Skill Development Training Ready 261
Project report

Submitted

In partial fulfillment of the requirements for the award of the

Degree of

BACHELOR OF TECHNOLOGY

In

AGRICULTURAL ENGINEERING

By

A.NAGARAJU
(CAES/2016-023)

PROFESSOR JAYASHANKAR TELANGANA STATE AGRICULTURAL UNIVERSITY.

COLLEGE OF AGRICULTURAL ENGINEERING, KANDI, SANGAREDDY, JUNE2018

COLLEGE OF AGRICULTURAL ENGINEERING, KANDI SANGAREDDY

B. TECH (AGRICULTURAL ENGINEERING)

CERTIFICATE

This is to certify that **A. NAGARAJU (CAES/2016-023)** B. Tech (Agricultural Engineering) students of College of Agricultural Engineering, Kandi, Sangareddy, Professor Jayashankar Telangana State Agricultural University, have successfully completed their project at College of Agricultural Engineering, Kandi, Sangareddy. The duration of the project was from JUNE to JULY of 2018.

DR.CH.BALACHAND
Dept. of FMPE

Er. CH. SHRAVAN KUMAR
HOD and Assistant Professor (Dept. of FMPE)

Dr. G. MANOJ KUMAR
Associate Dean and University Head (SWCE)
CAE, KANDI, SANGAREDDY

S.NO	DAY	ACTIVITY
1	22-06-2018	introduction

2	23-06-2018	sketcher(line,circle,rectangle,spline,arc,ellipse)
3	24-06-2018	practice
4	25-06-2018	sketcher(filllet,chamfer,text,offset,thickness,project)
5	26-06-2018	sketcher(modify tools)
6	27-06-2018	projection views,extrude,revolve
7	28-06-2018	sweep,helical sweep,swept blend
8	29-06-2018	blend ,rotational blend,round,hole,chamfer
9	30-06-2018	draft,shell,rib,pattern,mirror,trim,intersect,boundary blend,falltern
10	1-07-2018	practice
11	2-07-2018	assembly
12	3-07-2018	assembly
13	4-07-2018	drafting
14	5-07-2018	drafting
15	6-07-2018	sheetmetal
16	7-07-2018	sheetmetal
17	8-07-2018	practice
18	9-07-2018	ansys introduction
19	10-07-2018	engineering data
20	11-07-2018	geometry
21	12-07-2018	static structural
22	13-07-2018	modal analysis
23	14-07-2018	steady state thermal,couple field analsis
24	15-07-2018	practice
25	16-07-2018	harmonic response
26	17-07-2018	explicit dynamics
27	18-07-2018	cfx fluent
28	19-07-2018	project
29	20-07-2018	project
30	21-07-2018	project

INTRODUCTION:

Parametric technology corporation (PTC) is a U. S based company founded in year1985, it develops markets and supports software for product development. Its main product is for CAD /CAM, engineering calculations, and products lifecycle management

In design we get the perfect analysis of the object before manufacturing to avoid errors.

CAD: Computer Aided Drawing

Computer Aided Drafting

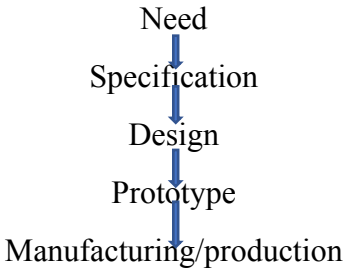
Computer Aided Design

Drawing: A rough sketch without any dimensions.

Drafting: Giving dimensions to the above rough sketch as per standards.

Design: Her we add the 3rd dimension to the above rough sketch.

DESIGN CYCLE



Need:

Requirement according to the market. Ex: Water bottle.

Specification:

Specification of the water bottle is specified her. (ex: Capacity, Quantity, Budget, color)

Design: It is divided into 3 types

- CAD (computer aided design
- CAE (computer aided engineering)
- CAM (computer aided manufacturing)

CAD:

Here we design the water bottle Using the Desired software's likes Auto Cad, Pro-E, CATIA, UNIGRAPHICS, Fusion 360

CAE

Here we do analysis for the water bottle by using the software like Ansys by applying the external loads on it.

Cam

Here we generate the CNC programming for the manufacturing. Here we can analysis the time taking process for the manufacturing

Prototype:

Here we manufacture the sample water bottles for the testing.

Manufacturing/ Production

After we go for bulk manufacturing. After the prototype testing.

PRO E Advantages

- Parametric features
- Feature based
- Association and model centric
- Parent and child relationship

Parametric features

Here we use the dimensions for creating the figures and edit the dimensions as per the requirement.

Feature based

The features are applied individually to the parts.

Association and model centric:

Here there will be a chain connection Between the Sketcher, Part design, Assembly, Drafting.

If dimension is changed in sketcher automatically the dimensions are changed in part deign, assembly, drafting.

Parent and relationship Child:

Here sub components are depending on the main component. If main component is deleted the sub component is also deleted.

Pro E

How to start

Go to start all programs PTC Creo parametric 3. 0 mo1o click on new select the part click on ok now select the solid part mms

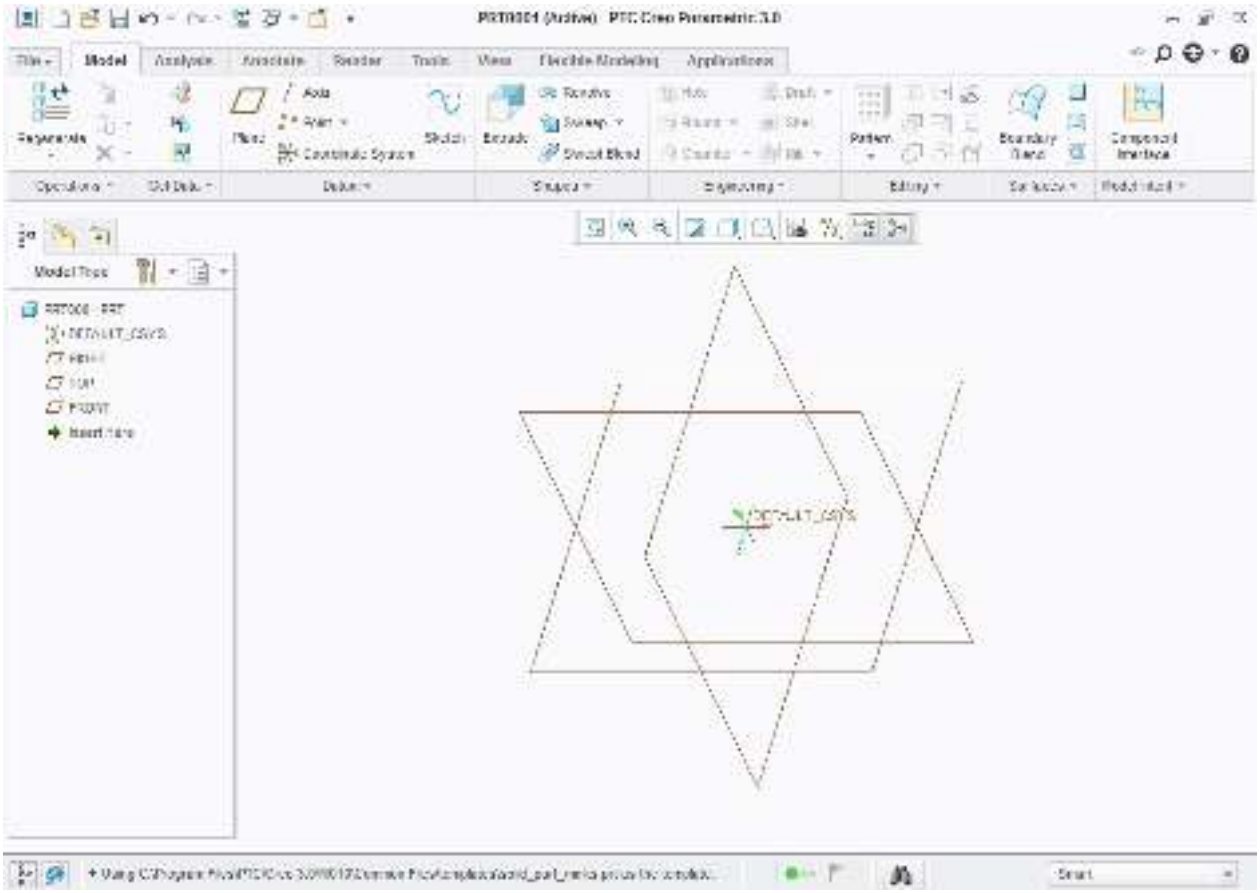
Navigation tree: here step step by step process of the diagram is recorded here we will be having three different type of planes known as right, top, front now select any one plane click on sketch

How to operate the mouse?

To rotate, click on center button and drag the mouse.

To Pan click on shift button and click on center button.

To Zoom in & Zoom out scroll the center button.



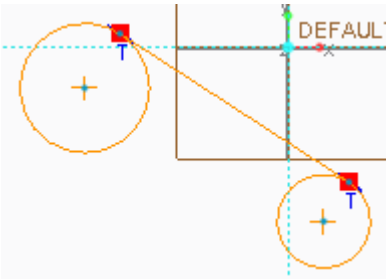
SKETCHER

Her we draw all the 2D profiles for the 3D objects

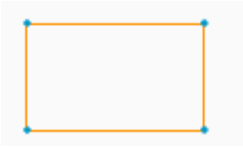
For sketcher select the required plane (Front, Right, Top) and click on sketch

Line: create continues line

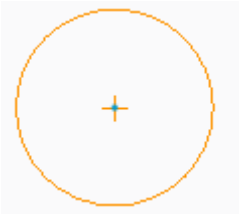
Line tangent: its connect the circle and line



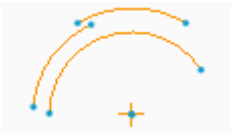
Rectangle: slanted rectangle, corner rectangle, center rectangle, parallelogram.



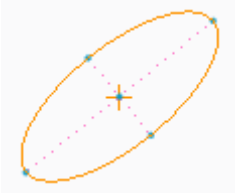
Circle: center and point, concentric ,3-point, 3 tangents.



Arc: 3-point arc, center and ends ,3 tangents, concentric, conic

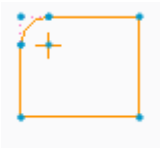


Ellipse: axis and ellipse, center and axis ellipse

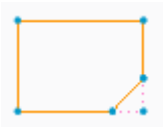


Spline: it is a curved line used to draw pipe shapes.

Fillet: circular, circular trim elliptical, elliptical trim



Chamfer: it used to make equal distance for easy measurement.



Chamfer trim: we assume it is a angle

Text: for writing the text

Offset: creating the multiple lines by taking the reference from the original lines according to the direction of arrow you need the dimensions for the loop select the line give the value single: offset created for the single line chain: offset created for the multiple lines & closed lines loop: creating the offset for the number of times

Thickness: to add the thickness to the drawn profile

types single open circular

End caps: open flat circular

Center line: it is a construction line or reference line

modify: here we modify the dimension click on the modify option select the dimension to modify it

delete segment: here we remove the unwanted lines

vertical and horizontal line: we can make an inclined line to vertical and horizontal

tangent: to draw the tangent line

mid-point: select the midpoint option select line one and line two point

mirror: draw the object center lines now select the object click on mirror option

