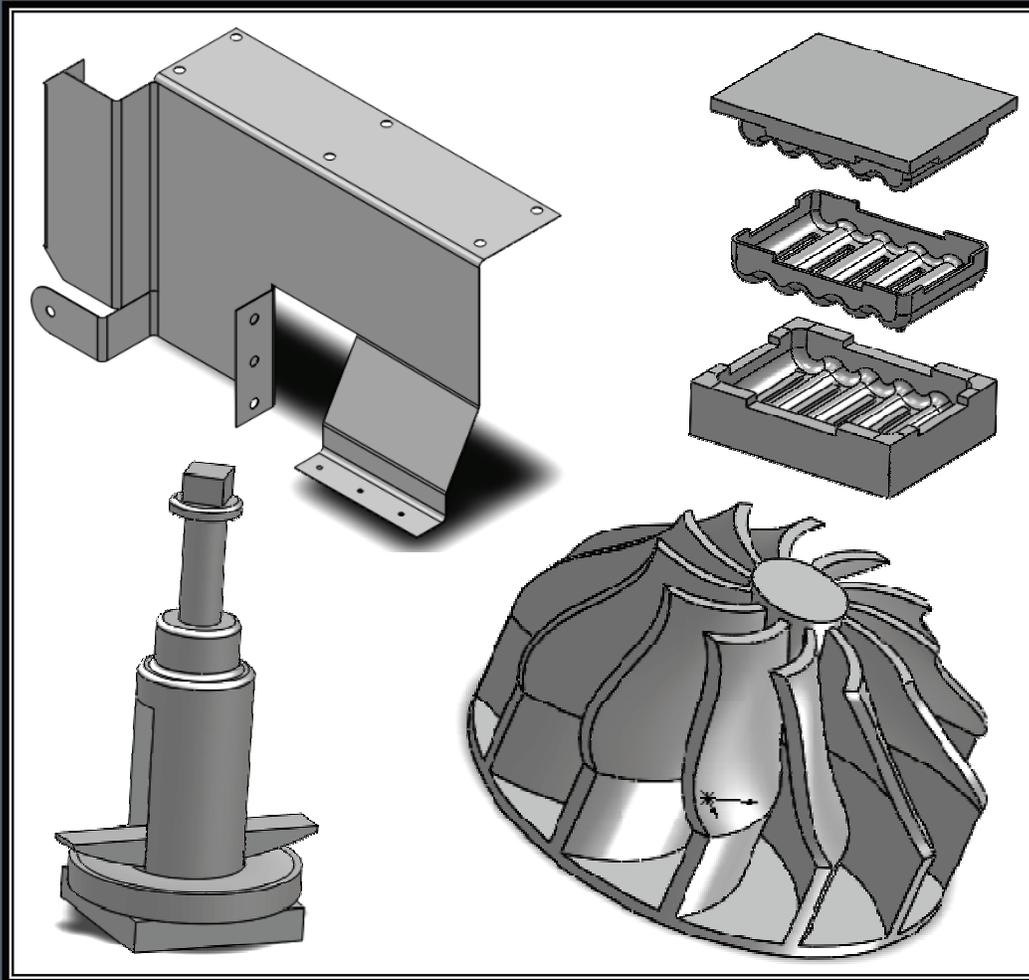


A Comprehensive Introduction to SolidWorks 2011



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Chapter 2

Geometric Construction Tools

Objectives:

When you complete this session you will have:

- Learnt about sketch entities in SolidWorks
- Learnt about sketch tools in SolidWorks
- Used the sketch tools to modify sketch entities in order to produce parts

INTRODUCTION

SolidWorks uses *sketch entities* and *tools* to facilitate the creation of parts. While *sketch entities* have specific geometries (line, rectangle, parallelogram, slot, polygon, circle, arc, ellipse, parabola, spline, etc.), some of the *sketch tools* (fillet, chamfer, offset, convert entities, intersection curves, trim, extend, split, jog line, constructive geometry, mirror, stretch, move, rotate, scale, copy, pattern, etc.) are used to modify the shapes of sketch entities. Entities are grouped to define a boundary or profile, which are closed regions needed for creating extruded or revolved parts. SolidWorks sketch entities and tools can be accessed by clicking the Tools bar.

Sketch Entities include **line**, **rectangle** (2-opposite vertices, centre, 2-opposite vertices, 3-point corner, 3-point centre), **parallelogram**, **slot** (straight, centre-point straight, 3-arc, centre-point arc), **polygon**, **circle** (centre-1-point, 3-points on perimeter), **arc** (centre-2-point, tangent, 3-point arc), **ellipse** (full, partial), **parabola**, and **spline**.

Sketch Tools include **fillet**, **chamfer**, **offset**, **convert entities**, **intersection curves**, **trim**, **extend**, **split** entities, **jog line**, **construction geometry**, **make path**, **mirror** (straight, dynamic), **stretch** entities, **move** entities, **rotate** entities, **scale** entities, **copy** entities, and **pattern** (linear, circular).

SKETCH ENTITIES

Sketch entities are commonly used geometric shapes which are building blocks for modeling simple and complicated shapes. In general, these entities are grouped into line, rectangle, polygon, slot, arc, circle, ellipse, parabola, and spline. Figure 2-1 shows the **Sketch Entities** toolbar.

For most of the entities definitions in this section, it assumed that a new **Part** has already been started, and the appropriate **Plane** chosen from the **Feature Manager**. Therefore, a generic format for sketch entities presented in this chapter is as follows:

1. Start a new **Part** document, click **Sketch** group on the **Command Manager**, and select (appropriate) **Plane** from the **Feature Manager**.

