Chapter 6: DRAFTING FUNDAMENTALS

Tutorial 6.1: Generative Drafting

Featured Topics & Commands

Prerequisite Knowledge & Commands

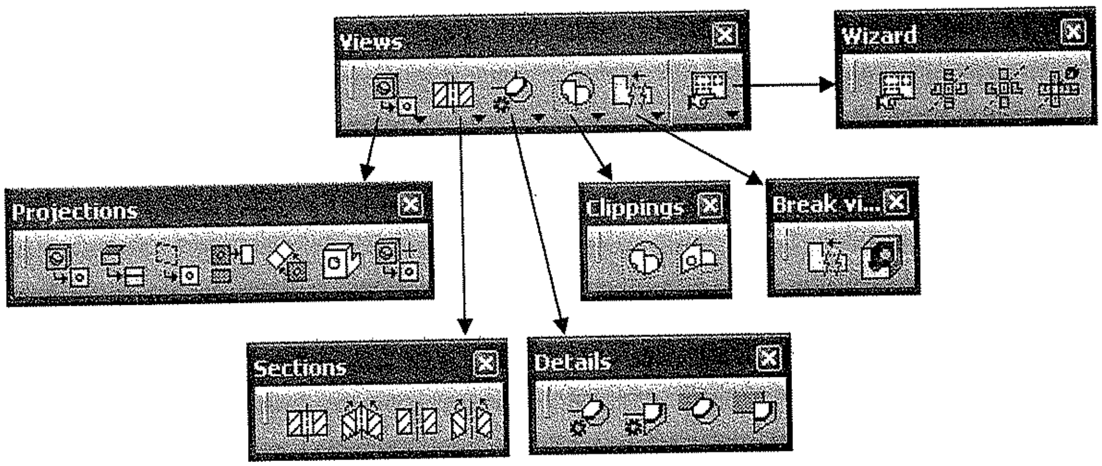
- Entering workbenches
- The *Sketcher* workbench and associated commands
- The *Part Design* workbench and associated commands

The Drafting Workbench

The *Drafting Workbench* allows you to create an orthographic projection or drawing (CATDrawing) directly from a 3D part (CATPart) or assembly (CATProduct). A CATDrawing contains a structure listing similar to a specification tree. The structure listing shows all the sheets and views contained in the document. CATIA enables you to create *generative views* that are associative with the 3D part, and to create *drawn views* which are not associative.

The Views toolbar

The commands located in the *Views* toolbar enable you to create a variety of views and view configurations. The sub-toolbars located within the *Views* toolbar, reading left to right, are

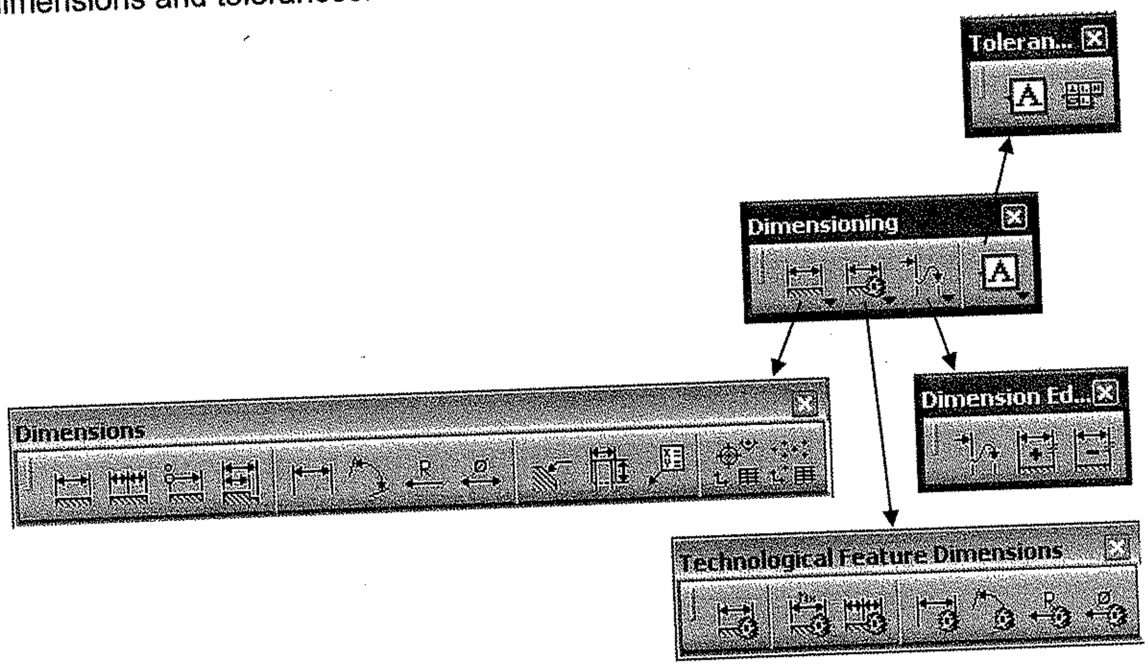


- Projections toolbar: The *Projections* toolbar contains commands that allow you to create different types of views. The commands, reading from left to right, are; *Front View, Unfolded View, View from 3D, Projection View, Auxiliary View, Isometric View, Advanced Front View.*
- Sections toolbar: This toolbar contains commands that allow you to create a variety of section and cut views.
- Details toolbar: The *Details* toolbar contains commands that allow you to create views that are a small portion of an existing view. The detail view is usually drawn at an increased scale.
- Clippings toolbar: The command located in the *Clippings* toolbar allow you to create removed views or removed section views. This is a small area of the part that is shown apart from the original view and is usually shown at an increased scale.

- Break view toolbar: The commands located in the *Break view* toolbar allow you to break the part in a specified location. This is usually done to save drawing space. The missing section of the part is usually uninteresting and not worth showing. There is also a command that enables you to create a broken out section.
- Wizard toolbar: This toolbar contains commands that allow you to select from several predefined view configurations or to define your own custom configuration. Reading from left to right, the commands are; '*View Creation Wizard*', which allows you to create custom view configurations, '*Front, Top and Left*', '*Front, Bottom and Right*', and '*All Views*'.

The Dimensioning toolbar

The commands located in the *Dimensioning* toolbar allow you to manually create dimensions and tolerances. The sub-toolbars, reading from left to right, are

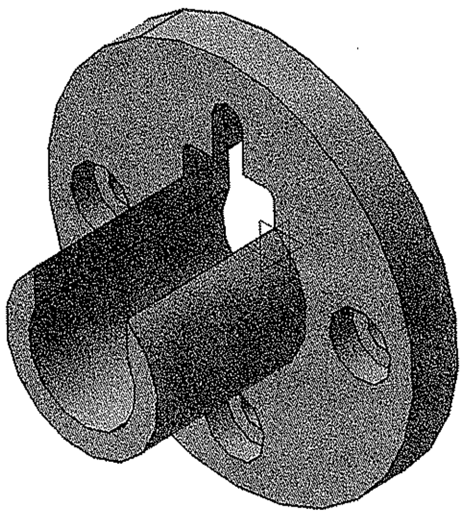


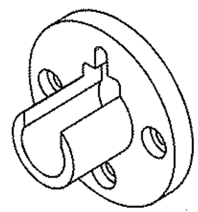
- Dimensions toolbar: The *Dimensions* toolbar contains commands that allow you to manually create dimensions. The commands, reading from left to right, are; *Dimensions, Chained Dimensions, Cumulated Dimensions, Stacked Dimensions, Length/Distance Dimensions, Angle Dimensions, Radius Dimensions, Diameter Dimensions, Chamfer Dimensions, Thread Dimensions, Coordinate Dimensions, Hole Dimension Table,* and *Coordinate Dimension Table*.
- Technological Feature Dimensions toolbar: These commands allow you to dimension technological features. Technological feature dimensioning relies on the fact that technological features can specify the way they should be dimensioned. This allows you to create only realistic and customized dimensions, based on the know-how of a given field.

