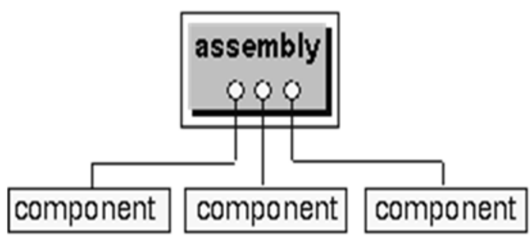
CHAPTER 6 – ASSEMBLY MODELING

Every day, we see many examples of components that are assembled together into one model such as bicycles, cars, and computers. All of these products were created by designing and manufacturing individual parts and then fitting them together. The designers who create them have to carefully plan each part so that they all fit together perfectly in order to perform the desired function.

In this chapter, you will learn two kinds of approaches used in *Assembly* modeling. We will practice assembly modeling using the impeller assembly as an example. Some parts of this assembly have already been modeled in earlier chapters.

NX 12 *Assembly* is a part file that contains the individual parts. They are added to the part file in such a way that the parts are virtually in the assembly and linked to the original part. This eliminates the need for creating separate memory space for the individual parts in the computer. All the parts are selectable and can be used in the design process for information and mating to insure a perfect fit as intended by the designers. The following figure shows how components are added to make an assembly.



6.1 TERMINOLOGY

Assembly

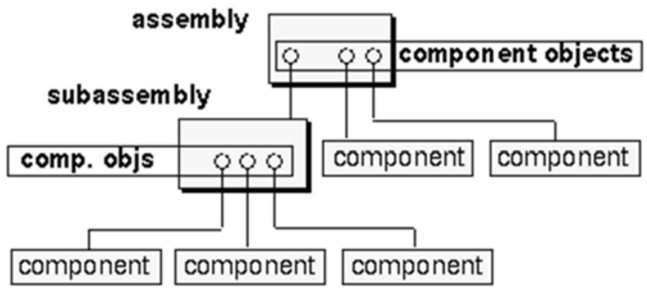
An assembly is a collection of pointers to piece parts and/or subassemblies. An assembly is a part file, which contains component objects.

Component Object

A component object is a non-geometric pointer to the part file that contains the component geometry. Component object stores information such as the *Layer*, *Color*, *Reference set*, *position data for component relative to assembly* and *path of the component part on file system.*

Component Part

A component part is a part file pointed to by a component object within an assembly. The actual geometry is stored in the component part and is **referenced**, **not copied** by the assembly.



Component Occurrences

An *occurrence* of a component is a pointer to geometry in the component file. Use component occurrences to create one or more references to a component without creating additional geometry.

Reference Set

A *reference set* is a named collection of objects in a component part or subassembly that you can use to simplify the representation of the component part in higher level assemblies.

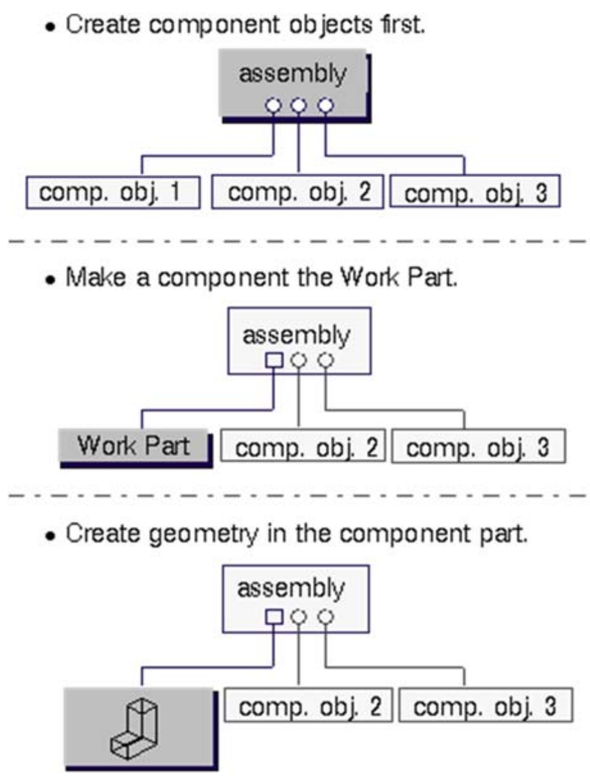
6.2 ASSEMBLING APPROACHES

There are two basic ways of creating any assembly model.

- Top-Down Approach
- Bottom-Up Approach

6.2.1 Top-Down Approach

In this approach, the assembly part file is created first and components are created in that file. Then individual parts are modeled. This type of modeling is useful in a new design.

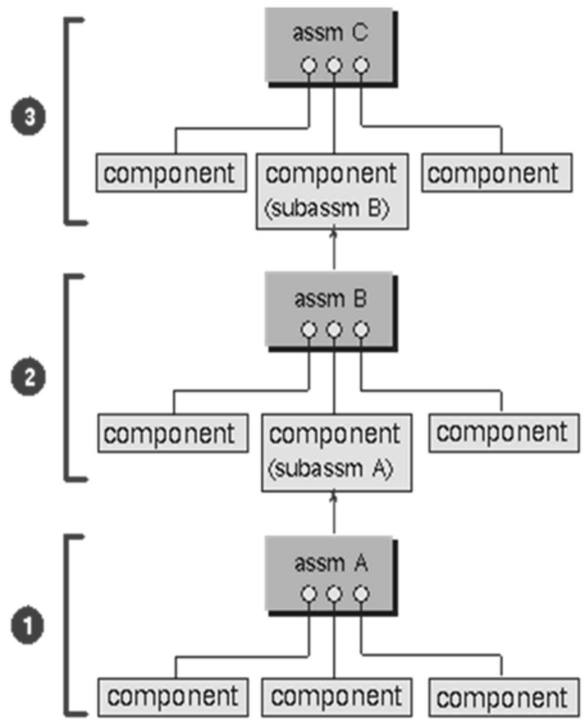


6.2.2 Bottom-Up Approach

The component parts are created first in the traditional way and then added to the assembly part file. This technique is particularly useful, when part files already exist from the previous designs, and can be reused.

6.2.3 Mixing and Matching

You can combine these two approaches, when necessary, to add flexibility to your assembly design needs.

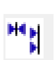


6.3 ASSEMBLY AND CONSTRAINT NAVIGATORS

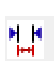
The *Assembly Navigator* and *Constraint Navigator* are located on top of the *Part Navigator* in the *Resource Bar* on the left of the screen. These navigators show you various things that form the assembly, including part hierarchy, the part name, information regarding the part such as whether the part is read only, count of objects, and the constraint status.

6.4 MATING CONSTRAINTS

After the *Component Objects* are added to the assembly part file, each *Component Object* is mated with the existing objects. By assigning the mating conditions on components of an assembly, you establish positional relationships, or constraints, among those components. These relationships are termed *Mating Constraints.* A mating condition is made up of one or more mating constraints. There are different mating constraints as explained below:

 **Touch/Align**: Planar objects selected to align will be coplanar but the normals to the planes will point in the same direction. Centerlines of cylindrical objects will be in line with each other.

 **Concentric**: Constrains circular or elliptical edges of two components so the centers are coincident and the planes of the edges are coplanar.

 **Distance**: This establishes a +/- distance (offset) value between two objects

 **Parallel**: Objects selected will be parallel to each other.

 **Perpendicular**: Objects selected will be perpendicular to each other.

 **Bond**: Creates a weld and welds components together to move as a single object.

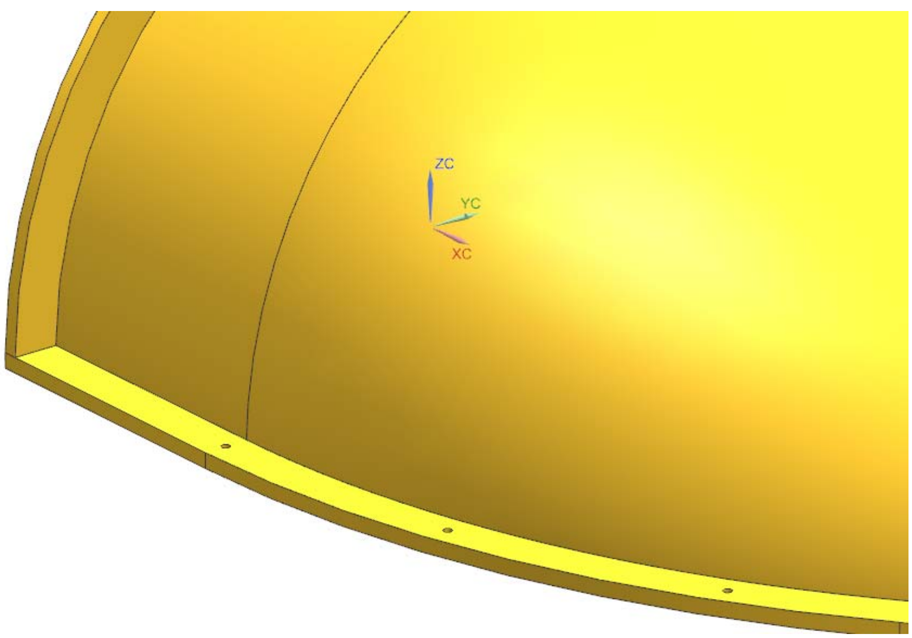
 **Center**: Objects will be centered between other objects, i.e. locating a cylinder along a slot and centering the cylinder in the slot.

