Experiment No.1

Object:- To draw the sketch for the model shown in figure.

Mode of Object: Sketch

Expected time: 45 Min

Steps Used:

a. Set the working directory and create a new object file.

Open the navigator, click on the plus symbol. Now right click on the C01 folder to display a shortcut menu. Choose the Set Working directory option

b. Draw The sketch using the line tool

the sketch in figure consists of only lines draw all the lines with the help of 2 points lines button in the sketcher toolbar.

c. Applying constraints to sketch

constraints are applied to the sketch to maintain the design intent of feature and this might sometimes results in less dimensions in the sketch. Choose the equal lengths, equal radii, or same curvature constraints button for applying the constraints to the sketch.

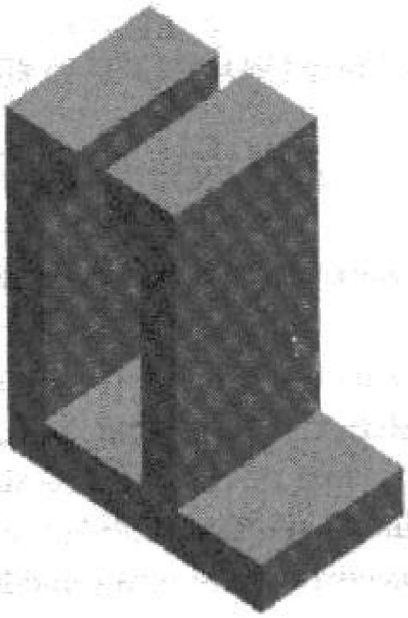
d. Dimensioning the sketch

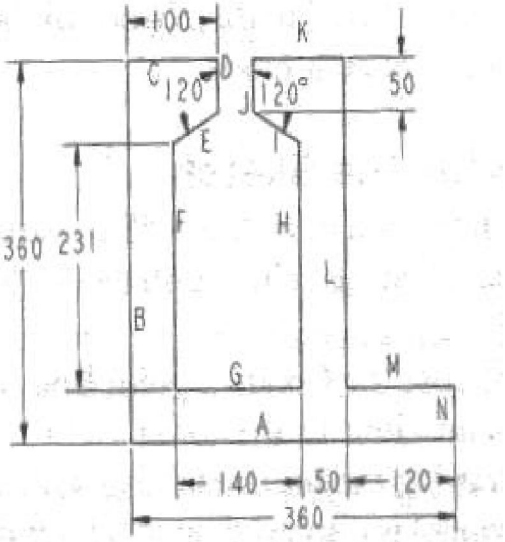
weak dimensions are already applied to the sketch while drawing. You need to dimension only the angle b/w the lines by using create defining dimension button.

e. modifying the dimensions.

Select the item button and then select all the dimensions by specifying a window around them. When the dimension turned red in color, choose the modify the values of dimensions, geometry of splines, or text entities button; the modify dimension dialog box is displayed.

f. save and sketch and close the file




100
K
C
120°
D
J
120°
50
E
360
231
F
H
L
B
G
M
N
A
140
50
120
360

Experiment No.2

Object:- To draw the sketch for the model shown in figure.

Mode of Object: Sketch

Expected time: 45 Min

Steps Used:

a. Set the working directory and create a new object file.

Open the navigator, click on the plus symbol. Now right click on the C02 folder to display a shortcut menu. Choose the Set Working directory option

b. Draw the sketch using the sketch tool

the sketch in figure consists of lines, arc, and circles. Draw all the lines with the help of 2 points lines button in the sketcher toolbar. Create an arc by 3 points or tangent to an entity at its endpoint button. Complete the arc at the end point of the right vertical line. Create circle by picking the center and a point on the circle button.

c. Filleting the corners

choose the create a circle fillet between two entities button from the sketcher tools toolbar. Select two entities one by one using the left mouse button. The corners of the selected lines are filtered.

d. Applying constraints to sketch

constraints are applied to the sketch to maintain the design intent of feature and this might sometimes results in less dimensions in the sketch. Choose the equal lengths, equal radii, or same curvature constraints button for applying the constraints to the sketch.

