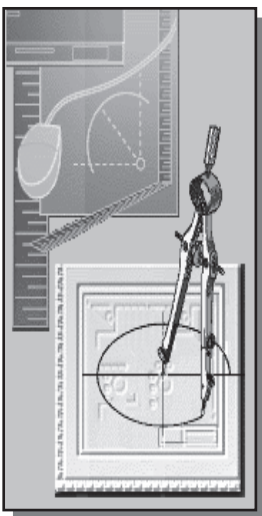
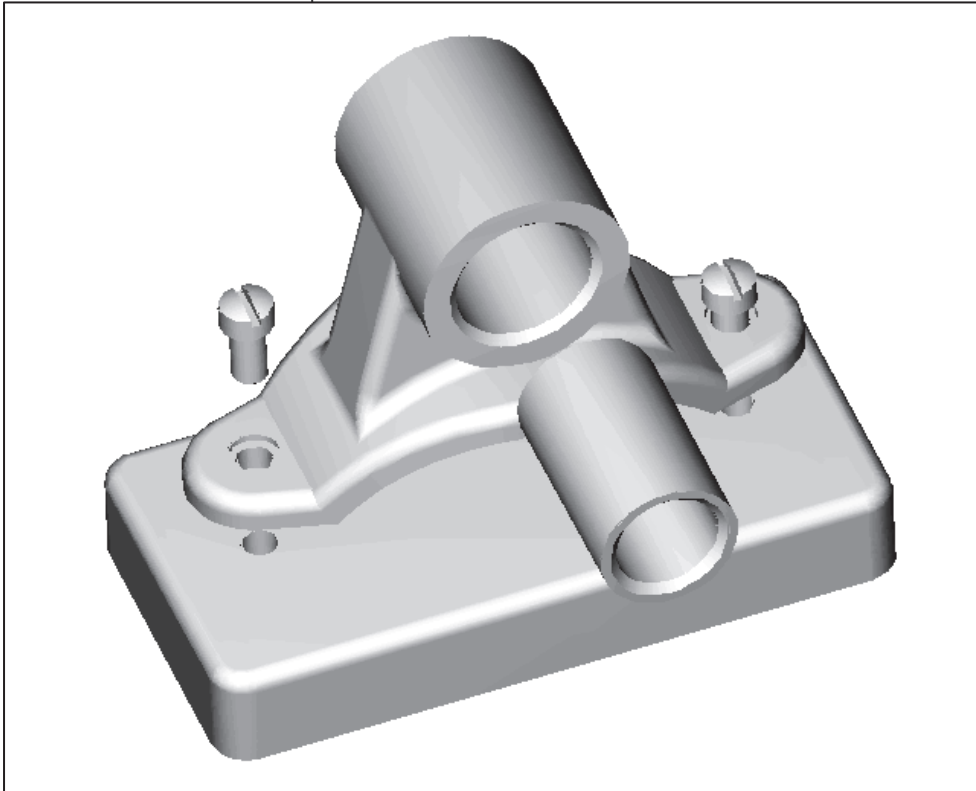


## Chapter 13

# Assembly Modeling – Putting It All Together



### Learning Objectives

- ◆ Understand the Assembly Modeling Methodology
- ◆ Create Parts in the Assembly Modeler Mode
- ◆ Understand and Utilize Assembly Constraints
- ◆ Understand the Autodesk Inventor DOF Display
- ◆ Utilize the Autodesk Inventor Adaptive Design Approach
- ◆ Create Exploded Assemblies

## Autodesk Inventor Certified Associate Exam Objectives Coverage

### Section 5: Placing, Creating, and Constraining Components

Objectives: Describe how *Assembly Constraints* affect individual parts in the assembly; Describe the *Content Center* consumer environment and what a *Content Center* consumer can do; Basic *Part Design* in an Assembly.

<b>Place Component</b> .....	13-9
<b>Degrees of Freedom</b> .....	13-11
<b>Assembly Constraints</b> .....	13-12
<b>Mate Constraint</b> .....	13-14
<b>Flush Constraint</b> .....	13-18
<b>Insert Constraint</b> .....	13-20
<b>Create Component</b> .....	13-26
<b>Adaptive</b> .....	13-28
<b>Delete Constraints</b> .....	13-30

### Section 6: Interacting with an Assembly

Objectives: Use the selection *Priority* and *Component Selection* tools to simplify and enhance component selection; Calculate physical properties; Describe *Presentation Files* and their role in creating exploded *Drawing Views*.

<b>Assembly Modeling Methodology</b> .....	13-4
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<b>Constrained Move</b> .....	13-17
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### Section 8: Dimensioning, Annotations and Tables

Objectives: Describe the use and function of *Retrieved Dimensions*; Create and edit *Hole Tables*; Create *Centerline* and *Center Marks*; Add and edit *Revision Tags*.

<b>Parts List</b> .....	13-34
<b>Edit Parts List</b> .....	13-35
<b>iProperties</b> .....	13-37
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### Section 9: Annotating Assembly Drawings

Objectives: Describe the *Bill of Materials* and its uses; Use the *Edit Parts List* dialog box to modify your *Parts Lists*.

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<b>Export Bill of Materials</b> .....	13-41

## Introduction

In the previous lessons, we have gone over the fundamentals of creating basic parts and drawings. In this lesson, we will examine the assembly modeling functionality of *Autodesk Inventor*. We will start with a demonstration on how to create and modify assembly models. The main task in creating an assembly is establishing the assembly relationships between parts. To assemble parts into an assembly, we will need to consider the assembly relationships between parts. It is a good practice to assemble parts based on the way they would be assembled in the actual manufacturing process. We should also consider breaking down the assembly into smaller subassemblies, which helps the management of parts. In *Autodesk Inventor*, a subassembly is treated the same way as a single part during assembling. Many parallels exist between assembly modeling and part modeling in parametric modeling software such as *Autodesk Inventor*.

*Autodesk Inventor* provides full associative functionality in all design modules, including assemblies. When we change a part model, *Autodesk Inventor* will automatically reflect the changes in all assemblies that use the part. We can also modify a part in an assembly. **Bi-directional full associative functionality** is the main feature of parametric solid modeling software that allows us to increase productivity by reducing design cycle time.

One of the key features of *Autodesk Inventor* is the use of an assembly-centric paradigm, which enables users to concentrate on the design without depending on the associated parameters or constraints. Users can specify how parts fit together and the *Autodesk Inventor assembly-based fit function* automatically determines the parts' sizes and positions. This unique approach is known as the **Direct Adaptive Assembly** approach, which defines part relationships directly with no order dependency.

In this lesson, we will also illustrate the basic concept of *Autodesk Inventor's Adaptive Design* approach. The key element in doing **Adaptive Design** is to *under-constrain* features or parts. The applied *assembly constraints* in the assembly modeler are used to control the sizes, shapes, and positions of *under-constrained* sketches, features, and parts. No equations are required and this approach is extremely flexible when performing modifications and changes to the design. We can modify adaptive assemblies at any point, in any order, regardless of how the parts were originally placed or constrained.

In *Autodesk Inventor*, features and parts can be made adaptive at any time during the creation or assembly. The features of a part can be defined as adaptive when they are created in the part file. When we place such a part in an assembly, the features will then resize and change shape based on the applied assembly constraints. We can make features and parts adaptive from either the part modeling or assembly modeling environments.

The *Adaptive Design approach* is a unique design methodology that can only be found in *Autodesk Inventor*. The goal of this methodology is to improve the design process and allows you, the designer, to *design the way you think*.

## Assembly Modeling Methodology

The *Autodesk Inventor* assembly modeler provides tools and functions that allow us to create 3D parametric assembly models. An assembly model is a 3D model with any combination of multiple part models. *Parametric assembly constraints* can be used to control relationships between parts in an assembly model.

*Autodesk Inventor* can work with any of the assembly modeling methodologies:

### The Bottom Up Approach

The first step in the *bottom up* assembly modeling approach is to create the individual parts. The parts are then pulled together into an assembly. This approach is typically used for smaller projects with very few team members.

### The Top Down Approach

The first step in the *top down* assembly modeling approach is to create the assembly model of the project. Initially, individual parts are represented by names or symbols. The details of the individual parts are added as the project gets further along. This approach is typically used for larger projects or during the conceptual design stage. Members of the project team can then concentrate on the particular section of the project to which he/she is assigned.

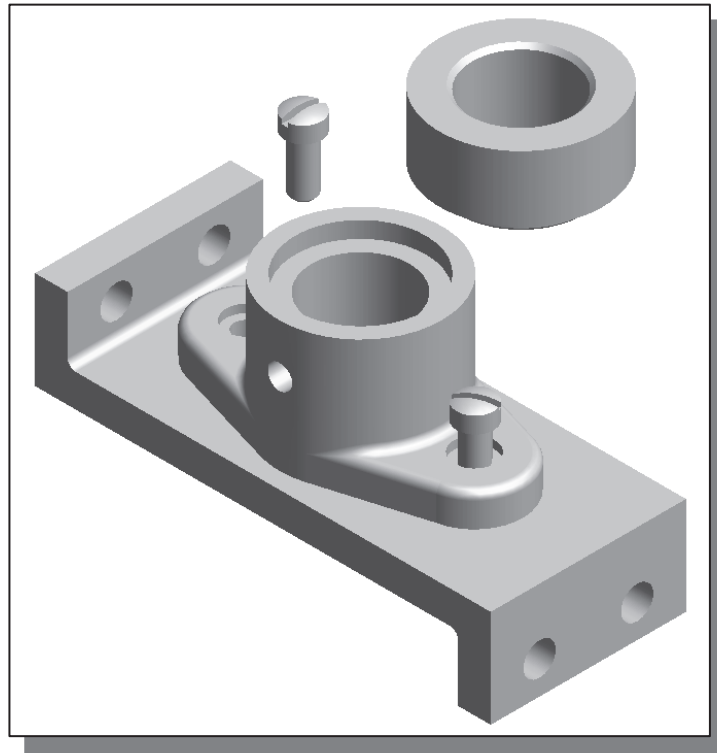
### The Middle Out Approach

The *middle out* assembly modeling approach is a mixture of the bottom-up and top-down methods. This type of assembly model is usually constructed with most of the parts already created, and additional parts are designed and created using the assembly for construction information. Some requirements are known and some standard components are used, but new designs must also be produced to meet specific objectives. This combined strategy is a very flexible approach for creating assembly models.

The different assembly modeling approaches described above can be used as guidelines to manage design projects. Keep in mind that we can start modeling our assembly using one approach and then switch to a different approach without any problems.

In this lesson, the *bottom up* assembly modeling approach is illustrated. All of the parts (components) required to form the assembly are created first. *Autodesk Inventor's* assembly modeling tools allow us to create complex assemblies by using components that are created in part files or are placed in assembly files. A component can be a subassembly or a single part, where features and parts can be modified at any time. The sketches and profiles used to build part features can be fully or partially constrained. Partially constrained features may be adaptive, which means the size or shape of the associated parts are adjusted in an assembly when the parts are constrained to other parts. The basic concept and procedure of using the adaptive assembly approach is demonstrated in the tutorial.

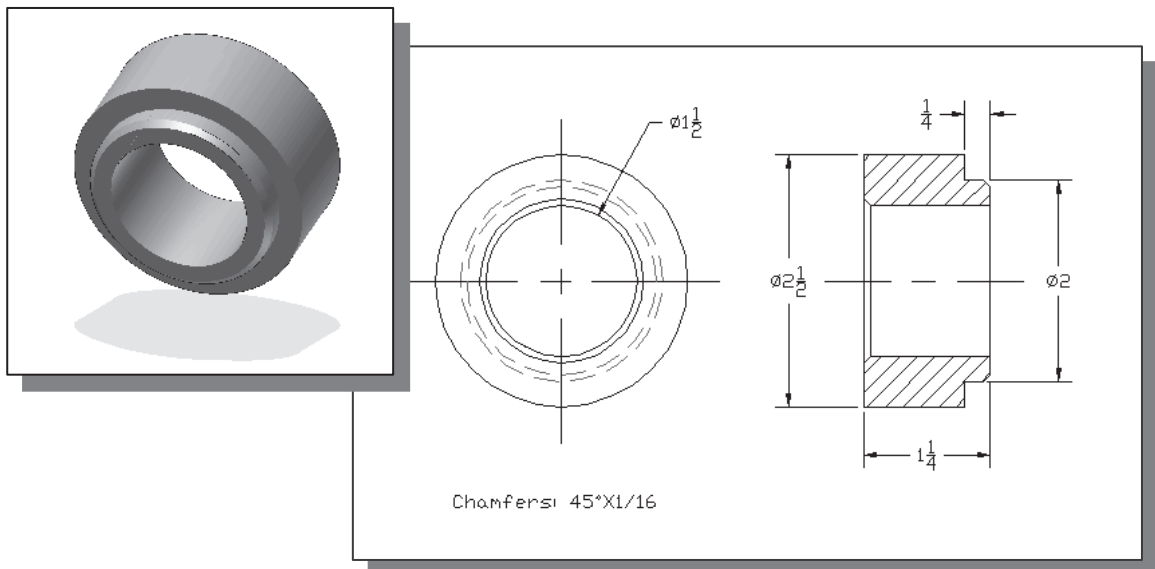
## The Shaft Support Assembly



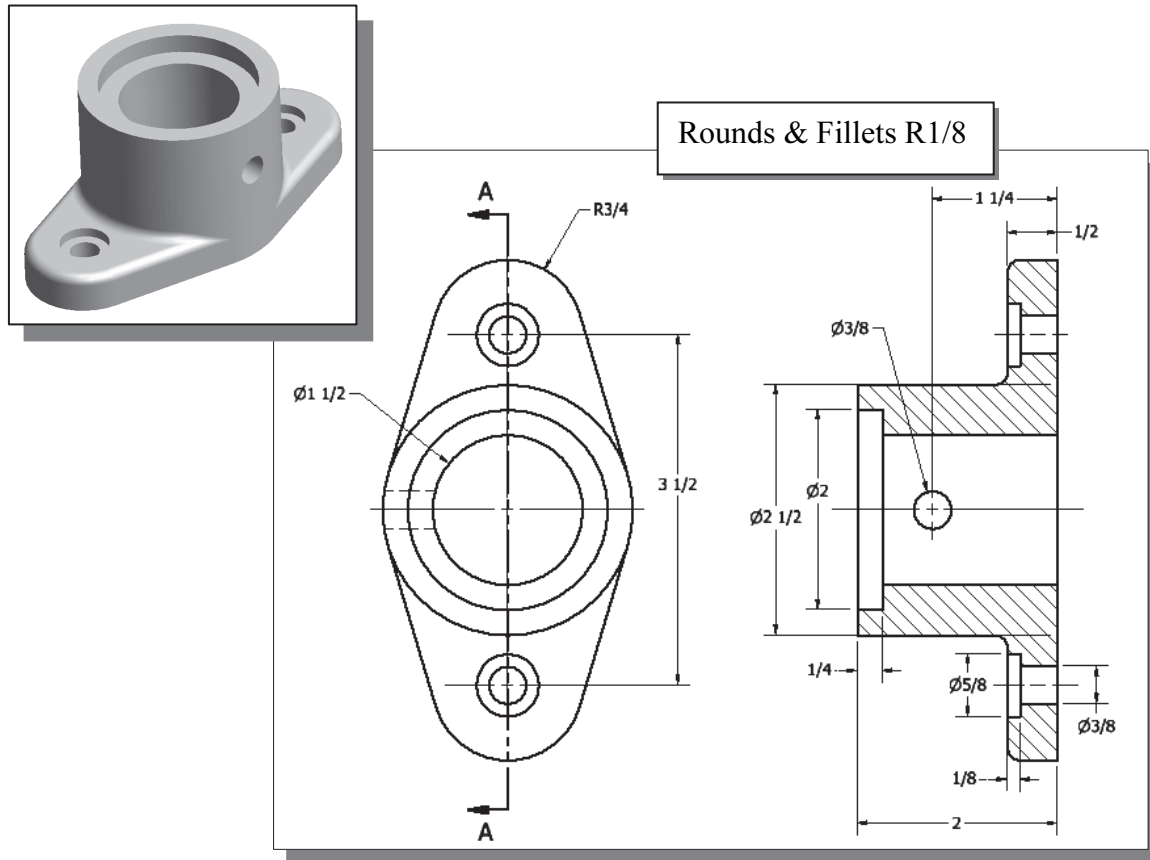
### Additional Parts

- Four parts are required for the assembly: (1) **Collar**, (2) **Bearing**, (3) **Base-Plate** and (4) **Cap-Screw**. On your own, create the four parts as shown below; save the models as separate part files: *Collar*, *Bearing*, *Base-Plate*, and *Cap-Screw*. (Place all parts in the **Chapter13** folder and close all part files or exit *Autodesk Inventor* after you have created the parts.)

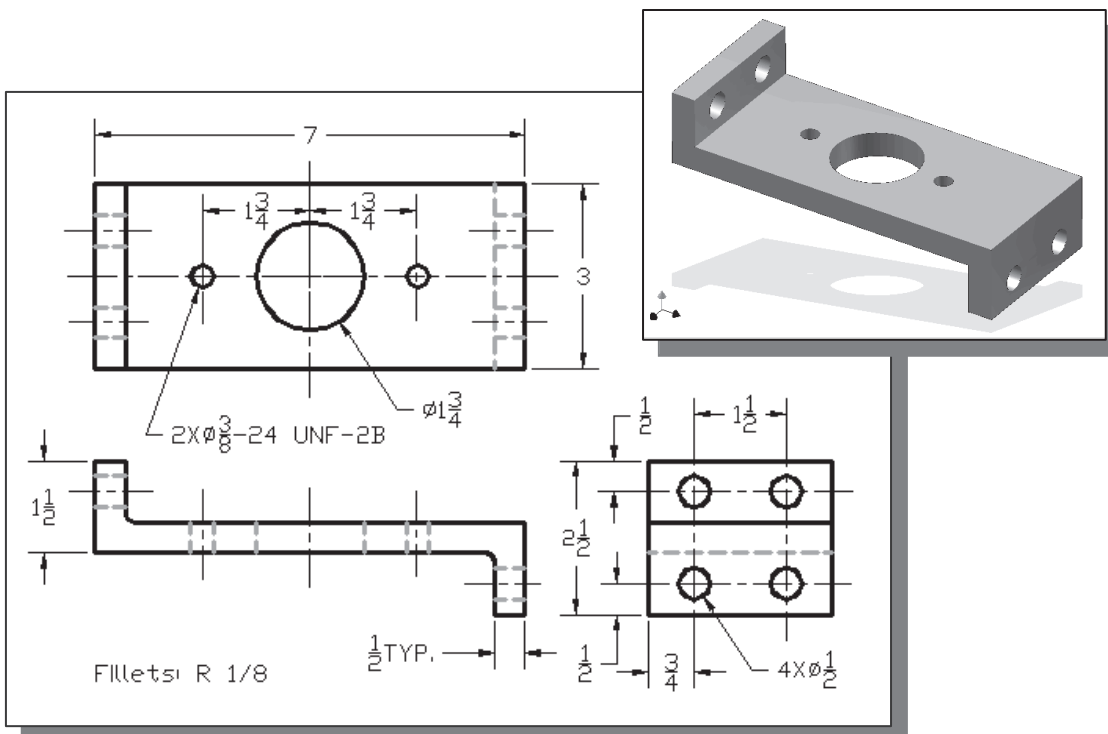
#### (1) **Collar**



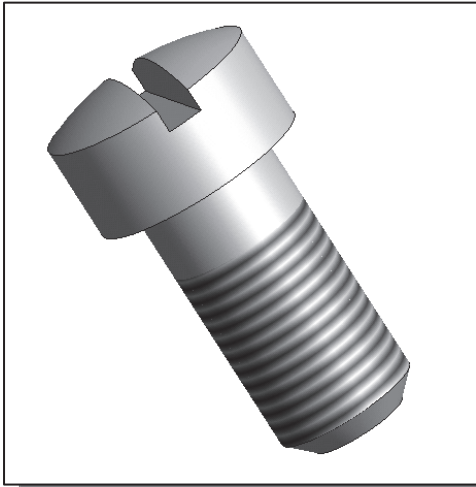
(2) **Bearing** (Construct the part with the datum origin aligned to the bottom center.)