pallet: it is for different polygon shapes

corner: select line then joint at corner

rotate

coincidence

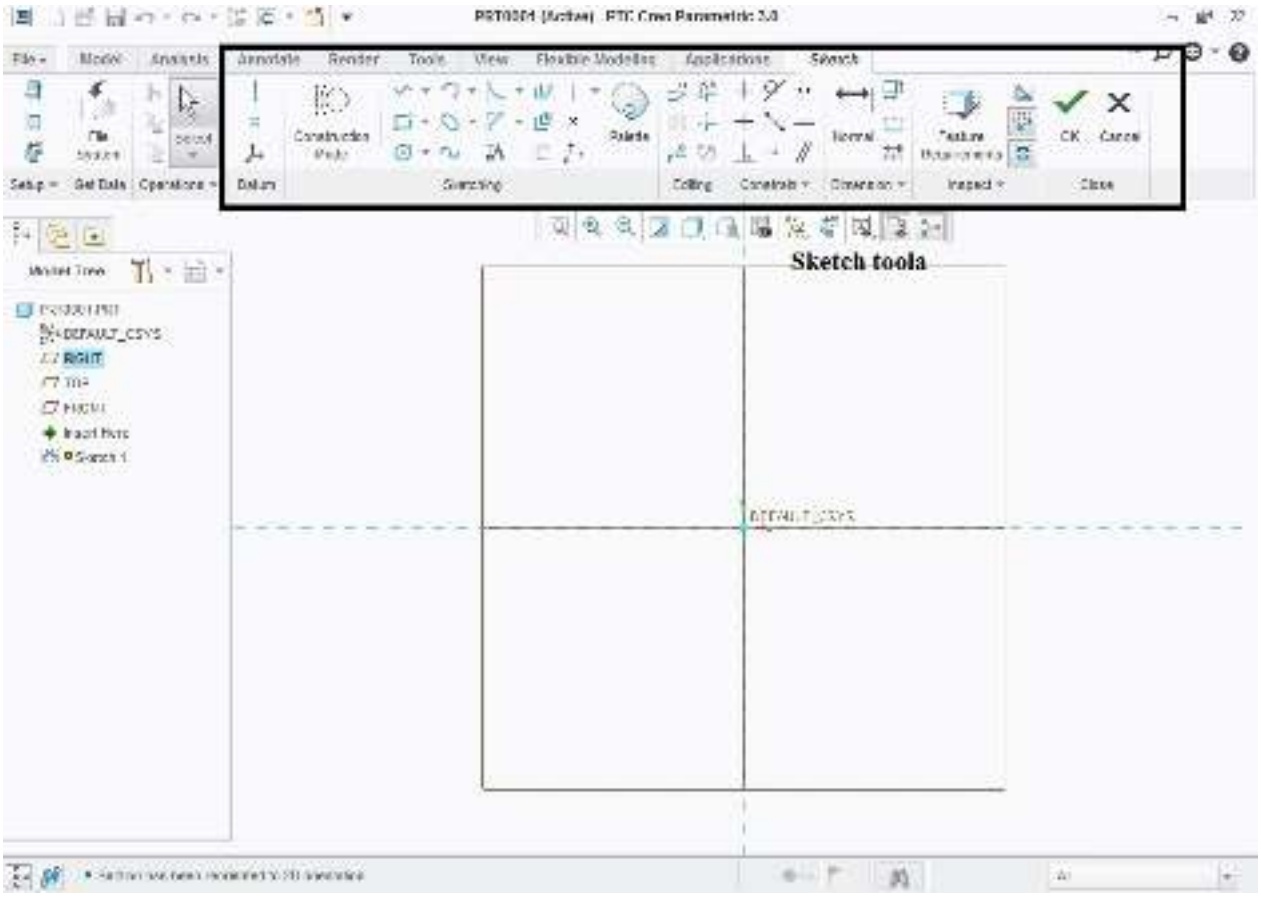
equal

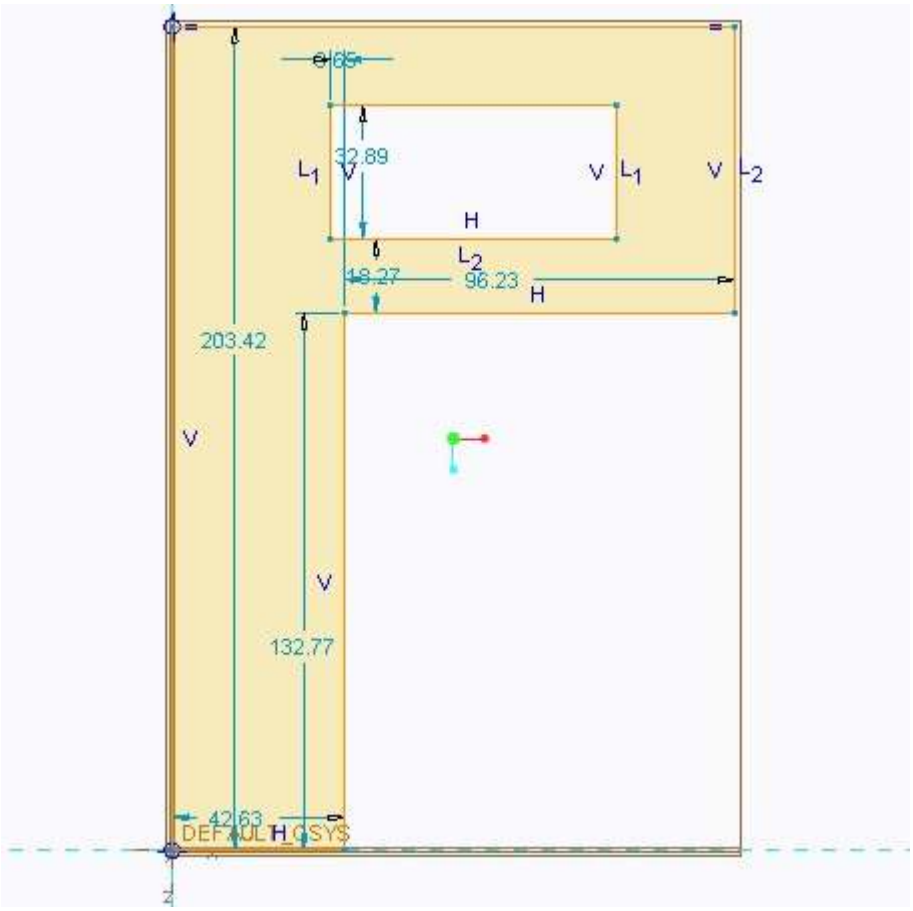
divide: break the profile

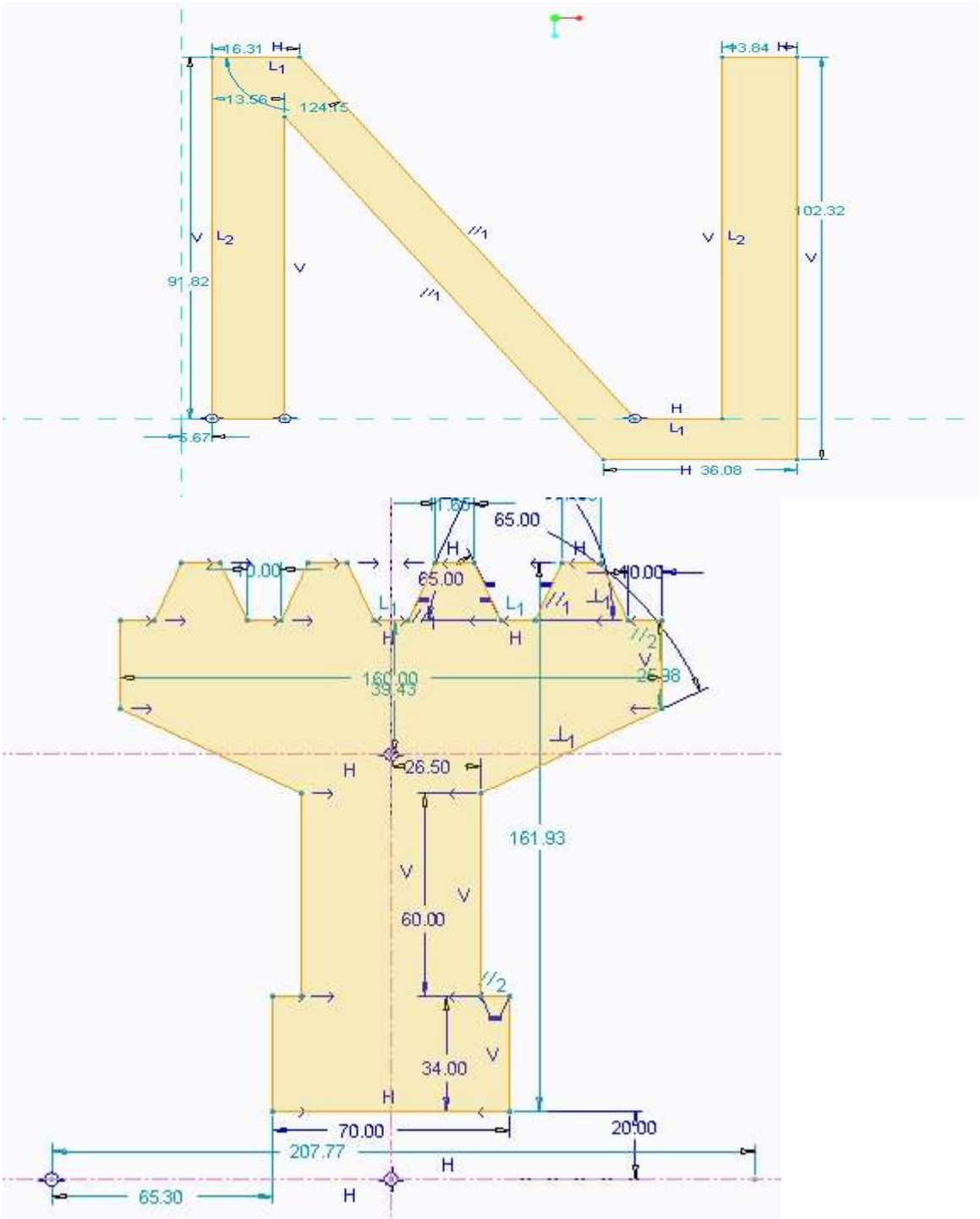
symmetry: in symmetry dimensions will be equal halves in the both sides

normal: here we give constants to the object with respect to the center line, after giving the dimensions using normal option, dimension color changes to violet

overlap: it overlaps the two circles, combined and appears to be one.

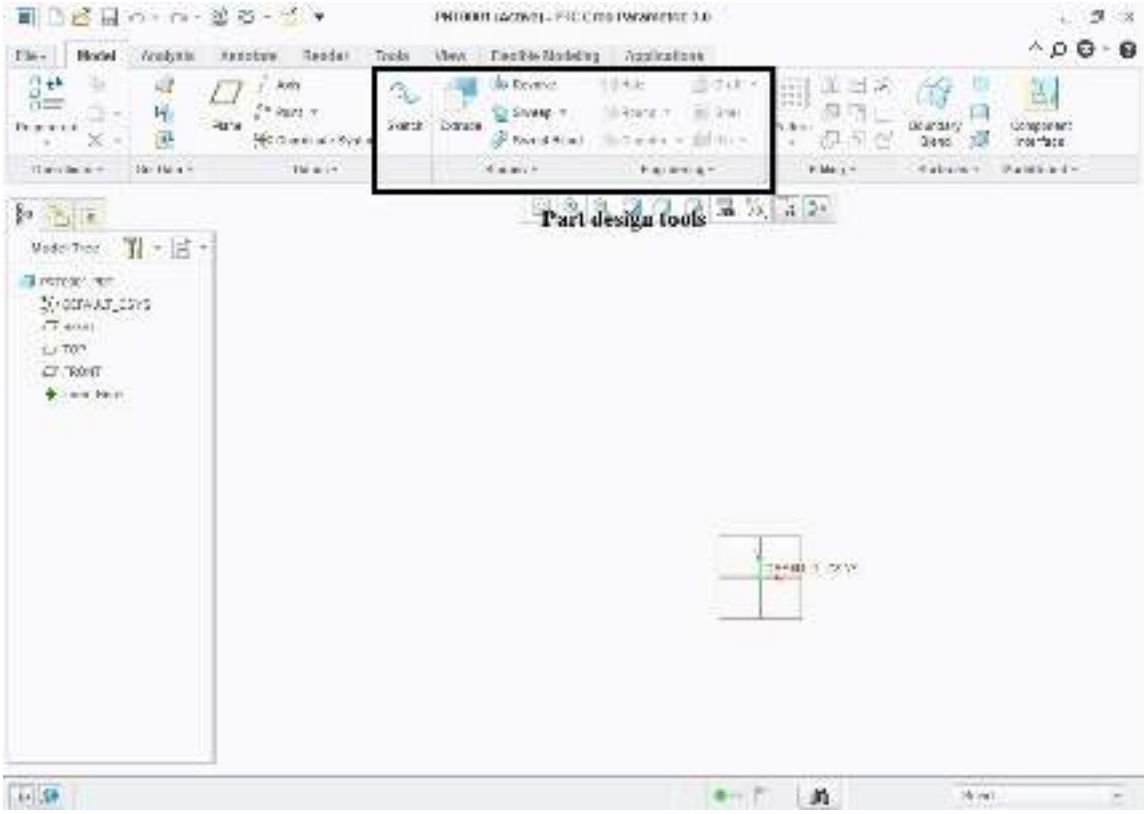



6.65
32.89
L1
V
V
L1
V
L2
H
L2
48.27
96.23
H
203.42
V
V
132.77
42.63
DEFAULT CSYS



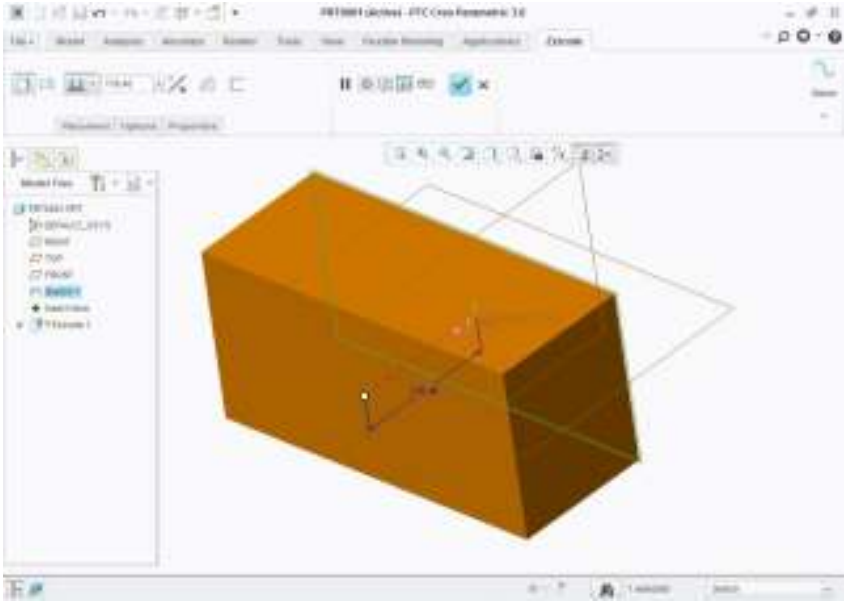
Practice of 2d profiles

Part Design:



Extrude

Creating a 3d geometry from the 2d profile in linear directions



Step by step Processes for extrude

1. Select a plane

2. go to sketch

3 draw a profile

4 click ok (convert into 3d)

5 go to extrude option

6 give dimensions

7 click ok

Types of extrudes

1. Depth value: here we specify the value

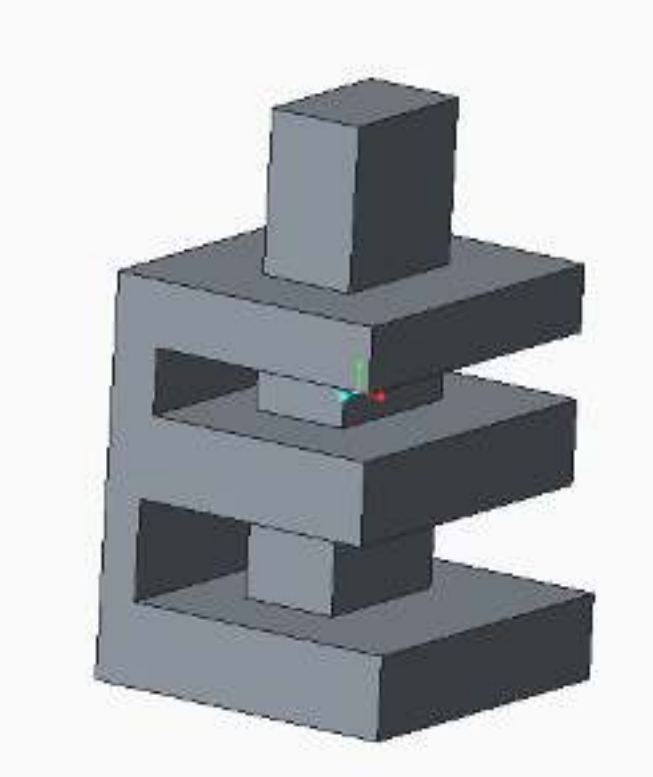
2 Symmetries: here materials have symmetry structure

