VINAYAKA MISSION'S RESEARCH FOUNDATION
(Deemed to be University)
AARUPADAI VEEDU INSTITUTE OF TECHNOLOGY

DEPARTMENT OF MECHANICAL ENGINEERING

COMPUTER INTEGRATED MANUFACTURING LAB MANUAL

R2021

Prepared by

Mr. P. KUMARAN
ASSISTANT PROFESSOR – II / MECH

INTRODUCTION TO CATIA

In cad laboratory we are going to study about how to create a model of engineering objects and also how to create an assembly of modeled objects. The modeling software's like CATIA, Pro e, Unigraphics are generally used in mechanical engineering field for the modeling. In this lab catia-v5r18 software is used to do the exercises.

CATIA (Computer Aided Three-dimensional Interactive Application) is a multi-platform CAD/CAM/CAE. It is written in the C++ programming language. Commonly referred to as a 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAx), from conceptualization, design (CAD), manufacturing (CAM), and engineering (CAE).CATIA can be customized via application programming interfaces (API). V4 can be adapted in the Fortran and C Programming languages under an API called CAA (Component Application Architecture). V5 can be adapted via the Visual Basic and C++ programming languages, an API called CAA2 or CAA V5 that is a component object model (COM)-like interface. Although later versions of CATIA V4 implemented NURBS, V4 principally used piecewise polynomial surfaces.

AN OVERVIEW OF CATIA DESIGN SOFTWARE

Optimal Sharing

Catia V6 users will get access to a unique, collaborative 3 dimensional environments that can be access by an unlimited number of people online. This allows people from across the globe to collaborate in a virtual environment. It has been designed to not only allow for online cooperation, but also makes offline sharing and designing easy to integrate as well.

Simplified Product Development

Creating a new product can be a long and complex process. It encompasses multiple design phases including the initial design, overall development, and manufacturing. Catia V6 decreases the complexity and length of the entire project because it integrates various stages of the development process so that they can be controlled and modified on a single platform. It does this by using an approach to systems engineering known as RFLP. This allows you to create several versions of the same product using different sets of requirements. This gives you a comprehensive look at what the final product could be Seamless Transitioning.

Every version of the CATIA design software is designed to allow for seamless integration with previous versions. This makes upgrading a simple process and can be completed without losing any of the information that has already been stored. CATIA design software has found itsway into more and more industries with each passing year. Traditionally, it gained notoriety through 3 main industries; however, every industry that is involved in engineering has found ituseful.

CATIA has become a leader in product development software and may be exactly what is needed to overcome the shortcomings of CAD software.

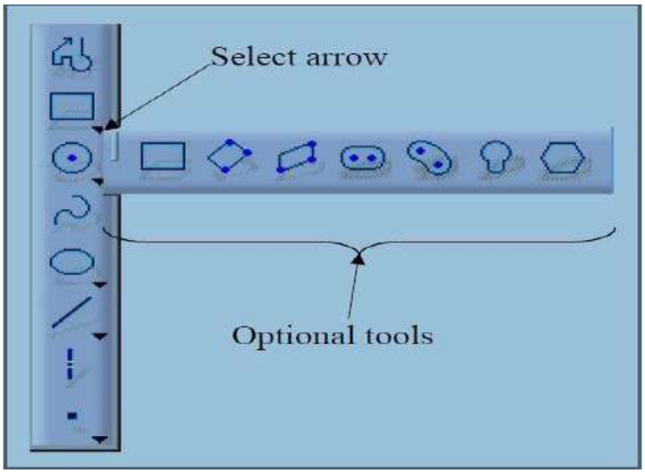
Tool Bars

Many standard toolbars are used in the different modes like sketcher mode operational mode etc, in CATIA software. Here we discuss about the tool bars used in sketcher with some examples.

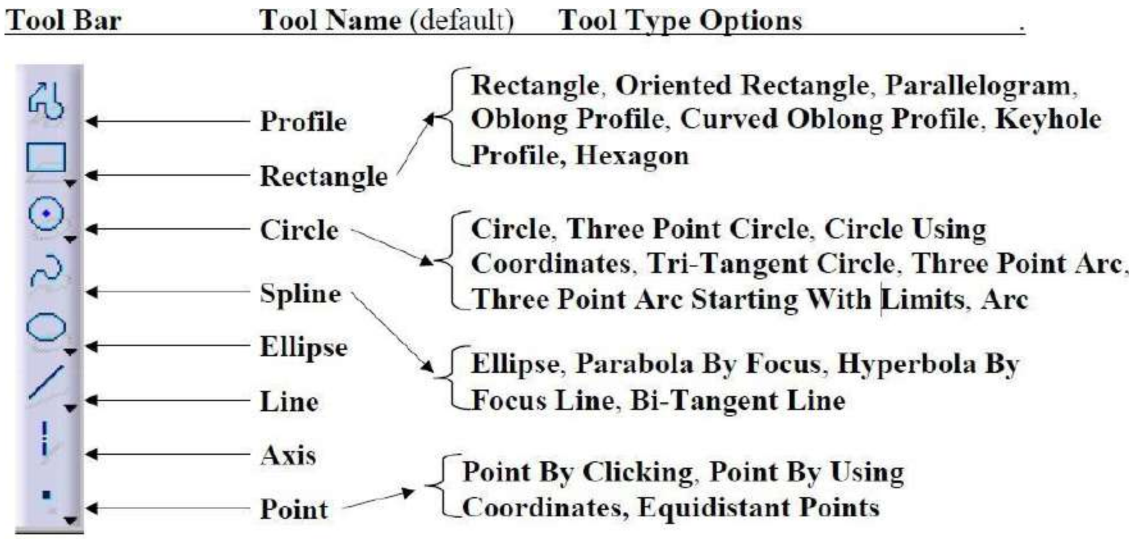
Sketcher Work Bench Tool Bars

There are three standard tool bars found in the Sketcher Work Bench. The three toolbars are shown below. The individual tools found in each of the three tools are labeled to the right of the tool icon. Some tools have an arrow located at the bottom right of the tool icon. The arrow is an indication that there is more than one variation of that particular type of tool. The tools that have more than one option that are listed to the right of the default tool. To display the other tool options you must select and hold the left mouse button on the arrow as shown in Figure1.1. This will bring up the optional tools Select Arrow Optional tools.

THE OPERATION TOOL BAR



THE PROFILE TOOL BAR



Tools covered in this lesson: **Profile, Rectangle, Circle, Line** and **Point**.

THE CONSTRAINTS TOOL BAR

Tool Bar	Tool Name (default)	Tool Type Options

Constraints Defined In Dialog Box

Constraint

Auto Constraint

Animate Constraint

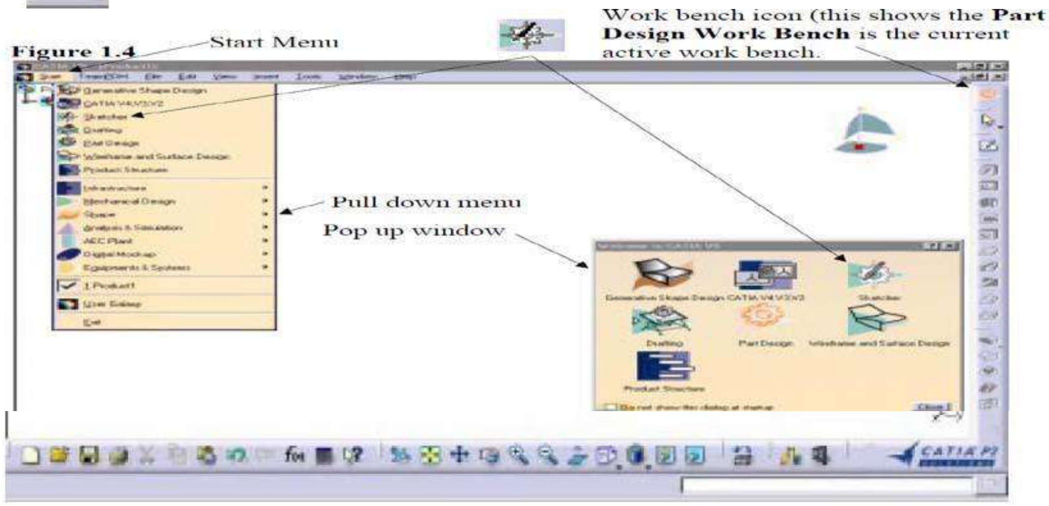


Figure 1.4

Specify a Working Plane

The next step is to create a 2 dimensional profile of the part. The Sketcher Work Bench is a two dimensional (planar) work area. To use the Sketcher Work Bench, you must specify which plane the profile is to be created on. Specifying, the plane can be done several differentways.

ZX plane

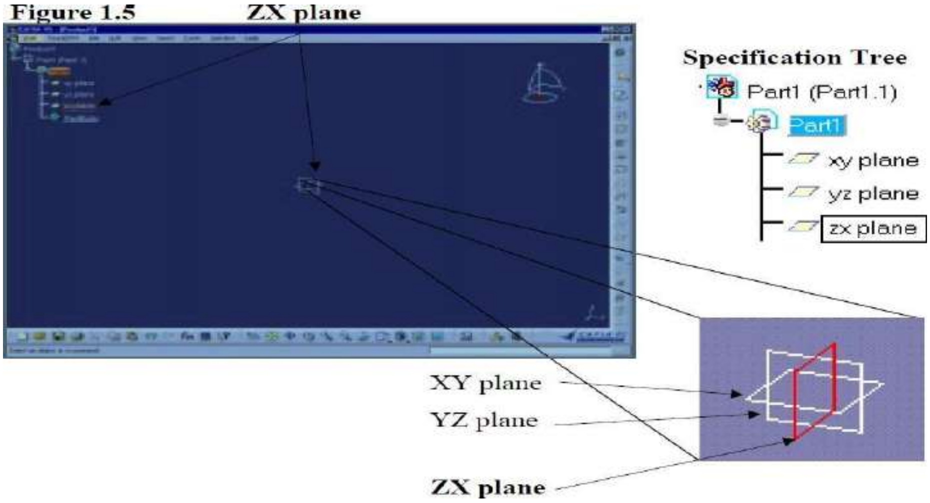
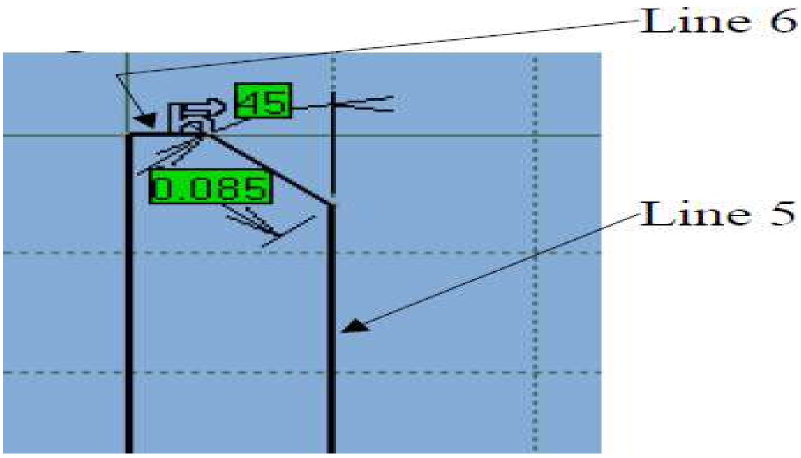


Figure 1.5

MODIFYING THE PROFILE USING CHAMFER

The Chamfer icon is also located in the Operations tool bar. This procedure assumes you know what a chamfer is. The steps required to create a chamfer are almost identical to creating a corner. Select the Chamfer icon.
The command prompt at the bottom left hand of the screen, will prompt you with the following: "Select the first curve, or a common point". For this exercise select line 5. The next command prompt will ask you to "Select the second curve".



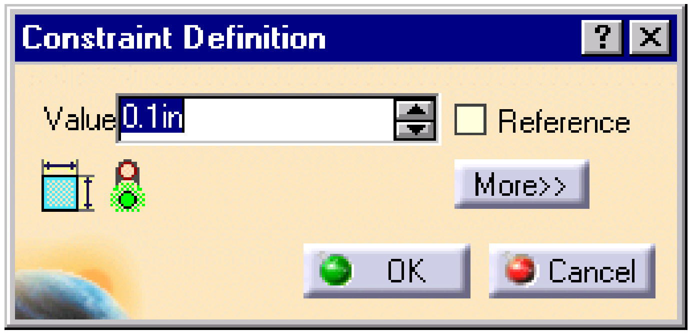
Constraint

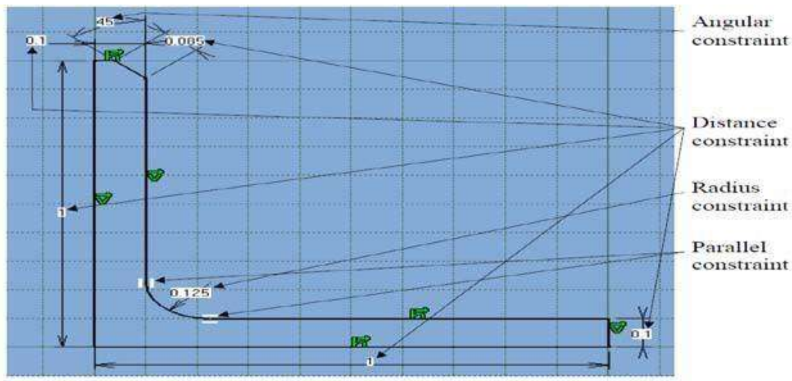
This tool allows you to create individual constraints, one at time. You have already applied a constraint and may not even know it. The Anchor icon is a constraint. The values attached to the Chamfer and Corner is constraints. To apply Dimensional Constraints, complete the following steps:

Select the Constraint icon.
Select the line and/or Sketcher element to be constrained.
The Sketcher element will turn green (constraint symbol) along with the appropriate dimension and box with the value in it. To re-locate the constraint value, select the value box and drag the mouse to the desired location.

If the initial location of the constraint is not satisfactory re-select the dimension and drag and drop it at the new location.
To edit the value of the constraint double click on the value box. This will bring up the Constraint Definition pop up window shown in. This window shows the existing value for the Sketcher element. This value can be edited by typing the new value over the existing value. Then select OK or hit the Enter key. The entities linked to the constraint will automatically be updated to the new value.

If the constraint is between two different entities, such as lines, select the first line and then the second line. CATIA V5 will constrain the distance between the two entities. The constraint value will appear near the constraint to move the constraint value. For this lesson constrain your "L Shaped Extrusion" similar to the one shown in Figure.





Suggested Steps for practice exercises:

1. Select the XY plane (the plane the profile will be sketched on). Enter the **SketcherWork Bench**.
2. Sketch the profile of the part. Hint: use the **Profile** tool.
3. Anchor the lower left hand corner of the sketch. For anchoring a profile.
4. Constrain the profile to match the dimensions given in the profile.
5. Exit the **Sketcher Work Bench**, return to the **Part Design Work Bench** (the 3D environment). **Sketcher Work Bench** and entering the **Part Design Work Bench**.
6. Once in the **Part Design Work Bench** extrude the profile to the dimension. It'sExtrude or cut the profile.
7. Finally save a part drawing.

****

TUTORIAL - 1

Creating the "Swivel. CATPart" Using Multiple Sketches

1.1 Start CATIA V5.
1.2 Verify that you are in the **Part Design Work Bench** and the default **Properties** are set the way you want them such as **Units**. For this step, set the Units to mm.
1.3 In the **Specification Tree** rename **Part.1** to "**Swivel**".
1.4 Enter the **Sketcher Work Bench** using the **ZX Plane** as shown in figure 1.1

ZX Plane

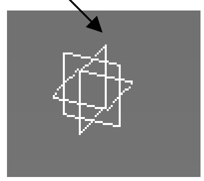


Figure 1.1

1.5 Create a circle at the coordinates (0,0) with a diameter of 25 mm as shown in figure 5.2

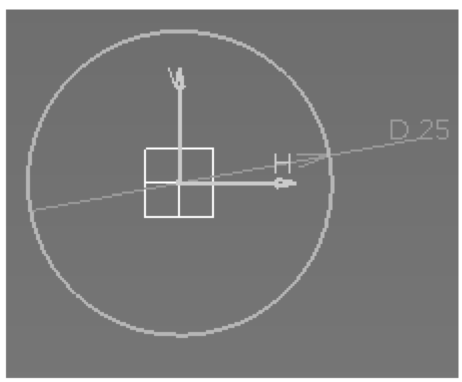


Figure 1.2

1.6 Exit the Sketcher Work Bench. Remember, this will put you back into the **Part Design Work bench**.
1.7 Select the **Pad** tool.
1.8 When the **Pad Definition** window appears, , select the **More** button. This will expand the **Pad Definition** window to show the **Second Limit** box. Figure 1.3 shows the **Pad Definition** window expanded to include the **Second Limit.**

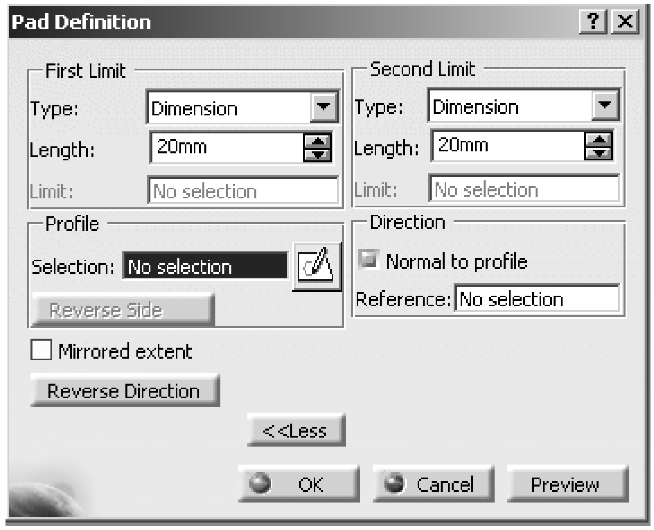


Figure 1.3

1.9 In the **First Limit** area enter “**20mm**” for the **Length** Box. Leave the **Type** box set at **“Dimension”,** as shown in figure 1.5.

1.10 In the **Second Limit** area, enter ”**20mm”** for the **Length** box, as shown in figure 1.5.

1.11 Select the **OK** button. Notice that the circle has been extruded 1 inch in both directions, reference figure 1.4

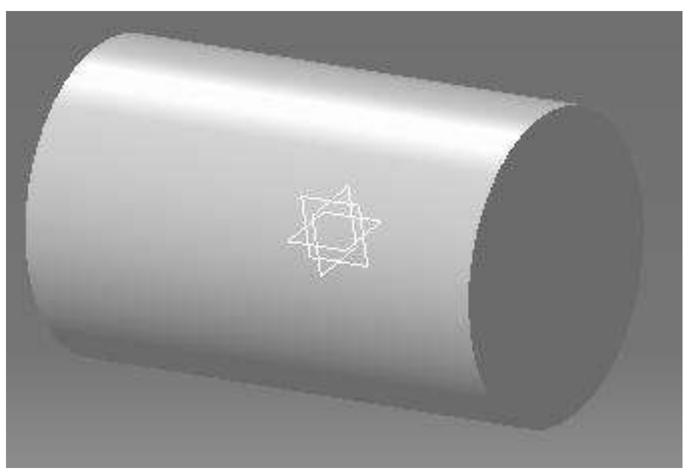


Figure 1.4

1.12 Select the ZX Plane (figure 5.1) and enter the **Sketcher Work Bench**.

1.13 Create another circle at the Coordinates (0,0) with a diameter of 18 (Figure 1.5).

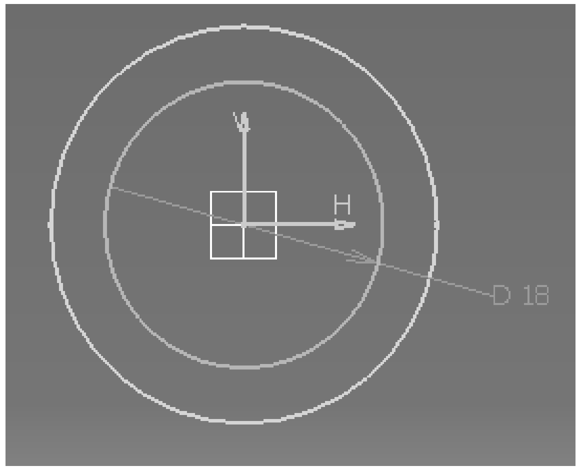


Figure 1.5

1.14 Exit the **Sketcher Workbench**. Select the **Pad** tool and select the **More** button to expand the Pad Definition window.

1.15 Enter “40” as the First Length and “40” as the Second length.

1.16 Select the OK button. The part should look similar the part shown in figure 1.6.

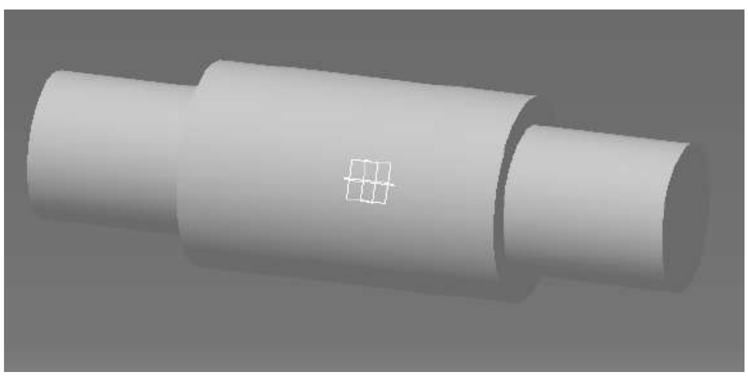


Figure 1.6

1.17 Select the YZ Plane (Figure 1.7) and enter the Sketcher Work Bench.

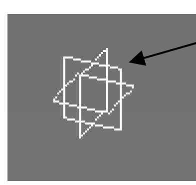


Figure 1.7

1.18 Create a circle at the coordinates(0,0) with a diameter of **25mm**(Figure1.2)
1.19 Exit the **Sketcher Work Bench** and select the **Pad** tool.
1.20 Select the More button from the Pad Definition window.
1.21 Enter "20" for both the First and Second lengths.
1.22 Select the OK button. The part should look similar the one shown in figure 1.8

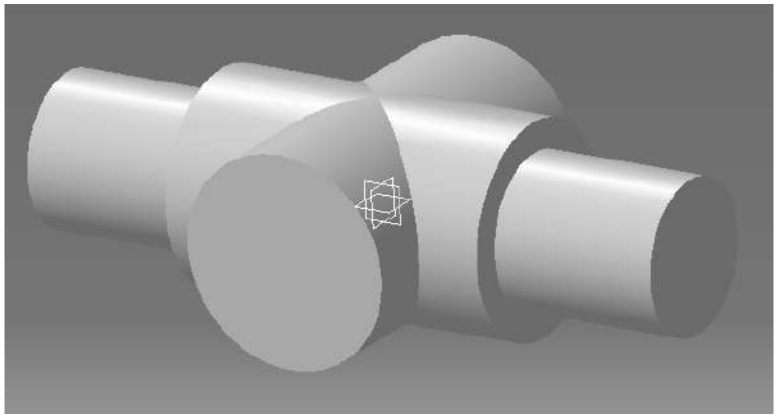


Figure 1.8

1.23 Select the **YZ Plane** reference figure 1.7
1.24 Enter the **Sketcher Work Bench.**
1.25 Create a circle at the coordinates (0,0) with a diameter of **18mm**, as shown in figure 1.5
1.26 Exit the **Sketcher Work Bench**.
1.27 Extrude the part using the **Pad** tool and **More** button. Enter **"40"** mm for both the First and Second Lengths.
1.28 Select the **OK** button. At this point your part should look similar to the one shown in Figure 1.9.

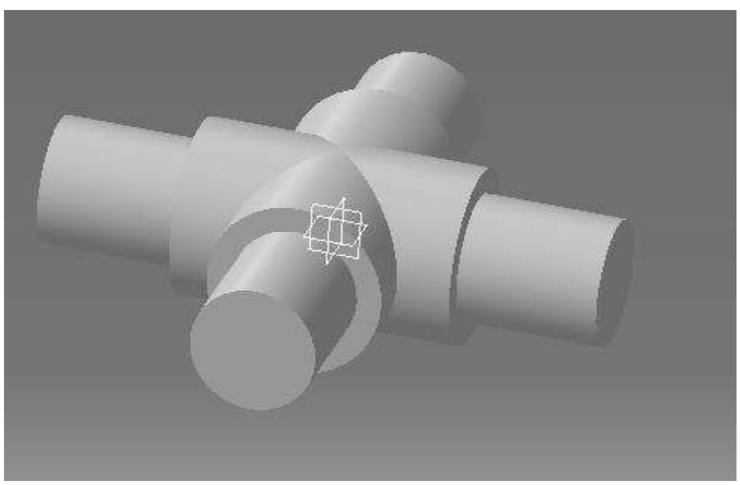


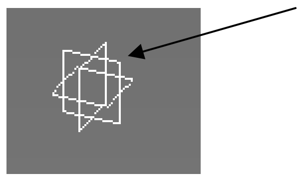
Figure 1.9

Save the part as "Swivel.CATPart".

TUTORIAL - 2

Creating The "Top U-Joint" Using Multiple Sketches

2.1 The **"Top U-Joint"** part is a new and completely separate part from the **"Swivel"** part you just created. Since the **"Top U-Joint**" part is a new part, you will need togo to the file option in the top left pull down menu. Select New.
2.2 Selecting the **Part** option automatically puts you in the **Part Design Work Bench**.
2.3 Select the **YZ Plane**, shown in Figure 2.1



YZ Plane

Figure 2.1

2.4 Enter the **Sketcher Work Bench.**
2.5 Sketch the Profile shown in Figure 2.2
2.6 **Constrain** the profile as shown in Figure 2.2

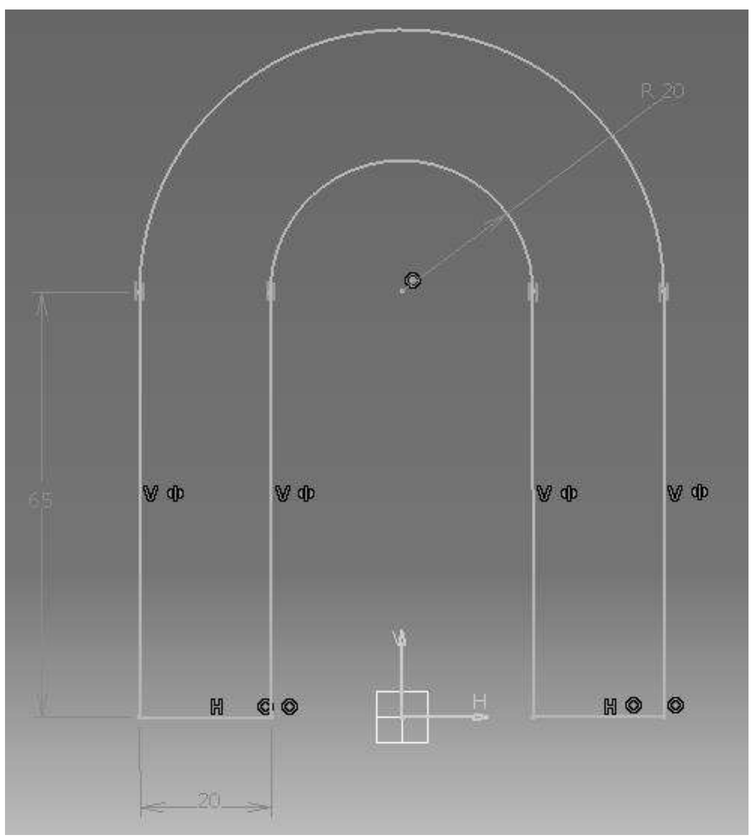


Figure 2.2

2.7 Exit the **Sketcher Work Bench**

2.8 Use the **Pad** tool to extrude the profile 40mm. Your part should look similar to the one shown in figure 2.3

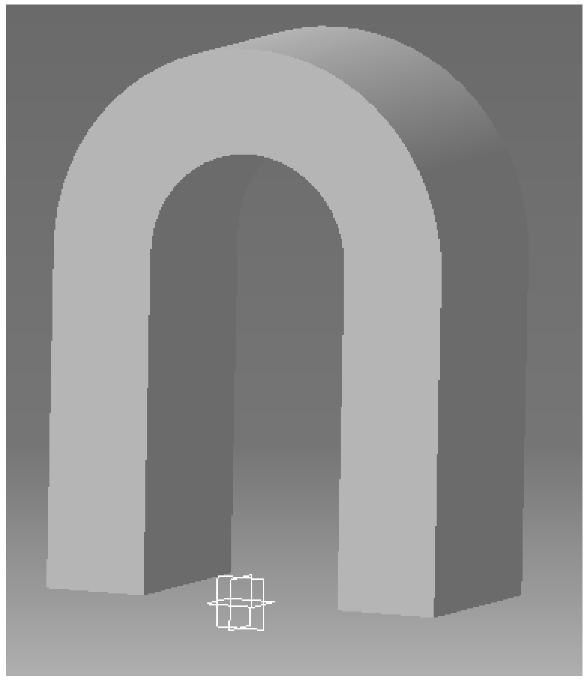


Figure 2.3

2.9 The next step is to create a 40mm diameter rounded edge on the bottom of both legs. This process can simplified by using the **Tritangent Fillet tool**. By default the Tritangent Fillet tool will be a sub option under the Edge Fillet tool .

Select the **Tritangent Fillet tool**.

2.10 Selecting the **Tritangent Fillet** tool will bring up the **Tritangent Fillet Definition** window (Figure 2.4). The first box is the **Faces To Fillet** box. This box allows you to select two faces to be joined with a fillet. The second box is **Face To Remove** box. This box allows you to select the face that will be removed and replaced with the fillet.

