

# Assembly Drawings

## Objectives

- Create an assembly drawing from a presentation file.
- Complete a parts list for an assembly drawing.
- Change the material type of a component.
- Extract a bill of materials from an assembly model.

ITEM	PART NUMBER	MATERIAL	QTY
5	Plate	Rubber	1
4	Handle	Brass, Soft Yellow	1
3	Stem	Galvanized Steel	1
2	Top Bar	Copper Alloy	1
1	Base	Steel, Mild	1

DATE	ISSUED BY	YOUR INSTITUTION
<b>PRESS ASSEMBLY</b>		
DATE	ISSUED BY	YOUR INSTITUTION
Press Assembly Drawing		

## Vocabulary

assembly drawings  
balloons

parts list

bill of materials (BOM)

## Creating an Assembly Drawing

Two-dimensional **assembly drawings** are used much like exploded assemblies: they provide information about part identification within an assembly. Assembly drawings can be generated from unexploded assemblies, but exploded assemblies usually provide a better view. In general, therefore, the first step in creating an assembly drawing in Inventor is to build an exploded assembly using the presentation modeler. In this chapter, you will generate an assembly drawing based on the exploded assembly you created in Chapter 13.

1. Open the Press Exploded Assembly.ipn file from Chapter 13.
2. Pick the arrow next to the New button on the Inventor Standard toolbar and pick Drawing from the list.
3. Pick the + beside Drawing Resources and Sheet Formats, and double-click to insert a new B size, 2 view sheet. The Press Exploded Assembly.ipn file should be selected by default.

A front view of the exploded assembly appears, but it extends off the drawing sheet. Fig. 14-1.

4. Delete the two views.
5. The default title block is ANSI - Large. You do not need a large title block for this drawing, so pick the + next to Sheet:2, select the title block, and delete it.
6. Pick the + next to Title Blocks and double-click ANSI A to insert a new title block.
7. Select Base View in the Drawing Views Panel.
8. In the Orientation list box, select Current to insert the position shown in the presentation file.
9. Set the scale to .5 and pick a point on the drawing sheet to place the view. The assembly should appear as shown in Fig. 14-2.

Fig. 14-1

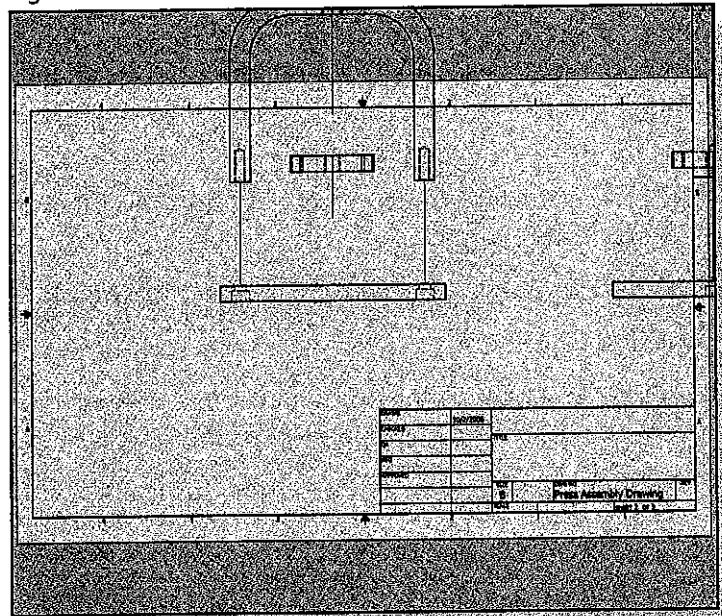
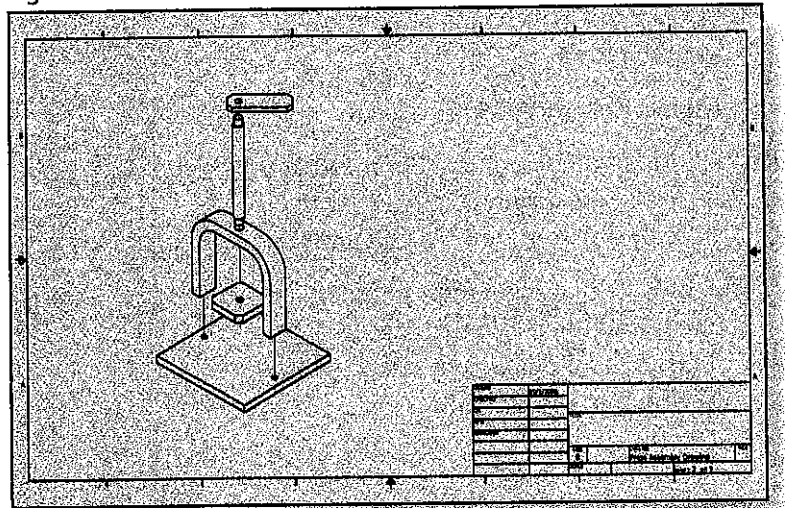


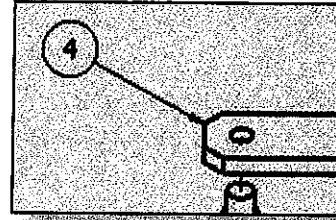
Fig. 14-2



## Adding Balloons

Like all drawings, assembly drawings need annotation to clarify the views. However, dimensions, notes, and labels tend to clutter the drawing and may become confusing, especially for assemblies that contain many parts. To keep them uncluttered and easy to read, assembly drawings contain **balloons**—small circles that contain the part numbers and are connected by leaders to various parts of an assembly. Fig. 14-3.

Fig. 14-3



One balloon is assigned to each type of component in the assembly drawing. For example, an assembly that consists of a plate, a clamp, and two screws will have three balloons: one for the plate, one for the clamp, and one for the screws.

The parts are numbered consecutively starting with the number 1. By convention, the first number is assigned to the largest or most important piece in an assembly; the second is assigned to the second-largest, and so on. Inventor can assign part numbers automatically to the components you select.

1. Pick the down arrow next to Drawing Views Panel and select Drawing Annotation Panel.
2. Scroll down in the Drawing Annotation Panel and select Balloon.
3. Pick the base.

The BOM Properties dialog box appears. The options it provides apply to complex assemblies and are not relevant to a simple assembly such as this one.

4. Without changing any of the settings, pick OK to close the BOM Properties dialog box.
5. Pick OK in the BOM View Disabled confirmation box.
6. Pick a point on the drawing sheet to place the balloon, as shown in Fig. 14-4.
7. Right-click and pick Continue.

The balloon, which contains the number 1, appears at the location you chose. Notice that the **Balloon** command is still active.

8. Continue picking components to place balloons. Refer to Fig. 14-5 for their general placement and the order in which to place them.

