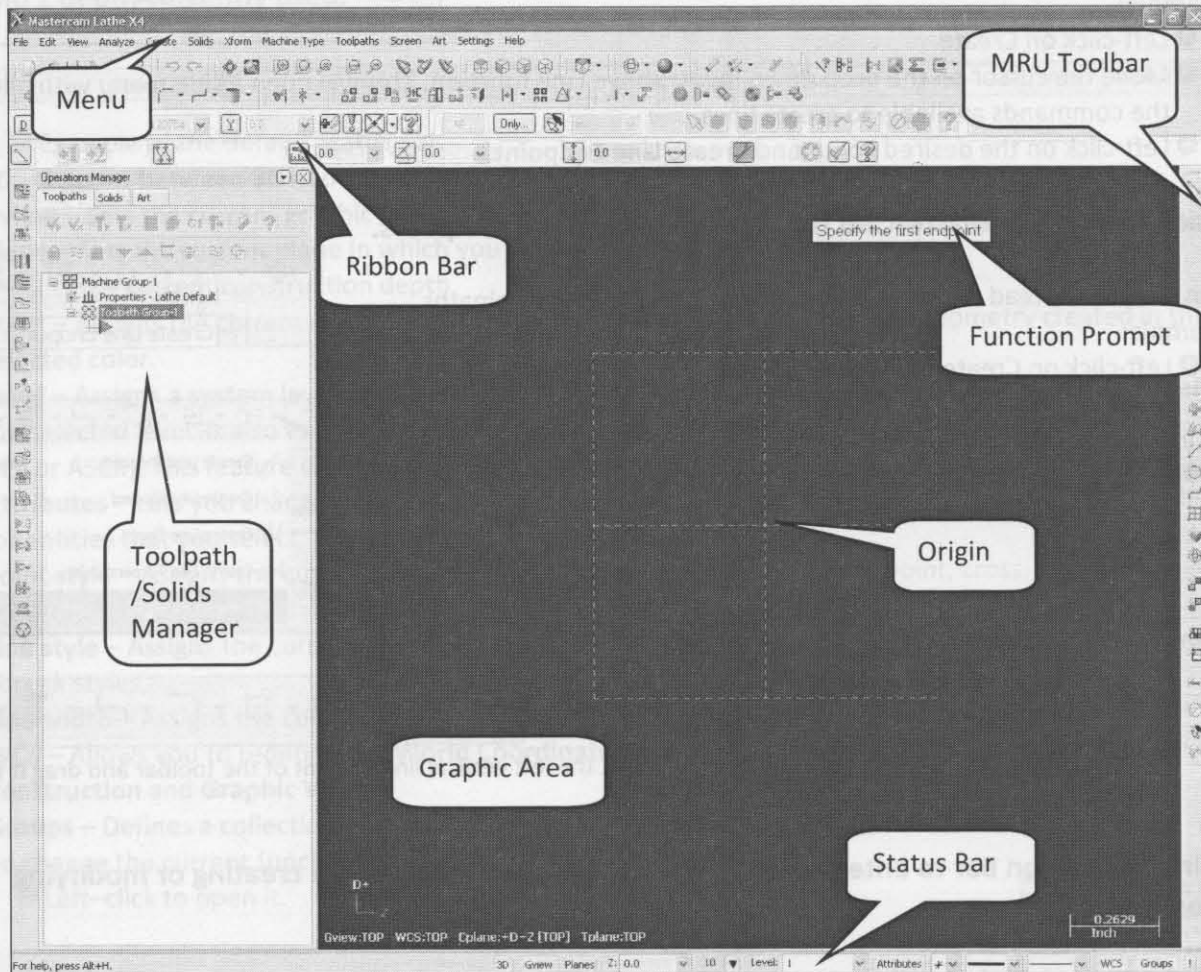


TUTORIAL SERIES FOR

Mastercam X⁴

GETTING STARTED

GUI-GRAPHIC USER INTERFACE



Menu – Allows you to select all the functions in Mastercam to create geometry and toolpaths.

Toolbars – Can be used instead of the menu to create geometry and toolpaths

Ribbon Bar – Allows you to enter the values and settings that define the entity that you are currently creating or modifying.

Function prompt – Prompts the user for info.

Status Bar – Allows you to set the attributes (color, level, style and width) and the View/Plane and Z depth currently used.

Toolpaths/Solids/Art Manager – Lists the history of the toolpath operations, solids and art.

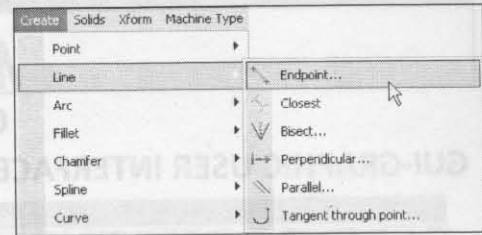
Origin – Geometry origin from which the system measures the point coordinates in X, Y and Z-axis in the current plane.

Graphic area – Workspace area in Mastercam where the geometry displays.

MRU Toolbar – List of the most recently used functions.

NAVIGATE THROUGH MASTERCAM

Using Menu functions in Mastercam to create geometry and toolpaths.



Example:

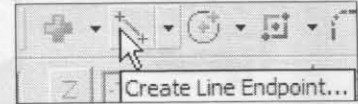
- Left-click on Create
- Move the cursor on the drop down menu above Line function. This will open a side menu with all the commands available to create lines.
- Left-click on the desired command **Create Line Endpoint**.

Using Toolbars in Mastercam to create geometry and toolpaths.

Can be used instead of the Menu to create geometry and toolpaths.

Example:

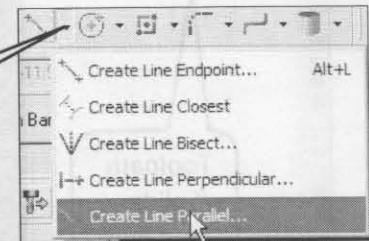
- Left-click on Create line endpoint icon.



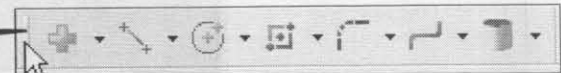
To list all Line commands;

- Select the drop-down arrow to the right of the icon.

Select the Drop down Arrow



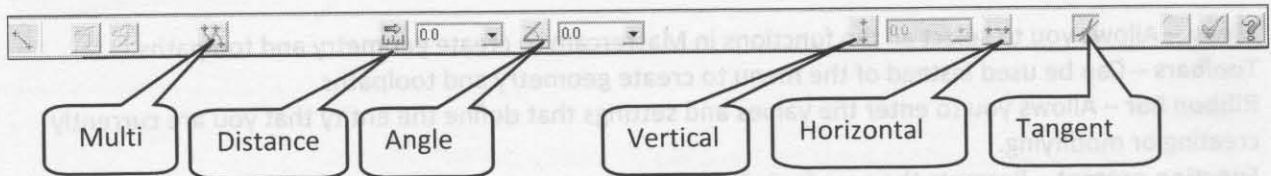
Select this line



- To reorganize the toolbars, if desired, select the left vertical line in front of the toolbar and drag it to the desired location.

Using the Ribbon Bar to enter the value and define the settings while creating or modifying geometry.

- Example of the Ribbon bar used in Create line endpoint command.



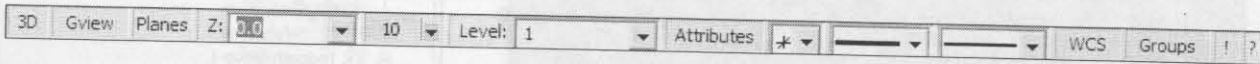
- To continue creating geometry using the same command select the **Apply** button from the ribbon bar. To exit the current command select the **OK** button or you can start a new command.
- To undo the last command use the **Undo** button. . The undo button can be used as many times as needed. Mastercam has a **Redo** button available too.

Function prompt – Prompts the user for info.

Specify the first endpoint

Example of the prompt used in **Create Line Endpoint** command.

Using the Status Bar to set the attributes (color, level, style and width) and the View/Plane and Z depth currently used.



Example of the default Status Bar.

3D – Toggles between 3D and 2D settings.

Gview – Sets the current graphic view to be able to view the geometry from several different angles.

Plane – Sets the current plane in which you will create and manipulate geometry.

Z – Sets the current construction depth.

Color – Assigns the current color. This instructs the system to display any new geometry created in the selected color.

Level – Assigns a system level to created/converted geometry. Level stores any new geometry created in the selected level. It also applies the chosen level to a converted file that does not support levels (i.e., NFL or ASCII). This feature also controls what levels are visible.

Attributes – Lets you change one or more attributes (color, level, line style, line width, and point style) for entities that you select, without affecting system default settings.

Point style – Assigns the current point style. You can choose between star, point, cross, X, circle or square styles.

Line style – Assigns the current line style. You can choose between solid, hidden, center, phantom and Zbreak styles.

Line width – Assigns the current line width. You can choose between 5 different widths.

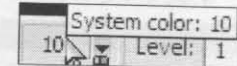
WCS – Allows you to redefine the **World Coordinate System**, and groups the functionality of **Tool**,

Construction and **Graphic Views**.

Groups – Defines a collection of entities or operations that can be manipulated as a single entity.

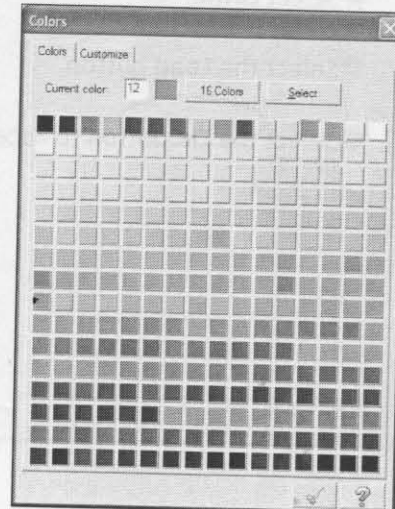
To change the current function;

Left-click to open it.



Example of the changing the current Color.

- Left-click on color
- Select the desired color from the dialog box.
- Select the **OK** button to exit.



Using the Toolpaths Manager

- The Toolpaths Manager lists all operations in the current job.
- Use this dialog box to sort, edit, regenerate, verify and post any operations.

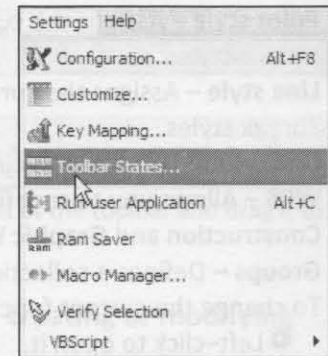
For more information on the Toolpaths Manager, please check the General Notes.



- Toolpaths/Solids manager can be hidden to gain more space in the graphic area for design. Press **Alt + O** to remove it.

Setting the Toolbar States

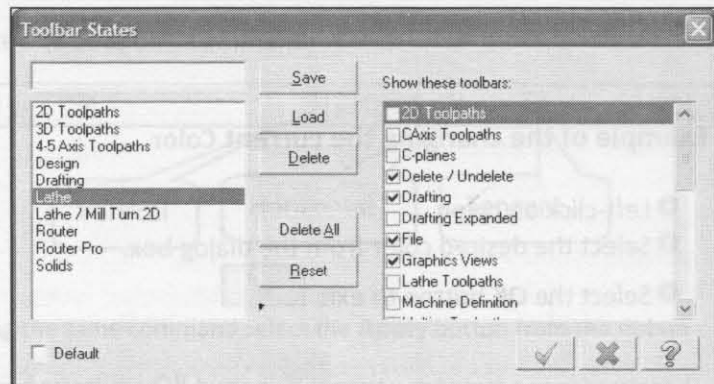
- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part.



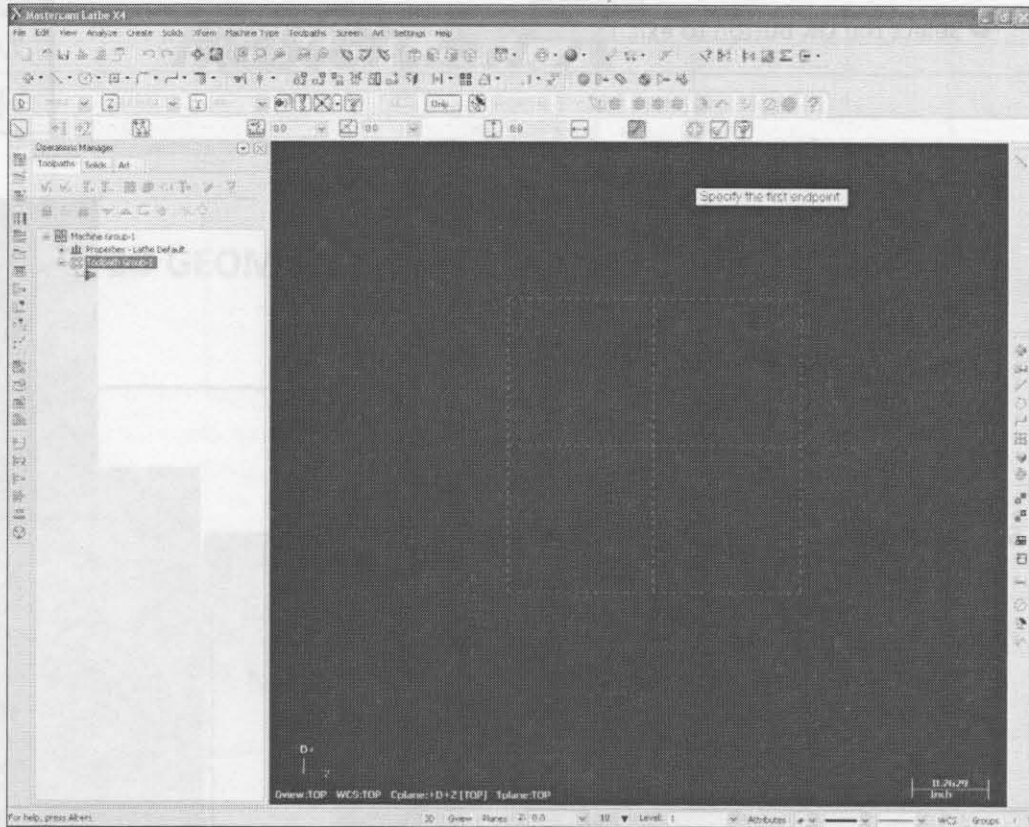
Settings

• **Toolbar States**

- Select **Lathe**.
- Select the **Load** button.
- Select the **OK** button to accept it.