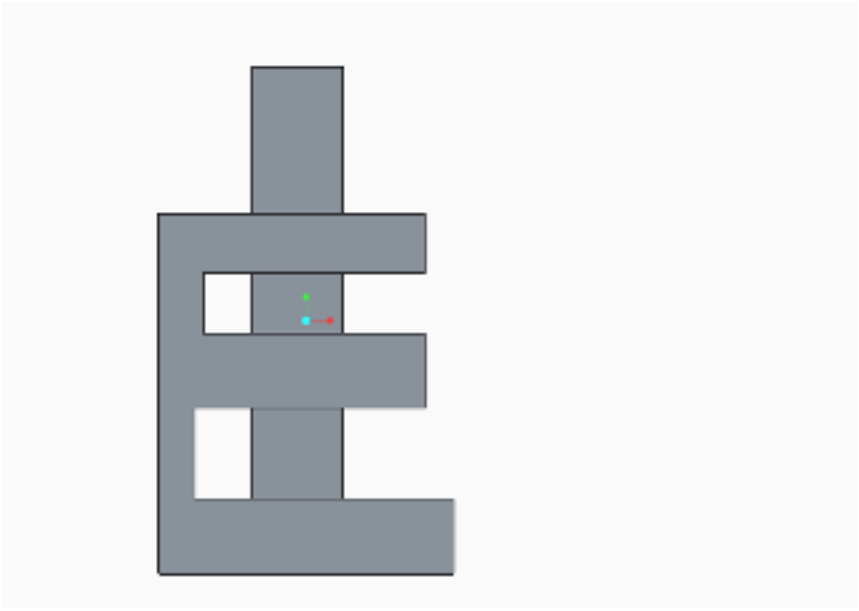
3 Up to next surface: here automatically material is added to the next corresponding surface

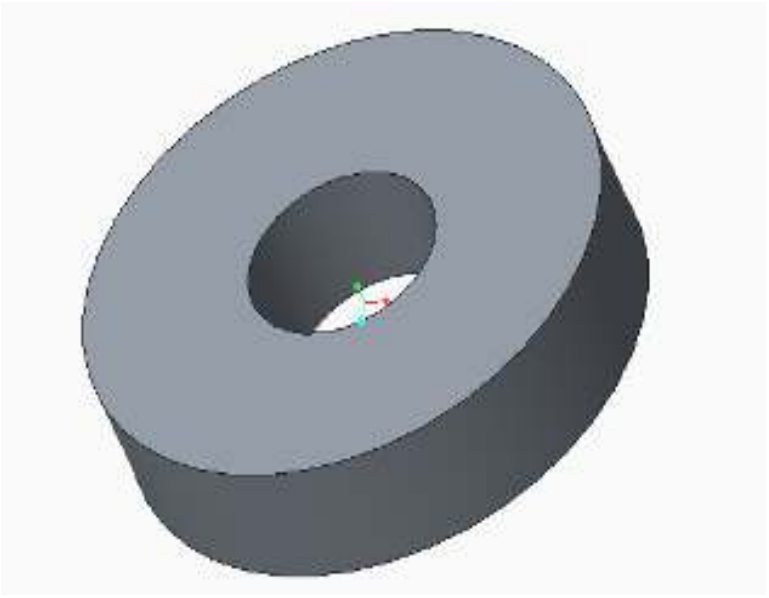
4 Through all surfaces: here material is added to the last surface which passes through all surfaces.

5 Through selected surfaces: here material is added to specified surface

6. Up to point, curve or plane surfaces: Here material is added to the selected plane.

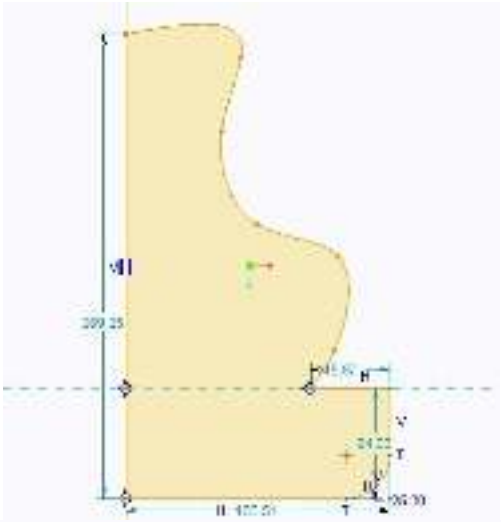


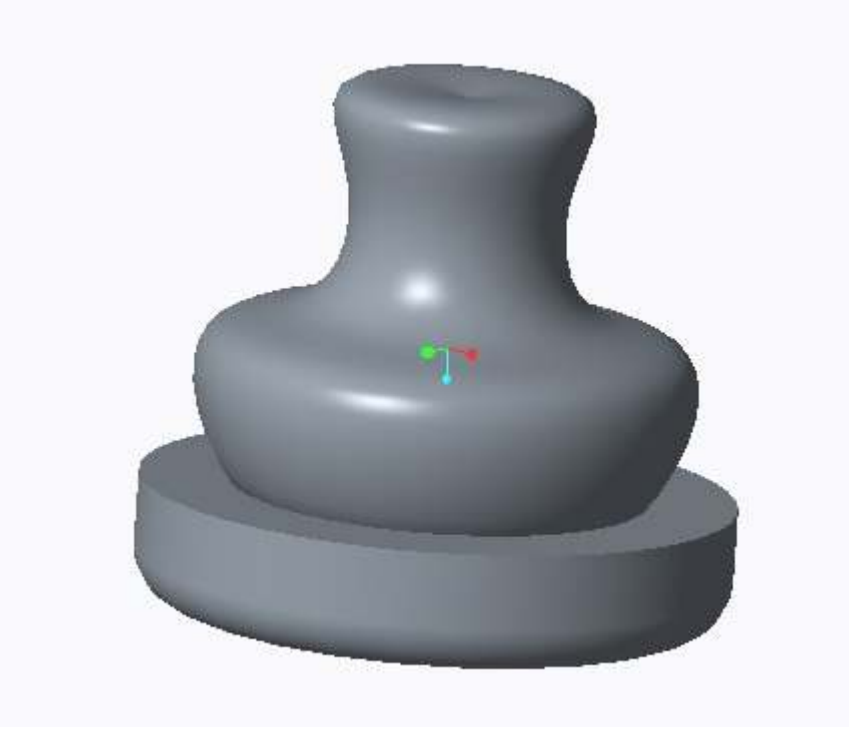




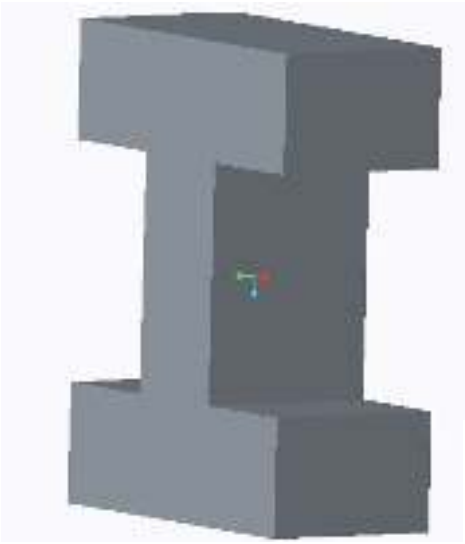
Revolve:

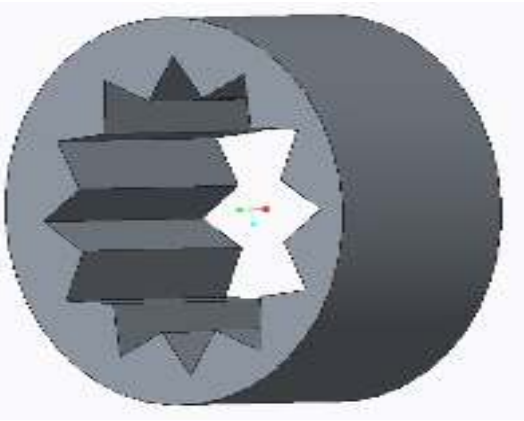
1. Here we need to draw the one to the fourth view of the diagram
2. Creating a 3D profile in 360 degrees by selecting line as your center axis.
3. Go to revolve option select the profile automatically diagram is created hollow with reference to axis.
4. If you required solid diagram, select the required axis.

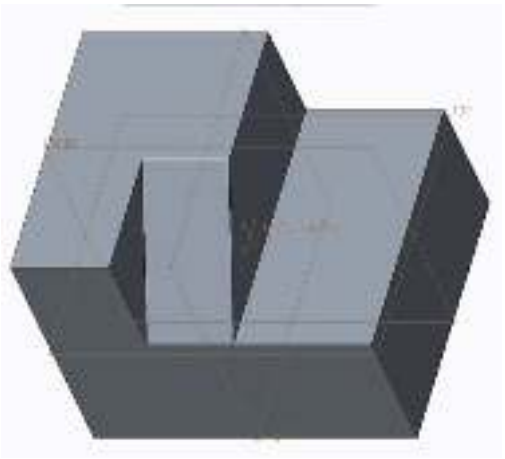


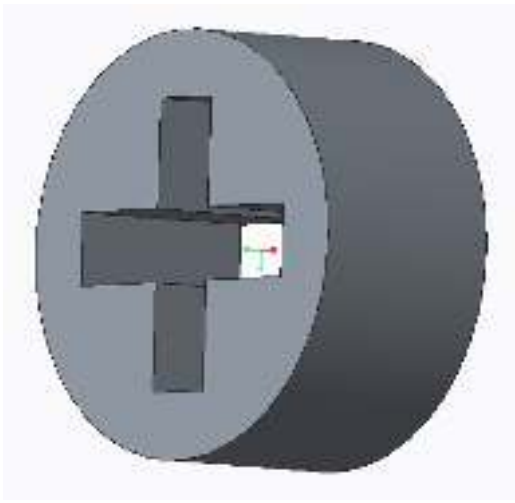


Extrude practice:

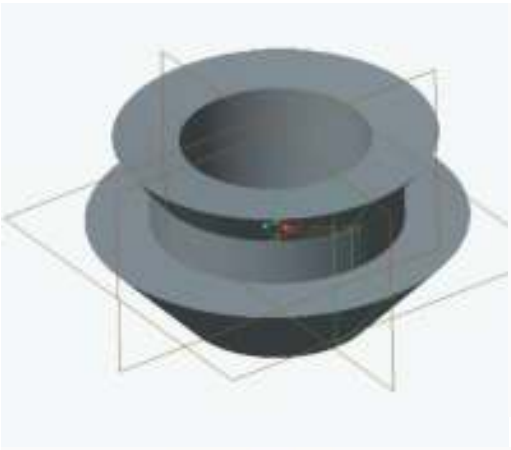


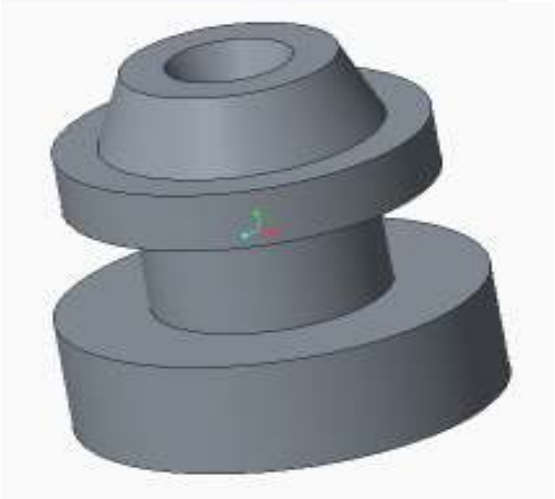






Revolve practice



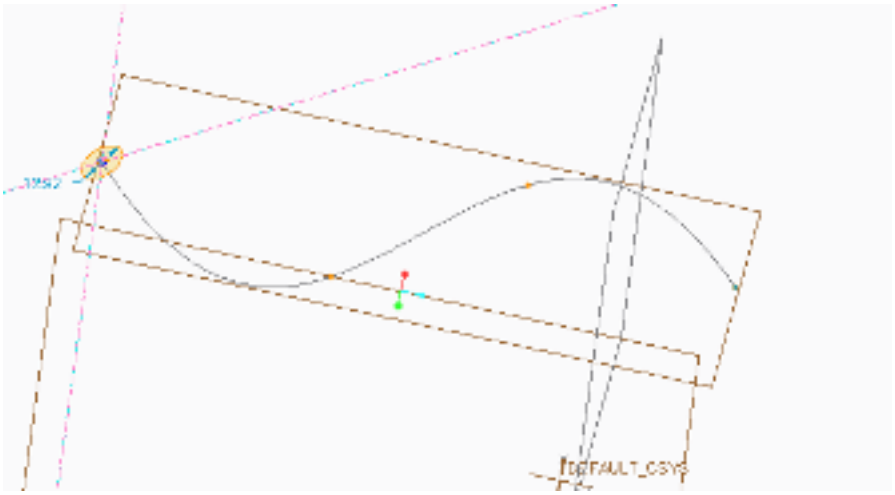


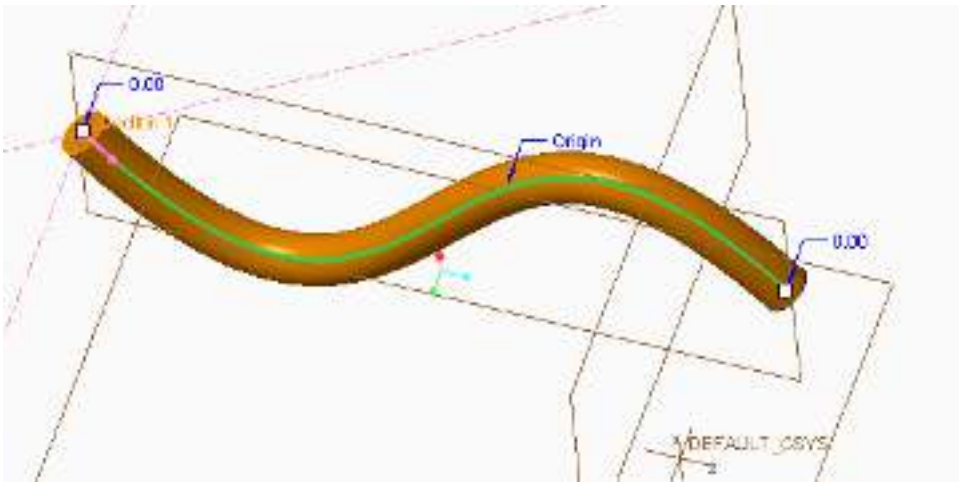
Sweep

Creating a 3D model using the center path or guide curve.

Process:

1. Select a plane
2. Go to sketch.
3. Select sweep option.
4. Select curve
5. Go to sketch.
6. Draw a profile.
7. Click ok.





Multiple sweep:

1. Draw the three lines
2. Select the sweep option
3. select the lines (by clicking on control)
4. go to reference, select the central line as the neutral and click on sketch.
5. draw the profile but the profile must be need the end points.

   (first guide curve and last guide curve)
6. click on ok.

