Inventor curved surface construction tools include Extrude, Revolve, Loft, Sweep and Coil (Helix).

These tools allow for generation of complex curved surfaces which can be used to define most of the surfaces needed for consumer goods, machine parts, automobile bodies, aircraft shapes, etc. When needed, curves can be imported from other programs such as Rhino, Autodesk Alias and other CAD softwares.
Cylindrical Surfaces

Cylinders are very common elements of a part model. A solid (positive) cylinder will appear as a circle in one view and as a rectangular outline in other views. The centerline through the middle of the front and side views in this example are very important. The centerlines convey information about the shape of the object.

Contour lines are needed to describe the extent of the part.

- Contour point “A” in the top view is contour line 2,4 in the front view.
- Contour point “C” in the top view is contour line 3,5 in the front view.
- Contour point “D” in the top view is contour line 6,8 in the side view.
- Contour point “B” in the top view is contour line 7,9 in the side view.

Contour lines are not sharp edges on the part - they only show the extent of the surface in a particular view.

Contour lines for holes (negative cylinders) follow the same rules. These contour lines show as hidden lines in the front and side views in this example.

A (right circular) cylinder may be created in several ways.

- Extrude a circle. (Protrusion / Cutout).

Extruding concentric circles will create a positive and a negative cylinders.

- Revolve a line parallel to an axis.
Surfaces of Revolution

- The profile in this example has lines which are parallel to or perpendicular to the axis.

Three cylindrical surfaces and three normal surfaces were generated.

Axis Icon is used to set the axis of revolution.

- Lines at an angle to the axis will create conical surfaces.

- Circular arcs will generate a sphere or torus.
Problem 1. Surface of Revolution

A. Draw the Overhead Trolley Wheel cross-section. Revolve about the centerline 360 degrees.
B. Create a drawing of the Trolley Wheel. Show Front, right and isometric views. Use “B” size format.
   Do not dimension at this time. It will be dimensioned later.
   Save the model and the drawing - they will be used in a later chapter.
   Material: Maleable Iron (Cast Iron)

Trolley attaches to a overhead “I” beam. A chain hoist attaches to the hook. Device is used to lift and move heavy items on a shop floor or construction site.

Grooves are for snap-rings to hold bearing in place.
Problem 2.
Loft shape.

A. Create the shapes shown.
   __Sketch the rectangle.
   2.50 x 4.00 centered on the reference planes.
   __Create a reference plane parallel to the front plane 8” back.
   __Draw a 7” diameter circle centered on the reference planes.
   __Draw a path curve line on the top (or side) reference plane from the center of the rectangle to the center of the circle.

   __Loft the surface.
   Select multiple cross-sections.
   __Shell the part. Select both ends.
   Thickness .125
   Part: Cooler Duct
   Material: Plastic

   __Create a “B” size drawing. Show front, right and isometric views.
   No dimensions.

NOTE: SHELL THICKNESS = .125
REMOVE BOTH ENDS.
Problem 3. **Loft shape.** Material: Brass.

A. Model the *Mixer Base* as shown.
B. Create a drawing. Just show front, top, right and isometric views for now. No dimensions. **Save. This model will be used again later.**
Problem 4. Lofted shape.

A. Model the bowl shown. 2.50 in. total height. Wall thickness = .125 in. Use Shell to hollow the bowl. Top surface only.

Elliptical cross-sections.

B. Create a drawing - Front, Top, Side and Isometric views. No dimensions.
**Problem 5. Lofted shape.**

Create a lid for the bowl in the previous problem *as a separate part*.

The example shown uses five cross-sections.
- 2 elliptical
- 3 circular.

The knob at the top of the lid must not be too narrow at the bottom of the knob otherwise the **Shell** command will not work.

Hollow the lid.

**Assembly View**

- Start an Assembly drawing.
- Insert both the bowl and the lid.
- Use the move command to align the parts.
- Create an isometric drawing.
Problem 6.
Helical cutout.

Helical Cam. Matl: Steel

Cylinder diameter = 2.00
Cylinder length = 5.00

Cutout axis length = 4.00
Cutout pitch = 0.50

Create a drawing. Show front, right, isometric views.
Problem 7.
Helical cutout.

Revolved cutouts.

Model a 1-8UNCx3.00 Hexagon Head Bolt.
Cylinder diameter = 1.00in
Cylinder length = 3.00in
Hexagon = 1.50in across flats
Hexagon height = 0.66in
Thread length = 2.25in
(8 threads / inch = .125 pitch)
Thread pitch = 0.125
Right-hand thread.

Make a drawing. Show front. Top, right and isometric

Material: Steel

Triangle = .124 in on a side. (.125 in. might result in “Intersecting Curve” error.)
End of bolt. Position triangle just above corner to avoid “submarine” cut.

Note: The computer may stop responding for an extended time due to the amount of processing required for this model. File size approx 1.5mb

Create revolved cutouts for the thread and the head of the bolt.

Alternate Problem:
Model a Left-Hand Thread.

Model in 2-face view not 3-face view.
**Problem 8. Swept Shape**

Frame Rail for a golf cart. This part might be hydro-formed. A round tube is capped at both ends. It is placed in a form and water is pumped in at 100,000 psi. The resulting part has very high strength and rigidity. This process is used for auto and truck frames as well as high strength posts for lights and signs.

Start a Inch part file.

Turn on the three reference planes.

Make the sketch shown on the Right plane. This will be the path curve.
Inventor Curved Surfaces

Draw the cross-section on the Front plane.

**Sweep** the shape.

Make a dimensioned drawing as shown on the previous page.

*Look for examples where sweep shapes can be used to model handrails, piping and wiring etc.*