## 2D Caster Drawing

Lesson 1: Sketching a Drawing

1. Open a new Solid Edge 2D document.

2. Under the View menu, click on 2D Model Sheet and then click the "2D Model" tab at the bottom of the page.

3. From the Home Tab, Relate section, make sure that the "Maintain Relationships" and "Relationship Handles" are selected.

4. From the Tools menu, select IntelliSketch.

5. On the Intelliskecth section of the home tab, make sure the following checkboxes are set: End point, Center point, Point on element, Horizontal or Vertical, Parallel, Tangent.
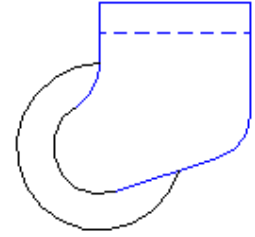
Sketch a wheel

6. On the Draw toolbar, click the Circle By Center Point button.

7. On the Circle By Center Point ribbon bar, click the Line Colour button and select the colour black from the palette.

8. On the drawing sheet, click near the center of the page to start the circle.

9. Type "70 mm" in the Diameter box and press ENTER (the circle is now placed).

10. Type "40 mm" in the Diameter box and press ENTER in the Command bar.

11. Move the mouse cursor over the circle that you just placed, at approximately the center point of that circle and the circle center point symbol will highlight. Click to place the circle position.

12. Click the right mouse button to reset out of the circle command.

Design Check

At this point, your drawing should look like the following drawing. If your sketch does not look similar, you should review the previous steps to correct the drawing before going on to the next step.
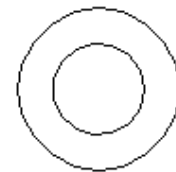
Sketch a support bracket

13. In the Draw Section of the home tab, click the Line button.

14. On the Line Command bar, click the Line Colour button and select the colour blue from the palette.

15. Move the mouse cursor over the top of the outer circle. Notice the point on symbol and a highlight from the center of the circle is displayed and click when the indicators is displayed.
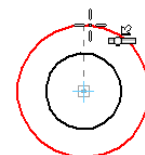
16. Type "25 mm" in the Length box (this should already be highlighted, so just type in the value directly) and press the TAB key. Type "90" in the Angle box and press ENTER.

17. Move the mouse cursor to the right so that the line is horizontal. Notice the indicator next to your mouse cursor, indicating a horizontal relationship. ype "65 mm" in the Length box and press ENTER. Click on the drawing sheet to place the line.
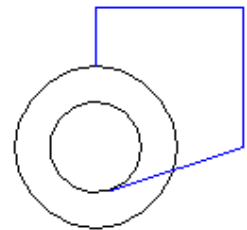
18. Move the mouse cursor down so that the line is vertical and aligned with the center of the circle (move the cursor over the circle to activate the geometry association). Notice the indicator next to the mouse cursor, indicating a vertical relationship, and the dashed line, indicating center alignment with the circle (the circle will also highlight). Click on the drawing sheet when the vertical and alignment indicators are displayed.
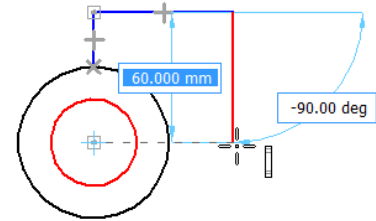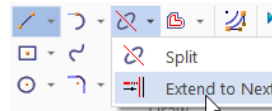
19. Move the mouse cursor to the bottom of the inner circle. Click when the point on and tangent indicators are displayed. Click the right mouse button to exit the line command.

Design Check

At this point, your drawing should look like the following drawing. If your sketch does not look similar, you should review the previous steps to correct the drawing before going on to the next step.

# Extend a line

20. Click the Extend To Next button.

21. Move the mouse cursor to the bottom of the central vertical line. Notice that the line extends down to the top of the inner circle. Click to extend the line.

Trim
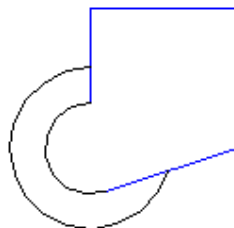
22. In the Draw section, click the Trim button.

23. Move the mouse cursor inside the inner circle. Click and drag the cursor up and to the right until it crosses both circles. Release the mouse button to trim away those portions of the circle.
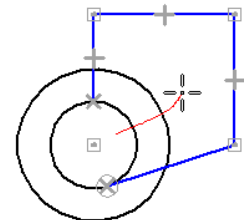
Design Check

At this point, your drawing should look like the following drawing. If your sketch does not look similar, you should review the previous steps to correct the drawing before going on to the next step.

