

Chapter 2

Engraving Tutorial

Engraving is used to inscribe (a design, writing, character, etc.) onto a block, plate or other surface by carving, etching with acid or other process.

The Engraving process is a three axes milling machining process.

Engraving is used when tool follows a specified trajectory.

The following step-by-step guide will be covered in this Engraving tutorial.

- Introduction to 3D Modelling of Part
- Adding Colour Visualisation to the Part
- Adding Material to the Part
- Activating the Manufacturing application
- Importing and constraining the 3D Part
- Machine Coordinate (Programme Zero) creation
- Adding Cutting Tool
- Add Engraving Parameters (example feed rate, spindle speed, depth of cut, etc.)
- Generate Cutter Location (CL) Data
- G-codes generation

2.1 Step-by-Step Guide on How to Create a 3D Model Part (REC_PRT1)

This step-by-step guide on how to create an Engraved 3D Part is to aid new users to Creo Parametric.

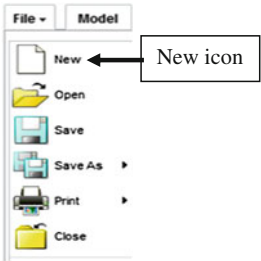
2.1.1 *How to Activate the Modelling Application*

Start Creo Parametric either from your Computer desktop or from Programme.

Click on New icon as shown below.

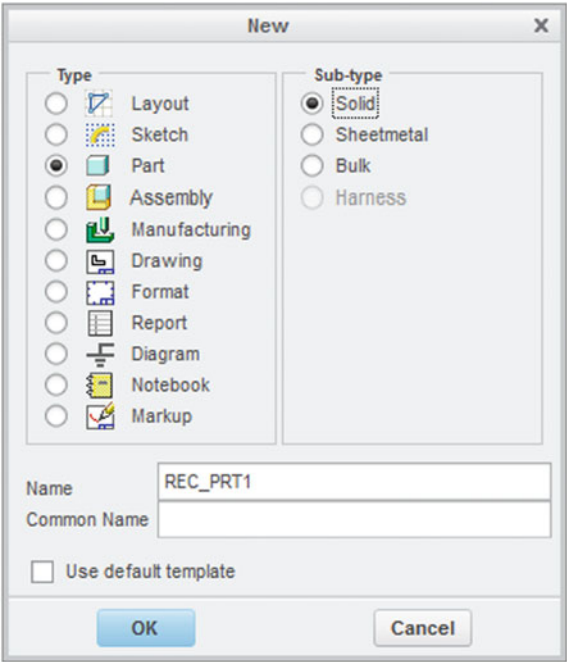


Alternatively, Click on File to activate its drop-down menu list >> Now click on New icon on the drop-down list as indicated by the arrow below.



The New dialogue box is activated >> Click in the Part radio button on the Type group >> Click in the Solid radio button on the Sub-type group >> Type REC_PRT1 in the Name section box >> Clear the Check Mark icon in the Use default template square box and the New dialogue box will look as shown in Fig. 2.1.

Fig. 2.1 New dialogue box with Part and Solid highlighted

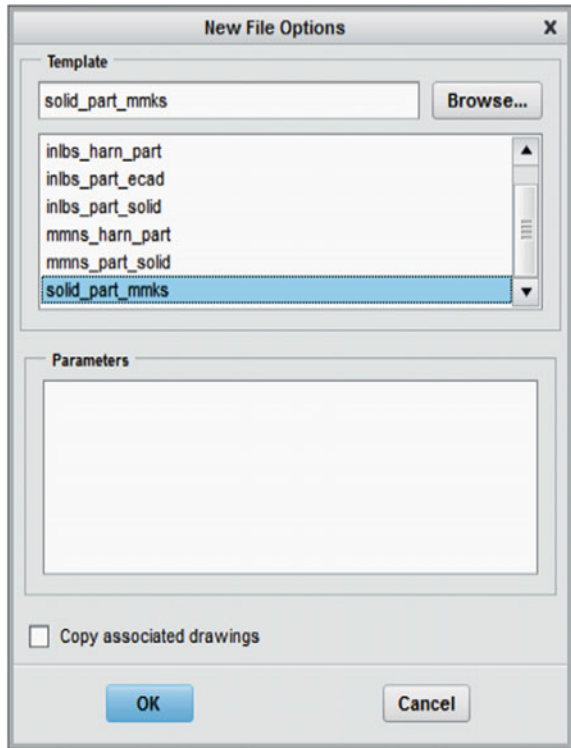


Click on OK tab to exit.

The New File Options dialogue window is activated >> On the Template group select “solid_part_mmks” as the unit to be used in this tutorial >> Clear the Copy associate drawings Check Mark icon in the square box. The New File Options dialogue window is as shown in Fig. 2.2.

Note: You can also select either “mmns_part_solid” or “solid_part_mmks” if you are working in SI units.

Fig. 2.2 New File Options dialogue box with solid_part_mmks highlighted



Now click on OK tab to exit.

Note: Any of the standard units of measurement above can be selected based on the unit you are working with.

The Model Graphic User Interface (GUI) window is activated as shown (See Figs. 2.3).

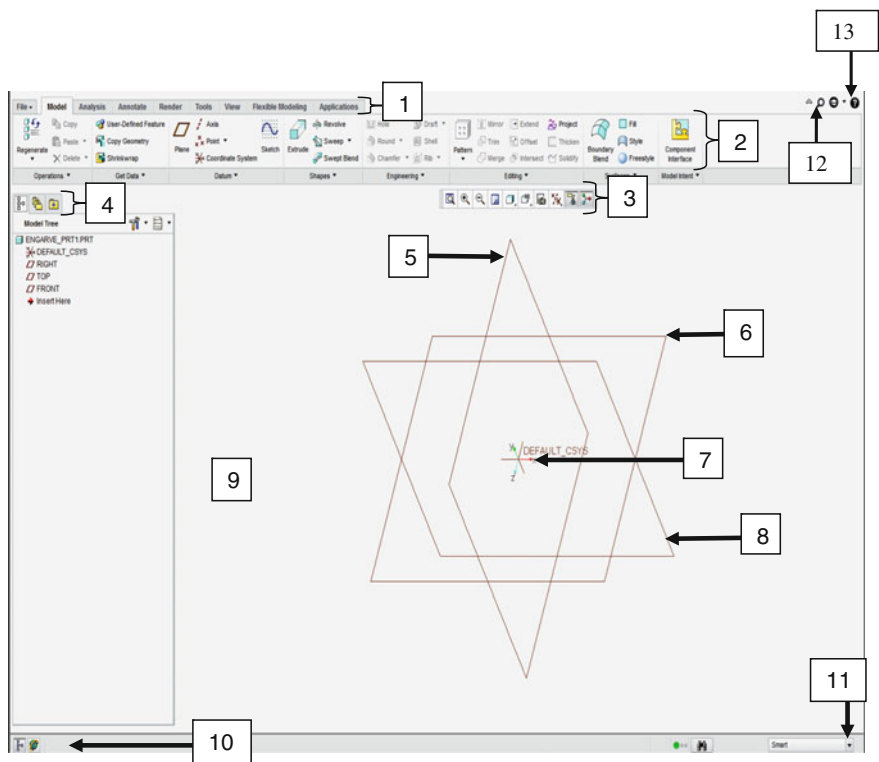


Fig. 2.3 The Model main Graphic User Interface Window

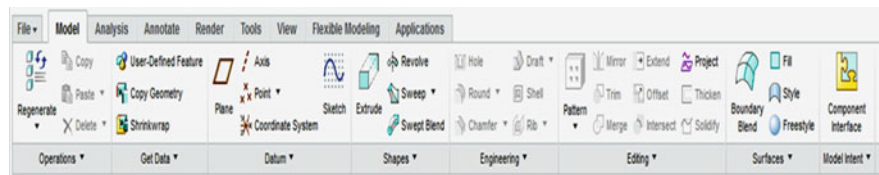


Fig. 2.4 Activated Model ribbon/tools in concise form

where

- 1 represents the Menu Bar (Main Tab)
- 2 represents the Model Ribbons (Model menu list)
- 3 represents the Default Graphics Toolbar (View and Display Tools)

- 4 represents the Navigator (Contains the Model Tree, Folders and Favourites)
- 5 represents the Right Datum Plane
- 6 represents the Top Datum Plane
- 7 represents the Default Coordinate System
- 8 represents the Front Datum Plane
- 9 represents the Main Graphic Window (Graphic User Interface Window (GUI))
- 10 represent the Message Bar
- 11 represent the Filter Selection drop-down list (Active filter)
- 12 represent the Command Search
- 13 represent the Creo parametric Help

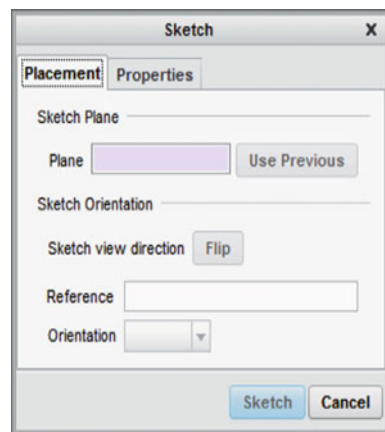
2.1.2 Activate 2D Sketch Application

Click on the Sketch icon on the Model menu list as shown below.

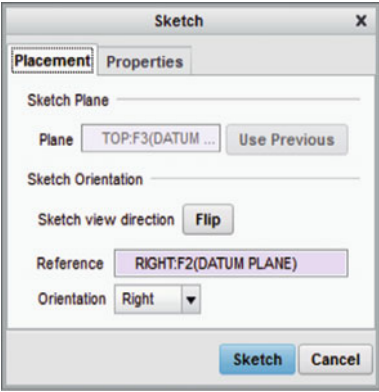


The Sketch dialogue box is activated on the main graphic window as shown in Fig. 2.5.

Fig. 2.5 Activated Sketch dialogue box



Click on the Placement tab >> On the Sketch Plane group, click in the Plane section box >> Now click on the Top Datum Plane on the Model Tree and the Sketch dialogue box is automatically updated. Make sure that the Orientation is set to Right as shown. See Fig. 2.6.



Click on Sketch tab to exit.

Fig. 2.6 Updated Sketch dialogue box

The Sketch ribbon tools are activated as shown in Fig. 2.7.

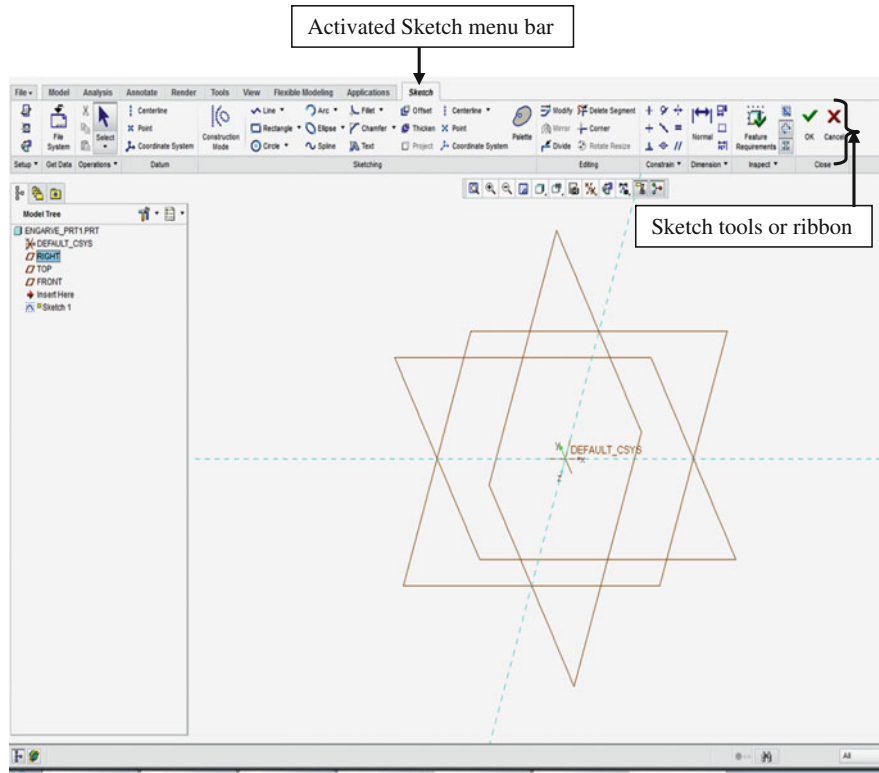


Fig. 2.7 Activated Sketch tools and datum plane display on the main Graphic User Interface (GUI) window

The concise Sketch ribbon tools are activated. See Fig. 2.8.

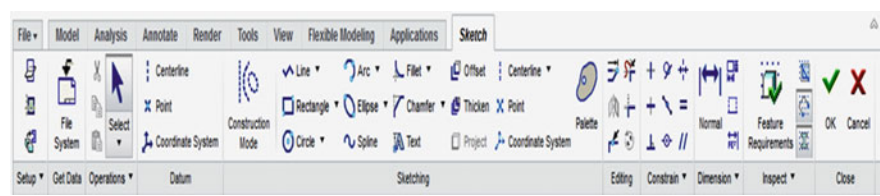


Fig. 2.8 Concise Sketch tools in concise form

Now click on the Sketch view icon  to position the Top Datum Plane as the sketch plane (orient the sketch plane parallel to the screen) as shown in Fig. 2.9.

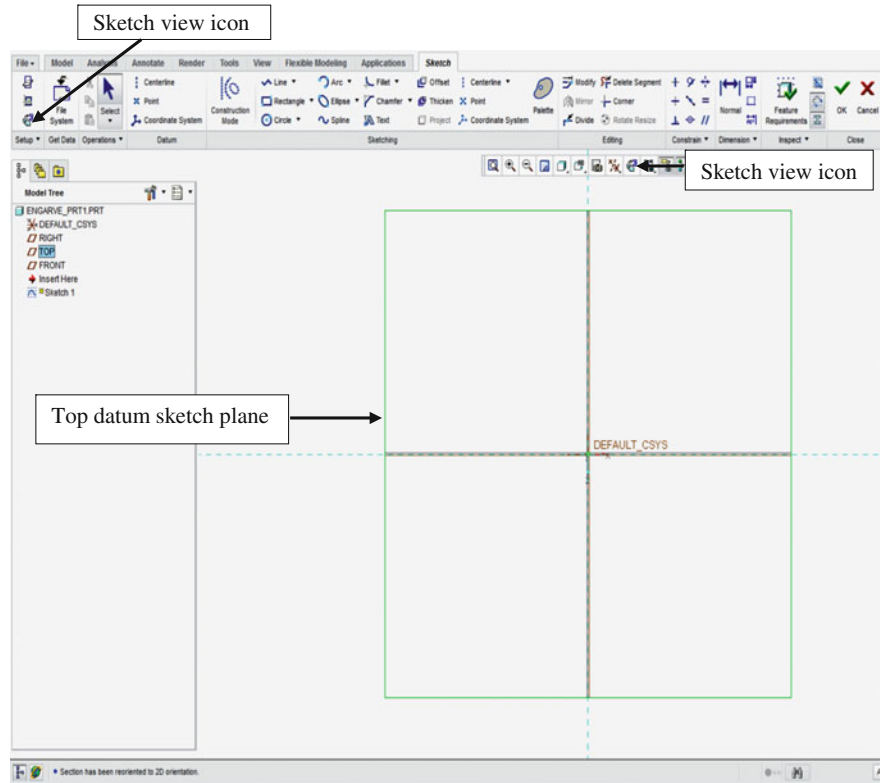


Fig. 2.9 Orientation of Sketch view on the main graphic window

Click on the Rectangle icon downward arrow to activate its drop-down menu >> Now click on Centre Rectangle on the drop-down list as shown in Fig. 2.10.