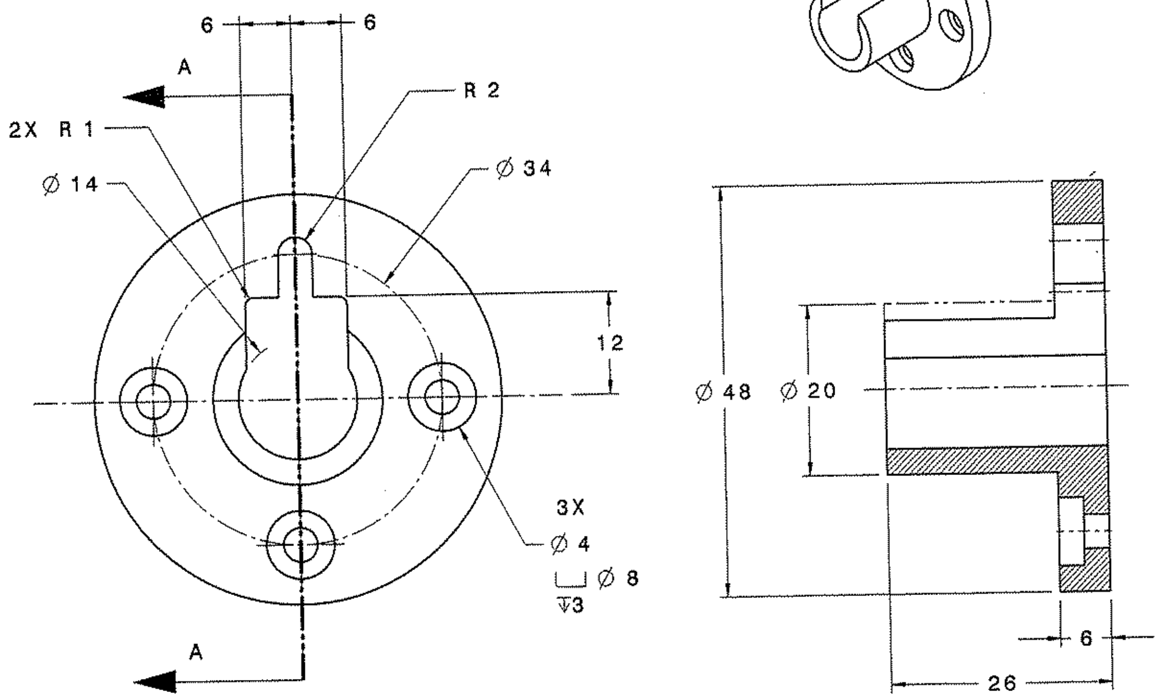
- Dimension Editor toolbar: The commands in this toolbar allow you to re-route dimensions, and create or eliminate interruptions.
- Tolerancing toolbar: The Tolerancing toolbar contains commands that allow you identify Datum features and to apply feature control frames with associated GD&T symbols.

Part/Drawing Modeled

The part modeled in this tutorial is used to illustrate the basic commands available in the *Drafting* workbench. You will learn how to create an orthographic projection that contains a section view, and how to create and place dimensions.







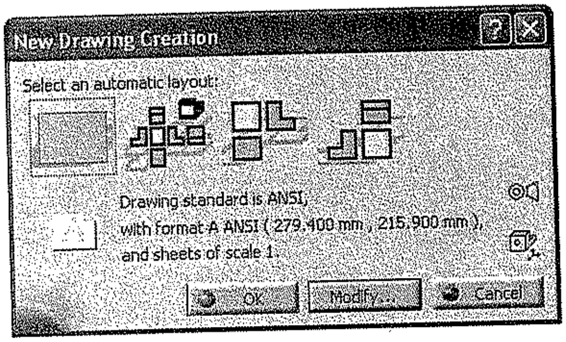
Section 1: Modeling the part

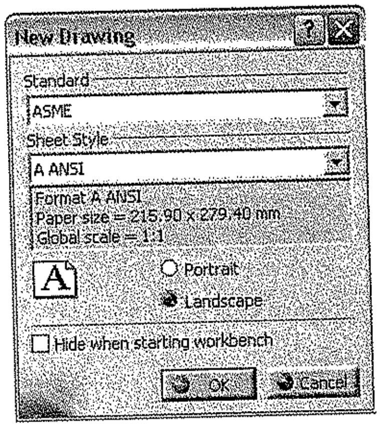
1) Open a **New ...** Part, name it ***Jig*** and save the file as ***T6-1.CATPart***.

2) Model the part shown on the previous page in the *Part Design* workbench and save.

Section 2: Creating standard views

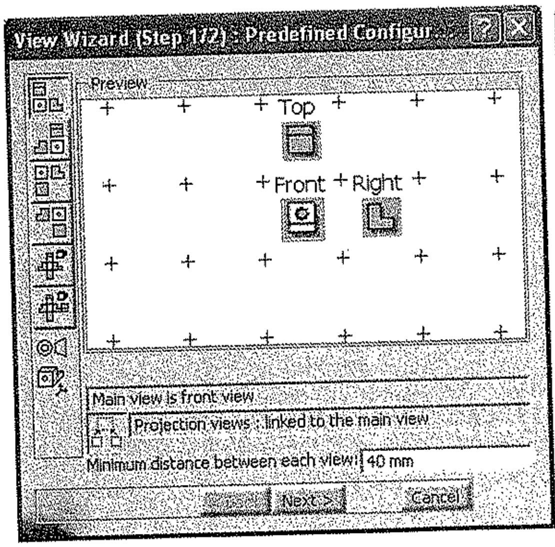
1) With the **PartBody** of your part selected switch to the *Drafting* workbench. In the *New Drawing Creation* window, select **Empty sheet** layout and then the **Modify...** button. In the *New Drawing* window select the **ASME** standard, an **A ANSI** format, and **Landscape** orientation. Select **OK** in both windows.

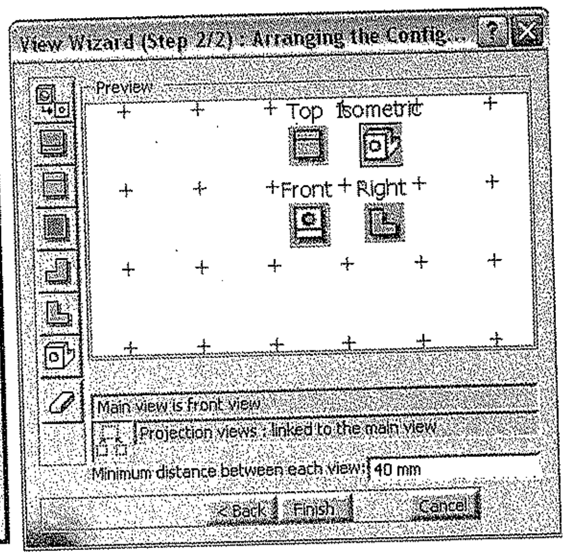




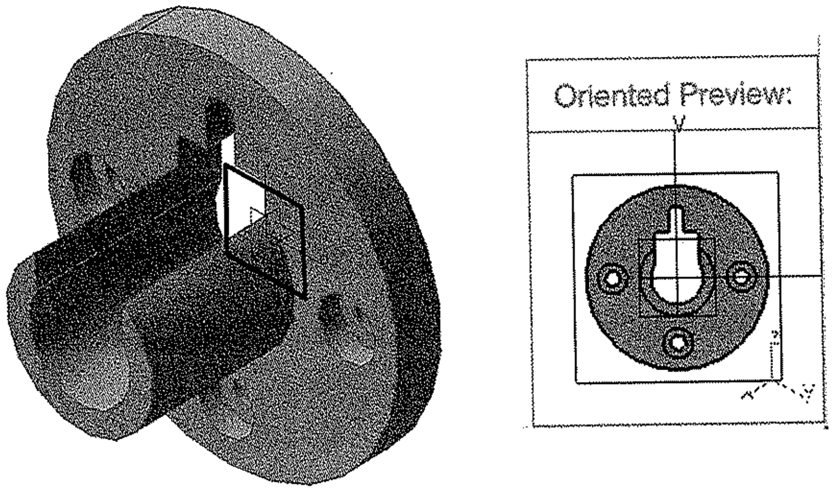
2) When the blank sheet appears, right click on **Sheet.1** in your structure listing (on the left) and select **Properties**. In the Properties window make sure that the **third angle standard** is active.