Fig. 14-4

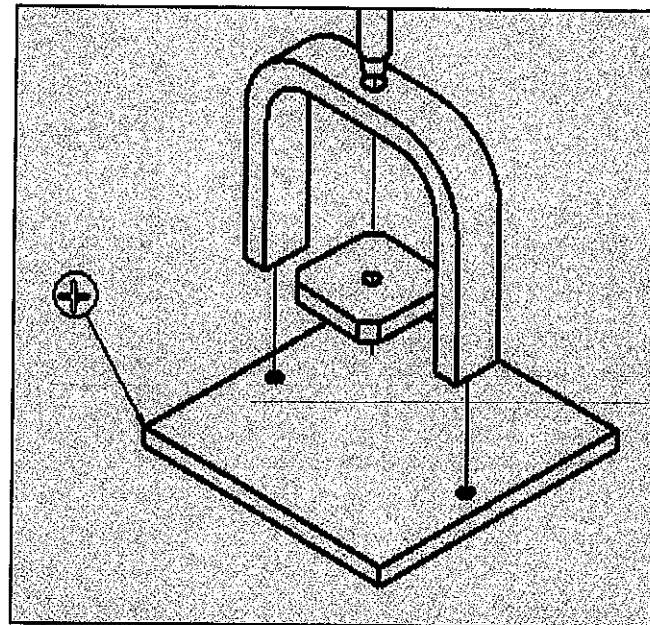
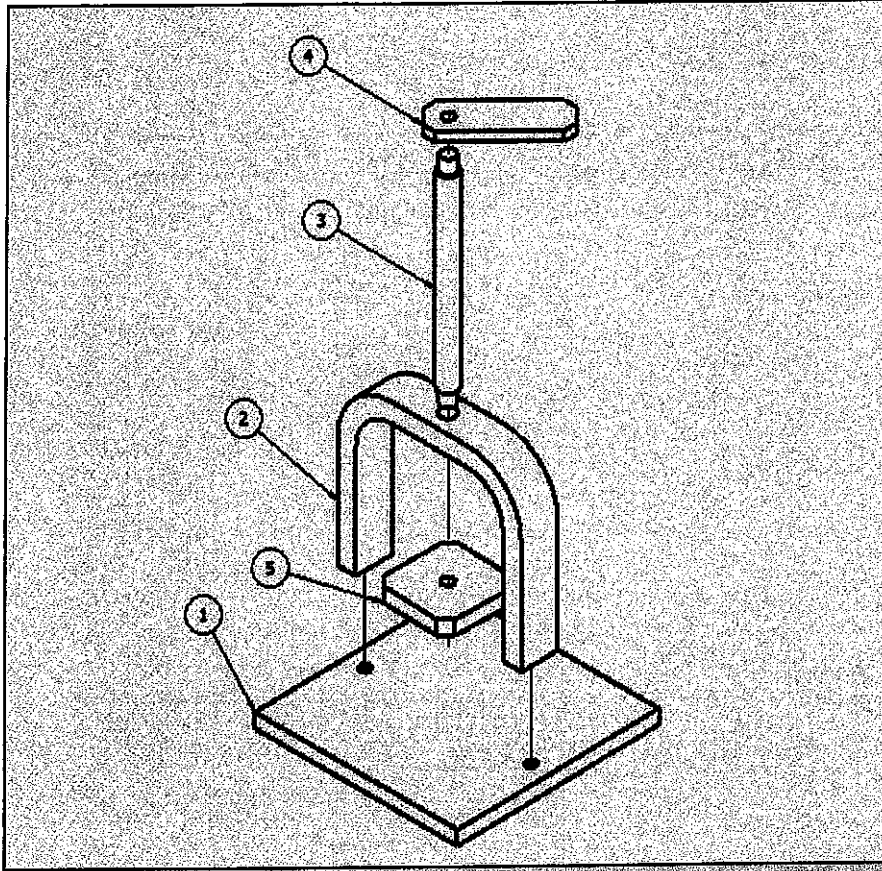


Fig. 14-5



9. Save the drawing file as Press Assembly Drawing.idw.

Notice that the line of balloons is a bit ragged. This is perfectly acceptable in many industries. However, in a formal assembly drawing, the balloons are generally aligned and grouped in series of three.

10. Move the cursor over the number 5 balloon to activate its grips.
11. Pick the grip in the balloon and move it over the number 1 balloon to acquire the center point of that balloon. Fig. 14-6.
12. Drag the number 5 balloon upward so that a dotted line extends between the two balloons. The balloons are now exactly aligned. Pick a point as shown in Fig. 14-7 to place the number 5 balloon.

Fig. 14-6

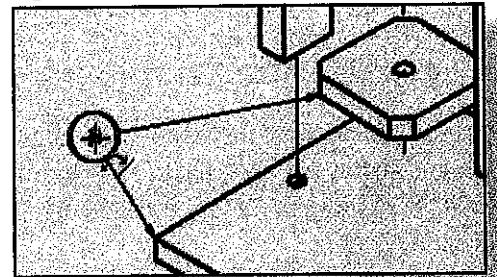
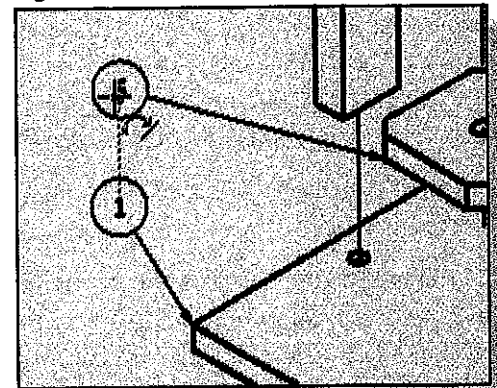
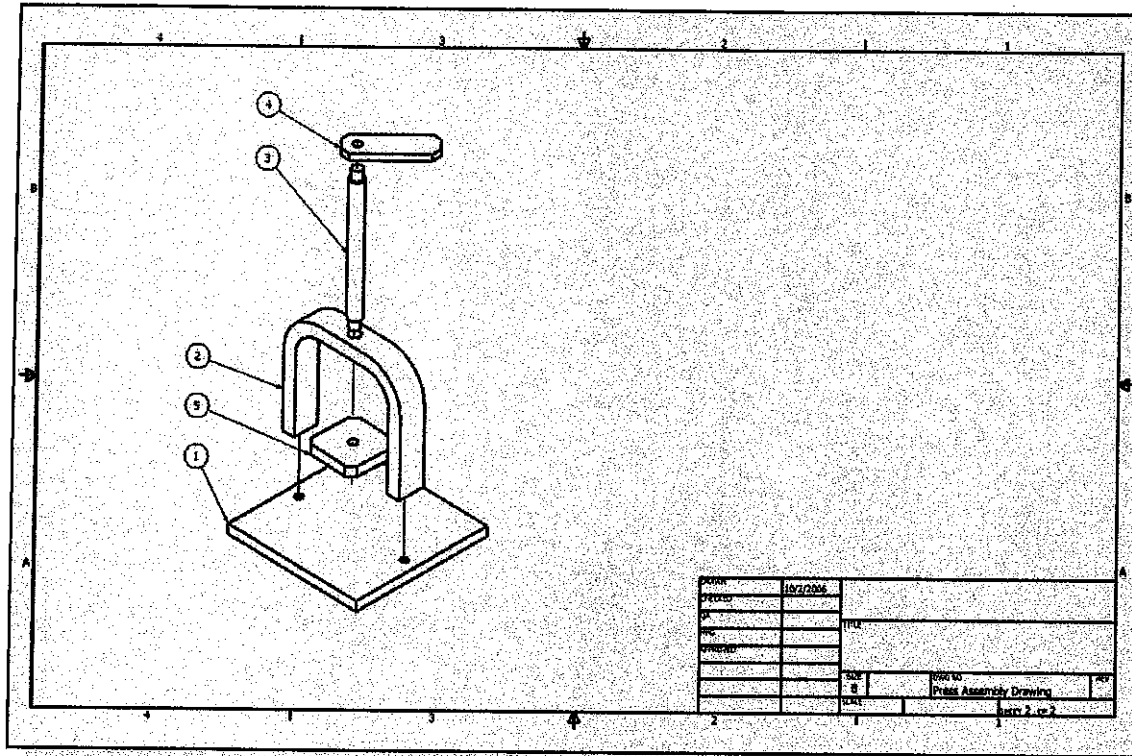