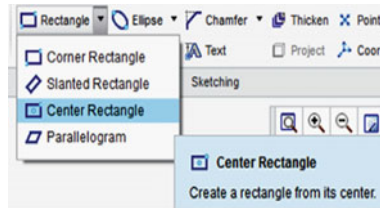


Fig. 2.10 Activated Rectangle drop-down menu list

The mouse pointer changes to an X symbol at its tip.

Place the arrow at the intersection point between the vertical and horizontal dotted line on the sketch plane and draw/sketch a rectangle of 400 × 250 mm.

Click on the Chamfer icon tab to activate its drop-down menu list >> Now click on Chamfer on the drop-down list as highlighted and shown in Fig. 2.11.

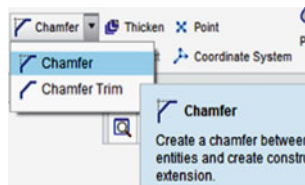


Fig. 2.11 Activated Chamfer drop-down menu list

Click on the vertical and horizontal lines of both ends of the bottom/corner sketch to add chamfer to corner sketch >> To Change the chamfer dimension click on the Select icon in the Operation group on the active Sketch ribbon tools as shown below.



Now click on the dimension that you want to change its value and type 20 mm as the new chamfer value/length >> Press the Enter key on your key board to accept.

Click on the Fillet icon tab to activate its drop-down menu >> Now click on Circular on the drop-down list as highlighted as shown in Fig. 2.12.

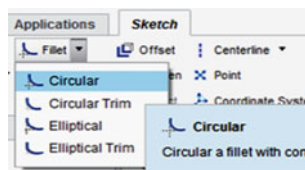


Fig. 2.12 Activated Fillet drop-down menu list

Click on both the vertical and horizontal lines of the upper sketch to add the fillet to both top end corners/edges >> To change the fillet dimension, click on the Select icon as shown below.



Now click on the dimension that you want to change its value and type 20 mm as the new fillet radius >> Press the Enter key on your key board to accept.

The changes made to the sketch are as shown in Fig. 2.13.

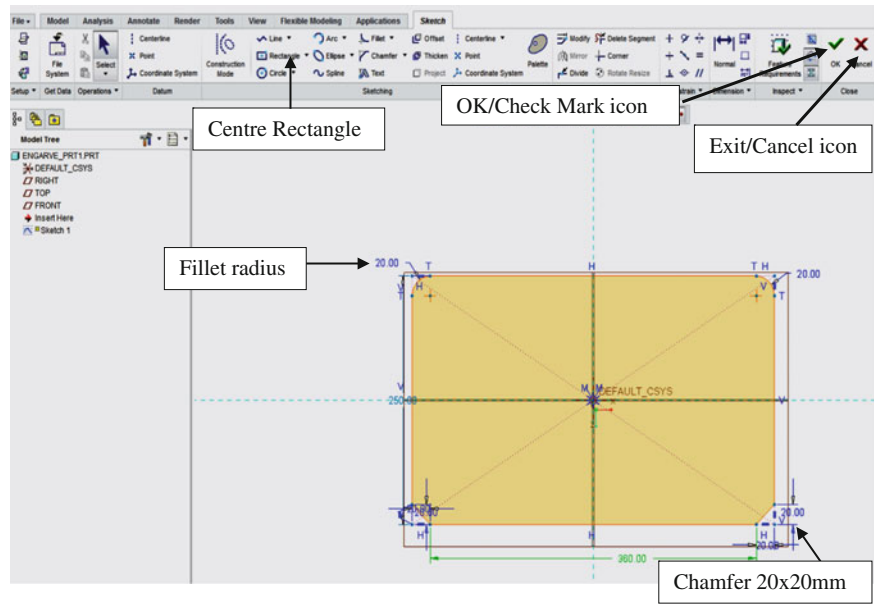


Fig. 2.13 Sketching of the rectangle on the main graphic window

Click on the OK icon as indicated by the arrow shown in Fig. 2.13; if you are satisfied with your sketch to exit the Sketch application.

The main GUI window changes as shown in Fig. 2.14.

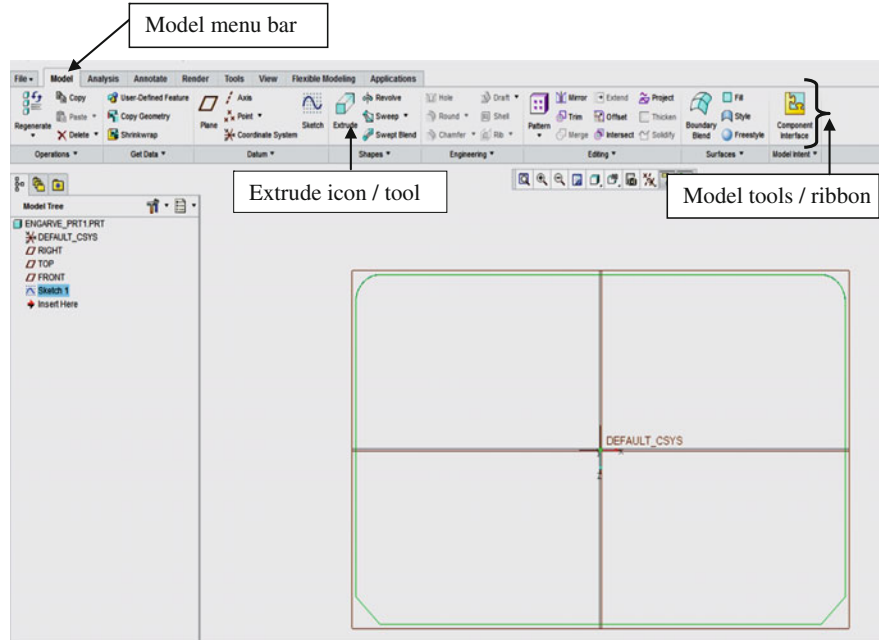


Fig. 2.14 Finished Sketch with corner chamfers and fillets

2.1.3 Activate Extrude Application

Now click on the Extrude icon on the Model tools/ribbons as shown below.



The Extrude dashboard tools are activated as shown in concise form (Fig. 2.15).

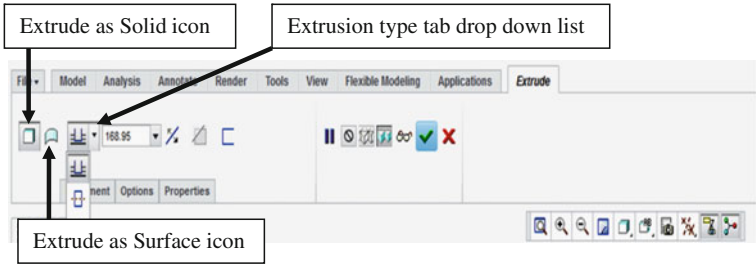


Fig. 2.15 Concise Extrude dashboard

Make sure that the Extrude as Solid icon tab is activated as indicated in Fig. 2.15
» Click on the Extrusion type tab downward arrow to activate its drop-down menu, now click on “Extrude from sketch plane by a specified depth value” on the drop-down menu list. » In the “Extrude from sketch plane by a specified depth value” value section box, type 50 mm to be depth of extrusion as illustrated in Fig. 2.16.

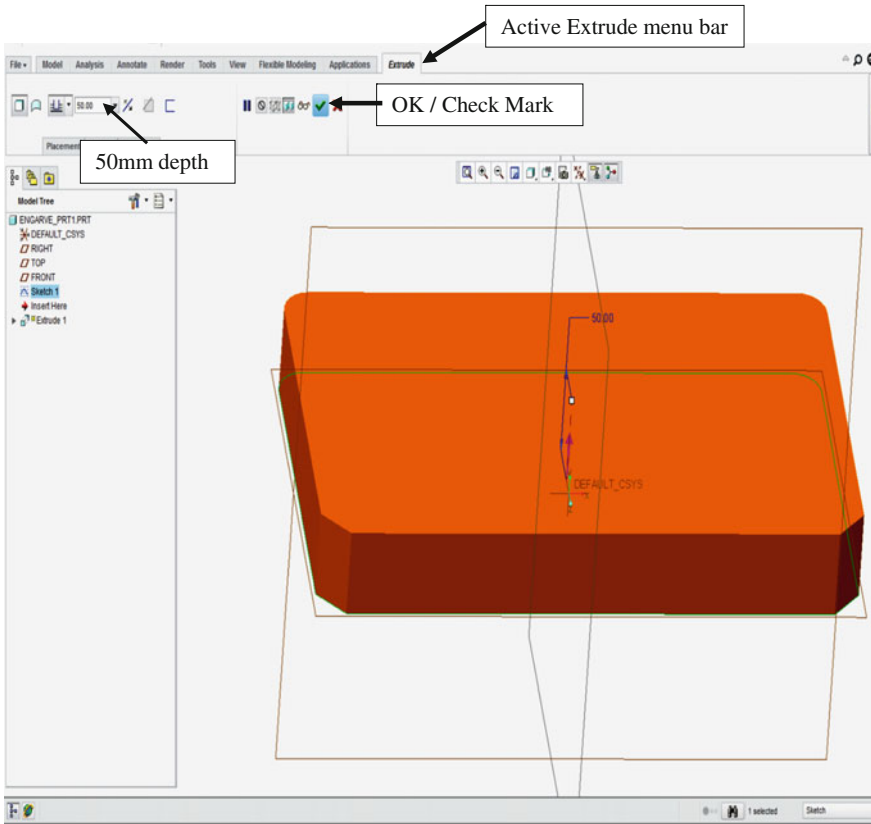


Fig. 2.16 Extruded sketched Part

Click on the OK/Check Mark icon.

After clicking on Check Mark, the Part changes on the main GUI window as shown in Fig. 2.17.

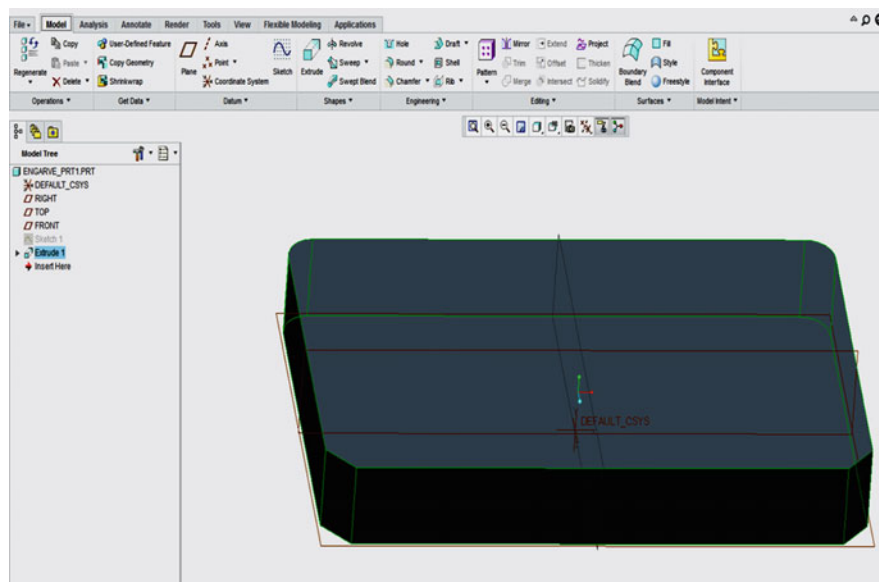


Fig. 2.17 Extruded 3D Part with corner chamfers and fillets

2.2 Add Material and Colour Visualisation to the 3D Part

2.2.1 Add the Part Visualisation Colour to Polished Brass Colour

Click on the View menu bar to activate the View ribbon as shown in Fig. 2.18.

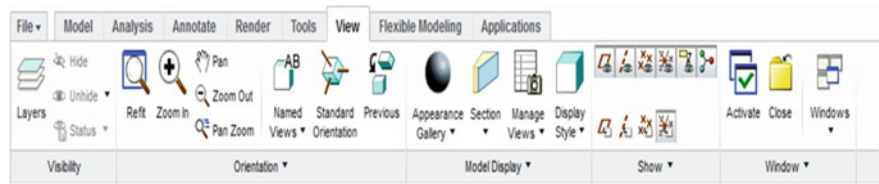
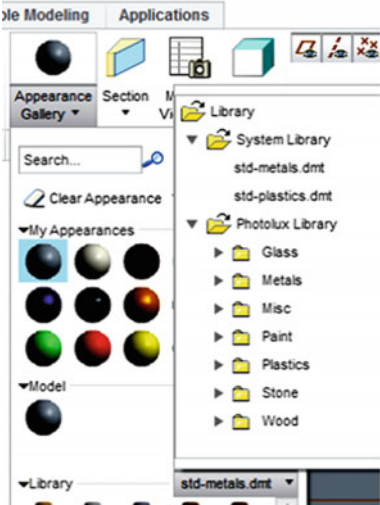


Fig. 2.18 Activated View ribbon in concise form

Click on the Appearance Gallery icon  to activate its content.

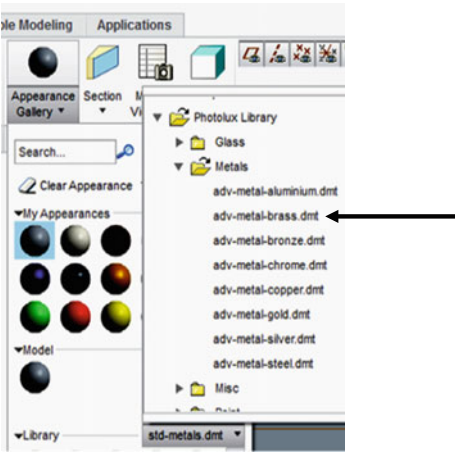
Go to Library, and click on the “std-metals.dmt” tab to activate its content as illustrated in Fig. 2.19.

Fig. 2.19 Activated
std-metals.dmt group



On the drop-down menu list, click on the Metals folder arrow to expand its contents >> Now click on the “adv-metal-brass.dmt” on the list as indicated by the arrow shown in Fig. 2.20.

Fig. 2.20 Activated Metals
folder drop-down list



On the “adv-metal-brass.dmt” list, now click on the “adv-brass-polished” photolux appearance as indicated by the arrow as shown in Fig. 2.21.

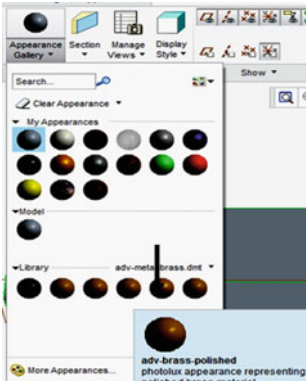


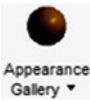
Fig. 2.21 Activated adv-metal-brass.dmt group

The mouse pointer changes from an arrow to a blush icon >> The Select dialogue box is activated on the main window as shown in Fig. 2.22.



Fig. 2.22 Activated Select dialogue box

The Appearance Gallery icon will change to “adv-brass-polished” colour as shown below.



Add the “adv-brass-polished” photolux appearance

To add the “adv-brass-polished” material colour to the Part >> Click on the “adv-brass-polished” icon on the Appearance Gallery tab, and the Select dialogue box is activated of the main window as shown in Fig. 2.23. If it is not active already.

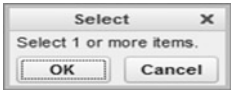
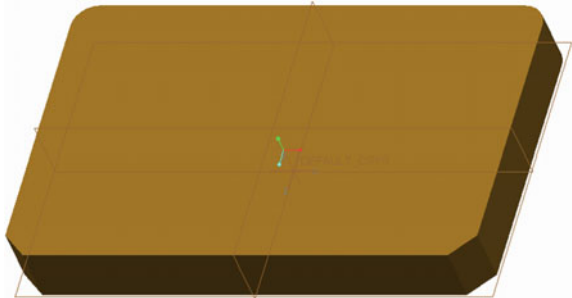


Fig. 2.23 Activated Select dialogue box

Now start clicking on the Part surfaces and edges while holding down the Ctrl key on your key board >> Click on the OK tab when you are done to exit the Select dialogue box. The Part changes colour as shown in Fig. 2.24.