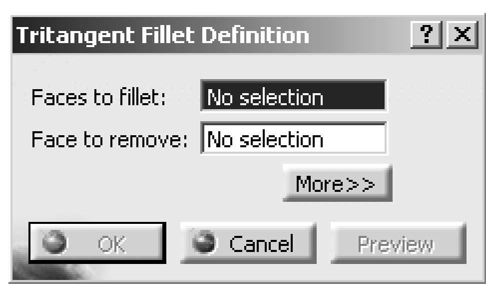


Figure 2.4

2.11 Select the front and back surfaces as the **Faces to Fillet** on the **"Top U-Joint",** as shown in Figure 2.5

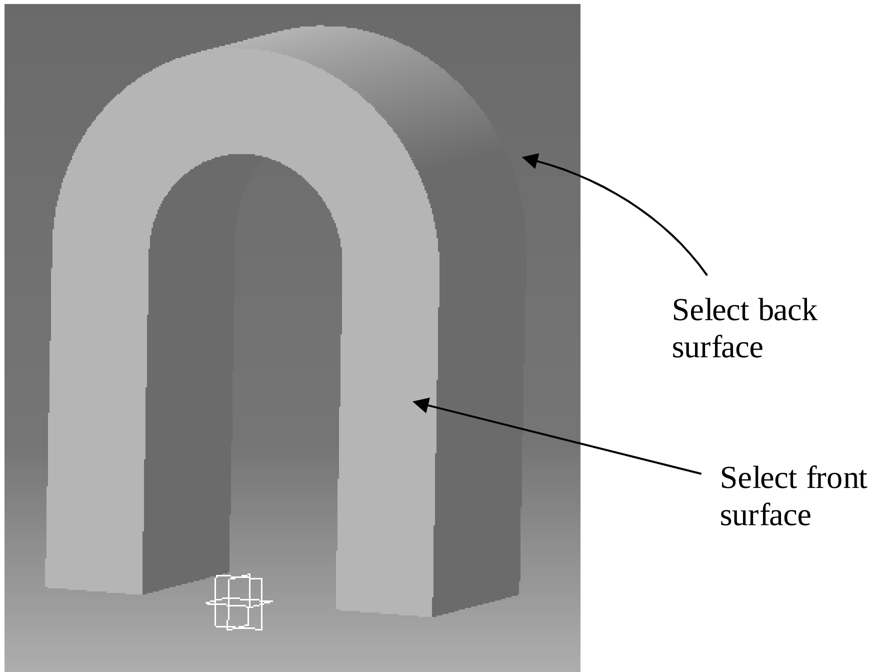


Figure 2.5

2.12 Select the bottom surface of the right leg. This surface joins the front and back surfaces. It is the **Face to Remove**, as shown in figure 2.6

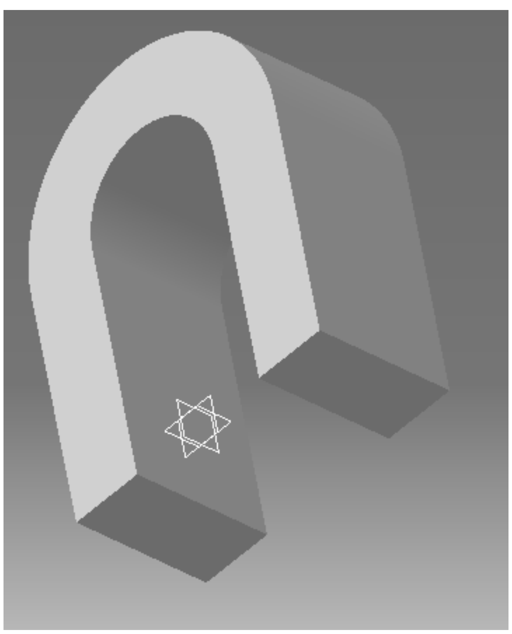


Figure 2.6

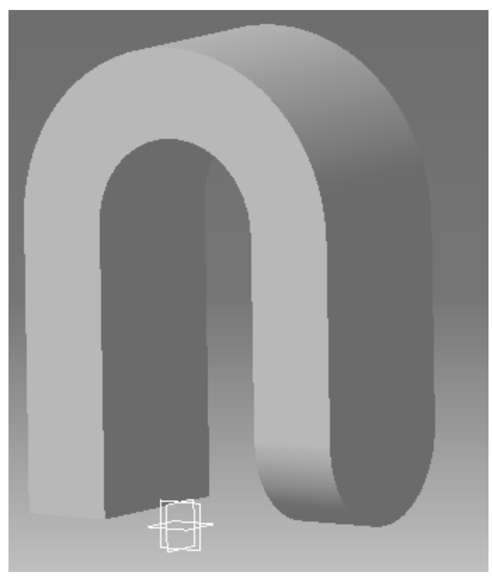


Figure 2.7

2.13 Select the **OK** button. The bottom surface selected will be removed and replaced with a radius. The radius will be the same size as the length of the surface it replaced, reference Figure 2.7.

2.14Repeat steps 2.9 thru 2.13 to create the radius on the other leg. With the radii created on both legs, the part should look similar to the one shown in figure 2.8

2.15To put the 18mm diameter hole in thetwo leg of the part, multi select the edge and surface shown in figure 2.8

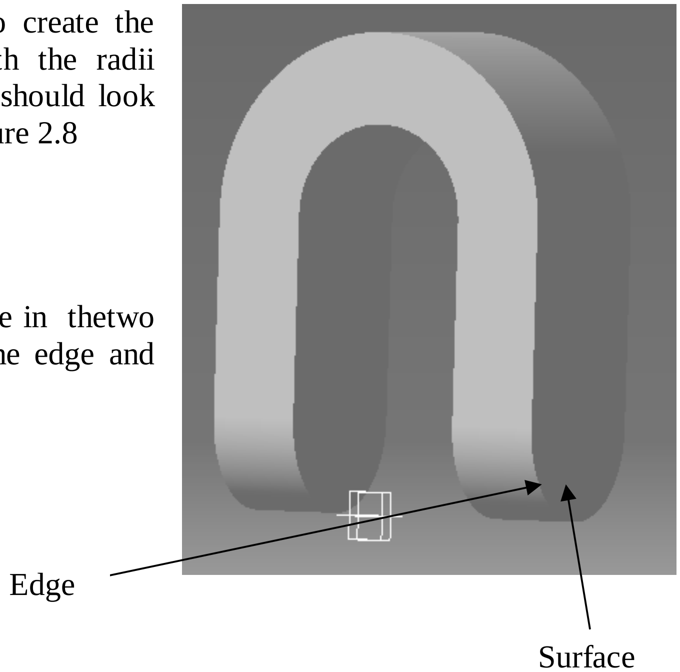


Figure 2.8

2.16Select the **Hole** tool. When the **Hole Definition** Window appears, enter **"18 mm"** for the **Diamete**r box and select "Up To Last" as the Hole Type. This will create the Hole in both legs at the same time. Select OK to create the hole. Your part should look similar to the one in Figure 2.9.

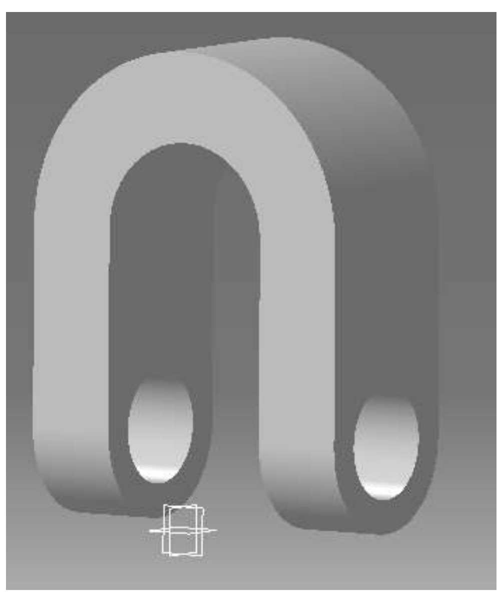
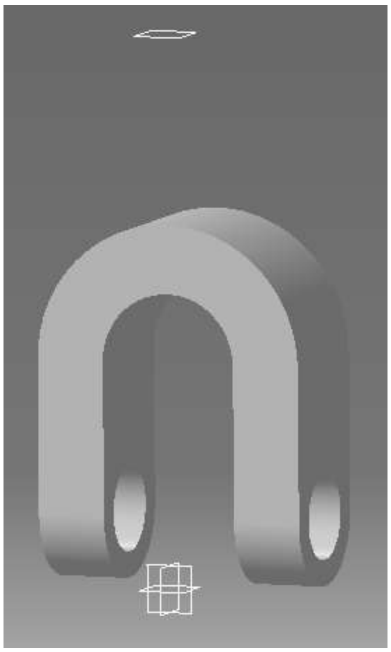


Figure 2.9

2.17The next step is to create the shaft on the top of the **“Top U-Joint**”. To accomplish this you will need to create a **Plane** that will represent the top of the shaft. This **Plane** is where you will create the sketch for the shaft.

2.18Select the **Plane** tool from the **Reference element tool bar** and with the **Plane Definition** window set **to “Offset from Plane”,** create a plane **150mm** from the **XY Plane** as shown in Figure 2.10



. **Figure 2.10**

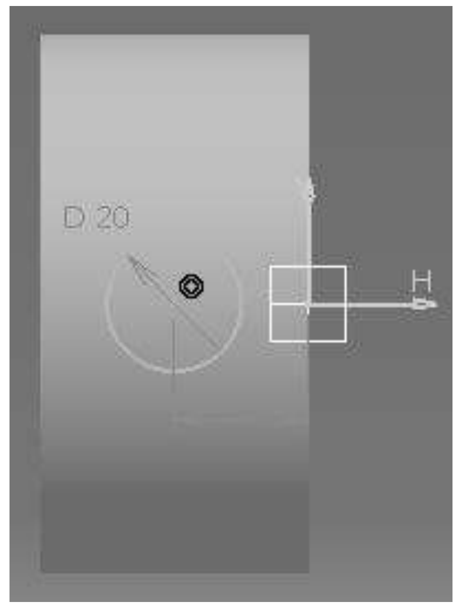


Figure 2.11

2.19 Select the new **Plane** and enter the **Sketcher Work Bench.**
2.20 Create a **Circle,** with a radius of 20mm and **Constrain** it as shown in figure 2.11
2.21Exit the **Sketcher Work Bench** and extrude the circle down to the top surface of the **“Top U-Joint”** as shown in figure 2.12

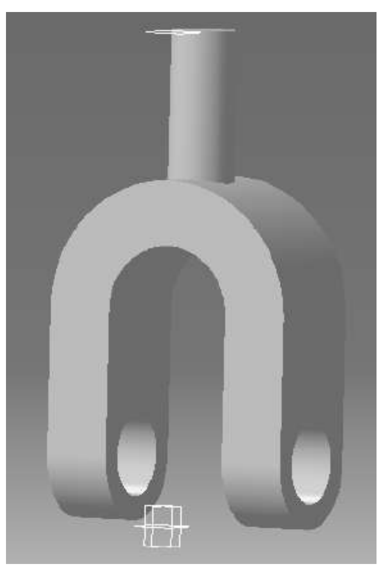


Figure 2.12

2.22 Add a **2mm Chamfer** to the top edge of the shaft as shown in figure 2.13

2.23 Add a **5mm** radius **Fillet** to all of the exterior edges of the solid as shown in Figure 2.13

2.24 The **"Top U-Joint"** has now been created. Rename **Part.1** in the **Specification Tree** to **"Top U-Joint"**

2.25 Save the part as **'Top U-Joint.CATPart."**

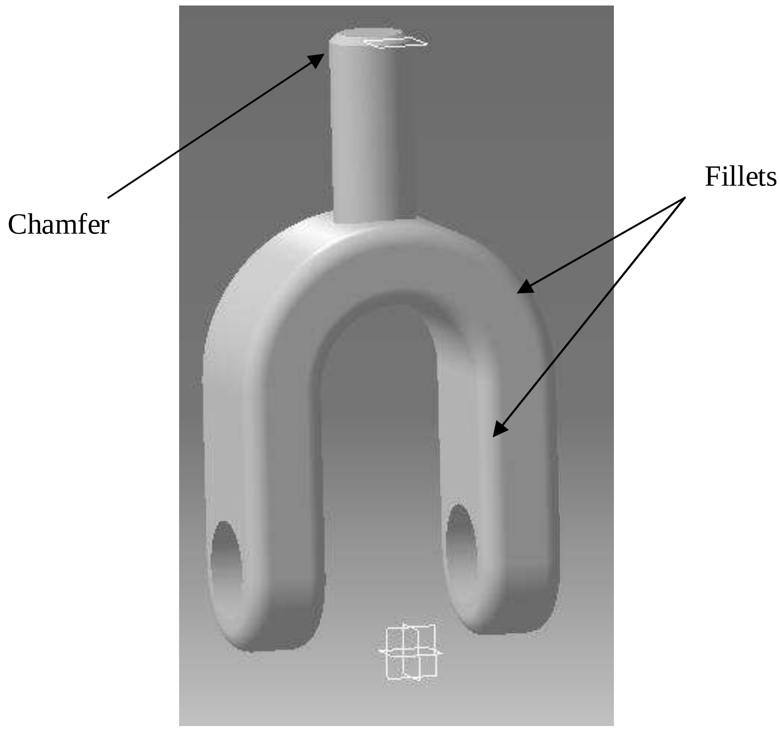


Figure 2.13

TUTORIAL – 3

Creating the "Bottom U-Joint" using Boolean Geometry.

The third and last part you will create in this lesion is the Bottom U-Joint; The Bottom U-Joint is identical to the Top U-Joint. In industry you would create the Bottom U-Joint as efficiently as possible, which would be by duplicating the Top U-Joint. Since it is this books objective to show you how to step will use another method of creating the Bottom U-Joint. At the moment this may not be the most efficient method, but it will help you be a more efficient and knowledgeable CATIA V5 user. The method still has solids using Boolean geometry. The following instructions step you through the process of creating the Bottom U-Joint using Boolean Operations.

3.1 Start a new part. rename it to Bottom U-Joint.
3.2 Select the ZX Plane and enter the Sketcher Work bench.
3.3 Create a rectangle of 80mm x 85mm in the Sketcher Work Bench .
3.4 Exit the Sketcher Work Bench and extrude the box 40mm as shown in figure 3.1

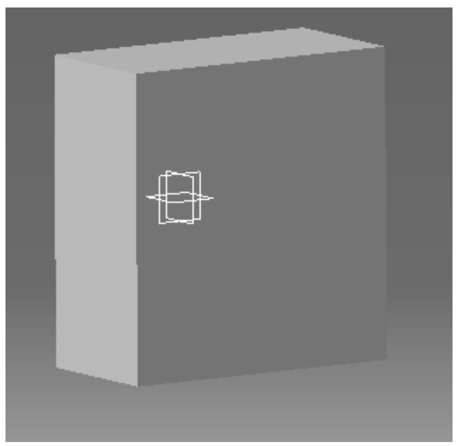


Figure 3.1

3.5 Select the Insert tab from the top pull down menu as shown in figure 3.2

3.6 From the Insert window, select the Body option. This will insert a new body into the Specification Tree (Figure 3.3). The purpose of this step will be explained later.

3.7 Select the front surface of the box as shown in figure 3.4 and enter the Sketcher Work Bench. Selecting the surface is the same as selecting a plane; the selection is where the sketch will be created.

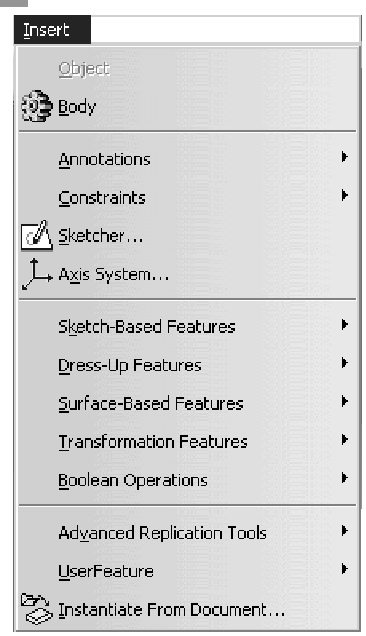
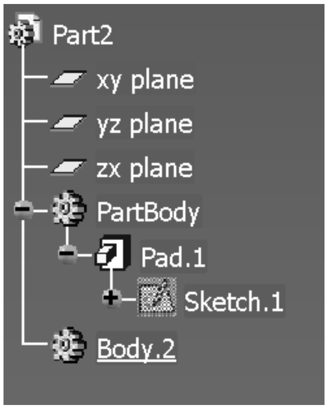
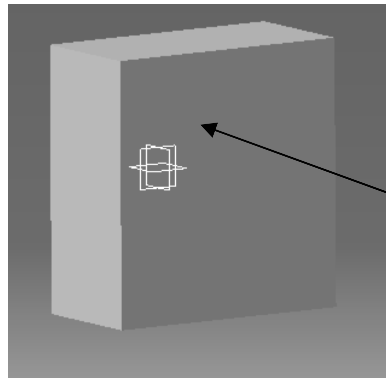


Figure 3.2

3.8 Sketch the profile shown in figure 3.5 CATIA V5 will allow you to Constrain the new sketch to the edges of the existing box (Sketch.1) as shown in figure 3.5





Select this face

Figure 3.3 **Figure 3.4**

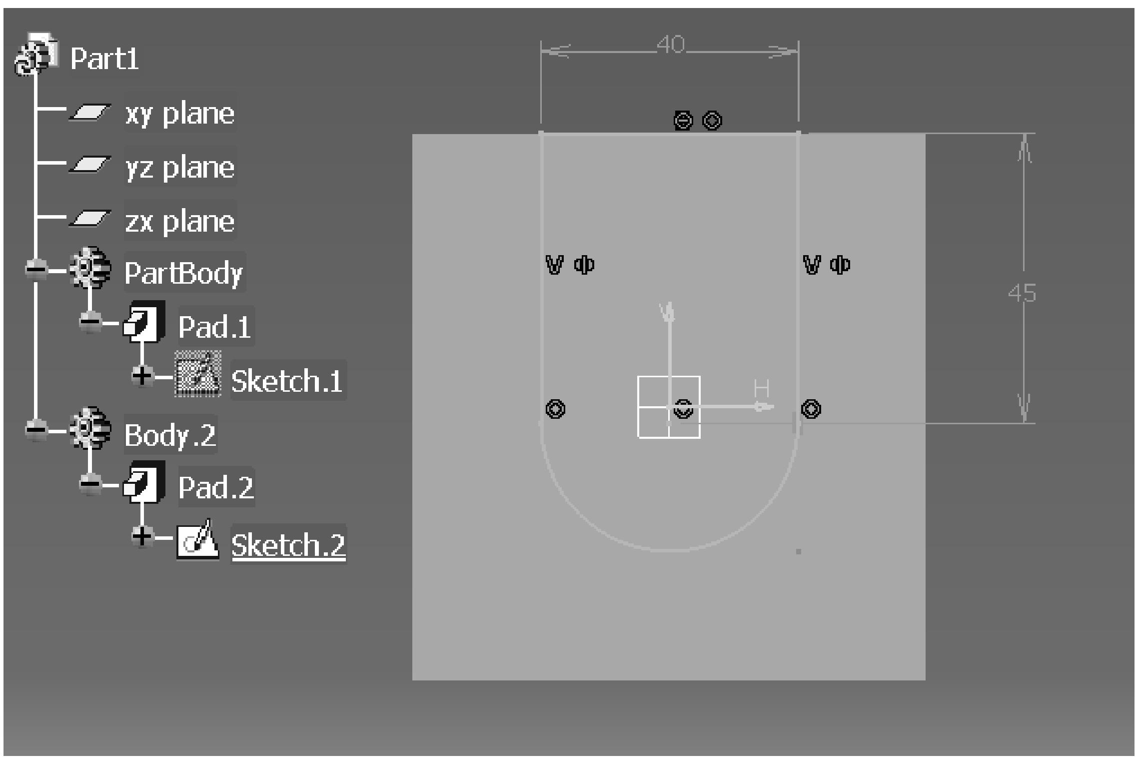


Figure 3.5

3.9 Exit the Sketcher Work Bench and select the Pad tool.

3.10 In the Pad Definition window, select the More button. Figure 3.6 shows the Pad Definition window already expanded.

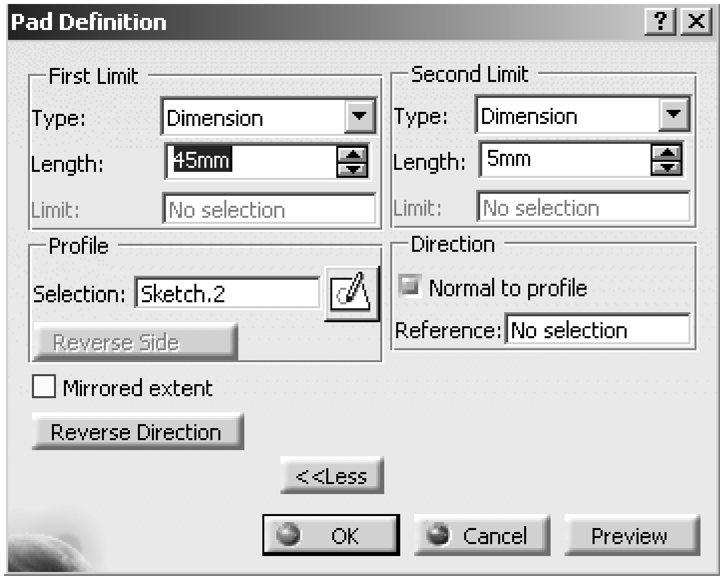


Figure 3.6

3.11In the First Limit box, for the Length, enter “45mm” as shown in figure 3.6

3.12In the Second Limit box, for the Length enter “.5mm” as shown in figure 3.6. Select the OK button. The extruded solid should look similar to the one shown in figure 3.7. if the solid was extruded in the wrong direction you may need to hit the Reverse Direction button to reverse the direction of the extrusion.

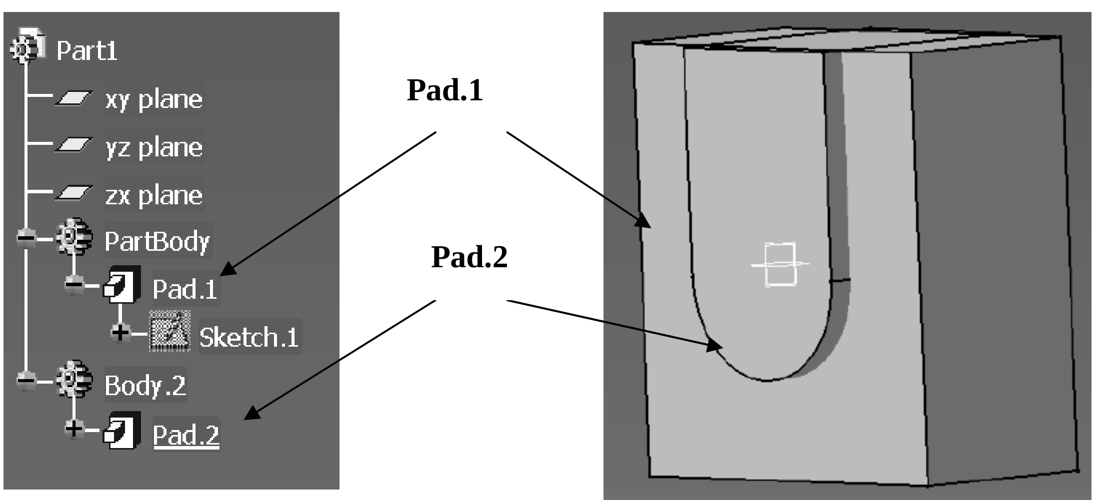


Figure 3.7

3.13Select the branch, Body.2 from the Specification Tree. This should highlight indicating it has been selected. The corresponding solid on the screen Body.2 will also highlight. If Body.2 is not selected, the following steps will not work.

3.14 Select the Edit tab from the top pull down menu as shown in figure 3.8

3.15 Select the body.2 Object from the bottom of the edit window Figure 3.8.

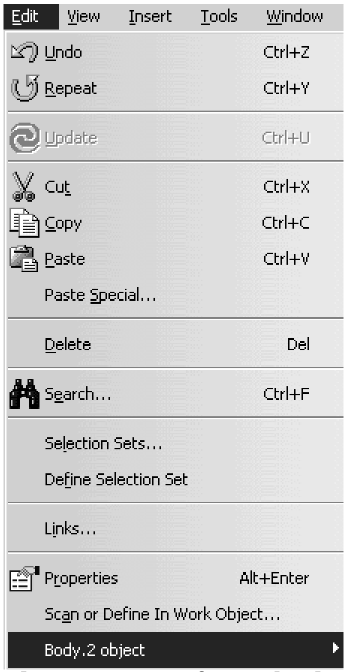


Figure 3.8

3.16Select Remove from the list of Boolean Operations as shown in figure 3.8. this will remove the second profile from the first profile. Reference figure 3.9. selecting the Insert, Boolean Operation, Remove option will give you the same result.

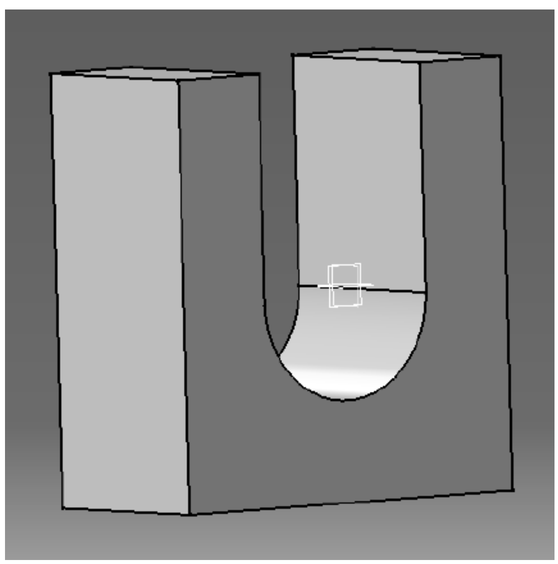


Figure 3.9

3.17Create a 40mm radius Fillet on the bottom two edges of the par. Reference figure 3.10

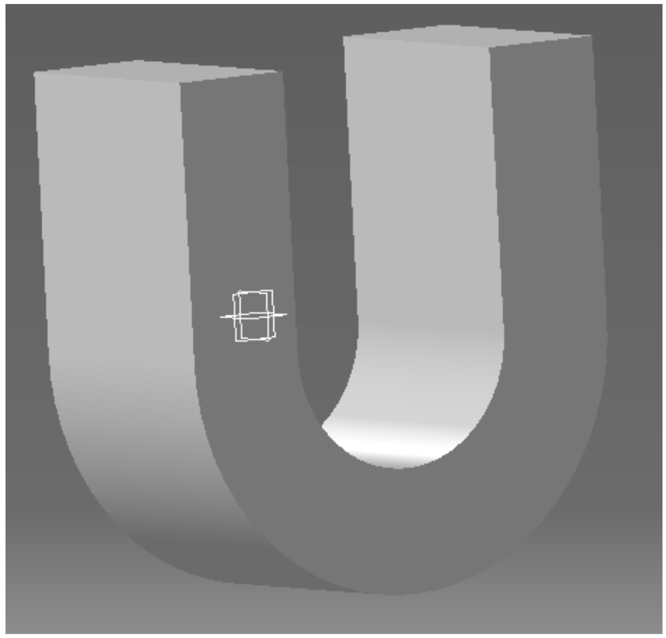


Figure 3.10

3.18 By using step 2.9 thru 2.23 then your can construct part, which will be like fig 3.11.

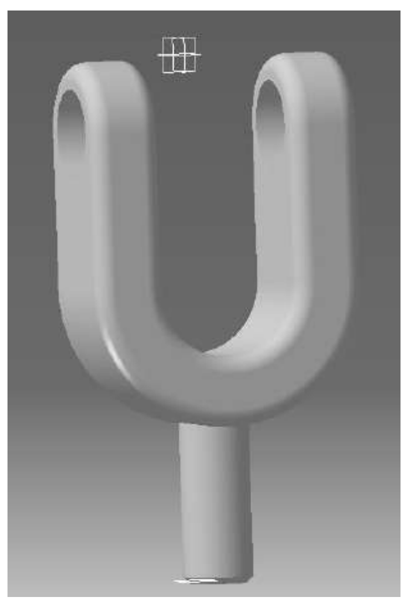
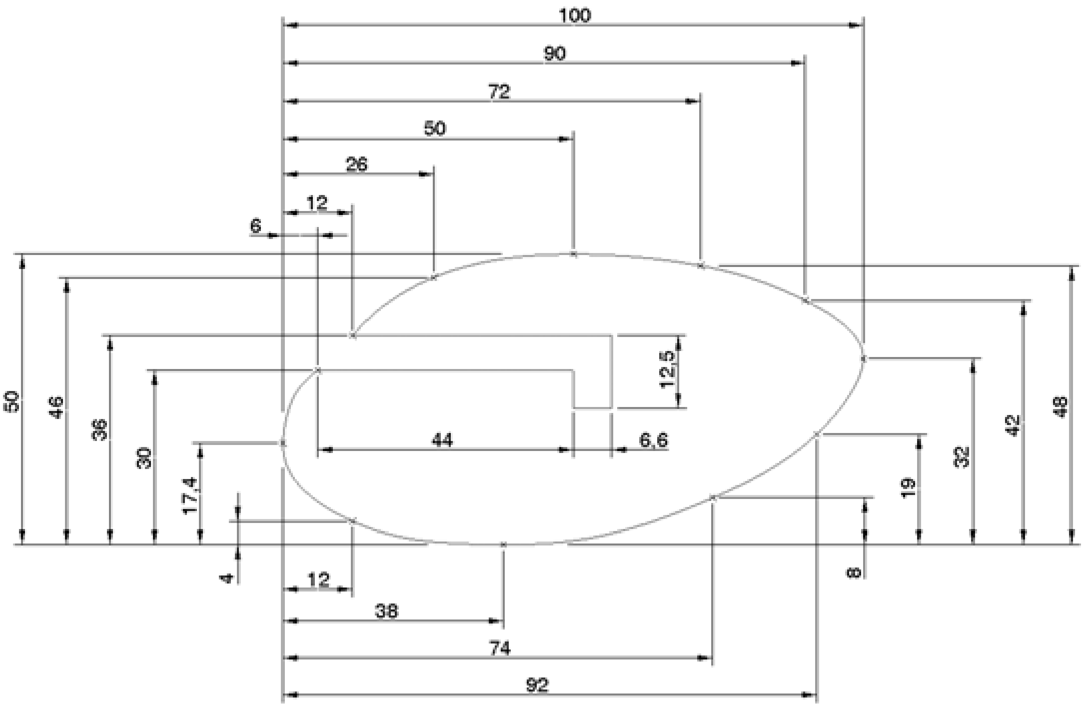


Figure 3.11

3.19If your part looks similar the one shown in figure 3.11, you are ready to save your newly created CATPart. Save the part as Bottom U-Joint.CATOPart.


100
90
72
50
26
12
6
50
46
36
30
17,4
4
12,5
44
6,6
12
38
74
92
48
42
32
19
8

EX. NO. 1 **2D Geometry- Splines**

DATE:

AIM:

Preparation of 2D model using CATIA V5.18 software

TOOLS USED:

Pad, Pocketing etc.,

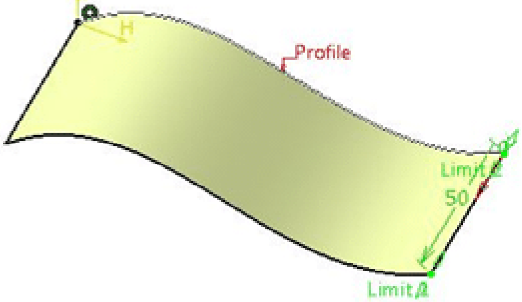
PROCEDURE:

1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench isdisplayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.

RESULT:

Thus the given 2D model as per the drawing is modelled using CATIA V5.18 software.