Helical sweep:

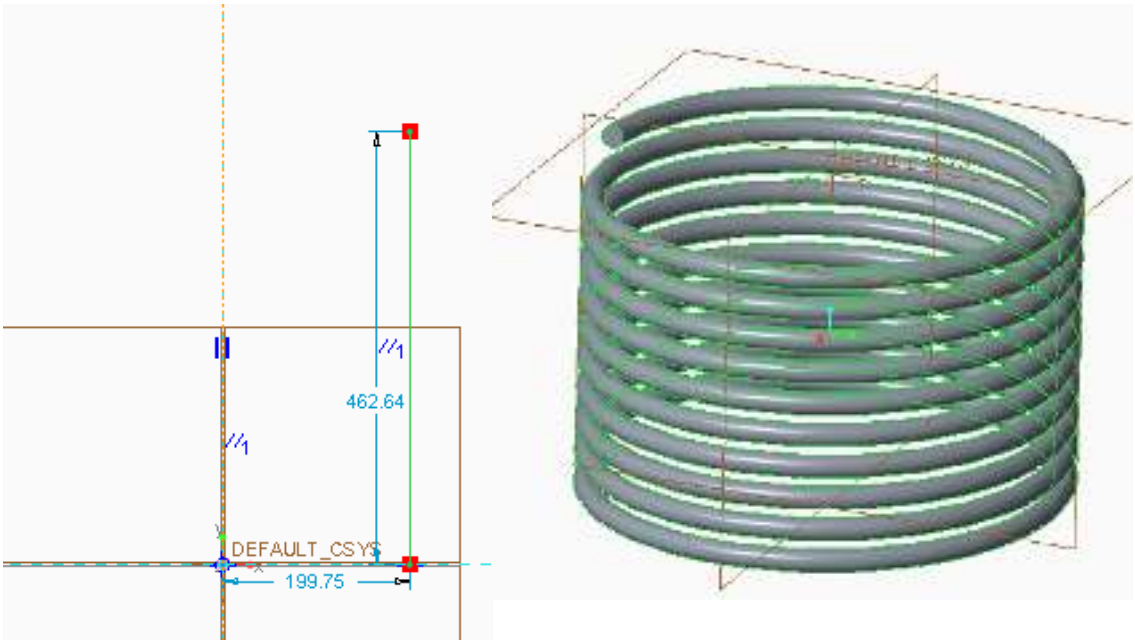
Draw the line and Center line

Go to helical sweep.

Select the line and click on sketch.

Draw the profile (circle)

Click on ok.



Swept blend:

Draw the center curve, click ok.

Go to swept blend option, select the curve.

Go to tool bar click on sketch (first profile)

Now draw the profile rectangle (rectangle having 4 vertices), click on ok.

Go to section insert (for the new sketch).

Click on sketch (second profile).

Draw another profile (number of vertices of the second profile should meet the count of the first profile vertices).

For the creating number of vertices for the profile select the divide option in sketcher and create the vertices and click on ok.

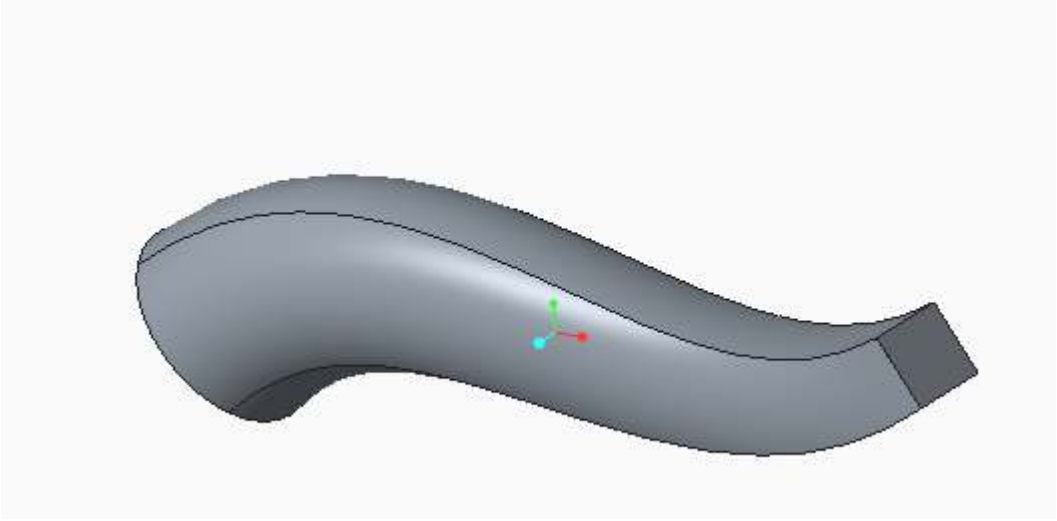
If the is not smooth, check the starting point of the first profile and check starting point of the second

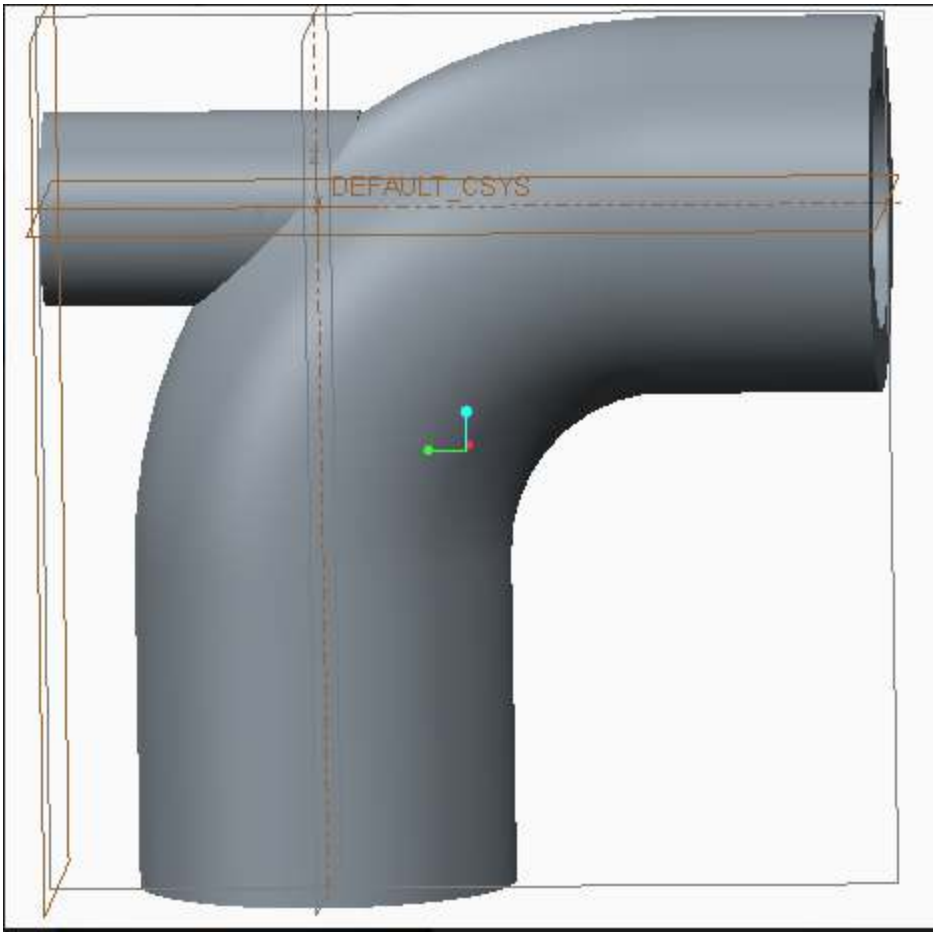
To change the starting point, go to section select the required profile click on sketch.

Change the starting point by clicking on another point right click starting point.

If you want to create any profile at center of the curve.

By selecting datum, select the real option and click on that then select the pointed line.





Blend:

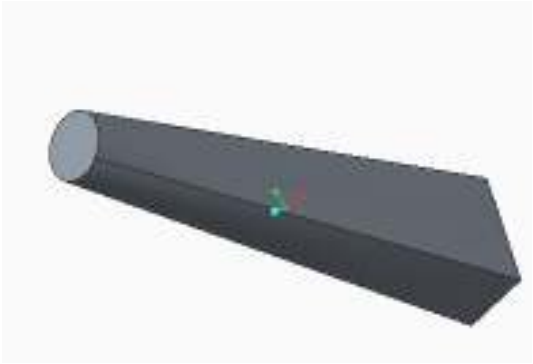
Select the blend, go to sections, click on define.

Now select the plane where you want to draw the profile click on ok

Automatically center distance has been formed go to sections specify offset distance, click on sketch

Now draw another profile (there should be equal number of vertices) click on ok

If you want to create the odd another profile clicks on sketch and draw .

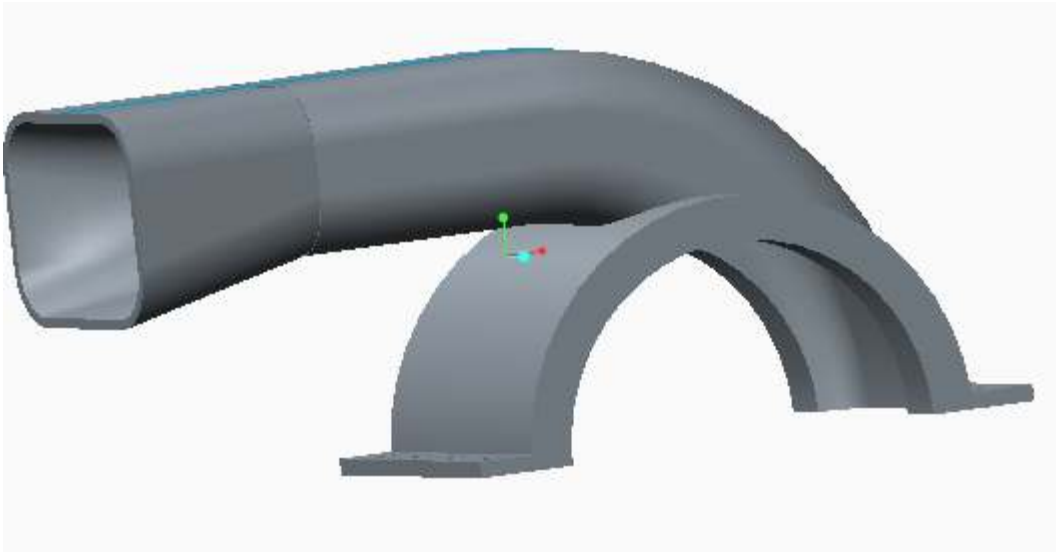


Rotational blend:

1 select the blend, click on sketch draw the profile, (1st profile) and draw the central axis (it acts as a center for the profile too create an arc).

2.click on ok. Select the rotational blend option.

3.select the rectangle, automatically arc will form from the rectangle center. here arc is formed up to 120 and then go to sections and click on sketch then draw the profile, click on ok.



Hole:

This option is used to create predefined holes like counter bore, countersunk, counter drilled

1. Create a solid body go to hole option, select the face where you require hole.
2. Now, give the dimensions to that by fixing the green color points to edges of the diagram. so that here the standard hole is formed.
3. You can specify, diameter, depth value and angle of v point.(in the shape option you can edit everything).

Predefined standard holes:

1) Counter sunk

2) Counter bore

3) Counter drilled: combination of counter bore and counter sunk.

Round: It is used to create to smooth edges for the solid diagram.

Face fillet:

1. select the first face and control
2. Select the opposite parallel face and row
3. Click on control and select any one face

Variable radius face fillet:

1. Select the edge of the diagram, here we can create the fillet with different radius
2. Go to sets option at the bottom of that radius dialogue box, give right click at radius and radius will add at the corner of the edge.
3. here we can add second radius to the corner of the edge. if we require fillet at specified point create the point by datum. again, go to radius tool bar for the 3rd radius select the location as the reference, now select the that point

Auto round:

fillet will come all the edges except the selected edge.

Chamfer: create the sharp edges by specifying the length and angle.

Types:

Distance * distance

Angle * distance

Different distance

45 * distance

Corner chamfer: the chamfer is created at the corner of the solid body

Draft: go to reference select the draft surface, and, select the draft hinge and row select the pull direction.

Draft split: after crating the draft go to split option, select the option split by split object. this option is used to split by drawing the line. this option is applicable for square object.

After selecting the split by split object select the define and select the plane where you have taken the draft surface area.

Shell: it is used to create the hollow from the solid diagram by adding the equal thickness

Rib:

profile rib: go to the profile rib option, go to reference, click on define, select the plane, click on sketch, now draw the open profile.

Trajectory rib: draw the solid body, create the shell for it, go to the trajectory rib, go to placement, click on define select the face click on sketch, draw the open lines, click ok

Pattern: creating the instantons from the original object

Types:

Dimension: here we can give the values to the profile

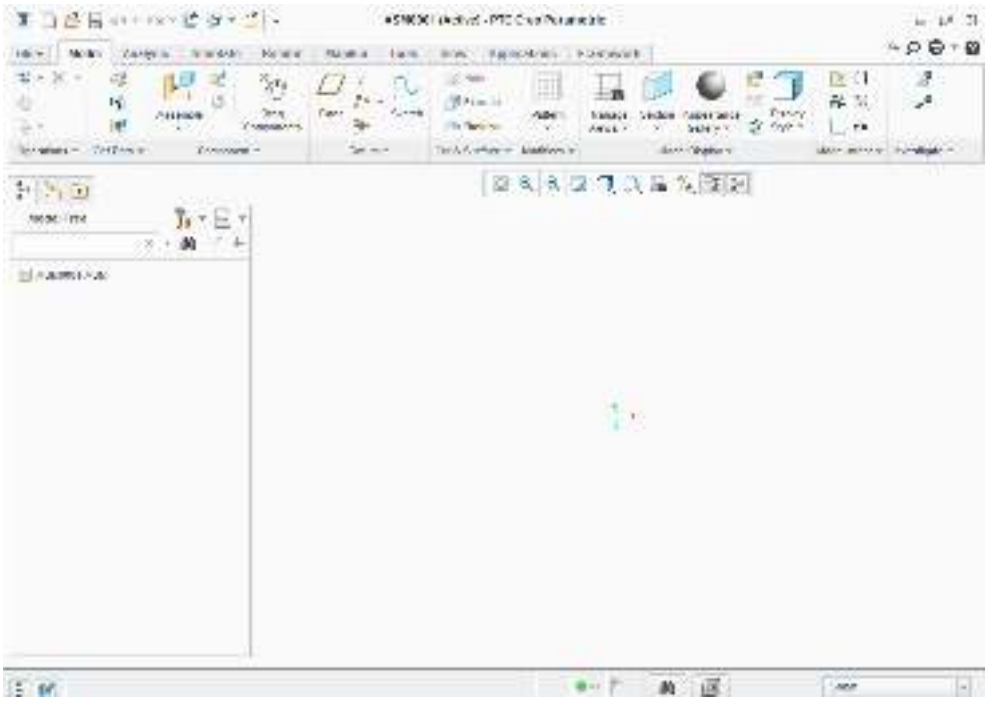
Direction: this option is used to create the instantons.. in linear directions.