Fig. 2.24 The “adv-brass-polished” material colour added to the Part

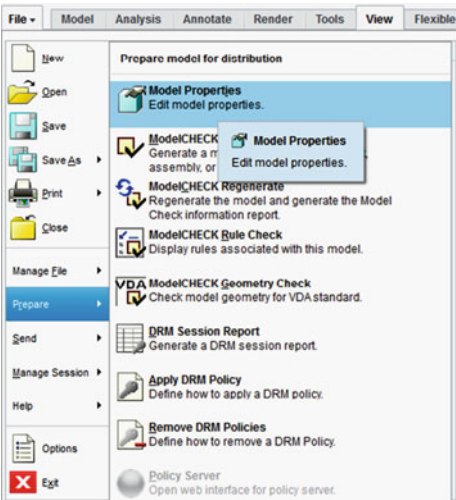


Note: Remember to release the Ctrl tab while selecting the Part surfaces and edges, by doing so you will be able to rotate the Part.

2.2.2 Add Material to the Part

To add the “adv-brass-polished” material to the Part >> Go to File on the menu bar and click on its downward arrow to activate its drop-down menu >> Now click on Prepare to activate its content >> On the Prepare model for distribution list, click on Model Properties as highlighted in Fig. 2.25.

Fig. 2.25 Activated Prepare panel content



The Model Properties dialogue window is activated as shown in Fig. 2.26.

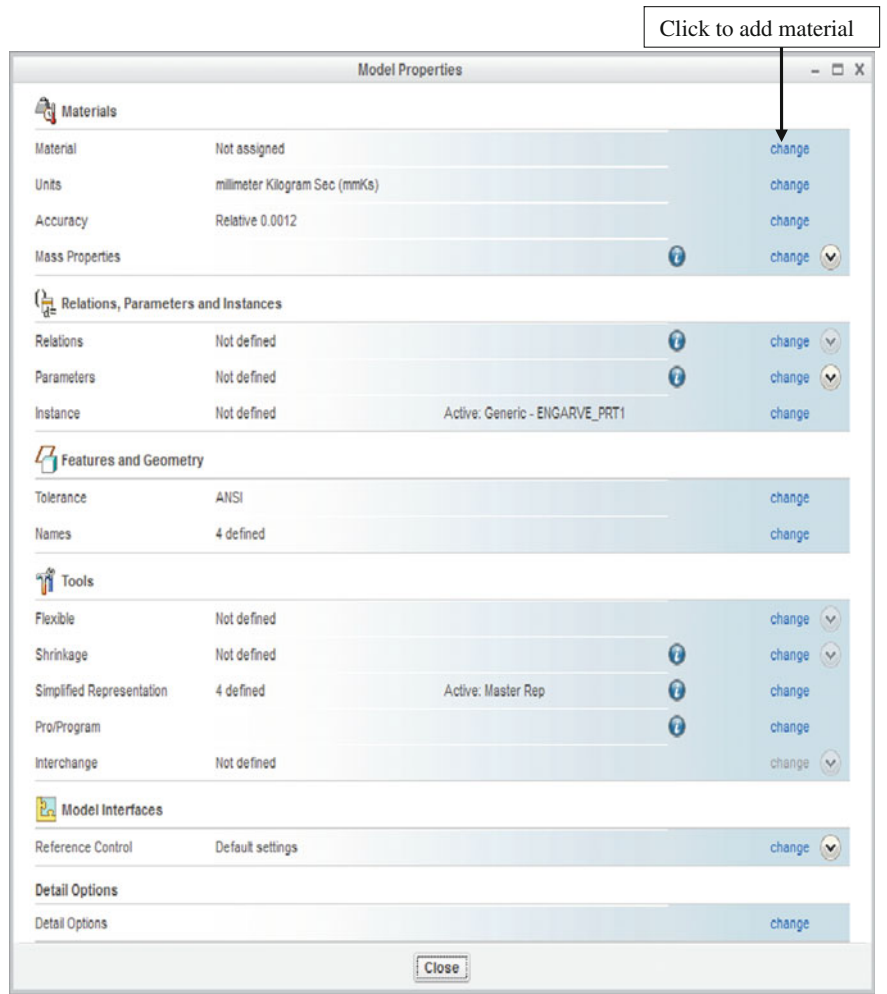


Fig. 2.26 The Model properties dialogue window

Click on Change on the Material section, under the Materials group as indicated by the arrow shown in Fig. 2.27.

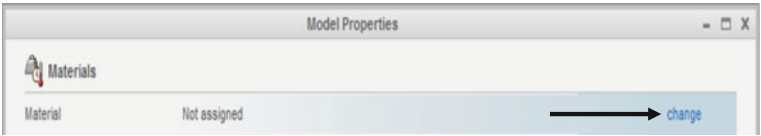


Fig. 2.27 Activating Change on material section

The Materials dialogue window is activated as shown in Fig. 2.28.

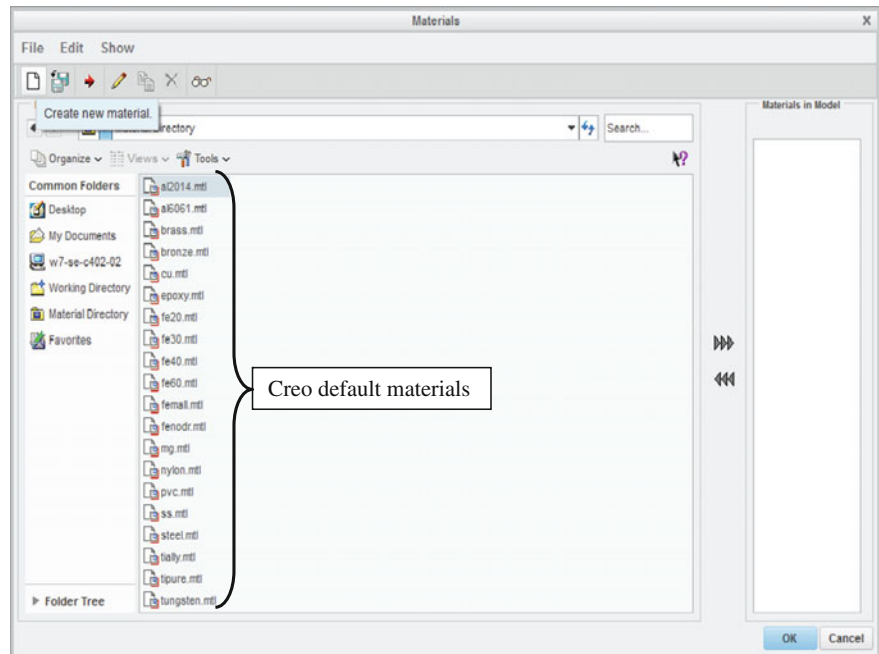


Fig. 2.28 Activated Materials dialogue window

Choose from the lists of default materials if you want to. And click on the red arrow as shown below at the top to assign material to the Part.



The material assigned now appears on the Material in Model section box as indicated by the arrow shown in Fig. 2.29.

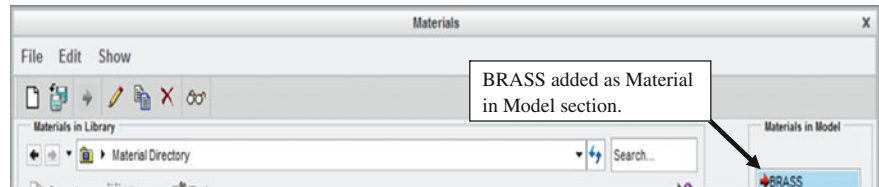
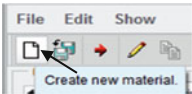


Fig. 2.29 BRASS in Materials in Model section

Now click on the Ok tab to exit the Materials dialogue box.

Alternatively, add material to Part through Create new material as will be illustrated and used in this tutorial.

Click on the Create new material icon on the Material dialogue window as shown below.



The Material Definition dialogue window is activated on the main graphic window >> Type “adv-brass-polished” in the Name section as shown in Fig. 2.30.

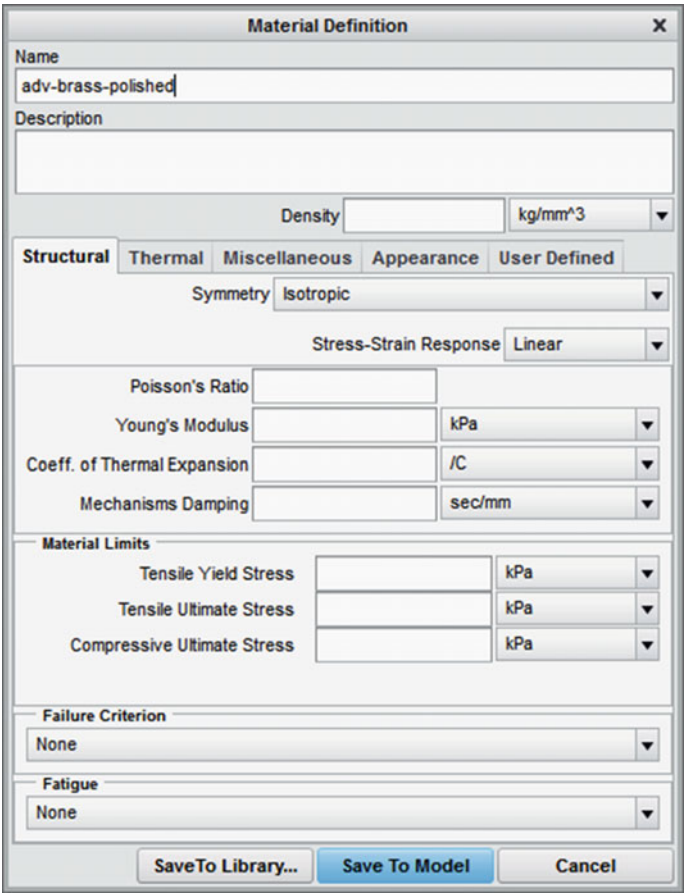


Fig. 2.30 The Material Definition dialogue window

Note: For advanced users, you can create your own new Material properties if you know the material Poisson’s Ratio, Young’s Modulus etc.

Click on the Appearance tab >> Now click on the New tab as indicated by the arrow in Fig. 2.31.



Fig. 2.31 Activating new appearance

The Material Appearance Editor dialogue window is activated as shown in Fig. 2.32.

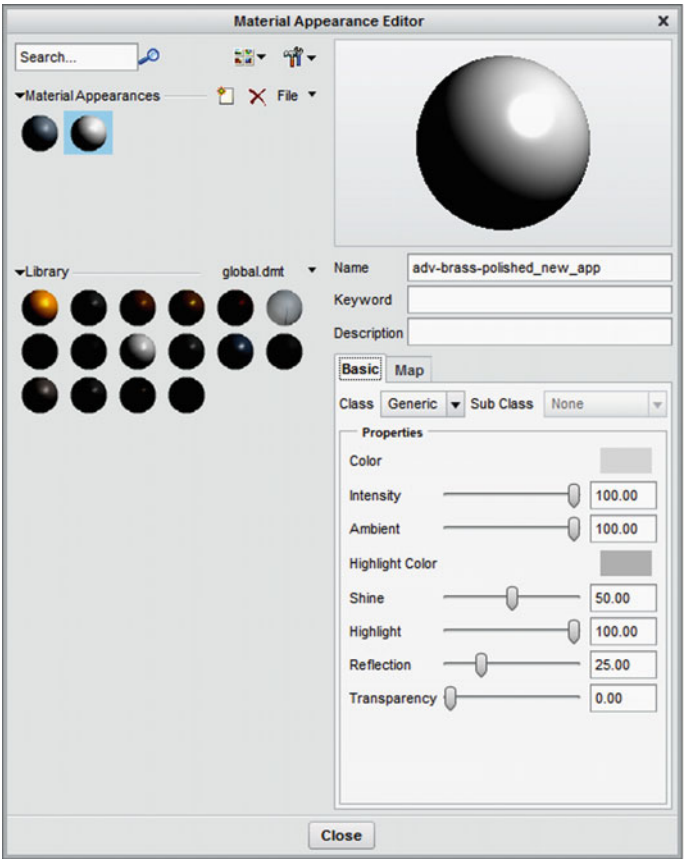


Fig. 2.32 The Material Appearance Editor dialogue window

On the Material Appearance Editor dialogue window >> Click on the “global.dmt” downward pointing arrow to activate its content >> On the drop-down menu list, click on the Metal folder arrow to activate its content.

Now click on the “adv-metal-brass.dmt” as indicated by the arrow shown in Fig. 2.33.

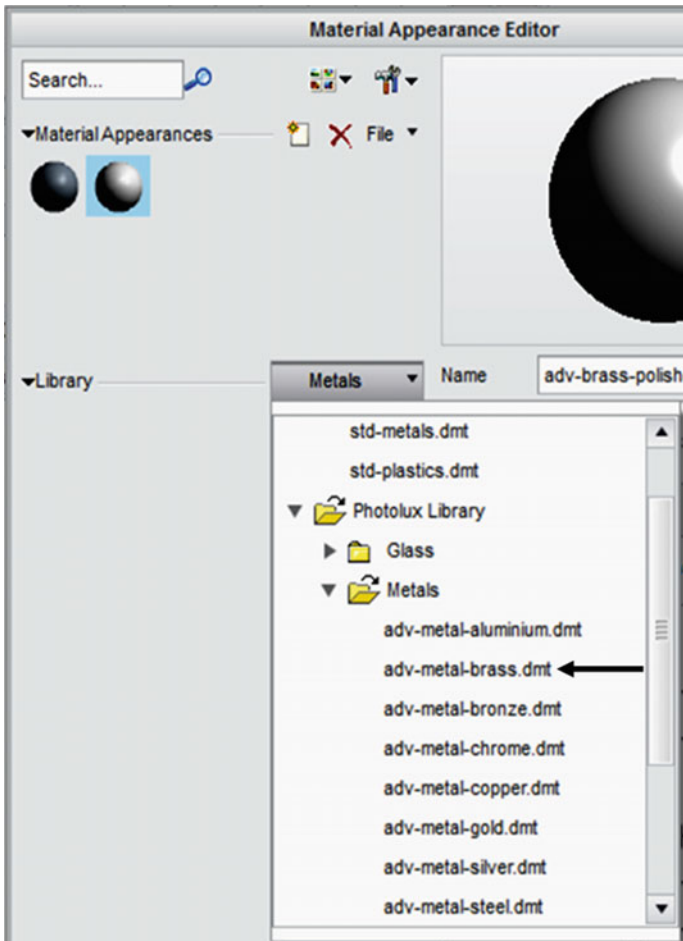


Fig. 2.33 The activated Metals folder panel

The “adv-metal-brass.dmt” group is activated as shown in Fig. 2.34. On the “adv-metal-brass.dmt” group list, click on the “adv-brass-polished” tab indicated by the arrow as shown in Fig. 2.34.

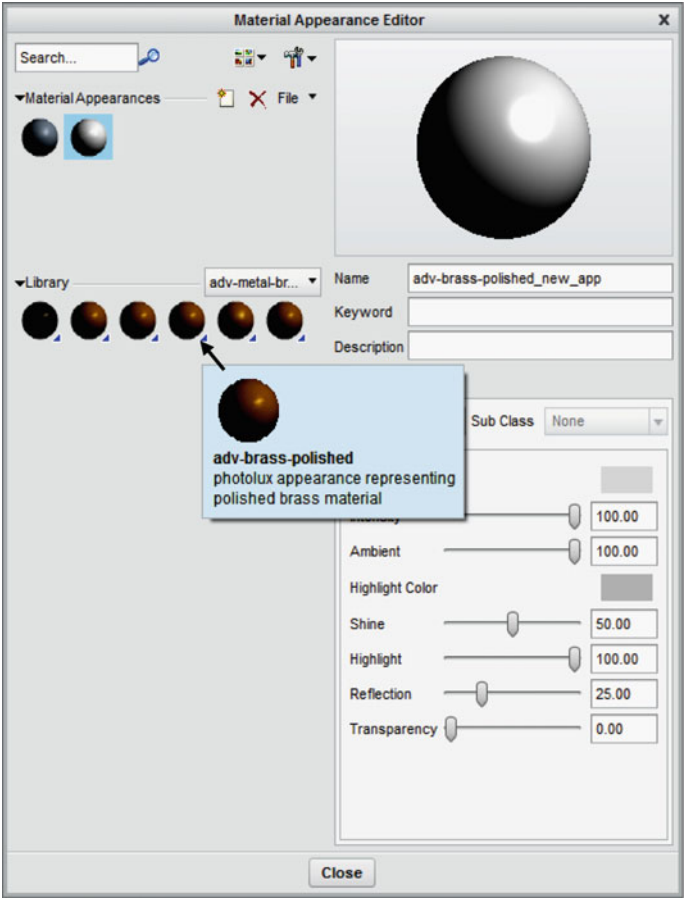


Fig. 2.34 The selected “adv-brass-polished” material to be added to Part

Now click on Close tab to close the Material Appearance Editor dialogue window.

