ROHINI COLLEGE OF ENGINEERING AND TECHNOLOGY

Kanyakumari Main Road, Near Anjugramam, Palkulam, Tamil Nadu 629401

Department of Mechanical Engineering

VALUE ADDED COURSE ON MODELING AND MACHINING PRACTICE FOR CNC MACHINES

MODELING AND MACHINING PRACTICE FOR CNC MACHINES – MVA021

UNIT – I

Introduction Modeling - Introduction about CNC Machines - CAD/CAM/CAE, Job opportunity in CAM - Introduction on NC Manufacturing - Expert Machinist CMM - Sheet metal - Cad, Cavity - Mold Cavity - Process Plan - Additive Manufacturing.

UNIT – II

WC Model Creation - Using, Sketching – Constrain – Dimensions - Shapes, Extrude - Revolve - Engineering tools - Hole, Round, Chamfer, Datum Coordinate System.

UNIT – III

Reference Finished Part Model - Work Piece - Automatic Work Piece - Coordinate Creations - Machine Tool Setup - Work Center - Mill, Parameters Setting- Add Tools, Cutting Tool Setup - Mill Operation, CSYS Selection - Clearance Type, Reference Surface - Mill – Face, Cut – Feed, Slep – Depth. Step over, Spindle – Speed

UNIT IV

Display Tool path - Tool Preview, Milling – Play path, Material Removal Simulation Display NC Tool Path - G-Codes used in CNC Programming Colmmon M-Codes - Reading Manufacturing Drawings - Work Steeing and Offsets, work Cordinates Milling Tool types, Face Mill, Slot Mill, Hole Making tools.

UNIT V

Work Center Lathe - Lathe Tool Setting Lathe Coordinate Setting - Clearance Setting, Turning Profile Settings - Turning Tool Path, Turning Material Removal Simulation Turning NC Tool Path

REFERENCE

CNC Programming using fanus Custom Macro B, Sinha S.K.
CNC Machining Hand book, Alan Overby.
CNC setting and Operation Workbook, Tom Renshaw.
CNC Programming Student work book, Mill & Lathe.
Machining Fundamentals, John R. Walker BOB Dixon.

MODELING AND MACHINING PRACTICE FOR CNC MACHINES – MVA021

UNIT – I

Introduction Modeling

CAD Modelling or Computer-aided Design is an important part of the design process. Before expending any physical resources, CAD brings your idea to life in the digital world. Computer Aided Designs are used for a variety of applications ranging from 3d printing prototypes to promotional photo-realistic renderings.

Introduction about CNC Machines

CNC Machining is often the last step in metal manufacturing, or sometimes the only process involved. Compared to other metalworking techniques, CNC machining is capable of meeting the tightest tolerances, and producing the most accurate, precise products over and over again

CAD/CAM/CAE

CAD:

Engineers and draftsmen use computer-aided design software to create illustrations or 3D models. While most engineers should be proficient with CAD, the amount of time they spend using it may depend on their specific role and level of experience.

Best CAD Software for Engineers:

- **AutoCAD**
- **CREO**
- **CATIA**
- **Fusion 360**
- **NX**
- **SolidWorks**

CAE

Computer-aided engineering refers to the use of software to simulate the effects of different conditions on the design of a product or structure using simulated loads and constraints. CAE tools are often used to analyze and optimize the designs created within CAD software.

CAE Software Leaders:

- **Abaqus**
- **Ansys**
- **CFX**
- **Comsol**
- Excel
- Fluent
- HEEDS
- HyperWorks
- LS-DYNA
- Matlab
- Nastran
- Simulink
- STAR-CCM+

The best CAE product choice will depend to some extent on the skills you want to build and your specific career path. For example, niche products like OpenFOAM are open source, which can provide a great deal of flexibility. However, many users feel the software could be improved because it relies solely on a command line and has no graphical user interface (GUI).

Computer-aided Manufacturing (CAM)

Computer-aided Manufacturing (CAM) is commonly defined as the use of software to automate manufacturing processes. CAM software is able to translate

CAD designs into instructions for machines, increasing the efficiency of producing parts and optimizing the amount of materials used.

Best CAM Software

Because CAM is so intertwined with CAD, some software companies produce both types of applications. SolidWorks, for example, offers a full-suite of tools for CAD, CAM and other engineering processes. Similarly, companies like AutoDesk offer combined CAD and CAM tools. Some of the leaders in CAM software include: SolidWorks CAM, Fusion 360 and NX, which combines elements of CAD, CAE, and CAM

CAD vs. CAM vs. CAE: How Familiarity Helps Your Career

Most engineers will benefit from exposure to CAD and CAE software tools, but they should also be familiar with CAM. Knowledge of how CAD and CAM software interact can help to make improvements in the design and analysis phase of planning products and structures.

Engineers whose work is focused on seeing the big picture, such as optimizing systems or managing engineering teams, can especially benefit from building familiarity across all three types of software.

Job opportunity in CAM

There are many careers in Computer Aided Manufacturing or similar fields. These include:

Mechanical Engineer

CNC Operator

CNC Programmer

Design Engineer

Machinist

Industries using CAM include aerospace, automotive, chemical and medical technology, military, or engineering.

CAM jobs can pay around $34,000 to $60,000 a year on average. You will usually require a qualification or technical certification. Some positions may be available at entry-level without experience but some knowledge would be advantageous.

Different jobs will be varied. You could be manufacturing nuts, bolts, and screws. Or you could be creating tools and dies for another manufacturing process. CAM could even lead to work with jewelry and precious stones or metals.

Introduction on NC Manufacturing

Numerical control represents the use of programming, in the form of numbers, letters, and symbols, to guide the operation of a machine. In that sense, NC is a machine control concept more than a manufacturing or machining process.

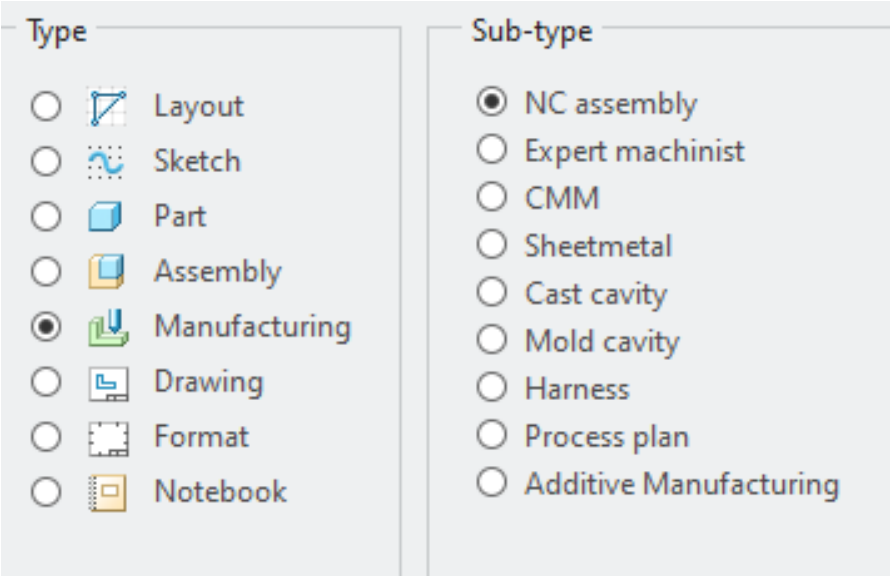
History. The first NC machines were built in the 1940s and 1950s, based on existing tools that were modified with motors that moved the tool or part to follow points fed into the system on punched tape.

Expert Machinist CMM

Coordinate Measuring Machine (CMM) Operator operates CMM equipment to perform measurement, inspection, and testing of production materials and components using previously prepared inspection programs. Mounts or installs tools, attachments, fixtures, or work pieces on CMM.

A coordinate measuring machine (CMM) is a device that measures the geometry of physical objects by sensing discrete points on the surface of the

object with a probe. There are different types of CMMs and probes that manufacturers can use to ensure and maintain quality.



Sheet Metal Manufacturing

Sheet metal fabrication is the process of turning flat sheets of steel or aluminium into metal structures or products, by cutting, punching, folding and assembling. Sheet metal can be cut, bent or stretched into nearly any shape, which is generally done by cutting and burning the metal.

Casting Manufacturing

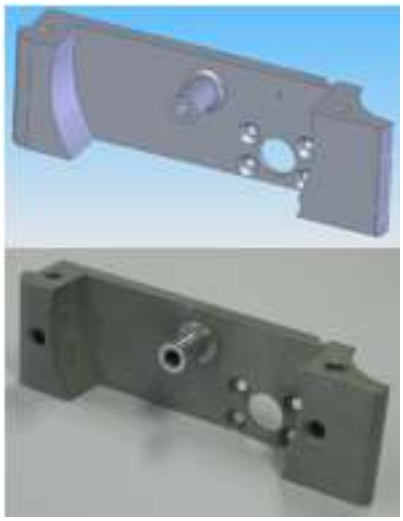
Casting is a manufacturing process used by the forging industry to produce an object (often metal) of a specific desired shape. This is achieved by pouring hot molten metal into a mold that contains a hollow cavity of the exact required shape.

Additive Manufacturing.

Additive manufacturing is the process of creating an object by building it one layer at a time. It is the opposite of subtractive manufacturing, in which an object is created by cutting away at a solid block of material until the final product is complete

UNIT – II

NUMERICAL MODEL CREATION:



CAD model and CNC machined part

Sketching – Constrain – Dimensions

Creo Sketch is a free 2D CAD application that offers the easiest way for anyone to quickly sketch out product design ideas and share them electronically with customers, suppliers, and co workers. It lets you add color and other visual enhancements to transform quick sketches into finished artwork.

Constraints are rules enforced by Creo Parametric on your sketched entities.

- Constraint types include:
  - Vertical
  - Horizontal
  - Perpendicular
  - Tangent
  - Mid-point
  - Coincident
  - Symmetric
  - Equal
  - Parallel

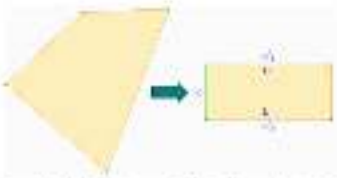


Figure 1 Sketch Before and After Constraints Applied

Figure 2 – Constrain Group

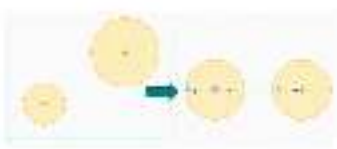


Figure 3 – Sketch Before and After Constraints Applied

Sketched entities are the basis for a solid face or surface of a 3-D model.

- There are two types of lines:
  - Line Chain
  - Tangent Line

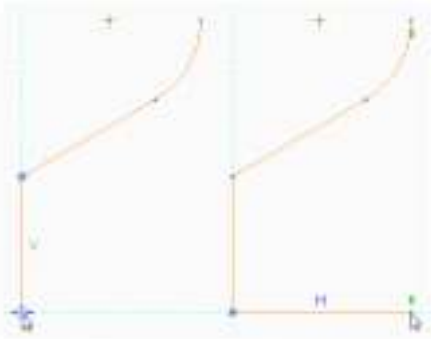


Figure 1 – Line Chain

Figure 2 – Tangent Line

You can quickly sketch four-sided shapes.

- The four lines are independent.
- You can delete, trim, and align each line individually.
- You can create symmetric rectangles using Center Rectangles.

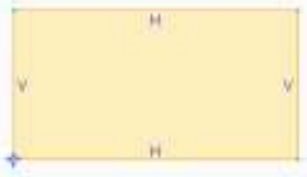


Figure 1 – Corner Rectangle

Figure 3 – Center Rectangle

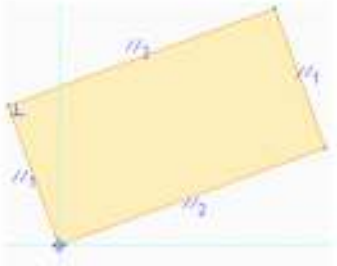


Figure 2 – Slanted Rectangle

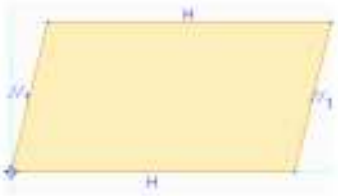


Figure 4 – Parallelogram

You can quickly sketch various types of circles.

- There are four types of Circles:
  - Center and Point
  - Concentric
  - 3 Point
  - 3 Tangent

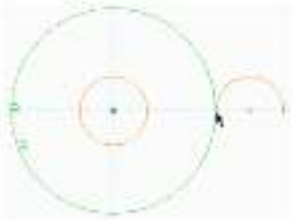


Figure 1 – Concentric Circle

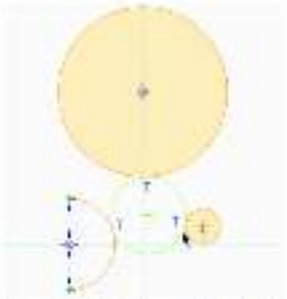


Figure 2 – Circle Tangent to 3 Entities

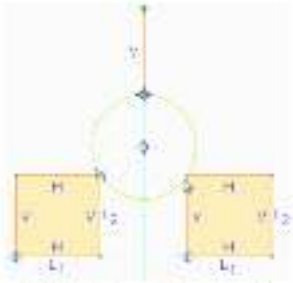


Figure 3 – Circle Created by Picking 3 Points

You can create numerous types of arcs within Sketcher.

- There are five types of Arcs:
  - 3-Point
  - Tangent End
  - Concentric
  - Center and Ends
  - 3 Tangent

Figure 1 – 3-Point Versus Tangent Arc Creation

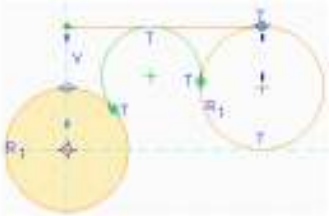


Figure 2 – Arc Tangent to 3 Entities

Figure 3 – Center and Endpoints Arc

You can extrude a sketch to many different depth options.

- Extrude depth options:
  - Blind
  - Symmetric
  - To Next
  - Through All
  - Through Until
  - To Selected
  - Side 1/Side 2
- Set using dashboard or right-clicking drag handle

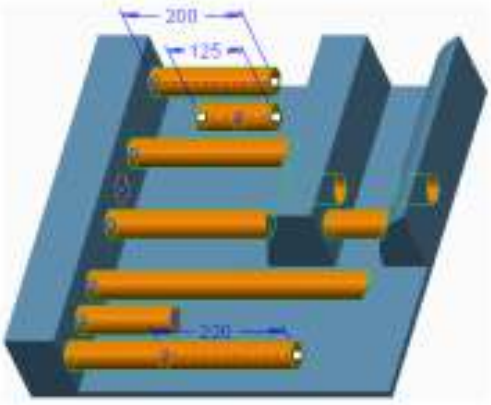


Figure 1 - Extrude Depth Options

You can create extruded features from 2-D sketches.

- Extrude sections perpendicular to the sketching plane.
- Add or remove material from the model.

Figure 1 - Viewing 2-D Sketches

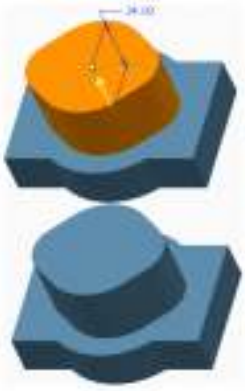


Figure 2 - Extrude Feature Adding Material

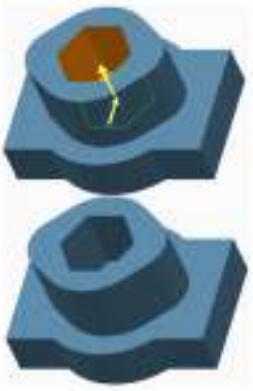


Figure 3 - Extrude Feature Removing Material

You can create revolved features from 2-D sketches.

