Chapter

17

Creating a 3D Assembly Drawing

In this chapter, you will learn the following to World Class standards:

- **Making your first 3D Assembly Drawing**
- **The XREF command**
- **Making and Saving Part One, the Rail**
- **Making and Saving Part Two, the Leg**
- **Making and Saving Part Three, the Brace**
- **Making and Saving Part Four, the Foot Rest**
- **Creating the Assembly File**
- **Using the XREF Tool to Import the First Part Drawing**
- **Using the XREF Tool to Import the Second Part Drawing**
- **Using the XREF Tool to Import the Third Part Drawing**
- **Using the XREF Tool to Import the Fourth Part Drawing**

Making Your First Assembly Drawing

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In the world of three dimensional computer aided design, you create a single product by concentrating on a group of solid parts, bringing them together into a collection of joining components. This system of design has replaced the older and antiquated two-dimensional layout. The many views and tools allow the designer to examine parts and look for interferences. With the ability to add material and lighting, you can inspect the assembly for cosmetic appearance as well. If developing solid parts in earlier chapters inspired you, and generating multiple orthographic views in seconds gave you a sense of modern-day efficiency, you join all true designers in experiencing the breathtaking ability to create inexpensive and complex assemblies. An architect or engineer who has control of 3D assembly technology can take the driver's seat in the design process.

In the history of assembly drawings, such as those originated by Leonardo Da Vinci, designers penciled artistic renderings by hand to portray assemblies such as a church or defensive structures. In the 20th century, layout drawings showed each part to half or full scale to identify the relationship between objects. Even though designers began to adapt to two-dimensional computer aided design systems, many of the similar processes used in the previous decades and centuries proved adequate… until three-dimensional modeling came on the scene.
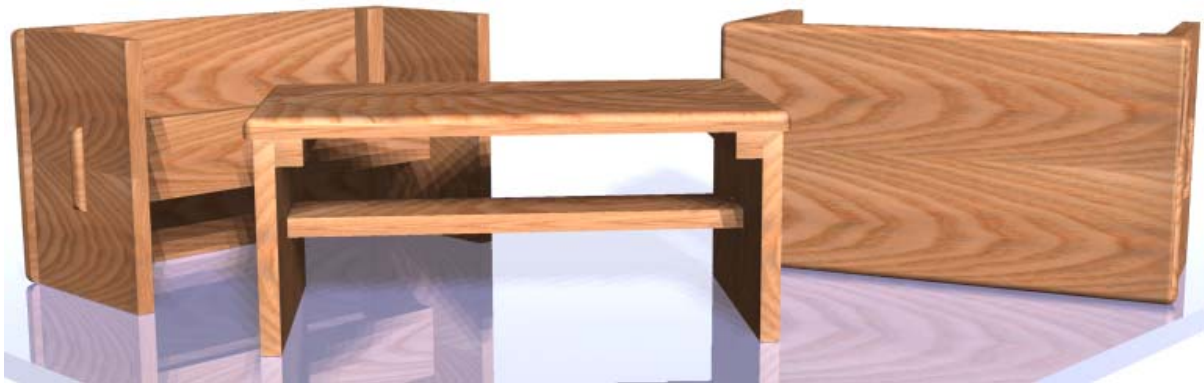

Figure 17.1 – A marketing image of your first simple three-dimensional assembly

In this chapter, you will be creating a straightforward compilation of wooden pieces like those shown in Figure 17.1. The assembly procedure for a three-dimensional design is to create each part in a separate computer file, and then unite the solids in the assembly-drawing file using the External Reference function. You will still be able to build an orthographic layout in Paper Space, enabling you to add overall dimensions, a bill of material, and assembly notes explaining how to build the manufactured good. In this lesson, you will construct a simple assembly that has four relatively trouble-free parts.

The XREF Command

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The concept of using a drawing reference as opposed to a drawing block is very important to understand when beginning to build assemblies. Imagine a house with thousands of components. If you insert each file using copy and paste or the insert block function, each solid enters into the home assembly with all the part's computer definitions, which existed in the individual part-drawing file. After adding multiple pieces into the assembly drawing, the computer file size will show an enormous quantity of bytes. In other words, as you construct the virtual house in Model Space, if you periodically check the properties of the drawing file you will soon notice your drawing has significantly expanded in file size, slowing your personal computer's performance dramatically.

The External Reference function solves this dilemma for you. Instead of inserting the external block, you insert the reference material of the data into the assembly drawing. This process is akin to a modern web site, which has links between the home page and the other web pages in the server. AutoCAD creates a link to the part's computer file so the assembly drawing does not grow to enormous size. You will need to use the Reference panel under the Insert tab as shown in Figure 17.2 to find the tools to place references in your drawing.
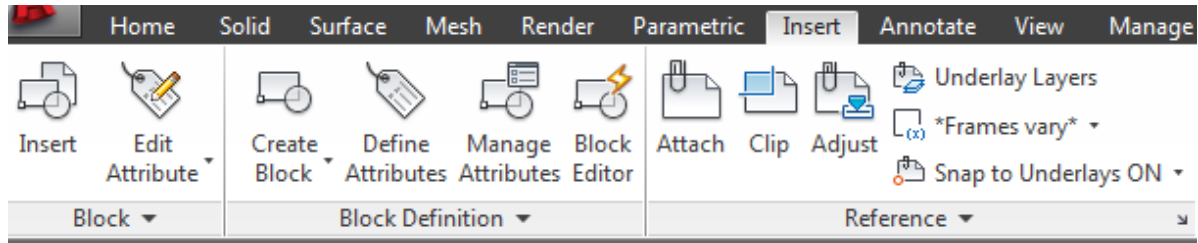

Figure 17.2 – Reference panel

Making and Saving Part One, the Rail

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As with the practice solid problems, usually an engineer or designer will provide you the drafter with a sketch. The first solid part in the assembly is just a rectangular box. The dimensions of the geometric solid are 1.0 inch by 1.0 inch by 11.0 inches as shown in Figure 17.3. Again, basic tools like the Box command will come in handy to create simple solids.

Many junior designers will receive a set of sketches from their supervisor and pick the most difficult geometric shape to load into model space first. In a perfect world where the customer or production is not in a hurry, this technique would be satisfactory because you will release the production package, containing the assembly drawing and the four part drawings, to a manufacturer as a single package. In this case your manager would be running a serial production system by completing each design step before phasing into the building of the prototype. However, in the last decade many supervisors have transitioned to using a parallel method to speed through the design process. In such a case, you will want to select the easiest part, or a component with a long lead or manufacturing time, to draw first so that

as you finish the virtual prototype in Model Space the first production unit will simultaneously be finished on the shop floor.

This design does not have any lengthy deliverable items, so you can start with the easiest part first. You need to follow the 3D modeling procedures learned from your earlier work. Always refer to the sketch first and use the written procedures as a reference.
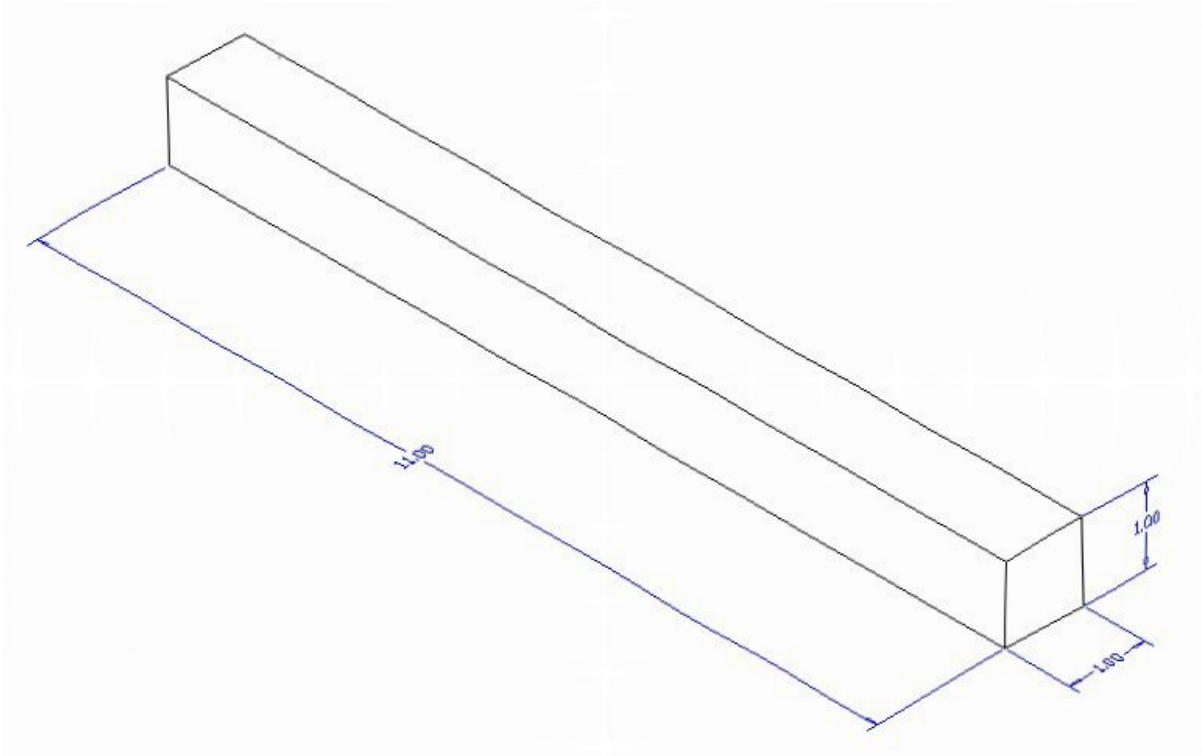

Figure 17.3 – Part one sketch

Starting the first 3D Part Drawing, the Rail
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In Figure 17.4, the window "Select template" is loaded by selecting the New tool on the Quick Access Toolbar. Scroll down the list and until you see the Mechanical.dwt template and select it.

Figure 17.4 – Starting the drawing using the Mechanical template

To draw the solid rectangular box, select the Box tool on the Modeling panel. At the command line, a prompt asks you to specify the corner of the box or to type CE to specify the center of the object. You will need to identify the corner of the box by picking anywhere in the middle of the graphical display. After picking the starting point for the box, the system asks you to specify the other corner. Type **L** for the length option of the Box command and hit **ENTER**. For part one, type **1.0** for the length, **11.0** for the width, and **1.0** for the height. Select the southwest isometric view to see all three dimension of the part as shown in Figure 17.5.

Figure 17.5 – Creating part one and moving the part to the origin

To move part one to the drawing origin of 0,0,0 on the x, y, and z axis, pick the Move tool on the Modify panel, select the 3D solid, and hit ENTER. At the prompt, "specify base point or displacement", select the endpoint shown in Figure 17.5 at the lower right hand side of the entity, which will become part one's insertion point into the assembly drawing. At the "second point of displacement", type a # symbol (with Dynamic Input toggled on) and then **0,0,0** and **ENTER**.

To save Problem 1 in your Fundamentals of 3D Drawing Folder, select the Save tool on the Quick Access toolbar. The "Save Drawing As" window will appear in your graphical display. In the Save In list box, select your drawing folder. You will need to create an assembly folder. Select the New Folder tool, which is the yellow folder with a sparkle in the upper right corner as shown in Figure 17.6. When the folder with the name "new folder" appears in the Save Drawing As display, type the words "Foot Rest Assembly" and hit ENTER. Double click on your new assembly folder to open the Foot Rest Assembly folder. At the File Name textbox, type "Rail" and hit the Save button to save the drawing (Figure 17.7).

