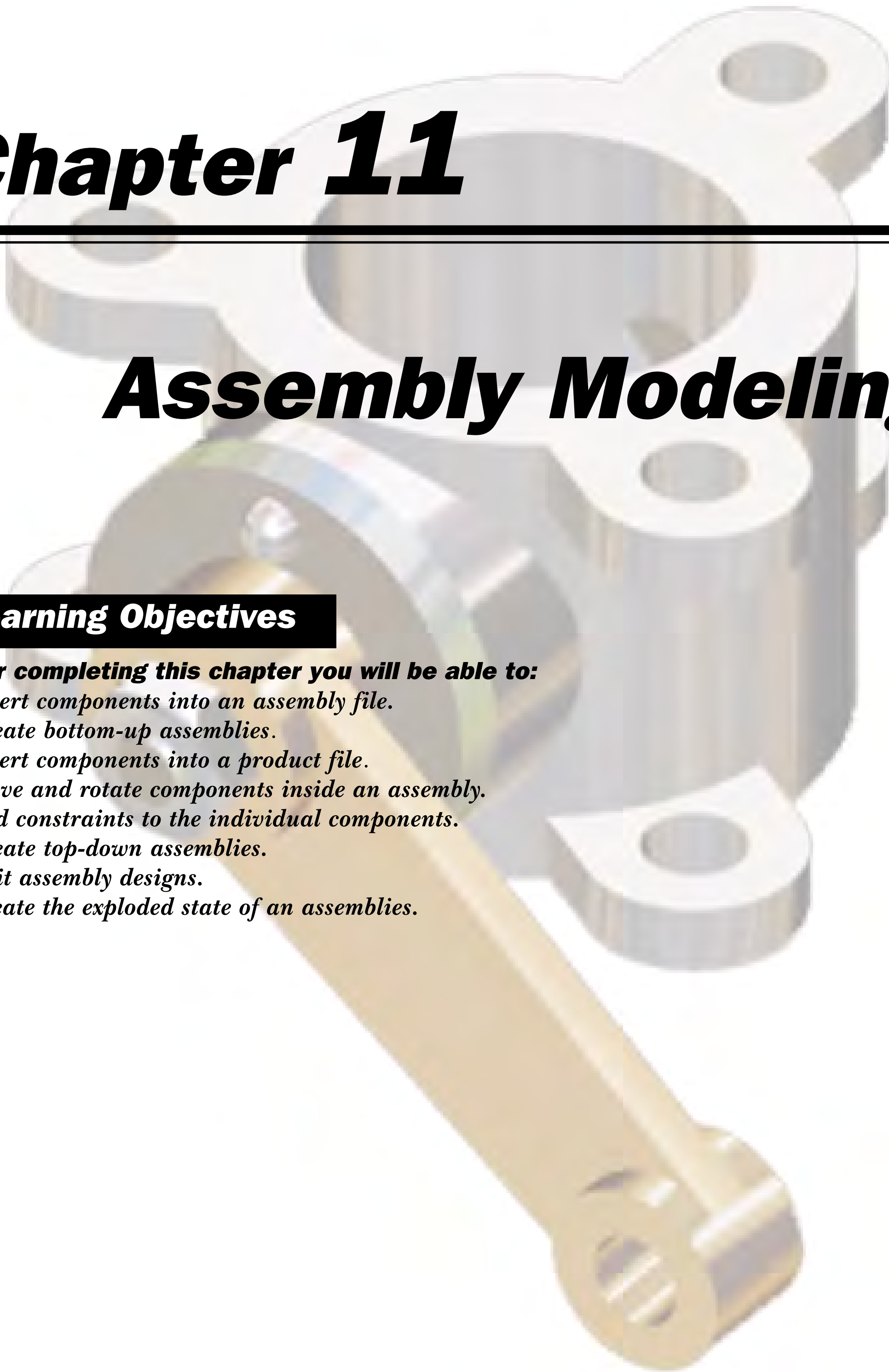
Chapter 11

Assembly Modeling



Learning Objectives

After completing this chapter you will be able to:

- *Insert components into an assembly file.*
- *Create bottom-up assemblies.*
- *Insert components into a product file.*
- *Move and rotate components inside an assembly.*
- *Add constraints to the individual components.*
- *Create top-down assemblies.*
- *Edit assembly designs.*
- *Create the exploded state of an assemblies.*

ASSEMBLY MODELING

Assembly modeling is the process of creating designs that consist of two or more components assembled together at their respective work positions. The components are brought together and assembled in **Assembly Design** workbench by applying suitable parametric assembly constraints to them. The assembly constraints allow you to restrict the degrees of freedom of components on their respective work positions. The assembly files in CATIA are called Product files. There are two methods to invoke the **Assembly Design** workbench of CATIA. The primary method to start a new product file is by selecting **File** > **New** from the menu bar to open the **New** dialog box From this dialog box select **Product**, as shown in Figure 11-1. The other method of invoking the **Assembly Design** workbench is by choosing **Start** > **Mechanical Design** > **Assembly Design** from the menu bar.

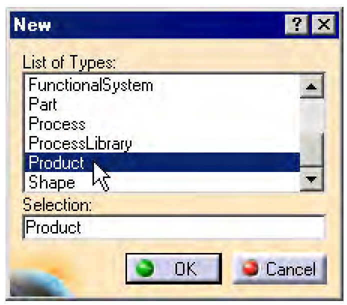


Figure 11-1 *The* ***Product*** *option selected from the* ***New*** *dialog box*

A new file is started in the **Assembly Design** workbench. The screen display of CATIA after starting the new file in the **Assembly Design** workbench is as shown in Figure 11-2. You will notice that the toolbars related to assembly are displayed. The tools available in these toolbars will be discussed later in this chapter.

Types of Assembly Design Approach

In CATIA you can create assembly models by adopting two types of approaches. The first design approach is the bottom-up approach, and the second one is the top-down approach. Both these design approaches are discussed below.

Bottom-up Assembly

The bottom-up assembly is the most preferred approach for creating assembly models. In this of approach, the components are created in the **Part Design** workbench as (*.*CATPart*) file. Then the product (*.*CATProduct*) file is started and all the previously created components are inserted and placed in it using the tools provided in the **Assembly Design** workbench. After inserting each component, constraints are applied to position them properly in the 3D space with respect to other components.

Adopting the bottom-up approach gives the user the opportunity to pay more attention to the details of the components as they are designed individually. Because the other components are not present in the same window, it becomes much easier to maintain a relationship between

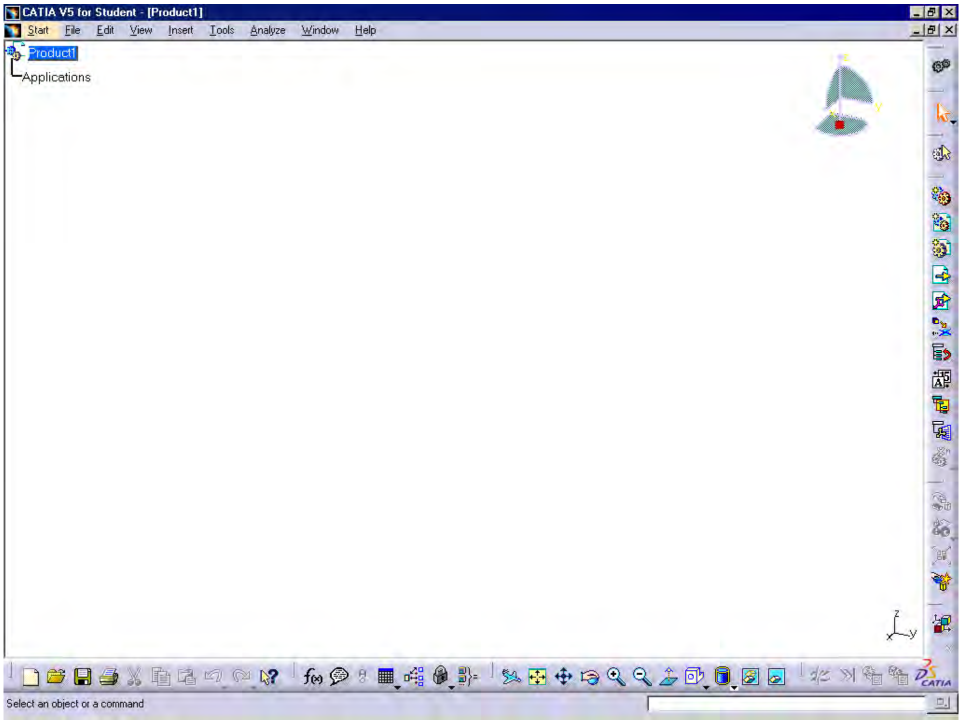


Figure 11-2 *Screen display after starting a new file in the* ***Assembly Design*** *workbench*

the features of the current component. This approach is preferred for large assemblies, especially those having intricate individual components.

Top-down Assembly

In the top-down assembly design approach, components are created inside the **Assembly Design** workbench. Therefore, there is no need to create separate part files of the components. This design approach is completely different from the bottom-up design approach. Here you have to start the product file first and then, one by one, create all components. Note that even though the components are created inside the product file, they are saved as individual part files and can be opened separately later.

Adopting the top-down design approach gives the user the distinctive advantage of using the geometry of one component to define the geometry of the other. Here the construction and assembly of the components takes place simultaneously. As a result of this, the user can view the development of the product in real time. This design approach is highly preferred, while working on a conceptual design or a tool design where the reference of previously created parts is required to develop a new part.

Note

An assembly can also be created by using the combination of both the top-down and bottom-up assembly design approaches.

CREATING BOTTOM-UP ASSEMBLIES

As mentioned earlier, while creating an assembly using the bottom-up approach, the components are created in separate part files and are then inserted into the assembly file. They are assembled at their working position by applying assembly constraints to them. To create an assembly using this approach, it is recommended to insert the first component and fix its position after properly orienting it in the 3D space. The other components can be inserted and positioned with reference to the first component. The method used for placing components inside the product file is discussed below.

Inserting Components in a Product file

Menu:	Insert > Existing Component
Toolbar:	Product Structure Tools > Existing Component

To insert the first component in the product file, choose the **Existing Component** button from the **Product Structure Tools** toolbar. You are prompted to select a component into which the existing component will be inserted. You need to select **Product1** from the **Specification Tree**. After you do so, the **File Selection** dialog box is displayed. Browse the location where the part files are saved and double-click on the component to be inserted; the component will be inserted in the current product file. You will notice a new entry in the **Specification Tree**, which is referred to as **Part1**. **Part1** is a default part number assigned by the software to the component. A default part number is assigned to each component that is inserted in the assembly, unless it is changed by the user. Inside the assembly, the components are referred to by their part number and not by their file name. The process of changing the part number of the component is discussed later in this chapter. It is always recommended to fix the first component using the **Fix** constraint after inserting. The method of applying the **Fix** constraint to the component is discussed later in the chapter.

The above procedure needs to be repeated for inserting the next component. When you insert additional components, the **Part number conflicts** dialog box is displayed, as shown in Figure 11-3. This dialog box is displayed because there is a clash between the part numbers of the previously inserted component and the currently inserted component. Note that in the selection area of the dialog box, the numbers of both the components are displayed as Part1, but the names of the files are different. You can change the part number of the

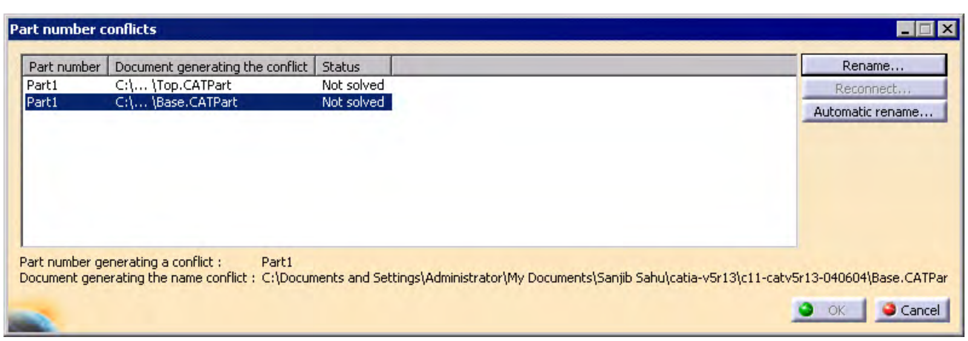


Figure 11-3 *Part number conflict dialog box*

component using the options available in this dialog box. There are two active buttons available on the right of the selection area of this dialog box: **Rename** and **Automatic Rename**.

If you select the component to be renamed and choose the **Automatic rename** button, the part number of the selected component is renamed from **Part1** to **Part1.1**. Choose the **OK** button from the **Part number conflicts** dialog box to insert the second component into the Product file. Follow the same procedure to rename the part number, while inserting other components. Note that while inserting the third component, the first time when you rename the component using the **Automatic Rename** option the part number is changed to **Part1.1**. Because this part number is already assigned to the second component, the **Part number conflicts** dialog box is again displayed after choosing the **OK** button and shows the conflict between the second and third component. You need to choose the **Automatic rename** button again to change the part number of the third component. Now, the third component will be renamed from **Part1.1** to **Part1.1.2**. Choosing the **OK** button will insert the third component into the product file. This means if you are inserting the n^{th} component, the automatic rename button has to be used n-1 times. This way the part number of every new component keeps on changing in a similar fashion, and the same is represented in the **Specification Tree**, as shown in Figure 11-4.

***Figure 11-4 Specification Tree** showing four components*

If you choose the **Rename** button from the **Part number conflicts** dialog box, the **Part Number** dialog box is displayed, as shown in Figure 11-5.

***Figure 11-5** The **Part Number** dialog box*

In this dialog box, you can enter the new part number for the selected component based on your requirement. After typing the new part number in the text box provided in the dialog box, choose the **OK** button to exit the **Part Number** dialog box. Now, choose the **OK** from the **Part number conflicts** dialog box to insert the component in the product file. Ideally the part number entered should be the same as the file name. If you enter the same part number for two different components, it will not be accepted by the software, and the **Part number conflicts** dialog box will again be displayed. Again choose the **Rename** button and enter a unique name for that part such that it does not conflict with any other part number. The advantage of using this option is that the user can enter the desired part number, which can be useful especially when the individual components are referred to in the an assembly using number coding. The **Specification Tree** showing individual part numbers is shown in Figure 11-6.

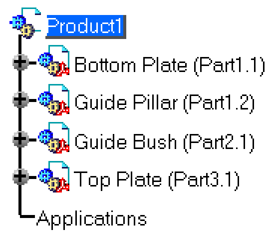
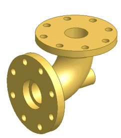


***Figure 11-6 Specification Tree** showing four components with unique part numbers*

Note that in the **Specification Tree**, the part numbers of each component is suffixed by the instance number, which is displayed within parenthesis. This instance number is generated by the software itself and is unique for each component.

When a component is inserted into a product file, its placement in the 3D space depends on the location of its default planes. The default planes of the component are placed over the default planes of the product file. The default planes of the product file are not visible, but are present at the center of the screen, unless moved by panning. When more than one components are inserted into the product file, the default planes of all components are placed one over the other hence appearing as one set of default planes. When the components are moved away, the defaults planes of each component are distinctly visible. You will learn more about moving the components later in this chapter.

Note
If the default planes of the inserted components are not visible, this means its visibility is turned off in its part file. Therefore, you need to turn on the visibility of the reference planes in the part file to display them in the assembly file.



Tip. *You can also insert components in the product file using the Copy and Paste method. To insert the components using this method, open the part file of the component that you need to insert. Select the name of the component from top of the **Specification Tree**, and choose **Copy** from the contextual menu. Now, switch to the product file, and select the name of the assembly on top of the **Specification Tree**. Invoke the contextual menu, and choose **Paste** from it; the component will be placed in the assembly.*

Moving Individual Components

Generally, the components when inserted in a product file are overlapped by other components placed earlier. As a result, their visualization is hampered, and it becomes difficult to apply constraints to them. Therefore, it is necessary to reposition the components in the 3D space such that they are distinctly visible, and the mating references are accessible in the assembly. CATIA allows you to move and rotate the individual unconstrained components inside the product file without affecting the position and location of the other components. The reorientation of the component can be carried out using three different methods, which are discussed in the following sections.

Moving and Rotating Using the Manipulation Tool

Toolbar:	Move > Manipulation
Menu:	Edit > Move > Manipulate

The **Manipulation** tool is used to move or rotate the component freely by dragging the cursor. To translate or rotate any component, choose the **Manipulation** button from the **Move** toolbar; the **Manipulation Parameter** dialog box is displayed, as shown in Figure 11-7.

Figure 11-7 *Manipulation Parameter dialog box*

This dialog box contains buttons arranged in three rows. The currently active button is displayed on the top of the dialog box. The buttons in the first row are used to translate the component along a particular direction. There are four buttons in this row, which are discussed below:

The **Drag along X axis** button is the first button and is chosen by default. This button is used to translate the selected component along the X-axis of the assembly coordinate system. To move the component, select the component to move and then drag it. After moving the component to the desired location, release the left mouse button.

The **Drag along Y axis** button is used to translate the component along the Y-axis of the assembly coordinate system. It works similar to the button discussed above. After choosing the **Drag along Y axis** button, select the component to move and then drag it.

The **Drag along Z axis** button is used to translate the component along the Z-axis of the assembly coordinate system.

The **Drag along any axis** button is used to move the component along a selected direction. After choosing the **Drag along any axis** button, you need to select a direction to define the translation axis. This direction can be a line, an edge, or an axis of a cylindrical feature. After selecting the axis of translation, drag the selected component along the selected direction.

The buttons in the second row of the **Manipulation Parameter** dialog box are used to move the selected component along a particular plane. These planar translation buttons are discussed below.

The **Drag along XY plane** button is used to translate the selected component parallel to the XY plane of the assembly coordinate system.

The **Drag along YZ plane** button is used to translate the selected component parallel to the YZ plane of the assembly coordinate system.

Drag along XZ plane button is used to translate the selected component parallel to the XZ plane of the assembly coordinate system.

Drag along any plane button is used to move the selected component parallel to a specified plane. After choosing the **Drag along any plane** button, you need to select a plane for planar translation. This plane can be a construction plane, a planar face, or a surface. After selecting the plane for translation, select the component to move and then drag it on the selected plane.

The buttons on the third row of the **Manipulation Parameter** dialog box are used to rotate the selected component around an axis. These rotation buttons are discussed below.

The **Drag around X axis** button is used to rotate the selected component around the X-axis of the assembly coordinate system.

The **Drag around Y axis** button is used to rotate the selected component around the Y-axis of the assembly coordinate system.

The **Drag around Z axis** button is used to rotate the selected component around the Z-axis of the assembly coordinate system.

The **Drag around any axis** button is used to rotate the selected component around a specified axis. After choosing the **Drag around any axis** button, you need to select a line to define the rotation axis. This line can be an edge of the component or an axis of a cylindrical feature. After selecting the axis for rotation, select the component to rotate and then drag it.

The **With respect to constraints** check box is selected to move or rotate the components within its available degrees of freedom after applying constraints. You will learn more about applying constraints later in this chapter.

Moving Components Using the Snap Tool

Toolbar:	Move > Snap > Snap
Toolbar:	Edit > Move > Snap

The **Snap** tool is used to move the component by snapping the geometric element of first component on the other component or on the same component. The movement of the component depends on the selection of the geometric elements. The element selected first will move to snap the second element. For example, if you first select a line and then a point, the line will be reoriented in such a way that it passes through the selected point.

To move a component using the **Snap** tool, choose the **Snap** button from the **Move** toolbar. You are prompted to select on a component the first geometric element: an axis system, a point, a line or a plane. Select a suitable geometrical element that will be projected on the next selection. In Figure 11-8, the upper right edge of the left component is selected as the first geometrical element.