- Revolve a section about the axis of revolution in a sketching plane.
- Add or remove material from the model.
- Select different axes of revolution.
  - First geometry centerline
  - Axis or edge

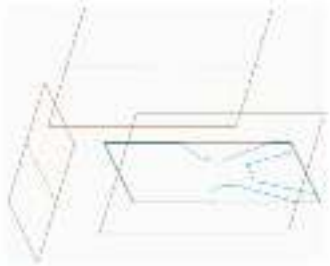


Figure 1 – Viewing 2-D Sketches

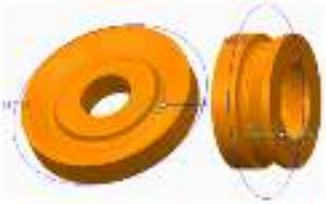


Figure 2 – Same Revolved Sketch using Different Axes of Revolution

Figure 3 – Removing Material using a Revolve Feature

You can revolve a sketch to many different angle depths.

Revolve angle options:

- Variable
- Symmetric
- To Selected
- Side 1/Side 2

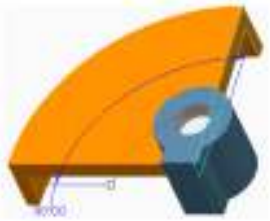


Figure 1 - Variable Revolve Angle Depth

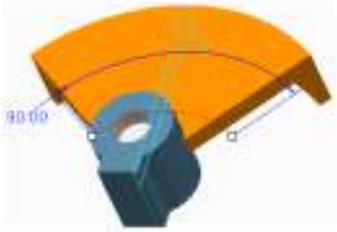


Figure 2 - Symmetric Revolve Angle Depth

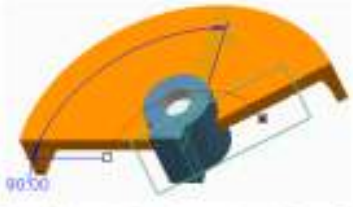


Figure 3 - Side 1 Revolve Angle To Selected, Side 2 Revolve Angle Variable

Rounds add or remove material by creating smooth transitions between existing geometry.

- Rounds can add or remove material.
- You can select edges or surfaces.

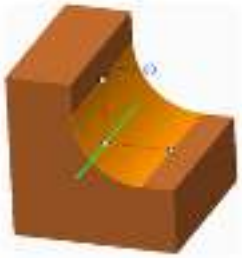


Figure 1 – Round Preview: Adding Material

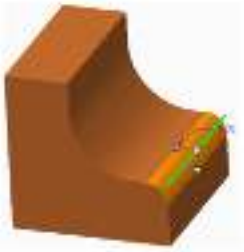


Figure 2 – Round Preview: Removing Material

Figure 3 – Completed Rounds

A coaxial hole is placed at the intersection of an axis and a surface.

- Placement references:
  - Datum axis
  - Surface or datum plane
- Offset references:
  - None

Figure 1 – Selecting Placement References

Figure 2 – Coaxial Holes

A linear hole is created by selecting one placement reference and two offset references.

- Placement references:
  - Datum plane or surface
- Offset references:
  - Datum plane or surface
  - Edge
  - Datum axis

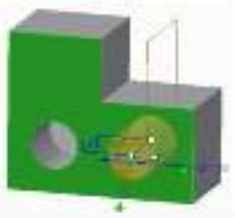


Figure 1 – Selecting Placement Reference

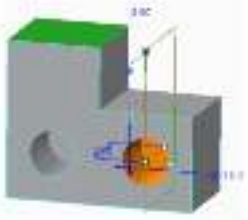


Figure 2 – Selecting Offset References

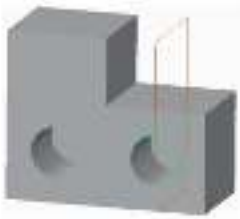


Figure 3 – Completed Hole

Rounds add or remove material by creating smooth transitions between existing geometry.

- Rounds can add or remove material.
- You can select edges or surfaces.

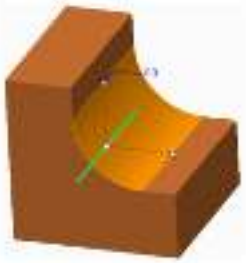


Figure 1 – Round Preview: Adding Material

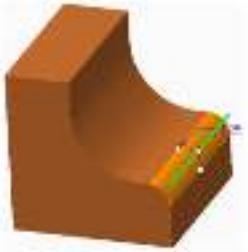


Figure 2 – Round Preview: Removing Material

Figure 3 – Completed Rounds

The rounds created by selecting edges are constructed tangent to the surfaces adjacent to the selected edges.

- You can select one or more edge.
- Rounds propagate around tangent edges.

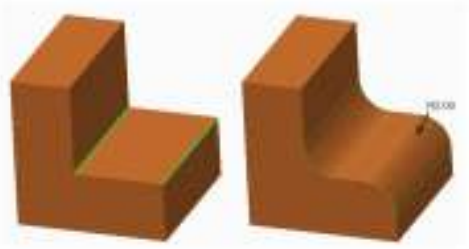


Figure 1 – Rounds Created by Selecting Two Edges

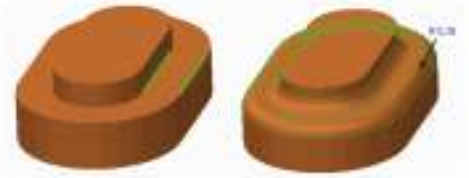


Figure 2 – Rounds Created by Selecting Two Edges

Rounds created by selecting a surface and edge are constructed tangent to the surface and pass through the edge.

- You can select a surface and an edge.
- Rounds propagate around tangent edges.

Figure 1 – Round Created by Selecting a Surface and Edge

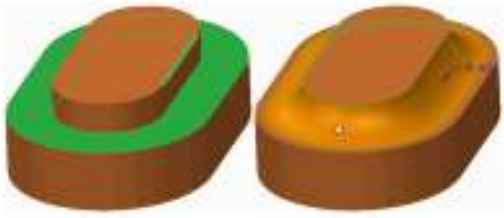


Figure 2 – Round Created by Selecting a Surface and Edge

Full rounds replace a surface with a round that is tangent to the surface it replaces.

- You can select two edges.
- You can select three surfaces.

Figure 1 – Full Round Created by Selecting Two Edges

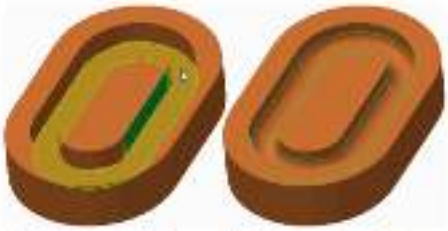


Figure 2 – Full Round Created by Selecting Three Surfaces

Rounds created by selecting two surfaces can span gaps or engulf existing geometry.

Chamfers add or remove material by creating a beveled surface between adjacent surfaces and edges.

- You can select one or more edge.
- Chamfers can add or remove material.
- Chamfers propagate around tangent edges.

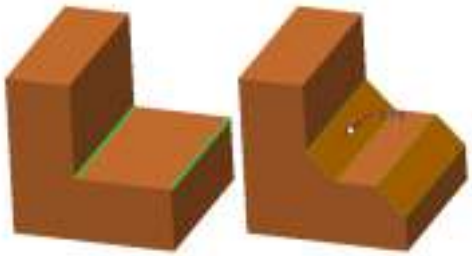


Figure 1 – Chamfers Created by Selecting Two Edges

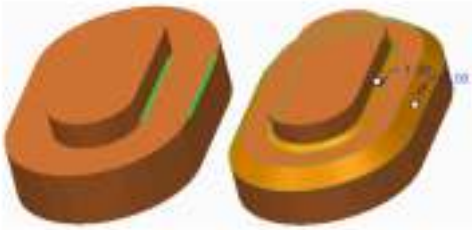


Figure 2 – Chamfers Created by Selecting Two Edges

There are several different ways to dimension a chamfer to capture desired design intent.

- Dimensioning schemes include:
  - D x D
  - D1 x D2
  - Angle x D
  - 45 x D

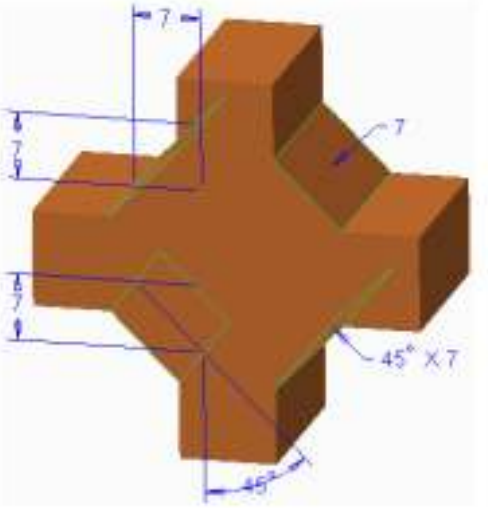


Figure 1 – Four Different Chamfer Dimensioning Schemes with the Same Geometry

Datum features are commonly required as references when creating other features.

- The following types of datum features can be created:
  - Datum Planes
  - Datum Axes
  - Datum Points
  - Datum Coordinate Systems

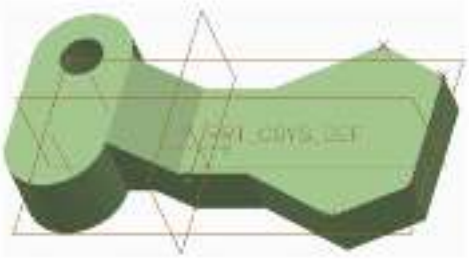


Figure 1 — Examples of Datum Features

Datum axes are particularly useful for making datum planes, placing items coaxially, and creating axis patterns.

- **Definition:**
  - No mass, infinite linear reference
  - Display length can be changed
- **Uses:**
  - Construction geometry
  - Reference
- **Types:**
  - Auto axis
  - Axis feature
  - Geometry point
  - Geometry centerline

Figure 1 — Various Datum Axis Types

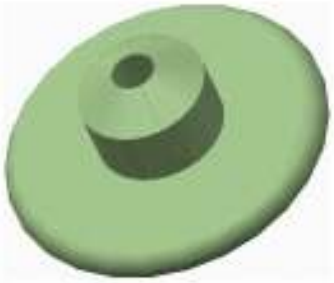


Figure 2 — Geometry Axis (Centerline) Created for Revolve Feature

Datum planes are 2-D reference geometry that you use to build feature geometry.

- **Definition:**
  - No mass, infinite planar reference
  - Display size can be changed
  - Two sides
- **Uses:**
  - Default datum planes
  - Construction geometry
  - Reference
- **Types:**
  - Through
  - Normal
  - Parallel
  - Offset
  - Angle
  - Tangent
  - Blend section

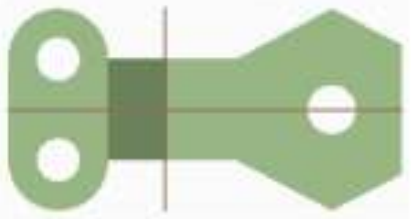


Figure 1 — Viewing Datum Plane Sides

Figure 2 — Datum Plane Types

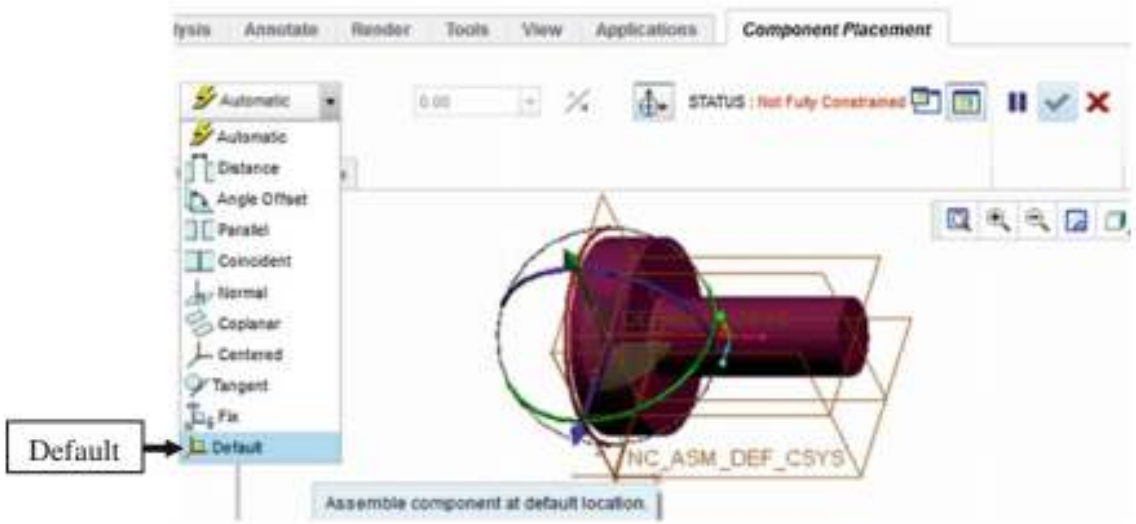
UNIT V

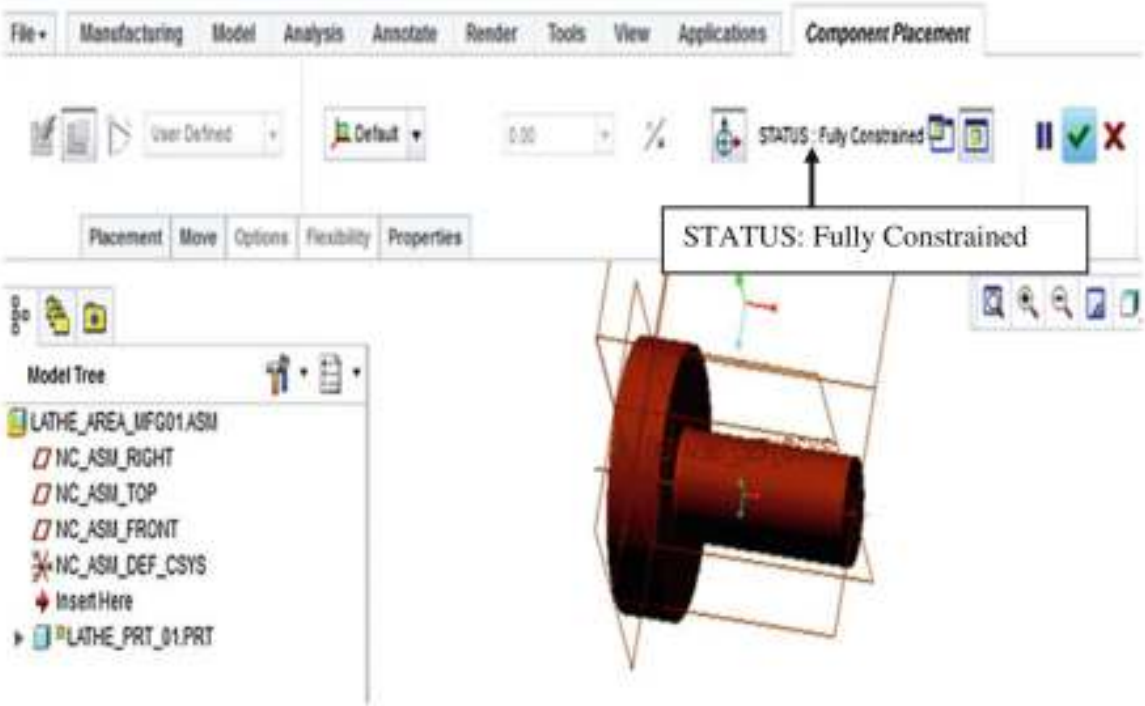
Work Center Lathe - Lathe Tool Setting Lathe Coordinate Setting - ClearanceSetting, Turning Profile Settings

Area Turning is used to remove excess material from stock.
Area Turning is a two-axis machining process.
The following steps will be covered in this tutorial.