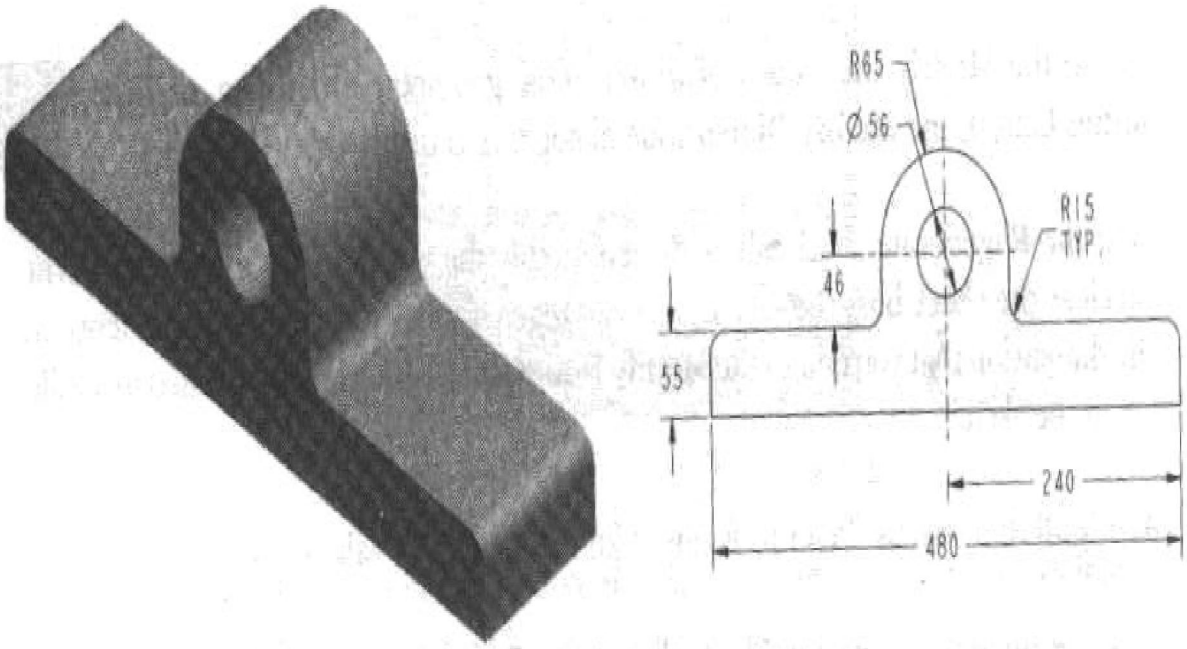
e. Dimensioning the sketch

weak dimensions are already applied to the sketch while drawing. You need to dimension only the angle b/w the lines by using create defining dimension button.

f. modifying the dimensions.

Select the item button and then select all the dimensions by specifying a window around them. When the dimension turned red in color, choose the modify the values of dimensions, geometry of splines, or text entities button; the modify dimension dialog box is displayed.

g. save and sketch and close the file



Experiment No.3

Object:- Select the A4 size drawing sheet and generate all drawing views shown in figure.

Mode of Object: drawing mode

Expected time: 45 Min

Steps Used:

a. Starting a new drawing file.

Select the drawing radio button and then enter the name of file then click on OK button. Chose brows button to select the file for generating views. select empty button and choose landscape area. Select A4 from standard size drop down list chose OK.

b. Generating the top view

choose the create a general view from top tool chest. Select the TOP option and choose the apply button.

c. Generating the front view

Choose insert. Drawing View . Projection from the menu bar. Specify the center point for front view below the top view.

d. Generating the section view

Choose insert. Drawing view. projection from the menu bar. Select the top view as the projection parent view specify the center point. Select the sections from the categories list box in drawing views dialog box. Select the 2D cross section radio button and choose the (+) button choose offset. Both Sides. Single. Done from the XSEC CREAT menu. Specify the name of cross section as X in the message Input Window. Choose datum planes ON/OF button from datum display toolbar. Select the Top datum plane from the sub window. Choose OK THE sketch view submenu is displayed choose the bottom option from the SKET VIEW submenu and select the front datum plane exit the reference dialog box. Sketch the line for defining the section plane and align the lines and modify the angular dimension to 120.

e. Modifying the hatching

select X section Filter. Select the hatching from the drawing sheet. Chose the properties from the shortcut menu. Choose the spacing option and modify the hatching.

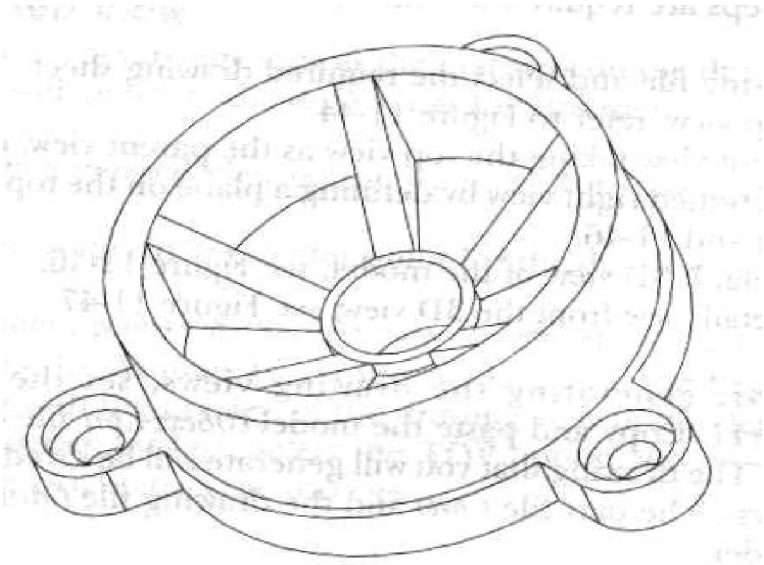
f. Generating the general view.