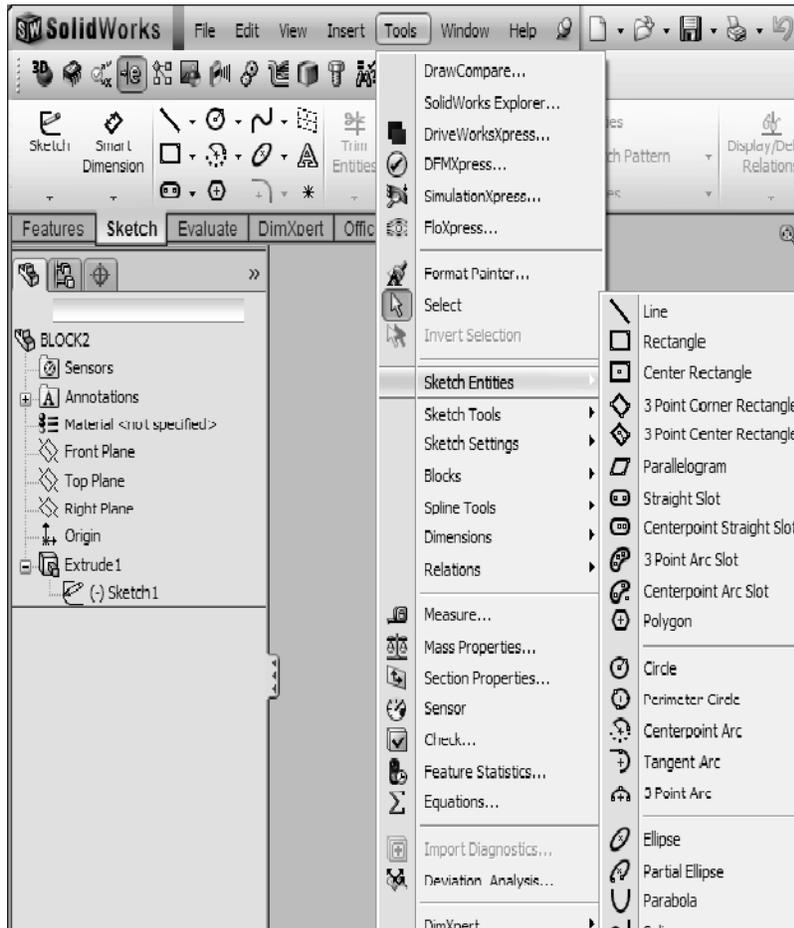


Figure 2-1 Sketch Entities toolbar

Line

Line is one of the most frequently used sketch entities in part modeling. There are basically two main types of line: solid and construction lines as shown in Figure 2-2.

1. Click the **Line** group on the **Command Manager**.
2. Select a starting point for the line and click the point.
3. Select other point and extend the line.
4. When the line is complete, right-click the mouse and click the **End chain** option.

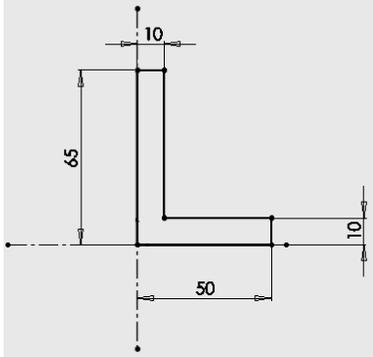


Figure 2-2 Solid and construction lines

Rectangle

There are a number of options for a rectangle entity: 2-opposite vertices, centre, 2-opposite vertices, 3-point corner, 3-point centre as shown in Figure 2-3. When a user is more interested in centering the rectangle about the origin of the graphics screen, then the **centre rectangle** is the one to go for. Let us illustrate the centre rectangle.

1. Click the **Rectangle** group on the **Command Manager**.
2. Select a centre click the point.
3. Drag the cursor away from the centre point to create a rectangle.
4. When the rectangle is complete, click the OK check mark on the PropertyManager to complete the rectangle construction.

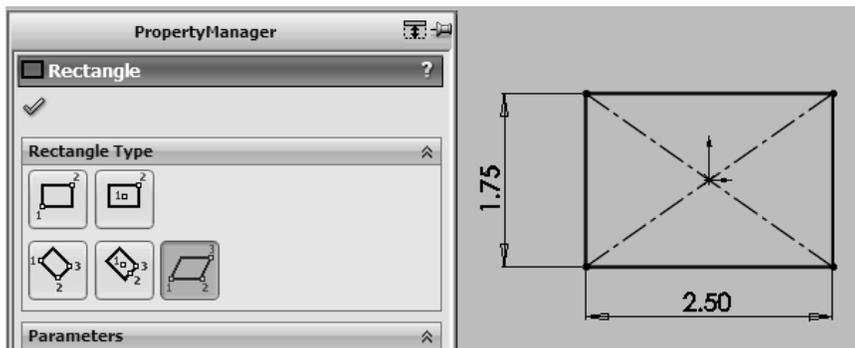


Figure 2-3 Rectangle entities

Parallelogram

Parallelogram is considered a special type of rectangle, hence it is grouped as a rectangle type as shown in Figure 2-4.

1. Click the **Rectangle** group on the **Command Manager**.

2. Select and click any two horizontal points.
3. Drag the cursor away from the second point to create a parallelogram.
4. When the parallelogram is complete, click the OK check mark on the PropertyManager to complete the parallelogram construction.

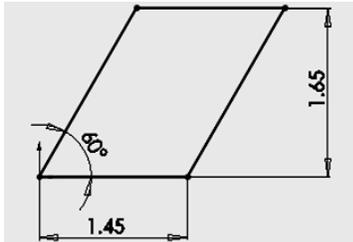


Figure 2-4 Parallelogram entities

Slot

There are a number of options for a slot entity: straight, centre-point straight, 3-arc, centre-point arc as shown in Figure 2-5.

1. Click the **Slot** group on the **Command Manager**.
2. Select and click any two horizontal points.
3. Drag cursor to the top or bottom of two points for the third point, to get the width of the slot.
4. When the slot is complete, click the OK check mark on the PropertyManager to complete sketching the slot.

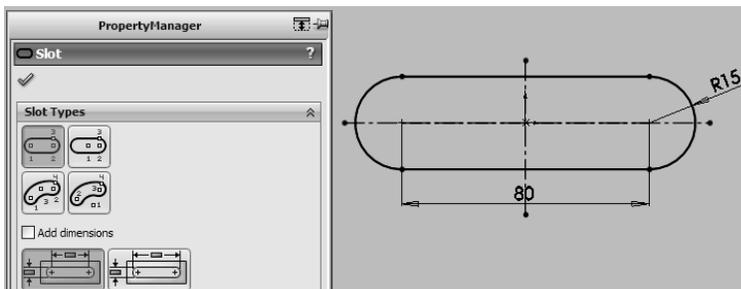


Figure 2-5 Slot entities

Polygon

Polygon has n -number of sides as shown in Figure 2-6. To sketch a polygon:

1. Click the **Polygon** group on the **Command Manager**.
2. Define the number of sides as six (hexagon); check Inscribed circle.

3. Click the OK check mark on the PropertyManager to complete sketching the polygon.

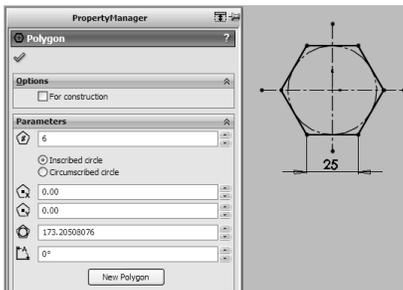


Figure 2-6 Polygon entities

Circle

There are generally two ways of defining a circle: centre-1-point, 3-points on perimeter. See Figure 2-7.

1. Click the **Circle** group on the **Command Manager**.
2. Select and click a point.
3. Drag the cursor away from the centre point to create a circle.
4. When the circle is complete, click the OK check mark on the PropertyManager.

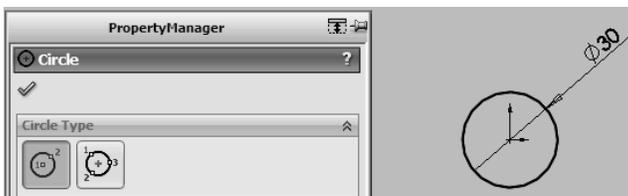


Figure 2-7 Circle entities

Arc

There are generally three ways of defining an arc: centre-2-point, tangent, 3-point arc. See Figure 2-8.