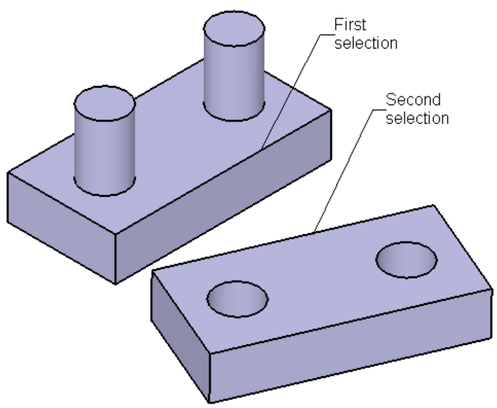


Figure 11-8 *Geometric elements selected to snap*

Next, you are prompted to select on the same component or on another component the second geometric element: a point, a line or a plane. Select the second geometric element on which you need the first selection to be snapped. In Figure 11-8, the upper left edge of the right component is selected as the second geometric element. The first selection will be snapped to the second selection, and a green arrow will be displayed at the snapping location, as shown in the Figure 11-9. You can click on the arrow to reverse the snapping direction, else click anywhere in the geometry area to exit the tool. Figure 11-10 shows the components, after the snapping direction is reversed.

In some cases the arrows are not displayed when you snap the two elements such as snapping a point to a point, a point to surface, a point to a cylindrical surface or a planar surface, and so on.

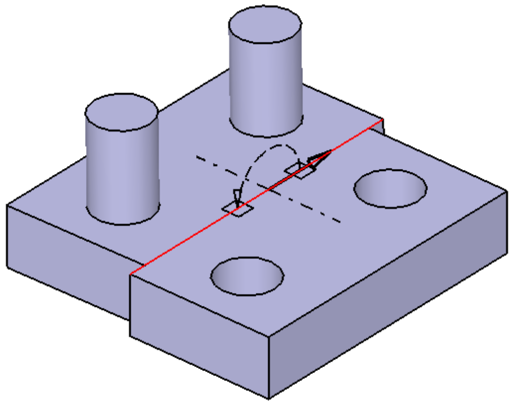


Figure 11-9 *Position of components after snapping*

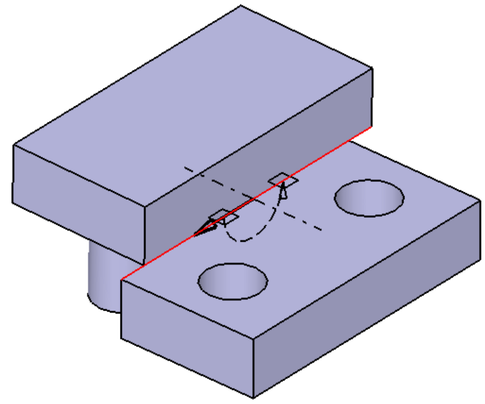


Figure 11-10 *Position of components after snapping direction is reversed*

Moving Components Using the Smart Move Tool

Toolbar:	Move > Snap > Smart Move
Menu:	Edit > Move > Snap

The **Smart Move** tool works as a multipurpose tool. This tool has the capability of manipulating and snapping components and can also apply constraints to them, if required. To invoke this tool, choose the down arrow besides the **Snap** button to invoke the **Snap** toolbar. Choose the **Smart Move** button to invoke the **Smart Move** dialog box. Choose the **More** button to expand it, as shown in Figure 11-11.

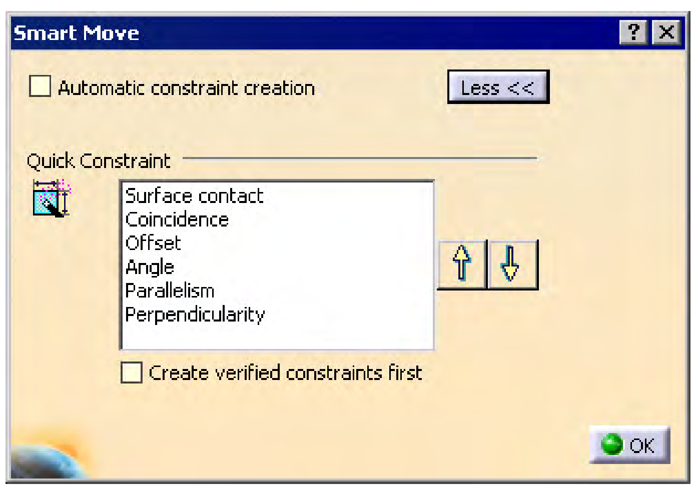


Figure 11-11 *The expanded **Smart Move** dialog box*

If the **Automatic constraint creation** check box is selected, a permanent constraint will be applied between the selected elements of the components to be snapped and the same will be displayed in the **Specification Tree**. If this check box is not selected, the components will be repositioned but no permanent constraint will be applied. You will learn more about applying constraints later in this chapter.

The **Quick Constraint** area displays the constraints in a hierarchial order. These constraints will be applied to the components, while they are being snapped. If more than one constraint can be applied to the current selection set, then priority will be given to the constraint, which on the top in the hierarchy. To change the position of a constraint, select it and choose the **Up** or **Down** arrows on the right side of the **Quick Constraint** area.

After settings the options in this dialog box, select the first geometric element on a component, which can be a point, line, plane, planar face, or circular face. Now, select the second geometric element on the other component. The suitable constraint will be applied between the two components, depending on the current selection set. A green arrow may be displayed at the constraint location. You can click on this green arrow to reverse the orientation of the mating components. The component from which the first selection is made will move to snap the component on which the second element is selected. After the components are reoriented, choose the **OK** button from the **Smart Move** dialog box to complete the operation. Figure 11-12 shows two cylindrical surfaces to be selected and Figure 11-13 shows the resulting concentric constraint applied between the two surfaces.

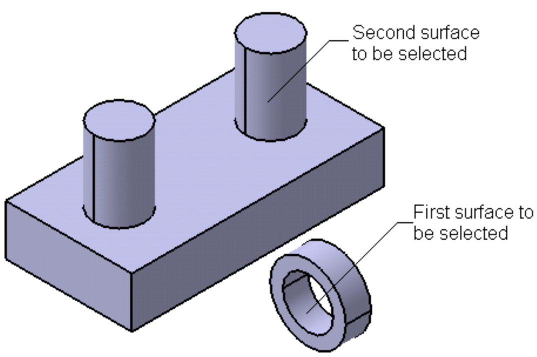


Figure 11-12 *Surfaces to be selected*

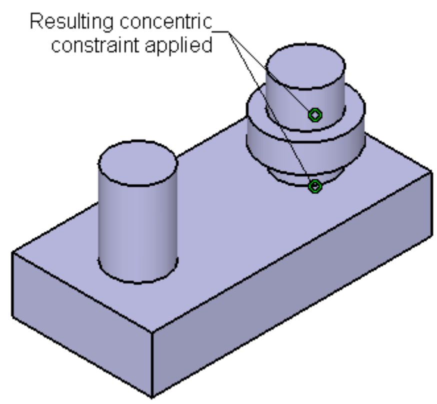


Figure 1-13 *Resulting constraint applied*

The **Smart Move** tool can be invoked along with a viewer, which makes the selection of geometric elements easier. To invoke the viewer, first select the components that need to be moved and then choose the **Smart Move** button from the **Snap** toolbar. This time the **Smart Move** dialog box is displayed with a viewer on top of it. The partial view of the **Smart Move** dialog box is shown in Figure 11-14. The component selected first is displayed in the viewer. The part number of the displayed component is displayed on top of the viewer. Only one component is displayed in the viewer and its geometric element can be easily selected as you can zoom in and rotate the component in it. This is especially helpful when there are a number of components in the geometry area and some of the components are fully or partially placed inside another component. After selecting the geometric element of the first component, choose the **Next component** button from the **Smart Move** dialog box. Note that this button will not be available if you select only one component before invoking the **Smart**

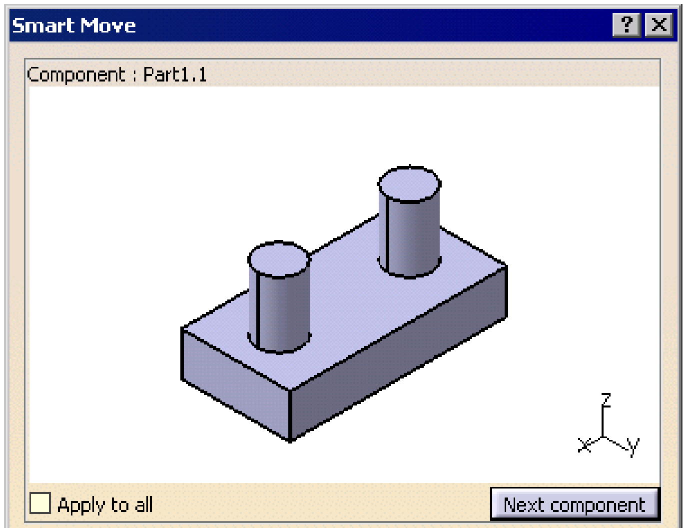


Figure 11-14 *The partial view of the* ***Smart Move*** *dialog box with viewer*

Move dialog box. Now, the other component is displayed in the viewer and you can select its geometric element. Note that while you zoom or rotate the component in the viewer, the actual orientation of the component in the geometry area is not changed. After the selections are made from the viewer, the components are reoriented in the geometric area. Now, choose the **OK** button to close the **Smart Move** dialog box.

Manipulating Components using the Compass

The orientation of the components can also be manipulated using the compass available on the top right corner of the geometry area. To move a component using the compass, you first need to associate it with the component that needs to be moved or rotated. To associate the compass to a part, move the cursor over the red square displayed on the base of the compass. When the selection cursor is replaced by the move cursor represented by four directional arrows, hold down the left mouse button, and drag the compass on the surface of the component to be manipulated. Once the compass is moved, a black dot appears at its original location. To associate the compass with a component, place it on the surface of the component by releasing the left mouse button. The black dot is no longer displayed. To move the component, place the cursor on any of the straight edges of the compass. The edge are highlighted in orange and the cursor is replaced by the hand symbol. Press and hold down the left mouse button and drag the cursor along the highlighted edge to move the component in that direction. After moving it to the desired location, release the left mouse button. Similarly, to rotate the component, place the cursor over any of the circular edges and when the hand symbol is displayed, drag the cursor along that edge.

Once the manipulation is over you need to place the compass back at its original position. To do so, move the cursor on the red square on the base of the compass. Once the move cursor symbol is displayed, drag the cursor anywhere in the geometry area away from all the components in the assembly and release the left mouse button.

Note

When the compass is placed back in its original location, the orientation of the compass remains the same as it was after manipulating the component. This may lead to confusion. To bring the compass back to its default orientation, place the compass over a perfectly horizontal surface. Then place the compass back at its default location.

Applying Constraints

After placing the components in the product file, you need to assemble them. By assembling the components, you will constrain the degree of freedom of the components. As mentioned earlier, the components are assembled using the constraints. Constraints help you to precisely place and position the components with respect to the other components and the surroundings in the assembly. If all degrees of freedom of all components of the assembly are restricted, it is called a fully constrained assembly. Else it is called a partially constrained assembly. If some mechanism needs to be created after assembling the components, some degrees of freedom of the assembly needs to be kept free intentionally, so that movements can be achieved in that direction. Various types of constraints available in CATIA are discussed below.

Fix Component Constraint

Menu:	Insert > Fix
Toolbar:	Constraints > Fix Component

The **Fix Component** constraint is used to fix the location of the selected component in the 3D space. Once the orientation of the component is fixed, its orientation cannot be changed. To invoke this tool, choose the **Fix Component** button from the **Constraints** toolbar; you are prompted to select the component to be fixed. You can select the component from the geometry area or from the **Specification Tree**. Once the component is selected, an anchor symbol  is displayed on the component. Now, other components can be constrained with respect to the fixed component. While doing so, the orientation of the base component will not be altered and the other components will be reoriented to apply the constraints. It is always advisable to fix the base component at its default location so that it can be used as a reference for other components. The **Fix Component** constraint is displayed in the **Specification Tree**. To view the applied constraints, expand the **Constraints** option from the **Specification Tree**.

Coincidence Constraint

Menu:	Insert > Coincidence
Toolbar:	Constraints > Coincidence Constraint

The **Coincidence Constraint** is applied to coincide the central axis of the cylindrical features that are selected from two different components. This option can also be used to apply **Coincident** constraint between edges, points, planes or planar faces. To invoke this tool, choose the **Coincidence Constraint** button from the **Constraints** toolbar; the **Assistant** dialog box is displayed that provides information about the selected constraint. You can select the **Do not prompt in the future** check box, if you do not want to display this dialog box again. Now, move the cursor over a cylindrical surface to display the central axis. When the preview of the central axis is displayed, as shown in Figure 11-15, click the left mouse button to select it. Similarly, select the axis of the second component, as shown in Figure 11-16.

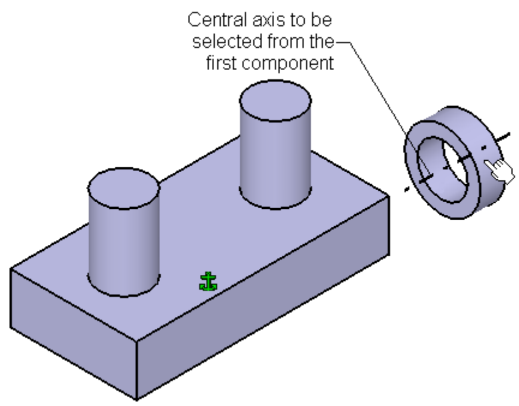


Figure 11-15 *Central axis of the first component to be selected*

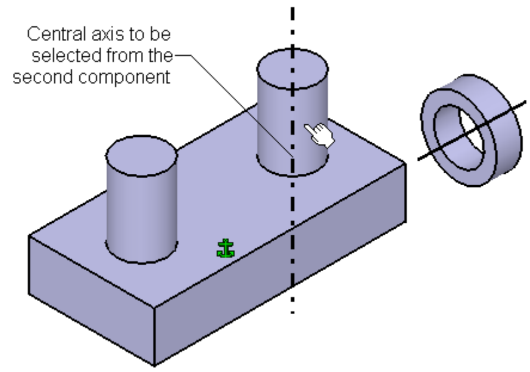


Figure 11-16 *Central axis of the second component to be selected*

Once the two axes are selected, the **Coincidence** constraint is applied between them and the coincidence symbol is displayed, as shown in Figure 11-17. You will notice that although the **Coincidence** constraint is applied between the two components, the components are not assembled with respect to the constraint applied. Instead, a line connecting the two constraint is displayed. To position the components, choose the **Update All** button from the **Tools** toolbar or press CTRL+U from the key board. Now, the components will be placed such that the two selected cylindrical surfaces become concentric, as shown in Figure 11-18. Click once in the geometry area to remove the constraint from the current selection set. The symbol of the constraint is displayed in green on the assembled components.

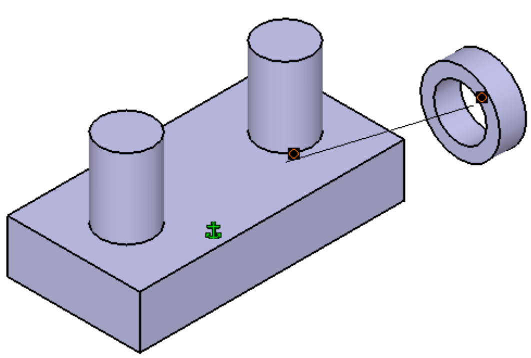


Figure 11-17 *Coincidence constraint applied between two components*

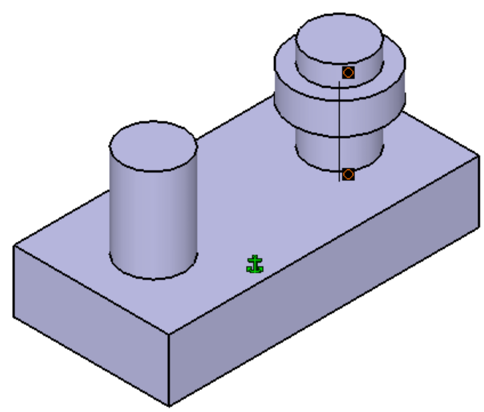
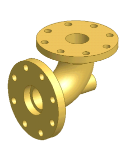


Figure 11-18 *Position of the components after updating*



Tip. *If you select planar faces or planes to apply the coincident constraint, the* ***Constraint Properties*** *dialog box is displayed. Choose the* ***OK*** *button from this dialog box. You will learn more about this dialog box later in this chapter.*

Contact Constraint

Menu:	Insert > Contact
Toolbar:	Constraints > Contact Constraint

The **Contact Constraint** is applied to make a surface to surface contact between two selected elements from two different components. The elements to be selected can be planes, planar faces, cylindrical faces, spherical faces, conic faces, or circular edges. To invoke this tool, choose the **Contact Constraint** button from the **Constraints** toolbar; you are prompted to select the first geometric element of the **Contact** constraint. Select the element from the first component. Next, you are prompted to select the geometric element to place in contact with the first selection. Select the element from the second component. A **Contact** constraint will be applied between the two elements and the component will be placed with respect to the constraints after updating. The **Contact** constraint symbol will be displayed on the assembled components. Figure 11-19 shows the faces to be selected and Figure 11-20 shows the resulting constraint applied to the components.

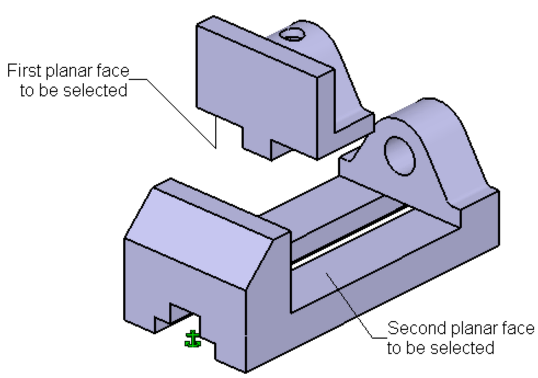


Figure 11-19 *Planar faces to be selected*

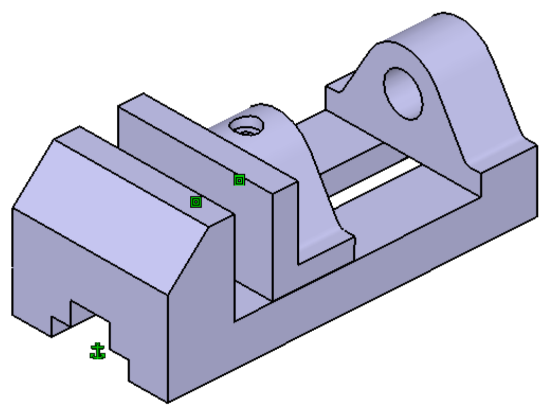


Figure 11-20 *Position of components after constraint is applied and updated*

Note

*If you apply **Contact** constraint between two cylindrical surfaces or a cylindrical surface and a planar face or plane, the **Constraint Properties** dialog box is displayed. You can set the orientation of the constraint using the option available in the **Orientation** drop-down list. After setting the orientation option, choose the **Constraint Properties** dialog box.*

Offset Constraint

Menu:	Insert > Offset
Toolbar:	Constraints > Offset Constraint

The **Offset Constraint** is used to place selected elements at an offset distance from each other. It also makes the two planar faces parallel to each other. To invoke this tool, choose the **Offset Constraint** button from the **Constraints** toolbar. After invoking the tool, you are prompted to select the first geometric element for the **Offset** constraint. Select a planar face, circular face, plane, axis, or a point from the geometry area. You are prompted to select the second geometric element. Select a planar face of another component; the **Constraint Properties** dialog box is displayed, as shown in Figure 11-21.

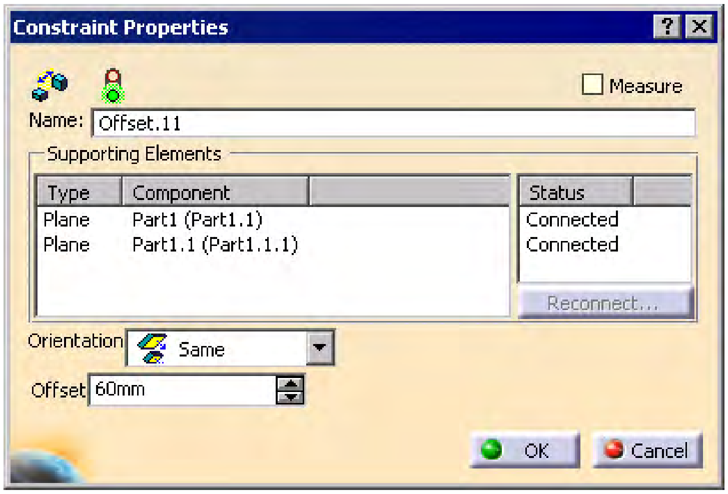


***Figure 11-21** The **Constraint Properties** dialog box*

If you select two planar faces, then two arrows will be displayed on them. The arrows represent the orientation of the plane with respect to each other. The planes can either face in the same direction or opposite direction. To flip the direction or arrows, click on any one of the arrow, or use the options available in the **Orientation** drop-down list. The options available in the dialog box are discussed next.

