Router-CIM 2012



© 2011 CIM-TECH.COM, Inc



by CIM-TECH.COM, Inc

We are pleased to announce the release of Router-CIM 2012 Automation Suite. This state-of-the-art programming software combines the latest advances in CNC machine tool programming with the industry-standard CAD features found only in AutoCAD®, the world's premier Computer Aided Design software

Router-CIM 2012

© 2011 CIM-TECH.COM, Inc

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Applied Topics Include:

Router-CIM 2012

Geoshape and Cut

Expert Nurbs Cutter

Pocketing

Profile cutting

Drilling

Configuration

Modification

Knowledge

CIM-TECH.COM, Inc.
7512 Dr. Phillips Blvd.
Suite 50-859
Orlando, FL 32819

Toll-Free Technical Support:
(877) 549-8211

Table of Contents

Foreword 0

Part I System Requirements and Installation 10

1 Product Overview 11
2 Installation Quick Start 12
3 Critical Issues 12
4 Installation 14

Part II Configuration Wizard 27

1 Instructions 28
2 Knowledge 29
3 Database 30
Create Knowledge Base 31
Import Knowledge Base 31
Delete Knowledge Base 32
Save Knowledge Base 32
Default Knowledge Base 34
4 Measurements 34
5 Show Wizard 35
6 Help 36
7 How to configure Router-CIM for your machine 36
8 Customizing the Knowledge Base 38

Part III Geoshape, Group, and Start Point Edit 40

1 Geoshape 40
2 Start Point Edit (Start Pt) 41
3 Open Shape Issues 42
4 RCIM Group Command 43
5 Troubleshooting Geometry 48

Part IV Tool Information 50

1 Tool Number and Comment 52
2 CRC Offset 53
3 Spindle Direction 54
4 Diameter 54
5 Radius 56
6 Tool Length 57
7 4 Axis Safe 58
8 Tool Type and Tool Category 59

© 2011 CIM-TECH.COM, Inc

9 Vertical and Horizontal Offset 59
10 Aggregate Offset 59
Spindle Offset 60
Collet Offset 66
Combination of Spindle AND Collet Offset 74
11 Cutter Compensation 77

Part V Modifying a Tool 80

1 Temporary Editing 81
2 Permanent Editing 84
3 Moving an Item in the List 93
4 Changing Tool Parameters 98

Part VI Cutting Cycles 101

1 Cycle Descriptions 105
Profiling Cycles 108
Plunge-Outside 108
Plunge-Inside 116
Plunge-Outside Line Leads 122
Ramp-Outside 128
Ramp-Inside 134
Heli-Lead Outside 140
Heli-Lead-Inside 145
Multi-Pass-Outside 152
Multi-Pass-Inside 159
Incline-Outside 168
Incline-Inside 173
Incline-Outside-Ramp 179
Incline-Inside-Ramp 184
No-Retract-Outside 189
No-Retract-Inside 194
Taper-Plunge-Outside 199
Taper-Plunge-Inside 205
Profile-Inside-Auto 211
Profile-Inside-Interactive 216
Drilling and Hole Cutting Cycles 221
Drill Motions 221
Fast Drill 224
Drill-Interpolation 227
Spot-Bore 232
Peck-Bore 236
Pattern Recognition 240
Fast Pattern Recognition 245
Deep-Bore 250
Helical-Arcs-Center 253
Helical-Arcs-Edge 257
Hole-Interpolation 1 260
Hole-Interpolation 2 265
Pocketing Cycles 270
Linear-Pocket 270
Spiral-Pocket 274

© 2011 CIM-TECH.COM, Inc

Linear Pocket Ramp-In ... 278
Spiral Pocket Ramp-In ... 282
Linear Pocket Lead Hole ... 286
Spiral Out2In ... 290
Spiral In2Out ... 294
Linear Pocket ZigZag ... 298
Linear Pocket Unidirectional ... 302
Open Shape and Center Cutting Cycles ... 306
Heli-Lead Center ... 306
Center-Line Cut ... 311
Center-Line Ramp ... 316
3D Poly Center ... 321
Profile ZigZag ... 326
Incline-Center ... 331
Right Hand Cut ... 336
Left Hand Cut ... 341
NURBS Surface Machining Cycles ... 346
NURBS Cycles Supplied ... 347
NURBS Cycle Parameters ... 348
Knowledge Editor in NURBS ... 352
Tools in XNURBS ... 356
Establishing the Feed Plane ... 357
Z Level Completion Option ... 357
Fall over Mode in Vertical Cutting ... 359
Containment Cutting ... 362
Constant Scallop Height ... 365
Intersect Cutting ... 367
Roughing ... 371
Vertical Roughing Mode 1 ... 372
Vertical Roughing Mode 2 ... 375
Vertical Roughing Mode 3 ... 378
Examples ... 380
NURBS XZ ... 381
Vertical Finishing at Angle 0.0 ... 383
Vertical Finishing at Angle Minus 45 ... 384
NURBS YZ ... 386
Vertical Finishing in YZ ... 388
NURBS CCW XY ... 389
Horizontal Finishing XY ... 391
NURBS Flowcut ... 392
Open Polyline Cross Flow Cutting ... 394
Closed Polyline Step Over Flow Cutting ... 398
Closed Polyline Spiral Flow Cutting ... 400
Closed Polyline Cross Flow Cutting ... 402
NURBS Project ... 405
Polyline Project Cutting ... 407
Specialty Cycles ... 409
4 Axis Cutting ... 409
4 Axis Saw ... 414
4 Axis Moulder Head ... 419
DO-IT ... 423
Tap-Matic ... 424
Square Corners ... 427
2 Cycle Parameters ... 431

© 2011 CIM-TECH.COM, Inc

Offset Dim 431
Cut Side 432
Cut Direction 435
Round Corners 438
Lead-In 439
Lead-Out 446
Lead Size 453
Lead Angle 454
Lead Feed 455
Lead Ratio 456
MaxOffset 456
XYStep 457
XY Stock Allowance 458
Z Stock Allowance 459
LastOffset 460

Part VII Status Information 461

1 Safety Plane 461
2 Depth Per Pass 462
3 Total Cut Depth 462
4 Material Thickness Compensation 462
5 Feedrate 463
6 Spindle Speed 463
7 Surface FPM and Units Per Rev 463
8 Before Codes and After Codes 465
9 Oscillation Amt. 465
10 Sort by Rank # 467
11 Acc-N-Dec 468
12 Metric 470
13 Plane Detect 470
14 Inline 472
15 Ramp Amt 473
16 Overlap Amt 473
17 DOIT File 474

Part VIII Cutting 475

1 Layer Control 475
2 Collect and Restore 475
3 Cutting Text 480
4 Toolcheck 482
5 Erasecheck 484

Part IX Sequence and Make Code 485

1 Sequence Definition 485
Clear All 486

© 2011 CIM-TECH.COM, Inc

Develop a Cut Sequence ... 487
Sort Cuts By ... 487
2 Tooling Requirements ... 501
Make Subroutines ... 501
Develop Tooling Motions ... 503
Block Sequence ... 503
Start and End Codes ... 503
3 Information Development ... 505
Make NC Code ... 505
Make NC Code Options ... 506
Acceleration/Deceleration ... 508
Perform Machine Calc. ... 512
Report Information ... 513
Edit Tooling Motions ... 514
Sequence Emulator Options ... 516

Part X Knowledge 517

1 Retrieve Knowledge ... 518
2 Save Knowledge ... 520
3 Export Knowledge ... 521
4 Import Knowledge ... 524
5 Edit Knowledge ... 526
6 Group Knowledge ... 532
7 Save Without Knowledge ... 536
8 Knowledge Notes ... 537

Part XI NC Vars 537

1 Setting NC Vars ... 538
NC Var Notes ... 539
Adding NC Vars ... 539
2 NC Vars Appendix ... 542
3 Geometric ... 542
4 Layer ... 543
5 System ... 544
6 Other ... 546

Part XII Tabbing 546

1 Tabbing Settings ... 547
2 2D or 3D Tabs ... 551

Part XIII Cutting Solids in Router-CIM 558

1 NCExtrude ... 559
2 NCExtract ... 563
3 NCDepth ... 564
4 NCUpdate ... 568

© 2011 CIM-TECH.COM, Inc

5 NCFlat 570
6 Shapehole 571
7 Parametric Toolpath in Router-CIM 571

Part XIV 4th Axis Programming 572

1 Geometry 572
2 Cutting 574
3 4th Axis Interpolation Cycle 577
Drawing Requirements 577
Cycle Corner Treatments 578
Outside Corners 578
Inside Corners 579
Position and Direction Control 580
Lead Styles 580

Part XV Horizontal Boring 588

1 Horizontal Boring 589
2 Using FS, RS, BS, LS 590
3 Setting up Knowledge 592
4 Using the Sample Drawing (COPY) 606
5 Avoiding Extra Safety Moves 608

Index 611

© 2011 CIM-TECH.COM, Inc

1 System Requirements and Installation

Hardware Requirements

The system requirements for Router-CIM 2012 are as follows:

Minimum Requirements	Recommended Requirements
Intel® or AMD® Dual Core processor, 1.7 GHz or greater	Intel® or AMD Quad Core processor 2.0 GHz or greater
2 GB RAM	4 GB of RAM or greater
750 MB free disk space for installation	2 GB of disk space available, not counting installation
1280x1024 VGA display with true color	1280x1024 32 bit color video display adapter (true color)
Microsoft® Internet Explorer 8.0 (SP1 or higher)	Microsoft® Internet Explorer 8.0 (SP1 or higher)
Microsoft® Windows® XP Professional (SP2 or higher), Windows Vista 32, or Windows 7 (32 bit).	128 MB or greater, OpenGL®-capable workstation class graphics card
DVD Drive to install Cam Companion	Microsoft® Windows® XP Professional (SP2 or higher), Windows Vista or Windows 7 (32 or 64 bit).

Supported Autocad Versions

Router-CIM 2012 supports the following Autodesk products:
AutoCAD 2010
AutoCAD 2011
AutoCAD 2012
Additionally, we provide an Inventor Link to the following versions of Autodesk Inventor:
Autodesk Inventor 2010
Autodesk Inventor 2011
Autodesk Inventor 2012

You should check your hardware against the Autodesk Certified Hardware List to be sure of compatibility with your version of Autocad.
Click here to check Autodesk Certified Hardware List

Hardware Locks

In addition to the hardware requirements listed above, it is important to note that Router-CIM and AutoNEST require hardware locks (often called 'dongles') in order to function. These can be either a USB type or a Parallel Port (printer port) type. Since there are two hardware locks required if you are using the nest software, you should have two USB ports available. For the parallel port type, only one port is necessary since these locks can plug into each other.

© 2011 CIM-TECH.COM, Inc

1.1 Product Overview

The main purpose of Router-CIM Automation is to provide a CAD/CAM solution for both single part machining and also to automate the programming task for quantities of parts. Router-CIM offers both nested based manufacturing technology, used to automatically produce NC Code for parts that are produced or imported into the AutoCAD environment, as well as any other single part that can be parametrically defined, or produced as a drawing or DXF file.

Parametric macros that are defined using Router-CIM's Parametric Macro Builder can be automatically sized and either nested or cut singly, and also from different materials.

With Router-CIM Automation, you can select macros, DXF files, and AutoCAD DWG files, associate a material and specify a quantity of them and Router-CIM will do the rest, automatically.

In Automation, typically, a job file is created that contains all the machining data necessary to perform the programming tasks. Each job contains the macro, drawing, or DXF file name, size, quantity, material, and other variables relevant to your cutting needs. Each part is analyzed, and a layer to knowledge association automatically creates the tooling paths. The part and tool paths are serialized and stored in the database as necessary. Once all parts are cut or cut and/or nested, then sorted NC Code is made. The parts or nests are printed, and external files, like a label file and also a machine schedule file are made.

Each job is stored in an independent folder for easy storage and retrieval. The locations of these folders can be specified either locally on your hard drive, a network drive, or even the machine tools drive if it is able to exist on the same network.

For new users of Router-CIM, there are a few concepts which need to be understood in order to use the product efficiently. These are:

- A basic understanding of the Microsoft Windows environment, and file handling with programs such as My Computer, or Windows Explorer.
- A working knowledge of the AutoCAD environment. How to create and name layers, and assign geometry to those layers.
- Some knowledge of the types of tools, cutting conditions, and materials you are likely to use on your parts.
- Knowing the difference between the file types Router-CIM uses such as DWG, DXF, and SCN.
- Understanding the Layer to Knowledge association feature in Router-CIM.

If you will be using primarily macros for your part definitions, then a solid understanding of Router-CIM's Parametric Macro Builder and how to create a macro is truly necessary. This takes practice and is best done on paper first!

Also solid understanding of the variable types, Global, Dynamic, Tagged, and Local, and how to use them on individual parts or assemblies of parts will be of primary importance in a truly parametric macro. You should know how to incorporate the variables into formulas to define parts or part features.

© 2011 CIM-TECH.COM, Inc

1.2 Installation Quick Start

Installation Quick Start:

These installation notes assume that AutoCAD 2010 or higher is installed and functioning properly.

Make sure you are logged in with at least Local Administrator privileges on your system.

The Router-CIM 2012 program is installed in the following manner.

- Unplug any USB hardware locks previously installed.
- Insert the Router-CIM Installation CD and it will start automatically.
- If the Installation doesn't start automatically, pick Start, Run, D:\SETUP. EXE (where D: is the CD ROM drive letter).
- The Router-CIM installation serial number will determine the installation options of Router-CIM. The serial number is included on the CD that Router-CIM is installed from. If Nesting was purchased, the Router-CIM installation will automatically install the product.
- The Install will place the Router-CIM files in the default locations.
- The install procedure searches for AutoCAD on your hard disk, and uses this location to build icons to run the Router-CIM program.
- Once the install is finished, reboot the computer, plug in the USB hardware locks and you will be able to run the product.
- If any custom post processors are necessary, install them now. The procedure to install the post processors is to simply run the .exe file that contains the post processor, and answer a few default questions during the install.

1.3 Critical Issues



After Router-CIM is properly installed and configured for use, all pertinent folders should be backed up. A simple method is by copying them and placing them in a new folder. The proper way would be to make a CD, or back up to tape or another easy to restore method.

CIM-Tech will store a backup of your system if you burn it to a CD and mail it to us. We will archive that as an off-site backup in case you lose your own. This is NOT a substitute for your own backup, as you will likely make changes and edits on a more regular basis than the backup that is shipped to us.

CD recorders, DVD Recorders, External Hard Drives, and Tape backups are all valid ways to back these folders up. Floppy disks are not a good choice. It would take a large number of them to get each folder in its entirety. Typically all of the folders can fit on one CD. After using the system for a while, the backups that Router-CIM stores on your system could mean that more than one CD is necessary.

A Note on CD Recorders: Each file that is backed up in this fashion may become read-only. This means you can only look at the documents without altering or editing them. It will be necessary to change the properties of the files back from read-only after the folder is copied. The method of doing this varies depending on your operating system.

The folders to back up are as follows:

© 2011 CIM-TECH.COM, Inc

1.C:\Router-CIM
2.C:\Rcim_work
3.C:\Anest
4.C:\Automation_code

(C: is assumed here to be the installation drive.)

CIM-Tech is NOT Responsible for your loss of productivity or data! If you do not back up your system completely and you lose data without a backup, your only option is to re-install the software and start over.

© 2011 CIM-TECH.COM, Inc

1.4 Installation

Installation Guide

The Router-CIM program is installed in the following manner:

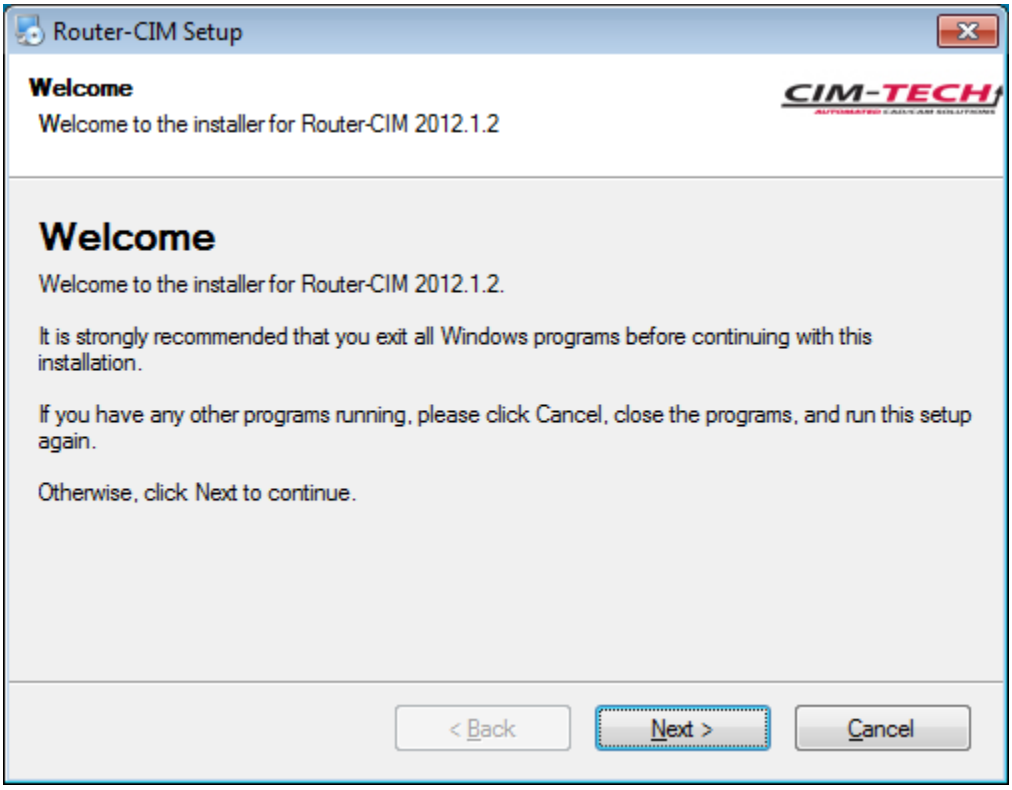
Insert the Router-CIM Installation DVD and it will start automatically. If it does not start automatically, pick Start > Run then pick or type D:\setup.exe (where D: is the DVD ROM drive letter).

The Router-CIM serial number will determine the installation options. If Nesting was purchased, it will be installed automatically.

The install will search for Router-CIM files in the default location. If your version of Router-CIM was installed to a drive other than C, you can re-direct the setup process to locate the folder in the proper drive by clicking on Browse and then selecting the correct drive location.

The install procedure searches for AutoCAD on your hard drive, and uses this location to build icons to run the Router-CIM program.

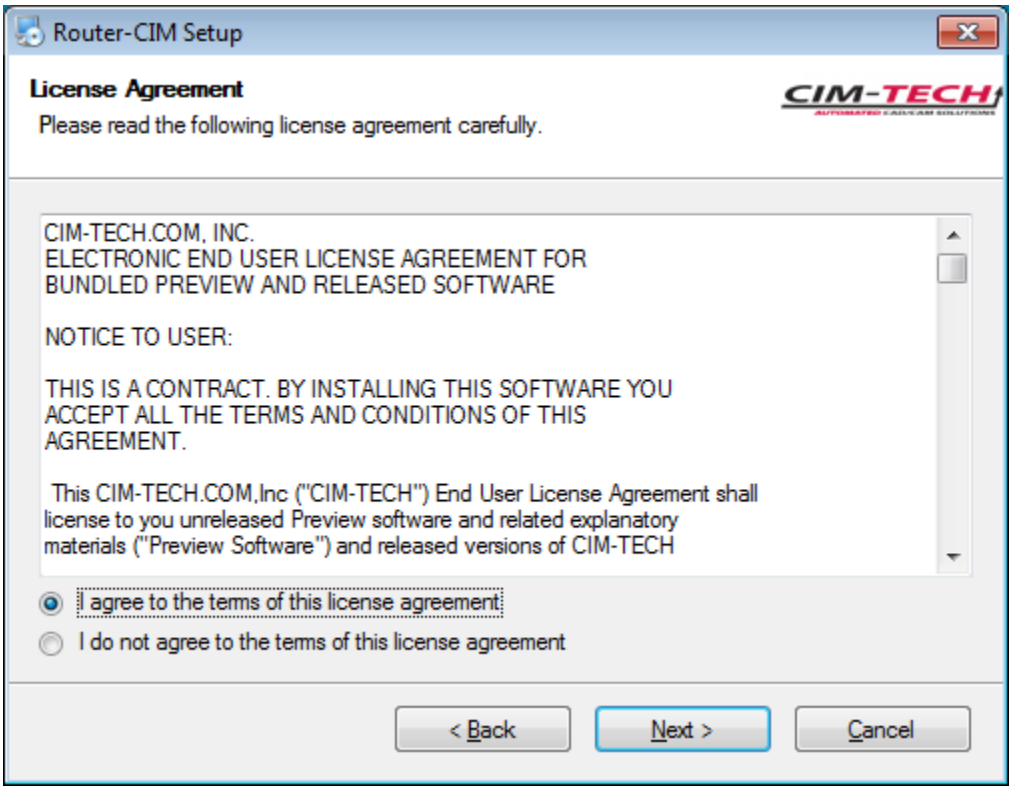
When the installation program starts, the following screen will appear:



From here, all of the default selections have been made for you. We recommend that you leave the defaults, as it will be simpler for you later if technical support is needed and you need to know the locations of the Router-CIM files.

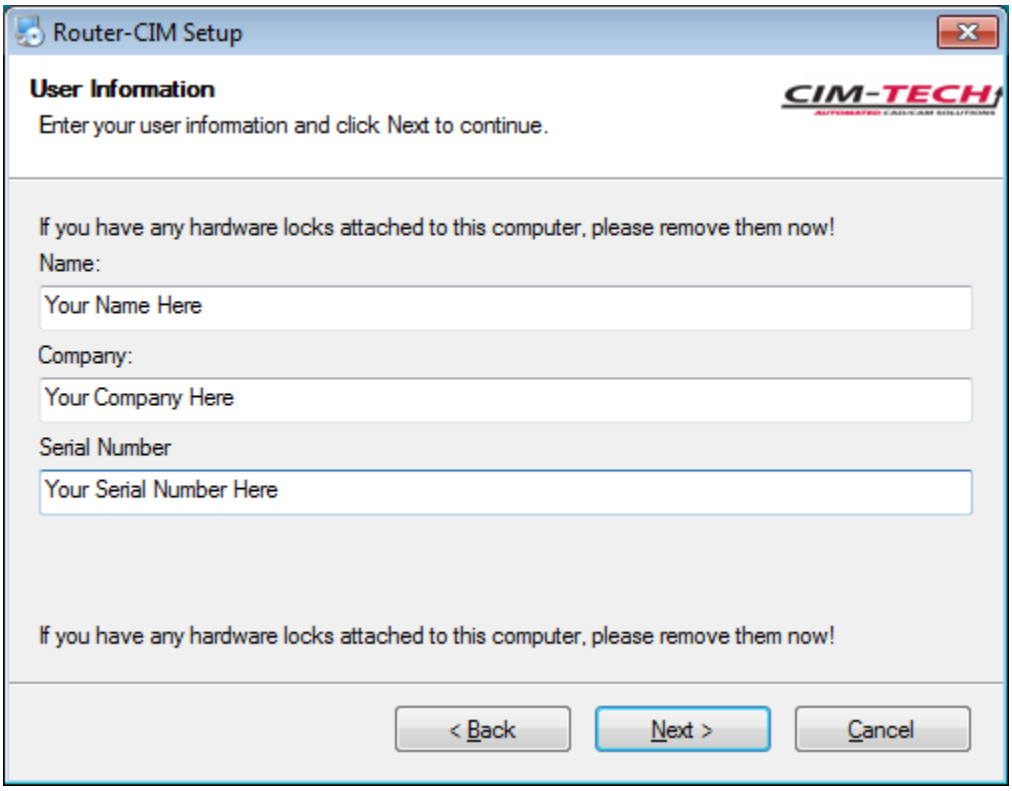
© 2011 CIM-TECH.COM, Inc

Select Next to continue.

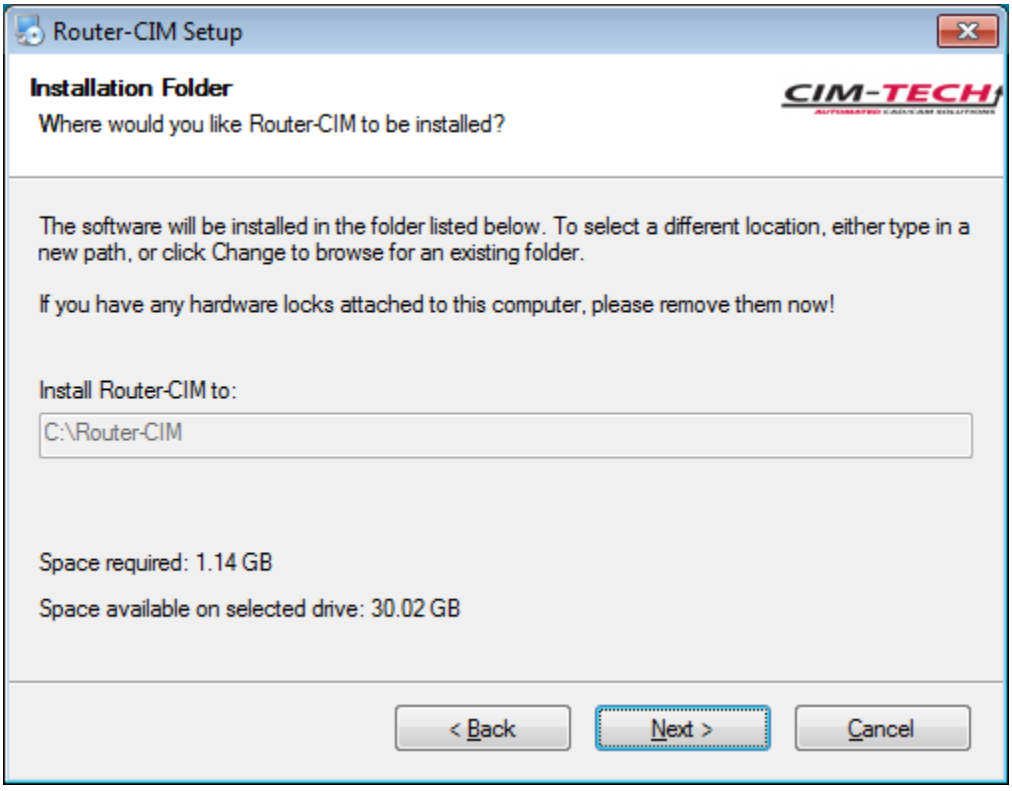


This is the license agreement for Router-CIM. Be sure you read the agreement. Selecting 'I do not agree' will allow you to exit the program without installing it to your computer. Selecting Back will return you to the previous window. Select 'I agree to the terms' and Next to continue.

© 2011 CIM-TECH.COM, Inc



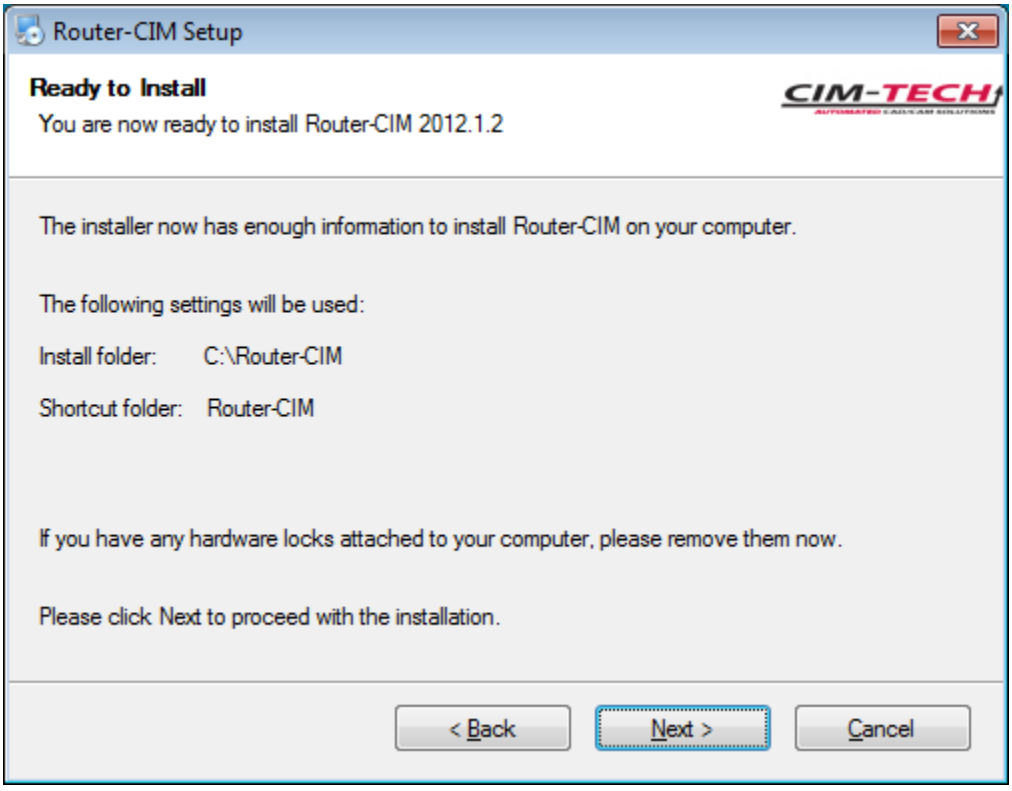
Enter your name, your company name, and the serial number that is supplied to you with the software. Select Next to continue.



© 2011 CIM-TECH.COM, Inc

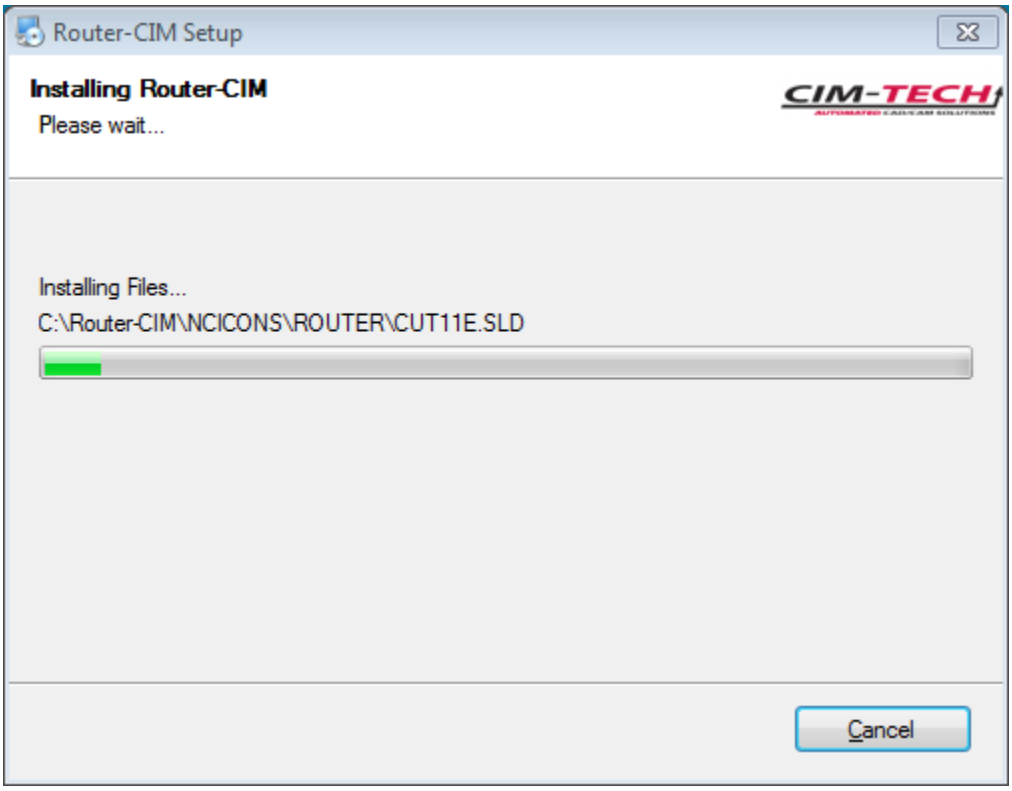
Router-CIM will specify the location for installation. You may select Next to continue, or either Back to go to the previous window or Cancel to exit from the install.

You should remove any hardware locks that have been plugged into the computer.

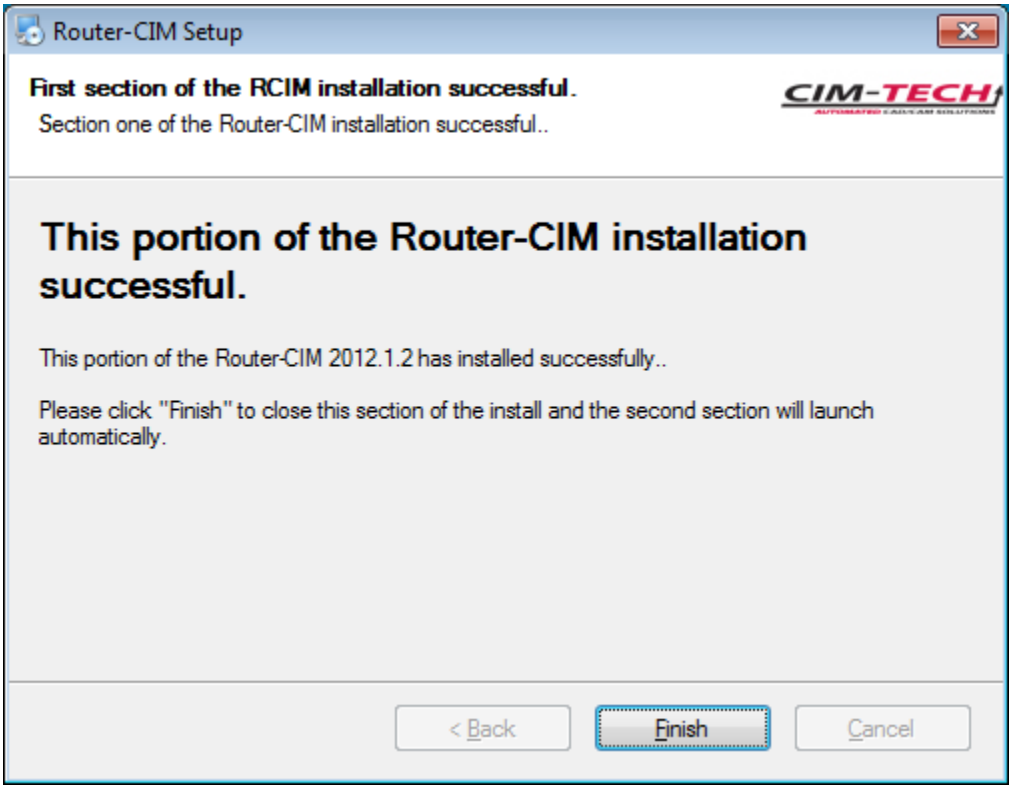


This screen will recap where Router-CIM is going to be installed to.

© 2011 CIM-TECH.COM, Inc

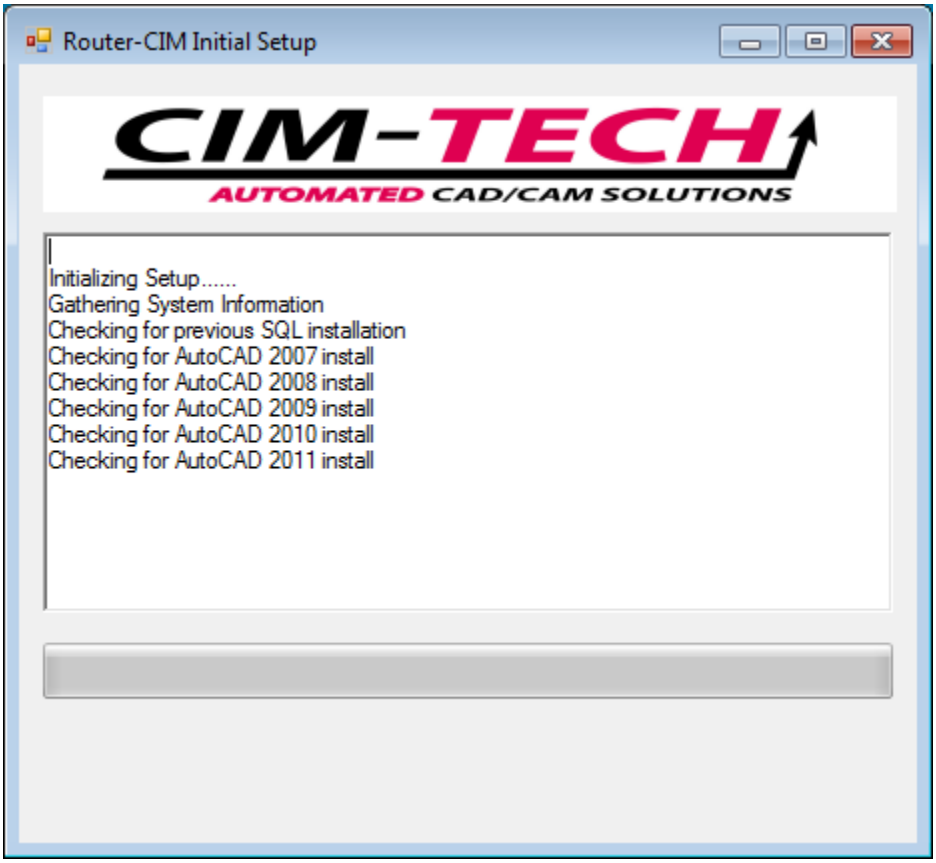


The installer will start copying files to the necessary locations in this section of the install. As soon as this is finished you will see the following screen:



© 2011 CIM-TECH.COM, Inc

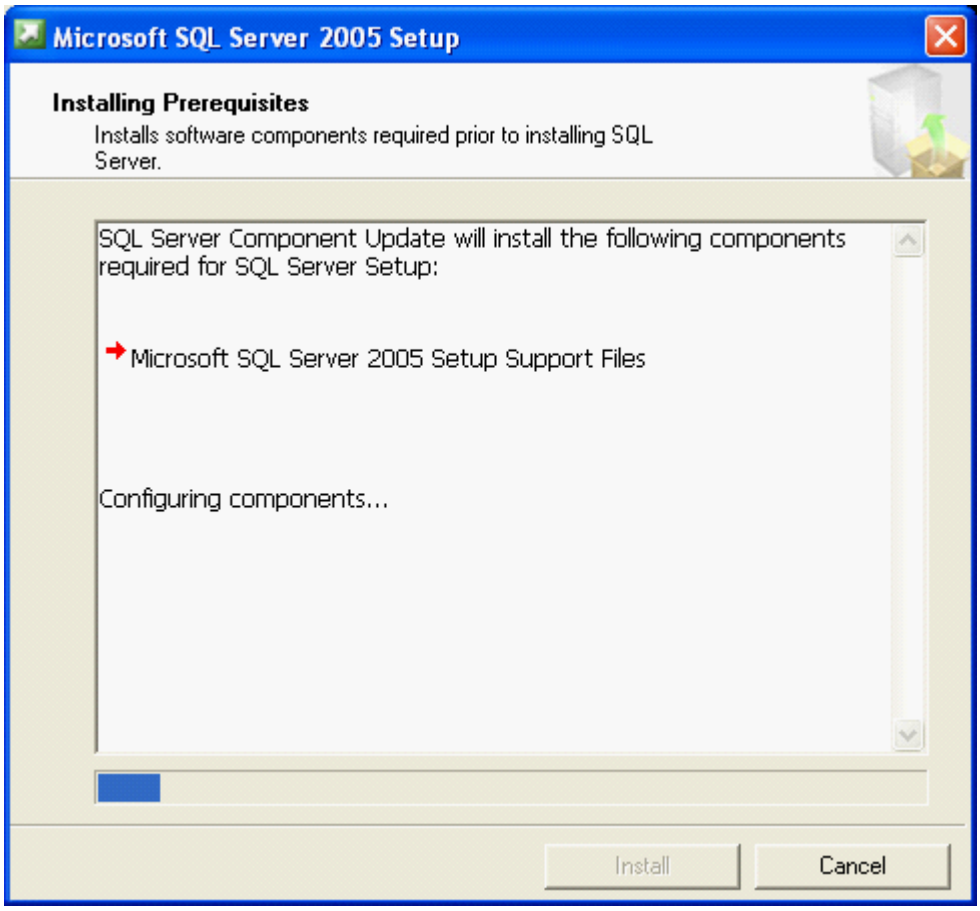
Click on Finish to start the next section of the installation.



This is the second section of the installation. The install will extract some more files and then check your computer for versions of AutoCAD, SQL Server and Microsoft .NET Framework versions.

This section may take a while, please be patient and allow the installer to finish.

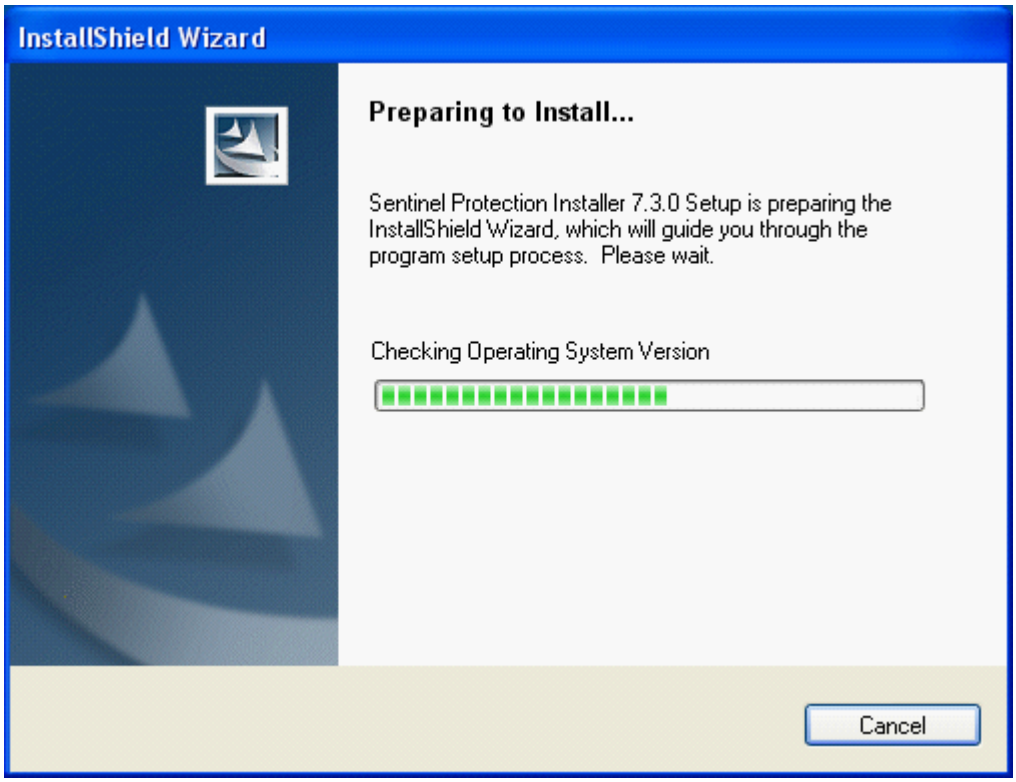
© 2011 CIM-TECH.COM, Inc



The next screen to appear will be the .NET Framework. If the proper version of the .NET Framework is not detected, then it will install.

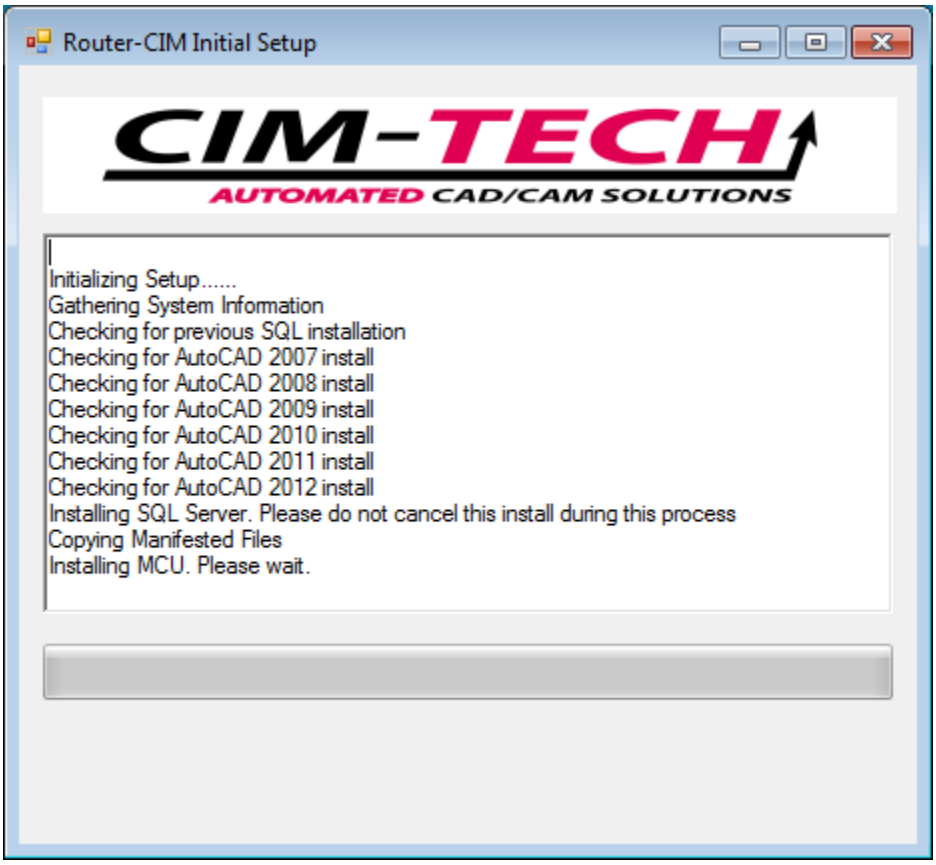
There is no visible progress meter during this install, so just leave the window open and do not cancel the installation. The next step will appear when this is finished.

© 2011 CIM-TECH.COM, Inc



The Sentinel Protection Installer will run next to install the hardware lock drivers. This is usually pretty quick and you may see some windows open with a black background while this is installing.

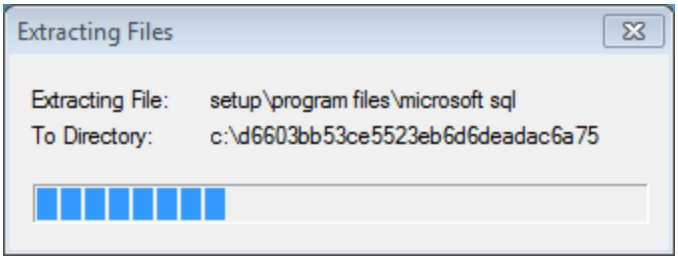
© 2011 CIM-TECH.COM, Inc



During the Sentinel install, the Router-CIM Progress screen will look like the one above.

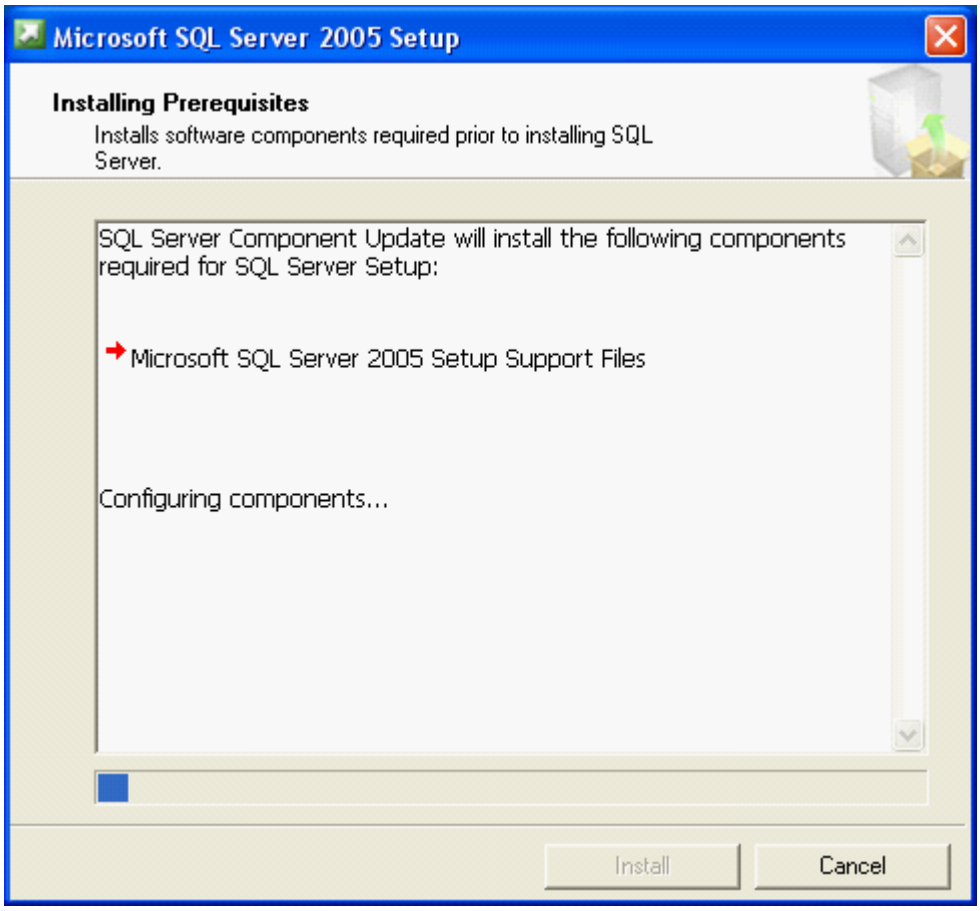
After the Sentinel Protection Server is finished installing, the next section of the install is the SQL Server Express install.

During this install, you will likely see a small progress bar like this:



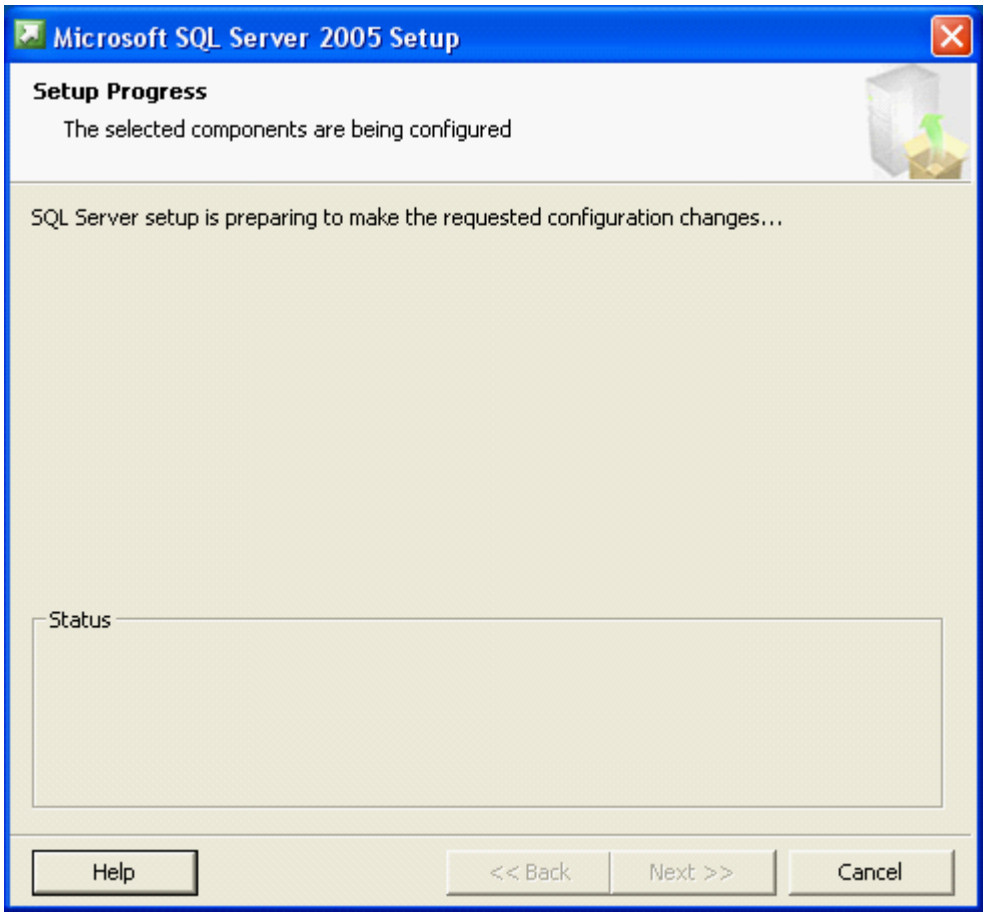
The progress will continue until the SQL Server is installed and configured.

© 2011 CIM-TECH.COM, Inc



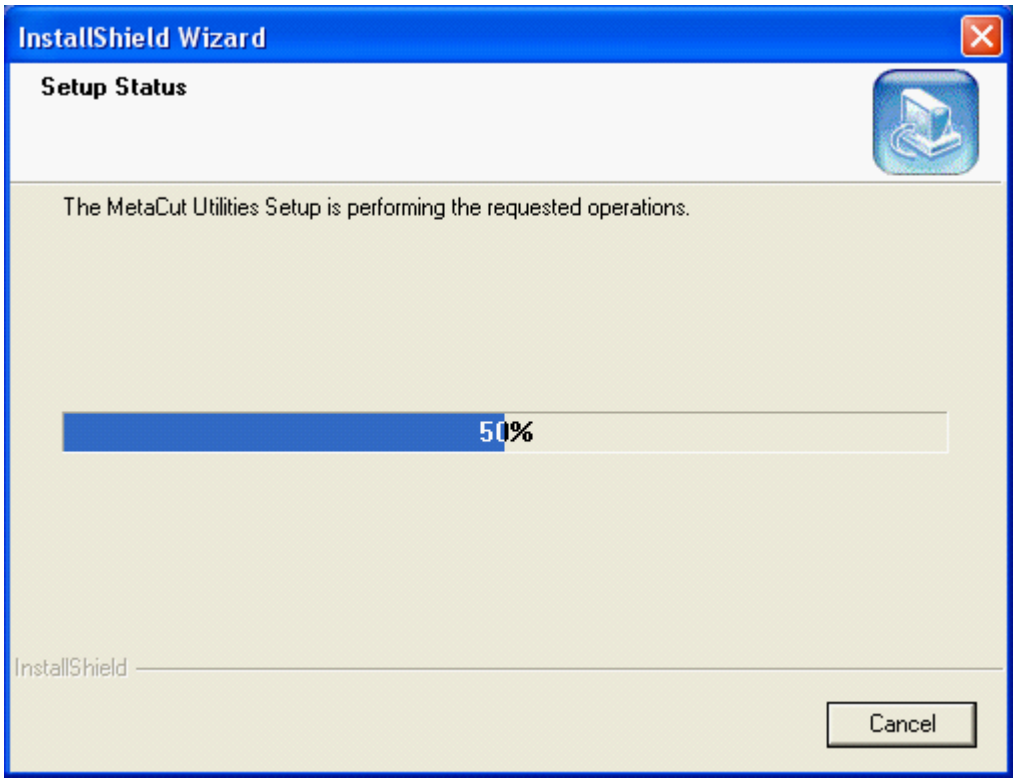
During the configuration of the SQL Server, the window above will appear. Do not hit Cancel, the SQL will continue to install and configure.

© 2011 CIM-TECH.COM, Inc



The SQL Server install screen will change slightly as it finishes its configuration.

© 2011 CIM-TECH.COM, Inc

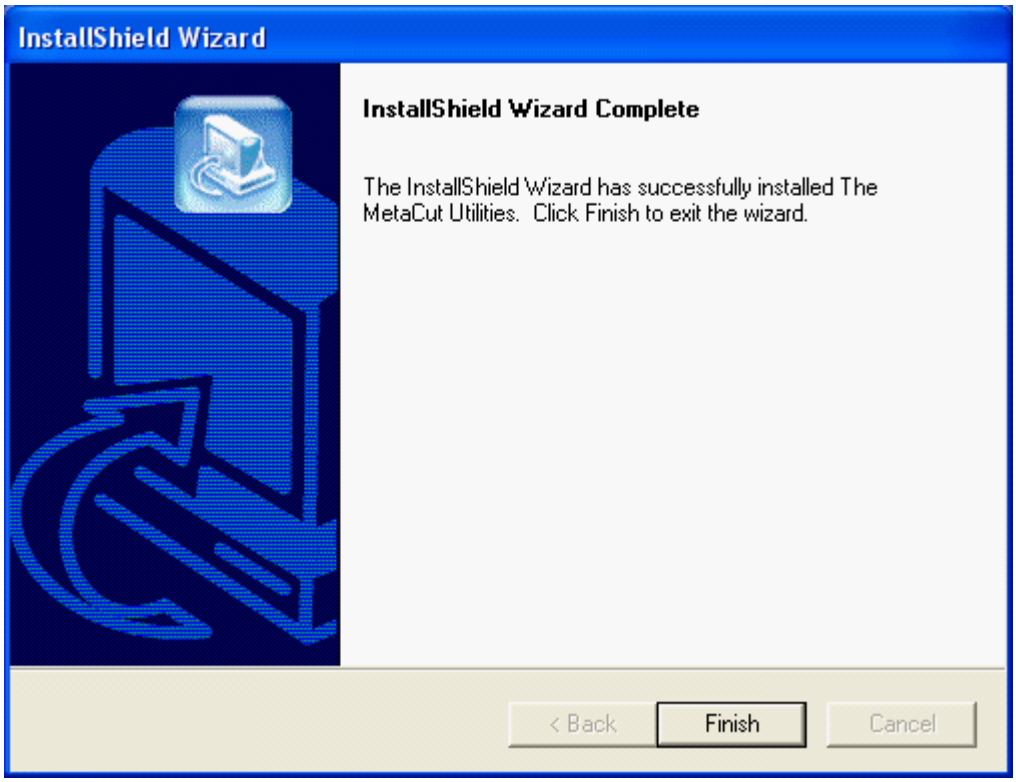


Tool Path Verification Software Install / Re-Install.

If the Tool Path Verification software (MCU) was purchased, the above screen will appear during the install.

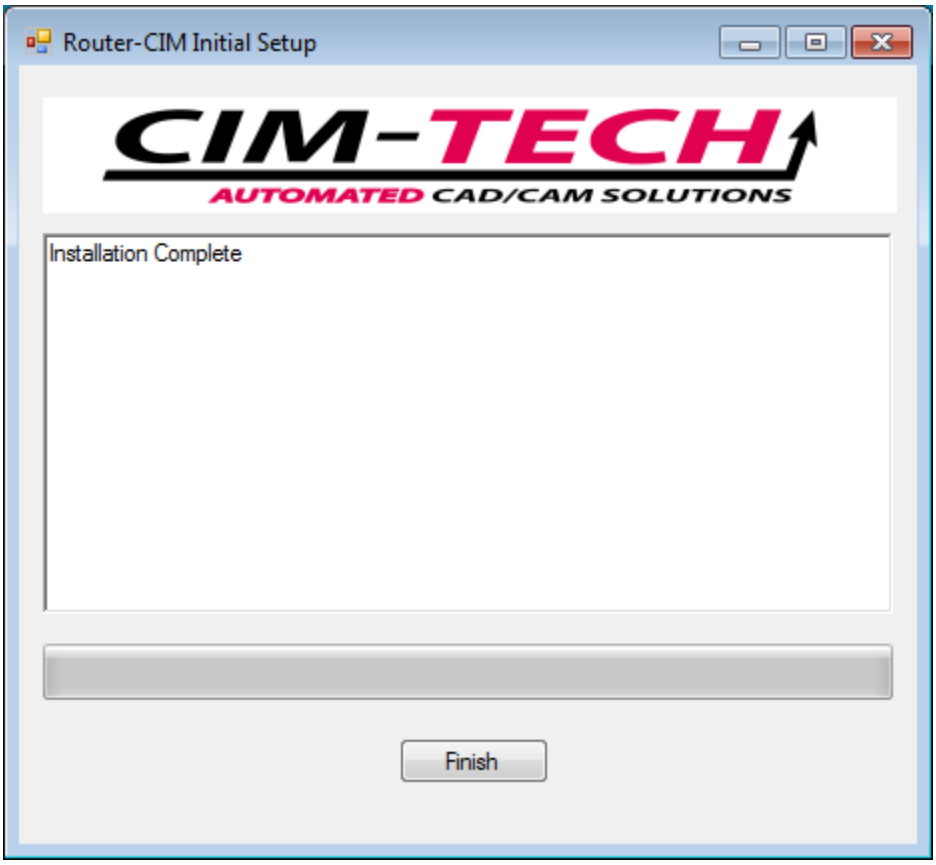
Once MCU is done installing, the following screen will appear.

© 2011 CIM-TECH.COM, Inc

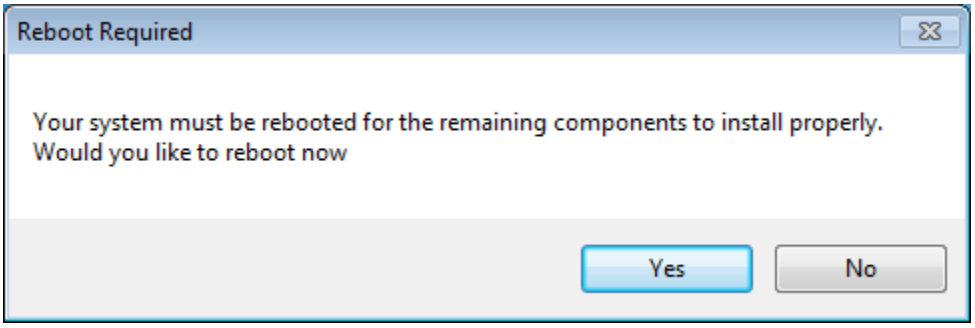


Select Finish to continue.

© 2011 CIM-TECH.COM, Inc



Once the rest of the Automation components are installed, the installation is finished. You will see a message requesting you to Reboot your computer. The installation is not finished until you reboot your computer.



After the Reboot is finished, plug the hardware lock into the computer. There is either a parallel port lock (25 pin device) or a USB lock for a USB port.

2 Configuration Wizard

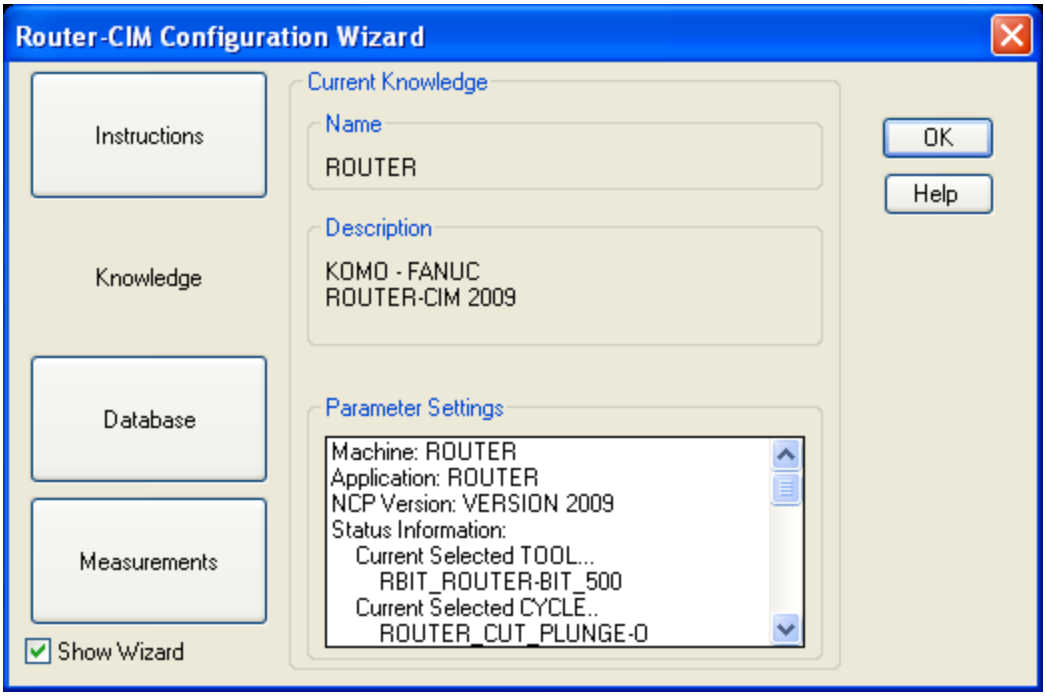
The Configuration Wizard

A knowledge base is created when a machine (Router) is linked to a postprocessor. A knowledge base name will always default to the name of the postprocessor. However, you are free to choose a name to your liking. Knowledge bases are saved as part of the current AutoCAD drawing.

© 2011 CIM-TECH.COM, Inc

As you use Router-CIM you will find that altering knowledge base settings and/or defaults to suit your particular situation can be advantageous. These alterations can be saved under a new knowledge base name. The Configuration Wizard is your tool to managing knowledge bases.

The Configuration Wizard dialog appears when Router-CIM is launched. The Current Knowledge base appears in the grayed out area at the center of the dialog screen. It contains the Name and Description of the knowledge base that was last defined by the Configuration Wizard, or it will display the most recently dated postprocessor found when Router-CIM is started for the first time.



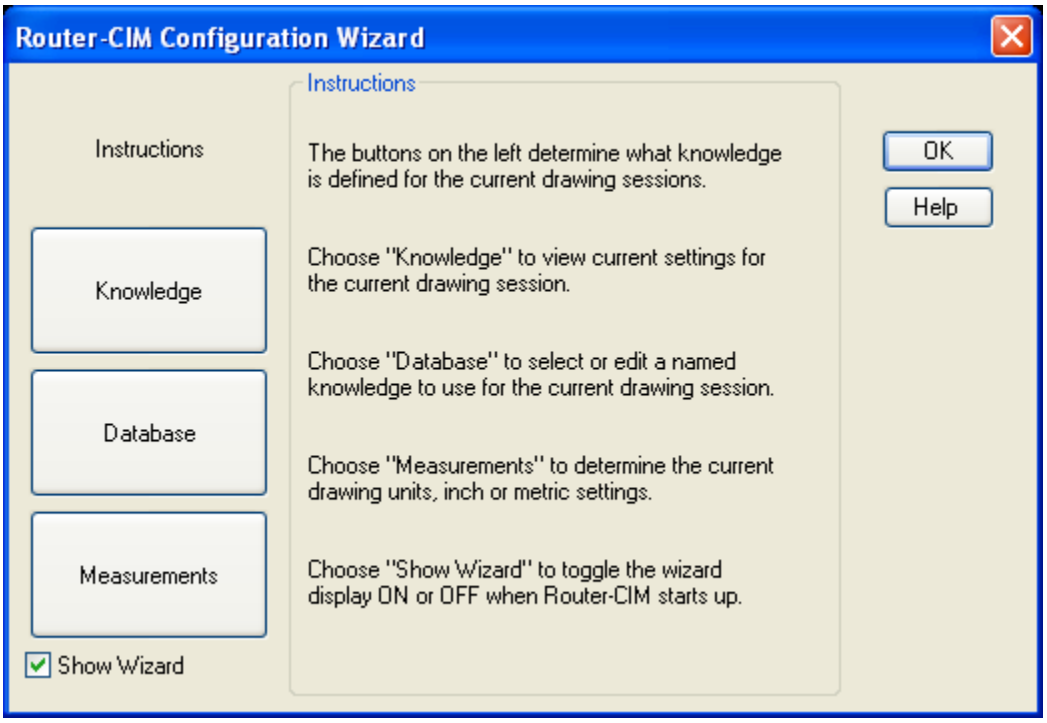
View the Parameter Settings in the window of the dialog screen. If the Machine and/or Application parameters are not what you want for a configuration, use the Database button to retrieve a different machine or postprocessor.
See "How to Configure Router-CIM for Your Machine" (in this chapter) for step-by-step instructions.
To customize knowledge base settings and/or defaults see "Customizing the Knowledge Base" (in this chapter).

Click OK to exit the Configuration Wizard and establish the selected knowledge base in the current drawing session. After clicking OK you can return to the Configuration Wizard to select a new postprocessor or to modify a knowledge base. To return to the Wizard, you must go to the RCIM pull-down menu and choose Configuration Wizard.

2.1 Instructions

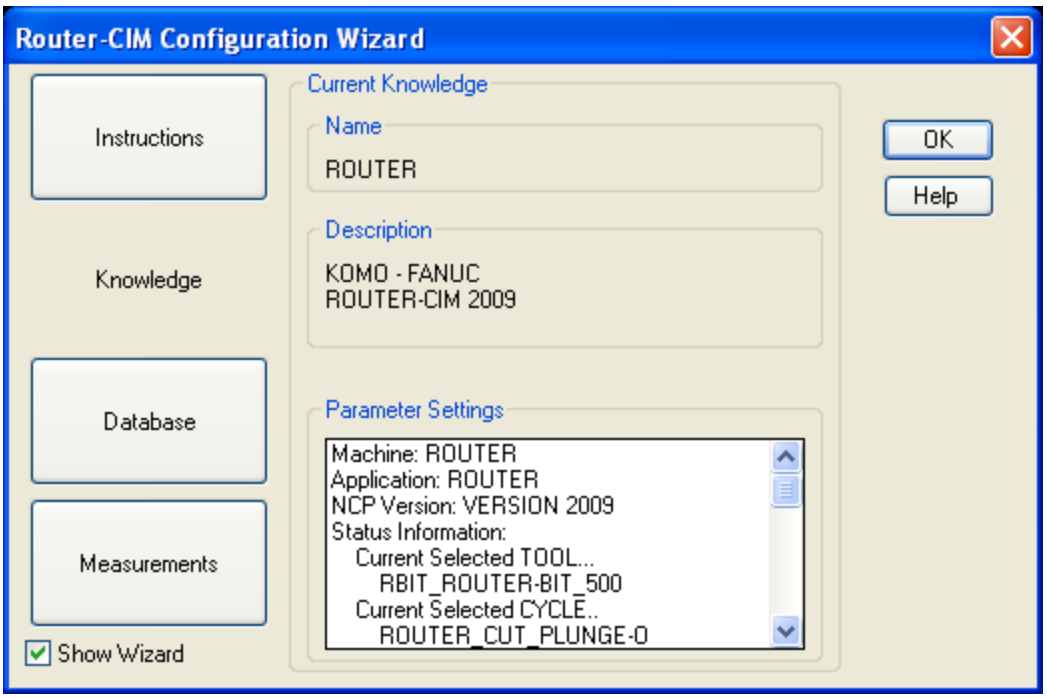
Choose "Instructions" to see a brief description of each button choice.

© 2011 CIM-TECH.COM, Inc



2.2 Knowledge

Knowledge is the default choice when the Configuration Wizard appears.
This window will show information for the currently selected post processor.



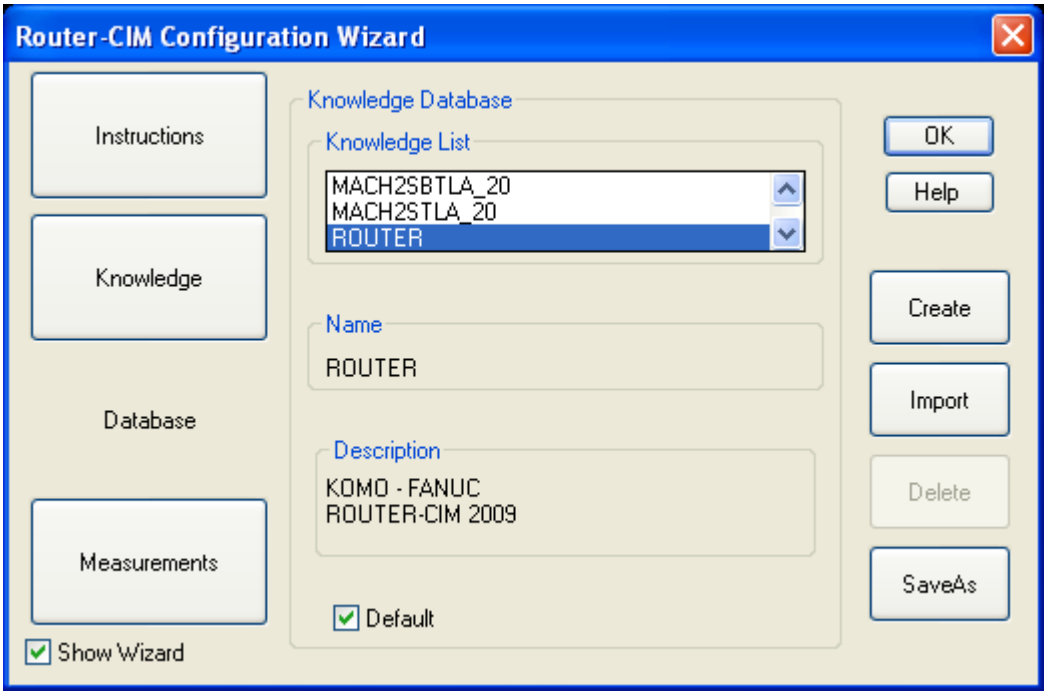
© 2011 CIM-TECH.COM, Inc

The Name and Description fields contain the information found in the postprocessor file as it was delivered to you. If you have made custom knowledge bases, and made one of them the Default, it will display the information for that system.

The Parameter Settings are obtained by the system, so information in this screen cannot be edited. Status Information displays the default configuration for the current knowledge base and it displays information such as the Machine Post name and the version of Router-CIM the post and post drawing were configured from. The default Tool, Cycle and Status information for the knowledge drawing are shown here as well.

2.3 Database

Click Database to display the available Knowledge List, Name and Description for any named knowledge bases you may have created or installed. In Database mode, new buttons appear that allow you to Create, Import, Delete and Save knowledge bases for use in Router-CIM.



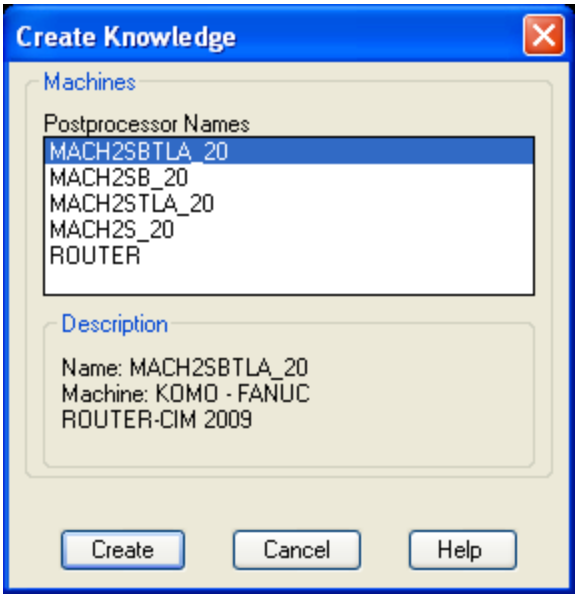
The Database screen shows the available Knowledge database. Here you can change from one knowledge base to another. If you are just starting a session, and there are multiple entries in the Knowledge List, you can change from one machine to another.
By selecting a machine from the Knowledge List, all available post processors and named knowledge bases for the selected machine appear. If no post processors have ever been accessed by the Configuration Wizard, you will be asked to create a named knowledge base by selecting a postprocessor. This will create a Knowledge List of one knowledge base using the selected postprocessor. If you have additional post processors you can create additional knowledge bases using the Create button. The Create button will generate additional knowledge bases and they will appear in the Knowledge List. By selecting an item in the Knowledge List, the Name and Description edit boxes are filled in.

You can select the buttons from the picture above to move to those topics.

© 2011 CIM-TECH.COM, Inc

2.3.1 Create Knowledge Base

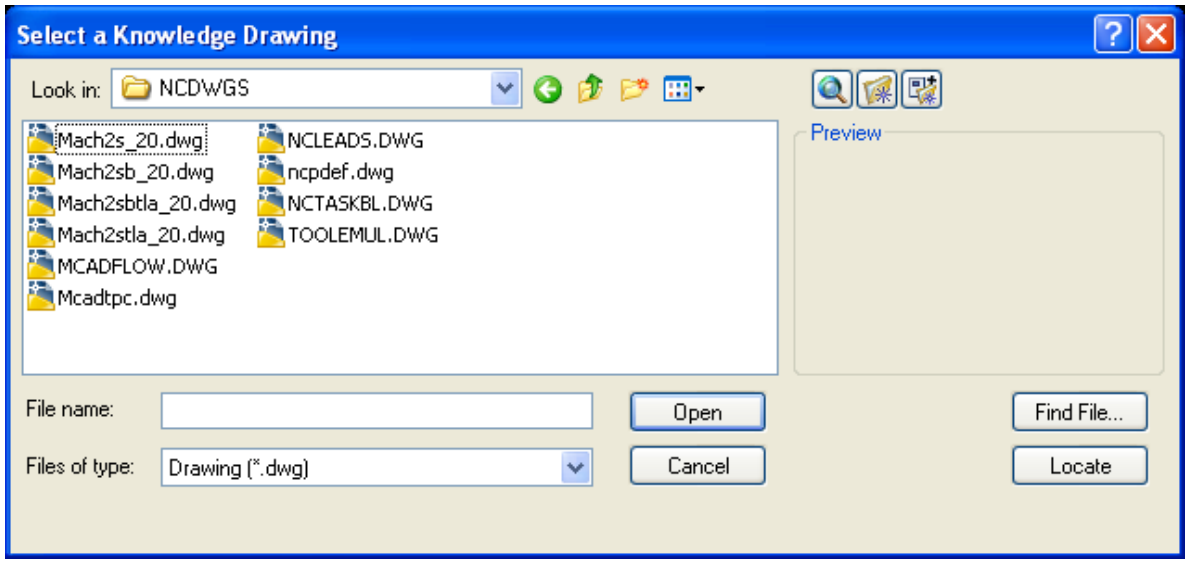
Create will open a selection window that allows you to create a new configuration from an existing configuration that is in the Router-CIM\Ncpost folder.



The Create button creates a new knowledge base, based on the information found in a selected postprocessor. Select the desired postprocessor name and the knowledge base is created. The Configuration Wizard will return you to the Database display.

2.3.2 Import Knowledge Base

Import will open a file selection window and you can select a knowledge drawing from there to define a new configuration. Any drawing that Router-CIM has been successfully run in and saved is potentially a new knowledge drawing.



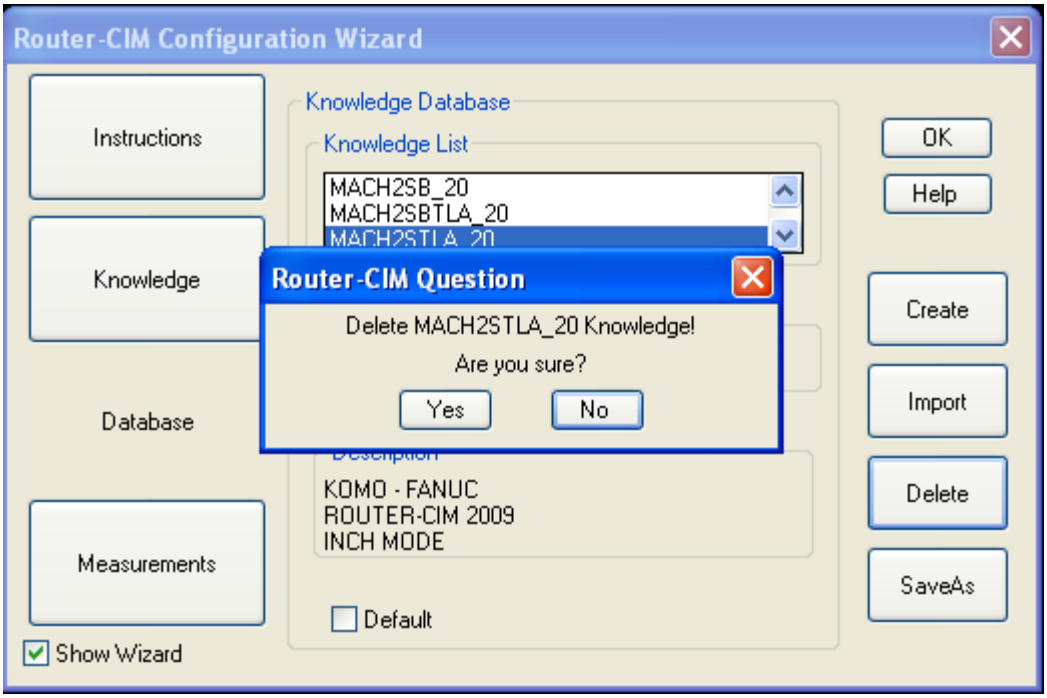
The Import button imports a knowledge base defined in another drawing. Existing Router-CIM users

© 2011 CIM-TECH.COM, Inc

can import the knowledge bases used in previous versions of Router-CIM. Select a drawing that you know has a knowledge base, and the Knowledge list will be updated with the imported knowledge base using the name of the drawing selected. If the selected drawing has a knowledge base that does not agree with the current machine choice, the knowledge base in the selected drawing will be rejected.

2.3.3 Delete Knowledge Base

The Delete button is the opposite of the SaveAs button. The Delete button removes named Knowledge Bases from the configuration database.
This function will allow you to delete a selected knowledge base from the list. You will be prompted to be sure that this is the action you wish to perform.



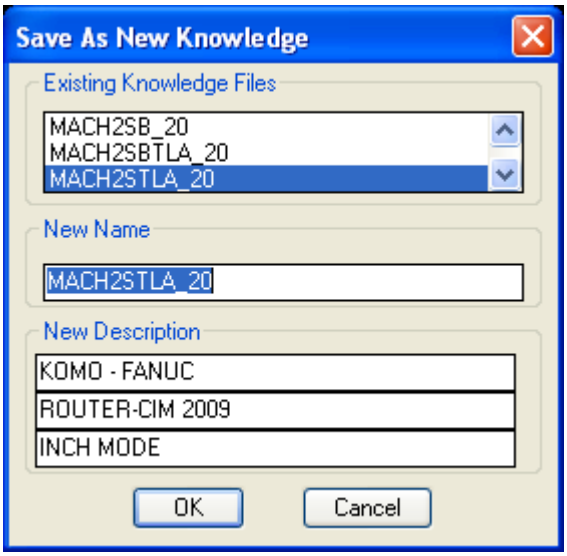
Once a knowledge base is deleted, you will have to create or import it and save it again to get it back.

If a Default knowledge is deleted, the Router becomes the default the next time Router-CIM loads.

2.3.4 Save Knowledge Base

The SaveAs button saves a selected knowledge base by name into the configuration database. The configuration database contains all the named knowledge bases that you see in the Knowledge List for each machine.

© 2011 CIM-TECH.COM, Inc



Existing Knowledge Files

The Existing Knowledge Files list displays the existing knowledge bases available for the current machine.

New Name

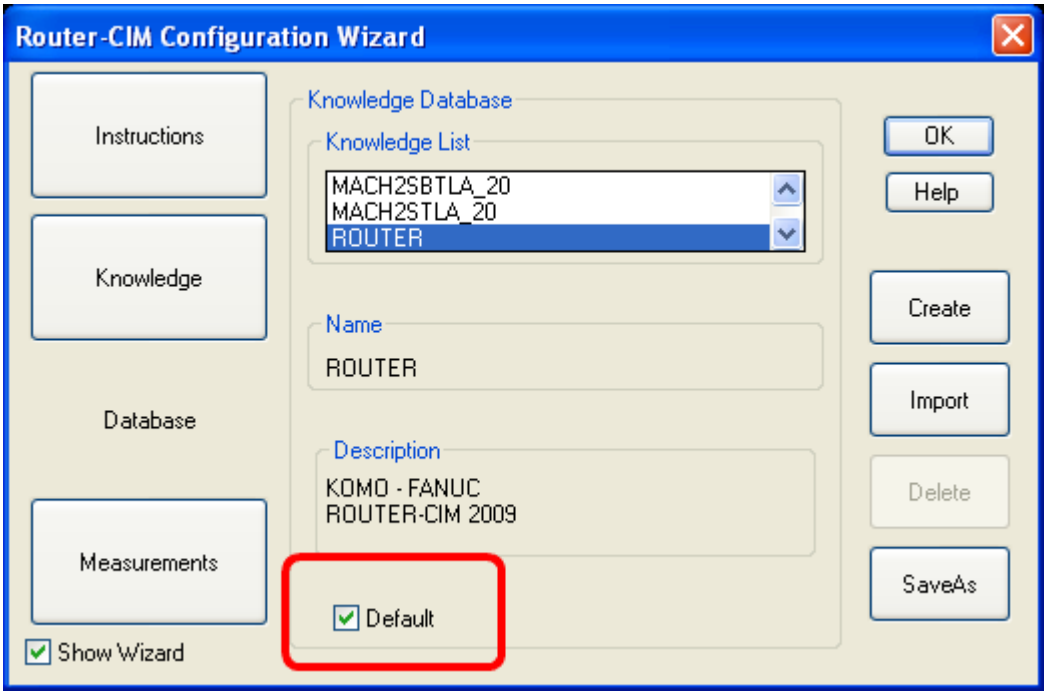
The New Name edit box is used to create a new knowledge base name for the Knowledge List. By typing a new name in the edit box and clicking OK, a new knowledge base name is created in the Knowledge List. This new name will have the same Description as the original unless you edit the Description fields.

New Description

The New Description fields are used to describe the named knowledge base. There are three lines of information available.

© 2011 CIM-TECH.COM, Inc

2.3.5 Default Knowledge Base



The Default check box indicates which knowledge base will be the default when Router-CIM starts. Check this box when the desired knowledge base is selected in the Knowledge List. Only one default can exist at any time. When you check this box, the previous default is removed and a new one established for the current named Knowledge.

2.4 Measurements

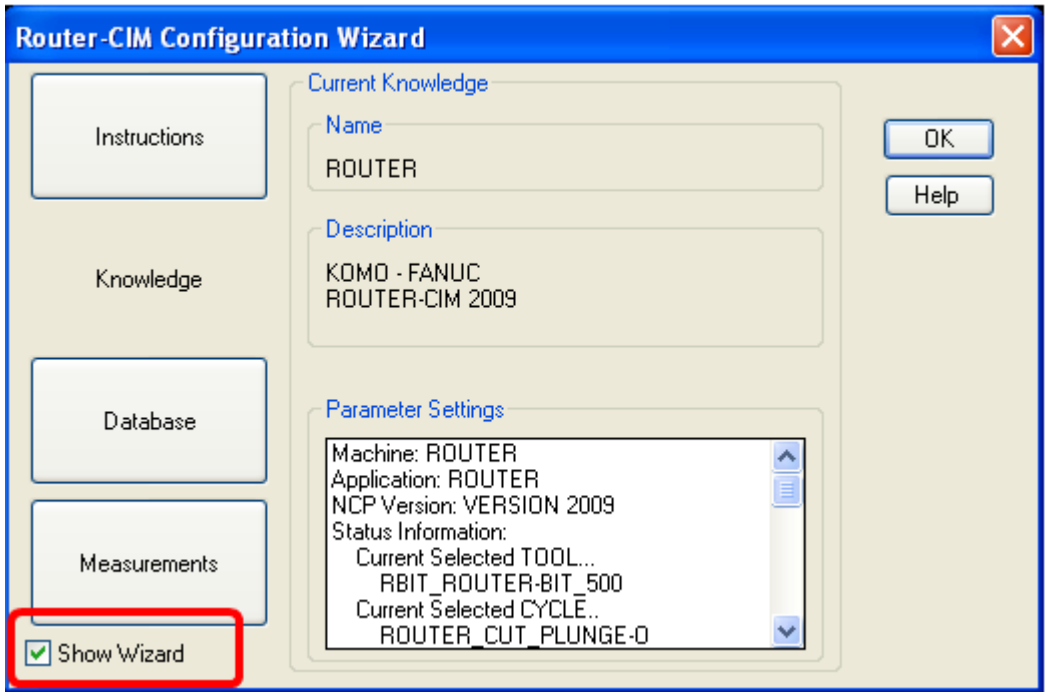
This button allows the setting of metric or inch units for the current drawing session. All the Router-CIM variables that depend on units are updated to this selection. If a drawing already exists, the units are predefined. You can change the unit settings, but they will not agree with the settings of the original drawing. The metric or inches setting in NC Vars is not affected.
In other words, changing this setting will not scale your drawing, but only change the default units that the post processor reports when making NC Code.

© 2011 CIM-TECH.COM, Inc



2.5 Show Wizard

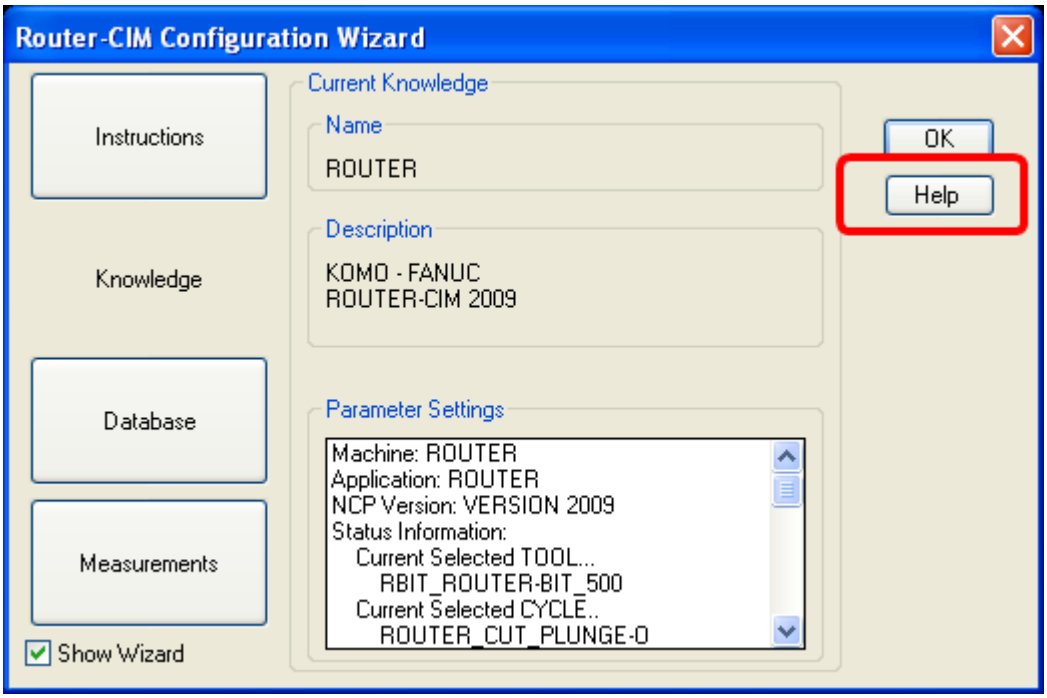
Choosing this will allow the Configuration Wizard to be displayed each time a Router-CIM session is begun. If this is unselected (remove check) then Router-CIM will load without the Configuration Wizard each time a drawing is begun that has a knowledge base in it.
The wizard will still be displayed when no knowledge is present in the drawing.



© 2011 CIM-TECH.COM, Inc

2.6 Help

Selecting the Help button displays a popup help covering the basic topics in this chapter.



2.7 How to configure Router-CIM for your machine

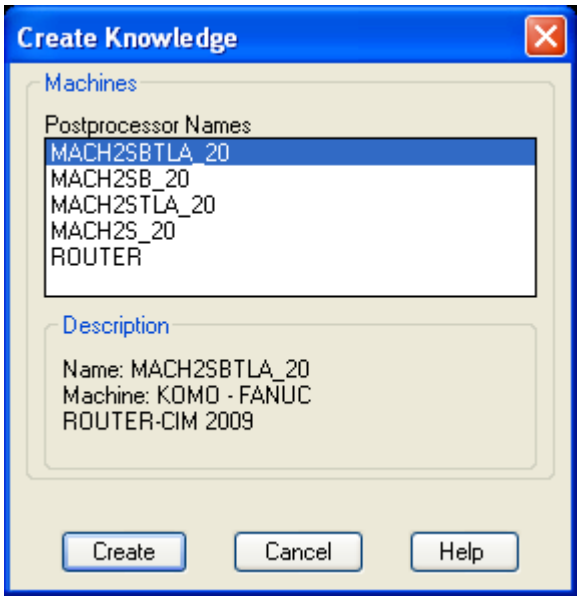
Included here are basic instructions on how to configure Router-CIM for your machine tool. It assumes that you have a post processor for your machine installed, and available to Router-CIM.

1. View the Database
After you launch Router-CIM and the Configuration Wizard dialog appears, click on Database to view the knowledge database. If the desired Knowledge is displayed, you are ready to begin using Router-CIM. If this is not the case, proceed to step 2.

2. Select a Machine
If the correct machine is displayed (Knowledge List) but the wrong postprocessor (Name) is displayed, proceed to step 3.
If the wrong machine is displayed (Knowledge List) select the correct machine from the list.

3. Create a New Knowledge Entry
Click on Create to be presented with a list of all of the post processors that exist in your system.

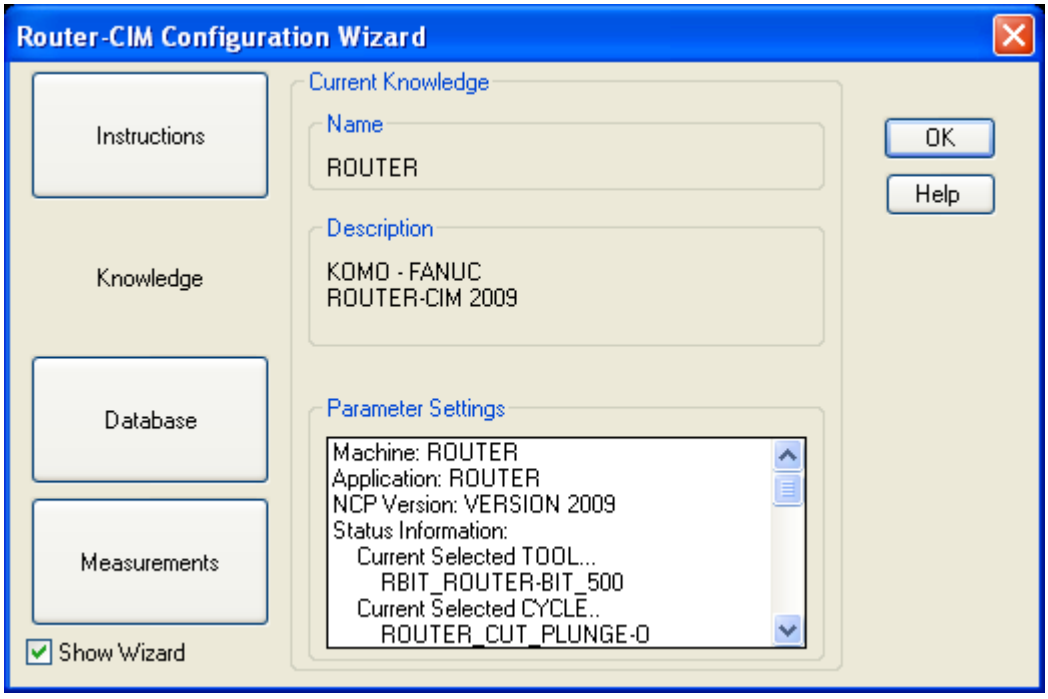
© 2011 CIM-TECH.COM, Inc



Select the post processor from the list that fits your machine and click Create to link the selected post processor to the selected machine.

5. View the Information for the New Selected Knowledge.

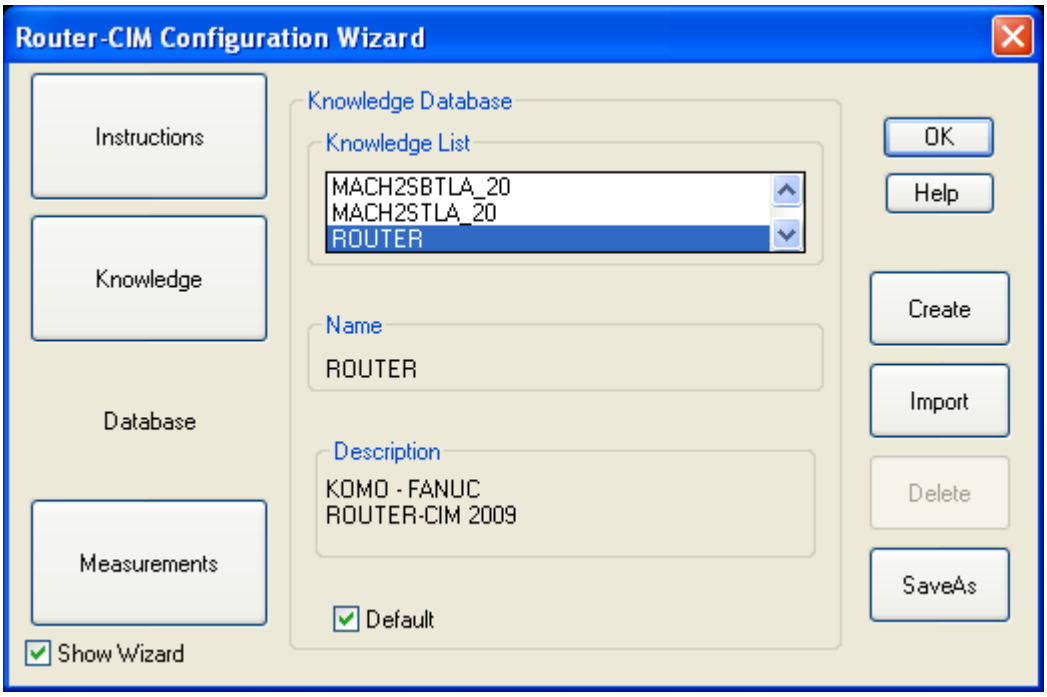
After validating that the correct machine and post processor have been selected, click OK to use Router-CIM.



© 2011 CIM-TECH.COM, Inc

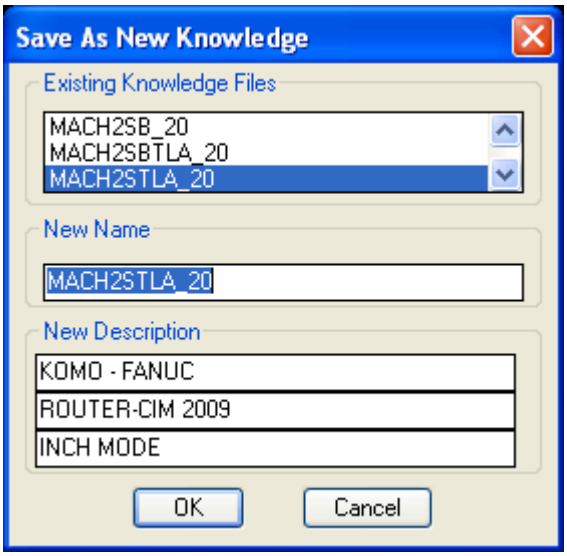
2.8 Customizing the Knowledge Base

As you use Router-CIM you may find it desirable to change the system defaults of your knowledge base. To do so, make the desired changes to settings found in the Knowledge Editor, Tool information, Cycle information, or Status information. Use the Update Current Conditions command under NC Commands on the Router-CIM menu after each Cycle or Tool edit. This makes these changes available to be permanently saved by the Configuration Wizard.
When you have completed all of your edits, click on the Router-CIM pull-down menu and select Configuration Wizard to view your defaults.
If you want to save these settings to a named knowledge base, use the Database button.



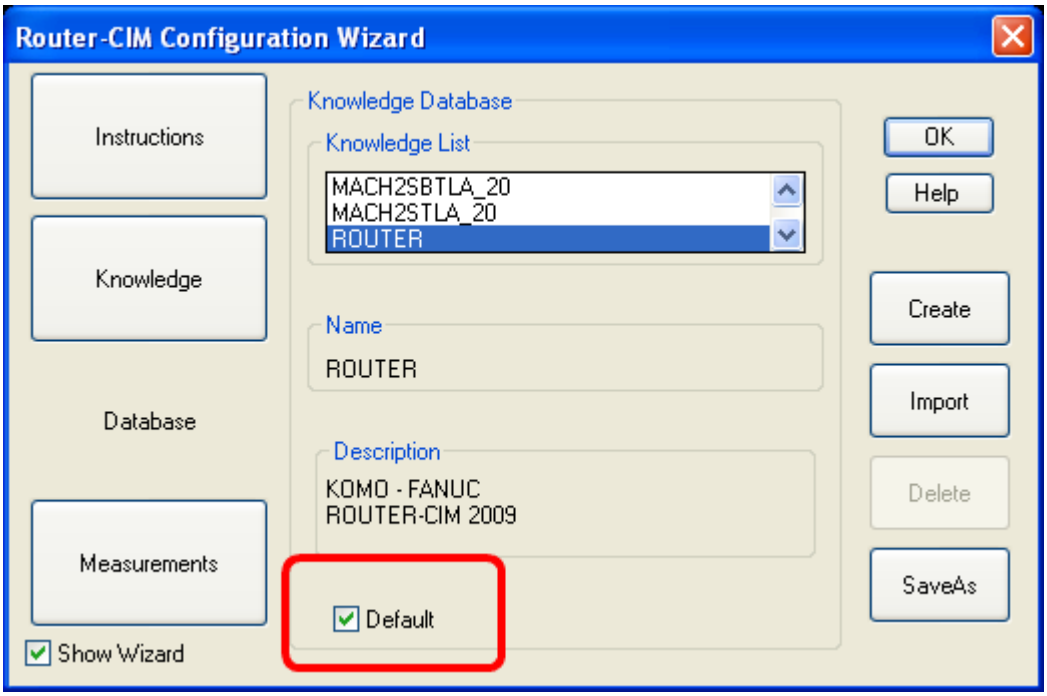
Click on SaveAs. When the dialog window appears, type a new Name and a new Description as appropriate.

© 2011 CIM-TECH.COM, Inc



Select OK when you are finished editing the Name and New Description.

If you would like this to become the default every time you launch Router-CIM, click the Default check box.



© 2011 CIM-TECH.COM, Inc

3 Geoshape, Group, and Start Point Edit

Geoshape, Group, and Start Point Edit

Geoshape

The Geoshape command is used to convert your drawing elements into usable 2D Polylines that Router-CIM uses to create tool paths. When you pick the Geoshape command, you will be prompted to select objects. Select items that represent the geometry that you wish to use to create a tool path or several tool paths from. Do not select objects that are not related to the tool path. You can use a window or crossing window, or a single pick selection. If using DO-IT, Pattern Recognition, Cutting a Solid, or a Surface; do not use Geoshape. The DO-IT command automatically defines shapes before Cutting.

The Geoshape command runs a task called Geoshape. The result of the Geoshape command depends on the geometry selected. The primary operation of this command is to make closed, counter-clockwise Polylines on layer NC_SHAPE from the selected geometry. Geoshape is the Router-CIM command that will changes lines, arcs and other geometry in Polylines. This is also the command that Router-CIM uses to blend or heal open or crossing geometry.

Group

Group is a command that lets you take several Geoshaped elements and place them into a single set or group. This way one cut command can be used on several shapes that are cut in the same way. Such as drilling many holes that are the same size. Grouping will work on holes, and on sperate pieces of geometry. There is no limit to the number of items in a group, but note that once the items are in a group, all of them are selected at once for any command given, like CUT.

Start Point Edit

Start Point Edit is a command used to move or change the location where the tool starts it cut in a shape. Some start points are selected by default when the Geoshape command is run. These locations can be changed to suit your particular cutting needs.

3.1 Geoshape

Router-CIM Toolbar: 

Keyboard: **GS**

Router-CIM must have geometry to make toolpaths from. Valid types of geometry are Lines, Arcs, Circles, Polylines, 3D Polylines, Splines, Ellipses, and Solids.
Blocks should be broken down into usable geometry. There are many instances where the Geoshape command will do this for you, but nested blocks can cause the Geoshape command to leave some geometry in a block form and that is not acceptable for cutting.

The first operation that the Geoshape command executes is to erase any existing objects on layer NC_GEO. This step is important because the objects on layer NC_GEO are used to make polylines on layer NC_SHAPE. These existing objects (if any) on NC_GEO are erased so that you don't get two or more NC_GEO objects on top of each other.

© 2011 CIM-TECH.COM, Inc

After the NC_GEO layer is cleared, Router-CIM issues the Geometry command. Geometry copies what you have selected onto layer NC_GEO. If you select Polylines or blocks, they will first be copied onto layer NC_GEO then exploded into their smallest element.

The second part of the Geoshape command is running the Shapes command.

The Shapes command now converts the NC_GEO objects into CCW Polylines on layer NC_SHAPE. The Polylines will be closed, if possible. The Geoshape command will fix contingencies or gaps within .02". This fix gap tolerance is adjustable in NCVARS, Geometric, *brng*.

These steps are necessary because only Polylines and surfaces on layer NC_SHAPE are suitable for cutting. Since there is blending, any lines that are not tangent or are open by more than .02" will be considered separate, open elements, and will generate their own start points (see the section on Start Point Edit later in this chapter for more information about start points). Gaps are fixed because the NCVAR called NCJOIN is set to True, and the NCVAR *brng* is set at 0.02 this is the largest gap to fix.

After completion of Start Point Edit, the Geoshape command will delete any geometry on layer NC_GEO. The Geoshape command will place NC_SHAPE Polylines over the original objects. The original geometry produced is not altered in any way by Router-CIM.

Since the qualities of your tool paths are determined by your drawing, only clean, continuous objects should be selected. Proper geometry definition is important to making accurate tool paths. If you do not like the shapes created, delete geometry on layer NC_SHAPE, fix the geometry on your original layer, and try again.

You can create your own Polylines on layer NC_SHAPE and use them for making tool paths as well. Certain rules apply if you make your own Polylines:

- Any polyline that you want to machine must be on layer nc_shape.
- If it is a closed shape it must go in the CCW direction.

Therefore, if you create a 2D polyline that goes in the CCW direction, you may place that polyline on layer NC_SHAPE (using either the AutoCAD Change or Chprop commands) and then Cut it.

3.2 Start Point Edit (Start Pt)

Router-CIM Toolbar: 

Keyboard: **SP**

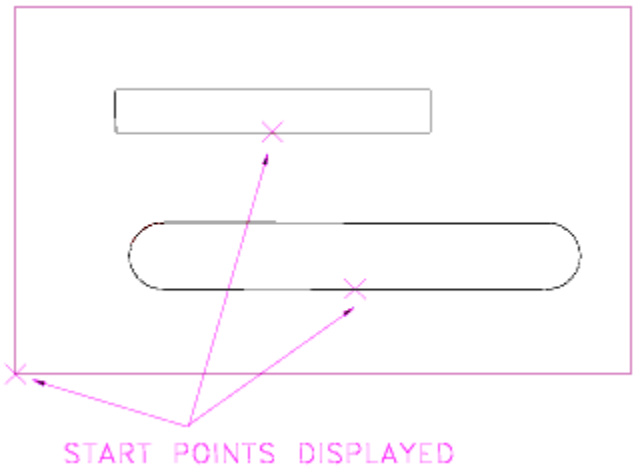
This command allows you to set or change the locations for the STARTING POINT of the cutting cycle. This is the point where the tool leads-in or leads-out of the material.
You will be prompted to select the items you wish to change the start points on, at this point the start points appear as magenta "X's on the part.

Multiple selection is allowed, so if there are several shapes you wish to change the start points on, select them all, and Router-CIM will highlight them one by one for you to select the start point positions.

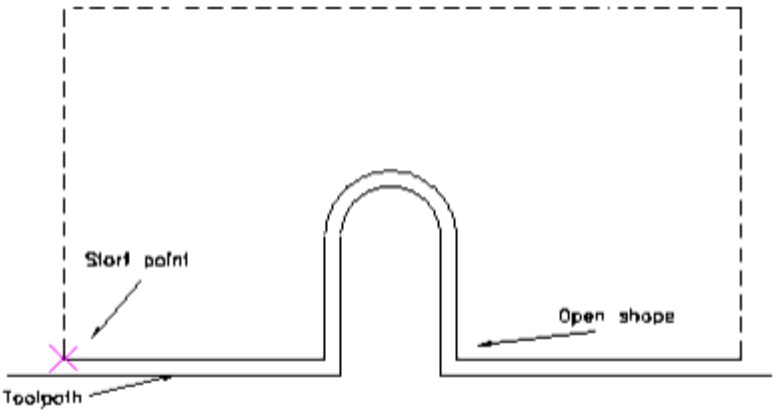
If there are no start points you wish to change, then press <Enter> two times with no selection and you will exit the Start Point Edit command.

© 2011 CIM-TECH.COM, Inc

If you make a mistake picking a start point you can re-enter the Start Point Edit command by picking the StartPT button from the Control Panel, and selecting the geometry you wish to change start points on.



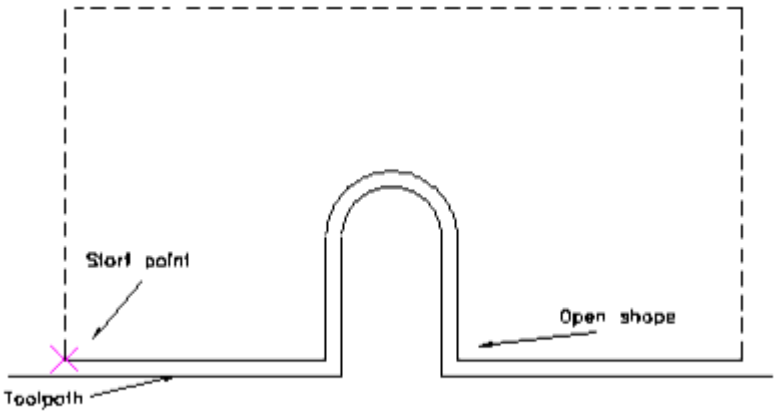
Start Point Edit will reverse the direction of an open shape (see figure 2). You can change the cycle's Cut Side and/or Cut Direction in the Control Panel to make a correct Cut on an open shape.



3.3 Open Shape Issues

The location of the start point indicates the break in the open shape. Drawings that have lines on top of other lines will create open shapes. If you were expecting a closed shape, but Start Point Edit does not let you change the start point, you have an open shape. Every time you pick your shape with Start Point Edit, it shows you the problem areas. The X indicates where the shape is not continuous, or has a gap that is larger than the gap tolerance (Geometric NCVAR brng). You can erase the open shapes (on layer nc_shape); fix the original geometry on the original layer, then Geoshape the objects again.

© 2011 CIM-TECH.COM, Inc



3.4 RCIM Group Command



Above is the Group toolbar used for shapes or holes which have been previously Geoshaped. The icons, from left to right are: Group Shapes, Group Holes and Edit Group. The end result will block the shapes or holes together so only one item needs to be selected in the future. This is particularly helpful with large groups of holes or any combination of shapes you wish to cut using the same knowledge.

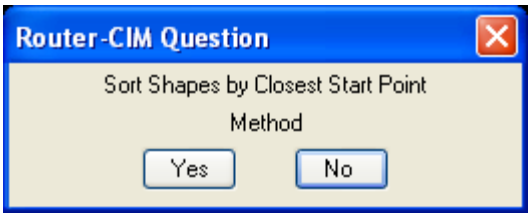
Group Shapes

The first icon in the Group toolbar is for Grouping Shapes. Following are the steps involved to group different shapes together so they may be cut with the same knowledge.

1. Geoshape the parts or pieces of geometry first.

2. Click on the Group Shapes icon.
At the command line you will be prompted:
SELECT Shapes to Group Together
Select Objects

3. Window around all the object you want in a group or pick them in the order you wish them to be cut. Left Click then Enter. You will be prompted with:



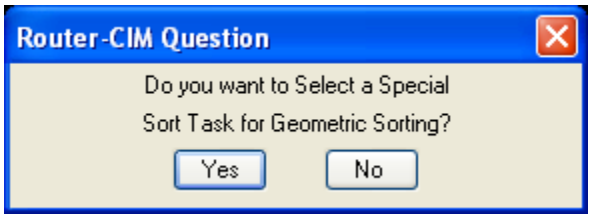
The shapes will turn Red. Choose Yes. You may choose No if you have only a few shapes which you have already chosen in the order you want them cut.

4. At the command line:
Choose First Shape in Packet

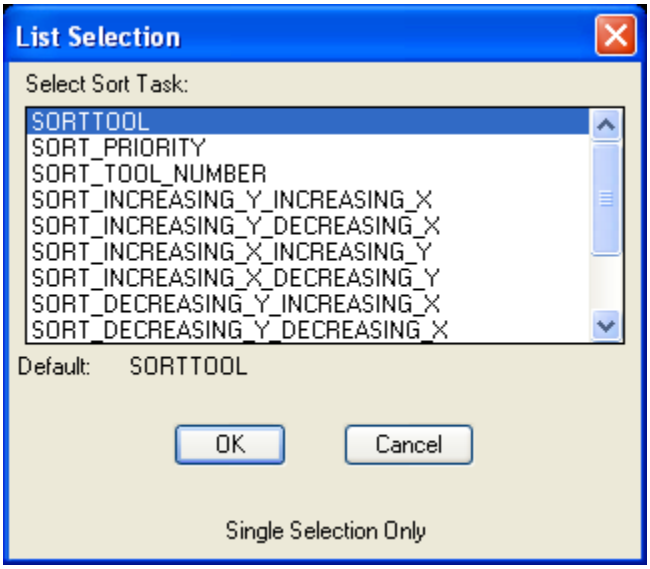
© 2011 CIM-TECH.COM, Inc

Select Objects

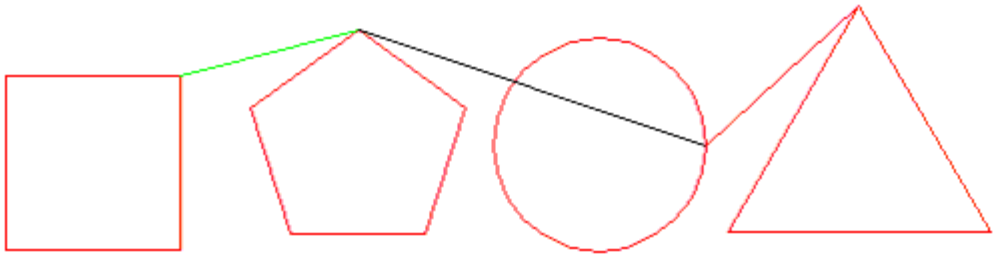
Pick on which one you want to be first in the group. Then Enter. You will be prompted with:



Choosing No will sort the shapes by Closest Start Point method. Choosing Yes will bring up a dialog box allowing you to choose different sorting methods:



5. After selecting either No or Choosing Yes followed by a different sorting method, your parts will be grouped together enabling you to cut them with the same knowledge.



In the picture above the Green line coming off the square indicates the square is the first object in the group and the Red line leading to the triangle indicates the triangle is the last object in the group.

Group Holes

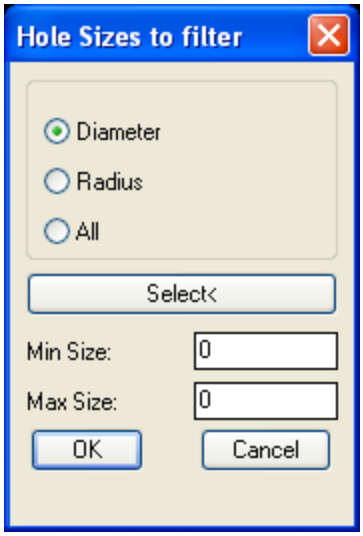
The second icon in the Group toolbar is for Grouping Holes. This option can be very helpful in situations with line boring or any operation involving a large amount of holes.
1. Geoshape the holes first.

2. Click on the Group Holes icon

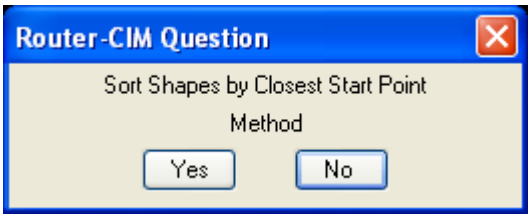
© 2011 CIM-TECH.COM, Inc

At the command line you will be prompted:
Select shapes to search for circles:
Select objects:

3. Window around all the holes in your group or pick them in the order you wish them to be cut. Left Click, then Enter. You will be prompted with the following dialog:



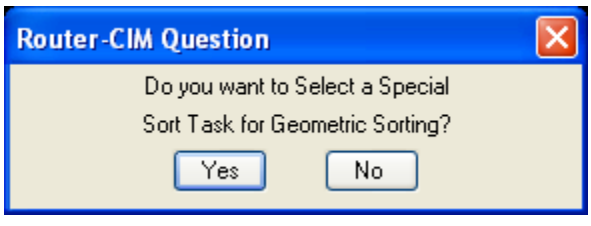
4. At this point you may select the holes in two different ways. If you know all of your holes are the same size and you want them in one group you may change the hole parameters from Diameter or Radius to All and then click on OK and you will be prompted with:



5. The holes will turn Red. Choose Yes. You may choose No if you have only a few holes which you have already chosen in the order you want them cut.

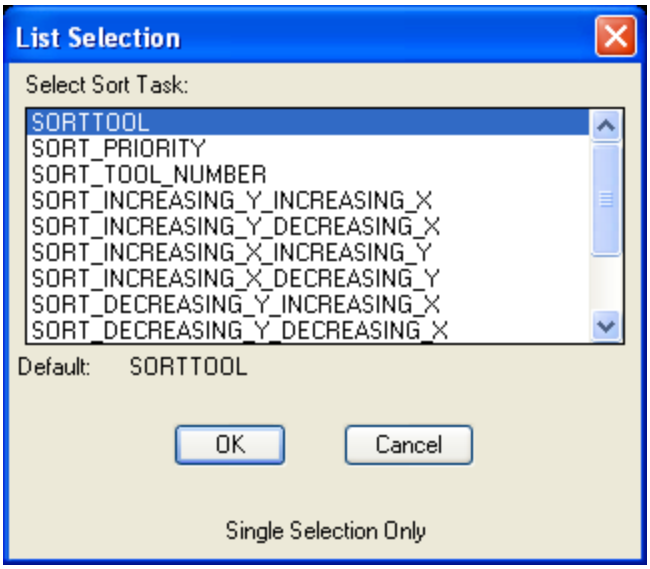
6. At the command line:
Choose First Shape in Packet
Select Objects

Pick on which one you want to be first in the group. Then Enter. You will be prompted with:

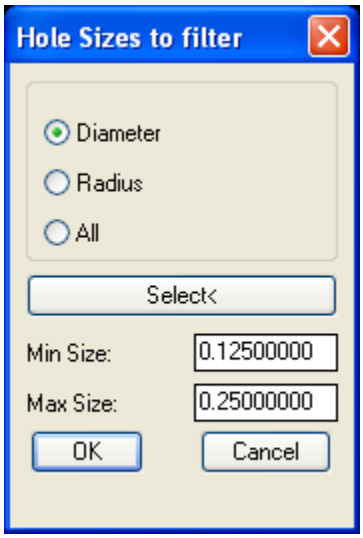


Choosing No will sort the shapes by Closest Start Point method. Choosing Yes will bring up a dialog box allowing you to choose different sorting methods:

© 2011 CIM-TECH.COM, Inc

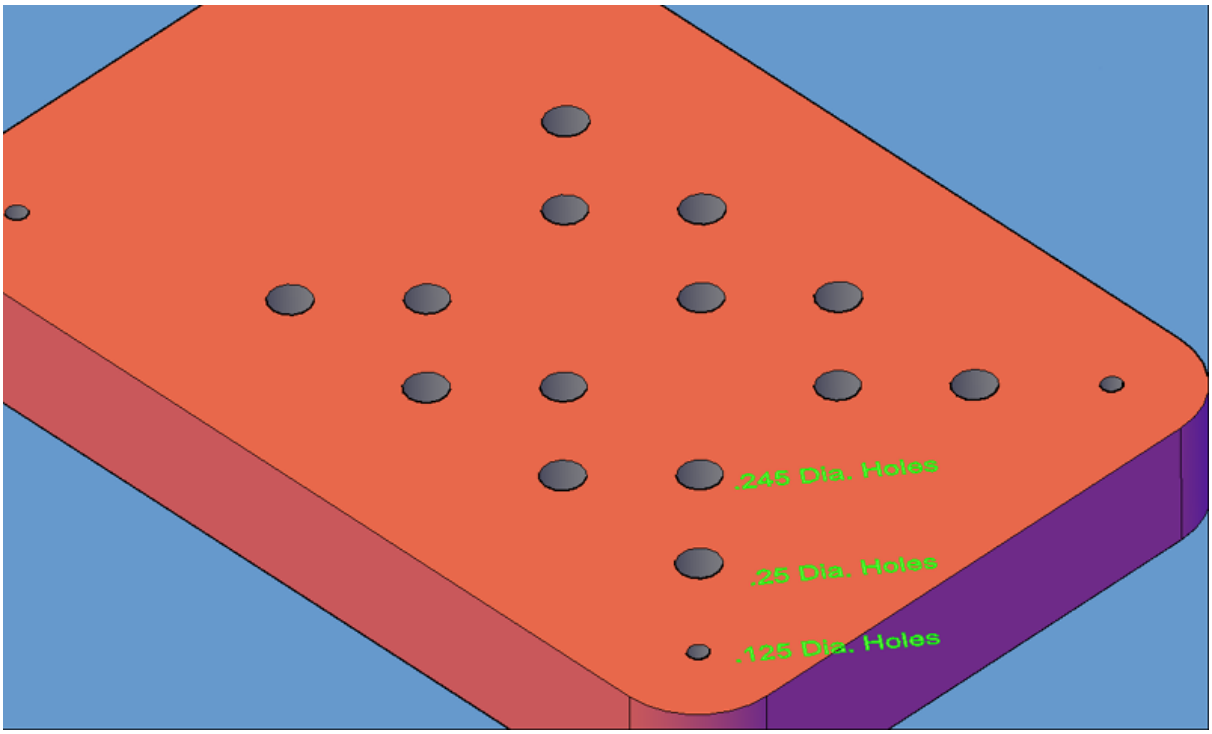


7. If you have a group of holes which are close to the same size, but differ by a small amount and you wish to have them all grouped together, you may do this by using the Min Size Max Size setting to search for holes within a specified range.

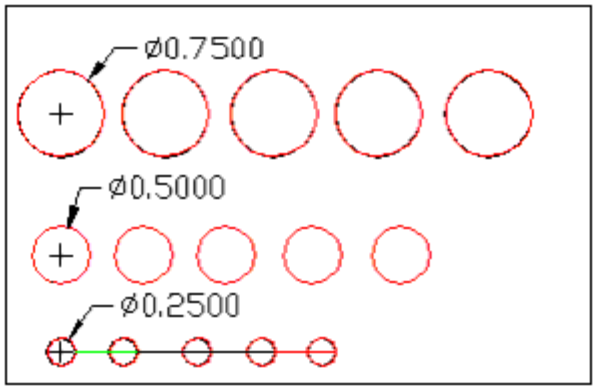


If you had a few holes which happened to be 0.125 in diameter in addition to the ones which are 0.245 and 0.250 and you wanted to center drill all of them at once, you could enter these values into the Min Size and Max Size fields and Router-CIM will search for them and include them in the group.

© 2011 CIM-TECH.COM, Inc



Note: If you have multiple groups of holes in a part which differ in size (i.e. .25, .50, .75 etc.) you can create each group individually based on size if that is preferred.

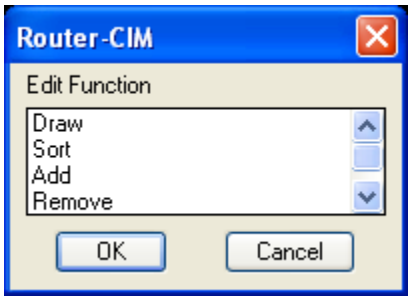


Edit Group

This is the third icon on the Group Shapes tool bar and allows you to Edit the features of a previously created group. This function allows you to Add, Remove, or Sort objects of an existing group.
After you have determined which group it is you want to edit. Click on the Edit Group icon. You will be prompted with:
Select Shape Pack to Edit:

Pick on one of the objects which is part of the group.
If you try to pick on the line connecting the pieces, the group will not be recognized.
After you select one you will be prompted with the following dialog box:

© 2011 CIM-TECH.COM, Inc



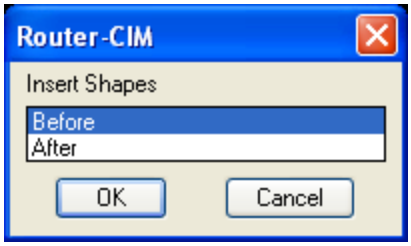
Draw
The Draw function will re-draw the group sorting line on the selected group. The sorting line shows which shape is first (green) which shapes are next (white) and which shape is last (red).

Sort
Allows you to pick a different sorting method for the group. By default, the objects will be sorted by the Closest Start Point method.

Add
If you have added additional objects to the drawing you are working on and wish to add them to a current Group, you may do so with this feature. Click on Add, then OK and you will be prompted with
Select Shapes to ADD
Select Objects

Choose the new shape(s) and Enter.

Select Position Shape
The Position shape is the object in the current Group you want to use as a reference for adding the new one. Once selected you will be prompted with a new dialog:



Choose the positioning you want and the object will be added to the existing group.

Remove
If you wish to remove objects from a Group in the current drawing you are working on, you may do so with this feature. Click on Remove, then OK and you will be prompted with
Select Shapes to Remove
Select Objects
Click on the one(s) you wish to Remove, then hit Enter. The shape will turn from Red back to Blue (Geoshape color) and will no longer be part of the Group.

3.5 Troubleshooting Geometry

Some geometry could cause issues that are not readily detectible until the part is run on the machine. Other geometry can cause issues that are readily apparent when a tool path is created.

© 2011 CIM-TECH.COM, Inc

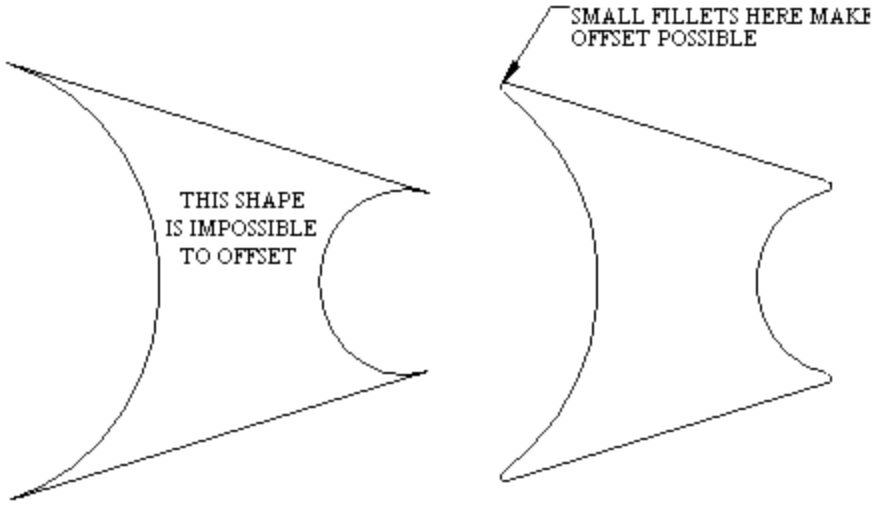
Crossing Geometry:

It is important to note that lines and arcs that cross other lines and arcs could confuse the Shapes command.
When lines and arcs have the same end points, the shape could go in several directions. Keep all geometry to be Geoshaped clean and continuous without ambiguous end points.

The easiest way to see incorrectly shaped geometry is with the Start Point Edit command. If you have a shape that is supposed to be closed and Router-CIM is finding more than one Start Point, or if the Start Point will not change, then the geometry is not closed and should be repaired before going any further. The location of the X is where a problem exists in the geometry.

Impossible Offsets:

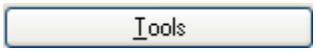
Certain geometry is impossible to offset. Under certain circumstances, you may have to insert a Fillet or Chamfer to allow offset. These situations occur when arcs and lines cannot connect when offset. The Fillet or Chamfer will usually solve this problem. Use the smallest acceptable radius or distance.



© 2011 CIM-TECH.COM, Inc

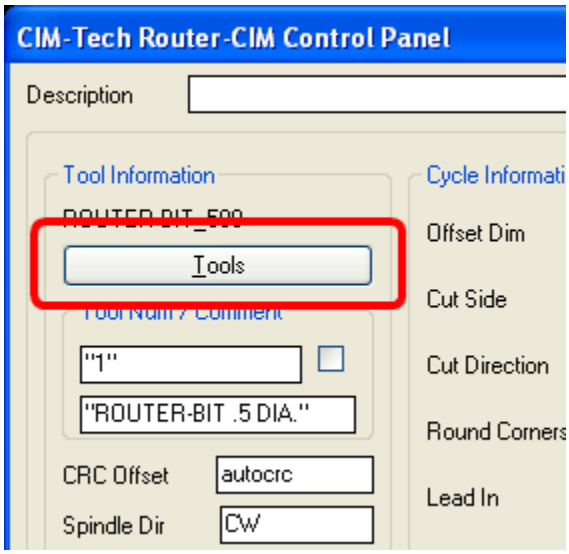
4 Tool Information

Tool Information

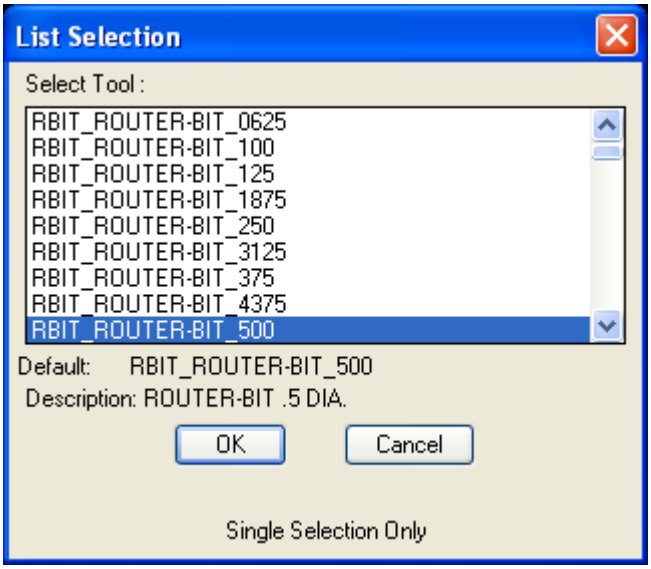


Following is a brief explanation of Tool Information, the first column in the Control Panel/All Stats page.

Select a tool from the Rbit Tool List by picking on the Tools Button in the Control Panel.

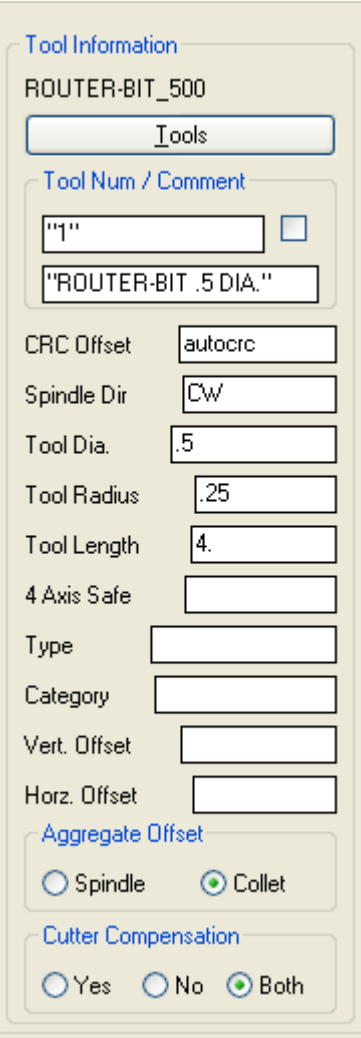


A dialog box showing the tool names found in the Rbit list will appear. Move the cursor over the tool to use and pick the tool you wish to Cut with.



The system will use the current attributes of the tool you select.

© 2011 CIM-TECH.COM, Inc



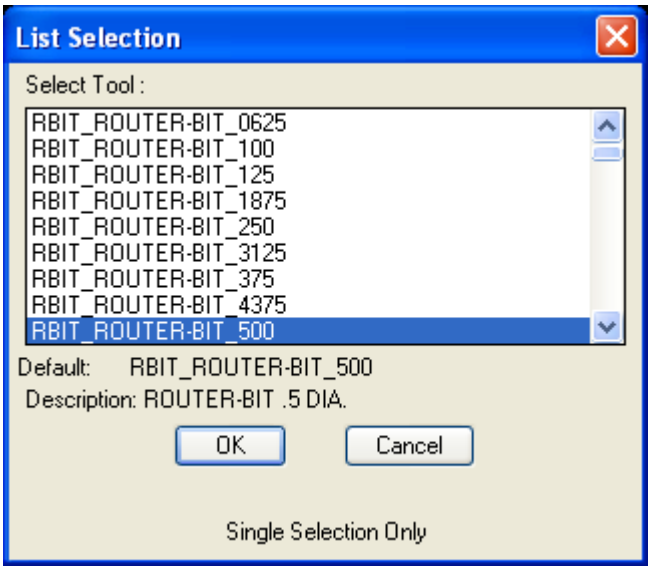
The standard Tool list contains most of the common size router bits, drill bits, ball mills, as well as some generic shaping tools (e.g. round over bits).

These tools are listed in English Units at the top of the list and in Metric Units at the bottom of the list.

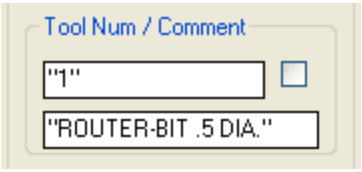
After selecting a tool, the Control Panel will reflect all the current information for that selected tool in the Tool Information column.

This list of tools is may be customized by the user, and can contain all the tools you will use on a regular basis. You may add tools to any position in the list, however setting or keeping a specific order to the list is usually helpful in finding and/or changing tools in the future.

© 2011 CIM-TECH.COM, Inc



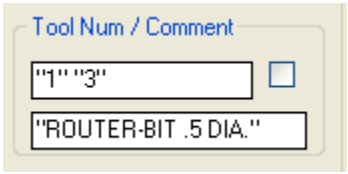
4.1 Tool Number and Comment



The top field is the Tool Number field, and indicates any tool or spindle numbers that are to be used for the current cut.
The bottom field is the Tool Comment field, and indicates any comment you want placed in the code at the start of the cut.

Tool Number

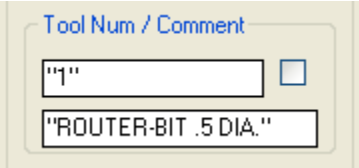
The main tool or spindle number, and any slave spindles to use for the current cut are entered into this space as strings (in quotation marks).
Enter the main spindle as the first tool number in quotes, enter each slave spindle in quotes following the main spindle, separate with a space.
Example1: Tool Num/Comment: "1" "3" uses Main spindle #1 and slaves spindle #3.



Example 1

Example2: Tool Num/Comment: "1" uses Main spindle #1 and no slaves.

© 2011 CIM-TECH.COM, Inc



Example 2

Your router may use a different numbering scheme, but the tool number will be entered in the same way and in the same place. The correct tool numbers for your machine will be shown in the Application Notes for your post processor.

Comment

The tool comment parameter is simply a comment field that will generate a comment in the NC Code file prior to the tool change to the designated tool.

- You can place any syntax in this field you like up to 80 characters.
- Some machine tool controllers (like Fanuc) cannot represent lower case characters. So you should use capital letters in this parameter.
- The entire string placed in this parameter must be contained in quotation marks (" ").

4.2 CRC Offset



This location generates a D value for using Cutter Compensation (G42) or (G41). The task called Autocrc will automatically use the specified tool number for the CRC Offset number only when the tool attribute Cutter Compensation is set to Yes or Both.

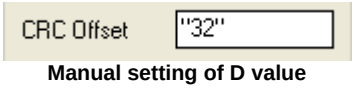
If Cutter Compensation is set to No this field is ignored.

When the CRC Offset field is set to autocrc, then Router-CIM will automatically generate a D number for you as long as the specified tool number is a valid number for the post and is not a drill.



Automatic setting of D value

You can put any number into this field if you don't want to use the Autocrc task (if you don't want the system to automatically generate a D number). The number must be in quotes to be considered valid.



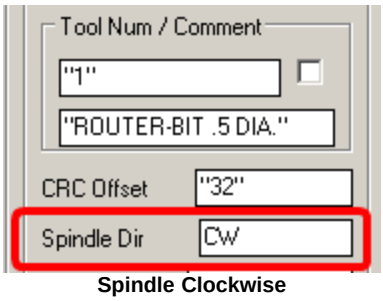
Manual setting of D value

Remember that the Post processor system will automatically assign a D value for each tool as long as autocrc is listed in this location. Check the Application notes for the offset number for each tool. Replacing the number in the autocrc field with any number (in quotes) will set that number for the D value in the cut, and can also be stored in a knowledge so that you do not have to reset the number again.

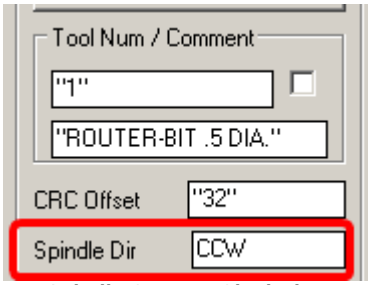
© 2011 CIM-TECH.COM, Inc

4.3 Spindle Direction

This field allows you to change spindle rotation from clockwise (CW) to counter-clockwise (CCW) by placing the correct text in this box. The only valid entries are CW or CCW. Few machines support the CCW option. Check your machine's specifications to see if it can output the code necessary to start the spindle in either CW or CCW. If your machine tool does not accept the CCW command, then you will likely get an error when the nc code file is run.



Spindle Clockwise



Spindle Counter-Clockwise

4.4 Diameter



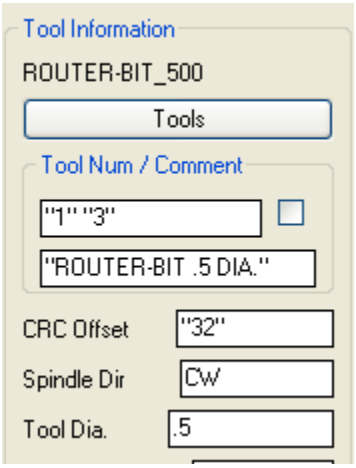
This parameter holds the Diameter of the currently selected tool.

The field can be edited if you wish to temporarily change the value of the currently selected tool. Changing this field does not change the value of the tool in the tool list! It is for temporary editing only.

This value will affect the size of the lead-in and lead-out on most cut cycles, but will not affect the offset of the tool paths.

If you are using the Tool Path Verification software (MCU), this will change the size of the tool reported to MCU and result in a back plot that is inaccurate.

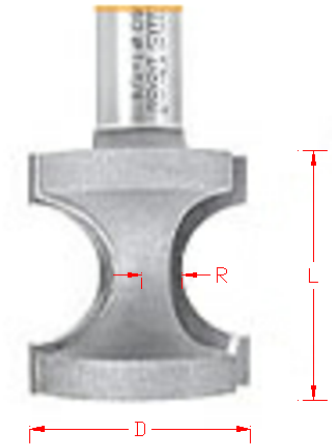
© 2011 CIM-TECH.COM, Inc



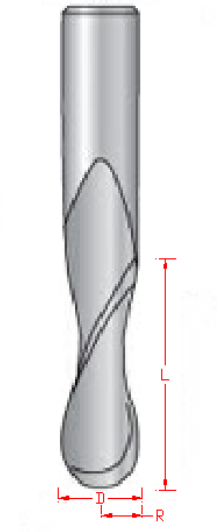
The diameter value is the largest diameter value of the tool if more than 1 option is available, for instance on a shaper type tool. Some common tools and their measurements are shown here.



Cove Bit

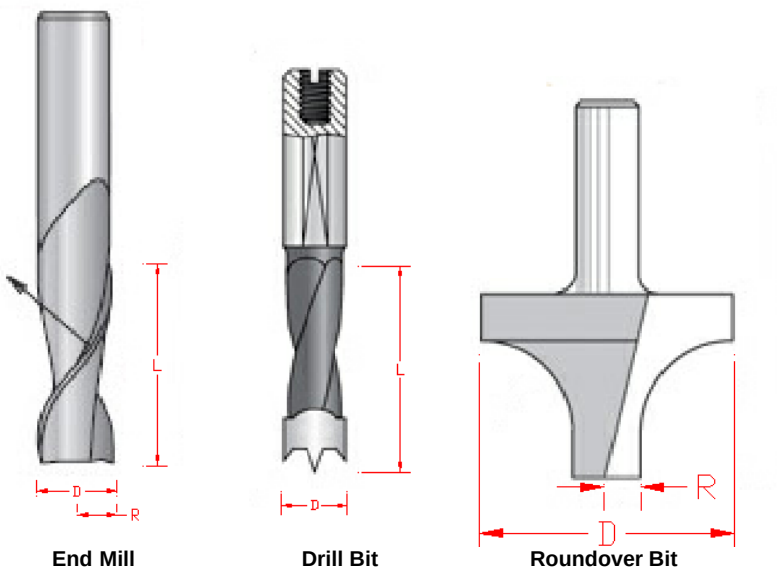


Roundover Bit



Ball Mill

© 2011 CIM-TECH.COM, Inc



End Mill **Drill Bit** **Roundover Bit**

In each of these tools L is the Tool Length, D is the Tool Diameter, and R is the Tool Radius.

4.5 Radius



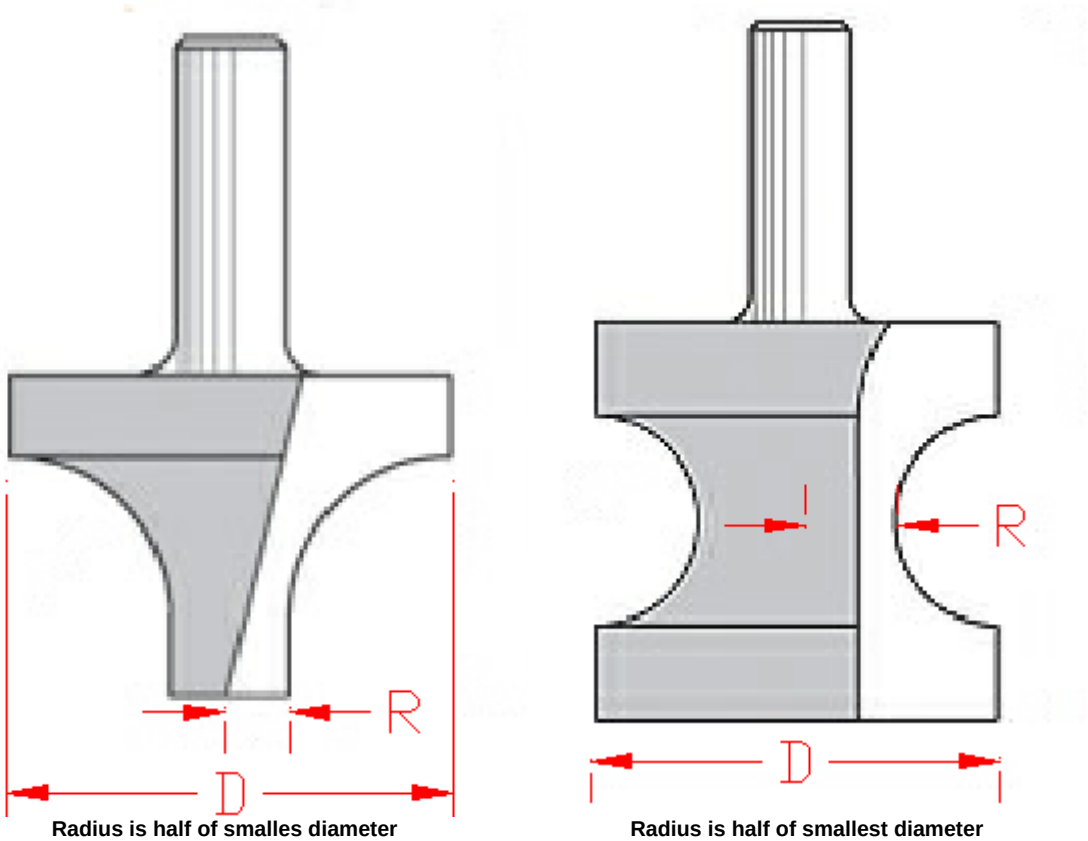
This parameter holds the Radius of the currently selected tool.
The field can be edited if you wish to temporarily change the value of the currently selected tool.
Changing this field does not change the value of the tool in the tool list! It is for temporary editing only.
This value will determine the tool path offset distance from the geometry.

If you are using the Tool Path Verification software (MCU), this will change the tool size as well as changing the offset amount of the toolpath resulting in a completely inaccurate back plot.

You should instead use the Offset Dim in the cycle information to change the amount of the tool offset.

When setting the tool radius, it is important to remember that this is used as the offset for the tool path and if you are using a shaper cutter, the radius may not be half of the diameter.
Look at the following images:

© 2011 CIM-TECH.COM, Inc



Radius is half of smalles diameter **Radius is half of smallest diameter**

In both of these cases you would only want to offset the tool path by half of the smallest diameter. However for lead-in and lead-out clearance, the diameter is the largest diameter so that you account for the entire size of the tool in the tool path. In this case the radius is not half of the diamter.

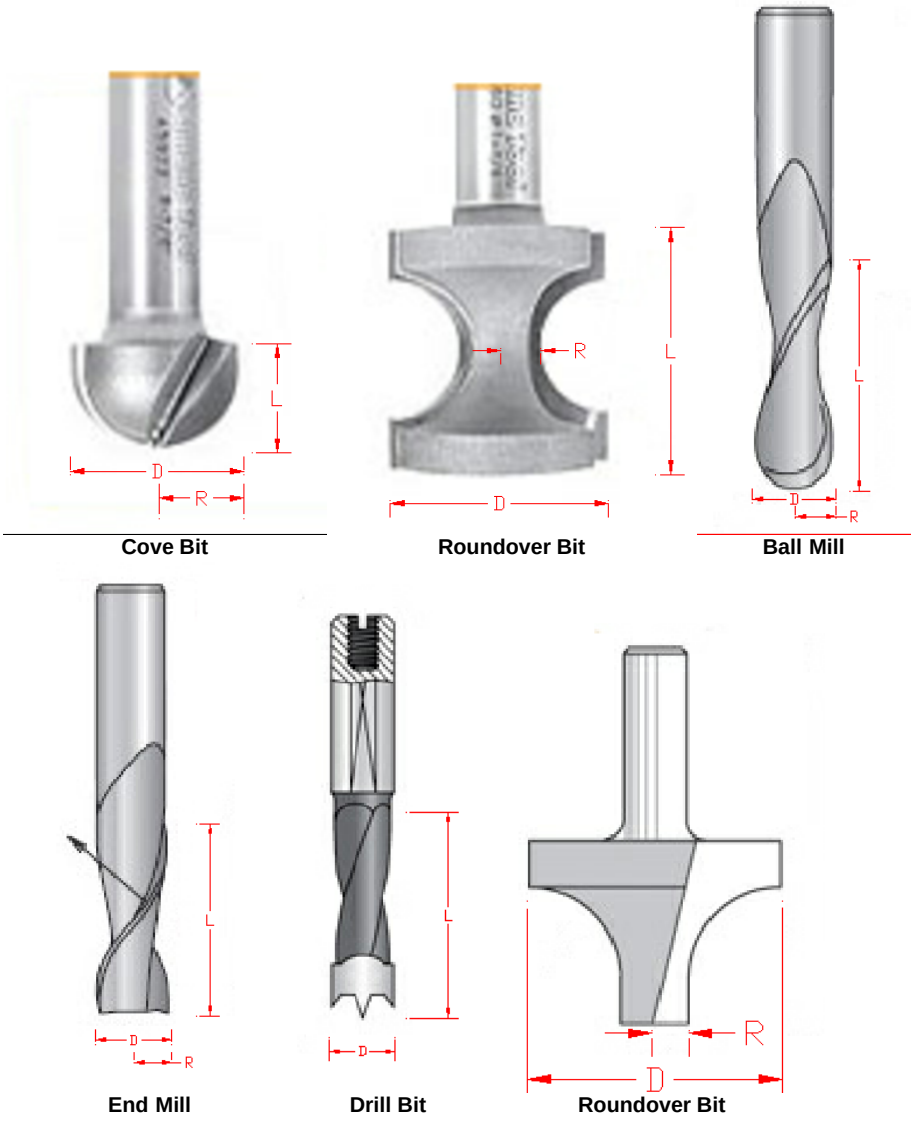
4.6 Tool Length



The Tool Length field is used to set the length of cutting flute on a tool. It is also the maximum depth that Router-CIM will allow this tool to move in Z.
For temporary editing, you may change this value on the Control Panel.

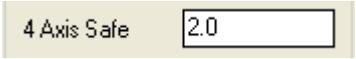
When setting this value, you should pay some attention to how deep each tool can cut, especially if you are going to make a cut at full depth in one pass.
A typical setting of tool length for common tools would be the L setting in the following examples.

© 2011 CIM-TECH.COM, Inc



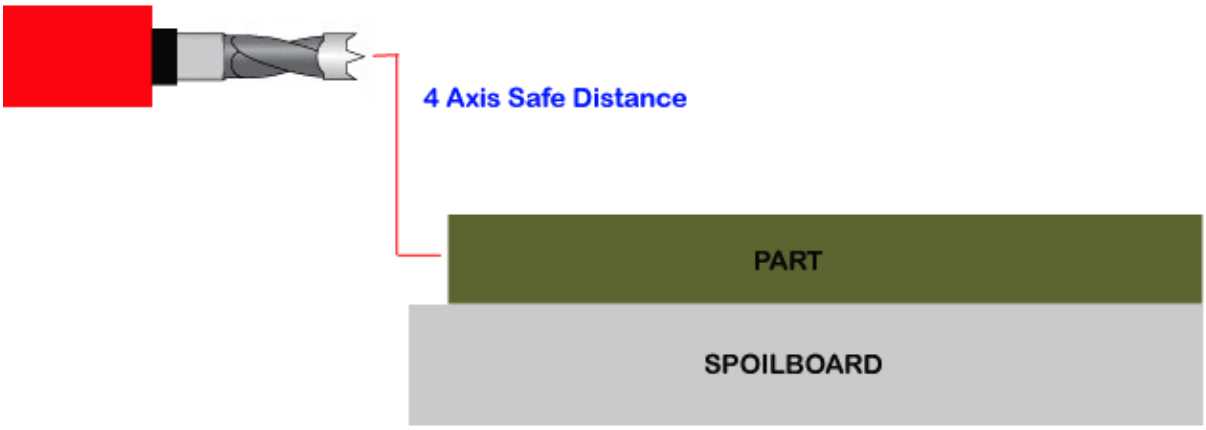
In each of these tools L is the Tool Length, D is the Tool Diameter, and R is the Tool Radius.

4.7 4 Axis Safe



This field allows for a safety retract height for a horizontal tool such as a horizontal boring head. This value will show the height to which the tool will return after each Cut, and also the height it will traverse at between cuts.

© 2011 CIM-TECH.COM, Inc



This parameter is used for horizontal drilling and for horizontal machining only. If no value is input, and one is necessary, then you will be prompted on the screen for the 4 Axis Safe amount at the command line.

Positive values only are expected.

4.8 Tool Type and Tool Category



Type and Category are used for certain types of Point to Point Boring machines. Currently these fields are used by the system, and no user input is needed. Your Router-CIM Application notes will give you further information on these fields if necessary.

4.9 Vertical and Horizontal Offset



The Vertical and Horizontal Offsets are used for certain types of machines. Currently these fields are used by the system, and no user input is needed. Your Router-CIM Application notes will give you further information on these fields if necessary.

4.10 Aggregate Offset



© 2011 CIM-TECH.COM, Inc

This parameter is the offset type of a horizontal tool path and applies only when making Horizontal Cuts. This type of offset is dependent on the settings that are available to the programmer/operator when programming and running the machine. The control of the cut and tools can be solely placed in the program to accomplish all the machine offset moves or it can be placed in the machine, or a combination of both.

Spindle

If Spindle is chosen as the aggregate offset, the cut path will appear on the screen to be offset by the amount designated in the Tool Length parameter.

Collet

If Collet is chosen, the cut path will appear inside the part the distance designated in the Total Depth parameter on the Control Panel. This method requires that you either set the work coordinate to the tip of the tool, or set the distance from the tip of the tool to the center of the spindle in the horizontal tool length offset and use Plane Detect.

The Combination of Spindle and Collet methods

There is a third choice for using a horizontal drill. It is probably the easiest for both the programmer and operator alike. This method requires that you are cutting on one of the 4 faces of the part and not some arbitrary angle, and also for the programmer to know the distance from the center of the drill block to the face of the collet that the tool fits into. It then requires the operator to measure how much tool is sticking out of the collet in the drill block.
The programmer places the distance from the center of the spindle (where the work coordinate is set) to the face of the collet in the Tool Length.
The operator placed the tool length (the amount of the tool sticking out of the collet) into the horizontal length offset.

This method allows for the programmer to only need one number that never changes for a tool length, and the operator can measure the amount from the collet to the tip of the tool (with almost any measuring device) and put it in an offset, just like he/she would for any other tool touched off.

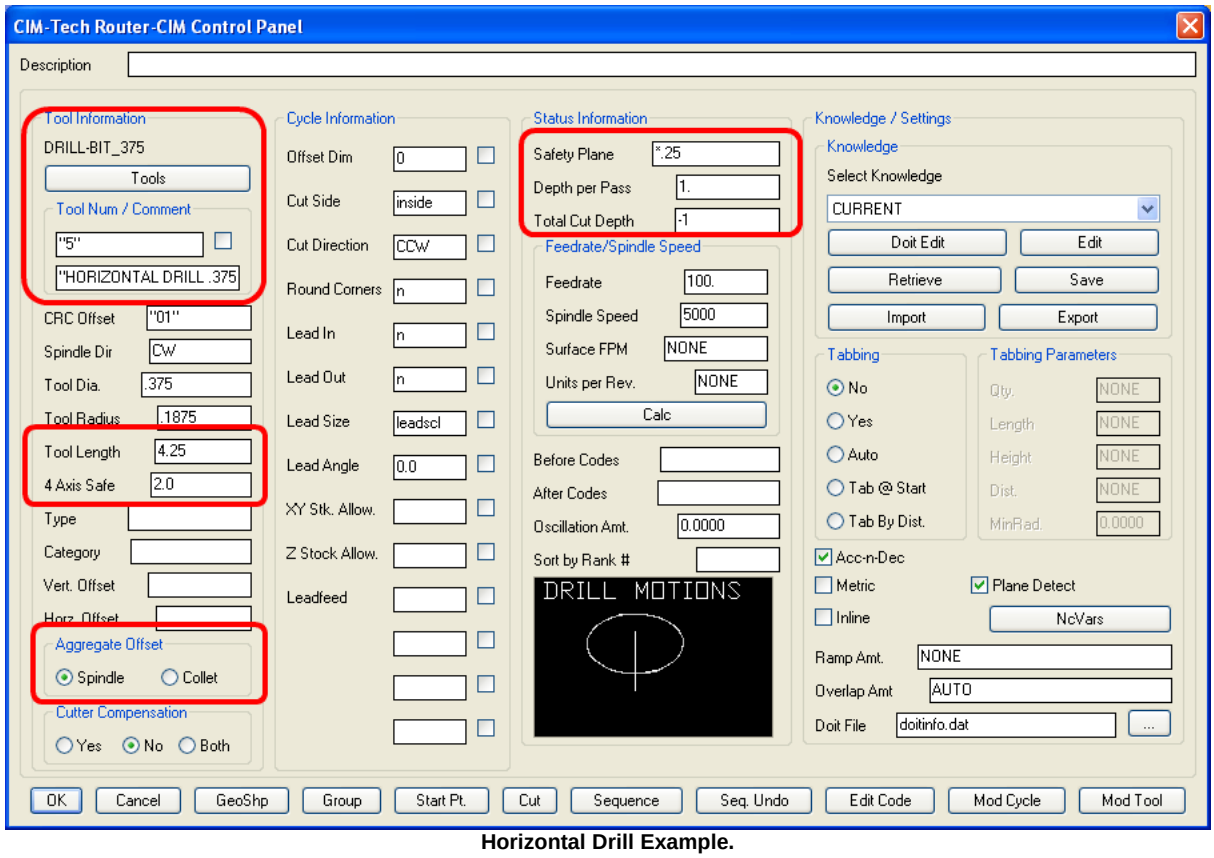
4.10.1 Spindle Offset

Spindle

If Spindle is chosen as the aggregate offset, the cut path will appear on the screen to be offset by the amount designated in the Tool Length parameter.
The Plane Detect is not necessary, but it it is used, then the offsets will be set to 0.
For an example of how this method would work, the following explanation is offered.

Looking at the picture below, the tools is a 3/8" Drill Bit, in a horizontal drill spindle.
The 4th axis Safe Plane is set to 2.0.
The Tool Length is set to 4.25.
The Aggregate Offset is set to Spindle.
The Safety Plane is *.25, and the Cut Depth is -1.0.

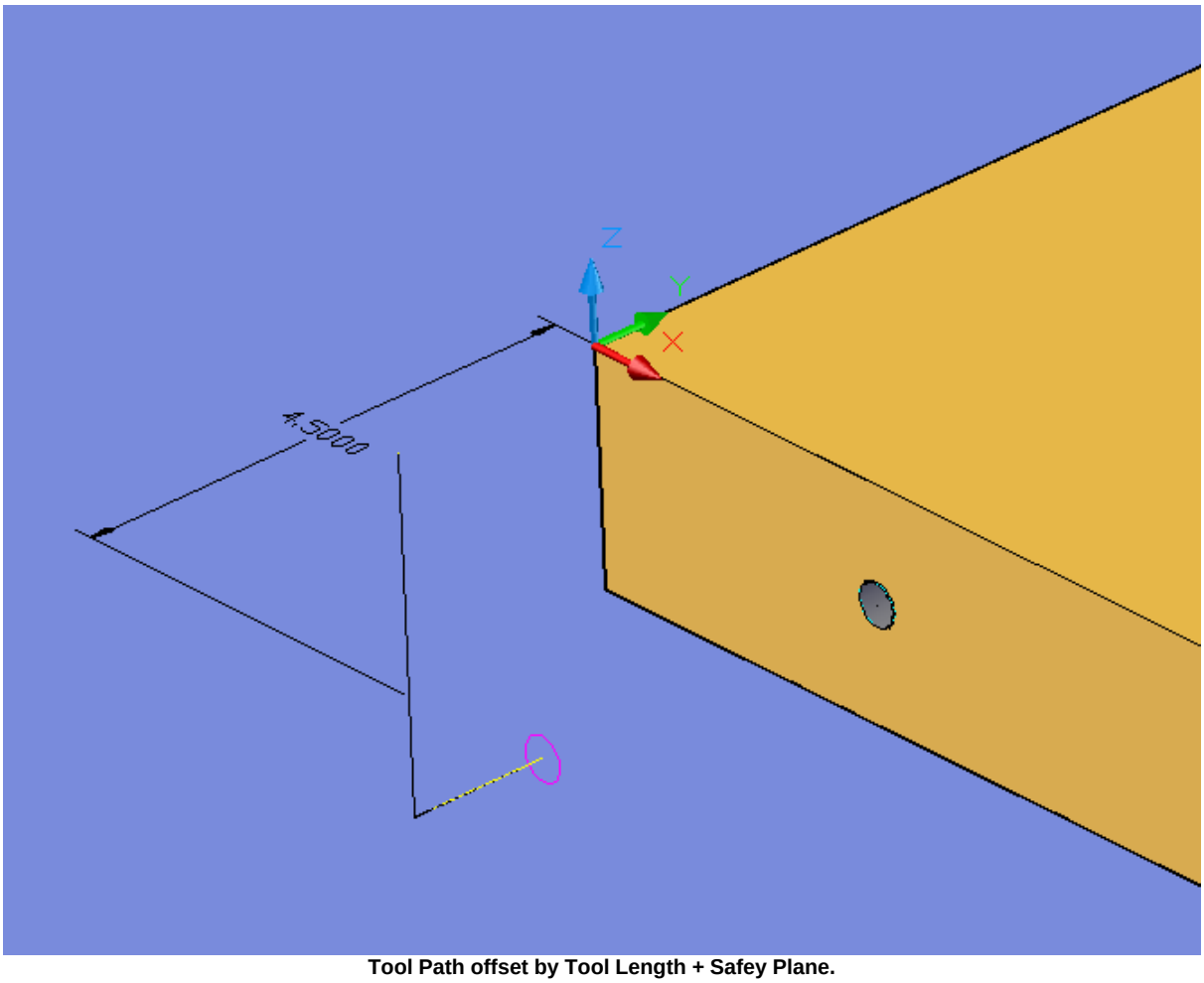
© 2011 CIM-TECH.COM, Inc



Horizontal Drill Example.

The effect of this is that the Tool Path will be offset from the geoshape by 4.5". This dimension is 4.25 for the tool length PLUS .25 for the Safety Plane.

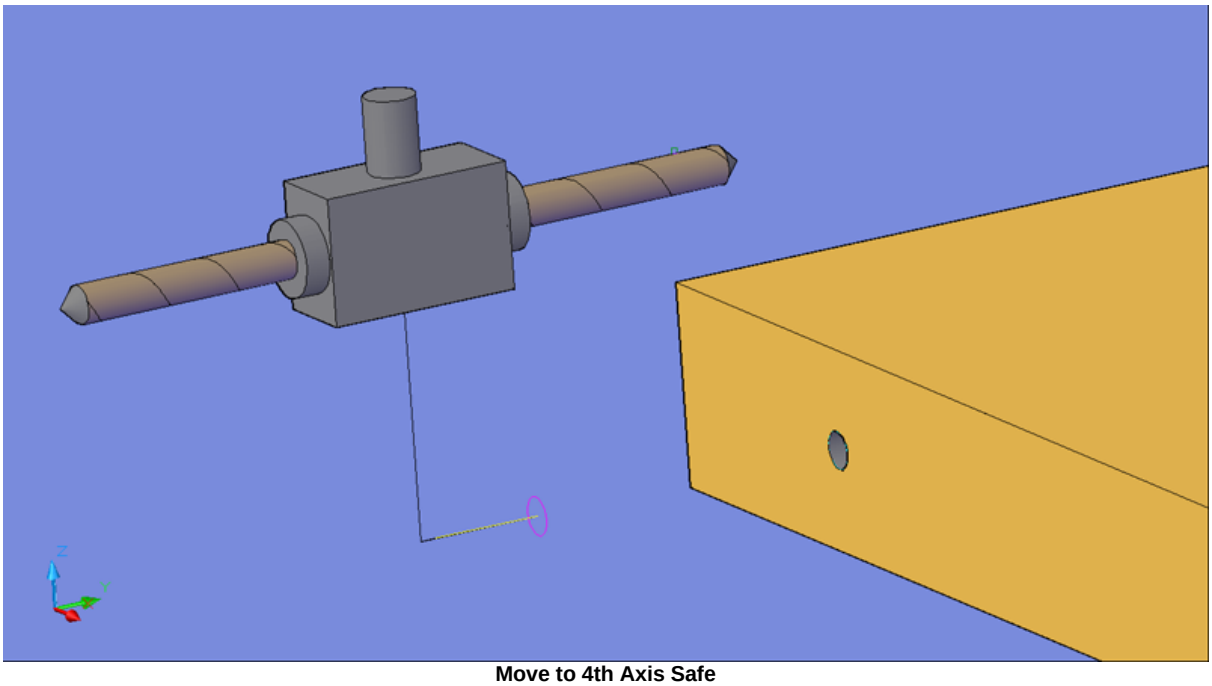
© 2011 CIM-TECH.COM, Inc



Tool Path offset by Tool Length + Safey Plane.

The tool path will represent the center of the drill block in this example. That means that the center of the drill block will move to the end of this tool path which starts at Z2.0 (4th Axis Safe) and the XY position 4.5 inches from the face of the drill hole.

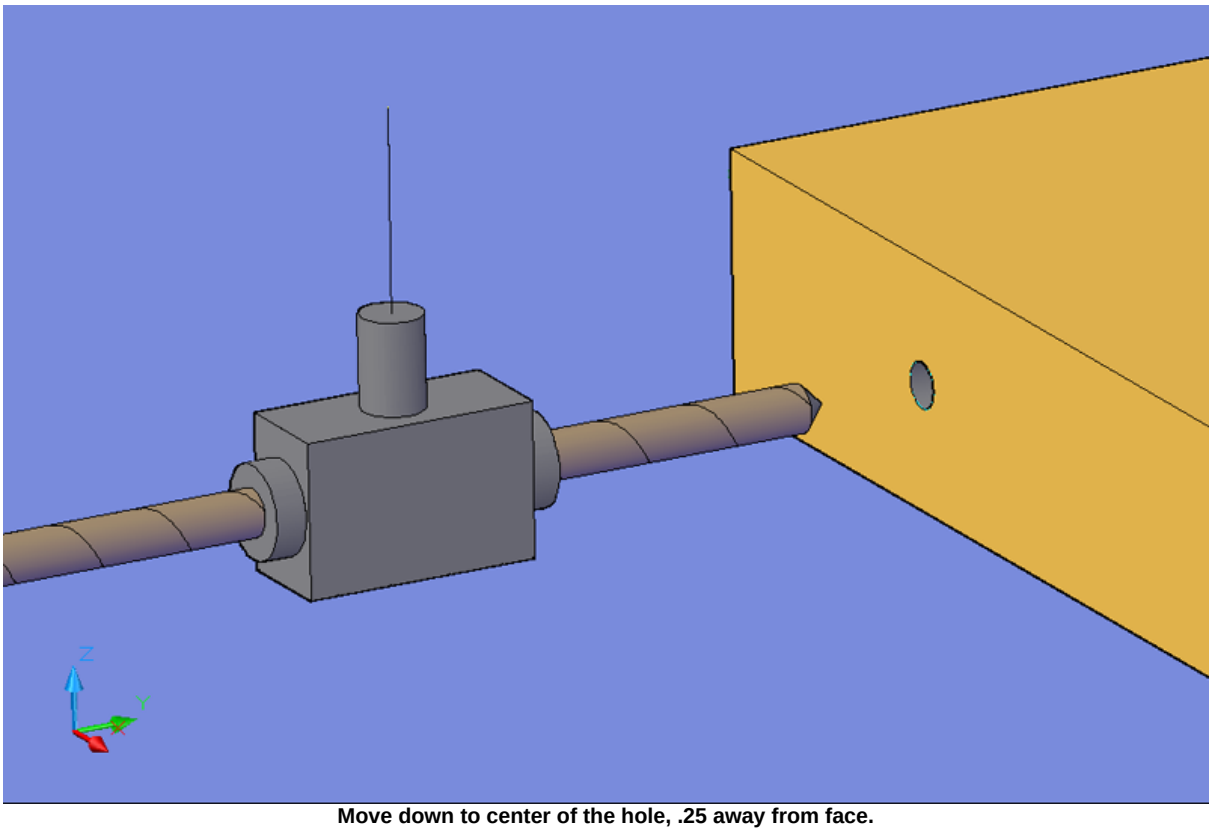
© 2011 CIM-TECH.COM, Inc



Move to 4th Axis Safe

Next the tool will move straight down to the center of the drill hole in Z. It will still be .25 (Safety Plane) away from the center of the hole. This is the small black section of the tool path shown above. The tool will move from that .25 away from the hole, drill all the way to the bottom of the hole (1" into the material) and then move back out to the point 4.5 inches away from the center of the hole, then lift back up to the 4th Axis Safe point (Z2.0).

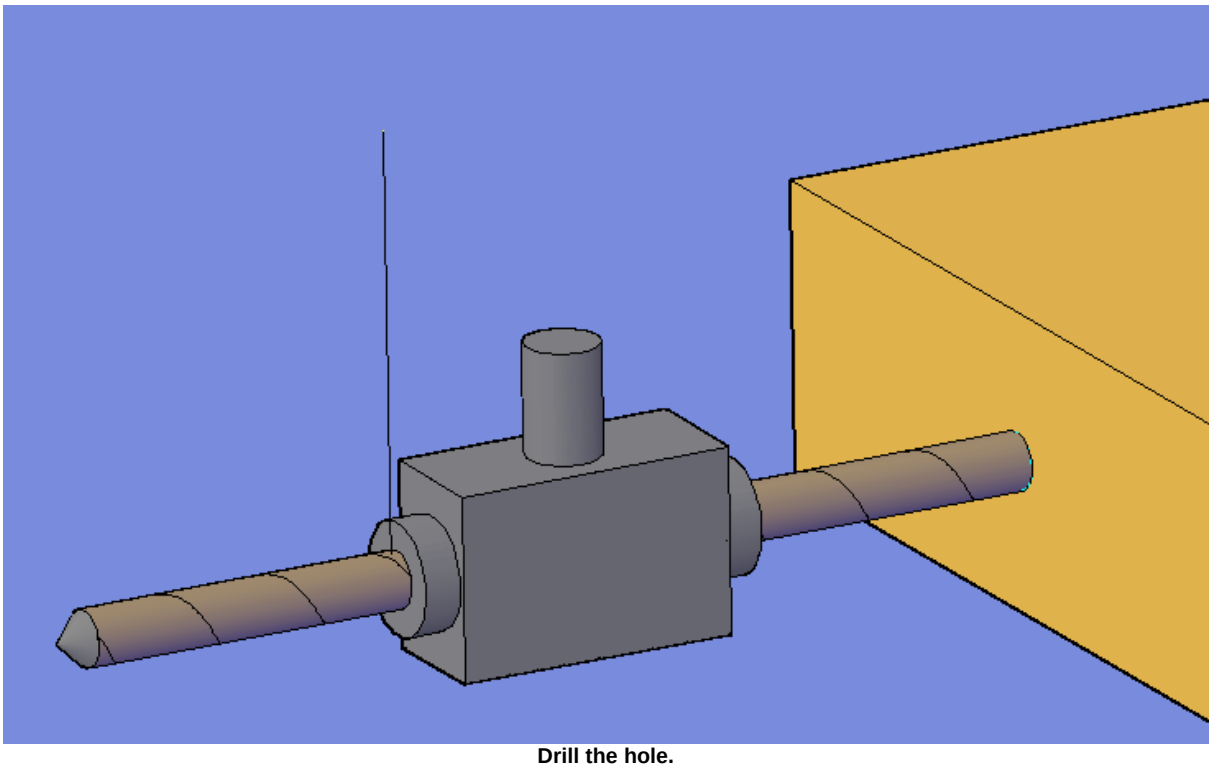
© 2011 CIM-TECH.COM, Inc



Move down to center of the hole, .25 away from face.

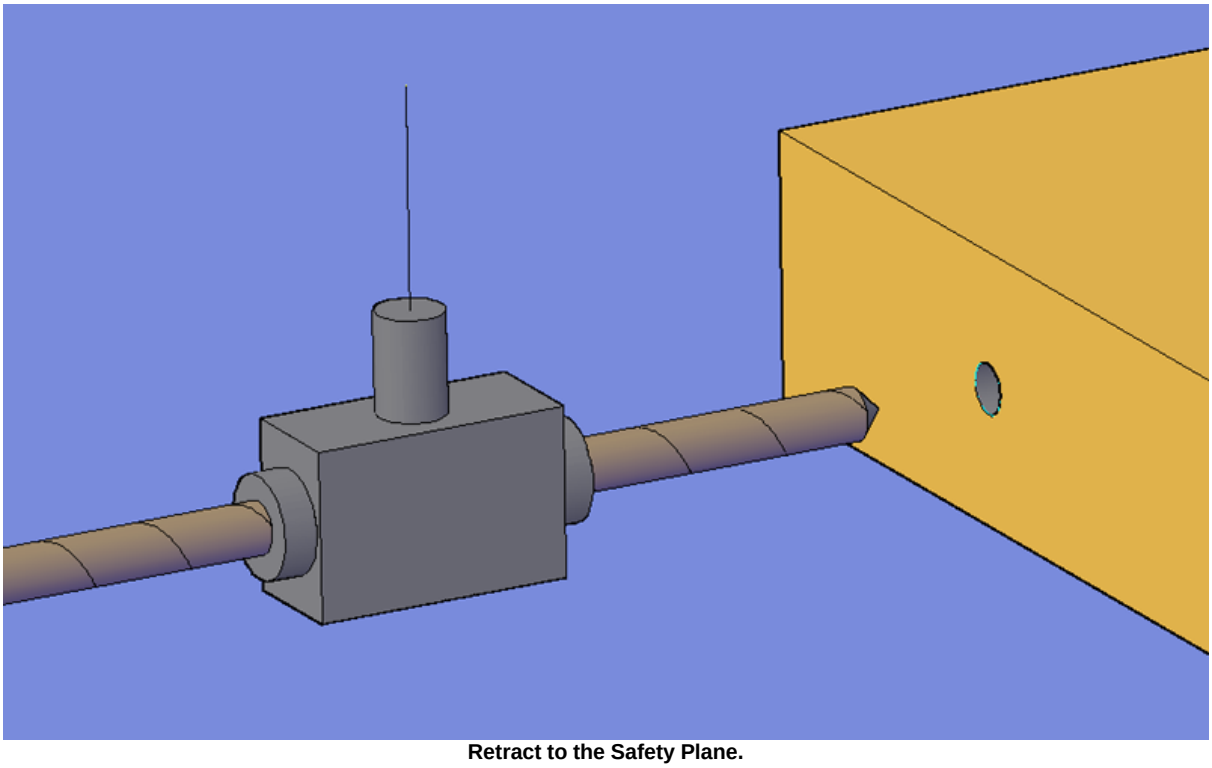
Here is the move down in Z to the center of the hole, still .25" away from the face of the part. This is a Rapid Traverse (G0) move.
Next the hole will be drilled 1.0" deep...

© 2011 CIM-TECH.COM, Inc



Drill the hole.

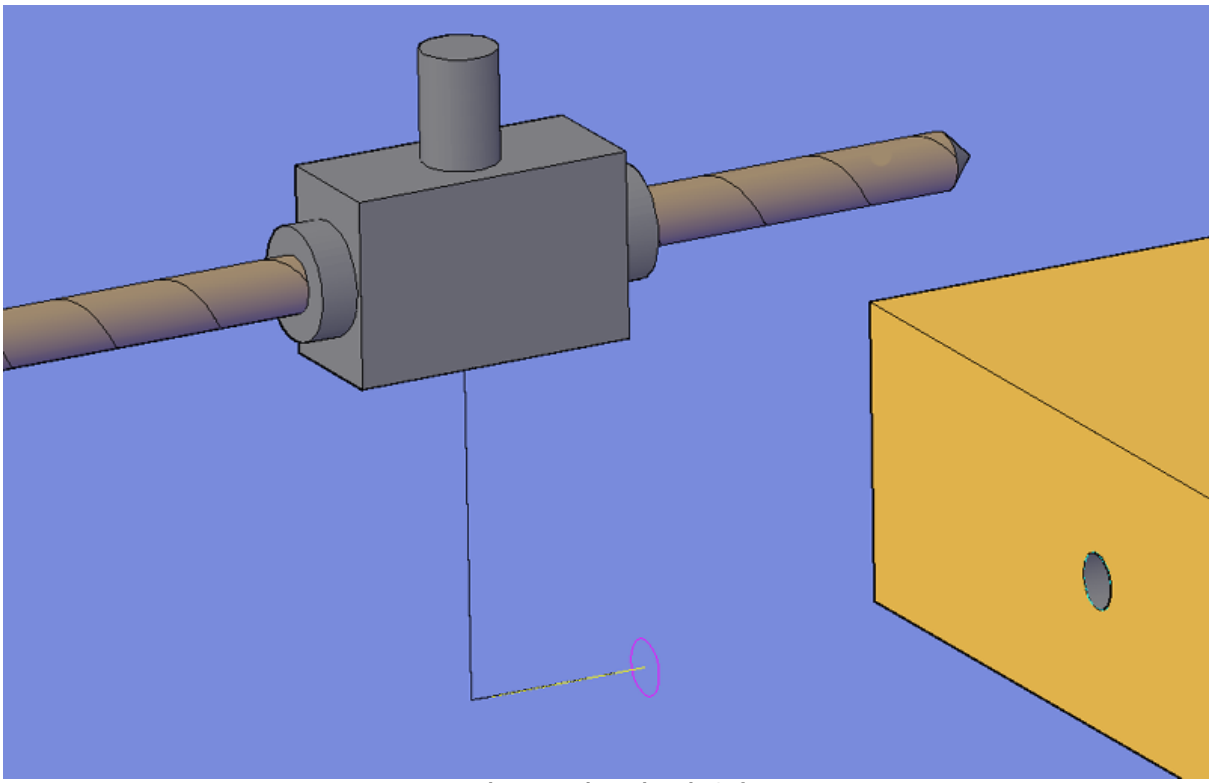
Drill the hole 1.0" deep. This is a feed move (G1) at the programmed feedrate.
Next the tool will retract back to the Safety Plane.



Retract to the Safety Plane.

© 2011 CIM-TECH.COM, Inc

The tool retracts to the Safety Plane. This is a Rapid Traverse (G0) move.
Next the tool will raise back up in Z to the 4th Axis Safe.



Tool retracted to 4th Axis Safe.

And the tool is now retracted to the 4th Axis Safe. This is also a Rapid Traverse (G0) move.

This method only require that the programmer have an idea of the distance from the center of the spindle to the tip of the drill bit. That is an important measurement, but is really the only one required.

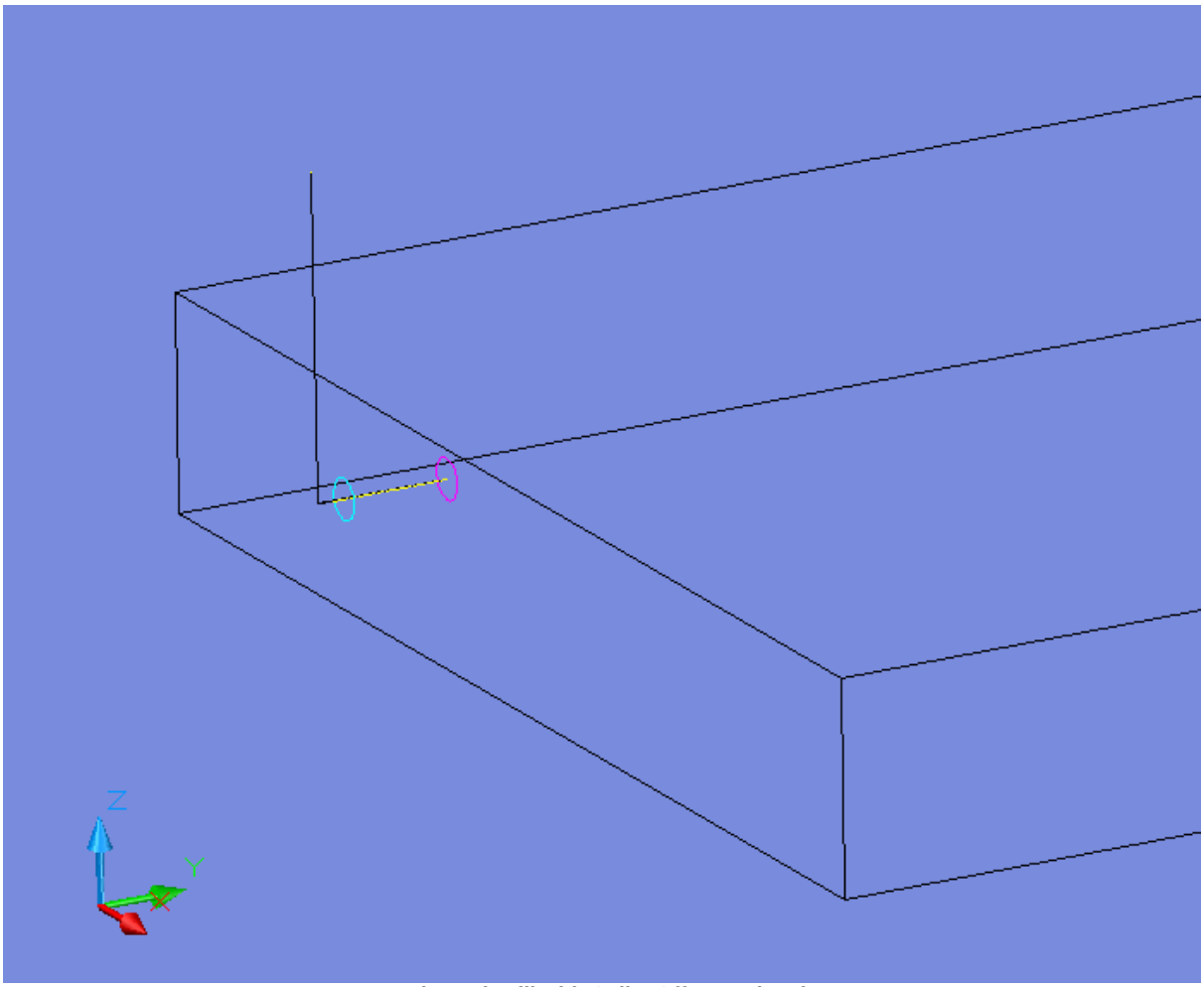
In the picture above, the tool path has been shifted by the value in the tool length field to emulate the spindle center, where the work coordinate or origin is set.

4.10.2 Collet Offset

Collet

If Collet is chosen, the cut path will appear inside the part the distance designated in the Total Depth parameter on the Control Panel. This method requires that you either set the work coordinate to the tip of the tool, or set the distance from the tip of the tool to the center of the spindle in the horizontal tool length offset and use Plane Detect.

© 2011 CIM-TECH.COM, Inc

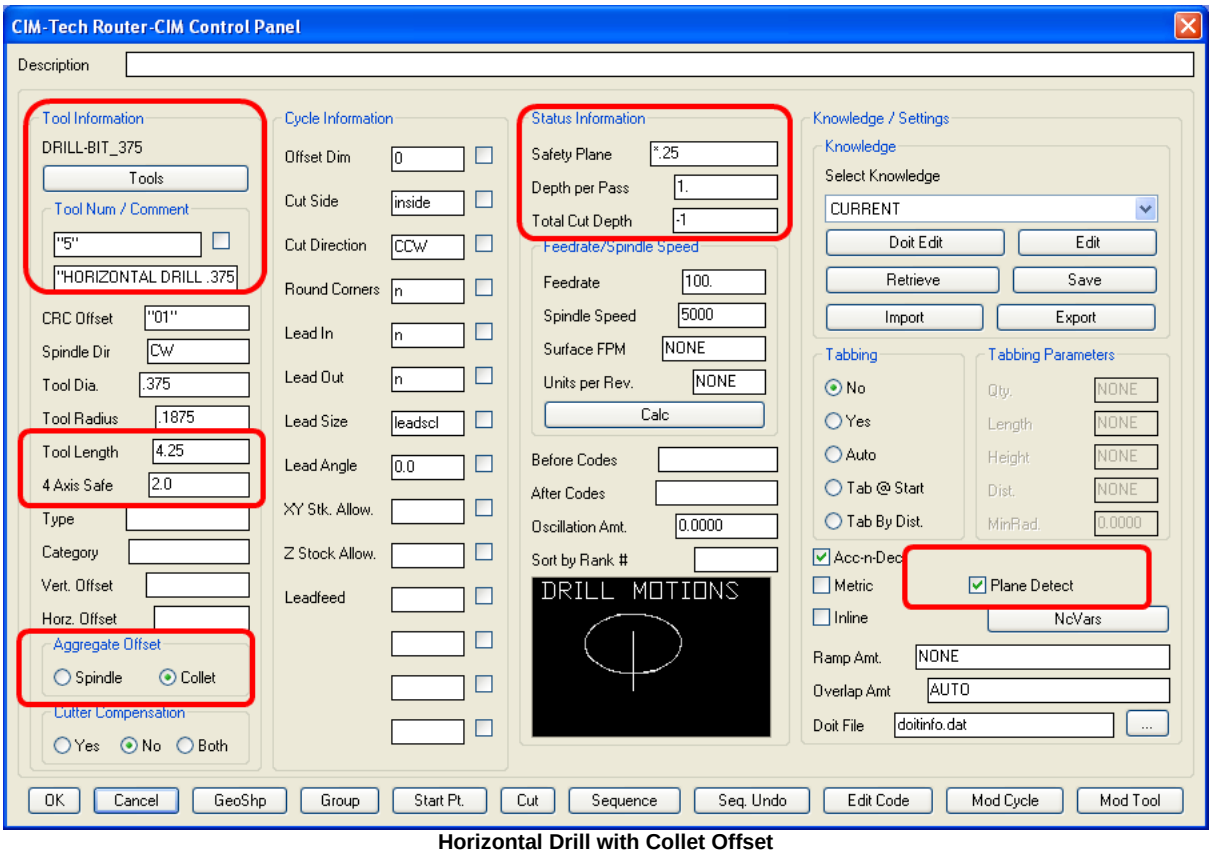


Horizontal Drill with Collet Offset tool path.

For an example of how this method would work, the following explanation is offered.

Looking at the picture below, the tools is a 3/8" Drill Bit, in a horizontal drill spindle.
The 4th axis Safe Plane is set to 2.0.
The Tool Length is set to 4.25.
The Aggregate Offset is set to Collet.
The Safety Plane is *.25, and the Cut Depth is -1.0.

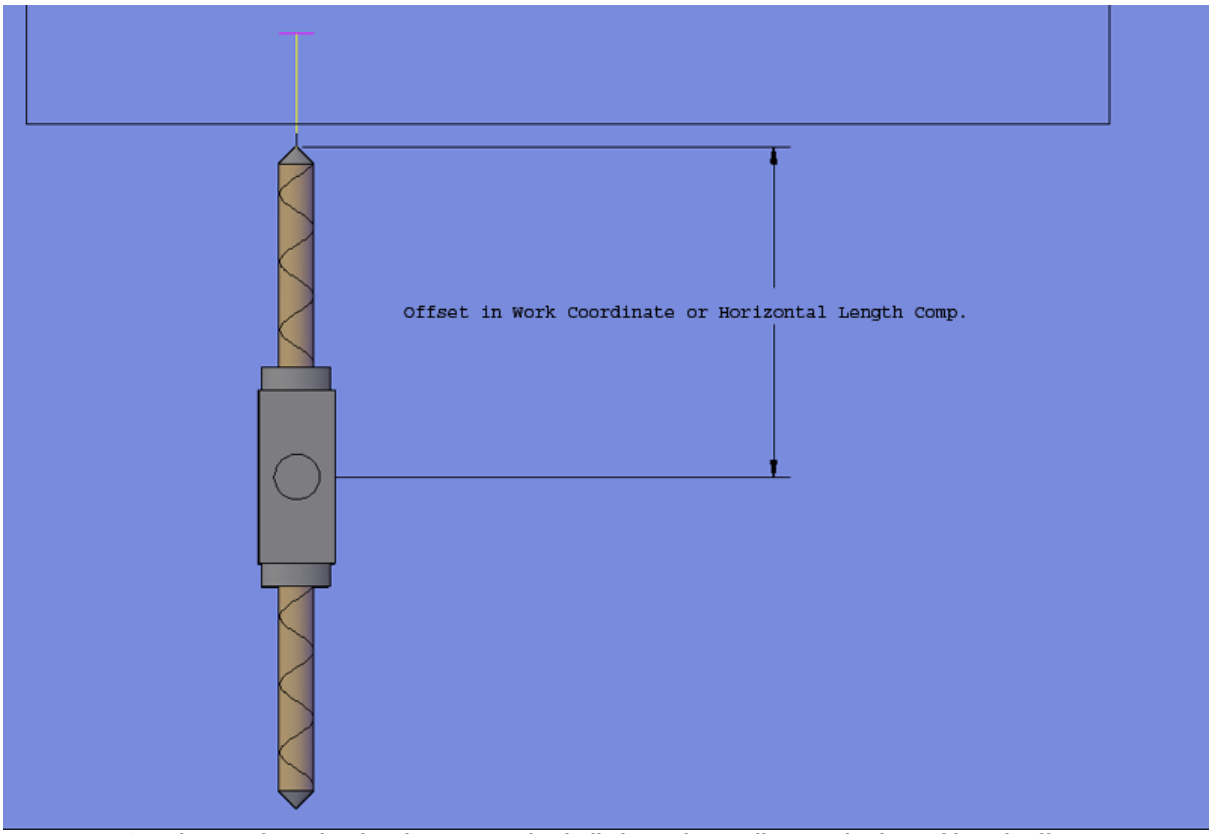
© 2011 CIM-TECH.COM, Inc



Horizontal Drill with Collet Offset

The only caveat to this system is that you must set the work coordinate to the tip of the tool, or horizontal length comp offset to the distance from the center of the drill or aggregate to the tip of the tool.

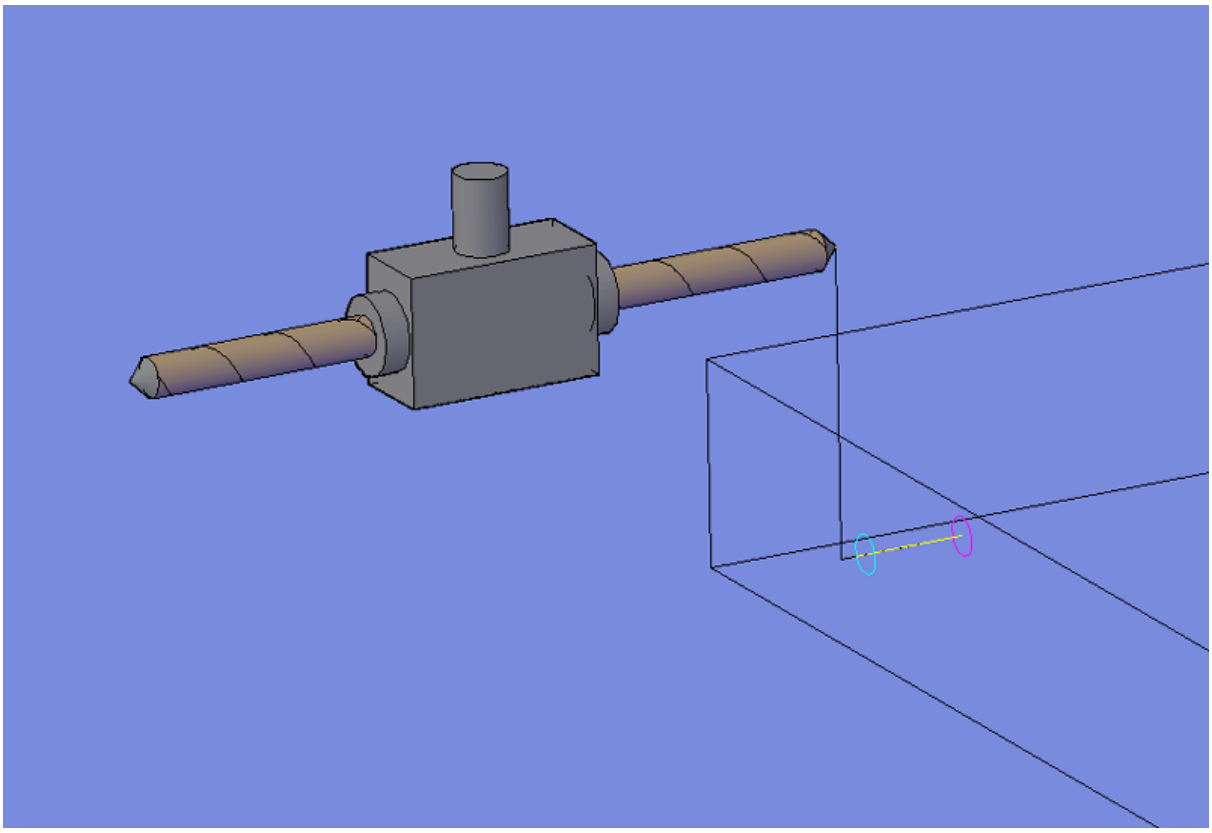
© 2011 CIM-TECH.COM, Inc



Set Distance from tip of tool to center of spindle in work coordinate or horizontal length offset.

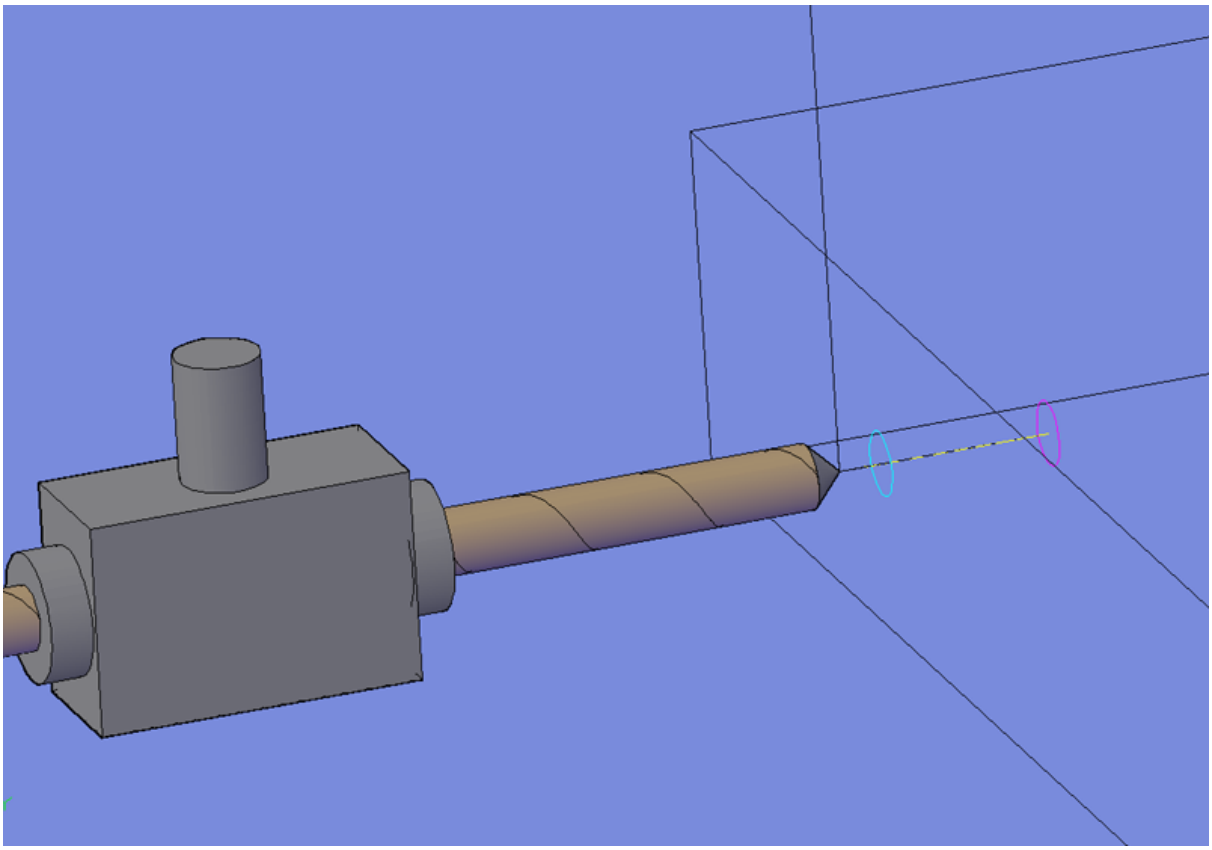
The tool will start above the part at the 4th Axis Safe point.

