Design 0100, Weekly Design Guide
Week 8, The Chair

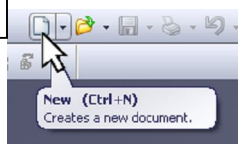
This week we will build a chair complete with legs and a back. This exercise will teach the student how to build a model based off of a "Layout Sketch". A *Layout Sketch* allows a designer to sketch out a design on a plane (this one will be on the Top Plane) using this sketch throughout the model. All of the usual sketch elements and restraints and definitions will be based in the single sketch.

Perform the following steps:

Open a new document:

1. Choose "New" then press "OK"
2. Select "Part" from the "Templates" tab. Save this part (even before you do any work) to your thumb drive in the Week 7 folder and name it "Chair".
3. To start choose the "Sketch" tab from the "Command Manager"

1



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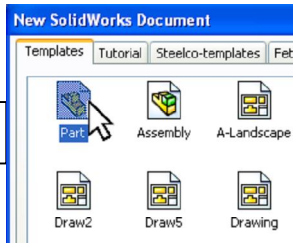

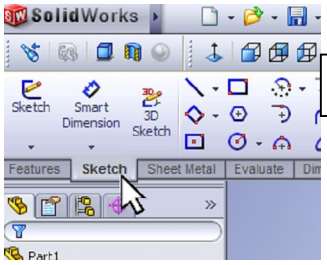

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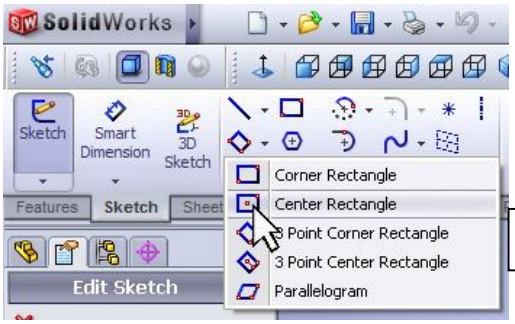
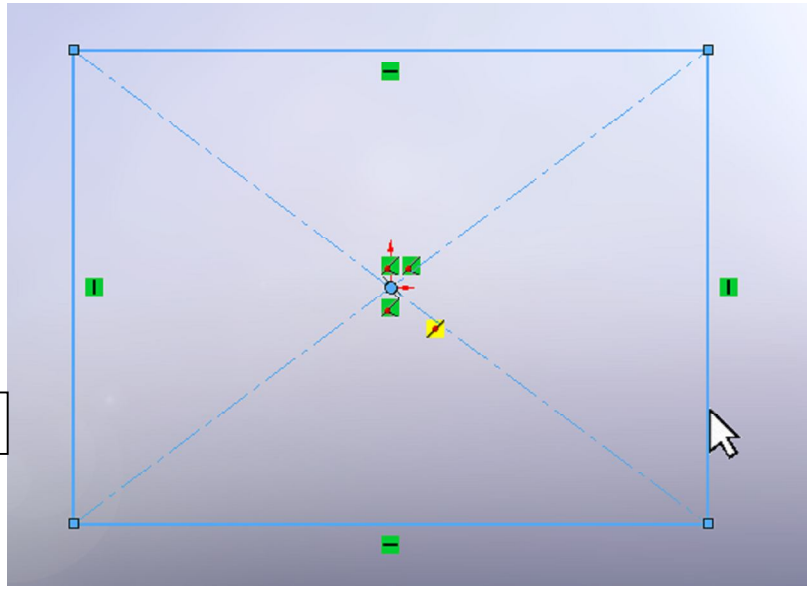


4

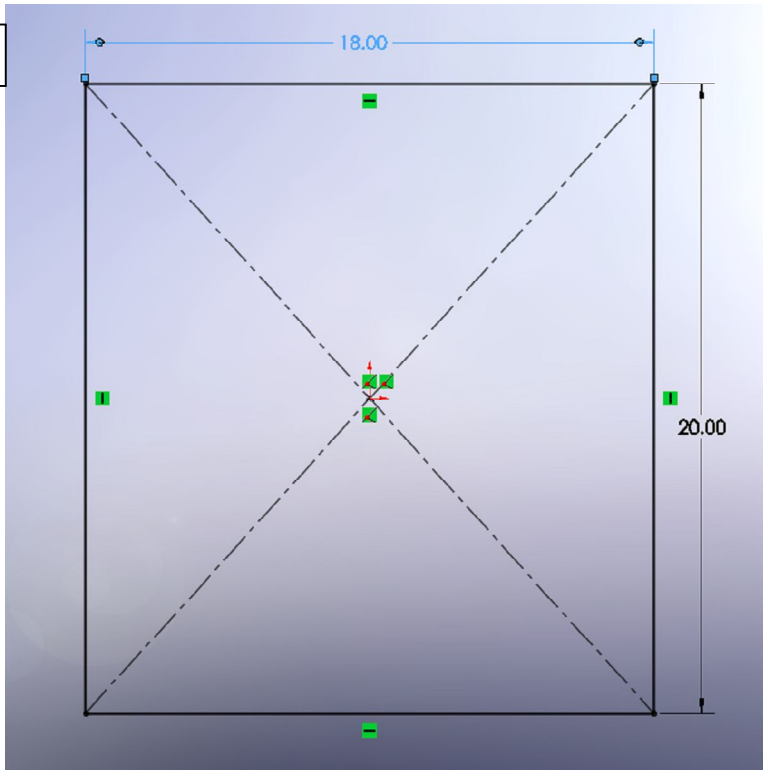
4. Click on the top plane first then choose the "Center Rectangle" sketch element on the "Command Manager". Notice that the Top Plane becomes perpendicular to the screen. This will allow for easier sketching.
5. Place the center at the origin and stretch it to a reasonable size. Notice the sketch relations: coincident, midpoint, horizontal and vertical relations. Now add "Smart Dimensions". Before we do any modeling lets define where the legs will go.