You can finish many simple parts in every assembly in minutes. Imagine how productive you can be if you start to follow these simple procedures.
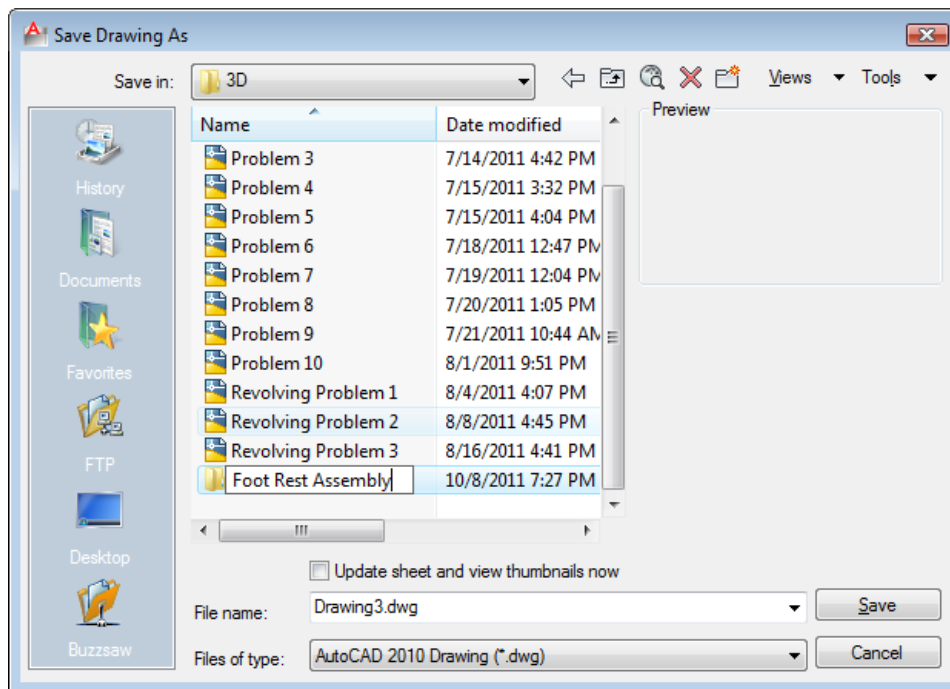

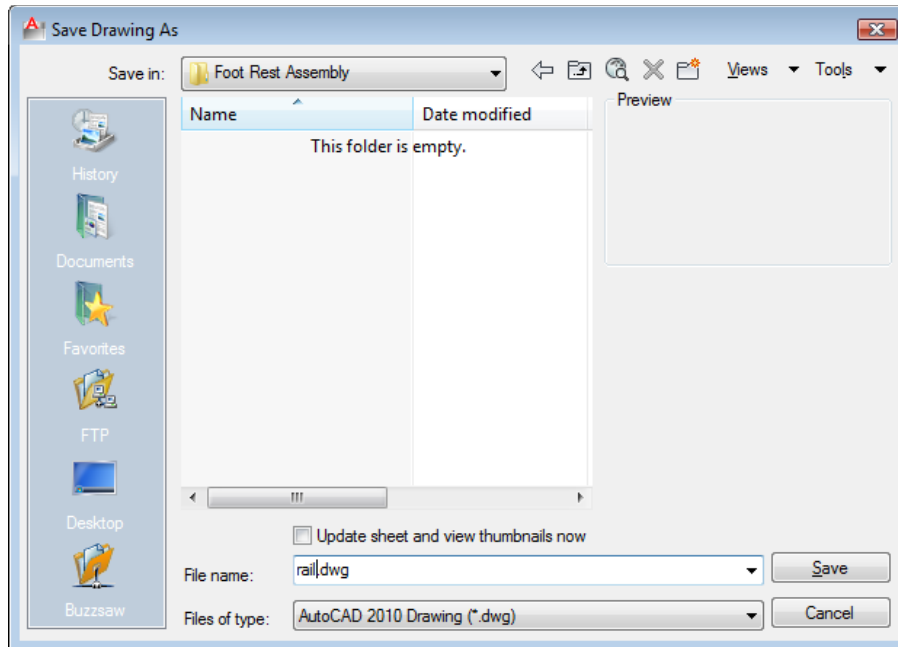**Figure 17.6 – Creating a folder for the Foot Rest Assembly drawing**

Figure 17.7 – Saving the rail file in the Foot Rest Assembly folder

Making and Saving Part Two, the Leg

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Part two has a more complex shape as seen from the side view. You will use the Polyline function to draw the perimeter and extrude the shape. Start the problem with the mechanical template (Figure 17.9).
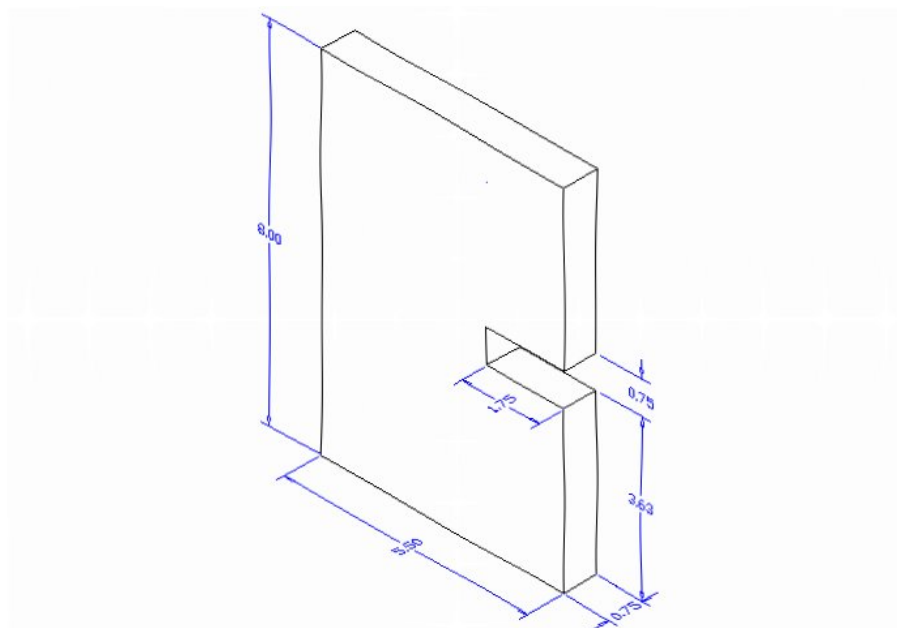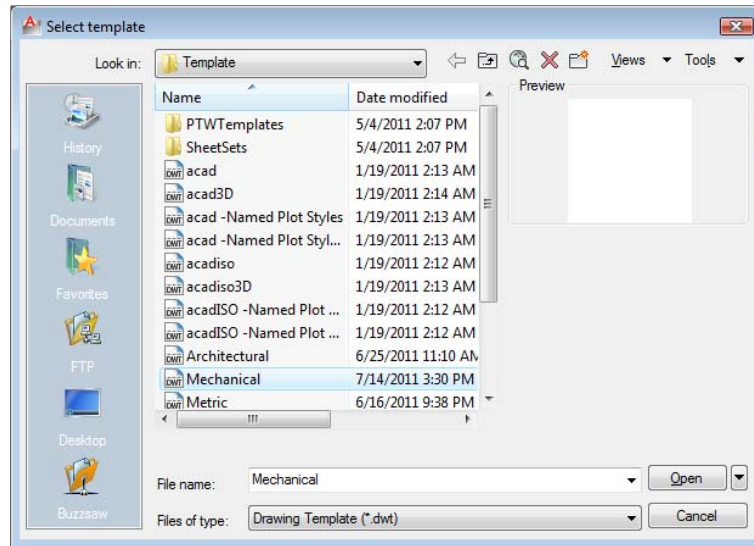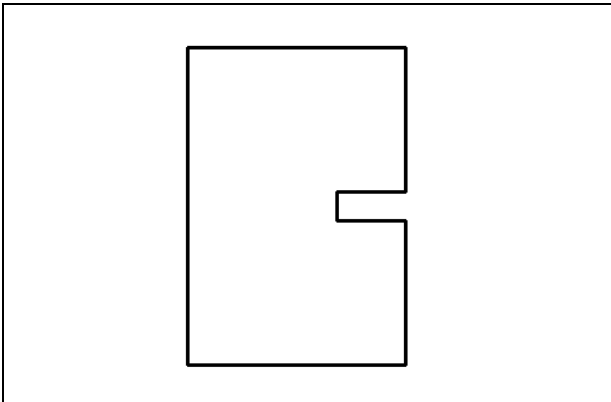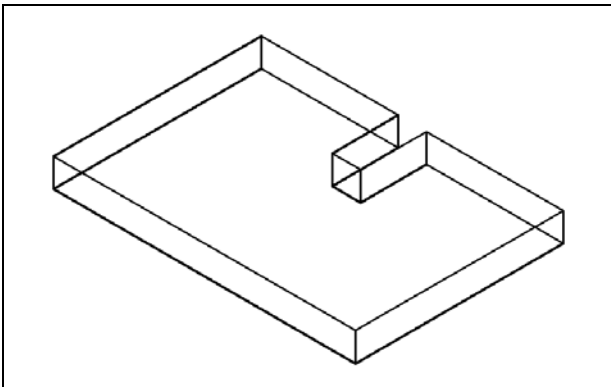

Figure 17.8 – Part two sketch

Figure 17.9 – Starting the drawing using the Mechanical template

You will draw the contour of this solid similarly to those drawn in previous chapters. On the Draw panel, select the Polyline tool and specify a start point in the lower left hand corner of the graphical display.
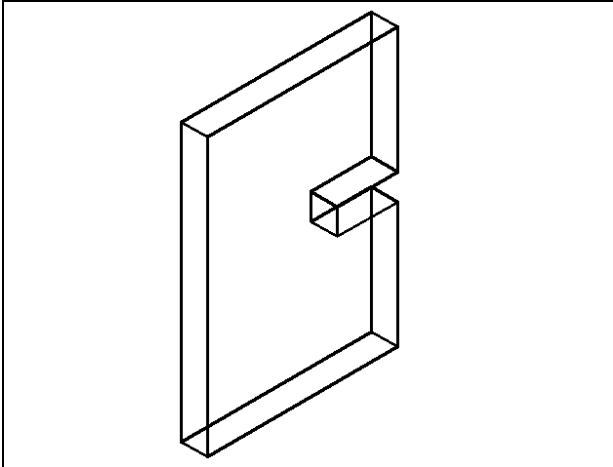

With the Ortho mode "on", draw a **5.5** unit line to the right, a **3.625** line up, a **1.75** line to the left, a **0.75** line upward, a **1.75** line to the right, a **3.625** line upward, a **5.5** to the left, and then type "**C**" and ENTER to close the Polyline. A closed entity will appear in the graphical display as shown in Figure 17.10. Select the southwest isometric view to prepare to extrude the solid shape.
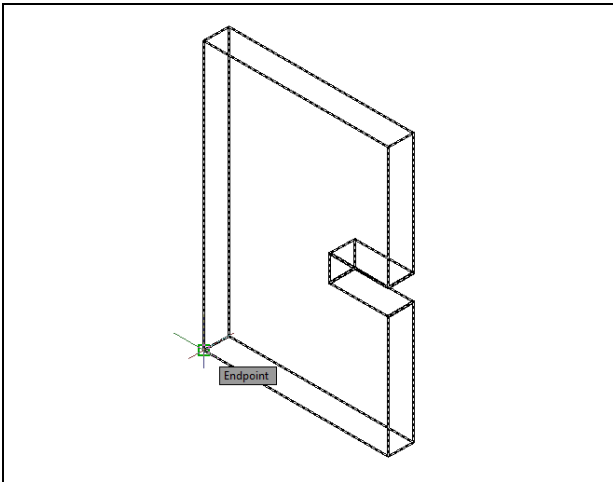
Figure 17.10 – Drawing the shape

Select the Extrude tool on the Modeling panel and the command line will prompt you to "**select objects**". Select the closed polyline and the command line will return with "**1 found**". Hit **ENTER** to specify the height or path of the solid. Type "**0.75**" for the height of the extrusion and hit **ENTER** to accept the zero degree default for the angle of taper. A more complex 3D solid will appear in the graphical display as shown in Figure 17.11.

Figure 17.11 – Extruding the shape

Select 3D Rotate from the Modify panel. At the command, "select objects", pick the solid on the graphical display and hit **ENTER** to proceed to the second part of the function. You are going to rotate the solid part on the x-axis, so pick the midpoint of the bottom right edge as the base point and then select the red diameter to pick the x-axis as the rotation axis. With the Ortho mode still "On" specify an angle start point and then rotate the part 90 degrees so it appears as shown in Figure 17.12.

Figure 17.12 – 3D Rotate the solid

To rotate the solid on the XY plane, select the regular Rotate command on the Modify panel and pick the solid. The selected entity will appear dashed, so to continue with the command, hit **ENTER** to select the base point of rotation. Select any point at the base of the solid with the left mouse button and then with the Ortho mode on move the mouse to rotate the shape 90 degrees clockwise (Figure 17.13). Before you save, make sure to move the part to the origin (0,0,0). Use the back corner of the part as shown in Figure 17.13 as the base point.