Mastercam new toolbars are display as shown in the following screenshot.



Toolpaths/Solids manager to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to remove it.

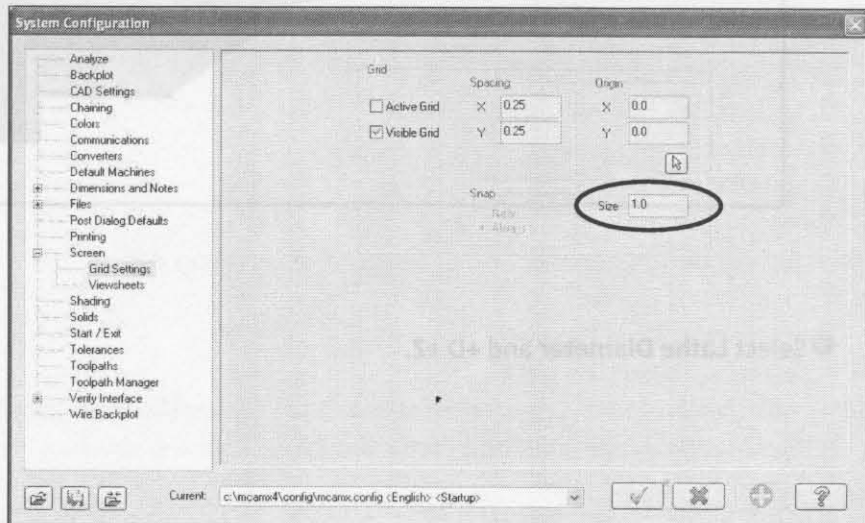
Setting the Grid

Before starting the geometry enable the **Grid**. It will show you at each moment where is the part origin.


Settings

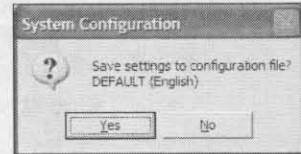
Configuration.

- Select **Screen** from the configuration **Topics**.
- Select the + beside **Screen**

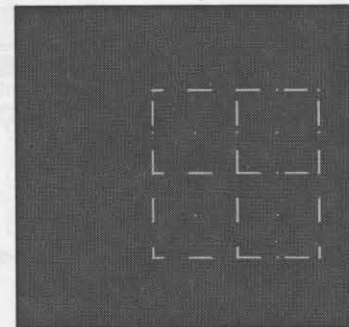


Enable **Visible Grid** and change the **Size** to **1**.

Select the **OK** button to exit. 

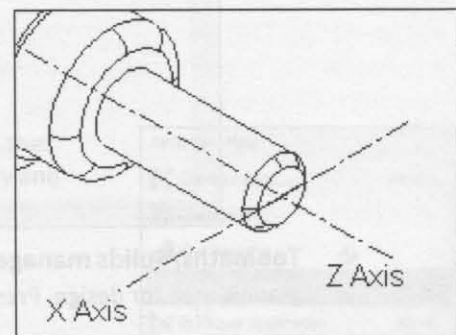


Select **Yes** button to save the setting.

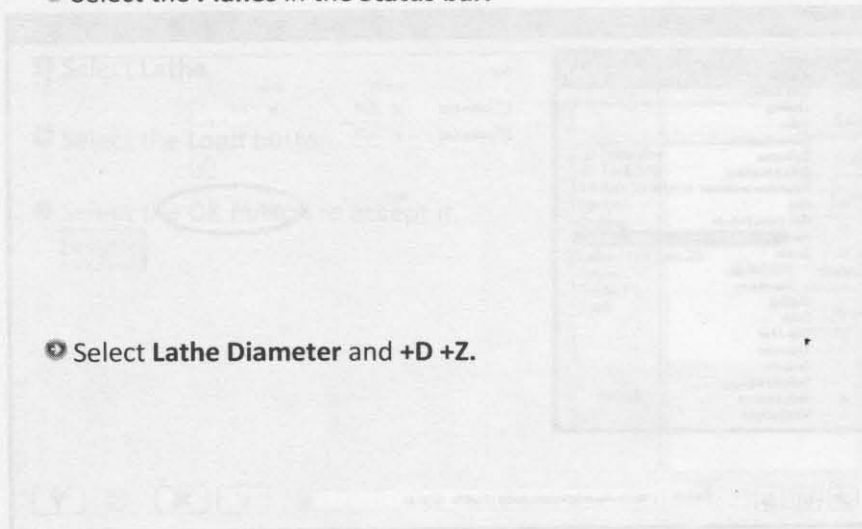


Setting the Construction and Tool Plane to DZ

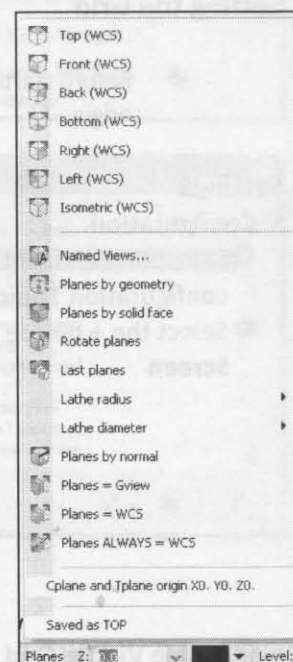
- ◆ The **Planes** settings **+D+Z** apply to construction methods for geometry requiring two axes of motion, the Z axis and a diameter value (the D value represents the diameter of X). Set the Cplane to +D+Z and Mastercam interprets X axis values as diameter values (as opposed to radius values).
- ◆ The absolute zero or origin of the part is normally at the front face and centreline of the part. The centerline forms the Z axis as shown in the following example.



Select the **Planes** in the **Status bar**.



Select **Lathe Diameter** and **+D+Z**.

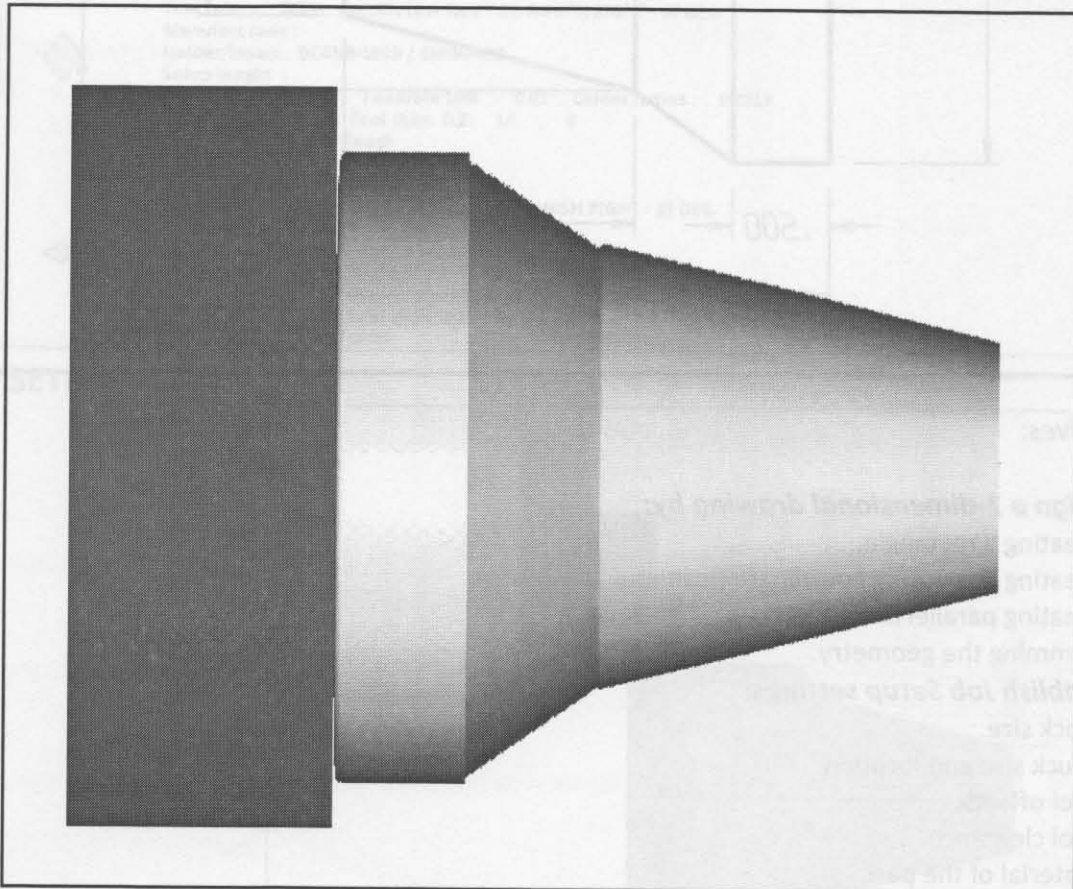


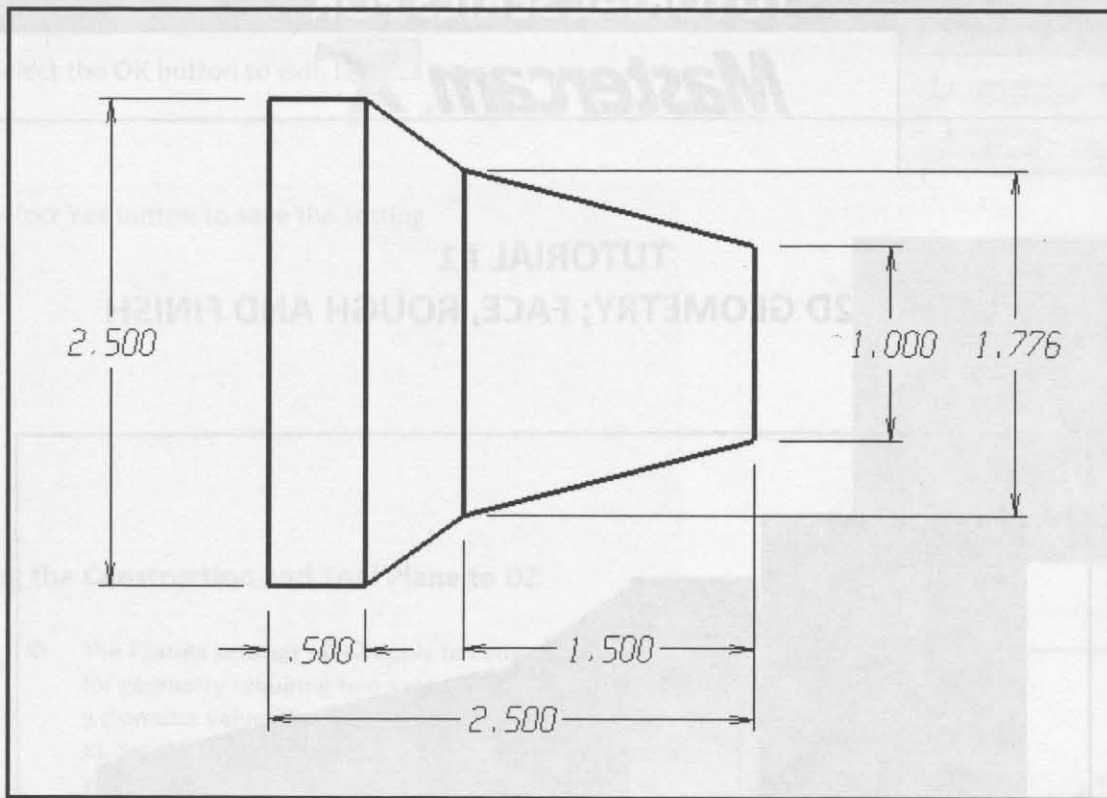
TUTORIAL SERIES FOR

Mastercam X⁴

TUTORIAL #1

LATHE - 2D GEOMETRY; FACE, ROUGH AND FINISH





Objectives:

To design a 2-dimensional drawing by:

- ☉ Creating a rectangle.
- ☉ Creating lines using coordinate positioning.
- ☉ Creating parallel lines.
- ☉ Trimming the geometry.