5



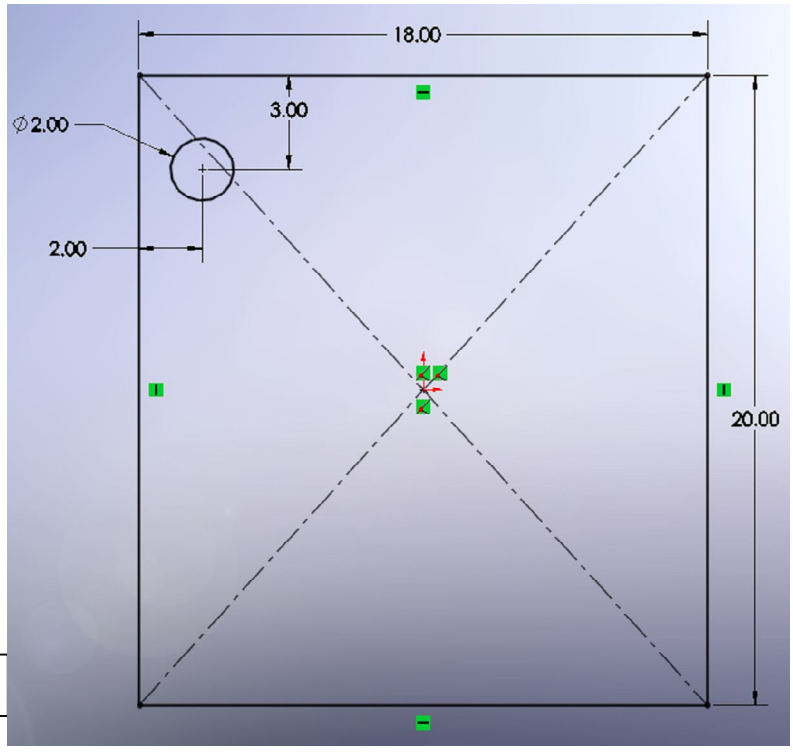
6. Add a horizontal “Smart Dimension” on the top or bottom line of 18 for 18 inches (this will be the chair seat width). Add a vertical dimension of 20 for 20 inches (this will be the seat length). Notice that the lines of the rectangle are now black meaning that it is fully restrained or defined.

6



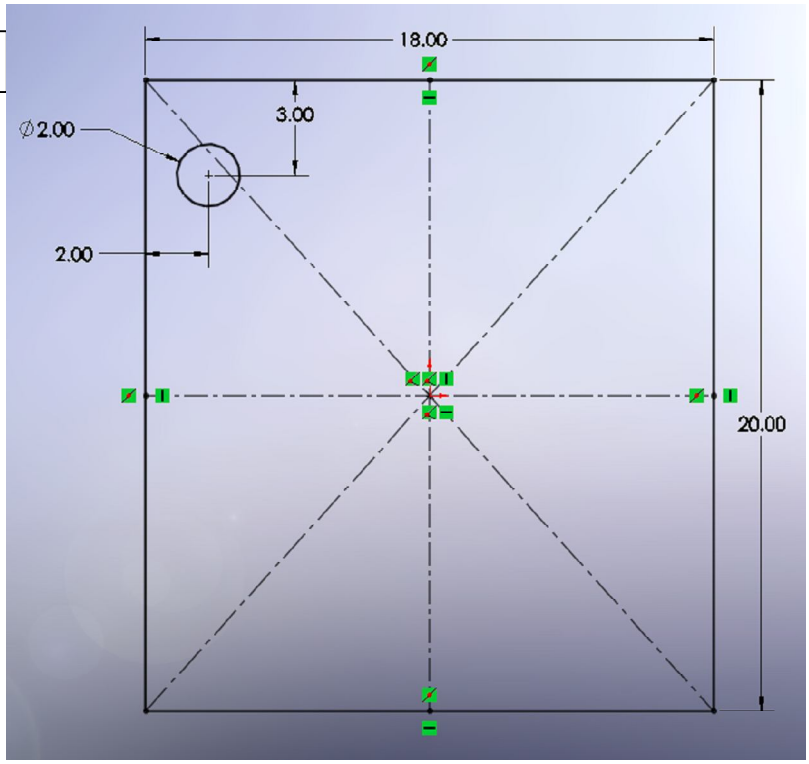
7. Place a circle in the approximate location shown. Add a “Smart Dimension” of 2 for a 2 inch diameter leg. Then place a dimension from the edge of the circle to the top line of 3 inches and 2 inches from the side as shown. Our next step is to mirror the circle to the right side of the seat then both of the top circles to the bottom of the chair. To do this we need to place two centered Centerlines both horizontally and vertically.

