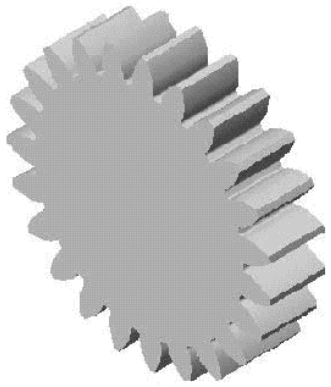
3.15.4 Entity Editing

Some CAD/CAM systems provide their users with specialized operations which enable them to edit existing entities in special ways. By using these operations, the user does not have to delete entities and re-create them. While some of these commands can be replaced by the already introduced editing operations, some cannot. Some CAD/CAM systems, for example, provide their users with operations to edit existing fillets or change existing circles. A user can change the radius or diameter of an existing circle. These operations are useful, and you should investigate your own system for some of these useful commands.

3.16 Tutorials

3.16.1 Create a Spur Gear

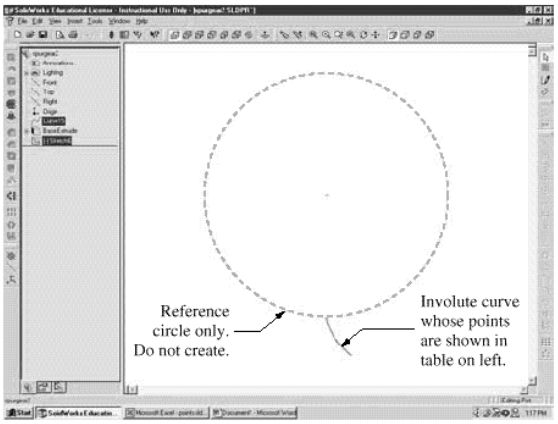
This tutorial shows how to create a spur gear as an example of a circular array. The gear teeth form the circular array. We create one tooth using involute curves. We use the tooth to generate the array. The gear model is a 2½D uniform thickness model. After we create the front face with the teeth, we extrude it a thickness of 25 mm. All dimensions are in mm. The screenshot on the right shows the final model.



Modeling strategy:

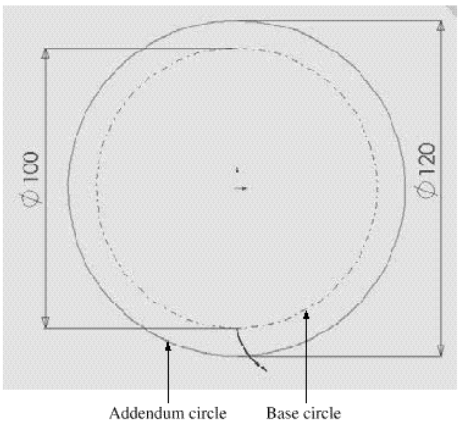
1. **Create the tooth involute curve.** Using the `Front` sketch plane, create a spline curve through free points and enter the points shown in the table below, to generate the curve as shown.

x	*y*	*z*
0	−50	0
0.011068	−50.19	0
0.08834	−50.7558	0
0.297012	−51.6842	0
0.700275	−52.954	0
1.358341	−54.5355	0
2.327502	−56.3912	0
3.659237	−58.4765	0
5.399372	−60.7397	0
7.587302	−63.1233	0
10.2553	−65.5644	0

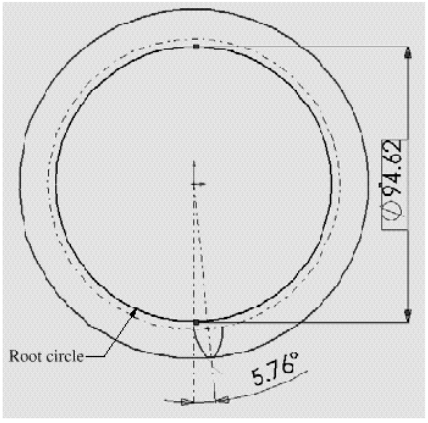


2. **Create the gear base and addendum circles.** Sketch the two circles in the `Front` sketch plane as shown below. Also trim the involute curve to the addendum circle.
3. **Create the tooths other curve and the gear root circle.** Mirror the trimmed curve from step 2 with respect to the center line shown in the screenshot below. Also create the root circle shown below.