© 2011 CIM-TECH.COM, Inc



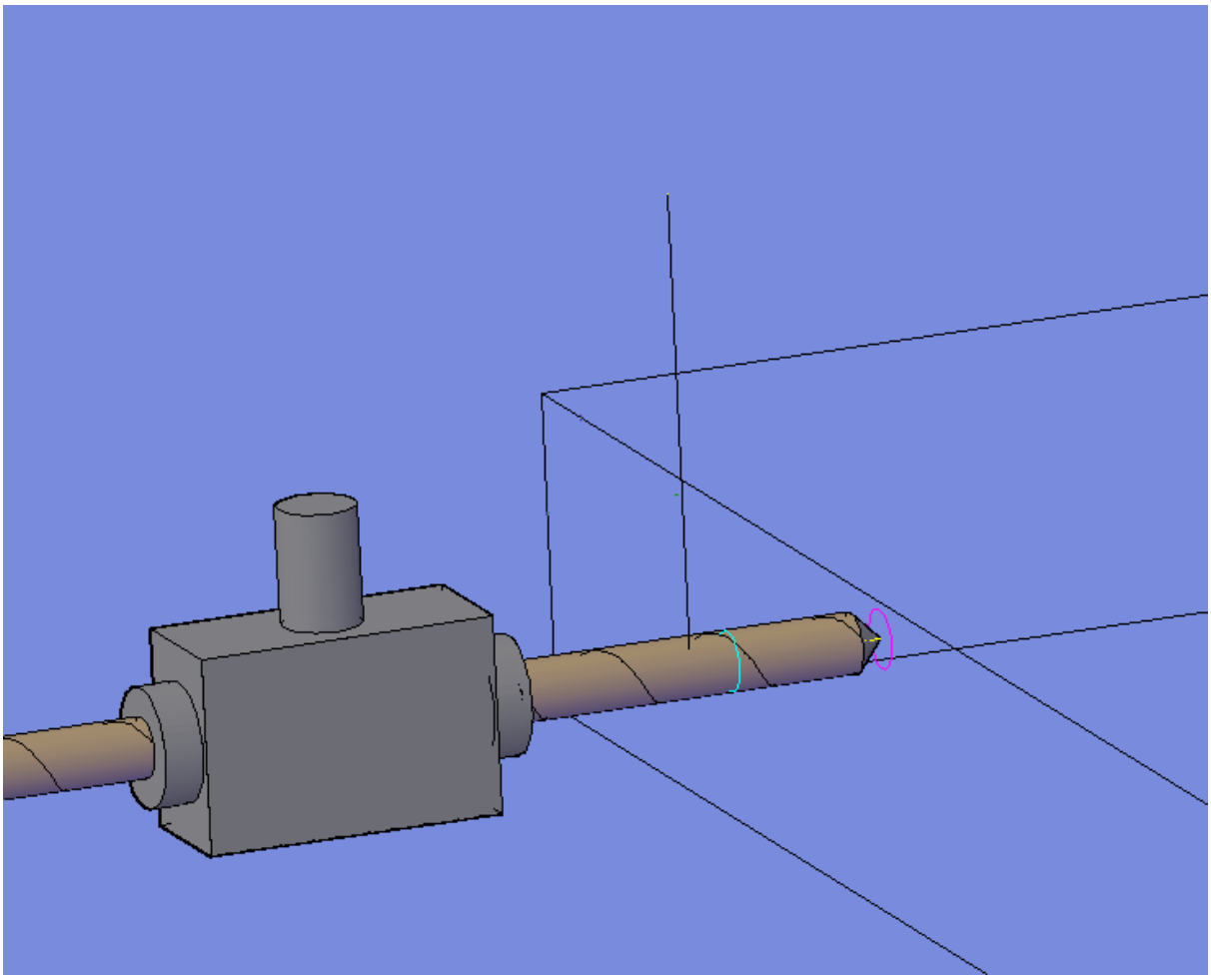
Next the tool will move down to the center of the hole, but still .25" away from the part as set by the Safety Plane. This will be a Rapid Traverse (G0) move.

© 2011 CIM-TECH.COM, Inc



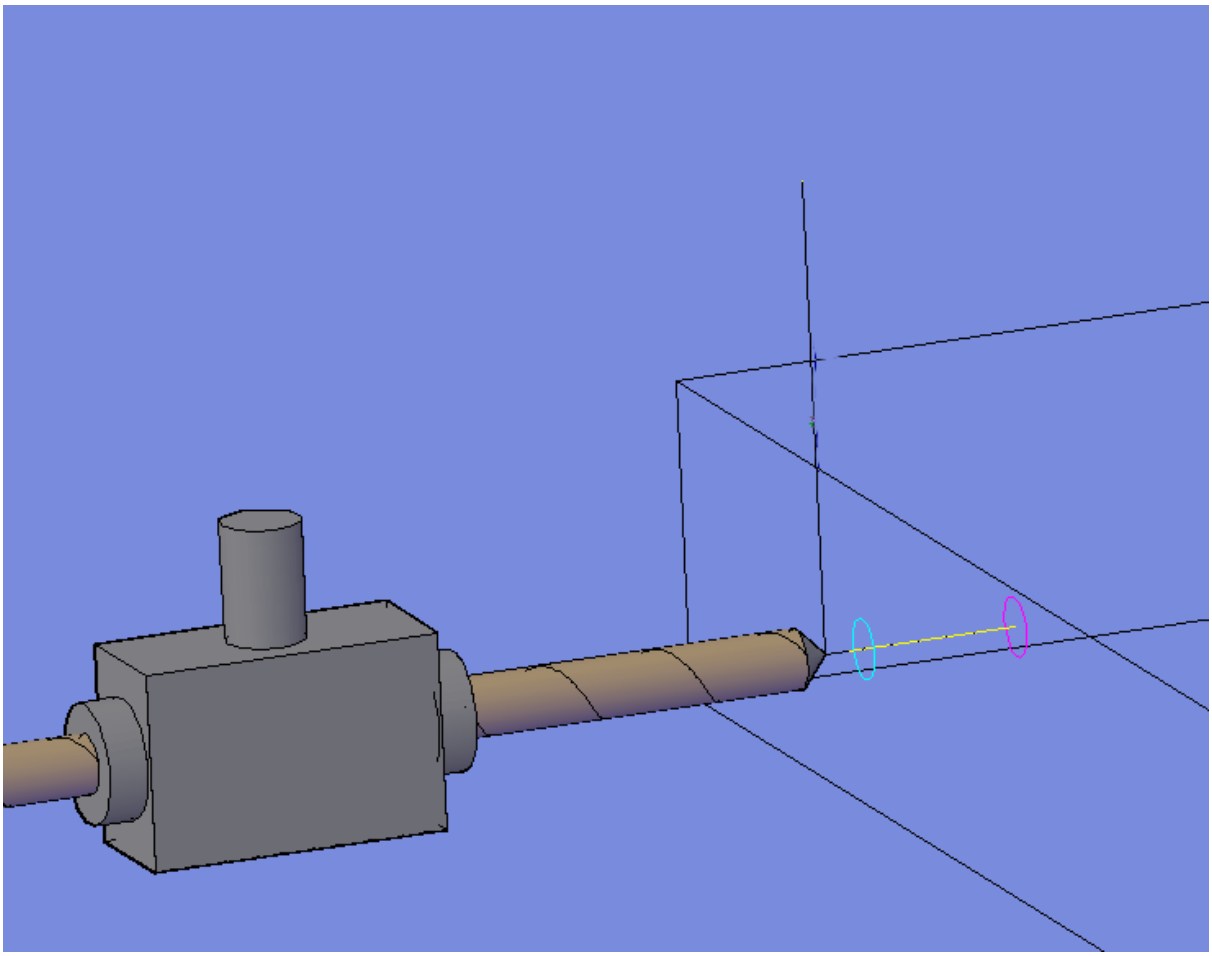
Next, the tool will feed into the part from the Safety Plane to the depth of cut (or depth of first pass if set to multiple passes) in feed mode (G1).

© 2011 CIM-TECH.COM, Inc



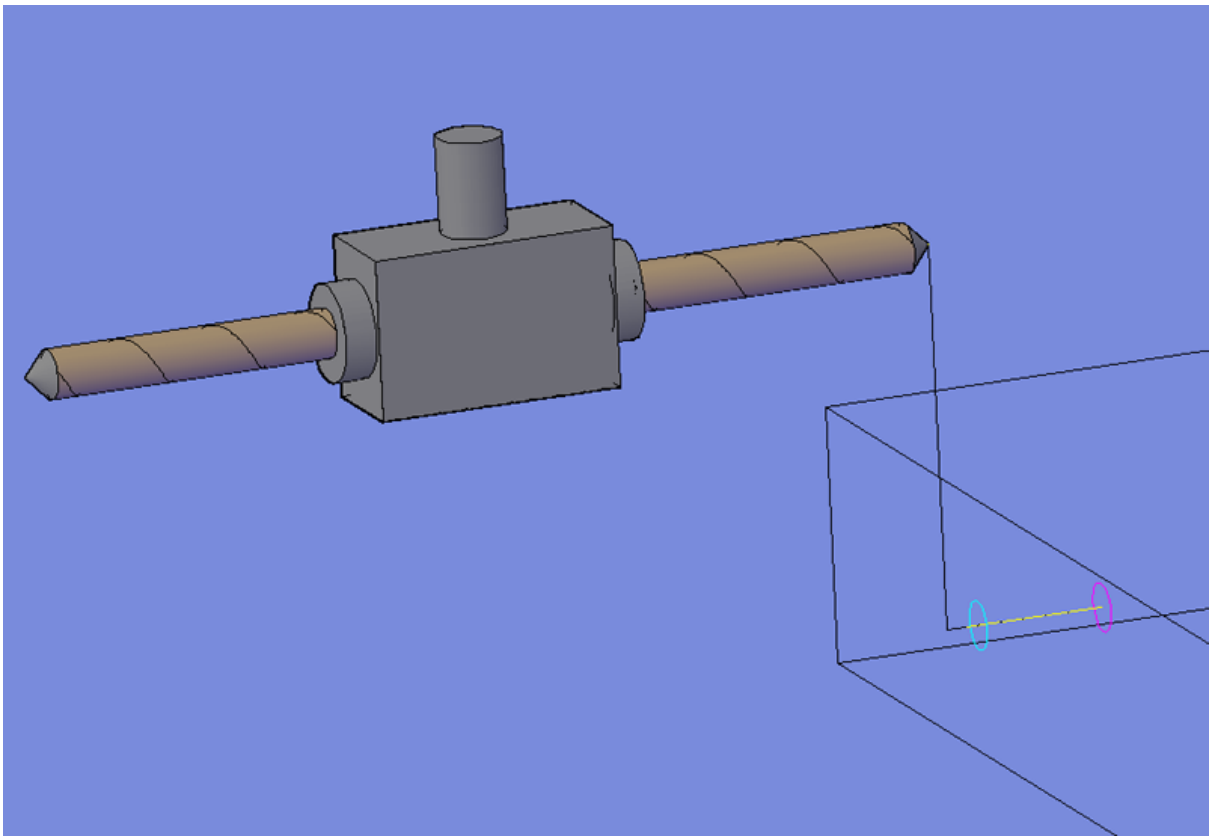
Once the tool has reached the bottom of the hole, it will retract back to the Safety Plane. This will be a Rapid Traverse move (G0) since there is no material to remove and the tool is retracting back to the point it just came from.

© 2011 CIM-TECH.COM, Inc



The last move will be to retract back to the 4th Axis Safe point. This will also be a Rapid Traverse (G0) move. This is the position the tool started from.

© 2011 CIM-TECH.COM, Inc



Using this method of offset only requires you to use either the Plane Detect with a Horizontal Tool Length offset or set the Work Coordinate to the tip of the tool.

4.10.3 Combination of Spindle AND Collet Offset

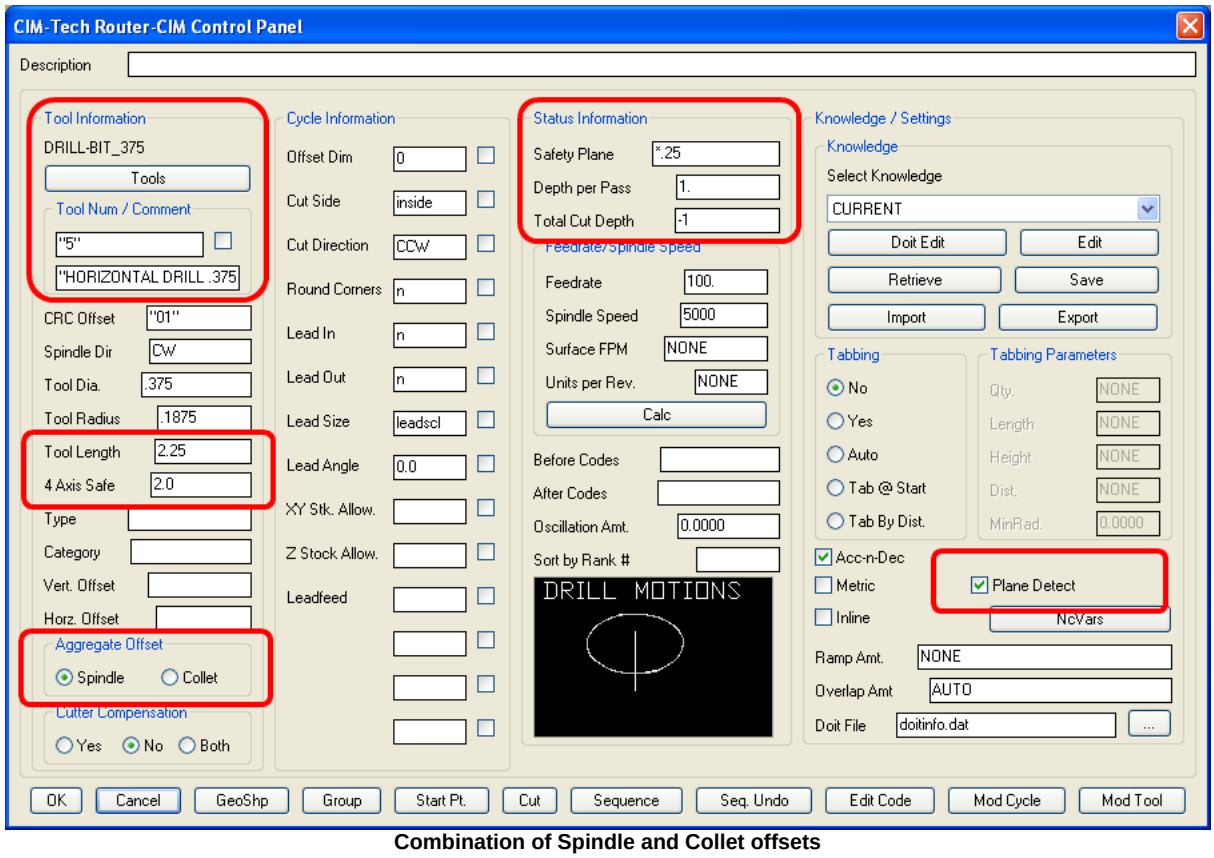
The Combination of Spindle and Collet methods

There is a third choice for using a horizontal drill. It is probably the easiest for both the programmer and operator alike. This method requires that you are cutting on one of the 4 faces of the part and not some arbitrary angle, and also for the programmer to know the distance from the center of the drill block to the face of the collet that the tool fits into. It then requires the operator to measure how much tool is sticking out of the collet in the drill block.
The programmer places the distance from the center of the spindle (where the work coordinate is set) to the face of the collet in the Tool Length.
The operator placed the tool length (the amount of the tool sticking out of the collet) into the horizontal length offset.

This method allows for the programmer to only need one number that never changes for a tool length, and the operator can measure the amount from the collet to the tip of the tool (with almost any measuring device) and put it in an offset, just like he/she would for any other tool touched off.

In Router-CIM, the knowledge would be set to Spindle, Plane Detect would be ON, and the setup would look like this for example:

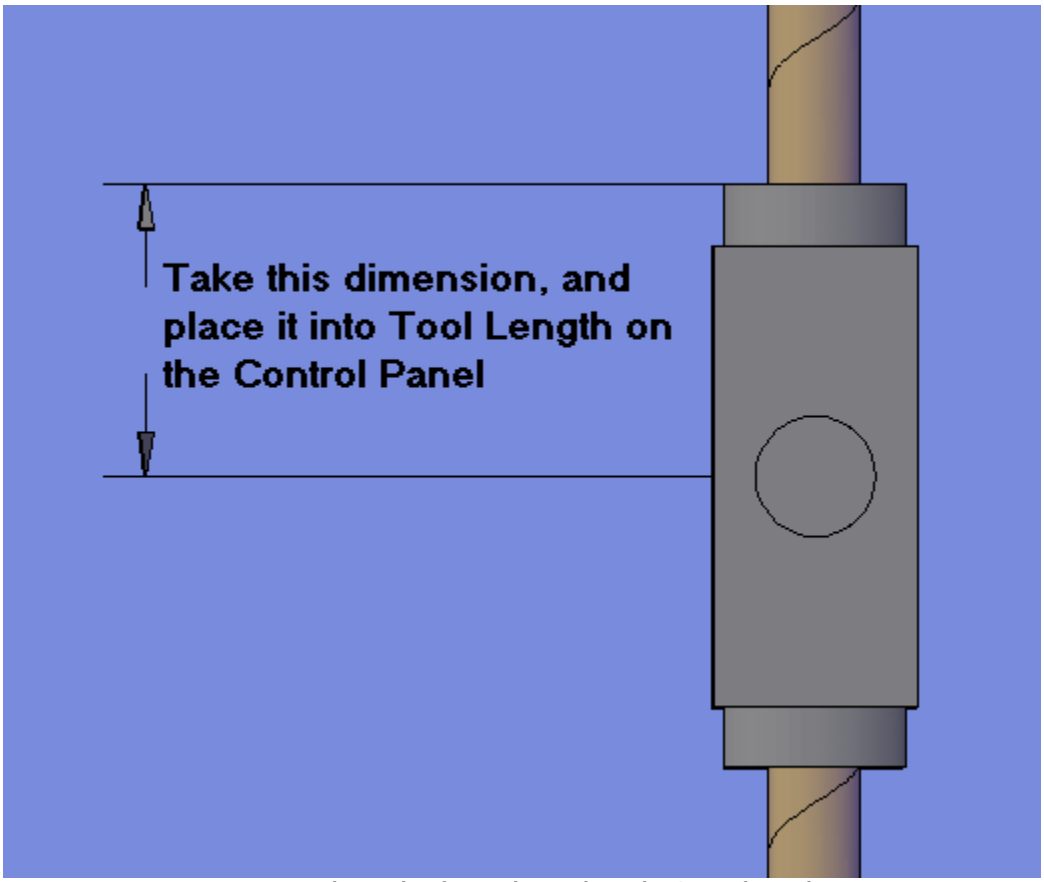
© 2011 CIM-TECH.COM, Inc



Combination of Spindle and Collet offsets

The Tool Length is set to the dimension of the center of the spindle to the face of the collet, which really only needs to be taken once. Set Plane Detect on. This is important.

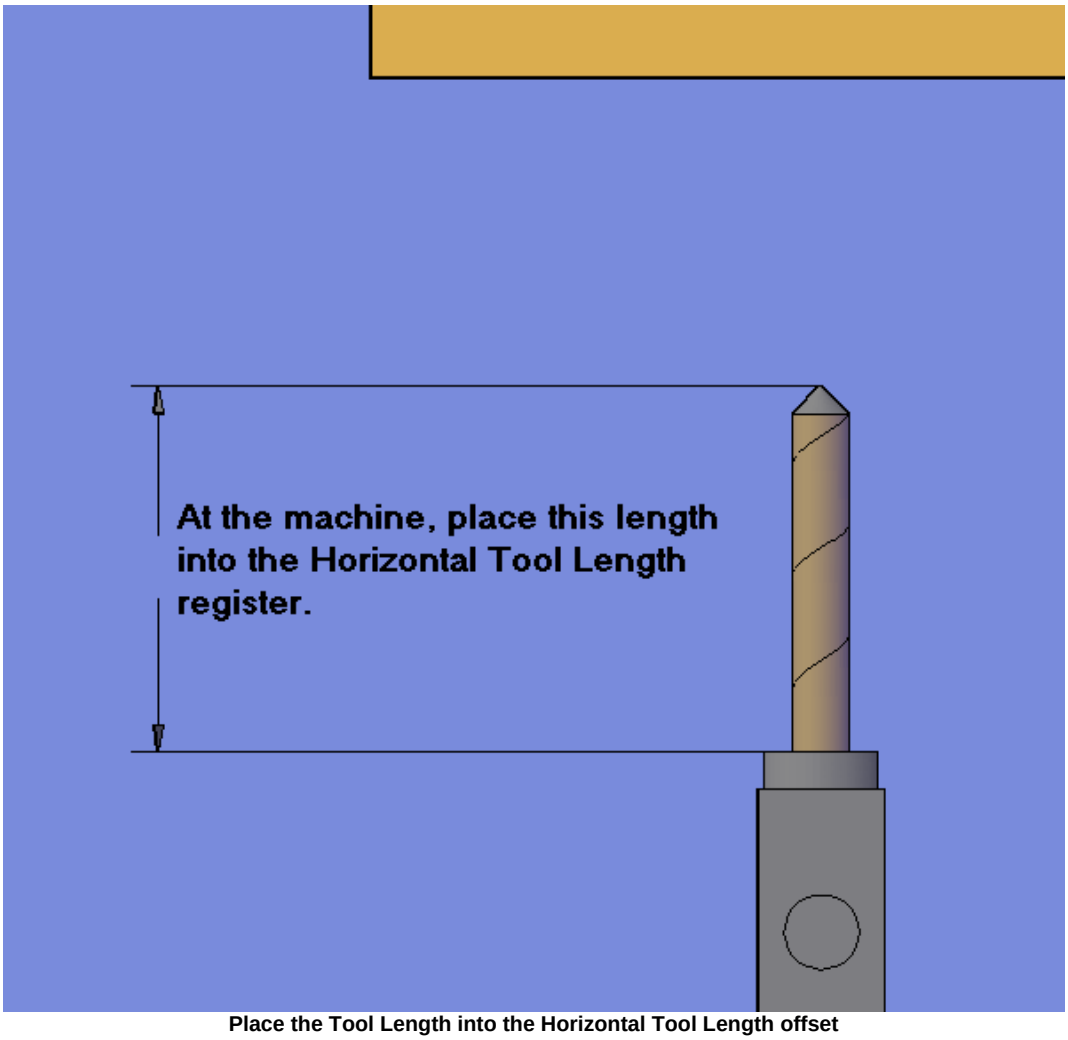
© 2011 CIM-TECH.COM, Inc



Measure the setting for Tool Length on the Control Panel

Then make the cut, with the same settings as a Spindle offset tool path. The cut will be placed out away from the shape by the amount in the Tool Length parameter, and then at the machine, the operator must place the length of the tool in the Horizontal Tool Length offset register.

© 2011 CIM-TECH.COM, Inc

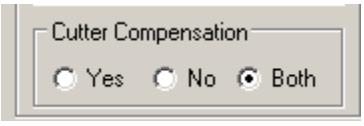


Place the Tool Length into the Horizontal Tool Length offset

When the machine sees the Horizontal Offset and the plane command (G18/G19) it will offset the tool path further by the length of the tool and place the tip of the cutter in the proper position.

This method is easier for both the programmer and operator from the standpoint that the programmer needs only one setting (Spindle to Collet distance) which will always stay the same, and for the operator that sets the tool offsets, the tool can be measured as it would normally be and that amount placed in an offset register just like a vertical tool.

4.11 Cutter Compensation



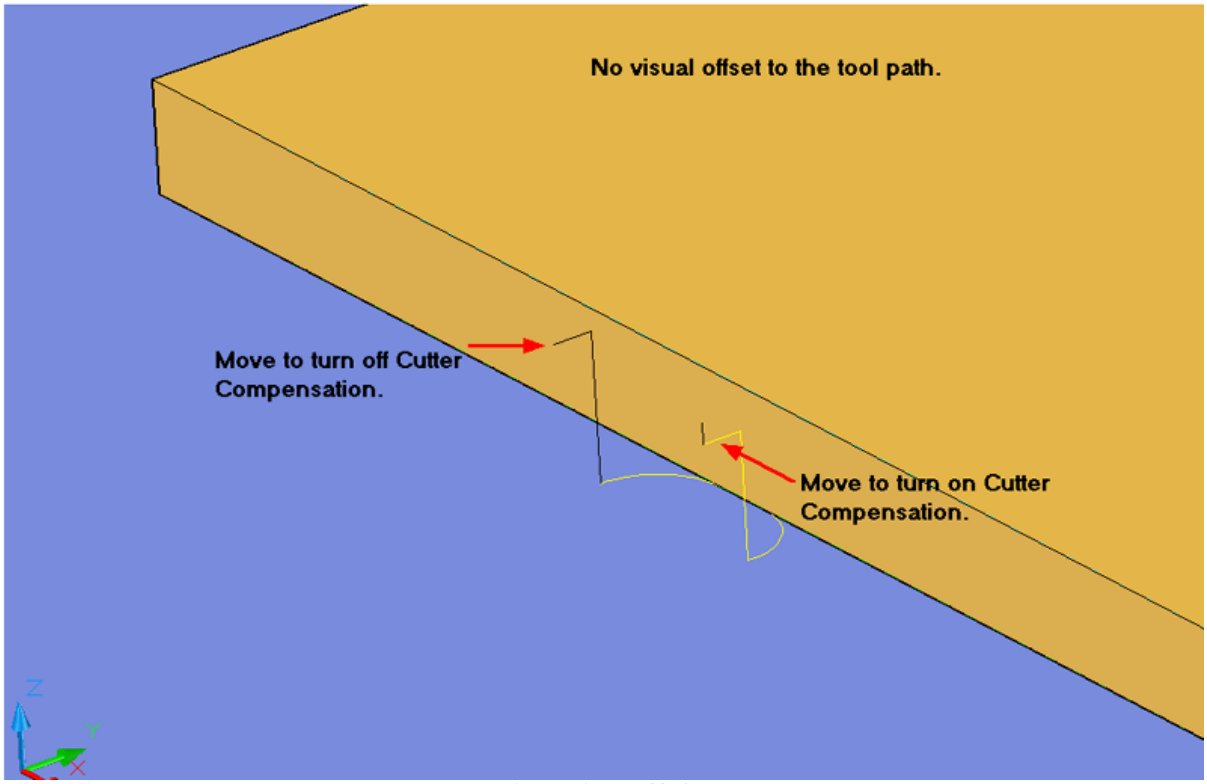
Cutter Compensation will accept 1 of 3 answers: Yes, No, or Both, explained below.

Cutter Comp YES

When using this option you will see the tool path directly on top of your part geometry.
The code that is produced reflects the part profile with compensation. At the machine, the entire tool

© 2011 CIM-TECH.COM, Inc

radius must be placed into the offset register.
Your code will have the G41 or G42 with an offset register number, such as D01, and a move to turn off the compensation with the G40 code.

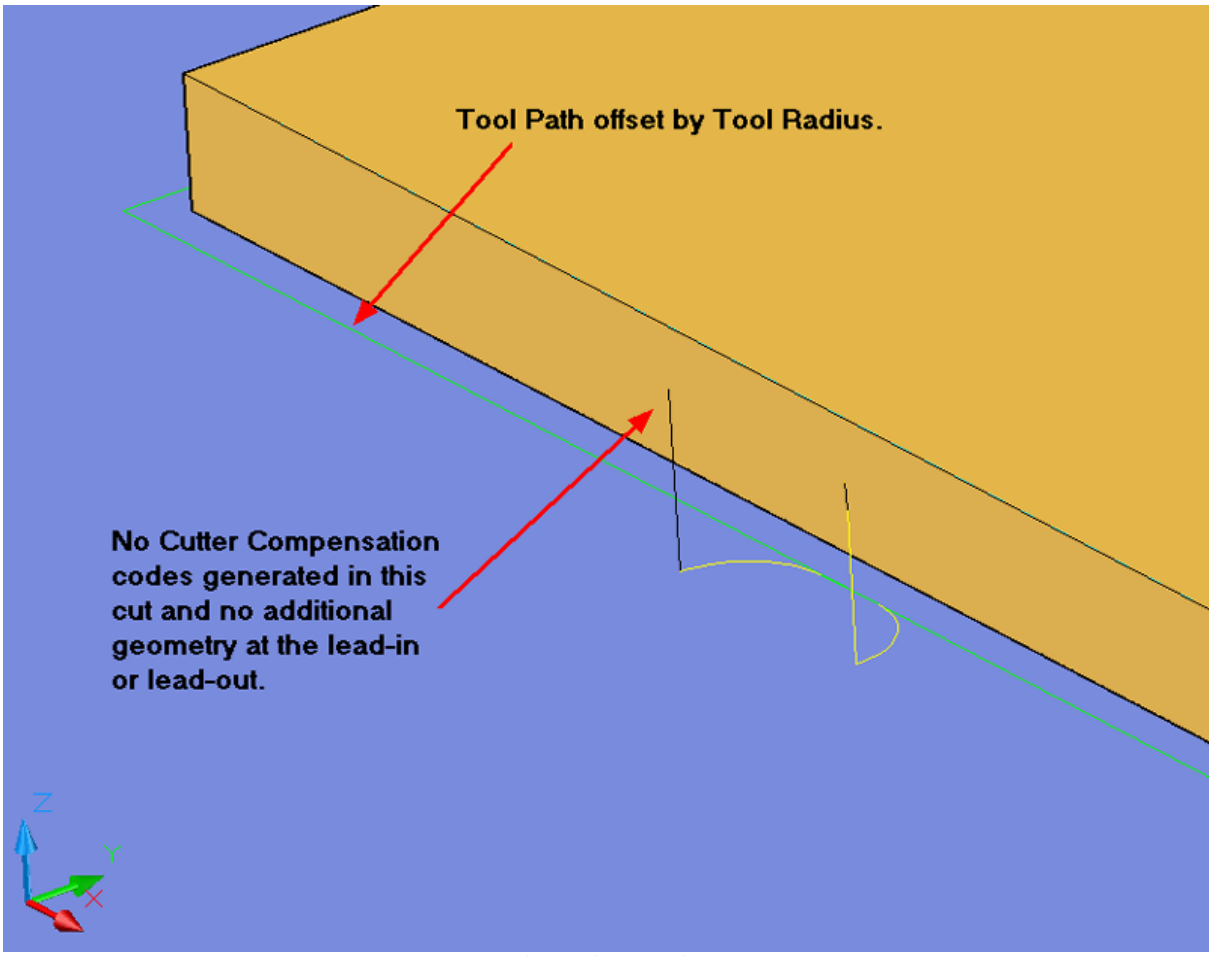


Cutter Comp. YES

Cutter Comp NO

Using this option will generate a tool path that is offset by the radius of the tool as it appears on the Control Panel in the Tool Radius parameter.
This option does not generate a Cutter Compensation code (no G41/G42/G40 or D numbers).
Typically No is used for drilling, surface cutting, pocketing and engraving.

© 2011 CIM-TECH.COM, Inc

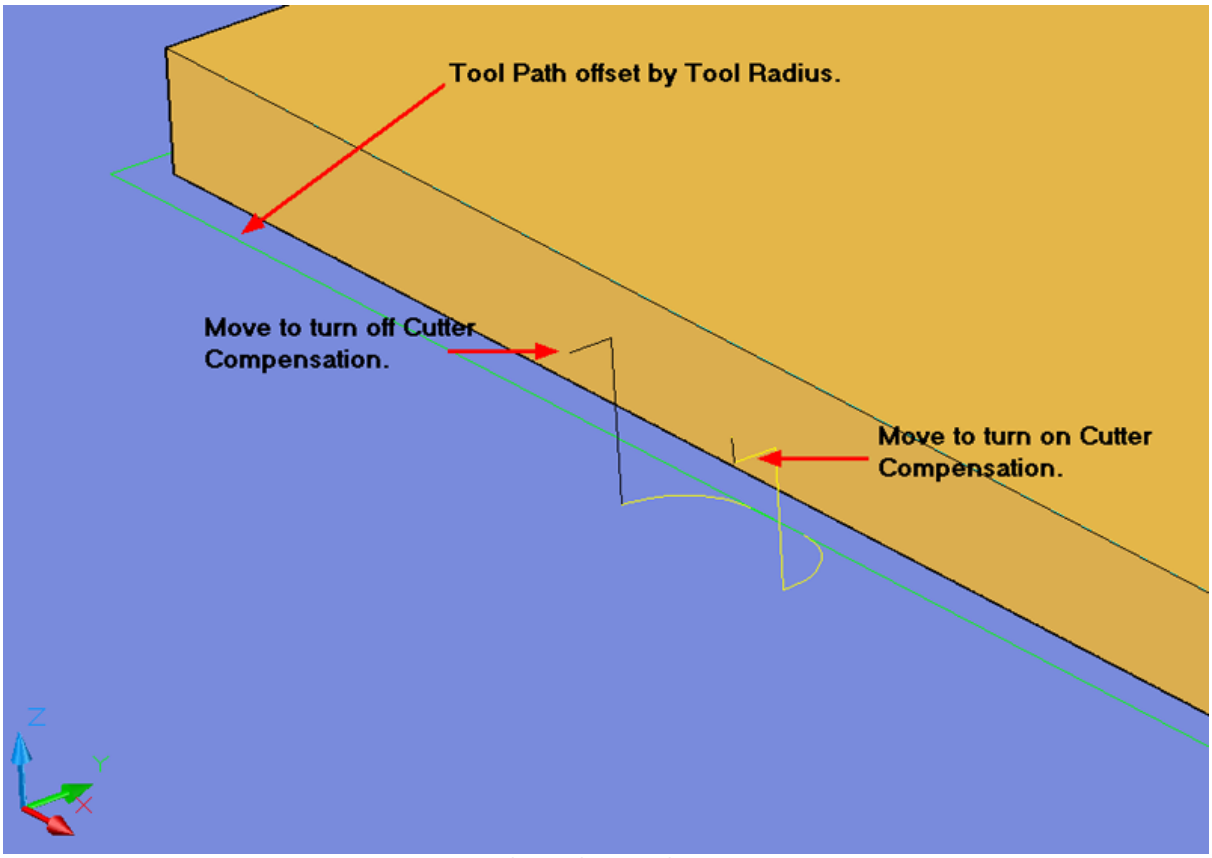


Cutter Comp. NO

Cutter Comp BOTH

This option will show the tool path offset by the radius of the tool as it appears on the Control Panel in the Tool Radius parameter.
Both will generate a G41/G42 also, along with a D number and a G40 to cancel the compensation.
The wear amount of the tool is then input into the offset register (D#) at the machine.
This option provides the best results because you can visually inspect the offset on the screen, and compensate for tool wear. Typically used for all contour cutting.
The B for Both choice is useful, because you input a D value at the machine that is the difference between what Router-CIM offsets and what the tool radius actually is (usually a very small amount). Measure the tool radius and add or subtract that value from what was offset (nominal radius), and then place that difference into the appropriate D register (the offset page at the controller). You should program most cuts with Cutter Compensation using Yes or Both. Only by using Cutter Compensation can you adjust for sharpened or worn bits as well as to parts cut with different sized bits.

© 2011 CIM-TECH.COM, Inc



Cutter Comp. BOTH

Note: A thorough understanding of Cutter Compensation is very important. Consult your control's Operator's Guide for more information on G41, G42, & G43.

5 Modifying a Tool



Modifying Tools is a common and necessary practice in Router-CIM. You can make these changes temporarily, in a current drawing, or permanently, in your machine specific default drawing. You can even create and use a new tool list if desired.

There will be times when just changing the diameter and radius are enough to get the job done. However if you find that you are making changes often, making a new tool may be a better solution.

There are three sections to this chapter.

- Temporary Editing
- Permanent Editing
- Creating a new Tool List

© 2011 CIM-TECH.COM, Inc

5.1 Temporary Editing

When you start Router-CIM, a blank drawing is inserted into your current drawing session. That blank drawing contains all of the blocks and lists that Router-CIM needs in order to display the tool, cycle and status information on the Control Panel. Once the drawing is saved with that information in it, any temporary edits you made to Router-CIM are available in that drawing for future use. Router-CIM retains in the cut knowledge, any edits you made in order to create a tool path for the cutting condition you needed.

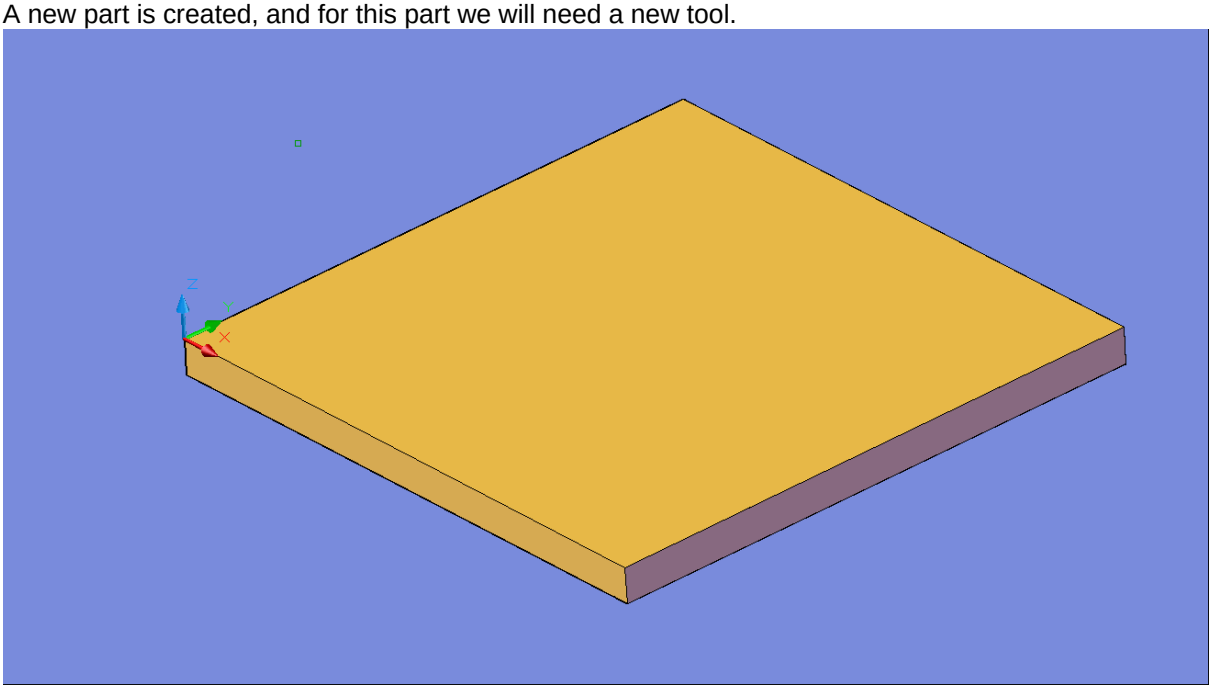
If you modify a tools' parameters, then make a cut, that cut contains all the modified parameters for that tool.

The key here is that this information is only available in the drawing if the tool paths are saved with the modified data in them, and only that particular drawing has the changed parameters.

This might be enough if you are just modifying a tool number, offset, comment or temporarily changing a tool size.

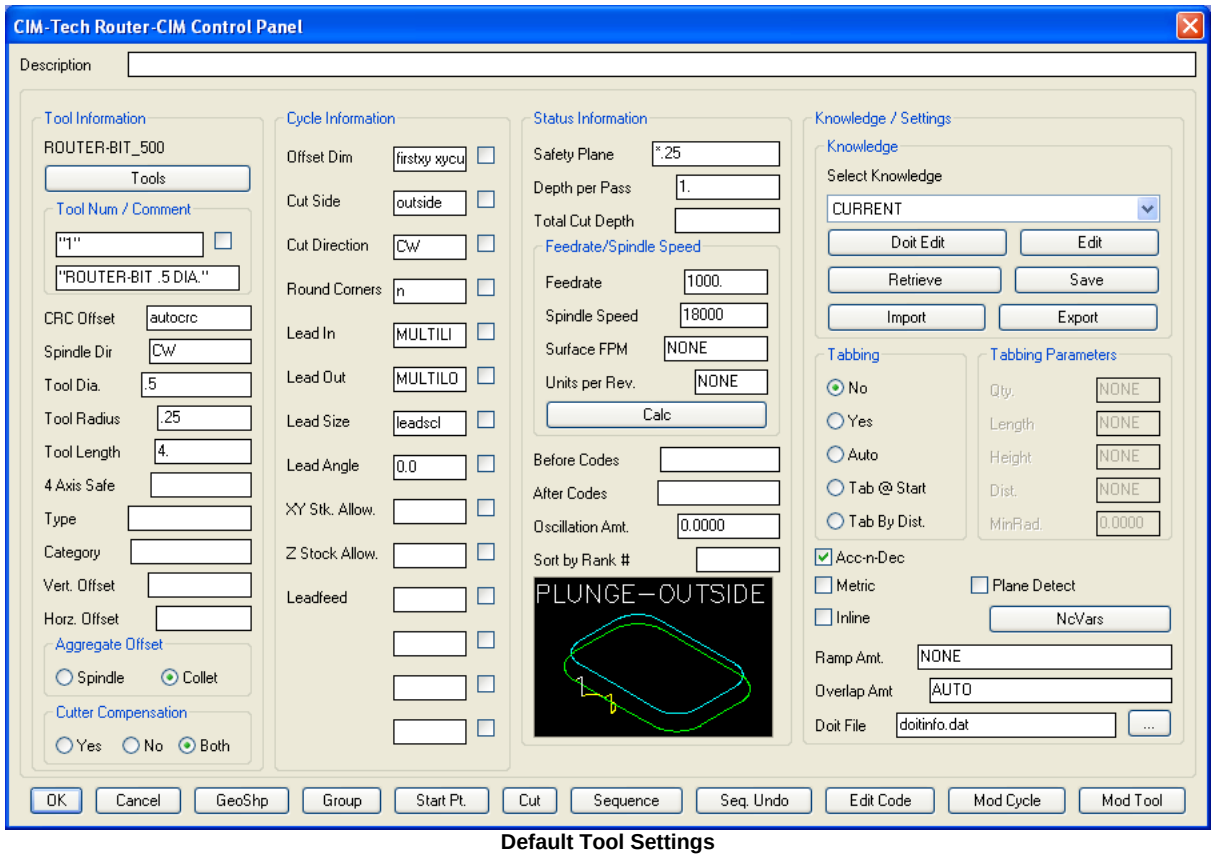
To demonstrate the changes that are stored and how they are stored, the following example is shown.

A new part is created, and for this part we will need a new tool.



The default tool show in Router-CIM in our current session is a .5" Router-BIT. This is the tool we are going to temporarily modify into a 1.5" diameter shaper tool.

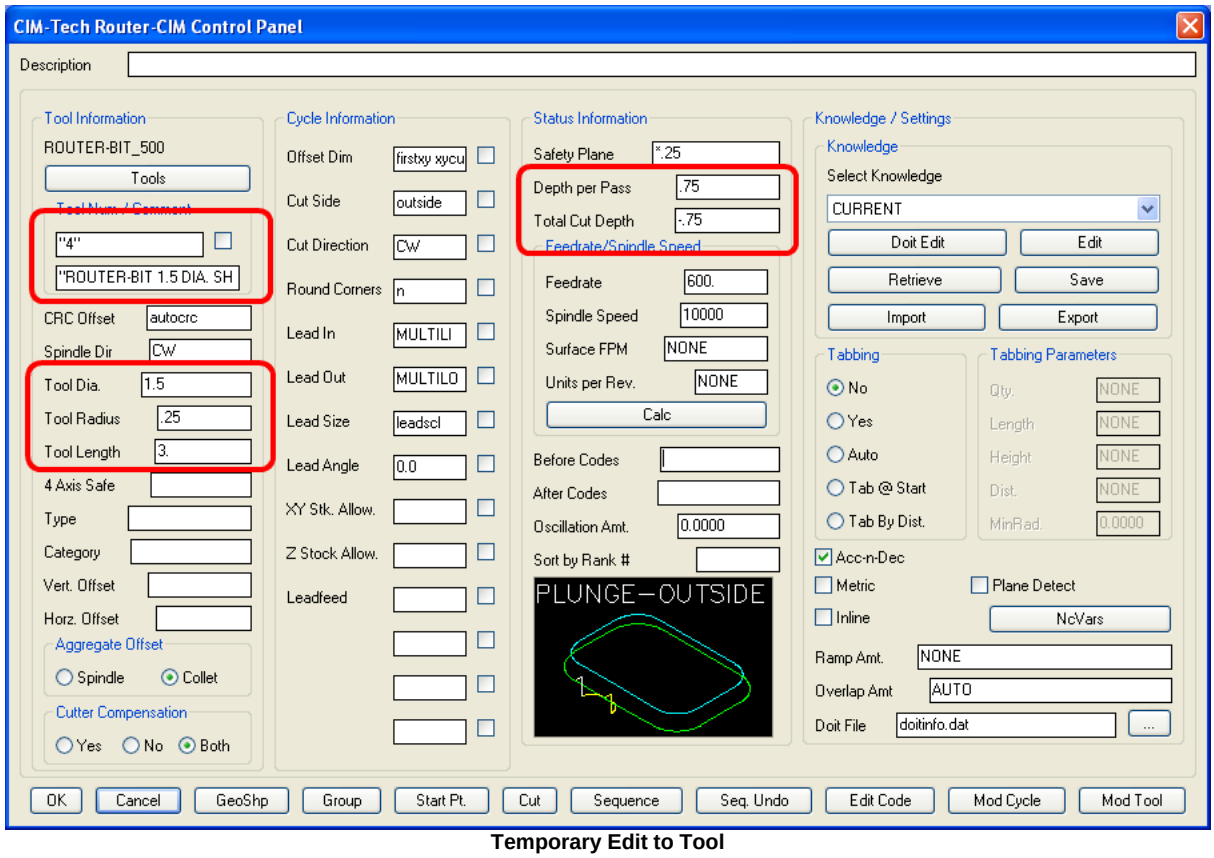
© 2011 CIM-TECH.COM, Inc



Default Tool Settings

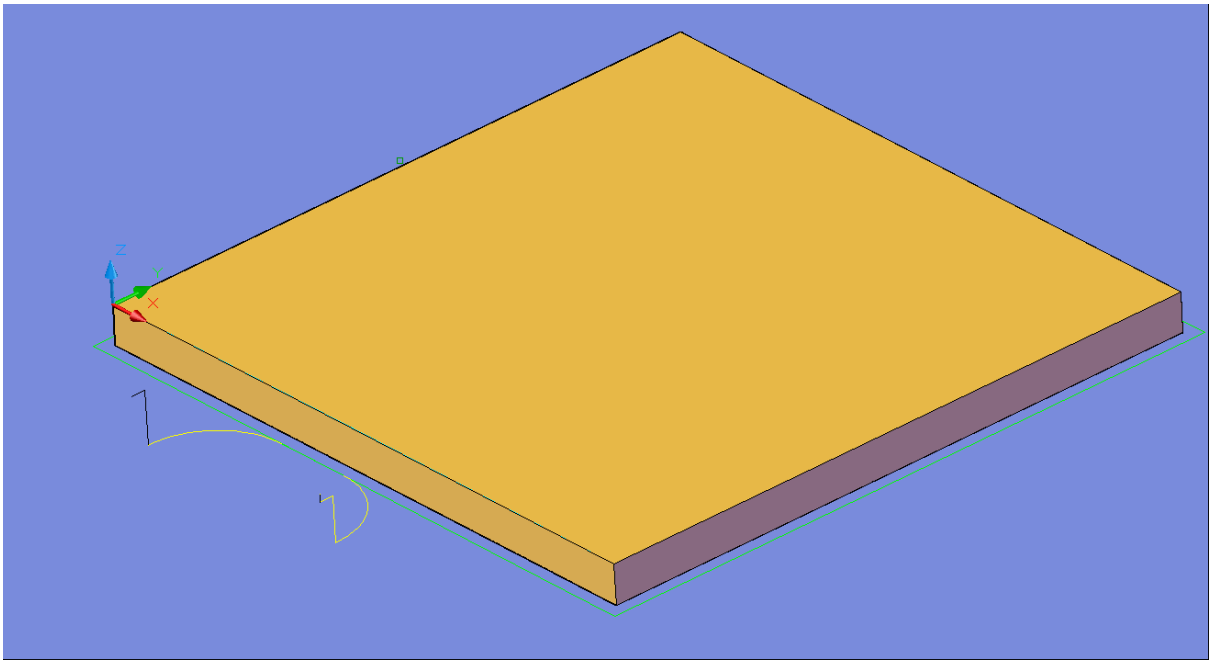
If you were to simply modify the Tool Number, Comment, Tool Diameter, Radius, Length, and Depth Per Pass, you would create a new tool that is the size needed for the current operation.

© 2011 CIM-TECH.COM, Inc



Temporary Edit to Tool

Once those changes are made, we can set the Feedrate, Spindle Speed, Safety Plane and any other cycle or status parameters and then make the cut.



New, temporary, parameters applied to tool path

© 2011 CIM-TECH.COM, Inc

When the drawing is saved, with the tool path, all the parameters that were changed will be available again by retrieving the knowledge from the cut.

If a New drawing is opened, however, none of the changes that were made will be available because they were only edited in the control panel and only saved within the drawing where they were used.

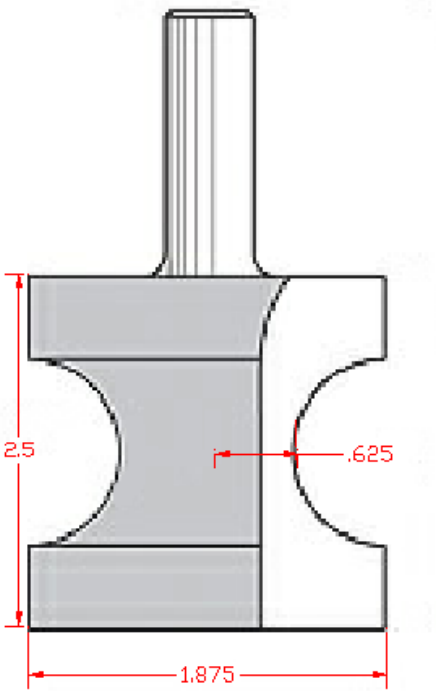
If the tool you modified is going to be used over again at a later time, a more permanent solution is needed to avoid having to enter the tool parameters over and over again.
Saving these parameters requires Permanent Editing.

5.2 Permanent Editing

Making permanent edits to the tool list involves making the changes to the tool in the permanent editing section, and then adding the tool to the list and filing the list so that it is available again in other drawings.

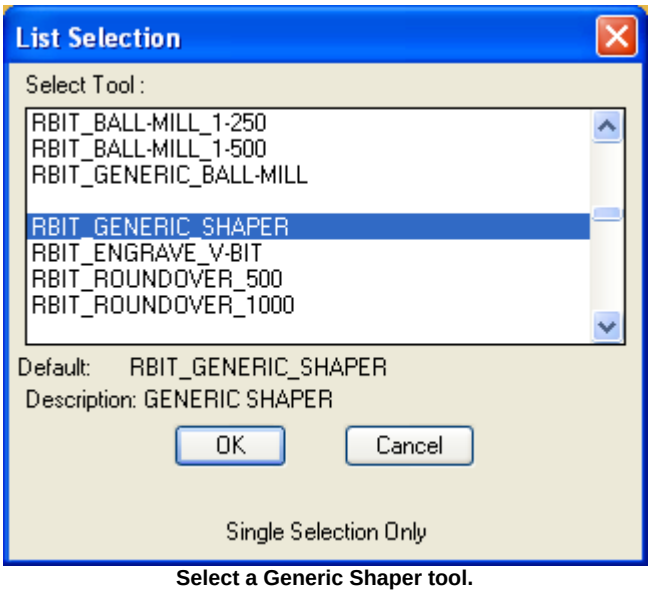
In order to show Permanent Editing, we will create a new tool in the tool list, then file the list and open a new Router-CIM session to show the tool available. You should read through all the steps involved before trying this and then come back and step through them one at a time until your results are the same.

The new tool created in this example is going to be a 1.875" Diameter shaper type tool that looks like the following illustration.



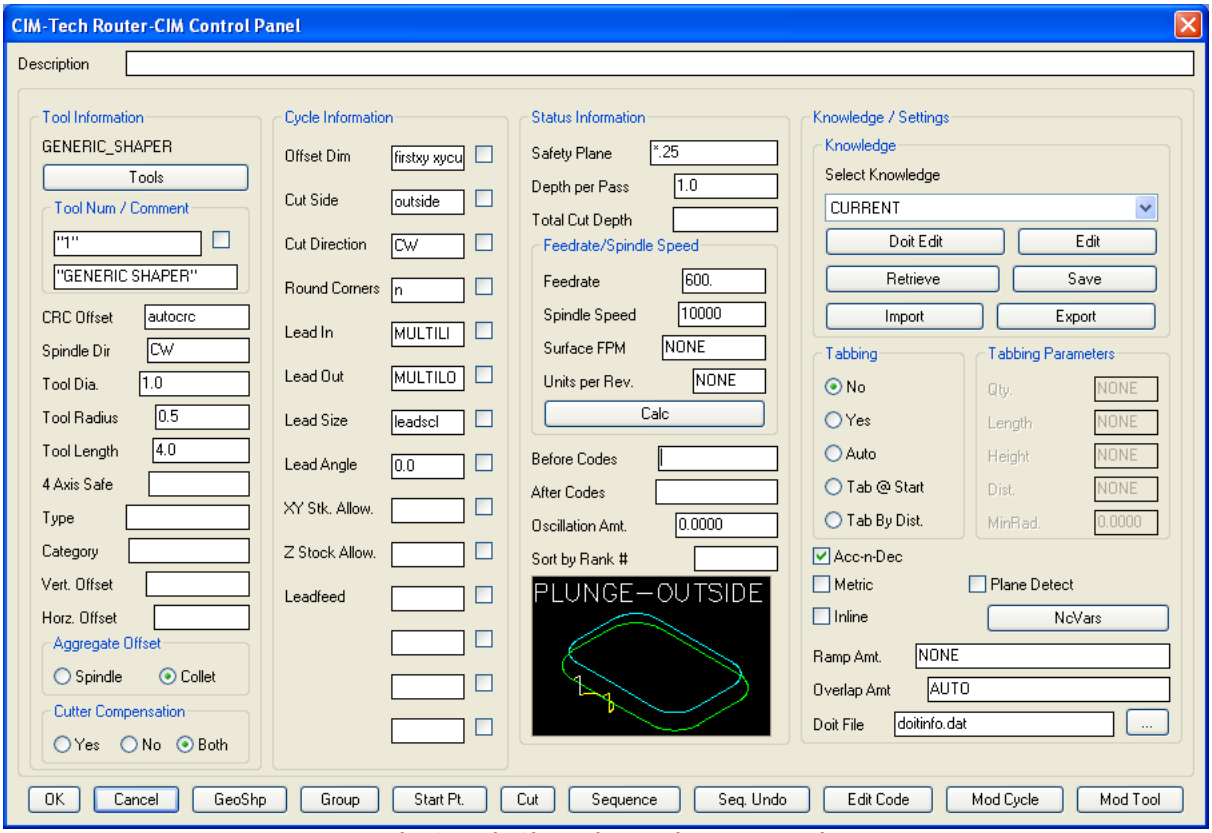
In order to make this tool, a new, blank, drawing is opened up and Router-CIM is loaded.
Then we start with the Control Panel and pick a tool from the tool list that is somewhat close to what we want.
Select the TOOLS button and in this case the RBIT_GENERIC_SHAPER is selected.

© 2011 CIM-TECH.COM, Inc



Select a Generic Shaper tool.

Once that tool is selected, click on OK to move back to the Control Panel with that tool as the current tool.



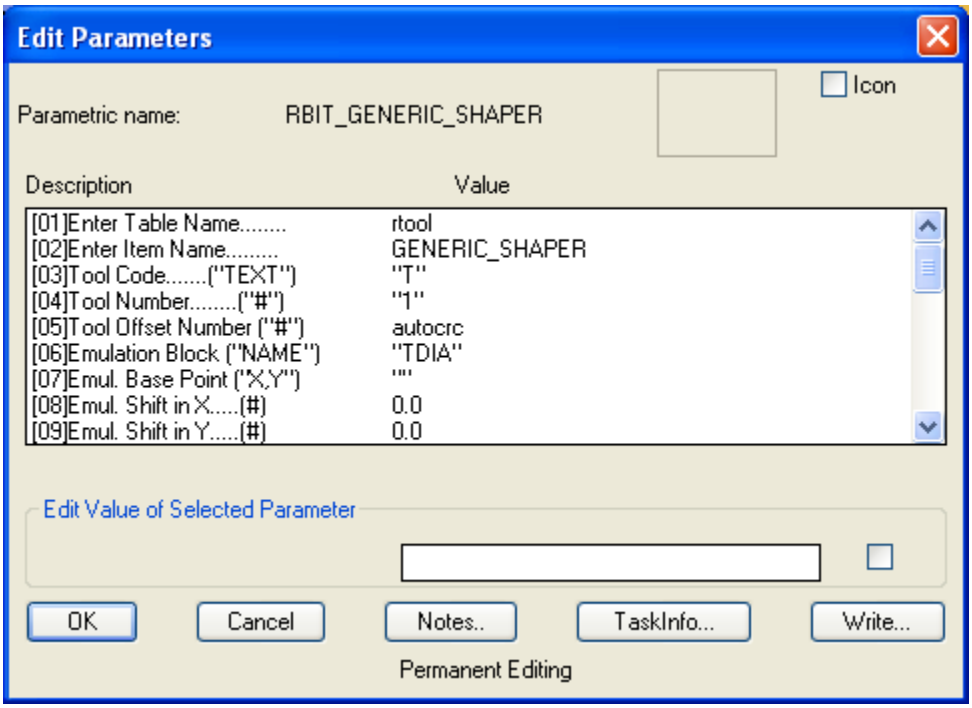
The Generic Shaper is now the current tool

Next we will modify the tool parameters to create the new tool for the list.

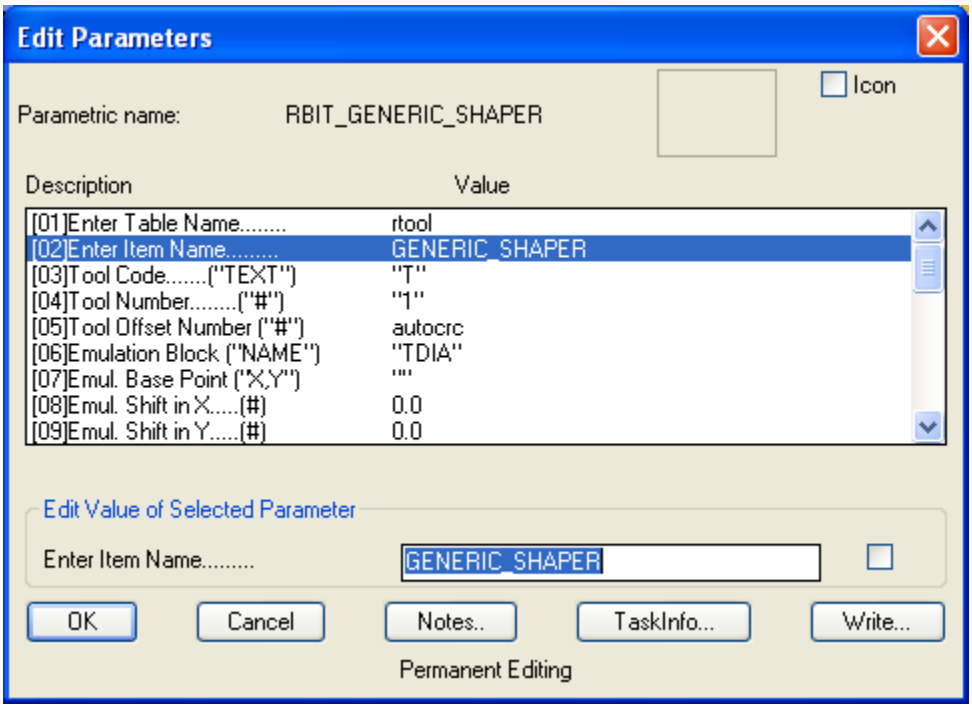
© 2011 CIM-TECH.COM, Inc

Select Mod Tool. 

Mod Tool displays all the tool parameters, some of which will be changed to create the new tool.

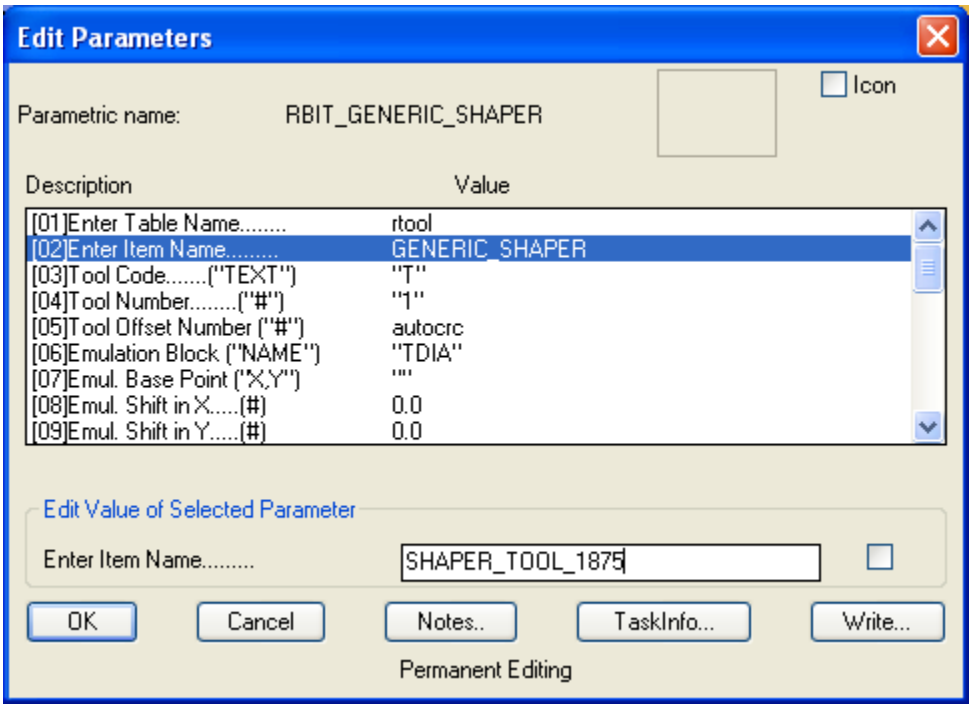


The first item to change is the Item Name (#2 in the list). Select this item with your mouse and then at the bottom of the window you will see the value to be changed.

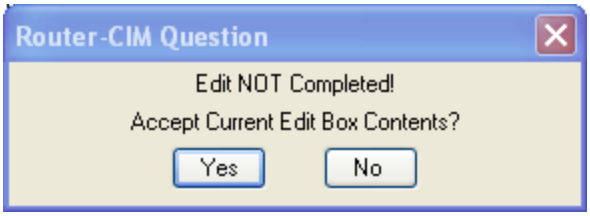


© 2011 CIM-TECH.COM, Inc

Turn on the Caps Loc and type in SHAPER_TOOL_1875 and then Press ENTER on the keyboard.



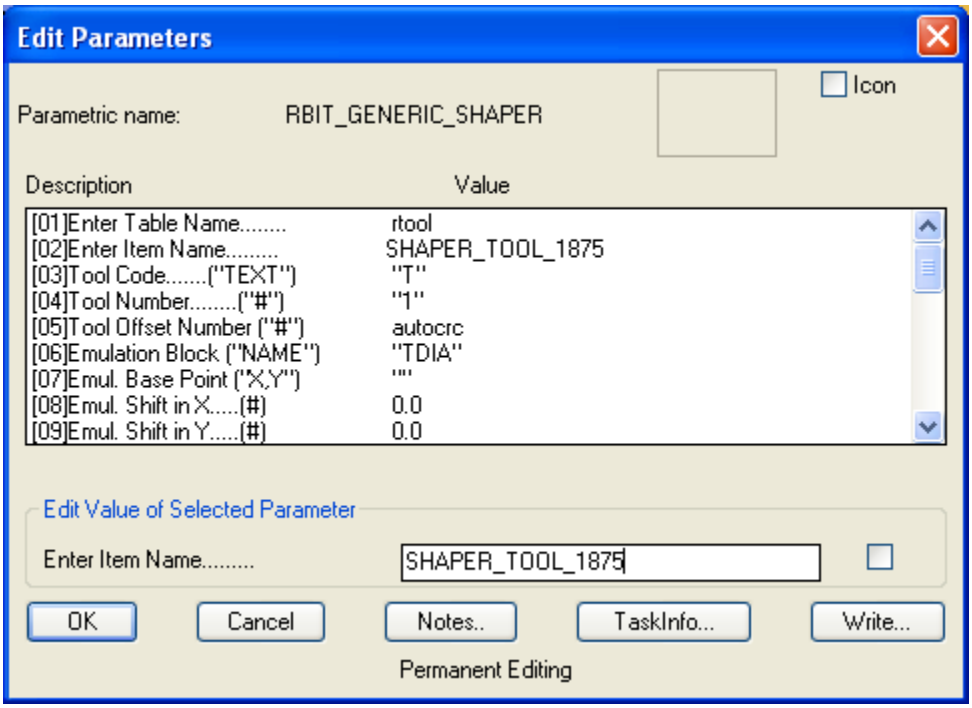
If you do not press ENTER, you will see the following notice:



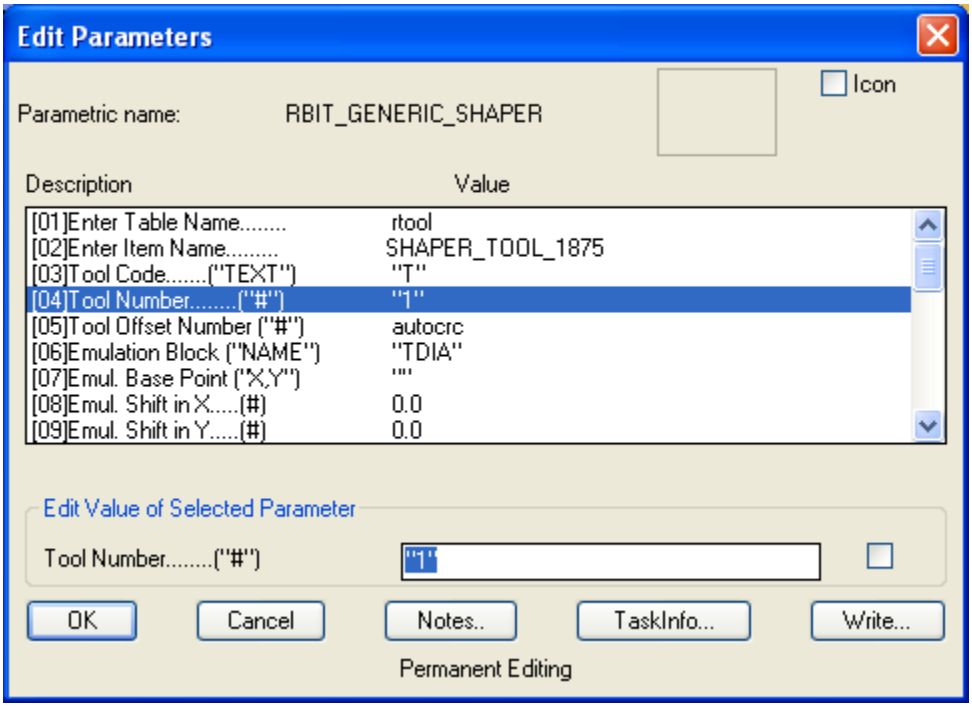
Select YES if you see this notice.

The parameter window should then show the changed value.

© 2011 CIM-TECH.COM, Inc

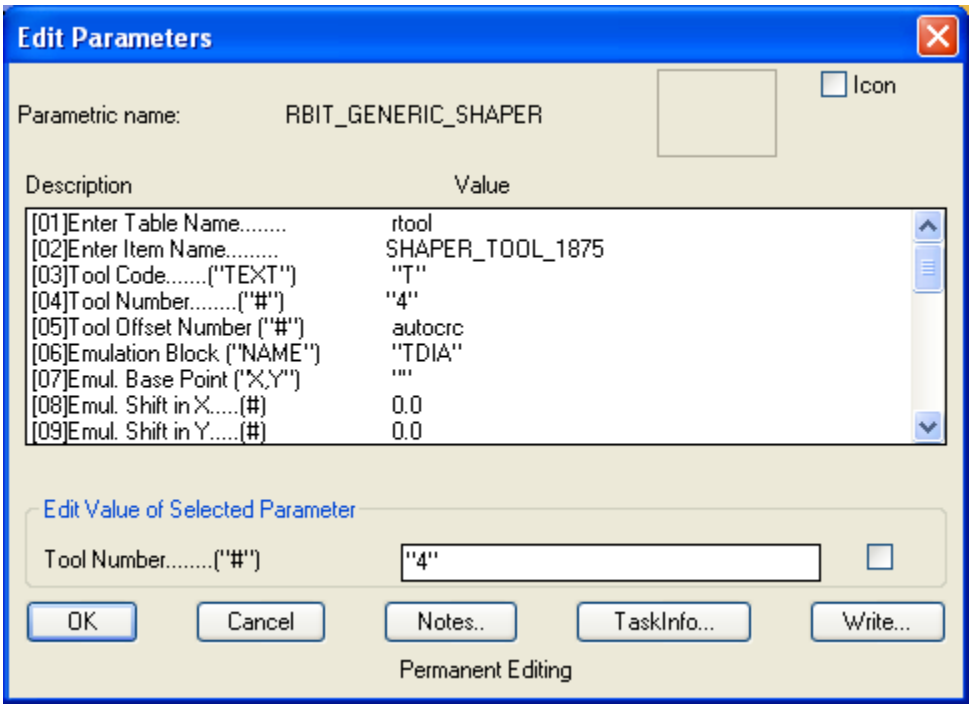


Next we will select the Tool Number (#4 in the list) and change the number in quotes from "1" to "4". Again, select the item with your mouse and it will be available below to edit.

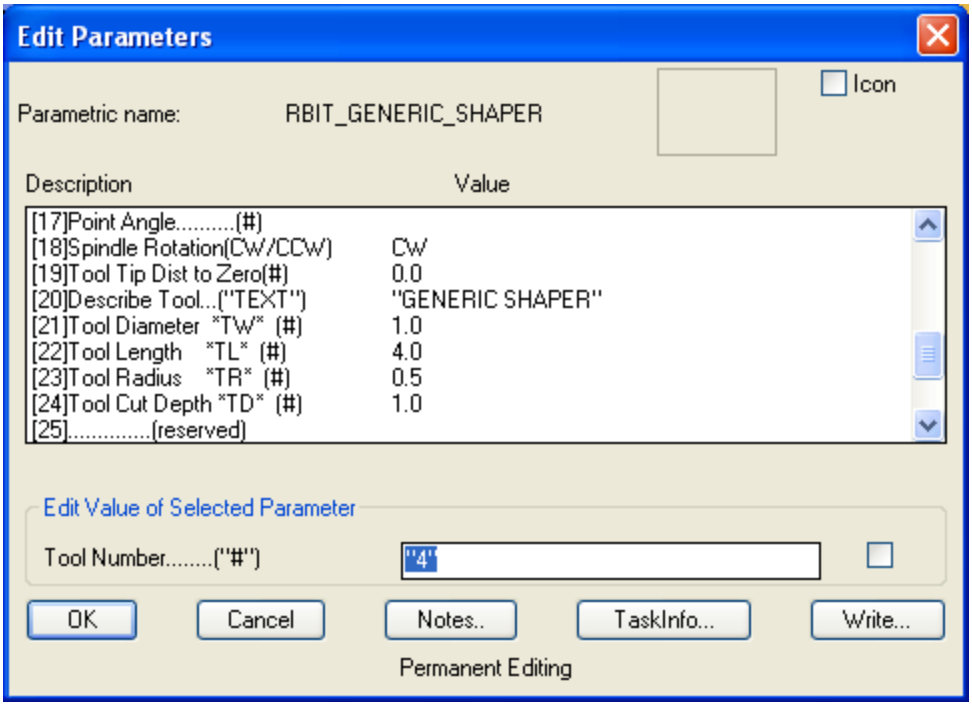


Change the number inside the quotes from "1" to "4" and then press ENTER and you will see the change in the list.

© 2011 CIM-TECH.COM, Inc

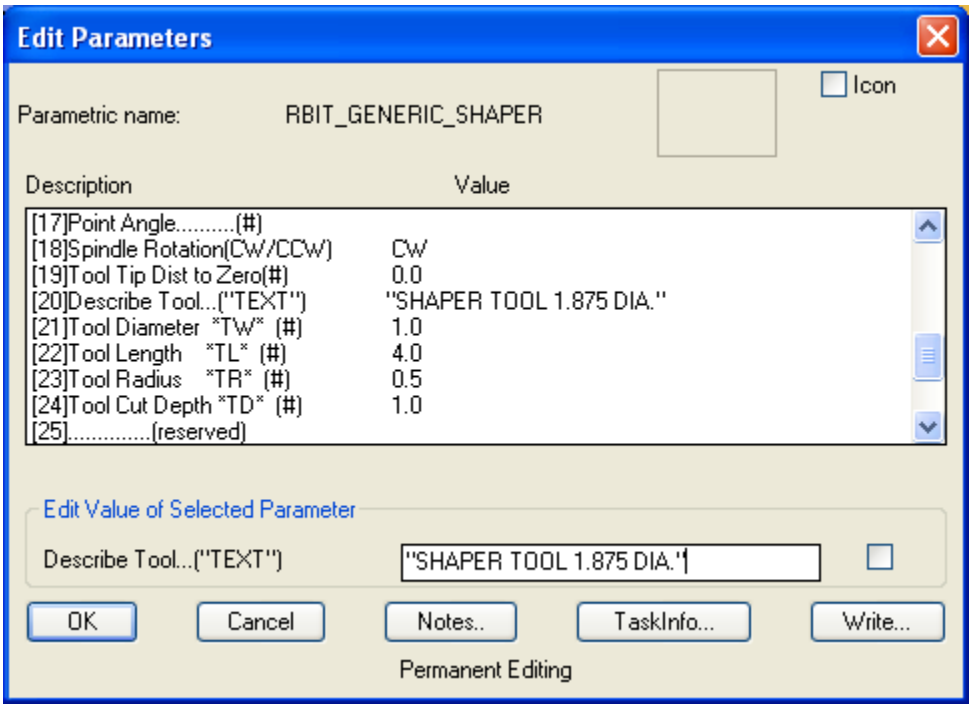


The next item we need to change in this case is the Tool Description (#20 in the list).



Select that and change it from "GENERIC SHAPER" to "SHAPER TOOL 1.875 DIA.", then press ENTER to see the change in the list.

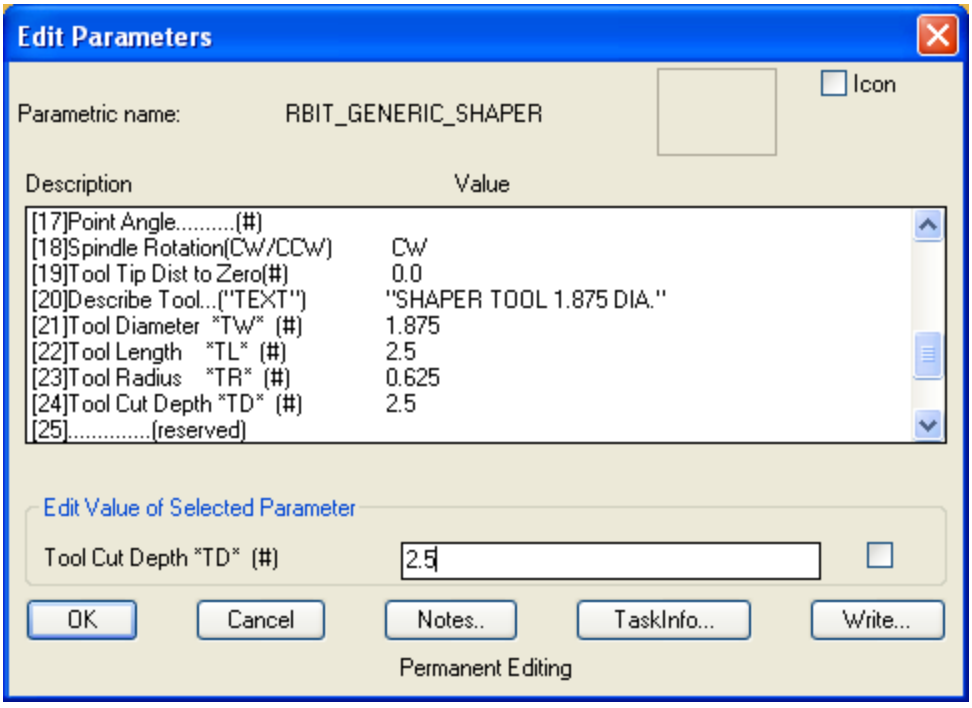
© 2011 CIM-TECH.COM, Inc



We will then change:

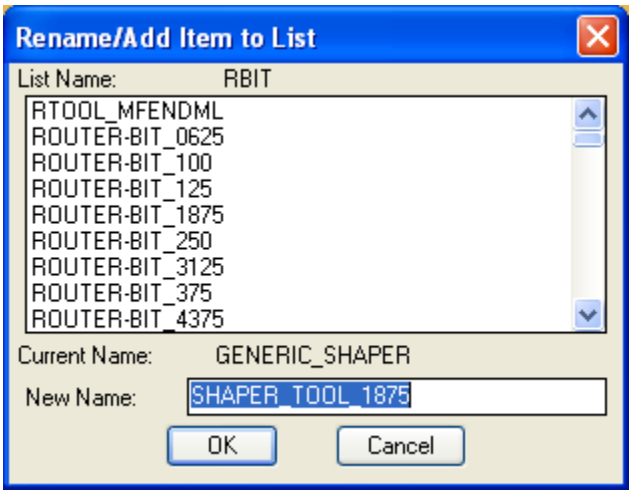
- Tool Diameter to 1.875.
- Tool Length to 2.5
- Tool Radius to .625
- Tool Cut Depth to 2.5

The list should then look like this:

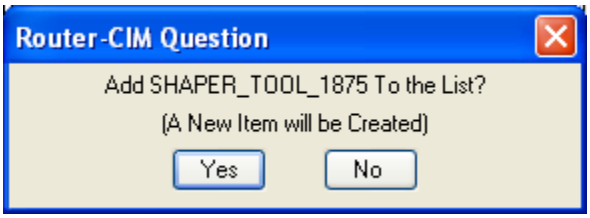


© 2011 CIM-TECH.COM, Inc

Once the changes in this editor are made, select OK to add a new tool to the list. The following window appears.



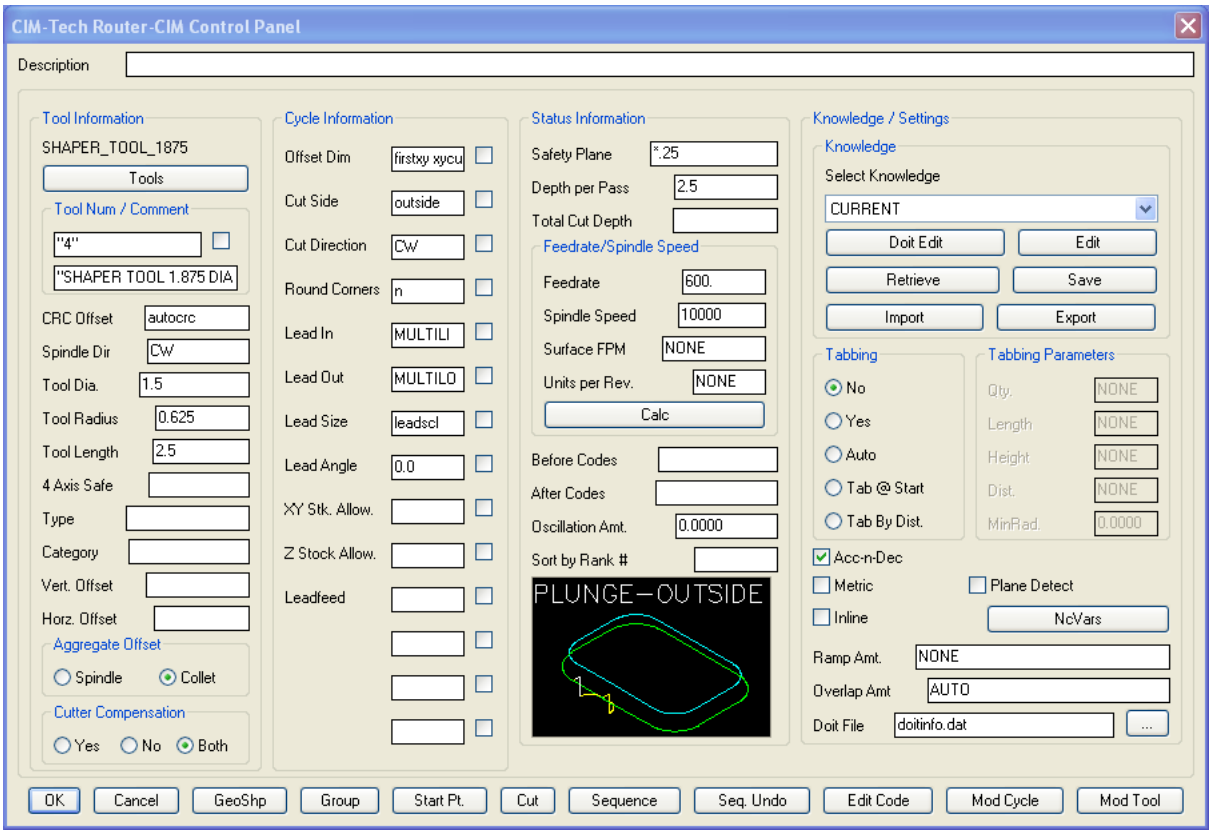
This shows you that you are going to either Rename or Add a new item to the list. In this case it will be a new item.
Select OK.



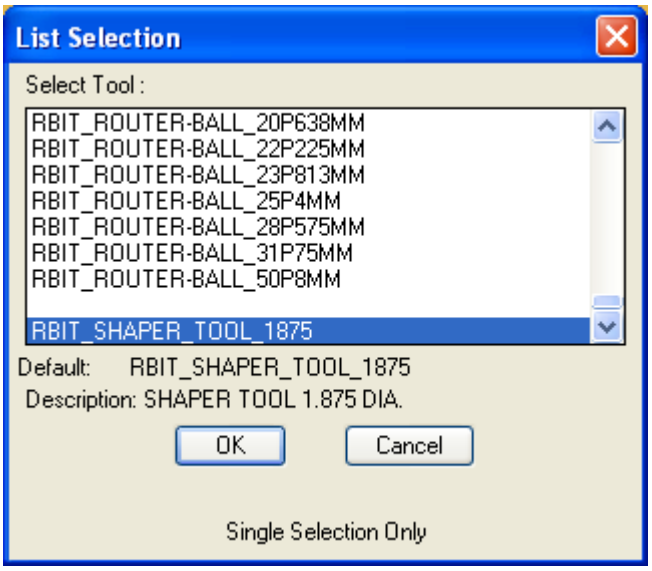
You get a final window asking if you want to add a new item to the list. Select YES.

Then you will see the Control Panel with the new tool displayed.

© 2011 CIM-TECH.COM, Inc



At this point the tool is added to the list, and the list is filed to the hard drive. If you select the tool list now, you can see the new tool at the very bottom of the list.



In the next section we will move that tool up in the list so that it is easier to find.

© 2011 CIM-TECH.COM, Inc

5.3 Moving an Item in the List

To move tools to new locations or organize tools in the Tool List you must use Layout Edit.

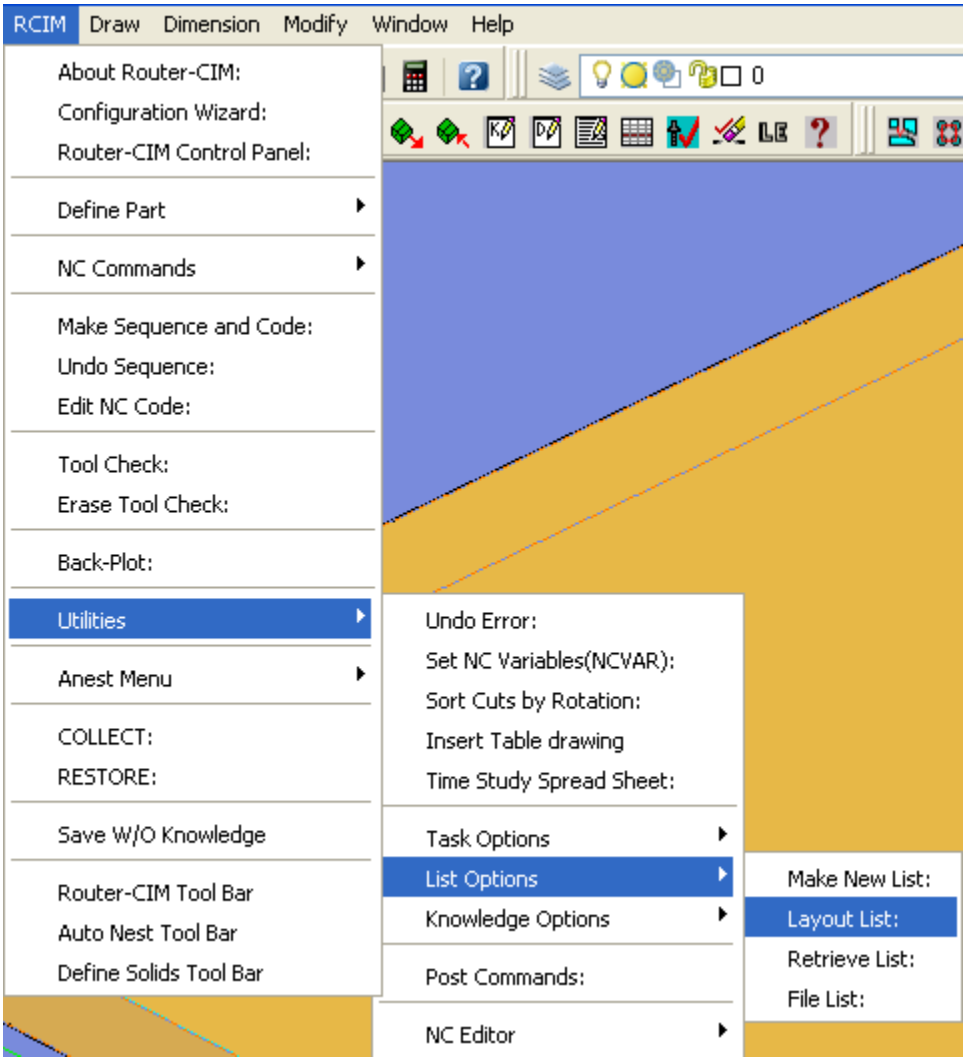
The lists can be re-arranged and annotated so that the list has a sense of grouping items together and commenting as to what the items are in the list. This is accomplished by the "List Editor". It provides for the arrangement of items in the list and the adding of comments in the list.

The List Editor uses the "clip board" concept to move items around in the list. You can take items out of the list and place them on the clip board (Clip command) or place items into the list from the clip board (Paste command). You can additionally edit lines in the list that are not names of items in the list. This provides a way for comments to appear in the list.

Layout Edit

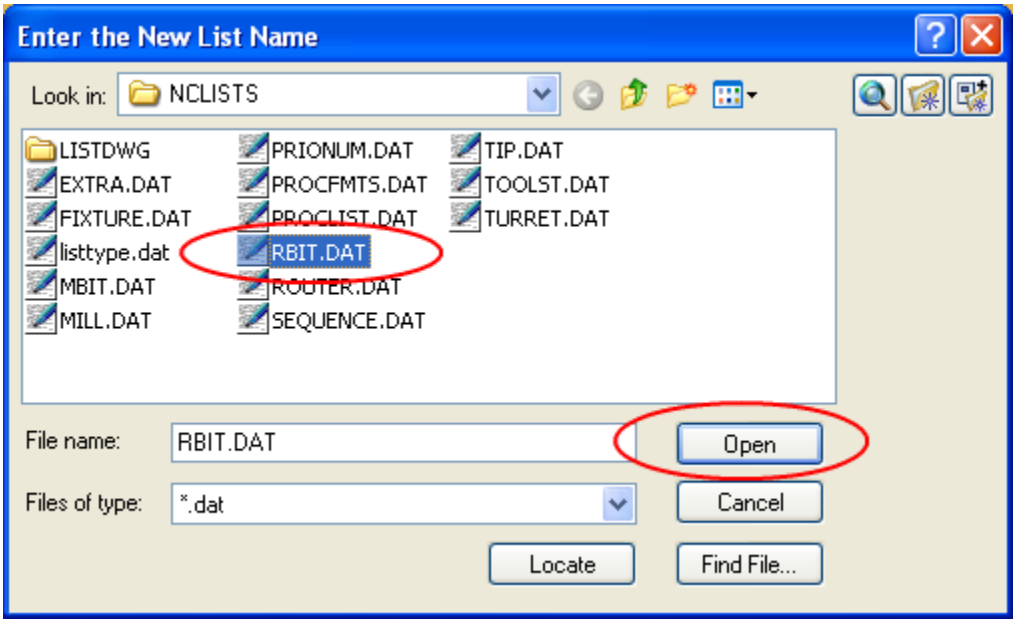
In the last section, we created a new tool and added it to the list.
In this section, we will move that item from the bottom of the list to a location below where the Generic Shaper tool is in the tool list.

Select Layout List from the pulldown menu:



© 2011 CIM-TECH.COM, Inc

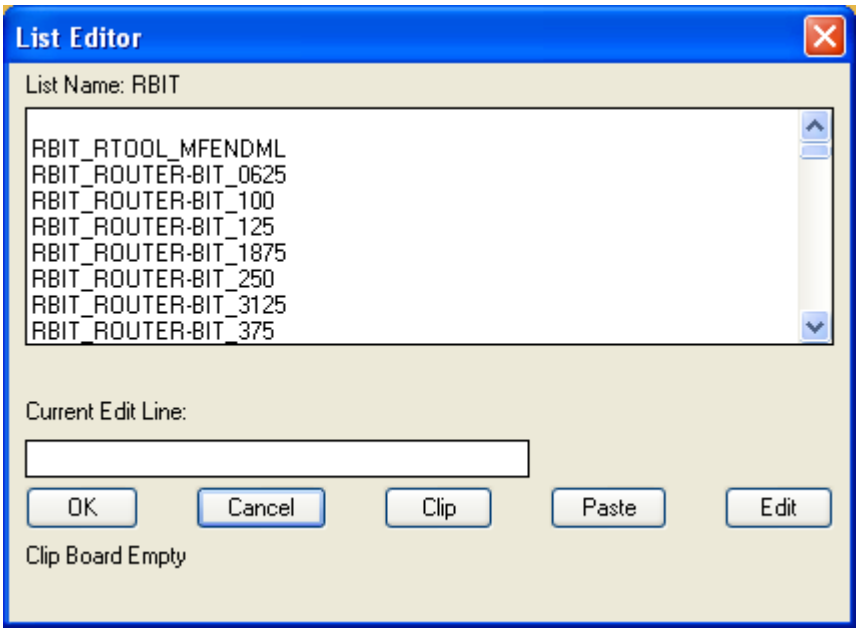
When you select Layout List you will be shown a selection of lists to modify, select RBIT.Dat.



The RBIT.DAT file is the Tool List that Router-CIM uses by default, and the one you will normally edit.

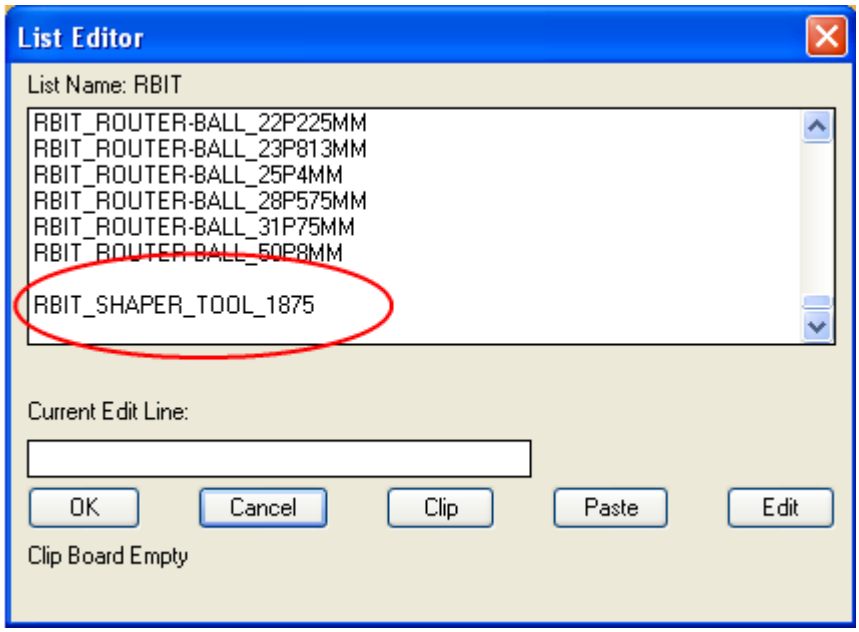
Select Open to continue.

The List Editor will be shown, and you will be looking at the top of the list.

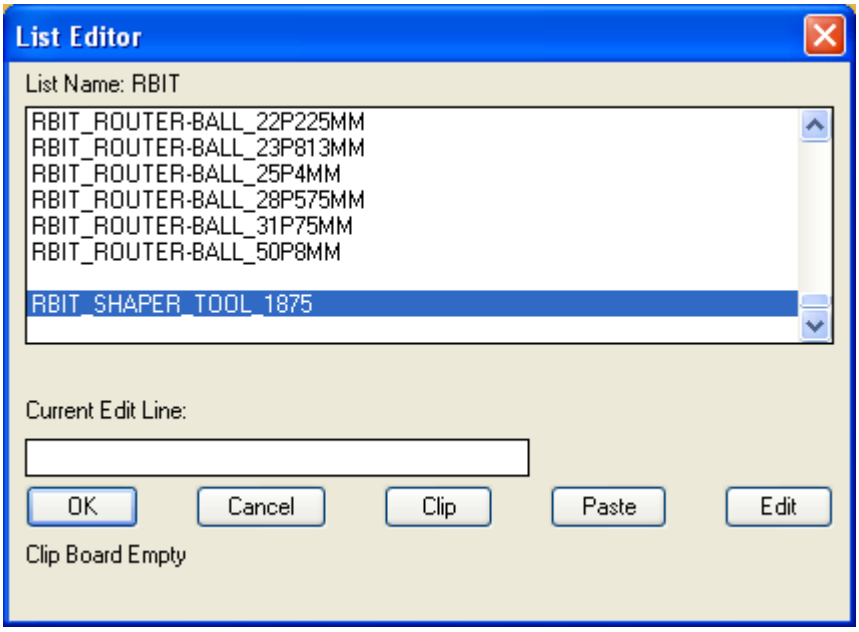


Scroll down to the very bottom to find your new tool...

© 2011 CIM-TECH.COM, Inc

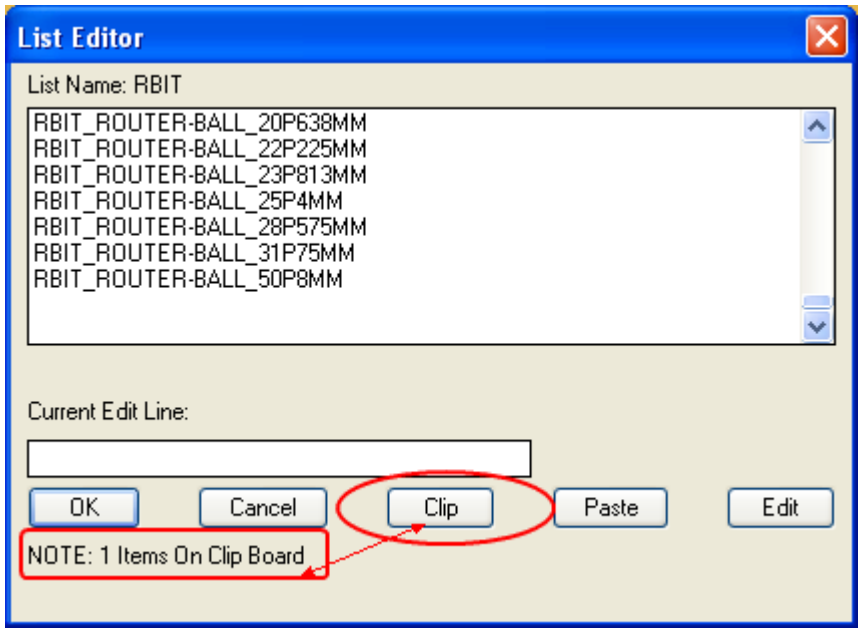


The Clip Board will be empty at this point, so select your new tool...

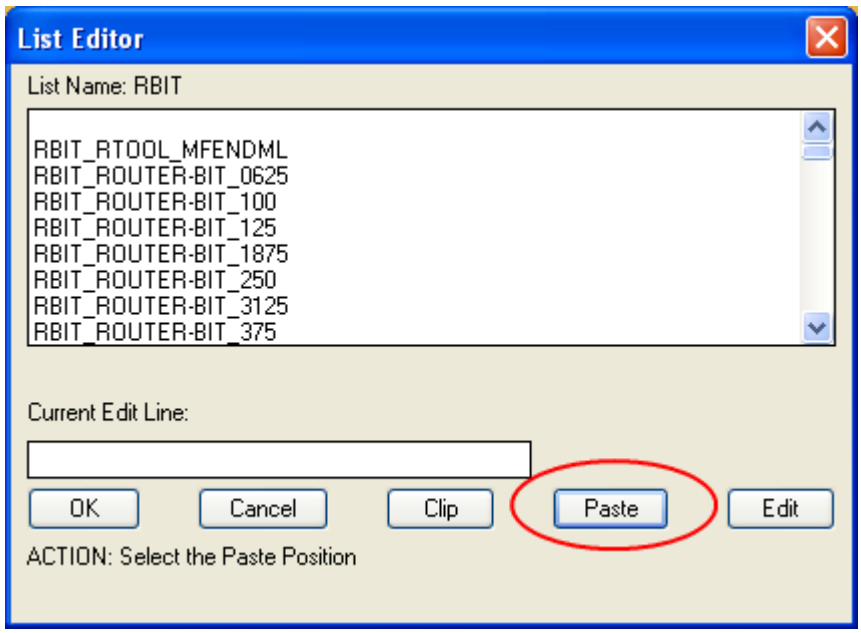


Then pick CLIP.

© 2011 CIM-TECH.COM, Inc



The tool will disappear from the list and the message at the bottom will show that you have a new item on the Clip Board.

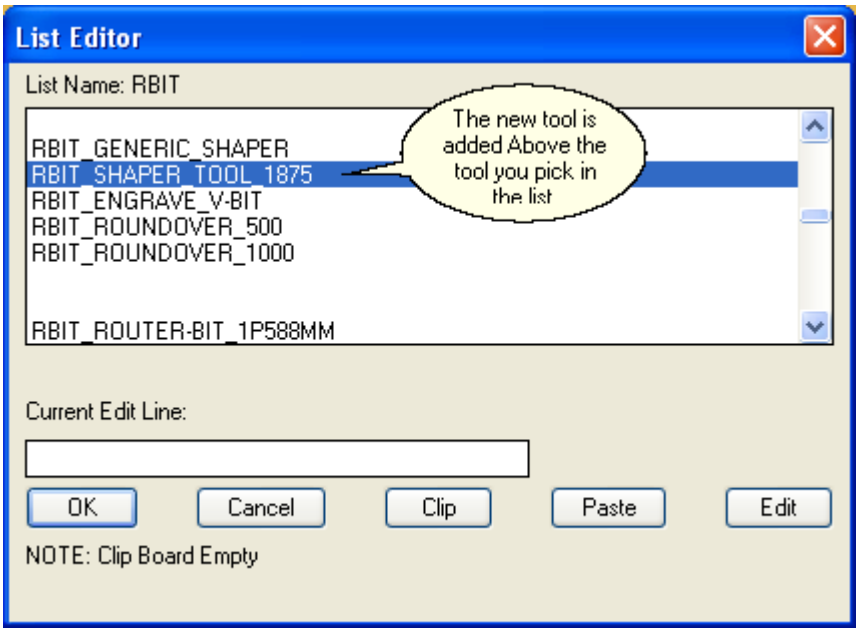


Next pick on PASTE. The list will scroll up to the top.

WARNING ... The next position you pick in the list will force to the tool to be pasted ABOVE that location.

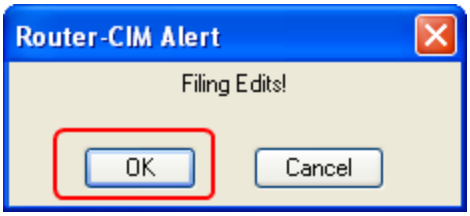
In this case, we scrolled down to where the Generic Shaper was located and selected the RBIT_ENGRAVE_V-BIT in the list.

© 2011 CIM-TECH.COM, Inc



The tool was pasted directly ABOVE that tool.

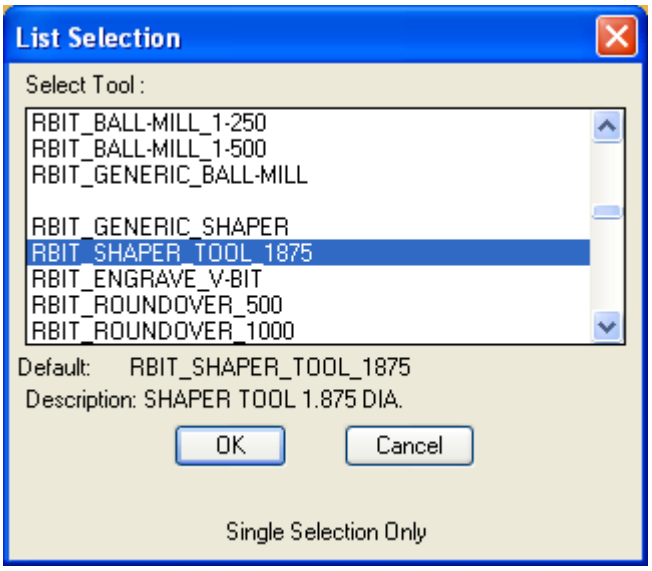
Next select OK to finish the edit.



When the Filing Edits window appears, select OK to make the change permanent to the hard drive RBIT.DAT file.

If you check the tool list from the Control Panel now, you should see your tool in the proper position:

© 2011 CIM-TECH.COM, Inc



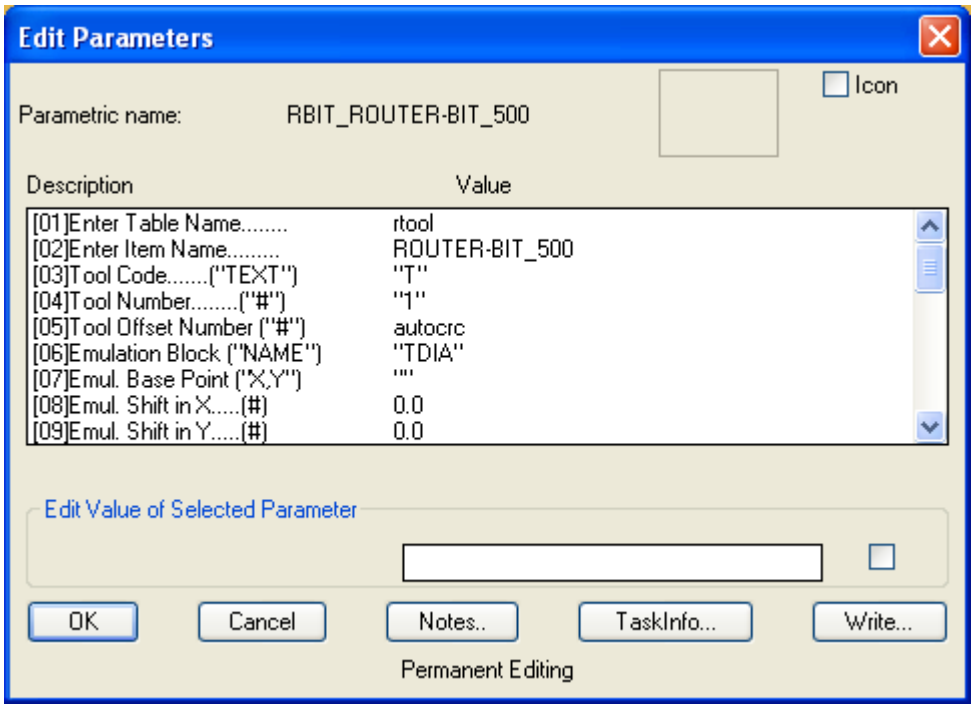
Edit

The Edit feature of the Layout Editor provides for the insertion of comments into the list. This can be used to describe groups of like tools in the list for clarity when searching for the particular tool you need.
Simply highlight the line to edit and press [Edit] from the menu. If a tool name is selected , an error message will appear in the message area stating that the selected item cannot be edited.
In the edit box field the selected line will appear. Make any desired changes in the edit box and then press ENTER when finished. The edited line will appear in the tool list at the selected location.
If you need more lines for comments, clip and paste a blank line at the desired location and then edit the blank line to add your comment.

5.4 Changing Tool Parameters

When changing the tool parameters, it is important to know the definition of the parameters you need to change to get the desired results. Not all of these parameters will have to be changed with every tool, but you should be familiar with the settings and their definitions.

© 2011 CIM-TECH.COM, Inc



(01) ENTER TABLE NAME...This is the table name of the tool list and normally is never changed.

(02) ENTER ITEM NAME... This is the name of the tool as it will appear on the tool list. You should try to keep the name format of similar tools similar to each other, so they are easy to find in the list (refer to current list for reference). You cannot use spaces, or special characters in this version of the tool name. Only Underscore and Hyphen are allowed.

(03) TOOL CODE... This is the code put out for individual tools, usually the tool definitions are accounted for in the post processor so changes are not necessary. Unless you are directed to change this parameter in your application notes, you should not change this parameter at all.

(04) TOOL NUMBER...This is the default number of the tool as it appears in the Control Panel. If you have a tool that is always the same number, place the number here so that it is the default. The number must be set inside quotation marks in order to generate the correct code.

(05) TOOL OFFSET NUMBER... This is the same as the offset number on the Control Panel. Normally you will not need to change the field from 'autocrc' unless you wish to have a specific offset number for a particular tool. This will output as the "D" value with the number you specify in this field.

(06) - (17) EMULATION BLOCK NAME, AND EMULATION XY SCALEs... These values control the block to insert and the size of the Tool Check tool. You can show a detailed Tool Check by drawing a 3D tool and storing it as a Block in your default or current drawing. Put the block name in the Emult Block Name field. During the Tool Check command, that block will be inserted. A task called Tdsize will use the tool's diameter for the emulation X and Y scale factors. These are for advanced use of the Tool Check function and are not normally used for regular programming.

(18) TOOL ROTATION... This is the same field that appears in the Control Panel as Spindle Dir. If you have counter-clockwise spindles, you should change this field when you are using CCW bits in your spindle. This field generates an M03 or M04 code for clockwise and counter-clockwise normally.

© 2011 CIM-TECH.COM, Inc

(19) TOOL TIP DISTANCE TO ZERO... This is the distance that you need to compensate for if you have a tool whose tip is not flat. If you touch off the tip of the tool to zero, but there is an angled or rounded surface to the end of the tool, such as a drill bit or a ball mill, you can input the distance to the desired cutting surface here.

(20) DESCRIBE TOOL... This will be the description of the current tool as it will appear in the NC code file (of your router supports comments). This field can hold any description or character, however it must be enclosed in quotation marks.

(21) TOOL DIAMETER...This is the field that appears in the Control Panel as Tool Diam. The diameter of the tool input here should be the outside diameter of the carbide.

(22) TOOL LENGTH... This is the same field that appears in the Control Panel, if you wish to set the default Tool Length input a number in this field.

(23) TOOL RADIUS...This is the same field that appears in the Control Panel as Tool Rad.

(24) TOOL CUT DEPTH... This is the same field that appears in the Control Panel as the Depth/Pass field. This is the maximum depth you wish the tool to be able to cut in the Z direction in any single pass, thus extending tool life.

(25) - (26) These fields are reserved and should not be edited in normal use.

(27) This field should normally be set to "Y" and should not be edited in normal use.

(28) MACHINE CUTTER COMPENSATION... This is the field where you can specify the default value for the Cutter Comp. Selection in the Control Panel. Valid entries are "Y", "N", or "B" for Yes, No, or Both. Please refer to the section on Cutter Compensation for further details.

(29) FEED DISTANCE TO MATERIAL... This is the point at which your machine tool switches from rapid traverse to programmed feed rate. This field is set to .1 and should not be set any lower than this! The machine tool will rapid to this point and then the next line of code will have to use a feed rate code if it is going to enter the material.

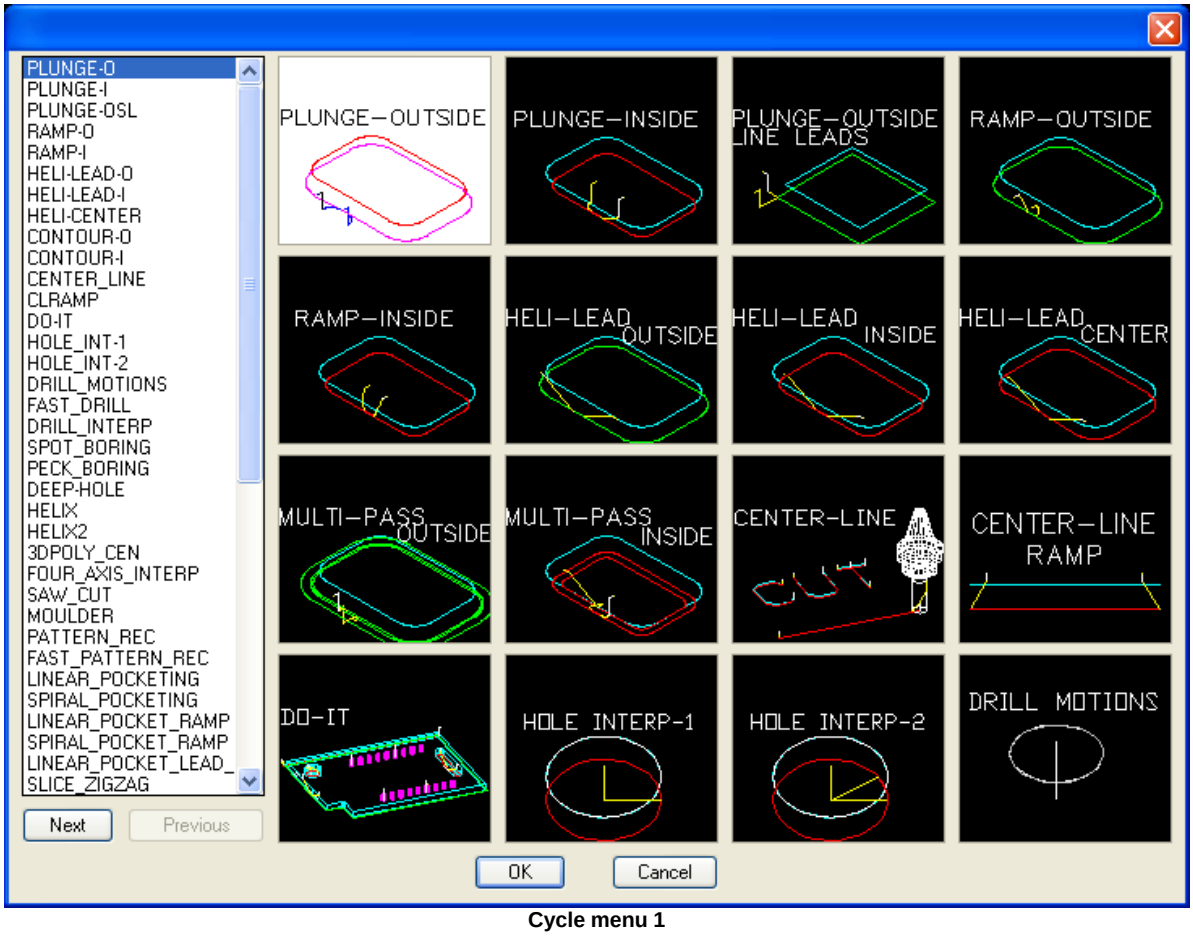
© 2011 CIM-TECH.COM, Inc

6 Cutting Cycles

Cycle Selection

Sixty cutting cycles are available in Router-CIM, each designed for a slightly different purpose. Cycles are selected by clicking the cycle icon in the Control Panel. The Cycle Selection screen will appear. Select a cycle by either double-clicking to make it the current cycle, or by clicking a cycle once and clicking OK on the bottom of the screen. After picking a cycle from this list you will return to the Control Panel, where the center column shows the parameters required by that cycle.

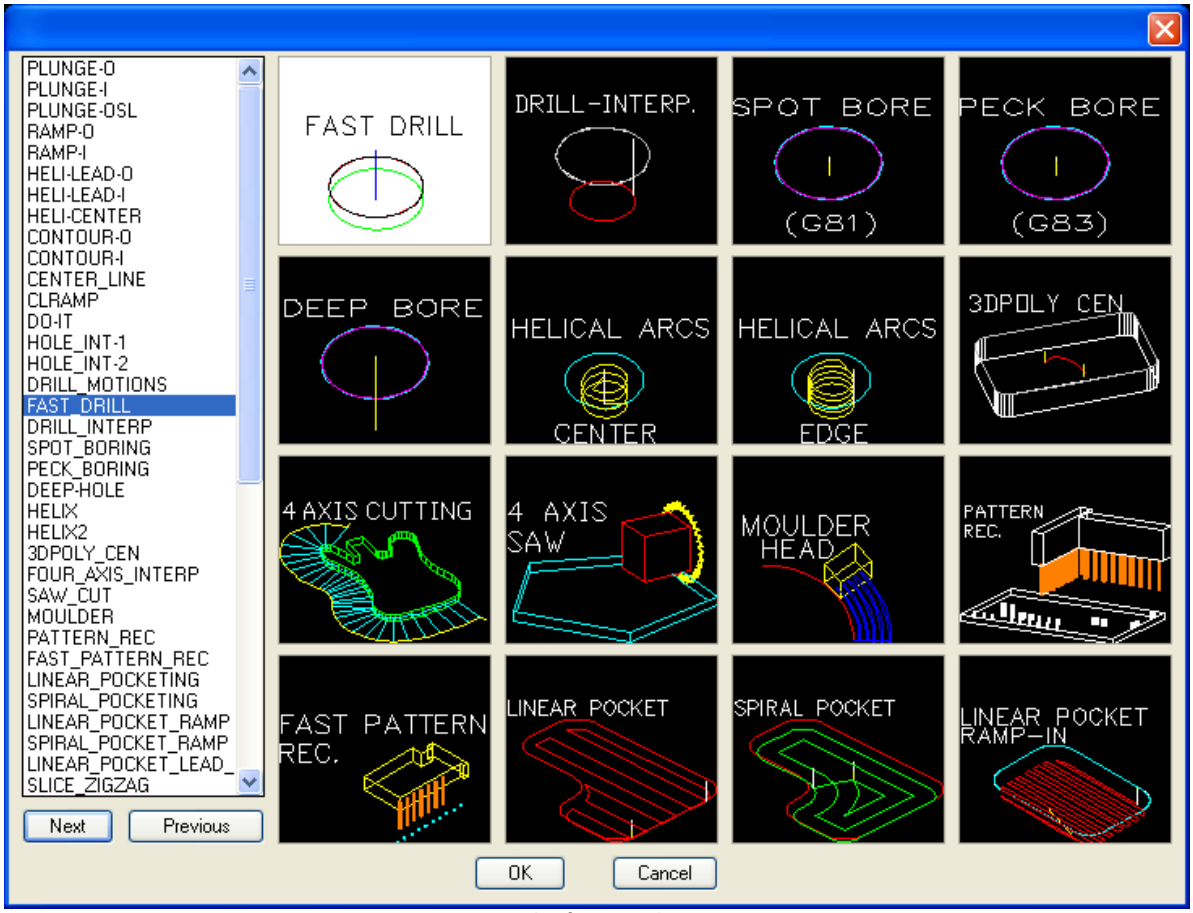
Each of the four Cycle screens can show up to 16 cycles. You may toggle back and forth between the four cycle screens with the Next and Previous buttons.



Cycle menu 1

© 2011 CIM-TECH.COM, Inc

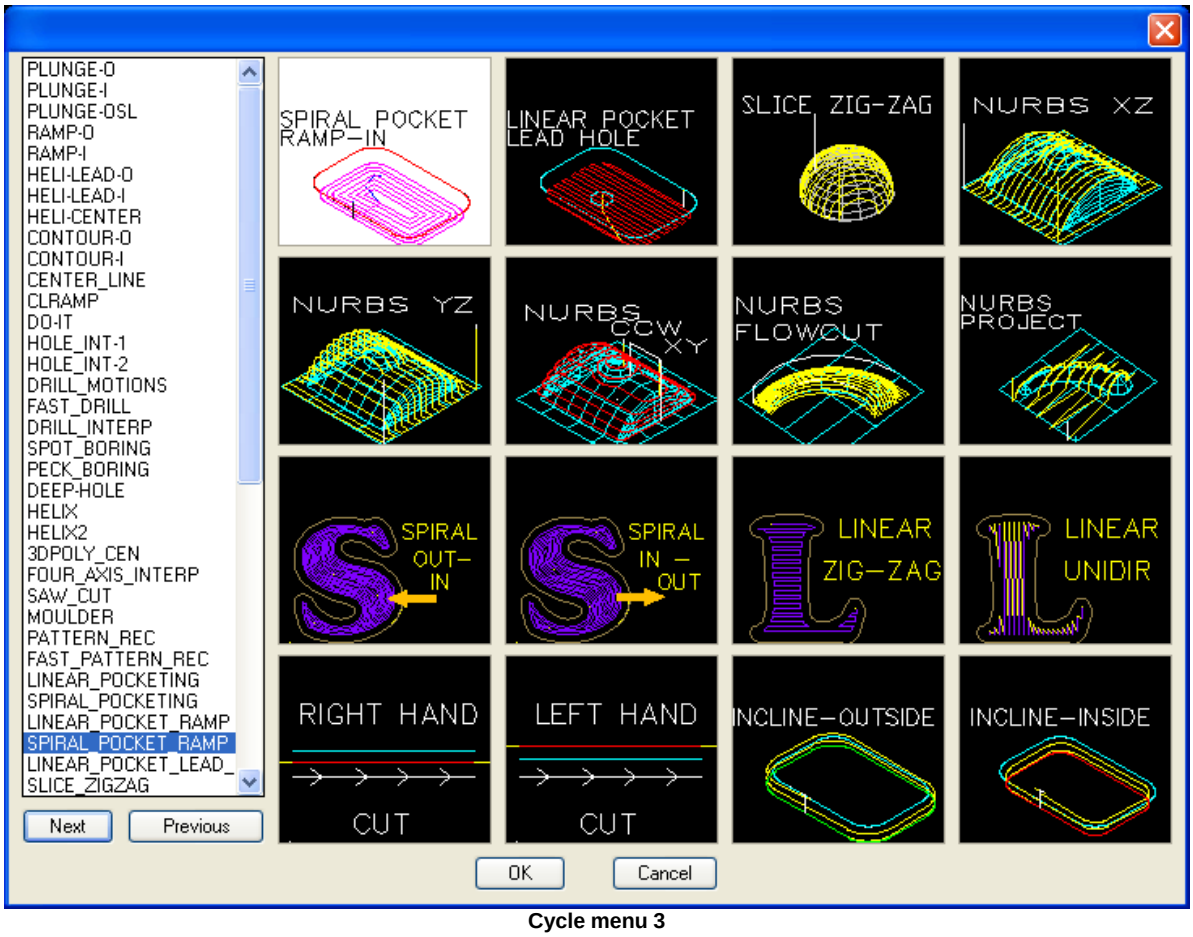
The second Cycle screen is shown below:



Cycle menu 2

© 2011 CIM-TECH.COM, Inc

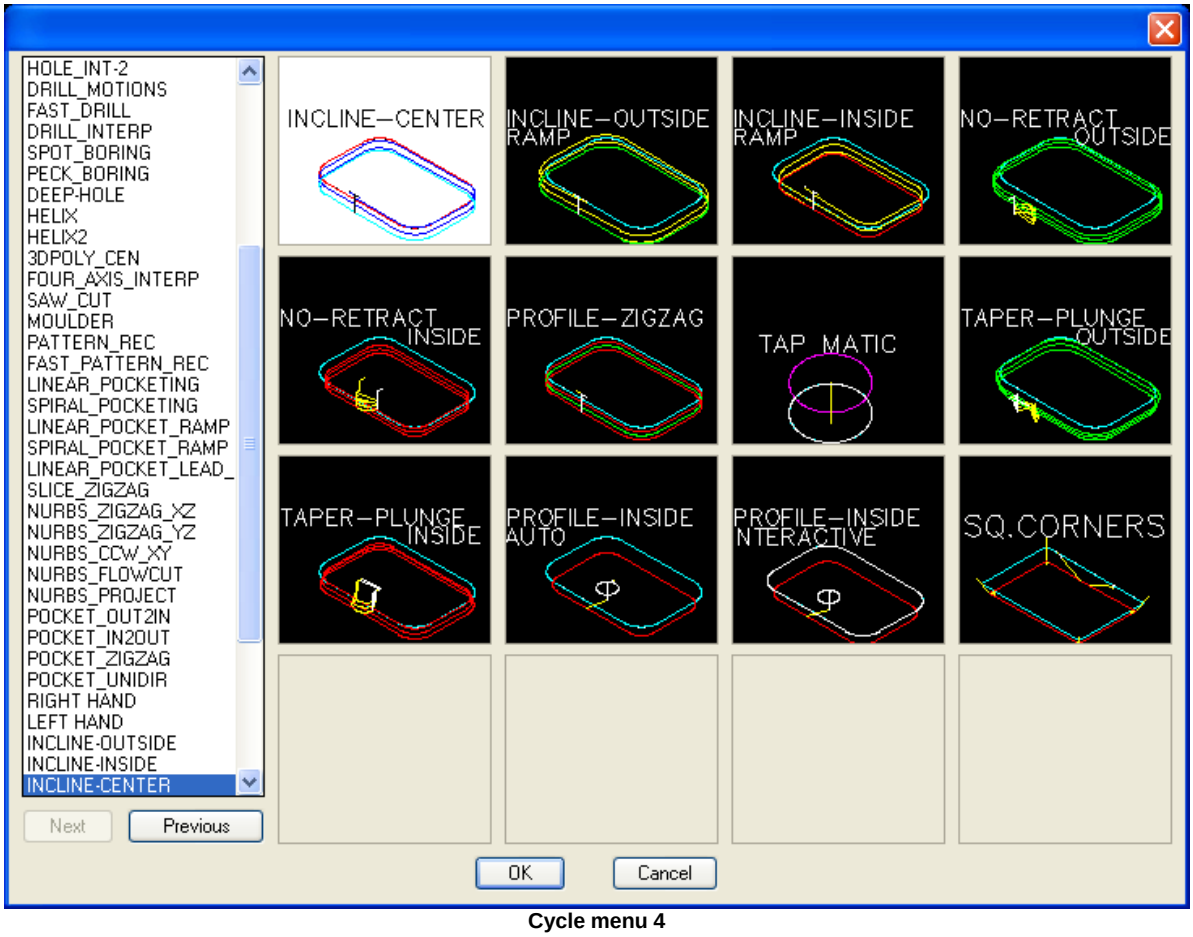
The third Cycle screen is shown below:



Cycle menu 3

© 2011 CIM-TECH.COM, Inc

The fourth Cycle screen is shown below:



Cycle menu 4

To understand the data that must be input to make each cycle work correctly, we need to analyze each cycle along with the Cycle Information column. The cycles may be broken into common groups which require similar inputs in the Cycle Information column.

© 2011 CIM-TECH.COM, Inc

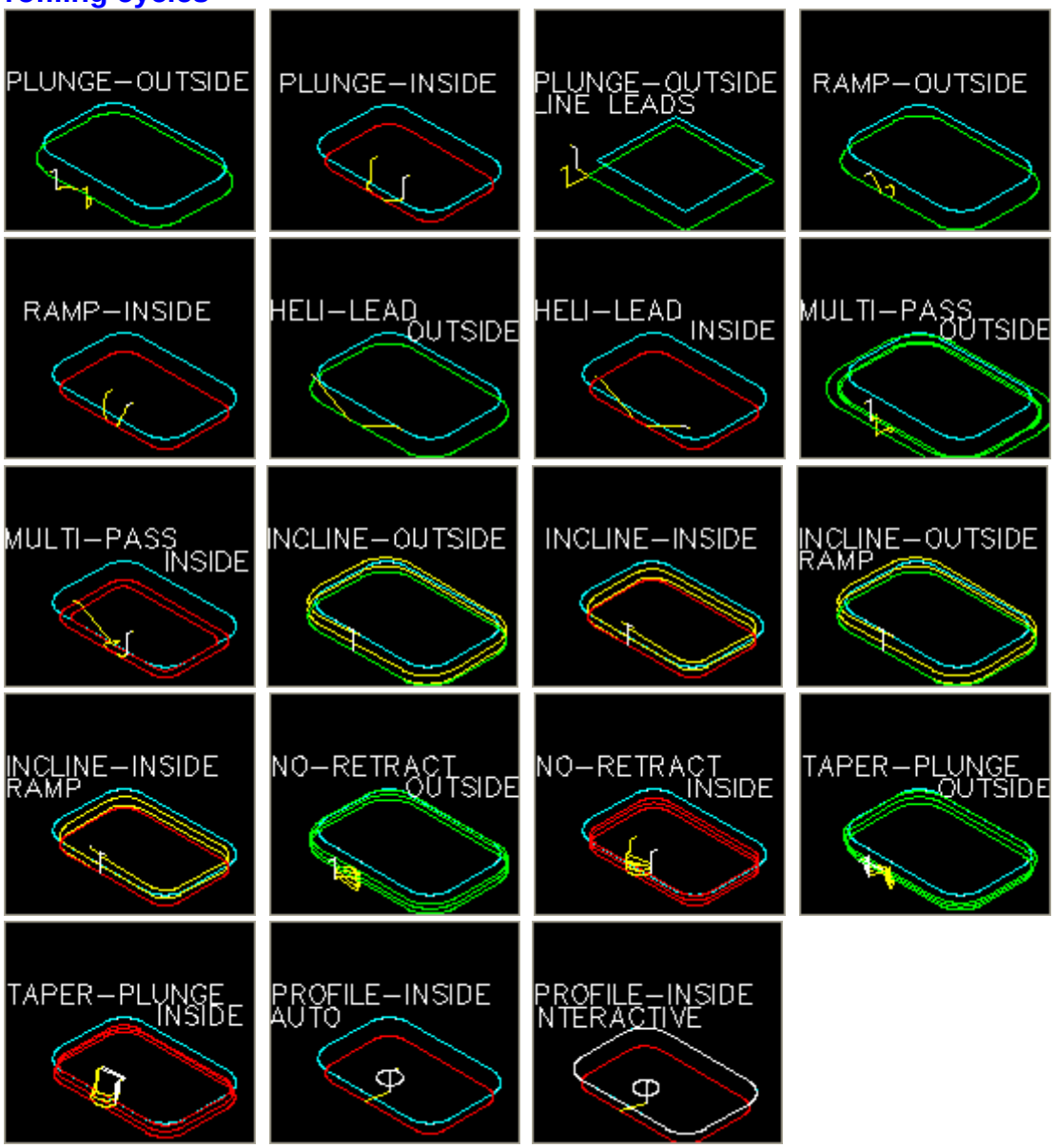
6.1 Cycle Descriptions

Each of the cut cycles in Router-CIM have particular settings to allow their use with minimal setup. Still, some explanation of these cycles is necessary to enable a full understanding of their parameters.

Each of the parameters available for each cycle is listed and the possible entries shown.

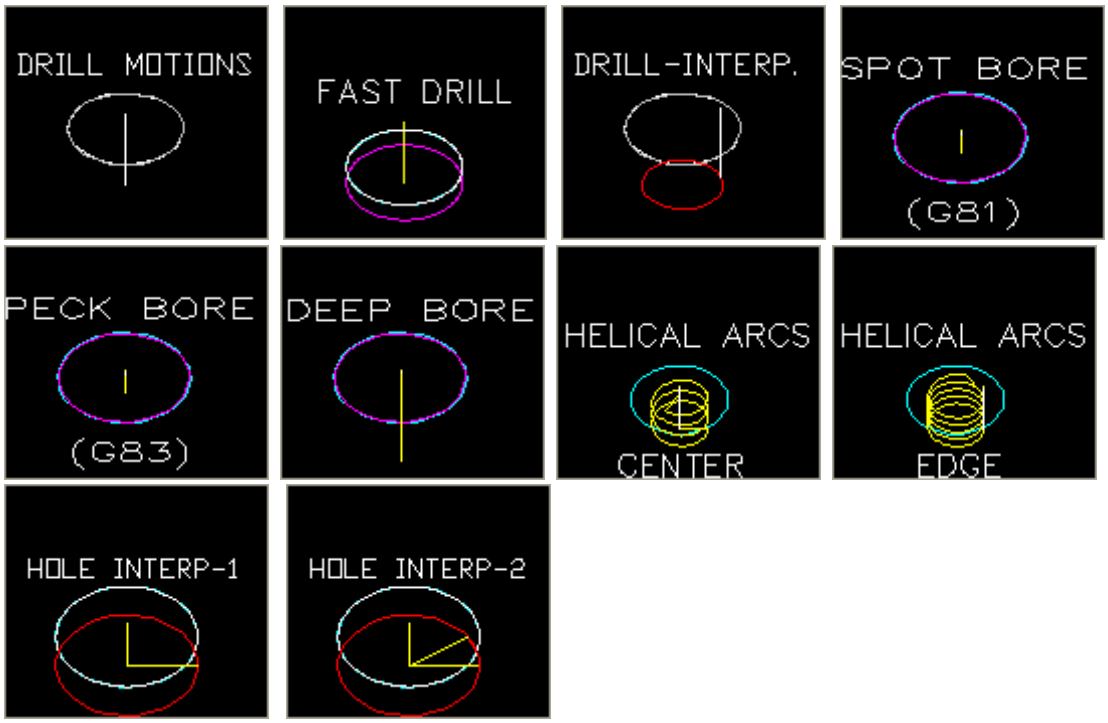
The Cycles are listed according to their functionality. Profiling cycles, Drilling and Hole Cutting cycles, Pocketing cycles, Open Shape cycles, NURBS cycles, and then Specialty cycles.

Profiling cycles

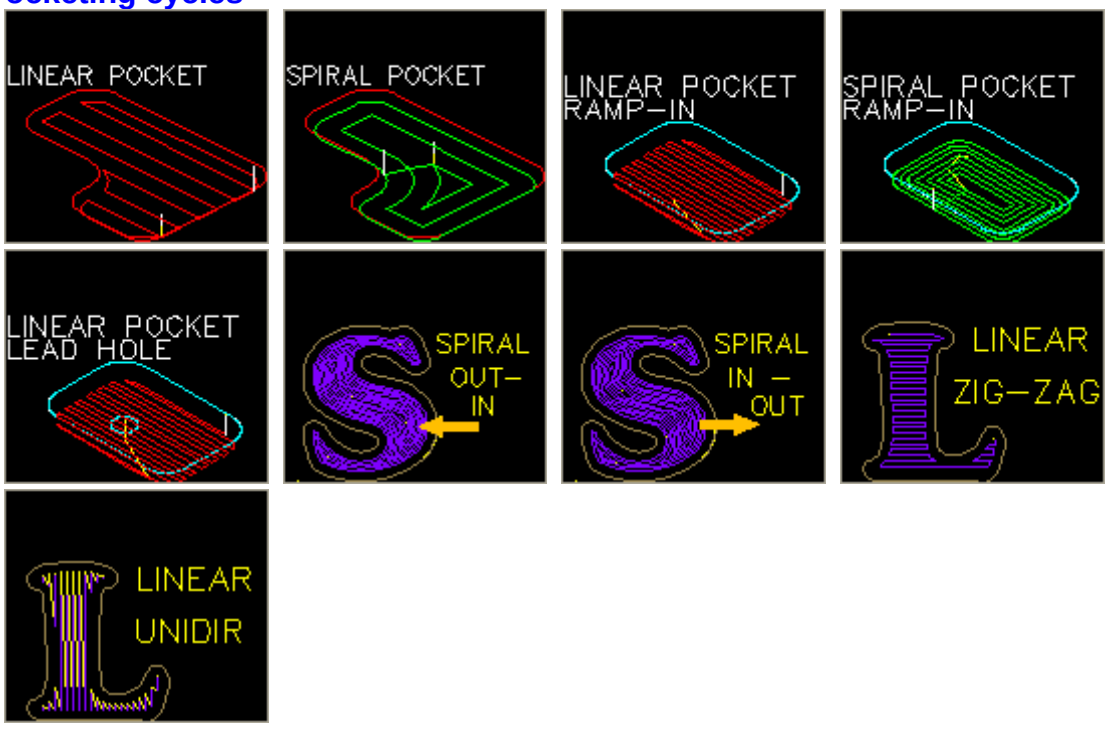


Drilling and Hole Cutting cycles

© 2011 CIM-TECH.COM, Inc

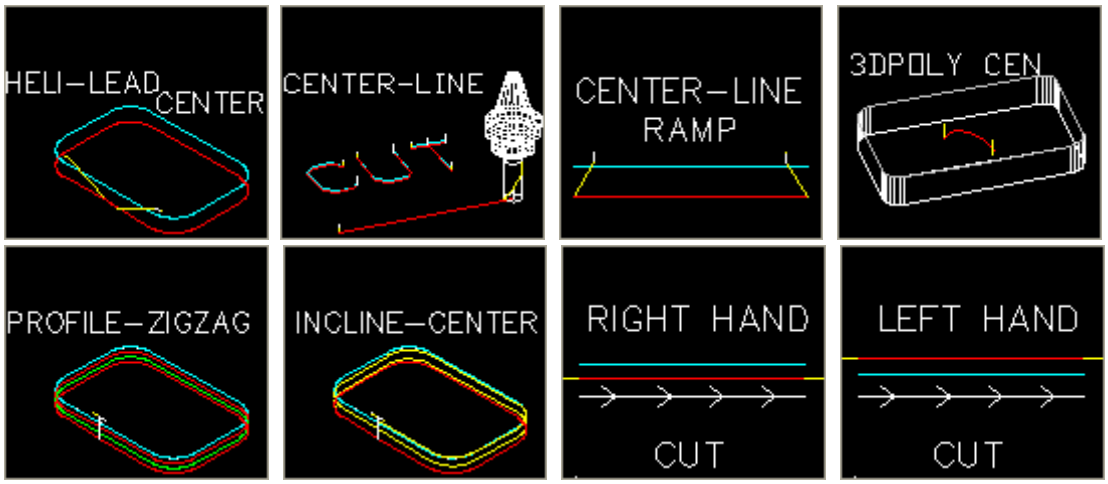


Pocketing cycles

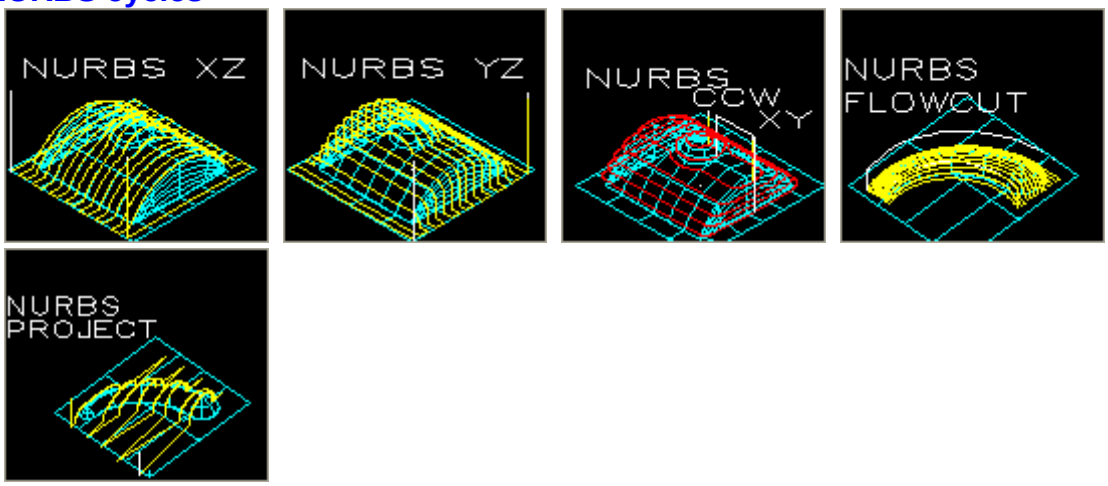


Center Line and Open Shape cycles

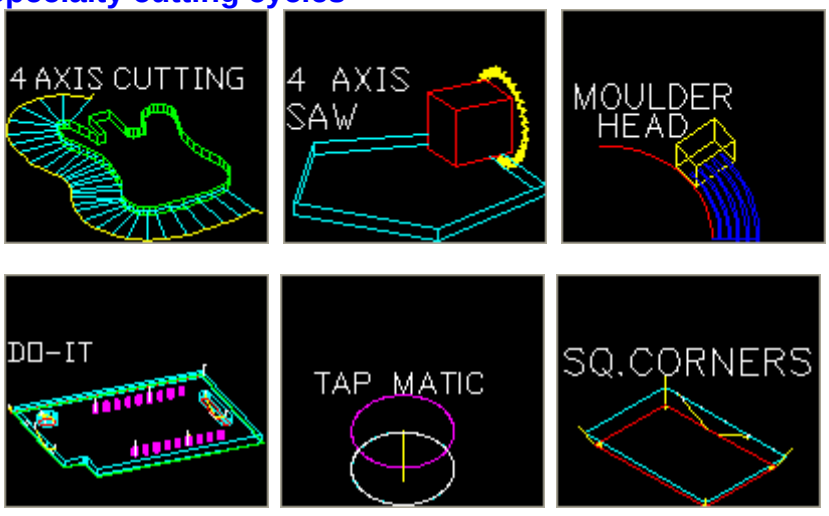
© 2011 CIM-TECH.COM, Inc



NURBS cycles



Specialty cutting cycles



© 2011 CIM-TECH.COM, Inc

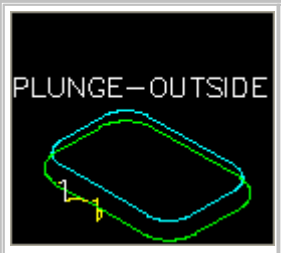
6.1.1 Profiling Cycles

Profiling cycles are Router-CIM cut cycles that are meant to cut a closed shape either on the outside or the inside. These cycles are only meant for closed shapes.
There are several types of profile cuts available to suit a wide variety of tool and material conditions. Each cycle has parameters that control aspects of the cut that you may change in order to manipulate the cut cycle to fit your particular needs.

In the following section, the cut cycles will be explained and each of the parameters available will be described in some detail. At the end of the section there are one or more examples of the use of the cut cycle with the settings used to make the tool path.

The example parts used in each of the sections that follow are available to download to follow the descriptions.
You may download the files from the links at the end of each section.

6.1.1.1 Plunge-Outside

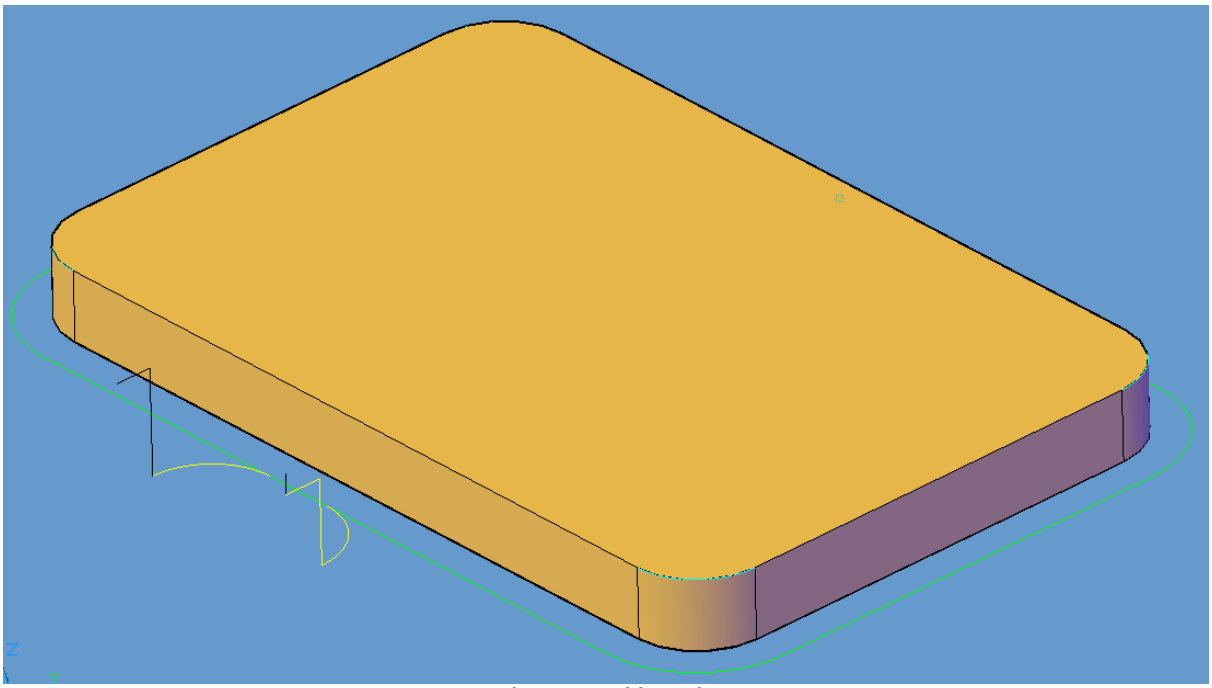


The Plunge-Outside profile cycle will start at the Z height of the Safety Plane, plunge to the point set by Total Cut Depth or Depth per Pass and then make a 90° lead-in arc to the start point, cut the profile shape on the outside back to the start point, then overlap the start point, make a 90° lead-out arc and retract (in rapid) back to the safety plane in Z.
The size of the arc lead-in and lead-out are set by the size of the tool selected and may be overridden by the user as necessary. The overlap amount is also able to be modified. There are special parameters to control the lead-in feedrate separately from the cut feedrate, and the lead out move is done in Rapid Traverse (as there is no material there typically.

Each of the parameters that control the cycle are explained in more detail below.

In use the cycle looks like this:

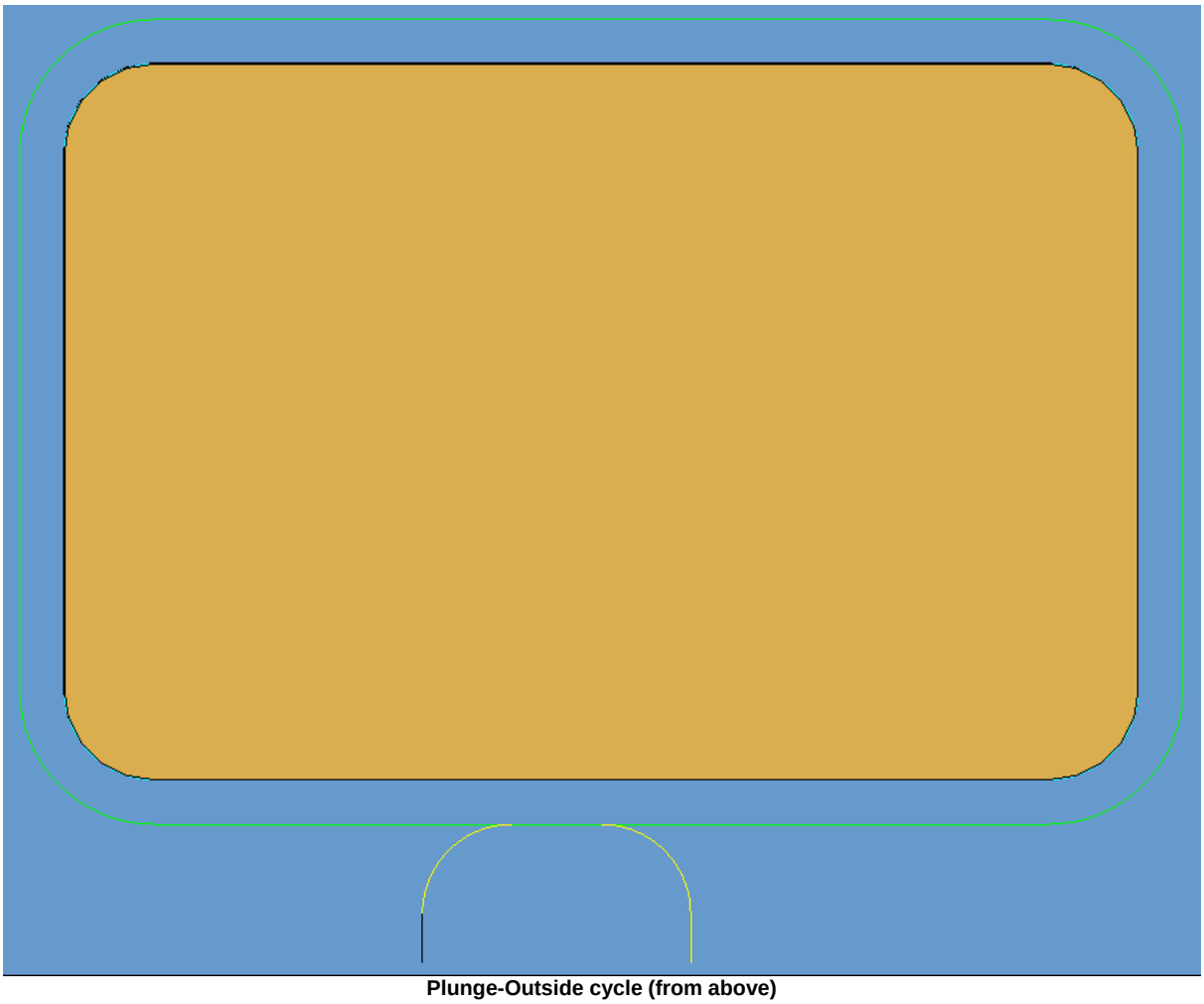
© 2011 CIM-TECH.COM, Inc



Plunge-Outside cycle

© 2011 CIM-TECH.COM, Inc

From the top the cycle looks like this:

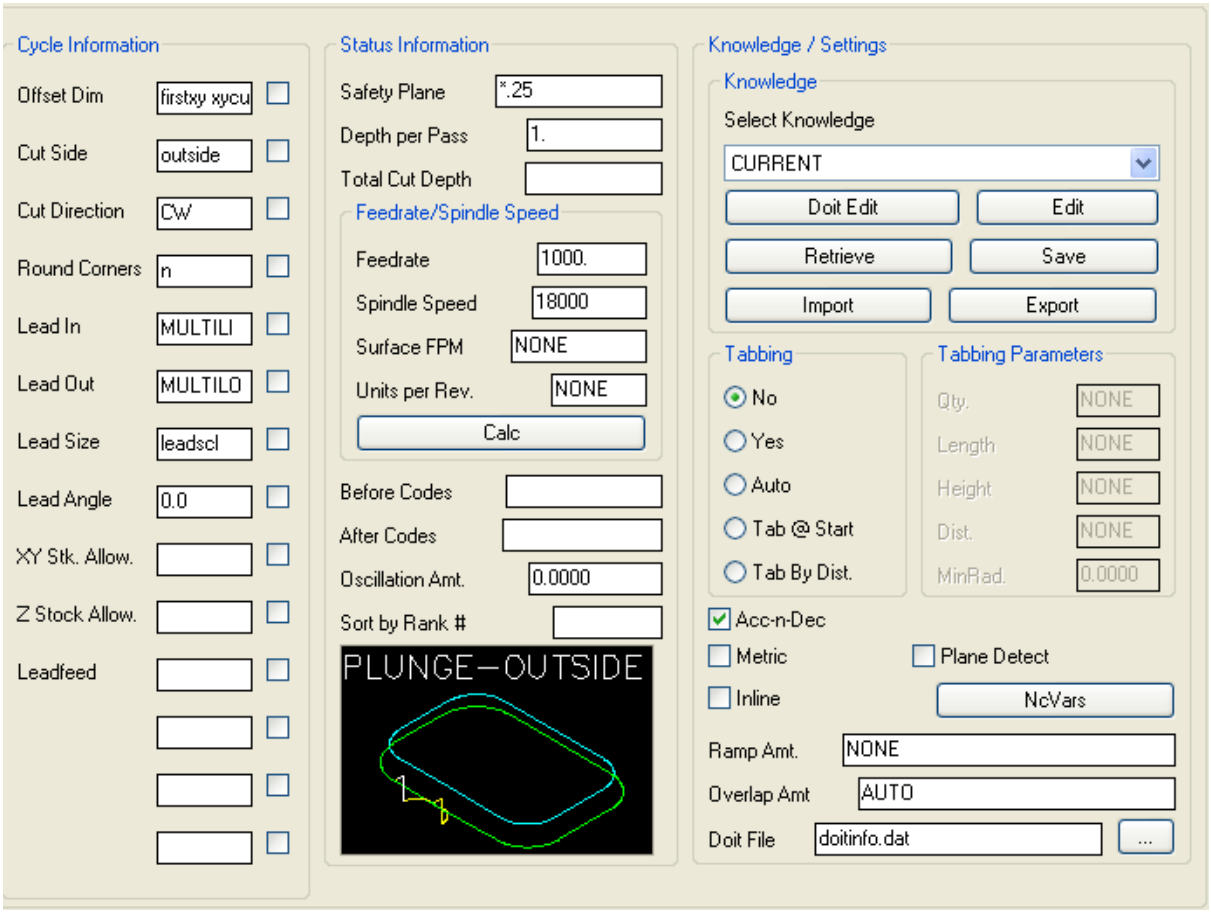


Plunge-Outside cycle (from above)

© 2011 CIM-TECH.COM, Inc

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:



Plunge-Outside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See Offset

© 2011 CIM-TECH.COM, Inc

Dim for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

© 2011 CIM-TECH.COM, Inc

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.
Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).
Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

© 2011 CIM-TECH.COM, Inc

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth
The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

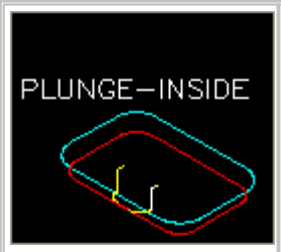
See the Overlap Amt section for more information.

© 2011 CIM-TECH.COM, Inc

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

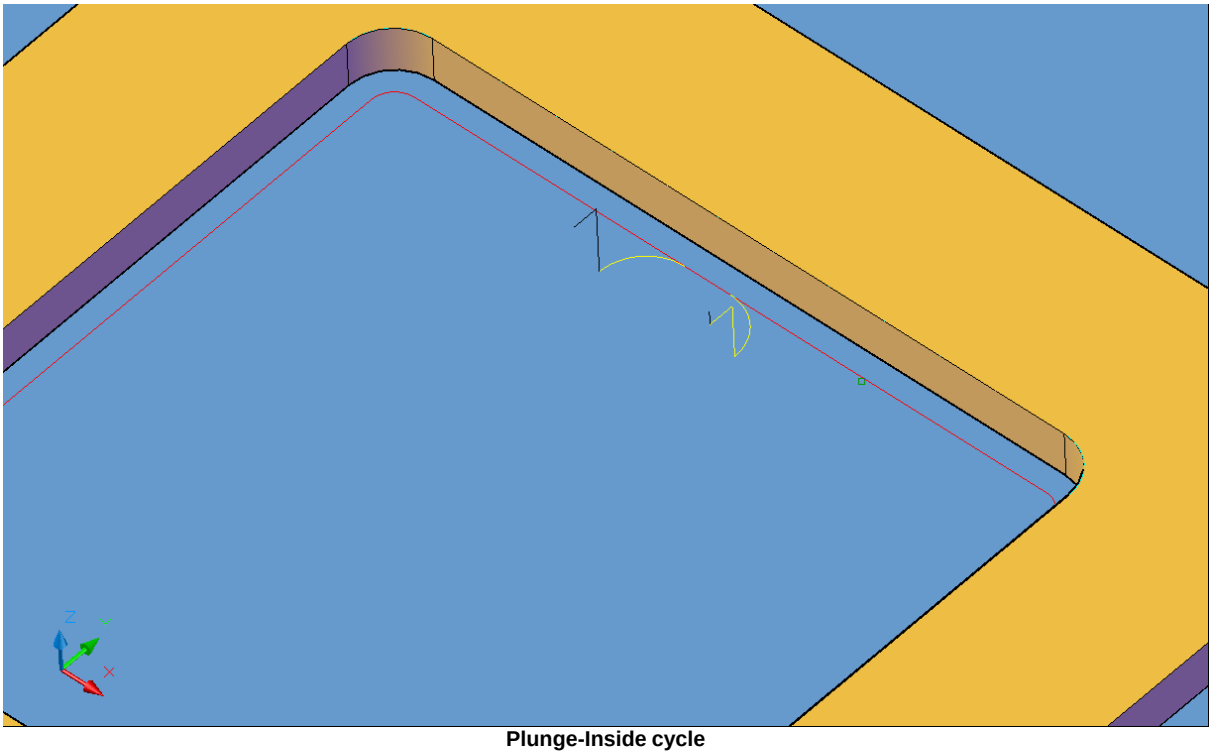
6.1.1.2 Plunge-Inside



The Plunge-Inside profile cycle will start at the Z height of the Safety Plane, plunge to the point set by Total Cut Depth or Depth per Pass and then make a 90° lead-in arc to the start point, cut the profile shape on the inside back to the start point, then overlap the start point, make a 90° lead-out arc and retract (in rapid) back to the safety plane in Z.
The size of the arc lead-in and lead-out are set by the size of the tool selected and may be overridden by the user as necessary. The overlap amount is also able to be modified. There are special parameters to control the lead-in feedrate separately from the cut feedrate, and the lead out move is done in Rapid Traverse (as there is no material there typically.

Each of the parameters that control the cycle are explained in more detail below.

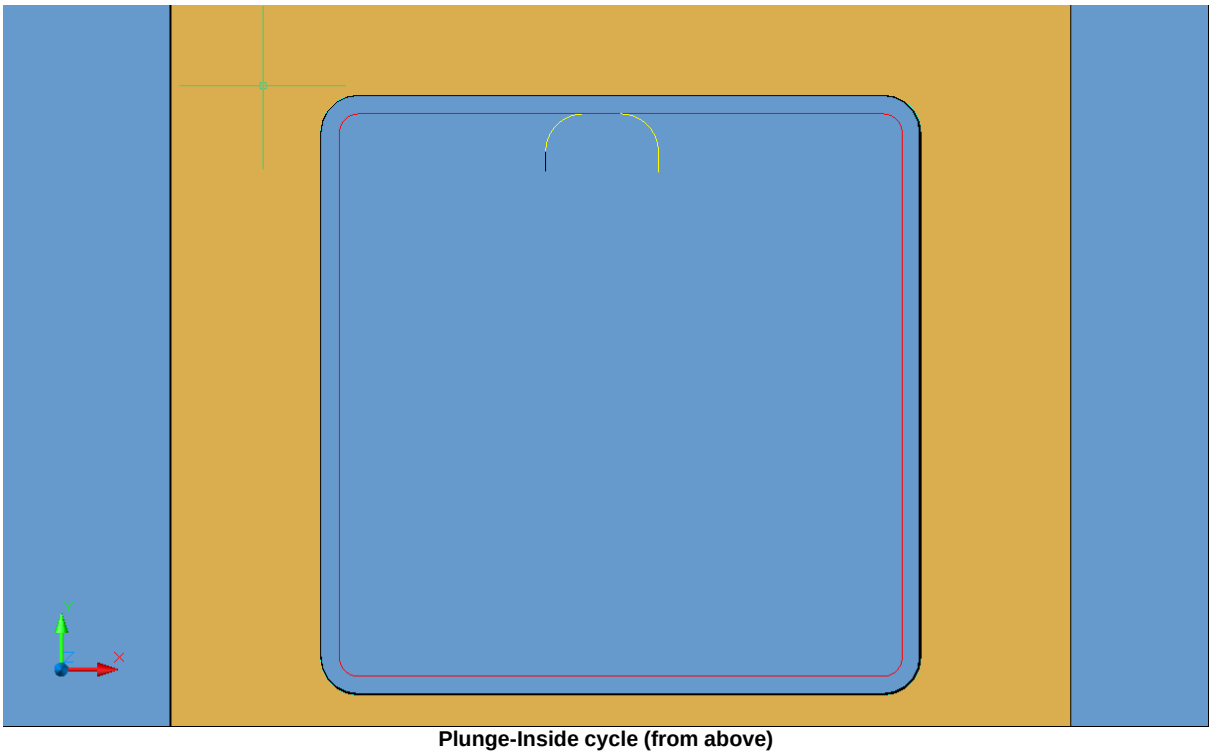
In use the cycle looks like this:



Plunge-Inside cycle

© 2011 CIM-TECH.COM, Inc

From the top the cycle looks like this:

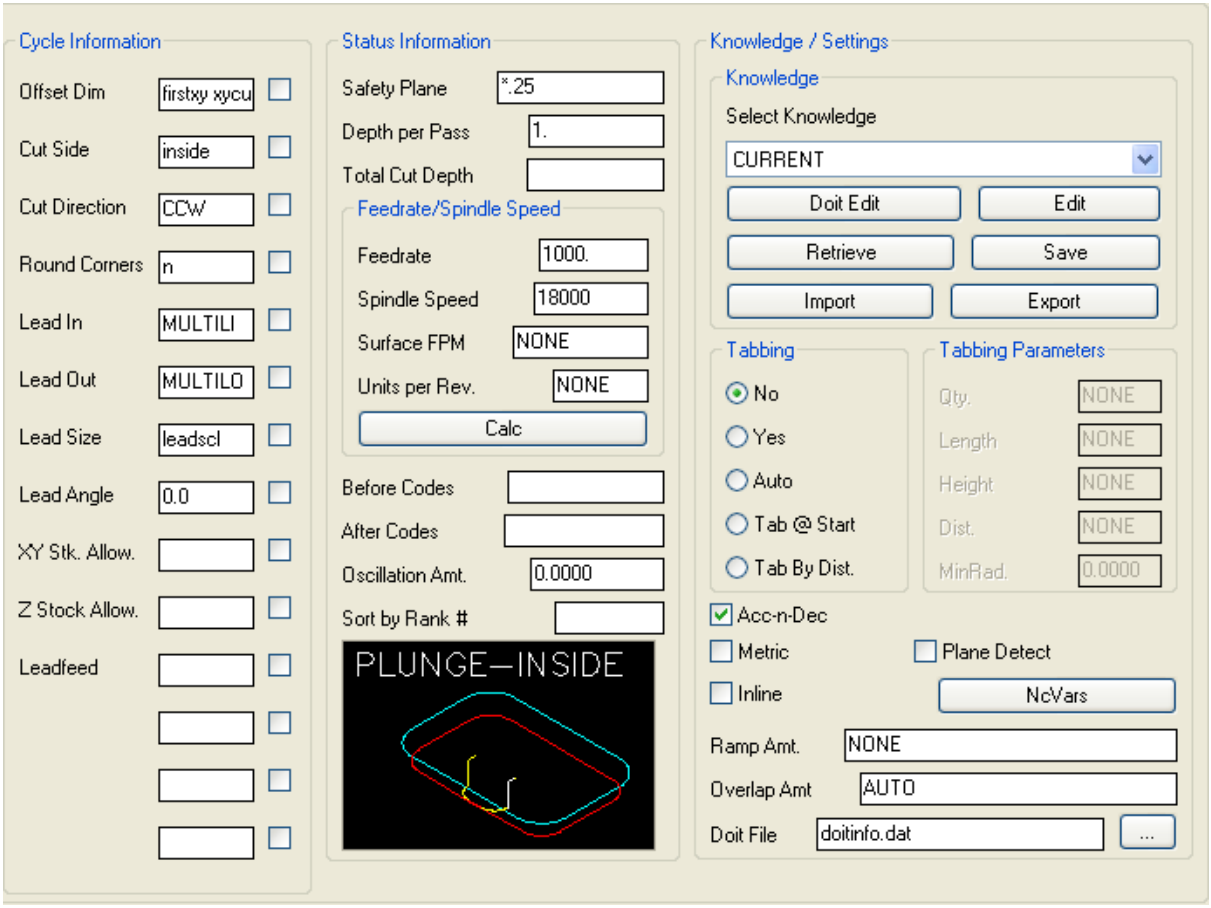


Plunge-Inside cycle (from above)

© 2011 CIM-TECH.COM, Inc

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:



Plunge-Inside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside

© 2011 CIM-TECH.COM, Inc

(Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

© 2011 CIM-TECH.COM, Inc

See the Lead Angle section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you

© 2011 CIM-TECH.COM, Inc

give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

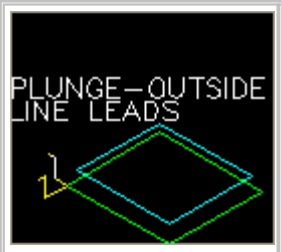
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.1.3 Plunge-Outside Line Leads



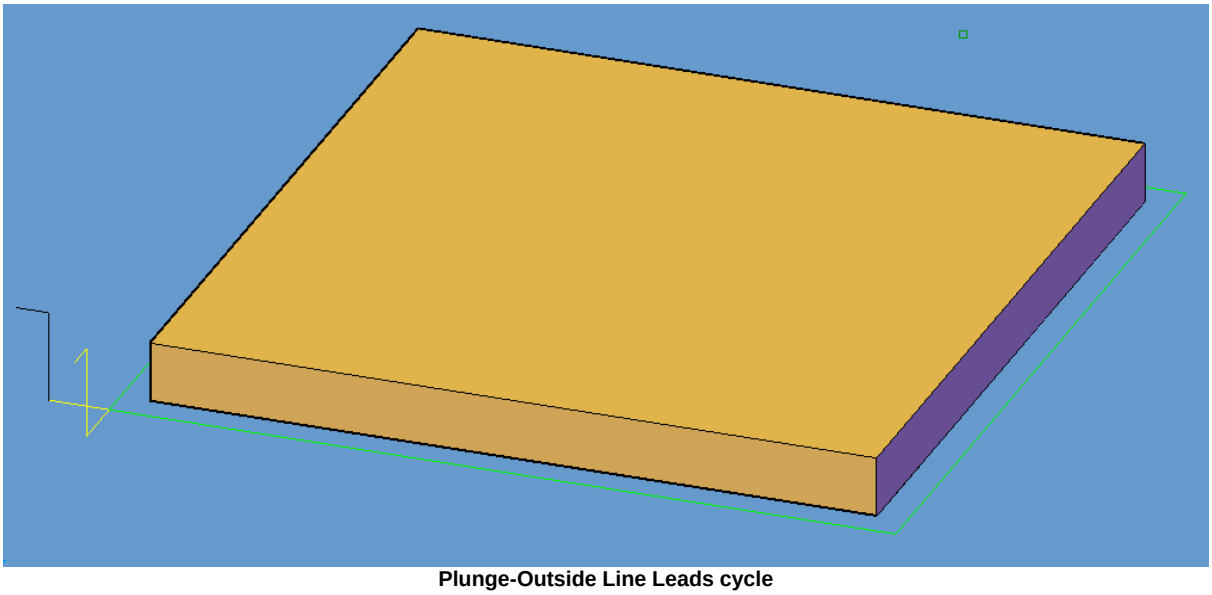
The Plunge-Outside Line Leads cycle will start at the Safety Plane, plunge to the Total Cut Depth or Depth per Pass and then make a lead-in move tangent to the first cut, then follow the entire shape given, and make a lead-out tangent to the last move, and finally retract back to the Safety Plane. There is no overlap by default.

This cycle is used most often on outside shapes where the tool is plunged to full depth outside of the part, then lead into the shape to cut the outside and led out away from the part before retracting. The size of the lead-In and lead-out may be controlled by the user, as well as the geometry of the lead-in or lead-out as desired.

The Plunge-Outside Line Leads cycle is a somewhat generic profile cutting cycle in that the parameters used for this cut cycle are able to be modified to create many types of profile cuts. Each of the parameters shown in the section below will be explained in detail for a better explanation of their capabilities.

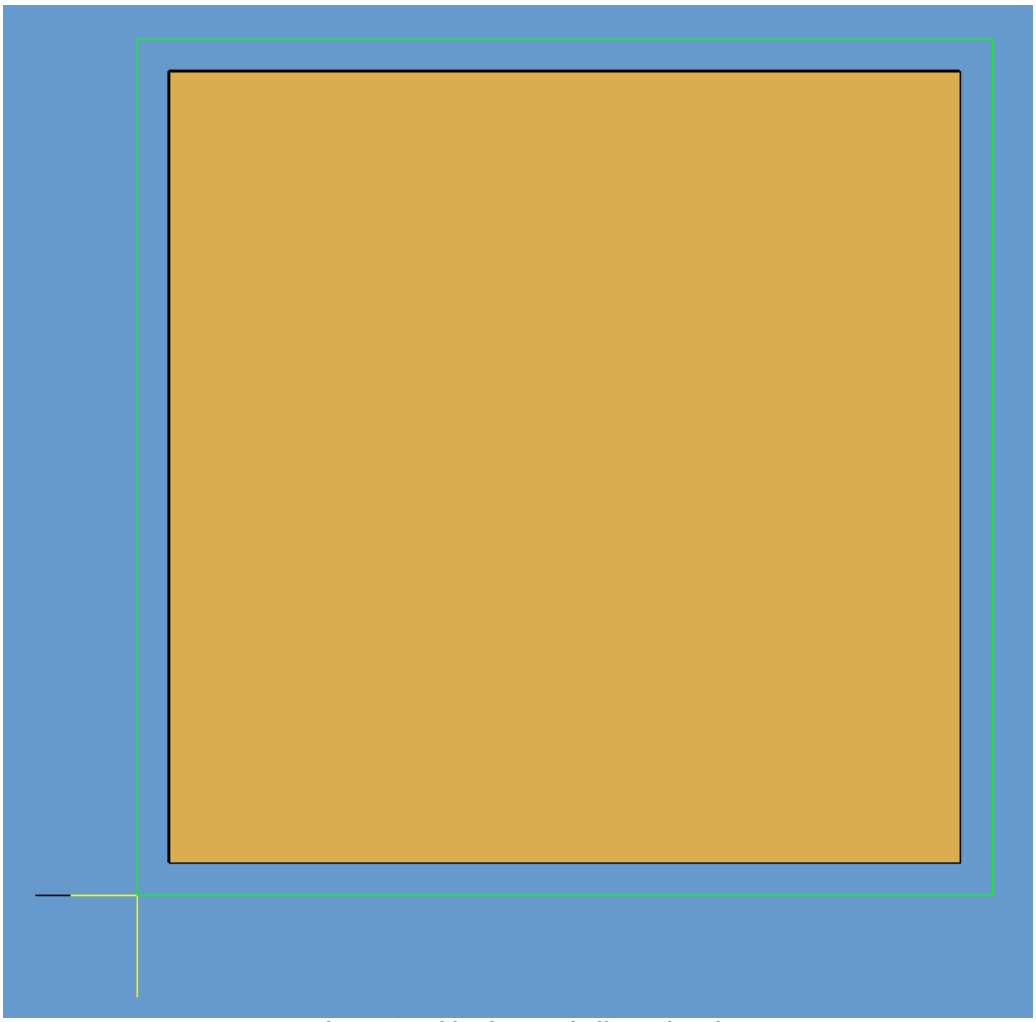
Remember that this cycle has no overlap, so if the Start Point is along an edge you will see a mark on the part where the cutter started and ended. Use this cycle for cutting open shapes, because it allows the user to modify the Lead-In, Lead-Out, Cut Side, Cut Direction, Offset Amount, and Lead Size parameters to fit a specific cutting task.

This cycle is used mostly for cutting with shaper or profile tools, when you want to lower the cutter to full depth away from the finished part, and allow the cutter to rotate up to speed before engaging the finished part.
In use the cycle looks like this:



Plunge-Outside Line Leads cycle

© 2011 CIM-TECH.COM, Inc

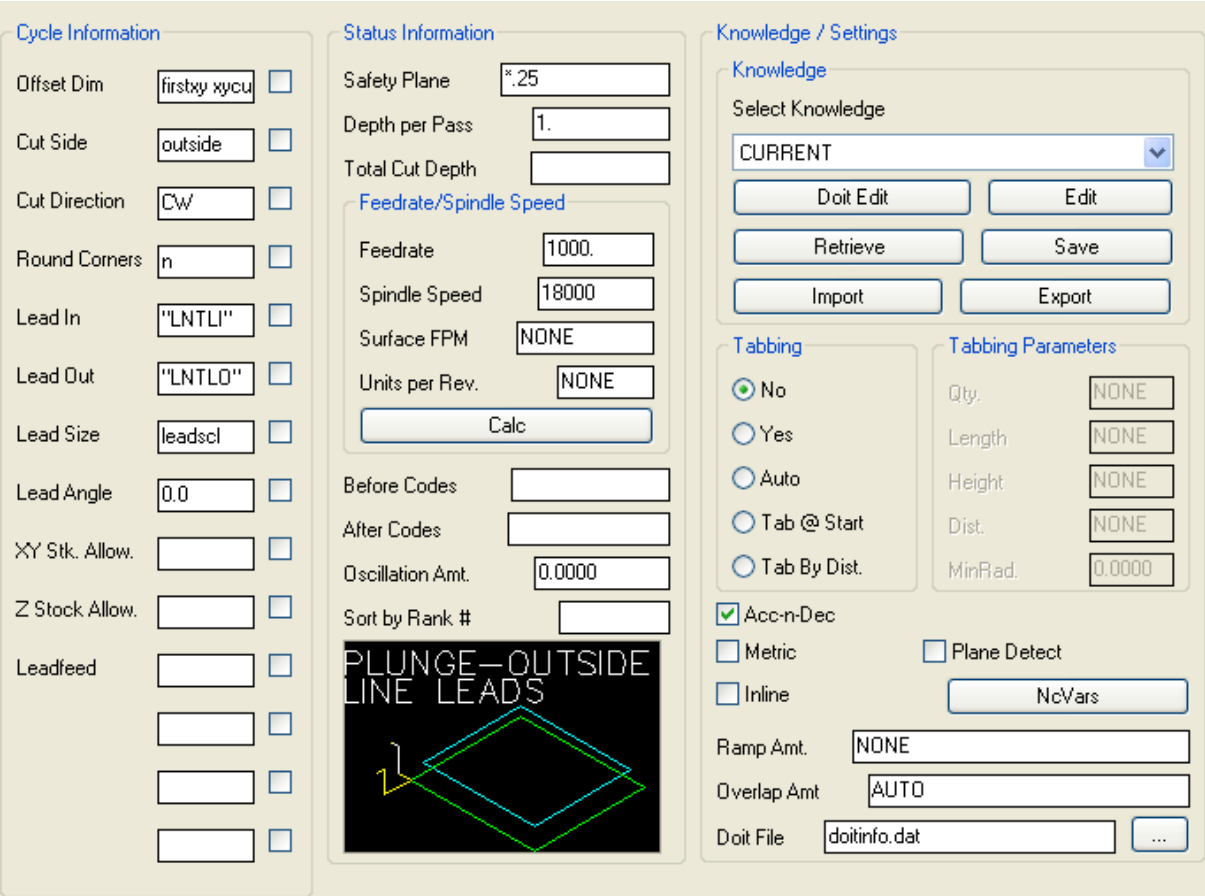


Plunge-Outside Line Leads (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

© 2011 CIM-TECH.COM, Inc



Plunge-Outside Line Leads cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See Offset Dim for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*,

© 2011 CIM-TECH.COM, Inc

Inside, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

© 2011 CIM-TECH.COM, Inc

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.
Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).
Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach

© 2011 CIM-TECH.COM, Inc

this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

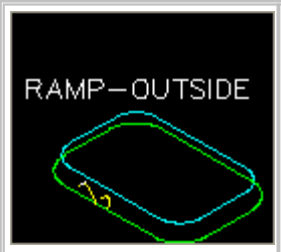
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

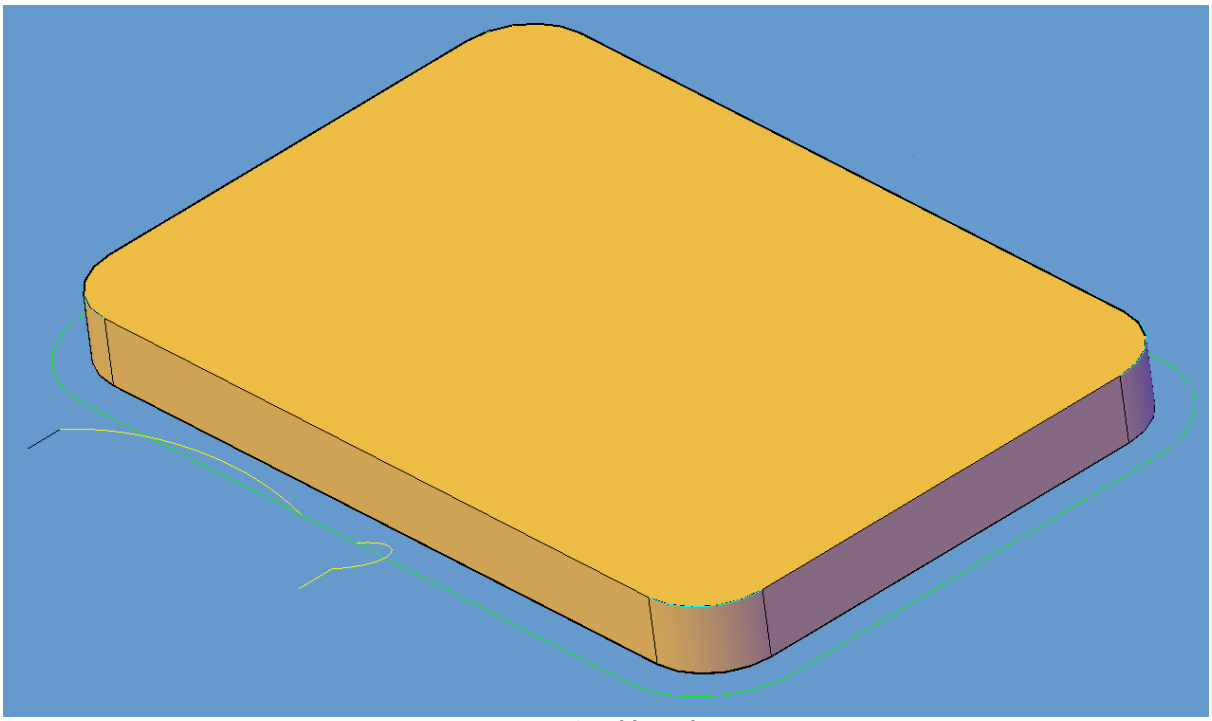
© 2011 CIM-TECH.COM, Inc

6.1.1.4 Ramp-Outside



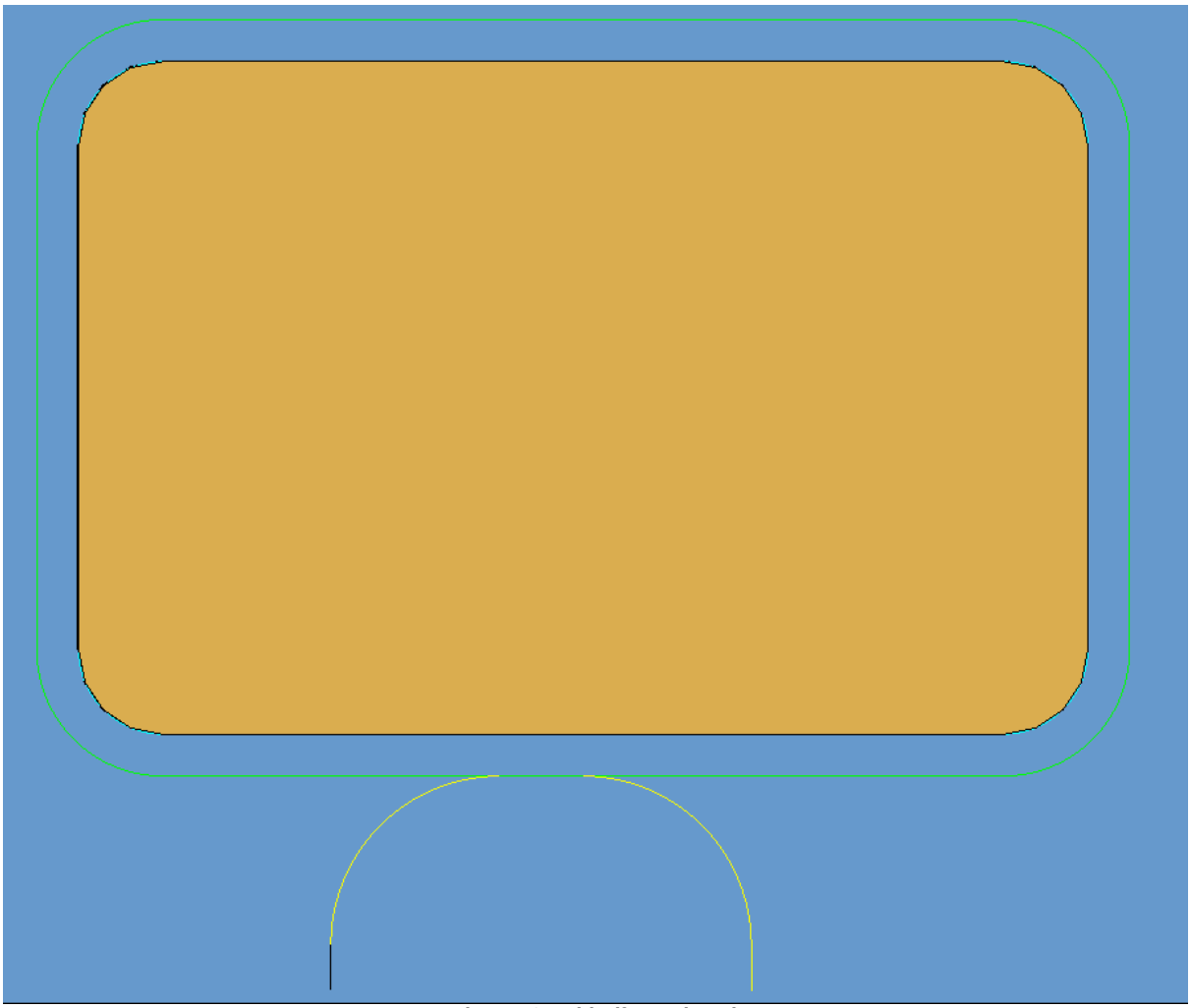
Ramp Cutting cycles allow you to have a helical lead-in and lead-out from a point perpendicular to the part.
This cycle is especially useful to save on tool wear, and in circumstances where the cutter cannot plunge into the material.
Ramp Outside and Ramp Inside only work on closed shapes, and produce cuts with an offset to the outside or inside of the geometry according to the cycle.

The tool will start above the part at the Safety Plane, and then make a 90° arc, ramping lead-in to the start point. Once at depth the cutter will follow the shape on the outside and cut back to the start point, overlap the start point and then make a 90° arc, ramping lead-out back up to the Safety Plane.



Ramp-Outside cycle

© 2011 CIM-TECH.COM, Inc

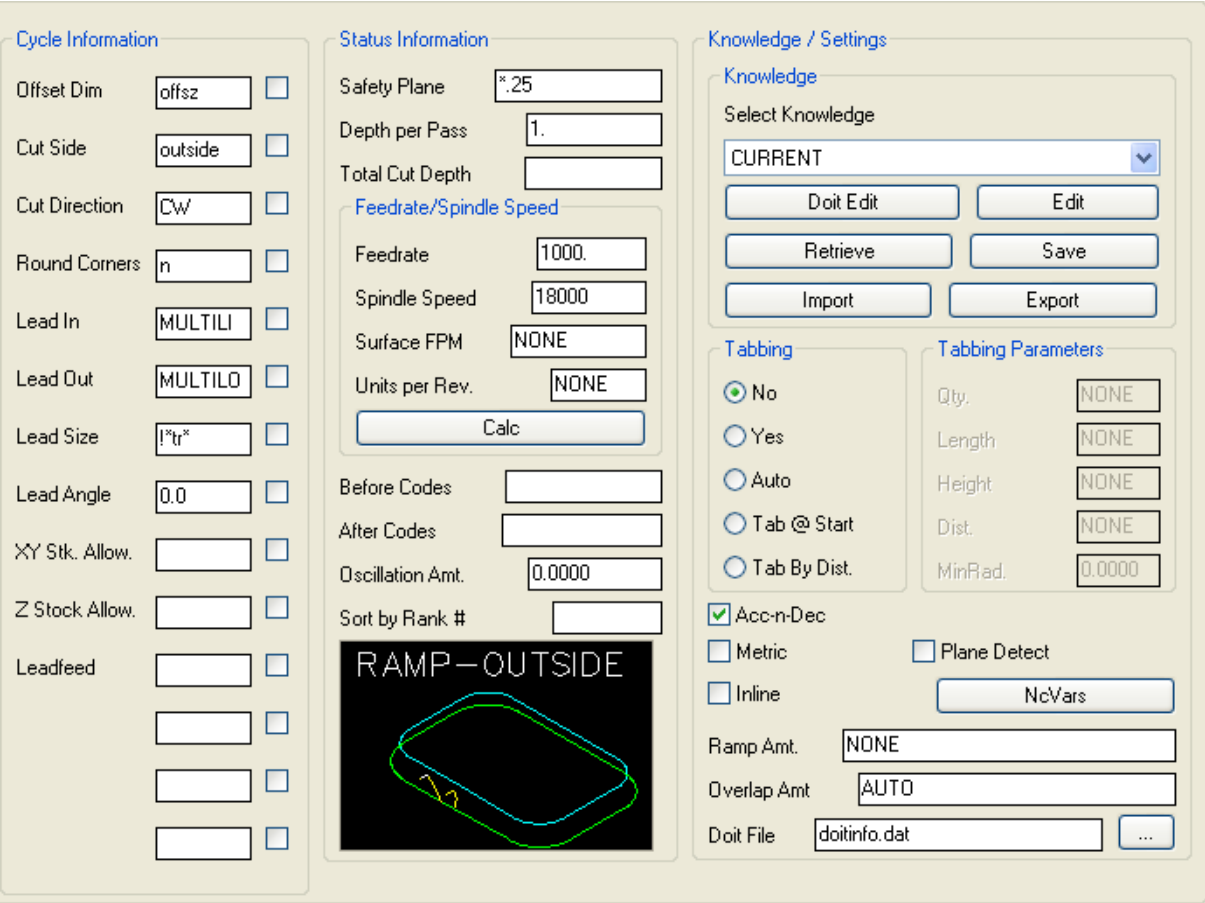


Plunge-Outside (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:

© 2011 CIM-TECH.COM, Inc



Plunge-Outside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See Offset Dim for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*,

© 2011 CIM-TECH.COM, Inc

Inside, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

© 2011 CIM-TECH.COM, Inc

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.
Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).
Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach

© 2011 CIM-TECH.COM, Inc

this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

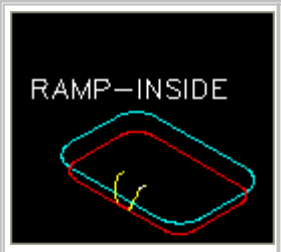
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

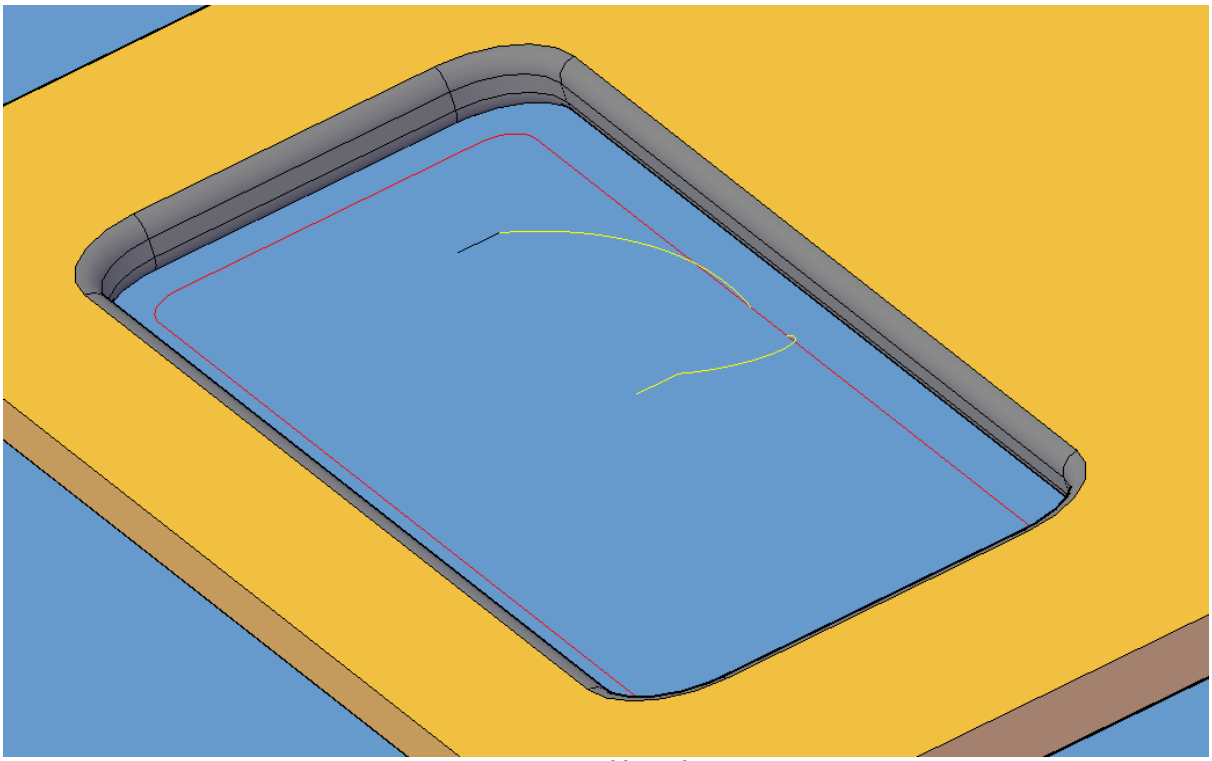
© 2011 CIM-TECH.COM, Inc

6.1.1.5 Ramp-Inside



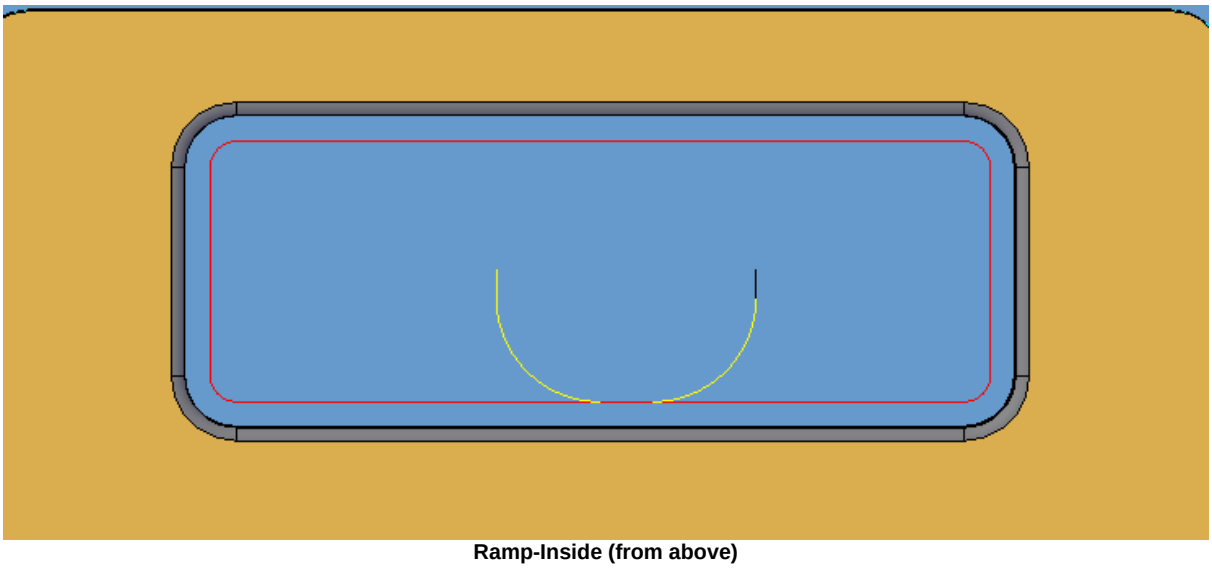
Ramp Cutting cycles allow you to have a helical lead-in and lead-out from a point perpendicular to the part.
This cycle is especially useful to save on tool wear, and in circumstances where the cutter cannot plunge into the material.
Ramp Outside and Ramp Inside only work on closed shapes, and produce cuts with an offset to the outside or inside of the geometry according to the cycle.

The tool will start above the part at the Safety Plane, and then make a 90° arc, ramping lead-in to the start point. Once at depth the cutter will follow the shape on the outside and cut back to the start point, overlap the start point and then make a 90° arc, ramping lead-out back up to the Safety Plane.



Ramp-Inside cycle

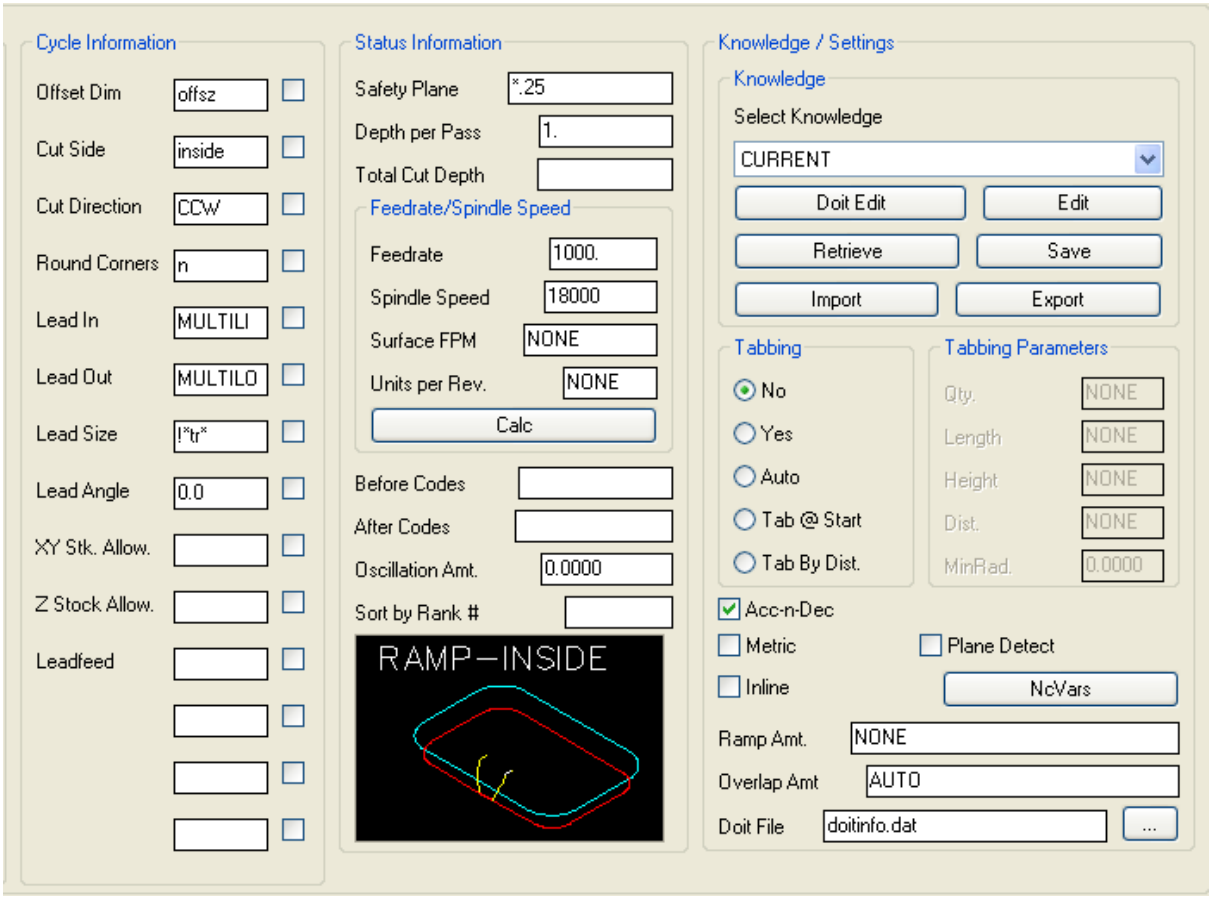
© 2011 CIM-TECH.COM, Inc



Ramp-Inside (from above)

Cycle Parameters

There are several parameters set by the cycle as defaults, and most will not need to be changed. The valid parameters are shown below:



Ramp-Inside cycle parameters

© 2011 CIM-TECH.COM, Inc

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See Offset Dim for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

© 2011 CIM-TECH.COM, Inc

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance
Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass.
Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed
This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).
Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the Lead Feed section for more information.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.

© 2011 CIM-TECH.COM, Inc

This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes

Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes

Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank

© 2011 CIM-TECH.COM, Inc

A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt

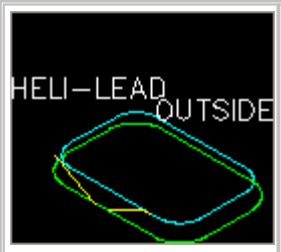
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.1.6 Heli-Lead Outside

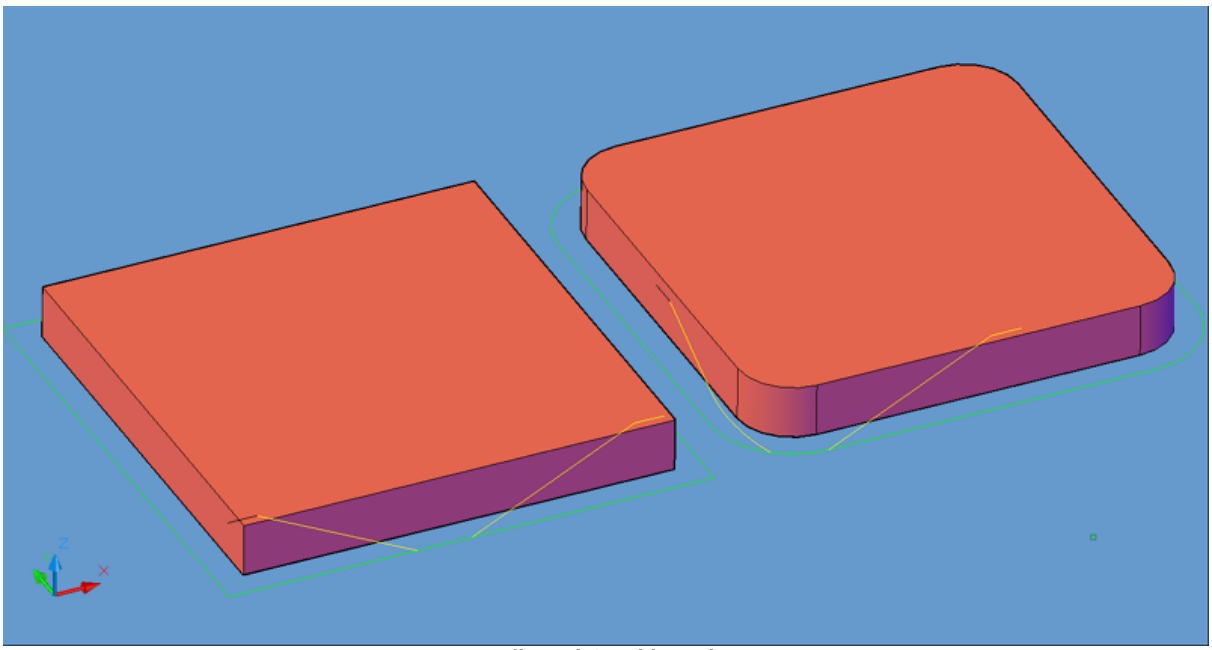


Heli-Lead Cutting cycles allow you to have a contour following lead-in and lead-out. This means that on a straight cut, the lead in and lead out would be a ramp (XZ or YZ motion) and on a non-straight shape the lead-in and lead-out will follow the contour of the shape, no matter the complexity.

These cycles are especially useful when cutting interior shapes or parts in a nest; the scrap will not move during the lead-out. These cycles only work on closed shapes.

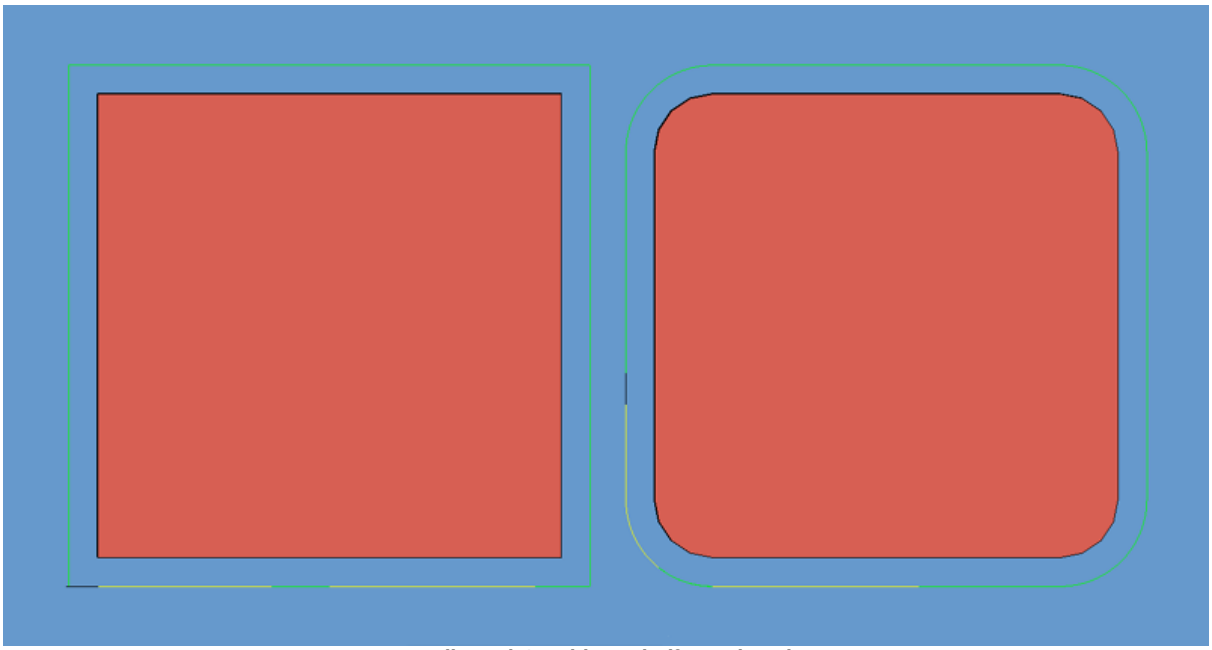
Typically, Heli-Lead Outside will start above the part at the Safety Plane and then ramp into the part along the geoshape, following the contour, until it reaches full cut depth. The cycle will follow the shape back to the start point, overlap by the tool diameter, then ramp out of the shape, again following the contour, back up to the Safety Plane.