 **Angle**: This fixes a constant angle between the two object entities chosen on the components to be assembled.

6.5 EXAMPLE

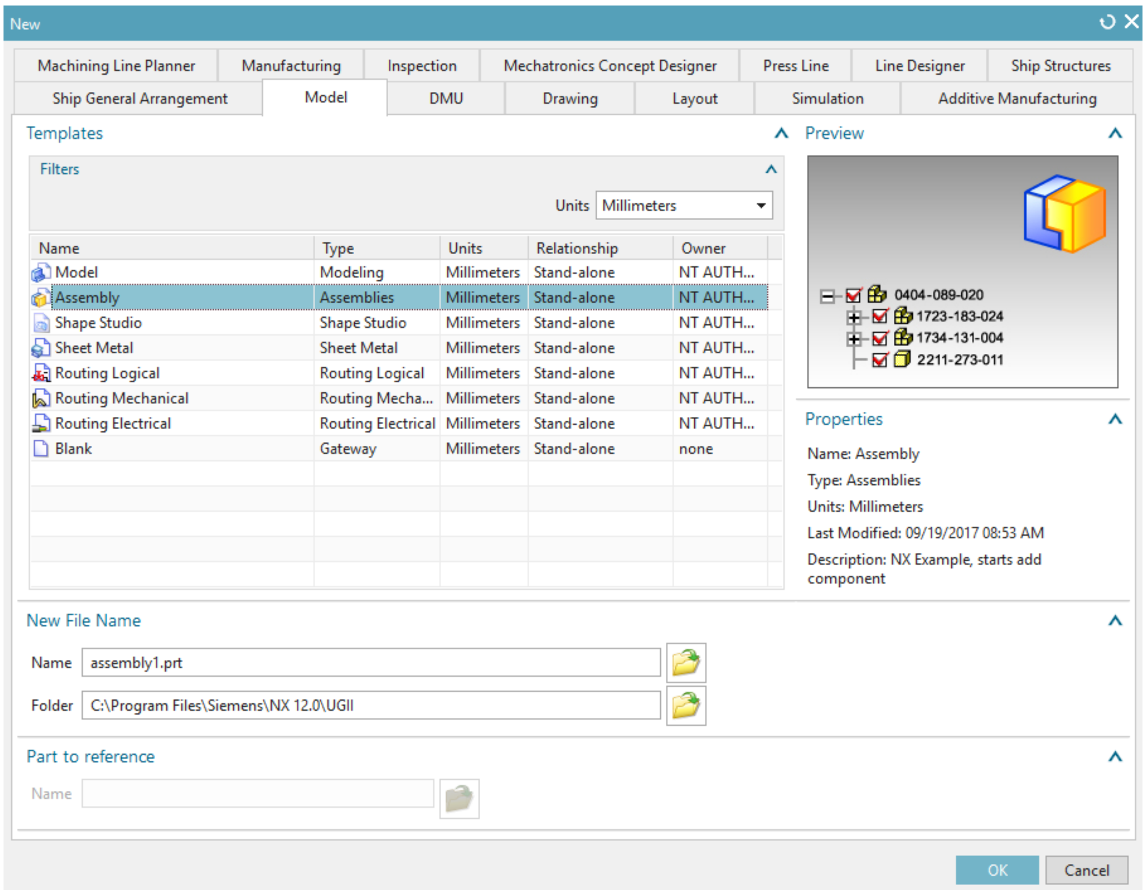
We will assemble the impeller component objects. You have modeled all the components in previous chapters. Now we have to insert them into the assembly environment and apply constraints to locate them relative to each other. Once the assembling is completed, we can create an exploded view and prepare the drafting.

Before starting the assembly modeling, make three through-holes on each side of the *Impeller-lower-casing* and *Impeller-upper-casing* (a total number of 6 holes for each casing) for the *Hexa-bolt*. Diameter of the holes should be 0.25 and their location should be similar to the figure below. Make sure to create the holes in the same places for both lower and upper casing, so that when they are assembled they match.



6.5.1 Starting an Assembly

- Create a new file
- Choose **Assembly** under the **Model** tab
- Set the **Units** to **Inches**
- Name it as **Impeller_assembly.prt**

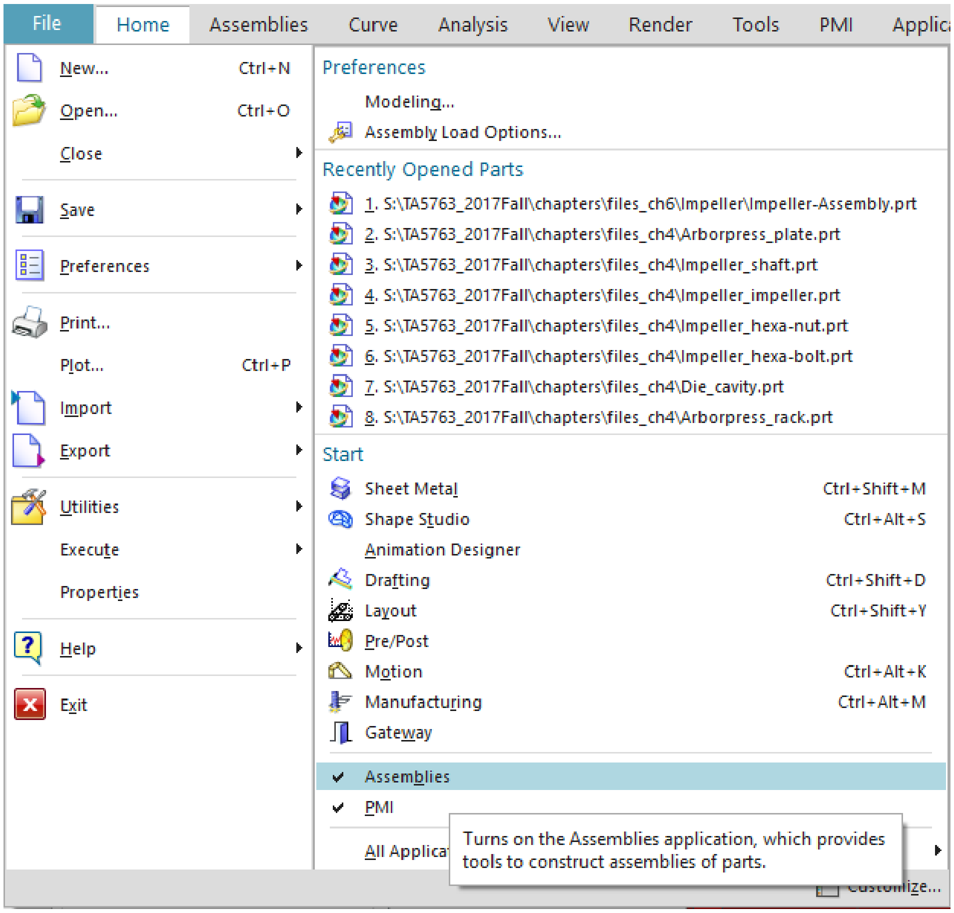


OR, if you are in the *Modeling Application* and want to start assembling,

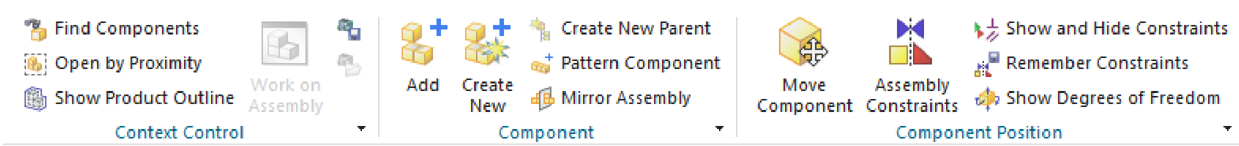
- Turn on **Assemblies** option in **Application** tab and a new **Assemblies** tab shows up

OR

➢ Click **File** →**Assemblies** as shown below



➢ The **Home** menu bar will now display tools for assembly



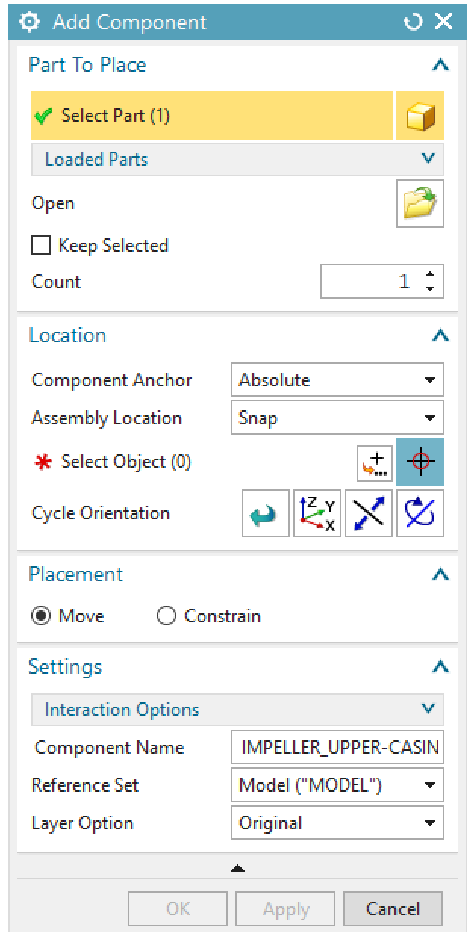
In the **Components** option,

- *Add* option adds new component objects whose part files are already created.
- *Create New* lets you create new component geometries inside the assembly file when you are using *Top-Down* approach of assembly.

The *Assembly Constraints* allows you to create assembly constraints and *Move Components* allows you to reposition the components wherever you want them in the assembly.

6.5.2 Adding Components and Constraints

➢ Choose **Add**



The dialogue box shown on the right side will pop up. You can select the part files from those existing (should be already shown in *Loaded Parts* tab) or you can load the part files using the *Open* file options in the dialog box. This will load the selected part file into the *Loaded Parts* dialog box.