3) Select the **View Creation Wizard**  icon. In the *View Wizard (Step 1/2) Predefined Configuration* window, select the **Front, Top and Right** configuration icon. Select the **Next >** button. In the *View Wizard (Step 2/2): Arranging the Configuration* window, select the **Isometric view** icon. Select the **Finish** button.

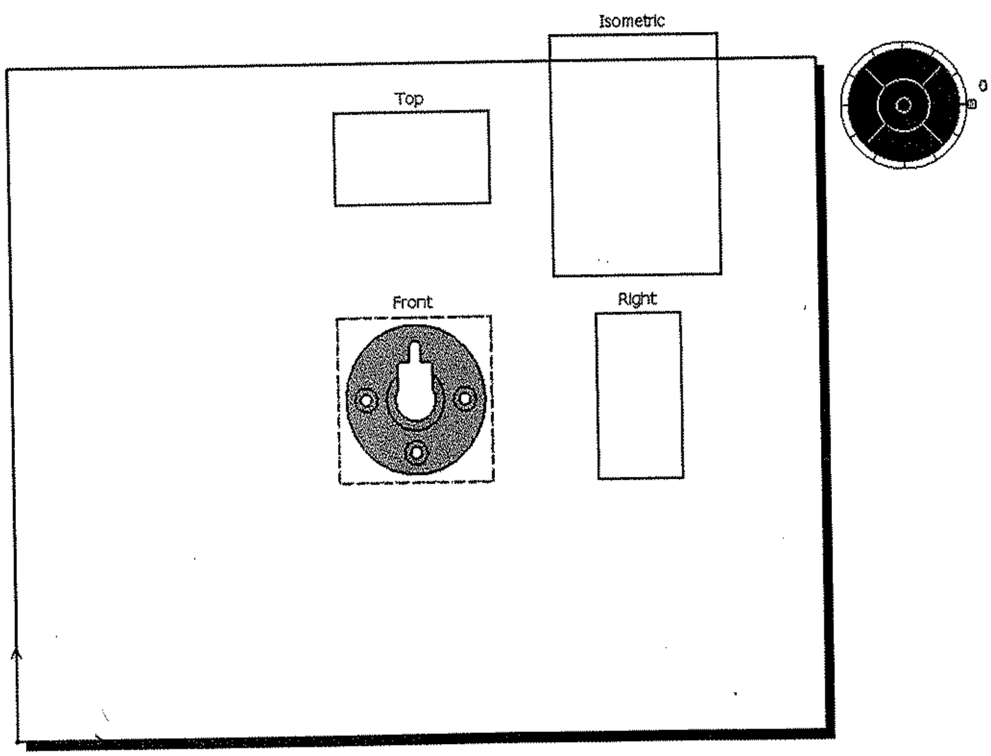




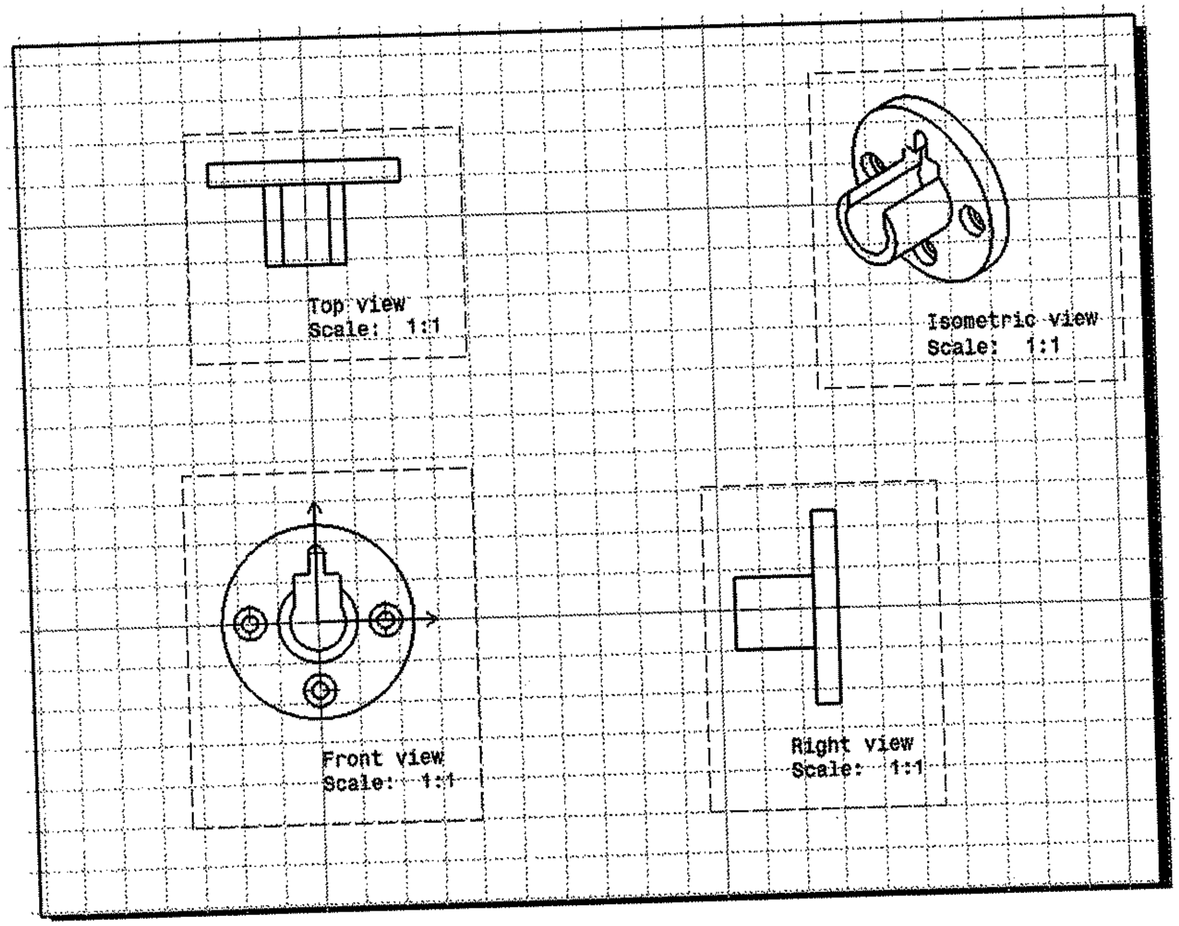
4) The prompt line will read, *Select a reference plane on a 3D geometry*. At the top pull down menu, select **Window** and then your part drawing. Move your mouse over the plane that is parallel with the back of the part. An Oriented Preview will appear in the corner showing you what the front view will look like. If this is acceptable, click on the plane.