Figure 17.13 – Rotate and move the solid

To save Problem Two in your Foot Rest Assembly folder, select the Save tool on the Quick Access toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. Type "Leg" in the File Name textbox and hit the Save button to save the drawing (Figure 17.14).
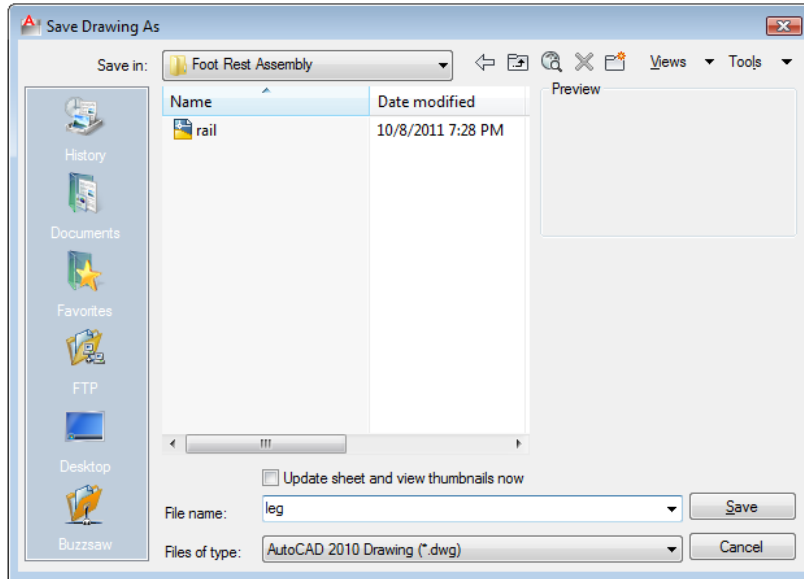
Figure 17.14 – Saving the Leg file in the Foot Rest Assembly folder

Making and Saving Part Three, the Brace

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Part three is just a rectangular box with rounded ends. The dimensions of the geometric solid are 19.875" x 3.5" x 0.75" as shown in Figure 17.15. Again, basic tools like the Box command will come in handy to create simple solids. You will use the Fillet command to round the corners of each end.
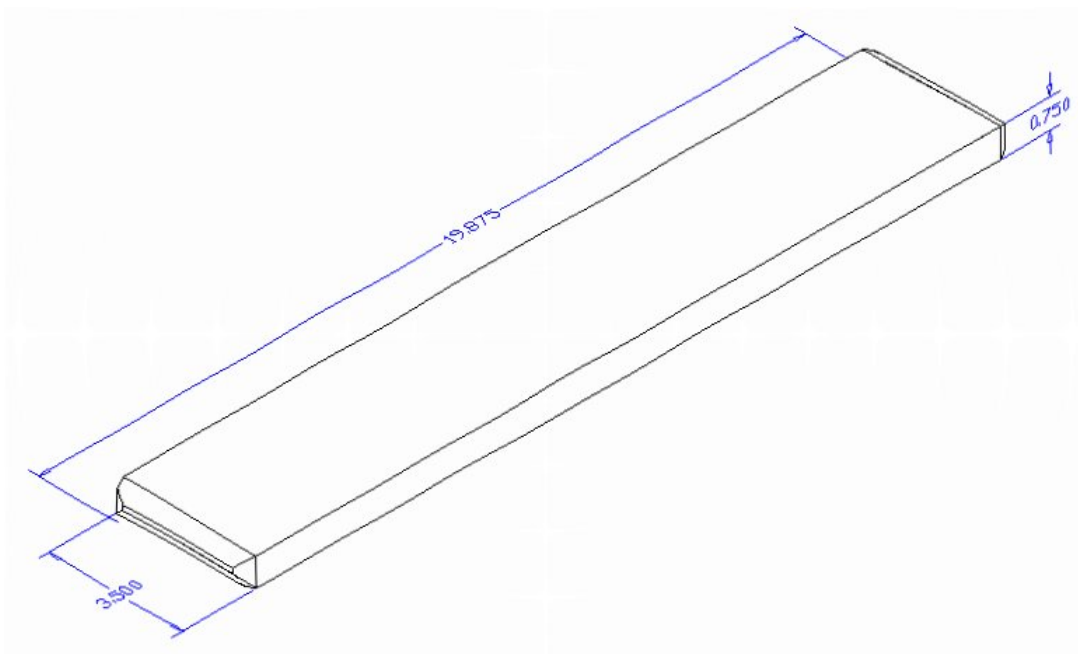

Figure 17.15 – Part Three sketch

In Figure 17.16, the window "Select template" is loaded by selecting the New tool on the Quick Access toolbar. Scroll down the list and until you see the template, Mechanical.dwt and select it.
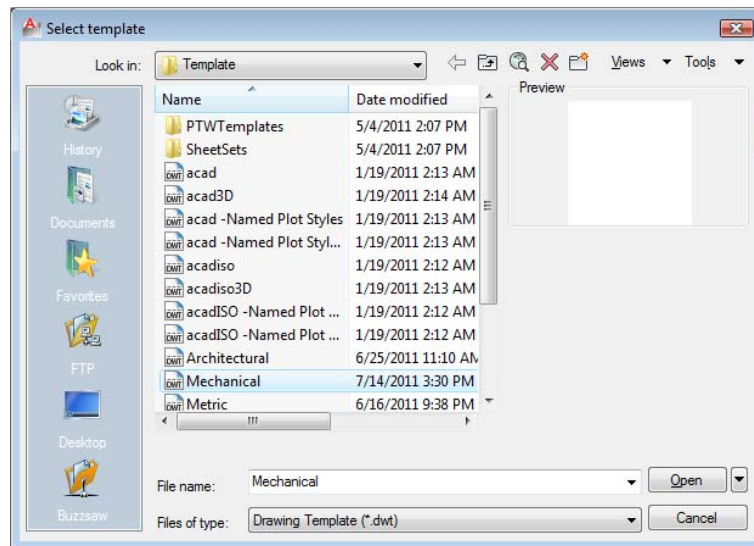

Figure 17.16 – Starting the drawing using the Mechanical template

To draw the solid rectangular box, select the Box tool on the Modeling panel. At the command line, a prompt asks you to specify the corner of the box or to type CE to specify the center of the object. You will need to identify the corner of the box by picking anywhere in the middle of the graphical display. After picking the starting point for the box, the system will ask you to denote a corner; type **L** for length and hit **ENTER**. Type **19.875** for the **X** coordinate, **3.5** for the **Y** coordinate, and **0.75** for the **Z** coordinate. Switch to the SW Isometric view and the solid will appear in the graphical display as shown in Figure 17.17.
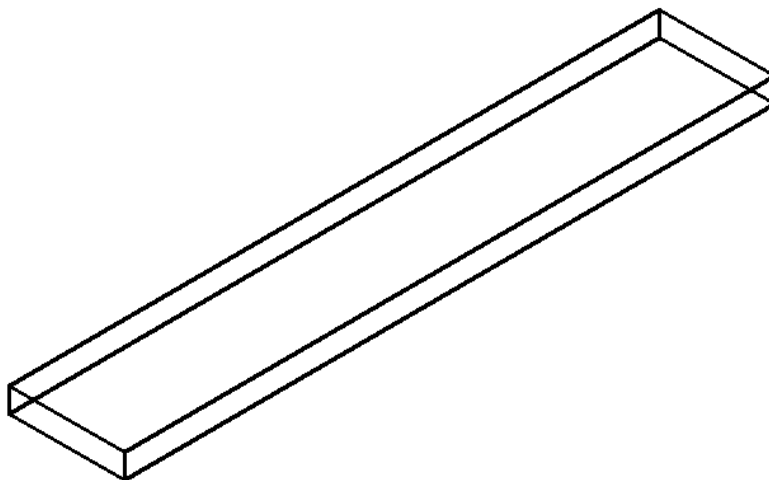

Figure 17.17 – Part Three Master Solid

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Select the Fillet tool on the Modify panel. After picking the tool, type "**R**" for radius and then pick the vertical line on the back corner (shown in red in Figure 17.18). Enter "**0.313**" as the fillet radius and hit **ENTER**. At the prompt, "**Select an edge or [Chain/Loop/Radius]:**" pick the other three lines of the face of the solid (shown in blue in Figure 17.18). Hit **ENTER** and the rounded edges will appear as shown in Figure 17.19. Repeat the process for the other side of problem three and you will have symmetrical fillets on both ends. Before you save, make sure to move the part to the origin (0,0,0). Use the front bottom corner of the part as shown in Figure 17.20 as the base point.
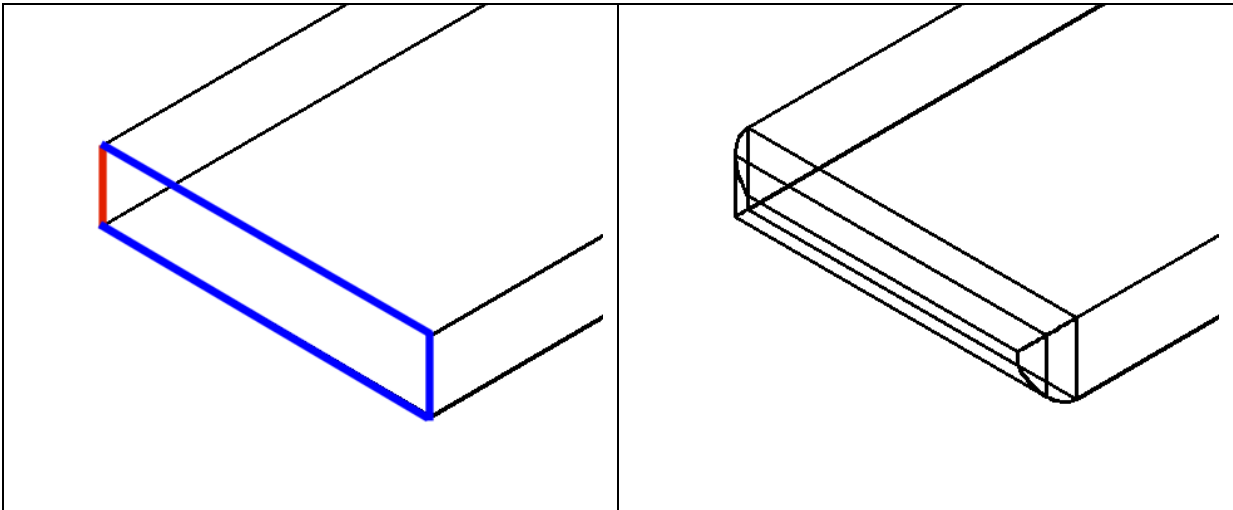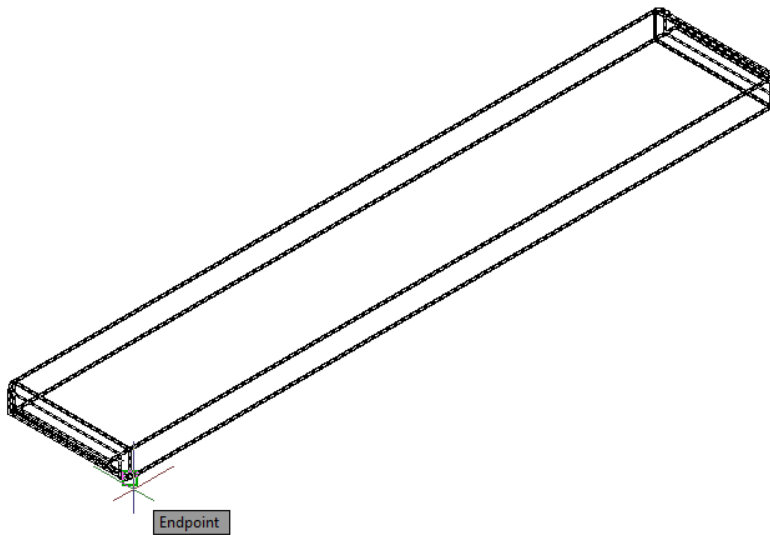

Figure 17.18 – The three edges   **Figure 17.19 – The 0.313 fillet**

To save Problem Three as shown in Figure 17.20 in your Foot Rest Assembly folder, select the Save tool on the Quick Access toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. In the File Name textbox, type "Brace" and press the Save button to save the drawing (Figure 17.21).

Figure 17.20 – The finished Brace

In larger projects like assemblies, you will want to save more often. In architectural floor plans and mechanical assemblies, plan on saving on a regular basis. Once you initially save the file, to repeat the process, choose the Save tool on the Quick Access toolbar. The Save As

window will not appear because the file already has a name. So, save when you leave your desk. Save when you answer the phone. Save when you are in a discussion with someone. In every case, always save your file, whether distracted or the last save time exceeds 10 minutes.
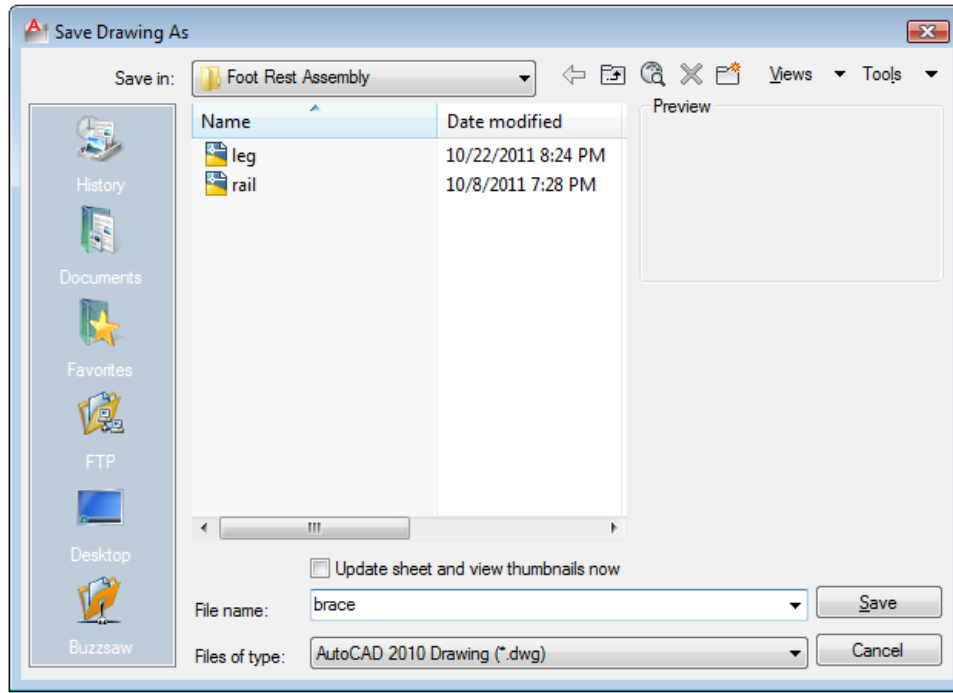