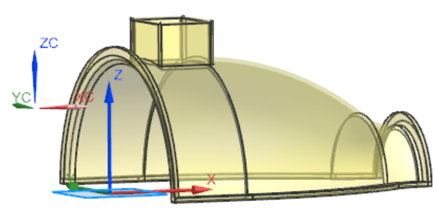
➢ Click on the **Open** icon and select the file **Impeller_upper-casing.prt**

➢ Click **OK** in the part name dialog box

Then we need to set a location to place the coordinate system of the first component. In the *Location* group box, keep the default *Snap* for the *Assembly Location* option.

➢ Click **Select Object**

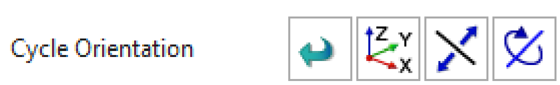
Now you should be able to see the part in a transparent mode, as shown in the figure on the right side.



➢ Click the **Point** Dialog icon and create the coordinates of [0, 0, 0]

➢ Click **OK** to exit the **Point** Dialog

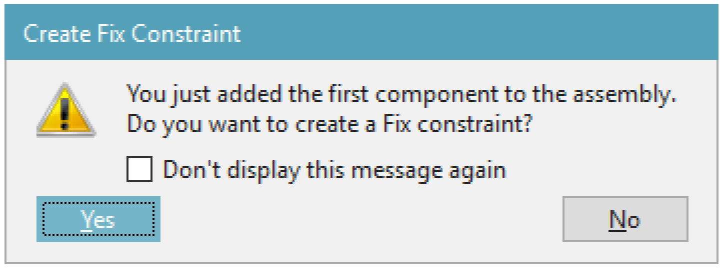
Note: Feel free to play with the **Cycle Orientation** options to set different orientations.

➢ 

In the *Placement* group box, we can define where and how we place this component. In this case, we will leave the options as default.

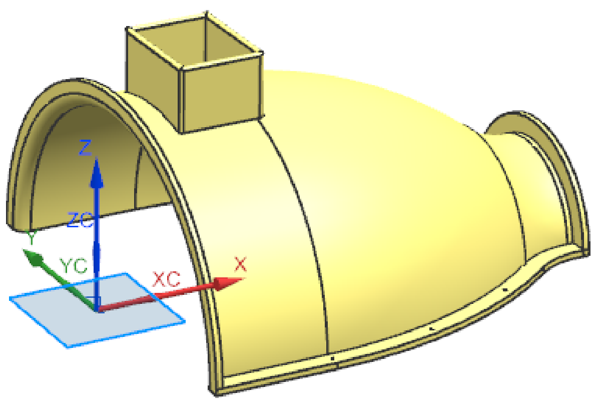
➢ Click **OK** to exit the **Add Component** Dialog

Then, you will see a pop-up dialog appears as shown in the figure below.



➢ Click **Yes** to create a Fix constraint to this part.

Now what you have should like the figure below.



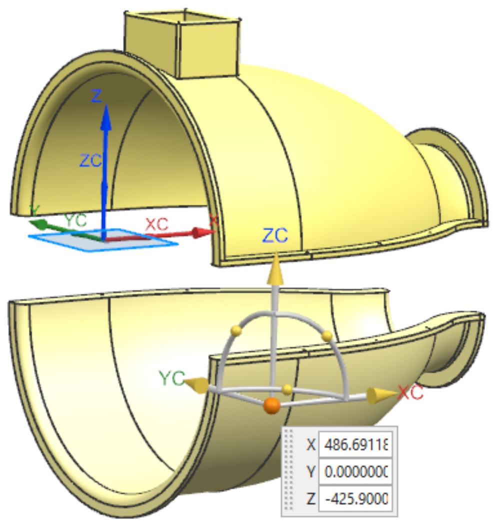
Let's move on to add the second component, the lower casing.

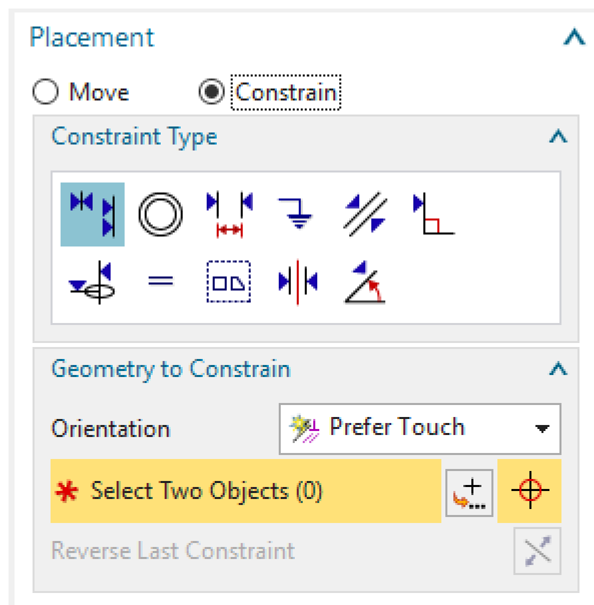
➢ Click on **Add** in the assembly section

➢ Select the file **Impeller_lower-casing.prt** from the *Loded Parts* or *Open*

➢ In the **Location** group box change the option to **Absolute – Work Part** to place the new part at the absolute origin of the current work part

➢ In the **Placement** group box, first toggle the **Move** radio button and move the lower casing away from the upper casing to have enough space for selecting the mating surfaces. What you will have should like the figure below.





➢ Then, toggle the **Constrain** radio button.

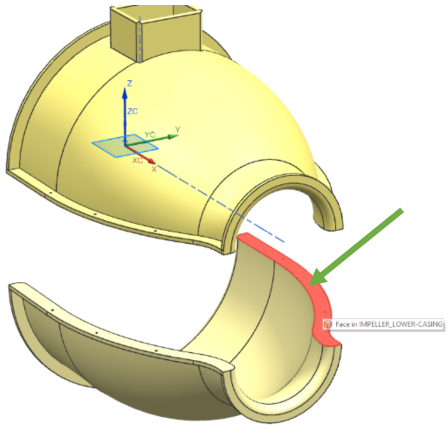
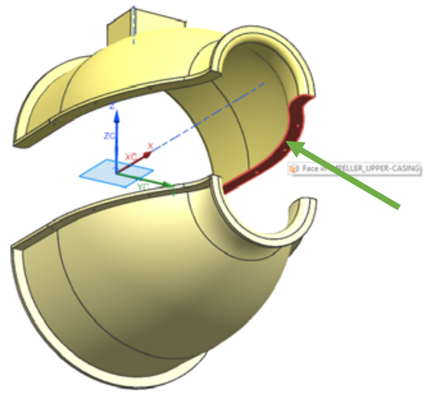
Now let's mate the upper and the lower casings. You can access all the constraints in the drop-down menu in the *Constraint Type* box.

Here you can see the different Mating Types, which were explained above in the previous section.

➢ Make sure the **Touch Align** icon  is selected in the **Type** dialog box

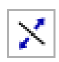
➢ First, select the face that the arrow is pointing to as shown below in the figure on the left.

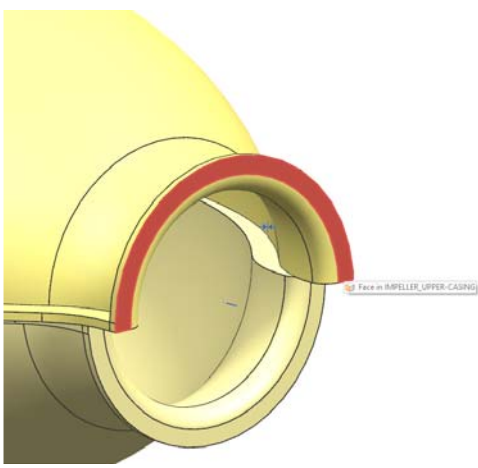
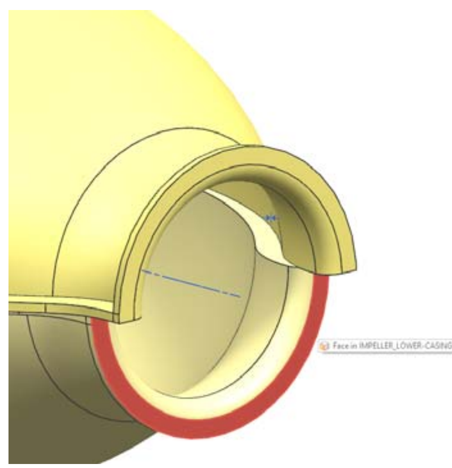
➢ Click on the face of the upper casing in the screen as shown in the figure on the right.

You may have to adjust the view in order to select the faces. After you choose these two faces, the Touch Align constraint will be automatically added.

Let’s add another **Touch Align** constraint.

- Click on the **Flange** of the upper casing
- Click on the **Flange** of the lower casing, you may need to inverse the direction of constraint by click on the **Inverse** icon 

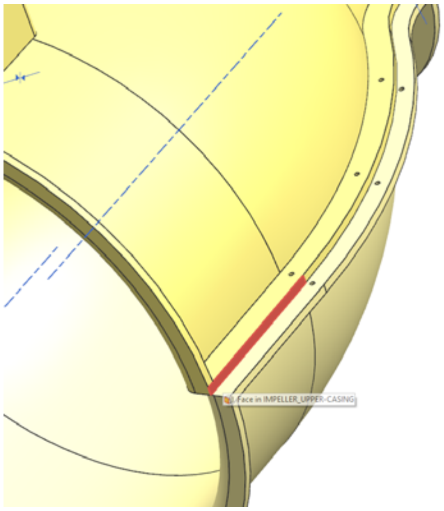
 

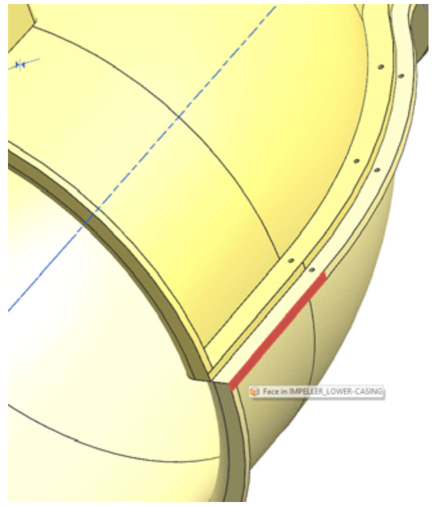
Note: If it is difficult for you to select the faces because of the position of the parts, you can move them by toggling the *Move* in the Placement group box and manipulate its handler.