On the Material Definition dialogue window, click on the “adv-brass-polished_new_app” >> Now click on the Default Red arrow tab as shown in Fig. 2.35.

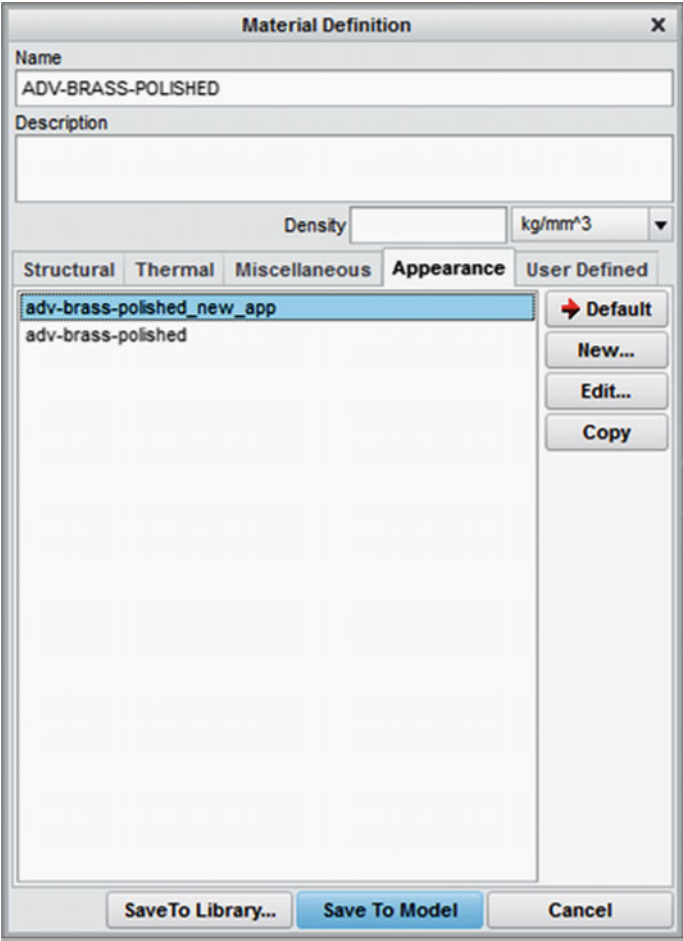


Fig. 2.35 The “adv-brass-polished_new_app” material is now added to Appearance section box

Click on the Save To Model tab as highlighted above to save the created material in the Materials Directory.

Note: When saving to a different material library folder click on Save To Library tab.

The ADV-BRASS-POLISHED is now in the Material in Model section as indicated by the arrow shown in Fig. 2.36.

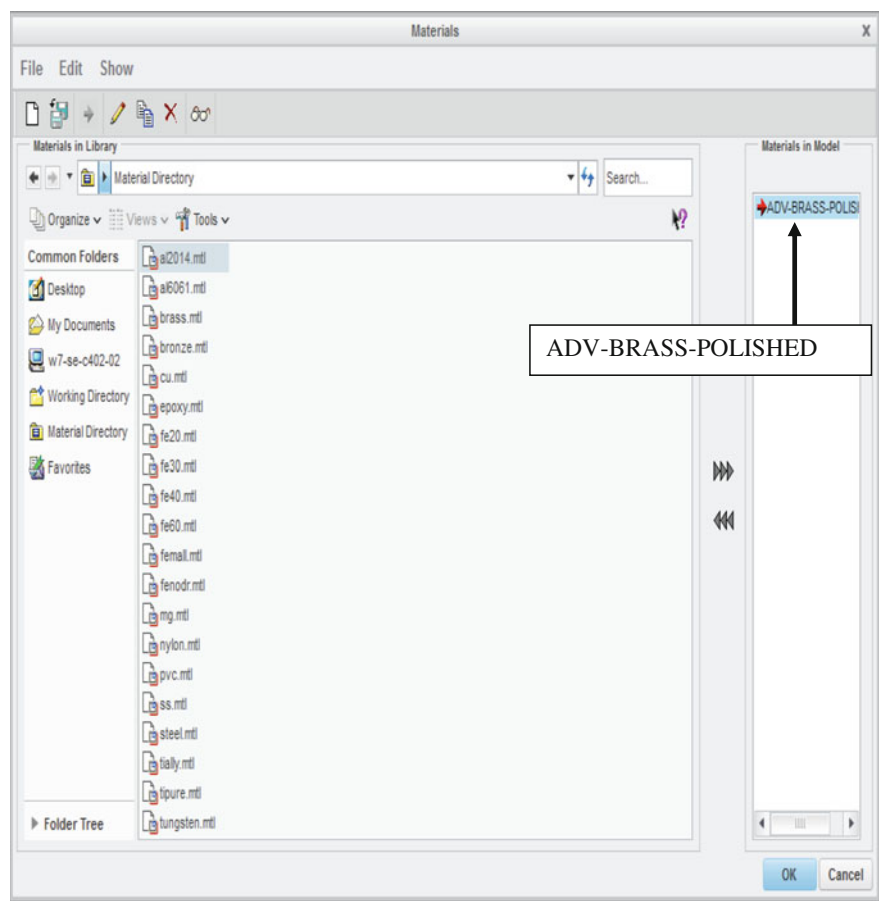


Fig. 2.36 The “adv-brass-polished_new_app” material is now added on the Model section box

Now click on the OK tab to exit.
The Model Properties dialogue window is activated indicating that the “adv-brass-polished” material is now the material added to the 3D Part as indicated by the arrow in Fig. 2.37.

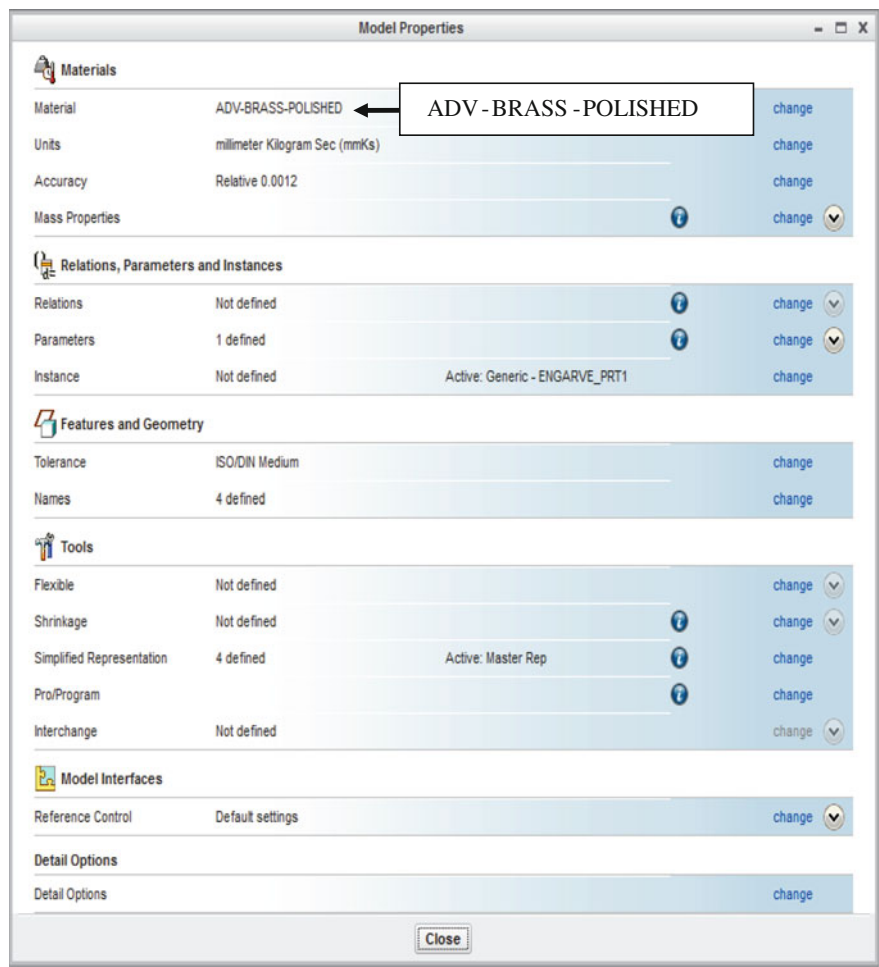


Fig. 2.37 Model Properties dialogue box showing the added material on the Material section

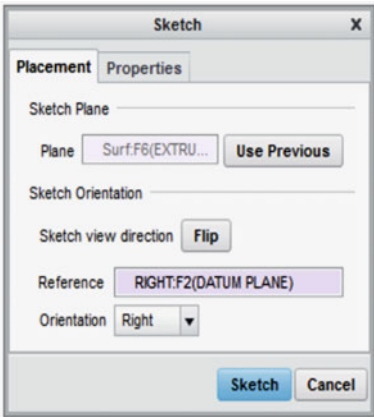
Now click on the Close tab to close the window.
Click on the Save icon to save your work.

2.3 Create the Engraving Imprint on the Part Surface

On the Model menu bar >> Click on the Sketch icon as shown below.



The Sketch dialogue box is activated on the main graphic window >> Click on the Placement tab >> Now click on the Top surface of the Part. The Plane, Reference and Orientation section boxes are updated automatically by the system. Make sure that Orientation is set to Right. See Fig. 2.38.



Click on the Sketch tab to exit.

Fig. 2.38 Updated Sketch dialogue

The Sketch tools are activated with all the sketching tools (See Fig. 2.39).

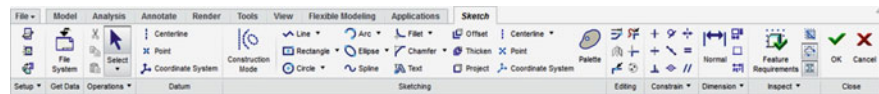
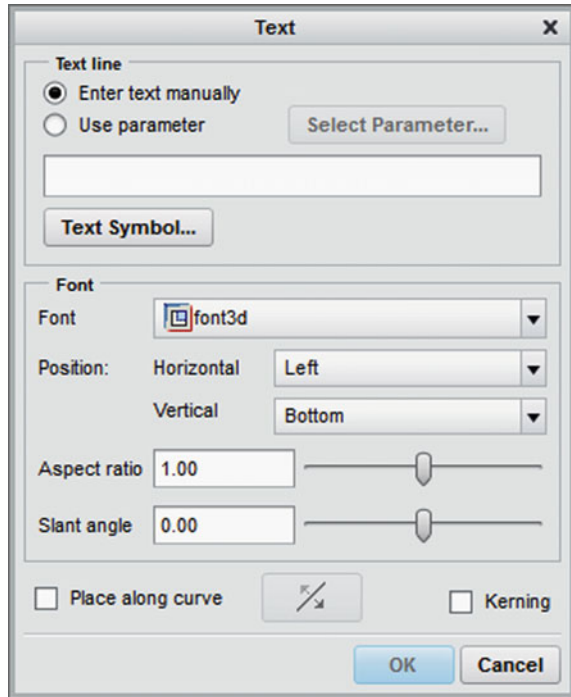


Fig. 2.39 Activated Sketch tools

To imprint “MANUFACTURE” on the Part surface >> Click on the Text icon and now click on the Part surface to determine the start point, click again to determine the text height. The Text dialogue box is activated as shown in Fig. 2.40.

Fig. 2.40 Activated Text dialogue box

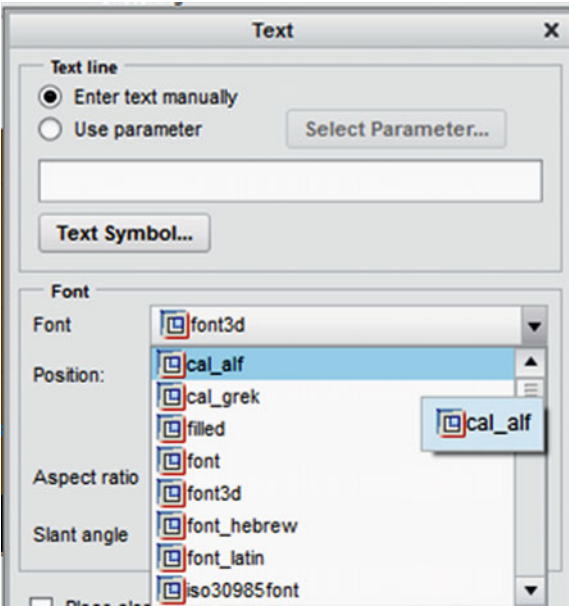


Note: Remember that the length of the line will determine the height of your text, but do not worry because it can be edited easily to the correct height. Note also, that the method of creating the text height on Part surface will determine the orientation of the text. It is advisable to click “bottom to top” and not top to bottom on the Part surface.

On the Text dialogue box, click in the “Enter text manually” radio button on the Text line group to make it active >> Now type “MANUFACTURE” in the empty section box.

On the Font group, click on the downward pointing arrow on the Font section box >> On the activated drop-down list click on cal_alf as shown in Fig. 2.41.

Fig. 2.41 Activated Font drop-down menu list



On the Font group, set the Horizontal, Vertical and Aspect ratio section boxes as illustrated in Fig. 2.42.

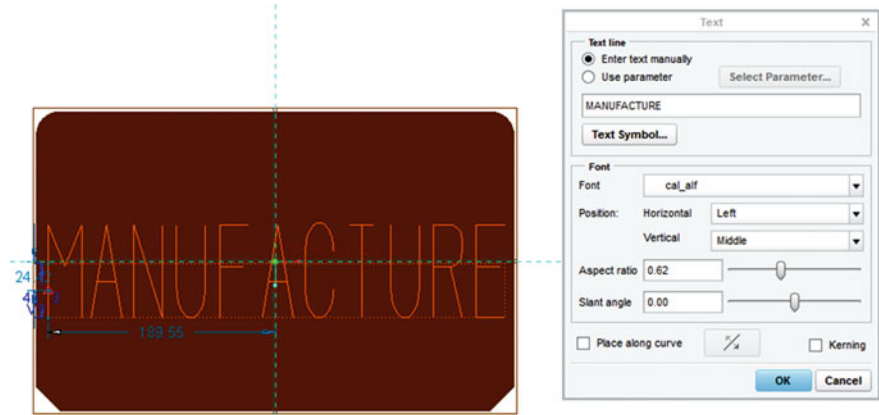
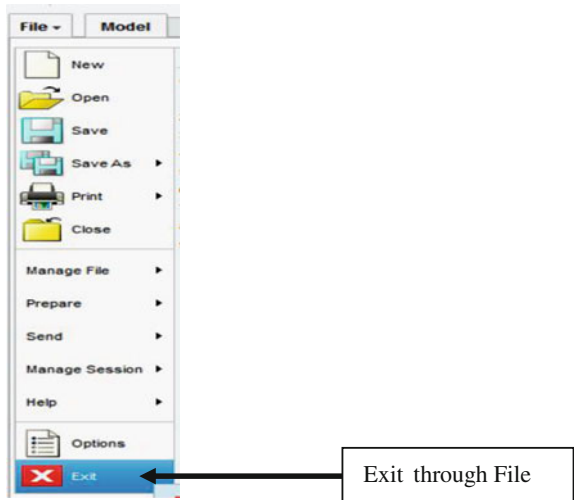


Fig. 2.42 MANUFACTURING imprint on Part surface and Text dialogue box

Click on the OK tab to exit the Text dialogue box.
Now click the Check Mark icon to exit the Sketch application.
Save the Engraved Part in the selected directory, folder or drive as ENGRAVED_PRT1.