Figure 2-8 Arc entities

When the centre point and two other points are known, the **centre-2-point** should be used. When two lines are already sketched and two end points of an arc are to coincide with the end points of the two lines, then **tangent** options should be used. For general an arc where the centre point is not initially known, then the **3-point arc** option should be used. For example in Figure 2-9, only two points (end points of the lines) are defined for the **tangent** arc. However, the centre and the two end points of the lines are needed when the **centre-2-point arc** option is used as in Figure 2-9.

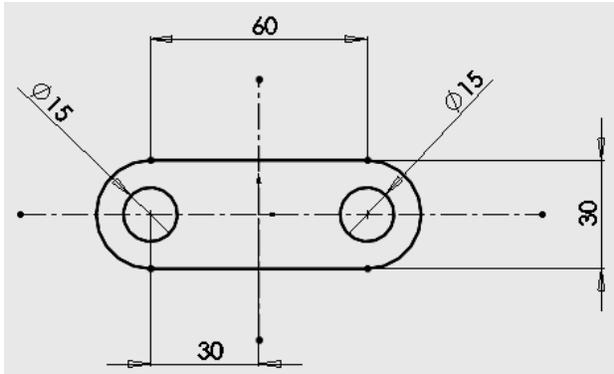


Figure 2-9 Arc entities for sketch definitions

Ellipse

An ellipse has a major diameter corresponding to its major axis and a minor diameter corresponding to its minor axis as shown in Figure 2-10. Two types are available: full and partial.

To create an ellipse:

1. Click **Ellipse** on the Sketch toolbar, or **Tools, Sketch Entities, Ellipse**.
2. Click in the graphics area to place the center of the ellipse.
3. Drag and click to set the major axis of the ellipse.
4. Drag and click again to set the minor axis of the ellipse.

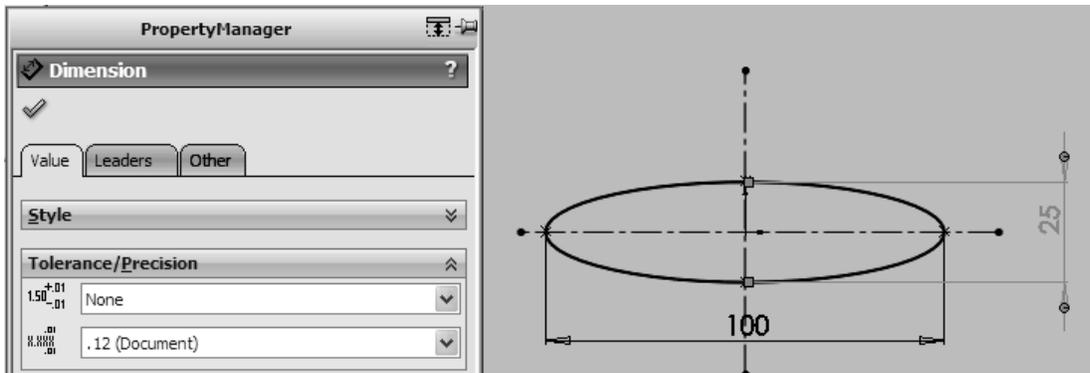


Figure 2-10 Ellipse entities

Ellipse can be used to model rugby ball by following these steps:

1. Trim half of the ellipse.
2. Sketch a line to close the shape.
3. Revolve the shape through the centre line.

Parabola

A parabola is a loci of points in which the distance between a fixed point, the focus, and a fixed line, the diretrix, are always equal. See Figure 2-11. It is a well known geometry in elementary mathematics. To create a parabola:

1. Click Parabola (Sketch toolbar) or **Tools, Sketch Entities**, parabola.
2. Click to place the focus of the parabola and drag to enlarge the parabola.
The parabola is outlined.
3. Click on the parabola and drag to define the extent of the curve.

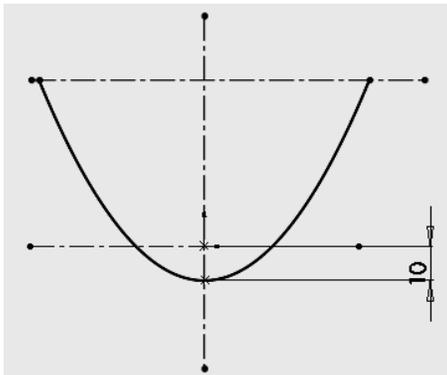


Figure 2-11 Parabola entity

Parabola can be used to model a satellite dish by following these steps (see Figure 2-12):

1. Trim half of the parabola.
2. Sketch horizontal line at the top and a vertical line through the focus.
3. Revolve the shape through the vertical centre line.
4. Shell the object with thickness of 1 mm.

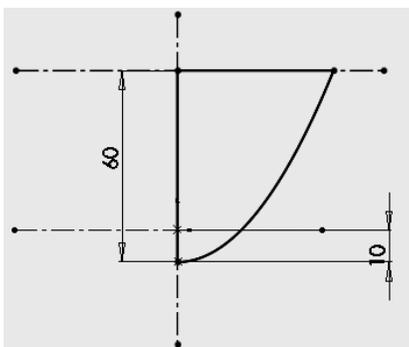


Figure 2-12 Half-parabola used for modeling satellite dish

Spline

Splines are geometries for modeling complex, general-form curves or surfaces as shown in Figure 2-13. Understanding the concept of splines is challenging and could be found in advanced CAD documentations. Free-form surfaces are extensively used in the automobile industry. The construction of aircraft, turbo-machinery, and automobiles, and recent applications in the advertising and animation industries, has become driving force behind the research into techniques for surface design. Splines have been heavily used in the automotive and aircraft industries and early work in the area began in those sectors for modeling of automotive bodies and fuselages.

Free-form surfaces are of different forms: quadratic or cubic spline (nonuniform rational B-spline) curve. To sketch a quadratic or cubic spline, specify:

1. First point
2. Next point (loops so that you can specify as many points as you want)
3. Complete the spline. The spline realized is a single entity.

The shapes modeled using splines can be “controlled” as shown using controlling polygons.