V
H


V
H
Profile
50

EX. NO. 2 **Surface Modeling- NURBS**

DATE:

AIM:

Preparation of surface model using CATIA V5.18 software

TOOLS USED:

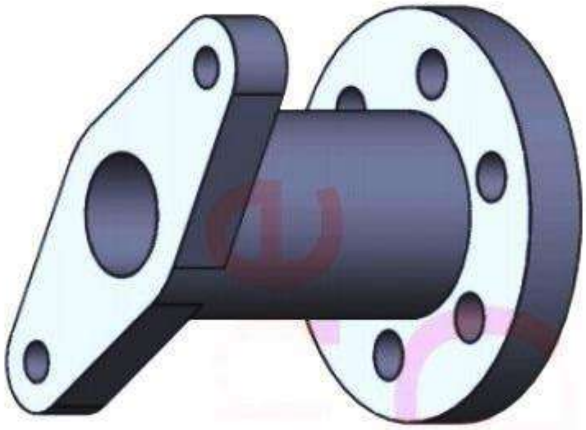
Pad, Pocketing etc.,

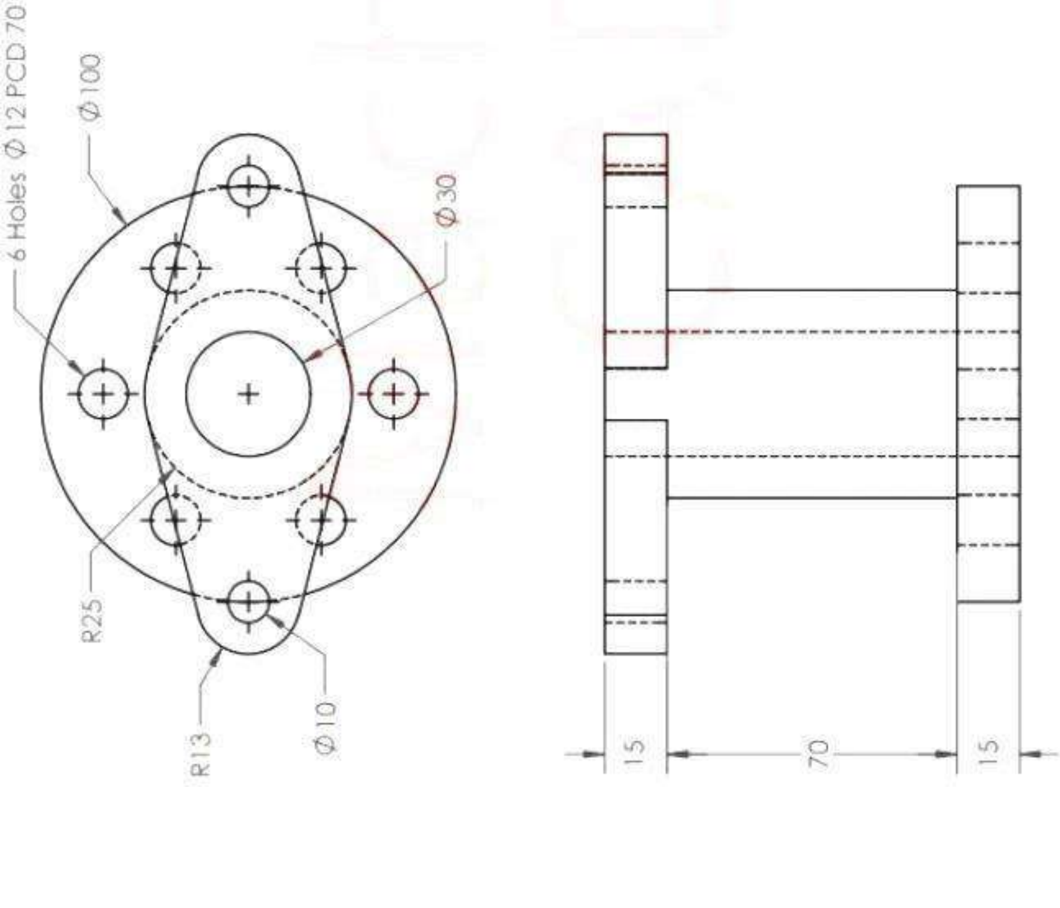
PROCEDURE:

1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench is displayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click *Pad* and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.

RESULT:

Thus the given surface model as per the drawing is modelled using CATIA V5.18 software.




6 Holes Ø12 PCD 70
Ø100
Ø30
R25
R13
Ø10
15
70
15

EX. NO. 3 **Solid Modelling**

DATE:

AIM:

Preparation of solid model using CATIA V5.18 software

TOOLS USED:

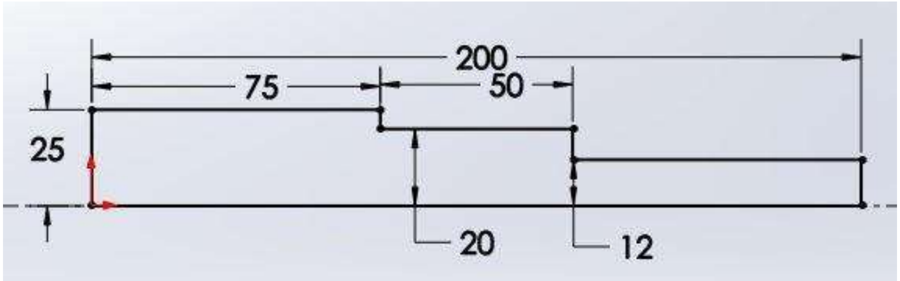
Pad, Pocketing etc.,

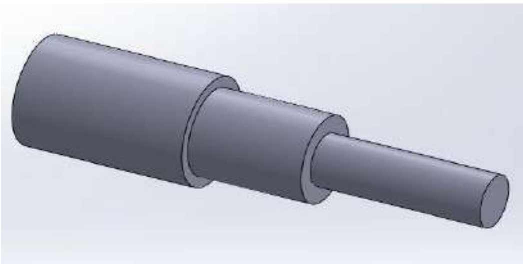
PROCEDURE:

1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench isdisplayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click *Pad* and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.
7. Using *Pocket* command material is removed and the final model is created.

RESULT:

Thus the given solid model as per the drawing is modelled using CATIA V5.18 software.


200
75
50
25
20
12



EX. NO. 4 **Preparing solid model for analysis**

DATE:

AIM:

Preparation of solid model using CATIA V5.18 software for analysis

TOOLS USED:

Pad, Pocketing etc.,

PROCEDURE:

1. Click the Sketcher icon to start the *Sketcher workbench*.
2. Select XY, YZ, ZX plane to define the sketch plane, now the Sketcher workbench is displayed, it contains the tools needed for sketching any profile.
3. Select the profile and draw the part which is given in the model.
4. Using *Constraint* command the dimensions are modified as per the given model.
5. Exit the Sketcher workbench, click *Pad* and give the thickness for the part.
6. Select the face to define the work plane and draw the second element.
7. Using *Pocket* command material is removed and the final model is created.
8. Select Save As tab from file menu and save the as IGS/ STEP file format for analysis

RESULT:

Thus the given 3D model as per the drawing is modelled using CATIA V5.18 software.

STUDY OF ANALYSIS AND ITS BENEFITS

Some Basic Concepts

The finite element method (FEM), or finite element analysis (FEA), is based on the idea of building a complicated object with simple blocks, or, dividing a complicated object into smaller and manageable pieces.

Why FEA?

Computers have revolutionized the practice of engineering. Design of a product that used to be done by tedious hand drawings has been replaced by computer-aided design (CAD) using computer graphics. Analysis of a design used to be done by hand calculations and many of the testing have been replaced by computer simulations using computer-aided engineering (CAE) software. Together, CAD, CAE, and computer-aided manufacturing (CAM) have dramatically changed the landscape of engineering.

Among all the computational tools for CAE, the FEM is the most widely applied method or one of the most powerful modern “calculators” available for engineering students and professionals. FEA provides a way of virtually testing a product design. It helps users understand their designs and implement appropriate design changes early in the product development process. The adoption of FEA in the design cycle is driven by market pressure since it brings many benefits that will help companies make better products with reduced development costs and time-to- market.

Finite Element Applications in Engineering

The FEM can be applied in solving the mathematical models of many engineering problems, from stress analysis of truss and frame structures or complicated machines, to dynamic responsesof automobiles, trains, or airplanes under different mechanical, thermal, or electromagneticloading. There are numerous finite element applications in industries, ranging from automotive, aerospace, defense, consumer products, and industrial equipment to energy, transportation and construction, as shown by some examples in Table. The applications of the FEA have also been extended to materials science, biomedical engineering, geophysics, and many other emerging fields in recent years.

FEA with *ANSYS Workbench*

Over the last few decades, many commercial programs have become available for conducting the FEA. Among a comprehensive range of finite element simulation solutions provided by leading CAE companies, *ANSYS® Workbench* is a user-friendly platform designed to seamlessly integrate *ANSYS, Inc.*'s suite of advanced engineering simulation technology. It offers bidirectional connection to major CAD systems. The *Workbench* environment is geared toward improving productivity and ease of use among engineering teams. It has evolved as an indispensible tool for product development at a growing number of companies, finding applications in many diverse engineering fields

A General Procedure for FEA

To conduct an FEA, the following procedure is required in general:

- ❖ Divide the CAD/geometric model into pieces to create a "mesh" (a collection of elements with nodes)
- ❖ Describe the behavior of the physical quantities on each element.
- ❖ Connect (assemble) the elements at the nodes to form an approximate system of equations for the entire model.
- ❖ Apply loads and boundary conditions (e.g., to prevent the model from moving).
- ❖ Solve the system of equations involving unknown quantities at the nodes (e.g., the displacements).
- ❖ Calculate the desired quantities (e.g., strains and stresses) at elements or nodes.

In commercial FEA software, this procedure is typically rearranged into the following phases:

- ➢ Preprocessing (build FEM models, define element properties, and apply loads and constraints)
- ➢ FEA solver (assemble and solve the FEM system of equations, calculate element results)
- ➢ Postprocessing (sort and display the results)

Overview of *ANSYS Workbench*

ANSYS Workbench is a simulation platform that enables users to model and solve a wide range of engineering problems using the FEA. It provides access to the *ANSYS* family of design and analysis modules in an integrated simulation environment. This section gives a brief overview of the different elements in the *ANSYS Workbench* simulation environment or the graphical-user interface (GUI). Readers are referred to *ANSYS Workbench* user's guide for more detailedinformation.

The User Interface

The *Workbench* interface is composed primarily of a *Toolbox* region and a *Project Schematic* region. The main use of the two regions is described next.

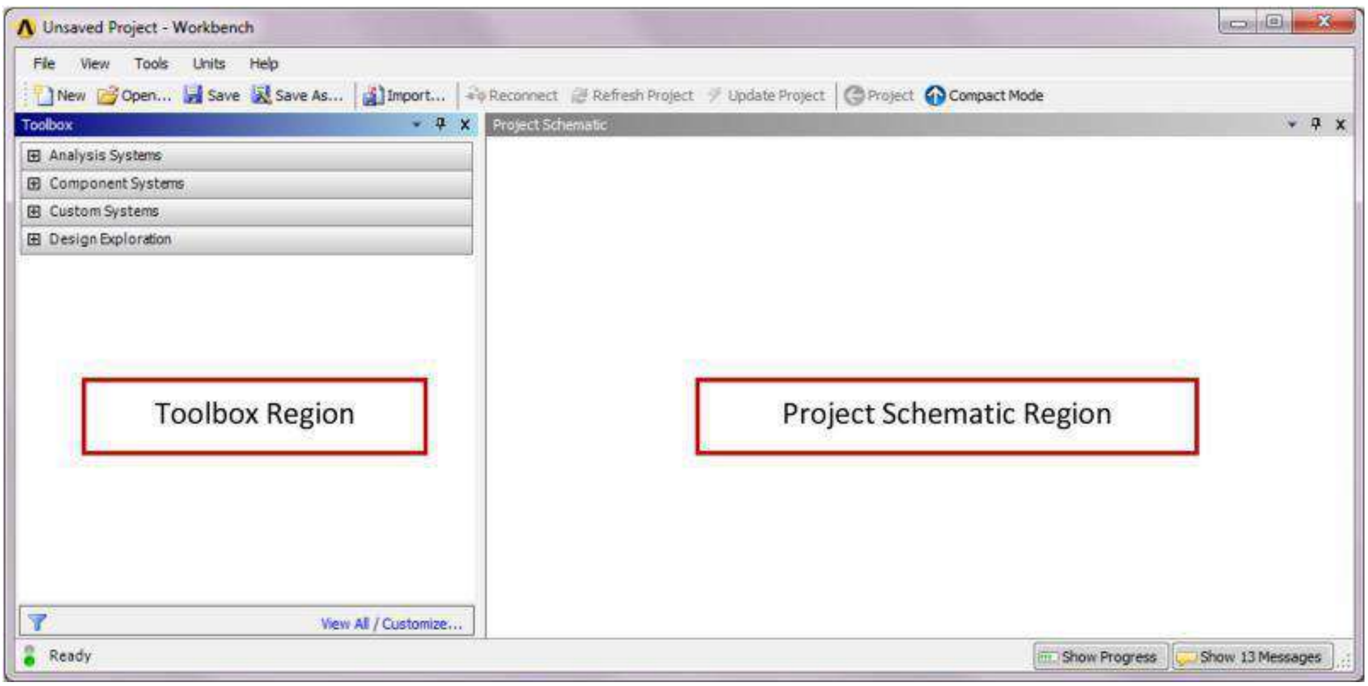


***Figure**: ANSYS Workbench* user interface.

The Toolbox

The *Toolbox* contains the following four groups of systems:

***Analysis Systems**:* Predefined analysis templates to be used to build your project, including static structural, steady-state thermal, transient thermal, fluid flow, modal, shape optimization, linear buckling, and many others.

***Component Systems**:* Component applications that can be used to build or expand an analysis system, including geometry import, engineering data, mesh, postprocessing, and others.

***Custom Systems**:* Coupled-field analysis systems such as fluid solid interaction, prestress modal, thermal-stress, and others.

***Design Exploration**:* Parametric optimization studies such as response surface optimization, parameters correlation, six sigma analysis, and others.

The Project Schematic

A project schematic, that is, a graphical representation of the workflow, can be built by dragging predefined analysis templates or other components from the *Toolbox* and dropping them into the *Project Schematic* window. “Drag” here means to move the mouse while holding down the left mouse button, and “drop” means to release the mouse button.

To build a project for static structural analysis, for instance, drag the *Static Structural* template from the *Toolbox* and drop it into the rectangular box that appears in the *Project Schematic* window. A standalone analysis system that contains the components needed for static structural analysis is added to the project schematic as shown in Fig. The system consists of sevenindividual components called cells.

Alternatively, a standalone analysis can be created by double-clicking. For example, double- click the *Steady-State Thermal* template from the *Toolbox*, and an independent

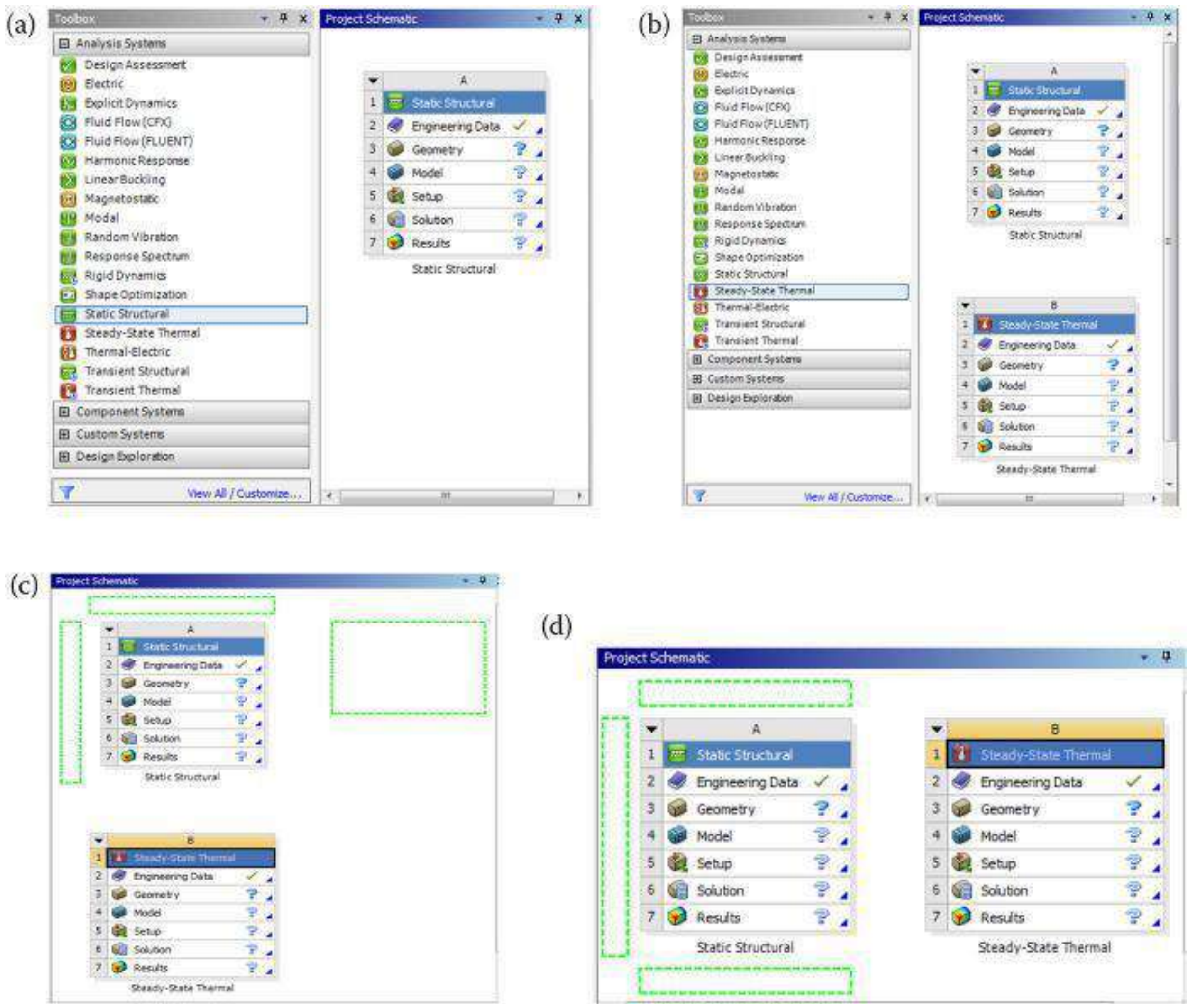


Figure: *Defining standalone analysis systems in the project schematic: (a) a standalone system; (b) two independent standalone systems; (c) moving a system in a top-bottom configuration; and (d) moving a system in a side-byside configuration.*

Steady-State Thermal system will be placed in the default location below the existing *Static Structural* system.

A system can be moved around another system in the project schematic. To move a system, click on the header cell (i.e., the cell titled *Steady-State Thermal* for the thermal system) and drag it to a new place. Once you drag the header cell, dashed rectangles appear for the possible new locations to drop the system. This is illustrated in Fig c and d for two systems with initial top– bottom and side-by-side configurations, respectively.

To delete a system, click on the down arrow button at the upper left corner of the system from the *Project Schematic* window, and then choose *Delete* from the drop-down context menu.

In somecases, a project may contain two or more analysis systems that share data. For example, a downstream modal analysis may use the same material, geometry, and model data from the preceding structural analysis. To build such a project, create a standalone system for *Static Structural* analysis. Then, drag the *Modal* analysis template from the *Toolbox* and drop it onto the *Model* cell of the *Static Structural* system. Immediately before the subsequent system is dropped, bounding boxes will appear on the *Engineering Data*, *Geometry,* and *Model* cells of the first system, as shown in Fig a. After the system is released, a project including two linked systems is created, as shown in Fig b, where the linked cells indicate data sharing at the *Model* and above levels.

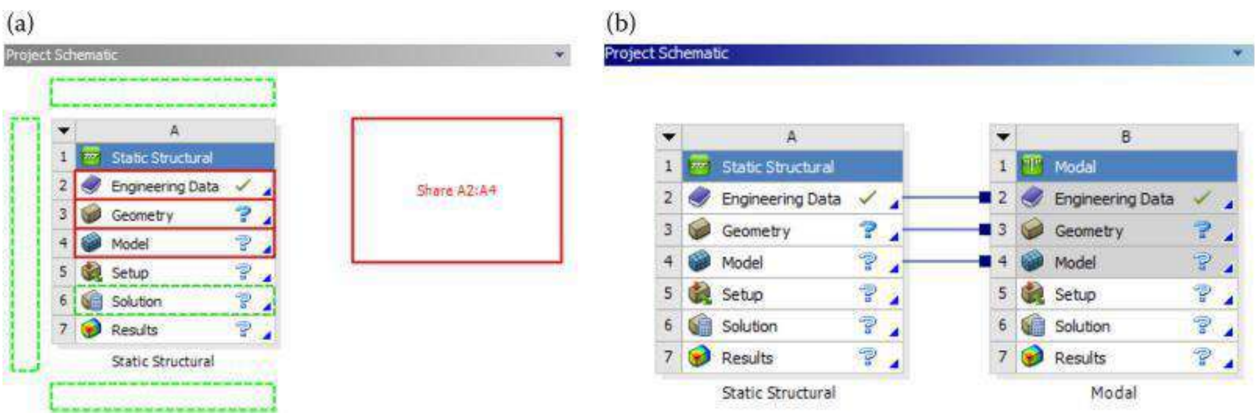


Figure: *Defining linked analysis systems in the project schematic: (a) dropping the second (subsequent) system onto the Model cell of the first system to share data at the model and above levels; (b) two systems that are linked.*

Working with Cells

Cells are components that make up an analysis system. You may launch an application by double-clicking a cell. To initiate an action other than the default action, right-click on a cell to view its context menu options. The following list comprises the types of cells available in *ANSYS Workbench* and their intended functions:

Engineering Data: Define or edit material models to be used in an analysis.

Geometry*:* Create, import, or edit the geometry model used for analysis.

***Model/Mesh**:* Assign material, define coordinate system, and generate mesh for the model.

***Setup**:* Apply loads, boundary conditions, and configure the analysis settings.

***Solution**:* Access the model solution or share solution data with other downstream systems.

***Results**:* Indicate the results availability and status (also referred to as postprocessing).

As the data flows through a system, a cell's state can quickly change. *ANSYS Workbench* provides a state indicator icon placed on the right side of the cell. Table describes the indicator icons and the various cell states available in *ANSYS Workbench*. For more information, please refer to *ANSYS Workbench* user's guide.

The Menu Bar

The menu bar is the horizontal bar anchored at the top of the *Workbench* user interface. Itprovides access to the following functions:

***File Menu**:* Create a new project, open an existing project, save the current project, and so on.

View Menu**:* Control the window/workspace layout, customize the toolbox, and so on.Tools Menu**:* Update the project and set the license preferences and other user options.***Units Menu**:* Select the unit system and specify unit display options.

***Help Menu**:* Get help for *ANSYS Workbench*.

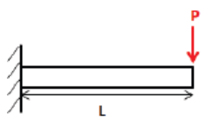
Indicator Icons and Descriptions of the Various Cell States

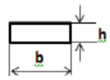
Cell State	Indicator	Description
Unfulfilled		Need upstream data to proceed
Refresh required		A refresh action is needed as a result of changes made on upstream data
Attention required		User interaction with the cell is needed to proceed
Update required		An update action is needed as a result of changes made on upstream data
Up to date		Data are up to date and no attention is required
Input changes pending		An update or refresh action is needed to recalculate based on changes made to upstream cells
Interrupted		Solution has been interrupted. A resume or update action will make the solver continue from the interrupted point
Pending		Solution is in progress

Ex. No :5 **STRESS ANALYSIS OF CANTILEVER BEAM**

Date :

Problem Description





Nomenclature:

L =110m	Length of beam
b =10m	Cross Section Base
h =1 m	Cross Section Height
w=20N/m	Distributed Load
E=70GPa	Young's Modulus of Aluminum at Room Temperature
ν=0.33	Poisson's Ratio of Aluminum

This is a simple, single load step, structural analysis of a cantilever beam. The left side of the cantilever beam is fixed while there is a distributed load of 20N/m. The objective of this problem is to demonstrate a simple ANSYS Workbench problem with a textbook solution: finding Von Mises' stresses and total deflection throughout the beam. The beam theory for this analysis is shown below:

Theory

Von Mises Stress

Assuming plane stress, the Von Mises Equivalent Stress can be expressed as:

$$\sigma' = (\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}}$$

Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

$(\tau_{xy} = 0,\ \sigma_y = 0)$.

Using these simplifications, the Von Mises Equivalent Stress from equation 1 reduces to:

$$\sigma' = \sigma_x$$

Bending Stress is given by:

$$\sigma_x = \frac{P(x-L)c}{I}$$

Where $I = \frac{1}{12}bh^3$ and $c = \frac{h}{2}$. From statics, we can derive:

$\sigma_x = \frac{6P(x-L)}{bh^2} = 66kPa$

Beam Deflection

As in module 1.1, the beam equation to be solved is:

$$\frac{d^2y}{dx^2} = \frac{M(x)}{EI}$$

Using Shigley's Mechanical Engineering Design, the beam deflection is:

$$\delta(x) = \frac{Px^2(x-3L)}{6EI}$$

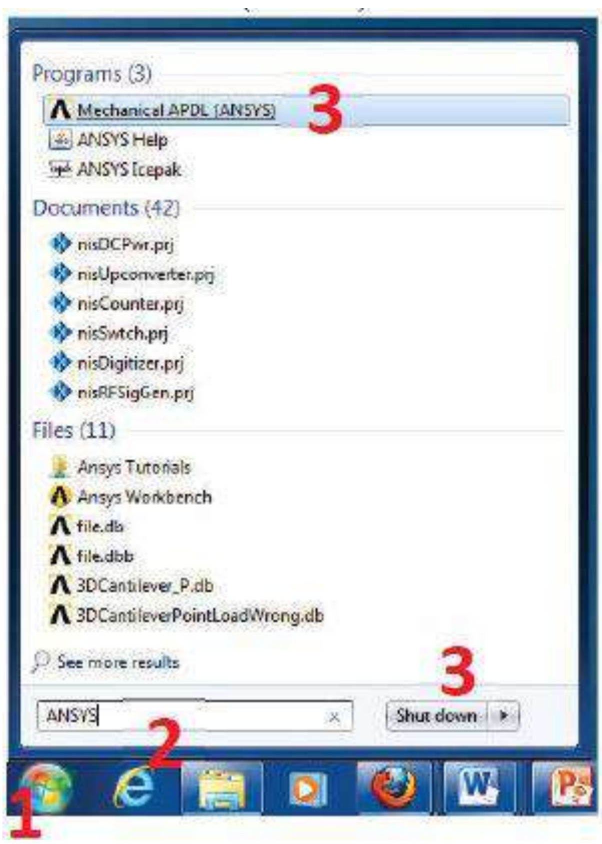
With Maximum Deflection at:

$$\boldsymbol{\delta = \frac{PL^3}{3EI} = 7.61mm}$$

Workbench Analysis System

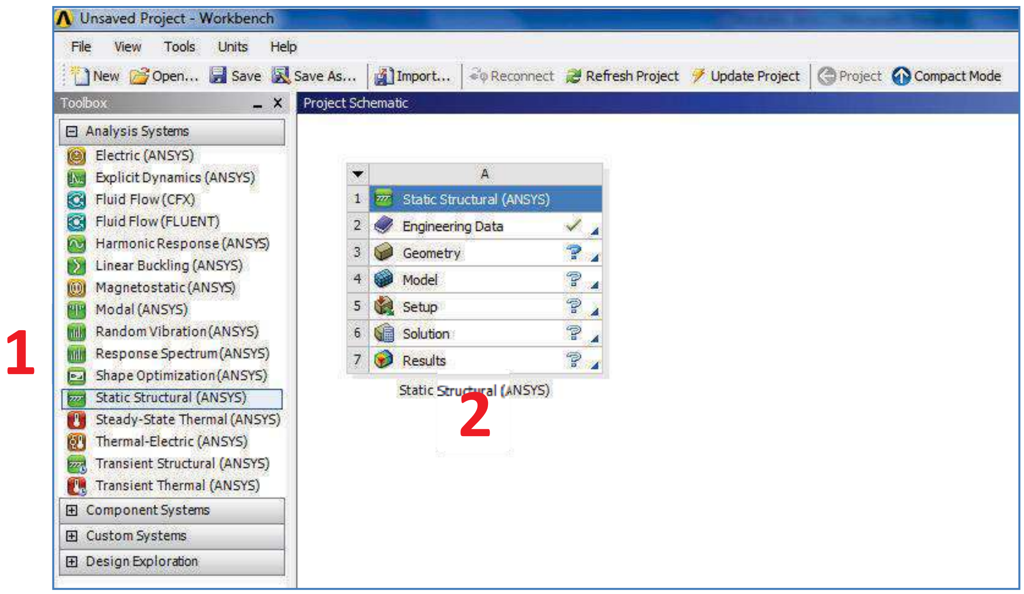
Opening Workbench

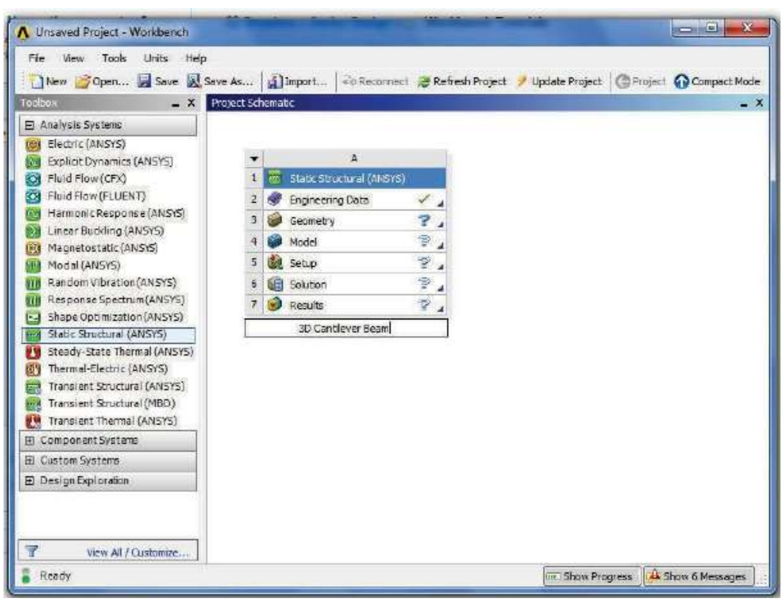
1. On your Windows Desktop click the Start button.
2. Under Search Programs and Files type "ANSYS"
3. Click on ANSYS Workbench to start workbench. This step may take time.