Let’s add another **Touch Align** constraint.

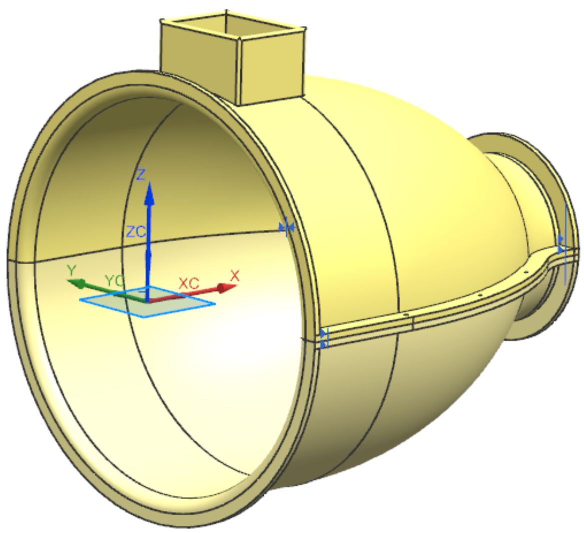
- Make sure you are still using **Touch Align**

- Click on the flat face of the upper casing as shown and then the corresponding face on the lower casing





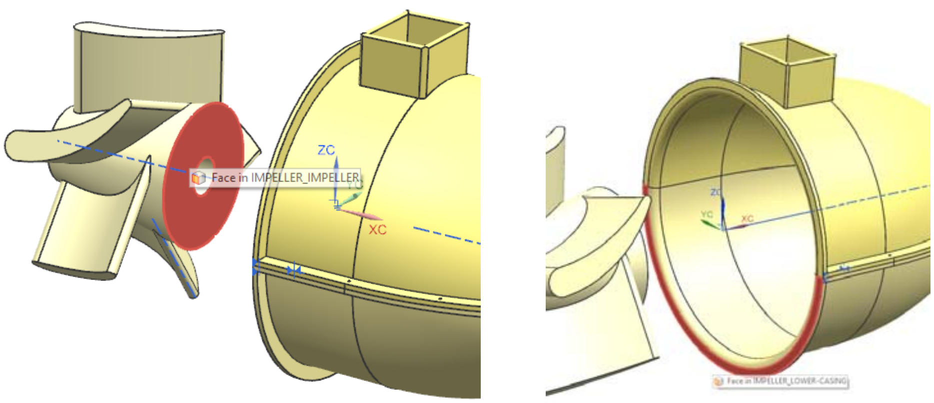
The two assembled components will be seen as shown in the figure below.



The lower casing is constrained with respect to the upper casing. Now let us add the impeller.

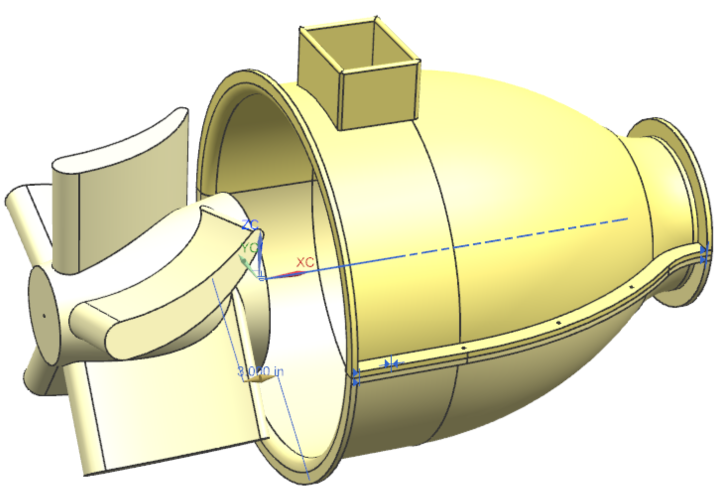
- Choose **Assemblies** →**Component** →**Add** to add a component to the current assembly
- Open the file **Impeller_impeller.prt**
- Click **OK** on the dialog box

- Choose *Absolute – Work Part* for Assembly Location
- Toggle the **Constrain** button
- Click on the **Distance** icon  in the **Constraint Type** box
- Select the two faces, first on the impeller and then on the casing, as shown in the figure below

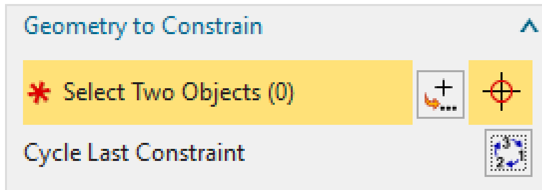


- In the **Distance** box in the **Placement** group, enter the value of **3**
- Press **Enter** to preview the current assembly

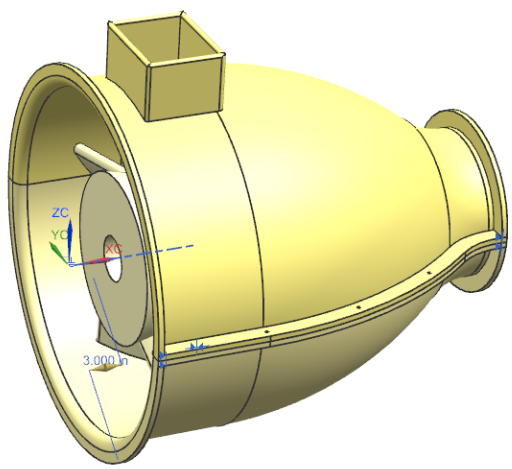
The preview may show the impeller oriented in the direction opposite to the one we want.



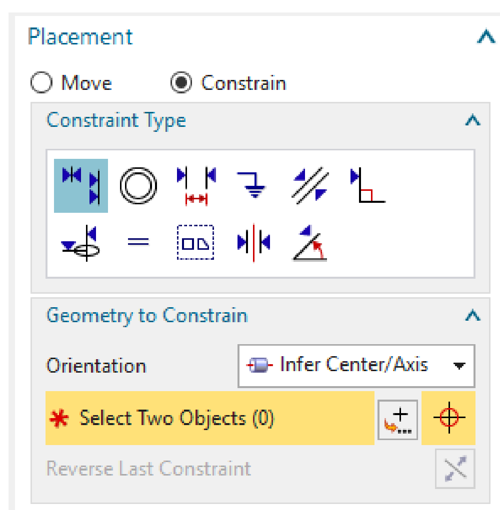
➢ To change the orientation of the part or the distance direction, in the **Placement** window, click on the **Cycle Last Constraint**  button in the **Geometry to Constrain** box, as shown below



Check the assembly status from its preview, you may have to click on the  button for several times to get the desired result. Now the impeller will be oriented in the right direction.



We will now add the shaft using the *Center* constraint.



➢ Click on **Assemblies** →**Components** →**Add**

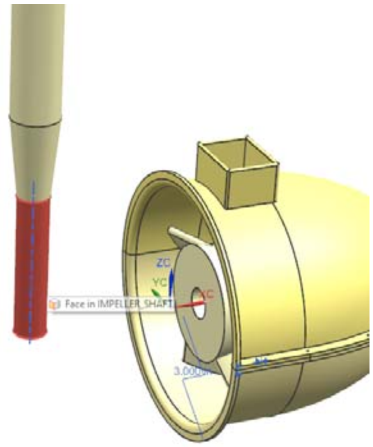
➢ Open the file **Impeller_shaft.prt**

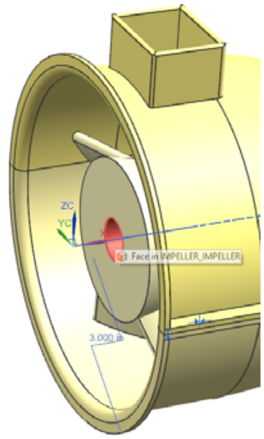
➢ Click **OK** on the dialog box

➢ Choose the **Touch Align** icon in the **Constraint Type** box

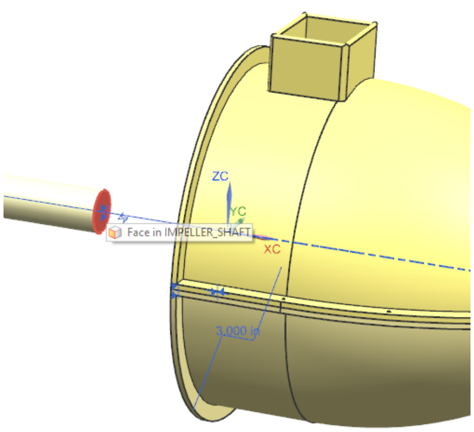
➢ Choose the **Infer Center/Axis** option in the **Geometry to Constrain** box