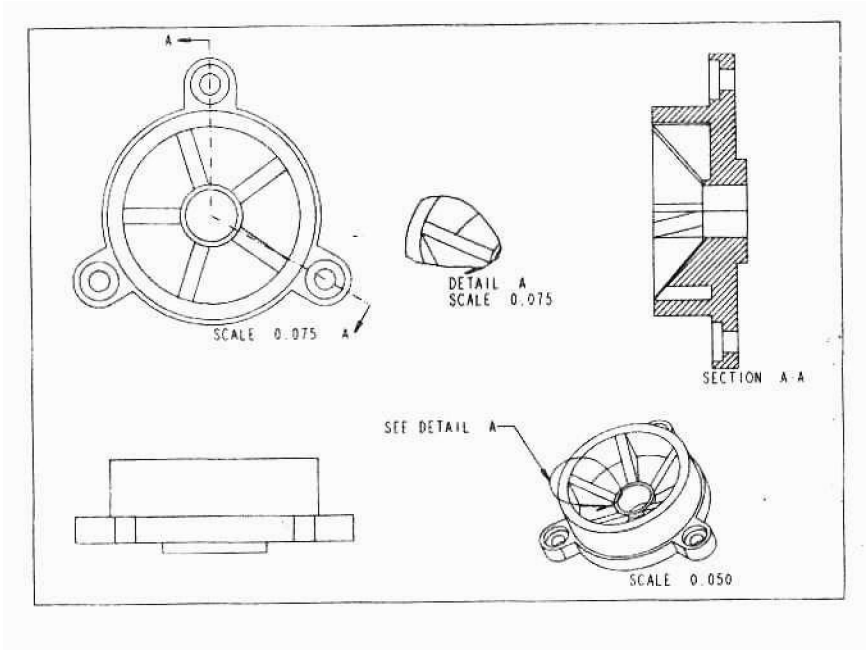
Choose the create a general view button from the top toolchest. Specify the center point for the placement of the general view below the section view.

g. Generating the Detail View.

Select the center point for detail on one ribs in the trimetric view. Draw the spline. Select the center point for the placement of the drawing view. The detailed view is generated.

h. save and sketch and close the file.




A
SCALE 0.075
A
DETAIL A
SCALE 0.075
SECTION A-A
SEE DETAIL A
SCALE 0.050

Experiment No.4

Object:- Select the A4 size drawing sheet and generate all drawing views shown in figure.

Mode of Object: drawing mode

Expected time: 45 Min

Steps Used:

a. Starting a new drawing file.

Select the drawing radio button and then enter the name of file then click on OK button. Chose brows button to select the file for generating views. select empty button and choose landscape area. Select A4 from standard size drop down list chose OK.

b. Generating the top view

choose the create a general view from top tool chest. Select the TOP option and choose the apply button.

c. Generating the front view

Choose insert. Drawing View . Projection from the menu bar. Specify the center point for front view below the top view.

d. Generating the section view

Choose insert. Drawing view. projection from the menu bar. Select the top view as the projection parent view specify the center point. Select the sections from the categories list box in drawing views dialog box. Select the 2D cross section radio button and choose the (+) button choose offset. Both Sides. Single. Done from the XSEC CREAT menu. Specify the name of cross section as X in the message Input Window. Choose datum planes ON/OF button from datum display toolbar. Select the Top datum plane from the sub window. Choose OK THE sketch view submenu is displayed choose the bottom option from the SKET VIEW submenu and select the front datum plane exit the reference dialog box. Sketch the line for defining the section plane and align the lines and modify the angular dimension to 120.

e. Modifying the hatching

select X section Filter. Select the hatching from the drawing sheet. Chose the properties from the shortcut menu. Choose the spacing option and modify the hatching.

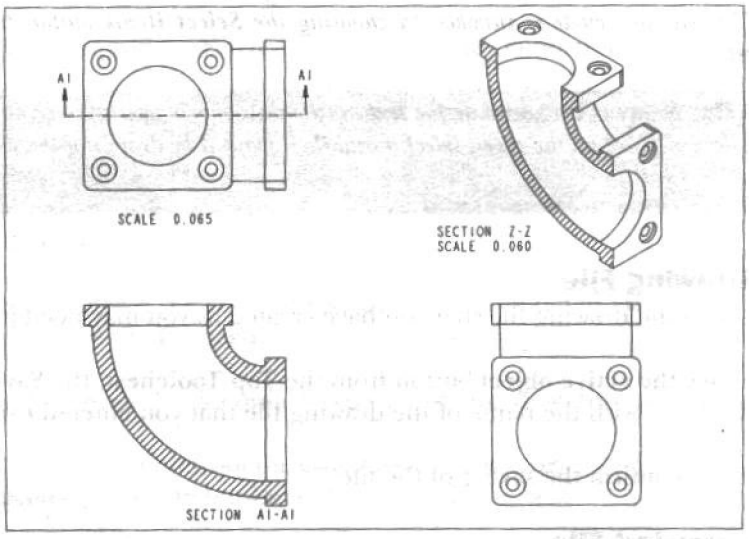
f. Generating the general view.

