START UP I-DEAS

1. Bring up I-DEAS Master Series, and start a new model file as shown below in the I-DEAS Start window:

   Project Name: Your account name
   Model File Name: Truck_wheel
   Application: Design
   Task: Master Modeler

   Click OK button, and OK again in the I-DEAS Warning window

2. Make sure the units are in mm (milli Newton); mm is the default units.

3. Use Workplane Appearance and setup the following:
CREATE THE BASE FEATURE

4. Pick Line icon to create a single horizontal line. The horizontal line will be used as an axis of revolution.

5. Use Polyline icon and create a sketch as shown below. Use Pre-Select to delete all dimensions and re-create the dimensions as indicated in the figure, and then modify the dimensions.

To change the decimal places of dimensions, pre-select all dimensions. Click the Appearance icon, and Product & Manufacturing Information window appears.
Click Units/Decimal Places button,
Enter the desired decimal places in Decimal Places box.

CREATE A REVOLVED FEATURE

6. Select Revolve icon – the icon is located at the same icon stack as the Extrude icon.
7. Pick the section we just created.

8. Press Enter or middle-button-mouse to continue.

9. Pick the horizontal line (single line) we created as the axis of revolution.

10. The Revolve Section window appears, and make sure you have 360 degrees and New Part settings as shown below:

![Revolve Section window](image)

11. Then click OK.

12. To view the 3-D model, select Isometric icon or F3 + move the mouse to for dynamic rotation function, and then Shaded icon.

13. Reset the display to wireframe.
CREATE A CIRCULAR PATTERN


15. Choose Circle – Center Edge icon, and create a circle below and toward the right of the 3-D model.

16. Modify the diameter of the circle to 22.

17. Extrude the circle to make a new part with distance 40.

18. Choose Isometric View icon.

19. Rotate the newly created cylinder 90-degree about Y-axis. The Rotate icon is located in the same stack of Move icon.

20. Pick Circular Pattern icon.

21. When prompted with “Pick part to make a pattern of”, click the cylinder we just created.

22. Pick the right side plane of the cylinder when prompted with “Pick patterning plane”.

23. In the prompt window, the message “Pick a center point for circular pattern (done)” is displayed.

24. Select Translated option by pressing and holding down the right-mouse-button.

25. Pick the center of the right-side plane of the cylinder when prompted with “Pick point to translate from”.

26. The message “Enter translation X, Y, Z (0.0, 0.0, 0.0) is displayed. We will select a point that is directly below the center of the cylinder. In the prompt window, type in: 0,-43,0 and hit Enter. Bring the cursor in the graphic area, and press Enter again, and Circular Pattern window appears. Make sure you have the following settings:
27. The resulted operation is shown below.

28. Click Y-Z plane icon, and the orientation is shown as the following:
29. We will now move the center of the circular pattern to the center of the Truck_wheel.

30. Choose Move icon, and select the cylinders. Hit Enter.

31. Select Move To from pop-up window.

32. Pick the center of patterned cylinders when prompted with “Pick point to move from”.

33. Pick the center of Truck_wheel when prompted with “Pick point to move to”.

34. Press Enter or middle-button-mouse to finish the command.

35. Click X-Y icon (Front View) and Y-Z icon (Side View) to make sure the patterned cylinders are correctly positioned or oriented.

36. Note that to cut the hole, we will perform Boolean operation CUT using the cylinders as the cutting tool.

37. Select Isometric View icon.

38. Choose Cut icon.

39. Pick the right-side planar plane of one of the patterned cylinders when prompted with “Pick planar surface from the moveable part”.

40. Pick the right-side planar plane of the hub of the Truck_wheel when prompted with “Pick planar surface from the part to cut”.

Move To
Move Along
Slide On Screen
Copy sw
Measure
Backup
Cancel

Click X-Y icon (Front View) and Y-Z icon (Side View) to make sure the patterned cylinders are correctly positioned or oriented.

Note that to cut the hole, we will perform Boolean operation CUT using the cylinders as the cutting tool.

Select Isometric View icon.

Choose Cut icon.

Pick the right-side planar plane of one of the patterned cylinders when prompted with “Pick planar surface from the moveable part”.

Pick the right-side planar plane of the hub of the Truck_wheel when prompted with “Pick planar surface from the part to cut”.
41. Click Done in the pops-up menu.

42. Pick Rounds icon to create rounds on the edges as required by the problem. Make sure you select the right round’s radius and edges to be rounded – see Figure in the first page. You can Pre-Select the edges that have the same radius of rounds.

43. The resulted object is shown below.

![Truck Wheel with no rounds](image1)

![Truck Wheel with rounds](image2)