Figure 17.21 – Saving the Brace file in the Foot Rest Assembly folder

Making and Saving Part Four, the Foot Rest

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Part four is just a rectangular box with rounded ends on three sides. The dimensions of the geometric solid are 20.0 inches by 5.5 inches by 0.75 inches as shown in Figure 17.22. Again, the basic tools like the Box command will be useful in creating simple solids. After making the simple solid, you will add a fillet on two 0.75-inch-tall corners. You will also learn to create a continuous fillet along three edges.
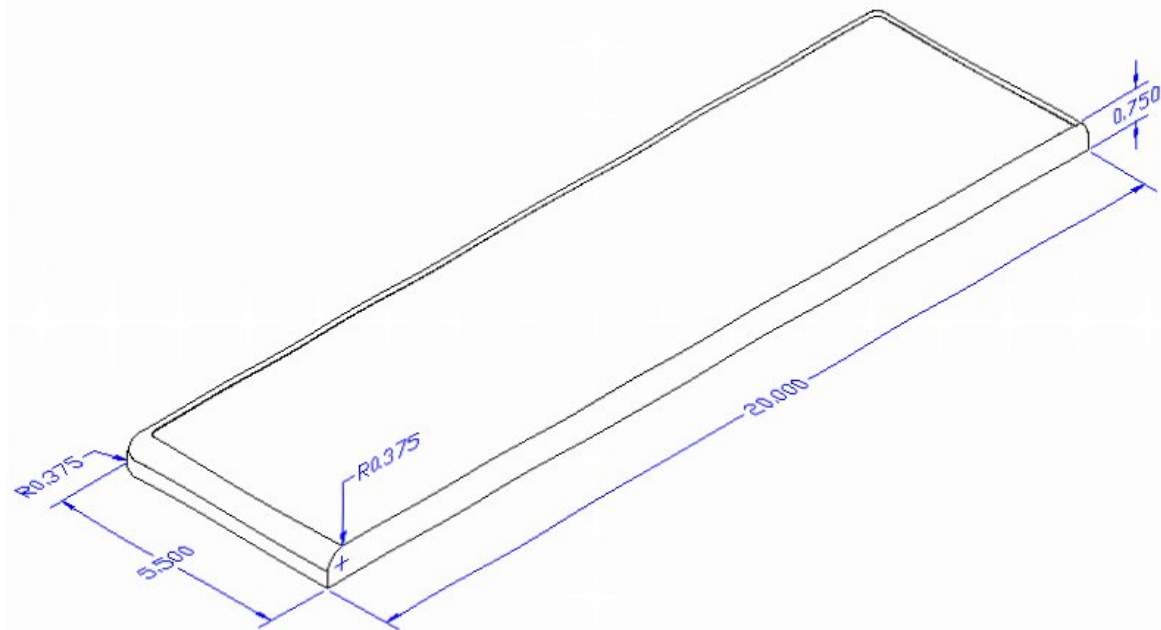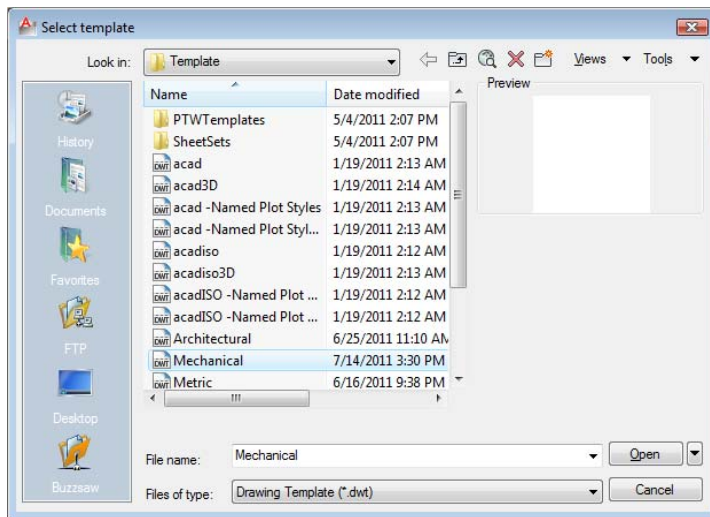
Figure 17.22 – Part Four sketch

In Figure 17.23, the window "Select template" is loaded by selecting the New tool on the Quick Access toolbar. Scroll down the list and until you see the template, Mechanical.dwt and select it.

Figure 17.23 – Starting the Drawing Using the Mechanical Template

To draw the solid rectangular box, select the Box tool on the Modeling panel. At the command line, a prompt asks you to specify the corner of the box or type CE to specify the center of the object. You will need to identify the corner of the box by picking anywhere in the middle of the graphical display. After picking the starting point for the box, the system will ask you to denote a corner; type **L** for length and hit **ENTER**. For part four, type **20.0** for the **X** coordinate, **5.5** for the **Y** coordinate, and **0.75** for the **Z** coordinate. The solid will appear in the graphical display as shown in Figure 17.24.
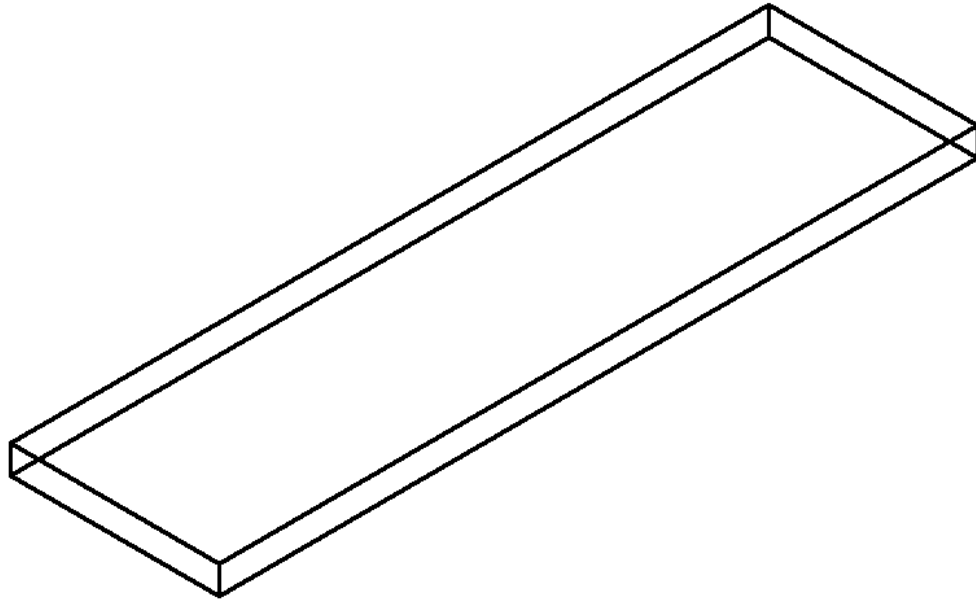
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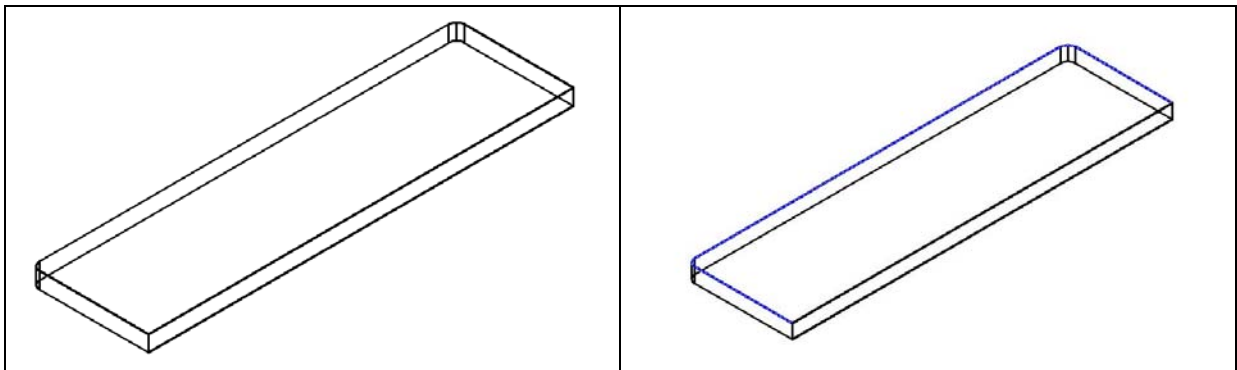**Figure 17.24 – Creating Part Four**

Figure 17.25 – Rounding the 0.75 corner   **Figure 17.26 – Selecting the top edge**

Select the Fillet tool on the Modify panel. Type "**R**" for radius and then **0.375** for the measurement. Now pick the vertical edge of the leftmost corner of the box and hit **ENTER**. At the prompt, "**Select an edge or [Chain/Radius]:**" pick the other vertical edge on the far corner of the part. Hit **ENTER** and the rounded edges will appear as shown in Figure 17.25. To place a continuous rounded edge on the top of three edges, select the Fillet tool again from the Modify panel. Because the radius is the same as the previous fillet, just pick one of the top 5.5" edges and hit **ENTER**. At the prompt, "**Select an edge or [Chain/Radius]:**" type "**C**" for chain and select the same 5.5" edge as shown in blue in Figure 17.26. Hit **ENTER** and the rounded edges will appear as shown in Figure 17.27.
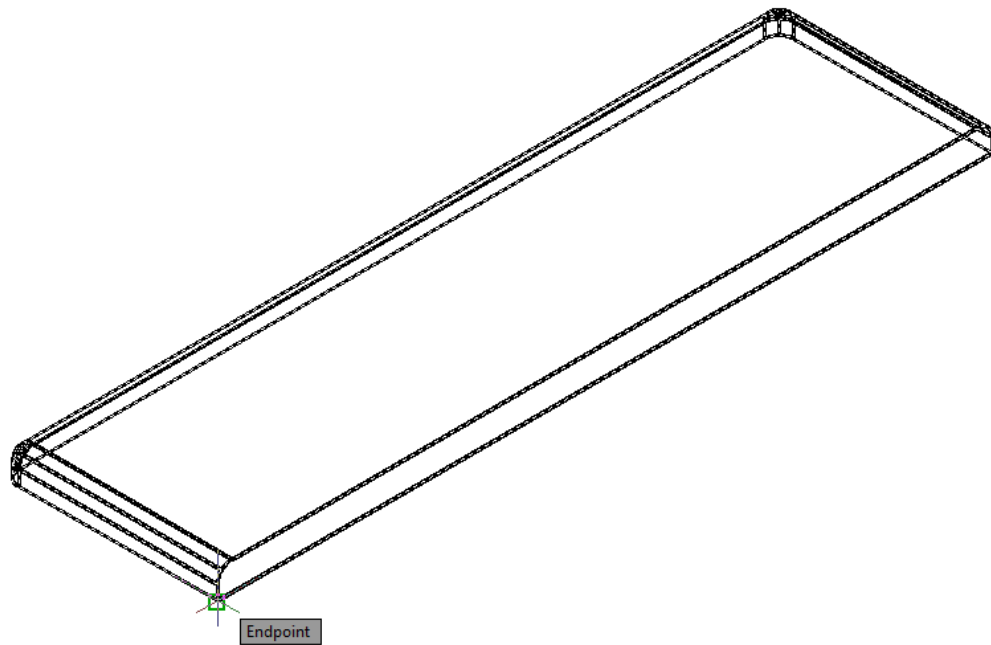
Figure 17.27 – The rounded edges from the chain fillet

Before you save, make sure to move the part to the origin (0,0,0). Use the front bottom corner of the part as shown in Figure 17.27 as the base point.