Choose the create a general view button from the top tool chest. Specify the center point for the placement of the general view below the section view.

g. Generating the Detail View.

Select the center point for detail on one ribs in the trimetric view. Draw the spline. Select the center point for the placement of the drawing view. The detailed view is generated.

h. save and sketch and close the file.


A1
A1
SCALE 0.065
SECTION Z-Z
SCALE 0.060
SECTION A1-A1

Experiment No.5

Object: - Create a model as per the shown in the diagram.

Mode of Object: Part mode

Expected time: 40 Min

Steps Used:

a. Set the working directory and create a new object file.

Open the navigator, click on the plus symbol. Now right click on the C02 folder to display a shortcut menu. Choose the Set Working directory option

b. Starting a new object file

Start a new object file in the part mode. Name the file.

c. Selecting the protrusion option

the revolve button will be used to crate the model. Choose insert. Revolve from the menu bar. The revolve dashboard is displayed below the drawing area.

d. Selecting the sketching plane

Here you have to select the Front datum plane as sketching plane. Choose the placement tab and then define. Then select the front datum plane as sketching plane. The Right datum plane will be perpendicular to the sketching plane will be parallel to the screen.

e. Drawing the sketch

The sketch in figure consists of only lines draw all the lines with the help of 2 points lines button in the sketcher toolbar. also draw the center line by choosing the create 2 point centerline button.

f. Dimensioning the sketch.

weak dimensions are already applied to the sketch while drawing. You need to dimension only the angle b/w the lines by using create defining dimension button.

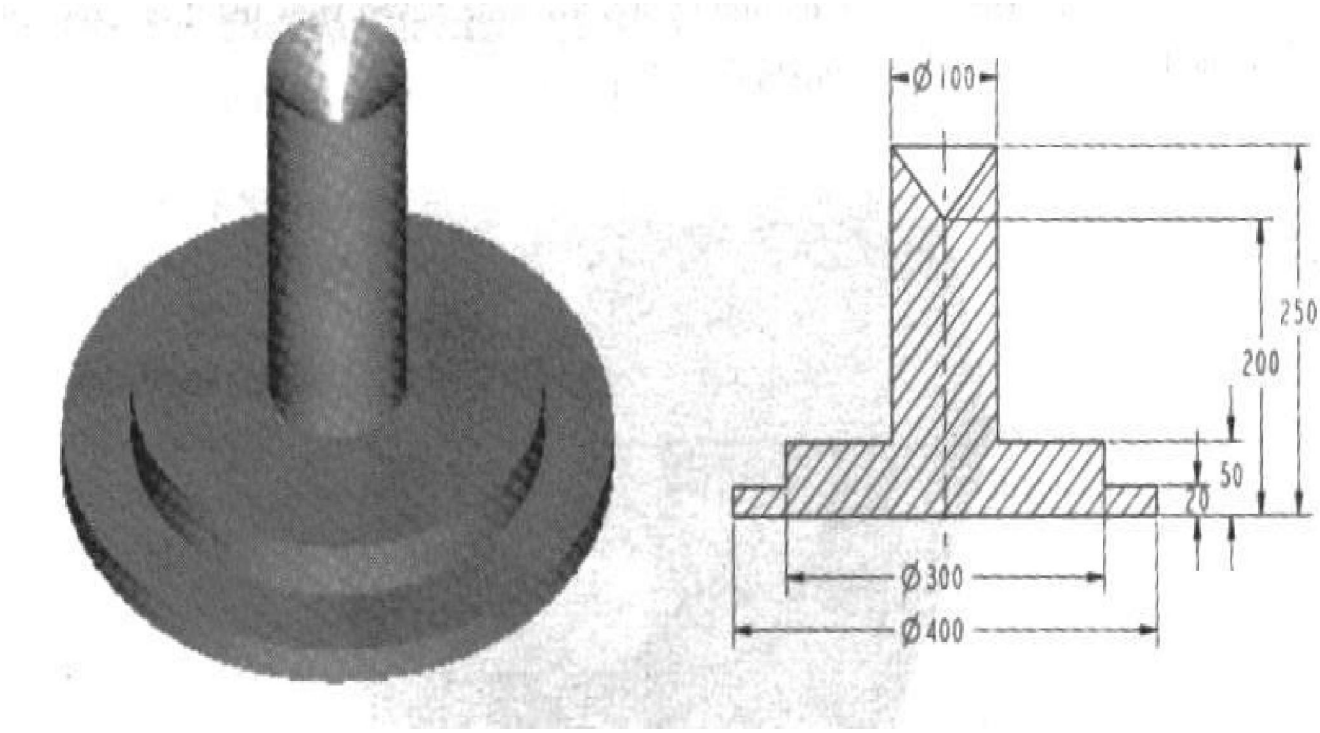
g. modifying the dimensions.