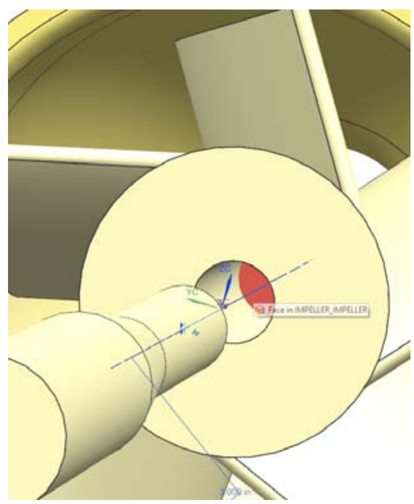
- Select the two surfaces, first on the shaft and then on the impeller as shown in the figures below



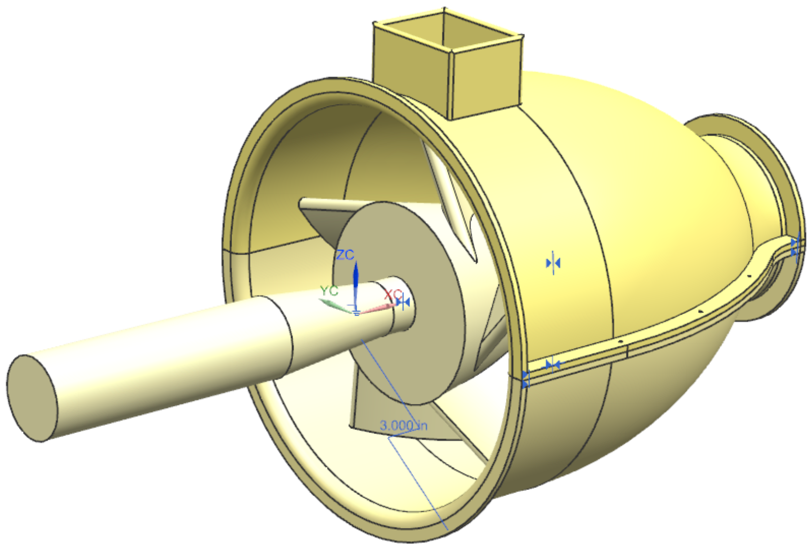


- Stay in the **Touch Align**  constraint
- Choose the **Prefer Touch** option in the **Geometry to Constrain** box
- First, select the face on the shaft and then select the bottom face of the hole in the impeller as shown





The assembly will now look like the figure below.

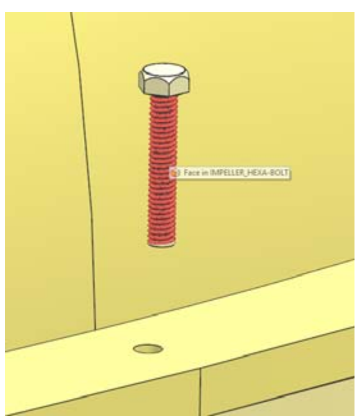


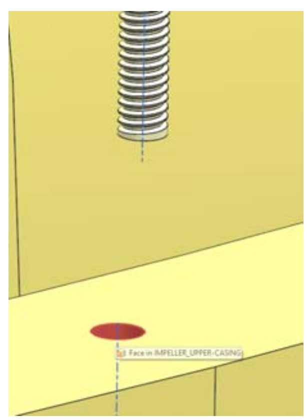
➢ Click on **Assemblies** →**Components** →**Add**

➢ Open the file **Impeller_hexa-bolt.prt**

➢ Choose the **Touch Align** constraint. Use the **Infer Center/Axis** option in the **Geometry to Constrain** box

➢ First, select the outer cylindrical threading on the bolt and then select the inner surface of the hole on the upper casing as show in the figures below.

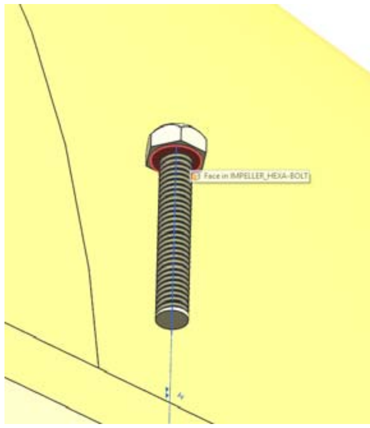


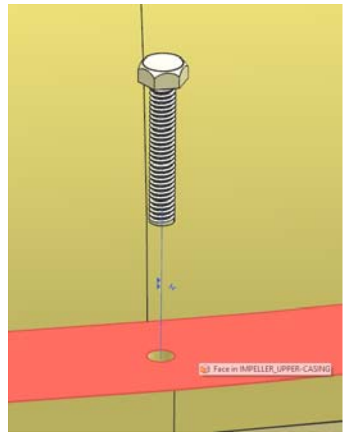


➢ Again in the **Touch Align** constraint change the **Geometry to Constrain** option to **Prefer Touch**

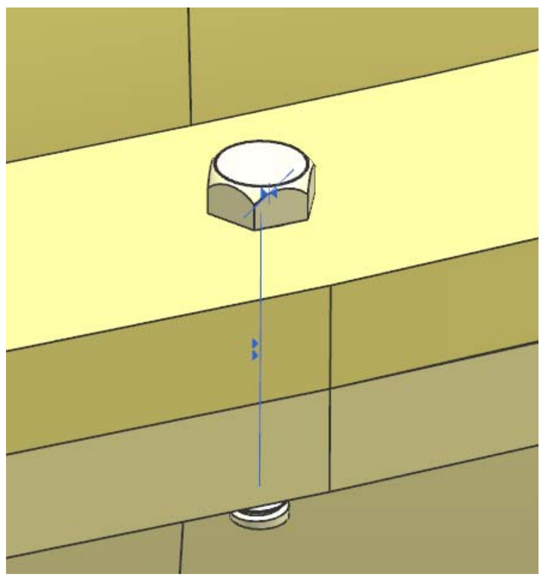
➢ Select the flat face on the bolt and the face on the rib of the upper casing as shown

➢ Click **OK**





The assembly is shown below.

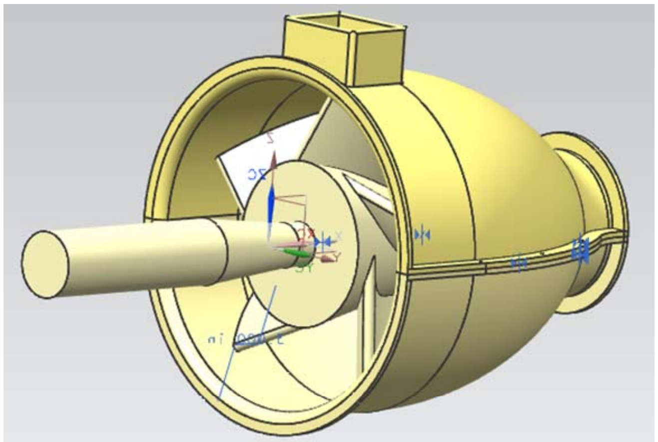


➢ Repeat the same procedure to add bolts and nuts to all the holes in the assembly.

This completes the assembly of the impeller.

Note: There is a simpler way to assemble the bolt and nut set. Instead of adding the three parts individually, you can assemble these components separately in another file. This will be a sub-assembly. You can insert this subassembly and mate it with the main assembly.

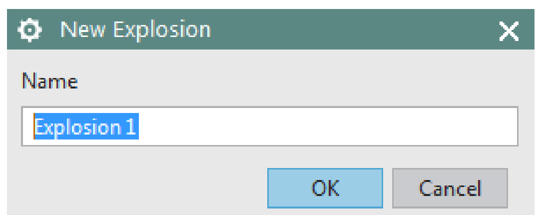
The Final Assembly will look as the shown below. Save the Model.



6.5.3 Exploded View

In this section, we are going to create an *Exploded View* of the assembly to show a separated part-by-part picture of the components that make the assembly. In today's industrial practice, these kind of views are very helpful on the assembly shop floor to get a good idea of which item fixes where. The user should understand that exploding an assembly does not mean relocation of the components, but only viewing the models in the form of disassembly. You can *Unexplode* the view at any time you want to regain the original assembly view. Let us explode the Impeller Assembly.

- Choose **Menu →Assemblies →Exploded Views →New Explosion**

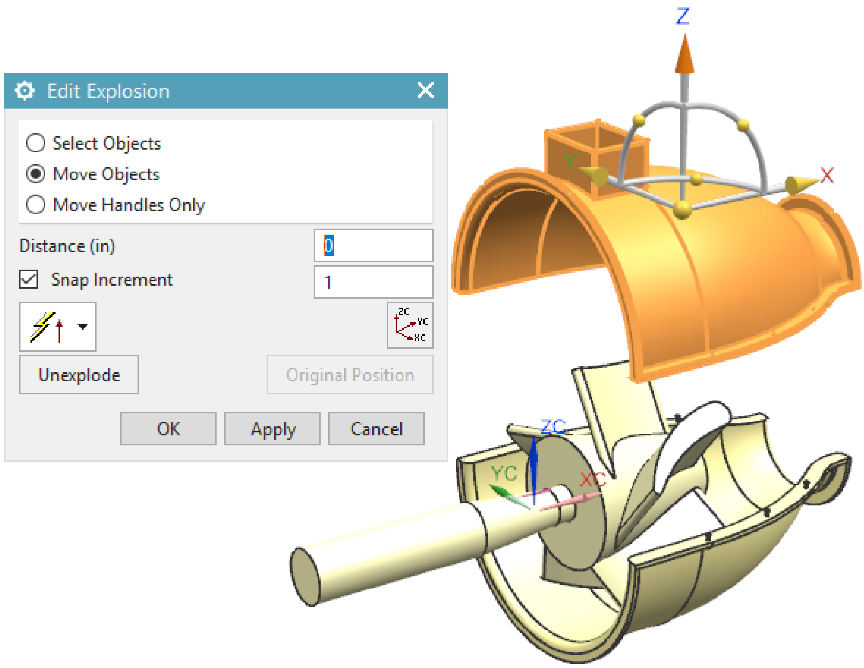


This will pop up a dialog box asking for the name of the *Explosion* view to be created. You can leave name as the default name and choose *OK*

Now the NX environment is in *Exploded View* environment though you do not find any difference. When we start exploding an assembly, we should decide upon a component to keep that component as the reference. This component should not be moved from its original position. In the case of the impeller assembly, the impeller will be the right option as it is central to the entire assembly. Now let us start exploding the components.