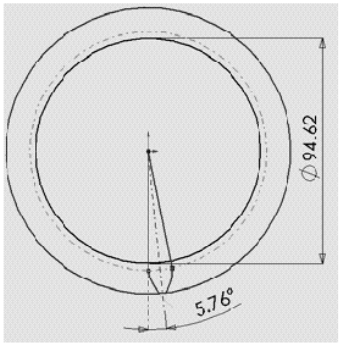
Step 2

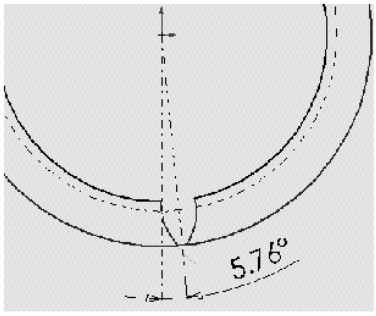


Step 3

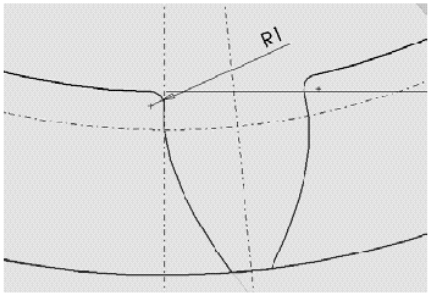


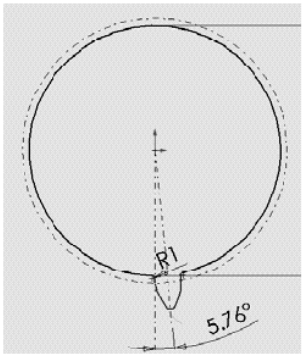
4. **Create the tooth shoulder and trim base circle.** Draw the two radial lines to join the end points of the involute curves with the origin. Trim the radial lines and the root circle to obtain the profile shown below.



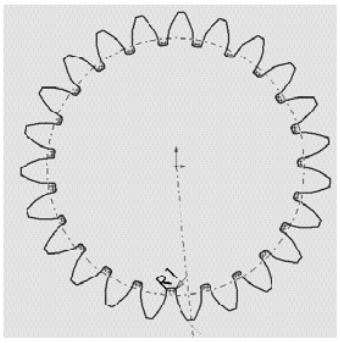


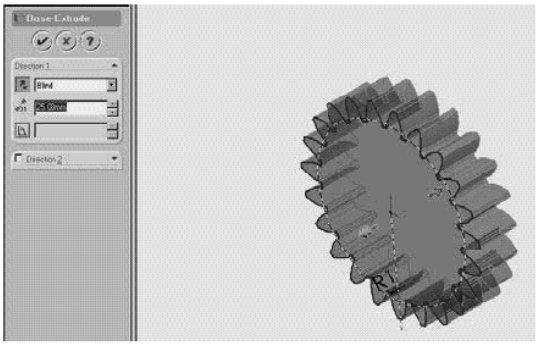
5. **Fillet the corners of the tooth shoulder and trim circles**. Zoom into the profile and fillet the edges shown. Use a fillet radius of 1 mm. Trim the outer circle to obtain the tooth profile.





6. **Create the circular array.** Using the circular pattern option, array the tooth profile about the center of the gear. Using the number of teeth as 23, create the cross section of the gear. Trim the root circle at appropriate locations to get the final gear profile.
7. **Extrude the gear cross section.** Extrude the profile to get the solid model of the spur gear. Extrusion thickness is 25 mm.





Tutorial 3.16.1 discussion:

The creation of the involute curve uses the involute equations. Any point on the involute has the following coordinates:

$$\cdot = \cdot \sin - \cos \quad {}_{0} \quad \ldots \tag{3.1}$$

$$\cdot = -\cdot \cos + \sin \quad {}_{0} \quad \ldots \tag{3.2}$$

We use $= 0$ and $\ldots = 50$. We use an increment of $= 5$ degrees to generate points on the involute curve. The maximum value, 50, of is chosen to ensure that the involute curve clears the addendum circle. Note that we can write a C or Java program that uses Eqs. (3.1) and (3.2) to generate the table shown at the beginning of the tutorial. The program saves the points in

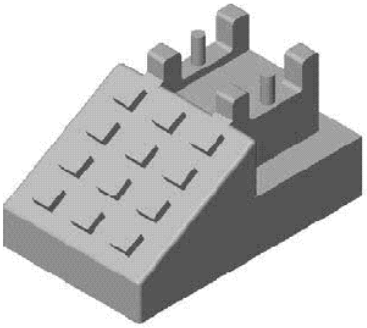
a file that we can open to read from in a CAD system, such as SolidWorks. C trigonometric functions require the angles to be in radians and not degrees.

Tutorial 3.16.1 hands-on exercise:

Change the addendum, the base, and the root circles. This changes the number of teeth. Find out the new correct number by working out the geometry and using your CAD system.