Select the item button and then select all the dimensions by specifying a window around them. When the dimension turned red in color, choose the modify the values of dimensions, , or text entities button; the modify dimension dialog box is displayed.

h. Specifying the model attribute.

Choose the saved view list button from the view toolbar. Choose the default orientation option from the dropdown list. All attributes that are needed to create the solid model are selected by default in the revolve dashboard. choose the build feature button in the extrude dashboard. The model appears.

h. save the model.



Experiment No.6

Object: - Create a model as per the shown in the diagram. The front and the right side view of the solid model are shown in fig.

Mode of Object: Part mode

Expected time: 40 Min

Steps Used:

a. Set the working directory and create a new object file.

Open the navigator, click on the plus symbol. Now right click on the C02 folder to display a shortcut menu. Choose the Set Working directory option

b. Starting a new object file

Start a new object file in the part mode. Name the file.

c. Selecting the sketching plane for the base feature.

Here you have to select the Front datum plane as sketching plane. Choose the placement tab and then define. Then select the front datum plane as sketching plane. The Right datum plane will be perpendicular to the sketching plane will be parallel to the screen. select the top datum plane from the drawing area and then select top option from the orientation drop down list.

d. creating and dimensioning the sketch

Draw the sketch using various sketcher tools and add the required constraints and dimensions. When you initially draw the sketch. It is dimensioned automatically and some weak dimensions are assigned to it. modify the dimensions values. choose the build feature button from the extrude dashboard. The base feature is completed.

e. selecting the sketching plane for the second feature.

After choosing define button in placement tab of extrude feature, select use previous button from the sketch plane area in the sketch dialog box.

f. drawing the sketch for second feature.

The second feature has a rectangular section that will be extruded to a depth of 14. To improve the clarity of the edges of the base feature, choose the no hidden button from the model display toolbars before you start sketching the second feature. draw the figure as shown in diagram.

g. creating the datum plane for the third feature.

A new datum plane, at an offset of 2 from front face of the second feature, is required to create the next feature. Choose the datum plane tool button from datum toolbar. The datum plane dialog box is displayed. In the datum plane dialog box, under the reference collector the offset constraints is displayed. Now you need to specify the offset distance. In the translation dimension box enter a value of -2 and press enter. Choose OK button.

h. Creating third feature on DTM1

Choose extrude tool, then placement tab and define button. Select DTM1 as sketching plane for the third feature. Sketch the section for the third feature of the model and add constraints and dimensions to the sketch. As shown in figure. choose continue with current section button. Change depth direction of extrude to other side of sketch button. Enter the value of 10 in the dimension box present on the extrude dashboard.

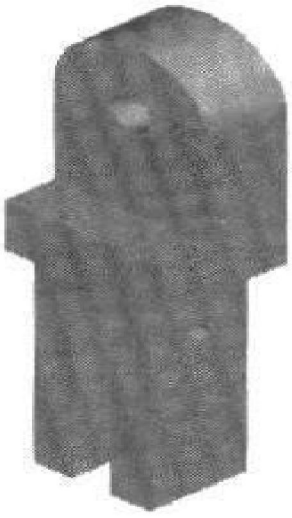
i. selecting the sketching plane for cut feature.

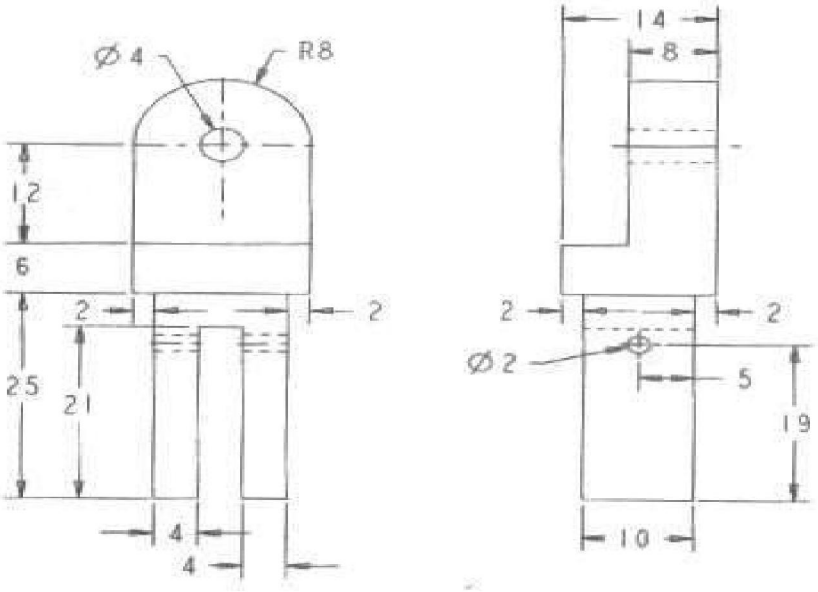
Choose extrude> remove material. Choose placement tab and define button. The sketch dialog box is displayed. Select the face for sketching.