This is one of the most commonly used cycles in Router-CIM, because they produce no scrap - the leads are always tangent to the part geometry.



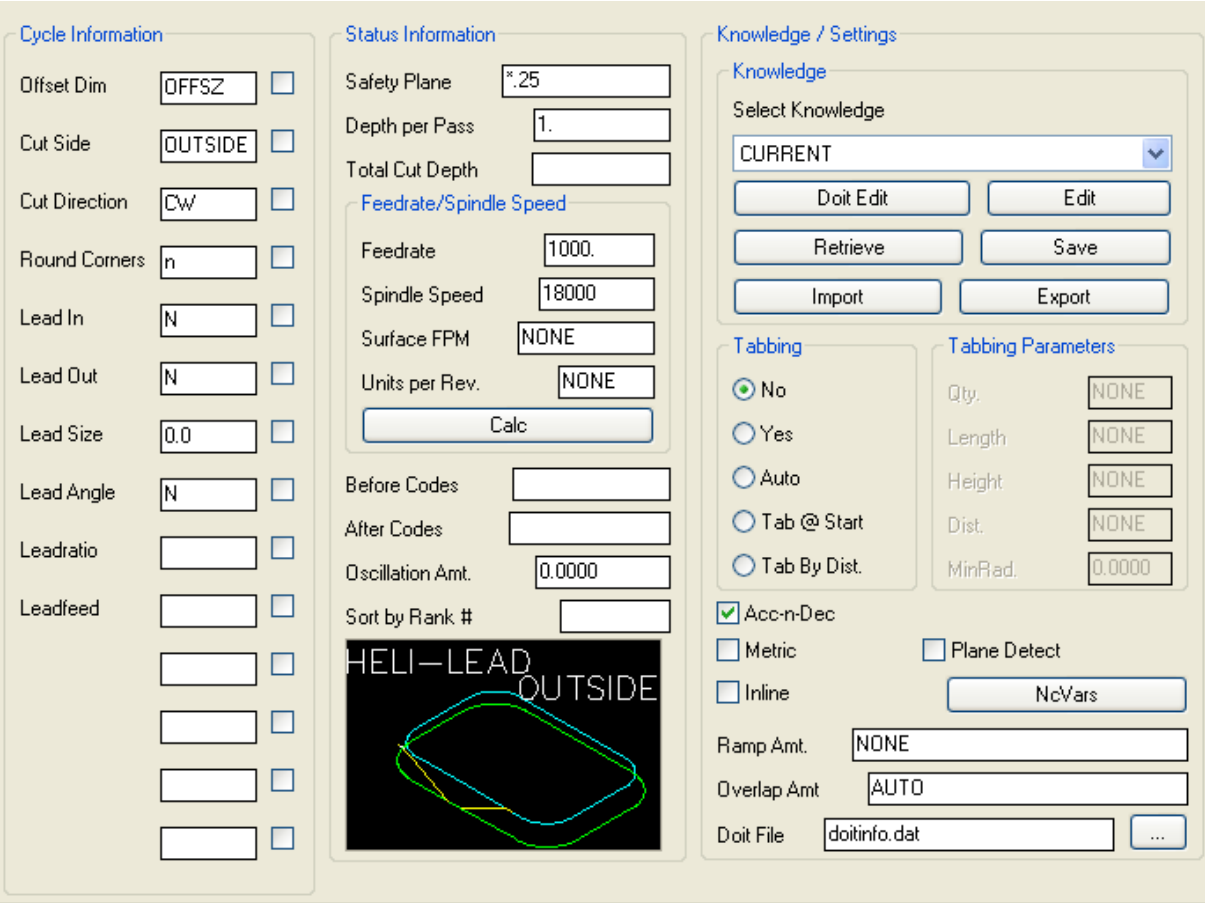
Heli-Lead-Outside cycle

© 2011 CIM-TECH.COM, Inc



Heli-Lead-Outside cycle (from above)

© 2011 CIM-TECH.COM, Inc



Heli-Lead-Outside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (OFFSZ) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

Select the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

Select the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

Select the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

Select the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

Select the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

Select the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

Select the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

Select the Lead Angle section for more information.

Lead Feed

© 2011 CIM-TECH.COM, Inc

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

Select the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

Select the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

© 2011 CIM-TECH.COM, Inc

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

6.1.1.7 Heli-Lead-Inside



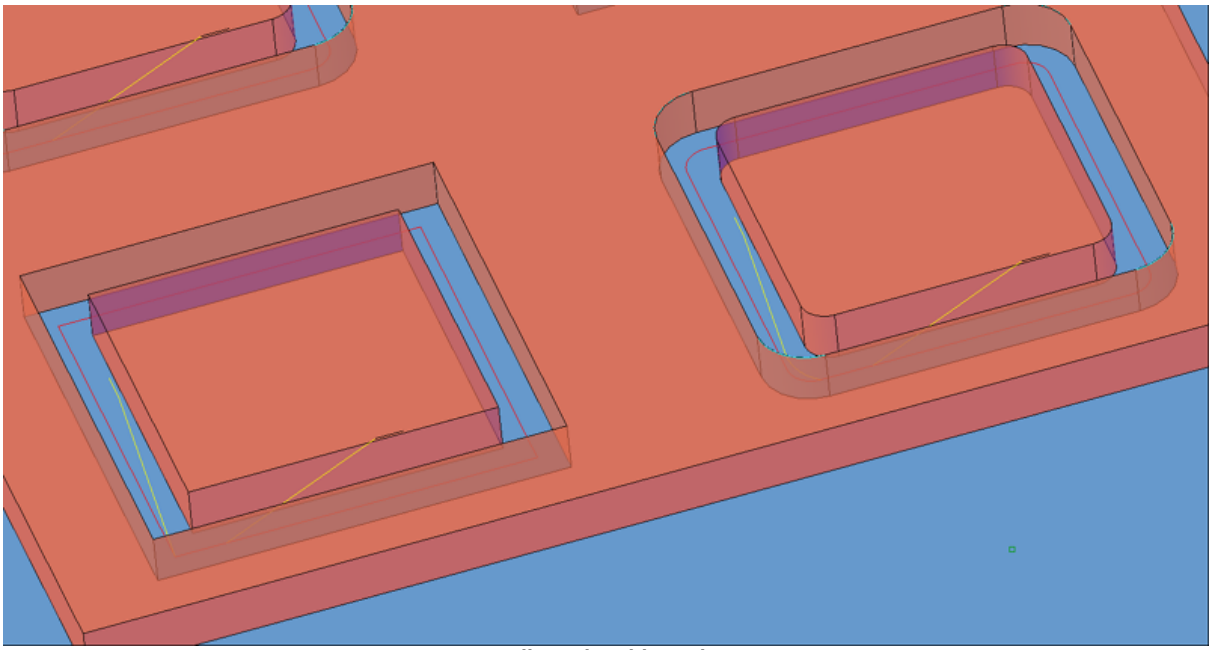
Heli-Lead Cutting cycles allow you to have a contour following lead-in and lead-out. This means that on a straight cut, the lead in and lead out would be a ramp (XZ or YZ motion) and on a non-straight shape the lead-in and lead-out will follow the contour of the shape, no matter the complexity.

These cycles are especially useful when cutting interior shapes or parts in a nest; the scrap will not move during the lead-out. These cycles only work on closed shapes.

Typically, Heli-Lead Inside will start above the part at the Safety Plane and then ramp into the part along the geoshape, following the contour, until it reaches full cut depth. The cycle will follow the shape back to the start point, overlap by the tool diameter, then ramp out of the shape, again following the contour, back up to the Safety Plane.

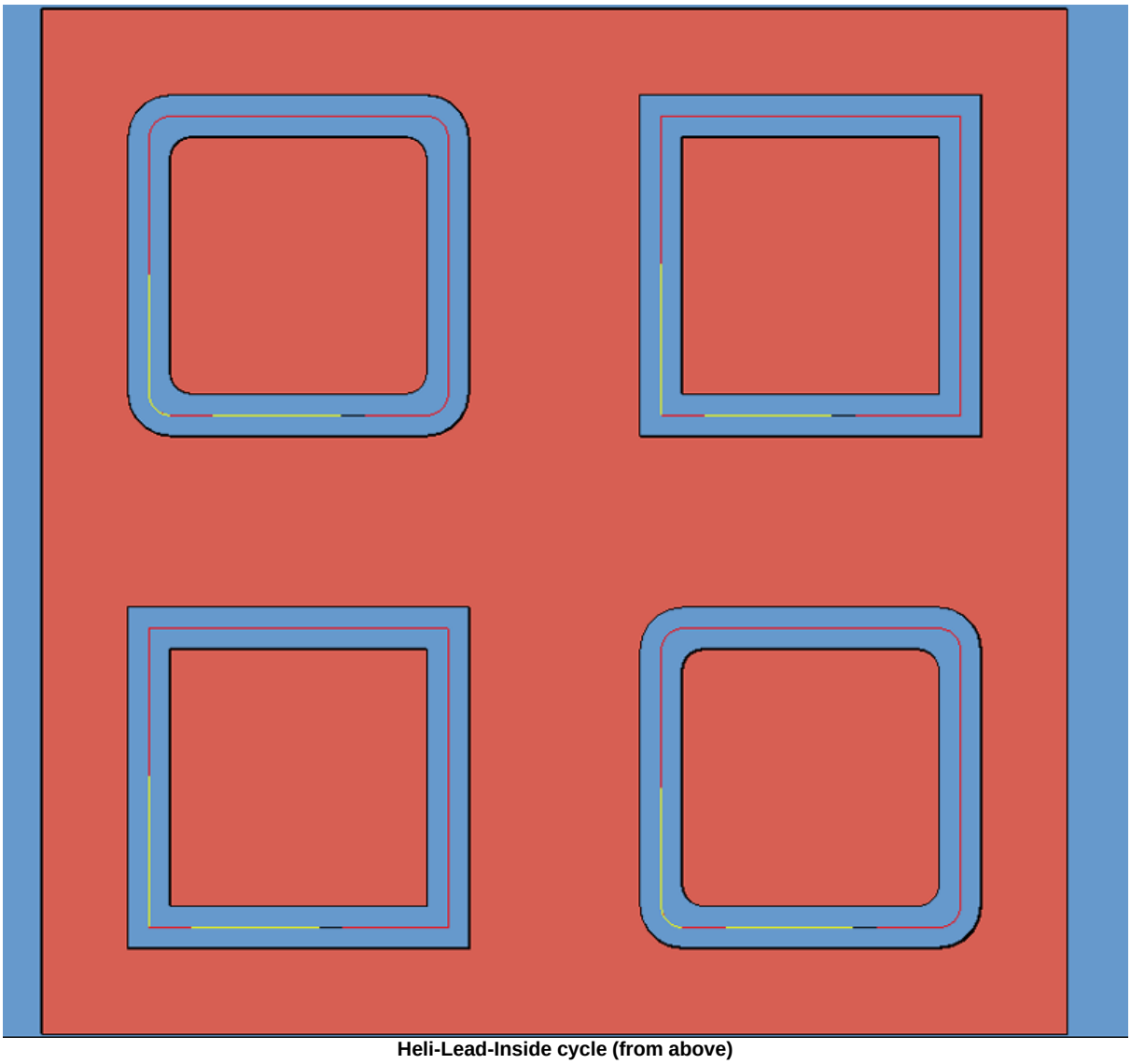
This is one of the most commonly used cycles in Router-CIM, because they produce no scrap - the leads are always tangent to the part geometry.

© 2011 CIM-TECH.COM, Inc



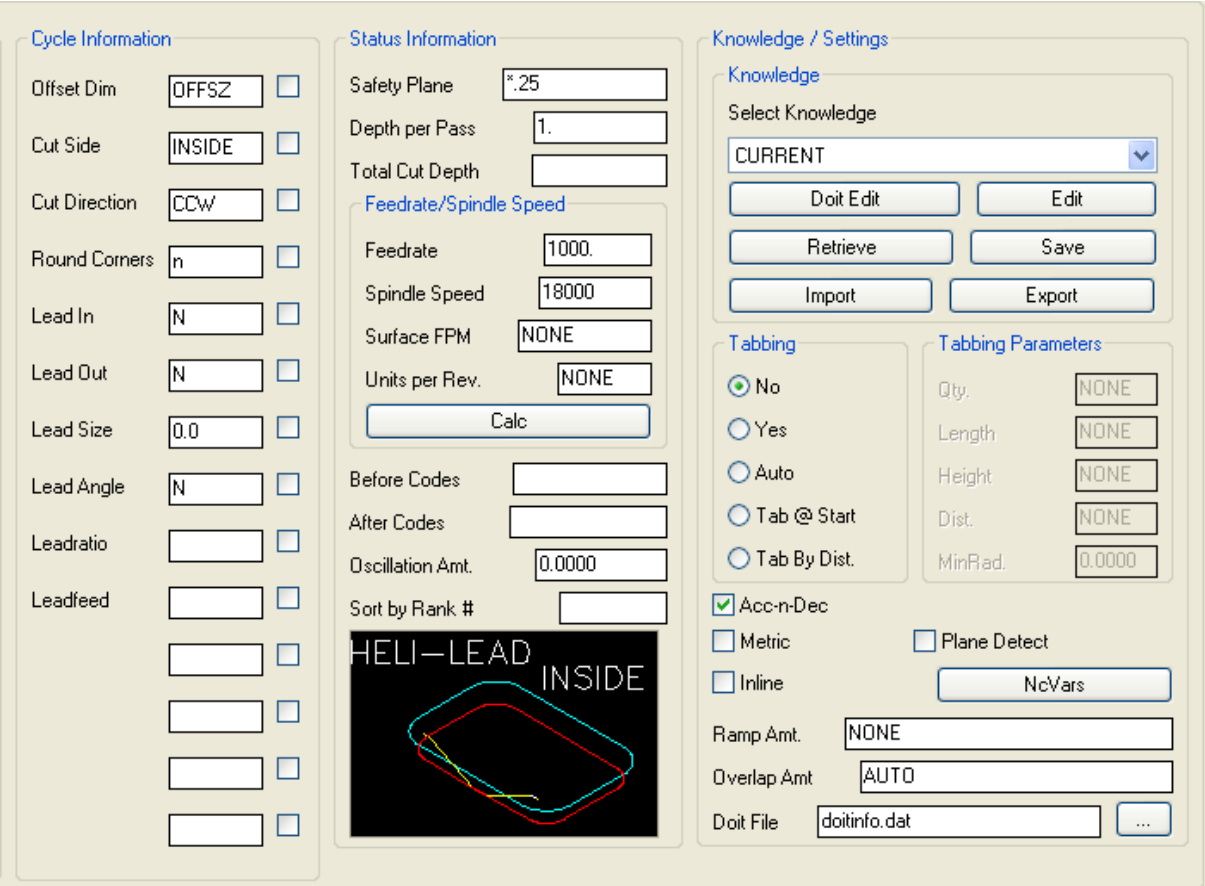
Heli-Lead-Inside cycle

© 2011 CIM-TECH.COM, Inc



Heli-Lead-Inside cycle (from above)

© 2011 CIM-TECH.COM, Inc



Heli-Lead-Inside cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (OFFSZ) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

© 2011 CIM-TECH.COM, Inc

See the Cut Side section for more information.

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

© 2011 CIM-TECH.COM, Inc

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

© 2011 CIM-TECH.COM, Inc

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

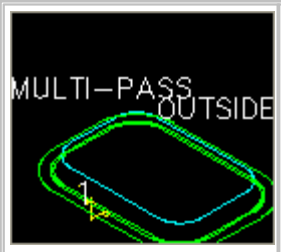
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

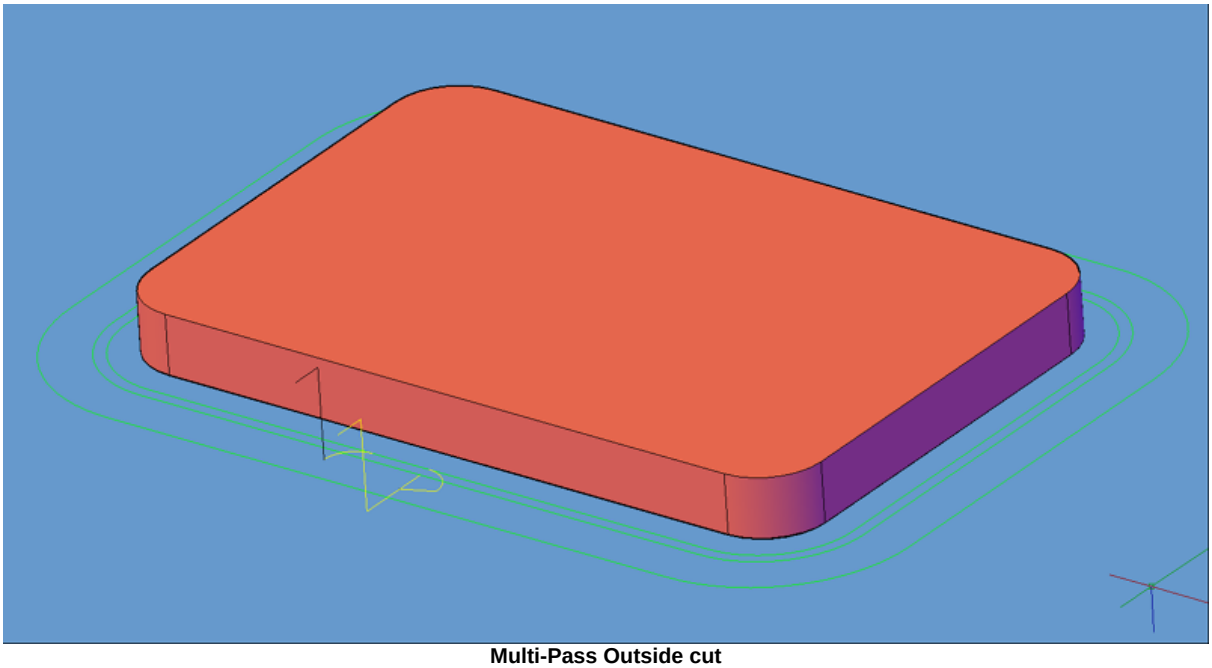
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.1.8 Multi-Pass-Outside

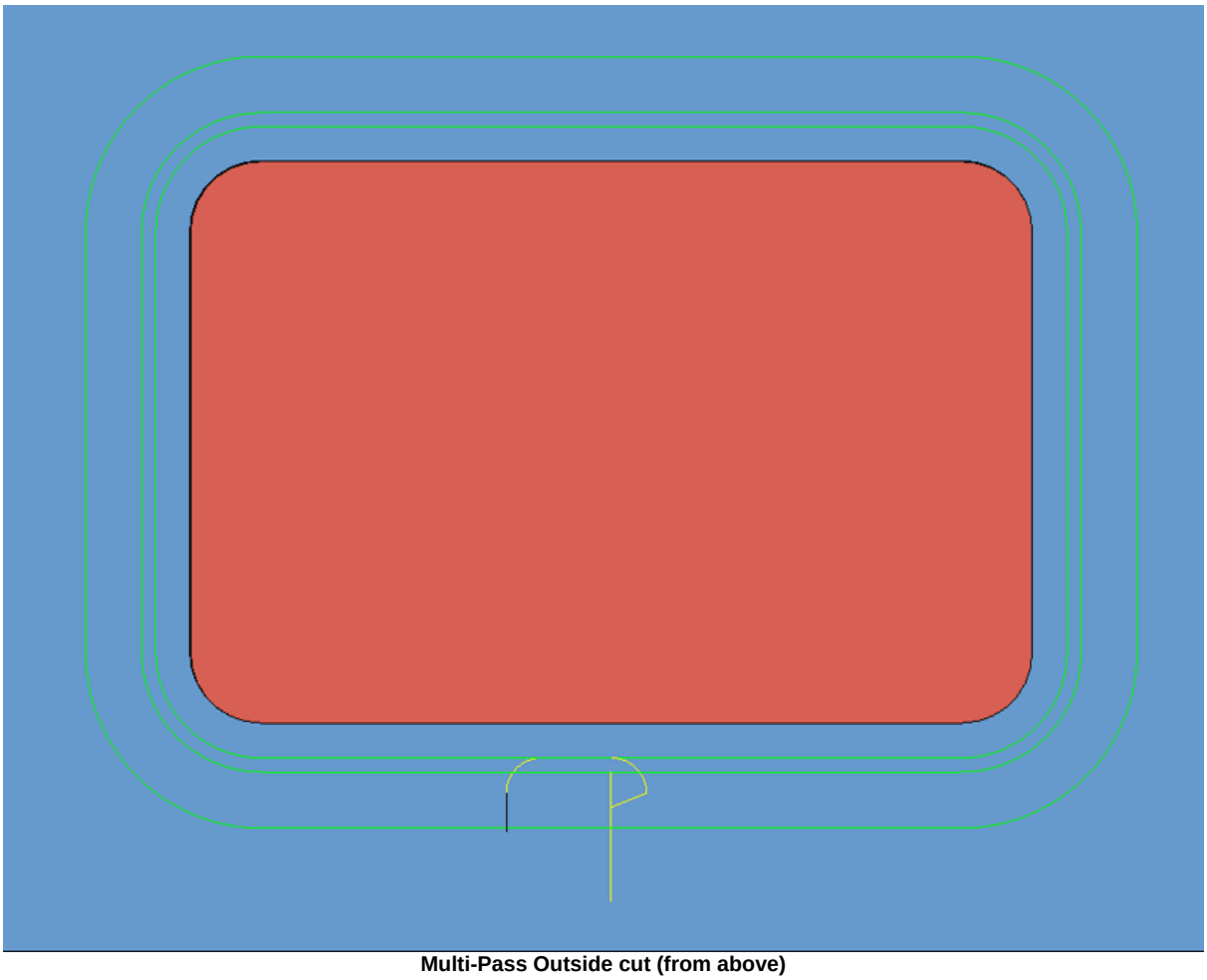


This cycle allows for a rough cut and finish cut on the outside of a shape according to dimensions that you specify in cycle parameters. The cycles creates multiple, offset toolpaths from the outside of the shape towards the finished edge, with specified dimensions for the amount of the first pass, last pass, and cut spacing. You also specify how far away from the finished edge the finish pass is created. The tool will start at the Safety Plane, Plunge to the depth specified in Total Cut Depth or Depth per Pass and then make a straight lead in to the location of the first pass set by the Max Offset. The cutter will make the first pass, then step in towards the finished edge according to the XY Step amount and make another pass until it reaches the offset for the Last Offset. On the Last Offset, the tool will move back and over to a point where it can make a 90° lead-in arc, then cut the finish pass back to the start point, overlap the start point by the tool diameter and make a 90° lead-out arc before retracting back up to the Safety Plane.



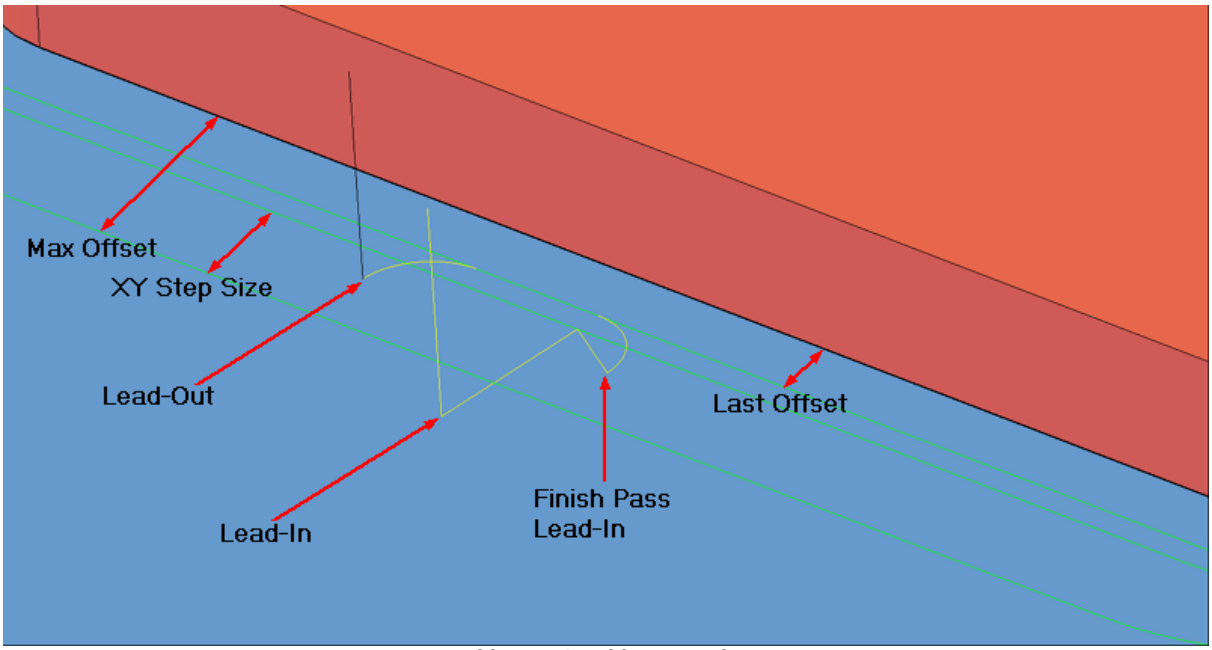
Multi-Pass Outside cut

© 2011 CIM-TECH.COM, Inc

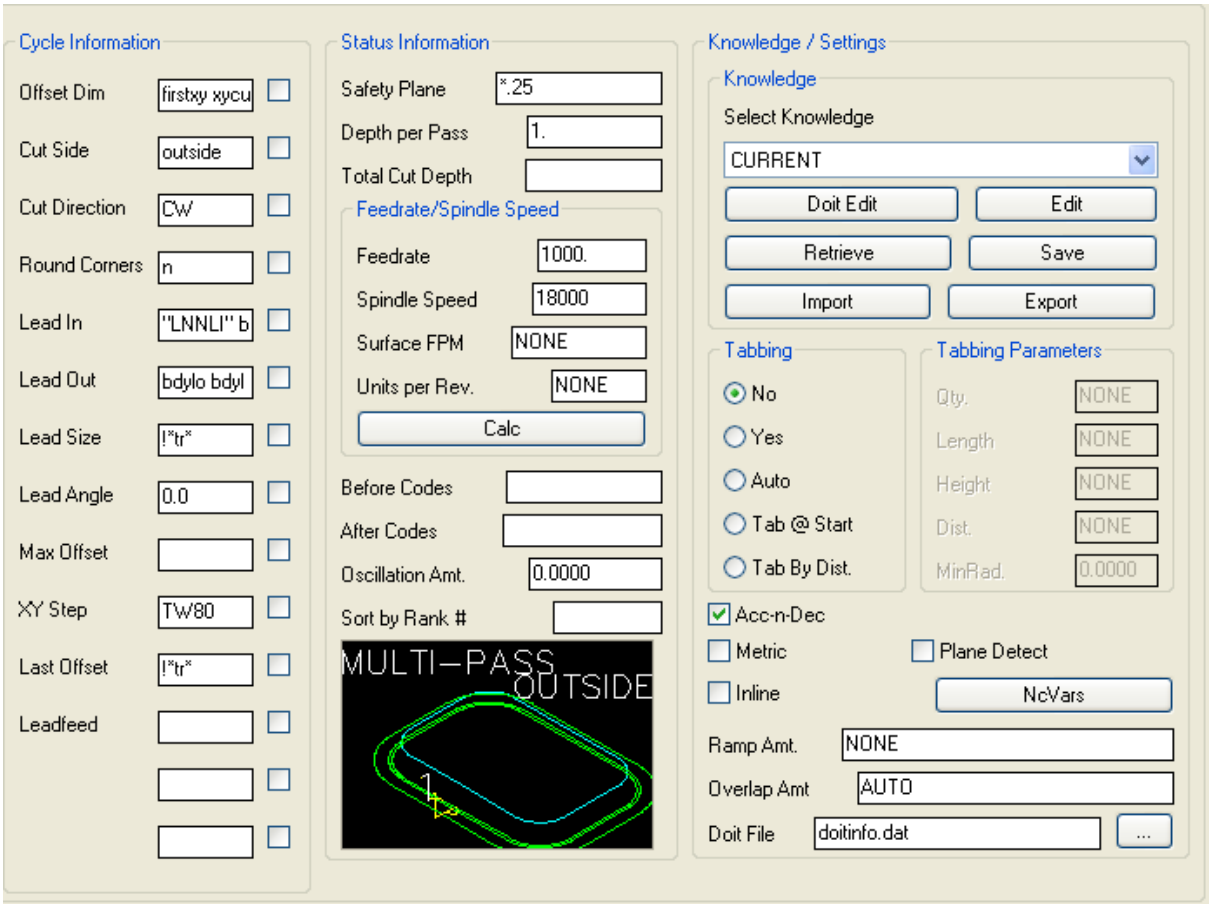


Multi-Pass Outside cut (from above)

© 2011 CIM-TECH.COM, Inc



Multi-Pass Outside cut settings



Multi-Pass Outside cut parameters

© 2011 CIM-TECH.COM, Inc

This cycle will only offset shapes that can be offset in AutoCAD, and if the AutoCAD offset command fails, so will this cutting cycle. There is no provision for leaving 'islands' or separate areas that are uncut in the shape. This is not a pocketing cycle. There are some cut properties unique to this cycle, each of them is explained below.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

© 2011 CIM-TECH.COM, Inc

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

Max Offset
Max Offset is the distance from the start point to the start of the first pass on the cut. If the cut cycle is Multi-Pass Inside, then the Max Offset will be to the inside of the shape by the specified amount. If the cycle is Multi-Pass Outside, then the Max Offset will be from the start point to the outside of the shape by the Max Offset amount. This is only the distance for the first cut. All subsequent cuts will be determined by either XY Step or Last Offset.

XY Step Size
The XY Step is the distance between each tool path from the first to the last (excluding the finish pass) on the Multi-Pass Inside and Multi-Pass Outside cycles.

You can specify the distance as a numeric value, or you can make a task to calculate a step over amount and place the task name in the XY Step field. An example of a task is TW80, which looks at the tool diameter field and then places an amount equal to 80% of that value in this location.

Last Offset
The Last Offset is the distance from the finish pass to the edge of the shape being cut. This can be any numeric value, but if it exceeds the XY Step value, the cutter will not contact the part on the finish pass. Typically this is set to the radius of the tool in a rough-cut, finish-cut scenario. You can leave some material on the part if you want to clean up the part with a finish cutter.

Lead Feed
This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

© 2011 CIM-TECH.COM, Inc

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes

Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes

Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

© 2011 CIM-TECH.COM, Inc

Sort by Rank

A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt

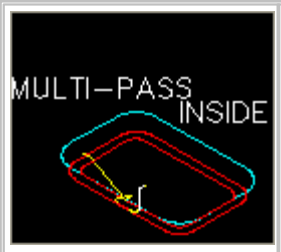
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

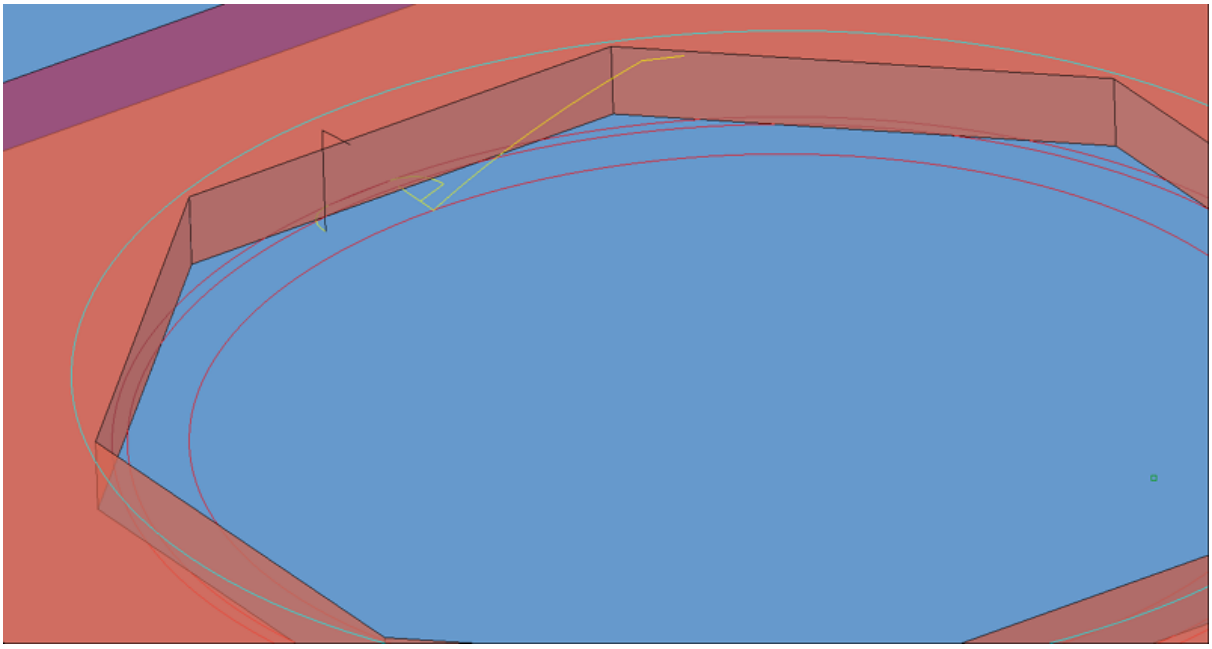
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

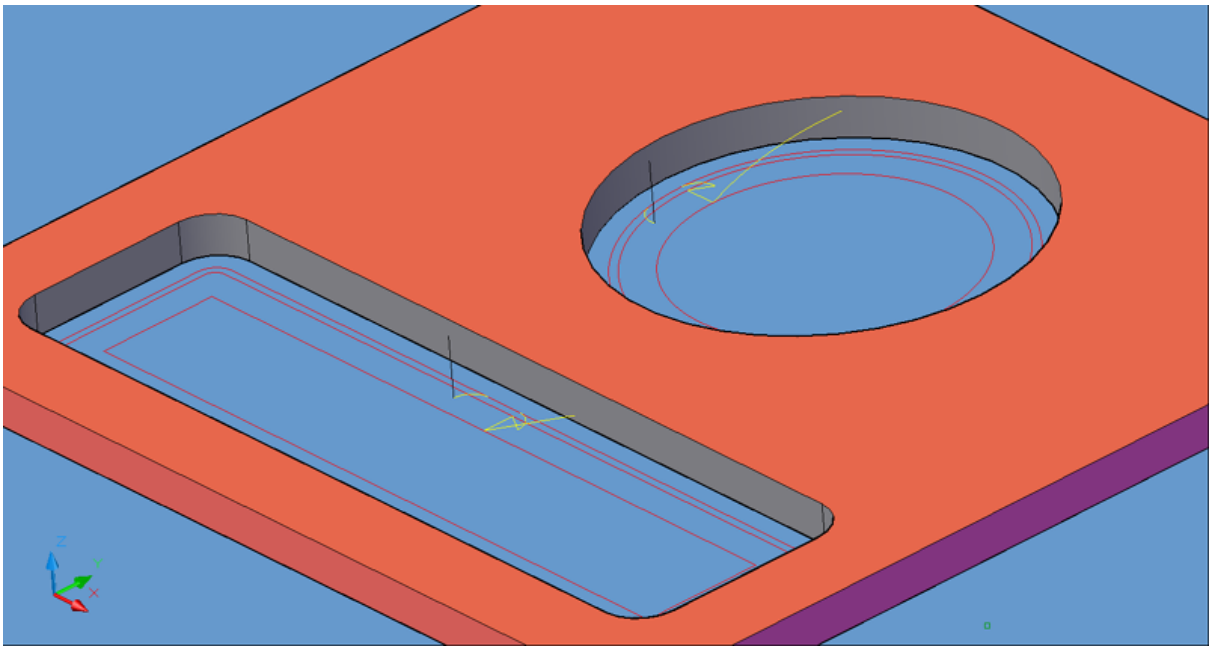
6.1.1.9 Multi-Pass-Inside



This cycle allows for a rough cut and finish cut on the inside of a shape according to dimensions that you specify in cycle parameters. The cycles creates multiple, offset toolpaths from the inside of the shape towards the finished edge, with specified dimensions for the amount of the first pass, last pass, and cut spacing. You also specify how far away from the finished edge the finish pass is created. The tool will start at the Safety Plane, ramp down to the depth specified in Total Cut Depth or Depth per Pass and then make a straight lead in to the location of the first pass set by the Max Offset. The cutter will make the first pass, then step in towards the finished edge according to the XY Step amount and make another pass until it reaches the offset for the Last Offset. On the Last Offset, the tool will move back and over to a point where it can make a 90° lead-in arc, then cut the finish pass back to the start point, overlap the start point by the tool diameter and make a 90° lead-out arc before retracting back up to the Safety Plane.

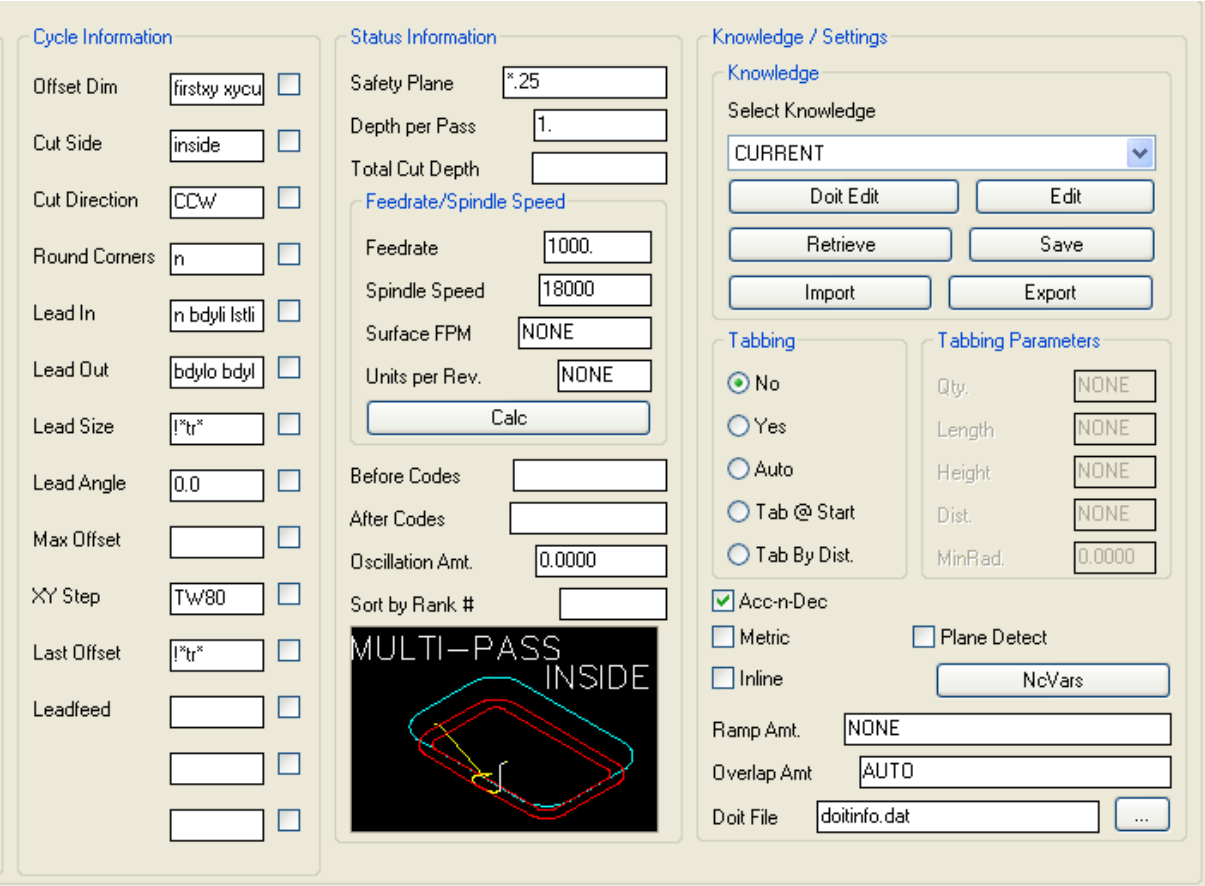


© 2011 CIM-TECH.COM, Inc



One of the main differences between the Multi-Pass Inside and Multi-Pass Outside is that the Inside cycle will make a contour following ramp-in for the first pass. This means that this cycle can be used to remove material from an area inside a shape without a rougher tool having to remove any material first.

© 2011 CIM-TECH.COM, Inc



Multi-Pass Inside cycle parameters

This cycle will only offset shapes that can be offset in AutoCAD, and if the AutoCAD offset command fails, so will this cutting cycle. There is no provision for leaving 'islands' or separate areas that are uncut in the shape. This is not a pocketing cycle. There are some cut properties unique to this cycle, each of them is explained below.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*,

© 2011 CIM-TECH.COM, Inc

Inside, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

© 2011 CIM-TECH.COM, Inc

Max Offset
Max Offset is the distance from the start point to the start of the first pass on the cut. If the cut cycle is Multi-Pass Inside, then the Max Offset will be to the inside of the shape by the specified amount. If the cycle is Multi-Pass Outside, then the Max Offset will be from the start point to the outside of the shape by the Max Offset amount. This is only the distance for the first cut. All subsequent cuts will be determined by either XY Step or Last Offset.

XY Step Size
The XY Step is the distance between each tool path from the first to the last (excluding the finish pass) on the Multi-Pass Inside and Multi-Pass Outside cycles.

You can specify the distance as a numeric value, or you can make a task to calculate a step over amount and place the task name in the XY Step field. An example of a task is TW80, which looks at the tool diameter field and then places an amount equal to 80% of that value in this location.

Last Offset
The Last Offset is the distance from the finish pass to the edge of the shape being cut. This can be any numeric value, but if it exceeds the XY Step value, the cutter will not contact the part on the finish pass. Typically this is set to the radius of the tool in a rough-cut, finish-cut scenario. You can leave some material on the part if you want to clean up the part with a finish cutter.

Lead Feed
This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth
The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

© 2011 CIM-TECH.COM, Inc

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

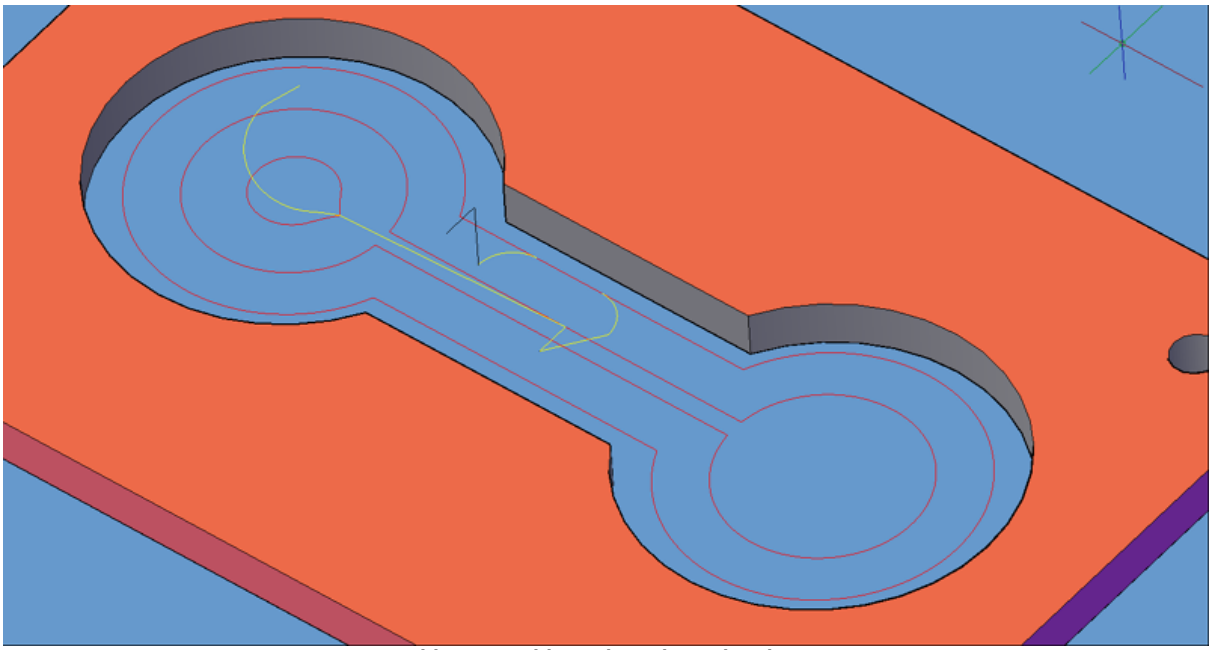
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

Additional notes for Multi-Pass Inside:
If you are using this cycle to remove all of the material from the inside of a shape, then there are two things to consider.

First, the shape you are cutting must be a regular shape that the AutoCAD offset command can work on to make regular offsets that will clean out the entire area. If the AutoCAD offset command fails to be able to make enough offsets, then this cut cycle will not make the proper tool paths. Shown below is an example of how an irregular shape could fail the offset. For these types of shapes, the Pocketing cycles are appropriate.

© 2011 CIM-TECH.COM, Inc



Multi-Pass Inside cycle on irregular shape.

Second, if you have a regular shape (rectangle, circle, etc.) and you must remove all the material inside the shape with the Multi-Pass Inside cycle, you will have to do some calculations for the Max Offset. Since Max Offset is the offset distance from the edge of the finished shape to the start of the first cut, you will have to calculate the distance of that first offset tool path.
There are some simple formulas to use for regular shapes.

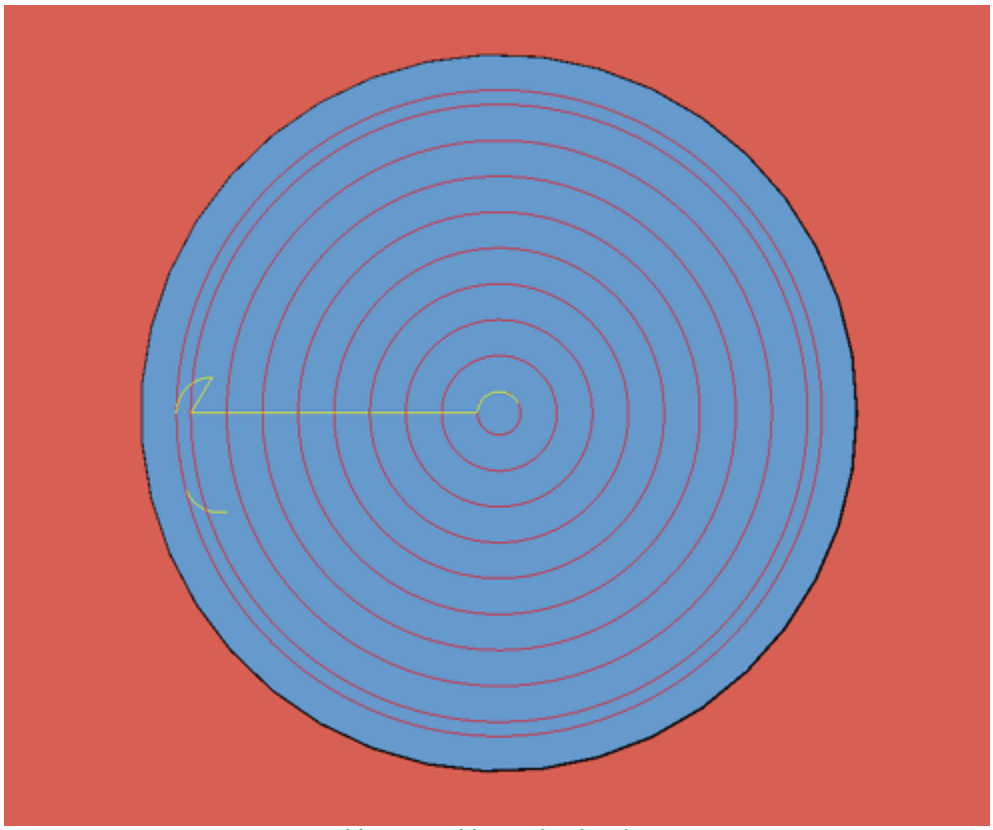
Circular Shapes

To calculate how far to make the Max offset on a circular shape, you can take the Radius of the circle, which would get you to the exact center, then subtract the tool radius to get an offset from the center (otherwise the tool would move down to the center, kind of like a drill with no other motion to make), then add how much of an overlap you want on the first pass.
So, in example to cut a 5" diameter circle, with a .5" Router-Bit, you would start with a Max offset of about 2.35. That is using an overlap of .10 on the first pass.

Diameter / 2 = Circle Radius	5.0 / 2 = 2.5
Tool Diameter / 2 = Tool Radius	0.5 / 2 = .25
Overlap = .10	2.5 - .25 = 2.25
Circle Radius - Tool Radius + Overlap = **Max Offset**	2.25 + .10 = **2.35**

The result of using the Max Offset of 2.15, an XY Step of .25 and a Last Offset of .25 would look like this:

© 2011 CIM-TECH.COM, Inc



Multi-Pass Inside on circular shape.

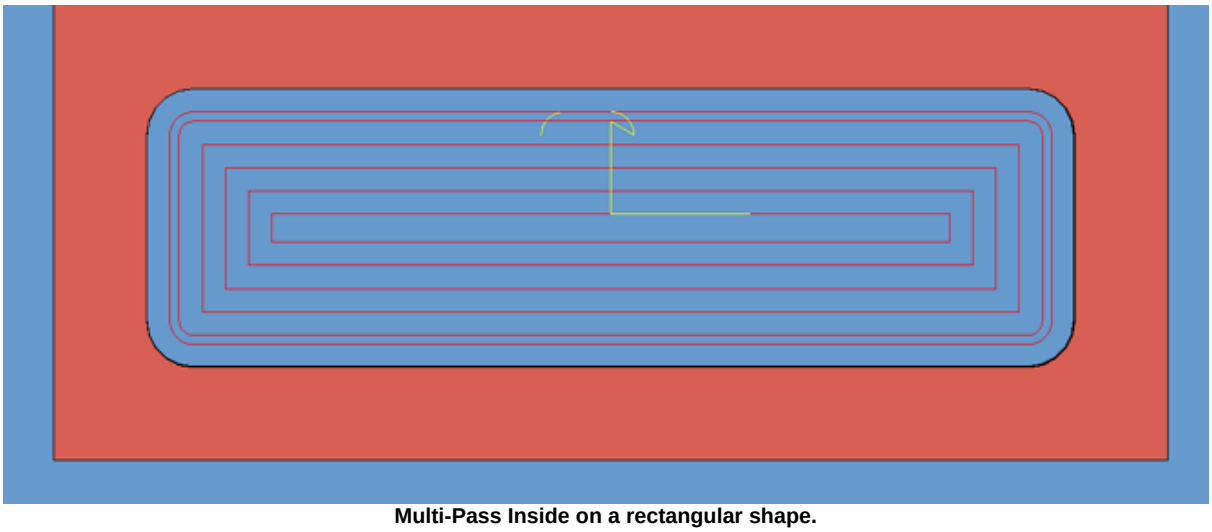
Rectangular Shapes

To calculate the distance of the Max Offset on a rectangular shape, you can take the width of the narrow side divided by 2 (to get to the middle of the shape) then subtract the tool radius and add the overlap amount you want on the first pass.
So, in example, to cut a 10" x 3" rectangular shape, you would need a Max Offset of about 1.35, using an overlap of .10 on the first pass.

Width of narrowest side / 2	3.0 / 2 = 1.5
Tool Diameter / 2	0.5 / 2 = .25
Overlap on first pass = .10	1.5 - .25 = 1.25
Width / 2 - Tool Radius + Overlap = **Max Offset**	1.25 + .10 = **1.35**

The result of using the Max Offset of 1.35, an XY Step of .25 and a Last Offset of .25 would look like this:

© 2011 CIM-TECH.COM, Inc



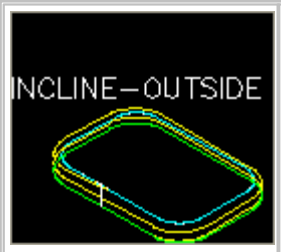
Multi-Pass Inside on a rectangular shape.

The reason for using a smaller first offset amount than the XY Step size or Last offset is only because you need to remove the overlap on the starting pass, each step after that will remove the overlap from the pass before it, so a smaller amount was used. Feel free to use any amount that is just short of the tool radius, as some tools are slightly undersized even when new, and worse as they wear.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

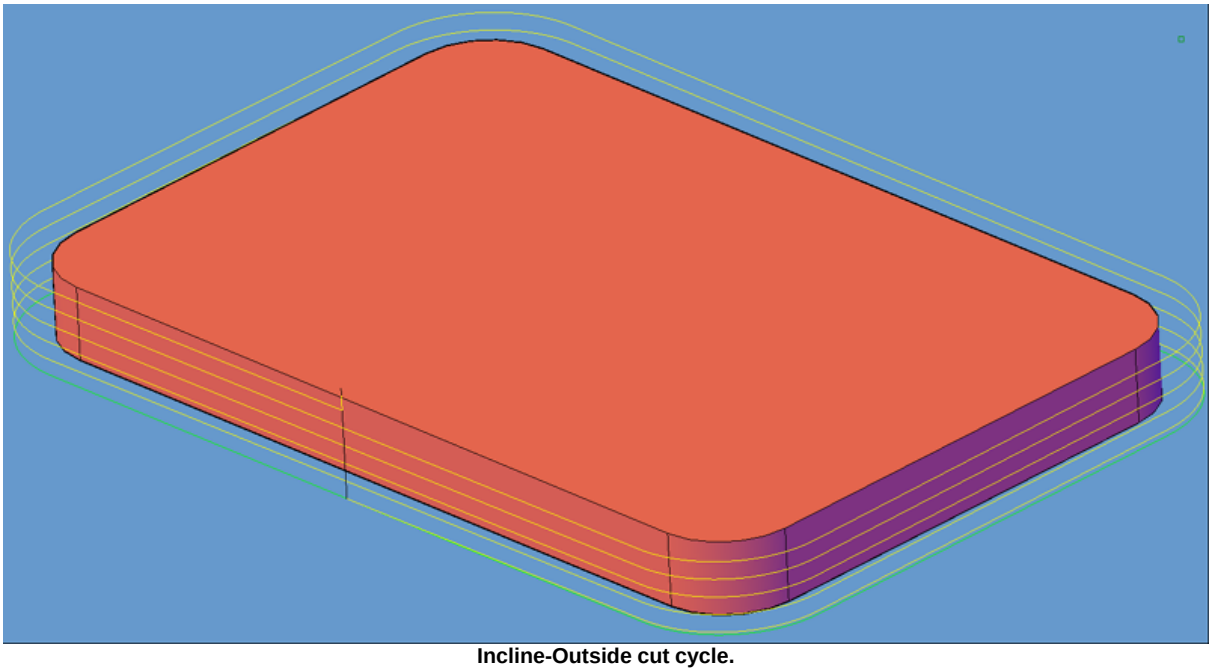
© 2011 CIM-TECH.COM, Inc

6.1.1.10 Incline-Outside



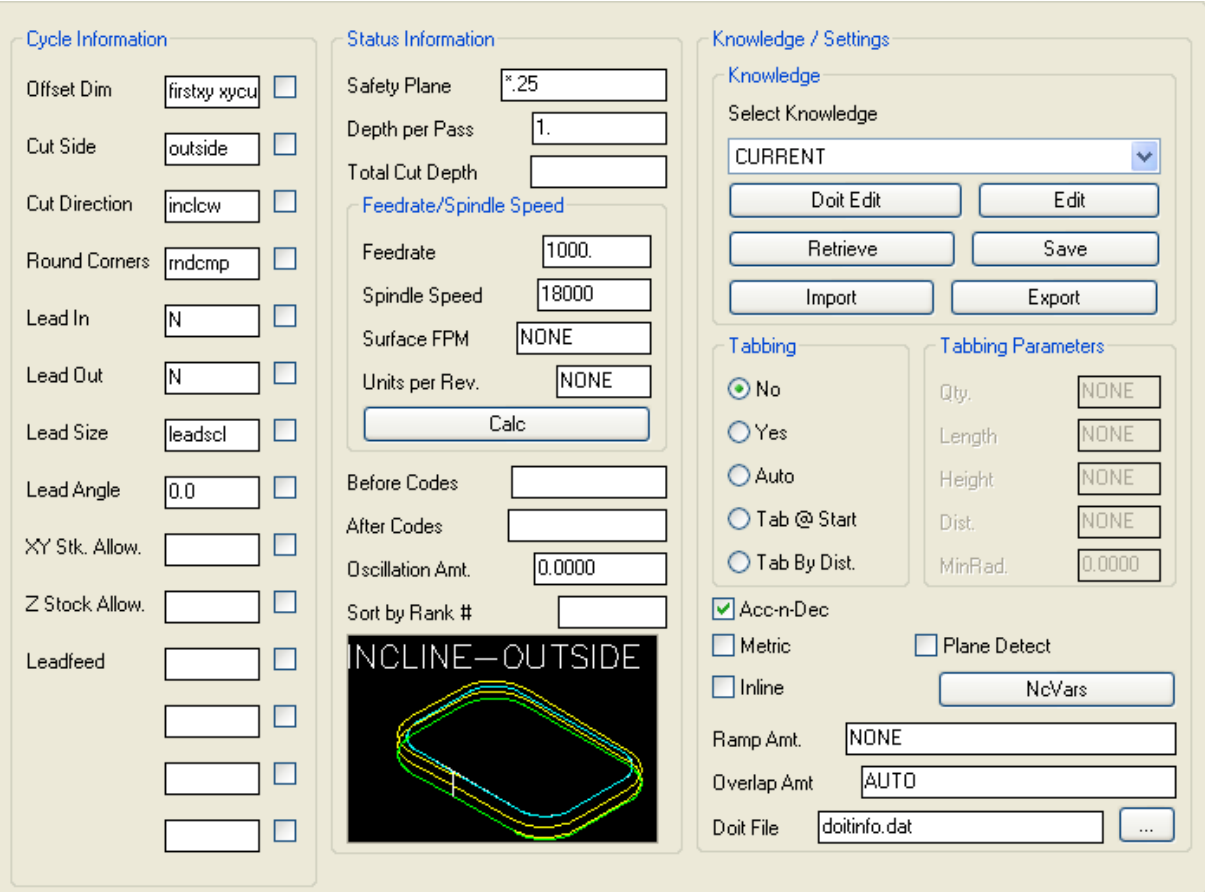
Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.
The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

Incline-Outside will default to cutting on the outside of a closed shape.



Incline-Outside cut cycle.

© 2011 CIM-TECH.COM, Inc



Incline-Outside cycle parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.
Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.
For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then 1.1 divided by .25 = 4.4. We have to round that number up to 5 otherwise using 4 would give us 1.1 divided by 4 which is .275 and that is more than the depth per pass. So rounding up to 5 gives us 1.1 divided by 5 which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you

© 2011 CIM-TECH.COM, Inc

give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

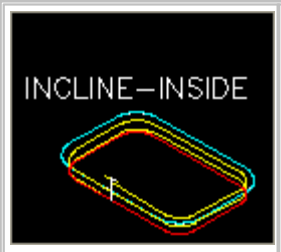
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

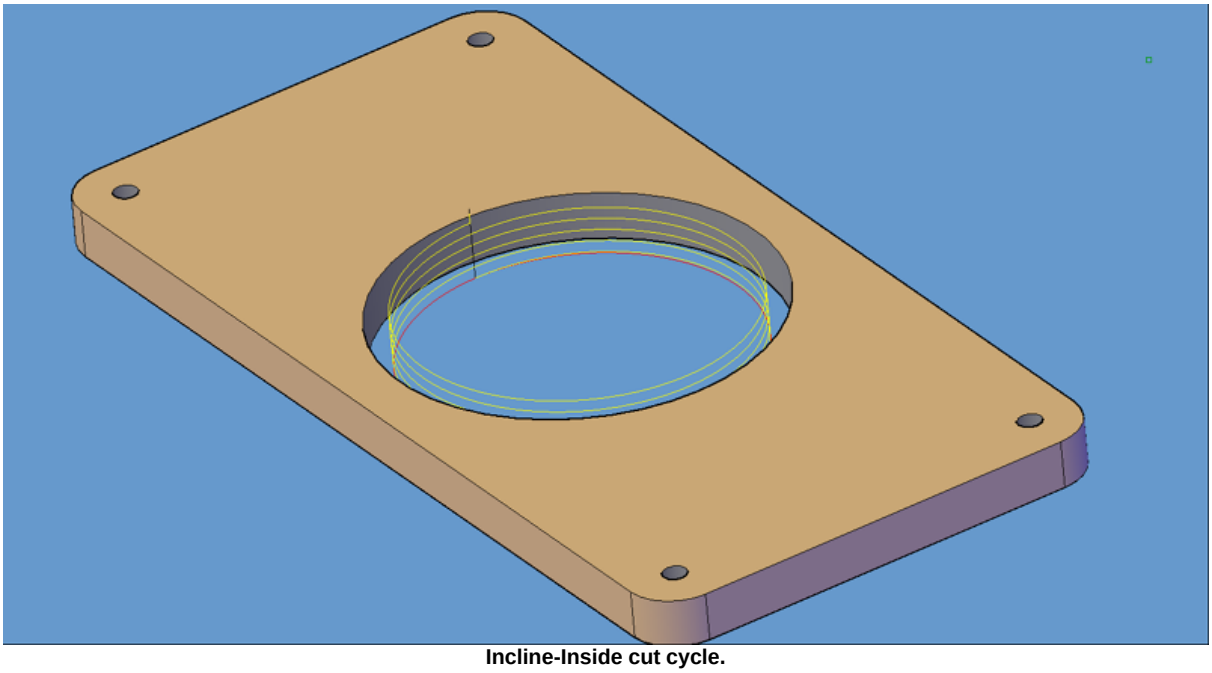
© 2011 CIM-TECH.COM, Inc

6.1.1.11 Incline-Inside



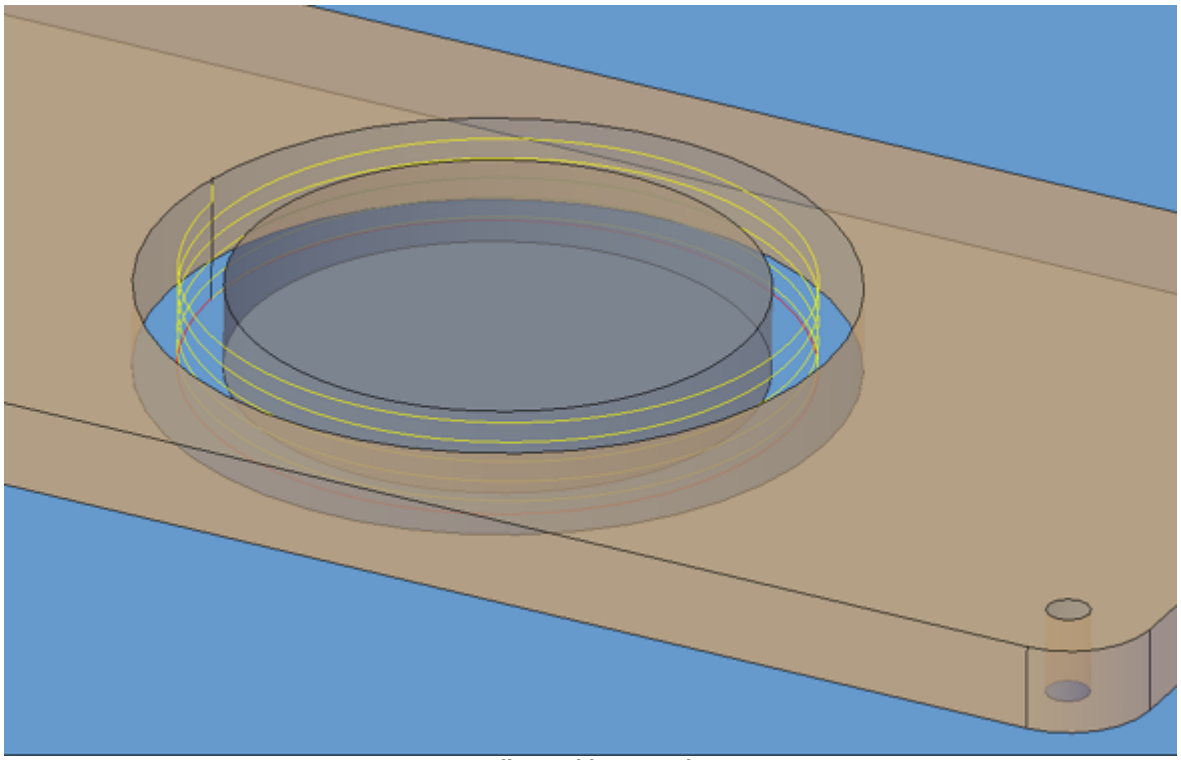
Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.
The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

Incline-Inside will default to cutting on the inside of a closed shape.



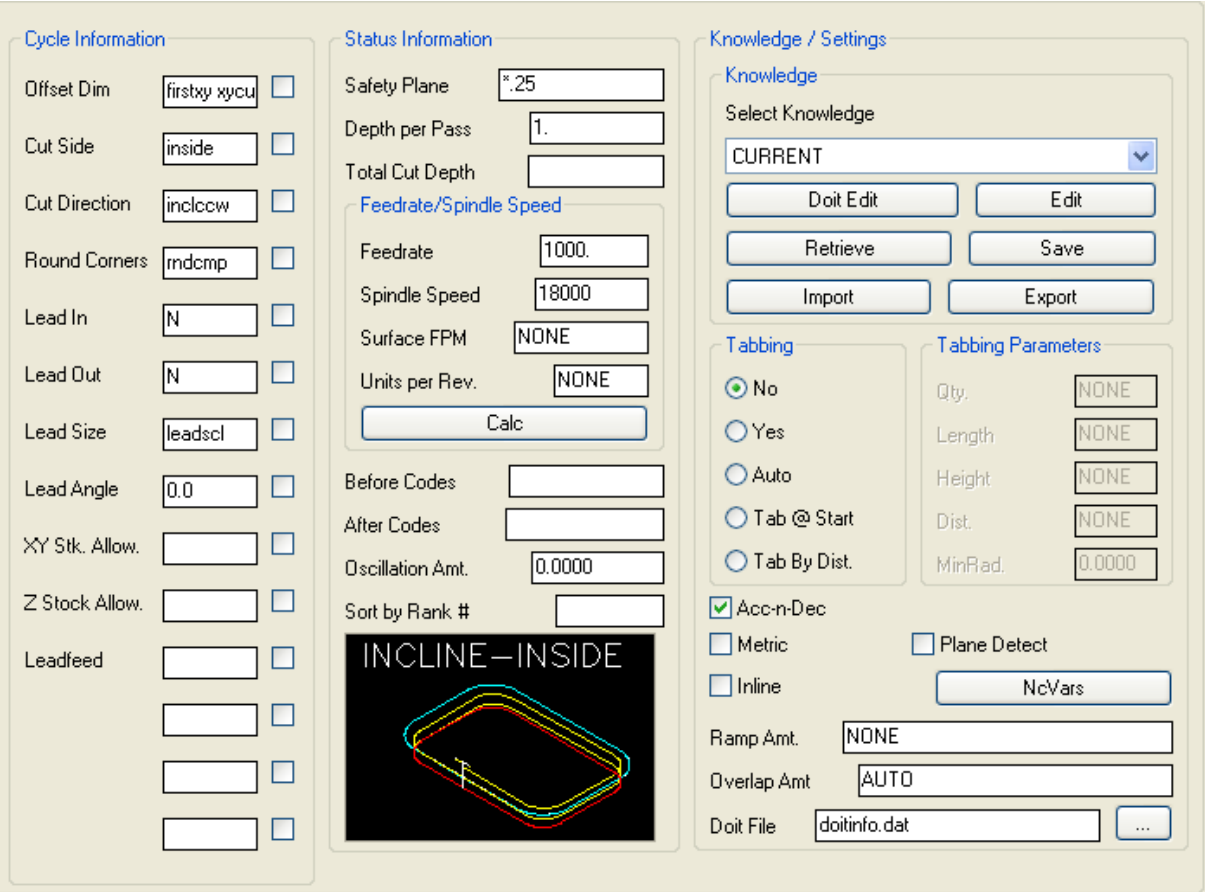
Incline-Inside cut cycle.

© 2011 CIM-TECH.COM, Inc



Incline-Inside cut cycle.

© 2011 CIM-TECH.COM, Inc



Incline-Inside cut parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.
Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.
For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then 1.1 divided by .25 = 4.4. We have to round that number up to 5 otherwise using 4 would give us 1.1 divided by 4 which is .275 and that is more than the depth per pass. So rounding up to 5 gives us 1.1 divided by 5 which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you

© 2011 CIM-TECH.COM, Inc

give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

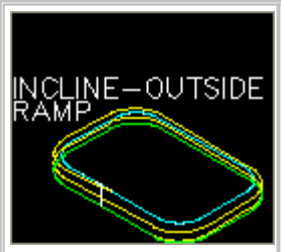
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

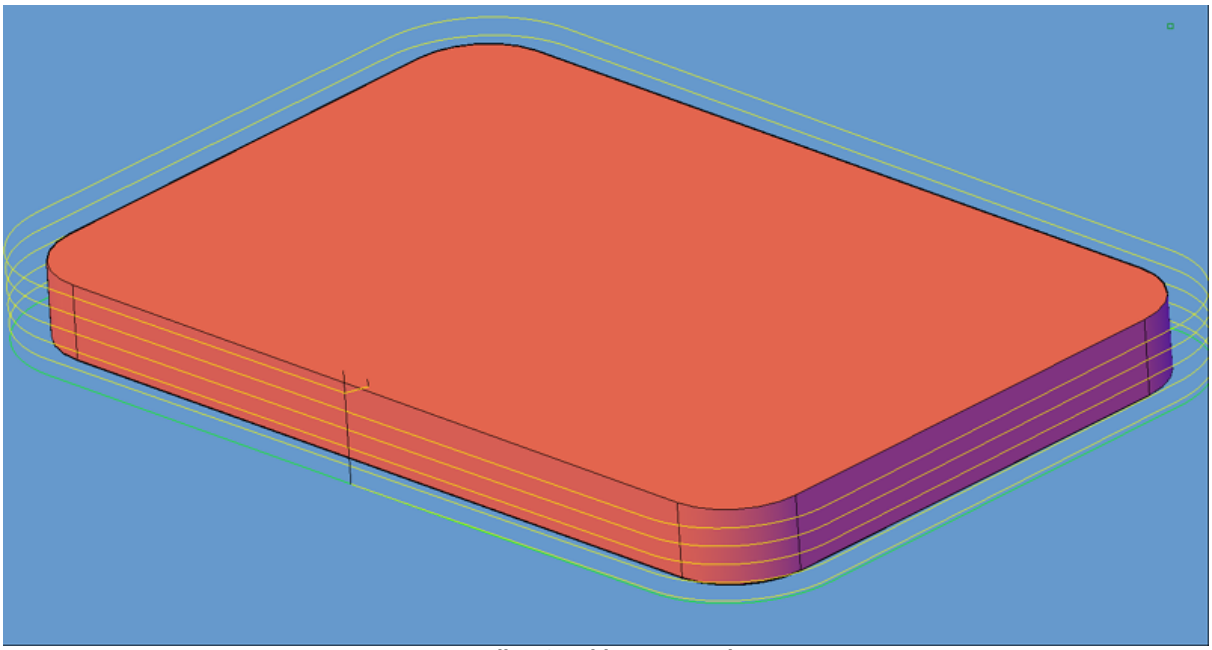
6.1.1.12 Incline-Outside-Ramp



Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.
The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

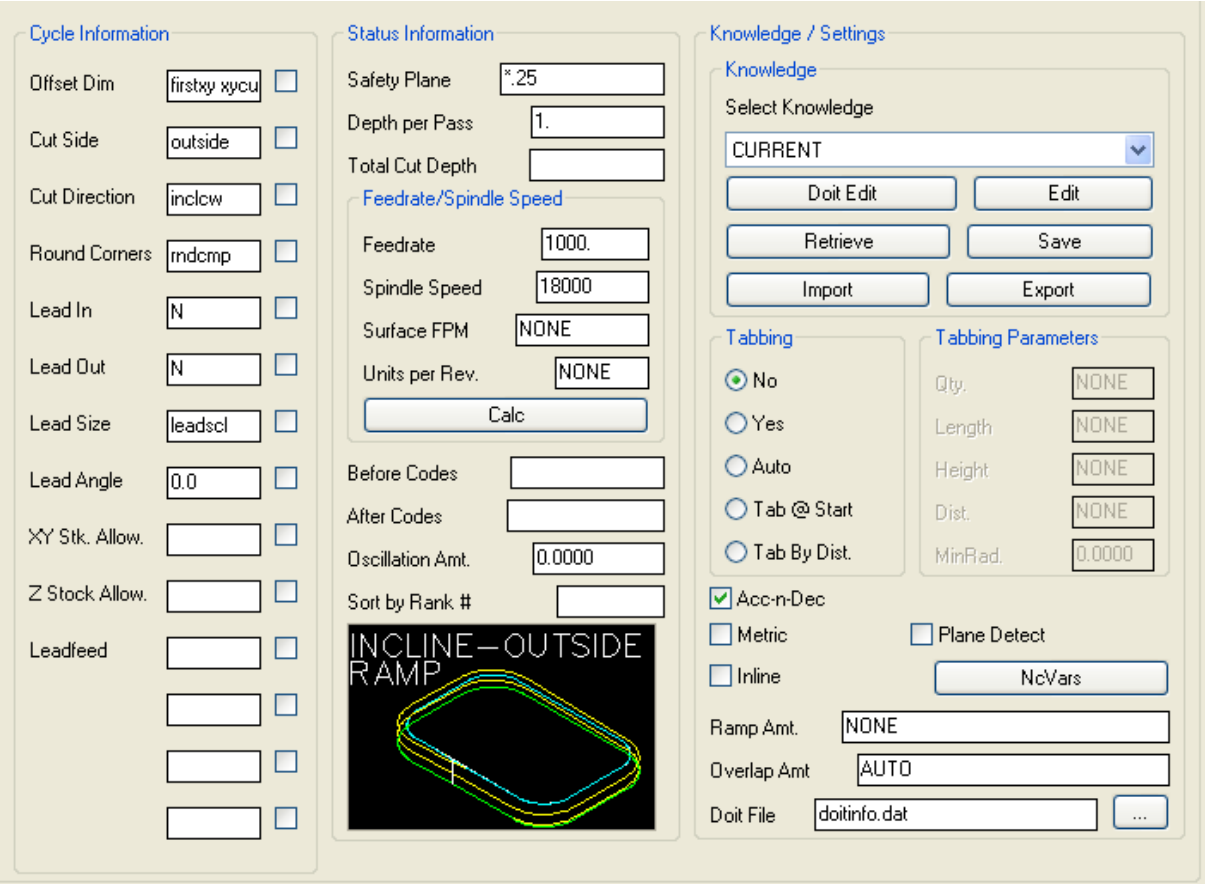
Incline-Outside will default to cutting on the outside of a closed shape.

The only difference between Incline-Outside and Incline-Outside Ramp is that the initial move is a ramping lead in to the start of the first pass instead of a plunge move. This is only necessary if the start of the cut will be inside of uncut material. Typically the start of the first cut is above the top of the part. If the start of the first cut cannot be above the top of the part, it may be better for the tool to make a ramp into the material instead of a plunge.



Incline-Outside Ramp cycle

© 2011 CIM-TECH.COM, Inc



Incline-Outside Ramp cycle parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part

© 2011 CIM-TECH.COM, Inc

that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.
Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.
For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then 1.1 divided by .25 = 4.4. We have to round that number up to 5 otherwise using 4 would give us 1.1 divided by 4 which is .275 and that is more than the depth per pass. So rounding up to 5 gives us 1.1 divided by 5 which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

© 2011 CIM-TECH.COM, Inc

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

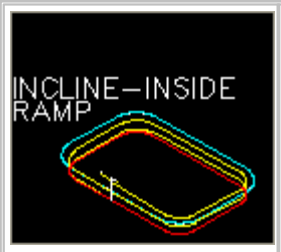
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

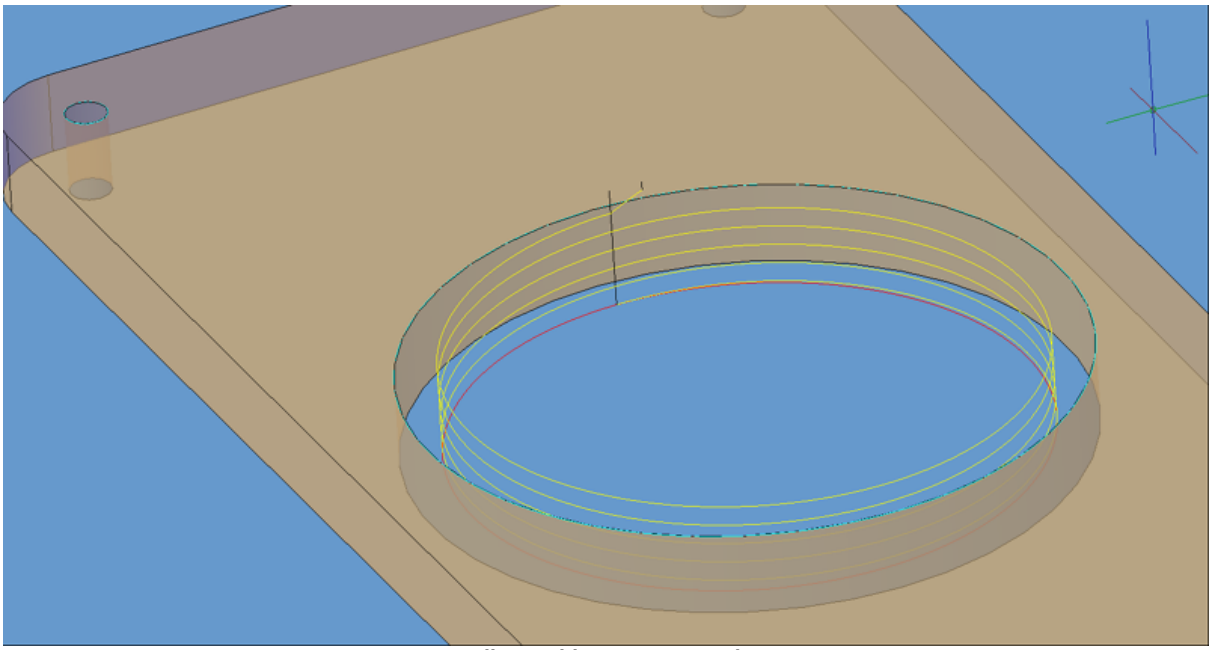
6.1.1.13 Incline-Inside-Ramp



Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.
The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

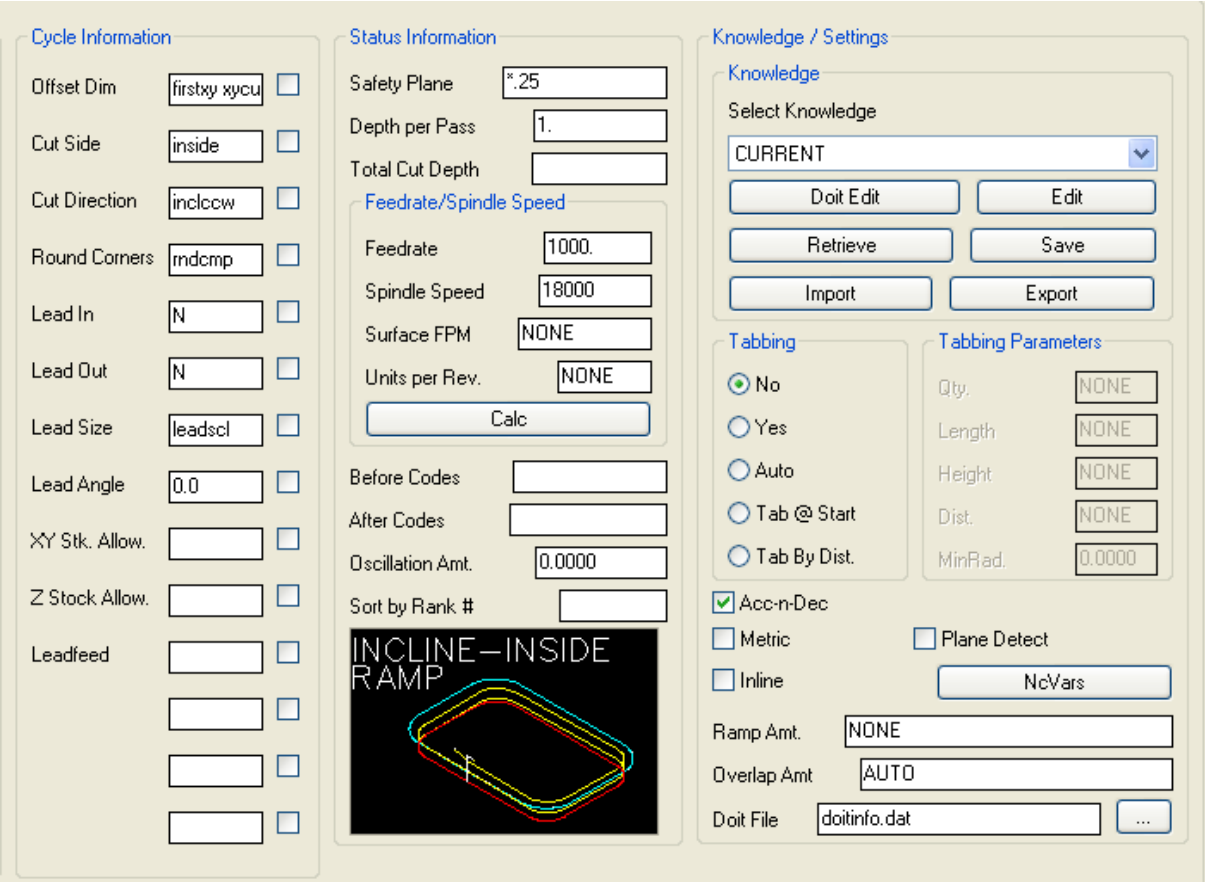
Incline-Inside Ramp will default to cutting on the inside of a closed shape.

The only difference between Incline-Inside and Incline-Inside Ramp is that the initial move is a ramping lead in to the start of the first pass instead of a plunge move. This is only necessary if the start of the cut will be inside of uncut material. Typically the start of the first cut is above the top of the part. If the start of the first cut cannot be above the top of the part, it may be better for the tool to make a ramp into the material instead of a plunge.



Incline-Inside Ramp cut cycle.

© 2011 CIM-TECH.COM, Inc



Incline-Inside Ramp cut parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part

© 2011 CIM-TECH.COM, Inc

that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.
Since this cycle is a constant ramp, for the entire length of the cut, there is a little more involved with the depth per pass. If the tool is starting from a point above the part (if tool tip distance to zero is set to any value above 0, which it normally is) then you should take that amount and add it to the total depth and then divide that number by the depth per pass. If you get an even number (no decimal) then that will be the number of passes, otherwise, round up to the next number of passes and then divide the total depth by that number. The result is the depth per pass for each level of the ramp.
For instance, if the tool tip distance to zero is .1 (in the tool parameters) and the total depth is -1.0, then the total movement of the tool is 1.1. If the depth per pass is set to .25, then 1.1 divided by .25 = 4.4. We have to round that number up to 5 otherwise using 4 would give us 1.1 divided by 4 which is .275 and that is more than the depth per pass. So rounding up to 5 gives us 1.1 divided by 5 which is .22 and that is less than the .25 depth per pass, and so you will see 5 levels to the total cut at a spacing of .22 before the final finish pass.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

© 2011 CIM-TECH.COM, Inc

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

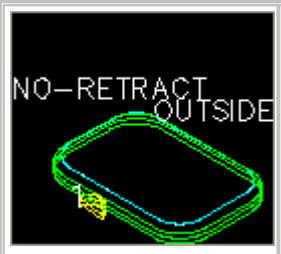
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

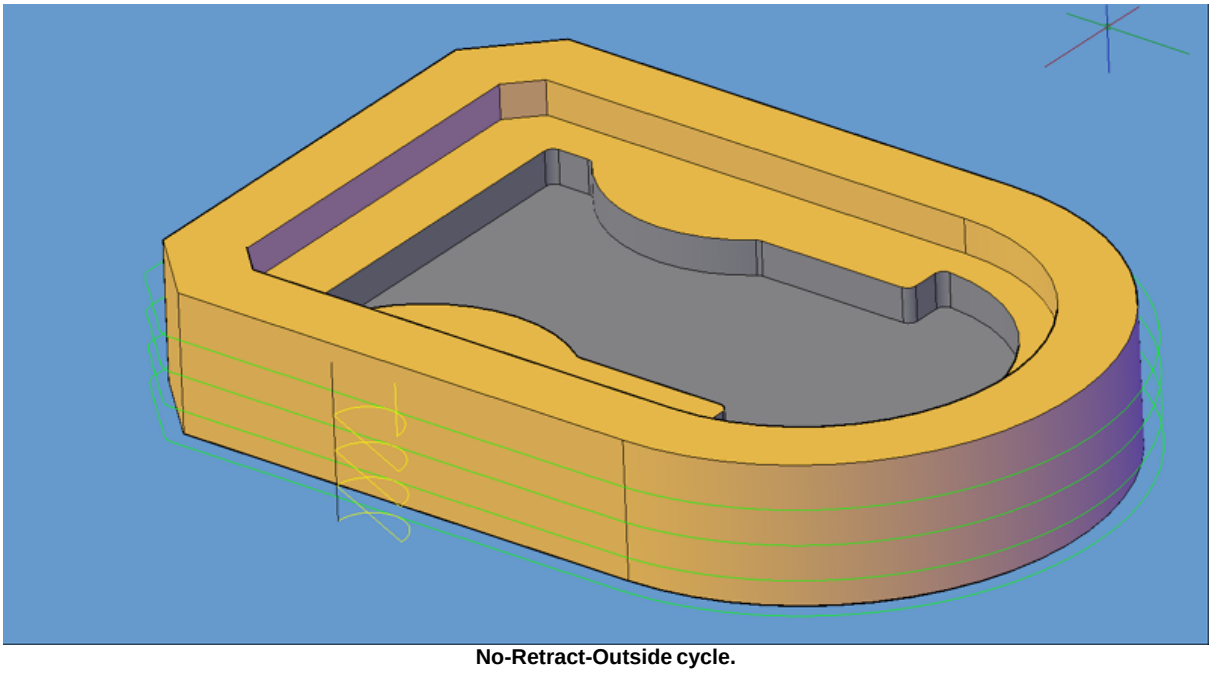
© 2011 CIM-TECH.COM, Inc

6.1.1.14 No-Retract-Outside



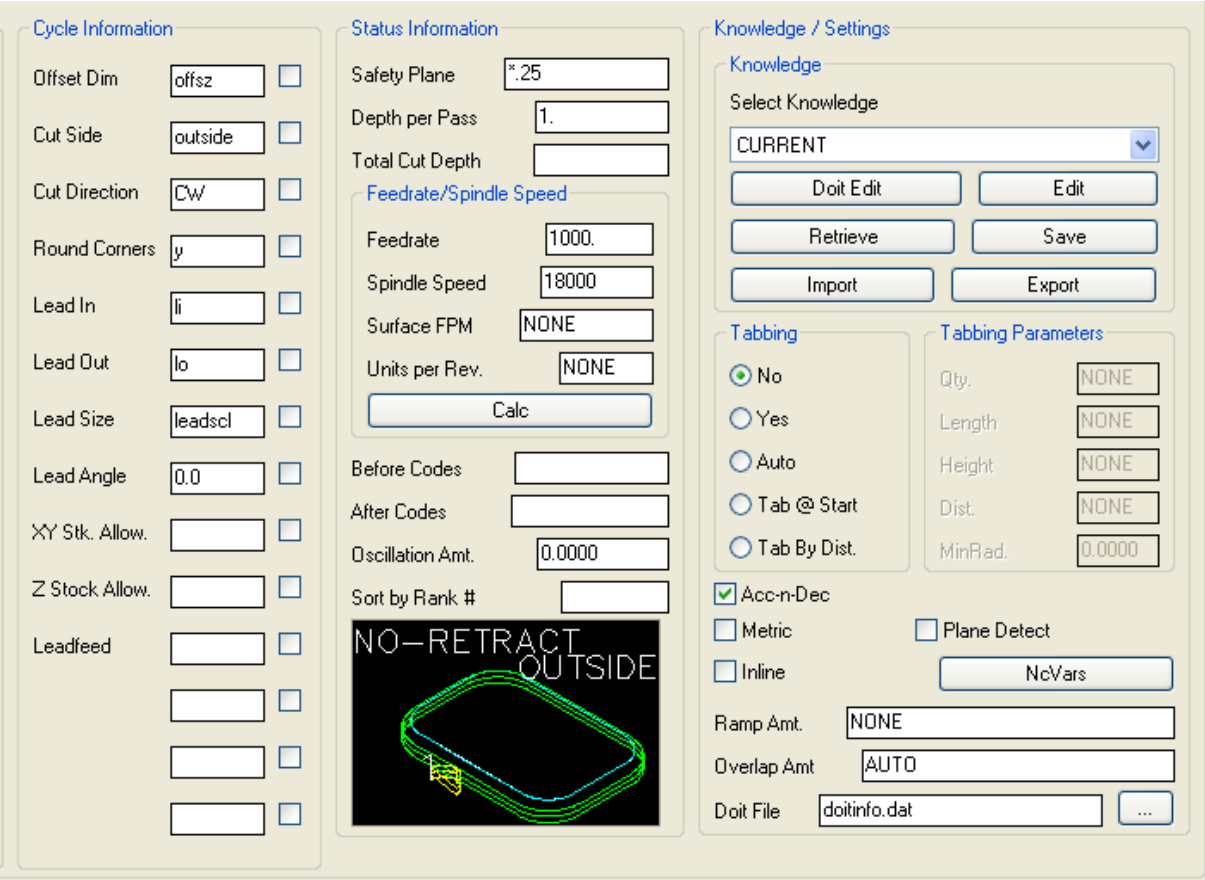
The No-Retract Outside cycle is a profile cutting cycle meant for cuts that need multiple passes in Z. This cycle uses a separate lead-in and lead-out per pass, but instead of retracting the cutter to the Safety Plane between each pass in Z, the tool will move from the end of the lead-out and ramp down to the start of the lead-in on the next pass. Each pass in Z will contain a 90° lead-in and lead-out, and the ramp down between each pass will be performed at the programmed feedrate.

There is no overlap on this cycle by default, as the cutter is constantly overcutting the start point, but an overlap is allowed to be specified in the parameters if desired. This cycle should only be used with multiple depths per pass, as there is no benefit to using it on a single pass in Z.



No-Retract-Outside cycle.

© 2011 CIM-TECH.COM, Inc



No-Retract-Outside parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a

© 2011 CIM-TECH.COM, Inc

negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

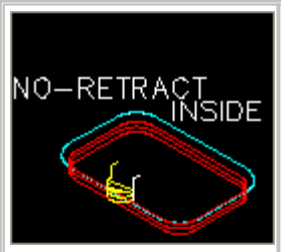
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

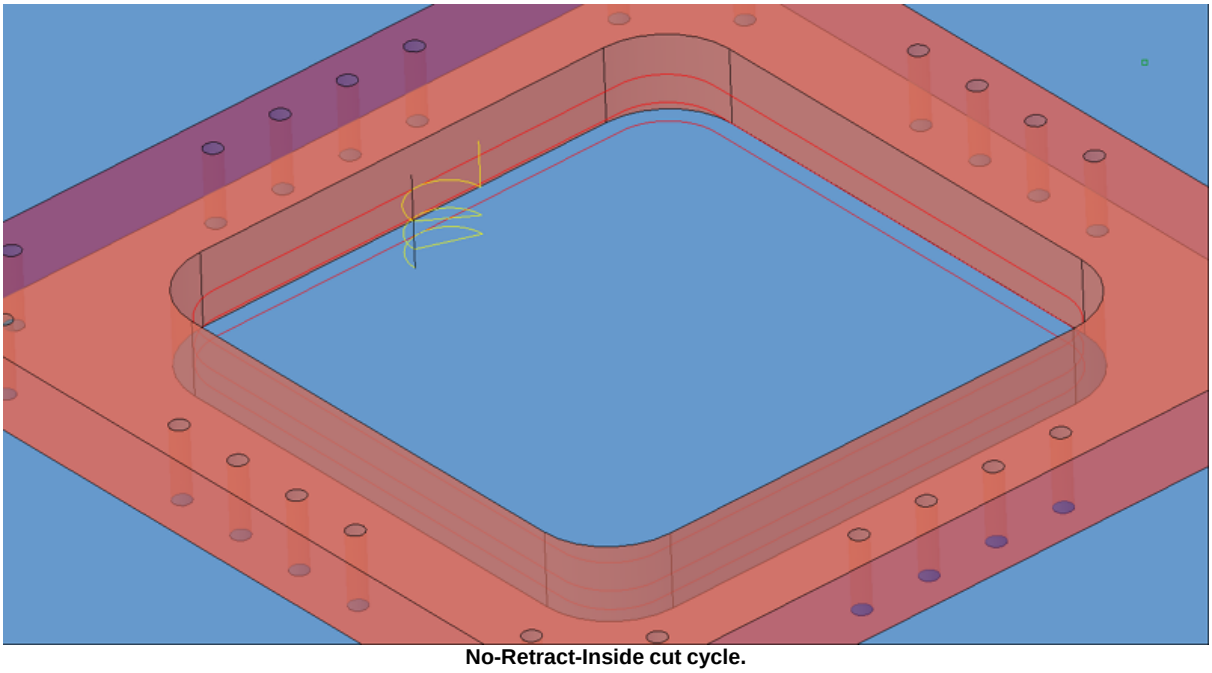
© 2011 CIM-TECH.COM, Inc

6.1.1.15 No-Retract-Inside



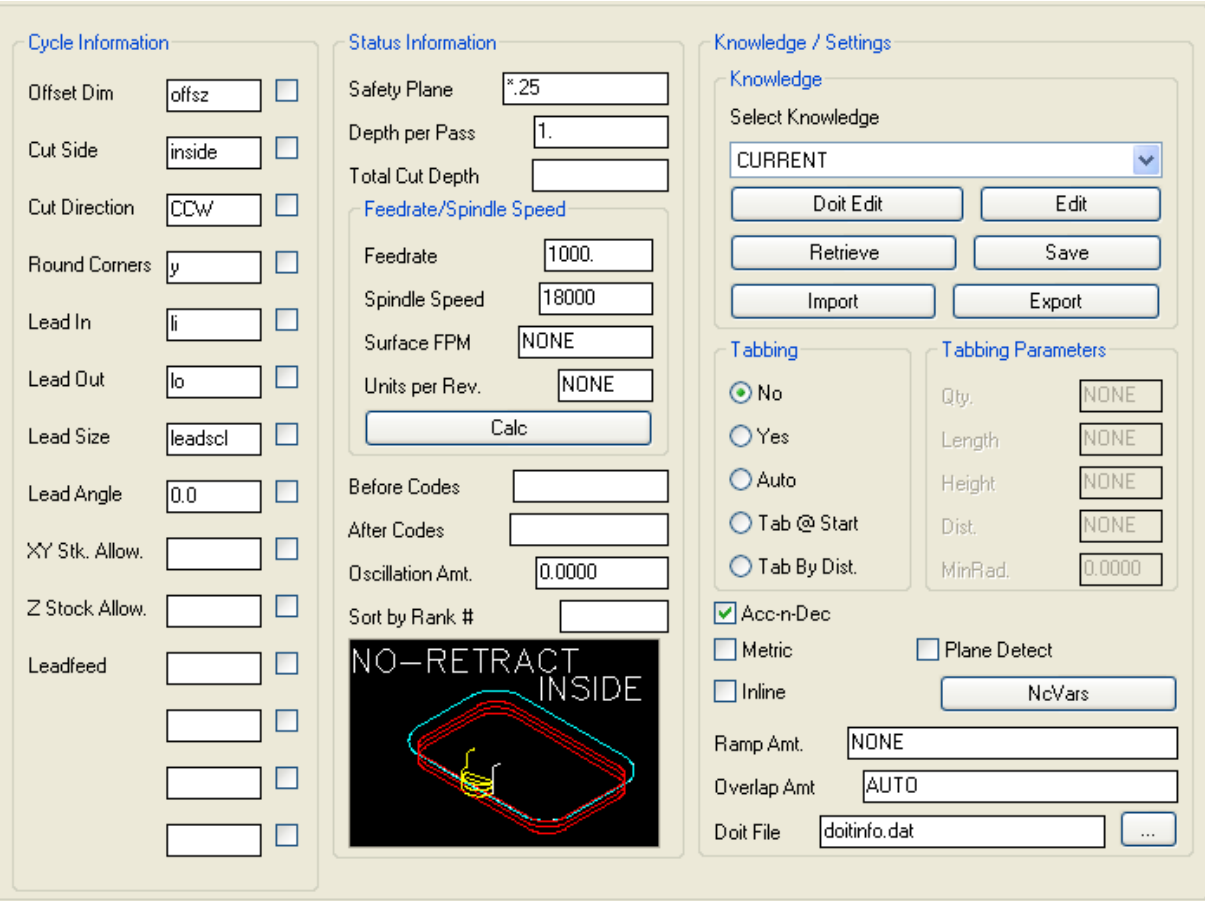
The No-Retract Inside cycle is a profile cutting cycle meant for cuts that need multiple passes in Z. This cycle uses a separate lead-in and lead-out per pass, but instead of retracting the cutter to the Safety Plane between each pass in Z, the tool will move from the end of the lead-out and ramp down to the start of the lead-in on the next pass. Each pass in Z will contain a 90° lead-in and lead-out, and the ramp down between each pass will be performed at the programmed feedrate.

There is no overlap on this cycle by default, as the cutter is constantly overcutting the start point, but an overlap is allowed to be specified in the parameters if desired. This cycle should only be used with multiple depths per pass, as there is no benefit to using it on a single pass in Z.



No-Retract-Inside cut cycle.

© 2011 CIM-TECH.COM, Inc



No-Retract-Inside parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a

© 2011 CIM-TECH.COM, Inc

negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

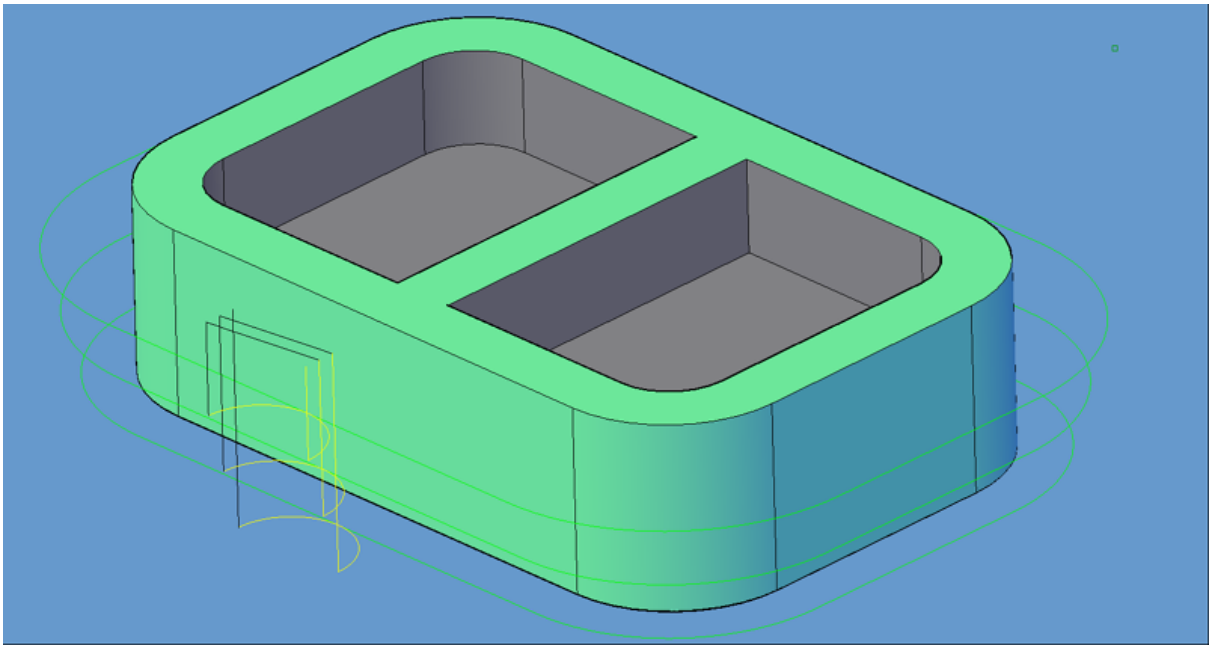
© 2011 CIM-TECH.COM, Inc

6.1.1.16 Taper-Plunge-Outside

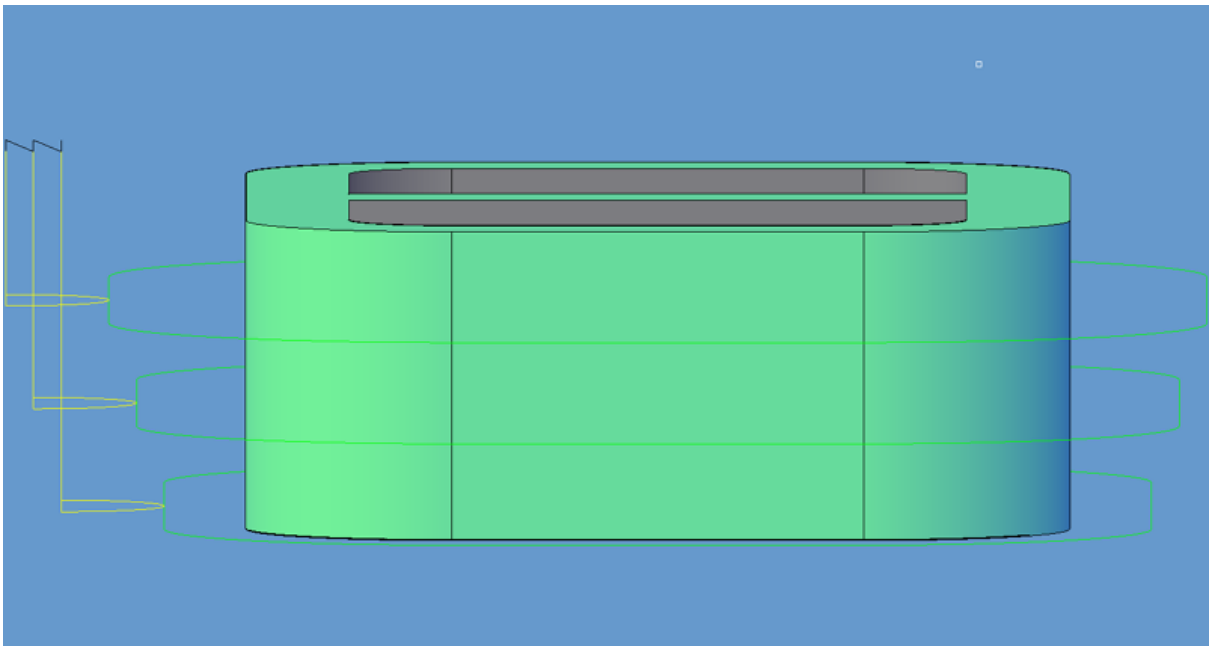


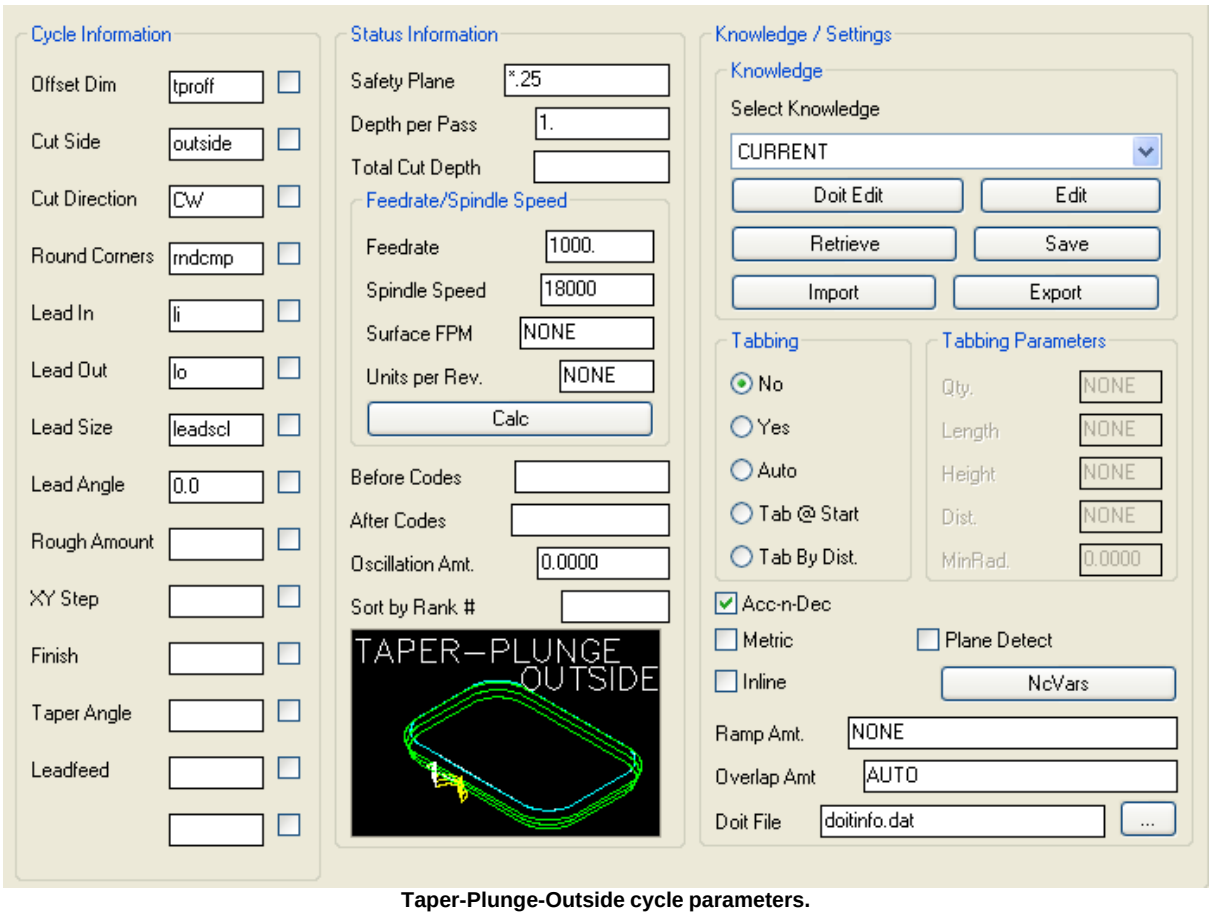
Taper-Plunge cutting is a method of rough and finish cutting with the same tool where the roughing is not only in XY but in Z as well. The cycle will start at the Safety Plane and plunge to the first cut depth and then make a 90° lead-in to make the first pass. The offset of the first pass is typically offset in XY away from the edge of the finished shape. After the first pass takes place, the cutter makes a 90° lead out, retracts the cutter and then moves over to the start of the second cut, which will be closer in XY to the finished edge and lower in Z. It will make the same lead-in move, cut the profile shape and then lead-out and retract, next moving over closer to the finished edge in XY and then plunging in Z down to the next depth. Depending on the set parameters this can occur many times, each time stepping in and down closer to the finished shape of the part, which will be the last pass.

You must use multiple depths per pass to have the cycle parameters create any effect on the tool path.



© 2011 CIM-TECH.COM, Inc





Taper-Plunge-Outside cycle parameters.

The following parameters effect the toolpath creation:

© 2011 CIM-TECH.COM, Inc

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

© 2011 CIM-TECH.COM, Inc

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Step

This parameters specifies how far each pass is from one another in the XY plane. Even though the cuts can step in the Z axis, they have an XY step over amount as this would control the amount of material removed by the tool on each pass.
The Z axis step is controlled by the Depth per Pass and Taper Angle.

Finish

Finish is the offset amount from the edge of the part where the finish pass is placed, in the XY plane. Typically for a finish cut this would be the radius of the tool, for a roughing cut it would be the tool radius plus the amount of material you desire to leave for the finish cutter to remove.

Taper Angle

The angle in degrees to try and step each cut as they get closer or further away from the part. This would be the angle between the passes in Z.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

© 2011 CIM-TECH.COM, Inc

Depth Per Pass
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth
The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce

© 2011 CIM-TECH.COM, Inc

any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

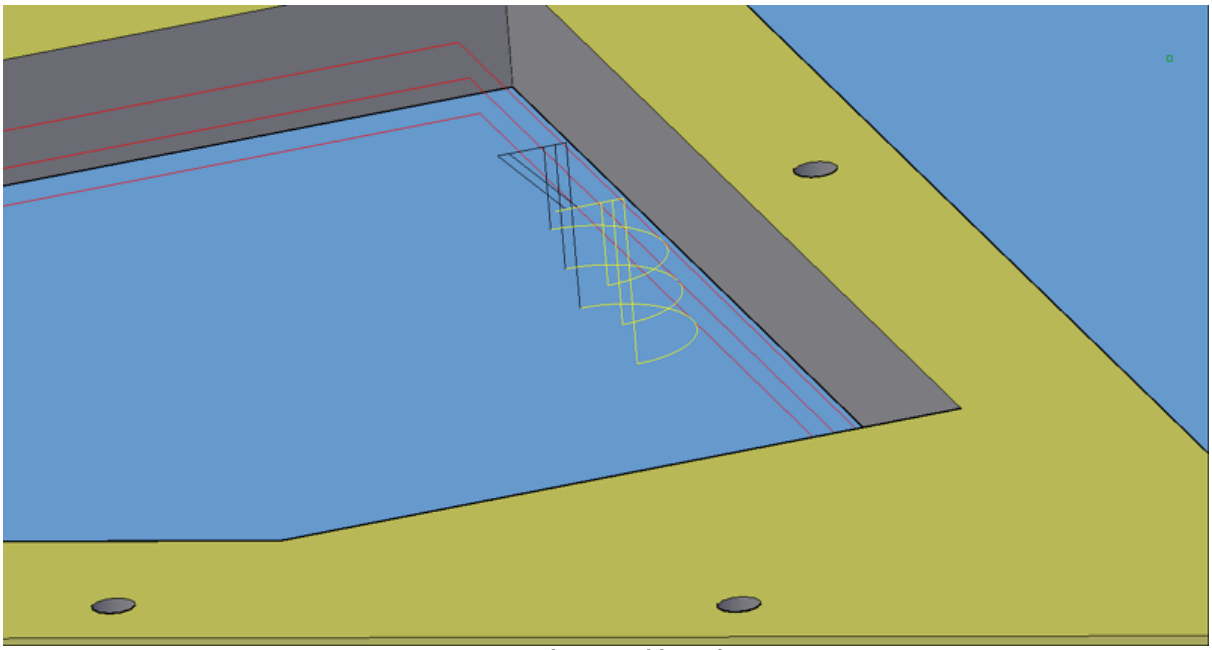
© 2011 CIM-TECH.COM, Inc

6.1.1.17 Taper-Plunge-Inside



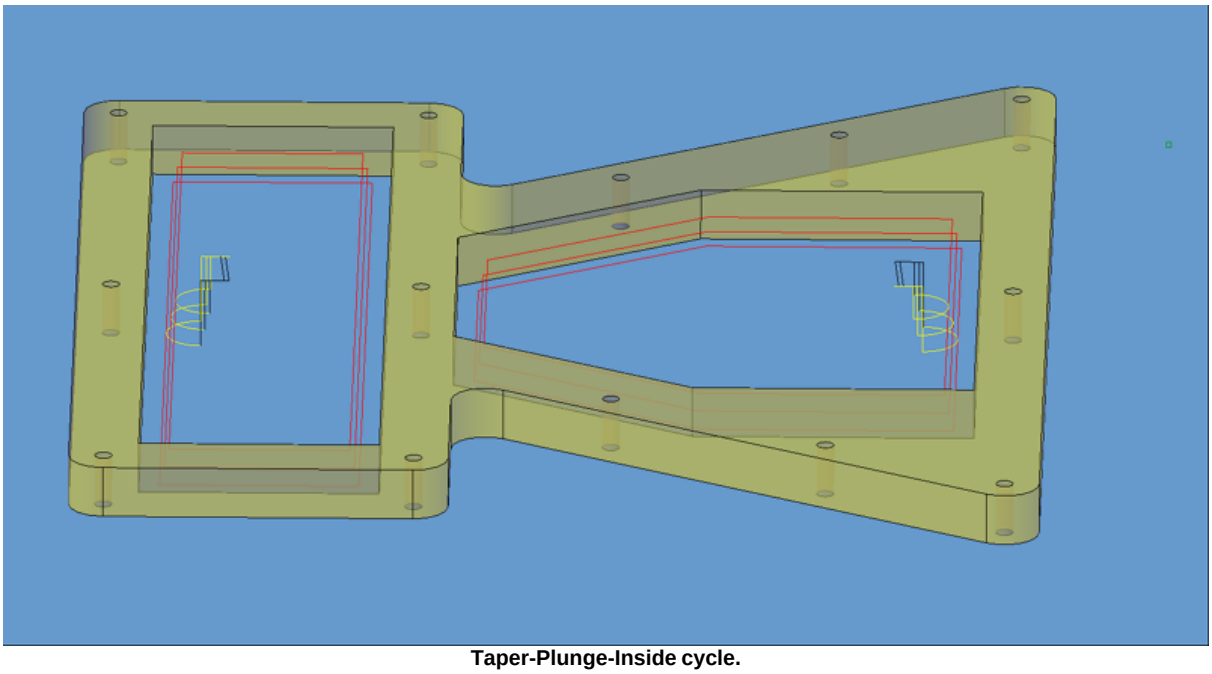
Taper-Plunge cutting is a method of rough and finish cutting with the same tool where the roughing is not only in XY but in Z as well. The cycle will start at the Safety Plane and plunge to the first cut depth and then make a 90° lead-in to make the first pass. The offset of the first pass is typically offset in XY away from the edge of the finished shape. After the first pass takes place, the cutter makes a 90° lead out, retracts the cutter and then moves over to the start of the second cut, which will be closer in XY to the finished edge and lower in Z. It will make the same lead-in move, cut the profile shape and then lead-out and retract, next moving over closer to the finished edge in XY and then plunging in Z down to the next depth. Depending on the set parameters this can occur many times, each time stepping in and down closer to the finished shape of the part, which will be the last pass.

You must use multiple depths per pass to have the cycle parameters create any effect on the tool path.

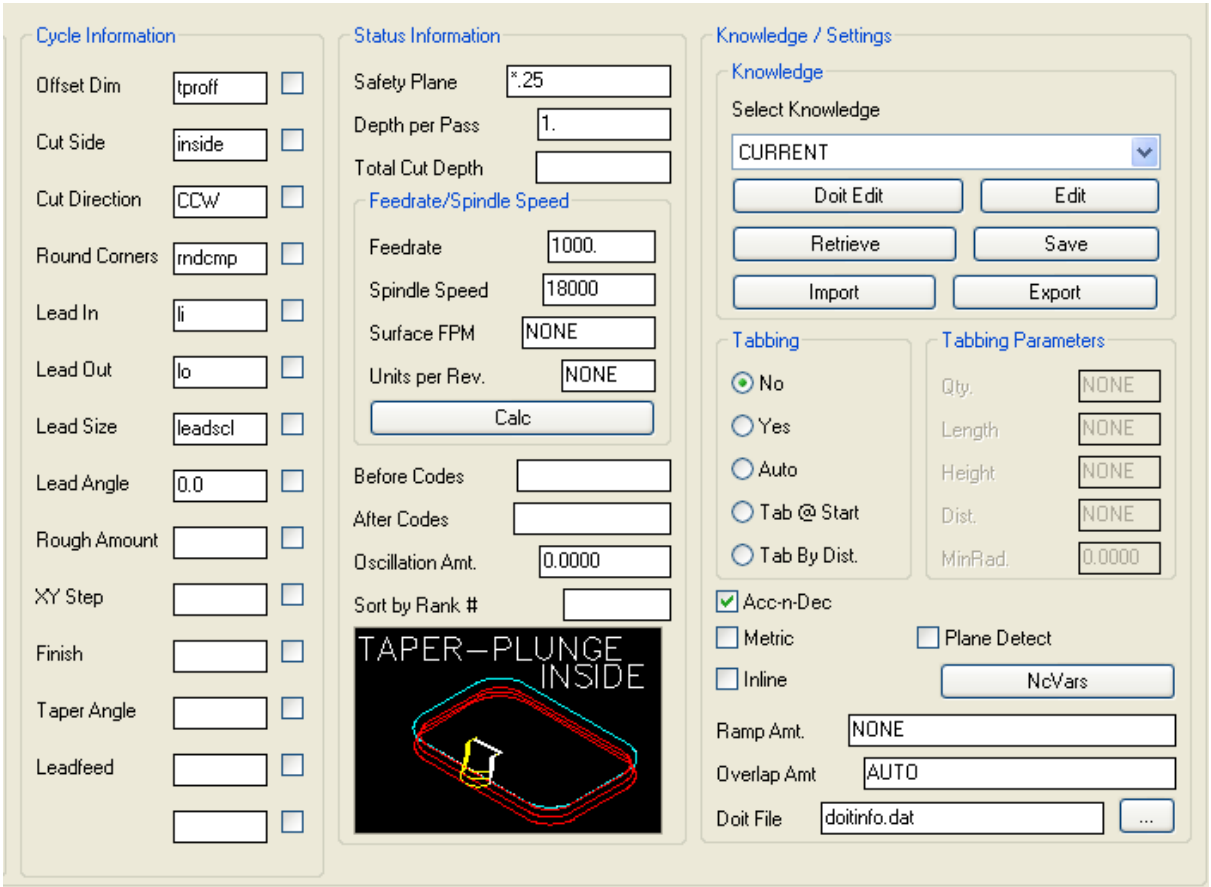


Taper-Plunge-Inside cycle.

© 2011 CIM-TECH.COM, Inc



Taper-Plunge-Inside cycle.



Taper-Plunge-Inside cycle parameters.

© 2011 CIM-TECH.COM, Inc

The following parameters effect the toolpath creation:

Offset Dim
The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side
Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will

© 2011 CIM-TECH.COM, Inc

respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Step
This parameters specifies how far each pass is from one another in the XY plane. Even though the cuts can step in the Z axis, they have an XY step over amount as this would control the amount of material removed by the tool on each pass.
The Z axis step is controlled by the Depth per Pass and Taper Angle.

Finish
Finish is the offset amount from the edge of the part where the finish pass is placed, in the XY plane. Typically for a finish cut this would be the radius of the tool, for a roughing cut it would be the tool radius plus the amount of material you desire to leave for the finish cutter to remove.

Taper Angle
The angle in degrees to try and step each cut as they get closer or further away from the part. This would be the angle between the passes in Z.

Lead Feed
This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

© 2011 CIM-TECH.COM, Inc

See the Safety Plane section for more information.

Depth Per Pass
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth
The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount

© 2011 CIM-TECH.COM, Inc

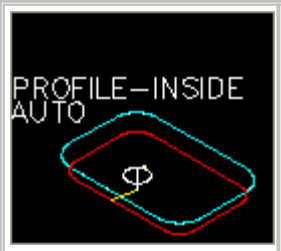
is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

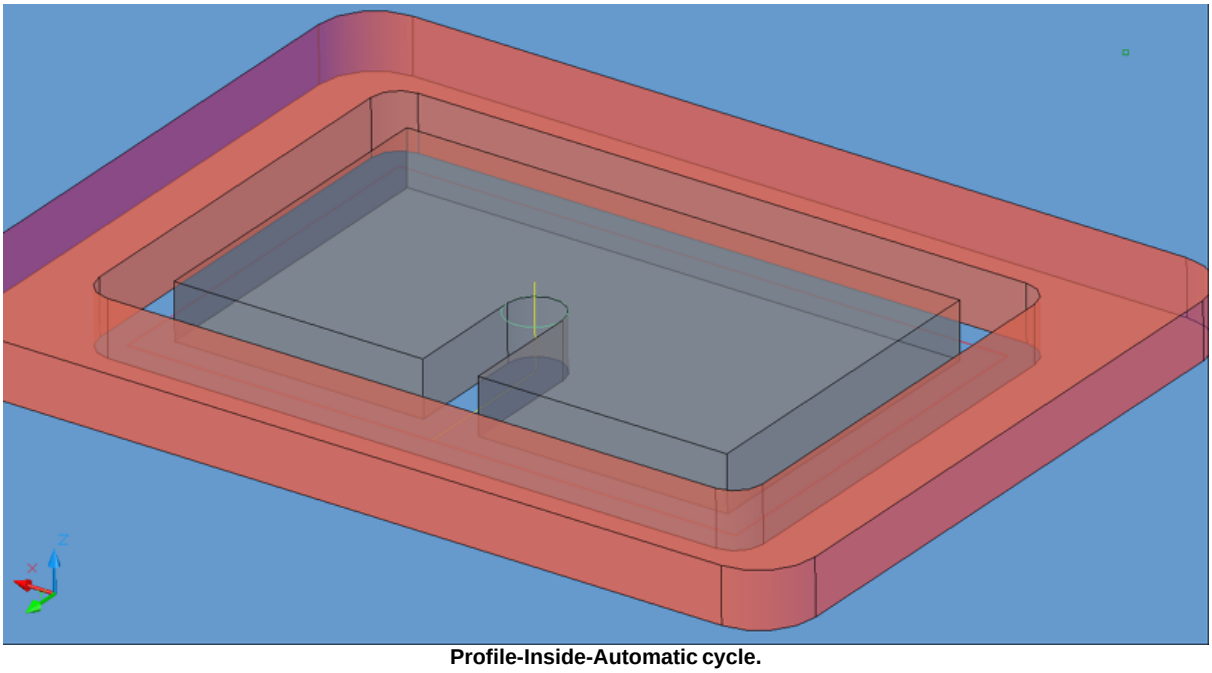
© 2011 CIM-TECH.COM, Inc

6.1.1.18 Profile-Inside-Auto



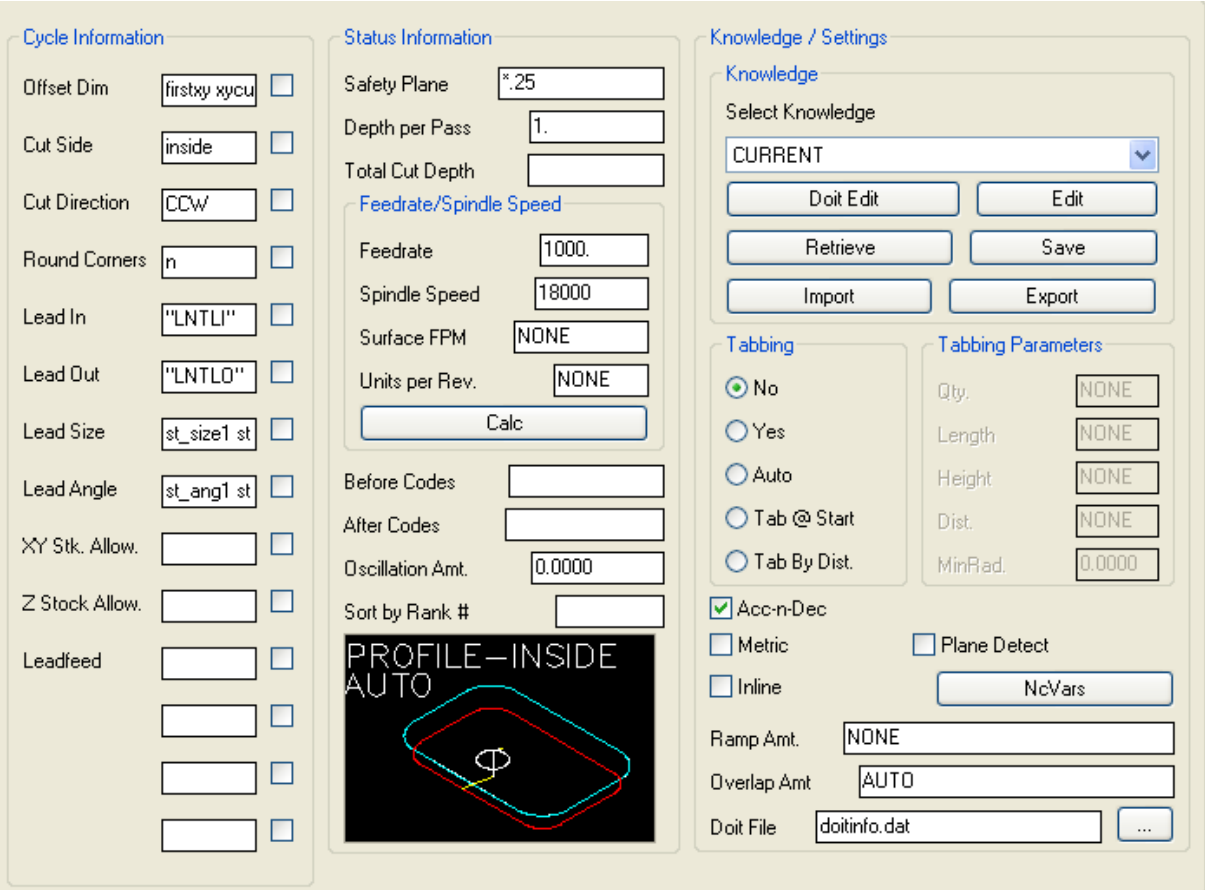
Profile-Inside cycles will start the cut in a hole that has been created for the purpose of getting the tool down to the proper cut depth without plunging the cutter into the material. The cycle will start at the Safety Plane, plunge down to the Total Cut Depth inside the hole that has been created for that purpose. Then the cutter will machine a path to the start point of the shape and cut around the inside of the profile back to the start point and then back to the lead hole where it started and at that point it will retract back up to the Safety Plane.

Profile-Inside Auto requires a circle on layer 'CSTART' in order to automatically have a starting hole location. The circle on layer CSTART should be drilled or milled out before this cut cycle occurs, insuring that the cutter has a safe place to plunge.



Profile-Inside-Automatic cycle.

© 2011 CIM-TECH.COM, Inc



Profile-Inside-Automatic parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

© 2011 CIM-TECH.COM, Inc

See the Cut Side section for more information.

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance

© 2011 CIM-TECH.COM, Inc

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

© 2011 CIM-TECH.COM, Inc

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

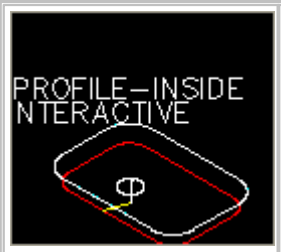
Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

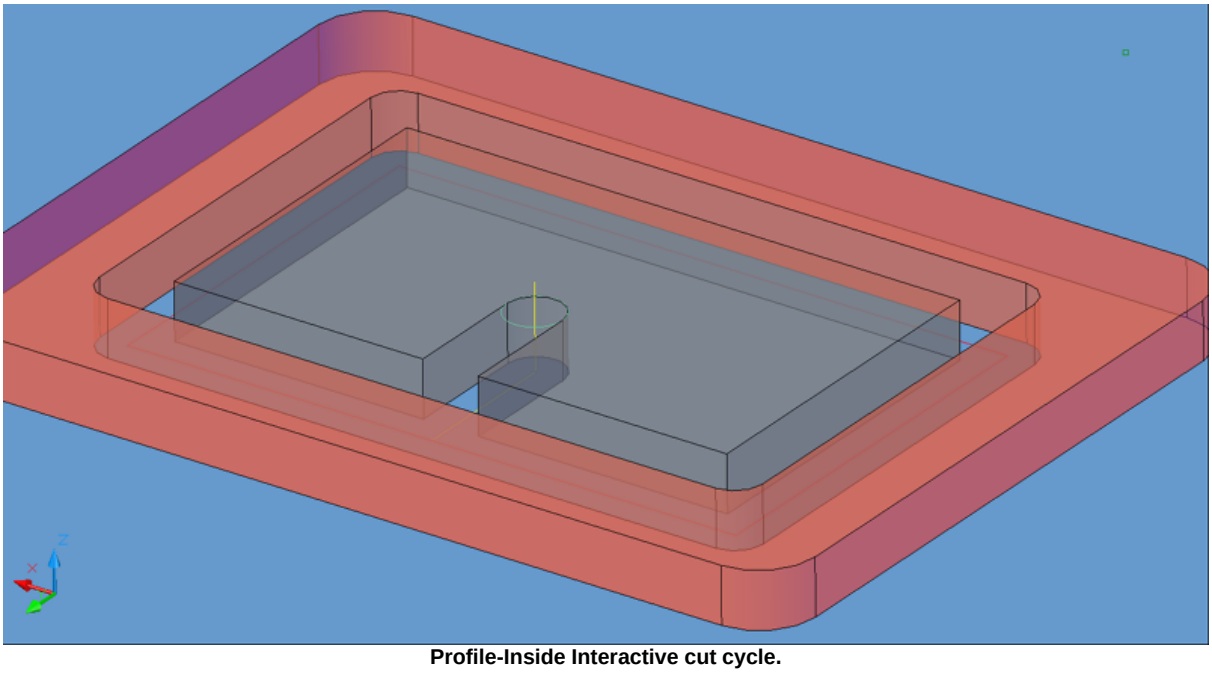
© 2011 CIM-TECH.COM, Inc

6.1.1.19 Profile-Inside-Interactive



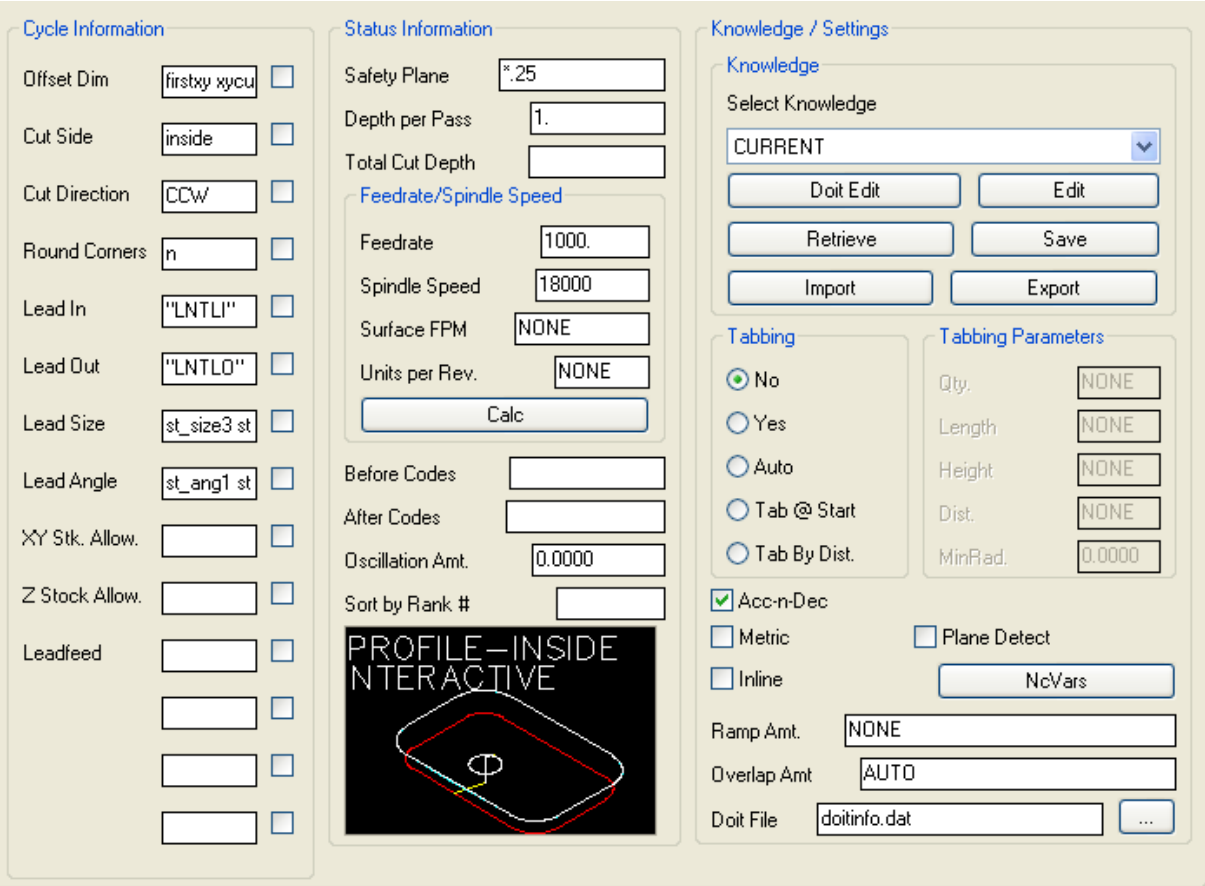
Profile-Inside cycles will start the cut in a hole that has been created for the purpose of getting the tool down to the proper cut depth without plunging the cutter into the material. The cycle will start at the Safety Plane, plunge down to the Total Cut Depth inside the hole that has been created for that purpose. Then the cutter will machine a path to the start point of the shape and cut around the inside of the profile back to the start point and then back to the lead hole where it started and at that point it will retract back up to the Safety Plane.

Profile-Inside Interactive requires you to pick the center of a hole to start in during the cut. The circle on should be drilled or milled out before this cut cycle occurs, insuring that the cutter has a safe place to plunge.



Profile-Inside Interactive cut cycle.

© 2011 CIM-TECH.COM, Inc



Profile-Inside-Interactive parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case

© 2011 CIM-TECH.COM, Inc

the part would be cut to -.76.

Feedrate
This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequencer to place cuts in a specific order when the code is created.

Overlap Amt
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.2 Drilling and Hole Cutting Cycles

There are various drilling and hole cutting cycles inside Router-CIM. These cycles are designed with specific purposes and are all explained in as much detail as possible to insure you can select the ideal cycle for the task at hand.

There are different cycles for drilling than there are for hole cutting as there are instances where you must machine a hole with a router-bit or end mill instead of drilling with a drill-bit. These cycles are meant for the instances where the job is performed with a tool suitable to remove all the material in the hole and not leave a slug which could be ejected from the material and cause injury.
If it is not possible to cut the hole with a large enough tool to remove the slug, sometimes it is better to pocket cut the slug first and then finish cut the inside if the hole to obtain the best finish, and the safest practice.

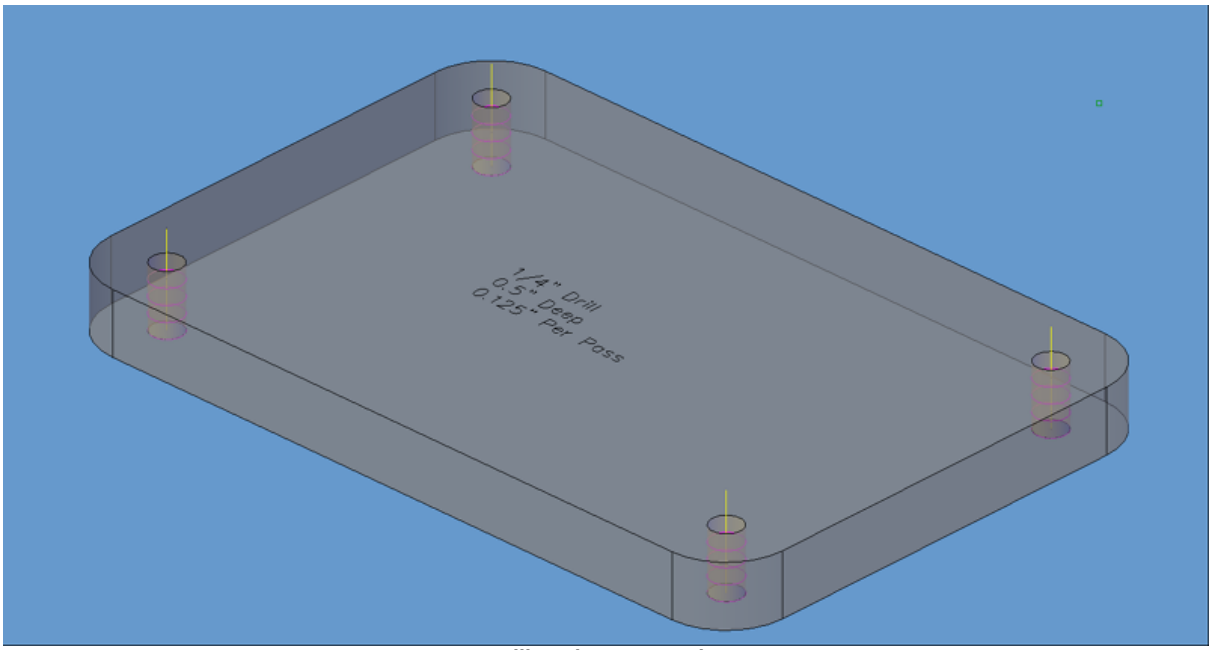
6.1.2.1 Drill Motions



The Drill Motions cycle is the standard drilling cycle in Router-CIM. With Drill Motions, you can drill in one pass, or in several passes with pecking motions.

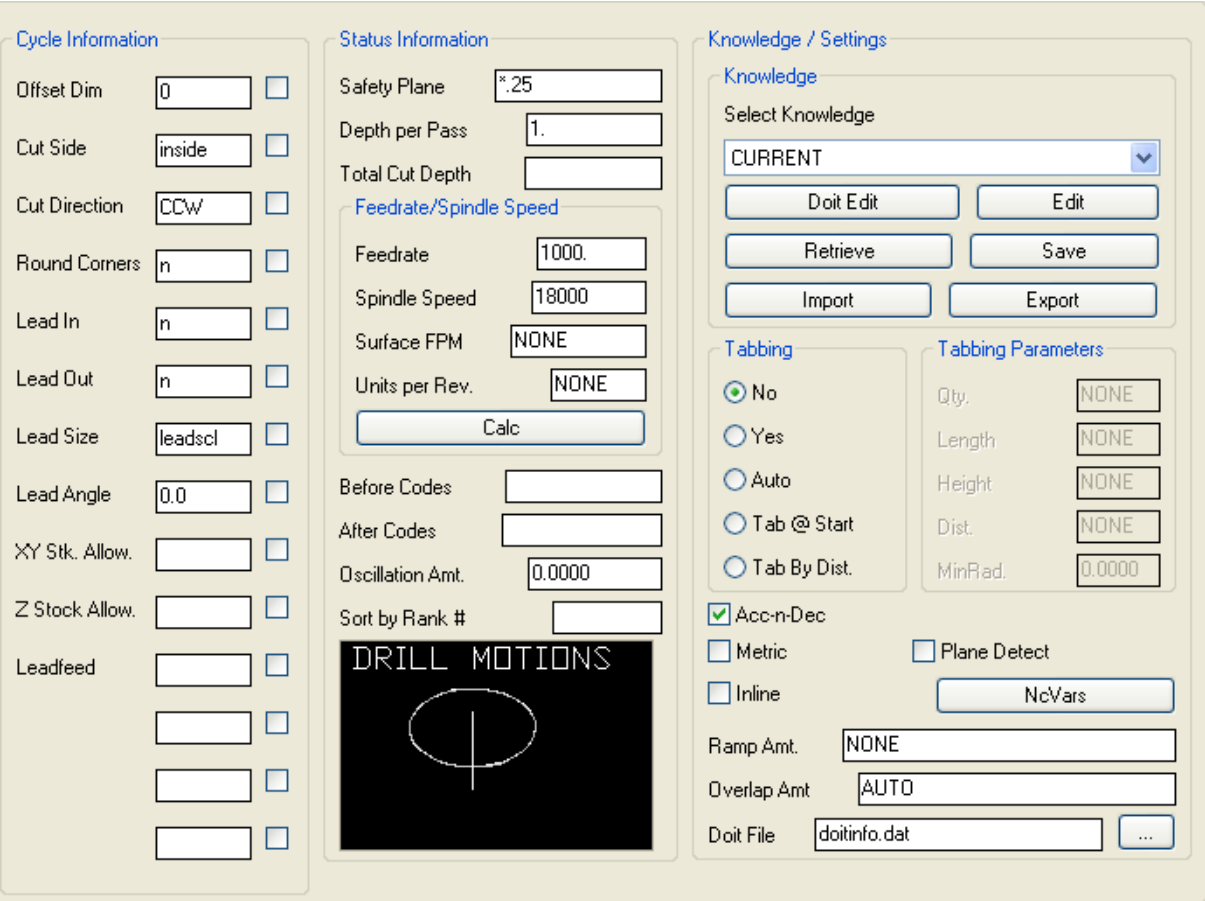
By default the tool will rapid to the safety plane above the hole, then feed in to the depth of the cut and then rapid out of the hole back up to the safety plane. If you wish to peck drill the hole, you can set multiple depths per pass (peck increments) and the tool will rapid to the hole, feed down to the first depth, the rapid back to the safety plane, then rapid back down to .1 above the next material to be cut, then feed down to the next depth of cut, rapid up to the safety plane, etc. until it finishes the hole.

The drill motions cycle has very few parameters that can be changed as it is the most basic drilling cycle in Router-CIM.



Drill Motions cut cycle.

© 2011 CIM-TECH.COM, Inc



Drill Motions parameters

The following parameters effect the toolpath creation:

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

© 2011 CIM-TECH.COM, Inc

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

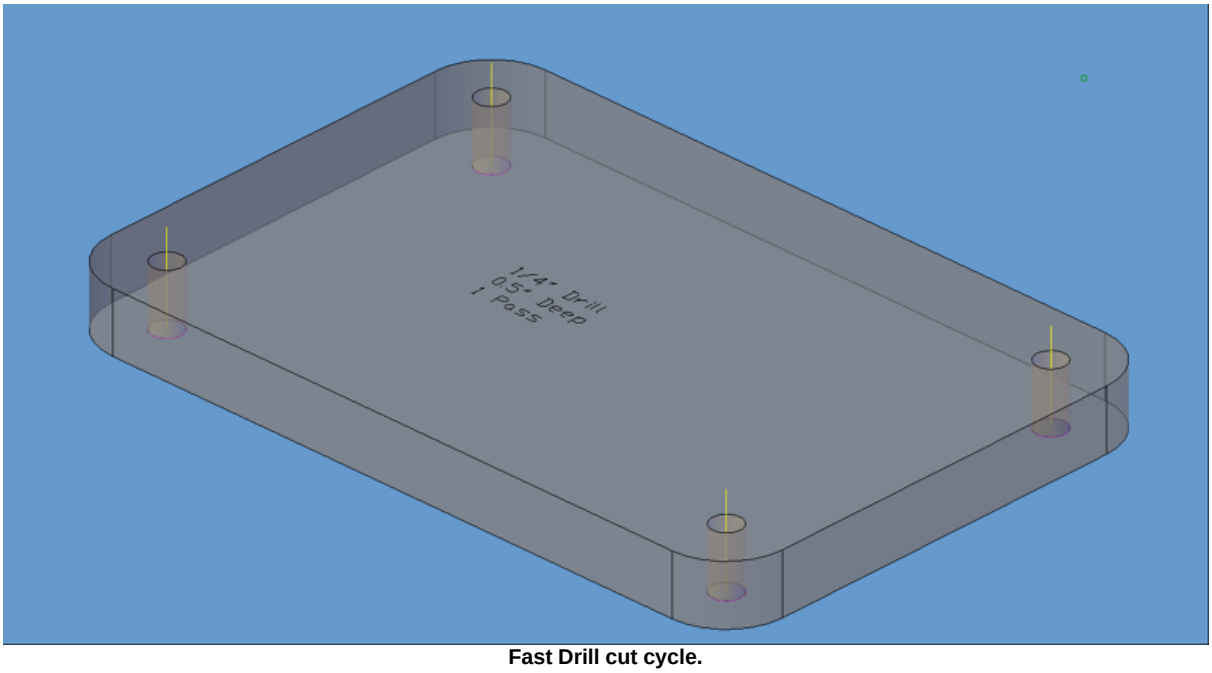
© 2011 CIM-TECH.COM, Inc

6.1.2.2 Fast Drill



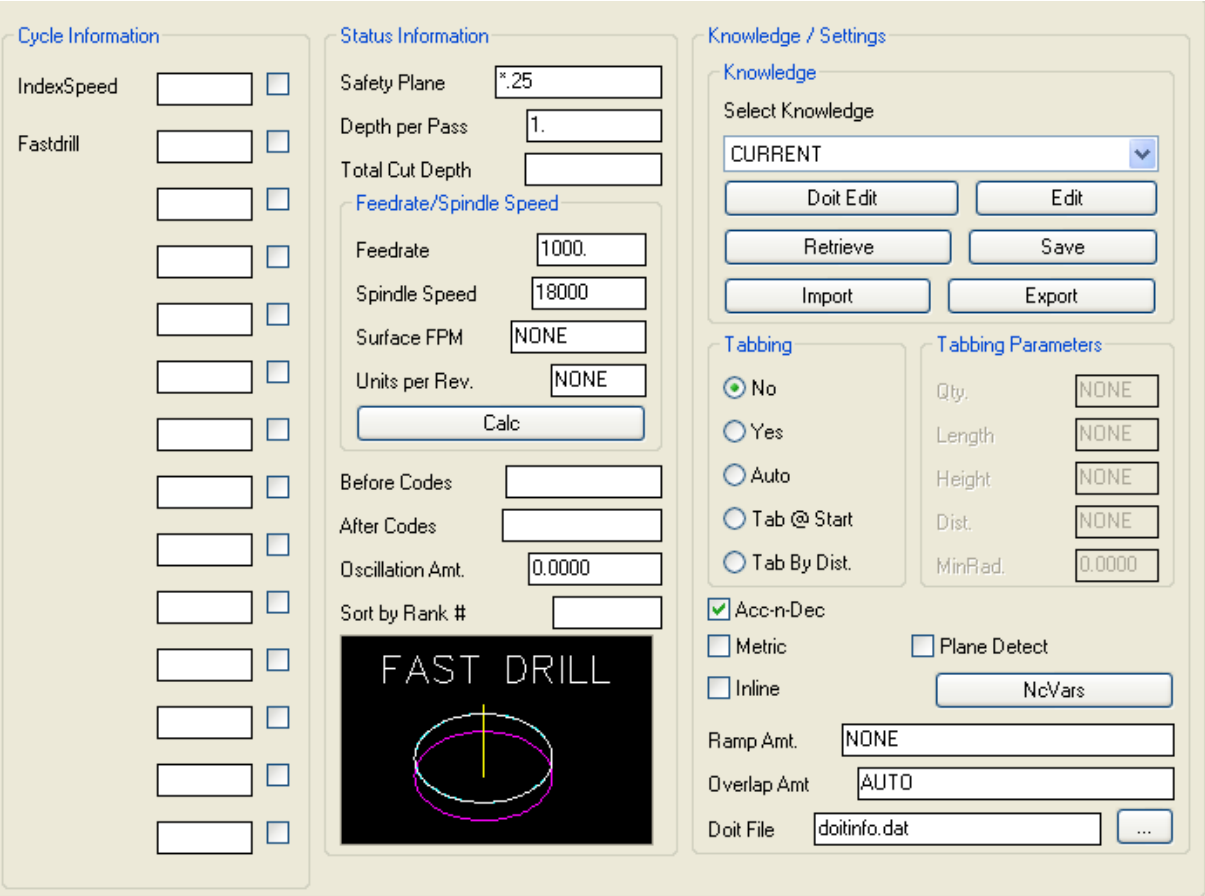
There is a Fast Drill cycle included with Router-CIM. Not all machines support this feature, but most Fanuc controlled machines will. The basic idea is to remove the dwell caused by switching from rapid to a programmed feedrate, and back to rapid again on each hole. This causes a small dwell on each change from G0 to G1. Instead the code will keep the machine in G1 mode and change the feedrate from the speed programmed for the drill to the fastest feedrate available for the machine on the moves where there would normally be a rapid traverse move. This causes the machine to move much faster from hole to hole, by avoiding the dwell.

In use the cycle appears the same as Drill Motions, but the resulting NC Code in the program is very different.



Fast Drill cut cycle.

© 2011 CIM-TECH.COM, Inc



Fast Drill cut parameters.

The following parameters effect the toolpath creation:

Index Speed

This is the fastest speed that you want the machine to achieve between the drilled holes. This feedrate will take place of the rapid traverse move between cuts and during the retract of the hole. If the machine can make a fast linear move between the cuts, usually this will reduce the overall cycle time of the drill moves.

Fastdrill

This engages the fast drill cycle. Entering Y will turn it on. Leaving it blank or setting to N will turn it off.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

© 2011 CIM-TECH.COM, Inc

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

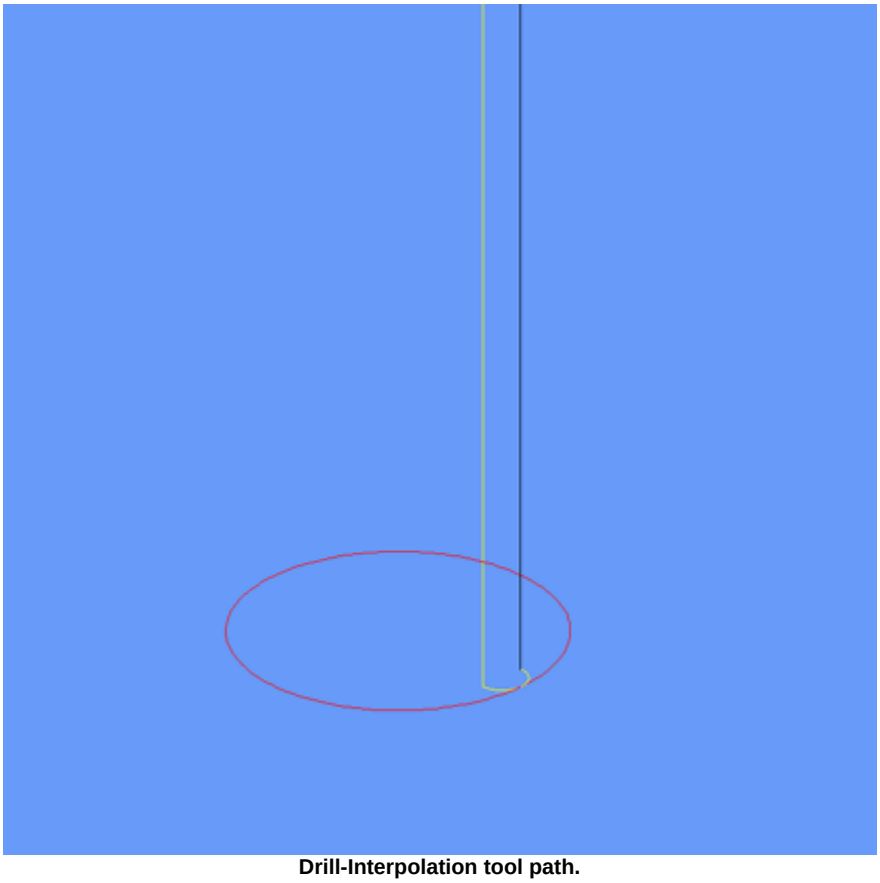
© 2011 CIM-TECH.COM, Inc

6.1.2.3 Drill-Interpolation



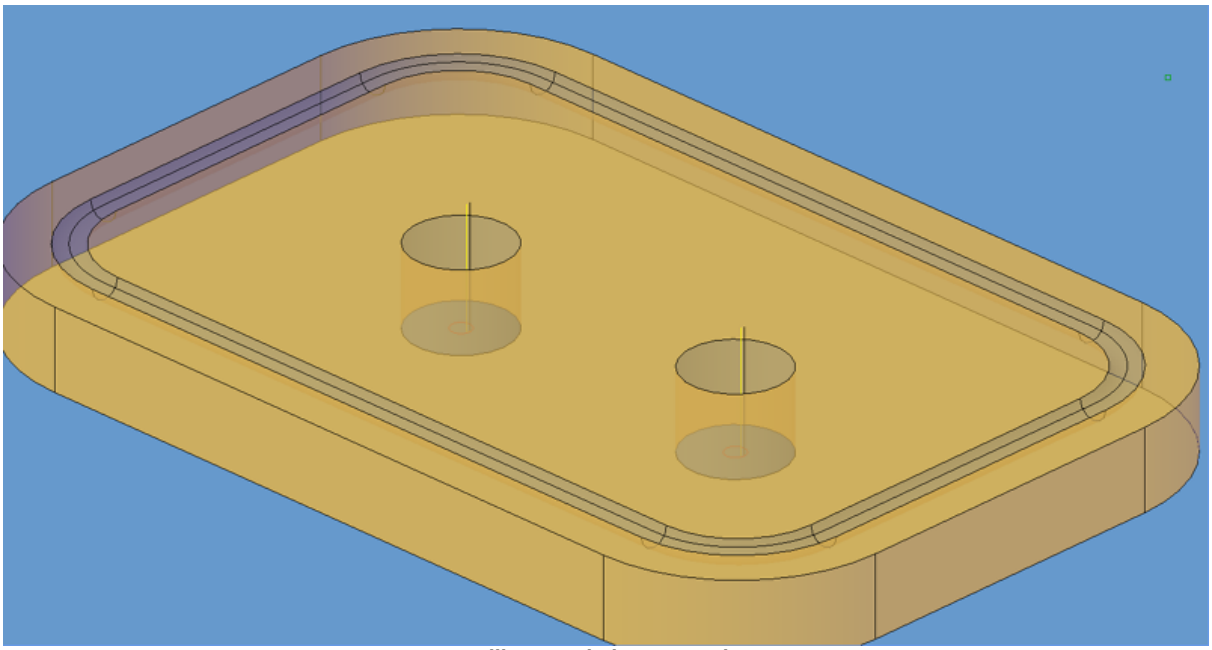
Drill Interpolation is a somewhat misleading term. This is actually a ROUTING cycle, but was created to allow for a way to cut a hole when there was no drill bit of the correct size available and there was a router-bit that was some amount smaller than the hole to be cut.

It is important to note that the router-bit should be smaller than the hole. It is also important to note that you should use a somewhat slower speed to interpolate the hole, or you may end up with an egg shaped hole. Sometimes it is helpful to set the safety plane a little higher if you are using Cutter Compensation to allow the machine time to make the cutter comp move before it starts moving the cutter down into the hole to interpolate.

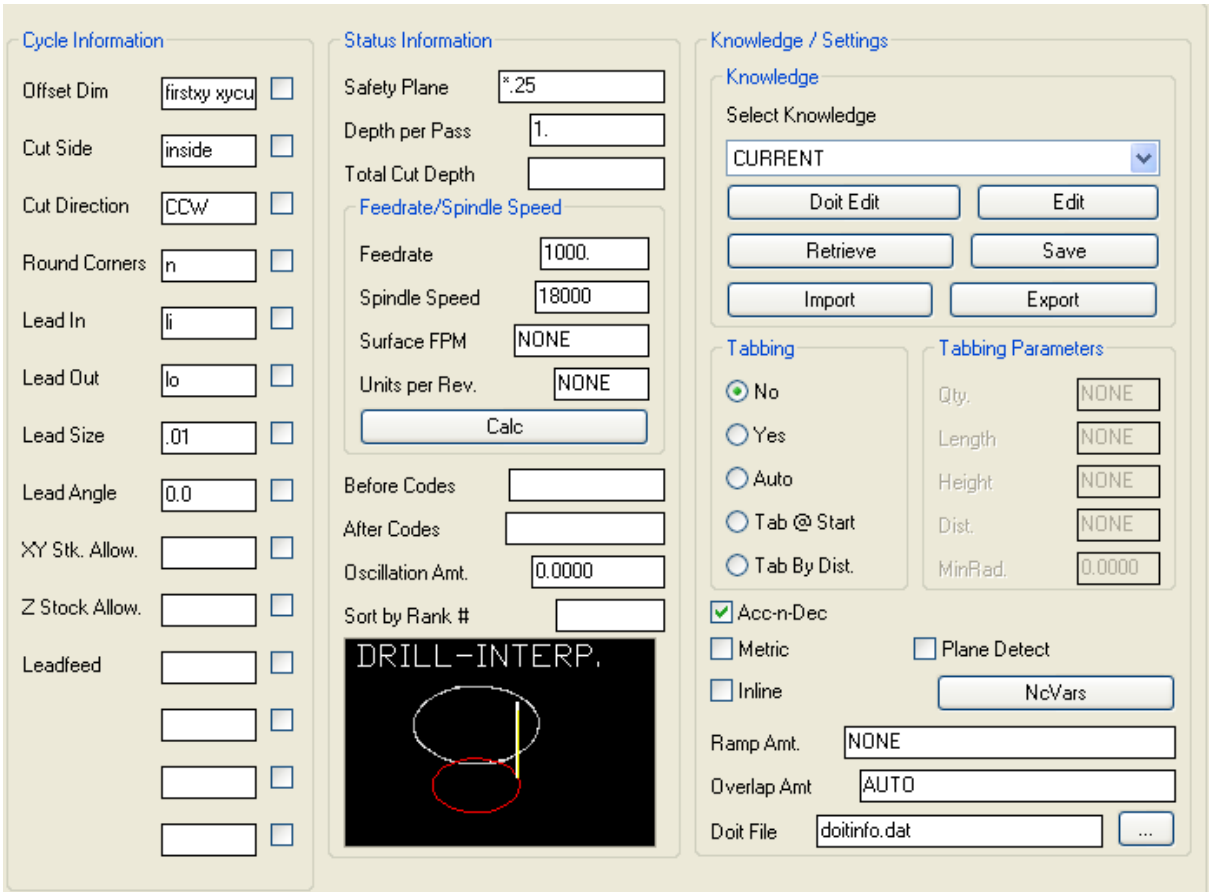


Drill-Interpolation tool path.

© 2011 CIM-TECH.COM, Inc



Drill-Interpolation cut cycle.



Drill-Interpolation cycle parameters.

The following parameters effect the toolpath creation:

© 2011 CIM-TECH.COM, Inc

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

© 2011 CIM-TECH.COM, Inc

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See the Z Stock Allowance section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

© 2011 CIM-TECH.COM, Inc

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.2.4 Spot-Bore

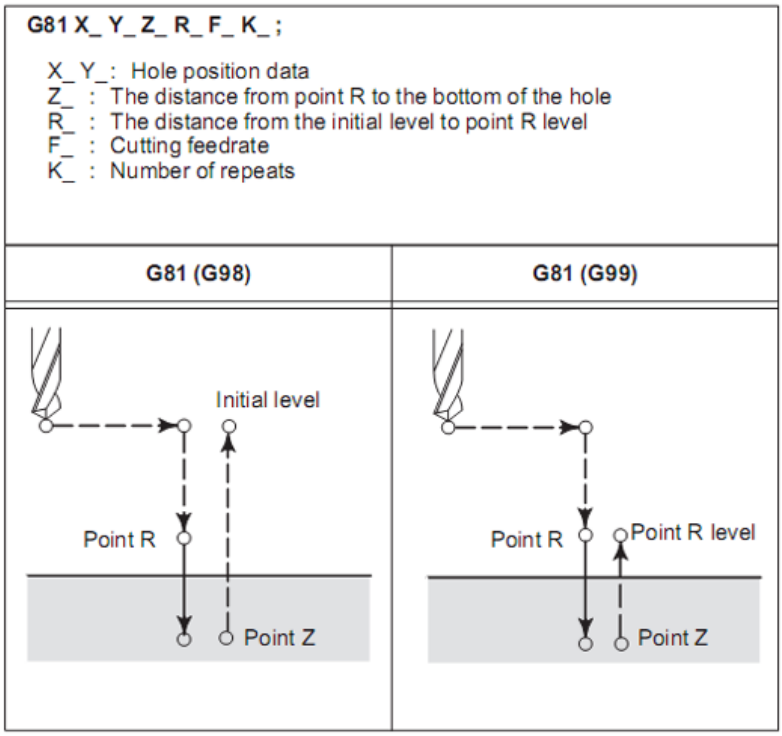


Spot Bore is a Canned Cycle. That really means that it is a text based cycle that has several parameters which are fed to the machine controller so that all the holes chosen are cut in exactly the same manner, like a macro. In fact on most machines, Canned Cycles are an option and you should check your machine to see if it is equipped with this cycle before trying to use it. Typically, Canned Cycles are used to lessen the amount of NC Code necessary in a program when drilling a large amount of holes. There is a code savings because the Z moves up and down in each hole are not necessary to call out in the code. Another reason to use Canned Cycles is if you are writing code by hand, as there is less code to generate. Finally, if there is a chance that you are going to edit the code later by hand, a Canned Cycle leaves you with only one line to edit to change all the drilling parameters.

Spot Bore is typically a normal one-pass drilling operation performed to the bottom of the hole.

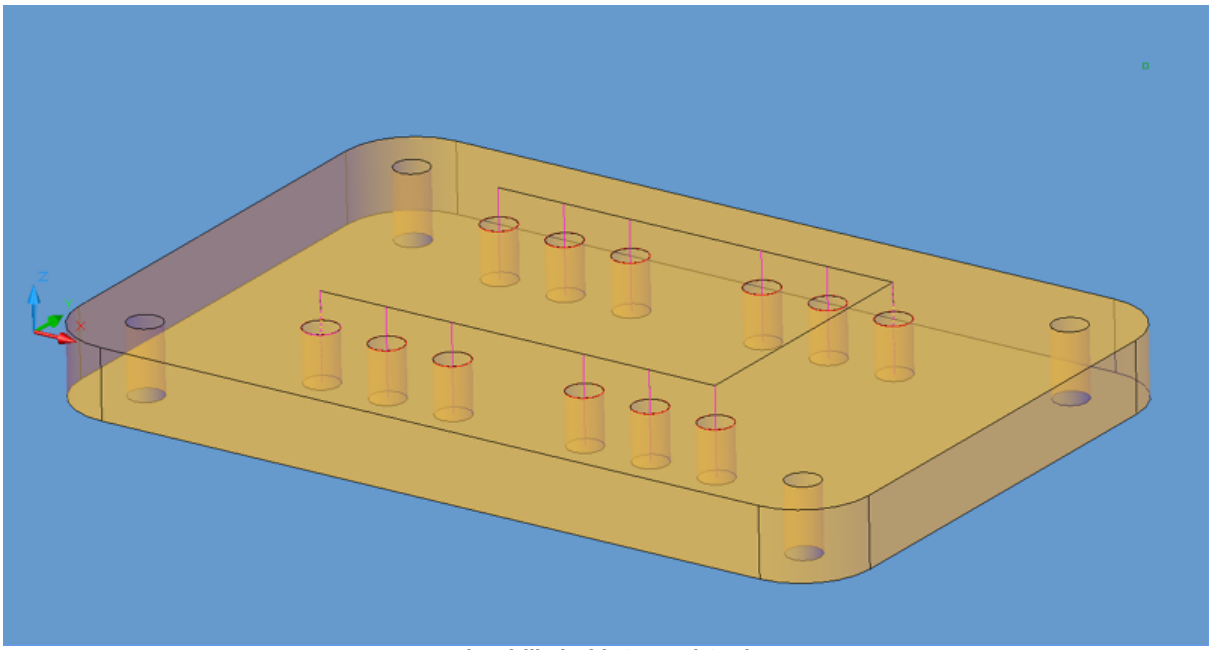
Canned Cycles use a series of commands on the first hole chosen and then perform those same commands on all other holes in the group. To use Canned Cycles, some explanation of them is necessary. This data is from the Fanuc control manual and will be specific to Fanuc controls or controls that can emulate the Fanuc code.

Format

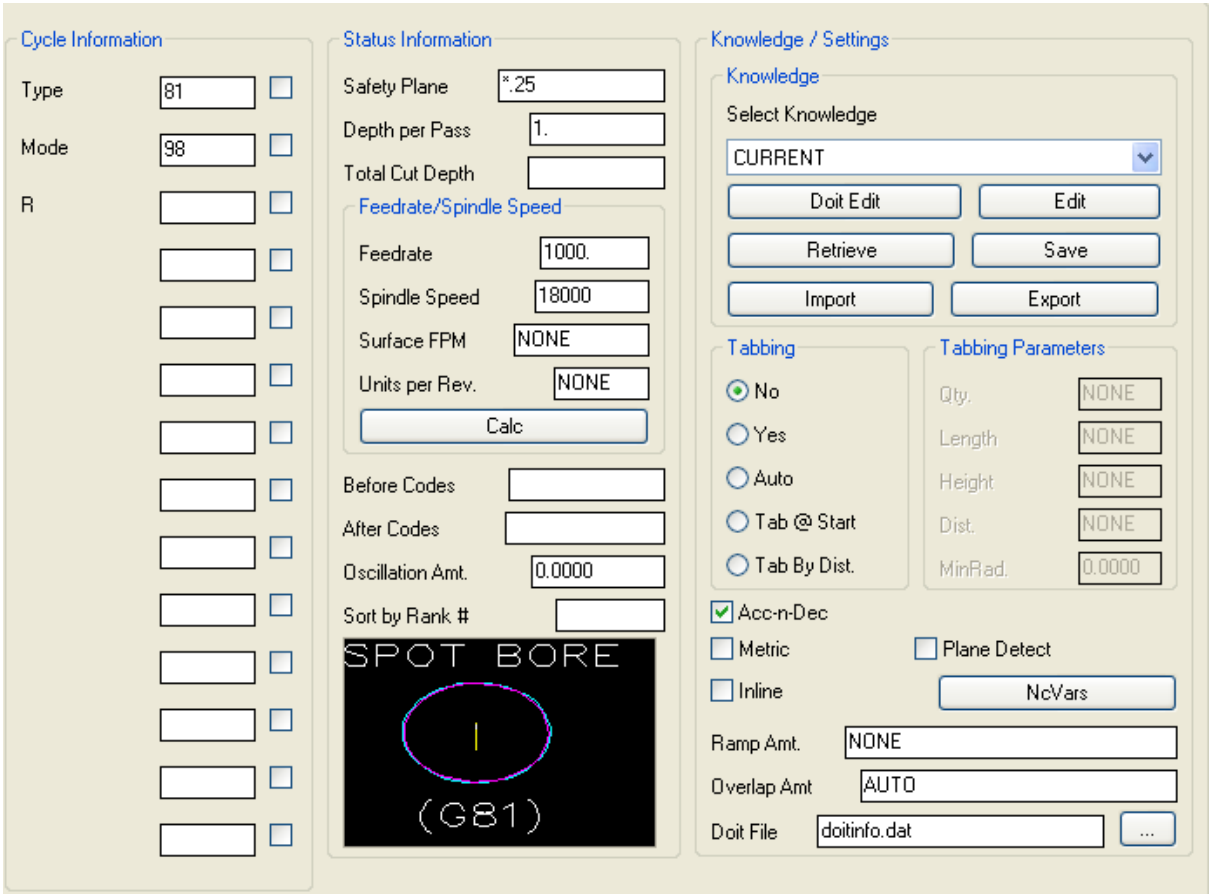


G81 Canned Cycle

© 2011 CIM-TECH.COM, Inc



Holes drilled with Canned Cycles



Canned Cycle parameters.

© 2011 CIM-TECH.COM, Inc

The following parameters effect the toolpath creation:

Type
The type field for Canned Cycles indicated the Cycle Type or method of the canned cycle. In this case G81 is the desired Canned Cycle, so 81 is the answer in Type.
You can substitute any valid Canned Cycle type in this field to create code for another cycle type.

Mode
Mode refers to the Retract mode of the canned cycle. In the pictures above, there is either G98 Mode, where the tool retracts to the Initial Point, or G99 Mode, where the tool retracts to the location specified by the R value (the next parameter).

R
This is the absolute point where you want the tool to retract to during pecks, between holes, and when the cycle is finished.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth
The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

© 2011 CIM-TECH.COM, Inc

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.2.5 Peck-Bore



Peck Bore is a Canned Cycle. That really means that it is a text based cycle that has several parameters which are fed to the machine controller so that all the holes choosen are cut in exactly the same manner, like a macro. In fact on most machines, Canned Cycles are an option and you should check your machine to see if it is equipped with this cycle before trying to use it. Typically, Canned Cycles are used to lessen the amount of NC Code necessary in a program when drilling a large amount of holes. There is a code savings because the Z moves up and down in each hole are not necessary. Another reason to use Canned Cycles is if you are writing code by hand, as there is less code to generate. Finally, if there is a chance that you are going to edit the code later by hand, a Canned Cycle leaves you with only one line to edit to change all the drilling parameters.

Peck Bore is typically a multiple depth per pass drilling operation.

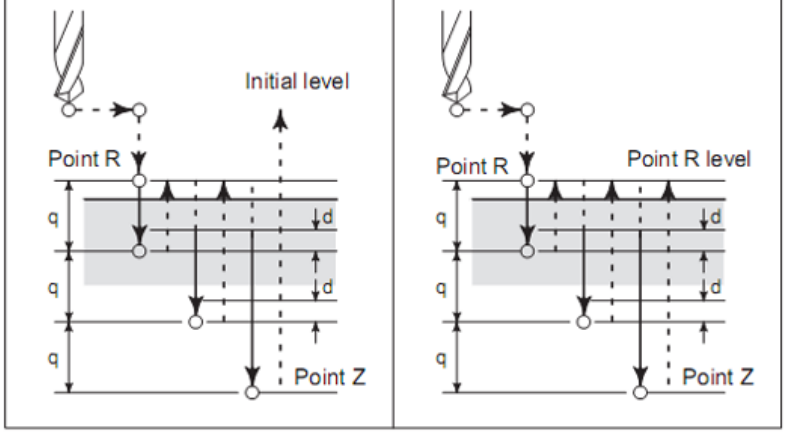
Canned Cycles use a series of commands on the first hole chosen and then perform those same commands on all other holes in the group. To use Canned Cycles, some explanation of them is necessary. This data is from the Fanuc control manual and will be specific to Fanuc controls or controls that can emulate the Fanuc code.

© 2011 CIM-TECH.COM, Inc

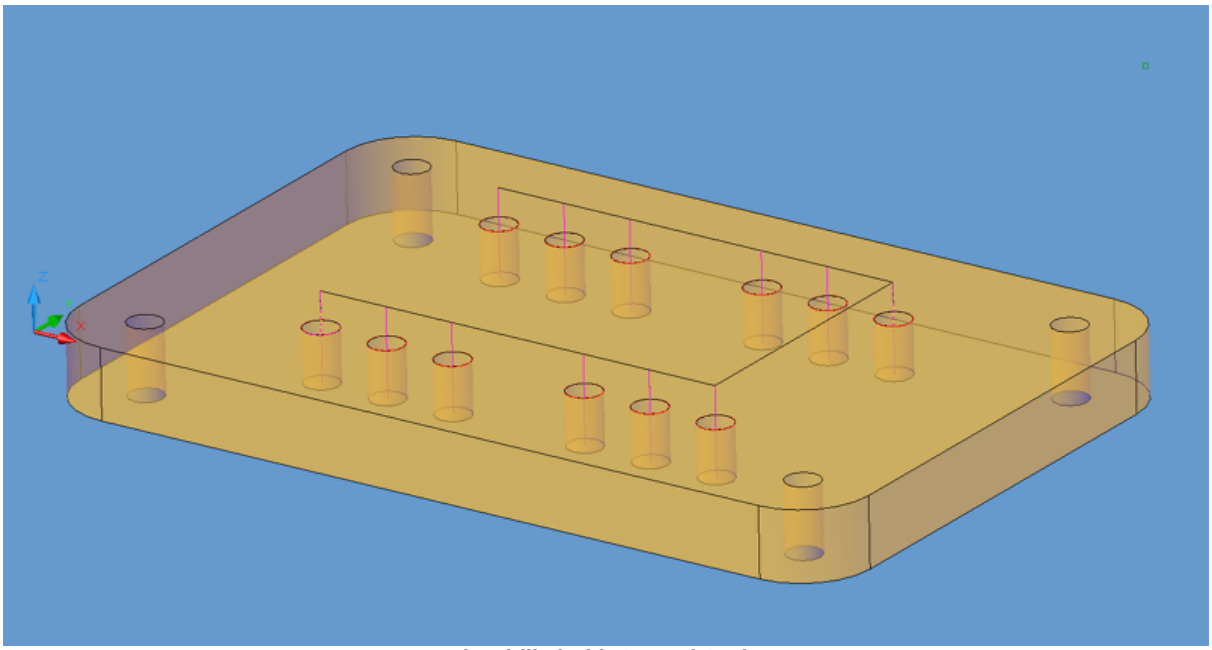
Format

G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_: Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
Q_ : Depth of cut for each cutting feed
F_ : Cutting feedrate
K_ : Number of repeats

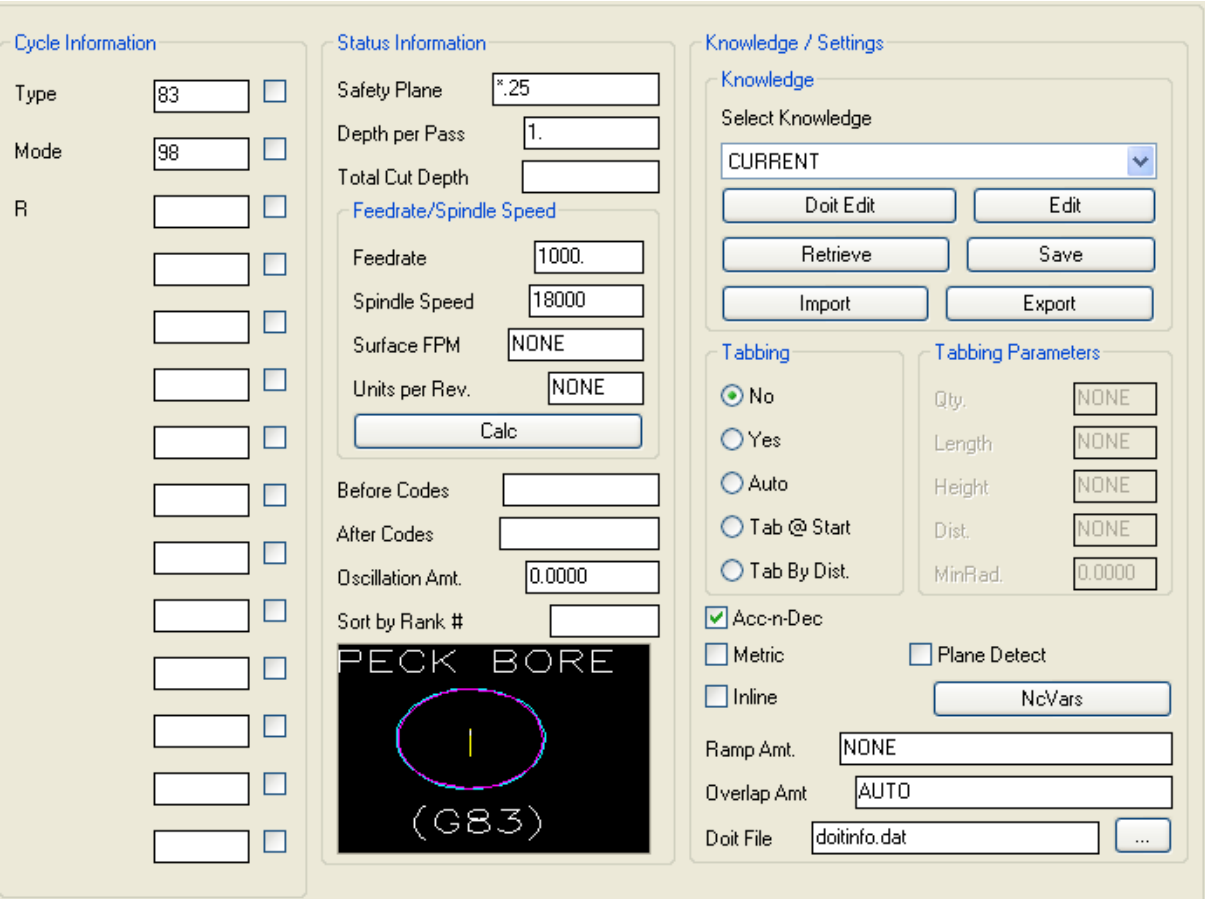


Peck Bore Cycle.



Holes drilled with Canned Cycles.

© 2011 CIM-TECH.COM, Inc



Peck Bore parameters.

The following parameters effect the toolpath creation:

Type
The type field for Canned Cycles indicated the Cycle Type or method of the canned cycle. In this case G81 is the desired Canned Cycle, so 81 is the answer in Type.
You can substitute any valid Canned Cycle type in this field to create code for another cycle type.

Mode
Mode refers to the Retract mode of the canned cycle. In the pictures above, there is either G98 Mode, where the tool retracts to the Initial Point, or G99 Mode, where the tool retracts to the location specified by the R value (the next parameter).

R
This is the absolute point where you want the tool to retract to during pecks, between holes, and when the cycle is finished.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.

© 2011 CIM-TECH.COM, Inc

This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

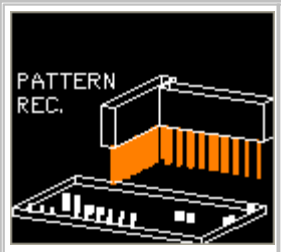
Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

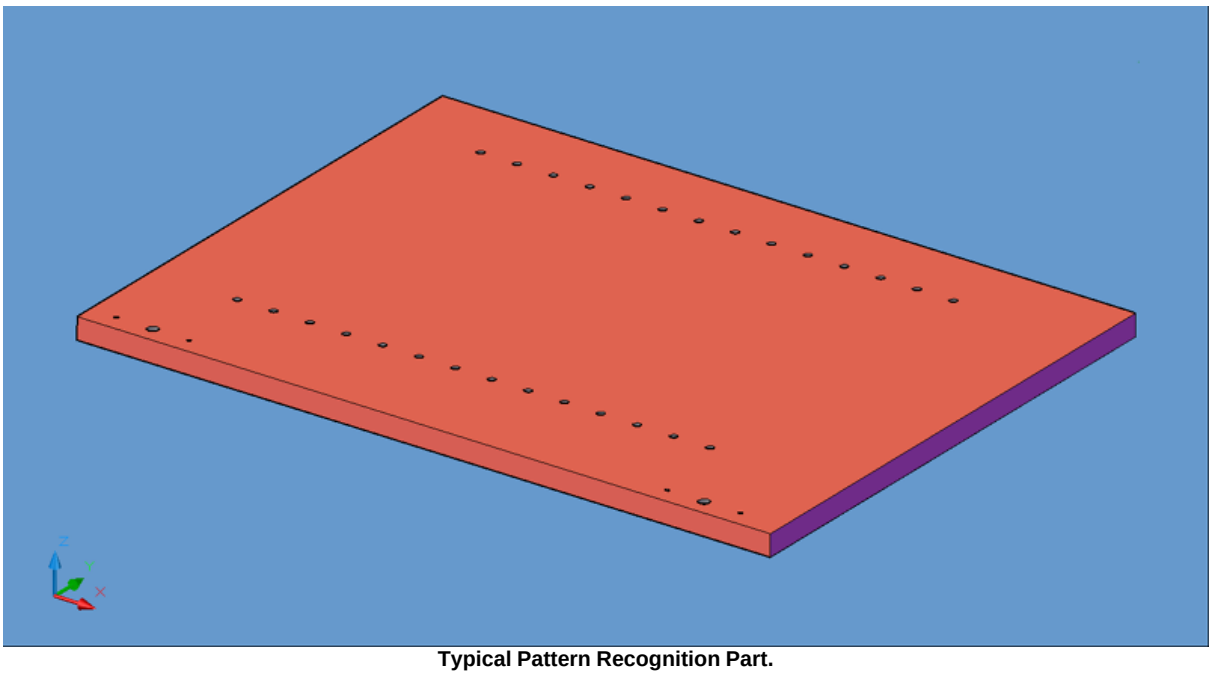
© 2011 CIM-TECH.COM, Inc

6.1.2.6 Pattern Recognition



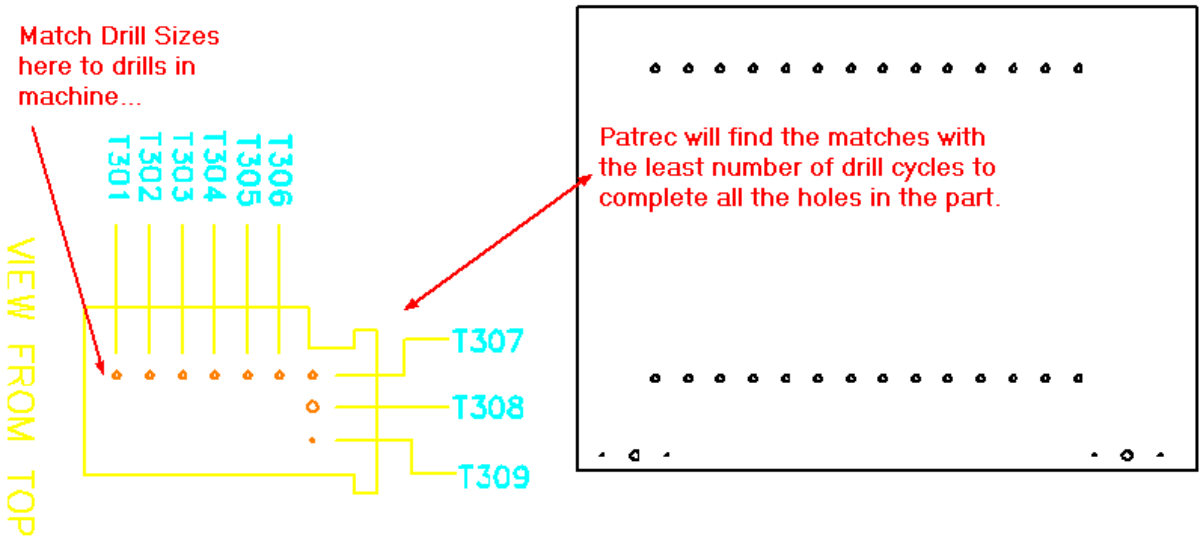
The Pattern Recognition cycle examines the drill holes on your part and the configuration of your drilling head(s) to determine the most efficient means of boring the selected holes. In order to use this function properly, the configuration of your drill heads must be set before you attempt to use this cycle. (See Note below.) Once configured, pick Pattern Recognition from the Cycle menu, choose Cut, and then select the holes to be bored. Pattern Recognition will load into memory, determine the most efficient route, and then proceed to bore the holes. The lead spindle in the group will have the tool path displayed on the screen; the others in the group will change colors to show they have been processed. The gang drill drawing will be provided to you by CIM-Tech following the specifications of your machine tool. Drill holes processed with Pattern Recognition need not be Geoshaped, and should be given thickness equal to their depth (negative Z value).

Note: The size of the holes on your gang drill drawing in your machine-specific default drawing MUST BE THE SAME SIZE as the holes in your current drawing for Pattern Recognition to work. You do not need to add any tool numbers in the Control Panel, as Pattern Recognition will find the tools it needs and add them automatically.

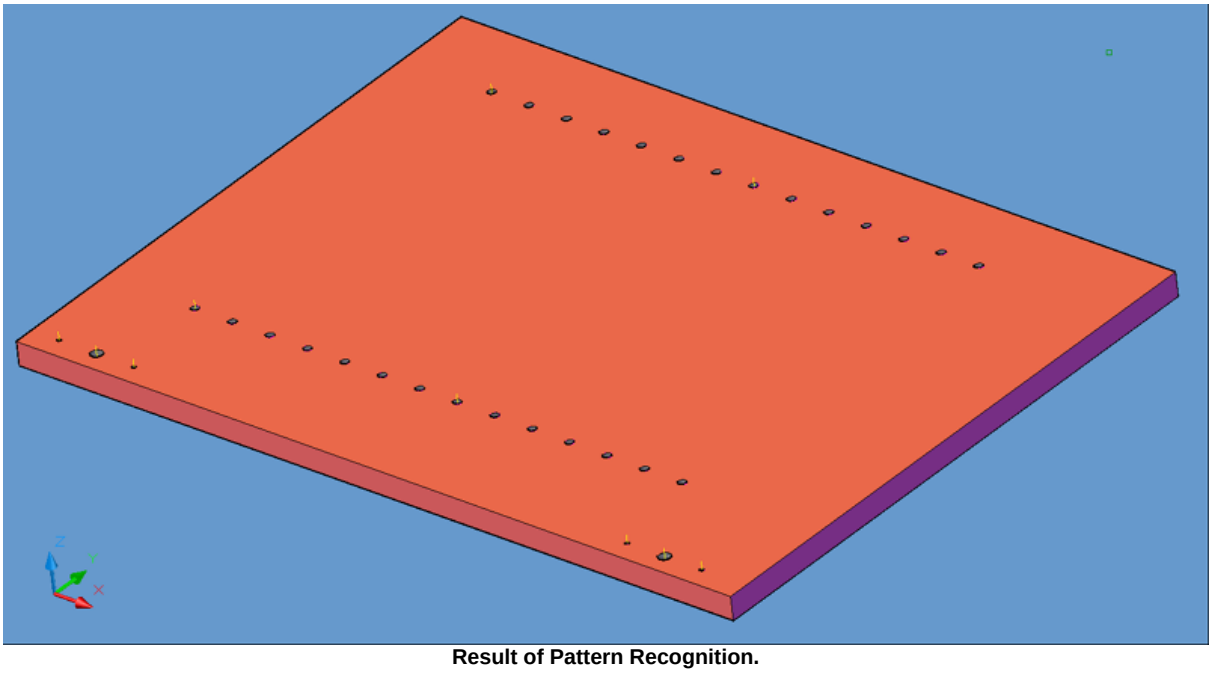


Typical Pattern Recognition Part.

© 2011 CIM-TECH.COM, Inc



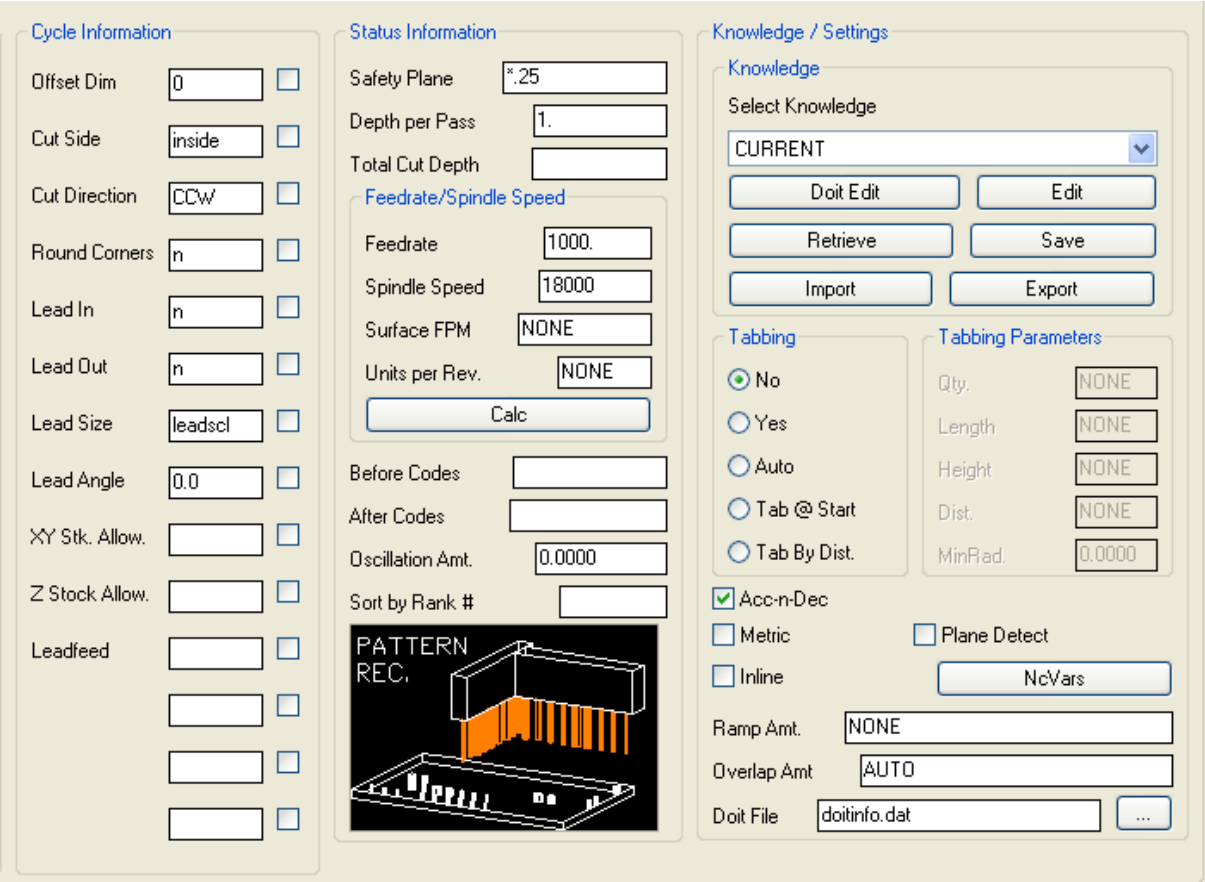
Match the drill block drawing to the drills in the machine.



Result of Pattern Recognition.

In the example above, the two rows of shelf holes are 32mm apart (1.2598") and there are two rows of 14 each. There are also two sets of hardware holes with sizes that do not match the shelf holes. With the drill block shown above, there are 7 drills in a row that match the shelf holes, so the drill block only drops twice on each row, drilling 7 holes each time. It then drills each of the hardware holes one at a time. In this instance, there are 10 drill cycles for a total of 34 holes.

© 2011 CIM-TECH.COM, Inc



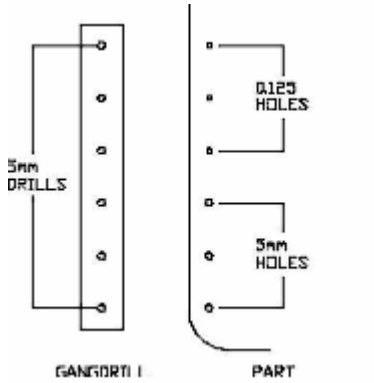
Pattern Recognition parameters.

Pattern Recognition - Tolerance Variable (*pat_fuzz*)

For Pattern Recognition to work properly, the diameter of the holes in your current drawing must be the same size as the holes on your gang drill drawing in your machine-specific default drawing (which is loaded into your current drawing upon loading Router-CIM). There is a tolerance built into the system, for the diameter of the holes, that is set to 0.05. This variable is *pat_fuzz* in the NCVARS. Under normal circumstances this need not be changed. But, if you have tools of a similar size (i.e. within .05 in radius or diameter) then Pattern Recognition may choose the wrong tools to complete its boring operation. Generally, if you are using all inch or all metric this will not be a problem.

Example: There are six drills on the gang drill with 5mm diameter drills in them. The part has both 5mm and 0.125 diameter holes in it. The difference between these two diameters is within the 0.05 tolerance, so Pattern Recognition will see them all as one size. Therefore, all six drills will drop and drill all six holes and all hole diameters will be 5mm, which is incorrect.

© 2011 CIM-TECH.COM, Inc



The following parameters effect the toolpath creation:

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

© 2011 CIM-TECH.COM, Inc

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

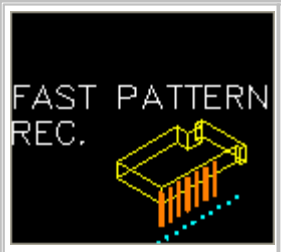
Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

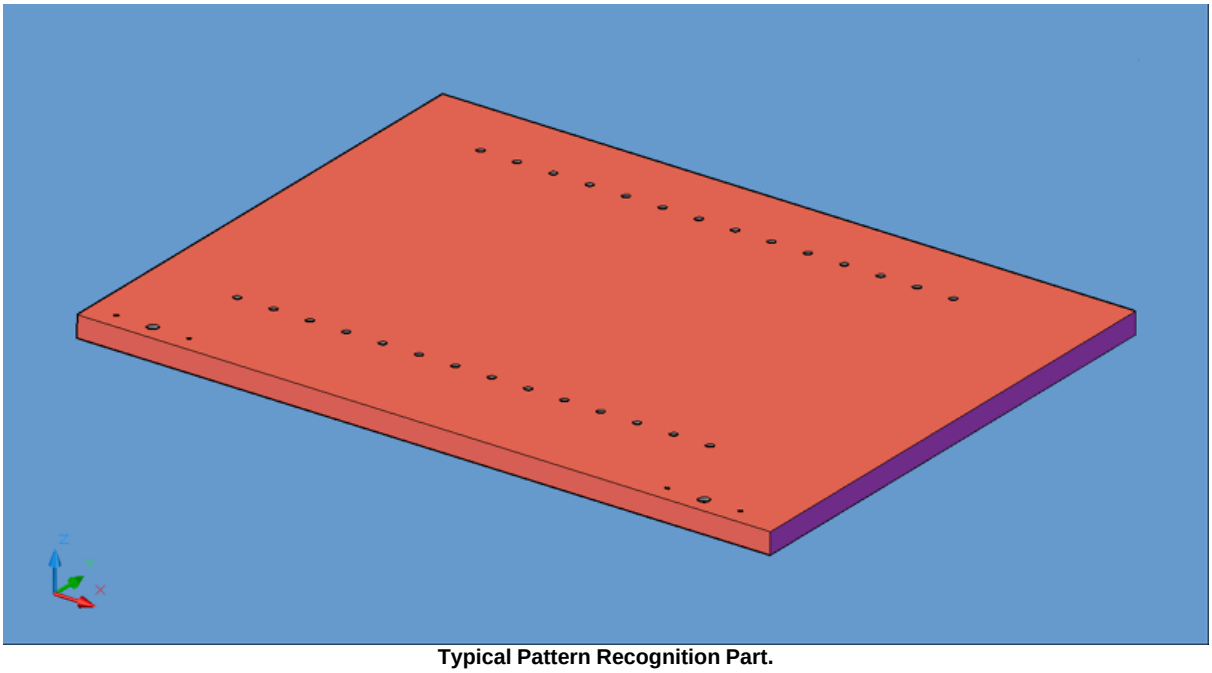
6.1.2.7 Fast Pattern Recognition



Fast Pattern Recognition is the same as regular Pattern Recognition, except that it substitutes fast feed moves for the rapid index moves between drills, which actually speeds up the overall drilling operations.

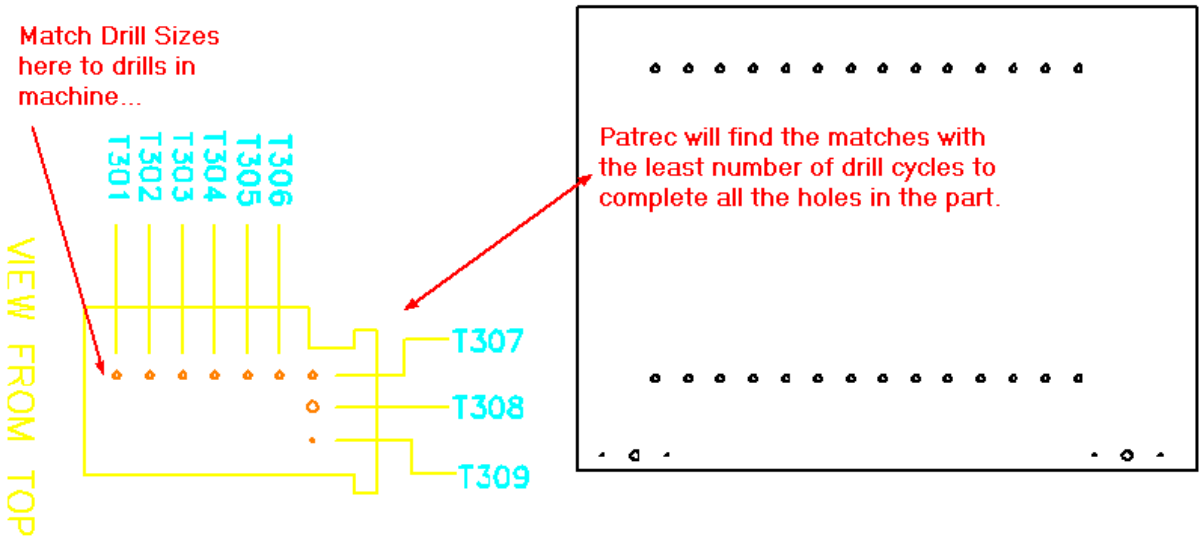
The Pattern Recognition cycle examines the drill holes on your part and the configuration of your drilling head(s) to determine the most efficient means of boring the selected holes. In order to use this function properly, the configuration of your drill heads must be set before you attempt to use this cycle. (See Note below.) Once configured, pick Pattern Recognition from the Cycle menu, choose Cut, and then select the holes to be bored. Pattern Recognition will load into memory, determine the most efficient route, and then proceed to bore the holes. The lead spindle in the group will have the tool path displayed on the screen; the others in the group will change colors to show they have been processed. The gang drill drawing will be provided to you by CIM-Tech following the specifications of your machine tool. Drill holes processed with Pattern Recognition need not be Geoshaped, and should be given thickness equal to their depth (negative Z value).