To save Problem Four as shown in Figure 17.27 in your Foot Rest Assembly folder, select the Save tool on the Quick Access toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type "Foot Rest" and press the Save button to save the drawing (Figure 17.28).
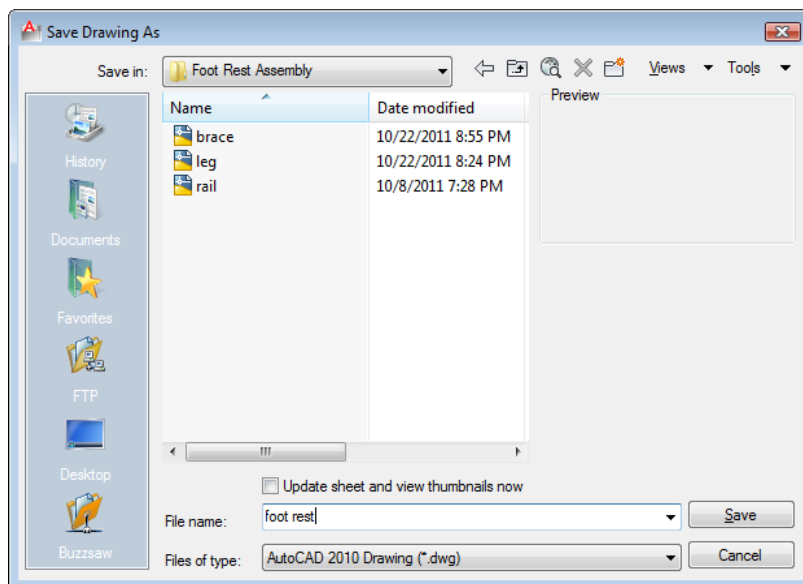

Figure 17.28 – Saving the foot rest file in the foot rest assembly folder

Making an Assembly Drawing

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You will make the Foot Rest Assembly drawing from the four solids parts in your Foot Rest Assembly Folder. Following the sketch as shown in Figure 17.29 and using the Endpoint and the Midpoint Osnaps, you will easily construct your product.
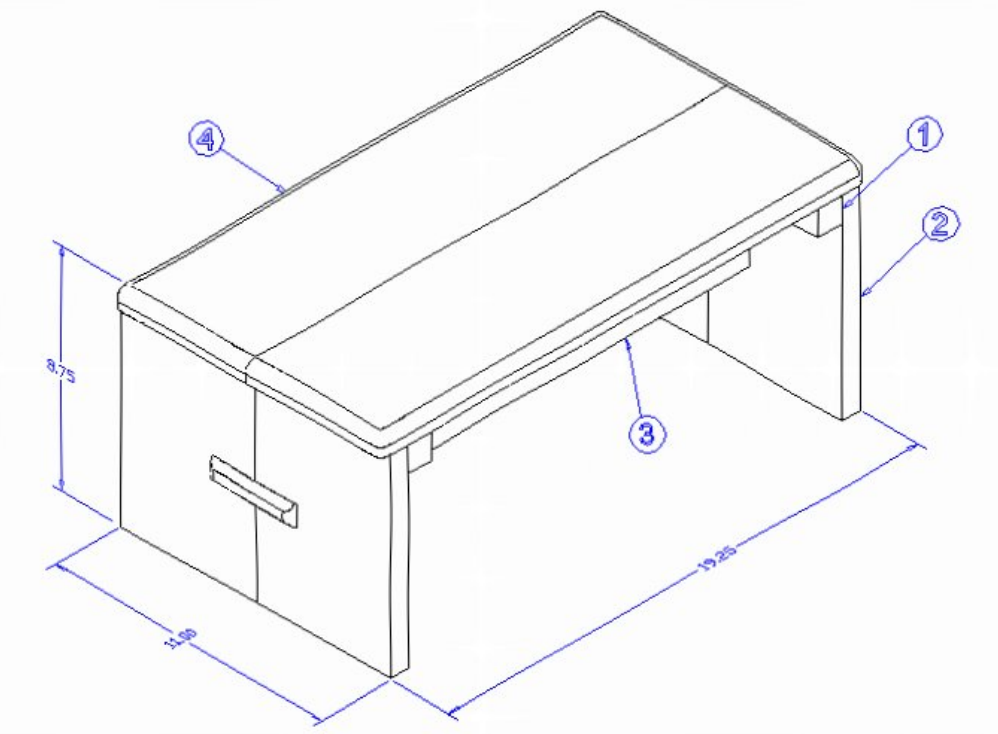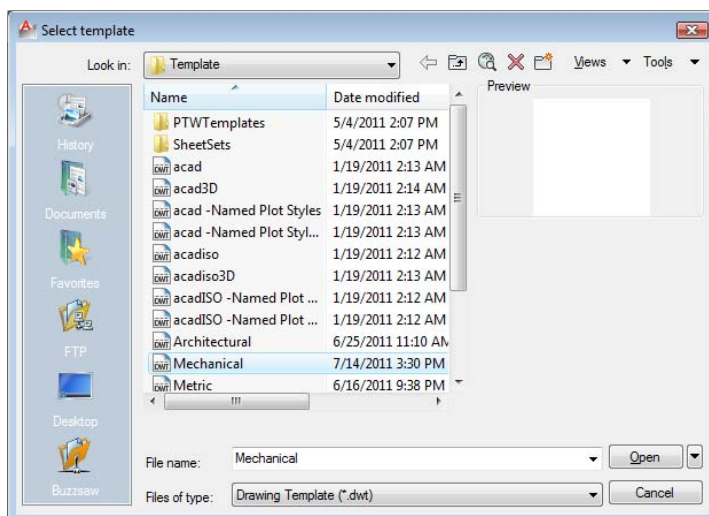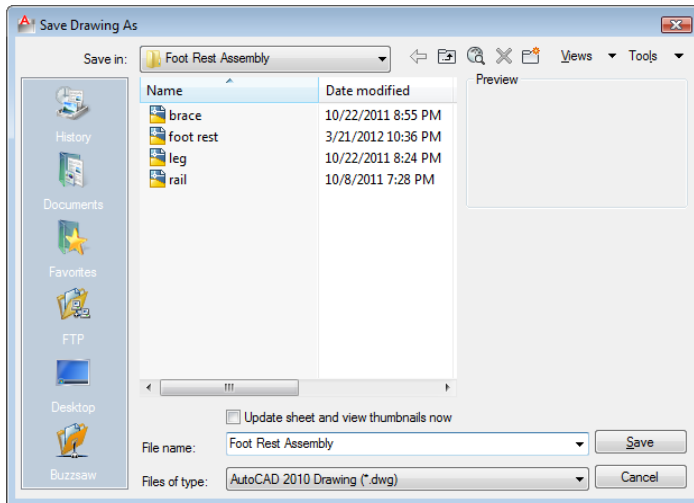

Figure 17.29 – The Foot Rest assembly sketch

In Figure 17.30, the "Select template" window is loaded by selecting the New tool on the Quick Access toolbar. Scroll down the list and until you see the template, Mechanical.dwt and select it.

Figure 17.30 – Starting the drawing using the mechanical template

Figure 17.31 – Saving the Foot Rest assembly file in the foot rest assembly folder

To save the Foot Rest Assembly in your Foot Rest Assembly folder, select the Save tool on the Quick Access toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type "Foot Rest Assembly" and press the Save button to save the drawing (Figure 17.31).

Using the XREF Tool to Import the First Part Drawing

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All of the hard work to make the product has been done by creating each individual solid and saving them into the Foot Rest Assembly folder. Inserting a reference into the solid component is as simple as inserting a block or external file. On the View panel, select the Southwest Isometric View tool to allow you to see the part on the XY plane and the vertical Z-axis will rise from the level surface.

Pick the arrow on the lower right corner of the Reference panel under the Insert tab and the Xref Manager Window will appear in the center of the graphical display. Presently it shows the Foot Rest Assembly drawing, but the Xref Manager as shown in Figure 17.32does not show any attached files. To attach an externally referenced file, click on the Attach button, which will open the Attach External Reference window.
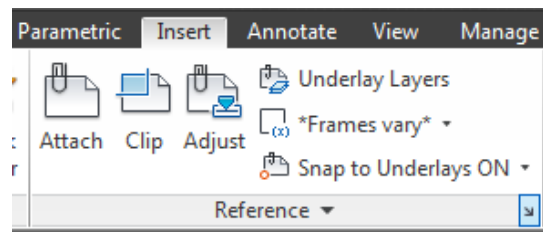

Figure 17.32 – Reference Panel

The first externally referenced drawing file that you will want to insert into the assembly drawing is the Leg, so select the Leg file from the list of drawings in the Foot Rest Assembly folder. Pick the Open button to close the dialog box and the name textbox in the External Reference window will list the Leg file. Select the OK button to insert the solid into the assembly drawing.
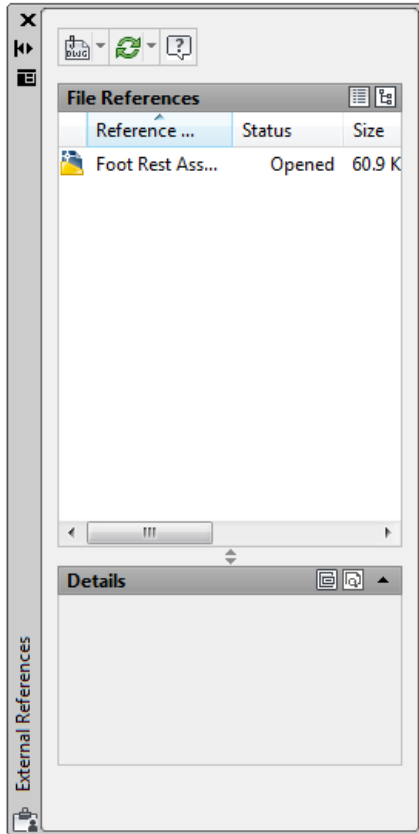
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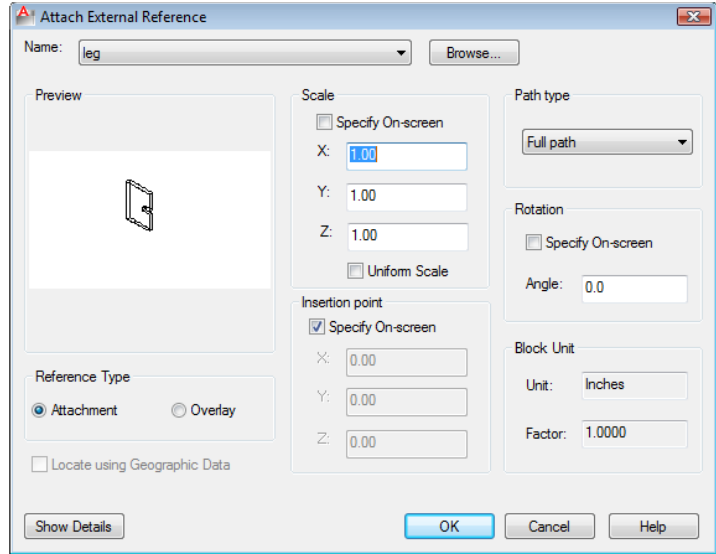**Figure 17.33 – The Xref Manager window**

Figure 17.34 – Selecting the leg in the Attach External Reference window

Next, you will specify the insertion point, so at the Command line prompt "Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]:", you will want to pick a point towards the lower left hand corner of the graphical display. The solid Leg will appear in the graphical display as shown in Figure 17.35.
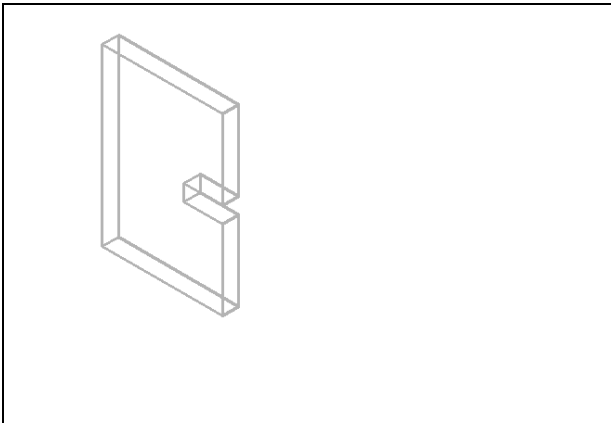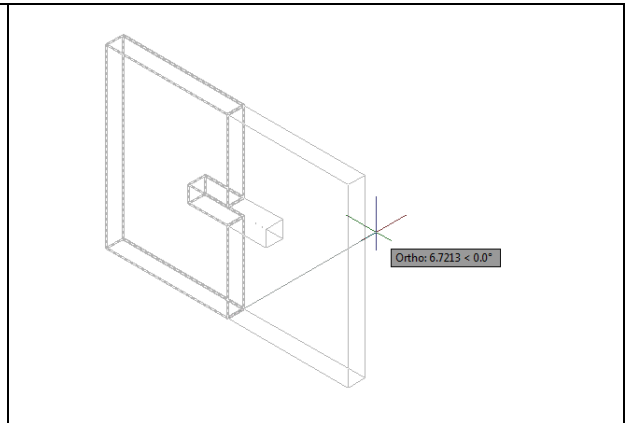

Figure 17.35 – Inserting the Leg

Figure 17.36 – Mirroring the Leg

You will use the Mirror tool on the Modify panel to copy the Leg across the Endpoint itself. At the prompt to select objects, pick the Leg and then hit **ENTER** proceed to the second half of the Mirror command. "Specify first point of mirror line:" by picking the right Endpoint as shown in Figure 17.36 and "Specify second point of mirror line:" by picking to the right of the part with your Ortho "On". The default to delete the source objects is no, so at the prompt "Delete source objects? [Yes/No] <N>:", hit **ENTER** and the mirrored and duplicated entity will appear on the graphical display.

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Using the XREF Tool to Import the Second Part Drawing

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Again, pick the External Reference tool on the Reference panel and the Xref Manager Window will appear in the center of the graphical display. Now the Xref Manager shows the attached Leg file as shown in Figure 17.37. To attach another externally referenced file, click on the Attach button, which will open the Attach External Reference window.

The second externally referenced drawing file that you will want to insert into the assembly drawing is the Brace, so select the Brace file from the list of drawings in the Foot Rest Assembly folder. Pick the Open button to close the dialog box and the name textbox in the External Reference window will list the Brace file as shown in Figure 17.38. Select the OK button to insert the solid into the assembly drawing.

Next, you will specify the insertion point, so at the Command line prompt "Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]:", You will want to pick a point anywhere to the right of the Legs on the graphical display. The Brace will appear in the graphical display as shown in figure 17.39.