J Sketching the cut feature

Choose option tab in the extrude dashboard. From the side1 dropdown list, choose through all option the cut feature is completed and can now be previewed in the drawing area. choose the build feature.

k. save the model.




Ø4
R8
12
6
2
2
25
21
4
4
14
8
2
2
Ø2
5
19
10

Experiment No.7

Object: Create a model as per the shown in the diagram. The front , top and the right side view of the solid model are shown in fig.

Mode of Object: Part mode

Expected time: 40 Min

Steps Used:

a. Set the working directory and create a new object file.

Open the navigator, click on the plus symbol. Now right click on the C02 folder to display a shortcut menu. Choose the Set Working directory option

b. Starting a new object file

Start a new object file in the part mode. Name the file.

c. Selecting the sketching plane for the base feature.

Here you have to select the Front datum plane as sketching plane. Choose the placement tab and then define. Then select the front datum plane as sketching plane. The Right datum plane will be perpendicular to the sketching plane will be parallel to the screen. select the top datum plane from the drawing area and then select top option from the orientation drop down list.

d. creating and dimensioning the sketch

Draw the sketch using various sketcher tools and add the required constraints and dimensions. When you initially draw the sketch. It is dimensioned automatically and some weak dimensions are assigned to it. Modify the dimensions values. Choose the build feature button from the extrude dashboard. The base feature is completed.

e. selecting the sketching plane for the second feature.

After choosing define button in placement tab of extrude feature, select use previous button from the sketch plane area in the sketch dialog box.

f. drawing the sketch for second feature.

The second feature has a rectangular section that will be extruded to a depth of 14. To improve the clarity of the edges of the base feature, choose the no hidden button from the model

display toolbars before you start sketching the second feature. draw the figure as shown in diagram.

g. creating the datum plane for the third feature.

A new datum plane, at an offset of 2 from front face of the second feature, is required to create the next feature. Choose the datum plane tool button from datum toolbar. The datum plane dialog box is displayed. In the datum plane dialog box, under the reference collector the offset constraints is displayed. Now you need to specify the offset distance. In the translation dimension box enter a value of -2 and press enter. Choose OK button.

h. Creating third feature on DTM1

Choose extrude tool, then placement tab and define button. Select DTM1 as sketching plane for the third feature. Sketch the section for the third feature of the model and add constraints and dimensions to the sketch. As shown in figure. choose continue with current section button. Change depth direction of extrude to other side of sketch button. Enter the value of 10 in the dimension box present on the extrude dashboard.

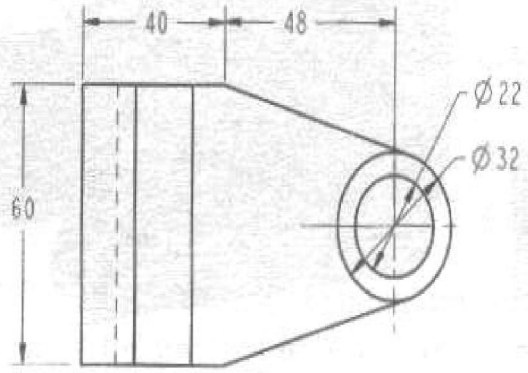
i. selecting the sketching plane for cut feature.

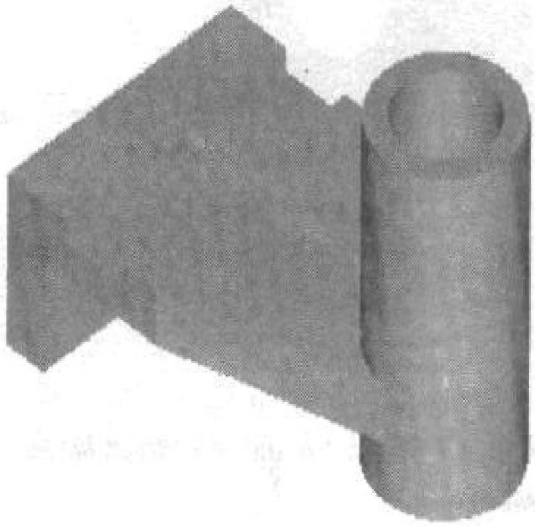
Choose extrude> remove material. Choose placement tab and define button. The sketch dialog box is displayed. Select the face for sketching.

