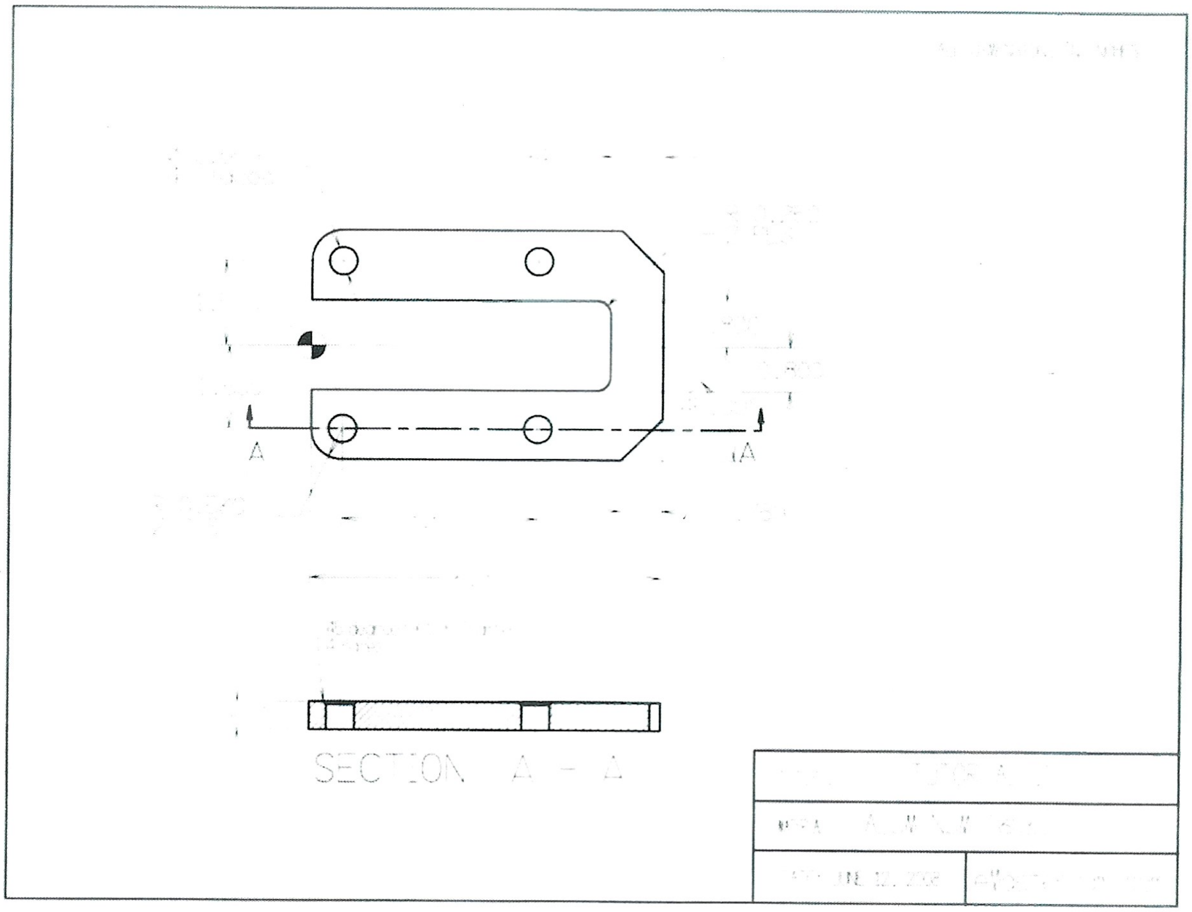
Mastercam 2020 Mill Essentials tutorial (ME363)

Milling Part preview

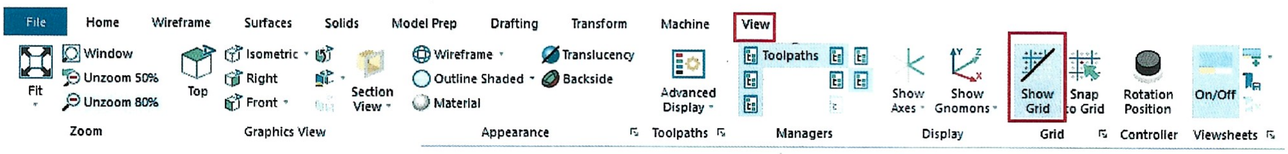


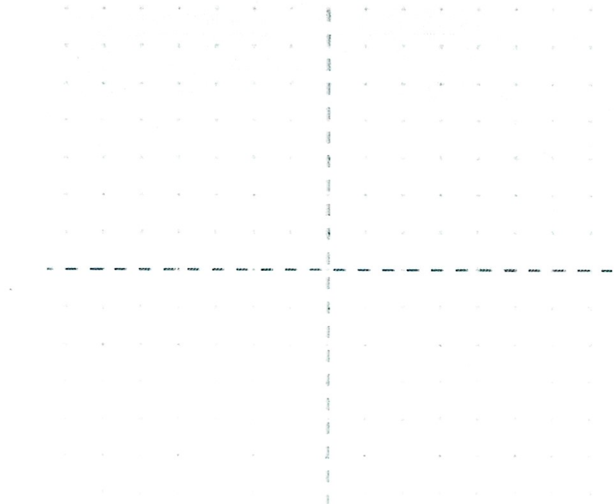
Step 0: Setting the grid

Before starting the geometry enable the Grid. It will show you where the part origin is.

View

- From the grid group, select show grid

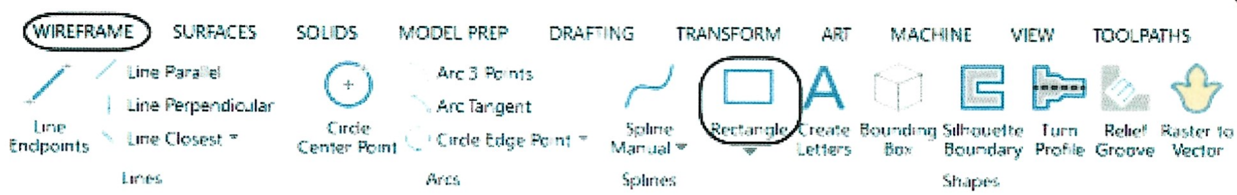




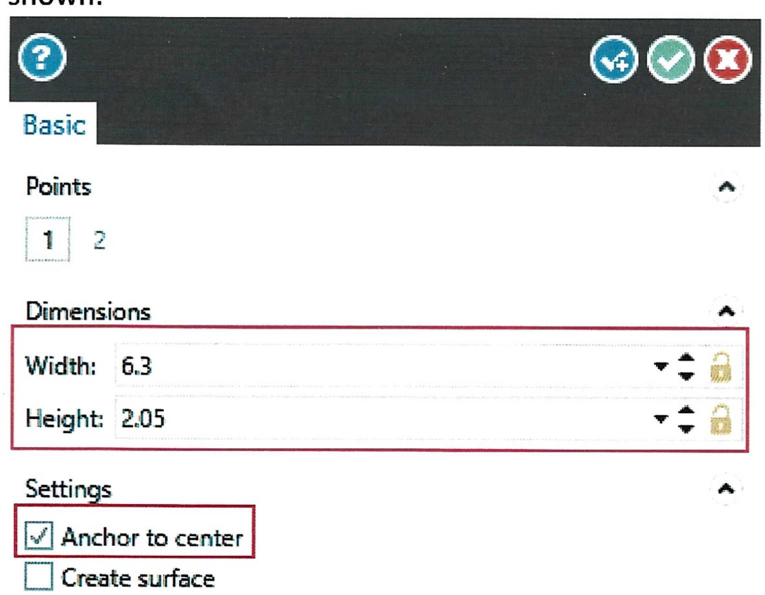
Step 1: Create a rectangle knowing the width, height and base point

Wireframe

- From the shapes group, select rectangle

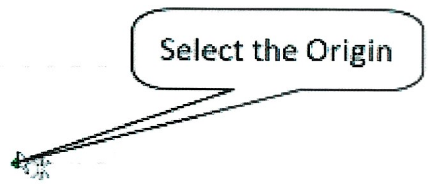


- In the rectangle panel, enter the Width and Height and enable Anchor to center as shown.

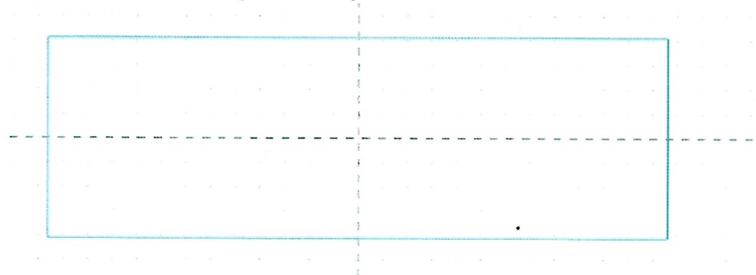


Note: Make sure that Create surface is not selected. Anchor to center sets the base point of the rectangle to its center and draws the rectangle outward from the center.

- Select the position of the base point as shown



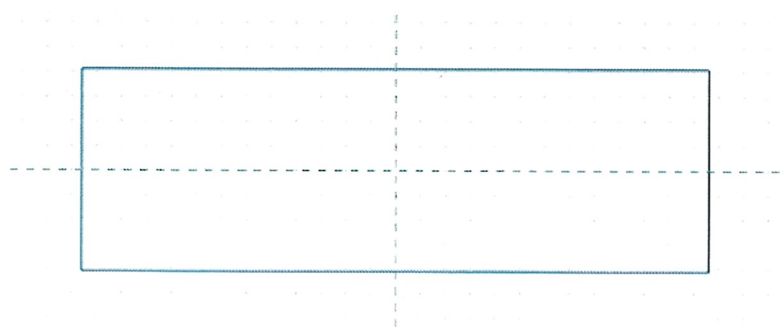
- A preview of the geometry should look as shown.



Note: The geometry should appear in cyan blue color which is the color for the live entities. While the rectangle is live, you can adjust the dimensions or select a new base point.

- Select the OK button to exit the Rectangle command. 

- The geometry should look as shown.

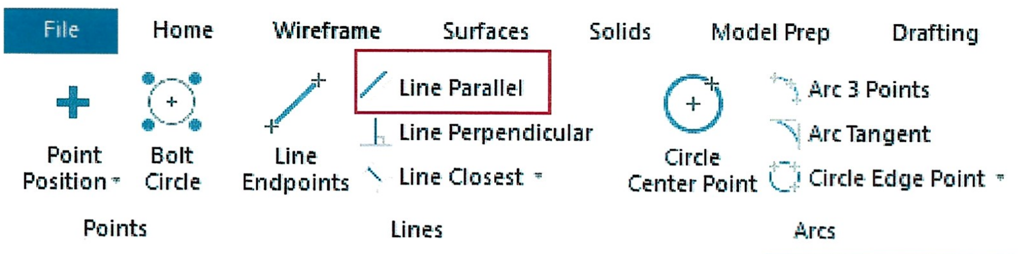


Note: While creating geometry for this tutorial, if you make a mistake, you can undo the last step using the Undo icon . You can undo as many steps as needed. If you delete or undo a step by mistake, just use the Redo icon . To delete unwanted geometry, select the geometry first and then press Delete from the keyboard. To zoom or un-zoom, move the cursor in the center of the geometry and scroll up or down the mouse wheel.

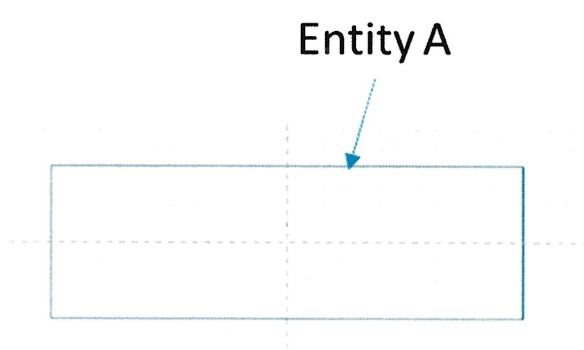
Step 2: Create the inside geometry

Wireframe

- From the lines group, select line parallel



- Select Entity A

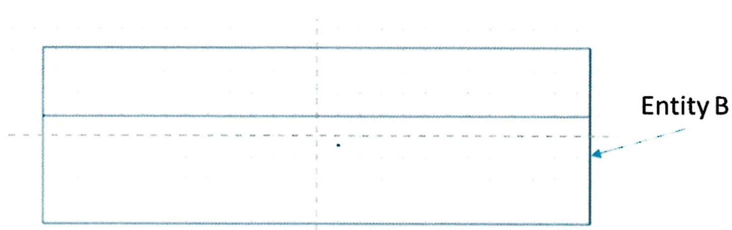


- [Select the point to place a parallel line through]: Pick a point below the selected line. Note: that the color of the geometry is cyan which means that the entity is "live" and you can still change
- Enter the Offset Distance 0.8 {Press Enter).

  Note: that to continue using the same command you have to select the Apply button  or press Enter.

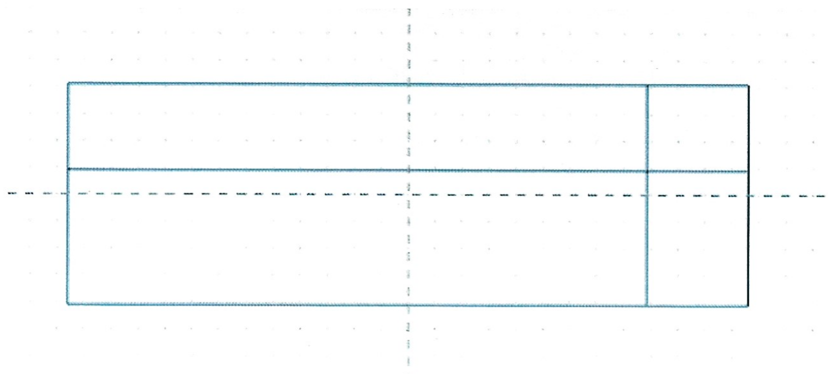
  To exit the command you can either start a new command or select the OK button .

- Select the Apply button to continue.
- [Select a line): Select Entity B.



- [Select the point to place a parallel line through]: Pick a point left to the selected line.
- Type the Offset Distance 0.94 (Enter)
- Select the OK button .

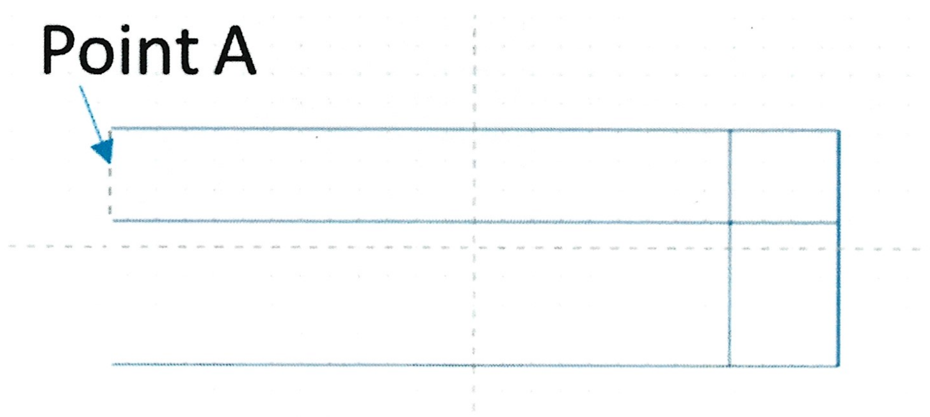
The drawing should look as shown.



Step 2: Trimming the lines.

Wireframe

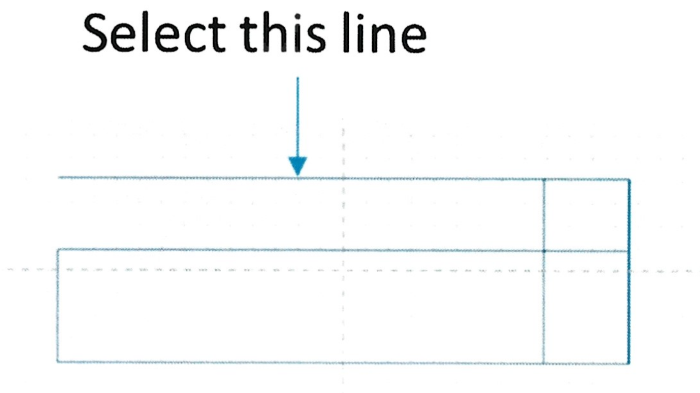
- From the Modify group, select Divide Divide .
- [Select the entity to trim/extend]: Select the entity at Point A (select all entities exactly as shown in the drawing).
  Note: that the dashed line represents what will be removed.



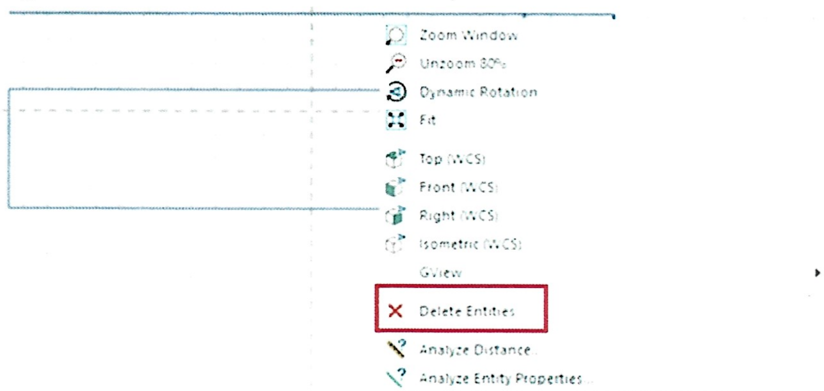
- Select the OK button .

Step 3: Deleting a line.

- Select the horizontal line as shown.



- Right click and select the delete entity icon.

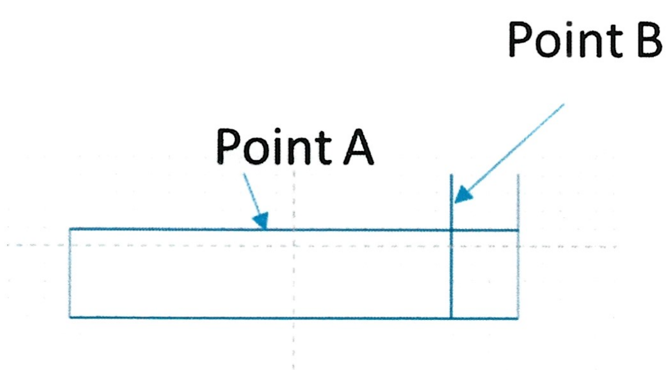


Step 4: Create the 0.25 radius fillet.

Wireframe

- From the Modify group, select Fillet Entities 

.



- Enter the fillet Radius 0.25.
- [Select an entity]: Select Point A.
- [Select another entity]: Select Point 8.

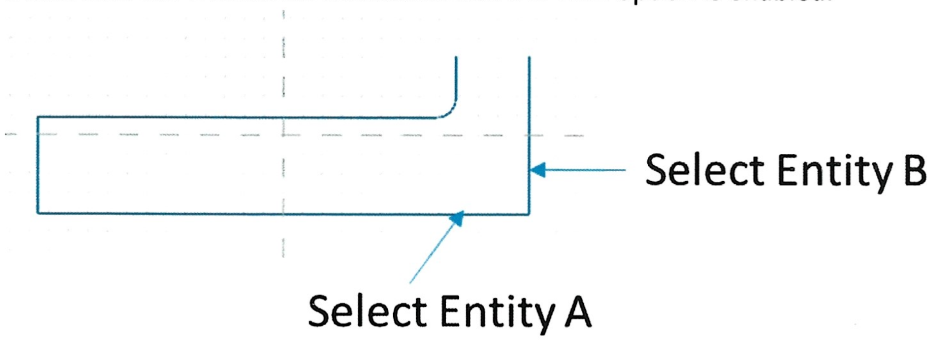
- Select the OK button .

Step 5: Complete the outside profile (Create the 45 degree chamfer)

Wireframe

- From the Modify group, select Chamfer Entities 

.
- Make sure the 1 Distance is selected and the Trim option is enabled.



- Enter the Distance 1 0.75 (Press Enter).
- [Select line or arc]: Select Entity A.
- [Select line or arc]: Select Entity B.

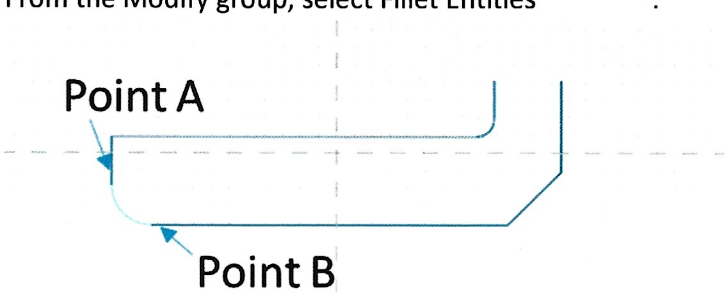
- Select the OK button  to exit chamfer command.

Step 6: Complete the outside profile (Create the 0.56 radius fillet)

Wireframe

- From the Modify group, select Fillet Entities 

.

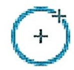


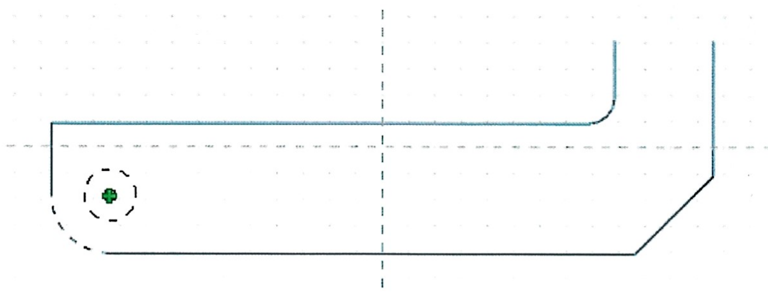
- Enter the fillet Radius 0.56.

- [Select an entity]: Select Point A.
- [Select another entity]: Select Point B.
- Select the OK button .

Step 7: Create the two 0.5" diameter circles knowing the center point and the diameter

Wireframe

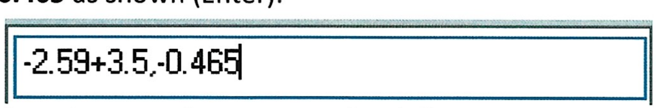
- From the Arcs group, select Circle Center Point  Circle Center Point .
- Enter the Diameter value 0.5 (Enter).
- [Enter the center point]: Mover the cursor to the center location of the 0.56 radius fillet. Select the point when the circle center icon appears.



- Select the Apply button .
- Enter the Diameter value 0.5 (Enter).

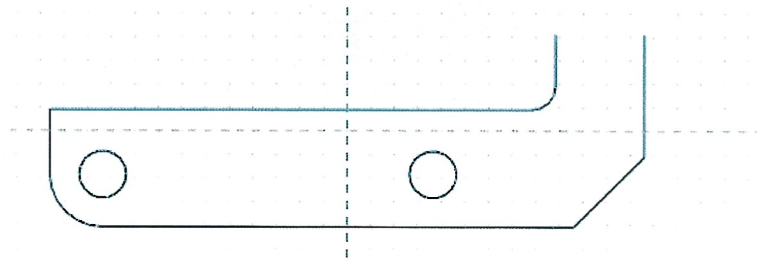
[Enter the center point]: Select the **AutoCursor Fast Point** icon from the **General Selection** toolbar and the field where you can type the coordinates will open at the upper left side of the graphics window as shown.

- Type **-2.59+3.5, -0.465** as shown (Enter).



Note: Mastercam will perform basic math operations (+,-,*,/). You can enter the values without any of the corresponding coordinate letters (X, Y, Z) as long as you enter them in this order and separate them by commas.

- Select the OK button .

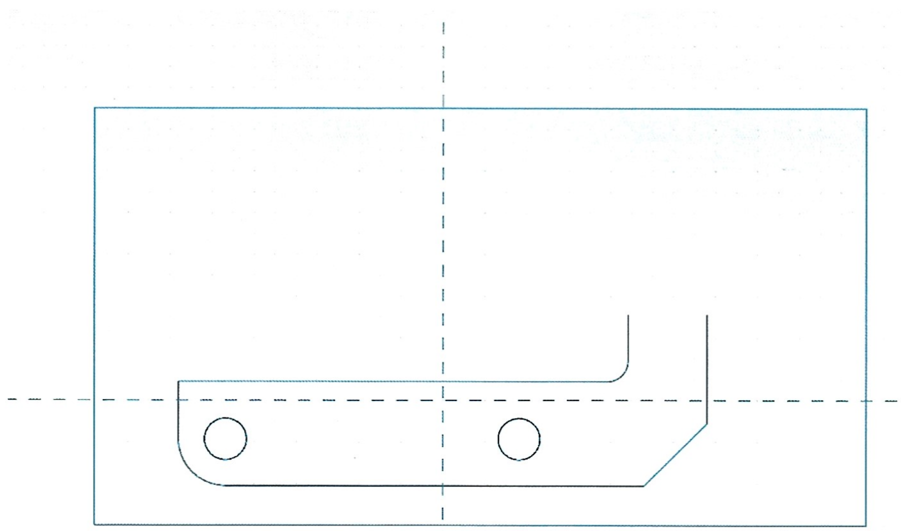


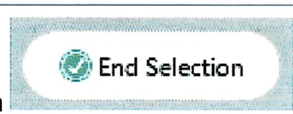
Step 8: Transform the geometry to represent the whole part

Transform

- From the Position group, select Mirror 

.
- Select all entities as shown below



- Click End selection 

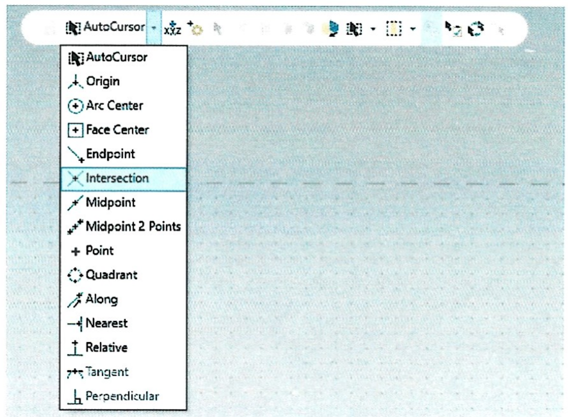
.
- Input Y offset of 1.025.
- Select the OK button .

Step 9: Move to origin

This command allows you to quick move the part to have the origin in the same location that you will set the part zero at the machine. In our case we will move the lower left corner to the origin.

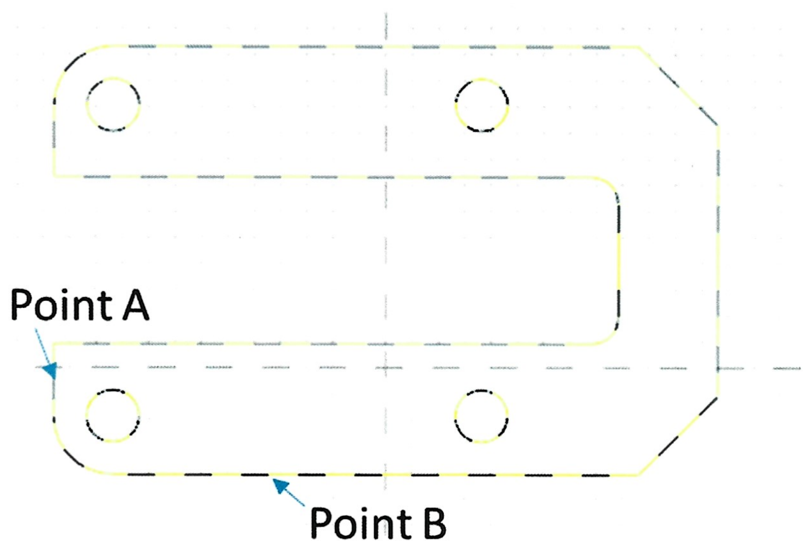
* From the Position group, select Move to Origin 

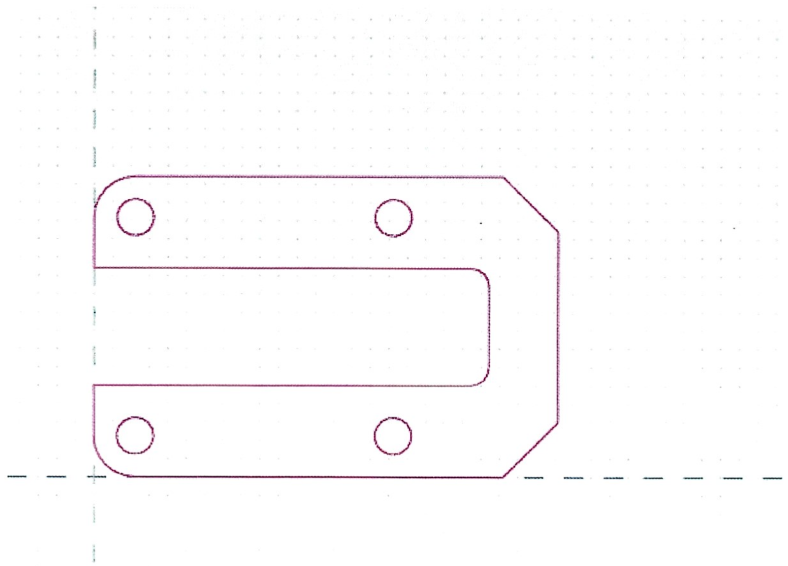
.
* [Select the point to translate from]: Select the drop-down arrow next to the AutoCursor's override and select from the list Intersection as shown.



