

# Assemblies

## Objectives

- Learn how to create assembly drawings.
- Learn how to create exploded assembly drawings.
- Learn how to create a parts list.
- Learn how to animate an assembly.
- Learn how to edit a title block.

## 5-1 INTRODUCTION

This chapter introduces the **Assembly** tools. These tools are used to create assembly drawings. Assembly drawings can be exploded to form isometric assembly drawings that when labeled and accompanied by a parts list become working drawings. Assembly drawings can be animated.

## 5-2 STARTING AN ASSEMBLY DRAWING

Figure 5-1 shows a test block. The overall dimensions for the block are  $80 \times 80 \times 80$  mm. The cutout is  $40 \times 40 \times 80$  mm. The test block will be used to help introduce the **Assembly** tools.

1. Start a new drawing.
2. Click the **Assembly** icon.
3. Click **OK**.

Draw and save as **Block, Test**.

The **Begin Assembly** box will appear. See Figure 5-2.

5. Click the **Browse . . .** box.

The **Open** box will appear. See Figure 5-3.

6. Click **Block, Test**, then click **Open**.

The test block will appear on the screen. See Figure 5-4.

7. Click the **Insert Components** tool and insert a second block.

See Figure 5-5. The first block inserted is fixed in place. In any assembly drawing the first component will automatically be fixed in place. Note the **(f)** notation to the left of **Block, Test <1>**. See Figure 5-6. As the assembly is created, components will move to the fixed first component.

### TIP

To remove the fixed condition, locate the cursor on the **(f) Block, Test <1>** callout, right-click the mouse, and click the **Float** option. To return the block to the fixed condition or to fix another component, right-click the component name callout and select the **Fix** option.

## 5-3 MOVE COMPONENT

See Figure 5-7.

1. Click the **Move Component** tool.
2. Click the second block inserted and hold down the left mouse button.

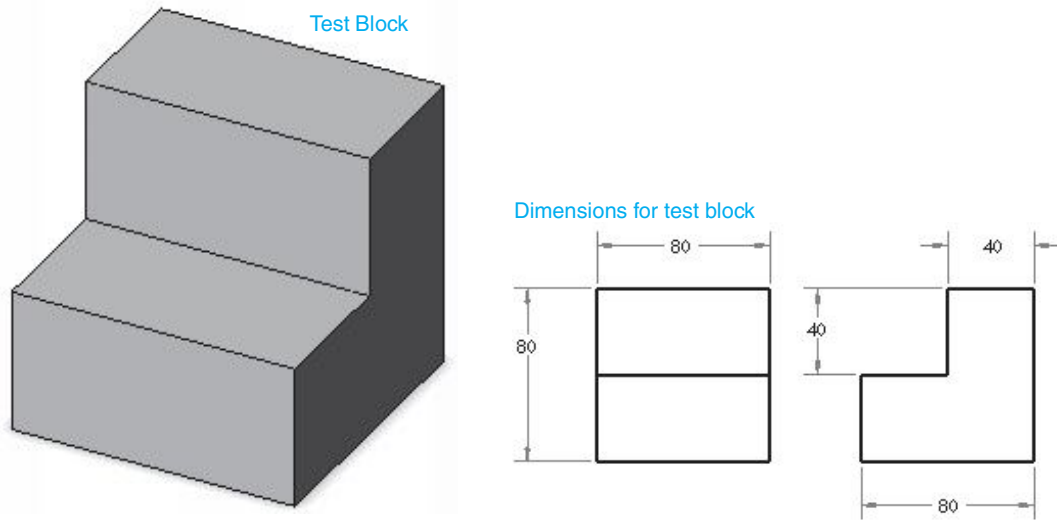


Figure 5-1

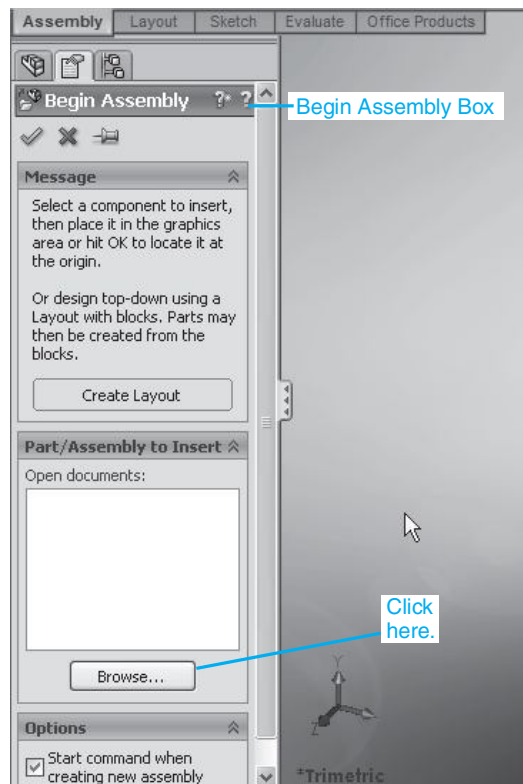


Figure 5-2

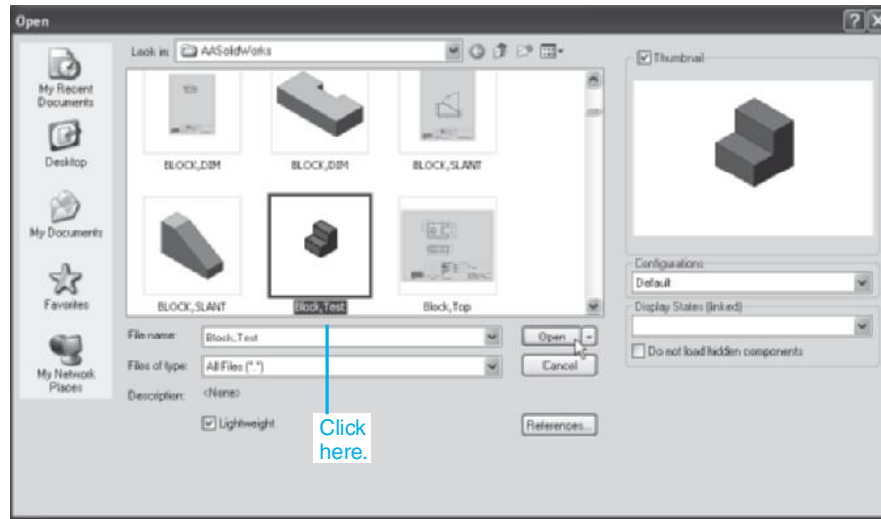


Figure 5-3

3. While holding down the left button, move the block around the screen by moving the mouse.
4. Release the button and click the OK check mark.

3. While holding down the left button, rotate the block around the screen by moving the mouse.
4. Release the button and click the OK check mark.
5. Use the **Undo** tool to return the block to its original position.

## 5-4 ROTATE COMPONENT

See Figure 5-8.

1. Click the **Rotate Component** tool.
2. Click the second block inserted and hold down the left mouse button.

## 5-5 MATE

The **Mate** tool is used to align components to create assembly drawings. See Figure 5-9.

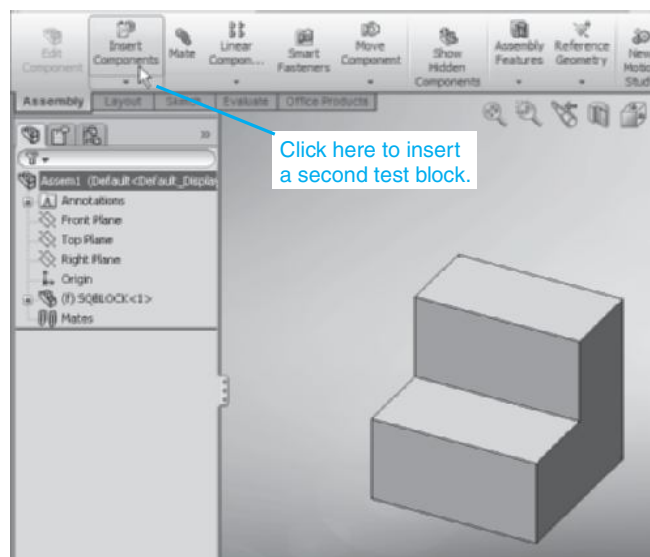
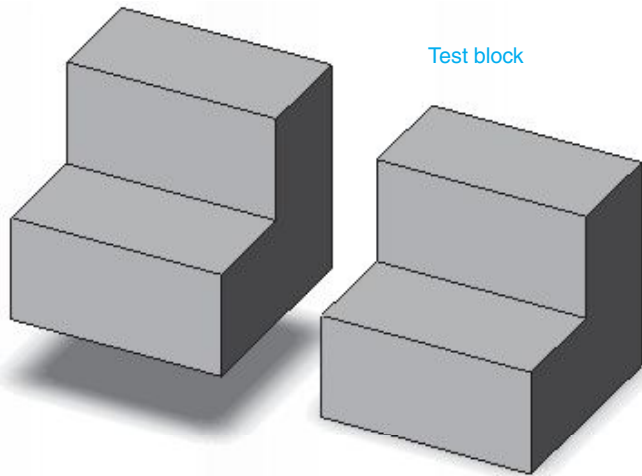


Figure 5-4



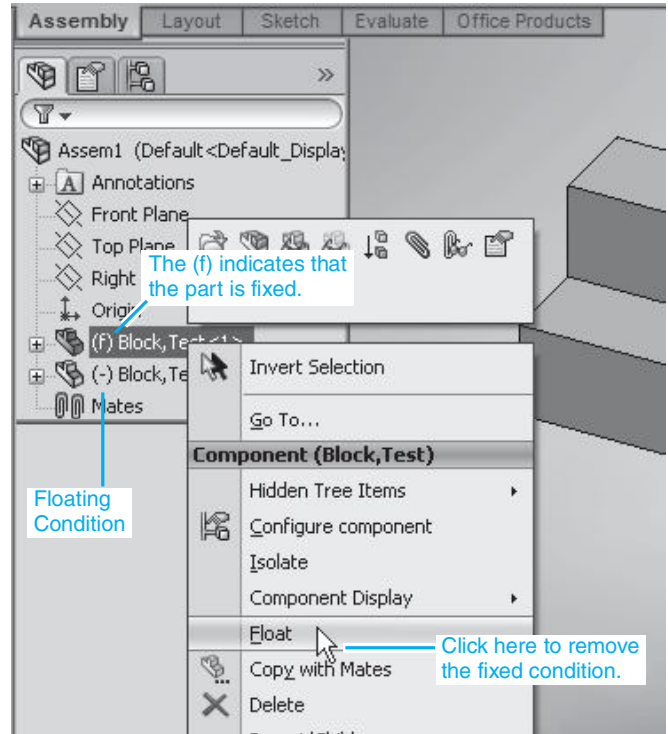
**Figure 5-5**  
**First Assembly**

Mate the two test blocks side by side.

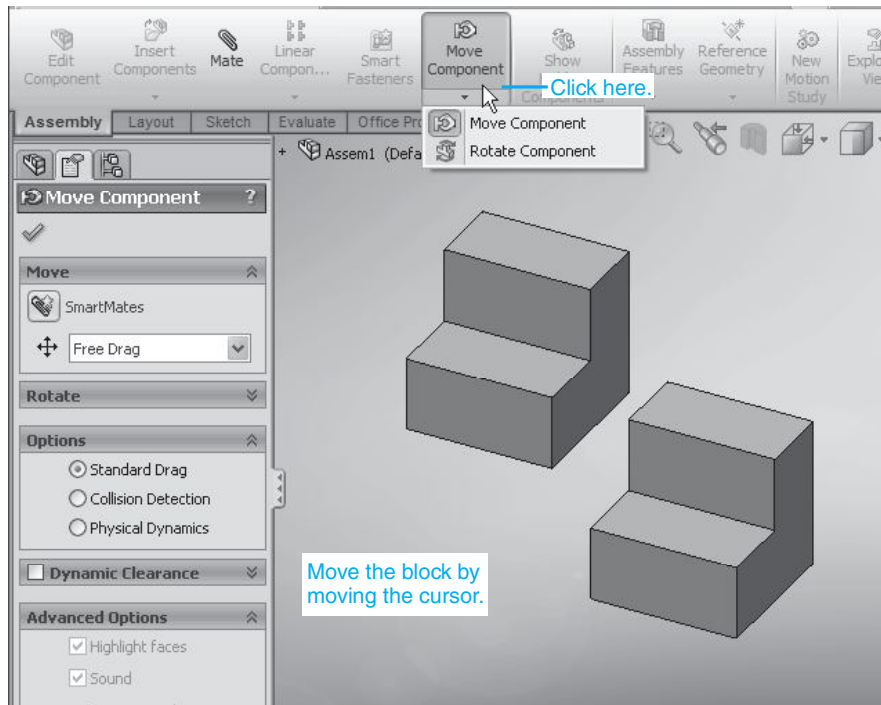
1. Click the **Mate** tool.
2. Click the upper right edge of the second block inserted.

See Figure 5-10. Note that the second block is listed in the **Mate Selections** box after it is selected.

3. Click the upper left edge on the first block.



**Figure 5-6**



**Figure 5-7**

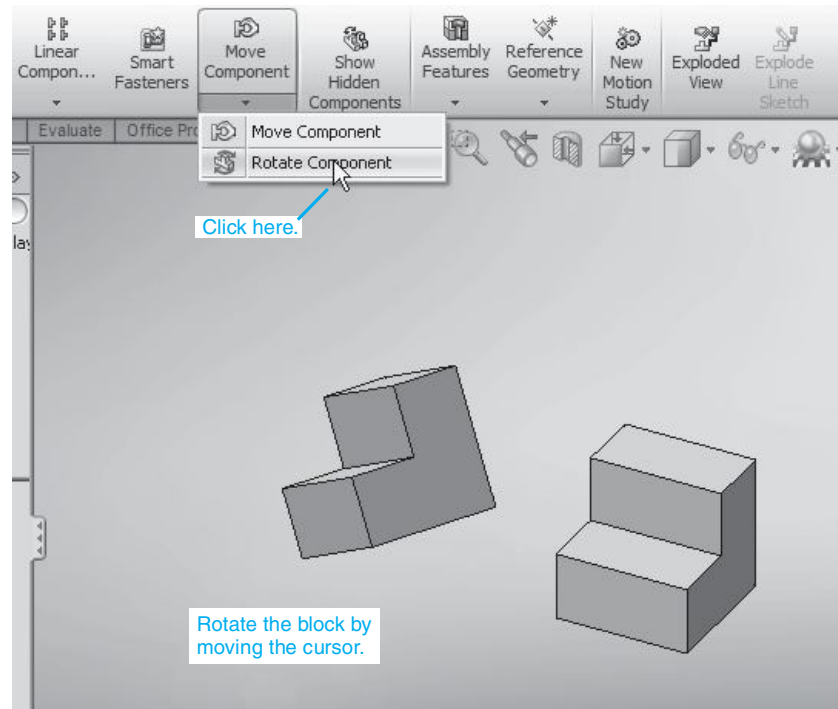


Figure 5-8

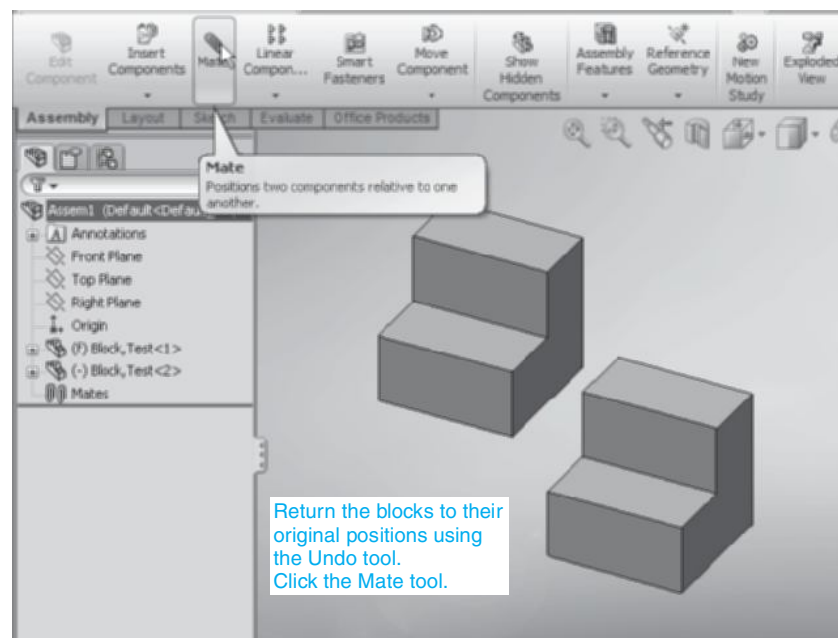


Figure 5-9

- The edges will align. See Figure 5-11.
4. Click the OK check mark to clear the tools.
- The **Mate Selections** box should be clear.

5. Click the upper front surface of the second block.
  6. Click the upper front surface of the first block.
- See Figure 5-12. The surfaces will align. See Figure 5-13.

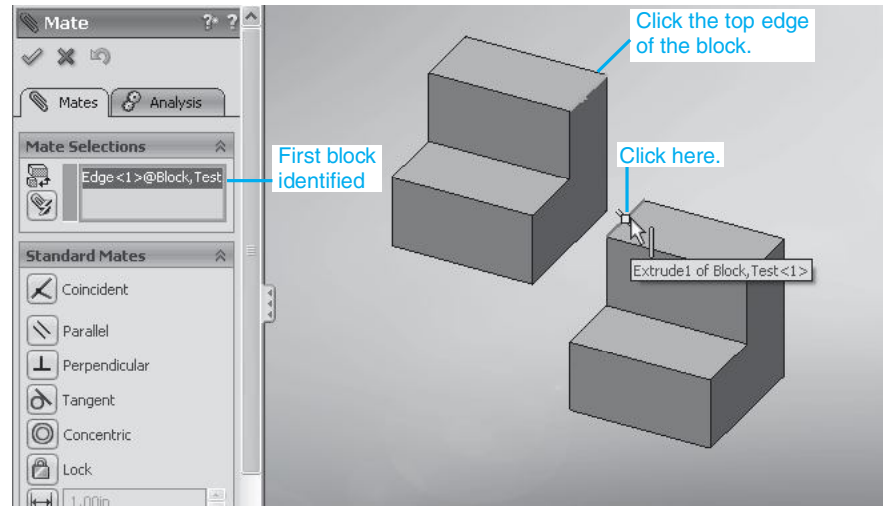


Figure 5-10

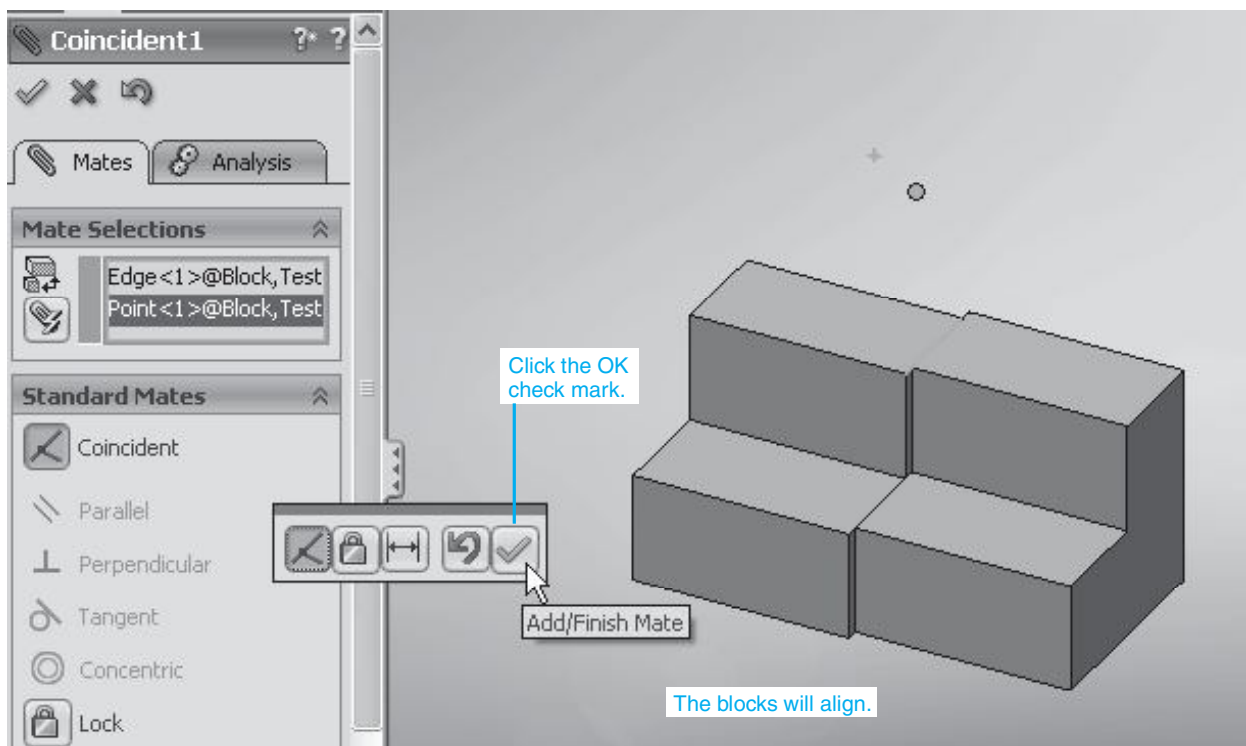


Figure 5-11

### Second Assembly

Mate the two test blocks face-to-face.

1. Use the **Undo** tool and return the blocks to their original positions.

See Figure 5-14. The second test block must be rotated into a different position relative to the first test block.

2. Use the **Rotate Component** tool and position the second block as shown.  
See Figure 5-15.
3. Use the **Mate** tool and click the lower front face of each block.