- Activate the Manufacturing application
- Import the Part into the Manufacturing environment
- Constrain the Part into the Manufacturing environment
- Create Automatic Workpiece (Stock)
- Create Programme Zero (Machine Coordinate System)
- Create Work Centre
- Create Operation
- Create Cutting Tool
- Create Area Turning using Profile Method
- Generate the Cutter Location (CL) Data File
- Generate the G-code Data

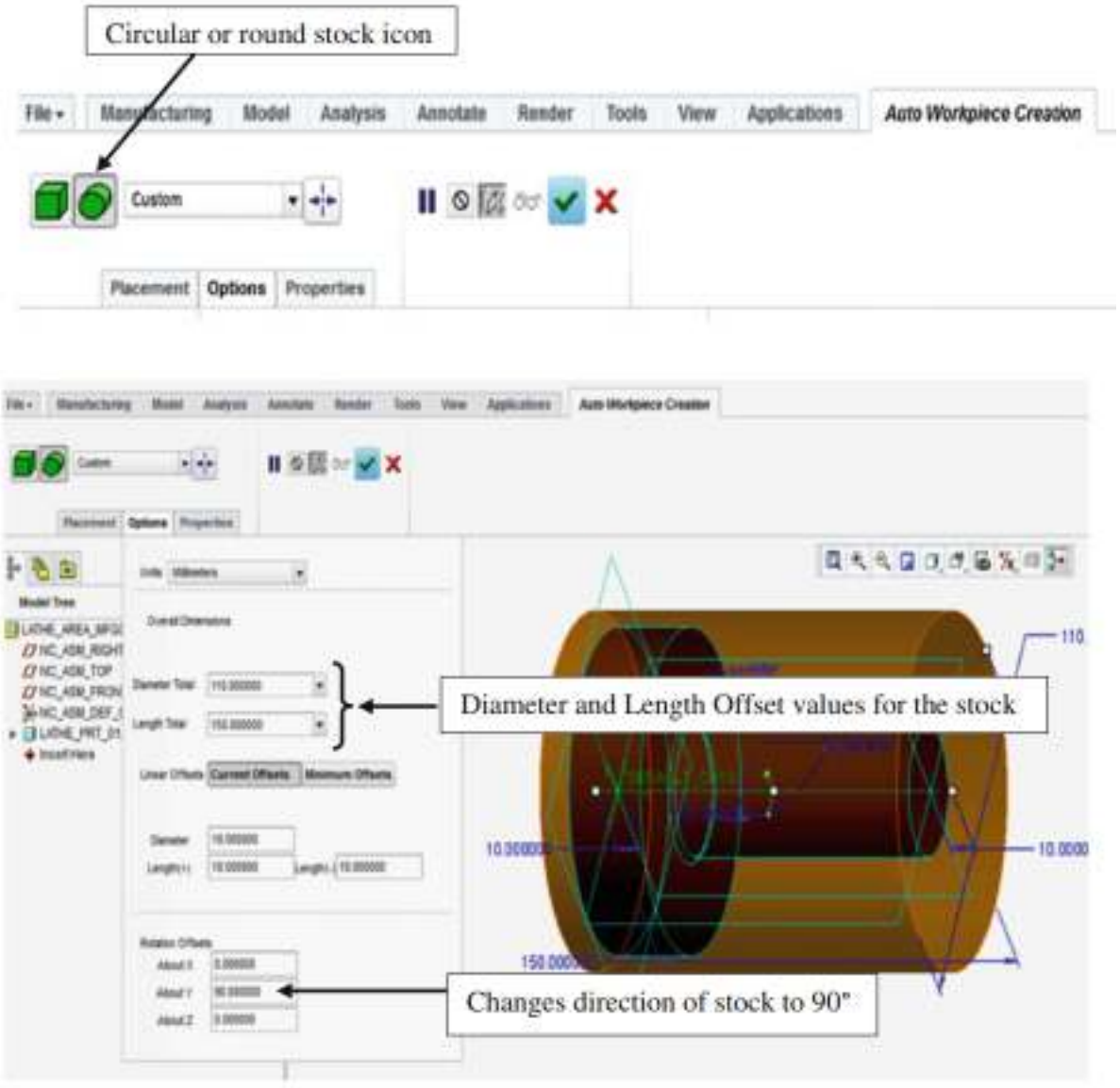
Once the Component Placement tools are activated » Click on the Automatic downward pointing arrow and on the drop-down list, and click on Default as indicated by the arrow in Fig. 10.5.

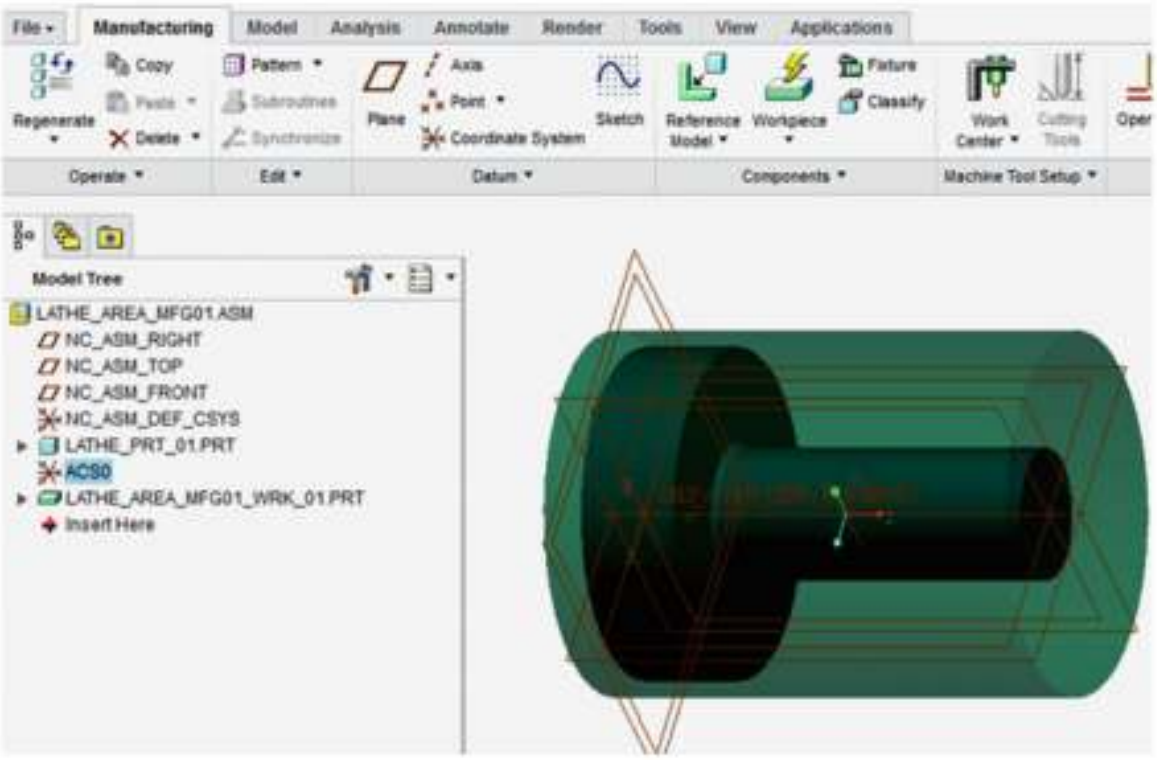




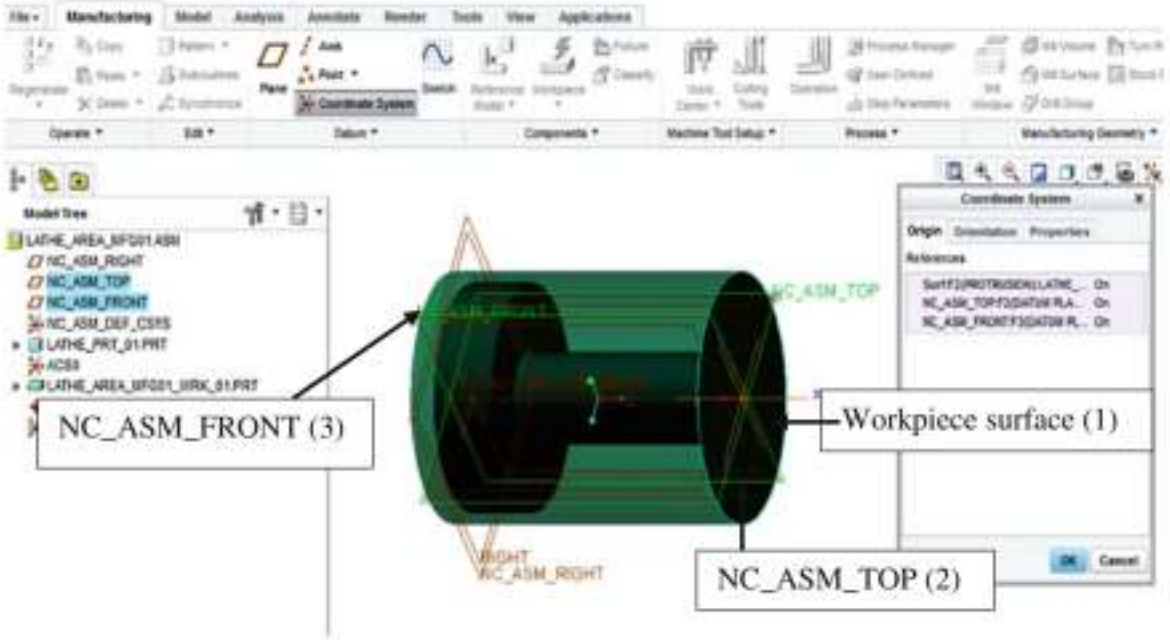
Click on Workpiece icon ≫ Now select Automatic Workpiece on the drop-down menu list as shown below.

The Auto Workpiece Creation dashboard tools are activated. See Fig. 10.7.

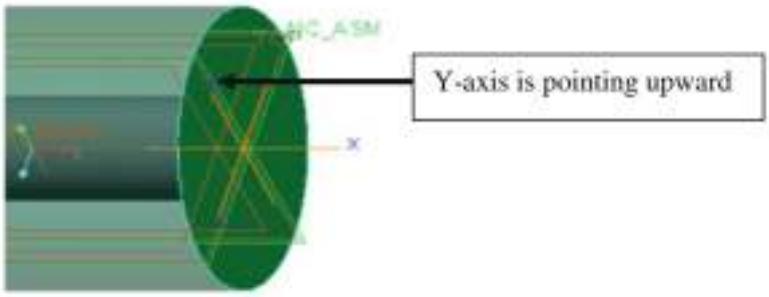




The Coordinate System dialogue box is activated on the main graphic window » Now click on the Origin tab on the Coordinate System dialogue box » Click on the Workpiece surface, NC_ASM_TOP and NC_ASM_FRONT while holding down the Ctrl key as indicated by the arrows in Fig. 10.10.

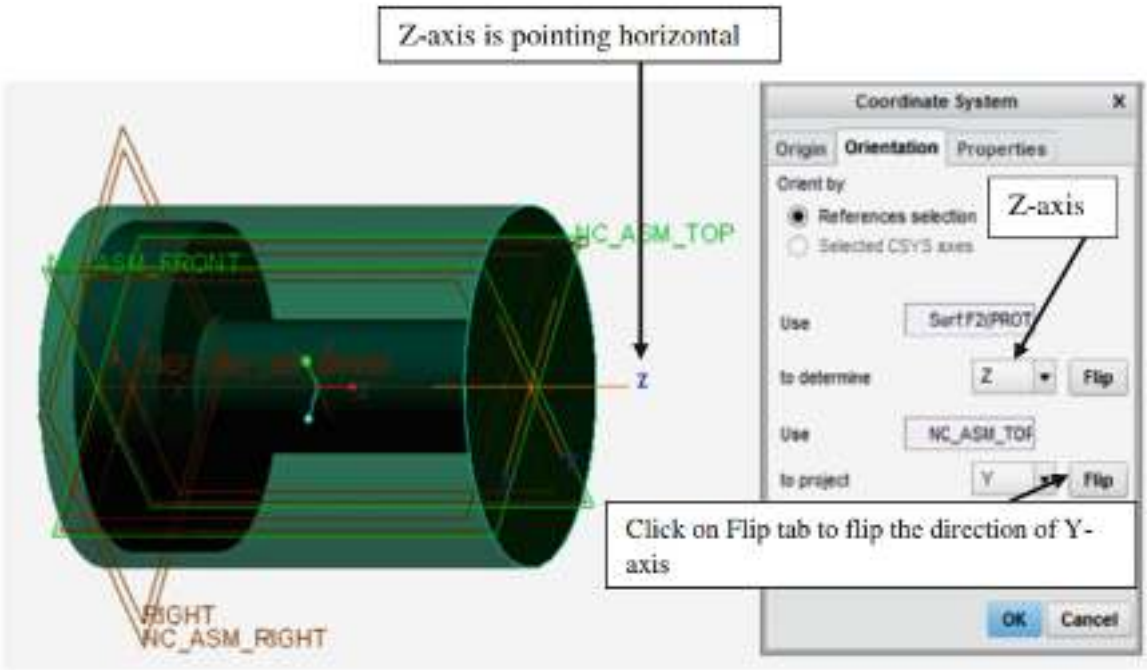


The Orientation of the created Coordinate System is not correct as shown in Fig. 10.11.



To orient the X, Y and Z coordinate axes to the correct orientation

Click on the Orientation tab on the Coordinate System dialogue box to activate its content ≫ On the "Orient by" group, click on the radio button of "References selection" ≫ Click on the "to determine" section box to activate its drop-down menu list, now click on Z axis on the activated drop-down list ≫ Click on the "to project" section box, and now click on Y axis on the activated drop-down list. Now click on the Flip tab to flip Y axis orientation as illustrated in Fig. 10.12.



The Lathe Work Centre dialogue window is activated on the main graphic window as shown in Fig. 10.13.

Fig. 10.13 Activated Lathe Work Centre dialogue window

Click on the Check Mark icon.

Turning Tool Path, Turning Material RemovalSimulation Turning NC Tool Path

Click on the Operation icon as shown below.

The Operation application tools are activated ≫ LATHE01 and ACS1 are automatically generated and added by the system as indicated by the arrows in Fig. 10.14.

Note: If LATHE01 and ACS1 are not automatically generated by the system in their respective section boxes as indicated by the *arrows*, manually select and input them yourself.