Note that the AutoCursor Override list allows you to choose only one type of point to detect and snap to. In our case we do not have an intersection point to snap to, so we will let Mastercam determine the intersection point of two existing lines.

* Select the two lines as shown.

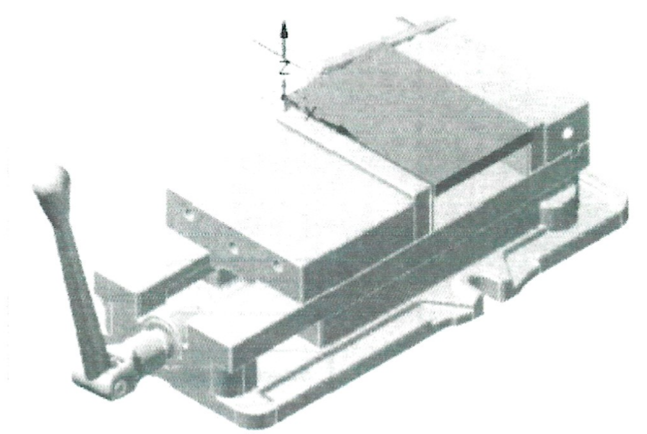




Step 10: Save the file

- Go to file and click save as.
- File name: "Your Name_2"

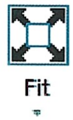
SUGGESTED FIXTURE FOR SETUP 1:



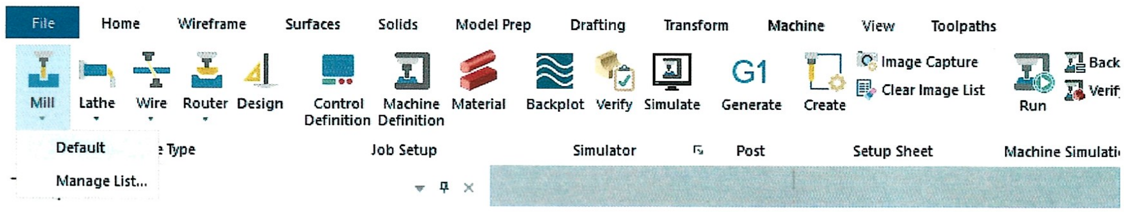
Step 11: Select the machine and set up the stock to be machined.

In Mastercam, you select a Machine Definition before creating any toolpaths. The Machine Definition is a model of your machine tool's capabilities and features and acts like a template for setting up machining jobs. The machine definition ties together three main components: the schematic model of your machine tool's components, the control definition that models your control unit's capabilities and the post processor that will generate the required machine code (G-code).

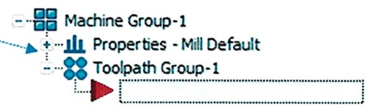
- To display the Toolpaths Manager press Alt+ 0.

- Use the Fit icon  from the View tab to fit the drawing to the screen.

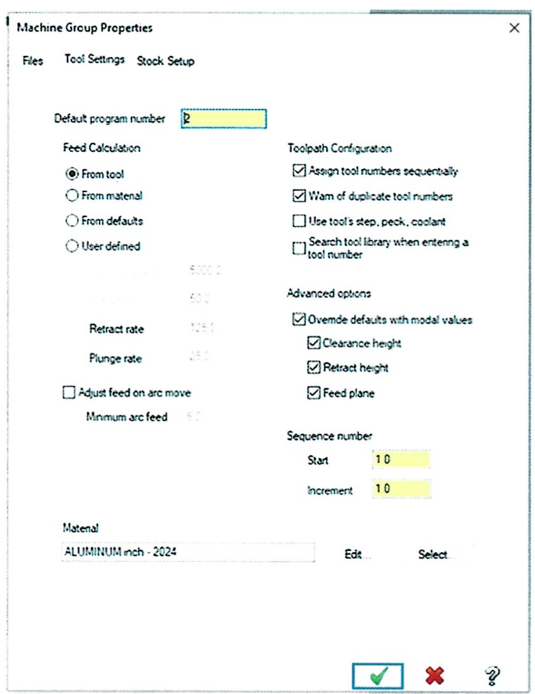
- Select the machine tab and click the Default mill.



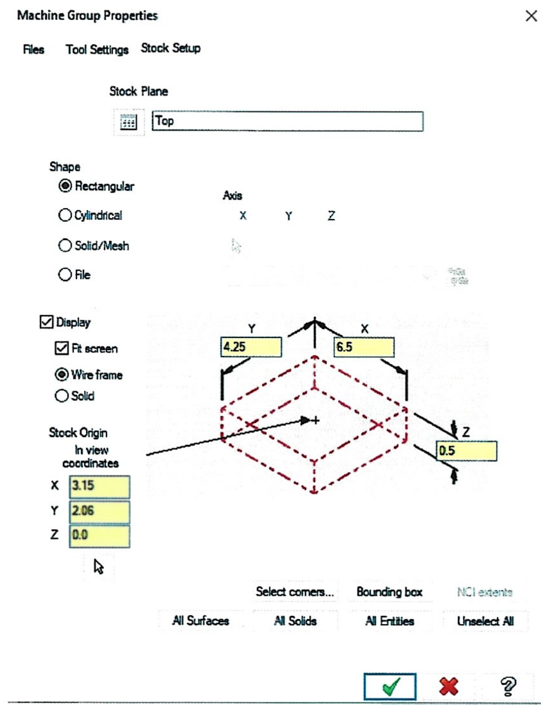
- Select the plus sign in front of Properties to expand the Toolpaths Group Properties

Select the plus 

- Select **Tool Settings** and change the parameters to match the screenshot below.
  Assign tool numbers sequentially allow you to overwrite the tool number from the library with the next available tool number. (First operation ->tool number 1; Second operation ->tool number 2, etc.)
  Warn of duplicate tool numbers allows you to get a warning if you enter two tools with the same number.
  Override defaults with modal values enable the system to keep the values that you enter.
  Feed Calculation set From tool uses feed rate, plunge rate, retract rate and spindle speed from the tool definition.



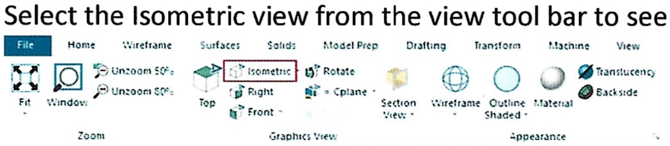
- Select Stock setup tab and define the stock by setting the stock shape (Rectangular) and entering the stock dimensions as shown in the screenshot below.



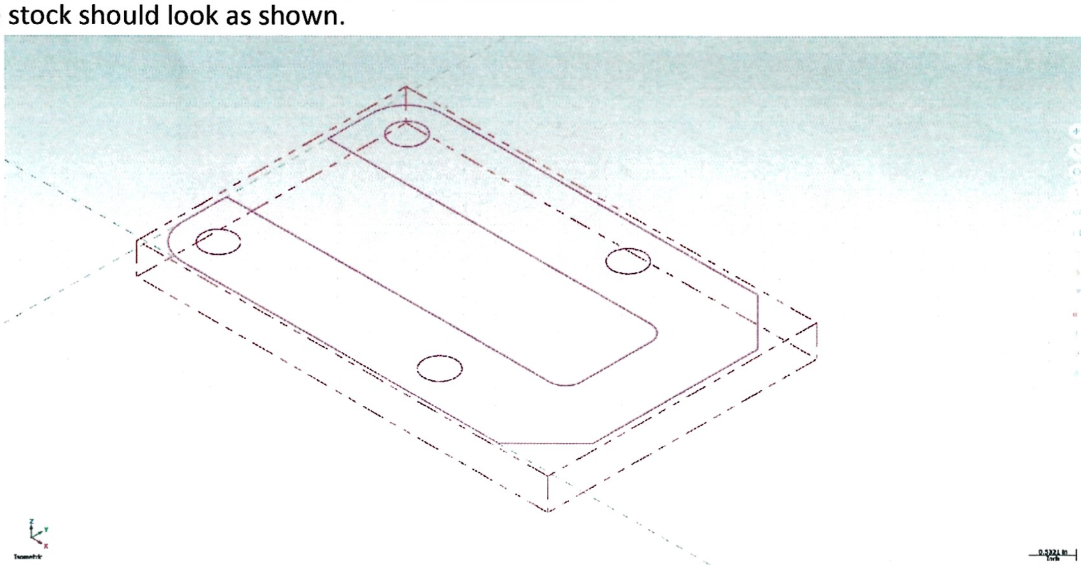
The Stock Origin values adjust the positioning of the stock, ensuring that you have equal amount of extra stock around the finished part.

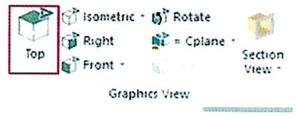
Display options allow you to set the stock as Wireframe and to fit the stock to the screen (Fit Screen) to better visualize the stock.

- Select the OK button to exit Machine Group Properties.
- Select the Isometric view from the view tool bar to see the stock



- The stock should look as shown.



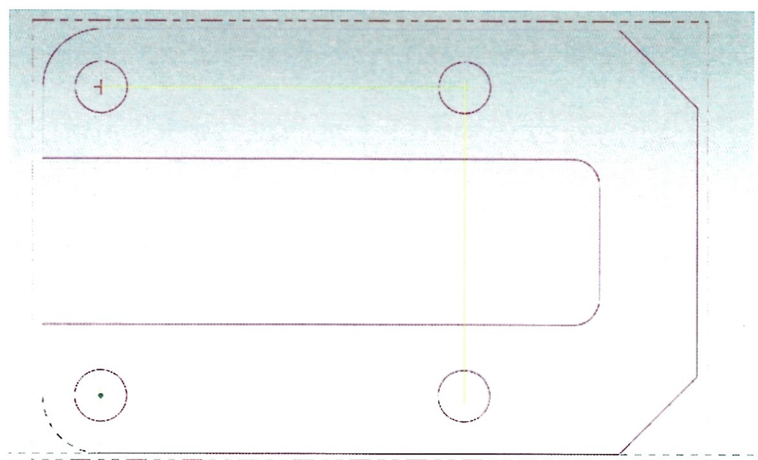
- Select the Top View from the view toolbar  to see the part from the top.

Step 12: Spot drill the four ½" diameter holes

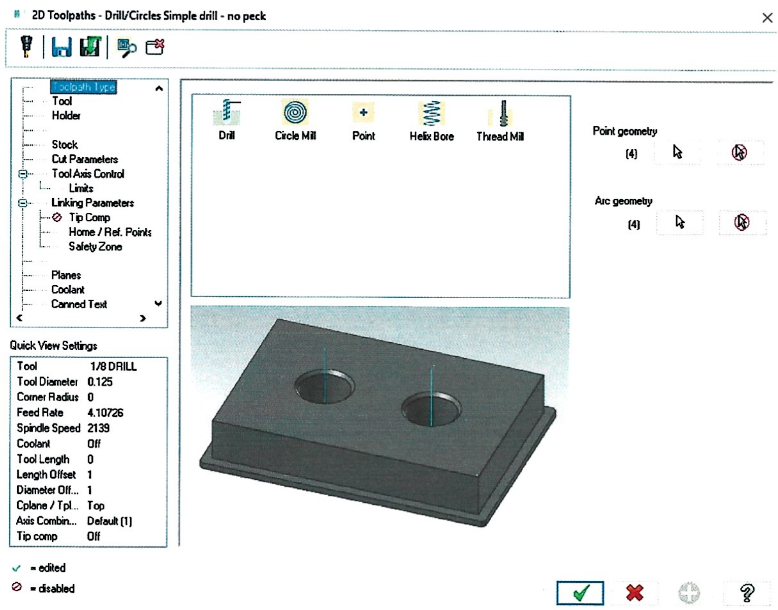
Drill center points selection

- Select the Toolpaths tab and click Drill icon.

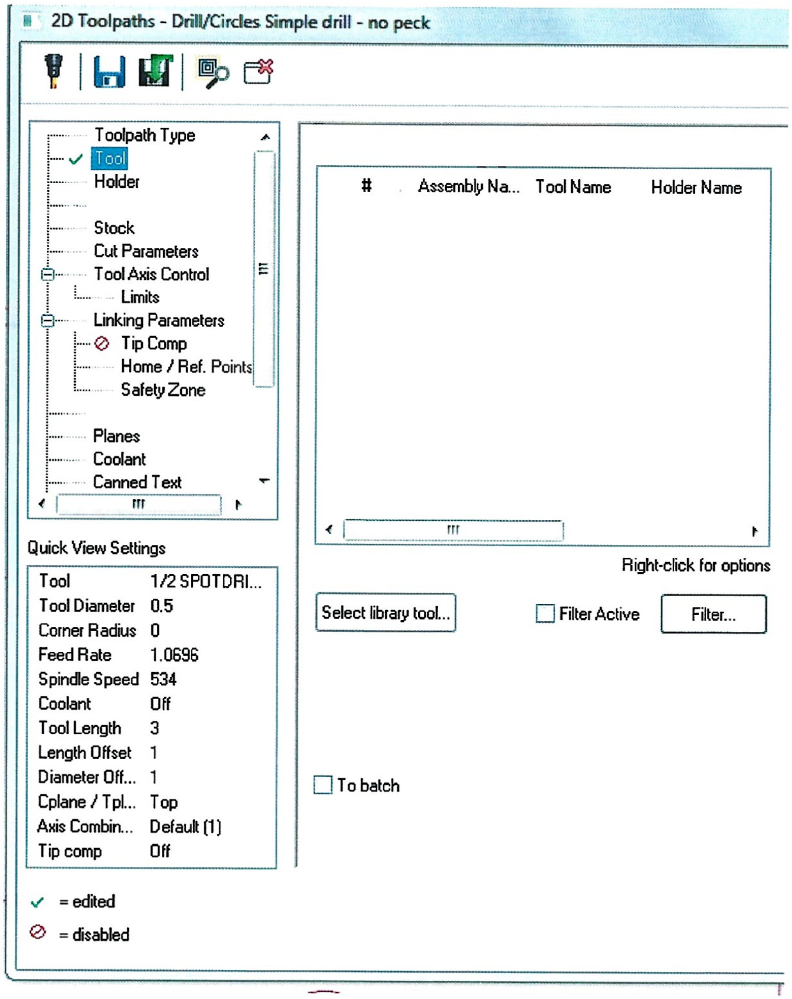
- Select the OK four circle centers by clicking within the center of the circles as shown in the following picture.



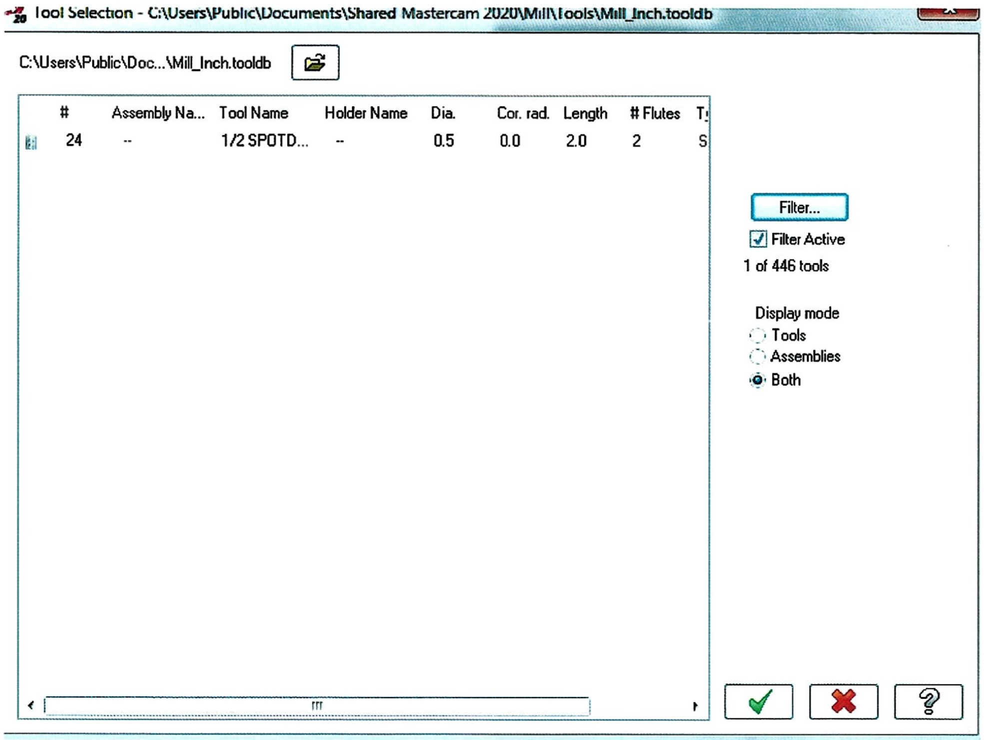
- In the Toolpath Type page, note that the Drill icon should be selected and in the Point/Arc geometry you should have 4 points as shown.


2D Toolpaths - Drill/Circles Simple drill - no peck
Toolpath Type
Tool
Holder
Stock
Cut Parameters
Tool Axis Control
Limits
Linking Parameters
Tip Comp
Home / Ref. Points
Safety Zone
Planes
Coolant
Canned Text
Drill
Circle Mill
Point
Helix Bore
Thread Mill
Point geometry
(4)
Arc geometry
(4)
Quick View Settings
Tool 1/8 DRILL
Tool Diameter 0.125
Corner Radius 0
Feed Rate 4.10726
Spindle Speed 2139
Coolant Off
Tool Length 0
Length Offset 1
Diameter Off... 1
Cplane / Tpl... Top
Axis Combin... Default (1)
Tip comp Off
= edited
= disabled

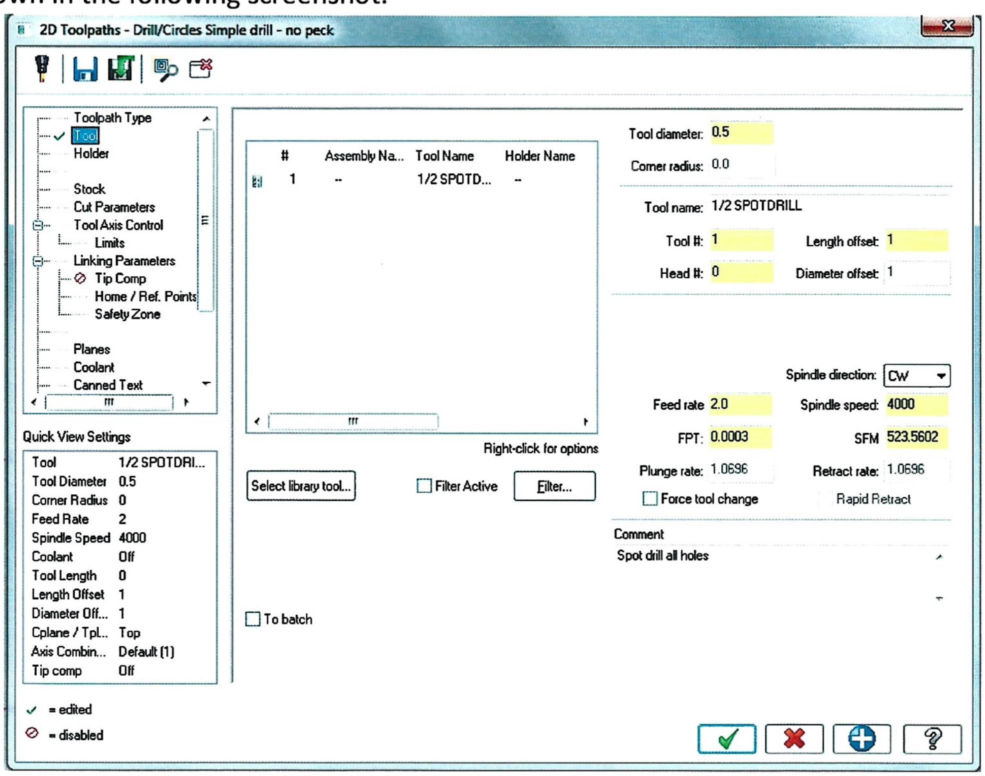
- From the Tree view list select Tool.



- Click on the Select library tool button.
- Disable Filter Active if needed to see all the tools available in the current tool library.
- Select the 1/2" Spot Drill from the list (#24). We will use this tool to chamfer the holes too.

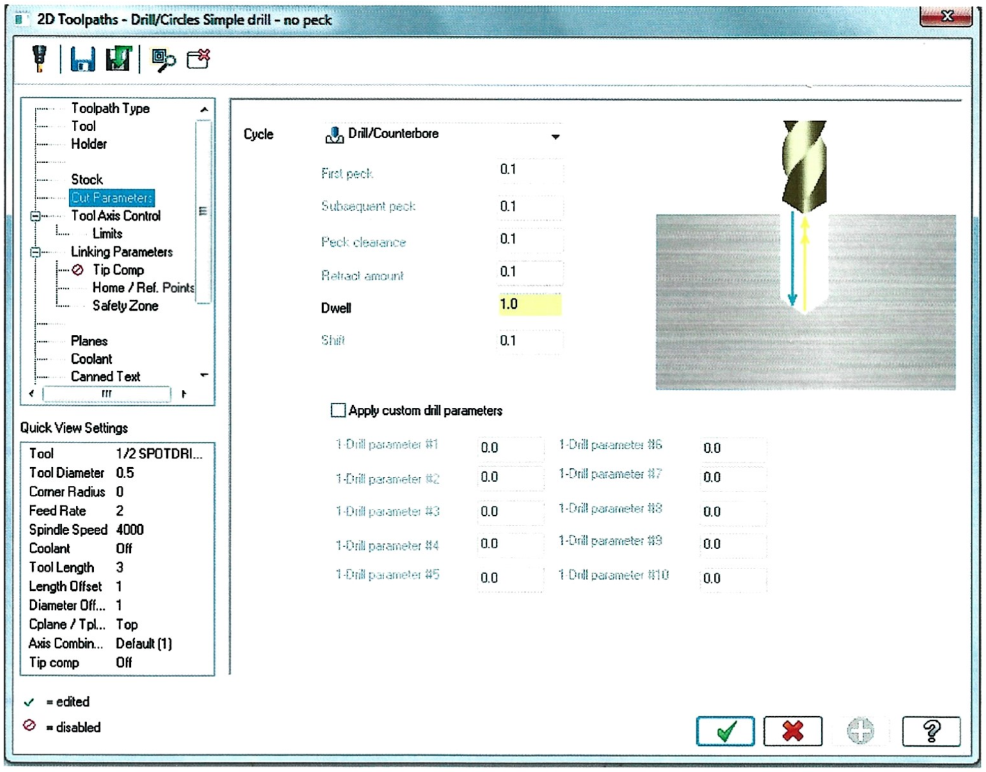


- Select the OK button.
- Set the parameters of the spot drilling operation: Make the changes in the Tool parameters as shown in the following screenshot.

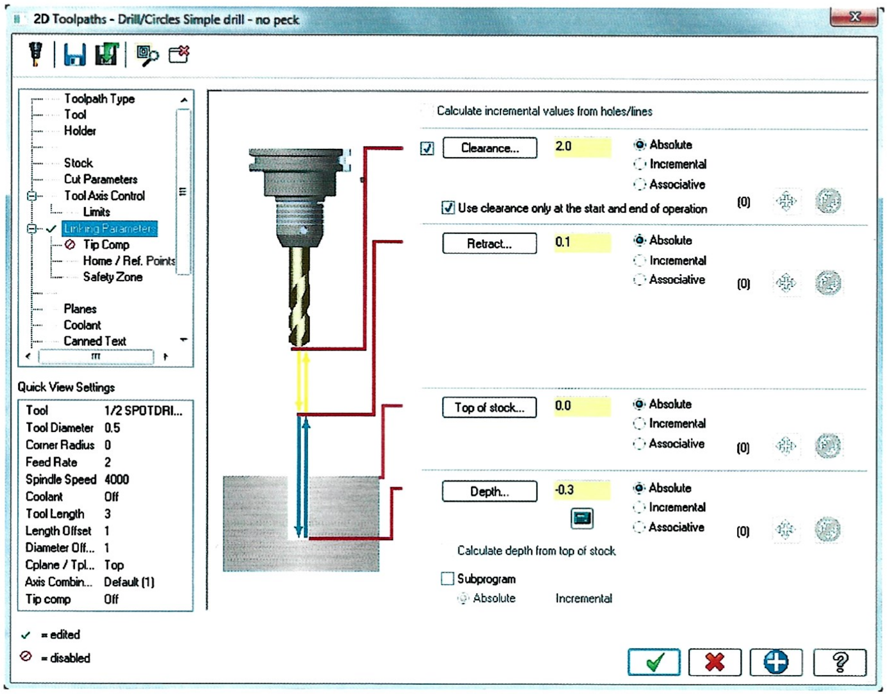


Note that Feed rate, Plunge rate, Retract rate and Spindle speed are roughly based on the part material aluminum and the tool material HSS. Change them if needed.

- In the Comment field enter a comment to help identify the toolpath in the Toolpaths/Operations Manager
- Set the Cut Parameters: From the Tree view list select Cut Parameters.
  Make sure that Cycle is set to Drill /Counterbore and set the Dwell to 1 second to cleanup the chamfer.



- Set the Linking Parameters: From the Tree view list, select Linking Parameters. Make sure the parameters are set as shown.



Note: Clearance value sets the height at which the tool rapids to or from the part. Retract value sets the height at which the tool rapids/feed-rates up to, before the next step down. Depth value sets the final machining depth for the drilling operation.

- Select the OK button to exit drilling parameters.
- Press Alt + T keys to remove the toolpath display from the screen.

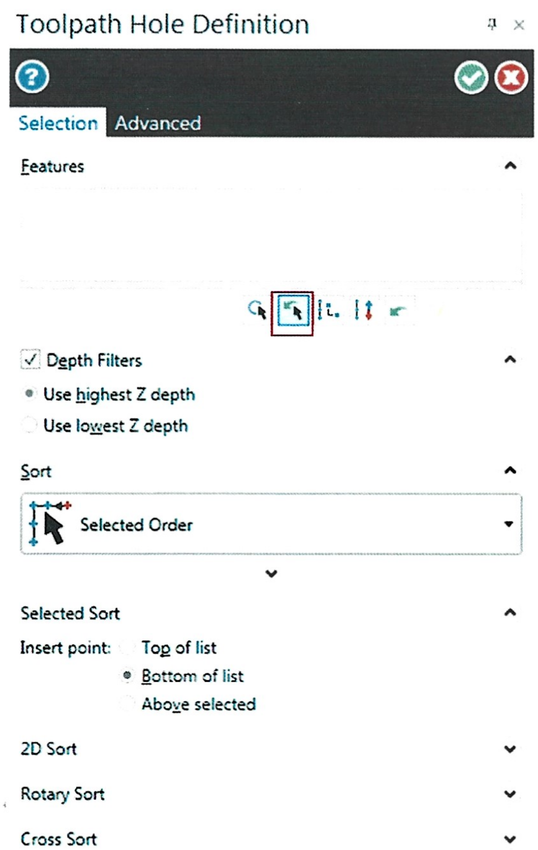
Step 13: Drill all four ½" diameter through holes

Drill center points selection

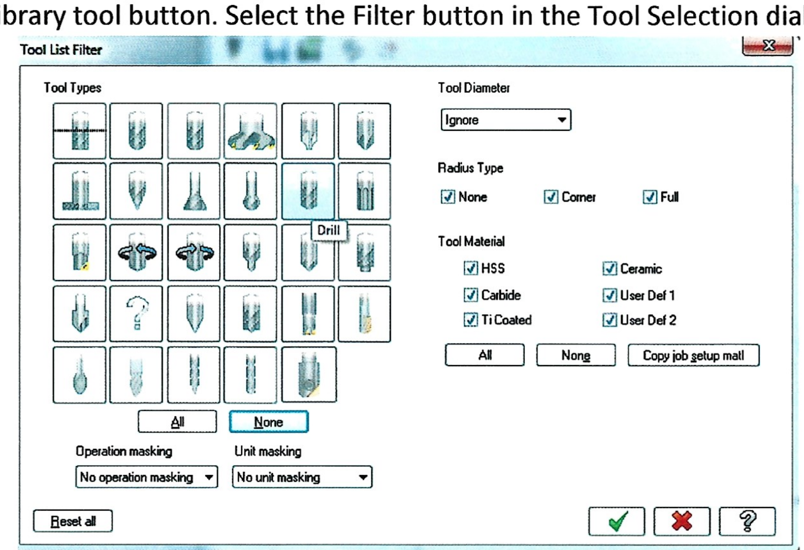
Toolpaths

- Select the Toolpaths tab and click Drill icon.

- Select the Copy previous button in the Drill Point Selection dialog box to select the same center points as you did before.