To establish Job Setup settings:

- ☉ Stock size.
- ☉ Chuck size and location.
- ☉ Tool offsets.
- ☉ Tool clearance.
- ☉ Material of the part.
- ☉ Feed calculation.

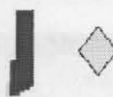


To create a 2-dimensional Lathe toolpath consisting of:

- ☉ Face cutting the part.
- ☉ Rough machining the part.
- ☉ Finish machining the part.

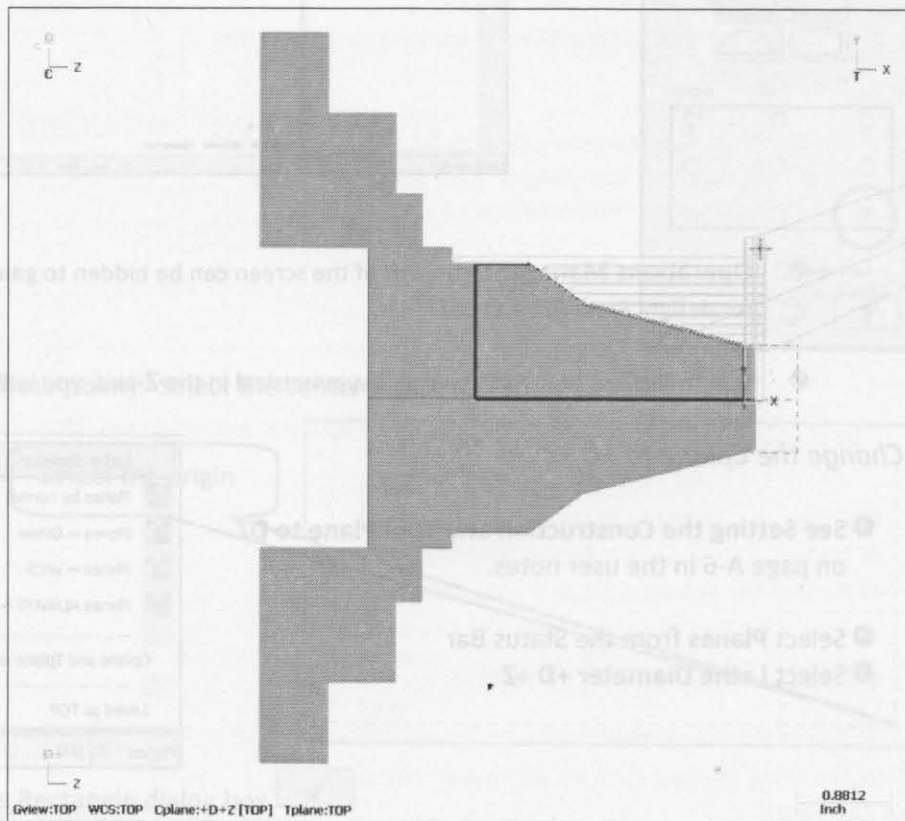
To check the toolpath using Mastercam's Verify verification module by:

- ☉ Running the Verify function to machine the part on the screen.
- ☉ Generate the NC- code:
- ☉ Running the post processor.

SETUP SHEET:

Operation List of TUTORIAL 1.MCX	
Proj./Part No.: Drawing No.: Prog. No.: 0	Date : 03/03/09 Customer : 1 Programmer : Tutorial 1
	Tool type : T0101: specific tool type - OD ROUGH RIGHT - 80 DEG. Manufact code : Holder/Insert : DCGNR-164D / CNMG-432 Setup length : Spindle RPM : 200 Feedrate UPR : 0.01 Corner radius : 0.0313 Length offset : 1 Tool chan. D,Z: 14 , 8 Comment : Lathe Face
	Tool type : T0101: specific tool type - OD ROUGH RIGHT - 80 DEG. Manufact code : Holder/Insert : DCGNR-164D / CNMG-432 Setup length : Spindle RPM : 200 Feedrate UPR : 0.01 Corner radius : 0.0313 Length offset : 1 Tool chan. D,Z: 14 , 8 Comment : Lathe Rough
	Tool type : T0202: specific tool type - OD FINISH RIGHT - 35 DEG. Manufact code : Holder/Insert : MVJNR-164D / VNMG-431 Setup length : Spindle RPM : 200 Feedrate UPR : 0.01 Corner radius : 0.0156 Length offset : 2 Tool chan. D,Z: 14 , 8 Comment : Lathe Finish

PART SETUP:

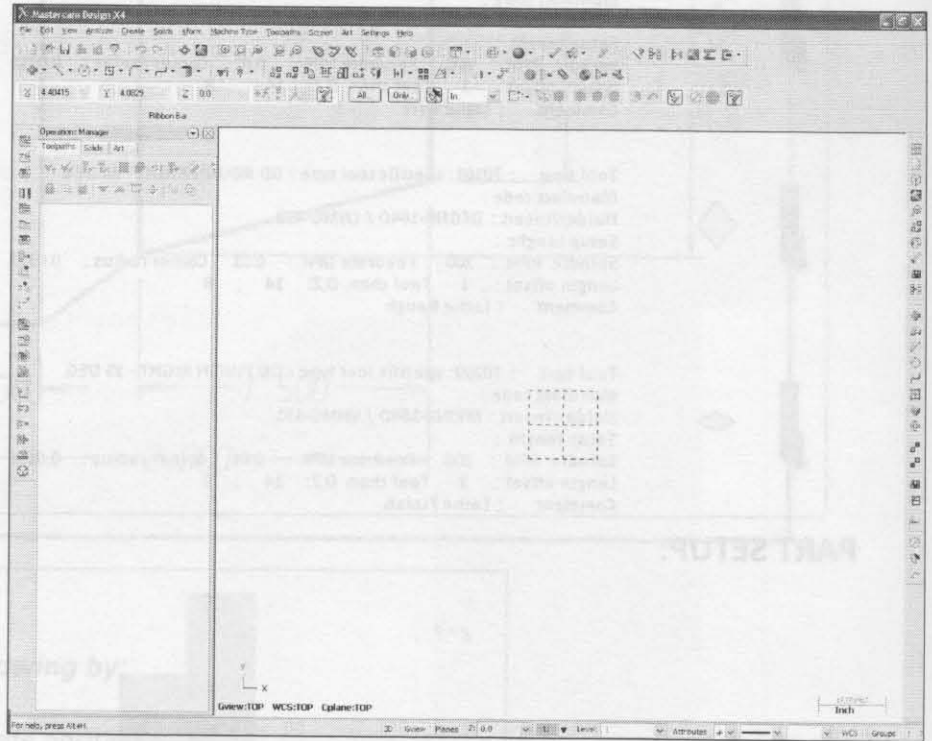


GEOMETRY CREATION

Setting the toolbar states

Before starting the geometry creation we should customize the toolbars to create the geometry and machine a 2D part. See **Setting the Toolbar States** on page A-4 in the **User Notes**.

Make sure that the **Grid** is enabled as it will show you where the part origin is. See **Setting the Grid** on page A-5



➤ **Operations Manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to hide it.

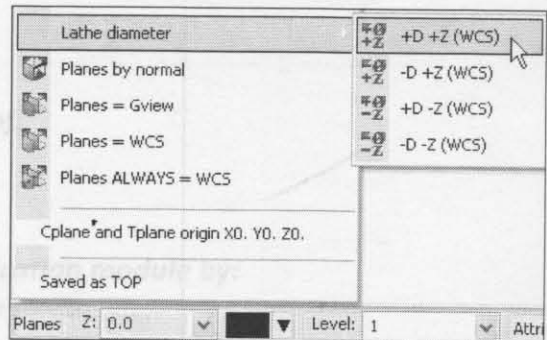
➤ Due to the fact that this drawing is symmetrical in the **Z**-axis, you will only draw 1/2 of the total part.

Change the Cplane to +D +Z

➤ See **Setting the Construction and Tool Plane to DZ** on page A-6 in the user notes.

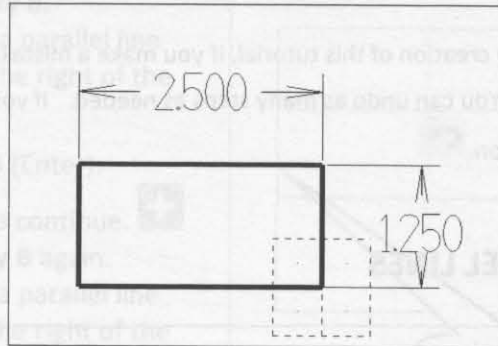
➤ Select **Planes** from the Status Bar

➤ Select **Lathe Diameter +D +Z**



STEP 1: CREATE ½ THE OUTSIDE BOUNDARY

Step Preview:

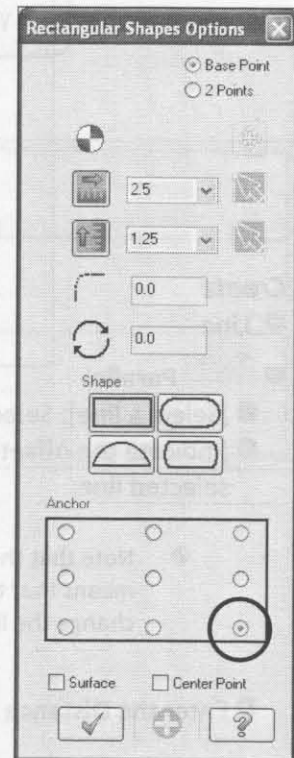


Create

- 1. **Rectangular Shapes**
 - Enter the **Width** and the **Height** as shown below.

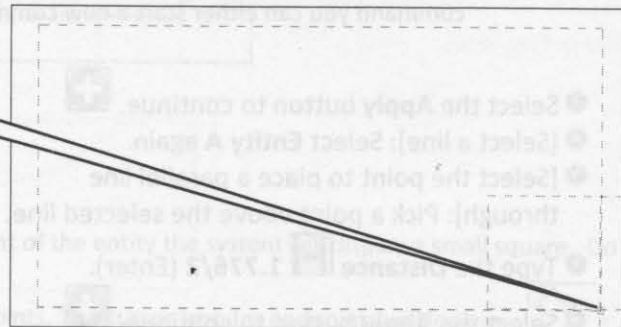
The Drawing should look as shown in the picture to the right.

- 2. Select the lower right corner radio button as the anchor



- 3. [Select position for the base point]: Select the center location of the grid (the origin).

Select the origin

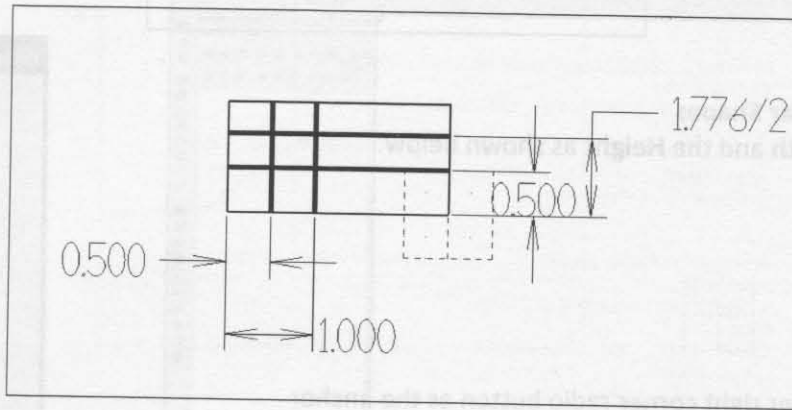


- 4. Select the **OK** to exit the Rectangle dialog box

- Use the **Fit** icon to fit the drawing to the screen.
- During the geometry creation of 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.

STEP 2: CREATE PARALLEL LINES

Step Preview:



Create

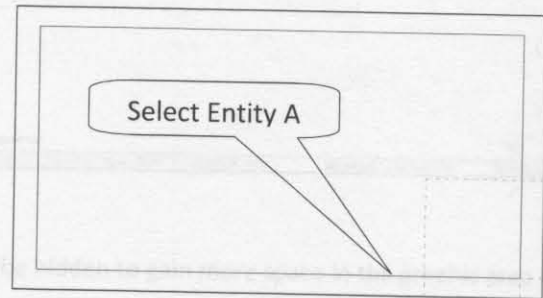
Line

Parallel

- [Select a line]: Select **Entity A**
- [Indicate the offset direction]: 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 the line parameters if needed.

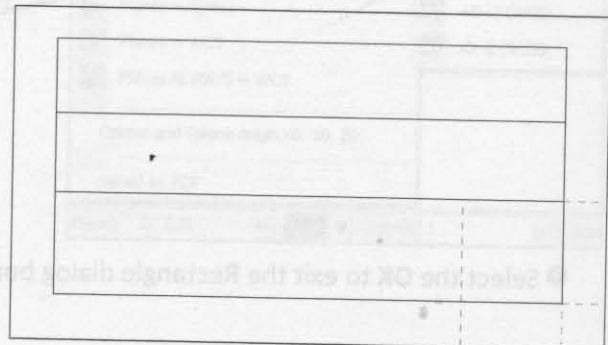
Enter the **Distance** 0.5 (Press Enter).