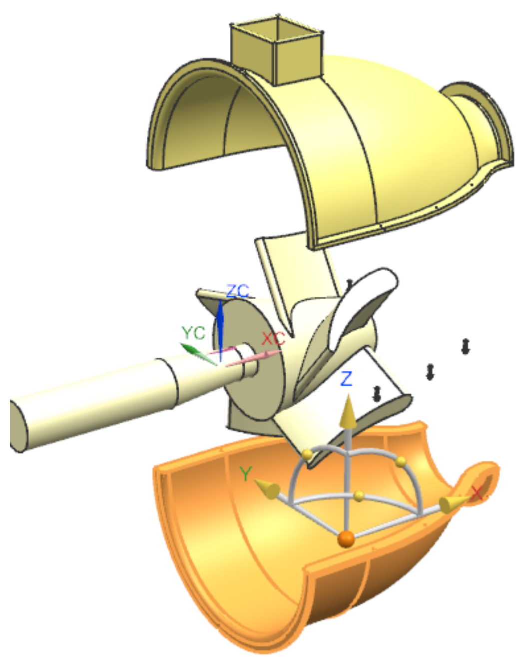
- Right click on the upper casing and choose **Edit Explosion**

The *Edit Explosion* window will pop up along with coordinate handles on the component.



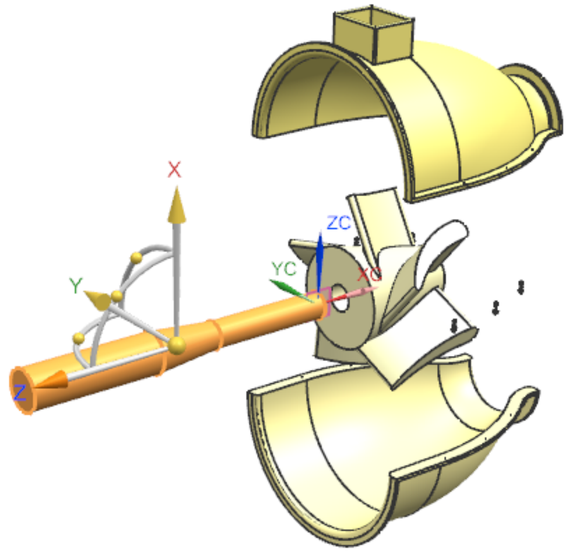
- Click on the **Z** axis; hold the mouse and drag upwards until the reading in the **Distance** shows **-20 (substitute +20 if you have designed in opposite direction)**
- Click **OK**
- Right click on the lower casing and choose **Edit Explosion**

Again, this will pop up a dialog window for *Edit Explosion* and a coordinate system on the component.



- Click on the **Z-axis**; hold the mouse and drag downwards until the reading in the Distance shows **20** as shown in the figure on the right side.
- Right click on the shaft and choose **Edit Explosion**
- This time click on the **X-axis**; hold the button and drag to the right side until the reading in the distance shows **-25** as shown in the following figure

➢ Choose **OK**

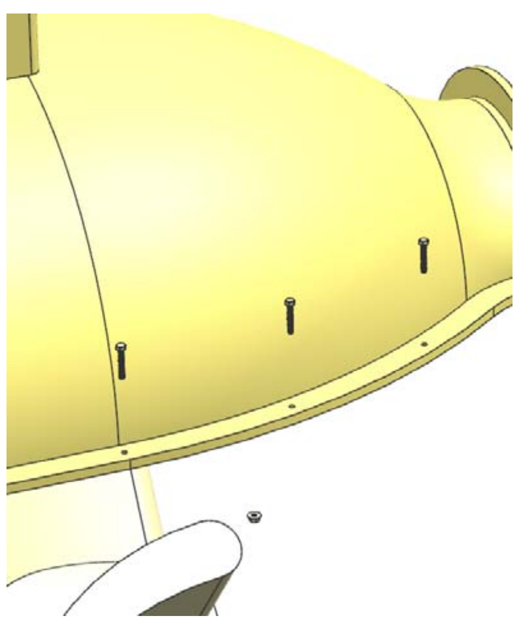


➢ Select all the six hexagonal bolts in the assembly by clicking on them

➢ Right click on one of them and choose **Edit Explosion**

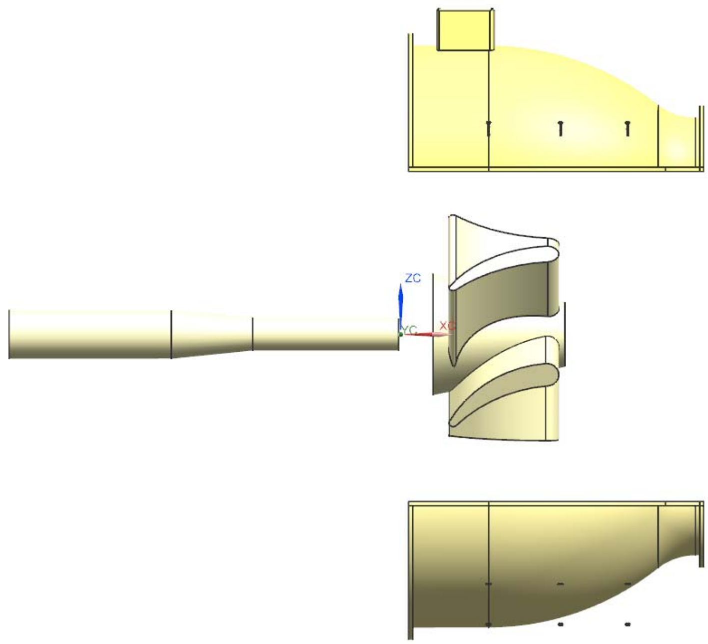
➢ This time click on the **Z-axis**; hold the button and drag upwards until the reading in the Distance shows **25** as shown in the following figure. This will move all the six bolts together to the same distance.

➢ Choose **OK**



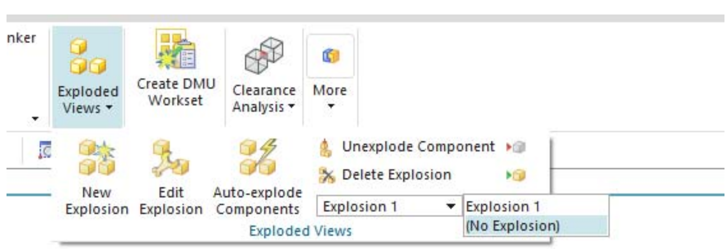
- ➢ Likewise, move the six washers and the six hexagonal nuts downwards to -30 and -35, respectively.

This is the Exploded view of the assembly. You can rotate and see how it looks like.



If you want to go back to the original unexploded view,

- ➢ Click **Exploded Views** in the ribbon → choose *(No Explosion)* from the drop-down menu as shown below



Or, you can *Unexplode* any component in the assembly,

- ➢ Right click on the component and choose **Unexplode**.

If you want to unexplode all the components,

- ➢ Choose **Assemblies** → **Exploded Views** → **Unexplode Component**

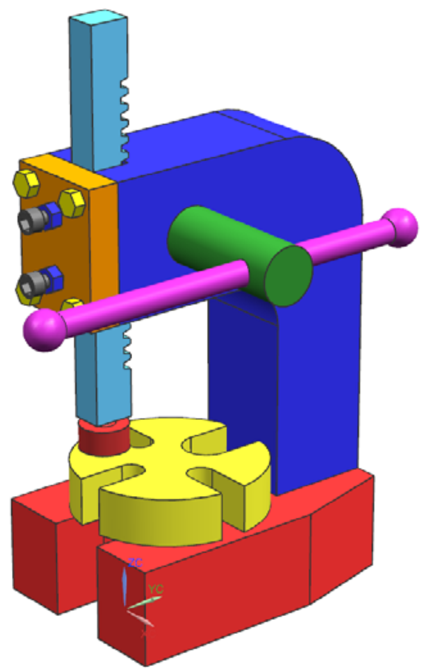
➢ Select all the components and choose **OK**

6.6 EXERCISES

6.6.1 Arbor Press

In the previous sections of this tutorial, we have modeled various parts, some of which are components of the arbor press shown below. Assemble the arbor press using the components that you have modeled in addition to ones that are provided to you, which you have not modeled before. The complete list of parts of the arbor press assembly is provided below. All these parts are provided in a folder that can be accessed along with this tutorial in the same internet address (https://web.mst.edu/~mleu/).