The **Name** edit box displays the name assigned to this particular constraint. You can also enter a new name of your choice. If the **Measure** check box available on the top left corner of the dialog box is selected, the present distance between the two selected surfaces will be measured from the geometry, and will be assigned the same value as the offset distance.

The **Supporting Elements** area displays the type of geometrical element selected for applying the constraint, the name of the component on which the geometrical elements are present, and the status of the constraint. The status should display **Connected**. If it is displaying **Disconnected**, you need to choose the **Reconnect** button, and select the geometric element again.

The **Orientation** drop-down list has three options namely, **Undefined**, **Same**, and **Opposite**. If the **Undefined** option is selected, then the software automatically orients the component either in the same or in opposite direction, depending on the orientation of the planes. Otherwise, you can select the required option from the drop-down list. In the **Offset** spinner, you need to enter the required offset distance between the planes. After setting all parameters, choose the **OK** button from the **Constraint Properties** dialog box. Select the **Update** button to place the components defined by the constraint. Now, the two selected planes will be placed parallel to each other and will have the specified separation between them. Figure 11-22 shows the faces to be selected. The orientation arrows are shown in the Figure 11-23.

Figure 11-24 shows the orientation of the selected planes after updating. After the **Offset** constraint is applied, the constraint symbol is displayed as the offset distance between the two selected faces.

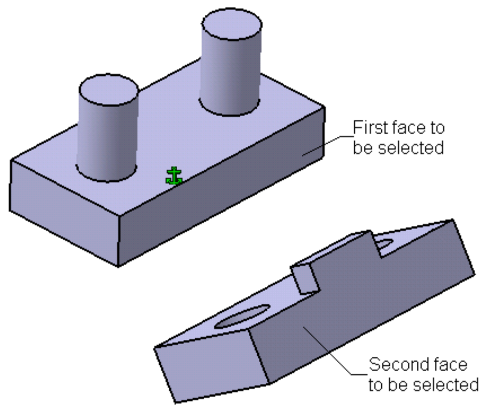


Figure 11-22 *Faces to be selected*

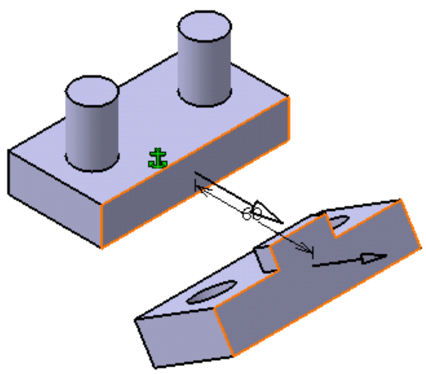


Figure 11-23 *Arrows in the same direction*

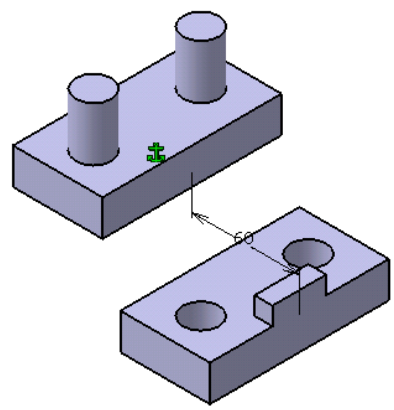


Figure 11-24 *Components after updating*

Angle Constraint

Menu:	Insert > Angle
Toolbar:	Constraints > Angle Constraint

The **Angle Constraint** is used to position two geometric elements at a particular angle with respect to each other. You can also use this tool to make two selected elements parallel or perpendicular to each other. To invoke this tool, choose the **Angle Constraint** button from the constraint toolbar. Now, select the two planar faces from the two different components that you need to place at some angle from each other. You can also select a plane, circular face or an edge as the geometric element. Once the selection is complete, the **Constraint Properties** dialog box is displayed, as shown in Figure 11-25. You will note that in the **Constraint Properties** dialog box, the **Angle** radio button is selected by default. If required, you can select the **Parallelism** or **Perpendicularity** radio buttons. If the **Perpendicularity** radio button is selected, the angle between the faces is automatically

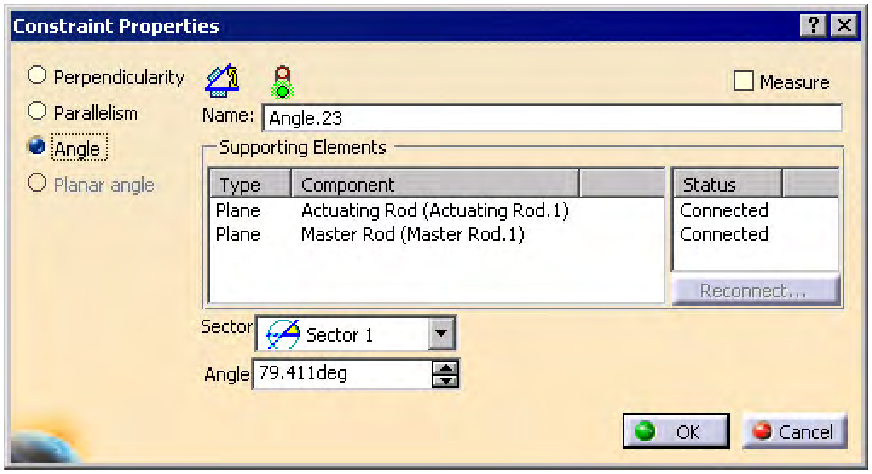


Figure 11-25 ***Constraint Properties*** *dialog box for* ***Angle*** *constraint*

set to 90-degree and the display of the **Angle** spinner is turned off. If the **Parallelism** radio button is selected then the **Orientation** drop-down list is displayed. From this drop-down list, select the required orientation option. If the **Angle** option is selected then the **Angle** spinner and the **Sector** drop-down list is displayed. Before specifying the angle, you need to select the appropriate sector in which it will be applied. Select the sector from the **Sector** drop-down list. The selected sector will also be displayed in the geometry area. After setting all parameters, choose the **OK** button from the **Constraint Properties** dialog box. Figure 11-26 shows the faces to be selected and Figure 11-27 shows the resulting orientation of the faces after applying the **Angle** constraint and updating.

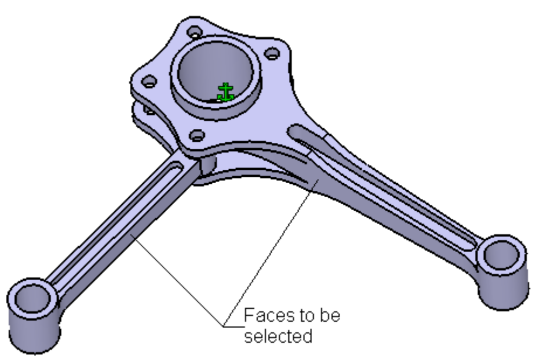


Figure 11-26 *Faces to be selected*

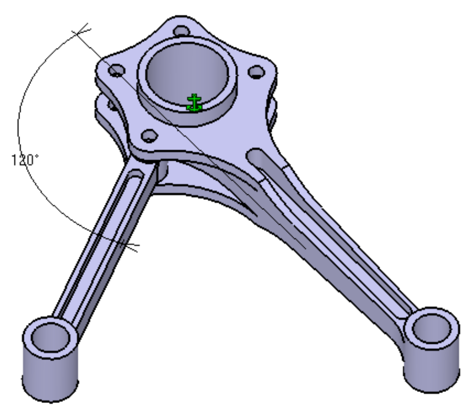


Figure 11-27 *The orientation of the faces after applying the* ***Angle*** *constraint and updating*

After the constraint is applied, the angle value is displayed attached to the selected faces. In case the perpendicularity constraint has been applied, the perpendicular symbol appears between the selected faces. The parallel symbol is displayed between the selected faces, if the parallelism constraint is applied. The name of the resulting constraint is also displayed in the **Specification Tree**.

Fix Together

Menu:	Insert > Fix Together
Toolbar:	Constraints > Fix Together

The **Fix Together** constraint is used to fix the position of the selected components with respect to each other. Once the selected components are fixed together, they can be moved as a single component such that the position of one component with respect to another component remains the same. To invoke this tool, choose the **Fix Together** button from the **Constraints** toolbar; the **Fix Together** dialog box is displayed. Now, select components to be linked. The part number of the selected components are displayed in the **Fix Together** dialog box, as shown in the Figure 11-28. If you want to remove a particular component from the list, click on its part number in the **Components** selection area. Choose the **OK** button from the **Fix Together** dialog box to apply the **Fix Together** constraint.

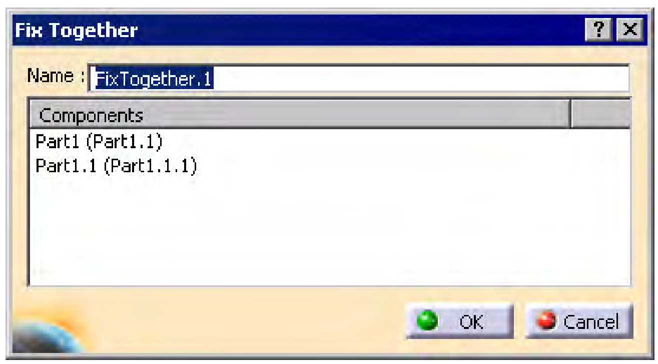
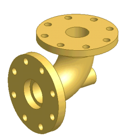


Figure 11-28 *The **Fix Together** dialog box*



Tip. *After applying the **Fix Together** constraint, if you need to move the linked components by using the **Manipulation** tool, select the **With respect to constraints** check box. If you are moving the linked components using the compass, hold down the SHIFT key on the keyboard, and then move the component. Else the components will move separately.*

Quick Constraint

Menu:	Insert > Quick Constraint
Toolbar:	Constraints > Quick Constraint

In CATIA, there is an option in which the software applies the most appropriate constraint to the entities in the current selection set. To apply constraints using this method, choose the **Quick Constraint** button from the **Constraints** toolbar. The possibility of applying constraints depends on the priority of constraints in the **Quick Constraints** priority list. You can invoke this list to set the priority by choosing **Tools** > **Options** from the menu bar. From the **Options** dialog box, choose **Assembly Design** from the **Mechanical Design** on the left of the **Options** dialog box. The **Quick Constraint** priority list is displayed.

Now, select the two geometric elements to be constrained. The software will automatically apply the most appropriate constraint between the selected geometric elements. This saves time in assembling components. However, you need to be careful while selecting the geometric elements because the orientation of the components depends on it.

Reuse Pattern

Menu:	Insert > Reuse Pattern
Toolbar:	Constraints > Reuse Pattern

Sometimes, while assembling the components, you may need to assemble more than one instance of the component in a specified arrangement. Consider a case of a flange coupling where you need to assemble eight instances of nuts and bolts to fasten the coupling. This is very tedious and time-consuming process. Therefore, to reduce the time in the assembly design cycle, CATIA provides you with the **Resume Pattern** tool to insert and constrain multiple copies of a component over an existing pattern. The pattern can be rectangular, circular, or a user pattern.

The first step of using this tool is to insert the first instance of the component and constrain it with any instance of the pattern in the other component. Figure 11-29 shows a Plate with holes created using circular pattern and a Pin that needs to be placed in each instance of the hole. After inserting the Plate and the Pin into the product file, constrain the Pin to any one instance of the holes on the Plate, as shown in Figure 11-30. To assemble the Pin with a hole in the Plate, apply a **Coincident** constraint to the central axis of the Pin and the instance of the hole on the Plate. Next, apply the surface **Contact** constraint between the bottom face of the head of the Pin and the top face of the Plate.

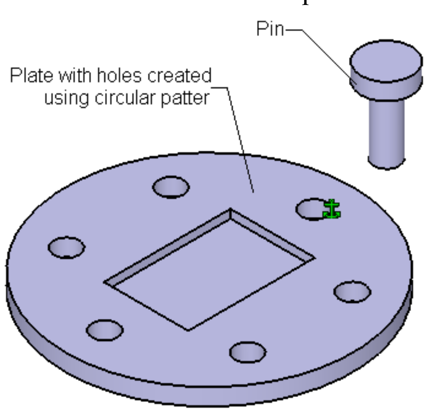


Figure 11-29 *The Pin and the Plate having patterned holes*

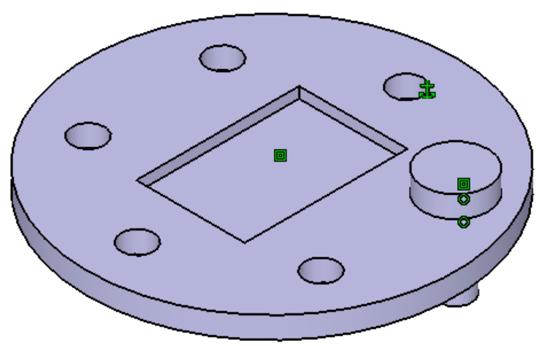


Figure 11-30 *The Pin assembled to one of the instances of the patterned hole*

Next, you need to select the constraint that associates the position of the pin with the pattern instance. Therefore, in this case the **Coincidence** constraint needs to be selected. After selecting the constraint, choose the **Reuse Pattern** button from the **Constraints** toolbar. The preview of Pins assembled with all instances of hole is displayed in the geometry area. The **Instantiation on a pattern** dialog box is displayed, as shown in the Figure 11-31. Note, that the **Pattern** area of the dialog box indicates the name of the pattern, the number of instances

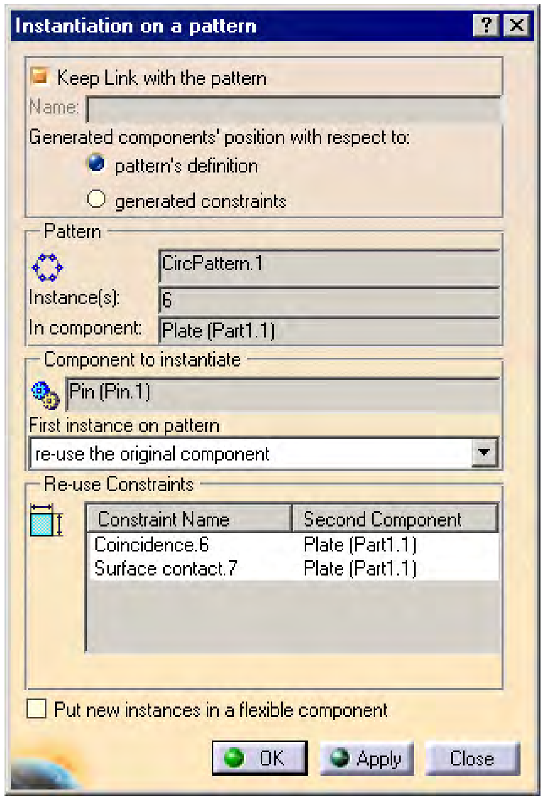


Figure 11-31 *The* ***Instantiation on a pattern*** *dialog box*

to be created, and the name of the component in which the pattern has been created. The name of the component to be repeated is displayed in the **Component to instantiate** area. The **Keep Link with the pattern** check box, at the top of the dialog box, is selected by default. This makes the newly created instances of the pattern associative with the pattern geometry.

The **First instance on pattern** drop-down list has three options. These options are used to define the first instance of the component to be duplicated, and are discussed below.

re-use the original component
This option is used to retain the original component at its location, generate the instances, and populate only the vacant locations.

create a new instance
This option is used to place the new instance of the component on all the patterned instances, including the original instance. As a result, there will be two over lapping instances at the location of the original component.

cut and paste the original component

This option is used to remove the original component from its location and place the new instance of the component at all locations.

By default, the **pattern's definition** radio button is selected in the **Generated component's position with respect to** area of the **Instantiation on a pattern** dialog box. This option facilitates in placing and constraining all instances based on the selected reference pattern. If you select the **generated constraints** radio button, the constraints applied to the parent instance are also applied individually to all pattern instances.

After setting all options, choose the **OK** button from the **Instantiation on a pattern** dialog box. Figure 11-32 shows the resulting assembly after using the **Reuse Pattern** tool. The list of instances created is displayed in the **Specification Tree** shown in Figure 11-33. Note that the part number of every instance remains the same, but the instance number displayed in parenthesis is different. A new entity, called **Assembly Features**, will be created in the **Specification Tree** and the **Reused Circular Pattern.1** is displayed under it.

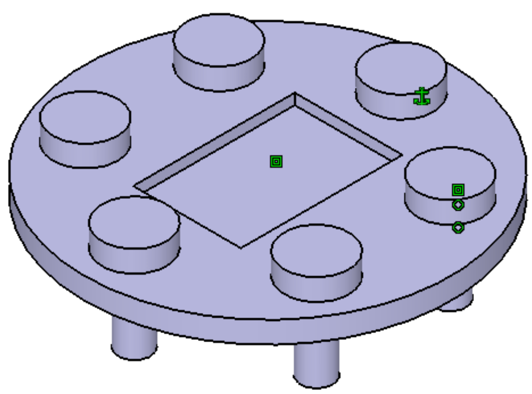


Figure 11-32 The assembly after the selected component is patterned

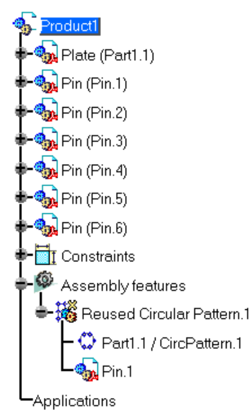


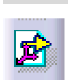
*Figure 11-33 The **Specification Tree** after creating the component pattern*

Note

*You can choose the **Reuse Pattern** button without selecting any constraint. The dialog box will not display any selections. Now, select the pattern instance and the component to instantiate from the **Specification Tree** or from the geometry area.*

Inserting Existing Components With Positioning

Menu:	Insert > Existing Component With Positioning
Toolbar:	Product Structure Tools > Existing Component With Positioning



The **Existing Component With Positioning** tool is used to insert, position, and apply constraints to a component in a single operation and is an enhanced form of the **Insert Existing Component** tool. To invoke this tool, choose the **Existing Component With Positioning** button from the **Product Structure Tools** toolbar. Now, click on the **Product1** in the **Specification Tree**. The **File Selection** dialog box is displayed. Select the part to be inserted and choose the **Open** button. The **Smart Move** dialog box, along with the viewer, will be displayed. Use the **Smart Move** dialog box to position and constraint the newly inserted component. You need to make sure that the **Automatic constraint creation** check box is selected. Else, the component will only be placed and the constraint will not be applied. The applied constraint is displayed in the **Specification Tree**. By using this tool, you can save the assembly creation time.

Note

*By using the **Existing component** and **Existing Component With Positioning** tool you can insert an existing product file into the currently active product file as a subassembly. You can use the individual parts of the subassembly to apply the constraints. The parts and the subassemblies are associative with the parent product file. Therefore, if any modifications are made to the part or the subassembly, they will be visible in the product file.*

CREATING TOP-DOWN ASSEMBLIES

As discussed earlier, in the top-down assembly design approach, all components of the assembly are created inside the **Assembly Design** workbench. To create the components, you need to invoke the **Part Design** workbench within the **Assembly Design** workbench to draw the sketches and then use them for creating features.

Creating Base Part in Top-Down Assembly

Menu:	Insert > New Part
Toolbar:	Product Structure Tools > Part

To start working on the top-down assembly, start a new product file. Click on **Product1** in the **Specification Tree** and select the **Part** button from the **Product Structure Tools** toolbar. A new component named **Part1** is displayed in the **Specification Tree** and a default name is assigned to it. Once a new part is inserted into the product file, the geometry area will display the default planes. These planes belong to the new part and can be used to draw sketches and create features. By default, the origin of these planes is placed over the origin of the assembly coordinate system. .