Static Structural Analysis

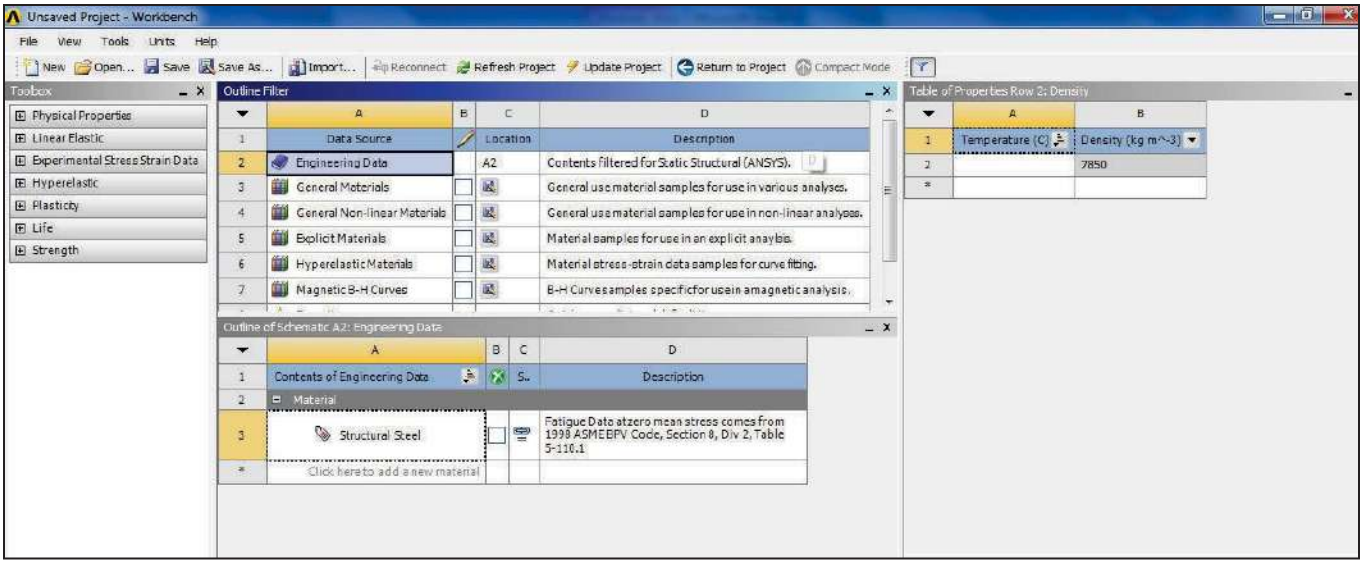
1. As you open ANSYS you can see the entire array of problems on the left had side this software can help you solve. The problem at hand is a *Static Structural* problem. Double click **Static Structural (ANSYS)** to open the task manager for your problem set in the Project Schematic area.
2. ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see **Static Structural (ANSYS),** double click this to change the name. For this problem choose "*3D Cantilever beam*."



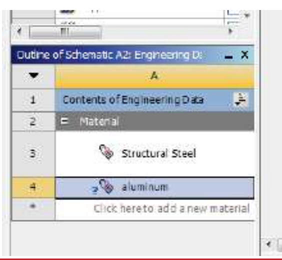


Engineering Data

To begin setup for your cantilever beam, double click or right click on ***Engineering Data*** and click ***edit***. This will bring up another screen.



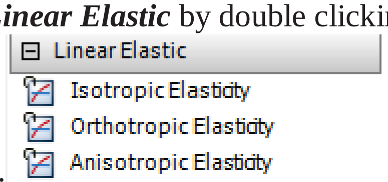
This new window will allow you to alter the material properties of your cantilever beam. Under **<u>Outline of Schematic A2: Engineering Data</u>**, it shows ***click here to add a new material***, this menu allows you to input the material of your cantilever beam, double click and type ***Aluminum***.



<u>WARNING</u> Do not delete or change the Structural Steel, just another material.

Now expand ***Linear Elastic*** by double clicking on 

or on the plus symbol shown.

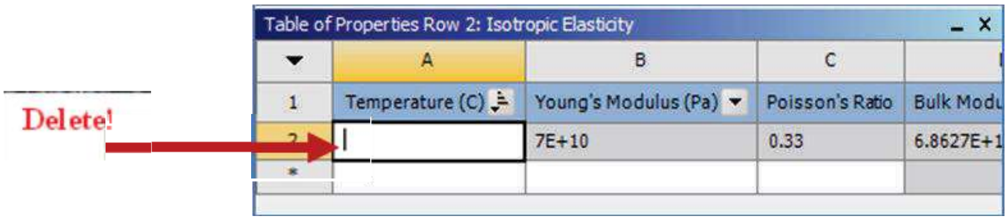


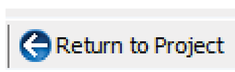
Double click on Isotropic Elasticity to give the material the same properties across the beam. This action brought up a new table on the right; this allows us to add necessary properties. As show on the top right of the screen in ***Table of Properties Row 2: Isotropic Elasticity***:

1. Click in Temperature and type 25
2. Click in Young's Modulus and type 70E9 or 7E10
3. Click in Poisson's Ratio and type 0.33

	A	B	C
1	Temperature (C)	Young's Modulus (Pa)	Poisson's Ratio
2	25	7E+10	0.33
*			

WARNING Make sure to DELETE the Temperature entry after property input before continuing! Failure to do so will lead to errors later

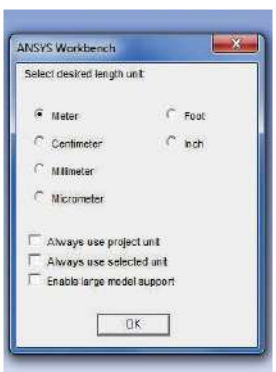


After filling in the properties, this concludes the Engineering Data, to return to the project schematic area, click on  seen on the upper tab.

Geometry

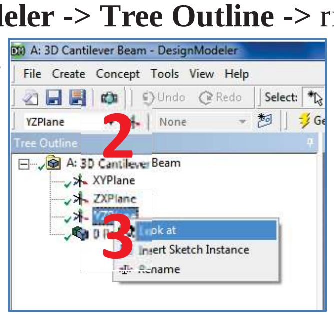
Base Geometry

1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for *ANSYS Design Modeler* where the Geometry will be created.

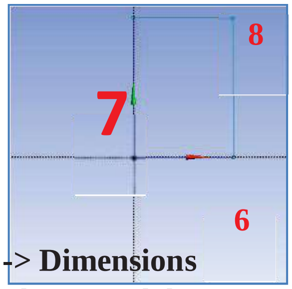


Note: Select meters and hit ok

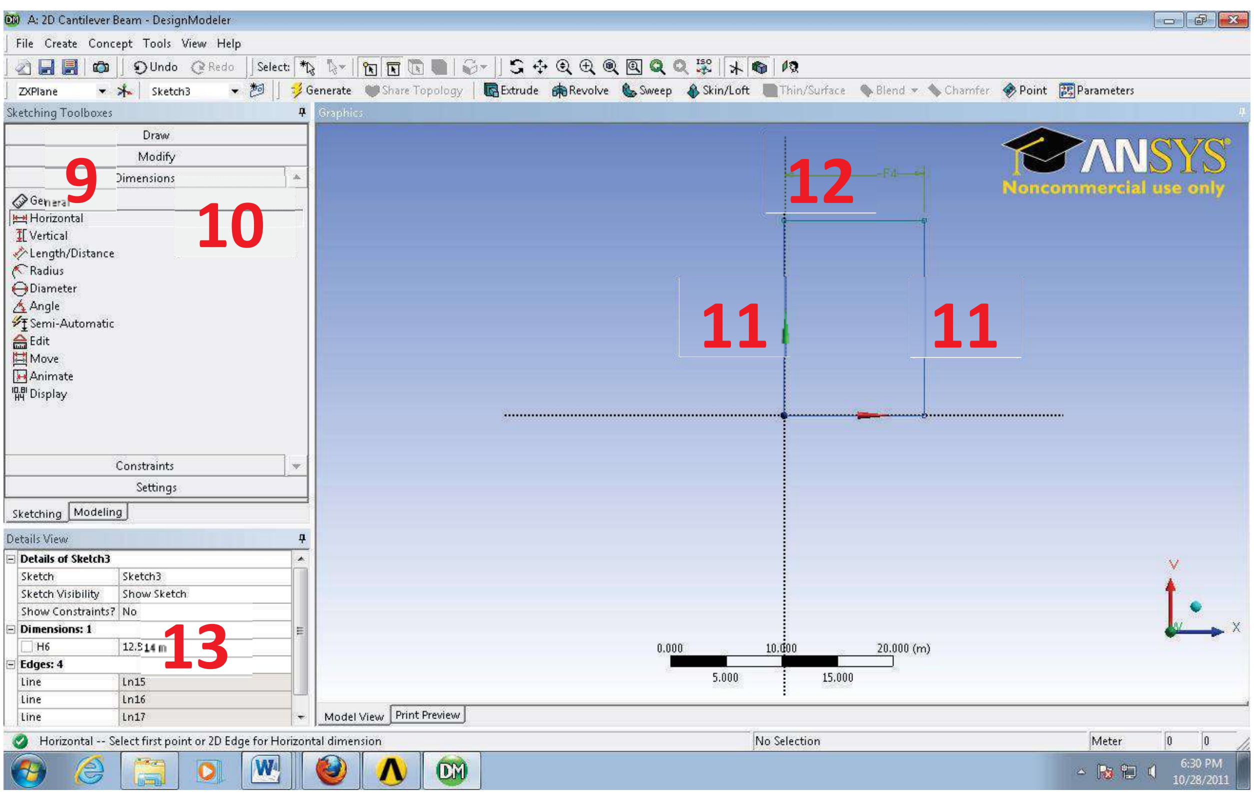
2. In the new window, click the  **Display Plane** icon to toggle the coordinate system.
3. Go to **Design Modeler -> Tree Outline ->** right click on YZPlane. Click **Look At** to view the YZ plane.



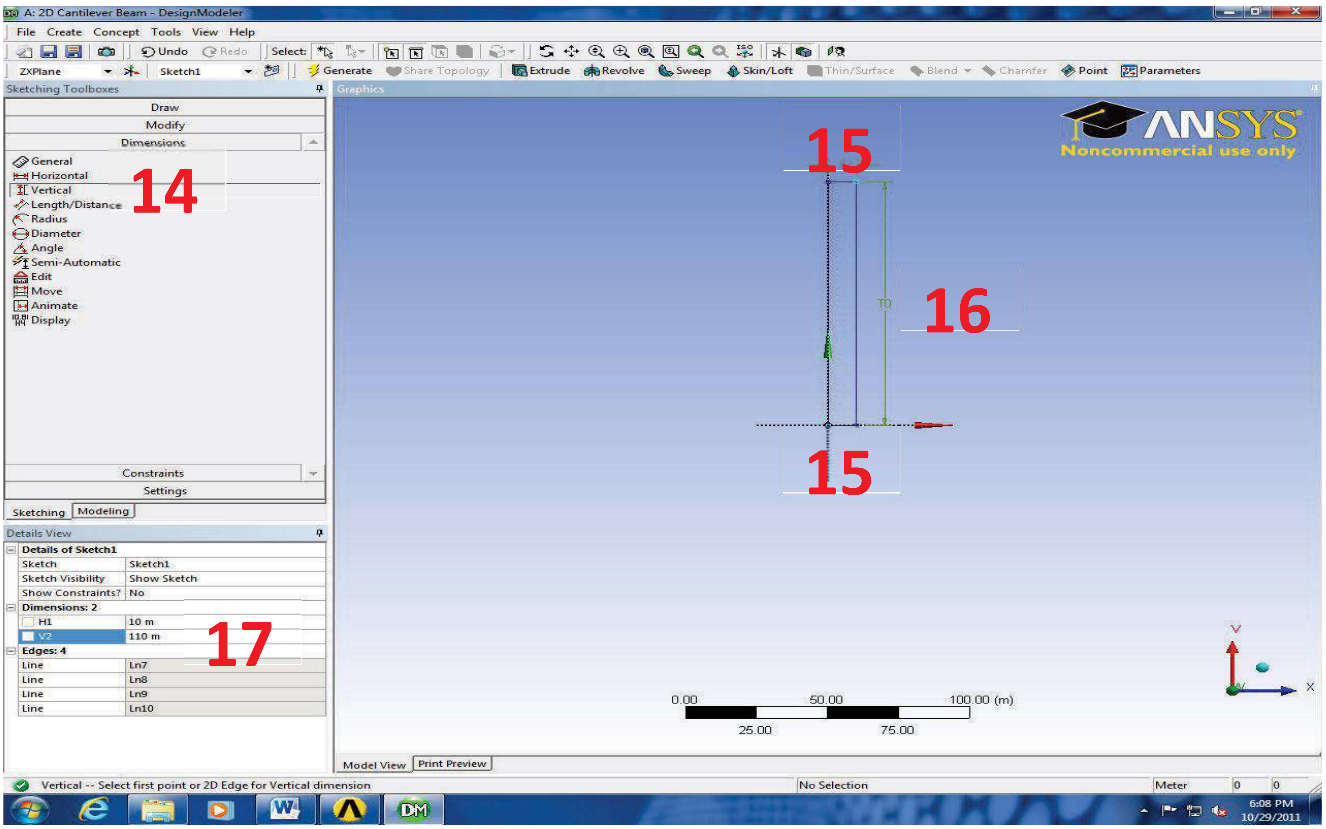
4. Go to **Design Modeler -> Tree Outline -> Sketching**
5. Click on **Rectangle** and Click off **Auto-Fillet:**
6. Bring your cursor into the workspace at point 0,0, over the origin until 'P' appears directly above the origin.
7. Click on the origin to place the lower left corner of our rectangle on the origin.
8. Click on a point in the first quadrant to define the top right corner of our rectangle. The point is arbitrary as we will be fixing dimensions momentarily.



9. Go to **Sketching Toolboxes -> Dimensions**
10. Click **Horizontal** to specify a horizontal dimension.
11. Click the left and right faces of the rectangle in the sketch to specify that we will be dimensioning this horizontal length. A green line with a symbol should appear.
12. Drag the green line above the sketch and click to set its location.
13. Go to **Detail View -> Dimension 1.** In the first subcategory, replace the current dimension with 10. The units should populate automatically.



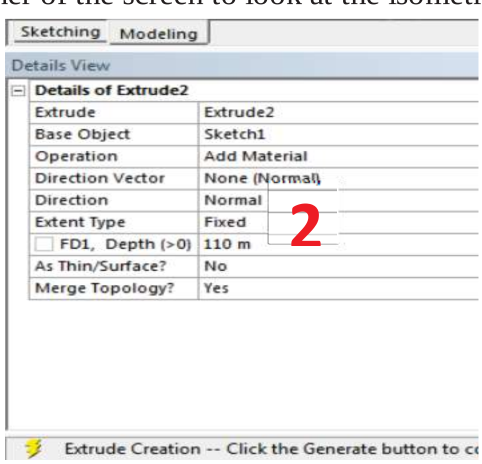
14. Go to **Sketching Toolboxes -> Dimensions -> Vertical** to specify the vertical dimension.
15. Click the bottom and top faces of the sketch to specify the vertical dimension. A green line should appear.
16. Drag the green line to the right of the sketch and click.
17. Go to **Detail View -> Dimension 2**. Replace the value with 10. The units should populate automatically (meters).



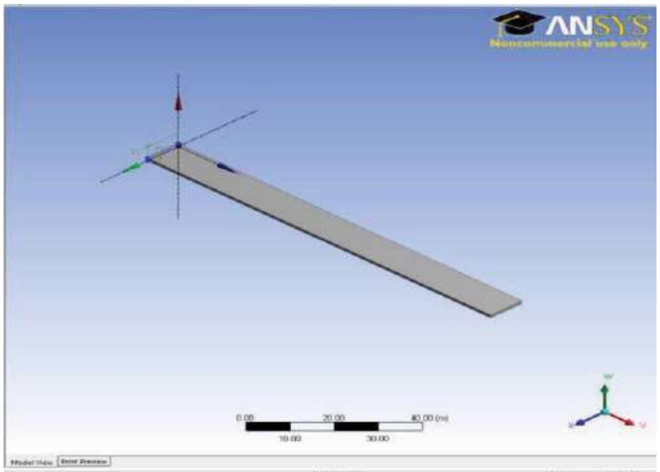
Now that we have modeled the base geometry, we will extrude it to create a 3D volume.

<u>Extrude Sketch</u>

1. Go to **Main Toolbar ->** and select **Extrude**
2. Go to **Modeling -> FD1, Depth (>0) ->** enter in **110**
3. Go to **Design Modeler ->** **Generate**.
4. To verify our geometry, look at the isometric view. Click the *blue dot* in the **triad** in the lower right corner of the screen to look at the isometric view.



Your 3D surface should look like this:

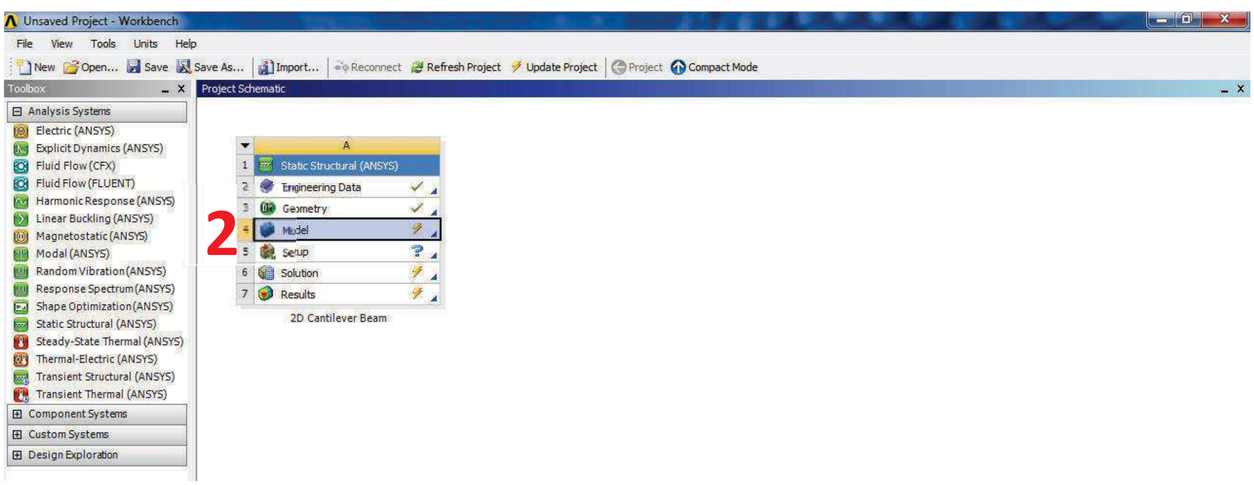


Now that we have the geometry, we will mesh the beam using 3D Elements.

<u>Model</u>

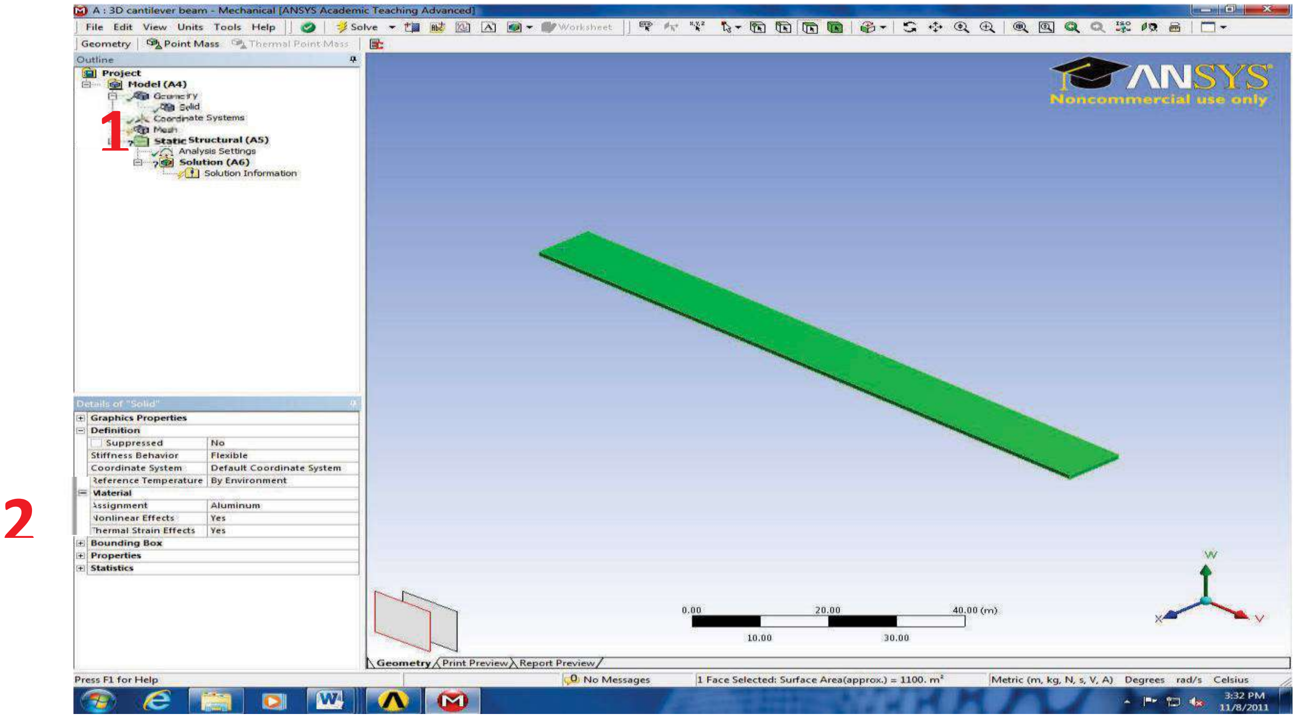
<u>Open ANSYS Mechanical</u>

1.  out of Design Modeler. Don't worry, your work will be saved.
2. Go to **Workbench -> Project Schematic -> Model** This will open *ANSYS Mechanical*



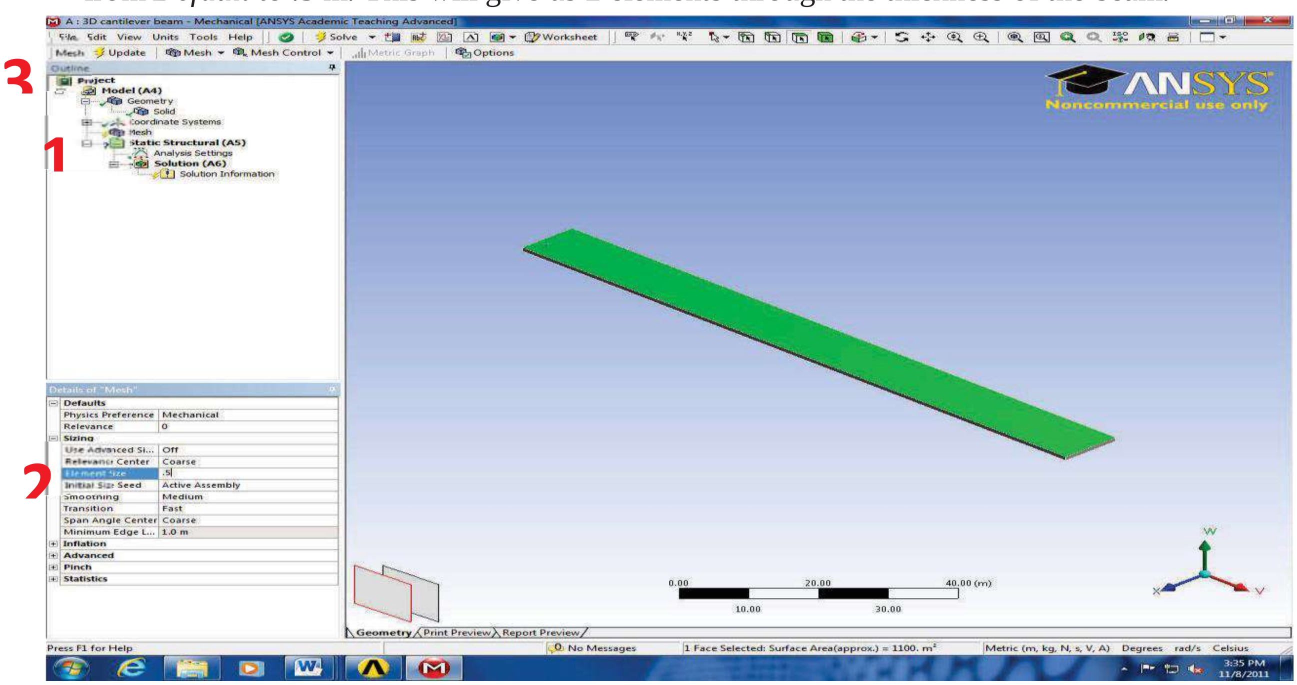
Material Assignment

1. Go to **Mechanical -> Outline -> Project -> Model -> Geometry -> Surface Body**
2. Under **Mechanical -> Details of "Surface Body" -> Material -> Assignment**, change *Structural Steel* to *Aluminum.*

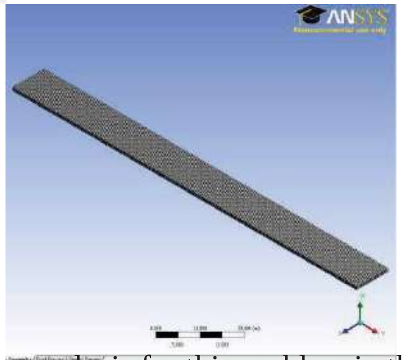


Mesh

1. Go to **Mechanical -> Outline -> Project -> Model -> Mesh**
2. Go to **Mechanical -> Details of 'Mesh' -> Sizing -> Element Size** and change the value from *Default* to .5 m. This will give us 2 elements through the thickness of the beam.



3. Click **Mechanical ->**  **Update**. This may take some time. Your mesh should look as shown below:

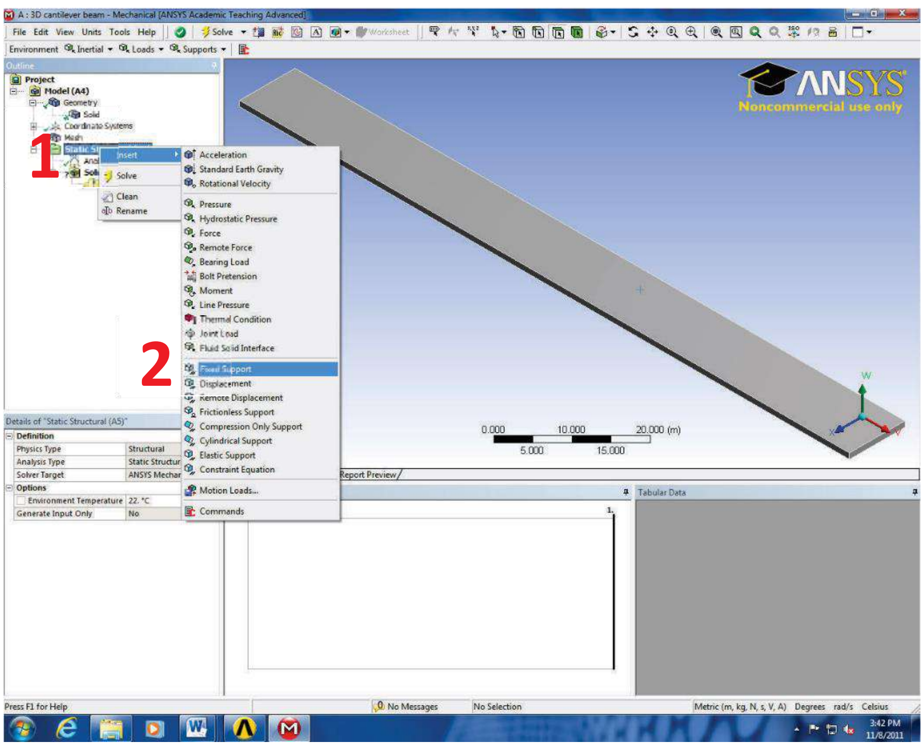


Setup

You can perform the rest of your analysis for this problem in the *ANSYS Mechanical* window. The other options in the *Workbench* window will link you back to the same screen (i.e. Setup, Solution, Results)

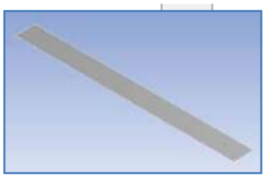
Fixed Support

1. Go to **Mechanical -> Outline ->** right click **Static Structural (A5)**
2. Go to **Insert -> Fixed Support**



 We are going to fix the elements at the left end of the beam. In order to do this, we will use the **Edge** tool to select the left edge. However, from the current orientation of the beam, it is difficult to select this surface.

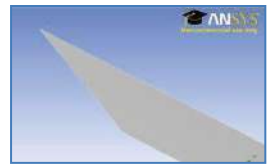
3. Using the  **Rotate** tool click on the graphic area and move the mouse to the right. This will cause the left end of the beam to be oriented in a manner that can be clicked



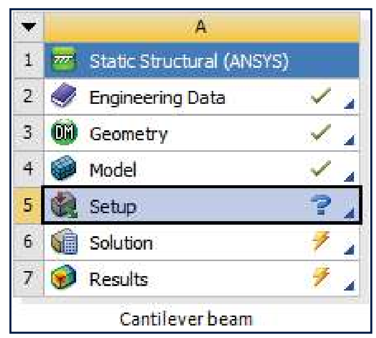
Rotate

4. Using the **Pan** tool, click the graphic area and drag the left face to the center of the graphic window. Use the *mouse scroll* to zoom in on the left face

Pan



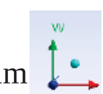
5. Click the **Edge** tool.
6. Go to **Mechanical -> Outline -> Static Structural (A5) -> Fixed Support**
7. Run the cursor across the left end face. When it becomes red, click it to select it.
8. Go to **Mechanical -> Details of "Fixed Support" -> Geometry** and select **Apply**

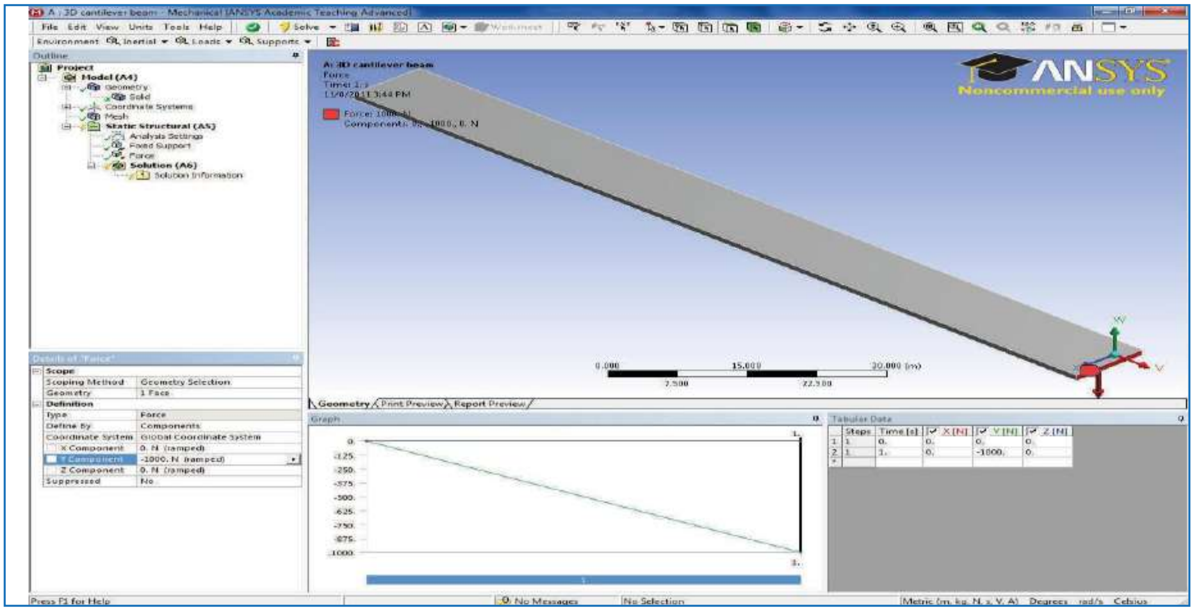


<u>Setup</u>

While in the Project Schematic double click Setup
This will , en a new window similar to Model Space

<u>Loads</u>

1. Click the x-axis icon to get a side view of the cantilever beam 
2. Click Fixed end On the tool bar, make sure vertex option is selected.
3. Go to **Mechanical -> Outline -> Static Structural (A5) -> Fixed Support**
4. Run the cursor across the left end face. When it becomes red, click it to select it.
5. Click the left side of the geometry; this will add a green box to select the point.
6. Right click Static Structural (A5)
7. Click insert, and Fixed Support
8. This will add a fixed end to your cantilever beam in the work space.
9. Point Load On the tool bar, change selection option to Edge: edge instead of vertex.
10. Click on the geometry, this will highlight
11. Right click Static Structural (A5)
12. click insert , and A table will appear "Details of Line Pressure" Line Pressure
13. Under "Definition" you will see "Defined by"→Change this to "Components"
14. As shown, Y Component force is zero. → Change this to value to -20
15. This will show your cantilever beam with a load applied as shown. Leave the Setup screen open this time.