Note: The size of the holes on your gang drill drawing in your machine-specific default drawing MUST BE THE SAME SIZE as the holes in your current drawing for Pattern Recognition to work. You do not need to add any tool numbers in the Control Panel, as Pattern Recognition will find the tools it needs and add them automatically.

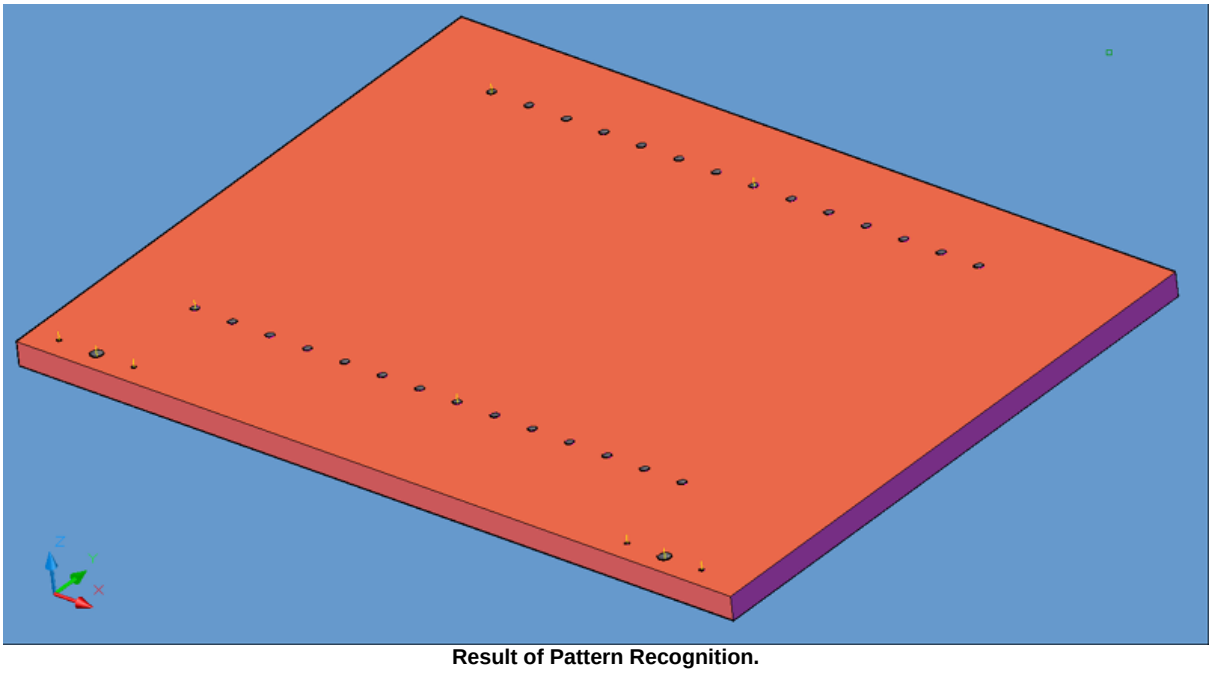


Typical Pattern Recognition Part.

© 2011 CIM-TECH.COM, Inc



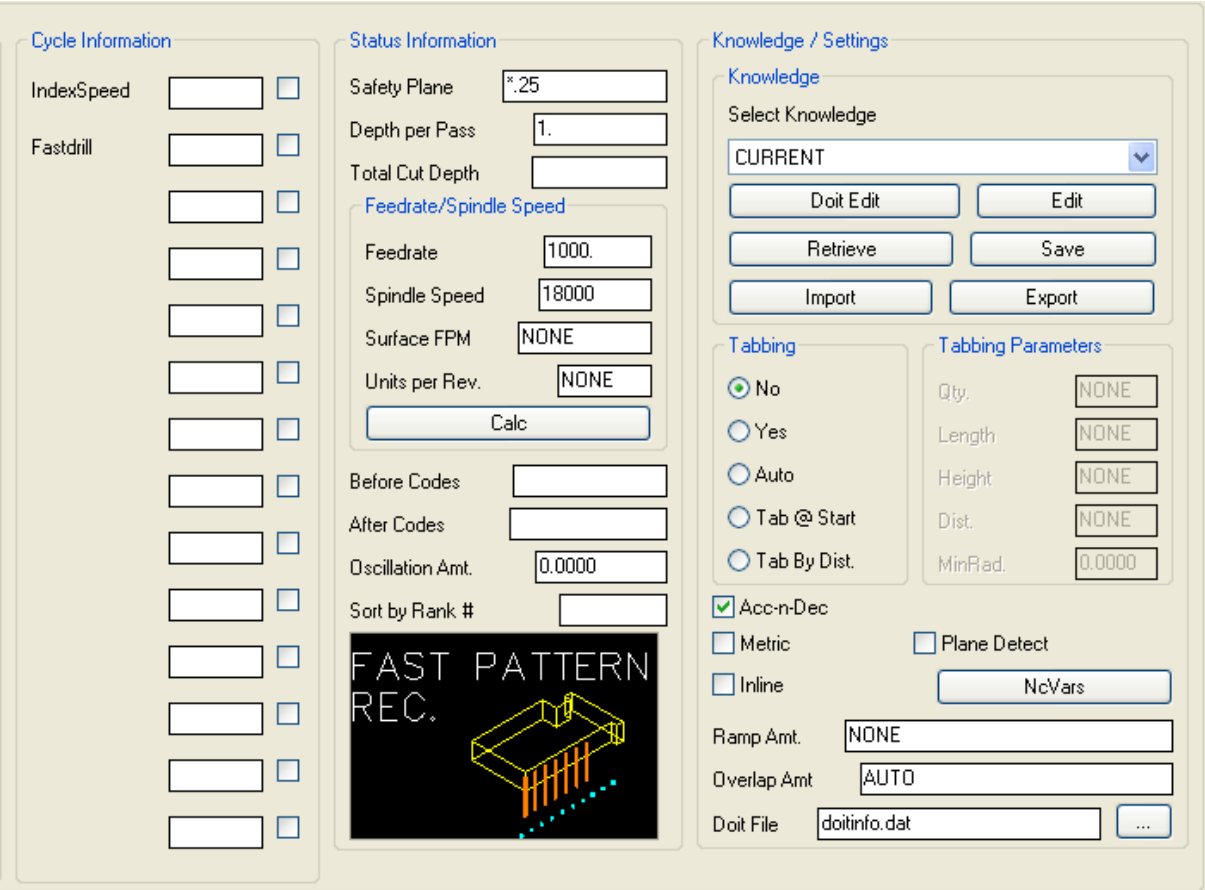
Match the drill block drawing to the drills in the machine.



Result of Pattern Recognition.

In the example above, the two rows of shelf holes are 32mm apart (1.2598") and there are two rows of 14 each. There are also two sets of hardware holes with sizes that do not match the shelf holes. With the drill block shown above, there are 7 drills in a row that match the shelf holes, so the drill block only drops twice on each row, drilling 7 holes each time. It then drills each of the hardware holes one at a time. In this instance, there are 10 drill cycles for a total of 34 holes.

© 2011 CIM-TECH.COM, Inc



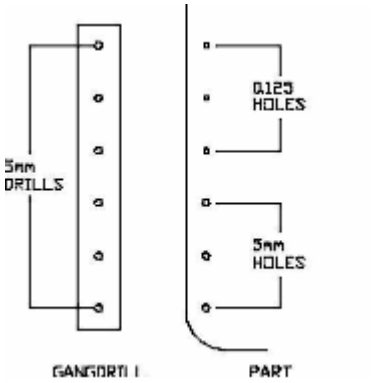
Fast Pattern Recognition parameters.

Pattern Recognition - Tolerance Variable (*pat_fuzz*)

For Pattern Recognition to work properly, the diameter of the holes in your current drawing must be the same size as the holes on your gang drill drawing in your machine-specific default drawing (which is loaded into your current drawing upon loading Router-CIM). There is a tolerance built into the system, for the diameter of the holes, that is set to 0.05. This variable is *pat_fuzz* in the NCVARS. Under normal circumstances this need not be changed. But, if you have tools of a similar size (i.e. within .05 in radius or diameter) then Pattern Recognition may choose the wrong tools to complete its boring operation. Generally, if you are using all inch or all metric this will not be a problem.

Example: There are six drills on the gang drill with 5mm diameter drills in them. The part has both 5mm and 0.125 diameter holes in it. The difference between these two diameters is within the 0.05 tolerance, so Pattern Recognition will see them all as one size. Therefore, all six drills will drop and drill all six holes and all hole diameters will be 5mm, which is incorrect.

© 2011 CIM-TECH.COM, Inc



The following parameters effect the toolpath creation:

Index Speed
Specify here the fastest feedrate your machine can achieve in X and Y. This will be the feedrate used to move between holes and index out of the holes.

Fastdrill
Valid answers are Y for yes, or N for no. If it is left blank, N is filled in.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth
The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

© 2011 CIM-TECH.COM, Inc

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

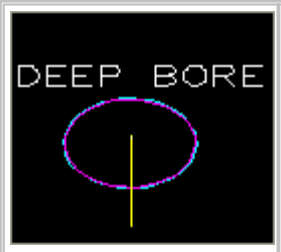
Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

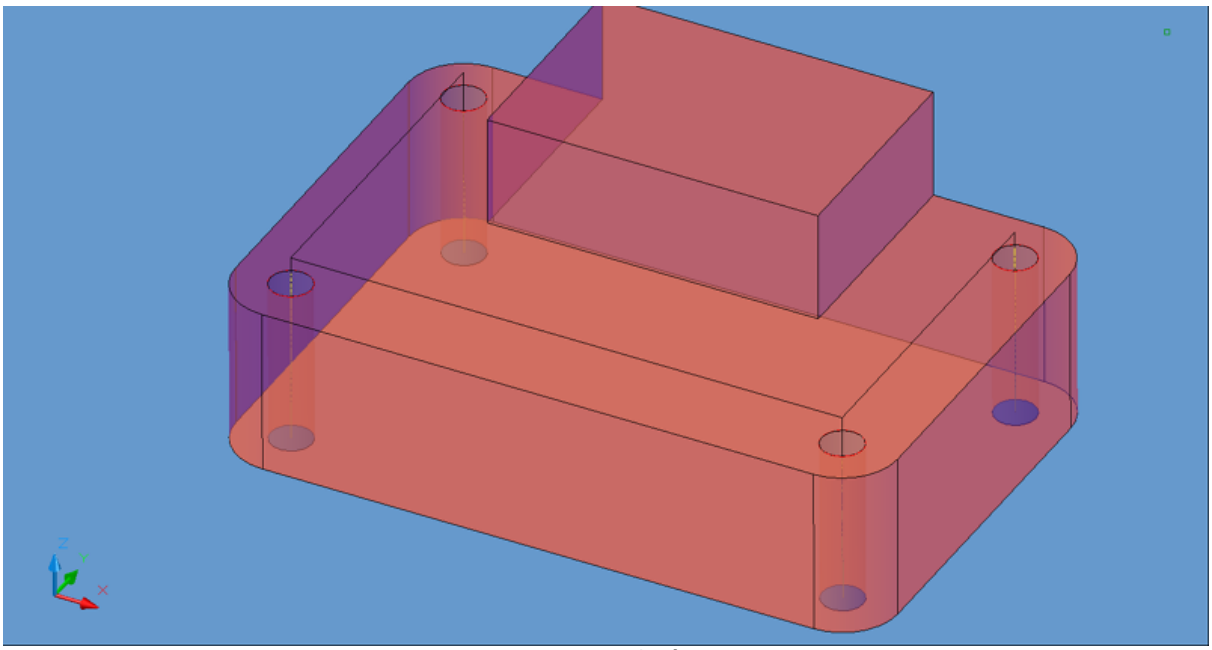
6.1.2.8 Deep-Bore



Deep Bore is a multiple pass drilling cycle that offers some flexibility with regards to peck increments, chip clearing, and changing feedrates during the cut cycle.
Typically this cycle will start at the Safety Plane, drill down to the first peck depth, retract enough to break the chip and then bore to the next peck depth. There is a parameter setting for the Chip Clear Increment and Chip Break Height where the tool can retract to the clearance plane between pecks to remove chips from the hole. There are also parameters available to change the feedrate at a certain point during the cut and allowances to change the peck increment at a specific depth, as well as new chip break and chip clear increments.

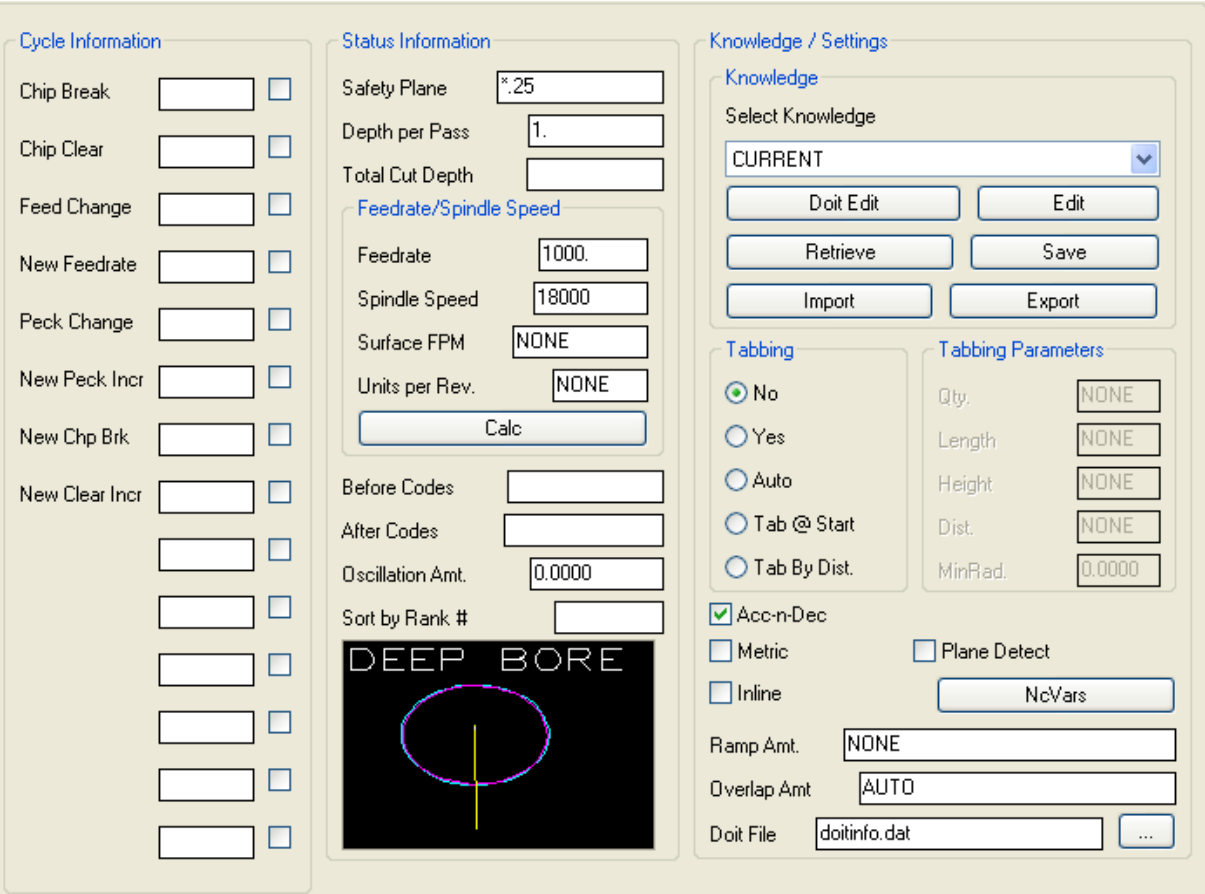
This cut cycle is meant for drilling in very thick material or when the tool needs to be retracted several times to keep the chips from clogging the tool.

Holes drilled with this cycle should be grouped together like canned cycles so that each cut task can run on each hole.



Deep Bore Cycle

© 2011 CIM-TECH.COM, Inc



Deep Bore parameters.

The following parameters effect the toolpath creation:

Chip Break Height

How far to back up on a chip break move. A chip break move is usually a short motion to fracture the chip, and does not clear the hole.

Chip Clear Increment

Usually a multiple of the peck increment. When this secondary increment is passed, the drill is retracted to the clear plane.

Chip Break and Chip Clear work together making short strokes until the drill is loaded, then clearing the drill and starting over.

Feed Change

Entered as a depth in Z (typically negative). Change to a new feedrate when this depth is reached or passed.

New Feedrate

New feedrate to change to when the Feed Change depth is reached or passed.

Peck Change

Depth to modify the Peck Increment, and chip handling values. Change to new pecking and chip handling values when this depth is reached or passed.

© 2011 CIM-TECH.COM, Inc

New Peck Increment
The value to use as the Peck Increment once the Peck Change depth is reached or passed.

New Chip Break
The value to use as the Chip Break Height once the Peck Change depth is reached or passed.

New Clear Increment
The new Chip Clear Increment to use once the Peck Change depth is reached or passed.

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a '*' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.2.9 Helical-Arcs-Center

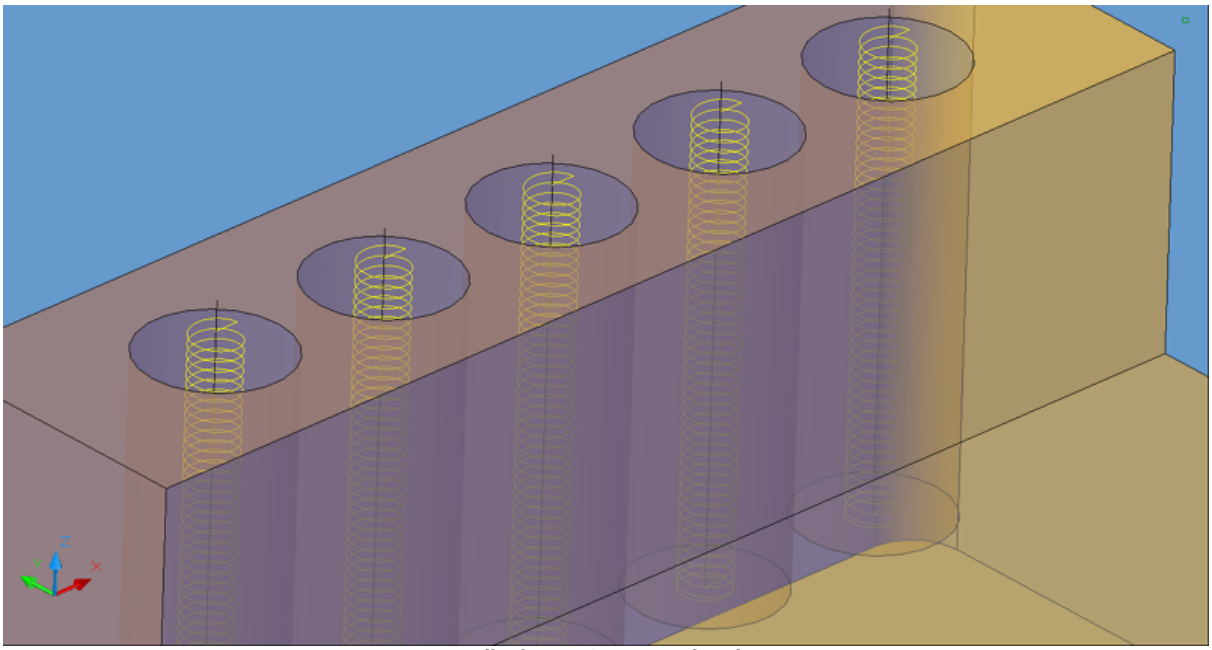


Helical Arcs can be used to make deep holes with a cutter as well as making threads with a thread mill. The benefit of making a regular hole cut with this cycle is that the tool chip load is constant and the tool does not switch back and forth between being loaded and unloaded, which minimizes tool deflection. There are parameters that allow you to have a bottom clean up pass so that there is no small wedge left at the bottom of the hole.

Thread milling can be accomplished with this cycle as well, by using a thread mill and entering in the parameters for the thread pitch. There are parameters that let you start at either the bottom or top of the hole.

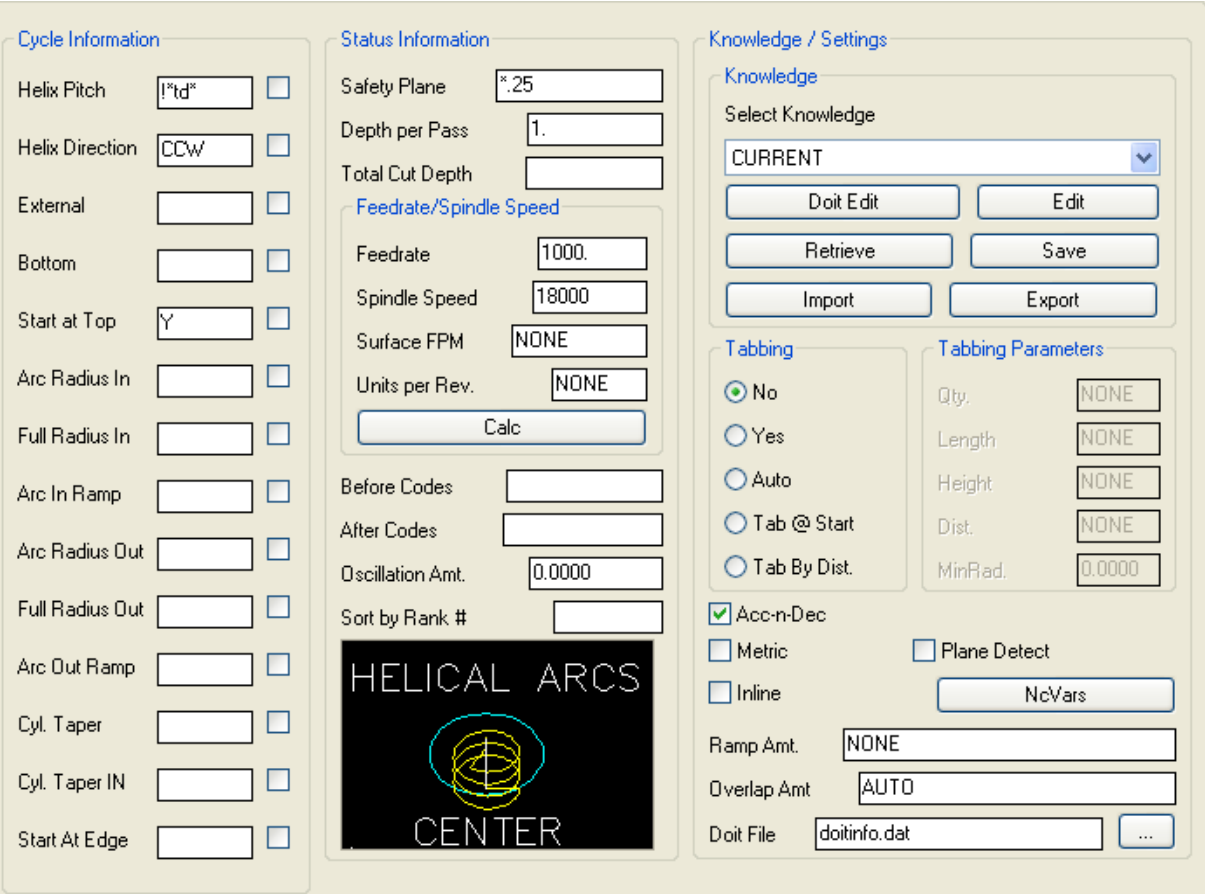
Helical Arcs Center will start the tool in the center of the hole and move towards the edge in the lead-in move.

This cycle will produce X, Y and Z moves on an arc at the machine. The objects used to create this tool path are Geoshaped circles.



Helical Arcs Center tool path.

© 2011 CIM-TECH.COM, Inc

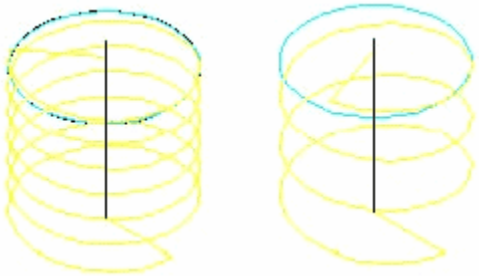


Helical Arcs Center parameters.

The following parameters effect the toolpath creation:

Helix Pitch

The desired pitch, in units per one revolution. If you picture the spirals of the helix like threads on a screw, the Helix Pitch is the distance from one thread to the next.



Helix Direction

The rotation direction of the pitch, clockwise or counterclockwise. Inputs are either CW or CCW. If you are threading, this is the setting to make right or left hand threads.

External

Do an external or internal cut. The possible inputs for this are either Y or N (blank). Y will give you a tool path on the outside of the geo-shaped geometry.

© 2011 CIM-TECH.COM, Inc

Bottom
This setting is for a Bottom Clean Up Pass in the cut. Possible inputs are Y or N. If Y is chosen, an extra cleanup pass is made at the bottom of
the hole to remove excess material. This pass is only made at the Total Depth of Cut.

Start at Top
Y is the default. If No is chosen, the tool will feed to the bottom of a pre-cut hole
and start its helical motion upward.

Arc Radius In
N will leave the lead-in arc radius at its default value. For a different lead-in
radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50, .60).

Full Radius In
Choose Y or N.
If Y is chosen, the cycle will have a 180° arc lead-in move based on the full radius of the tool.

Arc In Ramp
Choose Y or N.
If Y is selected, the cycle will have a helical lead-in.

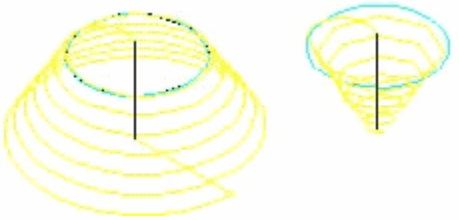
Arc Radius Out
N will leave the lead-out arc radius at its default value. For a different lead-out
radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50 .60).

Full Radius Out
Choose Y or N.
If Y is chosen, the cycle will have a 180° arc lead-out move based on the full radius of the tool.

Arc Out Ramp
Choose Y or N.
If Y is selected, the cycle will have a helical lead-out.

Cyl. Taper
Degrees of angle of cylinder taper. Possible inputs are positive whole number angles in degrees in 1 degree increments (i.e. 10, 20, 45 etc.).

Cyl. Taper IN
Choose Y or N. N will taper the cylinder outwards or bigger towards the bottom of the hole; Y will taper the cyclinder in towards the bottom of the hole.



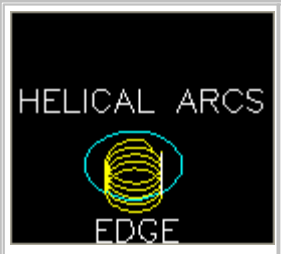
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if**

© 2011 CIM-TECH.COM, Inc

you change these default settings.

© 2011 CIM-TECH.COM, Inc

6.1.2.10 Helical-Arcs-Edge

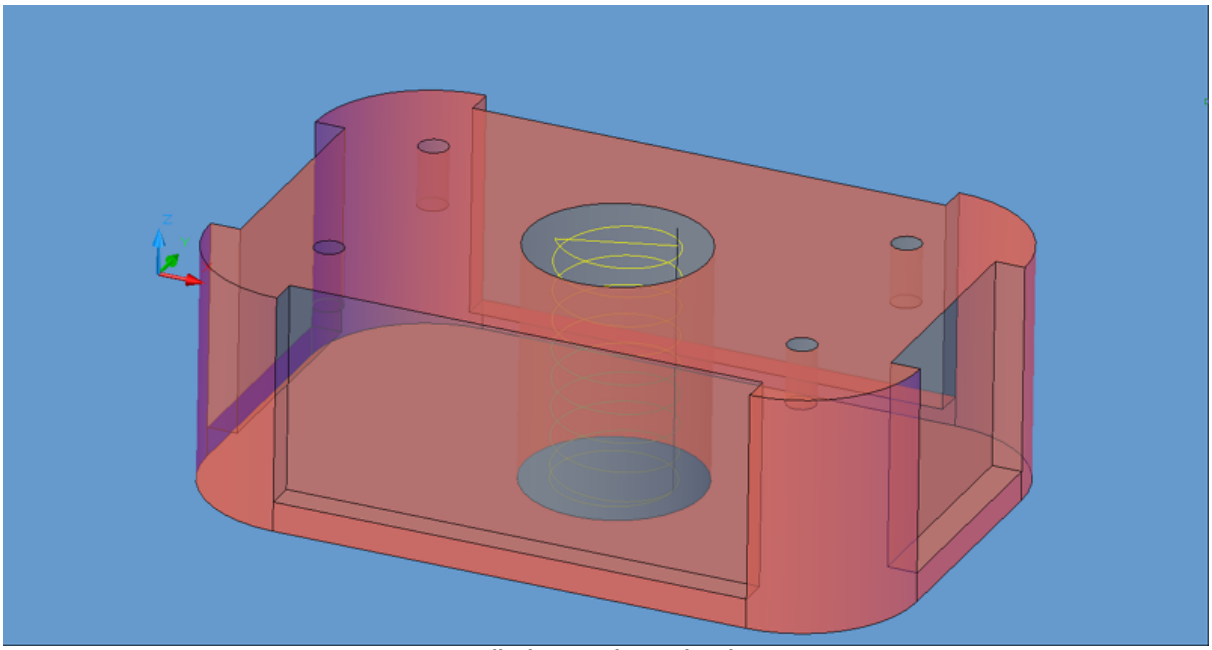


Helical Arcs can be used to make deep holes with a cutter as well as making threads with a thread mill. The benefit of making a regular hole cut with this cycle is that the tool chip load is constant and the tool does not switch back and forth between being loaded and unloaded, which minimizes tool deflection. There are parameters that allow you to have a bottom clean up pass so that there is no small wedge left at the bottom of the hole.

Thread milling can be accomplished with this cycle as well, by using a thread mill and entering in the parameters for the thread pitch. There are parameters that let you start at either the bottom or top of the hole.

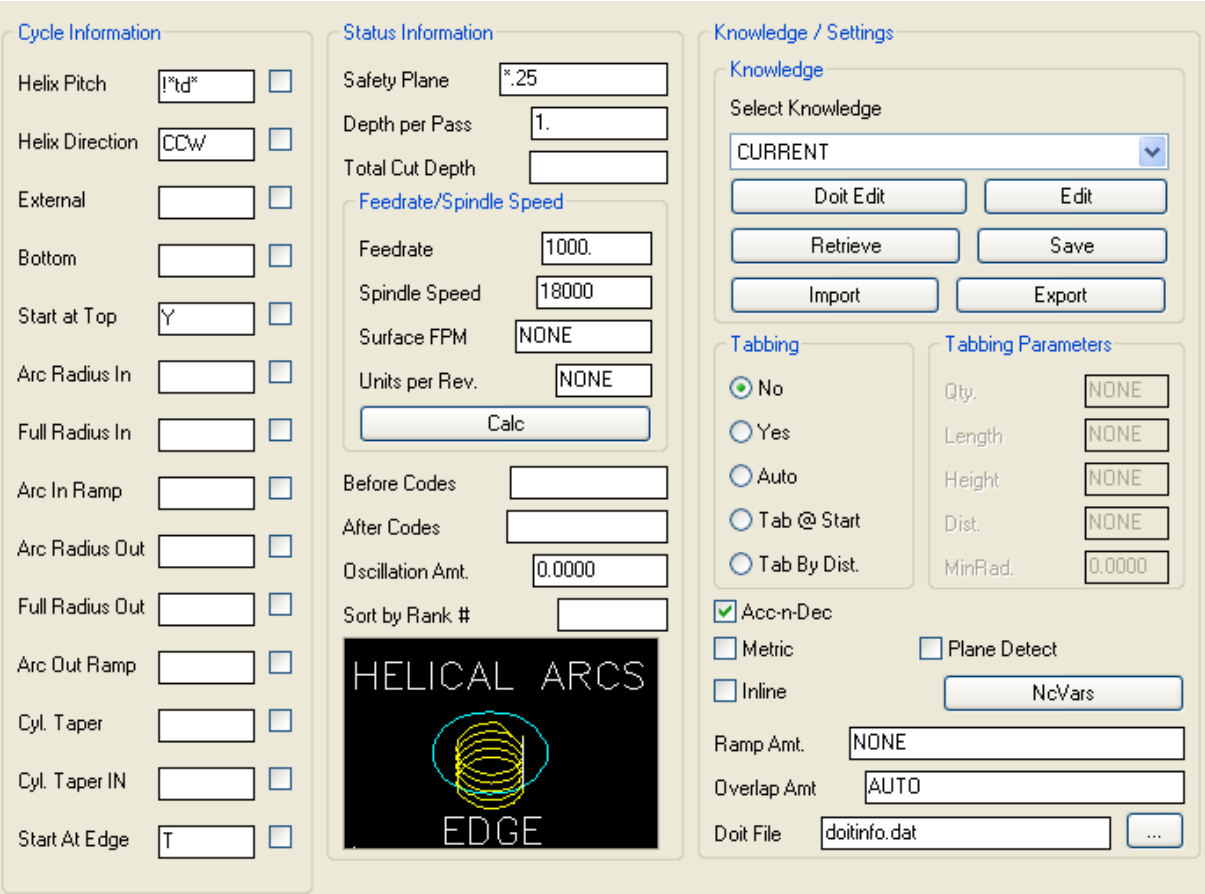
Helical Arcs Edge will start the tool at the edge of the hole where the start point is chosen.

This cycle will produce X, Y and Z moves on an arc at the machine. The objects used to create this tool path are Geoshaped circles.



Helical Arcs Edge tool path.

© 2011 CIM-TECH.COM, Inc

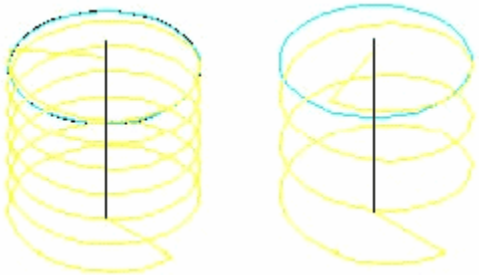


Helical Arcs Edge parameters.

The following parameters effect the toolpath creation:

Helix Pitch

The desired pitch, in units per one revolution. If you picture the spirals of the helix like threads on a screw, the Helix Pitch is the distance from one thread to the next.



Helix Direction

The rotation direction of the pitch, clockwise or counterclockwise. Inputs are either CW or CCW. If you are threading, this is the setting to make right or left hand threads.

External

Do an external or internal cut. The possible inputs for this are either Y or N (blank). Y will give you a tool path on the outside of the geo-shaped geometry.

© 2011 CIM-TECH.COM, Inc

Bottom
This setting is for a Bottom Clean Up Pass in the cut. Possible inputs are Y or N. If Y is chosen, an extra cleanup pass is made at the bottom of
the hole to remove excess material. This pass is only made at the Total Depth of Cut.

Start at Top
Y is the default. If No is chosen, the tool will feed to the bottom of a pre-cut hole
and start its helical motion upward.

Arc Radius In
N will leave the lead-in arc radius at its default value. For a different lead-in
radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50, .60).

Full Radius In
Choose Y or N.
If Y is chosen, the cycle will have a 180° arc lead-in move based on the full radius of the tool.

Arc In Ramp
Choose Y or N.
If Y is selected, the cycle will have a helical lead-in.

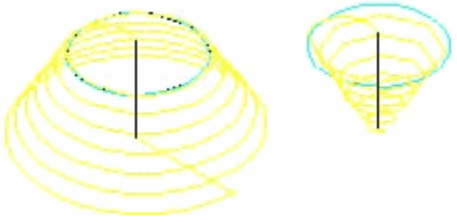
Arc Radius Out
N will leave the lead-out arc radius at its default value. For a different lead-out
radius, acceptable inputs for this field would be radius values in the form of a decimal (i.e. .25, .50 .60).

Full Radius Out
Choose Y or N.
If Y is chosen, the cycle will have a 180° arc lead-out move based on the full radius of the tool.

Arc Out Ramp
Choose Y or N.
If Y is selected, the cycle will have a helical lead-out.

Cyl. Taper
Degrees of angle of cylinder taper. Possible inputs are positive whole number angles in degrees in 1 degree increments (i.e. 10, 20, 45 etc.).

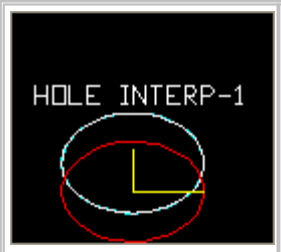
Cyl. Taper IN
Choose Y or N. N will taper the cylinder outwards or bigger towards the bottom of the hole; Y will taper the cyclinder in towards the bottom of the hole.



****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

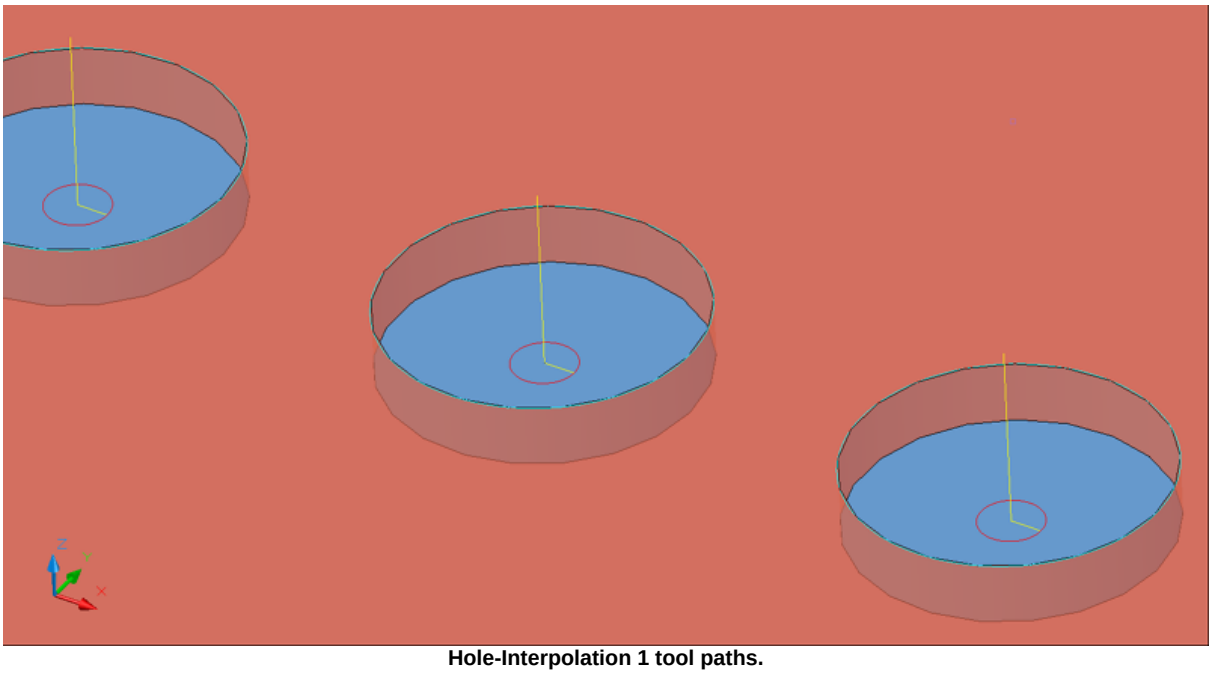
© 2011 CIM-TECH.COM, Inc

6.1.2.11 Hole-Interpolation 1



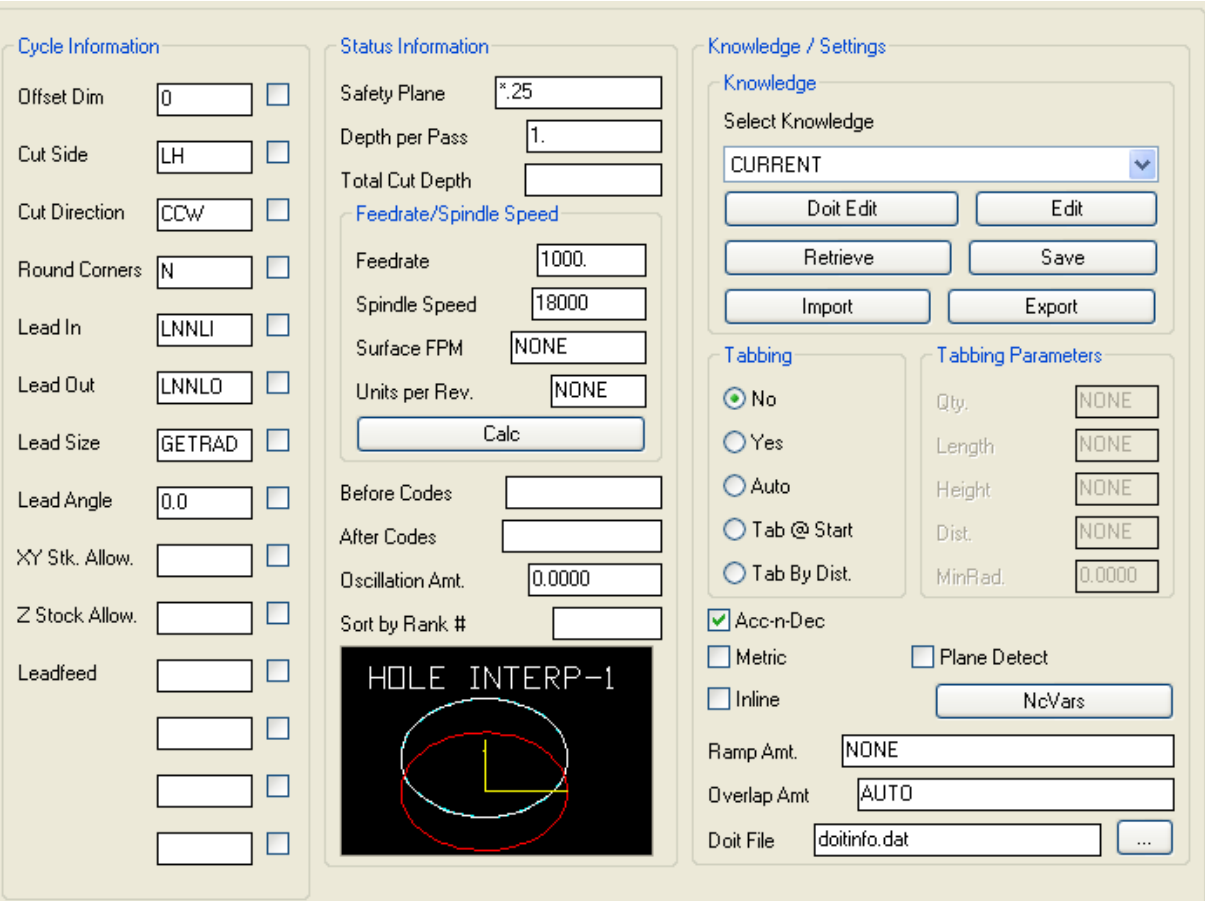
These cut cycles will allow the tool to start at the center of a circle, plunge down to the cut depth, interpolate the circle back to the starting point and then move back to the center of the circle and retract to the safety plane. The difference between them is that Hole Interpolation 2 will have an overlap in the cut past the start point before the tool moves back to the center of the circle.

This cycle only works on circles or arcs.
You should only use this cycle when there will be no slug left behind in the hole that could be expelled from the part and cause injury.



Hole-Interpolation 1 tool paths.

© 2011 CIM-TECH.COM, Inc



Hole-Interpolation-1 parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

© 2011 CIM-TECH.COM, Inc

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance

© 2011 CIM-TECH.COM, Inc

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

© 2011 CIM-TECH.COM, Inc

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

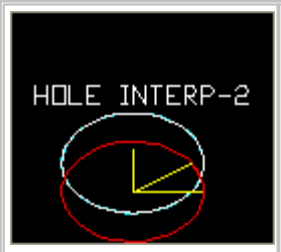
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

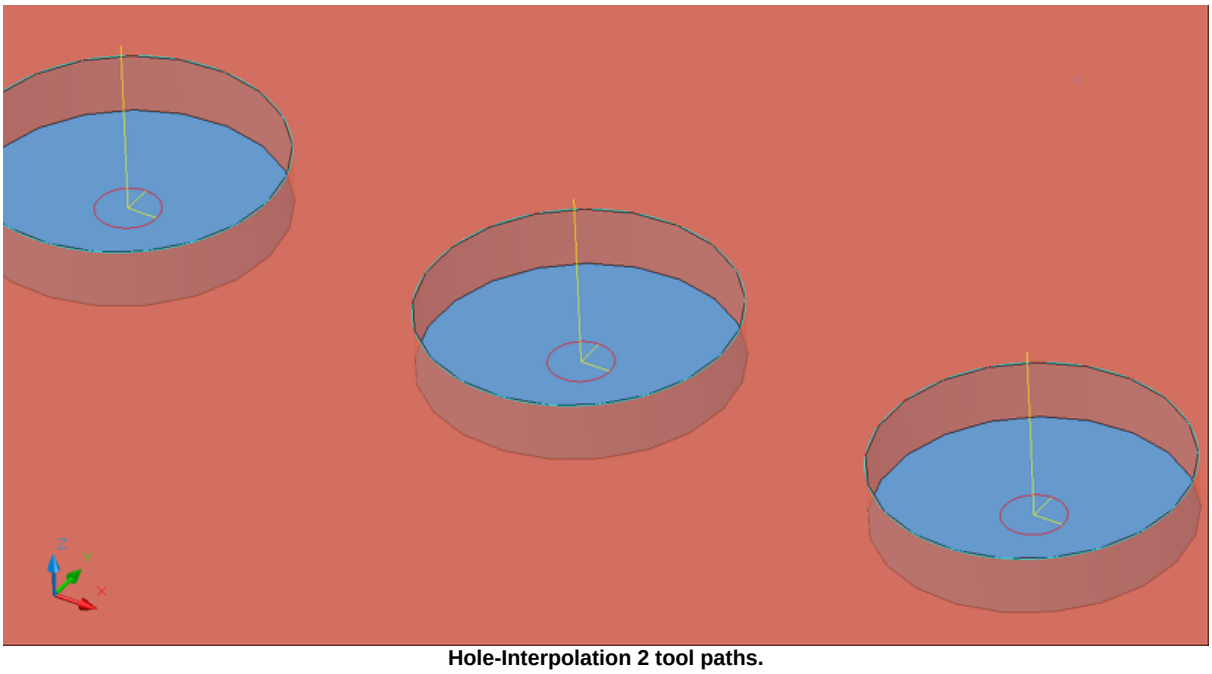
© 2011 CIM-TECH.COM, Inc

6.1.2.12 Hole-Interpolation 2



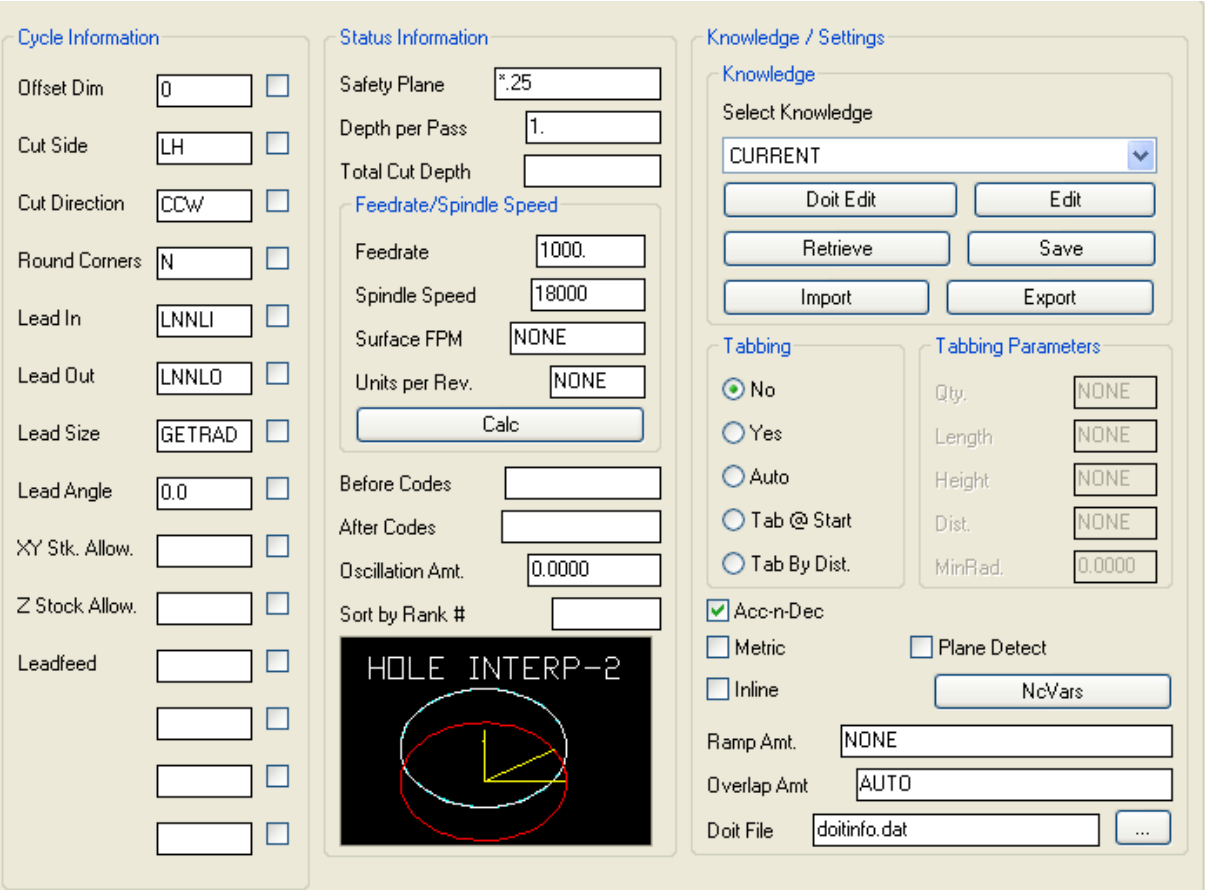
These cut cycles will allow the tool to start at the center of a circle, plunge down to the cut depth, interpolate the circle back to the starting point and then move back to the center of the circle and retract to the safety plane. The difference between them is that Hole Interpolation 2 will have an overlap in the cut past the start point before the tool moves back to the center of the circle.

This cycle only works on circles or arcs.
You should only use this cycle when there will be no slug left behind in the hole that could be expelled from the part and cause injury.



Hole-Interpolation 2 tool paths.

© 2011 CIM-TECH.COM, Inc



Hole-Interpolation 2 parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z. Entering a negative number will move the tool path DOWN in Z.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a

© 2011 CIM-TECH.COM, Inc

negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.3 Pocketing Cycles

In pocketing, you have to remove material from the interior of a closed geometry. Because there are many types of tools and materials, there are several pocketing cycles in Router-CIM. Each is intended to behave slightly differently to accomplish the task of pocketing with various results. Some of the cycles will make a linear motion to clear the pocket, and some will make pattern following motions to clear the pocket. Each cycle has the ability to make clean up passes around the perimeter of the shape and also separate passes to clean up around islands that may be in the shape.

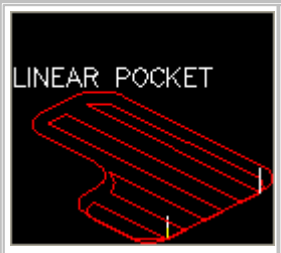
When the geometry consists of one or more profiles and none of them are enclosed or intersect with one another, each is cut as a separate pocket without islands.

When the geometry consists of one or more profiles, any of them which are enclosed or intersect with another are treated as an island. You can define several islands within a single pocket.

The descriptions given for each cycle will explain the various parameters available and how they function.

There are some examples used for each cycle and those drawings are available to download. A link is provided in each section.

6.1.3.1 Linear-Pocket

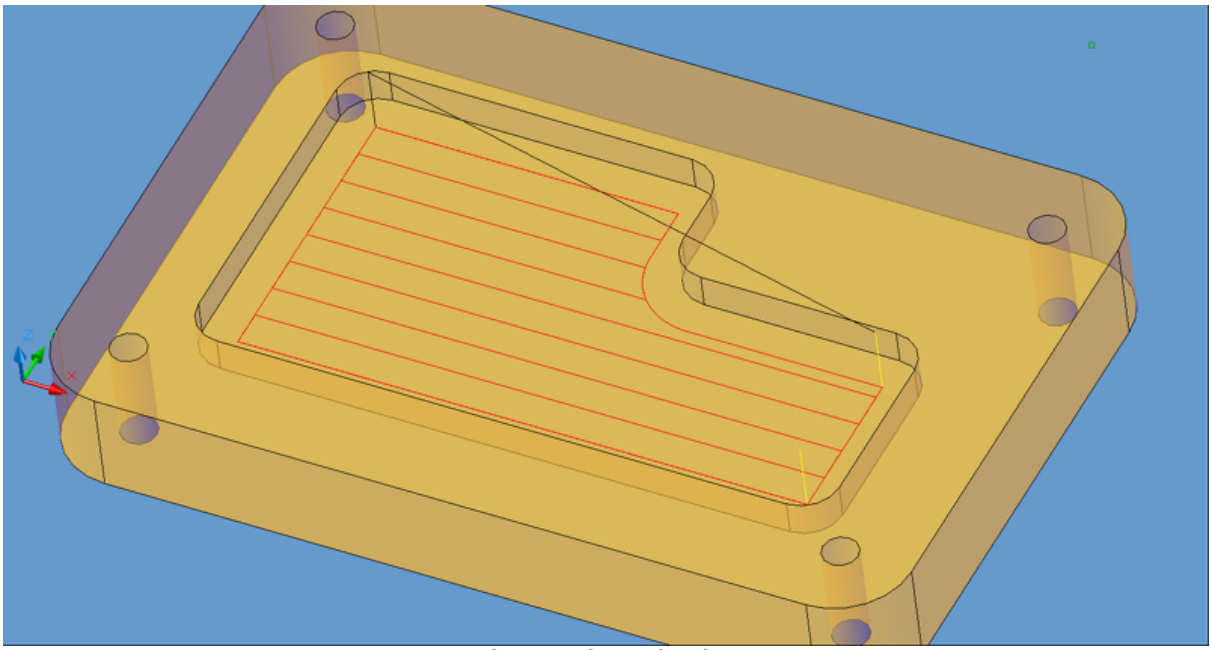


Linear Pocketing will move the tool across the shape to be cut in a straight back and forth pattern, removing material with each pass. There is a provision for a finish pass to clean up the profile of the shape and also provisions for clean up passes around islands in the shape.

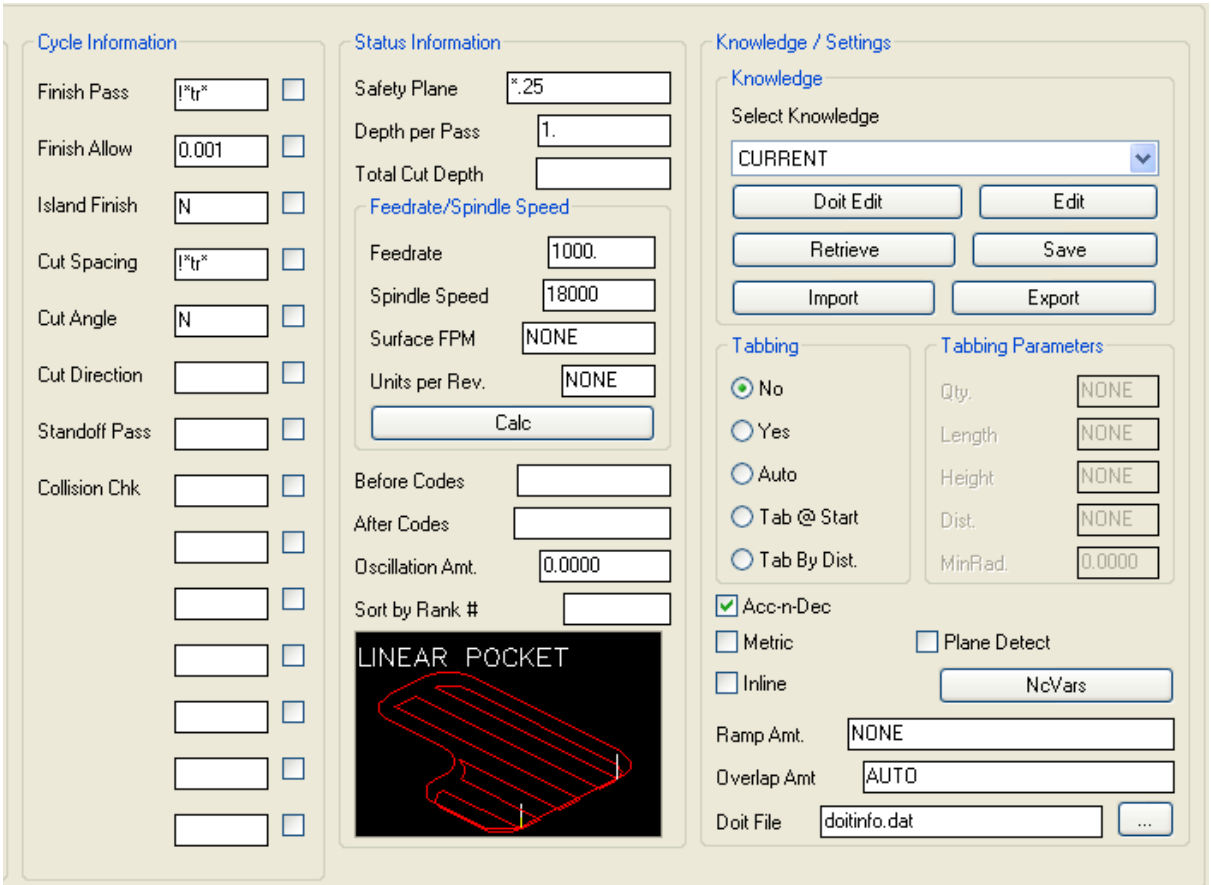
The direction of the cut (in degrees) can be controlled give a cut in the X direction, Y direction, or at any angle.

Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.

© 2011 CIM-TECH.COM, Inc



Linear Pocket tool paths.



Linear Pocket parameters.

© 2011 CIM-TECH.COM, Inc

The following parameters effect the toolpath creation:

Finish Pass
The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.
The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance
The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish
Either Y or N are valid values here.
N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.
If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing
This value is the spacing between each pass of the tool in the pocket.
Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle
Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut.
The value given is in degrees.
N tells Router-CIM to determine the direction on its own.

Cut Direction
Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass
A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.
A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

Collision Check
The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.
When set to Y, the pocketing routine will run slower than normal.

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

© 2011 CIM-TECH.COM, Inc

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

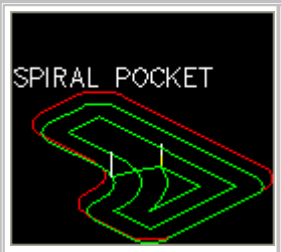
After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

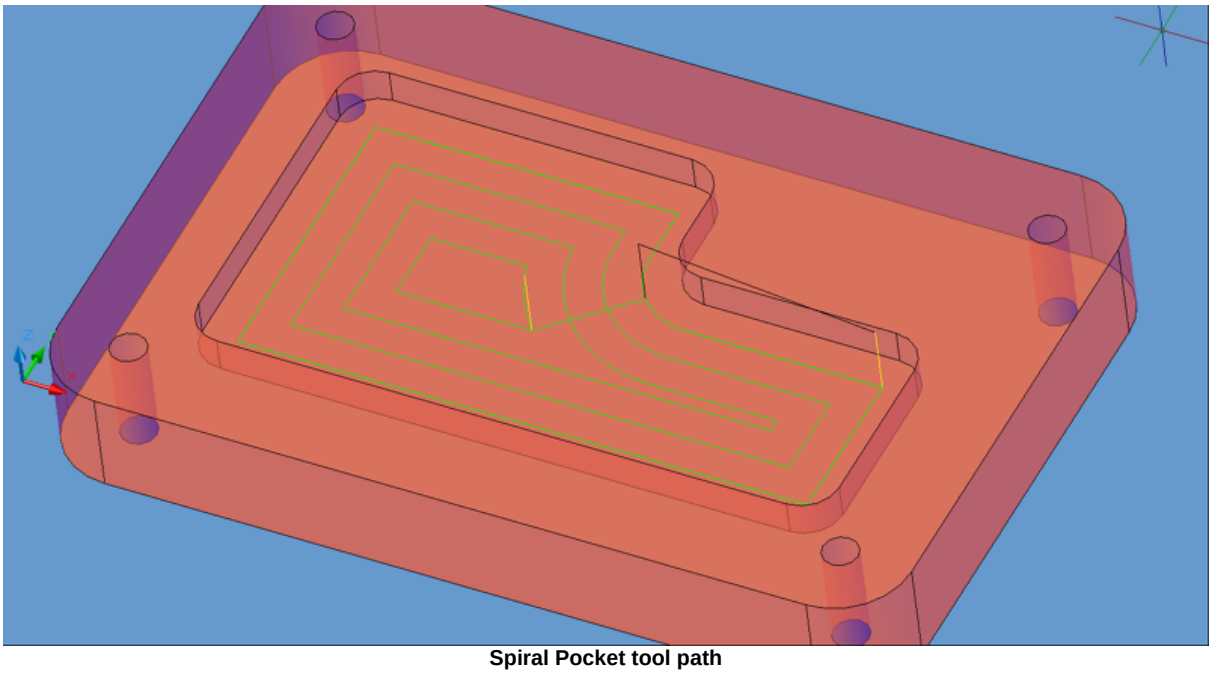
© 2011 CIM-TECH.COM, Inc

6.1.3.2 Spiral-Pocket



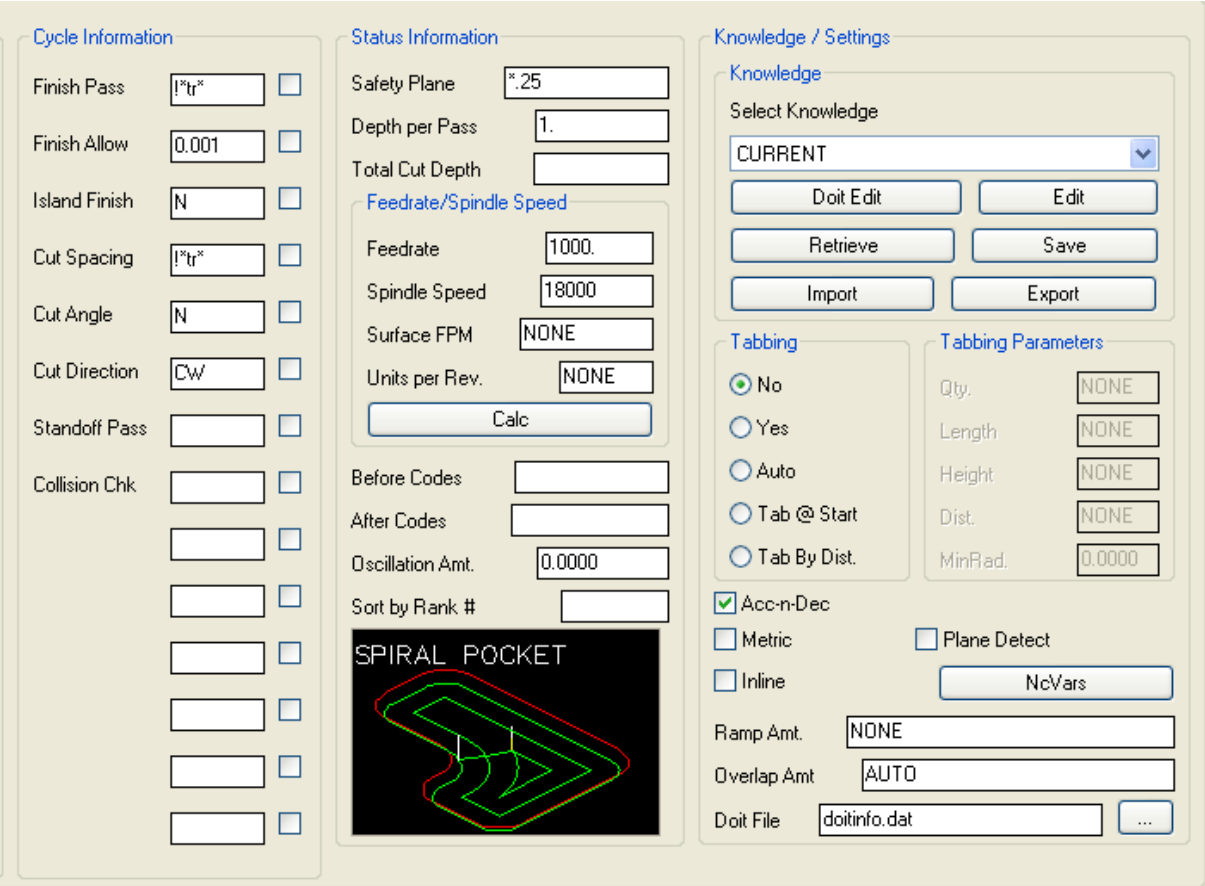
Spiral Pocketing will create pocketing tool motions that will be offsets of the shape of the pocket. A Spiral Pocket can contain islands and there are parameters in the cycle to allow for a finish pass around islands and also around the perimeter of the pocket.

Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.



Spiral Pocket tool path

© 2011 CIM-TECH.COM, Inc



Spiral Pocketing Parameters.

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.
The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.
N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.
If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.

© 2011 CIM-TECH.COM, Inc

Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle
N tells Router-CIM to determine the direction on its own. This is the default for Spiral Pocketing where the angle of each pass is determined by the geometry of the shape.
Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut with Linear Pocketing. It has no effect for Spiral Pocketing.

Cut Direction
Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass
A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.
A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

Collision Check
The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.
When set to Y, the pocketing routine will run slower than normal.

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order

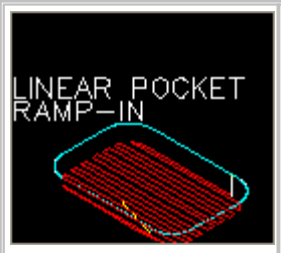
© 2011 CIM-TECH.COM, Inc

when the code is created.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.3.3 Linear Pocket Ramp-In

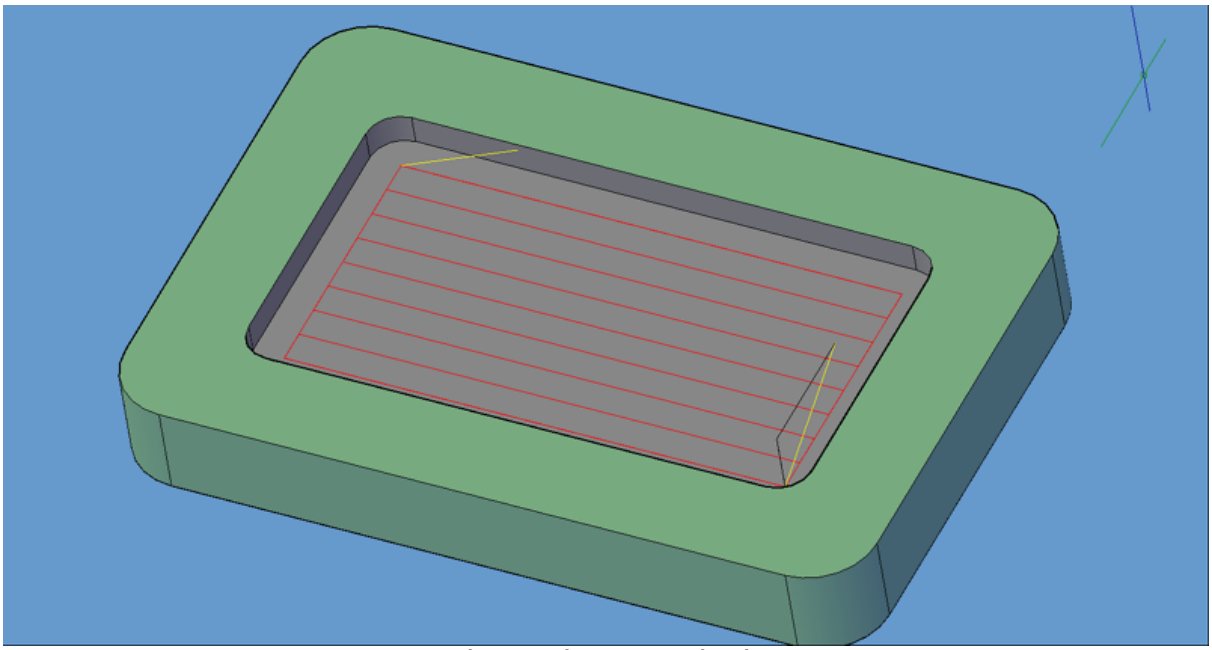


Linear Pocketing will move the tool across the shape to be cut in a straight back and forth pattern, removing material with each pass. There is a provision for a finish pass to clean up the profile of the shape and also provisions for clean up passes around islands in the shape.

The difference with Linear Pocket Ramp-In is that the first pass will ramp into the material instead of making a plunge cut.

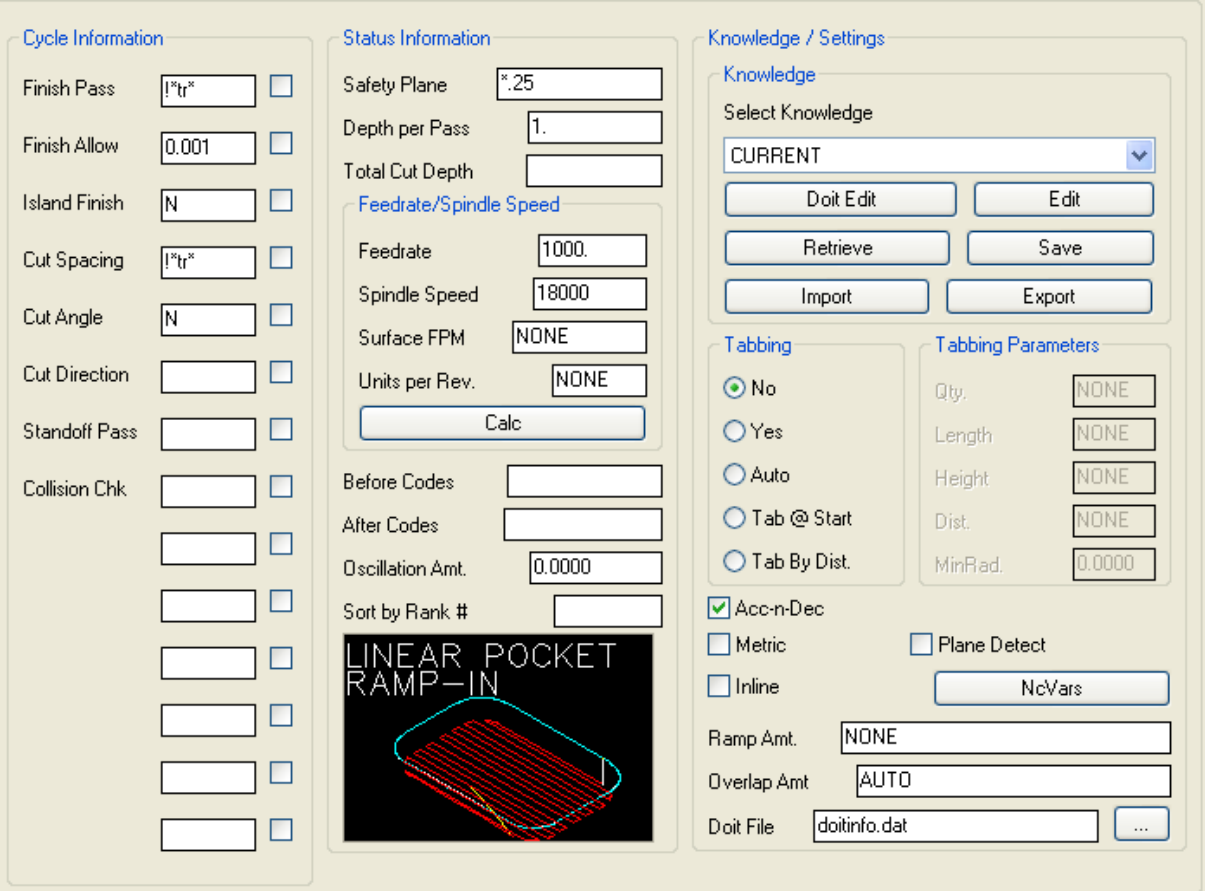
The direction of the cut (in degrees) can be controlled give a cut in the X direction, Y direction, or at any angle.

Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.



Linear Pocket Ramp tool path

© 2011 CIM-TECH.COM, Inc



Linear Pocket Ramp parameters.

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.
The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.
N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.
If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.
Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass

© 2011 CIM-TECH.COM, Inc

spacing to that number.

Cut Angle
Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut. The value given is in degrees.
N tells Router-CIM to determine the direction on its own.

Cut Direction
Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass
A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.
A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

Collision Check
The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.
When set to Y, the pocketing routine will run slower than normal.

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

© 2011 CIM-TECH.COM, Inc

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

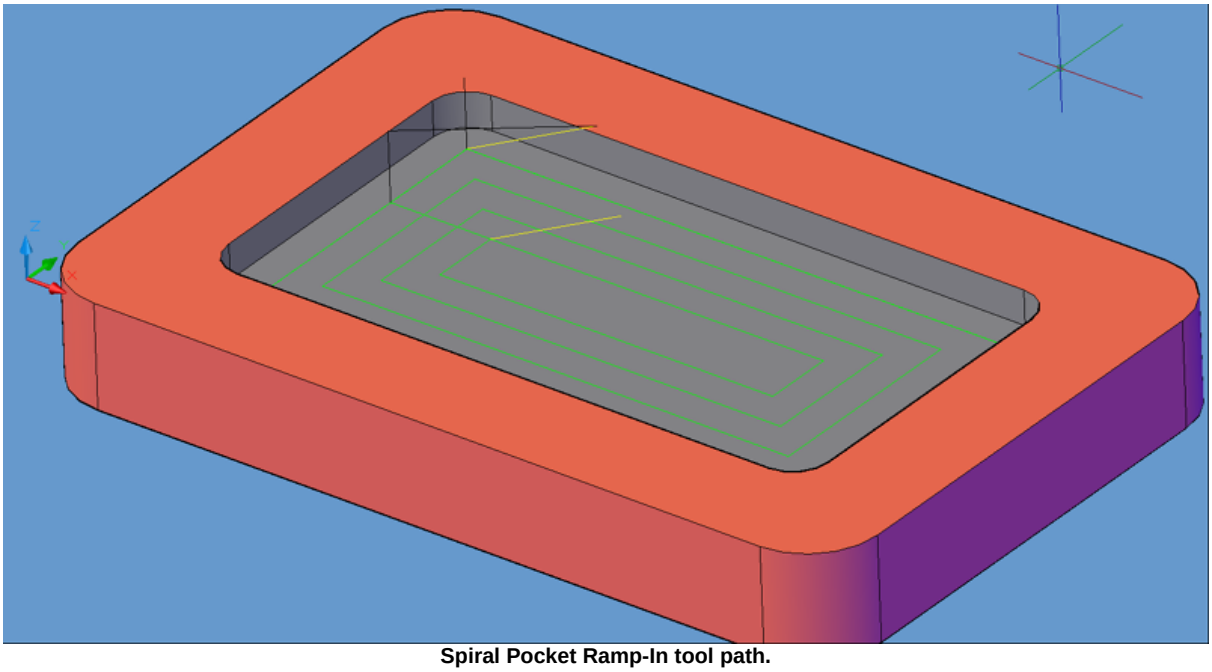
6.1.3.4 Spiral Pocket Ramp-In



Spiral Pocketing will create pocketing tool motions that will be offsets of the shape of the pocket. A Spiral Pocket can contain islands and there are parameters in the cycle to allow for a finish pass around islands and also around the perimeter of the pocket.

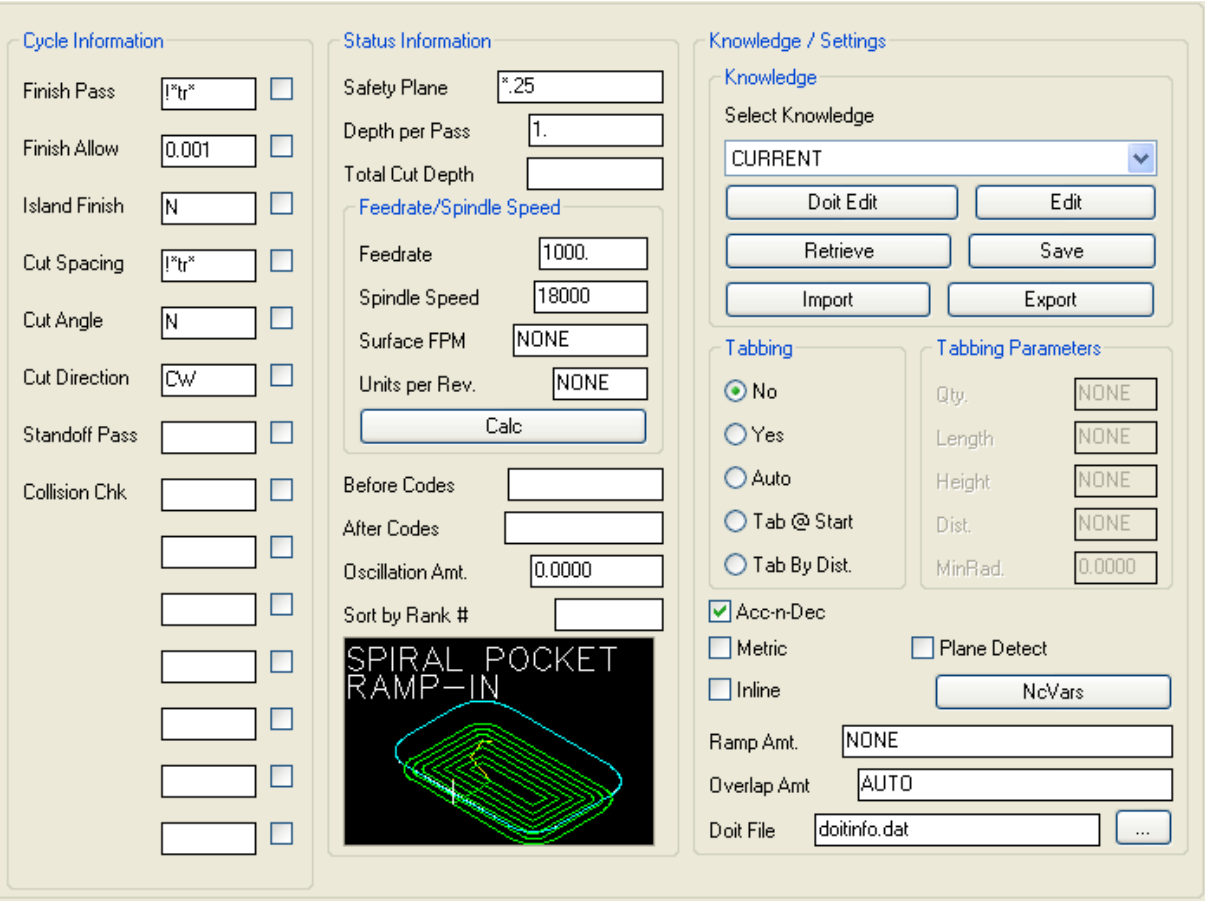
The difference with Spiral Pocket Ramp-In is the move into the material will be a ramping lead in, instead of a plunge cut.

Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.



Spiral Pocket Ramp-In tool path.

© 2011 CIM-TECH.COM, Inc



Spiral-Pocket-Ramp parameters.

The following parameters effect the toolpath creation:

Finish Pass

The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.
The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance

The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish

Either Y or N are valid values here.
N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.
If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing

This value is the spacing between each pass of the tool in the pocket.
Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass

© 2011 CIM-TECH.COM, Inc

spacing to that number.

Cut Angle
N tells Router-CIM to determine the direction on its own. This is the default for Spiral Pocketing where the angle of each pass is determined by the geometry of the shape.
Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut with Linear Pocketing. It has no effect for Spiral Pocketing.

Cut Direction
Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass
A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.
A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

Collision Check
The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.
When set to Y, the pocketing routine will run slower than normal.

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

© 2011 CIM-TECH.COM, Inc

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.3.5 Linear Pocket Lead Hole



Linear Pocketing will move the tool across the shape to be cut in a straight back and forth pattern, removing material with each pass. There is a provision for a finish pass to clean up the profile of the shape and also provisions for clean up passes around islands in the shape.

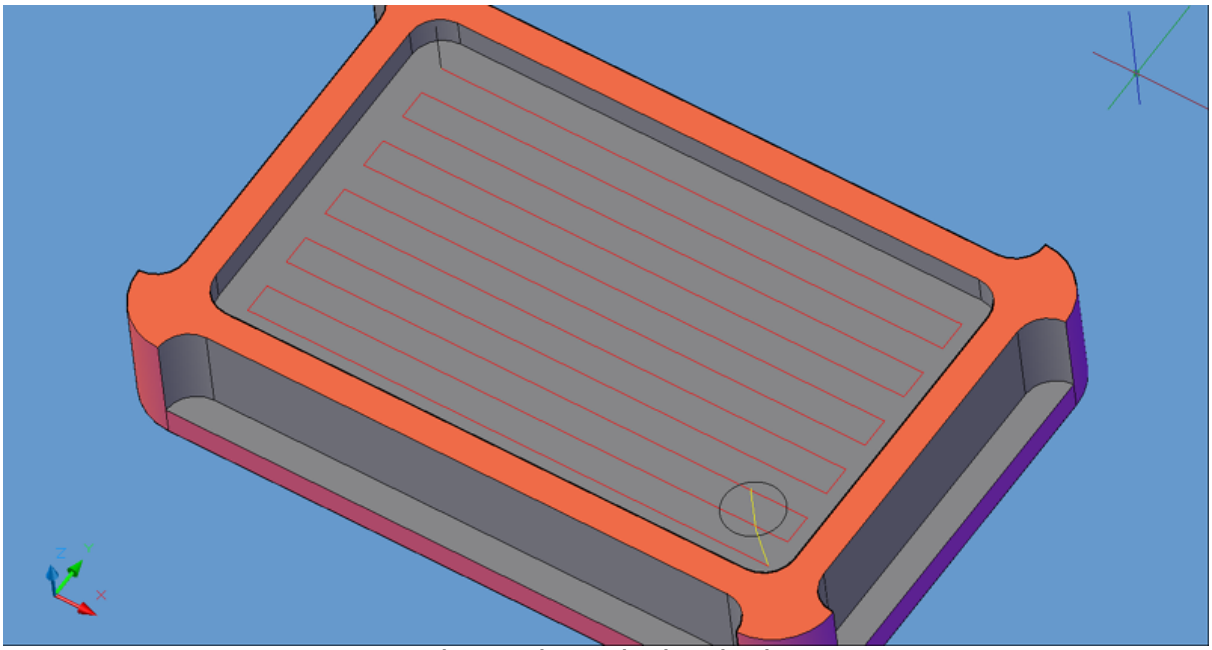
The direction of the cut (in degrees) can be controlled give a cut in the X direction, Y direction, or at any angle.

Linear Pocket Lead-Hole will allow the tool to plunge into a hole that has already been created in the part to avoid the tool plunging into the material and breaking.

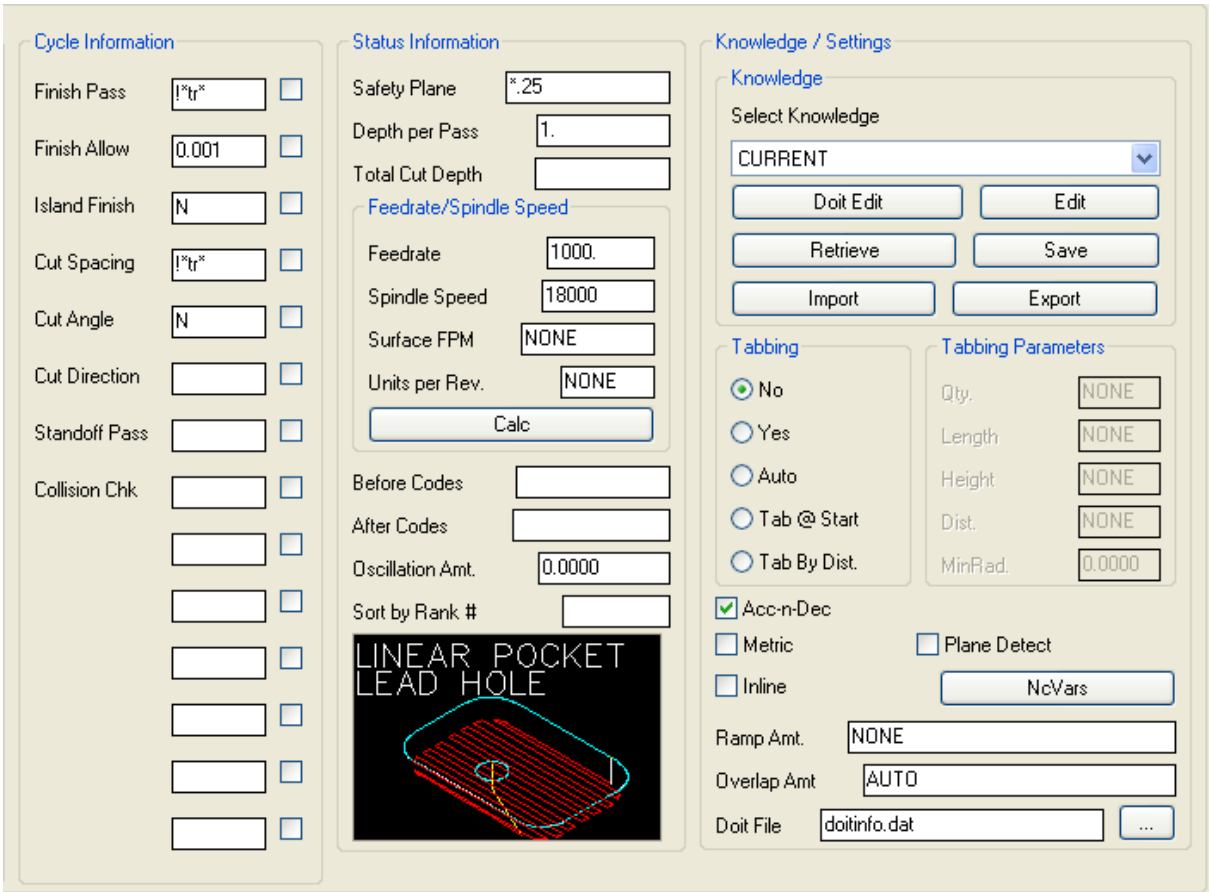
Pocketing does not use cutter radius compensation due to the fact that areas of the cut may exist where the tool could be cutting on both the left and right side and cutter radius compensation would ruin the part. If a finish pass is needed with cutter radius compensation, leave material in the pocket with the finish allowance and make a separate pass with another cut cycles.

During the cut, you will be prompted to select the lead hole location. Use the center point OSNAP and pick the center of the hole into which you want the tool to plunge to make the lead in. Note that the tool will plunge into this hole and make a linear move at feedrate to the start of the shape, so the hole should be placed fairly close to the start point to avoid unnecessary cycle time.

© 2011 CIM-TECH.COM, Inc



Linear-Pocket-Lead-Hole tool path.



Linear Pocket Lead-Hole parameters.

© 2011 CIM-TECH.COM, Inc

The following parameters effect the toolpath creation:

Finish Pass
The normal response here is !*TR* (Tool Radius). The pocketing tool will stay away from the inside of the pocket and the outside of any islands by the value of this parameter.
The value here is added to Finish Allowance to provide for a finishing tool to clean up the pocket if necessary.

Finish Allowance
The value entered here will be added to Finish Pass above to provide material left for a clean up pass on the pocket with a separate tool.

Island Finish
Either Y or N are valid values here.
N tells Router-CIM to apply the same values specified in Finish Pass and Finish Allowance to all islands contained in the pocket.
If Y is entered, you will be prompted for different values to use for the Island pass offset during the cut.

Cut Spacing
This value is the spacing between each pass of the tool in the pocket.
Using !*TR* will offset each pass by the Tool Radius. Entering a numeric value will set the pass spacing to that number.

Cut Angle
Changing Cut Angle will change the direction of the linear passes made inside the pocket during cut.
The value given is in degrees.
N tells Router-CIM to determine the direction on its own.

Cut Direction
Valid entries are CW or CCW for clockwise or counter-clockwise. Leaving the parameter blank will default to CCW in a linear pocket.

Standoff Pass
A Standoff pass is described as a tool path that travels around the island(s) and the inside of the pocket after the pocket roughing tool path has been created. The default response is blank or N (no). No Standoff pass will be created.
A Standoff pass will be produced when a value is entered in this parameters. If Y (yes) is entered, you will be prompted during cut for the Standoff Pass amount.

Collision Check
The default entry is left blank or N (no). When Y (yes) is entered, the routine will check to see if islands collide (on first offset) with other islands, or if islands collide (on first offset) with the pocket. This collision detection prevents a tool path from being created when a tool has a diameter too large to traverse between islands and/or between islands and the pocket.
When set to Y, the pocketing routine will run slower than normal.

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth

© 2011 CIM-TECH.COM, Inc

This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

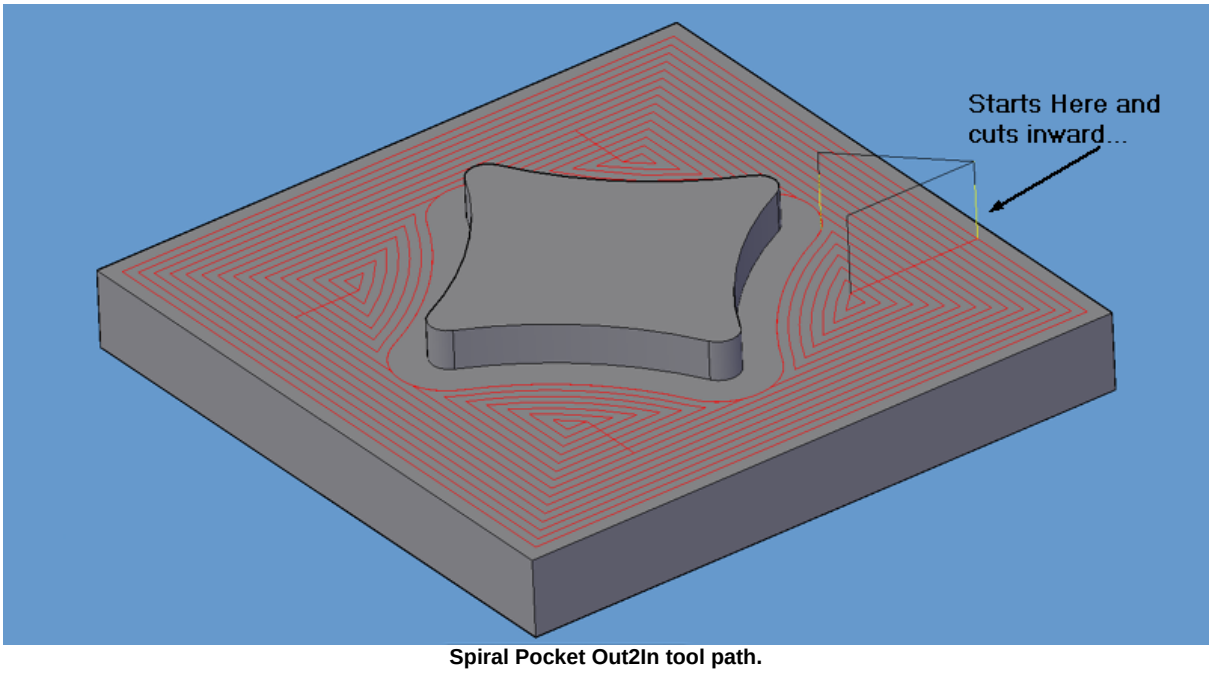
© 2011 CIM-TECH.COM, Inc

6.1.3.6 Spiral Out2In



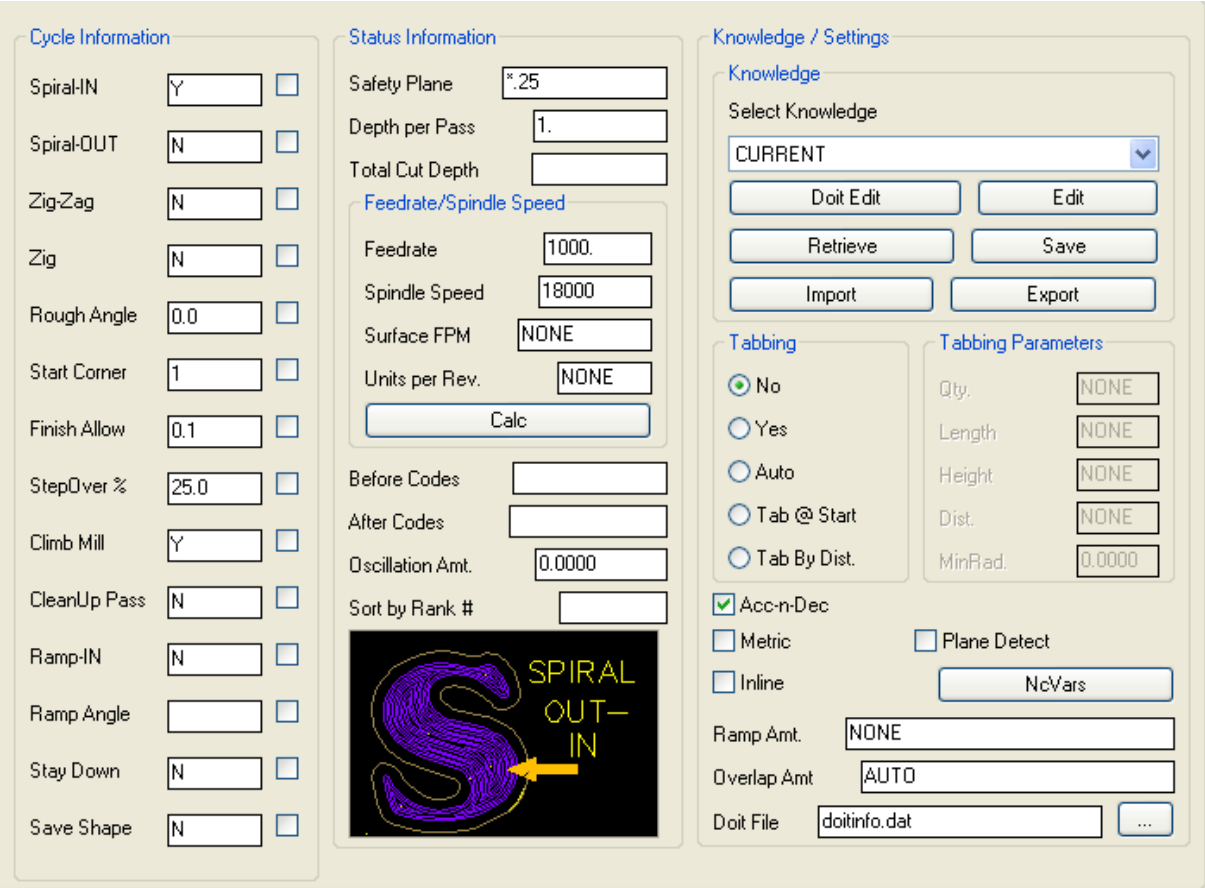
This version of Spiral Pocketing (in both the In-Out and Out-In cycles) uses a different algorithm and contains many options to control the creation of the tool path than the other Spiral Pocketing cycles. This method allows for more complex geometry to be handed to the tool path generator.

Spiral Pocket Out-In will start at a point at the outside edge of the pocket and create offset tool paths, moving toward the inside of the pocket. There are parameters to allow a plunge start or a ramp start, and also options for keeping the tool down while in the pocket or allowing the tool to pick up and move to another are of the pocket to continue the shape.



Spiral Pocket Out2In tool path.

© 2011 CIM-TECH.COM, Inc



Spiral Out2In parameters.

The following parameters effect the toolpath creation:

Spiral-IN

Spiral – OUT

Zig-Zag

Zig

Rough Angle

Start Corner

Finish Allow

© 2011 CIM-TECH.COM, Inc

Step Over %

Climb Mill

CleanUp Pass

Ramp-IN

Ramp Angle

Stay Down

Save Shape

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

**Changing values in the cycle parameters may yield unexpected

© 2011 CIM-TECH.COM, Inc

results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.

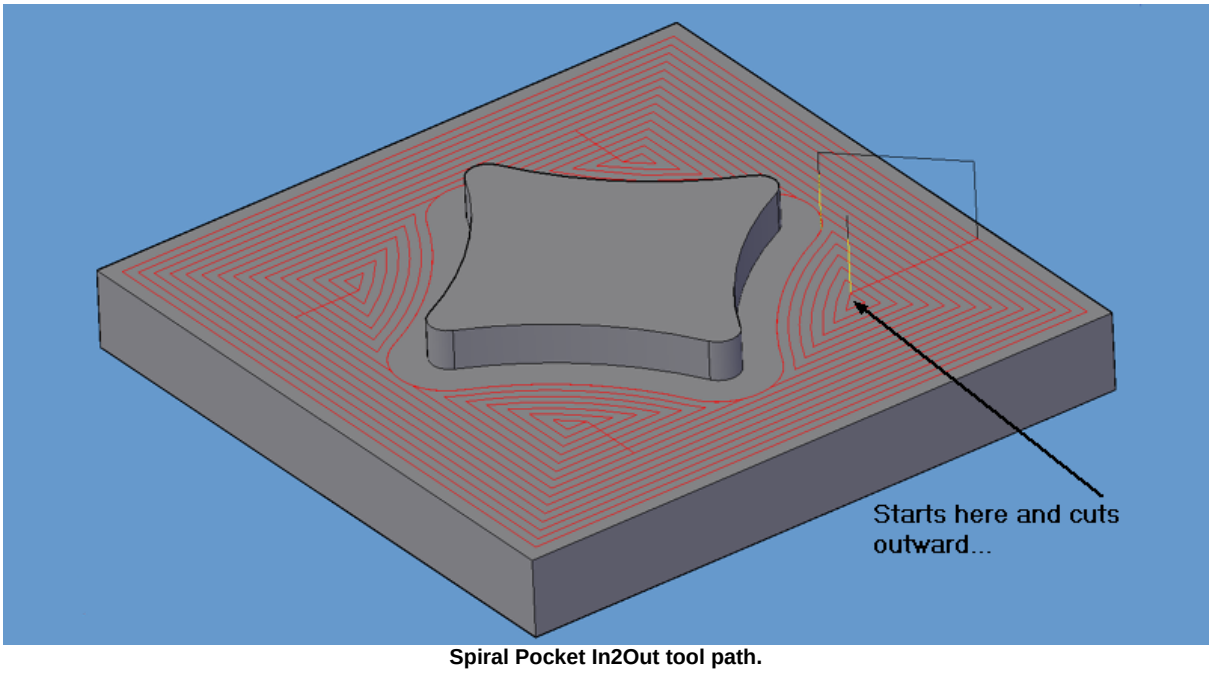
© 2011 CIM-TECH.COM, Inc

6.1.3.7 Spiral In2Out



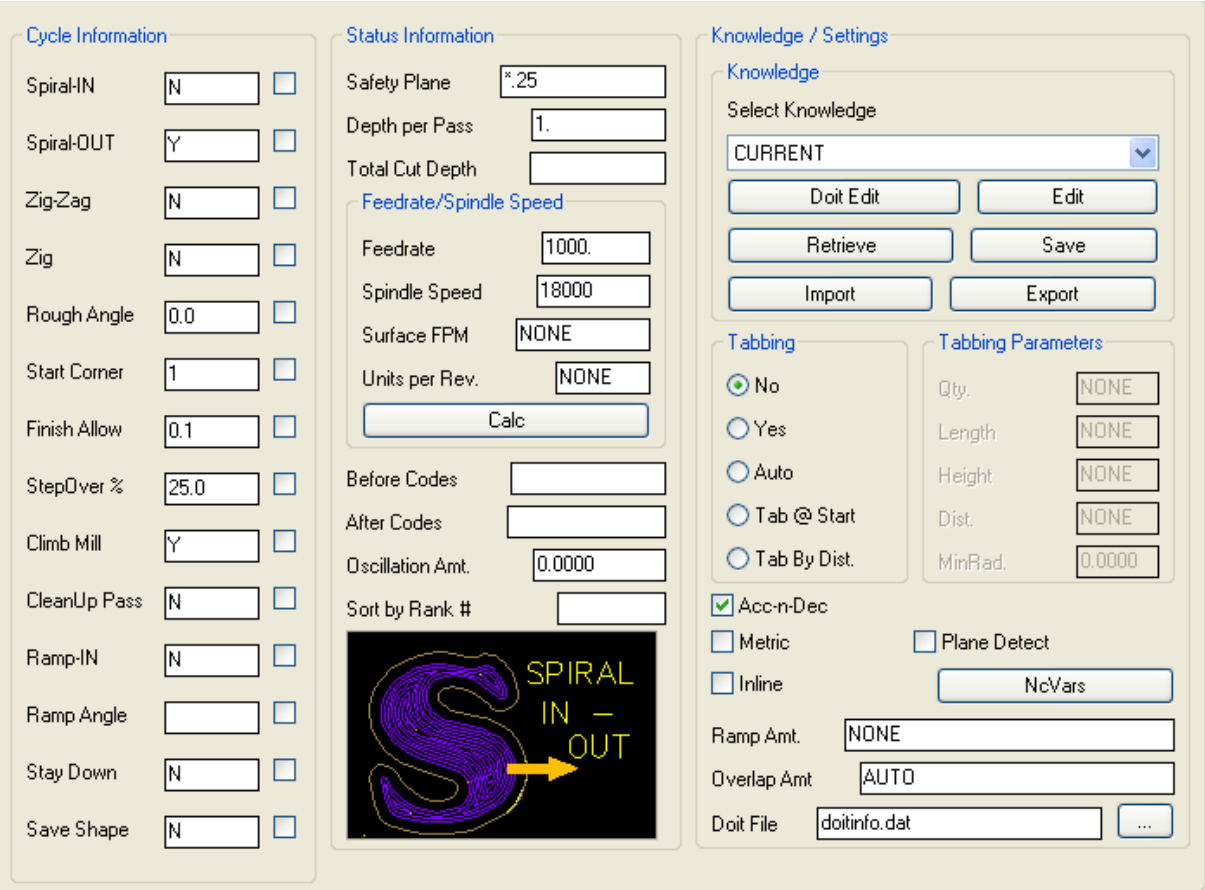
This version of Spiral Pocketing (in both the In-Out and Out-In cycles) uses a different algorithm and contains many options to control the creation of the tool path than the other Spiral Pocketing cycles. This method allows for more complex geometry to be handed to the tool path generator.

Spiral Pocket In-Out will start at a point inside the pocket and create offset tool paths, moving toward the outside of the pocket. There are parameters to allow a plunge start or a ramp start, and also options for keeping the tool down while in the pocket or allowing the tool to pick up and move to another are of the pocket to continue the shape.



Spiral Pocket In2Out tool path.

© 2011 CIM-TECH.COM, Inc



Spiral In2Out parameters.

The following parameters effect the toolpath creation:

Spiral-IN

Spiral – OUT

Zig-Zag

Zig

Rough Angle

Start Corner

Finish Allow

© 2011 CIM-TECH.COM, Inc

Step Over %

Climb Mill

CleanUp Pass

Ramp-IN

Ramp Angle

Stay Down

Save Shape

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

**Changing values in the cycle parameters may yield unexpected

© 2011 CIM-TECH.COM, Inc

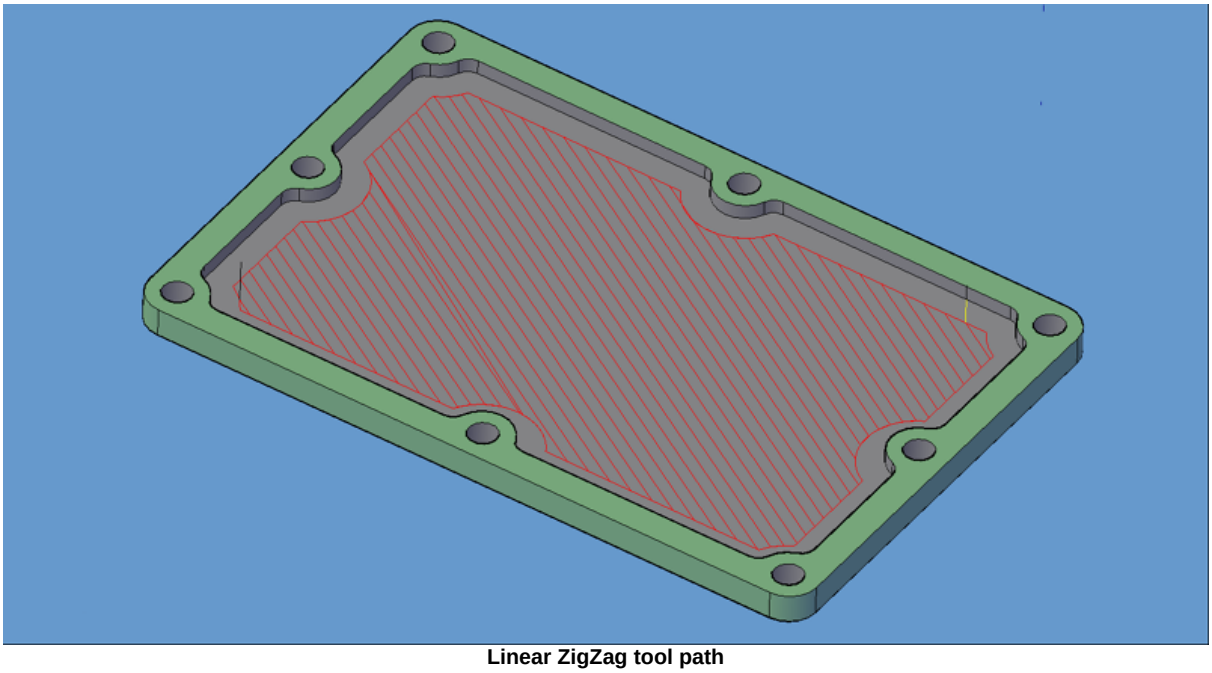
results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.

© 2011 CIM-TECH.COM, Inc

6.1.3.8 Linear Pocket ZigZag

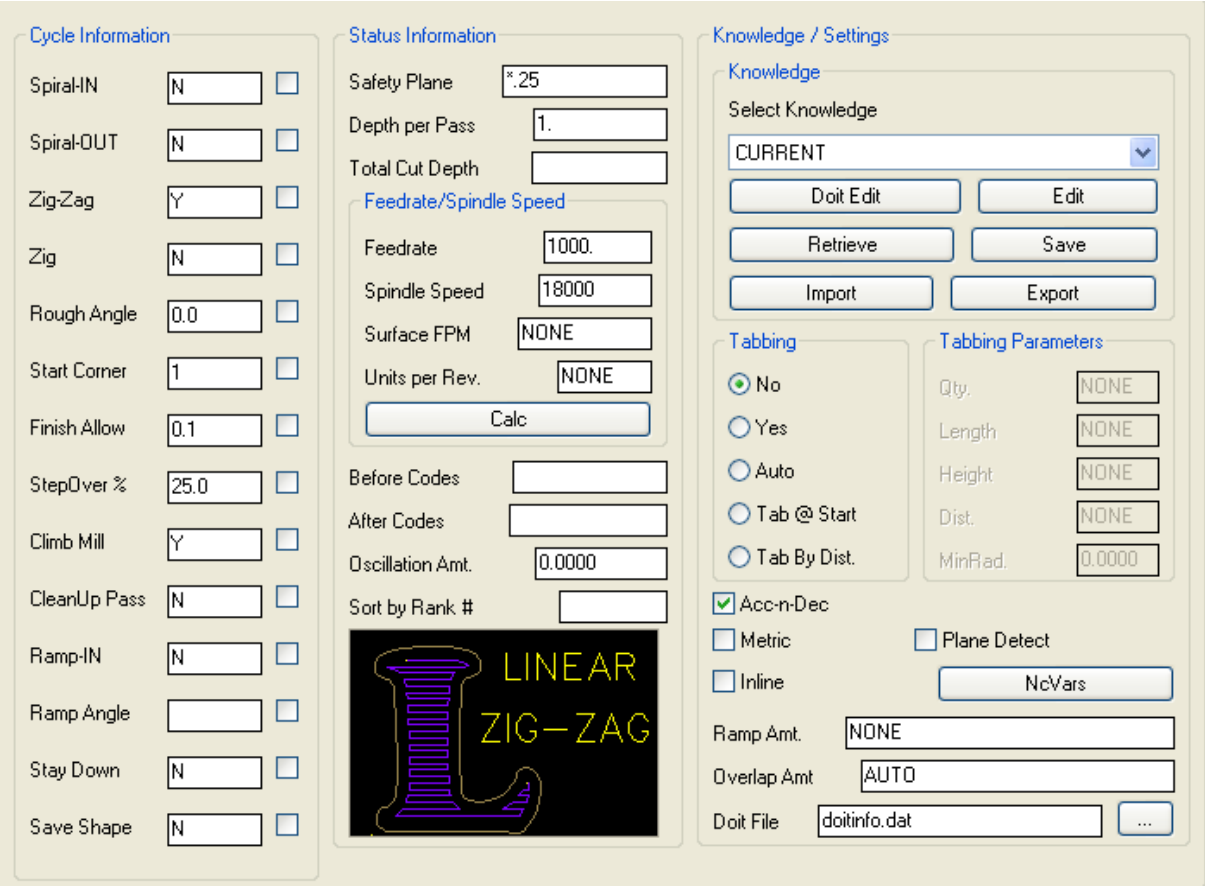


This version of Linear Pocketing uses a different algorithm and contains many options to control the creation of the tool path than the other Linear Pocketing cycle. This method allows for more complex geometry to be handed to the tool path generator.
Linear Zig-Zag will cut a pocket in a linear, back and forth motion, cutting in both directions. This version of Linear Pocketing has several options in the setup of the cut, such as cut direction, finish pass allowances, and also tool stay down during the cut.



Linear ZigZag tool path

© 2011 CIM-TECH.COM, Inc



Linear Zig Zag parameters.

The following parameters effect the toolpath creation:

Spiral-IN

Spiral – OUT

Zig-Zag

Zig

Rough Angle

Start Corner

Finish Allow

© 2011 CIM-TECH.COM, Inc

Step Over %

Climb Mill

CleanUp Pass

Ramp-IN

Ramp Angle

Stay Down

Save Shape

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

© 2011 CIM-TECH.COM, Inc

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

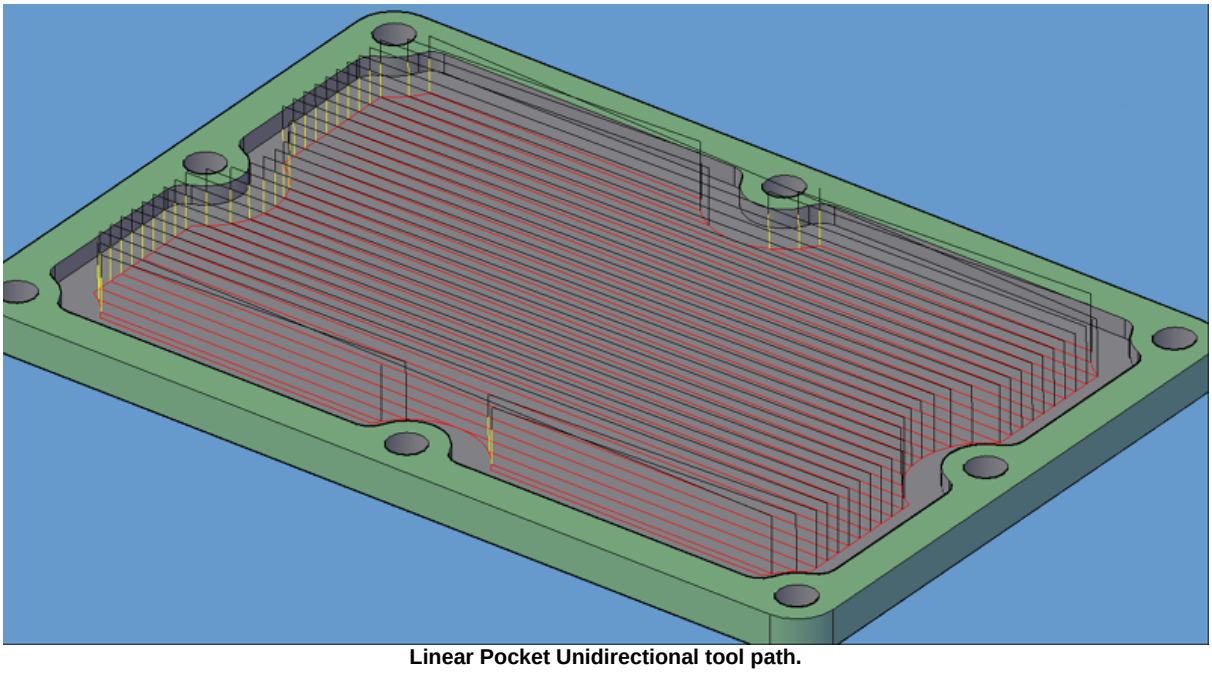
6.1.3.9 Linear Pocket Unidirectional



Linear Unidirectional Pocketing will create a linear pocket cut with the tool cutting in one direction. At the end of the cut the tool will retract, move back to the start of the next pass, then plunge and cut in the same direction as the last cut, retract, and start over.

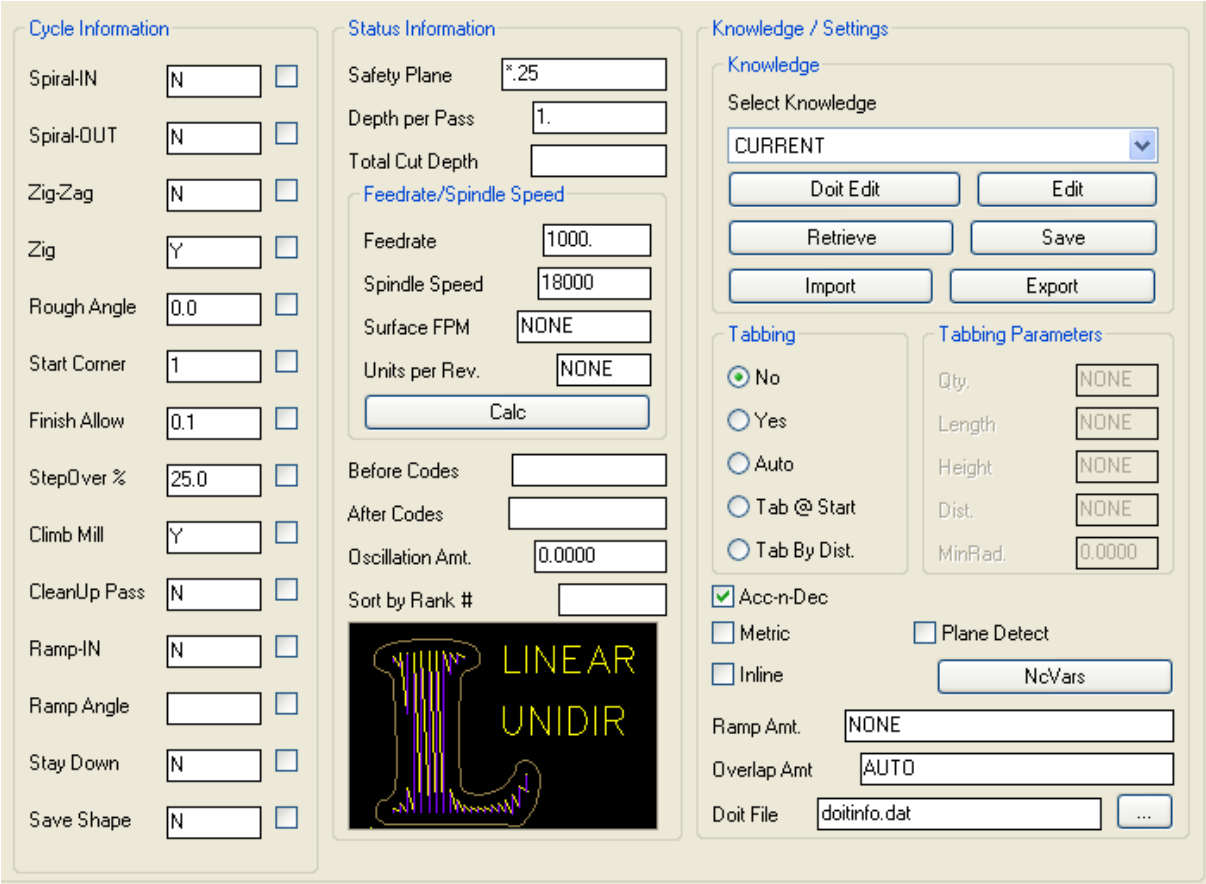
This is a useful cycle when the geometry of the tool or condition of the material will only permit a satisfactory finish with the tool cutting in one direction (climb or conventional milling).

There is a provision for a separate finish pass at the end, which will follow the contour of the entire shape to clean up any tool marks on the edge of the cut.



Linear Pocket Unidirectional tool path.

© 2011 CIM-TECH.COM, Inc



Linear Pocket Unidirectional parameters.

The following parameters effect the toolpath creation:

Spiral-IN

Spiral – OUT

Zig-Zag

Zig

Rough Angle

Start Corner

Finish Allow

© 2011 CIM-TECH.COM, Inc

Step Over %

Climb Mill

CleanUp Pass

Ramp-IN

Ramp Angle

Stay Down

Save Shape

Safety Plane
The safety plane is the index plane Z location. If a ' * ' is used as the first character, that position is absolute in world Z coordinates, otherwise it is considered to be the distance above the shape.

Depth Per Pass
This controls the depth per pass in Z. It is also the initial Peck Increment.

Total Cut Depth
This parameter controls the total depth of the cut. If a ' * ' is used as the first character, that position is absolute in world Z coordinates. If it does not, then that distance is considered to be the distance below the initial shape.

Feedrate
Initial feedrate to start the drilling operation.

Spindle Speed
The RPM value to use for the spindle for this tool path.

Before Codes
Values placed here will be output in the cut cycle before the tool enters the material, typically at the height of the Safety Plane once the tool length compensation is set.

After Codes
Values placed here will be output in the cut cycle after the tool has retracted from the cut, typically at the height of the Safety Plane after the cut is finished.

Sort by Rank
A numeric value to use for the tool path created to allow the Sequence to place cuts in a specific order when the code is created.

© 2011 CIM-TECH.COM, Inc

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.4 Open Shape and Center Cutting Cycles

There may be several instances where cutting an open shape is desirable. One instance may be to use a profile or shaper tool to cut a visible edge, leaving the profile shape in the material. In that instance you need a cycle where you can control the cut side and direction of the cut to perform either a climb or conventional cut.

Center cutting cycles are also in this section. There are many instances where cutting a shape on center with no offset is necessary, such as engraving or slotting. Router-CIM has several cycles available to allow complete control over the cutting conditions.

The descriptions given for each cycle will explain the various parameters available and how they function.

There are some examples used for each cycle and those drawings are available to download. A link is provided in each section.

6.1.4.1 Heli-Lead Center

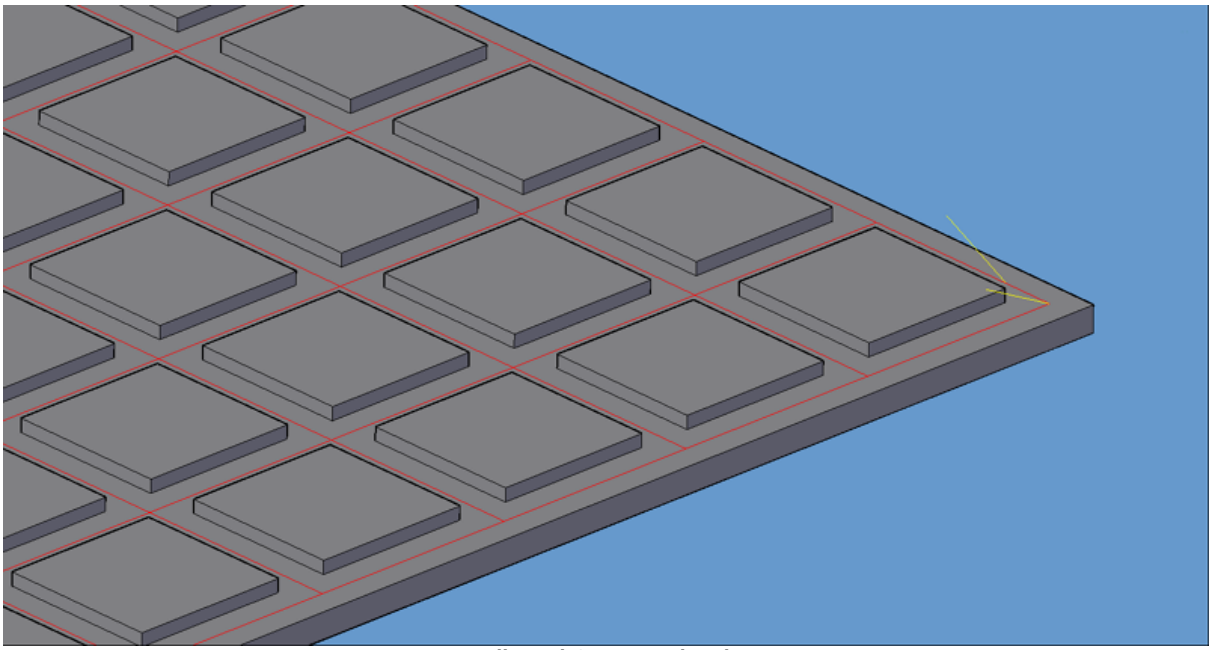


Heli-Lead Cutting cycles allow you to have a contour following lead-in and lead-out. This means that on a straight cut, the lead in and lead out would be a ramp (XZ or YZ motion) and on a non-straight shape the lead-in and lead-out will follow the contour of the shape, no matter the complexity.

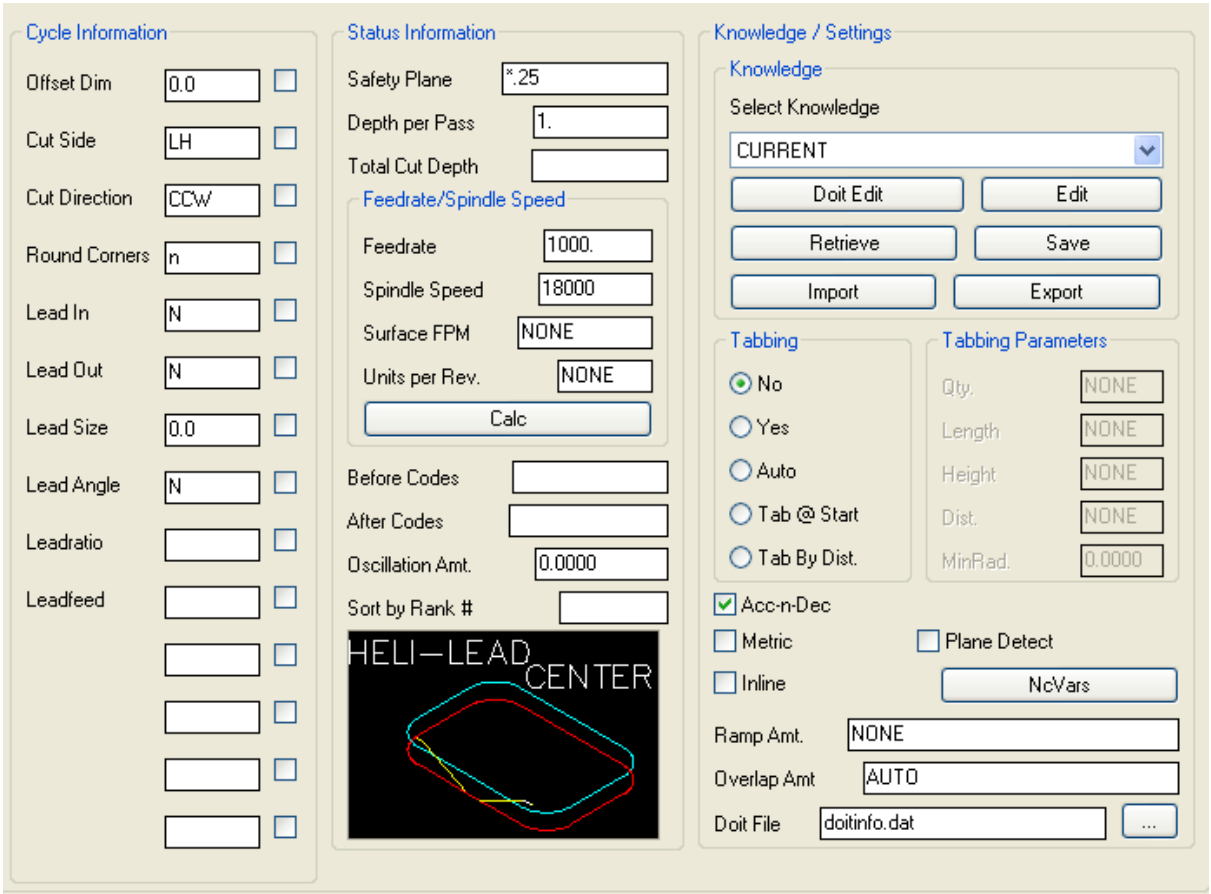
This cycle is especially useful when cutting stay-down tool path nests.

Typically, Heli-Lead Center will start above the part at the Safety Plane and then ramp into the part along the geoshape, following the contour, until it reaches full cut depth. The cycle will follow the shape back to the start point, overlap by the tool diameter, then ramp out of the shape, again following the contour, back up to the Safety Plane.
Heli-Lead Center is a center cutting (no offset) version of Heli-Lead Outside and Heli-Lead Inside.

© 2011 CIM-TECH.COM, Inc



Heli-Lead-Center tool path.



Heli-Lead-Center parameters.

© 2011 CIM-TECH.COM, Inc

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

© 2011 CIM-TECH.COM, Inc

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

Lead Feed
This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth
The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is

© 2011 CIM-TECH.COM, Inc

usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

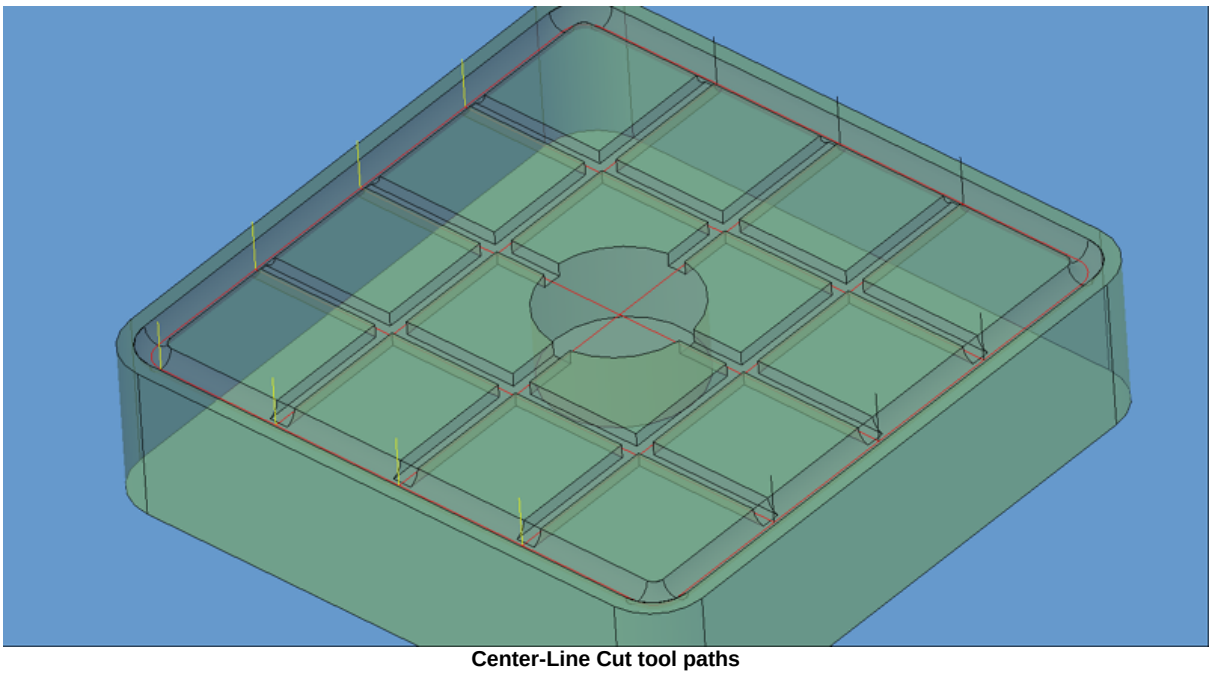
6.1.4.2 Center-Line Cut



These cycles allow you to create tool paths directly from any defined shapes. Use this cycle to make a fixture or to engrave geometry. This cycle is also used to cut tool-width slots. You can draw the gasket grooves and vacuum grooves for a spoil board, and use this cycle to follow that geometry. This cycle would normally be used with a tool that does not use Machine Cutter Compensation, since there is no offset created.

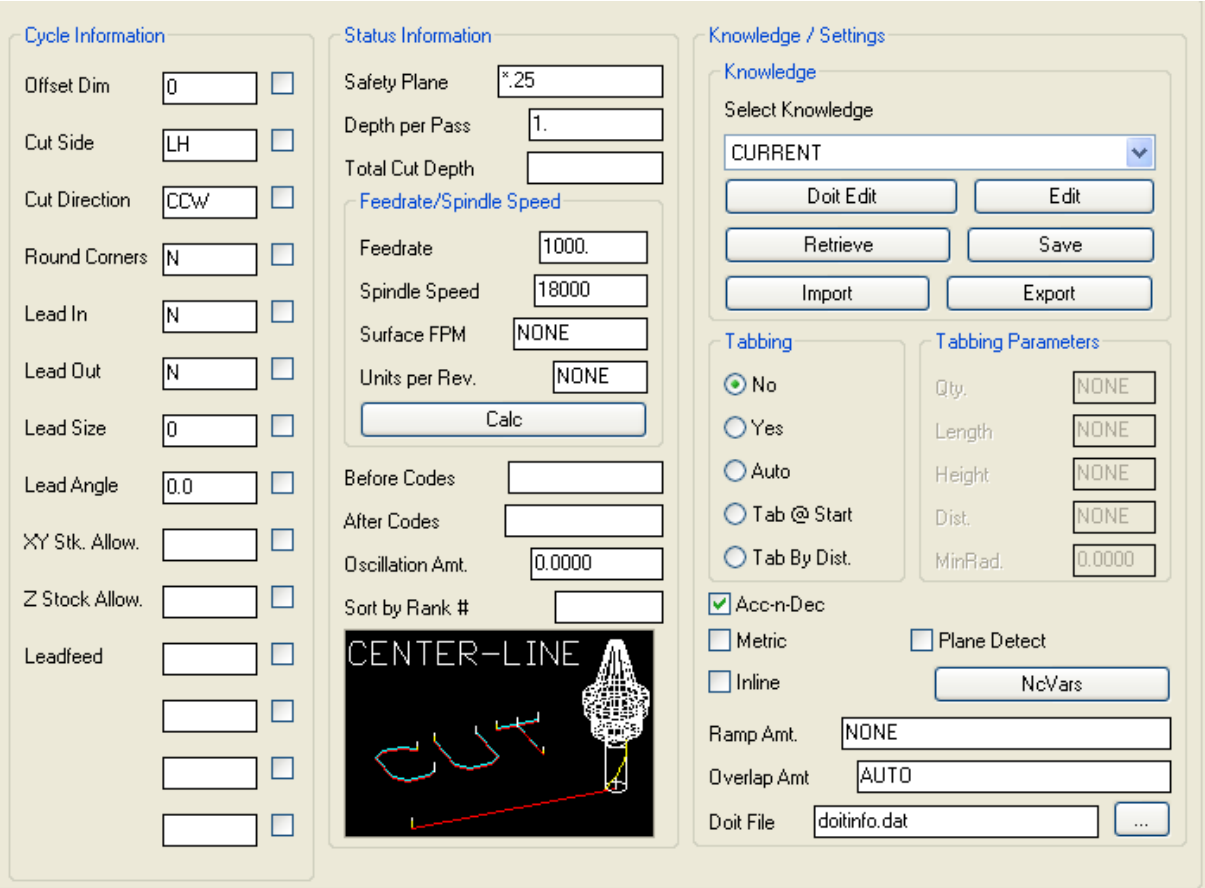
Cycle Start and End positions determine where to rapid to and where to feed to in the Z axis. Tool Depth of Cut affects the number of cutting passes.

Center-Line Cut will plunge at the beginning of the cut and retract straight up out of the shape at the end of the cut.



Center-Line Cut tool paths

© 2011 CIM-TECH.COM, Inc



Center-Line Cut parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset. In this instance, there should be NO offset so 0 is set by default.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (0) is a setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed unless you want to force an offset for the tool path.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a

© 2011 CIM-TECH.COM, Inc

negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

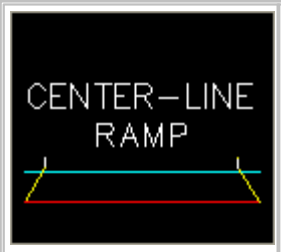
Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

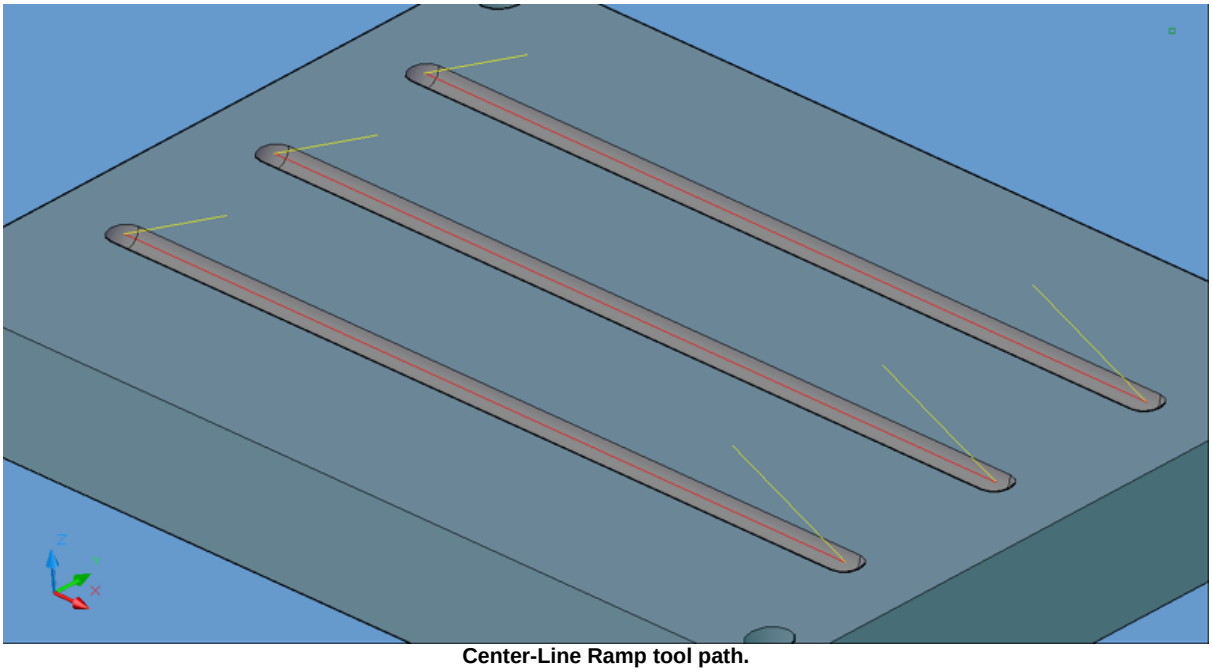
6.1.4.3 Center-Line Ramp



These cycles allow you to create tool paths directly from any defined shapes. Use this cycle to make a fixture or to engrave geometry. This cycle is also used to cut tool-width slots. You can draw the gasket grooves and vacuum grooves for a spoil board, and use this cycle to follow that geometry. This cycle would normally be used with a tool that does not use Machine Cutter Compensation, since there is no offset created.

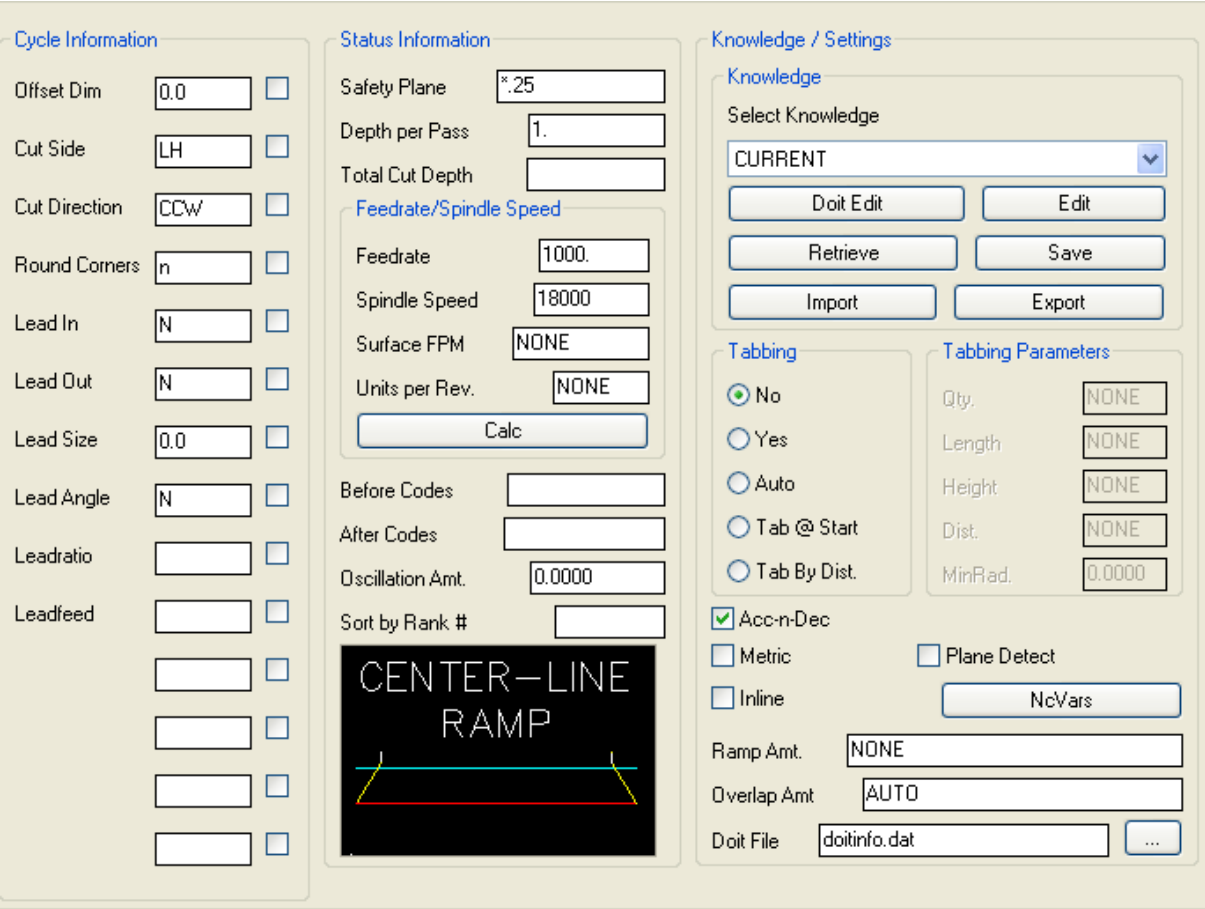
Cycle Start and End positions determine where to rapid to and where to feed to in the Z axis. Tool Depth of Cut affects the number of cutting passes. You must manually offset the defined geometry to determine the centerline of the tool if you turn on cutter compensation.

Center-Line Ramp will make a ramp in reverse direction to the cut (so it does not violate the shape) at the beginning and end of the shape. This cycle can be used on open or closed geometry.



Center-Line Ramp tool path.

© 2011 CIM-TECH.COM, Inc



Center-Line Ramp parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset. In this instance, there should be NO offset so 0 is set by default.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (0) is a setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed unless you want to force an offset for the tool path.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside

© 2011 CIM-TECH.COM, Inc

(Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

© 2011 CIM-TECH.COM, Inc

See the Lead Angle section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you

© 2011 CIM-TECH.COM, Inc

give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

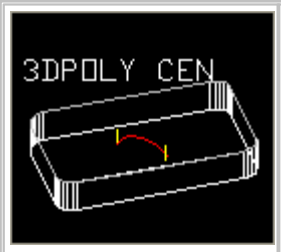
Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings**

© 2011 CIM-TECH.COM, Inc

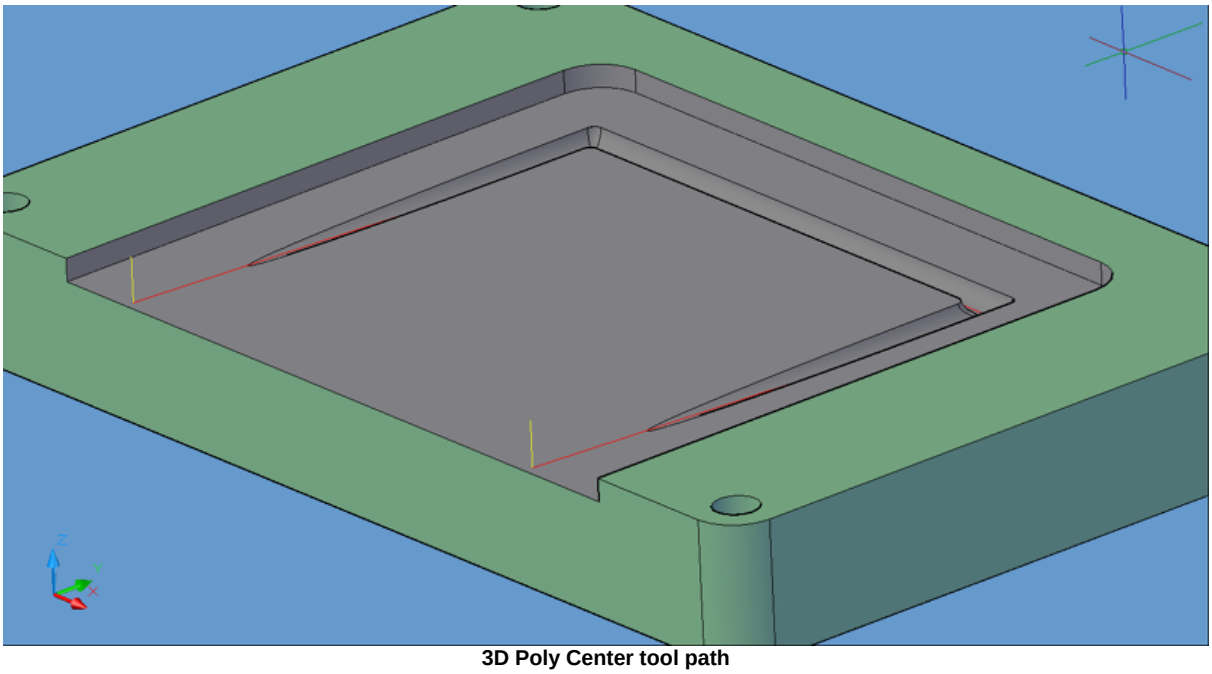
6.1.4.4 3D Poly Center



3D Poly Center will follow a 3D Polyline drawn and geo-shaped. The tool path will start at the Safety Plane, plunge to the start of the cut and then follow the 3D Polyline to the end and finish by retracting straight up to the Safety Plane.

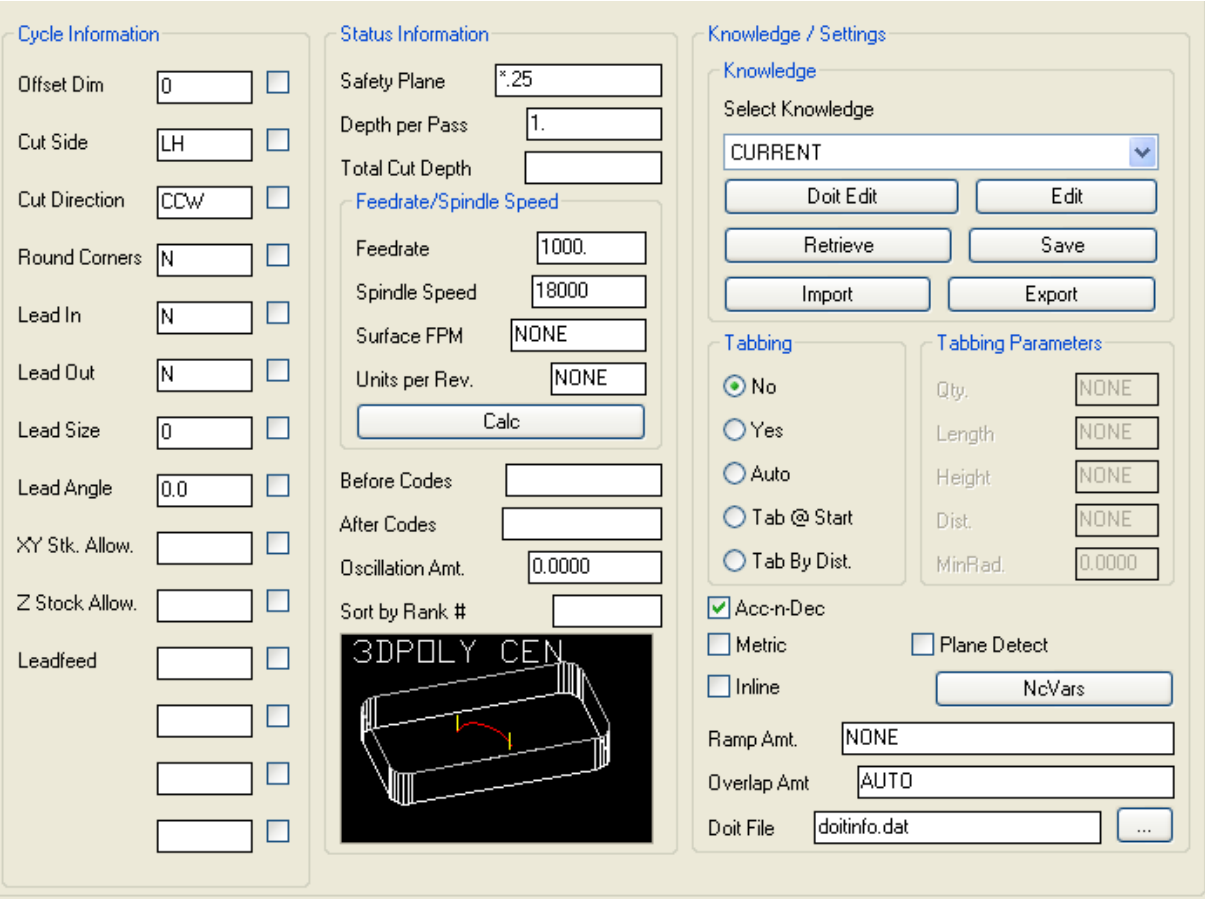
This is a center cutting cycle and no tool radius compensation should be used.

This cycle is used to turn any 3D polyline into a tool path. Draw the 3D polyline on layer NC_SHAPE (or Geoshape a 3D polyline), ensure Cutter Comp is set to No, and click Cut. This cycle will add vertical lead-in and lead-out at the start and end points on the polyline. This cycle is used for cutting sloped slots or varying depth engraving cuts.



3D Poly Center tool path

© 2011 CIM-TECH.COM, Inc



3D Poly Center parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

© 2011 CIM-TECH.COM, Inc

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance

© 2011 CIM-TECH.COM, Inc

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

© 2011 CIM-TECH.COM, Inc

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

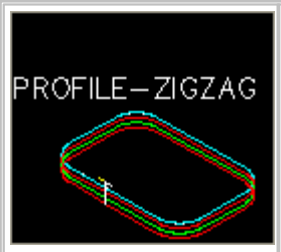
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

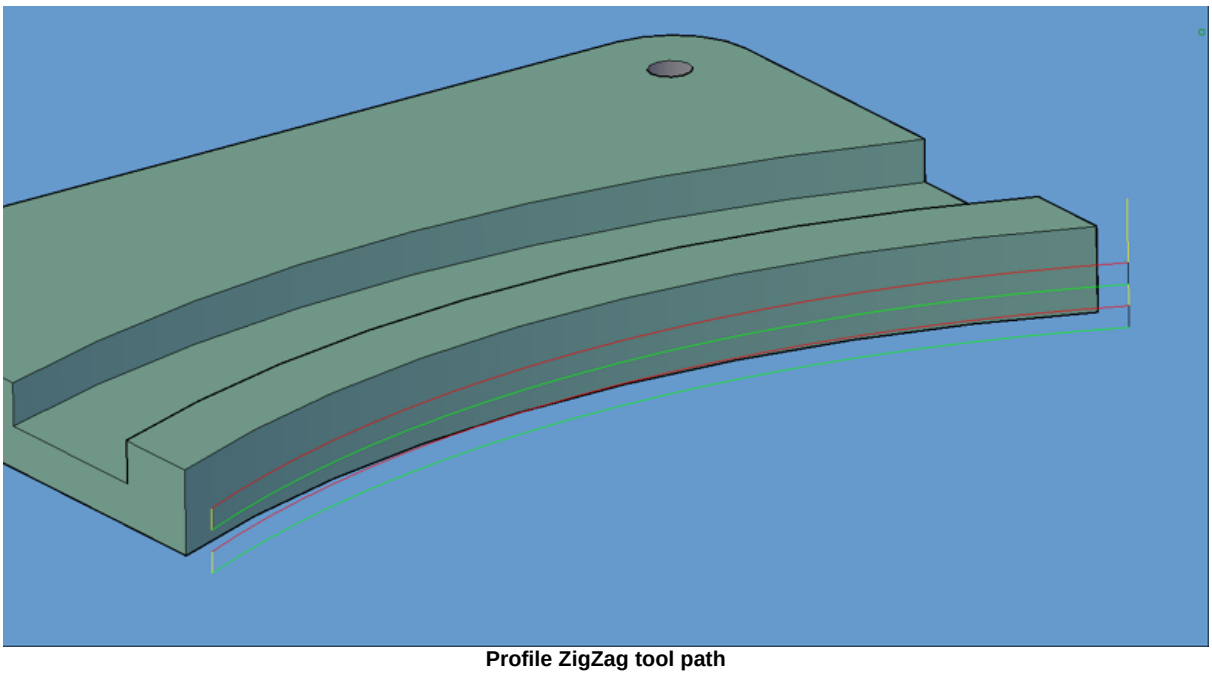
© 2011 CIM-TECH.COM, Inc

6.1.4.5 Profile ZigZag



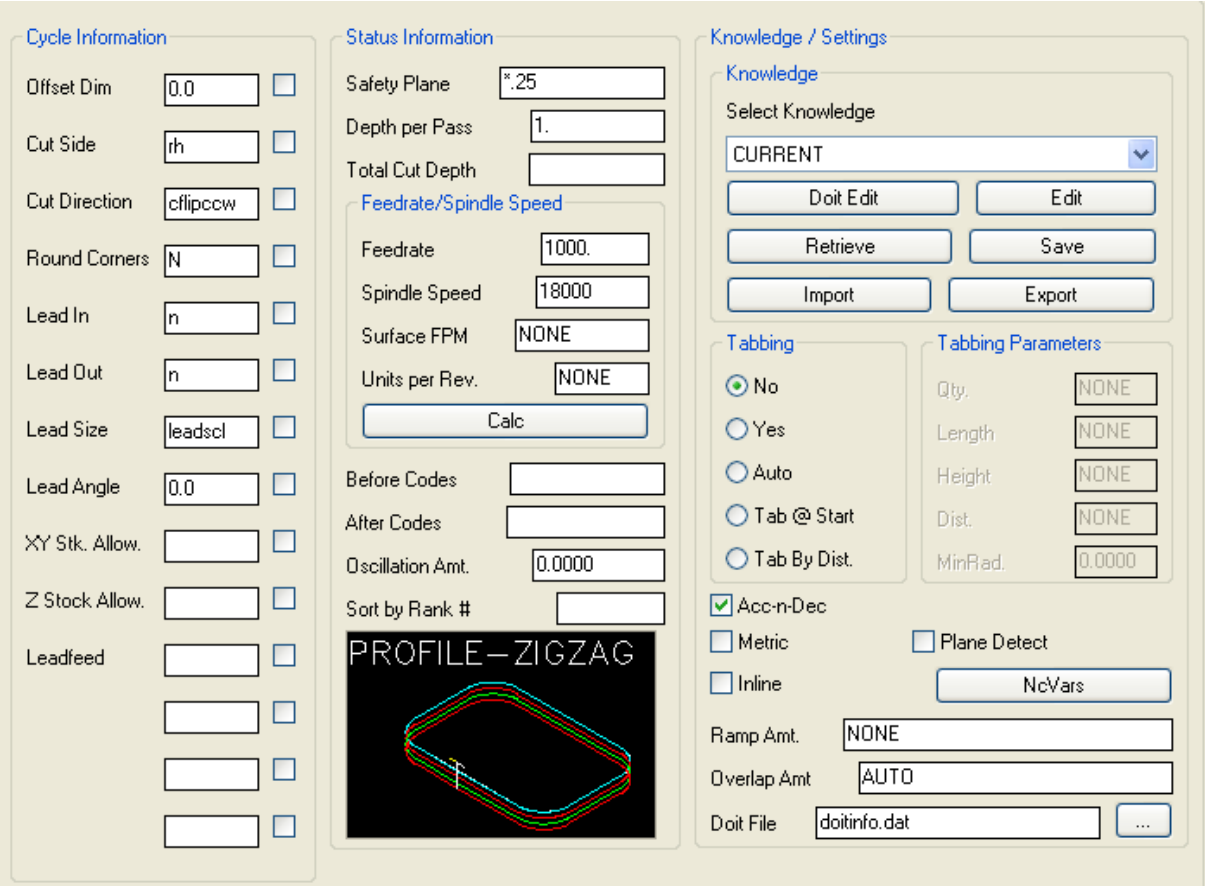
Profile Zig-Zag is a cycle best suited for cutting open shapes. The tool will start at the Safety Plane, plunge to the first cut depth, then follow the shape to the other end, move down in Z to the next depth, and then cut the shape back to the start, repeating the actions until it has reached the Total Cut Depth. Since it is a center cutting cycle, there is no offset by default, so the geometry should be offset or placed in the proper position for the tool. It is possible to have an offset of the tool path, by changing some cycle parameters, described below.

Typically, if a cycle is needed to follow a geoshape and cut multiple depths per pass without the tool lifting at the end of each pass and moving back to the start, then Profile ZigZag is well suited for this cut.



Profile ZigZag tool path

© 2011 CIM-TECH.COM, Inc



Profile ZigZag parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part

© 2011 CIM-TECH.COM, Inc

that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for

© 2011 CIM-TECH.COM, Inc

example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

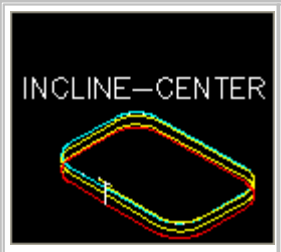
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

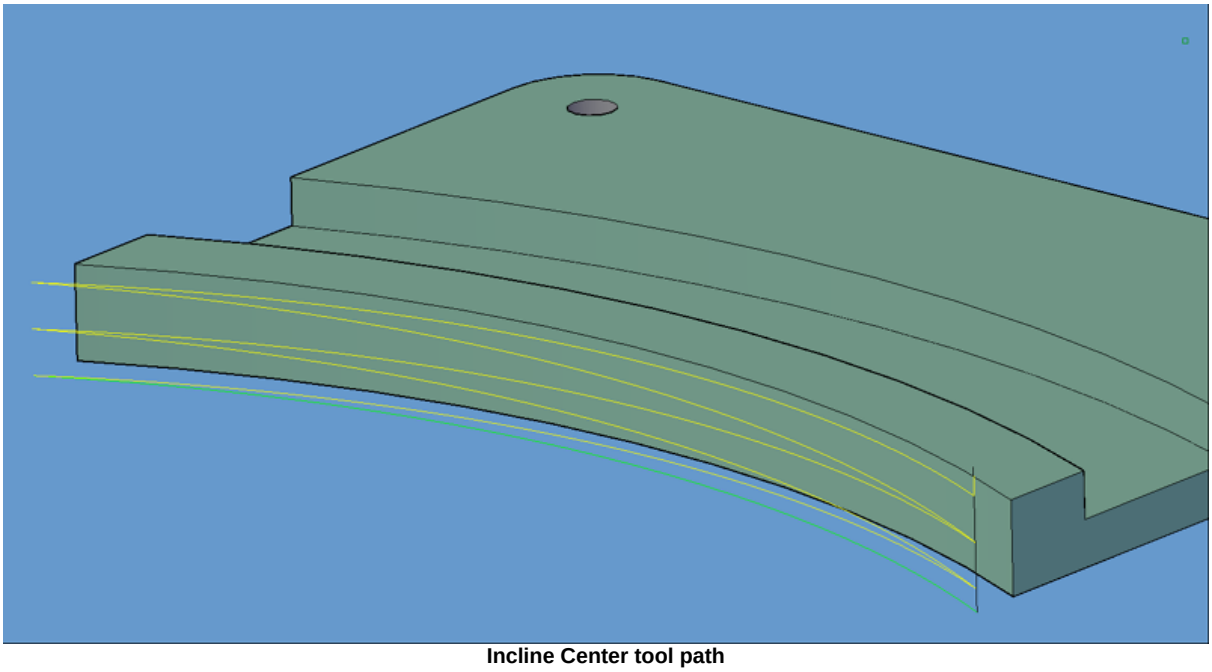
© 2011 CIM-TECH.COM, Inc

6.1.4.6 Incline-Center



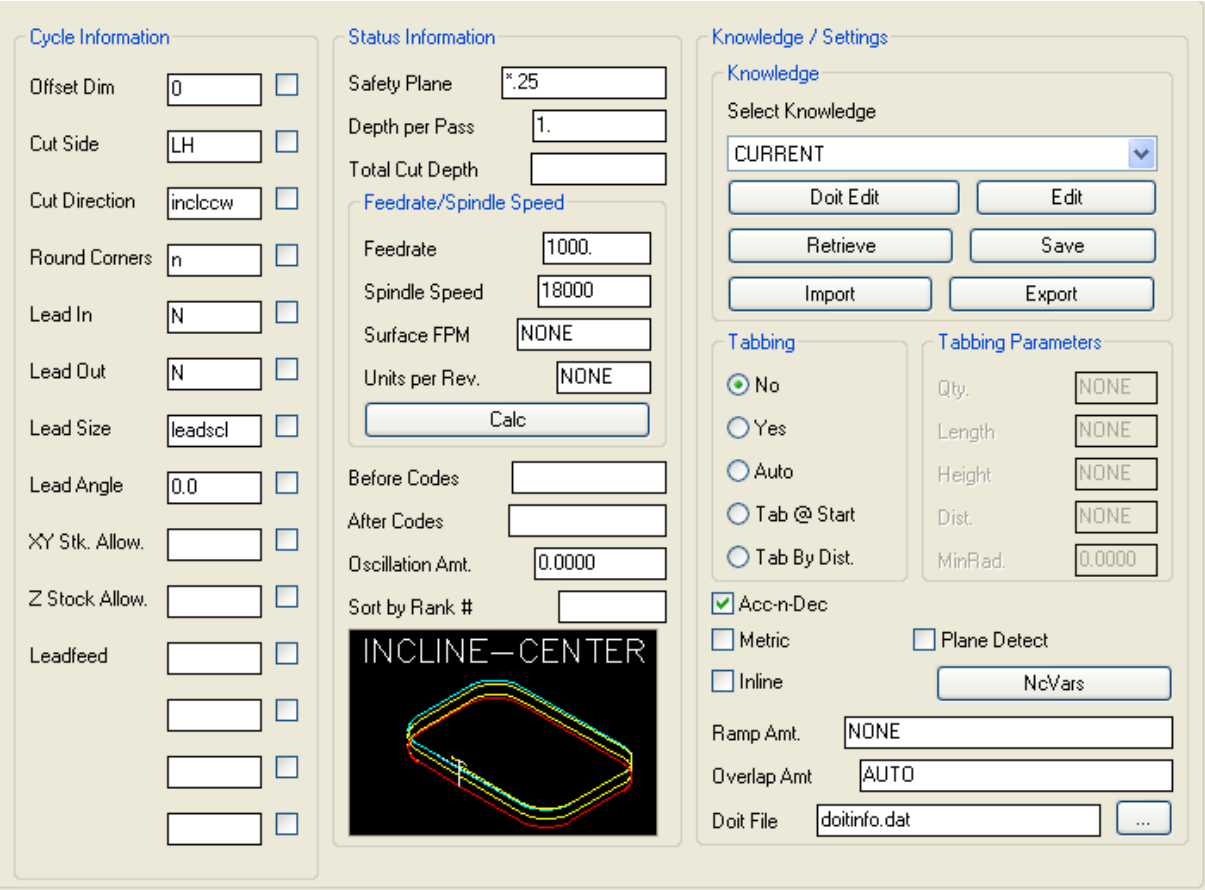
Incline cutting will make a constant ramping motion as the tool path moves around the profile. At the bottom of the cut there is a finish pass to remove the wedge of material left by the ramping motion. This type of cutting is useful when you have a material and tool that need a constant load or chip during the cut. Since the tool is continuously ramping, the tool load is never released, or increased during the entire cut, until the finish pass at the bottom.
The amount of material removed by the cutter, and thus the number of passes in Z, are controlled by the Total Cut Depth and Depth per Pass parameters. The Total Cut Depth is the depth of the cut overall and the Depth per Pass parameter controls how deep each pass is in Z, which controls the chip load.

Incline-Center will default to cutting on the centerline of an open or closed shape.



Incline Center tool path

© 2011 CIM-TECH.COM, Inc



Incline Center parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For

© 2011 CIM-TECH.COM, Inc

instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a

© 2011 CIM-TECH.COM, Inc

negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

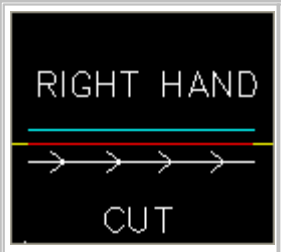
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

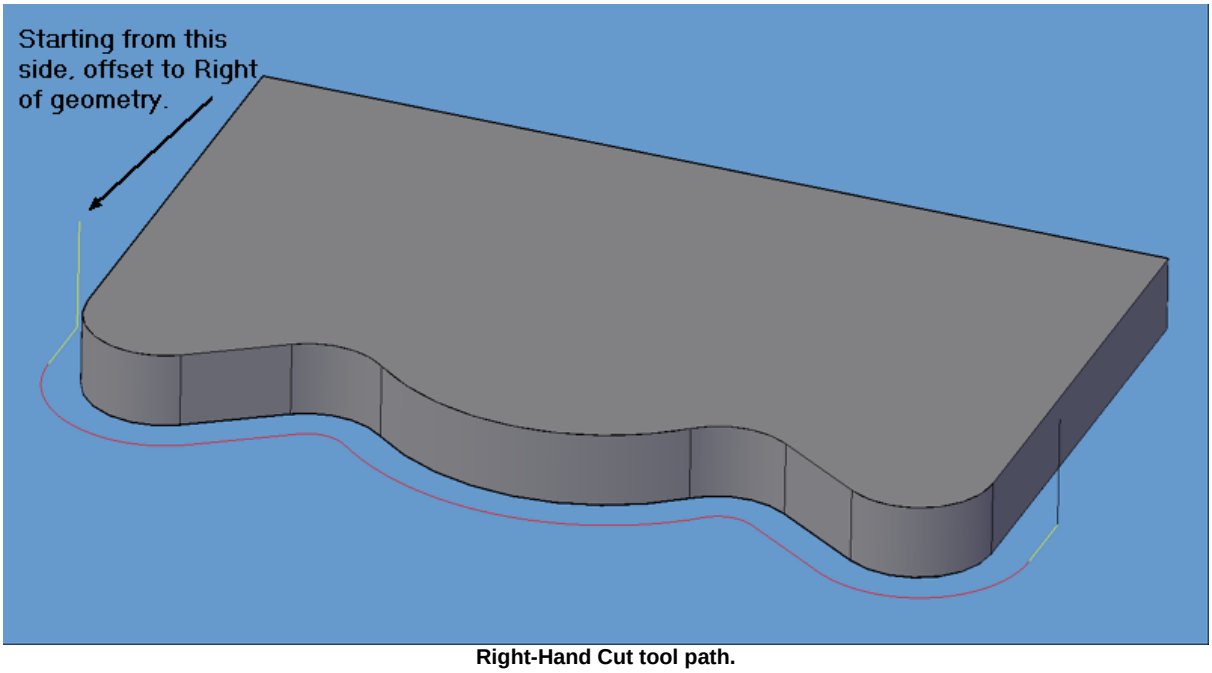
© 2011 CIM-TECH.COM, Inc

6.1.4.7 Right Hand Cut



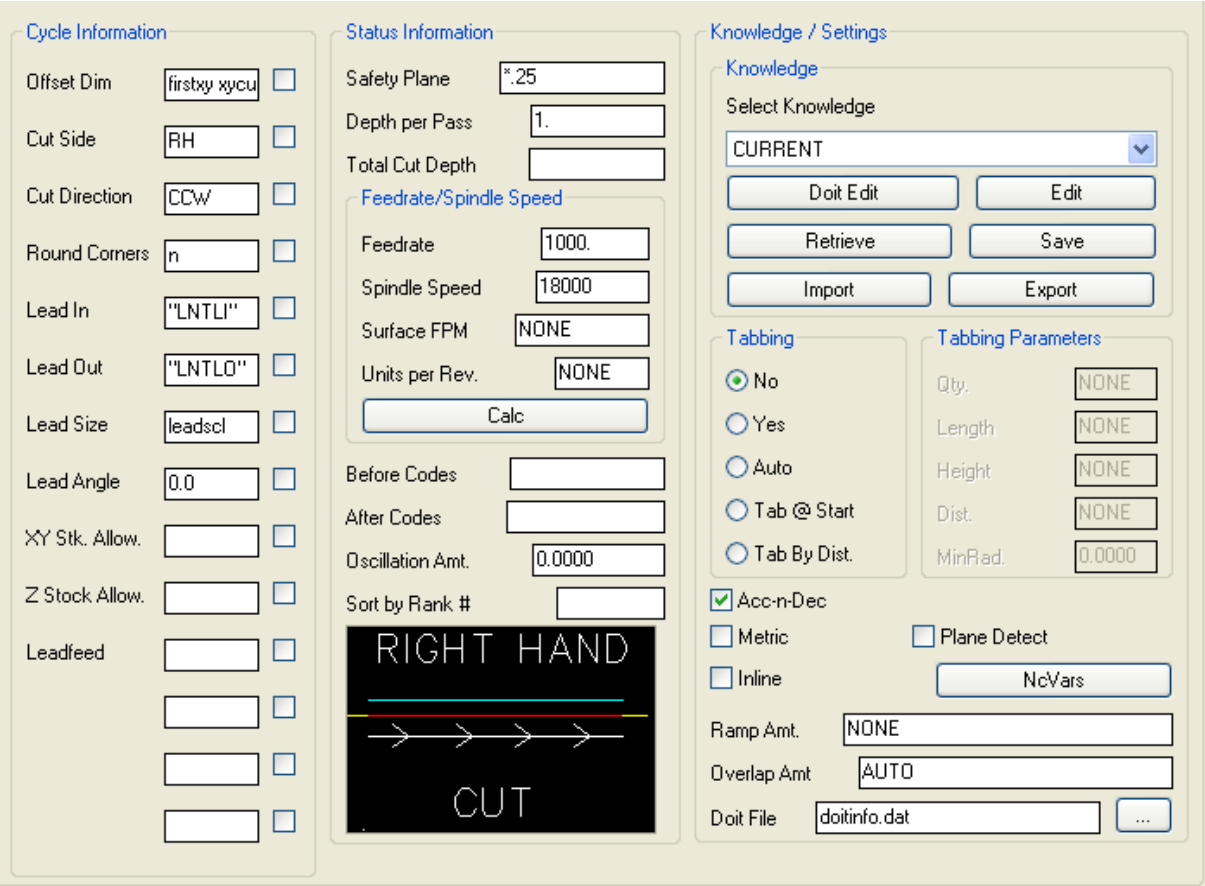
Right Hand Cut is an open shape cut cycle. The cycle will start at the Safety Plane, plunge to the Total Cut Depth and then follow the shape to the end, lead straight out and then retract back to the Safety Plane.
The offset by default is to the right side of the shape, starting from the start point location.

This cycle is only useful in interactive Router-CIM. Automation uses a different method of determining the right from the left side of a shape during geoshape.



Right-Hand Cut tool path.

© 2011 CIM-TECH.COM, Inc



Right-Hand Cut parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners
If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In
This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out
This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle
Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance
Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part

© 2011 CIM-TECH.COM, Inc

that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for

© 2011 CIM-TECH.COM, Inc

example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

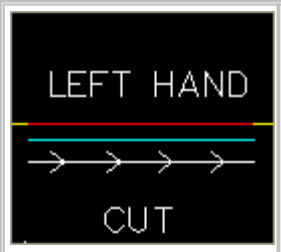
Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

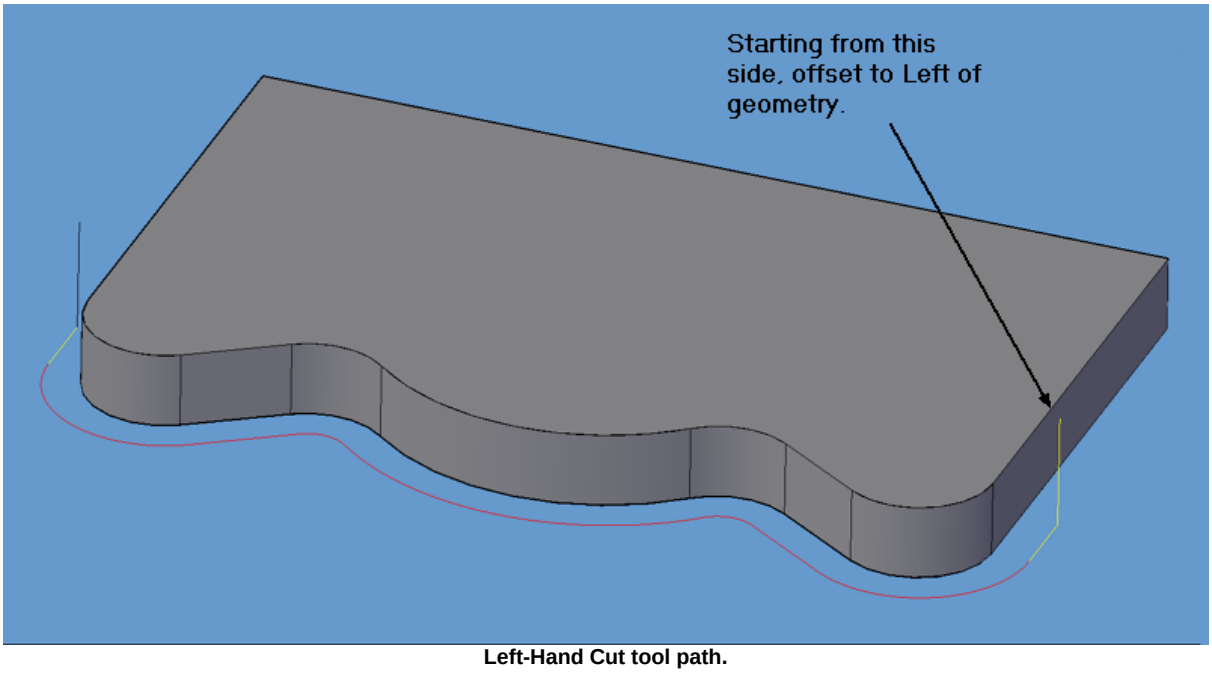
© 2011 CIM-TECH.COM, Inc

6.1.4.8 Left Hand Cut



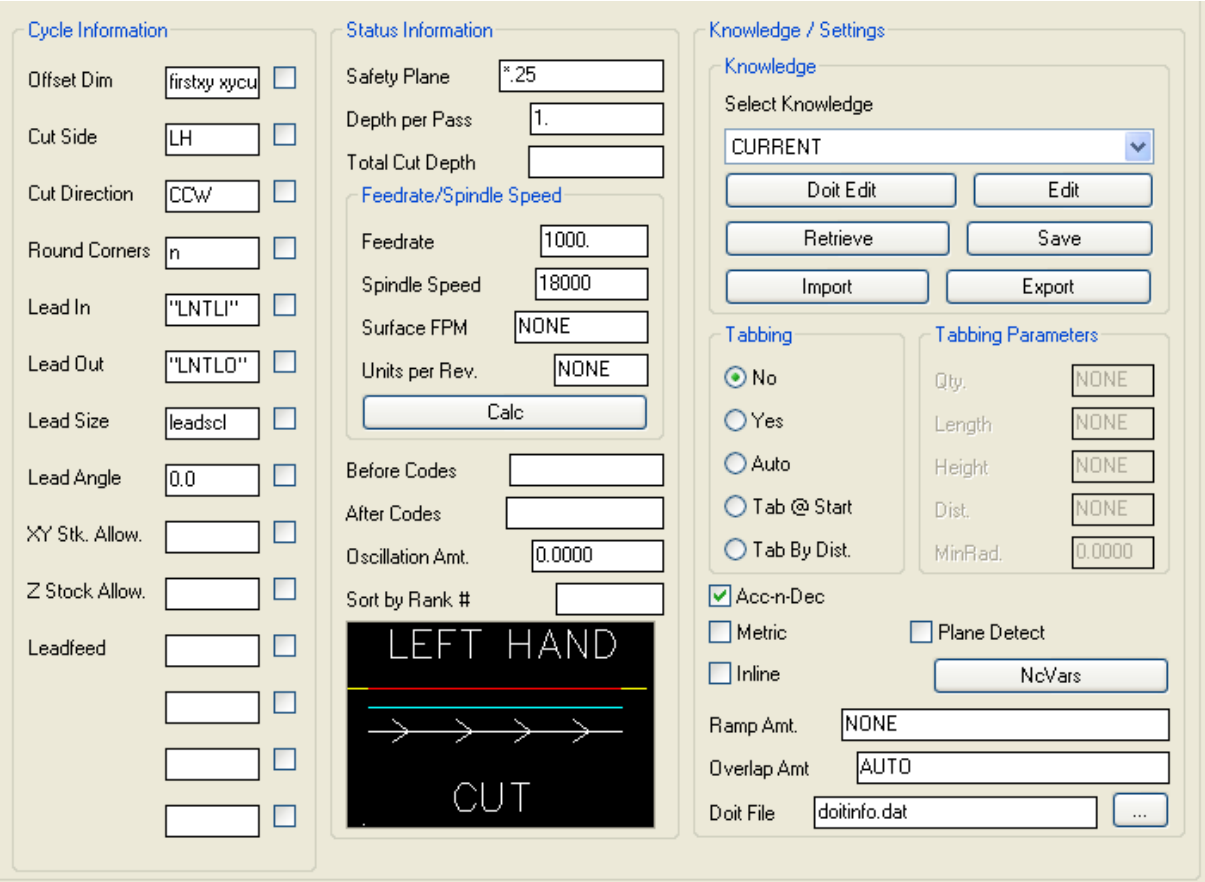
Left Hand Cut is an open shape cut cycle. The cycle will start at the Safety Plane, plunge to the Total Cut Depth and then follow the shape to the end, lead straight out and then retract back to the Safety Plane.
The offset by default is to the left side of the shape, starting from the start point location.

This cycle is only useful in interactive Router-CIM. Automation uses a different method of determining the right from the left side of a shape during geoshape.



Left-Hand Cut tool path.

© 2011 CIM-TECH.COM, Inc



Left-Hand Cut parameters.

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See the Offset Dim section for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

© 2011 CIM-TECH.COM, Inc

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

XY Stock Allowance

Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part

© 2011 CIM-TECH.COM, Inc

that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

See XY Stock Allowance for more information.

Z Stock Allowance

Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

See Z Stock Allowance for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out. Whatever number you set this variable to is a percentage of max feedrate set in the Control Panel. Setting the number to a value greater than 1.0 will give you an exact feedrate.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

See the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for

© 2011 CIM-TECH.COM, Inc

example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

Overlap Amt

Overlap is the movement of the cutter past the starting point of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0". This is typically done to reduce any witness mark in the material left by the tool on the lead-in maneuver.

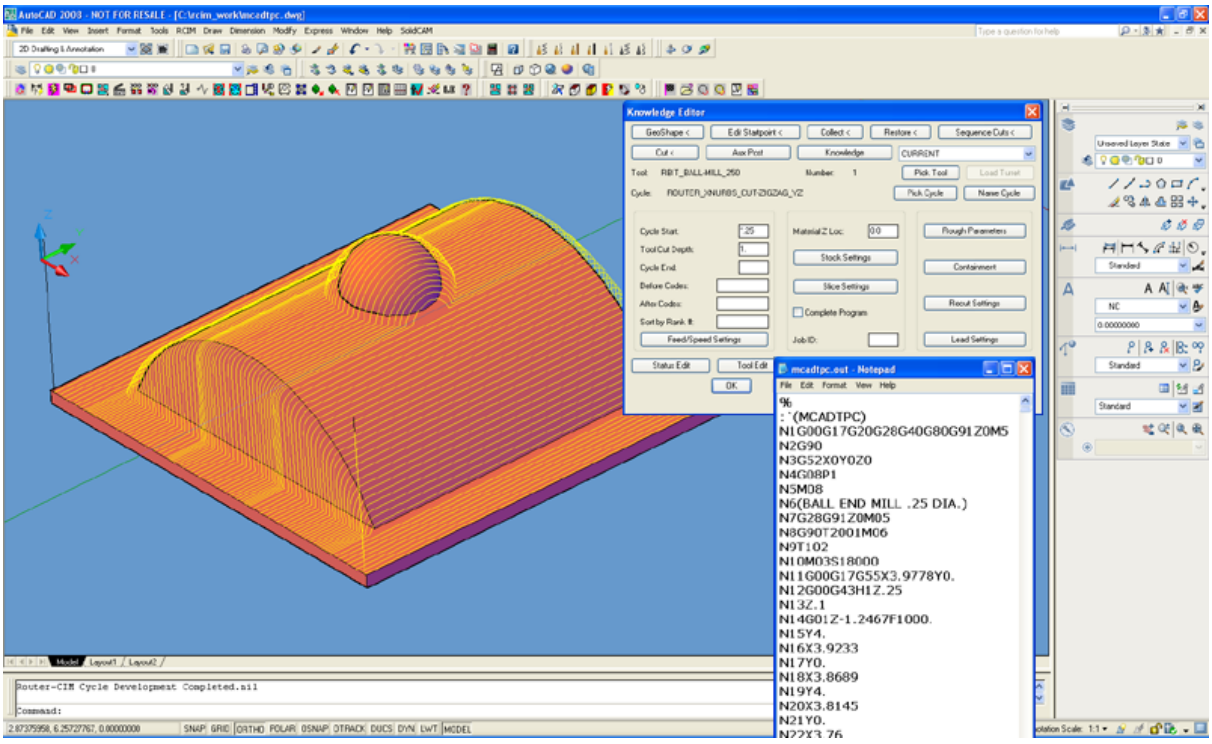
See the Overlap Amt section for more information.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.5 NURBS Surface Machining Cycles

The Expert NURBS Cutter Extension was developed to service the complex 3D machining needs of Router users. The 3D Toolpath Calculator is seamlessly integrated with Router-CIM. Toolpaths can be created from 3D solids or surfaces.
Toolpath results from the Expert NURBS Cutter Extension are displayed in either the AutoCAD graphic windows or the NCSurfer module. NCSurfer is a Windows Open GL Application. NCSurfer provides maximum performance. When using NCSurfer, NC code is produced at the same time toolpath is being calculated and displayed.



The graphic examples used in this section of the manual were captured from AutoCAD 2008.
Toolpath displayed in the NCSurfer Open GL Window will look similar.
Toolpath results displayed in NCSurfer look similar to the figure shown.

Overview:

Router-CIM communicates with the toolpath calculator, or the Expert NURBS Cutter, by using files external to the drawing. You may find some, or all, of the following files being created when using the Expert NURBS Cutter.
"Filename.GEO" contains the NURBS description.
"Filename.PAR" contains parameter controls.
"Filename.O" contains toolpath data that is read by NC Polaris when toolpath is being returned to the drawing.
"Filename.JOB" contains instructional data when toolpath is calculated and displayed using NCSurfer.
"Filename.OB" contains data that enables immediate viewing of toolpath when using NCSurfer.

When multiple machining steps are required for the same group of surfaces or solid (i.e., roughing and finishing), you would run "Cut" multiple times. After running "Cut" the first time, the system recognizes that the <B> ".GEO" file already exists for the drawing name and during subsequent "Cut" requests you are prompted, "Define Surfaces YES/NO". If the operation is intended to address the identical surfaces (or solid), using identical surface tolerances and complimenting entities, you can respond

© 2011 CIM-TECH.COM, Inc

"NO", to the prompt. If a different combination of surfaces (or solid), surface(s) tolerances, or different complimenting entities are required, you must respond "YES", and select the new combination of surfaces (or solid) and complimenting entities for the next operation. Defining Surfaces is a relatively fast function, and it is always safer to respond "YES", and select the surfaces again.
As a general housekeeping rule, after completing an entire job, you can delete the external files to avoid unnecessary buildup of files on the hard disk drive in your computer.

Surface Construction Rules:
A surface must not contain C0 continuity. If unusual toolpath results occur on a given surface, use AMD's Surface Editing Commands to break at C0 to insure proper performance from the Expert NURBS Cutter Extension.
Occasionally IGES translators will create surface or solid anomalies. Be prepared to use AMD editing features to repair these surface or solid anomalies when unusual toolpath results.
Surfaces created from the 3D Solid should be truncated to insure proper operation of the Expert NURBS Cutter Extension. Toolpath can be produced directly from the 3D Solid.
A Cavity (female part) must contain a horizontal plane(s) that represents the top of the Cavity or Parting Plane. This Plane(s) can be very narrow to avoid unnecessary toolpath motions. Without the Parting Plane(s) the software can not establish an approach to vertical surfaces, or surface areas that approach near vertical at termination in a Cavity, and results can be unpredictable. A narrow horizontal plane at the top of Cavity wall gives the software an established location to start the toolpath.

6.1.5.1 NURBS Cycles Supplied

The Expert NURBS Cutter Extension can be operated by using one of the Cycles in the Cycle List containing the word "XNURBS". The appropriate Cycle, used in conjunction with entries in the Knowledge Editor GUI, will generally provide an intuitive method for entering machining parameters. All five XNURBS Cycles share the same parameters. The differing default parameter entries cause them to perform differently.

The five Cycles supplied are:

NURBS CUT-ZIGZAG XZ
Default settings will produce vertical toolpath at a constant step over equal to tool radius in the XZ plane (angle 0).

NURBS CUT-ZIGZAG YZ
Default settings will produce vertical toolpath at a constant step over equal to tool radius in the YZ plane (angle 90).

NURBS CUT-CCW XY
Default settings will produce horizontal climb milling toolpath at a constant step over equal to tool radius in the XY plane.

NURBS CUT-FLOWCUT
Default settings will produce vertical flowcut toolpath at a constant step over equal to tool radius in the XZ plane (angle 0) or YZ plane (angle 90).

NURBS CUT-PROJECT
Default settings will produce vertical toolpath that follows the direction of the projection polyline.

This section of the manual is supported by a series of graphics, showing tool path results in the drawing. Tool paths displayed using NCSurfer will appear similar to these graphics. Each of the NURBS Cycles are used. A variety of entries in the Knowledge Editor will give different toolpath results. Drawings MCADTPC.DWG, and MCADFLOW.DWG, will be used to describe the capabilities of the NURBS Cutter. The toolpath results shown in all examples were created by using a.25

© 2011 CIM-TECH.COM, Inc

Diameter Ball End Mill.

The NURBS Cutter supports machining with Ball, Bull or Flat End Mill Tools, on solids or surfaces. Once you have familiarized yourself with the operation of the NURBS Cutter using the .25 diameter Ball End Mill Tool, you are free to experiment using Bull or Flat End Mill Tools.

Drawings used for the examples will be made available via hyperlinks so that you may download the completed drawings if desired.

6.1.5.2 NURBS Cycle Parameters

The Expert NURBS Cutter is rich in 3D machining features. The Mill Cycle List has been expanded to include a special parametric table to accommodate the Expert NURBS Cutter. The Expert NURBS Cutter parametric table is described as follows. Default parameter entries are shown for the MILL_XNURBS_CUT-ZIGZAG_XZ Cycle. Graphic toolpath examples using each of the XNURBS Cycles with different Knowledge Editor entries are shown in the EXAMPLES section.
Your focus when operating the Expert NURBS Cutter will be entering values in the Knowledge Editor. The following list of Cycle Parameters is for reference when altering parameter defaults, or making additional Cycles.

[01] Enter Table Name..... mltpc
Table name. Can not be edited.

[02] Enter Item Name..... XNURBS_CUT-ZIGZAG_XZ
Cycle name. Editing name enables making of new Cycles.

[03] Cycle Type.....(MLTPC) MLTPC SCUT2
Defines the type of cutting cycle and should not be edited.

[04] .....
Reserved, do not edit.

[05] Vert/Horz/Flow/Proj Cut V
Enter V for Vertical cutting, H for Horizontal cutting, F for Flow cutting, and P for polyline projection cutting.

[06] Rough Z Cut Increment (#)
An entry in this parameter automatically enables roughing. The value entered controls the maximum cutting depth for each roughing pass.

[07] Roughing Type (1/2/3)
A total of 6 roughing methods are available with the Expert NURBS Cutter. They are divided into two categories, Vertical and Horizontal.
Vertical Cutting + Mode 1 = Offsets each roughing pass by [06] "Rough Z Cut Increment" + [22] "Stock Amount to Leave", moving in rapid when machine motions exceed [19] "Working Area Z Maximum".
Vertical Cutting + Mode 2 = Offsets each roughing pass by [06] "Rough Z Cut Increment" + [22] "Stock Amount to Leave", and proportionally distributes Z depth throughout the roughing activity without allowing tool to leave the stock material.
Vertical Cutting + Mode 3 = Shifts roughing motions in "Z" based on [06] "Rough Z Cut Increment" while compensating for [22] "Stock Amount to Leave", moving in rapid when machine motions exceed [19] "Working Area Z Maximum".
Horizontal Cutting + Mode 1 = Plunges straight down to each Z level, compensates for [22] "Stock Amount to Leave", roughs without following collective surface contours before proceeding to next lower Z level.
Horizontal Cutting + Mode 2 = Ramps down to each Z level, compensates for [22] "Stock Amount to

© 2011 CIM-TECH.COM, Inc

Leave", roughs and makes a contour following pass before ramping down to next Z level.
Horizontal Cutting + Mode 3 = Plunges straight down to each Z level, compensates for [22] "Stock Amount to Leave", roughs and makes a contour following pass before plunging straight down to next Z level.

[08] Surface Intersect (Y/N)

Y enables intersect cutting. Intersect cutting can be applied to all surface intersections in a drawing, or applied only to the intersections of selected sets of surfaces. If the surface drawing has already been defined to the system by previously making toolpath (a.GEO already exists), respond YES to the "Define Surfaces" prompt. You are then prompted "Group Surfaces into Sets?".
1. If your objective is to apply surface intersect toolpath on all surface intersections in the drawing, you should respond NO, and select all surfaces in the drawing when prompted to select objects.
2. If your objective is to apply surface intersect toolpath on intersections of selective sets of surfaces, you should respond YES. You are then given the opportunity to "Select Surfaces in First Set" and "Select Surfaces in Second Set". The "First Set" will disappear from the screen when you have completed the selection process for user feedback.
3. Upon completion of selecting objects you are prompted to, "Specify Minimum Angle that Defines Intersection Deg:". Respond according to your requirements.

[09] CrossCut.....(Y/N)

Y enables CrossCut. Cross cutting only applies to vertical finishing motions. Cutting motions are produced on any surface(s) area(s) that meet the angle criteria, in the direction of the cutting motions. You are prompted during "Cut" for surface(s) angle criteria.

[10] Recut.....(Y/N)

Y enables Recut. Recut, sometimes referred to as rest cutting, removes material left by a prior operation or tool. In Vertical cutting mode (entry V in [05]), the system will prompt for information regarding previous tool used. Toolpath is then produced on all surface areas not cut by the prior tool. In Horizontal cutting mode (entry H in [05]), no prompt occurs regarding previous tool, and vertical toolpath is produced on all surface areas that are nearly horizontal where XY motions could not be generated.

[11] Define Contain Surf(Y/N)

Y enables Surface Containment. If the surface drawing has already been defined to the system by previously making toolpath (a .GEO file exists), respond YES, to the "Define Surfaces" prompt. You are prompted during "Cut", to select surfaces to cut, followed by selection of surfaces to use for toolpath containment. This feature is not available when cutting directly from a solid.

[12] Def. Contain String(Y/N)

Y enables String Containment. If the surface drawing has already been defined to the system by previously making toolpath (a .GEO file exists), respond YES, to the "Define Surfaces" prompt. You are prompted during "Cut" to select surfaces to cut and to select 2-D polyline(s) to use to contain toolpath. The XY boundary of a single closed polyline will contain the toolpath limits.

[13] Define Plunge Pts.(Y/N)

Y enables Plunge points. Plunge points apply only to roughing. In the roughing mode the system will make every attempt to plunge at the defined plunge locations. You are prompted during "Cut" to define plunge points.

[14] Working Area X Minimum.

[15] Working Area Y Minimum.

[16] Working Area Z Minimum.

© 2011 CIM-TECH.COM, Inc

[17] Working Area X Maximum.

[18] Working Area Y Maximum.

[19] Working Area Z Maximum.
Parameters [14] through [19] are used to define a working area (clip box). Normal system operation requires that parameter [19] "Working Area Z Maximum", contains a value equal to the value entered in parameter [21] "Material Z Loc(#)". Entry of a value different than the entry in [21] "Material Z Loc.(#)", is used only under special conditions.

[20] Description.....("TEXT") "X-Nurbs Cutting, Zigzag XZ Plane"
Text string used to describe Cycle.

[21] Material Z Plane Loc.(#)
Requires entry of where the top of the stock material is located. Cycle start position uses this entry to calculate feed distance to material. Normal system operation requires that parameter [19] "Working Area Z Maximum", contains a value equal to the value entered in parameter [21] "Material Z Loc(#)".

[22] Stock Amount to Leave... 0.0 Y
The first entry controls the amount of stock to leave. The second entry "Y" (without the quotes), indicates that you would like multiple stock allowances. You are prompted during CUT to select surface(s) for different stock allowance, and you are then prompted for a value.

[23] Slice Step Dir (POS/NEG) POS
Determines whether vertical cutting motions begin at X or Y zero and proceed positive, or begin at X or Y maximum and proceed negative.
IMPORTANT NOTE<R>The determination for CONVENTIONAL or CLIMB milling is automatically made by designating a POS or NEG answer to this parameter when using Horizontal finishing, as long as there is no entry in Parameter [29]CW Horz. Cuts....(Y/N). A POS answer results in CONVENTIONAL milling, a NEG answer results in CLIMB milling. An entry of Y or N in Parameter [29]CW Horz. Cuts....(Y/N), will override the automatic computation.

[24] Constant Step Over Dist. !*tr*
This entry defines the fixed distance between cuts. This entry is overridden when [25] "Define Scallop Size" is enabled with a Y entry.

[25] Define Scallop Size(Y/N)
Y enables Scallop Height Control for Vertical finishing. You are prompted during "Cut" for desired scallop height. Enter the desired scallop height at that time. You are then prompted for minimum step size. Minimum step size is a clamp to protect from excessive toolpath on unusually steep surface areas. A response of .001 for Inch, or .01 for Metric, is appropriate in most cases. A response too large will circumvent scallop height calculation.

[26] Vertical Cutting Angle. 0.0
Defines angle of Vertical toolpath. 0.0 = XZ, 90 = YZ. Any angle between 0 and 90 is valid. Negative angles are allowed. This parameter entry works in conjunction with [23] "Slice Step Dir (POS/NEG).

[27] Lace Cutting Paths (Y/N) Y
Y enables lace or zigzag cutting motions in Vertical and Horizontal cutting. Enter N to achieve uni-directional cuts in Vertical cutting. Enter N to achieve continuous direction cutting when using Horizontal cutting.

[28] Fall Over Mode (0/1/2/3)
Fallover Mode settings accommodate special Vertical finishing situations. The system default is Fallover Mode 3. In Fallover Mode 3, vertical finishing toolpath ends at the extents of the selected

© 2011 CIM-TECH.COM, Inc

surfaces. Fallover Mode 1 and 2 cause toolpath to roll over the extents of the selected surfaces by tool radius. Mode 2 stops at tool radius, Mode 1 continues down in Z to "Working Area Z Minimum". Fallover Mode 0 causes toolpath to roll over as in mode 1, but also move away from the surface by a distance equal to tool radius. Fallover Mode settings are managed by the system for normal operation.

[29] CW Horz. Cuts.....(Y/N)
This parameter applies only to Horizontal finishing. The default is no entry. No entry allows the entry in [23]Slice Step direction to automatically control CLIMB or CONVENTIONAL cutting. N forces counter clock wise cutting. Y forces clock wise cutting direction. CCW toolpath is RED and CW toolpath is GREEN.
Lace cutting set to Y will cause zigzag (back and forth) horizontal finishing motions. This is particularly useful when cutting single or open multiple surfaces. Lace cutting should be set to N for normal multiple surface finishing.

[30] Lead-In Name ("NAME"/N)

[31] Lead-Out Name ("NAME"/N)

[32] Size of Leads.....(#)

[33] Angle of Leads.....(#/N)

[34] Task @ Lead-In (name/N) TPC-FEED

[35] Task @ Cut Start (name/N) TPC-CUT

[36] Task @ Cut End (name/N)

[37] Task @ Lead-Out (name/N)

[38] Run Tasks in Slices(Y/N) Y
Parameters [30] through [38] follow normal Router-CIM convention.

[39] .....