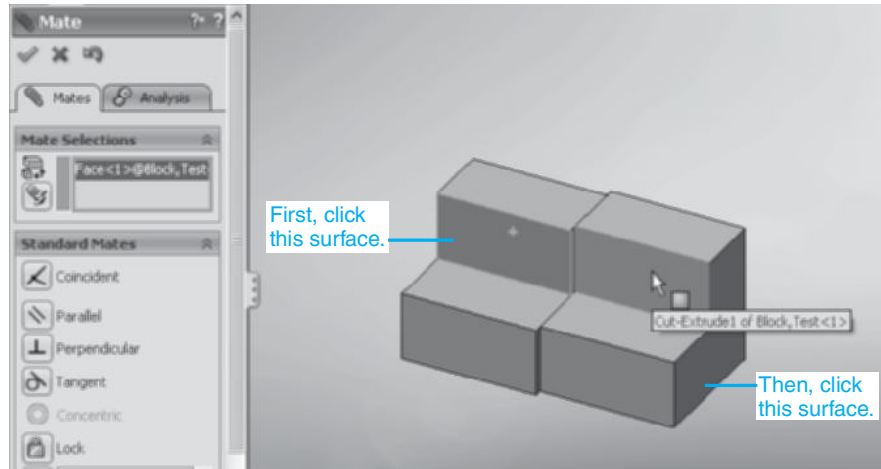


Figure 5-12

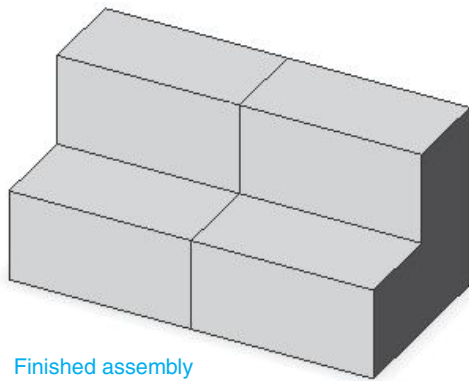


Figure 5-13

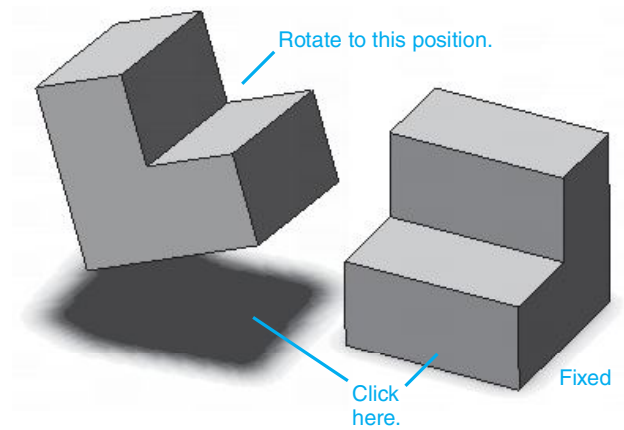


Figure 5-15

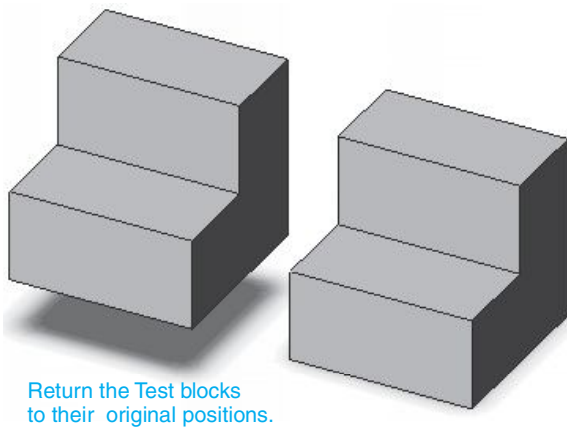


Figure 5-14

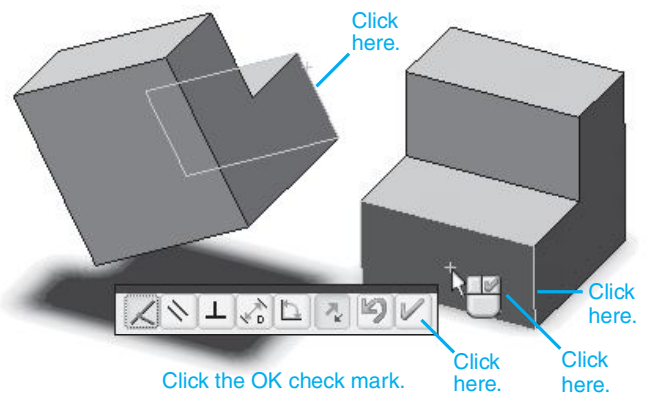


Figure 5-16

The second block will rotate relative to the first block. See Figure 5-16. Recall that the first block is in the fixed condition, and the second block is in the floating condition.

4. Click the **Add/Finish** check mark on the toolbar, then click the OK check mark.
5. Use the **Mate** tool and click the two faces of the test block as shown.

See Figure 5-17.

6. Click the **Add/Finish Mate** check mark on the toolbar then click the OK check mark.
7. Use the **Mate** tool and click the two edge lines as shown.

See Figure 5-18.

8. Click the **Add/Finish Mate** check mark on the toolbar, then click the OK check mark.

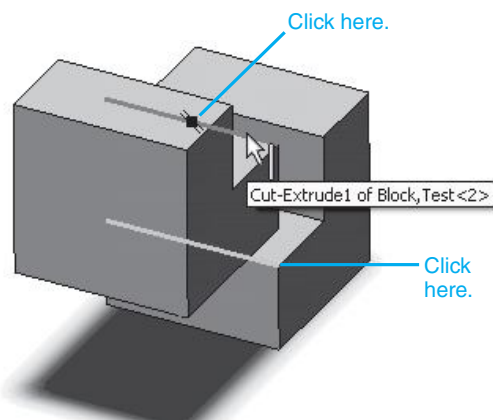


Figure 5-17

### Third Assembly

Mate the two test blocks to form a rectangular prism.

1. Use the **Undo** tool and return the blocks to their original positions.

See Figure 5-14.

2. Use the **Rotate Component** tool and position the second test block as shown.

See Figure 5-19.

3. Click the **Add/Finish Mate** check mark on the toolbar, then click the OK check mark.
4. Use the **Mate** tool and click the right surfaces of the blocks.
5. Click the **Add/Finish** check mark on the toolbar, then click the OK check mark.

See Figure 5-20.

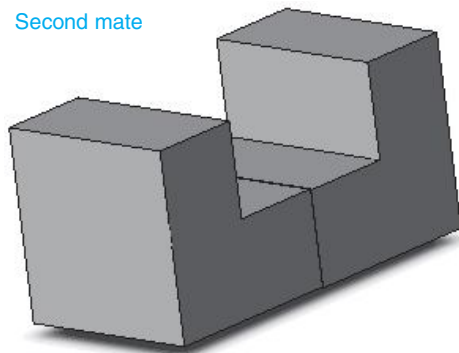


Figure 5-18

Use the **Rotate Component** tool and reorient the second test block.

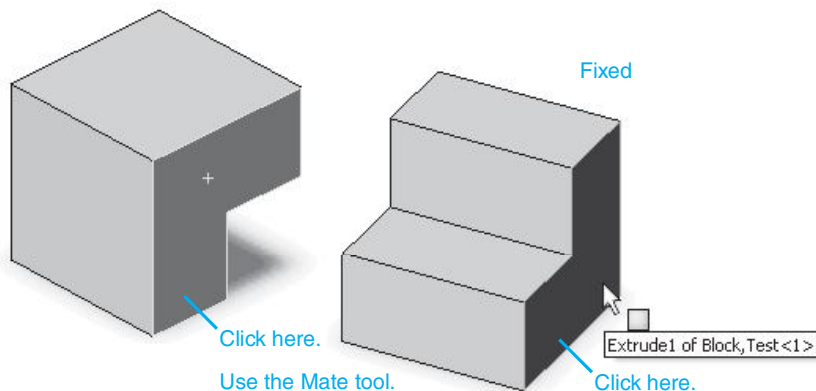


Figure 5-19



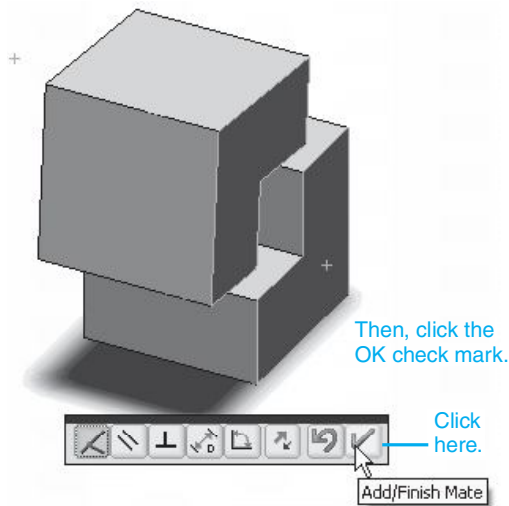


Figure 5-20

6. Click the upper edge of the blocks as shown.  
See Figure 5-21.
7. Click the **Add/Finish Mate** check mark on the toolbar, then click the OK check mark.  
See Figure 5-22.
8. Reorient the blocks so that the bottom surfaces are visible.  
See Figure 5-23.
9. Use the **Mate** tool and click the bottom surfaces of the blocks.  
Figure 5-24 shows the finished assembly.

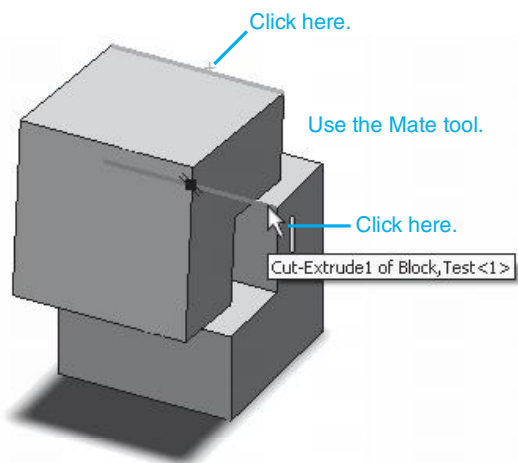


Figure 5-21

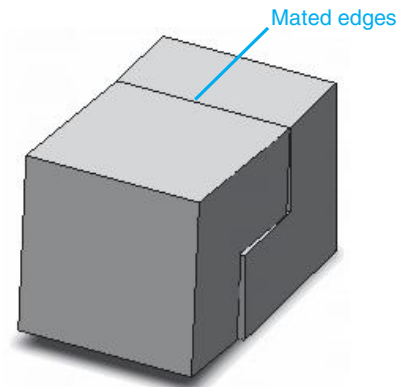


Figure 5-22

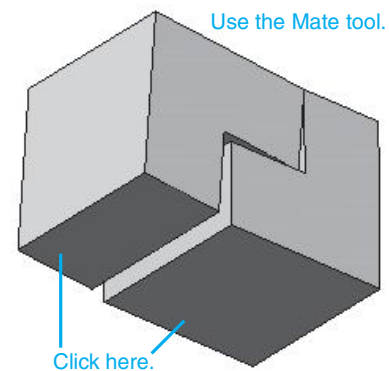


Figure 5-23

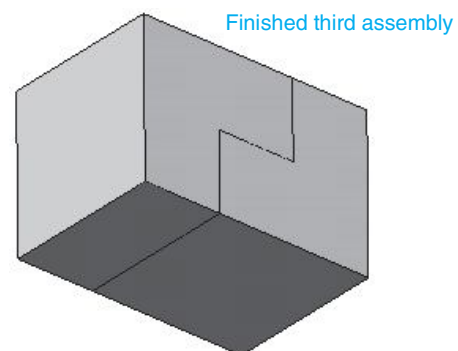


Figure 5-24

## 5-6 BOTTOM-UP ASSEMBLIES

*Bottom-up assemblies* are assemblies that are created for existing parts; that is, the parts have already been drawn as models. In this example the three parts shown in Figure 5-25 have been drawn and saved.

1. Start a new drawing and select the **Assembly** format.

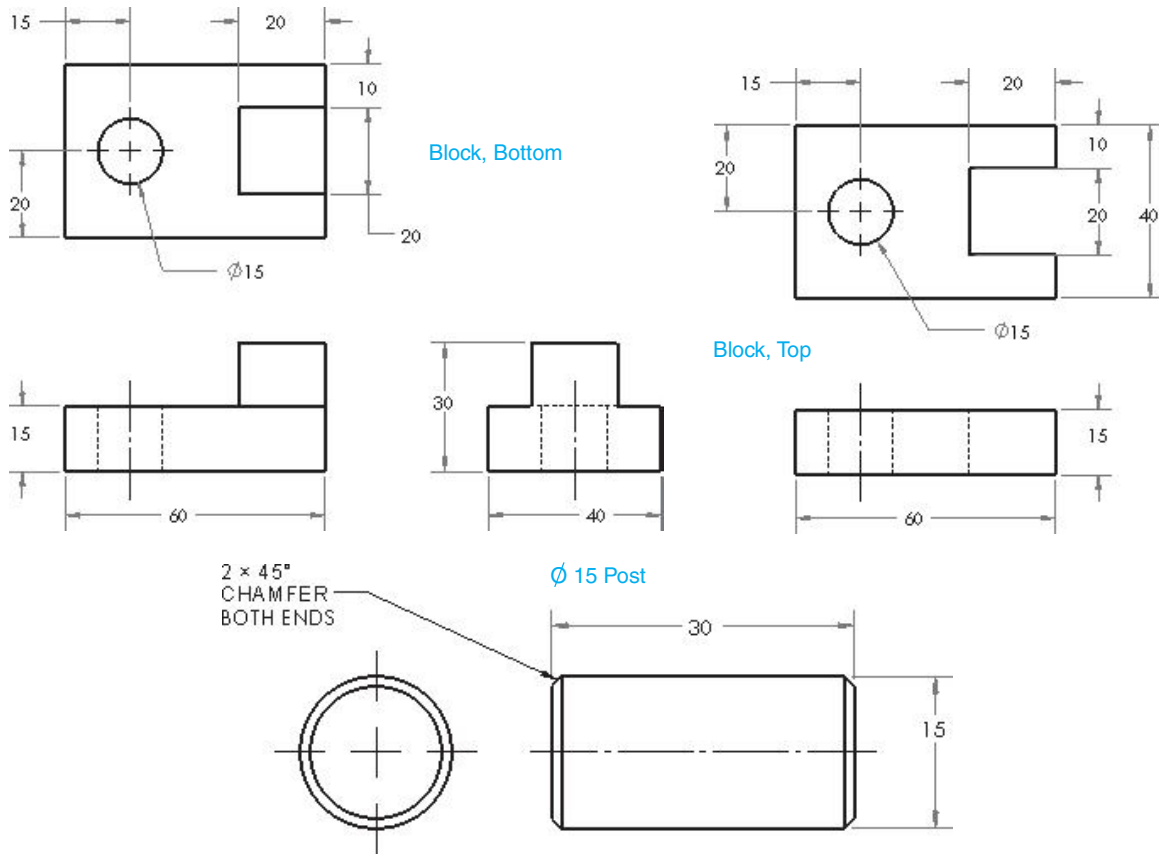


Figure 5-25

2. Click the **Browse . . .** box, then click the **Block, Bottom** component.

See Figure 5-26. The Block, Bottom will appear on the screen. See Figure 5-27.

3. Click the **Insert Component** tool, click the **Browse . . .** box, and select **Block, Top**.
4. Repeat the sequence and select **Ø15 Post**.

See Figure 5-28.

### Note:

Note that the Block, Bottom was the first part entered and is fixed in its location, as designated by the (f) symbol in the Properties Manager box.

5. Use the **Mate** tool and click the center point of the edge line of the Block, Bottom and the Block, Top as shown.

See Figure 5-29. A dot will appear when the cursor is on the center point of the edge. Figure 5-30 shows the resulting mate. If the blocks do not align, use the **Mate** tool again to align the blocks.

6. Click the OK check mark.
7. Click the **Mate** tool and click the **Concentric** option.

See Figure 5-31.

8. Click the side of the Ø15 Post and the inside of the hole in the Block, Top.
9. Click the **Add/Finish Mate** check mark on the toolbar, then click the OK check mark.
10. Use the **Mate** tool and click the top surface of the Ø15 Post and the top surface of the Block, Top.
11. Click the **Add/Finish Mate** check mark on the toolbar, then click the OK check mark.

## 5-7 CREATING AN EXPLODED ISOMETRIC ASSEMBLY DRAWING

1. Click the **Exploded View** tool.

See Figure 5-32.

2. Click the top surface of the Ø15 Post.

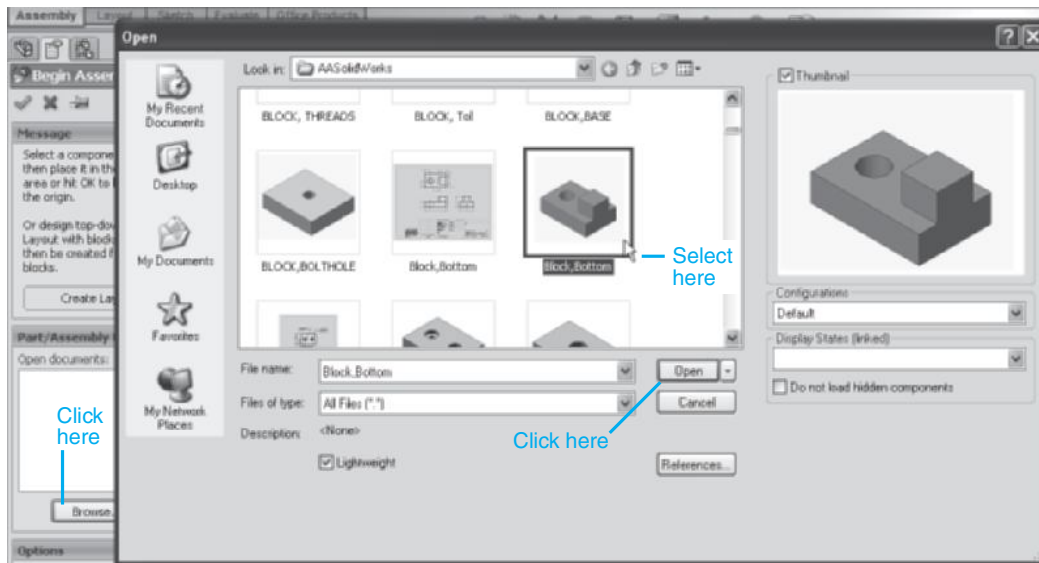


Figure 5-26

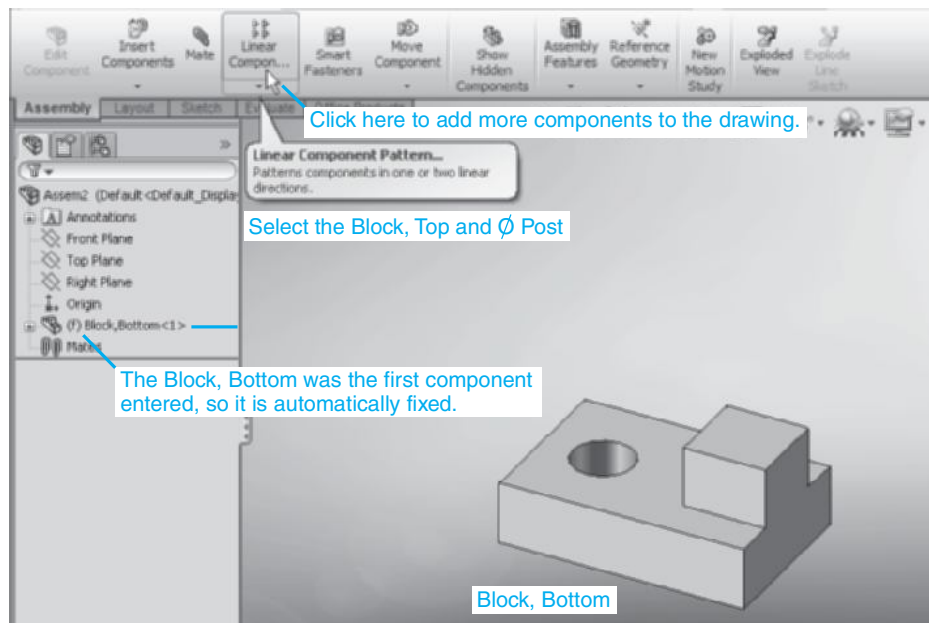


Figure 5-27

An axis system icon will appear. See Figure 5-33. The arrow in the Z-direction (the one pointing vertically) will initially be green.

3. Move the cursor onto the Z-direction arrow and hold down the left mouse button.

The arrow will turn yellow when selected.

4. Drag the  $\varnothing 15$  Post to a location above the assembly as shown.

5. Click the OK check mark.
6. Repeat the procedure and drag the Block, Top away from the Block, Bottom.
7. Click the OK check mark.
8. Save the assembly.

In this example the assembly was saved as **Block Assembly**. Figure 5-34 shows the final assembly.

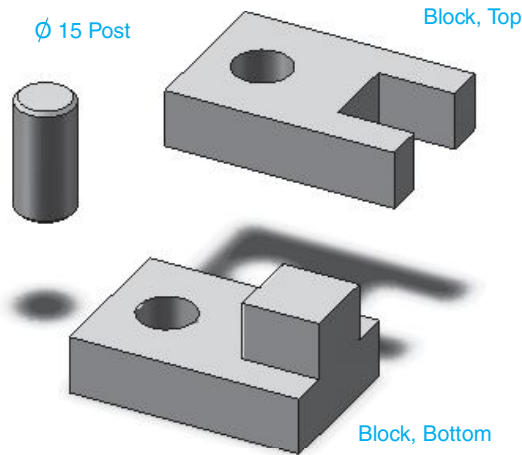


Figure 5-28

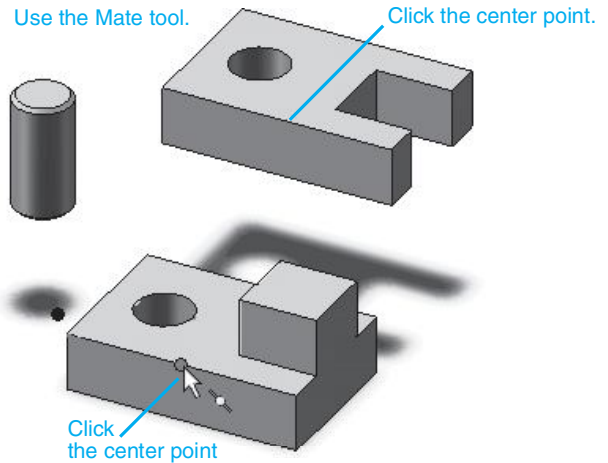


Figure 5-29

## 5-8 CREATING AN EXPLODED ISOMETRIC DRAWING USING THE DRAWING FORMAT

1. Create a new drawing using the **Drawing** format.
2. Select the **A-Portrait** sheet format.
3. Click the **Browse ...** box in the **Model View Properties Manager**.

See Figure 5-35.

4. Select **Block Assembly**; click **Open**.
5. Set the **Orientation** for **Isometric** and the **Display Style** for **Hidden Lines Removed**.

See Figure 5-36.