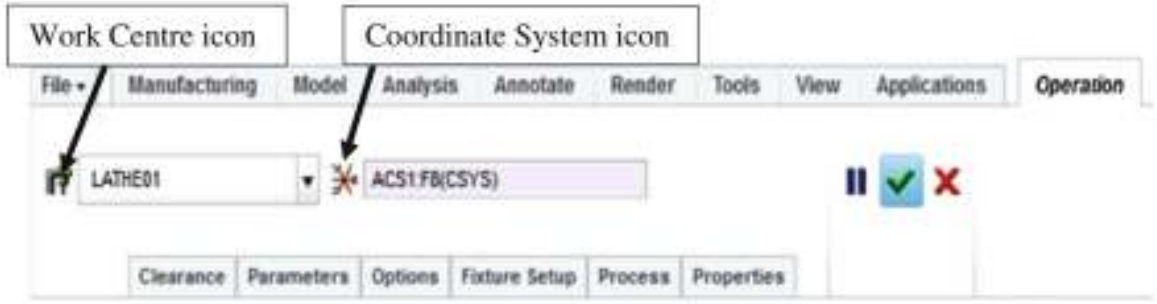


Fig. 10.14 Activated Operation dashboard

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

Fig. 10.15 Activated Tools Setup dialogue window

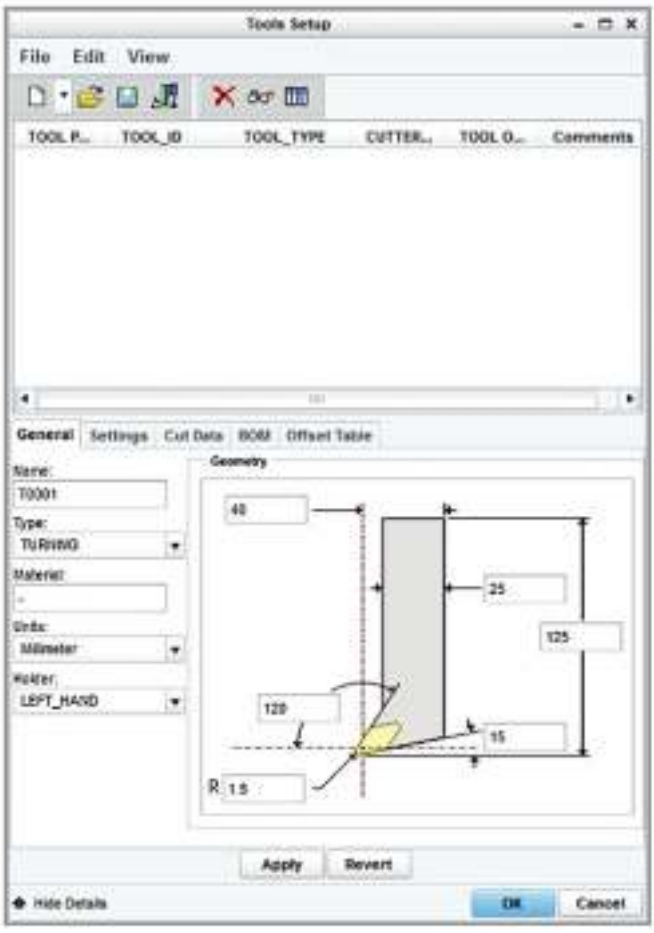
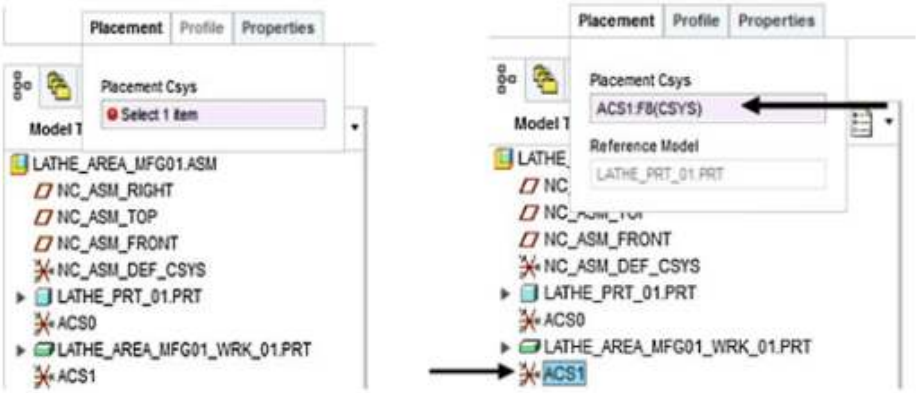


Fig. 10.17 Activated Turn Profile dashboard

Click on the Placement tab to activate its content ≫ Click on the Placement Csys section box, now go to the Model Tree and click on the ACS1 as the Coordinate System as indicated by the arrows in Fig. 10.18.

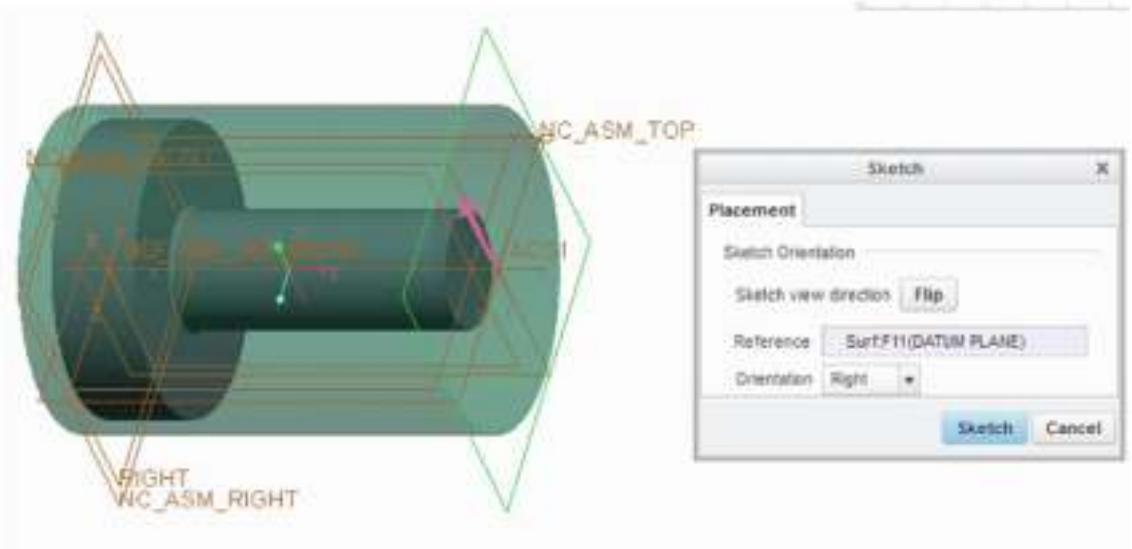


Click on the Use sketch to define turn profile icon in Fig. 10.17 as indicated by the arrow in Fig. 10.19.

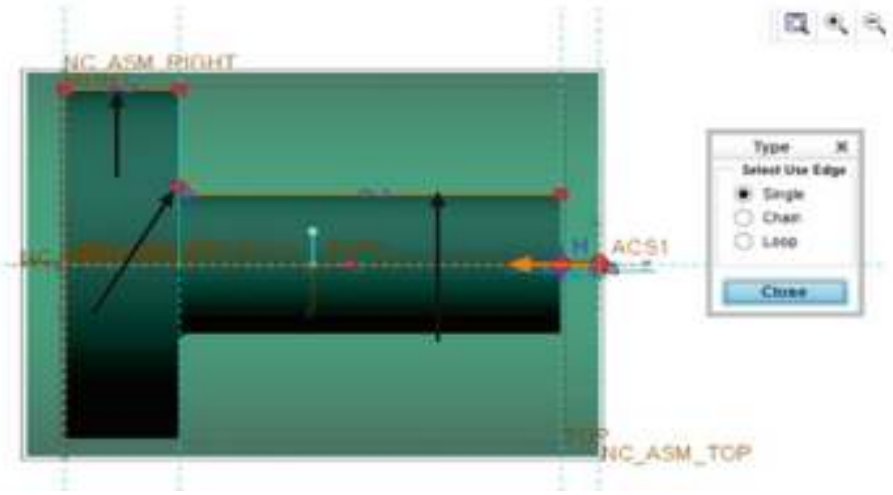
Fig. 10.19 Activating Turn Profile sketch

Click on the Define an Internal Sketch icon as indicated by the arrow in Fig. 10.20.

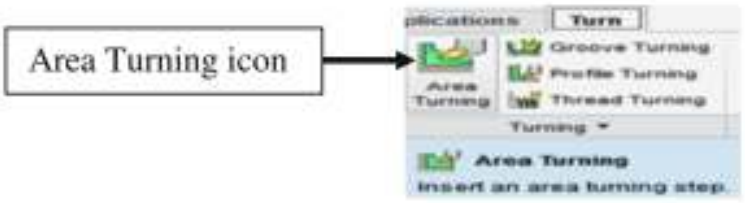
The Sketch dialogue box is activated on the main graphic window. On the Placement group, the Reference and Orientation section boxes are automatically updated by the system on the Sketch Orientation section as shown on the Sketch dialogue box in Fig. 10.21.



Click on Lines and Curves on the Part to be projected as indicated by the arrows as illustrated in Fig. 10.26.

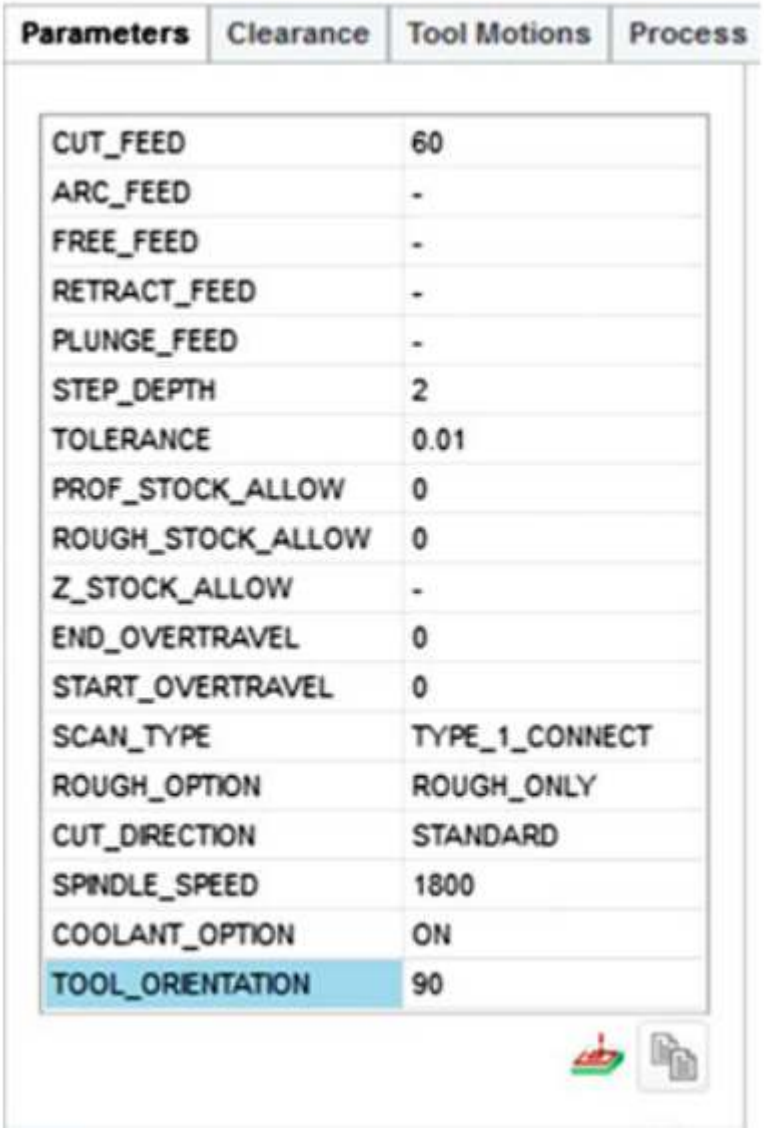


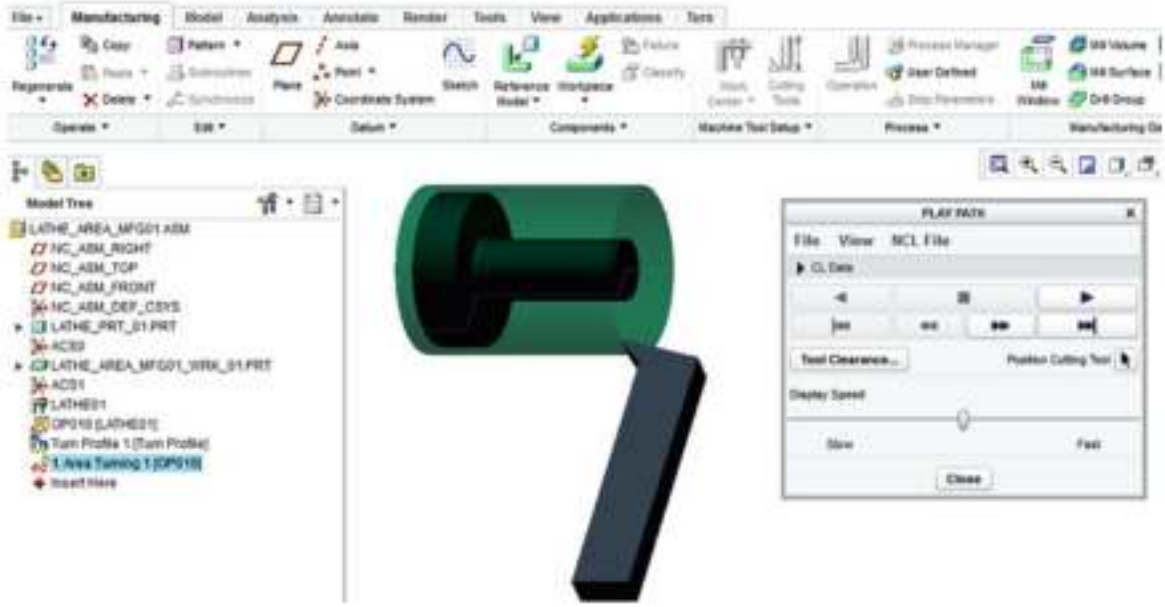
Click on the Turn menu bar to activate its menu list ≫ Now click on the Area Turning icon as indicated by the arrow below.



The Area Turning application tools are activated ≫ If T0001 and ACS1 are not automatically added by the system, manually add them yourself by following the step-by-step guide below.

Click on the Tool icon section box to activate its menu bar, now select T0001 on the activated drop-down list ≫ Click on the Coordinate System section box, and go to the Model Tree and click on ACS1. See Fig. 10.29.





Click on the Play tab to start the on-screen Area Turning operation. The end of the Area Turning operation is shown in Fig. 10.32.

Click on the Save a CL File icon as shown below.

The SELECT FEAT Menu Manager dialogue box is activated on the main graphic window » Click on Operation, and now click on OP010 on the SEL. MENU group as highlighted on the Menu Manager dialogue box shown in Fig. 10.42.