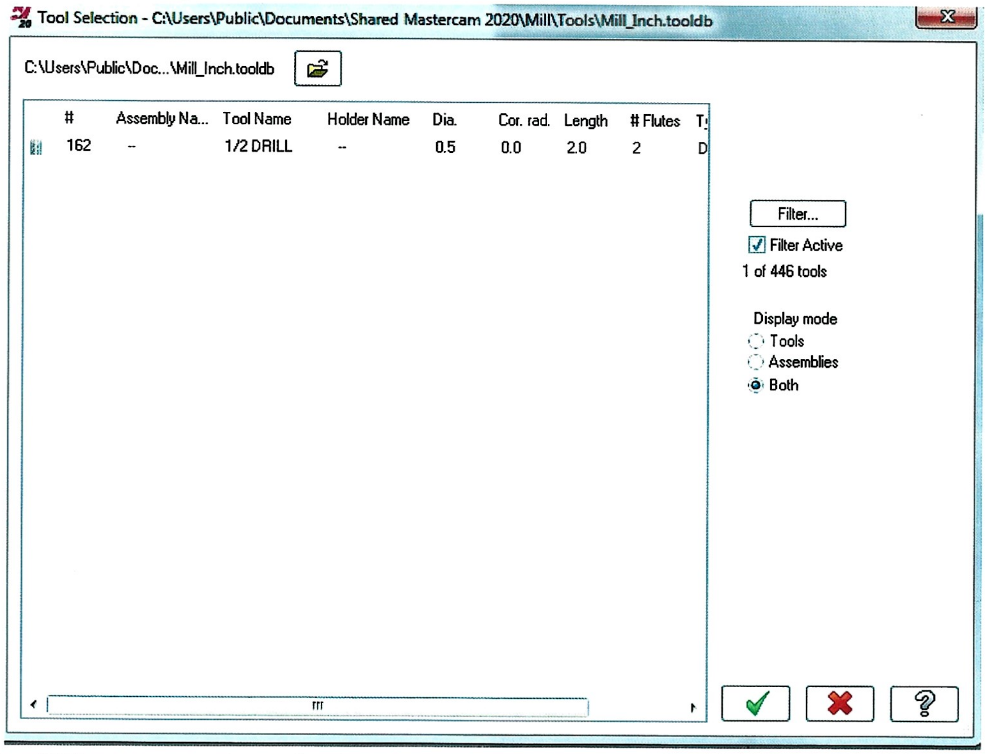


- Select OK button.
- Select the ½" diameter drill and set the Tool parameters: From the Tree view select Tool. Click on the Select library tool button. Select the Filter button in the Tool Selection dialog box.



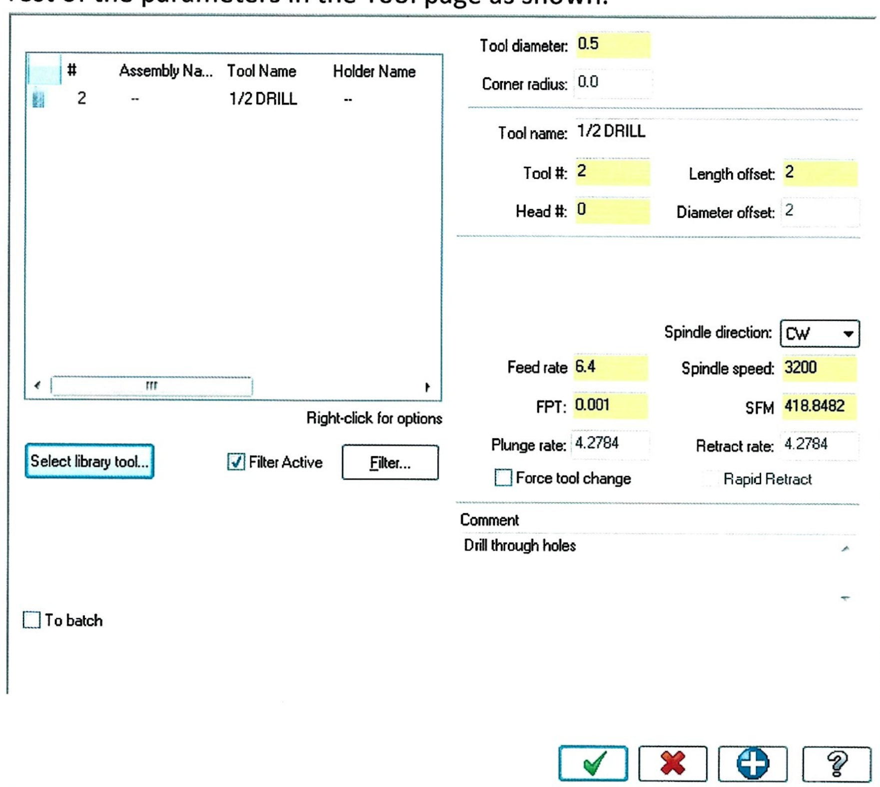
- Select the None button to disable any previous tool selection as shown.
- Select the Drill in the Tool Types list.
- Select the drop-down arrow in the Tool Diameter field and select Equal.

- Enter 0.5 in the Tool Diameter valux box.
- Select the OK button to exit Tool List Filter.
- Select the select library tool in the Tool Selection window

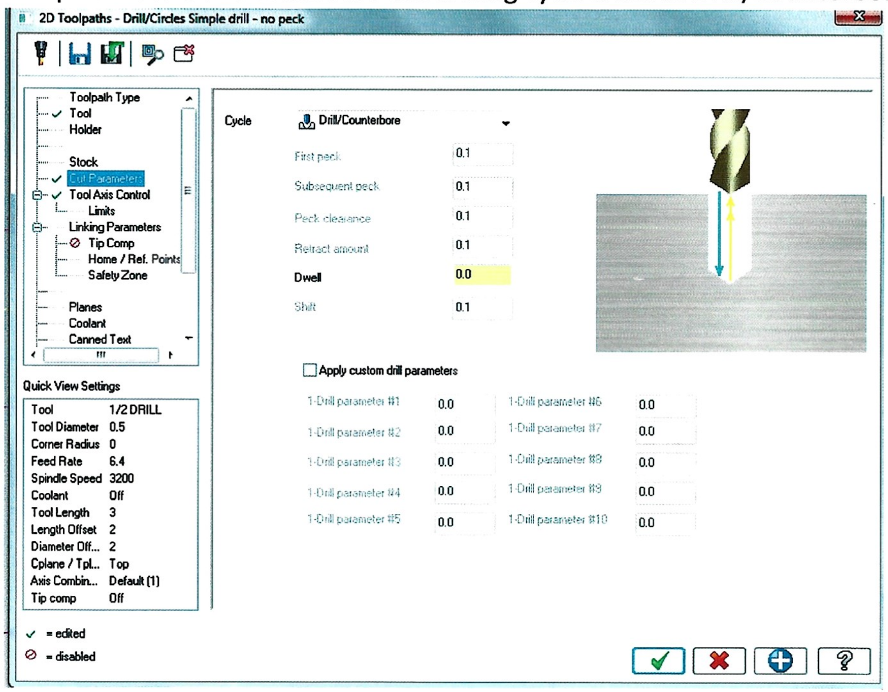


Select the OK button to exit Tool Selection.

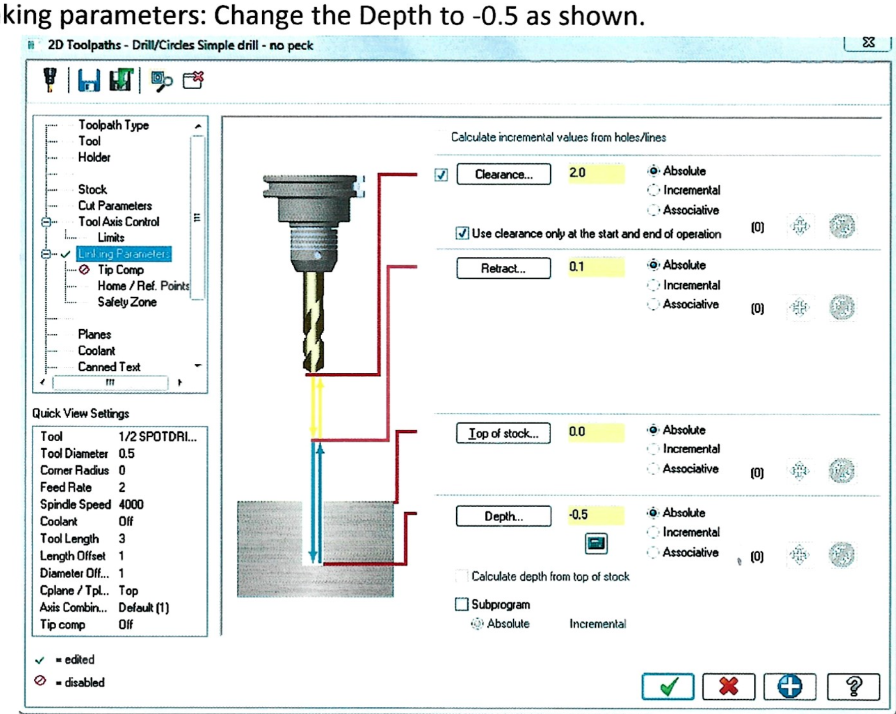
- Change the rest of the parameters in the Tool page as shown.



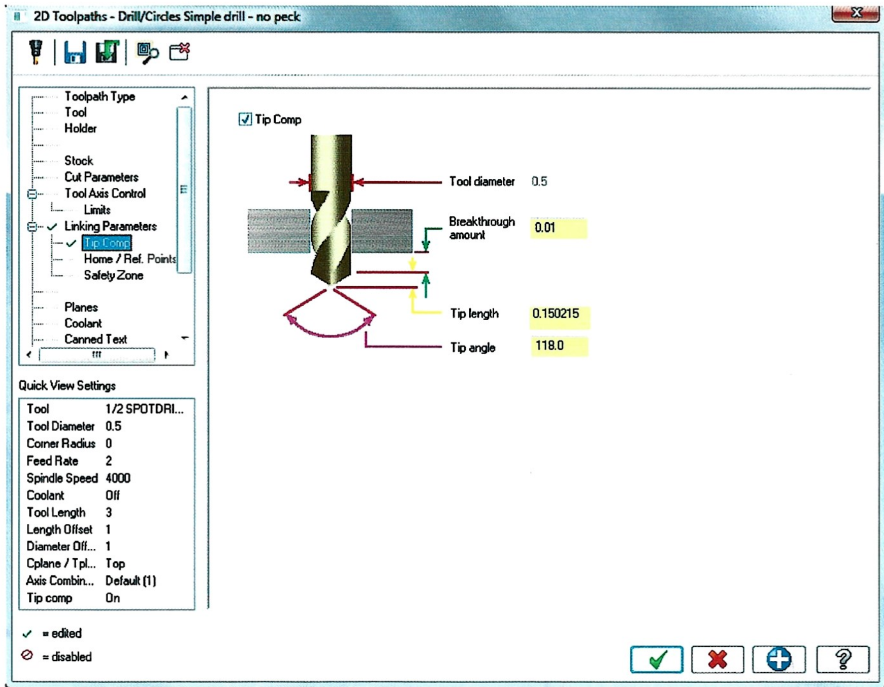
- Click on the Cut parameters and make sure the drilling cycle is set to Drill/Counterbore



- Set the Linking parameters: Change the Depth to -0.5 as shown.



- From the Tree view list, select the plus sign in front of the Linking Parameters and then select Tip comp.
- Enable the Tip Comp to cut deeper than the final depth with the tip of the drill.



Note: Breakthrough amount value allows you to give an extra amount for the tool to go deeper than the final depth to prevent any remaining material for the cut-outs. Tip length value is automatically calculated by the system based on the diameter and tip angle of the tool. The value is added to the final depth.

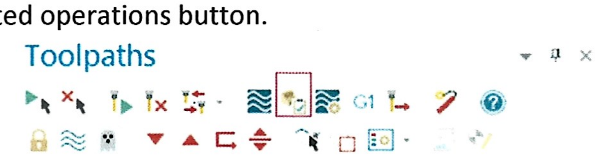
- Select the OK button to exit
- Press Alt + T to remove the toolpath display.

Step 14: Verify the toolpath

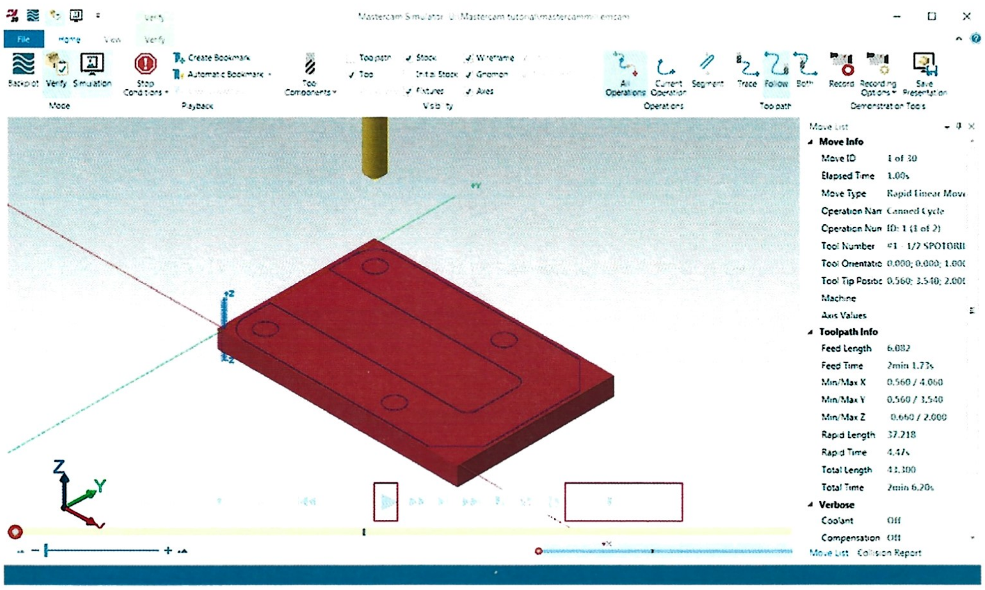
Verify simulate the machining of a part from a stock model display. The stock dimension are based on the values that we specified in the Stock Setup.

- Click on Select all operations icon

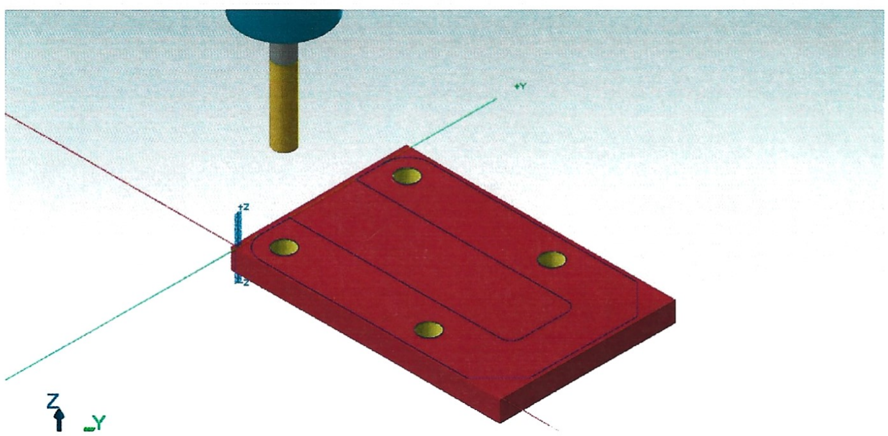
- Select the Isometric view from the view toolbar to see the stock
- Select the Fit button
- Select the Verify Selected operations button.



- Set the Verify speed by moving the slider bar in the speed control bar.



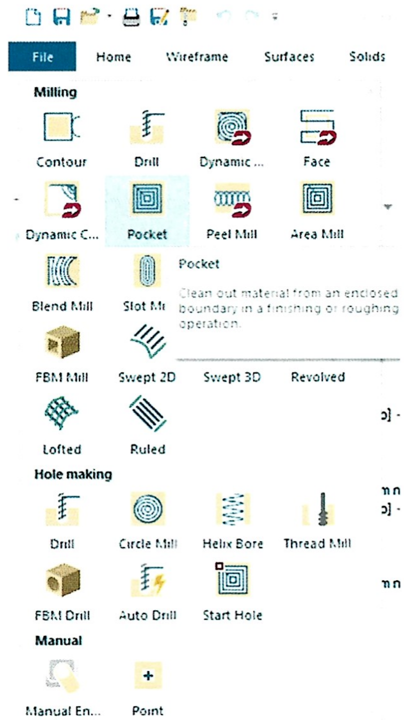
- Select the play button to start simulation.
  The part should appear as shown in the following picture.



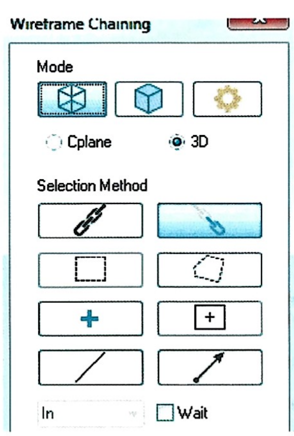
Step 15: Remove the inside material using open pocket

Pocket toolpaths remove material inside an enclosed area. Open pockets are created from chains that do not have the same start and end point. Mastercam creates and imaginary line between the start and end points of the chain to close the geometry, allowing you to overlap the open area with a percentage of the tool diameter.

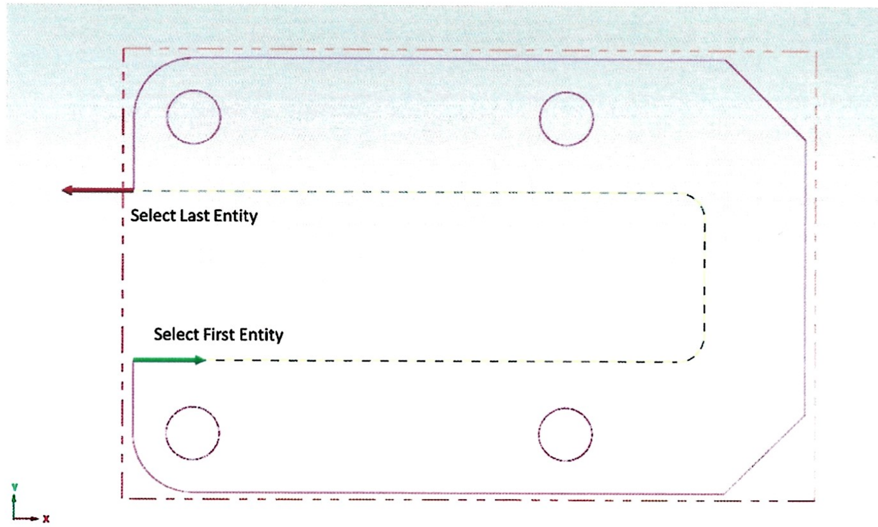
- Pocket selection: Select pocket in Toolpaths 2D Milling (Do not select Pocket from 3D Roughing)



- Enable Partial button in the chaining dialog box to be able to select the first and the last entity of the chain.

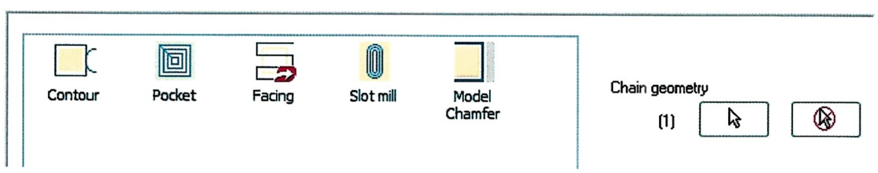


- Select the first entity in the chain, as shown.
- Select the last entity as shown.

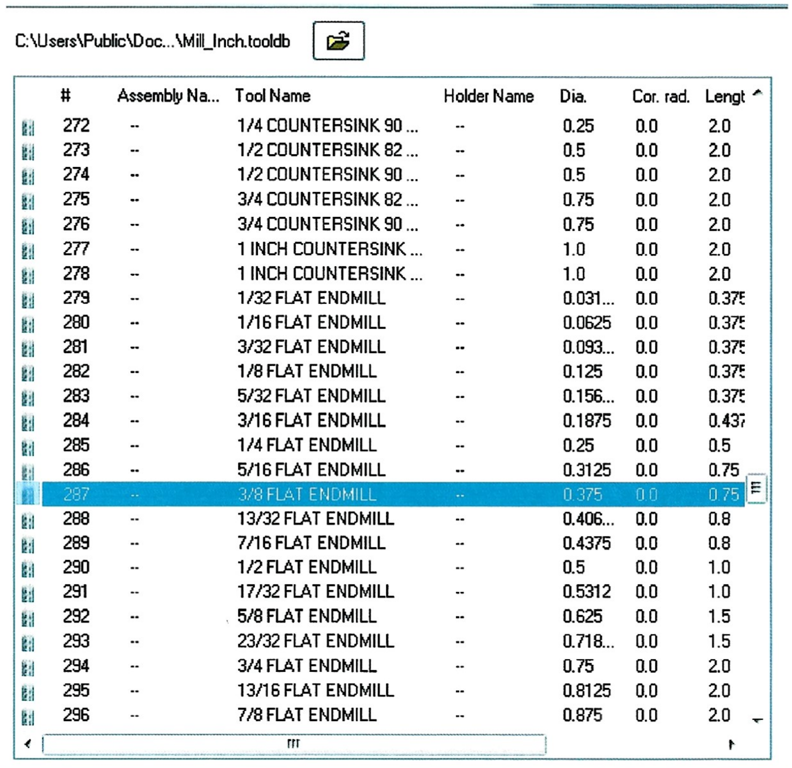


Note: For climb milling, be sure to chain the contour in a CCW direction. Otherwise, select the Reverse button.

- Select the OK button to exit chaining.
  Note that in the Toolpath Type page the Pocket icon is automatically selected by Mastercam and the Chain geometry shows one (1) chain.



- Select a 3/8" Flat endmill from the current library
- From the Tree view list, select Tool.
- Click on the Select library tool button.
- Disable Filter Active to see all the tools available in the current tool library.
- Scroll down and select the 0.375 diameter Endmill form the Tool Selections list.

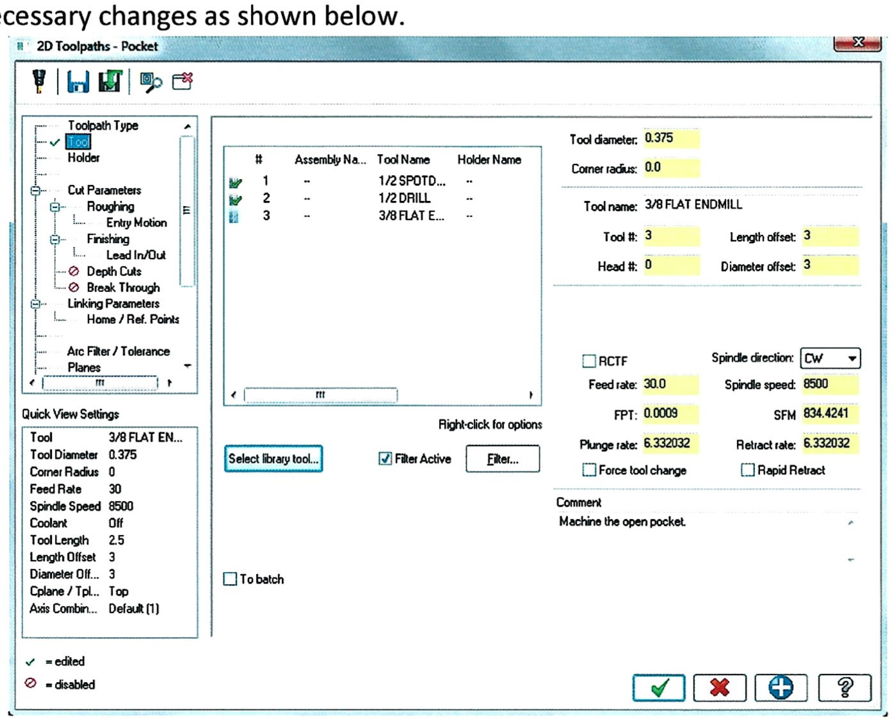


C:\Users\Public\Doc...\Mill_Inch.tooldb

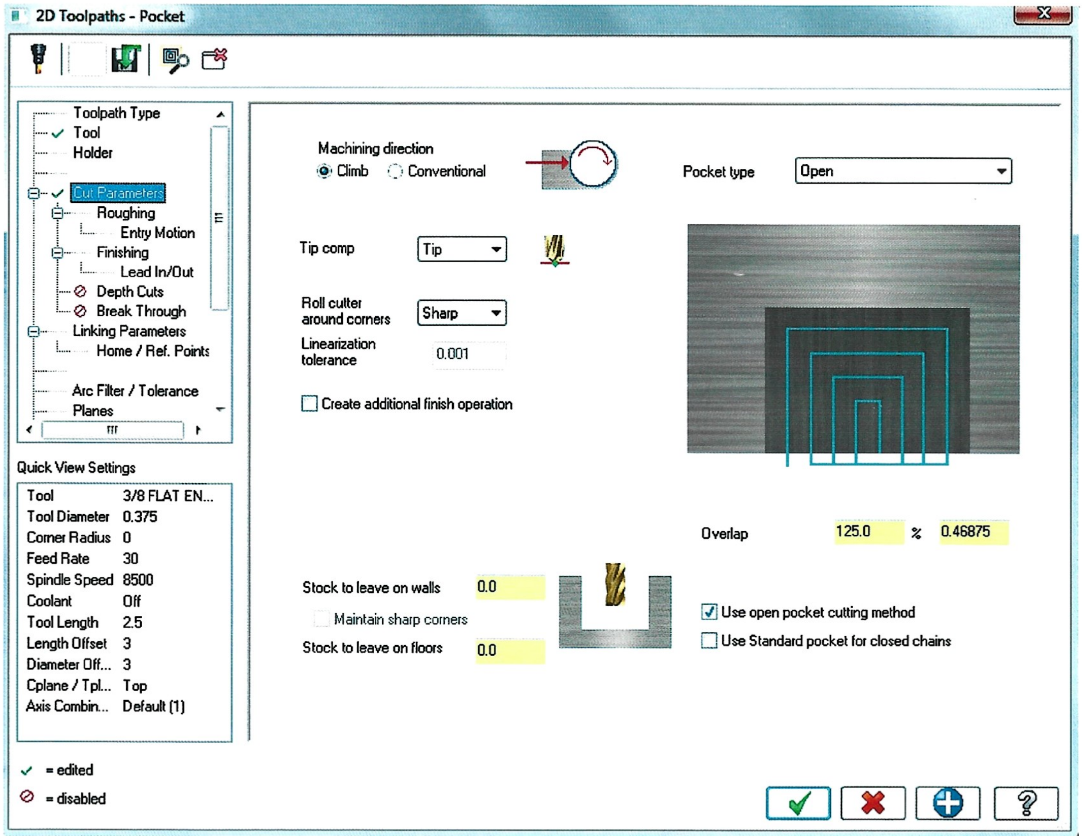
#	Assembly Na...	Tool Name	Holder Name	Dia.	Cor. rad.	Lengt
272	--	1/4 COUNTERSINK 90 ...	--	0.25	0.0	2.0
273	--	1/2 COUNTERSINK 82 ...	--	0.5	0.0	2.0
274	--	1/2 COUNTERSINK 90 ...	--	0.5	0.0	2.0
275	--	3/4 COUNTERSINK 82 ...	--	0.75	0.0	2.0
276	--	3/4 COUNTERSINK 90 ...	--	0.75	0.0	2.0
277	--	1 INCH COUNTERSINK ...	--	1.0	0.0	2.0
278	--	1 INCH COUNTERSINK ...	--	1.0	0.0	2.0
279	--	1/32 FLAT ENDMILL	--	0.031...	0.0	0.375
280	--	1/16 FLAT ENDMILL	--	0.0625	0.0	0.375
281	--	3/32 FLAT ENDMILL	--	0.093...	0.0	0.375
282	--	1/8 FLAT ENDMILL	--	0.125	0.0	0.375
283	--	5/32 FLAT ENDMILL	--	0.156...	0.0	0.375
284	--	3/16 FLAT ENDMILL	--	0.1875	0.0	0.437
285	--	1/4 FLAT ENDMILL	--	0.25	0.0	0.5
286	--	5/16 FLAT ENDMILL	--	0.3125	0.0	0.75
287	--	3/8 FLAT ENDMILL	--	0.375	0.0	0.75
288	--	13/32 FLAT ENDMILL	--	0.406...	0.0	0.8
289	--	7/16 FLAT ENDMILL	--	0.4375	0.0	0.8
290	--	1/2 FLAT ENDMILL	--	0.5	0.0	1.0
291	--	17/32 FLAT ENDMILL	--	0.5312	0.0	1.0
292	--	5/8 FLAT ENDMILL	--	0.625	0.0	1.5
293	--	23/32 FLAT ENDMILL	--	0.718...	0.0	1.5
294	--	3/4 FLAT ENDMILL	--	0.75	0.0	2.0
295	--	13/16 FLAT ENDMILL	--	0.8125	0.0	2.0
296	--	7/8 FLAT ENDMILL	--	0.875	0.0	2.0

Select the OK button to exit Tool Selection.

- Make all necessary changes as shown below.

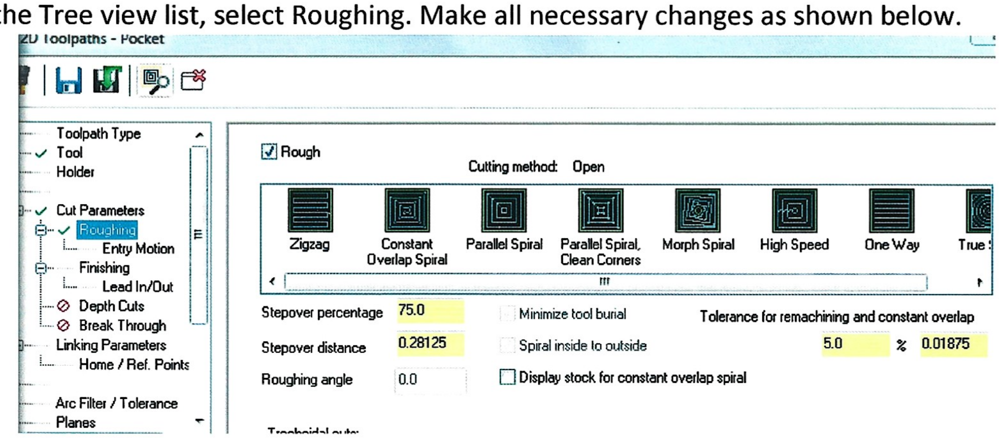


- Set the Cut Parameters: Make all necessary changes as shown below.