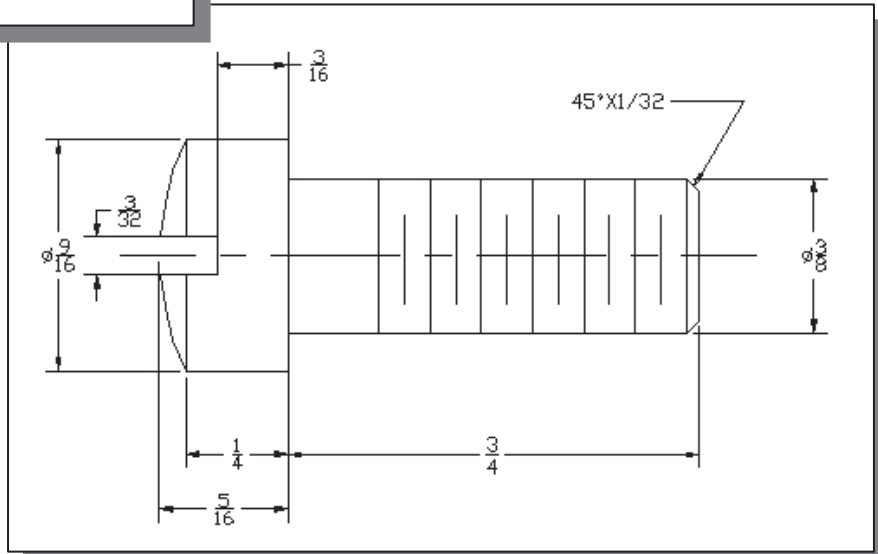
(3) **Base-Plate** (Construct the part with the datum origin aligned to the bottom center of the large hole.)



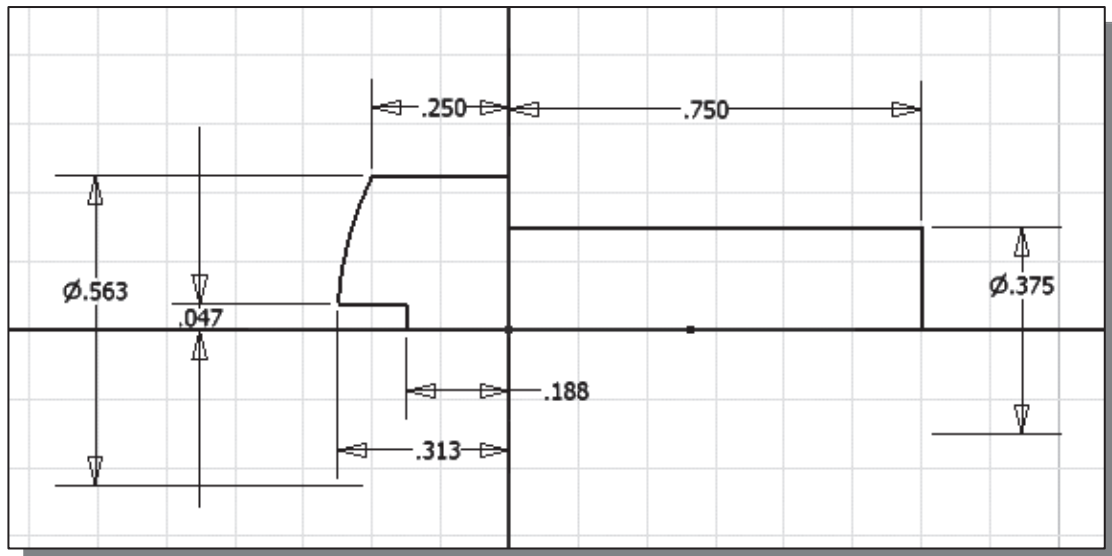
(4) **Cap-Screw**



- *Autodesk Inventor* provides two options for creating threads: **Thread** and **Coil**. The **Thread** command does not create true 3D threads; a pre-defined thread image is applied on the selected surface, as shown in the figure. The **Coil** command can be used to create true threads, which contain complex three-dimensional curves and surfaces. You are encouraged to experiment with the **Coil** command and/or the **Thread** command to create threads.

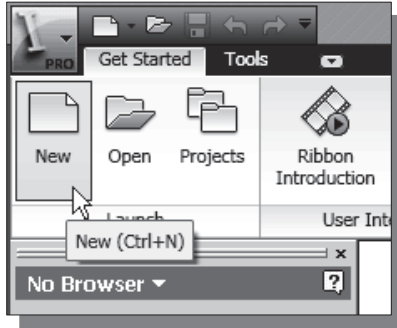


- Hint: First create a revolved feature using the profile shown below.



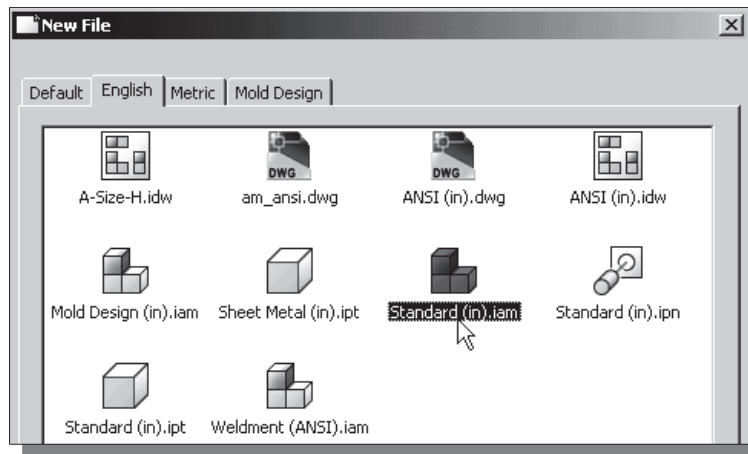
## Starting Autodesk Inventor

1. Select the **Autodesk Inventor** option on the *Start* menu or select the **Autodesk Inventor** icon on the desktop to start *Autodesk Inventor*. The *Autodesk Inventor* main window will appear on the screen.

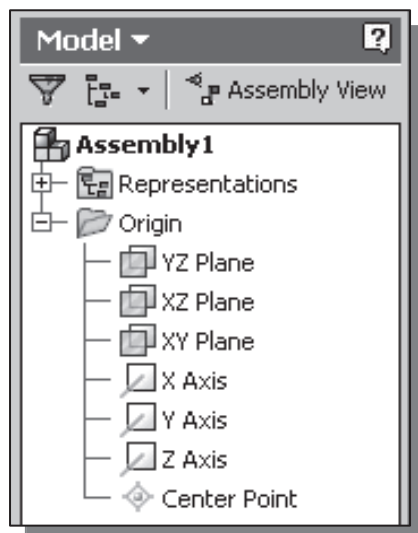


2. Select the **New File** icon with a single click of the left-mouse-button in the *Launch* toolbar as shown.
3. Confirm the **Parametric-Modeling-Exercises** project is activated; note the **Projects** button is available to view/modify the active project.

4. Select the **English** tab and in the *Template* list, select **Standard(in).iam** (*Standard Inventor Assembly Model* template file).



5. Click on the **OK** button in the *New File* dialog box to accept the selected settings.

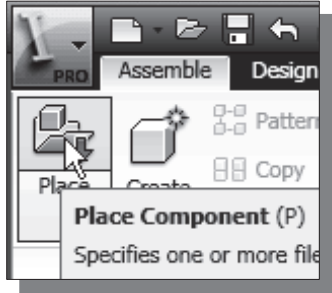


- In the *browser* window, **Assembly1** is displayed with a set of work planes, work axes and a work point. In most aspects, the usage of work planes, work axes and work point is very similar to that of the *Inventor Part Modeler*.
- Notice, in the *Ribbon* toolbar panels, several *component* options are available, such as **Place Component**, **Create Component** and **Place from Content Center**. As the names imply, we can use parts that have been created or create new parts within the *Inventor Assembly Modeler*.

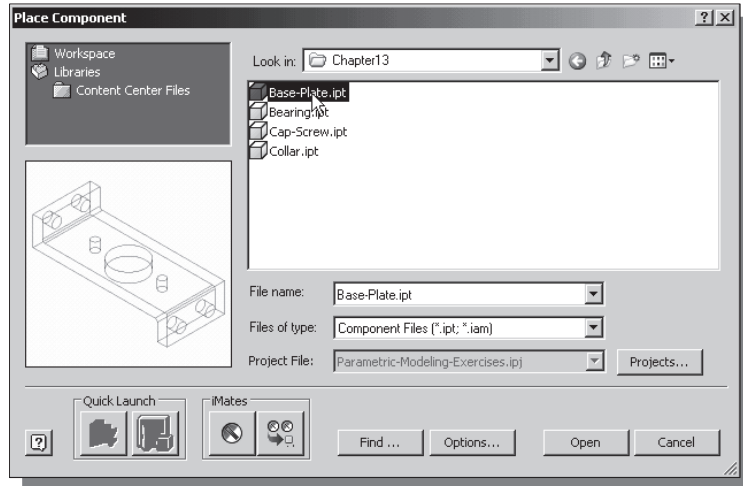


## Placing the First Component

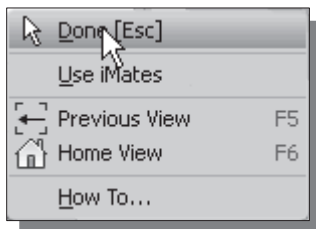
- The first component placed in an assembly should be a fundamental part or subassembly. The first component in an assembly file sets the orientation of all subsequent parts and subassemblies. The origin of the first component is aligned to the origin of the assembly coordinates and the part is grounded (all degrees of freedom are removed). The rest of the assembly is built on the first component, the **base component**. In most cases, this *base component* should be one that is **not likely to be removed** and **preferably a non-moving part** in the design. Note that there is no distinction in an assembly between components; the first component we place is usually considered as the *base component* because it is usually a fundamental component to which others are constrained. We can change the base component to a different base component by placing a new base component, specifying it as grounded, and then re-constraining any components placed earlier, including the first component. For our project, we will use the **Base-Plate** as the base component in the assembly.



1. In the *Assembly* panel (the toolbar that is located to the left side of the graphics window) select the **Place Component** command by left-mouse-clicking the icon.
2. Select the **Base-Plate** (part file: *Base-Plate.ipt*) in the list window.



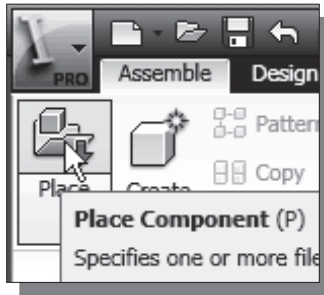
3. Click on the **Open** button to retrieve the model.



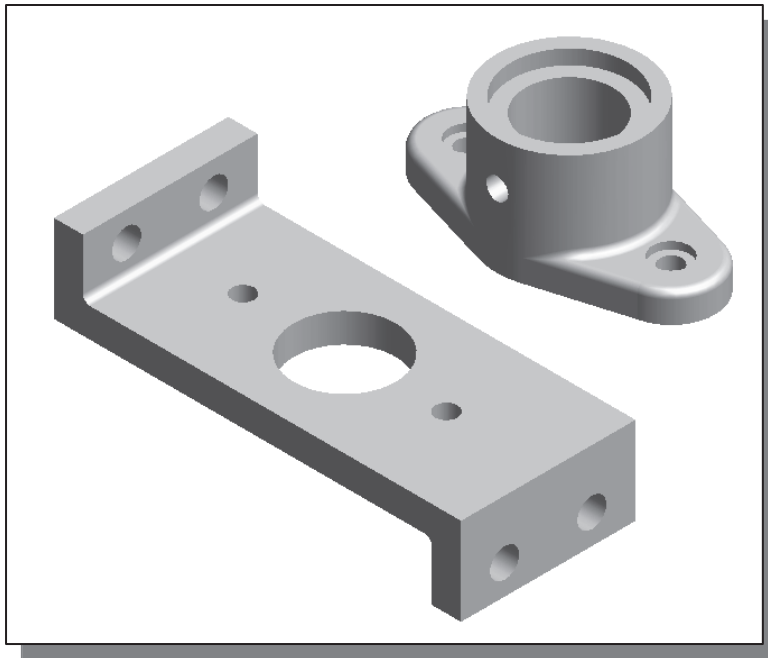
4. By default, the first component is automatically aligned to the origin of the assembly coordinates. We can also place multiple copies of the same component. Right-mouse-click once to bring up the option menu and select **Done** to end the placement of the *Base-Plate* part.

## Placing the Second Component

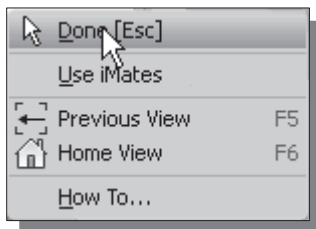
➤ We will retrieve the *Bearing* part as the second component of the assembly model.



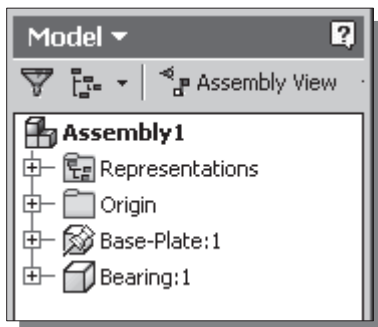
1. In the *Assembly* panel (the toolbar that is located to the left side of the graphics window) select the **Place Component** command by left-mouse-clicking the icon.
2. Select the ***Bearing*** design (part file: *Bearing.ipt*) in the list window. And click on the **Open** button to retrieve the model.



3. Place the *Bearing* toward the upper right corner of the graphics window, as shown in the figure.



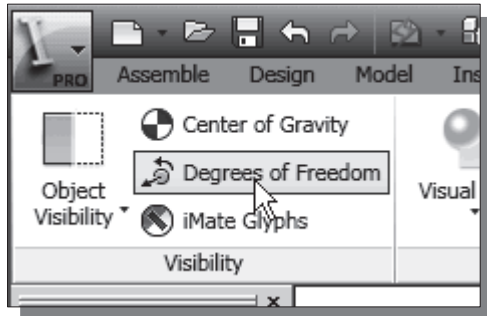
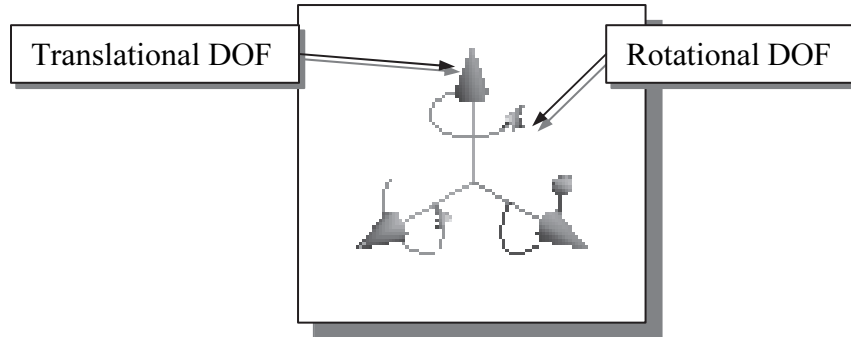
4. Inside the graphics window, right-mouse-click once to bring up the option menu and select **Done** to end the placement of the *Bearing* part.



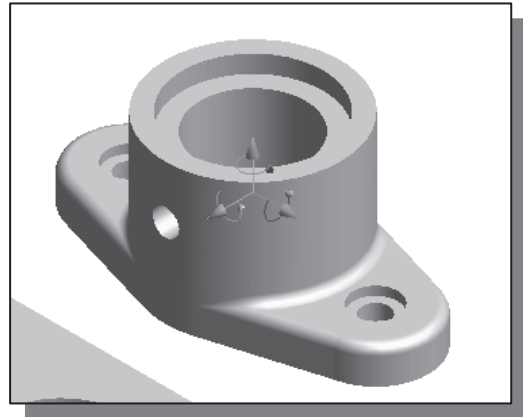
- Inside the *browser* window, the retrieved parts are listed in their corresponding order. The **Pin** icon in front of the *Base-Plate* filename signifies the part is grounded and all **six degrees of freedom** are restricted. The number behind the filename is used to identify the number of copies of the same component in the assembly model.

## Degrees of Freedom and Constraints

- Each component in an assembly has six **degrees of freedom (DOF)**, or ways in which rigid 3D bodies can move: movement along the X, Y, and Z axes (translational freedom), plus rotation around the X, Y, and Z axes (rotational freedom). *Translational DOFs* allow the part to move in the direction of the specified vector. *Rotational DOFs* allow the part to turn about the specified axis.



- Select the **Degrees of Freedom** option in the **View** tab to display the DOF of the unconstrained component.

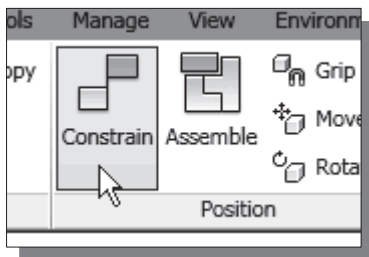


- In *Autodesk Inventor*, the degrees-of-freedom symbol shows the remaining degrees of freedom (both translational and rotational) for one or more components of the active assembly. When a component is fully constrained in an assembly, the component cannot move in any direction. The position of the component is fixed relative to other assembly components. All of its degrees of freedom are removed. When we place an assembly constraint between two selected components, they are positioned relative to one another. Movement is still possible in the unconstrained directions.
- It is usually a good idea to fully constrain components so that their behavior is predictable as changes are made to the assembly. Leaving some degrees of freedom open can sometimes help retain design flexibility. As a general rule, we should use only enough constraints to ensure predictable assembly behavior and avoid unnecessary complexity.

## Assembly Constraints

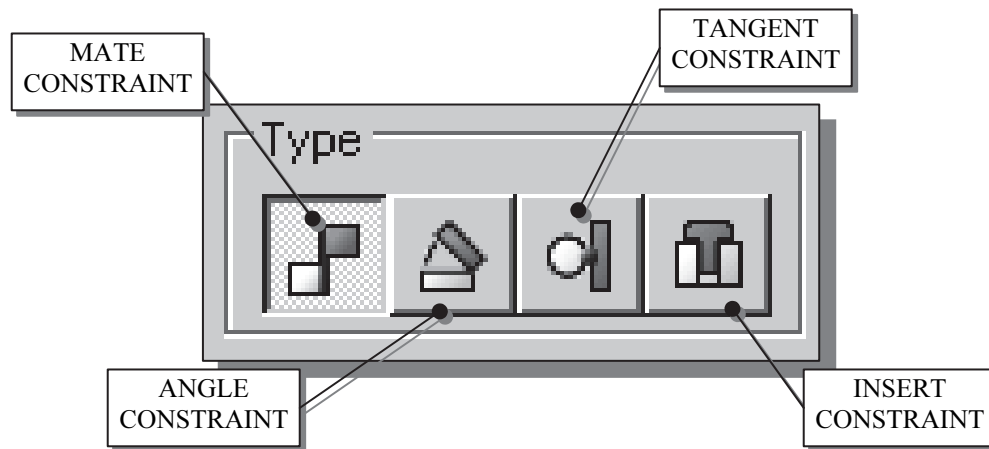
- We are now ready to assemble the components together. We will start by placing assembly constraints on the **Bearing** and the **Base-Plate**.