5) Your sheet should contain the front view and areas for the top, right and isometric views. These are not necessary in their final positions. The prompt line will read, *Click on the sheet to generate the view or redefine the view orientation using the arrows*. Look at the front view. If the orientation is not correct, you can use the blue circle with four arrows to reorient the front view. If the orientation is correct, click on the sheet.

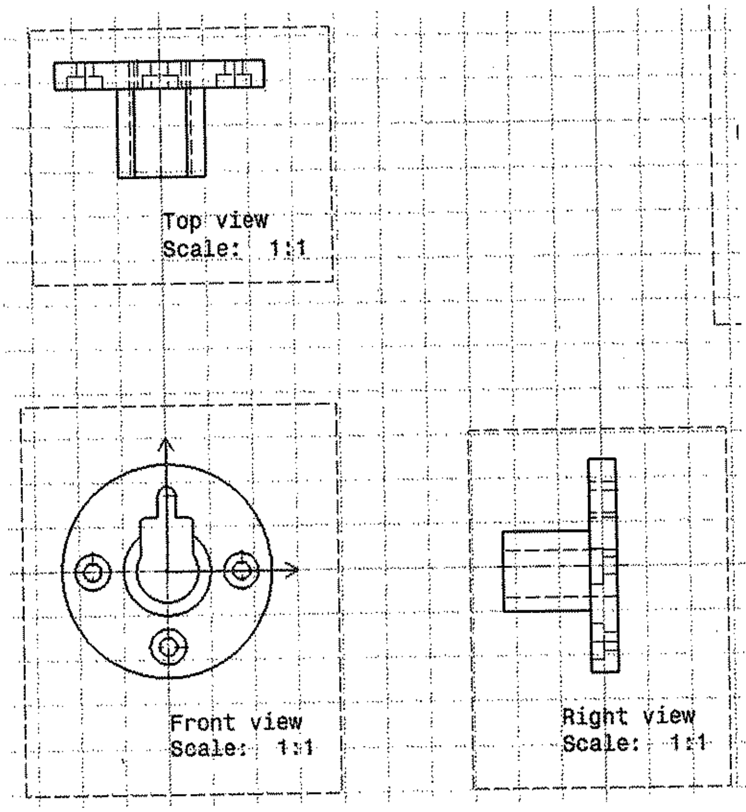


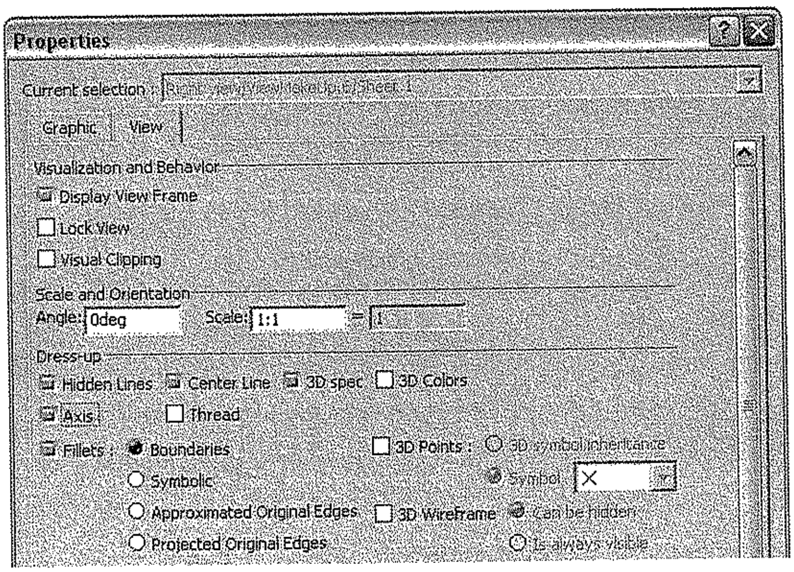
6) After you accept the front view orientation by clicking on the screen, the other views will appear. The front view should be the view that is active (in a red border). If it is not active, double click on it. Move the front view towards the bottom left corner, and then move the other views as shown in the figure. To move a view, click on the view border and move the mouse. Notice that when you try to move the isometric view it is constrained by the location of the front view. Right click on the isometric view's border and select **View Positioning – Position Independently of Reference View**. Then, move the isometric view to the position shown.



7) **Save** your drawing as ***T6-1.CATDrawing***.

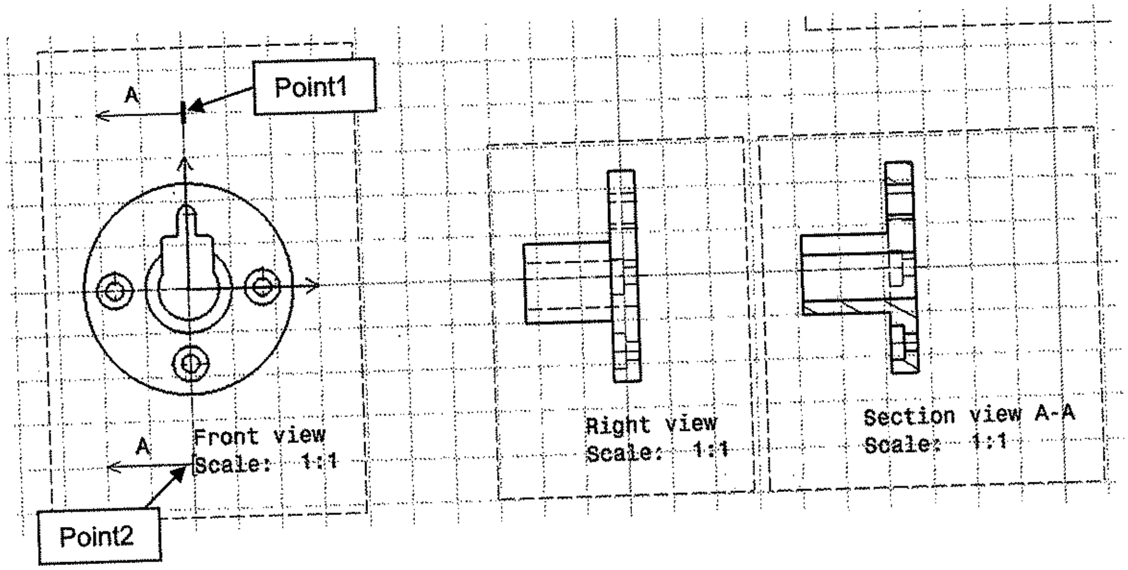
8) Notice that none of the views contain hidden or center lines. Right click on the **Right view** identifier in the structure listing and select **Properties**. In the *Properties* window, activate the **Hidden Lines, Center Lines,** and **Axis** toggles in the *Dress-up* area. Repeat for the front and top views.