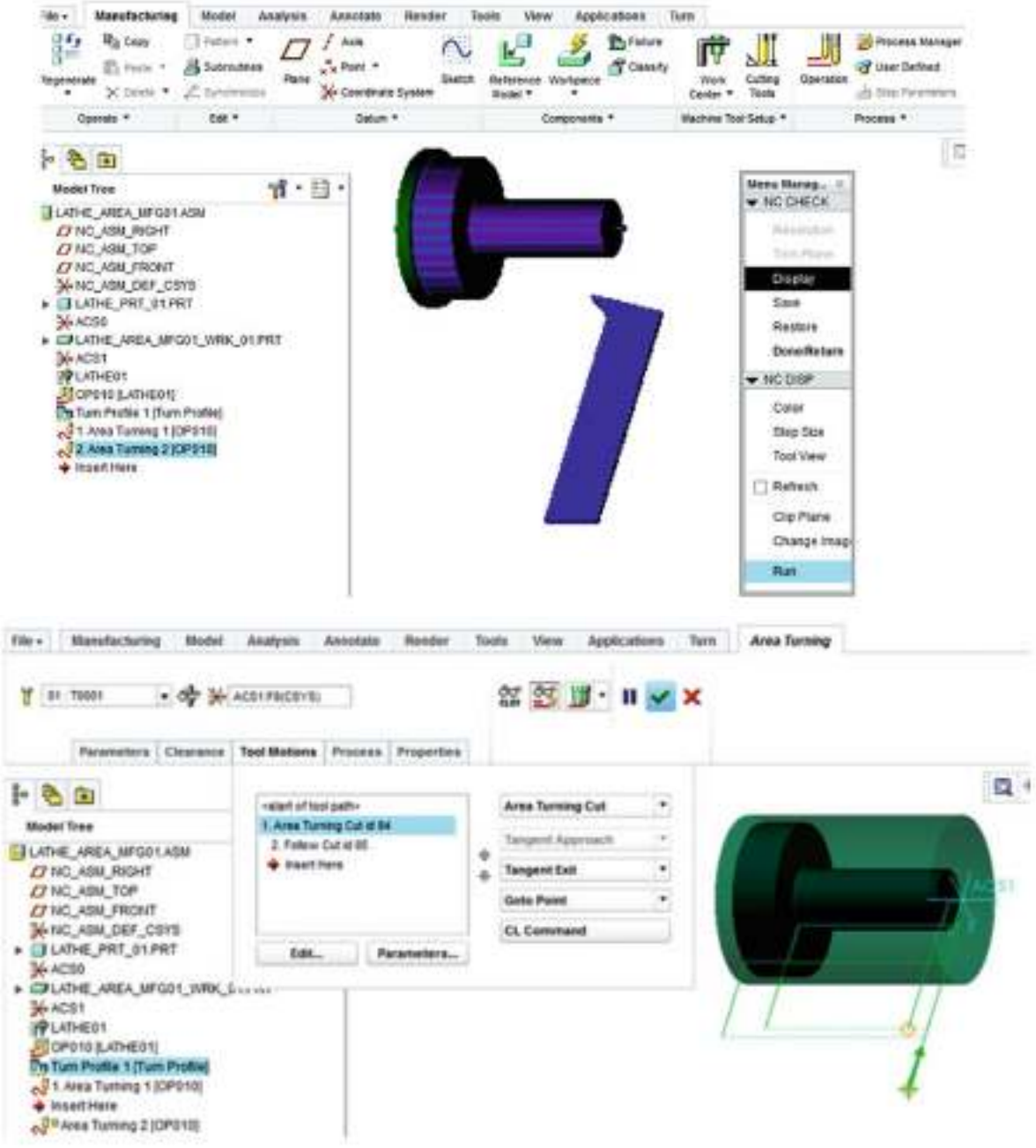


Fig. 10.42 Activated SELECT FEAT Menu Manager dialogue box

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 CL File and Interactive square boxes on the OUTPUT TYPE group as shown in Fig. 10.43.

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 CL File and Interactive square boxes on the OUTPUT TYPE group as shown in Fig. 10.43.

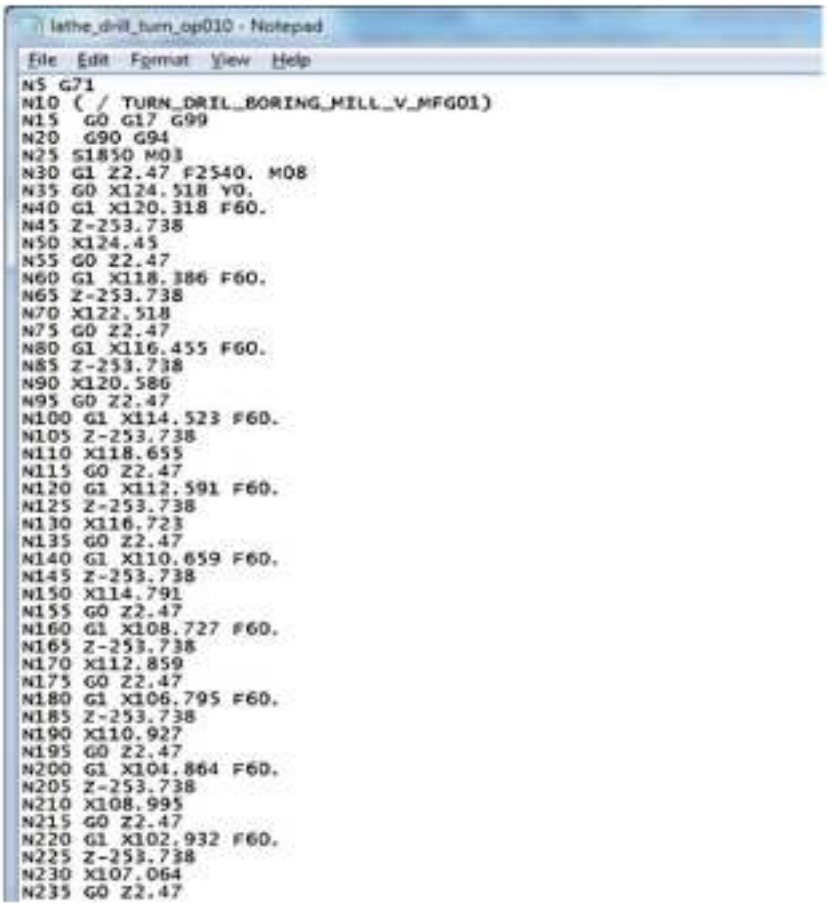
Fig. 10.43 Activated PATH Menu Manager dialogue box

Click on Done.

The Save a Copy dialogue window is activated on the main graphic window. Make sure that the correct Part name is on the New Name section box. See Fig. 10.44.

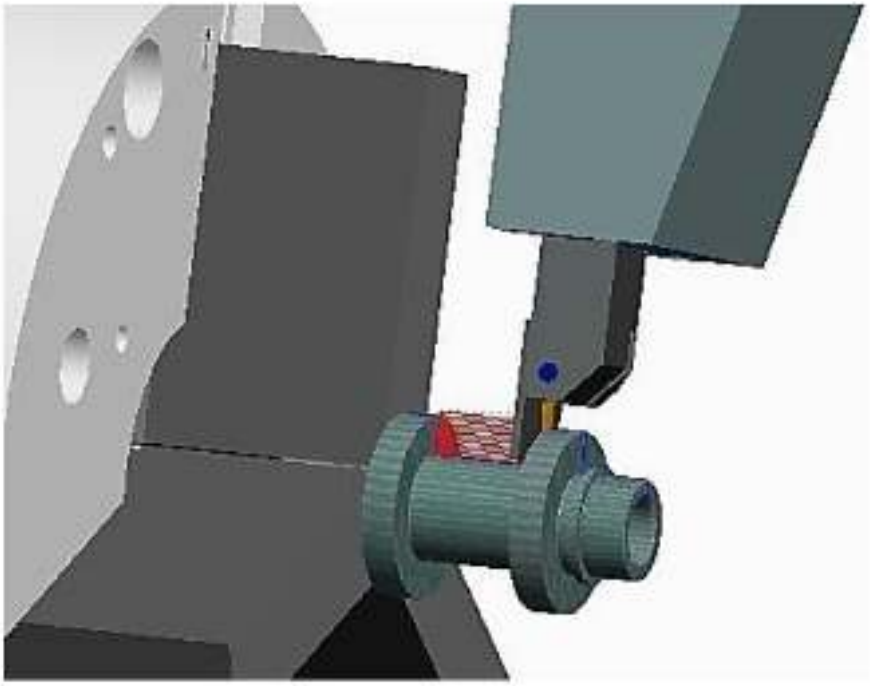
The generated G-codes data are stored as a TAP File in the chosen work directory. Open the TAP File using Notepad to view the generated G-codes.

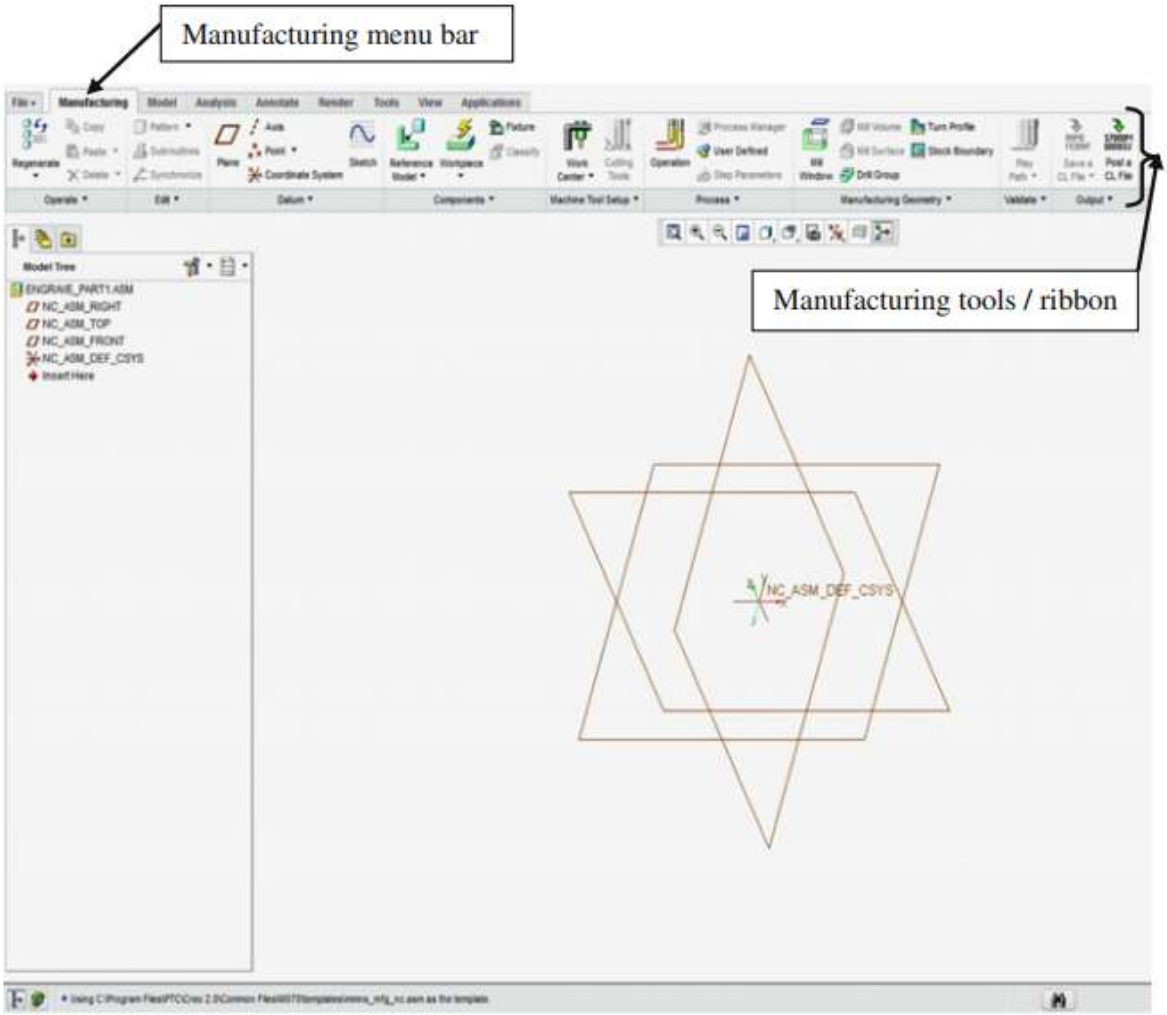
Some of the G-codes data generated in this tutorial are as shown below and on the proceeding pages.



UNIT – III

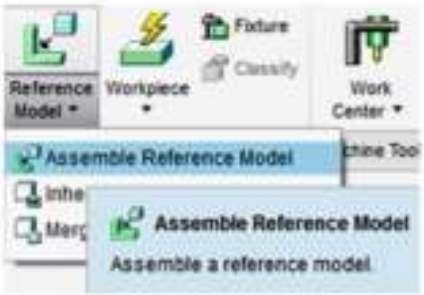
Reference Finished Part Model - Work Piece - Automatic Work Piece – CoordinateCreations





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.



The 3D Part (engrave_prt1) 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.

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.

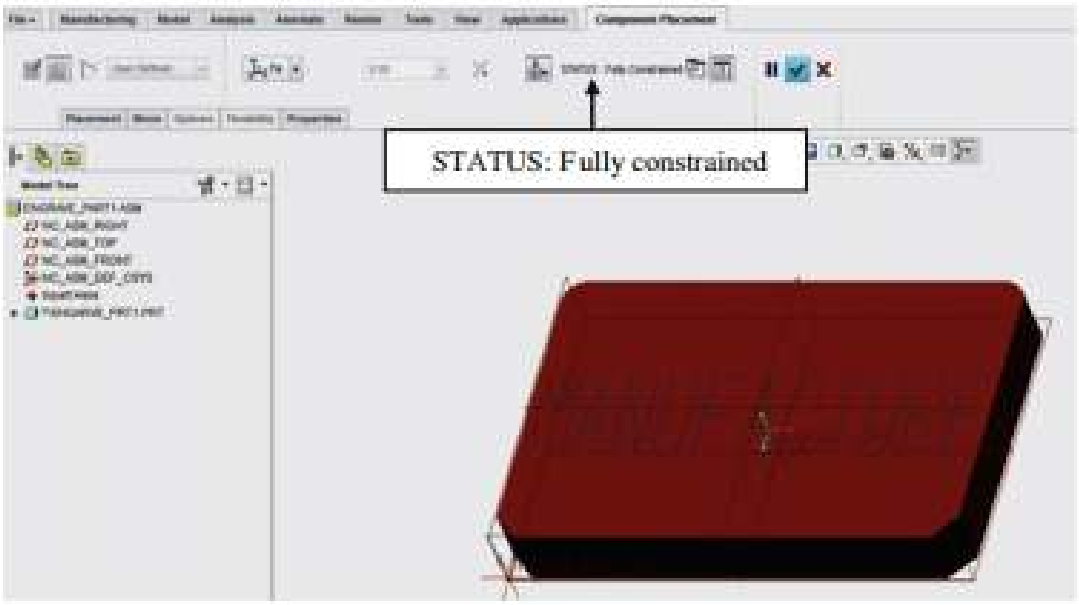


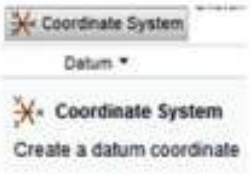
Fig. 2.53 Fully constrained Part on the Manufacturing main GUI window

Click on the Check Mark icon to exit.

2.5 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.