7



8. Draw Centerlines from the midpoint of the back (top) line to the midpoint of the front (bottom) line and also from the midpoint of the left line to the midpoint of the right line. Make certain the Sketch Relations for the Centerlines are established with the symbols shown.
9. Select "Mirror Entities" from the "Sketch" tab on the "Command Manager".
10. Select the circle for the "Entities to mirror:" and the vertical centerline for the "Mirror about:" option. Click the green checkmark.

8



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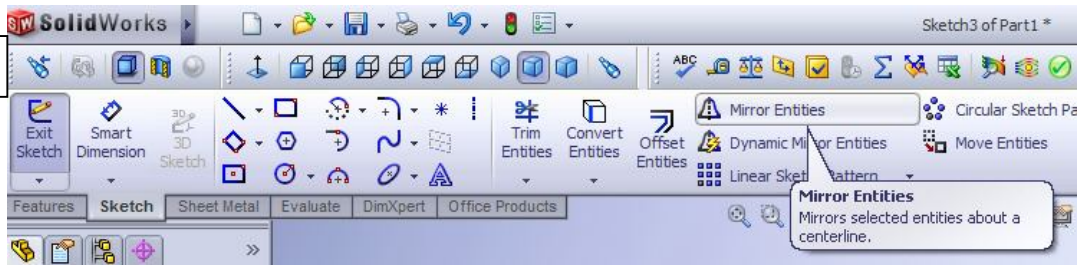
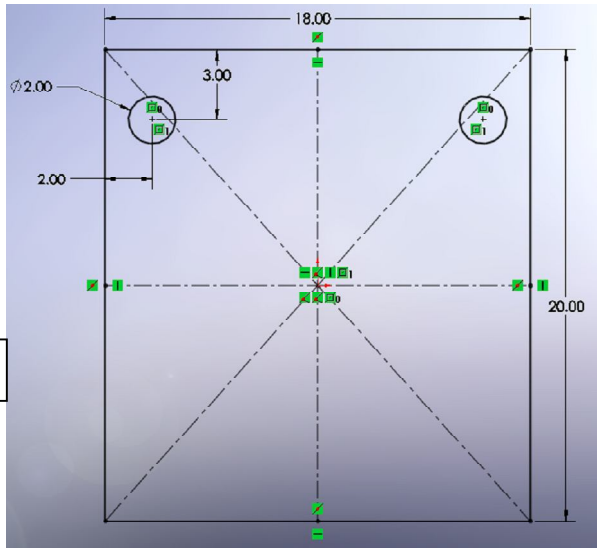


11. The result of this move should resemble the image to the right.

11



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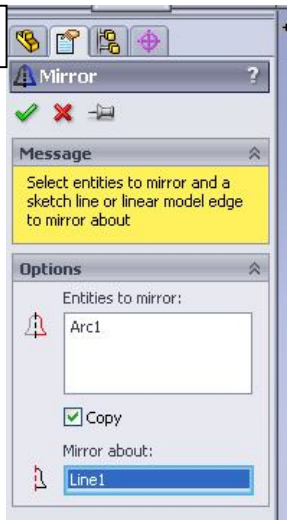
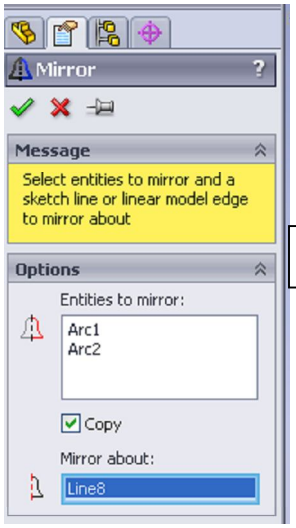


12. Select the 2 circles in the back (top of the image) for the "Entities to mirror:" and the horizontal centerline for the "Mirror about:" option. Click the green checkmark.
13. The result of this move should resemble the image. Now we have the outline of the seat (which can be modified for a better look later) and four legs. The next step we will refer to this sketch to model the seat and legs. Later we will model the back of the chair using the same sketch.
14. Press the "Exit Sketch" button or the "Rebuild" button.
15. Select a material of your choosing. I will choose "pine" for this demonstration.
16. Now we will make the seat of the chair. The edges of the seat will be defined by the sketch we just drew. Choose the "Top Plane" in the Feature Manager Design Tree. Choose the "Sketch" button in the Sketch portion of the Command Manager.
17. Go back to the *Layout Sketch still* visible in the background colored gray and select an edge of the chair.