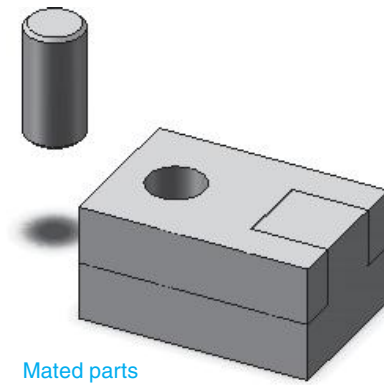


Figure 5-30

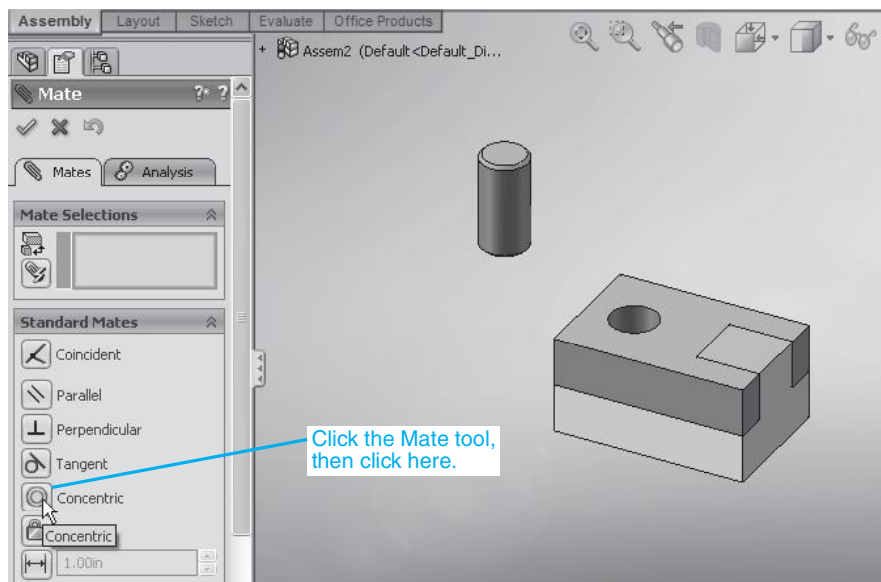
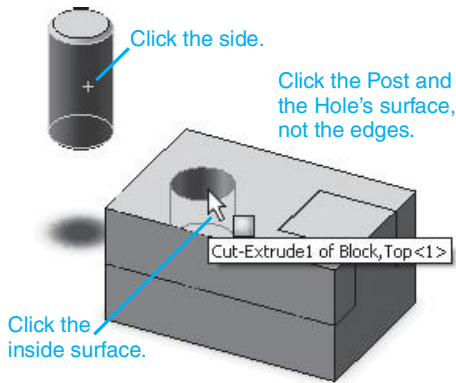


Figure 5-31



Use the Mate tool.

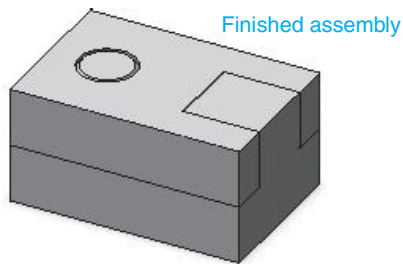
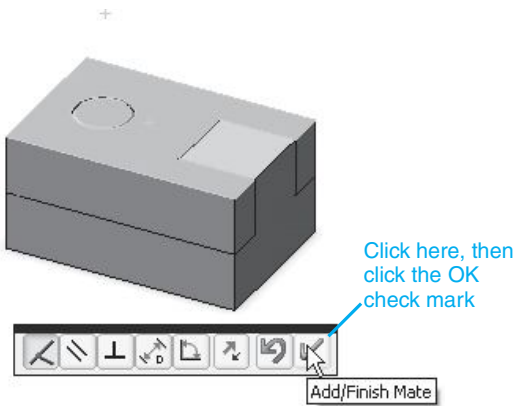
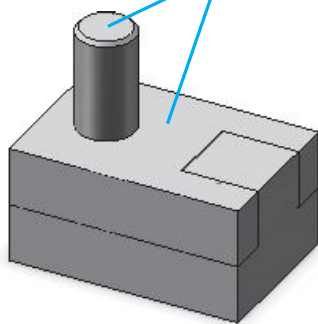


Figure 5-31 (continued)

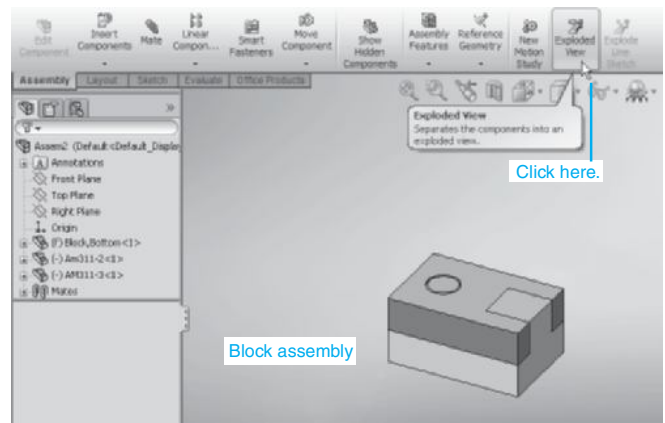


Figure 5-32

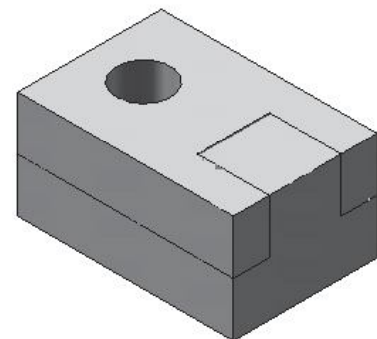
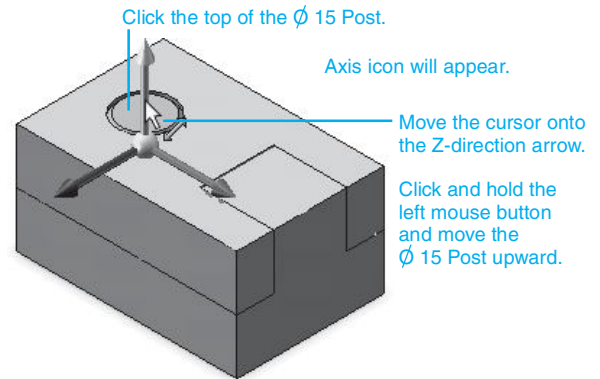


Figure 5-33

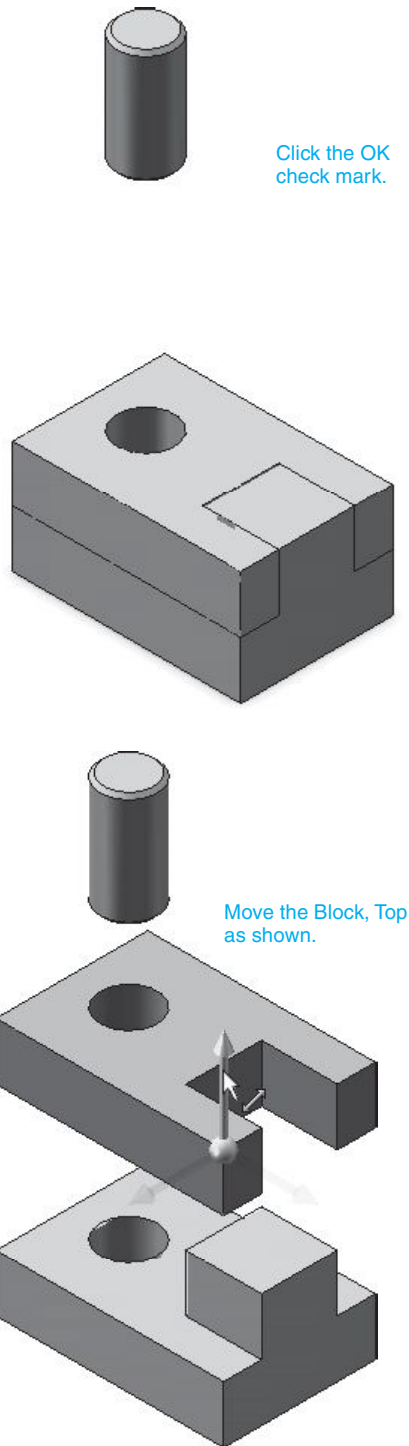


Figure 5-33 (continued)

**TIP**

As a general rule, hidden lines are not included on isometric drawings.

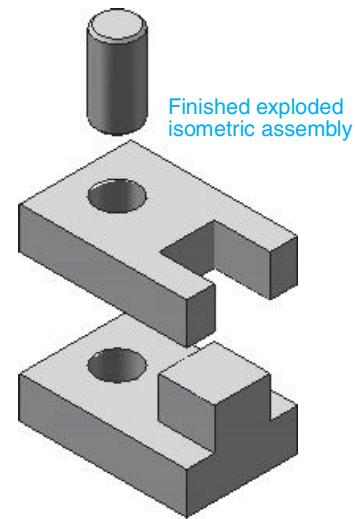


Figure 5-34

6. Move the cursor into the drawing area.  
A rectangular outline of the view will appear.
7. Locate the view and click the left mouse button.  
See Figure 5-37.

**5-9 ASSEMBLY NUMBERS**

*Assembly numbers* are numbers that identify a part within an assembly. They are different from part numbers. A part number identifies a specific part, and the part number is unique to that part. A part has only one part number but may have different assembly numbers in different assemblies.

Assembly numbers are created using the **Balloon** or **AutoBalloon** tools located on the **Annotation** tool panel.

1. Click the **Annotation** tool.
2. Click the **AutoBalloon** tool.

Figure 5-38 shows the results. The **AutoBalloon** arrangements may not always be the best presentation. Balloons can be applied individually.

**TIP**

Balloons can be moved by first clicking them. They will change color. Click and hold either the balloon or the box that will appear on the arrow. Either the balloon or the arrow may be dragged to a new location.

3. Undo the auto balloons.
4. Click the **Balloon** tool.

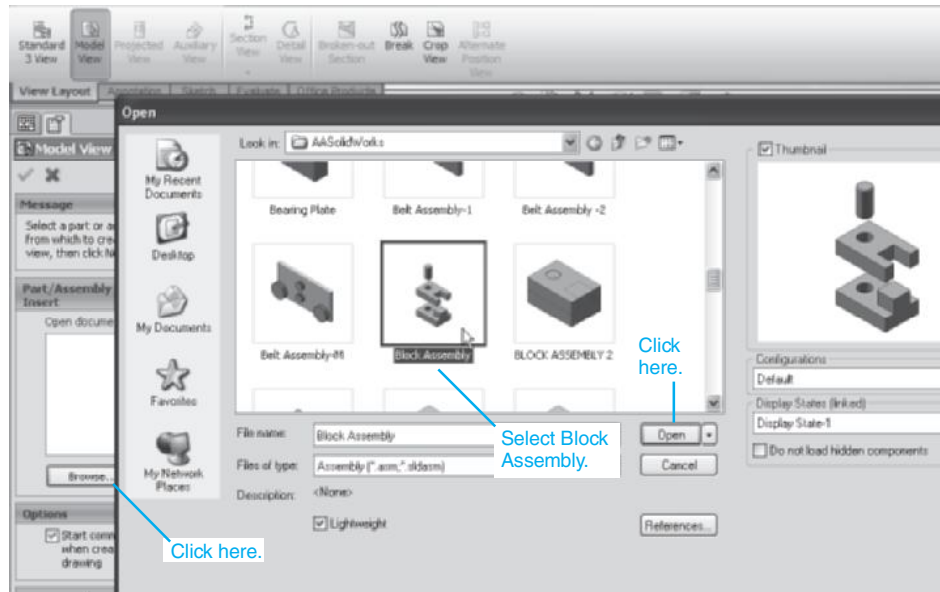


Figure 5-35

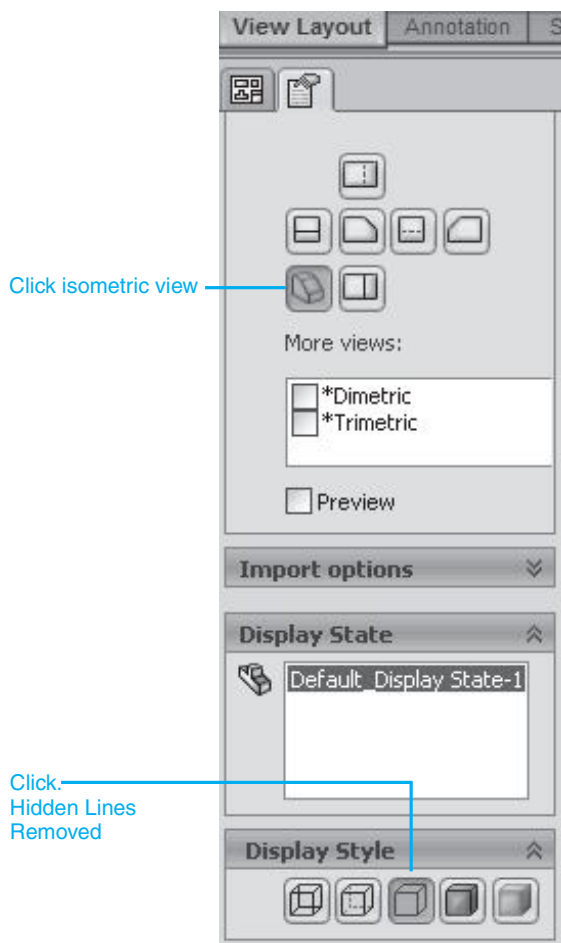


Figure 5-36

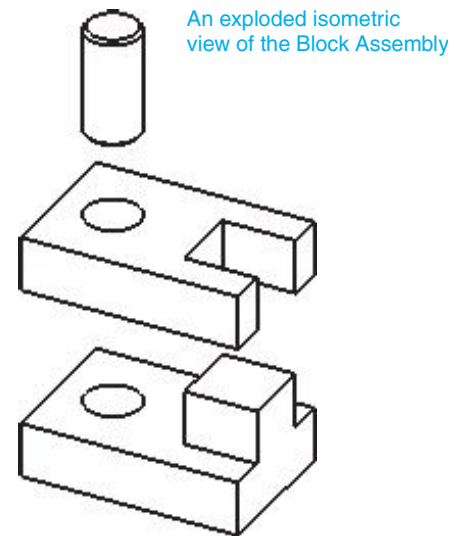


Figure 5-37

5. Click each part and locate the balloon.
6. Click the OK check mark.

See Figure 5-39.

## 5-10 BILL OF MATERIALS (BOM OR PARTS LIST)

A *bill of materials* is a listing of all parts included in an assembly drawing.

1. To access the **Bill of Materials** tool click the **Annotation** toolbox, **Tables**, and **Bill of Materials** tools.

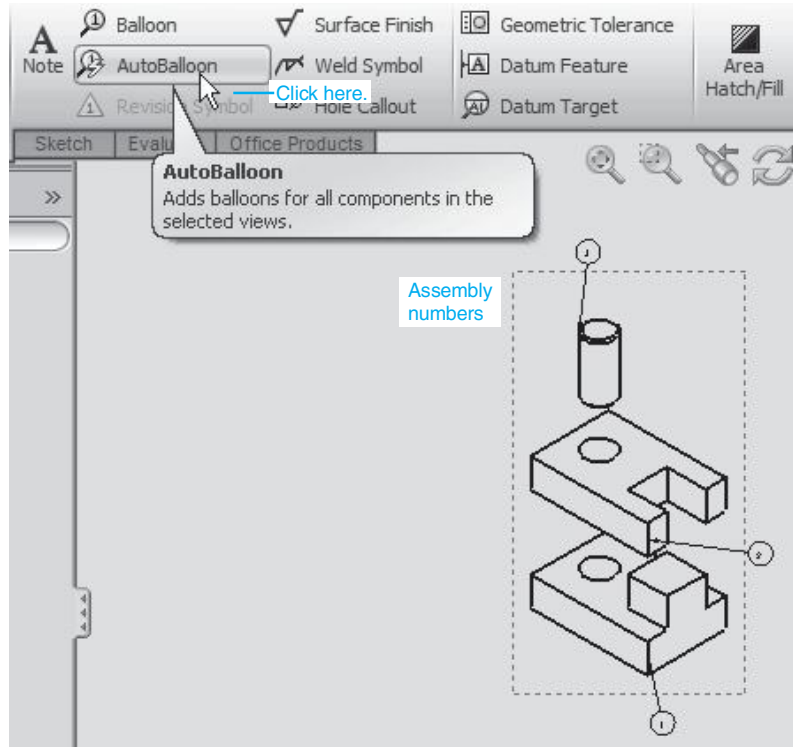


Figure 5-38

See Figure 5-40.

2. Click the area of the Block Assembly.

A red box will appear around the Block Assembly. Click within that box.

3. Click the OK check mark.

Pull the cursor back into the drawing area. The BOM will follow. Select a location for the BOM and click the mouse. See Figure 5-41.

Individually placed assembly numbers

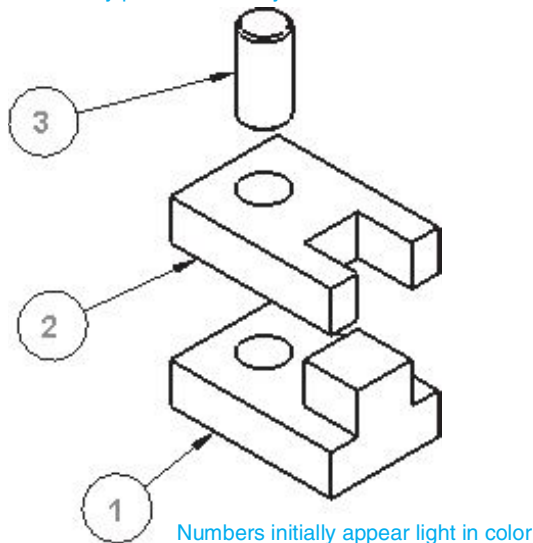


Figure 5-39

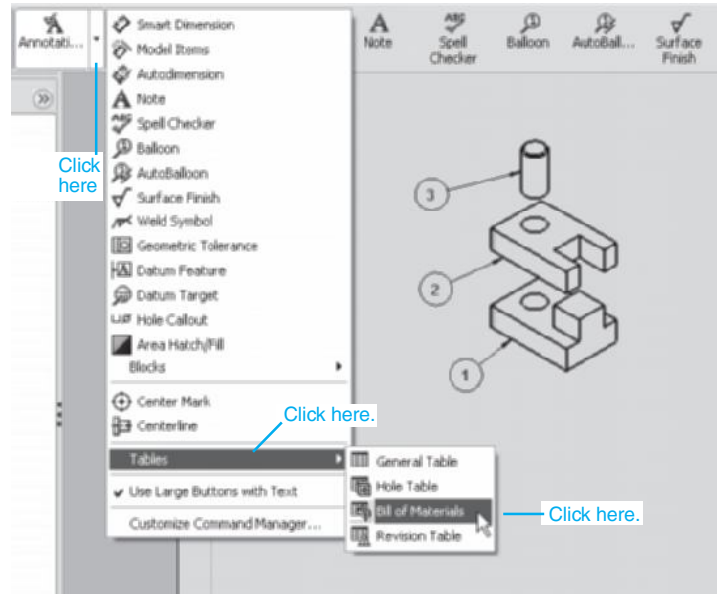


Figure 5-40



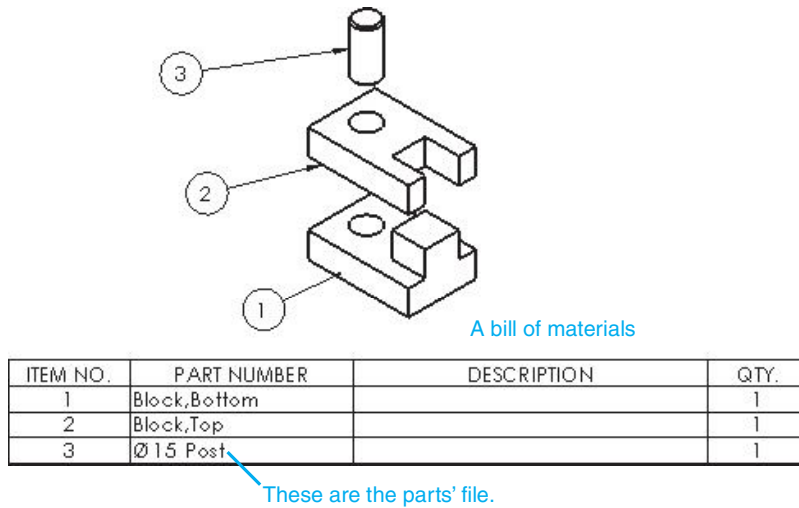


Figure 5-41

**Note:**

Note that the information in the **PART NUMBER** column is the parts' file names. These names are directly linked by SolidWorks to the original part drawings. They can be manually edited, but if the assembly is changed and regenerated, the original part names will appear. In this example the original parts were renamed using part numbers as their file names. See Figure 5-42.

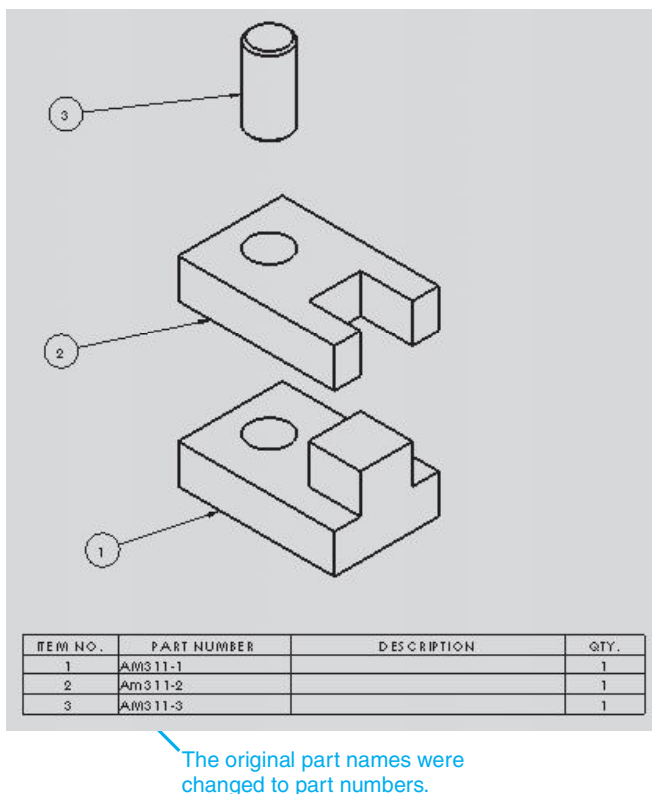


Figure 5-42

**To Edit the BOM**

1. Double-click the box directly under the heading **DESCRIPTION**.  
A warning dialog box will appear.
2. Click the **Keep Link** box.  
The **Formatting** dialog box will appear.
3. Type in a description of the part.  
See Figure 5-43.
4. Click the box below the one just edited.  
A warning box will appear.
5. Click the **Yes** box.  
A second warning box will appear.
6. Click the **Keep Link** box.
7. Edit the remaining boxes as shown.  
See Figure 5-44.