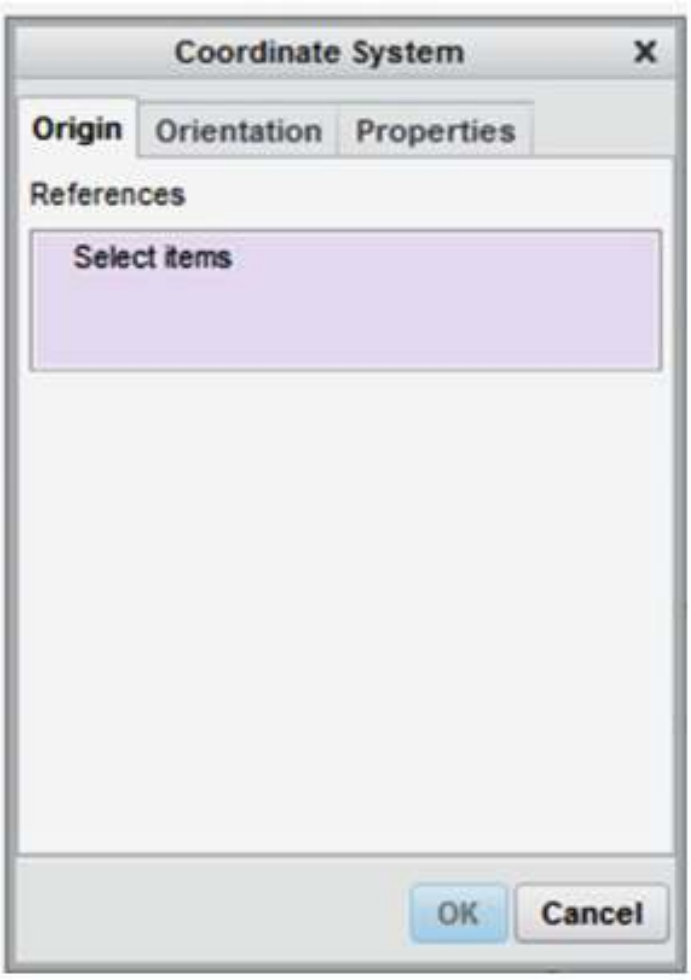


Fig. 2.54 Activated Coordinate System dialogue box

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.

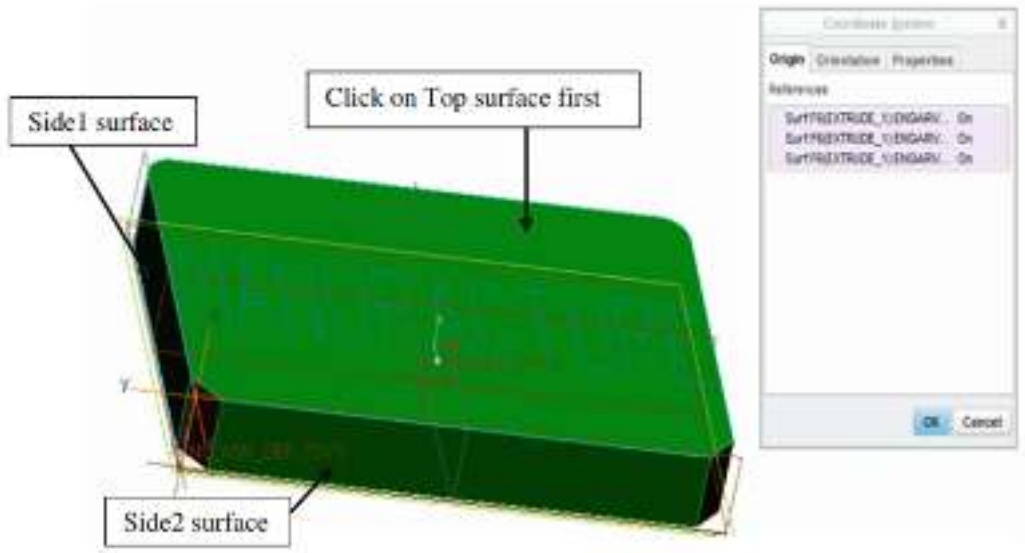


Fig. 2.55 Selected surfaces and the Coordinate Systems dialogue box

> 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.

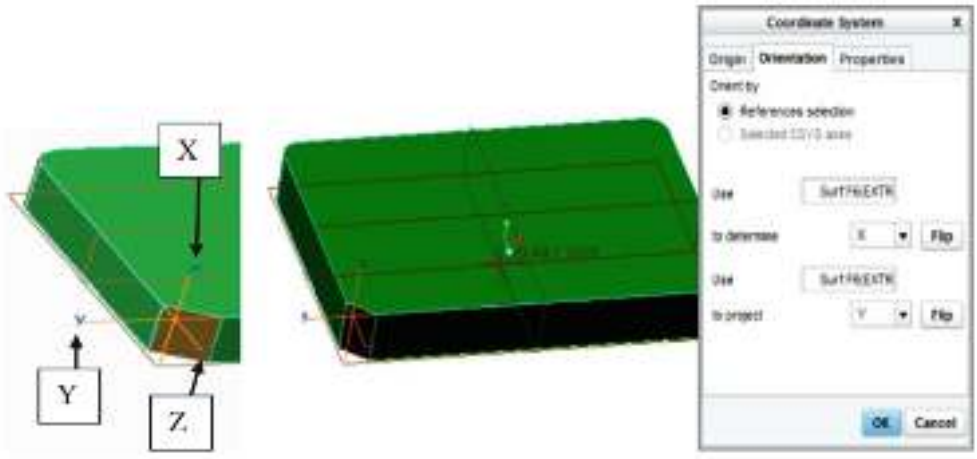
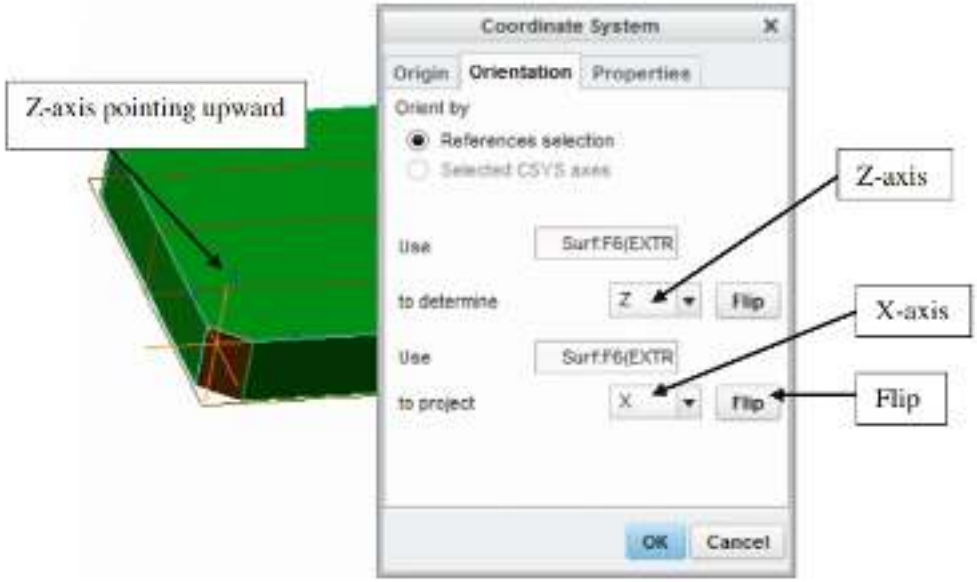


Fig. 2.56 Wrong orientation of *X*, *Y* and *Z*-axes of the Coordinate Systems

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

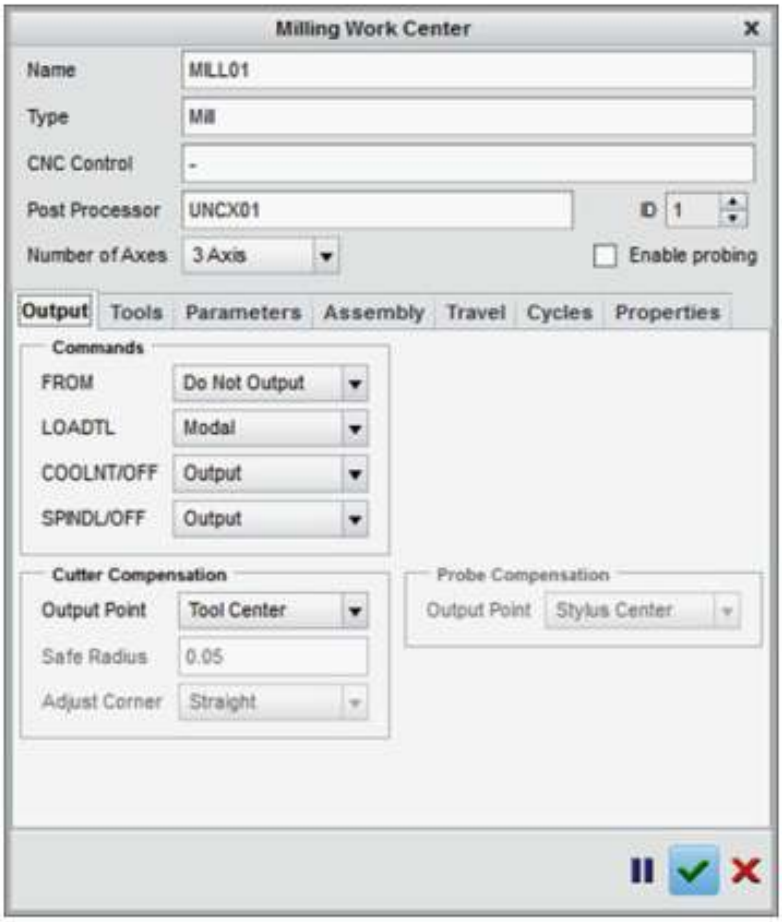
Click on the Orientation tab ≫ On the Orient by group, click in the "References selection" radio button ≫ Click on "to determine" section box and click on *Z* axis on the drop-down list ≫ Click on "to project" section box and click on *X* axis on the drop-down list ≫ Now click on the Flip tab, to flip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



Machine Tool Setup - Work Center - Mill, Parameters Setting- AddTools, Cutting Tool Setup

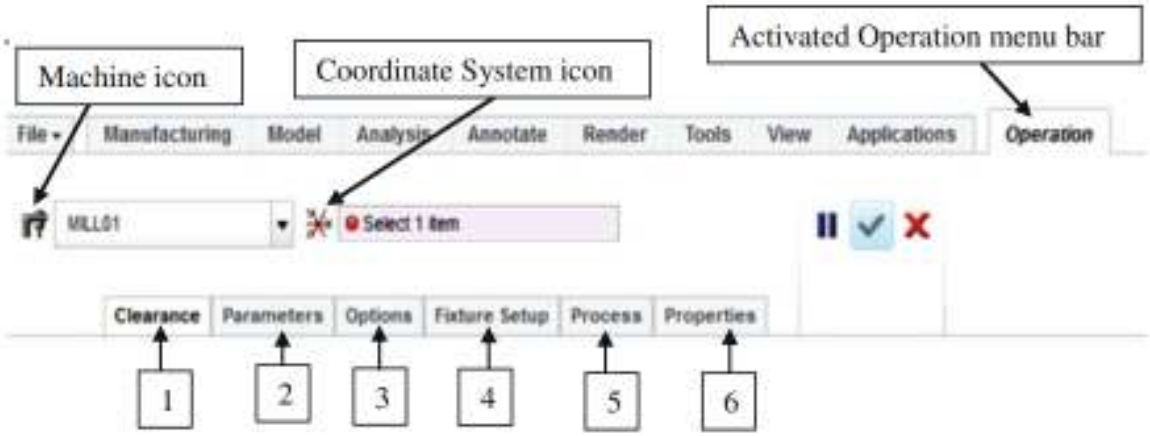
Click on the Work Centre icon on the Manufacturing menu bar ≫ On the activated drop-down menu list, click on Mill as shown below.

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.



Click on Operation icon as shown below

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



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)

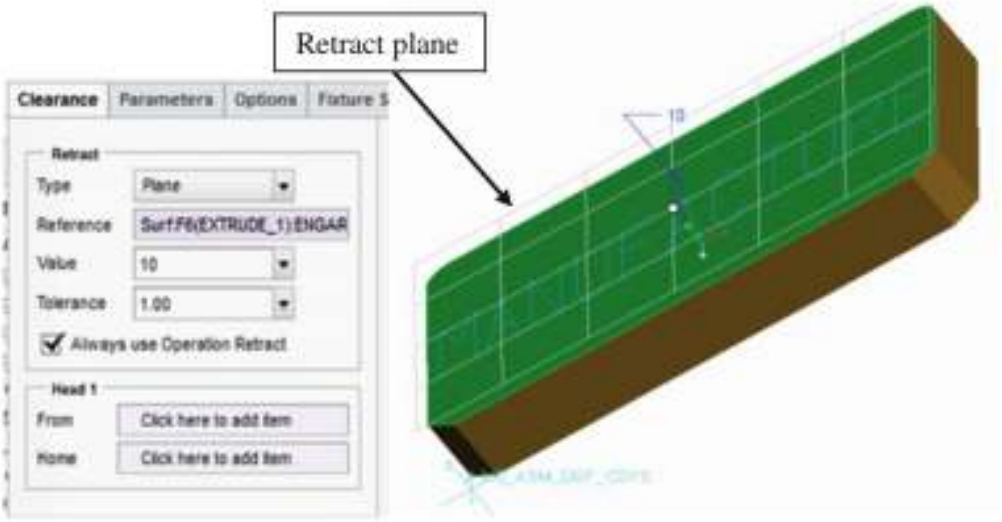
Click in the Coordinate System icon section box ≫ Go to the Model Tree and click on ACS1. ACS1 is automatically added onto the Coordinate System icon section box.

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.



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.

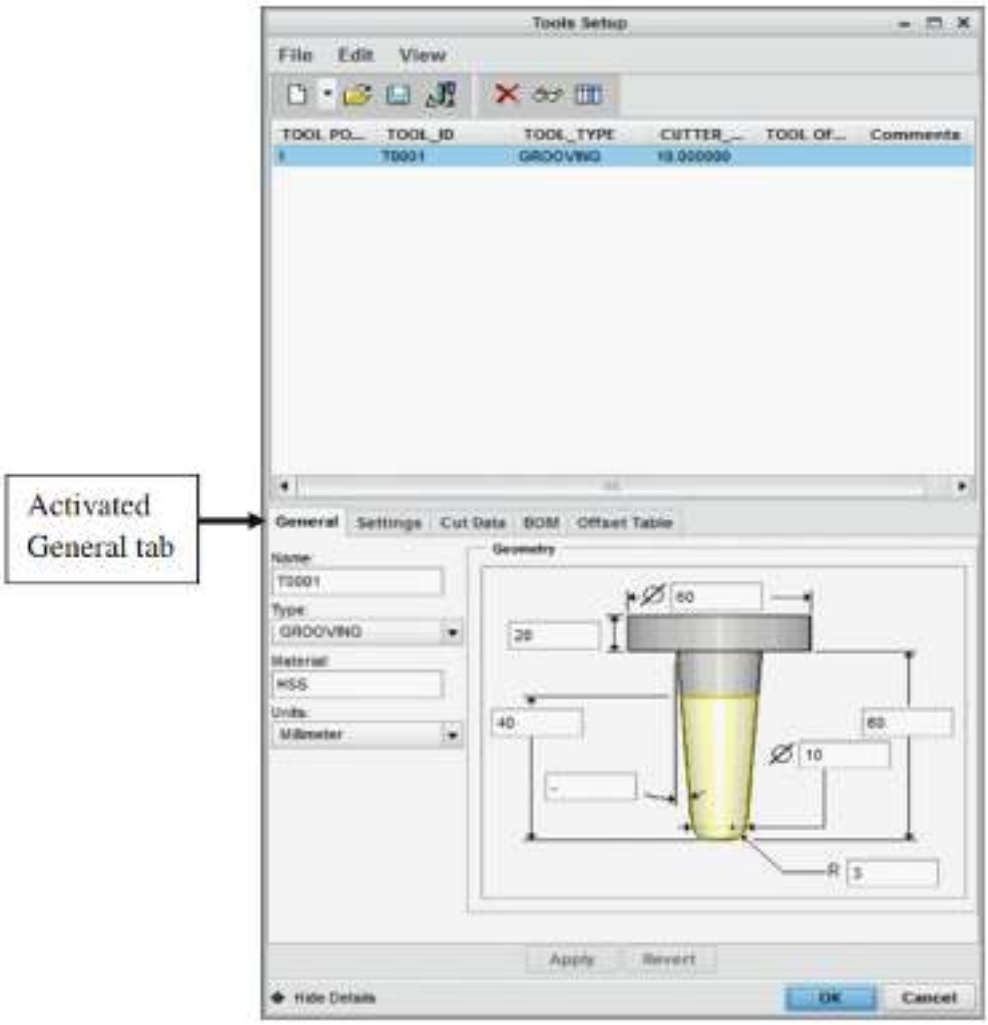
Check the Model Tree to make sure that the ACS1, MILL01 and OP010 [MILL01] as created are on the Model Tree as shown in Fig. 2.62.