Axis: this option is used to create the instantons in 360 degrees. We need to select the circle center axis.

Fill: use to fill the intaniuus in a specified shape, for this we need to create the define shape for instantons.

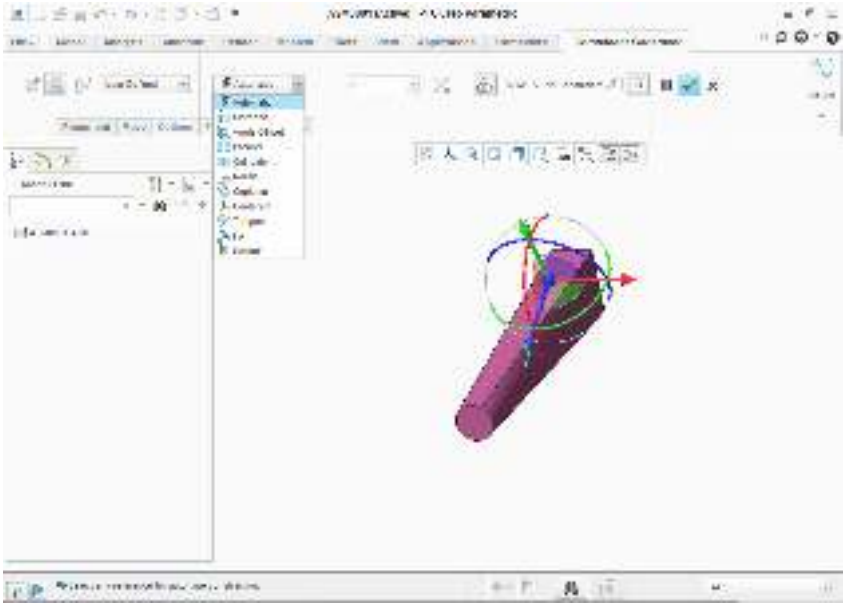
ASSEMBLY

First select the moving object, next the fixed one here we need to combine all the 3d parts together to form final products. For the assembly we need to open the main body, we need to fix that component. Here we can move the components.



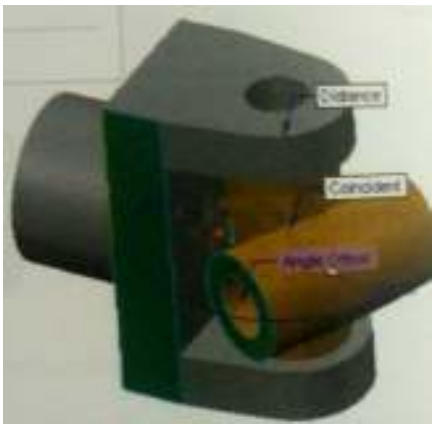
Types of constraints:

1. Coincident
2. Distance
3. Parallel
4. Angle offset
5. Normal



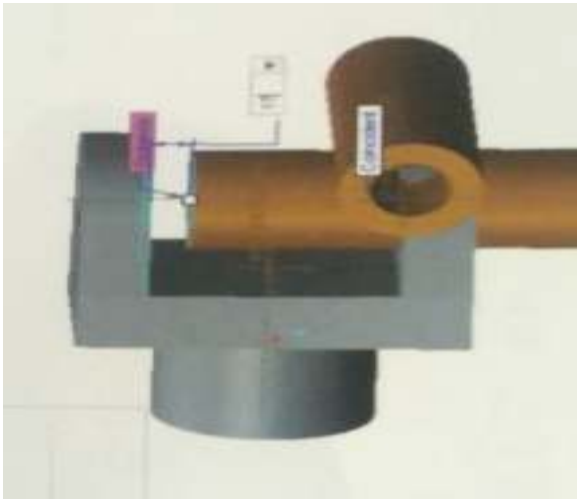
Distance here we can do assemble for two components at a certain distance by selecting two parallel faces or two parallel planes.

Angle offset: here angle is created between two faces

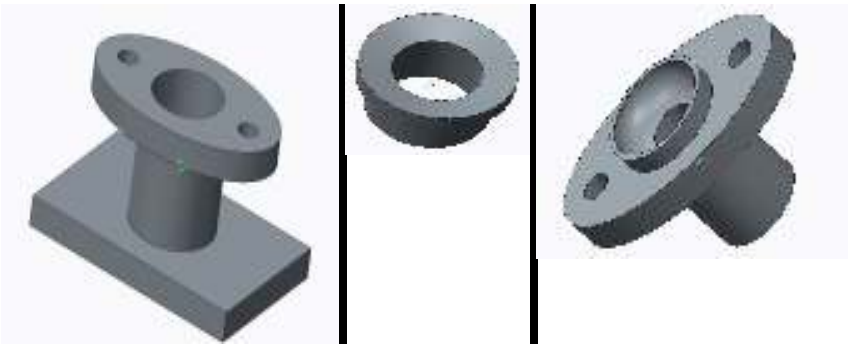


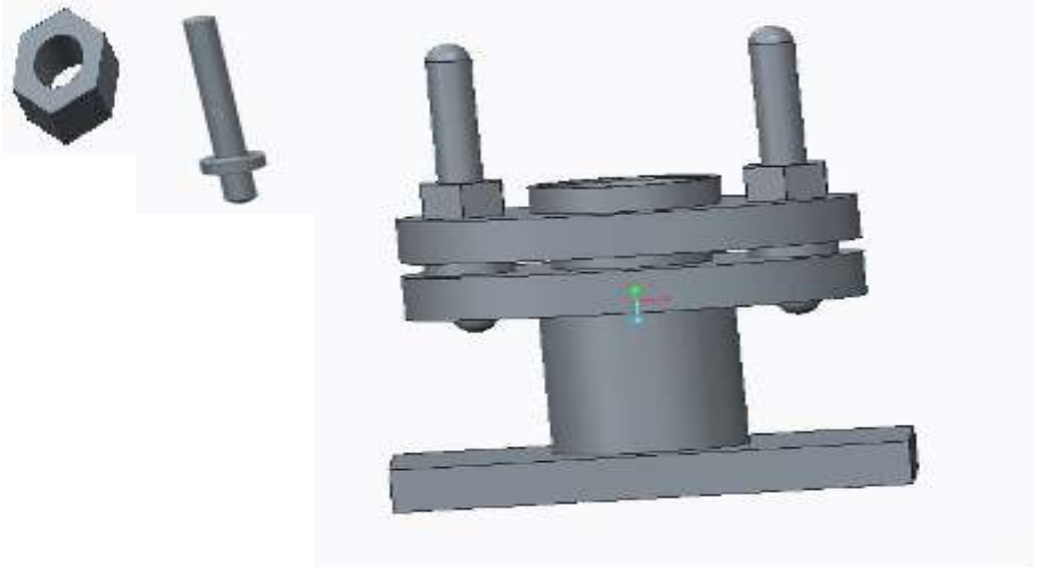
Parallel: here we create the two faces parallel

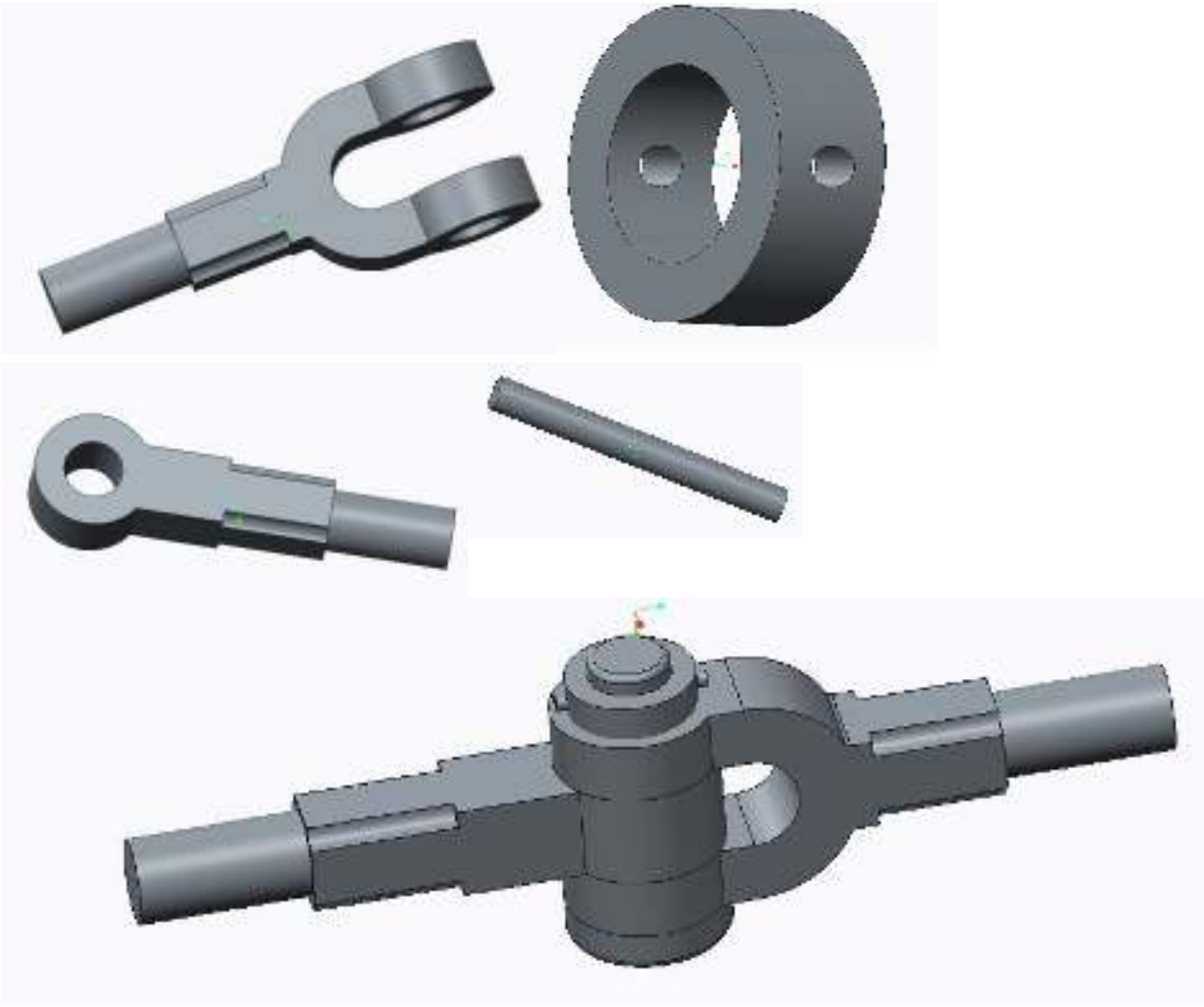
Coincident: this option is mainly used for meeting the two faces at zero distance and meeting two circular center axis



Tangent: this option is used for one circular object and one straight object.







DRAFTING:

Here we can generate the views from the 3d component to add the datum planes for 3d object

1.open the 3d model go to annotate select the datum target annotation feature

2. select the feature set datum, click on ok type the name select option as geometry select the face where do you want to apply the datum click on ok

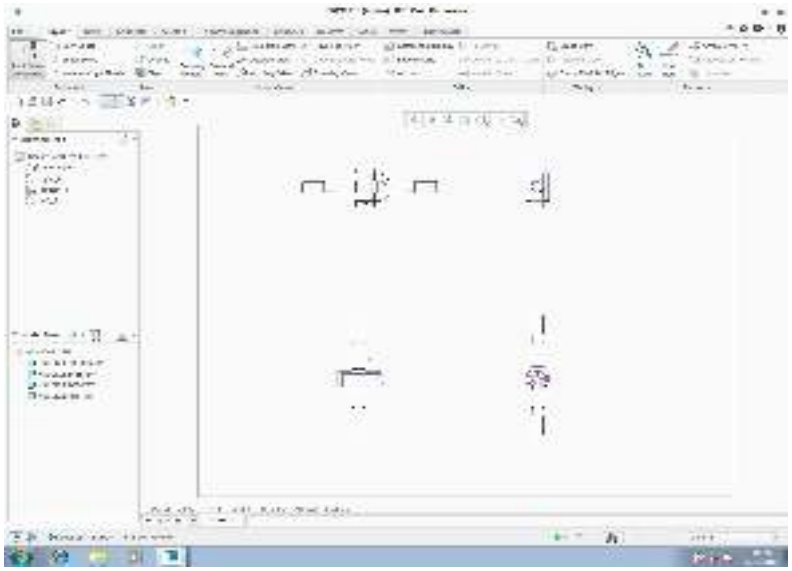
3. open file new drawing then click on ok select the empty with format select the appropriate sheet, click ok, after selecting the sheet, go to general view, select the no combine state and click ok.

4 click on page automatically the view is generated go to view display select the display as no hidden and select tangent edges displays style as dimmed apply.

PROJECTION VIEW: here the projection is generated as per the selection of view

DETAILED VIEW: this option is used to show smaller parts in detailed after selecting the option, click on point where we create, then create a profile click on object, after creating profile click on center button where do we want to place the view just click there.

ANNOTATE: go to annotate tool bar, select the show model annotate symbol.



SHEET METAL: to convert the solid diagram into sheet metal create the solid part in part design

Go to operation select the convert to sheet metal option, in that select the shell option

Select the face where you require to apply the shell

Automatically page is converted to sheet metal

Conversion: this option is used to convert the solid into sheet metals

Tools:

Planar: here we add the thickness to the profile

Flat: creating the bend for only one edge

Flange: creating the bend for multiple edges

Rip:

Edge rip: it divides the solid body into sheets by selecting the edges

Form:

Punch form: create the solid body in part design save the file go to the sheet metal create the sheet, go to the punch form, open the punch, assemble the punch to the sheet .it should be fully constraints

Sketched form: here we can draw the punch as you required.

Bend:

Edge bend: creating the radius at the edge.

ANSYS

Introduction:

Ansys: It was introduced by “swanson “in the year 1974.

Experimental:

Here we get the accuracy results using meshing for various objects due to conditions applied. If we need to change the material properties of object manufacturing cost increases for testing and for the number of components.