To assemble components into an assembly, we need to establish the assembly relationships between components. It is a good practice to assemble components the way they would be assembled in the actual manufacturing process. **Assembly constraints** create a parent/child relationship that allows us to capture the design intent of the assembly. Because the component that we are placing actually becomes a child to the already assembled components, we must use caution when choosing constraint types and references to make sure they reflect the intent.



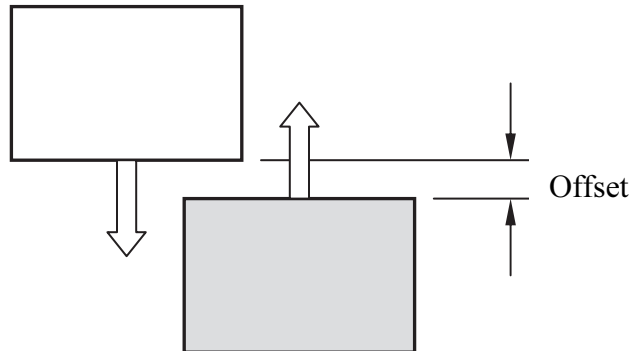
- Switch back to the *Assemble* panel; select the **Constrain** command by left-mouse-clicking once on the icon.

- The *Place Constraints* dialog box appears on the screen. Four types of assembly constraints are available.

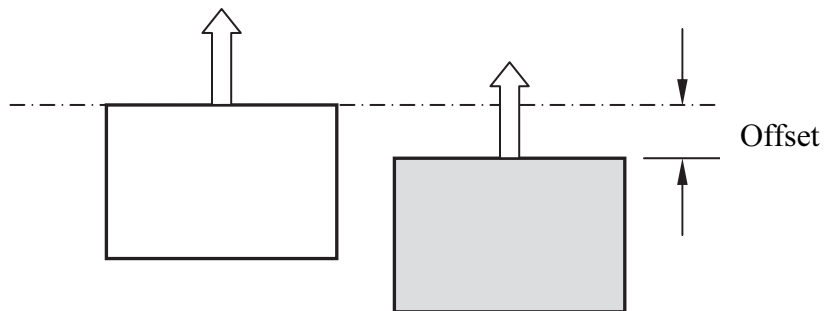


- Assembly models are created by applying proper *assembly constraints* to the individual components. The constraints are used to restrict the movement between parts. Constraints eliminate rigid body degrees of freedom (**DOF**). A 3D part has *six degrees of freedom* since the part can rotate and translate relative to the three coordinate axes. Each time we add a constraint between two parts; one or more DOF is eliminated. The movement of a fully constrained part is restricted in all directions. Four basic types of assembly constraints are available in *Autodesk Inventor*: **Mate**, **Angle**, **Tangent** and **Insert**. Each type of constraint removes different combinations of rigid body degrees of freedom. Note that it is possible to apply different constraints and achieve the same results.

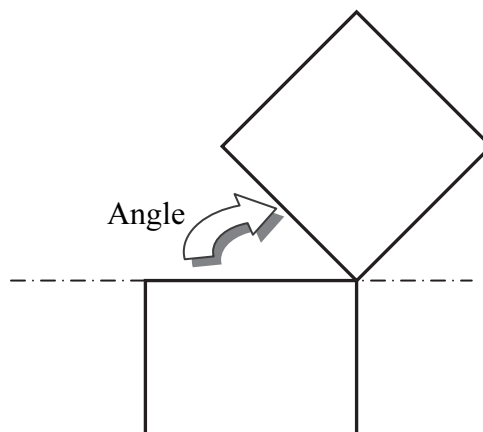
- **Mate** – Constraint positions components face-to-face, or adjacent to one another, with faces flush. Removes one degree of linear translation and two degrees of angular rotation between planar surfaces. Selected surfaces point in opposite directions and can be **offset** by a specified distance. **Mate** constraint positions selected faces normal to one another, with faces coincident.



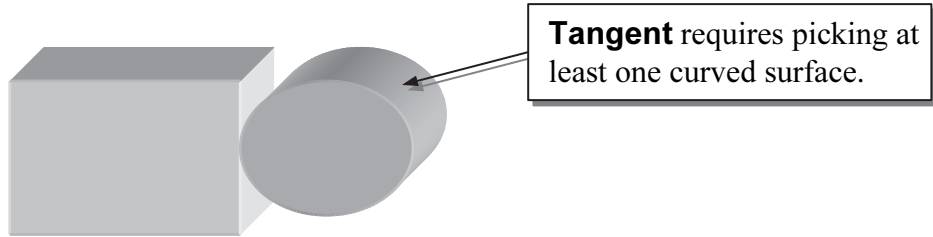
- **Flush** – Makes two planes coplanar with their faces aligned in the same direction. Selected surfaces point in the same direction and are offset by a specified distance. **Flush** constraint aligns components adjacent to one another with faces flush and positions selected faces, curves, or points so that they are aligned with surface normals pointing in the same direction. (Note that the **Flush** constraint is listed as a selectable option in the **Mate** constraint.)



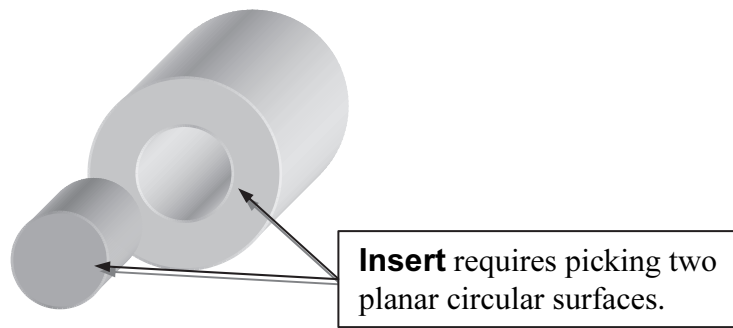
- **Angle** – Creates an angular assembly constraint between parts, subassemblies, or assemblies. Selected surfaces point in the direction specified by the angle.



- **Tangent** – Aligns selected faces, planes, cylinders, spheres, and cones to contact at the point of tangency. Tangency may be on the inside or outside of a curve, depending on the selection of the direction of the surface normal. A **Tangent** constraint removes one degree of translational freedom.

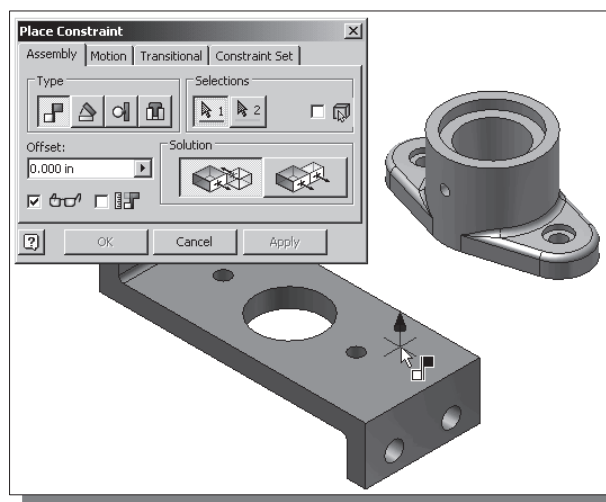


- **Insert** – Aligns two circles, including their center axes and planes. Selected circular surfaces become co-axial. **Insert** constraint is a combination of a face-to-face **Mate** constraint between planar faces and a **Mate** constraint between the axes of the two components. A rotational degree of freedom remains open. The surfaces do not need to be full 360-degree circles. Selected surfaces can point in opposite directions or in the same direction and can be offset by a specified distance.

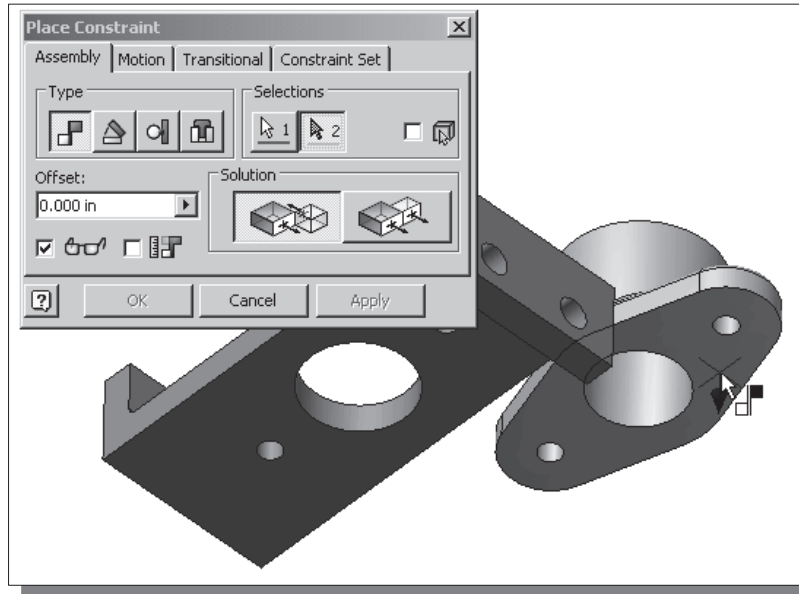


## Apply the First Assembly Constraint

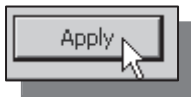
1. In the *Place Constraint* dialog box, confirm the constraint type is set to **Mate** constraint and select the **top horizontal surface** of the base part as the first part for the **Mate** alignment command.



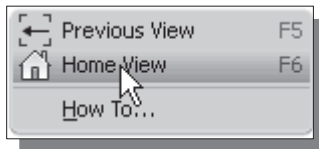
- On your own, dynamically rotate the displayed model to view the bottom of the *Bearing* part, as shown in the figure below.



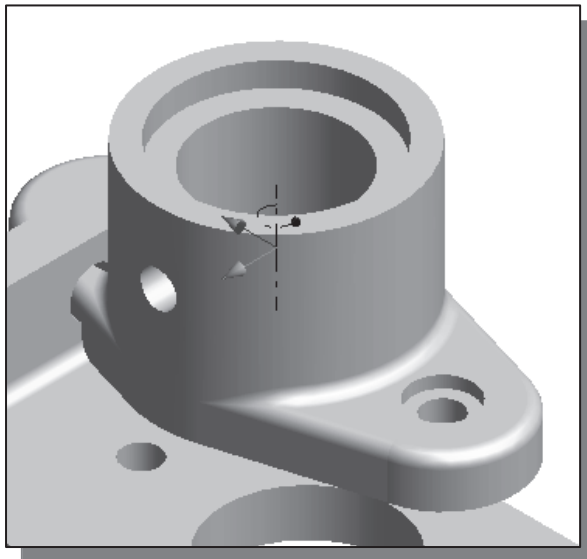
- Click on the bottom face of the *Bearing* part as the second part selection to apply the constraint. Note the direction normals shown in the figure; the **Mate** constraint requires the selection of opposite direction of surface normals.



- Click on the **Apply** button to accept the selection and apply the **Mate** constraint.



- Move the cursor inside the graphics window, and then right-mouse-click to bring up the option menu.
- Select the **Isometric View** to adjust the display of the assembly model.

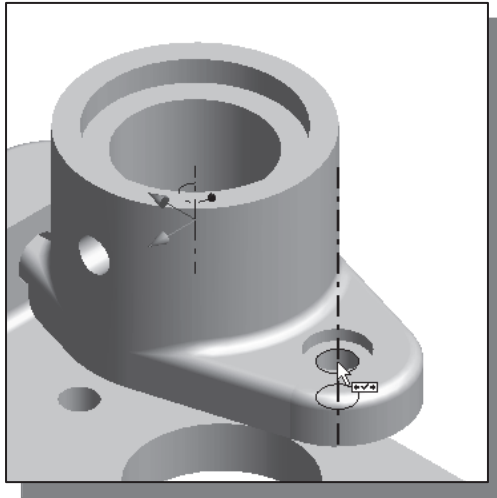


- ❖ Notice the DOF symbol is adjusted automatically in the graphics window. The **Mate** constraint removes one degree of linear translation and two degrees of angular rotation between the selected planar surfaces. The *Bearing* part can still move along two axes and rotate about the third axis.

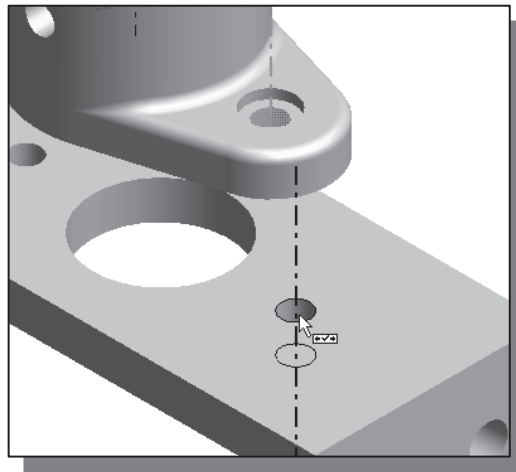
## Apply a Second MATE Constraint

❖ The *Mate* constraint can also be used to align axes of cylindrical features.

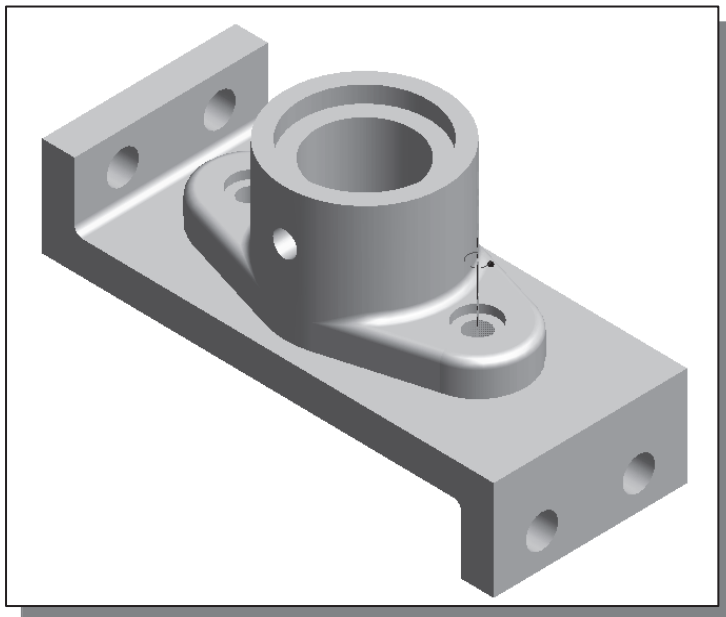
1. In the *Place Constraint* dialog box, confirm the constraint type is set to **Mate** constraint and the *Offset* option is set to **0.00**.



2. Move the cursor near the cylindrical surface of the right counter bore hole of the *Bearing* part. Select the axis when it is displayed as shown. (Hint: Use the *Dynamic Rotation* option to assist the selection.)



3. Move the cursor near the cylindrical surface of the small hole on the *Base-Plate* part. Select the axis when it is displayed as shown.
4. In the *Place Constraint* dialog box, click on the **Apply** button to accept the selection and apply the *Mate* constraint.



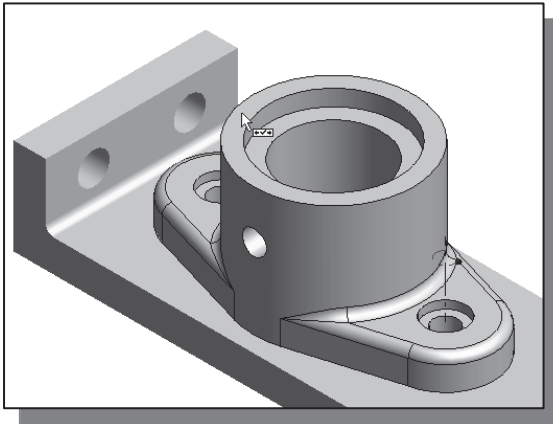
5. In the *Place Constraint* dialog box, click on the **Cancel** button to exit the *Place Constraint* command.

❖ The *Bearing* part appears to be placed in the correct position. But the **DOF** symbol indicates that this is not the case; the bearing part can still rotate about the displayed vertical axis.



## Constrained Move

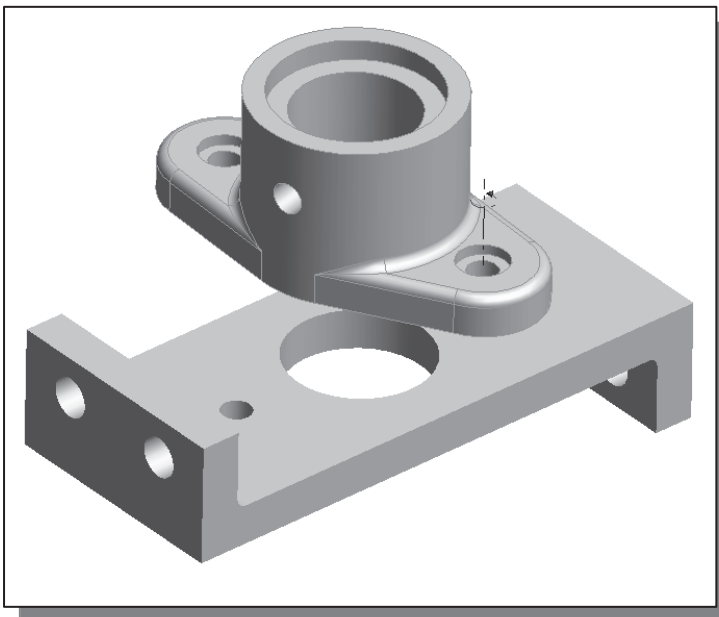
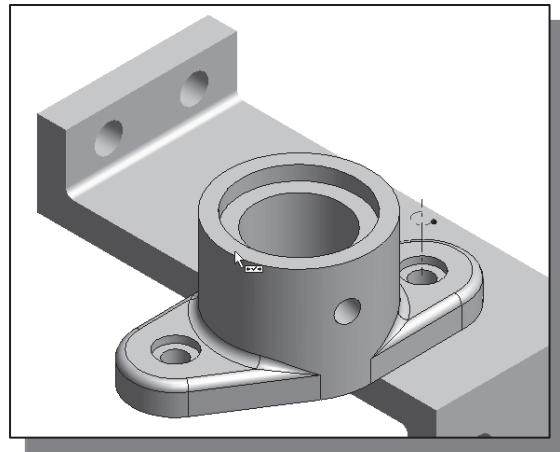
- ❖ To see how well a component is constrained, we can perform a constrained move. A constrained move is done by dragging the component in the graphics window with the left-mouse-button. A constrained move will honor previously applied assembly constraints. That is, the selected component and parts constrained to the component move together in their constrained positions. A grounded component remains grounded during the move.



1. Inside the graphics window, move the cursor on top of the top surface of the *Bearing* part as shown in the figure.

2. Press and hold down the left-mouse-button and drag the *Bearing* part downward.

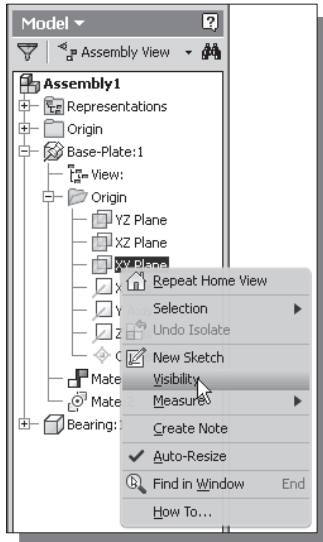
- ❖ The *Bearing* part can freely rotate about the displayed axis.