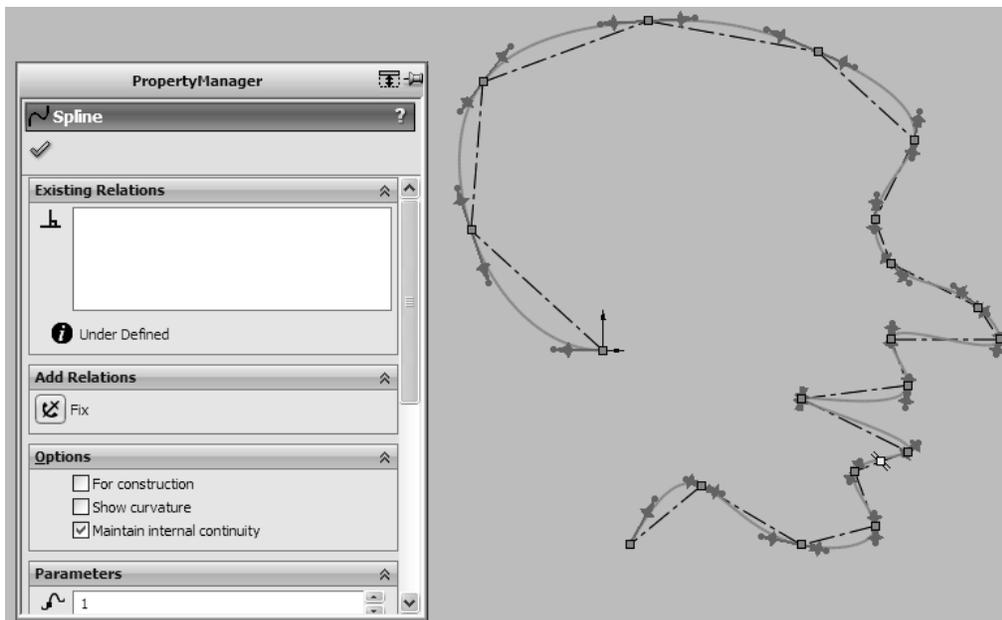


Figure 2-13 Spline entity

Parametric B-spline curves and surfaces have several advantages:

- 1) Ability to control the degree of continuity at the joints between adjacent curve segments, and at the borders between surface patches, independent of the order of the segments or the number of control vertices being approximated;

- 2) Ability to guarantee that smooth shapes would be generated from smooth data.

SKETCH TOOLS

Sketches are 2D in nature and Figure 2-14 shows the SolidWorks **Sketch Tools** toolbar mainly used for modifying them.

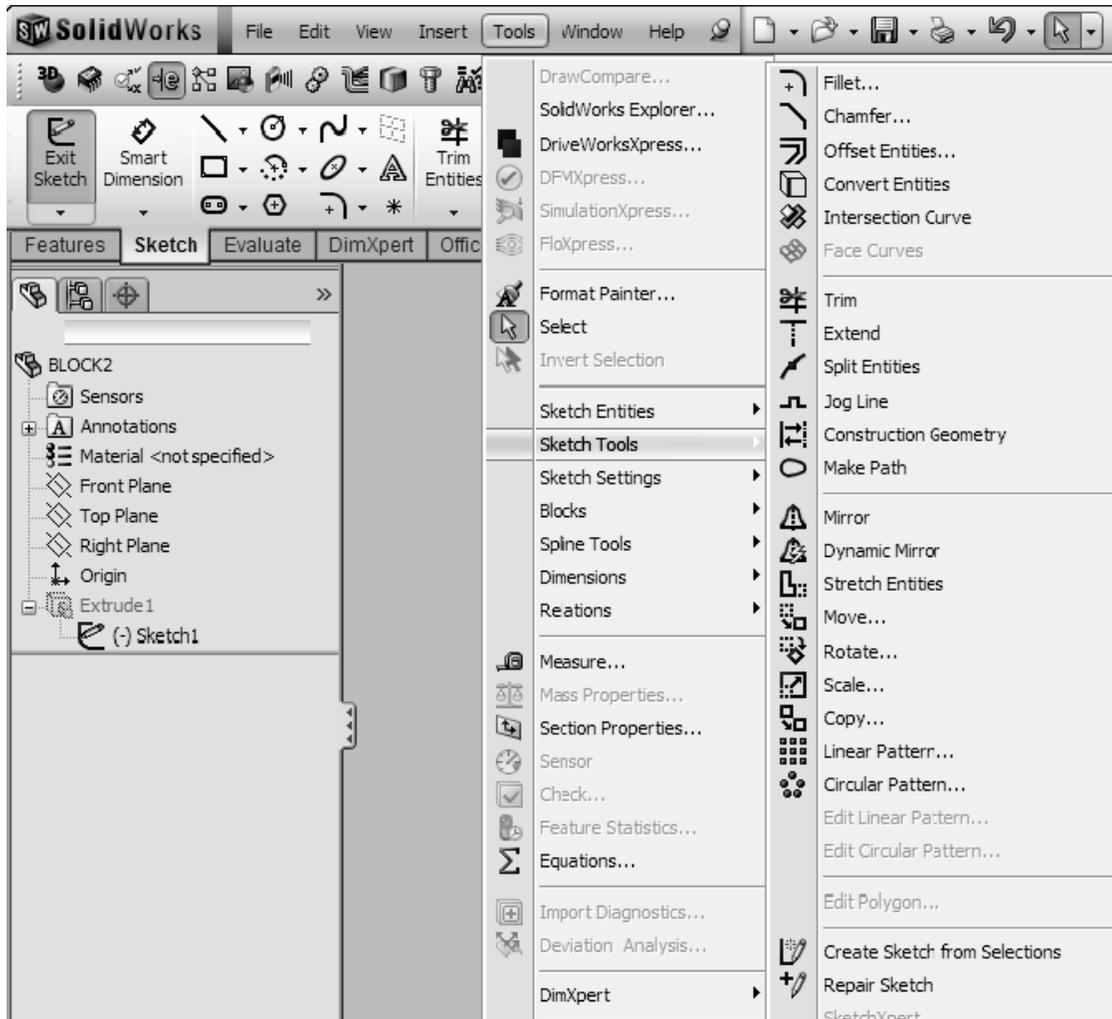


Figure 2-14 Sketch Tools toolbar

Fillet

The Fillet tool is used to create fillet between lines as desired in a model in order to remove sharp edges. The sketch shown in Figure 2-15 is filleted at a constant value of 5 mm radius between adjacent lines.

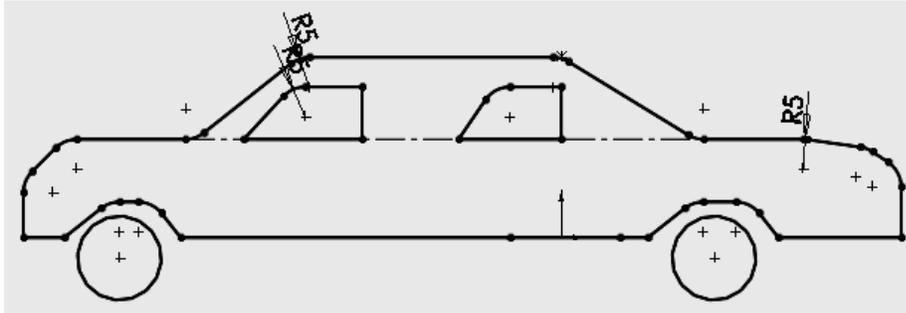


Figure 2-15 Illustrating fillet tool

Chamfer

The chamfer tool is used to create chamfer in a model. Generally, a chamfer of 45 degrees is common but other angles could be used. Let us illustrate chamfering by considering an object having all side horizontal or vertical. A chamfer is needed between the vertical and horizontal lines to the right of the object at an angle of 45 degree (see Figure 2-16).