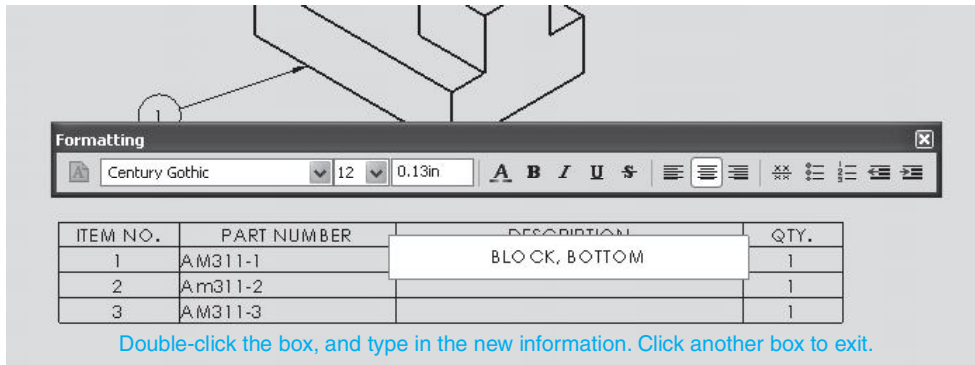


Figure 5-43

ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	AM311-1	BLOCK, BOTTOM	1
2	Am311-2	BLOCK, TOP	1
3	AM311-3	Ø15 POST	1

An edited BOM

Figure 5-44

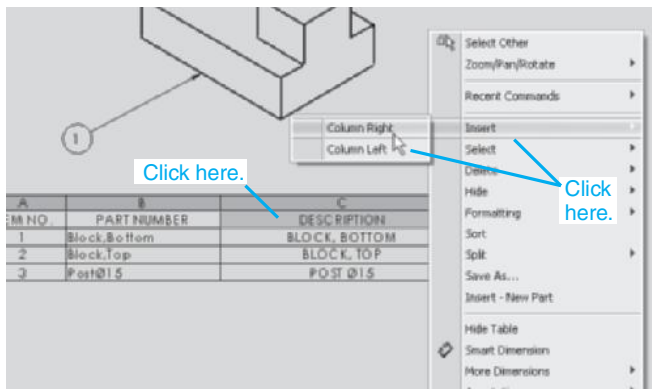


Figure 5-45

**To Add Columns to the BOM**

1. Right-click the **BLOCK, BOTTOM** box.  
See Figure 5-45.
2. Click the **Insert** option, then click **Column Right**.  
A new column will appear to the right of the BLOCK, BOTTOM box. See Figure 5-46.
3. Right-click one of the boxes in the new column.
4. Select the **Formatting** option; click **Column Width**.

See Figure 5-47. The **Column Width** dialog box will appear. See Figure 5-48.

5. Enter a new value.

In this example a value of **1.75 in** was entered. Figure 5-48 shows the new column width.

	DESCRIPTION		QTY.
	BLOCK, BOTTOM		1
	BLOCK, TOP		1
	Ø15 POST		1

A new column has been added.

Figure 5-46

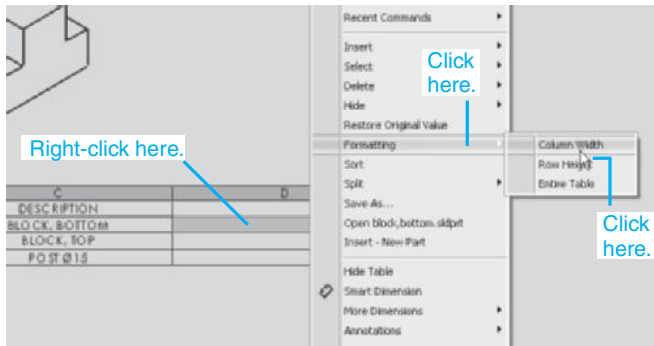


Figure 5-47

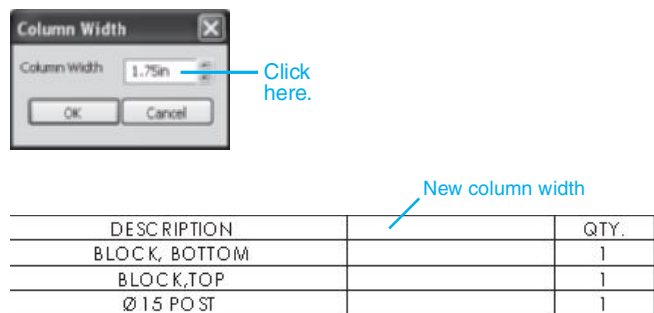


Figure 5-48

### To Edit a BOM

1. Click the top box of the new 1.75-wide column.
  2. Type the heading **MATERIALS**.
- See Figure 5-49.

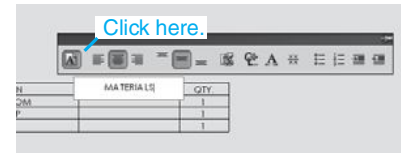


Figure 5-49

3. Add material specifications as shown in Figure 5-50.

SAE 1020 is a type of mild steel.

4. Add another new column, **NOTES**, to the right of the materials column. Make the column 2.00 wide.

See Figure 5-51. Notes are included in BOMs to define information about the part that is not visual.

5. Add the notes as shown.

See Figure 5-52.

6. Edit the BOM so that all columns are justified to the right.

See Figure 5-53.

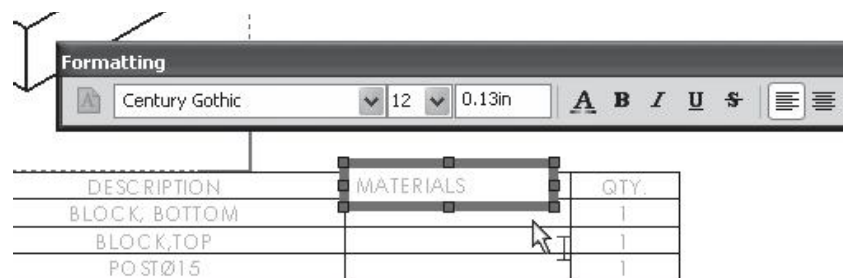
## 5-11 ANIMATE COLLAPSE

Assembly drawings can be animated. In this example the **Animate Collapse** tool will be used.

1. Open the **Block Assembly**.
2. Right-click the **Block Assembly** heading in the **Properties Manager**.

See Figure 5-54.

3. Click the **Animate Collapse** option.



Locate the note outline box in the top box of the new column, click the mouse and enter a new heading.

Click another box, press the ESC key

Figure 5-50

DESCRIPTION	MATERIALS	QTY.
BLOCK, BOTTOM		1
BLOCK, TOP		1
Ø15 POST		1

New heading

Figure 5-51

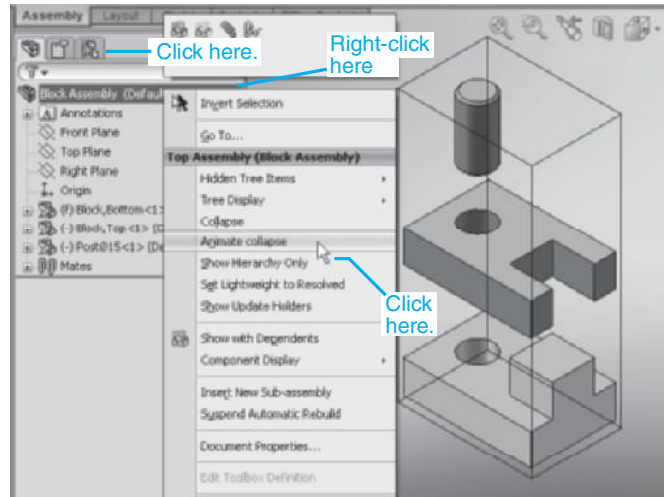
	MATERIALS	QTY.
	SAE 1020	1
	SAE 1020	1
	MILD STEEL	1

Enter notes

Figure 5-52

The assembly will automatically be animated and start to move. See Figure 5-55.

4. Click the **Start** button to stop the animation and return the assembly to the exploded position.
5. Close the **Animation Controller**.



Block assembly

Figure 5-54

In this example the **Animate Collapse** tool was used because the assembly was shown in the exploded position. Had the assembly been in a closed assembled position, the **Animate Explode** option would have been used.

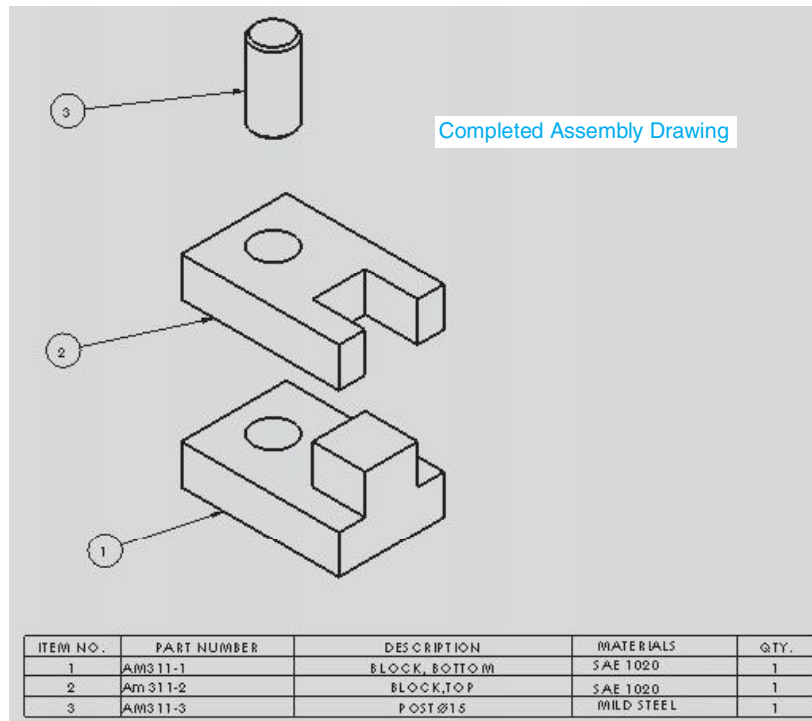


Figure 5-53

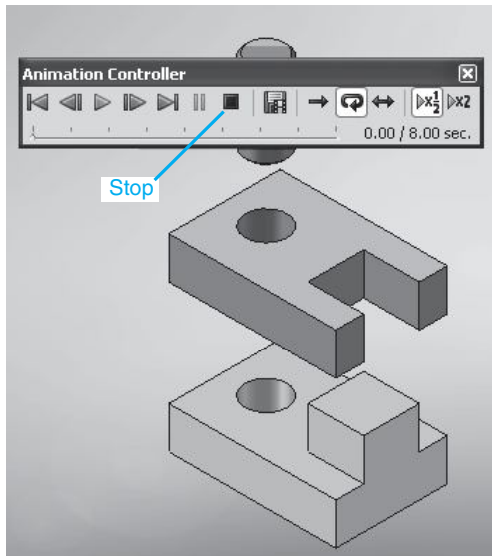


Figure 5-55

## 5-12 MOTION STUDY

The **Motion Study** tool is used to animate an assembly. Animating an assembly allows the viewer to see and better understand how an assembly moves. The motion study tool will also display any interferences in the motion. There are four types of motion available: motor, spring, contact, and gravity.

Figure 5-56 shows the **Motion Study Manager**. It is accessed by clicking the **Motion Study** tab at the bottom of the screen. Additional motion studies may be added by clicking the **New Motion Study** tool at the top of the screen.

Section 5-12 demonstrates how to apply the **Motion Study** tool to an assembly.

## 5-13 SAMPLE PROBLEM 5-1: CREATING THE ROTATOR ASSEMBLY

Figure 5-57 shows the components for the Rotator Assembly. The dimensions for the components can be found in Project P5-10 at the end of the chapter. Draw and save the four Rotator Assembly components as **Part** documents.

1. Click the **Mate** tool.
2. Click the **Concentric** tool.
3. Click the side of the bottom post of the Link L&R and the inside of the left hole in the Plate.

See Figure 5-58. The Link L&R and Plate will align.

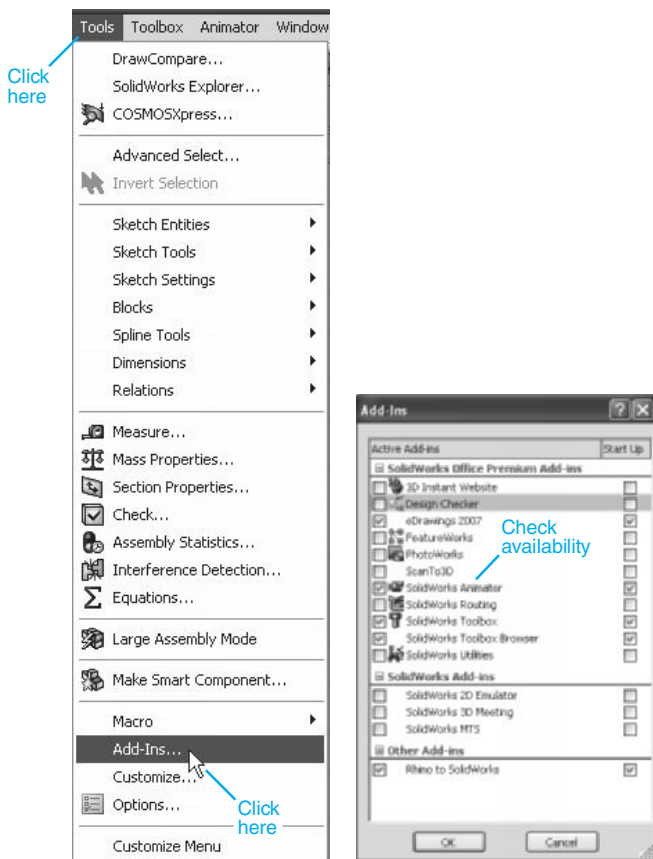


Figure 5-56

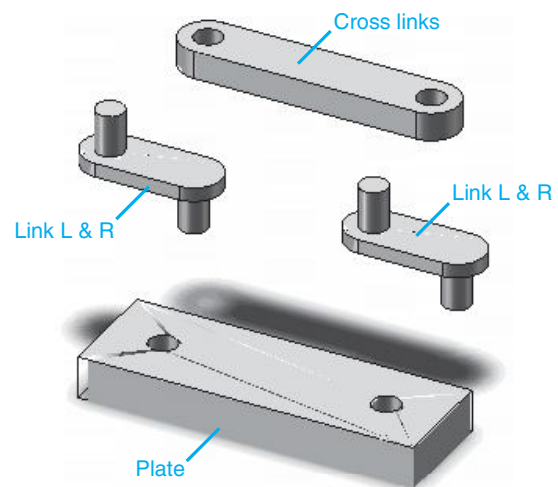


Figure 5-57

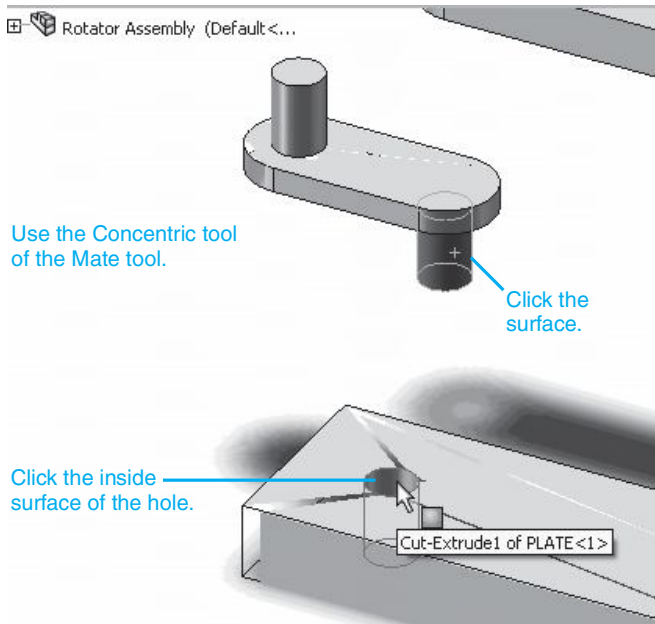


Figure 5-58

**TIP**

Click the post's and hole's surfaces. Do not click the edge lines.

4. Click the OK check mark.
5. Click the **Mate** tool.
6. Click the top surface of the Plate and the bottom surface of the Link L&R.

Access the Mate tool.

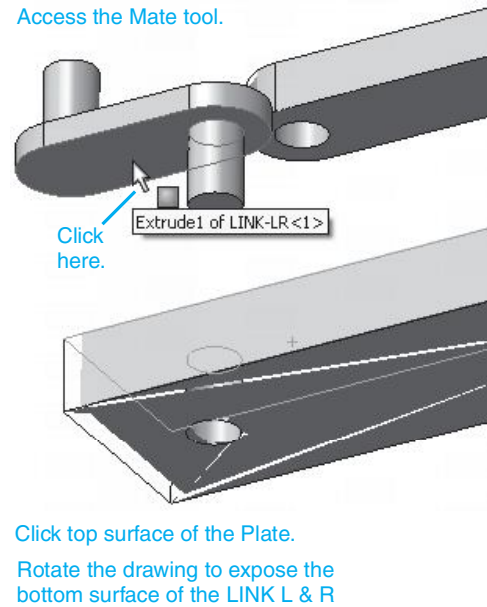


Figure 5-59

See Figure 5-59. Use the **Rotate View** tool to manipulate the view orientation so that the bottom surface of the Link L&R is visible. The part also can be rotated by holding down the mouse wheel and moving the cursor.

7. Click the **Distance** box and enter a value. In this example a value of **2 mm** was entered.

See Figure 5-60. The initial offset values may be in inches. Enter the new values followed by **mm**, and the system will automatically change to metric (millimeter) distances.

8. Click the OK check mark twice.

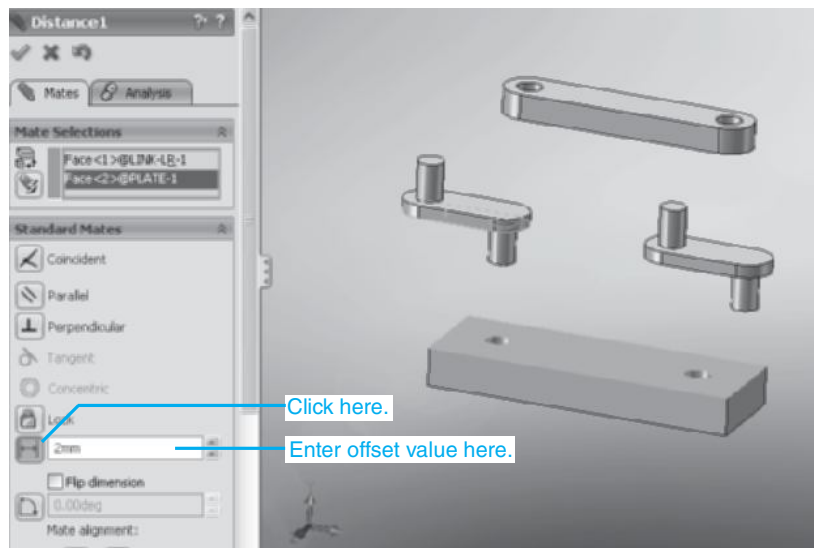


Figure 5-60

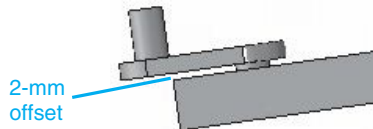


Figure 5-61

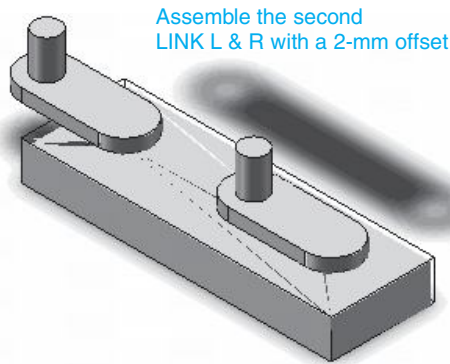


Figure 5-62

Figure 5-61 shows the 2-mm offset between the plate and the link.

9. Return the drawing to the **Isometric** orientation.
10. Repeat the procedure for the other Link L&R.

See Figure 5-62.

11. Access the **Mate** tool and use the **Concentric** tool to align the top post of the left Link L&R with the left hole in the Crosslink.
12. Use the **Mate** tool and click the top surface of the Link L&R's post and the top surface of the Crosslink.
13. Click the **Distance** box and define the offset distance as **2.00 mm**.

See Figure 5-63.

14. Use the **Flip** tool and offset the Crosslink so that the posts protrude 2 mm above the top surface of the Crosslink.
15. Click the OK check mark.

See Figure 5-64.

16. Access the **Mate** tool, click the **Concentric** tool, and align the right hole in the Crosslink with the post on the right Link L&R.

## TIP

This is an important step to tie the entire assembly together.

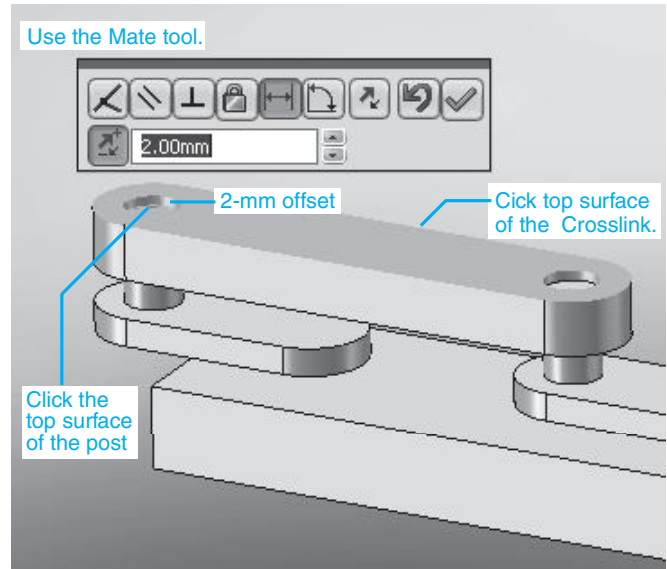


Figure 5-63

17. Locate the cursor on the Crosslink and move it around.

The Crosslink and two Links L&R should rotate about the Plate.

See Figure 5-65.

18. Save the Rotator Assembly.

## 5-14 USING THE SOLIDWORKS MOTION STUDY TOOL

Figure 5-65 shows the Rotator Assembly created in the last section.

1. Click the **Mate** tool, then click the **Parallel** tool.
2. Make the front surface line of the Crosslink parallel to the front edge of the Plate.