Theoretical

By using the FEM methods by the hand calculations deformation values are obtained but it is difficult for the complex shapes to obtain accuracy results but it takes lot of time for the results.

Numerical:

It is by using finite element analysis we can obtain the more accuracy results within no time using axis software. But it is one-time investment.

FINITE ELEMENT ANALYSIS (FEA):

It is a numerical behavior of engineering components accused by machine in mechanical components into finite number of building blocks called nodes and elements and analyzing its durability based on its boundary conditions.

Element:

It is small regular geometric shape.

Nodes:

Elements which are interconnected at joints.

Uses of FEA:

Cost saving (reduces the amount of prototype)

Time saving (reduces time to market)

Finite Element Methods(FEM)

It is a numerical method for solving problems of engineering and mathematical physics for type of analytical solutions of these problems generally require the solution to boundary value problems for partial differential equation.

Methods of FEM:

1. Galkerin method
2. R.R method
3. C.S.T method

Types of mesh:

Free mesh:

Here mapping is generated with irregular shapes.

Mapped mesh:

Here for meshing the number of mesh elements are indicated.

FEA design:

Problem classification:

here the analysis is determined whether it is time dependent (dynamic) and time independent(static).

Preliminary analysis:

In this before solving a model we need to have a hand calculation.

FEA analysis:

There are three steps in FEA

Preprocessing:

Here we generate the machine for the analysis. Nodes and elements are generated.

Analysis:

here the data is prepared for the preprocess.

$$\{K\}\{U\} = \{F\}$$

Where,

K = stiffness matrix

U = nodal displacement vector

F = nodal force vector.

Post processing:

Here the stress results are obtained.

Check results:

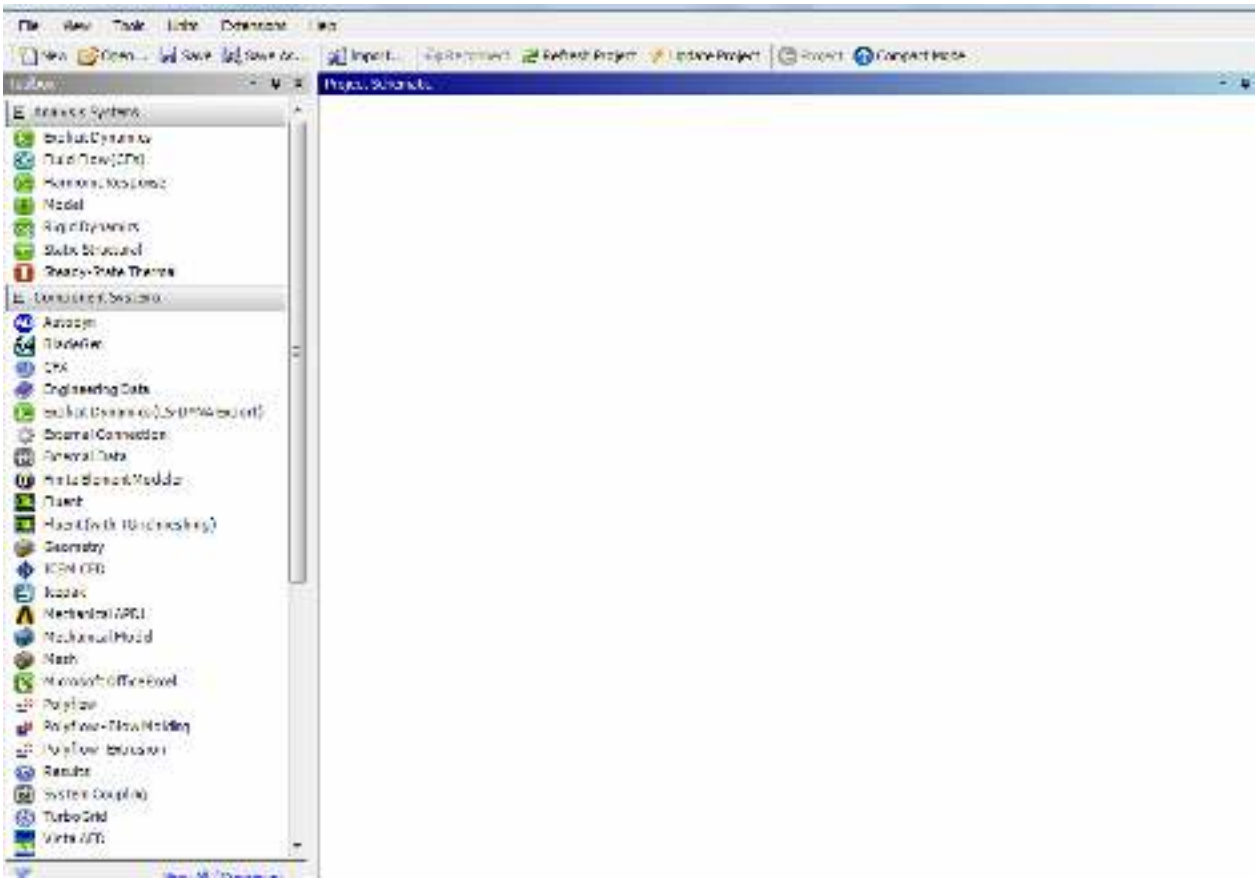
Thus the structure deforms as per the boundary condition verification.

Model division:

here changes are done for better results.

SYLLABUS:

1. Static structural
2. Steady state thermal
3. Modal analysis
4. Harmonic response
5. CFX (Fluid flow)
6. Explicit dynamic
7. Couple field analysis



HOW TO START ANSYS:

1. Start all program
2. Ansys (14.5) (workbench 14.5)
3. Right click on page
4. New analysis system

Engineering data:

Here we set the material for the analysis depends on the material properties results are obtained.

Double click on engineering data

Structural steel material properties;

1. Density
2. Isotropic elasticity:
3. Young's modulus
4. Poisson's ratio (0.3)
5. Tensile yield strength

To add a new material:

To add a new material to the library, go to engineering data (that option is located at the top right-side corner, symbol is like three books)

Types of data source:

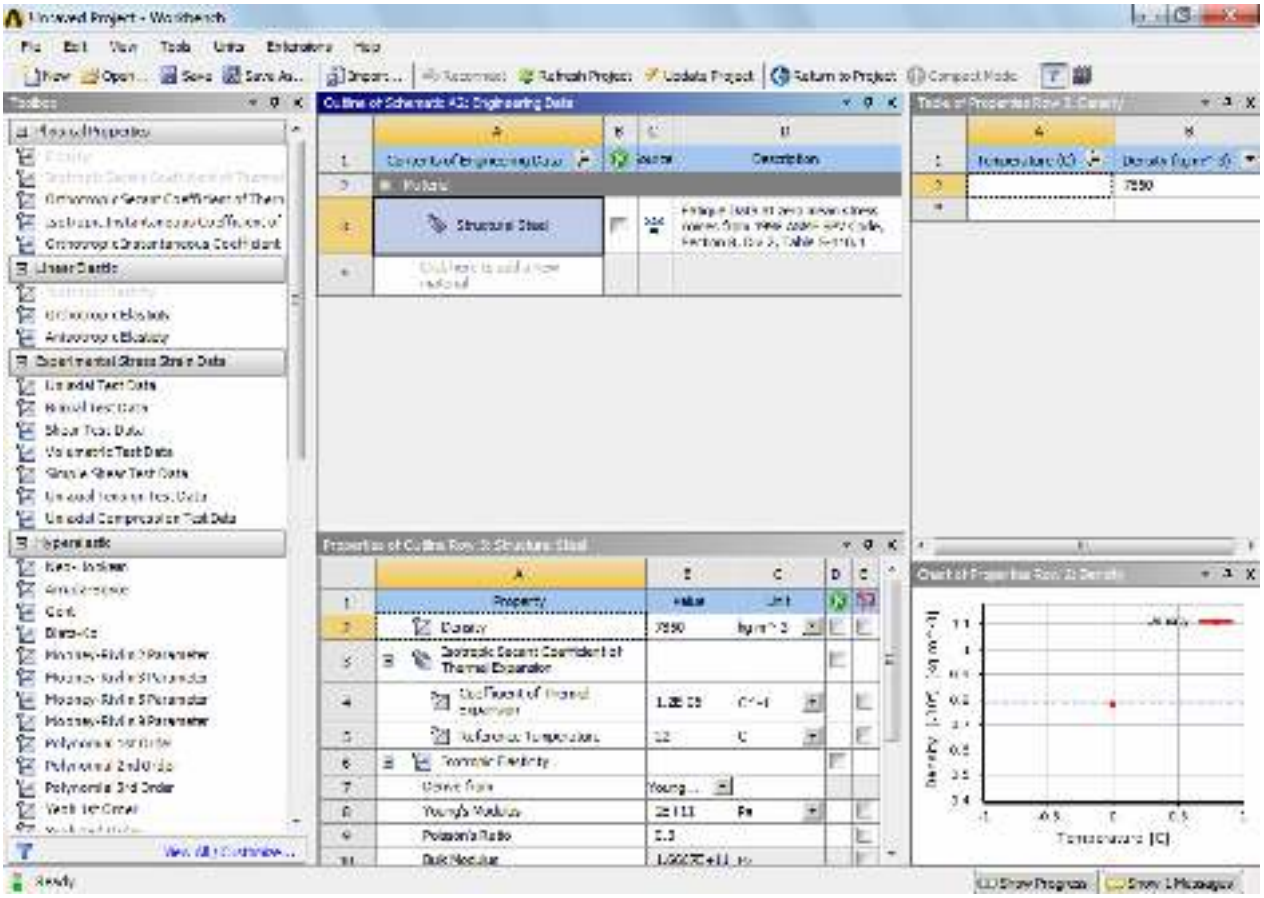
1. General materials
2. General nonlinear materials
3. Explicit materials
4. Now click on general materials
5. Aluminum alloy
6. Concrete alloy
7. Copper alloy
8. Gray cast-iron
9. Magnesium alloy

Select the appropriate material what we do require. Click on add (+) option.

After selecting the materials again go back to the engineering data source option (off that option).

How to create a new material: At the engineering data, type the new material name, at the side of it there will be material properties. Select required material property drag to it and specify the exact value.

After creating the materials click on return to project option.



Geometry: Here we can import the geometry from the 3D modeling software by converting the file into IGES or IGS format.

Right click on geometry

Import geometry

Browse

How to create geometry in geometry module?

Right click on geometry

New geometry

Set the units(mm)

Select any one plane, right click, look at.

Draw a rectangle go to extrude specify the extrude after clicking the extrude then click on the generate.

Face split:

This option is used to divide the faces by using the lines.

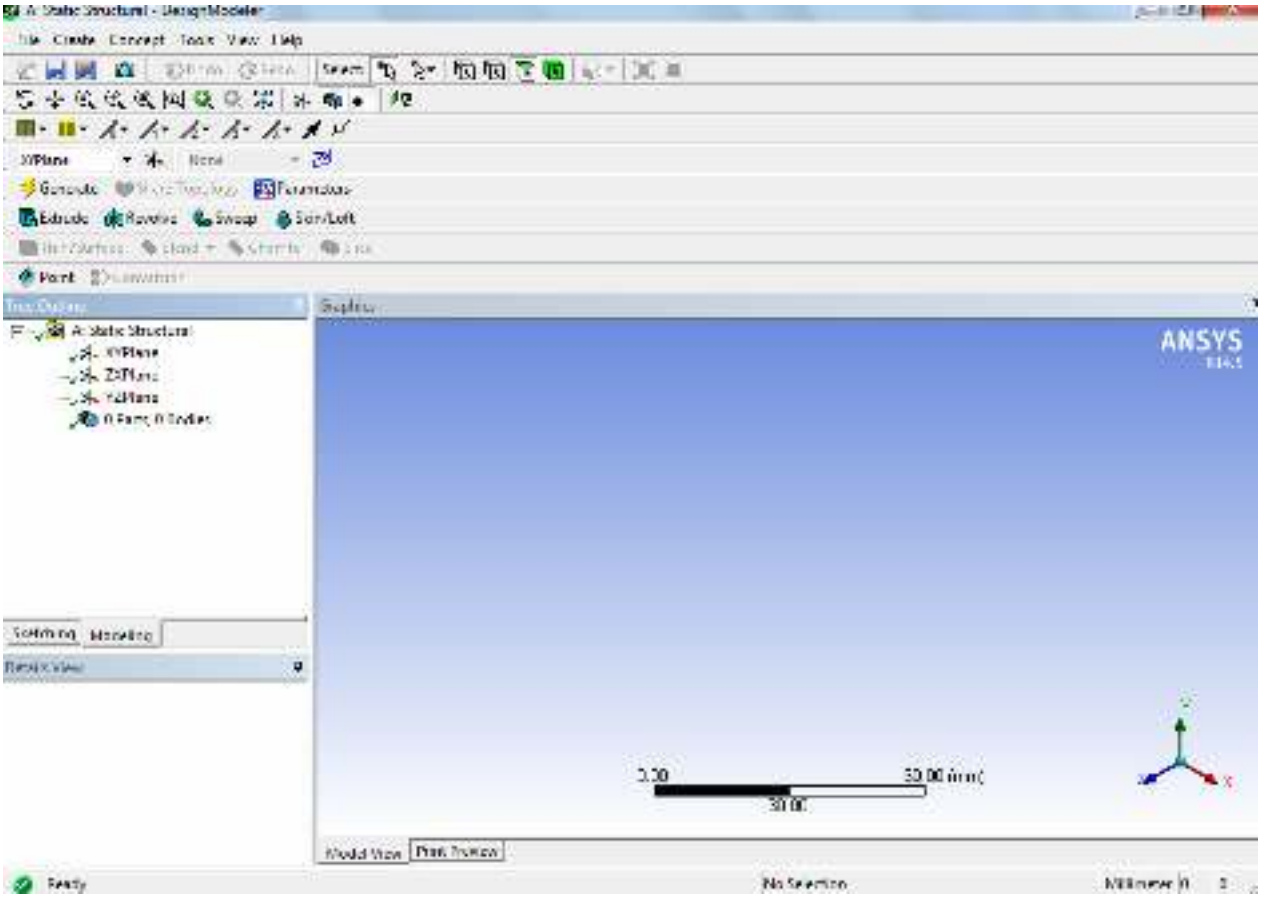
Select the appropriate face for the face split according to that select the plane look at.

Go to sketching, draw the lines give the dimensions.

Go to tools face split, select the face, apply (that is known as target geometry).