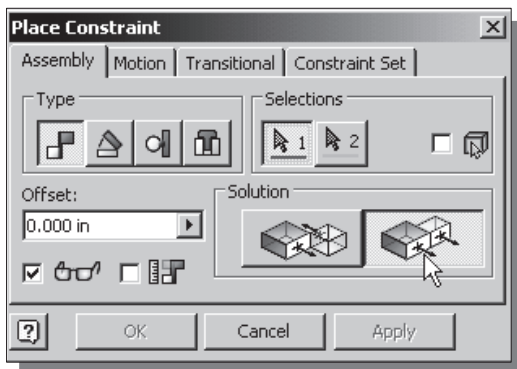
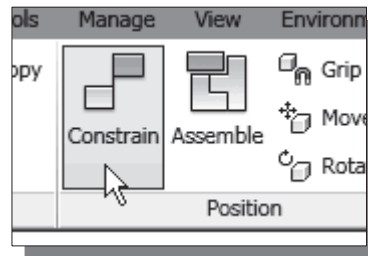
3. On your own, use the dynamic rotation command to view the alignment of the *Bearing* part.
4. Rotate the *Bearing* part and adjust the display as shown in the figure.
5. Switch to the wireframe display mode before proceeding to the next section.

## Apply a Flush Constraint

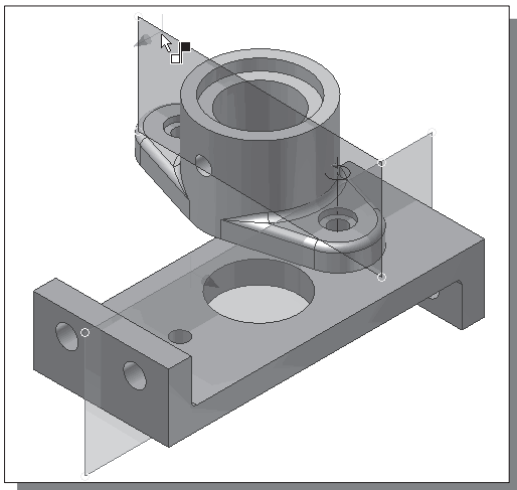
- Besides selecting the surfaces of solid models to apply constraints, we can also select the established work planes to apply the assembly constraints. This is an additional advantage of using the *BORN technique* in creating part models. For the *Bearing* part, we will apply a **Mate** constraint to two of the work planes and eliminate the last rotational DOF.



- On your own, inside the *browser* window toggle *ON* the **Visibility** for the corresponding work planes that can be used for alignment of the *Base-Plate* and the *Bearing* parts.
- In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.

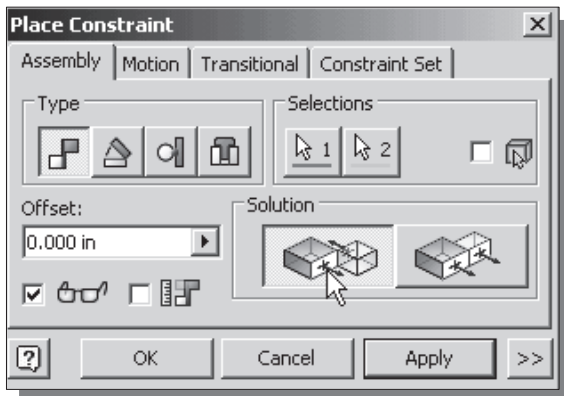
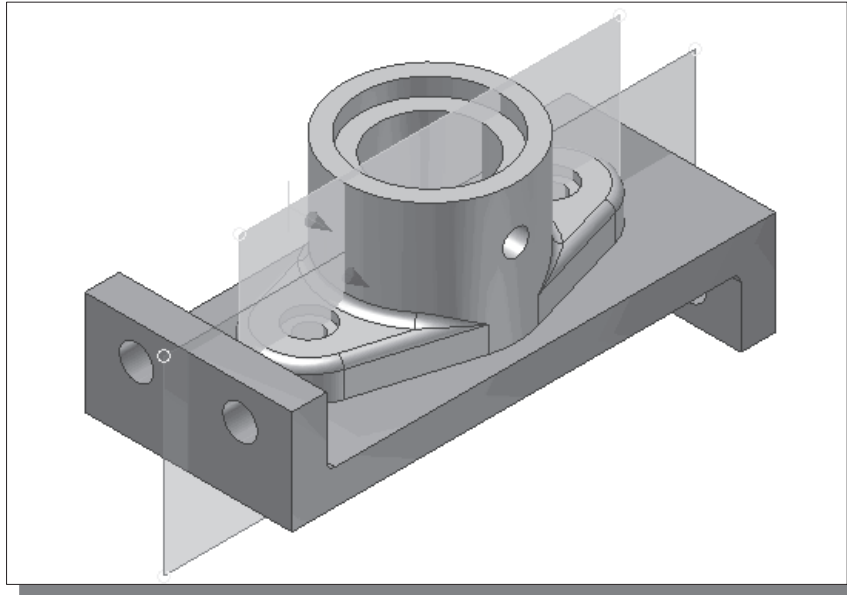


- In the *Place Constraint* dialog box, confirm the constraint type is set to **Mate** constraint and the *Offset* option is set to **0.00**.



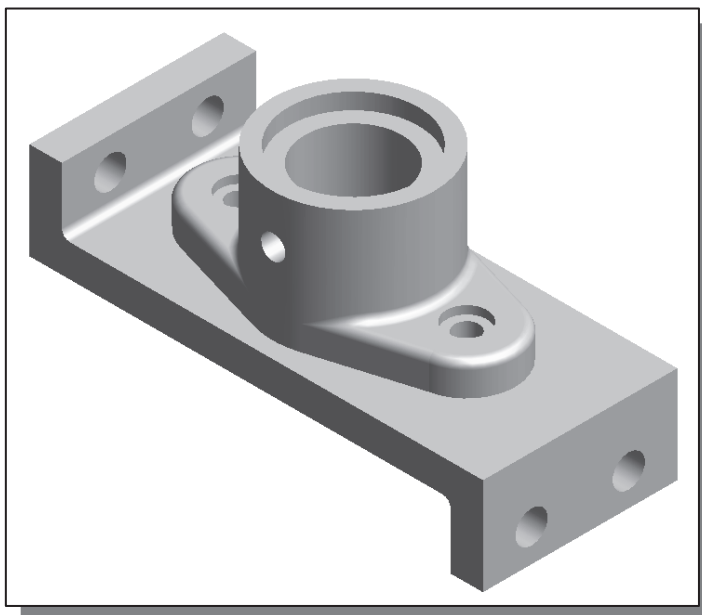
- In the *Place Constraint* dialog box, switch the *Solution* option to **Flush** as shown.
- Select the *work plane* of the *Base-Plate* part as the first part for the Flush alignment command.
- Select the corresponding *work plane* of the *Bearing* part as the second part for the Flush alignment command.

- ❖ Note that the **Flush** constraint makes two planes coplanar with their faces aligned in the same direction.



7. In the *Place Constraint* dialog box, switch the *Solution* option to **Mate** and notice the *Bearing* part is rotated 180 degrees to satisfy the **Mate** constraint.

- ❖ Note that the *Show Preview* option allows us to preview the result before accepting the selection.



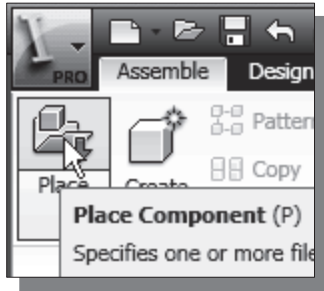
8. On your own, switch back to the **Flush** option and click on the **Apply** button to accept the settings.

9. In the *Place Constraint* dialog box, click on the **Cancel** button to exit the *Place Constraint* command.

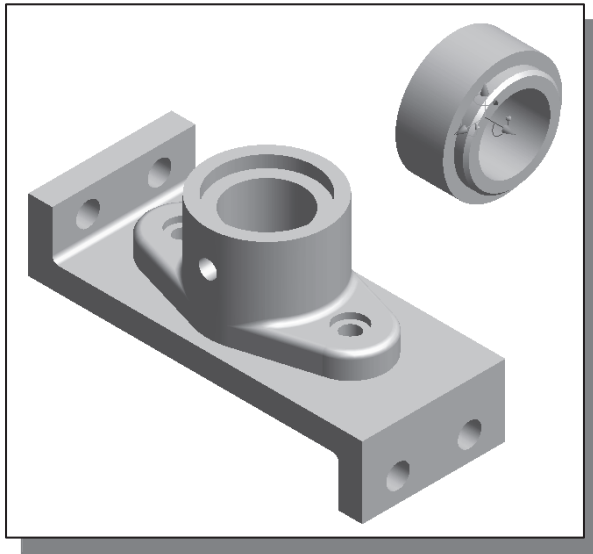
- ❖ Note the *DOF* symbol disappears, which indicates the assembly is fully constrained.

## Placing the Third Component

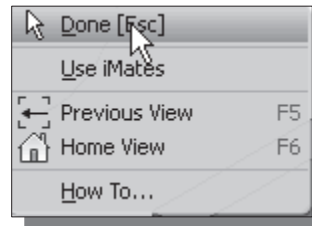
➤ We will retrieve the *Collar* part as the third component of the assembly model.



1. In the *Assembly* panel (the toolbar that is located to the left side of the graphics window) select the **Place Component** command by left-mouse-clicking the icon.
2. Select the **Collar** design (part file: *Collar.ipt*) in the list window. And click on the **Open** button to retrieve the model.

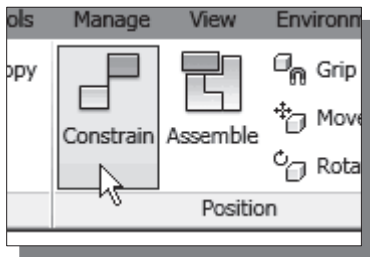


3. Place the *Collar* part toward the upper right corner of the graphics window, as shown in the figure.
4. Inside the graphics window, right-mouse-click once to bring up the option menu and select **Done** to end the placement of the *Collar* part.

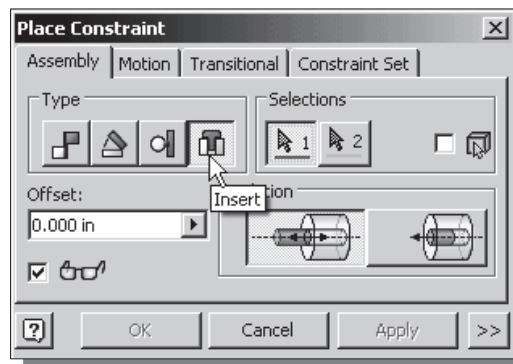


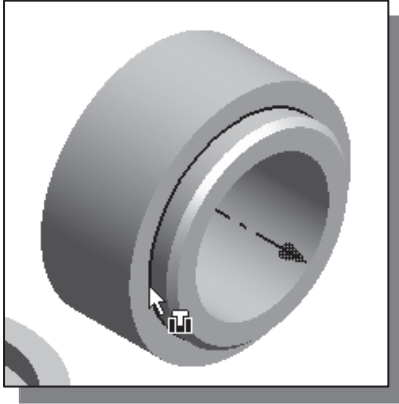
❖ Notice the DOF symbol displayed on the screen. The *Collar* part can move linearly and rotate about the three axes (six degrees of freedom).

## Applying an Insert Constraint

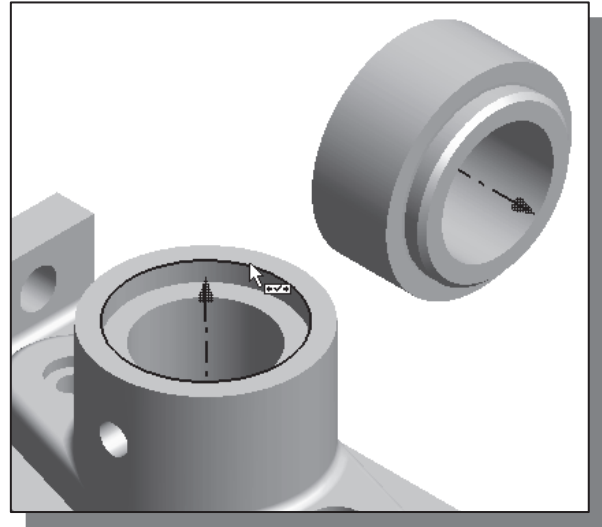


1. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.
2. In the *Place Constraint* dialog box, switch to the **Insert** constraint.

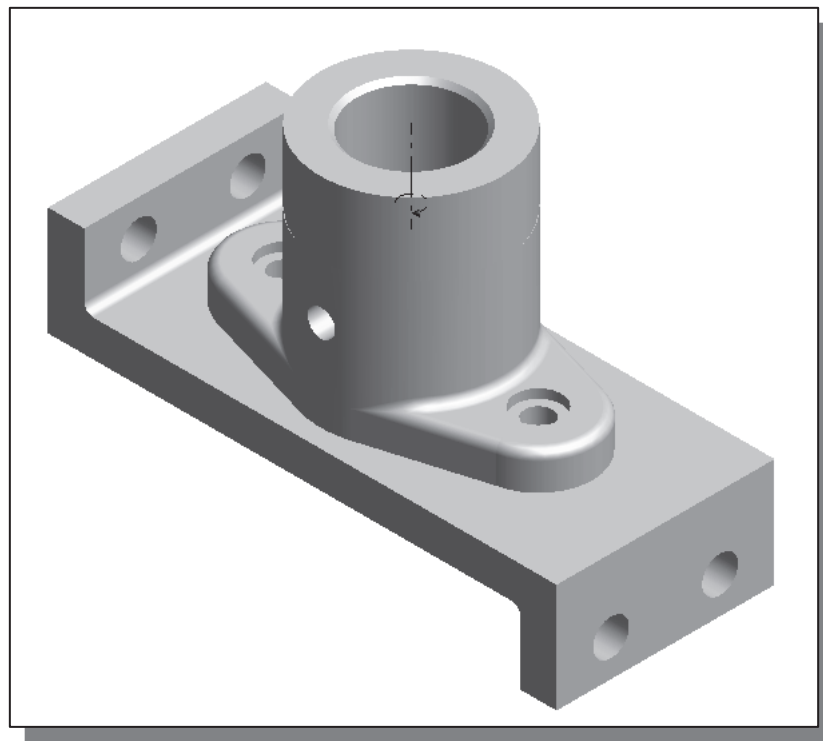




3. Select the inside corner of the *Collar* part as the first surface to apply the **Insert** constraint, as shown in the figure.



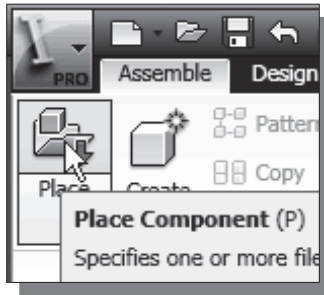
4. Select the inside circle on the top surface of the *Bearing* part as the second surface to apply the **Insert** constraint, as shown in the figure.
5. Click on the **Apply** button to accept the settings.



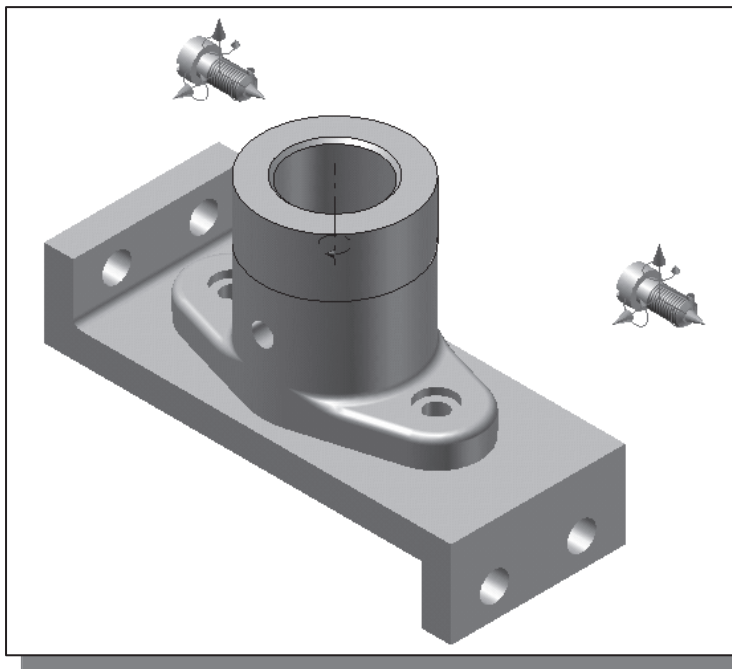
- Note that one rotational degree of freedom remains open; the *Collar* part can still freely rotate about the displayed DOF axis.

## Assemble the Cap-Screws

- ❖ We will place two of the *Cap-Screw* parts to complete the assembly model.

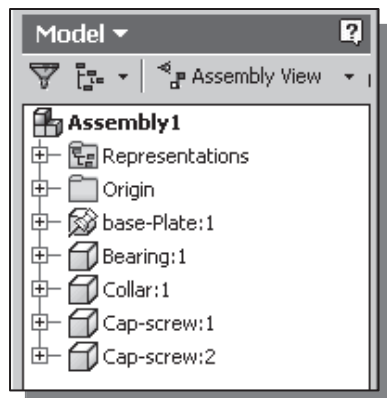


1. In the *Assembly* panel (the toolbar that is located to the left side of the graphics window) select the **Place Component** command by left-mouse-clicking once on the icon.
2. Select the **Cap-Screw** design (part file: *Cap-Screw.ipt*) in the list window. And click on the **Open** button to retrieve the model.



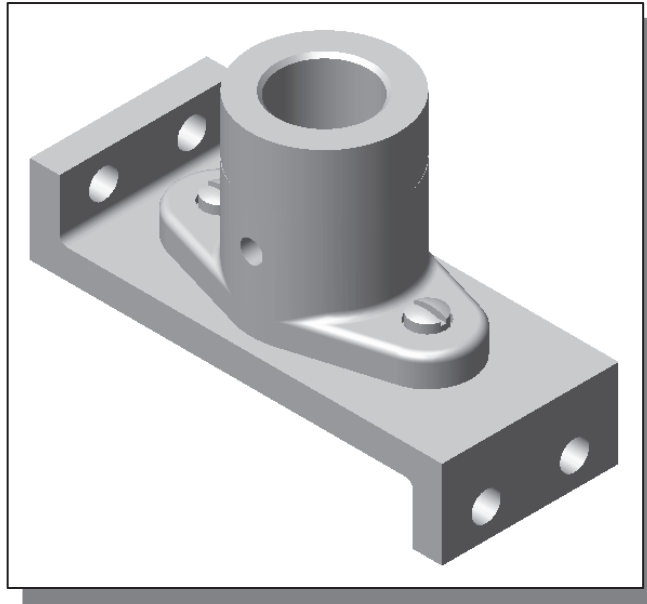
3. Place two copies of the *Cap-Screw* part on both sides of the *Collar* by clicking twice on the screen as shown in the figure.
4. Inside the graphics window, right-mouse-click once to bring up the option menu and select **Done** to end the **Place Component** command.

- ❖ Notice the DOF symbols displayed on the screen. Each *Cap-Screw* has six degrees of freedom. Both parts are referencing the same external part file, but each can be constrained independently.



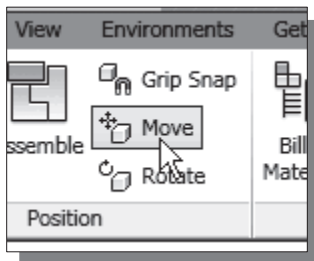
- Inside the *browser* window, the retrieved parts are listed in the order they are placed. The number behind the part name is used to identify the number of copies of the same part in the assembly model. Move the cursor to the last part name and notice the corresponding part is highlighted in the graphics window.

- On your own, use the **Place Constraints** command and assemble the *Cap-Screws* in place as shown in the figure below.