Machining Direction set to Climb cuts in one direction with the tool rotating in the opposite direction of the tool motion. Pocket type set to Open can be used with open and closed chains. The distance between the start and end points of the open contour is determined and internally Mastercam creates a line to close the chain. Overlap sets the amount that the tool overlaps the open area. Use open pocket cutting method forces the tool to start cutting from the open area, avoiding plunges in the middle of the part.

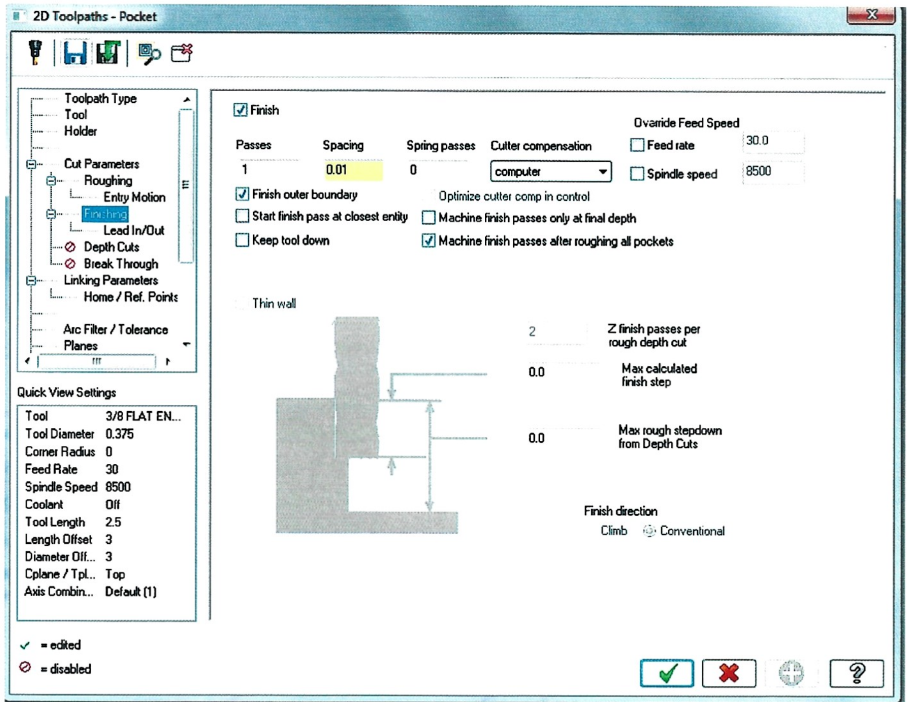
- From the Tree view list, select Roughing. Make all necessary changes as shown below.



- Stepover percentage sets the distance between roughing passes in the XY axis as a percentage of the tool diameter and will automatically update the stepover distance. Roughing angle sets the orientation for the roughing passes when you use the One Way or Zigzag cutting methods for pocket tool paths.

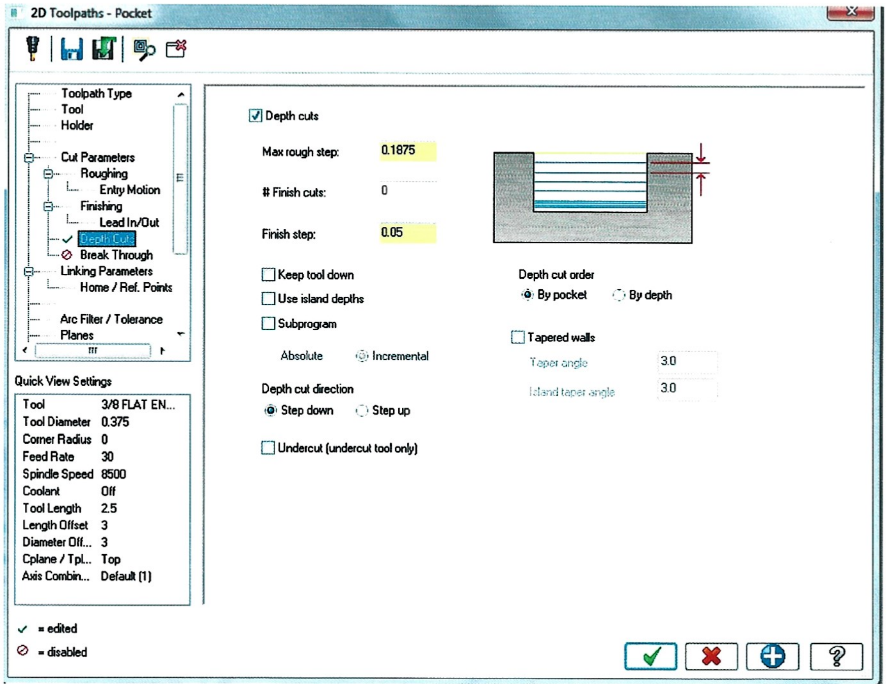
Note that Zigzag cutting method will not be used in this toolpath; instead, the open pocket cutting method will be used as set in the Cutting Parameters page.

- From the Tree view list, select Finishing

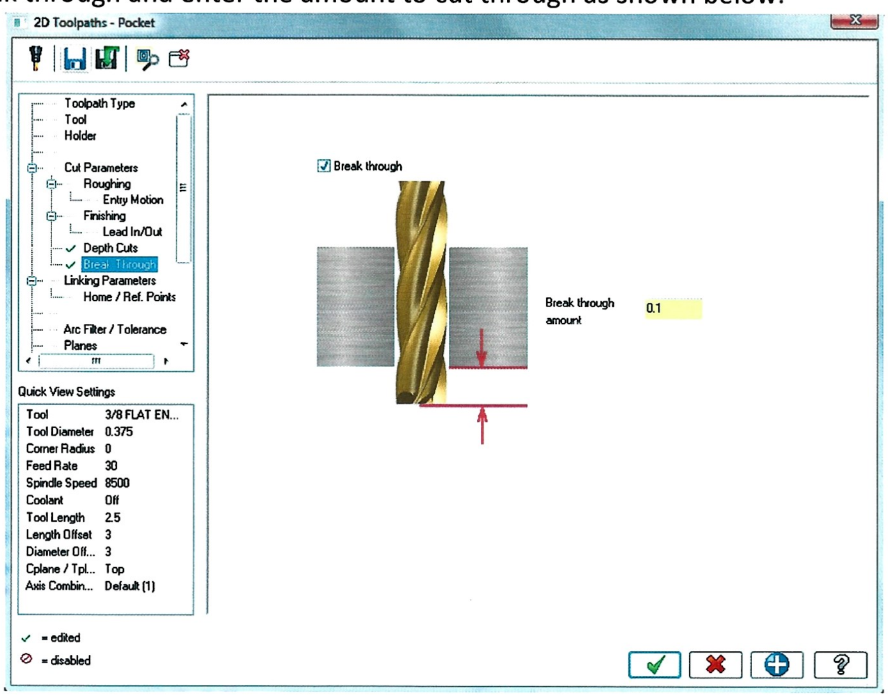


The current settings in the Finish area enables one finish pass of a 0.01 amount, performed around the pocket walls at the final depth.

- From the Tree view list, select Depth Cuts.
- Enable the box in front of the Depth cuts and change the parameters to divide the total depth in increments not bigger than the tool radius as shown below.

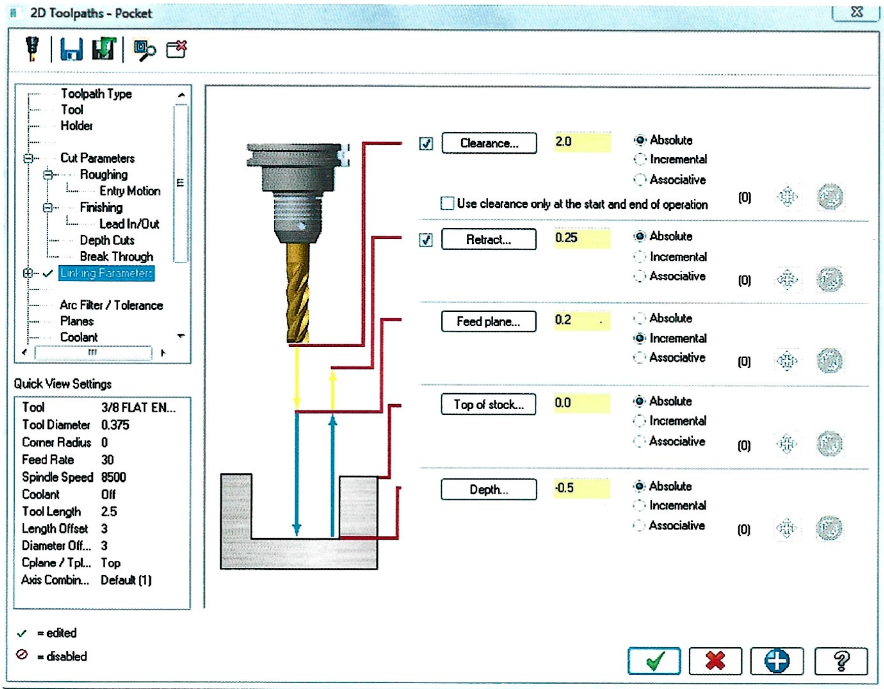


- From the Tree view list, select Break Through.
- Enable Break through and enter the amount to cut through as shown below.



Break Through causes the tool to cut through the material. Always enter the break through amount as a positive number. Mastercam adds the break through amount to the final depth of the toolpath to ensure through-cutting.

- Set the Linking Parameters: From the Tree view list, select Linking Parameters.
- Enable Clearance and enter the Depth value as shown below.



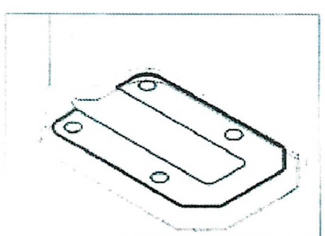
Clearance value sets the height at which the tool rapids to or from the part. Retract value sets the height at which the tool rapids/feed-rates up to, before the next step down. Feed plane value sets the height from which the tool plunge-rates into the part. Top of stock value sets the height of the material. Depth value sets the final machining depth for the pocket operation.

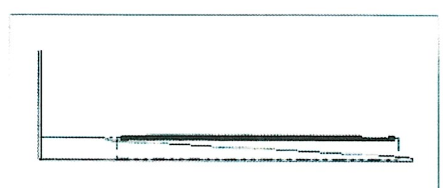
- Select the OK button to exit 2D Toolpaths- Pocket parameters

Step 16: Contour the outside profile using ramp

Contour toolpaths remove material along a path defined by a chain of curves. Contour toolpaths only follow a chain; they do not clean out an enclosed area.

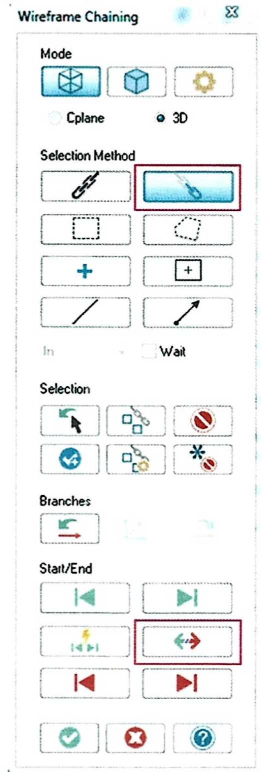
Toolpath Preview:



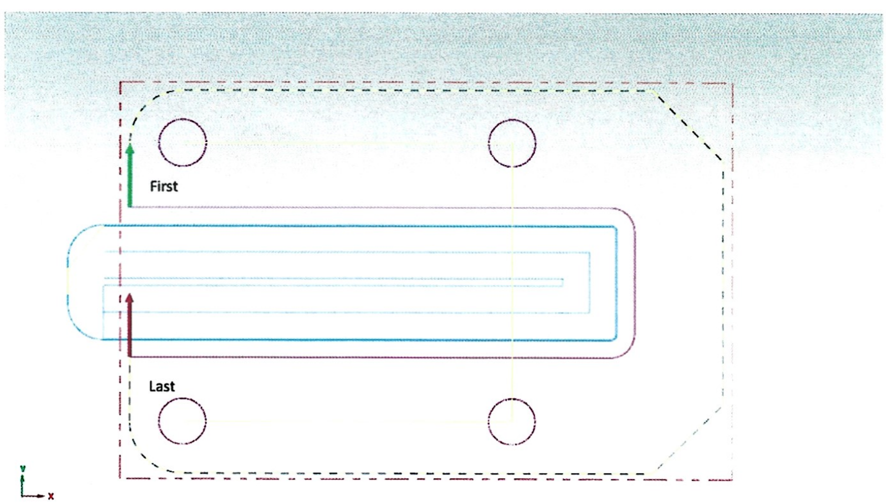


- Toolpaths: Select Contour

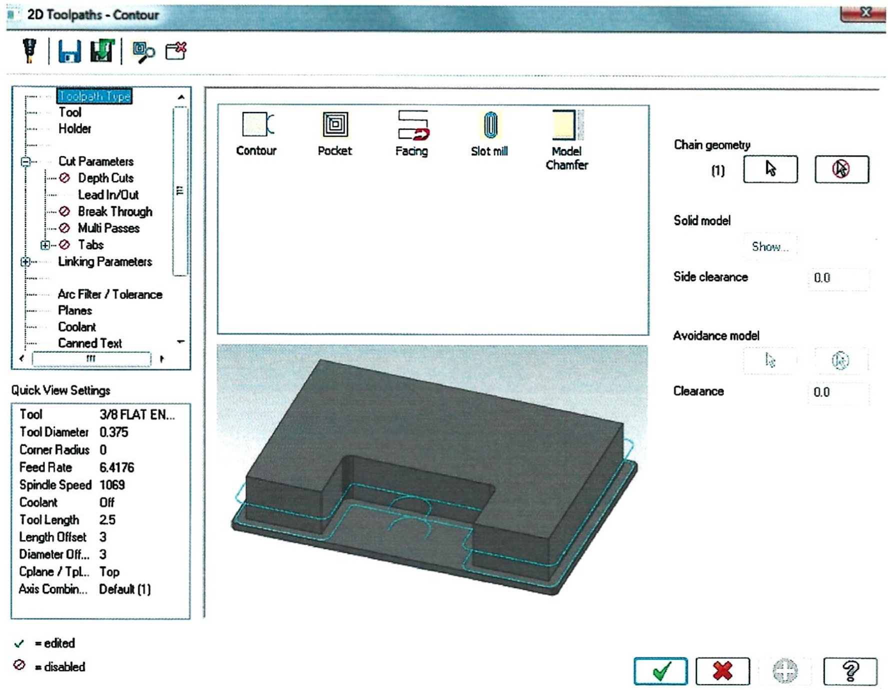
- Enable Partial button in the chaining dialog box to be able to select just the outside contour.



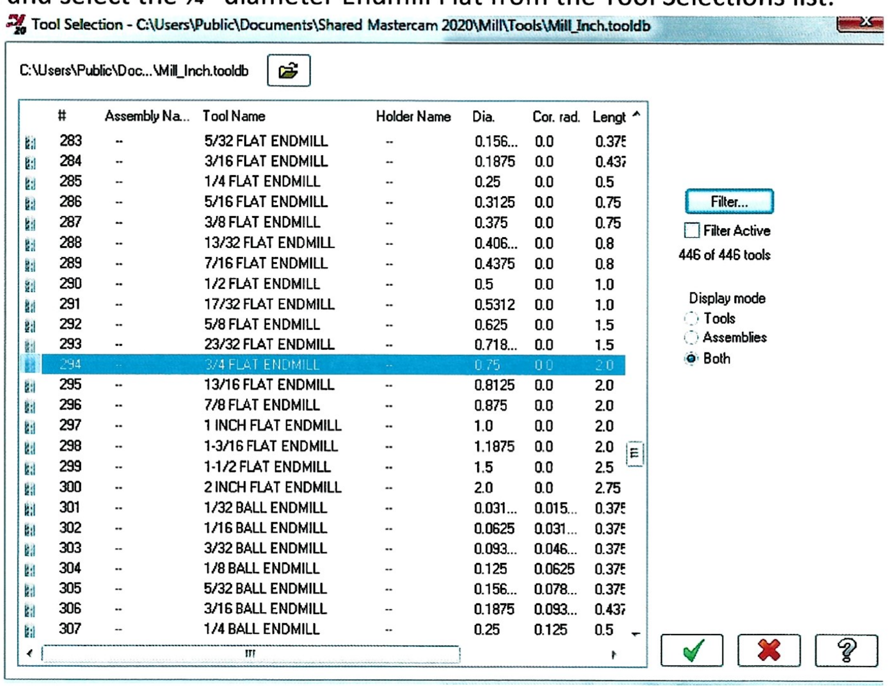
- Select the first entity in the chain, as shown.
  Be sure to chain the contour in a CW direction. Otherwise, select the Reverse button.
- Select the last entity as shown.



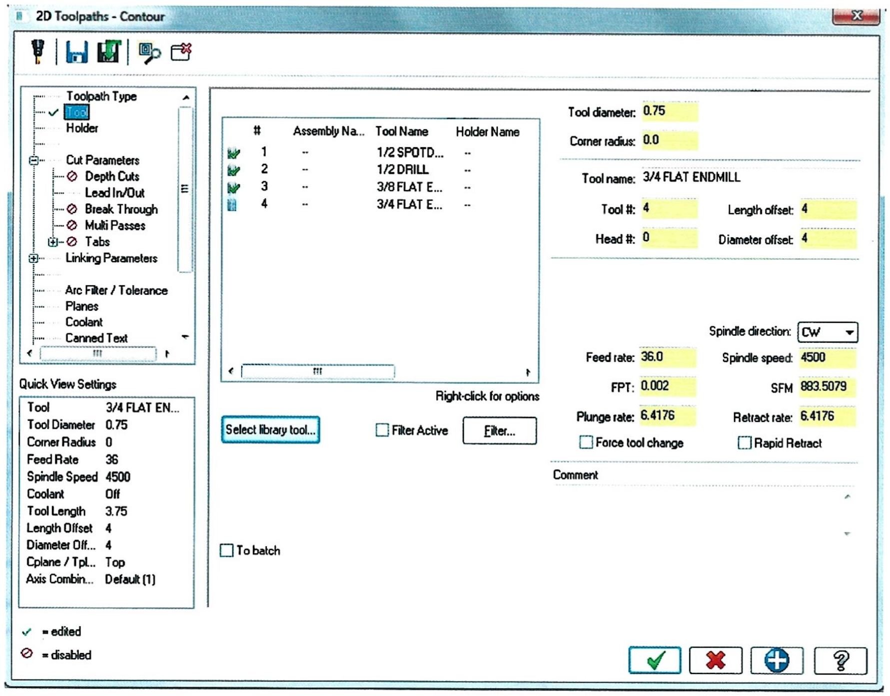
- Select the OK button to exit chaining.
- Check the Toolpath Type to ensure that the contour icon is selected as shown.



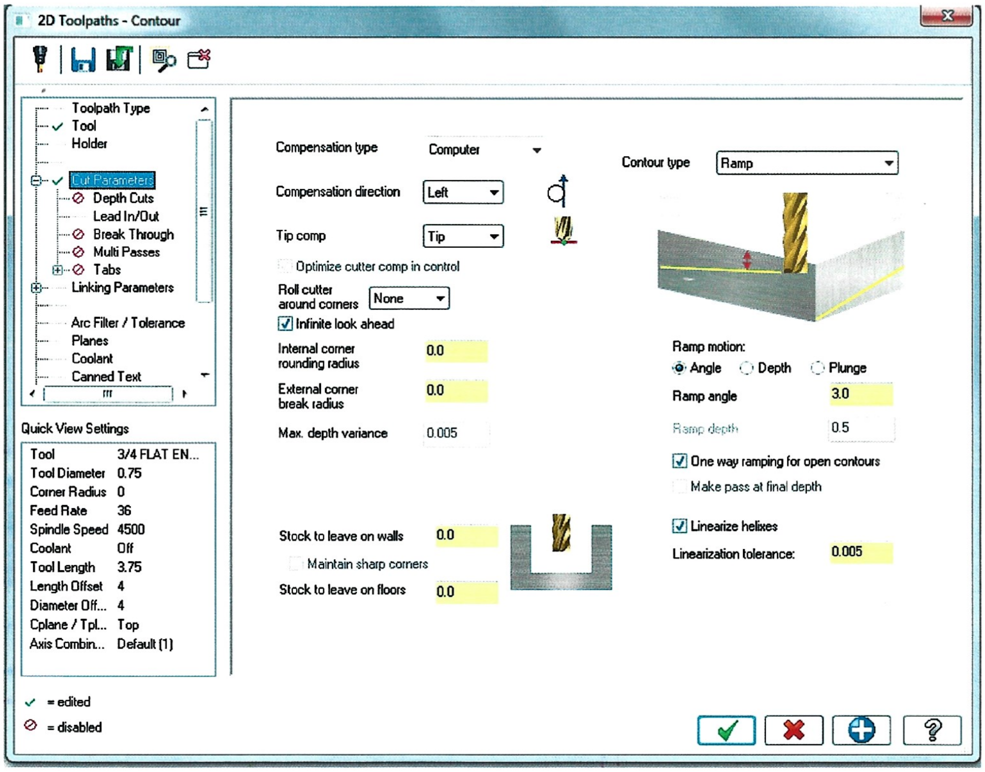
- From the Tree view list, select Tool.
- Click on the Select library tool button.
- Disable Filter Active to see all the tools available in the current tool library.
- Scroll down and select the ¾" diameter Endmill Flat from the Tool Selections list.



- Select the OK button to exit Tool Selection.
- Change the parameters to match the following screenshot.

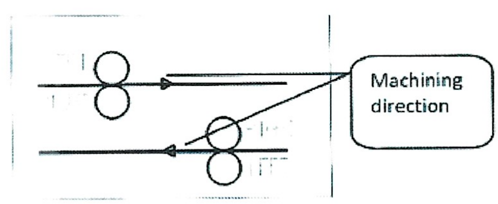

2D Toolpaths - Contour
Toolpath Type
Tool
Holder
Cut Parameters
Depth Cuts
Lead In/Out
Break Through
Multi Passes
Tabs
Linking Parameters
Arc Filter / Tolerance
Planes
Coolant
Canned Text
Quick View Settings
Tool 3/4 FLAT EN...
Tool Diameter 0.75
Corner Radius 0
Feed Rate 36
Spindle Speed 4500
Coolant Off
Tool Length 3.75
Length Offset 4
Diameter Off... 4
Cplane / Tpl... Top
Axis Combin... Default (1)
= edited
= disabled
Assembly Na... Tool Name Holder Name
1 -- 1/2 SPOTD... --
2 -- 1/2 DRILL --
3 -- 3/8 FLAT E... --
4 -- 3/4 FLAT E... --
Right-click for options
Select library tool...
Filter Active
Filter...
To batch
Tool diameter: 0.75
Corner radius: 0.0
Tool name: 3/4 FLAT ENDMILL
Tool #: 4
Length offset: 4
Head #: 0
Diameter offset: 4
Spindle direction: CW
Feed rate: 36.0
Spindle speed: 4500
FPT: 0.002
SFM 883.5079
Plunge rate: 6.4176
Retract rate: 6.4176
Force tool change
Rapid Retract
Comment

- Set the Cut Parameters: Make all the necessary changes as shown below.

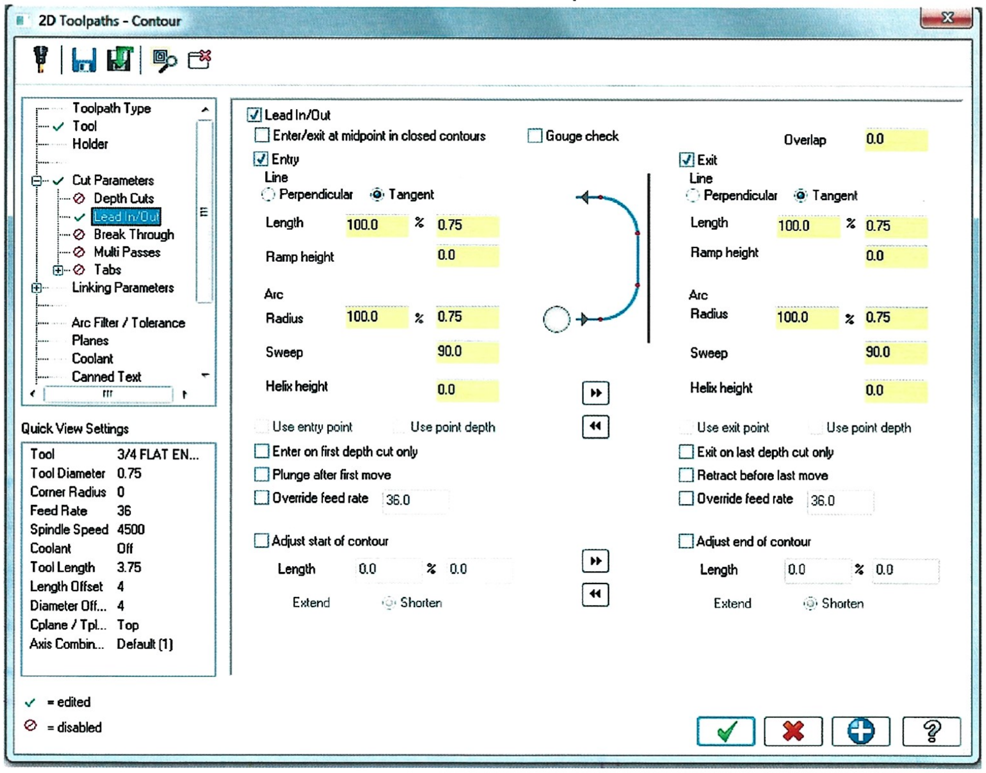


Compensation type set to Computer allows Mastercam to compensate the toolpath based on the tool diameter and does not output G41/G42 in the code. Compensation direction set to Left compensates the toolpath to the left of the chain based on the chaining direction. See the graphic below. Roll cutter around corners set to Sharp inserts arc moves around corners in the toolpath. The radius of the arc moves is equal with the radius of the tool. Set to None to not create any extra arcs. Infinite look ahead prevents the toolpath from crossing itself (fish tail). Contour ramp type allows you to use a continuous ramp to transition smoothly between depth cuts, instead of individual plunge cuts.

Cutter compensation:

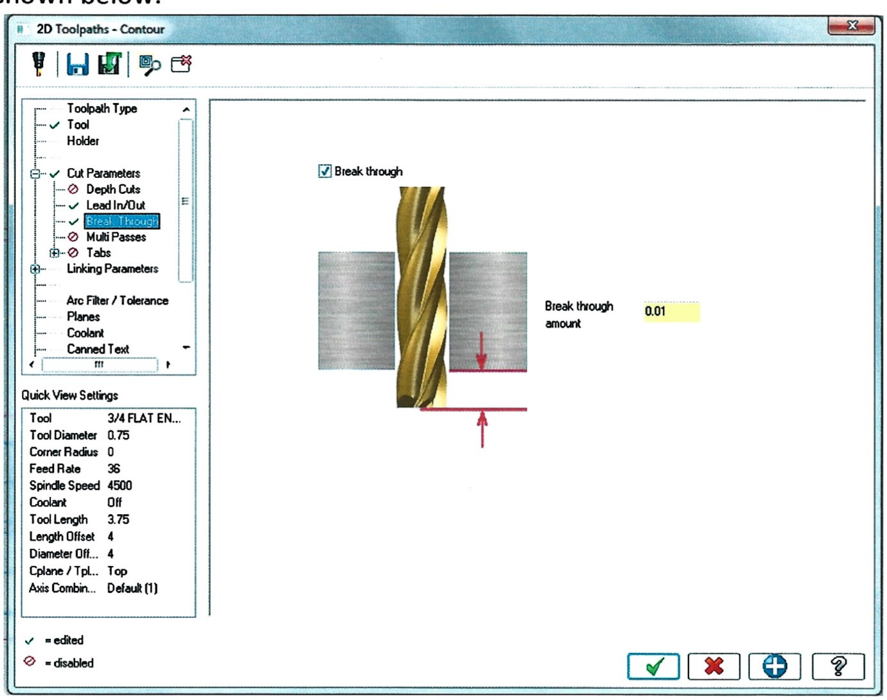


- From the Tree view list, select Lead In/Out and set the parameters as shown below.