To change the name of the part, choose **Properties** from the contextual menu invoked by right-clicking on the part name displayed in the **Specification Tree**. The **Properties** dialog box is displayed. The **Product** tab is selected by default in the **Properties** dialog box. Specify the name of the part in the **Instance name** and **Part Number** edit box. After making the changes, choose the **OK** button. The part names are modified in the **Specification Tree**.

To create the model, you need to invoke the **Part Design** workbench. Click on the plus sign (+) displayed on the left of part name in the **Specification Tree** to expand it. Now, double-click on the part name that is displayed inside the expanded branch to expand it further and simultaneously the **Part Design** workbench is invoked. The fully expanded **Specification Tree** is shown in Figure 11-34.

After the part is completed, double-click on the **Product1** in the **Specification Tree** to switch

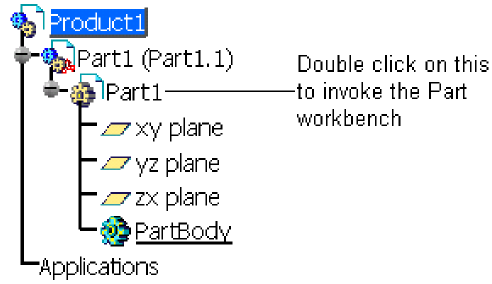


***Figure 11-34** The fully expanded **Specification Tree** after inserting a part in the product file*

back to the **Assembly Design** workbench. Now, you can move and apply constraints to the base component.

Creating Subsequent Components in the Top-down Assembly

After the base component is created inside a product file, you need to create other components of the assembly. The process of creating subsequent components is similar to that of creating the base component. Click on the **Product1** in the **Specification Tree** and then choose the **Part** button. The **New Part: Origin Point** dialog box is displayed, which prompts you whether to define a new origin point for the new part. Select the **No** button to define the origin point of the assembly as the origin part for new part. A new part is created and its name is displayed in the **Specification Tree**. Now, invoke the **Part Design** workbench.

While creating the subsequent components using the top-down approach, you can refer to the geometry of components already created in the assembly to extract the geometry of the sketches of the current component. You can also refer to the geometry of the already created components, while creating the features of the current component. For example, you can sketch a circle and then apply the tangent constrain to the circle with an edge of another part in the assembly or extrude a sketch up to surface that belongs to another part. To retain this kind of relation between the external references, you need to activate the **Keep link with selected object** option. Choose **Tools > Options** from the menu bar to invoke the **Options** dialog box. Select the **Infrastructure** option from the left of the dialog box to expand the branch and select the **Part Infrastructure** option. Now, select the **General** tab if it is not selected by default, to display the entries under it. In the **External References** area, select the **Keep Link with selected object** check box. Now, choose the **OK** button from the **Options** dialog box. After activating this option, the relations with external references will be maintained.

You can select a planar surface of the base component or the default planes as the sketching plane to draw sketches for creating the features of the new part. The edges of the base feature can be used to constraint the sketch and its surfaces can work as limits, while creating the extrude feature. Once the part creation is complete, double-click on **Product1** to switch back to the **Assembly Design** workbench.

Similarly, you can create more parts in the current product file. Note that if you reorient a part that has some relation with external references, then the product file needs to be updated to re-establish the relation. Figure 11-35 shows two parts created inside the assembly file. The cylinder is extruded up to the surface that belongs to the base part. Figure 11-36 shows the **Up to surface** relation still maintained even after moving the cylinder base along Z-direction.

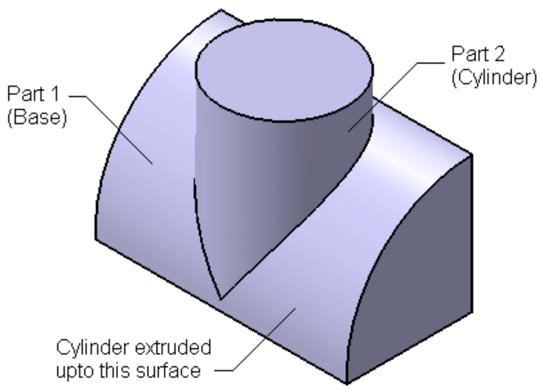


***Figure 11-35** Two different parts created in a product file and the cylinder extruded up to surface*

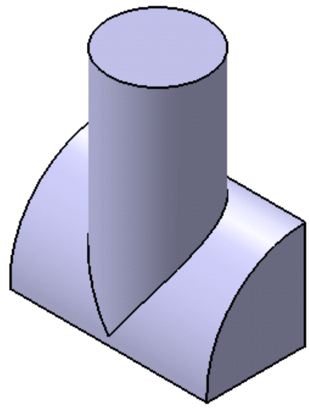


***Figure 11-36** Up to surface relation maintained even after moving the base downward*

Creating Subassemblies in Top-down Assembly

While creating complicated assemblies, you may need to have subassemblies inside an assembly. While working in the top-down approach, you can directly create a subassembly inside the product file. In CATIA V5, there are two types subassemblies that can be created in the **Assembly Design** workbench: **Product** and **Component**. Both these subassemblies are discussed next.

Product Subassemblies

Menu:	Insert > New Product
Toolbar:	Product Structure Tools > Product

If you create a subassembly using the **Product** tool, the resulting subassembly and the parts created within it are saved as a separate product and part files within the folder in which the main assembly file is saved. This gives the benefit of managing the subassembly or the part files individually. You can also open these files separately and work on the design changes, which results in greater flexibility. Once the modifications are made and the files are saved, the changes are automatically reflected in the main assembly file. To create a subassembly inside the product file, select **Product1** from the **Specification Tree** and choose the **Product** button from the **Product Structures Tools** toolbar. The new subassembly named **Product2** is displayed in the **Specification Tree**. The name of this subassembly can also be modified using the method similar to that discussed while renaming the parts. Because the newly created subassembly is already highlighted, choose the **Part** button to create a new part inside it. In this way, you can create more subassemblies inside the main assembly or inside the subassembly itself.

Note

*You can activate the subassemblies by double-clicking on the name of the subassembly in the **Specification Tree**. To switch back to the main assembly, double-click on the name of the main assembly in the **Specification Tree**.*

Component Subassemblies

Menu:	Insert > New Component
Toolbar:	Product Structure Tools > Component

If you create a subassembly using the **Component** tool, the resulting subassembly becomes an integral part of the main assembly file and will not be saved as separate product file. However, the individual parts are saved as separate part files. If you need to make any modification in the subassembly, you need to access it from the main assembly because the subassembly file is not saved separately. To create this type of subassembly inside the product file, select **Product1** from the **Specification Tree** and choose the **Component** button from the **Product Structures Tools** toolbar. The newly created subassembly named **Product3** is displayed in the **Specification Tree**, as shown in the Figure 11-37. Now, you can rename this file, if required, and create parts inside it.

***Figure 11-37** The **Specification Tree** having a product and a component within an assembly file.*

Note

*Save the product file once all parts and subassemblies are created inside it. There is no need to save the parts and subassemblies separately. They will be automatically saved as separate files inside the folder in which the main product file will be saved. The file names will be the same as that given to parts and the subassemblies in the **Properties** dialog box.*

EDITING ASSEMBLIES

After creating the assembly, you may need to modify the parts, subassembly, or the applied constraints. You may also need to replace the existing part with another part. These editing operations are discussed below.

Deleting Components

While working in the **Assembly Design** workbench, you may need to delete some of the constituent parts and subassemblies. To delete a part or a subassembly, right-click on its name in the **Specification Tree** and choose the **Delete** option from the contextual menu. You can also delete a part or subassembly by selecting it from the **Specification Tree** and pressing the DELETE key on the keyboard. If there are some relations associated with the selected part, the **Delete** dialog box will be displayed, as shown in the Figure 11-38.

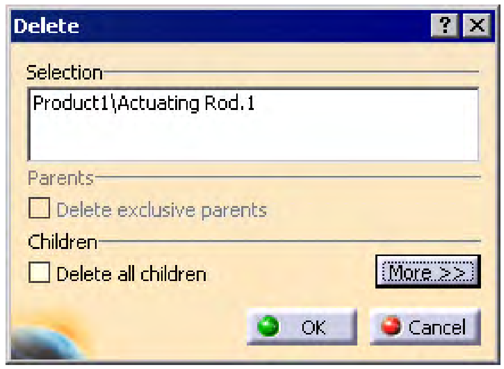


Figure 11-38 *The **Delete** dialog box*

The **Selection** area of the **Delete** dialog box will display the names of the parts to be deleted and the name of the assembly to which the parts belong. Choose the **OK** button to complete the deletion process. The associated constraints now become inconsistent and a yellow error symbol is displayed in the **Specification Tree**. These constraints have to be deleted separately.

If you select the **Delete all children** check box in the **Delete** dialog box, all relations associated with the selected part will be deleted along with it. Similarly, way the subassemblies can also be deleted from the main assembly.

Note
In a Product file with more than one subassembly, you cannot delete the currently activated subassembly. To do so, first activate any other subassembly and then delete the subassembly that was active earlier.

Replacing Components

Menu:	Edit > Components > Replace Component
Toolbar:	Product Structure Tools > Replace Component

In CATIA you can replace an existing component with another component inside an assembly. If the new component being placed has the basic geometry which is the same as the original component, then it will be placed exactly at the location where the original component was placed. Otherwise the replaced component will be placed arbitrarily in space with no association with the location where the earlier component was present. If the component to be replaced has more than one instance and you replace any one of the instance, then all the instances of the component are replaced.

To replace a component, select it from the **Specification Tree** and choose the **Replace** button from the **Product Structure Tools** toolbar. The **File Selection** dialog box will be displayed. Select the component and choose the **OK** button from the **File Selection** dialog box. The **Impacts On Replace** dialog box will be displayed. Choose the **OK** button from this dialog box to replace the existing component with the selected component. Note that the constraints that were earlier applied on the previous component now become inconsistent. You can either reattach these constraints, or delete them and apply new constraints. The process of reattaching a constraint is discussed later in this chapter. Figures 11-39 and 11-40 show the example of replaced components. Note that you cannot undo a **Replace Component** operation. Therefore, you need to be careful while performing this operation. If there is a need to undo the **Replace Component** operation, use the same command again to replace the new component with the previous component.

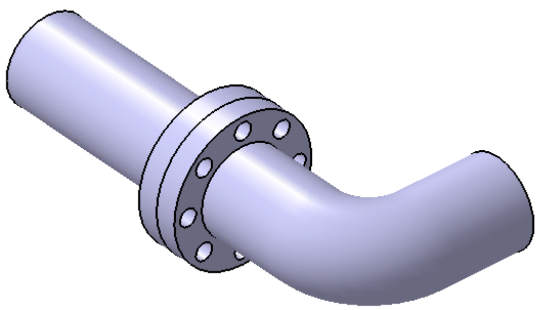


Figure 11-39 *The original component*

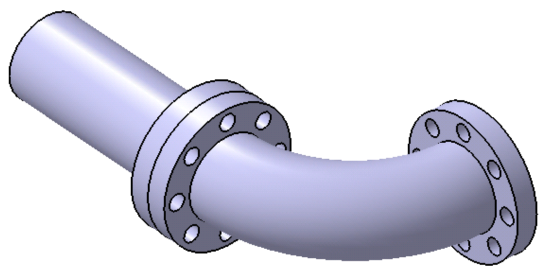


Figure 11-40 *The replaced component*

Editing Components Inside an Assembly

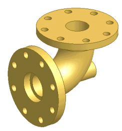
You can also edit the features and modify the sketches of the parts of assembly within the **Assembly Design** workbench. For this you need to activate it by invoking the **Part Design** workbench. To invoke the **Part Design** workbench for editing a part, click on the plus sign (+) displayed on the left of the part name to expand it in the **Specification Tree**. Now, double click on the part name that is displayed inside the expanded branch to expand it further and simultaneously invoke the **Part Design** workbench. In the **Part Design** workbench, you can make the required modifications to the features and sketches of the part. After you have made all changes, double-click on the Product name to return back to the **Assembly Design** workbench. Note that in the **Part Design** workbench, all parts of the assembly are visible, but changes are made only to part that is active. Similarly, you can also edit the components in the subassemblies.

Editing Subassemblies Inside an Assembly

You can also edit subassemblies that are placed inside the main assembly. To edit a subassembly, double-click on the name of the subassembly in the **Specification Tree**; the subassembly will be activated. You can insert or remove the components from the subassembly, or you can edit the constraints applied to the components of the subassembly. After making the necessary changes, double-click on the main assembly to switch back to the main assembly.

Note

The changes made to a part or subassembly inside the product file are also reflected in their respective part and product files. Therefore, the changes will take place whereever these parts and subassemblies have been used.



Tip. *If you double-click on the sketch of the feature of a component in the* ***Specification Tree****, the* ***Sketcher*** *workbench is invoked directly. Modify the sketch and exit the* ***Sketcher*** *workbench. You will be switched to the* ***Part Design*** *Workbench. To switch back to the* ***Assembly Design*** *workbench, double-click on* ***Product*** *in the* ***Specification Tree****.*

Editing the Assembly Constraints

In an assembly, the constituent parts are positioned at their respective locations using the constraints. Many a times, you need to replace the existing constraint with another constraint or to change the entities to which the constraints are applied. The methods to modify the constraints are discussed below.

Editing the Constraint Definition

All assembly constraints need to be associated with entities of two different components. These entities can be planes, surfaces, axes, edges, and so on. In the **Offset** and **Angle** constraints, some numeric values, which define the offset distance and the rotation angle, are also specified. These associated entities and the numerical values can be modified by editing the definition of the constraint. The definition of a constraint can be accessed by double-clicking on its name in the **Specification Tree** or its symbol from the graphics area. The **Constraint Definition** dialog box is displayed. Choose the **More** button from this dialog box to expand it, as shown in the Figure 11-41.

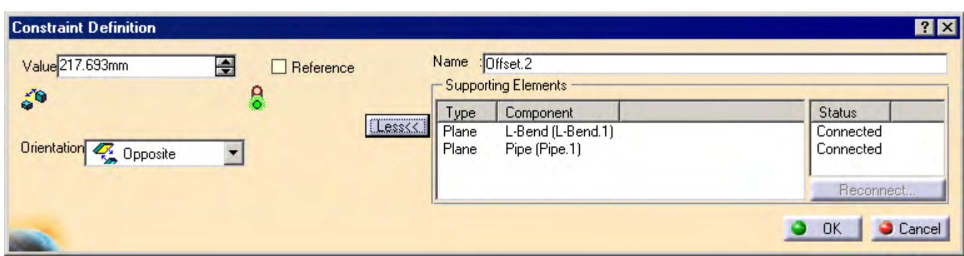


Figure 11-41 *The* ***Constraint Definition*** *dialog box*

The **Value** spinner is available on the left of this dialog box. You can modify the displayed value to change the offset distance. The appropriate option from the orientation drop-down list can be selected to change the position of the faces between which the **Offset** constraints have been created. In the expanded region of the dialog box, the name of the constraint is displayed. If required, you can enter a new name for the constraint and the same will be displayed in the **Specification Tree**.

The **Supporting Elements** area displays the type of entities and their corresponding components between which the constraint is applied. Ideally, the **Status** area should display **Connected**. This means that the association of the constraint with the entity is present. If the status shows **Disconnected**, the association has been broken and hence the constraint has become inconsistent. In this case, the constraint has to be reconnected with the proper entity. The **Reconnect** button is used to select the entity on which the constraint will be connected. This button can also be used to replace an existing element with another element, even if the status is connected. By default, the **Reconnect** button is disabled. To enable it, select the reference that you need to replace from the **Supporting Element** display area. Next, select the element from the geometry area; the new selection will be displayed in the dialog box. Choose the **OK** button to complete the constraint editing operation. You need to update the model to incorporate the changes. Figure 11-42 shows the **Offset** constraint applied between faces of two components. Figure 11-43 shows the position of components after the **Offset** constraint is reconnected to another surface, and the offset distance is modified.

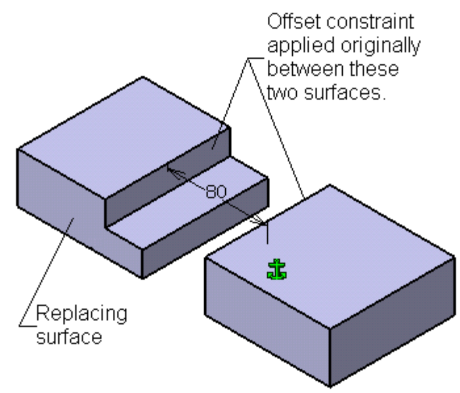


*Figure 11-42 The associated and replacing surface for the **Offset** constraint*

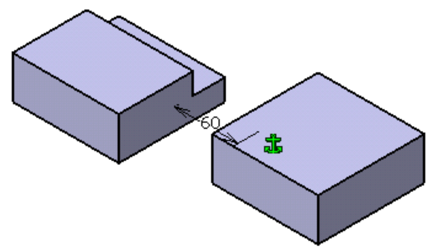
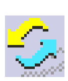


Figure 11-43 Components after editing the offset constraint and updating it

Every type of constraint can be modified in a similar manner by editing its definition.

Replacing a Constraint

Toolbar:	Constraints > Change Constraint



To replace an existing constraint with another constraint, select it from the **Specification Tree** or from the geometry area. Choose the **Change Constraint** button from the **Constraints** toolbar to display the **Change Type** dialog box. This dialog box displays all the possible constraints that can be used to replace the selected constraint. Select the appropriate constraint and choose the **OK** button from the **Change Type** dialog box. The previously applied constraint will now be replaced by the new constraint. You can change the definition of the replaced constraint as per requirement. After making the changes, update the assembly to bring the newly applied constraint into effect. Figures 11-44 and 11-45 show the replacement of the **Contact** constraint that is applied between the flange of the pipes by the **Offset** constraint after modifying its definition and entering some offset distance.

Simplifying the Assembly

While working on large assemblies consisting of a large number of parts and subassemblies,

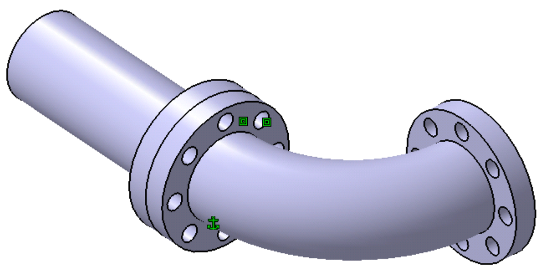


*Figure 11-44 Contact constraint to be replaced by the **Offset** constraint*

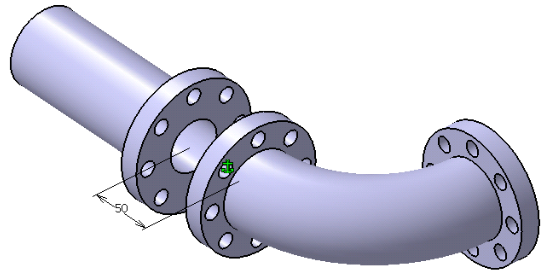


*Figure 11-45 Components after applying the **Offset** constraint*

you may face a difficulty while managing the components of the assembly. Therefore, it is recommended to hide some of the parts to improve the visibility of other parts. While working with a large assembly, you may experience some difficulty in updating the assembly because all the parts of the assembly are rebuilt during updation. Therefore, it is recommended to suppress the parts that are not required at that particular stage of design cycle. This reduces the regeneration time of the assembly. Hiding and suppressing the components are discussed next.

Hiding a Component

Menu:	View> Hide/Show > Hide/Show
Toolbar:	View> Hide/Show

The **Hide/Show** tool is used turn off the display of the selected component of the assembly. But the component exists in the hierarchy of the assembly and participate in assembly updation. The symbol of the hidden component is displayed in light grey color in the **Specification Tree**. The **Hide/Show** tool can also be accessed from the contextual menu by right clicking on the name of the component to hide.

Deactivating a Component

Deactivating the component, removes it temporarily from the assembly. This substantially decreases the regeneration time of the model. To deactivate a part, invoke the contextual menu by right-clicking on the name of the component in the **Specification Tree**. Now, place the cursor over the instance name to open the cascading menu and choose the **Activate/Deactivate Component** option. A red symbol () is displayed on the left of the name of the deactivated component in the **Specification Tree**. Follow the same procedure to unsuppress the component and make it active. Note that once a component is suppressed the constraints associated to it become inconsistent and a yellow symbol is displayed against them in the **Specification Tree**. These constraints are no longer displayed in the geometry area.