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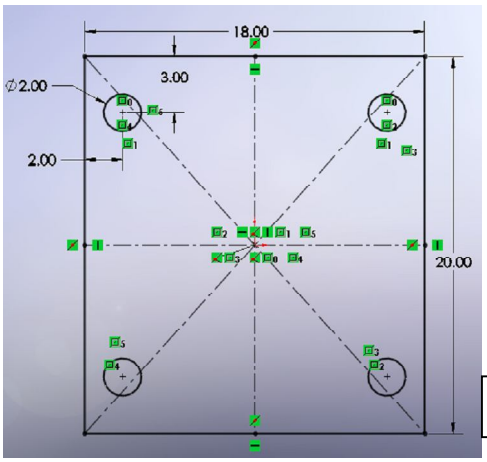
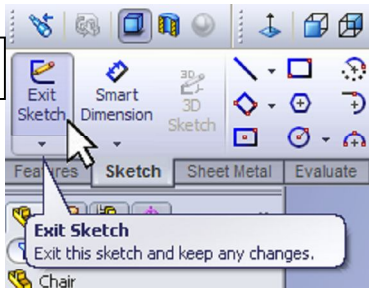


13



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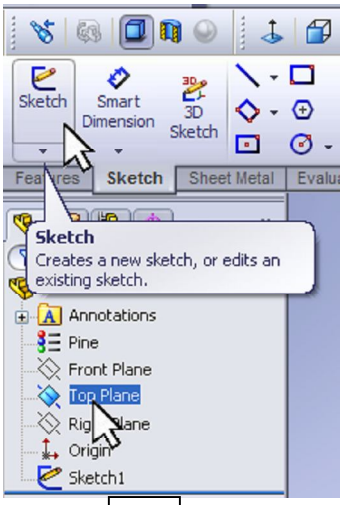
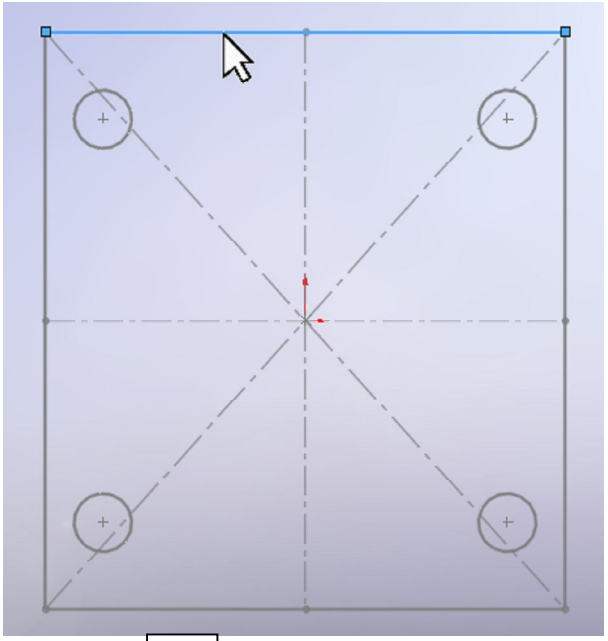
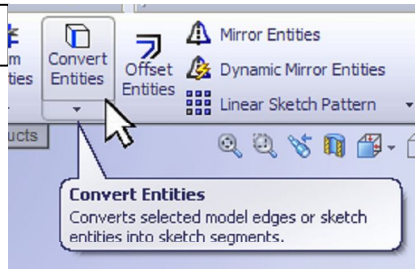