Fig. 14-7



13. Position the remaining balloons so that they are grouped and aligned as shown in Fig. 14-8.

Fig. 14-8



## Preparing the Parts List

Assembly drawings include a **parts list**, which includes details such as part numbers and quantities for each part in the assembly. Parts lists are usually located either in the upper right corner of the drawing sheet or in the lower right corner, just above the title block.

If the parts list is in the upper right corner, parts are listed by part number in order from 1 to the highest number. If the parts list is above the title block, however, parts are listed in the reverse order, with part 1 being closest to the title block. This is done so that there is space after the highest number in case other parts need to be added later. Inventor can place the parts in either order. For the press assembly drawing, we will place the parts list above the title block.

1. In the Drawing Annotation Panel, scroll down and pick the Parts List command.

The Parts List dialog box appears. At the top of the dialog box, the Select View button is selected, allowing you to select the view or views to use in creating the parts list. Fig. 14-9. Alternatively, you can select a document from the drop-down list.

Fig. 14-9

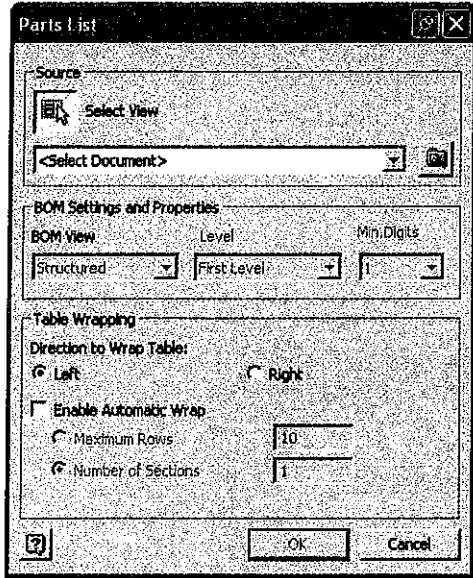
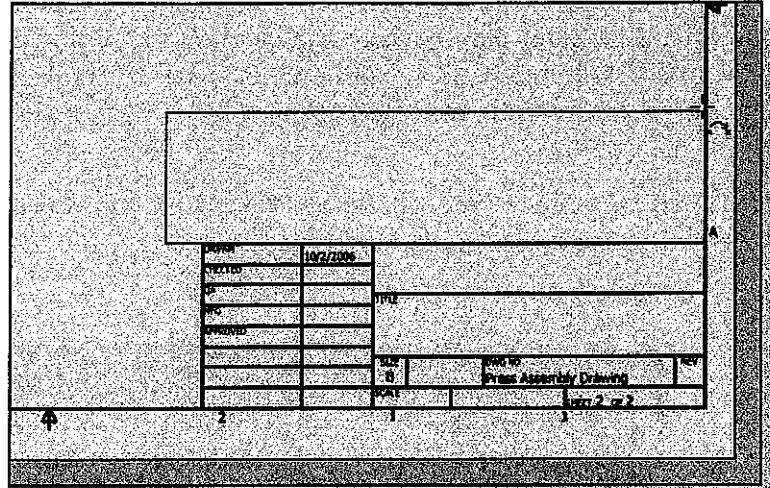


Fig. 14-10

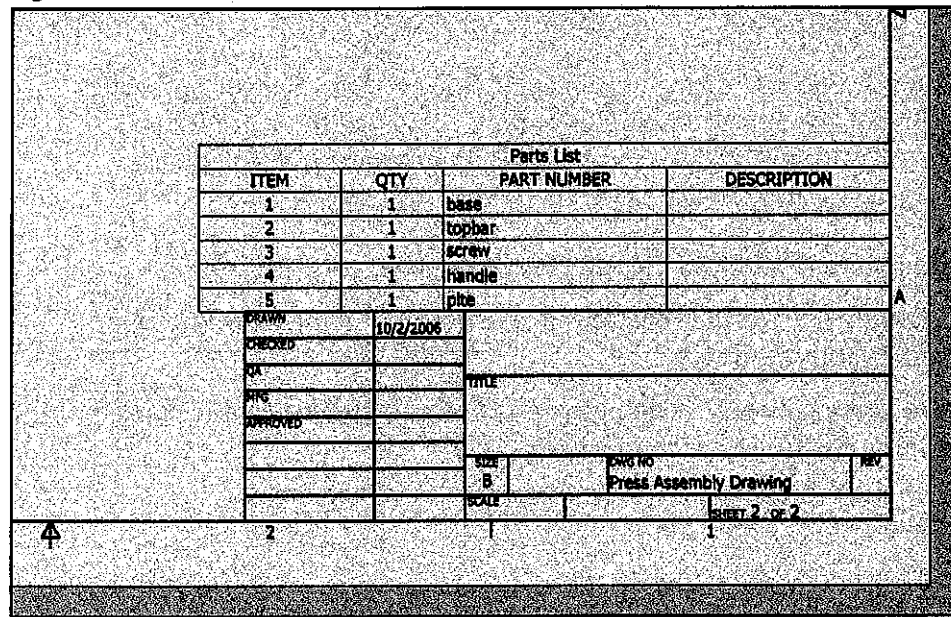


2. Pick the exploded view in the drawing as the view to be used for the parts list and pick OK to close the dialog box.

3. Position the parts list outline box on the right edge of the drawing so that it is just above the title block, coincident with the border, as shown in Fig. 14-10.

The parts list generates, as shown in Fig. 14-11. Inventor has counted the number of each component used and has listed beside the part number the quantity needed to build the part. The part numbers correspond to the balloon numbers in the drawing.