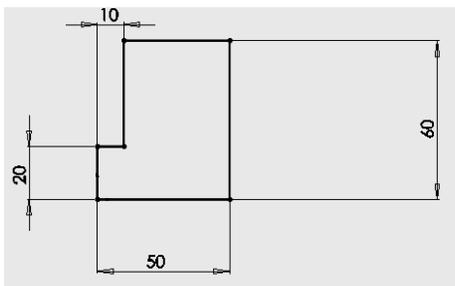


Figure 2-16 Initial geometry to apply chamfer tool

The resulting chamfered object is shown to have a vertical dimension of 30 mm and an angle of 45 degrees as shown in Figure 2-17.

1. Click the **Chamfer** group on the **Command Manager**.
2. Check the **Angle-Distance** option.
3. For the Distance, click the right vertical line; for the angle, specify 45 degrees.
4. When the chamfer is complete, click the OK check mark on the PropertyManager.

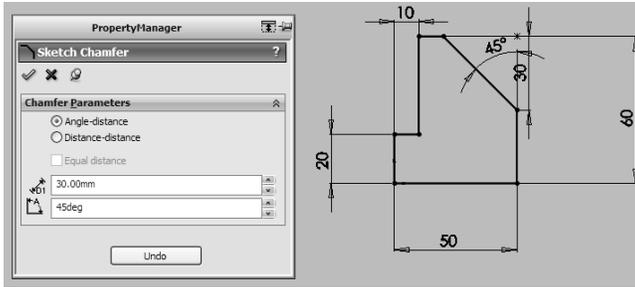


Figure 2-17 Chamfer tool

Offset

The **Offset** tool is used to draw entities parallel to existing entities of a sketch as shown in Figure 2-18. This tool is particularly useful and it can cut down time of modeling since all that is needed is the selection of the entities and the definition of the amount of offset.

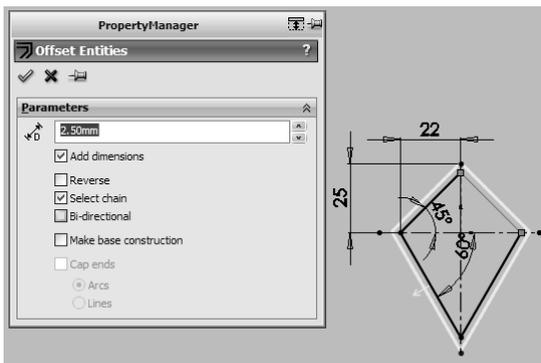


Figure 2-18 Offset tool

1. Click the **Offset** group on the **Command Manager**.
2. Select and click edges to be offset.
3. Specify the offset desired (2.5 mm in this case).
4. When the offset is complete, click the OK check mark on the PropertyManager.

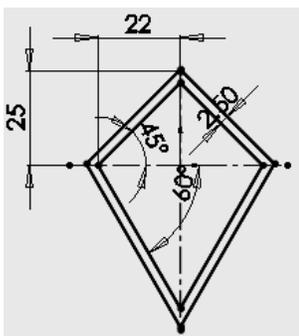


Figure 2-19 Offset geometry realized

Convert Entities

The **Convert Entities** tool is used to extract portions of an entity onto a plane and then used for further modeling as shown in Figure 2-20. For example, a cut-out is realized from a circle by using this tool to describe a sector. This sector could then be used to cut through the length or portion of the part being modeled (say, a cylinder).

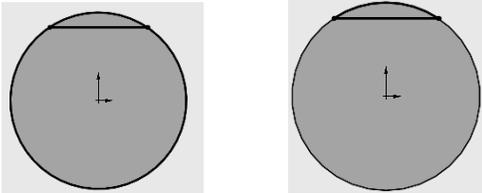


Figure 2-20 Illustrating Convert Entities tool

Trim

The **Trim** tool is used to clean up lines or arcs that are not needed as shown in Figure 2-22.

1. Click the **Trim** group on the **Command Manager**.
2. Select the **Power Trim** option.
3. Hold down and drag the cursor across the entities to be trimmed.
4. When the trim is complete, click the OK check mark on the PropertyManager.

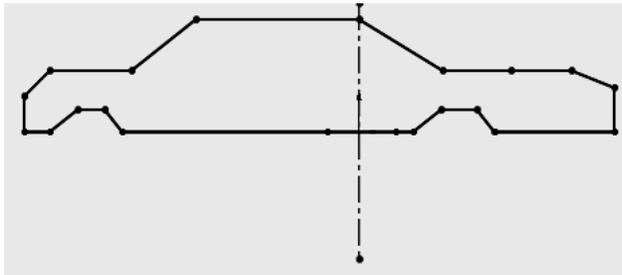
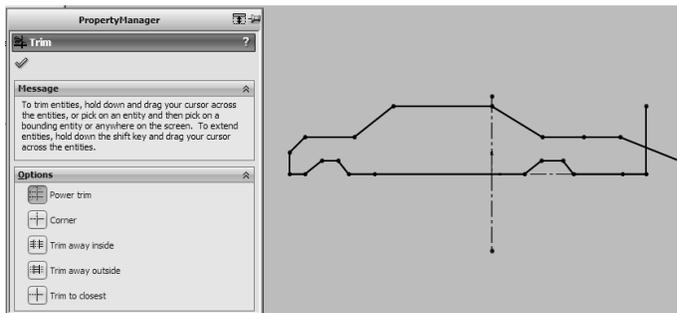


Figure 2-22 Trim entities tool

Extend

The **Extend** tool is used to extend an existing line to increase its length. See Figure 2-23. It is found in the Trim Entities group in the **Command Manager**.

1. Click the **Extend** group on the **Command Manager**.
2. Click the top and bottom lines to be extended.
3. When the extend task is complete, click the OK check mark on the PropertyManager.

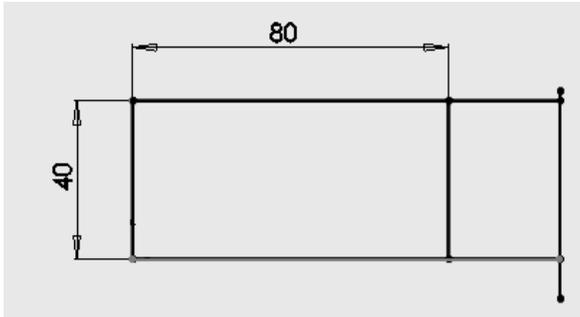


Figure 2-23 Extend entities tool

Split Entities

The Split Entities tool is found on the Sketching Tools. When it is chosen, click on the entity to trim approximately where you want the split. Figure 2-24 shows how this tool is used.