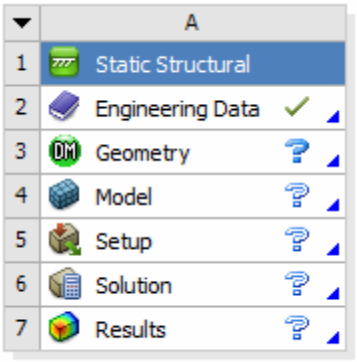
Now select the point and select another point by clicking control and click on apply (it is known as tool geometry).

After creating the face split click on generate



Static structural:
Set the engineering data, import the geometry, generate mesh.
In this analysis system it is used to determine the response of the structure subjected to static loading conditions which does not include inertia and damping effect.

This is used for calculation of displacement, deformation, stress, strain forms under different loading conditions.



Static Structural

Analysis process

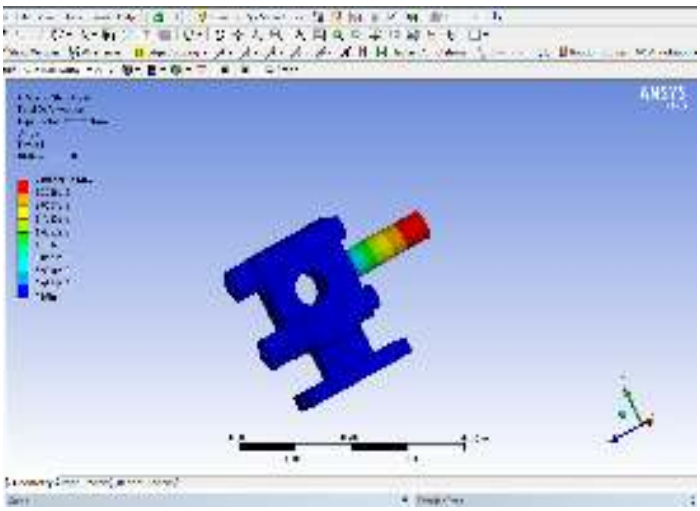
Set the engineering data import or draw the geometry

Generate the mesh for the geometry in the model.

Set the boundary conditions i.e apply force or pressure and fixed supports as you required

Solve all the solutions

Select the required solutions



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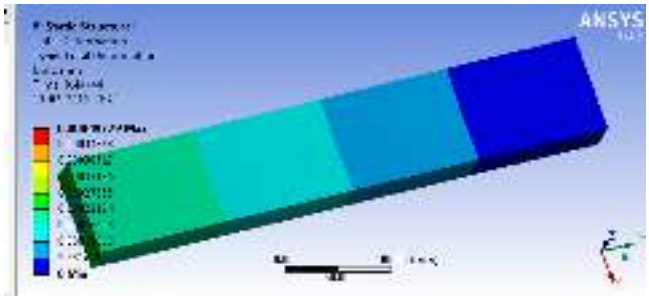
Ex; Analysis for a square rod:

- Create a square rod in Creo save the file as IGES.
- Right click on the page new analysis system, static structural.
- Set the engineering data.
- Import the geometry in geometry module, double click on model
- Go to mesh, right click generate mesh. If we require accurate mesh or user defined mesh Right click on mesh, insert, sizing.
- Specify the element size and select the body apply
- After generating the mesh, right click on static structural, insert and then select the required load
- At this bottom left side, you need to select “define by” as components. Here according to the Compass give t After applying the boundary conditions right click on static structural then solve.
- Then right click on solution, insert and select the require solutions as you required. Again, right click on solution evaluate all results.
- he values again right click on static structural select the fixed support. After applying the boundary conditions right click on static structural then solve.
- Then right click on solution, insert and select the require solutions as you required. Again, right click on solution evaluate all results



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Beams:

- Set the engineering data
- double click on geometry
- Select any one plane, right click, look at, draw a line and generate.
- Go to concept, toolbars, cross section rectangular, generate.
- Again, go to concept, lines from sketches.
- Select the two lines click on apply again click on generate.
- Go to part, line body, assign rectangle to it.
- To view the complete beam, go to the view option select the cross section solid option.
- Double click on model
- Generate mesh
- Right click on static structural
- Insert
- Fixed support
- Apply the force
- Right click Solve
- Select the required results

Beams (shear-moment diagram):

Process:

- Model
- Construction geometry
- Right click on construction geometry
- Insert
- Path
- Select the path type as edge
- Now select the two lines
- Right click on solutions
- Insert

Beam results

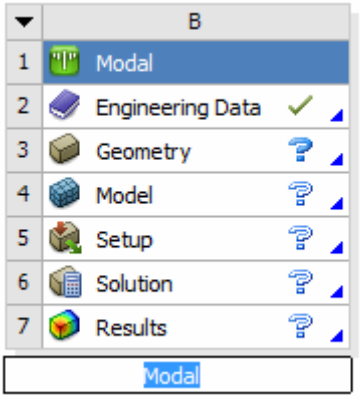
- Shear moment diagram
- Select the path as path
- Right click on solutions
- Evaluate all the results

How to set the graph?

- Go to analysis setting
- In step control model column set the step end time as you required.
- Edit the define by as sub steps.
- Enter the initial sub steps you required
- Minimum sub steps should be less than or equal to initial sub steps.
- Maximum sub steps should be equal to initial sub steps.
- After setting the analysis setting select any two results. Then click on new chart and table

Model analysis:

It is used to determine the vibrational characteristics natural frequencies and node shapes of a structure for a dynamic loading condition



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Set the engineering data, import the geometry in the geometry model, generate mesh

Go to analysis setting.

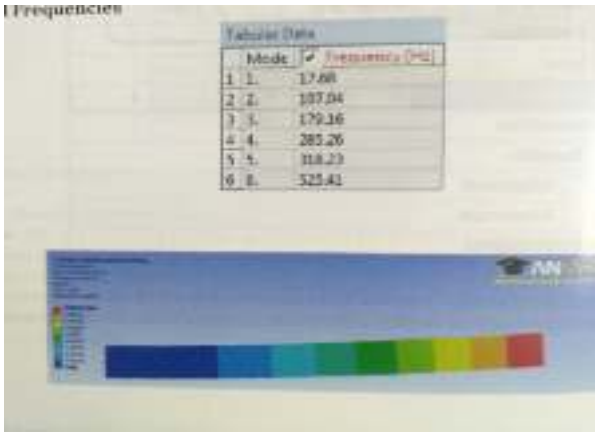
Give the number to maximum nodes.

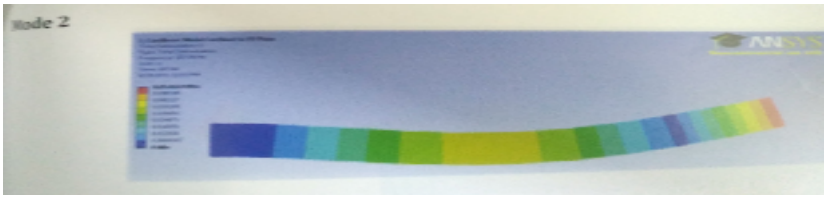
Apply the fixed support to one face.

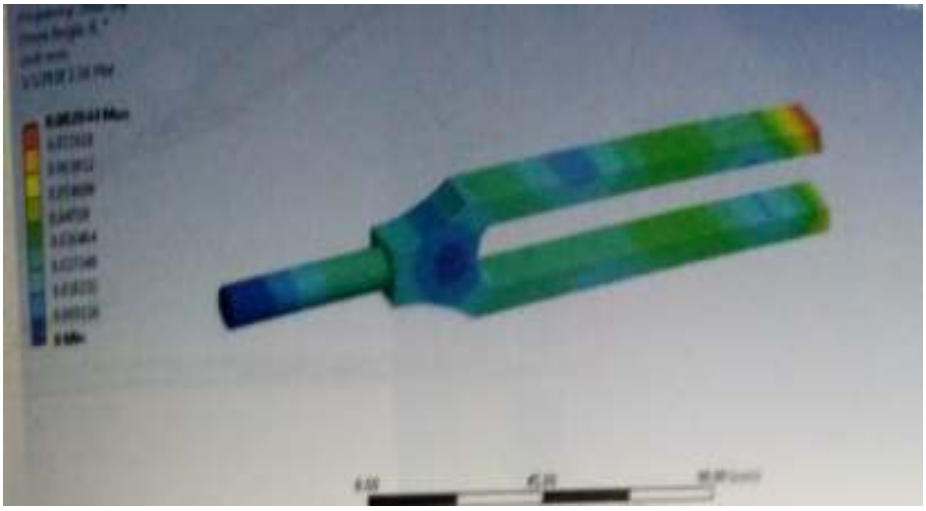
Right click on solutions

Total deformation

Evaluate all results



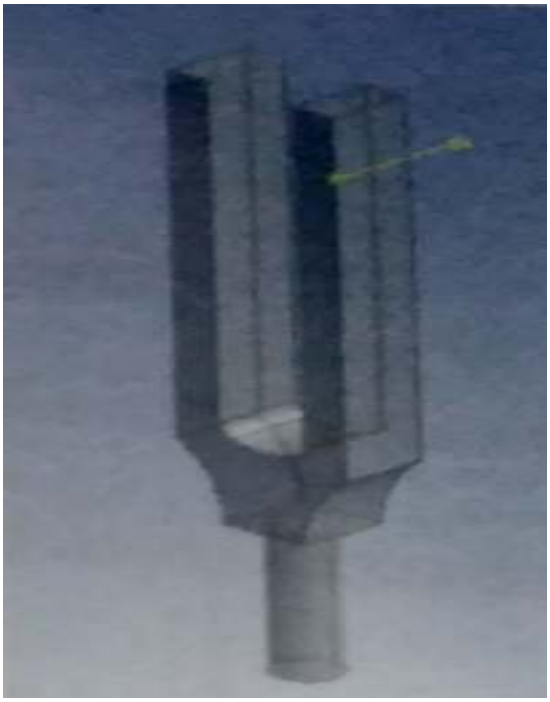


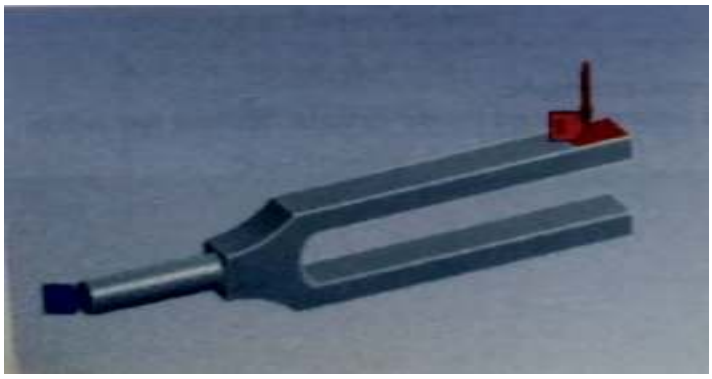


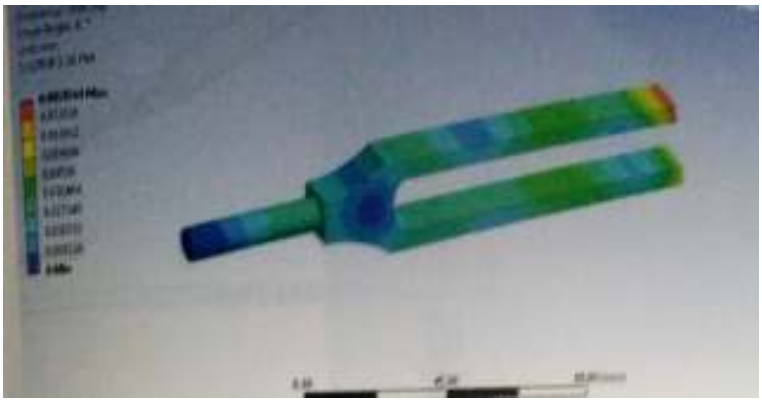
Harmonic:

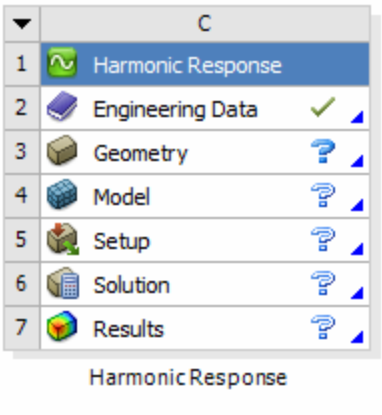
Set the engineering data, import the geometry in the geometry model, generate mesh.

Go to analysis setting give the frequency value apply the fixed support give the pressure at the certain area solve the solution results are obtained



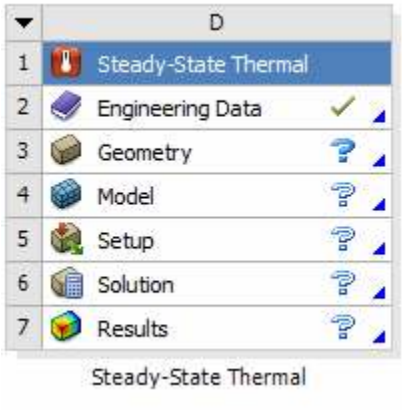




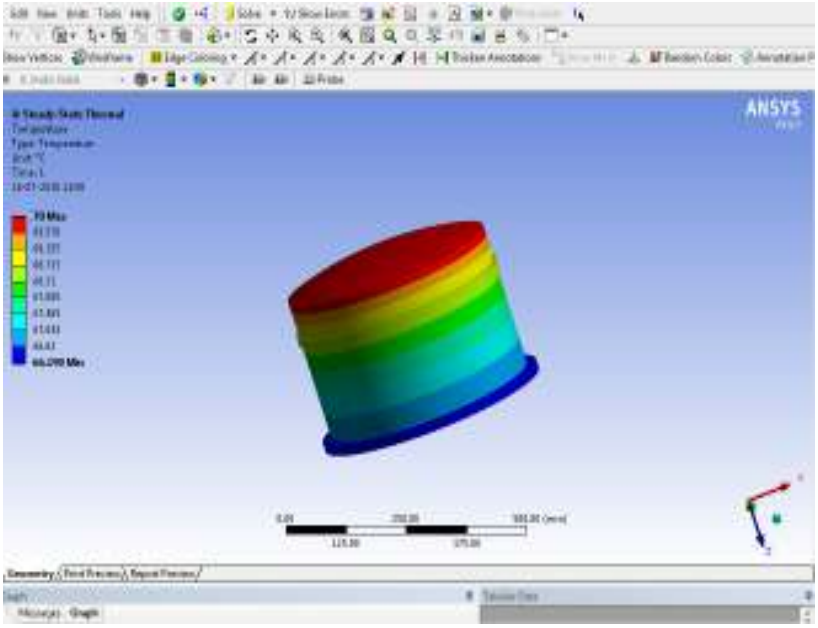


Steady state thermal:

It is used to determine the heat flow rate under the influence of thermal loading conditions which remains constant with time

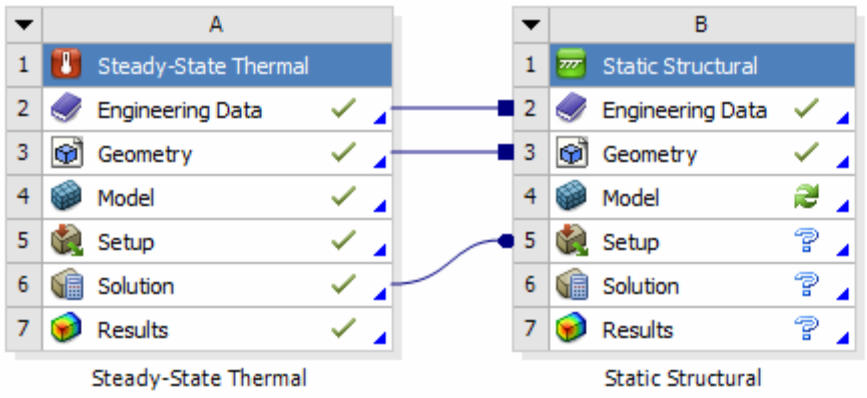


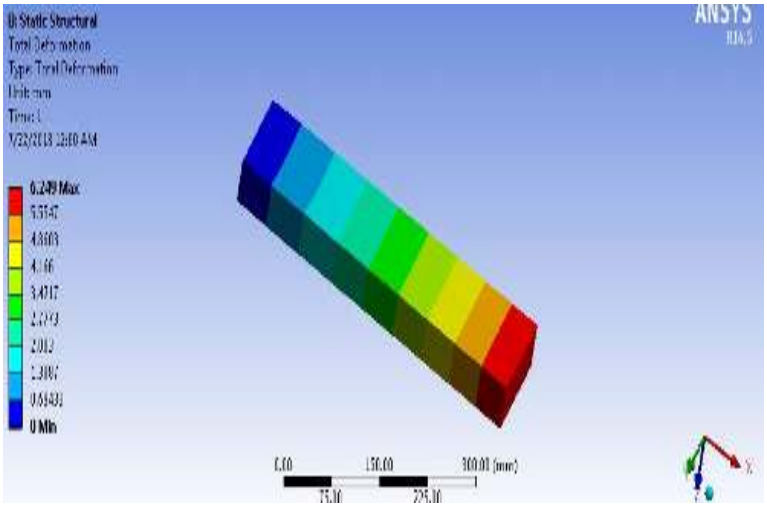
- After importing the profile into geometry generate the mesh.
- Right click on steady state thermal
- Insert, temperature
- Select the face where do we want to apply the temperature.
- Give the specific value for it, apply
- Again, right click on steady state thermal, insert, convection.
- Select the face where temperature should flow
- Then go to stagnated air simplified
- Click on ok, go to initial temperature
- Change the ambient value
- Right click on solution
- Select the temperature, total heat flux
- Solve



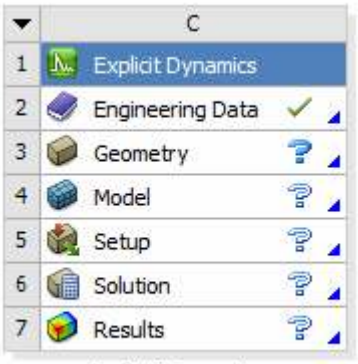
Couple field analysis

Do the analysis for steady state thermal and import the data into static structure



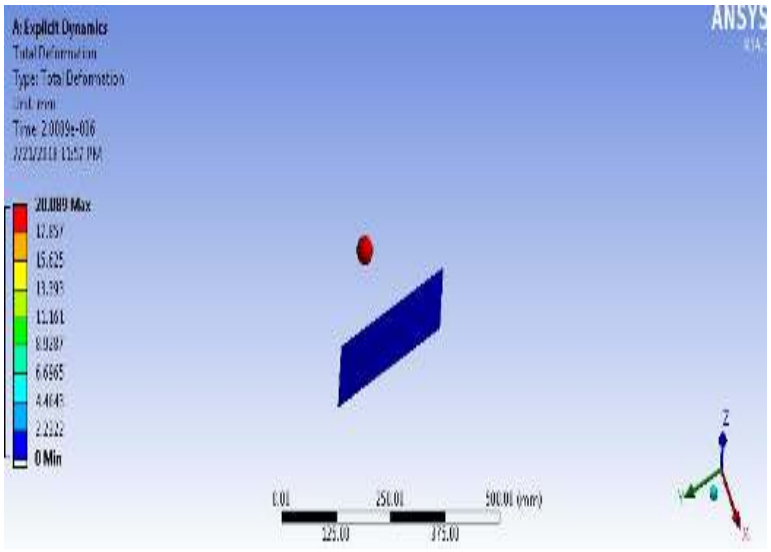


Explicit dynamic: this type of analysis is known as dynamic analysis. It is used to see the collision between two objects.



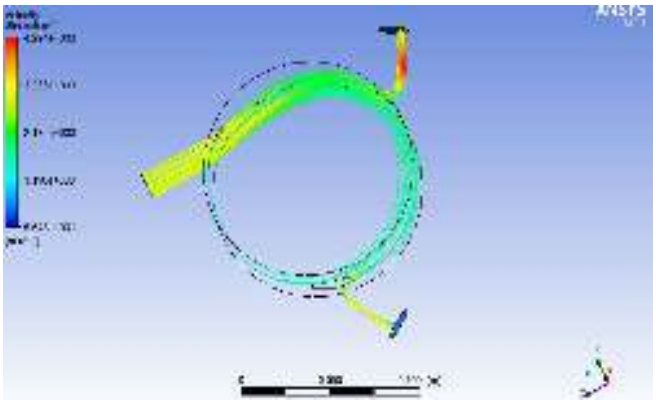
Explicit Dynamics

- Set the engineering data with different materials.
- Import the geometry.
- Open the model.
- Assign the different materials to different bodies and then generate mesh.
- Right click on initial conditions. Select velocity.
- Give the velocity speed.
- Right click on explicit dynamic, fixed support.
- Give the fixed support to the sheet and go to analysis setting.
- In analysis setting go to end time, output control.
- Give the equal values and right click on solutions, total deformation



Fluid flow (CFX):

- It is the flow of water through pipes.
- Draw the profile in pro/E. import geometry in ANSYS and generate the mesh.
- Select the face and right click, create the named selection.
- Double click on setup allow access and go to tools and pic set up mode and then select the problem type as single phase and fluid as water.
- Click on next and then click on default domain.
- Right click on boundary and add boundary, specify the speed.
- Click on start run after completion of results click on ok. Then click on results.
- Go to default domain select inlet and outlet select stream line option and click ookAgain, click on animate select the streamline and click on play.

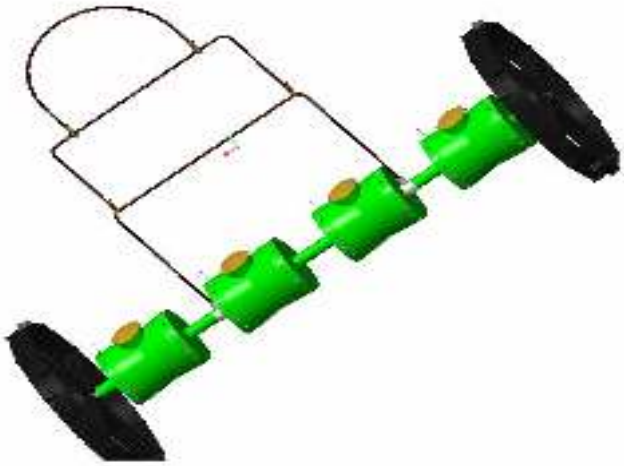


RICE SEED SOWING MACHINE:-

- **Introduction:-**

- This is a one type of rice seed sowing machine
- It is used to sowing the rice . the machine is offering in tamilnadu
- It gets helpful to reduce the manual effort and help farmer
- Rice seed sowing machine proves to be practical for uniformity seed sowing
- The shape of the seed drum :-hyperboloid diameter of the metering hole :-9mm
- Level of filling the seed drum :-half volume
- It ids very easy to care and work
- This is for sowing germinated
- It is manual operated
- The Nakhan sawan rice seed centre hopes to introduced a new rice sowing technology
- To small scale farmer in an attempt to reduce labour and cost our team evaluated
- The current rice planting practices and obtained information
- The small farmer ,the rice sowing machine inventor and rice seed centre representatives in order
- The recommendation regarding improvements

The current machine and suggest way to initiate the adoption and diffusion process



FEATURES :-

- Crop will matures within 7-10 days earlier than the transplanted paddy
- Uniformity in seed sowing and plant population will be seen
- Continuous drilling of seed is eliminated
- Light in weight machinery
- It provides consistency
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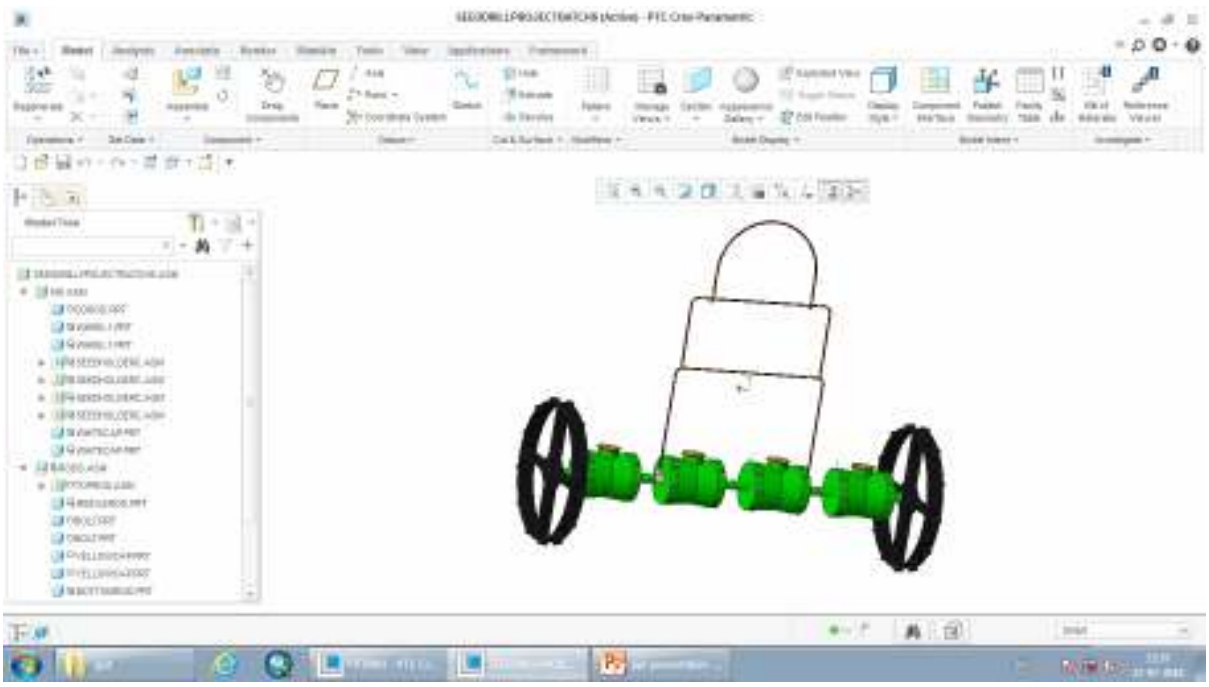
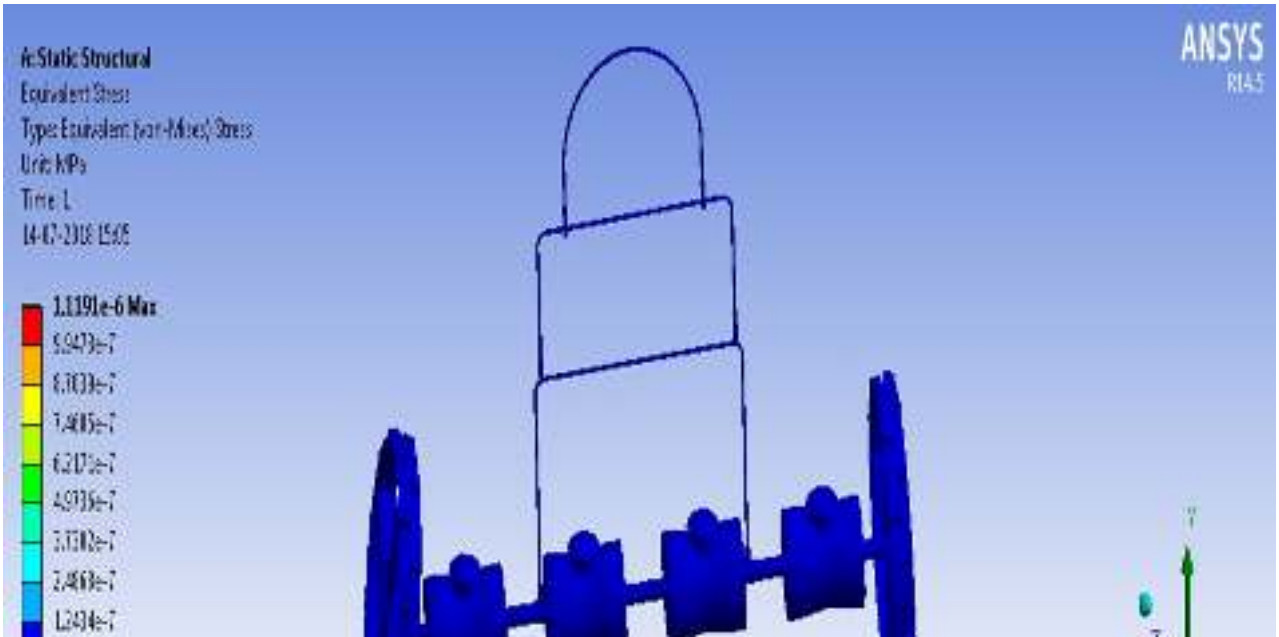
- Continuous drilling of seed is eliminated
- Light in weight machinery
- It provides consistency

ADVANTAGES OF SEED SOWING MACHINE:-

- Labour cost is less
- Seeds are exposed
- Seeds are presented to fowls ,rats and snails
- Seed pre-requisite is more /hectare
- Nursery arrangement are not required




A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
14-07-2018 15:05
1.1191e-6 Max
ANSYS
R14.5