Fig. 14-11

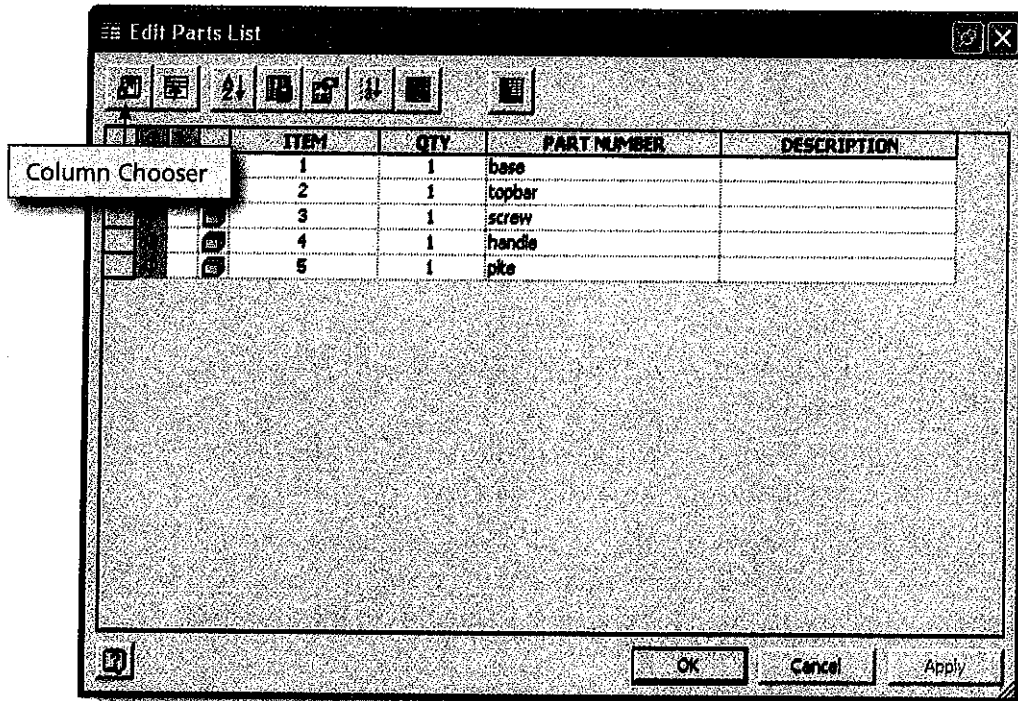


## Editing the Parts List

By default, Inventor parts lists contain columns for item number, quantity, part number, and description. However, other types of information are available. You can edit the parts list to include items such as material, project, revision number, and many other items.

1. Place the cursor on the parts list, right-click, and select Edit Parts List.
2. In the Edit Parts List dialog box, pick the Column Chooser button. Fig. 14-12.

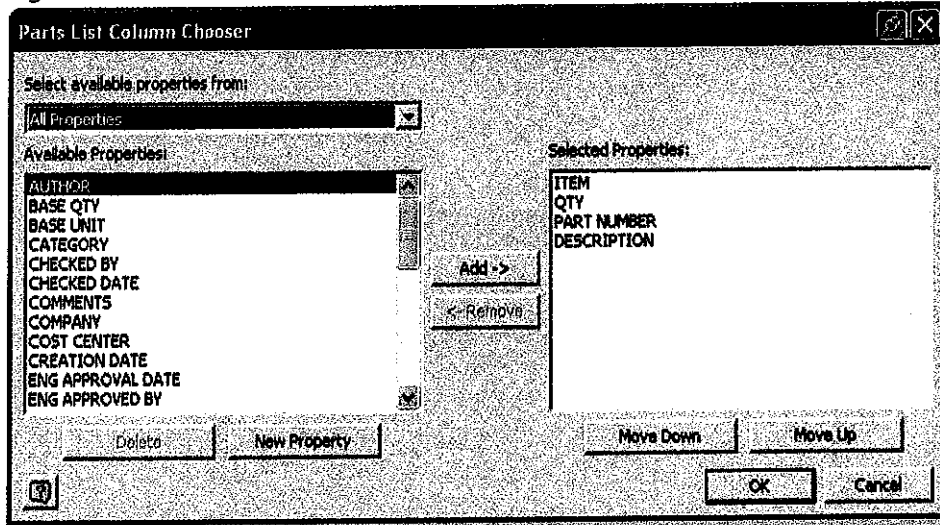
Fig. 14-12



The Parts List Column Chooser dialog box appears. Fig. 14-13. Notice the properties in the Available Properties list. You can add columns for any of these properties to the parts list. You can also specify the order in which they appear in the parts list.

3. Scroll down the Available Properties list and select MATERIAL.
4. Pick the Add button to add it to the Selected Properties list.
5. Highlight DESCRIPTION in the Selected Properties list and pick the Remove button to remove it from the list.
6. Use the Move Down and Move Up buttons as necessary to display the items in the Selected Properties list in the following order:  
ITEM  
PART NUMBER  
MATERIAL  
QTY

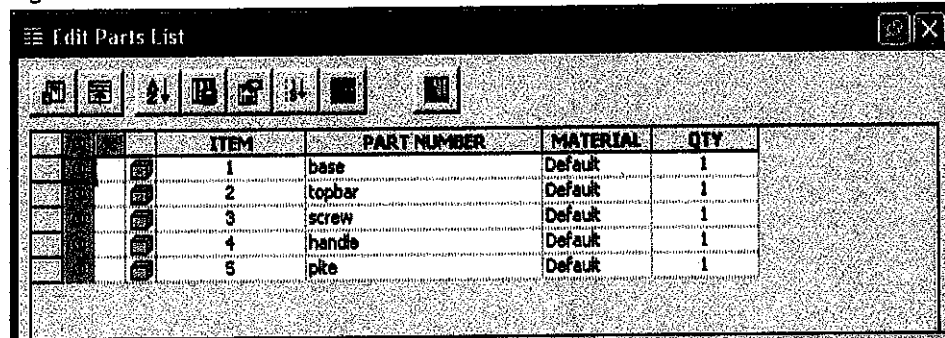
Fig. 14-13



7. Pick OK to accept the settings.

The parts list adjusts automatically to the new settings in the Edit Parts List dialog box, as shown in Fig. 14-14.

Fig. 14-14



In the Part Number column, Inventor picks up part of the file name you chose when you created the individual component models. Depending on how you named your files, these names may be difficult to understand. You can change the text to make it more readable.

8. Double-click the text in the first line to activate the text cursor and change the part name (if necessary) to Base.
9. Repeat step 8 to rename the other components Top Bar, Screw, Handle, and Plate.

Refer again to Fig. 14-11 on page 321. Notice that the parts are in the wrong order; they should be listed with part number 1 at the bottom. This is easy to change.