Now to exit the Modelling application >> Go to File and click on the downward arrow >> Click on Exit on the drop-down menu list as indicated by the arrow shown in Fig. 2.43.

Fig. 2.43 Activated File drop-down menu list



Alternatively, click on the Close icon on the GUI main window as indicated by the arrows shown below.



The CONFIRMATION dialogue box is activated as shown in Fig. 2.44.

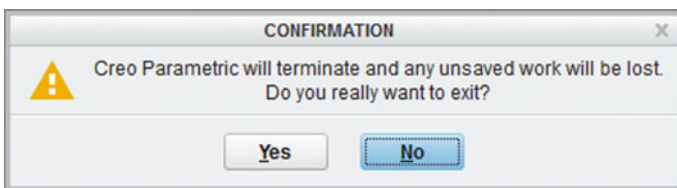


Fig. 2.44 The CONFIRMATION dialogue box

Click on the Yes tab to confirm and exit Creo Parametric completely.

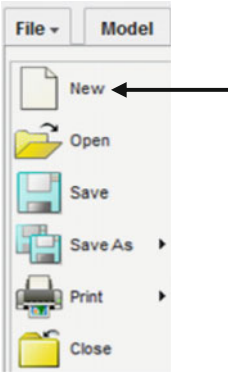
Congratulations for getting to this stage in this tutorial. For a new user, all that is required is patience and to practice as much as you can and play with different application/toolbar icon to get familiar with using them.

2.4 Manufacturing Procedure of the Engraved 3D Part

2.4.1 Activate the Manufacturing Application

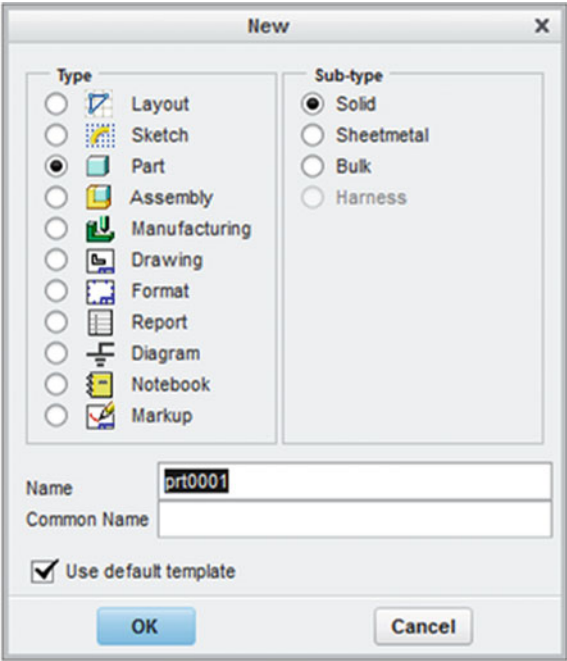
Start Creo Parametric either from your Computer desktop or from Programme.
Click on File to activate its drop-down menu list >> Now click on New on the drop-down menu list as indicated below by the arrow as shown in Fig. 2.45.

Fig. 2.45 Activated File drop-down menu list



The New dialogue box is activated on the main graphic window as shown in Fig. 2.46.

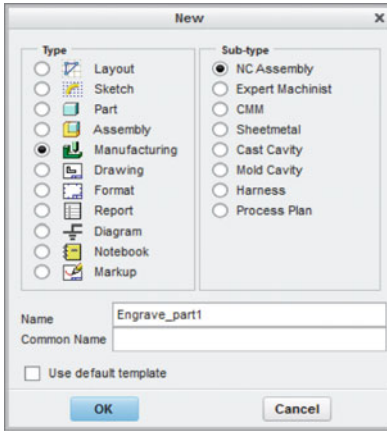
Fig. 2.46 The activated New dialogue box



Change the Radio button on the Type and Sub-type groups on the New dialogue box

On the Type group, click in the Manufacturing radio button to make it active >> On the Sub-type group, click in the NC Assembly radio button to make it active.

Type the name (Engave_PRT1) in Name section box. Clear the Use default template as shown in Fig. 2.47.



Now click on OK tab to exit.

Fig. 2.47 Manufacturing and NC Assembly radio buttons activated

The New File Options dialogue window is activated >> Under the Template group, click on “mmns_mfg_nc” >> On the Parameters group, you can type your name and a brief description on the “MODELLED_BY” and “DESCRIPTION” section boxes, respectively, if you want and clear the copy associated drawings square box as shown in Fig. 2.48.

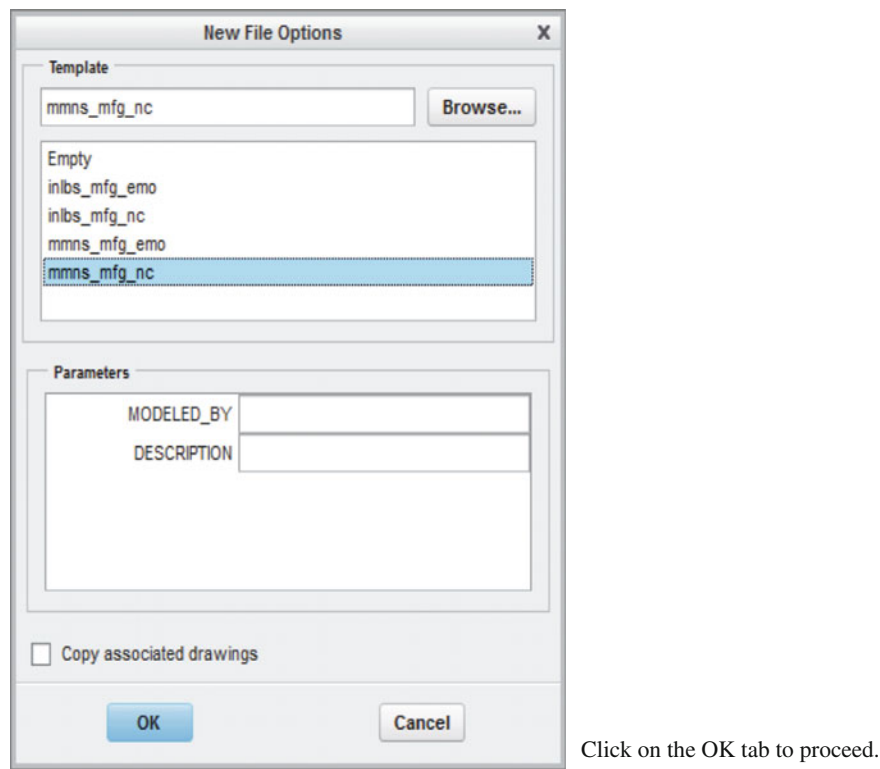


Fig. 2.48 The New File Options dialogue box

The Manufacturing main Graphic User Interface (GUI) window is activated as shown in Fig. 2.49.

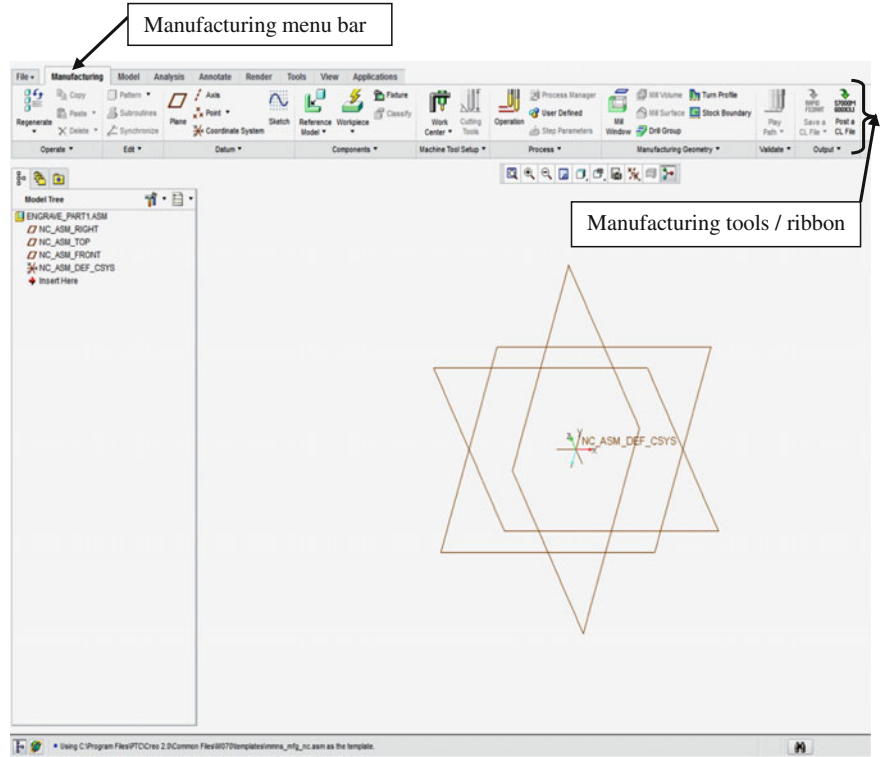
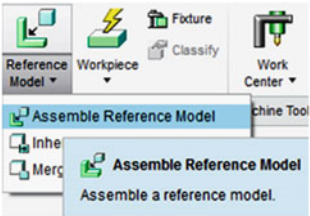


Fig. 2.49 Activated Manufacturing main GUI window

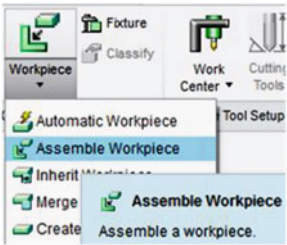
2.4.2 Import and Constrain the Reference Model

Import 3D Part

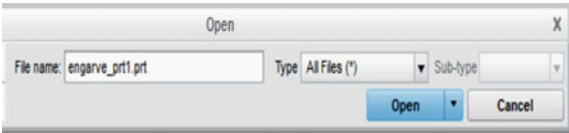
Click on the Reference Model icon to activate its drop-down list >> Now select Assembly Reference Model on the drop-down list as illustrated below.



Note: The Assemble Workpiece option as shown below is used to assemble together two Parts, where one is the imported Part and the other is the imported Workpiece.



The Open dialogue window is activated. Make sure that the file name you want to open is the correct one on the File name section box as shown in Fig. 2.50.



Click on the Open tab.

Fig. 2.50 The activated Open dialogue window in concise form

The 3D Part (engraveprt1) is now imported on the main graphic window.

Constrain the Imported Part

Constrain the imported Part with the ASM datum plane by right clicking on the right mouse on the main GUI window and then click on Default Constraint on the drop-down menu list as illustrated in Fig. 2.51.

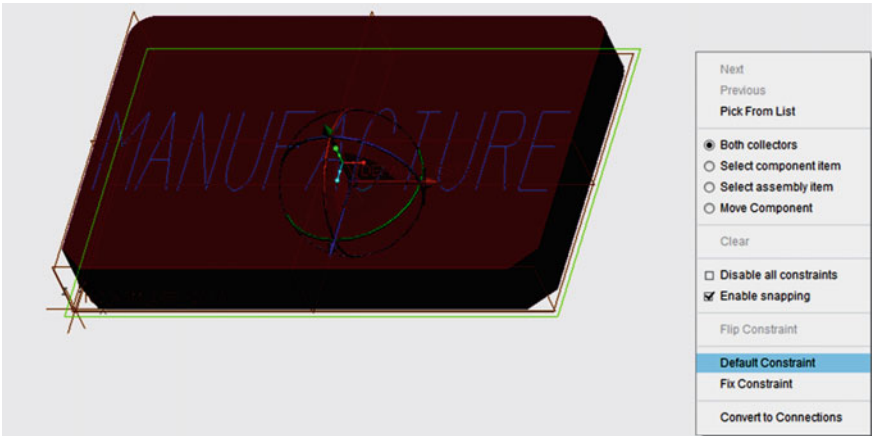


Fig. 2.51 Constraining imported Part method 1

Alternatively, Click on the Automatic section box to activate its drop-down menu list >> Now click on Default on the drop-down menu list as indicated by the arrow in Fig. 2.52.

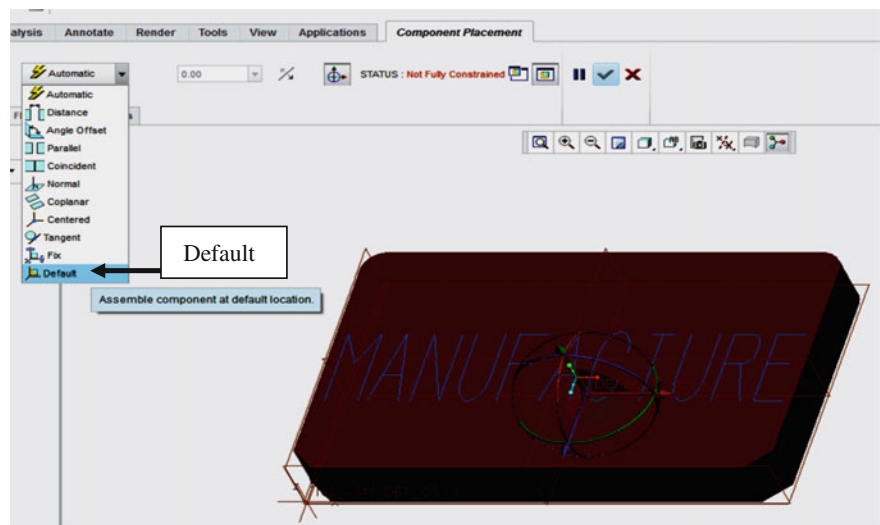


Fig. 2.52 Constraining imported Part method 2

The Part is now fully constrained on the main graphic window as shown in Fig. 2.53. The information on the STATUS section, reads Fully Constrained as indicated by the arrow in Fig. 2.53.

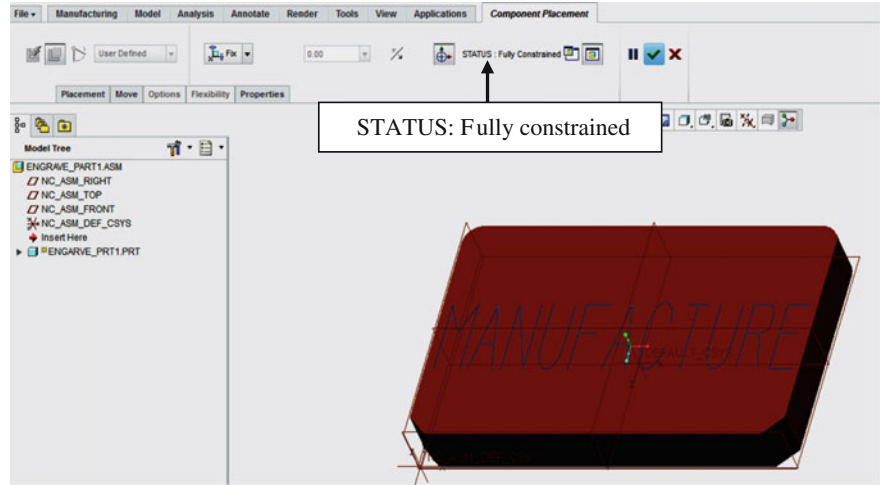


Fig. 2.53 Fully constrained Part on the Manufacturing main GUI window

Click on the Check Mark icon  to exit.

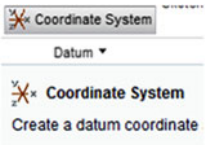
2.4.3 Adding Stock to the Part

No Stock will be created/added to Part in this Engraving process, because Reference Part and Stock (Workpiece) have the same dimensions.