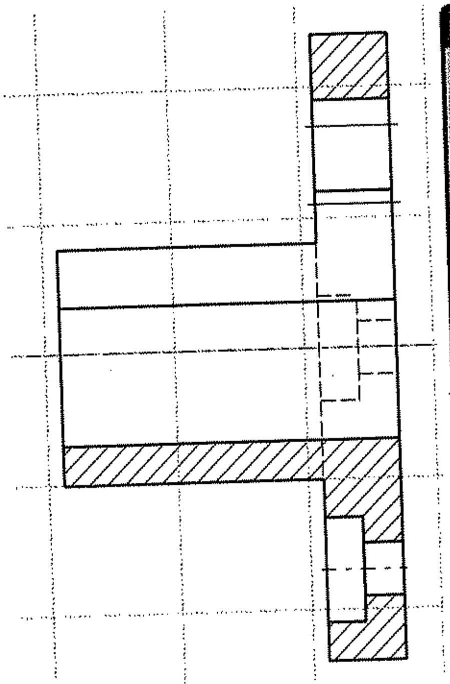


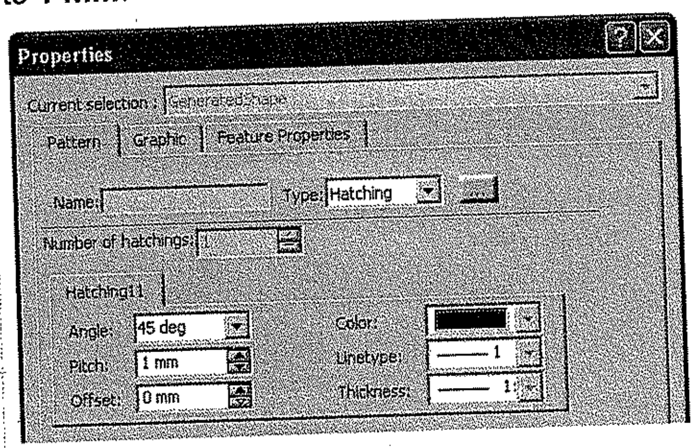
Section 3: Creating section views

1) Create a full section right side view. Select the **Offset Section**  icon. Choose the location of your cutting plane in your front view by selecting point1 as indicated in the figure and then double clicking at point2. A shaded view will appear. Place it to the right of the right side view and click.



2) Change the look of the section lines. In the section view, double click on the section lines. In the *Properties* window click on the **Pattern** tab and change the angle to ***45°*** and the pitch to ***1 mm***.

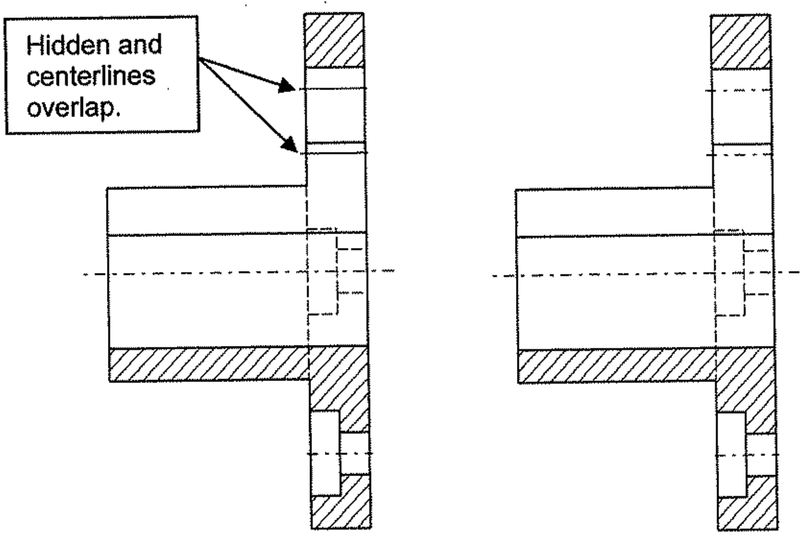




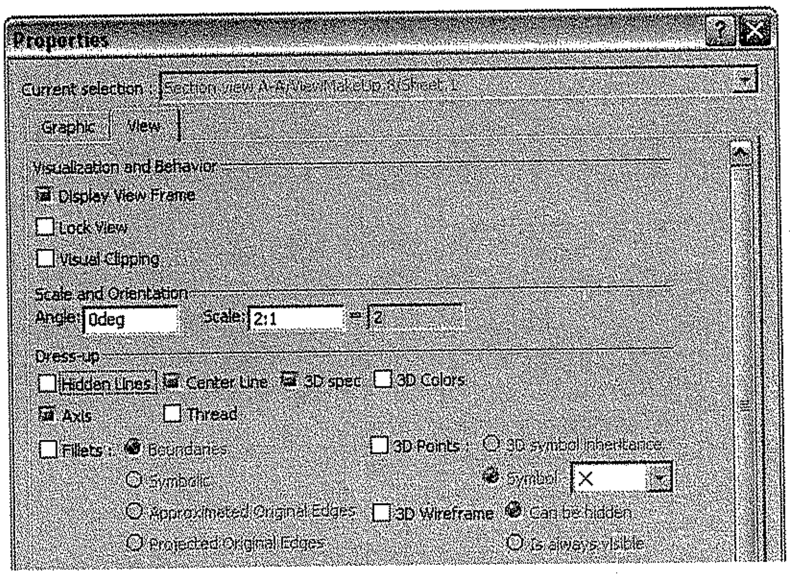
Section 4: Cleaning up the views

1) Delete the right side unsectioned view and the top view for these views are not needed. Right click on the border of the view and select **Delete**.

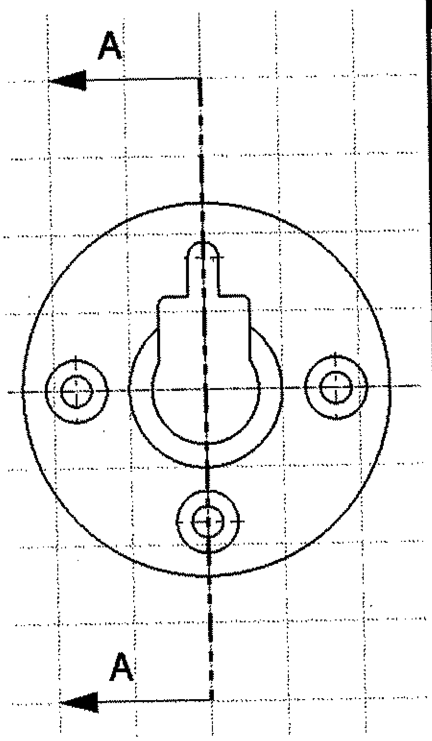
2) Notice that in the right side sectioned view there are two places where two lines (center and hidden) are coincident. According to drafting standards only one of those lines should be shown. In this case the hidden lines have no reason for being there (they are used to show a fillet). Enter the *Properties* window for the right side view. Deselect the **Fillet** toggle.

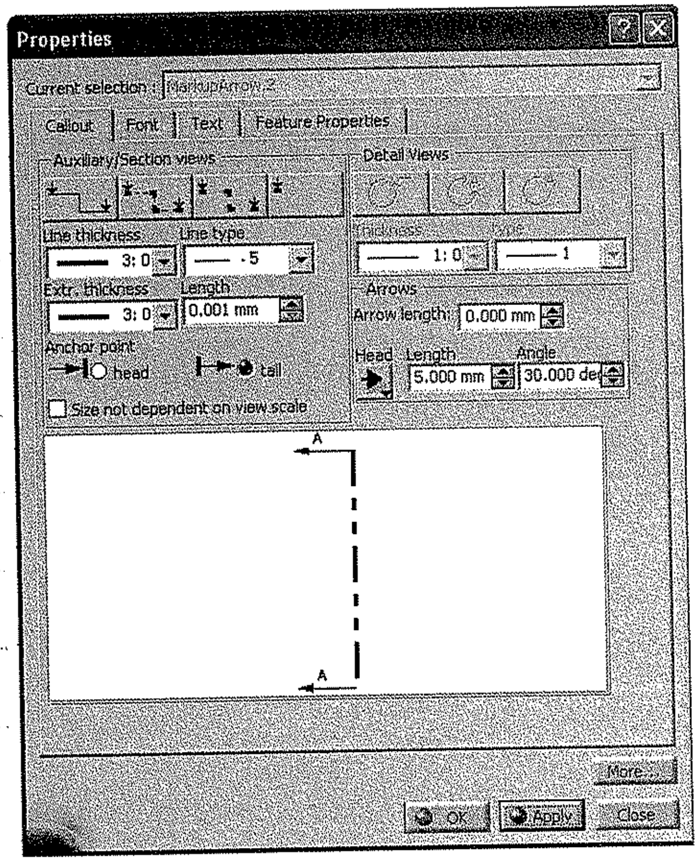


3) Section views usually do not contain hidden lines. Eliminate the hidden lines in the right side section view.