Interference Detection

Menu:	Analyze > Clash
Toolbar:	Space Analysis > Clash

It is recommended to check the interference and clearance between the components of the assembly to make sure that the components are not interfering with each other and the right type of fit is maintained between the mating parts. The interference is detected using the **Clash** tool, which is invoked by choosing the **Clash** button from the **Space Analysis** toolbar. The **Check Clash** dialog box is displayed, as shown in Figure 11-46.

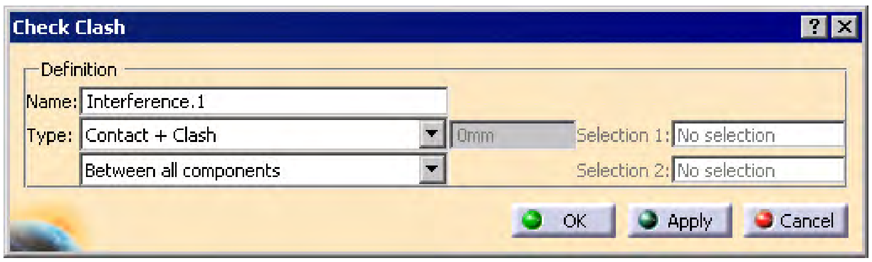


Figure 11-46 *The* ***Check Clash*** *dialog box*

There are two drop-down lists available in the **Type** area. From the upper drop-down list, select the type of analysis that you need to perform. From the lower drop-down list, you can select the option for defining the selection of components between which the interference will be calculated. Choose the **Apply** button to perform the analysis. The **Check Clash** dialog box will expand and display the result of the analysis. The result display area serially lists the name of components and the type of interference between them, which can be clearance, contact, or clash. Selecting a particular interference from the list area will display the corresponding interference value and the **Preview** window will be displayed to show the location of the interference. Figure 11-47 shows the **Preview** window displaying the location and value of interference between the two components.

There are three tabs available on top of the **List** area. They are used to change the display format of the list. You can also use the drop-down lists of the **Filter list** area to display specific type of interferences in the list area. After checking the required interferences, choose the **OK** button to close the **Check Clash** dialog box.

Sectioning an Assembly

Menu:	Analyze > Sectioning
Toolbar:	Space Analysis > Sectioning

Sometimes it is required to section an assembly model to view its cross-section. This is required to analyze the clearance and interference of internal parts, which may not be visible from outside. To section an assembly model, choose the **Sectioning** button from the **Space Analysis** toolbar; the **Sectioning Definition** dialog box is displayed, as shown in the Figure 11-48.

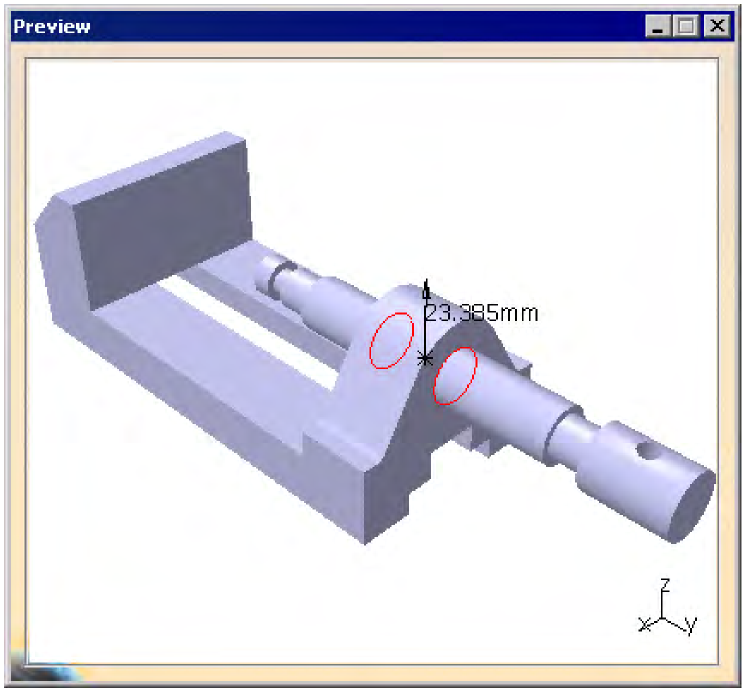


Figure 11-47 The ***Preview*** *window*

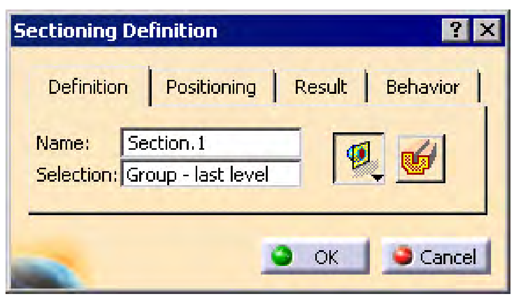


Figure 11-48 The ***Sectioning Definition*** *dialog box*

The 2D representation of the section view is represented in another window, which is tiled vertically with the assembly window. The assembly window displays the sectioning plane. By default, the sectioning plane is coincident to the YZ plane. To change the position of the sectioning plane, place the cursor over it to display a bidirectional arrow. This arrow will be pointing normal to the sectioning plane. After the blue arrow is displayed, drag the plane to reposition it. You can also use the red compass to rotate the sectioning plane in the same way as it is used for reorienting parts. The size of the sectioning plane can be modified by dragging its edges. You can also position the sectioning plane using the options available in the **Positioning** tab of the **Sectioning Definition** dialog box. Figures 11-49 and 11-50 show the 2D sectional view generated by sectioning the model and the plane used to create the section, respectively.

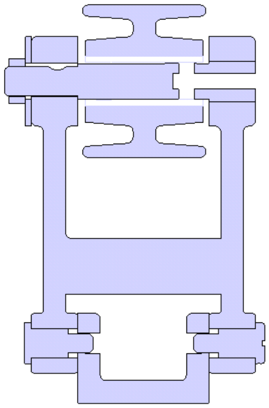


Figure 11-49 *The 2D section view of the complete assembly*

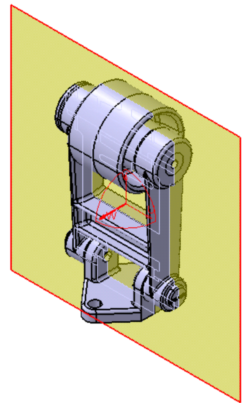


Figure 11-50 *The sectioning plane*

The **Definition** tab is selected by default when the **Sectioning Definition** dialog box is invoked. Under this tab, there are two buttons. The button on the left is used to select the type of sectioning required. Choose the down arrow displayed on the right of this button to open the flyout. This flyout displays the three sectioning options. The section can be created using a plane or a slice of the model can be generated by sectioning it between two planes, or a portion of the model can be sectioned out by placing it in a bounding box. The next is the **Volume Cut** button. If this button is selected, the solid section view of the assembly is displayed in the assembly model window. If you need to view only the 2D section view of some selected parts, then click in the **Selection** selection area and select the required parts from the **Specification Tree**. Selecting the same part again will remove it from the current selection set. To view the section of the whole assembly again, remove all parts from the current selection set and select the product, if it is not available in the current selection set. After viewing the required sectional view, choose the **OK** button to exit the **Sectioning Definition** dialog box. The cross-section that is generated after sectioning the model is now displayed in the geometry area and the corresponding section name is displayed in the **Specification Tree** inside the **Applications** heading. Now, close the window with the section view and maximize the product file.

Hide the section, if you do not want it to be displayed.

Exploding an Assembly

Menu:	Edit > Move > Explode in assembly design
Toolbar:	Move > Explode

Generally, an assembly model consists of a large number of parts. Some of the parts are assembled inside the other parts. Therefore, these parts are not visible and the user is unable to see all components present in the assembly. To resolve this problem, the assembly is exploded such that all components are moved from their original position to a location where they are clearly visible. To explode an assembly, choose the **Explode** button from the **Move** toolbar; the **Explode** dialog box will be displayed, as shown in the Figure 11-51.

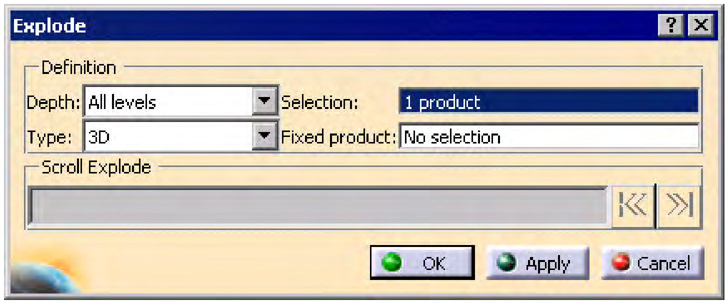


Figure 11-51 *The* ***Explode*** *dialog box*

Make sure **Product1** is activated in the **Specification Tree** before invoking the **Explode** tool. If there are multiple assemblies in the product file, you can select any one of them to explode.

You can set the parameters for exploding the assembly in the **Explode** dialog box. The options of this dialog box are discussed below:

The **Depth** drop-down list is provided with two options. If the **First level** is selected from this drop-down list, the parts of the subassembly are not exploded. Rather, the subassembly will be treated as a single component. The components of the subassembly will be exploded only if the **All level** option is selected from the **Depth** drop-down list. The **Selection** selection area displays the number of products that have been selected for explosion. The **Fixed Product** selection area is used to select a part of the assembly that needs to be fixed, while exploding the assembly. All other parts will be moved with respect to it. In the **Type** drop-down list, there are three options. By default, the **3D** option is selected, which enables the assembly model to explode in the **3D** space and the components are placed arbitrarily in it. The assembled view of the Belt Tightener assembly is shown in Figure 11-52 and Figure 11-53 shows the position of components, after the assembly is exploded using the **3D** option.

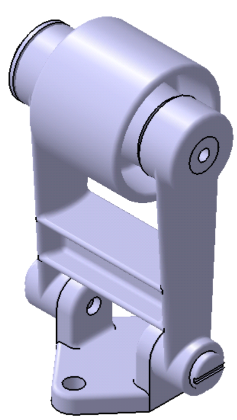


Figure 11-52 *The Belt Tightener in the assembled state*

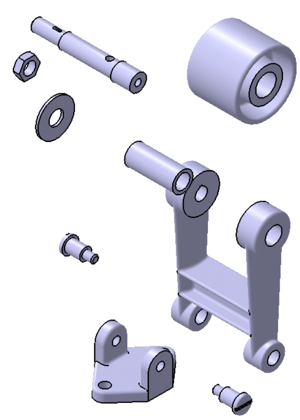


Figure 11-53 *Overlapping components in 3D explosion of the assembly*

If you select the **2D** option from the **Type** drop-down list, the components are exploded and placed parallel to the viewing plane. Figure 11-54 shows the Belt Tightener assembly exploded using the **2D** option, with the front plane parallel to screen. Figure 11-55 shows the top view of the same exploded assembly.

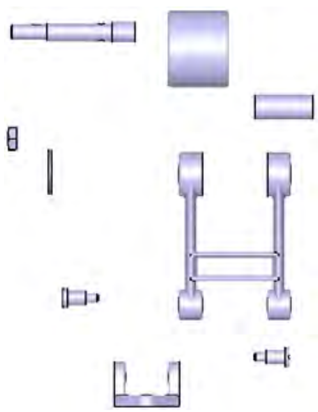


Figure 11-54 Front view of the exploded Belt Tightener assembly exploded using the ***2D*** *option*

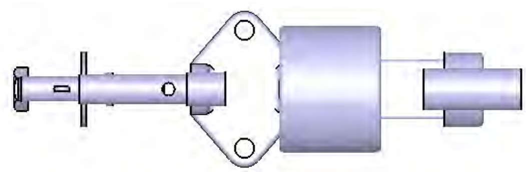


Figure 11-55 Top view of the exploded Belt tightener assembly

The third option provided in the **Type** drop-down list is the **Constrained**. This option is selected to explode the assembly in such a way that some of the constraints applied to the parts are maintained. This results in a more organized explosion, as shown in the Figure 11-56.

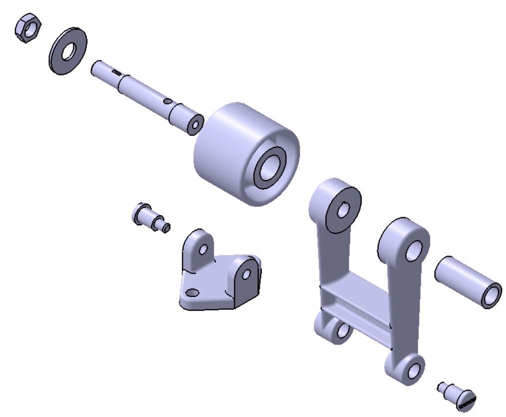


Figure 11-56 Figure showing the exploded assembly with Constrained selected as the type

After all selections are made in the **Explode** dialog box, choose the **Apply** button; the assembly will be exploded and the **Information Box** will be displayed. This box intimates you that the exploded parts can now be moved using the 3D compass. Move the components to arrange them in a more realistic manner, if required. Choose the **OK** button to close this **Information Box**. You can clear the **Show this message next time** check box to prevent the **Information Box** from appearing every time you explode a model. Finally, choose the **OK** button from the **Explode** dialog box to close it and then choose **Yes** from the **Information Box**. The exploded assembly is shown in the geometry area.

To switch back to the assembled view, choose the **Update** button from the **Tools** toolbar.

TUTORIALS

Tutorial 1

In this tutorial you will create all the components of the Blower assembly and then assemble them together. The Blower assembly is shown in Figure 11-57. After creating the assembly, you will generate the exploded view. The exploded view of the Blower assembly is shown in Figure 11-58. The dimensions of all components are given in Figures 11-59 through 11-64.

(Expected time: 2.5 hrs)