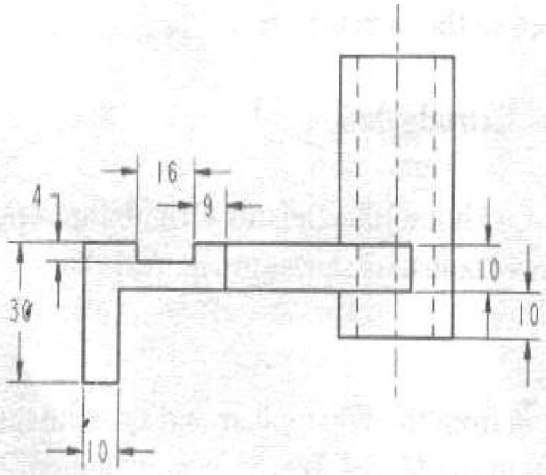
J Sketching the cut feature

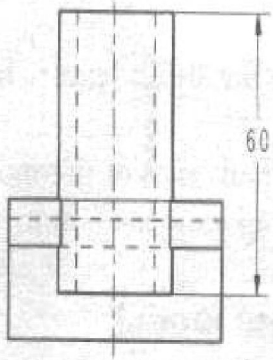
Choose option tab in the extrude dashboard. From the side1 dropdown list, choose through all option the cut feature is completed and can now be previewed in the drawing area. choose the build feature.

k. save the model.


40
48
Ø22
Ø32
60




16
9
4
10
10
30
10


60

Experiment No.8

Creating Patterns.

Object: - Create a model as per the shown in the diagram. The front and the top view of the solid model are shown in fig.

Mode of Object: Part mode

Expected time: 40 Min

Steps Used:

a. Set the working directory and create a new object file.

Open the navigator, click on the plus symbol. Now right click on the C02 folder to display a shortcut menu. Choose the Set Working directory option

b. Starting a new object file

Start a new object file in the part mode. Name the file.

c. creating the base feature.

To create the sketch for the base feature, you first need to select the sketching plane for the base feature. In this model you need to draw the base feature on the top datum plane. This is because the direction of extrusion of this feature is perpendicular to the top datum plane.

d. creating and dimensioning the sketch

Draw the sketch using various sketcher tools and add the required constraints and dimensions. When you initially draw the sketch. It is dimensioned automatically and some weak dimensions are assigned to it. Modify the dimensions values. choose the build feature button from the extrude dashboard. The base feature is completed.

e. Creating the second feature.

The second feature is also an extruded feature and will be created on the top face of the base feature. Therefore you need to define the top face as the sketching plane for the second feature.

f. drawing the sketch for second feature.

The second feature has a circular section that will be extruded to a depth of 58.. draw the Sketch as shown in diagram. enter a value of 58 in the dimension box in the extrude dashboard and choose build feature button.

g. creating the third feature.

The third feature is a through hole that is coaxial to the cylindrical feature. The hole feature will be created using hole dashboard.

h. Creating fourth feature.

The fourth feature is a through hole and will be created on the top planer surface of the base feature. The hole feature will be created using hole dashboard.

i. Patterning the hole feature.

Select the hole feature and then choose the pattern tool button from the edit features toolbar. select the identical option . select the dimension value 25 from the drawing area. After you select the dimensions in the first direction the edit box is displayed. Enter the value of 100 in the edit box. Hold down the right mouse button to display the shortcut menu. Choose the direction 2 dimensions option from the shortcut menu. select the dimension value 25 and enter the value 250 in the corresponding edit box. Choose the build feature the rectangular pattern of the hole is displayed.

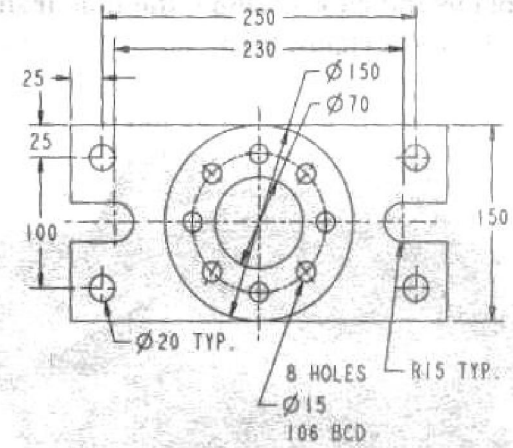
J Crating a hole on the cylindrical feature.

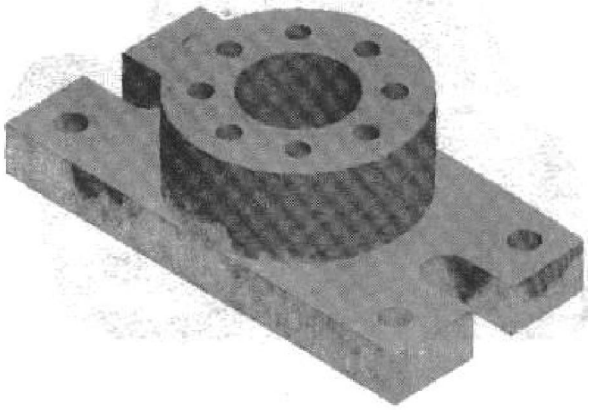
The hole on the cylindrical feature will be created diametrically using the hole dashboard.

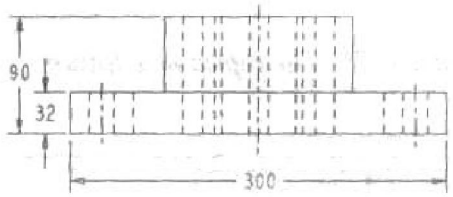
k. Creating the rotational pattern of the hole feature.