**10.** Pick the Table Layout button, as shown in Fig. 14-15.

Fig. 14-15

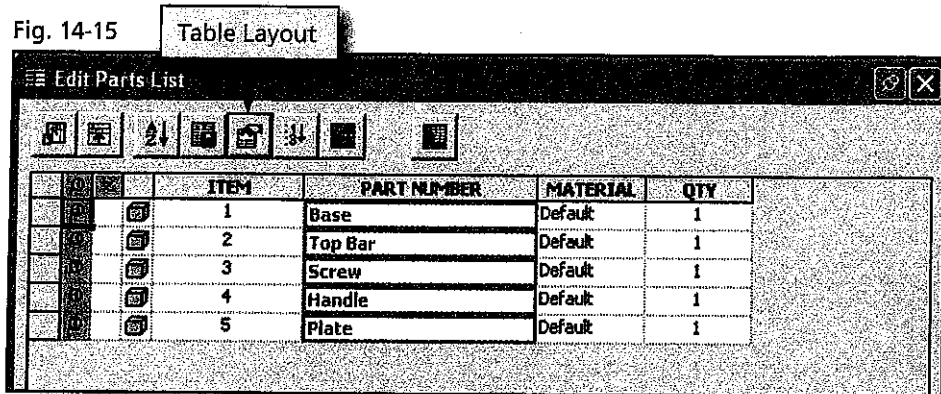
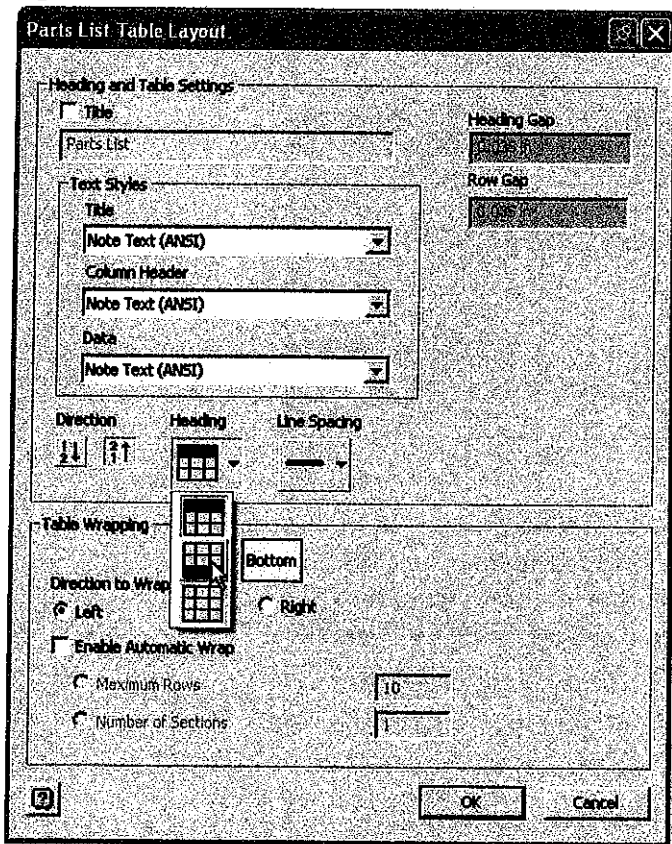


Fig. 14-16



**11.** In the Direction area of the Parts List Table Layout dialog box, pick the second button. (The tooltip for this button is Add new parts to top.)

**12.** Pick the down arrow under Heading and select the icon that shows the green heading bar at the bottom of the table. Fig. 14-16.

**13.** Pick to remove the check mark next to Title.

**14.** Pick OK to accept the settings.

**15.** Pick OK again to close the Edit Parts List dialog box.

The changes take effect in the assembly drawing. Because the title is no longer being shown, there is now a gap between the parts list and the title block.

**16.** Move the parts list down so that it rests against the title block, as shown in Fig. 14-17.

Fig. 14-17

ITEM	PART NUMBER	MATERIAL	QTY
5	Plate	Default	1
4	Handle	Default	1
3	Screw	Default	1
2	Top Bar	Default	1
1	Base	Default	1

DRAWN Jody James	10/2/2006	
CHECKED		
QA		TITLE
WFS		
APPROVED		
	SIZE B	PART NO Press Assembly Drawing
	SCALE	REV
		SHEET 2 of 2

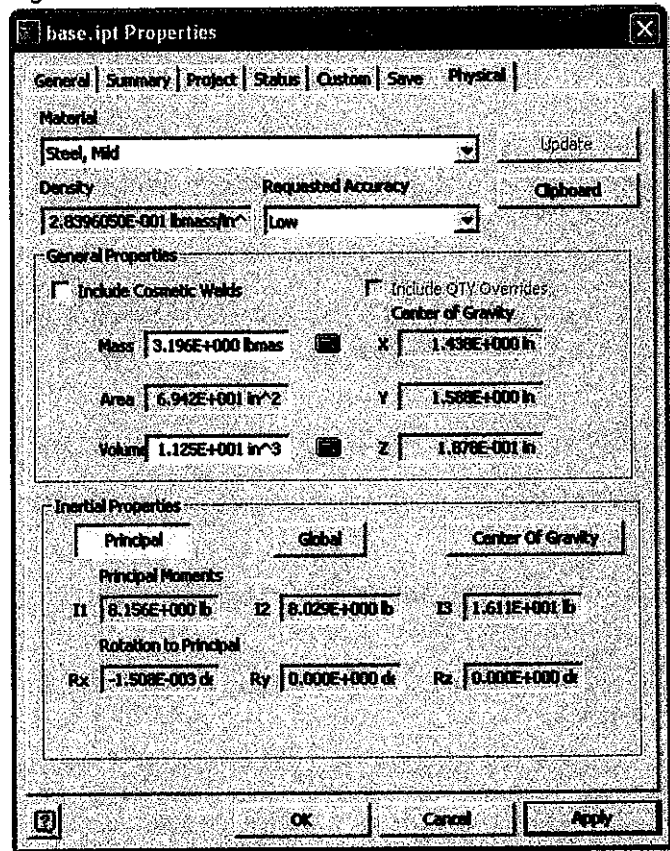
## Changing the Material Type

Notice in the parts list that the material for every part in the assembly is listed as Default. Inventor provides many standard material types that can be attached to individual parts in an assembly.

1. Switch to the Press Exploded Assembly.ipn window.
2. In the part browser, pick the + next to Explosion1 and Press Assembly.iam to display the components in the assembly.
3. Right-click Base:1 and pick Open to open the Base.ipt file for editing.
4. Right-click the name of the file in the part browser and select iProperties to display the properties of the base model.
5. In the Base.ipt Properties dialog box, pick the Physical tab.
6. Pick the down arrow in the Material area and select Steel, Mild.

Notice the values in the Density, General Properties, and Inertial Properties areas. Fig. 14-18. These values reflect the actual material properties for mild steel.

Fig. 14-18



7. Change the material to Titanium and notice the changes in the properties.
8. Change the material back to Steel, Mild.
9. Pick OK to accept the settings.
10. In the drop-down box on the Inventor Standard toolbar, pick the down arrow and change it back to As Material.
11. Save and close the Base.ipt file.

The base will now display in the assembly in a color and finish that resembles mild steel.

12. Switch back to the Press Assembly Drawing.idw window and notice that the material information in the parts list has been updated.
13. Open each of the other components and repeat the process to change the materials for the other parts in the assembly to the materials shown in Fig. 14-19.

Fig. 14-19

5	Plate	Rubber	1
4	Handle	Brass, Soft Yellow	1
3	Screw	Galvanized Steel	1
2	Top Bar	Copper Alloy	1
1	Base	Steel, Mild	1
ITEM	PART NUMBER	MATERIAL	QTY

14. Switch back to the Press Assembly Drawing.idw file.

Because the names of some of the materials are longer than the column width, they may now take two lines. You can fix this by adjusting the column widths.

15. Position the cursor over the vertical line on the left side of the MATERIAL column until a double-headed arrow appears. Move the line to the left until the name of each material occupies only one line.