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.



Mill Operation, CSYS Selection - Clearance Type,Reference Surface - Mill – Face, Cut – Feed, Slep – Depth. Step over, Spindle –Speed

Click on the Operation icon as shown below.

The operation dashboard tools are activated as shown in a concise form in Fig. 3.19.

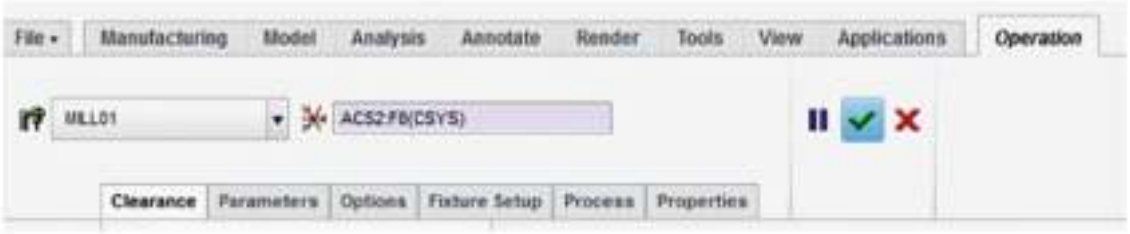


Fig. 3.19 Activated Operation dashboard

To add ACS2, if it is not automatically added by the system » Click on the Coordinate System icon section box » Go to the Model Tree and click on the created Coordinate System (ACS2).

Click on the Clearance tab to activate its panel » On the retract group, click on the Type section box and click on Plane on the activated drop-down list as shown in Fig. 3.20.

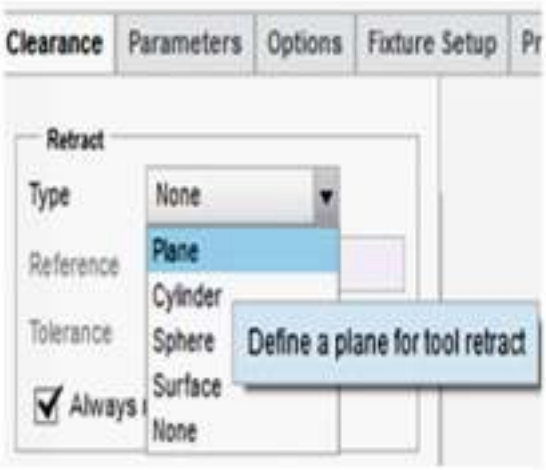


Fig. 3.20 The Type drop-down menu list on the Retract group

Click on the workpiece top surface and the Reference section box is automatically updated by the system ≫ Type 10 mm in the Value section box, as illustrated by the arrow in Fig. 3.21.

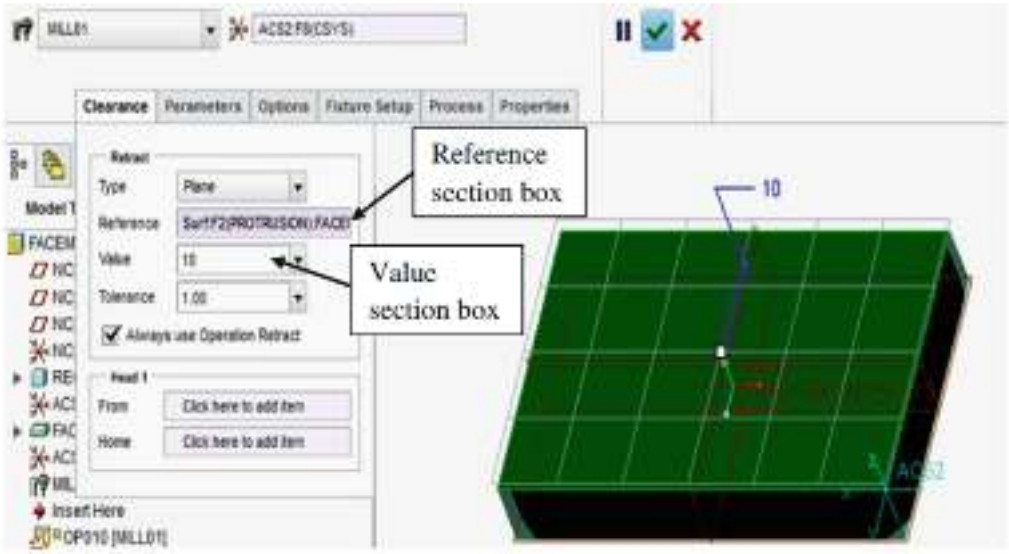


Fig. 3.21 Activated Clearance panel indicating the Type and References updates

Now click on the Check Mark icon to exit.

Note that the retract surface is the surface that the tool will move up to, after each machining process.

The workpiece changes on the Manufacturing main graphic window. See Fig. 3.22.

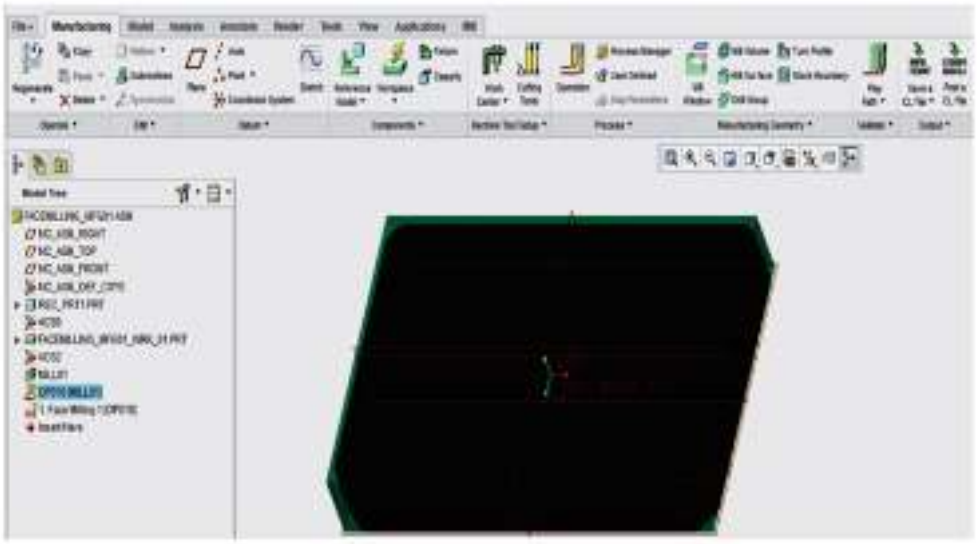


Fig. 3.22 Workpiece after adding the work centre and operation parameters

Check the Model Tree as shown in Fig. 3.23 to make sure that the ACS2, MILL01, and OP010 [MILL01] are in the Model Tree as created.

Fig. 3.23 Model Tree after creating Auto-Workpiece, ACS2, work centre, and operation

3.7 Setup 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 to activate its content ≫ Type the tool name in the name section box or leave the generated default Tool name ≫ Click on the downward arrow on the Type section box and now click on End Mill Tool on the activated drop-down menu list ≫ Click on the Material section box, and type the tool material. Add dimensions to the tool. See Fig. 3.24.

3.8 Setup the Face Milling Operation

Click on the Mill menu bar to activate the Mill ribbon ≫ Now click on Face milling icon on the milling group as indicated by the arrow shown below.

Fig. 3.25 Activated Face Milling dashboard in concise form

To add "01:T0001", click on the Tool icon section box to activate its drop-down menu list ≫ Now click on "01:T0001" on the activated drop-down menu list.

Click on the Coordinate System section box; now click on the created Coordinate System (ACS2) on the Model Tree, if it is not add automatically by the system.

Click on the Reference tab to activate its panel as shown in Fig. 3.26.

Now click on the "Type" section box; and now click on Surface on the activated drop-down menu list ≫ Click on the workpiece top surface and the Machining References section box is automatically updated by the system.

Note: You can also use the filter selection to select the top surface of the workpiece. Go to the bottom of the main graphic window and click on the downward arrow on the active filter to activate its drop-down menu list. Now, click on reference model surface as highlighted in the drop-down list in Fig. 3.27.

Now click on the top surface of the Workpiece and the Machining References section box is automatically updated/added.

Click on the "Parameters" tab to activate its panel. Type values for the CUT_FEED, STEP_DEPTH, STEP_OVER, CLEAR_DISTANCE, and SPINDLE_SPEED as shown in Fig. 3.28.

Fig. 3.28 Parameter values for face milling process

Click on the Check Mark icon ✔ to exit.

Note: The CUT_FEED represents the speed of the tool on the Part surface being machined.

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 motions before CLEAR_DISTANCE are in rapid mode (G00). The CLEAR_DISTANCE must always be less than the values of the height of retract plane/distance.

The TOLERANCE controls the accuracy to which the cutter path follows the shape of the component.

The STEP_OVER is the distance the tool moves between adjacent tool path tracks. The distance or Step-over value determines whether the surface finish on a Workpiece is rough or smooth. While using a flat bottom tool such as an End Mill, the Step-over value normally ranges between 70 and 80 % of the cutter diameter. When using a Ball nose cutter, the Step-over will be considerably smaller, when roughing and finishing mainly due to the geometry of the tool. A larger Step-over will ultimately give a rough surface finish than a small Step-over.

UNIT IV

Display Tool path - Tool Preview, Milling – Play path, Material Removal Simulation Display NC Tool Path - G-Codes used in CNC Programming Colmmon

Click on the Manufacturing menu bar >> Click on the Play Path icon to activate its drop-down menu list, now click on Play Path on the drop-down menu list as indicated by the arrow in Fig. 3.29.

Fig. 3.29 Manufacturing ribbon toolbar with Play Path tab activated

Alternatively, right click the mouse on Face Milling 1[OP010] on the Model Tree indicated by the arrow as shown below.

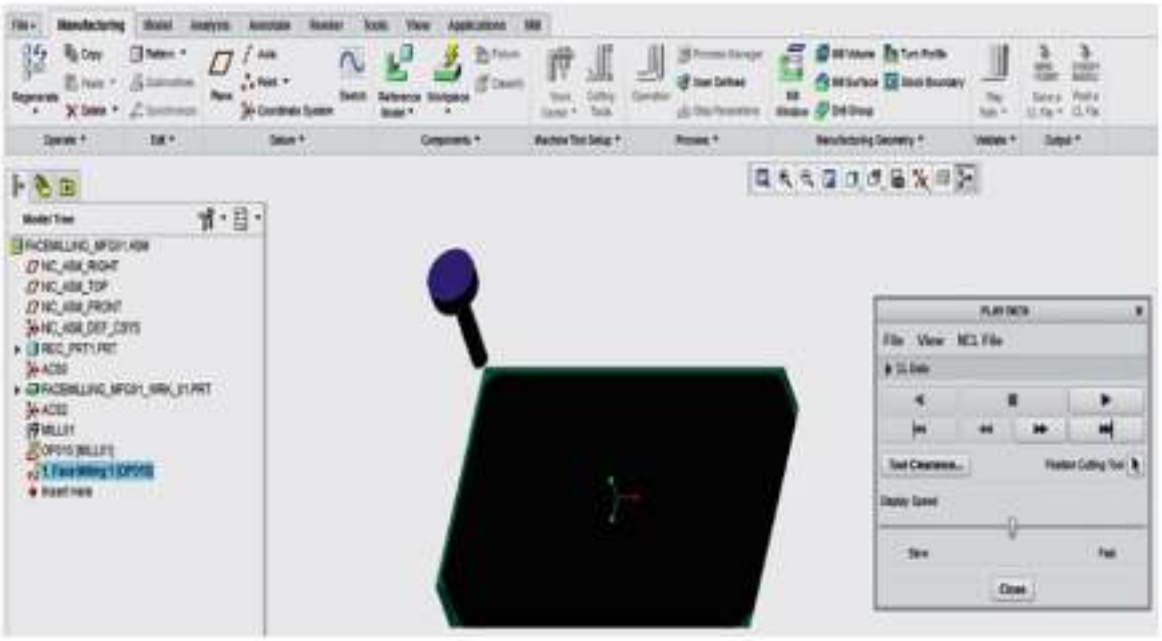


Fig. 3.30 Activated End Mill Tool and PLAY PATH dialogue box

Note: The tool speed can be adjusted to enable better view of the on screen face milling operation.

Click on the play tab to start the on screen face milling operation. The end of the face milling operation is as shown in Fig. 3.31.

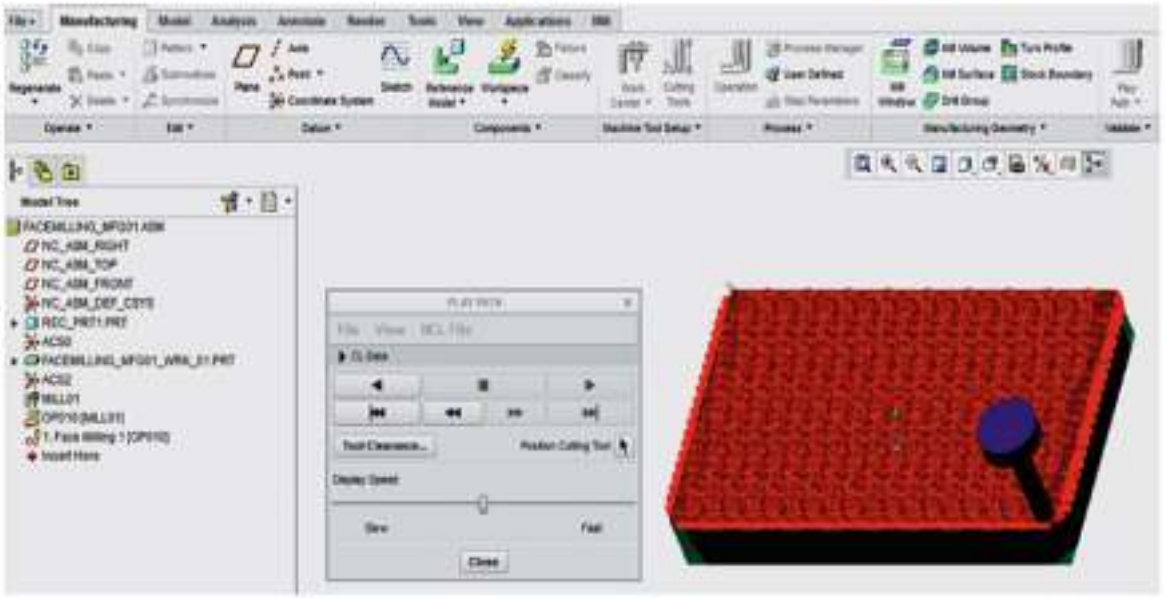


Fig. 3.31 End of on screen Face milling operation Face Milling operation

Click on the Close tab to exit the on screen face milling operation.