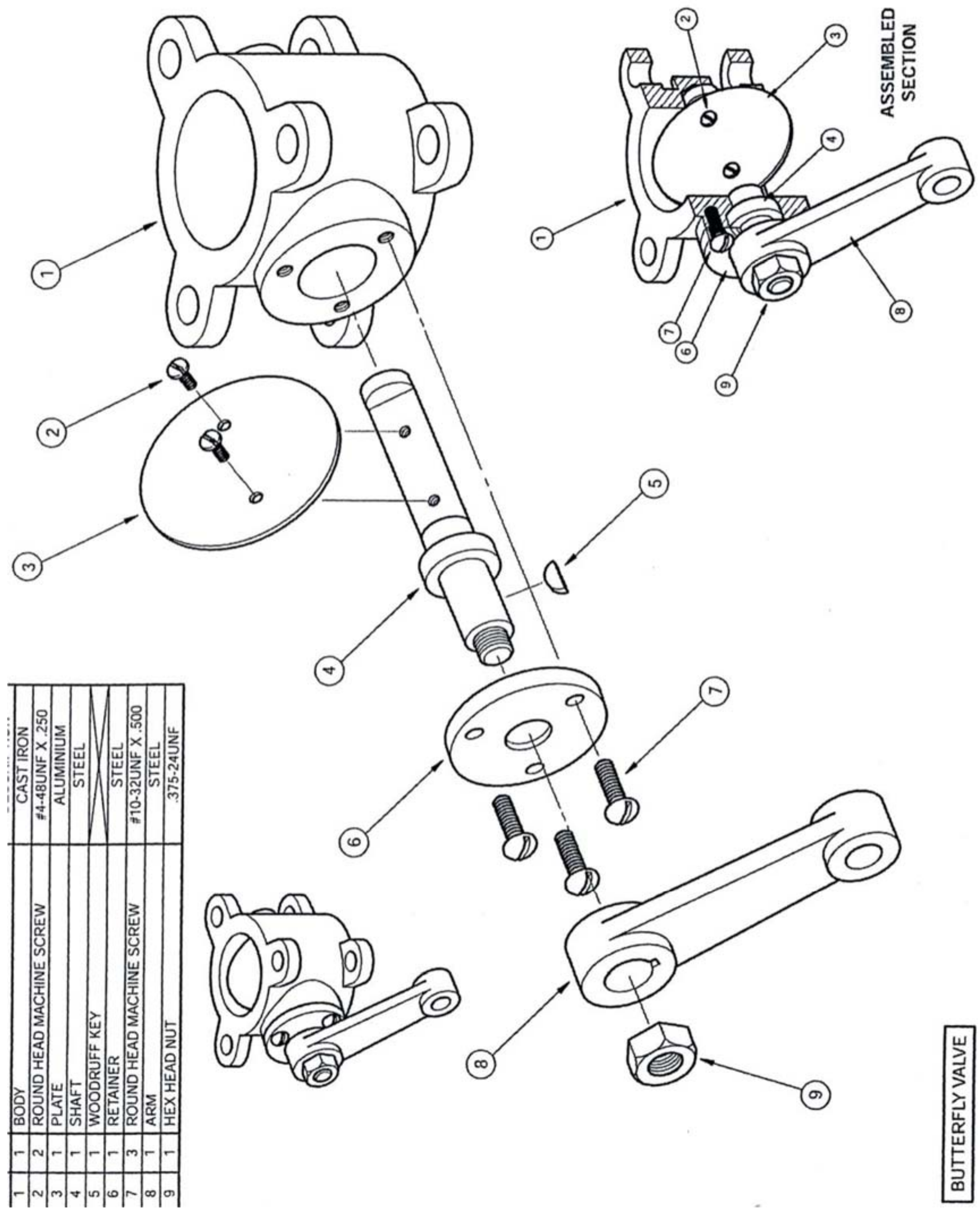
- Allen Bolt
- Allen Nut
- Base
- Circle base
- End clip
- Handle
- Hexagonal Bolt
- L-bar
- Pin
- Pinion
- Pinion handle
- Plate
- Rack
- Sleeve

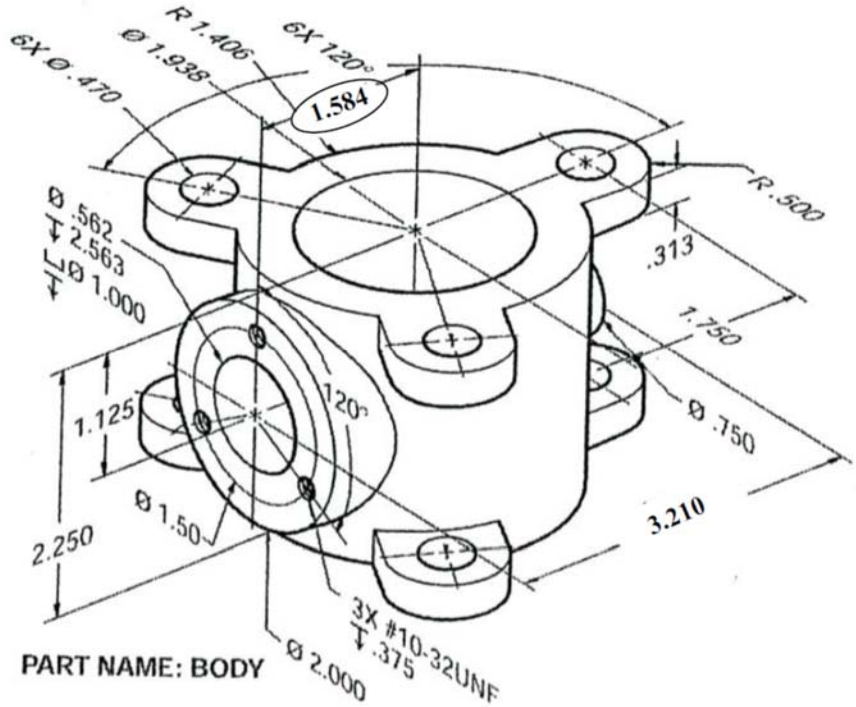


6.6.2 Butterfly Valve

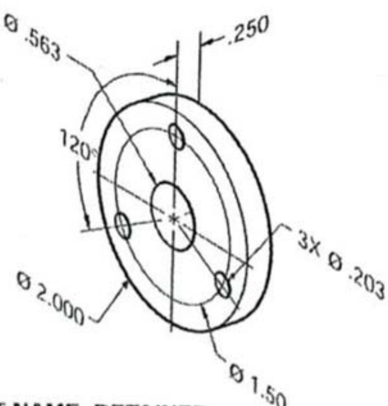
Butterfly valve is one of the most commonly used devices for controlling flow. The Butterfly valve consists of a rotating disk positioned in the pipe. The disc is attached to the shaft that is connected to an actuator on the outside of the valve. Rotating the actuator turns the disc either parallel or perpendicular to the flow. When the valve is closed, the disc is turned so that it completely blocks off the passageway. When the valve is fully open, the disc is rotated a quarter turn so that it allows an almost unrestricted passage of the fluid. The valve may also be opened incrementally to regulate flow.

In this exercise, you will model and assemble the Butterfly valve consisting of 9 different parts for a total of 12 components. The names, drawings and dimensions of the different part items, as well as how the various components fit together, are provided. The parts for which the drawings are not given are to be designed or selected, for example, the round head machine screw. All dimensions are in inches.

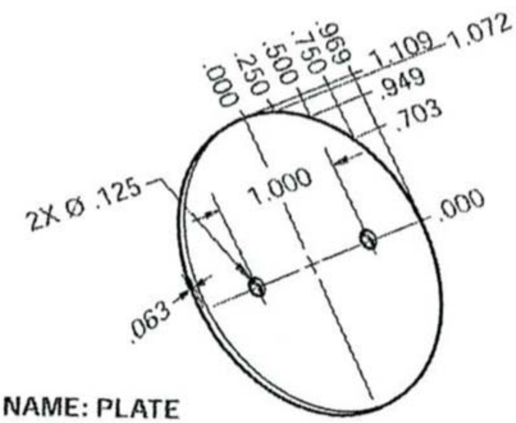




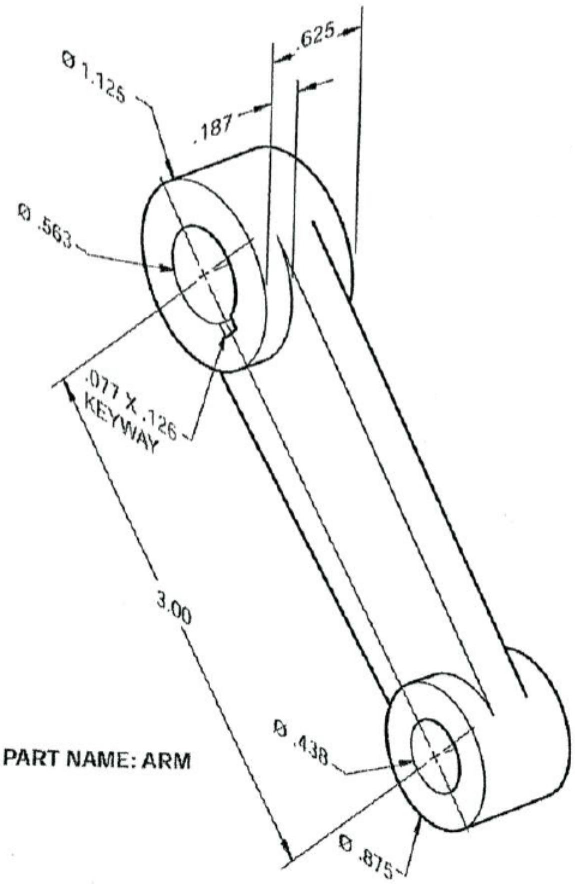
PART NAME: BODY



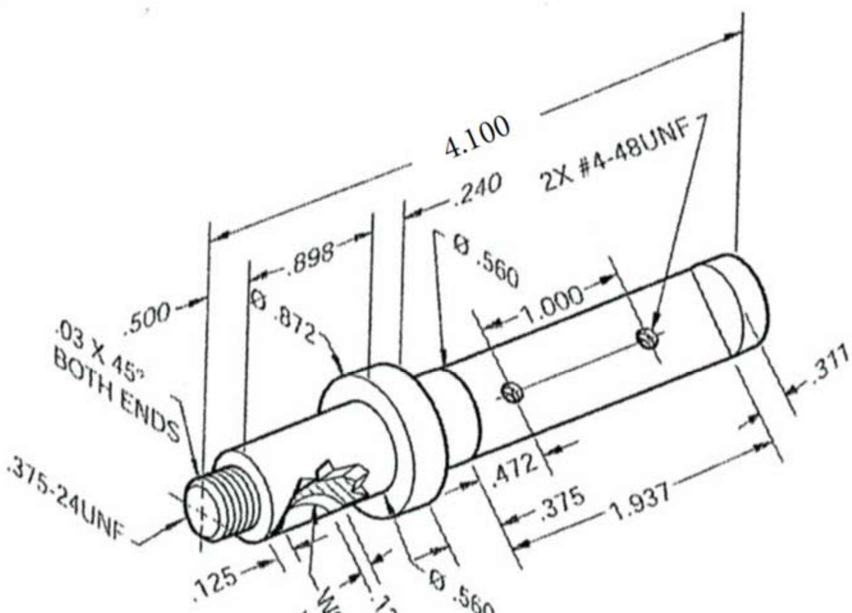
PART NAME: RETAINER



PART NAME: PLATE (ELLIPTICAL)