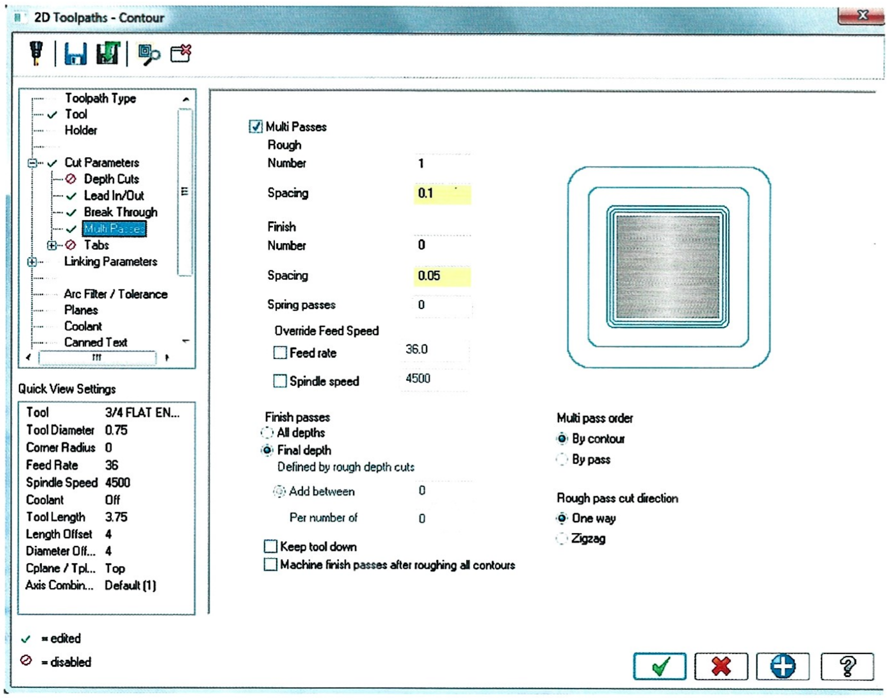


Lead In/Out allows you to select a combination of a Line and an Arc at the beginning and/or end of the contour toolpath for a smooth entry/exit while cutting the part.

- From the Tree view list, select Break Through, enable the option and enter the Break through amount as shown below.

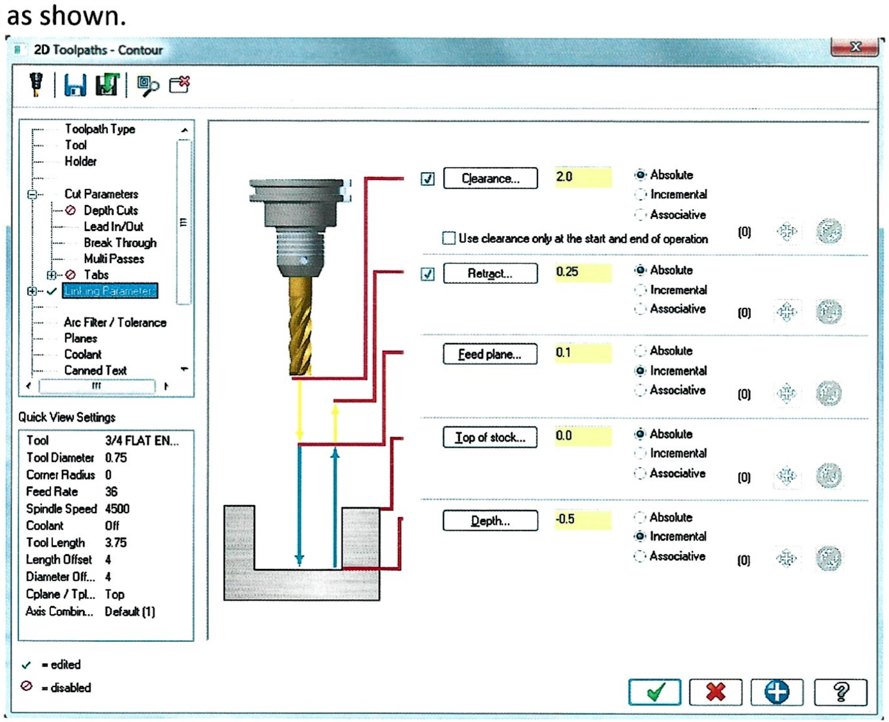


- From the Tree view list, select Multi Passes
- Enable Multi passes to set a rough pass and a finish pass at the final depth.



Multiple cutting passes allows you to follow the contour geometry in steps in the current cutting plane (XY). Setting to 1 the Number of rough passes will remove the material around the outside contour leaving the 0.05 amount untouched for the finish pass. The finish pass will remove the 0.05 step at the final depth.

- Set the Linking Parameters: From the Tree view list, select Linking Parameters and change the parameters as shown.



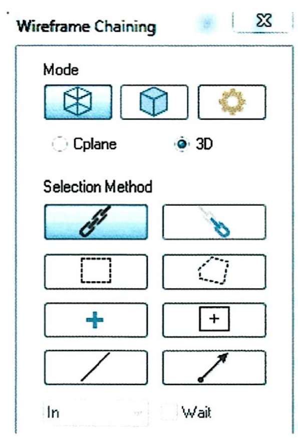
- Select the OK button to exit 2D Toolpaths- Contour parameters.

Step 17: Chamfer the profile using 2D chamfer contour type

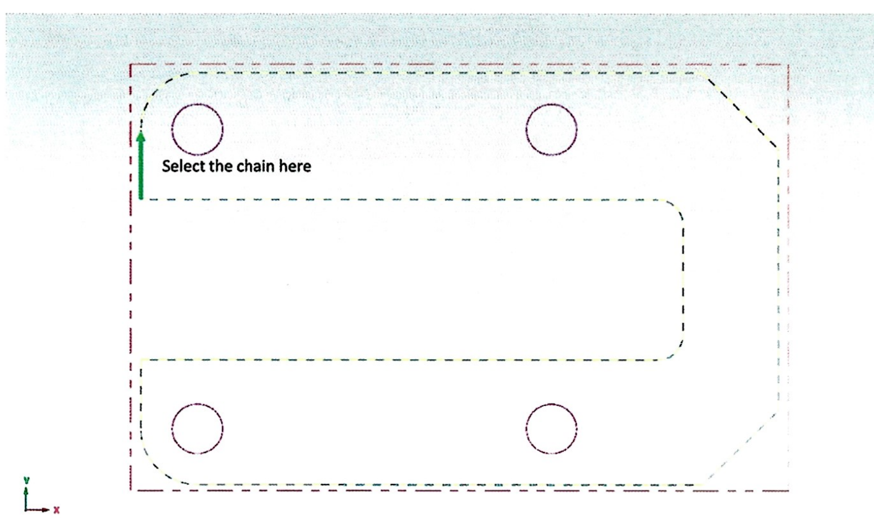
2D Chamfer Contour toolpaths allows you to automatically cut a chamfer around a contour. Typically, when creating a chamfer toolpath, you will set the Depth to 0.0 relative to the chained geometry, and let Mastercam calculate the tool depth from the chamfer dimensions.

- Toolpaths: Select Contour

- Make sure the Chain button in the chaining dialog box is enabled to be able to select the entire contour.

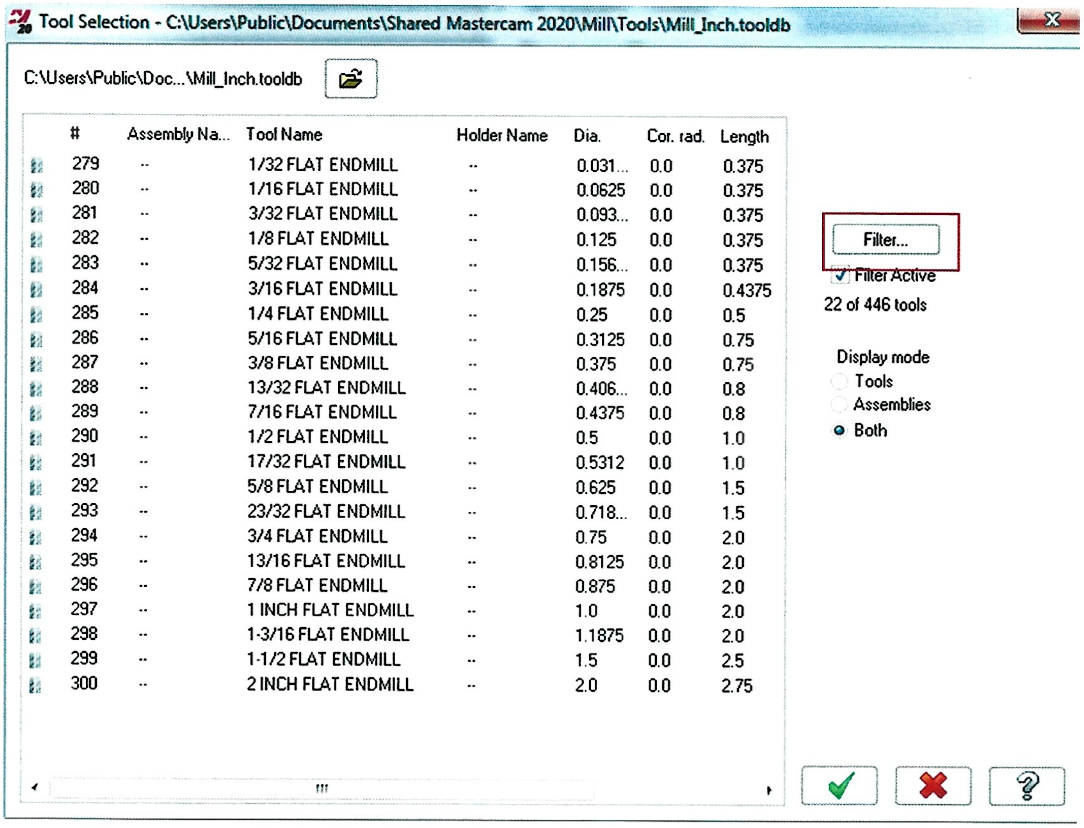


- Select the contour, as shown. Be sure to chain the contour in a CW direction. Otherwise, select the Reverse button.

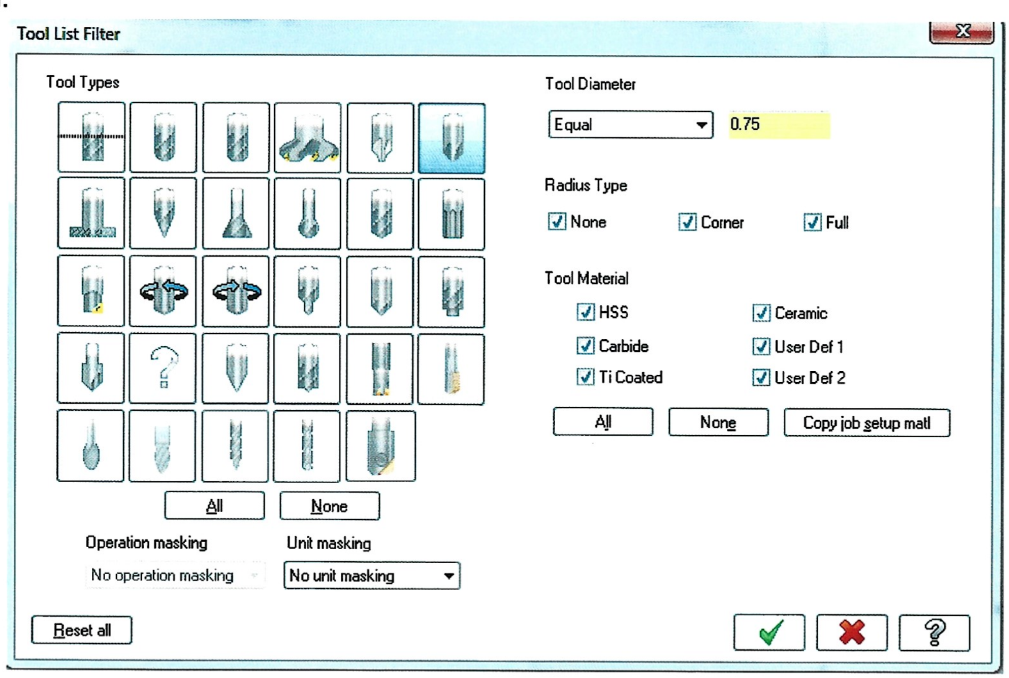


- Select the OK button to exit Chaining.

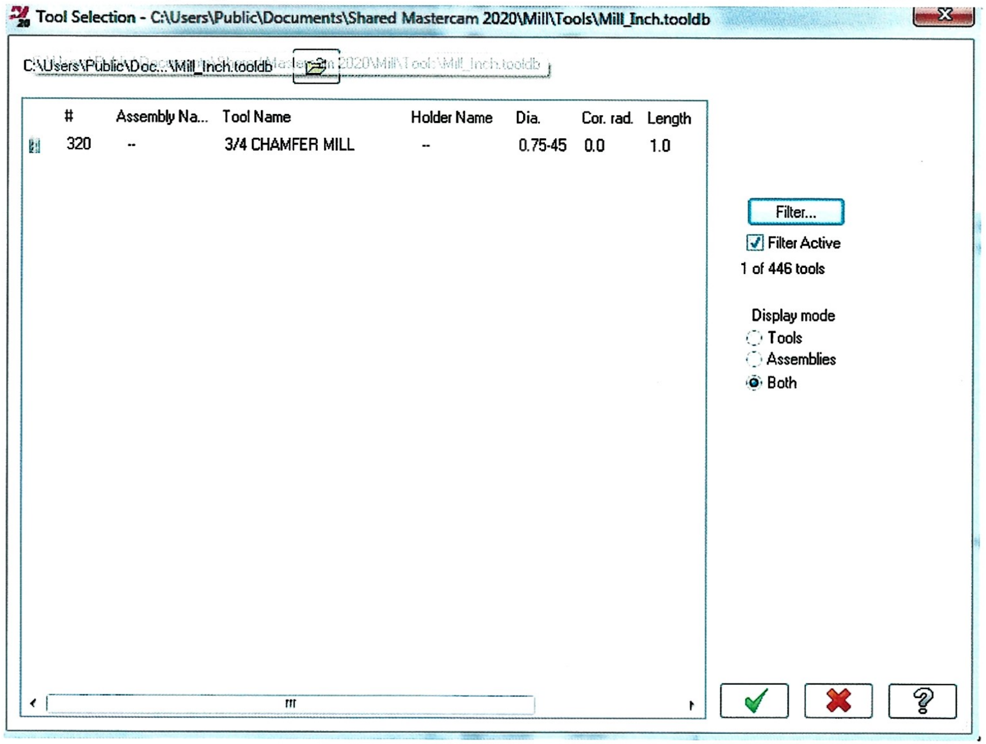
- Select the ¾" Chamfer mill from the library and set the Tool page parameters
- From the Tree view list, select Tool.
- Click on Select library tool button
- Select the Filter button.



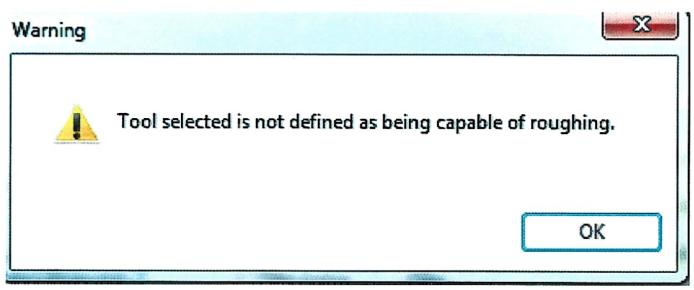
- Change the parameters to select only the chamfer mill with the diameter equal to 0.75 as shown.



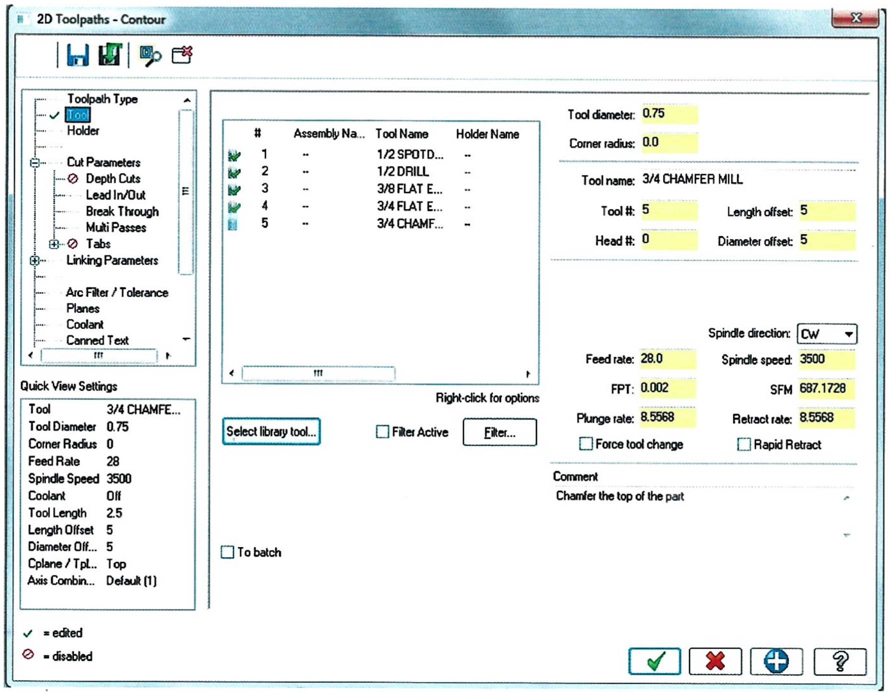
- Select the OK button to exit.
- Click on the tool to highlight it, and then select the OK button to exit Tool Selection.



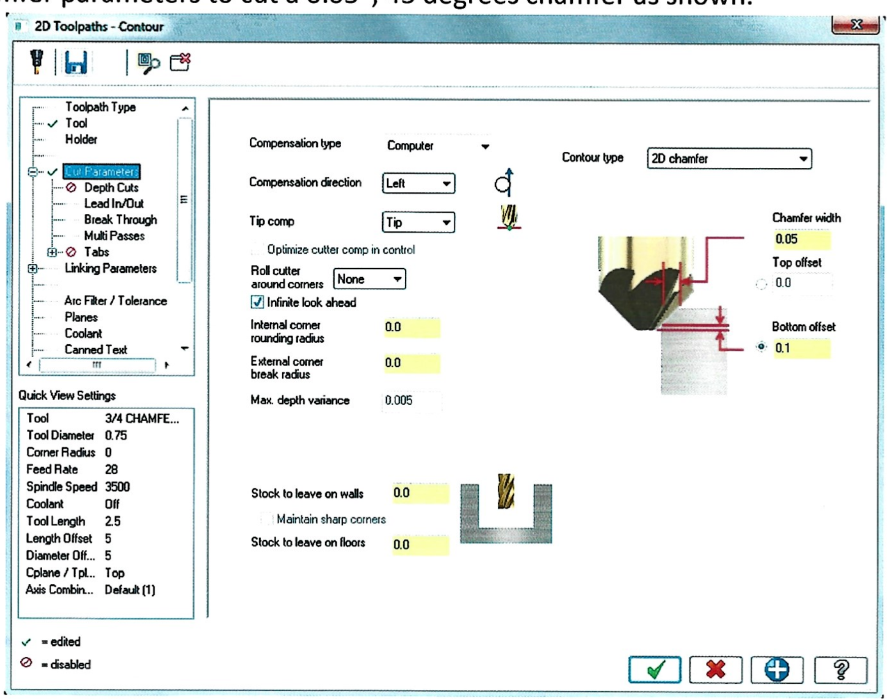
- Select the OK button to accept the warning that tells you that the tool cannot be used for both roughing and finish.



- Set the Tool page parameters as shown.

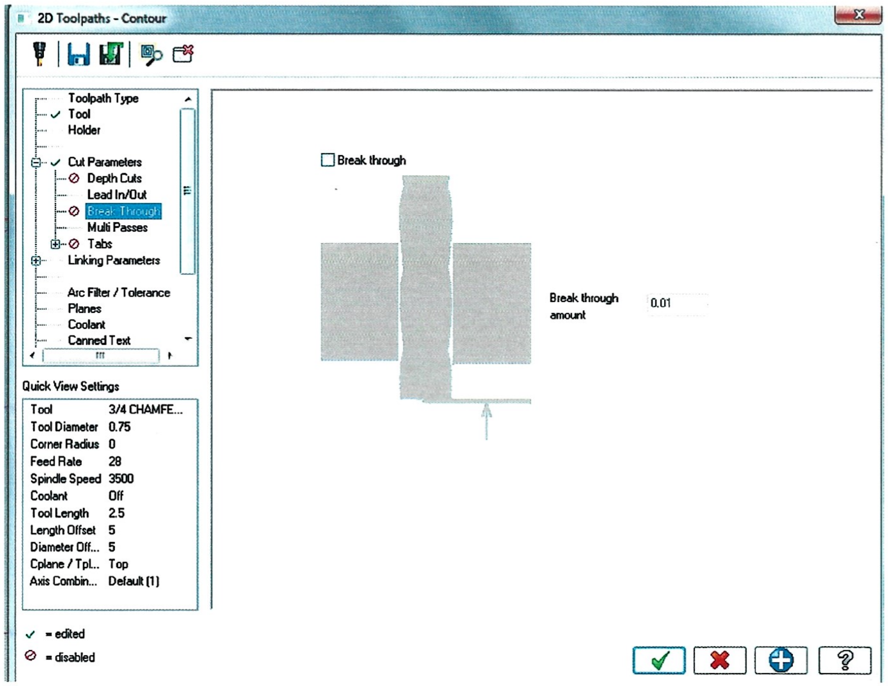


- Set the Cut Parameters: From the Tree view list, select Cut Parameters
- Click on the drop down arrow in the Contour type and select 2D chamfer
- Set the chamfer parameters to cut a 0.05", 45 degrees chamfer as shown.

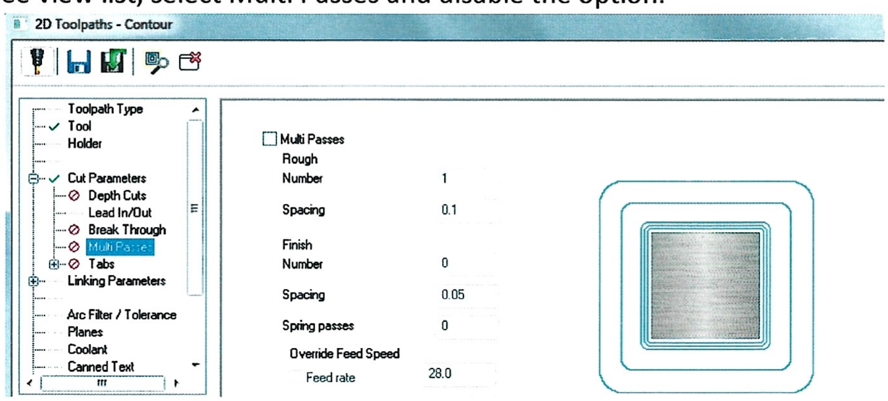


Tip offset amount ensure that the tip of the tool clear the bottom of the chamfer.

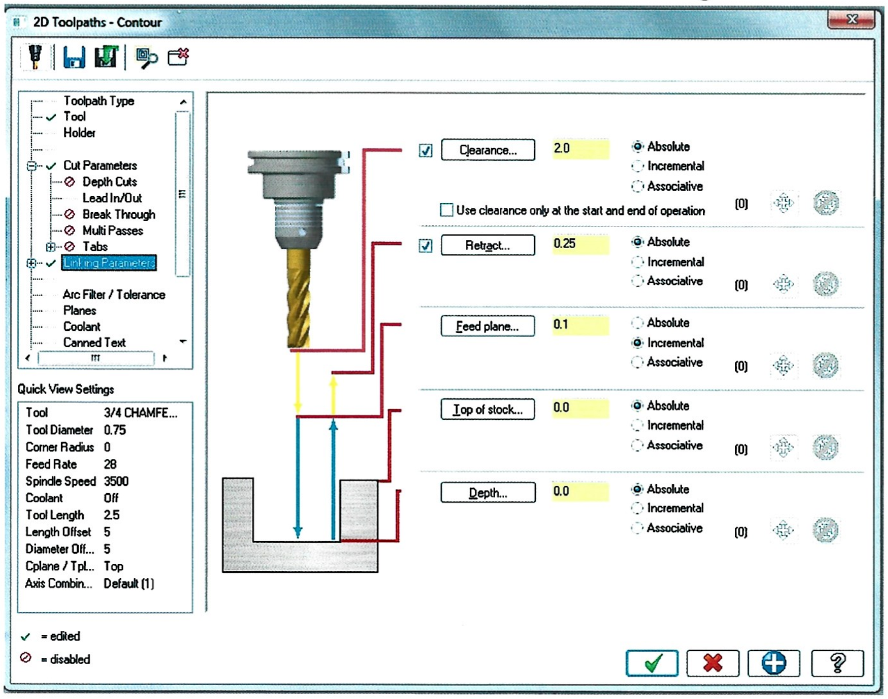
- From the Tree view list, select Break through and disable the option.



- From the Tree view list, select Multi Passes and disable the option.



- From the Tree view list, select Linking Parameters and make the changes as shown below.



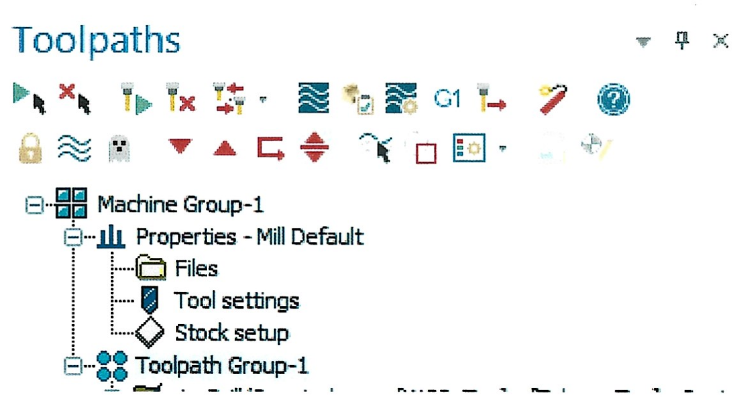
Note that the Depth is set to 0 relative to the top of the stock to let Mastercam to calculate the tool depth from the chamfer dimensions.

- Select the OK button to exit 2D Toolpaths- Contour parameters.

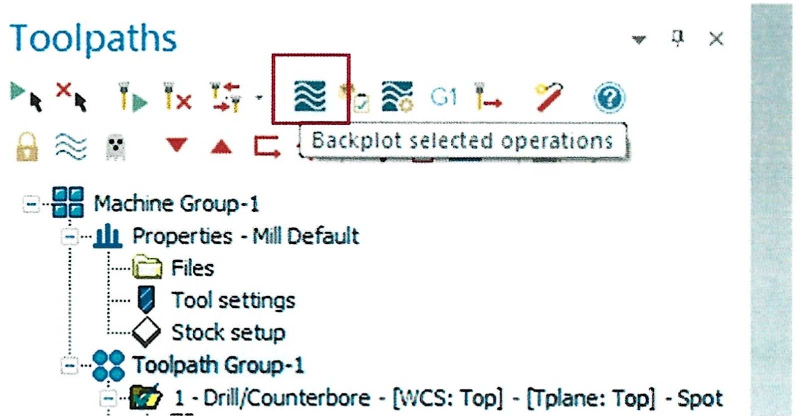
Step 18: Backplot the toolpaths

Backplotting shows the path the tools take to cut the part. This display lets you spot error in the program before you machine the part. As you backplot toolpaths, Master displays the current X, Y, and Z coordinates in the lower left corner of the screen.

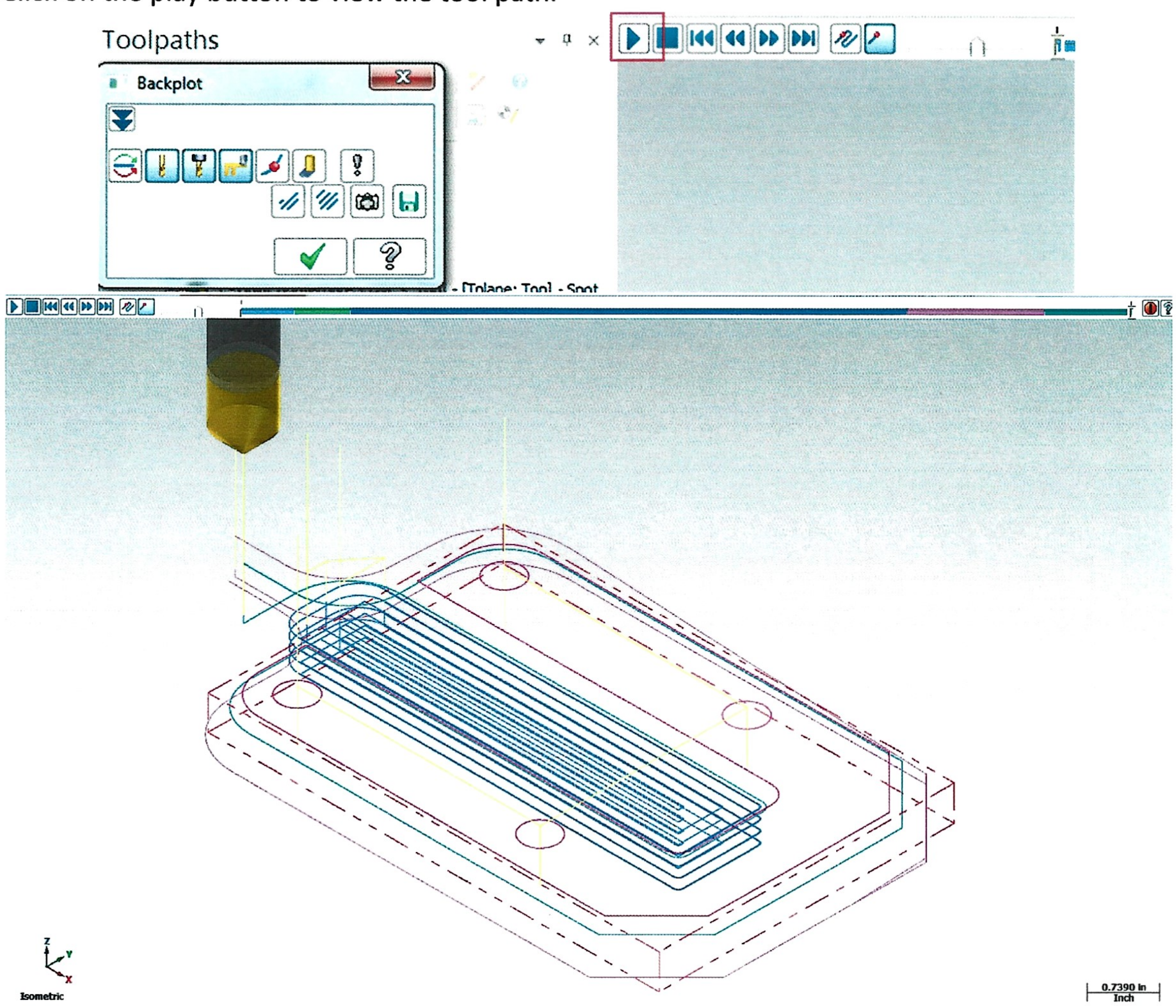
- Click on the Toolpath Group in the Toolpaths Manager to select all operation.