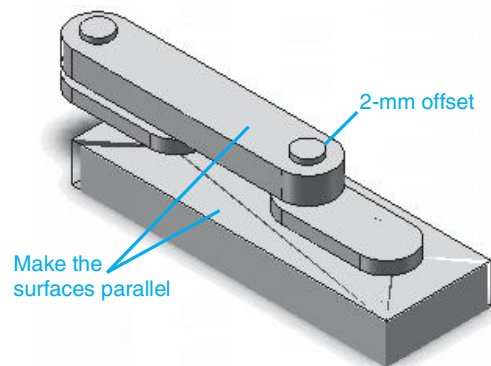


Figure 5-64

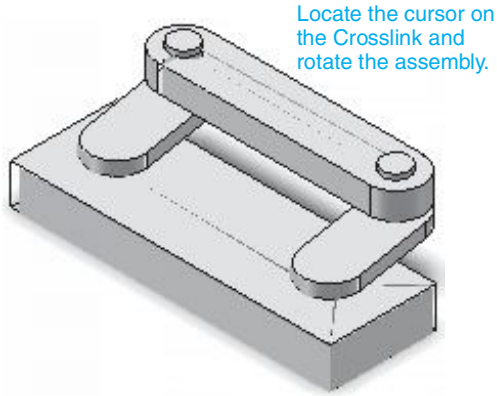


Figure 5-65

This step will assure that the Crosslink rotates in an orientation parallel to the front edge of the Plate. See Figure 5-66.

3. Click the **Motion Study** tab at the bottom of the screen.

See Figure 5-67.

Compare the component listing on the left side of the **Motion Study manager** in Figure 5-67 with the component listing in Figure 5-68. Note that the icons to the left of the components in Figure 5-68 include feathers. The feather indicates that this is a lightweight assembly; that is, only a subset of the model data has been loaded. This is done to save file space and to make the loading faster, particularly for large assemblies. When preparing an assembly for animation use the fully resolved assembly drawing.

To assure that you have a fully resolved assembly when opening a saved assembly make sure the **Lightweight** box is turned off; that is, no check mark appears.

4. Click the **Motor** tool.

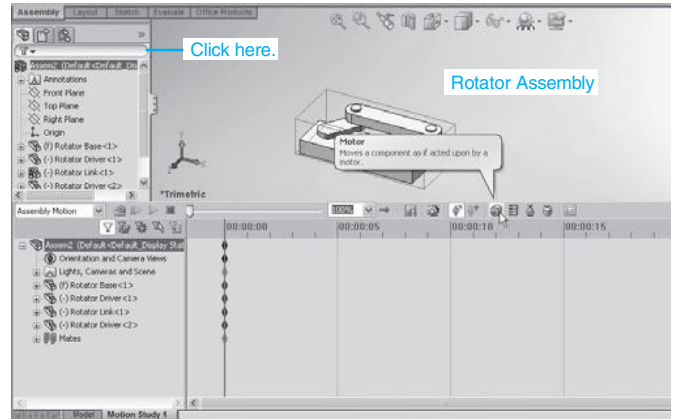


Figure 5-67

The **Motor Manager** box will appear.

See Figure 5-69.

5. Click the **Rotary motion** tool.
6. Click the box under the **Component/Direction** heading, then click the left Link L&R.

The left Link L&R is now the driver link. It will drive the other components.

**Motion**

1. Go to the **Motion** box and define the assembly's motion.

See Figure 5-70. In this example the default values of **Constant Speed** and **100 RPM** were accepted.

2. Click the OK check mark and return to the **Motion Study Manager**.

See Figure 5-71.

3. Click the **Play** tool.

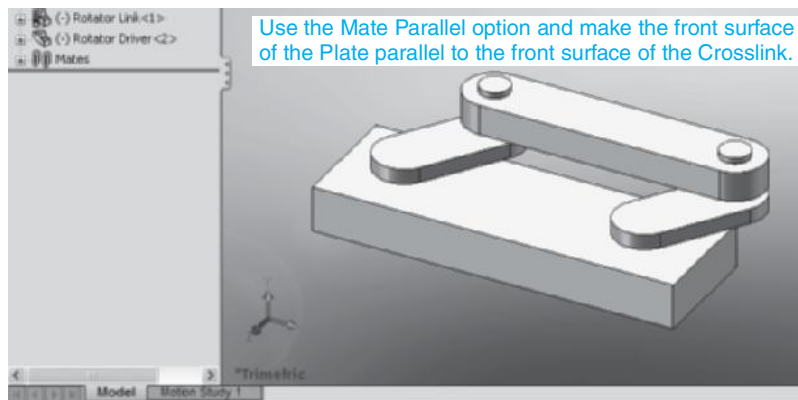


Figure 5-66



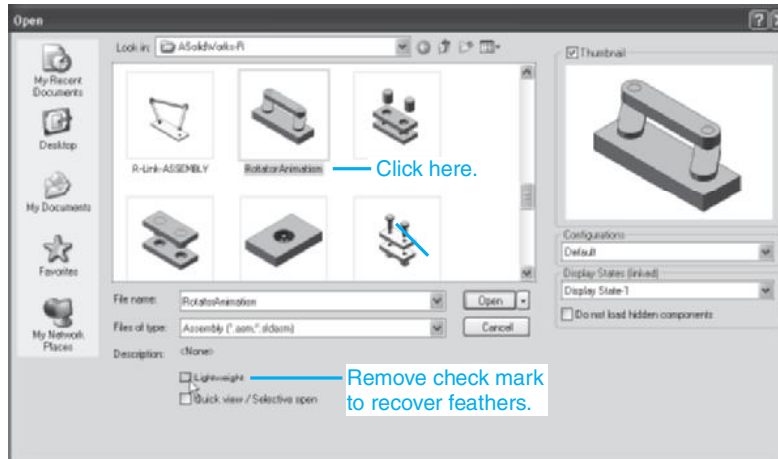


Figure 5-68

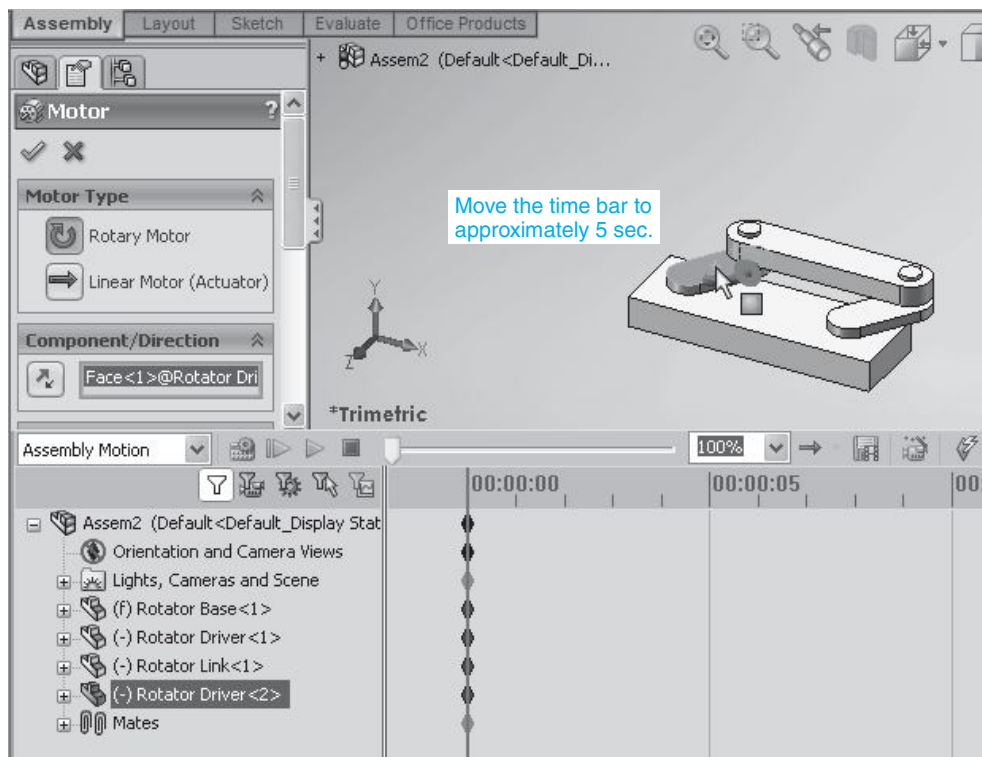


Figure 5-69

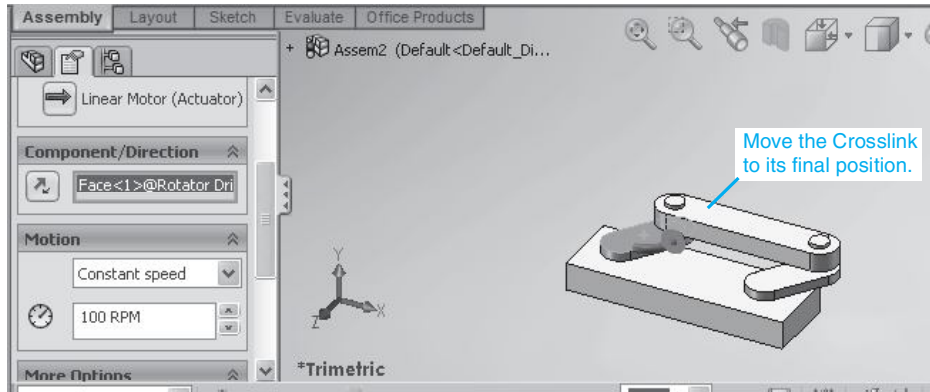


Figure 5-70

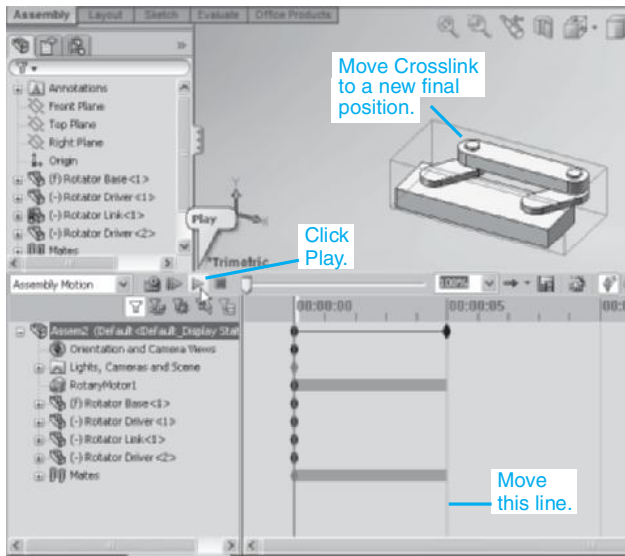


Figure 5-71

### 5-15 EDITING A PART WITHIN AN ASSEMBLY

Parts already inserted into an assembly drawing can be edited. Figure 5-72 shows the Block Assembly created earlier in the chapter.

1. Right-click the Block, Top.
  - Say we wanted to change the hole in the top block.
2. Click the + sign to the left of the **Cut-Extrusion 1** for the **Block, Top** heading in the **Properties Manager**.

In this example the **Cut-Extrusion 1** is the hole in the top block.

3. Right-click the **Sketch** heading and click the **Edit Sketch** tool.

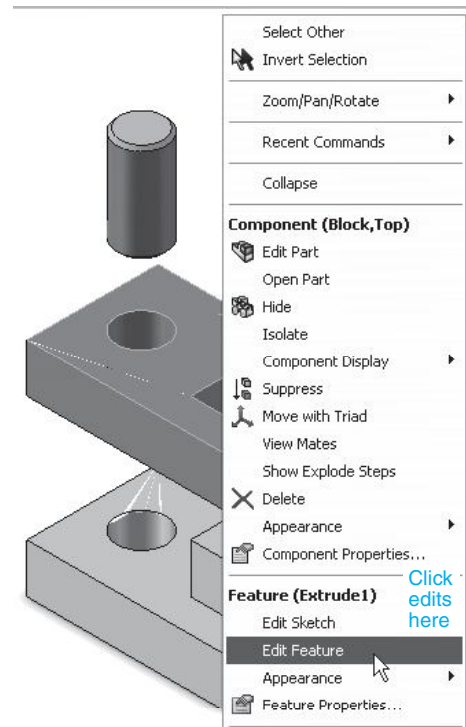


Figure 5-72

4. Double click the  $\varnothing 15$  hole value and enter a new value.
5. Click the OK check mark.
6. Click the **Exit Sketch** tool.
7. Click the **Edit Component** tool.

### 5-16 TITLE BLOCKS

A title block contains information about the drawing. See Figure 5-73. The information presented in a title block varies from company to company but usually includes the company’s name, the drawing name and part number, the drawing scale, and a revision letter.

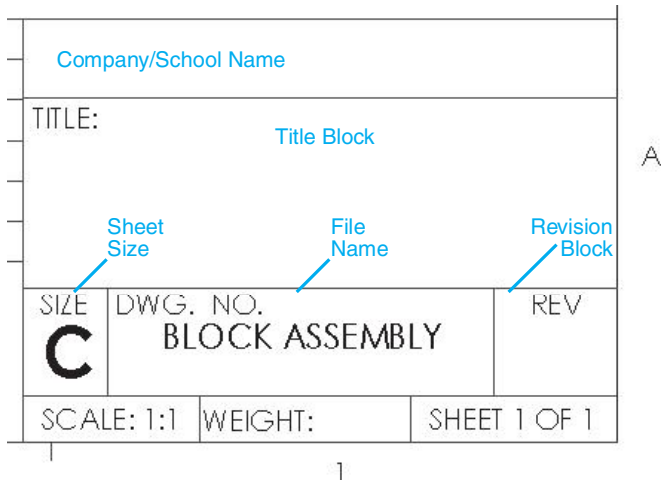


Figure 5-73

### Revision Letters

As a drawing goes through its production cycle changes are sometimes made. The changes may be because of errors but they may also be because of the availability of new materials, manufacturing techniques, or new customer requirements. As the changes are incorporated onto the drawing a new revision letter is added to the drawing.

#### Note:

SolidWorks will automatically enter the file name of the document as the part number. In the example shown in Figure 5-73 the drawing number BLOCK ASSEMBLY is not the document's part number. The title block will have to be edited and the correct part number entered.

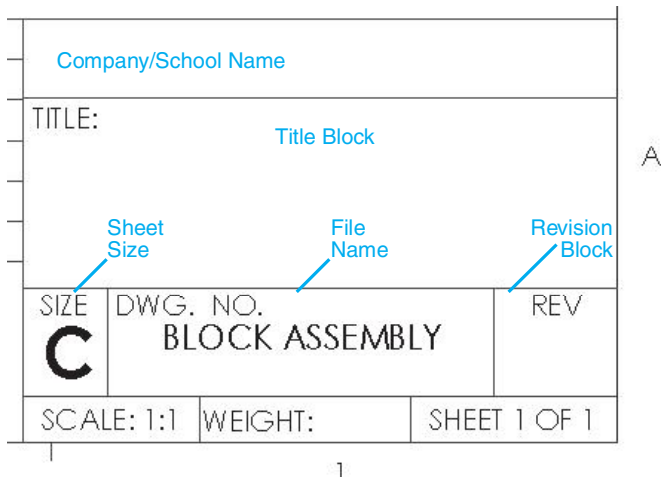


Figure 5-74

### To Add Information to a Title Block

Figure 5-74 shows the title block presented in Figure 5-73. The Block Assembly drawing title has been relocated.

1. Click the **Note** tool located on the **Annotation** toolbar.
2. Locate the **Note** box in the upper portion of the title block as shown.
3. Type the company or school name.
4. Click the check mark.
5. Fill in other information as needed.

### To Change Fonts

1. Click the **Tools** heading at the top of the screen.  
See Figure 5-75.
2. Click the **Options . . .** tool.
3. Click the **Document Properties** tab.
4. Click the **Annotations Font** tool.
5. Click the **Note** tool.
6. Select the **Times New Roman** font.
7. Click **OK** and return to the drawing.

#### Note:

The default font for SolidWorks is Century Gothic. If you change fonts, choose one that is easy to read. Some very stylish fonts are difficult to read and can cause errors.

### TIP

The **Annotations Font** tool can also be used to change the height and style of the font.

### Release Blocks

A finished engineering drawing is a legal document that goes through a release process before it becomes final. The release block documents the release process. For example, once you have completed a drawing, you will sign and date the **Drawn by** box located just to the left of the title block. The drawing will then go to a checker, who, after reviewing and incorporating any changes, will sign and date the **Checked** box.

### Tolerance Block

The tolerance block will be discussed in Chapter 8, Tolerances.

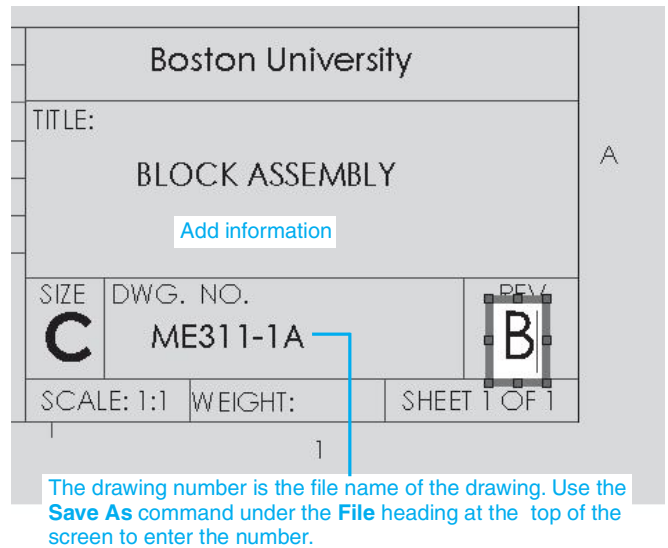
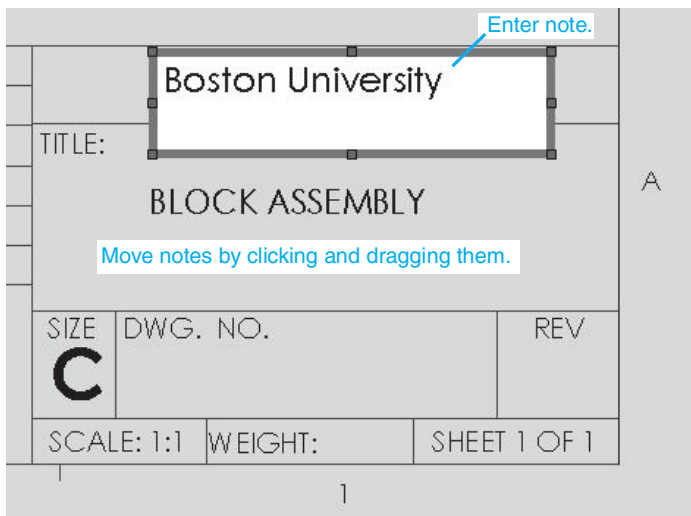
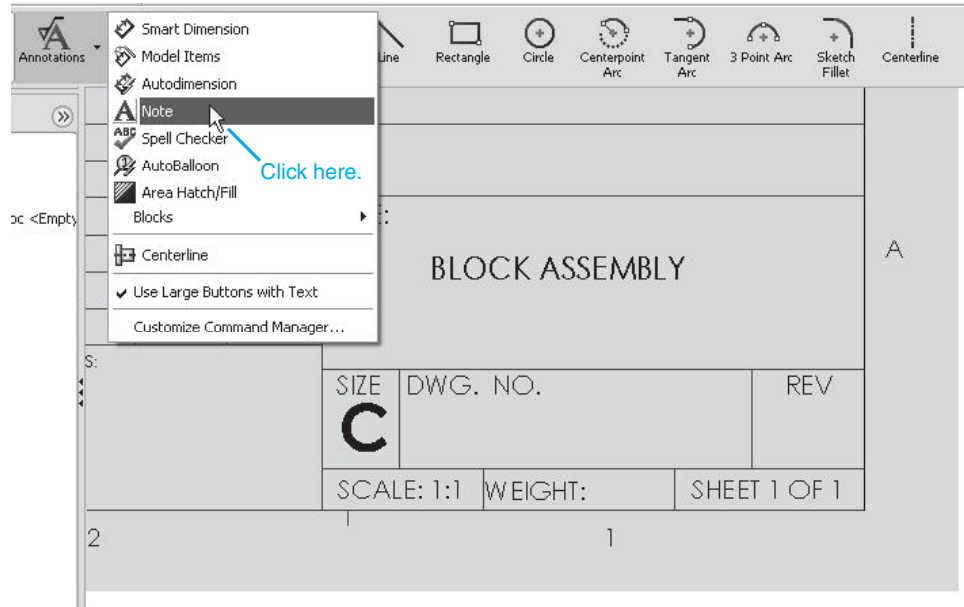


Figure 5-75

**Application Block**

See Figure 5-76.

This block is used to reference closely related drawings. In this example, we know that the Block assembly will be used on assembly ME-312A and that it was also used on EK131-46. This information makes it easier to access related drawings that can be checked for interfaces.

**Note:**

The note “DO NOT SCALE DRAWING” located at the bottom of the tolerance block is a reminder not to measure the views on the drawing. If a dimension is missing, do not measure the distance on the drawing, because the drawing may not have been reproduced at exactly 100% of the original.

## 5-17 PROJECTS

### Project 5-1:

Create a **Part** document of the SQBLOCK using the given dimensions. Create assemblies using two SQBLOCKS, positioning the blocks as shown in Figures P5-1A through P5-1G.

Pages 229 through 237 show a group of parts. These parts are used to create the assemblies presented as problems in this section. Use the given descriptions, part numbers, and materials when creating BOMs for the assemblies.