Place a fillet

24. In the Draw section, click the Fillet   button.

25. On the drawing sheet, click the intersection of the two lines on the lower right portion of the support bracket.

26. On the Fillet ribbon bar, type "20 mm" in the Radius box and press ENTER.

27. On the drawing sheet, click the intersection where the vertical line intersects with the inner circle to place another fillet with a 20 mm radius.

Place a hidden line

28. In the Draw section, click the Line button.

29. Change the line style on the line ribbon bar from Normal to Dash.
     Note: The dashed line represents a hidden line.

30. On the drawing sheet, draw a horizontal line below and parallel to the top line.

31. Click the right mouse button to finish the line command.

Design Check

At this point, your drawing should look like the following drawing. If your sketch does not look similar, you should review the previous steps to correct the drawing before going on to the next step.
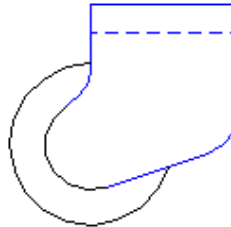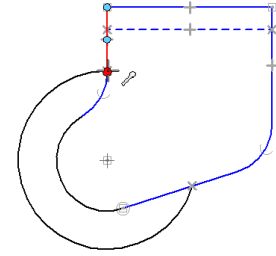
Apply relationships

32. In the Relationships Section, click the Horizontal/Vertical button.

33. Move the mouse cursor over the bottom end of the left vertical line. Notice the relationship indicator to the right of the mouse cursor, representing the end point relationship. Click to accept that end point.

34. Move the mouse cursor over the inner circle line and then back to the center of the circle. Click when the center point relationship indicator is displayed.
Note: This relationship ensures that the vertical line remains aligned with the center point of the hub.

**35.** In the Relationships section, click the Equal button.

36. Move the mouse cursor over the left fillet and click when the line highlights.

37. Move the mouse cursor over the right fillet and click when that fillet highlights.
Note: This relationship ensures that the radius of both fillets remain equal. The equal relationship indicators may not be visible until you zoom in.

Design Check

At this point, your drawing should look like the following drawing. If your sketch does not look similar, you should review the previous steps to correct the drawing before going on to the next step.
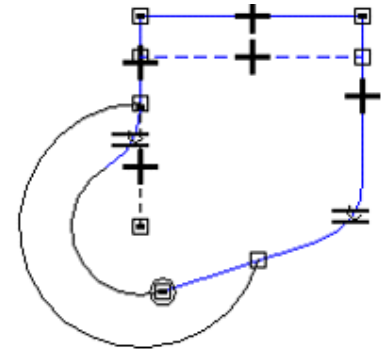
Lesson 2: Dimension a Drawing

Set dimension options

1. In the Dimension Section of the Home tab, click Style.

2. Ensure that the Style Type field is set to Dimension. Select the Iso (mm) dimension style and select the Modify button.

3. On the Text tab of the dialog box, type "5" in the Font Size box.

4. On the Units tab of the dialog box, in the Linear box, select ".12" from the Round-off list. Select the OK button and the Apply button.

5. While Iso (mm) is selected, select the New button. Name the dimension style as "ISO RAD".

6. On the Text tab of the dialog box, set the orientation to Horizontal. Click OK and Close.

7. On the View icons at the bottom, click the Fit button to fit the drawing sheet in your window.

Place dimensions

8. On the Left, click on the Layers tab.

9. Click the New Layer button. Type Dims as the Layer name. This now becomes the active layer.
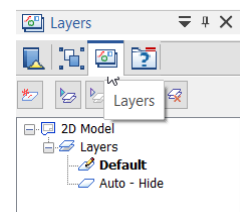
10. On the home tab in the Dimension sectio, click the
    SmartDimension button.

11. Move the mouse cursor over the top horizontal line and click when the
    line is highlighted.

12. Move the mouse cursor up to position the dimension and click to place
    the dimension.

13. Change the dimension style to ISO RAD on the ribbon bar. Move the
    mouse cursor over the upper left portion of the outer circle (wheel) and click when the arc
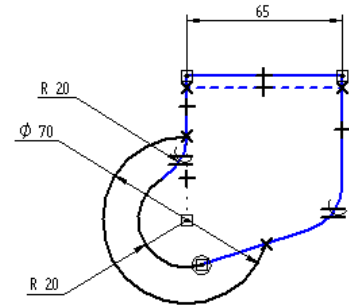   highlights.
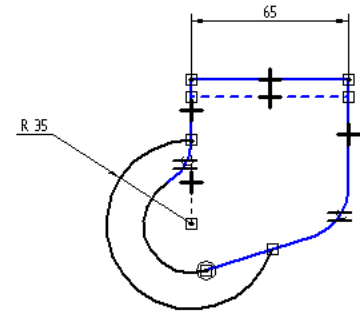
14. On the ribbon bar, click the Diameter   option. Move the mouse cursor up and
    to the left to position the dimension and click to place the dimension.