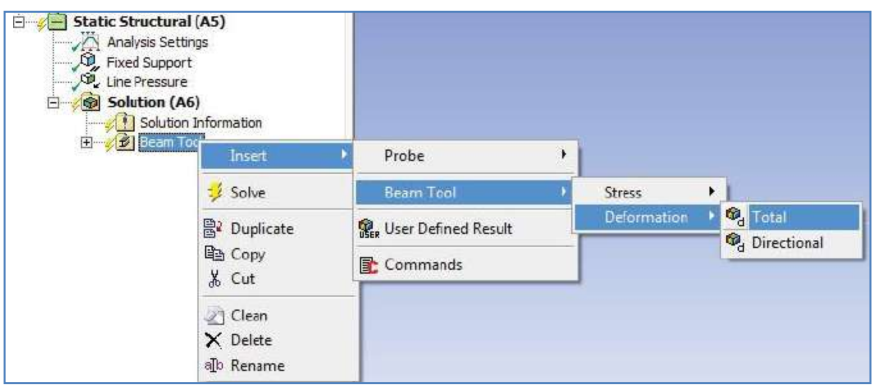


<u>Solution</u>

Go to **Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) ->** Right Click **Solution (A6) -> Insert -> Beam Tool**

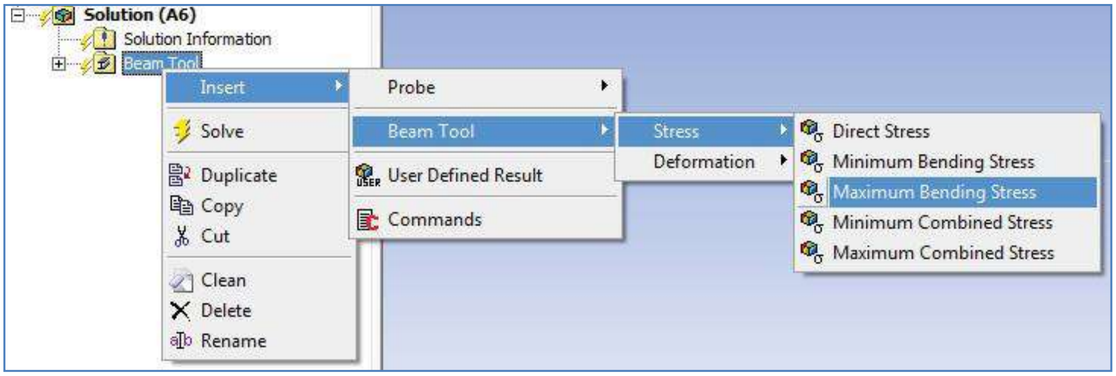
<u>Deformation</u>

Go to **Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Beam Tool -> Insert -> Beam Tool -> Deformation -> Total**



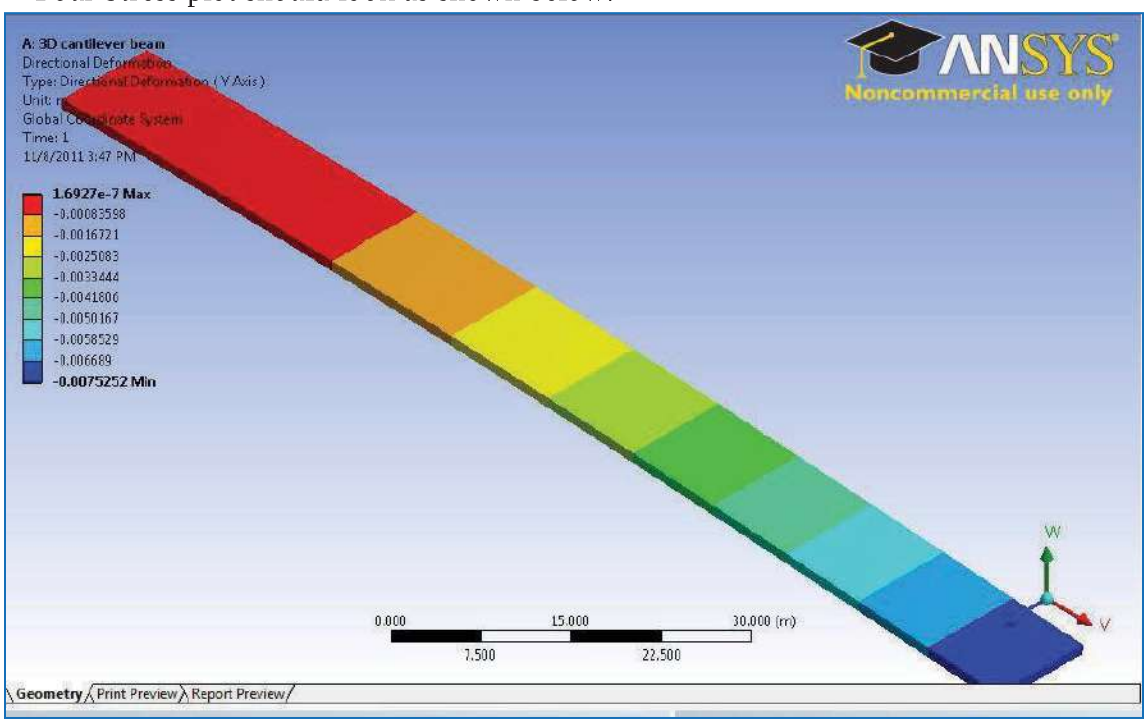
<u>Stress</u>

Go to **Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Beam Tool -> Insert -> Beam Tool -> Stress -> Maximum Bending Stress**



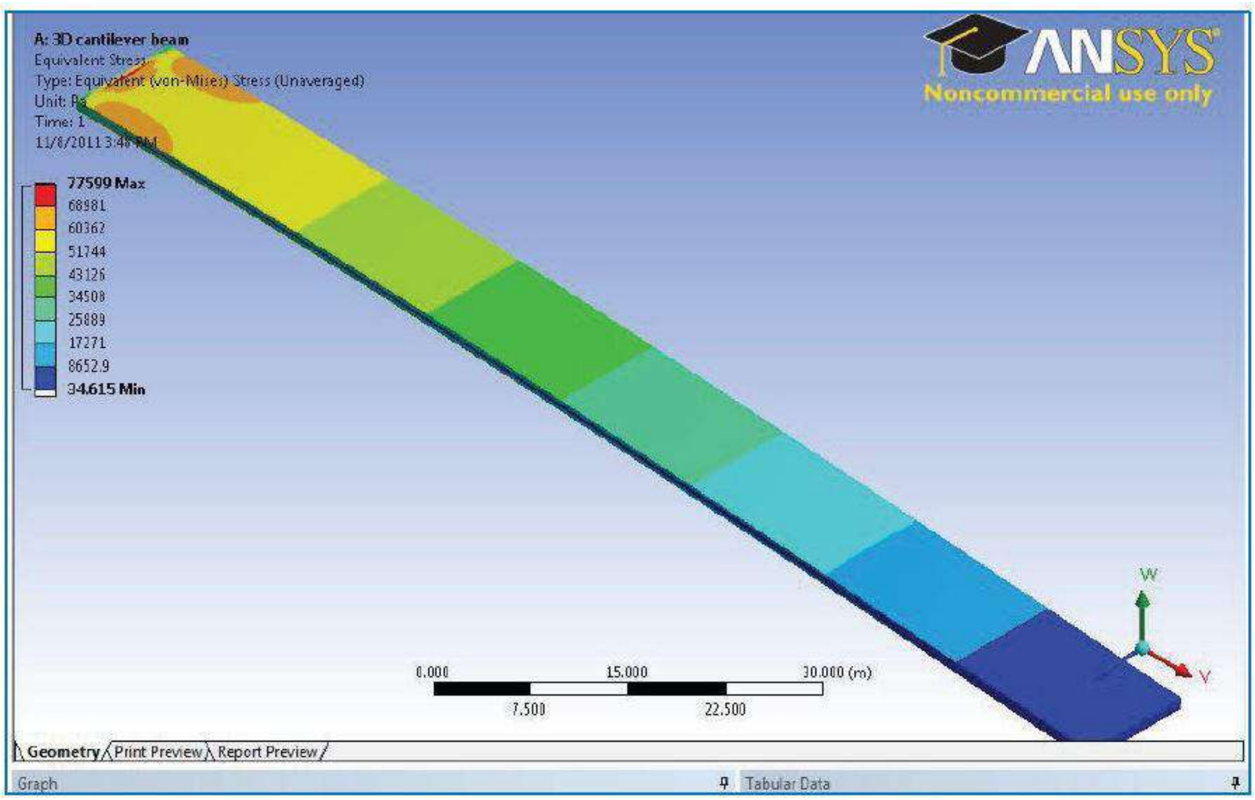
Now that our solvers have been defined, go to **Mechanical** -> **Solve**. The calculations in Workbench may take up to a minute to solve.

Go to **Mechanical** -> **Outline** -> **Project** -> **Model (A4)** -> **Solution (A6)** -> **Maximum Bending Stress**
Go to **Details of "Maximum Bending Stress"**-> **Integration Point Results** -> **Display Option** -> Change to **Unaveraged**
Your Stress plot should look as shown below:



Go to **Mechanical -> Outline -> Project -> Model(A4) -> Solution(A6) -> Total Deformation**

Your Von-Mises plot should look as shown below:



Results

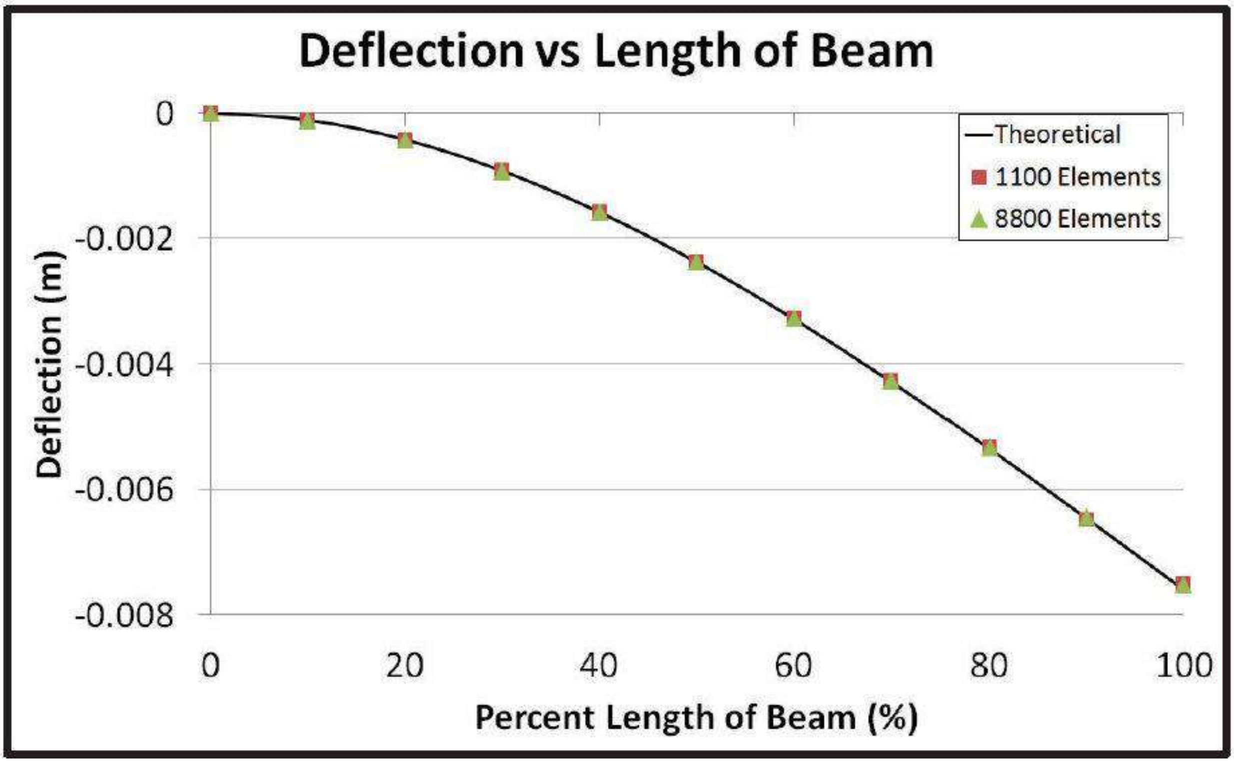
Max Deformation Error According to equation , the theoretical max deflection is 7.16 mm. The percent error (%E) in our model can be defined as:

$$\%E = abs\left(\frac{\delta_{theoretical} - \delta_{model}}{\delta_{theoretical}}\right) * 100 = \mathbf{1.28\%}$$

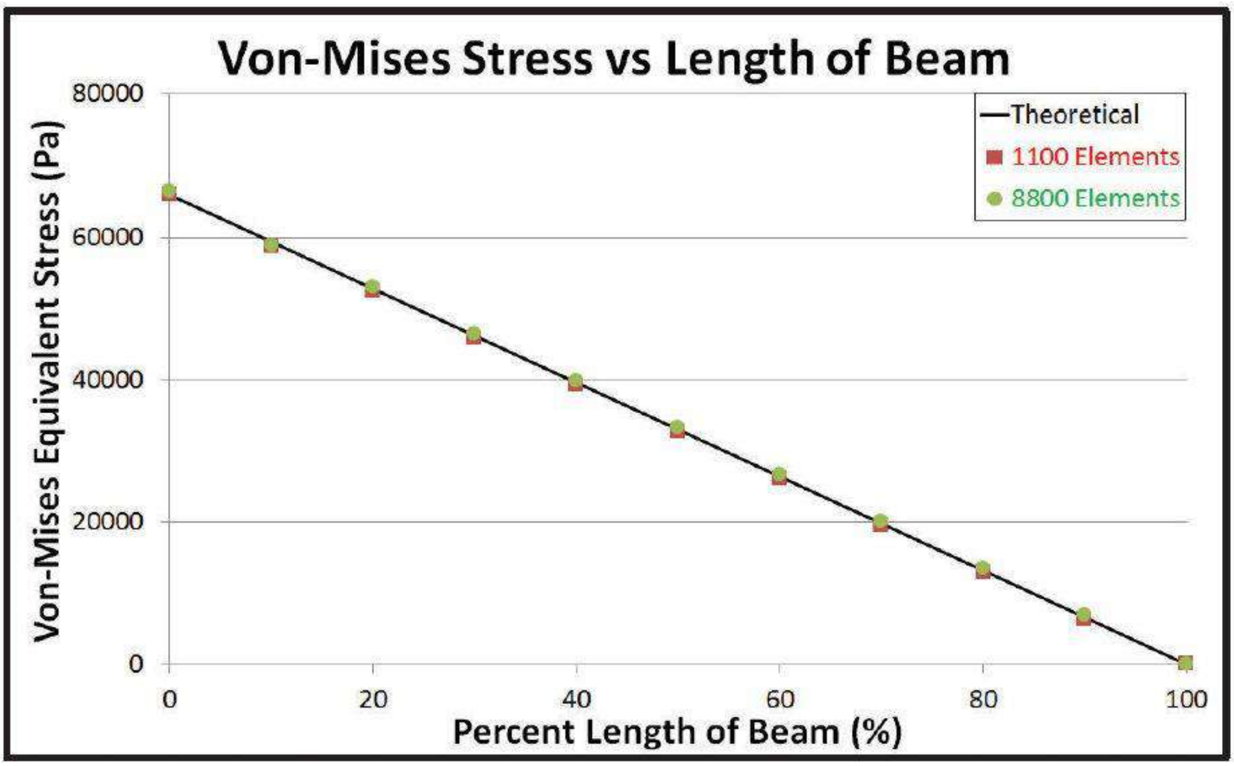
Max Equivalent Stress Error

According to equation, the theoretical max equivalent stress is 66000 Pa. Using the same definition of error as before, we derive that our model has **6.3%** error in the max equivalent stress. The reason for the elevated stress level is singularity resulting from Poisson's effect at the fixed support. In the validation section, it is shown that with increased mesh size, the analytical answers for Max Equivalent stress are closely represented in nodes close to but not at the region where singularity occurs. The effect of singularity is also reduced with the implementation of higher order elements.

Validation



Deflection vs Length of Beam
0
-0.002
-0.004
-0.006
-0.008
Deflection (m)
0
20
40
60
80
100
Percent Length of Beam (%)
Theoretical
1100 Elements
8800 Elements



Von-Mises Stress vs Length of Beam
80000
60000
40000
20000
0
Von-Mises Equivalent Stress (Pa)
0
20
40
60
80
100
Percent Length of Beam (%)
Theoretical
1100 Elements
8800 Elements

MODAL ANALYSIS OF STEPPED SHAFT

Ex. No : 6

Date **:**

INTRODUCTION

Modal analysis is an important type of structural analysis and play an important role in vibration prediction of engineering structures. Every structure has its number of natural frequenciesstarting from first, second to infinite. Modal analysis is used to predict these natural frequencies. These natural frequencies are used to avoid resonance as it occurs if applied load frequency matches with one of the natural frequencies of vibrating body.

PROBLEM DESCRIPTION

The geometric dimension of stepped shaft has been shown in fig-1. A 3D model has been shown in fig-2.

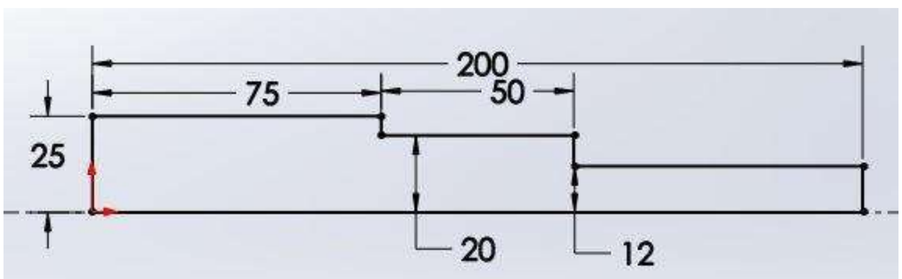


Fig1

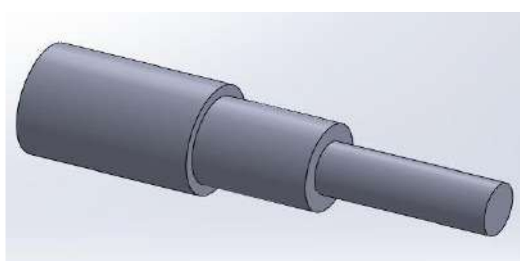


Fig 2

Step-1. Open Ansys workbench and double click on **moda**l analysis module from toolbox, it appears in **Project schematic**, fig-3.

Step-2. In project Schematic right click **Geometry>Import Geometry>Browse** the path where downloaded IGS file of stepped shaft is located, fig-4.

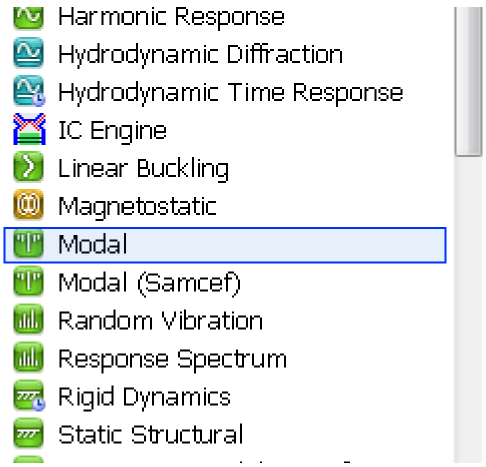


Fig 3

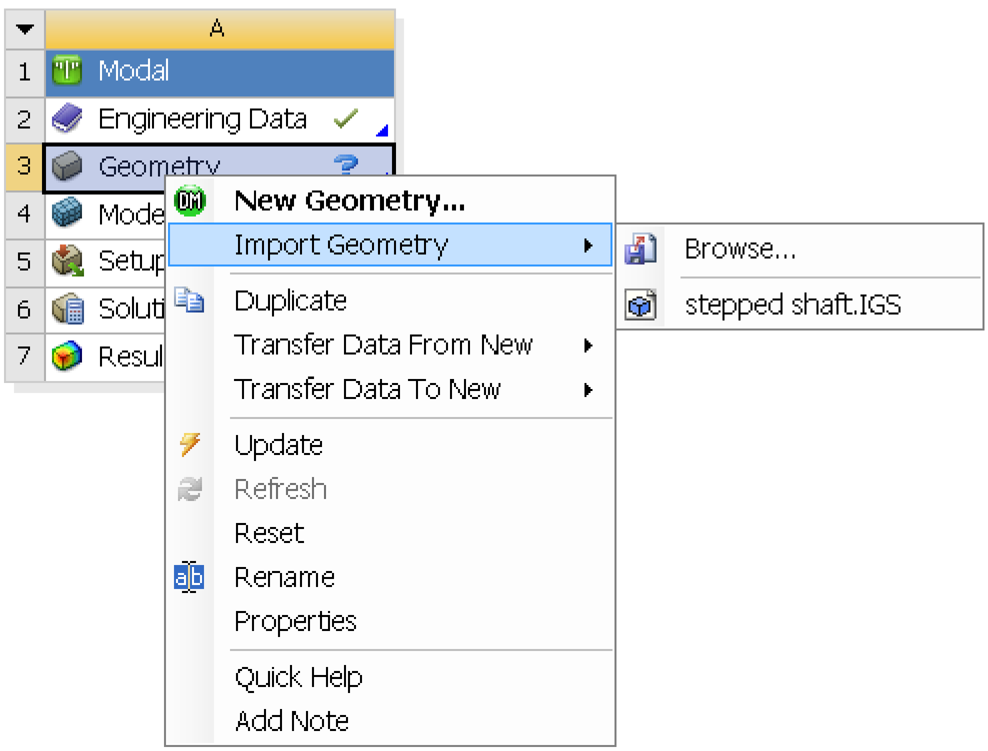


Fig 4

Step-3: Click **File>Save**, write **File Name** as "stepped shaft" and press **Save**.

Step-4: Right click on **Model>Edit**, mechanical window opens now.

Step-5: Click on **mesh** in **Outline** tree and go to **Details of mesh** and click '+' sign on the left of **Sizing** and change **relevance center** from **Coarse to fine**, fig-5.

Step-6: Right click on **mesh** and click on **Generate mesh**, fig-6. Meshing of shaft will be generated, fig-7.

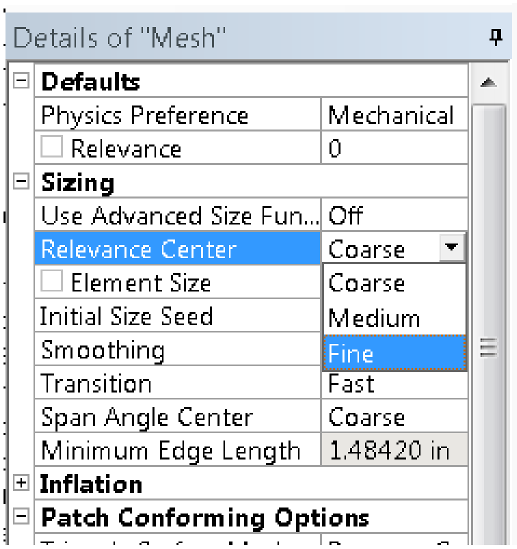


Fig 5

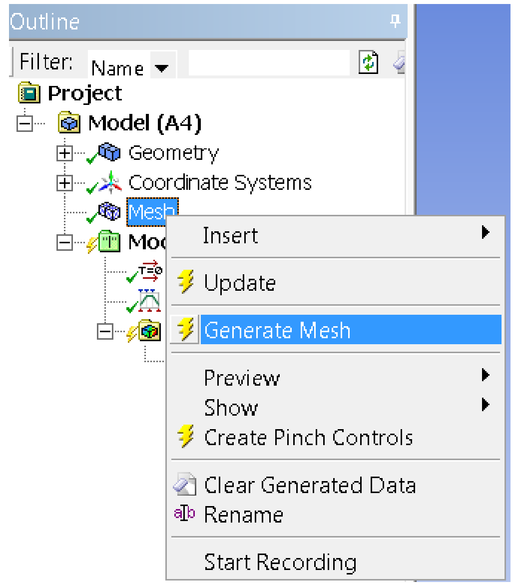


Fig 6

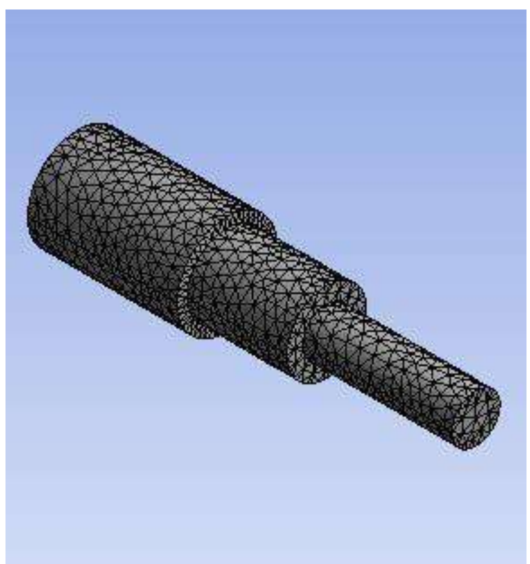


Fig 7

Step-7: Click on **Face selection** button to select faces, fig-8. From outline right click **Modal>Insert>Fixed Support**, fig-9. Now select left end face it turns green, fig-10 and "Details view of support" click apply, fig-11.

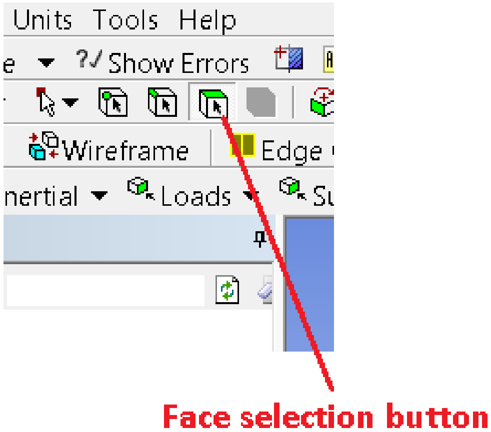


Fig 8

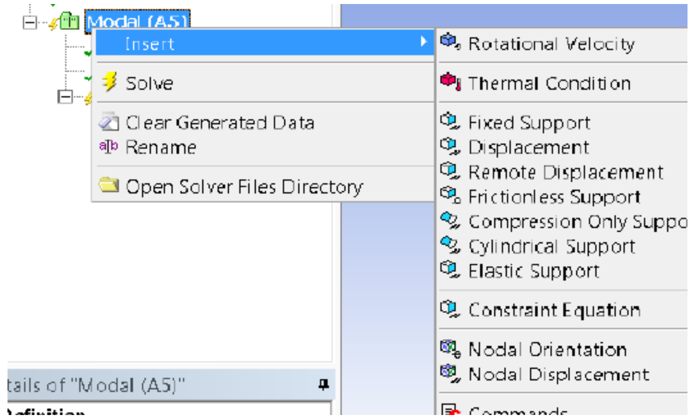


Fig 9

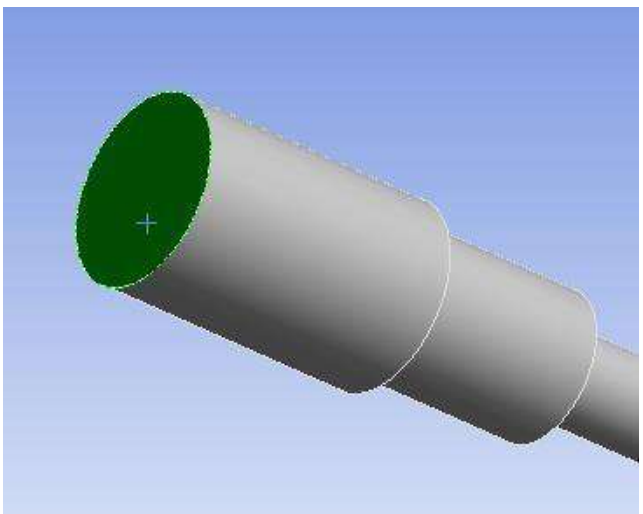


Fig 10

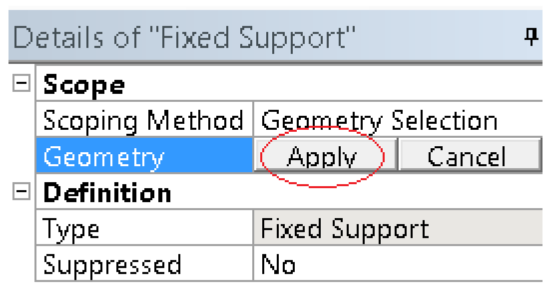


Fig 11

Step-8: Click on **analysis settings,** that is present under **Modal** in outline. Go to "Details of analysis settings" and set **Max Modes to Find** to 6, fig-12. Click **File>Save.** Right click **Solution>Insert>Deformation>Total and** set **Mode** to 1 in "Details of Deformation"., Fig-13.