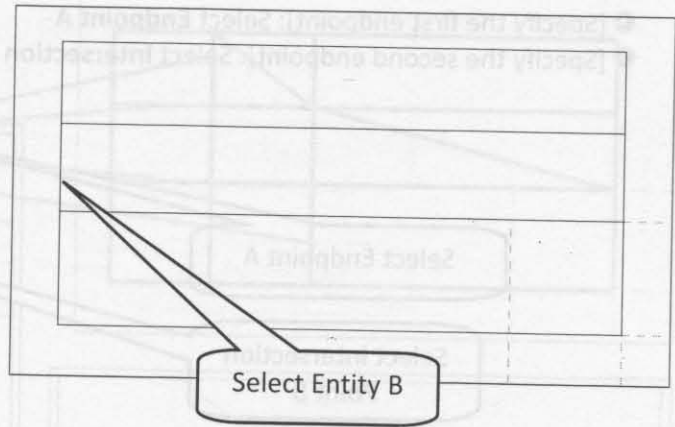
- Note that to continue using the same command you have to select the **Apply** button. To exit the command you can either start a new command or select the **OK** button.

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

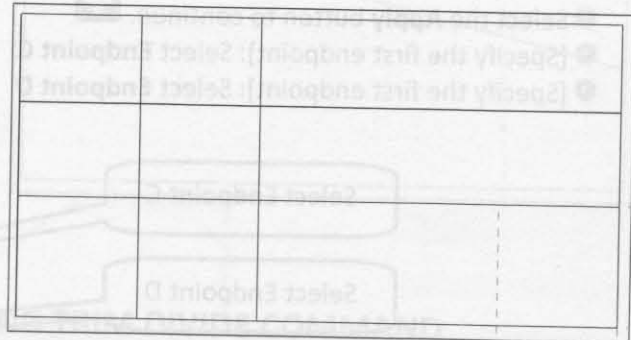
The drawing should look as shown in the picture to the right.



- [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 **Distance** **0.5** (Enter).
- Select the **Apply** button .
- [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 **Distance** **1.0** (Enter).
- Select the **OK** button.



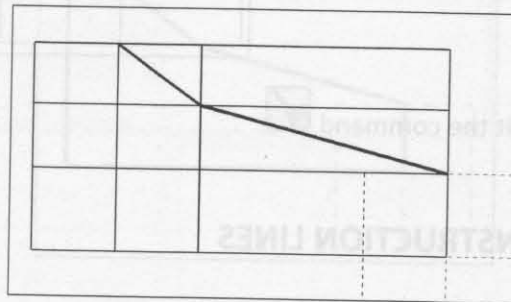
The Drawing should look as shown in the picture to the right.



STEP 5: TRIMMING THE GEOMETRY


STEP 3: CREATE LINES KNOWING THE ENDPOINTS

Step Preview:



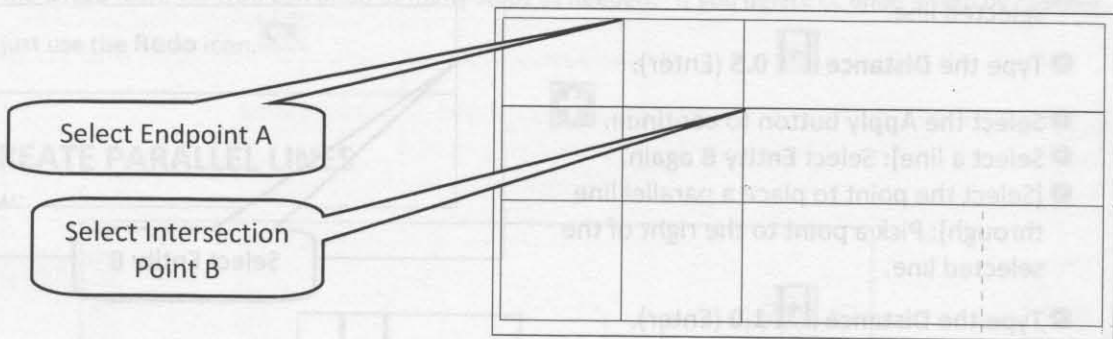
Create


- Line
-  **Endpoint**

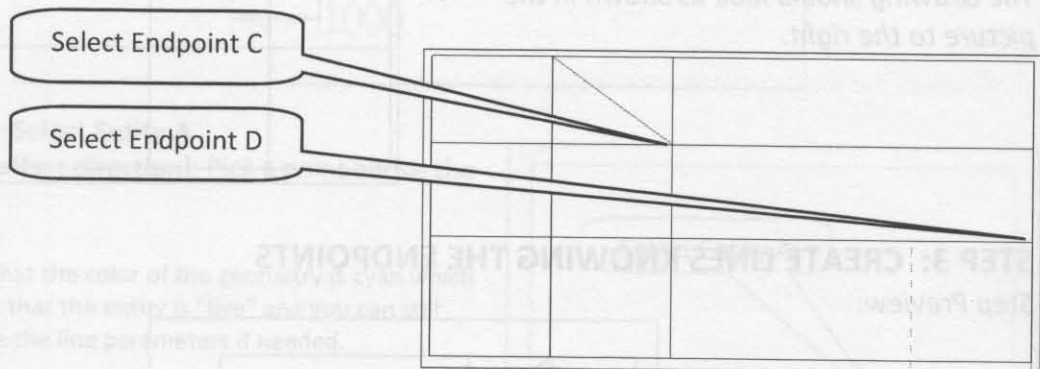
• By moving the cursor exactly on the endpoint of the entity the system will display a small square. Do the same thing to select the following endpoints. The cursor icon for endpoints will look as  ;


and for an intersection point 

- [Specify the first endpoint]: Select **Endpoint A**
- [Specify the second endpoint]: Select **Intersection point B**



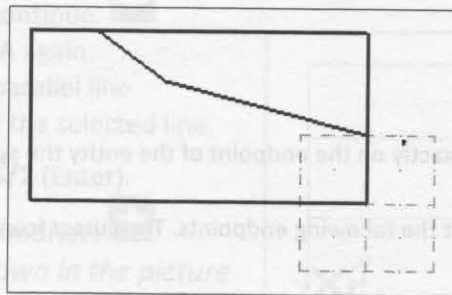
- Select the **Apply** button to continue. 
- [Specify the first endpoint]: Select **Endpoint C**
- [Specify the first endpoint]: Select **Endpoint D**




- Select the **OK** button to exit the command 

STEP 4: DELETE THE CONSTRUCTION LINES


Step Preview:

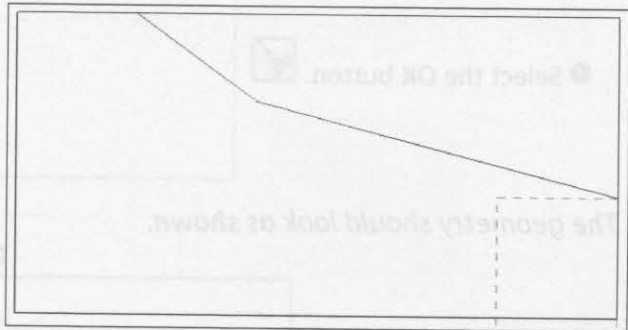
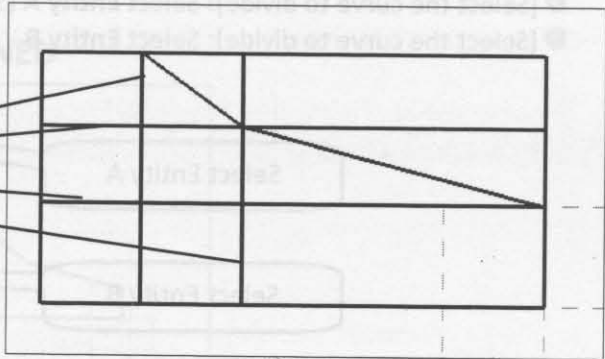


Select the lines as shown to the right.

Select the **delete** entity icon 

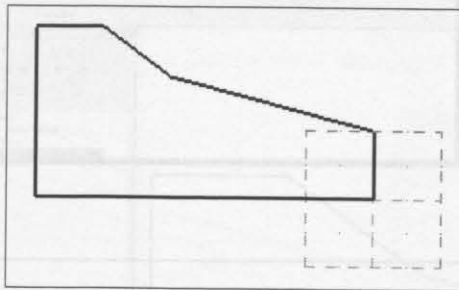
Delete these Lines

Select the **Repaint** button. 




STEP 5: TRIMMING THE GEOMETRY USING TRIM DIVIDE COMMAND

Step Preview:



Edit

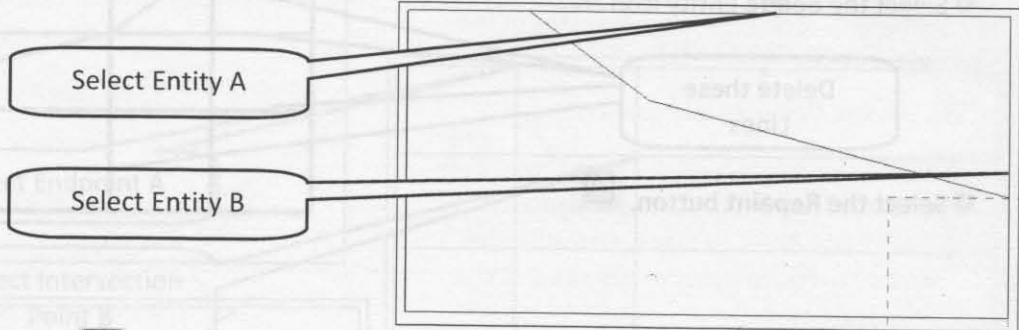
Trim/Break

Trim/Break/Extend 

Select Trim Divide

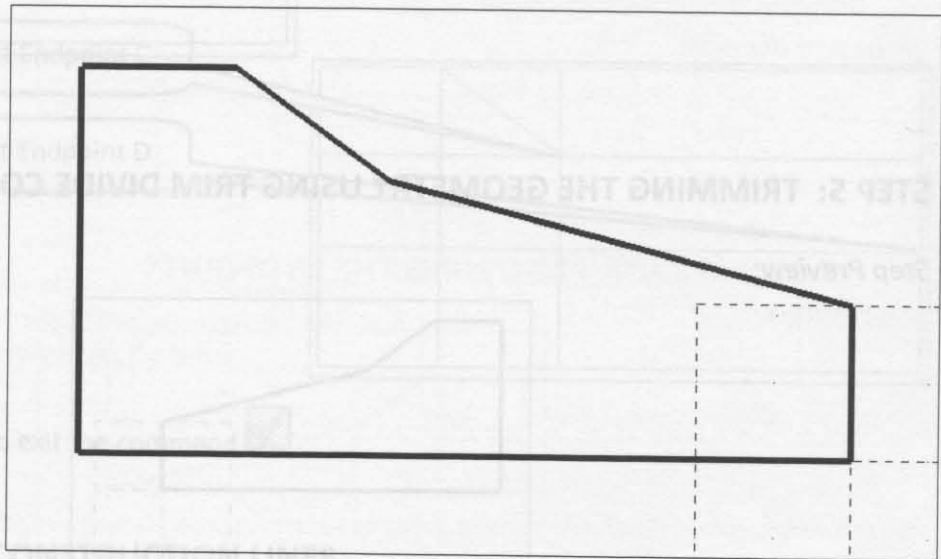


- [Select the curve to divide]: Select **Entity A** (Select the entities exactly as shown in the drawing)
- [Select the curve to divide]: Select **Entity B**



- Select the **OK** button.

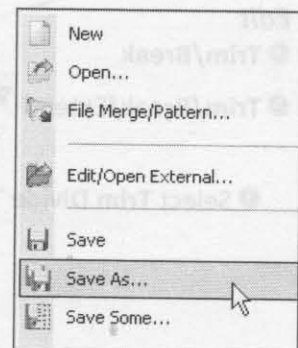
The geometry should look as shown.



STEP 6: SAVE THE FILE

File

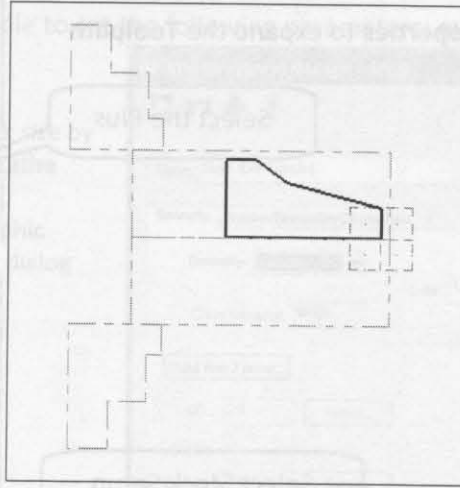
- **Save As**
 - File name: "Your Name_1"
 - Select **OK** button to accept it.