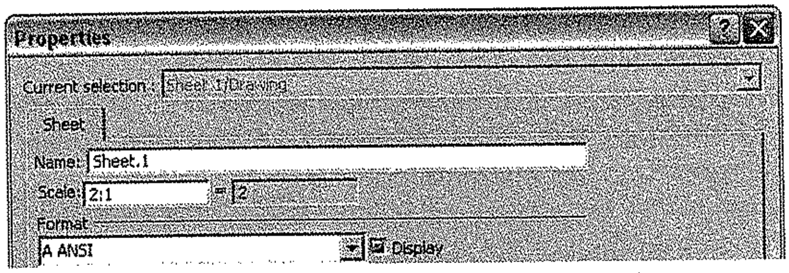
4) Change the properties of the cutting plane line. Right click on the cutting plane line and select **Properties**. In the *Properties* window, click on the **Call out** tab and set the line thickness to ***3*** and the line type to ***5***. Set the length of the arrow base to ***0.001 mm***. Set the arrow head type to solid, the length to ***5 mm***, and the angle to ***30°***.





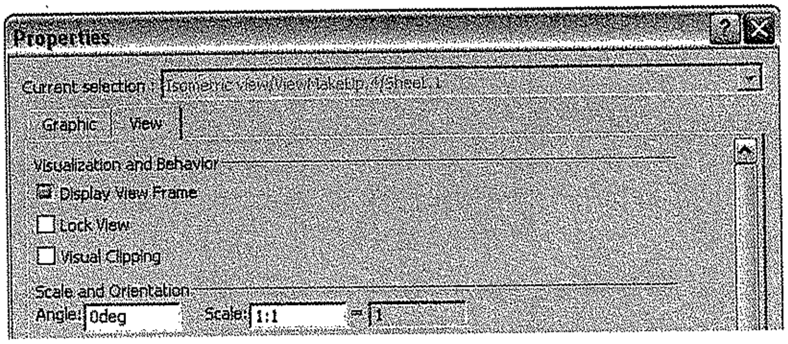
5) Hide the view names. Right click on each view name and select **Hide/Show**.

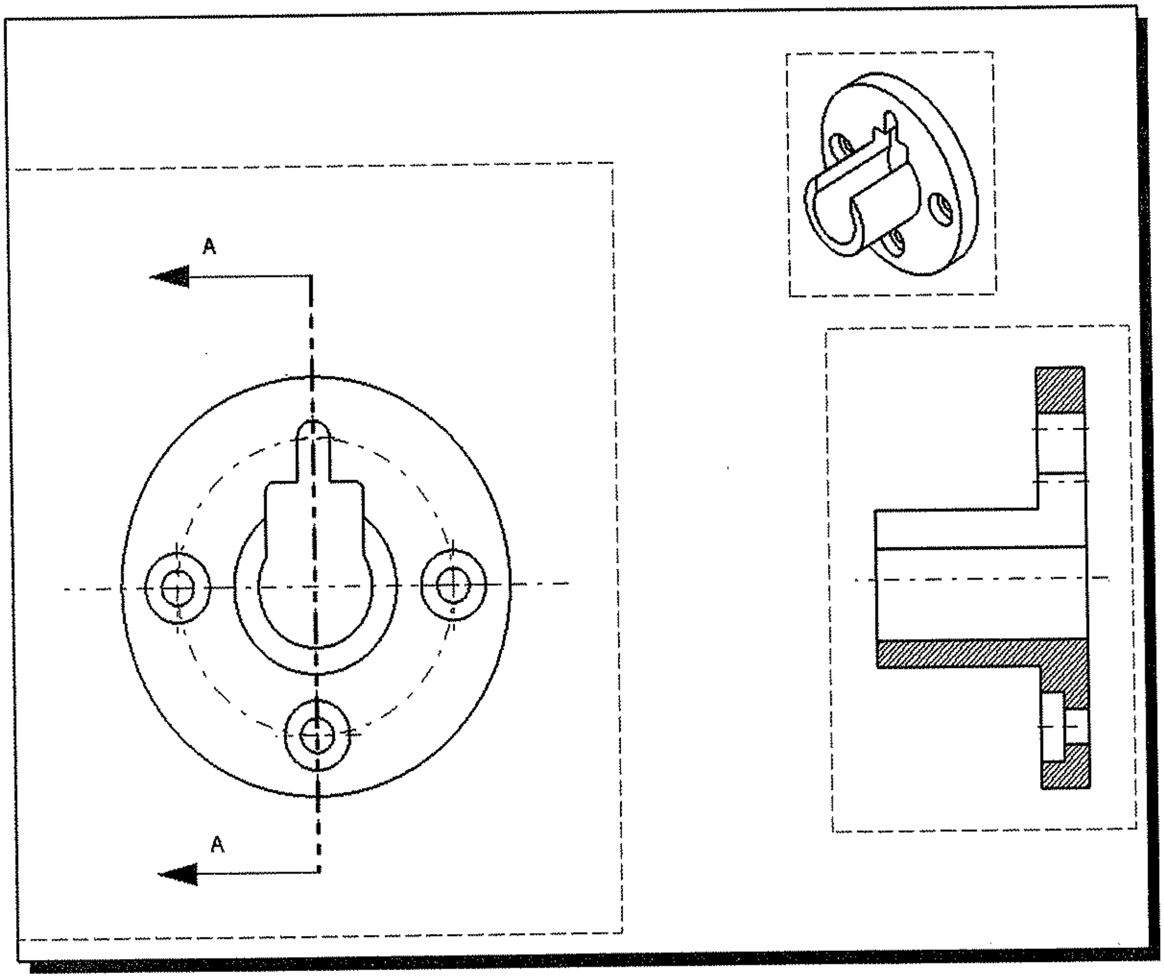
6) Change the scale of the views to 2:1. Right click on Sheet.1 and select **Properties**. In the *Properties* window set the scale to ***2:1***.



7) Change the scale of the isometric view back to **1:1**.

8) Save your drawing.