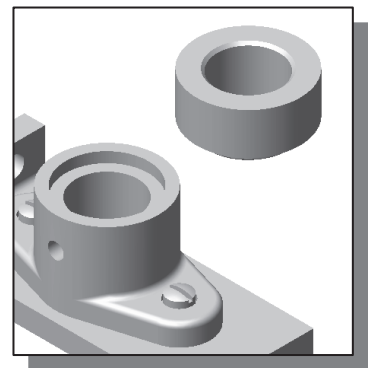
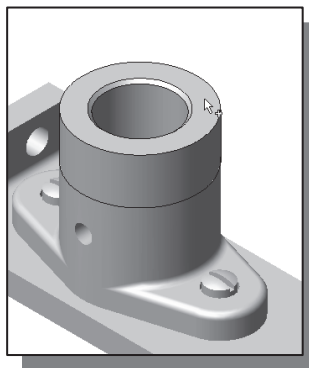
### Exploded View of the Assembly

- Exploded assemblies are often used in design presentations, catalogs, sales literature, and in the shop to show all of the parts of an assembly and how they fit together. In *Autodesk Inventor*, an exploded assembly can be created by two methods: (1) using the **Move Component** and **Rotate Component** commands in the *Assembly Modeler*, which contains only limited options for the operation but can be done very quickly; (2) transferring the assembly model into the *Presentation Modeler*. For our example, we will create an exploded assembly by using the **Move Component** command that is available in the *Assembly Modeler*.

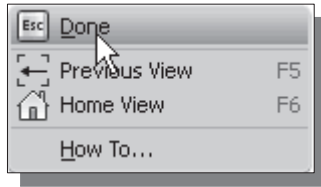


1. In the *Position* toolbar panel, select the **Move Component** command by left-mouse-clicking once on the icon.
2. Inside the graphics window, move the cursor on top of the top surface of the *Collar* part as shown in the figure.

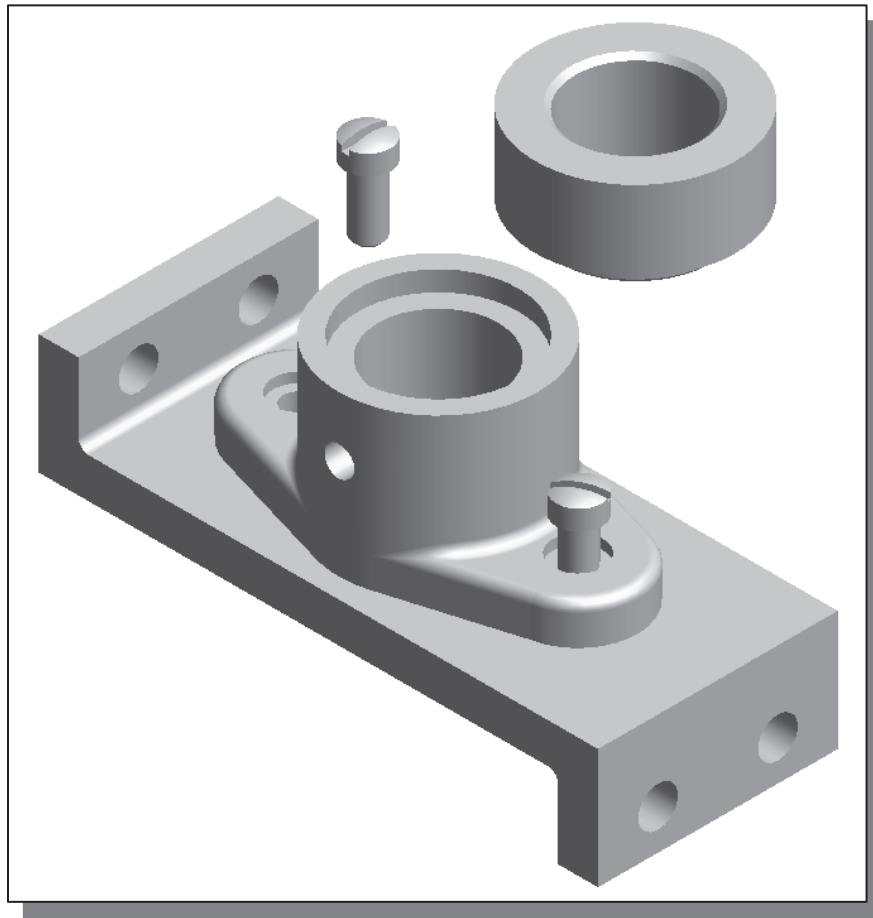
3. Press and hold down the left-mouse-button and drag the *Collar* part toward the right side of the assembly as shown.



4. On your own, repeat the above steps and create an exploded assembly by repositioning the components as shown in the figure below.



5. Inside the graphics window, right-mouse-click once to bring up the option menu and select **Done** to end the **Move Component** command.



- ❖ The **Move Component** and **Rotate Component** commands are used to temporarily reposition the components in the graphics window. The displayed image is temporary, but it can be printed with the **Print** command through the pull-down menu.

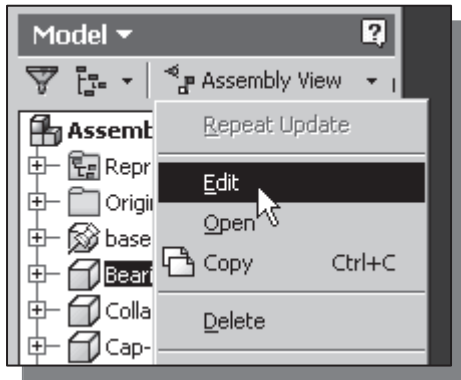


6. Click on the **Update** button in the *Standard* toolbar area.
- ❖ Note that the components are reset back to their assembled positions, based on the applied assembly constraints.



## Editing the Components

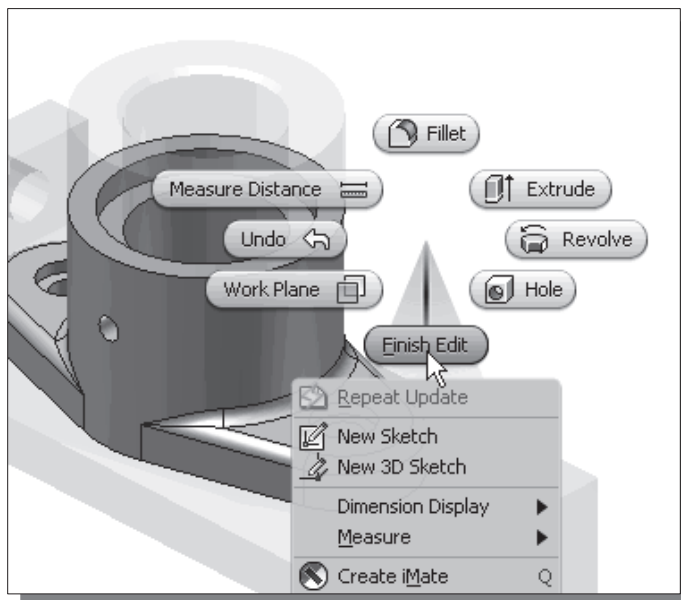
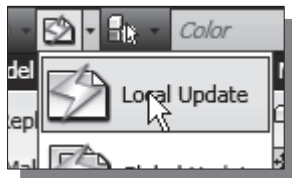
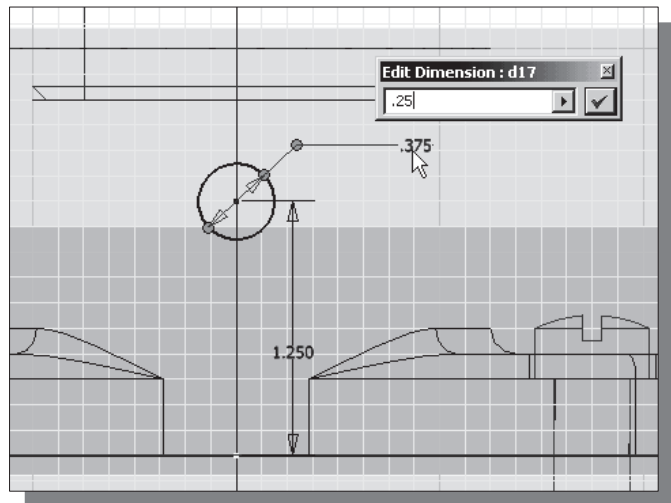
- The *associative functionality* of *Autodesk Inventor* allows us to change the design at any level, and the system reflects the changes at all levels automatically.



1. Inside the *Desktop Browser*, move the cursor on top of the **Bearing** part. Right-mouse-click once to bring up the option menu and select **Edit** in the option list.

❖ Note that we are automatically switched back to *Part Editing Mode*.

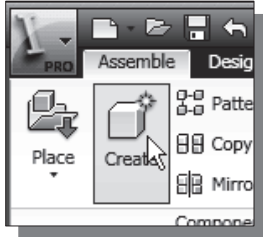
2. On your own, adjust the diameter of the small *Drill Hole* to **0.25** as shown.
3. Click on the **Update** button in the *Standard* toolbar area to proceed with updating the model.



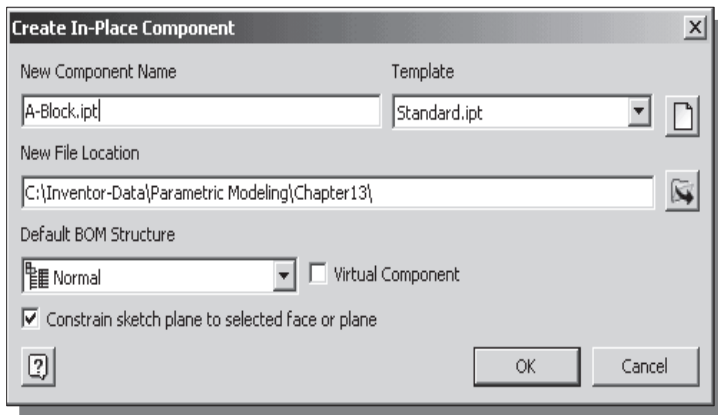
4. Inside the graphics window, click once with the right-mouse-button to display the option menu.
  5. Select **Finish Edit** in the popup menu to exit *Part Editing Mode* and return to *Assembly Mode*.
- *Autodesk Inventor* has updated the part in all levels and in the current *Assembly Mode*. On your own, open the *Bearing* part file to confirm the modification is performed.

## Adaptive Design Approach

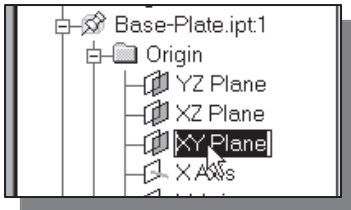
- *Autodesk Inventor's Adaptive Design* approach allows us to use the applied *assembly constraints* to control the sizes, shapes, and positions of **underconstrained** sketches, features, and parts. In this section, we will examine the procedure to apply the constraints directly on 3D parts; note that the adaptive design approach is also applicable to 2D sketches.



1. In the *Assemble* panel, select the **Create Component** command by left-mouse-clicking once on the icon.



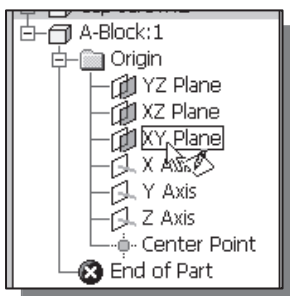
2. In the *Create In-Place Component* dialog box, enter **A-Block** as the new file name.
3. Set the *File Location* to **Chapter13** as shown.
4. Click on the **OK** button to accept the settings.



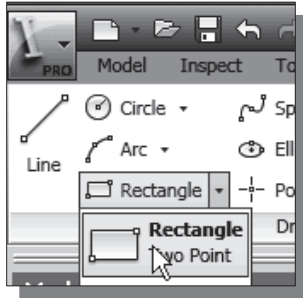
5. Click on the **XY Plane** of the *Base-Plate* part, inside the *browser* window, to apply a **Flush** constraint.



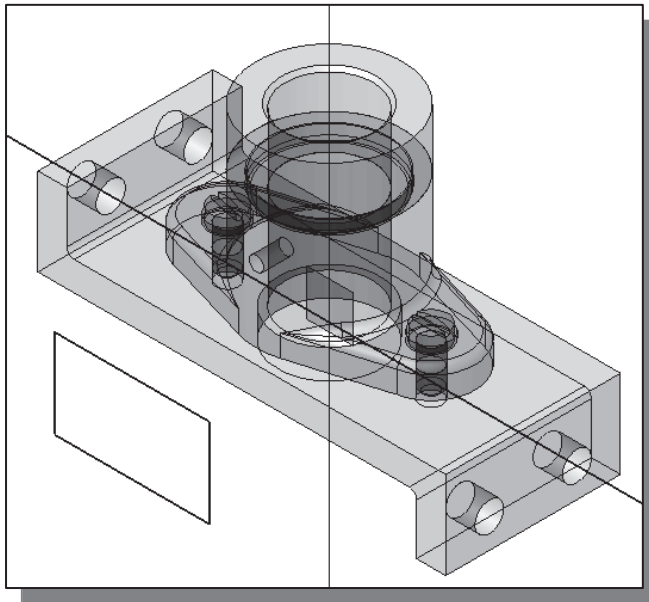
6. Activate the *Model* toolbar and select the **Sketch** command by left-clicking once on the icon.



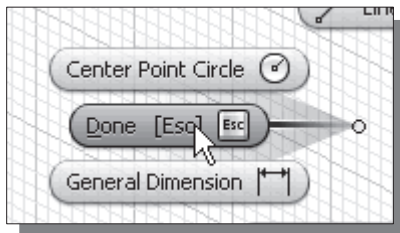
7. Select the **XY Plane**, in the *browser* window, of the *A-Block* part to align the sketch plane.



8. Select the **Two point rectangle** command by clicking once with the left-mouse-button on the icon in the *2D Sketch* panel.

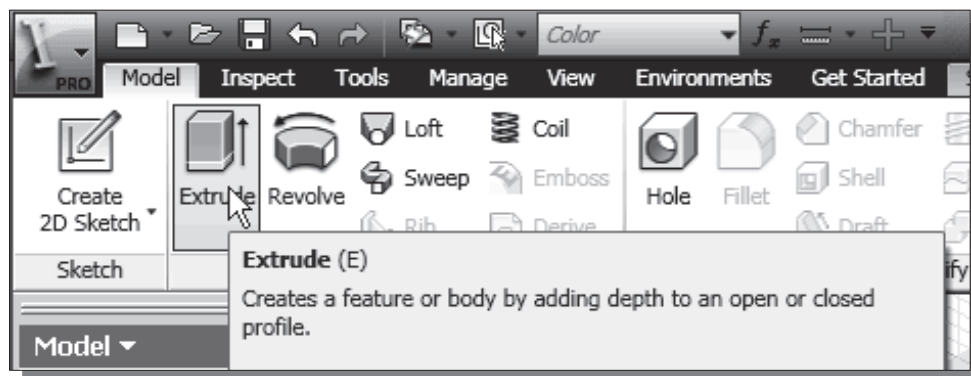


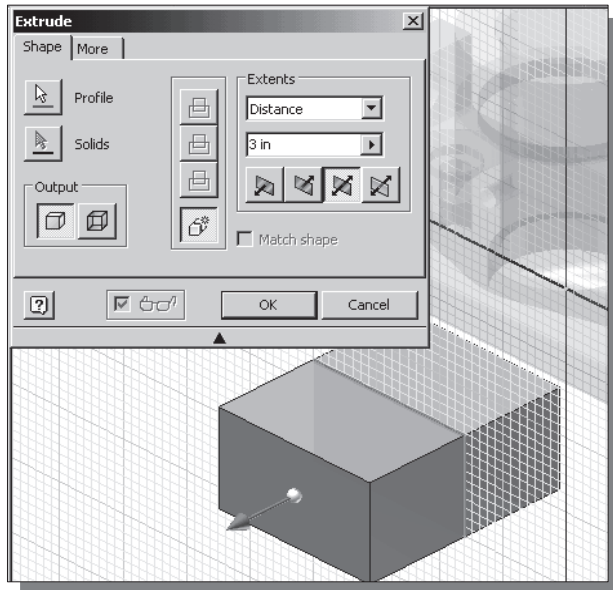
9. On your own, create a rectangle of arbitrary size below the assembly model as shown in the figure.



10. Inside the graphics window, click once with the right-mouse-button to display the option menu and select **Done** to end the **Rectangle** command.

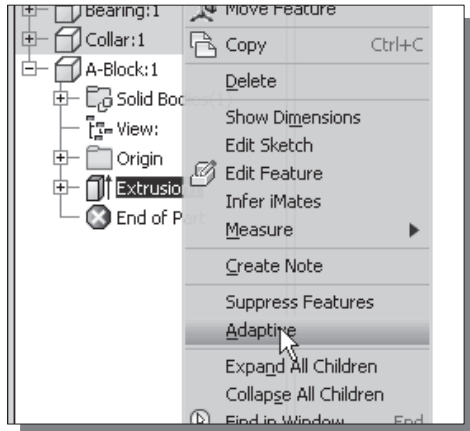
11. In the *Ribbon* toolbar panel, click on the **Model** tab then **Create** → **Extrude** as shown in the figure below.





12. Inside the *Extrude* dialog box, select the **Both Sides** option and set the *Extents Distance* to **3 in** as shown.

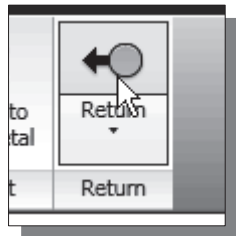
13. In the *Extrude* dialog box, click on the **OK** button to proceed with creating the feature.



14. Right-mouse-click on the **Extrusion1** feature of the *A-Block* part, inside the *browser* window, and select **Adaptive** to allow the use of the adaptive design approach.

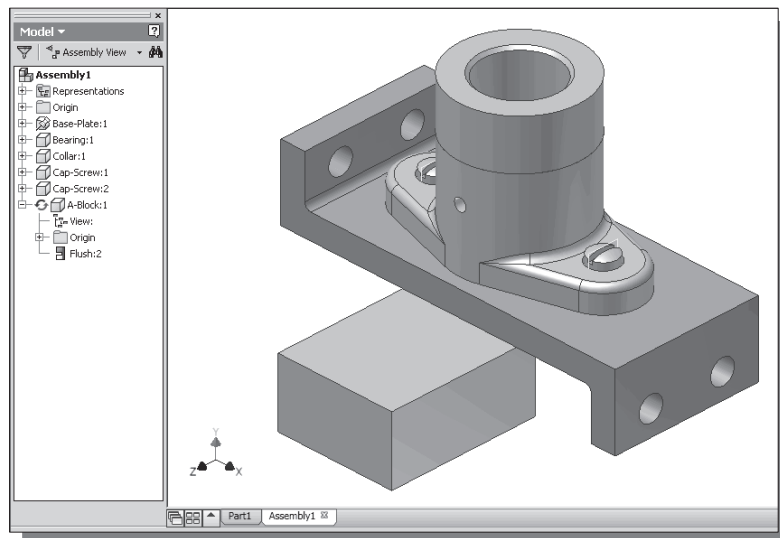
15. Click inside the graphics window to deselect any part or assembly.

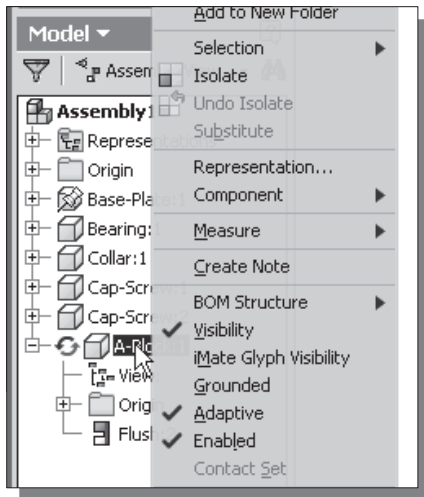
16. Inside the graphics window, click once with the **right-mouse-button** to display the option menu.