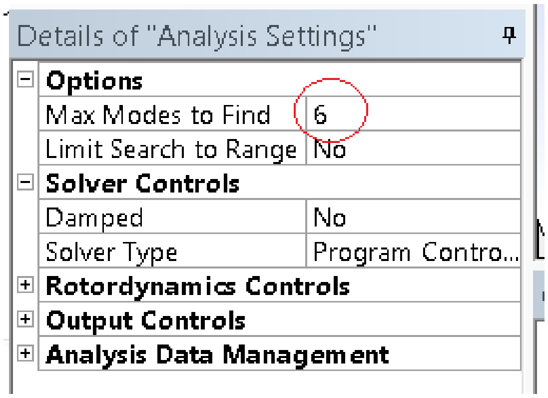


Fig 12

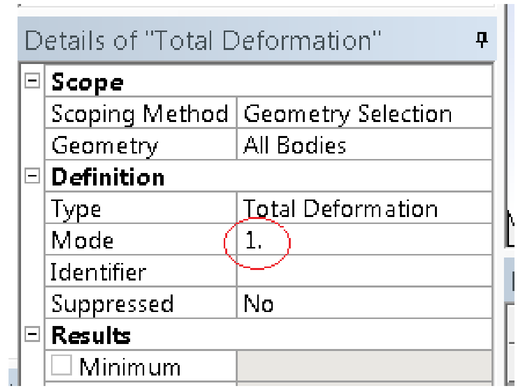


Fig 13

Step-9: Similarly repeat Step-9 again to set mode 2, 3, and so on up to 6.

Step-10: Right click on **Solution>Solve** and wait until solution status complete.

Step-11: Reduce the **view scale** from top right corner to 1.7e-2, fig-14.

Step-12: Click on **Total Deformation** under **Solution** in order to see contours for any Mode, fig-16. **Frequency Table** is also available from **Tabular Data**, Fig-15.

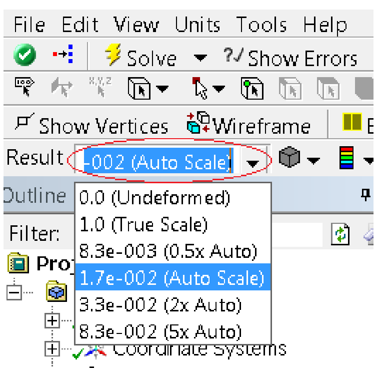
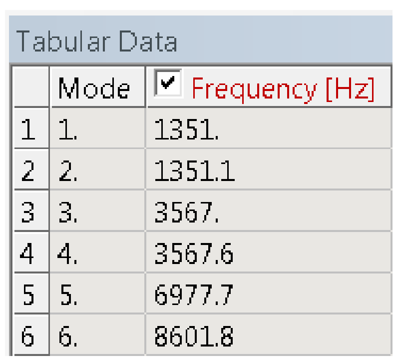


Fig 14


Tabular Data

	Mode	Frequency [Hz]
1	1.	1351.
2	2.	1351.1
3	3.	3567.
4	4.	3567.6
5	5.	6977.7
6	6.	8601.8

Fig 15

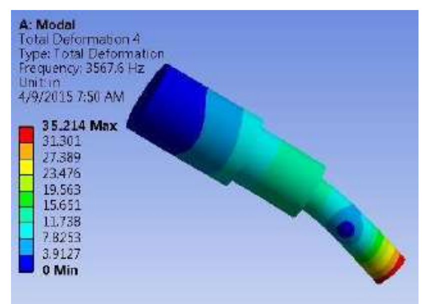


Fig-16: Mode 4 Shape

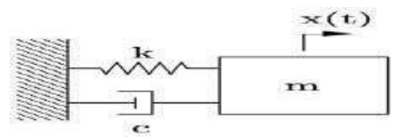
Result:

So in this modal analysis, we have found natural frequencies using Ansys workbench.

TRANSIENT ANALYSIS OF A SPRING MASS DAMPER SYSTEMUSING ANSYS

Ex: No: 7

Date:



Problem:

Show the effect of damper and its damped oscillation when the mass m is displaced from its equilibrium position and left free for its damped oscillation (as shown in Fig.7). Take spring stiffness (K)

= 100 N/m, and mass (m) = 10 kg. Show the response of the following cases, when damping coefficient (i) C = 0, (ii) C = 0.1 of C_c (iii) C = C_c, (iv) C = 1.2 of C_c where C_c = critical damping coefficient = $2\sqrt{K*m}$

ANSYS Analysis Procedure:

1. **Utility menu bar:** select **File > Change Title**:
   Enter the title (Give title of your choice) in the window appeared and click 'OK'.
2. **Main menu bar: Preferences > Structural>** Click 'OK'
3. **Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the** window click **Add>select combination(sub window 1)> spring damper14(sub window 2)>** Click 'OK'
   Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the window select "**type1 COMBIN14**> click on "**OPTIONS**" > K2 options (enter) longitude UY DOF
   Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the window click **Add**>select**sturcutral mass(sub window 1)>3D mass 21 (sub window 2)>** Click 'OK'
   Main menu bar: Preprocessor>Element Type > Add/Edit/Delete > in the window select "**type1 COMBIN14**> click on "**OPTIONS**" > K3options (select) 2-D w/o rot iner> click **'OK'**
4. **Main menu bar: Preprocessor>Real Constants > Add/Edit/Delete>** Click **Add...** and select '**COMBIN14**'> Click on 'OK'. The **Real Constants** window will appear> input **K** = 100>**CV1**=10> Click on 'OK'. 'Set 1' now appears in the dialog box.
   Select Mass21'> Click on 'OK'. The **Real Constants** window will appear> input **mass** = 10 Click on 'OK'. 'Set 2' now appears in the dialog box. Click on 'Close' in the 'Real Constants' window
5. **Main menu bar: Preprocessor>Modeling> Create >nodes> In Active CS, in** the create nodes in active coordinate system**, type key point no. and its coordinates. Here**

node	**x**	**y**	**z**
1	0	0	0 (Click 'apply')
2	0	-1	0 (Click 'OK')

6. **ANSYS Main Menu: Preprocessor>Modeling>Create>Nodes>In Active CS>** type the following in the Create Nodes in the Active Coordinate System window

Node number	**X**	**Y**	**Z**
1	**0**	**0**	**0 (Click Apply)**
2	**1**	**0**	**0 (Click OK)**

7. **ANSYS Main Menu: Preprocessor>Modeling>Create>Element>Auto Numbered> Through Nodes,** in the Elements from Nodes window, type **'1' (Enter)** and type **'2' (Enter) and Click 'OK'.**
8. **ANSYS Main Menu: Solution >Define Loads > Apply > Structural > Displacement** >**On nodes**>Type node **1** in the Apply U, ROT on Nodes window **click 'Apply'**, then **select 'UX'** in the Apply U, ROT on Nodes window (Check Apply as field with **' Constant value'** and **VALUE = 0**), then **Click 'OK'**
9. **ANSYS Main Menu: Solution >Define Loads > Apply >Initial Condition>Define>**in the Define Initial Conditions window type **'2' and Click 'Apply'.** Then in the Define Initial Conditions window, in the DOF to be specified field select **'UX'**and in the Initial value of DOF type **'-1'.Click 'OK'.**
10. **ANSYS Main Menu: Solution >Analysis Type>New Analysis**>**Transient**>**Click 'OK'>Click 'OK'.**
11. **ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Transient>**in the Full Transient Options **Tick Transient effects** and in the Algorithm Field>select **HHT algorithm. Click 'OK'.**

12. **ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Nonlinear>Click Set Convergence Criteria>Replace>Tolerance about VALUE>0.0001> OK>Close>Close>OK**
13. **ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Basic>**in the Write Items to Result field, **Tick User selected>Nodal DOF solution** and in the Frequency field> select **Write every Nth Substep,** then in the Time Control field >in the Time at end of load step type **'0.0002'** and **Tick No. of Substeps and type 5 in the No. of Substeps field. Click 'OK'**
14. **ANSYS Main Menu: Solution>Load step options>Output Ctrls> Solu Printout>**in the Solu Printout Controls window, in the Item for Printout Control field select **'Basic quantities'** and in the Print frequency field, **Tick None. Click 'OK'.**
15. **ANSYS Main Menu: Solution>Load step options>Load step opts>Write LS file> Click 'OK'**
16. **ANSYS Main Menu: Solution>Load step options >Output ctrls >Solu printout>>**in the Solu Printout Controls window, in the Item for Printout Control field select **'Basic quantities'** and in the Print frequency field, Tick **Last Substep**. **Click 'OK'.**
17. **ANSYS Main Menu: Solution >Analysis Type> Sol'n controls >Basic>**in the Time Control field>in the Time at end of load step type**'10'** and **type 40 in the No. of Substeps field. Click 'OK'.**
18. **ANSYS Main Menu: Solution>Load step options >Load step opts>Write LS file> Click 'OK'**
19. **ANSYS Main Menu: Solution>Solve>From LS file,** in the Solve Load Step Files, in the LSMIN field, type **'1',** in the LSMAX field, type **'2'** and in the LSINC field, type **'1'. Click 'OK'.**
20. **ANSYS Main Menu: Time Hist Postpro >Close the window**
21. **ANSYS Main Menu: Time Hist Postpro >Settings>Data>**in the Data Settings window, in the TMIN field, type **'0.003',** in the TMAX field, type '**10'** and in the TINC field, type **'1'. Click 'OK'.**
22. **ANSYS Main Menu: Time Hist Postpro >Define Variable>Add>** in the Add Time-History Variable window, **Tick Nodal Solution. Click 'OK',** then in the Define Nodal Data window, in the NVAR field, type **'2',** in the NODE field, type **'2',** and in the User specified Label field, type **'2UX'.** Then in the Item, Comp Data item, **Pick DOF solution (sub window 1) and Translation UX (sub window 2). Click 'OK',**
23. **ANSYS Main Menu: Time Hist Postpro >Graph Variables>**in the Graph Time- History Variable window, in the NVAR1 field, type **'2'. Click 'OK',**
24. **ANSYS main window shows the corresponding output graph.**
25. **With the above said step, Change the Damping Coefficient (in Step 5) as C = 0 (nodamper), 6.32 (under damped), 63.2 (critically damped), and 75.84 (over damped). See the corresponding output graph in the ANSYS main window.**

RESULT:

Thus the transient analysis of a spring mass damper system has been analyzed.

INTRODUCTION TO CNC

DEFINITION OF CNC

"A system in which the actions are controlled by direct insertion of numerical data at some point .The system must automatically interpret at least some portion of this data"

WHY IT IS CALLED AS CNC?

Since the information required to actuate and control slides of the machine are coded numerically, this technology came to be known as Numerical Control.

WHAT IS CNC?

CNC is acronym for <u>Computer Numerical Control.</u>

A dedicated computer is used to perform all the basic NC functions. The complete part programme to produce a component is input and stored in the computer memory and the information for each operation is fed to the machine tools. The program can be stored and used in future

AXIS IN CNC MACHINES

THE BASIS OF AXIS IDENTIFICATION IS THE 3-DIMENSIONAL CARTESIAN CO-ORDINATE SYSTEM AND THREE AXIS OF MOVEMENT ARE IDENTIFIED AS X,Y AND Z AXIS

Z AXIS.

Z- Axis The Z Axis of motion is always the axis of the main spindle of the machine.It doses not matters whether the spindle carries the work piece or the cutting tool . On vertical machining centers Z axis is vertical and on horizontal machining center and turning centers Z axis is horizontal

Positive Z Movement is away from the spindle

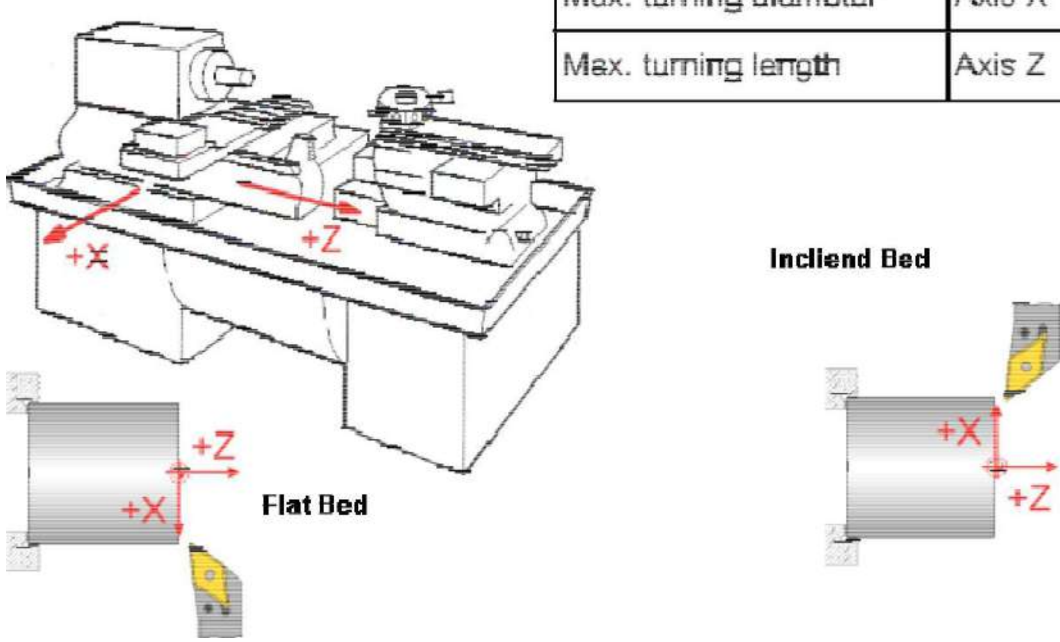
X-Axis The axis is always horizontal and is always parallel to the work holding surface. Positive X Axis movement is identified as being to the right, when looking from the spindle towards its supporting column.

Y- Axis The axis is always at right angle to both X-Axis and Z-Axis

Rotary axis .The rotary motion about the X,Y and Z-Axis are identified by A,B,C respectively .Clockwise is designated as +VE. .Positive rotation is identified looking in x ,y and z direction respectively

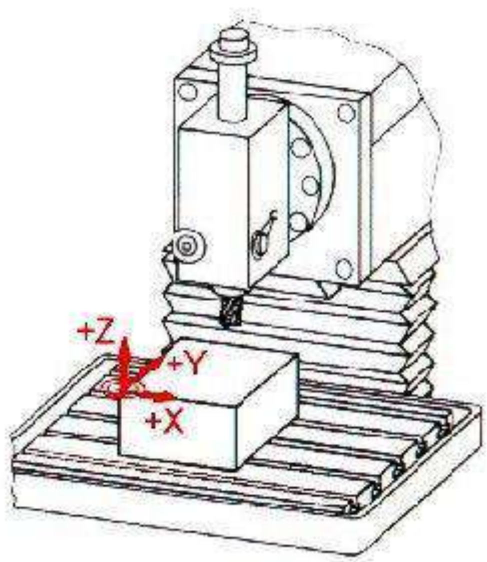
AXIS IN CNCLATHE

Max. turning diameter	Axis X
Max. turning length	Axis Z



AXIS IN MILLING MACHINE

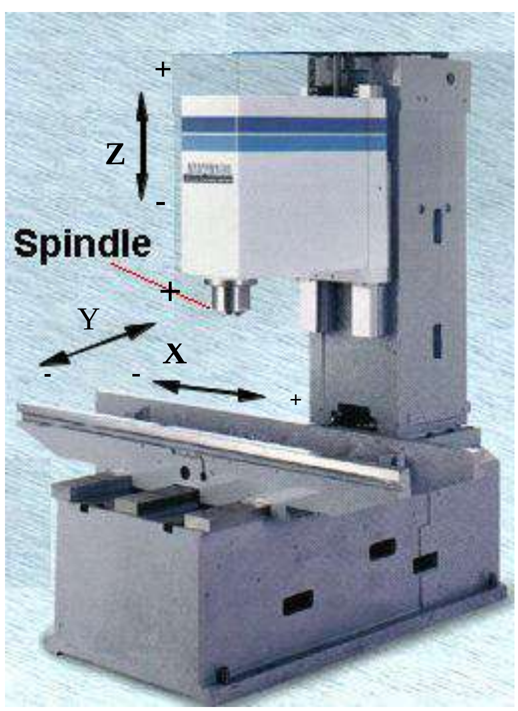
A milling machine has 3 axes of movement identified by X, Y & Z axes

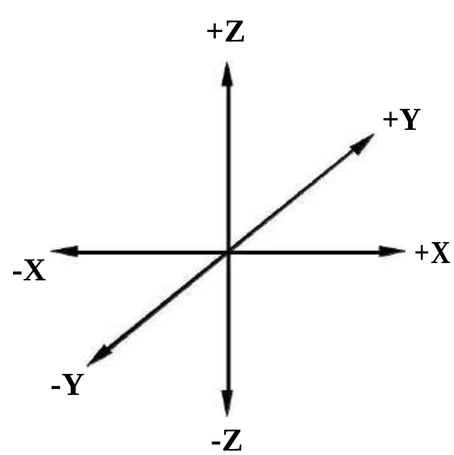


Max. workpiece length	Axis X
Max. workpiece width	Axis Y
Max. workpiece height	Axise Z

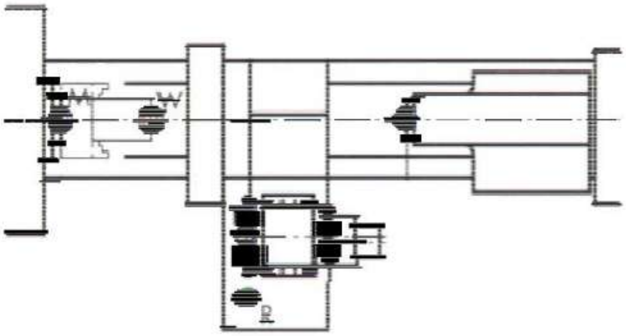
The maximum work piece dimensions correspond to the possible traversing path of the tool in the particular axis.

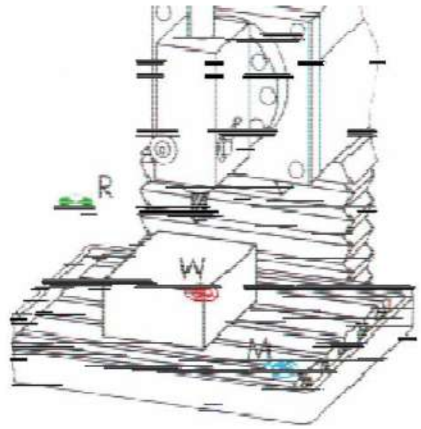
AXIS IN MILLING MACHINE





ZERO POINTS REFERENCE POINT





Machine zero M

The manufacturer defines the machine zero M and this cannot be changed.
It is located at the origin of the machine coordinate system.

Workpiece zero W

The workpiece zero W (also known as program zero) is at the origin of the workpiece coordinate system. It can be freely selected, and should be located at the point where most of the dimensions originate in the drawing.

Reference point R

The reference point R is approached to set the measuring system to zero, as the machine zero point can generally not be approached. The control starts to count in its incremental position measuring system.

The reference point R serves for calibrating and for controlling of measuring systems of the slides and tool traverses. The position of the reference point is accurately predetermined in every traverse axis by the trip dogs and limit switches. Therefore, the reference point coordinates always have the same, precisely known numerical value in relation to the machine zero point.

After initiating the control system, the reference point must always be approached from all axes to calibrate the traverse measuring system.

DIMENSION SYSTEM

Dimensional information in a work piece drawing can be stated in two ways Absolute Dimension System and Incremental Dimension System.

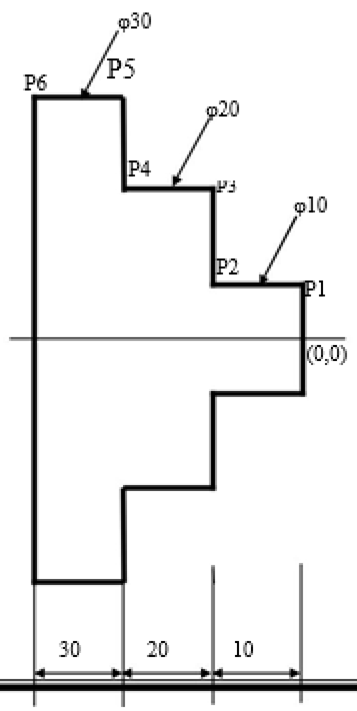
Absolute Dimension System

Data in absolute dimension system always refer to a fixed reference point. This point has the function of a coordinate zero point. The dimension lines run parallel to the coordinate axes and always start at the reference point. Absolute dimensions are also called as 'Reference dimensions'

Incremental Dimension System

When using Incremental Dimension system, every measurement refers to a previously dimensioned position Incremental dimensions are distance between adjacent points. These distances are converted into incremental coordinates by accepting the last dimension point as the coordinate origin for the new point. This may be compared to a small coordinate system, i.e., shifted consequently form point to point (P1..P2..through P9). Incremental dimensions are also frequently called 'Relative dimensions' or 'Chain dimensions'.

DIMENSION SYSTEM



ABSOLUTE DIMENSIONING			INCREMENIAL DIMENSIONING		
POINTS	X	Z	POINTS	U	W
P1	10	0	P1	10	0
P2	10	-10	P2	0	-10
P3	20	-10	P3	10	0
P4	20	-25	P4	0	-15
P5	30	-25	P5	10	0
P6	30	-35	P6	0	-10

STUDY OF ISO CODES

CNC MILLING

AIM: To study various CNC Milling codes and addresses

PREPARATORY FUNCTIONS (G CODES)

A number following address G determines the meaning of the command for the concerned block. G codes are divided into the following two types

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified
Modal G code	The G code is effective until another G code of the same group is specified

LIST OF G CODES

G00: Point to point positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation clockwise

G03: Circular interpolation counter clockwise

G04: Dwell, Exact stop

G17: X Y Plane selection

G18: Z X plane selection

G19: Y Z Plane selection

G20: Input in inch

G21: Input in metric (mm)

G28: Return to reference point

G40: Cutter compensation cancel

G41: Cutter Compensation left

G42: Cutter Compensation right

G43: Tool length compensation positive direction

G44: Tool length compensation negative direction

G49: Tool length compensation cancel

G50: Scaling OFF

G51: Scaling ON

G54: Datum shift

G68: Rotation ON

G69: Rotation cancel

G73: High speed Peck drilling cycle

G74: L.H Tapping cycle

G76: Fine Boring

G80: Canned cycle cancel

G81: Continuous drilling cycle, Stop boring

G82: Continuous drilling cycle, Stop boring with dwell

G83: Peck drilling cycle

G84: R.H. Tapping cycle

G85: Boring cycle with feed retraction

G86: Boring cycle with rapid retraction

G87: Back boring cycle

G88: Boring cycle

G89: Boring cycle with dwell & feed retraction

G90: Absolute coordinates

G91: Incremental coordinates

G92: Set Datum

G94: Feed per minute

G95: Feed per revolution

G98: Return to initial point in a canned cycle

G99: Return to R point in a canned cycle

G170, G171: Circular pocketing

G172, G173: Rectangular pocketing

MISCELLANEOUS FUNCTIONS (M CODES)

When a three digit figure is specified following address M, a 3-digit BCD code signal and a strobe signal are transmitted. These signals are used for ON/OFF control of a machine function such as tool change, spindle rotation change, coolant ON/OFF. One M code can be specified in one block. Selection of M codes for functions varies with the machine tool builder.

LIST OF M CODES

M00: Program stop

M01: Optional (planned) stop

M02: End of program

M03: Spindle ON clockwise

M04: Spindle ON counter clockwise

M05: Spindle stop

M06: Tool change

M07: Coolant 2 ON

M08: Coolant 1 ON

M09: Coolant OFF

M10: Vice open

M11: Vice close

M12: Synchronization code

M13: Spindle clockwise and coolant ON

M14: Spindle counterclockwise and coolant ON

M19: Orientates Spindle

M20: ATC arm IN (Towards spindle)

M21: ATC arm out (Retracts from spindle)

M22: ATC arm down

M23: ATC arm up

M24: Activates ATC draw bar

M25: Releases draw bar

M30: Program stop & rewind

M32: Rotates ATC clockwise

M33: Rotates ATC counter clockwise

M38: Opens the door

M39: Closes the door

M62: Output 1 ON

M63: Output 2 ON

M64: Output 1 OFF

M65: Output 2 OFF

M66: Wait input 1 ON

M67: Wait input 2 ON

M70: X mirror ON

M71: Y mirror ON

M76: Wait input 1 OFF

M77: Wait input 2 OFF

M80: X mirror OFF

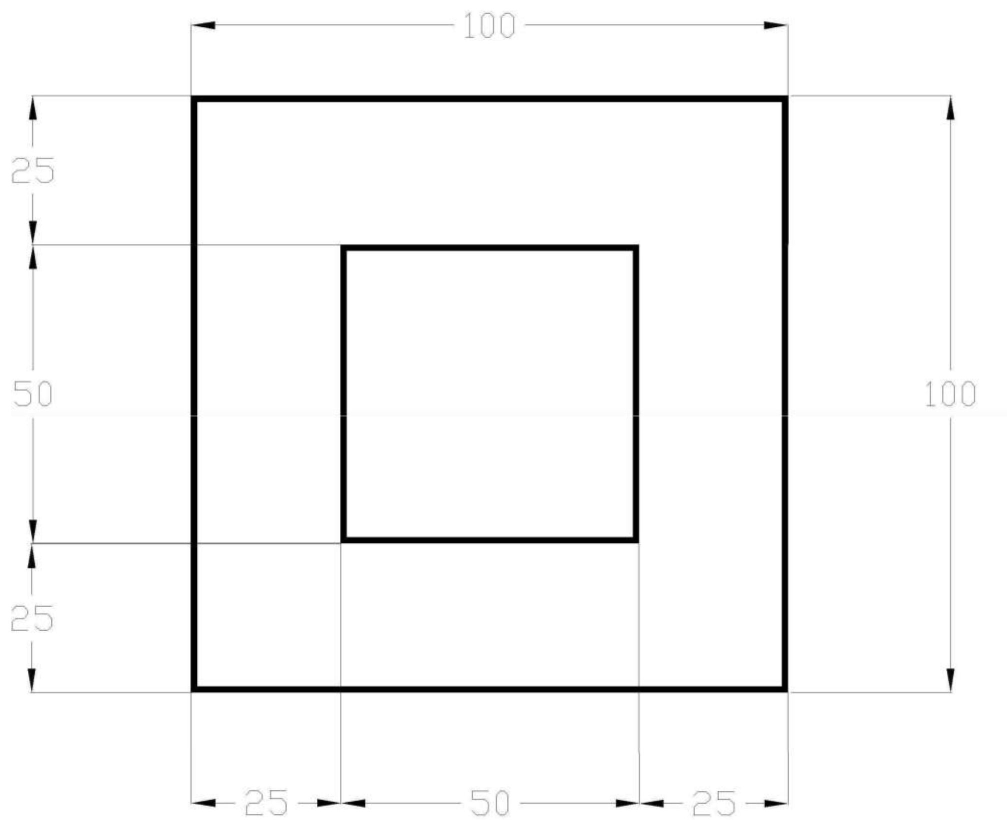
M81: Y mirror OFF

M98: Sub program Call

M99: Sub program Exit

Write a manual part program for machining the component shown in **DWG.NO.2**.Profile depth=1mm

LINEAR MOTIONS



ALL DIMENSIONS ARE IN "mm"

DWG.NO.2

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1002				**DWG.NO:2**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

LINEAR MOTIONS

EX.NO: 8
DATE:

AIM:
To write a manual part program for machining the component shown in the DWG. NO.2

MATERIAL REQUIRED:

Material Size

: Aluminium
: Length 100mm, Width 100mm, Thickness 10mm

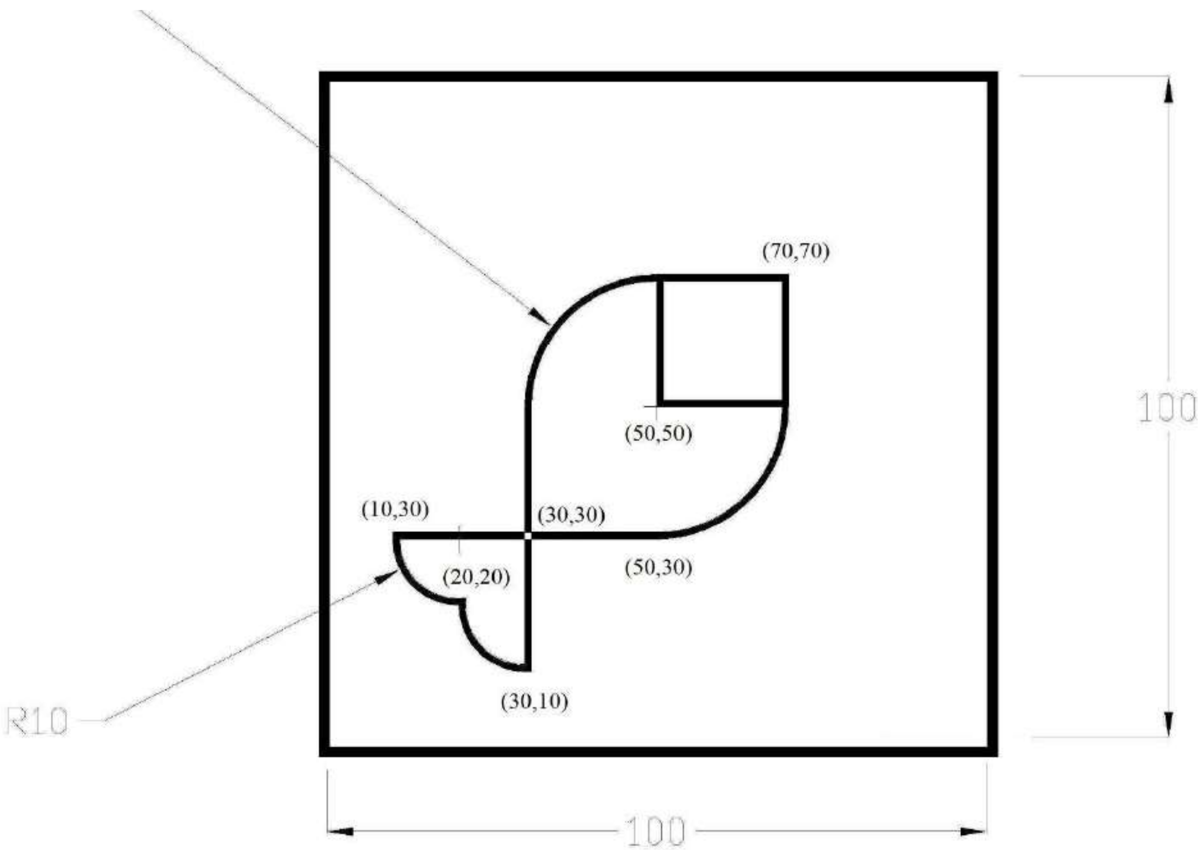
PROGRAM:

```
O0002 G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X25 Y25 Z5
 G01 Z-1 F50
G01 X75
G01 Y75
G01 X25
G01 Y25
G00 Z5
G91 G28 X0 Y0 Z0 M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for **Profile Milling operation** for the component shown in **DWG.NO.4**.
Profile depth=1mm



ALL DIMENSIONS ARE IN "mm"

DWG.NO.4

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1004				**DWG.NO:4**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Contouring	Slot cutter	5	1	1	1500	30-50

CONTOUR MOTIONS

EX.NO: 9

DATE:

AIM:

To write a manual part program for contouring operation for the component shown in the DWG. NO.4

MATERIAL REQUIRED

Material :Aluminium
Size : Length 100mm, Width 100mm, Thickness 10mm

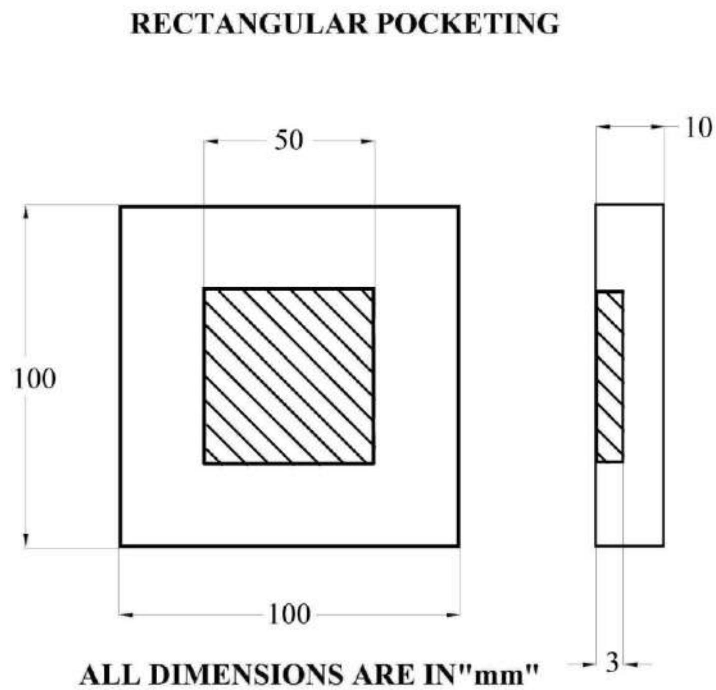
PROGRAM:

```
O0004
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1500
G90 G00 X50 Y70 Z5
G01 Z-1 F30
G01 X70
G01 Y50
G01 X50
 G01 Y70
G03 X30 Y50 R20
G01 X30 Y10
G02 X20 Y20 R10
G02 X10 Y30 R10
G01 X50
G03 X70 Y50 R20
G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling.