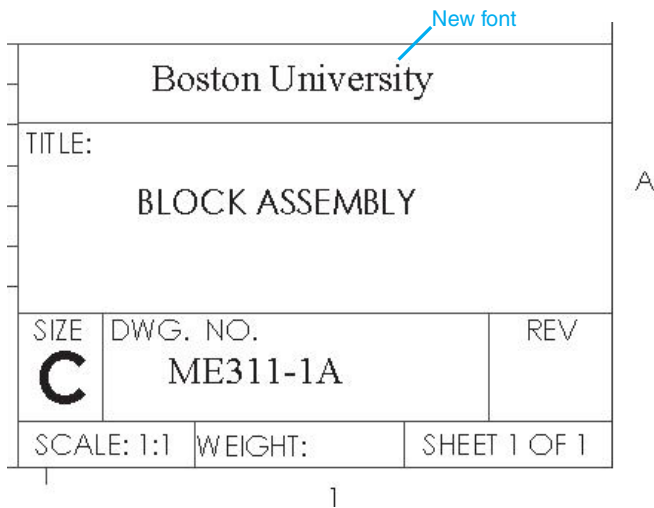
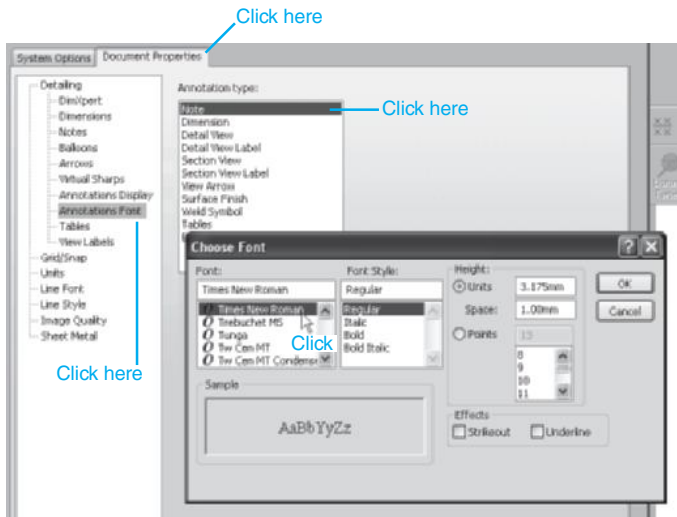


Figure 5-76

Referenced Drawings		See Chapter 8
ME-312A	EK 131-46	UNLESS OTHERWISE SPECIFIED:
		DIMENSIONS ARE IN INCHES
		TOLERANCES:
		FRACTIONAL ±
		ANGULAR: MACH ± BEND ±
		TWO PLACE DECIMAL ±
		THREE PLACE DECIMAL ±
		INTERPRET GEOMETRIC TOLERANCING PER:
		MATERIAL
		FINISH
NEXT ASSY	USED ON	
APPLICATION		DO NOT SCALE DRAWING

Figure 5-77

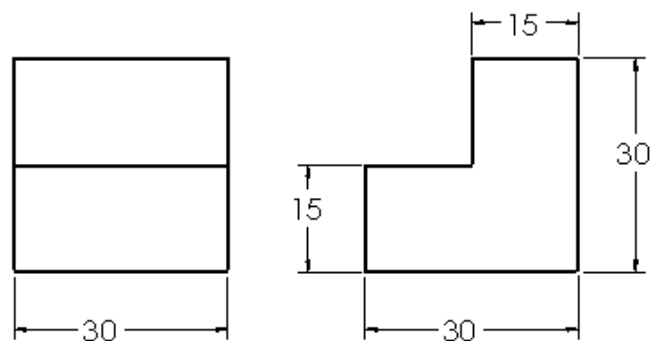
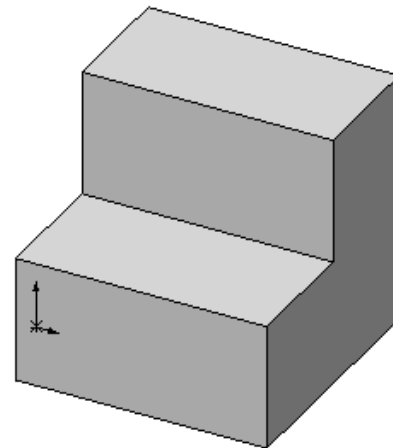
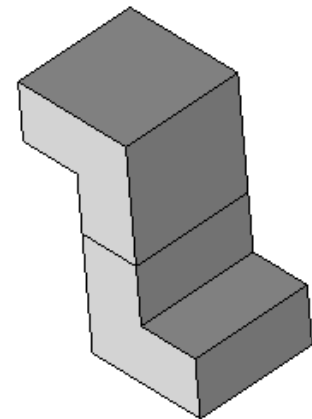


Figure P5-1 MILLIMETERS

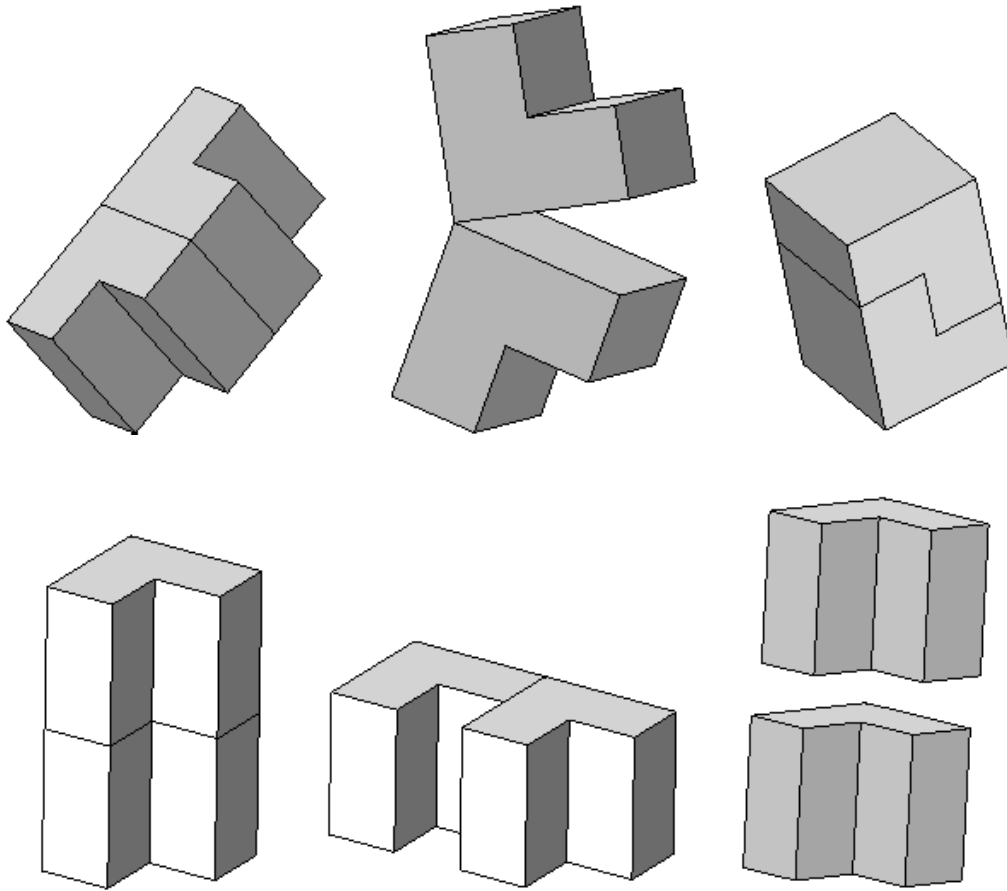


Figure P5-1 (continued)

**Project 5-2:**

Redraw the following models and save them as **Standard (mm).ipn** files. All dimensions are in millimeters.

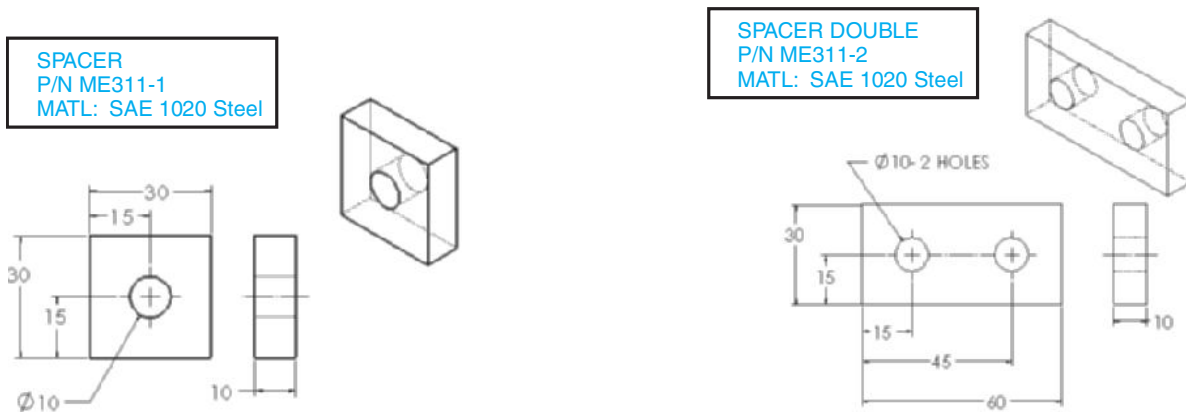
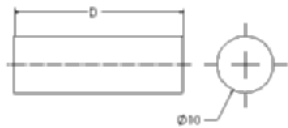
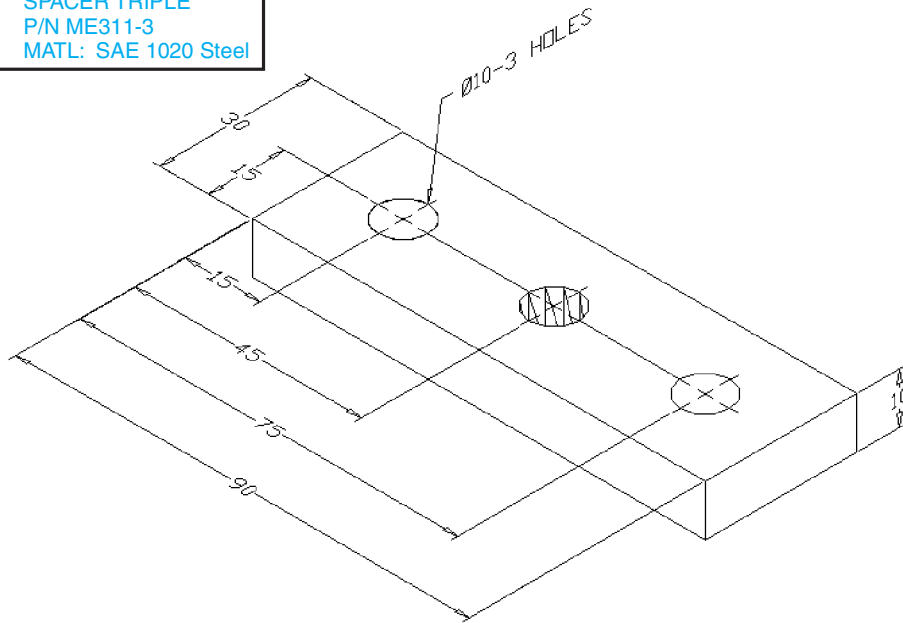


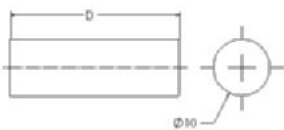
Figure P5-2 MILLIMETERS

SPACER TRIPLE  
P/N ME311-3  
MATL: SAE 1020 Steel

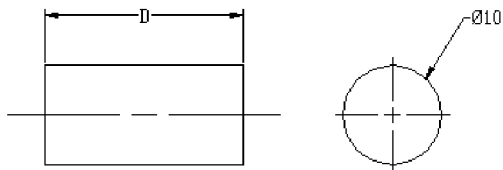


DESCRIPTION	PART NO.	D-VALUE
PEG, SHORT	PG20-1	20
PEG	PG30-1	30
PEG, LONG	PG40-1	40

PEGS  
MATL: Steel



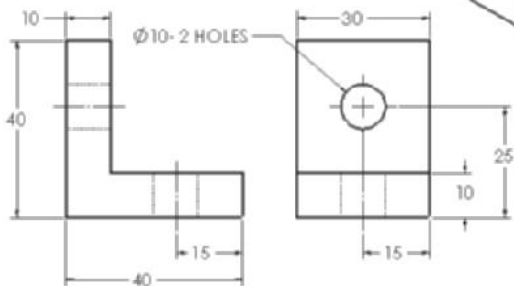
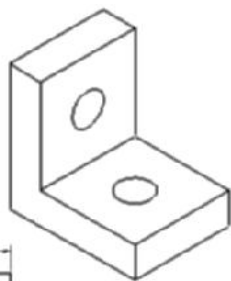
DESCRIPTION	PART NO.	D-VALUE
PEG, SHORT	PG20-1	20
PEG	PG30-1	30
PEG, LONG	PG40-1	40



DESCRIPTION	PART NO.	D
PEG, SHORT	PG20-1	20
PEG	PG30-1	30
PEG, LONG	PG40-1	40

ALL DISTANCES IN MILLIMETERS

L-BRACKET  
P/N BK20-1  
MATL: SAE 1040 Steel



Z-BRACKET  
P/N BK20-2  
MATL: SAE 1040 Steel

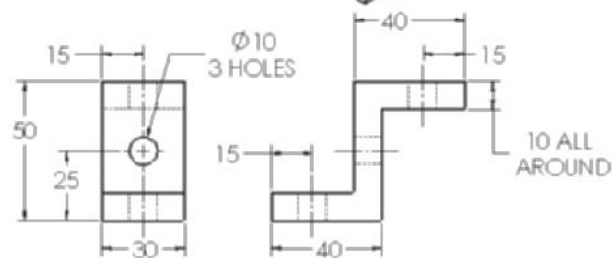
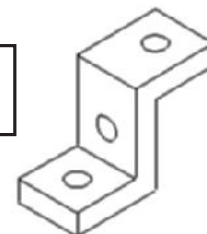


Figure P5-2 (continued)

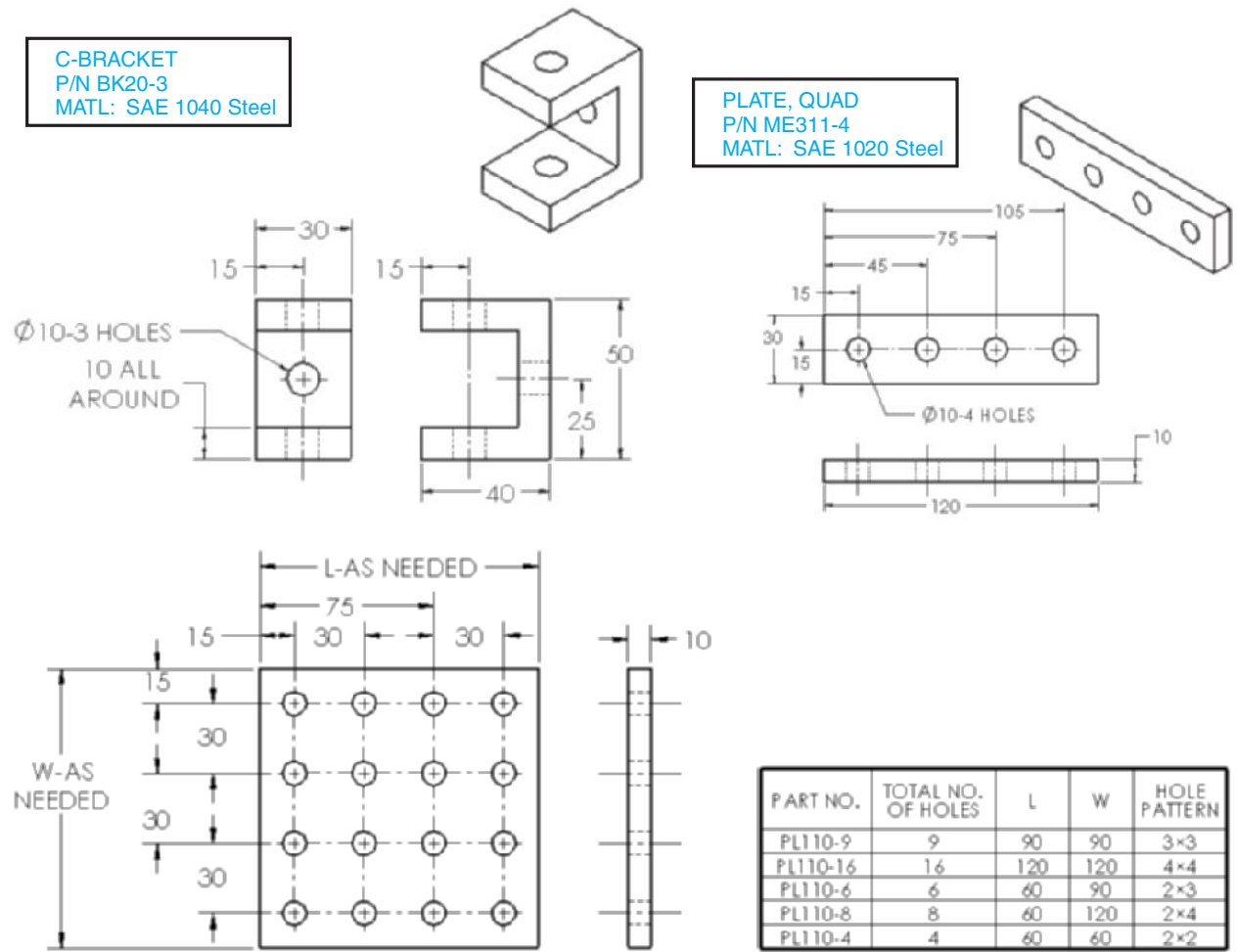


Figure P5-2 (continued)

**Project 5-3:**

Draw an exploded isometric assembly drawing of Assembly 1. Create a BOM.

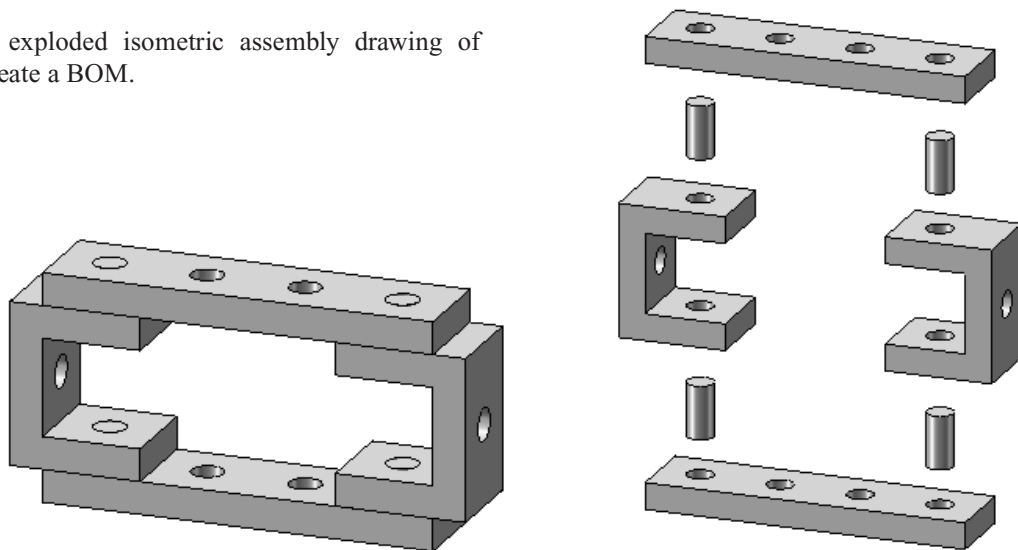
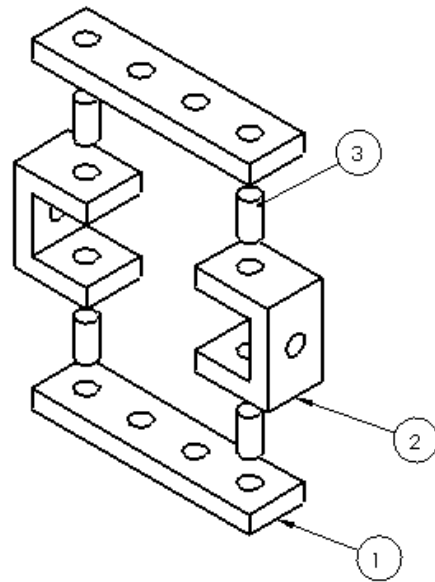


Figure P5-3 MILLIMETERS





ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	ME311-4	PLATE, QUAD	2
2	BK20-3	C-BRACKET	2
3	PG20-1	Ø12×20 PEG	4

Figure P5-3 (continued)

**Project 5-4:**

Draw an exploded isometric assembly drawing of Assembly 2. Create a BOM.

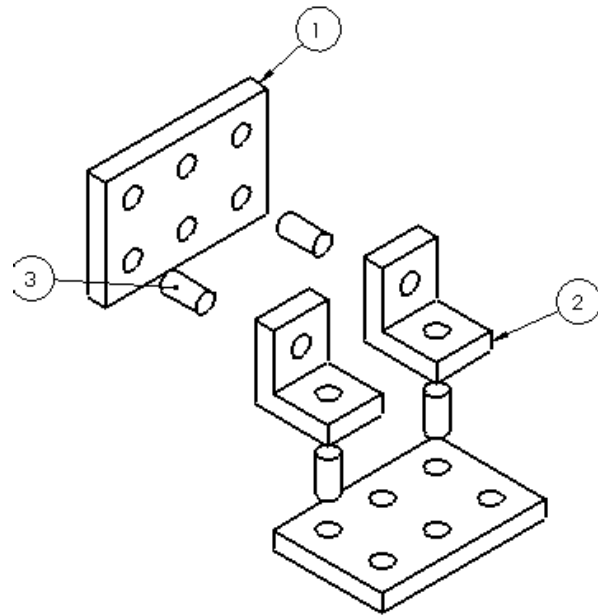
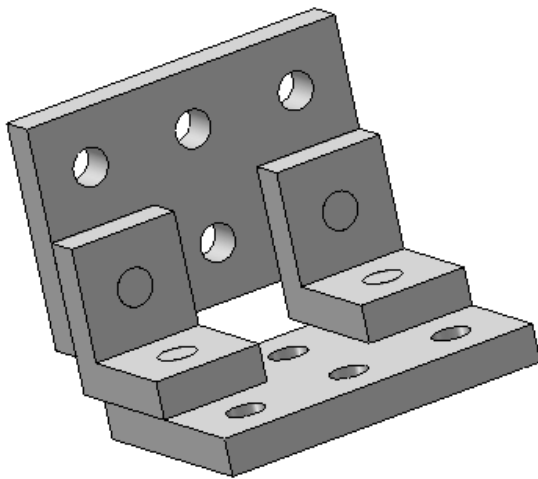


Figure P5-4 MILLIMETERS

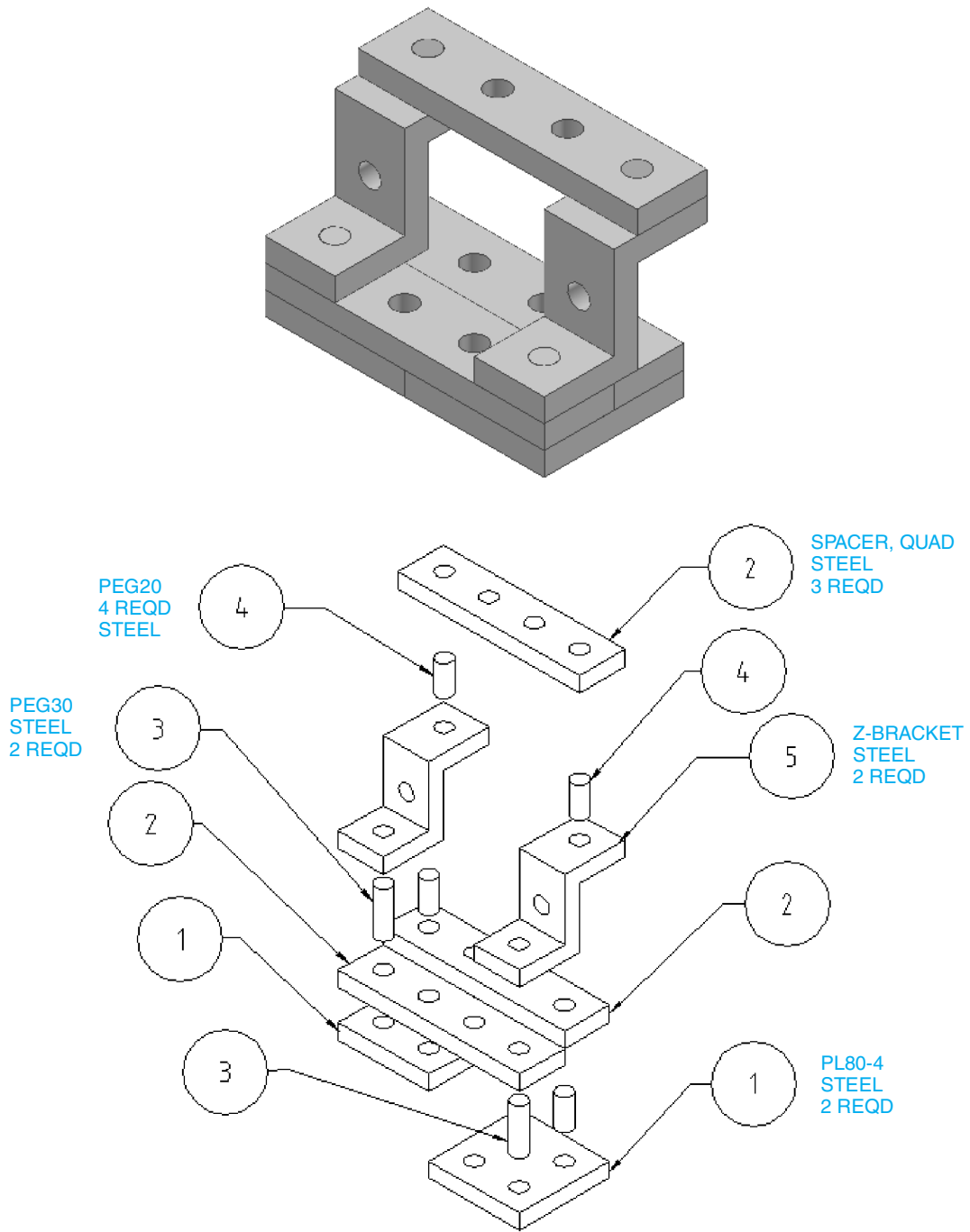


Figure P5-5

**Project 5-6:**

Draw an exploded isometric assembly drawing of Assembly 4. Create a BOM.