- Select the Backplot selected operations button.



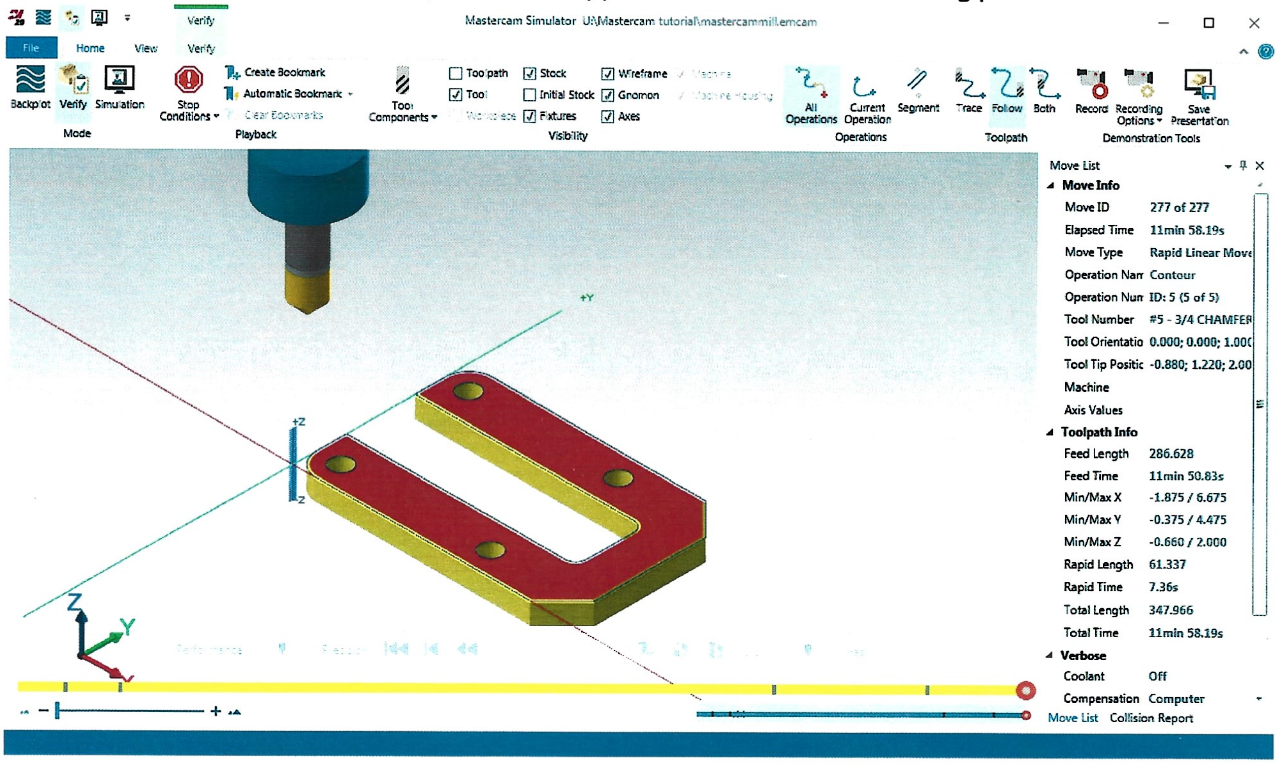
- Click on the play button to view the tool path.



Step 18: Verify the toolpath

- Select the Verify selected operations button.

- Click the play button. The finished part should appear as shown in the following picture.

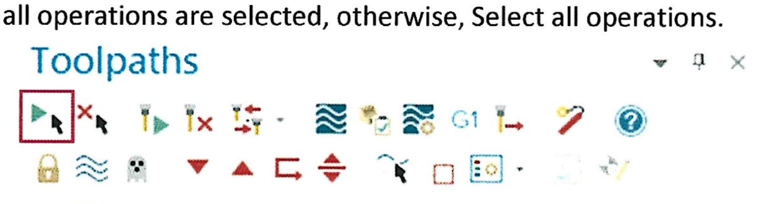


- Select the X button to exit.

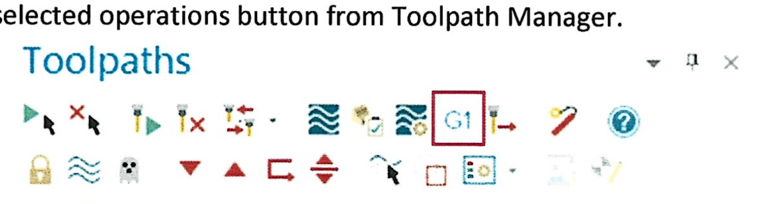
Step 19: Post the file

Post processing refers to the process by which the toolpaths in the Mastercam part files are converted to a format that can be understood by the machine tool's control such as G-codes. Generally, every machine controller will require its own post processor, customized to produce code formatted to meet its exact requirements.

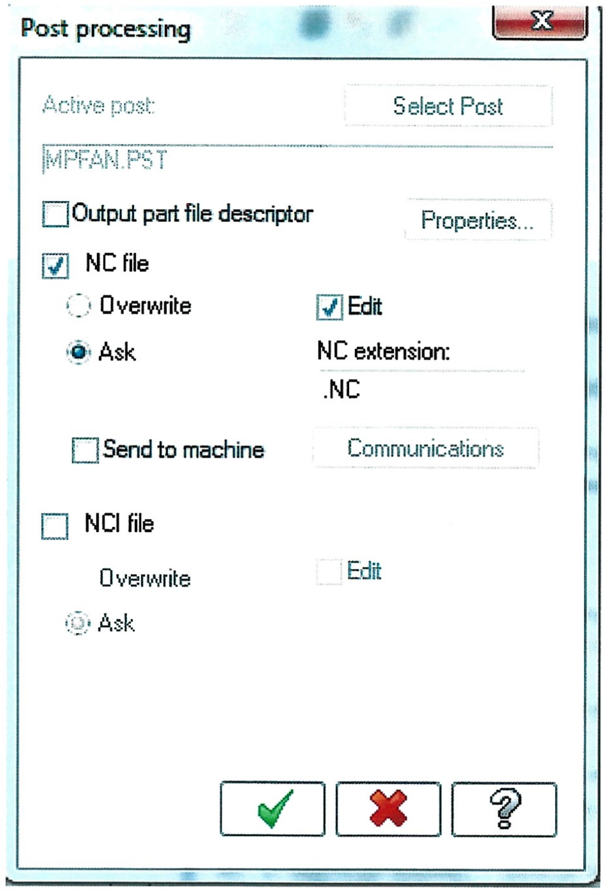
- Make sure that all operations are selected, otherwise, Select all operations.



- Select the Post selected operations button from Toolpath Manager.

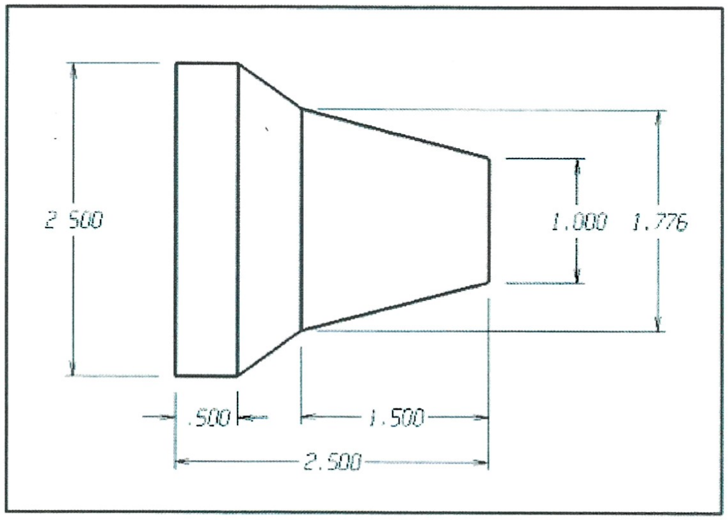


- In the Post processing window, make all the necessary changes as shown. Select the OK button to continue.



- Save the NC file to you own personal folder.
- Go to file and click save
- This completes the tutorial for the milling operation

Lathe Part preview



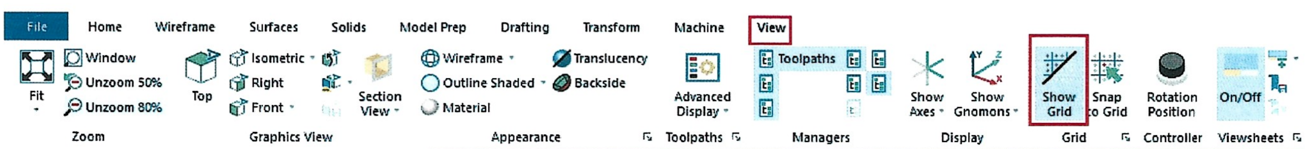
- Go to file and click new.

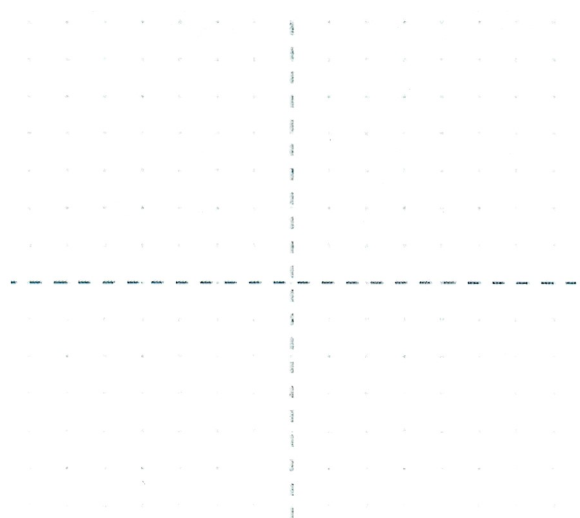
Step 0: Setting the grid

Before starting the geometry enable the Grid. It will show you where the part origin is.

View

- From the grid group, select show grid

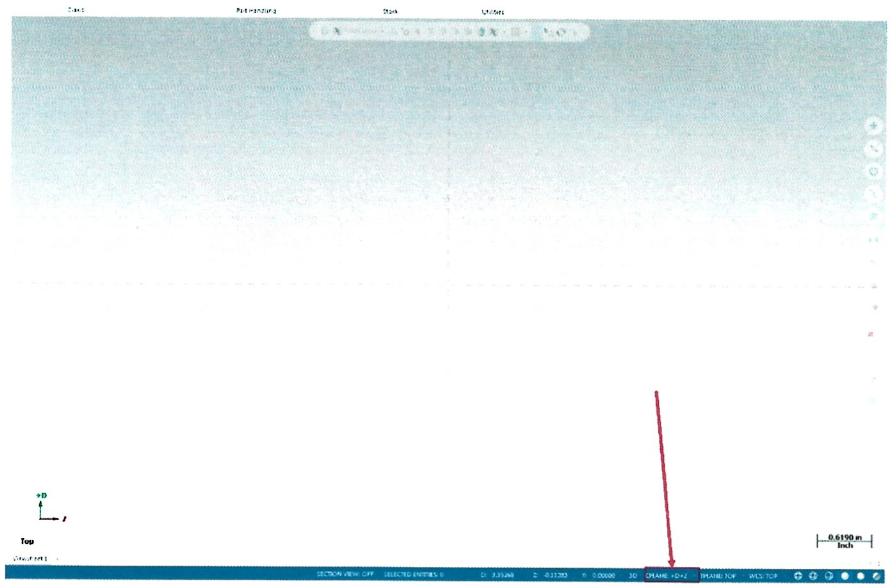




Step 1: Geometry creation

- Select Default lathe from the Machine tab

- Verify that the C plane changed to +D +Z (It is in the bottom of the window as shown below)



The plane settings +D+Z apply to construction methods for geometry requiring two axes of motion, the Z axis and a diameter value (the D value represents the diameter of X). Set the Cplane to +D+Z and Mastercam interprets X axis values as diameter values (as opposed to radius values)

- In wireframe tab and shapes group select Rectangle

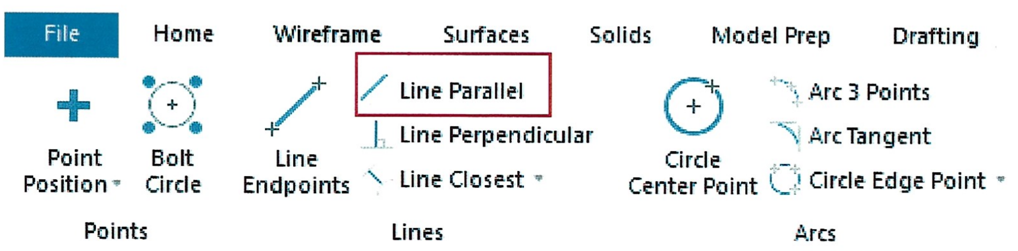
- Select the position for the base point: Select the center location of the grid (the origin)
- Enter width and the height as shown below.

- Select the Ok to exit the Rectangle dialog box.

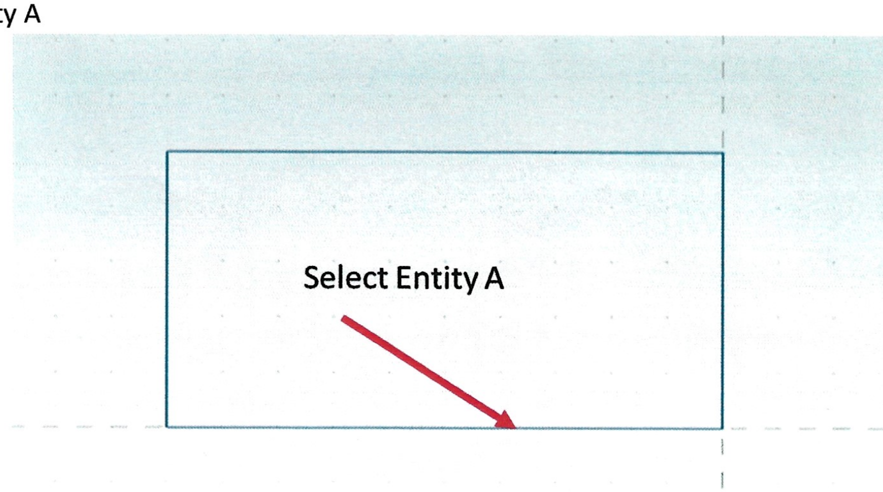
Step 2: Create parallel lines

Wireframe

- From the lines group, select line parallel



- Select Entity A



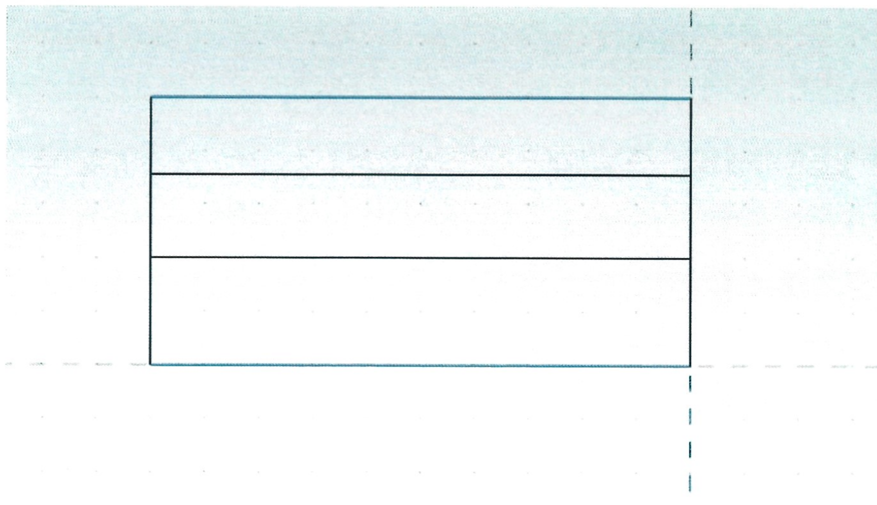
- [Select the point to place a parallel line through]: Pick a point above the selected line.

Note: that the color of the geometry is cyan which means that the entity is "live" and you can still change

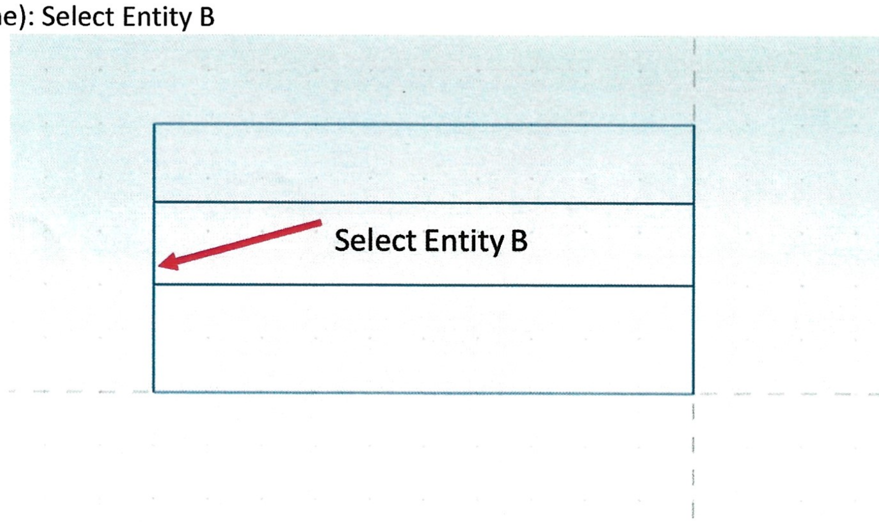
- Enter the Offset Distance 0.5 {Press Enter).
- Select the Apply button to continue.

- [Select a line): Select Entity A again.
- [Select the point to place a parallel line through]: Pick a point above the selected line.
- Type the Offset Distance 1.776/2 (Enter)
- Select the Apply button to continue.

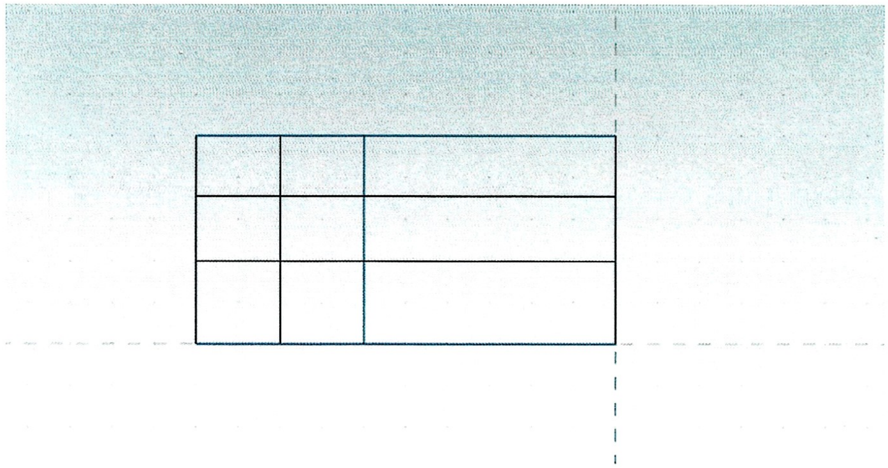
The drawing should look as shown.



- [Select a line): Select Entity B



- [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
- Type the Offset Distance 0.5 (Enter)
- Select the Apply button to continue.
- [Select a line): Select Entity B again.
- [Select the point to place a parallel line through]: Pick a point to the right of the selected line.
- Type the Offset Distance 1.0 (Enter)
- The drawing should look as shown.

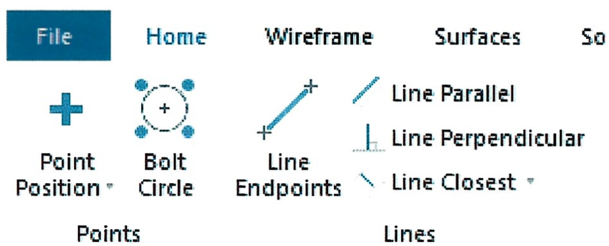


- Select the Ok to exit.

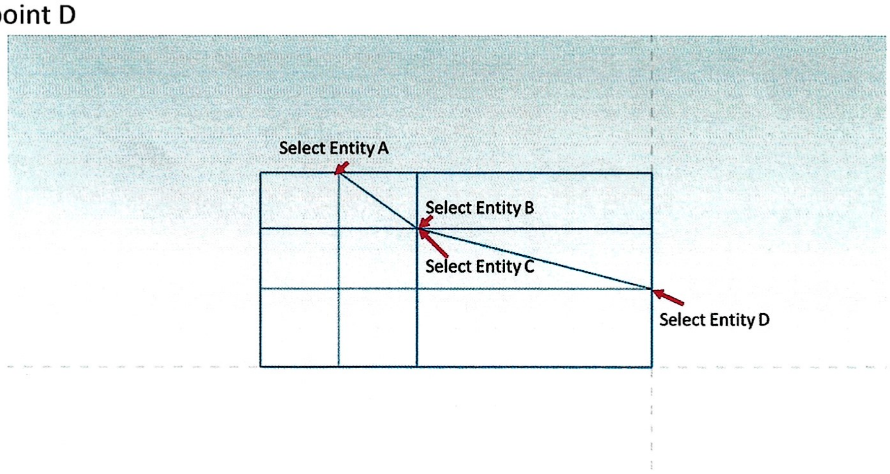
Step 3: Create lines knowing the endpoints

Wireframe

- From the lines group, select Line Endpoints



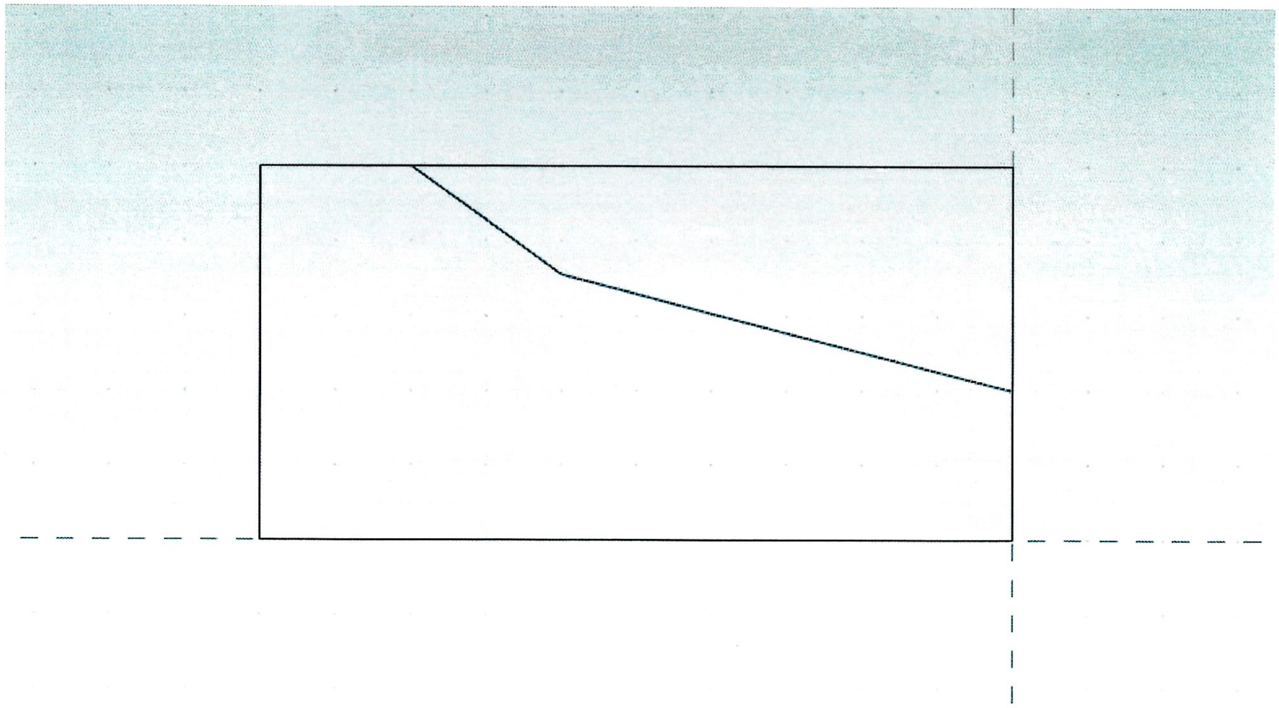
- Select Endpoint A
- Select Intersection point B
- Select the Apply button to continue.
- Select Endpoint C
- Select Endpoint D



- Select the OK button to exit the command.

Step 4: Delete the construction lines

- Right click on the lines and use the delete entity function to create the identical geometry as shown below.

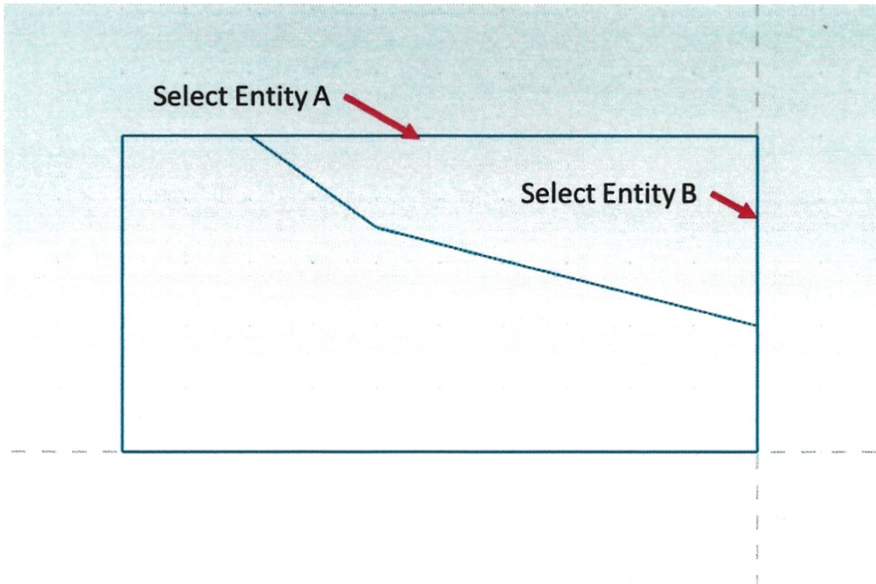


Step 5: Trimming the geometry using trim divide command

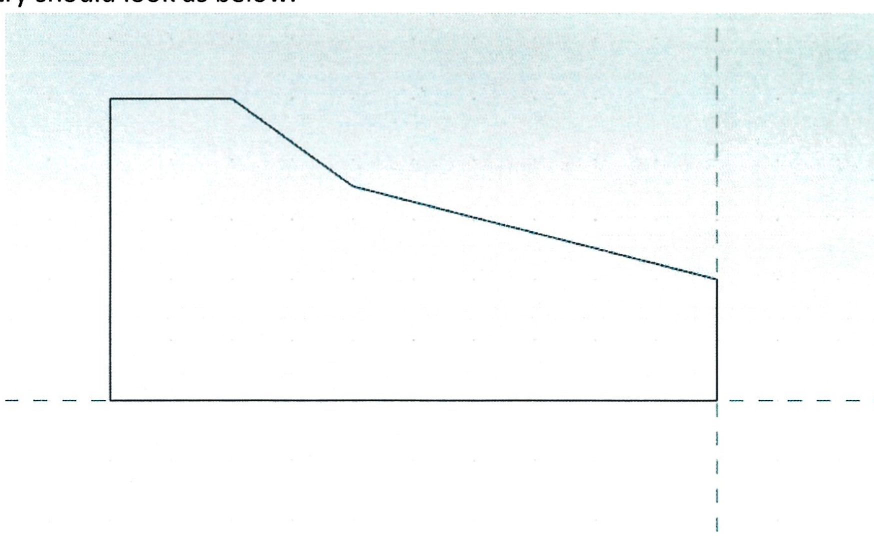
Wireframe

- From the Modify group, select Divide 

.
- [Select the entity to trim/extend]: Select the entity A and entity B (select the entities exactly as shown in the drawing).



- Select the OK button .
- The geometry should look as below.