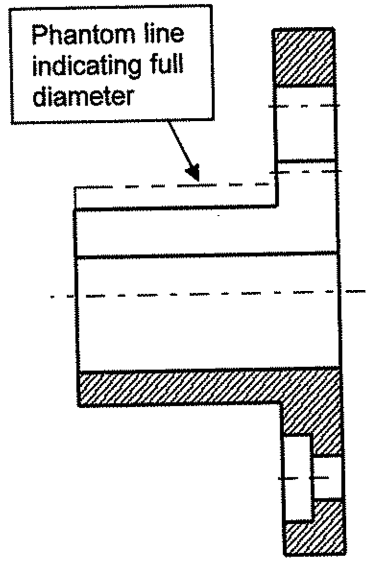


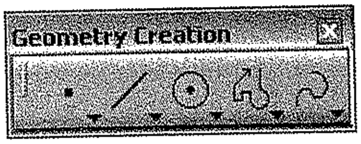


<u>Section 5:</u> Dimensioning

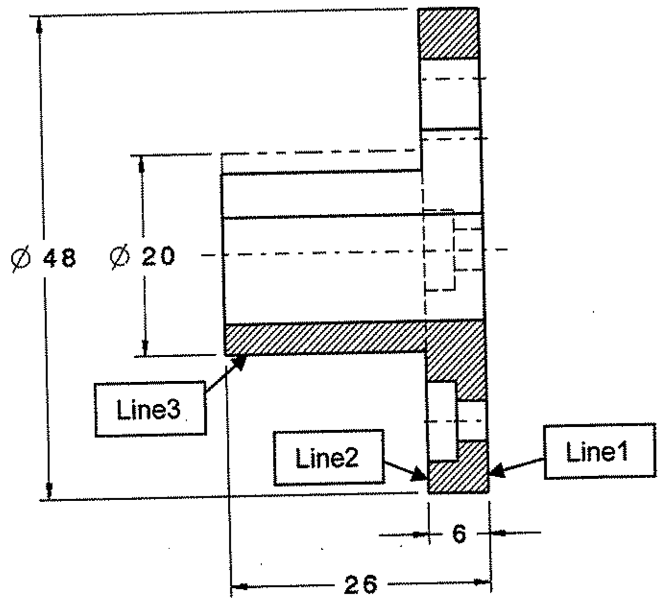
1) Make the right side view active.

2) Draw a phantom line that indicates the full diameter of the hub. Double click on the right side section view to make it active. Draw two lines as indicated in the figure. Select both lines, right click, and select **Properties**. In the *Properties* window, set the line thickness to **1** and the line type to **5**. Commands such as line, circle, etc... are located in the *Geometry Creation* toolbar. (Hint: Start the vertical line at the centerline so that you know how long to make it.)





3) Place the linear dimension in the right side view. Select the **Linear Dimension**  icon. Select *Line1* and then *Line2*. Repeat, using a similar procedure, for the 26 mm dimension. Note: You can move the arrows in and outside of the extension lines by clicking on them. You can also drag the texts outside the extension lines.

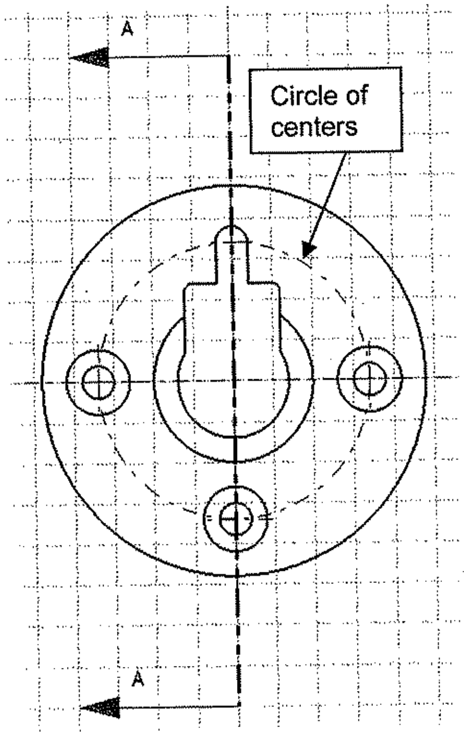


4) Select the **Diameter Dimension**  icon. Select *Line3*. The Ø20 mm diameter dimension should appear. If it does not, select the horizontal phantom line. Repeat for the Ø48 mm diameter dimension.

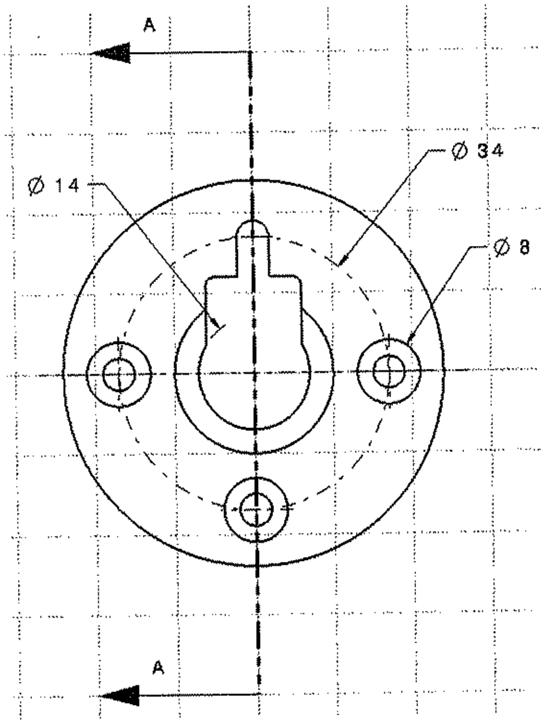
5) Activate the front view by double clicking on it.

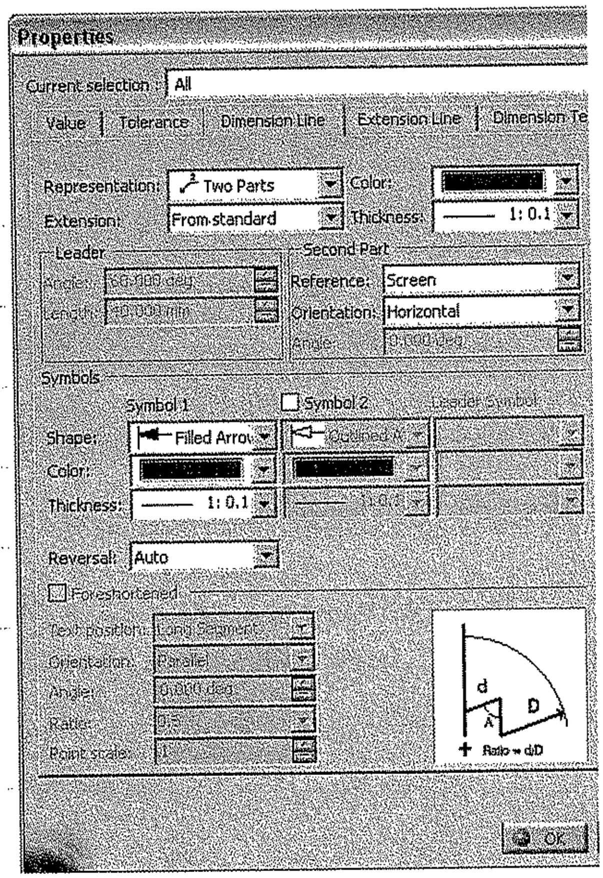
6) Draw a **Circle**  that identifies the circle of centers of the counterbored holes.

7) Right click on the circle and select **Properties**. Change the line type to ***4*** and the line thickness to ***1***.