Select the hole feature and then choose pattern tool from the edit feature. select identical option select the angular dimension 90 the edit box is displayed. Enter a value of 45 in the edit box and press enter. Enter 8 in the 1 box and press enter choose build feature button the rotational patteren is created.

l. save the model.


250
230
25
Ø150
Ø70
25
100
150
Ø20 TYP.
8 HOLES
Ø15
106 BCD
R15 TYP.




90
32
300

Experiment No.9

Creating Advance Blends, Sweeps, Swept Blend, ETC.

Object: - Create a model as per the shown in the diagram. The front and the top view of the solid model are shown in fig.

Mode of Object: Part mode (advance)

Expected time: 40 Min

Steps Used:

a. Set the working directory and create a new object file.

Open the navigator, click on the plus symbol. Now right click on the C02 folder to display a shortcut menu. Choose the Set Working directory option

b. Starting a new object file

Start a new object file in the part mode. Name the file.

c. Invoking blend option.

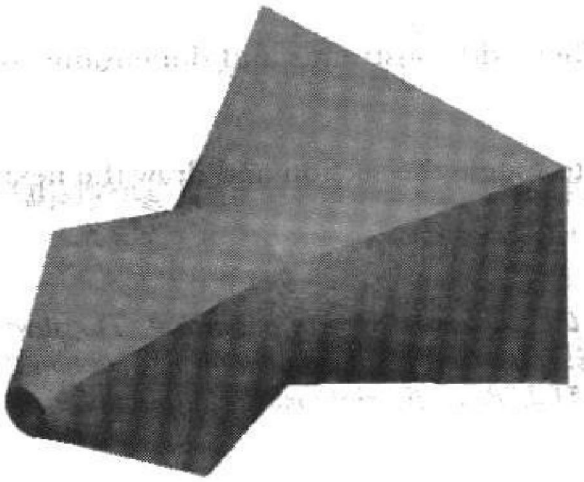
Choose Insert>Blend >Protrusion from the menu bar. From the blend option submenu choose parallel > Regular sec> sketch section. Attribute menu will ask to choose straight or smooth option. Choose straight > done from attribute menu

d. Selecting the sketch plane

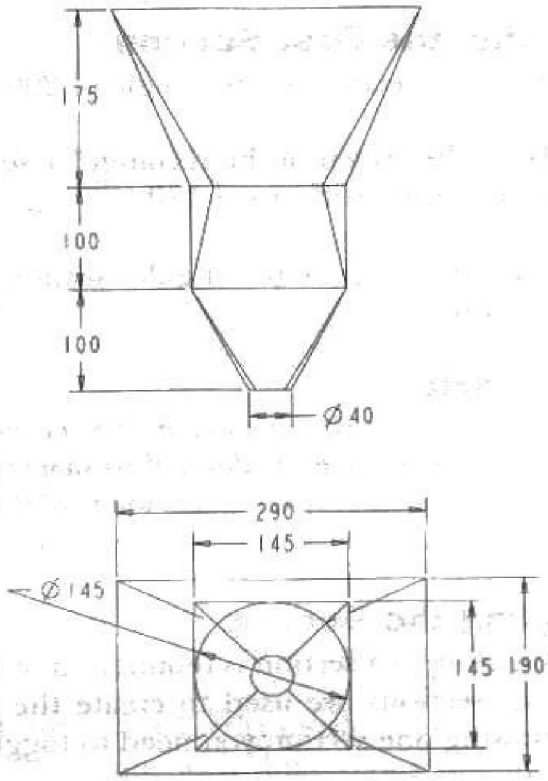
Select front datum plane as sketching plane. From sketch menu select top.

e. Drawing first section and toggle the section- draw the rectangular section and then add constraints and dimensions to it. Go to sketch menu and choose feature tools> toggle section.

f. Drawing circular section and dividing the circular section: draw the circular section and modify the diameter of the circle to 145. The circular section should be divided at four points because rectangle and square have four points, the no. of entities. Choose the black arrow on the right of dynamically trim section entity button and then choose the entity divide the entity at the point of selection button. Select the circle at four points.



Isometric view of the model



g. Applying constraints to the four points: Choose constraints from sketcher menu bar. Choose make line or two vertices vertical constraint button from constraint dialog box and select two division points on left side of the circle to lie in a vertical line. Similarly select two points on the right to apply the constraints. Choose make line or two vertices horizontal constraint button and select the two division points on the upper half and lower half to lie in horizontal line. Now toggle the section and create new section. The next section to be drawn is square section. After drawing it draw the circular section. Draw the circular section into four entities similar to section 2 and then constraint it and dimension it.

h. Giving depth to the section: Choose the continue with current section button. The message input window appears and prompt for depth of section 2. Enter the value 175 in the window and enter.

i. Save the model.