To move the Brace, select the Move tool on the Modify panel. When promted to "Select Objects," pick the Brace. The command line will respond with "**1 found**", so hit **ENTER** to go the second part of the command. Next, you need to "specify base point or displacement" on the graphical display. Pick your base point on the lower left Endpoint before the curvature of the end begins. Hit **ENTER** and select the "second point of displacement" as the endpoint on the corresponding corner point of the slot in the Legs as shown in Figure 17.40.
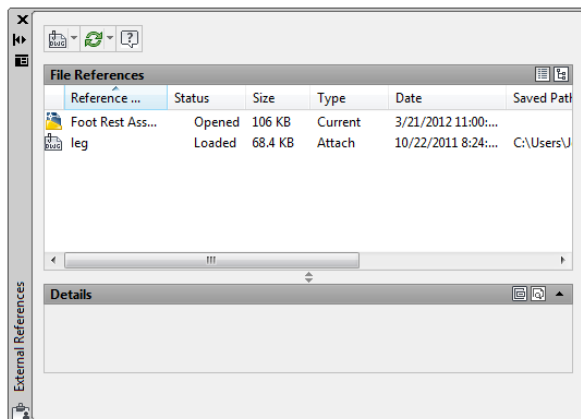

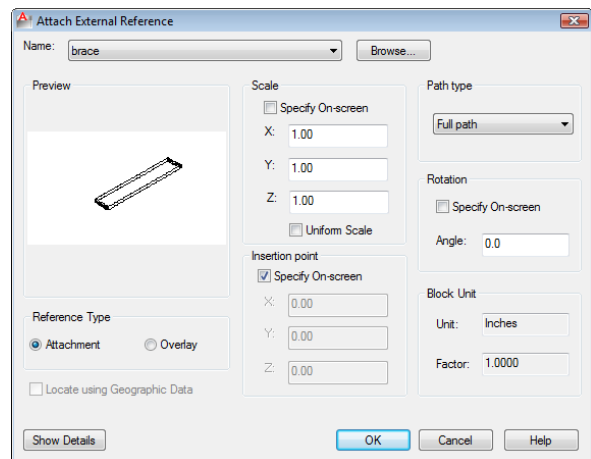**Figure 17.37 – The Xref Manager Window showing the attached Leg File**

Figure 17.38 – Selecting the Brace File in the Attach External Reference window
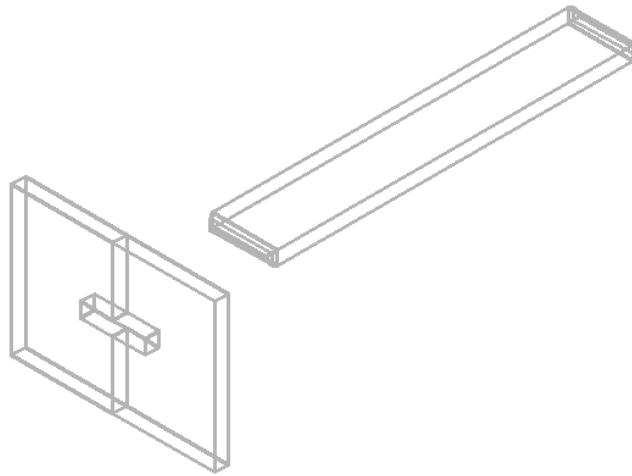
Figure 17.39 – Inserting the Brace in the Foot Rest Assembly

Specify second point or <use first point as displacement>: 12.6852 < 200.3°
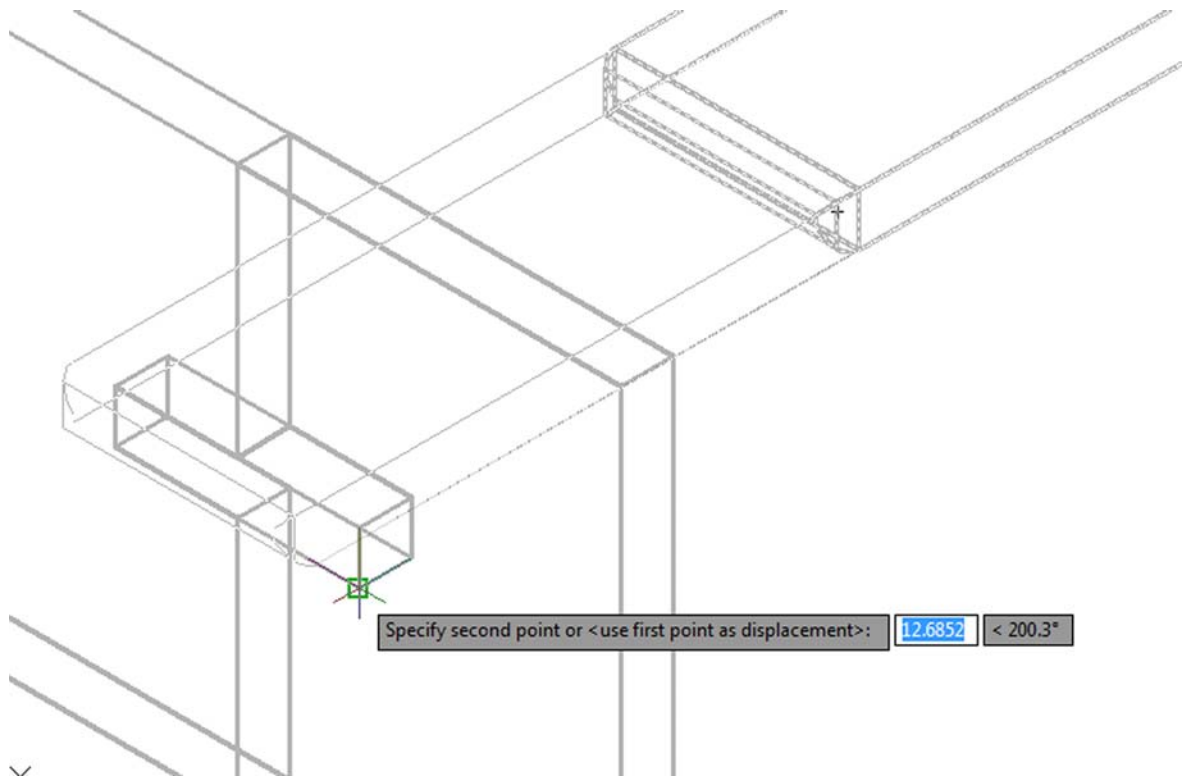
Figure 17.40 – Moving the Brace into position in the slot in the Leg

The Brace will move into the slot with a perfect fit. The curved end will stick out of the rectangular hole and you can check your work by changing to the Front View on the View panel or the Viewcube. If the Brace is in the correct position, return to your previous view, and if the Brace is not located properly, use the Move command to reposition the solid. The use of Object Snaps in the creation of assembly layouts is just as important as in part drawings.
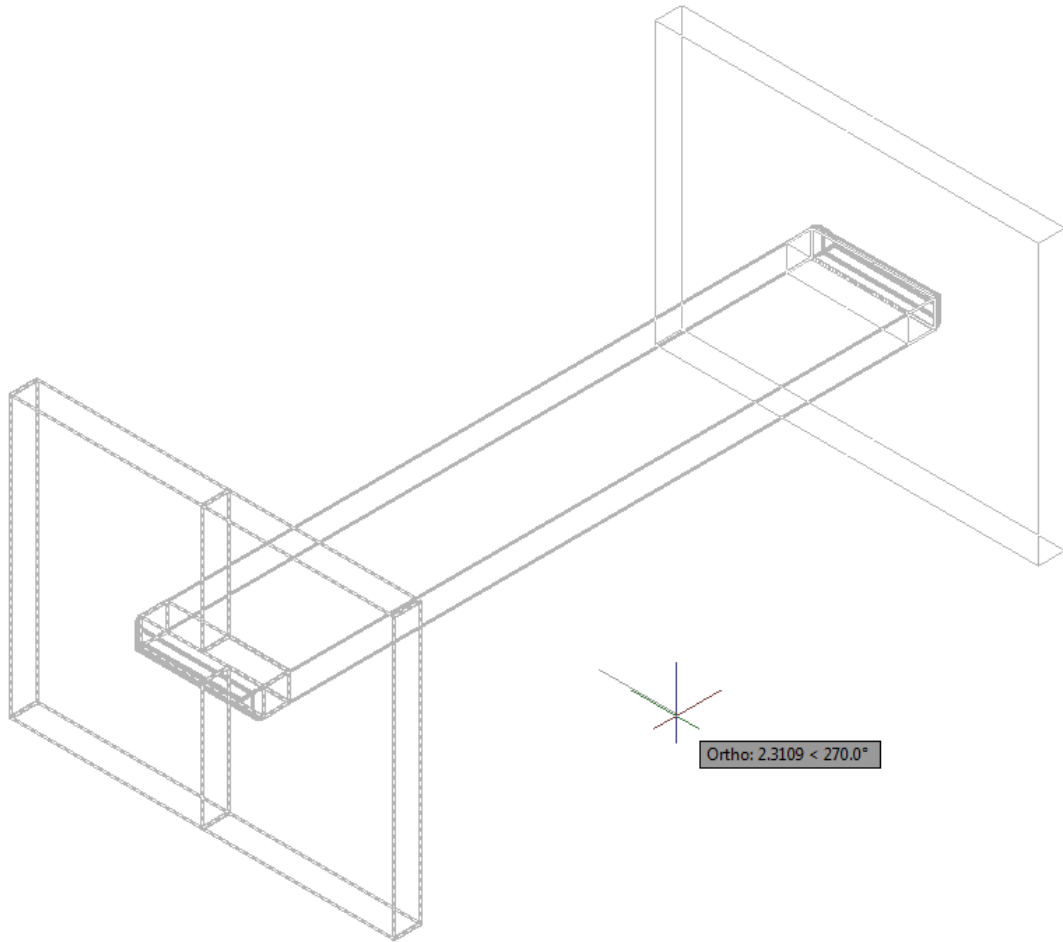
Figure 17.41 – Mirroring the Leg to the opposite side of the Brace

You will use the Mirror tool on the Modify panel to copy the Leg across the midpoint of the Brace. At the prompt to select objects, pick the two Legs and then hit **ENTER** proceed to the second half of the Mirror command. "Specify first point of mirror line:" by picking the Midpoint as shown in Figure 17.41 and "Specify second point of mirror line:" by picking to drawing a line to the left with your Ortho "On". At the prompt "Delete source objects? [Yes/No] <N>:" hit **ENTER** to accept the default "N" and a second set of legs will appear in the proper location on the other side of the brace. .

Using the XREF Tool to Import the Third Part Drawing

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Again, pick the External Reference tool on the Reference panel and the Xref Manager Window will appear. Now the Xref Manager shows the attached Leg and Brace files as shown in Figure 17.42. To attach another externally referenced file, click on the Attach button, which will open the Attach External Reference window.

The third externally referenced drawing file that you will insert into the assembly drawing is the Rail, so select the Rail file from the list of drawings in the Foot Rest Assembly folder. Pick the Open button to close the dialog box and the Rail file will appear in name textbox in the Attach External Reference window as shown in Figure 17.43. Hit the OK button to insert the solid into the assembly drawing.

Next you need to specify the insertion point, so at the Command line prompt "Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]:", pick a point anywhere to the right of the assembly on the graphical display. The Rail will appear in the graphical display as shown in figure 17.44.

To move the Rail, select the Move tool on the Modify panel. Once you choose the Move command, you will be prompted to "Select Objects". Pick the Rail and the command line will respond with "**1 found**", so hit **ENTER** to go the second part of the command. Next, you need to "specify base point or displacement" on the graphical display. Pick your base point on the upper left corner Endpoint of the closest face of the Rail. Hit **ENTER** and select the "second point of displacement" as the Endpoint on the right point of the Leg as shown in Figure 17.45.
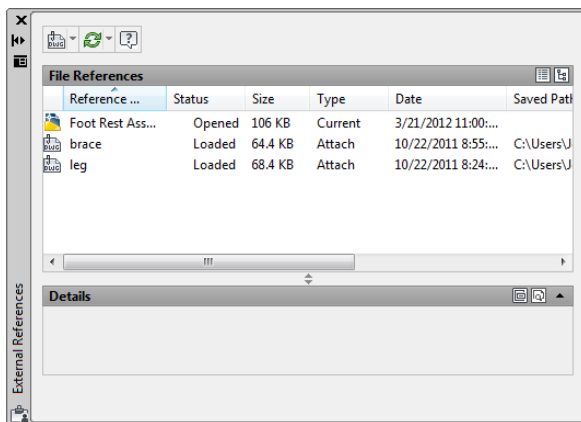

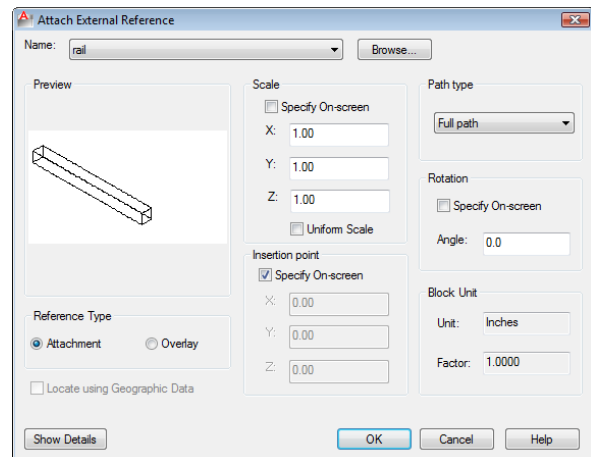**Figure 17.42 – The Xref Manager window showing the Attached Brace and Leg Files**