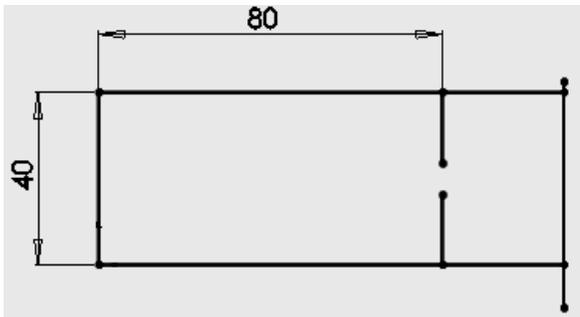


Figure 2-24 Split entities tool

Construction Geometry

It converts a solid line/circle/arc to construction line/ circle/arc.

Mirror

The **Mirror** tool is used to create a mirror image of an entity in a sketch. This tool is used when an axis, usually a line, exists about which an entity is to be mirrored. There are two forms of mirror tool: simple and dynamic. In the simple form, one-half of the entities are completed and mirrored about an axis as shown in Figure 2-25. In dynamic form, as an entity is sketch, it is dynamically mirrored about the chosen axis. The mirrored entities are show in Figure 2-26.

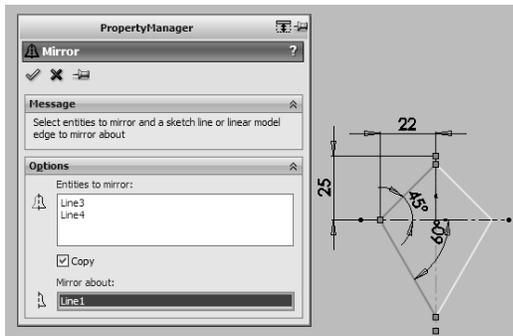


Figure 2-25 Mirror entities tool

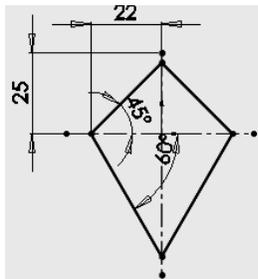


Figure 2-26 Mirrored entities

Stretch Entities

Stretch Entities tool stretches entities chosen; the value of stretch is based on the datum defined as shown in Figure 2-27. For example, the right-top, side and bottom lines are stretched about the left vertical line.

1. Click the **Stretch** group on the **Command Manager**.
2. Click the lines to be stretched.
3. Click the Base Define by clicking the left vertical line
4. When the stretch task is complete, click the OK check mark on the PropertyManager.

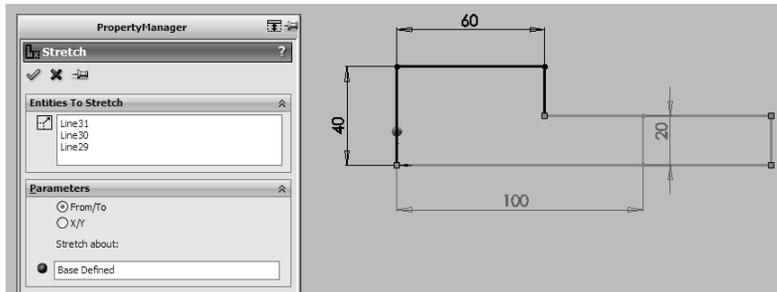


Figure 2-27 Stretch entities tool

The bottom line is stretched from 100 mm to 125 mm while the top-right line is stretched from 40 mm to 65 mm. See Figure 2-28 for stretched entities.

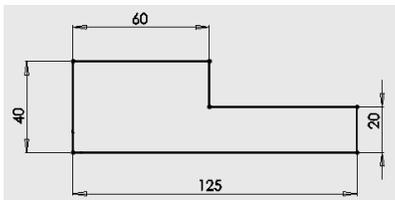


Figure 2-28 Illustrating stretched entities

Move Entities

The **Move Entities** tool is used to move entities chosen. An individual item or an entire object can be moved as shown in Figure 2-29.

1. Click the **Move** group on the **Command Manager**.
2. Click **Entities to Move** and Window the entities to be moved.
3. Uncheck the **Keep Relations**.
4. Click any point as starting point From Point Defined.
5. Drag object to another point and Click mouse at the End Point Defined.
6. When the move task is complete, click the OK check mark on the PropertyManager.

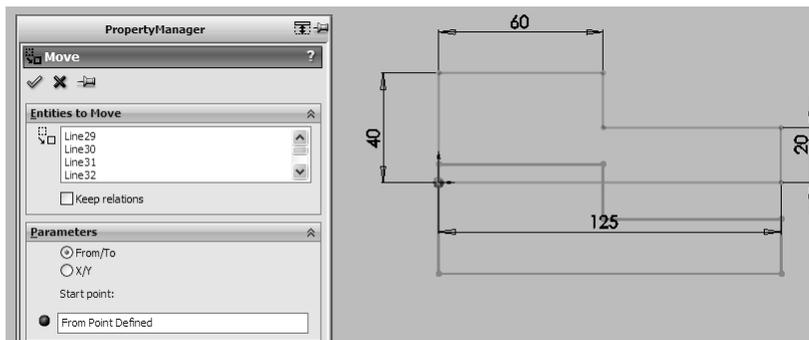


Figure 2-29 Move entities tool

Rotate Entities

The **Rotate Entities** tool is used to rotate entities chosen. This tool is illustrated in Figure 2-30.

1. Click the **Move** group on the **Command Manager**.
2. Click **Entities to Rotate** and Window the entities to be moved.
3. Uncheck the **Keep Relations**.
4. Click any point as Centre of Rotation.
5. Define the Angle of rotation.
6. When the rotation task is complete, click the OK check mark on the PropertyManager.

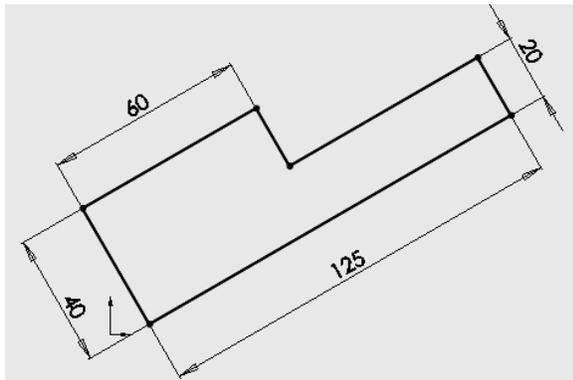
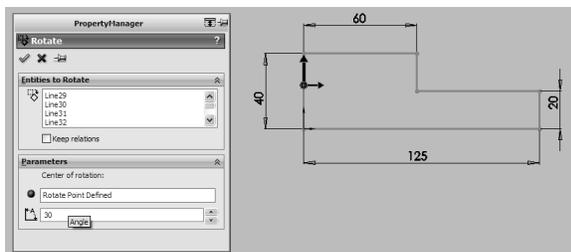


Figure 2-30 Rotate entities tool

Scale Entities

The **Scale Entities** tool is used to scale entities chosen. This tool is illustrated in Figure 2-31.