17. Select **Return** in the popup menu to exit *Part Editing Mode* and return to *Assembly Modeling Mode*.

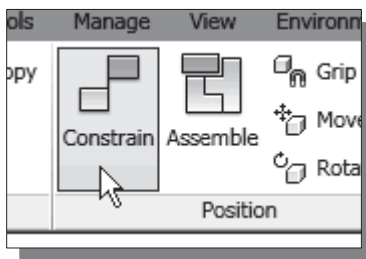
- Note that the rectangular block is created with only one dimension, the extrude distance. The 2D sketch of the part is intentionally under-constrained.





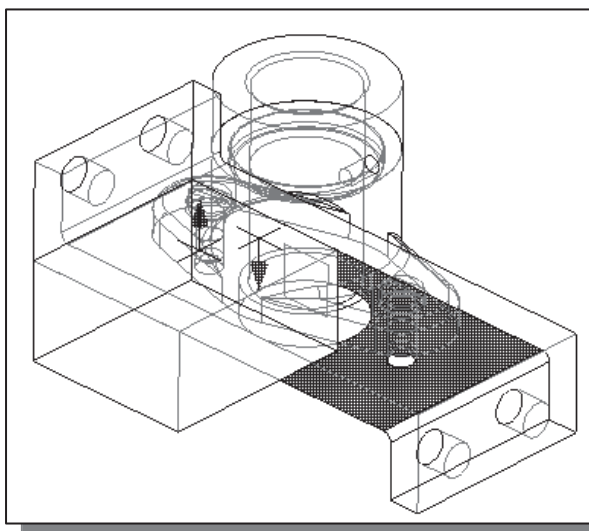
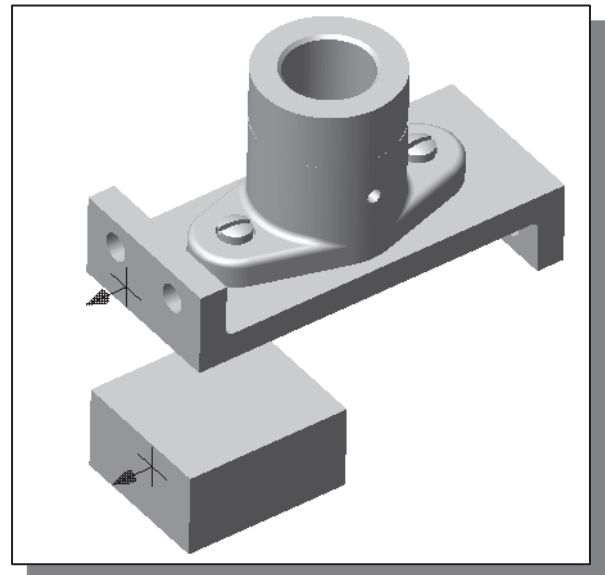
18. Right-mouse-click on the **A-Block** part, inside the *browser* window, and confirm the **Adaptive** option is turned **ON**. This option allows the use of the **Adaptive Design** approach in the current assembly model.

- Note the **Adaptive** icon, the two-arrow symbol, appears in front of the part name in the *browser* window.

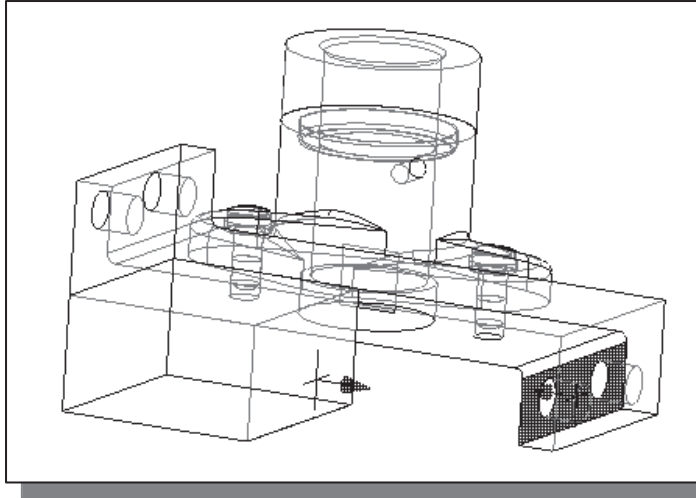


19. In the *Assembly* panel, select the **Constrain** command by left-mouse-clicking once on the icon.

20. On your own, use the **Flush** constraint to align the *A-block* to the left-edge of the *Base-Plate* as shown.



21. Create a **Mate** constraint to align the top of the *A-block* to the bottom of the *Base-Plate* part as shown.



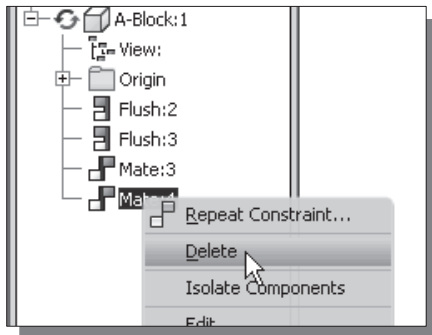
22. Use the **Mate** constraint and align the right surface of the *A-block* to the bottom left surface of the *Base-Plate* part as shown.

- Note that the length of the *A-block* part is automatically adjusted to fit the defined constraint.

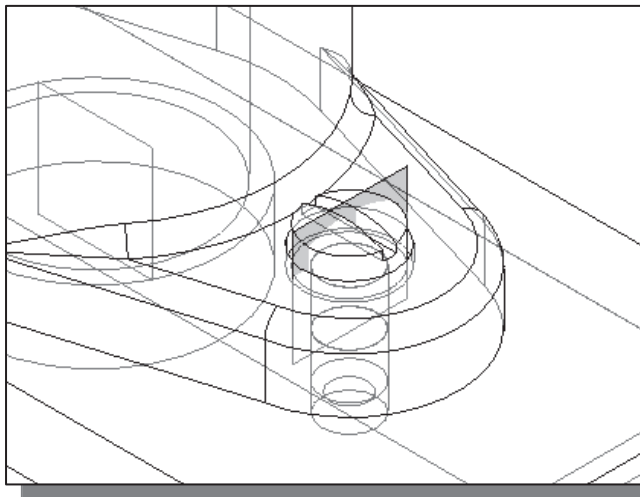


23. In the *Place Constraint* dialog box, click on the **Cancel** button to end the *Place Constraint* command.

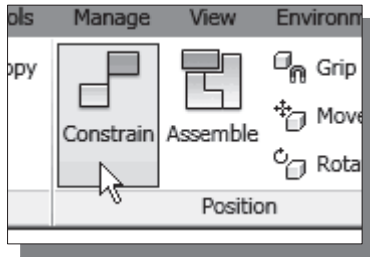
## Delete and Re-apply Assembly Constraints



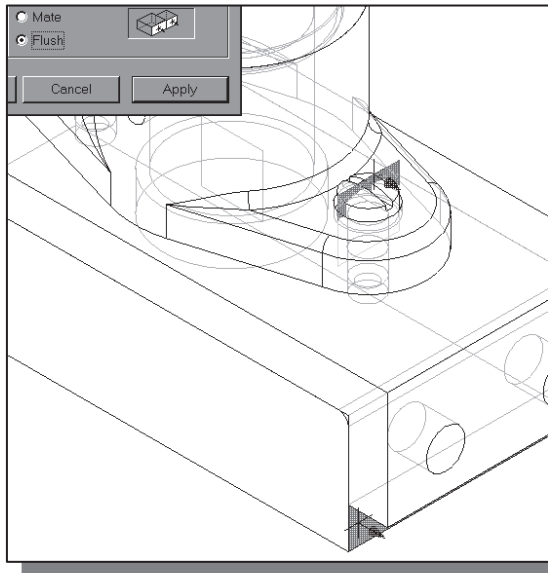
1. Inside the *browser* window, right-mouse-click on the last **Mate** constraint of the *A-Block* part to bring up the option menu.
2. Select **Delete** to remove the applied constraint.



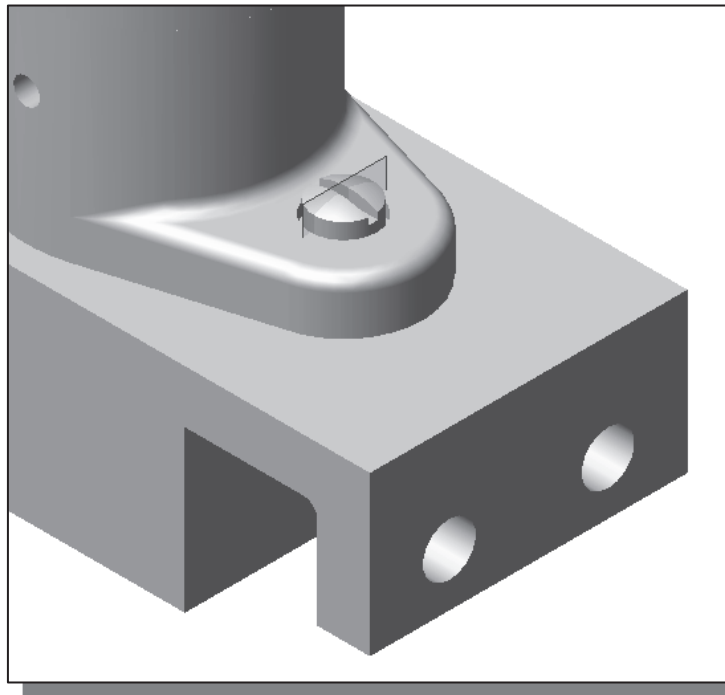
3. On your own, switch **ON** the **Visibility** of the vertical work plane of the *Cap-Screw* part, the plane that is perpendicular to the length of the *Base-Plate* part, as shown.



4. In the *Assembly* position panel, select the **Constrain** command by left-mouse-clicking once on the icon.

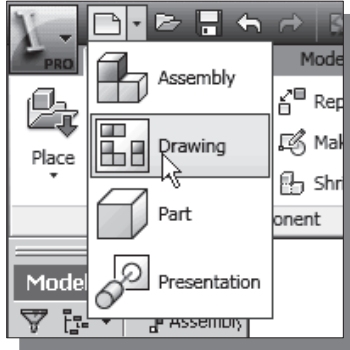


5. On your own, align the right vertical surface of the *A-block* part to the vertical work plane as shown in the figure.
6. On your own, experiment with aligning the right vertical surface of the *A-block* part to other vertical surfaces of the assembly model.
  - As can be seen, the length of the *A-block* is adjusted to the newly applied constraint. The **Adaptive Design** approach allows us to have greater flexibility and simplifies the design process.

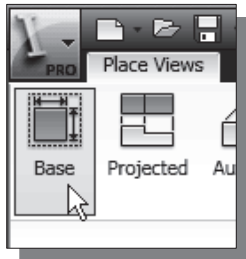


7. On your own, save the assembly model as **Shaft-Support.iam** under the **Chapter13** folder.

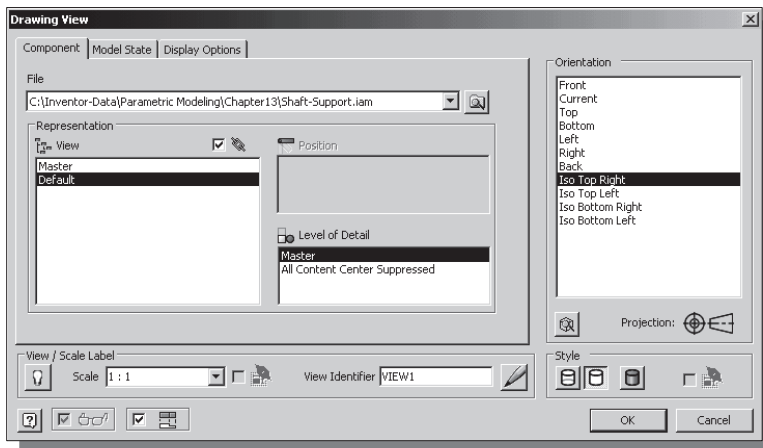
## Setup a Drawing of the Assembly Model



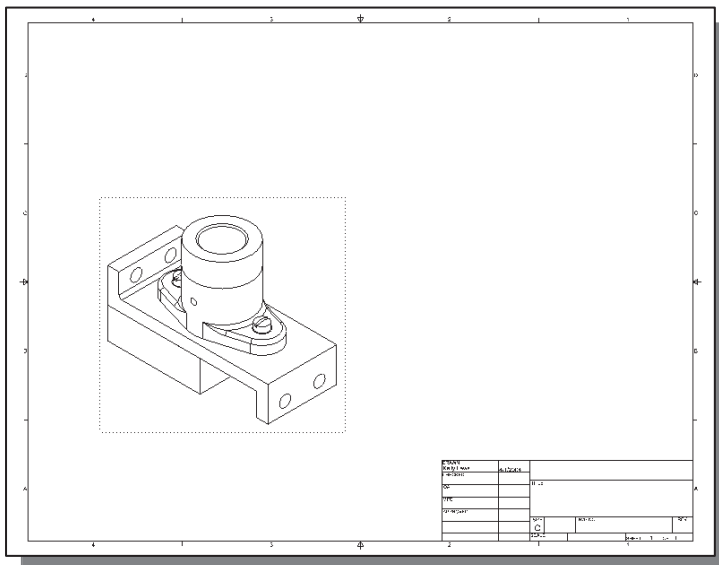
1. Click on the **drop-down arrow** next to the **New File** icon in the *Quick Access* toolbar area to display the available **New File** options.
2. Select **Drawing** from the option list.



3. Click on the **Base View** in the *Drawing Views* panel to create a base view.



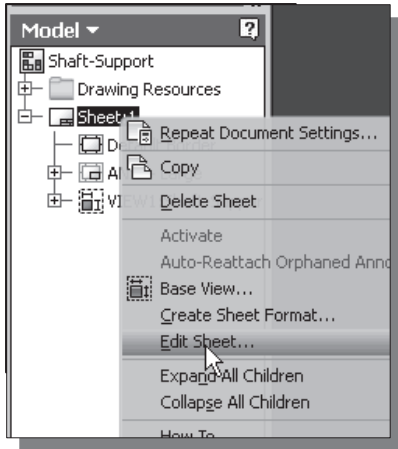
4. In the *Drawing View* dialog box, set *Orientation* to **Iso Top Right View** and **Hidden Line** as shown in the figure. (**DO NOT** click on the **OK** button at this point.)



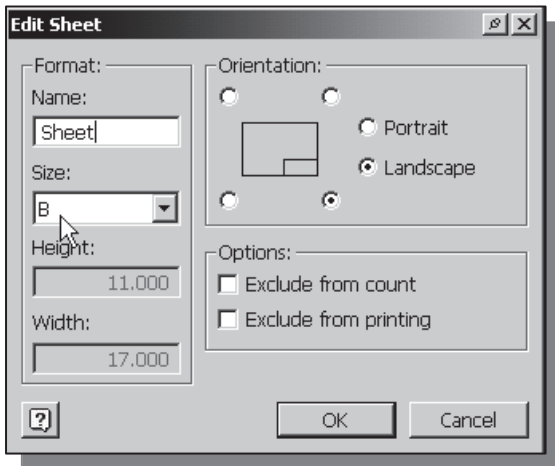
5. Move the cursor inside the graphics window and place the *base* view near the lower left side of the *Border* as shown.

- ❖ Note that the default sheet size is much bigger than the created view. *Inventor* allows us to adjust the sheet size even when views have been created.

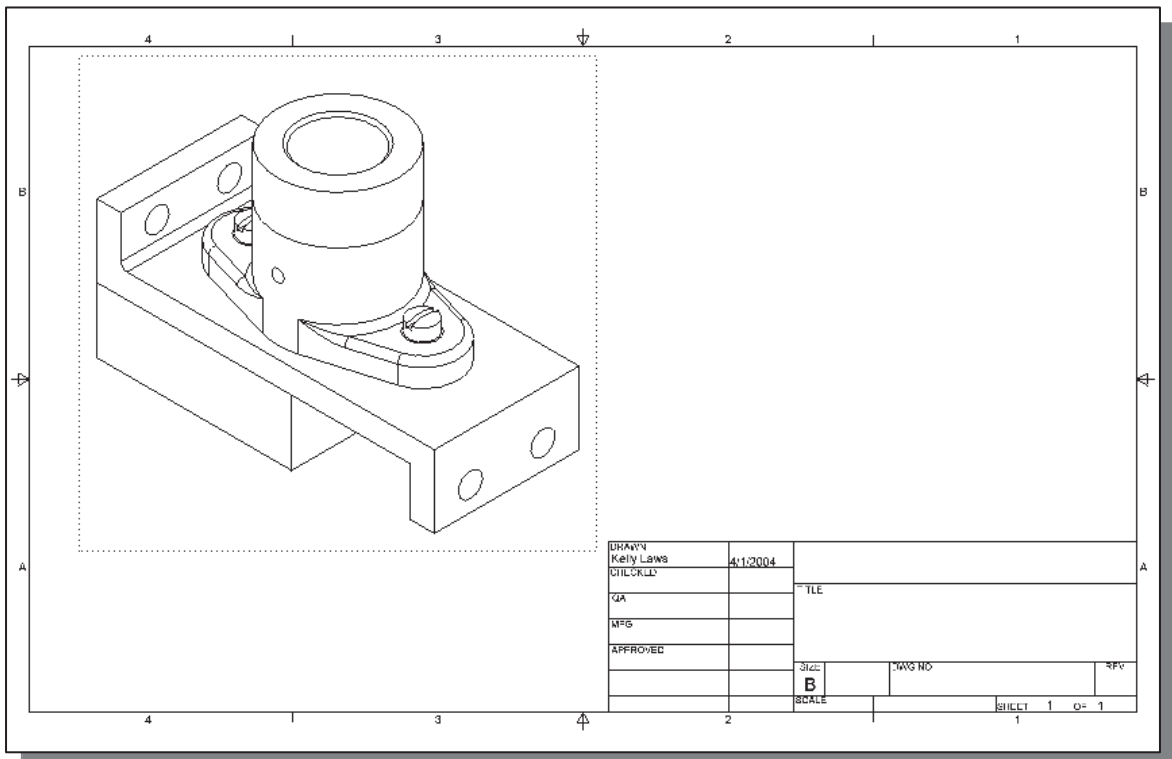




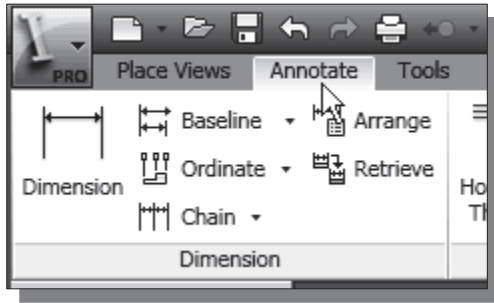
6. Inside the *Drawing Browser* window, right-mouse-click on **Sheet:1** to display the option menu.
7. Select **Edit Sheet** in the option menu to display the settings for the drawing.



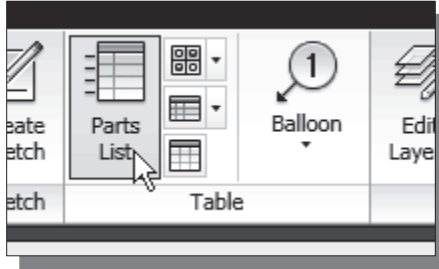
8. Set the sheet size to **B-size** as shown.
  9. Click on the **OK** button to accept the settings and exit the Edit Sheet command.
- ❖ Note that the *Border* and *Title Block* are automatically replaced as the sheet size is adjusted.