2.5 Add Coordinate System (Programme Zero) to the Part

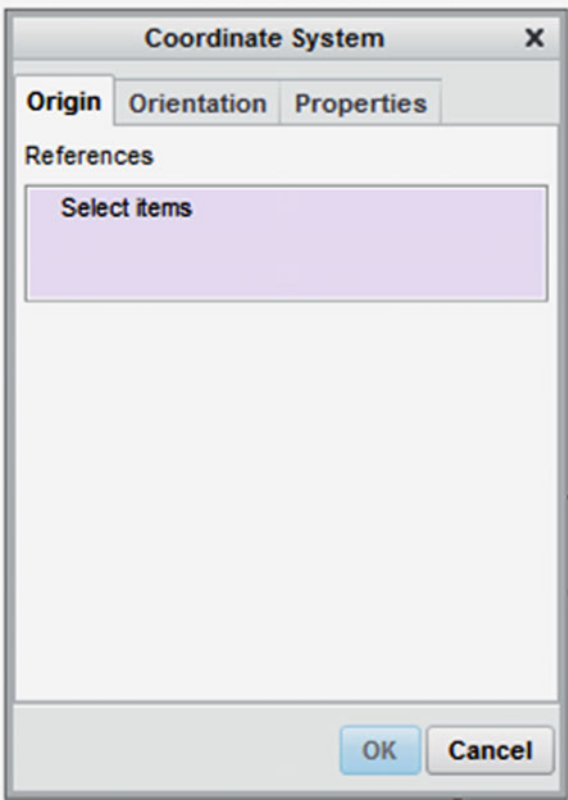
A Coordinate System or Programme Zero will be created on the Workpiece with respect to which all the Machine Coordinates will be referenced/obtained.

Click on the Coordinate System icon tab on the Datum group as shown below.



The Coordinate System dialogue box is activated on the main graphic window as shown in Fig. 2.54.

Fig. 2.54 Activated Coordinate System dialogue box



Now click on the Top surface and two Side surfaces of the Workpiece while holding down the Ctrl key when clicking. The selected Top and Sides surfaces will appear on the References section box on the Coordinate dialogue box as illustrated in Fig. 2.55.

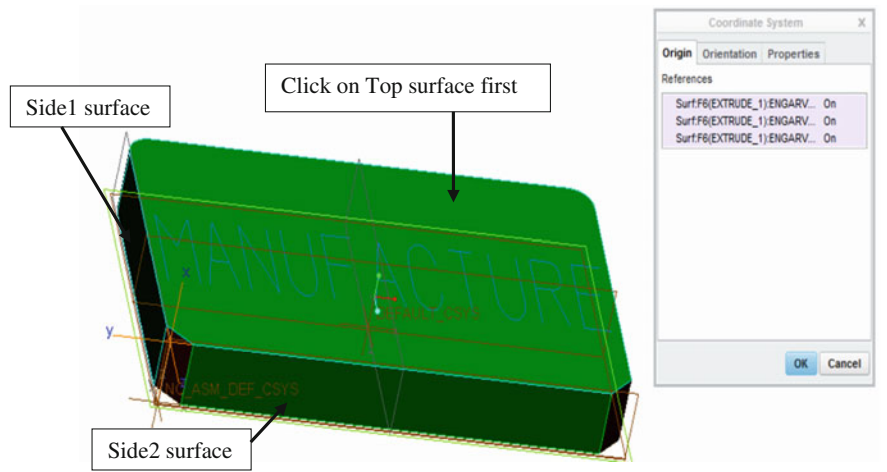


Fig. 2.55 Selected surfaces and the Coordinate Systems dialogue box

Note: The Coordinate System (Programme Zero) can be created anywhere on the Part as long as the correct procedures are followed. Your position of X, Y and Z axis may differ with reference to how you selected those surfaces.

Click on the Orientation tab on the Coordinate System dialogue box >> Notice that the created Programme Zero does not have the correct orientation because the X, Y and Z axis are facing in the wrong direction as indicated by the three arrows in Fig. 2.56.

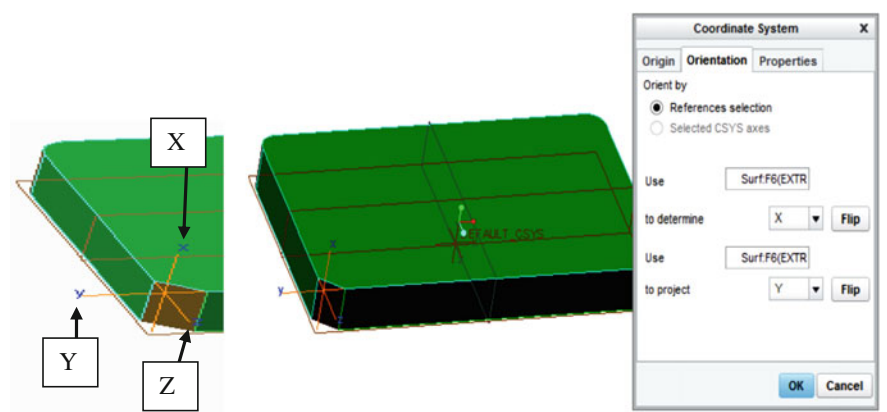


Fig. 2.56 Wrong orientation of X, Y and Z-axes of the Coordinate Systems

Orientate the X, Y and Z coordinate axes to the correct orientation

Click on the Orientation tab >> On the Orient by group, click in the “References selection” radio button >> Click on “to determine” section box and click on Z axis on the drop-down list >> Click on “to project” section box and click on X axis on the drop-down list >> Now click on the Flip tab, to flip X axis orientation. The arrows shown in Fig. 2.57 illustrate the above description, and show the correct orientation for the created Programme Zero.

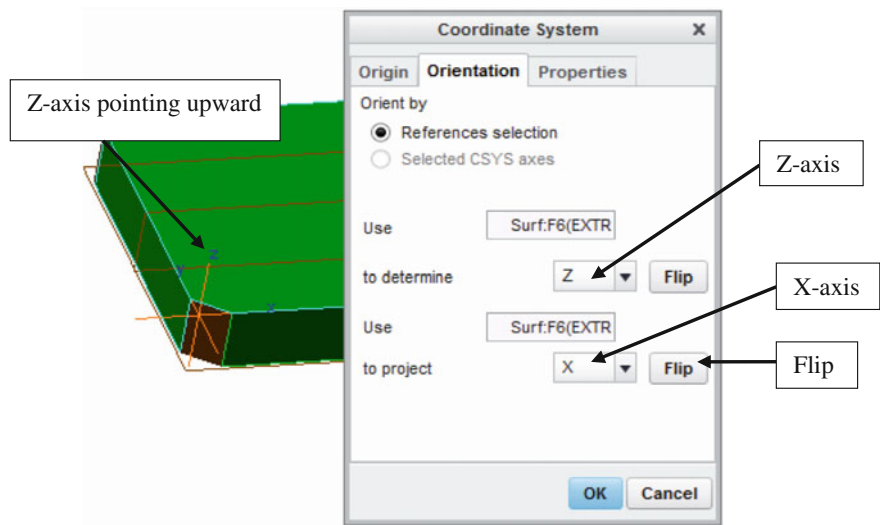
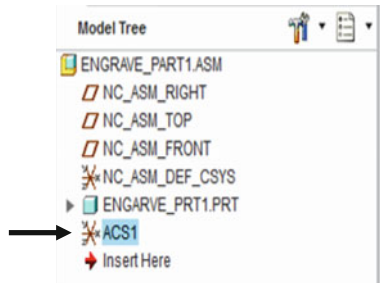


Fig. 2.57 Correct orientations of the Coordinate Systems X, Y and Z-axes

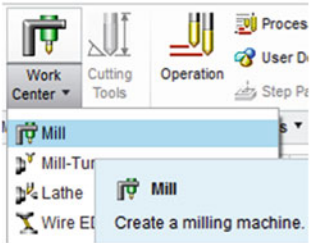
Click on the OK tab to exit the Coordinate System dialogue box.
A new Coordinate System (Programme Zero) called ACS1 is created. The new ACS1 created can be re-named by slow clicking on ACS1 on the Model Tree. Alternatively, right click on ACS1, now click on rename on the drop-down list. In this tutorial, ACS1 will not be re-named as indicated by the arrow on the Model Tree as shown in Fig. 2.58.

Fig. 2.58 Arrow indicating ACS1 on the Model Tree



2.6 Set up the Work Centre

Click on the Work Centre icon on the Manufacturing menu bar >> On the activated drop-down menu list, click on Mill as shown below.



The Milling Work Centre dialogue window is activated as shown in Fig. 2.59. Make note of the Name, Type, Post Processor and Number of Axis section boxes.

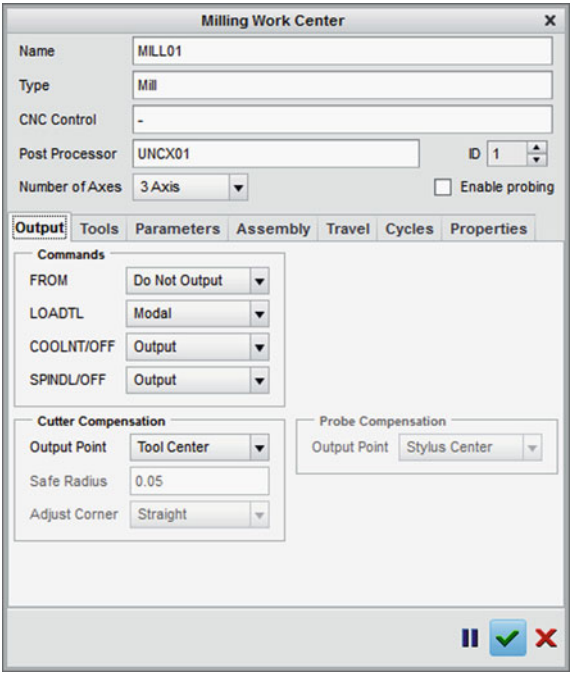


Fig. 2.59 Activated Milling Work Centre dialogue box

Click on the OK tab  to exit the Milling Work Centre dialogue window.

2.7 Set up the Operation

Click on Operation icon as shown below



The Operation dashboard tools are activated as shown in Fig. 2.60.

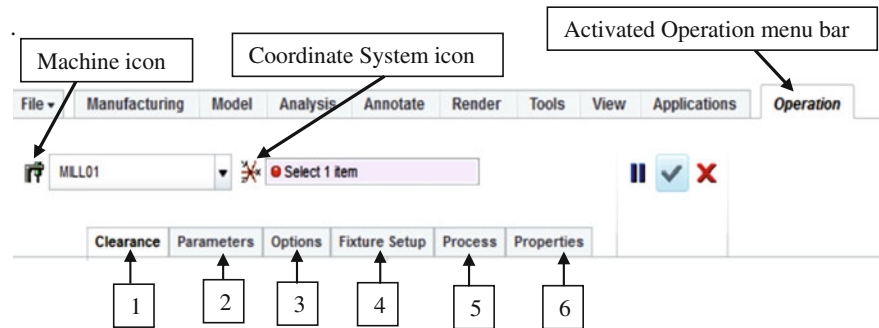


Fig. 2.60 Activated Operation tools in concise form

where

- 1. Represent Clearance tab
- 2. Represent Parameters tab
- 3. Represent Options tab
- 4. Represent Fixture Setup tab
- 5. Represent Process tab
- 6. Represent Properties tab

To add ACS1 (New Coordinate System)

Click in the Coordinate System icon section box >> Go to the Model Tree and click on ACS1. ACS1 is automatically added onto the Coordinate System icon section box.

Make sure that MILL01 is on the Machine icon section box. MILL01 is automatically added to the Machine icon section box by the system.

To activate the Clearance tab

Now click on the Clearance tab (that is 1 in Fig. 2.60) to activate the Clearance panel.

On the Retract group, click on the Type section box to activate its drop-down menu, >> Now click on Plane on the drop-down menu list >> Click on the Part surface and the Reference section box is automatically updated by the system >> Type 10 mm in the Value section box. This is now the Retract distance for the Tool as shown in Fig. 2.61.

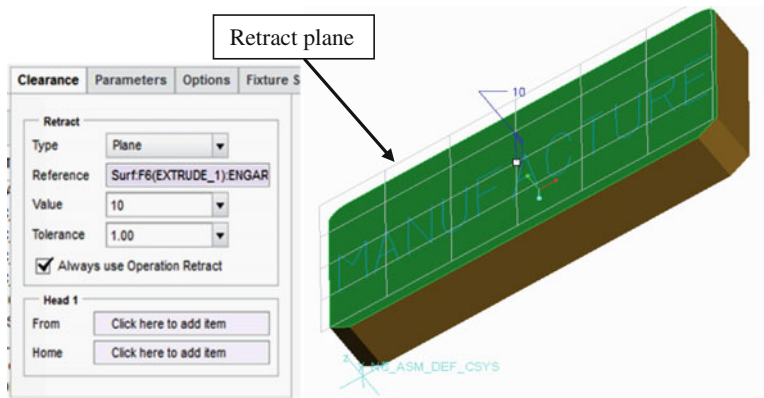


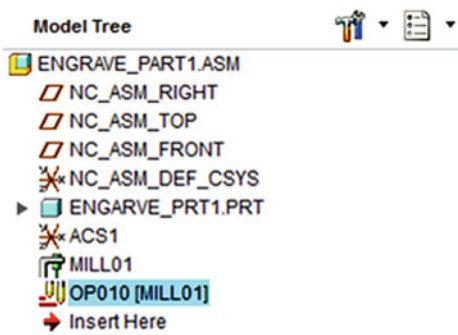
Fig. 2.61 Activated Clearance panel and the Retract plane

Click on the Check Mark icon  to exit the Operation setup.

Note: The Retract surface is the surface that the Tool will move up to after each machining process.

Check the Model Tree to make sure that the ACS1, MILL01 and OP010 [MILL01] as created are on the Model Tree as shown in Fig. 2.62.

Fig. 2.62 Model Tree showing all the steps achieved



2.8 Set up the Cutting Tool

Click on the Cutting Tools icon



The Tools Setup dialogue window is activated on the main graphic window >> Click on the General tab, type the Tool name in the Name box >> Click on the Type section box to activate its contents, now click on the Grooving tool on the menu list >> Click on the Material section box, type the Tool material name. Add dimensions to the Tool and click on the Apply tab. See Fig. 2.63.

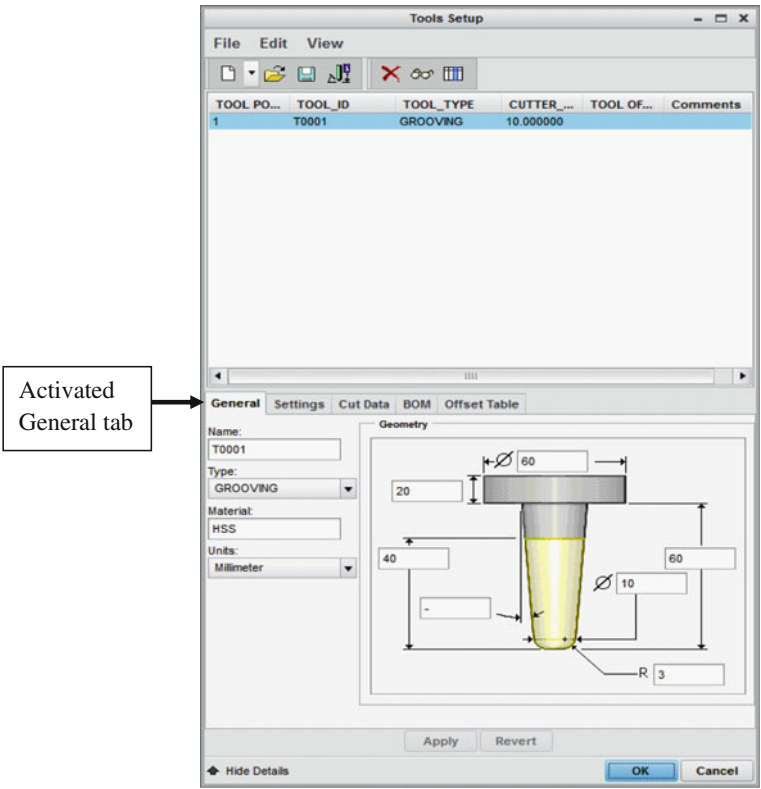


Fig. 2.63 Tools Setup dialogue box indicating the Grooving Tool

Click on OK tab to exit Tools Setup dialogue window.