Write a manual part program for **Rectangular Pocketing** operation for the component shown in **DWG.NO.5**. Profile depth=3mm



DWG.NO.5

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 100x100x10				**MATERIAL:** Aluminium			
PROGRAM NO:1005				**DWG.NO:5**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool length offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Slotting & Pocketing	Slot cutter	5	1	1	1500	35-45

RECTANGULAR POCKETING
G172 I(*i1) **J**(*j1) **K**(*k1) **P**(*p1) **Q**(*q1) **R**(*r1) **X**(*x1) **Y**(*y1) **Z**(*z1)
G173 I(*i2) **K**(*k2) **P**(*p2) **T**(*t2) **S**(*s2) **R**(*r2) **F**(*f2) **B**(*b2) **J**(*j2) **Z**(*z2)

INPUT DATA FOR G172	**INPUT DATA FOR G173**
I – Length of the pocket in X direction J – Length of the pocket in Y direction K – Corner radius (always zero) P – when P = 0 , roughing & when p = 1 , Finishing . Q **--** Depth of cut for each pass .i.e. Pocket Z increment. R**--** Absolute Z from surfaceX **--** Pocket corner X Y – Pocket corner Y Z – Absolute Z base of pocket	I – Pocket side finish allowance K**--**Pocket base finish allowance P—Cut width Percentage T—Pocket Tool S—Spindle speed for roughing R—Roughing feed for z F**--**Roughing feed XY B—Finishing spindle speed J—Finishing feed & Z—Safety Z

RECTANGULAR POCKETING

EX.NO: 10

DATE:

AIM:

To write a manual part program for Rectangular Pocketing operation for the component shown in the DWG. NO.5

MATERIAL REQUIRED:

Material: Aluminum
Size: Length 100mm, Width 100mm, Thickness 10mm

PROGRAM:

```
O0005
G21 G94
G91 G28 X0 Y0 Z0
M06 T01
M03 S1200
G90 G00 X0 Y0 Z5
G172 P0 Q1 R0.5 X-25 Y-25 Z-3 I50 J50 K0
G173 P80 T1 S1200 R30 I0.1 K0.1 F70 B2000 J40 Z5
G172 P1 Q1 R0.5 X-25 Y-25 Z-3 I50 J50 K0
G173 P80 T1 S1200 R30 I0.1 K0.1 F70 B2000 J40 Z5 G00 Z5
G91 G28 X0 Y0 Z0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC milling

CNC TURNING CODES

AIM: To study various CNC Turning codes and addresses

G Codes

G00: Point to point positioning (Rapid traverse)

G01: Linear interpolation

G02: Circular interpolation clockwise

G03: Circular interpolation counter clockwise

G04: Dwell, Exact stop

G17: X Y Plane selection

G18: Z X plane selection

G19: Y Z Plane selection

G20: Input in inch

G21: Input in metric (mm)

G28: Return to reference point

G32: Thread Cutting

G40: Cutter compensation cancel

G41: Cutter Compensation left

G42: Cutter Compensation right

G49: Tool length compensation cancel

G50: Work co-ordinate Change / Max Spindle speed setting

G70: Finishing cycle

G71: Stock removal in turning

G72: Stock removal in facing

G73: Pattern repeating

G74: Peck drilling in z axis

G75: Grooving in x axis

G76: Thread Cutting Cycle

G80: Canned cycle cancel

G90: Cutting cycle A

G92: Thread cutting cycle

G94: Cutting cycle B

G96: Constant surface speed control

G97: Constant surface speed control cancel

G98: Feed per minute

G99: Feed per revolution

M Codes

M00: Program stop

M01: Optional (planned) stop

M02: End of program

M03: Spindle forward clockwise

M04: Spindle forward counter clockwise

M05: Spindle stop

M06: Tool change

M08: Coolant ON

M09: Coolant OFF

M10: Chuck open

M11: Chuck close

M62: Output 1 ON

M63: Output 2 ON

M64: Output 1 OFF

M65: Output 2 OFF

M66: Wait input 1 ON

M67: Wait input 2 ON

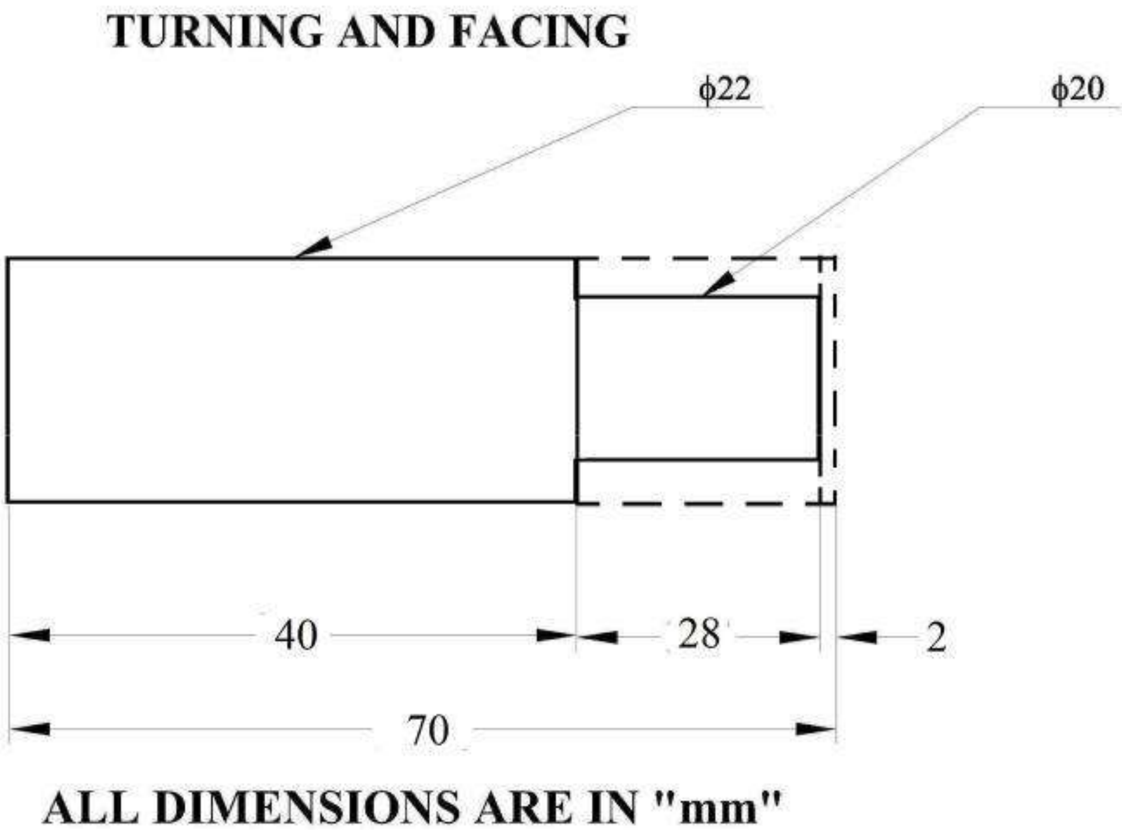
M76: Wait input 1 OFF

M77: Wait input 2 OFF

M98: Sub program Call

M99: Sub program Exit

Write a manual part program for simple **Turning and Facing** operation for component shown in **DWG.NO.T01**



DWG.NO.T01

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1001				**DWG.NO:1**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Simple Facing	SDJCR121H11	DCMT11T304	1	1	1200	45

PLAIN TURNING AND FACING

EX.NO: 11

DATE:

AIM:

To write a manual part program for simple turning and facing operation for component shown in DWG.NO.T01

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

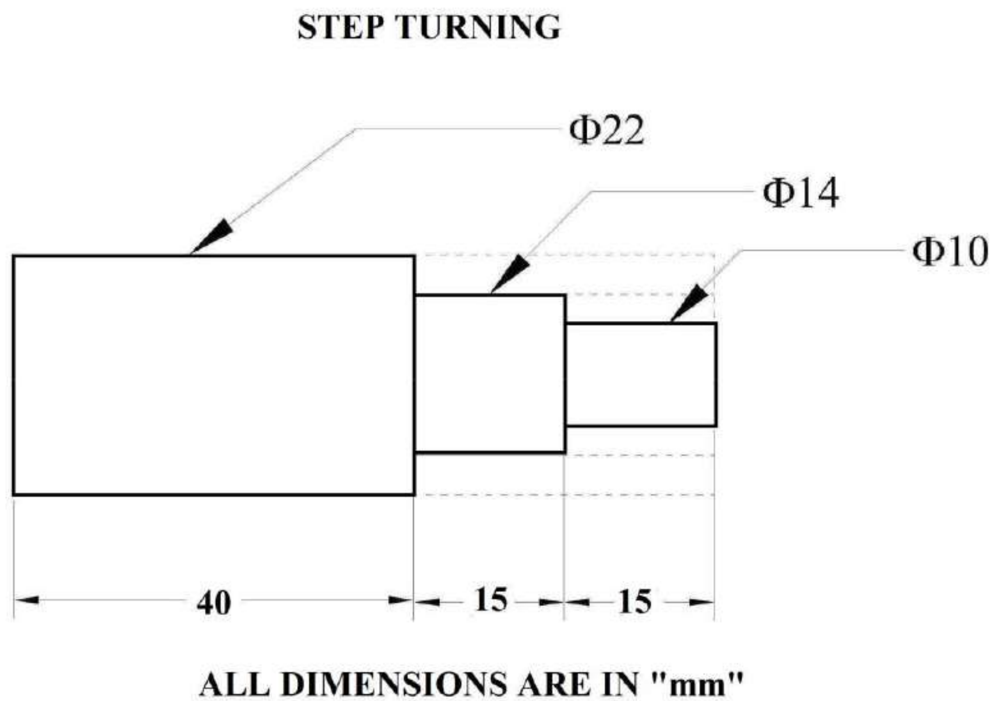
PROGRAM:

```
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X23 Z3
G00 Z-0.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-1.5
G01 X-1 F45
G00 Z3
G00 X23
G00 Z-2
G01 X-1 F45
G00 Z3
G00 X21.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X21
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20.5
G01 Z-30 F45
G00 X23
G00 Z3
G00 X20
G01 Z-30 F45
G00 X23
G28 U0 W0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Step Turning** operation with G90 cycle for component shown in **DWG.NO.T02**



DWG.NO.T02

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1002				**DWG.NO:2**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Step Turning	SDJCR121H11	DCMT11T304	1	1	1200	30

Cutting Cycle

G90 X(U) Z(W) F(*f)

X Diameter to which the movement is being made.

U The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.

W The incremental distance from the current tool position to the required Z axis Position fFeed rate

STEP TURNING

EX.NO: 12

DATE:

AIM:

To write a manual part program for step turning operation with G90 cycle for component shown in DWG.NO.T02

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

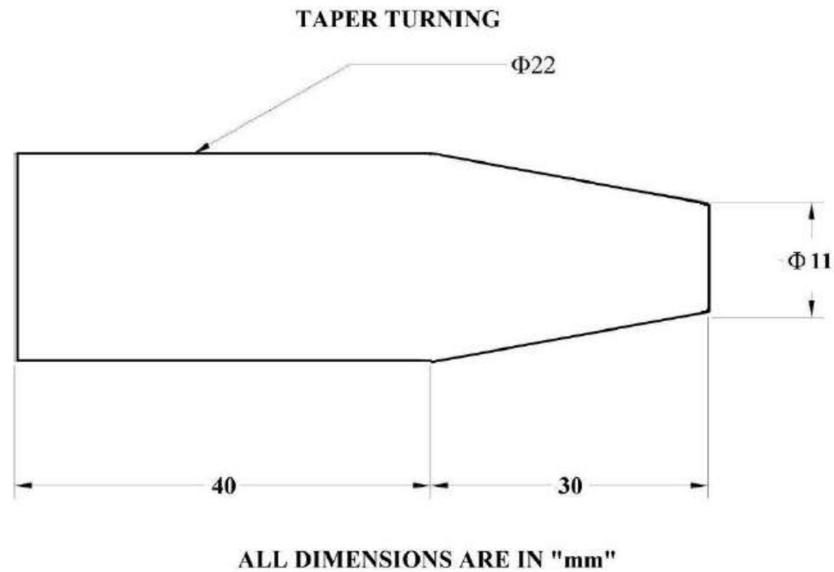
PROGRAM:

```
O0001
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X22 Z1
G90 X22 Z-30 F30
X21
X20
X19
X18
X17
X16
X15
X14
G00 X14 Z1
G90 X14 Z-15 F30
X13
X12
X11
X10
G28 U0 W0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC Lathe.

Write a manual part program for **Taper Turning** operation for component shown in **DWG.NO.T03**



DWG.NO.T03

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1003				**DWG.NO:3**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Taper Turning	SDJCR121H11	DCMT11T302	3	3	1000	30

Cutting Cycle
G90 X(U) Z(W) R F
X Diameter to which the movement is being made.
U The incremental distance from the current tool position to the required final Diameter Z The Z Axis Co-ordinate to which the movement is being made.
W The incremental distance from the current tool position to the required Z axis Position R The difference in incremental of the cut start radius value and the cut finish radius value.
fFeed rate

TAPER TURNING

EX.NO:13

DATE:

AIM:

To write a manual part program for taper turning operation for component shown in DWG.NO.T03

MATERIAL REQUIRED:

Material	: Aluminium
Size	: Length 70mm, Diameter 22mm

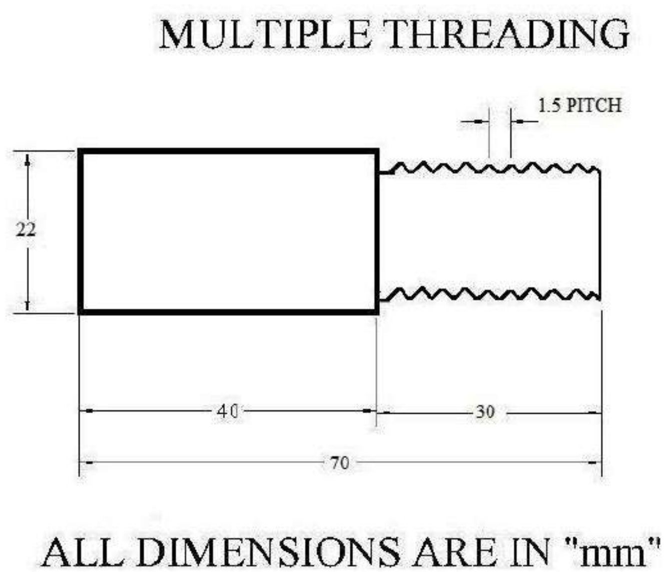
PROGRAM:

```
O0002
G21 G98
G28 U0 W0
M06 T0101
M03 S1200
G00 X22 Z1
G90 X22 Z-30 R0 F30
X22 R-0.5
X22 R-1
X22 R-1.5
X22 R-2
X22 R-2.5
X22 R-3
X22 R-3.5
X22 R-4
X22 R-4.5
X22 R-5
X22 R-5.5
G28 U0 W0
M05
M30
```

RESULT:

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **Multiple Threading** operations for component shown in **DWG.NO.T06**



DWG.NO.T06

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1006				**DWG.NO:6**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25
3	Grooving	HSS	3mm width	5	5	750	15

MULTIPLE THREADING CYCLE

G76 P(m) (r)(a) Q(*q1) R(*r1)
G76 X(*x) Z(*z) P(*p2) Q(*q2) F(*f)

Where
m- Repetitive count in finishing (1 to 99) r = Pull out angle
a= Angle of tool tip
*X = Minor diameter (core diameter)
*z = End position of thread
*p2 = Height of thread as a radius value x 1000, value in microns e.g. 1.02mm becomes p1020
*q2 = depth of the first cut as a radius value x 1000 , value is microns. E.g. 0.25 mm becomes Q250
*f = Lead or pitch of thread. E.g. lead 1.5 = F1.5
*q1 = Min. cutting depth
*r1 = Finishing allowance

THREADING

EX.NO:14

DATE:

AIM:

To write a manual part program for multiple threading operation for component shown in DWG.NO.T06

MATERIAL REQUIRED:

Material : Aluminium
Size: : Length 70mm, Diameter 22mm

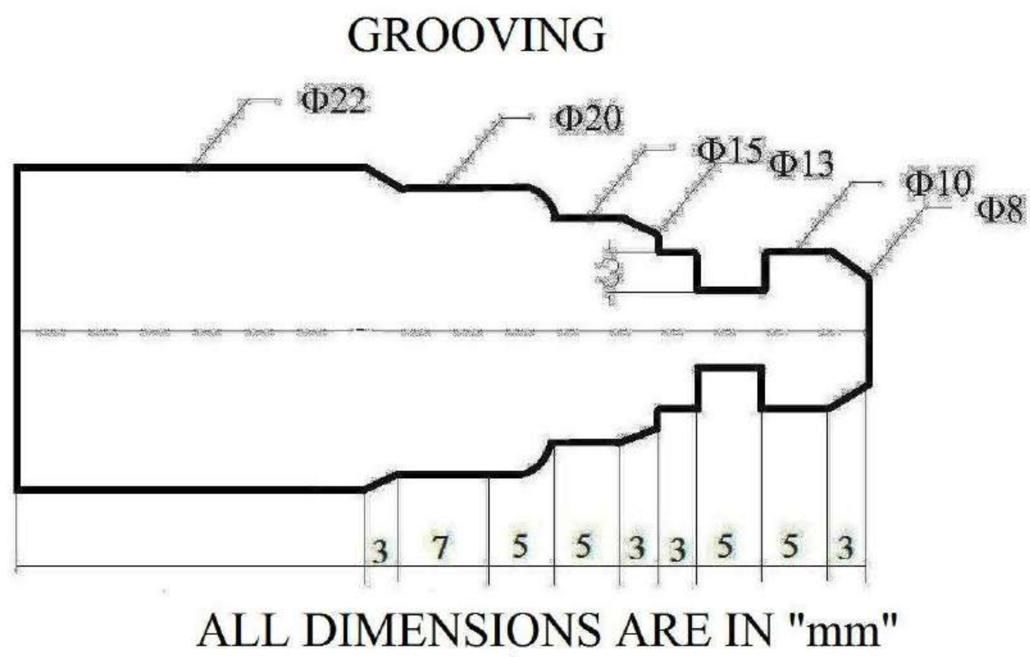
PROGRAM:

```
O0006
G21 G98
G28 U0 W0
M06 T0404
M03 S600
G00 X22.5 Z2
G76 P031560 Q050 R0.02
G76 X20.162 Z-20 P920 Q100 F1.5
G28 U0 W0
M05
M30
```

RESULT

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **External Grooving** operation for component shown in **DWG.NO.T05**



DWG.NO.T05

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1005				**DWG.NO:5**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25
3	Grooving	HSS	3mm width	5	5	750	15

GROOVING CYCLE

G75 R (*r1)
G75 X(u) Z(w) P(*p) Q(*q) F (*f)

Where, r1-Return Amount
X-Total Depth along X Axis (Absolute)
u-Total Depth along X Axis (Incremental)
z-Total Depth along Z Axis (Absolute)
w-Total Depth along Z Axis (Incremental)
p-Peck Increment in X Axis in microns
q -Stepping distance in Z Axis in microns
f -Feed Rate in mm

GROOVING

EX.NO:15

DATE:

AIM:

To write a manual part program for external grooving operation for component shown in DWG.NO.T05

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

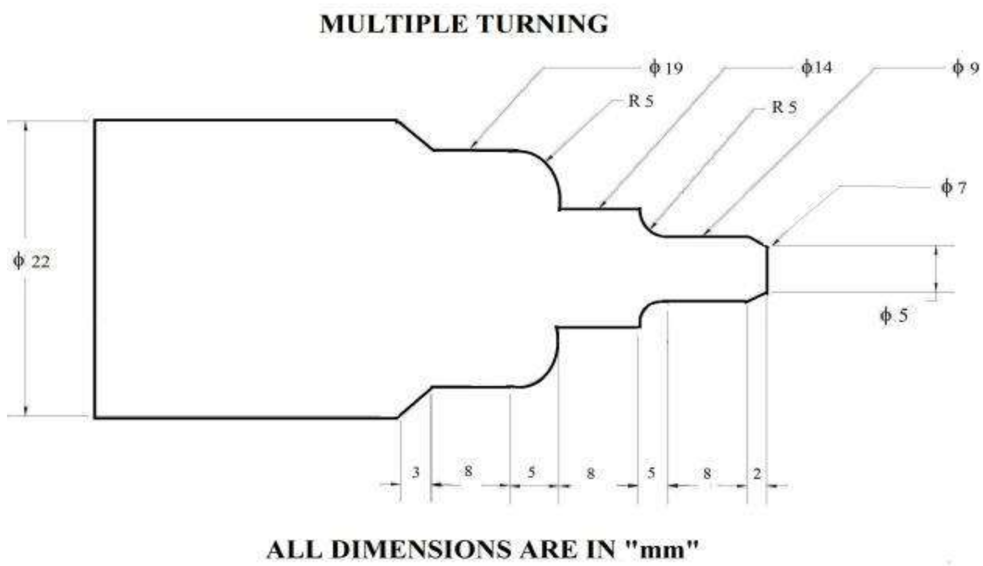
PROGRAM:

```
O0005
G21 G98
G28 U0 W0
M06 T03
M03 S1200
G00 X22 Z2
G71 U0.5 R1
G71 P10 Q20 U0.1 W0.1 F45
N10 G01 X8 Z0
G01 X10 Z-3
G01 Z-16
G01 X13
G01 X15 Z-19
G01 Z-24
G03 X20 Z-29 R5
G01 Z-36
N20 G01 X22 Z-39
M03 S1600
G70 P10 Q20 F30
G28 U0 W0
M06 T0707
M03 S600
G00 X10.5 Z-8
G75 R1
G75 X7 Z-10 P100 Q1000 F40
G28 U0 W0
M05
M30
```

RESULT

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

Write a manual part program for **Multiple Turning** operation for component shown in **DWG.NO.T04**



DWG.NO.T04

PLANNING AND OPERATIONS SHEET							
BILLET SIZE: 22x60				**MATERIAL:** Aluminium			
PROGRAM NO:1004				**DWG.NO:4**			
Sl.No	**Operation**	**Tool type**	**Tool dia., mm**	**Tool Station No.**	**Tool offset No.**	**Spindle speed, rpm**	**Feed, mm/min**
1	Multiple Rough turning	SDJCR121H11	DCMT11T304	1	1	1200	35
2	Finishing	SDJCR121H11	DCMT11T302	2	2	1450	25

MULTIPLE THREADING CYCLE

G76 P(m) (r)(a) Q(*q1) R(*r1)
G76 X(*x) Z(*z) P(*p2) Q(*q2) F(*f)

Where
m- Repetitive count in
finishing (1 to 99)r =
Pull out angle
a= Angle of tool tip
*X = Minor diameter (core diameter)
*z = End position of thread
*p2 = Height of thread as a radius value x 1000, value in microns e.g. 1.02mm becomes p1020
*q2 = depth of the first cut as a radius value x 1000 , value is microns. E.g. 0.25 mm becomes Q250
*f = Lead or pitch of thread. E.g. lead 1.5 = F1.5
*q1 = Min. cutting depth
*r1 = Finishing allowance

CANNED CYCLE – MULTIPLE TURNING

EX.NO: 16

DATE:

AIM:

To write a manual part program for multiple turning operation for component shown in DWG.NO.T04

MATERIAL REQUIRED:

Material : Aluminium
Size : Length 70mm, Diameter 22mm

PROGRAM:

```
O000
G21 G98
G28 U0 W0
M06 T0303
M03 S1200
G00 X22 Z2
G71 U0.5 R1
G71 P10 Q20 U0.1 W0.1 F45
N10 G00 X7 Z0
G01 X9 Z-2
G01 Z-10
G02 X14 Z-15 R5
G01 Z-23
G03 X19 Z-28 R5
G01 Z-36
N20 G01 X22 Z-39
M03 S1600 F30
G70 P10 Q20
G28 U0 W0
M05
M30
```

RESULT

Thus the manual part program was written to the given dimensions and executed in CNC lathe.

CNC MACHINE FAILURES AND TROUBLESHOOTING

Besides portraying decent durability, CNC machines are known for being long-lasting and can be relied upon due to their redundancy. However, even the most reliable CNC machines can frequently encounter malfunctions, be it minimal or substantial.

These deterrents, if minimal, can be addressed by a technician right away. But if the malfunction turns out to be substantial, it can lead to prolonged downtime and tax your business financially.
Here are the eight most common CNC machine failures and their solutions that you must be aware of if your company employs a benchtop CNC mill to perform mundane tasks.

1. Using Inappropriate Cutting Tools or Settings

Picking an inappropriate cutting tool can degrade the final build quality of the product, typically visible through rough edges, raised marks, or burn spots on the material's corners.

Poor material finishes directly result from either the tool being blunt or an improper feed speed ratio. Another leading cause can be choosing inaccurate tool dimensions for the task at hand in terms of the sizes, build quality or material.

Solution
Resolving this issue requires that you choose the appropriate tool and setting for the build quality and material. You must refer to proper instruction manuals that state the parameters for various tools and quality used.

2. Programming Errors

Since sophisticated equipment gets operated by CNC computers, several issues in CNC machining arise from incorrect programming.

This scenario might be due to a lack of understanding of the various G and M codes employed for the controller, improper set-up, or feeding erroneous data variables into the CNC controller.

Solution
New operators must be trained adequately in all different approaches in which CNC machines are programmable. Machine suppliers can provide new operators with comprehensive user manuals accompanied by training, motion sequencing, and machine operation.

3. Poor CNC Machine Tool Maintenance

Modern machines come with several mechanical parts that are constantly in motion. Therefore, CNC machine tools must be cleaned and maintained regularly to ensure optimal operation.

Failing to brush off the dirt, clean the material and other debris can lead to a build-up. This scenario could potentially lead to inaccuracies in machining or even machine failures.

Solution
It is pivotal for machine operators to adhere to a comprehensive maintenance regime for the machine tools. You must frequently check the coolant or airflow levels, for instance, air filters, to establish that the machine continues to operate smoothly.

4. Incompetent Workers and Lack of Training

With the growing implementation of computing and programming, CNC machine tool operators require a distinct set of knowledge and expertise.

Without relevant organizational, planning, and programming skills, workers cannot enhance the yield of these machines despite possessing machining skills and expertise using older machine models.

Solution

As a resolution, you must hire trained machine operators to visualize and design the machining process, choose the appropriate tools and sequences for the task, and write programs.

5. Issues With the Power Supply

The CNC machine tool's display or other parts sometimes might not operate due to problems with the primary power supply. This scenario could lead to the machine yielding inaccurate results or failing to operate at all.

Solution

Ensure that you are using the correct power and voltage for the input parameters. Subsequently, check if the output or secondary side is functional.

If voltage reading is low, disconnect the output wires with the power turned off, power up, and reassess the output side. Also, check if the LEDs on the machines are functional.

6. Problems with Automatic Tool Changer

Sometimes you might encounter issues with the automatic tool changer in your CNC Machine Tool. You can resolve this by learning each step of the tool-changing process.

Solution

Assess that the base, tool holder, gripper arm, support arm, and tool magazines are functioning smoothly. Check the swiveling and mechanical arm action to ensure that they are not causing the issue.

7. Machine Vibration or Chatter

If your CNC machine is vibrating when under operation, it could substantially shorten the life of your tool, negatively impact your CNC machine's durability, or sabotage the quality of your machined component.

Solution

You must diagnose if the noise is workpiece chatter or tool chatter. Consider adjusting the RPM of your machining process to ensure that the frequencies of the machining process do not resonate with the frequency of the material.

8. Machine Tool Overheating

Overheating can occur when dealing with high volume and prolonged durations of machining. The CNC machine tool might reach temperatures of over 150 degrees. This scenario could negatively affect your machining process's result, the tool used, and even the CNC machine.

Solution

You must be sure to clear all channels regularly and have the machine rid of dirt, soil, and debris. Plus, it's necessary to routinely clean up all metal shavings and the liquids utilized in cutting.

Consider using a spindle with Air-Oil Lubrication or an Oil-Jet Lubrication spindle as it has no issues operating for long hours at maximum rpm. Be sure to ventilate your CNC machine to provide some coolant from nature as well.

OPERATING PROCEDURE OF MASTER CAM

Step 1

The usual modification is that is done after initializing the master cam is the alteration of the available screen ie modification of grid size and as per the requirement of user screen,configure, selection grid. Select grid size (configure → grid) The dimensioning requirement is usually selected ie Metric or English etc.

Step 2

Once the screen and the dimensioning adjustments are done the basic required part geometry of the object is created by using the create command.

Create command: Rectangle or Circle or Line or Fillet.

Step 3

After defining the tool parameters and the machining operations parameter, the job setup is done. The job is usually defined in terms of length width and thickness (X, Y, Z). The selection of material for the job is done in jobsetup

Step 4

After the required part geometry is created, the tool path are defined on the selected geometry depending upon the milling operation.

Tool path → contour → save → chain → done

Tool path → pocket → save → area or chain or polygon → done

Once the tool path are defined, press done, tool parameter screen will appear where in the required tool diameter is entered or the tool can be selected from the tool manager.

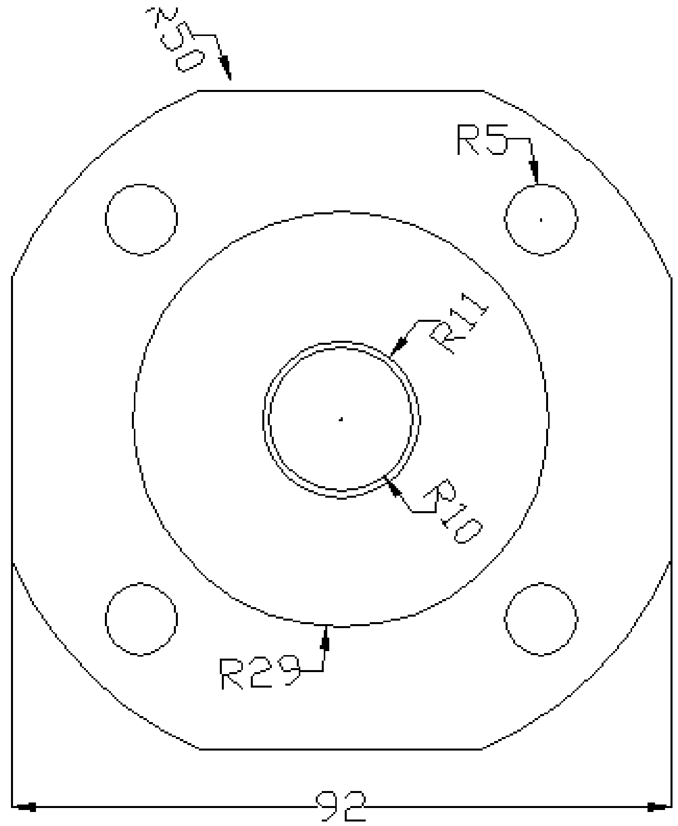
Similarly the above step is carried out for all operations the contour parameter such as clearance, retract, feed, top of stock, depth are given.

Step 5

Completion of the above setup leads to operation manager where in all the operation are selected by select all, paths are regenerated by Regenerate path, and by Verify, machining operation is done. Before machining, configuration is verified where in use of job setup values are highlighted.

Step 6

Press post button to the get the NC program


R50
R5
R11
R10
R29
92

GENERATION OF CNC PROGRAMME AND MACHINING USING MASTERCAM SOFTWARE

AIM

To machine the model as per the sketch.

Procedure:

Step 1

The usual modification is that is done after initializing the master cam is the alteration of the available screen ie modification of grid size and as per the requirement of user screen,

Choose-main menu → Screen → Configure → Current Configuration File

Select- Mill9.Mcfg(metric)

Choose-main menu → Screen → Next menu → Sel.Grid

Select-Active Grid, Visible Grid, Spacing x =1 and y =1, Grid Size= 110,Origin x=0,y=0 and choose suitable for Grid.

SCREEN
CONFIGURE
SYSTEM CONFIGURATION
CURRENT CONFIGURATION
FILE
MILL 7 MC FOR (METRIC)
SCREEN
SELECTION GRID

ACTIVE GRID
VISIBLE GRID
DONE
SAVE AS
FILE NAME
SAVE
OIL
BACK UP

Step 2

1. To create inner and outer circles.

- Choose-main menu → Create → Arc → circpt+dia.
- Typediameter100 → Enter
- Select origin
- Press Esc. And reselect circpt+dia
- Enter diameter58
- Select origin
- Press Esc. And reselect circpt+dia
- Repeat same procedure for diameter 20 and22mm

- xi. Press Esc. to exit circle function
- Choose it screen menu.

2. To create construction of rectangle /Square
   - Choose-main menu → Create → Rectangle → 1 point
   - Enter width 92 and height 92 (bcz. given figure is square) press OK
   - Select origin
   - Press Esc to exit linefunction

3. Trimming unwanted portion oflines
   - Choose-mainmenu → Modify → Trim → 3 entities
   - Select 1 entity as vertical line of square inside thecircle
     2 entity as Horizontal line of square inside the circle 3
     entity as arc of the circle inside thesquare.

MODIFY
FILLET
TRIM
BREAK
JOIN
MORMAL
CEPTS
NORBS X TO

TRIM
1ENTITY
2ENTITES
3ENTITES
TO POINT
MANY
CLOSE ARC
DIVIDE

FILLET
RADIS
ANGLE<180
TRIM Y
CHAIN
BACK

4. Copy and rotate the arc
   - Choose-main menu → X form → Rotate
   - Select anywhere on arc.
   - Choose done.
   - Select origin.
   - Select copy, enter the values. No of steps 3 and angle is90^0
   - Choose OK in the dialogue box remaining slots fo rcreated.

XFORM	ANALYSE	DELETE	FILE
MIRROR	POINT	CHAIN	NEW
ROTATE	CONTOUR	WINDOW	EDIT
SCALE	ONLY	POLYGON	GET

ACALEXYZ	BETWEEN PTS	AREA	MERG
TRANSLATE	ANGLE	ONLY	E LIST
OFFSET	DYNAMIC	ALL	SAVE
GS COLOUR	AREA/VOLUM	GROUP	SAVE SOME
GROUP	E NUMBER	RESUIT	BROWSE
STRETCH	CHAIN	DUPLICATE	CONVERT
ROLL	SURFACE	UNDELETE	ER NEXT
BACK UP	BACKUP	BACKUP	MENU

5. Trimming unwanted portion of lines(Square)

- Choose-main menu → Modify → Trim → 3entities
- Select 1 Entity vertical line of square inside the arc
  2 Entity Horizontal line of square inside the arc
  3 Entity arc inside the square.

6. To create construction lines

- Choose-main menu → Create → Line → Polar
- Enter the first co-ordinate origin
- Enter angle in degree45^0
- Enter the line length45
- Press Esc to exit line function

7. To create inner 10 mm diameter circles

- Choose- main menu → Create → Arc → circpt +dia.
- Type diameter10 → Enter
- Select end point of inclined line.
- Press Esc. And reselect circpt+dia

8. Copy and rotate the Circle

- Choose-main menu → Xform → Rotate
- Select anywhere on 10 mm diameter circle
- Choose done.
- Select origin.
- Select copy, enter the values. No of steps 3 and angle is90^0
- Choose OK in the dialogue box remaining 3 circles for created.

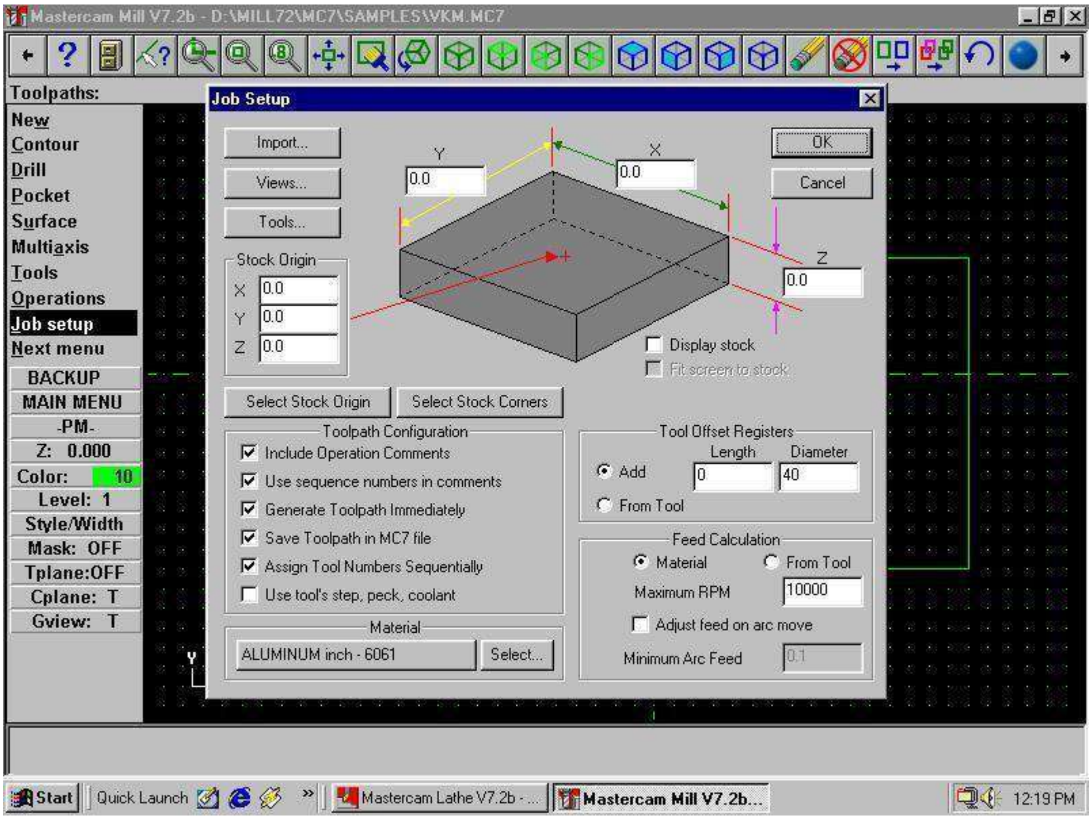
Now your drawing is ready for operations

The following operations to be conduct Using the geometry

- *Facing*
- **Contour**
- *Pocketing*
- **Drilling**

Step 3

1. Make job setup for given geometry
   - Choose-main menu → Toolpath → Job setup
   - Enter the X =110 Y=110 and Z=55 mm respectively
   - Select Display Stack and Fit to Screen.
   - Outside the drawing doted boundary red line is displayed
2. Another option for job setup
   - Choose-main menu → Toolpath → Job setup
   - Select stock origin (0,0)
   - select the stock corners.
   - Enter the value of Z = 55 (+ value).
   - Select the display stock checkbox.
   - Select fit stock checkbox.
   - Choose OK the stock should be enclosed by red dotted line.

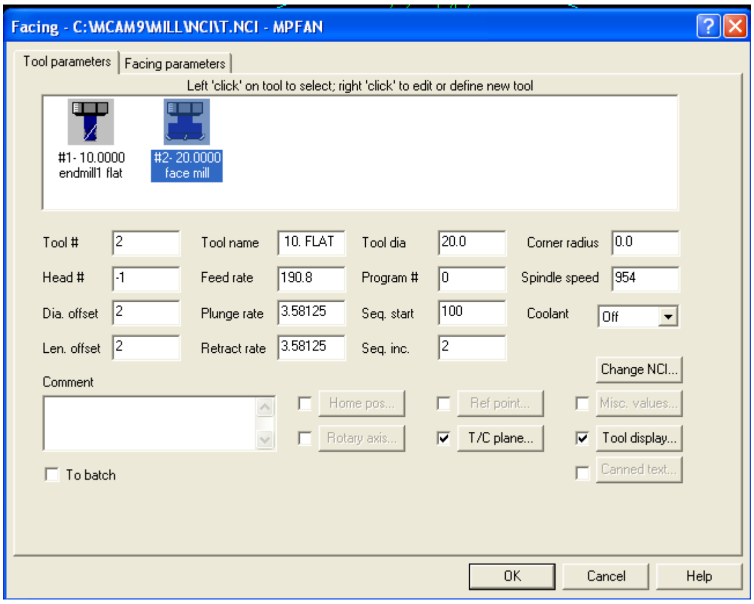


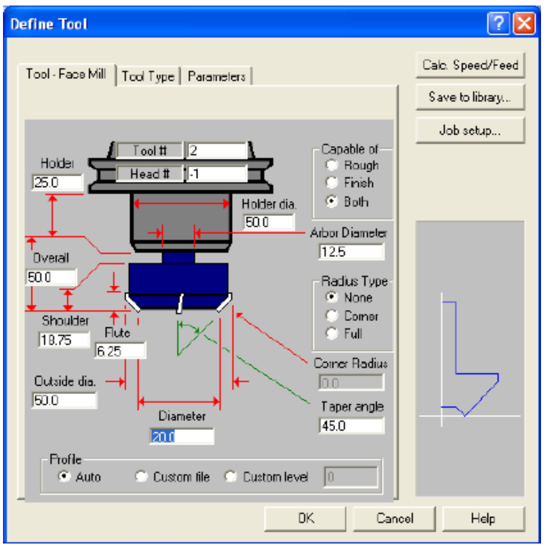
Mastercam Mill V7.2b - D:\MILL72\MC7\SAMPLES\VKM.MC7
Toolpaths:
New
Contour
Drill
Pocket
Surface
Multiaxis
Tools
Operations
Job setup
Next menu
BACKUP
MAIN MENU
-PM-
Z: 0.000
Color: 10
Level: 1
Style/Width
Mask: OFF
Tplane:OFF
Cplane: T
Gview: T
Job Setup
Import...
Views...
Tools...
OK
Cancel
Stock Origin
Select Stock Origin
Select Stock Corners
Display stock
Fit screen to stock
Toolpath Configuration
Include Operation Comments
Use sequence numbers in comments
Generate Toolpath Immediately
Save Toolpath in MC7 file
Assign Tool Numbers Sequentially
Use tool's step, peck, coolant
Material
ALUMINUM inch - 6061
Select...
Tool Offset Registers
Length
Diameter
Add
From Tool
Feed Calculation
Material
From Tool
Maximum RPM
10000
Adjust feed on arc move
Minimum Arc Feed
Start
Quick Launch
Mastercam Lathe V7.2b - ...
Mastercam Mill V7.2b...
12:19 PM

Step 4

1. Create tool path for Facing

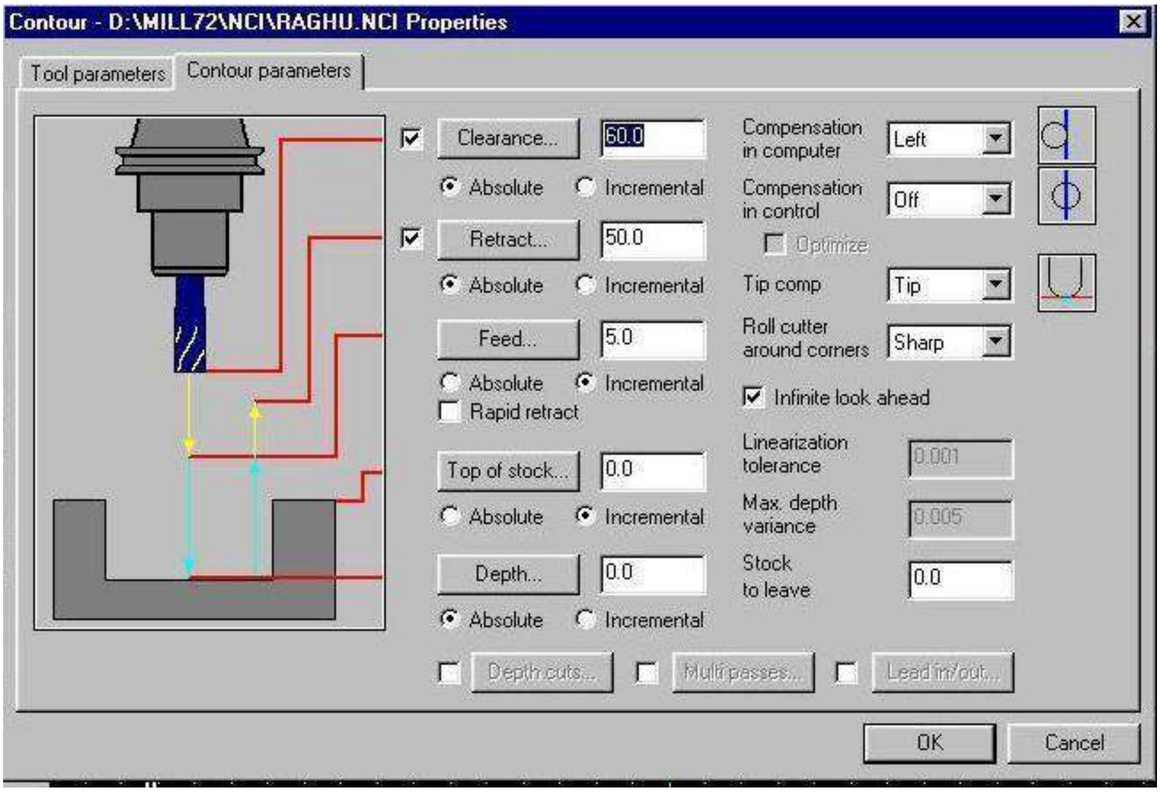
- Choose-main menu → Toolpath → Face
- Select start point for the chain at periphery of the figure.
- Choose done.
- Right click in the tool display area and select a 10 mm flat end mill
- from the tool library.
- Right click on the tool display , go to tool type select face mill tool,
- Select the facing parameters. Give Depth of cut -2mm (-negative)
- Select depth Cuts, Give Rough Cut 3, Finishing cut 1No. Finish Step0.5
- Choose OK twice in the tool path should be showing figure.





1. Create tool path for Contour

- Choose-main menu → Toolpath → Contour
- Select start point for the chain at position1.
- Choose done.
- Right click in the tool display area and select a 10 mm flat end mill from the tool library.
- Select the contour parameters , Give Depth of cut -53mm (- negative), Use Multi passes, Depth Cuts and Lead in/out options.
- Choose OK twice in the tool path should be showing figure.



3. Create tool path for pocket 1

- Choose-main menu → Toolpath → Pocket
- Select the 58 mm diameter circle.
- Choose done.
- Right click in tool display area and select 8mm flat end mill from tool library.
- Enter pocketing parameters and note that the depth should benegative say-10.
- Choose OK the tool path should look like the picture pattern.

4 Create tool path for pocket 2

- Choose-main menu → Toolpath → Pocket
- Select first 22 mm diameter circle.
- Choose done.
- Right click in tool display area and select 5mm flat end mill from tool library.
- Enter pocketing parameters and note that the depth should be negative-12.
- Choose OK the tool path should look like the picture pattern.

6. Create tool path for pocket 3

- Choose-main menu → Toolpath → Pocket
- Select first 20 mm diameter circle.
- Choose done.
- Right click in tool display area and select 3mm flat end mill from tool library.
- Enter pocketing parameters and note that the depth should benegative say-32.
- Choose OK the tool path should look like the picture pattern.

6. Create tool path for Drilling

- Choose-main menu → Toolpath → Drill
- Select Entities Choose 10mm diameter circles one by one.
- Choose done. Tool path is generated, Choose once again done
- Right click in the tool display area and select a 10 mm drill mill from the tool library.
- Select the drill parameters and note that the depth should be negative say-60.
- Choose OK twice in the tool path should be showing figure.

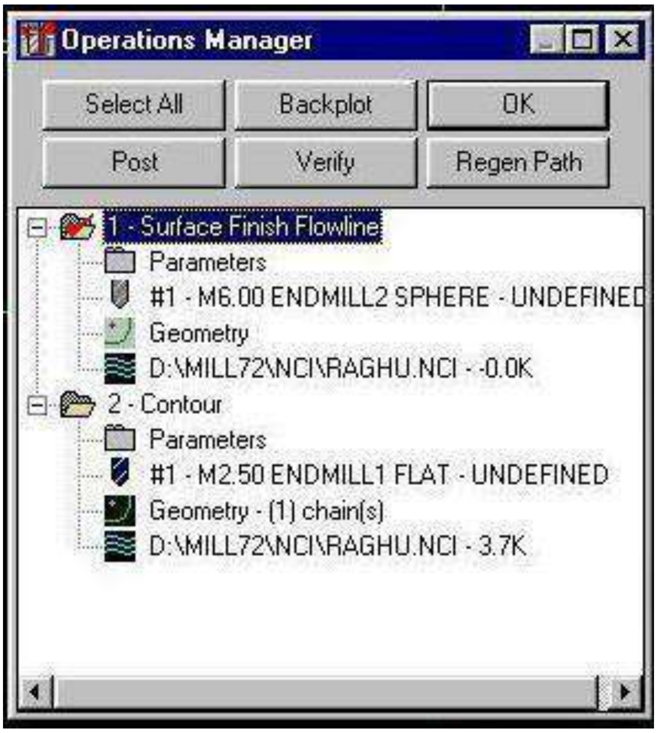
Step 5

Tool path Generator

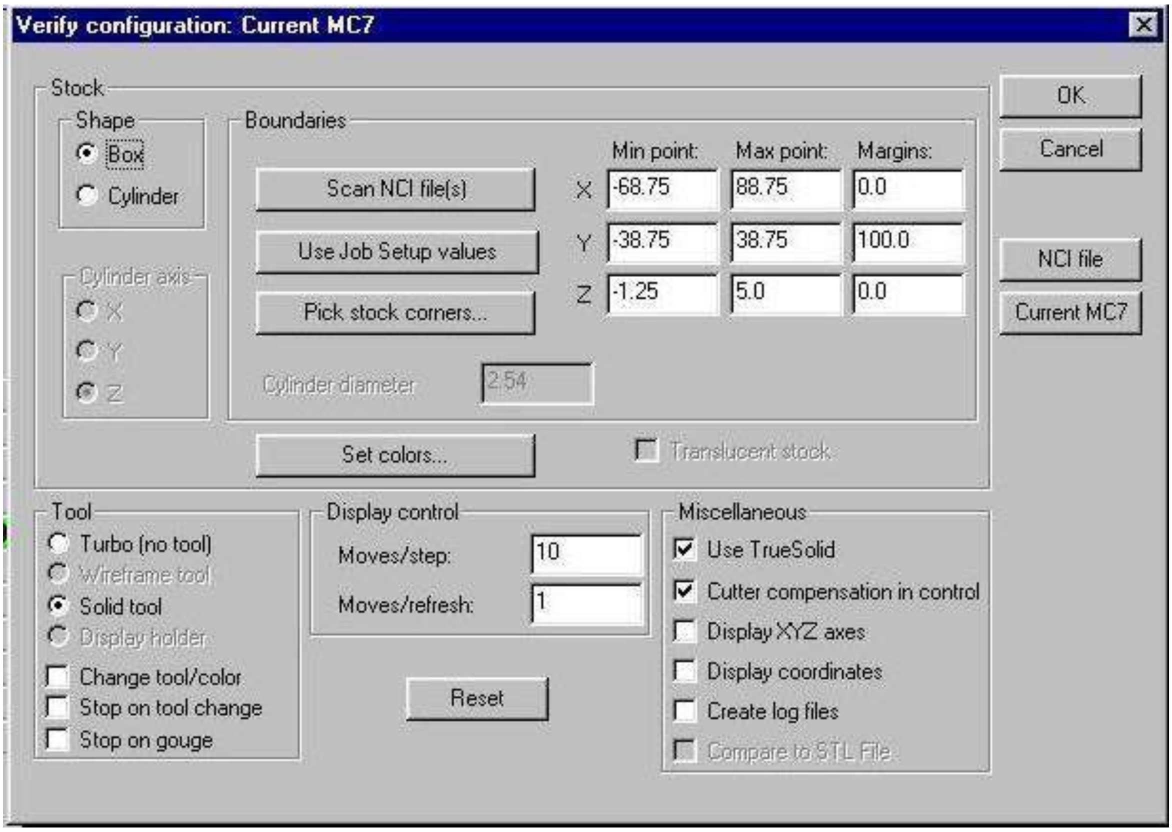
→ •

i. Choose - mainmenu Operation

- A dialogue box appears select all regenerate path
- When the tool path generation completes the dialogue box display
- choose verify, a deluge box is displayed select run (machine) item.

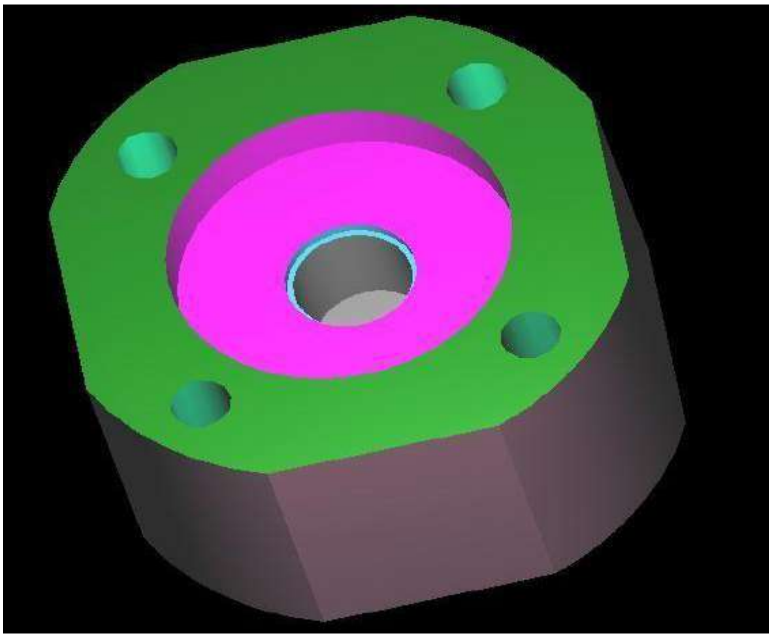


Operation management of Tool Path



Configuration of Tool path

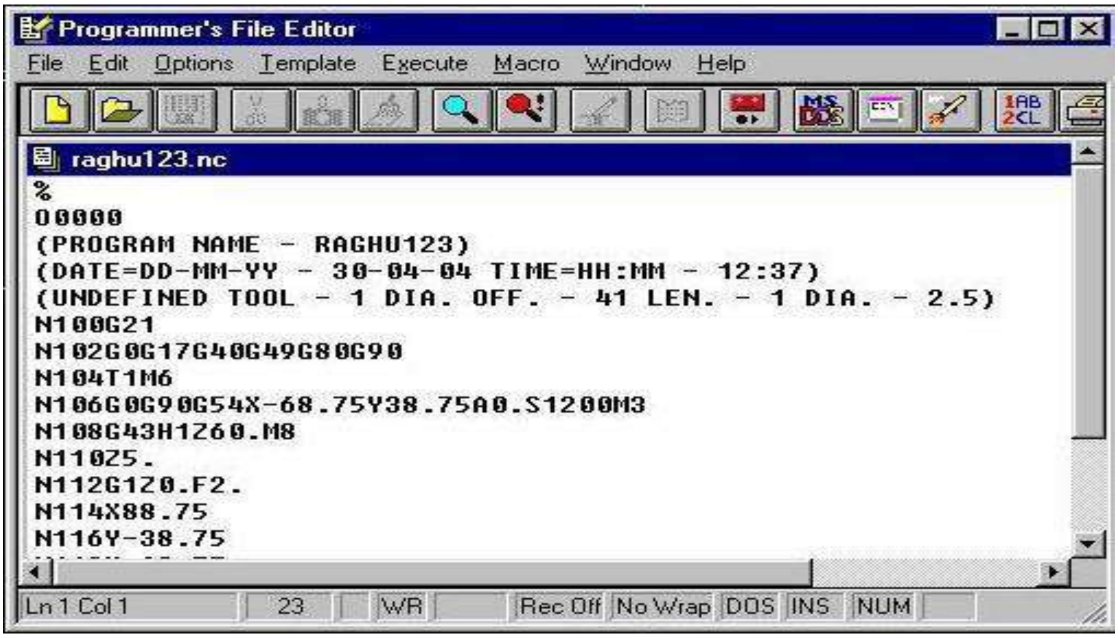
If your model is square/Rectangle go to shape, select box , suppose your model is Cylindrical go to shape, select cylinder



Step 6

Post processor NC Program

- Choose-main menu → Operation
- A dialogue box appears select all regenerate path, choose post
- Show path (desktop) for saving post processor Notepad file.



Result: The required geometry is created

VIVA QUESTIONS

1. **Why tolerances are given to the parts?**

   Because it's impossible to make perfect settings✓

   To reduce weight of the component

   To reduce cost of the assembly

   To reduce amount of material used

2. **What is bilateral tolerance?**

   Total tolerance is in 1 direction only

   Total tolerance is in both the directions✓

   May or may not be in one direction

   Tolerance provided all over the component body

3. **Which type of tolerance provided in drilling mostly?**

   Bilateral Unilateral✓

   Trilateral Compound

4. **What is mean clearance?**

   Maximum size of hole minus maximum size of shaft Minimum

   size of hole minus minimum size of shaft

   Mean size of hole minus mean size of shaft✓

   Average of both size of shaft and hole

5. **Which of the following is incorrect about tolerances?**

   Too loose tolerance results in less cost✓

   Tolerance is a compromise between accuracy and ability

   Too tight tolerance may result in excessive cost

   Fit between mating components is decided by functional requirements

6. **Bilateral tolerances are used in**

   Unitary production

   Mass production✓

   Both Unitary and mass production

   None of the above

7. **The overall width of a part is dimensioned as 3.00 ± 0.02. What is the upper limit?**

   3.00

   +0.02

   3.02✓

   0.04

8. **How may types of Assemblies are used in CATIA**

Top down

Bottom up

Both top down and bottom up✓

None of the above

9. **Abbreviation of CATIA**

Computer Aided Three Dimensional Interactive Animation Computer

Aided Three Dimensional Interactive Application✓ Computer Assisted

Three Dimensional Interactive Application Computer Aided Three

Dimensional Internet Application

10. **Polar coordinates are used mostly for drawing**

Circles

Arcs

Vertical lines Angled

lines✓

11. **When the interior of an object is complicated, which of the following view is used?**

Front view

Side view

Top view

Sectional view✓

12. **When the cutting plane cuts the entire object the section is known as____**

Full section✓

Half section

Revolved section Removed

section

13. **Inclined and offset cutting planes can be used if_____________**

all the hidden objects are not in one line

all the hidden objects are in one line

the single line nor offset sectioning is useful and shape of the object is inclined

it is used for combined objects✓

14. **Crane hook is to drawn by____________method.**

full section

half section

removed section revolved

section✓

15. The section which cuts the object at an angle is called

removed section

broken out section

auxiliary section✓

assembly section

16. The command which is used to set a new layer is called

LAYOFF

LAYVPI

LAYDEL

LAYER✓

17. Modifying a layer consist of

Line weight and line type✓

Thawing

Freeze

Deleting a layer

18. Thin parts like stiffeners, webs, bolts, rivets, etc. are___________if they are cut by the cutting plane along their axis.

Not hatched✓

hatched

sectioned

Not sectioned

19. Which command is used to divide the object into segments having predefined length?

Divide

Chamfer

Trim

Measure✓

20. Status bar do not contain

Snap

Grid

Erase✓

Polar

21. **Which mode allows the user to draw 90° straight lines**

Osnap

Ortho✓

Linear

Polar tracking

22. **To obtain parallel lines, concentric circles and parallel curves;____________is used.**

Array

Fillet

Copy

Offset✓

23. **The default grid spacing in both X and Y directions is:**

10✓

20

5

15

24. **Scale command can be accessed easily by typing**

SL
S
SC✓
C

25. **What is the save extension of the sketcher file in CATIA?**

CATPart✓

CATIAPart

CPart

None of the above

****