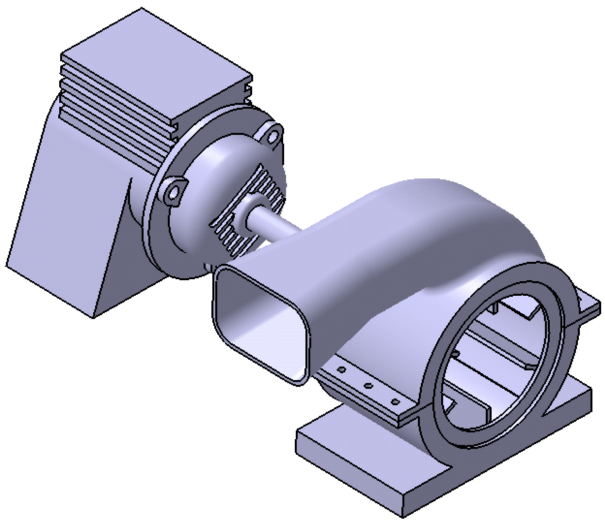


Figure 11-57 *The Blower assembly*

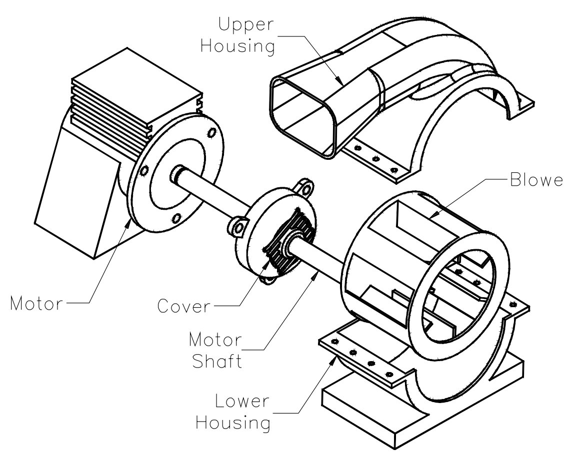


Figure 11-58 *Exploded view of the Blower assembly*

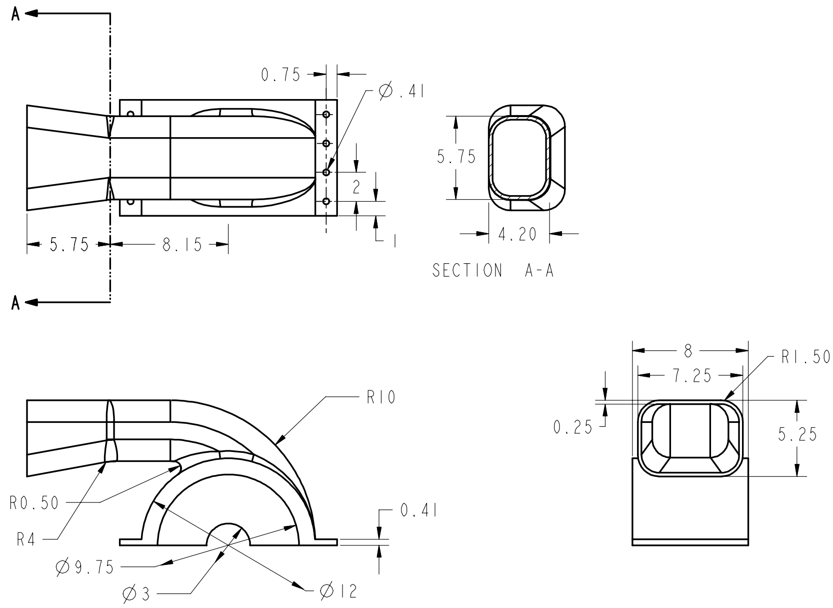


Figure 11-59 *Views and dimensions of the Upper Housing*

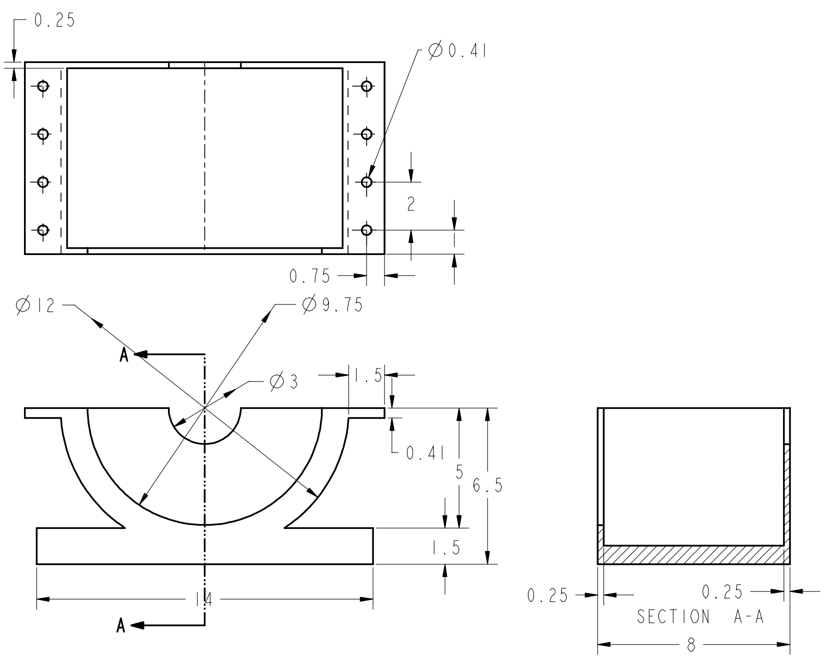


Figure 11-60 *Views and dimensions of the Lower Housing*

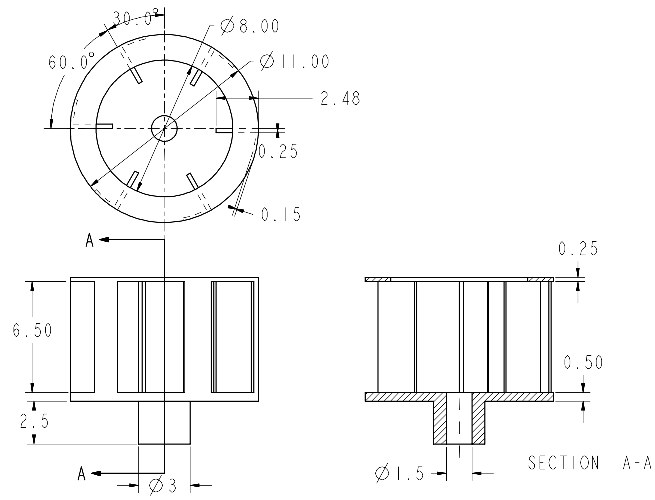


Figure 11-61 *Views and dimensions of the Blower*

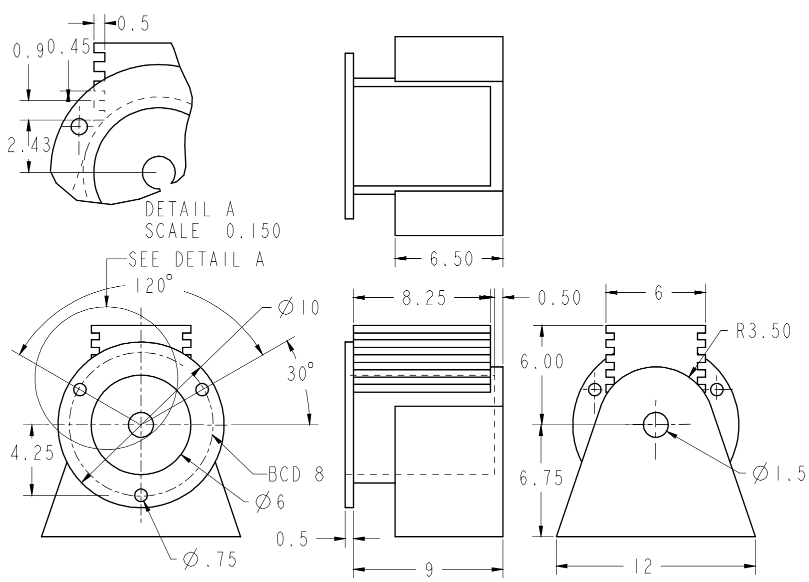


Figure 11-62 *Views and dimensions of the Motor*

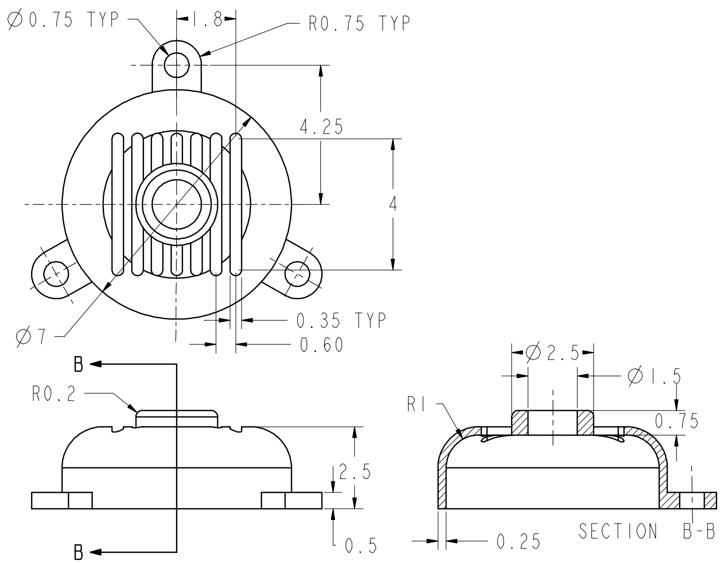


Figure 11-63 *Views and dimensions of the Cover*

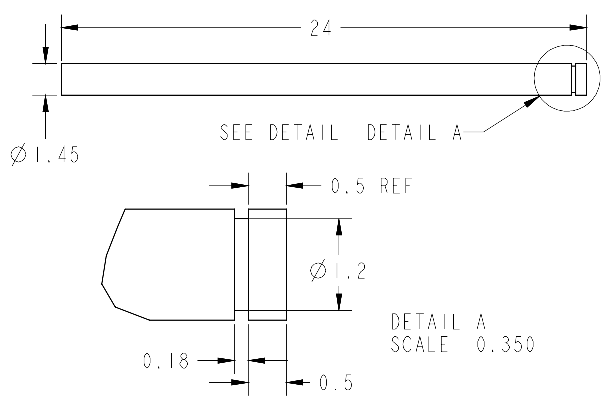


Figure 11-64 *Views and dimensions of the Motor Shaft*

The following steps are required to complete this tutorial:

a. Create all components of the assembly as separate part files in the **Part Design** workbench.
b. Start a new file in the **Assembly Design** workbench.
c. Insert the Lower Housing into the assembly as the base component, set its orientation, and apply the **Fix** constraint to it at its default location, refer to Figures 11-65 through 11-67.
d. Insert the Upper Housing into the assembly and place it over the lower housing by applying proper constraints, refer to Figures 11-68 through 11-70.
e. Hide the Upper Housing. Insert and place the blower inside the lower Housing.
f. Now, insert and constrain the Motor, the Motor Shaft, and the Cover refer to Figures 11-71 through 11-77.
g. Turn on the display of the Upper Housing, refer to Figure 11-78.
h. Create the exploded state of the assembly, refer to Figure 11-79.
i. Save the assembly file.

Before you start creating components for this tutorial, create *\My Documents\CATIA\c11\Blower Assembly* folder. You will save the parts of the blower assembly in this folder. Note that you should change the Part Number of every component before saving it. The process of changing the Part Number of a components has already been discussed earlier in this chapter.

Creating Components of the Assembly

The Blower assembly will be created using the bottom-up approach. As mentioned earlier, in bottom-up assemblies, all parts are first created as individual part files and then inserted in the assembly file.

1. Create all parts of the assembly and save them as separate part files in the above mentioned folder.

2. Close all the part files, if they are open.

Starting a New File in the Assembly Workbench

All components that you have created above need to be assembled in an assembly file. The assembly file has a file extension **.CATProduct*. You need to start a new file in the **Assembly Design** workbench to assemble the parts.

1. Choose the **New** button from the **Standard** toolbar. The **New** dialog box is displayed.

2. Choose the **Product** option from the **List of Types** list box.

3. Choose the **OK** button to start a new product file. A new file is started in the **Assembly Design** workbench, and the **Product1** is displayed on the top of the **Specification Tree**.

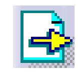
Note

*If you start a new session of CATIA, an assembly file is started automatically. Therefore, if you start another file, it will be named **Product2**.*

Inserting the First Component and Fixing it

After the new product file is started, you can insert the base component into the assembly. In this case, the Lower Housing is the base component. After inserting the Lower Housing, you need to set the orientation of the Lower Housing and then fix its location.

1. Choose the **Existing Component** button from the **Product Structure Tools** toolbar.



2. Select the **Product1** from the **Specification Tree**; the **File Selection** dialog box is displayed. From this dialog box browse the location of the file of Lower Housing and open it.

   The Lower Housing is displayed in the geometry area and its name is shown in the **Specification Tree**. The current orientation of the isometric view is not the same as that required in the assembly. Therefore, you need to set the orientation of the model. The orientation of the model will be set by using the **Snap** tool.

3. Choose the **Snap** button from the **Move** toolbar. Select the first element and the second element, as shown in Figure 11-65. The orientation of Lower Housing is changed and a flip arrow is displayed on it.

4. Click anywhere in the geometry area to exit the **Snap** tool. Set the orientation of the view of the assembly to Isometric. The Lower Housing is placed in the correct orientation, as shown in Figure 11-66.

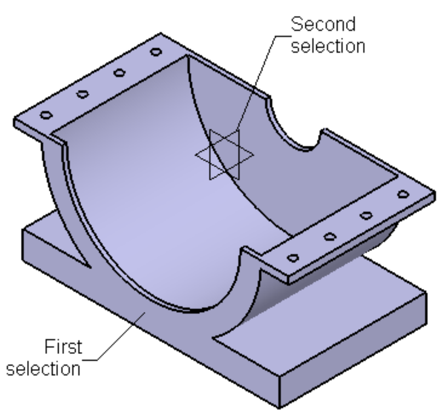


Figure 11-65 *First and second elements to be selected*

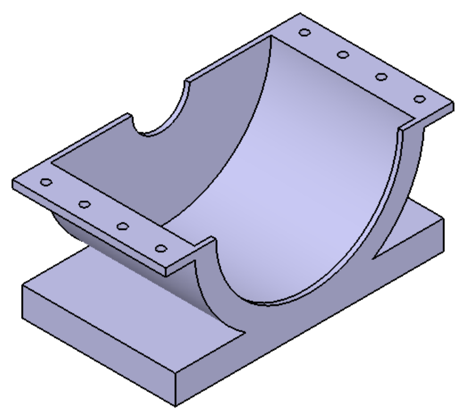


Figure 11-66 *Lower Housing after modifying its orientation*

Next, you need to apply the **Fix** constraint to lock it position.

5. Choose the **Fix Component** button from the **Constraints** toolbar and select the Lower Housing from the geometry area or from the **Specification Tree**.

   The symbol of the **Fix** constraint is displayed on the Lower Housing in the geometry area. Figure 11-67 shows the Lower Housing after fixing it at its location.

Inserting the Upper Housing and Constraining it

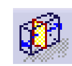
The sequence in which the parts should be inserted into the assembly depends on the user. In this case, the Upper Housing will be the second component to be inserted into the Blower assembly.

1. Insert the Upper Housing in the similar way as discussed earlier. Note that the Part Number of all components were modified before saving the part files. Therefore, the **Part Number Conflicts** dialog box will not be displayed for any component.

   The Upper housing is placed at its default location, as shown in the Figure 11-68. You need to apply constraints to place it properly over the Lower Housing.

   The first constraint that will be applied to the Upper Housing is the **Contact** constraint. This constraint will be applied between the upper face of the Lower Housing and the lower face of the Upper Housing.

2. Choose the **Contact Constraint** button from the **Constraints** toolbar. Select the two faces shown in Figure 11-68. You need to rotate the view of the assembly to select the surface that is not visible in the current display.



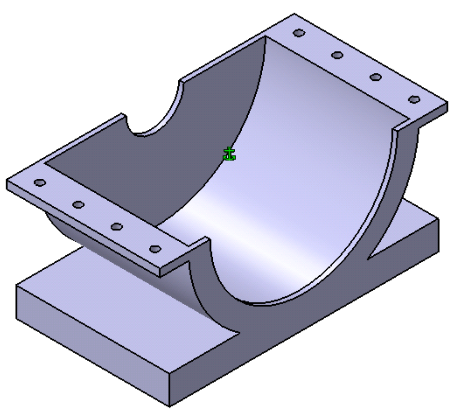


__Figure 11-67__ Lower Housing after it is fixed at its default location

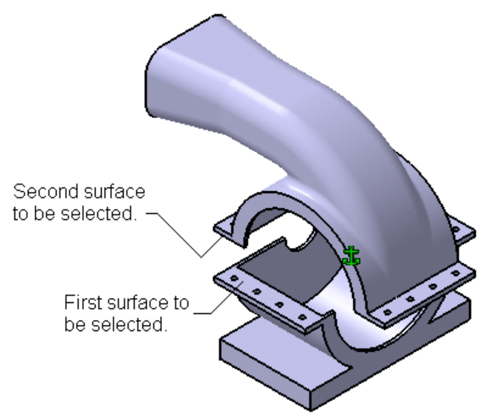


__Figure 11-68__ The surfaces to be selected to apply __Contact__ constraint

3. Choose the **Update All** button, if it is active. If this button is not active, the assembly does not need updation.

   The **Contact** constraint is applied between the two surfaces and its name is displayed in the **Specification Tree** under **Constraints**.

   Next, you need to apply the **Coincidence** constraint between the cylindrical surfaces of the two components to make them concentric.

4. Choose the **Coincidence Constraint** button from the **Constraints** toolbar. Now, select the two cylindrical surfaces, as shown in the Figure 11-69, to select the central axes of these surfaces.

5. Choose the **Update All** button to reorient the Upper Housing.

Note

To confirm the presence of the free degree of freedom, double-click on the ***Upper Housing*** *in the* ***Specification Tree****. The Upper Housing is activated. Now, choose* ***Analyze > Degree(s) of freedom*** *from the menu bar. The* ***Degrees of Freedom Analysis*** *dialog box is displayed along with a set of arrows in the x direction. This set of arrows displays the degree of freedom which is free. Choose the* ***Close*** *button from the* ***Degrees of Freedom Analysis*** *dialog box.*

Next, you need to align the right face of the Upper Housing with that of the Lower Housing. This can be done by applying the **Offset** constraint with 0 offset.

6. Choose the **Offset Constraint** button from the **Constraints** toolbar, and select the faces shown in Figure 11-70, to apply constraint between them.

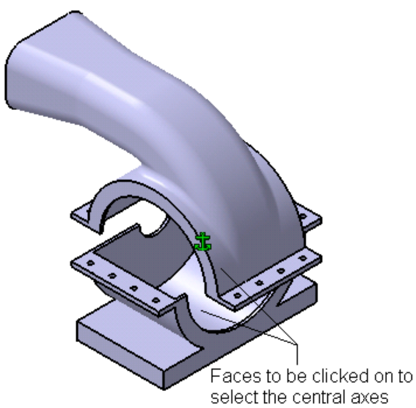


Figure 11-69 *Surfaces on which you need to click to select the central axes*

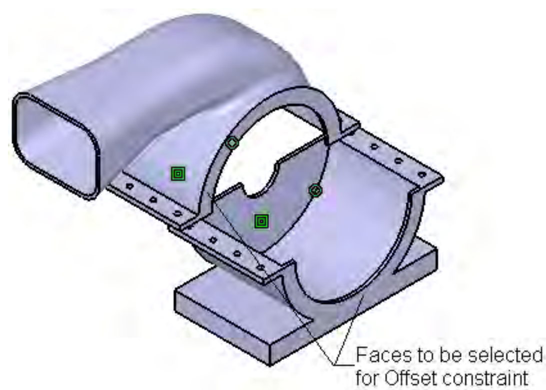
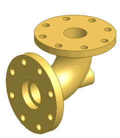


Figure 11-70 *The surfaces to be selected for applying Offset constraint*

The **Constraint Properties** dialog box is displayed. Make sure that the **Orientation** is set to **Same**, and the **Offset** value is set to 0.

7. Choose the **OK** button, and then choose the **Update All** button to apply the **Offset** constraint. The Upper Housing is fully constrained.



Tip. *To check whether a part is fully constrained, try to reorient it using the* ***Manipulation*** *tool with the* ***With respect to constraints*** *check box selected. If the part is fully constrained, it will not move or rotate in any direction.*

Assembling the Blower after Hiding the Upper Housing

The Blower needs to be assembled between the Upper and Lower Housings. To ease the process of assembling the Blower, you need to hide the Upper Housing.

1. Invoke the contextual menu by right-clicking on the name of the Upper Housing in the **Specification Tree** and choose the **Hide/Show** option to turn off the display of the selected component.

2. Now, insert the Blower in the assembly. Choose the **Coincidence Constraints** button and select the faces shown in Figure 11-71 to select the central axes of these faces.

3. Update the assembly.

Next, you need to place the left face of the Blower at an offset distance 0.635 from the inner left face of the Lower Housing using the **Offset** constraint.

4. Apply the **Offset** constraint between the faces shown in Figure 11-72. Enter **0.635** as the value in the **Offset** spinner and make sure that the **Orientation** is set to **Opposite** in the **Constraint Properties** dialog box. Update the model to bring the blower to its proper position.

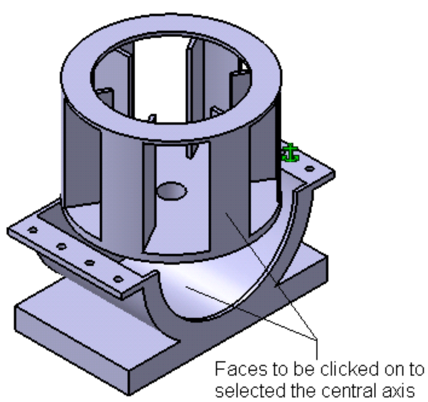


Figure 11-71 Faces to be clicked to select the central axes

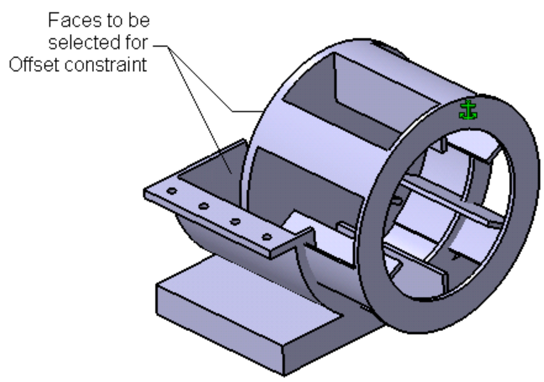


Figure 11-72 *Faces to be selected for applying* ***Offset*** *constraint*

Assembling the Motor Shaft

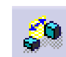
Next, you need to assemble the Motor Shaft.

1. Insert the Motor Shaft into the assembly file. By default, it is placed in the middle of the existing assembly, as shown in Figure 11-73. You need to move it out of the assembly to get a better view of the shaft.

2. Select the **Manipulation** button from the **Move** toolbar. Choose the **Drag along X axis** button and drag the Motor Shaft to move it out of the assembly, as shown in Figure 11-74. Exit the **Manipulation Parameters** dialog box.

   The direction of the Motor Shaft needs to be flipped. This is done because by default, the orientation of the Motor Shaft is not the same as that required. The direction of Motor Shaft will be flipped, while applying the constraint.

3. Choose the **Offset** button and then select the faces shown in the Figure 11-74. The **Constraint Properties** dialog box is displayed. Set the **Orientation** to **Same** and the **Offset** value to **0**. Update the model to place the Motor Shaft at its proper location.



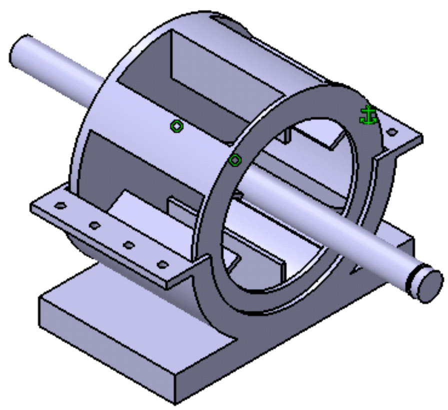


Figure 11-73 *Motor Shaft inserted at its default location*

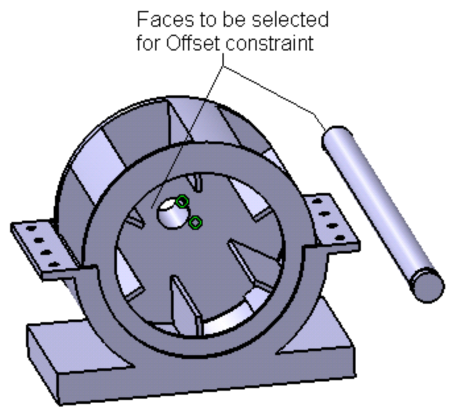


Figure 11-74 *Faces to be selected for* ***Offset*** *constraint*

   Next, you need to apply the **Coincidence** constraint between the axis of the Motor Shaft and the Blower hub. You will use the **Quick Constraint** tool to apply this constraint.

4. Choose the **Quick Constraint** button from the **Constraints** toolbar and move the cursor over the Motor Shaft. The axis of the shaft will be displayed as a center line. Select the axis by clicking over the center line. The axis will be now highlighted in orange. Similarly, select the axis of the Blower hub. The **Coincidence** constraint will be automatically applied between the two selected axes. Update the model to place the Motor Shaft inside the Blower hub, as shown in Figure 11-75.

Assembling the Motor

Next, you need to assemble the Motor with the Motor Shaft.

1. Insert the Motor in the Blower assembly. By default it will be placed in such a way that its body will overlap the existing assembly parts. Therefore, use the **Manipulation** tool to move it out into the open space.

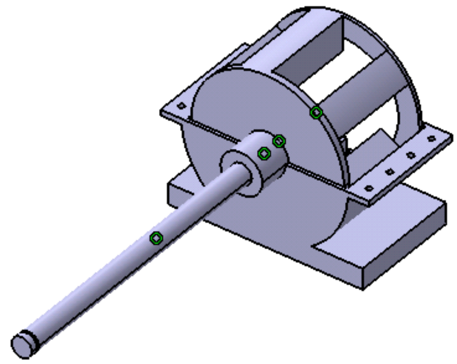


Figure 11-75 Position of the Motor Shaft with respect to the Blower shown from back side

2. Now apply the **Offset** constraint between the faces shown in the Figure 11-76. Set the **Orientation** to **Same** and the **Offset** value **zero**.