15. Move the mouse cursor over the lower left portion of the hub (the
    inner arc), click the Radius option and click when the line
    highlights.

16. Move the mouse cursor down and to the left to position the
    dimension and click to place it.

17. Move the mouse cursor over the upper left fillet and click when
    the line highlights.

18. Move the mouse cursor slightly up and to the left to position the dimension and click to place.

19. Move the mouse cursor over the bottom diagonal line that is connected with the lower
    portion of the hub and extends up to the right. Click when the line highlights.

20. On the SmartDimension ribbon bar, click the Angle   button.

21. Move the mouse cursor to position the dimension to the right and click to place it.

22. In the Dimension section, click the Distance Between button.

23. Change the dimension style back to ISO (mm).

24. Move the mouse cursor over the top horizontal line and click when the line highlights.

25. Move the mouse cursor over the hidden (dashed) horizontal line and click when the line
    highlights.

26. Move the mouse cursor to position the dimension to the right and
    click to place it.

27. In the dimension text box, type "3 mm" in the Dimension Value
    box and press ENTER. Notice the distance between the two lines
   changes to reflect the new distance.
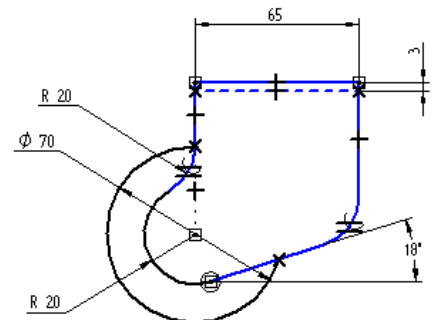
Design Check

At this point, your drawing should look like the following drawing. If your sketch
does not look similar, you should review the previous steps to correct the drawing before going on to the next step.

Lesson 3: Plan View

1. Select the Pan button from the view toolbar at the bottom left. You can also use the key combination of Ctrl+Shift+Right mouse button to pan.

2. Move the current drawing down to the bottom edge of the page (zoom out a bit if you need more space).

3. Draw a vertical line (change the colour to black), 25 mm long, approximately above the left side of the wheel.

4. Click the Tangent relationship, select the line just placed and then select the caster wheel in the front view.
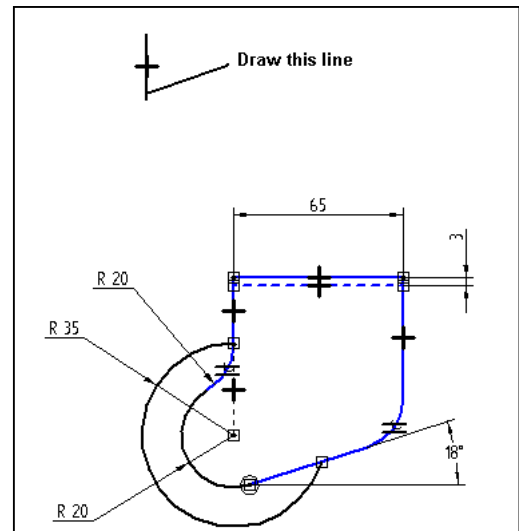
5. Click the Line button and place the first point at the bottom end of the vertical line just placed.

6. Move the cursor down to touch the left vertical line of the caster wheel casing and then place the second point of the line horizontal from the first point and vertically above the line just recognised. Right mouse click to reset the line operation.

7. Repeat this procedure to draw a horizontal line across the top to complete the plan of the wheel.
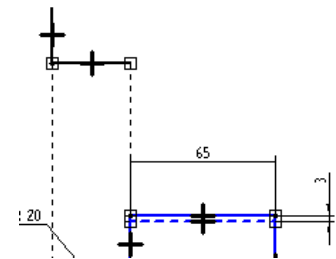
8. Select Show Grid from the Draw section of the Home tab. Reposition the origin at the free end of the last line placed. Select Grid Options and make sure the "Show readouts" is ticked. Turn on the XY keyin.

9. In the layers box on the left, double click Default so that the new lines will be placed on this layer.

10. Click the Line button. Change the Colour to Blue.

11. Type in the coordinates X:0.00, Y:10.00 and place a point. Create a vertical line down of length 45 mm using the length text box.

12. Turn off all of the grid options.

13. Create a horizontal line that is vertically above the right side of the casing (you will need to move over the right vertical line of the casing in the front view to recognise it).