Note: All dimensions of the cutting Tool are theoretical as shown in Fig. 2.63. They are used in this tutorial for illustration purposes only. Correct cutting Tool material must be selected for each machining applications in practical application.

2.9 Activate the Engraving Process

Click on the Mill menu bar >> Now click on Engraving icon tab on the Milling group as shown below.



The Engraving dashboard tools are activated as shown in Fig. 2.64.

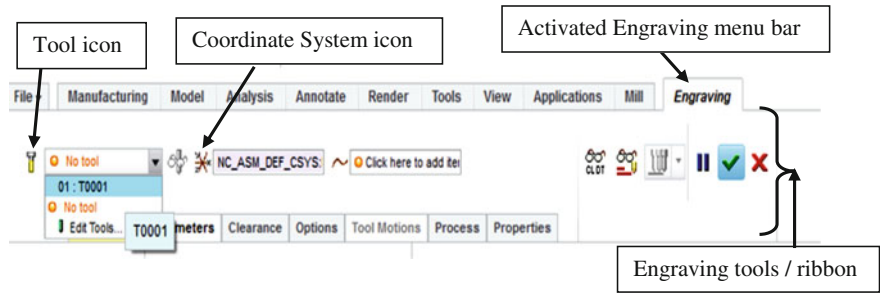


Fig. 2.64 Activated Engraving dashboard

Click on the Tool icon, now click on 01:T0001 on the drop-down list.
Click in the Coordinate System section box, go to the Model Tree and click on the created Coordinate System (ACS1).
Click on the Datum Curve or Curve Profile section box, now click on the “MANUFACTURE” imprinted text on the Part surface as indicated by the arrows in Fig. 2.65.

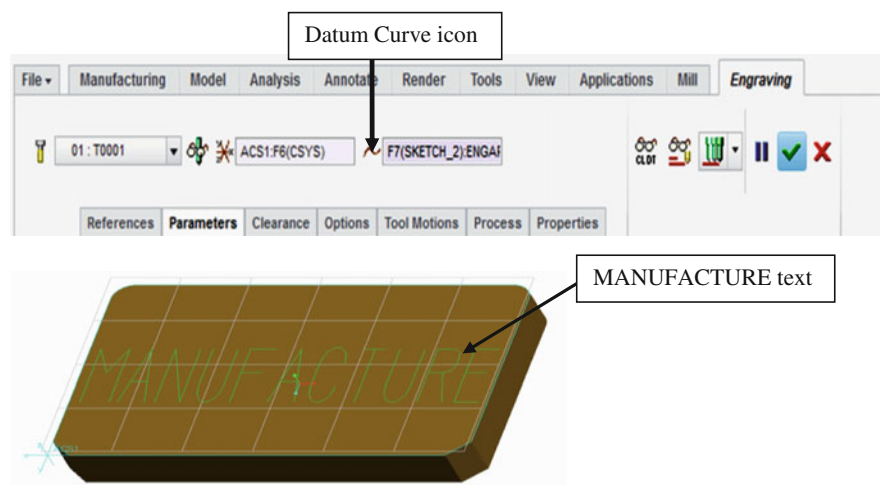


Fig. 2.65 T0001, ACS1, MANUFACTURE text added into their section boxes

Click on the Parameters tab in Fig. 2.65 to activate its panel, now type in the parameter values for the CUT_FEED, STEP_DEPTH, CLEAR_DIST, SPINDLE_SPEED as shown in Fig. 2.66.

Fig. 2.66 Engraving Parameters values

Parameters	Clearance	Options	Tool Motion
CUT_FEED	15		
ARC_FEED	-		
FREE_FEED	-		
RETRACT_FEED	-		
PLUNGE_FEED	-		
STEP_DEPTH	15		
TOLERANCE	0.01		
GROOVE_DEPTH	0		
NUMBER_CUTS	0		
CLEAR_DIST	5		
SPINDLE_SPEED	3500		
COOLANT_OPTION	OFF		

Click on the Check Mark icon  to exit the Engraving application.

Note: The CUT_FEED represent the speed of the Tool on the machined Part.

The STEP_DEPTH represent the amount of material to be removed in each pass of the Tool on the machined Part.

The CLEAR_DISTANCE is the distance when the Tool starts to follow the feed rate command (G01). All the Tool motion before CLEAR_DISTANCE are in rapid mode (G00). The CLEAR_DISTANCE must always be less than the value of the height of the Retract plane.

The SPINDLE_SPEED is the revolution per minute of the spindle/cutting Tool. The spindle speed is the biggest determiner of the Tools life. When Tool is running/turning too fast, it generates excess heat (there are others ways to generate heat too), which softens the Tool and ultimately allows the edge to dull.

2.9.1 Activate Play Path

Click on the Manufacturing menu bar >> Click on the Play Path icon to activate its drop-down list, now click on Play Path as indicated by the arrow in Fig. 2.67.

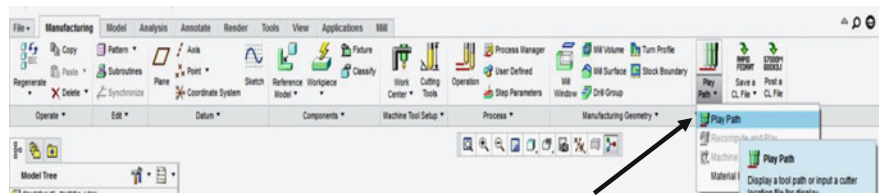
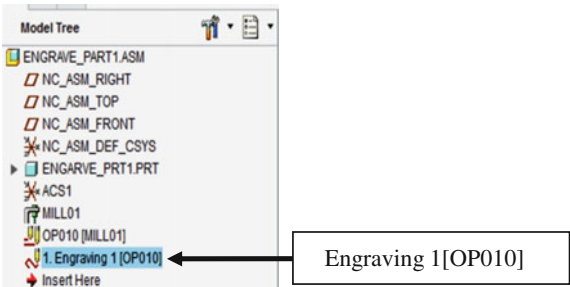


Fig. 2.67 Activated Manufacturing ribbon

Alternatively, the Play Path can be activated by right click the mouse on Engraving 1[OP010] on the Model Tree as shown in Fig. 2.68 >> Now click on Play Path on the activated drop-down menu list.

Fig. 2.68 Activating Engraving 1[OP010] on the Model Tree



The cutting Tool and the Play Path dialogue box are activated on the Manufacturing main graphic window as shown in Fig. 2.69.

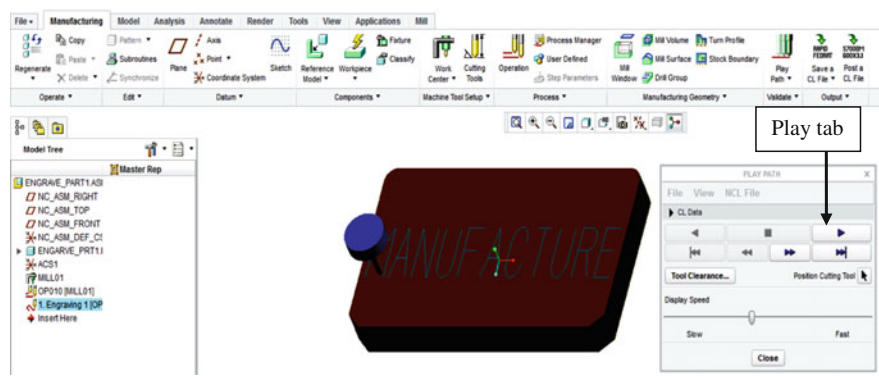


Fig. 2.69 Activated Engraving Tool and the PLAY PATH dialogue box

Click on the Play tab to start the Engraving operation.

Note: The Tool speed can be slowed or increased by adjusting the Display Speed dial/knob on the PLAY PATH dialogue box to enable better view of the Engraving operation.

The end of the Engraving process is as shown in Fig. 2.70.

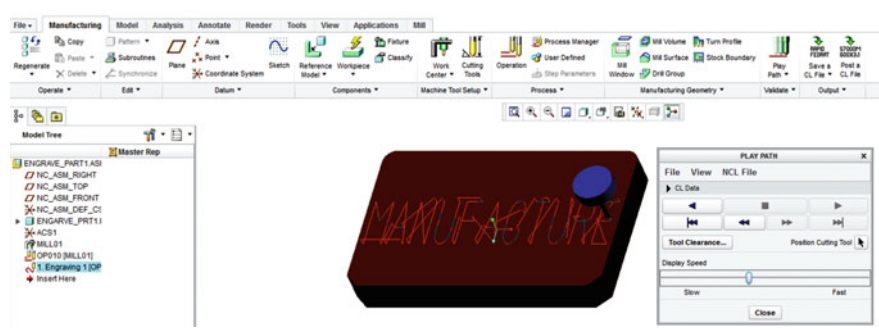


Fig. 2.70 End of on screen Engraving process

Note: When the mouse is right clicked on the MILL01, OP010 [MILL01] or Engraving 1 [OP010] on the Model Tree and Edit Definition is clicked on the drop-down menu list, there parameters can be edited.

2.10 Add Material Removal (nccheck_Type) Application

Click on File to activate its menu list >> Now click on Options on the drop-down list as highlighted in Fig. 2.71.

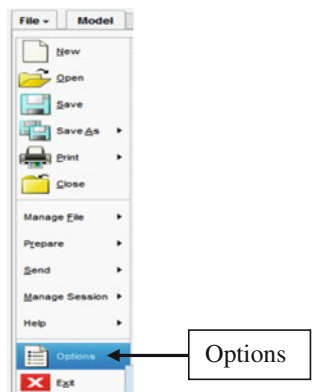


Fig. 2.71 Activated File group

The Creo Parametric Options dialogue window is activated as shown in Fig. 2.72.

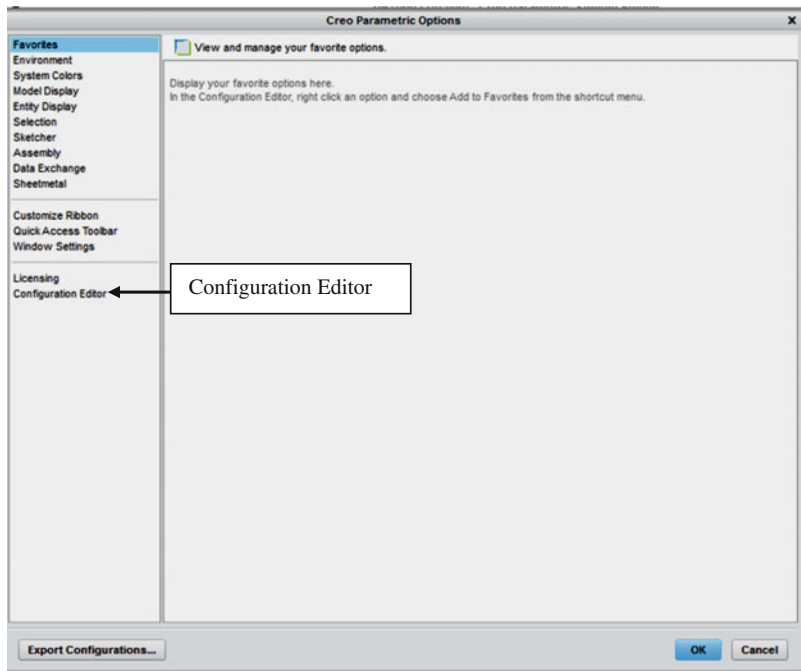


Fig. 2.72 Activated Creo Parametric Options dialogue window

Click on Configuration Editor on the menu list as indicated by the arrow in Fig. 2.72.

The Configuration Editor application tools are activated as shown in Fig. 2.73.

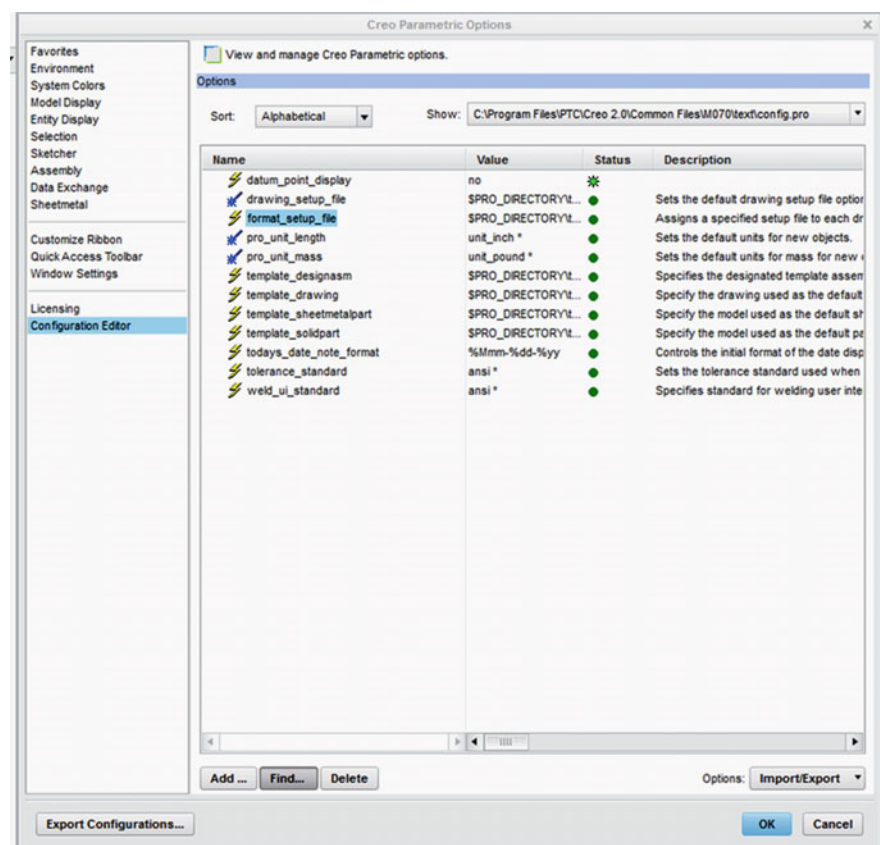


Fig. 2.73 Activated Configuration Editors application tools

Click on Find tab.
The Find Option dialogue window is activated on the main graphic window >> Now type “nccheck_type” in the Type keyword section box >> Click on Find Now tab >> Click on the downward pointing arrow on the Set value section box, now click on nccheck on the drop-down menu list. See Fig. 2.74.

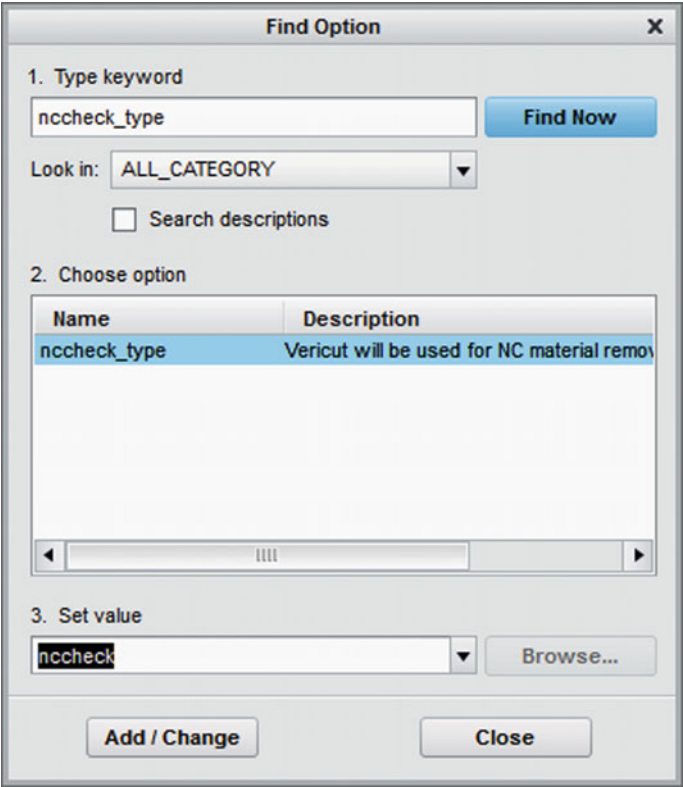


Fig. 2.74 Find Option dialogue window

- Click on Add/Change tab to add the nccheck to the system.
- Save into the configuration file when prompted to do so.
- Click on the Close tab to exit the Find Option dialogue window.