3. Apply the **Coincidence** constraint between the axis of the shaft and the axis of the hole on the back side of the Motor, refer to Figure 11-76. After applying both the constraints, update the model.

   You will notice that the base of the Motor and the Lower Housing appear to be parallel, but there is no constraint applied to both the faces. Therefore, you need to apply the **Angle** constraint to these two faces.

4. Choose the **Angle** button from the **Constraints** toolbar. Select the faces, as shown in Figure 11-76. Select the **Parallelism** radio button to make the selected faces parallel.

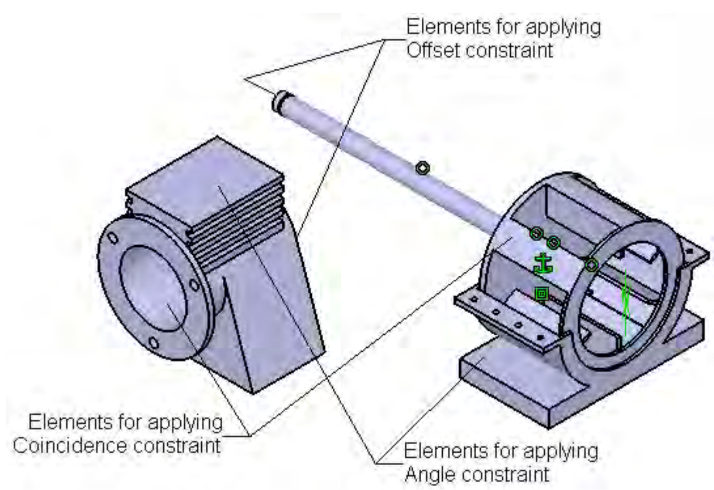


Figure 11-76 Elements to be selected for applying various constraints

Assembling the Cover and Turning on the Display of the Upper Housing

The last component to be assembled is the Cover. After assembling all components, you need to turn on the display of the hidden Upper Housing.

1. Insert the Cover into the Blower assembly. By default, it will be placed inside the blower. Use the **Manipulation** tool to move the Cover away from the assembly. Next, you need to apply constraints to the Motor Cover.

2. Apply the **Contact** constraint between the front face of the Motor and the bottom face of the Cover.

3. Apply the **Coincidence** constraint between the central hub of the Cover and cylindrical face of the Motor.

4. Apply another **Coincidence** constraint between one of the screw holes in the Cover and Motor. Various faces to be selected for applying these three constraints are shown in Figure 11-77. After all three constraints are applied, update the model to properly orient the cover in the Blower assembly.

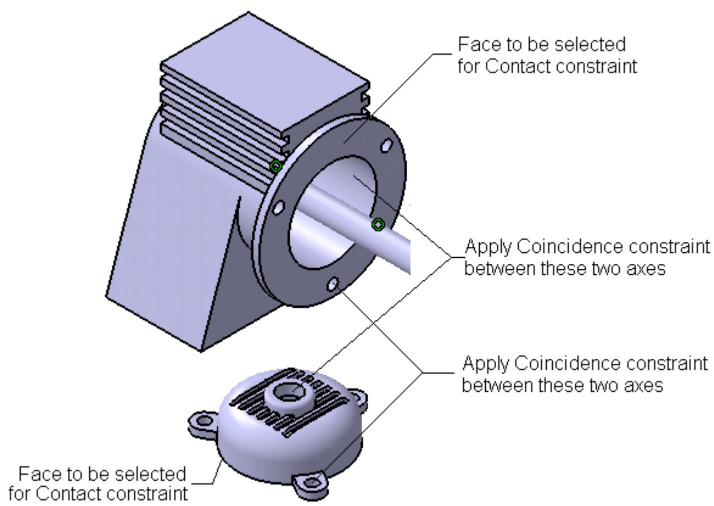


Figure 11-77 *Various faces to be selected for applying the constraints*

Next, you need to turn on the display of the Upper Housing.

5. Select the Upper Housing from the **Specification Tree** and choose the **Hide/Show** option from the contextual menu. The Upper Housing is displayed in the geometry area.

6. Select all constraints available under the **Constraints** heading in the **Specification Tree** and hide them. The Blower assembly is completed. The final assembly is shown in Figure 11-78.

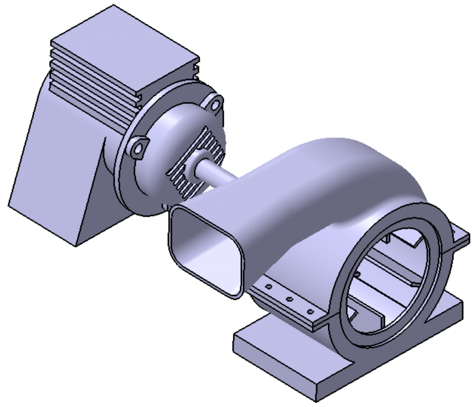


Figure 11-78 *The final Blower assembly.*

Creating the Exploded State of the Assembly

After creating the assembly, you can create the exploded state of the assembly. In the exploded state, all parts of the assembly are distinctly visible.

1. Select **Product1** from the **Specification Tree** and choose the **Explode** button from the **Move** toolbar. The **Explode** dialog box is displayed. 

2. Select the **All levels** option from the **Depth** drop-down list and select **2D** from the **Type** drop-down list. Click in the **Fixed product** area and then select the Lower Housing as the product to remain fixed, while exploding the assembly.

3. Choose the **Apply** button from the **Explode** dialog box to generate the exploded view. The **Information Box** will be displayed. Choose the **OK** button from the **Information Box** to close it. The exploded view of the Blower assembly is shown in Figure 11-79.

4. Choose the **OK** button from the **Explode** dialog box and then choose **Yes** from the **Warning** dialog box. The exploded state of the assembly is displayed in the geometry area.

5. To switch back to the assembled mode, choose the **Update** button from the **Tools** toolbar.

Saving the File

1. Choose the **Save** button from the **Standard** toolbar. The **Save As** dialog box will be displayed.

2. Browse for the *\My Documents\CATIA\c11\Blower Assembly* folder and save the file.

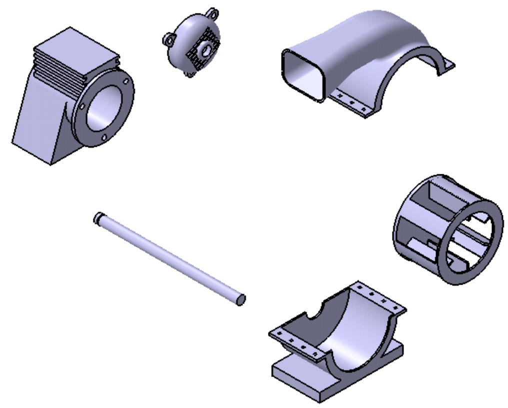


Figure 11-79 *The exploded view of Blower assembly*

Tutorial 2

In this tutorial, you will create some components of a Press Tool Base assembly using the top-down assembly approach. The Press Tool Base assembly is shown in Figure 11-80. The exploded state of this assembly is shown in the Figure 11-81. The dimensions of all components are shown in Figures 11-82 and 11-83. The drawing of the complete assembly is shown in Figure 11-80. **(Expected time: 45 min)**

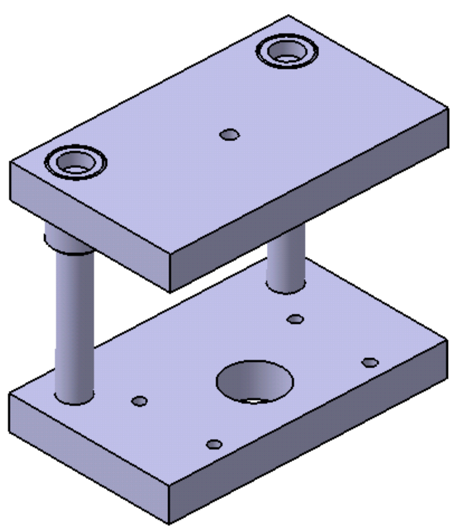


Figure 11-80 *The Press Tool Base assembly*

Note

Note that this is not a complete assembly of a Press Tool Base and is only created to explain the procedure of top-down assembly approach.

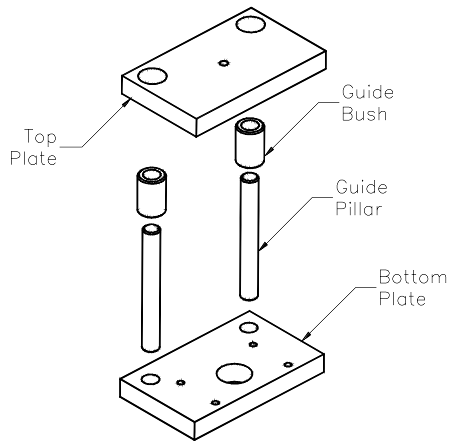


Figure 11-81 *The exploded state of the Press Tool Base assembly*

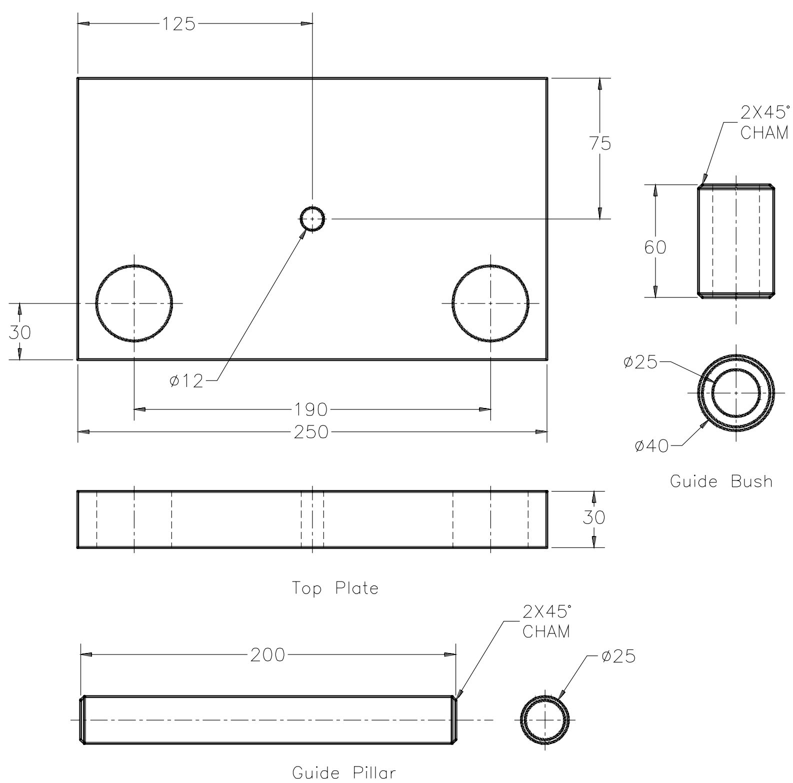


Figure 11-82 *Views and dimensions of the Top Plate, Guide Pillar, and Guide Bush*

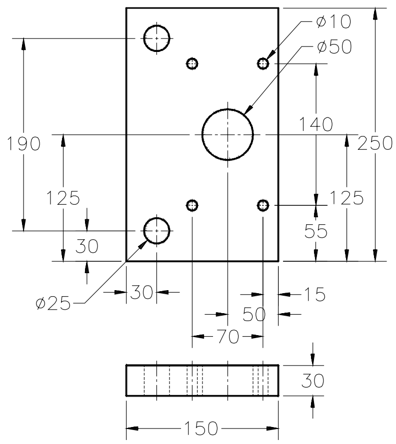


Figure 11-83 Views and dimensions of the Bottom Plate

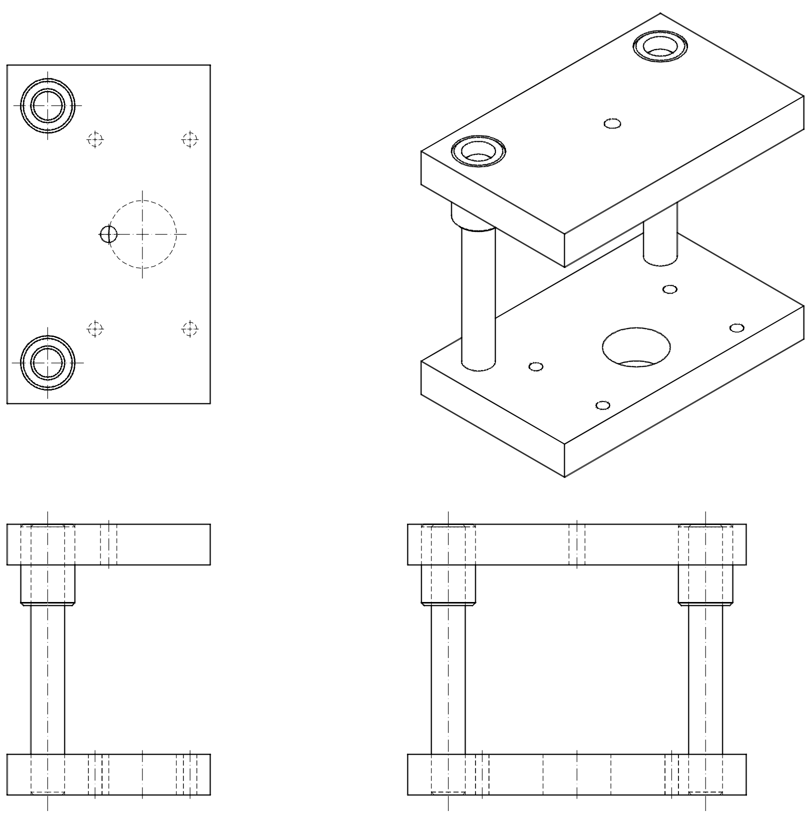


Figure 11-84 Drawing views of the Press Tool Base assembly

The following steps are required to complete this tutorial:

a. Start a new product file.
b. Create a new part inside the assembly. Modify its name and create features of the base component, refer to Figure 11-85. In this assembly the Bottom Plate will be the base component.
c. Create the Guide Bush and Guide Pillar as subsequent components inside the product file, refer to Figure 11-86 through 11-88.
d. Guide Pillar and Guide Bush are to be duplicated using the **Reuse Pattern** tool, refer to Figure 11-89.
e. Create the Top Plate, refer to Figure 11-90.
f. Finally save the Product file. The Part files will be saved automatically.

Before you start creating the top-down assembly, create the *\My Documents\CATIA\c11\Press Tool Base* folder. You will save the product file of the Press Tool assembly in this folder. All the Part files will also be automatically saved in the same folder.

Starting a New Product File

1. Choose the **New** button from the **Standard toolbar** and select the **Product** option from the **List of Types** area of the **New** dialog box. Choose the **OK** button to start a new Product file.

2. Invoke the contextual menu by right-clicking on **Product1** in the **Specification Tree**. Select **Properties** from the contextual menu to display the **Properties** dialog box.

3. Choose the **Product** tab, if it is not chosen, and change the part number to *Press Tool Base* in the **Part Number** edit box. Choose the **OK** button from the **Properties** dialog box. Note that now *Press tool* is displayed at the top of the **Specification Tree**.

   When the Product file is saved, it will automatically assign the name *Press Tool*.

Creating a New Part Inside the Assembly

1. Select **Press Tool Base** from the **Specification Tree** and choose the **Part** button from the **Product Structure Tools** toolbar.

   A new part is started inside the product file and is represented by **Part1** in the **Specification Tree**. Also, default planes are displayed in the geometry area. These are the default planes of the part and its origin is placed over the origin of the assembly coordinate system.

2. Set the part number and the instance name of the new part to *Bottom Plate* in the **Properties** dialog box invoked by right clicking on **Part1** in the **Specification Tree**.

3. Choose **Tools > Options** from the menu bar to invoke the **Options** dialog box. Select **Infrastructure** available on the left of this dialog box to expand this branch. Now, select the **Part Infrastructure** from the **Infrastructure** branch. Select the **Keep link with selected object** check box.

Creating Features of the Bottom Plate

After a new part file is created inside the assembly, you need to invoke the **Part Design** workbench to create the features. Since, the Bottom Plate is the base component, you will now create its features.

1. Click on the plus sign on the left of **Bottom Plate** in the **Specification Tree** to expand its branch. Now, double-click on the **Bottom Plate**, which is displayed inside the expanded branch to invoke the **Part Design** workbench.

2. Create the **Bottom Plate** using the part modeling tools. The final model of the Bottom Plate is shown in Figure 11-85.

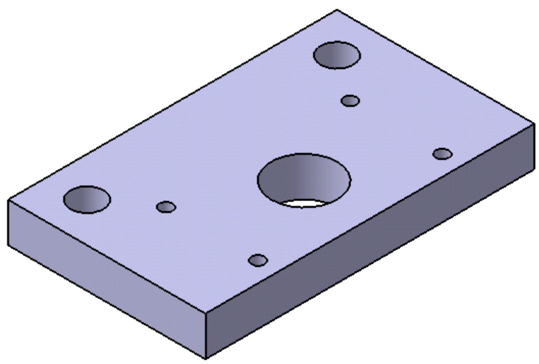


Figure 11-85 *The final model of the Bottom Plate*

3. Double click on **Press Tool Base** in the **Specification Tree** for switching to the **Assembly Design** workbench.

4. Apply the **Fix** constraint to the Top Plate.

Creating the Guide Pillar

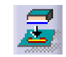
Next, you need to create the Guide Pillar by referring to the geometry of the Base Plate.

1. Choose the **Part** button from the **Product Structure Tools** toolbar to insert the second part in the assembly. The **New Part: Origin Point** dialog box is displayed.

2. Choose the **No** button from the **New Part: Origin Point** dialog box to place the origin of the second part over the origin of the assembly coordinate system.

3. Rename the second component to *Guide Pillar* and invoke the **Part Design** workbench by double-clicking on it.

4. For creating the Guide Pillar, you need to take the reference of the geometry of the Bottom Plate. Select the bottom face of the Bottom Plate as the sketching plane and invoke the **Sketcher** workbench.

5. Select the circular edge of left hole having the diameter of 25 and choose the **Project 3D Elements** button from the **Operation** toolbar. The geometry is extracted from the selected edge and is projected over the sketch plane. 

   Note that there is no need to provide any dimension to the circle. The size of the extracted circle is the same as that of the edge of hole from which it is extracted.

6. Exit the **Sketcher** workbench and extrude the sketch, refer to Figure 11-82 for dimensions.

7. Hide the Bottom Plate and apply chamfer on both the ends of the Guide Pillar. This will complete the feature creation of the Guide Pillar.

8. Turn on the display of the Bottom Plate and switch back to the **Assembly Design** workbench. The Guide Pillar in the assembly is shown in Figure 11-86.

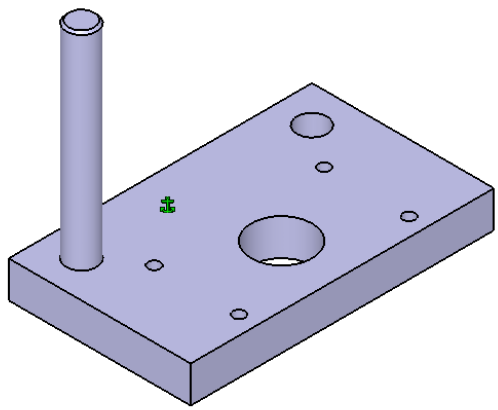


Figure 11-86 *Final Guide Pillar*

Creating the Guide Bush

It is evident from Figure 11-80, that Guide Bush will be placed over the Guide Pillar. Therefore, the geometry of the Guide Pillar will be used to create the Guide Bush.

1. Start another part file inside the assembly file and rename it as *Guide Bush.* Invoke the **Part Design** workbench.

2. Select the top face of the Guide Pillar as the sketching plane and draw two concentric circles. Make the inner circle coincident with the outer edge of the Guide Pillar and apply dimension to the outer circle, as shown in the Figure 11-87.

3. Exit the **Sketcher** workbench and extrude the sketch to 60 units. Now apply chamfers at both outer edges of the Guide Bush.