TOOLPATH CREATION

STEP 7: SET UP THE STOCK TO BE MACHINED

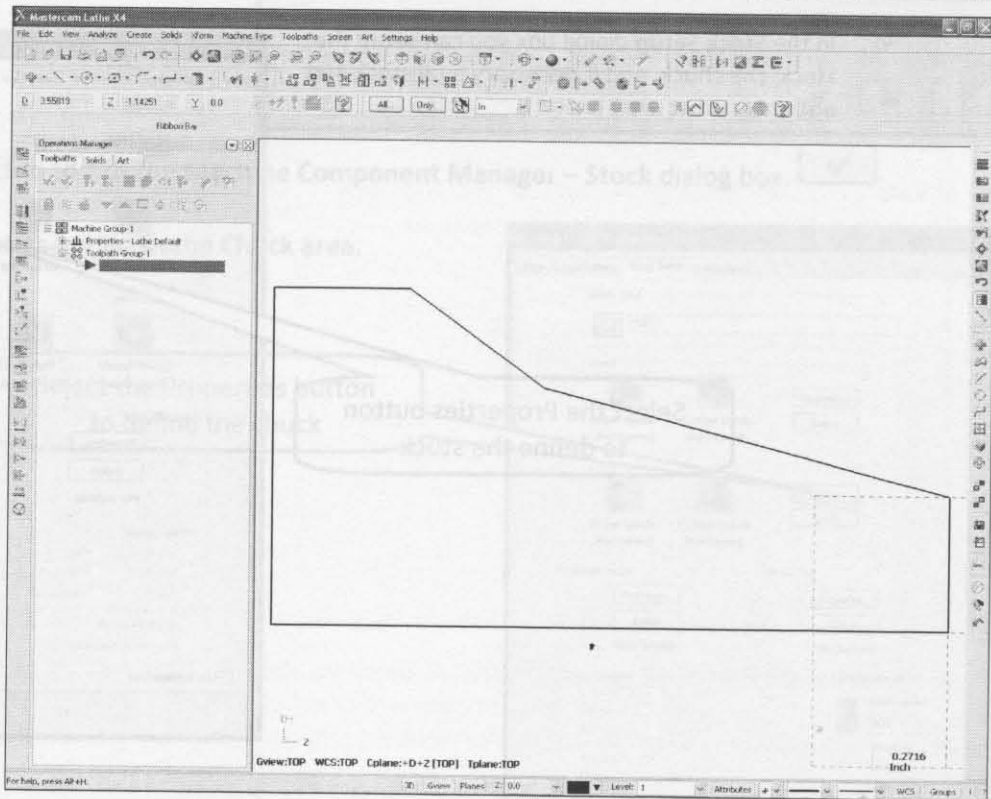
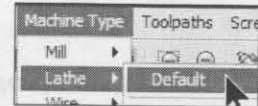
Step Preview:



- To display the **Toolpaths Manager** press **Alt + O**

Machine Type

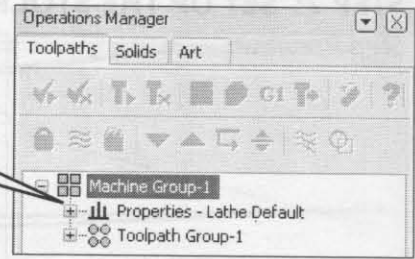
- Select **Lathe**
- Select **Default**



- Select the **Toolpaths Manager** tab to make it active

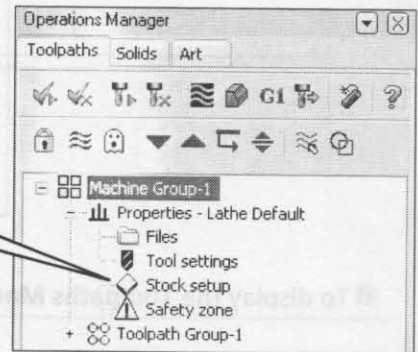
- Select the **plus** in front of **Properties** to expand the **Toolpath Group Properties**

Select the Plus



- Select **Stock Setup**.

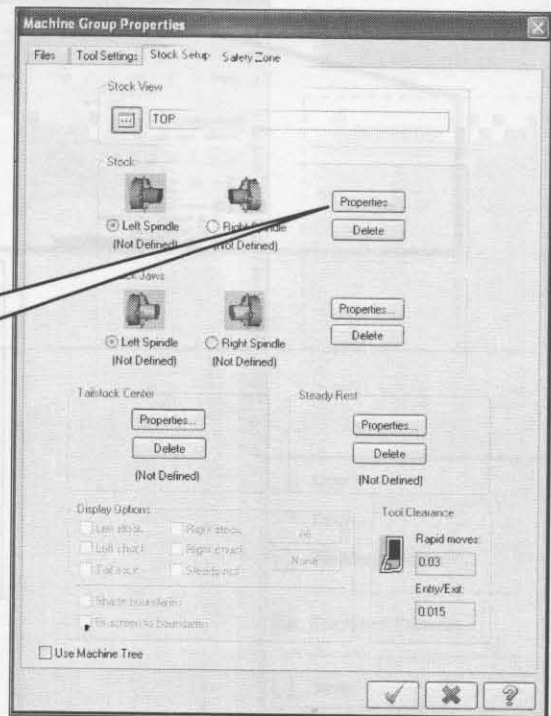
Select Stock Setup



- Change the **properties** to match the following screenshot.

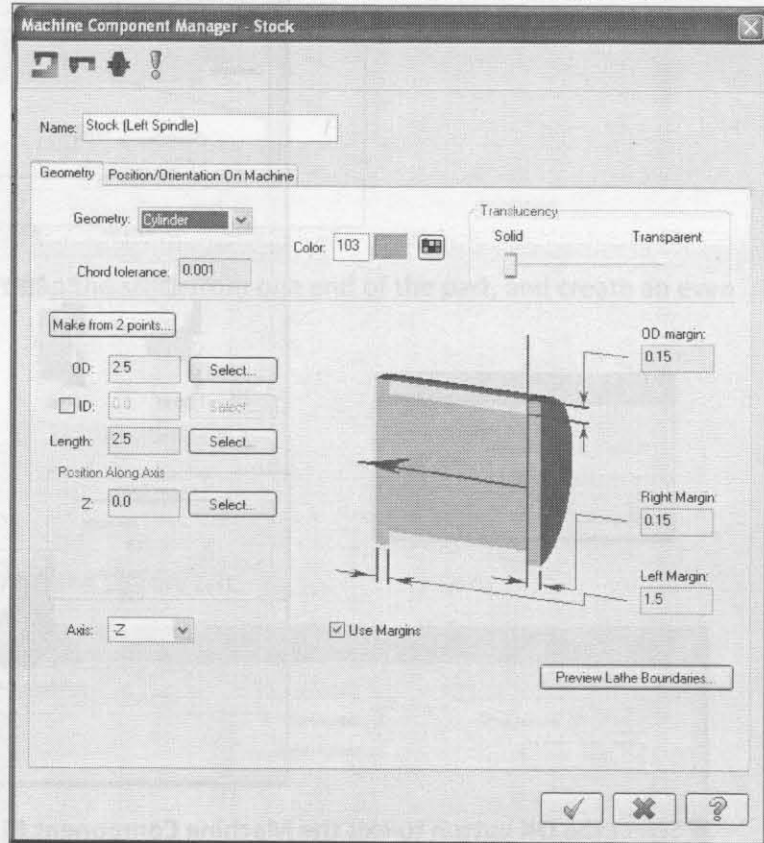
- In the **Stock Setup** dialog box you can setup the stock, the chuck, and if necessary a tailstock and a steady rest.

Select the Properties button to define the stock



- Select the **Properties** button in the **Stock** area to establish the stock size and make the necessary changes as shown below.
- Enable **Use Margins** to be able to set the following parameters; extra stock to the **OD, face, and back of stock**.

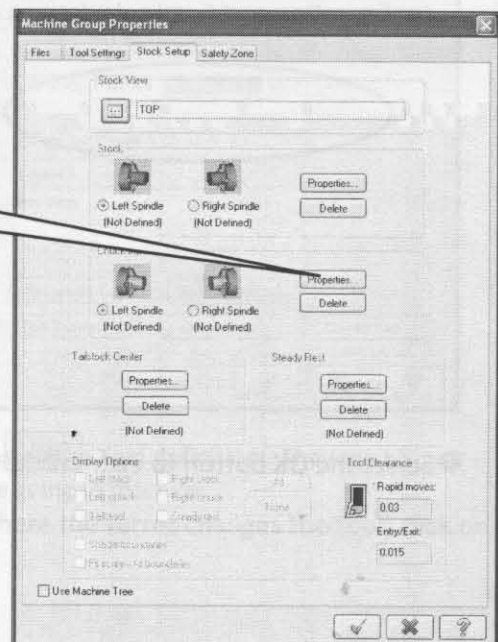
- You can check the stock size by selecting the **Preview Lathe Boundaries** button. To return from the graphic mode to the **Bar Stock** dialog box hit Enter.



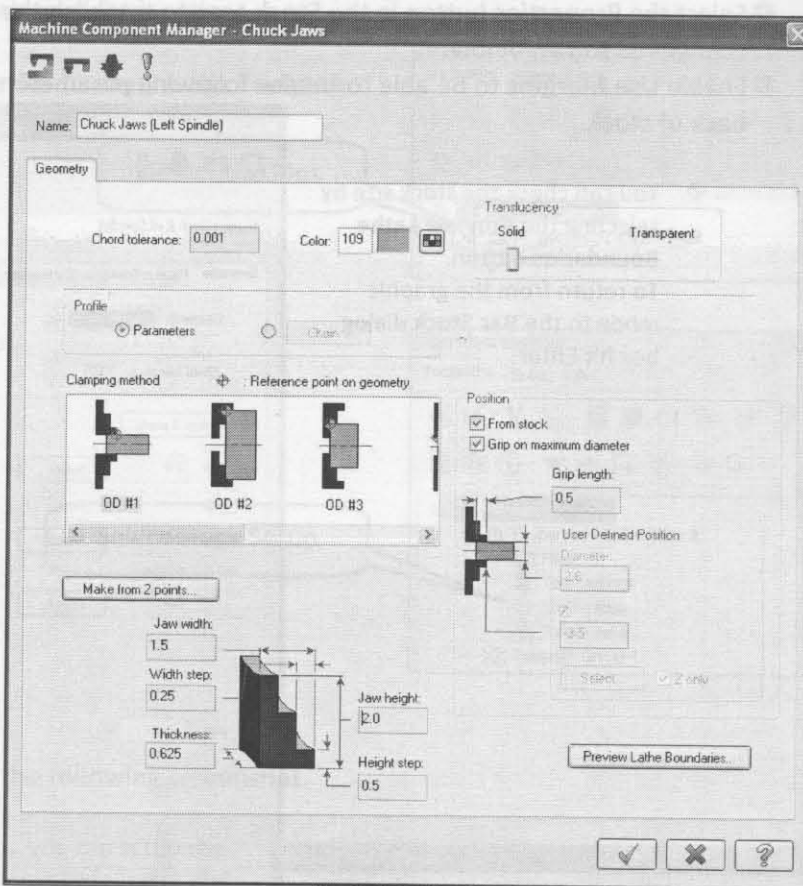
- Select the **OK** button to exit the **Machine Component Manager – Stock** dialog box.

- Select the **Properties** button in the **Chuck** area.

Select the Properties button to define the chuck

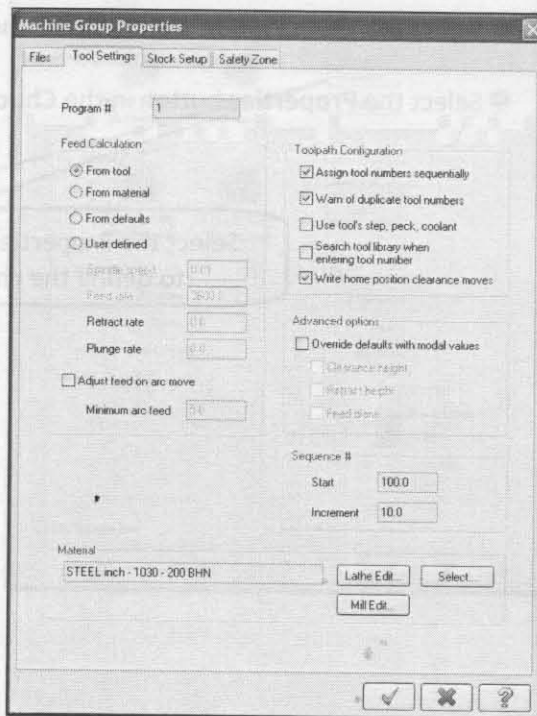


- Make the necessary changes to define the chuck size, clamping method and the stock position.



- Select the **OK** button to exit the **Machine Component Manager – Chuck Jaws** dialog box.

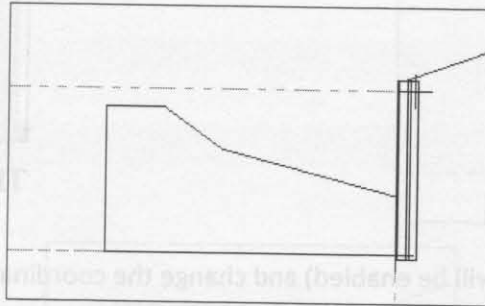
- Select the **Tool Settings** tab and set the Toolpath configuration and Feed calculations options.



- Select the **OK** button to exit.

STEP 8: FACE THE PART

Toolpath Preview:

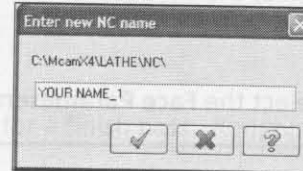


Face Toolpath: Allows the user to quickly clean the stock from one end of the part, and create an even surface for future operations.