## Creating a Parts List

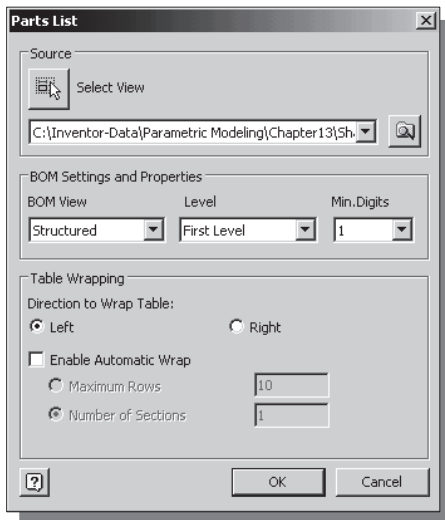


1. Left-mouse-click in the *title* area of the *Drawing Views* panel.
2. Select **Drawing Annotation Panel** by left-clicking once in the drop-down list.



3. In the *Drawing Annotation* window, click on the **Parts List** button.
4. In the prompt area, the message “*Select a view*” is displayed. Items in the selected view will be listed in the *Parts List*. Select the **base view**.

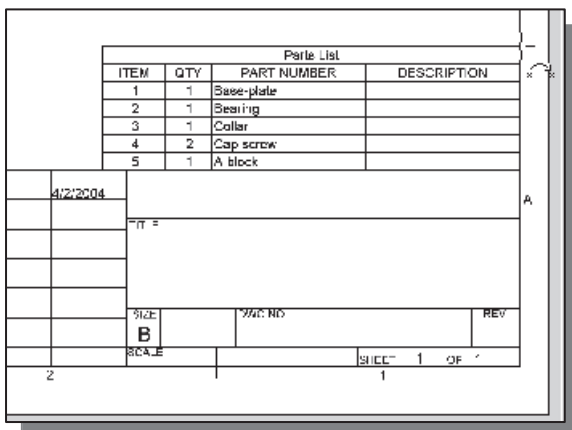
❖ The *Parts List* dialog box appears on the screen; options are available to make adjustments to the numbering system and table wrap settings.



➤ The *Parts List* – BOM options are as follows:

**Structured:** Creates a parts list in which subassemblies are assigned using a nested numbering system (for example, 1, 1.1, 1.1.1). The nested number extends as many levels as needed for the assembly levels in the model.

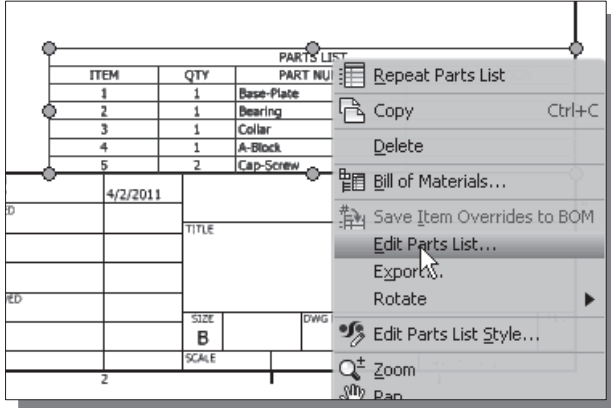
**Only Parts:** Creates a parts list that sequentially numbers all parts in the assembly, including parts that are contained in subassemblies.



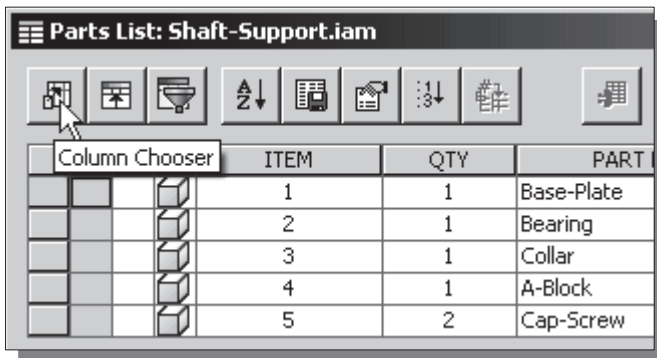
❖ Note that no subassemblies are currently used in the assembly; we will accept the default settings.

5. Click **OK** to accept the default settings.
6. Click **OK** again to enable the BOM view.
7. Place the *Parts List* above the *Title Block* as shown.

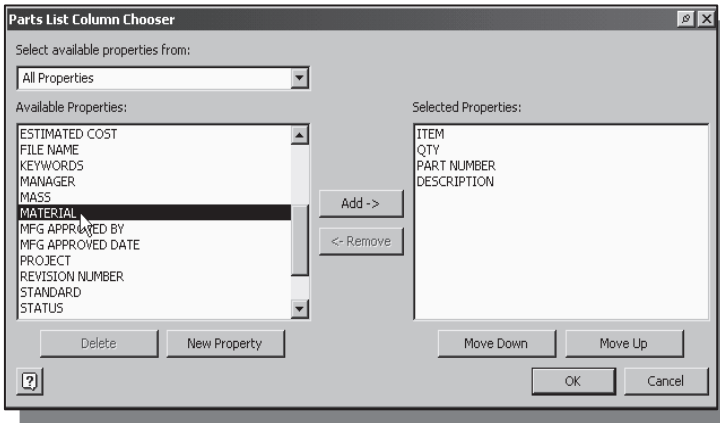
## Editing the Parts List



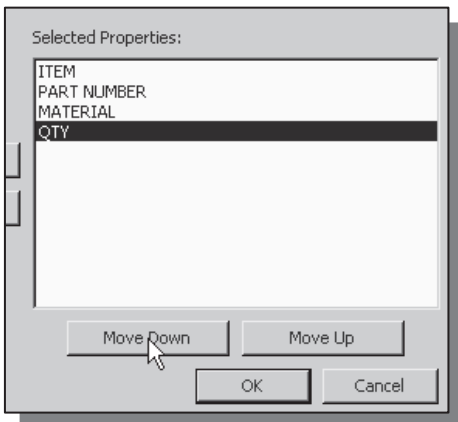
1. Move the cursor on top of the **Parts List** and right-mouse-click once to display the option menu.
2. Choose **Edit Parts List** in the option menu as shown.



3. Click on the **Column Chooser** button as shown.



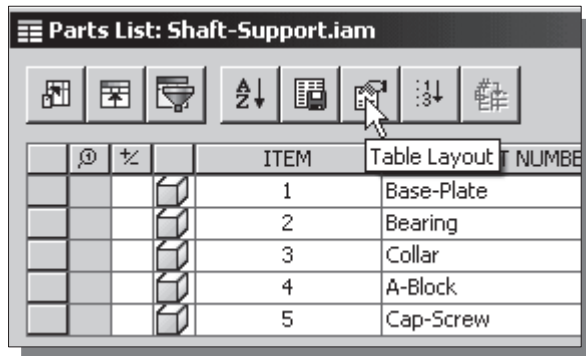
4. Select **MATERIAL** in the *Available Properties* list as shown.
5. Click **Add** to add the selected item to the *Selected Properties* list.



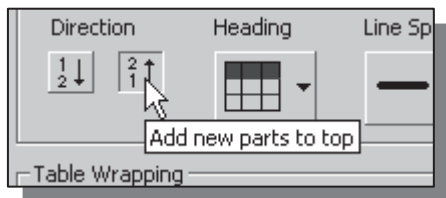
6. On your own, adjust the *Selected Properties* list as shown. (Hint: Use the **Move Up** and **Move Down** buttons to arrange the order of the list.)
7. Click **OK** to accept the settings.

- ❖ The **Parts List** is adjusted using the new settings. Note that, currently, all of the parts are using the **Default** material type.

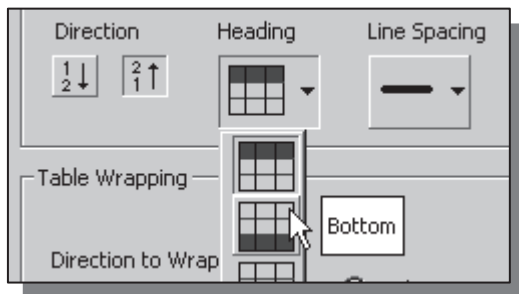
	ITEM	PART NUMBER	MATERIAL	QTY
	1	Base-Plate	Default	1
	2	Bearing	Default	1
	3	Collar	Default	1
	4	Cap-Screw	Default	2
	5	A-Block	Default	1



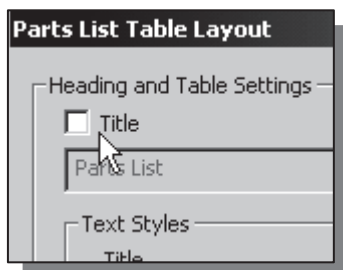
8. Click on the **Table Layout** button as shown.



9. Set the **Table Direction** to **Add new parts to top** as shown.



10. Set the **Heading Placement** to **Bottom** as shown.



11. Turn **OFF** the **Parts List Title**.

12. Click **OK** to accept the settings.



ITEM	PAR NUMBER	MATERIAL	QTY
5	A-block	Default	1
4	Cap-screw	Default	2
3	Collar	Default	1
2	Bearing	Default	1
1	Base-plate	Default	1

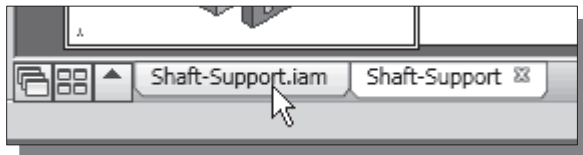
  

NAME	6173034		
CHANGED			
DATE			
BY			
APPROVED			
	SIZE	DWG NO	REV
	B	Shaft-Support	
		SHEET 1	OF 1

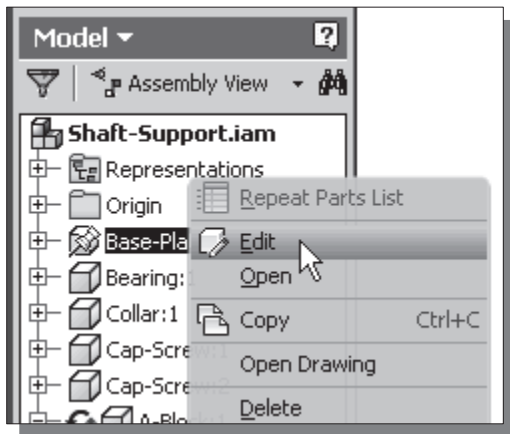
13. On your own, adjust the position of the **Parts List** so that it is aligned to the top edge of the *Title Block* as shown. (Hint: Look for the MOVE symbol next to the cursor.)

### Changing the Material Type

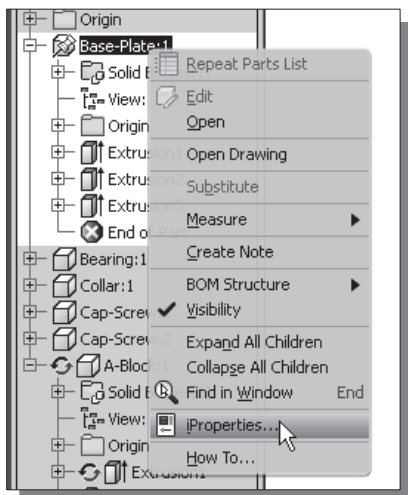
❖ We will switch back to the assembly model to change the assignments of the material type.



1. Click on the *Shaft-Support.iam* window to switch back to the assembly model.

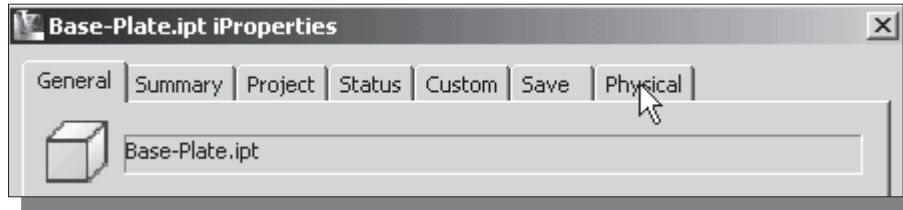


2. Inside the *Model Browser* window, right-mouse-click on **Base-Plate:1** to display the option menu.  
 3. Select **Edit** in the option menu to enter the *Edit Mode* for the selected part.

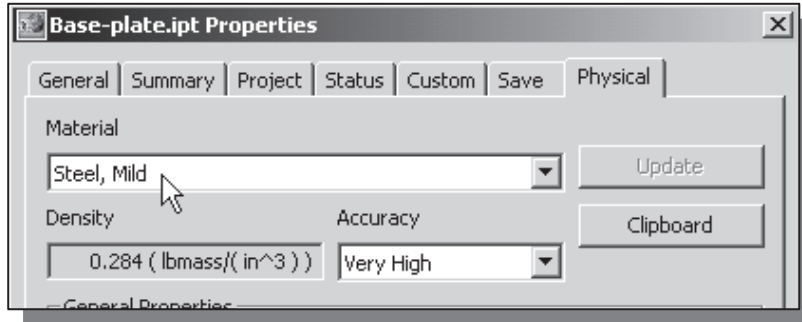


4. Inside the *Model Browser* window, select the **Base-plate** part by clicking once with the left-mouse-button.  
 5. Right-mouse-click on **Base-plate:1** to display the option menu and choose **iProperties** in the options list.

6. Click on the **Physical** tab in the *Base-plate.ipt Properties* window.



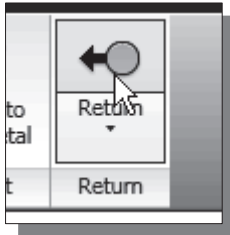
7. Choose the **Steel, Mild** in the *Material* list. Note the properties of the selected material are also displayed in the *Properties* list as shown in the figure.



❖ *Autodesk Inventor* comes with many materials that have pre-entered information; additional material types/properties can also be added/changed as well.



8. Click **OK** to accept the setting and exit the *Materials* dialog box.



9. Select **Return** in the Standard toolbar to exit the *Part Editing Mode*.

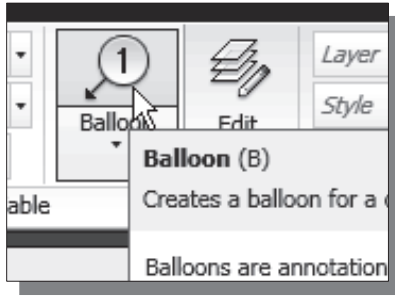


10. On your own, switch back to the *Shaft-Support.idw* window and notice the *Material* information for the **Base-plate** part is now updated

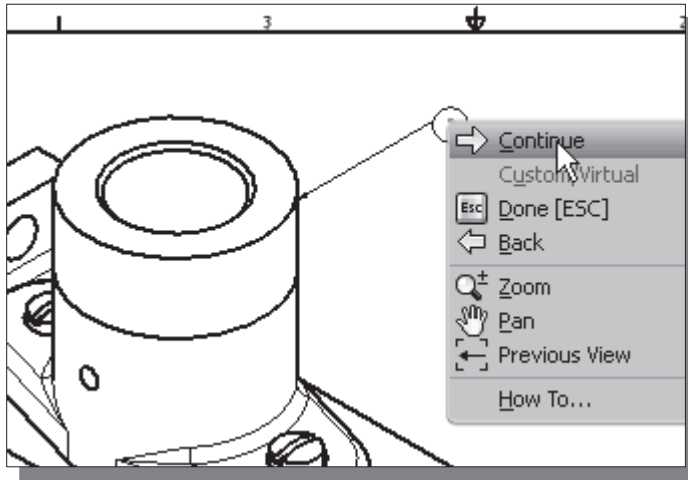
11. On your own, repeat the above steps to change the material information for the other parts as shown in the figure below.

5	A-block	Aluminum-6061	1
4	Cap-screw	Steel, Mild	2
3	Collar	Bronze, Soft Tin	1
2	Bearing	Cast Iron	1
1	Base-plate	Steel, Mild	1
ITEM	PART NUMBER	MATERIAL	QTY
DRAWN Randy		5/3/2005	A
CHECKED			

## Completing the Assembly Drawing

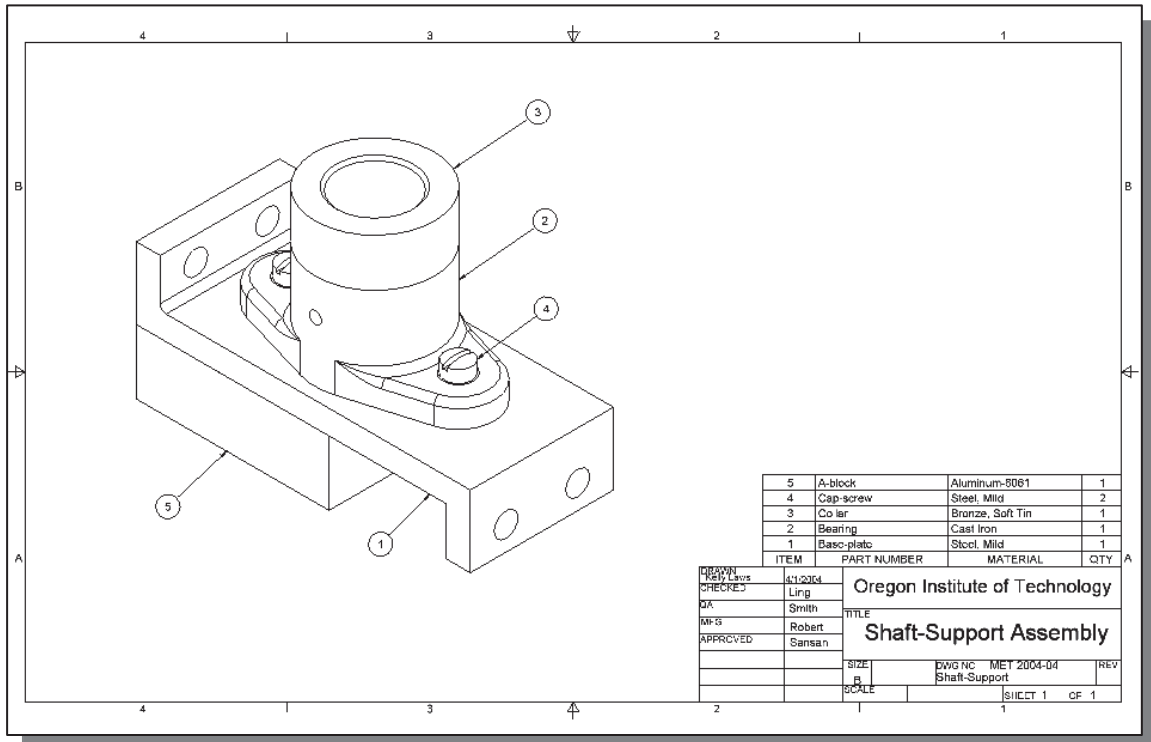


1. In the *Drawing Annotation* window, click on the **Balloon** button.
2. In the prompt area, the message “*Select a location*” is displayed. Click on the **Collar** part to attach an arrowhead to the part.
3. Pick another location to **place the balloon** as shown in the figure below.



4. Inside the graphics window, click once with the right-mouse-button to display the option menu.
5. Select **Continue** in the popup menu to proceed with the creation of the balloon.