4. This completes the feature creation of the Guide Bush as shown in Figure 11-88. Return to the assembly workbench by double clicking on Press tool in the **Specification Tree**.

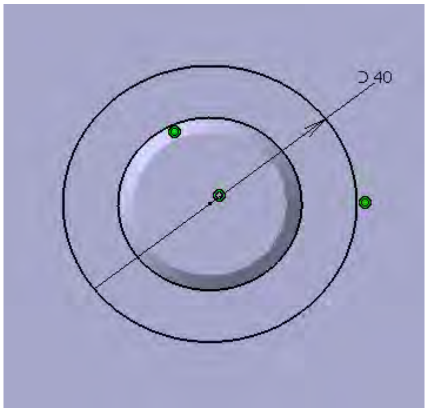


Figure 11-87 *Sketch of Pad feature for creating Guide Bush*

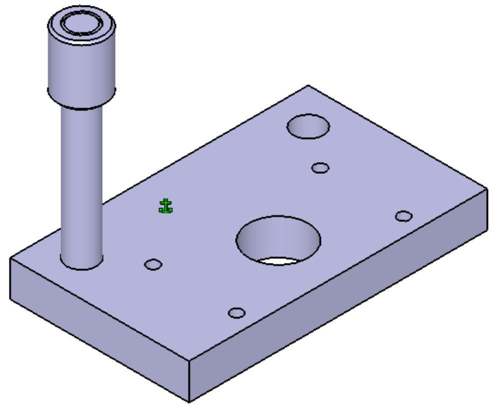


Figure 11-88 *Final Guide Bush*

Creating Second set of Guide Bush and Guide Pillar

In the Press Tool Base assembly, Guide Bush and Guide Pillar are used in pairs. Therefore to create the second set of Guide Bush and Guide Pillar, you will use the **Reuse Pattern** tool. The advantage of using this tool is that the second set of components will be places at the desired location without applying any constraints between them.

1. Expand the branch of Bottom Plate in the **Specification Tree** to display the **Rectangular Pattern** used for creating holes.

2. Press and hold down the CTRL key from the keyboard. Select the **RecPattern2** and **Guide Pillar** from the **Specification Tree**.

3. Choose the **Reuse Pattern** button from the **Constraints** toolbar; the **Instantiation on a pattern** dialog box is displayed. Choose the **OK** button to place another Guide Pillar in the second hole.

4. Similarly, assemble the second Guide Bush over the newly placed Guide Pillar using the **Resume Pattern** tool by selecting the rectangular pattern and Guide Bush from the **Specification Tree**.

   Once both the components are duplicated, you can close the expanded branch of the **Specification Tree**. Figure 11-89 shows the assembly model, after placing the second set of Guide Pillar and Guide Bush.

Creating the Top Plate

Top Plate is the last component that will be created in the Press Tool Base assembly. You will use the reference of the geometries of Bottom Plate and Guide Bush to draw the sketch for the Pad feature of the Top Plate.

1. Start another part inside the assembly file and rename it as *Top Plate.* Invoke the **Part Design** workbench.

2. Select the top face of the Guide Pillar as the sketching plane and invoke the **Sketcher** workbench.

3. Extract four side edges of the Bottom Plate and the outer circular edge of the Guide Bush using the **Project 3D Elements** tool, as shown in Figure 11-90.

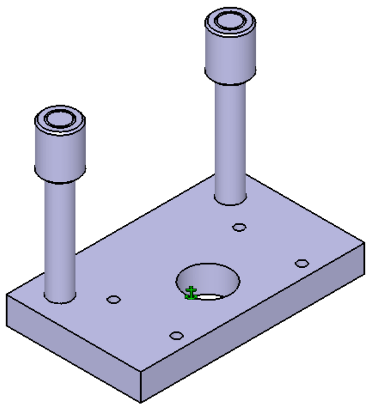


Figure 11-89 *Assembly after placing the second set of Guide Pillar and Guide Bush*

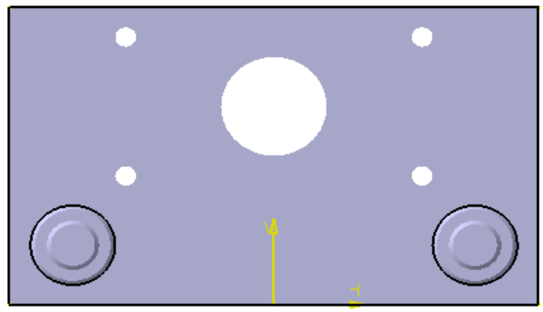


Figure 11-90 *Sketch of the Pad feature for creating Top Plate*

4. Exit the **Sketcher** workbench and extrude the sketch by 30 units by flipping its direction of extrusion. Now, create a hole on the top face of the Top Plate.

   This completes the feature creation of the Top Plate.

5. Switch back to the **Assembly Design** workbench. The final assembly is shown in the Figure 11-91.

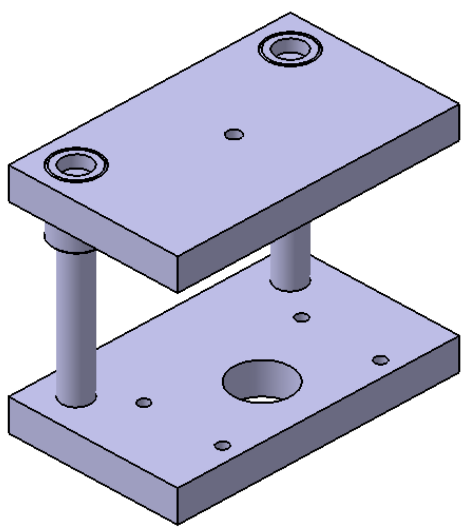


Figure 11-91 *The final Press Tool Base assembly*

Saving the Assembly File

1. Choose the **Save** button from the **Standard** toolbar. You need to make sure that you are in the **Assembly Design** workbench before saving the assembly. The **Save As** dialog box is displayed.

2. Browse the location of the *Press Tool Base* folder that you created in the beginning of this tutorial.

3. Choose the **Save** button from the **Save As** dialog box. The **Save As** confirmation box is displayed, as shown in Figure 11-92.

4. Choose the **Yes** button from this confirmation box to save the assembly file, along with all the Part files.

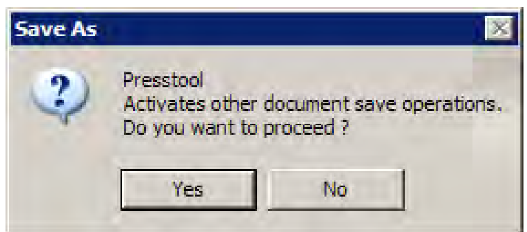


Figure 11-92 *The* ***Save As*** *confirmation box*

5. Close the assembly file by choosing **File** > **Close** from the menu bar.

SELF-EVALUATION TEST

Answer the following questions and then compare your answers with those given at the end of this chapter.

1. In the bottom-up assembly design approach all parts are created in separate part files and then inserted in to the product file. (T/F)

2. While creating a top-down assembly in CATIA V5, all individual parts created in the assembly needs to be saved separately, after saving the Product file. (T/F)

3. The **Angle** constraint can be used to make two surfaces parallel to each other. (T/F)

4. The **Manipulation** tool is used to move and rotate a part present inside an assembly. (T/F)

5. You cannot select a component to fix it at its original position, while exploding an assembly model**.** (T/F)

6. The __________constraint is used to place two planar faces at a specified angle with respect to each other.

7. You can place multiple instances of a component over a predefined pattern by using the __________tool.

8. The __________option is selected from the **Type** drop-down list in the **Explode** dialog box to explode all components of assembly along the **2D** plane, which is currently parallel to screen.

9. The edges of a component can be snapped on to an edge of another component by using the __________ tool.

10. If two parts having the same name are inserted into the assembly file then __________ dialog box is displayed.

REVIEW QUESTIONS

Answer the following questions.

1. The __________ constraint is used to make two cylindrical surfaces concentric.

2. Select the component and then choose the __________ button to hide it from the geometry area.

3. The __________ tool can be used to move a component and also apply a constraint to it.

4. The cross-section of an assembly model can be viewed using the __________ tool.

5. An existing part can be inserted into a Product file using the _________ button.

6. Which tool is used to calculate the interference between two mating components?

   (a) **Measure** (b) **Clash**
   (c) **Smart Move** (d) **Snap**

7. Which button is used to replace a constraint by another constraint?

   (a) **Reuse Pattern** (b) **Replace Component**
   (c) **Change Constraint** (d) **Replace Constraint**

8. Which button is used to apply the most appropriate constraint to the current selection set?

   (a) **Quick Constraint** (b) **Change Constraint**
   (c) **Contact Constraint** (d) None of these

9. After selecting the **Reuse Pattern** button, which dialog box is displayed?

(a) **Instantiation on a pattern** (b) **Reuse pattern**
(c) **Constraint properties** (d) **Constraint definition**

10. Which constraint is used to fix the position of a part in 3D space?

(a) **Contact** (b) **Fix**
(c) **Coincidence** (d) **Angle**

EXERCISE

Exercise 1

Create the assembly of the Radial Engine shown in Figure 11-93. The assembly in the exploded state is shown in Figure 11-94. Note that this exploded view is provided only for your understanding and has not been generated using CATIA. The dimensions of various parts of this assembly model are given in Figure 11-96 through Figure 11-99.

(Expected time: 3 hr 30 min)

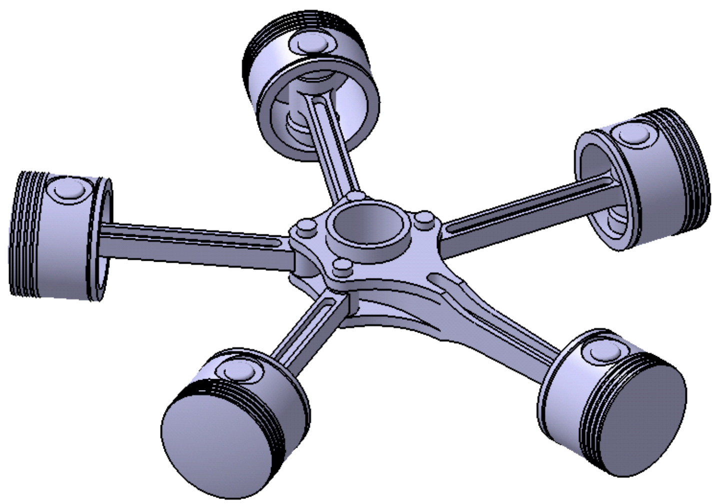


Figure 11-93 *The Radial Engine assembly*

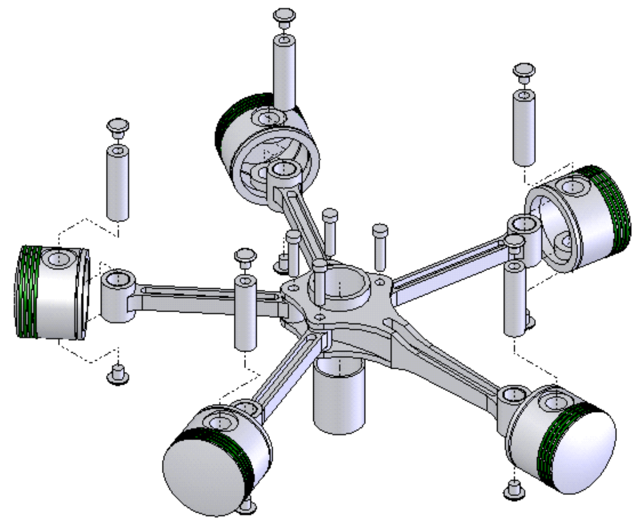


Figure 11-94 *Exploded view of the Radial Engine assembly*

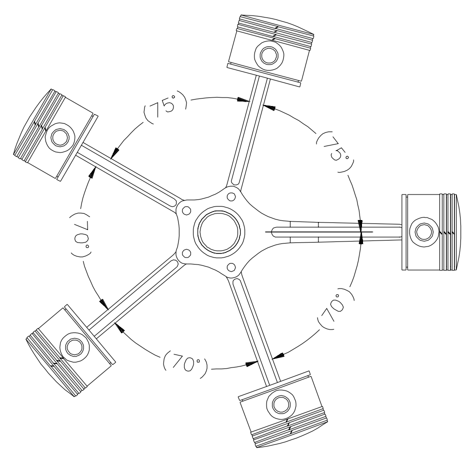


Figure 11-95 *Positioning of the Articulated Rods*

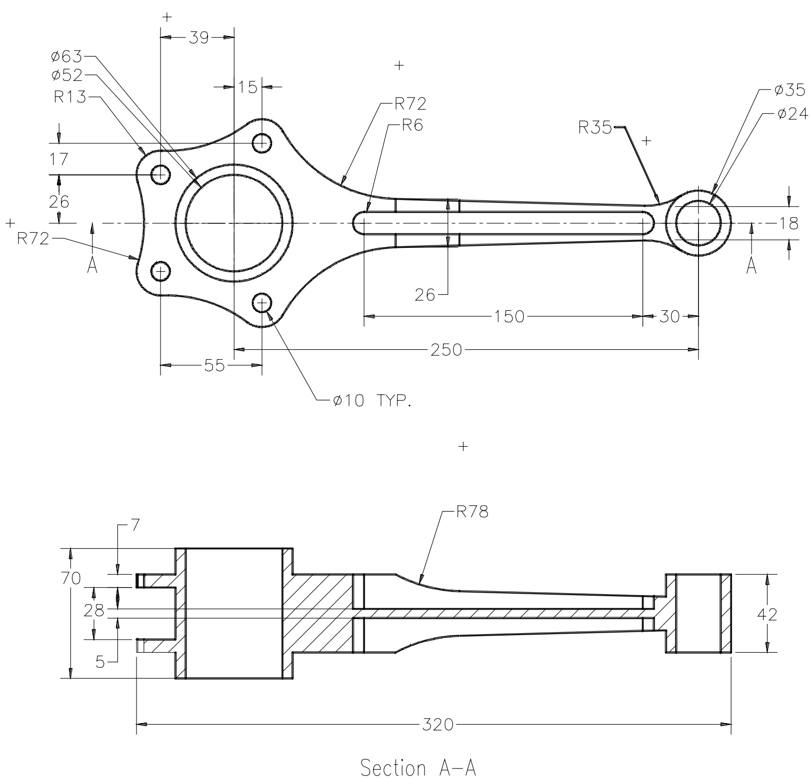


Figure 11-96 *Views and dimensions of the Master Rod*

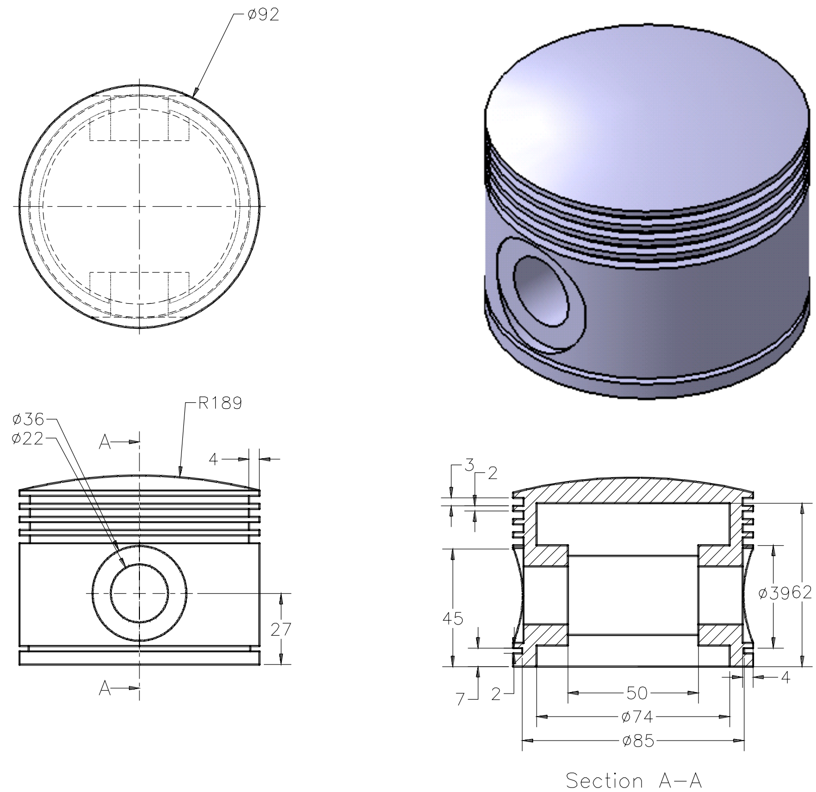


Figure 11-97 *Views and dimensions of the Piston*

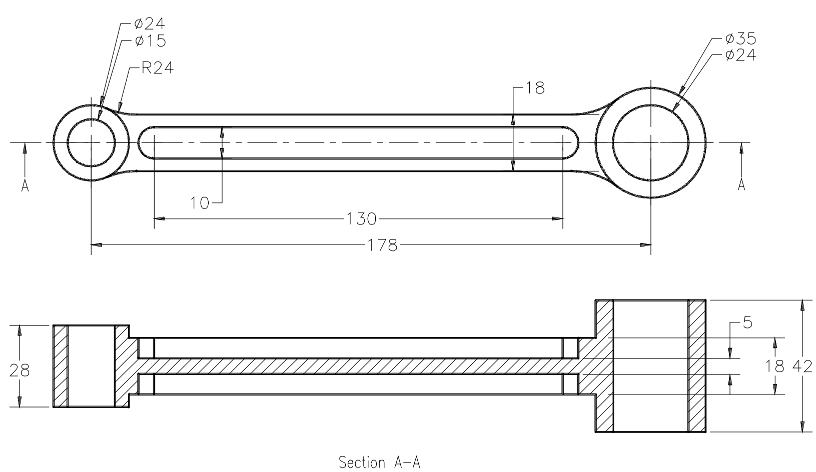


Figure 11-98 *Views and dimensions of the Articulated Rod*

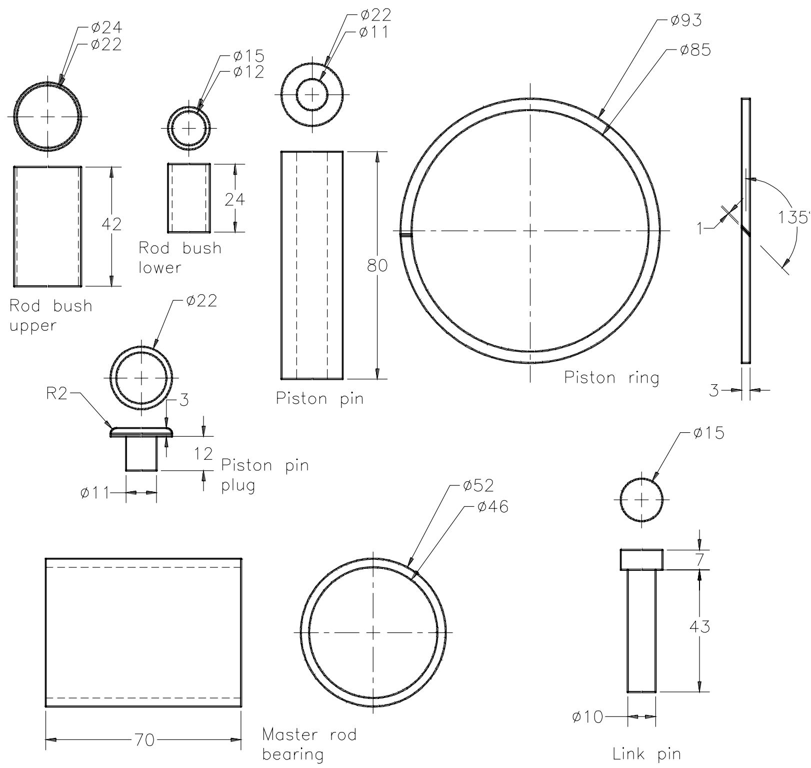


Figure 11-99 *Views and dimensions of the other components*

Answers to Self-Evaluation Test

1. T, 2. F, 3. T, 4. T, 5. F, 6. Angle, 7. Reuse Pattern, 8. 2D, **9. Snap, 10. Part number conflicts**