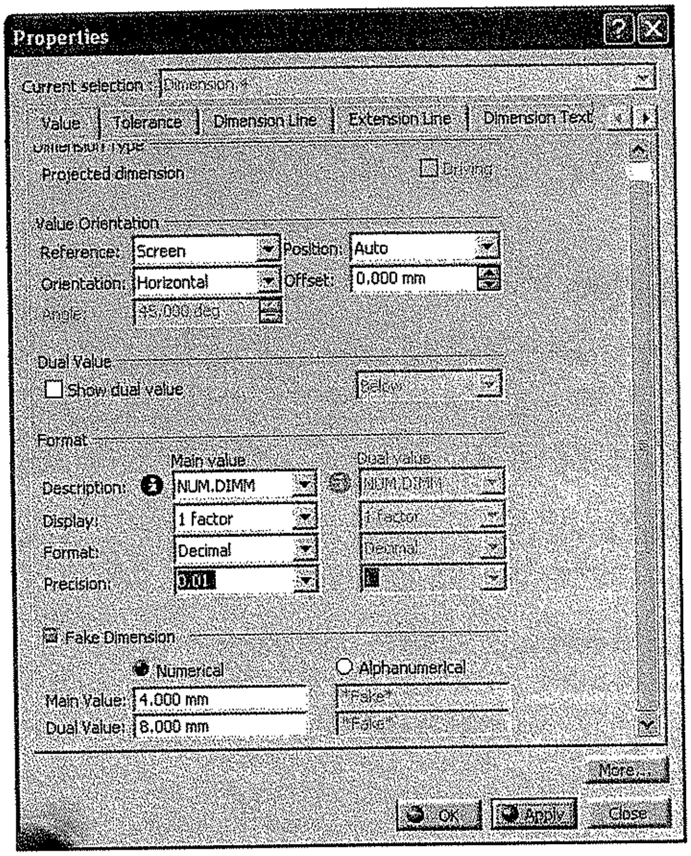


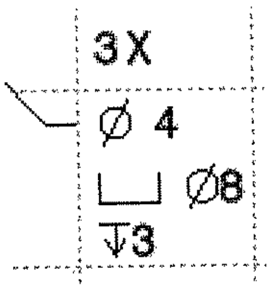
8) Dimension the 3 **Diameter Dimensions**  shown in the figure. Notice that the dimensions do not look like the dimensions shown in the figure. Select all 3 dimensions holding the Ctrl key and access the *Properties* window. Adjust the **Dimension Line** settings as shown. Position the dimensions as shown.

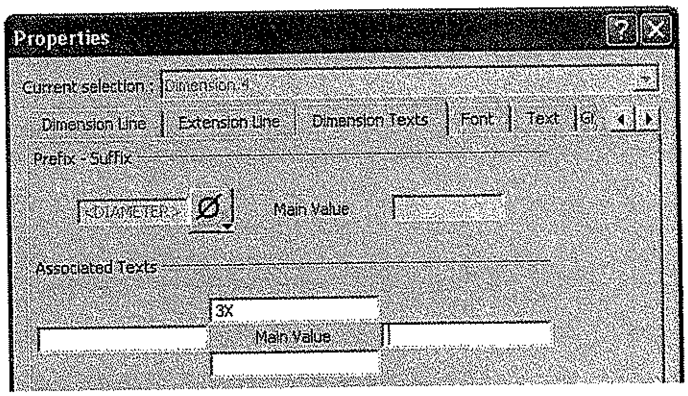




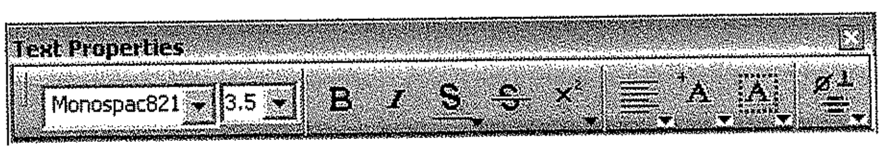
9) Notice that the dimension for the counterbored hole is not correctly stated. It shows only the diameter of the counterbore. Access the *Properties* window for that dimension. Select the **Value** tab and activate the **Fake Dimension** toggle. Enter a `Main Value:` of ***4 mm***. This is the diameter of the hole. Select the **Dimension Text** tab. Enter ***3X*** in the top box of the `Main Value` field. This indicates that there are 3 counterbored holes.

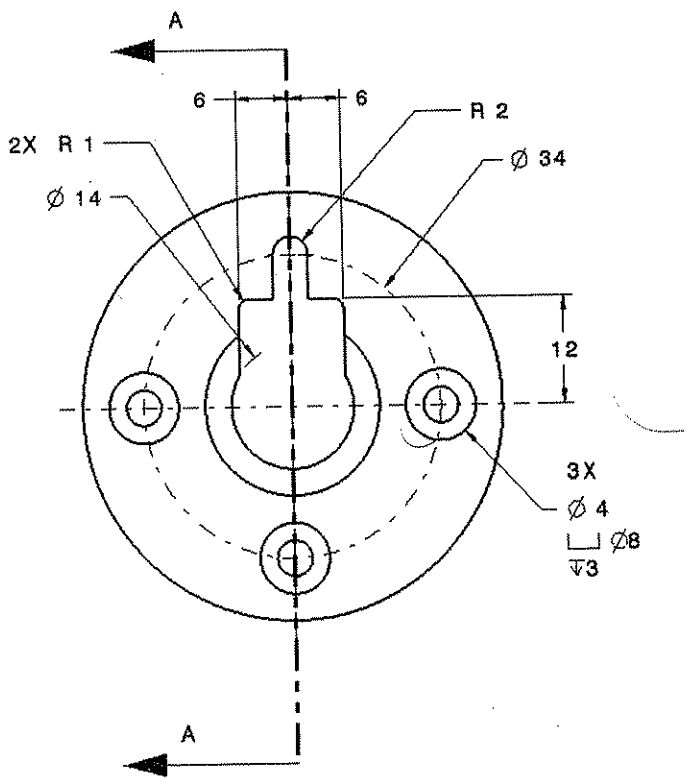






10) We still need to add the counterbore diameter and depth. Select the **Text** icon. Click your mouse under the Ø4 dimension. A *Text Editor* window will appear. At the top of the screen there should be a *Text Properties* toolbar. Click on the **Insert Symbol** icon and select the counterbore symbol, type a space, select the diameter symbol Ø, and then type ***8***. Hit Enter to start a new line. Select the deep symbol, and then type ***3***.



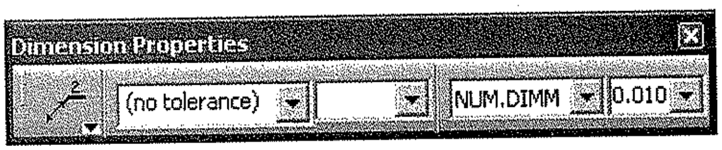


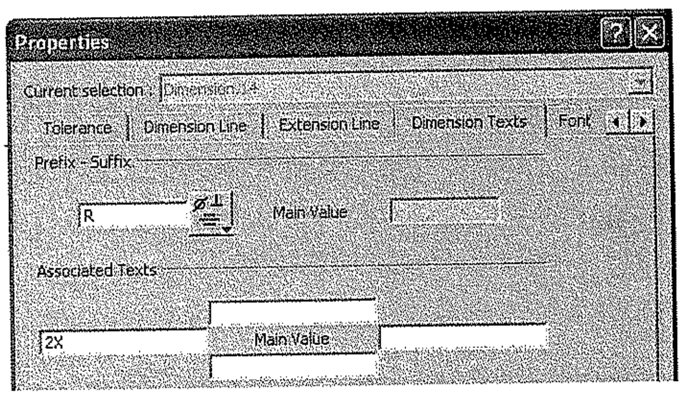
11) Move the counterbore dimension and text down as shown in the figure. Hold the Shift key down while moving the counterbore text to avoid a jerky motion.

12) **Save** your drawing.

13) Dimension the rest of the linear features using the **Linear Dimension** .

14) Dimension the radius dimensions using the **Radius Dimension**  commands. Notice that the radial dimension don't look like what is shown in the figure. At the top of the screen there should be a *Dimension Properties* window. Select one of the radial dimensions and select the **Dimension Line**  configuration. Right click on the R1 dimension and select **Properties**. In the *Properties* window, select the **Dimension Texts** tab and enter **2X** in the left box of the `Main Value` field.





15) Position your views and then select the **Display View Frame as Specified for Each View**  icon. This should turn off all of the view frames. This icon is usually located in the bottom toolbar area. If you are having trouble locating this icon, access the *Properties* window for each view and deactivate the **Display View Frame** toggle.

16) **Save** your drawing.