Figure 17.43 – Selecting the Rail File in the Attach External Reference window

You will use the Mirror tool on the Modify panel to copy the Rail across the midpoint of the Brace. At the prompt to select objects, select the rail and then hit **ENTER** to proceed to the second half of the Mirror command. "Specify first point of mirror line:" by picking the Midpoint and "Specify second point of mirror line:" by picking to the lower right of the part with your Ortho "On" as shown in Figure 17.46. The default to delete the source objects is no, so at the prompt "Delete source objects? [Yes/No] <N>:", just hit **ENTER** and the mirrored and duplicated rail will appear on the opposite side of the brace.
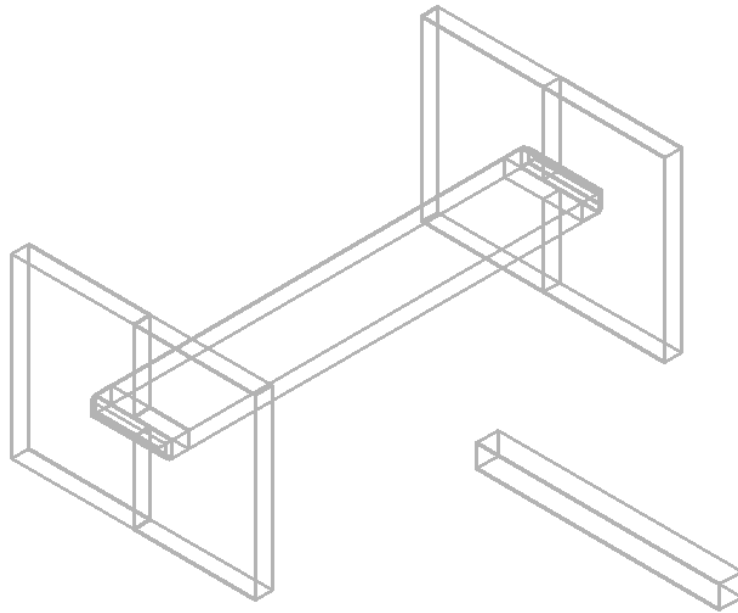
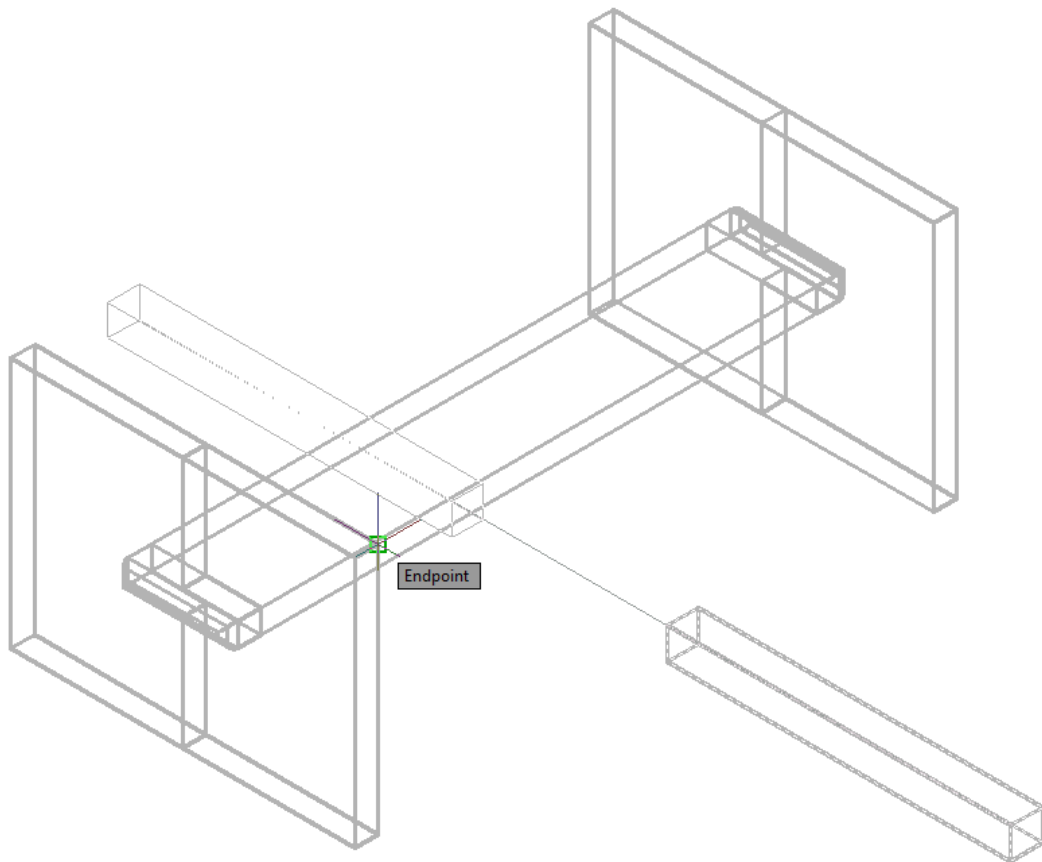**Figure 17.44 – Inserting the Rail in the Foot Rest Assembly drawing**

Endpoint

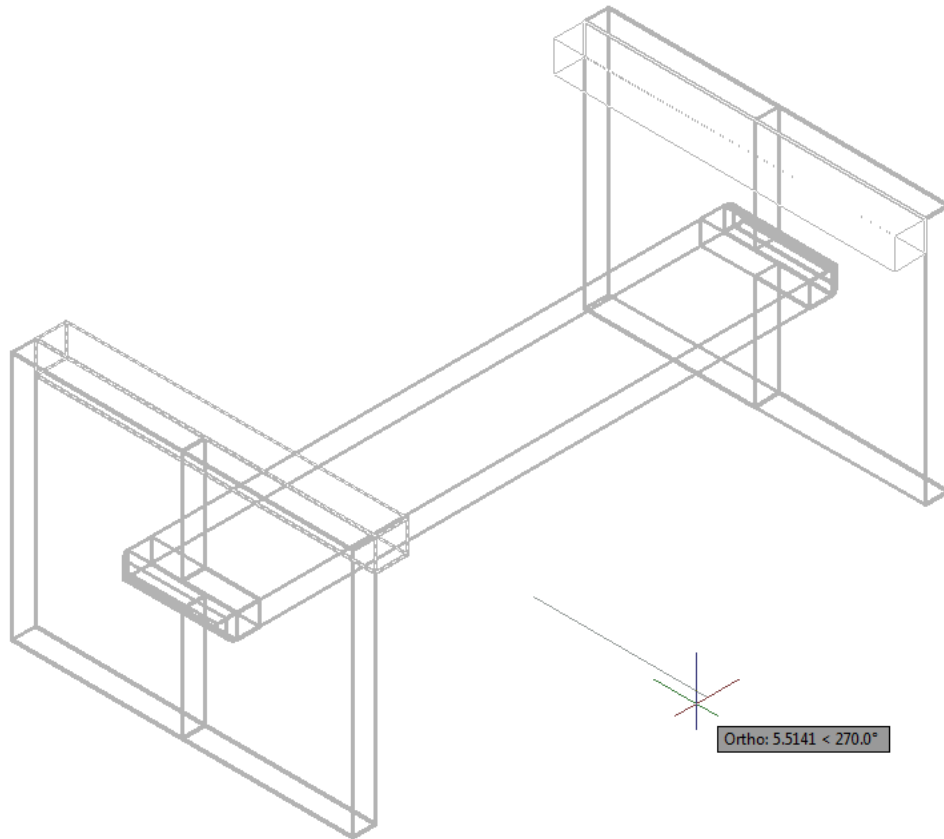**Figure 17.45 – Moving the Rail in the Foot Rest Assembly drawing**

Figure 17.46 – Mirroring the Rail in the Foot Rest Assembly drawing

Using the XREF Tool to Import the Fouth Part Drawing

One last time, pick the External Reference tool on the Reference panel and the Xref Manager Window will appear in the center of the graphical display. Now the Xref Manager shows the attached Leg, Brace and Rail files as shown in Figure 17.47. To attach another externally referenced file, click on the Attach button, which will open the Attach External Reference File window.

The fourth externally referenced drawing file that you will want to insert into the assembly drawing is the Foot Rest, so select the Foot Rest file from the list of drawings in the Foot Rest Assembly folder. Pick the Open button to close the dialog box, and the Foot Rest file will be listed in name textbox in the External Reference window as shown in Figure 17.48. Select the OK button to insert the solid into the assembly drawing.

Next you need to specify the insertion point, so at the Command line prompt "Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PRotate]:", you will want to pick an Endpoint on the outside joint of the Leg as shown in Figure 17.49. The Foot Rest will appear in the graphical display.
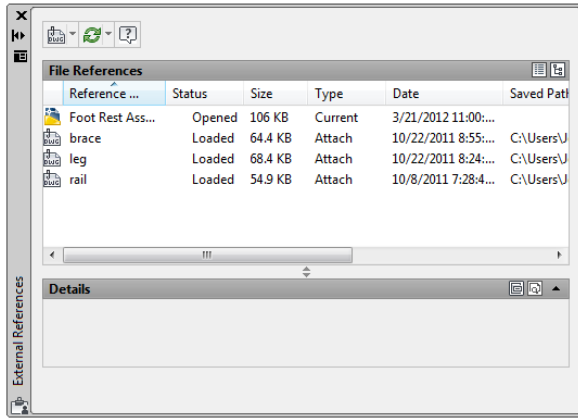
Figure 17.47 – The Xref Manager window showing the Attached Files
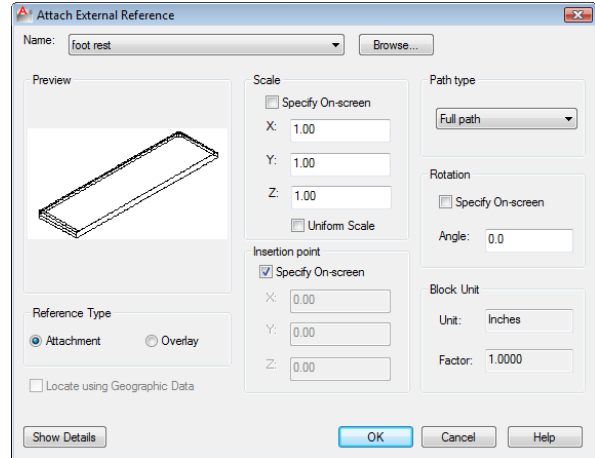

Figure 17.48 – Selecting the Foot Rest in the External Reference Window

To move the Foot Rest, select the Move tool on the Modify panel. Once you choose the Move command, the prompt will be to "Select Objects". Pick the Foot Rest and the command line will respond with "**1 found**", so hit **ENTER** to go the second part of the command. Next, you need to "specify base point or displacement" on the graphical display. Pick your base point anywhere of the graphical display. Hit **ENTER** and for the "second point of displacement", move your mouse to the left and type **0.375** and **ENTER.**
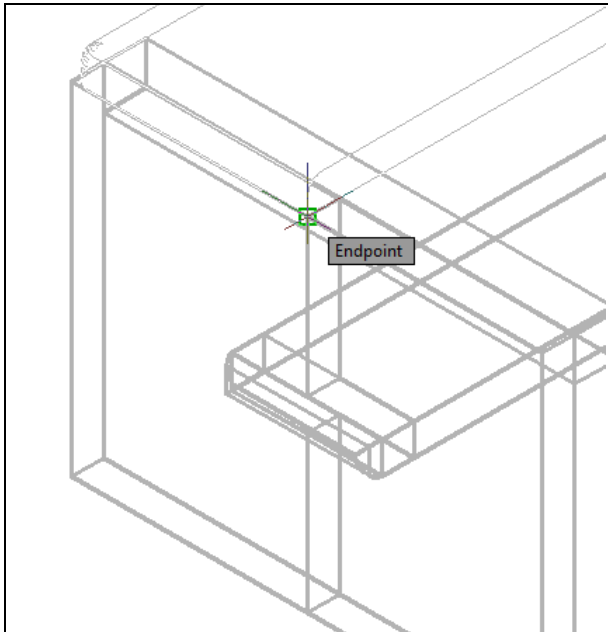

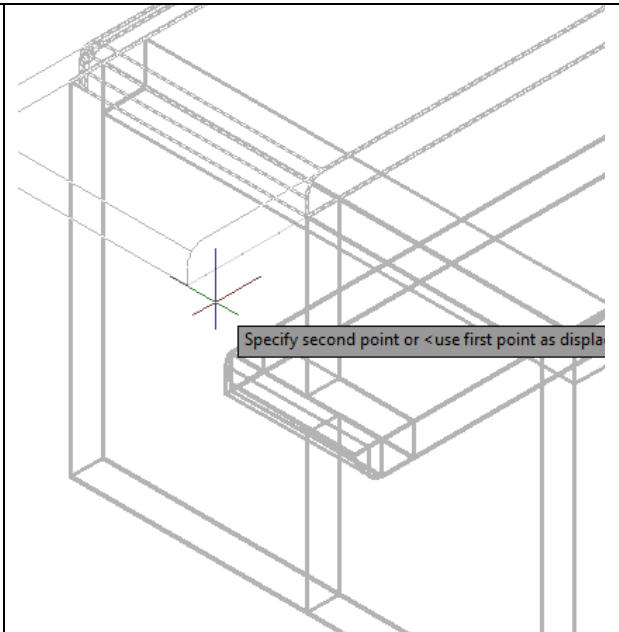**Figure 17.49 – Inserting the Foot Rest**