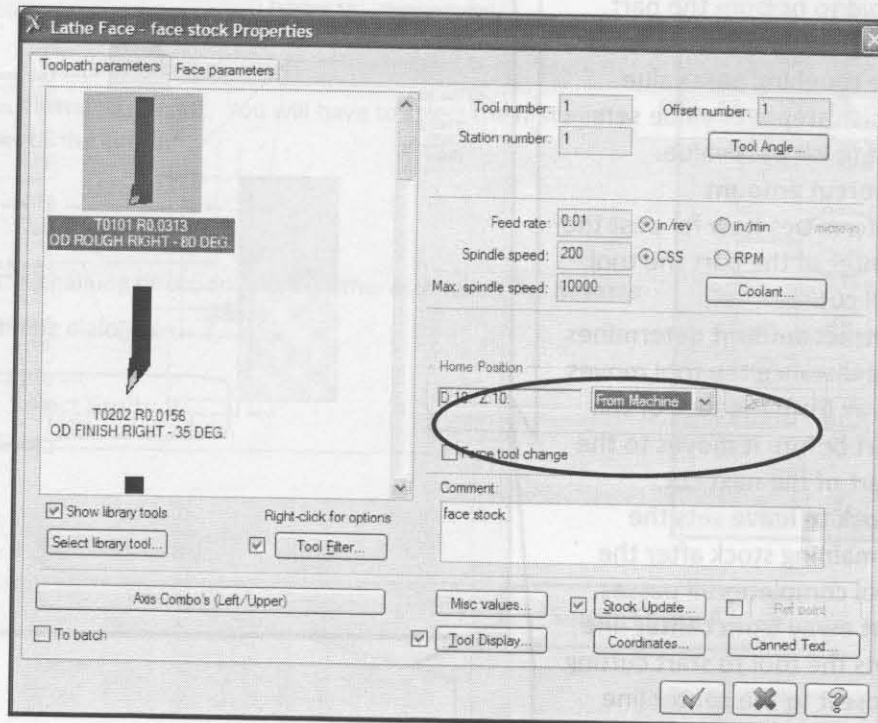
Face Toolpath Toolpaths

Face

- Select **OK** to accept NC name

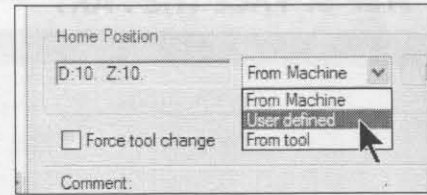


Select the **OD Rough Right -80deg** cutter from the Library List.

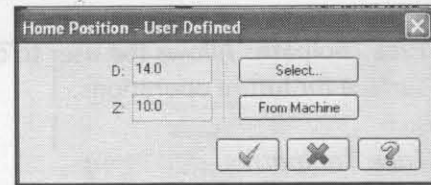


- The **Feed rate** and the **Spindle Speeds** are based on **Mastercam Tool definition**. They can be changed at any time, based on the material that you are going to machine.
- To change the **Home Position** coordinates (the position where the turret changes the tool) click on the drop down arrow next to **From Machine**.

- Select from the pull-down list select **User Defined**.



- Select the **Define** button (will be enabled) 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.

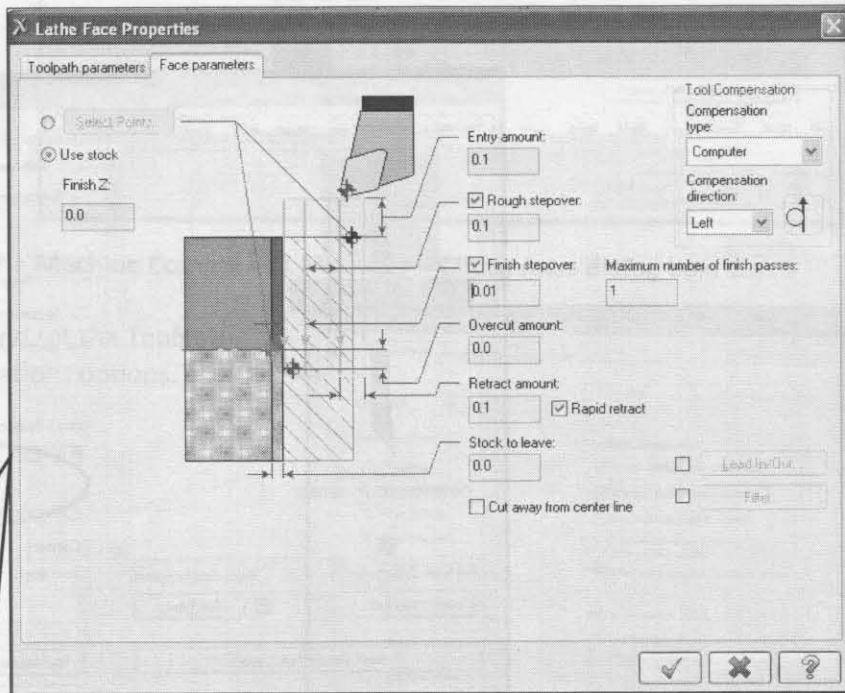
Finish stepover value sets the finish pass value.

Overcut amount determines how far past the center of the part the tool will cut.

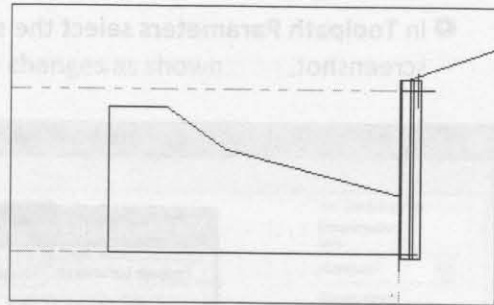
Retract amount determines the distance the tool moves away from the face of the part before it moves to the start of the next cut.

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.

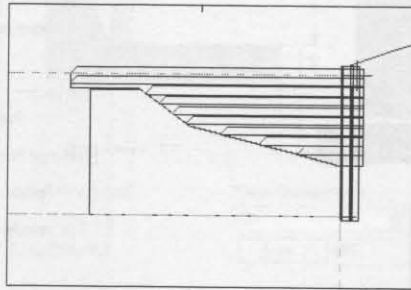


- Once complete select the **OK** button to exit **Face** properties.



STEP 9: ROUGH THE PART

Toolpath Preview:



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

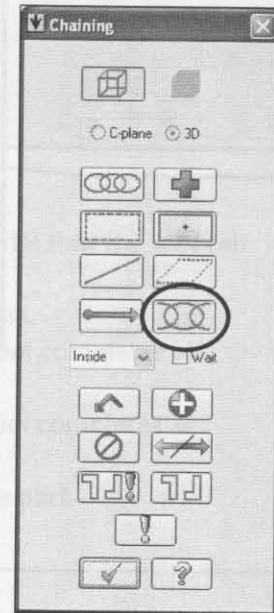
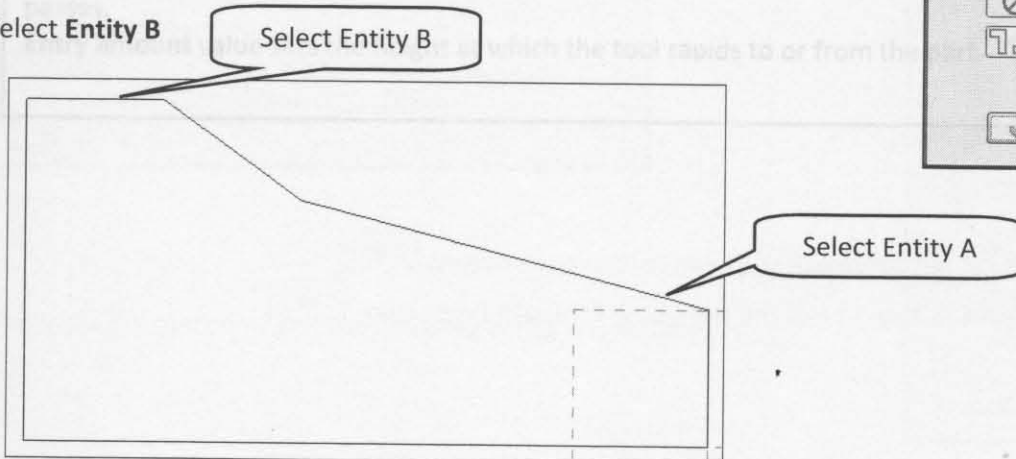
Rough

- 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.

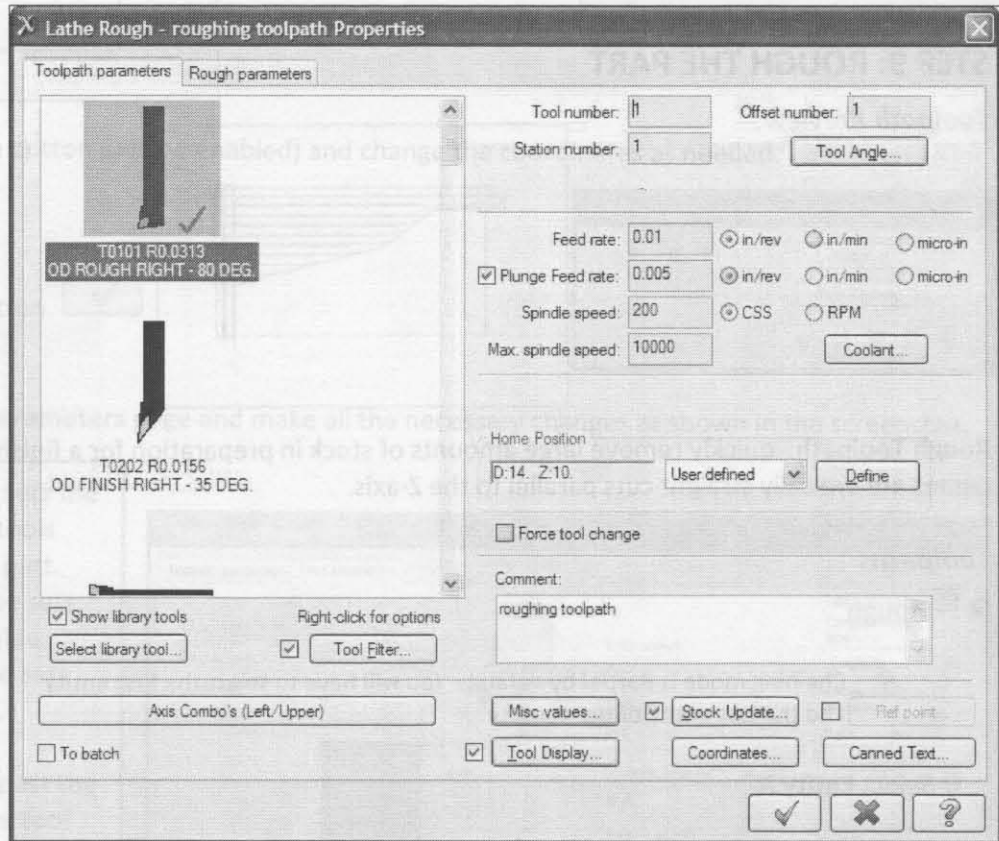
Select Entity B



- Select the **OK** button to exit **Chaining** dialog box.

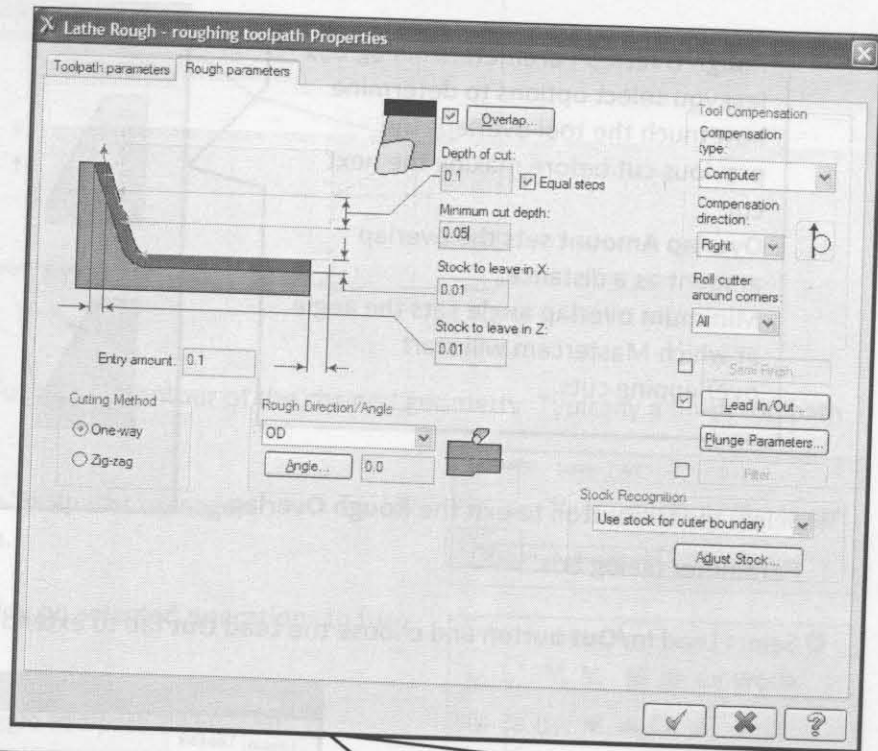
➤ In **Toolpath Parameters** select the same tool and make all the necessary changes as shown in the screenshot.