6. On your own, repeat the above steps and complete the drawing as shown.



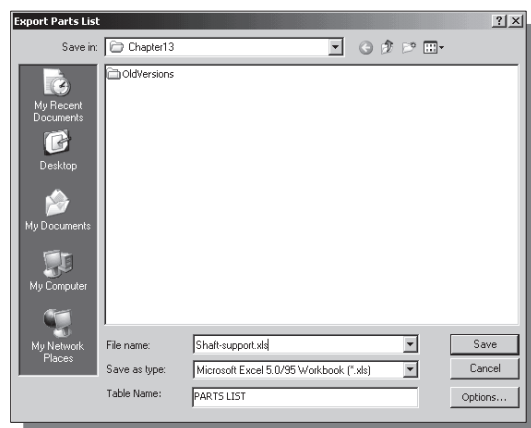
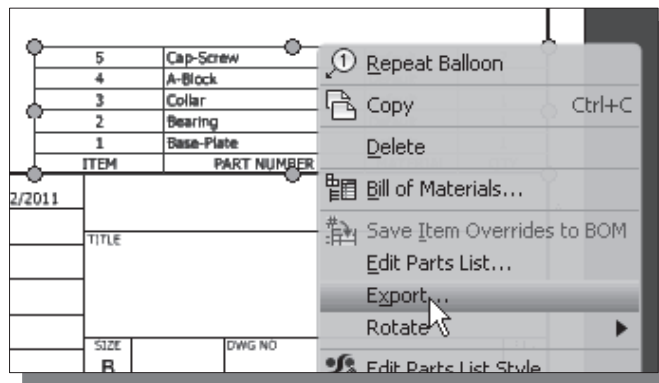
## Bill of Materials

- ❖ A bill of materials (BOM) is a table that contains information about the parts within an assembly. The BOM can include information such as part names, quantities, costs, vendors, and all of the other information related to building the part. The **parts list**, which is used in an assembly drawing, is usually a partial list of the associated BOM.

In *Autodesk Inventor*, both the *bill of materials* and *parts list* can be derived directly from data generated by the assembly and the part properties. We can select which properties to be included in the *bill of materials* or *parts list*, in what order the information is presented, and in what format to export the information. The exported file can be used in an application such as a spreadsheet or text editor.

### (a) BOM from Parts List

1. Move the cursor on top of the **Parts List** and right-mouse-click once to display the option menu.
2. Choose **Export...** in the option menu as shown.



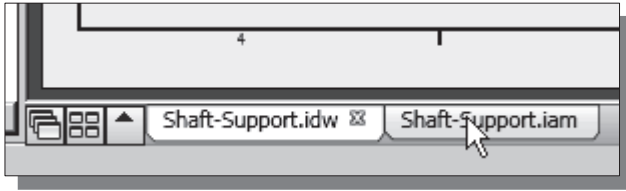
3. Confirm the **Save as type** list is set to **Microsoft Excel**.
4. Enter **Shaft-support.xls** as the filename and click **Save** to export the BOM.

BOM-Shaft-Support.xls				
	A	B	C	D
1	ITEM	PART NUMBER	MATERIAL	QTY
2	1	Base-plate	Steel, Mild	1
3	2	Bearing	Cast Iron	1
4	3	Collar	Bronze, Soft Tin	1
5	4	Cap-screw	Steel, Mild	2
6	5	A-block	Aluminum-6061	1
7				

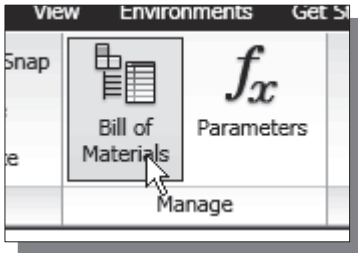
- ❖ On your own, examine the exported BOM by opening up the file in *Excel*.



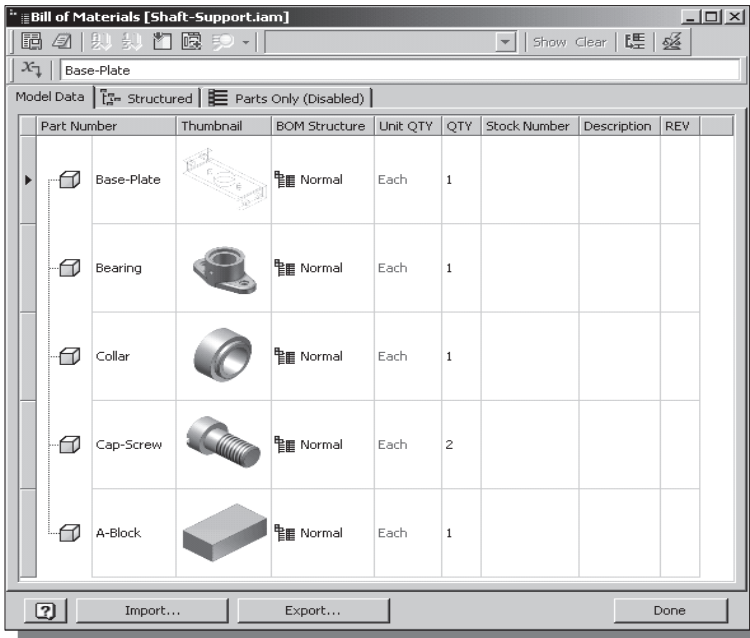
**(b) BOM from Assembly Model**



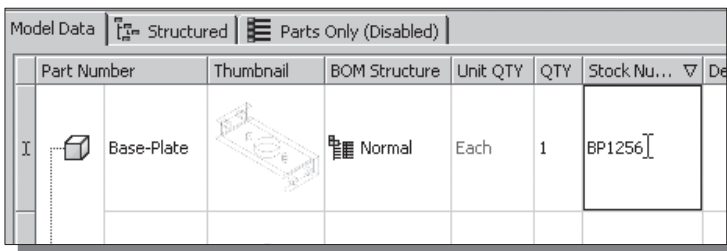
1. Click on the *Shaft-Support.iam* tab to switch back to the assembly model.



2. Select **Bill of Materials** in the *Assemble* tab of the *Ribbon* toolbar panel.



❖ Note that many of the controls and options are similar to those of the **Parts List** command in the *Drawing Mode*.



3. Click inside the **Stock Number** box to enter the *Edit Mode*.

4. Enter **BP1256** as the new *Stock Number*.



5. Click the **Export Bill of Materials** button.

6. On your own, export using the Microsoft Excel format and examine the *BOM* in Microsoft Excel.

**Questions:** (Time: 35 minutes)

1. What is the purpose of using *assembly constraints*?
2. List three of the commonly used *assembly constraints*.
3. Describe the difference between the **Mate** constraint and the **Flush** constraint.
4. In an assembly, can we place more than one copy of a part? How is it done?
5. How should we determine the assembly order of different parts in an assembly model?
6. How do we adjust the information listed in the **parts list** of an assembly drawing?
7. In *Autodesk Inventor*, describe the procedure to create a **bill of materials** (BOM)?
8. Create sketches showing the steps you plan to use to create the four parts required for the assembly shown on the next page:

Ex.1)

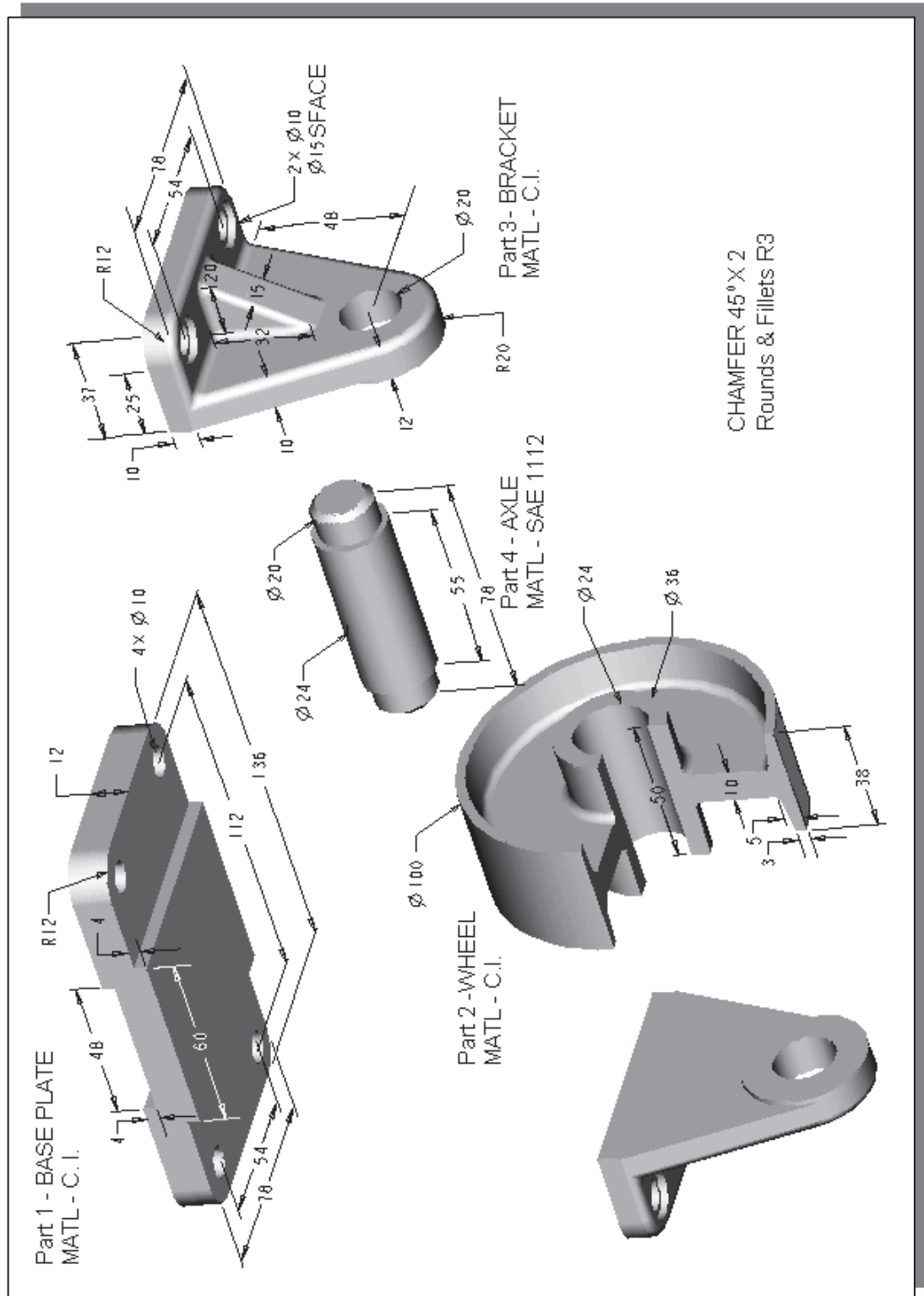
Ex.2)

Ex.3)

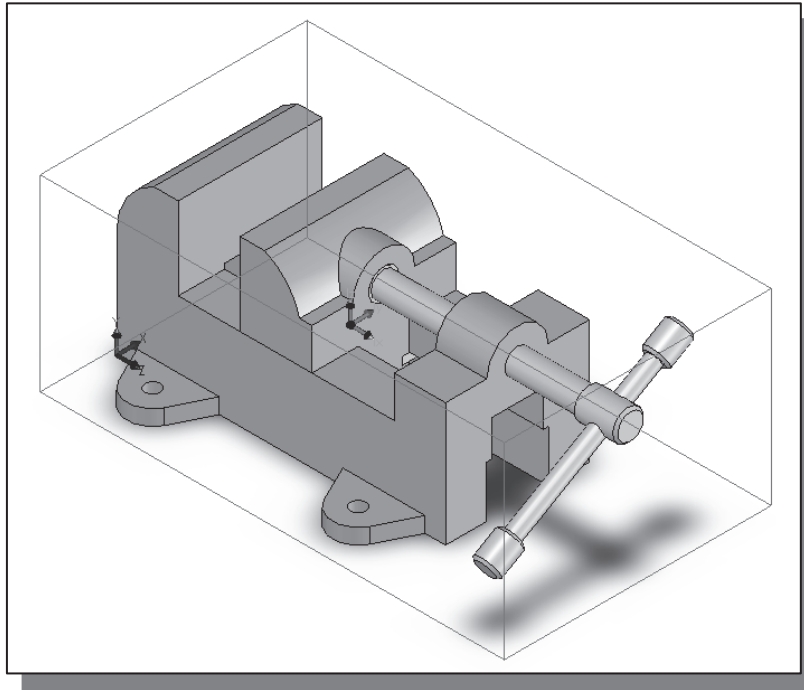
Ex.4)

**Exercises:** (Time: 60 minutes)

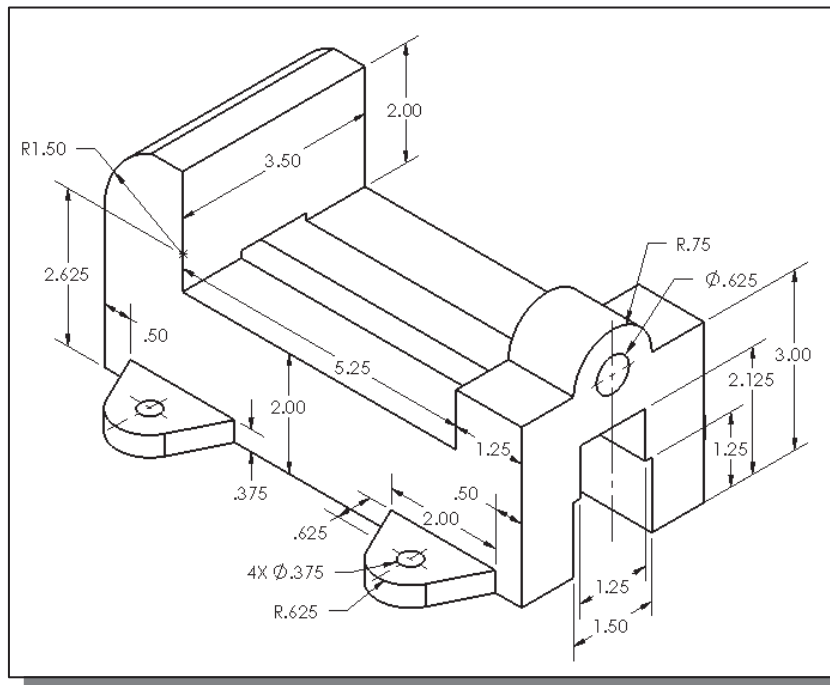
- Wheel Assembly** (Create a set of detail and assembly drawings. All dimensions are in mm.)



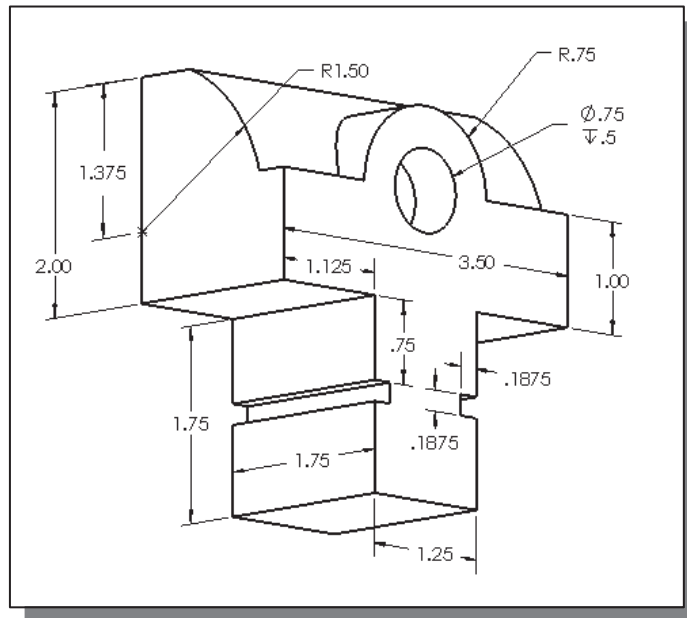
2. **Vise Assembly** (Create a set of detail and assembly drawings. All dimensions are in inches.)



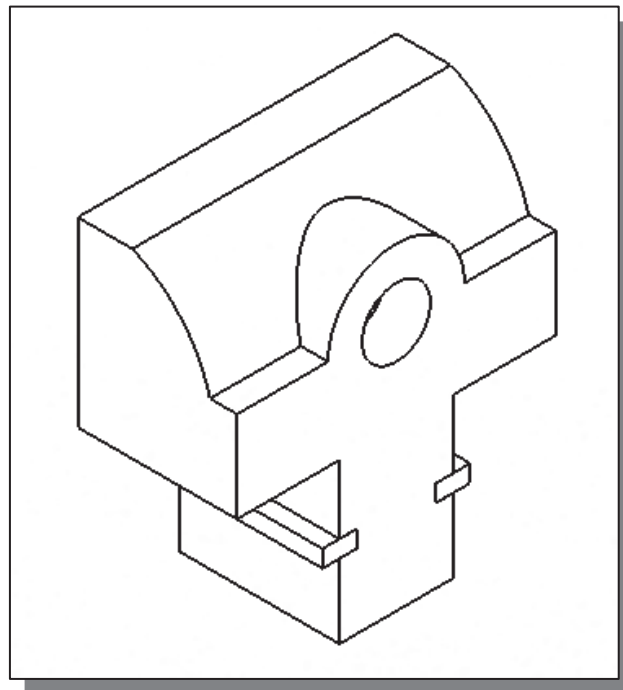
- (a) **Base:** The 1.5 inch wide and 1.25 inch wide slots are cut through the entire base.  
Material: **Gray Cast Iron.**



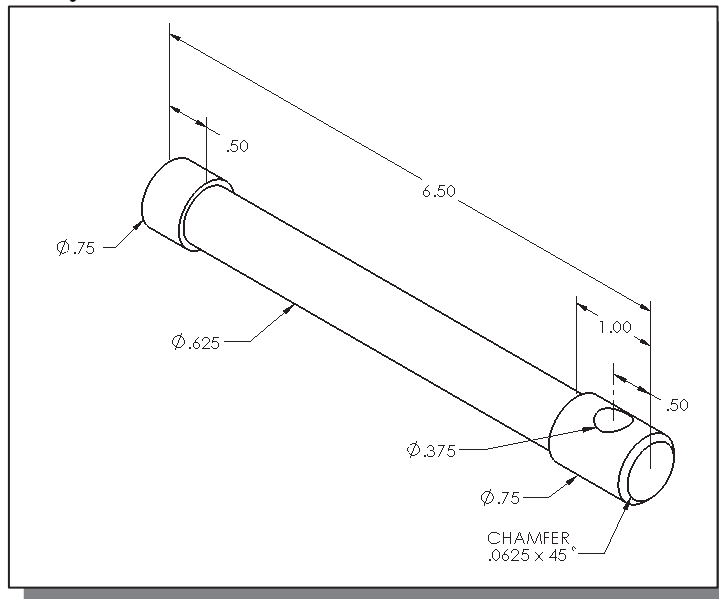
- (b) **Jaw:** The shoulder of the jaw rests on the flat surface of the base and the jaw opening is set to 1.5 inches. Material: **Gray Cast Iron.**



- (c) **Key:** **0.1875 inch H x 0.375 inch W x 1.75 inch L.** The keys fit into the slots on the jaw with the edge faces flush as shown in the sub-assembly to the right. Material: **Alloy Steel.**



- (d) **Screw:** There is one chamfered edge (0.0625 inch x 45°). The flat  $\varnothing 0.75$ " edge of the screw is flush with the corresponding recessed  $\varnothing 0.75$  face on the jaw.  
Material: **Alloy Steel.**



- (e) **Handle Rod:**  $\varnothing 0.375$ " x 5.0" L. The handle rod passes through the hole in the screw and is rotated to an angle of 30° with the horizontal as shown in the assembly view. The flat  $\varnothing 0.375$ " edges of the handle rod are flush with the corresponding recessed  $\varnothing 0.735$  faces on the handle knobs. Material: **Alloy Steel.**
- (f) **Handle Knob:** There are two chamfered edges (0.0625 inch x 45°). The handle knobs are attached to each end of the handle rod. The resulting overall length of the handle with knobs is 5.50". The handle is aligned with the screw so that the outer edge of the upper knob is 2.0" from the central axis of the screw. Material: **Alloy Steel.**