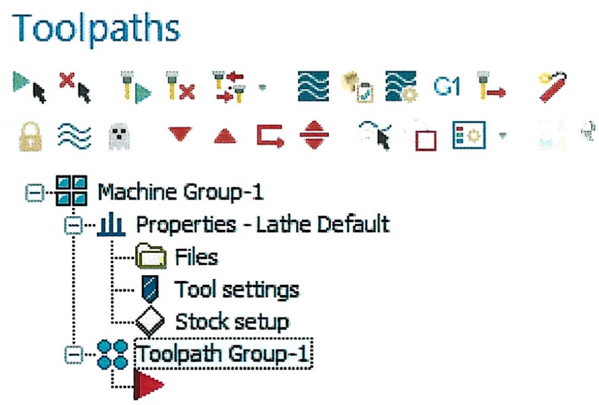


Step 6: Save the file

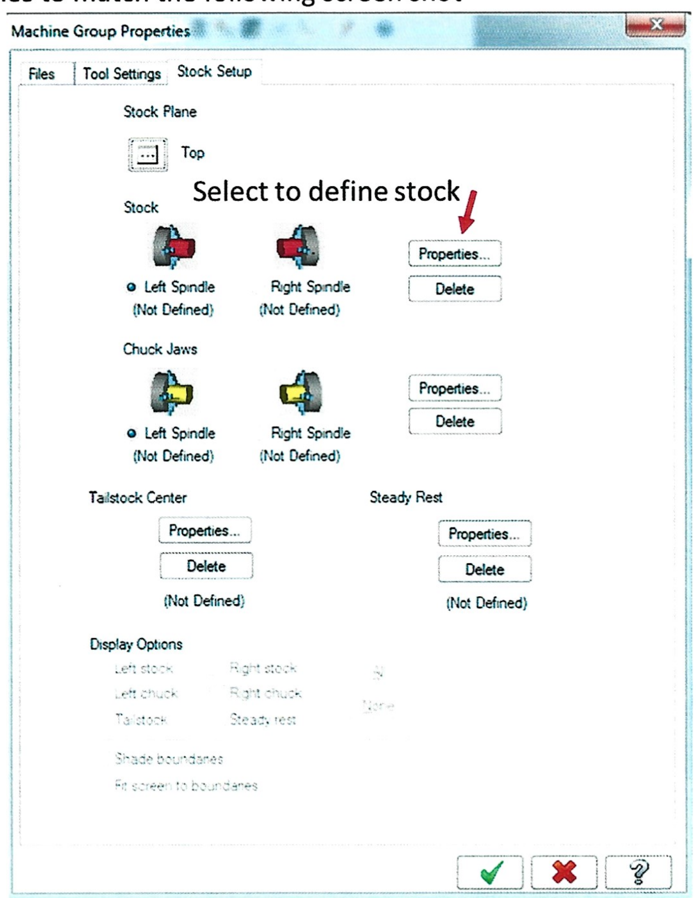
- Go to file and click save as.
- File name: "Your Name_1"

Step 7: Set up the stock to be machined

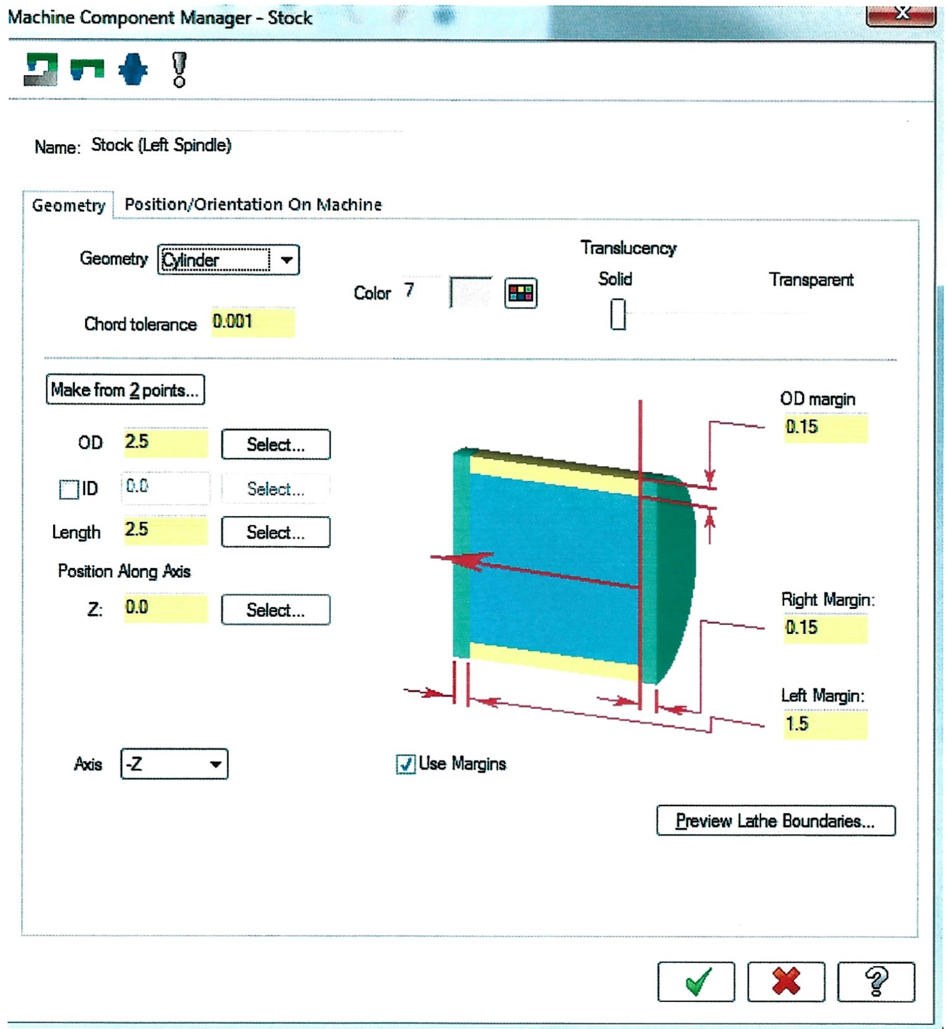
- To display the Toolpaths Manager press Alt+O
- Select the plus in front of Properties to expand the Toolpath Group Properties



- Select Stock setup
- Change the properties to match the following screen shot

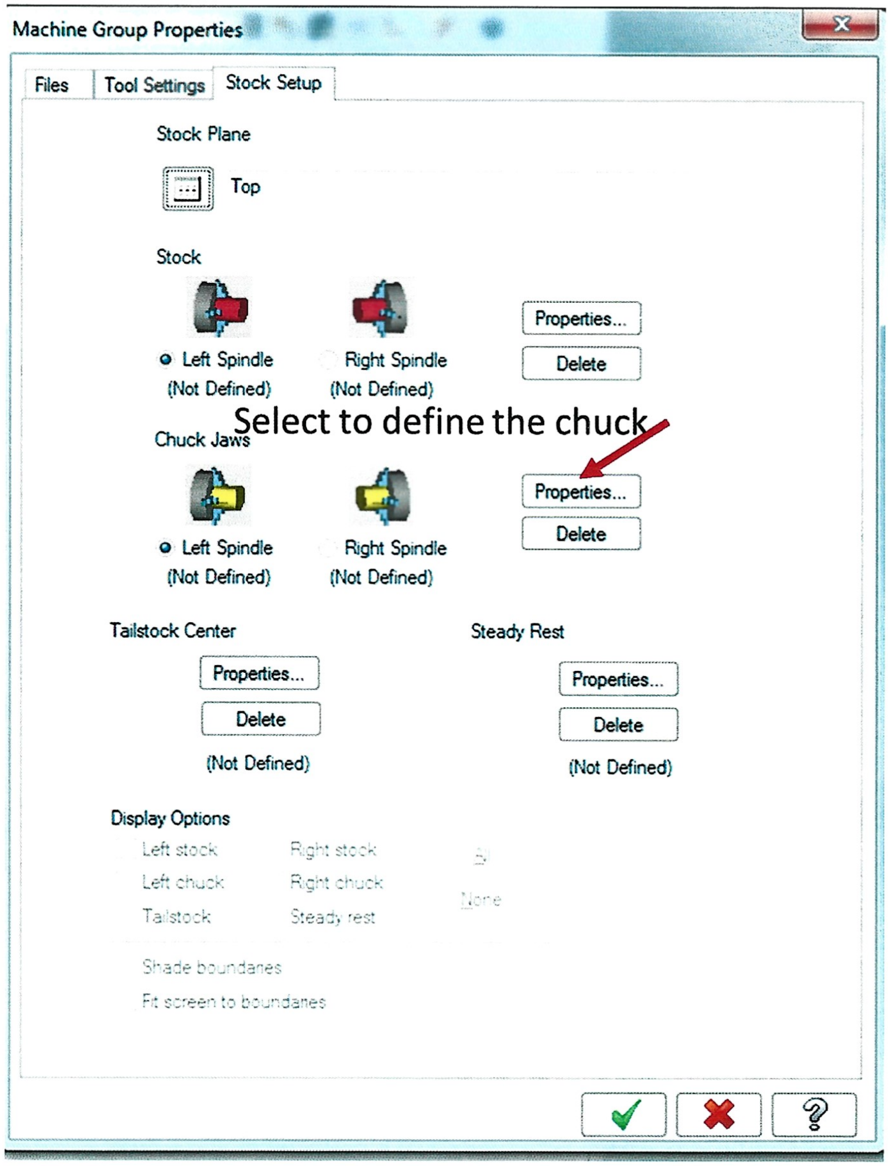


- Select the Properties button in the Sock area to establish the stock size and match the following screen shot

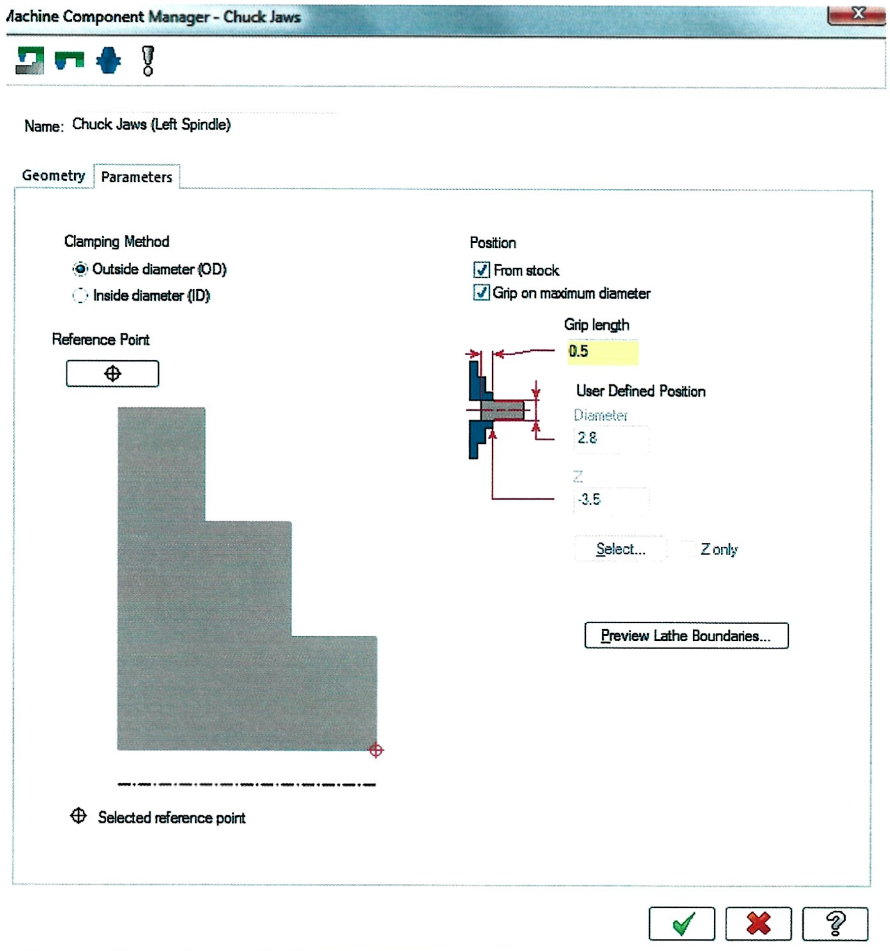


Enable Use Margin to be able to set the following parameters; extra stock to the OD, face, and back of stock.

- Select the OK button to exit the Machine Component Manager-Stock dialog box.
- Select the Properties button in the Chuck area.


Machine Group Properties
Files
Tool Settings
Stock Setup
Stock Plane
Top
Stock
Left Spindle
(Not Defined)
Right Spindle
(Not Defined)
Properties...
Delete
Select to define the chuck
Chuck Jaws
Left Spindle
(Not Defined)
Right Spindle
(Not Defined)
Properties...
Delete
Tailstock Center
Properties...
Delete
(Not Defined)
Steady Rest
Properties...
Delete
(Not Defined)
Display Options
Left stock
Right stock
Left chuck
Right chuck
Tailstock
Steady rest
None
Shade boundaries
Fit screen to boundaries

- Make the necessary change to define the stock position. (Click parameters tab)



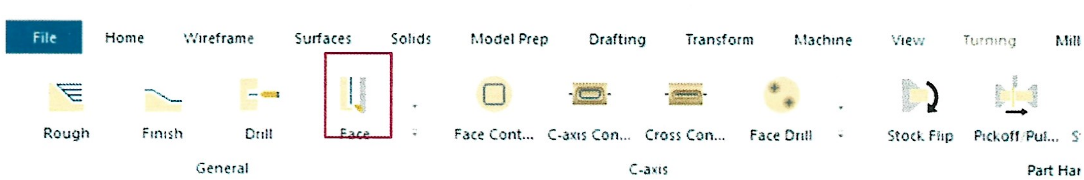
- Select the OK button to exit.
- Select the OK button again to exit the Machine group properties

Step 8: Face the part

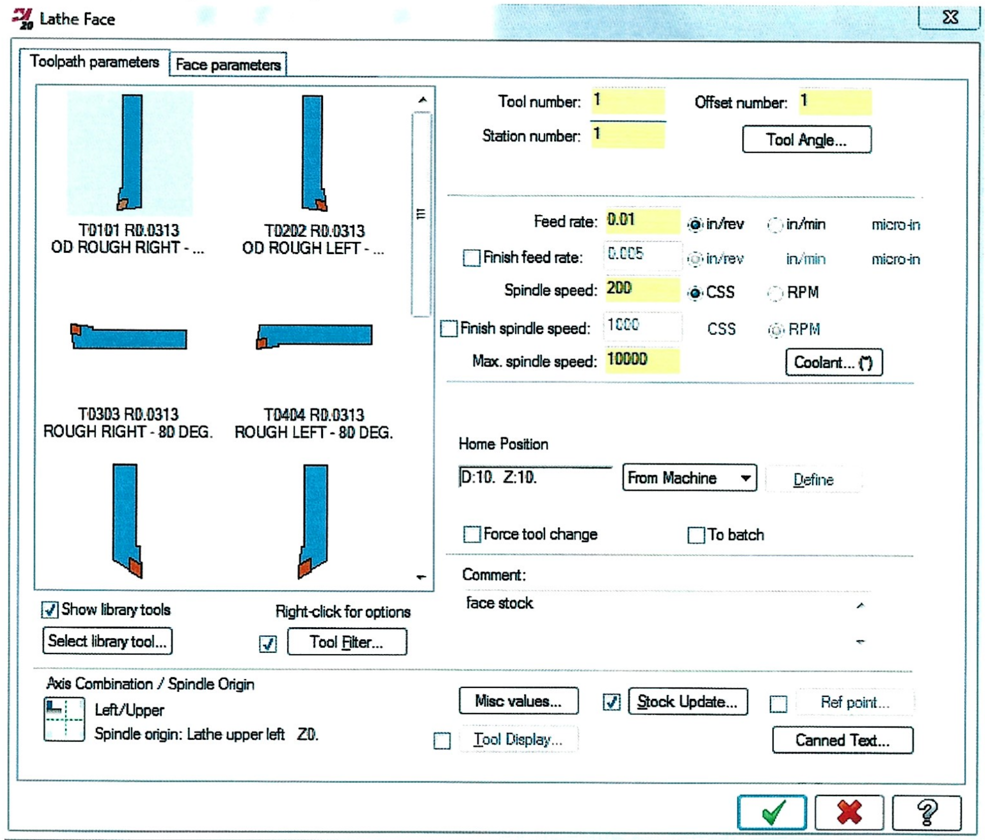
Face Toolpath: Allows the user to quickly clean the stock from one end of the part, and create an even surface for future operations.

Toolpaths

- Click Face from turning tab

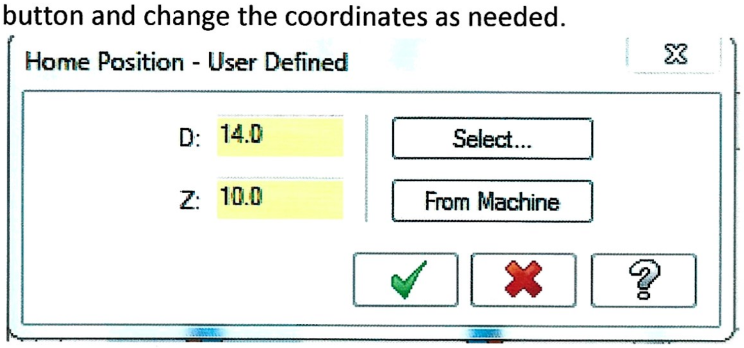


- Select the OD rough Right -80 deg cutter from the library list.

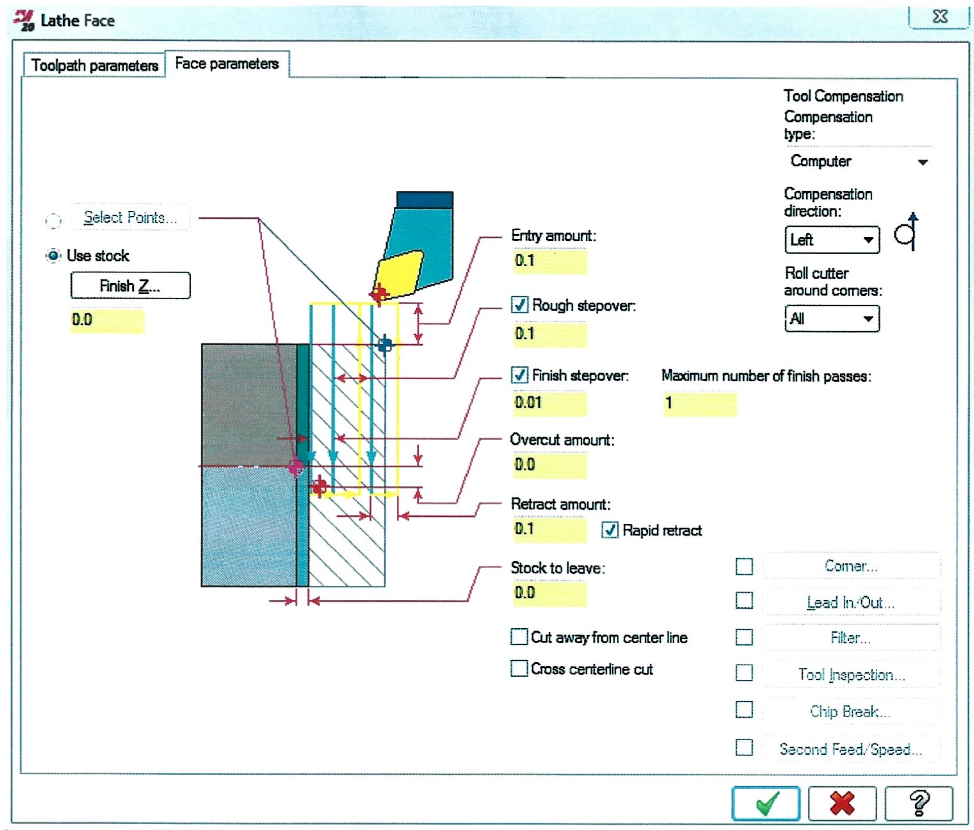


The Feed rate and the Spindles Speeds are based on Mastercam Tool definition. They can changed at any time, based on the material that you going to machine.

- To change the Home position coordinates (the position where the turret changes the tool) click on the drop down arrow next to From Machine.
- Select from the pull-down list select User defined.
- Select the Define button and change the coordinates as needed.

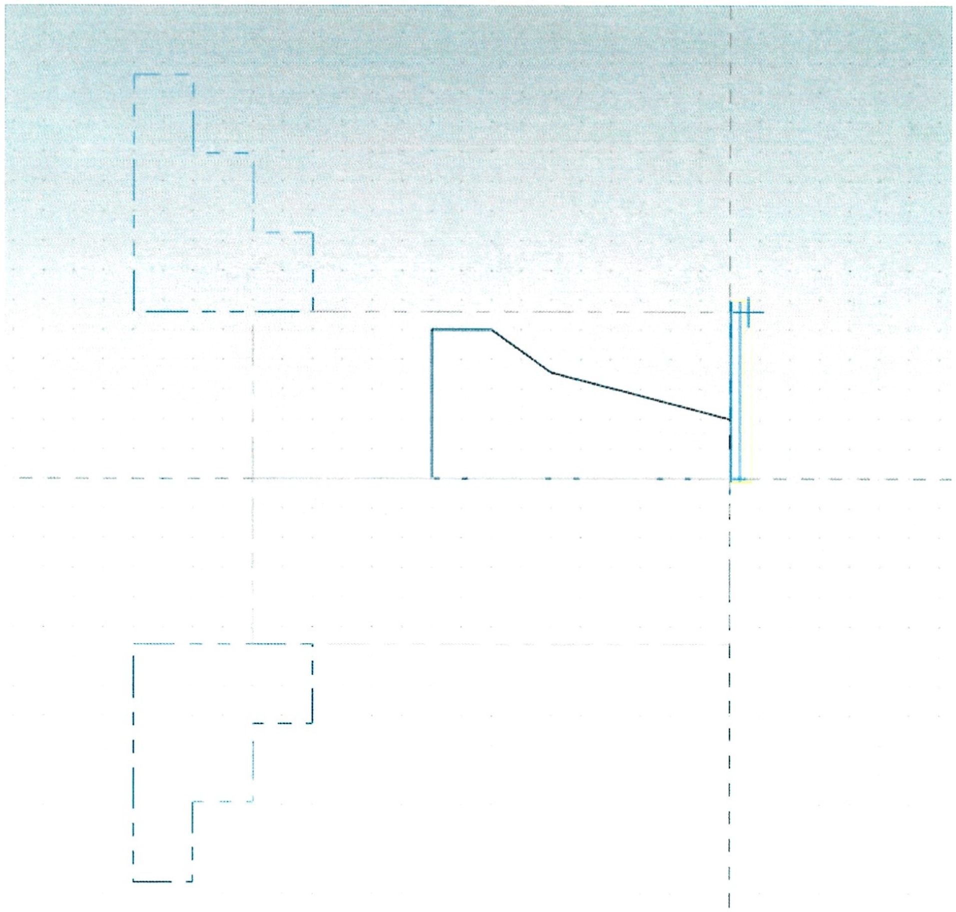


- Select the OK button.
- Select the Face Parameters page and make all the necessary changes as shown in the screenshot.



Entry amount value sets the height at which the tools rapid to or from the part. Rough stepover value sets the roughing pass value. Finish stepover value sets the finish pass value. Overcut amount determines how far past the center of the part the tool will cut. Retract amount determines the distance the tool moves away from the face of the part before it moves to the start of the next cut. Stock to leave sets the remaining stock after the tool completes all passes. Cut away from center line sets the tool to start cutting closest to the center line and cut away from the center line at each pass.

- Once complete select the OK button to exit Face properties.

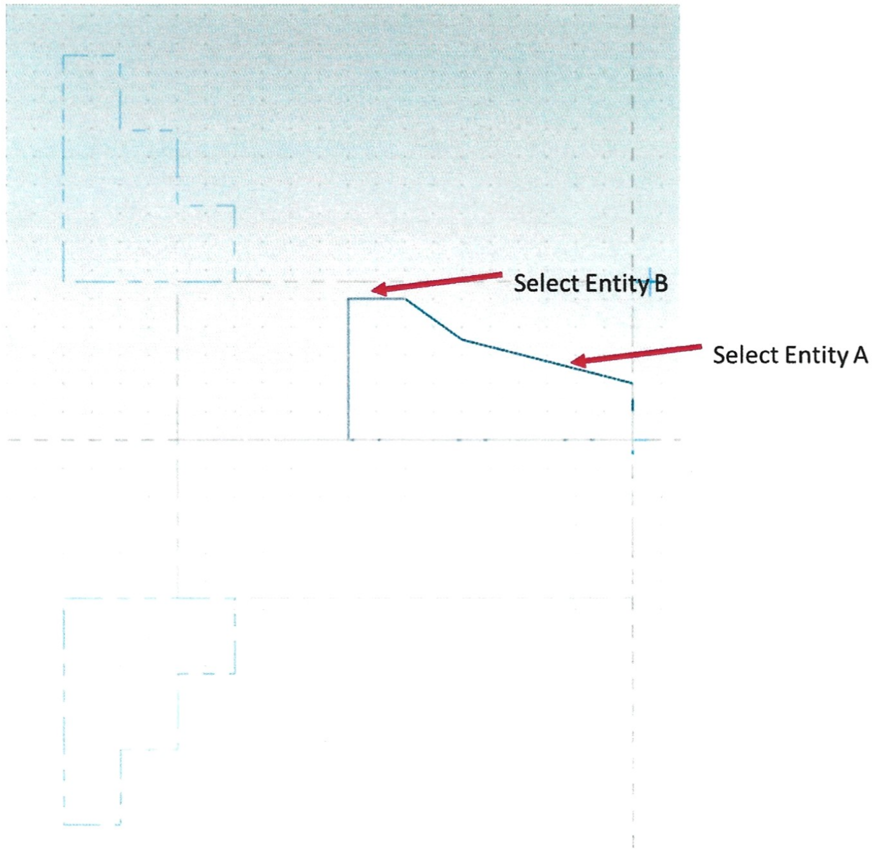


Step 9: Rough the part

- Rough Toolpath: Quickly remove large amounts of stock in preparation for a finish pass. Roughing passes are typically straight cuts parallel to the Z-axis

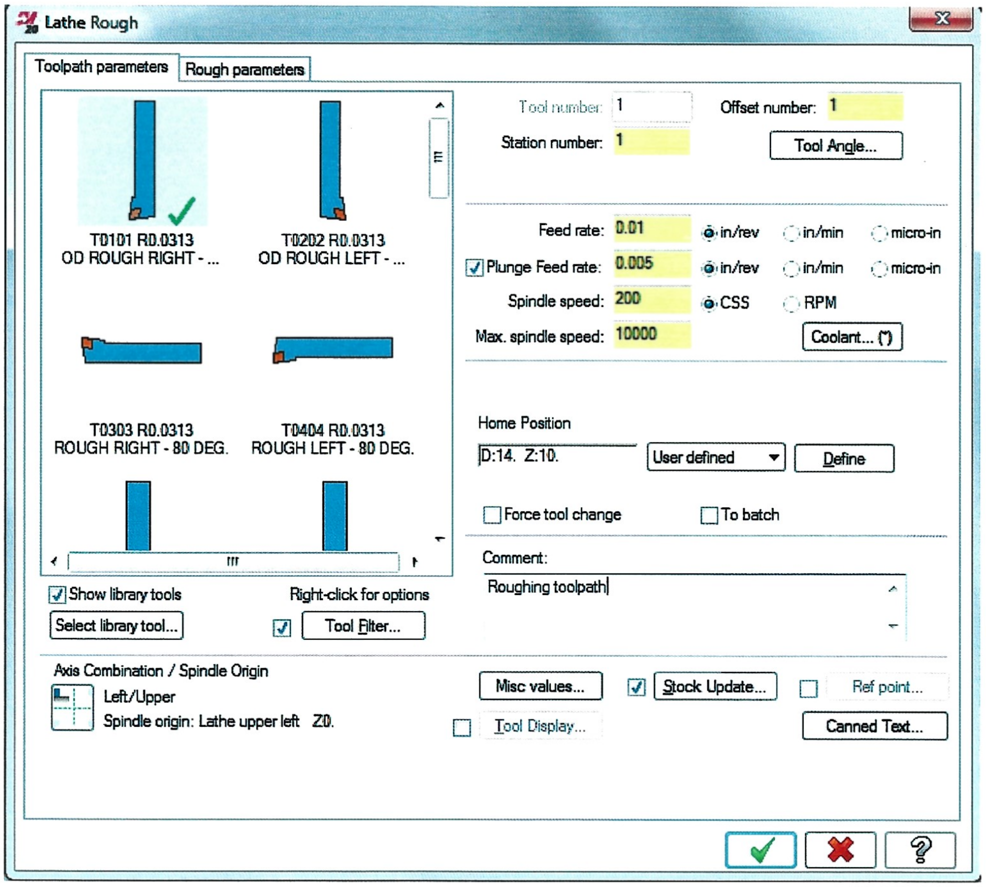
Toolpaths

- Click Rough from turning tab
- Chaining mode is partial by default. You will have to select the first entity and the last entity of the contour.
- Select entity A (Make sure that the chaining direction is CCW, otherwise select the reverse button from Chaining dialog box.