1. Click the **Scale** group on the **Command Manager**.
2. Click **Entities to Scale** and Window the entities to be copied.
3. Click any point as Scale Point Defined.
4. Define the scaling value.

- When the scale task is complete, click the OK check mark on the PropertyManager.

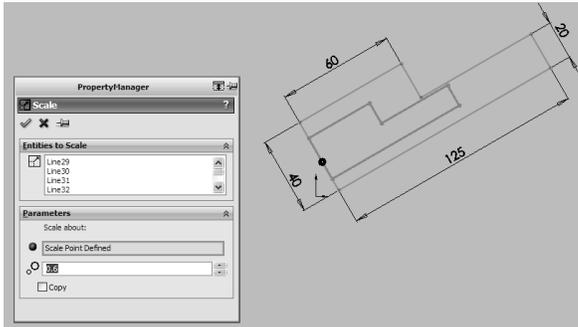


Figure 2-31 Scale entities tool

Copy Entities

The **Copy Entities** tool is used to copy entities chosen. An individual item or an entire object can be copied. This tool is illustrated in Figure 2-32.

- Click the **Copy** group on the **Command Manager**.
- Click **Entities to Copy** and Window the entities to be copied.
- Uncheck the **Keep Relations**.
- Click any point as starting point From Point Defined.
- Drag object to another point and Click mouse at the End Point Defined.
- When the copy task is complete, click the OK check mark on the PropertyManager.

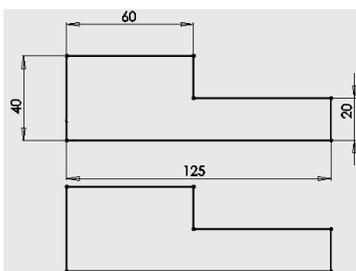
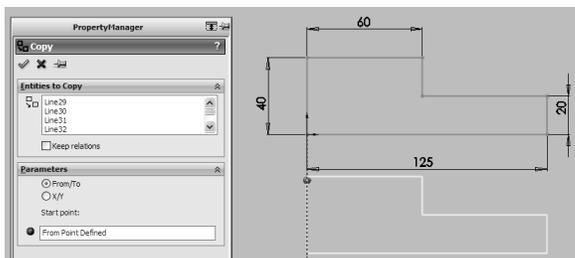


Figure 2-32 Copy entities tool

Pattern

The linear/circular pattern tool is used to create rectangular patterns based on a given model.

Linear Pattern

In linear pattern, directions for the patterns are linear. Let us consider an example as shown in Figure 2-33.

1. Sketch a geometry
2. Sketch the seed pattern.
3. Click the **Linear Pattern** tool.
The Linear Pattern Properties Manager appears.
4. Select the seed pattern.
5. For **Direction1**> select a horizontal edge for direction, 35 mm for Distance, and 4 for number of patterns. For **Direction2**> select a vertical edge for direction, 30 mm for Distance, and 3 for number of patterns.
6. Click OK to complete the patterns.

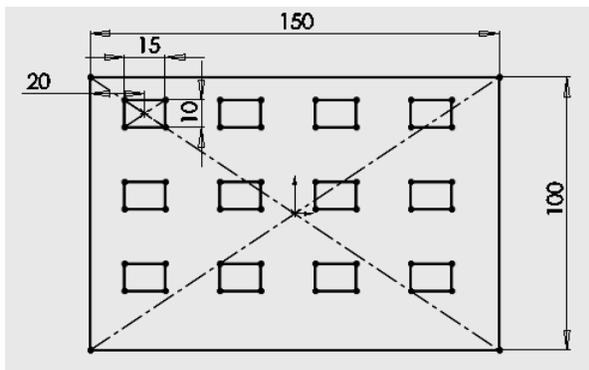
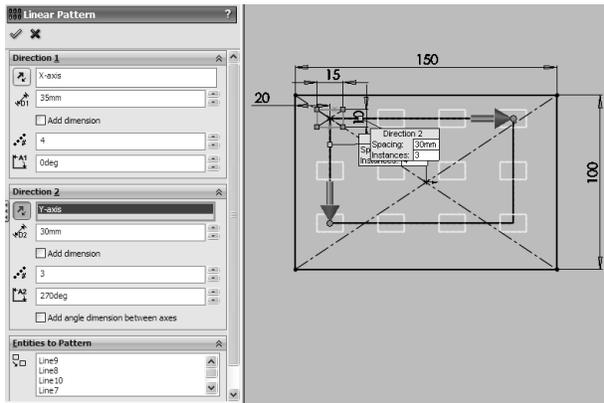


Figure 2-33 Linear patterns tool and patterns realized

Circular Pattern

In circular pattern, the patterns are created about an axis in a circular manner as shown in Figure 2-34.

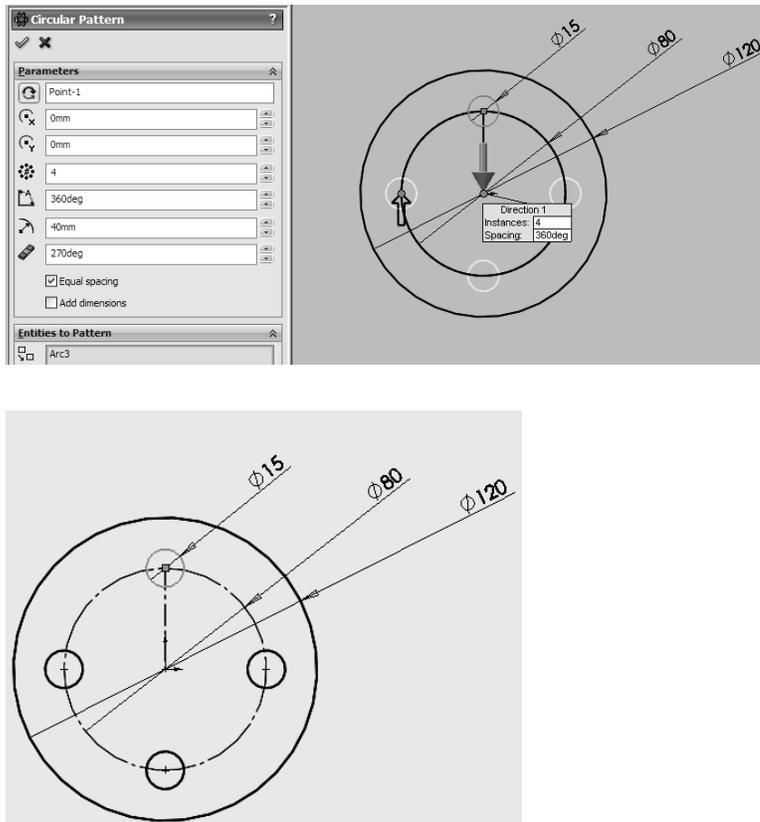


Figure 2-34 Circular pattern tool and patterns realized

Summary

This chapter discusses *sketch entities* and *tools* used in SolidWorks to facilitate the creation of parts. While *sketch entities* have specific geometries (line, rectangle, parallelogram, slot, polygon, circle, arc, ellipse, parabola, spline, etc.), some of the *sketch tools* (fillet, chamfer, offset, convert entities, intersection curves, trim, extend, split, jog line, constructive geometry, mirror, stretch, move, rotate, scale, copy, pattern, etc.) are used to modify the shapes of sketch entities. The *sketch entities* and *sketch tools* have been applied to a number of examples so that users can understand the principles involved.