➤ To change the **Home Position** coordinates use the same steps as shown in the previous operation.



STEP 10: FINISH THE PART

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



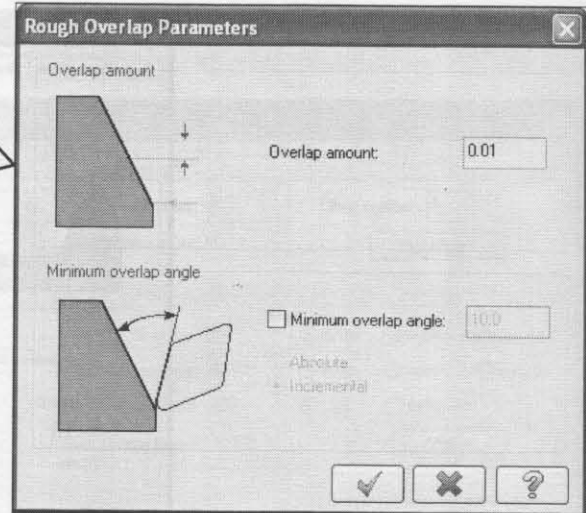
Depth of cut sets the amount of material to be removed during each pass.
Equal steps sets the **Depth of cut** value to the maximum amount of material that the tool can remove at each pass to ensure equal passes.
Minimum cut depth value sets the minimum cut that can be taken per pass.
Stock to leave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

Rough Overlap Parameters dialog box lets you select options to determine how much the tool overlaps the previous cut before making the next cut.

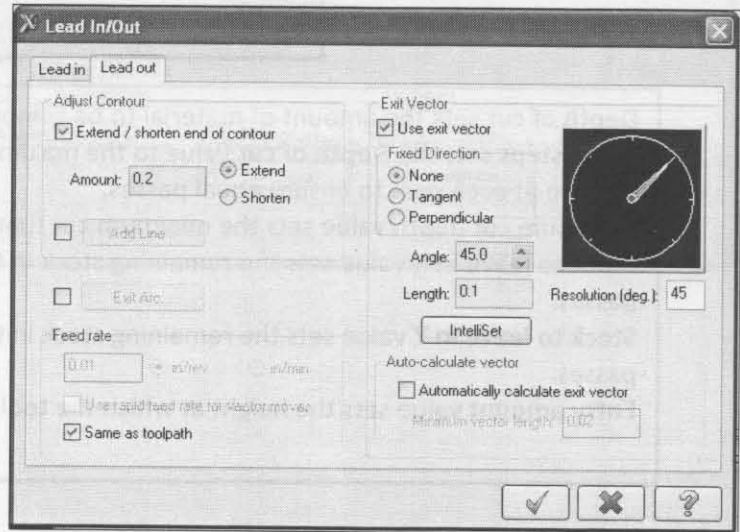
Overlap Amount sets the overlap amount as a distance.

Minimum overlap angle sets the angle at which Mastercam will start overlapping cuts.



- Select the **OK** button to exit the **Rough Overlap Parameter** dialog box.

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

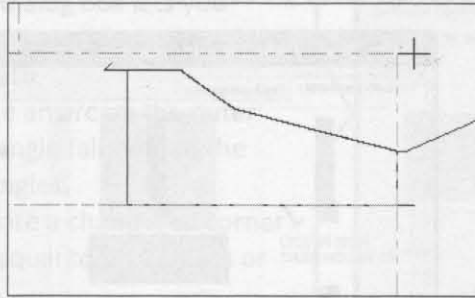


- Select the **OK** button to exit the **Lead In/Out parameters**.

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

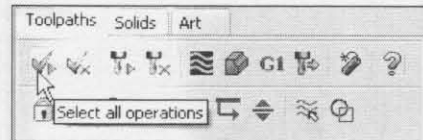
STEP 10: FINISH THE PART

Step Preview:



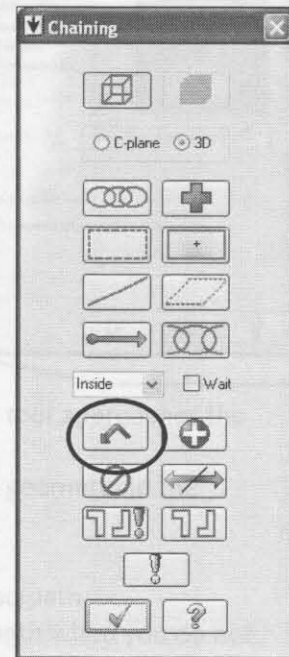
Finish Toolpath: Have the tool follow the contour of the chained geometry. Typically a finish toolpath follows a roughing toolpath.


- Select **Toolpaths** tab enable **Toolpaths manager**.
- Select **all operations** button.
- Select **Toggle toolpath display on selected operations** to turn the toolpath display off.
- Or you can use **Alt + T** to toggle toolpath display off.



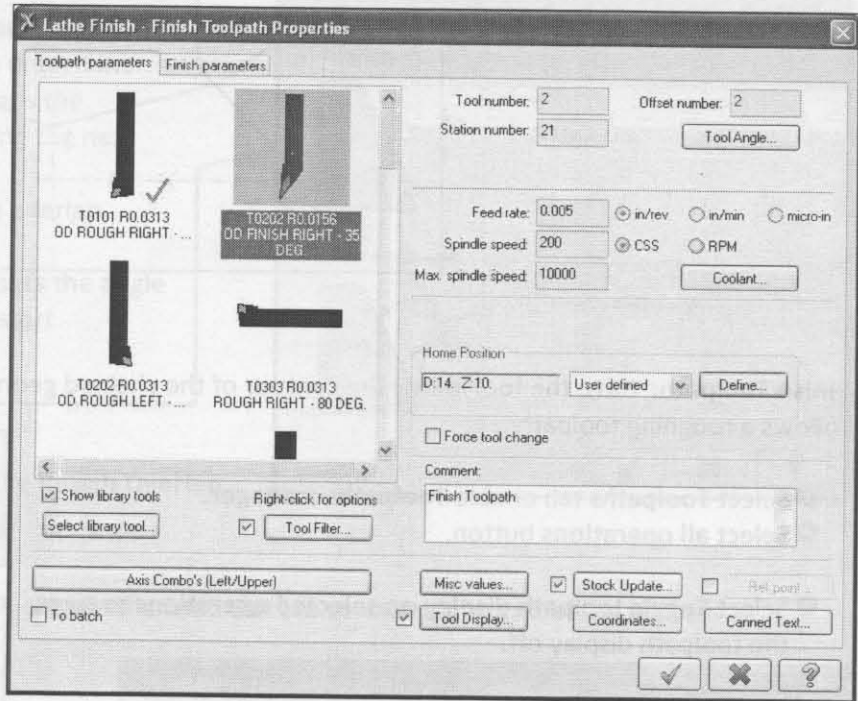
Toolpaths

- **Finish**
- Select **Last** button in the **Chaining** dialog box.

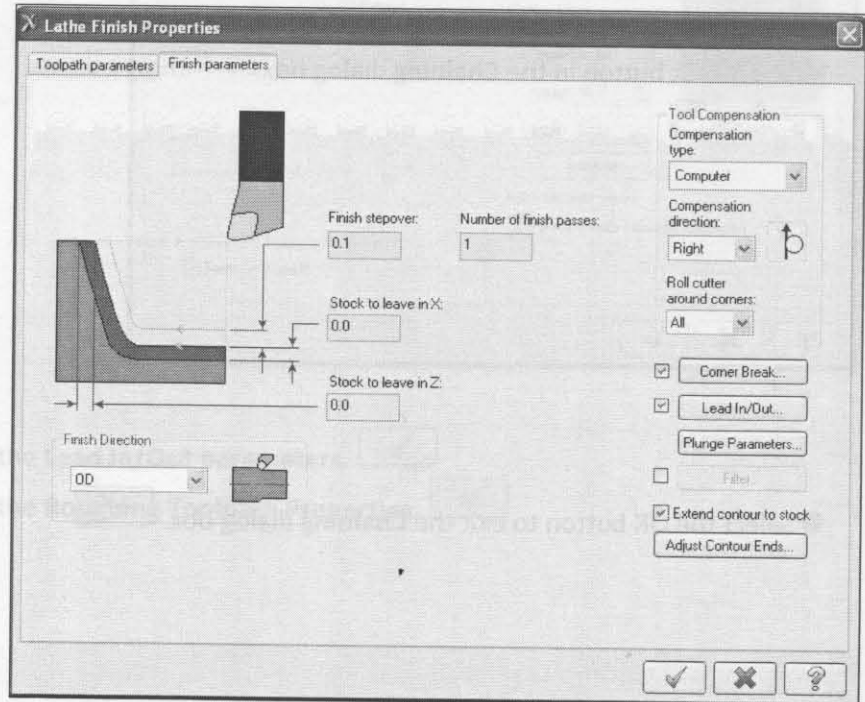


- Select the **OK** button to exit the **Chaining** dialog box. 

- Select the **OD Finish Right -35deg** cutter from the tool list.
- The Feed rate and the Spindle speed are based on Mastercam Tool definition.



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



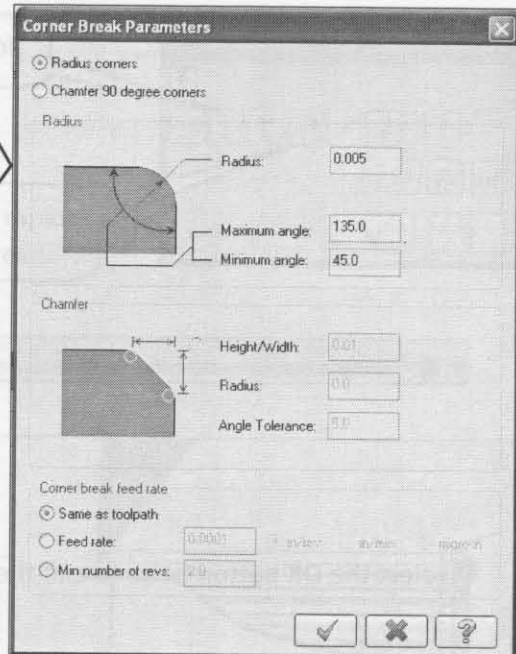
- Select the **Corner Break** button and make any necessary changes.

Corner Break Parameters dialog box lets you select options to create radii or chamfers on all outer corners of the toolpath.

Radius allows you to create an arc on the outer corners only if the corner angle falls within the maximum and minimum angles.

Chamfer allows you to create a chamfered corner only if the corner angle is equal to 90 degrees or in the **Angle tolerance**.

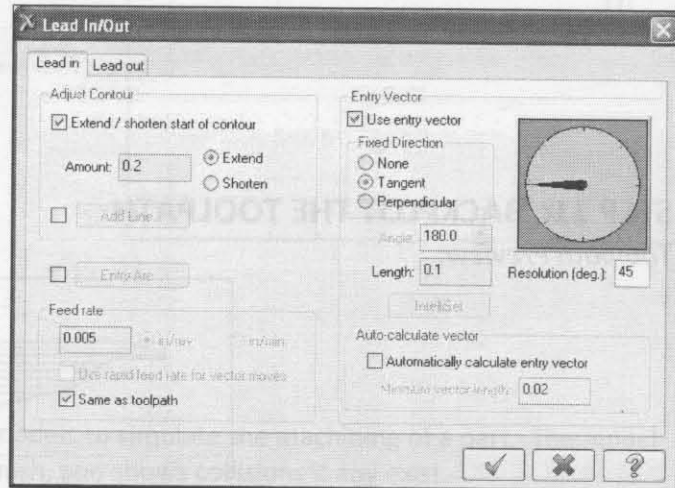
Corner break feed rate allows you to use a different feed rate while cutting the arc or the chamfer.



- Select the **OK** button to exit **Corner Break Parameters**.



- 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.

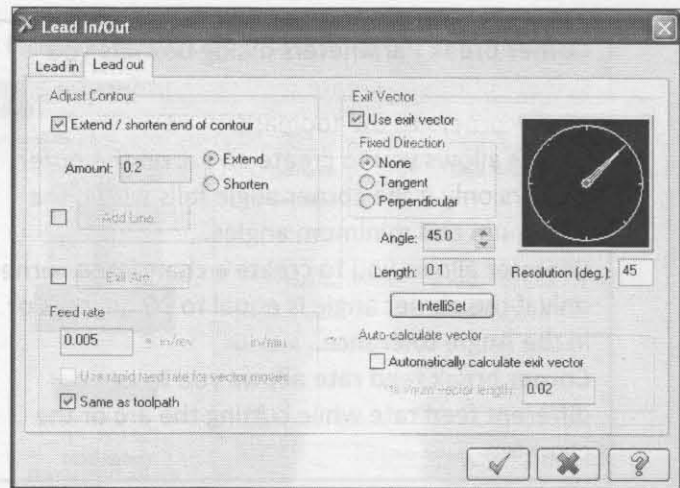
Entry Arc allows you to create a tangent arc move to the start of the toolpath.

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

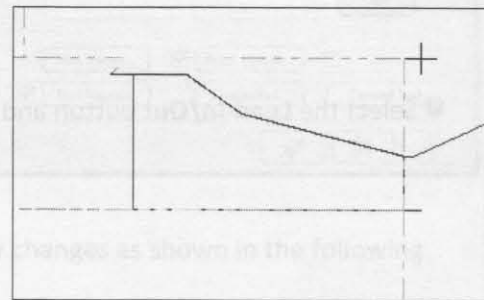
Tangent will create a line tangent to the first entity of the chained geometry when using a **Length** value.