## Completing the Assembly Drawing

The assembly drawing is now almost complete. You are now ready to add the finishing touches.

1. Close the Press Exploded Assembly.ipn file. (Do not close the Press Assembly Drawing.idw file.)
2. Open the Press Assembly.iam file and rotate the assembly to show an isometric view.
3. Switch back to the Press Assembly Drawing.idw file.
4. In the Drawing Views Panel, pick Base View. The File drop-down box should default to the open Press Assembly.iam file.

Fig. 14-20

ITEM	PART NUMBER	MATERIAL	QTY
5	Plate	Rubber	1
4	Handle	Brass, Soft Yellow	1
3	Screw	Galvanized Steel	1
2	Top Bar	Copper Alloy	1
1	Base	Steel, Mkd	1

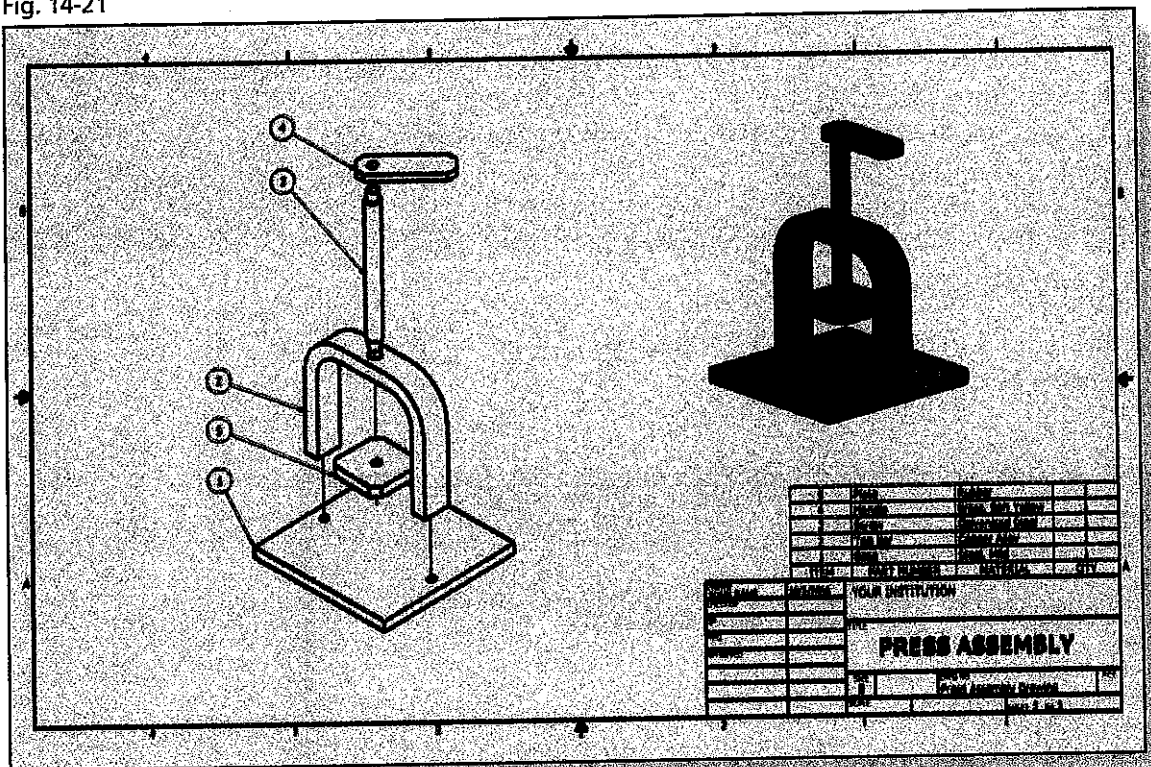
  

DATE	10/2/2006	YOUR INSTITUTION	
YOUR NAME		TITLE	
		<b>PRESS ASSEMBLY</b>	
		SIZE	DWG NO
		B	Press Assembly Drawing
		SCALE	Part 2 of 2

5. Change Orientation to Current and Style to Shaded.
6. Set the scale at .5 and pick a point above the parts list on the drawing sheet.
7. Fill in the information in the title block to complete the drawing, as shown in Fig. 14-20.

Your completed assembly drawing should appear similar to the one in Fig. 14-21.

Fig. 14-21



## 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. Although the terms *parts list* and *bill of materials* are often used interchangeably, a parts list is generally used in an assembly drawing and is a partial list of the associated BOM.

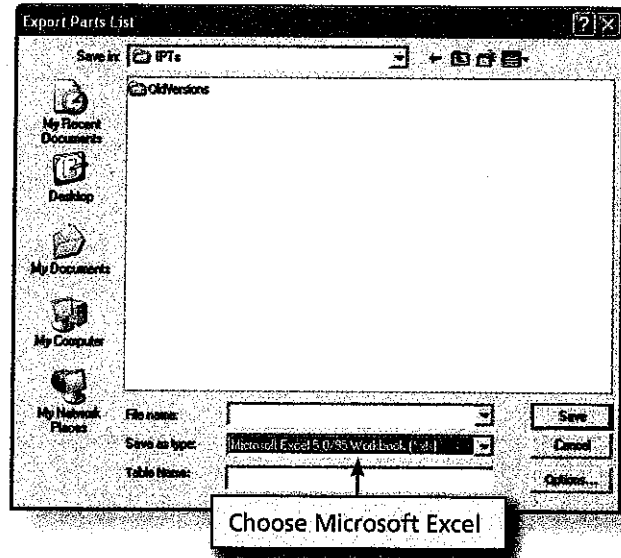
In Inventor, the bill of materials can be derived directly from the assembly model. You can select which properties are included in the bill of materials, in what order the information is presented, and in what format to export the information. The exported file can be used in an application such as a spreadsheet or text editor.

Inventor provides two methods for creating a BOM. You can derive it directly from the parts list on an assembly drawing, or you can create it from the assembly model.

### Deriving the BOM from a Parts List

1. Position the cursor over the parts list in the assembly drawing.
2. Right-click and select Export.
3. In the Export Parts List dialog box, pick the down arrow next to the Save as type box and select Microsoft Excel 5.0/95 Workbook (\*.xls), as shown in Fig. 14-22.
4. Enter Press Assembly.xls as the filename and pick Save to export the BOM.
5. If you have access to the Microsoft Excel® software, open the Press Assembly.xls file and examine it.

Fig. 14-22



### Creating a BOM from the Assembly Model

1. Switch to the Press Assembly.iam window.
2. From the Tools pull-down menu, select Bill of Materials to display the Bill of Materials dialog box. Fig. 14-23.

Fig. 14-23

The image shows the 'Bill of Materials' dialog box. The table contains the following data:

Part Number	BOM Struct...	Unit	QTY	Stock Num...	Description	REV
base	Normal	Each	1			
topbar	Normal	Each	1			
screw	Normal	Each	1			
handle	Normal	Each	1			
pte	Normal	Each	1			

Notice that many of the controls and options in the Bill of Materials dialog box are similar to those displayed when you enter the Parts List command in drawing mode. They are currently grayed out because we disabled the Structured BOM option earlier in this chapter.