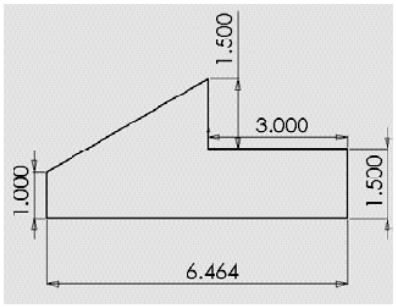
3.16.2 Create a Telephone

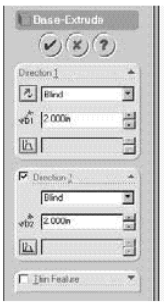
This tutorial shows how to create a telephone as an example of a rectangular array. The phone keys on the keypad form the rectangular array. We create one key and use it to generate the array. The phone model is a 2½D composite model. All dimensions are in inches. The screenshot on the right shows the final model.

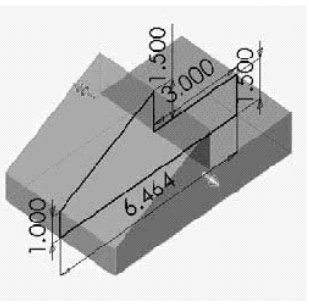


Modeling strategy:

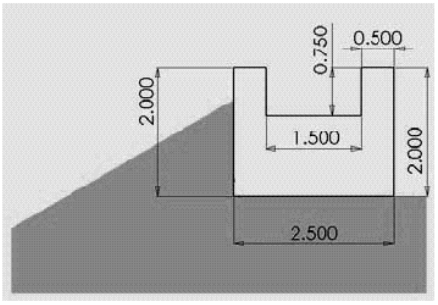
1. **Extrude the base profile.** Select the `Right` sketch plane and sketch the following profile. Extrude the profile a distance of 4 inches.

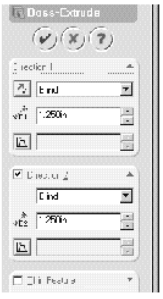


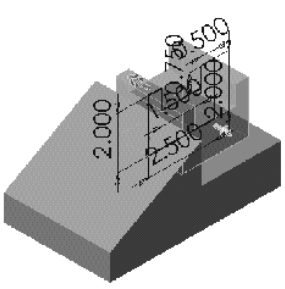




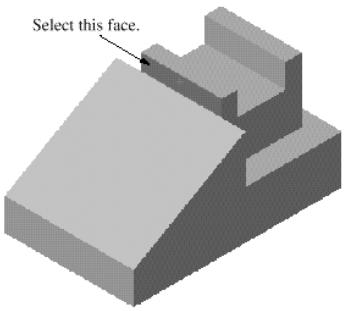
2. **Extrude the receiver holder.** Select the `Right` sketch plane and sketch the following profile. Extrude the profile a distance of 2.5 inches.

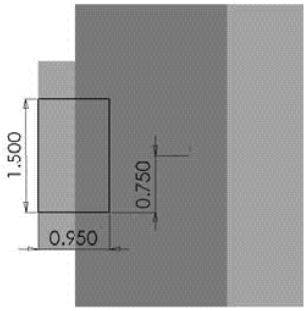




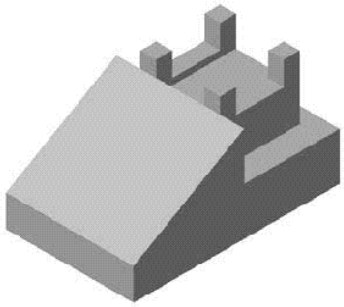


3. **Create the cut in the receiver holder.** Select the face of the receiver holder shown in the image on the left below as the sketch plane and draw a rectangle as in the image on the right below.

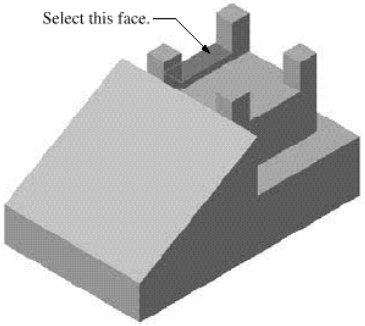


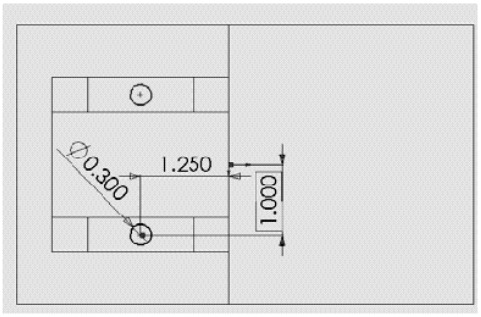


Now using the extruded-cut operation, cut the receiver holder using the parameters in the image on the left below to obtain the model as in the image on the right below.