Note: To edit the parameters of OP010 [MILL01], face milling 1 [OP010] and MILL01 on the Model Tree, just highlight and right click the mouse on each and then click on Edit Definition on their respective activated drop-down menu list.

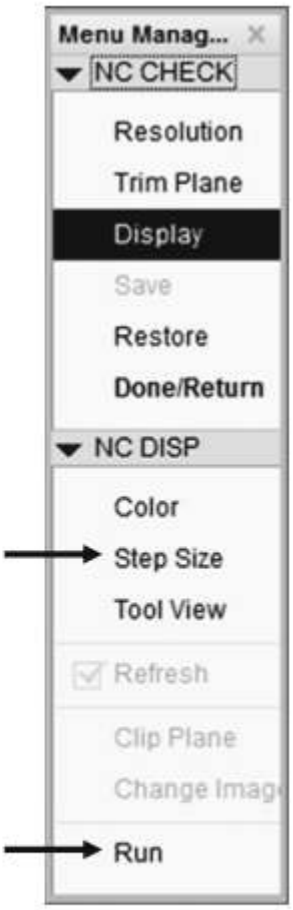
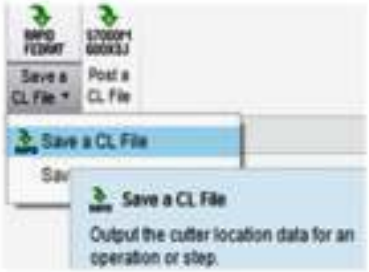


Fig. 3.36 Activated NC CHECK Menu Manager dialogue box

Click on Run to activate the Material Removal Simulation. Now click on the Step Size and change the automatic generated value to a lower value, this will slow down the on screen animation. Click on Done/Return on the NC CHECK group to exit the Material Removal Simulation

M-Codes - Reading Manufacturing Drawings - Work Steeing and Offsets, workCordinates Milling Tool types, Face Mill, Slot Mill, Hole Making tools.

Click on the Save a CL File icon on the Manufacturing menu bar ≫ Click on Save a CL File on the activated drop-down menu list as shown below.



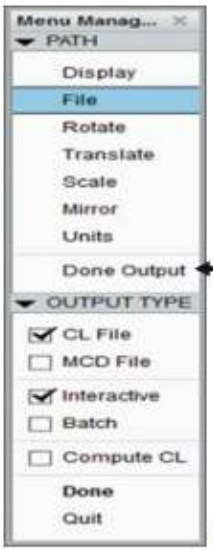
The SELECT FEAT Menu Manager dialogue box opens up on the main graphic window as shown in Fig. 3.37.

Fig. 3.37 Activated SELECT FEAT Menu Manager dialogue box

Now, click on Operation if it is not automatically selected by the system and click on OP010 on the SEL MENU group as highlighted above.

Once OP010 is clicked on the SEL MENU group, the "PATH" Menu Manager dialogue box opens up on the main graphic window.

On the PATH group, click on File and Check Mark the "CL File" and "Interactive" square boxes on the OUTPUT TYPE group as shown in Fig. 3.38, if they are not automatically selected by the system.

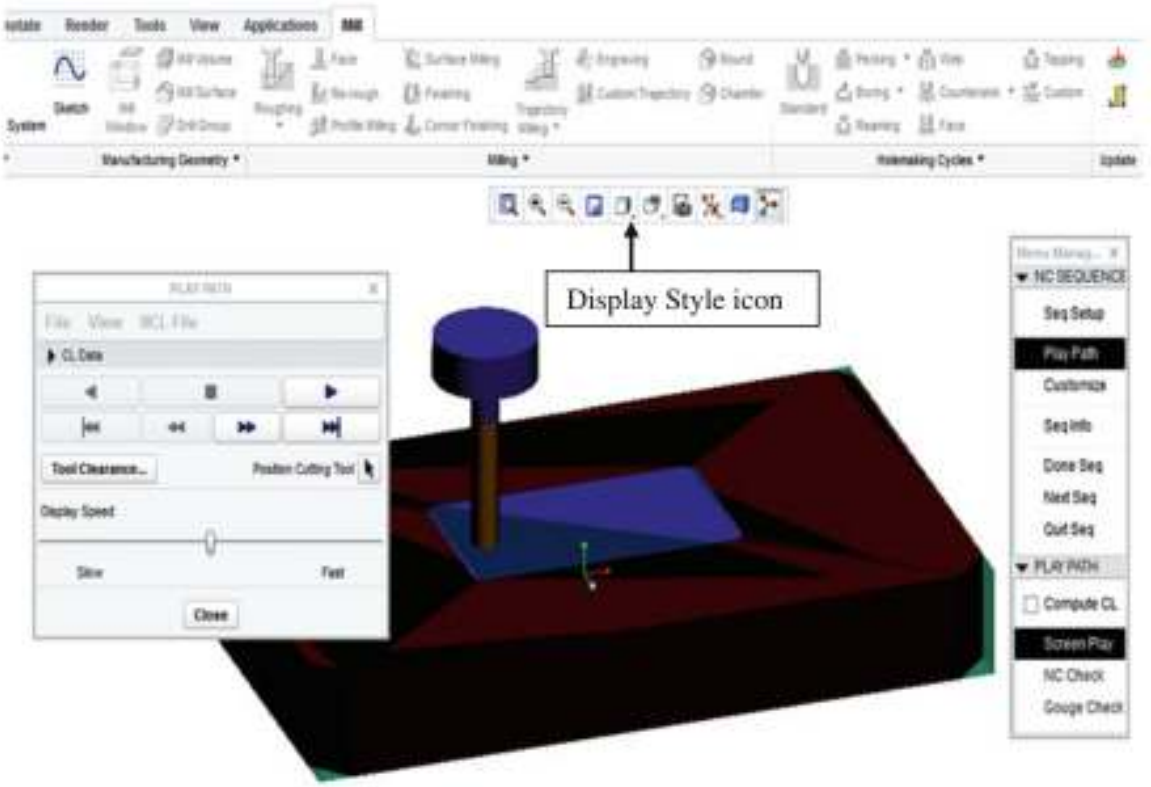


Done Output

Now click on Done on the OUTPUT TYPE group.

Fig. 3.38 PATH Menu Manager dialogue box

V



3.12 Creating the G-Code Data

Click on the Post a CL File icon tab 

.

The Open dialogue window pops up on the main graphic window. See Fig. 3.40.

Fig. 3.40 Activated Open dialogue window in a concise form

Click on the saved 

file on the Open dialogue window. Make sure that the correct file name and extension (.ncl) is in the "File name" section box.

Now 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" if they are not automatically Checked Marked by the system as shown in Fig. 3.41.

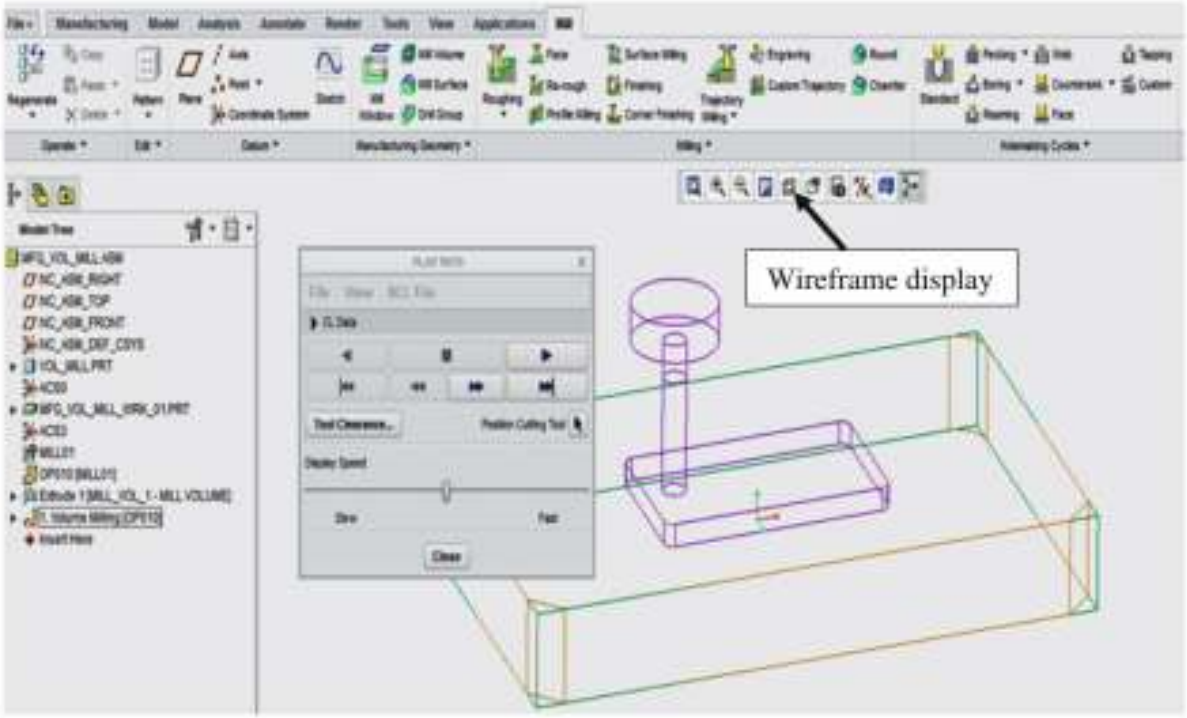
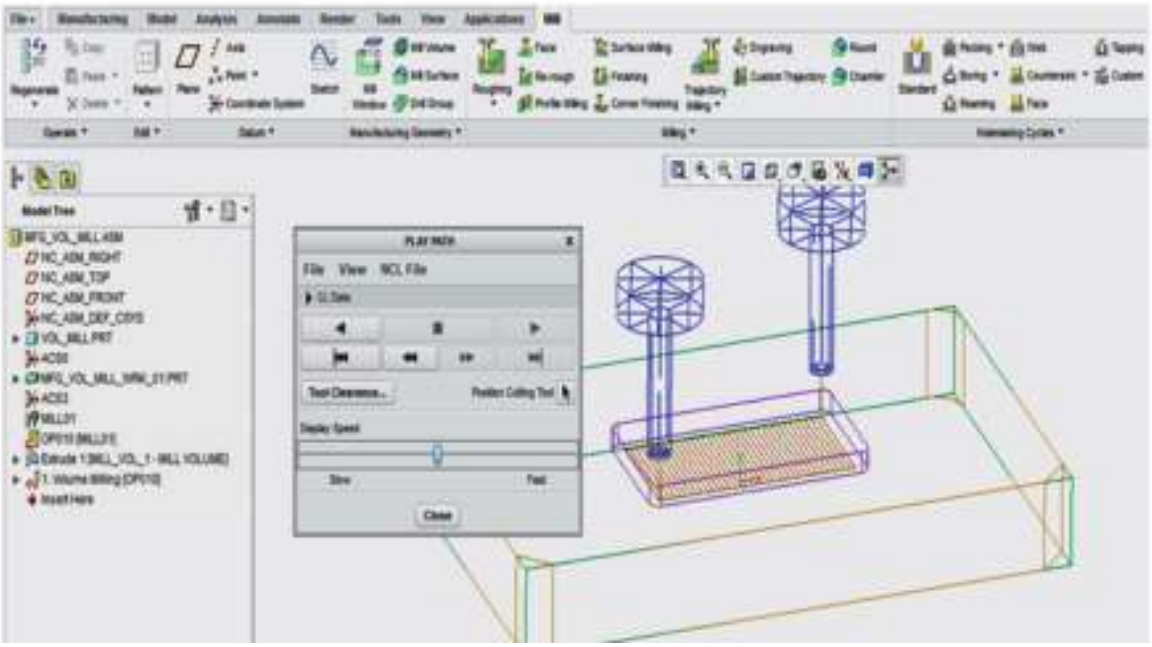
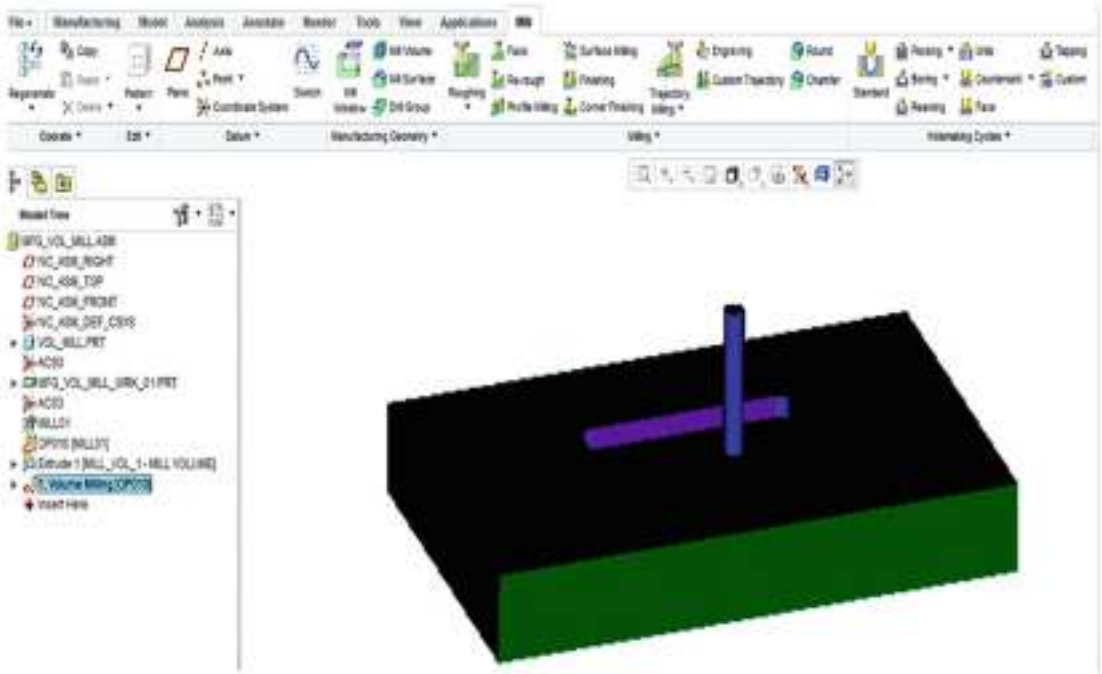


Fig. 4.50 Cutting Tool and Workpiece in wireframe display

Click the Play tab to activate the on screen simulation process.
The end of the Volume Rough milling operation is shown in Fig. 4.51.

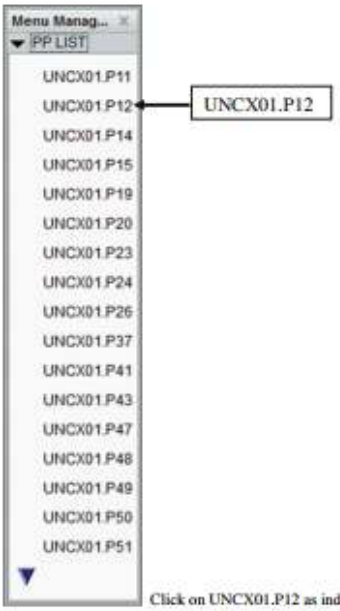




Click on Done.

Fig. 3.41 Activated PP Options Menu Manager dialogue box with Verbose and Trace Checked Marked

The system automatically generates the PP (Post Processor) LIST as shown (Fig. 3.42).



Click on UNCX01.P12 as indicated by the arrow.