3. Pick the Structure (Disabled) tab.
4. Right-click the tab and pick Enable BOM View.

The parts appear in the Structured tab and all of the options become available.

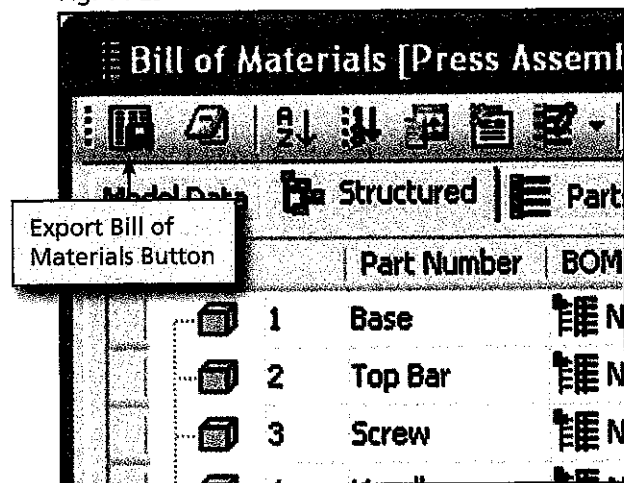
5. Clean up the names of the parts in the Part Number column.
6. Click inside the Stock Number box in the first row and enter BA0641 as the new stock number.
7. Add stock numbers for the other parts, as shown in Fig. 14-24.

Fig. 14-24

Item	Part Number	BOM Struct	Unit	QTY	QTY	Stock Num	Description	REV
1	Base	Normal	Each	1		BA0641		
2	Top Bar	Normal	Each	1		B78910		
3	Screw	Normal	Each	1		SW5548		
4	Handle	Normal	Each	1		HD0042		
5	Plate	Normal	Each	1		PL6938		

8. Pick the Export Bill of Materials button near the top of the Bill of Materials dialog box. Fig. 14-25.
9. In the Export Bill of Materials dialog box, accept the defaults by picking OK.
10. Save the file as Press Assembly BOM.xls.
11. If you have access to the Microsoft Excel software, open the Press Assembly BOM.xls file and examine it.

Fig. 14-25

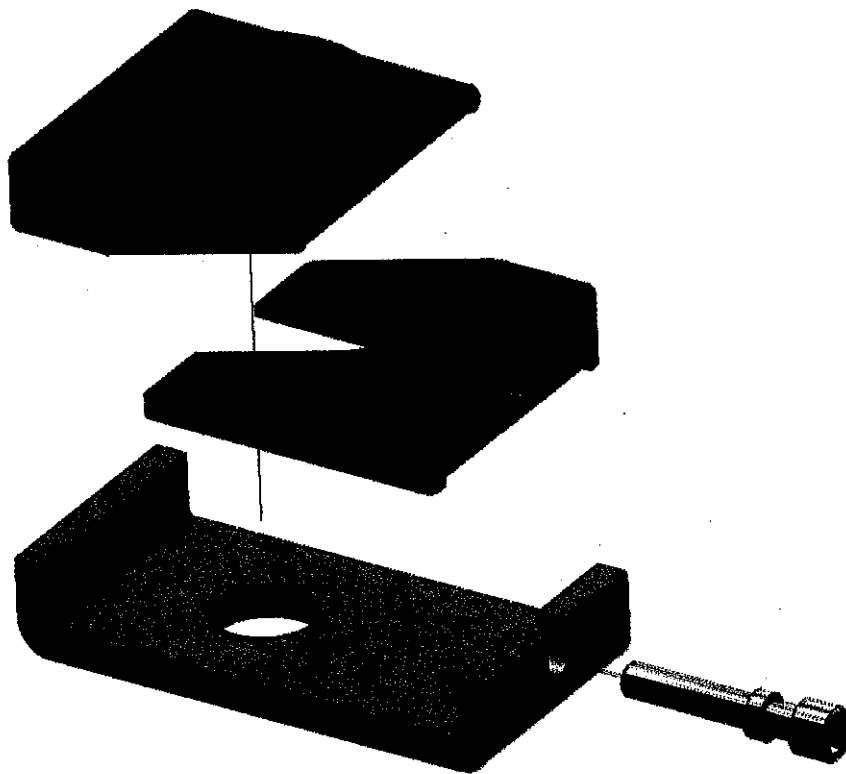
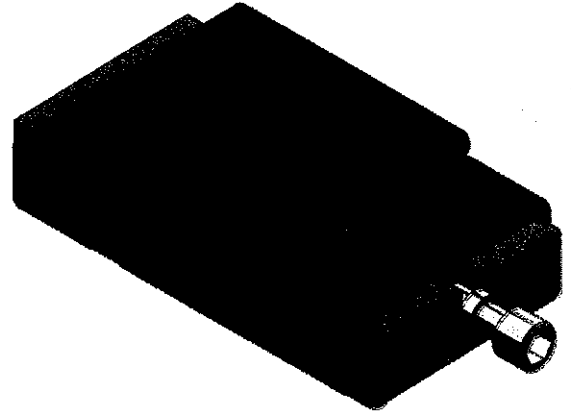


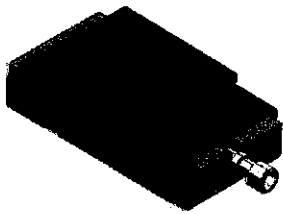
## Review Questions

1. What is the purpose of an assembly drawing?
2. What is a balloon in the context of an assembly drawing?
3. How are balloons grouped in a formal assembly drawing?
4. At what two locations on an assembly drawing might a parts list be placed?
5. In what order are parts listed in a parts list?
6. How can you adjust the information listed in the parts list of an assembly drawing?
7. Explain how to change the material properties of a part.
8. What is the difference between a parts list and a bill of materials?
9. Describe the procedure to create a bill of materials in Inventor.
10. Create sketches to show the steps you plan to use to create the four parts required for the leveling assembly in exercise 1 on page 331.

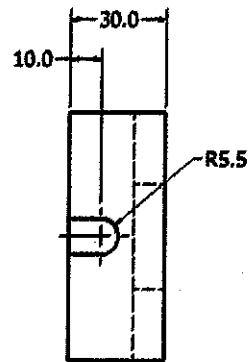
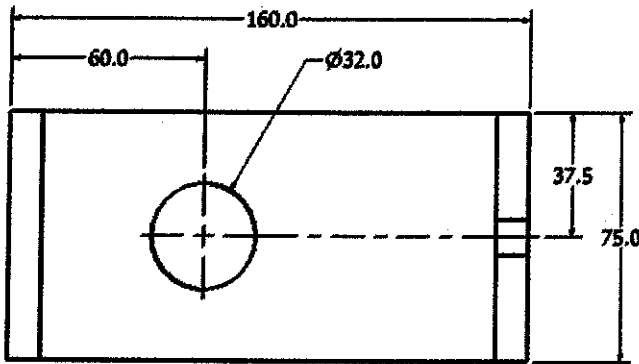
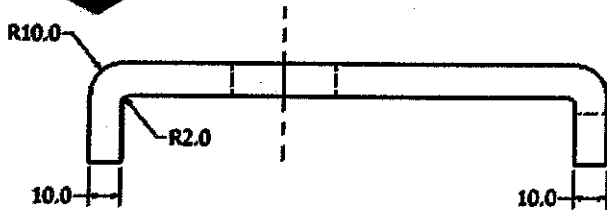
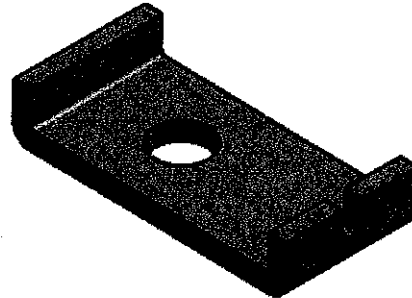
Create assemblies and assembly drawings as directed in exercises 1 and 2. *Note:* Both of these exercises use metric units.