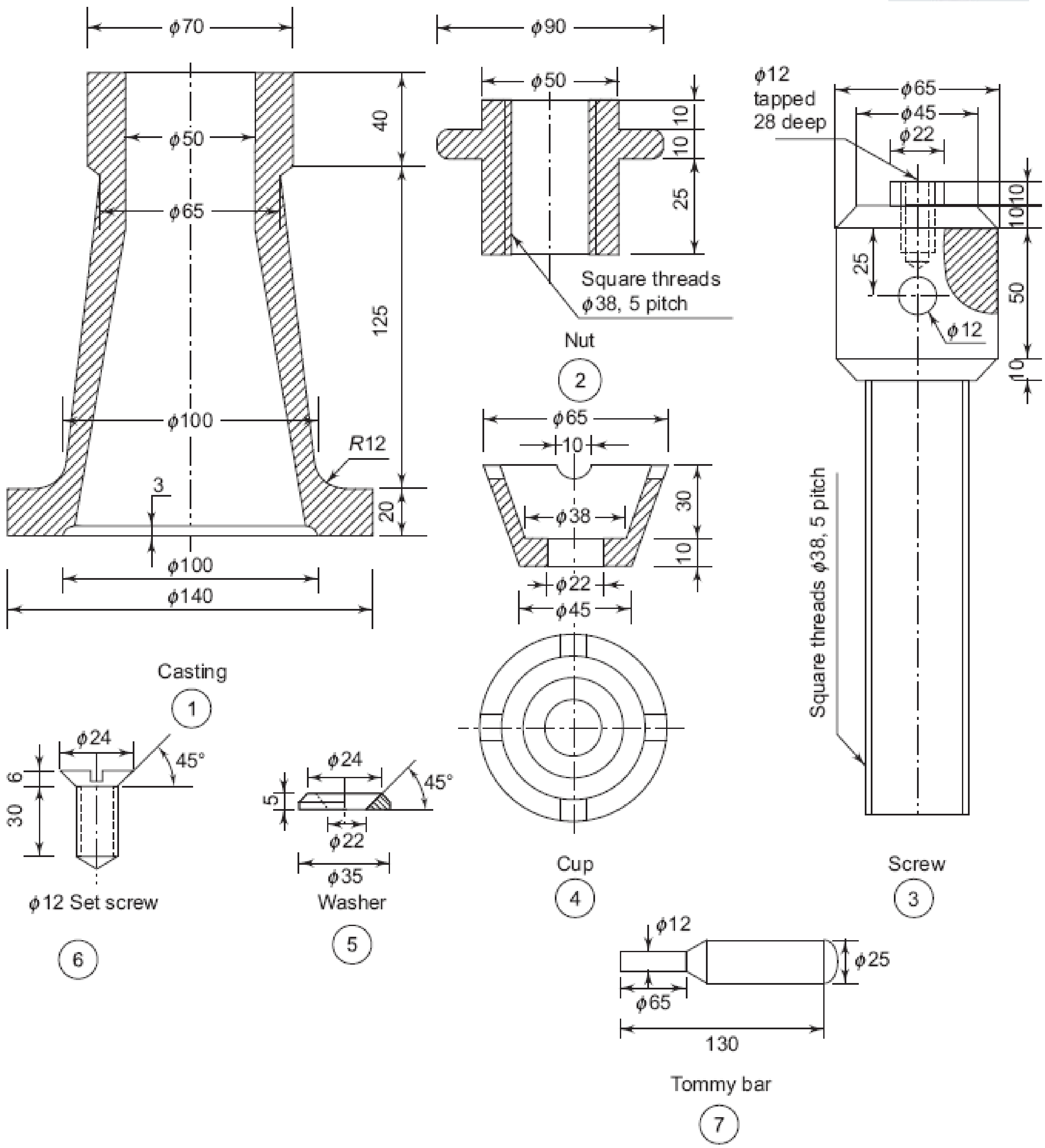
PART NAME: ARM

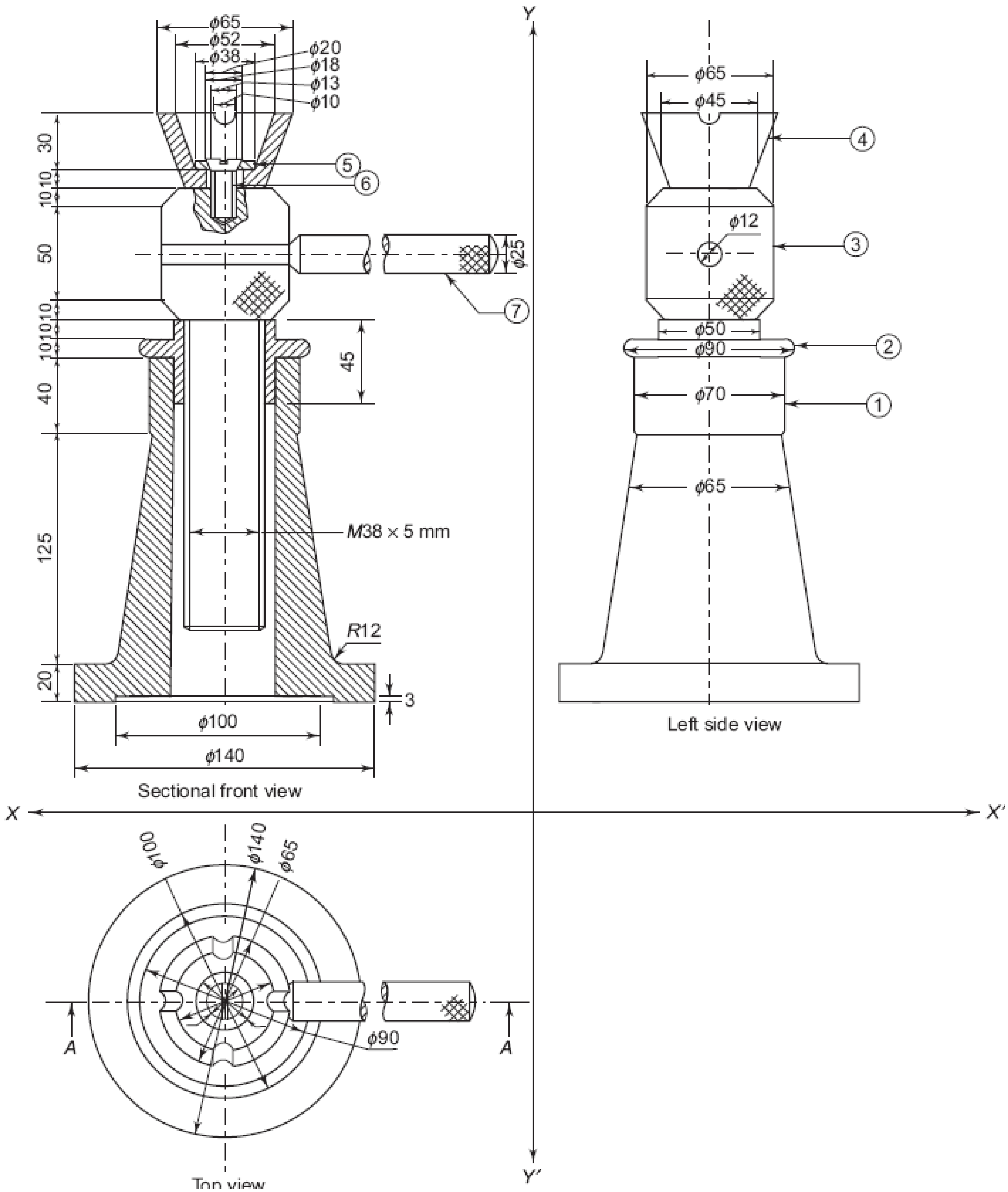


PART NAME: SHAFT

6.6.3 Jackscrew

A jackscrew is a type of jack that is operated by turning a leadscrew. In this exercise, you are asked to model, assemble and prepare drawings of parts. All dimensions should be in millimeters. Create individual drafts for each component. Draft the final assembly and make a table, listing out individual components. The Assembly draft should have an exploded view.





φ65
φ52
φ38
φ20
φ18
φ13
φ10
30
10
10
50
10
10
10
40
125
20
5
6
7
φ25
45
M38 × 5 mm
R12
3
φ100
φ140
Sectional front view
Y
Y′
X
X′
φ65
φ45
4
φ12
3
φ50
φ90
2
φ70
1
φ65
Left side view
φ100
φ140
φ65
φ90
A
A
Top view