[40] (reserved).....
Reserved

[41] Task @ Rough Str(name/N)

[42] Task @ Rough End(name/N)
Parameters [41] and [42] follow normal Router-CIM convention

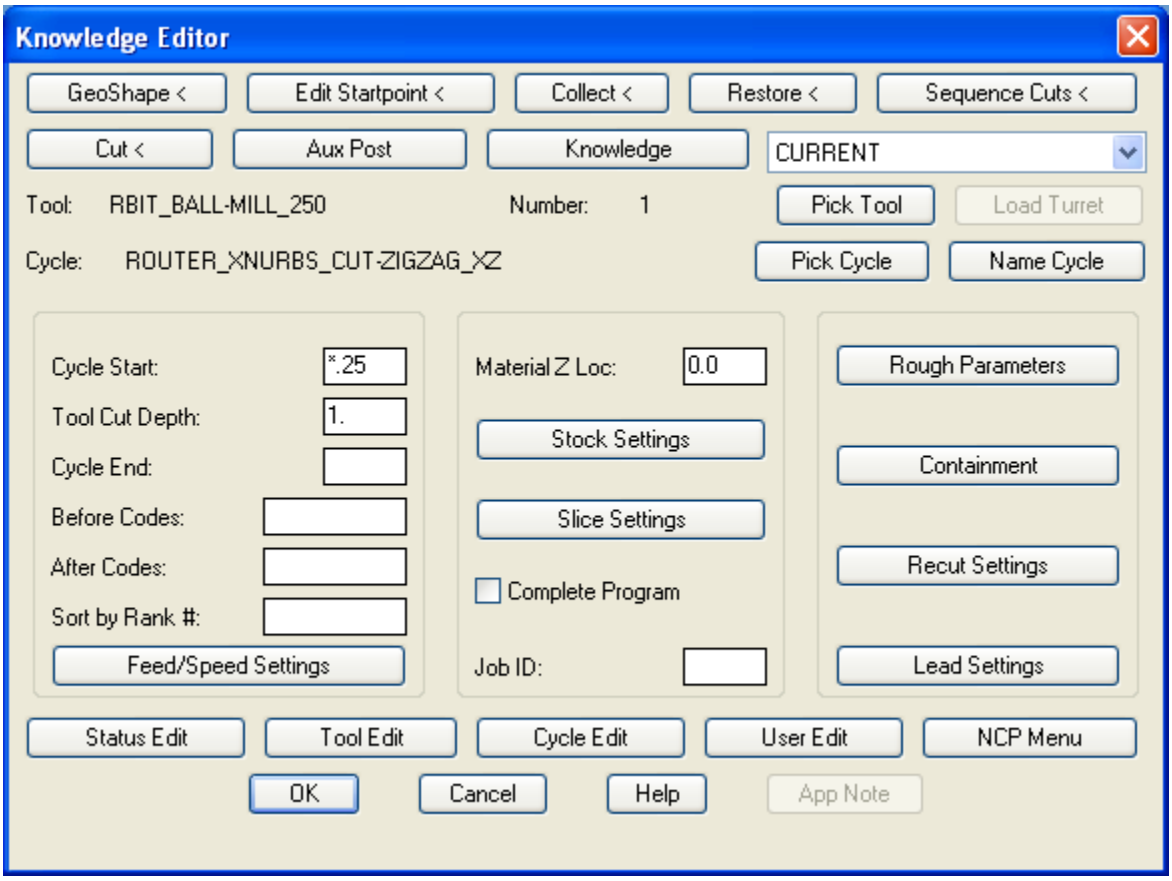
[43] Complete NC Program(Y/N) Y 1234
Y enables Complete NC Program which causes Router-CIM to go directly to NC Code after producing toolpath. The numeric value entered represents the Job Id.

[44] STR/END Tasks for NC Prg
This parameter allows the use of Start/End Codes when "[43]Complete NC Program(Y/N)" is enabled. Entry in this parameter to enable Start/End Codes is: PROCTSK1 PROCTSK2

© 2011 CIM-TECH.COM, Inc

6.1.5.3 Knowledge Editor in NURBS

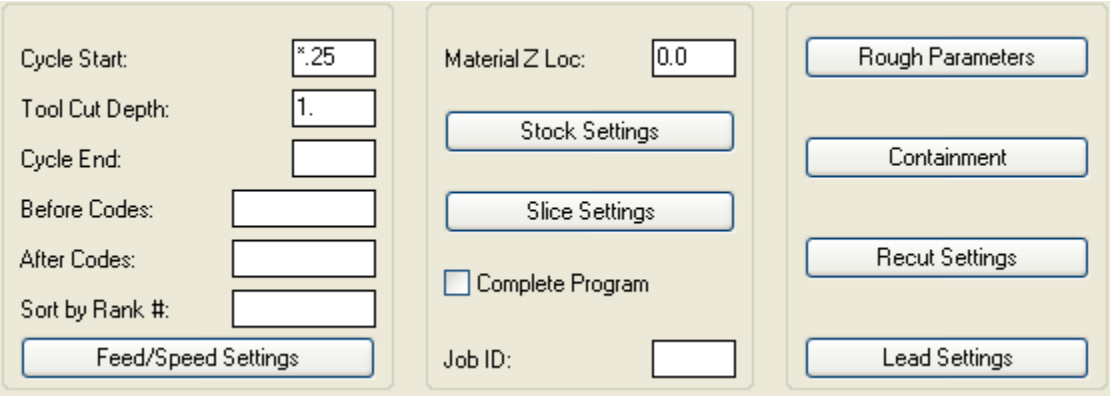
Knowledge Editor For NURBS



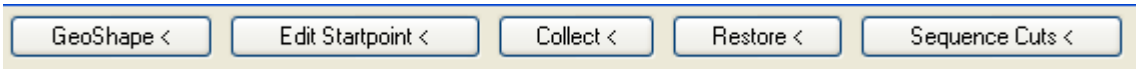
As you can see, the Control Panel changes into the Knowledge Editor when a NURBS cycle is selected. The NURBS configuration of the Knowledge Editor is customized to support the operation of the NURBS Cutter Cycle Editing. An explanation of the NURBS configuration of the Knowledge Editor follows.

NC code files for 3D machining are generally very large by comparison to NC code found in production machining. Because these files, as well as supporting files, tend to be so large, it is common to make a machining operation a separate NC code file. A "Check" in the Complete Program Toggle, will create separate NC code files for each operation. NC code files can be combined using the NCSurfer "Process" function.

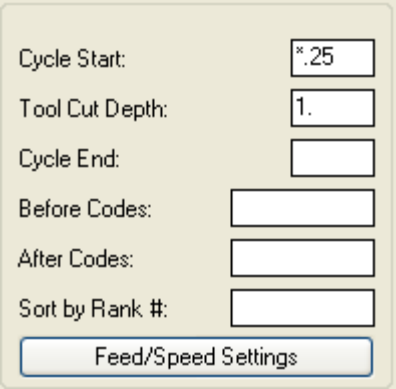
© 2011 CIM-TECH.COM, Inc



GeoShape and Edit Start Point can be used to control polyline direction and start point alignment when using Flow Cutting. Flow Cut Polylines must be going in the same direction. Closed Flow Cutting requires Start Point alignment. The Router-CIM command NCREVDIR will reverse the direction of Polylines. Flow line toolpath follows the direction of the Flow Polylines.

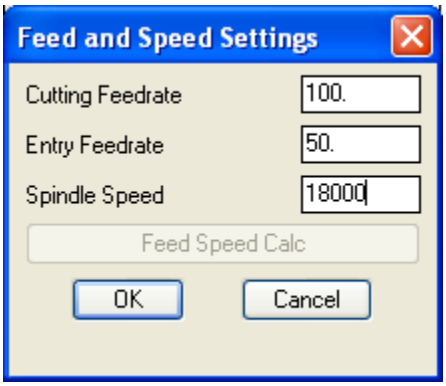


The Cycle Start entry defines the safe Z clearance plane. Tool Cut Depth is used as input for Project Cutting to limit cut depth per pass. Cycle End Position is ignored when using NURBS Cycles except when using Project Cutting. An entry in this parameter when using Project Cutting will control the depth of the final cutting depth.

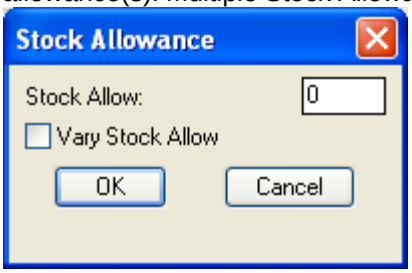


The Feed/Speed Settings opens an Edit Window for control of Cutting Feedrate, Plunging or Entry Feedrate, and Spindle Speed.
The Material Z Location entry indicates the location of the top of the stock material.

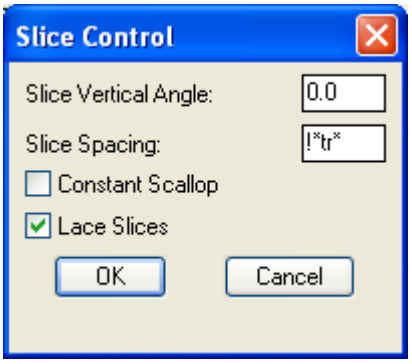
© 2011 CIM-TECH.COM, Inc



The Stock Settings Button opens an Edit Window for entry of Stock Allowance, sometimes referred to as finish allowance. A Check Box ON indicates that certain surfaces will require a different stock allowance(s). Multiple Stock Allowances are allowed.

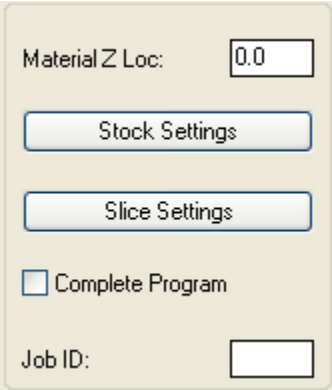


The Slice Settings Button opens an Edit Window for entry of Vertical Cutting Angle, Slice Spacing, Constant Scallop Toggle, and Lace or Zig-Zag Slices. If Constant Scallop is selected you are prompted for maximum scallop height during CUT. If Lace Slices is toggled on, toolpath will Zig-Zag. If toggled off, toolpath will be Unidirectional.

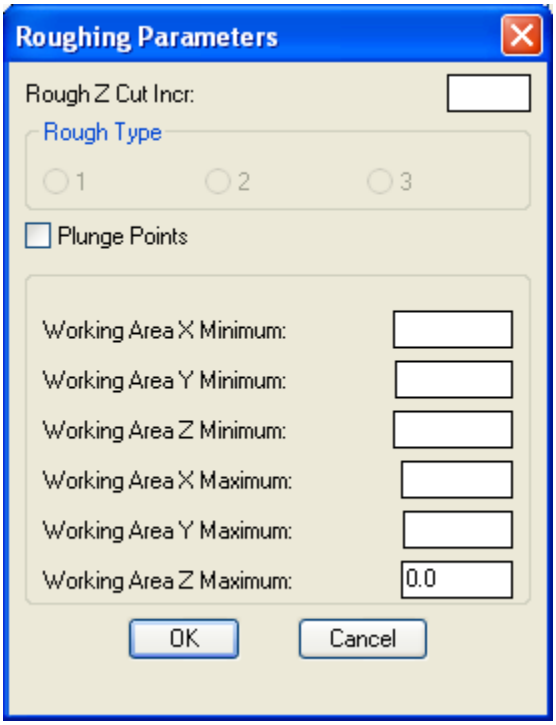


As stated earlier, a check in the Complete Program Toggle will cause the system to make NC code, including Start/End Codes. The Job Id entry will appear in the NC code.

© 2011 CIM-TECH.COM, Inc



The Rough Parameters Button opens an Edit Window for entry of roughing parameters. Rough Z Cut Increment controls the maximum depth of each roughing pass. Rough Type controls the type of roughing motions desired.

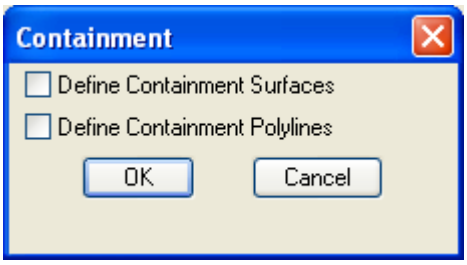


A total of 6 roughing methods are available with the Expert NURBS Cutter. They are divided into two categories, Vertical and Horizontal. When the XZ or YZ Cycle is selected, you are in Vertical Cutting. When the XY Cycle is selected you are in Horizontal Cutting.

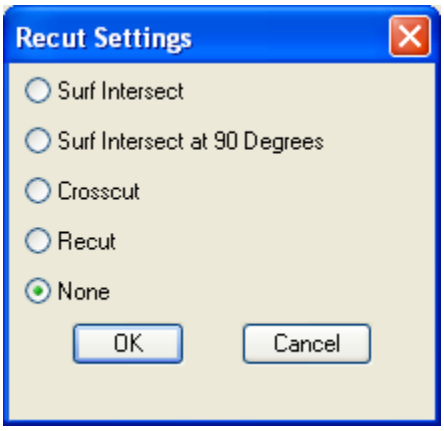
Plunge Points allows user selection of plunge locations. Always provide multiple plunge point locations. The system may create alternate plunge locations. Plunge points can be located inside or outside of the part.

The Containment Button opens an Edit Window for selection of Surface Containment or Polyline Containment.

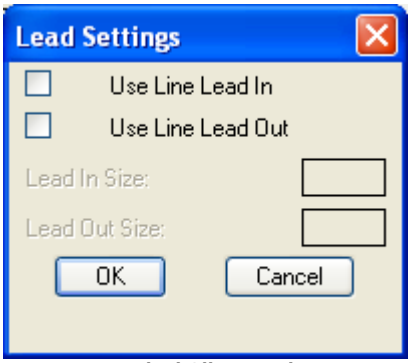
© 2011 CIM-TECH.COM, Inc



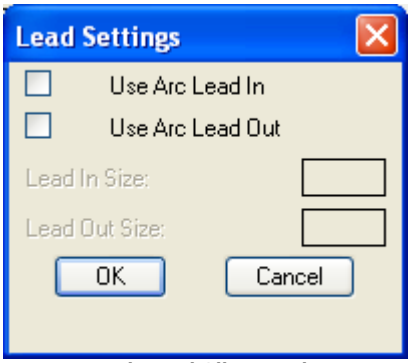
The Recut Settings Button opens an Edit Window for selection of Recut methods. Surface Intersect produces toolpath at surface(s) intersection(s). Crosscut produces toolpath on surfaces that meet angle criteria. Recut is used to produce toolpath, using a tool smaller than the previous tool, in areas that the larger tool could not access.



The Leads Button is to generate a line lead in and a line lead out of the tool path.
The Button opens an Edit Window for selection of Line Lead-In/Out's when using XZ or YZ Cutting.
Lead size is controlled by the Lead Size entry. Leads do not apply to XY Roughing.
The Button opens an Edit window for selection of Arc Lead-In/Out's when using XY Cutting.



Vertical Slice Leads



Horizontal Slice Leads

6.1.5.4 Tools in XNURBS

Tools

The NURBS Cutter supports BALL, BULL and FLAT End Mill Tools. Certain tool parameter entries are critical in order to achieve desired results.

© 2011 CIM-TECH.COM, Inc

The following tool parameter entries are used to calculate proper toolpath for the three types of tools supported.

Parameter [16]Tool Type (21,22,13,23)
Flat End Mill = 21
Ball End Mill = 22
Bull End Mill = 23
Drill = 13

Parameter [19]Tool Tip Dist to Zero(#)
This parameter will loft the tool path by the amount shown.
Flat End Mill = enter 0.0.
Ball End Mill = enter value of ball radius.
Bull End Mill = enter value of bull nose corner radius.

Parameter [19], "Tool Tip Dist to Zero(#)"
is also used to define corner radius of the tool for Bull End Mills.

Parameter [21]Tool Diameter *TW* (#)
Enter tool diameter regardless of tool type.

Parameter [23]Tool Radius *TR* (#)
Enter tool radius regardless of tool type.

Parameter [24]Tool Cut Depth *TD* (#)
Enter maximum cut depth per machining pass regardless of tool type.

6.1.5.5 Establishing the Feed Plane

Establishing The Feed Plane

The default Feed Plane of the Expert NURBS Cutter is based on the entry in Material Z Location of the Knowledge Editor. Feed Distance to Material (Tool Editor entry) is added to the entry in Material Z Location. An option exists to allow the Feed Plane to vary, based on where the next X or Y cutting motion(s) begins. The Feed Distance to Material entry is added to this position instead of the Material Z Location. This option is controlled by the NC Variable "NCS_CutPlaneMode". When "NCS_CutPlaneMode "is set to True (T) the toolpath will output rapid motions to within the value of Feed Distance to Material from where material is calculated to be already removed.

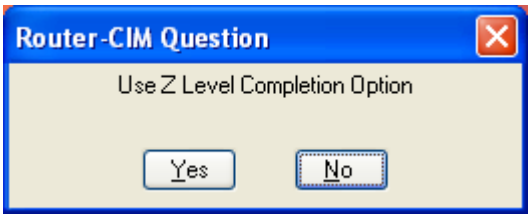
To change this NC Variable type NCVAR, select SYSTEM, and change "NCS_CutPlaneMode" to T. Although cycle time can be longer when feeding begins at Material Z Location it is generally considered safer to avoid rapid motions below this plane. You choose the method that best suits your company's policies and procedures. You can change this NC Variable at any time. The Examples in this tutorial are all run with this NC Variable set to nil.

6.1.5.6 Z Level Completion Option

Z Level Completion is used when you wish the tool path to move up to the clearance plane between each pass. Effectively completing one pass in Z, then retracting, moving to the start of the next pass, plunging and then finishing that Z level cut and retracting to the safety plane, and repeating the process.

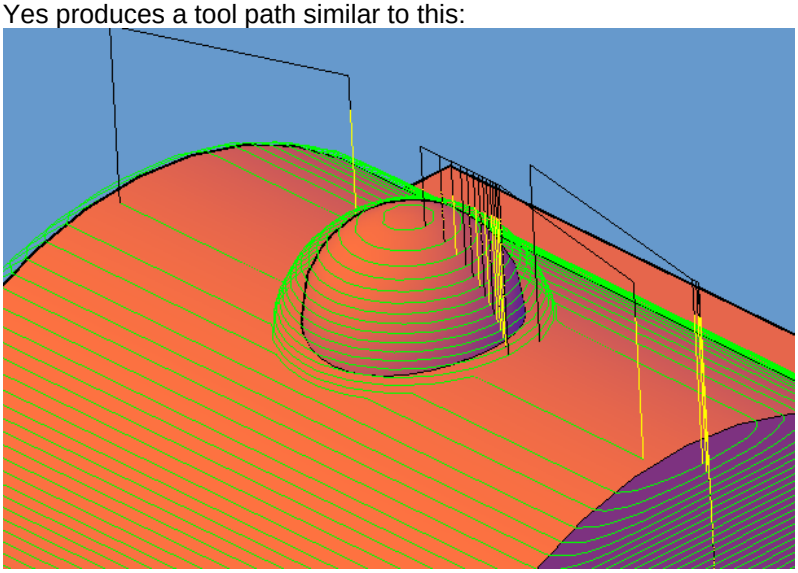
The Prompt will appear on the screen when using the NURBS_CCW_XY horizontal slicing cycle.

© 2011 CIM-TECH.COM, Inc



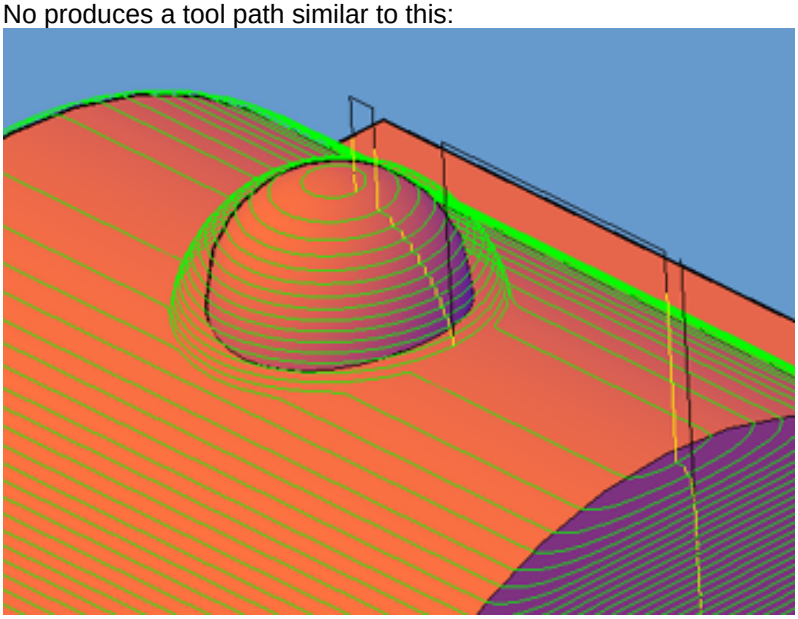
YES is used when a Cavity or Core contains multiple hills and valleys. YES will cause the toolpath calculator to complete all toolpath(s) required at each Z level, retract, and then go down to the next Z level.
NO continues to cut downward until complete.

Yes produces a tool path similar to this:



Where the tool is retracting between each slice it takes in XY at the current Z plane.

No produces a tool path similar to this:



© 2011 CIM-TECH.COM, Inc

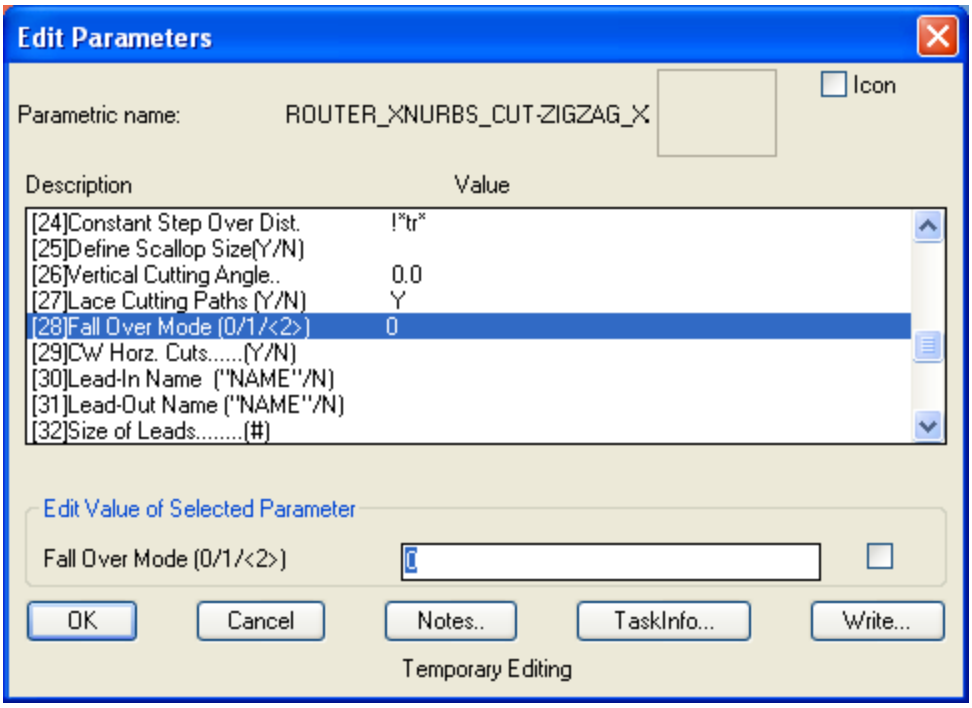
Once the tool completes a slice in the XY at the current Z plane, it simply moves down to the next Z plane and starts another slice in XY.

6.1.5.7 Fall over Mode in Vertical Cutting

Fallover is a term used to describe how the tool path reacts to the edge of the cutting surface. If the tool goes beyond the surface or solid edge, it will fall over by some amount. The control of the portion of the tool path behavior is set in the NURBS_XZ and NURBS_YZ cycles by means of a parameter (#28) named Fall Over Mode.

There are 4 possible settings for Fall Over Mode. You may specify 0, 1, 2, 3 for this parameter, which will change the behavior of the tool path as follows:

Setting Fall Over Mode:
Using the Modify Cycle button on the Knowledge Editor, select Parameter #28:

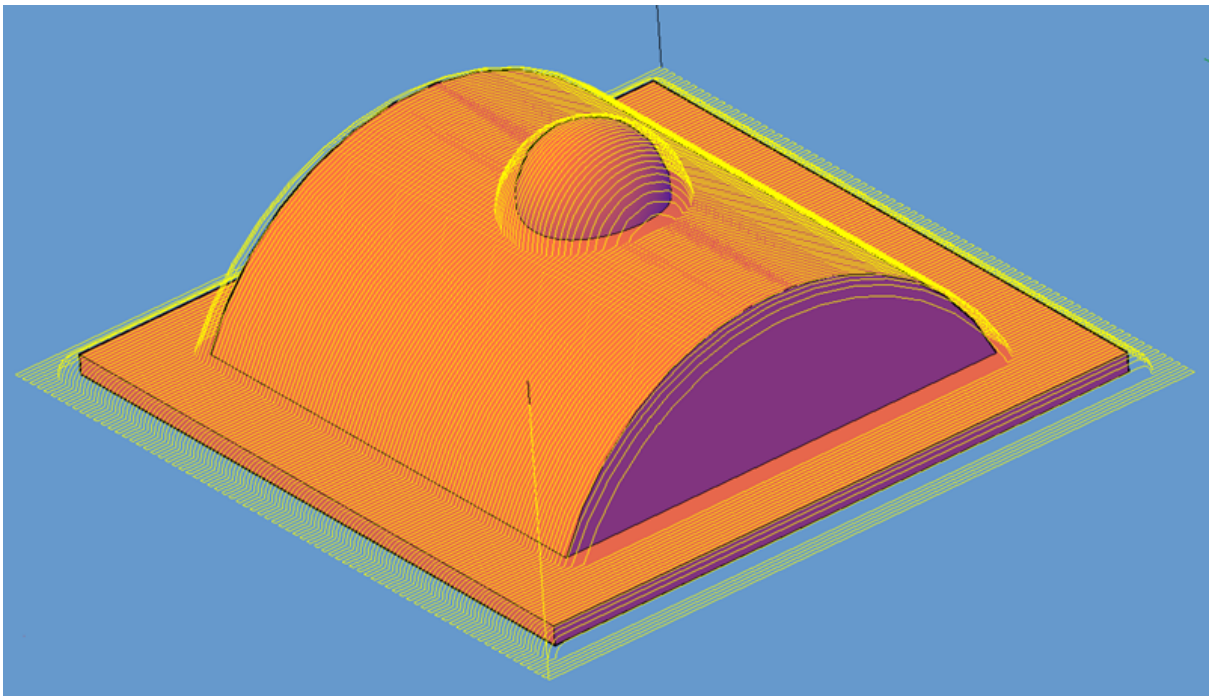


In the Edit Value box, type in the mode you want to set and then press Enter. You should see the list update with the number you typed in.

Fall Over Mode 0:

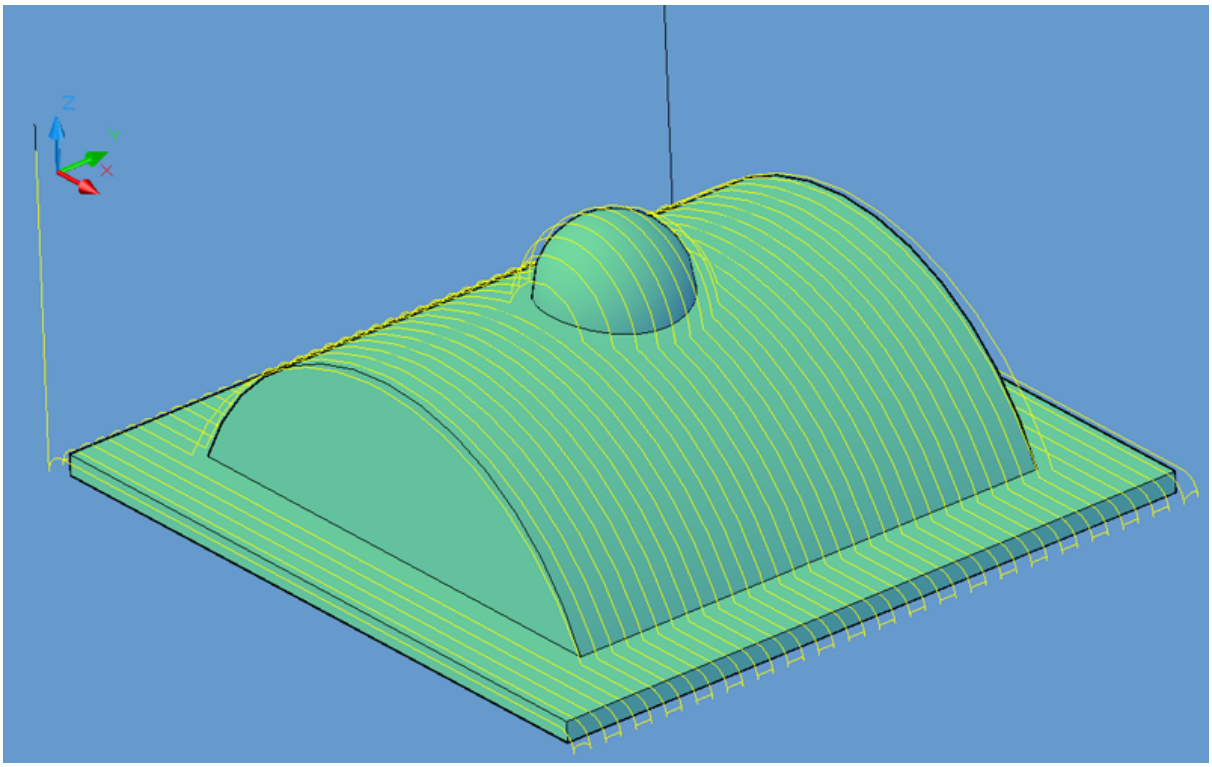
This method will cut down to the Maximum Z Depth set, then move over by a tool diameter in X or Y (depending on cycle) and then move over and return for the next pass.

© 2011 CIM-TECH.COM, Inc



Fall Over Mode 1 OR Fall Over Mode 2:

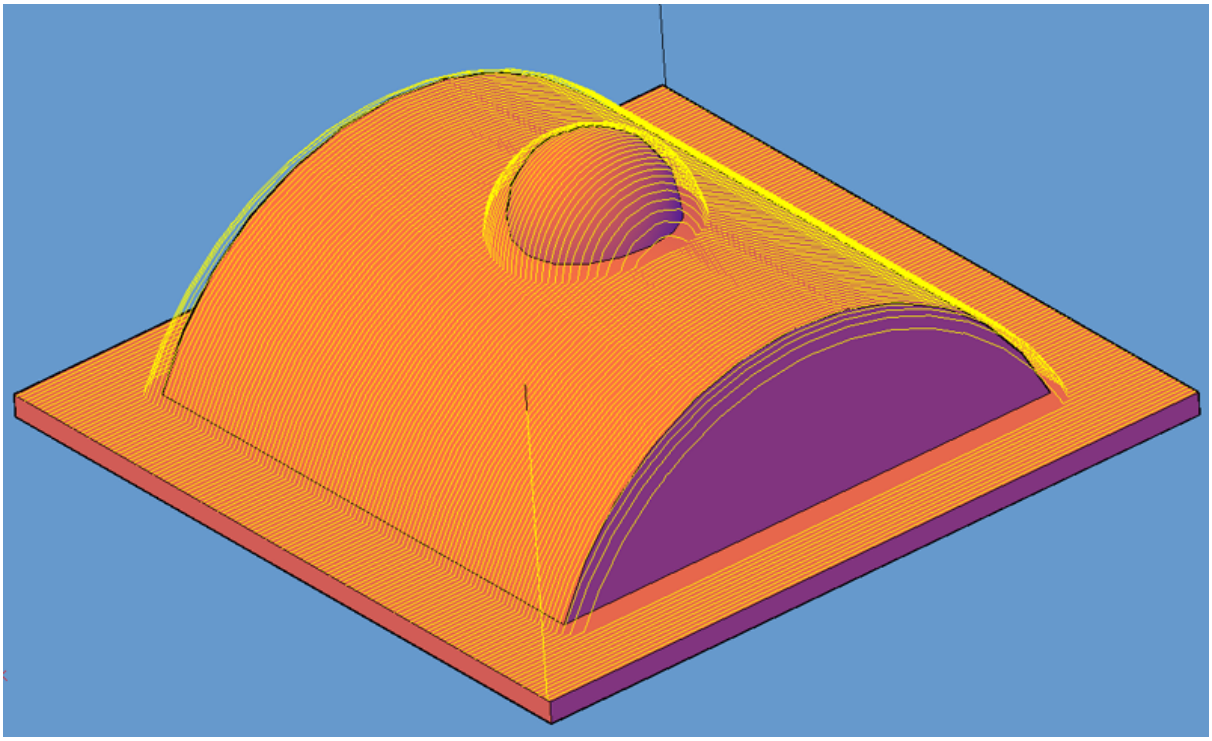
Fall Over Modes 1 and 2 are the same and switch between vertical slicing in X or Y. There is a slight overcut at the edge of the surface, before the tool path steps over and then returns.



© 2011 CIM-TECH.COM, Inc

Fall Over Mode 3:

This is the default method if none is specified. The tool path will cut to the edge of the solid or surface and stop, move over and return. There is no Fall Over actually done.



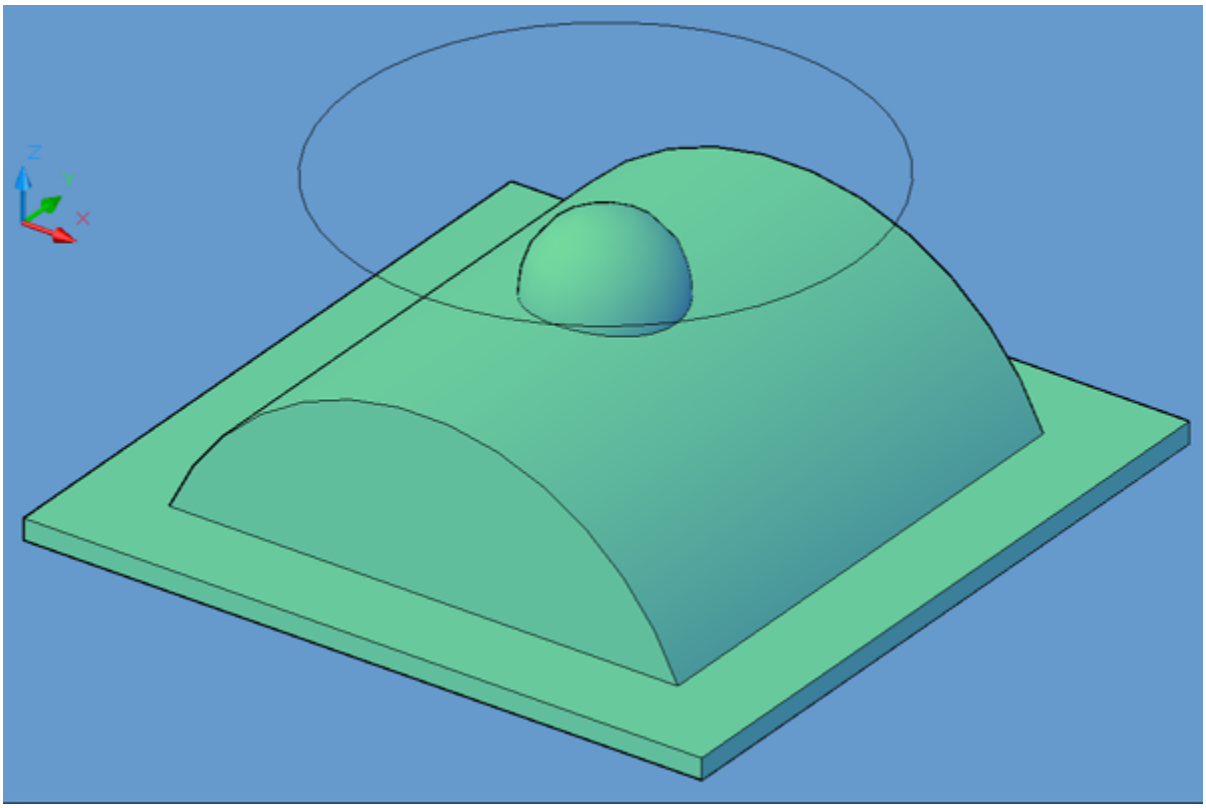
© 2011 CIM-TECH.COM, Inc

6.1.5.8 Containment Cutting

Containment cutting is the ability to control the generation of the tool path within a specified area of the part. This is done either by a polyline used as the containment area or by cutting a separate surface if you are drawing with MDT and have Nurbs surfaces on your part.

The example given here will show a containment polyline controlling the area where the tool path is created. The polyline must be closed, and actually be a polyline. This is easy to achieve if you are using a circle, ellipse or some other irregular shape or combination of lines and arcs, you can always Geoshape them and use the result from Geoshape for the containment polyline.

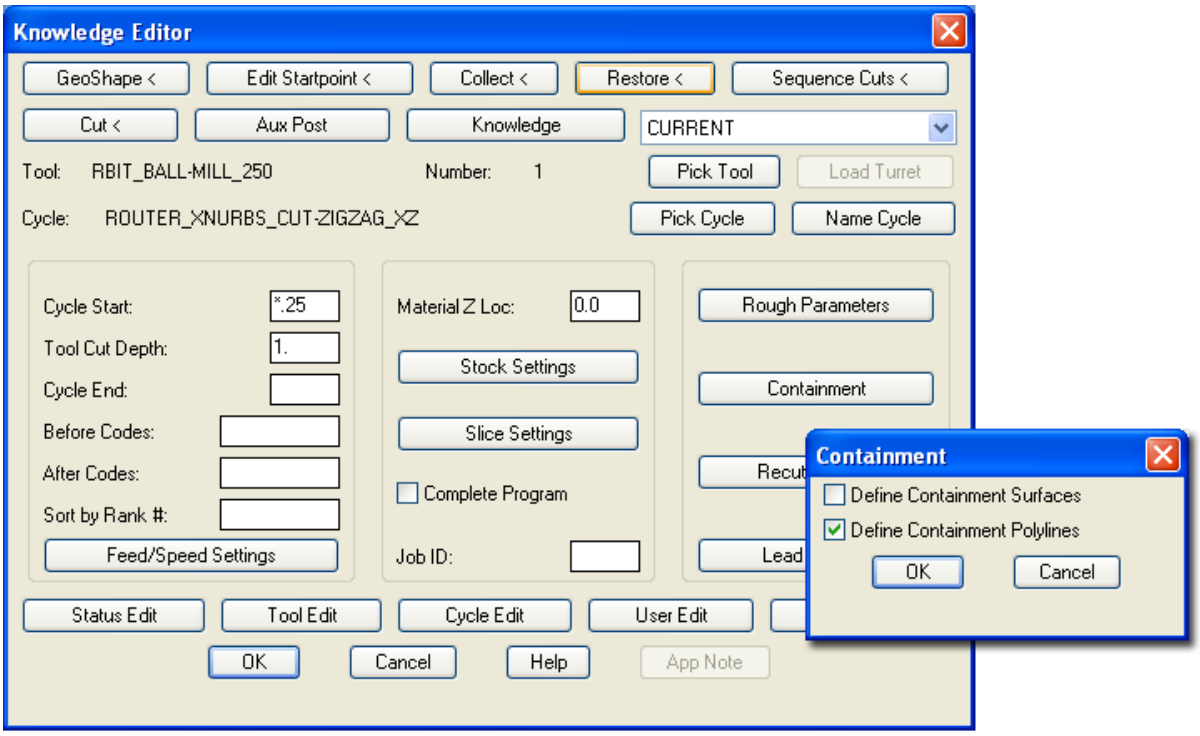
In this instance, the MCADTPC.dwg is used with a circular containment area created.



The containment polyline has been created at Z0 and so is above the part.

The cut settings are set up with the Nurbs XZ cycle defaults, but with a Containment Polyline selected in the Containment options.

© 2011 CIM-TECH.COM, Inc



Select Cut, and when prompted to redefine the surfaces, answer Yes. Select the green solid for the part.

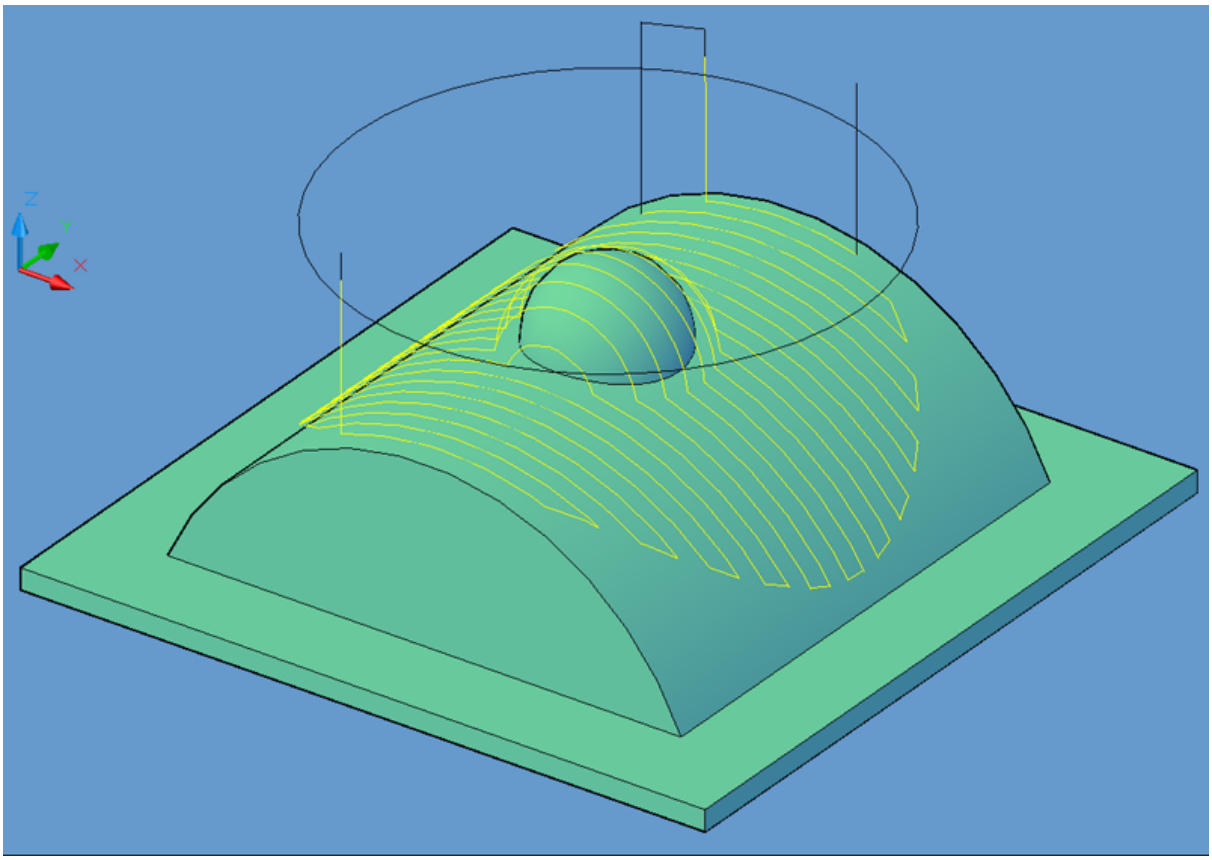
You will be prompted to input the Surface Tolerance, use .005.

```
Enter Surface Tolerance <0.00050000>: .005
```

Next you will be prompted to select the Containment Polyline(s), select the circular shape above the part.

You should see a tool path similar to the one shown here:

© 2011 CIM-TECH.COM, Inc



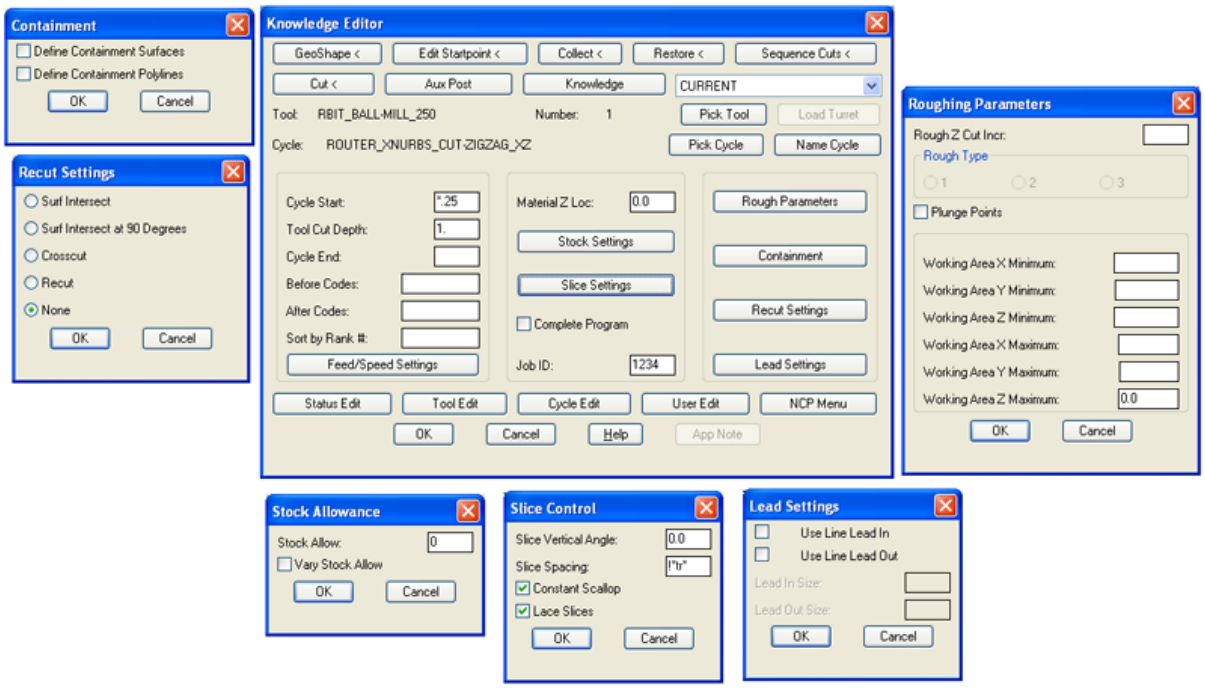
If there are any areas where the tool cannot continue its path without crossing the containment line, the tool will be forced up to the Safety Plane and then it will move over to the next area where it can continue the path. This is to avoid gouging the part or violating the containment polyline.

© 2011 CIM-TECH.COM, Inc

6.1.5.9 Constant Scallop Height

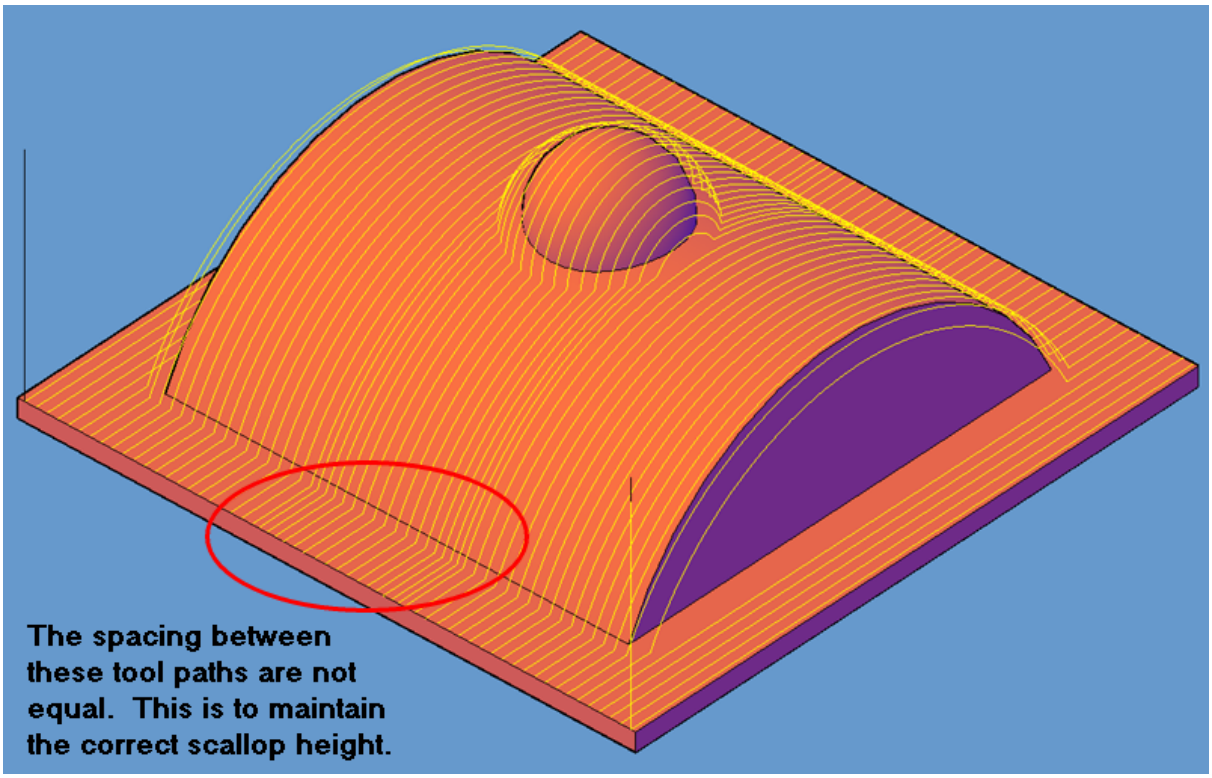
Scallops are the ridges left behind from the cutter stepping over in X or Y to cut the next pass. A Ball Mill will always leave a small scallop. The size of that scallop can be specified as a height, and Router-CIM will calculate the step over spacing between each pass to maintain that scallop height. Some tool paths will be placed closer together than others, especially if they are by a sloped surface where the tool has to maintain the scallop height while moving up or down as well as in X or Y.

Activate the Cut Button after duplicating the Knowledge Editor entries shown below using MCADTPC.DWG, to achieve similar results.
When prompted during CUT, enter .005 for Scallop Height and .001 for minimum step size.
If prompted to Define New Surfaces, select Yes.



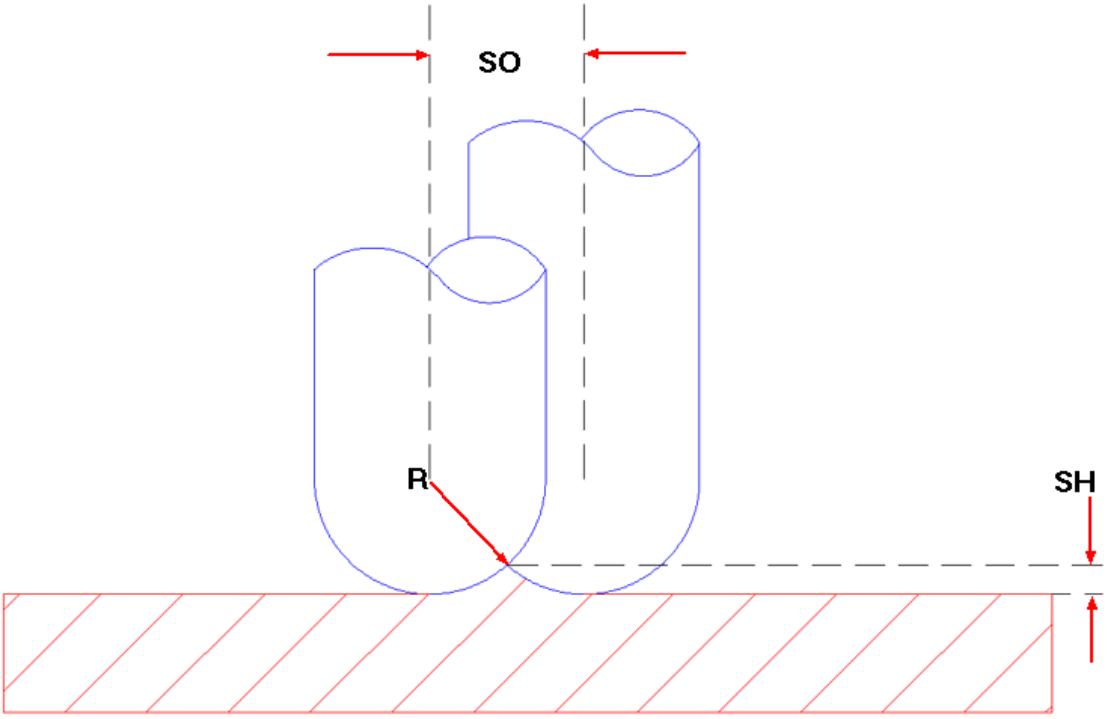
These settings will produce a tool path that has varied step over amounts wherever the tool needs to space the tool paths out differently to maintain the scallop height specified. Even though the Slice Spacing was set to !*tr* (Tool Radius) the same as the last two, the tool paths are clearly set to an amount very different from that specified so that they can maintain the scallop height of .005". In addition, they are much closer together at the edges of the sphere on top because the tool must step over less in this section to maintain the scallop height.

© 2011 CIM-TECH.COM, Inc



Router-CIM will maintain the Scallop Height indicated throughout the tool path created. If you want an idea of how much to set the scallop height to or what the step over amount would be, there is a formula to calculate the scallop height:

© 2011 CIM-TECH.COM, Inc



R = Cutter Radius
SO = Step Over
SH = Scallop Height

$$SH = R - \sqrt{R^2 - (SO/2)^2}$$

Or another way to express this would be:

SH=(R-(Sqrt ((R**2) - ((SO/2)**2))))

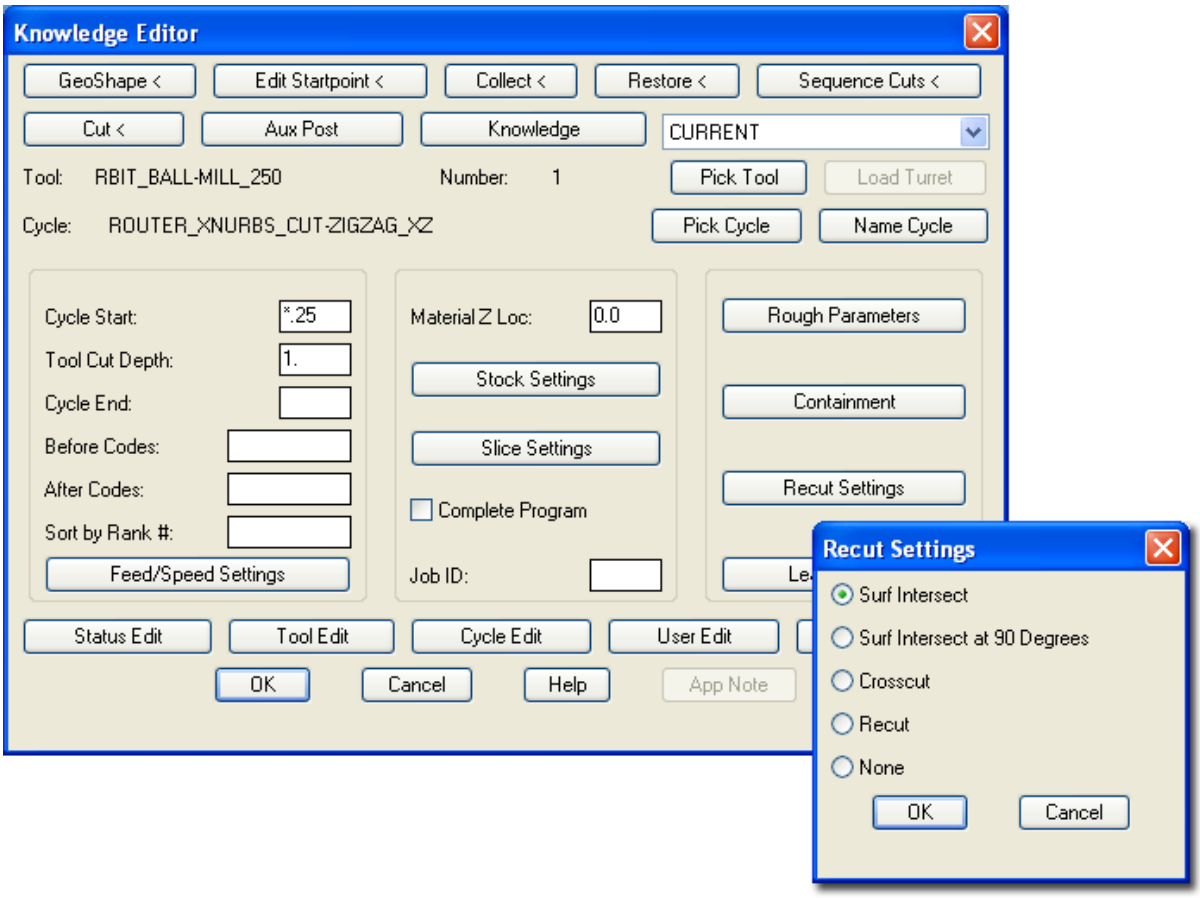
6.1.5.10 Intersect Cutting

Intersect Cutting will create a separate profile tool path at the intersection of each surface to create a smooth transition at these locations. This can also be useful to remove material in an area where a larger tool was used and left more material on the part than desired.

© 2011 CIM-TECH.COM, Inc

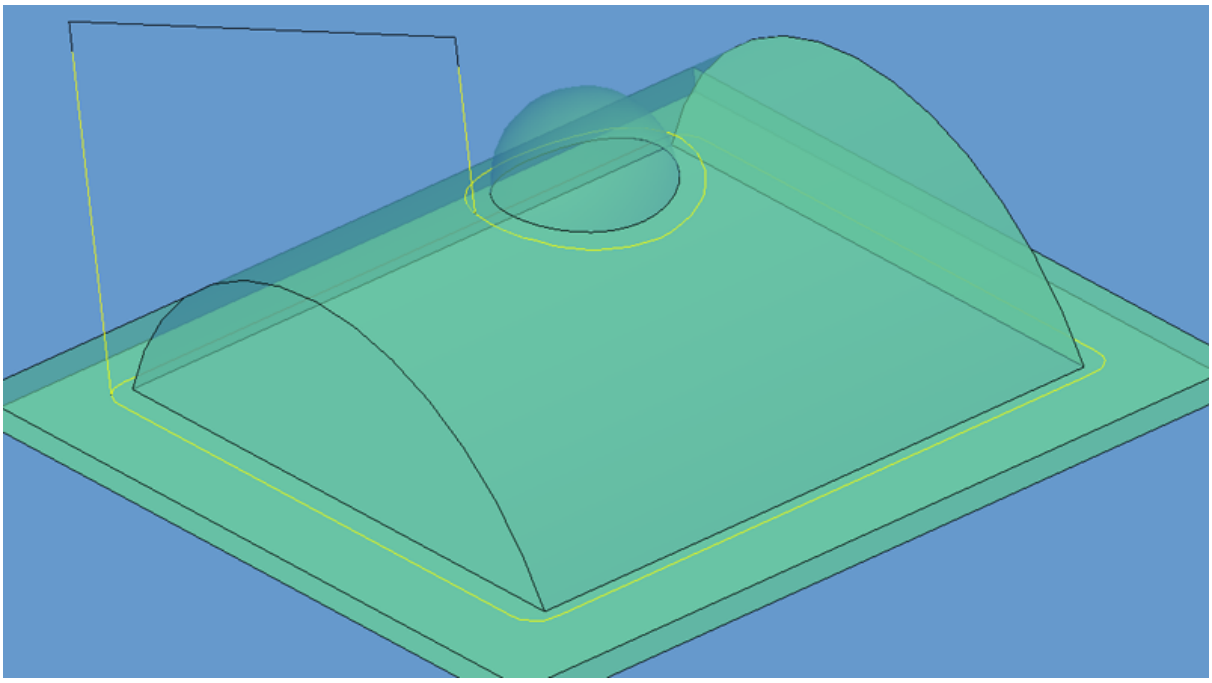
Surf Intersect

Shown below is the MCADTPC.dwg with the Surface Intersect turned on in Recut Settings.



Using these settings produces a tool path result similar to this:

© 2011 CIM-TECH.COM, Inc



This is a useful feature when a larger tool is used in a roughing or finishing operation and a smaller tool is needed only at the intersections of any geometry to create a smoother transition from one surface to the next.

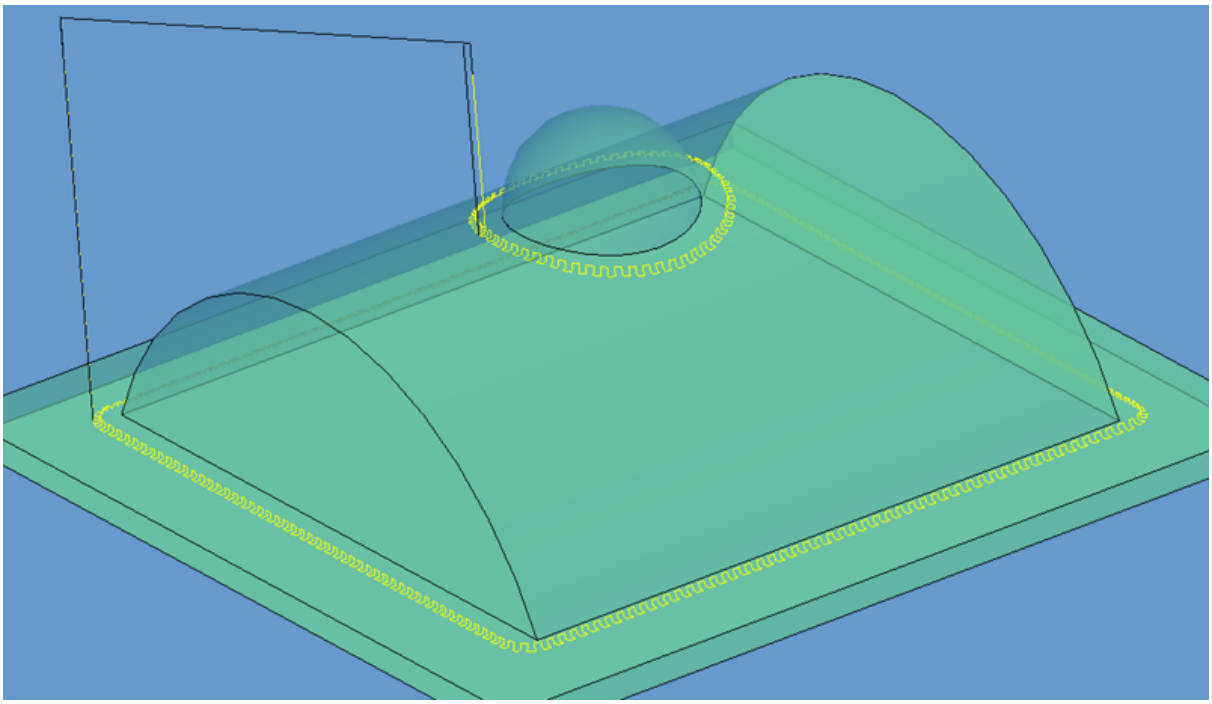
Be aware that this tool path does not consider the radius offset of the tool, so if you are creating this cut with a ball end mill, and the zero point for the end mill is the radius of the tool, then a Stock Allowance equal to the radius of the tool should be used to make sure the tool path is lofted and the part is not overcut.

Surf Intersect at 90 Degrees

Using the Surf Intersect at 90 Degrees creates a Zigzag tool path following the intersection of any of the contours on your part.

Changing the settings above to use Surf Intersect at 90 Degrees will yield a tool path with these results:

© 2011 CIM-TECH.COM, Inc

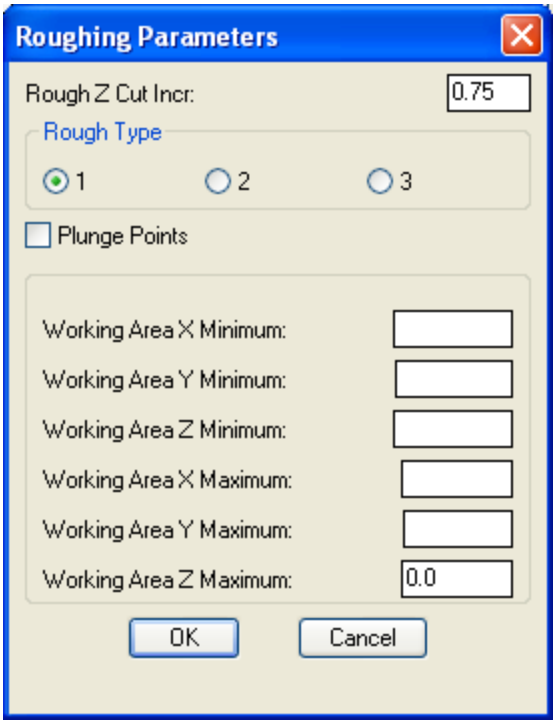


These motions will create many more moves than the first Surf Intersect, resulting in more NC Code, but possibly a smoother transition. The Slice Spacing will usually be changed to yield a much smaller scallop on the part.

© 2011 CIM-TECH.COM, Inc

6.1.5.11 Roughing

Using roughing passes, you can remove large volumes of material from a part with a large tool prior to making a finish pass (or passes) with a smaller tool. This helps lower the overall cycle time and save on tool wear. There are several methods of rough cutting in Router-CIM, and each will be described in as much detail as possible. There are example cuts shown for each method that employ settings that are intended to show the method and settings of the cycle, though are not necessarily the most efficient settings for that particular tool or cycle.



Vertical Cutting + Mode 1

Offsets each roughing pass by "Rough Z Cut Increment" + "Stock Allowance", moving in rapid when machine motions exceed "Working Area Z Maximum".
The available cycles for this method are Nurbs_XZ and Nurbs_YZ.

Vertical Cutting + Mode 2

Offsets each roughing pass by "Rough Z Cut Increment" + "Stock Allowance", and proportionally distributes Z depth throughout the roughing activity without allowing tool to leave material.
The available cycles for this method are Nurbs_XZ and Nurbs_YZ.

Vertical Cutting + Mode 3

Shifts roughing motions in "Z" based on "Rough Z Cut Increment" while compensating for "Stock Allowance", moving in rapid when machine motions exceed "Working Area Z Maximum".
The available cycles for this method are Nurbs_XZ and Nurbs_YZ.

Horizontal Cutting + Mode 1

Plunges straight down to each Z level, compensates for "Stock Allowance", roughs without following collective surface contours before proceeding to next lower Z level. Plunge Points can be used in conjunction with this method of roughing.
The available cycle for this methods is Nurbs_CCW_XY.

Horizontal Cutting + Mode 2

© 2011 CIM-TECH.COM, Inc

Ramps down to each Z level, compensates for "Stock Allowance", roughs and makes a contour following pass while ramping down to next Z level.
The available cycle for this methods is Nurbs_CCW_XY.

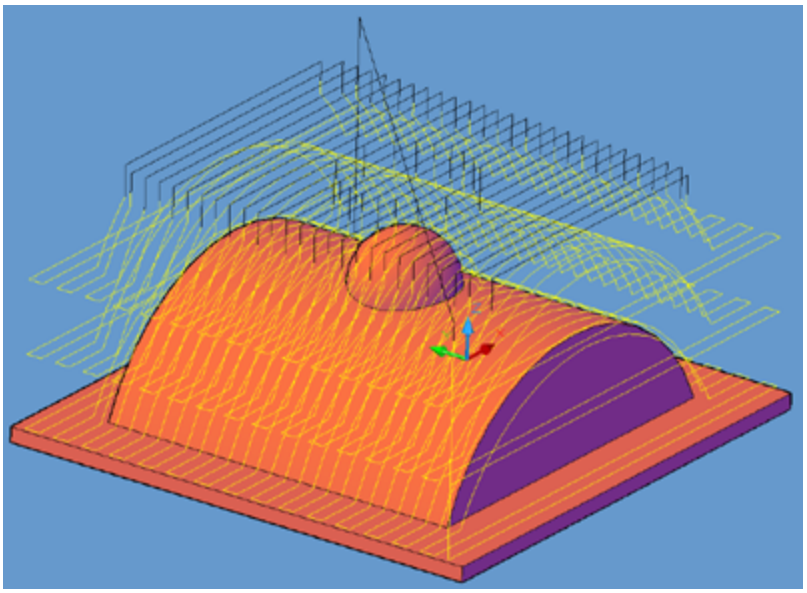
Horizontal Cutting + Mode 3

Plunges straight down to each Z level, compensates for "Stock Allowance", roughs and makes a contour following pass before plunging straight down to next Z level. Plunge Points can be used in conjunction with this method of roughing.
The available cycle for this methods is Nurbs_CCW_XY.

6.1.5.11.1 Vertical Roughing Mode 1

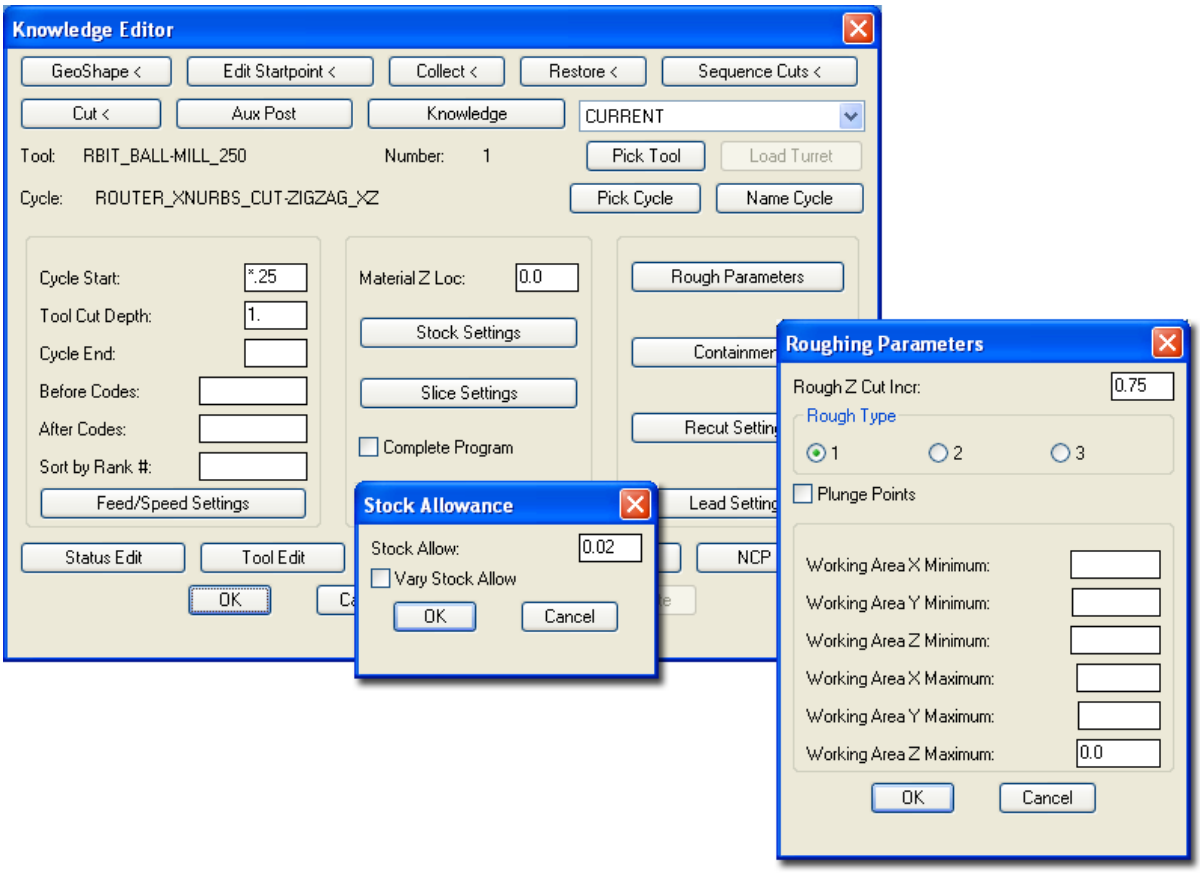
Vertical Roughing Mode 1 creates offsets of the shape at the various Z levels, and removes material where it exists at that Z level, indexing to the next location where there is no material to remove. The basic idea of this cycle is that the tool will be creating an approximation of the shape at each level, where the shape exists, and index where it does not to save time, and not cut 'air'.
This method is effecient if the tool can load and unload during the cut, and the indexing during some of the passes is not prohibitive to the tool or part.
In this case the first pass removes some material from the sides of the block, but spends some time indexing to the next cut position, before dropping down to the second level. The second level keeps the tool in the material for all but the top of the dome, and the third level keeps the tool in the material the entire time for the last rough pass.

These settings can be used with either the Zigzag XZ or Zigzag YZ cycles.

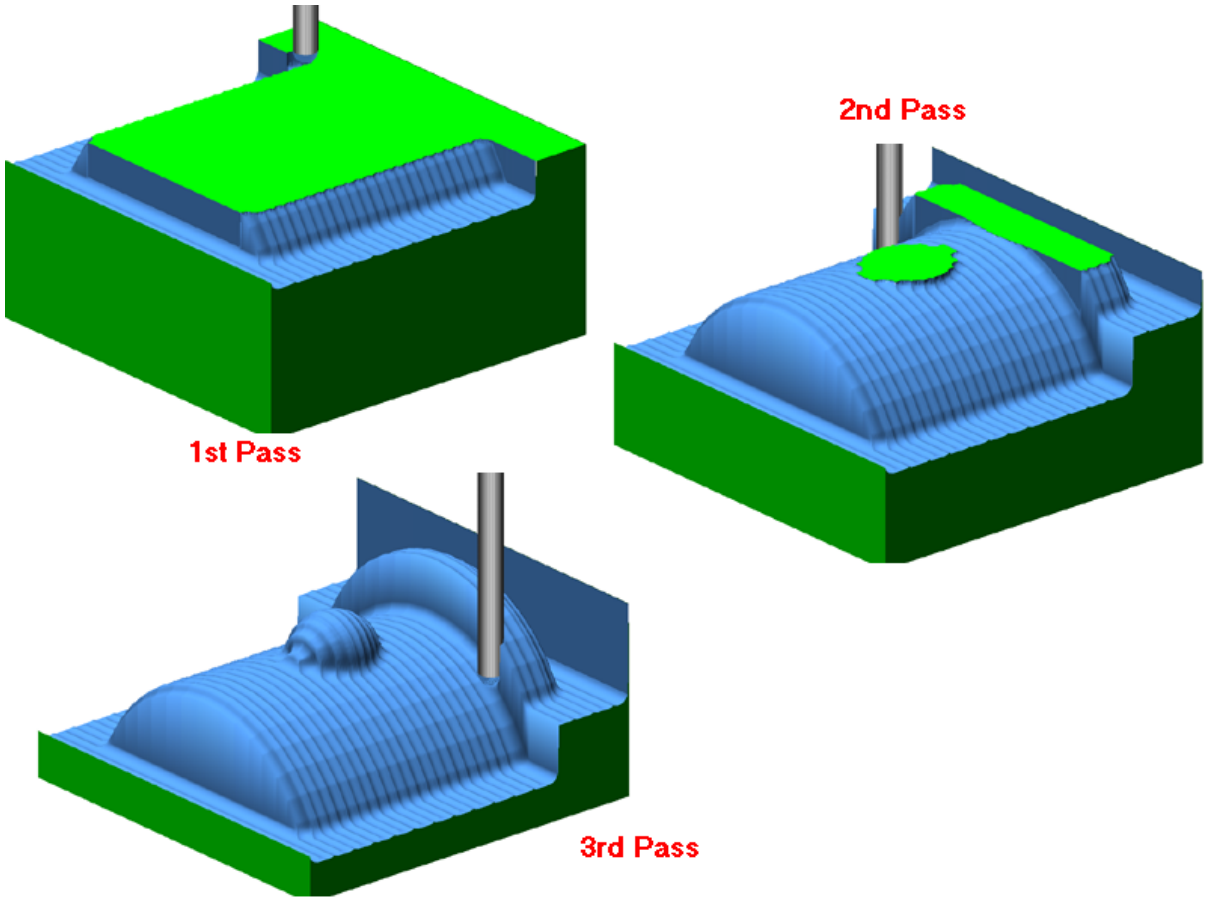


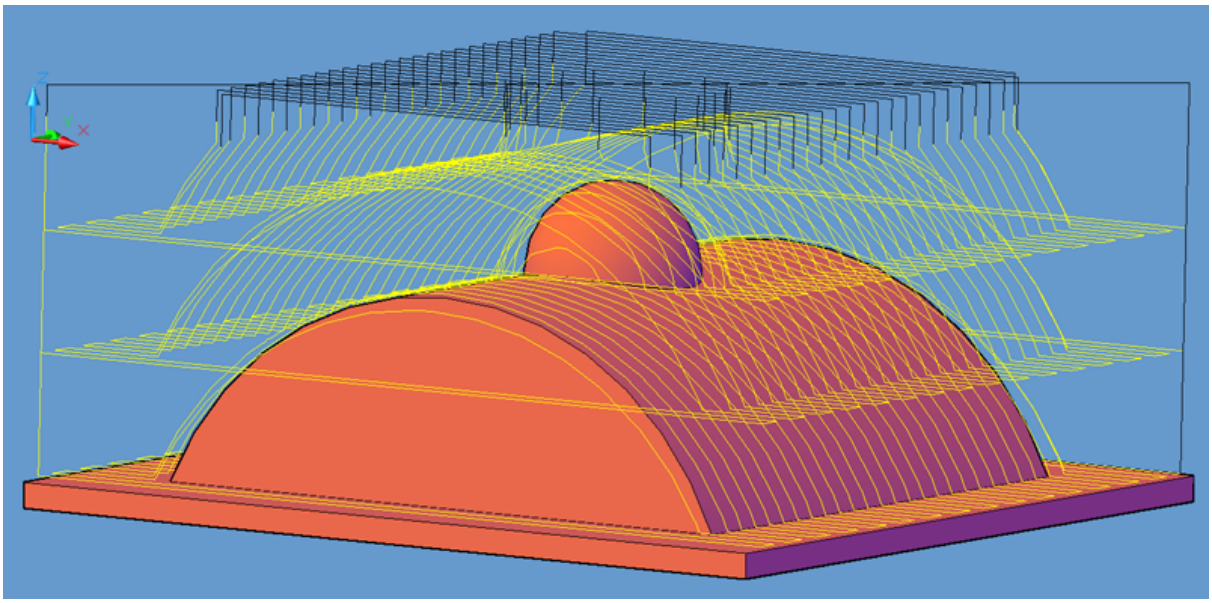
Using the MCADTPC.dwg and the following parameters, a similar roughing pass can be created.

© 2011 CIM-TECH.COM, Inc



© 2011 CIM-TECH.COM, Inc





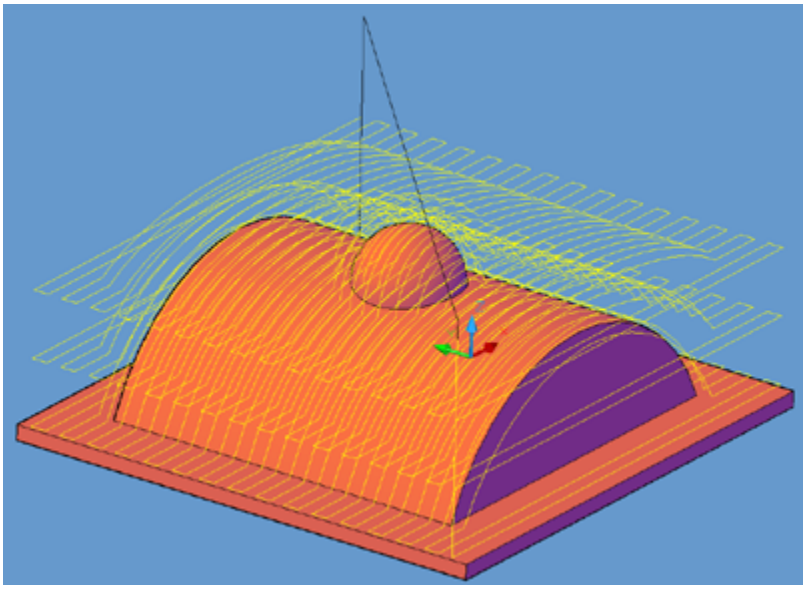
These roughing passes show where the tool lifts up and indexes over to the next cut position, only removing material from the part if it exists at that particular Z level. If the step down in Z was smaller, the top few passes would index at the Safety Plane for some of their cuts, but the bottom passes

© 2011 CIM-TECH.COM, Inc

would constantly be in material.

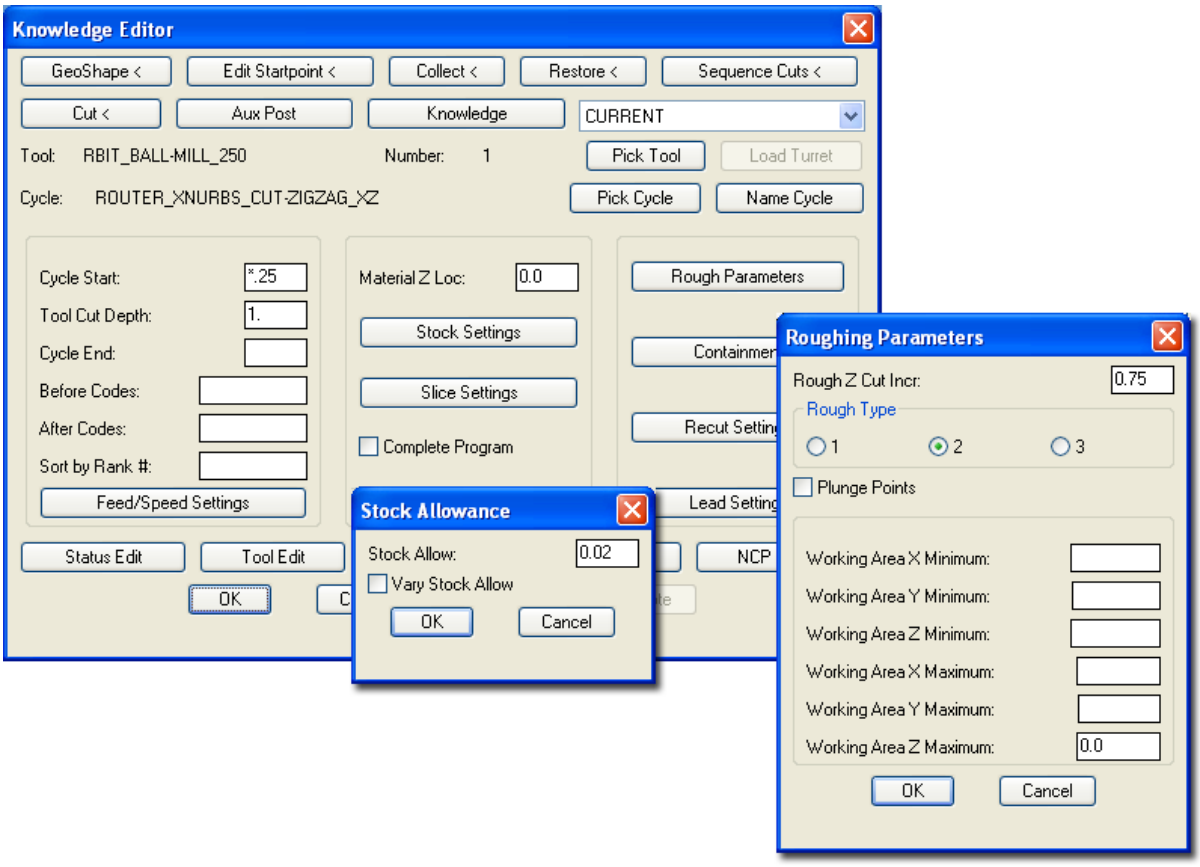
6.1.5.11.2 Vertical Roughing Mode 2

Vertical Roughing Mode 2 will remove material constantly from the part on each pass, with each pass approximating the part at that level. With this method the tool stays loaded during each pass, on each Z level until the cut is finished. This cycle cuts in a Zigzag fashion, so at the end of a pass the tool moves over and starts cutting back on the next pass, always in material.
This method of roughing is efficient when the tool must stay loaded up during the cut to get the most satisfactory performance. Since the tool is always cutting and does not index during any of the passes, this is the longest roughing cycle in terms of machine cycle time.
In this instance, the first pass removes material from the block, and cuts part of a contoured shape resembling the part at this Z level. The second pass removes more material leaving a more distinct impression of the part and still keeping the tool loaded during the entire pass. The last Z level finishes the roughing and removes the same volume of material as the previous passes.

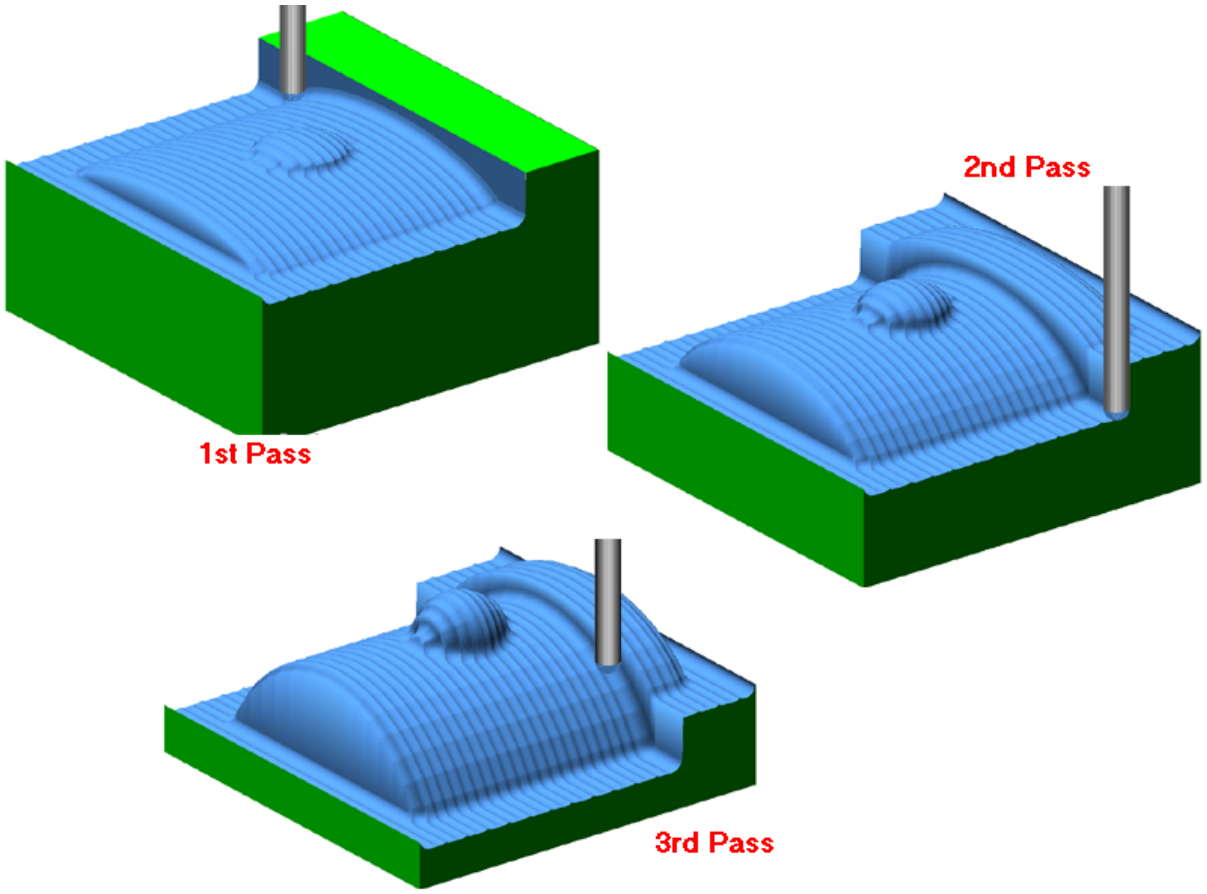


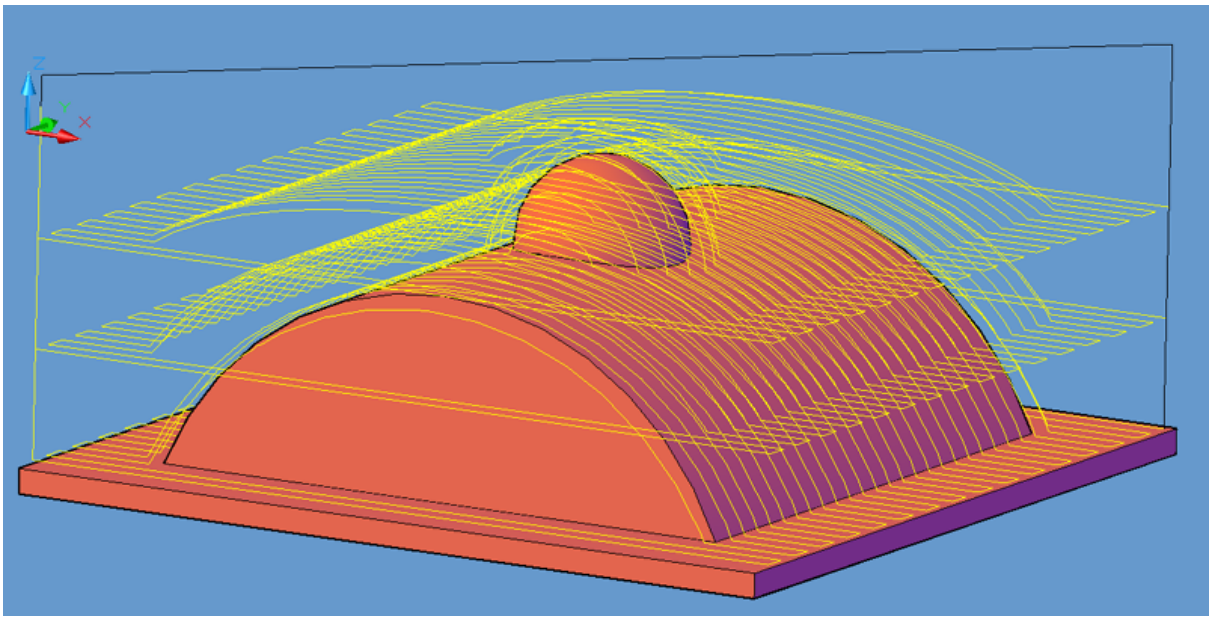
Using the MCADTPC.dwg and the following parameters, a similar roughing pass can be created.

© 2011 CIM-TECH.COM, Inc



© 2011 CIM-TECH.COM, Inc





These rouging tool paths show how the tool stays in the material cutting, never indexing during any of

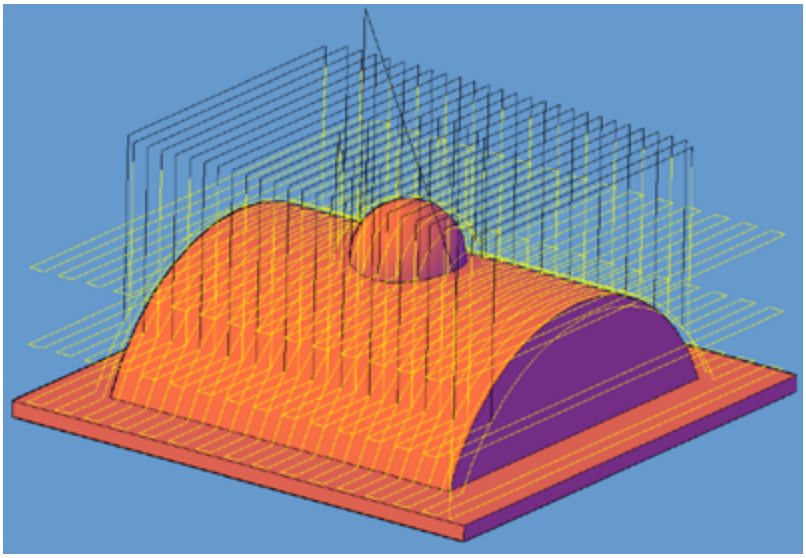
© 2011 CIM-TECH.COM, Inc

the passes.

6.1.5.11.3 Vertical Roughing Mode 3

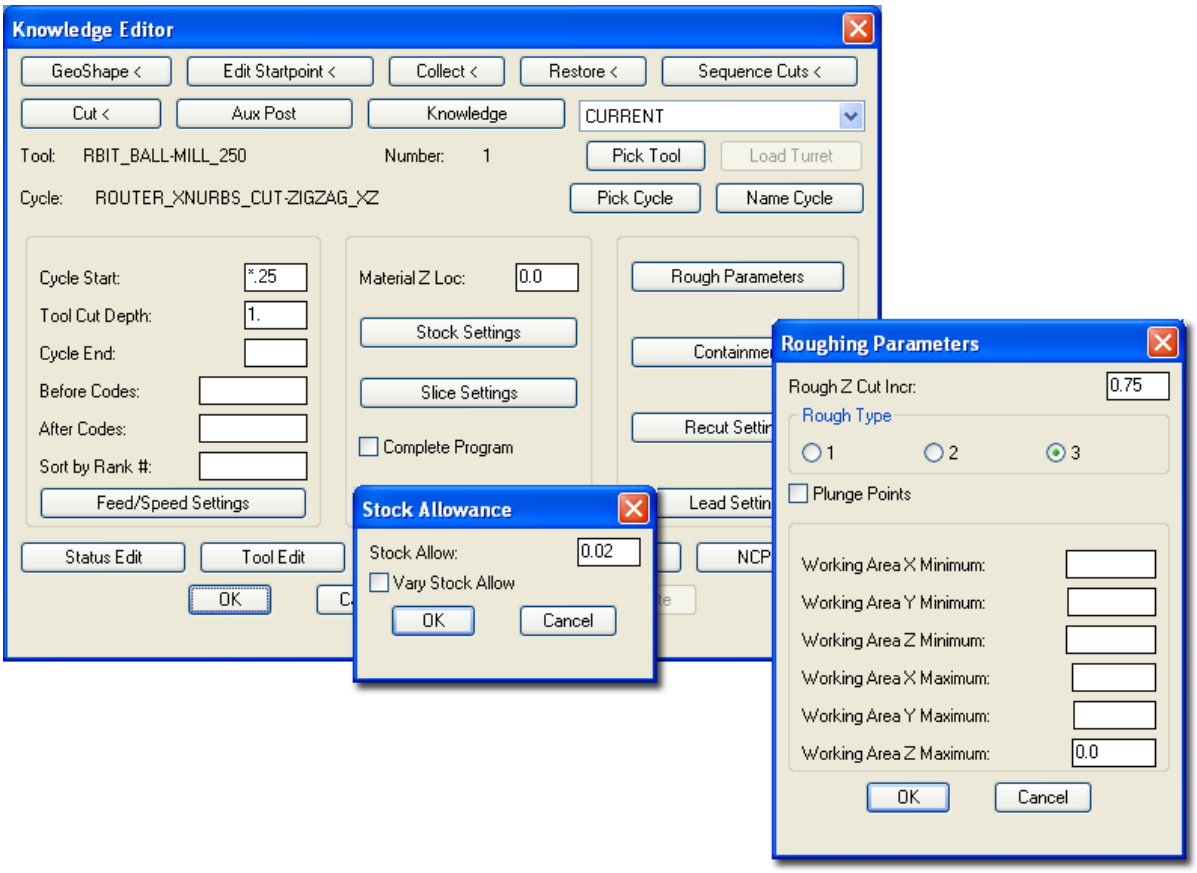
Vertical Rouging Mode 3 will remove material at the plane of the cut, and remove material from the part at the finished roughing size on each pass. Router-CIM will cause the tool to move to the Safety Plane and index to the next position if there is no material to remove from the part, or the block of material on the current pass. In the example that follows, the first pass is made removing material from the block and only the ball section at the top of the part. That ball section is never cut again on any of the subsequent passes, as the first pass removed all the material necessary. The second pass removes some material from the side of the part and indexes up to the Safety Plane and over to the other side of the part then down into the material again when there is material to remove from the block. The last pass removes material from the bottom section of the part but does not cut any of the part that was already cut on the first or second pass.

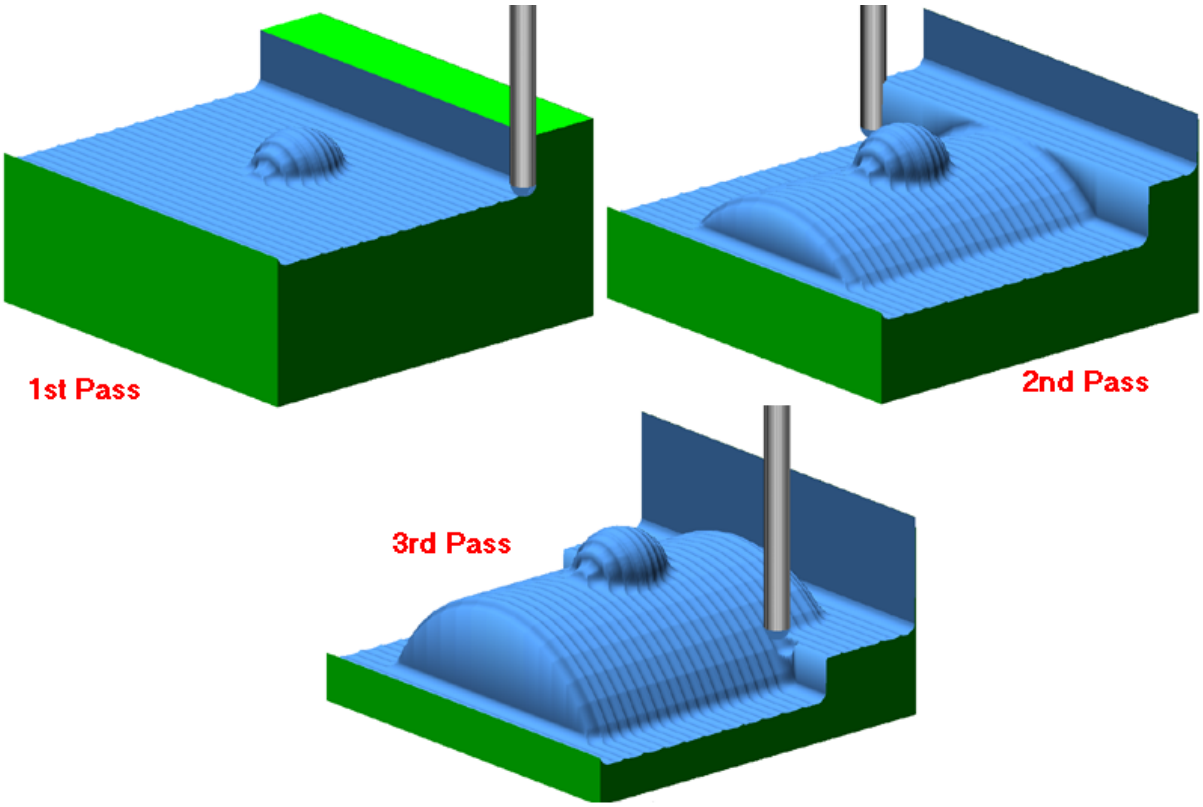
This is an efficient method of removing material if the tool can load/unload during the cut without affecting the part or the tool.



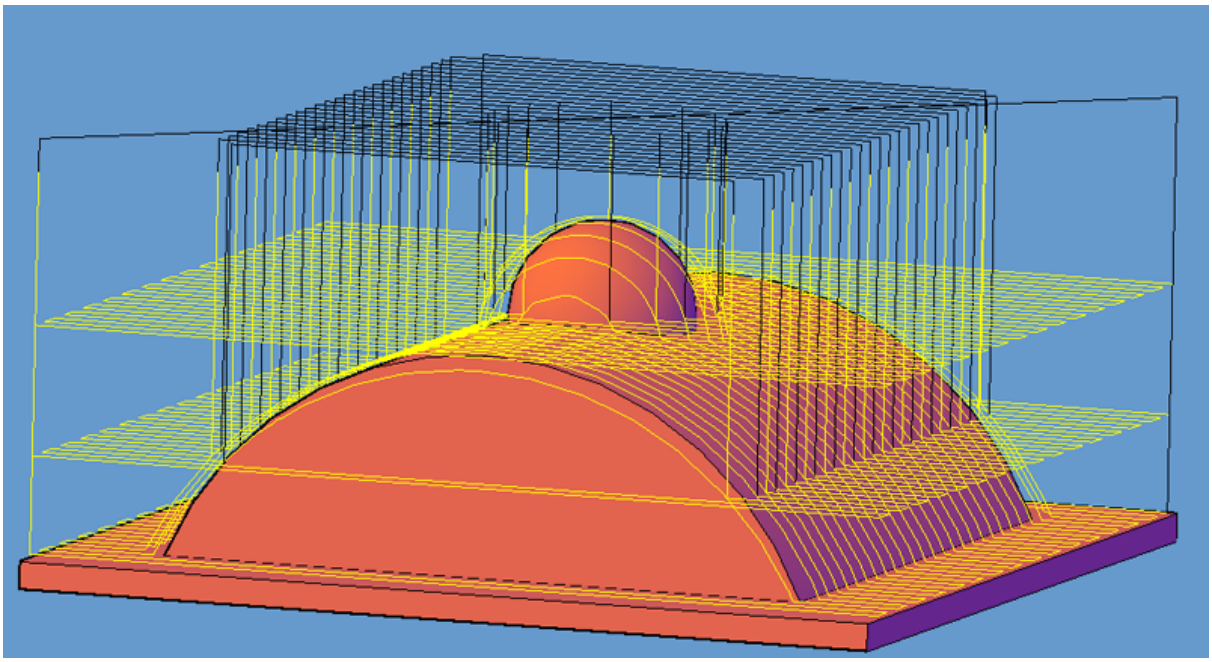
Using MCADTPC.dwg, and the following settings, a similar rouging tool path can be created.

© 2011 CIM-TECH.COM, Inc





© 2011 CIM-TECH.COM, Inc



These roughing passes show where the tool lifts up and indexes over to the next cut position, only removing material from the part itself once at each particular Z level.

6.1.5.12 Examples

To begin we will use the "MILL_XNURBS_CUT-ZIGZAG_XZ" Cycle. Proper operation of the Expert NURBS Cutter requires the establishment of a relationship between the surfaces to be cut, and the top of the stock material. The surfaces in the example parts MCADTPC.DWG and MCADFLOW.DWG, have been positioned as though the top of the stock material is located at Z 0.0. All surfaces are positioned below Z 0.0.
This is an important step as unreliable or confusing results can occur with geometry above Z0.0 in the cad drawing.

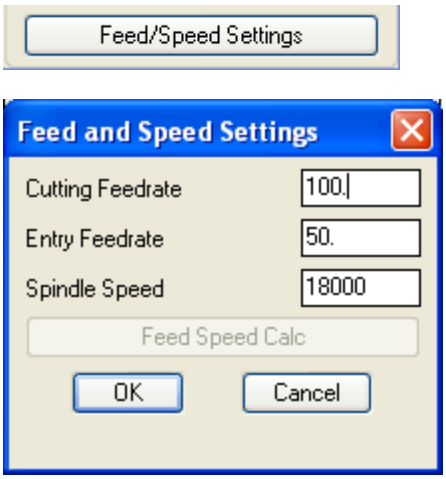
A ".GEO" file is produced by the system the first time you "Cut" the surfaces contained in the example drawings. When following along in the examples, always respond YES to the "Define Surfaces" prompt, because there are times when the original ".GEO" file is inappropriate for the exercise in the example.

Each Example will include screen captures of the Knowledge Editor, containing the proper entries to produce the results shown in the example. The only time you will be required to use the Edit Cycle function is in the Vertical Cutting Fallover Example #5. Fallover Mode is managed by the system for non finishing operations. The default Fallover Mode is Mode 3. Fallover Mode settings do not normally change on a day to day basis. Therefore, Fallover is not in the Knowledge Editor GUI to keep the operation of the system as simple as possible.

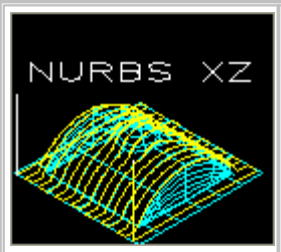
Detailed information on each of the Cycle Parameters is found in the previous section under NURBS CYCLE PARAMETERS.

The purpose of the Examples is to illustrate a variety of toolpath results, by varying the Knowledge Editor, or Cycle Parameter entries. Feed and Speed Settings do not effect the graphic results of toolpath. Therefore, you can enter these values one time, before beginning the exercises.

© 2011 CIM-TECH.COM, Inc



6.1.5.12.1 NURBS XZ

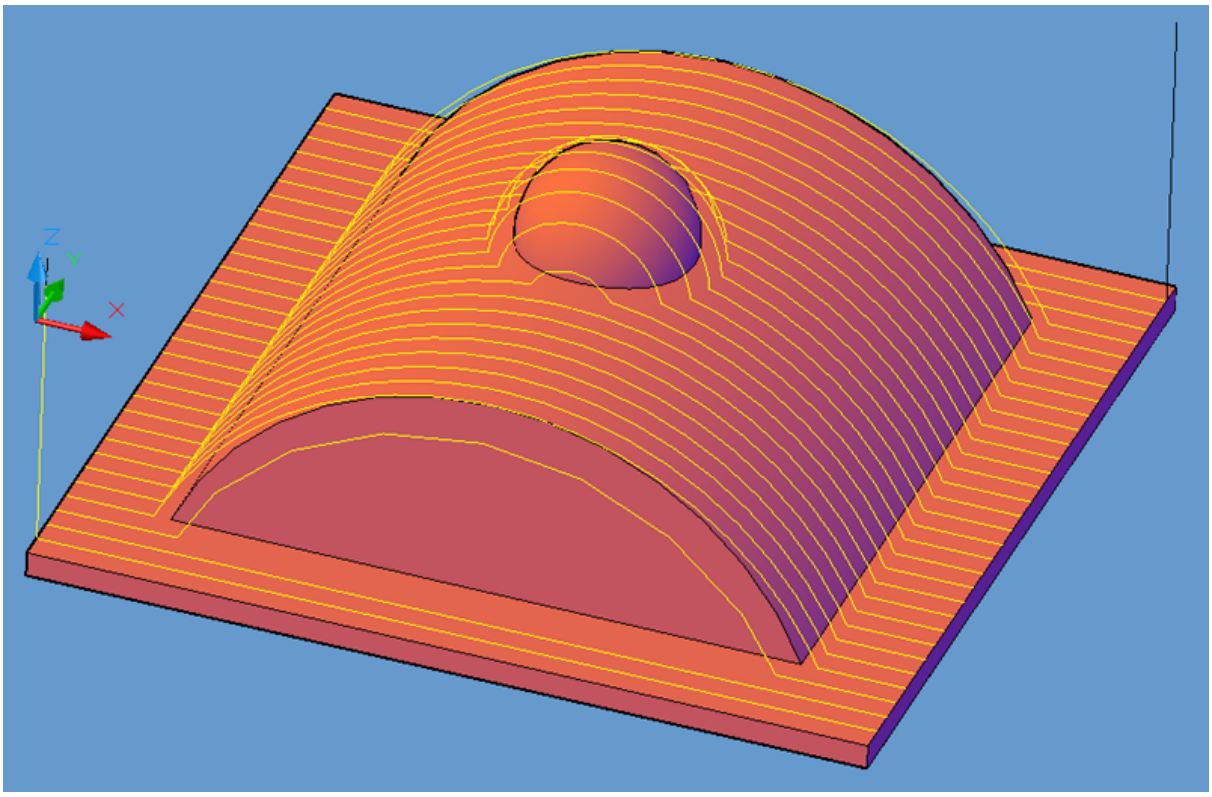


The NURBS XZ cycle is a Vertical Slicing cycle for cutting solids and surfaces inside of AutoCAD, or with the aid of the NCSurfer program. Tool paths will be generated in the X direction as slices following the contour of the shape in Z. The slices can be made in one direction, or laced together to allow slicing in both X directions (positive and negative), forming a zig-zag motion of the tool.

Essentially the NURBS XZ cycle is the same as the NURBS YZ cycle with a Slicing Angle of 0.0°.

There are many parameters available to fine tune this cycle to perform many operations and these will be described throughout the NURBS section of this document.

© 2011 CIM-TECH.COM, Inc



Shown is a tool path created with the default settings of the NURBS XZ cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

Enter .003

```
Enter Surface Tolerance <0.00050000>: .003
```

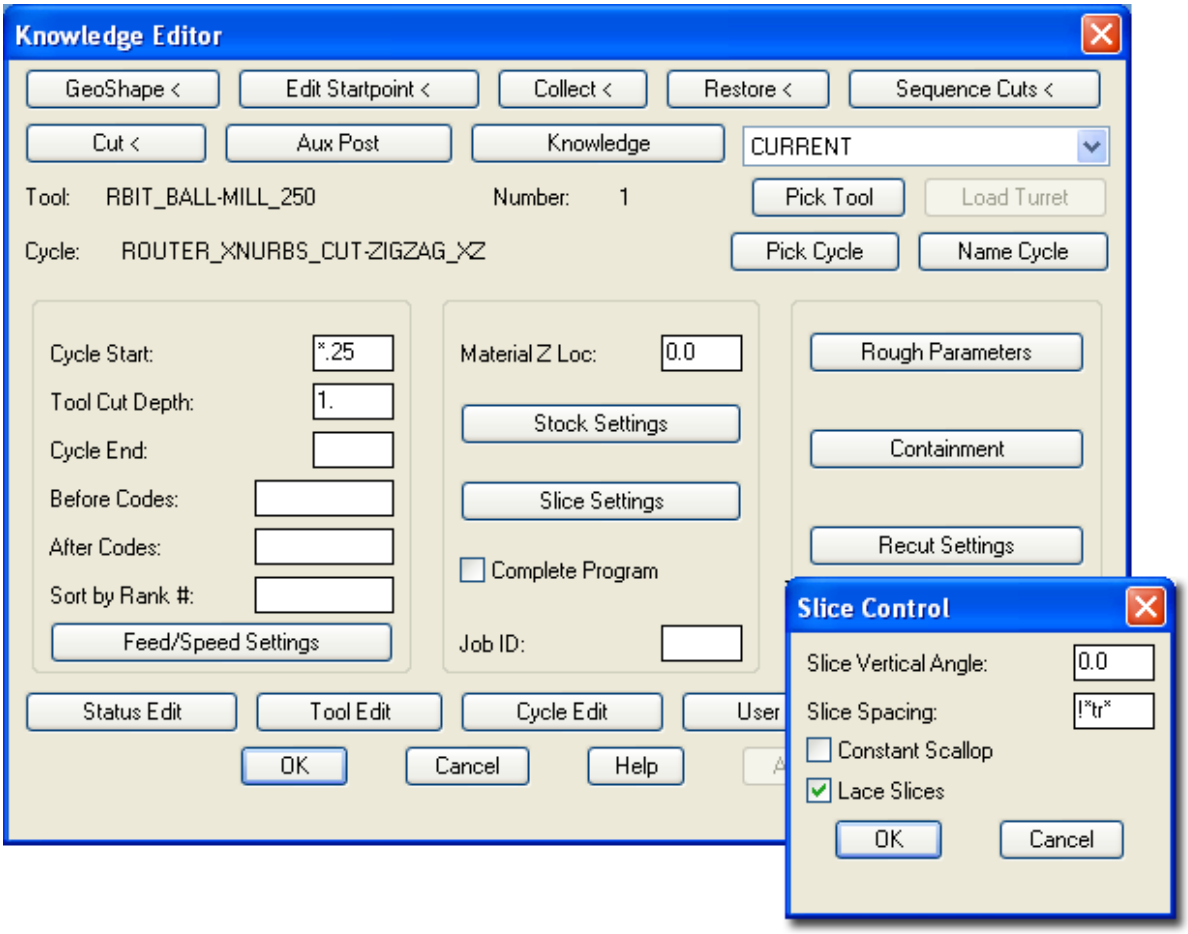
and you should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.5.12.1.1 Vertical Finishing at Angle 0.0

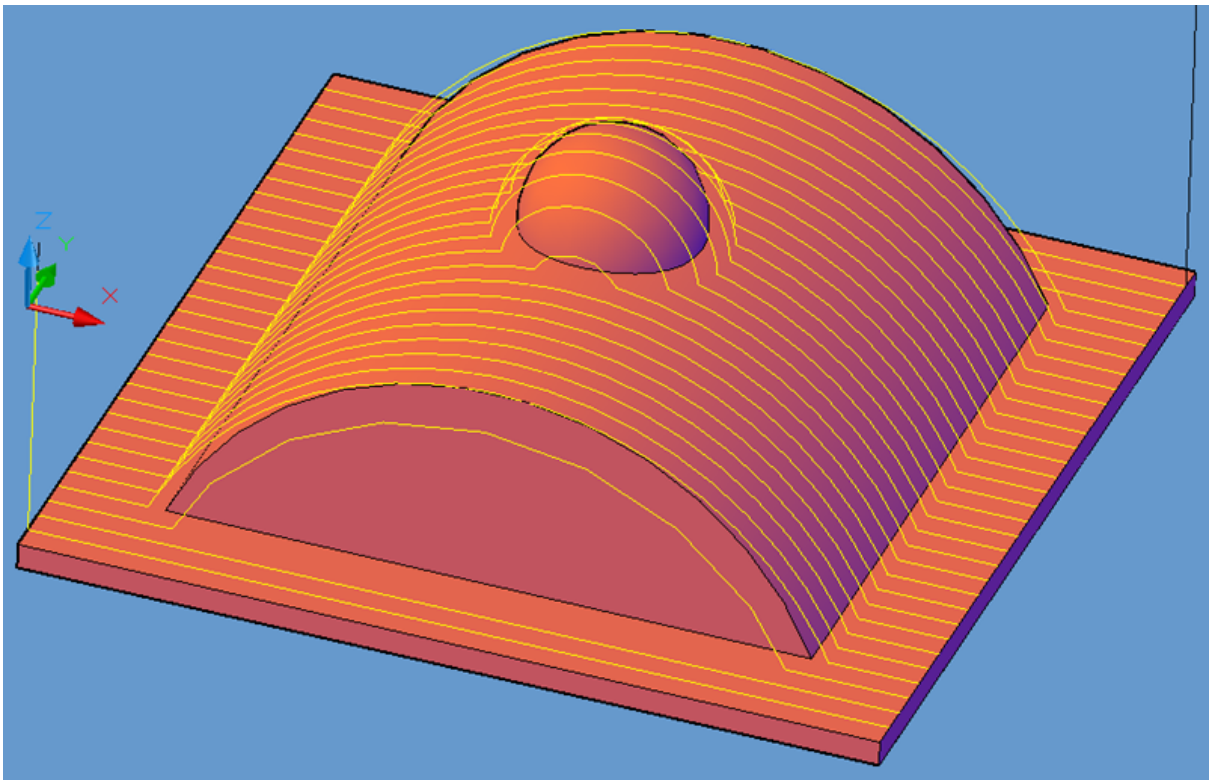
Using the MCADTPC.dwg file, select the XNURBS_CUT-ZIGZAG_XZ Cycle.
Select Slice Settings to be sure the Slice Vertical Angle is set to 0.0.



Press the Cut Button.

The output of these settings should produce a tool path like this:

© 2011 CIM-TECH.COM, Inc



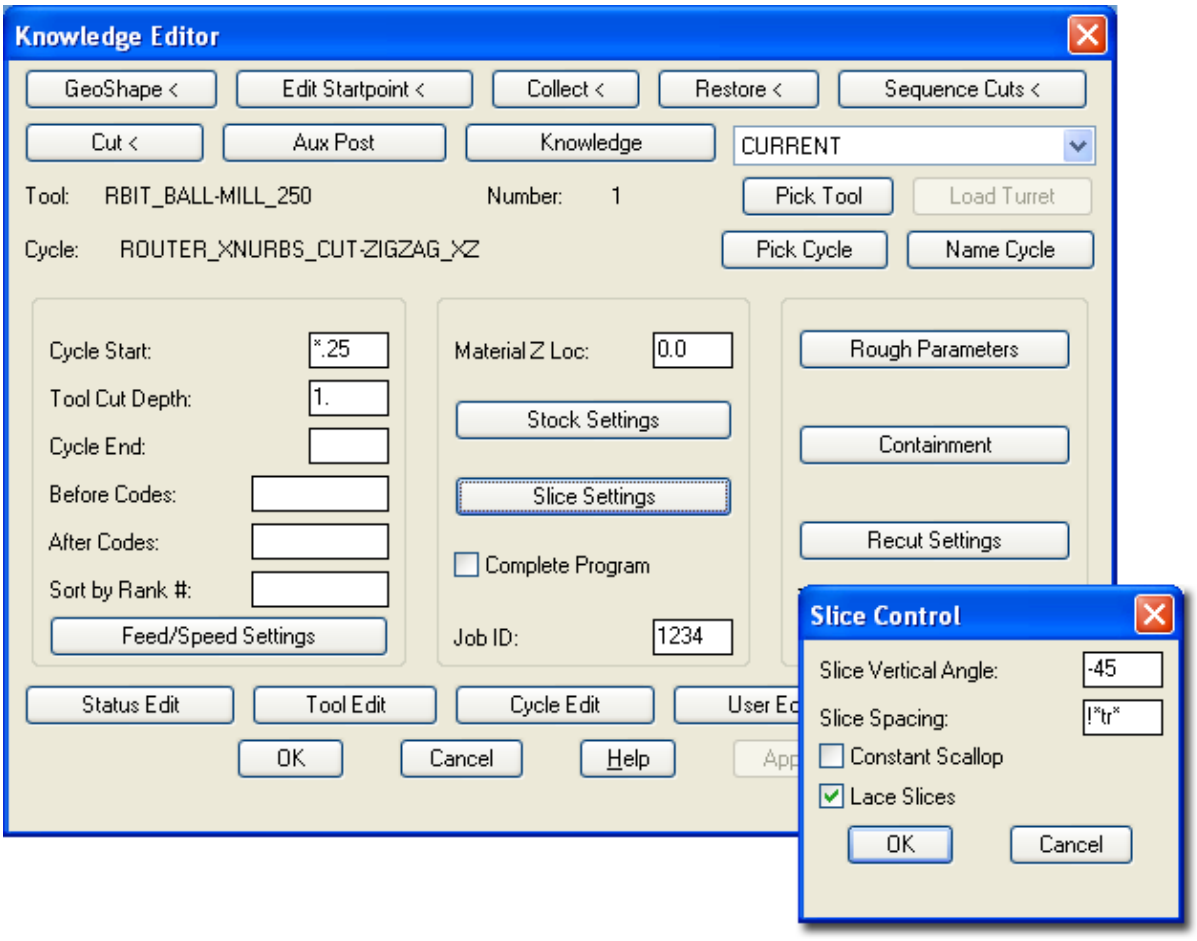
The settings used are finishing settings. There is no Stock Allowance used. There are no roughing passes, lead-in, or lead-out. Also Containment and Recutting are turned off. Each of those setting will be explained, just a little bit later in this document.

The main purpose here is that you understand how the XZ slice creates a basic tool path in the X and Z direction with step over in the Y.

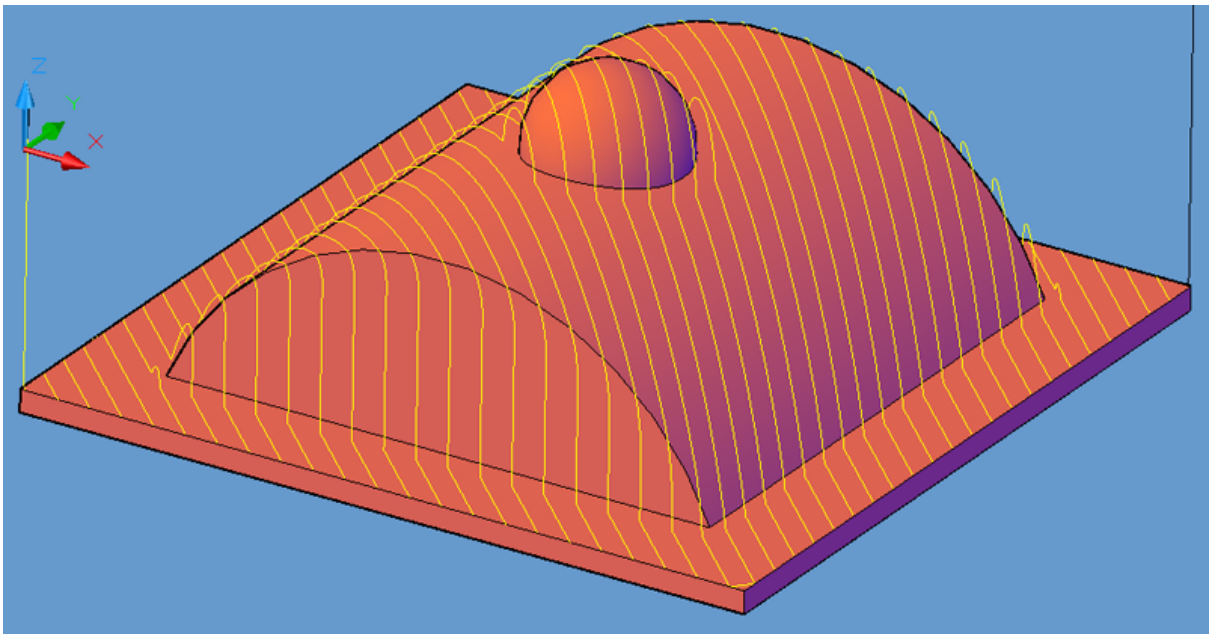
6.1.5.12.1.2 Vertical Finishing at Angle Minus 45

Using the MCADTPC.dwg file, select the XNURBS_CUT-ZIGZAG_XZ Cycle.
Select Slice Settings to be sure the Slice Vertical Angle is set to -45.

© 2011 CIM-TECH.COM, Inc



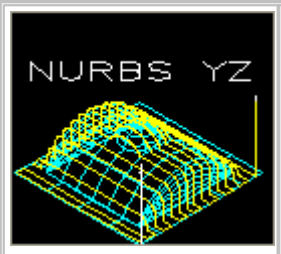
These settings should produce a tool path like this:



© 2011 CIM-TECH.COM, Inc

The only difference between this tool path and Vertical Finishing at Angle 0.0 is the setting for Slice Vertical Angle. Setting this to a value other than 0 produces slices on the specified angle.

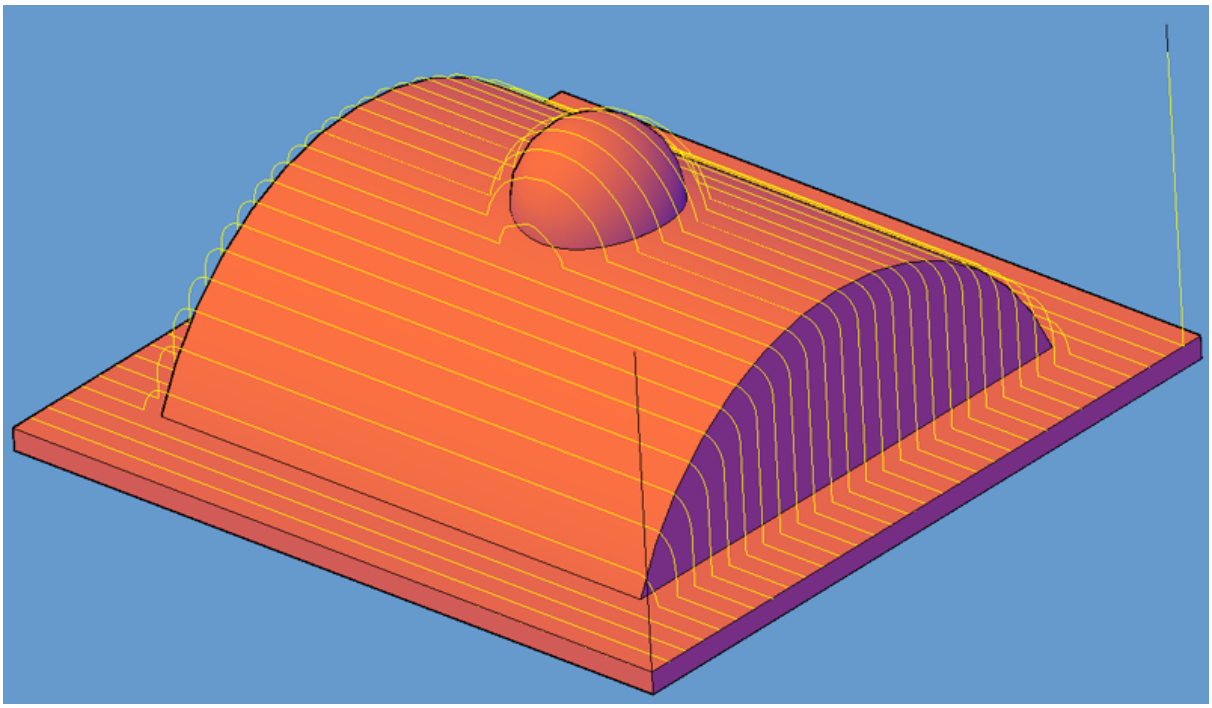
6.1.5.12.2 NURBS YZ



The NURBS YZ cycle is a Vertical Slicing cycle for cutting solids and surfaces inside of AutoCAD, or with the aid of the NCSurfer program. Tool paths will be generated in the Y direction as slices following the contour of the shape in Z. The slices can be made in one direction, or laced together to allow slicing in both X directions (positive and negative), forming a zig-zag motion of the tool.

Essentially the NURBS YZ cycle is the same as the NURBS XZ cycle with a Slicing Angle of 90°.

There are many parameters available to fine tune this cycle to perform many operations and these will be described throughout the NURBS section of this document.



Shown is a tool path created with the default settings of the NURBS YZ cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

© 2011 CIM-TECH.COM, Inc

```
Enter Surface Tolerance <0.00050000>:
```

Enter .003

```
Enter Surface Tolerance <0.00050000>: .003
```

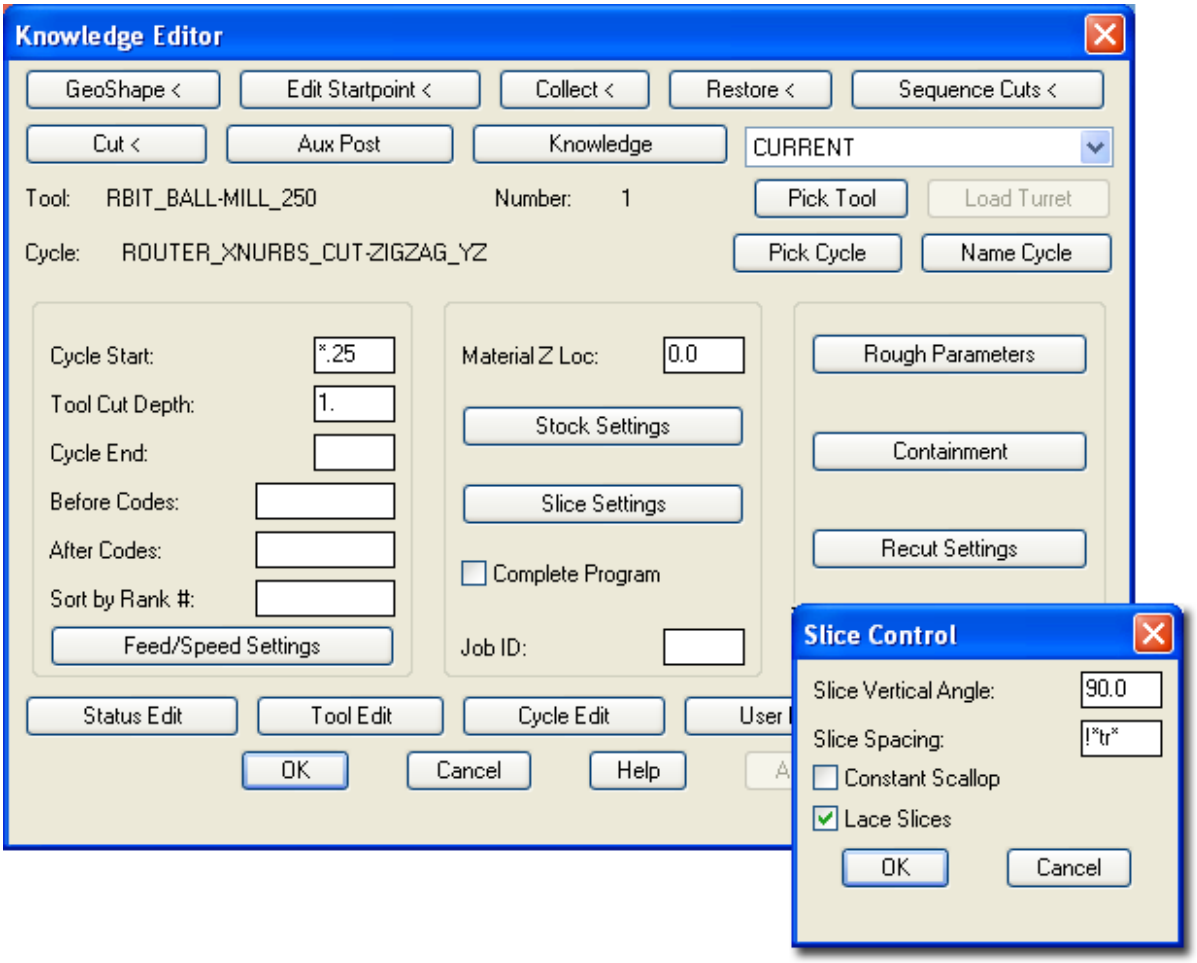
and you should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.5.12.2.1 Vertical Finishing in YZ

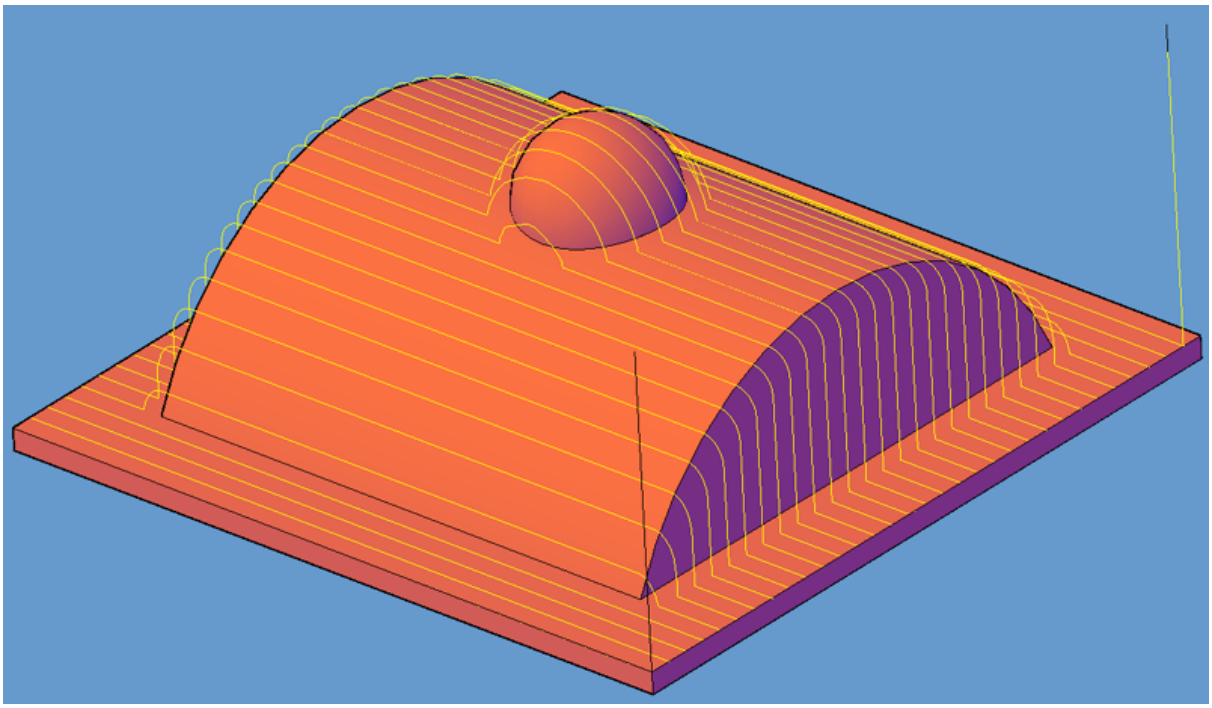
Using the MCADTPC.dwg file, select the XNURBS_CUT-ZIGZAG_YZ Cycle.
Select Slice Settings to be sure the Slice Vertical Angle is set to 90.0.



Press the Cut Button.

The output of these settings should produce a tool path like this:

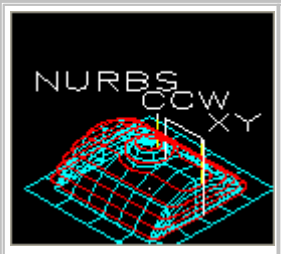
© 2011 CIM-TECH.COM, Inc



The settings used are finishing settings. There is no Stock Allowance used. There are no roughing passes, lead-in, or lead-out. Also Containment and Recutting are turned off. Each of those setting will be explained, just a little bit later in this document.

The main purpose here is that you understand how the YZ slice creates a basic tool path in the Y and Z direction with step over in X.

6.1.5.12.3 NURBS CCW XY

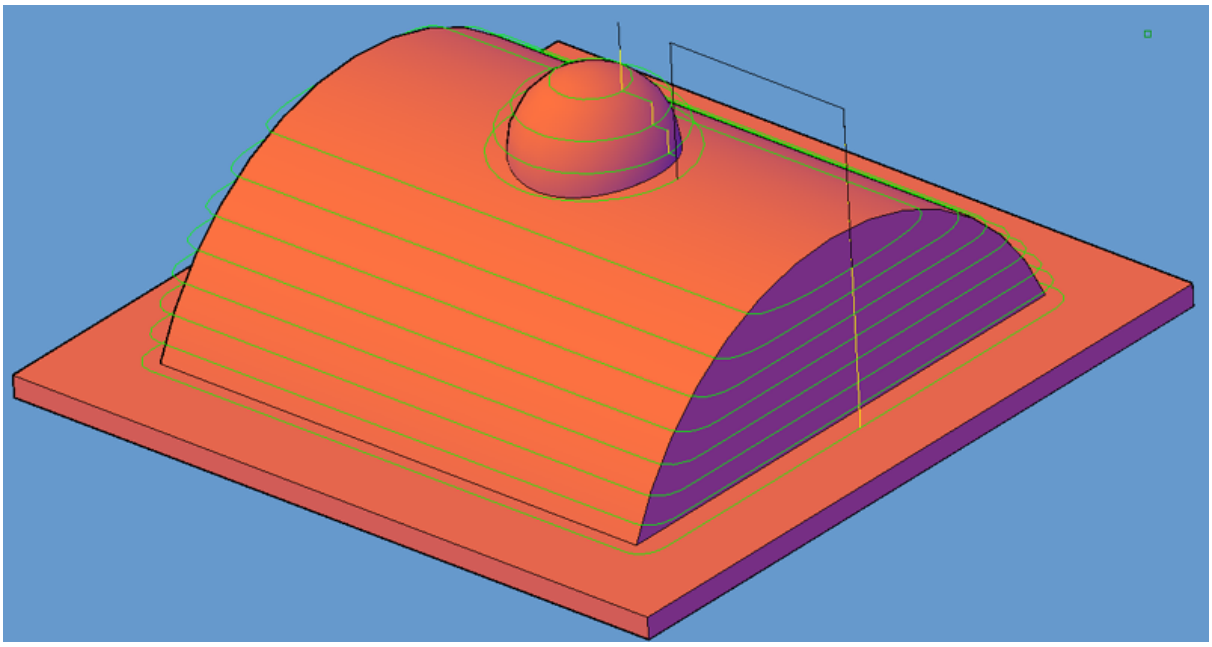


Nurbs CCW XY is a horizontal slicing cycle. Horizontal slicing will produce a tool path in the XY plane at each Z level, slicing horizontally instead of vertically in either X or Y. This is useful for some roughing and finishing operations where vertical slicing leaves an undesirable finish.

The tool path can be controlled in the Z plane between passes by setting the Z Level Completion option to either Yes or No.

Leads used on this cycle are linear by default with a straight plunge to the cut Z level, however, checking the leads option will allow arc lead-in and lead-out moves to be created for the beginning and end of the tool path. Each slice in between will still have a plunge move in Z.

© 2011 CIM-TECH.COM, Inc



Shown is a tool path created with the default settings of the NURBS CCW XY cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

Enter .005

```
Enter Surface Tolerance <0.00050000>: .003
```

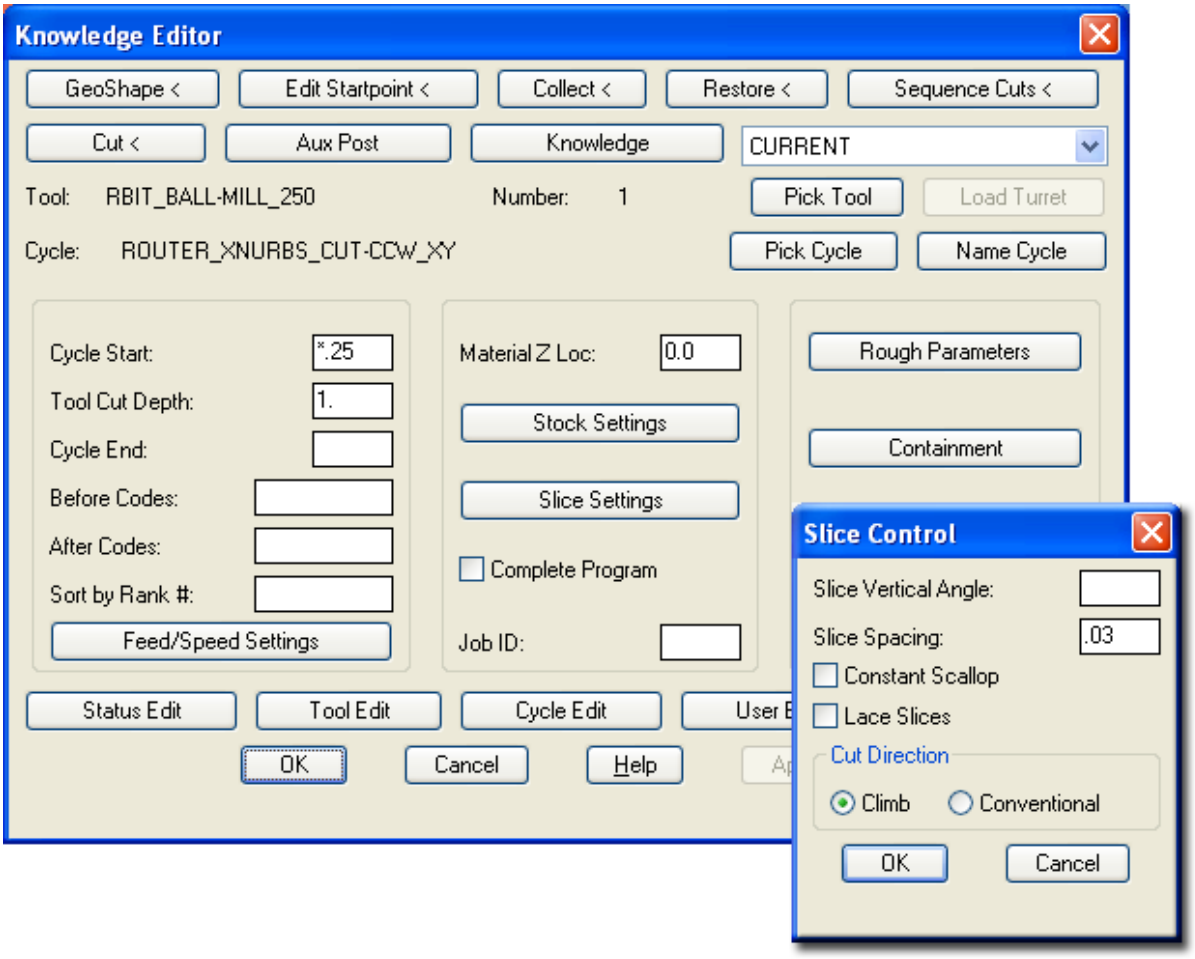
and you should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

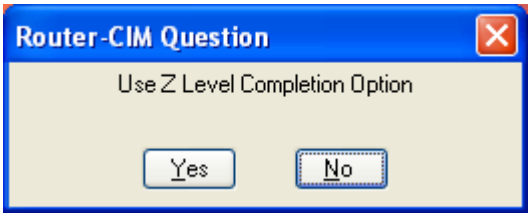
© 2011 CIM-TECH.COM, Inc

6.1.5.12.3.1 Horizontal Finishing XY

Select the XNURBS_CUT-CW_XY Cycle. Activate the Cut Button after duplicating the Knowledge Editor entries shown below using MCADTPC.DWG, to achieve similar results. The only parameter changed here is to set the Slice Spacing at .03 to show a more reasonable step over on the tool paths.



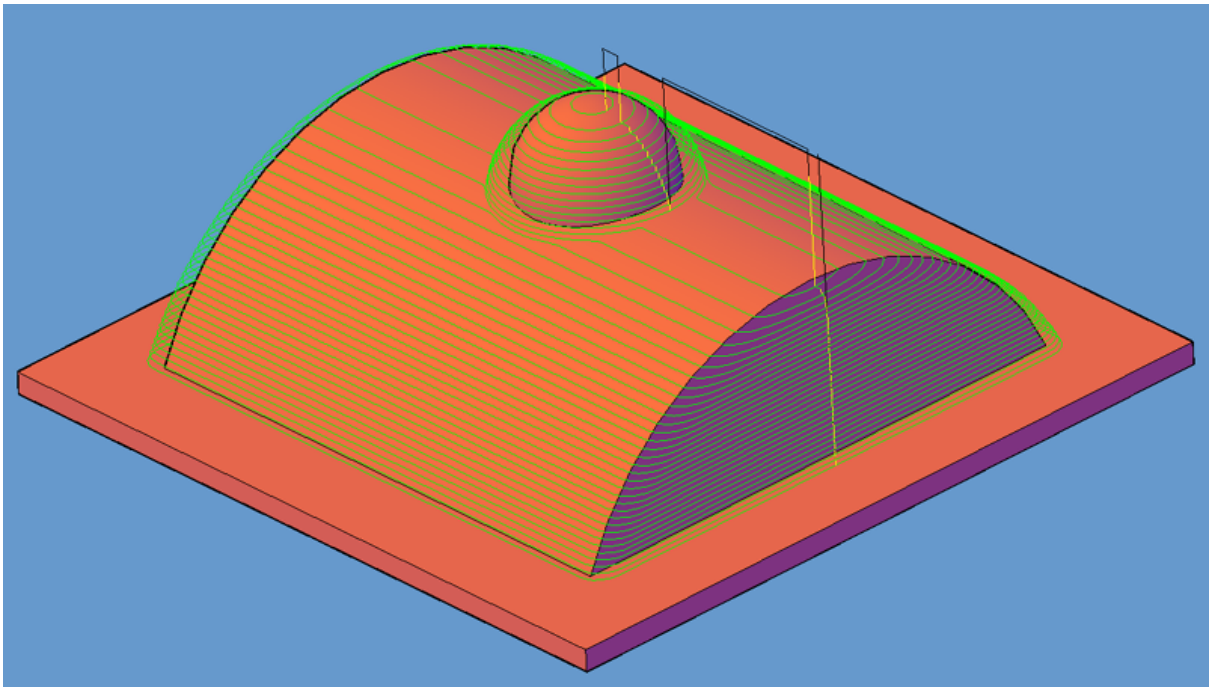
When Prompted for Z Level Completion:



Select NO to the "Z Level Completion Dialog".
YES is used when a Cavity or Core contains multiple hills and valleys. YES will cause the toolpath calculator to complete all toolpath(s) required at each Z level, retract, and then go down to the next Z level.
NO continues to cut downward until complete.

© 2011 CIM-TECH.COM, Inc

You should get a tool path like the following:

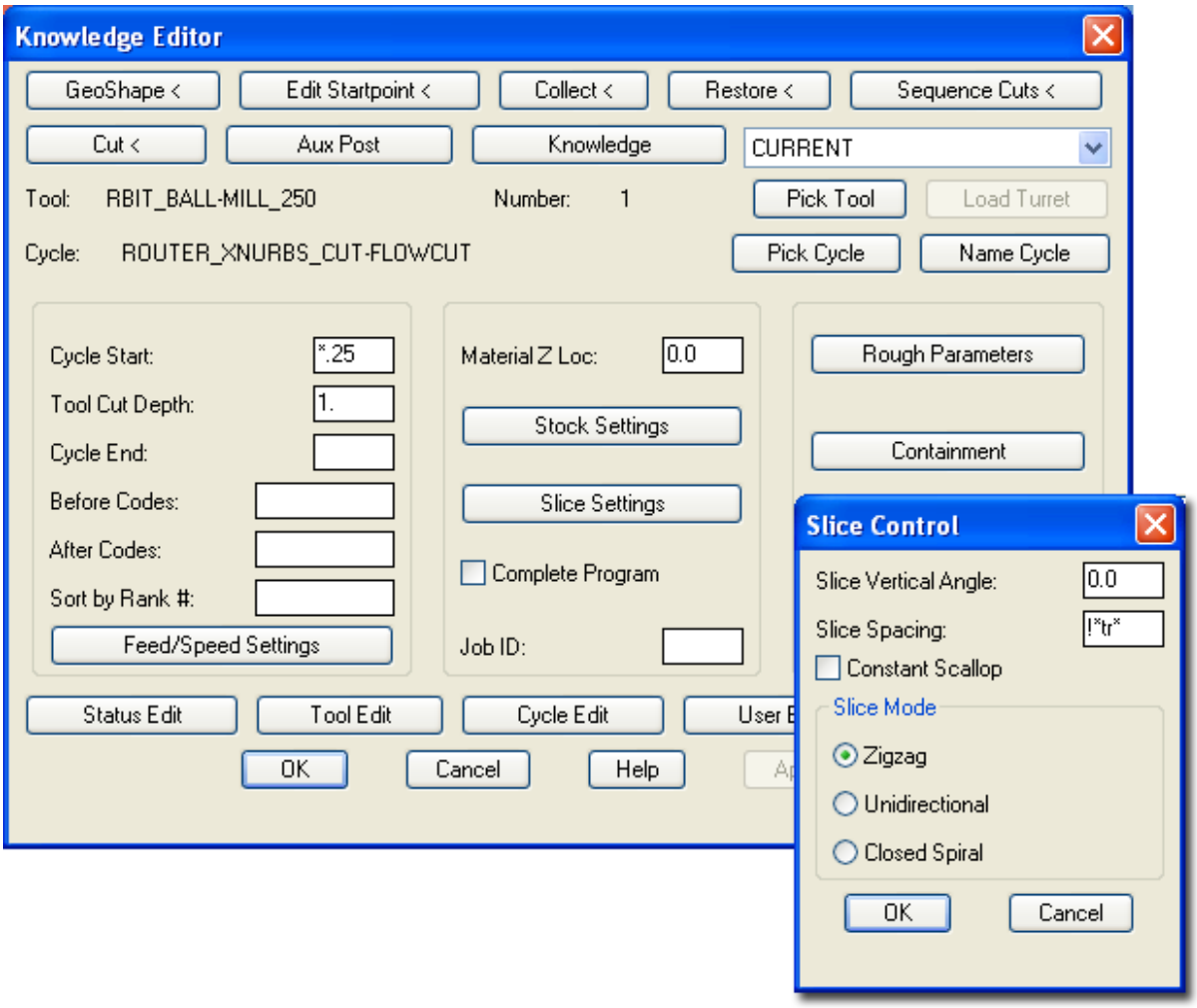


6.1.5.12.4 NURBS Flowcut

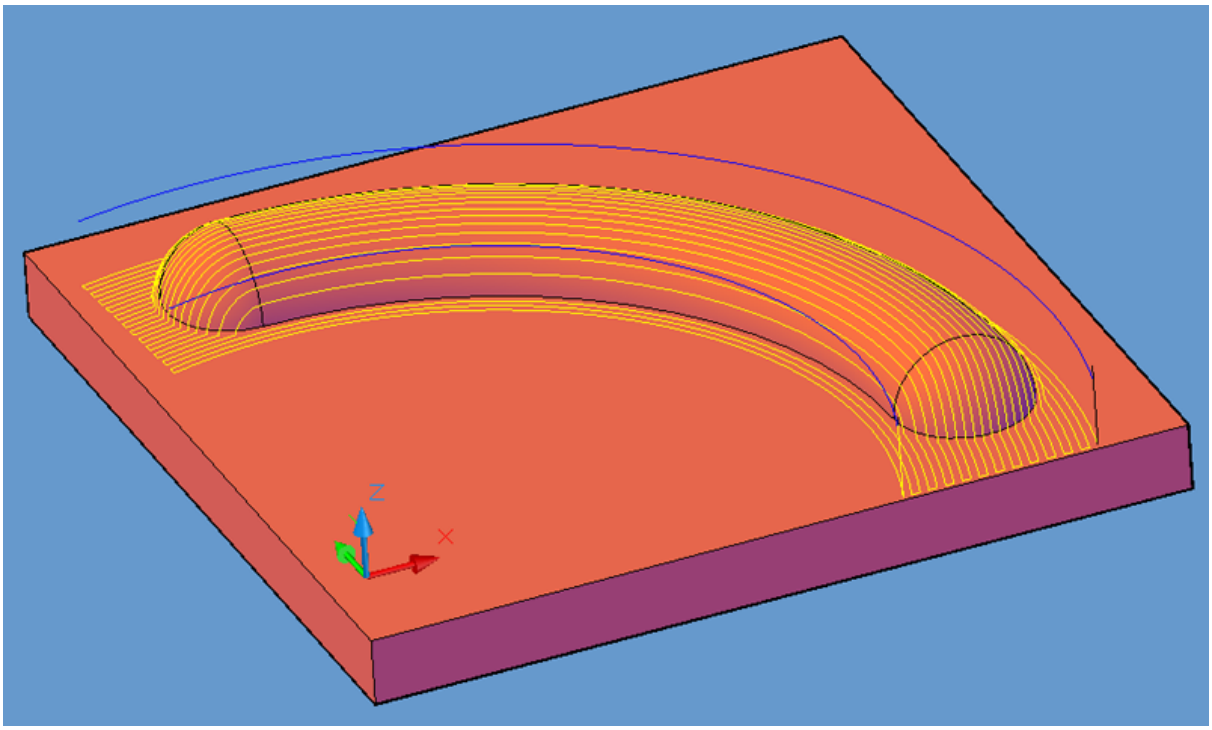


The Nurbs Flowcut cycle is a contour following slicing cycle. Once the shape has been created, you would typically create two 2D Polylines to be used as flow lines that follow the contour you wish to cut. The Nurbs cutter will then prompt you during the cut to select these flow lines and create a tool path that follows them, on the surface or solid below.
You may use open or closed Polylines for the flow lines and each method is described.
The two Polylines used must be going in the same direction, as the flow cutting tool path will follow these Polylines. This is easily accomplished by using Geoshape and Start Point Edit on both flow lines, which will allow you to make them both become counter-clockwise Polylines starting from the same location.

© 2011 CIM-TECH.COM, Inc



© 2011 CIM-TECH.COM, Inc



Shown is a tool path created with the default settings of the Nurbs Flowcut cycle on the MCADFLOW.dwg. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

Enter .005

```
Enter Surface Tolerance <0.00050000>: .003
```

and you should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

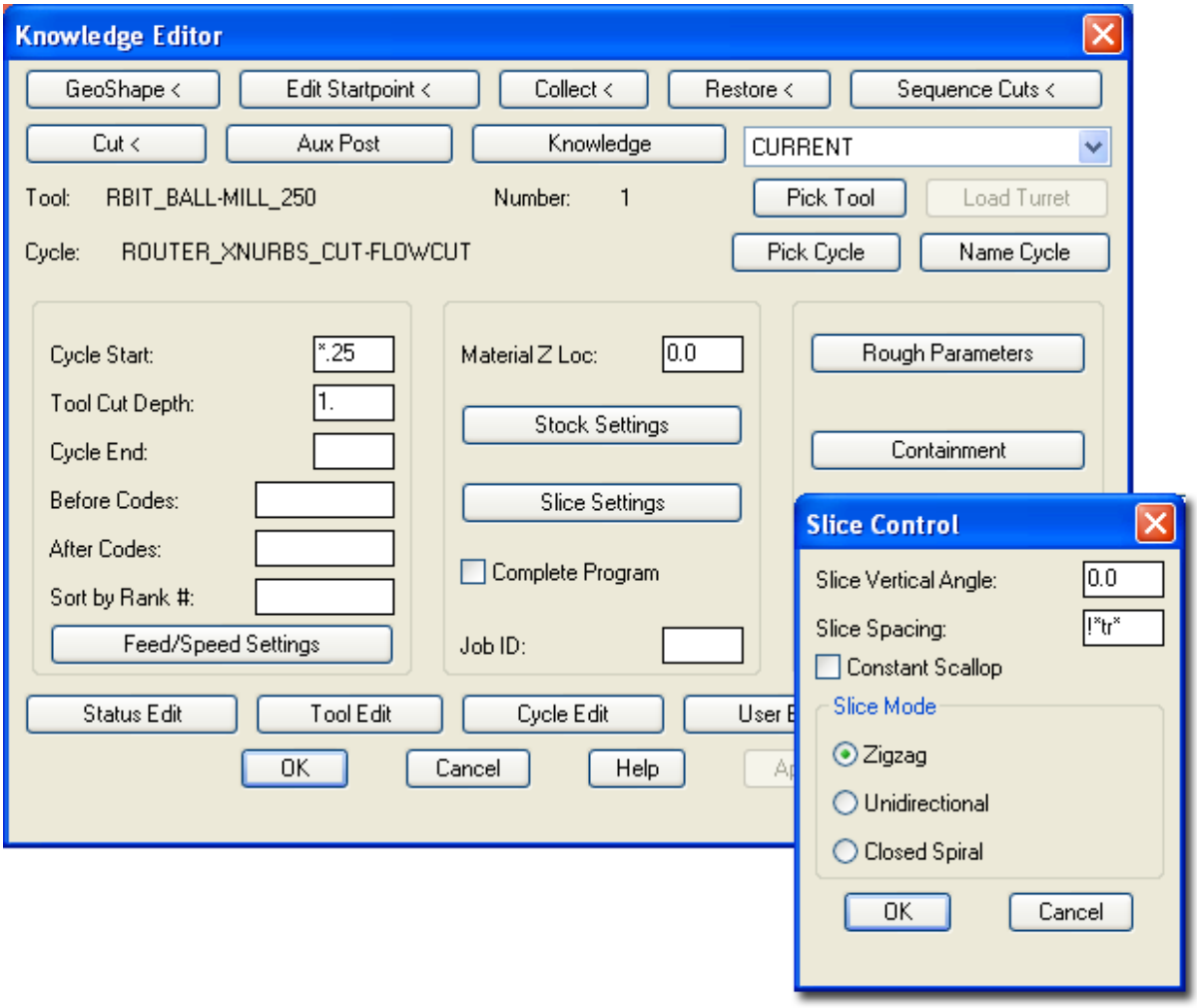
6.1.5.12.4.1 Open Polyline Cross Flow Cutting

Using open polylines that define the flow of your cuts is acceptable as long as both of the polylines are going in the same directions and are above the top of the part. You may use Geoshape and Start

© 2011 CIM-TECH.COM, Inc

Point Edit if you need to be sure of the direction of you flow lines.
During cut you will be prompted not only to select the surface or solid to cut, but for your flow lines as well.

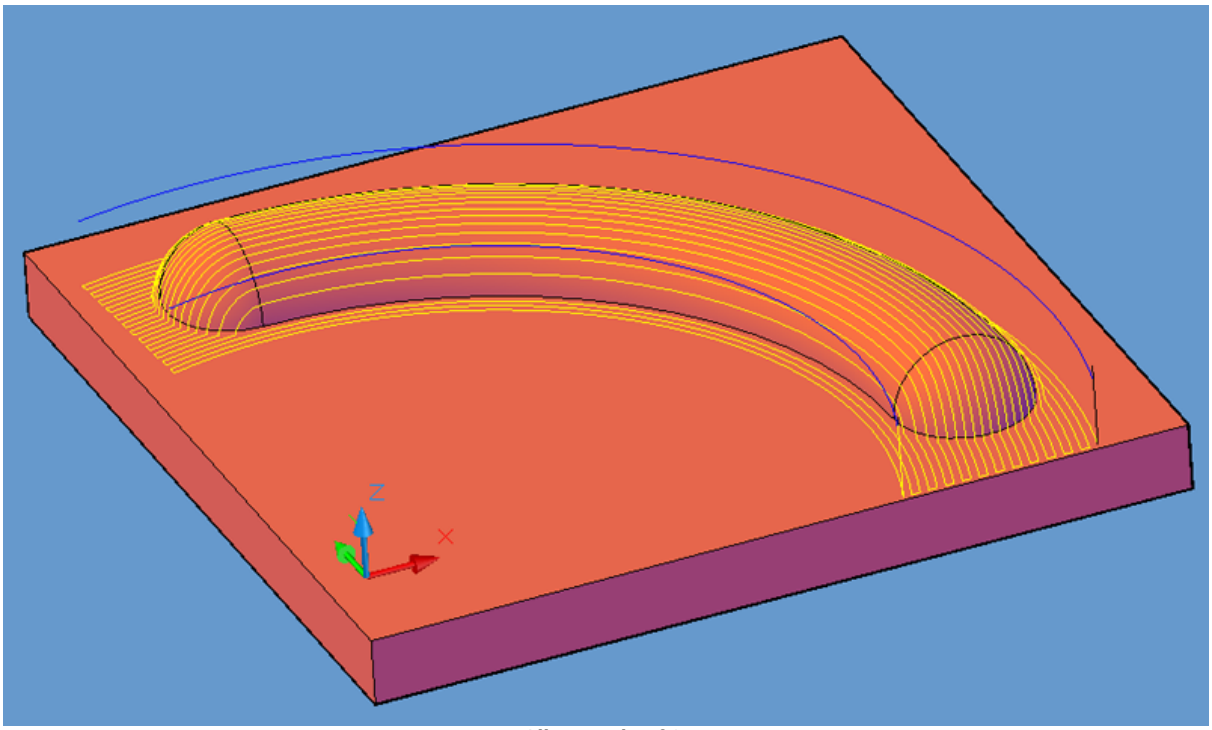
Using the MCADFLOW.dwg file, select the XNURBS_CUT-FLOWCUT Cycle.
Select Slice Settings to be sure the Slice Vertical Angle is set to 0.0, and the Slice Mode is Zigzag.



Press the Cut Button.

The output of these settings should produce a tool path like this:

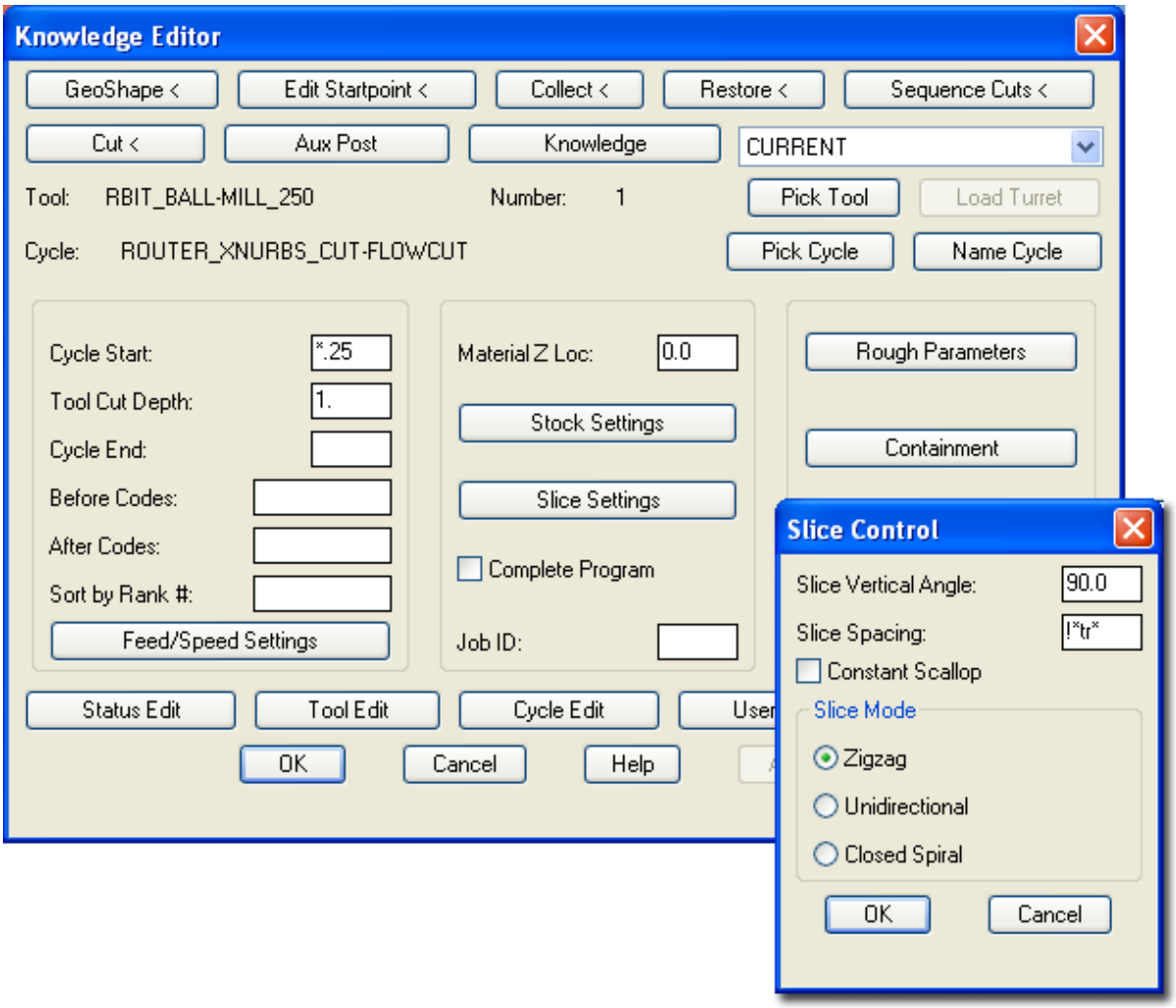
© 2011 CIM-TECH.COM, Inc



Slice Angle of 0.

Erase the previous tool path, then change the Slice Settings Vertical Angle to 90.0.

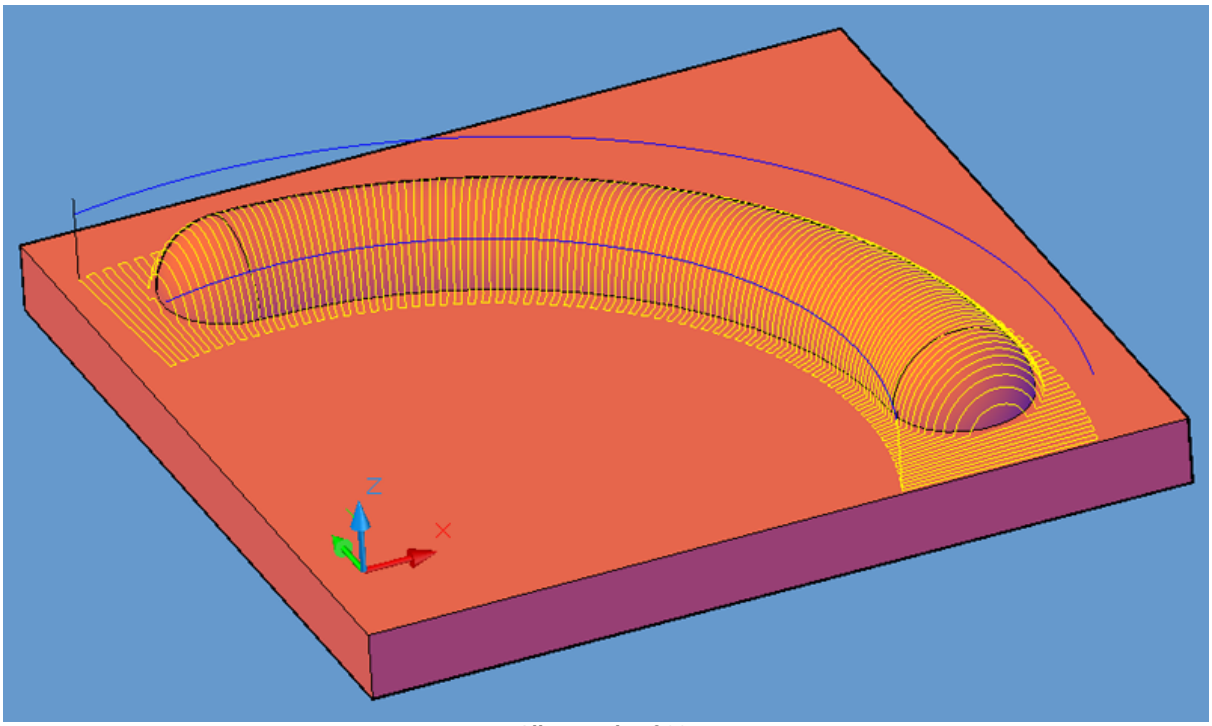
© 2011 CIM-TECH.COM, Inc



Press the Cut Button.

The output of these settings should produce a tool path like this:

© 2011 CIM-TECH.COM, Inc



Slice Angle of 90.

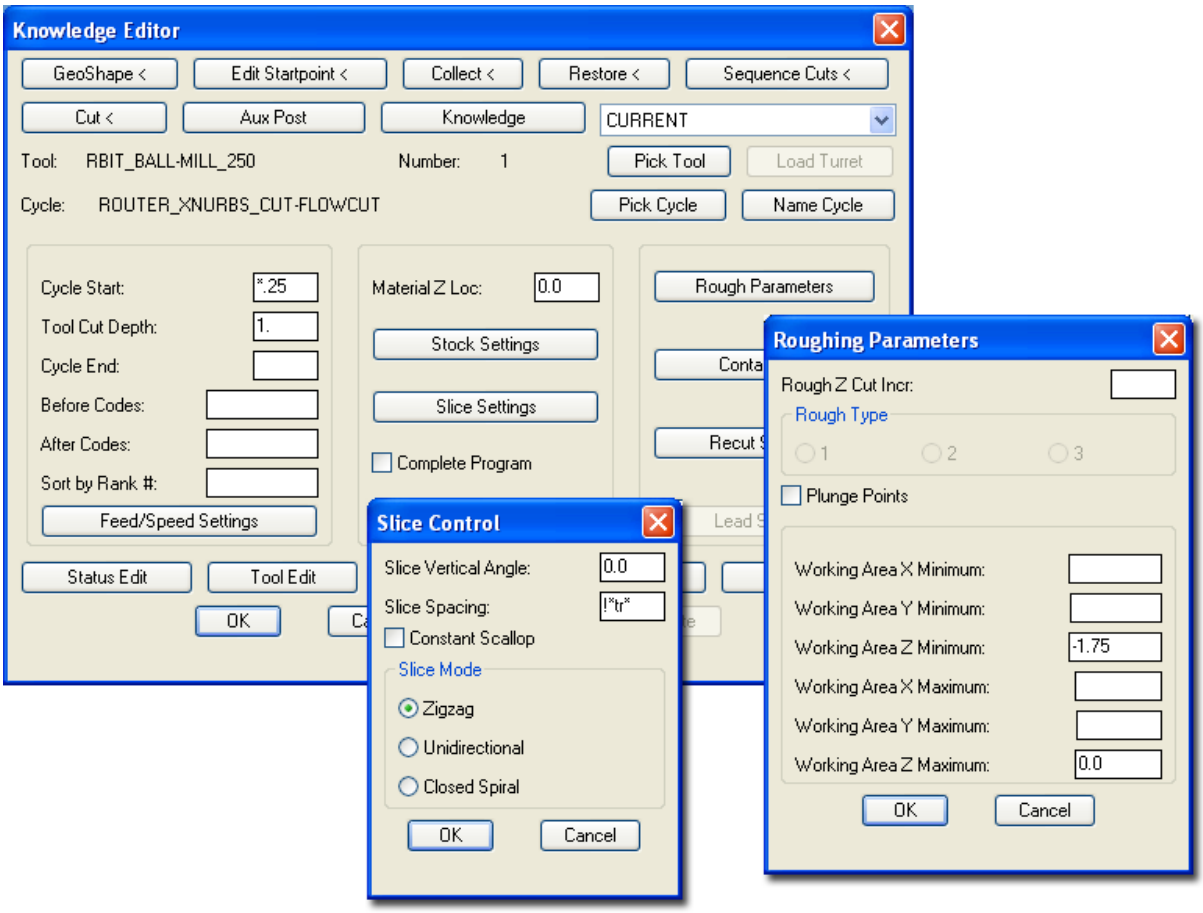
6.1.5.12.4.2 Closed Polyline Step Over Flow Cutting

Closed Polyline Flow Cutting is provided to satisfy cutting situations best handled by a continuous circulating toolpath, that is guided within an area defined by two closed polylines. Creating the two polylines is typically this is accomplished by creating a polyline from the edge of the solid and then offsetting it the appropriate amount for the tool. These closed polylines must be going in the same direction and the Start Points of both polylines must be aligned. These conditions are easily met by using Geoshape and Start Point Edit on the two polylines planned for use with this cycle. Geoshape will insure the polylines are going in the counterclockwise directions and Start Point Edit will allow you to align the two start points.
The Flow Cutting cycle will follow the direction of the two polylines, so you can use the command NCREVDIR to reverse the direction of the two polylines if desired.

Since there is nothing to contain the tool path when it meets the bottom of the shape, we will use the Nurbs Working Area settings to define the top and bottom of the shape so that the tool path does not go below the bottom of the part.

Open MCADCLFLOW.dwg and start Router-CIM. Change your knowledge settings to match the ones below:

© 2011 CIM-TECH.COM, Inc

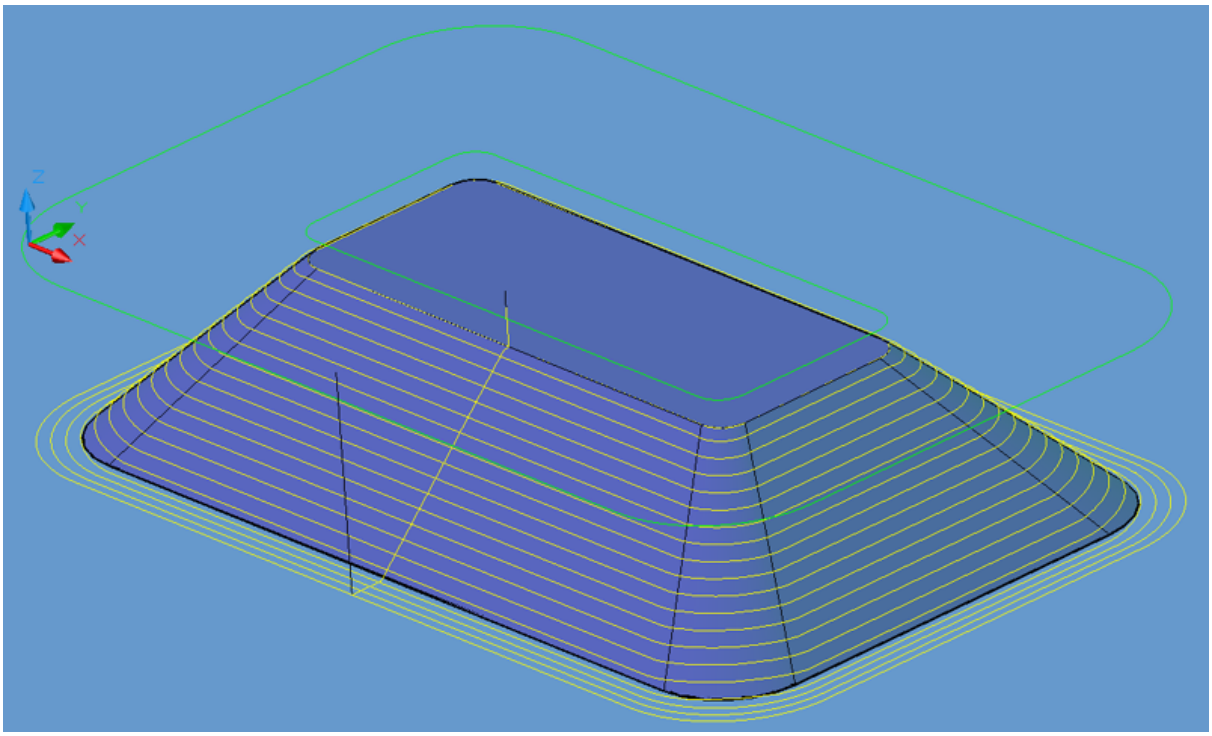


The Roughing Parameters window will allow you to set the Working Area minimum and maximum so that no tool paths cross beyond these areas. Set the Z Maximum to 0.0 which is the top of the part and the Z Minimum to -1.75 which is the bottom of the part and then no tool paths will fall below the part.

Setting the Slice Mode to Zigzag will allow the tool path to make a step down between each pass without lifting up to the Safety Plane between passes.

Change your settings to match the ones here and then make a cut, selecting the solid as the shape to cut and the two green polylines as the Flow Lines. If prompted to redefine the surfaces, answer Yes. Your results should look like the one below:

© 2011 CIM-TECH.COM, Inc



This tool path will step down in Z for each pass without lifting the tool up to the Safety Plane, and continue cutting the solid until it gets to the bottom of the shape at -1.75. Then, since it is still cutting until it meets the larger of the two closed flow lines, it will make passes in XY. Making a smaller outside flow line, that matches the shape would avoid the extra passes at the bottom.

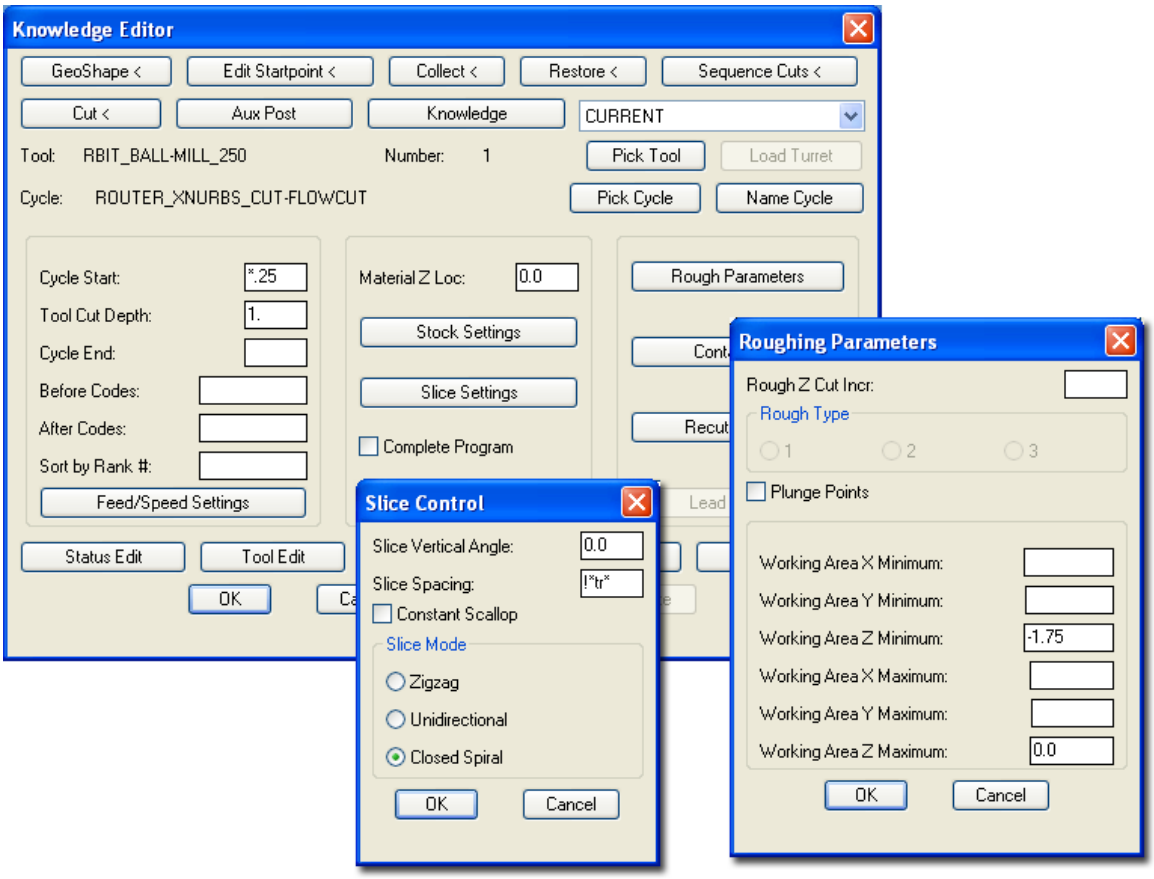
6.1.5.12.4.3 Closed Polyline Spiral Flow Cutting

Closed Polyline Flow Cutting is provided to satisfy cutting situations best handled by a continuous spiraling toolpath, that is guided within an area defined by two closed polylines. Creating the two polylines is typically this is accomplished by creating a polyline from the edge of the solid and then offsetting it the appropriate amount for the tool. These closed polylines must be going in the same direction and the Start Points of both polylines must be aligned. These conditions are easily met by using Geoshape and Start Point Edit on the two polylines planned for use with this cycle. Geoshape will insure the polylines are going in the counterclockwise directions and Start Point Edit will allow you to align the two start points.
The Flow Cutting cycle will follow the direction of the two polylines, so you can use the command NCREVDIR to reverse the direction of the two polylines if desired.

Since there is nothing to contain the tool path when it meets the bottom of the shape, we will use the Nurbs Working Area settings to define the top and bottom of the shape so that the tool path does not go below the bottom of the part.

Open MCADCLFLOW.dwg and start Router-CIM. Change your knowledge settings to match the ones below:

© 2011 CIM-TECH.COM, Inc

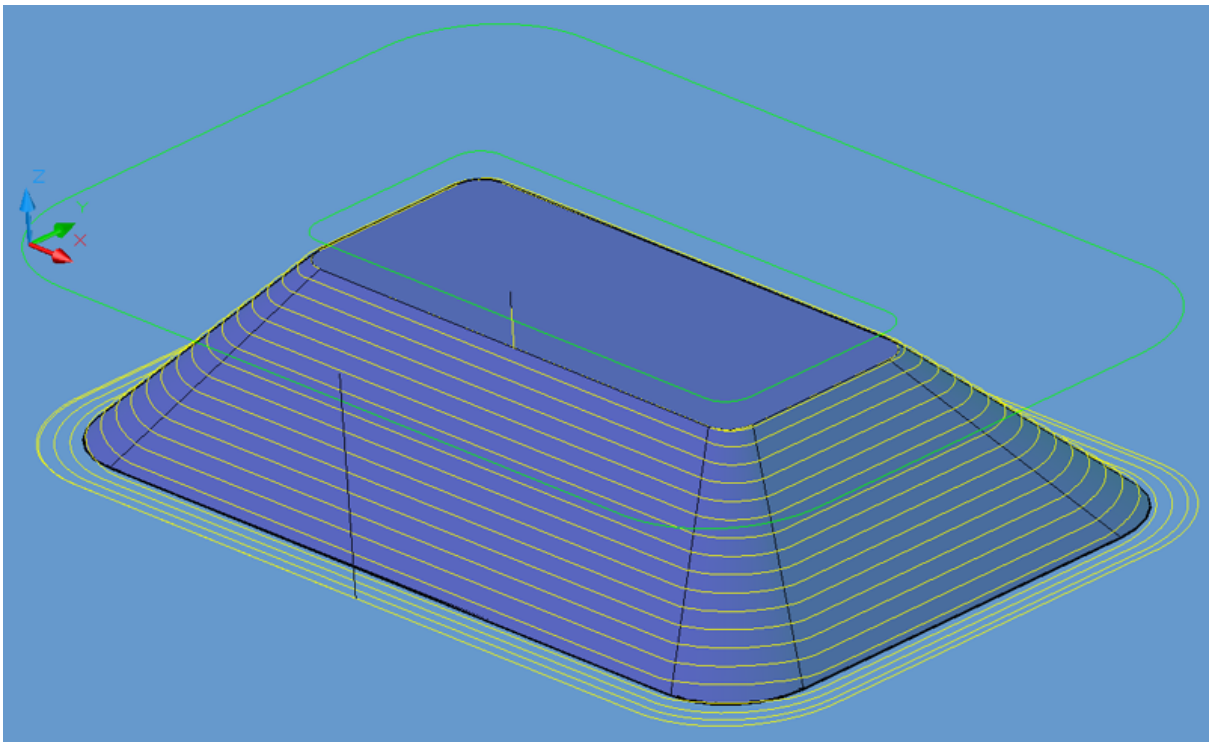


The Roughing Parameters window will allow you to set the Working Area minimum and maximum so that no tool paths cross beyond these areas. Set the Z Maximum to 0.0 which is the top of the part and the Z Minimum to -1.75 which is the bottom of the part and then no tool paths will fall below the part.

Setting the Slice Mode to Closed Spiral will allow the tool path to make a constant spiral pass without lifting up to the Safety Plane until it reaches the end of the path.

Change your settings to match the ones here and then make a cut, selecting the solid as the shape to cut and the two green polylines as the Flow Lines. If prompted to redefine the surfaces, answer Yes. Your results should look like the one below:

© 2011 CIM-TECH.COM, Inc



This tool path starts at the top of the part, on the boundary of the inner flow line, and makes one tool path constantly stepping down in Z while circling the part until it reaches the minimum Z depth set in the Roughing Parameters and then continues at that depth until it reaches the outside flow line.

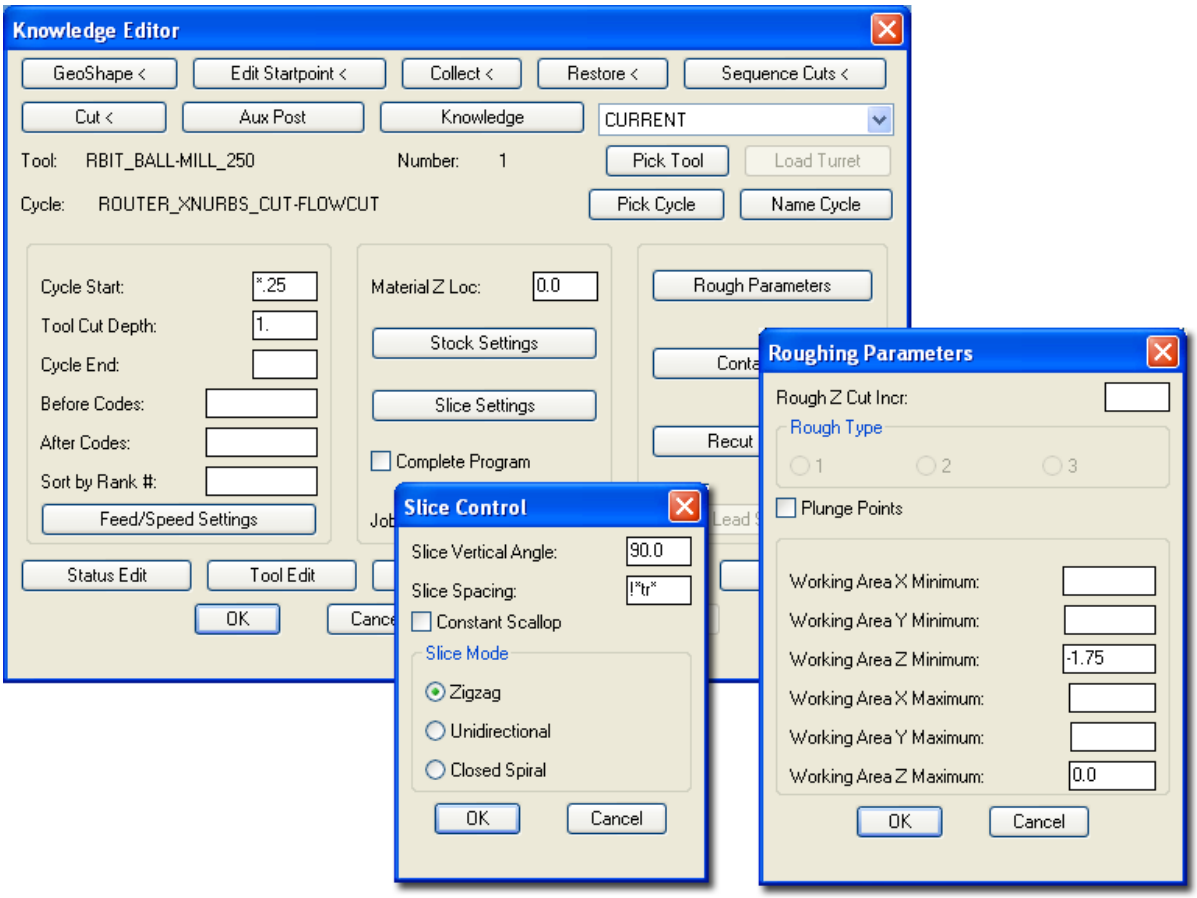
6.1.5.12.4.4 Closed Polyline Cross Flow Cutting

Closed Polyline Flow Cutting is provided to satisfy cutting situations best handled by a toolpath cutting at 90° to the flow lines provided on the part. Creating the two polylines is typically this is accomplished by creating a polyline from the edge of the solid and then offsetting it the appropriate amount for the tool. These closed polylines must be going in the same direction and the Start Points of both polylines must be aligned. These conditions are easily met by using Geoshape and Start Point Edit on the two polylines planned for use with this cycle. Geoshape will insure the polylines are going in the counterclockwise directions and Start Point Edit will allow you to align the two start points.
The Flow Cutting cycle will cut perpendicular to the flow lines, but cutting in the direction of the two polylines, so you can use the command NCREVDIR to reverse the direction of the two polylines if desired.

Since there is nothing to contain the tool path when it meets the bottom of the shape, we will use the Nurbs Working Area settings to define the top and bottom of the shape so that the tool path does not go below the bottom of the part.

Open MCADCLFLOW.dwg and start Router-CIM. Change your knowledge settings to match the ones below:

© 2011 CIM-TECH.COM, Inc

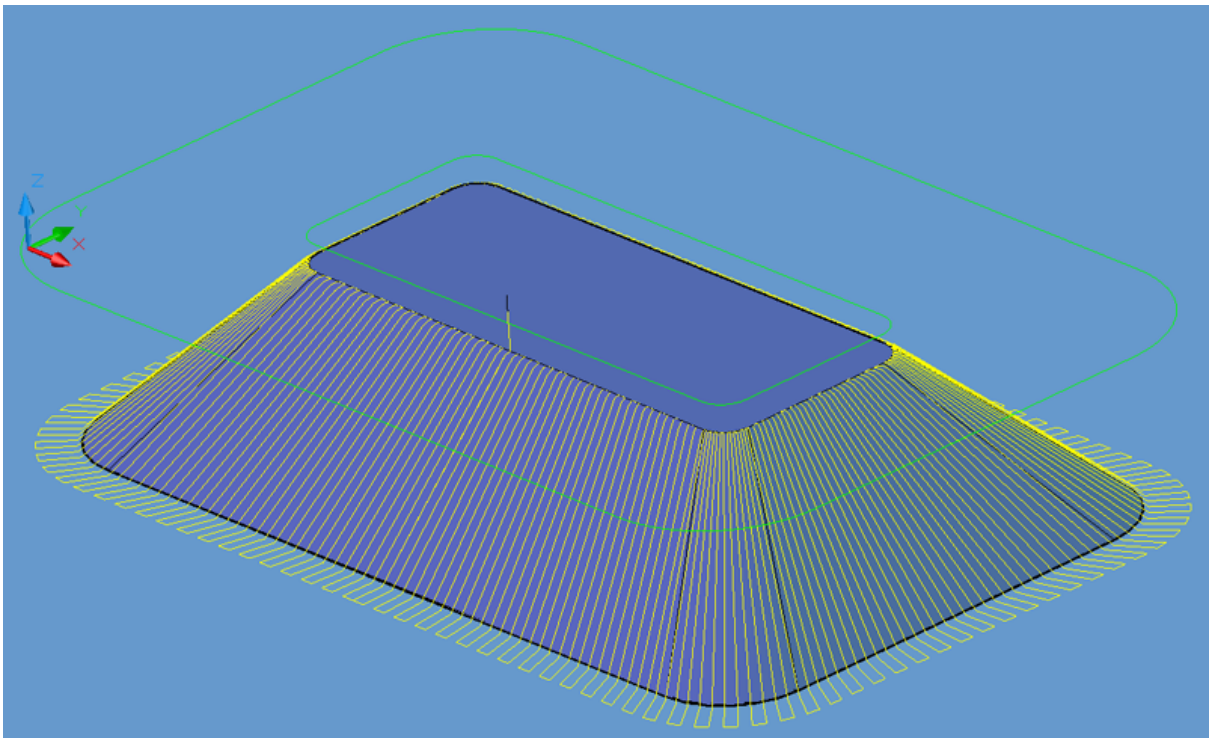


The Roughing Parameters window will allow you to set the Working Area minimum and maximum so that no tool paths cross beyond these areas. Set the Z Maximum to 0.0 which is the top of the part and the Z Minimum to -1.75 which is the bottom of the part and then no tool paths will fall below the part.

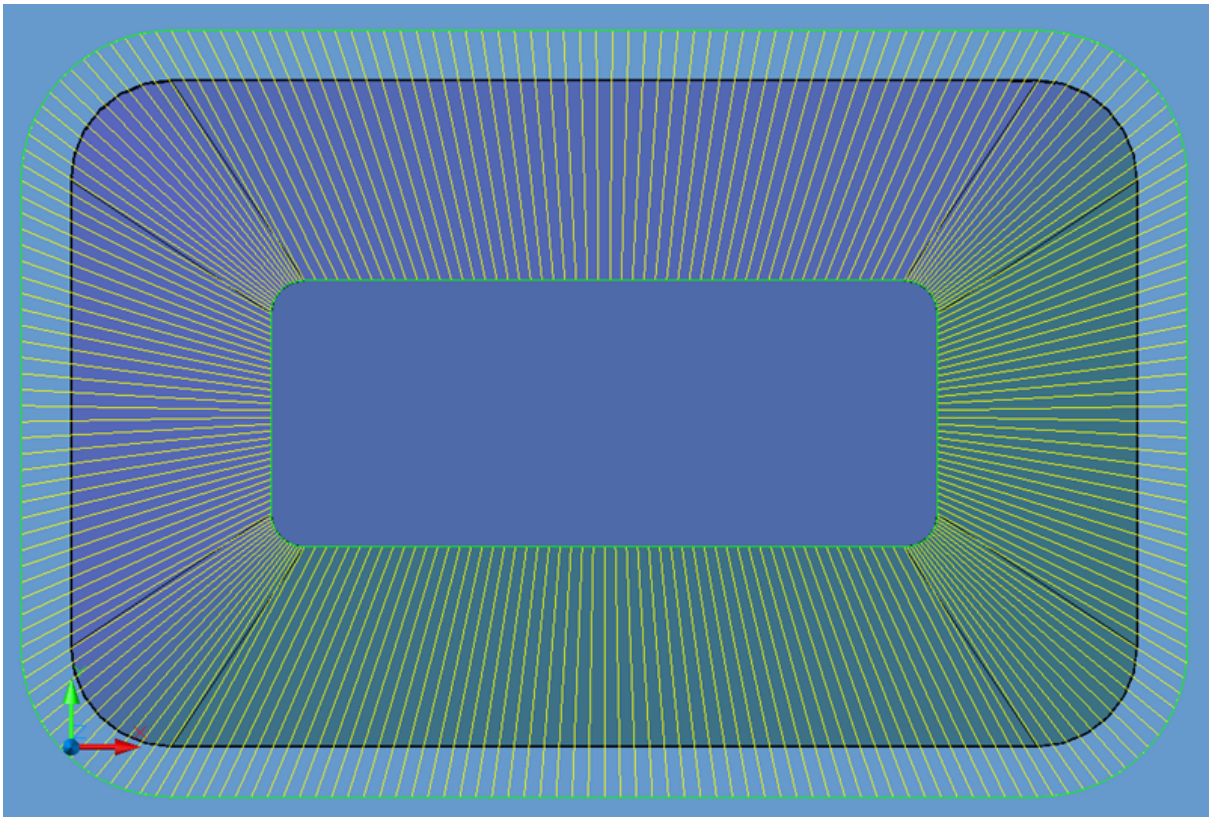
Setting the Slice Mode to Zigzag and the Slice Vertical Angle will allow the tool path to make a evenly spaced tool paths without lifting up to the Safety Plane until it reaches the end of the path.

Change your settings to match the ones here and then make a cut, selecting the solid as the shape to cut and the two green polylines as the Flow Lines. If prompted to redefine the surfaces, answer Yes. Your results should look like the one below:

© 2011 CIM-TECH.COM, Inc

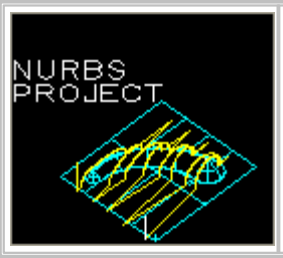


The result of this tool path is evenly spaced tool paths following the contours of the flow lines.



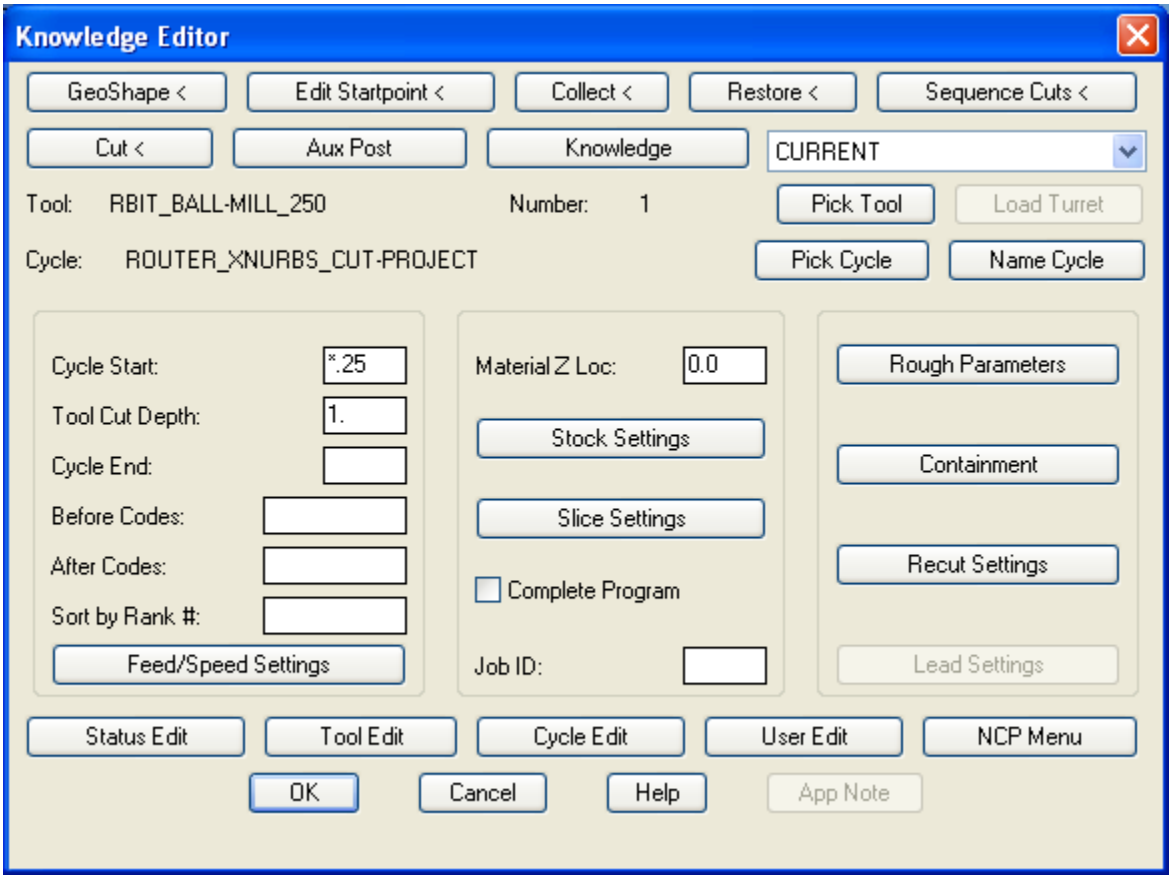
© 2011 CIM-TECH.COM, Inc

6.1.5.12.5 NURBS Project

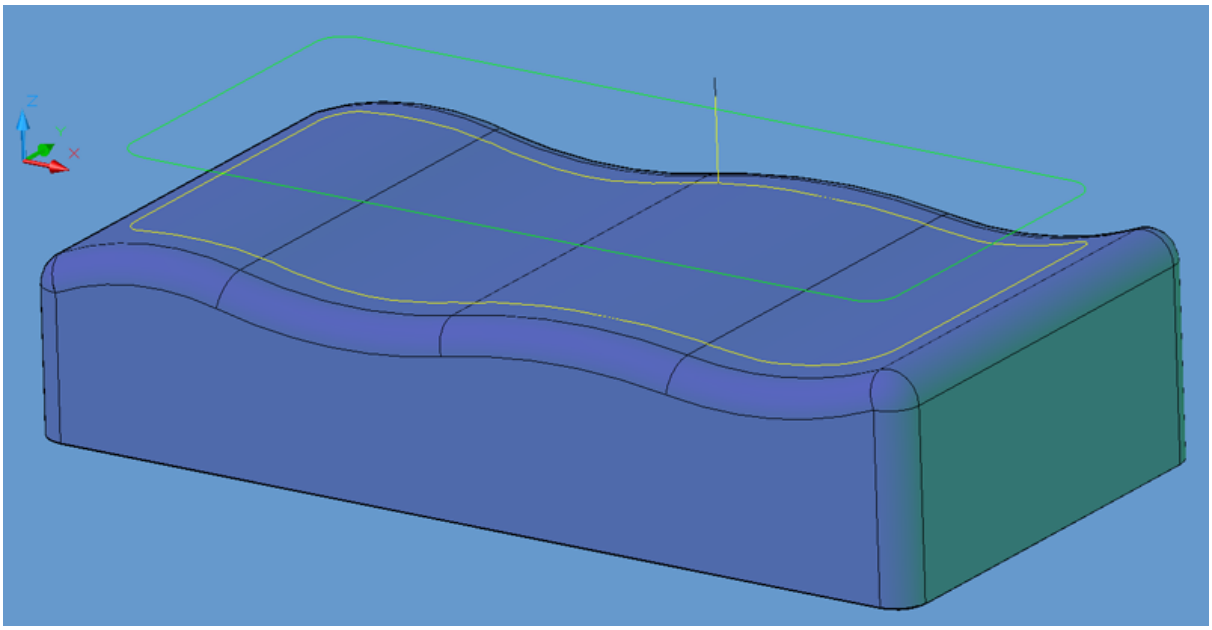


Project Cutting will create a tool path that is projected from a polyline(s) drawn above the part, down onto the surfaces or solid representing the part. Multiple polylines can be projected at one time. The resulting tool paths are vertical tool paths following the contour of the surface or solid.