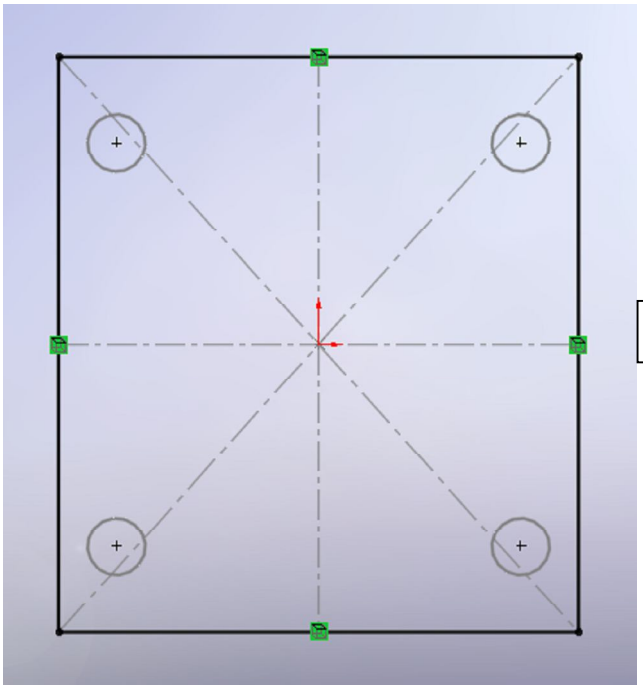
16

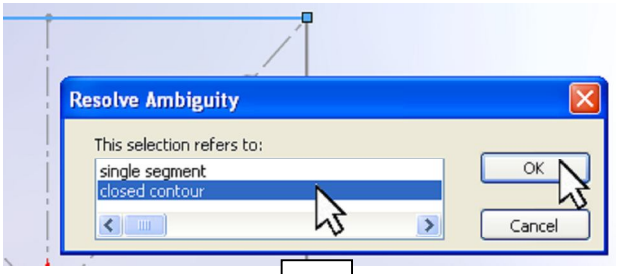

17

18. Choose “Convert Entities”.
19. Select the “closed contour” option in the dialog box that opens up. The closed contour will take the line that you selected and follow the connected lines back to the original line selected thus closing the sketch.
20. The former procedure should produce something similar to the image below.

18



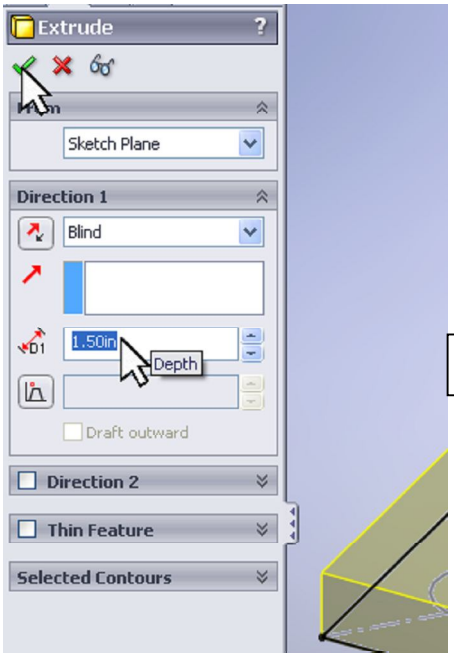




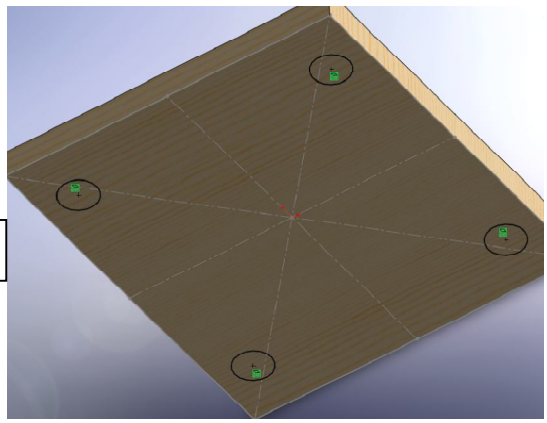
19

20

21. Go to the Command Manager and select “Extrude Boss/Base”, the condition should be “Blind”, orient the extrusion so that it extrudes up if necessary, enter 1.5 for 1 ½ inches and click the green checkmark. Add fillets as desired.
22. Now we will add the legs. Rotate the chair seat so that you will be sketching on the bottom. Select the bottom surface of the chair seat to sketch on. Click on “Sketch” in the Command Manager then click on each of the 4 circles (in the *Layout Sketch*) with the control “Ctrl” key depressed then choose “Convert Entities”. This will give you 4 circles to make the chair legs out of.