1. Create each of the components shown on pages 332 and 333. Then create an assembly model, assembling the parts as shown below. Both the assembly and the exploded assembly are shown for your reference. Create the assembly, using appropriate constraints to constrain the parts together. Then explode the assembly and use it to create an assembly drawing.

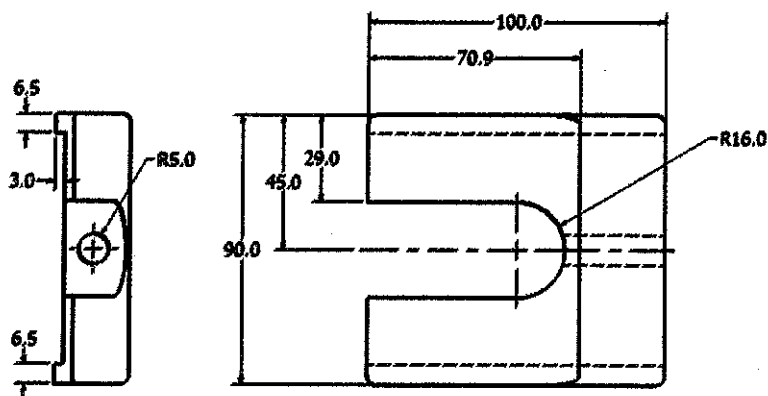
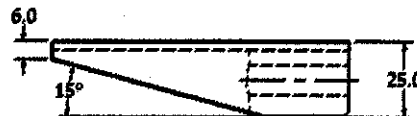
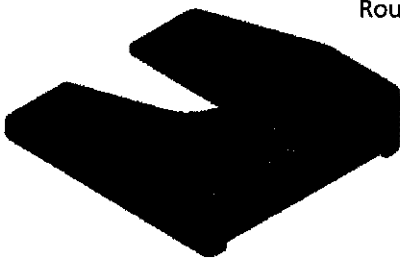




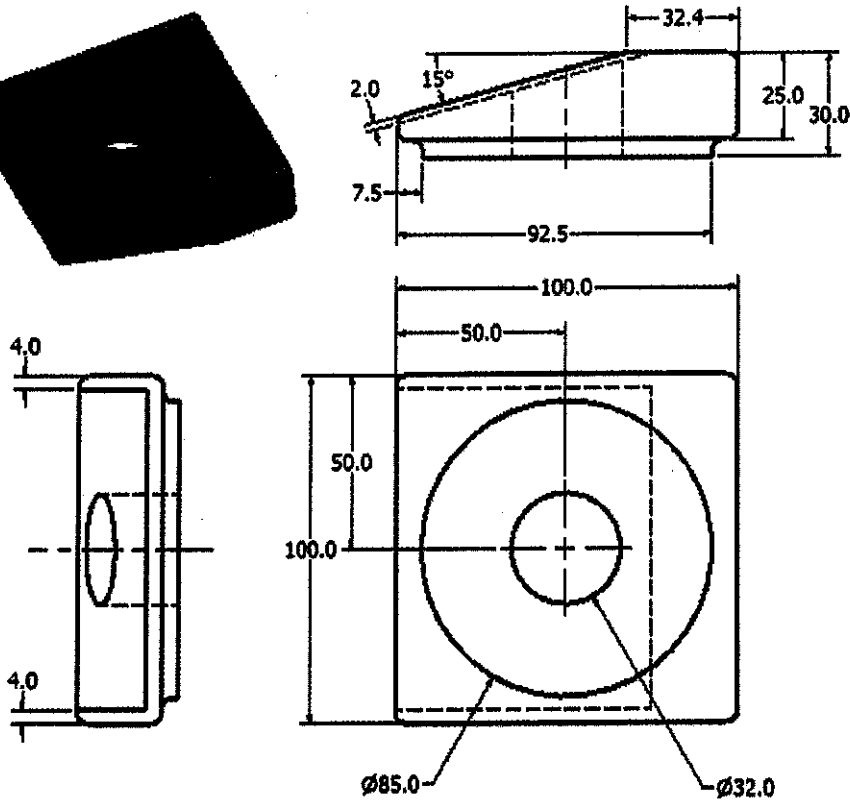
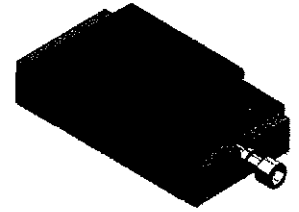
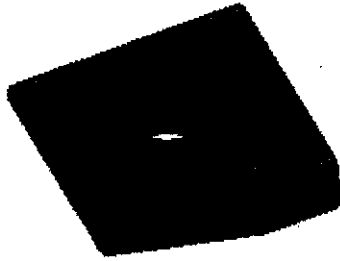
BASE PLATE



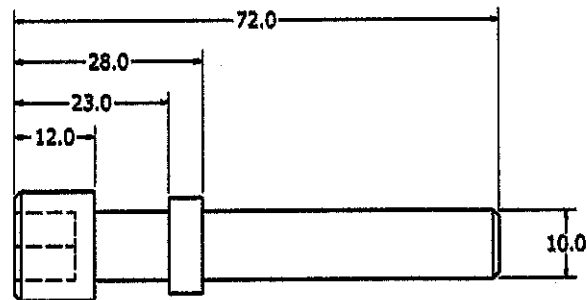
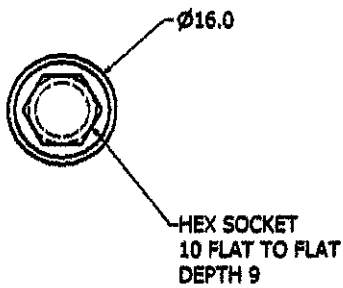
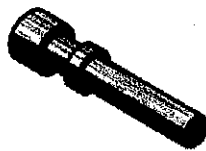
SLIDING BLOCK  
Rounds and Fillets R3



LIFTING BLOCK  
Rounds and Fillets R3



ADJUSTING SCREW  
(M10 X 15)  
Chamfers 45° X 1



2. Create each of the components for the leveling assembly using the specifications shown in the drawing. Then create an assembly model to show the pieces in their proper locations. Use constraints as necessary to place the components correctly. Then explode the assembly and create an assembly drawing.