The resulting tool path is on the top of the surface, but if you wish to move the tool path into the material, you can set a Stock Allowance of a Negative value to move the tool path down into the material.



© 2011 CIM-TECH.COM, Inc



Shown is a tool path created with the default settings of the Nurbs Project cycle. Similar results can be obtained by selecting the cycle and simply pressing the cut button and selecting the solid model. When prompted at the command line:

```
Enter Surface Tolerance <0.00050000>:
```

Enter .005

```
Enter Surface Tolerance <0.00050000>: .003
```

and you should see the same tool path as the picture above. The surface tolerance is the amount of deviation allowed for the tool path to follow the surface of the part as it breaks the moves up into small line segments for each pass. The smaller this number is, the more closely the tool path will follow the part contours, but the segments of the cut will be smaller, and thus produce more NC Code.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

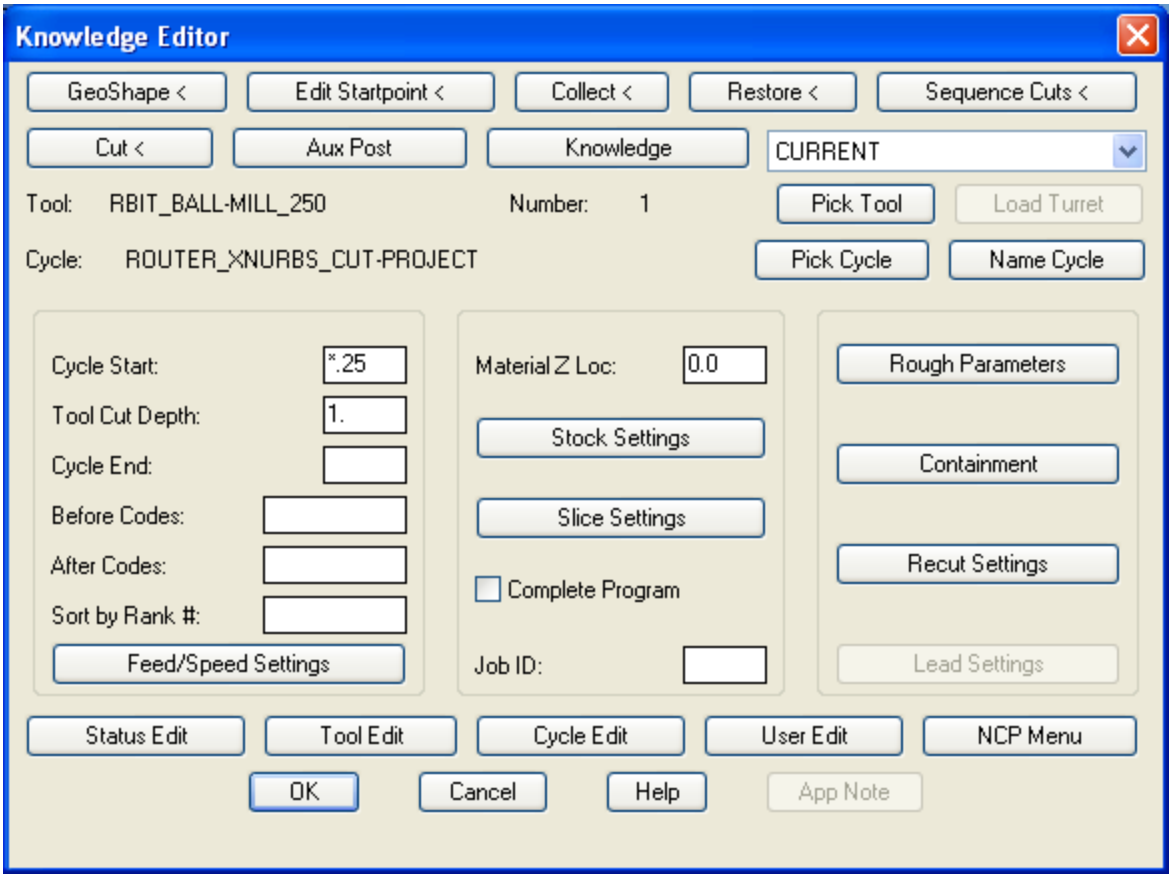
© 2011 CIM-TECH.COM, Inc

6.1.5.12.5.1 Polyline Project Cutting

Using Polyline Project Cutting, we can take several flat shapes and project them down onto an irregular surface or solid.

Open the drawing NurbsProject_1.dwg and start Router-CIM.

Using a .25 Ball End Mill and the default cycle settings, press CUT.



Select the blue solid when prompted to select the part.
When prompted to select the Surface Tolerance, enter .005.

```
Enter Surface Tolerance <0.00050000>: .005
```

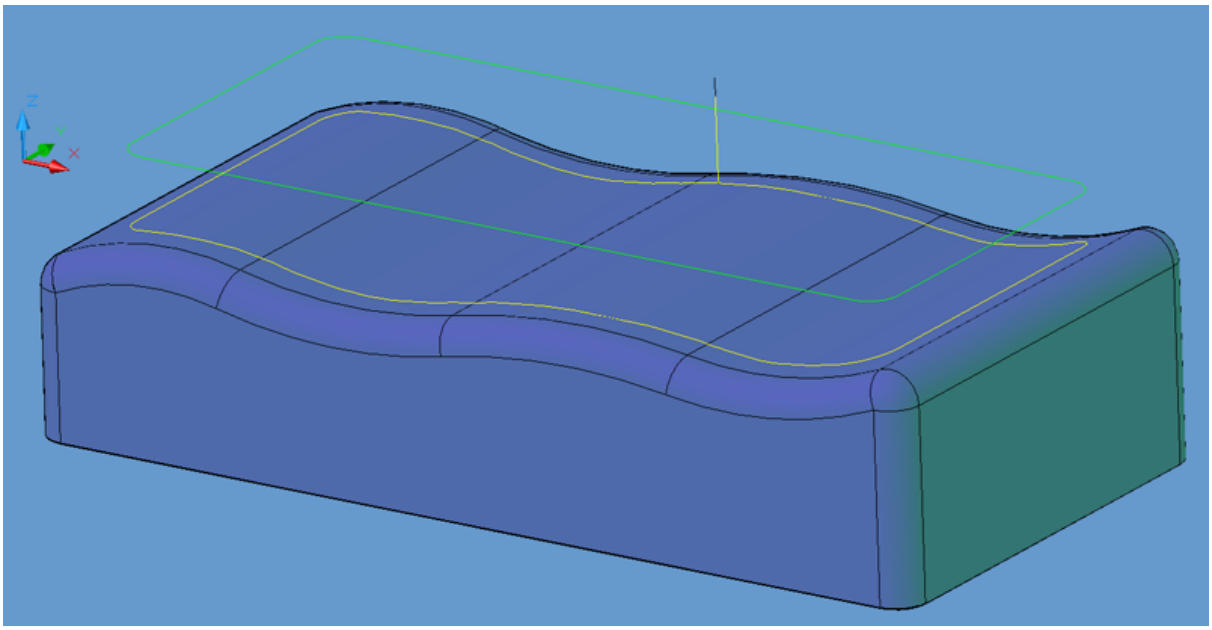
Next you will be prompted to select the Projection Polylines.

Select Projection Polylines/Cut Blocks:

Select the green polyline above the part.

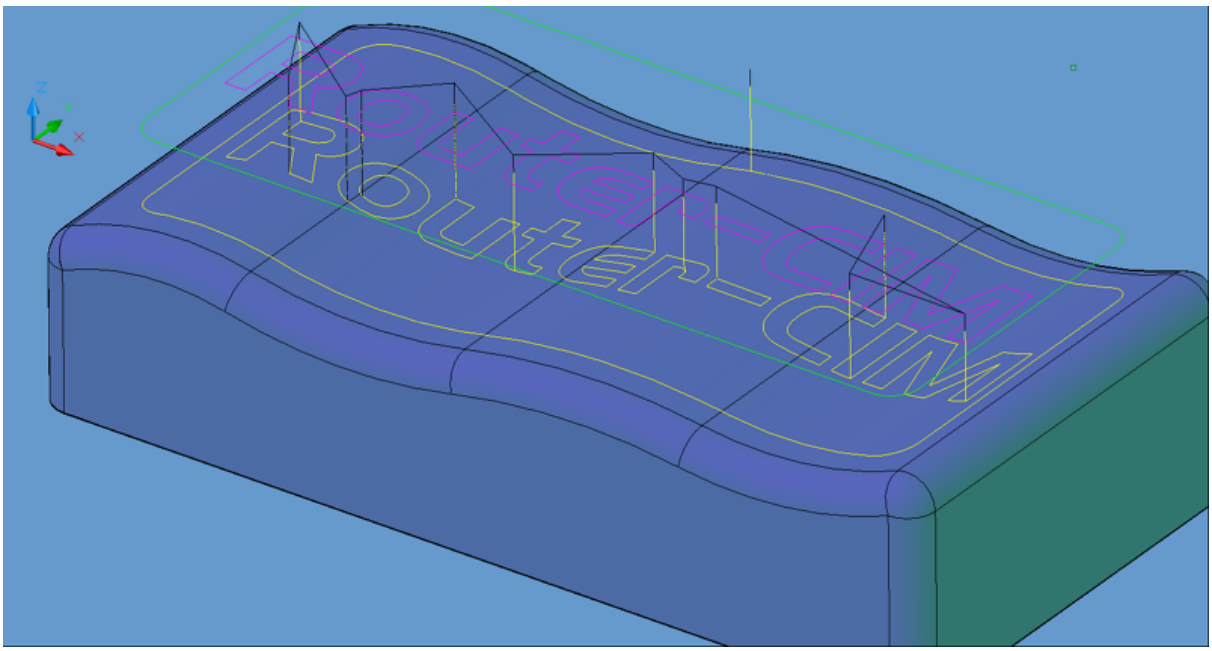
You should then see a tool path similar to the one shown here:

© 2011 CIM-TECH.COM, Inc



If you wish to make more cuts, Thaw layer ProjectText1, and then press Cut again, reselecting the solid and selecting the magenta text as the Projection Polylines.

You should end up with a tool path similar to this:



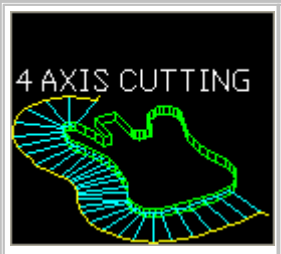
© 2011 CIM-TECH.COM, Inc

6.1.6 Specialty Cycles

There are some specialty cycles built into Router-CIM to accomplish specific tasks. These cycles are not useful on all machines. For instance there is a 4 Axis Saw cycle, and if your machine tool is not equipped with this type of spindle, then the cycle will not produce any suitable results.

Each of the specialty cycles are described in some detail to give and idea as to how the cycle works and when it is useful.

6.1.6.1 4 Axis Cutting

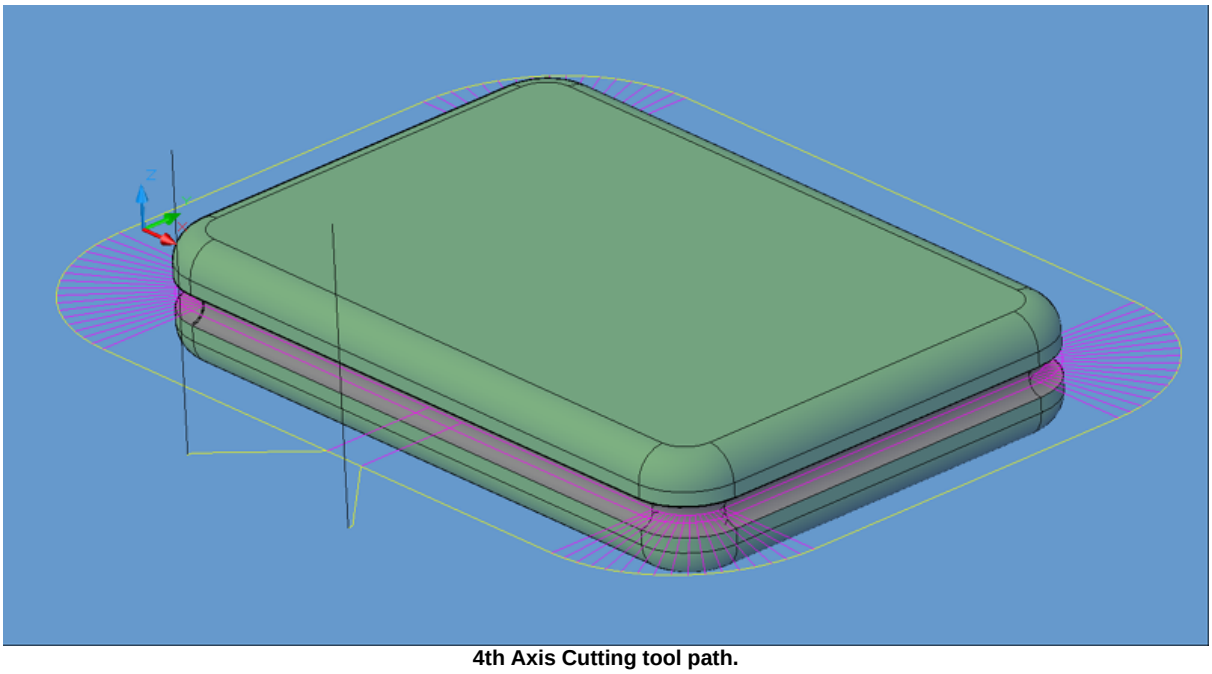


This cycle provides continuous four-axis motion for aggregate heads and 90° router heads.
Drawing Requirements: This cycle works on standard geometry created by the Geoshape command and is edited with the Start Point Edit command. The shape Polylines must be either a two dimensional polyline in the world plane or a three dimensional shape. Two dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create valid cutting motions for these shapes. If an invalid shape is selected, the cycle will notify you when you try to cut it. The shape drawn for a fourth-axis cycle must represent the location of the material to be cut. The tool tip will extend into the cut shape by the required depth. The tool path can be shifted down in Z during the cut cycle, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z shift.

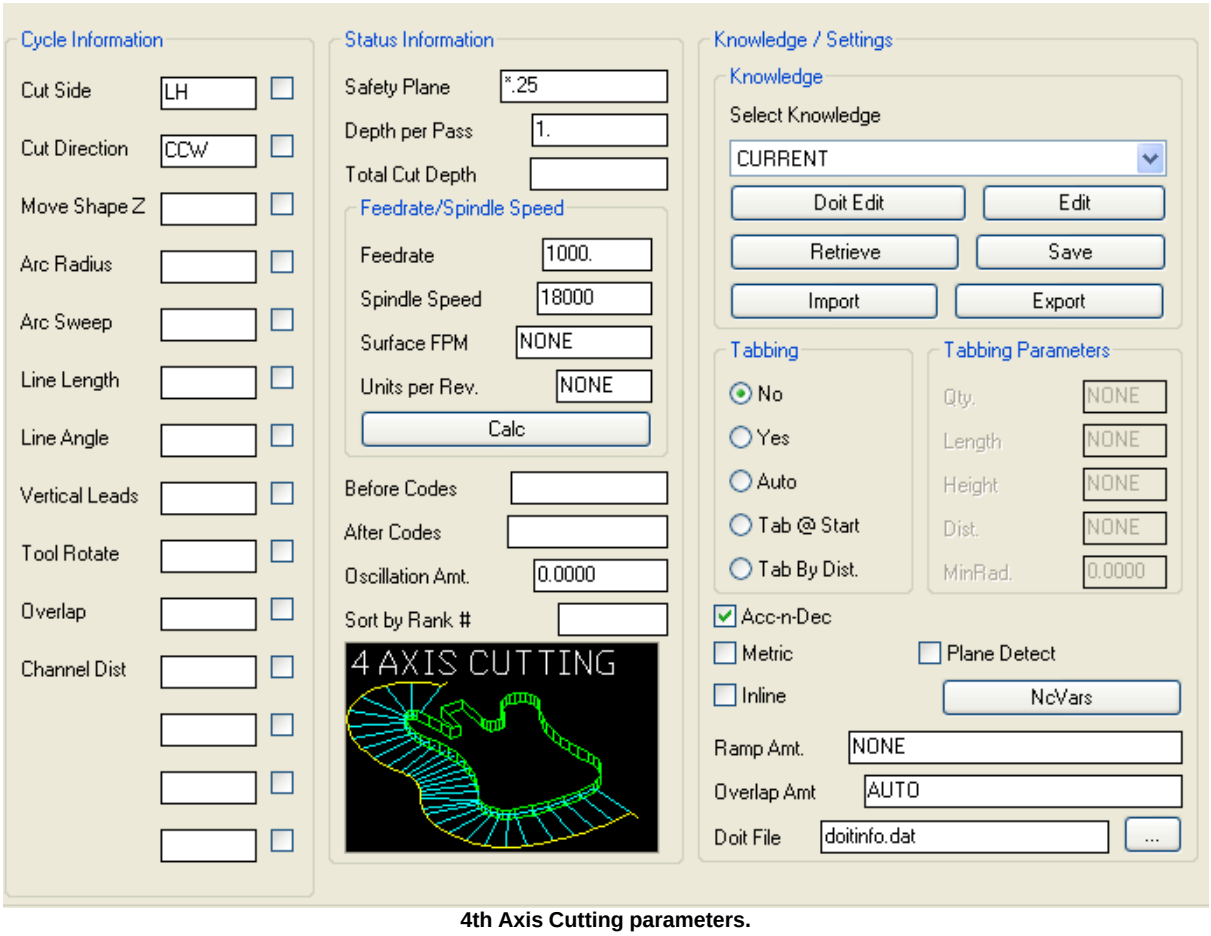
Drawing Requirements

This cycle works on standard geometry created by the Geoshape command and edited with the start point edit command. The shape polylines must be either a two dimensional polyline in the world plane or a three-dimensional shape. Two-dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create cutting valid cutting motions for these shapes.
The Shape drawn for the Forth Axis Cycle must represent the material location to be cut. The tool tip will extend into the cut shape by the depth specified in the Total Cut Depth specified. The tool path can be shifted down in Z during the cut cycle by specifying the depth in the Move Shape Z field, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z Shift for the Move Shape Z field.

© 2011 CIM-TECH.COM, Inc



4th Axis Cutting tool path.



4th Axis Cutting parameters.

© 2011 CIM-TECH.COM, Inc

The following parameters effect the toolpath creation:

Cut Side
Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

Select the Cut Side section for more information.

Cut Direction
The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

Select the Cut Direction section for more information.

Move Shape Z
This value sets the Z height of the tool path. The tool path can be moved up or down with a positive or negative number in this position. Note that this is the Z position in the world coordinate system. The distance into the material horizontally is the Total Cut Depth.

Arc Radius
Lead In and out moves can have an arc lead. The radius of the arc is specified here. There can be two values placed in this position, the first being the lead-in and the second being the lead-out.

Arc Sweep
The degrees of sweep in the arcs for the leads can be specified here. There can be two values placed in this position. The first would be for the lead-in and the second would be for the lead-out.

Line Length
The length of the lead-in and lead-out line. There can be two values here, one for the lead-in and one for the lead-out.

Line Angle
The angle of the leads, specified in degrees. There can be two values here, one for the lead-in and one for the lead-out.

Vertical Leads
Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be in the vertical plane, instead of the XY horizontal plane.

Tool Rotate
Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be cause the tool to rotate into the lead-in and lead-out. This is only valid for leads where Vertical Leads is set to N.

Overlap
Specified as a numeric distance. The overlap is the distance the tool travels as it crossed the start point. This is useful when removing witness marks from the lead-in.

© 2011 CIM-TECH.COM, Inc

Channel Distance

If Vertical Leads is set to Y, then you can specify a Channel Distance to make the cutter ramp into the cut following the contour. Setting the distance to a small number will cause a steep, sharp ramp, and setting it to a larger number will make a longer, more gradual ramp. There can be two values here, one for the lead-in and one for the lead-out.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

Select the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

Spindle Speed

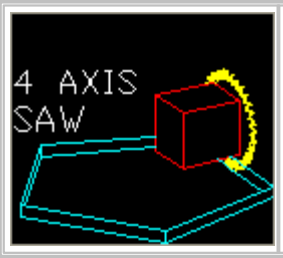
This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

© 2011 CIM-TECH.COM, Inc

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.6.2 4 Axis Saw

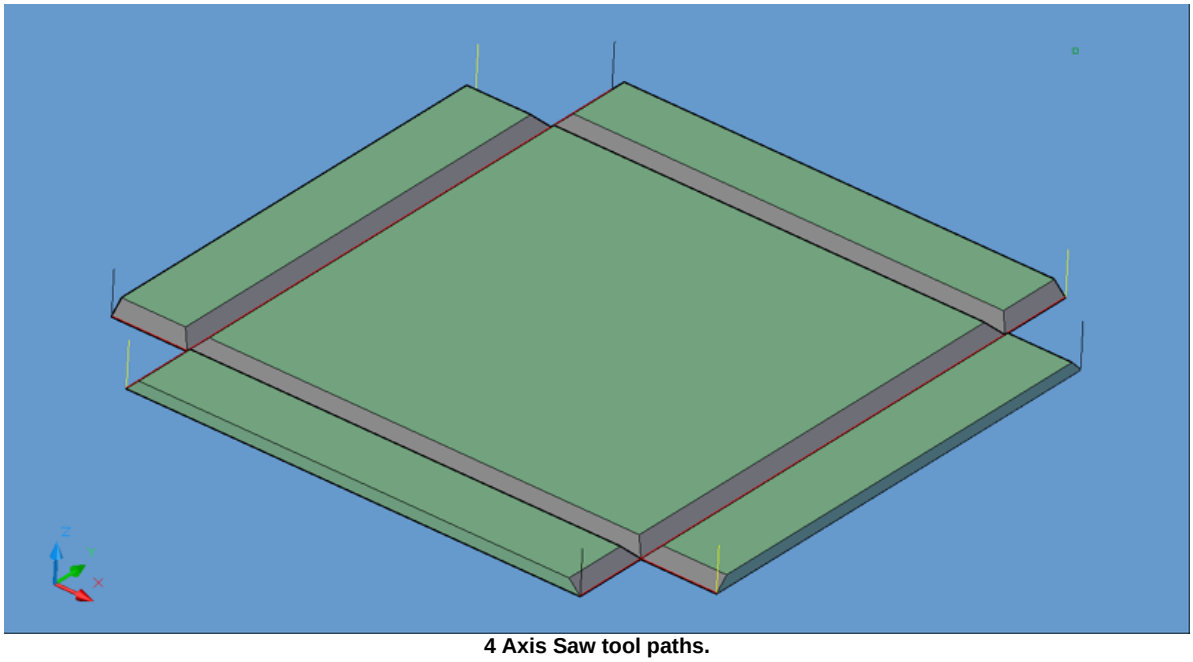


This cycle is set up much the same way as a normal straight cut, except use the tool number for the four-axis saw head. Router-CIM will interpret the axis geometry and all moves required to complete the cycle. The Control Panel settings for this cycle are the same as for a 2D cycle.

. • The 4 Axis Safe field has no function in this cycle.

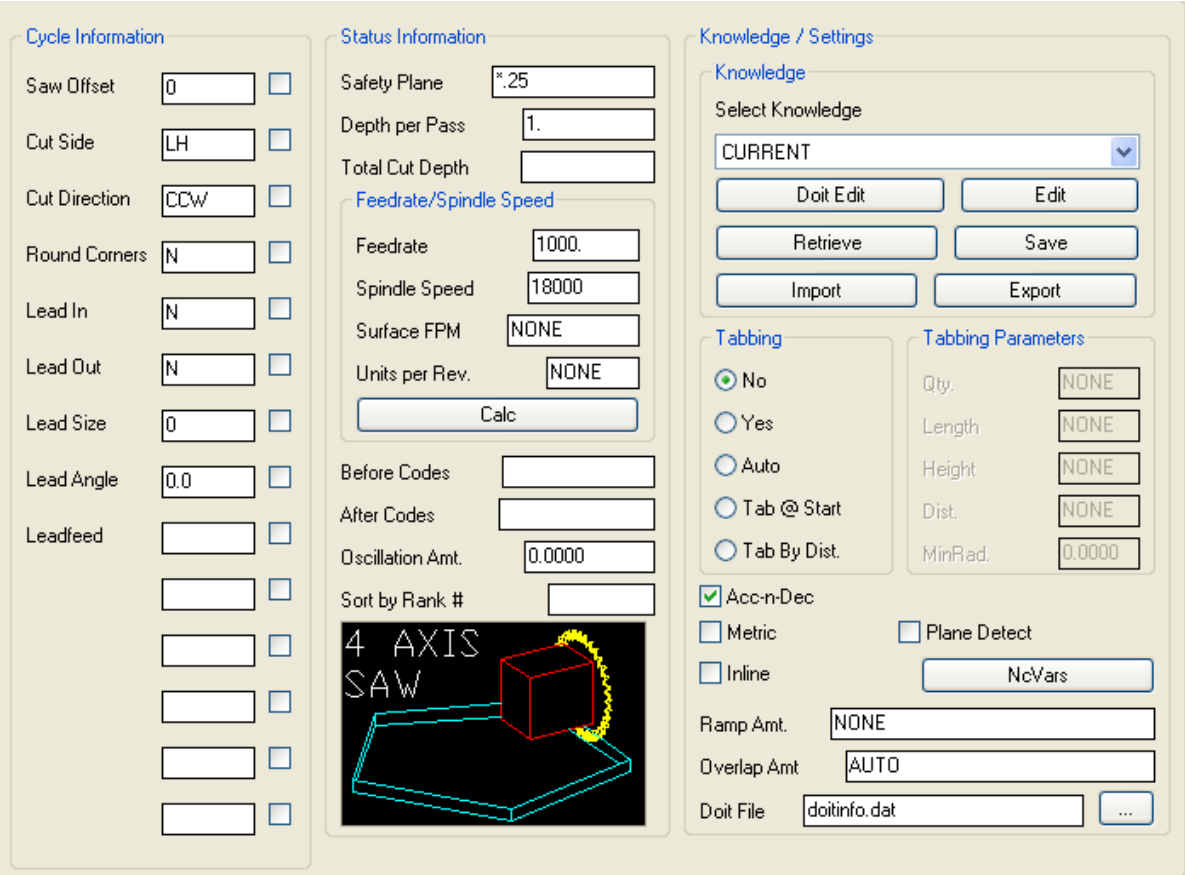
. • The Tool Length field is used to determine the distance from the center of the saw to the center of the blade.

. • Cutter Compensation is available in this cycle.



4 Axis Saw tool paths.

© 2011 CIM-TECH.COM, Inc



4th Axis Saw parameters.

The following parameters effect the toolpath creation:

Saw Offset

The saw offset is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.
For a saw, you might want the tool path set on top of your geometry with no offset (use 0) or offset by some amount relative to your saw spindle or aggregate.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

See Offset Dim for information that is relevant to Saw Offset as well.

Cut Side

© 2011 CIM-TECH.COM, Inc

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

© 2011 CIM-TECH.COM, Inc

See the Lead Angle section for more information.

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).
Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the Lead Feed section for more information.

Safety Plane

The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

Select the Safety Plane section for more information.

Depth Per Pass

This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

© 2011 CIM-TECH.COM, Inc

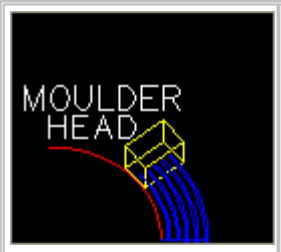
Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

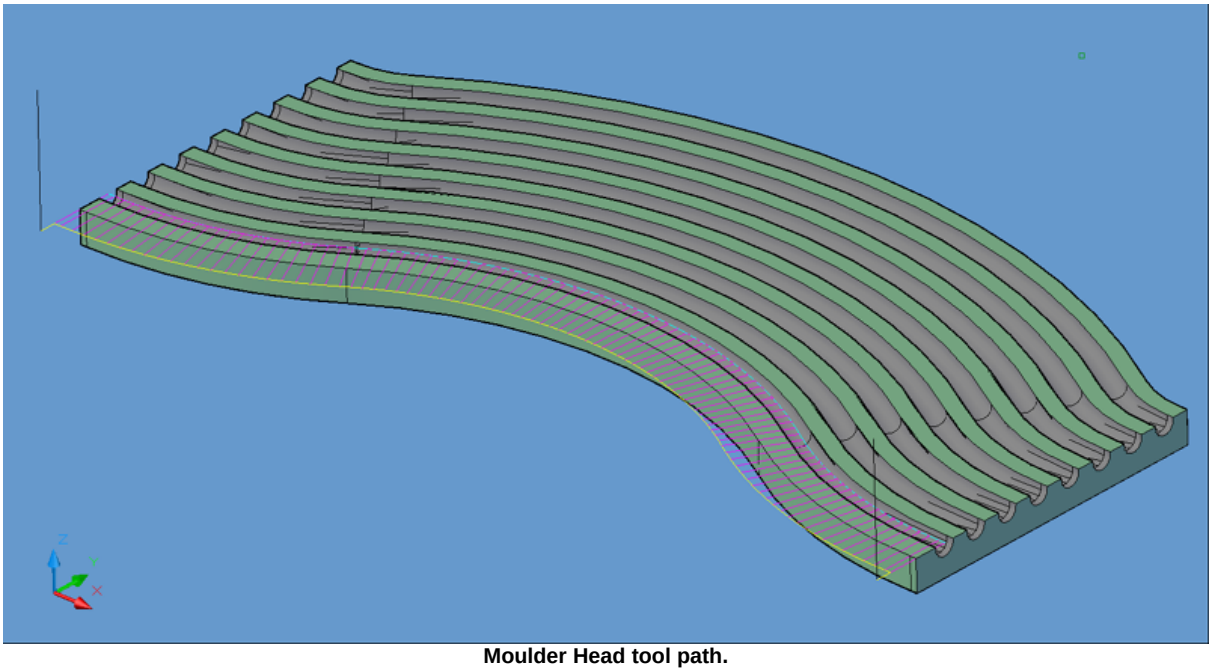
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.6.3 4 Axis Moulder Head

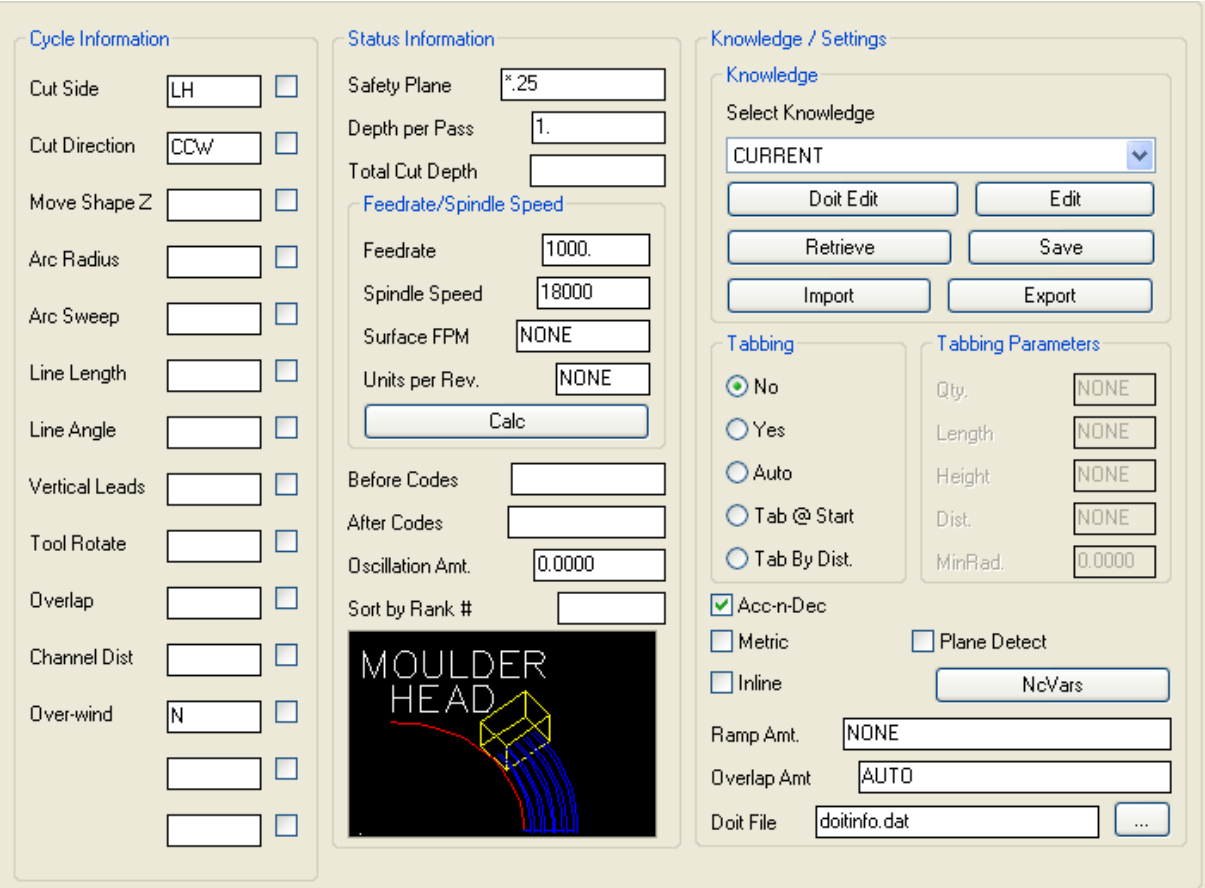


This cycle provides continuous four-axis motion for aggregate moulder heads and fixed moulder heads.
Drawing Requirements: This cycle works on standard geometry created by the Geoshape command and is edited with the Start Point Edit command. The shape Polylines must be either a two dimensional polyline in the world plane or a three dimensional shape. Two dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create valid cutting motions for these shapes. If an invalid shape is selected, the cycle will notify you when you try to cut it. The shape drawn for a fourth-axis cycle must represent the location of the material to be cut. The tool tip will extend into the cut shape by the required depth. The tool path can be shifted down in Z during the cut cycle, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z shift.



Moulder Head tool path.

© 2011 CIM-TECH.COM, Inc



Moulder Head parameters.

The following parameters effect the toolpath creation:

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

Select the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

Select the Cut Direction section for more information.

Move Shape Z

This value sets the Z height of the tool path. The tool path can be moved up or down with a positive or negative number in this position. Note that this is the Z position in the world coordinate system. The

© 2011 CIM-TECH.COM, Inc

distance into the material horizontally is the Total Cut Depth.

Arc Radius
Lead In and out moves can have an arc lead. The radius of the arc is specified here. There can be two values placed in this position, the first being the lead-in and the second being the lead-out.

Arc Sweep
The degrees of sweep in the arcs for the leads can be specified here. There can be two values placed in this position. The first would be for the lead-in and the second would be for the lead-out.

Line Length
The length of the lead-in and lead-out line. There can be two values here, one for the lead-in and one for the lead-out.

Line Angle
The angle of the leads, specified in degrees. There can be two values here, one for the lead-in and one for the lead-out.

Vertical Leads
Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be in the vertical plane, instead of the XY horizontal plane.

Tool Rotate
Specify Y for yes, or N for no. If Yes is selected, then the lead-in and lead-out moves will be cause the tool to rotate into the lead-in and lead-out. This is only valid for leads where Vertical Leads is set to N.

Overlap
Specified as a numeric distance. The overlap is the distance the tool travels as it crossed the start point. This is useful when removing witness marks from the lead-in.

Channel Distance
If Vertical Leads is set to Y, then you can specify a Channel Distance to make the cutter ramp into the cut following the contour. Setting the distance to a small number will cause a steep, sharp ramp, and setting it to a larger number will make a longer, more gradual ramp. There can be two values here, one for the lead-in and one for the lead-out.

Safety Plane
The safety plane is the location in the Z axis where the tool can retract to between cuts.
This should always be a value that places the cutter above the part to be cut as each tool change, or index move between cuts is going to start from this point.
Placing an asterisk (*) before the number specifies that this value is an absolute point above the part, where leaving this out determines the point to be incremental.

Select the Safety Plane section for more information.

Depth Per Pass
This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to Cut through, you would set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0.

In most of the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

© 2011 CIM-TECH.COM, Inc

Total Cut Depth

The Total Cut Depth is the depth you wish to Cut to, regardless of the number of passes made. It is usually put in as a negative number because Z0 is set at the top of the part. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to Cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth.

You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and if you have given you part thickness, Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

Feedrate

This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

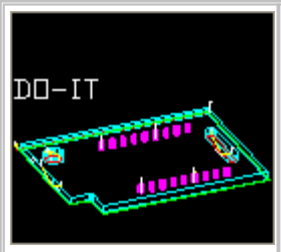
Spindle Speed

This field sets the spindle speed in rpm's (revolutions per minute). This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.1.6.4 DO-IT



DO-IT is automatic tool and cycle selection, accomplished using a knowledge base you define according to your individual cutting needs. This knowledge base and Layer to Knowledge level cutting is the basis for the Automation Suite inside of Router-CIM.

Using DO-IT you can place geometry on specific layers, and then associate those layers to specific cutting knowledges via a "DOIT File" or association file that stores the relationships of the layer to knowledge cutting. DO-IT will automatically search the file for the associations and then if the layer is present and the knowledge is present in the current drawing, it will place tool paths on the part according to the associations found.

****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

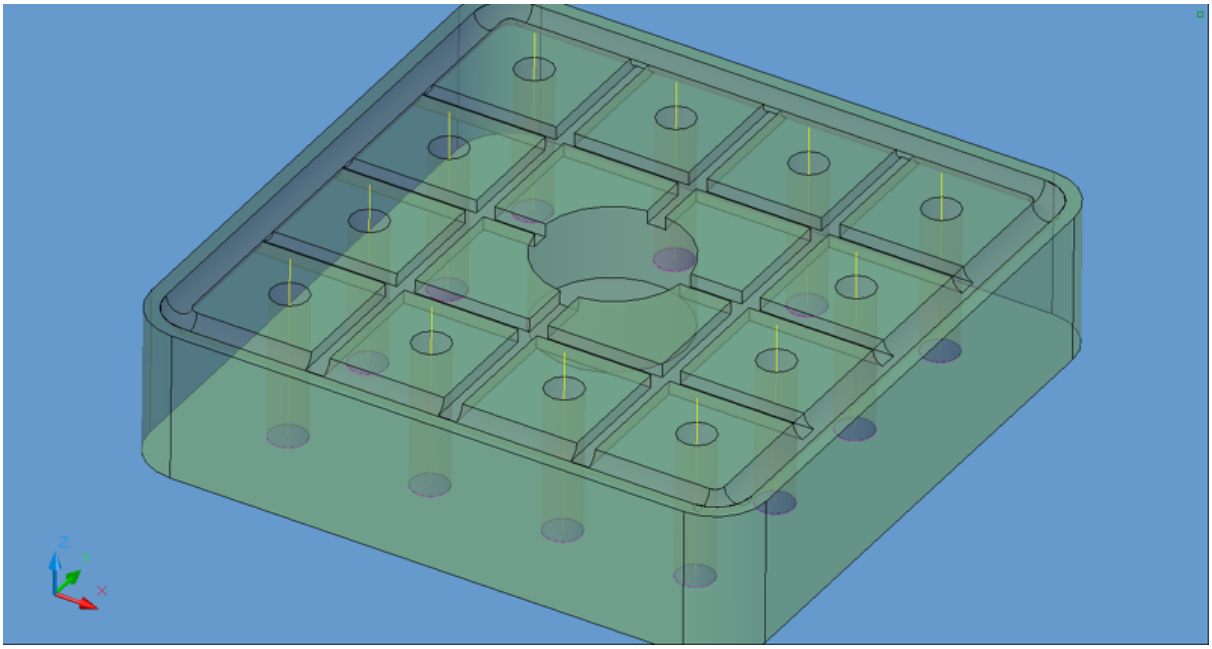
© 2011 CIM-TECH.COM, Inc

6.1.6.5 Tap-Matic

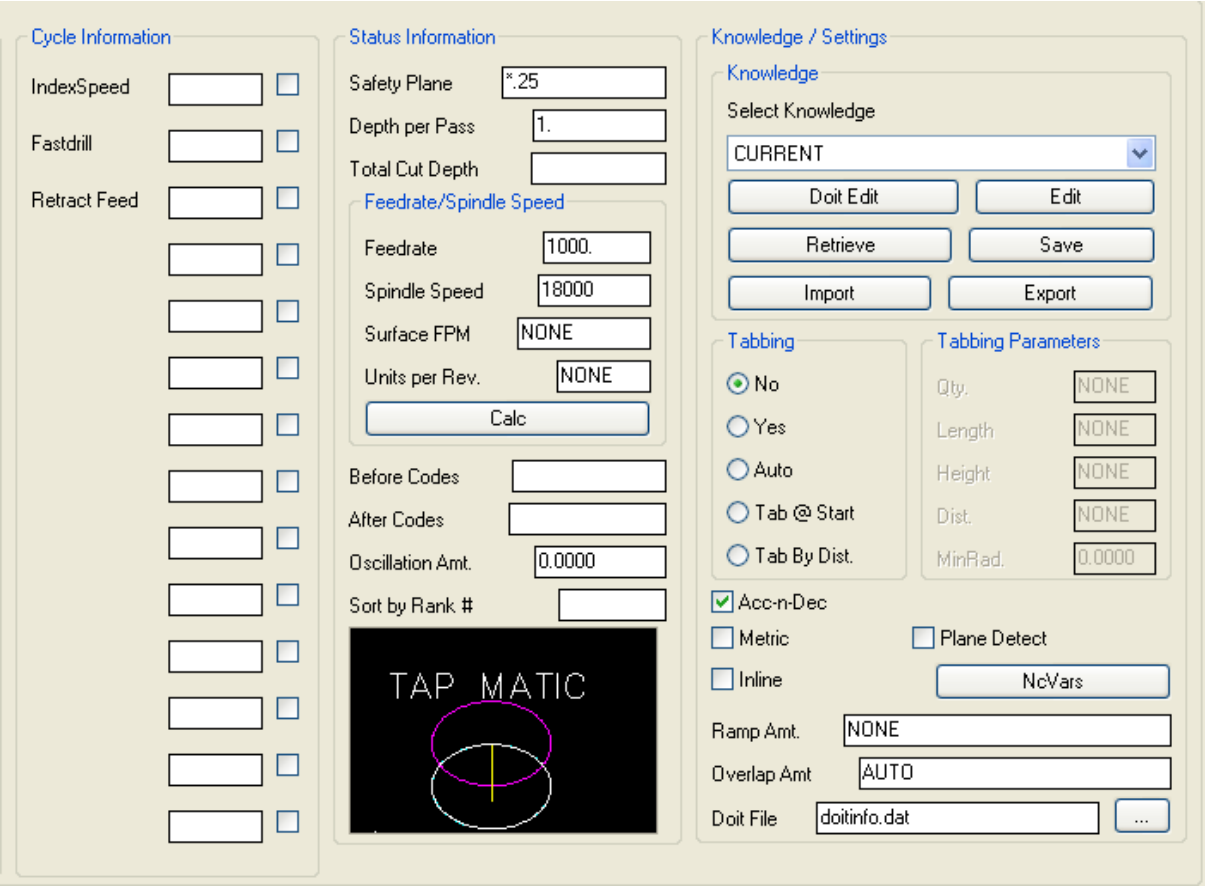


The Tap Matic cycle is used to program a drill type motion for a Tap Matic head fitted to a spindle on the machine. The Tap Matic head requires a feedrate for the motion down into the hole, and another feedrate on the way back up out of the hole.

Router-CIM provides for the separate feedrates in its drill motions using this cycle so that the Tap Matic head can be used. The geometry needed is simply a circle, geoshaped, and otherwise the cycle works the same as Drill Motions.



© 2011 CIM-TECH.COM, Inc



The following parameters effect the toolpath creation:

Retract Feed

This cycle has a separate feedrate for the lead-out. You should specify the lead-out feedrate in this position.

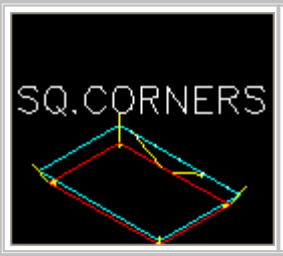
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the**

© 2011 CIM-TECH.COM, Inc

toolpath and NC Code carefully before running your machine tool if you change these default settings.

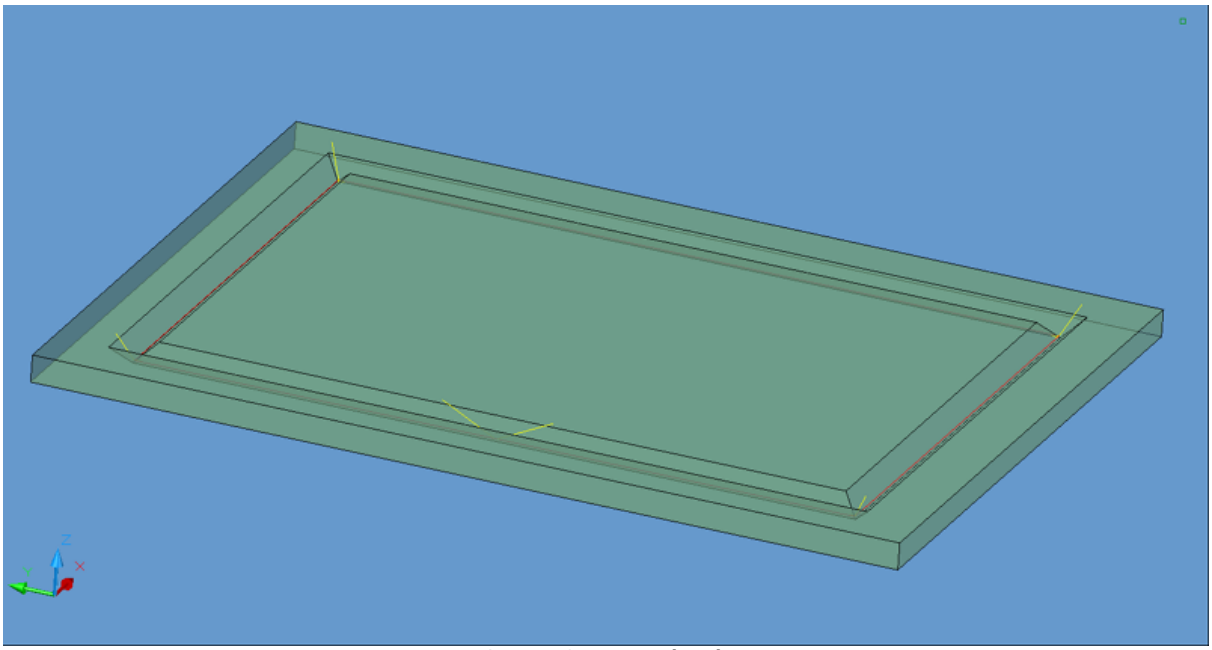
© 2011 CIM-TECH.COM, Inc

6.1.6.6 Square Corners



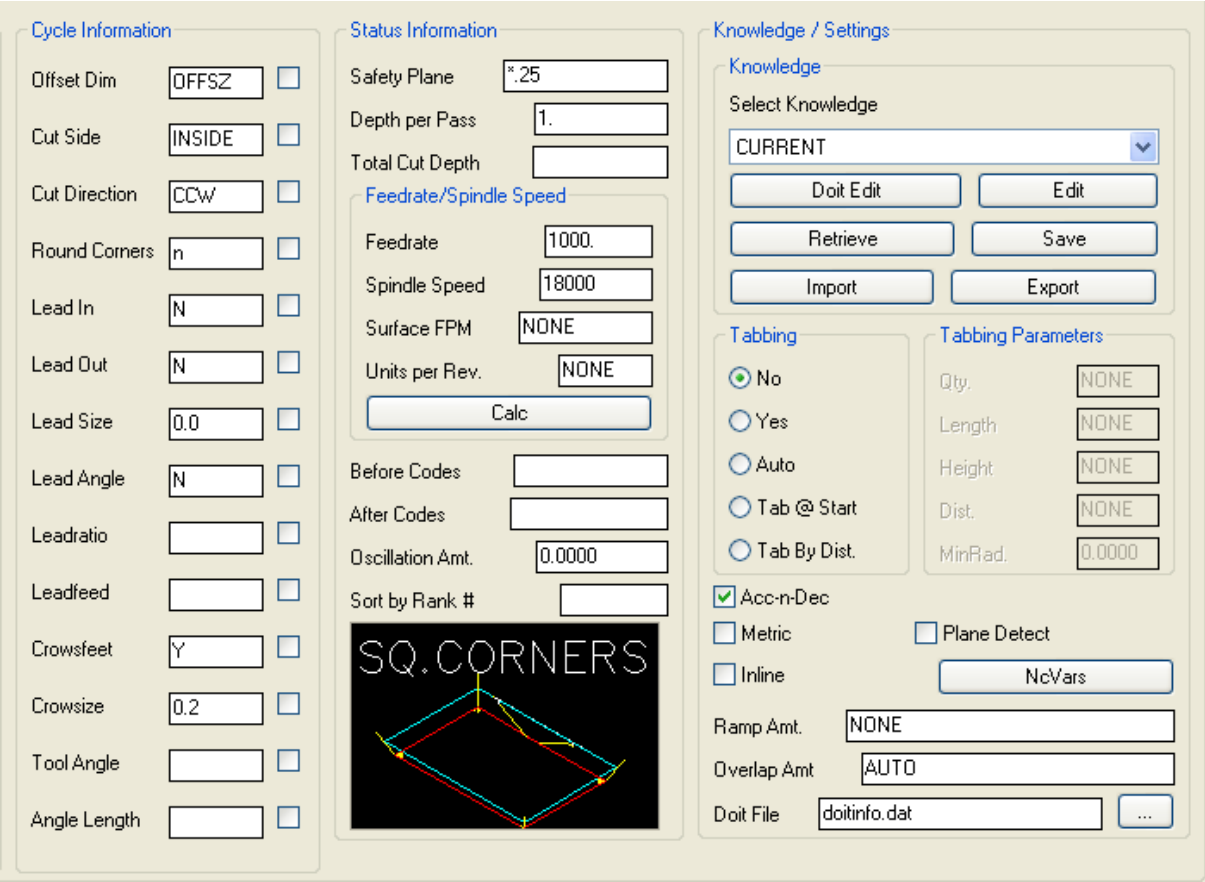
The Square Corners cycle is most often used when making solid doors (such as MDF single piece doors) and the desired effect is to have a squared off corner cut with an angled tool so that there is a somewhat sharp edge on the inside of the profile of the door.

This is accomplished by moving the angled tool up and down in each corner, using the angle of the tool to make a squared off edge.



Square Corner tool path

© 2011 CIM-TECH.COM, Inc



Square Corner parameters

The following parameters effect the toolpath creation:

Offset Dim

The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.

See Offset Dim for more information.

Cut Side

Cut Side is the side of the Geoshape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).

© 2011 CIM-TECH.COM, Inc

See the Cut Side section for more information.

Cut Direction

The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

See the Cut Direction section for more information.

Round Corners

If set to Yes, this option will round sharp corners with a radius of the value stored in the task *cutfil*. The default is 0.01 radius (in inch mode). This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

See the Round Corners section for more information.

Lead In

This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-In section for more information.

Lead Out

This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits. By default this cycle will usually not have the lead-in or lead-out changed as the defaults will accommodate multiple depths per pass and cutting on any plane.

See the Lead-Out section for more information.

Lead Size

Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field. You can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

See the Lead-Size section for more information.

Lead Angle

Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

See the Lead Angle section for more information.

© 2011 CIM-TECH.COM, Inc

Lead Feed

This sets lead-in and lead-out feed rates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.
Setting a number between 0 and 1.0 will give you a percentage of the max feedrate (for instance 0.4 would be 40%).
Setting the number to a value greater than 1.0 will give you an exact feedrate. For instance 250. would generate F250. in the code.

See the Lead Feed section for more information.

Crowsfeet

Possible answers here are Y for yes, or N for no. This parameters specifies whether or not to make tiny corner clean up moves at the bottom of the angled corner cut.

Crowsize
Setting this to a numeric value will allow you to control the size of the crowsfeet made at the bottom of the angled corner cut.
Crowsfeet must be set to Y in order for this setting to have any effect.

Tool Angle

Placing the tool angle in this parameter allows the square corner move to be at the correct angle to match the tool and the geometry.

Angle Length

This is a numeric value for the total length of the square corner move.

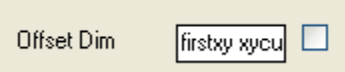
****Changing values in the cycle parameters may yield unexpected results with some settings or on some geometry. Examine the toolpath and NC Code carefully before running your machine tool if you change these default settings.**

© 2011 CIM-TECH.COM, Inc

6.2 Cycle Parameters

There are many parameters associated with the cutting cycles, but a majority of them share some parameters in common. This section will focus on some of the shared parameters and offer a more detailed explanation of them.

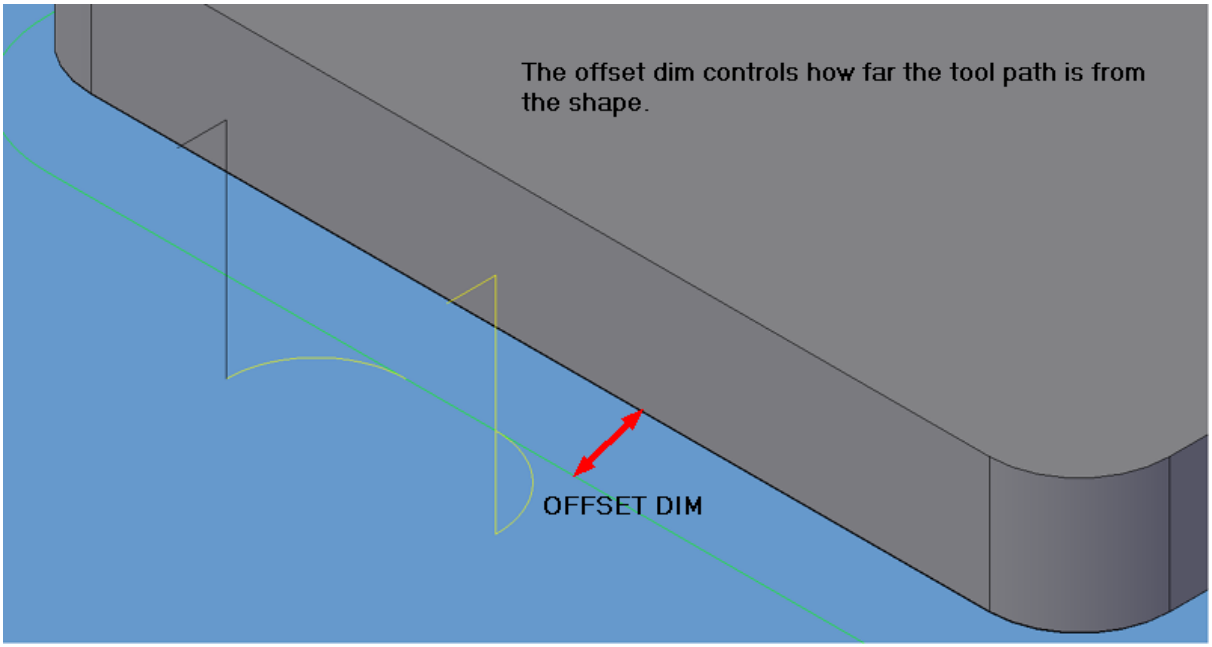
6.2.1 Offset Dim



The offset dim is the amount the toolpath is offset from the original geometry or Geoshape. Normally this is set by Router-CIM depending on a number of features such as the Cutter Compensation setting and the cut cycle itself. For instance if Cutter Comp is set to Yes, then the toolpath will lie directly on top of the Geoshaped geometry with no offset.

You may substitute the parameters here for numeric values to suit you particular cutting needs.

The value set by default (firstxy xycutloc) is a macro setting that allows Router-CIM to handle the offset automatically and will usually not need to be changed.



The options selected from the Options Box by default are:

firstxy xycutloc -- This is a macro which will allow Router-CIM to perform the normal offset depending on various settings on the cut such as cutter compensation.

0.0 -- This will set the offset amount to 0 and leave the toolpath directly on top of the geo-shaped entity.

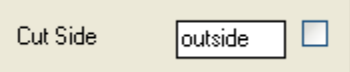
offsz -- This is also a Router-CIM macro, which allows the offset to be handled by Router-CIM.

!*tr* -- A Router-CIM macro which will set the offset to the tools radius.

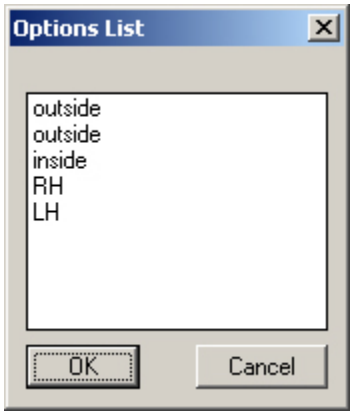
© 2011 CIM-TECH.COM, Inc

0 -- This will set the offset amount to 0 and leave the toolpath directly on top of the geo-shaped entity.

6.2.2 Cut Side



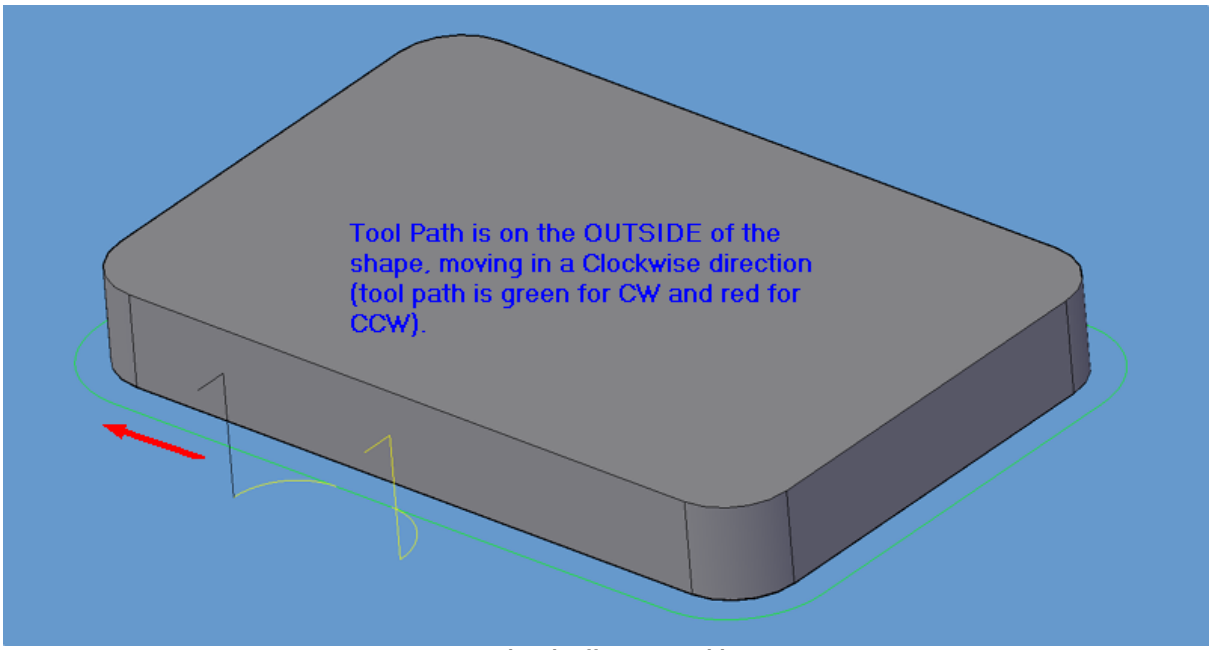
Cut Side is the side of the geo shape that the toolpath will be created on. For instance Plunge-Outside (Plunge-O) will have the toolpath on the outside of the shape. Valid entries for this field are *Outside*, *Inside*, *RH* (Right Hand) and *LH* (Left Hand).



The options selected from the Options Box by default are:

outside
Places the tool path on the Outside of a closed shape as determined by the Geoshape command.

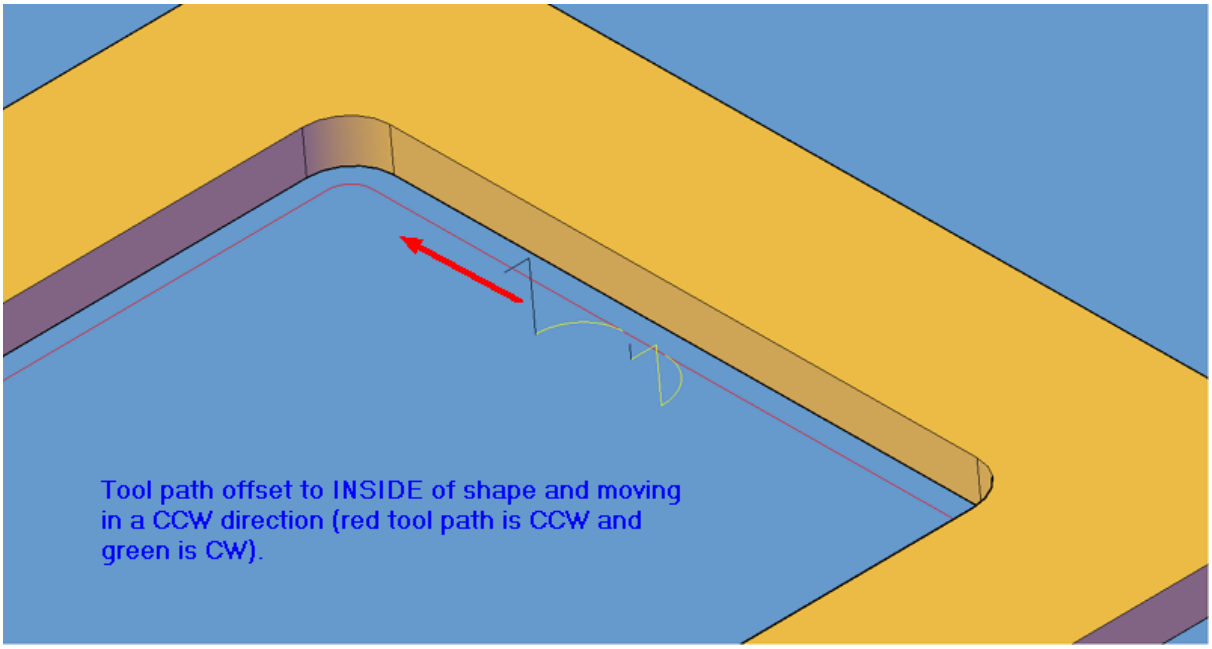
© 2011 CIM-TECH.COM, Inc



Tool path offset to outside.

inside

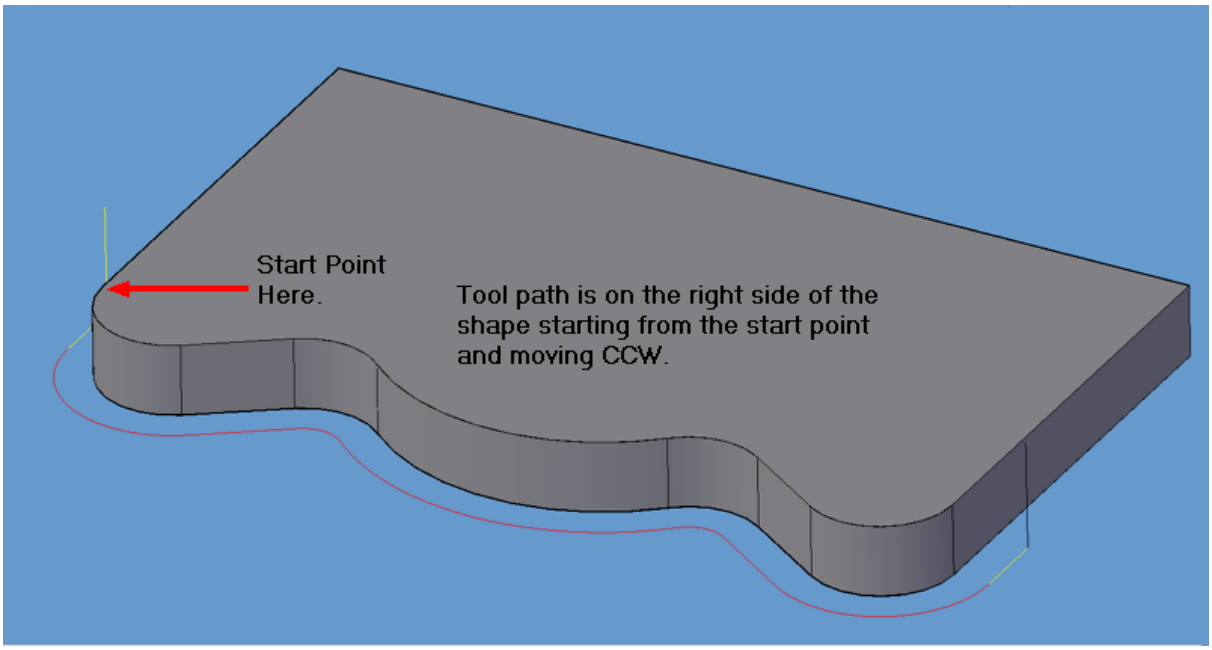
Places the tool path on the Inside of a closed shape as determined by the Geoshape command.



RH

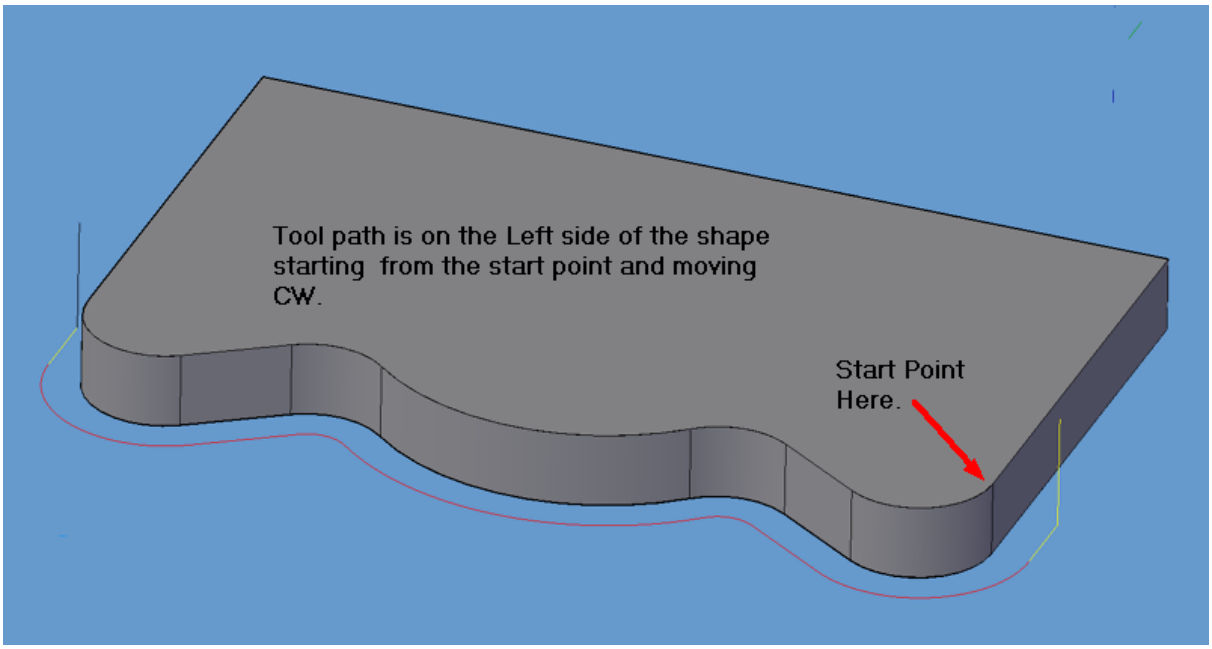
Places the tool path on the right side of a closed shape as determined by the Geoshape command.

© 2011 CIM-TECH.COM, Inc



LH

Places the tool path on the Left side of a closed shape as determined by the Geoshape command.



The drawing below shows different Cut Side/Cut Direction combinations based on the shape direction, to assist you in choosing a Start Point.

© 2011 CIM-TECH.COM, Inc



It is very common when cutting open shapes that you will need to change the side of the cut. Clicking the option box (small square to the right) allows you to choose the correct option: Inside, Outside, LH, and RH.
Inside and Outside work best on closed shapes; LH and RH work best on open shapes. To check the direction of an open shape use the AutoCAD command /pedit/Edit Vertex.

The best way to cut an open shape is to:
1. Select the Plunge-O Line Leads cycle.
2. Set the Cut Direction to CCW.
3. Set the Cut Side to RH or LH, depending on which side you want to cut.

OR

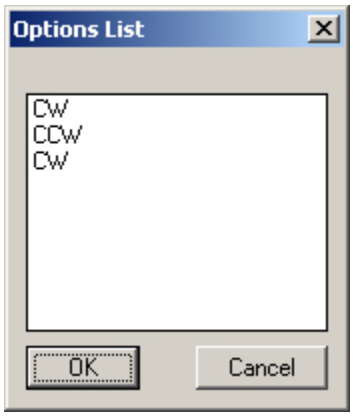
1. Use the Right Hand Cut cycle to cut on the right side of an open shape
2. Use the Left Hand Cut cycle to cut on the left side of an open shape

6.2.3 Cut Direction



The direction of the cut can only be clockwise (CW) or counter-clockwise (CCW). This even applies to open shapes where this direction really has no meaningful relationship to the geometry selected. Any closed shapes should have the direction set accordingly and any open shapes should be set to CCW as all shapes in AutoCAD and Router-CIM are CCW by default.

© 2011 CIM-TECH.COM, Inc



The options selected from the Options Box by default are:

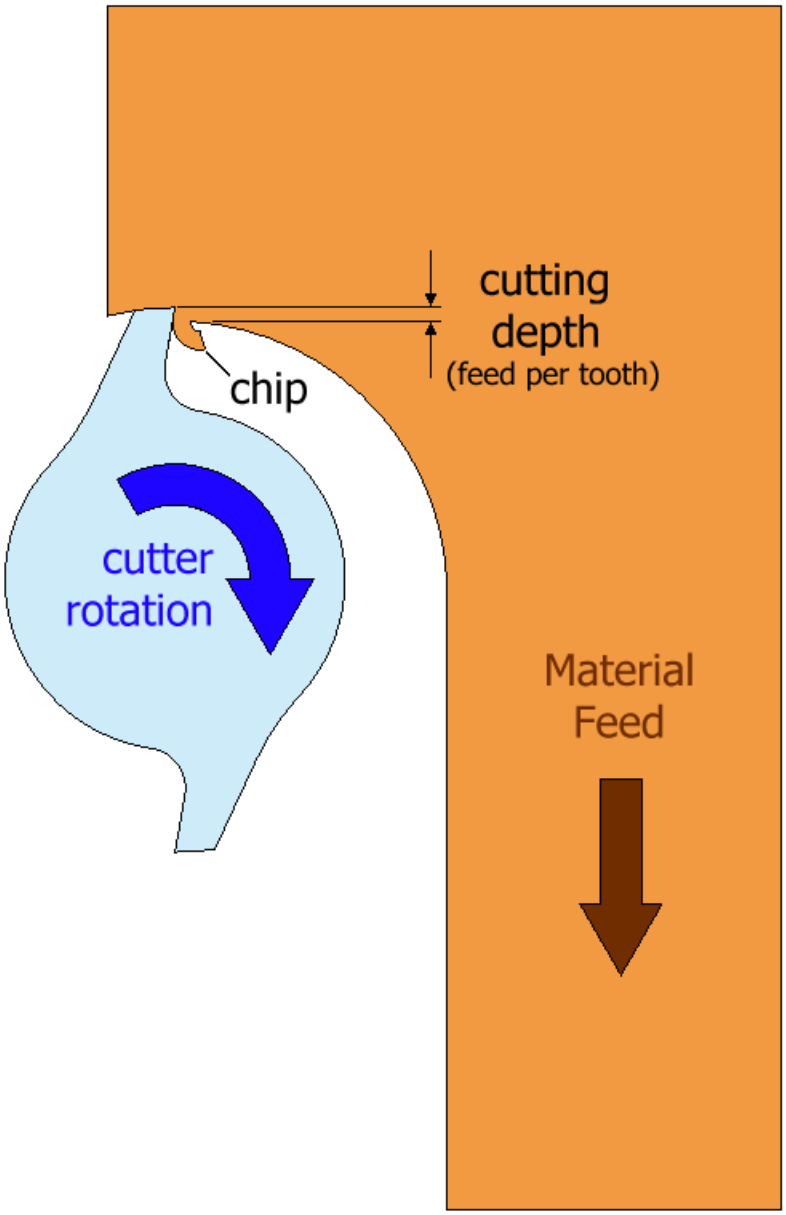
CW
This is the setting for Clockwise direction. CW tool paths in Router-CIM will appear in Green by default.

CCW
This is the setting for Counter-Clockwise direction. CCW tool paths in Router-CIM will appear in Red by default.

Climb Milling or Climb Cutting
Each tooth engages the material at a definite point, and the width of the cut starts at the maximum and decreases to zero. The chips are disposed behind the cutter, leading to easier swarf removal. The tooth does not rub on the material, and so tool life may be longer. However, climb milling can apply larger loads to the machine, and so is not recommended for older milling machines, or machines which are not in good condition.

The default Cut Direction in Router-CIM is Climb Cut, both inside (CCW) and outside (CW). Change this direction to suit your needs. The drawing below displays how the direction is determined.

© 2011 CIM-TECH.COM, Inc

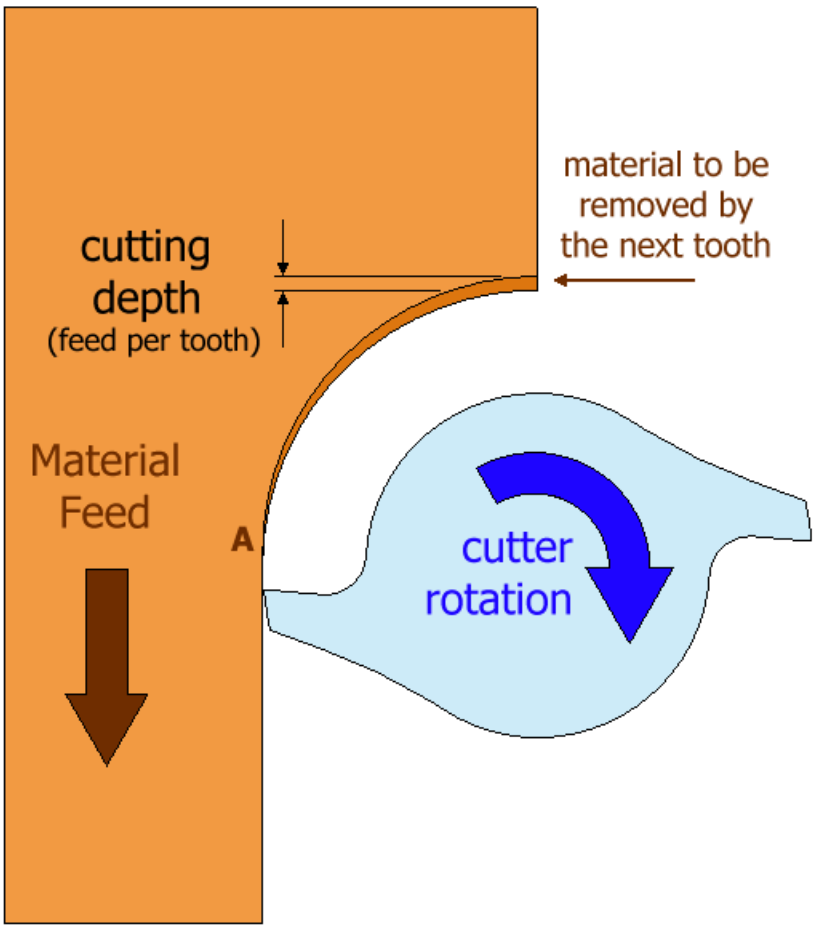


Climb Cut

Conventional Milling or Conventional Cutting

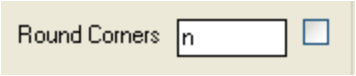
The depth of the cut starts at zero thickness, and increases up to the maximum. The cut is so light at the beginning that the tool does not cut, but slides across the surface of the material, until sufficient pressure is built up and the tooth suddenly bites and begins to cut. This deforms the material (at point A on the diagram, left), work hardening it, and dulling the tool. The sliding and biting behaviour can leaves a poor finish on the material, depending on material and tool types.

© 2011 CIM-TECH.COM, Inc



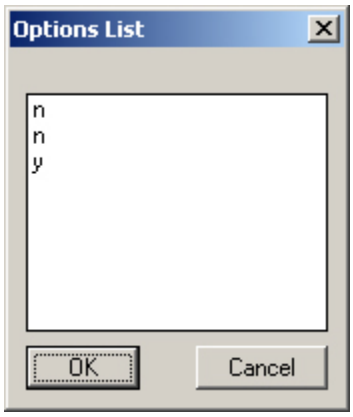
Conventional Cut

6.2.4 Round Corners



The Round Corners function, if set to Yes, will round sharp corners with a radius of the value stored in the variable *cutfil*. The default radius is 0.01 in Router-CIM. This option will insert a fillet in all corners, so if you have an inside cut you will most likely cause an error when the tool tries to fit into that radius or the corner will be cut larger than the intended radius. If you have inside and outside cuts on the same shape and need to fillet the corners, use the AutoCAD Fillet command, then Geoshape and Cut the shape.

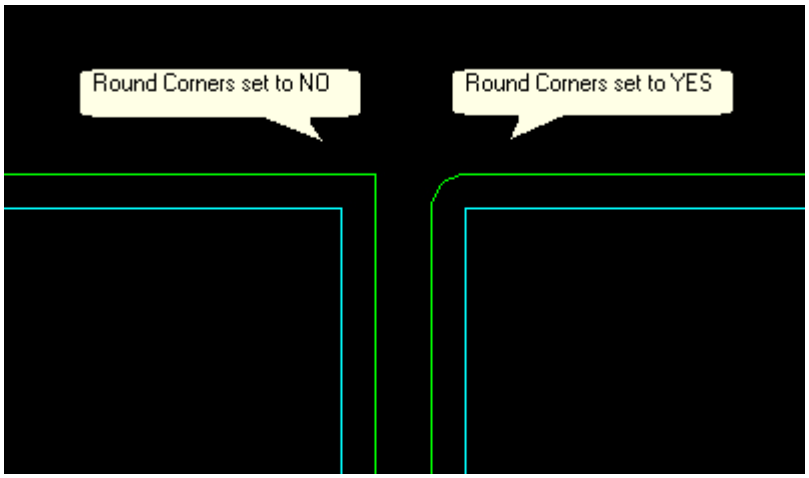
© 2011 CIM-TECH.COM, Inc



The options selected from the Options Box by default are:

n
This will set the option to NO, and no rounded corners will result.

y
This will set the option to YES, and rounded corners will result.

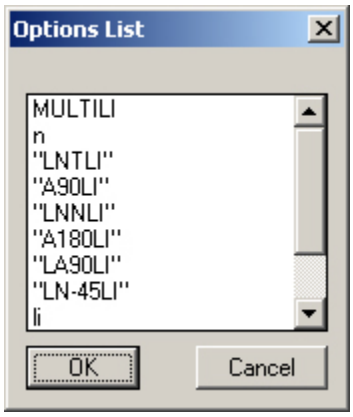


6.2.5 Lead-In



This field defines the lead-In block name. There are several available, but only some cycles will respond to the change of the Lead-In edits. Normally only the Plunge-Outside Line Leads Cycle is valid for setting the Lead-In or Lead-Out.

© 2011 CIM-TECH.COM, Inc



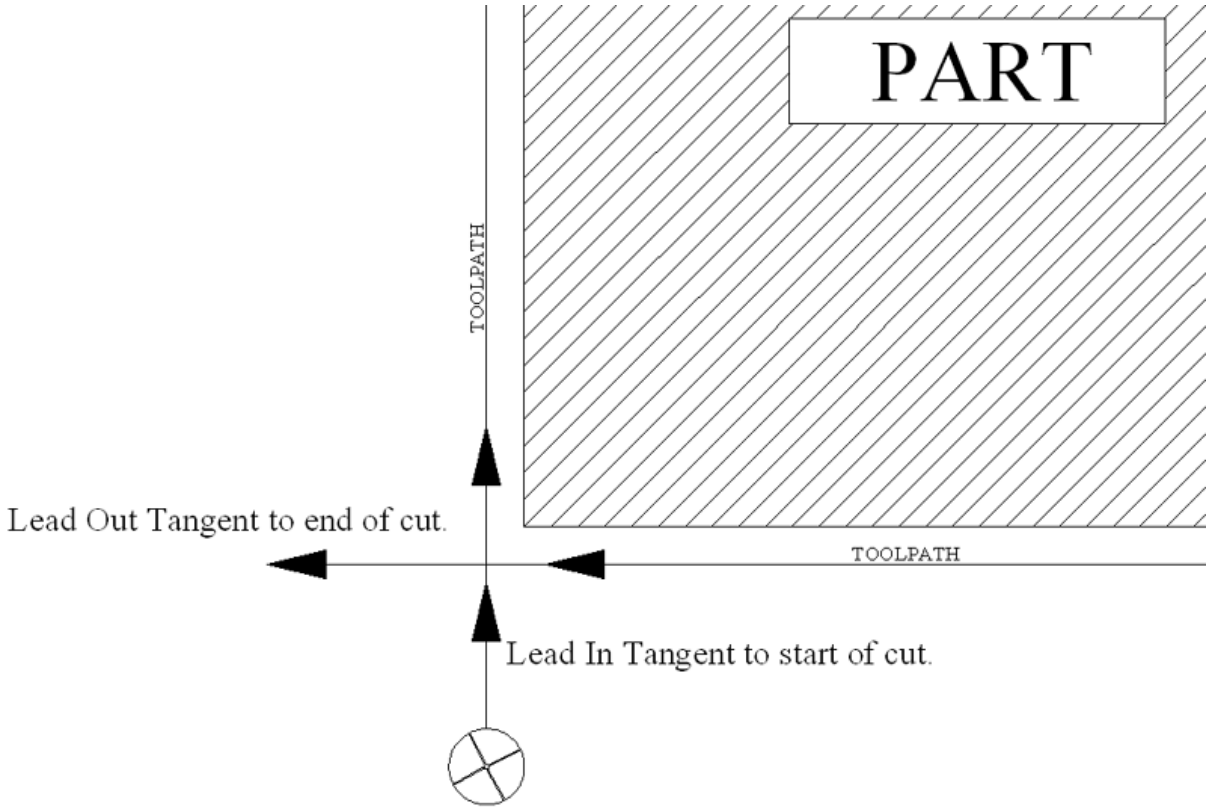
The options selected from the Options Box by default are:

MULTILI
Multili is the default lead type for some cycles where there are multiple depth per pass, and Router-CIM must figure out how to move each cut down (usually ramping) between passes. This is typically not a user selectable option.

n
None. There will be no lead in move generated.

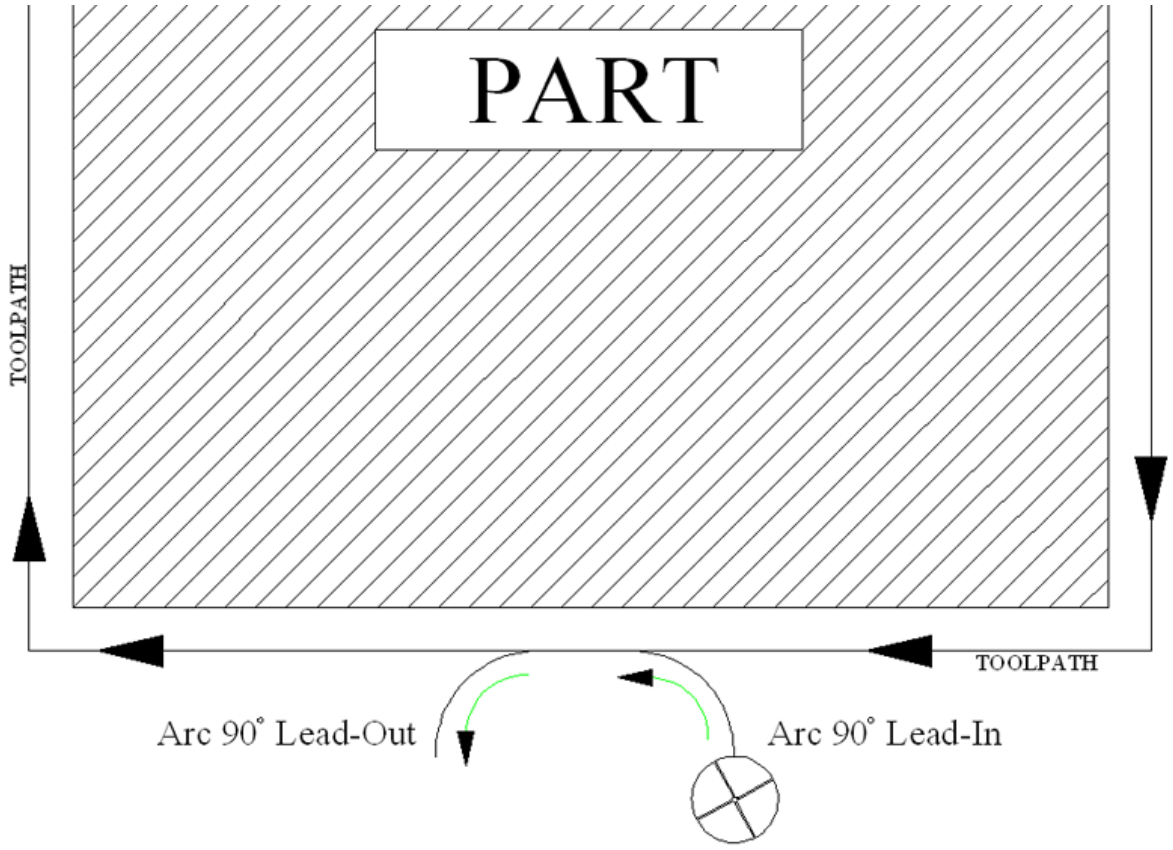
LNTLI
Line Tangent Lead In. This generates a move tangent to the start of the cut.

© 2011 CIM-TECH.COM, Inc



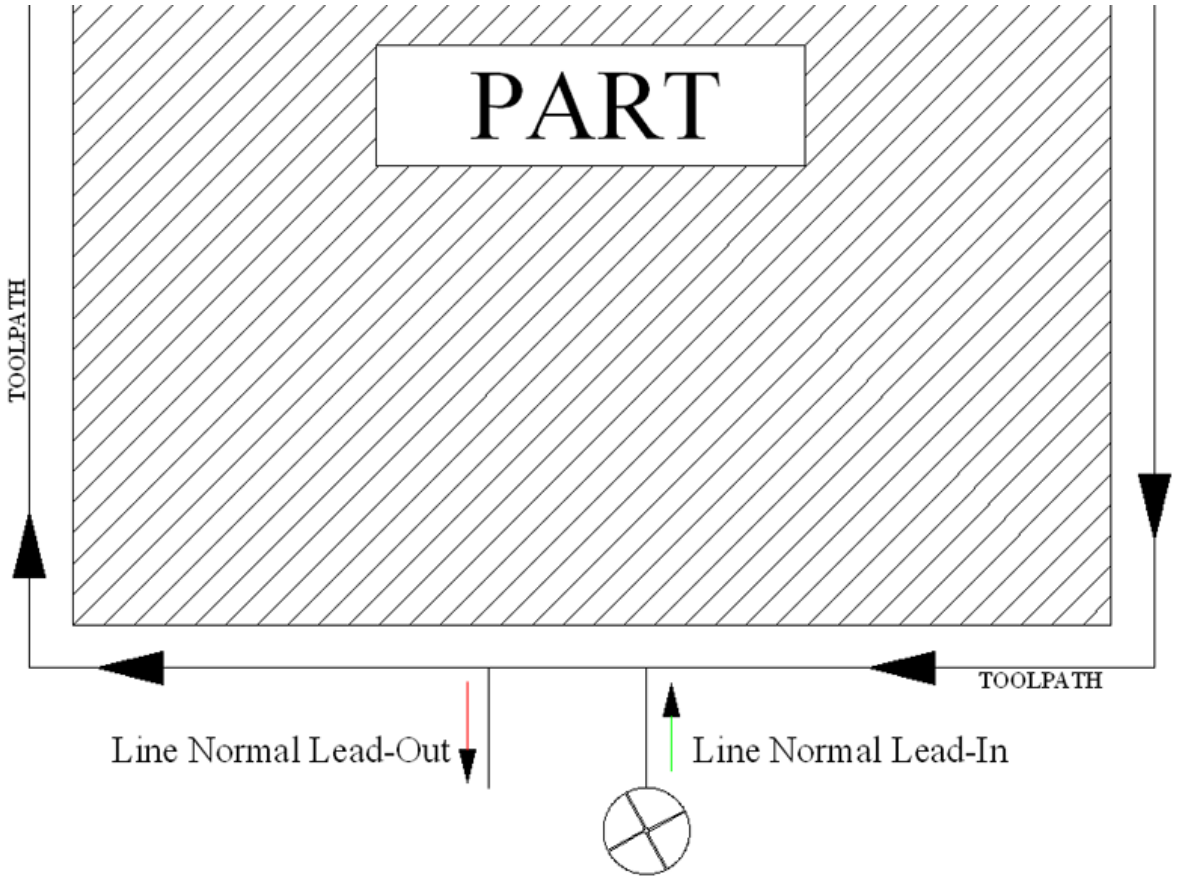
A90LI
Arc 90° Lead In. Generates an arc of 90°, ending tangent to the start of the cut.

© 2011 CIM-TECH.COM, Inc



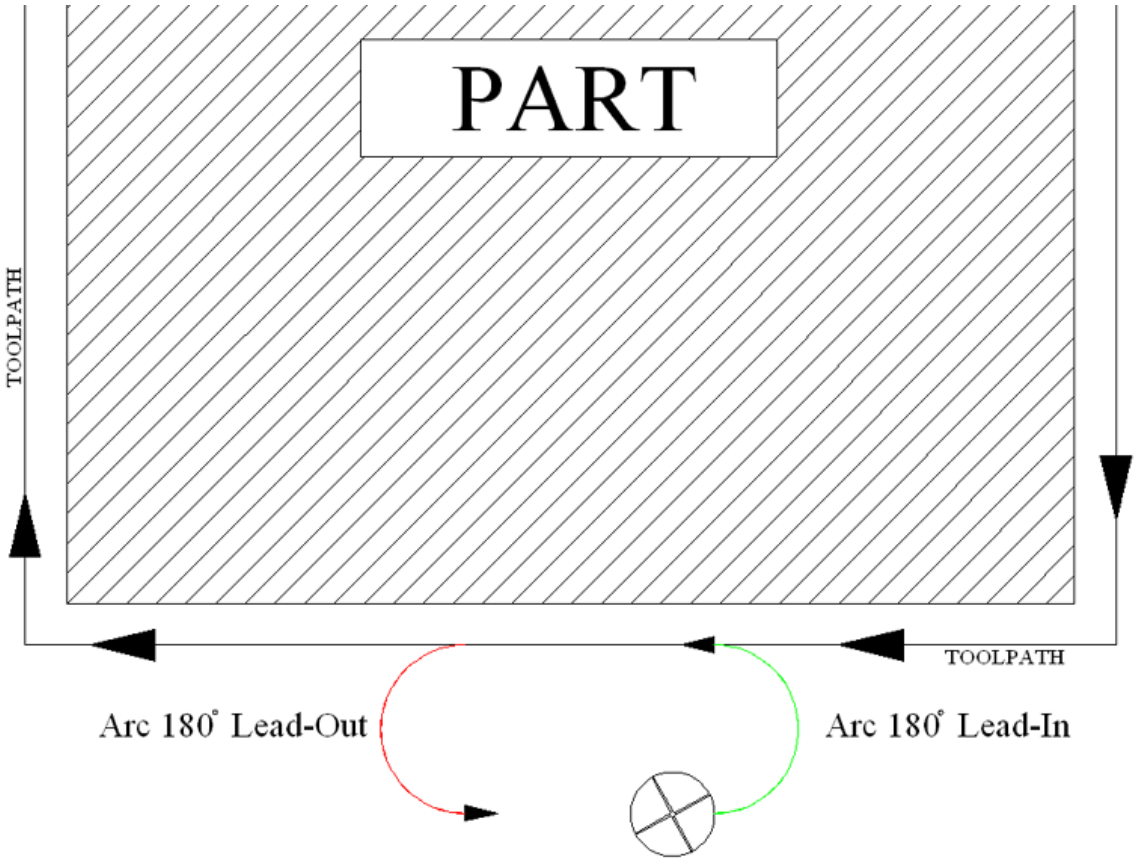
LNNLI
Line Normal Lead In. The Line Normal is perpendicular to the start of the cut.

© 2011 CIM-TECH.COM, Inc



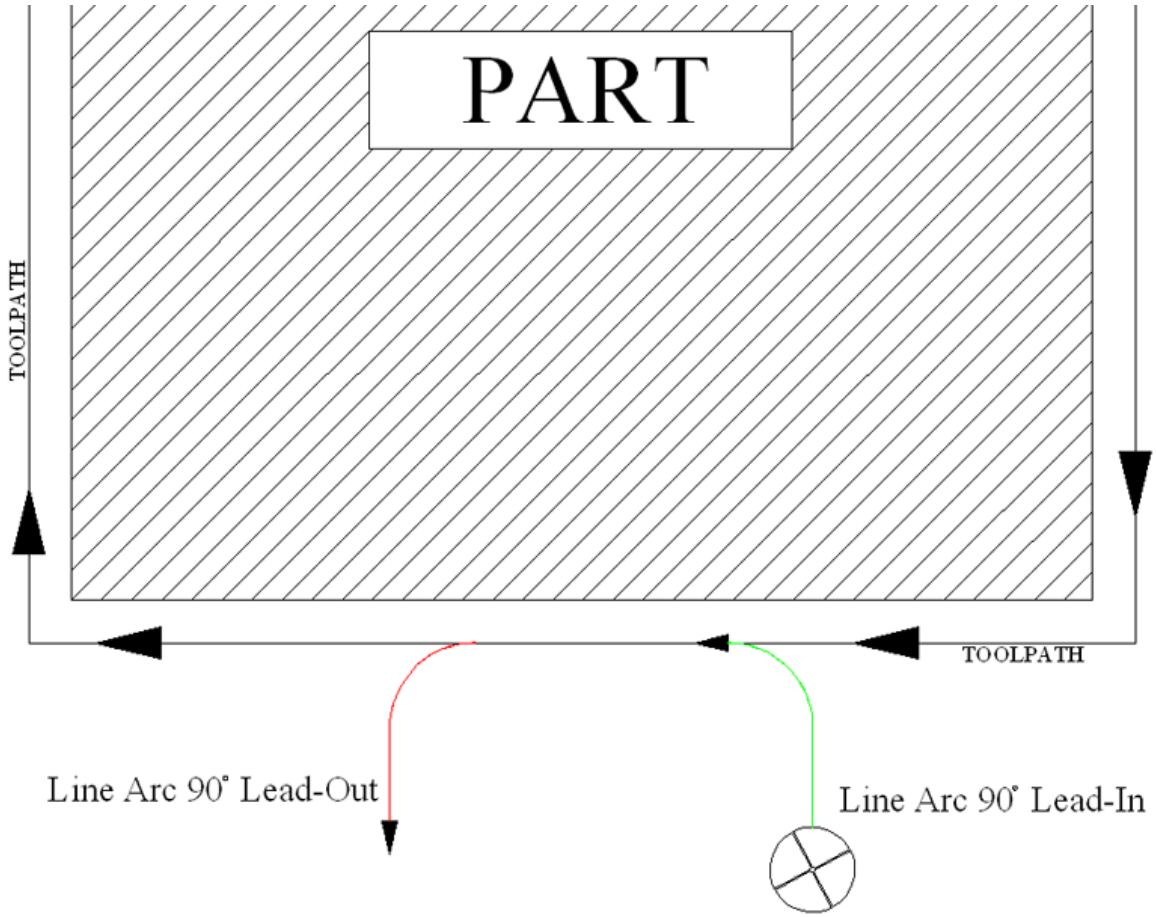
A180LI
Arc 180° Lead In. An arc will be generated with an angle of 180°, ending tangent to the start of the cut.

© 2011 CIM-TECH.COM, Inc



LA90LI
Line and Arc 90° Lead In. There will be a short line, then a 90° arc, ending tangent to the start of the cut.

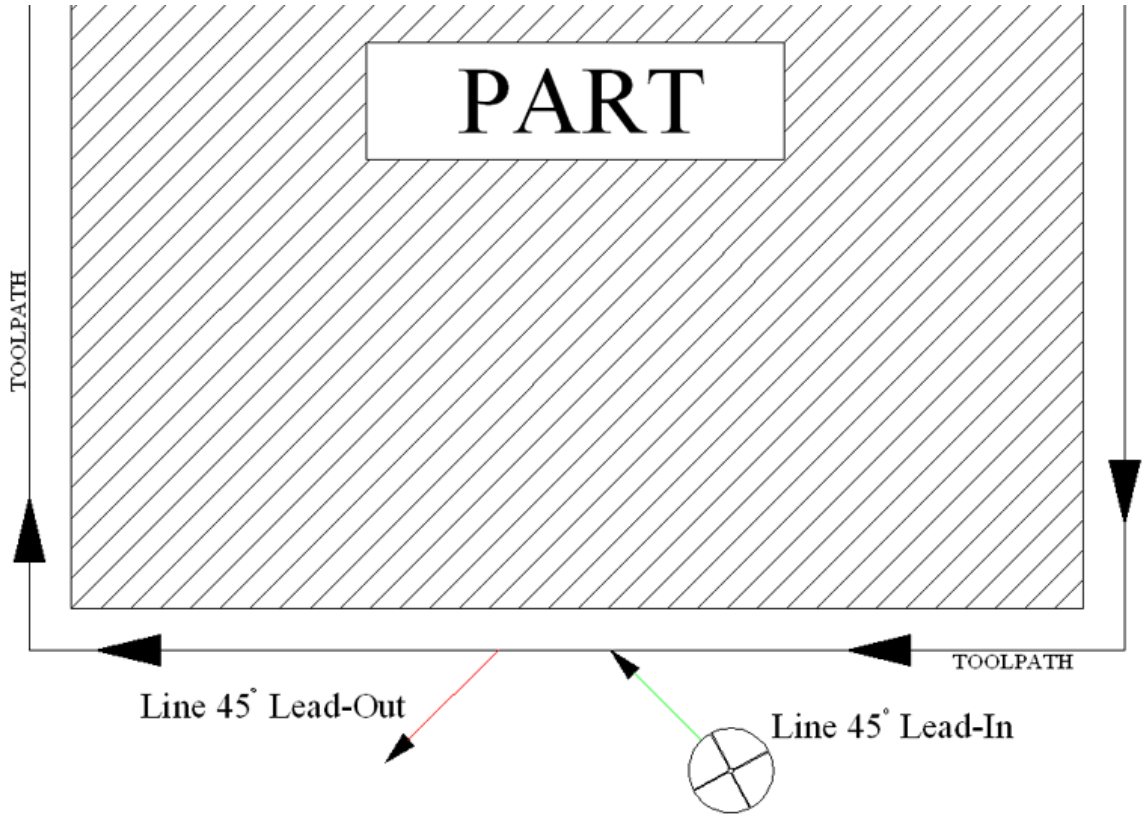
© 2011 CIM-TECH.COM, Inc



LN-45LI

Line 45° Lead In. This method generates a line at a 45° angle to the start of the cut.

© 2011 CIM-TECH.COM, Inc



li
This is a Router-CIM macro that will generate a line lead in, in various configurations depending on the cut settings.
This is not a user selectable option.

Creating Additional Leads

Additional Leads can be made in the /Router-CIM/Ncdwgs/(Knowledge Drawing based on the name of your postprocessor i.e. Mach1s, Mach2s, etc.) drawing by drawing a polyline on layer NC_LEADS ending at 0,0 for a lead-in. Make a block from the polyline with a 0,0 insertion point.

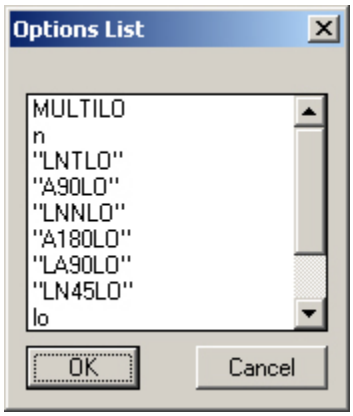
After the block is created you should erase the lead from the drawing and change the current layer back to Layer 0.
Save the drawing back to your /Router-CIM/Ncdwgs directory.
The next time you want to use the new lead, choose the Plunge Outside line leads cycle and enter the block name of the lead-in you chose to create, in the appropriate field in the Control Panel.

6.2.6 Lead-Out



This field defines the lead-Out block name. There are several available, but only some cycles will respond to the change of the Lead-Out edits.

© 2011 CIM-TECH.COM, Inc



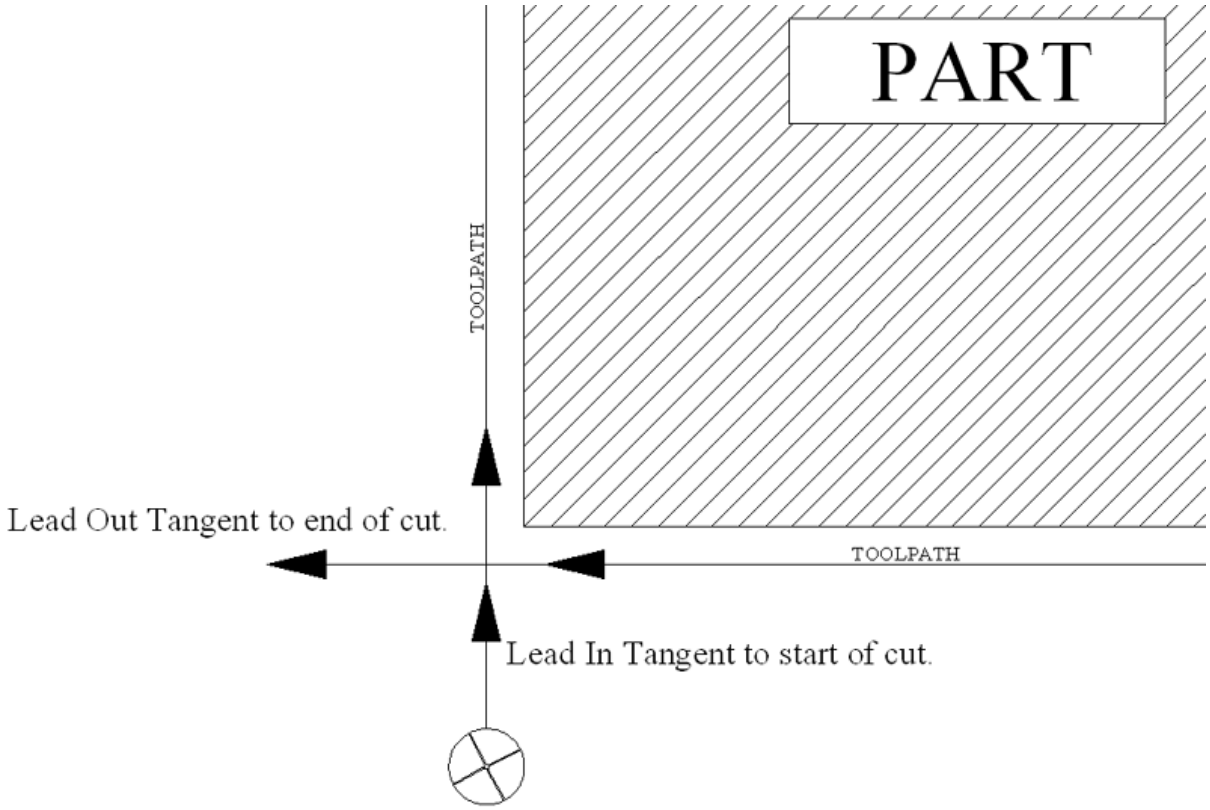
The options selected from the Options Box by default are:

MULTILO
Multilo is the default lead type for some cycles where there are multiple depth per pass, and Router-CIM must figure out how to move each cut down (usually ramping) between passes. This is typically not a user selectable option.

n
None. There will be no lead out move generated.

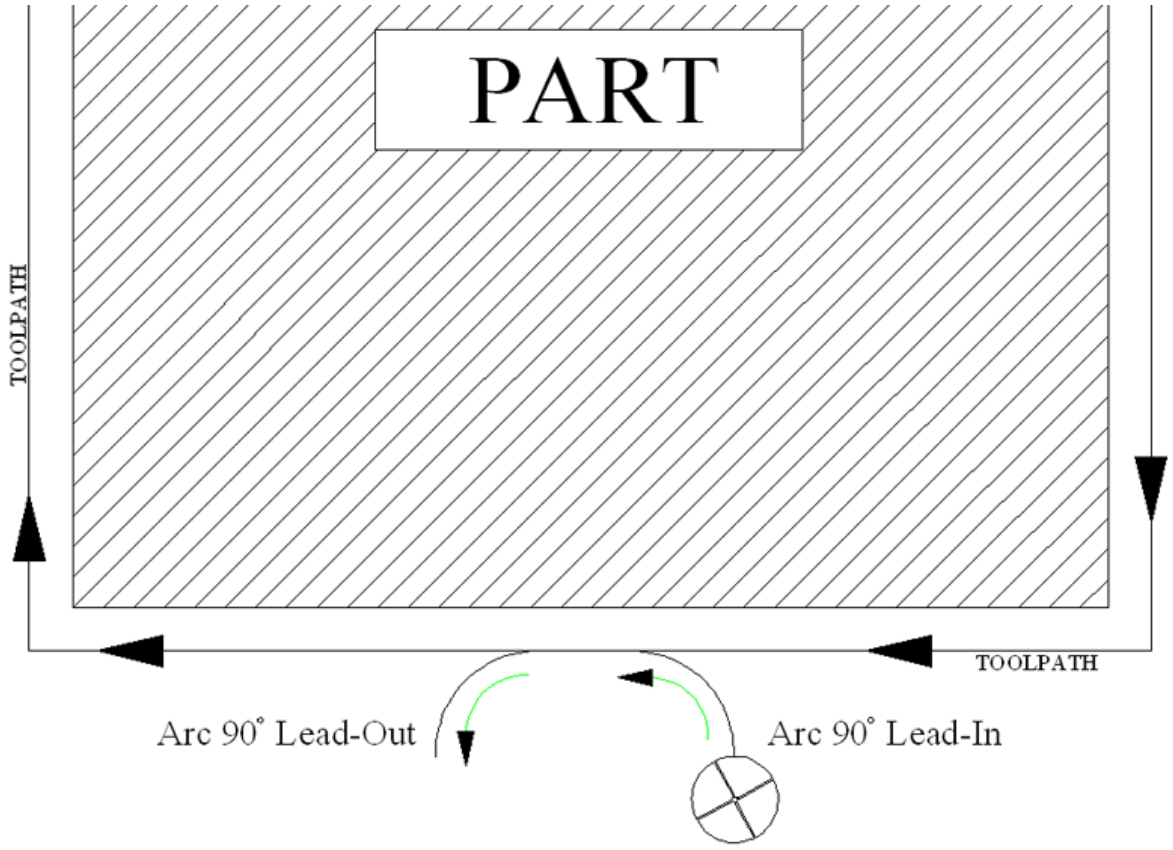
LNTLO
Line Tangent Lead Out. This generates a move tangent to the end of the cut.

© 2011 CIM-TECH.COM, Inc



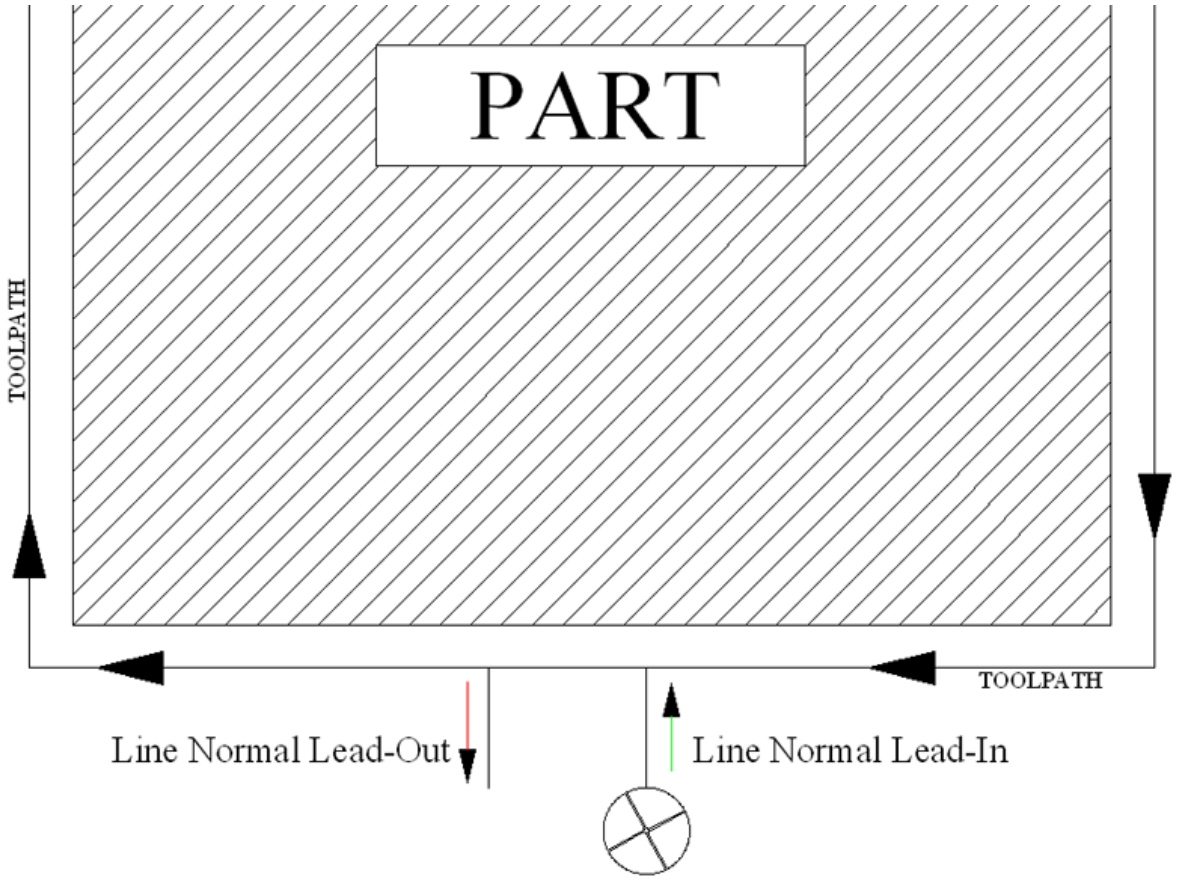
A90LO
Arc 90° Lead Out. Generates an arc of 90°, starting tangent to the end of the cut.

© 2011 CIM-TECH.COM, Inc



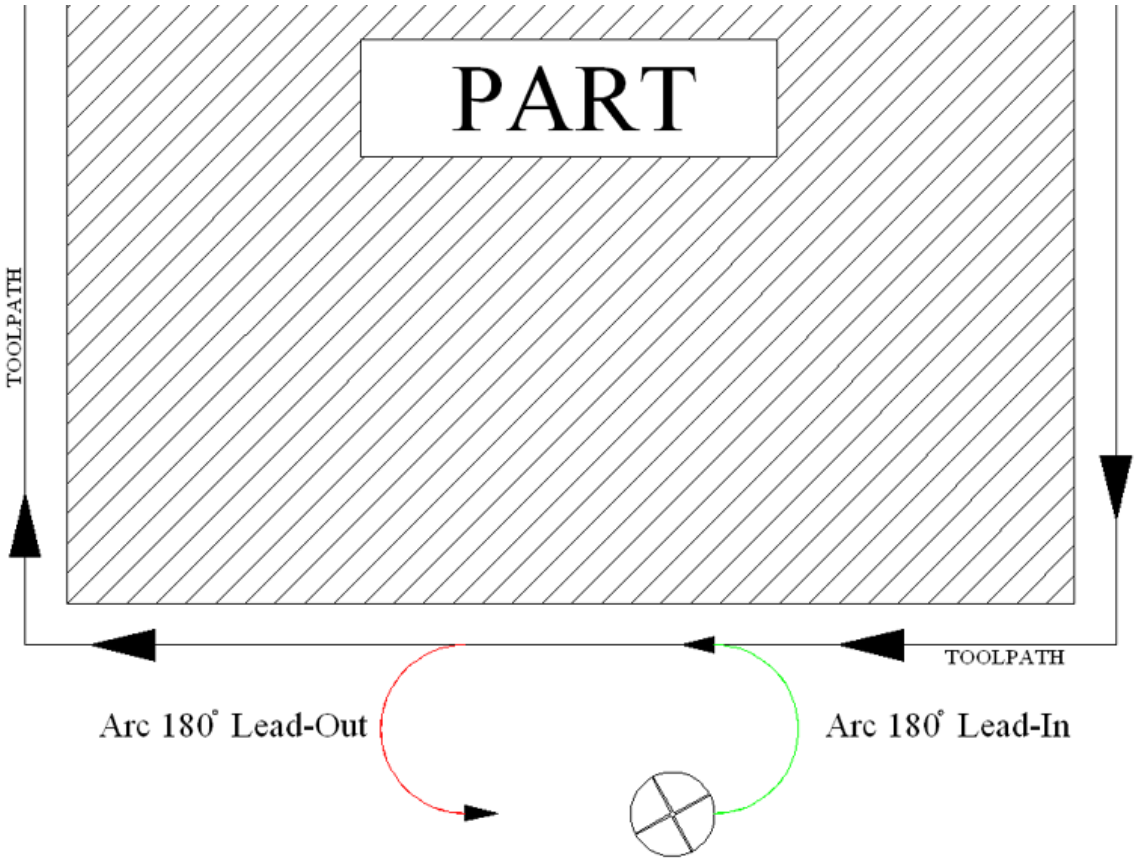
LNNLO
Line Normal Lead Out. The Line Normal is perpendicular to the end of the cut.

© 2011 CIM-TECH.COM, Inc



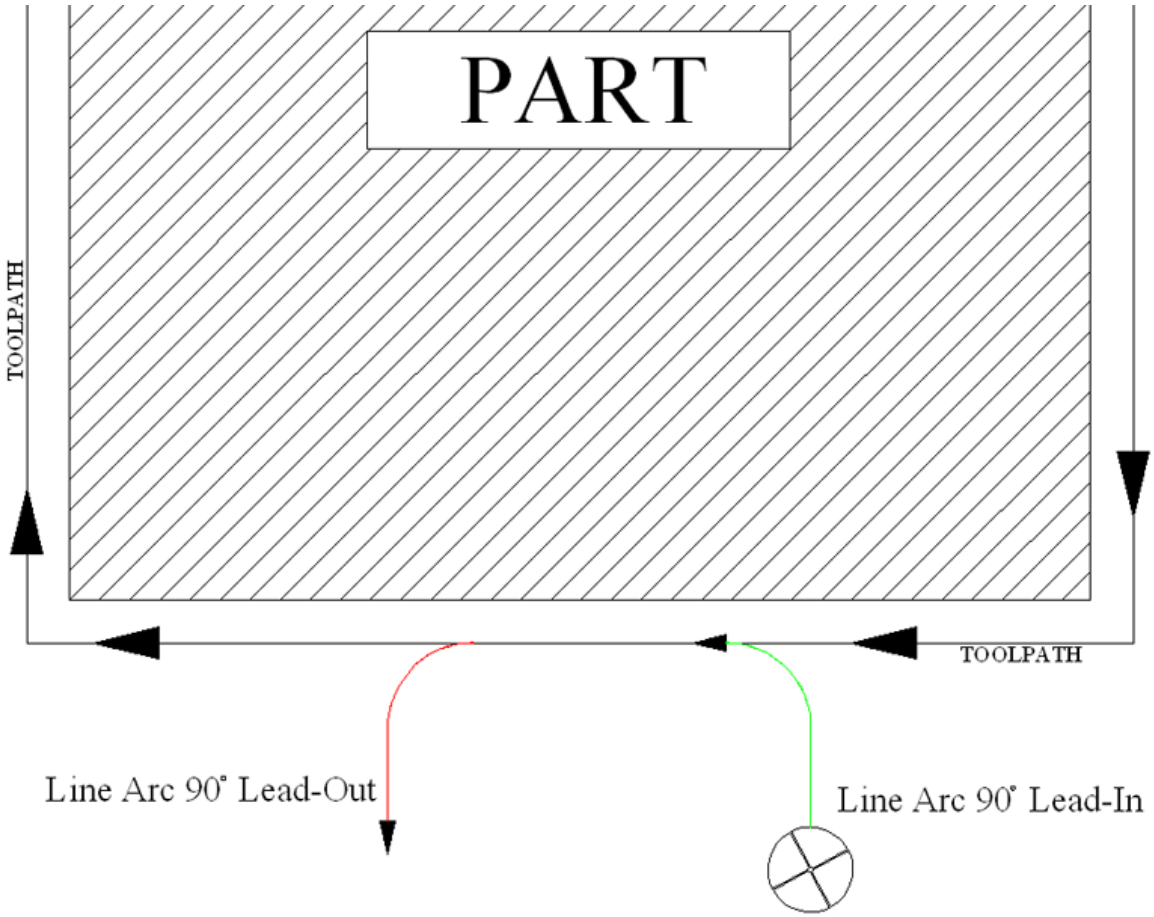
A180LO
Arc 180° Lead Out. An arc will be generated with an angle of 180°, starting tangent to the end of the cut.

© 2011 CIM-TECH.COM, Inc



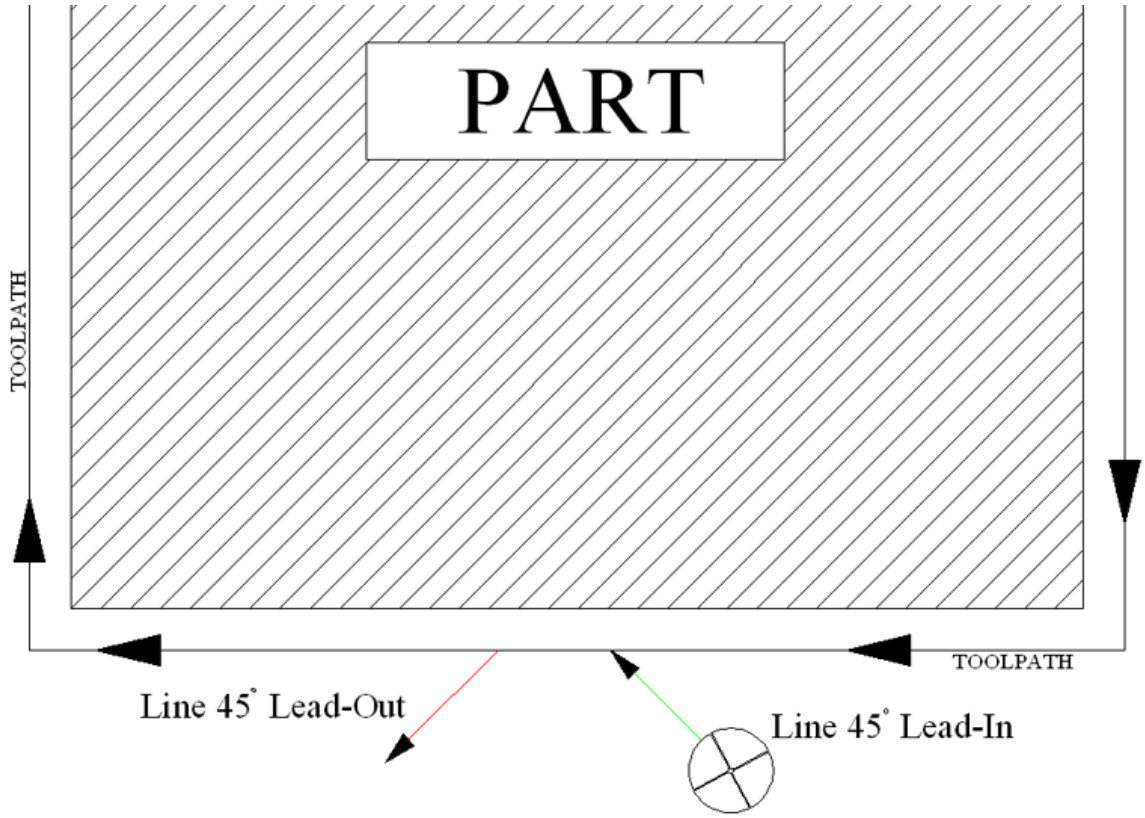
LA90LO
Line and Arc 90° Lead Out. There will be a short line, then a 90° arc, starting tangent to the end of the cut.

© 2011 CIM-TECH.COM, Inc



LN-45LO
Line 45° Lead Out. This method generates a line at a 45° angle to the end of the cut.

© 2011 CIM-TECH.COM, Inc



lo

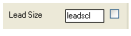
This is a Router-CIM macro that will generate a line lead out, in various configurations depending on the cut settings. This is not a user selectable option.

Creating Additional Leads

Additional Leads can be made in the /Router-CIM/Ncdwgs/(Knowledge Drawing based on the name of your postprocessor i.e. Mach1s, Mach2s, etc.) drawing by drawing a polyline on layer NC_LEADS beginning at 0,0 for a lead-out. Make a block from the polyline with a 0,0 insertion point.

After the block is created you should erase the lead from the drawing and change the current layer back to Layer 0.
Save the drawing back to your /Router-CIM/Ncdwgs directory.
The next time you want to use the new lead, choose the Plunge Outside line leads cycle and enter the block name of the lead-out you chose to create, in the appropriate field in the Control Panel.

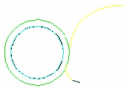
6.2.7 Lead Size



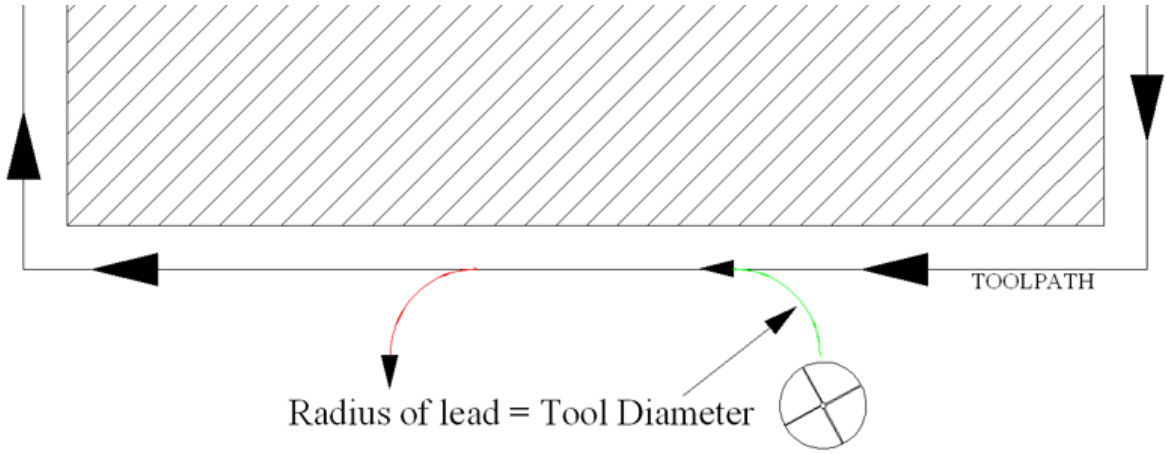
Use Lead Size to change the length of the leads. This field will affect both lead-in and lead-out if you put just one number in this field.
To change both leads separately, you can put two numbers in this field, separated by a space, and the first number will affect the lead-in and the second will affect the lead out.

© 2011 CIM-TECH.COM, Inc

Example: Lead Size = 2.0 0.5 will make the lead-in 2.0 and the lead-out 0.5. There must be a space separating the two numbers.
2.0 - lead-in
0.5 - lead-out



The default setting is a macro named leadscl, which will change the size of the leads according to the size of the tool.
On a Plunge Outside cut for instance, the arc leads will have a size equal to the diameter of the tool.

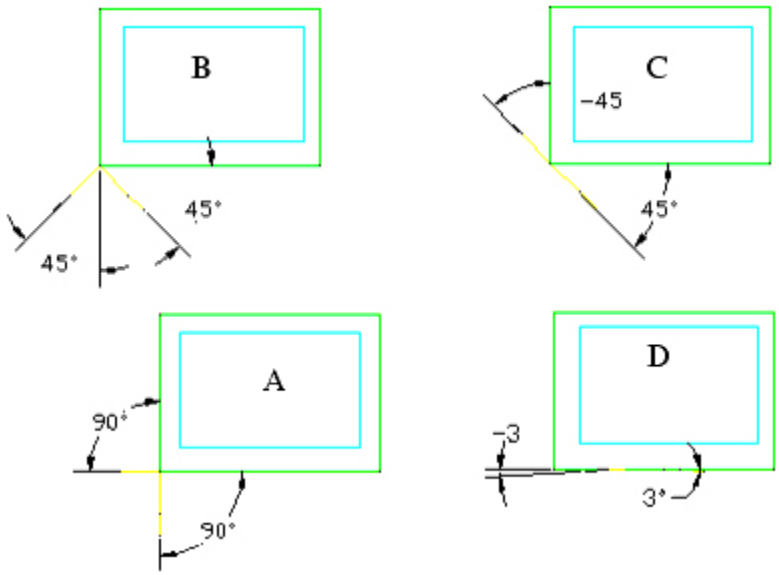


6.2.8 Lead Angle



Use Lead Angle to change the angle of the lead-in and lead-out. This field also will affect both lead-in and lead-out angles if you put just one number in the field. You can put two numbers in this field, separated by a space. The first number will affect the lead-in angle and the second will affect the lead-out angle.

© 2011 CIM-TECH.COM, Inc



The examples of lead-in and lead-out angles are:

. •A - Plunge Outside Line Leads cycle Lead Angle is left at 0 (default), which gives lead-in and lead-out of 90°.

. •B - Plunge Outside Line Leads cycle Lead Angle is set for 45° at lead-in and 45° at lead-out.

. •C - Plunge Outside Line Leads cycle Lead Angle is set for 45° at lead-in and -45° at lead-out.

. •D - Heli-Lead Outside cycle Lead Angle is set for 3° at lead-in and -3° at lead out. This would be used in a scenario in which you wanted the benefits of a Heli cycle, but you don't want the lead to bring the tool straight into the part. If the tool comes in at a slight angle it will prevent undesirable "witness marks."

6.2.9 Lead Feed



This sets lead-in and lead-out feedrates. The default is 0.5, Router-CIM's standard 50% feedrate for lead-in and lead-out.

Setting the parameter to a number less than 1.0 is a percentage of max feedrate set in the Control Panel.
Setting the parameter to a number greater than 1.0 will give you an exact feedrate.

Setting the Leadfeed to 180 on a cut where the normal feedrate is 600, would allow the feedrate to be slower on the lead in than normal. Normally the lead in would be 300. In the code the lead move feedrate would appear on the same line as the lead.

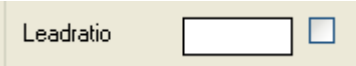
```
%
:1234
N1 G00 G17 G20 G28 G40 G80 G91 Z0 M5
N2 G90
N3 G52 X0 Y0 Z0
N4 G08 P1
N5 M08
N6 (ROUTER-BIT .5 DIA.)
N7 G28 G91 Z0 M05
N8 G90 T2001 M06
```

© 2011 CIM-TECH.COM, Inc

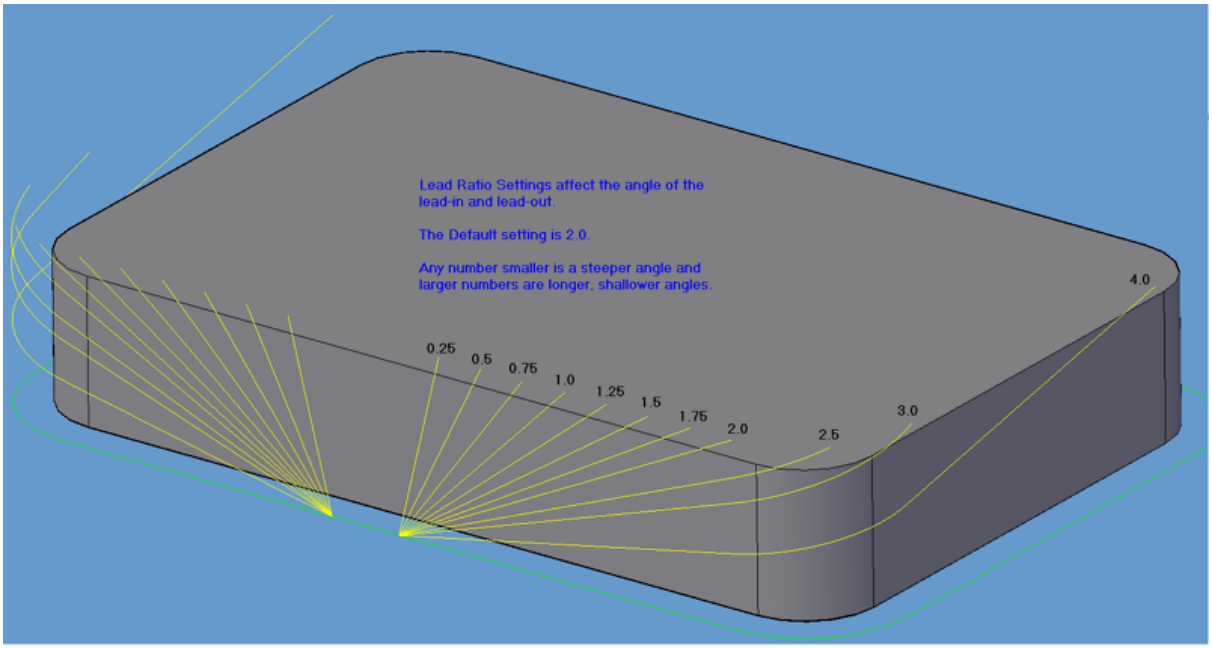
```
N9 T102
N10 M03 S18000
N11 G00 G17 G55 X5. Y-.249
N12 G00 G43 H1 Z.25
N13 G01 X3. Z-.75 F180.   <<<<<< Lead In feedrate here.
N14 X.5 F600.             <<<<<< Normal cut feedrate here.
N15G02 X-.249 Y.5 J.749
```

You could also set the Leadfeed in the example above to 0.3 and get the same result as 180 is 30% of 600.

6.2.10 Lead Ratio

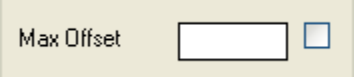


Lead Ratio determines the angle of the ramp in Z during the lead in and lead out. You can specify the Lead Ratio as a number that reflects the percentage of the angle from its default. That means that if you want a lead that is twice the normal ramp length (shallower angle) enter 2. If you want a lead that is steeper than the default, enter .5.



The Lead Ratio value effects both the lead in and the lead out. There is no way to have a separate angle for the lead-in and lead-out. If two numbers are input, only one is used.

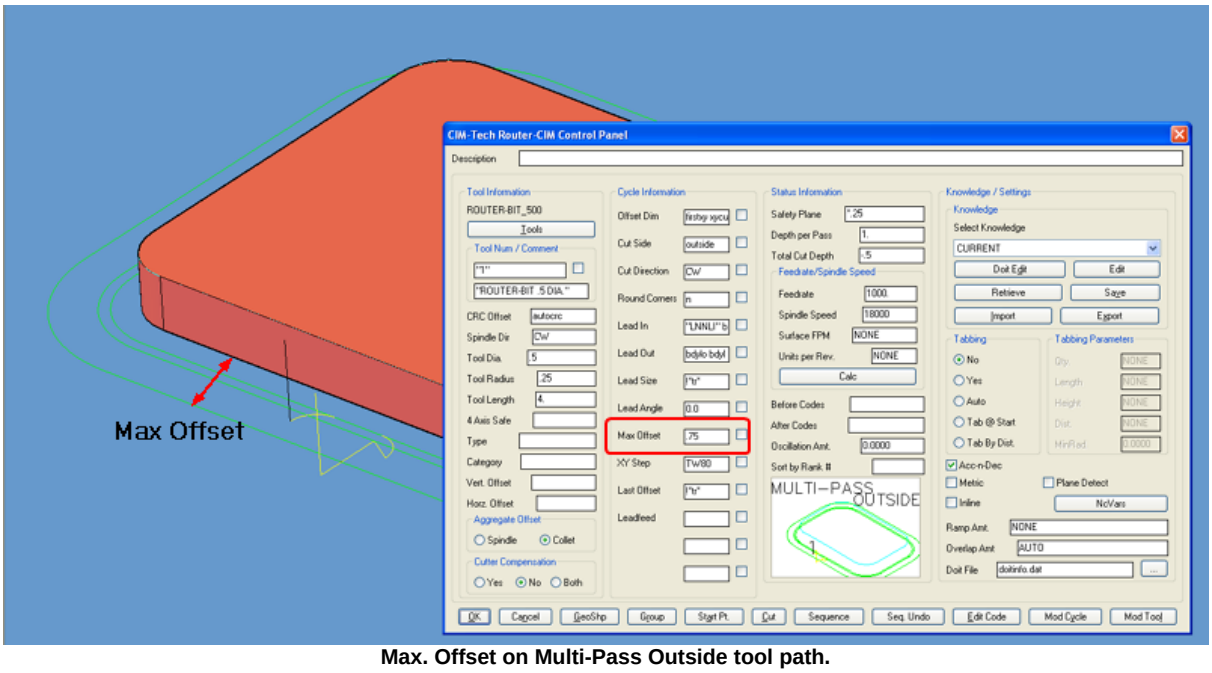
6.2.11 MaxOffset



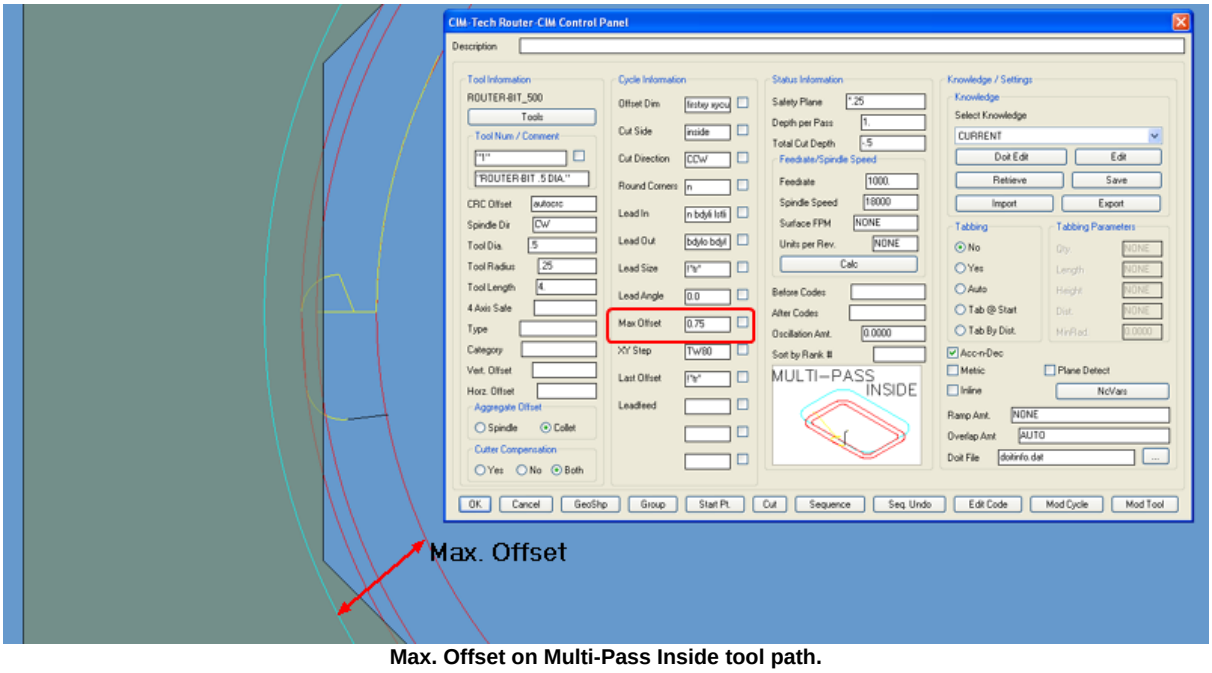
Max Offset is the distance from the start point to the start of the first pass on the cut. If the cut cycle is Multi-Pass Inside, then the Max Offset will be to the inside of the shape by the specified amount. If the cycle is Multi-Pass Outside, then the Max Offset will be from the start point to the outside of the shape by the Max Offset amount.

© 2011 CIM-TECH.COM, Inc

This is only the distance for the first cut. All subsequent cuts will be determined by either XY Step or Last Offset.



Max. Offset on Multi-Pass Outside tool path.



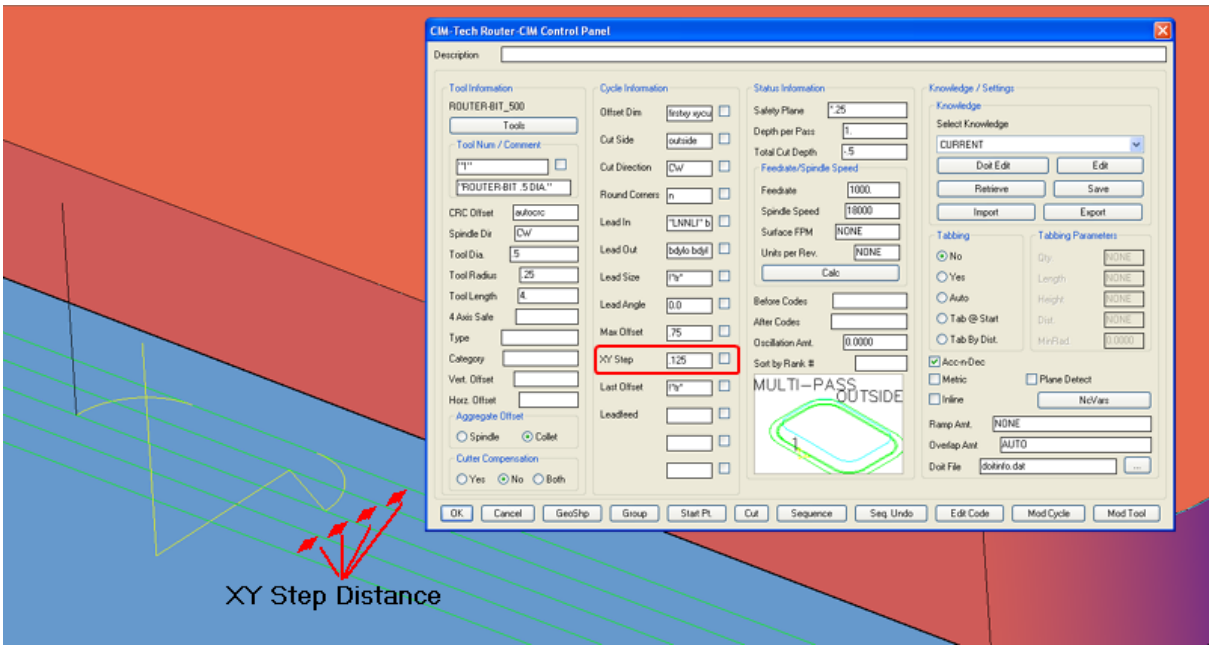
Max. Offset on Multi-Pass Inside tool path.

6.2.12 XYStep



The XY Step is the distance between each tool path from the first to the last (excluding the finish pass) on the Multi-Pass Inside and Multi-Pass Outside cycles.

© 2011 CIM-TECH.COM, Inc



You can specify the distance as a numeric value, or you can make a task to calculate a step over amount and place the task name in the XY Step field. An example of a task is TW80, which looks at the tool diameter field and then places an amount equal to 80% of that value in this location.

The task must be in proper lisp-task form and placed in a {taskname}.tsk file in the ncprog\ncsources folder. For example, make a file called TW60.tsk with Notepad and place it in the C:\Router-CIM\Ncprog\Ncsources folder. Place the following code inside the file.

(* *TW* 0.6)

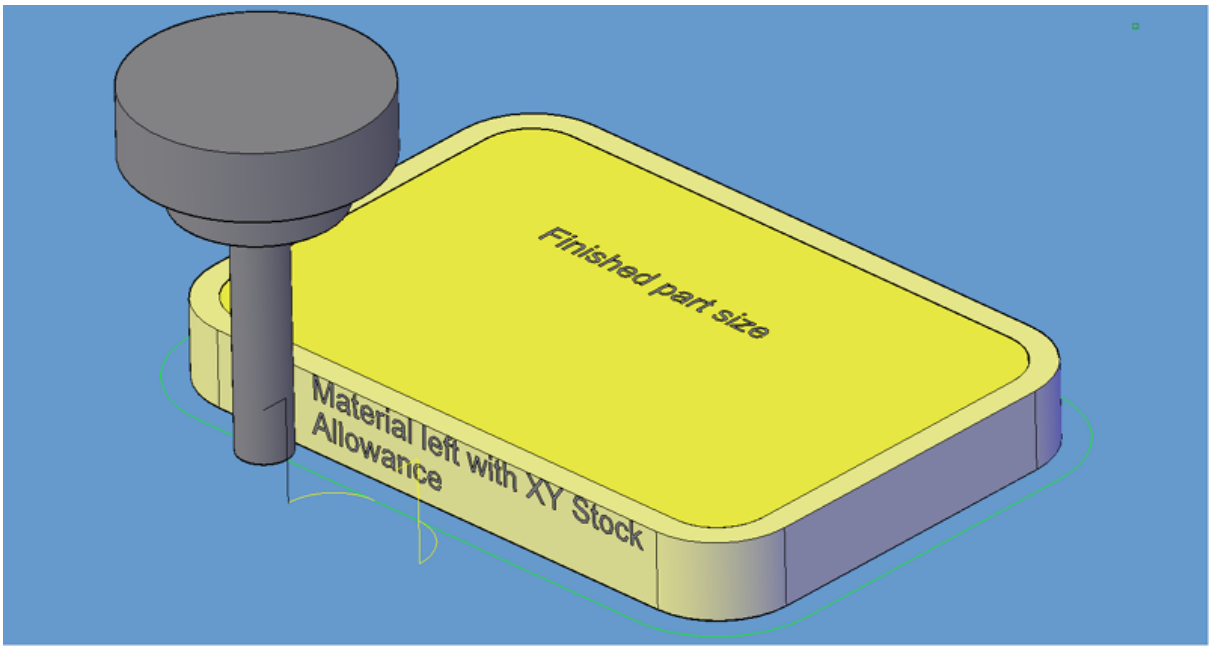
Save the file and then place the syntax TW60 in the XY Step field after starting Router-CIM again. The step over will then be 60% of the tool diameter. The code reads as follows:
(multiply ToolDiameter by 0.6) so (* 0.5 0.6) would be 0.3. So a step over of 0.3 would be set for the spacing for any pass that is not the finish pass.

6.2.13 XY Stock Allowance



Placing a value in this parameter will offset the tool path to leave material for a finish pass. For instance, placing .125 in the XY Stock Allowance and cutting a 6.4 x 4.0 shape will actually leave a part that is 6.25 x 4.25, by adding .125 to the offset of the tool path all the way around the part.

© 2011 CIM-TECH.COM, Inc



XY Stock Allowance.

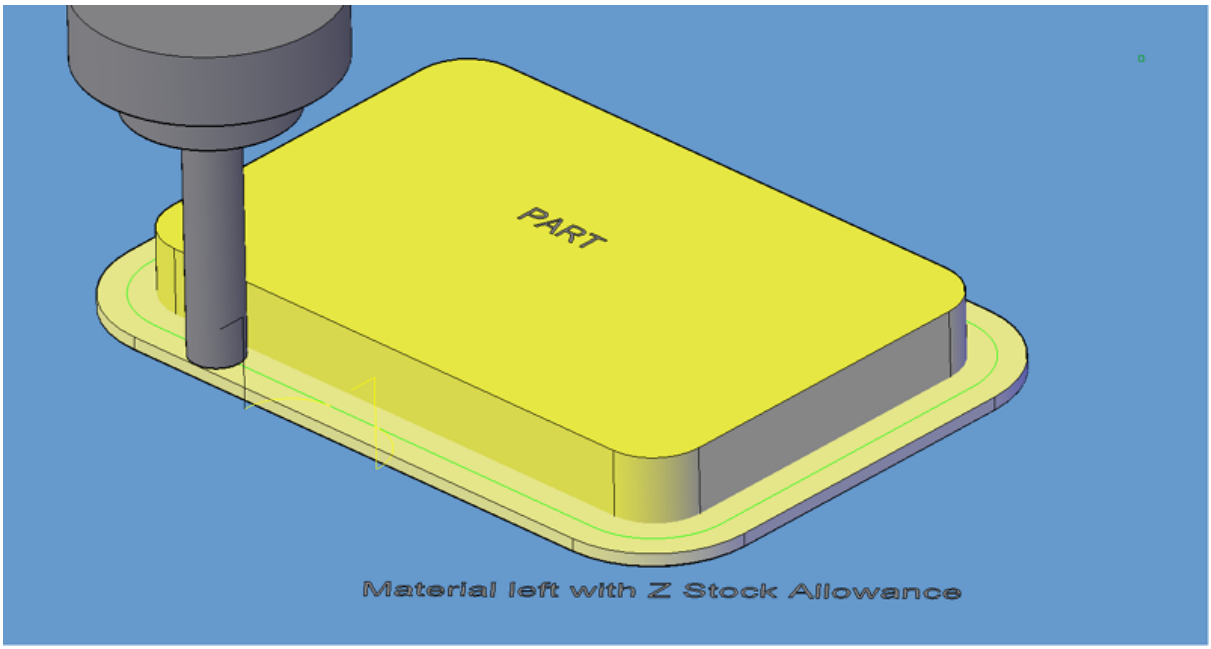
6.2.14 Z Stock Allowance



Placing a value in Z Stock Allowance will change the Total Cut Depth by the number entered. You can use this if you want to leave a small amount of material on the bottom of a part, or if you intentionally want to overcut a part to be sure it is cut all the way through.

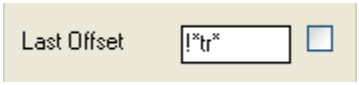
Entering a positive number will move the tool path UP in Z, leaving more material for a finish pass. Entering a negative number will move the tool path DOWN in Z, past the normal Total Cut Depth.

© 2011 CIM-TECH.COM, Inc

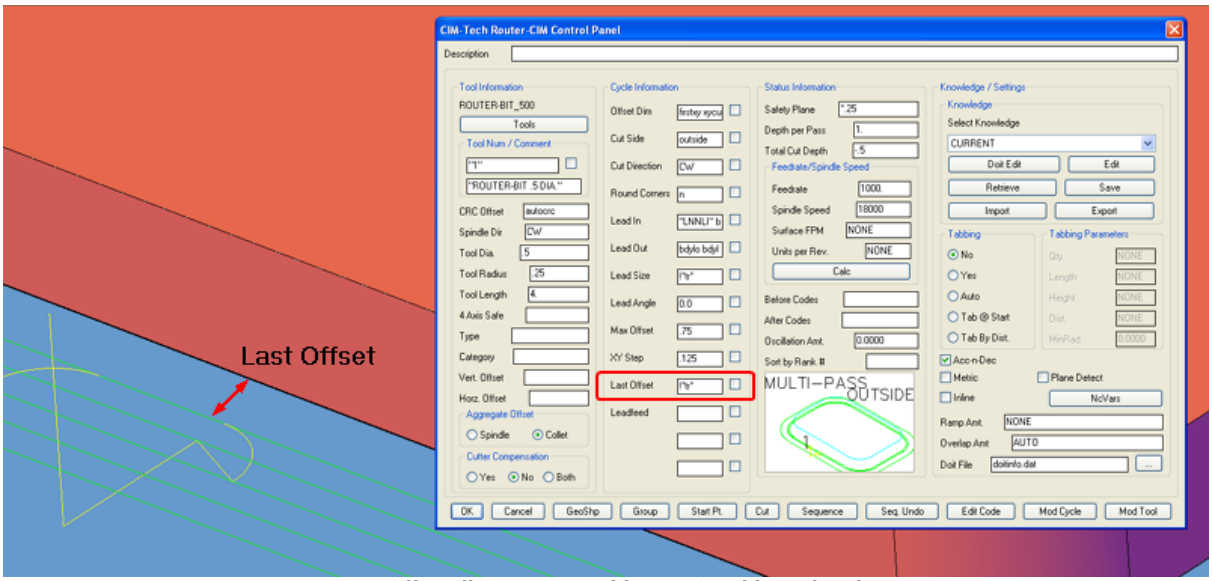


Z Stock Allowance

6.2.15 LastOffset



The Last Offset is the distance from the finish pass to the edge of the shape being cut. This can be any numeric value, but if it exceeds the XY Step value, the cutter will not contact the part on the finish pass. Typically this is set to the radius of the tool in a rough-cut, finish-cut scenario. You can leave some material on the part if you want to clean up the part with a finish cutter.

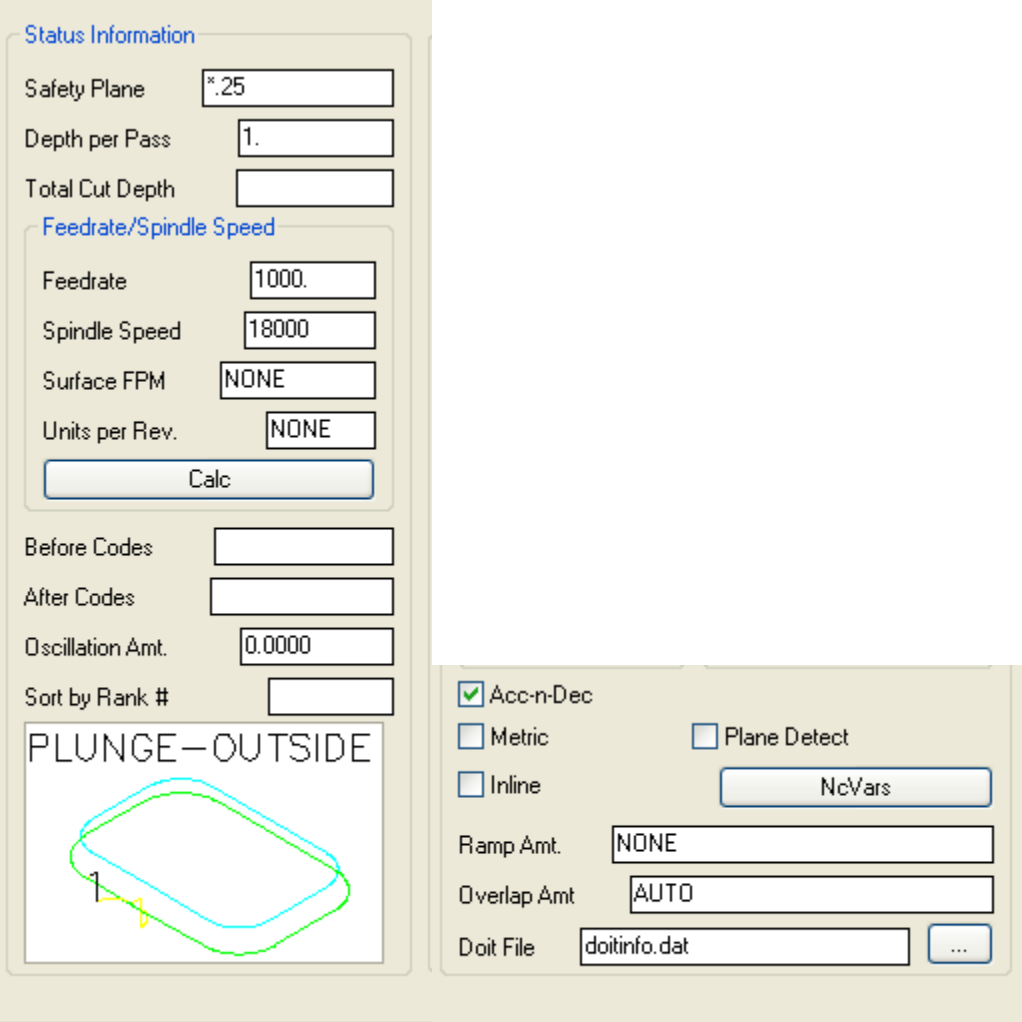


Last Offset distance on Multi-Pass Outside tool path.

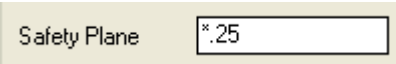
© 2011 CIM-TECH.COM, Inc

7 Status Information

Status Information is data related to the feed rates, spindles speeds, depths of cut etc. for the type of material and cutters available. The field has several parameters and each is explained in the following sections.



7.1 Safety Plane



This field is the Z axis SAFETY PLANE to which the tool retracts to between CUTS, or after a cutting cycle. The code generated will have a Z move to this location in rapid, then a Z move to the feed distance to material (parameter #29 in Modify Tool page), and the next Z axis move will have a feed rate attached to it. After CUTTING, the code will generate a move back to this location and then proceed either to the next Cut or to the next tool, etc.

Note Placing an asterisk (*) in this field before the value will insure that the move is absolute, instead of incremental. For instance using *0.25 will return the cutter to .25 above Z0, however, using 0.25 (no *) will return the cutter to a point .25 units above the last Z position only.

© 2011 CIM-TECH.COM, Inc

7.2 Depth Per Pass



This field allows multiple depths of Cut in a single tool path. By setting this number to a value less than the Total Depth of the Cut, you will have multiple passes in the material.

For example, if you have 1" thick material and need to take three passes to cut through the part, you could set the Depth/Pass field at .4 (any number between .35 and .5 is valid) and the Total Depth at -1.0. The code generated will produce the first pass at -.4, the second at -.8 and the third pass at -1.0. In the standard Router-CIM cycles the tool paths will ramp down between the Cuts.

7.3 Total Cut Depth



The Total Cut Depth is the final depth, in Z, you want the tool to cut down to, regardless of the number of passes made. Router-CIM uses this number to calculate the Z axis moves for the Total Depth to cut into the material. If the Depth/Pass field has a number smaller than this, Router-CIM calculates the number of passes necessary to reach this depth. You may enable Router-CIM to calculate the depth automatically for you based on the thickness you give a part. To do this place "A" in the Total Cut Depth field, and give you part thickness. Router-CIM will use that value for the Z depth. Remember to give your part negative thickness!

****Note****
Also, when you give your parts negative thickness, you can use a forward slash (/) followed by a negative value (/-.01 for example) in this field. Router-CIM will take the negative part thickness (-.75 for example), and the negative value following the slash and calculate the Total Cut Depth. In this case the part would be cut to -.76.

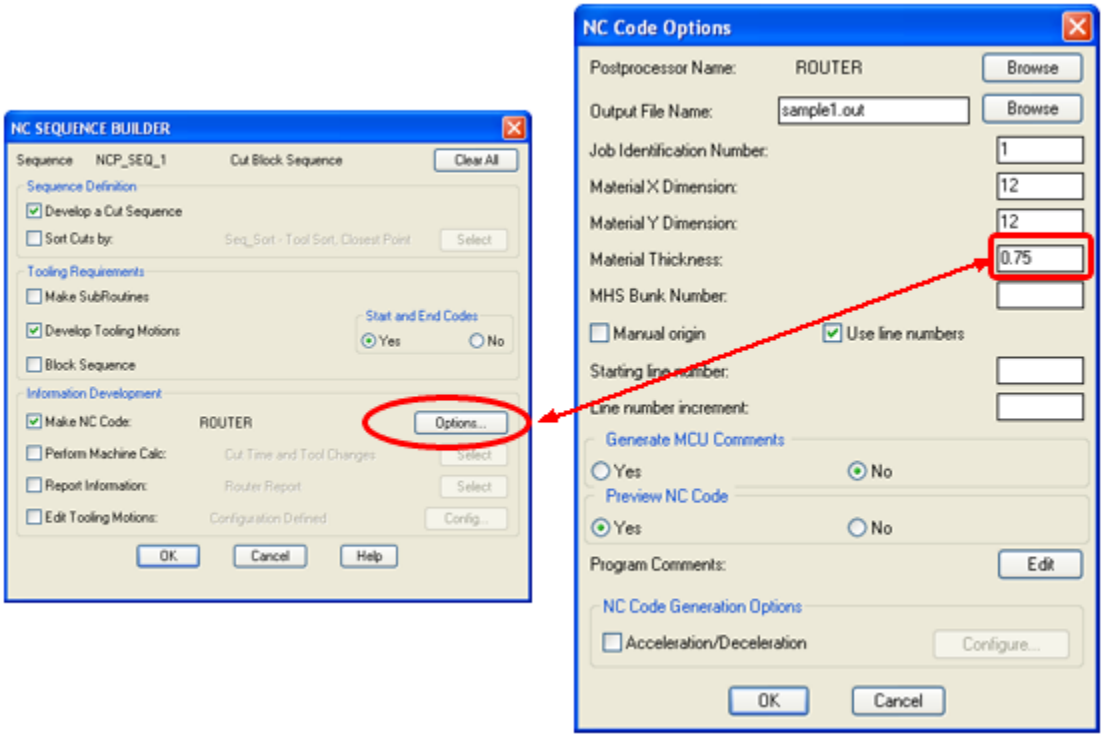
The following section on Material Thickness Compensation can directly affect Total Depth.
.

7.4 Material Thickness Compensation

If you program Z0 off the top of the spoil board instead of using the top of your part, you still program as if the top of part is Z0. To set Router-CIM for spoil board as Z0, put in your material thickness in the Material Thickness Compensation field shown below. The interface below is arrived at after a cut is made and you are ready to Sequence.

From the Sequence screen you choose the Options button to the right of "Make NC Code."

© 2011 CIM-TECH.COM, Inc



7.5 Feedrate



This field specifies the cutting maximum Feedrate in either inches per minute or millimeters per minute, depending on the mode you are programming in. See the chapter on Advanced Settings for information on how to program variable feed rates.

7.6 Spindle Speed



This field sets the spindle speed in rpm's. This is a modal field to many machine tools, so if you do not change this field for each Cut with the same spindle, you may only see the output for this setting once although you have made more than one Cut with the same spindle.

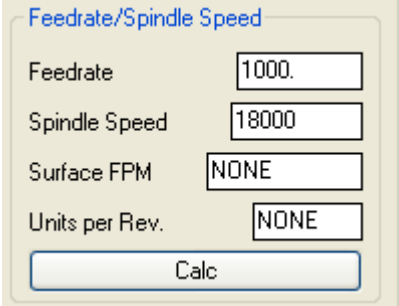
7.7 Surface FPM and Units Per Rev.



These fields are useful for calculating feed rates and spindle speeds based on a tooling/material parameter referred to as Constant Surface Speed.
Surface Feed per Minute and Units per Rev are variables based on the size of each tool, that your tool bit manufacturer can supply you with.
The calculation of Feedrate and Spindle Speed can now take place using the Surface FPM and the

© 2011 CIM-TECH.COM, Inc

Units per Rev fields. Simply input the Surface Feed per Min. and Units Per Revolution in their respective fields, click on the Calc button, and Router-CIM will input the correct feedrate and spindle speed automatically.



This Feedrate and Spindle Speed can be saved with your knowledge and used in future cutting operations.

****Note****
The actual settings input into the Surface Feed per Min. and the Units per Rev fields are not saved in the knowledge as they are used for calculation purposes only!)

Following is some information about Feeds and Speeds copied from the Onsrud Cutter© manual that may prove helpful in better understanding Surface Feed per Min. and Units per Rev.

To understand the concept of feeds and speeds, it is necessary to visualize what is occurring at the cutting edge of the tool. A chip of material is being removed from the base part. The size and thickness of the chip is controlled by the speed of the rotation speed of the spindle and the forward movement caused by feeding the tool into the material. If there is one flute, then the chipload is equal to the amount of travel in one revolution of the spindle. If there are two flutes, then there are two chips equal to one-half of the amount of travel in one revolution. If there are three flutes, then the chip load is one-third of the amount of travel in one revolution.

Most of the energy expended during these reactions is released as heat. Heat is one of the major factors in tool wear. The most effective way of getting rid of the heat is by having it carried away with the chip. This can be accomplished by cutting larger chips which both dissipate heat as well as yield a high quality part edge finish due to minimization of re-cut chips. This is possible if you have a tool that possesses a geometry that allows for both speed and finish characteristics.

After running a program, you can determine the actual feed value by timing one part or a total cycle time for a complete table of parts. The formula will be provided below.
There is another indication of proper feeds and speeds, and that is the tool temperature. After a run of parts, and after the spindle stops, check the temperature of the tool. If it is hot or warm to the touch, then the feed is too slow or the spindle speed is too high. If a proper speed and feed is used the tool should be at or near room temperature. Remember heat is what breaks down the cutting edge of a tool.

The first change to make is to the feed speed. This is the controlling factor in productivity. If the feed rate is at its maximum due to part configuration, hold down capabilities, software limits, or machine limitations, then the spindle speed should be lowered. This does two things; 1) It increases the chip thickness and 2) It lowers the number of times the cutting edge is presented to the material. This second factor can be a major factor in increased tool life if this tool in this material has a limited number of cuts per sharpening. This could increase tool life by 15 to 20%. It also reduces the spindle bearing temperatures by reducing heat transmitted into the spindle.

Chip Load (Inches) = Feed Rate (IPM)
RPM x No. of Flutes

© 2011 CIM-TECH.COM, Inc

Feed (IPM) = RPM x Number of Flutes x Chip Load

Spindle Speed (RPM) = Feed Rate (IPM)
Number of Flutes x Chip Load
For Time Studies and True Average Chip Loads Use The Following:
Actual Feed Rate (IPM) = Circumference of the Part (Inches) x 60 Run Time (Seconds)

7.8 Before Codes and After Codes



If a given code is needed at the beginning and end of a program (or part), such as a M08 (air blast on) and a M09 (air blast off), that code can be entered in the Before Code and After Code fields. This eliminates the need for inserting it manually in a text editor and the possibility of forgetting to add the code (Note: You must enter the codes correctly, for example M08 not M8).

7.9 Oscillation Amt.



When selected, this option will create a tool path that varies its depth in the Z Axis constantly throughout the Cut. This ensures that the cutter will not be subjected to sustained frictional pressure at any one location, such as at the point where the cutter contacts a laminated surface.

Entering an amount in this parameter will produce a tool path that continually ramps either up or down until it reaches the depth given in the Total Depth field if the number is positive , or the depth of the Cut minus the Oscillation Amt if the number is negative.

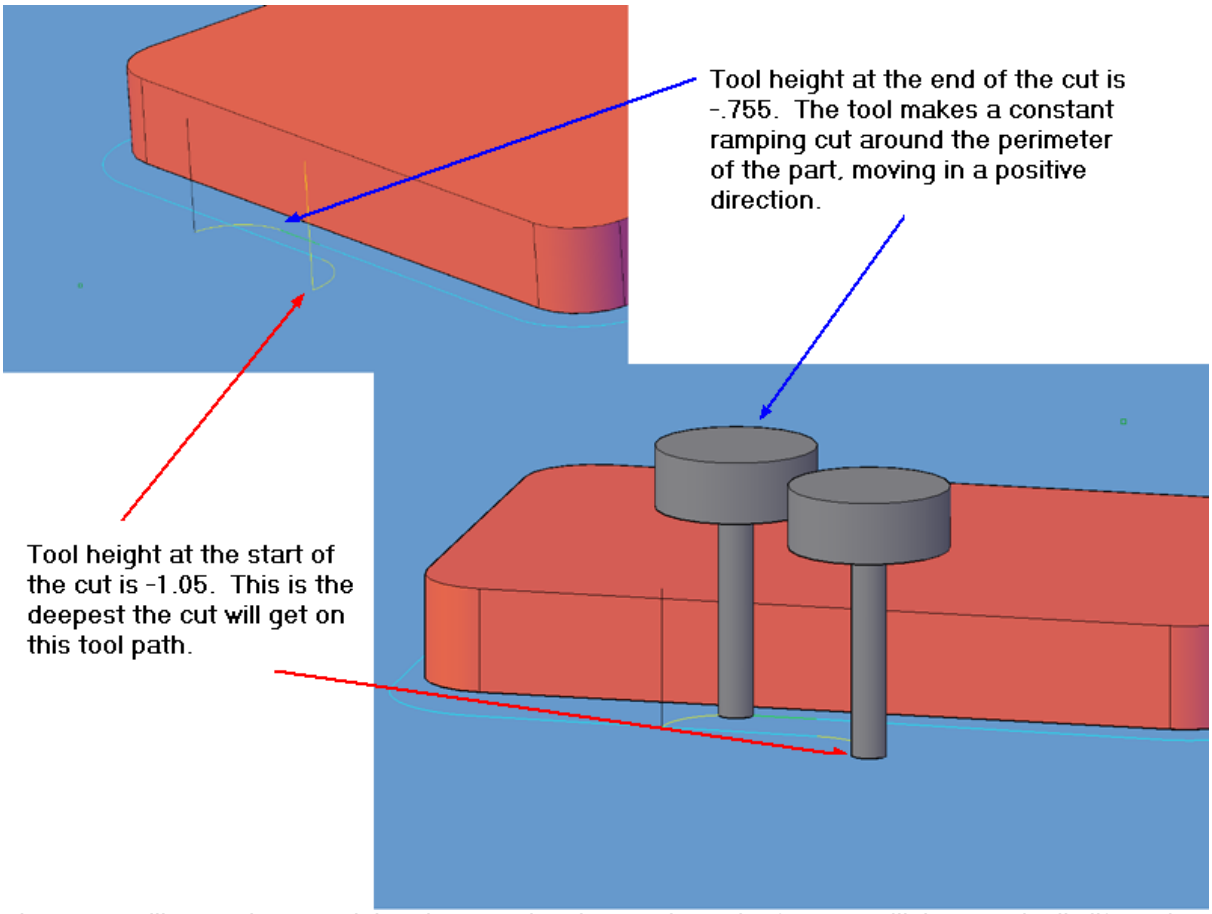
The following example will produce Cuts of various depths to allow for minimum tool wear and minimum removal of spoil board material.

EXAMPLE 1:

Material is ¾" thick.

- Set Total Depth to -1.05
- Set Oscillation Amt to 0.25

© 2011 CIM-TECH.COM, Inc



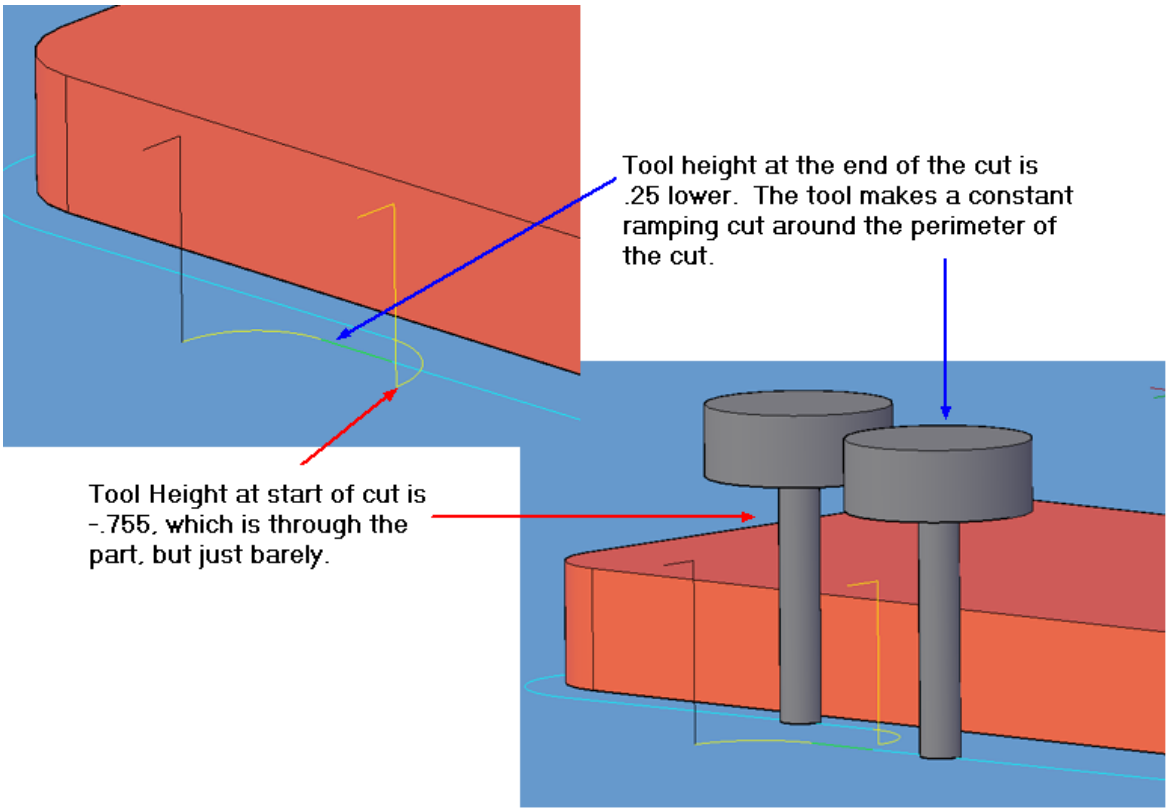
The cutter will enter the material and proceed to the Total Depth of -1.0. It will then gradually lift up, in the Z Axis until it has reached a Z depth of -.75 by the end of the Cut.
Any laminate or other characteristic of the material that would have caused heavy wear to one portion of the cutter has been distributed over ¼" of the cutting surface (which is the amount entered into the Oscillation Amt field).

EXAMPLE 2:

Material is ¾" thick.

- Set TOTAL DEPTH to -.755
- Set OSCILLATION AMT to -0.25

© 2011 CIM-TECH.COM, Inc



The cutter will enter the material and start the Cut at -.755. It will then increase the depth until a total of -1.05 at the Cut end, and then retract.

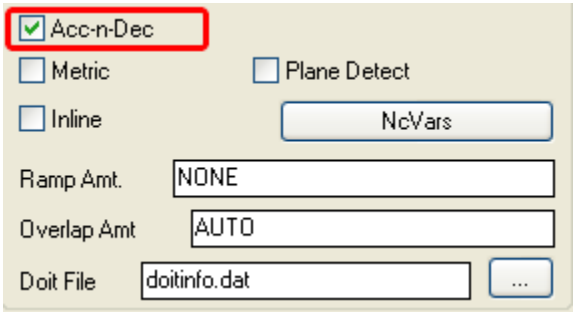
7.10 Sort by Rank



Rank is a numeric setting that allows tool paths and knowledges to be ordered in the code according to value, lowest to highest. When sorting by rank, the lowest rank number is cut first. You can make a tool path with rank 1, then another with rank 2. If you want a tool path to go between 1 & 2, make this tool path rank 1.5. Simply fill in a rank number before making a cut. Also note in the following sorting tasks, that area is defined as starting with the smallest and going to the largest.

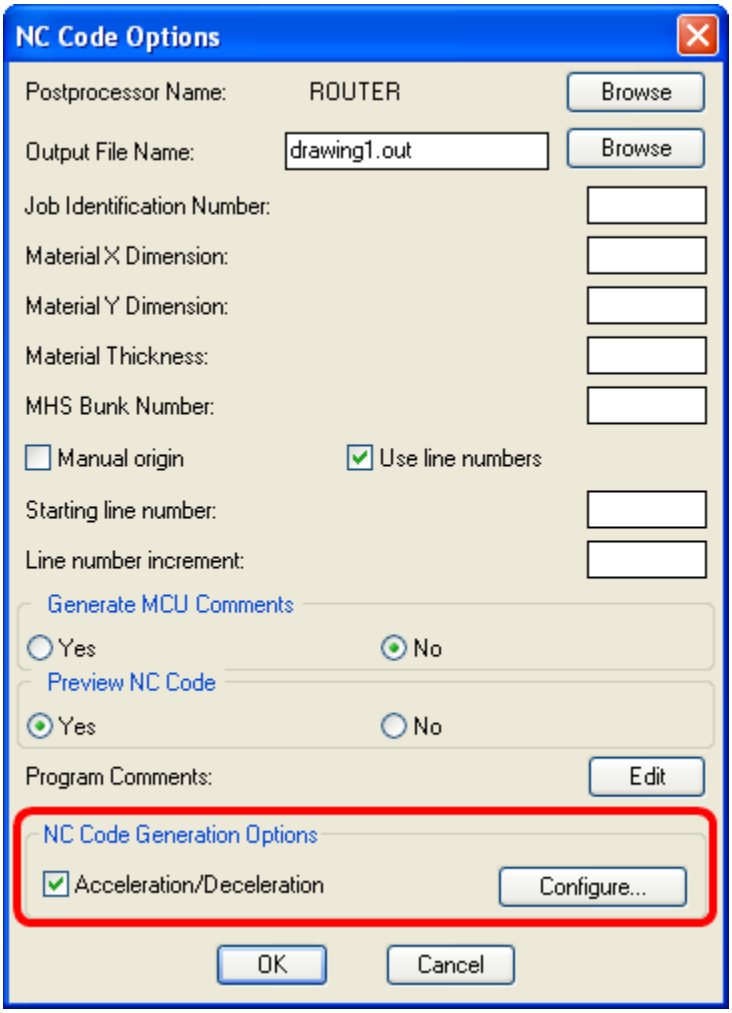
© 2011 CIM-TECH.COM, Inc

7.11 Acc-N-Dec



Checking this box enables the Acc-n-Dec feature at the cut level. Acc-n-Dec will output feedrate changes in the code depending on the geometry the tool is cutting and the maximum feedrate programmed.

Check this box to apply Acc-n-Dec feedrate changes to the next tool path you make. To get the feedrate changes in the code, you would also check the Acc-n-Dec button in the Sequence Options.



If you wish to disable Acc-n-Dec later, de-select Acc-n-Dec in the Sequencer to skip insertion of feedrate changes for all tool paths in the program.

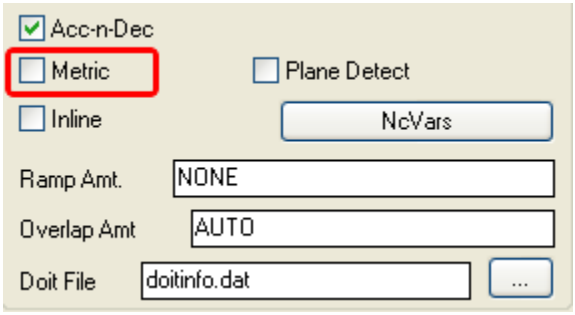
© 2011 CIM-TECH.COM, Inc

In the code example below, each of the feedrate changes have been highlighted.

```
%
: 1 (ACCDEC SAMPLE 1)
N1 G00 G17 G20 G28 G40 G80 G91 Z0 M5
N2 G90
N3 G52 X0 Y0 Z0
N4 G08 P1
N5 M08
N6 (ROUTER-BIT .5 DIA.)
N7 G28 G91 Z0 M05
N8 G90 T2001 M06
N9 T102
N10 M03 S18000
N11 G00 G17 G55 X43.9728 Y13.2444
N12 G00 G43 H1 Z.25
N13 G41 D01 G01 Y13.2559 F350.
N14 Y13.5078 F500.
N15 Y13.5194 F350.
N16 Z.2385
N17 Z-.4885 F500.
N18 Z-.5 F350.
N19 G03 X43.4728 Y14.0194 I-.5 F47.5
N20 G01 X43.4014 F71.16
N21 X25.5516 F1000.
N22 X25.4801 F71.16
N23 G02 X24.7311 Y14.7684 J.749
N24 G01 Y37.9181 F1000.
N25 Y37.9895 F71.16
N26 G02 X25.4801 Y38.7385 I.749
N27 G01 X62.2889 F1000.
N28 X62.3604 F71.16
N29 G02 X63.1094 Y37.9895 J-.749
N30 G01 Y14.8398 F1000.
N31 Y14.7684 F71.16
N32 G02 X62.3604 Y14.0194 I-.749
N33 G01 X43.4728 F1000.
N34 X42.9728
N35 G03 X42.4728 Y13.5194 J-.5 F47.5
N36 G00 Z.25
N37 G40 G00 Y13.2444
N38 G28 G91 Z0 M5
N39 G28 G91 X0 M09
N40 G90
N41 G52 X0 Y0 Z0
N42 G08 P0
N43 M30
%
```

© 2011 CIM-TECH.COM, Inc

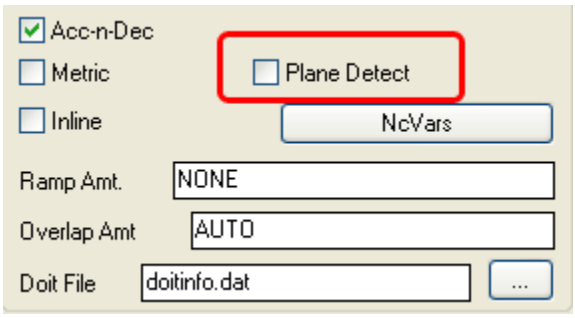
7.12 Metric



Checking the Metric box will change the format of the nc code to metric mode instead of inch mode. It will not change your feed rates or Cutter Compensation or your part size! If you draw in metric, you should use METRIC tools, feed rates in millimeters per minute, etc. The purpose is to insert the code that changes the way the control reads the program from inch to metric.

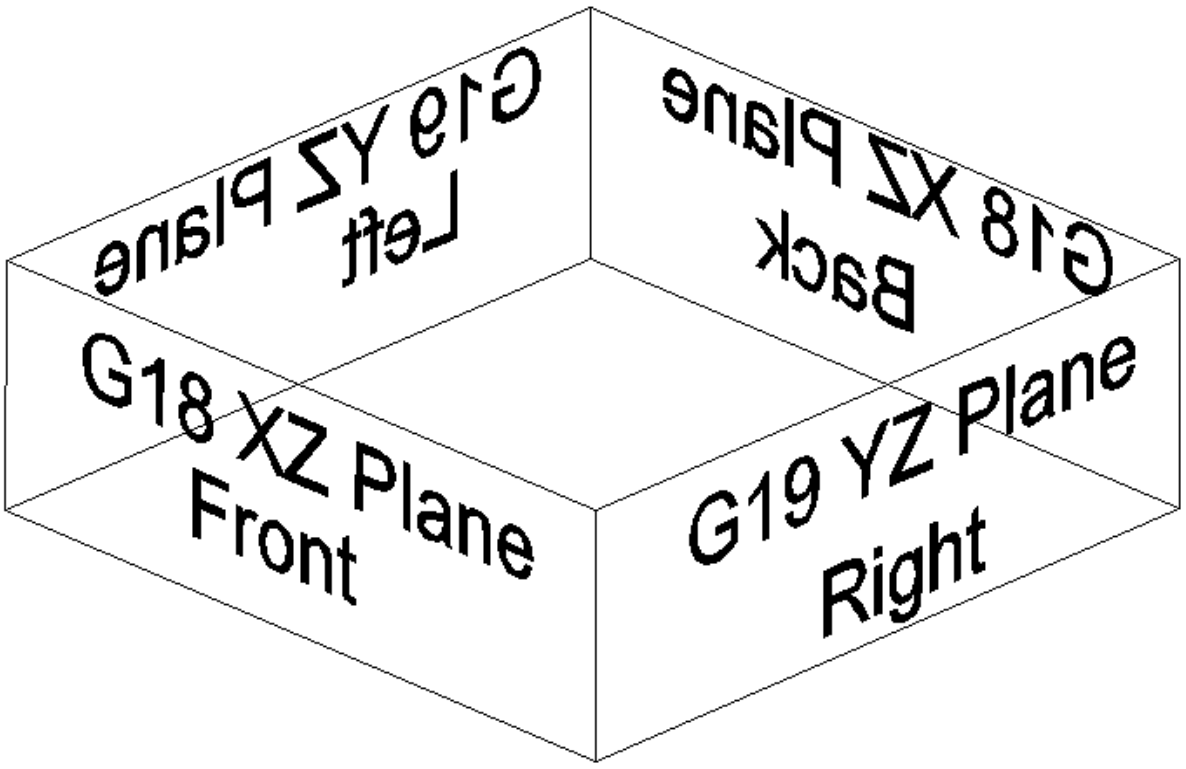
On most controls, the metric G-code is G21. The inch G-code is G20. Some controls use different G-codes and your post processor should output the code appropriately.

7.13 Plane Detect



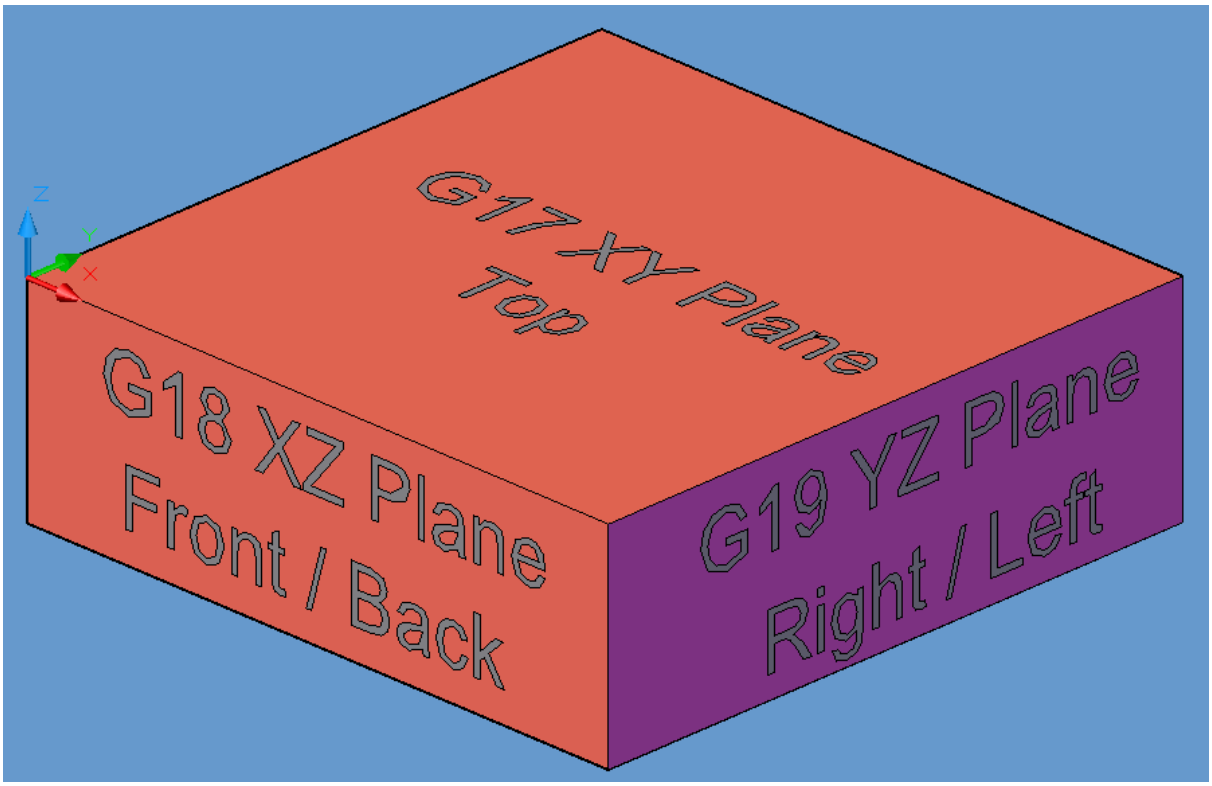
This parameter is used to check whether or not the cut is in the XZ (G18) or YZ (G19) plane. By checking the box, Router-CIM will produce the most efficient code possible for geometry outside of the XY plane. This is accomplished by automatically evaluating the tool path face (Coordinate System) for its eligibility to use Arc Interpolation vs. Point to Point. Plane detect will be reflected in your code by either a G18 (XZ plane selection) or G19 (YZ plane selection) if possible, or G17 (XY plane selection) otherwise.

© 2011 CIM-TECH.COM, Inc



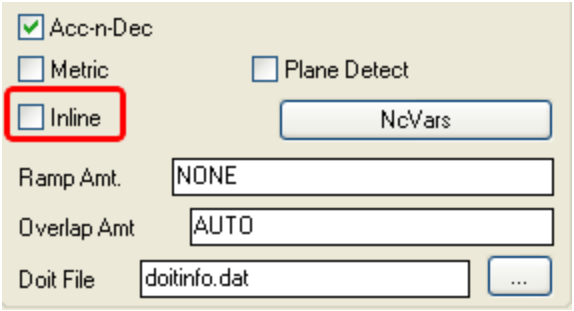
This drawing illustrates the fact that the front and back are both on the XZ plane and the right and left sides are both on the YZ plane.

© 2011 CIM-TECH.COM, Inc



The G17 plane is the XY plane. If any horizontal cuts do not like exactly on the G18 or G19 faces, then they will be output as G17 code.

7.14 Inline



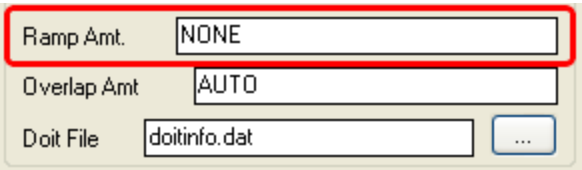
This option allows you to add specific commands to your code during the Cut phase. If this box is checked, when you make a Cut, you will be prompted with a question...Put Commands In Shape? You can then select either Yes or No. If you select Yes, you will then be prompted to Choose a Point Location: select a point on the geometry where you want to run, edit or insert a given task or command.
Next you will be prompted to Select/etc.... or <Exit> (Type in Select if selecting more than one point or press <Enter> to exit.
You will then be prompted to either RUN A TASK, EDIT A TASK, or INSERT A POST COMMAND.
If you select RUN A TASK, you will be prompted to enter the TASK NAME. This is usually only done if you have a specific TASK FILE (usually provided by CIM-Tech) to run.
If you select EDIT A TASK, you will be shown the TASK EDITOR, and you will be able to select the

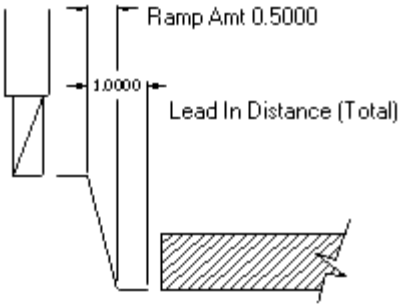
© 2011 CIM-TECH.COM, Inc

TASK you wish to modify. Again, this is usually on a specific TASK, and is not generally needed. The final option is to INSERT A POST COMMAND. When you choose this option, you get a SELECT POSTPROCESSOR COMMAND dialog box with several POST COMMANDS that you may INSERT into the drawing, at locations you choose. Click on the COMMAND you wish to insert then <OK> and you will either be prompted for a value or you will be prompted again as to whether or not you want to Put Commands In Shape?... This is useful if you need to insert a program stop, or a programmed pause into the Cut.

INLINE does not work with TABBING, or OSCILLATION.

7.15 Ramp Amt



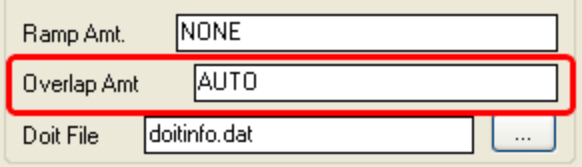


The RAMP AMOUNT is the distance during the lead-in that the cutter spends in the ramp to the Total Depth of the Cut.

For example, if you have a lead-in distance of 1.0" in 1.0" material, but want the cutter at full depth before it reaches the part, you may specify a RAMP AMT of 0.5." The cutter will ramp down to -1.0" in the first 0.5" of travel and will spend the next 0.5" of its lead-in at the maximum Cut DEPTH.

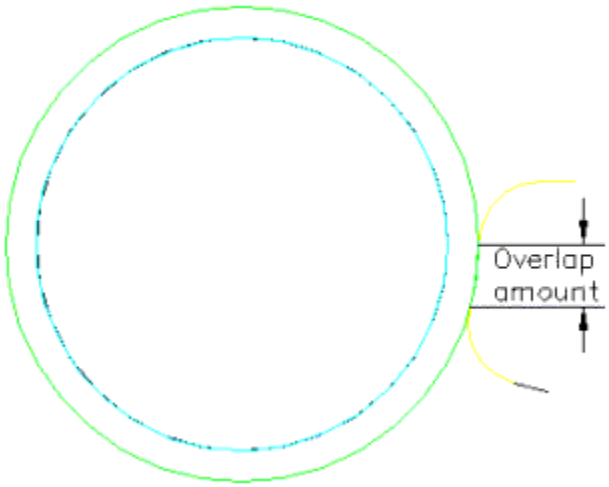
Used with Ramp Inside and Ramp Outside cycles only.

7.16 Overlap Amt

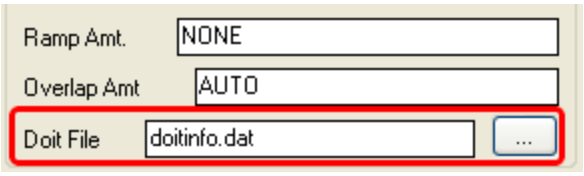


Overlap is the amount the tool travels in the cut beyond the start point before leading out of the cut. By default the Overlap amount is equal to the diameter of the tool. You are able to specify a larger or smaller amount for this by placing a value in this field. For instance, if you are using a 0.5" router bit, the Overlap distance is 0.5". If you put 1.0" in the Overlap Amt. field then the Overlap will be 1.0".

© 2011 CIM-TECH.COM, Inc



7.17 DOIT File



This field allows you to call up or choose which .dat file you want to make current for the DO-IT cycle. When you have several Doit files and wish to edit one, type the name into the Doit File box and then click on the Doit Edit button at the top right of the Control Panel/All Stats page.

The button to the right of the entry box is to allow you to browse your computer to select an existing file.

© 2011 CIM-TECH.COM, Inc

8 Cutting

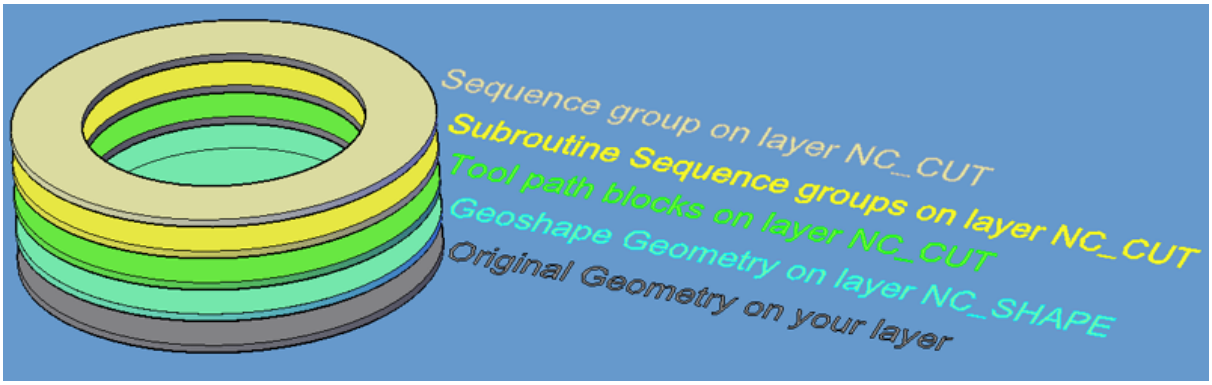
After selecting the Tool, Cycle and Status Information necessary to make a Cut, you must then select Cut from the Control Panel (or type Cut at the command prompt or select it from the Router-CIM toolbar) and you will be prompted to select objects. Pick the shape you wish to Cut and using the current settings from the Control Panel, a tool path will be generated using the parameters you have defined.
A Tool path will show up either as green (clockwise) or red (counter-clockwise) along with the Lead-In and the Lead-Out as yellow and the Rapid Moves in white.
The tool path will be generated at the Z depth you specified in the Total Cut Depth and/or Depth per Pass fields.
You may only Cut Polylines and surfaces that are on the NC_SHAPE layer.

8.1 Layer Control

Be aware of the Layering scheme that Router-CIM creates when tool paths are created.

The items on the screen will be on top of each other in this order:

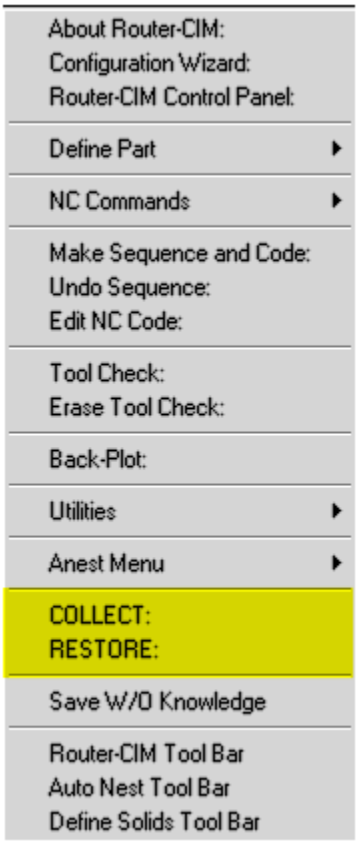
1. Your original geometry on whatever layer you built it.
2. Defined CCW Polylines from the geoshape command are on layer NC_SHAPE.
3. Tool Path blocks created from the CUT command are on layers starting with NC_Cut.
4. The Sequence List for each tool are on layers starting with NC_Cut, if using Sub Programs.
5. The Sequence List for all tool paths on layers starting with NC_Cut.



8.2 Collect and Restore

When CUTS have been made and you want to edit or redo some of the CUTS, it is sometimes difficult to see one Cut underneath another for the purpose of selection. Select the RCIM pull down menu, COLLECT and RESTORE will appear.

© 2011 CIM-TECH.COM, Inc



To use Collect:

Simply select COLLECT from the menu, and pick on the tool path you want to remove from the screen temporarily. It is collected to a temporary buffer in memory, available only as long as the drawing is open. The RESTORE command will clear the buffer and display the Cut on the screen once again.

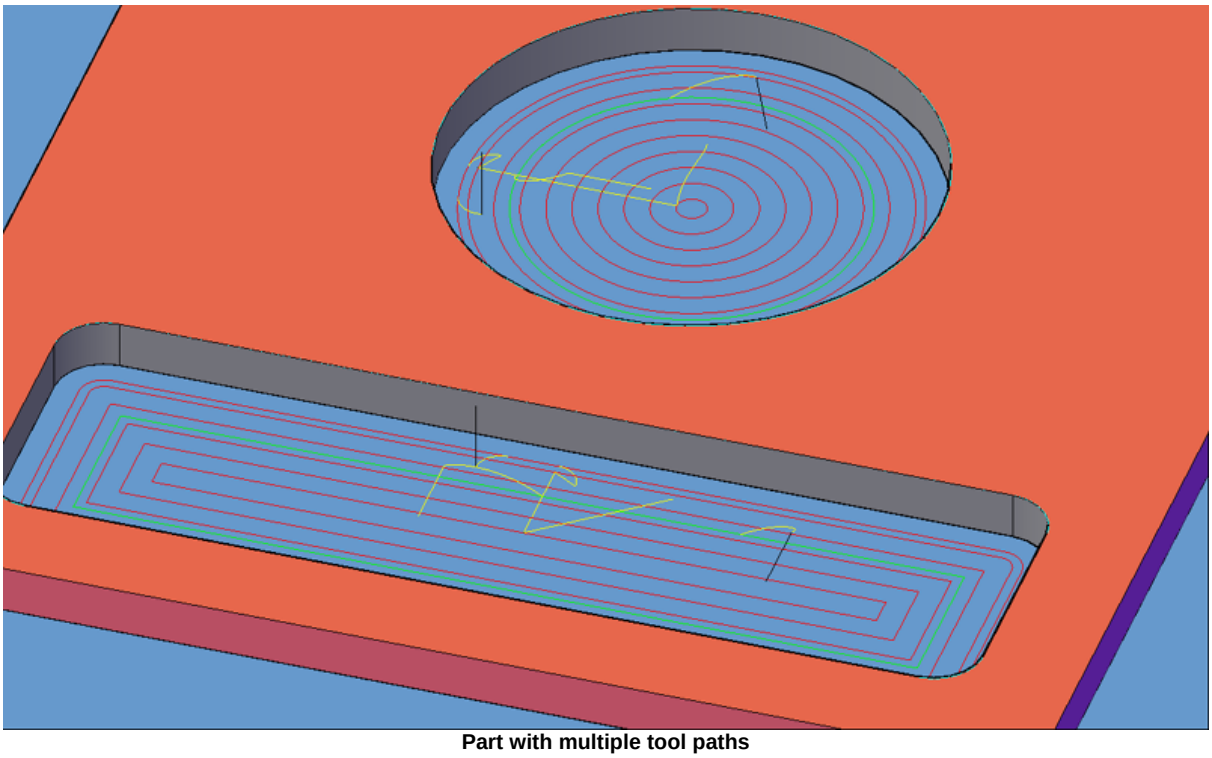
To use Restore:

Select the Restore command from the menu, and any cuts that have been Collected with the Collect command will be Restored to the drawing.