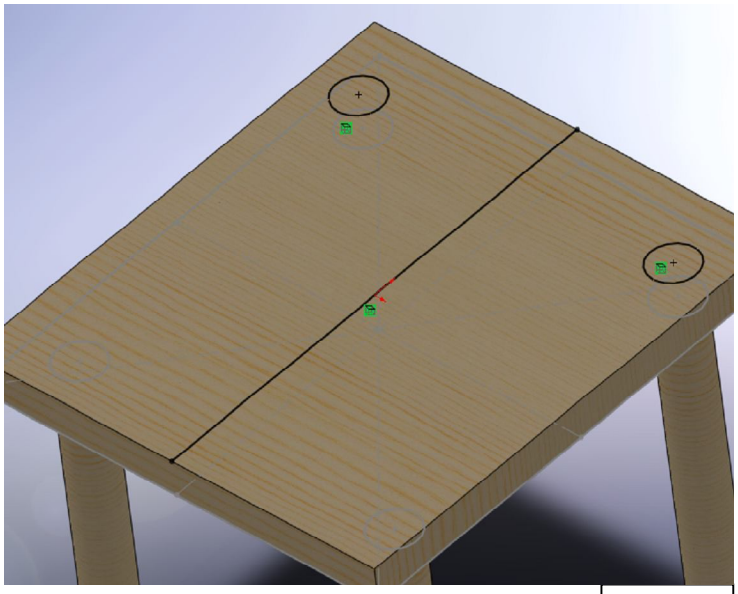


21



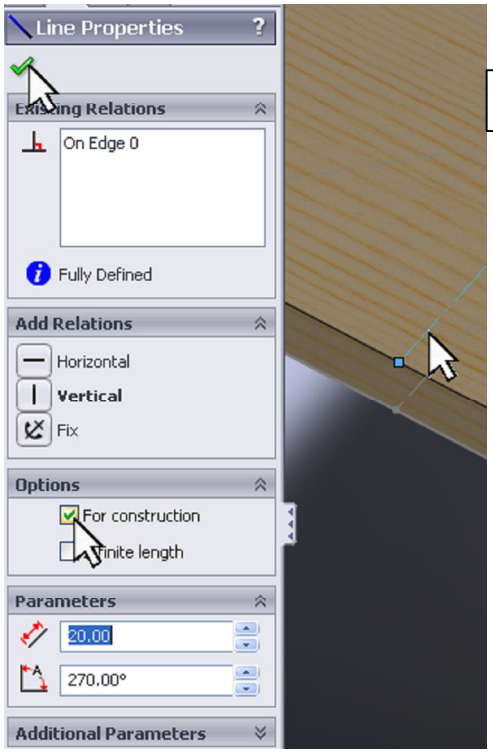
22

23. Go to the Command Manager and select “Extrude Boss/Base”, the condition should be “Blind”, orient the extrusion so that it extrudes down if necessary, enter 17 for 17 inches and click the green checkmark.
24. Now we will model the back of the chair. Select the top of the chair seat as a surface to sketch on. Go the “Sketch” button in the Command Manager. Select the two back circles from our *Layout Sketch* and select “Convert Entities”. Select the center line of our and again select “Convert Entities”.



24-23

25. Change the line selected above to a centerline by selecting the line and choosing “for construction” in the Properties Manager as shown.

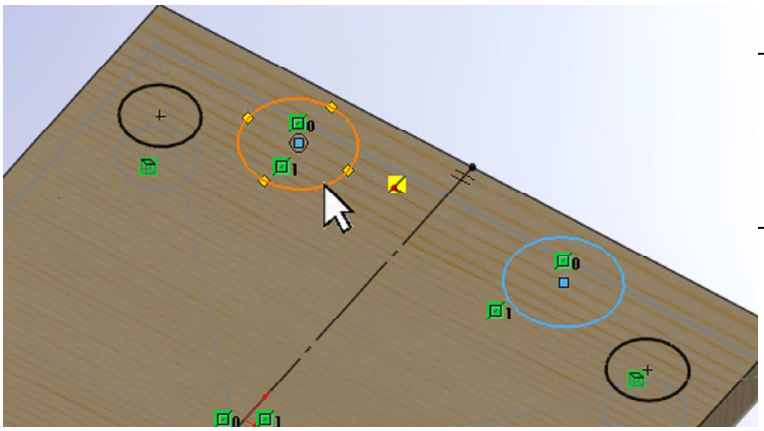


25

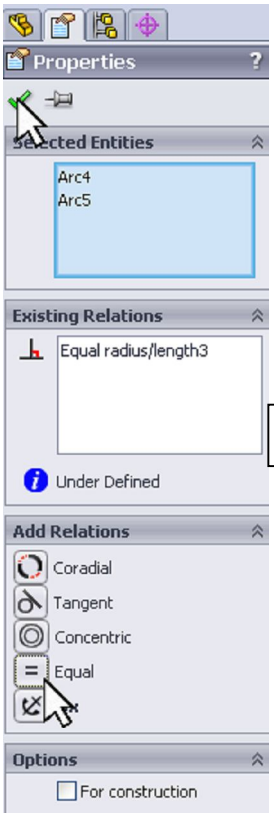
26. Select the centerline then go to the Sketch portion of the Command Manager and select “Dynamic Mirror Entities”. This is a fun feature because it is a very productive feature. The Dynamic Mirror will allow for the creation of mirrored objects on either side of the line these objects are drawn on. For example, we need to draw two additional circles in addition to the ones we have just converted. When we draw one circle on one side it will also appear mirrored on the other side of the centerline.
27. Draw one circle in the approximate location shown. The same circle (same size and distance from the centerline) will appear on the other side of the centerline.

26

28. Now make all of the circles equal to each other by performing the following procedure. Click on one of the recently sketched blue circles and with the control "Ctrl" key depressed click on one of the fully defined black colored circles. Go to the Properties Manager and select the "Equal" relation as shown. Now all four circles will be equal and symmetrical. The recently drawn circles are still blue because they have not been fully defined yet. This is the next step.

