Exercise 1. *In this exercise you are going to use the three options given for a circle and an arc construction.*

Open inventor, make a new part, start a new sketch and draw three lines as shown →

Click drop down arrow under the circle icon and select the ‘tangent’ circle option. Start constructing it by selecting the lines. The circle will appear after the third line is selected. It doesn’t work with two lines but three.
Select ‘ellipse’ option and construct few to see how that tool works. Try to add dimensions to the ellipses. Dimension tool will snap onto two semi-axes. The dimension value is not critical for this exercise.

Select the arc tool, **three point option**. Click anywhere in the sketch area and then drag the cursor away. Click again, drag and click once more to make a final selection. As you can see, it takes three clicks and two drags to construct an arc without any value input. Precision can be achieved by typing values in spaces highlighted in blue (press enter to move to the next space), or with dimensions and constraints which will be exercised later.
Select arc, **center point** option. Click, drag away (without clicking) type 7, press enter, type 120, press enter, right click and select OK.

Pick the **tangent** arc option and construct few of those.
Tangent arcs are always constructed at the end point of the selected geometry. That point will be highlighted as a green dot at the end closest to the cursor. For any other case where tangency is needed, arcs should be constructed with the three point or center point options and tangent constraint used.
Exercise 2.

*In this exercise you are going to apply various constraints.*

Use constraints from the Constrain palette and try apply them as shown:
End results may vary but here’s one possible outcome:
Exercise 3: Lifting lug. *In this exercise you will constrain and fully dimension geometry, trim and extend it, make separate sketches for different extrusions, sketch on an object’s face, project geometry, and mirror a solid object.*

Start with a new part → start 2D sketch (plane XY or whichever you prefer). Draw a circle, 2.25 inches diameter, constrain the center of it with the origin*, draw an arc (for example click on drop down arrow under the arc icon and pick the center point option).
When constructing an arc using the center point option, the first selection is going to be the center of the arc. Use origin point again, or the center of the 2.25 circle. The radius of the arc should be 4.5 inches.

Add 4 more lines as shown. Apply constraints so the lines 1 and 3 are horizontal, line 2 vertical and lines 1 and 4 tangent to the arc.
If horizontal and vertical (or any other) constraints were automatically grabbed while you were sketching, the software will return a message that those already exist if you try to add them.

Click cancel and continue with other constraints.

Trim and extend your geometry as shown:
Add dimensions and finish the sketch.
Extrude 0.5 inches using symmetric option.
Assign Steel, Mild as material.

Use iProperties to check the mass of your lifting lug (optional).
Start a new sketch on the same plane you used for the first one.
Project geometry

Once you started the new sketch, click on the project geometry icon shown below. Hover over the model and note how various edges get highlighted when the mouse pointer is over. Once you make your selection, the outline of it is going to become yellow. Projecting geometry enables a precise relationship between various extrusions or other design steps of a part.

Project the edge and sketch off of it as shown.

(Dimensions are 0.5, 2 and 20 inches)
Finish the sketch and symmetrically extrude 8 inches.
Make a new sketch. Select and click on the side face of your part, or select the face, right click and pick. Project the eye of the lug into your new sketch and add an 8 in circle that is concentric with it. Finish the sketch and extrude 0.25 in.
Mirroring solids

The red arrow below points to the mirror button. Click on it and note a dialog box (on the right). We want to mirror the last extrusion we made so we will use the default option – mirror individual features. Click on the second option – mirror a solid just to see what happens. Then click back on the first option.

Selections can be made either under the model tree or the model itself. Once you have finished selecting features you want to mirror, (black arrow points to it) you need to click on the mirror plane button inside the dialog box BEFORE you select the plane (red arrow points to the plane).
As you moved on, the arrow on the features button is not red but white and features that are to be mirrored are highlighted in blue. The arrow next to mirror plane turns white once you selected it and the preview of the result is highlighted in green. Click ok to finish mirroring.
The model is finished and all steps are shown in the model tree.
Expand the model tree as shown and hover over each entity to see how it is being highlighted on the model.
*The last note goes to the origin.

If the origin is accidentally deleted from a sketch, it can be retrieved with project geometry option. Expand the ‘origin’ folder of the browsing tree, select ‘center point’ and project it.