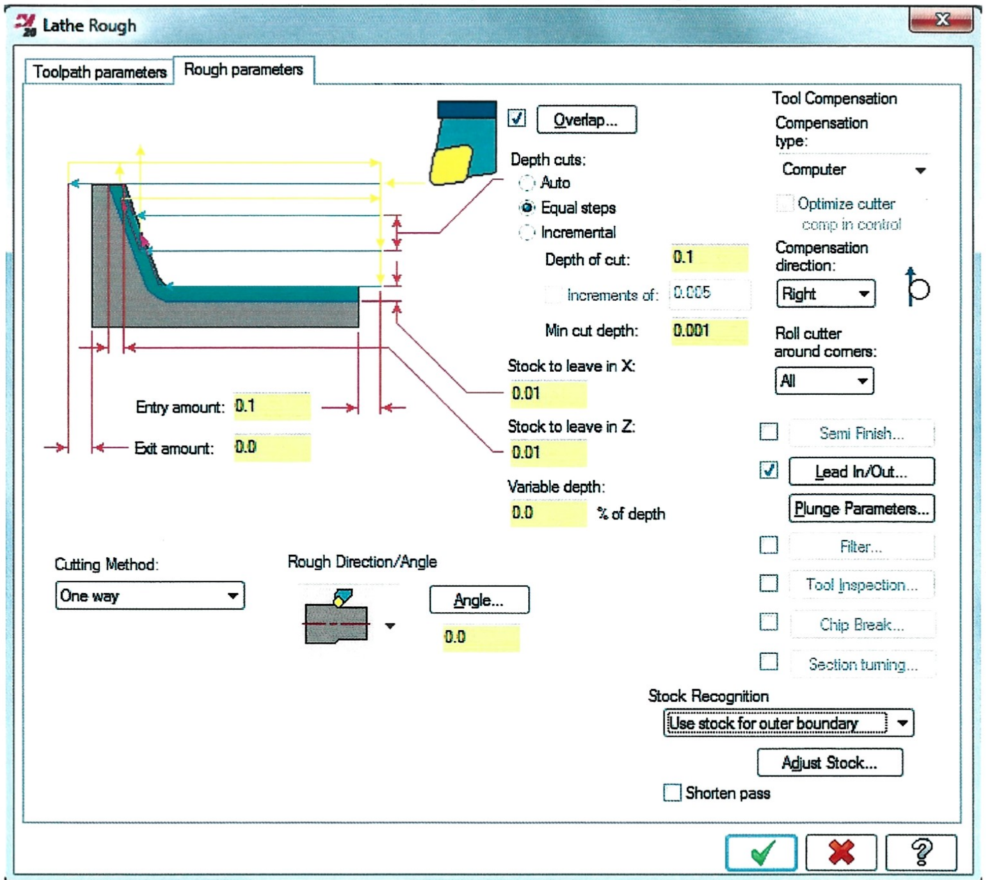


- Select entity B
- Select the OK button to exit Chaining dialog box.

- In Tool path Parameters select the same tool and make all the necessary changes as shown in the screenshot.

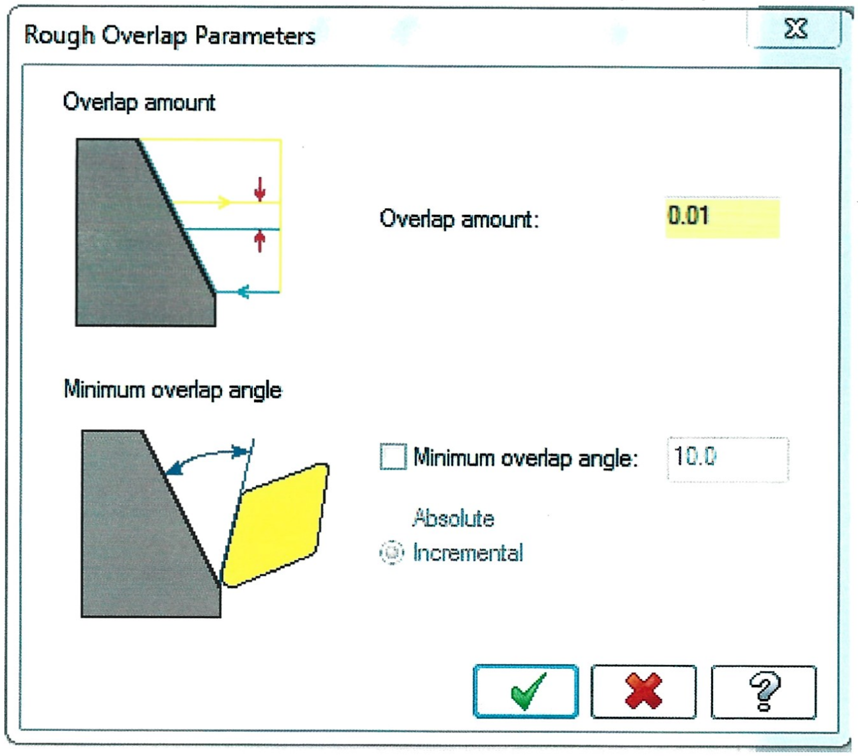


- Select the Rough Parameters tab and make any necessary changes as shown.

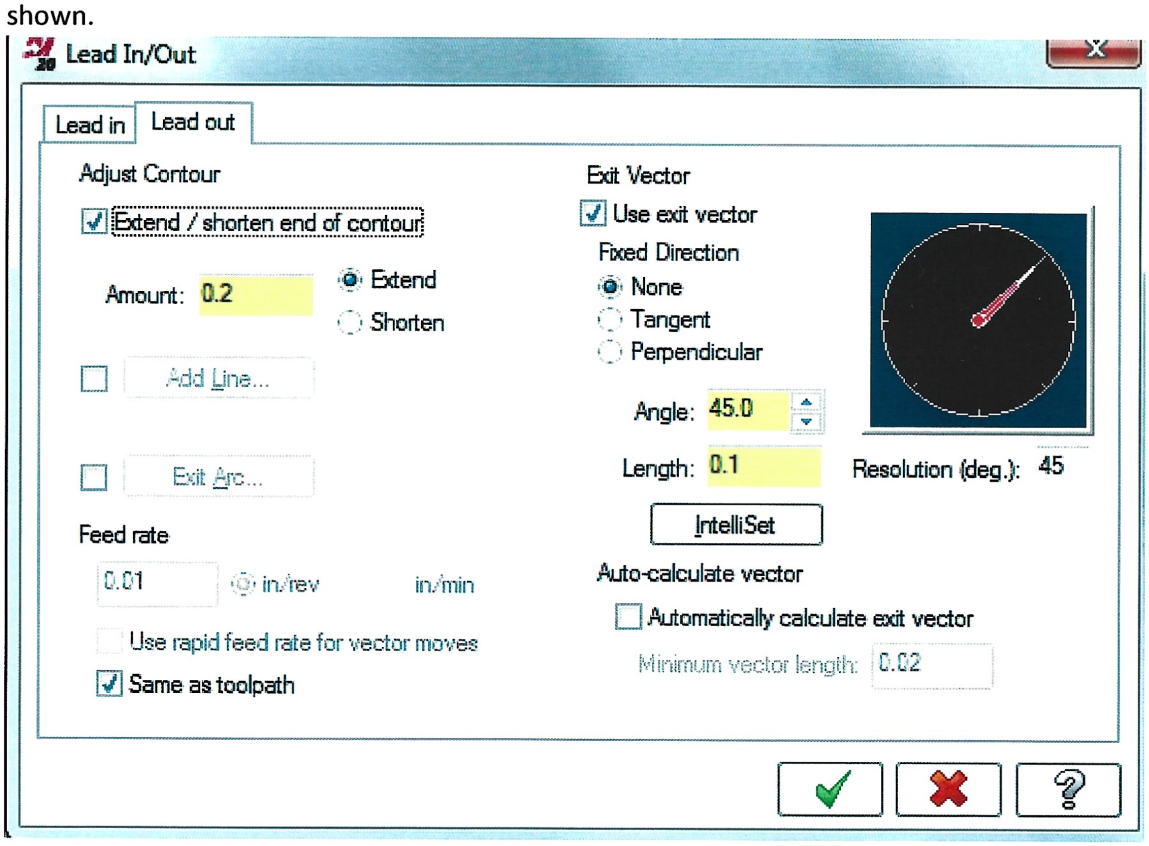


Depth of cut sets the amount of material to be removed during each pass. Equal steps sets the Depth of cut value to the maximum amount of material that the tool can remove at each pass to ensure equal passes. Minimum cut depth value sets the minimum cut that can be taken per pass. Stock to leave in X,Y value sets the remaining stock in the X, Y axis after the tool completes all passes. Entry amount value sets the height at which the tool rapids to or from the part.

- Select the overlap button to establish how much the tool overlaps the previous cut.



- Select Lead In/Out button and choose the Lead out tab to extend the end of the contour as shown.



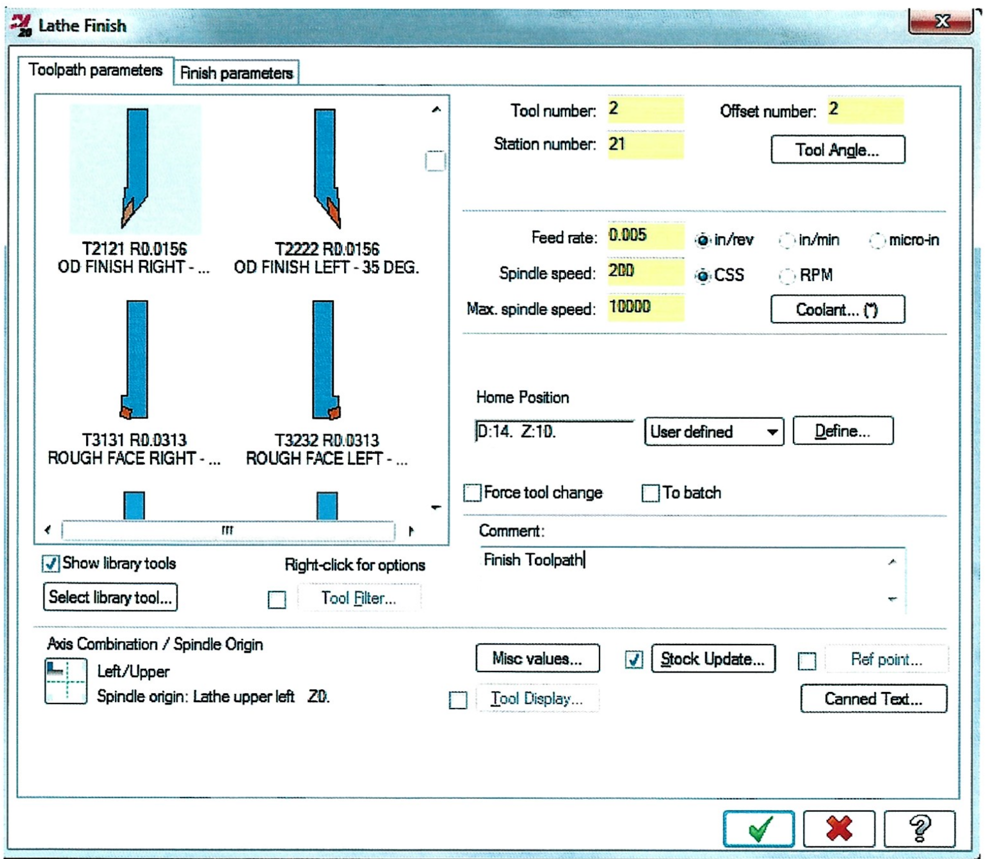
- Select the OK button to exit the Lead In/Out parameters

- Select the OK button to exit the Roughing Toolpath Properties.

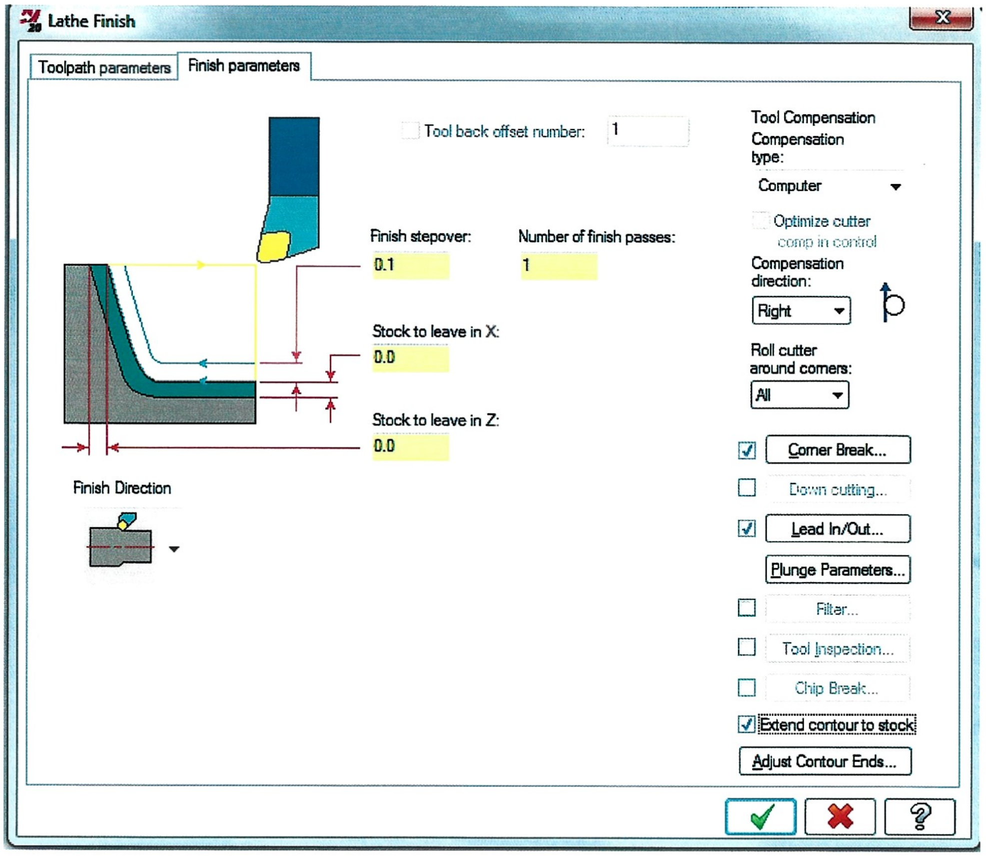
Step 9: Finish the part

Toolpaths

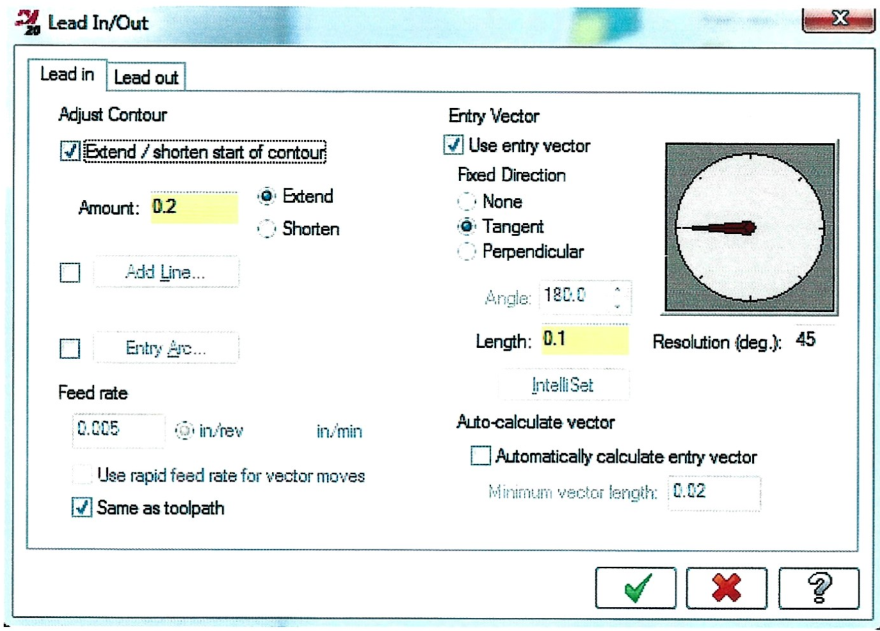
- Click Finish from turning tab
- Select Last button in the Chaining dialog box.
- Select OK.
- Select the OD Finish Right -35deg cutter from the tool list.



- Select the Finish parameters tab and make all the necessary changes as shown in the following screenshot.



- Select the Lead in/out button and change the Fixed Direction as shown in the Lead in tab.



Lead In parameters allow you to set the option for controlling how the tool approaches the part at the start of each pass in the toolpath.

Extend/Shorten start of the contour allows you to extend/shorten the geometry in the chained contour.

Add Line allows you to add a line to the start of the chained contour.

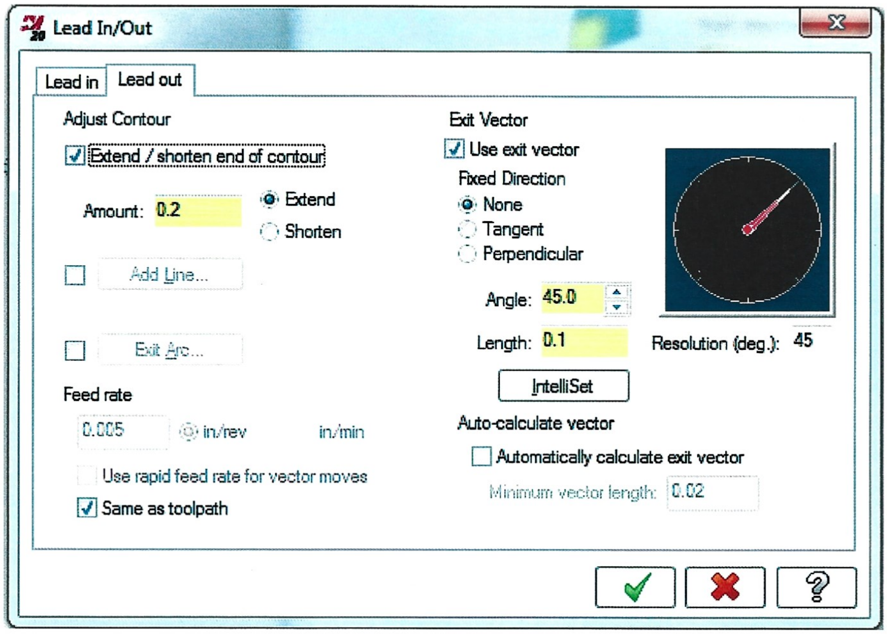
Entry Arc allows you to create a tangent arc move to the start of the toolpath.

Entry Vector allows you to define a vector by entering an angle and length when you do not use any **Fixed Direction**.

Tangent will create a line tangent to the first entity of the chained geometry when using a **Length** value.

Automatically calculates entry vector lets the system calculate an entry vector for you.

- Select the Lead out button and Extend the end of the contour with 0.2 as shown in the previous step.

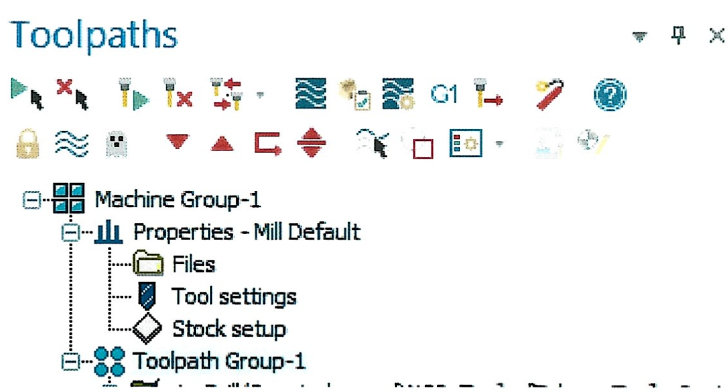


- Select the OK button twice to exit the Finish Parameters.

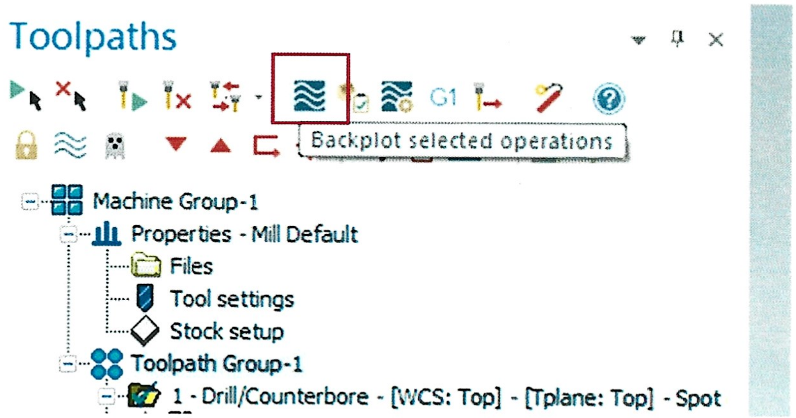
Step 10: Backplot the toolpaths

Backplotting shows the path the tools take to cut the part. This display lets you spot error in the program before you machine the part.

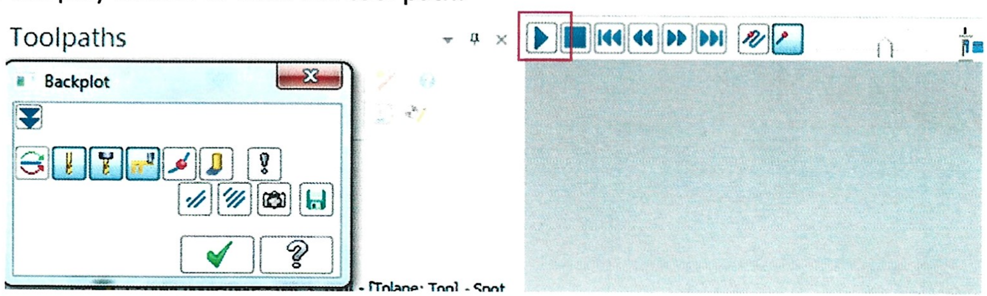
- Click on the Toolpath Group in the Toolpaths Manager to select all operation.

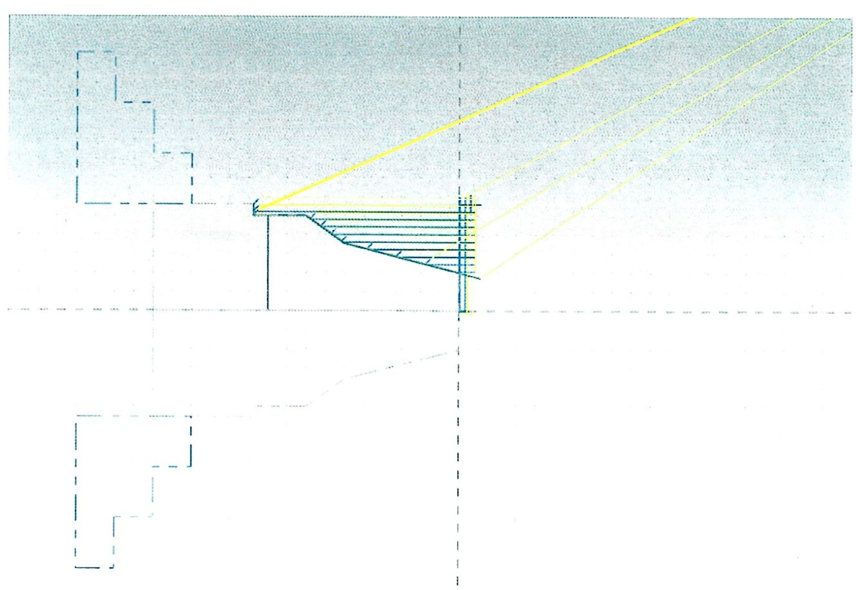


- Select the Backplot selected operations button.



- Click on the play button to view the tool path.

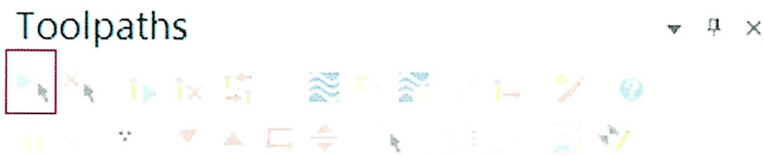




Step 11: Verify the toolpath

Verify simulate the machining of a part from a stock model display. The stock dimension are based on the values that we specified in the Stock Setup.

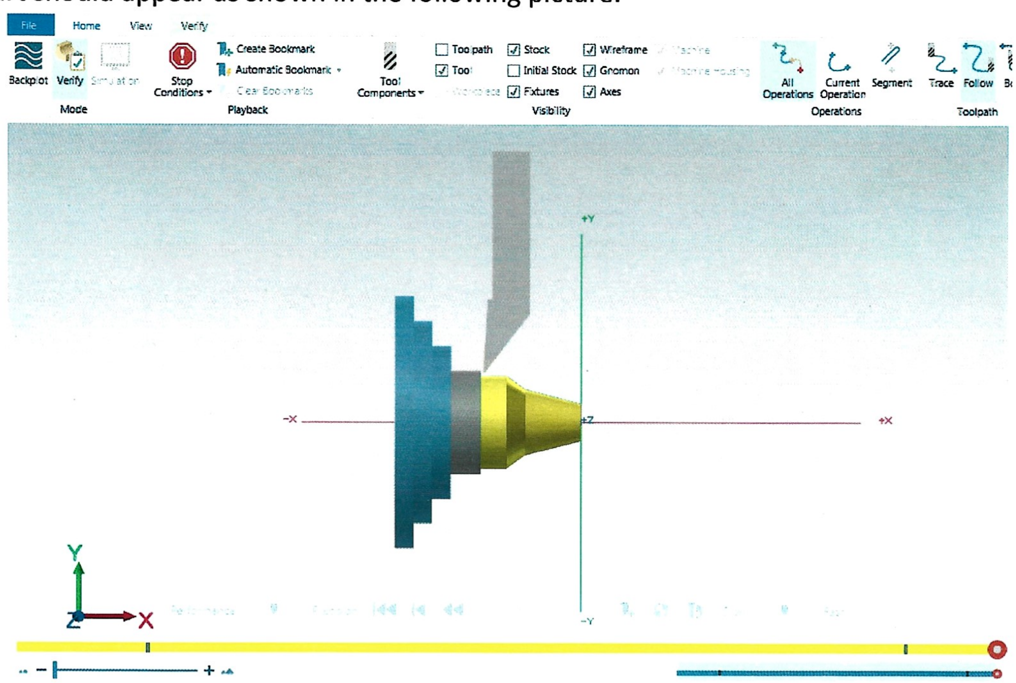
- Click on Select all operations icon



- Select the Isometric view from the view toolbar to see the stock
- Select the Fit button
- Select the Verify Selected operations button.

- Set the Verify speed by moving the slider bar in the speed control bar.

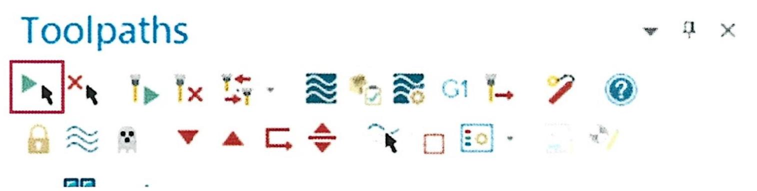
- Select the play button to start simulation.
  The part should appear as shown in the following picture.



Step 12: Post the file

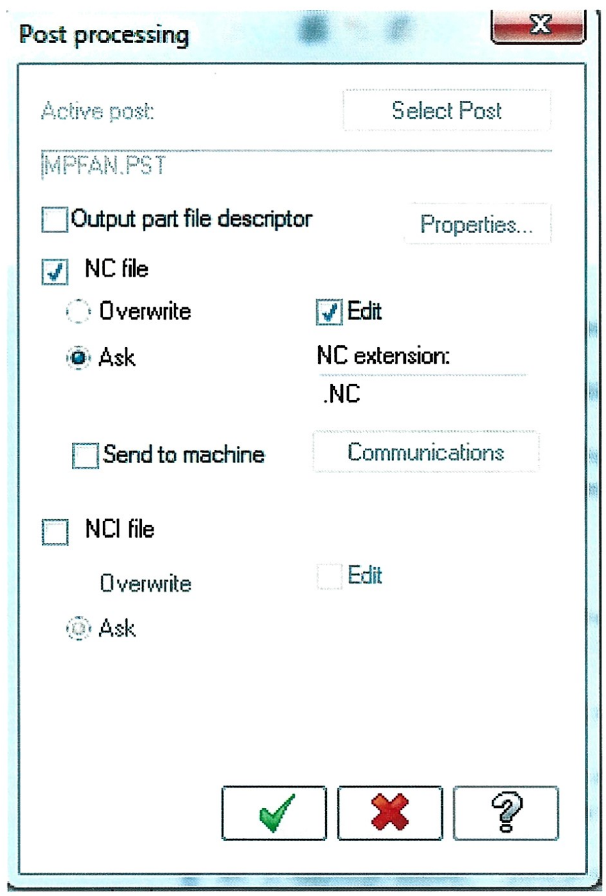
Post processing refers to the process by which the toolpaths in the Mastercam part files are converted to a format that can be understood by the machine tool's control such as G-codes. Generally, every machine controller will require its own post processor, customized to produce code formatted to meet its exact requirements.

- Make sure that all operations are selected, otherwise, Select all operations.



- Select the Post selected operations button from Toolpath Manager.

- In the Post processing window, make all the necessary changes as shown. Select the OK button to continue.



- Save the NC file to you own personal folder.
- Go to file and click save

This completes the tutorial for the turning operation