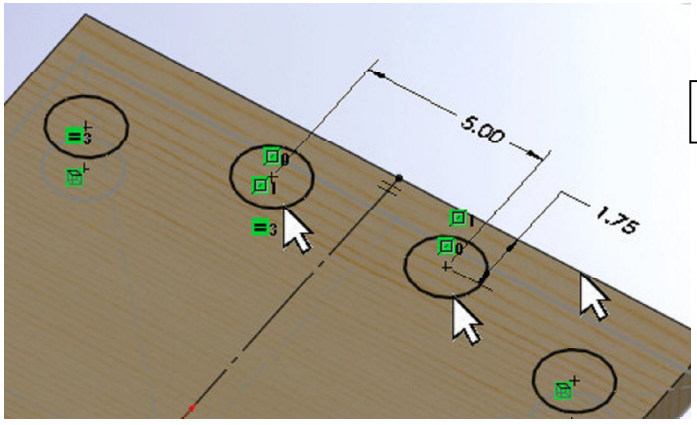


27 - 28



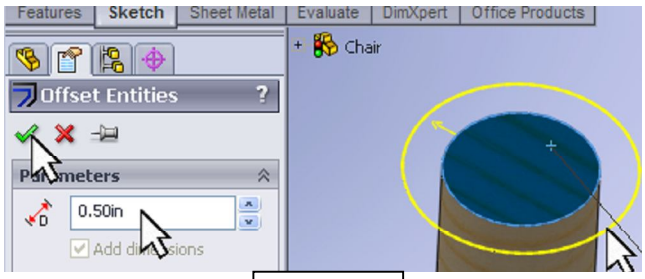
28

29. Establish a distance for the two recently drawn circles from each other. Go the Sketch portion of the Command Manager and select "Smart Dimension". Put a dimension between the two circles by clicking the sides not the center of the circles. Type in 5 for 5 inches. Now dimension between one of the recently drawing circles and the edge of the chair. Type in 1.75 for 1 ¾ inches.



29

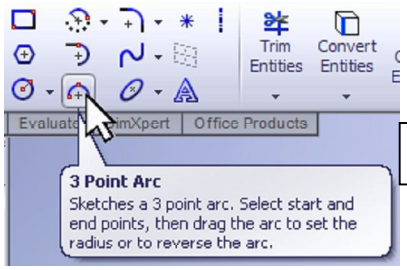
30. Go the Command Manager and extrude these circles to a distance of 12 inches.
31. As a last step we will create the top of the chair back. For a sketch surface, click on the top of any of the cylinders making up the bottom of the chair back. Click on "Sketch" in the Command Manager, click on the top edge of one of the outside cylinders then click on "Offset Entities" in the Command Manager,



30-31

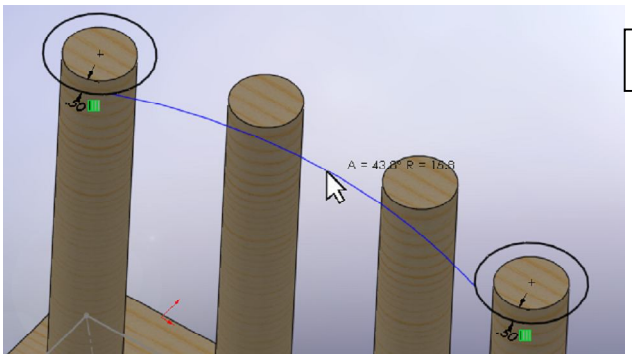
type in 0.5 for a half inch. Click the green checkmark. "Offset Entities" works similar to "Convert Entities". Do this same procedure to the other outside cylinder.

32. Go to the Sketch portion of the Command Manager and choose "3 Point Arc".

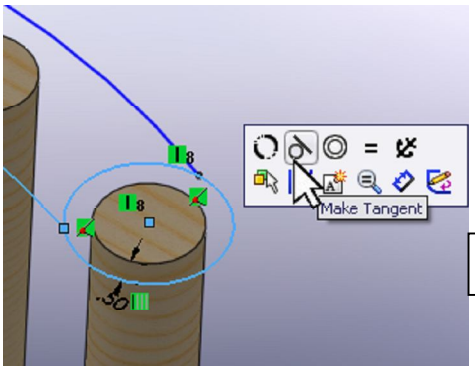


32

33. For the first point of the arc touch the first circle. The second point of the arc touch the second circle with the Third point in a place approximately where the arrow is a shown below. Perform the same procedure for the back.

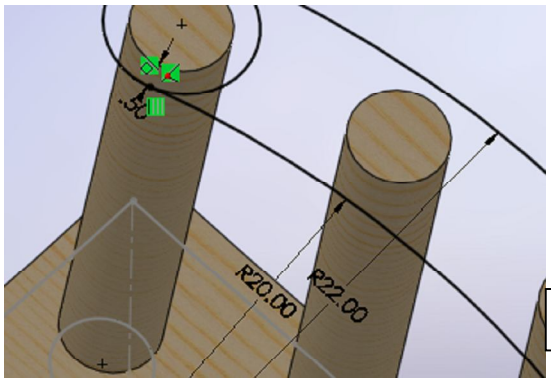

33

34. Touch one of the arcs and one of the circles and establish a "tangent" relation in the "Short Menu" shown or in the "Properties Manager". Do this for both sides and for the arc in the back on both sides.