- Use the given dimensions to redraw the given shape in Figure P-3. Create part models of the objects. Thickness = 20 mm

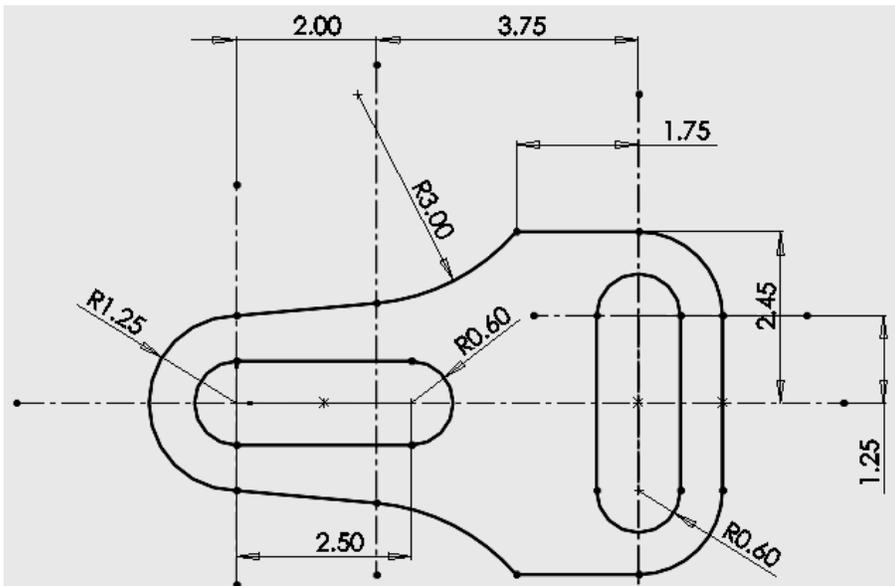


Figure P-3

- Use the information in the ellipse section (refer to Figure 2-10) to create the rugby ball model shown in Figure P-4.

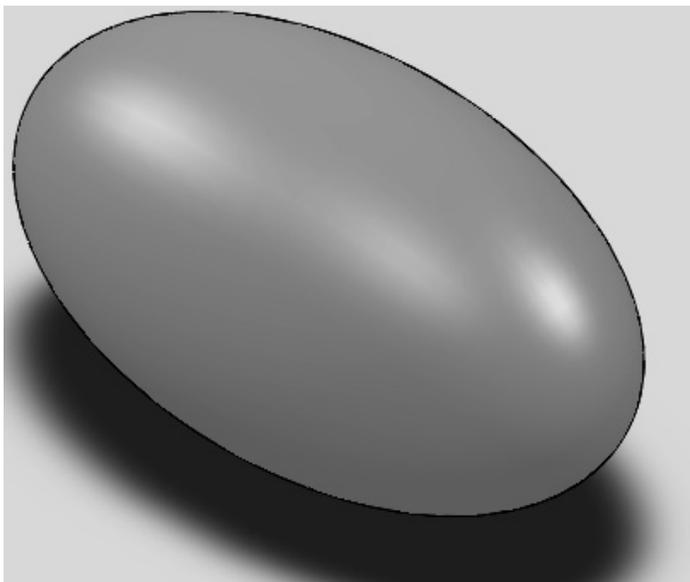


Figure P-4 Rugby ball

5. Use the information in the parabola section (refer to Figure 2-12) to create the satellite model shown in Figure P-5.

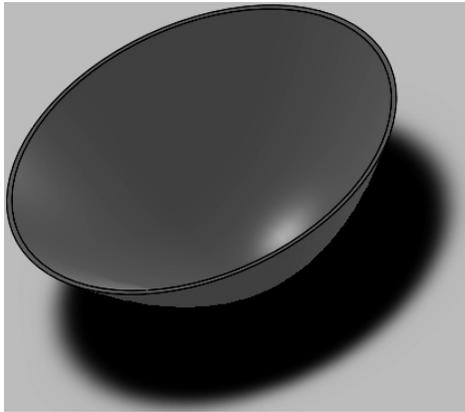


Figure P-5 Satellite dish

6. Use the information in the Convert Entities section to create the cut-out cylinder model shown in Figure P-6; start with a cylinder of 20 mm diameter and 50mm long.

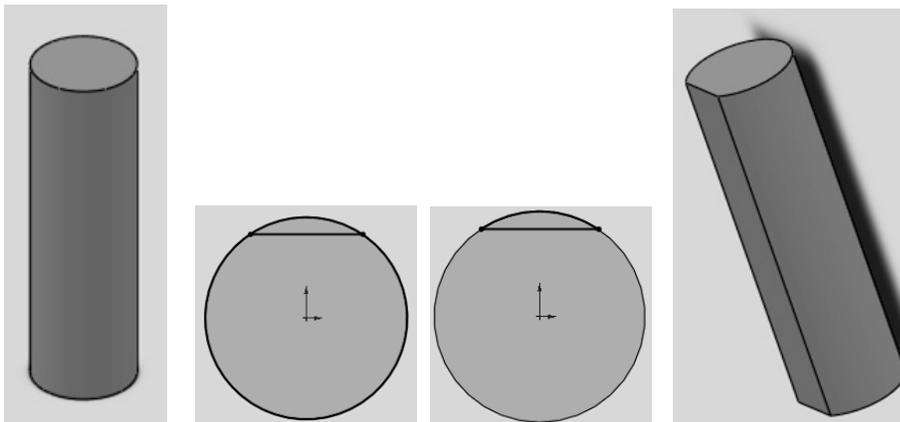


Figure P-6 Cut-out cylinder

7. Create the sketch shown in Figure 2-2 (line entities) and extrude it 10 mm Mid-Plane.
8. Create the sketch shown in Figure 2-3 (rectangle entities) and extrude it 10 mm.
9. Create the sketch shown in Figure 2-4 (parallelogram entities) and extrude it 10 mm.
10. Create the sketch shown in Figure 2-5 (slot entities) and extrude it 10 mm.
11. Create the sketch shown in Figure 2-6 (polygon entities) and extrude it 10 mm.
12. Create the sketch shown in Figure 2-9 Arc entities for sketch definitions