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.

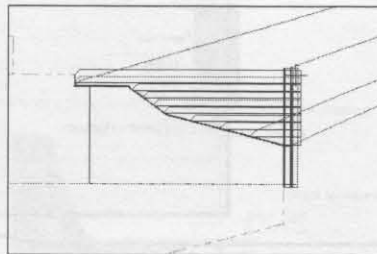


- Select the **OK** button twice to exit the **Finish Parameters**.



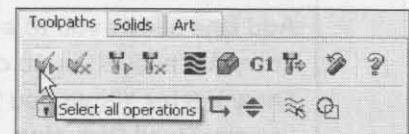
STEP 11: BACKPLOT THE TOOLPATH

Toolpath Preview:



Backplotting Toolpaths: Shows the path the tool takes to cut your part. This display helps you spot errors in your program before you machine the part.

- Select **All operations** button.

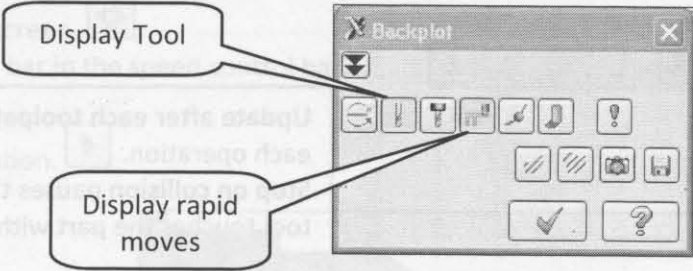


- Select **Backplot selected operations** button.

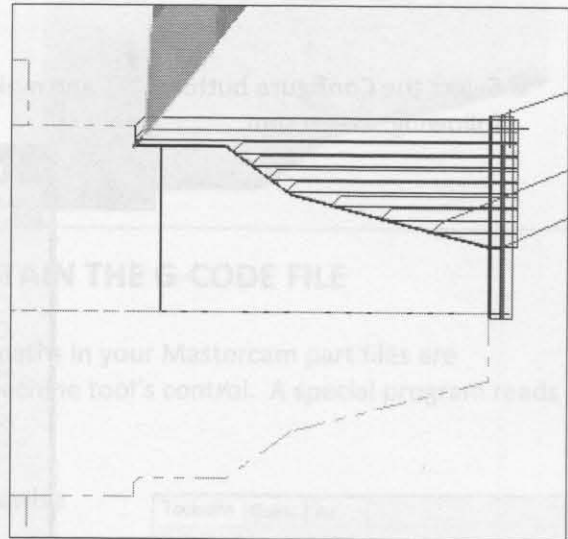


Make sure you have the following buttons turned on. (They look depressed when selected)

- Display tool
- Display rapid moves



Select Play button



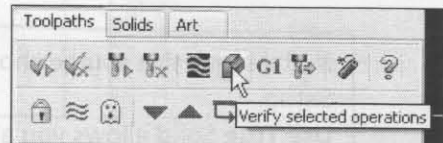
Select the OK button to exit Backplot.



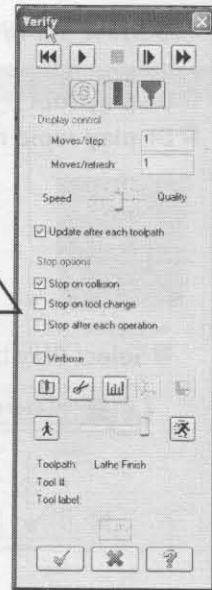
STEP 12: TOOLPATH VERIFICATION


Toolpath Verification: Allows you to use solid models to simulate the machining of a part. The model created by the verification represents surface finish, and shows collisions if any exist.

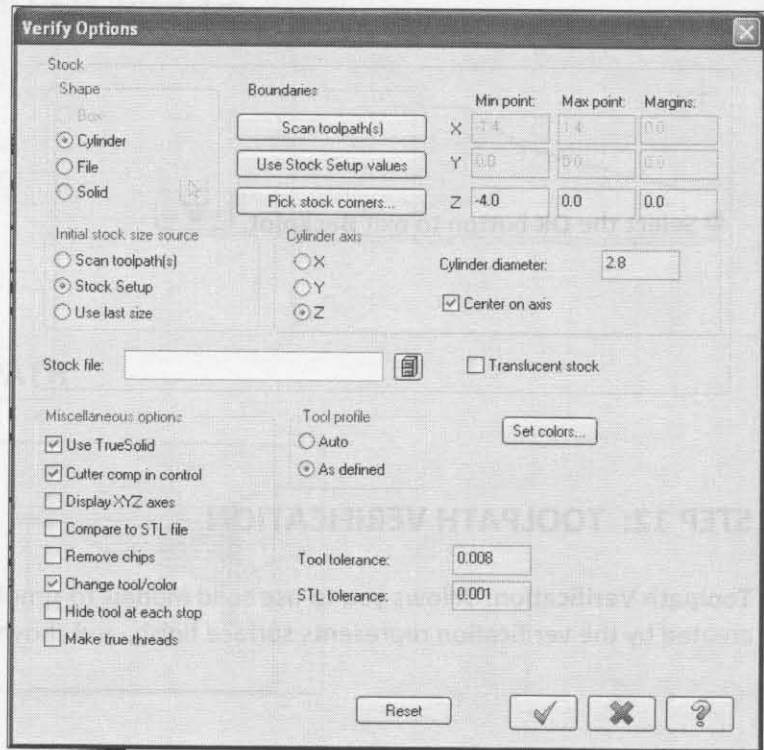
Select Verify all operations button.





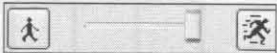

Update after each toolpath updates the stock after each operation.
Stop on collision pauses the verification when the tool touches the part with a rapid move



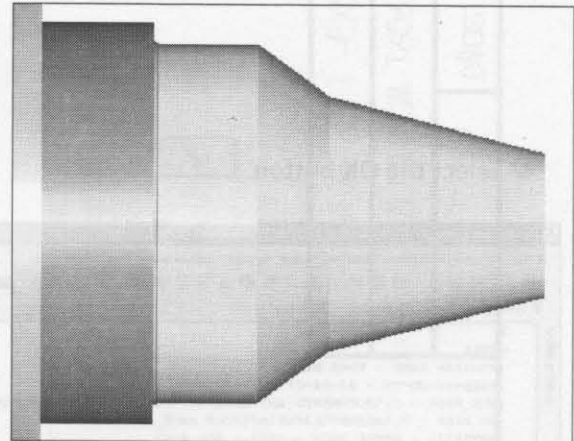
Select the **Configure** button  and make the necessary changes to the following screen shot.




Initial stock size source should be set to **Job Setup** to use the stock information from Stock Setup.
Use True Solid allows you after verifying the part, to rotate and magnify the part to more closely check features, surface finish, or scallops.
Cutter comp in control allows verify to use the information regarding the tool diameter and to simulate the cutter compensation
Change tool/color to change the color of the cut stock to indicate tool changes in the toolpath.
Simulate drill cycles allows the system to simulate peck drilling, chip break drilling cycles.

- ☉ Select the **OK** button to exit **Verify Options**. 
- ☉ Use **Fit Screen** icon to fit the solid to the screen. 
- ☉ Set the **Verify Speed** by moving the slider bar in the speed control bar. 
- ☉ Select **Machine** button to start the simulation. 

The finish part should appear as shown in the following picture.

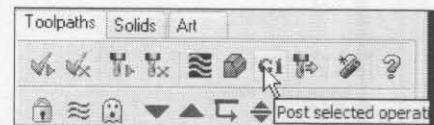


- ☉ Select the **OK** button to exit **Verify**. 

STEP 13: RUN THE POST PROCESSOR TO OBTAIN THE G-CODE FILE

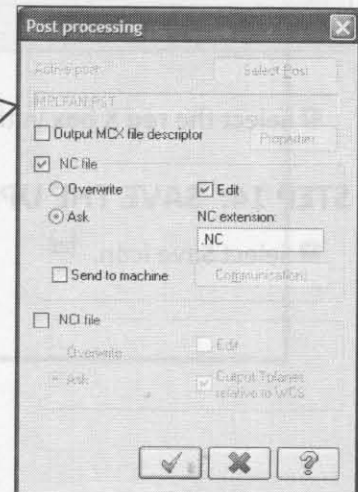
Post Processing: refers to the process by which the toolpaths in your Mastercam part files are converted to a format that can be understood by your machine tool's control. A special program reads your Mastercam file and writes the appropriate NC code.

- ☉ Make sure that all the operations are selected, otherwise
- ☉ **Select all visible operations** button.
- ☉ Select **Post selected operations** button from **Toolpath Manager**.



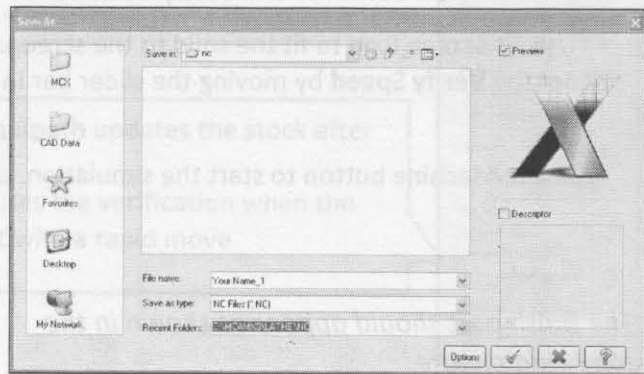
- ☉ In the **Post processing** window, make all the necessary changes as shown to the right.

NC file enabled allows you to keep the NC file and to assign the same name as the MCX file.
Edit enabled allows you to automatically launch the default editor.

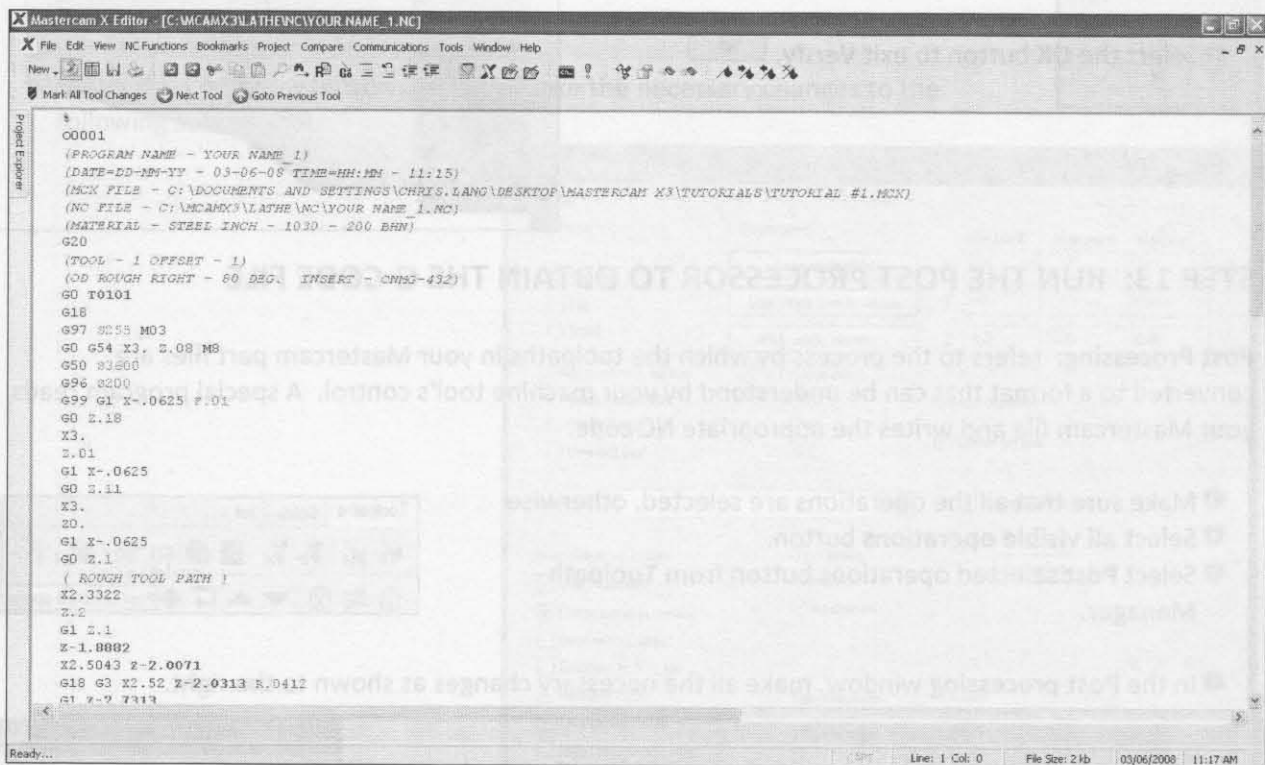


- ☉ Select the **OK** button to continue. 

Enter the same name as the geometry name in the NC file name field.



Select the OK button.



Select the red X box in the upper right corner to exit the editor.

STEP 14: SAVE THE UPDATED MCX FILE

Select Save icon.