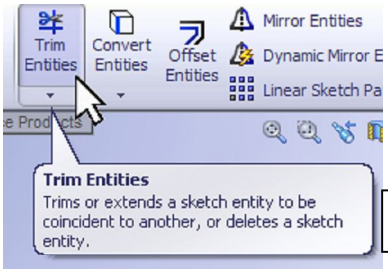
34

35. Use a "Smart Dimension" and place a dimension on the front arc of 20 inches and 22 inches for the back arc. Because they are fully defined they should now be black. The next step is to trim the portions of the two circles that we don't need before extruding this geometry.



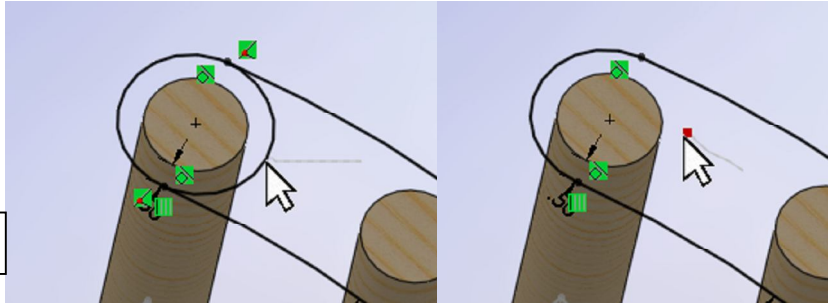
35

36. Go to the "Trim Entities" button in the "Sketch" portion of the Command Manager. This too is a fun tool, very productive.

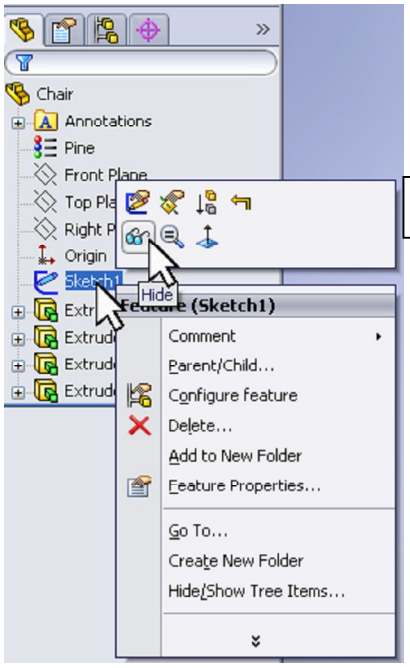


36

37. Take the cursor, place it inside the front and back arcs and with the left mouse button depressed move it toward the portion of the circle between the two arcs. As the cursor touches objects they disappear between other sketch entities. Do this for the other circle.


37

38. Extrude this sketch up 6 inches.
39. Go to the Feature Manager Design Tree and “Hide” the original layout sketch. Right click on “Sketch1” and choose the *eye glasses*, click with the left mouse button.
40. Modify some of the features for: fillets, length, draft angles, pads on the bottom of the legs, etc.



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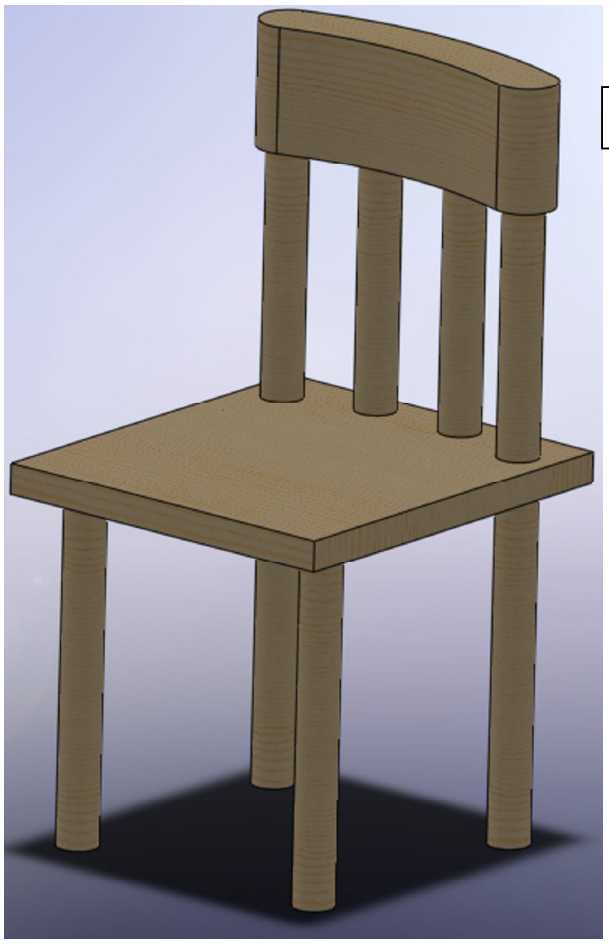

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