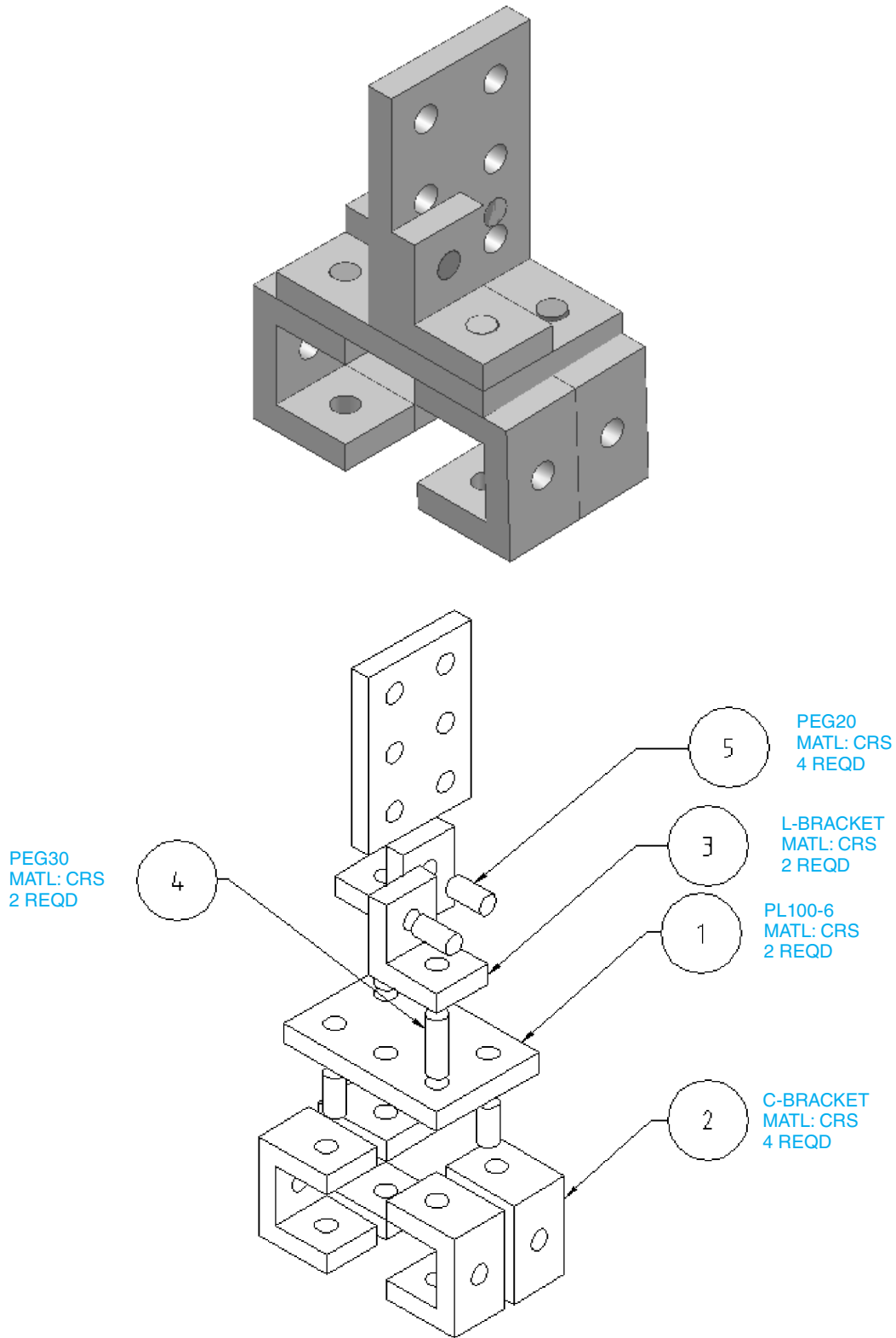
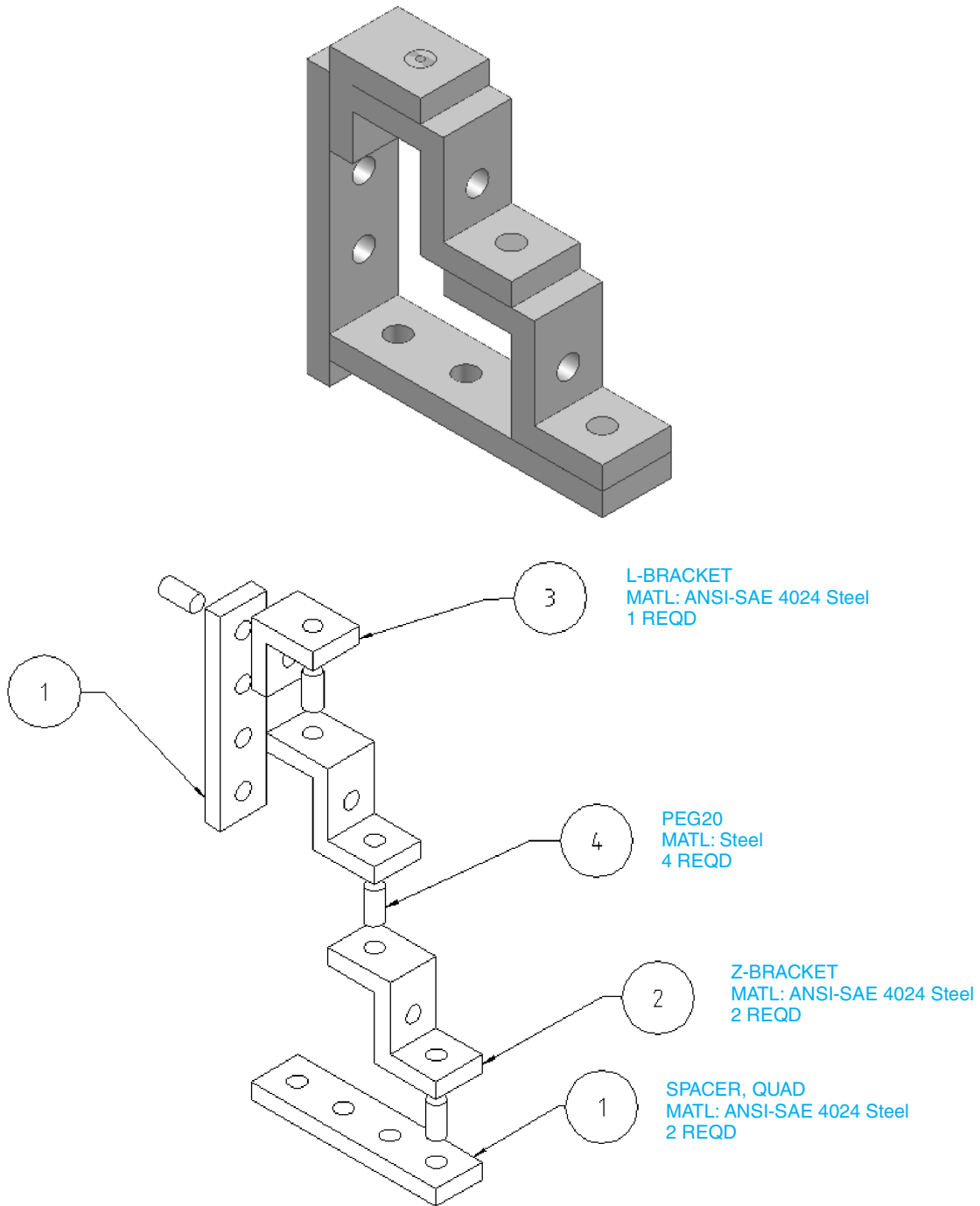


Figure P5-6 MILLIMETERS

**Project 5-7:**

Draw an exploded isometric assembly drawing of Assembly 5. Create a BOM.



**Figure P5-7 MILLIMETERS**

**Project 5-8:**

Draw an exploded isometric assembly drawing of Assembly 6. Create a BOM.

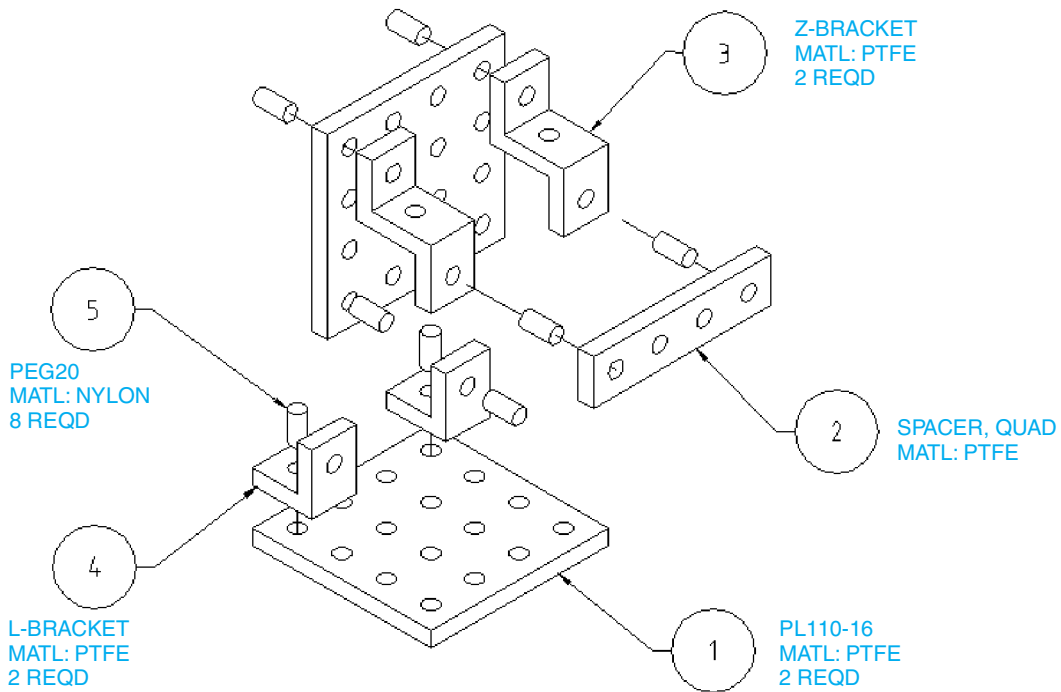
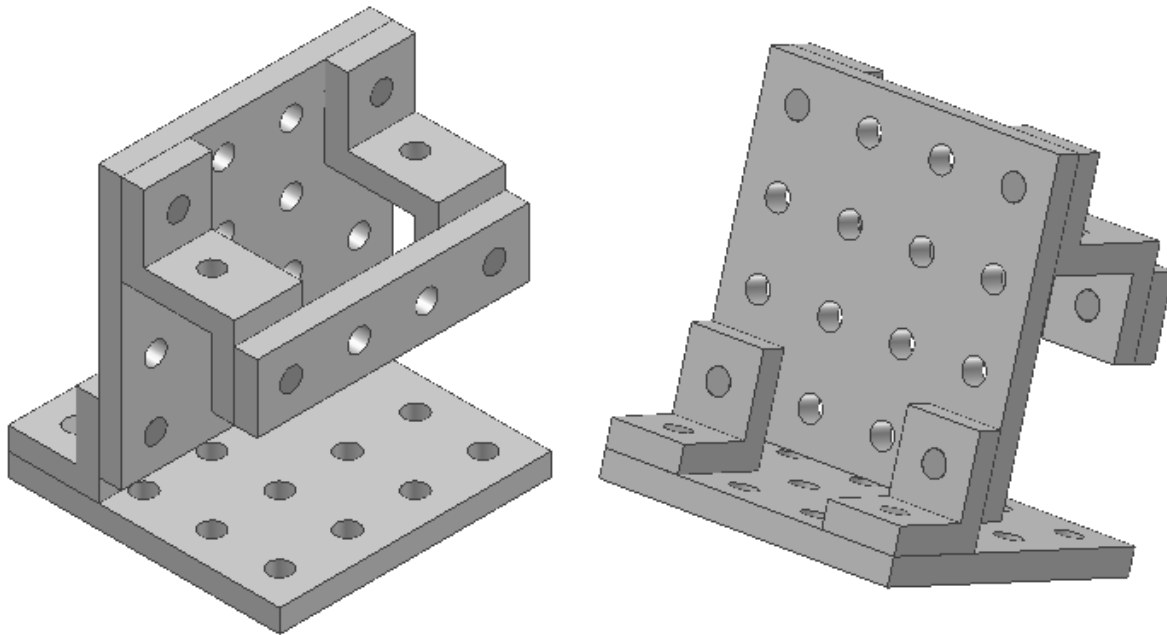


Figure P5-8

**Project 5-9:**

Create an original assembly based on the parts shown on pages 229–237. Include a scene, an exploded isometric drawing with assembly numbers, and a BOM. Use at least 12 parts.

**Project 5-10:**

Draw the ROTATOR ASSEMBLY shown. Include the following:

- A. An assembly drawing
- B. An exploded isometric drawing with assembly numbers

- C. A parts list
- D. An animated assembly drawing; the LINKs should rotate relative to the PLATE. The LINKs should carry the CROSSLINK. The CROSSLINK should remain parallel during the rotation.

**Note:**

This assembly was used in the section on animating assemblies See page 221.

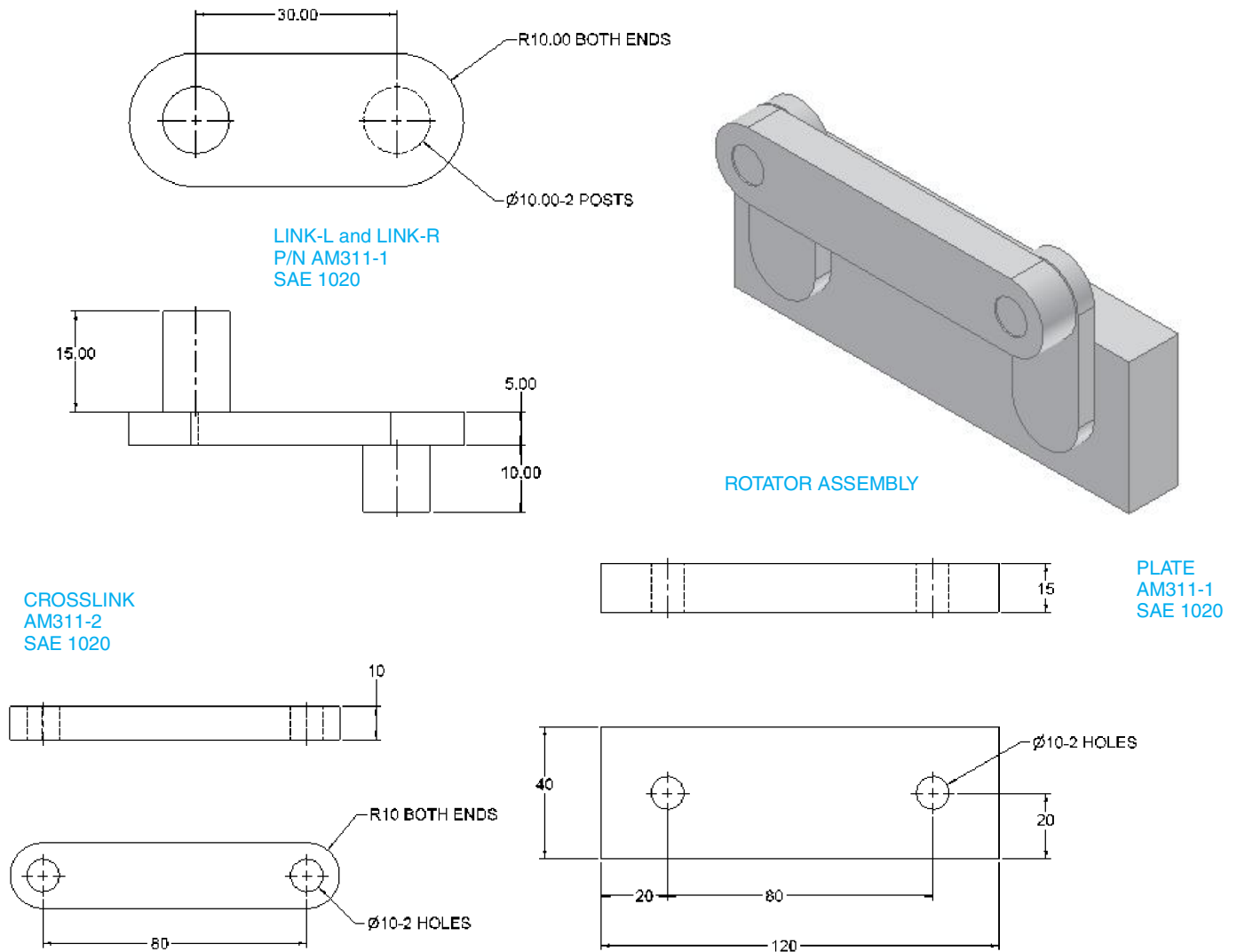


Figure P5-10

**Project 5-11:**

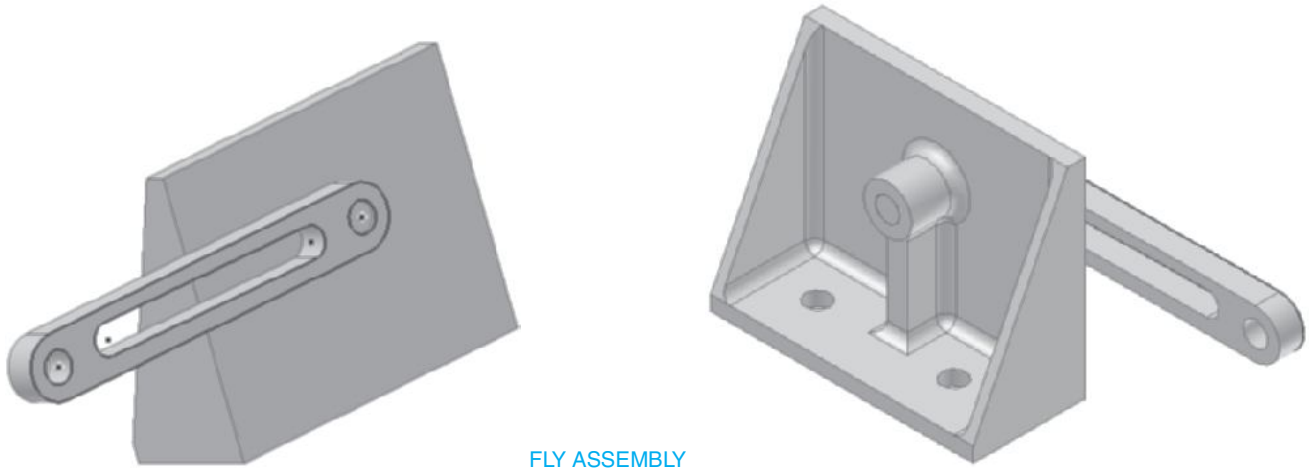
Draw the FLY ASSEMBLY shown. Include the following:

A. An assembly drawing

B. An exploded isometric drawing with assembly numbers

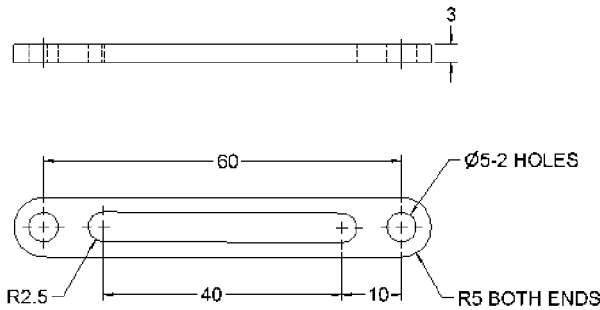
C. A parts list

D. An animated assembly drawing; the FLYLINK should rotate around the SUPPORT base.

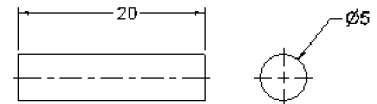


FLY ASSEMBLY

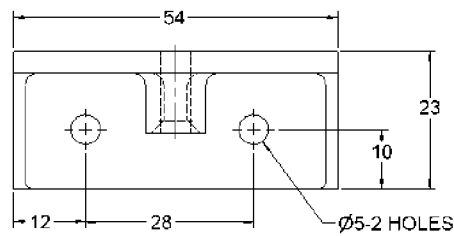
FLYLINK  
BU200A  
SAE 1040



PEGØ5  
BU-200C  
SAE1040



PLATE,SUPPORT  
BU200B  
SAE 1040



R2.0 FOR ALL FILLETS AND ROUNDS

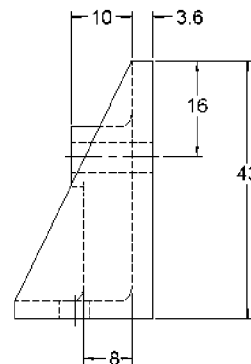
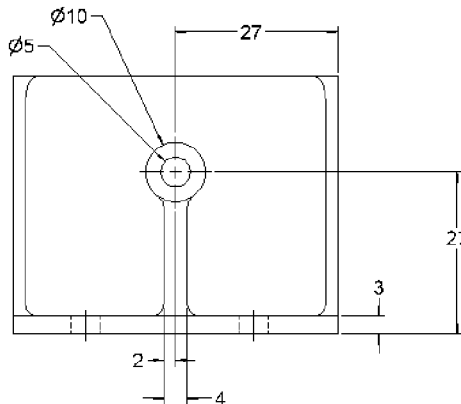
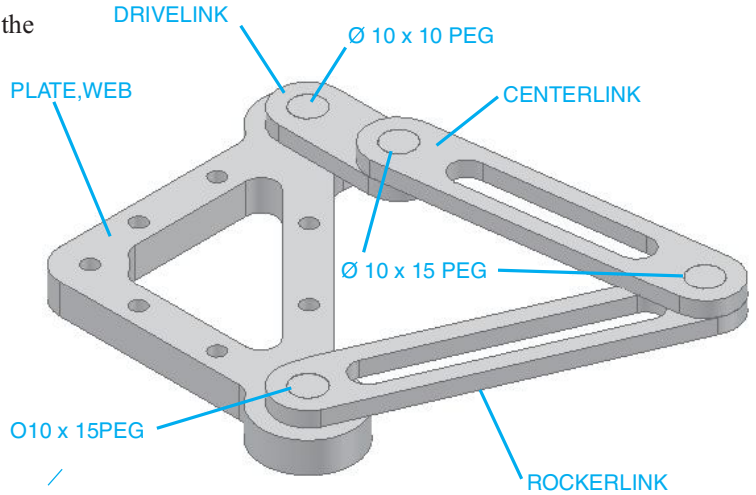


Figure P5-11

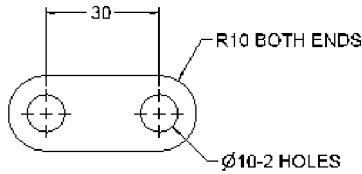
**Project 5-12:**

Draw the **ROCKER ASSEMBLY** shown. Include the following:

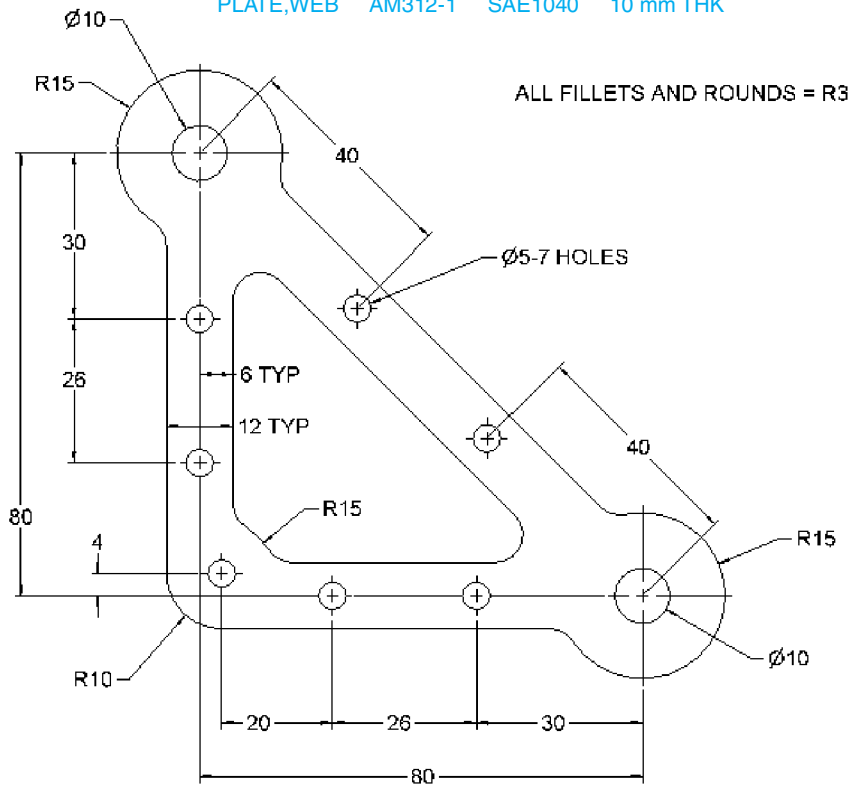
- A. An assembly drawing
- B. An exploded isometric drawing with assembly numbers
- C. A parts list
- D. An animated assembly drawing



DRIVELINK  
AM312-2  
SAE 1040  
5 mm THK

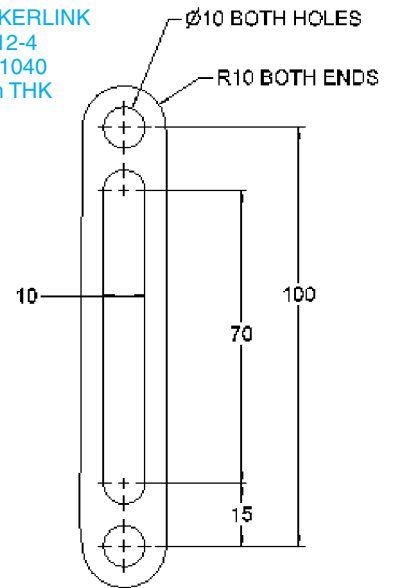


PLATE,WEB AM312-1 SAE1040 10 mm THK

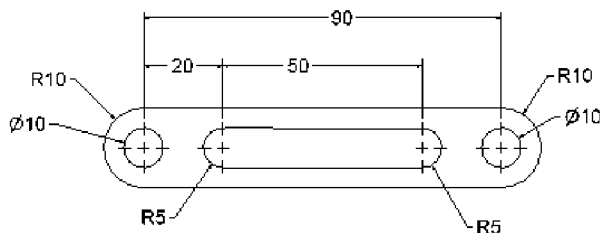


ALL FILLETS AND ROUNDS = R3

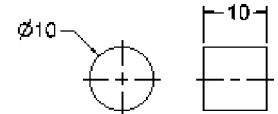
ROCKERLINK  
AM312-4  
SAE 1040  
5 mm THK



CENTERLINK  
AM312-3  
SAE1040  
5 mm THK



Ø 10 x 10 PEG  
AM312-5  
SAE 1020



Ø 10 x 15 PEG  
AM312-6  
SAE 1020

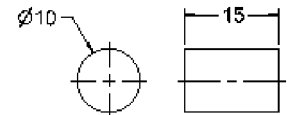


Figure P5-12



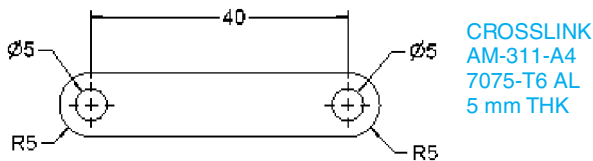
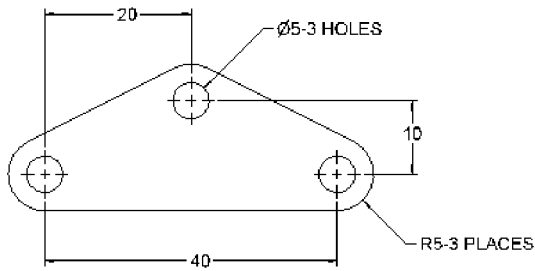
**Project 5-13:**

Draw the LINK ASSEMBLY shown. Include the following:

- A. An assembly drawing
- B. An exploded isometric drawing with assembly numbers

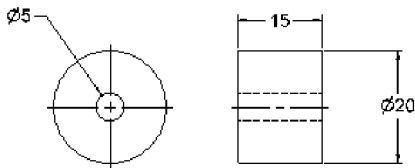
- C. A parts list
- D. An animated assembly drawing; the HOLDER ARM should rotate between  $-30^\circ$  and  $+30^\circ$ .

**HOLDER ARM**  
AM-311-A3  
7075-T6 AL  
5 mm THK

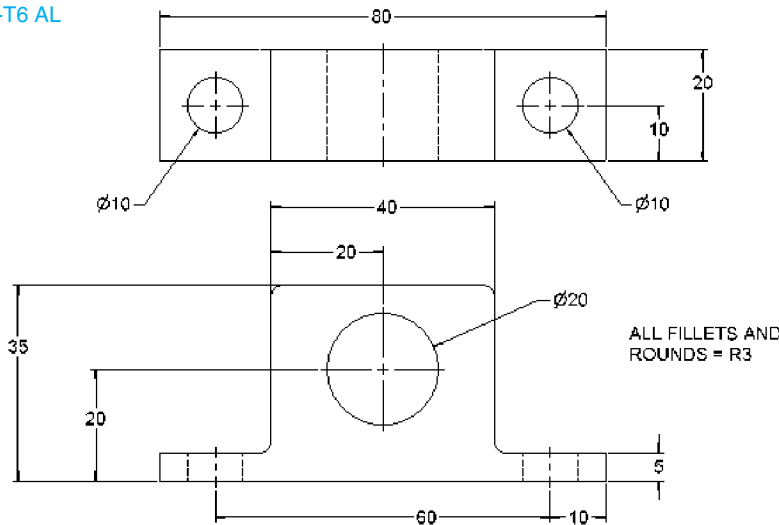


**CROSSLINK**  
AM-311-A4  
7075-T6 AL  
5 mm THK

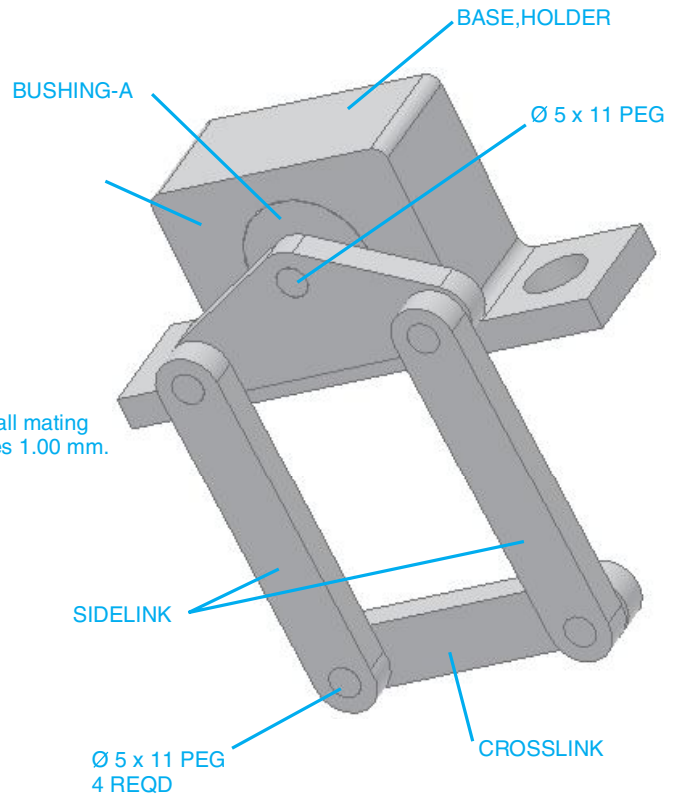
**BUSHING**  
AM-311-A5  
TEFLON



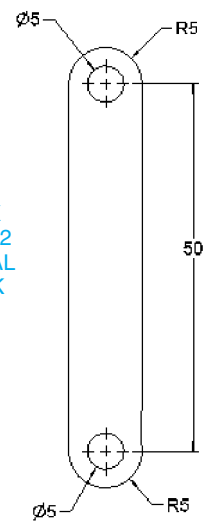
**BASE, HOLDER**  
AM-311-A1  
6061-T6 AL



**LINK ASSEMBLY**



**SIDELINK**  
AM-311-A2  
7075-T6 AL  
5 mm THK  
2 REQD

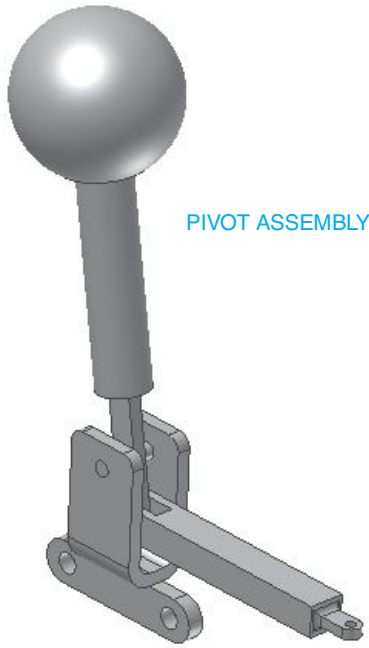


**Figure P5-13**

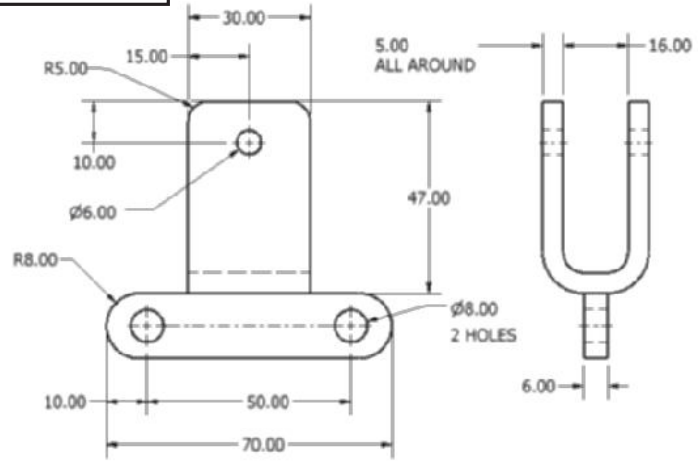
**Project 5-14:**

Draw the PIVOT ASSEMBLY shown using the dimensioned components given. Include the following:

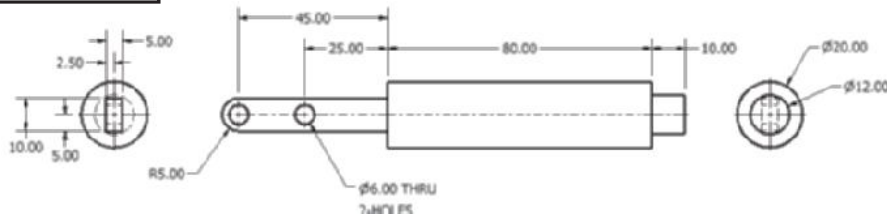
- A. A 3D exploded isometric drawing
- B. A parts list



BOX,PIVOT  
P/N: ENG-A43  
MATL: SAE 1020 STEEL



POST,HANDLE  
P/N: ENG-A44  
MATL: SAE 1020 STEEL



LINK  
P/N: ENG-A45  
MATL: SAE 1020 STEEL

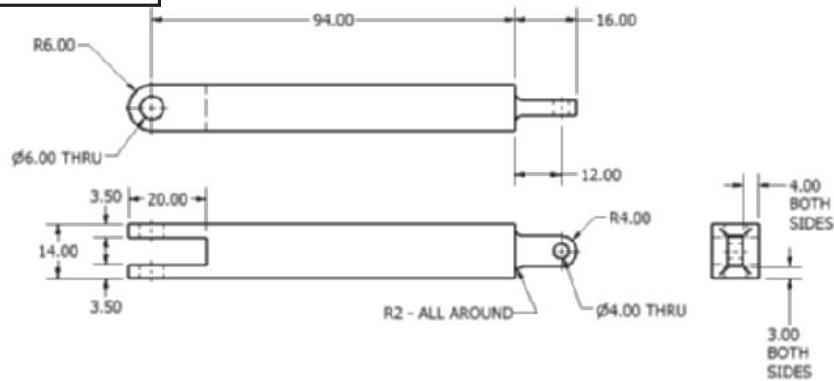


Figure P5-14 MILLIMETERS

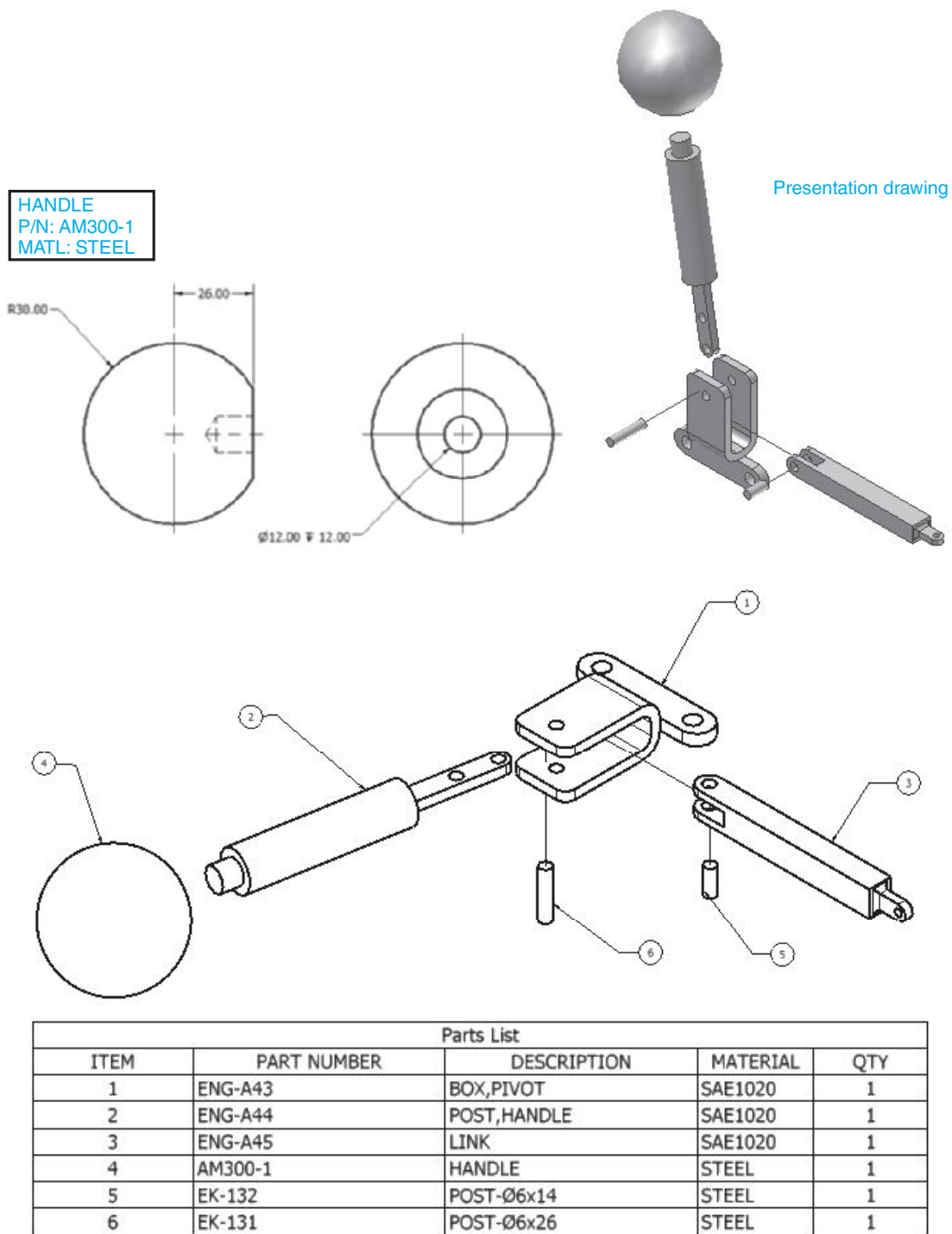


Figure P5-14 (continued)

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# Threads and Fasteners

## Objectives

- Explain thread terminology and conventions.
- Learn how to draw threads.
- Learn how to size both internal and external threads.
- Learn how to use standard-sized threads.
- Learn how to use and size washers, nuts, and set screws.

## 6-1 INTRODUCTION

This chapter explains how to draw threads, washers, and nuts. It also explains how to select fasteners, washers, nuts, and setscrews.

Internal threads are created using the **Hole Wizard** tool, which is located with the **Features** tools. Predrawn fasteners and other standard components may be accessed using the **Design Library**. See Figure 6-1.

### Note:

The **Design Library** is an add-in accessed through the **Tools** toolbar.

All threads in this book are in compliance with ANSI (American National Standards Institute) standards—ANSI Inch and ANSI Metric threads.

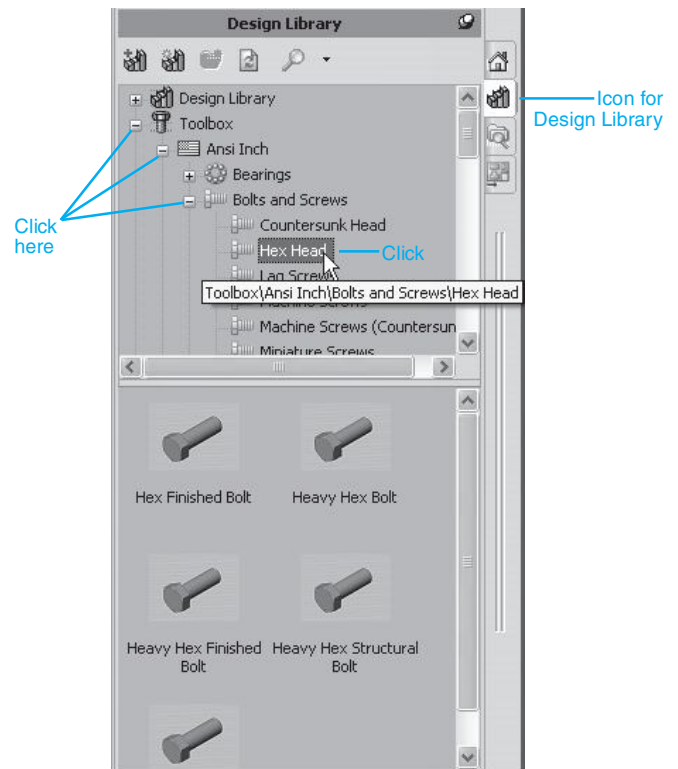


Figure 6-1