14. Create a vertical line the same length of the blue vertical line created above. Finish by closing the shape with a horizontal line.
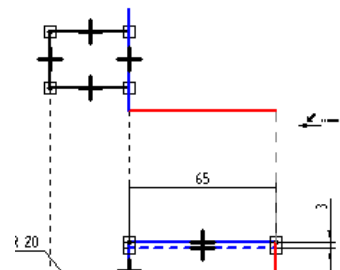
Design Check

At this point, your drawing should look like the following drawing. If your sketch does not look similar, you should review the previous steps to correct the drawing before going on to the next step.
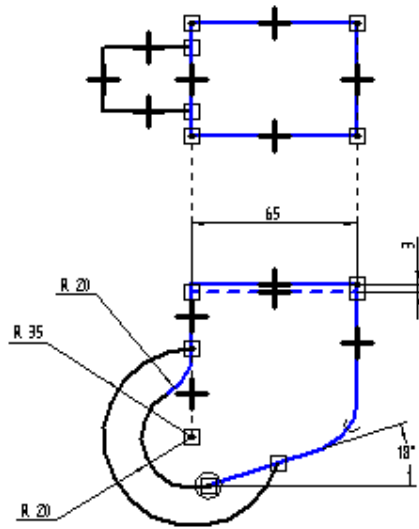
15. Click the Offset   button. On the ribbon bar, change the select option to single and the   Distance to 3mm.

16. Select the top horizontal, blue line of the plan view. Click the Tick button on the ribbon bar and click a point below the line just selected. Click the right mouse button to reset the command.
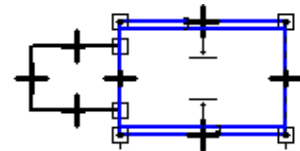
17. Select the bottom horizontal, blue line of the plan view. Right mouse click to activate the Tick button on the ribbon bar and click a point above the line just selected. Click the right mouse button to reset the command.

18. Click the Select button. Select the first offset line and then, holding down the Ctrl button, select the second offset line just created. On the ribbon bar, change the Line Type to Dash.

19. Click the Line button.

20. Click a point on the left end of the upper blue, horizontal line. Move left 20 mm and place a horizontal line. Place a vertical line of 3 mm (move the cursor down a long way to see the vertical icon and then type in 3) and place a third (horizontal) line back to the left end of the dashed line.
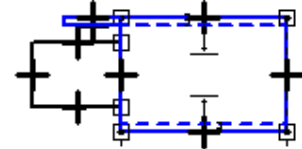
21. Click the Tangent relationship button. Click the 3mm vertical line just placed (to select the correct line, hover the cursor over the line and wait for Quickpick to display and then right mouse click to see each of the lines highlight) and then click the 20mm inner arc in the front view. This should adjust this profile to match the front view.

22. Repeat the procedure on the bottom edge of the casing.

23. Click on the 20 mm radius dimension and change its value to 25 mm and verify that the plan view alters to match it. Key-in Ctrl Z to undo the last change.

Design Check

The final drawing should look like the following drawing. If your sketch does not look similar, you should review the previous steps to correct the drawing before going on to the next step.
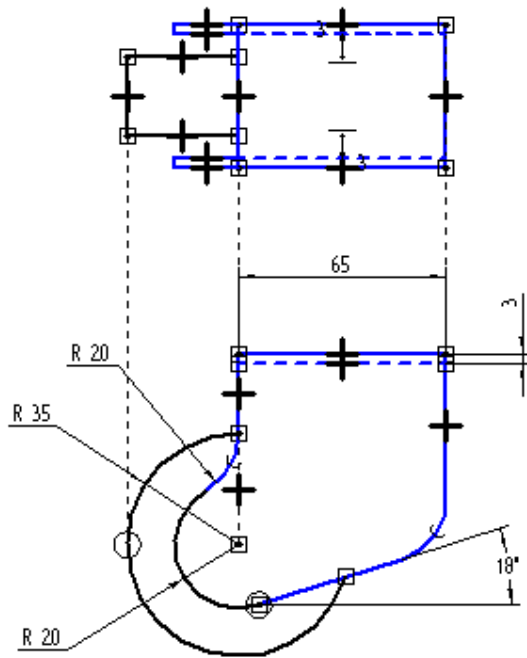
24. Click on Sheet 1 at the bottom of the page.

25. Click on the 2D Model View icon from the Drawing Views section of the Sketching tab. Put a box selection around the top view and place the drawing view to the top left of the page.

26. Click on the 2D Model View icon. Put a box selection around the bottom view and place the drawing view below the first.

27. Right mouse click on the top view and select Create Alignment. On the ribbon bar change Alignment position option to Keypoints and the option to Vertical Alignment.

28. Click the bottom right corner of the top view. Click the top right corner of the bottom view. The views are now linked so that if you move one, the other will also move.