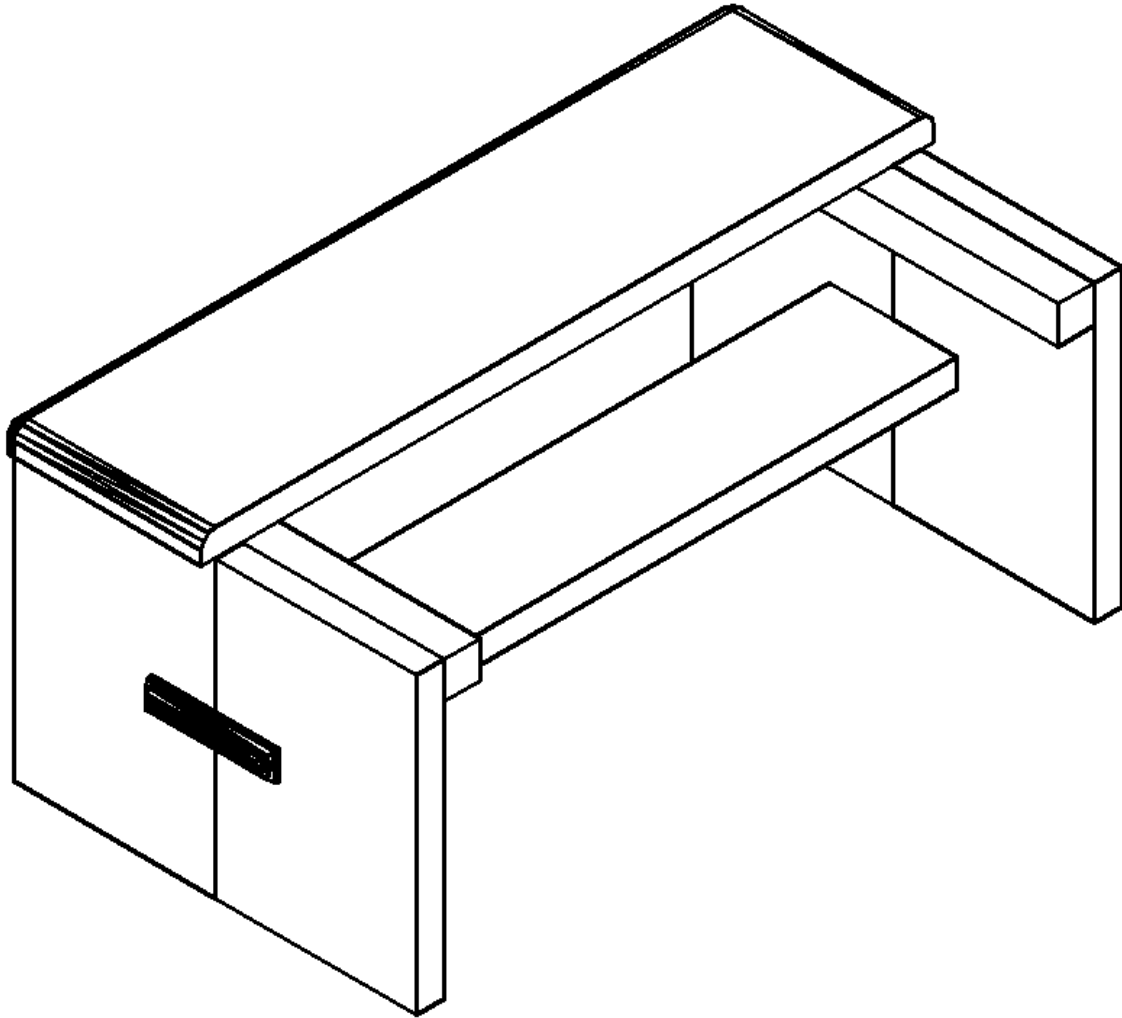**Figure 17.50 – Moving the Foot Rest**

17-26

Figure 17.51 – Using the Hide command to view the assembly

You will use the Mirror tool on the Modify panel to copy the Foot Rest across the endpoint of Leg joint. At the prompt to select objects, pick the Foot Rest and then hit **ENTER** proceed to the second half of the Mirror command. "Specify first point of mirror line:" by picking the Endpoint as shown in Figure 17.52 and "Specify second point of mirror line:" by picking to the lower left of the part with your Ortho "On". The default to delete the source objects is no, so at the prompt "Delete source objects? [Yes/No] <N>:", hit **ENTER** and a second footrest will appear as the final object of your Footrest Assembly drawing.
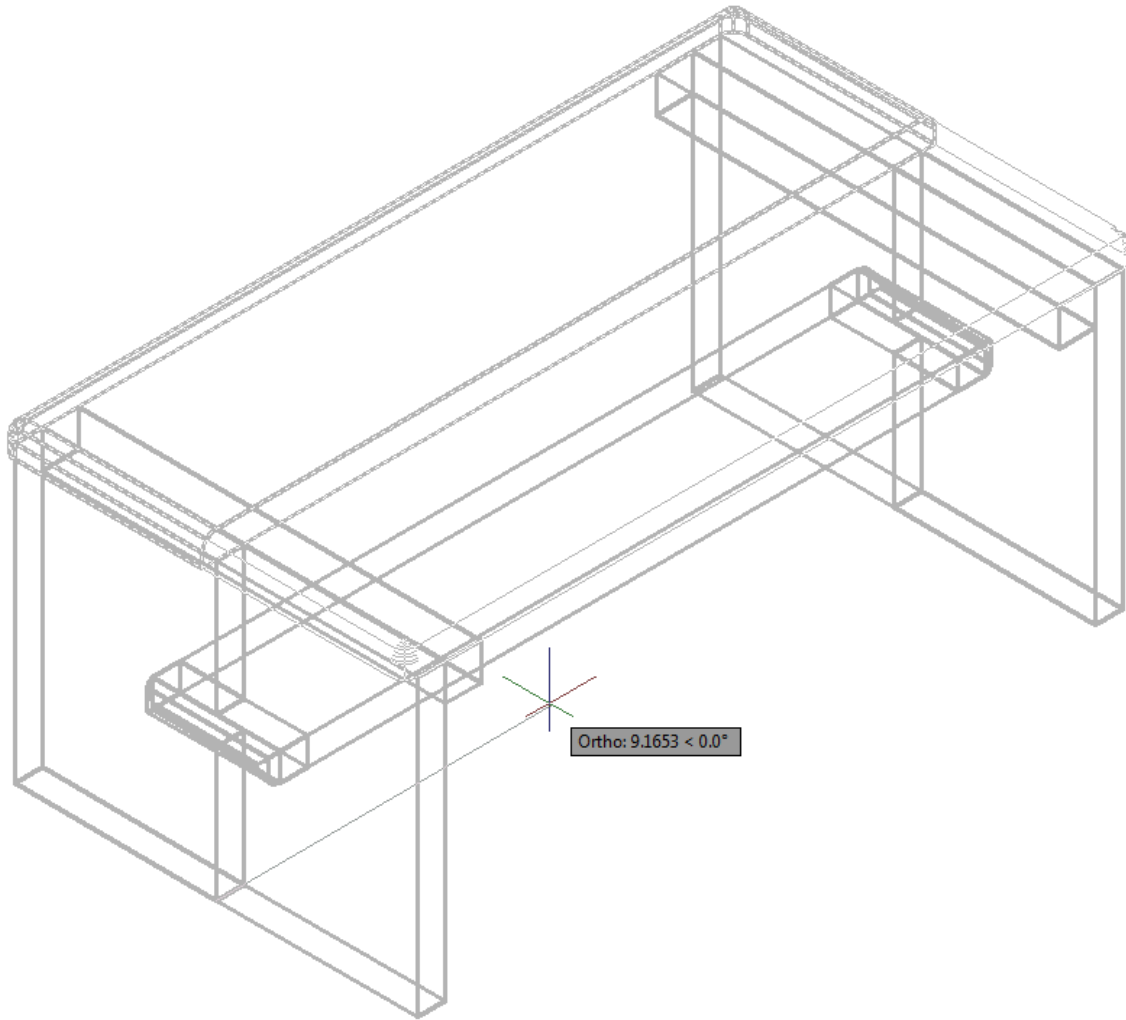
Ortho: 9.1653 < 0.0°

Figure 17.52 – Mirroring the Foot Rest in the Foot Rest Assembly drawing

To save the Foot Rest Assembly in your Foot Rest Assembly folder, select the Save tool on the Quick Access toolbar. The current and assembled version is now updated in the Foot Rest Assembly file. Your drawing should appear as shown in Figure 17.53.
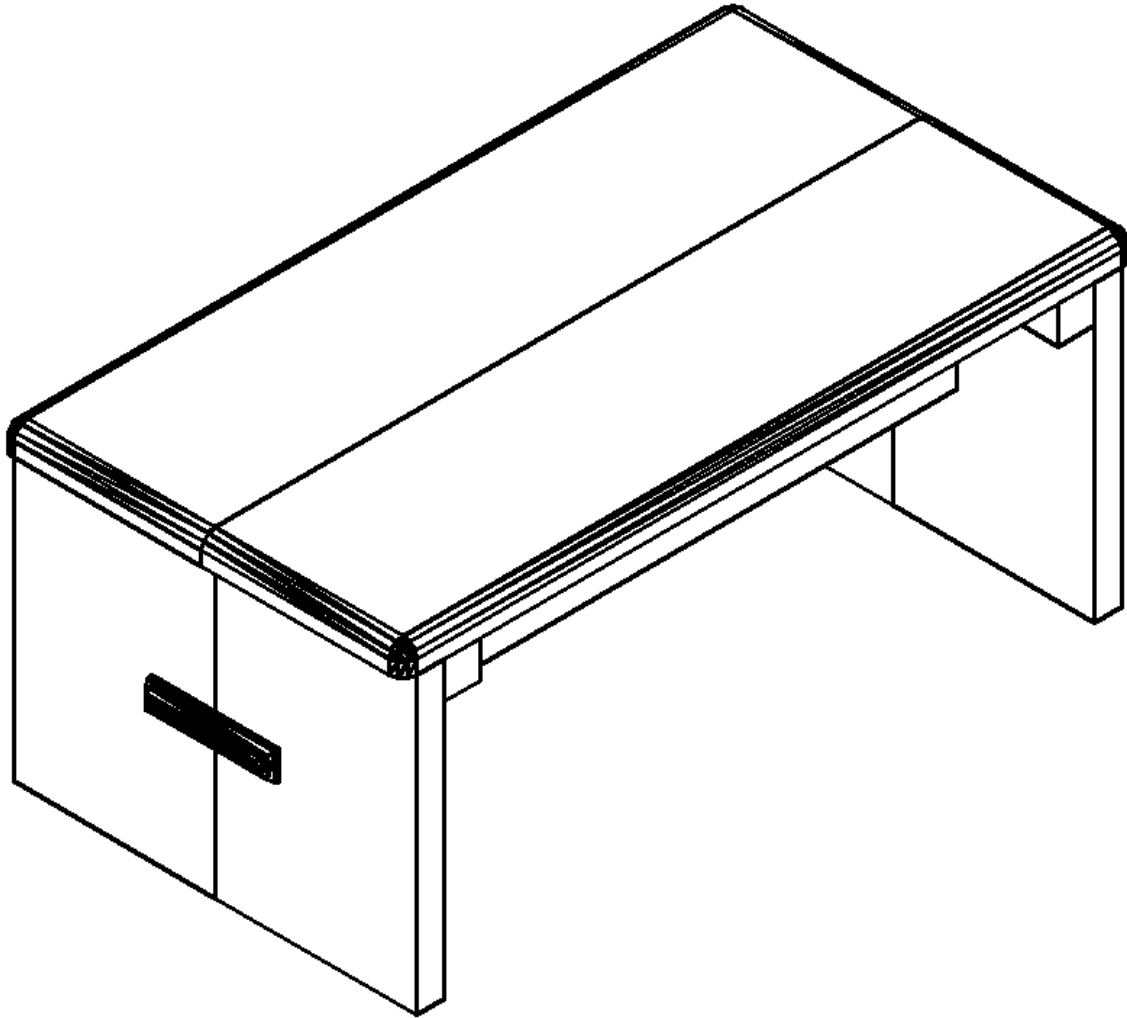
Figure 17.53 – The Finished Foot Rest Assembly Drawing

* World Class CAD Challenge 03-27* - Close this drawing file. Create four solid drawings in an assembly folder, and make an assembly file from the four externally referenced files. Complete the task in less than 60 minutes. Continue this drill four times, each time completing the drawing in less than 60 minutes to maintain your World Class ranking.

* World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.