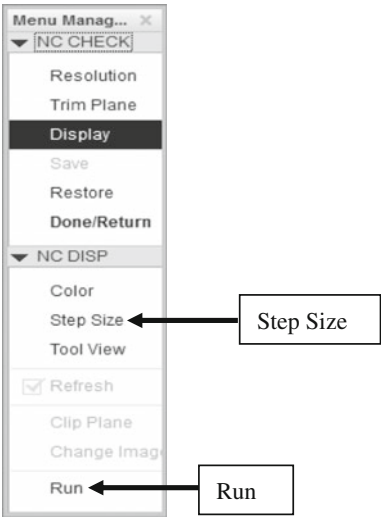
2.11 Material Removal Simulation Activation

To view the material removal simulation after adding the “nccheck_type” to the configuration file.

Got to the Model Tree and click on Engraving 1 [OP010] on the Model Tree to highlight it >> Now right click on the mouse to activate Engraving 1 [OP010] menu list, now click on Material Removal Simulation on the drop-down menu list.

The NC CHECK Menu Manager dialogue box is activated on the main graphic window as shown in Fig. 2.75.

Fig. 2.75 Activated NC CHECK and NC DISP groups on the Menu Manager dialogue box



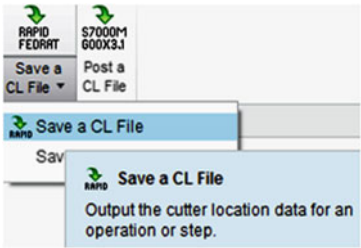
Click on Run to start the Material Removal Simulation.

To reduce the speed of the material removal process >> Click on the Step Size and add a lower step size value >> Click on Run again, and the simulation animation will run very slow.

Click on Done/Return on the NC CHECK group to exit the Material Removal Simulation process.

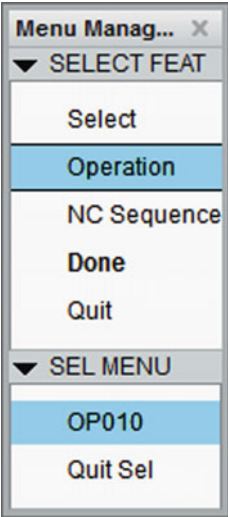
2.12 Creating the Cuter Location (CL) Data

Click on Manufacturing menu tab >> Now click on Save a CL File icon on the output group as illustrated below.



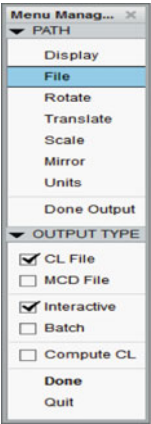
The SELECT FEAT Menu Manager dialogue box is activated >> Click on Operation on the SELECT FEAT group and click on OP010 on the SEL MENU group as highlighted in Fig. 2.76.

Fig. 2.76 Activated
SELECT FEAT and
SEL MENU groups on Menu
Manager dialogue box



Once OP010 is clicked, the PATH Menu Manager dialogue box is activated on the main graphic window >> Click on File on the PATH group and Check Mark the “CL File” and “Interactive” square boxes on the OUTPUT TYPE group if they are not automatically Check Marked by system as shown in Fig. 2.77.

Fig. 2.77 Activated PATH
and OUTPUT TYPE groups
on Menu Manager dialogue
box



Click on Done.

The Save a Copy dialogue window is activated on the main graphic window. See Fig. 2.78.

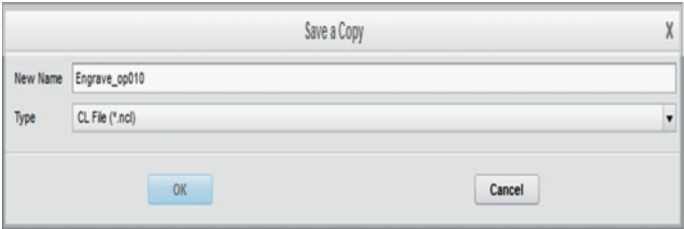


Fig. 2.78 Activated Save a Copy dialogue window in concise form

Make sure that the name on the New Name section box is correct. Alternatively, Type Engrave_op010 as the new name, if you did not type Engrave_part1 in the New dialogue box at the early stage.

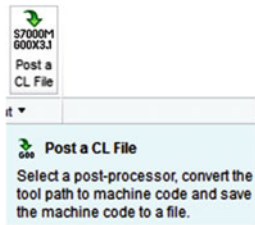
Click on the OK tab to exit.
The CL File(*.ncl) is now saved on the already chosen directory folder/drive.
Click on Done Output on the PATH group as indicated by the arrow shown in Fig. 2.79.

Fig. 2.79 Activated PATH group on the Menu Manager dialogue box



2.13 Create the G-Code Data

Click on Manufacturing toolbar >> Click on the Post a CL File icon on the output group as shown below.



The Open dialogue window is activated on the main graphic window. See Fig. 2.80.

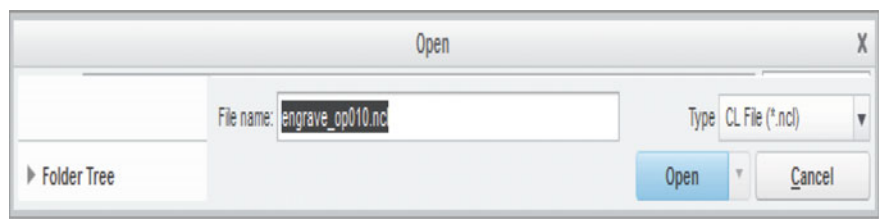
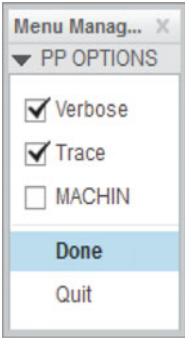


Fig. 2.80 Activated Open dialogue window in concise form

Make sure that the file name on the File Name section box is correct.
Click on the Open tab to exit.
The PP OPTIONS Menu Manager dialogue box is activated on the main graphic window >> Check Mark Verbose and Trace square boxes, respectively, if they are not automatically Checked Marked by the system as shown in Fig. 2.81.

Fig. 2.81 Activated PP
OPTIONS Menu Manager
dialogue box



Click on Done.

The PP LIST Menu Manager dialogue box is activated on the main graphic window with all the available Post Processor (PP) list as shown in Fig. 2.82.

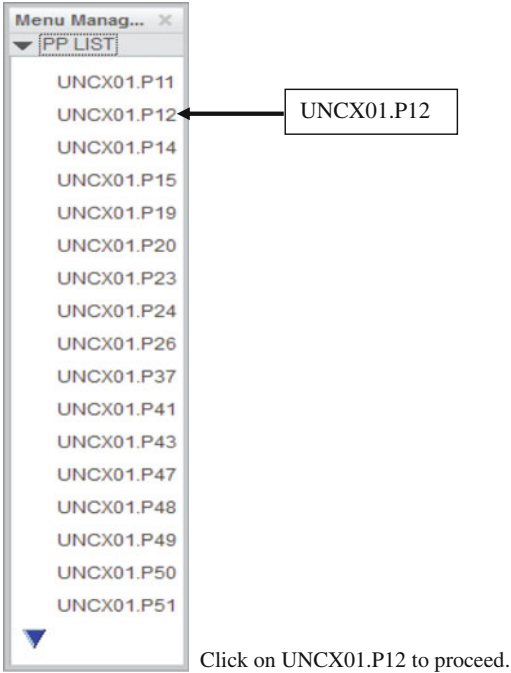


Fig. 2.82 Activated PP LIST Menu Manager dialogue box

Note: The UNCX01.P12 generates G-codes for four axes Milling Machine but can also generate G-codes for three axes Milling Machine.

The INFORMATION WINDOW opens up on the graphic window as shown in concise form. See Fig. 2.83.

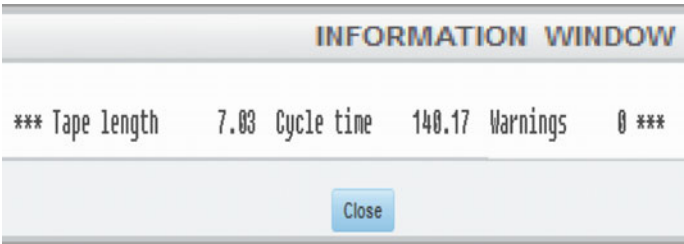


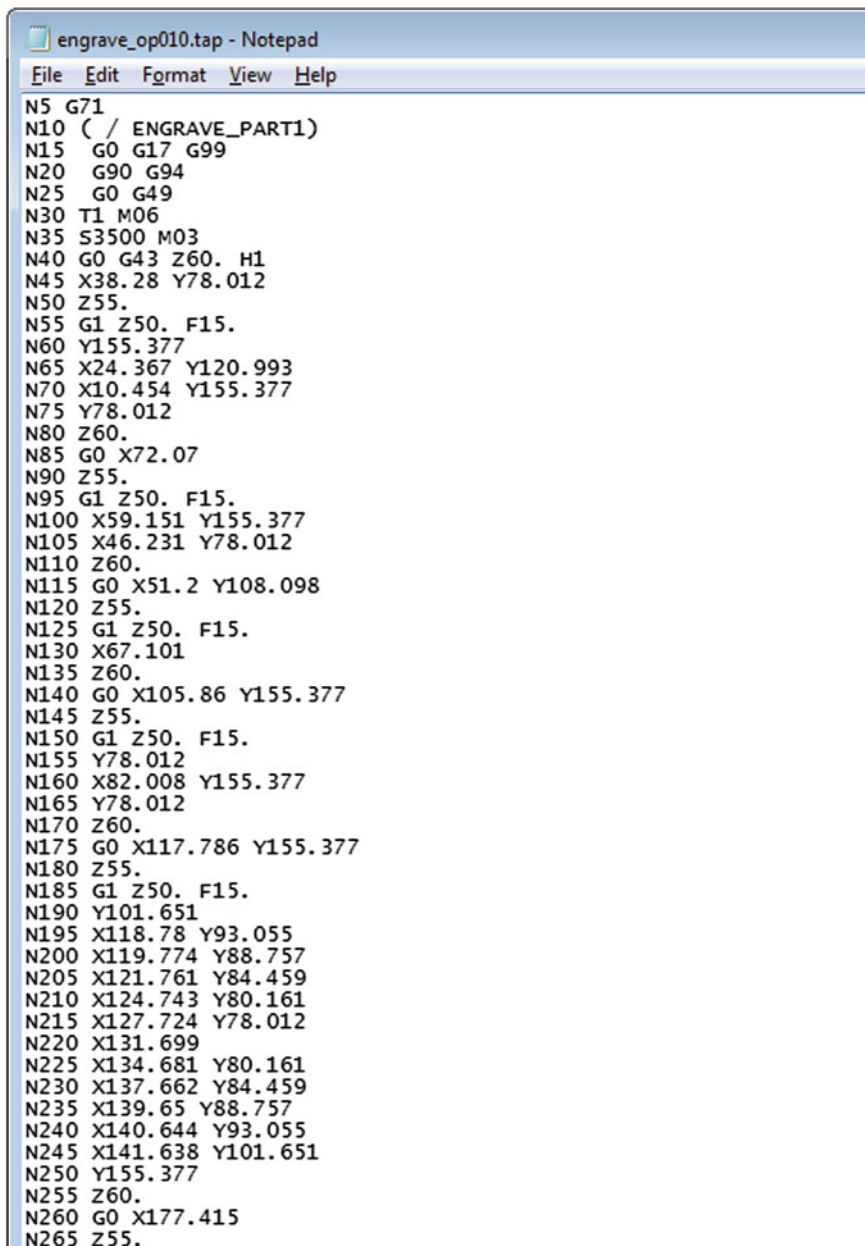
Fig. 2.83 Activated INFORMATION WINDOW in concise form

Check to see if there are any warnings in case the programme has encountered any problem during generating the G-codes.

Click on Close tab on the INFORMATION WINDOW dialogue window to exit.

The generated G-codes are stored as a TAP File on the chosen work directory. The TAP File will be opened using Notepad to view the generated G-codes.

The G-Codes command generated in this process, that Tool will follow during the Engraving process are shown below and on the proceeding pages.



```
engrave_op010.tap - Notepad
File Edit Format View Help
N5 G71
N10 ( / ENGRAVE_PART1)
N15 G0 G17 G99
N20 G90 G94
N25 G0 G49
N30 T1 M06
N35 S3500 M03
N40 G0 G43 Z60. H1
N45 X38.28 Y78.012
N50 Z55.
N55 G1 Z50. F15.
N60 Y155.377
N65 X24.367 Y120.993
N70 X10.454 Y155.377
N75 Y78.012
N80 Z60.
N85 G0 X72.07
N90 Z55.
N95 G1 Z50. F15.
N100 X59.151 Y155.377
N105 X46.231 Y78.012
N110 Z60.
N115 G0 X51.2 Y108.098
N120 Z55.
N125 G1 Z50. F15.
N130 X67.101
N135 Z60.
N140 G0 X105.86 Y155.377
N145 Z55.
N150 G1 Z50. F15.
N155 Y78.012
N160 X82.008 Y155.377
N165 Y78.012
N170 Z60.
N175 G0 X117.786 Y155.377
N180 Z55.
N185 G1 Z50. F15.
N190 Y101.651
N195 X118.78 Y93.055
N200 X119.774 Y88.757
N205 X121.761 Y84.459
N210 X124.743 Y80.161
N215 X127.724 Y78.012
N220 X131.699
N225 X134.681 Y80.161
N230 X137.662 Y84.459
N235 X139.65 Y88.757
N240 X140.644 Y93.055
N245 X141.638 Y101.651
N250 Y155.377
N255 Z60.
N260 G0 X177.415
N265 Z55.
```

```
N270 G1 Z50. F15.  
N275 X153.563  
N280 Y78.012  
N285 Z60.  
N290 G0 Y120.993  
N295 Z55.  
N300 G1 Z50. F15.  
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N310 Z60.  
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N335 X189.341 Y78.012  
N340 Z60.  
N345 G0 X194.31 Y108.098  
N350 Z55.  
N355 G1 Z50. F15.  
N360 X210.211  
N365 Z60.  
N370 G0 X248.97 Y133.887  
N375 Z55.  
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N390 X246.982 Y146.781  
N395 X244.995 Y151.079  
N400 X242.013 Y155.377  
N405 X239.032 Y157.526  
N410 X235.057  
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N425 X227.106 Y146.781  
N430 X226.112 Y142.483  
N435 X225.118 Y133.887  
N440 Y101.651  
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N595 X306.612 Y78.012
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N605 X313.568 Y80.161
N610 X316.55 Y84.459
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N630 Y155.377
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N645 Z55.
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N655 Y155.377
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N665 X352.327 Y153.228
N670 X354.315 Y151.079
N675 X355.309 Y148.93
N680 X356.303 Y142.483
N685 Y133.887
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N695 X354.315 Y125.291
N700 X352.327 Y123.142
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N725 Z55.
N730 G1 Z50. F15.
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