Using Collect and Restore with Sequence to create a Sort Order.

If the Cuts are Collected according to the order in which you wish them to be cut, then they can be Restored in order. If you immediately use the Sequence command, you can use the P for Previous option when prompted to Select Objects for the Sequence. The order the cuts were Collected in will be the order that they are stacked in the Sequencer, and no further sorting will be necessary.

© 2011 CIM-TECH.COM, Inc

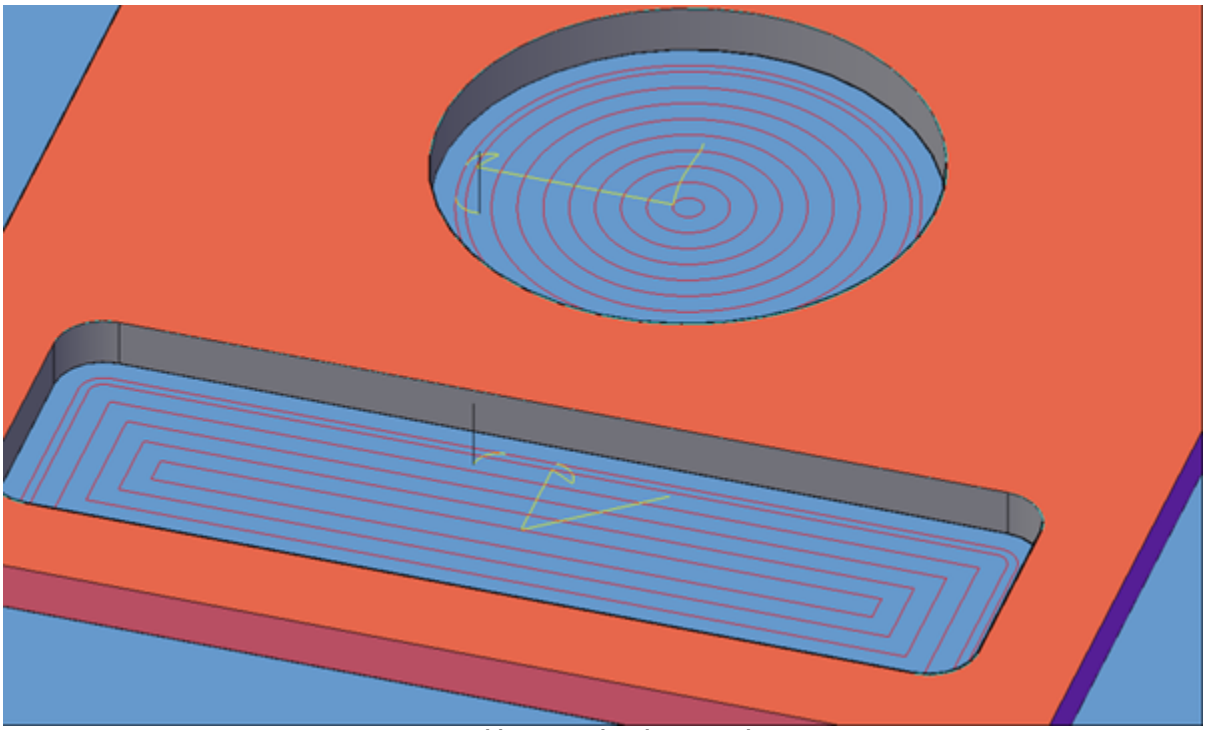


Part with multiple tool paths

For example, if in the following cuts you wish to use MULTI-PASS to cut the inside of the part first and then cut the inside shape all the way through with another cycle, you should pick on the MULTI-PASS cut first to COLLECT it, and then COLLECT the other inside cut. If no other action is taken between COLLECT and RESTORE the cuts will be restored in the order in which they were collected. If Sequence is the very next step, the Previous selection set will select the cuts in the same order, eliminating the need for further sorting.

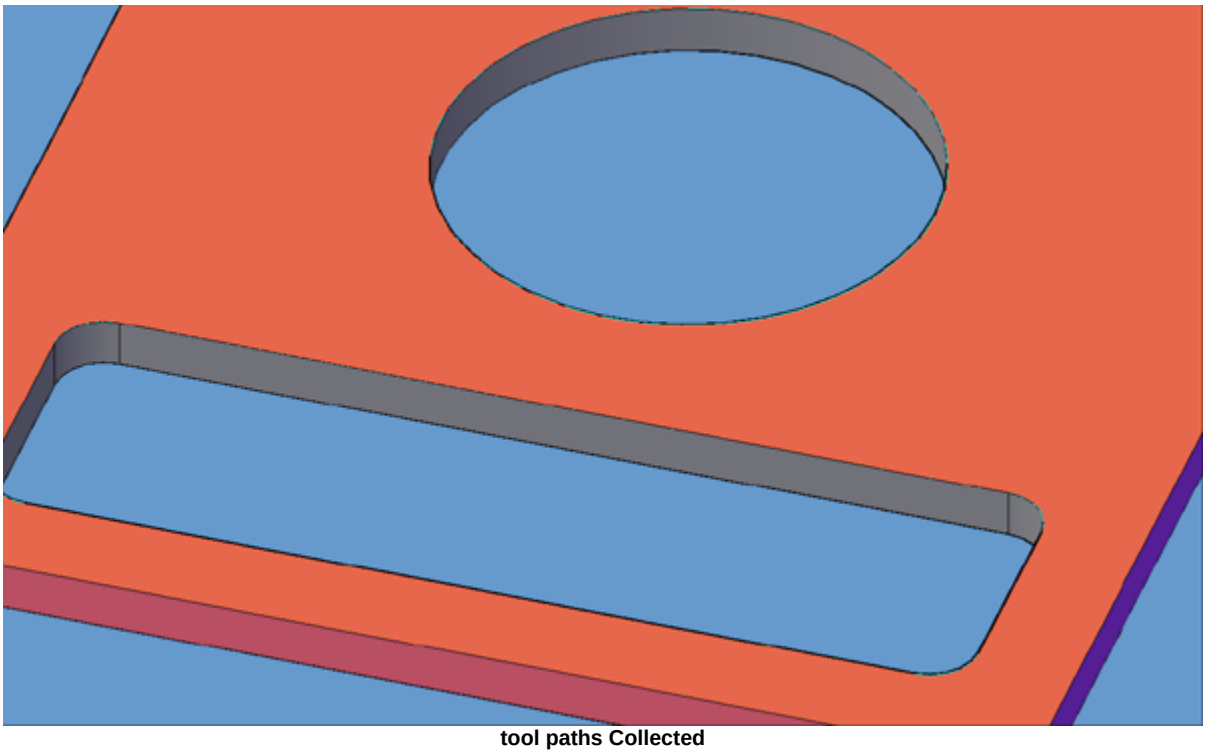
Make the First Cuts

© 2011 CIM-TECH.COM, Inc



Multi-Pass tool paths created

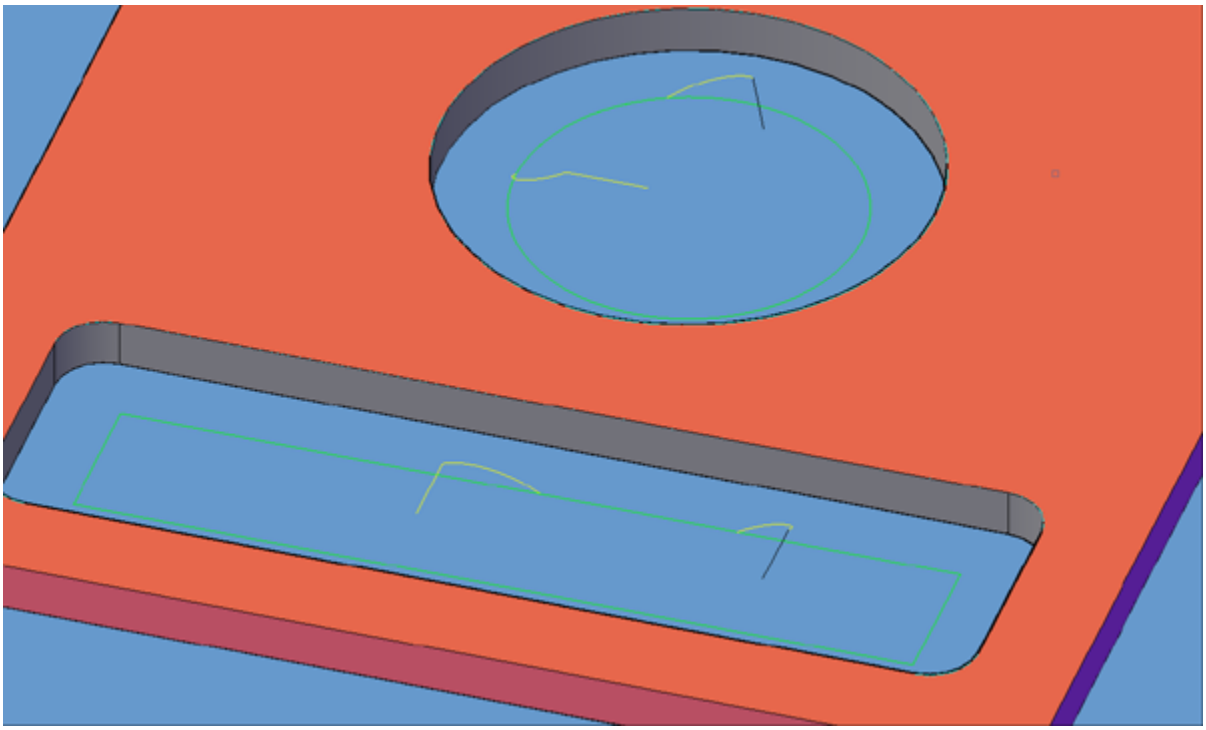
Then COLLECT them in the order you want them cut. The tool paths will be removed from the screen.



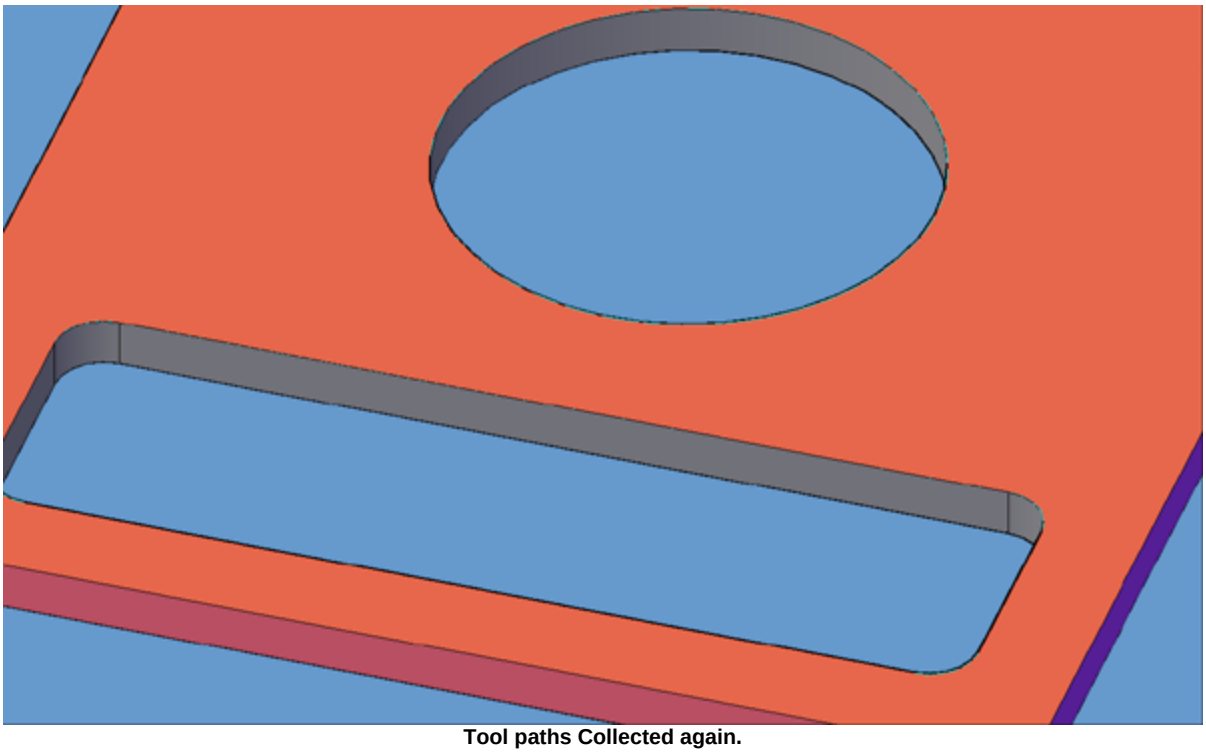
tool paths Collected

Next make the Shaper Tool cuts on the part.

© 2011 CIM-TECH.COM, Inc



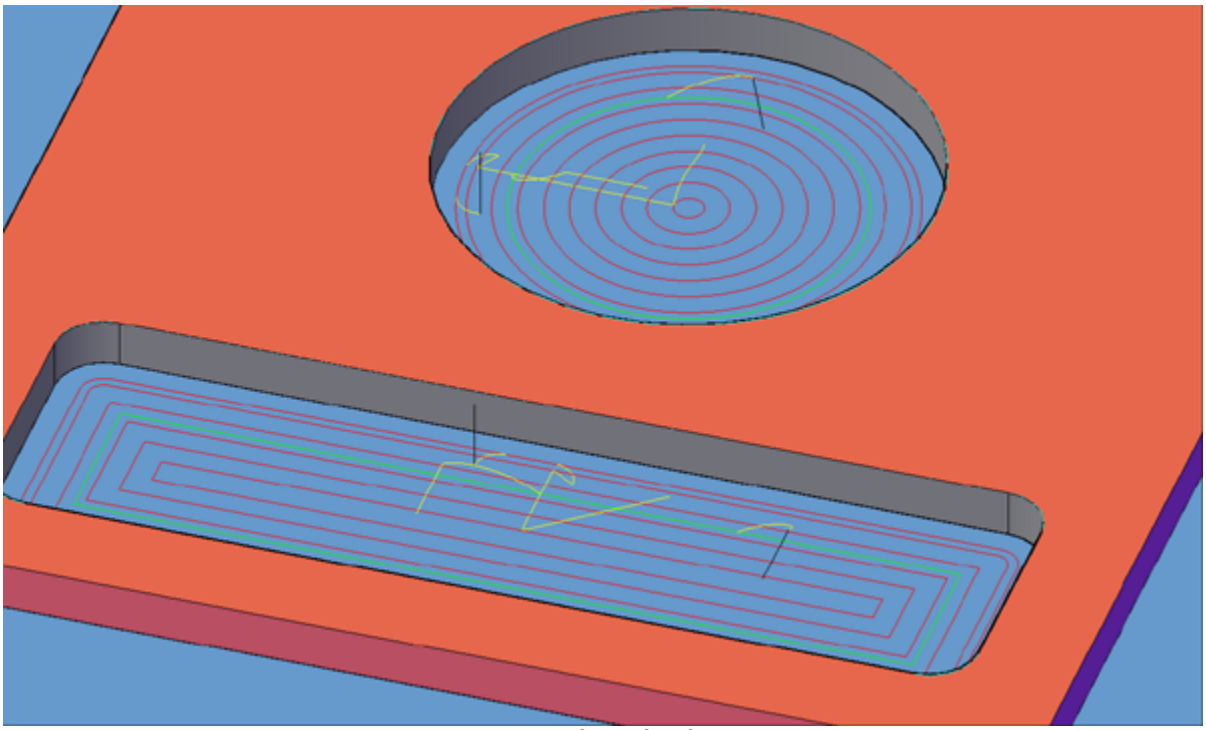
Then COLLECT these cuts in order.



Tool paths Collected again.

Now, RESTORE the tool paths.

© 2011 CIM-TECH.COM, Inc



Restore the tool paths.

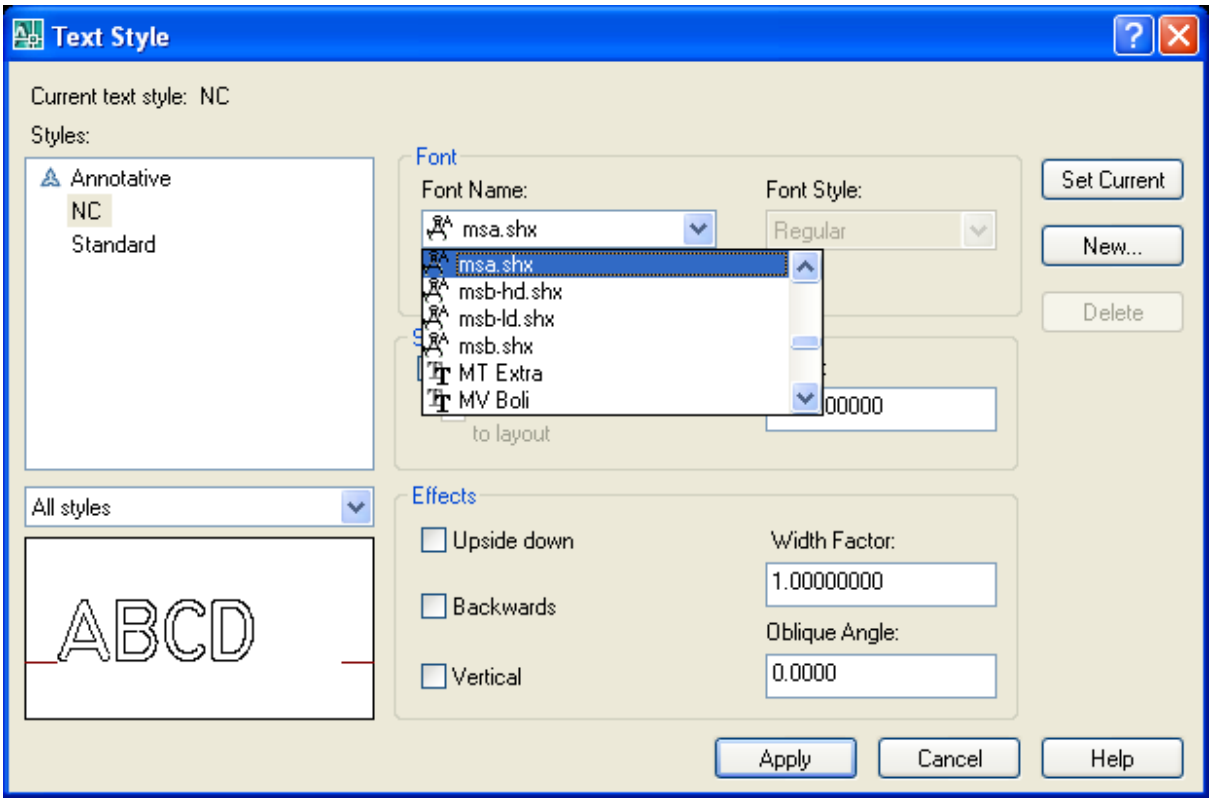
Finally, select Sequence and when prompted to Select Objects, type in P (for Previous) and then make your code. No other sorting tasks are necessary and the cuts will be Sequenced in exactly the order they were Collected in.

8.3 Cutting Text

Cutting text involves the use of a tool called Letter-Ease. This tool can be selected from the Router-CIM toolbar. Letterease allows you to select different text styles, and then explode them into lines and arcs so that you can Geoshape and cut.

To use the standard Fonts that come with AutoCAD, select a Text Style.

© 2011 CIM-TECH.COM, Inc



From the Text Style box, you can preview and select a new text syle, set the height, width, angle, etc. for your new lettering.

Next apply the lettering to the drawing with the DTEXT (Draw > Single Line Text).

```
Specify start point of text or [Justify/Style]:

Specify rotation angle of text <0.0000>:
```

You can Geoshape this text, after you explode it. To explode the text, pick the LE button on the Router-CIM toolbar. 

```
Command:

Baseline object/Circular/Edit/Justify/Spacing/eXplode/<Start point>:
```

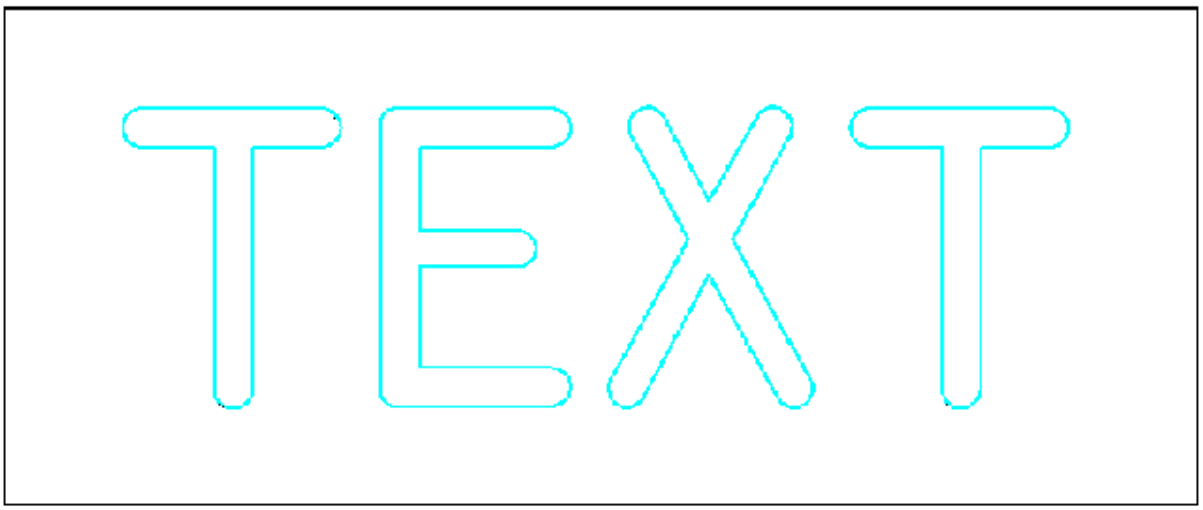
Type X for Explode to convert the text to polylines and polyline arcs.

```
Exploding Text Entry object 1....Done!

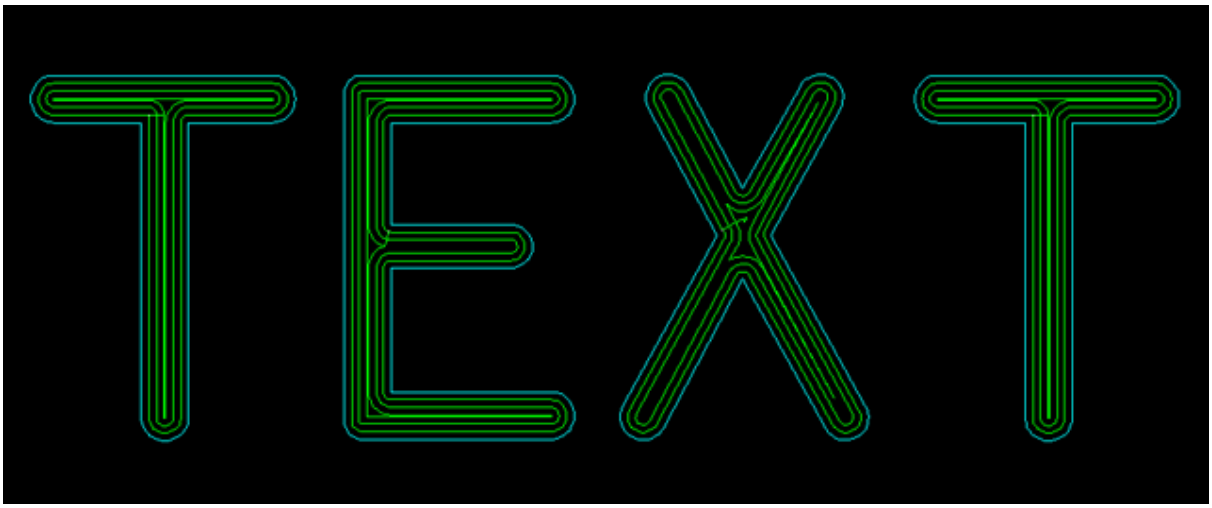
Command:
```

Once the text is exploded, you can Geoshape the text to make it ready for cutting.

© 2011 CIM-TECH.COM, Inc



Once the text is Geoshaped, select a tool, cycle and set the status information to create tool paths.



8.4 Toolcheck



There is a function built into Router-CIM to provide a display of the tool along a cut path. This function is called Toolcheck. This display can be used for visual verification and actual measurement checks. To select, click on the Toolcheck icon on the Router-CIM toolbar.

Toolcheck uses only cut cycle blocks as input. A cut cycle block has only one tool description. Toolcheck performs the tooling display using the tool described in a single cut block. If a Sequence group is selected and the Sequence was developed using cycles containing different tools, only the first tool in the Sequence will be displayed. Use Toolcheck on individual Cut cycle blocks for correct tooling display.

The display of the tooling is controlled by four options. These options are Endpoint, Measure, Task,

© 2011 CIM-TECH.COM, Inc

and Emulate. We recommend the use of Endpoint and Measure as the fastest, most reliable means of tool path verification.
The tooling geometry is on the Tool Check layer, and can be erased by selecting the Erasecheck icon off the Router-CIM toolbar. This is the only way to erase a Toolcheck!

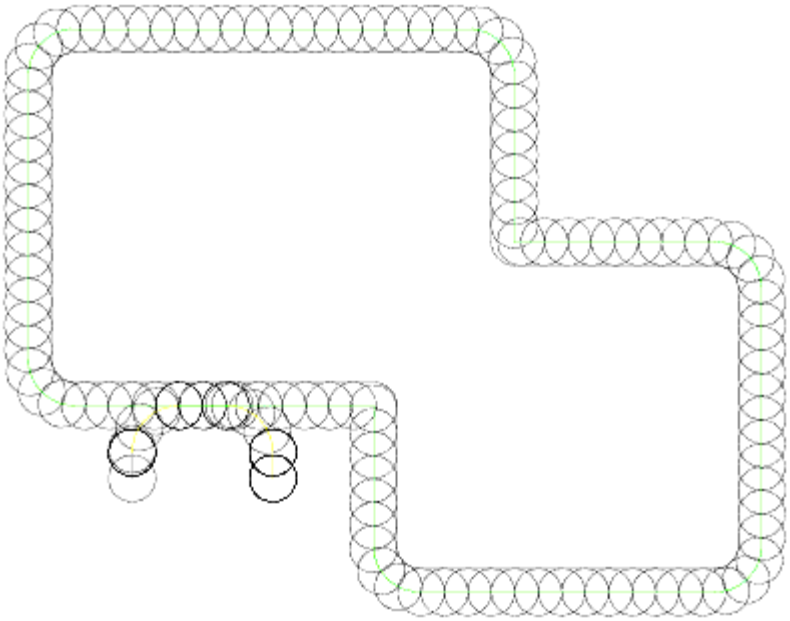
Endpoint

The tool block is inserted at each Endpoint of the tool path. The inserts remain on the screen after completion of Toolcheck.

Measure

The tool block is inserted at Measured points along the tool path. The AutoCAD "Measure" command is used to develop each of the insert points. The spacing of the insert points is entered at the prompt "Enter Tool Check Spacing:". The inserts remain on the screen after completion of Toolcheck. If you wish to use this function to test your tool path, or direction of Cut, etc., click on the Toolcheck icon off the Router-CIM toolbar. You will be prompted to select objects, pick the tool path you wish to have checked. The next prompt gives you specific choices for the Toolcheck mode.
Select 'M' for MEASURE, and you will be prompted for the Toolcheck spacing; usually the default is ¼ of the tool diameter. This is the spacing of the Toolcheck circles as they move around the part. To use the default press <Enter>, or change the value, usually setting the spacing to the tool radius is sufficient.

The Toolcheck will then proceed around your part (see diagram below). There will be a circle the same diameter as the tool moving along the center of your tool path at the circles (tools) center point at the spacing you specified.



This is very useful for checking the direction of Cut on open shapes, where it is often difficult to tell if the direction of the Cut is correct.

Task

This option will allow you to use a custom task or macro you design to perform a custom Toolcheck. You will be prompted at the command line with Enter Task Name: and you would type in the name of your task.
A default task would be TC. If you type in TC, the Toolcheck will continue.

© 2011 CIM-TECH.COM, Inc

Emulate

The Emulate mode is to allow a block representing the tool to move across the tool path at specific intervals.
You will be prompted at the command line for the Tool Block Name. If you have made a custom tool block you want to use, type in the name now, otherwise Router-CIM will insert the block named TDIA.
Next, you are prompted to Select Emulation Spacing. The default is usually fine, however, you can set any spacing you desire.
The Starting Element is the next prompt, and you should select the lead-in, then you will be prompted for the Stopping Element, and you should select the lead-out.
Router-CIM will show the tool block moving around the part at the spacing provided.

8.5 Erasecheck

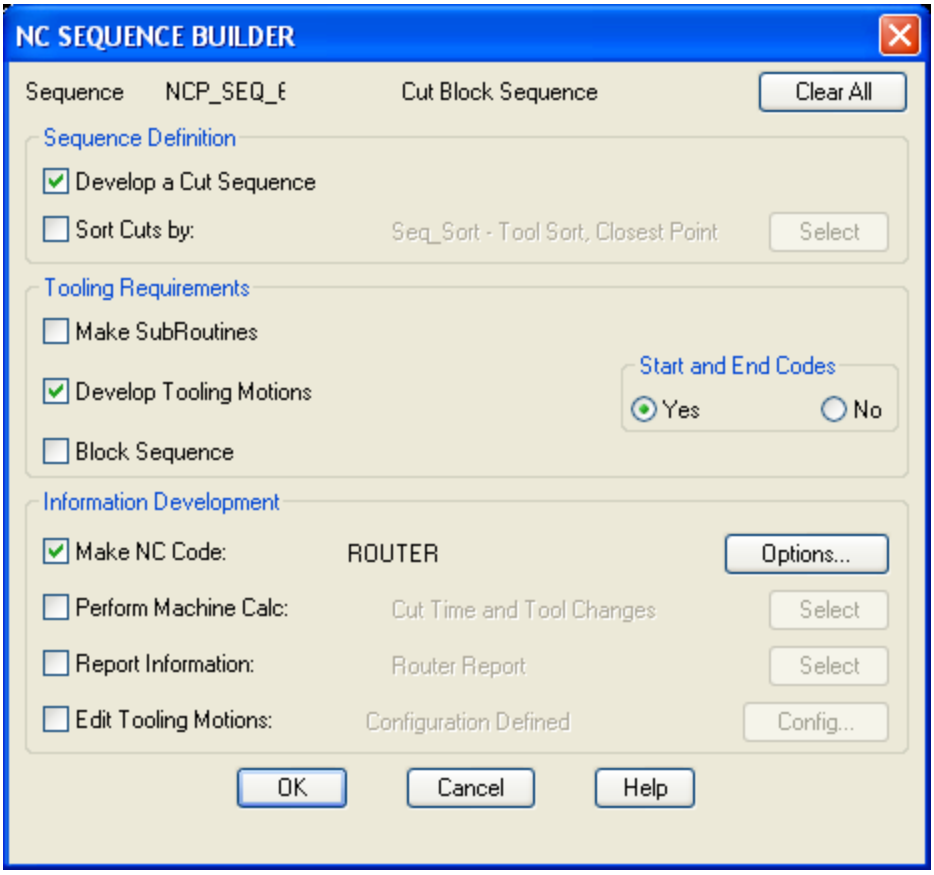


Use Erasecheck to erase the toolcheck. That is the only way to remove the TOOLCHECK! This can be found on the Router-CIM toolbar, or under the RCIM pull down menu. Please note that ERASECHECK will erase all the circles and then perform a redraw.

© 2011 CIM-TECH.COM, Inc

9 Sequence and Make Code

Upon selecting Sequence from either the Control Panel or off of the Router-CIM toolbar, an NC Sequence Builder dialog window will appear. With this you have access to such features as Sequencing, Sorting, Reporting, Tool Life Study, Automatic Sub-Programming, NC Code Documentation, and Tool Path Editing all from this one screen



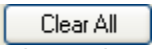
9.1 Sequence Definition



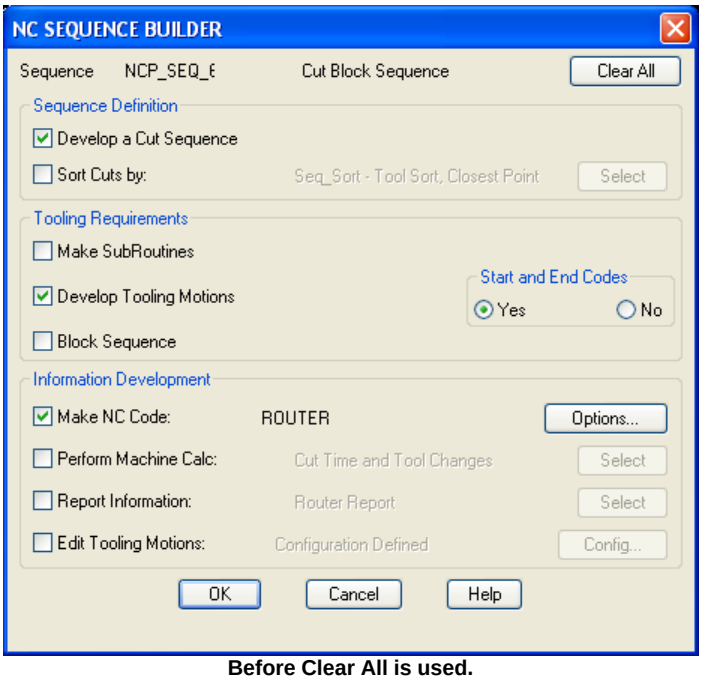
The NC Sequence Definition section allows you to develop a Cut Sequence and also to sort the cuts in the sequence according to several sorting keys available. There is also an option to clear out the sequence options so that you can start over or remove existing choices.

© 2011 CIM-TECH.COM, Inc

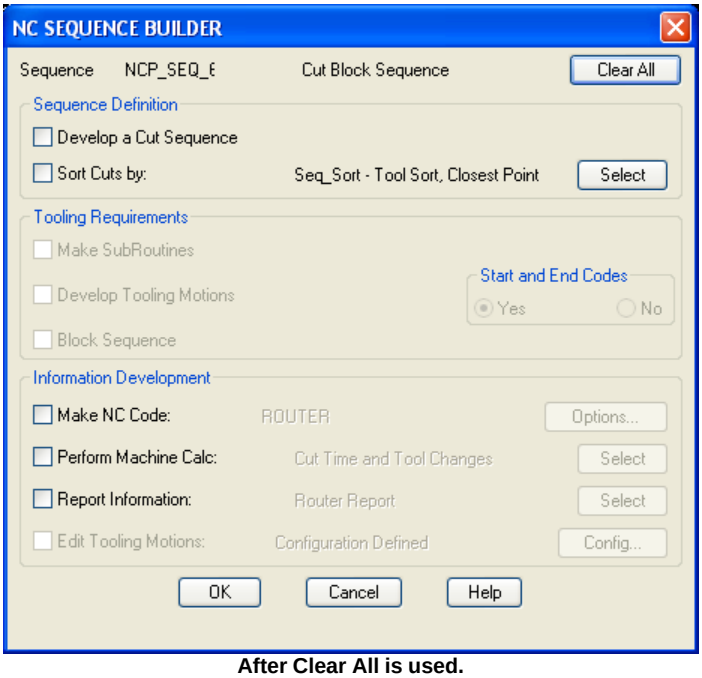
9.1.1 Clear All



When selected after making a Sequence, this button will clear all fields back to their default settings. This is useful for having Sequence write only Report Information or Make NC Code again (instead of re-writing everything) or for just clearing the screen to give you a fresh start



Before Clear All is used.



After Clear All is used.

© 2011 CIM-TECH.COM, Inc

9.1.2 Develop a Cut Sequence



Develops the Router-CIM database for selected cuts.

When you use 'Develop a Cut Sequence', you are defining all the cut cycle blocks, process list blocks and sequence blocks that will represent the basis of a sequence.

If you have selected an existing sequence that has developed tooling motions in it, the tooling motions are removed when the 'Develop a Cut Sequence' command is used.

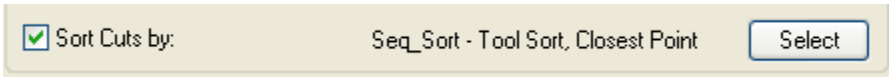
An internal database is established when cut cycles are defined for use by Router-CIM. The database is referred to as a sequence knowledge. A sequence knowledge is an internal list that contains information about the selected cuts and any other Cut or Sequence commands that have attached additional data to the cuts. This database remains with the sequence in the form of an AutoCAD block in the current drawing. When Router-CIM commands perform activity on a sequence, the current sequence database is used and updated.

If this is the only command chosen in the command options, the result would be a group containing the selected cut blocks.

If other command options are selected that require a new definition of cut cycles, the definition phase for the cuts will occur. This is true when you develop tool motions, sort the cuts, or define subroutines. The option dialog interface will disable these commands until you indicate that definition of cuts is to occur.

By default this option is check and should normally be left that way.

9.1.3 Sort Cuts By



Redefines the order of a sequence definition

Sort Cuts is the re-arrangement of the cut blocks in the sequence based on a set of sorting rules. The sorting rules are defined in a sort task. The option to select the sort task is provided in the dialog interface.

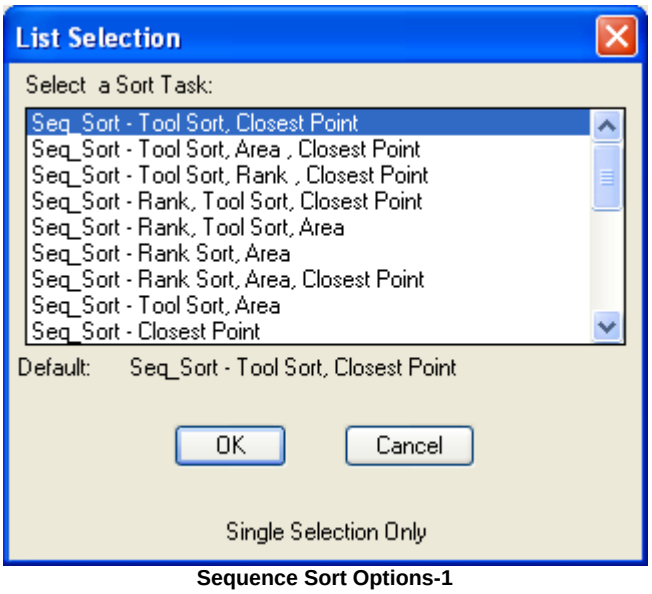
Since sorting can be used numerous times to determine the best sort, temporary vectors are generated between the cuts so that you can see the results. Do not confuse these temporary vectors with the index lines developed during tooling motions. These temporary lines will disappear upon the next redraw command.

If you are sorting and developing tooling motions, the temporary vectors are not generated since the index lines are produced during tooling motions development.

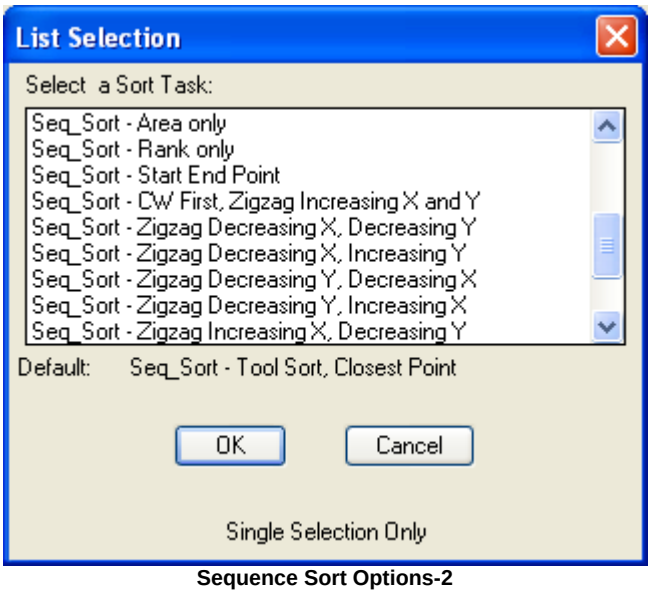
Sort simply rearranges the cut blocks in a sequence.

When you Sequence a Cut,
1) Pick the Sort Cuts by check box and click on the Select button.
2) Choose one of the Sort by Rank options as shown.

© 2011 CIM-TECH.COM, Inc

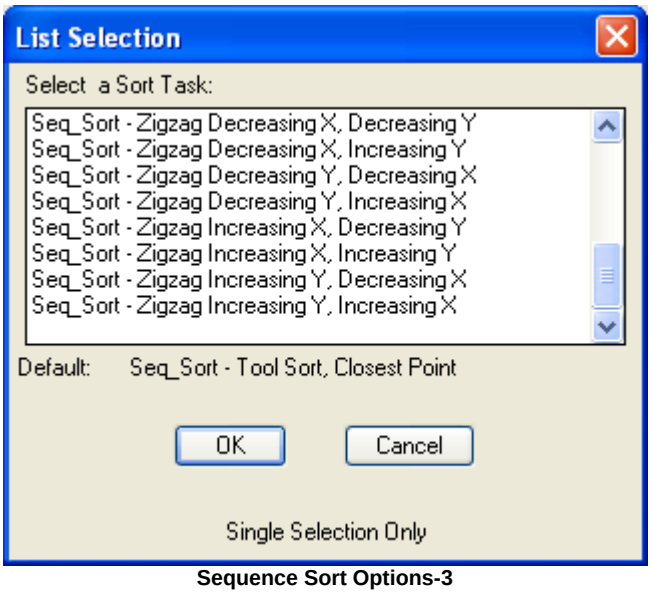


Sequence Sort Options-1



Sequence Sort Options-2

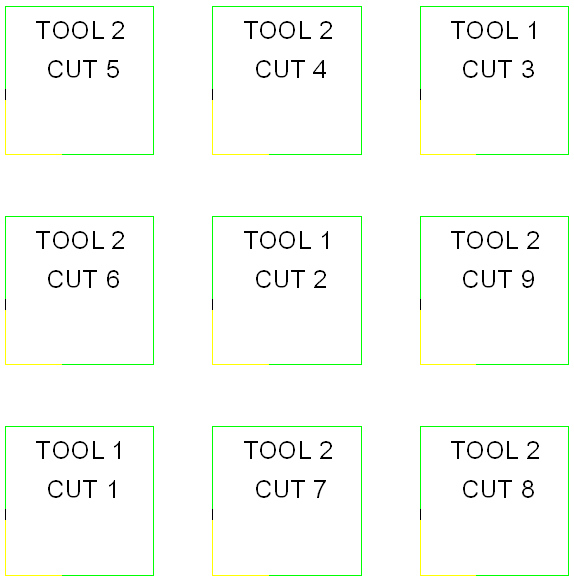
© 2011 CIM-TECH.COM, Inc



Sequence Sort Options-3

Seq_Sort -- Tool Sort, Closest Point

This will sort using the lowest number tool 1st, as many times as it can, sorting those cuts by closest point, then change tools to the next highest tool number and continue sorting by closest point, etc....

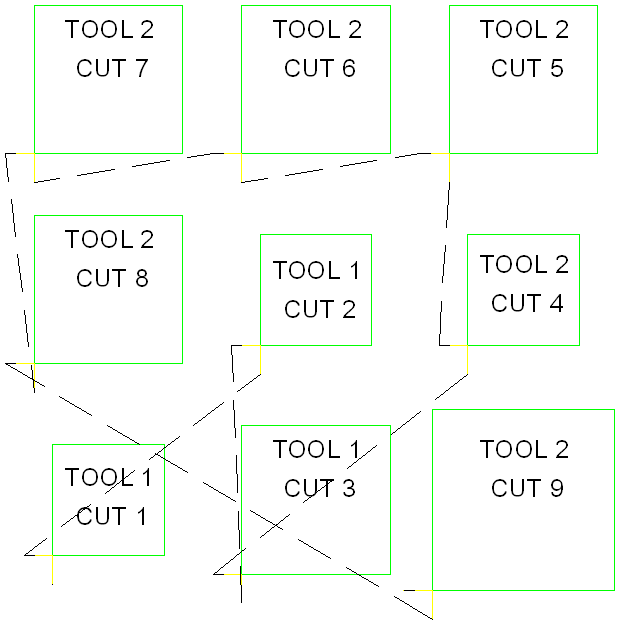


Tool Sort, Closest Point

Seq_Sort -- Tool Sort, Area, Closest Point

This will sort using tool number 1st, then area (smallest to largest), and then closest point.

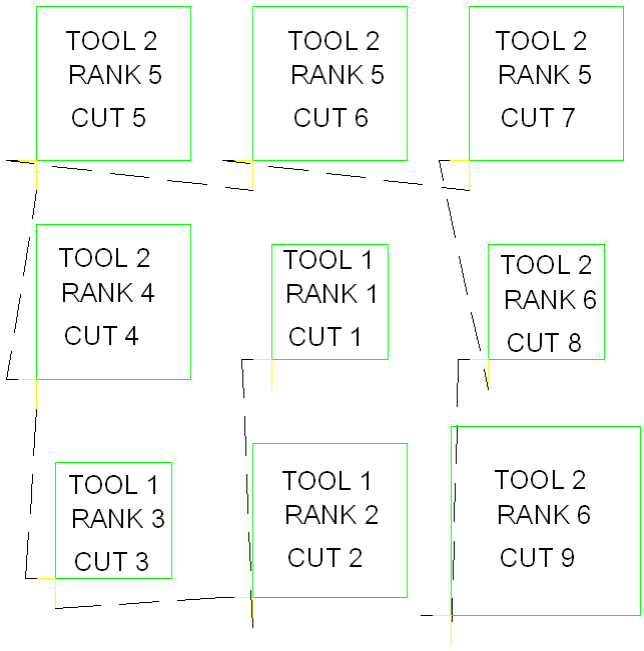
© 2011 CIM-TECH.COM, Inc



Tool Sort, Area, Closest Point

Seq_Sort -- Tool Sort, Rank, Closest Point

This will sort using lowest tool number 1st, then rank number, and then closest point. Looking at the example below, the cuts for tool 1 are gathered, then sorted by rank and placed in order according to closest point. Next, the tool 2 cuts are gathered, then sorted by rank and then selected according to closest point as some of them are the same tool and rank.



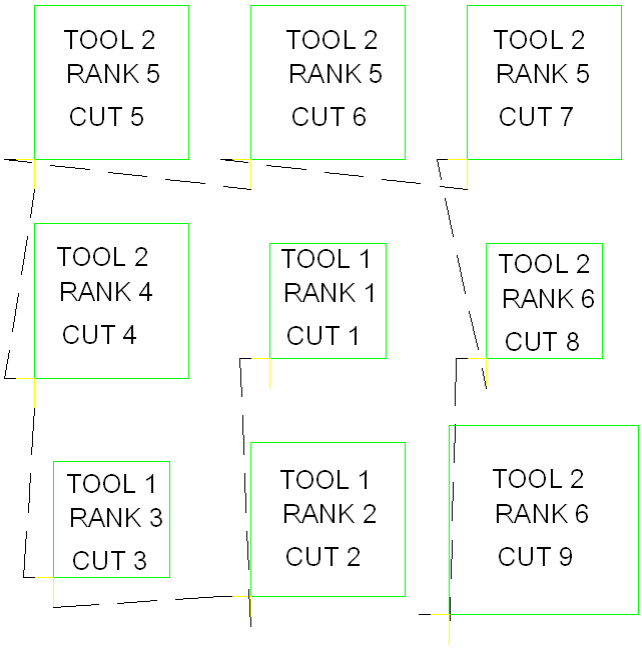
Tool Sort, Rank, Closest Point

Seq_Sort -- Rank, Tool Sort, Closest Point

This will sort using rank 1st, then lowest tool number, and then closest point. In fact the particular example shown below the cuts will be placed in the same order as the sort above. All cuts are sorted

© 2011 CIM-TECH.COM, Inc

by rank first, and then within the ranked cuts, the tool numbers are sorted, lowest first. Last the cuts are sorted according to location from one cut to another within the rank and tool sort.

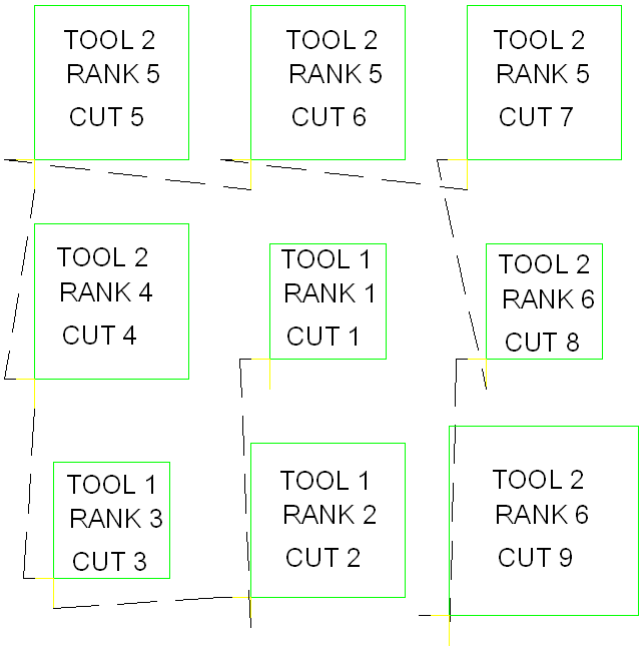


Rank, Tool Sort, Closest Point

Seq_Sort -- Rank, Tool Sort, Area

This will sort using rank 1st, then lowest tool number, then area (smallest to largest). Once again, setting the Sequence up in the same order as the last two since the area sort is last, cut 1, cut 2, and cut3 are placed in Rank and then Tool order first and since there is only 1 of each rank, the area sort is really canceled out. The tool 2 cuts (4-9) are sorted by rank and since there is only one cut with rank 4, it has to be first. There are 3 cuts with rank 5, but they are all the same tool number and area, so the sequence places them in an order (any order for these cuts would be valid, even if it was not according to closest point...we just got lucky there) and finally there are two rank 6 cuts and both are made with the same tool number, but one is smaller than the other so it must come first.

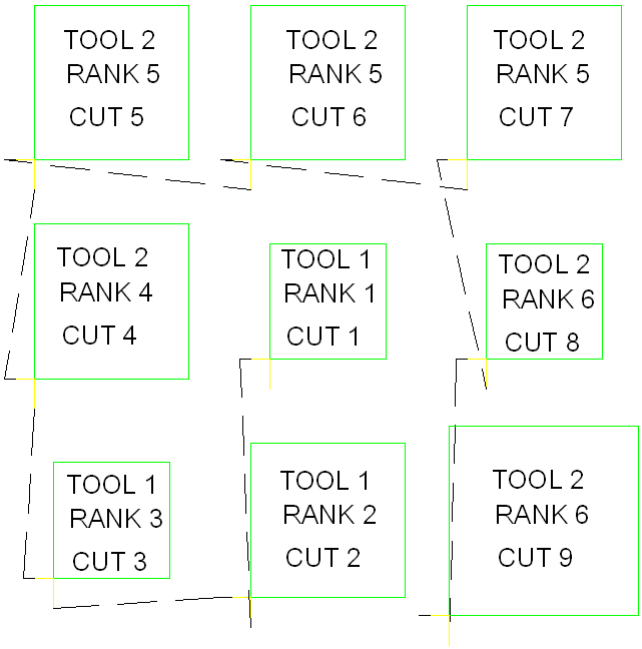
© 2011 CIM-TECH.COM, Inc



Rank, Tool Sort, Area

Seq_Sort -- Rank Sort, Area

This will sort using rank 1st, then area (smallest to largest). This order is still similar to the last few, as the rank sorting is the first key. Cuts 1-4 are ordered according to rank, then cuts 5-7 are made in any order since they are the same rank, and area. Finally cuts 8 and 9 are made in area order, since they are the same rank.

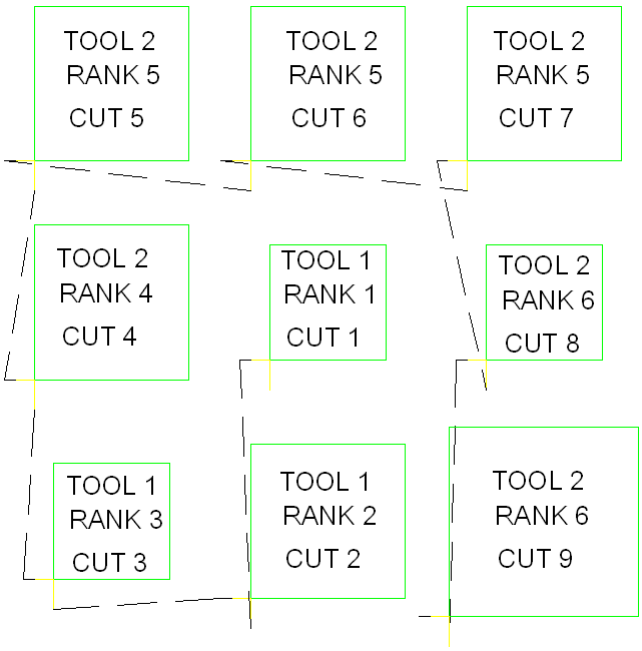


Rank Sort, Area

Seq_Sort -- Rank Sort, Area, Closest Point

© 2011 CIM-TECH.COM, Inc

This will sort using rank 1st, then area (smallest to largest), then closest point. This is once again the same order as the last few (you couldn't get this order in a real job, honest). Since the cuts 1-4 are in rank order and there are no duplicates, we move on to cuts 5-7 which are the same rank, and same area. They get sorted according to closest point (remember before we just got lucky, these cuts only have to appear in this order during this sort key). Finally, cuts 8 and 9 are made with the same rank, so they get sorted according to area and the sequence is finished.

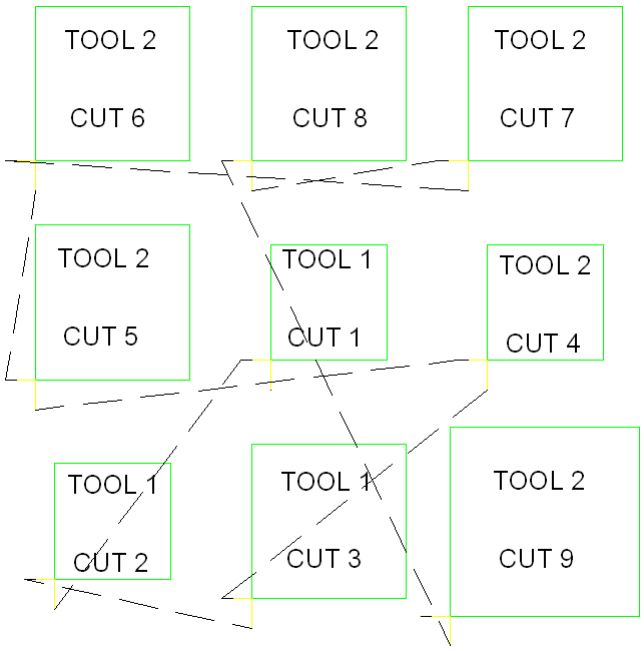


Rank Sort, Area, Closest Point

Seq_Sort -- Tool Sort, Area

This will sort using lowest tool number 1st, then area. Since the location of the cuts is not a factor, they are tool sorted, then sorted according to size. Cuts 1-2 are the same tool and area, cut 3 is made with tool 1 and is larger than cuts 1 and 2 so it comes after. Cut 4 is the smallest tool 2 cut, so it is next. Cuts 5-8 are all the same tool and area, so they get picked in no particular order, then on to cut 9 which is the largest cut made with tool 2 so it comes last.

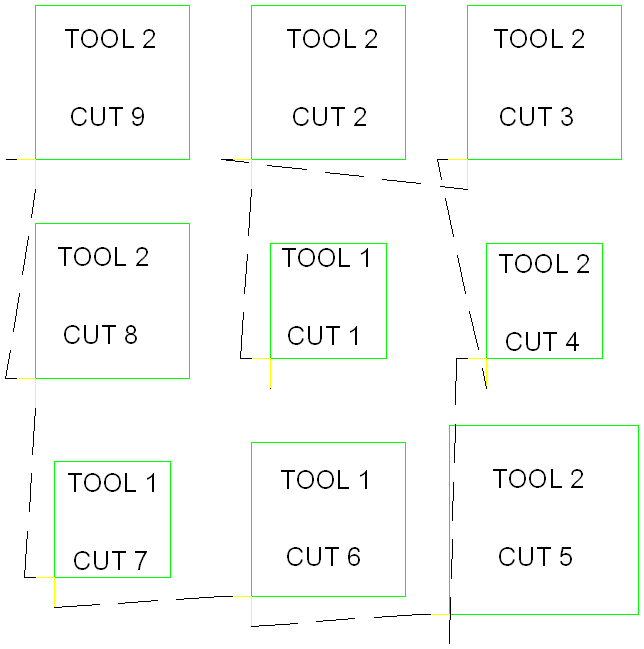
© 2011 CIM-TECH.COM, Inc



Tool Sort, Area

Seq_Sort -- Closest Point

This will sort using closest point only. The tool number, area, rank, etc are all ignored.

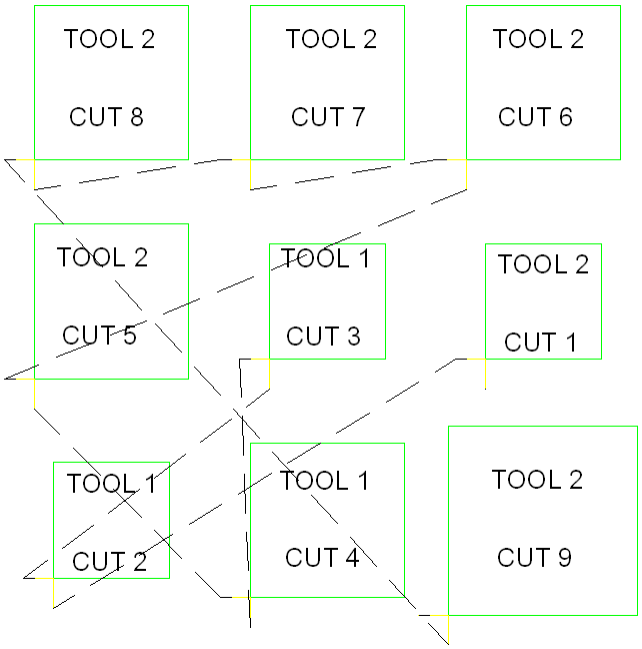


Closest Point only

Seq_Sort -- Area Only

This will sort using area only. It will start with the smallest and work towards the largest shape. No other factors are considered.

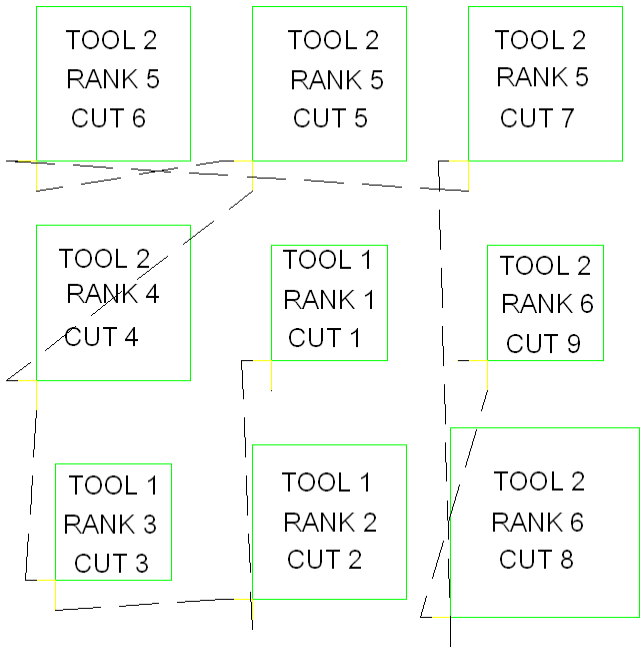
© 2011 CIM-TECH.COM, Inc



Area Only

Seq_Sort -- Rank Only

This will sort using rank only. The lowest rank is first, progressing toward the highest. Decimal and Integer numbers are allowed. There are no other considerations.

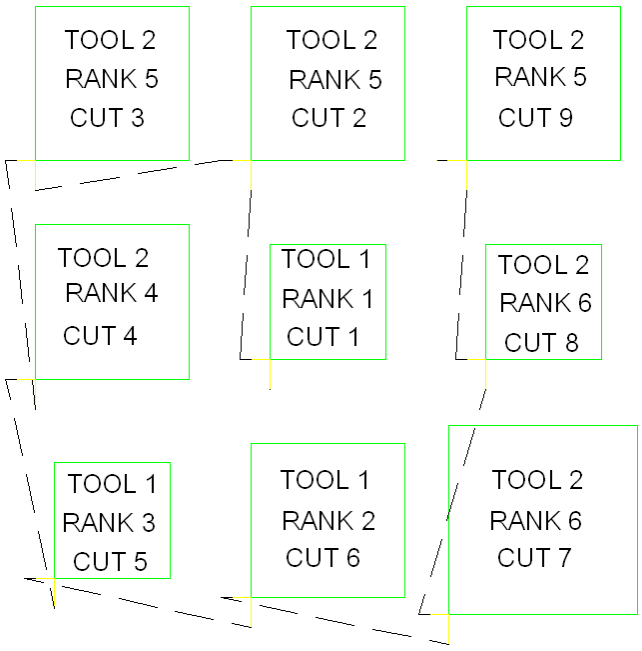


Rank Only Sort

Seq_Sort -- Start End Point

This option will sort by moving from the end point (lead-out) of a cut to the next closest start point (lead-in).

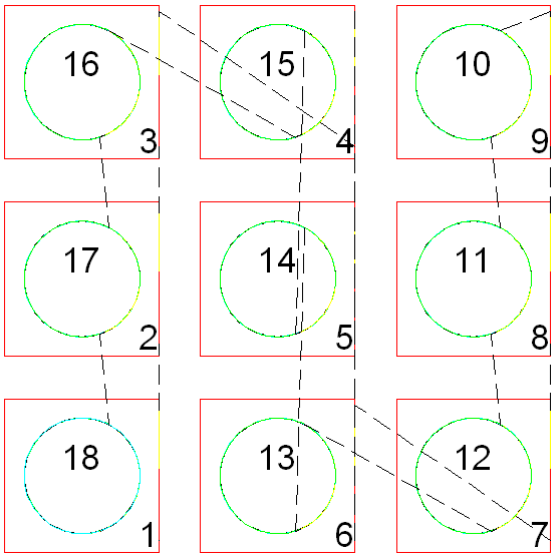
© 2011 CIM-TECH.COM, Inc



Start, End Point

Seq_Sort -- CW First, Zig Zag Increasing X and Y

This will sort cuts made in a clockwise direction starting with the lowest X and Y and cut in Increasing X and Y positions until it gets to the end of the CW tool paths and then reverse its direction walking the Sequence back to where it started.

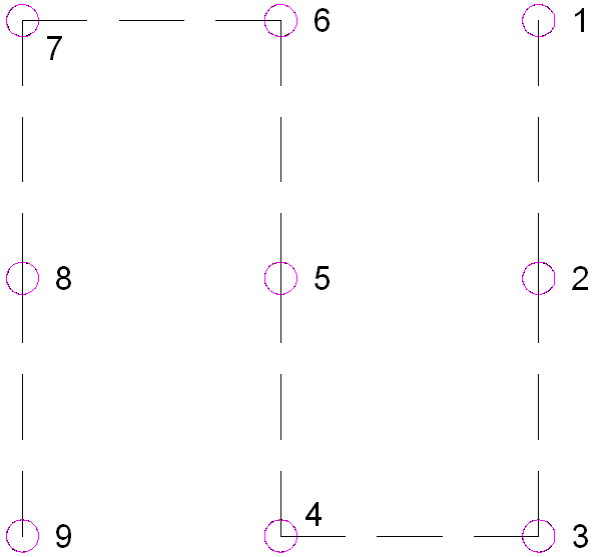


CW First, Zig Zag Increasing X and Y

Seq_Sort -- Zigzag Decreasing X, Decreasing Y

This will sort cuts starting at the farthest lead-in in X and Y and then proceed to the next closest lead -in while decreasing in X and decreasing in Y values.

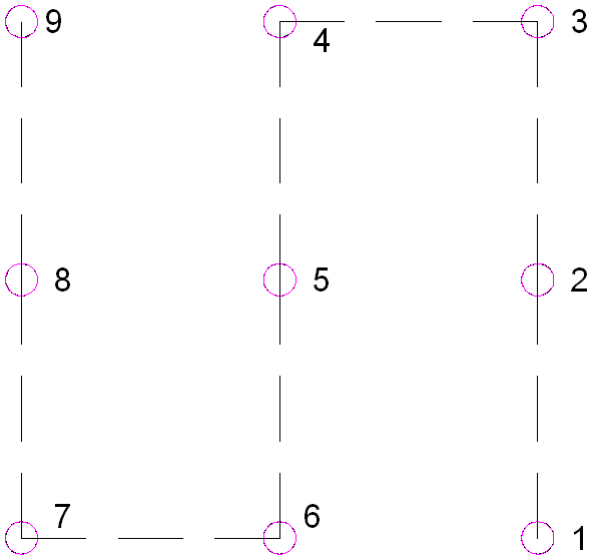
© 2011 CIM-TECH.COM, Inc



Decreasing X, Decreasing Y

Seq_Sort -- Zigzag Decreasing X, Increasing Y

This will sort cuts starting at the farthest lead-in in X, but lowest Y and then proceed to the next closest lead-in while decreasing in X and increasing Y values.

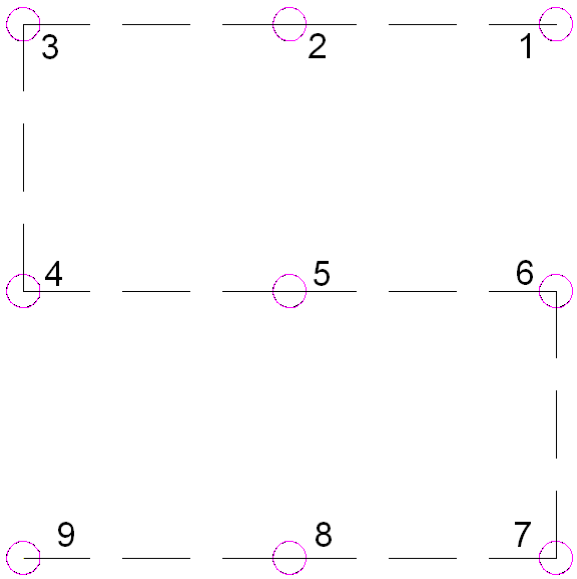


Decreasing X, Increasing Y

Seq_Sort -- Zigzag Decreasing Y, Decreasing X

This will sort cuts starting at the farthest lead in in X and Y and then proceed to the next closest lead-in while decreasing in Y and decreasing in X.

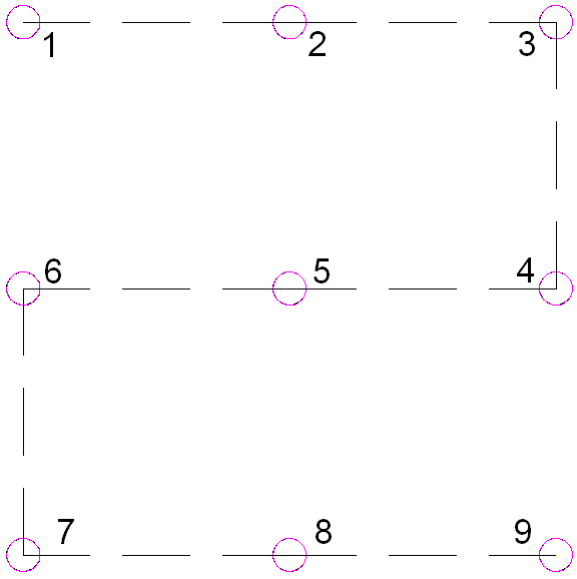
© 2011 CIM-TECH.COM, Inc



Decreasing Y, Decreasing X

Seq_Sort -- Zigzag Decreasing Y, Increasing X

This will sort cuts starting at the farthest lead-in in Y, but closest in X, and then proceed to the next closest lead-in while decreasing in Y and increasing in X.

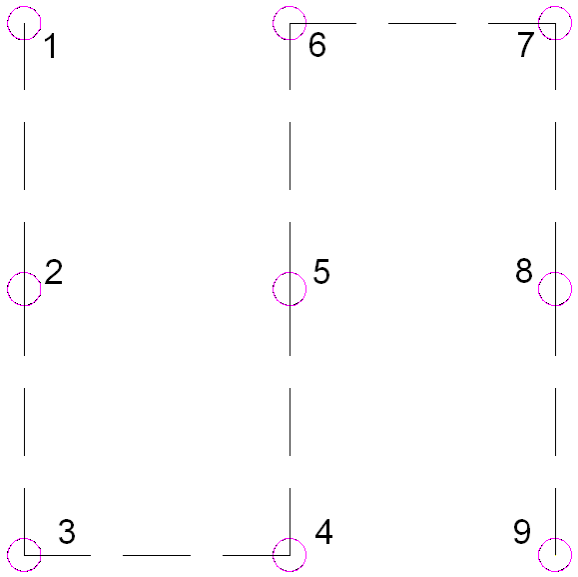


Decreasing Y, Increasing X

Seq_Sort -- Zigzag Increasing X, Decreasing Y

This will sort cuts starting at the closest X and farthest Y lead-in, then proceed to the next closest lead-in while increasing in X and decreasing in Y.

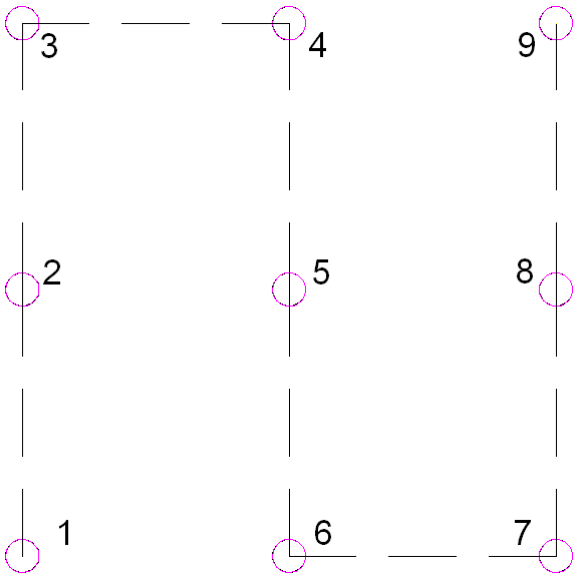
© 2011 CIM-TECH.COM, Inc



Increasing X, Decreasing Y

Seq_Sort -- Zigzag Increasing X, Increasing Y

This will sort cuts starting at the closest lead in in both X and Y and then proceed to the next closest lead-in while increasing in X and Y values.

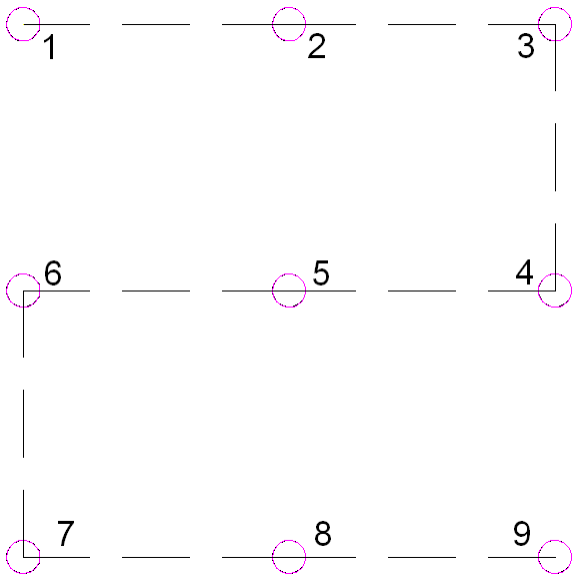


Increasing X, Increasing Y

Seq_Sort -- Zigzag Increasing Y, Decreasing X

This will sort cuts starting at the closest X, but furthest Y lead-in and then proceed to the next closest lead-in while increasing in Y and decreasing in X.

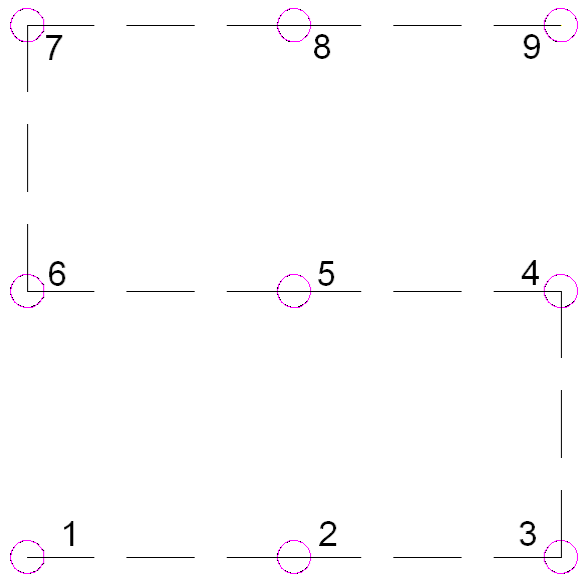
© 2011 CIM-TECH.COM, Inc



Increasing Y, Decreasing X

Seq_Sort -- Zigzag Increasing Y, Increasing X

This will sort cuts starting at the closest lead-in in both X and Y and then proceed to the next closest lead-in while increasing in Y and increasing X.



Increasing Y, Increasing X

© 2011 CIM-TECH.COM, Inc

9.2 Tooling Requirements



The Tooling Requirements section will cover items that affect the elements within the tool paths, like whether or not to use Sub Programs or Starting and Ending safety blocks.
Each section will be discussed in more detail.

9.2.1 Make Subroutines



Make subroutines is the development of subroutine information that is attached to the sequence and to each of the cut blocks. This information is then used by the task functions in 'Develop Tooling Motions' to form the correct commands representing subroutine construction.

The developed subroutine information is not easily seen since it resides in the Router-CIM internal database (NCPS) as lists of information to be used later by other commands. A user will not see the result until tooling motions are developed and NC code generated. A programmer can look at the results by examining the supplied NCPS Variable Definitions during the develop tooling motions command.

Variables

There is an NcVar (under System) called Auto_Sub_Num. When this variable is set to T for True (on), SubRoutines will increment in hundreds, starting at 100 and incrementing by 100 for each sub. If this variable is changed to nil for false (off), you will be prompted at the command line for the Sequence ID number (put in any number as you would normally to call out SubRoutines).

Sub-Programming

Sub-Programming is a method used to aid in the editing of NC Code and to minimize redundant code. To generate a program using Incremental Sub-Programming, Copy an existing Cut to the locations where you want to call up the Sub-Program. Each copied Cut will be part of the Sub-Program.
* *Note**: The Make SubRoutines box must be checked in the NC Sequence Builder (sequence command), for the program to generate Sub-Programs. If the box is left un-checked, then no Sub-Programs will be generated, even if the proper sub routine is followed. The code produced will repeat for each occurrence of the Cut.

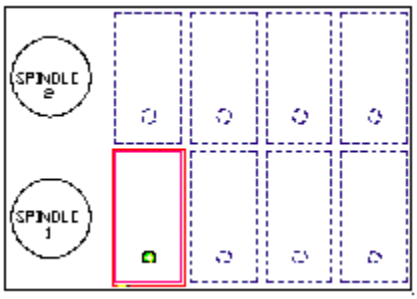


EXAMPLE: Make 8 parts using spindles 1 and 2 using ½" end mill in each spindle.

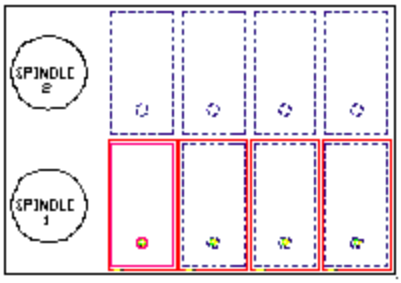
STEP 1. Geoshape and Cut parts. Do not include any slaving spindle parts or parts using the same

© 2011 CIM-TECH.COM, Inc

spindle(s) that the Cut can be copied to. The dashed lines(parts) are not included in the pick because they are either slaving spindle parts (parts in line with Spindle 2) or the cuts can be copied from the first part because they use the same spindle and they have the same part profile.



STEP 2. Copy or Array Cut(s) to new locations. In this example only the Cuts that spindle 1 will be cutting are copied. (Remember, Spindle 2 is slaving whatever Spindle 1 is doing).

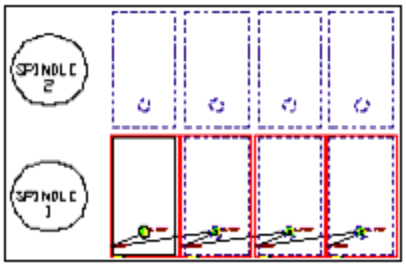


A Sub-Program is required for each different spindle (or spindle combination) used. There are an almost endless variety of Cut combinations that you can use to make multiple parts.

STEP 3. Sequence the parts. Select all parts in order to be Cut, or if you are going to select Sort Cuts By on the NC Sequence Builder dialog window, select the parts by putting a window around all the parts (this includes copied CUTS).

STEP 4. Press <Enter>. If selecting manually, this step is selection sensitive: the order in which you select the Cuts will be the order in which they will be cut in the program.

STEP 5. The NC Sequence Builder dialog window will appear. Select and enter all appropriate information required. Make sure to select the Make Subroutines box. When complete, click on <OK>. Now your drawing should be back on the screen and the command prompt will be asking you to select single cuts for sub-program creation. Always press <Enter> to this prompt and the Sequence command will begin gathering all information you entered previously in the NC Sequence Builder and will create NC code.



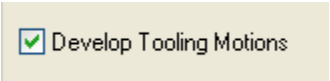
The code will then appear on the screen for you to view. (Only if you selected Yes to the Preview Code option under the NC Codes Options dialog window

Note

© 2011 CIM-TECH.COM, Inc

After entering in all the information required in the NC Sequence Builder, select <OK>. If you are making Incremental SubRoutines, you will just press <Enter> at the next prompt that follows, if you are making individual SubRoutines for a given part(s), you will select that part(s) individually now and then press <Enter>.

9.2.2 Develop Tooling Motions



Develops the tooling motions between cut cycles such as Index moves.

Develop Tooling Motions is the adding of NC objects into the sequence that represent tooling motions and commands required to traverse from one cut cycle to another. Typically these objects include index motions and tool change commands.

The option for the use of start and end codes is supplied to control what codes the task functions supply at the start and end of the sequence definition.

This process is a loop over all the cut blocks defined to the sequence. Two functions are performed. The SEQTSK1 task function develops NC objects that appear at the beginning of each cut block. The SEQTSK2 task function develops NC objects that appear at the end of each cut block.

If SEQTSK1 and SEQTSK2 functions are not defined, the Process MAKE Simulator is used to loop over all the cut blocks.

All the activity of determining what is appropriate NC objects is made through the use of task functions. These functions used by SEQTSK1 and SEQTSK2.

See NCPS Variable Definitions for programming variables.

The result of 'Develop Tooling Motions' is an expanded sequence with more objects in the appropriate locations in the sequence. Usually this can be seen by the index lines developed between each of the cut blocks.

9.2.3 Block Sequence



Block Sequence converts a sequence group into a NC object block

Block Sequence develops a block using the sequence information.

This is normally done when a sequence is fully developed with tooling motions and no start and end codes. The resulting blocked sequence is to be used as a cut block for another sequence. This is known as a nested sequence.

Sequences do not have to be blocked to produce NC code.

9.2.4 Start and End Codes



© 2011 CIM-TECH.COM, Inc

Yes is the default.

Yes will give you the safety blocks in the beginning and at the end of the program. It will also provide the tool calls for each cut. Setting this feature to No will not provide any safety blocks or tool changes, but just the XYZ motions of the cut.

For example, all of the cycles to be cut by the same spindles at the same time can be grouped into a Sequence without Start and End Codes (by selecting No). Each group of Sequences can then be grouped again into a final Sequence with Start and End codes.

Here is the code for a sequence with Start and End Codes set to NO:

```
%
:1
N1G41D01G01Z-.75F200.
N2G03X3.Y-.249I-.5F500.
N3G01X.5
N4G02X-.249Y.5J.749
N5G01Y3.5
N6G02X.5Y4.249I.749
N7G01X5.5
N8G02X6.249Y3.5J-.749
N9G01Y.5
N10G02X5.5Y-.249I-.749
N11G01X3.
N12X2.5F500.
N13G03X2.Y-.749J-.5
N14S18000
N15G00G43H0Z.25
N16G40G00Y-1.024
%
```

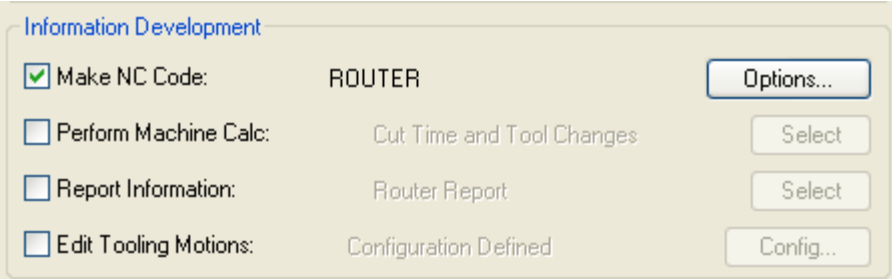
Here is the same tool path with the Start and End Codes set to YES. The code in red is what Start and End Code added to the program.

```
%
:1 (START AND END CODES ON)
N1G00G17G20G28G40G80G91Z0M5
N2G90
N3G52X0Y0Z0
N4G08P1
N5M08
N6(ROUTER-BIT .5 DIA.)
N7G28G91Z0M05
N8G90T2001M06
N9T102
N10M03S18000
N11G00G17G55X3.5Y-1.024
N12G00G43H1Z.25
N13G41D01G01Y-.749F200.
N14Z-.75
N15G03X3.Y-.249I-.5F500.
N16G01X.5
N17G02X-.249Y.5J.749
N18G01Y3.5
N19G02X.5Y4.249I.749
N20G01X5.5
N21G02X6.249Y3.5J-.749
N22G01Y.5
N23G02X5.5Y-.249I-.749
N24G01X3.
```

© 2011 CIM-TECH.COM, Inc

```
N25X2.5F500.
N26G03X2.Y-.749J-.5
N27G00Z.25
N28G40G00Y-1.024
N29G28G91Z0M5
N30G28G91X0M09
N31G90
N32G52X0Y0Z0
N33G08P0
N34M30
%
```

9.3 Information Development



The Information Development section of the Sequencer allows for the creation of the NC Code file, reporting, and tool emulation and editing.

9.3.1 Make NC Code

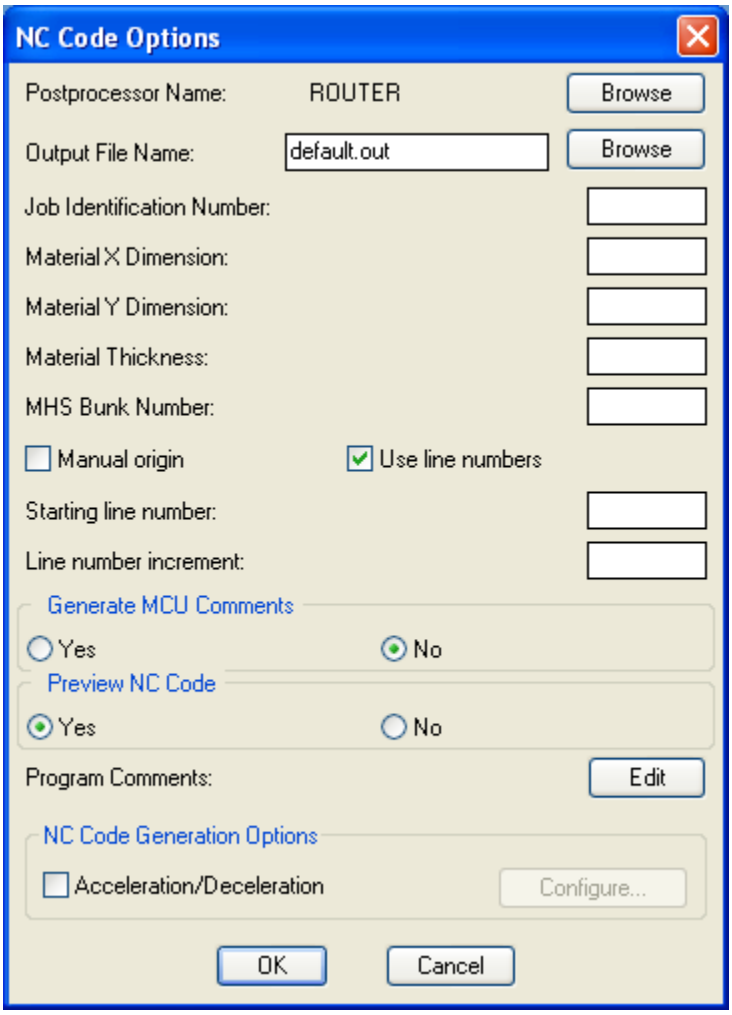
Make NC Code develops an NC Code file for a selected sequence

To define the options for developing the NC Code see the section NC Code Options section.

This command uses the programmable postprocessor through the task function ncpp_seq_nccode.

When selected, click on the Options button to the right and a NC Code Options dialog window will appear. From this screen you will have the following options:

© 2011 CIM-TECH.COM, Inc



Most of these settings have NC System Variables that control their use.
POST - postprocessor name
NCFIL - output file name
NCFILEXT - output file name extension
NCFILLOC - output file directory
JOBID - job identification number
VIEWCODE - Preview code option
_ACC_DEC - Acceleration/Deceleration option

Any or all of these can be defaulted by using the NCVAR command.

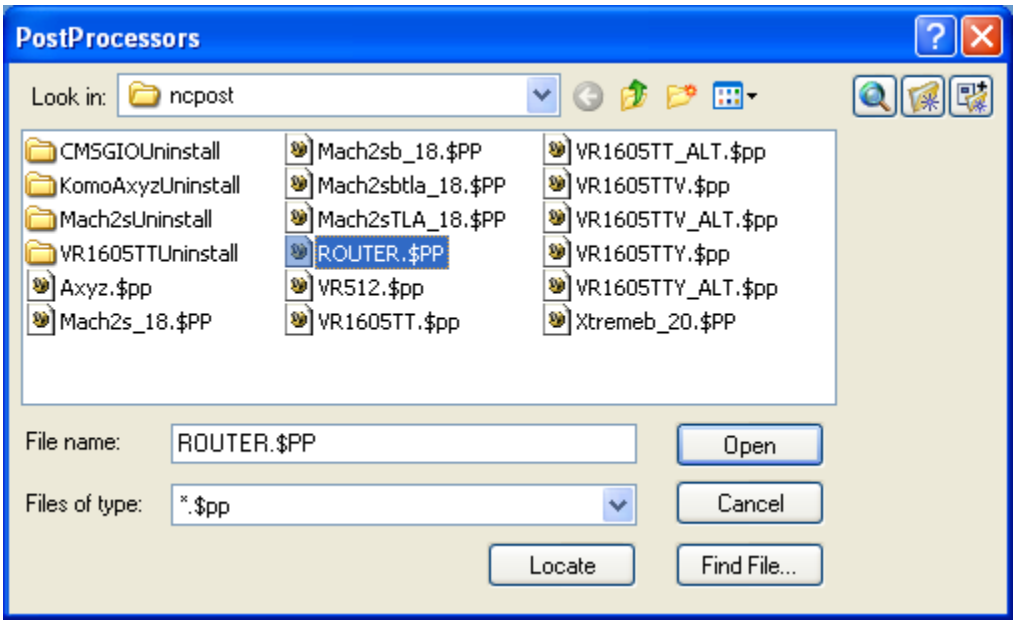
9.3.1.1 Make NC Code Options

Make NC Code Options provide all the machine specific options available during the creation of the sequence.

Postprocessor Name

A default name follows this field. If you want to change the Postprocessor, click on the Browse button to the right and you can select a different Postprocessor. The software is configured to direct you to the Ncpost folder if you do click on the browse button.

© 2011 CIM-TECH.COM, Inc



Output File Name

The default shows you the Path where the code will be written to and the name of the NC Code file (it defaults to the name of your current drawing) with an .OUT extension. You can change the name of the file or the extension to anything you want (as long as the Path is valid.)

Job Identification Number

This field is where you will enter the 4-digit Main Program Number (or whatever numbering system your machine typically uses).

Material Thickness Comp.

If you program your parts with 0,0 being the top of the spoil board you will need to enter a material thickness here. If you program 0,0 as the top of your part, you can leave this field blank.

Manual origin

When selected, you can tell Router-CIM where you want 0,0 to be on the screen (it can be anywhere) instead of it being at the lower left corner (by default). This is useful if you are programming multiple parts on your screen, in that you won't have to move the various parts to 0,0 then move them back out of the way to move another part down to 0,0. If you turn Manual origin on, you will be prompted for where you want 0,0 to be. (You can change the 0,0 location as many times as you need).

Use line numbers

If this box is checked, the NC Code produced will start with line number N1 and increment or increase by 1 for each line.

If this box is not checked, The NC Code produced will have no line numbers. This option might be chosen if you wish to save memory space at the controller.

Starting line number

If you do not want you your Starting line number to be 1, you may change it to what you wish in this first field (100 for example).
It will still increase or increment by 1.

Line number increment

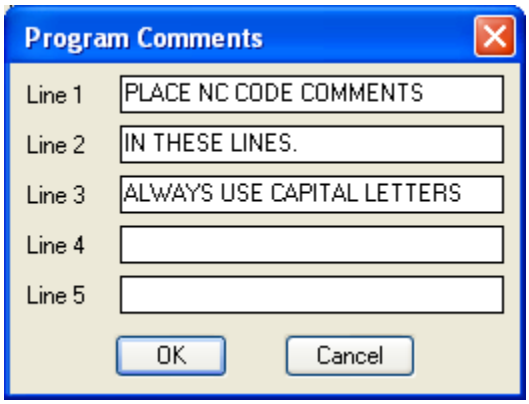
If you want to change how the numbers increment or increase, you may do this in the Line number

© 2011 CIM-TECH.COM, Inc

increment field. (You may want the numbers to increase by 10's rather than 1's).

Program Comment

Click on the Edit button to the right and a dialog window will appear. You are given five lines where you can enter any comments you wish to appear in the program.



Preview Code

This field gives the options, Yes or No, of whether or not you want the code to pop up on the screen for viewing after it has been written.

9.3.1.1.1 Acceleration/Deceleration

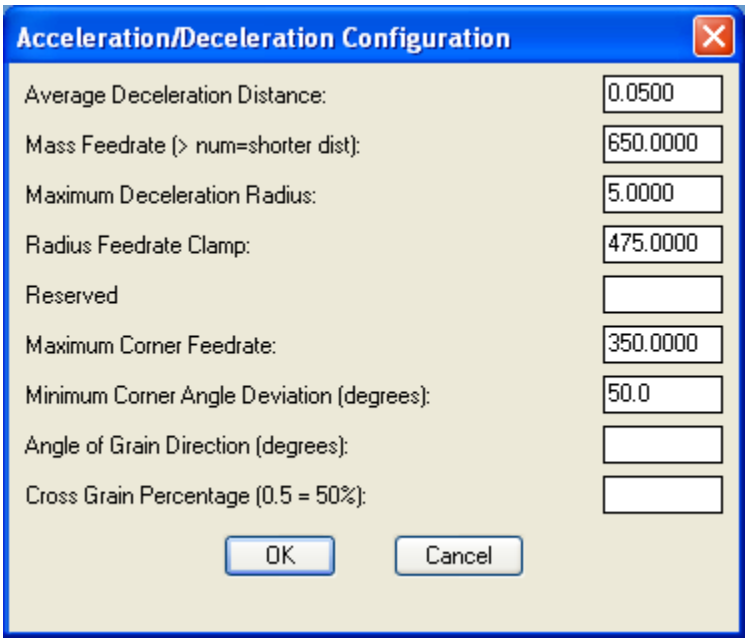


You can apply Acceleration and Deceleration to your NC program by selecting this option. Each motion in the NC Code will be examined for feed requirements and adjusted if necessary. The configuration button allows for the changes to the parameters that control acc/dec.

ACC 'n DEC anticipates change in direction, short moves and/or tight corners, and automatically inserts slow down or control moves into the machine code. ACC 'n DEC was designed to enhance the performance on all machine tools regardless of the controller's ability.

Select the option to turn on Acceleration/Deceleration, and then click on the Configure button to the right to display the Configuration options.
An Acceleration/Deceleration Configuration dialog window will appear.
A brief description for each field follows:

© 2011 CIM-TECH.COM, Inc



Average Deceleration Distance

Distance to travel to allow for the Mass Feedrate to occur. This is an average because some rounding can occur.
The NCVAR is _ACC_DECDIST.

Mass Feedrate(> num=shorter dist)

The decrease in feedrate from one element to the next over the distance specified by the Average Deceleration Distance. A ration defines the deceleration scale.
The NCVAR is _ACC_DECFEED.

Maximum Deceleration Radius

This parameter sets the maximum radius that can be cut at the Radius Feedrate Clamp. Any radius less than this will cause Acc/Dec to be applied.
The NCVAR is _ACC_CLAMPRADIUS.

Radius Feedrate Clamp

This is the maximum feed rate that is used on the largest arc setting set in Maximum Deceleration Radius.
The NCVAR is _ACC_CLAMPFEED.

Reserved

Various values are stored here when the system is in use. Not user configurable.

Maximum Corner Feedrate

This is the maximum feed rate that that will cut a corner with an angular deviation greater than the Minimum Corner Angle Deviation so that the cutter will not over/under shoot the corner.
The NCVAR is _ACC_CLAMPMIN.

Minimum Corner Angle Deviation (degrees)

Minimum Angle Deviation of a corner when acc/dec is applied. Any angle deviation between elements or arc sweep that is greater than this value will cause acc/dec to be applied.
Router-CIM will automatically vary the feed rate on angles that are equal or smaller than the angle size set in this field.

© 2011 CIM-TECH.COM, Inc

The NCVAR is _ACC_ANGDEV.

Angle of Grain Direction (degrees)

This is the angle of the grain on the part in degrees. Angles from grain direction are cut at a feedrate scale based on the Cross Grain Percentage.
The NCVAR is _ACC_GRAINDIR.

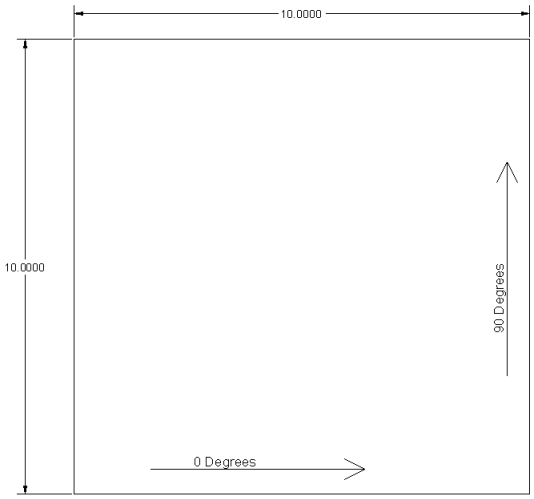
Cross Grain Percentage (0.5=50%)

The percentage of current feedrate to cut 90° across the grain of the part. The value input here (must be in decimal form) will give you a feed rate equal to the percentage, entered in this field, of the maximum feed rate set in the Control Panel.
The NCVAR is _ACC_CROSSGRAINPER.

How Acc-n-Dec works

To show how Acc-n-Dec works, we will use a simple example.

EXAMPLE:
A part with an outside corner that has a 90° change between two elements that define the corner. Each element is a line 10 units long. First element is at 0 degrees and the second element is at 90 degrees.



Acc-n-Dec part

Current setting are:

_ACC_DECDIST = 1.0
_ACC_DECFEED = 100
_ACC_CLAMPMIN = 5
_ACC_ANGDEV = 15
_ACC_GRAINDIR = 0.0
_ACC_CROSSGRAINPER = 1.0
_ACC_FEEDRATE = 200 (normal programmed feedrate)

Results:
The angle deviation is greater than _ACC_ANGDEV and the current feedrate exceeds the maximum corner feedrate expressed in _ACC_CORNFEED. Therefore, acc/dec will be applied.
The _ACC_DECDIST/_ACC_DECFEED ratio is applied to the current feedrate to determine a distance from the corner where the _ACC_CORNFEED will be applied. In this example, the distance from the

© 2011 CIM-TECH.COM, Inc

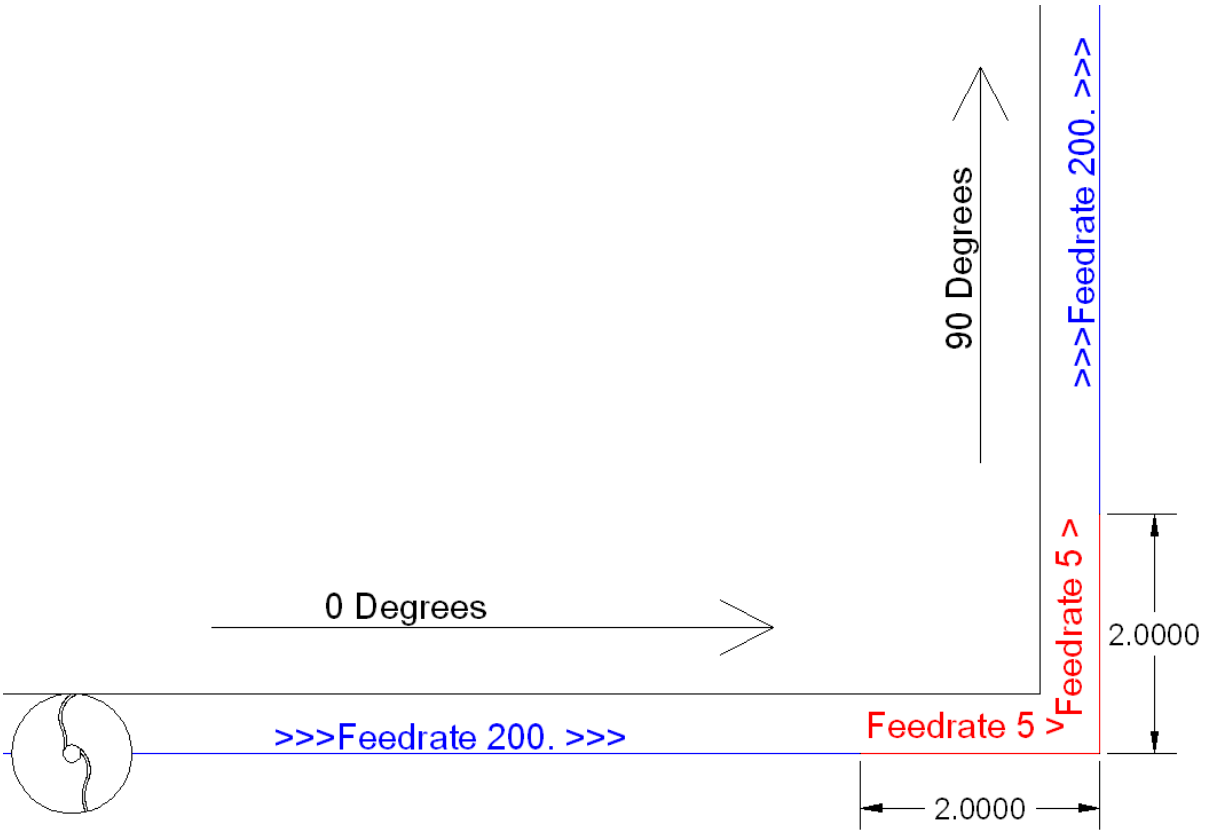
corner is

DIST = (Programmed Feedrate - Max Corner Feedrate) * Average Deceleration Distance / Mass Feedrate **OR**
DIST = (/ (* _ACC_DECDIST (- _ACC_FEEDRATE _ACC_CLAMPMIN)) _ACC_DECFEED) **OR**
DIST = (/ (* 1.0 (- 200.0 5.0)) 100.0) = 1.95 units

Each of these three formulas are the same.

The first element will be converted into two separate elements so that one can be the deceleration move. The first will be 8 units in length and the second (the deceleration segment) will be 2 units in length.
The first element will have a feedrate of _ACC_FEEDRATE (200), which is the normal programmed feedrate.
The second element will have a feedrate of _ACC_CLAMPMIN (5).
This will allow the machine to ramp down from a feedrate of 200 to a feedrate of 5 over the course of 2 units, just prior to the corner.

Since the _ACC_ACCDIST is zero in this example, the third element (originally the second element at 90°) will then be broken up into 2 pieces equal to the last two on the deceleration side. The first will be 2 units long and have a feedrate of 5 and the second have a feedrate of 200 and the length of 8 units.



Example Code:

```
N01 G91 G01 X8.0 Y0.0 F200.
N02 G01 X2.0 Y0.0 F5.
N03 G01 X0.0 Y10.0 F200.
```

© 2011 CIM-TECH.COM, Inc

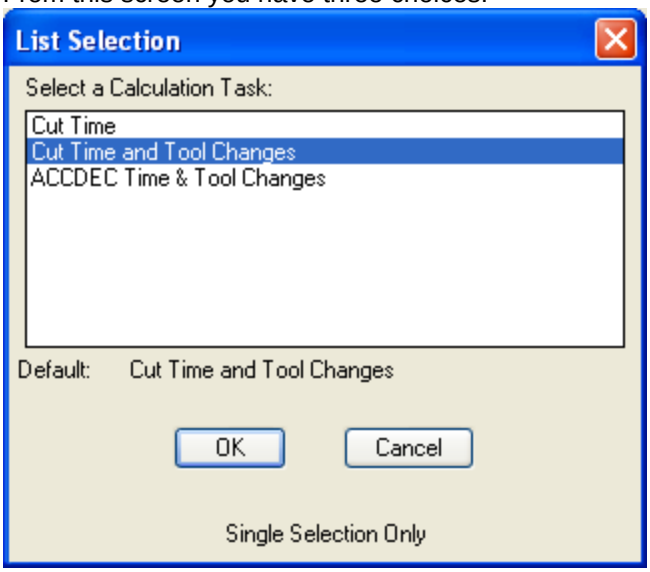
If the corner was filleted, the feedrate and length of the deceleration segment would be determined by using the radius feedrate scale ratio (_ACC_CLAMPRADIUS / _ACC_CLAMPFEED).

If _ACC_GRAINDIR is not NIL, then additional limits would be placed on all feedrates that went on the cross grain angle based on the cross grain percentage. By multipling the current feedrate by the cross grain percentage, a new feedrate is determined.

SPECIAL CONDITIONS:
If an element is not long enough to handle the required de-acceleration segment(dec-segment) length, then the feedrate will be applied to the start of the element. This will, at least, ramp down over the element's length even though full deceleration feedrate will not be achieved at the end of the element.

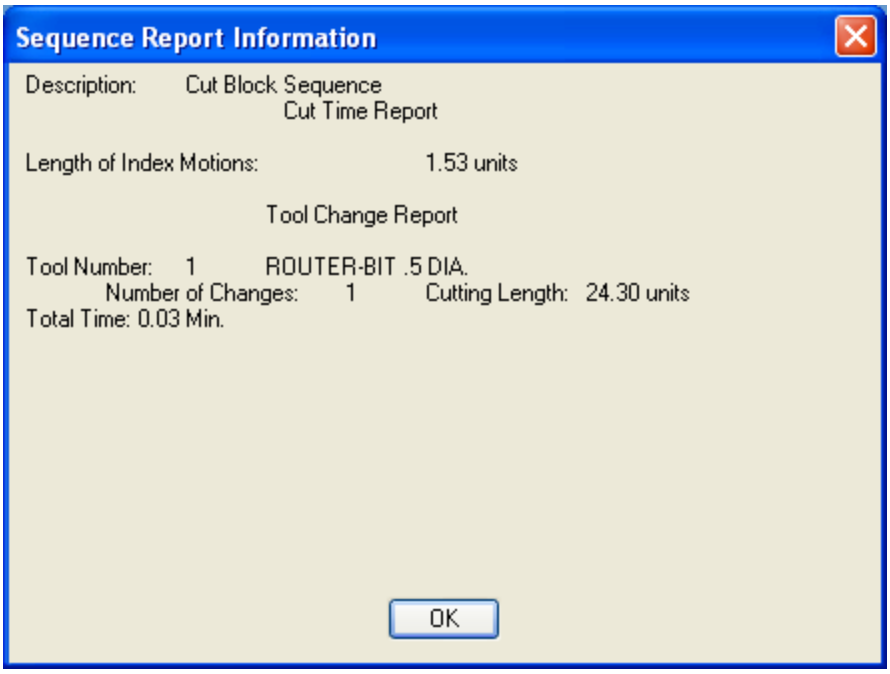
9.3.2 Perform Machine Calc.

After selecting, click on the Select button to the right. A List Selection dialog window will appear. From this screen you have three choices:



Cut Time
Selecting on this field, Router-CIM will figure the approximate cutting time, in minutes, for the Cut or Sequence selected.
Cut Time and Tool Changes
Selecting on this field, Router-CIM will figure the approximate cutting time, in minutes, and will add the time for a tool change for the Cut or Sequence selected. The time figured for the tool change is in a NcVar called *TOOL_TIME*. To change the value figured for the tool change:
1) Type in NCVAR at the command prompt.
2) Click on OTHER, scroll down and click on *TOOL_TIME* and enter the new value.
3) Press <Enter>, then click on <OK>.
ACCDEC Time & Tool Change
Selecting on this field, Router-CIM will figure the Acceleration/Deceleration, Cut Time and Tool Change Times for the Cut or Sequence selected. You will get a cut time based on acceleration and deceleration. This cut time is only valid if you make your code with ACCDEC on. If you did not use ACCDEC in your code, then select just Cut Time or Cut Time and Tool Changes.
Note: After running Sequence, a dialog window called Sequence Report Information will appear showing the time.

© 2011 CIM-TECH.COM, Inc



To see the report again after Sequence has been run, click on Sequence from the Control Panel or Router-CIM toolbar, click on the Clear All button at the top of the NC Sequence Builder screen, select Report Information, click on the Select button to the right, click on one of the two choices, click on <OK>, and then click on <OK>.

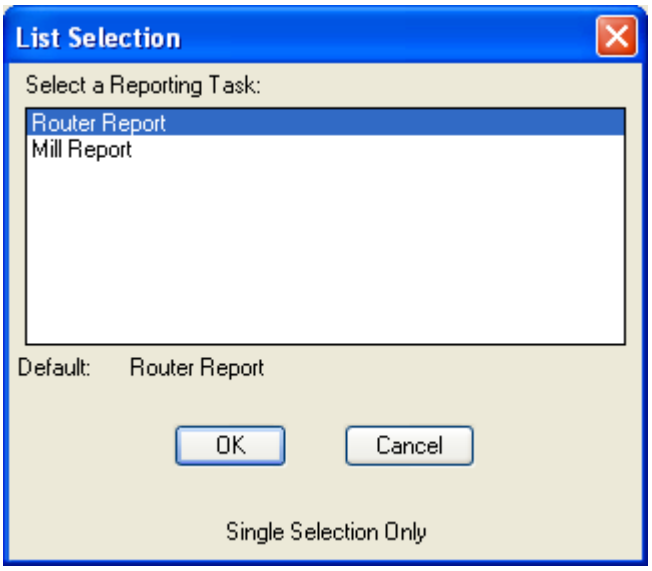
Using this option with Router Report turned on also will generate comments in the code like the ones shown below in red.

```
%
:12
(DRAWING NAME YOURDRAWINGNAME)
(PROGRAMMERS NAME HERE)
(11-06-2008 AT 11:03)
(ROUTER-BIT .5 DIA. TOOL 1 TOOL LENGTH 1 WORK CORD 55)
(CUTTER COMP 1)
(TOTAL TIME: 0.03 MIN.)
N1G00G17G20G28G40G80G91Z0M5
N2G90
N3G52X0Y0Z0
```

9.3.3 Report Information

After choosing this box, click on the Select button to the right. A List Selection dialog window will appear and from here you have two choices of output: Router Report or Mill Report.
Under normal use you should choose Router Report.

© 2011 CIM-TECH.COM, Inc



9.3.4 Edit Tooling Motions

Edit Tool Motions provide the editing and emulation of selected tool paths

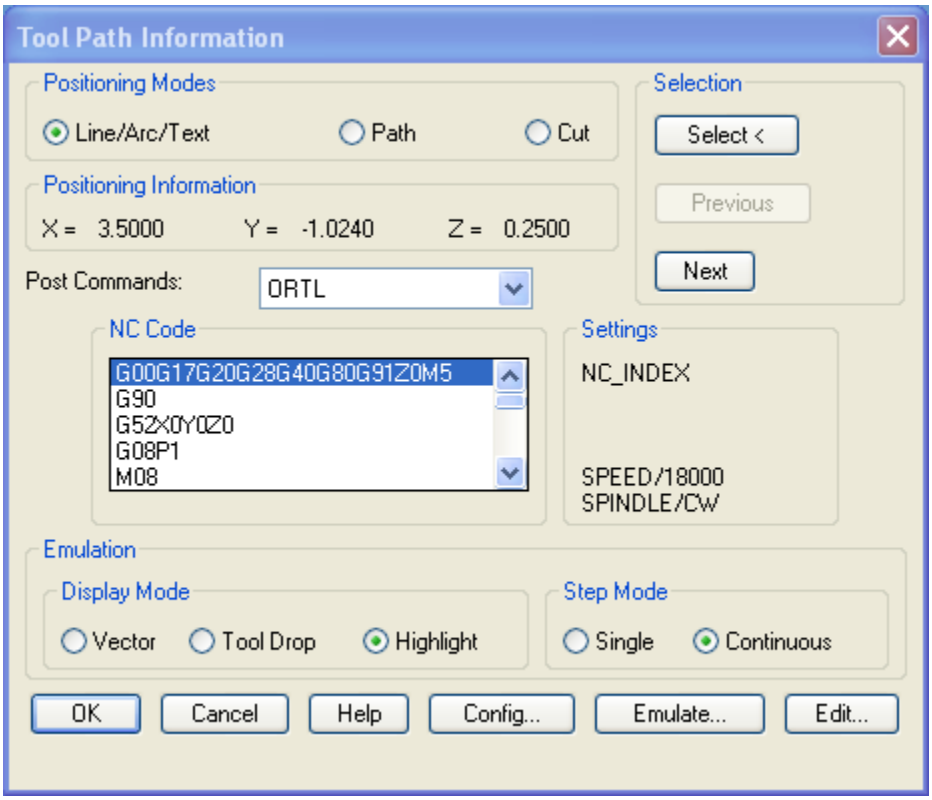
The Tool Path Editor provides for the displaying of information, emulation of tool path and the editing of tool paths. The three dialog interfaces are described as:

Tool Path Information - The main dialog screen is used to set modes and traverse the NC program. The display includes the NC Code and current settings that are in the NC Program. Selection modes are available to rapidly traverse through the NC Program. The modes for emulation are selected in this dialog.

Tool Path Emulation - This dialog provides for display settings and a traverse button to move through the NC Program during Emulation. The NC Program is emulated each time the NEXT button is activated. The mode of emulation was determined in the Information dialog display. Emulation methods include Tool Vector display, Tool Drop (inserting of tool block) and highlighting (drawing of the tool path in different colors).

Tool Path Editing - This dialog is used to pick the form of editing to apply and whether the edits are to be done globally or locally in the NC Program. When OK is selected, the actual edit takes place.

© 2011 CIM-TECH.COM, Inc



Areas in this dialog box that are not defined in other sections are described below:

Positioning Information

This area displays the current X,Y Z position at the location defined by the editor pointer. These coordinates at world space as defined by AutoCAD.

Post Commands

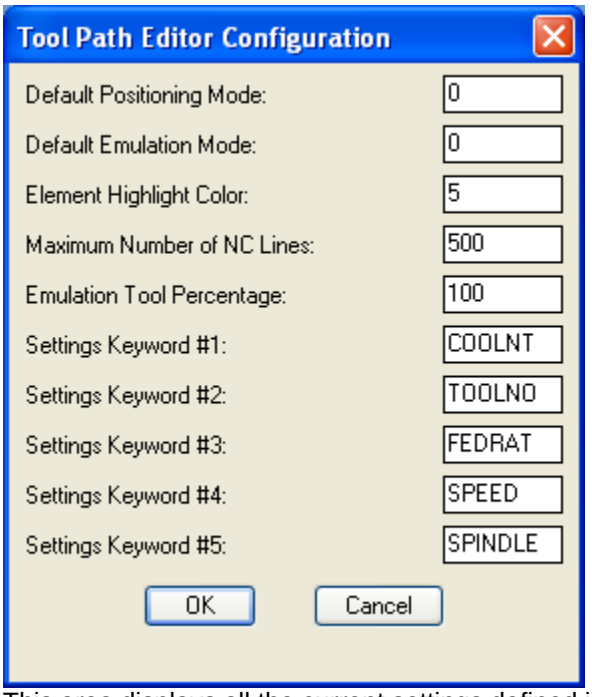
This area displays all the post processor commands used to develop the line of NC Code where the editor pointer is located.

NC Code List

This area displays the associated NC Code for the current Work Space. When the editor pointer is incremented the associated line of NC Code is selected in this list. This list can be used to move the editor pointer by scrolling to a location in the list and selecting the desired line of NC Code.

Settings

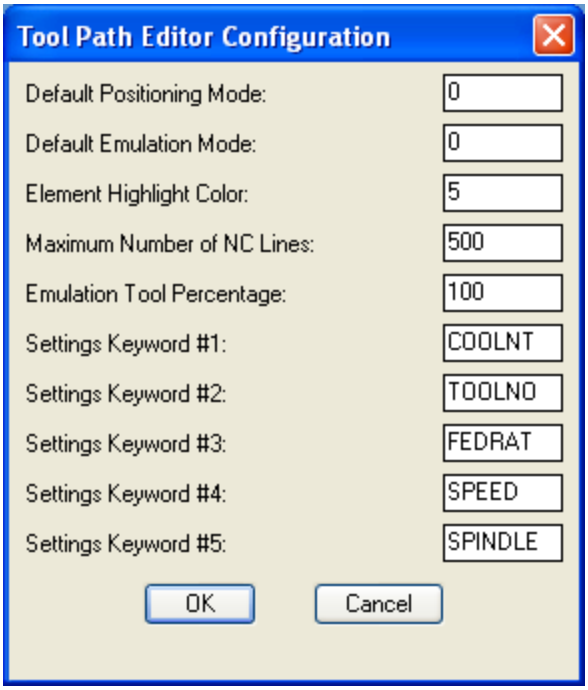
© 2011 CIM-TECH.COM, Inc



This area displays all the current settings defined in the Editor Configuration. Each time a Setting Keyword is encountered it is place in this area of the dialog. Each time the Editor pointer is moved this scan is performed.

9.3.4.1 Sequence Emulator Options

The Sequence Emulator Options Set different modes and values that control the Tool Path Editor



© 2011 CIM-TECH.COM, Inc

Default Positioning Mode
Positioning Modes: 0 = line/arc, 1 = path, 2 = cut
Line/Arc - positions along a cut by one single motion at a time.
Path - positions along a cut for an entire cut path at a time.
Cut - positions along an entire cut block at a time.

Default Emulation Mode
Emulation Modes: 0 = highlight, 1 = tool drop, 2 = vector

Highlight - draws the object using the Highlight color.
Tool Drop - inserts the tool geometry at intervals along the cut path.
Vector - draws vectors representing the tool alon the cut path.

Element Highlight Color
Highlight Color - a number representing an AutoCAD color. An object is highlighted by redrawing the object using this color.

Maximum Number NC Lines
Maximum of NC Code Lines - a buffer of information is stored in memory in which the editor works on. The size of this work space is determined by the approximate number of NC code lines that would be generated within the work space.

Emulation Tool Percentage
Emulation Tool Percentage - When using Vector or Tool Drop emulation, the spacing between each emulation motion is determined by the percentage of the tool radius.

Settings Keyword #1
A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #2
A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #3
A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #4
A postprocessor command word that will appear in the setting areas when encountered in the sequence.

Settings Keyword #5
A postprocessor command word that will appear in the setting areas when encountered in the sequence.

10 Knowledge

Knowledge

Defining Knowledge
Router-CIM knowledge blocks are the most powerful part of the Router-CIM and Router-CIM system.

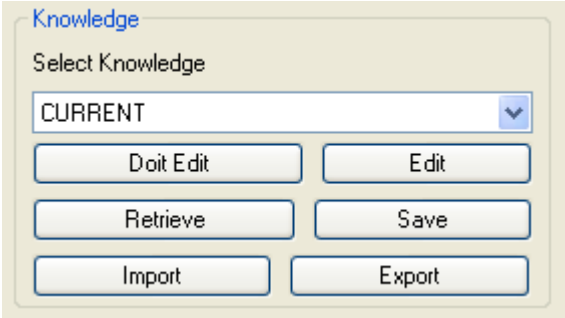
© 2011 CIM-TECH.COM, Inc

All of the information stored in Tool, Cycle,and Status Information (parameters) for *ANY* cut can be stored in a file, or, in a ***knowledge*** block. These cutting conditions can be stored and named within a drawing or stored and named on a hard or floppy disk. Disk based knowledge files can contain several knowledge blocks, making them a sort of library of cutting conditions.

You can store as many of these knowledge libraries on your computer as you wish and import them into a drawing when needed. Alternately, you may store several knowledges in your default drawing, and use them whenever Router-CIM or Router-CIM are running.

Router-CIM uses knowledge stored in individual drawings (in the knwdir folder) to allow you to use specific knowledge for a particular job. By keeping the knowledge for a specific function sperate, you keep the file sizes smaller, and the system runs faster.

Different forms of the knowledge command exist for building, extracting, and saving cutting conditions within a drawing or from a disk drive.



10.1 Retrieve Knowledge

Retrieve Knowledge

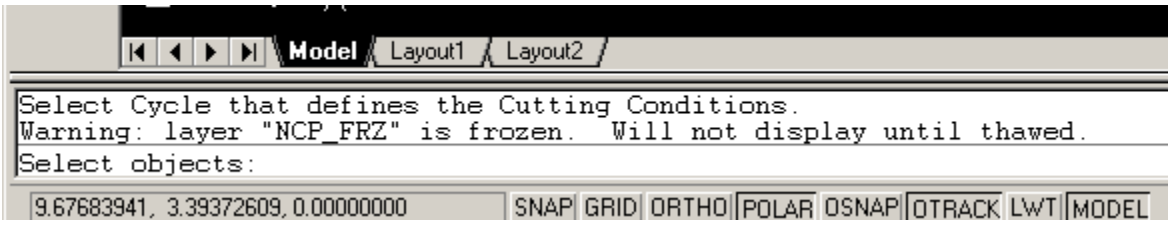


The Retrieve Knowledge ("K" at the keyboard) command is used to reset cutting conditions from an existing Cut. All the cuts that exist in a Router-CIM drawing have their knowledge stored in the drawing for retrieval.

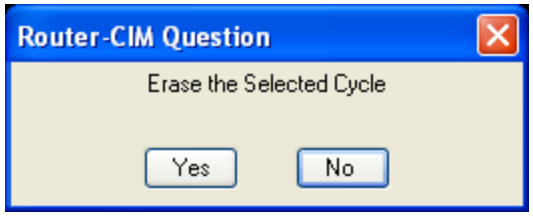
Retrieve

Selecting retrieve knowledge will first prompt you to Select Cycle that defines the Cutting Conditions that you want to retrieve.

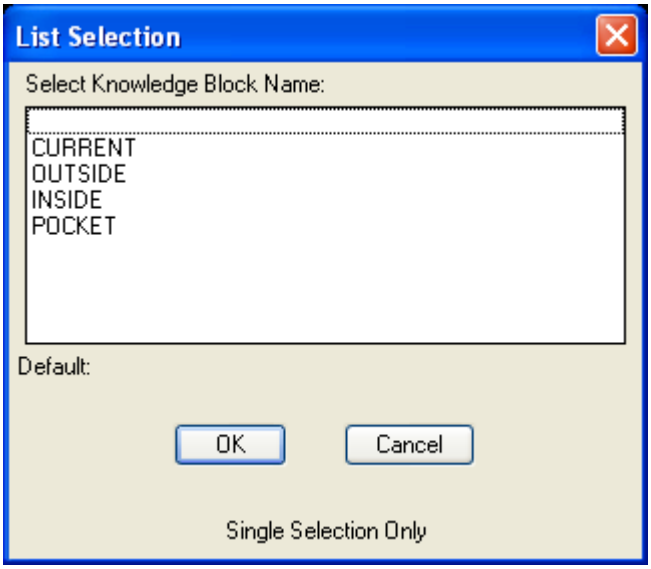
© 2011 CIM-TECH.COM, Inc



If there is a cut on the screen that you want the knowledge from, select it. You will then be prompted as to whether or not you want to erase the selected cut. If you wish to make edits to the cut for current use, answer Yes. Otherwise answer No. If you select ***No***, then the Control Panel will appear showing the attribute values for that condition - you can change any of the attributes or even change the tool or cycle and use any part of the condition in a new Cut.
If you answer ***Yes*** to the erase prompt, your Cut will be erased but the attribute values for that condition will still appear in the Control Panel.

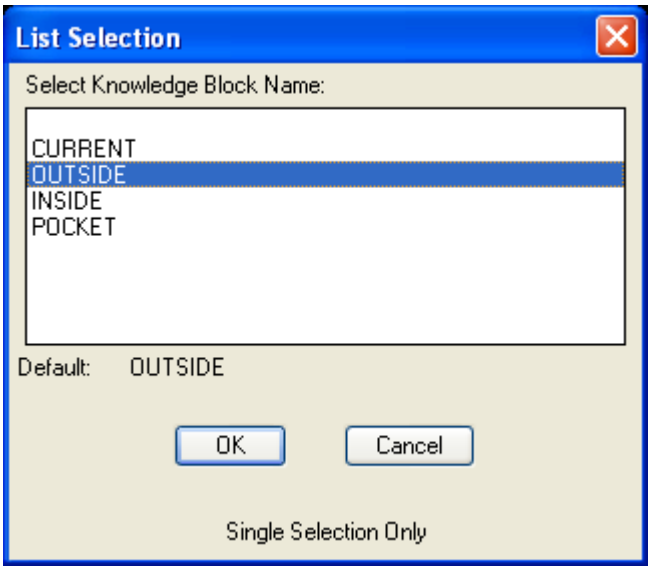


If there is no cut on the screen and you want to select the knowledge from a list of available knowledges, press <ENTER> and a list will be shown of available knowledges.

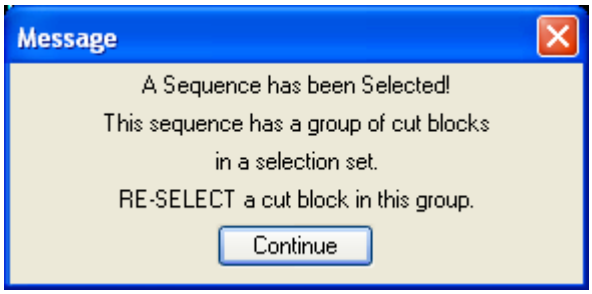


Select the one that you want the information from.

© 2011 CIM-TECH.COM, Inc



NOTE:
If you use Retrieve Knowledge and select a Sequence, you will see a message that a Sequence is selected and be asked to select a CUT block. If you select Continue, and select the cut again, you will get the knowledge from the cut and the Sequence will still exist.



10.2 Save Knowledge

Saving Knowledge



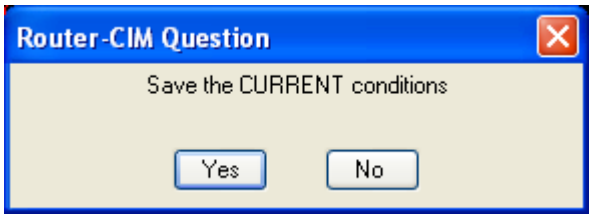
The Save Knowledge ("***SK***" at the keyboard) command is used to Save and name cutting conditions within a drawing.

How to Save a Knowledge

- Select the Save Knowledge button (shown above), enter SK from the keyboard, or use the Save button on the Control Panel to start the command.

© 2011 CIM-TECH.COM, Inc

- You are prompted to select a Cut cycle to Save. If you select a CUT that you want to save, you are then prompted to enter a knowledge block name (17 char. max.).



- You may want to only save the conditions on the Control Panel, and you don't have a cut on the screen to select. When you are prompted to select a Cut cycle to Save, press ***Enter*** and you will be prompted to Save the current cutting condition (screen above). Click on *YES* and you will be asked to give that knowledge a name.

```
Command: SK
Select Cycle that defines the Cutting Conditions
Select objects:
Enter Knowledge Block Name: MYKNOWLEDGE
```

- Enter a name and the knowledge is saved. In this case the name of the knowledge was MYKNOWLEDGE.

```
Enter Knowledge Block Name: MYKNOWLEDGE
Developing Knowledge Block...
MYKNOWLEDGE Knowledge Block Stored.
Command:
```

NOTE:
When you save knowledge, it is only saved in the current drawing. In order to make it available to other drawings, you must Export the knowledge. You can store several individual cut knowledges in one large disk knowledge file. This makes them easer to store and retrieve.

NOTE:
Use the Retrieve Knowledge ("K" at the keyboard) command to retrieve the Saved Knowledge (See Retrieve Knowledge).

10.3 Export Knowledge

Exporting Knowledge



The Export Knowledge ("EK" at the keyboard) command is used to Save and name cutting

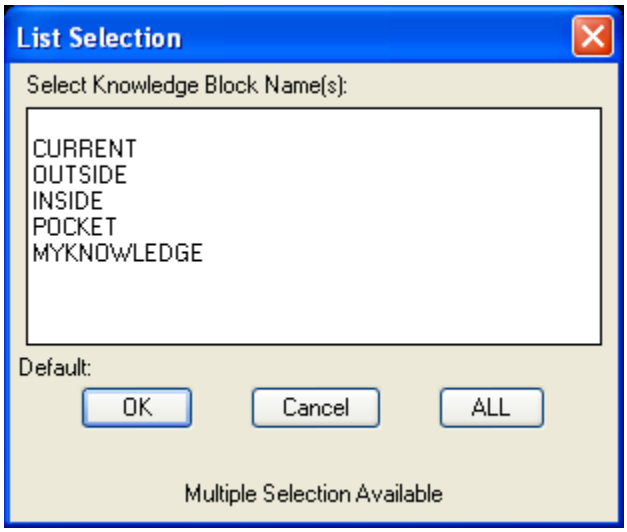
© 2011 CIM-TECH.COM, Inc

conditions to a disk drive. Use the Save Knowledge command first because you must Save the knowledge before you can Export it.

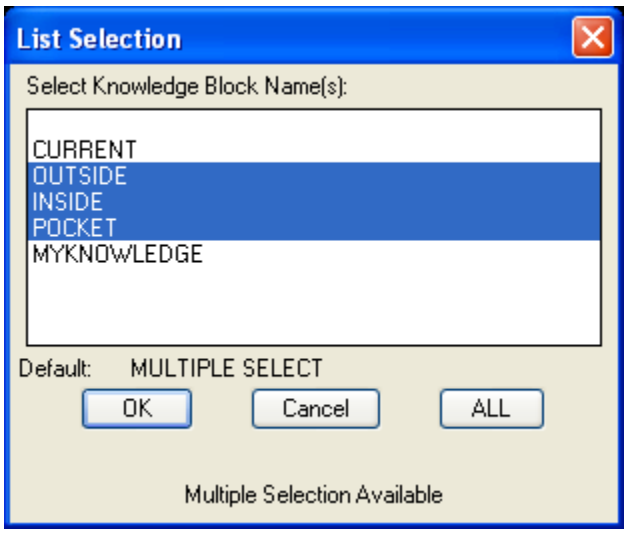
Export

In order to Export Knowledge the following steps must occur.

Select the **Export Knowledge** button, and a pop up menu showing the currently saved cutting conditions will appear.

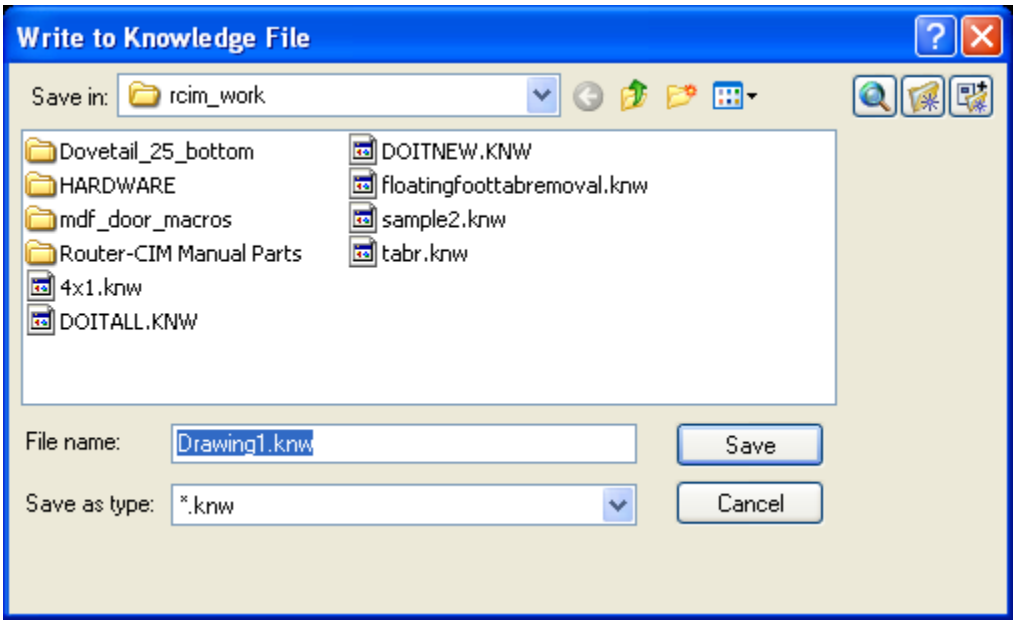


- Pick the cutting condition(s) to Export. Select ***only*** the ones you want to export. Selecting the blank line at the top here and the Current knowledge may cause issues because they are not a name you will want to retrieve later but they will still show up in the list.

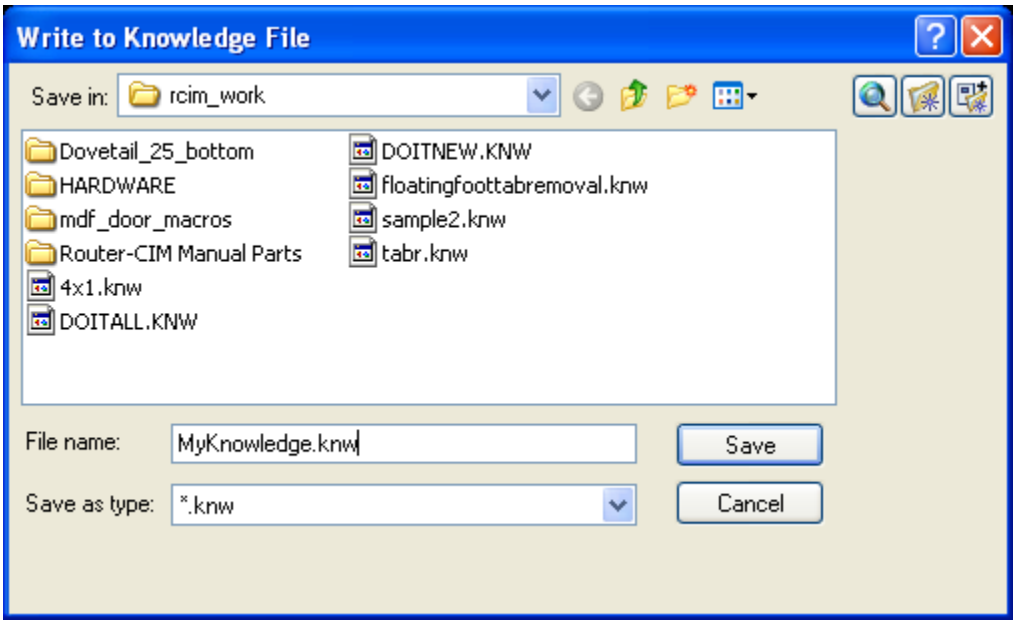


Multiple selection is available by selecting the first knowledge, holding the Shift Key, and selecting the last one. The selection above illustrates this.

© 2011 CIM-TECH.COM, Inc



- You will be shown a window to enter an Export File Name (above). The default name will be shown as the drawing name.



- If desired, enter a different file name.

```
Knowledge Selection - Examining Existing knowledge...
Exporting Knowledge...
Selected Knowledge Stored in the C:\RCIM_WORK\MYKNOWLEDGE.KNW file.
Command:
```

- The Exported file is stored on the disk drive as a .KNW file. The prompt above should appear in

© 2011 CIM-TECH.COM, Inc

your AutoCAD command line.

10.4 Import Knowledge

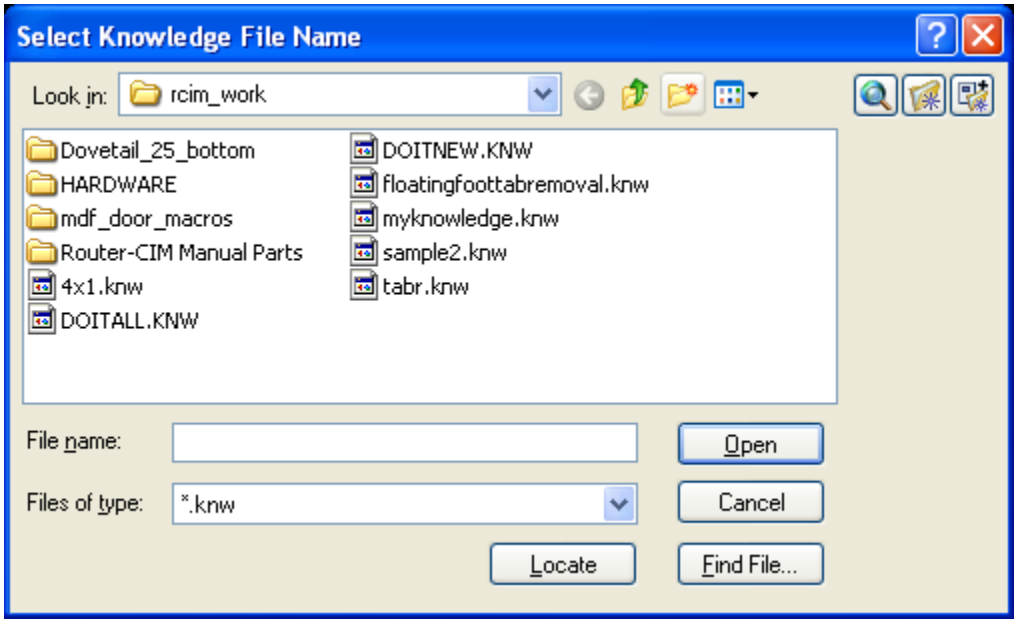
Importing Knowledge



The Import Knowledge ("IK" on the keyboard) command will insert cutting conditions from a disk drive knowledge file.

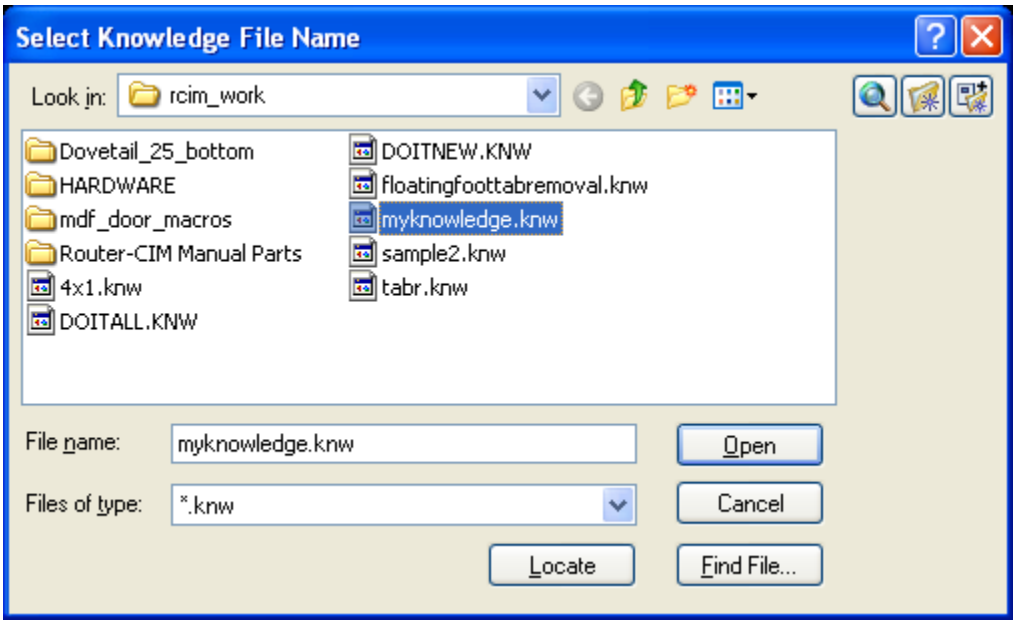
Import

Select Import Knowledge and you will be prompted to enter the name of the knowledge block to Import.

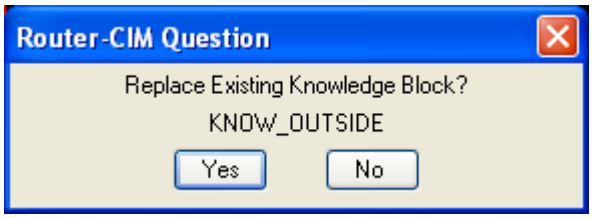


- You may then select the knowledge file that you wish to import into the current drawing.

© 2011 CIM-TECH.COM, Inc



- Once selected, press the Open button.
- If the knowledge file that you are importing contains cut knowledges that are already in the drawing that you are using, you will be prompted to overwrite the existing knowledges with the ones from disk.

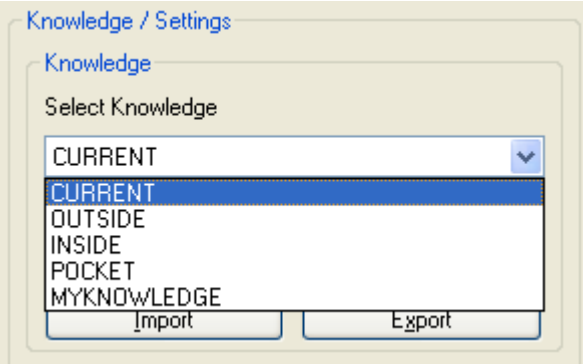


- Selecting Yes replaces the ones in the drawing with the ones from the file you are importing. Selecting No leaves the knowledge from the current drawing intact and the duplicate is not imported.
- After the knowledges have been imported the following message should appear at the command prompt.

```
Knowledge Selection - Examining Existing knowledge...
Importing Knowledge...
C:\RCIM_WORK\MYKNOWLEDGE.KNW Knowledge File Imported.
Command:
```

- The cut knowledges imported can now be seen in the Knowledge pull down menu on the control panel.

© 2011 CIM-TECH.COM, Inc



If you put multiple cutting conditions into a single Export file, the Import Knowledge command will insert all of the individual cutting conditions at one time. In the case above, there were several knowledges imported.

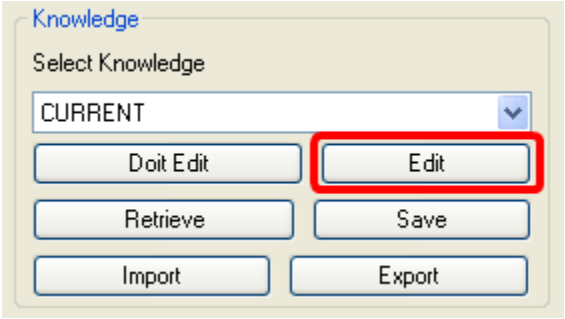
10.5 Edit Knowledge

Edit Knowledge



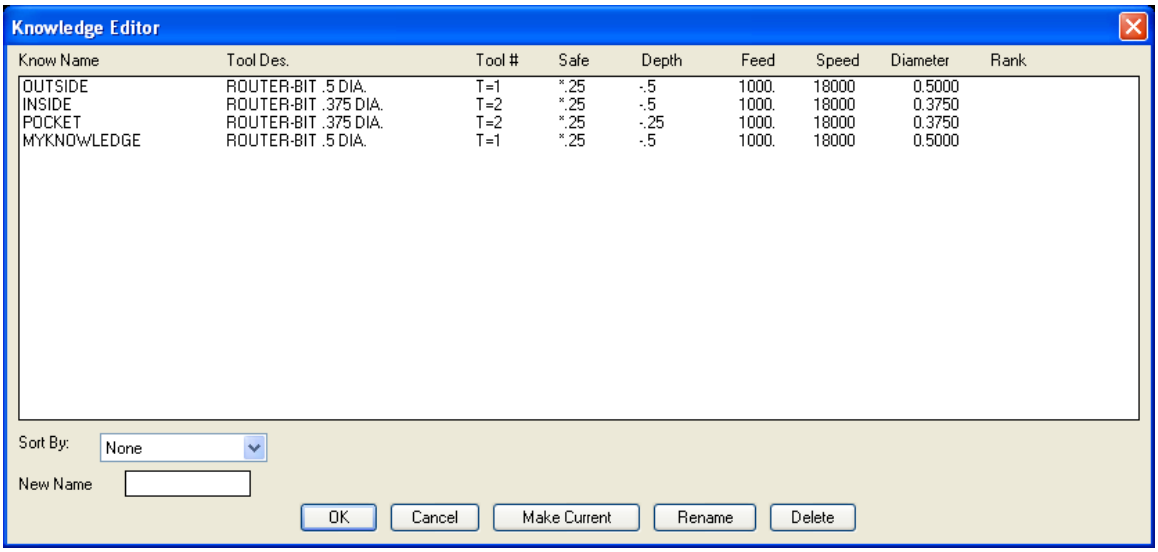
Editing a Selected Knowledge

Selecting this icon from the Router-CIM toolbar will bring you to the interface below. You may also select DOIT Edit from the Control Panel.



This displays the Knowledge Editor.

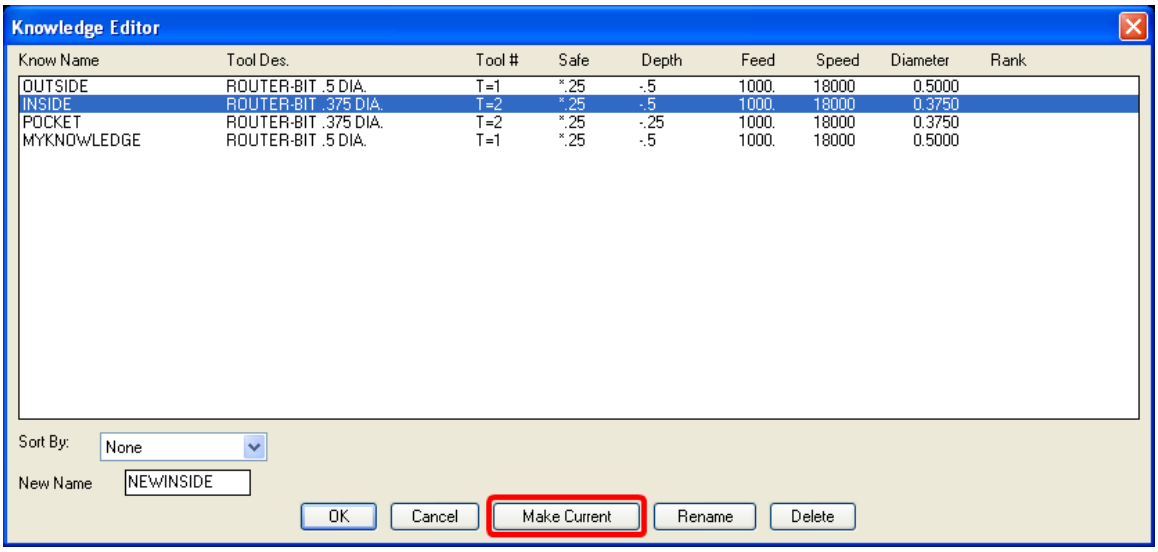
© 2011 CIM-TECH.COM, Inc



Using the Knowledge Editor

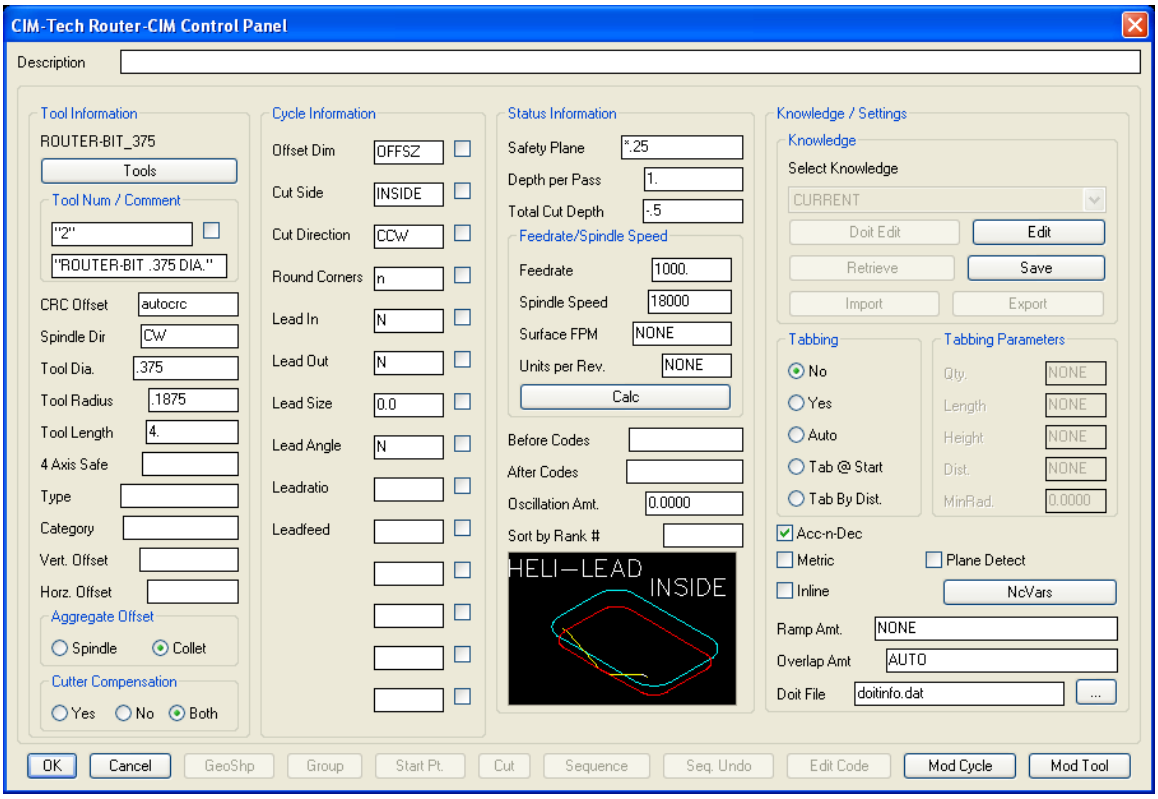
Making a knowledge current

- To make a Knowledge current in the Status Page, select it from the list so that it is highlighted.



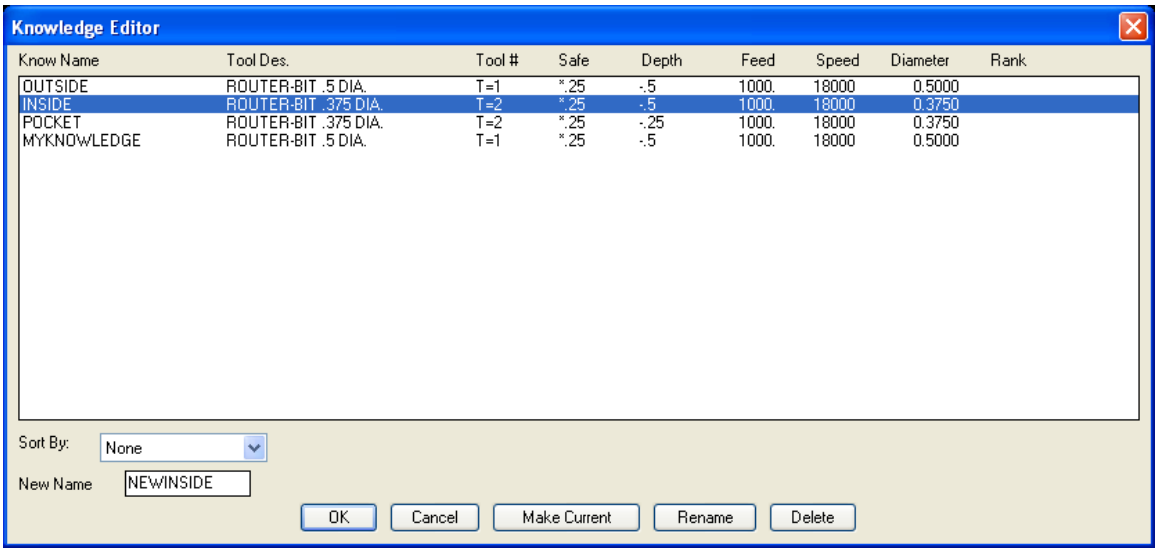
- Then select Make Current, and the knowledge will be shown on the Control Panel.

© 2011 CIM-TECH.COM, Inc



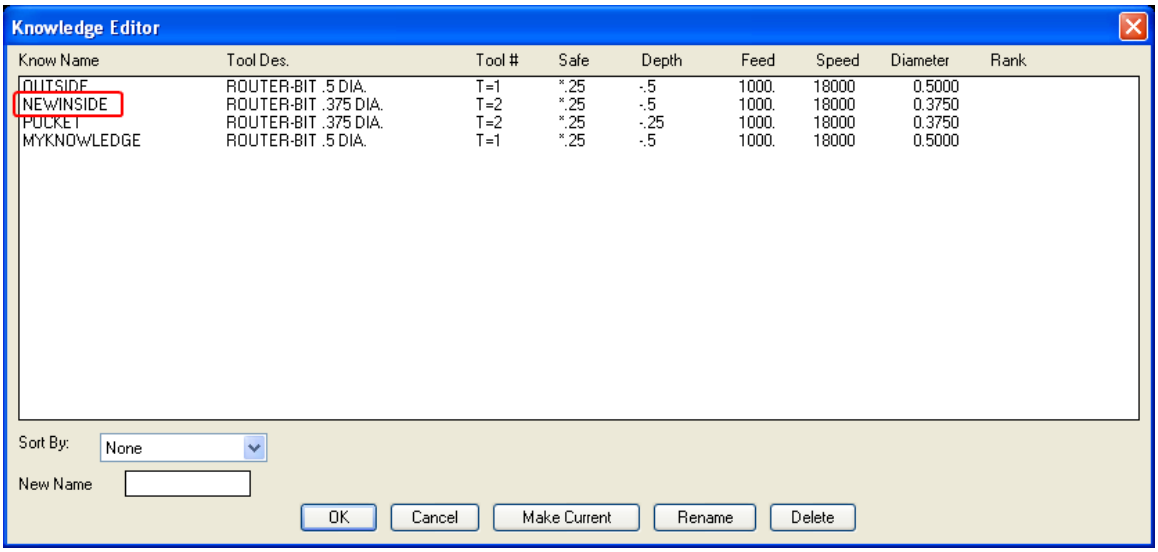
Rename a Knowledge

- To Rename a Knowledge, first select it in the window to make it current, then enter the new name in the appropriate box at the bottom left of the window.



- Click on the Rename button

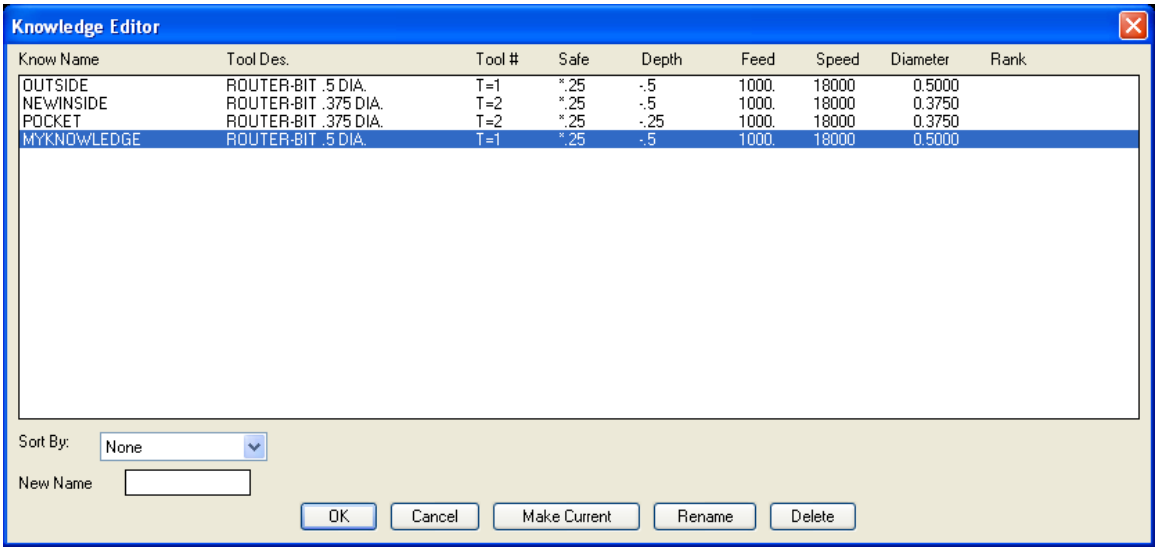
© 2011 CIM-TECH.COM, Inc



- The knowledge now appears in the list under the new name.

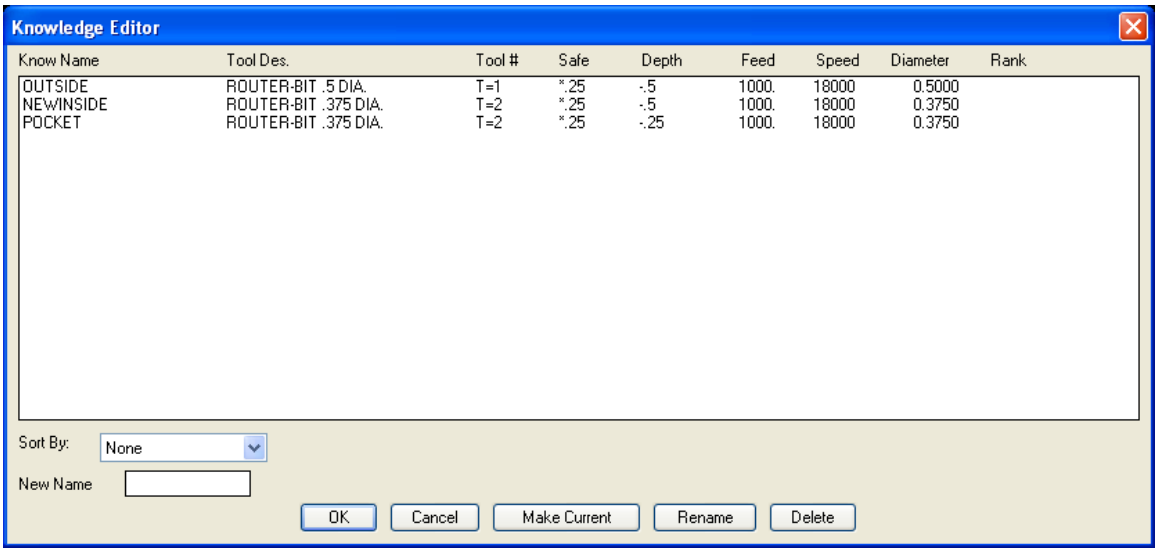
Deleting a Knowledge

- Delete a Knowledge by selecting it in the window to make it current and then click on the Delete button.



- The Knowledge is deleted from the available selections.

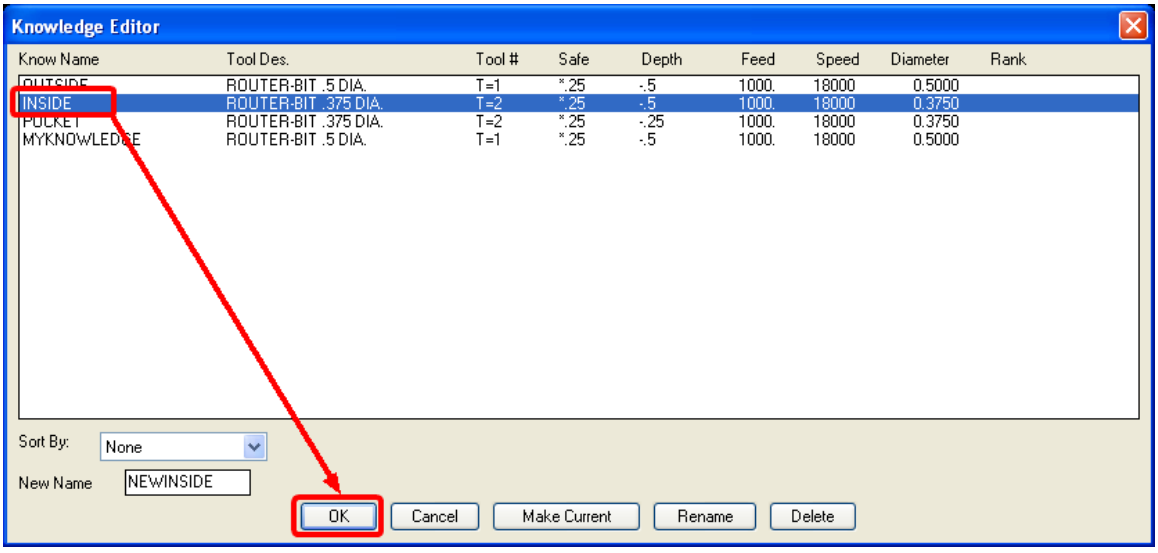
© 2011 CIM-TECH.COM, Inc



Knowledge Quick Edit

The Knowledge Quick Edit will allow you to change certain parameters of the selected knowledge and re save it to the list without changing its name. This has the benefit of allowing you to change items like a tool number or tool, and then be back in the Knowledge Editor with those changes saved.

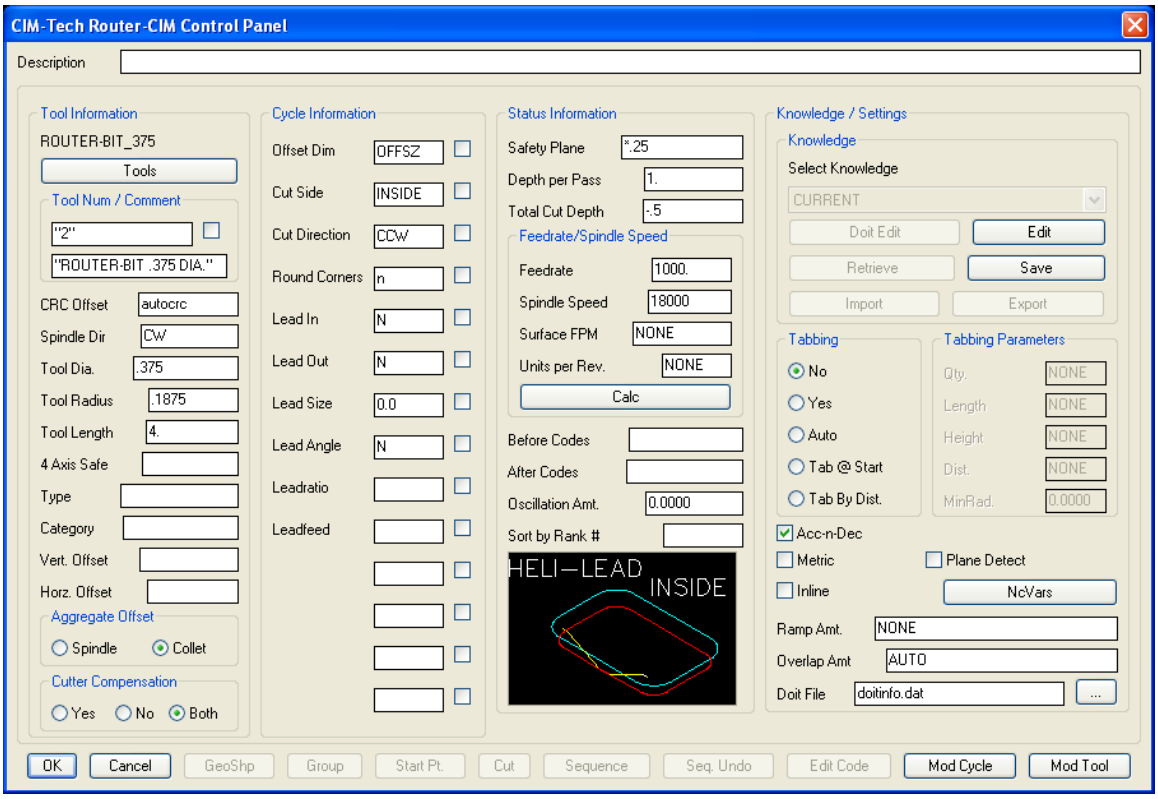
- Select a knowledge from the list so that it is highlighted.



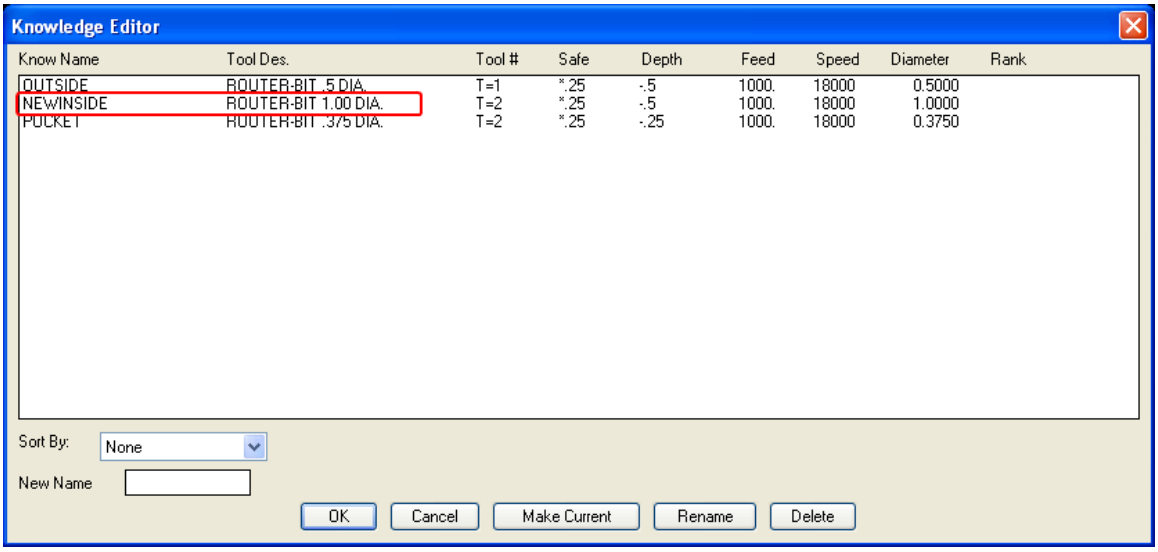
- Now select the OK button.

- The Control Panel is displayed, with only certain items available to you. You can change a tool, tool description, cycle, cycle parameters. You cannot Geoshape, Start Point Edit, even Cut is not allowed.

© 2011 CIM-TECH.COM, Inc



- Change the tool to a 1" end mill and the description, and click on Save in the Knowledge section, and you are returned to the Knowledge Editor with your changes intact.



- When no Knowledges are selected, click on OK to leave the Knowledge Editor.

© 2011 CIM-TECH.COM, Inc

10.6 Group Knowledge

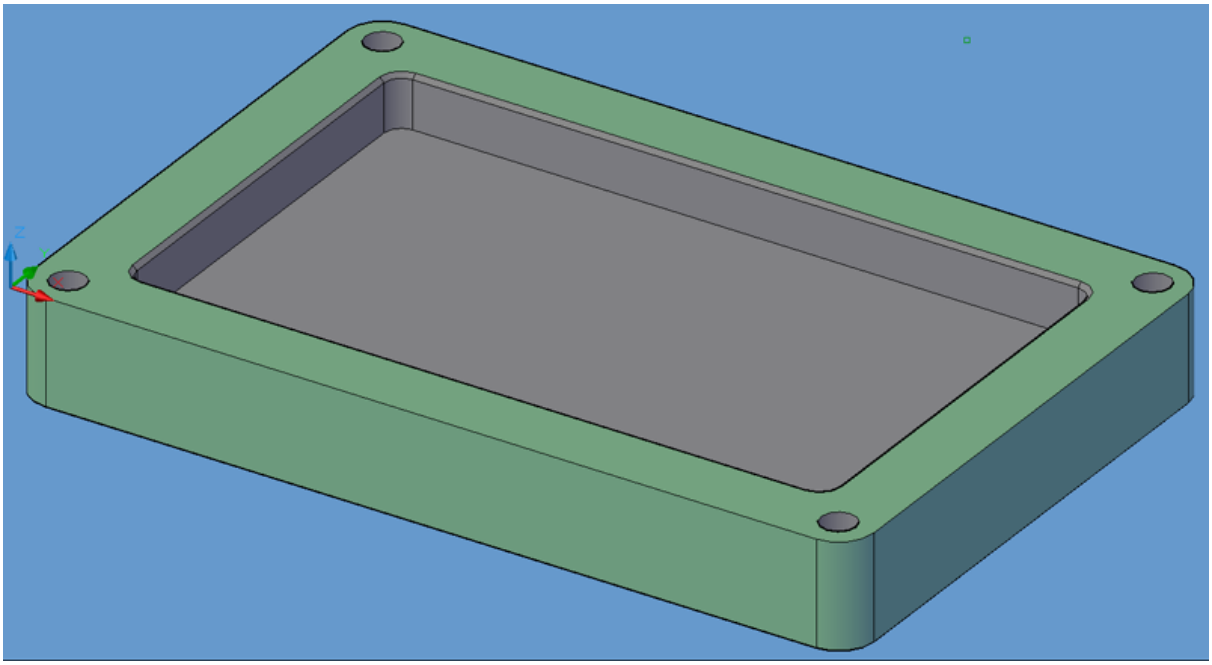
Group Knowledge



The Group Knowledge command will allow you to group several cutting conditions into one knowledge block so that all of the operations occur, in order, on a given shape by using one knowledge selection. If, for instance, you wanted to create a pocket of any given size with a roughing tool, finish with a different tool, then chamfer the edges, this can now be completed with a single selection using Group Knowledge.

You can only select Group Knowledge from the Toolbar, the pull-down menu, or typing "GK" on the keyboard. There is no selection from the Control Panel.

Consider the following part.



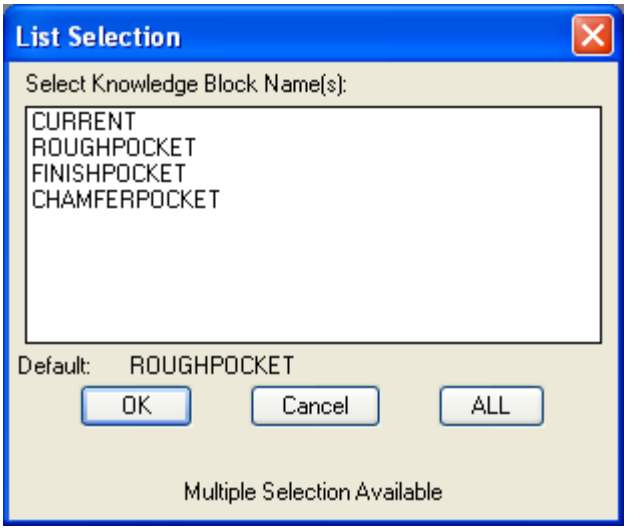
To complete this pocket with the specifications listed above, we need three knowledges. One to rough the pocket, one to finish the pocket, and one to make the chamfer along the top edge of the pocket.

While these could be completed in three operations, if you had several parts to make in this fashion, and they had pockets of varying geometry, then a Group Knowledge is easier to use per part.

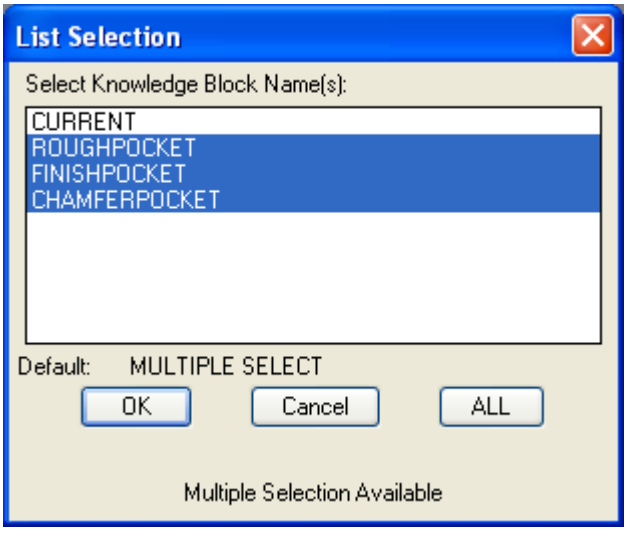
Making a Group Knowledge

© 2011 CIM-TECH.COM, Inc

- Select ***Group Knowledge*** from the toolbar. A window will appear with the current knowledges contained in the current drawing.



- Select the cuts ***IN ORDER*** for the Group Knowledge. This is the order that they will cut and sequence in.



- Select ***OK*** and you will be returned to the CAD screen.

```
Command:
Command:
Knowledge Selection - Examining Existing knowledge...
Enter Knowledge Block Name:
```

- Here you are prompted to give the Group Knowledge a name.

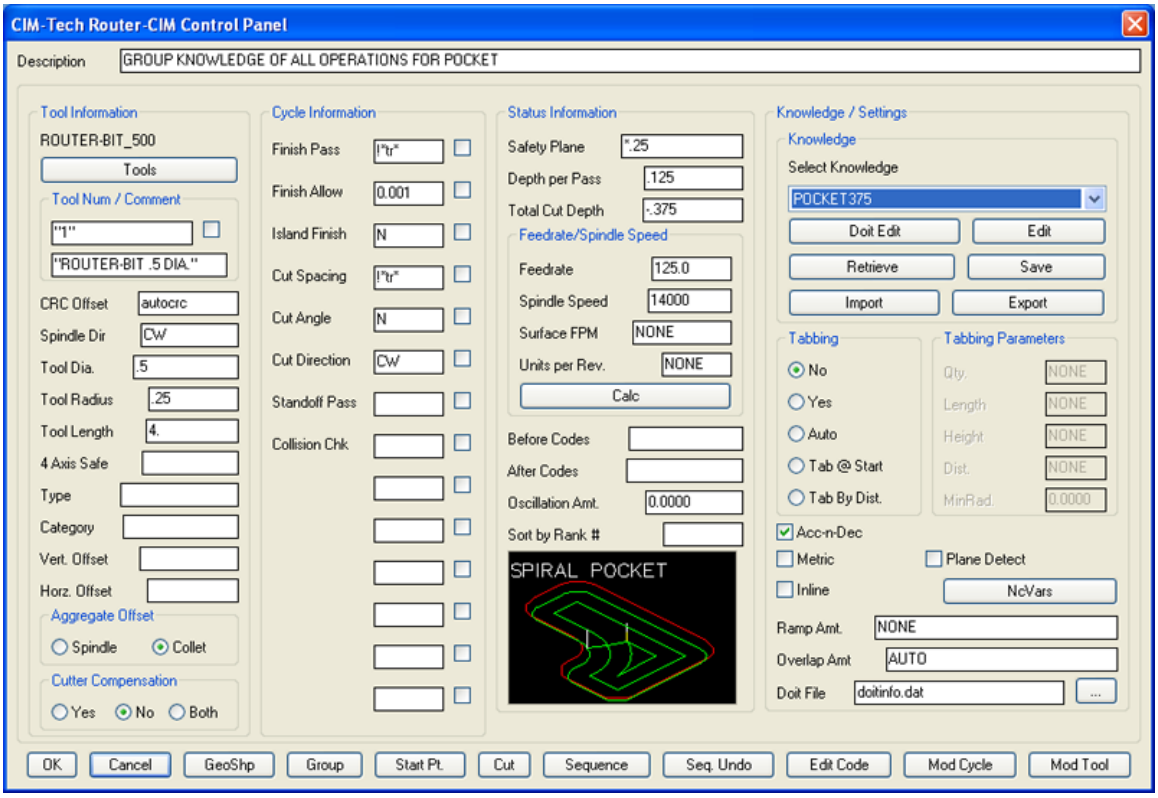
© 2011 CIM-TECH.COM, Inc

```
Command:
Command:
Knowledge Selection - Examining Existing knowledge...
Enter Knowledge Block Name: POCKET375
```

- Type in a name (up to 17 characters) and press ***ENTER***.

```
Enter Knowledge Block Name: POCKET375
Developing Multiple Knowledge Block...
POCKET375 Knowledge Block Stored.
Command:
```

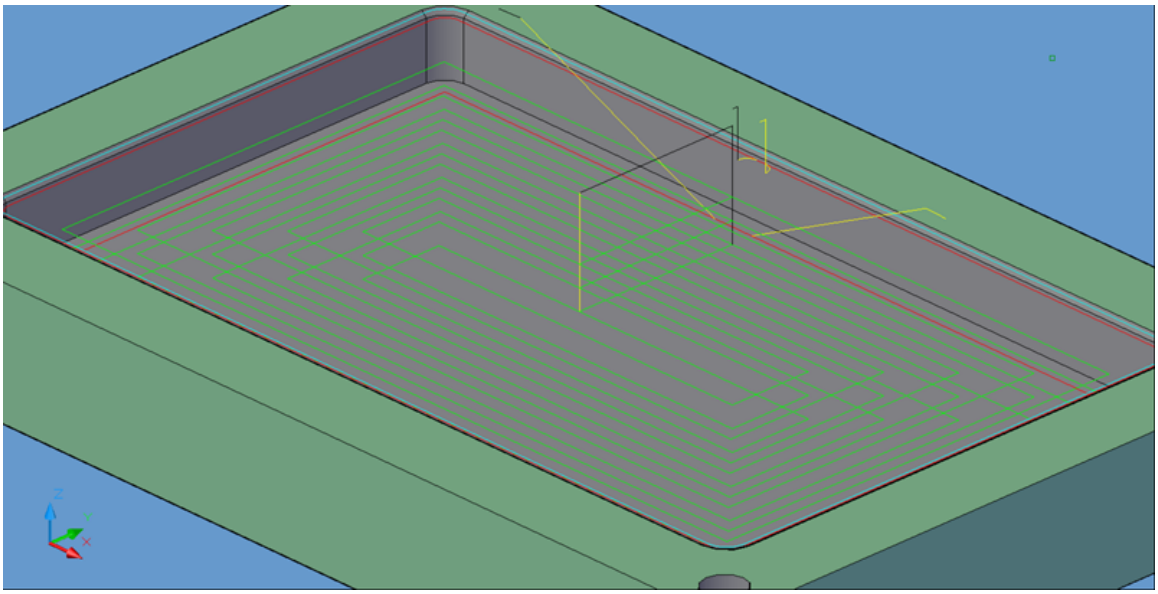
- The named Group Knowledge is stored in the drawing in the knowledge list.



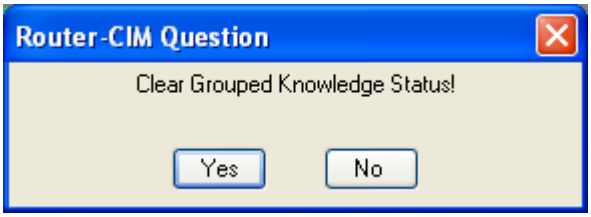
When a Group Knowledge is selected from the list, the first knowledge in the group will appear.

When a shape is selected to cut, all the knowledge operations will be performed in the order in which they are stored in the group. They will be Sequenced in that order as well, unless overridden by another sort key.

© 2011 CIM-TECH.COM, Inc



Then when you come back into the Control Panel, you will be prompted to clear the group knowledge status.



If you have more cuts to make with this same group right away, you can select ***NO***. Otherwise, it is advisable to click on ***Yes***, so that you don't accidentally make more than one cut on the next shape.

NOTES:
You can EXPORT Group Knowledge the same as regular knowledge.

When you use the Retrieve Knowledge command to retrieve the Group Knowledge conditions, a 'STATUS' dialog box will appear for each of the cut conditions contained in the group.

You CANNOT use Group Knowledge with DOIT, there is no way to determine if you want to clear the Group Status before it goes on to the next knowledge.

© 2011 CIM-TECH.COM, Inc

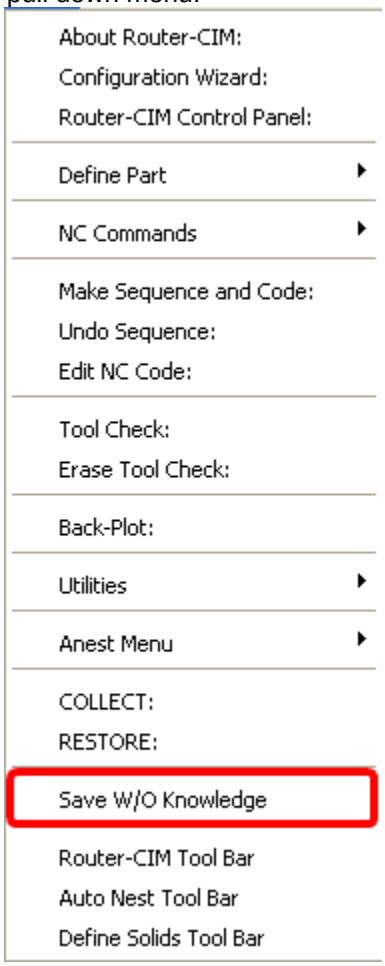
.

10.7 Save Without Knowledge

Save Without Knowledge

This command will save the drawing in the smallest file size. It will also remove machine dependent information from the knowledge in the drawing.

This command is found in the RCIM pull down menu.



Save Without Knowledge removes any unused layers, blocks, text styles, and dimension styles, and then saves the drawing. This is the same as the AutoCAD Purge command. This command will purge the drawing three times. By purging three times, any nested blocks are removed.

Your toolpaths will remain, and the knowledge attached to them will remain. However, if a cut is not on the screen, in use, it will be deleted with the rest of the information.

The benefit of this command is that it remove unused, machine dependent information from the drawing. If you are programming several machines and want to use knowledges between them, this helps keep the file size small, while retaining the cut information, and removes the post-processor specific details.

© 2011 CIM-TECH.COM, Inc

10.8 Knowledge Notes

Knowledge Notes

Use of the Knowledge commands will substantially reduce the time required to program a part(s). Once knowledge is created, it may be use repetitively without having to be re-created for each part or drawing.

Each new drawing becomes a knowledge base once you start Router-CIM in it. If the drawing is saved with cuts in it, it is a knowledge base. The \Router-CIM\NCDWGS\"Machine Name".DWG drawing file is inserted into the current drawing when you load Router-CIM. A knowledge base drawing can contain several hundred blocks. Tools, Cycles, Saved Knowledge, and lead-in/out drawings are among those blocks. Once the part program drawing is complete, the user can strip the extra blocks out of that drawing before saving by using Save Without Knowledge or the AutoCAD Purge command.

Note on Knowledge Base

Users should keep a growing knowledge base on the hard disk. Every time you make a Cut whose cutting condition can be used elsewhere, use the Save Knowledge command and name that condition. This will Save the cutting condition within the drawing. At the end of the drawing session, Export the knowledge(s) to the disk drive.

Important note about Purge

The Purge command will delete any Saved Knowledge that has not been Exported and is not present on the screen when the command is used.
Saved Knowledge is an invisible, not inserted, block that is stored within a drawing. You can see this block listed in the drawing by using the block command and looking for blocks that start with "know_"
The Purge command has to be used three times to completely remove data from the drawing. This strips all blocks that are not inserted, so it is important to Export the Saved Knowledge which writes the knowledge(s) to a file on the hard disk.

11 NC Vars

NC Vars

There are occasions when the "standard" installation of Router-CIM may not be applicable to your machine tool. Usually this can be fixed using the NcVars, which are specific variables that Router-CIM uses to control the program.

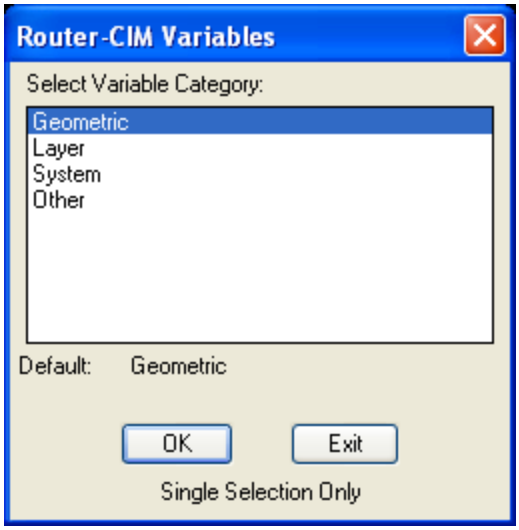
You should only change the NcVars when instructed to either by this manual or after consulting CIM-Tech. An improper setting could cause a failure in Router-CIM that may be hard to track down.

© 2011 CIM-TECH.COM, Inc

11.1 Setting NC Vars

Setting NC Vars

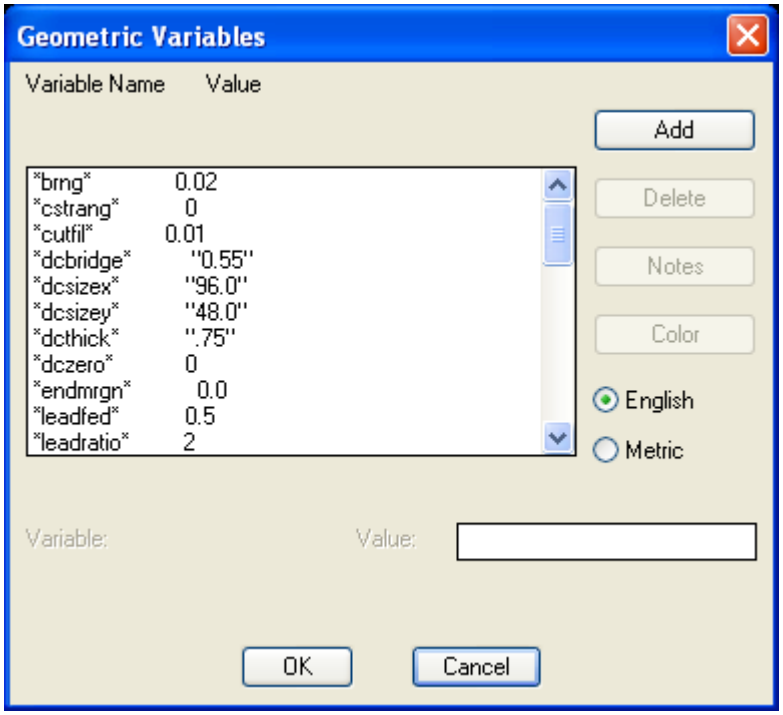
To change an NcVar, type in ***ncvar*** at the command prompt, press ***Enter*** and you will see the screen shown here. There are four types of NcVars.



You must select the type of variable you wish to change and pick ***OK***.

You should know the name of the NcVar you want to change, and the section that the variable is in, so that you lessen the chance of picking the wrong variable and making a mistake.

After selecting ***OK*** you will see the Geometric Variables dialog box.



© 2011 CIM-TECH.COM, Inc

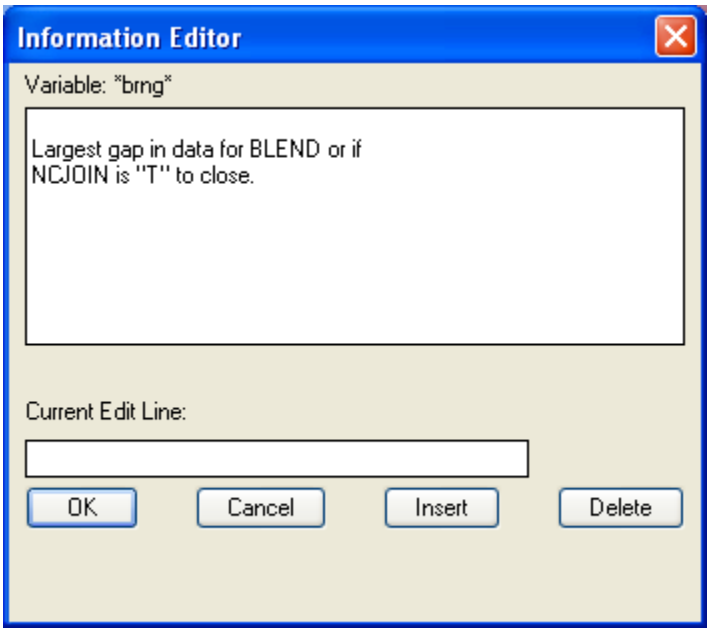
From this screen you can select the name of the NcVar you wish to change. Select the variable and the variable will show up in the Value box and you may input the new value for the variable and then press ***Enter*** to see the change, then click on ***OK***. You will then return to the Router-CIM Variables dialog box where you select ***Exit***, to return to the AutoCAD screen.

Note*
NcVars are extremely syntax sensitive! Enter the value exactly as it is presented, or the result may not be proper.

11.1.1 NC Var Notes

NC Var Notes

In the NcVar Geometric Variable dialog box, there is a button labeled Notes. This will allow you to read or input any notes you may have regarding the variable you wish to change. This can be helpful if you need to change an NcVar to a specific value, and then change it back later. Entering the value, and the reason for the change will make it much easier to change back later. There will usually be notes in the box already.



Click on a ***blank line*** in the Information Editor box, then click on the ***input line*** just below the Current Edit Line, enter the text you wish to input, and then click on ***Insert***. The line will be input.

If there is information you wish to delete, highlight it in the Information Editor box, then click on ***Delete***. The information will be removed.

11.1.2 Adding NC Vars

Adding NC Vars

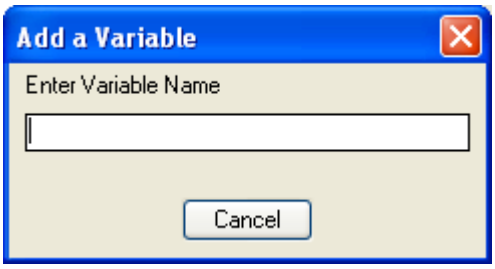
Router-CIM is a very comprehensive product that is continually being updated, and we may occasionally have an update, or an option that you may purchase that will have to be installed. This

© 2011 CIM-TECH.COM, Inc

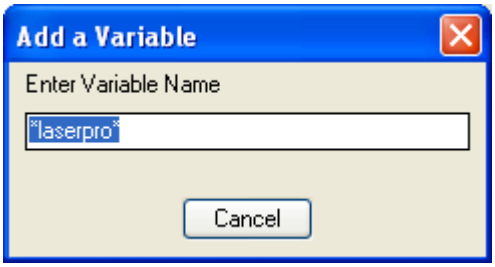
installation may require you to Add an NcVar to your system. The procedure is similar to the other NcVar functions, and can be easily completed by following these instructions. CIM-Tech will advise you of the NcVar you need to add.

To add an NcVar

- Type ***ncvar*** at the command prompt and press ***Enter***.
- Pick on the variable type, in this case System. Then pick ***OK***.
- Pick on ***Add***. An input box will appear.

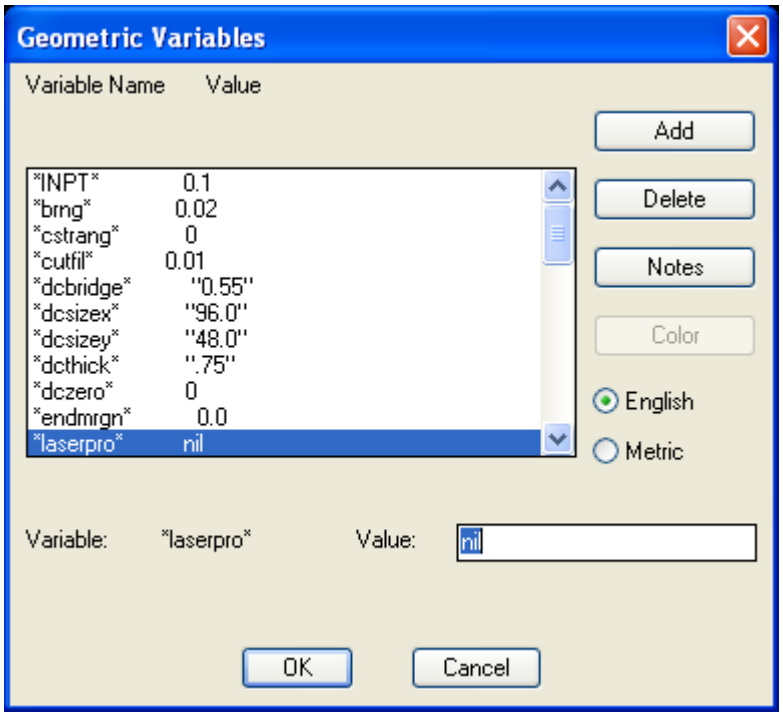


- Enter the variable name ****laserpro**** (or something of your choosing).

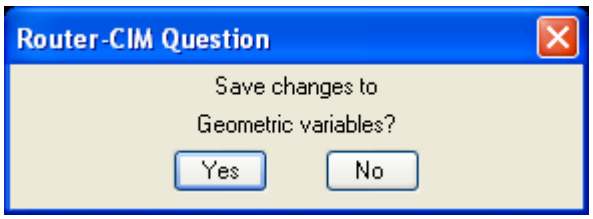


- Press ***Enter***.
- Input the value for the variable (***nil***), in the format necessary and press ***Enter***.

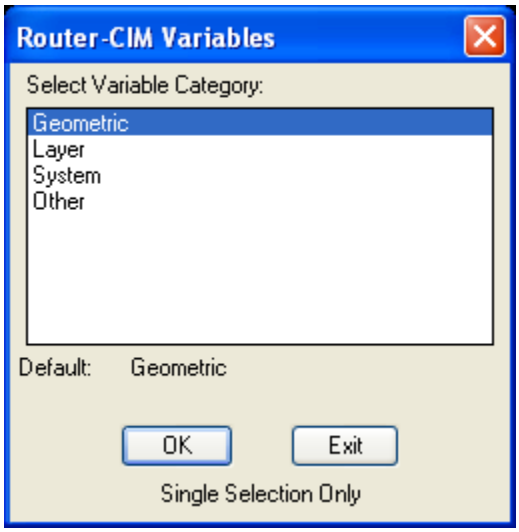
© 2011 CIM-TECH.COM, Inc



- Click on ***OK***.



- You will be prompted to Save changes to System Variables, click on ***Yes***.
- At the NcVar Router-CIM Variables dialog box, click on ***EXIT***.



© 2011 CIM-TECH.COM, Inc

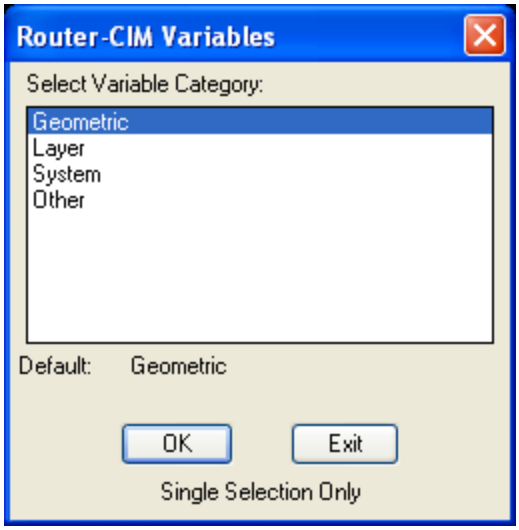
- Restart Router-CIM so the changes may take effect.

These steps are not unusually difficult, however they are fairly exacting. If you have any questions or wish for assistance in changing these variables, please call CIM-Tech for assistance.

11.2 NC Vars Appendix

NC Vars Appendix

There are some standard variables that ship with the program that may see regular use in customizing the system to your needs.
The NC Var will be first and in bold print. It's defintion will be below it and in regular print.



Geometric
Layer
System
Other

11.3 Geometric

GEOMETRIC VARIABLES

brng
Largest gap in data for BLEND if NCJOIN is "T" to close

cstrang
Start angle (expressed in degrees) for circular shapes. 0 starts at 3 o'clock.

© 2011 CIM-TECH.COM, Inc

cutfil
Offset fillet radius for rounded corner filleting in CUT.

endmrgn
End margin on slicer for 3D surface machining

leadfed
This is the percentage of the programmed federate that the lead in will be performed at.

leadratio
A value used in an equation to determine the angle of entry for Helilead cycles.

ncjoin
Use polyline join with *brng* tolerances to close gaps in geometry commonly caused by DXF or IGES transfers.

offvect
NOT USER CONFIGURABLE

pat_fuzz
Value for pattern recognition tolerance amount

r4x_corner_incr
This variable controls the Router-CIM 4th axis interpolation cycle. This value controls how long the segments should be when interpolating around an inside corner.

r4x_corner_offset
This varialble controls the Router-CIM 4th axis interpolation cycle. This is the distance from the inside corner the inside corner cuts will start.

reduceopt
To reduce surface machining tool path to arcs the response is "r" and "s" for no reduction to arcs.

reducezero
Data reduction tolerance of slicer for producing arcs in 3 axis machining.

11.4 Layer

LAYER VARIABLES

Bndclr
Boundary color

Cutclr
Cut line color for tool path other than CW and CCW.

Defclr
Geometry color for NC_GEO.

Inxclr
Index line color.

Inxfr
Index from layer.

© 2011 CIM-TECH.COM, Inc

lnxfrclr
Index from color.

lnxlt
Index line type.

lnxto
Index to layer.

lnxtoclr
Index to color.

Meshclr
Color for offset meshes.

Ncbackplot
Back plot layer.

Ncboundry
Boundary layer.

11.5 System

SYSTEM VARIABLES

3dtab
Turns Tabs into Helical ramps

3dhelix
If True, a polyline is created on layer NC_Helix and turned into a 3D arc in code.
If nil, a polyline is created on layer NC_leads and turned into point to point lines in code.

4aratioshp
Should always be T. Affects how 4 axis cutting treats complex geometry

4xramp
NOT USER CONFIGURABLE

5axis
"T" response will produce augmented data in the toolpath for full 5 axis solutions. This should only be set to "T" when a full 5 axis solution is provided by CIM-TECH.COM, Inc..

jobcomment
Always set to True. This provides the ability for Router-CIM to output comments in the code based on drawing parameters like drawing name, login name, date, etc.

no_fdtm
Eliminates the feed distance to material move which is the extra Z move before a cut starts. FEED DIST TO MATERIAL in the tool edit

sub_text
Text Height for Subroutine Labels.

adspost

© 2011 CIM-TECH.COM, Inc

If True, use the ADS version of the postprocessor.

auto_sub_num
Turns on automatic sub program numbering. When True, sub programs are automatically numbered, starting at 100 and increments by 100 for each additional sub.

cycmode
Canned cycle cutting mode, either 98 or 99. When set to 98, tools retract to Saftey plane, When set to 99, tool retracts to R point set in Router-CIM control panel.

cycname
If True, all cut blocks have a name based on the drawing name. If nil, all cut blocks have a name based on the cycle name.

cyctype
Canned cycle type, on most Fanuc controls, use either 81 (spot drill cycle) or 83 (peck drill cycle).

delgeos
If set to T, the geometry objects used to make shapes are deleted when the shapes are made. If set to NIL (the default), the geometry objects are left on the NC_GEO layer when shapes are made.

doorgrain
Door-CIM VAR

doorplot
Door-CIM VAR

editlist
Name of the text editor for List Edit. For R10 and R11 users only.

heli_rotate
If set to a non zero value, the lead in and lead out of heli cycles will be rotated away from the part by the specified amount. The leads will only rotate when a single pass cut is made.

knwedit_ncsurfer
Use NC Surfer when you cut a Nurbs cycle. Should be T to use the NCSurfer and *nil* if you do not want to use the NCSurfer.

linesort
Only use lines for open shape group sorting.

ncfilext
Extension to use on the output file.
Valid input is a string (value in quotes) up to 3 characters.

ncfilloc
The path to use when creating the NC Code file. If blank, uses the current drive and directory.

ncfont
Text font for NC text.

ncoffset
NIL = uses AutoCAD offset, T = uses Router-CIM offset

ncorigin
If True, Router-CIM will require a placement of an origin symbol and axis description during a CUT cycle. Mainly used for Nesting applications.

© 2011 CIM-TECH.COM, Inc

noshpchk
If set to T, no checks are made for duplicate definitions of selected objects. If set to NIL (the default), checks are made during the define process to see if the selected objects have already been defined. If so, messages instruct you exit or redefine the objects.

nsee
If T, enables the prompts for the Tape-to-Part extension. If you do not have NSEE or WNSEE, leave this nil.

offbias
This number is added to the offset value in Router-CIM. Sometimes this bias is required to created a specific offset.

11.6 Other

OTHER VARIABLES

TS_NO_NBL
NOT USER CONFIGURABLE

dwfmake
This variable will create a dwf image of a part created by the Parametric Macro Builder.Setting it to T will cause the drawing to zoom to extents and a print window will flash. Automation can then preview this part. Set to nil if Automation is not going to be used.

emul_mode
NOT USER CONFIGURABLE

emul_tool_percent
NOT USER CONFIGURABLE

geo_color
NOT USER CONFIGURABLE

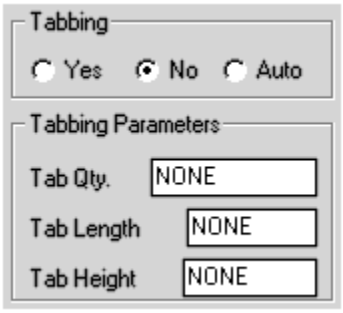
nc_lines
NOT USER CONFIGURABLE

nestvec
Scale factor for nest shape definition. Set to 1 for normal operation.

12 Tabbing

Tabbing

© 2011 CIM-TECH.COM, Inc



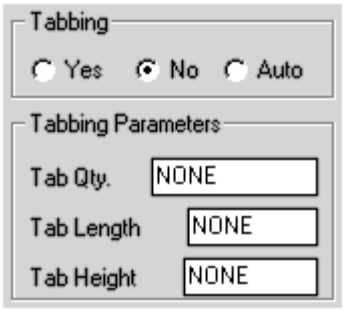
Tabbing is a technology that allows smaller or harder to hold parts stay together by allowing small pieces of material to remain holding the part to the sheet it is being cut from.

There are only a few tabbing settings, and these are easily described in the following section.

Tabbing Settings

12.1 Tabbing Settings

Tabbing Settings



Tabbing

Tabbing will lift the cutter slightly at certain places-selected by you or selected automatically-to leave parts connected to the sheet. Tabbing will allow you to set the Tabs for your part based on the parameters you set in the Control Panel.

New enhanced Tabbing will now allow Tabbing across even the most complex geometry-even when a single Tab may cross multiple elements. See options below:

YES to Tabbing

This will set up Tabbing to run and will prompt you for the locations for the Tabs before the Cut is finished. You may pick as many locations as you need for the Tabs, but be sure to pick on the tool path for the Tab location. If you select the geometry for a Tab location, you will not get a Tab placed in that spot.

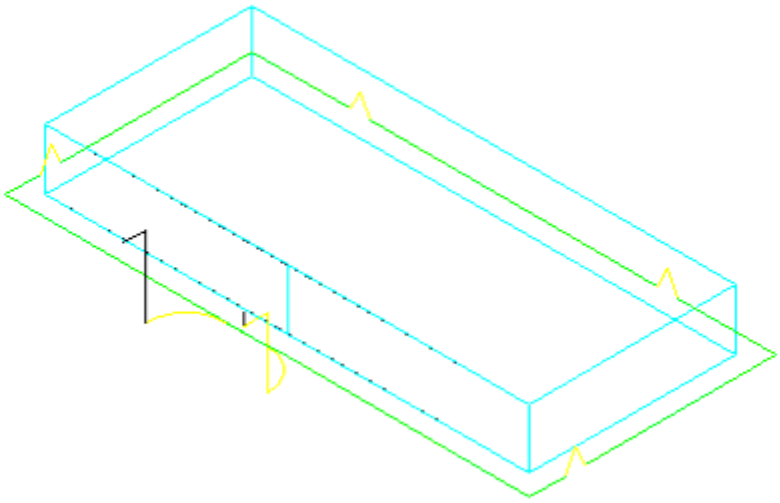
NO to Tabbing

If you select NO, then no Tabbing operations will occur even though you may have a Tabbing Length and a Tabbing Height set.

© 2011 CIM-TECH.COM, Inc

AUTOMATIC Tabbing

When Automatic is set, you will be prompted for the number of Tabs you wish to have on the part, and Router-CIM will divide the number of Tabs across the part evenly spaced.

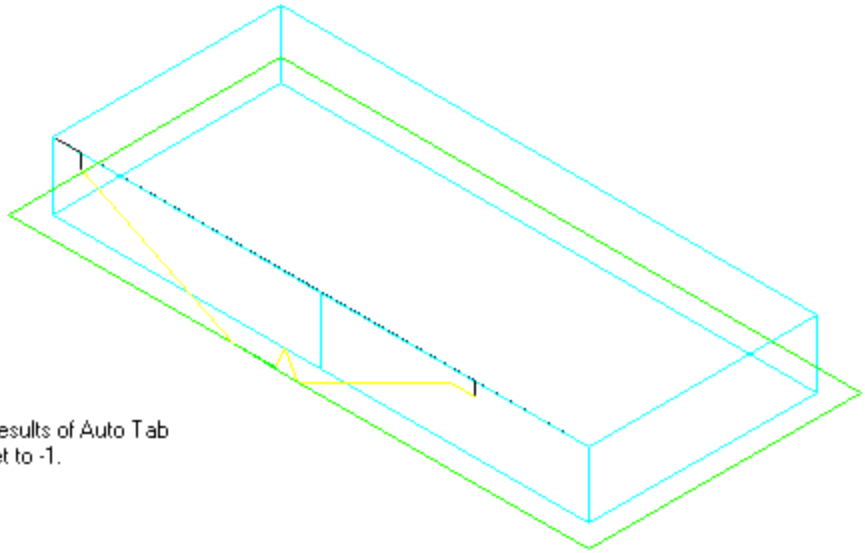


Auto Tab Qty

When Automatic is set, the number you put in this field will be the number of Tabs that Router-CIM will divide up evenly spaced across the part, based upon the length entered in Tabbing Length. If Auto Tab Qty is not filled in, the system will prompt the user at the Command line for "Quantity of Tabs?".

*****NOTE****:

Auto tab qty. set at -1 will produce a Tab immediately after the lead-in. The overlap will automatically cut the Tab off at the end of the cut. This will hold the part at the cut start for the duration of the cut.



Results of Auto Tab set to -1.

Tabbing Length

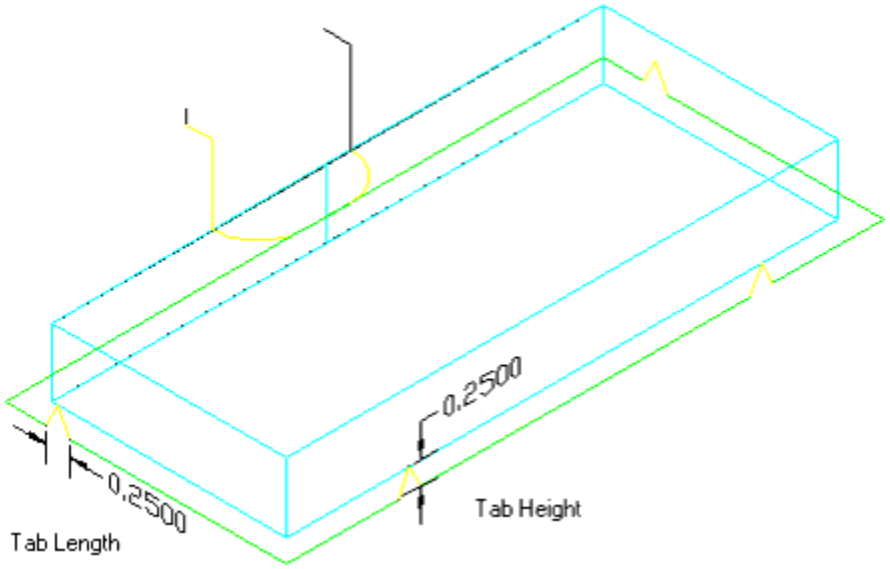
This sets the Length of the Tab, in units. If you leave this field blank, but select either Yes or Automatic to Tabbing, then you will be prompted during the Cut for the correct Tab Length. When

© 2011 CIM-TECH.COM, Inc

setting the Tab Length, you should try to make it at least twice the size of the cutter diameter.

NOTE*
Any Tab Length not at least the size(diameter) of the cutter will not leave any tabs after the part is cut.

Tabbing Height
This sets the Height of the Tab, in units. If you leave this field blank, but select either Yes, or Automatic to Tabbing, then you will be prompted during the Cut for the correct Tab Height.



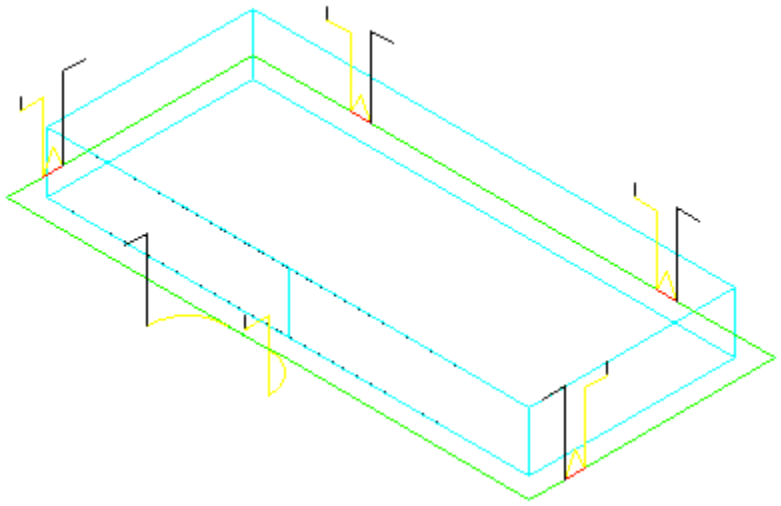
Tab Removal
After your parts are initially cut:
Bring up Control Panel/All Stats page

- Set Tabbing from Auto or Yes to ***No***.
- Choose *Center Line Cut* as the Cutting Cycle.



- Click ***OK*** at the bottom left of the Control Panel/All Stats page.
- Type tabr at the command line, hit ***Enter***.

© 2011 CIM-TECH.COM, Inc

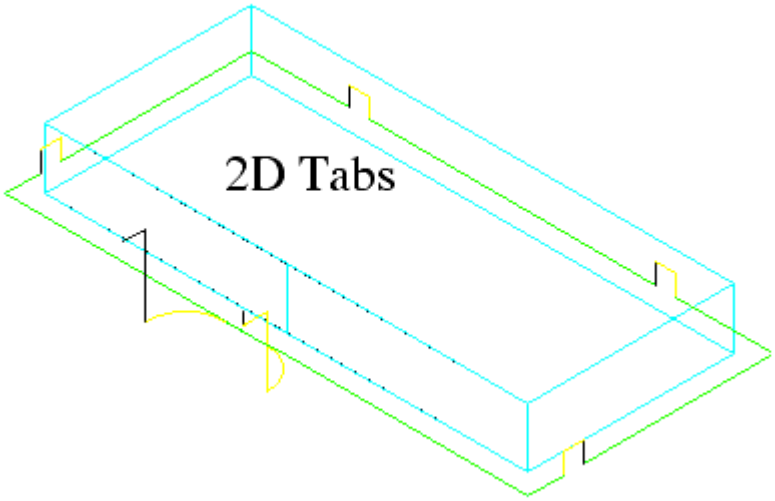


Ramped Tabs

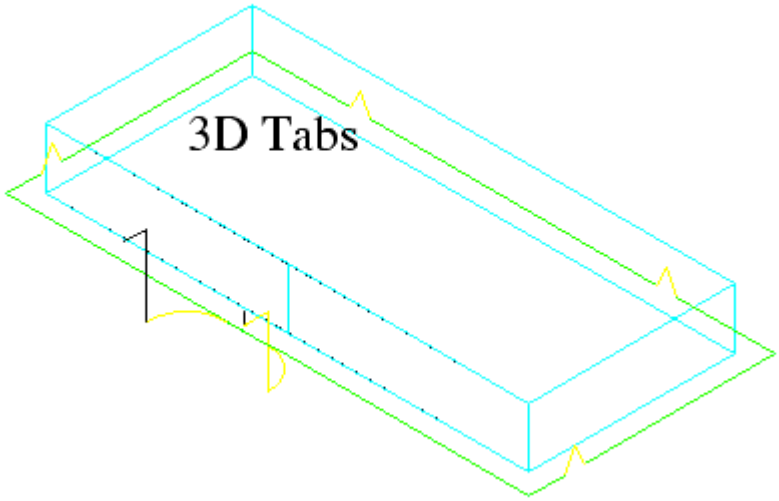
Router-CIM can make 3-D tabs that ramp up and ramp down. This is useful for modern machines that use high speed high precision machining.

Tabbing is either 2D or 3D based on the System NcVar:

3D TAB. If this variable is set to T, 3D tabs are produced, if set to nil, 2D tabs are produced.



© 2011 CIM-TECH.COM, Inc



12.2 2D or 3D Tabs

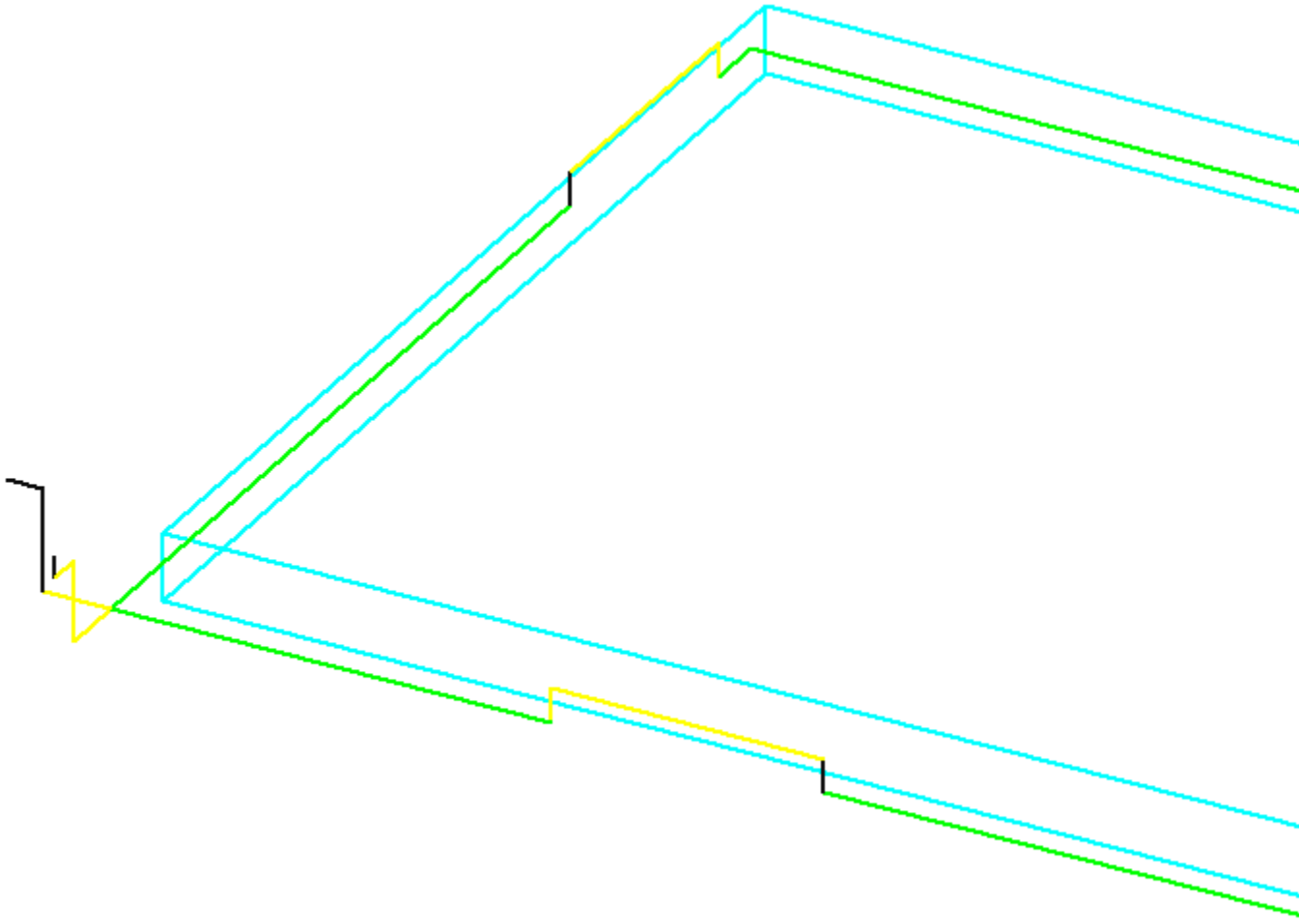
2D or 3D Tabs

There are two methods of tabbing in Router-CIM. You can use either 2D or 3D tabs. The differences are described

2D Tabs

These tabs are very square, and cause the cutter to stop and lift up in Z to the height of the tab, then move over by t
depth.

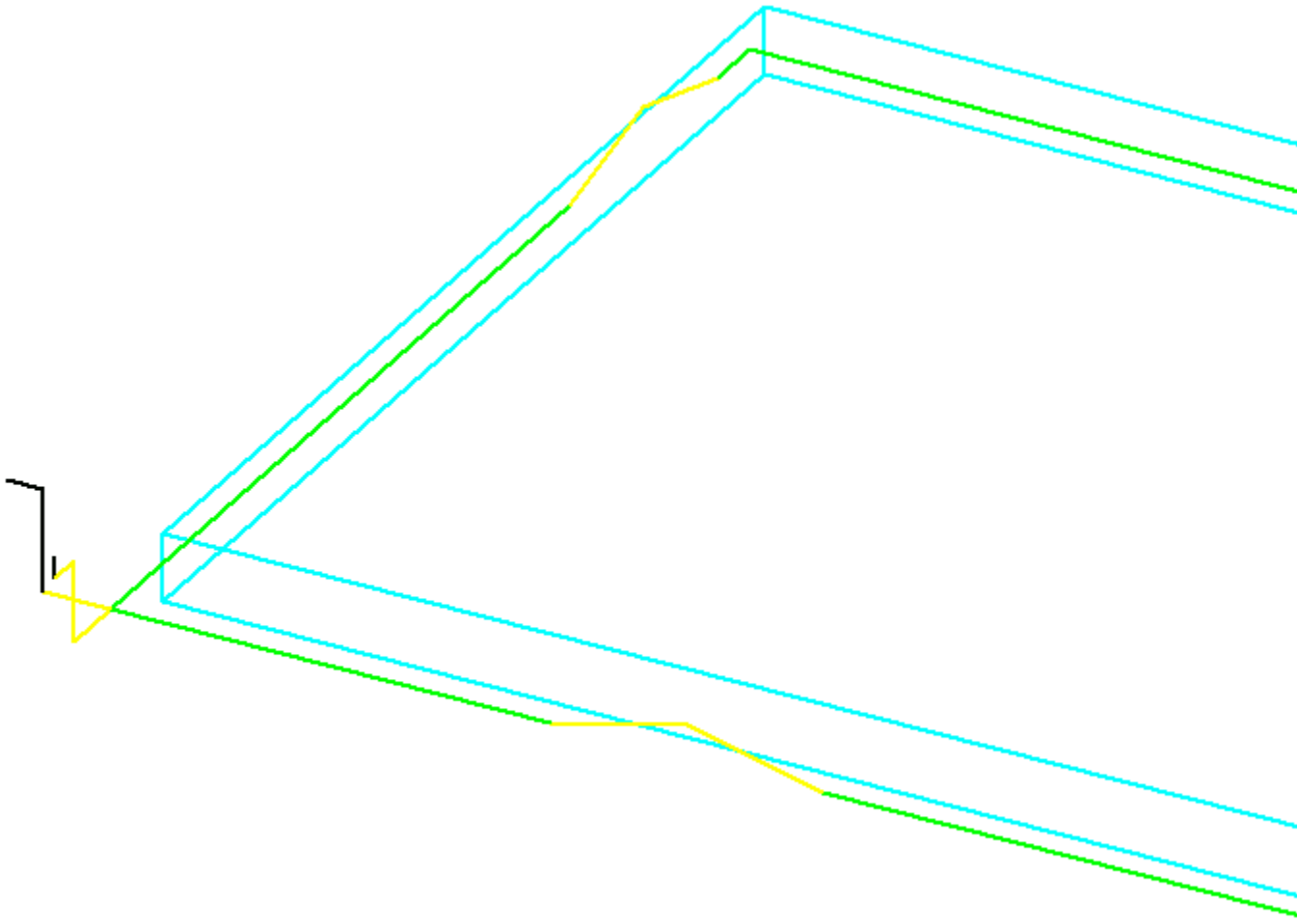
© 2011 CIM-TECH.COM, Inc



3D Tabs

3D tabs are usually faster, but leave a varied amount of material. The tool lifts up as it is cutting to the tab height in
stop and lift up in Z then move over and then move back down in Z in 3 separate moves, the machine will cut these

© 2011 CIM-TECH.COM, Inc



Changing from 3D tabs to 2D tabs.

3D tabs are the default in Router-CIM. You can change from 2D tabs to 3D tabs by changing the NCVAR *3DTAB*

3DTAB set to T will give you 3D tabs
3DTAB set to nil will give you 2D tabs.

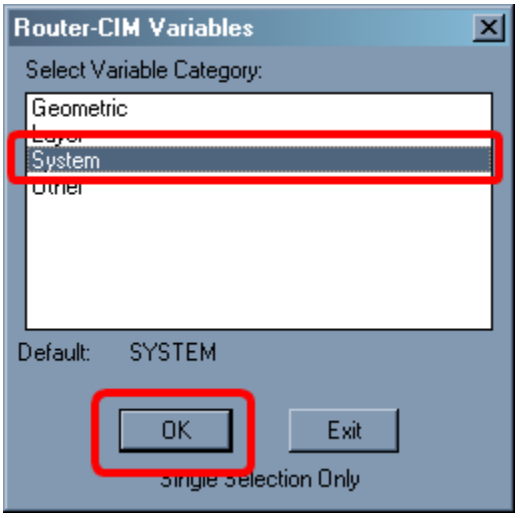
To change an NCVAR, you can select it either from the Control Panel or from the keyboard.

Using the keyboard to change NCVARs

 NCVAR

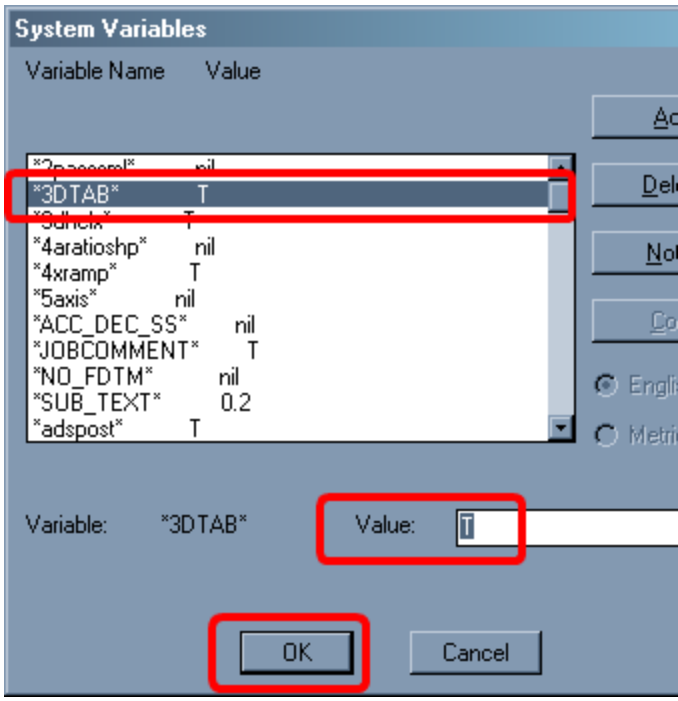
Type NCVAR and press Enter on the keyboard. You will see the following screen appear:

© 2011 CIM-TECH.COM, Inc



Select System from the choices listed and then select OK.

You will then be shown a list of NCVARS that you can change. The variable that you want is *3DTAB*

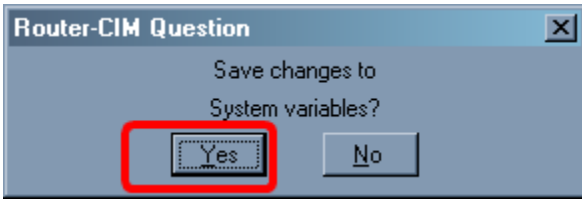


Once you select *3DTAB* you can change it to either nil or T.

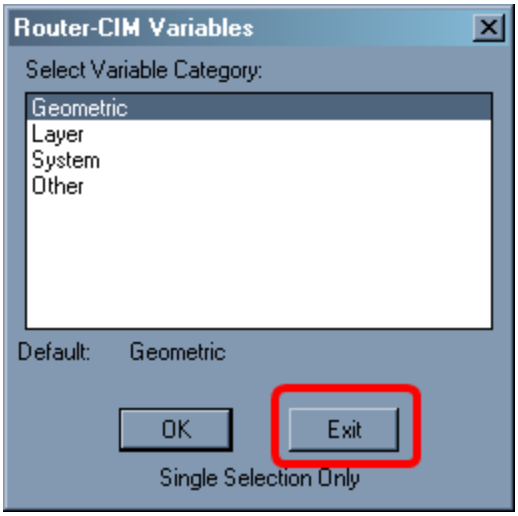
nil = 2DTAB
T = 3DTAB

Once you have changed the value, be sure to press ENTER on the keyboard. Then select OK to go back to the ma

© 2011 CIM-TECH.COM, Inc



This window will ask you if you want to save the changes, select Yes.



Then, at the main window, select Exit.

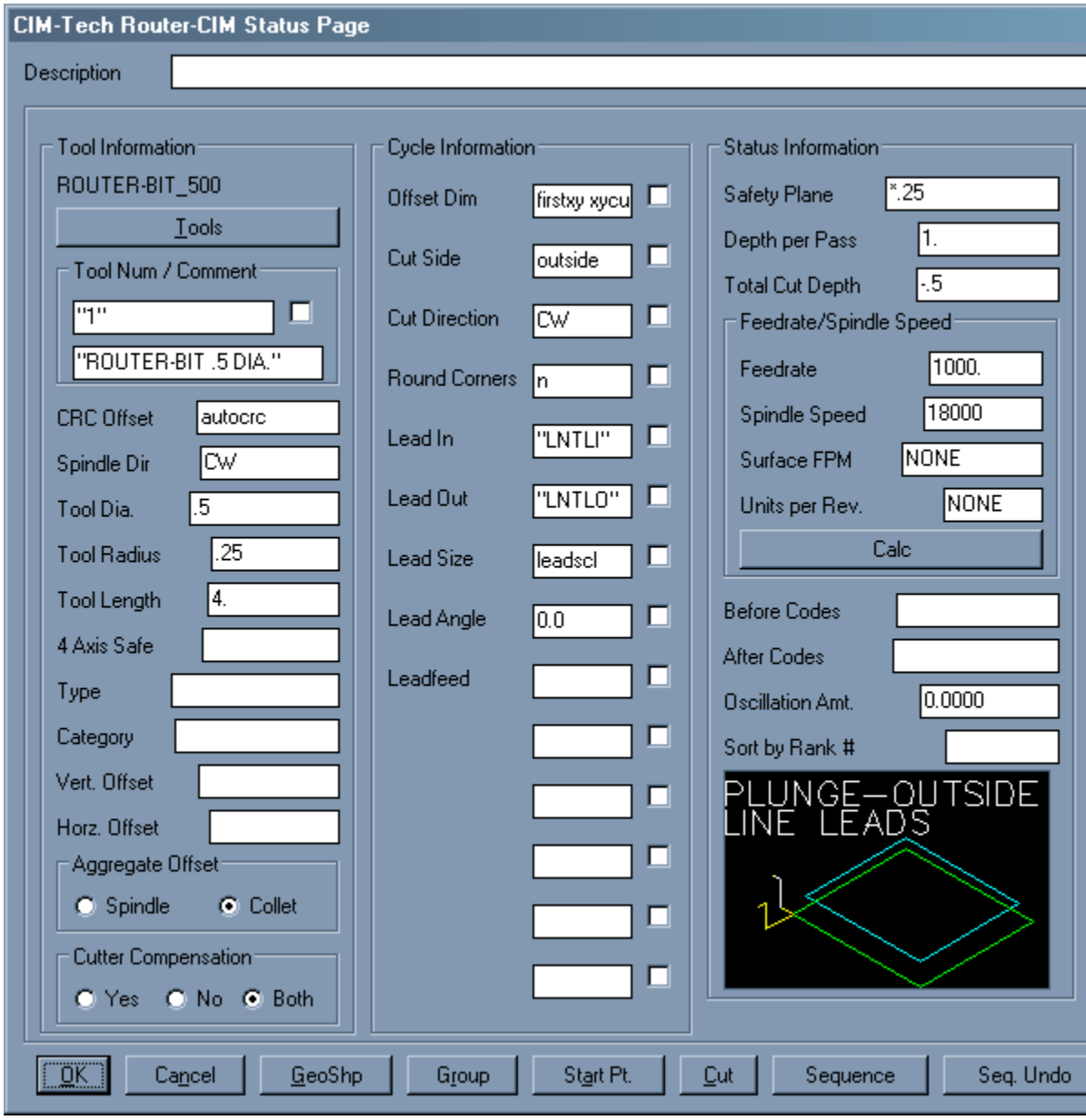
Using the Control Panel to change NCVARs



Select the Control Panel Icon from the Router-CIM toolbar.

The control panel will appear.

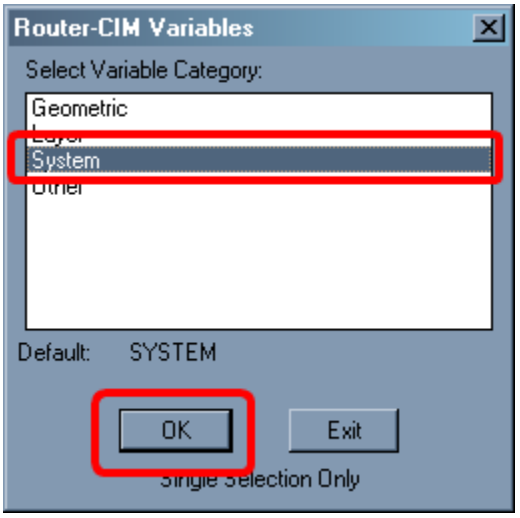
© 2011 CIM-TECH.COM, Inc



From the Control Panel select the button that says **NcVars**.

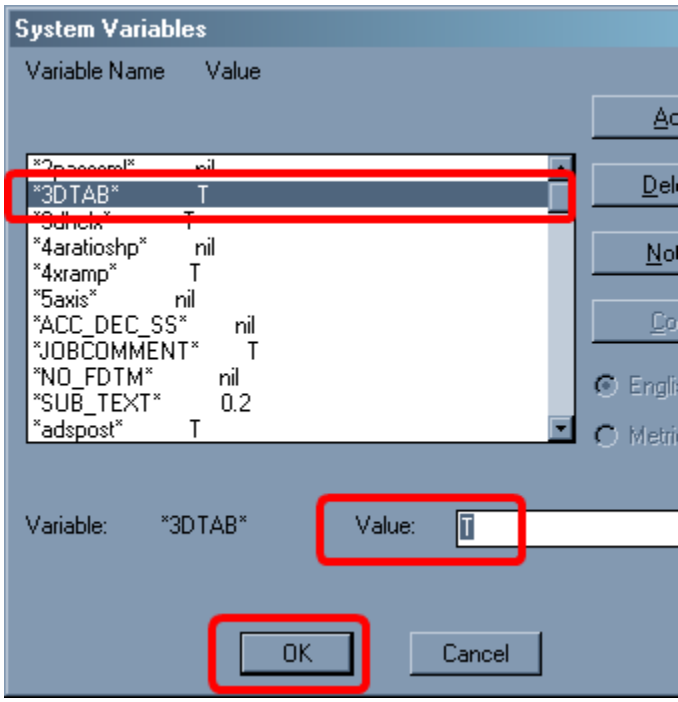
You will see the following screen appear:

© 2011 CIM-TECH.COM, Inc



Select System from the choices listed and then select **OK**.

You will then be shown a list of NCVARS that you can change. The variable that you want is ***3DTAB***

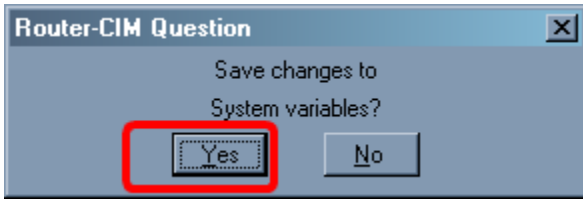


Once you select *3DTAB* you can change it to either nil or T.

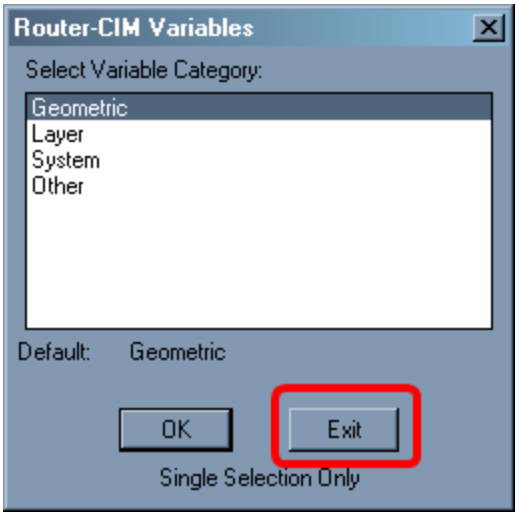
nil = 2DTAB
T = 3DTAB

Once you have changed the value, be sure to press ENTER on the keyboard. Then select OK to go back to the ma

© 2011 CIM-TECH.COM, Inc



This window will ask you if you want to save the changes, select Yes.



Then, at the main window, select Exit.

13 Cutting Solids in Router-CIM

Solids



The following is taken from the AutoCAD manual to give unfamiliar users some background on Solids.

A solid object represents the entire volume of an object. Solids are the most informational, complete and least ambiguous of the 3D modeling types. Complex solid shapes are also easier to construct and edit than wire frames and meshes.

You create solids from one of the basic solid shapes of box, cone, cylinder, sphere, torus, and wedge or by extruding a 2D object along a path or revolving a 2D object about an axis.

Once you have created a solid in this manner, you can create more complex shapes by combining solids. You can join solids, subtract solids from each other, or find the common

© 2011 CIM-TECH.COM, Inc

volume (overlapping portion) of solids.
Solids can be further modified by filleting, chamfering, or changing the color of their edges. Faces on solids are easily manipulated because they don't require you to draw any new geometry or perform Boolean operations on the solid. AutoCAD also provides commands for slicing a solid into two pieces or obtaining the 2D cross section of a solid.
Like meshes, solids are displayed as wire frames until you hide, shade, or render them. Additionally, you can analyze solids for their mass properties (volume, moments of inertia, center of gravity, and so on). You can export data about a solid object to applications such as NC (numerical control) milling or FEM (finite element method) analysis. By exploding a solid, you can break it down to mesh and wire frame objects.

 - **NCExtrude**

 - **NCExtract**

 - **NCDepth**

 - **NCUpdate**

 - **NCFlat**

 - **Shapehole**

You can cut AutoCAD or Mechanical Desktop solid models only if they have been passed through NCExtrude or NCExtract. Do Not Use Geoshape On A Solid!

13.1 NCExtrude

NCEXTRUDE

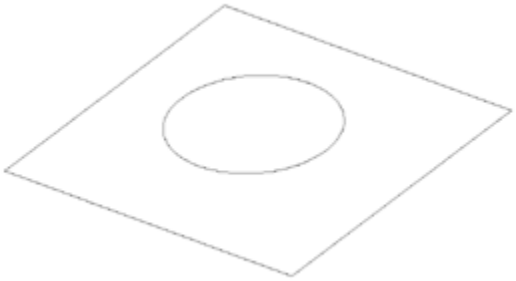


NCEXTRUDE converts a 3DSOLID entity into 2D Polylines that represent the extruded profiles. The thickness of the developed Polylines represent the depth of the profile in the 3D solid. The Polylines are placed on the NC_SHAPE layer and are immediately available for cutting.

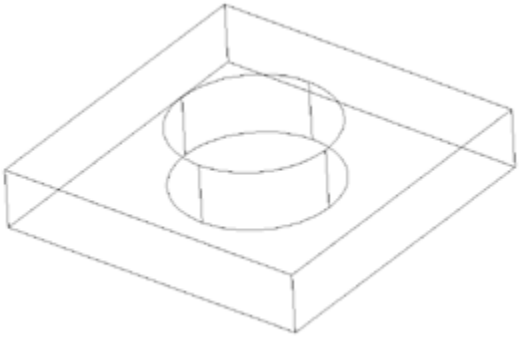
All bodies that are in the 3DSOLID that are not extrusions without taper are not converted. 3DSOLID entities that cannot be exploded are not converted.

Example: This will show a simple part being extruded to a 3d Solid and then NCExtrude will sperate the shapes into 2D Polylines with thickness suitable for cutting.

© 2011 CIM-TECH.COM, Inc



Step1. A simple 2d shape created (shown in an isometric view).



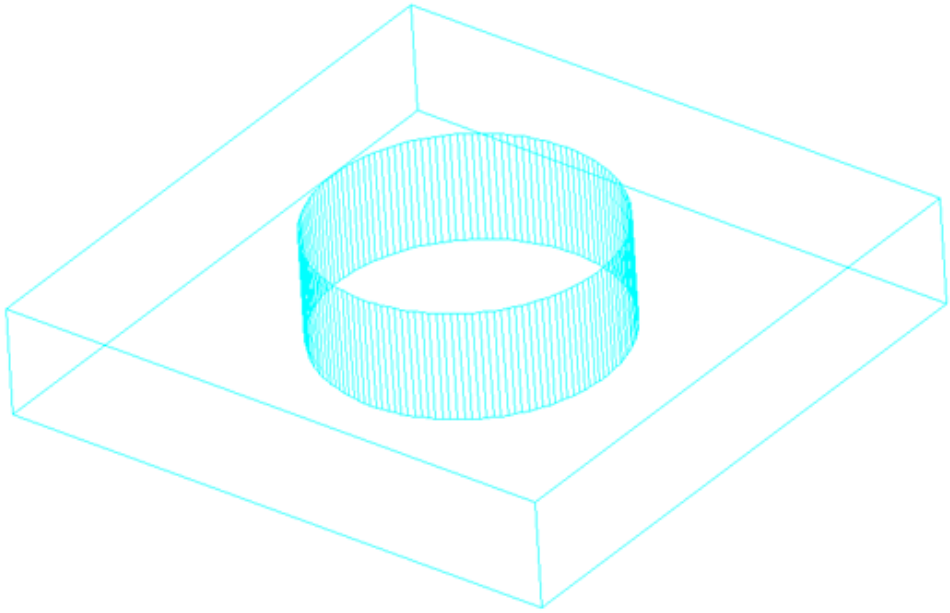
Step2. That same shape extruded and the center shape subtracted from the rectangular shape. This is now 1 solid model.

Step3. The NCEXTRUDE command is run and the following prompts appear:

Initializing...Select Extruded Solid to Define: ***Here you select the solid you wish to have extracted***
Examining Selected Solid...
Defining Regions...
Defining Shapes...
Defining Shape(s)...
Defining Shape(s)...

The resulting shape looks like this:

© 2011 CIM-TECH.COM, Inc

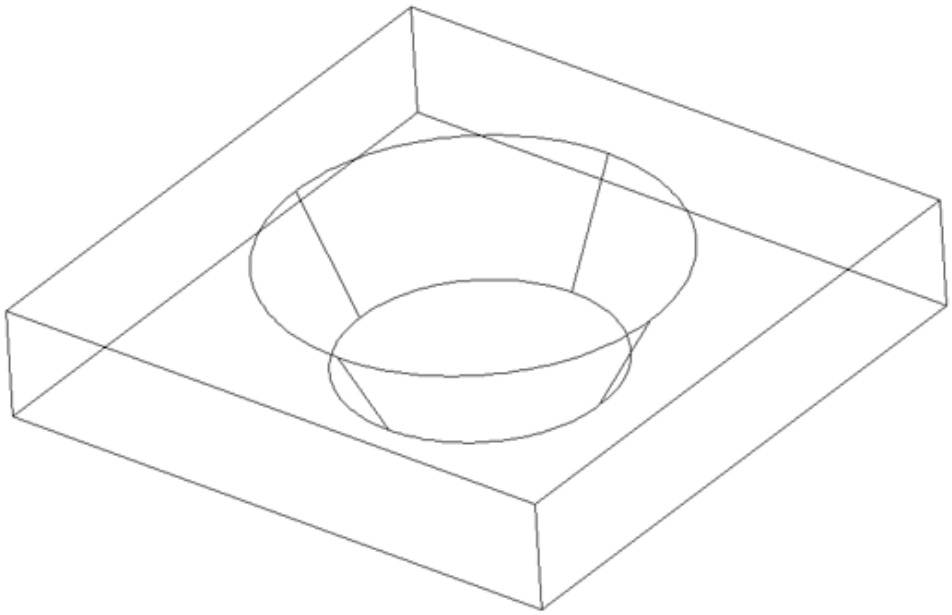


These resulting shapes are now reading for cutting in Router-CIM. If you wish to cut them with a layer to knowledge association, they resulting geometry shown here will have to be placed on the layers you wish to use for the layer to knowledge association. NCExtrude will only place the geometry on layer NC_Shape in preparation for cutting.

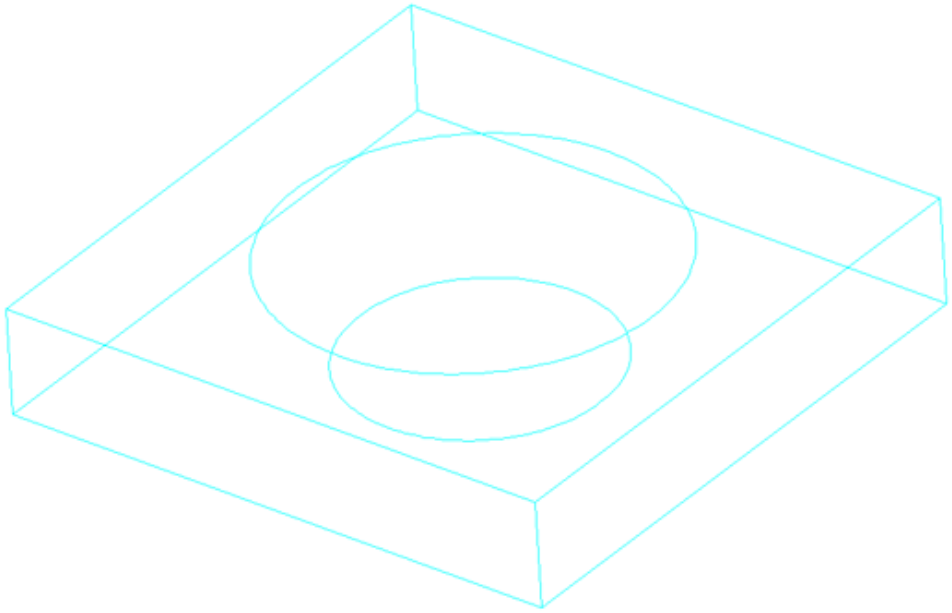
Objects with Taper

Only shapes that have been extruded and not tapered will have thickness assigned to the resulting 2D polyline created with the NCExtrude command.
If an object has a taper to it, NCExtrude will provide a 2D shape on the top and the bottom of the shape. Both will be on layer NC_Shape and suitable for cutting, but neither will have thickness.

© 2011 CIM-TECH.COM, Inc



This center shape clearly has a taper to it.



The resulting shape from NCExtrude is a 2D profile with thickness on the outside shape and two 2D profiles on the inside shape. One at the top of the shape and one at the bottom.

© 2011 CIM-TECH.COM, Inc

13.2 NCExtract

NCEXTRACT

Can be used on an AutoCAD or Mechanical Desktop Solid. You get a Geoshaped polyline from a planar face.

The NCEXTRACT command creates Shape Polylines from Solid models. NCEXTRACT works on core solids and parametric solid models.

Using NCEXTRACT:

From the RCIM pull down menu, pick Define Part >> Define Solids >> NCextract.

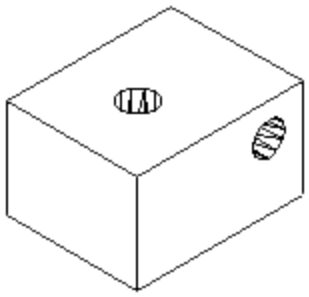
The NCEXTRACT command will extract a Geoshaped polyline from the face of an AutoCAD solid or Mechanical Desktop solid. That polyline is ready for start point edit or cut. By using this command, you create a NC friendly shape to be cut. This shape can be moved or the start point changed. This shape can also be used by the system to determine the positioning information for NCRotate and alignment of the cut.

Start the NCEXTRACT command from the toolbar, menu or the command line. The system prompts you to select an edge of the part. After selecting the edge, the system will highlight on of the two faces connected to that edge and ask you to accept this face. If it is the face you want, type *Y* for yes, otherwise type ***N*** for no or hit enter to accept the default. If you enter ***No***, the other face that is attached to the edge will highlight, and the question will repeat.

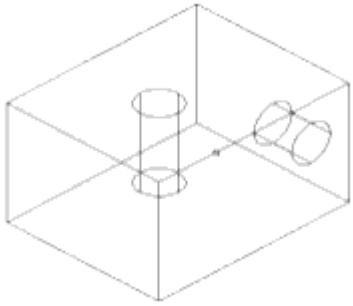
When you accept a face, all of the edges on that face will be converted to shape Polylines automatically. You may then start point edit, move or cut these shapes.

Example:

We would like to extract shapes on the top of this solid model:

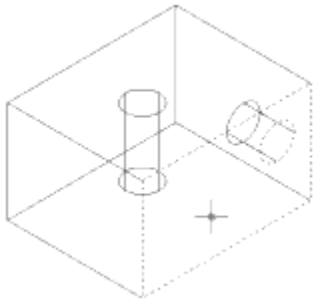


To extract from a Face, you pick the solid on an edge as shown:

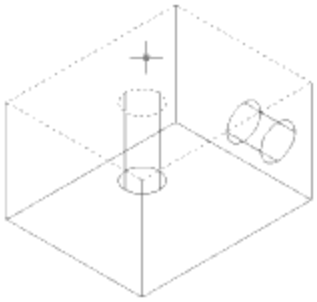


© 2011 CIM-TECH.COM, Inc

The first of the two faces will highlight. This is not the face we want, so we hit N on the Accept prompt.



This is the face we want, so we hit Y at the Accept prompt.



These are the shapes created. The solid has been removed for clarity.

When using NCEXTRACT, pick the face to extract and either Accept the selected face or toggle to the Next face. This face is automatically Geoshaped. You still need to use Start point Edit to choose where to lead in and out.

When using NCEXTRACT with any Mechanical Desktop solid, the resulting NC_shape polyline is parametrically attached to the solid. This attachment is two-dimensional. Whenever the solid changes size, the NCUPDATE command will update the NC_shape polyline. The NCDEPTH command makes the attachment three-dimensional. NCDEPTH is only necessary if you want the polyline to maintain a thickness associated with a solid. NCDEPTH is especially useful when using "A" for the total depth.

Known issues using NCEXTRACT: If you change the start point on a shape that has been extracted, the start point will revert back to an arbitrary location after NCUPDATE.

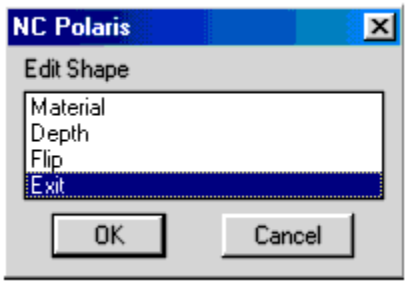
13.3 NCDepth

NCDepth

© 2011 CIM-TECH.COM, Inc

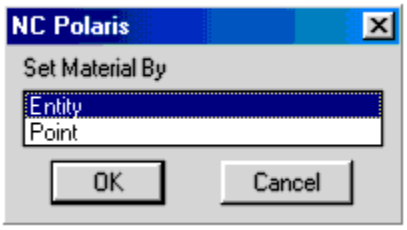
The NCDEPTH command allows the user to take existing shapes and link the material location and cut depth to existing entities, points in the drawing or specific values. The material location as well as the depth of the cut are defined using this command. For graphic feedback, the resultant shapes are moved up or down relative to the position of the polyline for the material location. The thickness of the shape is updated to show the depth of the cut. This depth of cut can be overridden by inserting a depth of cut in the Knowledge Editor. The cut depth can also be adjusted by a value. Use the / character before the number in the Knowledge Editor to add or remove from the depth of defined by NCDEPTH.

The NCDEPTH command has several options:



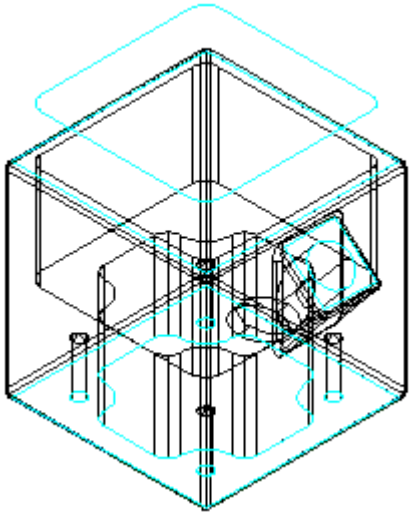
Material

The material location for a cut is defined by the location of the polyline. When you select Material, you are prompted for the shapes to edit. Select as many shapes as you wish. All shapes will be linked to the same material location. When you have selected the shapes, decide how to define the material location.

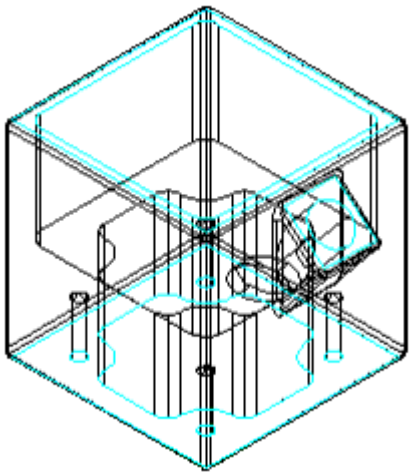


The ENTITY option prompts for an entity. Any entity that defines a point can be used to define the material location for a shape. If a 3D Solid object is selected, you will be asked to define the specific edge of the solid that should be used to define the material location.
The POINT option defines a specific point in WCS that defines the material location.
Regardless of how it is defined, by ENTITY or by POINT, the point does not change the work plane defined by the polyline. The polyline plane will be moved up or down to match the location of the defining point. Careful consideration should be made when cutting on the 4th and 5th rotational axis for proper positioning of the shapes.
For example, this part shows a shape polyline that defines the pocket geometry of the solid model to cut. The part model was moved down after the shape was created. The shape is now too far above the part to be used. The shape location should follow the top edge of the solid model. The material option of the NCDEPTH command allows you to link the material location of the shape polyline to the geometry of the solid model.

© 2011 CIM-TECH.COM, Inc



When you choose the Material Option, by Entity, the first prompt asks:
Select the Shapes to Define New Material Location:
Select objects: 1 found
The shape above the part is selected. Choose the Entity option and select the solid model.
Select Material Object:
Solid Selected!
Select Edge on Solid
Since this is a solid model, you cannot select a specific edge until the solid is selected. The solid model is highlighted at this time, so select the top edge of the pocket hole.

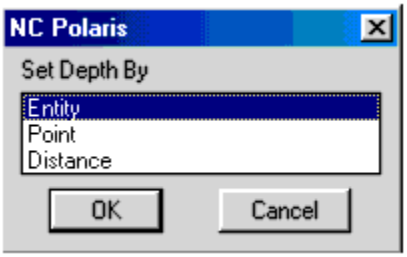


The shape moves down to the top of the hole.

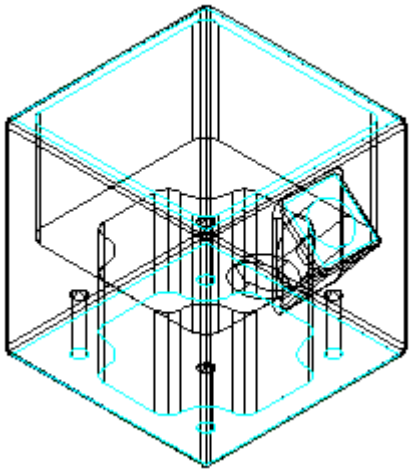
Depth

The depth of a cut is defined as the thickness of the polyline. The thickness is used by Router-CIM when you leave the Total Cut Depth blank, or fill in a plus or minus value that has a '/' <forward slash> in front of it. When blank, the exact thickness of the shape is used. When a Slash Value is used, it is added or subtracted from the actual part thickness. This can be used as an over cutting or undercutting technique.
In our example, the top shape is in position for the material location, and now we must define the cut depth. Cut depth can be defined in three ways:

© 2011 CIM-TECH.COM, Inc

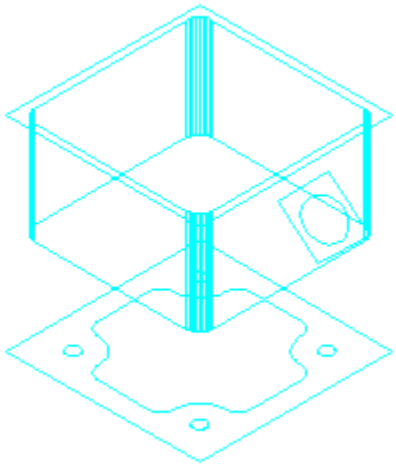


Entity and Point work just like defining the Material location. Distance prompts for a simple depth. For example:



Select the top pocket shape
Select the Shapes to Define New Depth:
Select objects: 1 found

Select the Entity Option, then select the Solid.
Select Depth Object:
Solid Selected!
Select Edge on Solid
The solid edge selected is an edge on the bottom of the pocket shape. The polyline is automatically updated to show the new shape depth. The solid has been removed from this picture for clarity.



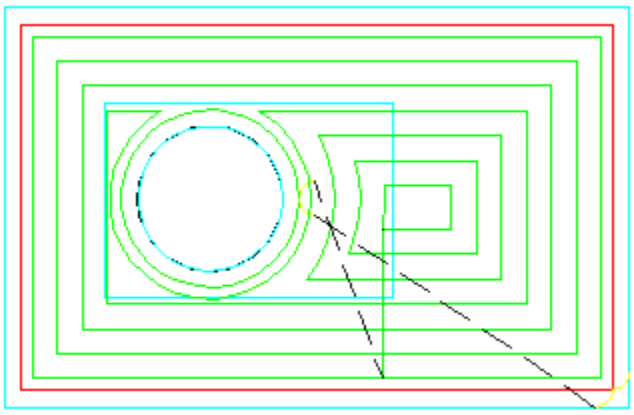
© 2011 CIM-TECH.COM, Inc

Flip
This option allows the user to flip the cutting direction of the polyline to the opposite side. Some shapes are easier to extract from the bottom or back side of a model and need to be flipped to cut properly.
Select the Shapes to Flip:
Select objects:
The selected Polylines will be flipped.

13.4 NCUpdate

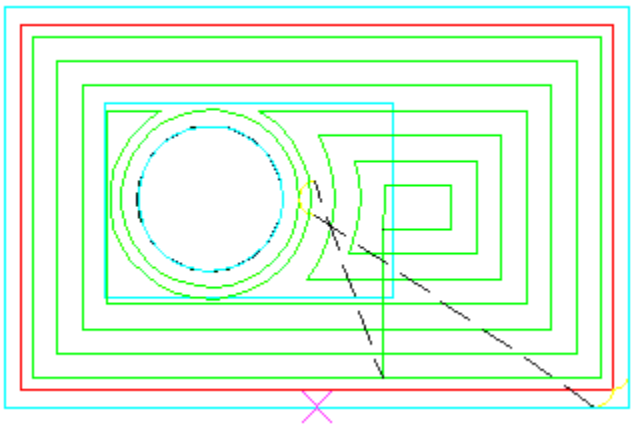
NCUpdate

The NCUPDATE command re cuts multiple cut blocks. A whole sequence can be updated to new geometry. Geometry that has links to other geometry will be updated as well. The NCEXTRACT command links geometry to the face of the solid object it came from. The NCDEPTH command can link shapes to objects that define the material location and cut depth. The STARTPT command can change the start point of the cut. The Move and Rotate commands can change the location and orientation of the shape. The NCUPDATE command searches for the changed shapes and updates them before updating the cuts. If a sequence of cuts is selected, the NCUPDATE command launches the NC Sequence Builder dialog box to allow the user to redefine the sequence, change sorting options, make new nc code, and generate new reports from the cut blocks.
The only input for NCUPDATE is:
Select Cuts to Update:
Select Objects:
Select the cut blocks to update. The cuts will be recreated on the screen. Any changes in the source geometry will affect the results of the new cut.
For example, this simple pocket has a new island shape defined for it:

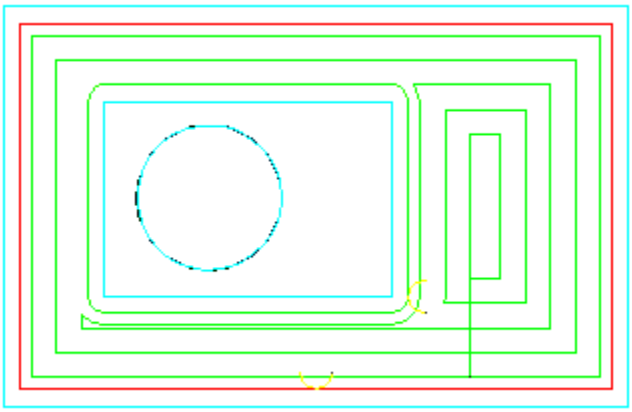


The rectangle is the new island shape. The circle is linked to the pocket and contour cut. Also, note that the start point of the contour cut on the large rectangle is in the wrong place. First, we use the Start Point Edit command to modify the outside shape start point to a better position.

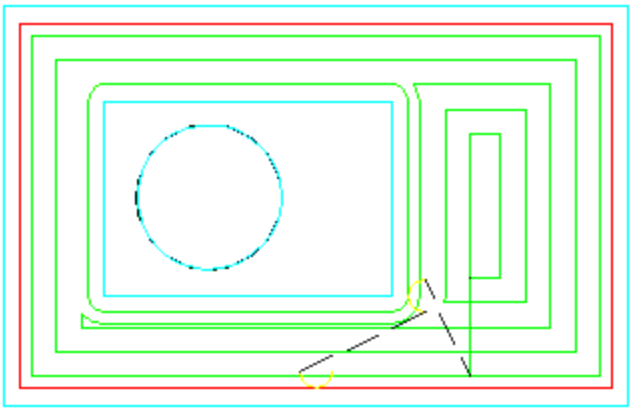
© 2011 CIM-TECH.COM, Inc



Then we use the NCLINKCUT command to link new shapes to the cut. Since NCLINKCUT only works on one cut block at a time, we select the pocket cut first. It is linked to the outer rectangle and the inner rectangle. The contour cut is only linked to the inner rectangle.
Now that the cuts are linked to the new shapes, the NCUPDATE command is run. The sequence of cuts is selected when you choose any one of the cuts, and the cuts are automatically updated to the new geometry.



The index lines are removed from between the cuts, and the sequence options dialog box is displayed. The options that were used to create the original sequence are displayed. They can be edited if you desire, or hit OK to recreate the sequence as is. New index motions are created between the cuts.



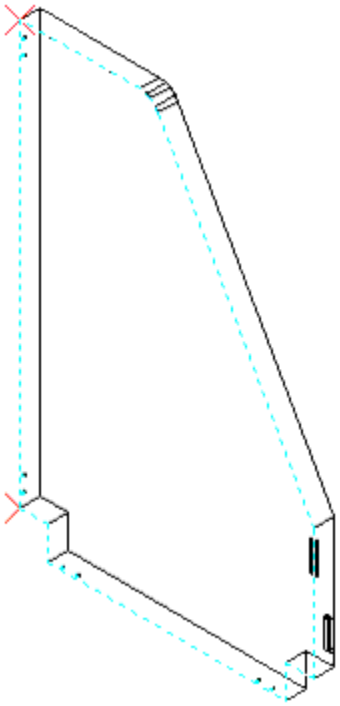
© 2011 CIM-TECH.COM, Inc

13.5 NCFlat

NCFLAT



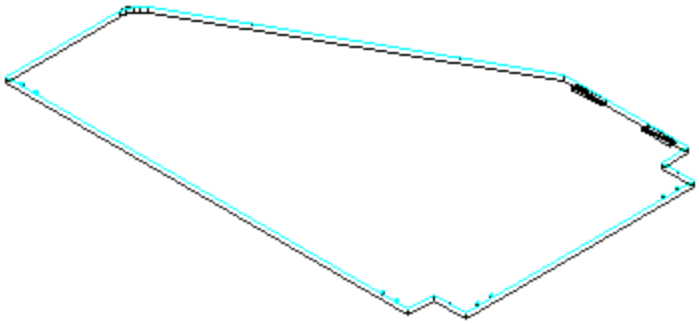
This icon is on the RCIM Solids toolbar. You may also access this function at the command line by typing ncflat. NCFLAT will rotate a part in space so that a selected face is pointing up to AutoCAD WCS (World Coordinate System). The part will also be extracted as if NCEXTRACT had been run. This command is used only on Solids and Assemblies in AutoCAD or Mechanical Desktop. If you wish to use NCFLAT on a part of an assembly, it is recommended you copy the part and run the command on the copy. This allows you to use the Update Assembly command in Mechanical Desktop and have any changes to features, constraints, or dimensions reflected in the update.
To run the command:
1. Click on the icon and you will be prompted:
Select Face
Select the face of the part to orient up. Depending on the part, you may need to choose Next to select the proper face. The selected face will highlight. Hit Enter once you have the proper face selected.
2. The face will appear as a blue dashed line with red X's appearing at the endpoints of the longest segment. At the command line you will be prompted:
Pick origin point <Longside>
Running object snaps will be changed to Endpoint for the next two prompts.
3. Choose the origin base point or hit Enter for automatic longest side selection. The longest side will be noted with a small X at the base point and a small > (arrow) at the x axis position.



If you chose a origin point, select a point that defines where the Positive X-axis should be.
4. Then choose the move to point. It defaults to Autocad World 0,0,0. Running object snaps will be turned off for this pick. A drag view of the part will be shown. World 0,0,0 is the base point of the dragging.
The part will be moved to the move to point at the appropriate orientation in three-d space. The part

© 2011 CIM-TECH.COM, Inc

will be extracted just as if NC Extract had been run.



Note: The longest side calculation only examines the ends of the outside profile. If the longest feature on the outside of the part is an arc, the ends of the arc will be lined up in the X-axis.

13.6 Shapehole

Shapehole

Shapehole is a command that was developed to allow the result of the ncextract geometry to be turned back into holes for Pattern Recognition in Router-CIM.

After an NCExtract or NCExtrude has been performed, and even after NCFlat has been run to put the part in the world coordinate system, use the Shapehole command and select all the circle shapes that you want to be recognized. The result will be circles placed on Layer Drilling to be sent to Pattern Recognition.

If you want to use Drill motions manually, then this step is not necessary.

13.7 Parametric Toolpath in Router-CIM

Parametric Toolpath in Router-CIM

Parametric Tool Path capability in Router-CIM and Mechanical Desktop.

Parametric tool paths work on Router-CIM 2D toolpaths and extruded solids in MCAD. The solid must start at Z0 and be extruded in the negative Z direction. To use this feature, follow these steps:
1) Draw, constrain, and extrude a shape to create a Mechanical Desktop Solid.
2) Load Router-CIM.
3) Use the NCEXTRACT command and select an edge on the solid that is to be cut. This command will make an NC_SHAPE polyline on the solid. Typing N for Next and A for Accept can toggle each edge.
4) Type NCDEPTH to associate the shape with the parametric solid.
5) Pick DEPTH, and at the prompt: Select the Shapes to Define New Depth: select the nc_shape polyline that was created by Ncextract, and then hit Enter.
6) Pick ENTITY, and at the prompt: Select Depth Object: select the bottom of the solid. At the prompt: Select Edge on Solid, pick the bottom edge of the solid again.
7) Pick OK to EXIT. The result of the NCDEPTH command is an Nc_shape polyline that has the same thickness as the solid.

© 2011 CIM-TECH.COM, Inc

8) To make a tool path, pick a tool and cycle, leave the Total Cut Depth field blank, and CUT the shape that was created by Ncdepth.
9) You can edit the solid by double picking on the Extrusion in the Browser. When you change the dimensions, the solid will update and get bigger or smaller.
10) Type or pick NCUPDATE, and select the tool path. This will update the shape and tool path to the newly sized solid. If the solid gets thicker, Depth per pass will start to make multiple Z level tool paths.

The same is true of a standard 2D polyline that was Geoshaped and cut in Router-CIM. Stretching the shape and using the NCUpdate command will change the toolpath.

14 4th Axis Programming

4th Axis Programming

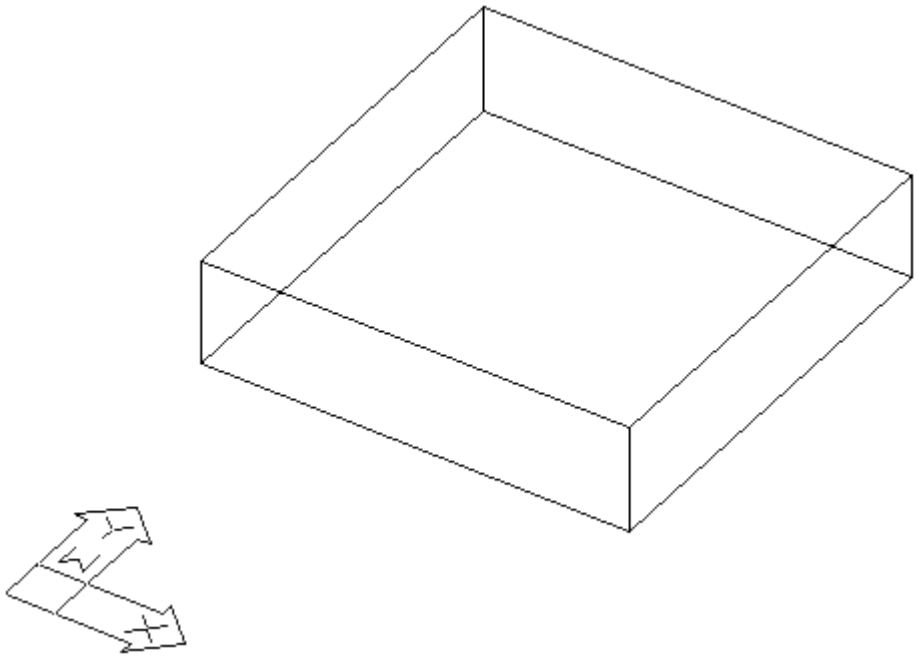
In Router-CIM many types of horizontal routing and drilling may be accomplished with very simple, standard, and familiar knowledge settings. The geometry allowed is the same type of geometry allowed for vertical cuts. For the sake of the machine tool, some care should be taken to insure that the tool selected is capable of the type of cut specified.

14.1 Geometry

Geometry

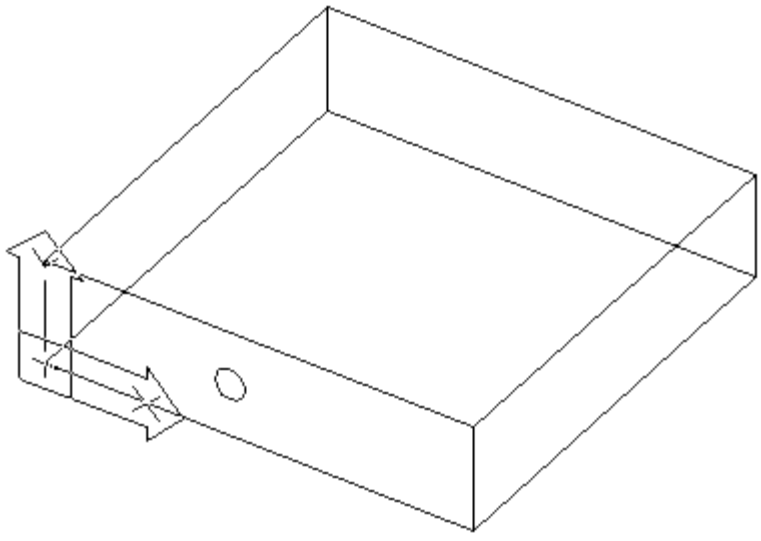
Geometry for horizontal applications must be drawn on the face of the part where the shape is desired and should be drawn to finished size. In other words, draw what you want cut, in the finished size. Parts may be drawn with thickness of the desired depth or the depth may be specified during cut. Depending on the type of cut desired the shape can be open or closed. Start points can be modified to suit the desired starting location of the tool.
Commands have been provided to allow you to draw geometry on the sides of the part easier. The commands are FS for Front Side, RS for Right Side, BS for Back Side, and LS for Left Side. You may return to the World coordinate system with the UCS, command or typing TS, for Top Side. These are simple UCS switches and you can always use the UCS, 3 point command to create a UCS on any face that does not fall into on one of the 4 sides of a part. If you create a UCS of your own, be sure to use the lower left corner of the part as the origin and that the positive Z direction is away from the part and negative Z is into the part. This way your cut will move into the part and not be inside out. The following example shows two pieces of geometry being created on two different sides of a part.

© 2011 CIM-TECH.COM, Inc



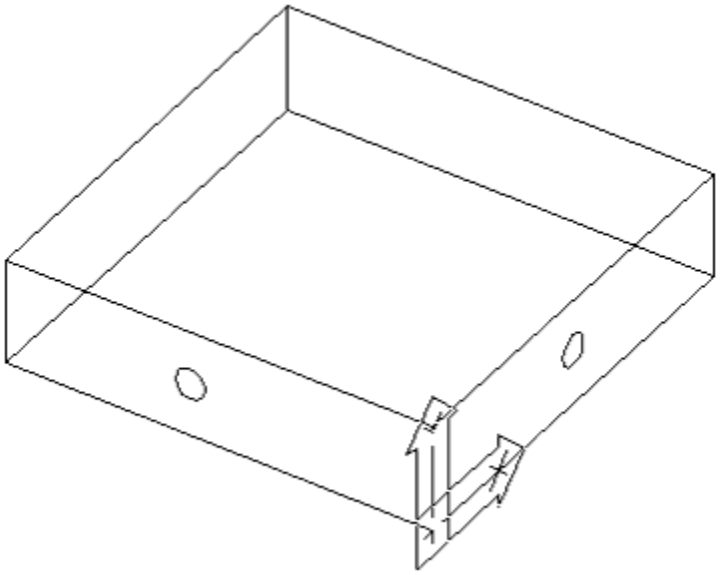
Standard part in World Coordinate System

This part is a standard rectangle drawn with thickness. The UCS Y is pointing at the front side of the part in the illustration above. To draw a shape on the front of your part use the FS command and when prompted for the origin, select the lower left intersection on the front face as your origin.
After the FS command is used, you will have a UCS Icon on the front of your part as shown. You can then draw your shape on the front of your part.



Using the same technique, you can also draw a shape on the right side of the part. Using the RS command, select the lower left corner of the right face. Then draw a circle on the right side.
The shapes you have drawn can now be geoshaped and cut using the standard cut cycles. The next section describes the cutting methods necessary.

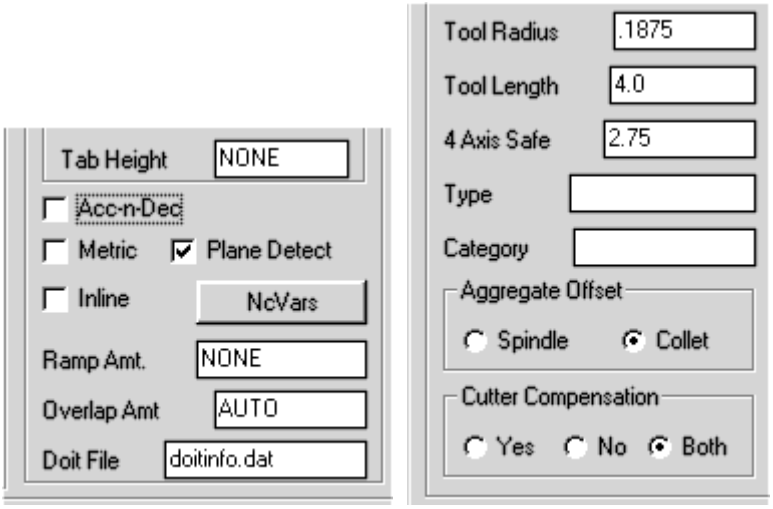
© 2011 CIM-TECH.COM, Inc



14.2 Cutting

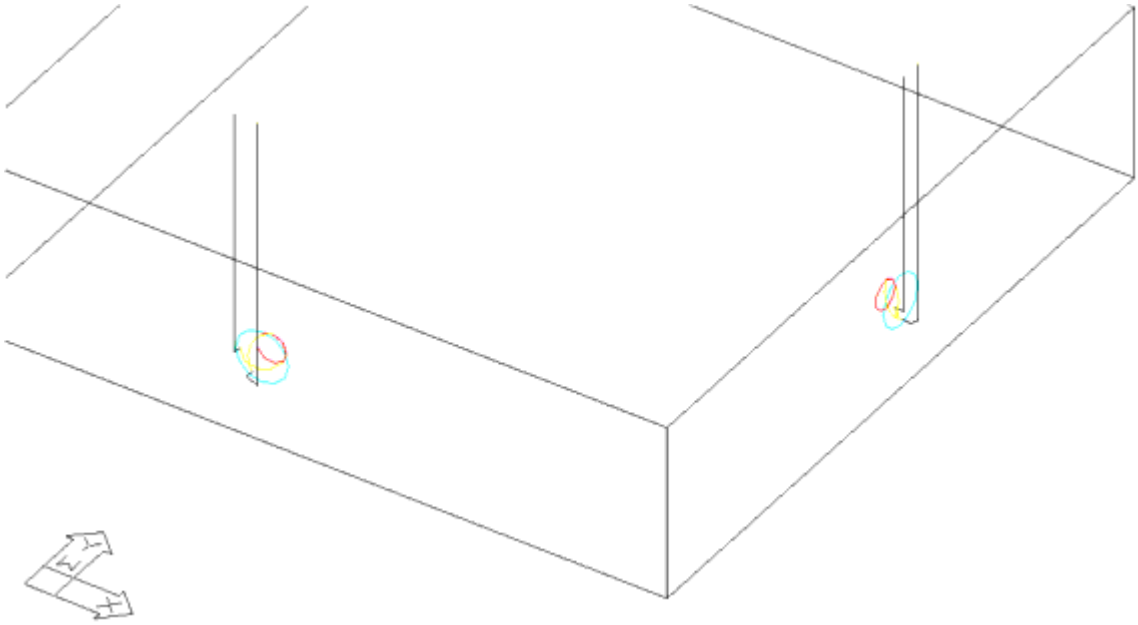
Cutting

Once the shapes have been geoshaped, you will be able to set up the cut knowledge necessary to cut your shapes. All of the standard cut cycle are available to use for horizontal cutting and only a few considerations are necessary to insure a proper cut. On the status screen, you should select your cutting tool and cycle. In order to insure that the horizontal tool will clear the part in the Z axis, a 4 axis safe height is necessary. The value stored here will be output as the Z clearance height generated before and after the cut. In the lower right side of the screen there is a Plane Detect checkbox that, when activated, will recognize which plane the geometry is drawn on, and generate the correct code for that particular face.
Both of these items, the 4 axis safe and the Plane Detect should be used when making a horizontal cut.

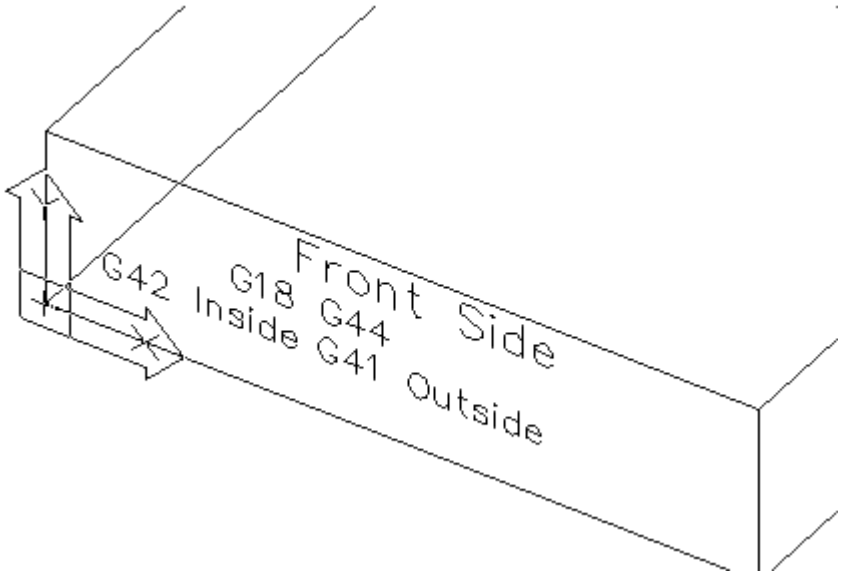


© 2011 CIM-TECH.COM, Inc

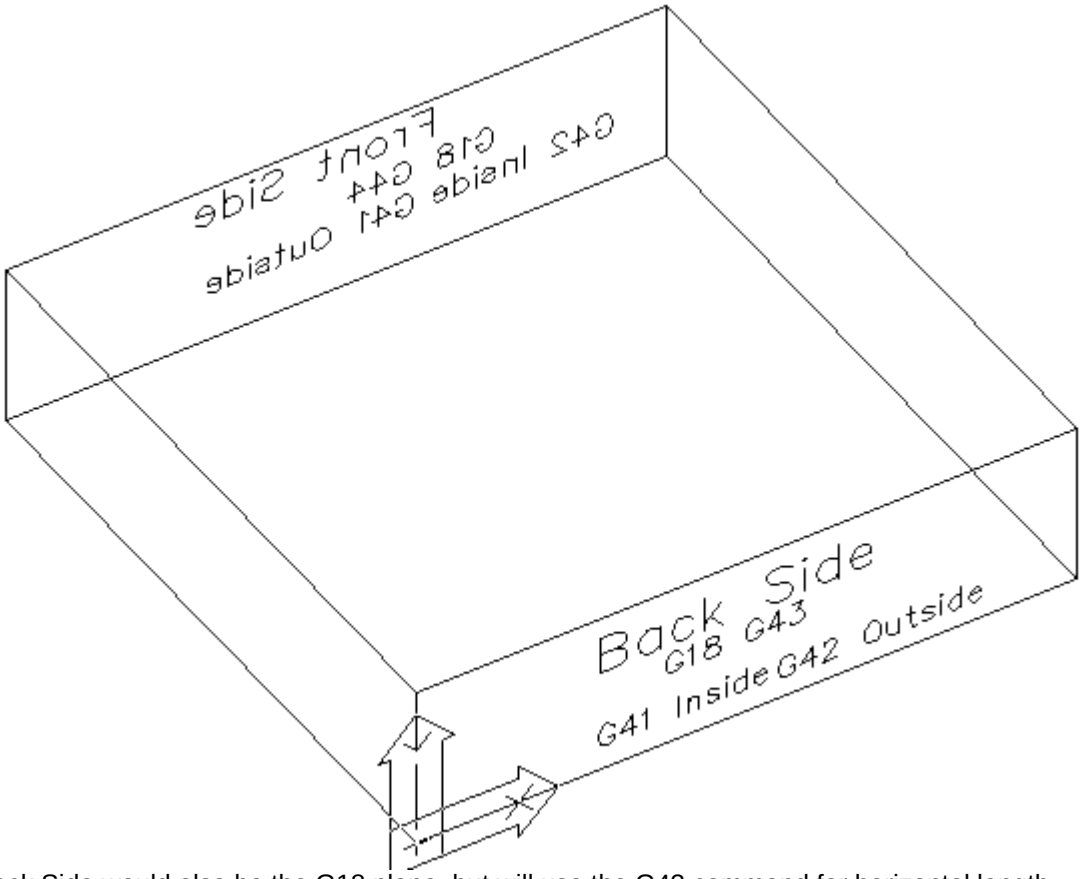
The 4 Axis Safe and Plane Detect settings on the Status Page.
Producing the cuts shown below.



Cutting Planes
When making cuts on the sides of a part, the geometry may lie on one of the planar faces that the machine tool can recognize.
On the Fanuc control, the front side of the part will be the G18 plane and use the G44 command for horizontal length compensation, and cutter compensation will use G42 for inside cuts and G41 for outside cuts.

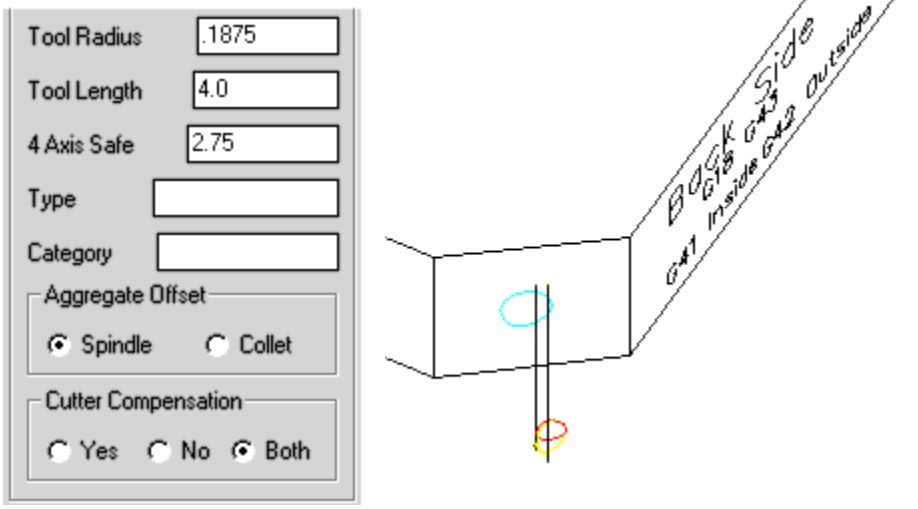


© 2011 CIM-TECH.COM, Inc



The Back Side would also be the G18 plane, but will use the G43 command for horizontal length compensation and G41 for cutter compensation on inside cuts and G42 on outside cuts.
The difference in the G43/G44 commands is so the action of the tool as it sets the horizontal length compensation is always the same and POSITIVE values are placed in the offset registers in the controller.
On the left and right sides of the part, the G19 Plane will be called. On the left side, the G19 plane uses the G44 command and G42 inside and G41 on the outside, just like the front face does. The right side of the part uses G19 with G43 and G41 on the inside and G42 on the outside cuts, just like the back side.
In addition to the commands listed some of the G02 and G03 commands for arcs will be reversed as well.
The post processor will generate arcs in the proper direction for the plane selected. If a helical cut is made on a G18 or G19 plane, then G01 point to point moves will be made for the helical portion of the cut and G02/G03 moves made for any flat, planar moves by the cutter.
If the geometry drawn does not lie directly on the G18 or G19 plane, and Plane Detect is checked, then the cuts will be generated as point to point G01 moves in the G17 plane. If this method of cutting is necessary, then another switch is located on the status page that will allow an offset value based on the Tool Length field. The G43/G44 cannot work if the cut is not planar. To accomplish the offset that would normally be done with the G43/G44 command, you must set the toggle Aggregate Offset to Spindle instead of Collet so that the cut is shifted by Tool Length away from the shape.
In the picture shown, the parameters are set as follows:

© 2011 CIM-TECH.COM, Inc



Setting the Aggregate Offset insures the offset shown. The Tool Length offset of 4.0 is the distance from the cut to the shape. If the safety plane value is *.25, the depth of cut is -.5, then the tool moves .25 inches away from the shape, then down to the cut position in X and Y, and then lead in from the .25 inch safety plane into the part .5 and then lead out to .25 away from the part and up to Z 2.75 inches (assuming Z0 is top of part). If the Z0 is the top of the table, then the Z value at the start and end will be 2.75 plus the part thickness.

14.3 4th Axis Interpolation Cycle

4th Axis Interpolation Cycle

This cycle is to provide continuous four axis motions for aggregate heads and 90º heads on routers. There is a provision to allow different tool geometry in the toolpaths, such as routers, moulder heads, angled heads.
Typically the geometry provided is still flat 2-D geometry, but 3-D geometry can be provided to all full 4th axis simultaneous programming if the machine allows for this motion.

14.3.1 Drawing Requirements

Drawing Requirements

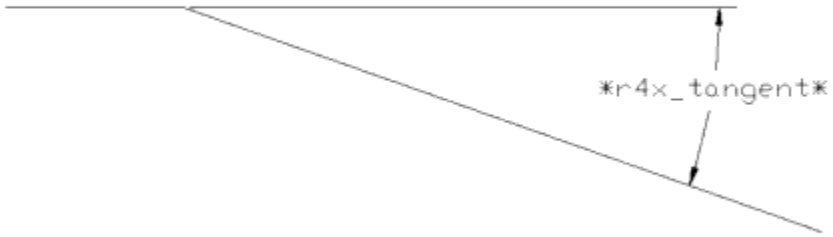
This cycle works on standard geometry created by the Geoshape command and edited with the start point edit command. The shape polylines must be either a two dimensional polyline in the world plane or a three-dimensional shape. Two-dimensional shapes on the side of the part do not need to be interpolated in the fourth axis and any or all of the existing cut cycles will work to create cutting valid cutting motions for these shapes. If an invalid shape is selected, the cycle will tell you when you try to cut it.
The Shape drawn for the Forth Axis Cycle must represent the material location to be cut. The tool tip will extend into the cut shape by the depth specified in the Total Cut Depth specified. The tool path can be shifted down in Z during the cut cycle by specifying the depth in the Move Shape Z field, or the shape can be drawn at the required Z depth. For clarity, we recommend that 3D shapes be drawn at the appropriate Z depth. If a shape has thickness, this will represent the required Z Shift for the Move Shape Z field.

© 2011 CIM-TECH.COM, Inc

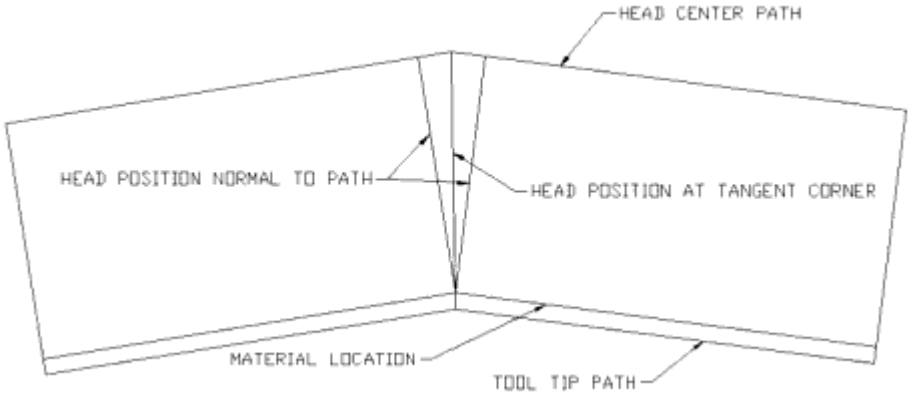
14.3.2 Cycle Corner Treatments

Cycle Corner Treatments

Tangent Corners
The system variable *r4x_tangent* defines how far from tangent a corner can diverge and still be called tangent. This variable can be edited with the NCVAR command. The following picture illustrates what is measured to determine tangency.



When a tangent corner is found, the motion around the corner smoothly changes the angle of the head to bisect the angles that would represent the position perpendicular to the material. The following illustration shows the head position at the tangent corner as well as the positions that would have been perpendicular to the material to be cut. For clarity, this illustration exaggerates the corner appearance. In normal use, this corner would be much flatter than the illustration.

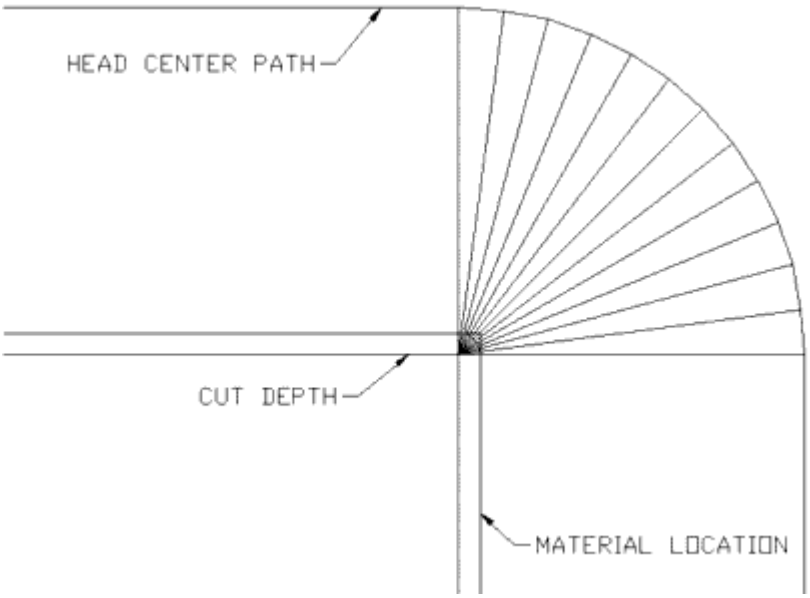


14.3.3 Outside Corners

Outside Corners

To get around an outside corner, the corner is cut with a rounded motion. The Head Center Path will swing around the corner while the tool tip remains stationary at the corner point. The following illustration shows this path.

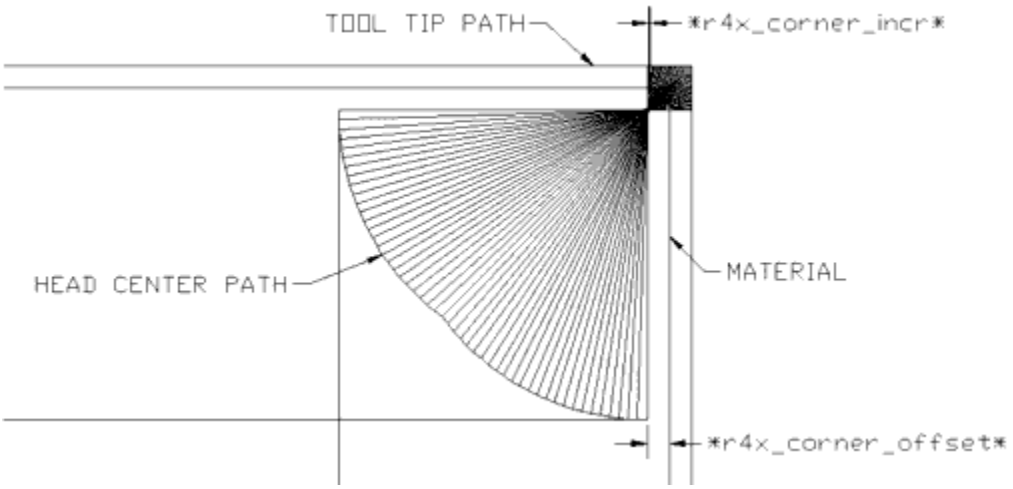
© 2011 CIM-TECH.COM, Inc



14.3.4 Inside Corners

Inside Corners

Inside corners in this application create the motion shown below.



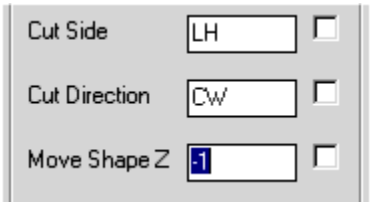
The inside corner logic uses two geometric variables to control the motions in the corner. The two variables are *r4x_corner_offset* and *r4x_corner_incr*. The illustration shows what these two variables control. The head moves perpendicular to the material until it is within *r4x_corner_offset* of the corner to be cut. At that point, the remaining segment to be interpolated is broken by *r4x_corner_incr* and the motions of the head are interpolated around while the angle of the head is changed as shown. When the tool reaches the actual corner, the tool direction will bisect the angle of the corner. Then, the process is reversed until the tool is again *r4x_corner_offset* from the corner and normal to the material.

© 2011 CIM-TECH.COM, Inc

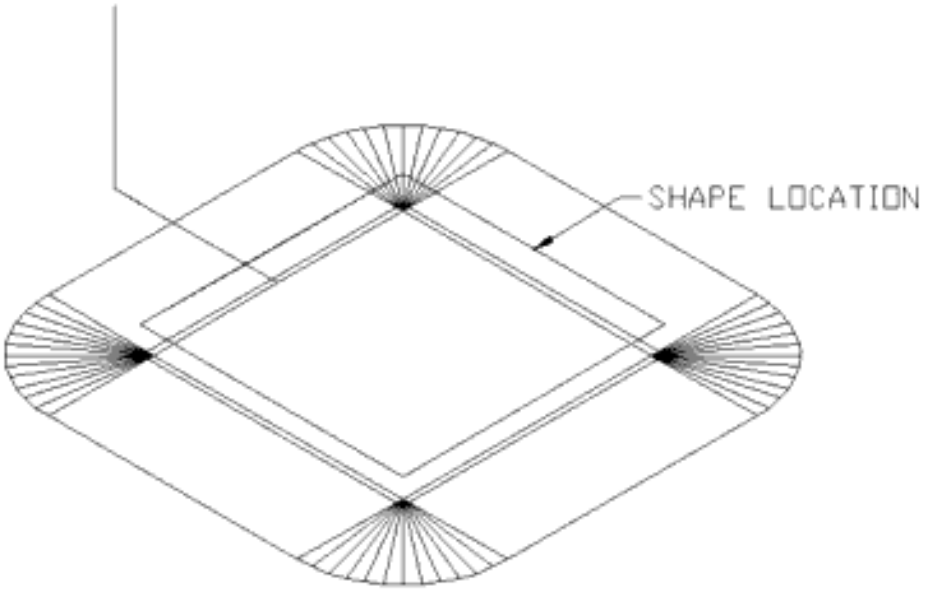
14.3.5 Position and Direction Control

Position and Direction Control

You can change the cut direction from CCW to CW and the cut side from RH to LH. These options work like any other cut cycle in Router-CIM.
Total Cut Depth controls how deep into the material the tip of the tool goes. Depth Per Pass can create multiple passes. The safety plane is measured as the distance from perpendicular to the end of the shape. It is defined in the plane of the leads. If Horizontal leads are specified, the safety moves are horizontal. If Vertical leads are specified, the moves are vertical. Due to concerns on operator and machine safety, ALL moves from the safety plane are made in feed motions. The 4 Axis Safe value must be specified for this cycle, and controls how far above the shape we index.
Move Shape Z is a special parameter for the Fourth Axis Interpolation cycle. It shifts the position of the cut up (positive) or down (negative) in Z by the amount specified. If no amount is specified, and the shape has thickness, the cut will move by this amount.



Moves the cut as shown:



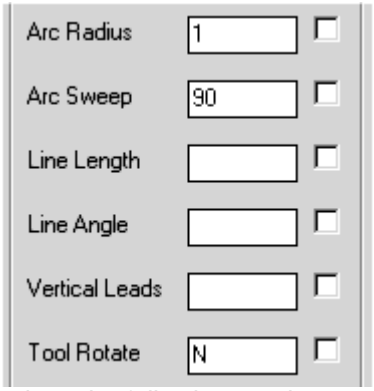
14.3.6 Lead Styles

Lead Styles

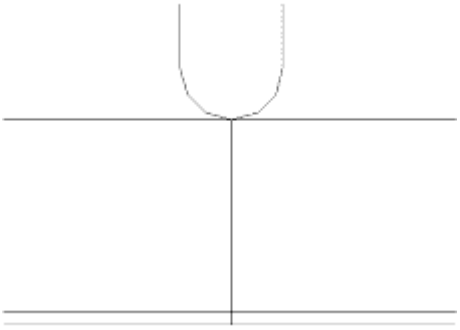
Several styles of leads are supported with the Fourth Axis Interpolation cycle.
Arc Leads
There are three styles of Arc Leads, depending on the choices made in the Status Page.

© 2011 CIM-TECH.COM, Inc

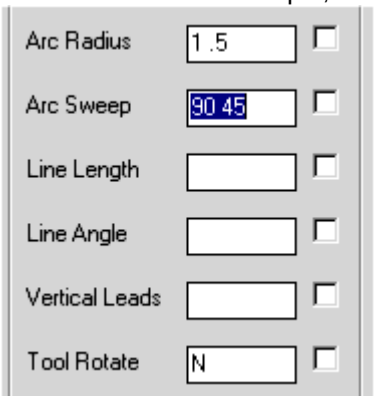
Horizontal Arc In



These settings in the Status Page produce the following results:

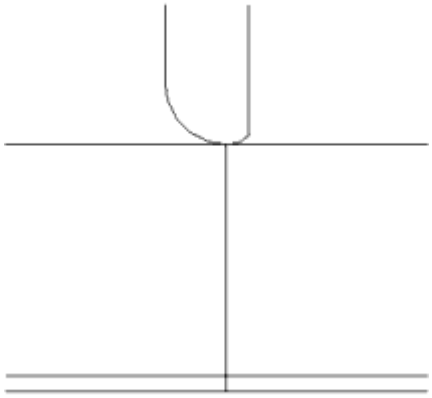


The tool is rotated to normal to the material before the lead in begins, then it moves in to the cut without rotating the tool. The Lead out moves away from the part without rotating the tool.
If a different lead in and lead out is required, two values separated by a space will apply the first value to the lead in and the second value to the lead out. For example, these settings:

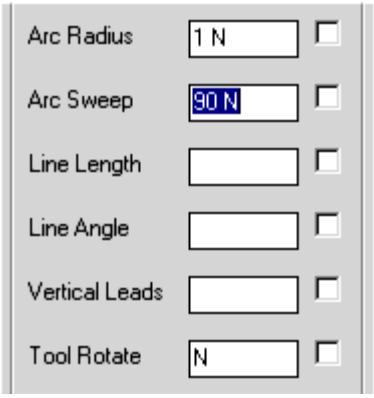


Produces the following result:

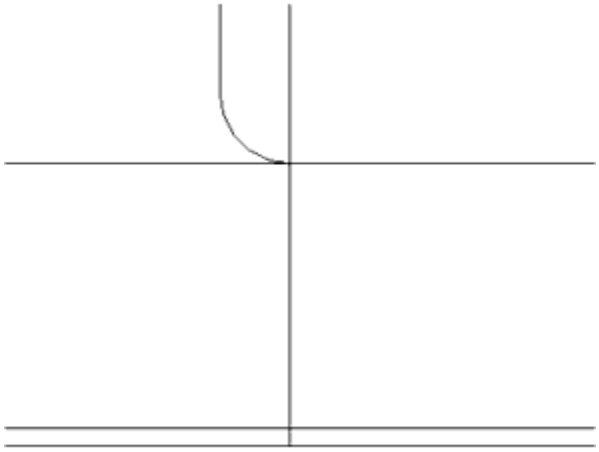
© 2011 CIM-TECH.COM, Inc



The lead in is the same; the lead out has changed. If you wish to suppress one or the other, put an N in place of a value as shown:



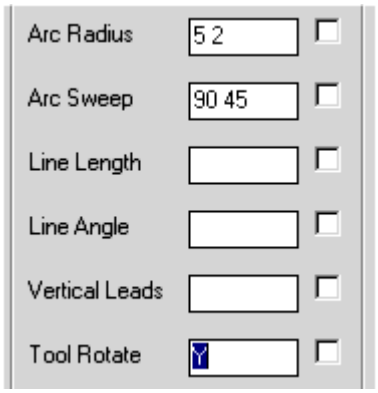
The N in the second position will suppress the lead out:



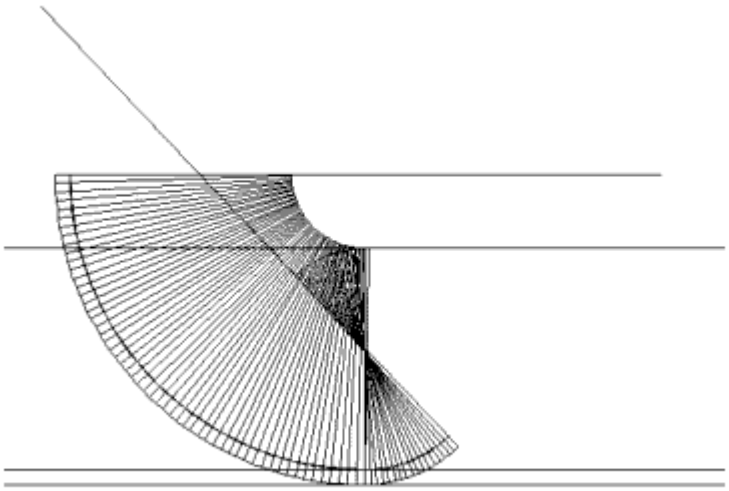
An arc lead in can only be created when both the radius and the sweep have been defined.

© 2011 CIM-TECH.COM, Inc

Horizontal Arc Leads with Tool Rotate
When you turn on tool rotate for the arc leads, the tool will sweep in on the arc. The following settings illustrate some of the actions that can be performed with the tool rotate option.



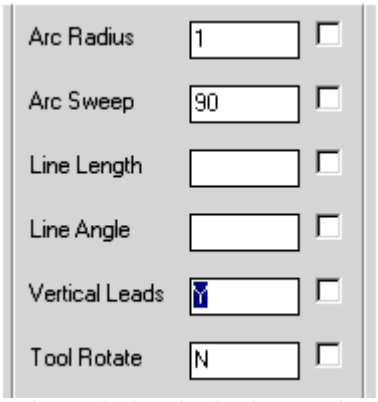
Produces the following results:



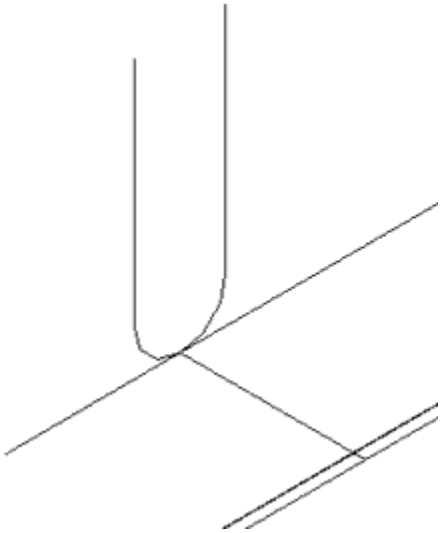
The tool is rotated 90 degrees to the shape, moves into the arc and sweeps by rotating the head as the tool center moves on the arc. In this example, the head is defined as 4 units long. On the lead out, the arc is smaller than the tool radius, so the reverse arc effect is shown as the tool pivots around 45 degrees before backing away from the part.
The tool rotate option can also be used selectively on the lead in and out by separating the values for the lead in and lead out by spaces. Tool Rotate only works on Horizontal Arc Leads.

Vertical Arc Leads
When the vertical option is turned on, the arc is created vertically instead of in the XY Plane.

© 2011 CIM-TECH.COM, Inc



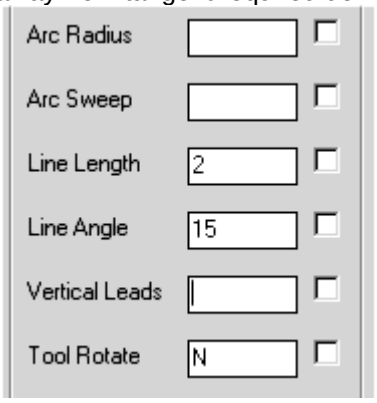
The results of setting vertical leads is shown below in the isometric view:



The tool is rotated perpendicular to the shape, then brought down onto the shape. As always, you can have different behaviors of the lead in and out by separating the values by spaces.

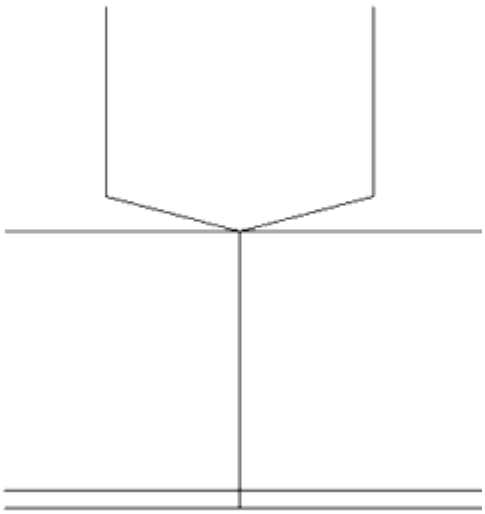
Horizontal Line Leads

The length of the line and the angle away from tangent required defines Line Leads. For Example:

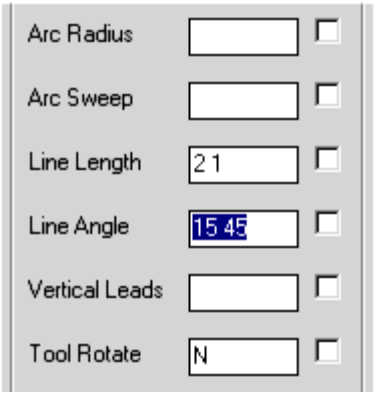


Produces the following results:

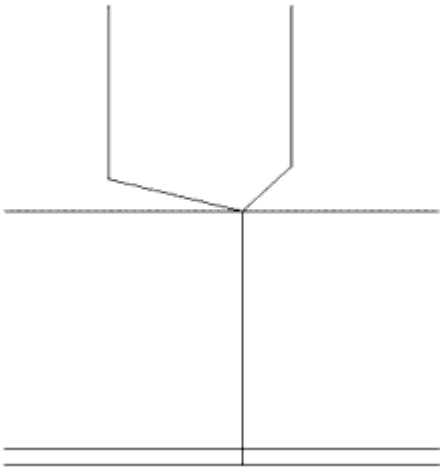
© 2011 CIM-TECH.COM, Inc



Like the arc leads, different values can be entered to achieve different results for the lead in verses the lead out.



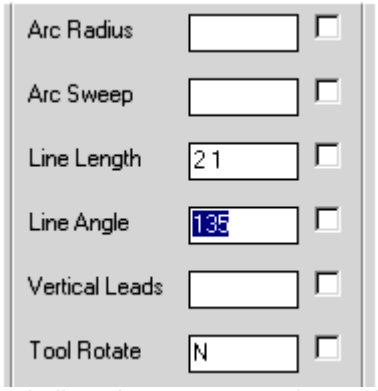
Produces



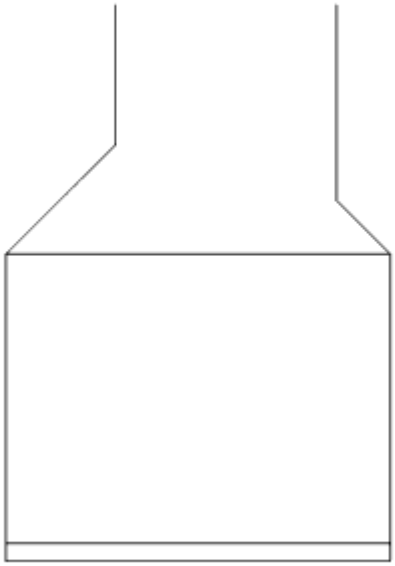
If the angle entered is greater than 90 degrees, the line will cut backwards to the lead point. This can

© 2011 CIM-TECH.COM, Inc

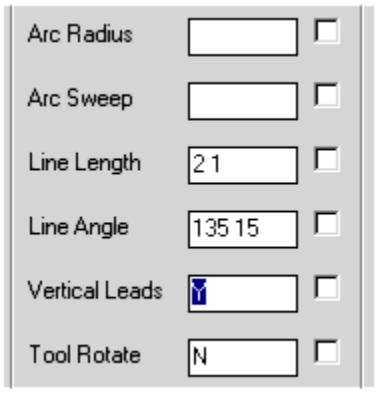
be useful for entering slot cuts on the side of the part with a ramp in motion:



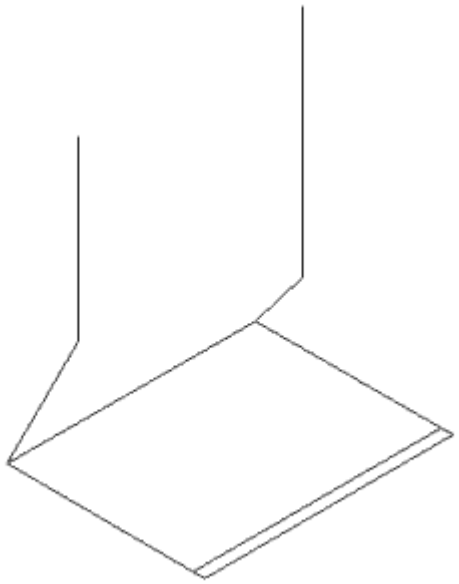
When these settings are cut on a simple line shape representing a slot cut, you get the following results:



Vertical Line Leads
Line leads can be vertical as well:



© 2011 CIM-TECH.COM, Inc

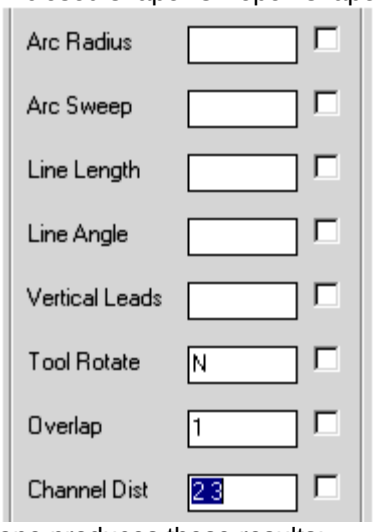


The lead in is a backward facing move, the lead out tapers off the end of the cut. Mixing vertical with horizontal only requires Y or N separated by a space as required.

Channel Leads

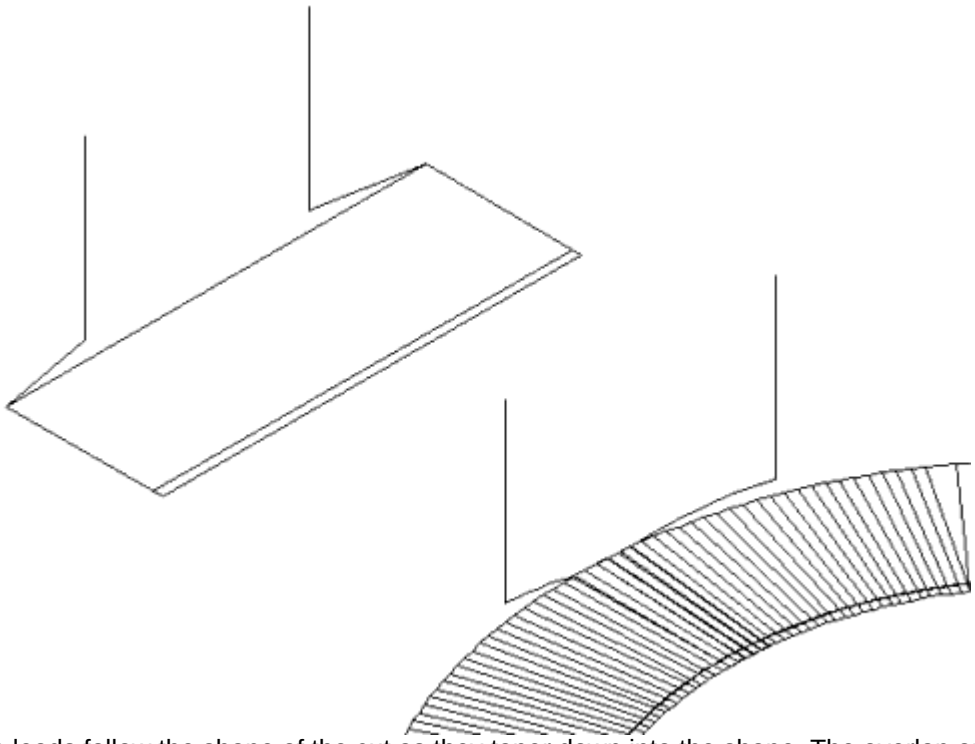
Channel lead in and out is available as well. They are always vertical, and do not appear when cutting 3D geometry. When the geometry is open, the lead in will cut back along the contour until it gets to the lead in point, then it will cut forward along the shape. The channel out will cut to the end of the shape then back up along the shape up and out.

Overlap is often used with channel style leads, but it can be used with the other lead in styles as well. The Overlap parameter only works on closed shape. On open shapes you do not get an overlap.



When cut on an open and closed shape produces these results:

© 2011 CIM-TECH.COM, Inc



Notice that the leads follow the shape of the cut as they taper down into the shape. The overlap only occurred on the closed shape. The direction of the lead in and lead out reversed on the open shape. Familiarity with the commands for you particular machine will insure that you can recognize the code as generated by the post processor.

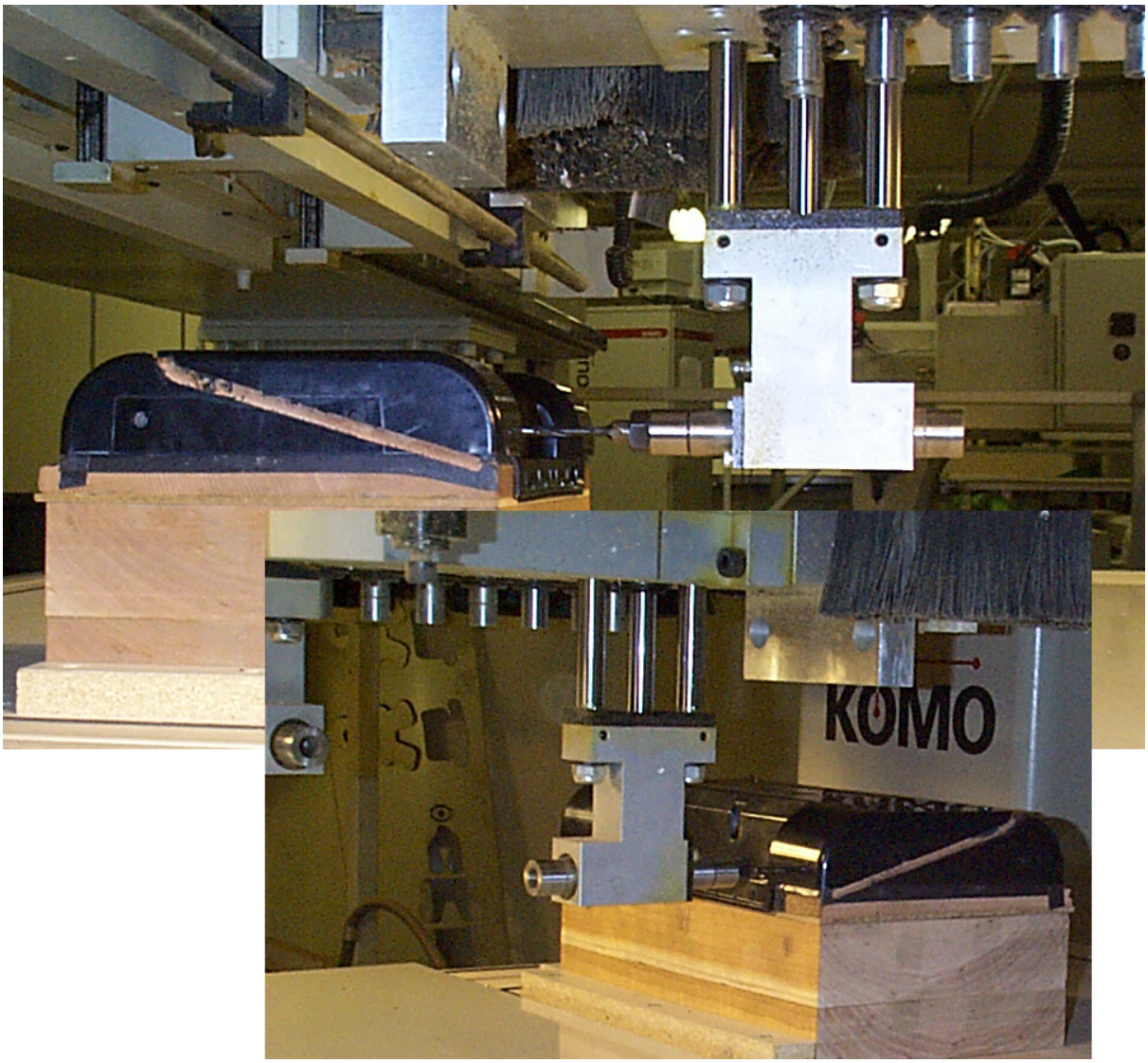
The use of specific aggregate tooling may require slight modification to these parameters, and CIM-Tech is available to assist you in configuring these parameters to suit your needs.

15 Horizontal Boring

Many machine tools have horizontal boring capabilities. Some machines have boring blocks with horizontal drills, some have aggregate tools with horizontal boring capability, and some have both.

The following section is a short tutorial covering some methods that you can use in Router-CIM to make horizontal boring tool paths. There is also some discussion about setting up offsets and work coordinates depending on the methods of cutting you use and how your machine tool is equipped.

© 2011 CIM-TECH.COM, Inc



15.1 Horizontal Boring

Horizontal Boring

Top Next

There are 4 possible ways to do horizontal boring operations in Router-CIM with a Boring Block. There is a drawing attached to this tutorial as a sample and there is a knowledge for each one of the 4 methods. This document will explain how they would work on the machine.

The drawing has holes on each side, and for horizontal boring you must draw the circle on the correct Face of the part.

To help you with this, Router-CIM has 4 UCS commands to help fix the UCS on the correct Face.

The FS command will ask for the origin point (lower left corner) of the Front side of the part.
The RS command will ask for the origin point (lower left corner) of the Right side of the part.
The BS command will ask for the origin point (lower left corner) of the Back side of the part.
The LS command will ask for the origin point (lower left corner) of the Left side of the part.

© 2011 CIM-TECH.COM, Inc

If the parts are not drawn on the correct face, the machine will not cut them properly.

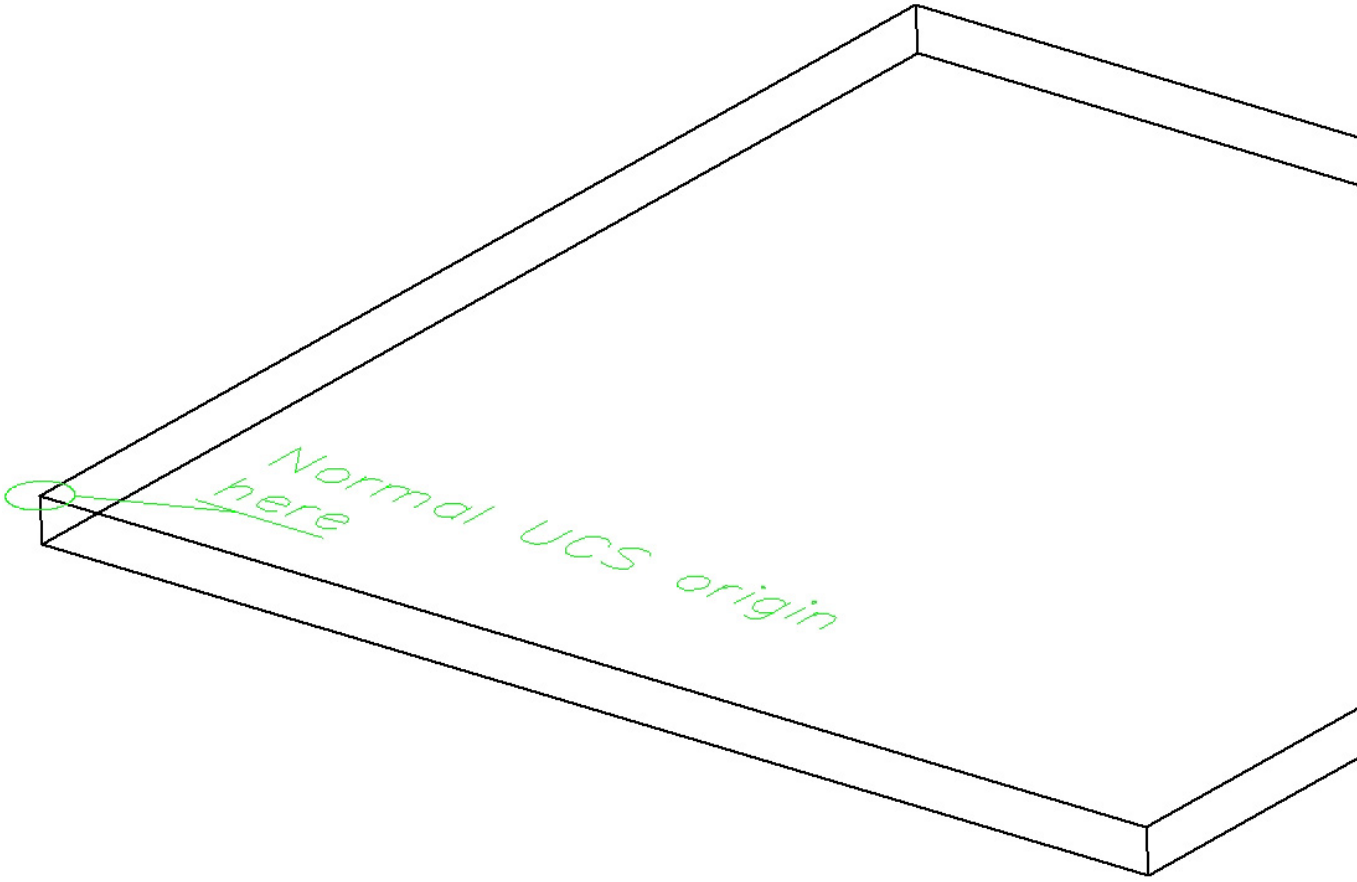
15.2 Using FS, RS, BS, LS

Using FS, RS, BS, LS

Top Previous Next

To use the Router-CIM quick UCS commands, you simply need to enter the command (FS, RS. BS. LS) at the com
location, select the Lower Left Corner of the part.

The normal World Coordinate system is set to the lower left corner of the top of the part:

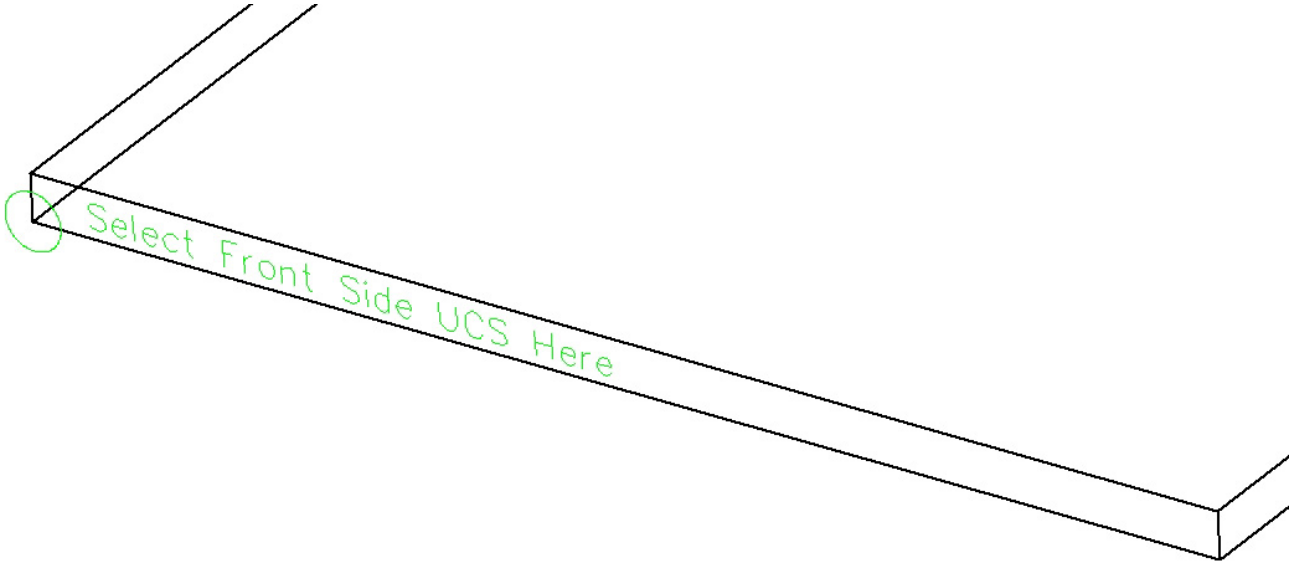


To use the side UCS commands, you need to select the lower left corner of the face you are going to work on. For e

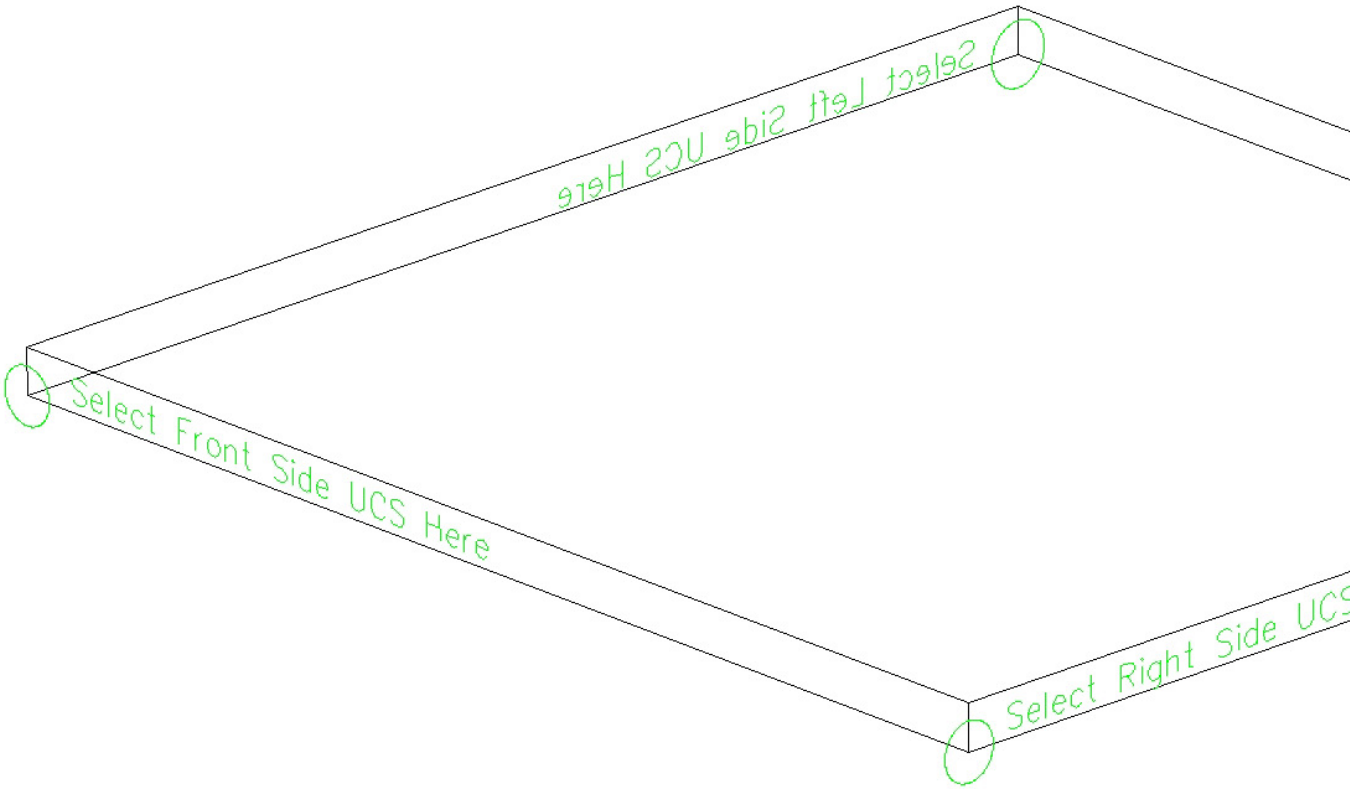
```
fs Origin point <0,0,0>:
```

From this prompt, select the Lower Left corner of the front side of the part

© 2011 CIM-TECH.COM, Inc



To demonstrate, there is a drawing showing the correct point of each of the 4 sides.



The basic point to make is that after entering any of the 4 quick UCS commands, you must choose the Lower Left C

© 2011 CIM-TECH.COM, Inc

15.3 Setting up Knowledge

Setting up Knowledge

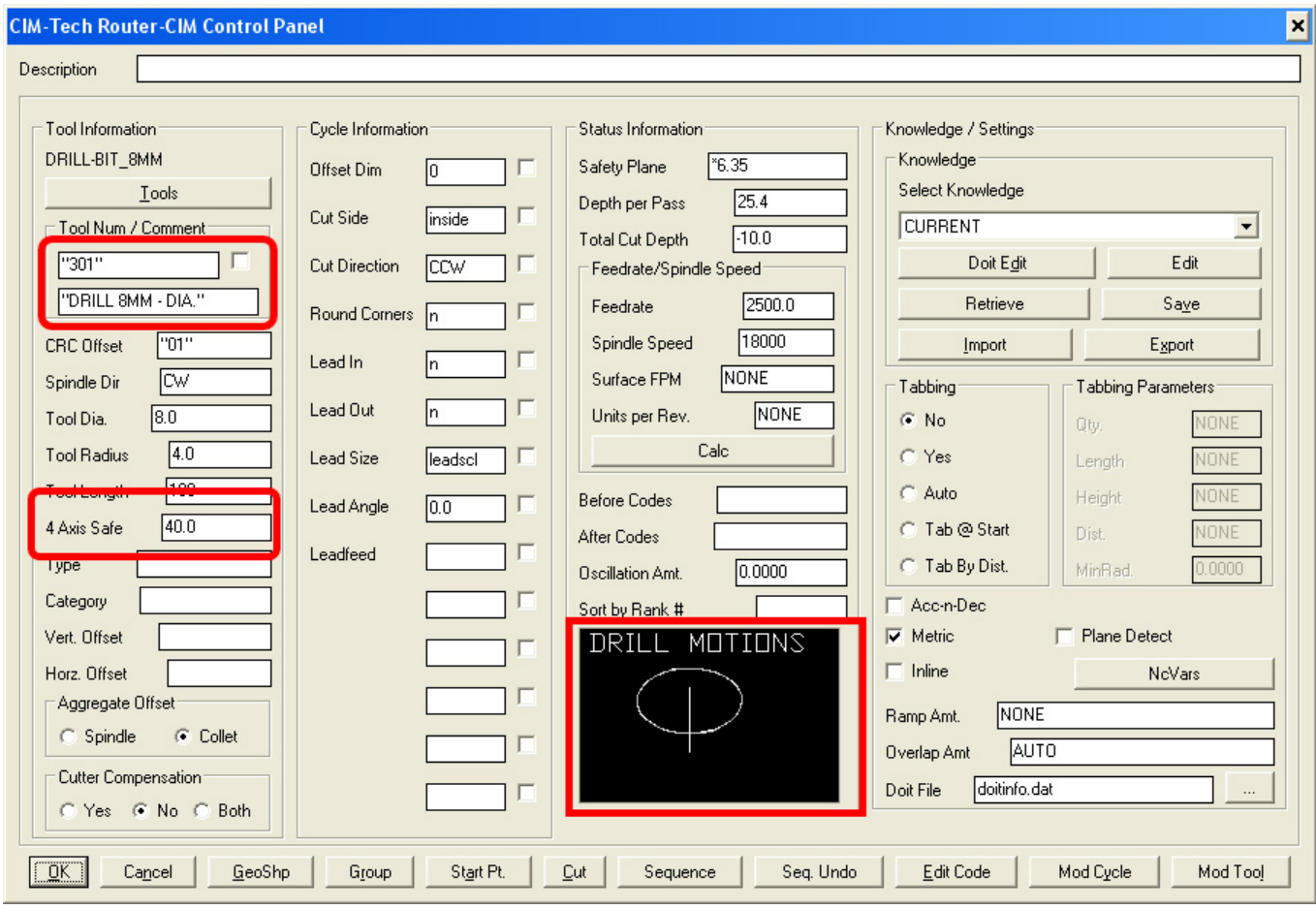
Top Previous Next

Setting up Knowledge for Horizontal Boring

As mentioned earlier, there are 4 ways to drill horizontally, each being slightly different both at the machine and in R

Each of the types of horizontal boring use the Drill Motions cycle, and you must select the correct Tool Number for th

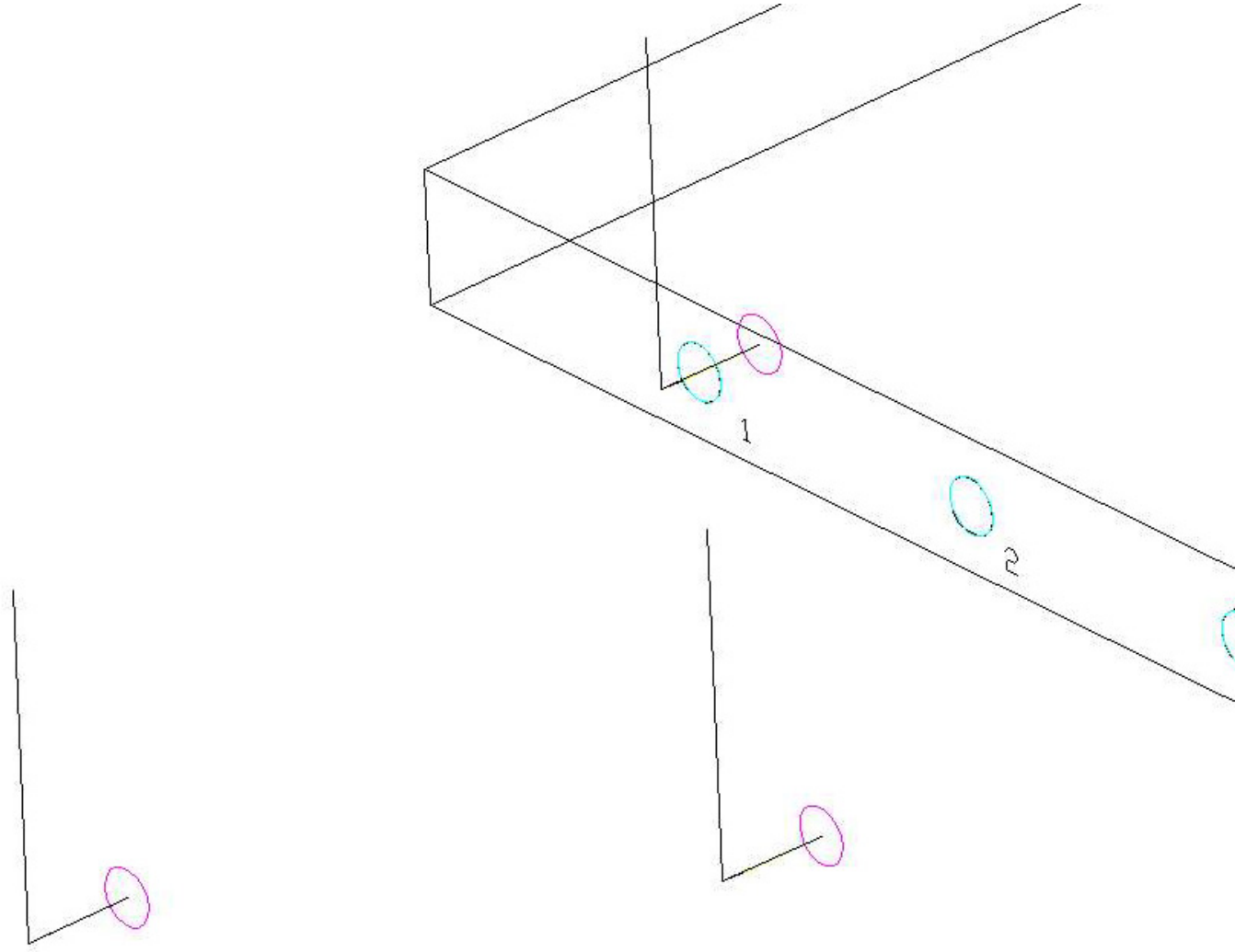
In addition, each uses the 4 Axis Safe position to move the drill up above the part between cuts.



Included in the Sample Drawing are 4 knowledges for each face. These represent the 4 different methods of cutting correct tool numbers for each face.

This document will use the Front Side Drills as an example.

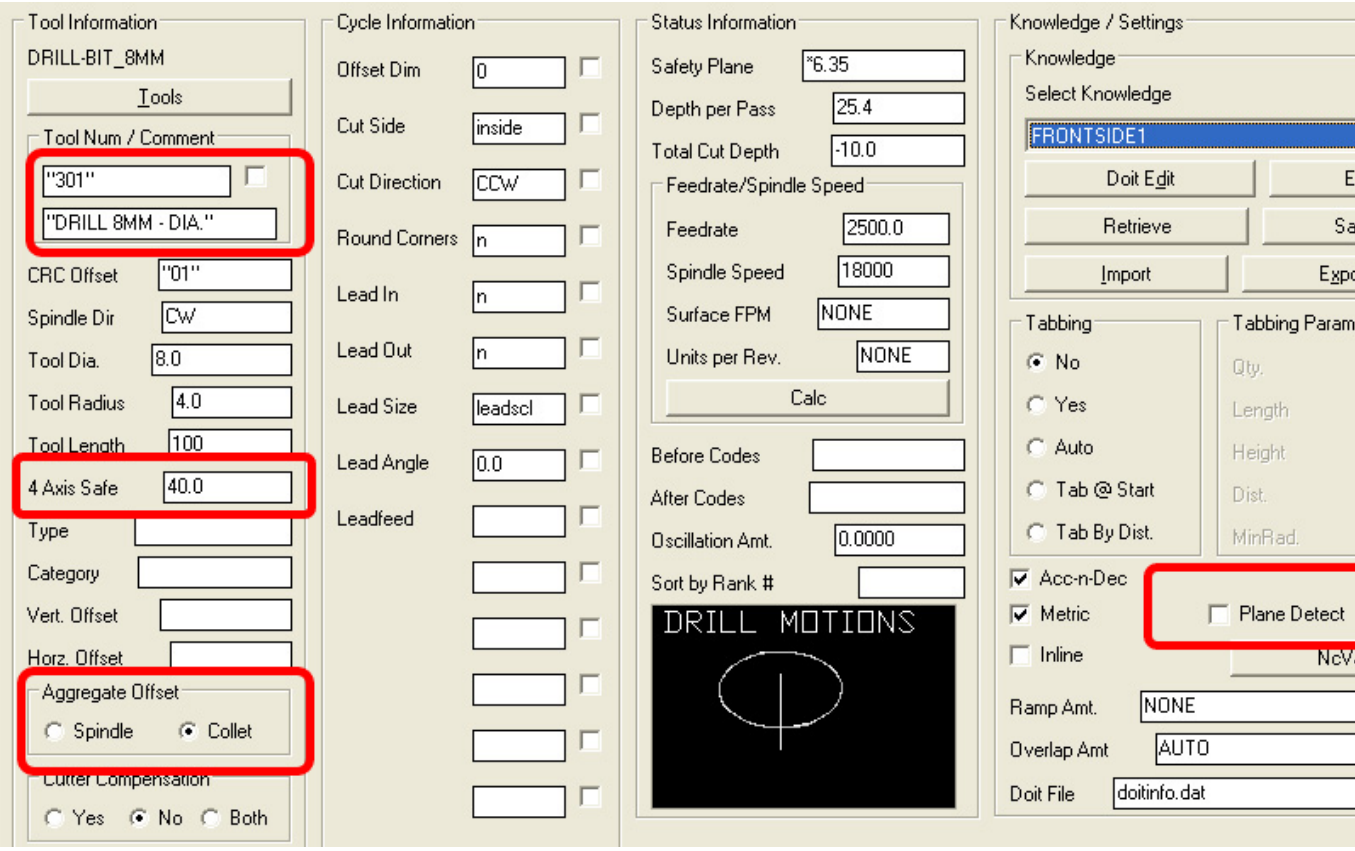
© 2011 CIM-TECH.COM, Inc



FrontSide1

The FrontSide1 knowledge is a very basic boring knowledge and is the easiest to set up in Router-CIM, but not the e

© 2011 CIM-TECH.COM, Inc



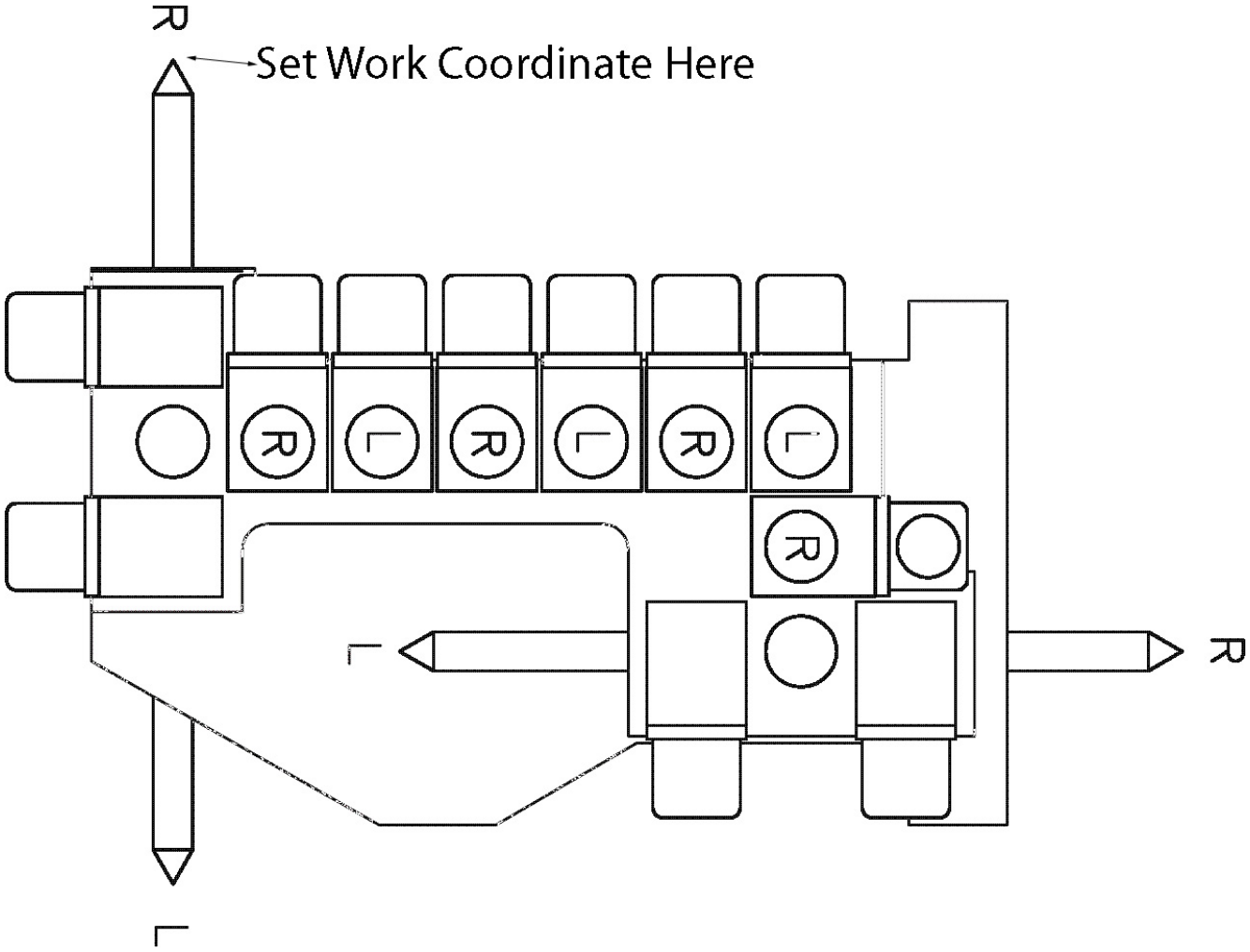
This knowledge uses a 4 Axis safe and has the Aggregate Offset set to Collet and Plane Detect is Not selected. When this knowledge performs a cut, the tool path shows up in a normal location with the cut inside the circle. The Axis Safe setting makes sure the tool moves to 40.0mm above the top of the part both before and after the cut. The away from the front face of the part.

© 2011 CIM-TECH.COM, Inc



The important idea with this knowledge is that for the cut to be in the correct position on the part, the Tip of the drill b
correct X, Y location or else this tool cannot reach the correct point of the cut. When you touch off the tip of the vert
horizontal drill tips must be set there too.

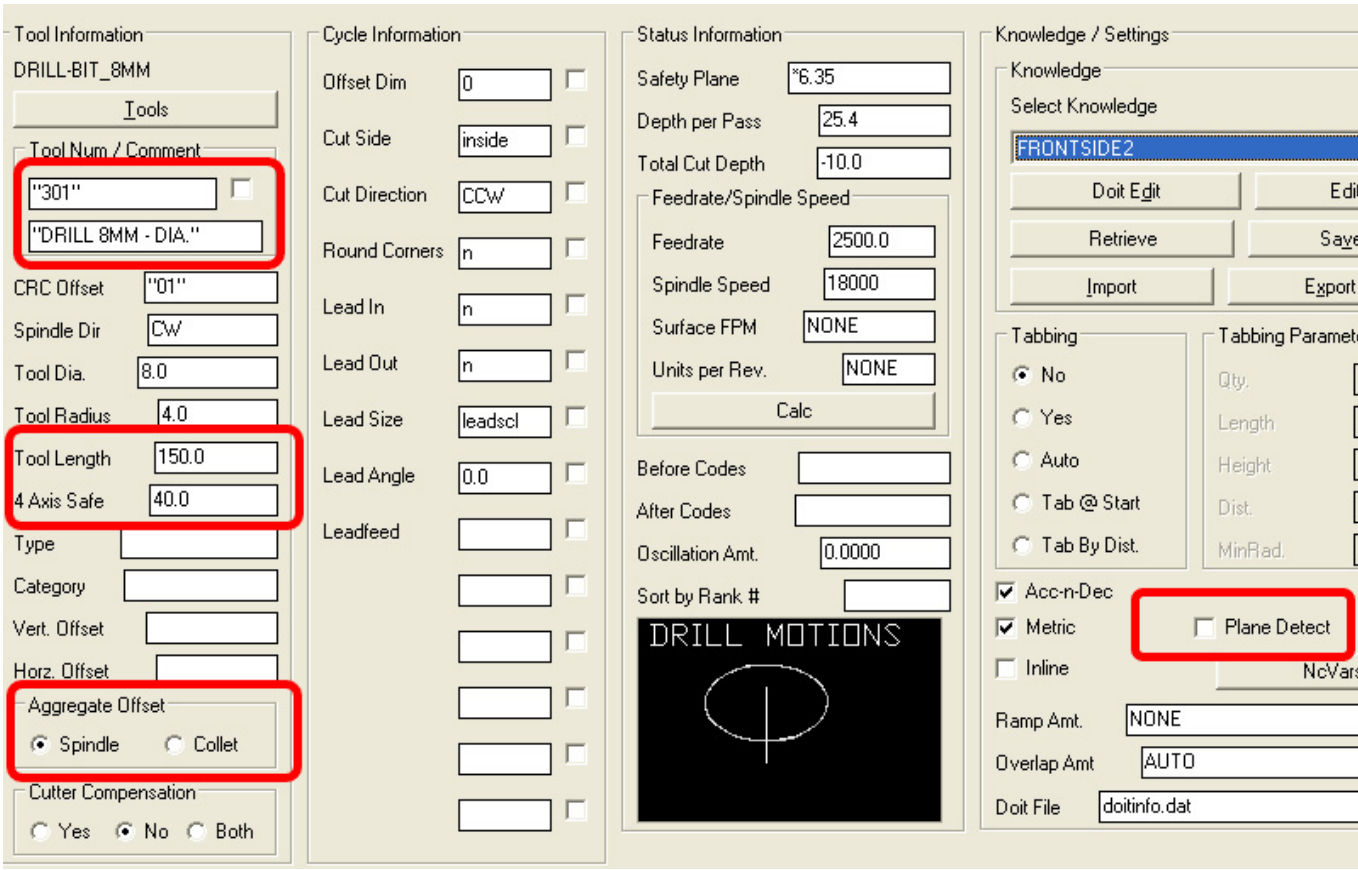
© 2011 CIM-TECH.COM, Inc



FrontSide2

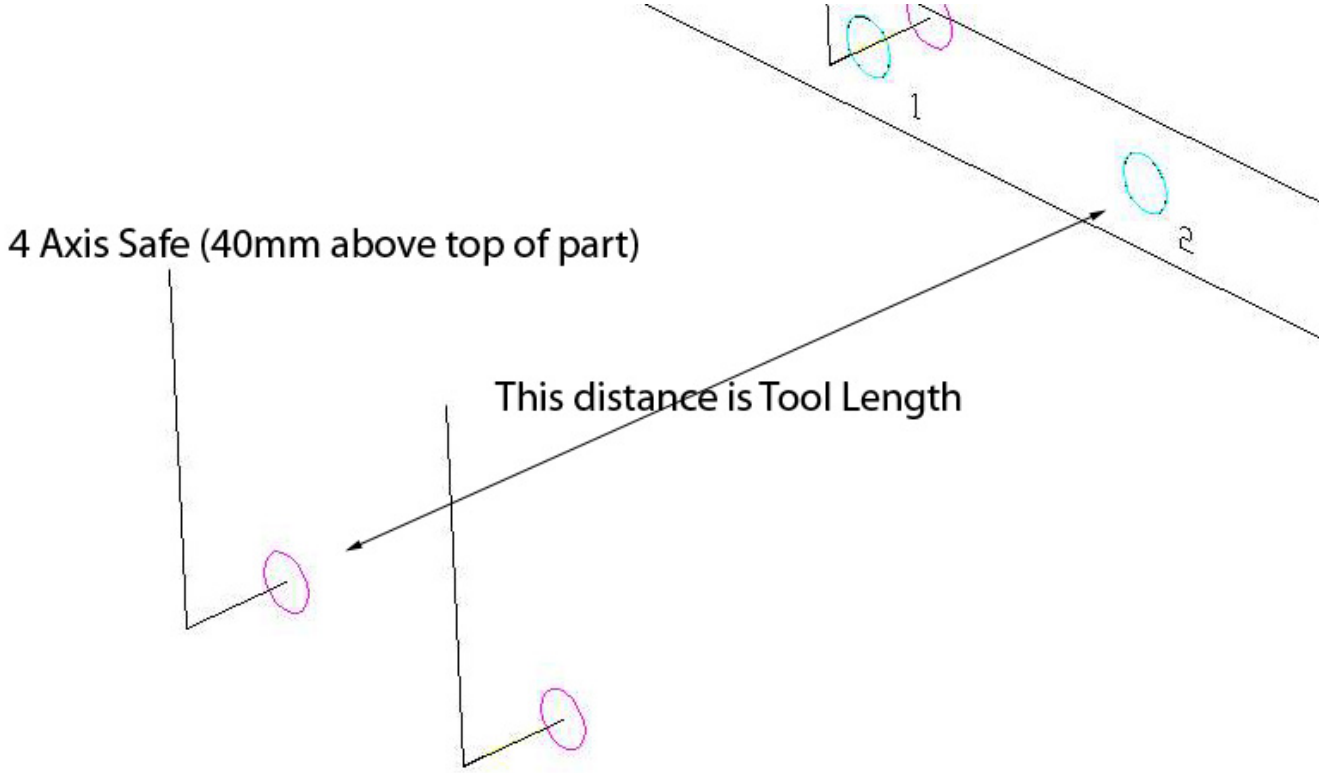
The FrontSide2 knowledge looks a little strange on the screen (it is away from the part) but is actually a little easier t
knowledge are:

© 2011 CIM-TECH.COM, Inc



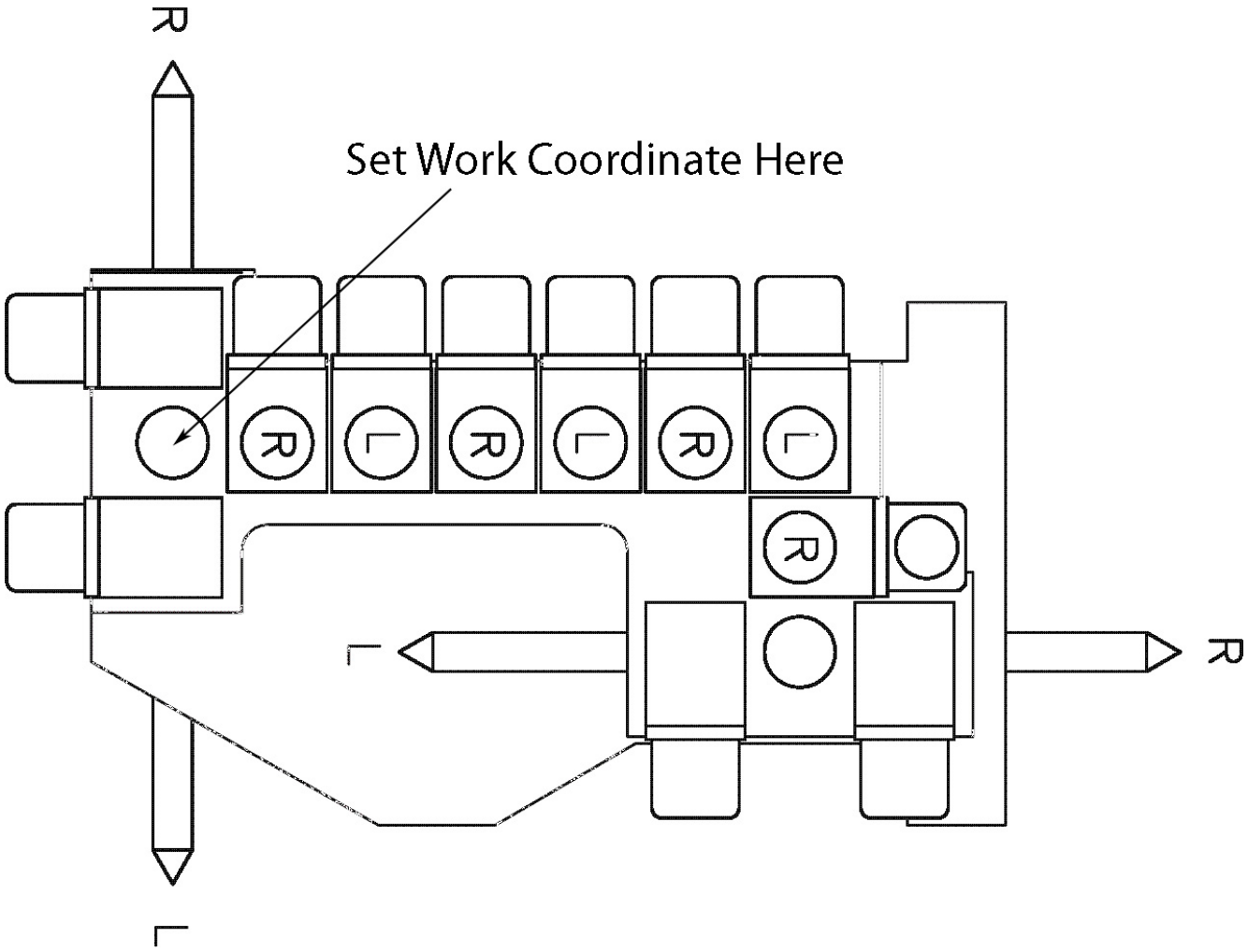
For this knowledge, you use the Tool Length to set the X, Y, offset from the center of the drill spindle and use the Sp
not used for this method either.

© 2011 CIM-TECH.COM, Inc



In order for this to work properly, you must know the distance from the tip of the drill to the center of the drill block lo
drill). This is where the Work Coordinate would normally be set to. Then, put that distance into the Tool Length loc
from the part by that amount so that the TIP of the tool is actually at the right location. This requires no input on the
informing the programmer of the length of the drill from the work coordinate location.

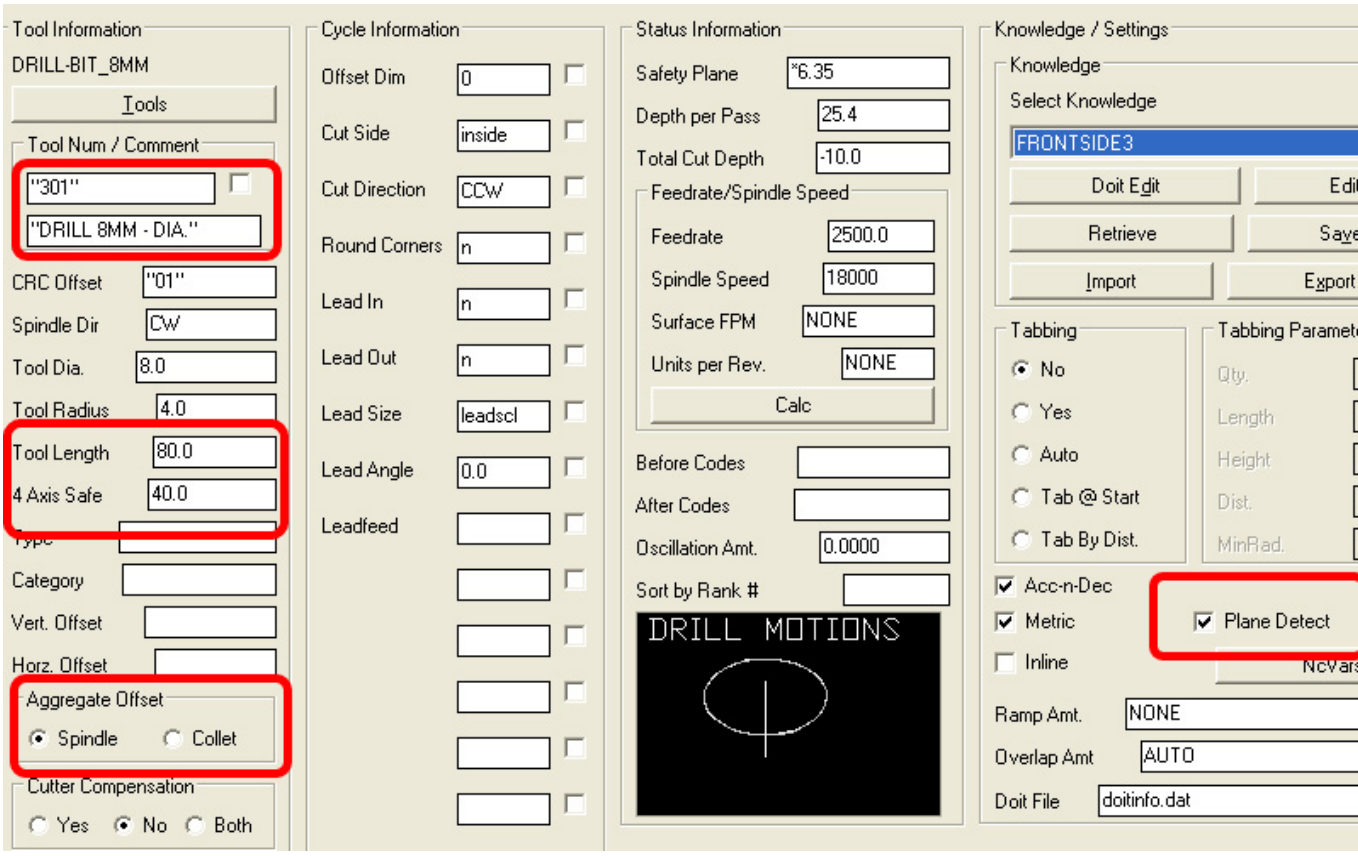
© 2011 CIM-TECH.COM, Inc



FrontSide3

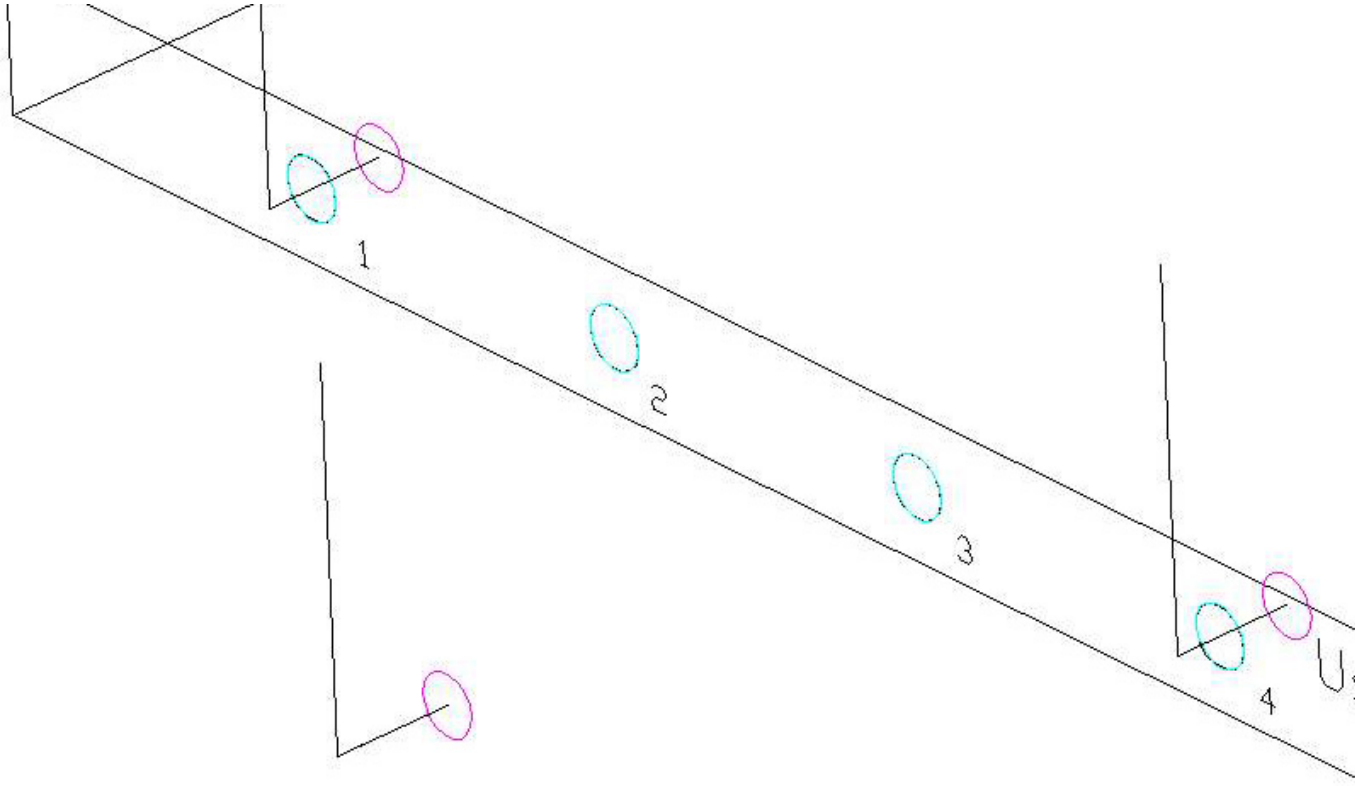
This method is slightly different from the other two in that it will use Plane Detect and the code will be slightly differer
horizontal offset to be input in the machine.

© 2011 CIM-TECH.COM, Inc



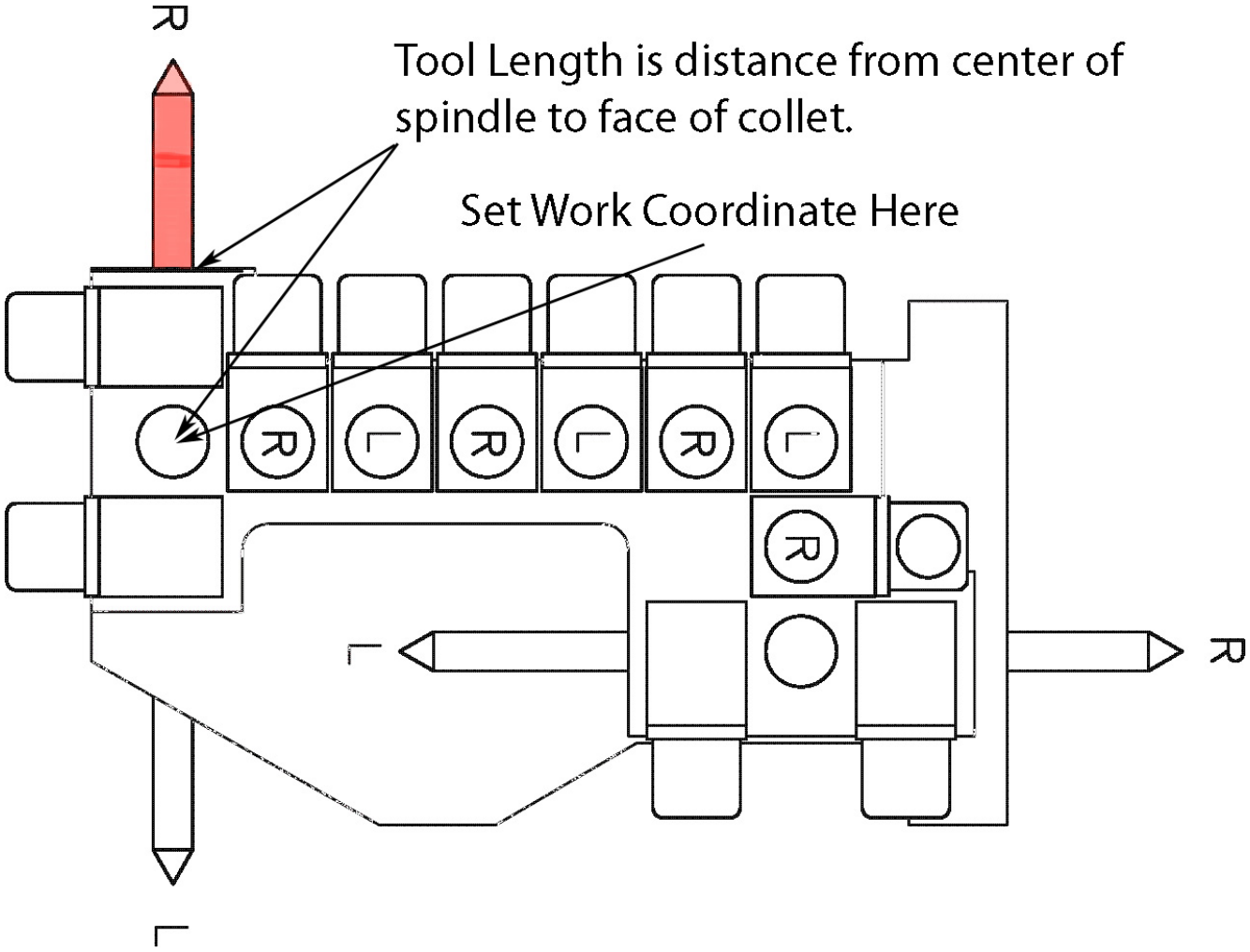
When using the Plane Detect option, Router-CIM will determine what plane the hole is being drilled on and will outpu
offset for the Horizontal Length.
The tool path will look similar to FrontSide2, but the code will be very different.

© 2011 CIM-TECH.COM, Inc



The setting of the Tool Length is now from the center of the spindle to the face of the collet that the drill fits into. The collet can be put in at the machine into a regular offset. This way if the drill length changes, a new program does no updated on the machine and the code run again.

© 2011 CIM-TECH.COM, Inc



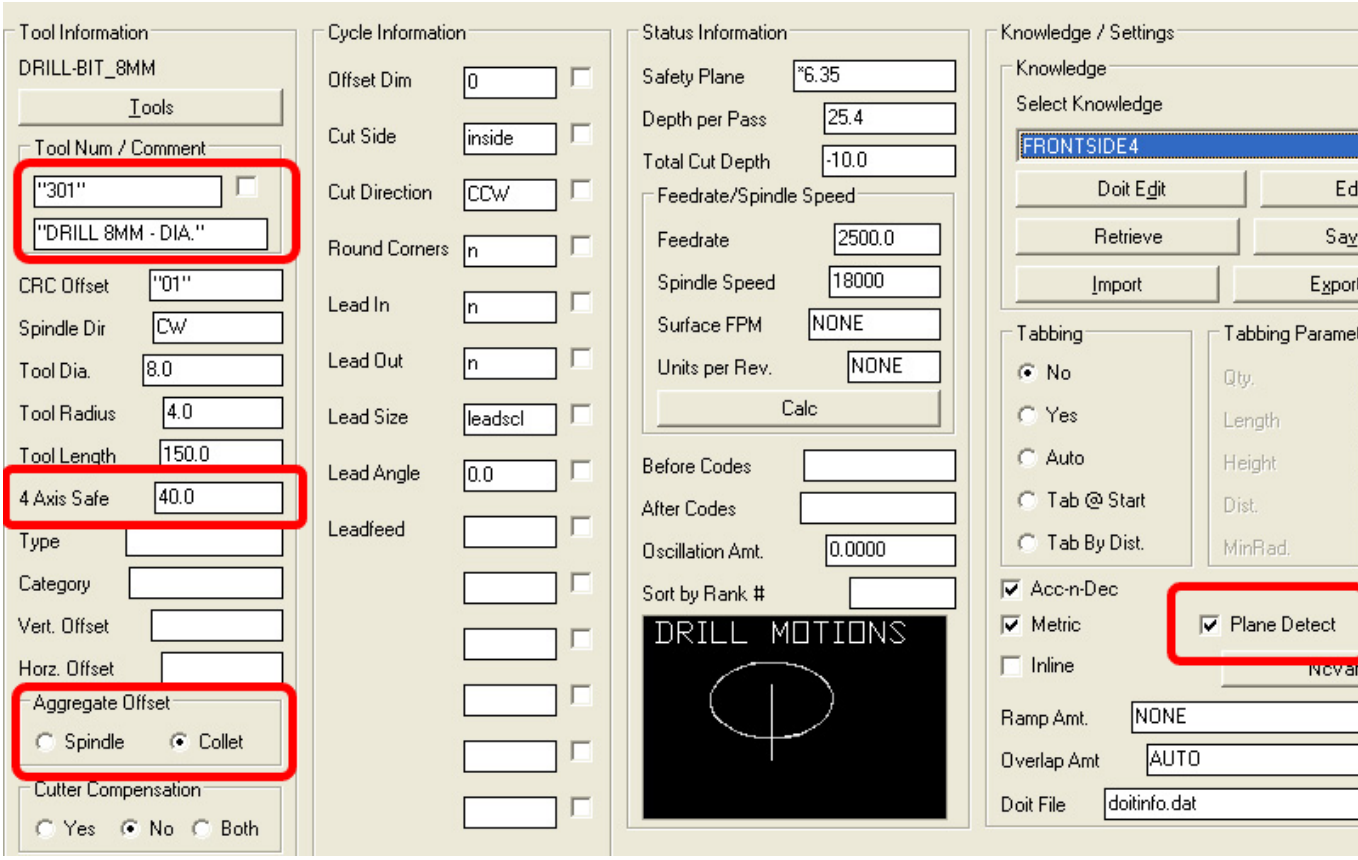
The code generated for this cut would show the normal work coordinate, the Vertical H offset (in this case it is H21) offset for the length of the tool (in this case it is H41).

```
(DRILL 8MM - DIA.)
G28 G91 Z0 M05
G90 T301
G00 G17 G54P1 X150. Y-156.35 M03 S18000
G00 G43 H21 Z59.  <--------------- Vertical Height Set in this offset
G18 G44 H41   <------------------ Horizontal Length Set in this offset
Z9.45
Y-152.54
G01 Y-140.F2500.
G00 Y-156.35
G00 Z59.
H0
```

FrontSide4

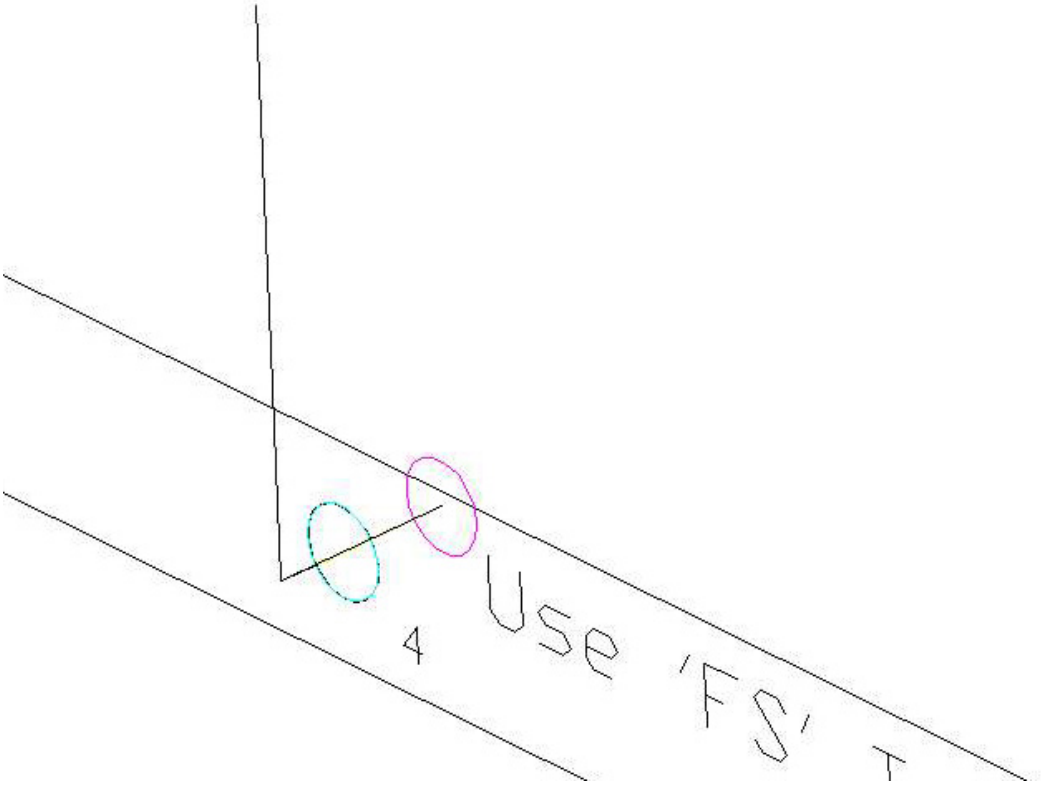
This last knowledge is almost exactly like FrontSide3, except that in since the tool path shows up in the normal cut p Length Offset must be the distance from the tip of the too to the center of the spindle. This is only useful if the same length is a known number for the operator. The programmer then does not need any data about the tool lengths and is in total control over where the tool actually cuts.

© 2011 CIM-TECH.COM, Inc



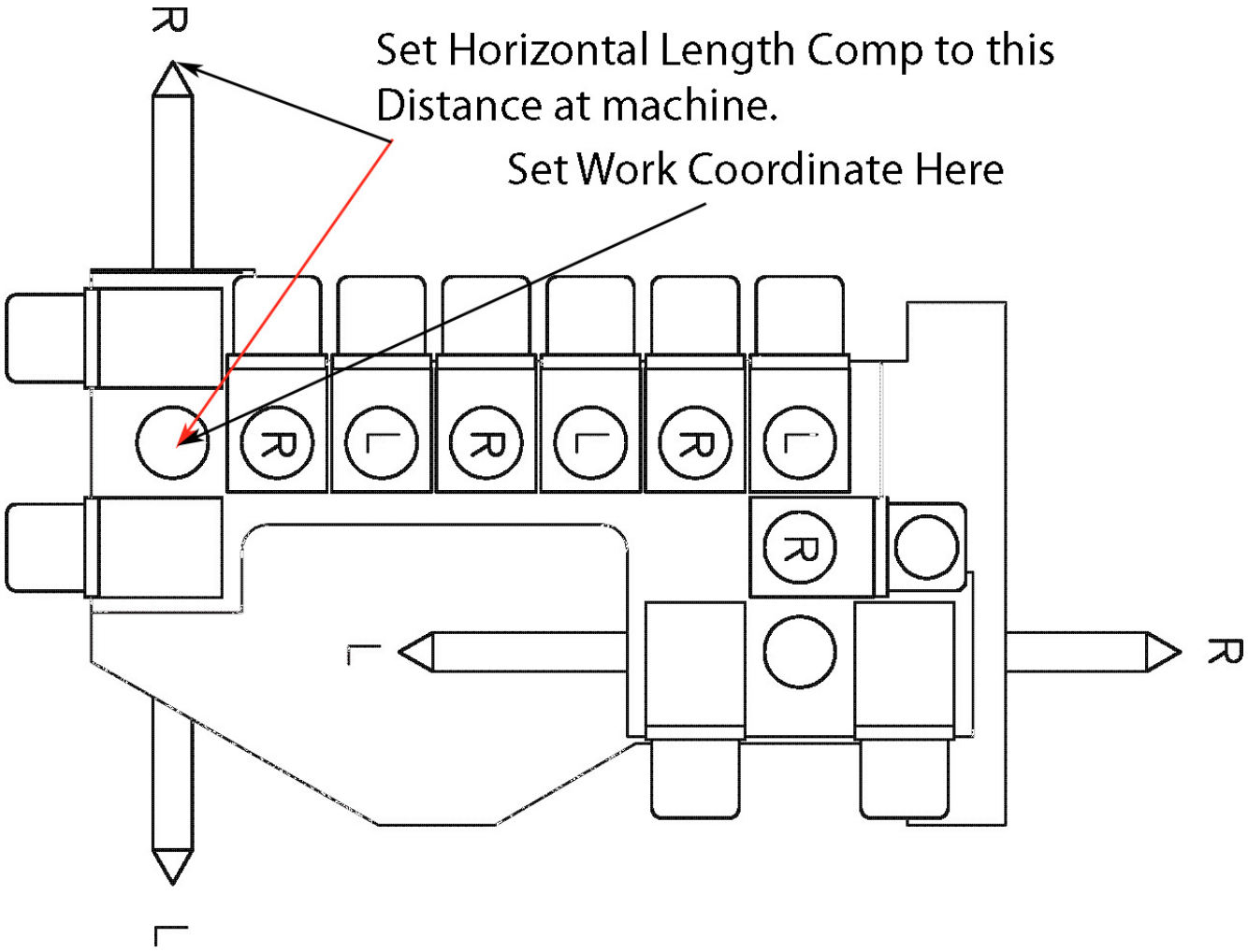
Because the Aggregate Offset is set to Collet, the tool path looks normal:

© 2011 CIM-TECH.COM, Inc



The real trick here is to set the Horizontal offset correctly:

© 2011 CIM-TECH.COM, Inc



The code will look almost exactly like the previous sample, except that the X, Y values will be different because the t

```
(DRILL 8MM - DIA.)
G28 G91 Z0 M05
G90 T301
G00 G17 G54P1 X200. Y-6.35 M03 S18000
G00 G43 H21 Z59.
G18 G44 H41
Z9.45
Y-2.54
G01 Y10. F2500.
G00 Y-6.35
G00 Z59.
H0
```

© 2011 CIM-TECH.COM, Inc

15.4 Using the Sample Drawing (COPY)

Using the Sample Drawing

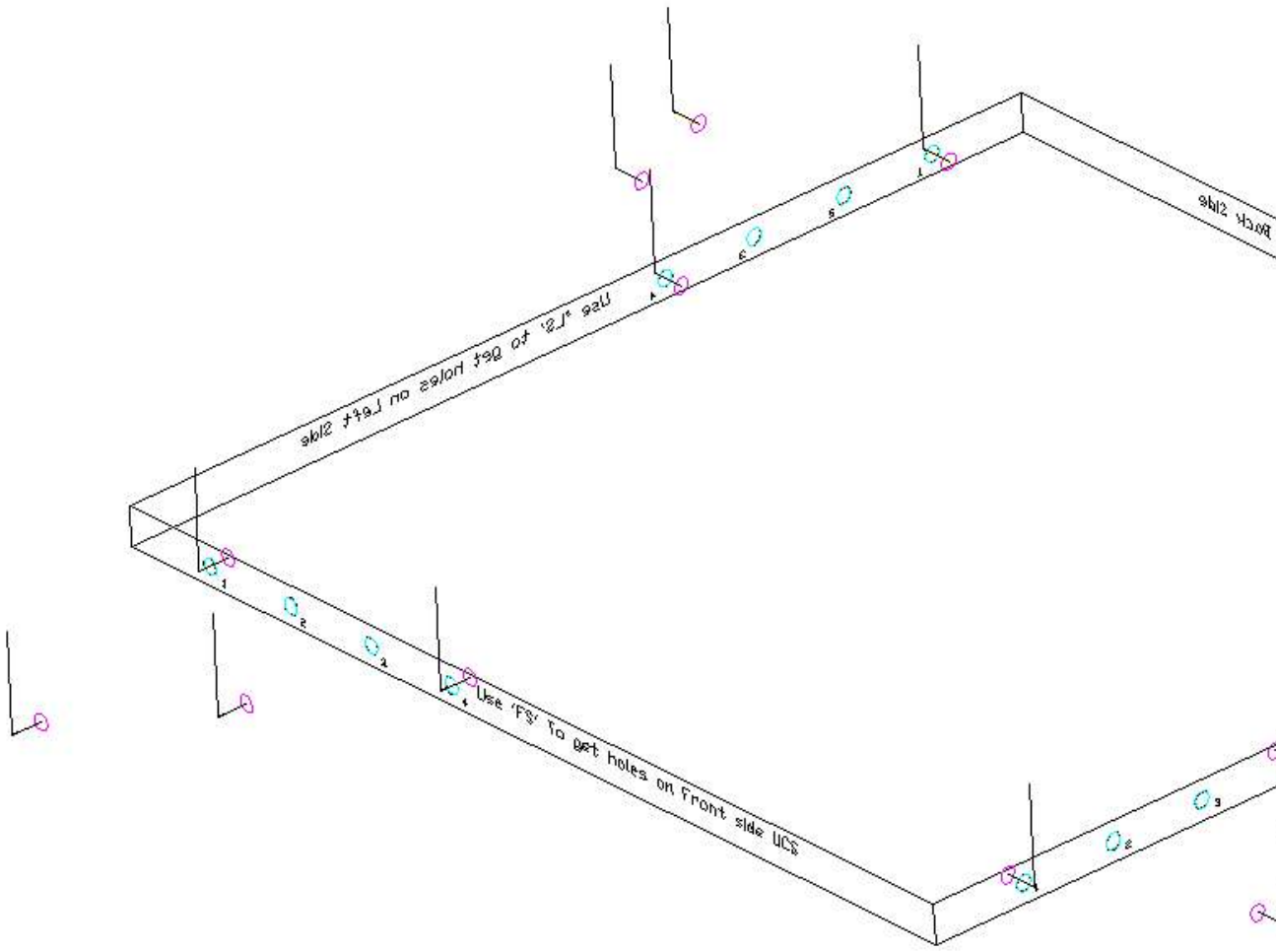
Top Previous

There is a drawing name HorizontalBoringSample1.dwg which contains knowledge for each of the 4 cutting methods or export these sample knowledges into your own drawings.

You can download the drawing here.

The sample has 4 holes drawn on each face, and one cut on each hole. Each shape is numbered to show which cu you are working on.

© 2011 CIM-TECH.COM, Inc



The sample knowledges contained in the drawing are:
FrontSide1
FrontSide2
FrontSide3
FrontSide4
RightSide1
RightSide2
RightSide3
RightSide4
BackSide1

© 2011 CIM-TECH.COM, Inc

BackSide2
BackSide3
BackSide4
LeftSide1
LeftSide2
LeftSide3
LeftSide4

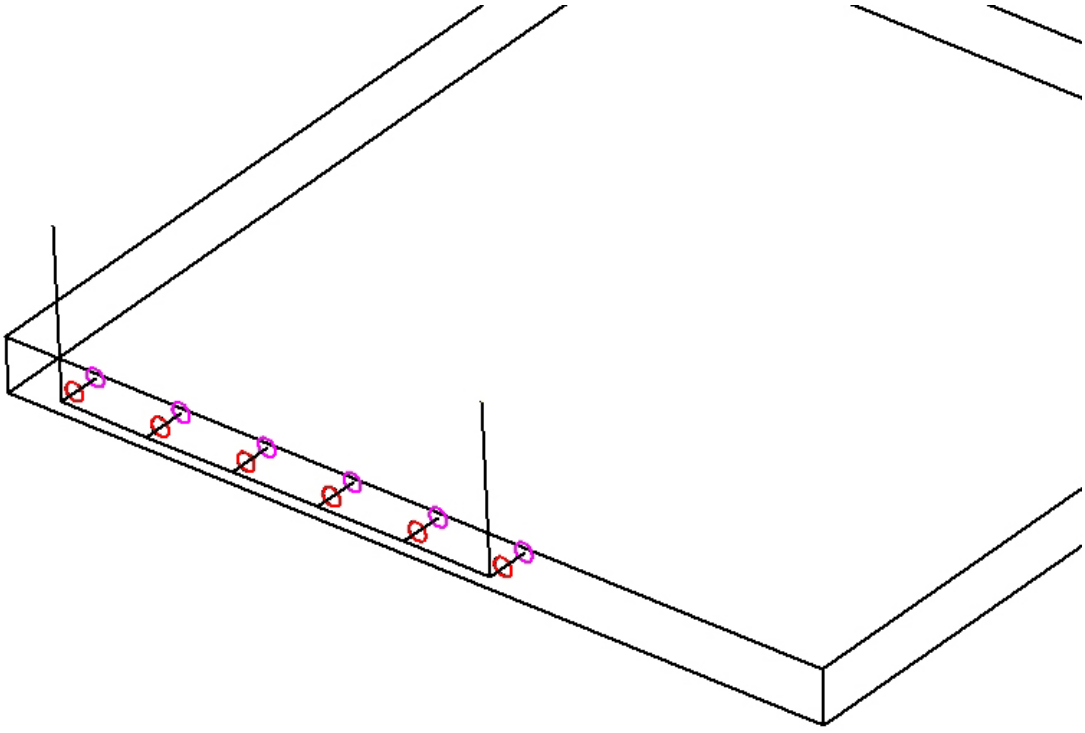
15.5 Avoiding Extra Safety Moves

Each time you make a cut on a horizontal plane, there is move from the safety plane, down to the depth of the cut in Z, then the cut is made in XY and the tool is moved back out of the hole and then back up to the safety plane in Z.

This means that if you were to drill several holes on the side of a shape, you would have the drill moving back up to the safety plane on each hole. This is mainly for safety as sometimes programs are stopped and restarted in locations that could cause a collision if the tool is left down at the hole depth.

If you want to avoid the extra moves in Z, you could simply group the holes on each side of the part after they are geo-shaped and then drill them. This will create one Z move down and leave the head at that Z location and then move between each of the holes, returning to the Z height of the safety plane when the holes are done.

This is how the cycle would look when the holes are drilled:



This would work with either a horizontal drill in a drill block, or an aggregate tool drilling on the side of a part. If Plane Detect is turned on, only one horizontal comp move will be generated as well.

The code would look like this:

© 2011 CIM-TECH.COM, Inc

```
%
:1234(HBORING)
N1G00G17G20G28G40G80G91Z0M5
N2G90
N3G52X0Y0Z0
N4G08P1
N5M08
N6(DRILL .25 DIA.)
N7G53C90.
N8G28G91Z0M05
N9G90T2003M06
N10T102
N11M03S18000
N12G00G17G54P43X1.Y-.25C90.
N13G00G43H3Z2.
N14G18G44H83
N15Z-.375
N16G01Y.375F150.
N17G00Y-.25
N18X2.2598
N19G01Y.375F150.
N20G00Y-.25
N21X3.5196
N22G01Y.375F150.
N23G00Y-.25
N24X4.7794
N25G01Y.375F150.
N26G00Y-.25
N27X6.0392
N28G01Y.375F150.
N29G00Y-.25
N30X7.299
N31G01Y.375F150.
N32G00Y-.25
N33G00Z2.
N34H0
N35G53C90.
N36G28G91Z0M5
N37G28G91X0M09
N38G90
N39G52X0Y0Z0
N40G08P0
N41M30
%
```

© 2011 CIM-TECH.COM, Inc

© 2011 CIM-TECH.COM, Inc

Index

- * -

JOBID 505
NCFIL 505
NCFILEXT 505
NCFILLOC 505
pat_fuzz 240
POST 505
VIEWCODE 505

- _ -

_ACC_ANGDEV 508
_ACC_CLAMPFEED 508
_ACC_CLAMPMIN 508
_ACC_CLAMPRADIUS 508
_ACC_CROSSGRAINPER 508
_ACC_DEC 505
_ACC_DECDIST 508
_ACC_DECFEED 508
_ACC_GRAINDIR 508

- 3 -

3DPOLY CEN 105

- 4 -

4 AXIS CUTTING 105
4 Axis Safe 58
4 AXIS SAW 105

- A -

Add 43
Aggregate Offset 59
Angle of Grain Direction 508
Area Only 487
Average Deceleration Distance 508

- B -

Block Sequence 501
BS 589

- C -

CCW 435
CENTER-LINE CUT 105
CENTER-LINE RAMP 105
Chip Break Height 250
Chip Clear Increment 250
CleanUp Pass 290, 294
Clear All 485
Climb Cutting 435
Climb Mill 290, 294
Climb Milling 435
Clockwise 435
Closest Point 487
Collect 475
Collet 59
Combination of Spindle and Collet methods 59
Comment 52
Configuration Wizard 27
Conventional Cutting 435
Conventional Milling 435
Counter-Clockwise 435
CRC Offset 53
Create Knowledge Base 31
Creating a new Tool List 80
Critical Issues 12
Cross Grain Percentage 508
Crossing Geometry: 48
Customizing the Knowledge Base 38
Cut Direction 116, 122
Cut Side 116, 122
Cutter Comp BOTH 77
Cutter Comp NO 77
Cutter Comp YES 77
Cutter Compensation 77
Cutting Cycles 101
CW 435

- D -

DEEP BORE 105

© 2011 CIM-TECH.COM, Inc

Default Knowledge Base 34
Defining Knowledge 517
Delete Knowledge Base 32
Depth Per Pass 116, 122
DESCRIBE TOOL 98
Develop a Cut Sequence 485
Develop Tooling Motions 501
Diameter 54
DO-IT 105
Draw 43
DRILL MOTIONS 105
DRILL-INTERPOLATION 105

- E -

Edit Group 43
Edit Knowledge 526
Edit Tooling Motions 505
Emulate 482
EMULT. BLOCK NAME 98
EMULT. XY SCALE 98
Endpoint 482
ENTER ITEM NAME 98
Enter Surface Tolerance 381
ENTER TABLE NAME 98
Erasecheck 484
Exporting Knowledge 521

- F -

Fall Over Mode 0 359
Fall Over Mode 1 359
Fall Over Mode 2 359
Fall Over Mode 3 359
Feed Change 250
FEED DISTANCE TO MAT'L 98
Feedrate 116, 122
Finish Allow 290, 294
FS 589

- G -

GEOMETRIC VARIABLES 542
Geometry 40
Geoshape 40
Group 40, 43
Group Holes 43
Group Knowledge 532
Group Shapes 43

- H -

HELICAL ARCS CENTER 105
HELICAL ARCS EDGE 105
HELI-LEAD-CENTER 105
HELI-LEAD-I 105
HELI-LEAD-O 105
HOLE-INTERP 1 105
HOLE-INTERP 2 105
horizontal boring 588, 589
Horizontal Offset 59
How Acc-n-Dec works 508

- I -

Import Knowledge Base 31
Importing Knowledge 524
Impossible Offsets: 48
Installation Quick Start 12

- K -

Knowledge 592
Knowledge Notes 537

- L -

LAYER VARIABLES 543
Lead Angle 116, 122
Lead Feed 116, 122
Lead In 116, 122
Lead Out 116, 122
Lead Size 116, 122
LINEAR POCKET 105
LINEAR POCKET RAMP IN 105
LINEAR POCKET UNIDIRECTIONAL 105
LINEAR POCKET ZIGZAG 105
LS 589

- M -

Make NC Code 505
Make Subroutines 501

© 2011 CIM-TECH.COM, Inc

Mass Feedrate 508
Max Offset 456
Maximum Corner Feedrate 508
Maximum Deceleration Radius 508
Measure 482
Minimum Corner Angle Deviation 508
MOULDER HEAD 105
Multi-Pass Inside 105
Multi-Pass Outside 105

- N -

NCVAR 505
New Chip Break 250
New Clear Increment 250
New Feedrate 250
New Peck Increment 250
NURBS XZ 381

- O -

Offset Dim 56, 116, 122
Open Shape Issues 42
OPEN SHAPE LEFT 105
OPEN SHAPE RIGHT 105
OTHER VARIABLES 546
Overlap Amt 116, 122

- P -

PATTERN RECOGNITION 105
PECK BORE 105
Peck Change 250
Perform Machine Calc 505
Permanent Editing 80
Plane Detect 66, 74
PLUNGE-I 105
Plunge-Inside 116
PLUNGE-O 105
PLUNGE-O LINE LEADS 105
Product Overview 11

- R -

Radius 56
Radius Feedrate Clamp 508
Ramp Angle 290, 294
RAMP-I 105
Ramp-IN 290, 294
RAMP-O 105
Rank Sort, Area 487
Rank Sort, Area, Closest Point 487
Rank, Tool Sort, Area 487
Rank, Tool Sort, Closest Point 487
RCIM Group Toolbar 43
Remove 43
Report Information 505
Restore 475
Retrieve Knowledge 518
Rough Angle 290, 294
Round Corners 116, 122
RS 589

- S -

Safety Plane 116, 122
Save Knowledge 520
Save Knowledge Base 32
Save Shape 290, 294
Save Without Knowledge 536
Saving Knowledge 520
Seq_Sort -- Area Only 487
Seq_Sort -- CW First, Zig Zag Increasing X and Y 487
Seq_Sort -- Rank Only 487
Seq_Sort -- Start End Point 487
Seq_Sort -- Zigzag Decreasing X, Decreasing Y 487
Seq_Sort -- Zigzag Decreasing X, Increasing Y 487
Seq_Sort -- Zigzag Decreasing Y, Decreasing X 487
Seq_Sort -- Zigzag Decreasing Y, Increasing X 487
Seq_Sort -- Zigzag Increasing X, Decreasing Y 487
Seq_Sort -- Zigzag Increasing X, Increasing Y 487
Seq_Sort -- Zigzag Increasing Y, Decreasing X 487
Seq_Sort -- Zigzag Increasing Y, Increasing X 487
Show Wizard 35
SLICE ZIG-ZAG 105
Sort 43, 487
Sort Cuts By 485
Spindle 59
Spindle Direction 54

© 2011 CIM-TECH.COM, Inc

Spindle Speed 116, 122
Spiral – OUT 290, 294
SPIRAL POCKET 105
SPIRAL POCKET IN-OUT 105
SPIRAL POCKET OUT-IN 105
SPIRAL POCKET RAMP IN 105
Spiral-IN 290, 294
SPOT BORE 105
Start Corner 290, 294
Start Point Edit 40, 41
Start Point Edit (Start Pt) 41
Stay Down 290, 294
Step Over % 290, 294
Surf Intersect 367
Surf Intersect at 90 Degrees 367
SYSTEM VARIABLES 544

- T -

Temporary Editing 80
Tool Category 59
TOOL CODE 98
TOOL CUT DEPTH 98
TOOL DIAMETER 98
TOOL LENGTH 57, 98
TOOL NUMBER 52, 98
TOOL OFFSET NUMBER 98
TOOL RADIUS 98
TOOL ROTATION 98
Tool Sort, Area 487
Tool Sort, Area, Closest Point 487
Tool Sort, Closest Point 487
Tool Sort, Rank, Closest Point 487
TOOL TIP DIST TO ZERO 98
Tool Type 59
Toolcheck 482
Total Cut Depth 116, 122
Troubleshooting Geometry 48

- U -

UCS 589

- V -

Vertical Roughing Mode 1 372
Vertical Roughing Mode 2 375
Vertical Rouging Mode 3 378

- X -

XY Stock Allowance 116

- Z -

Z Level Completion 391
Z Stock Allowance 116
Zig 290
Zig-Zag 290, 294
Zigzag Decreasing X, Decreasing Y 487
Zigzag Decreasing X, Increasing Y 487
Zigzag Decreasing Y, Decreasing X 487
Zigzag Decreasing Y, Increasing X 487
Zigzag Increasing X, Decreasing Y 487
Zigzag Increasing X, Increasing Y 487
Zigzag Increasing Y, Decreasing X 487
Zigzag Increasing Y, Increasing X 487

© 2011 CIM-TECH.COM, Inc



CIM-TECH, Inc.
7512 Dr. Phillips Blvd.
Suite 50-859
Orlando, FL 32819

(877) 549-8211