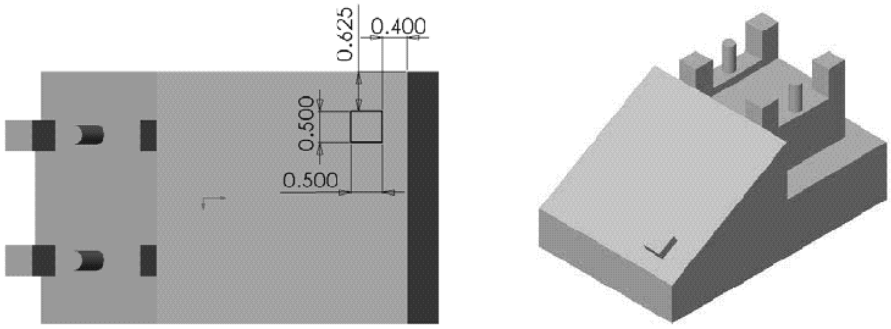


4. **Create the disconnectors.** Select the face shown in the image below as the sketch plane and sketch the two circles as shown below. Extrude these circles to a height of 0.7 inches.

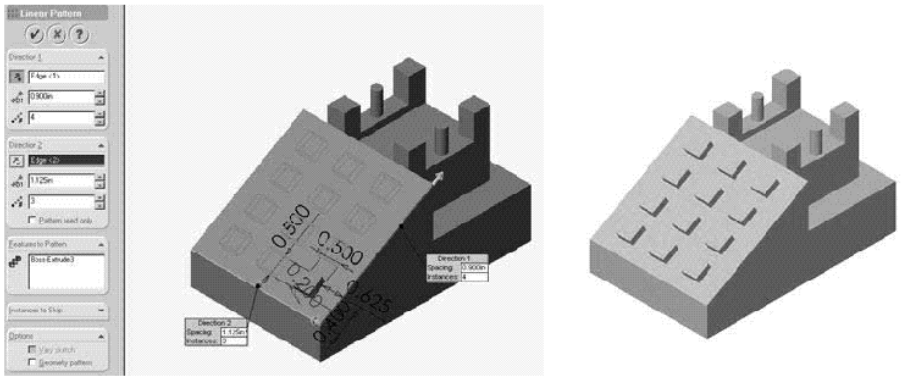




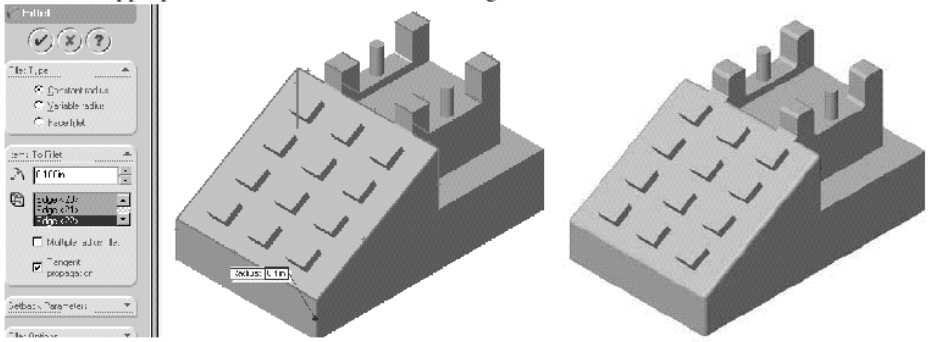
5. **Create the buttons.** Select the inclined plane of the model as the sketch plane and sketch the square as shown in the image below. Extrude this square profile of the button to a height of 0.2" to obtain the button as in the image below.



Now select the `Linear Pattern` command and set the parameters as in the image below. Finally, create the pattern. The phone model should appear as in the image below.



6. **Fillet the edges.** Use a 3D fillet operation and select the edges in the image below to fillet with appropriate radii also shown in the image. The final model is shown below.



Tutorial 3.16.2 discussion:

Step 5 shows how we can create a rectangular array. First, we create the base unit, a square, and extrude it. Second, we pattern the extruded feature in a 3 4 array.

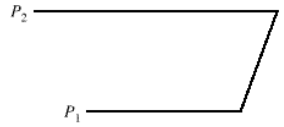
Tutorial 3.16.2 hands-on exercise:

Change the square cross section into a circle cross section, and regenerate the buttons. Also, add fillets to the other phone edges.

PROBLEMS

Part I: Theory

3.1 A user uses a line operation and clicks close to end points · 1 and · 2 to close the shown polygon. When the user utilized a later operation to calculate the area enclosed by the polygon, the system issued an error message that the polygon was not closed. How is this possible, and how can the user solve the problem?



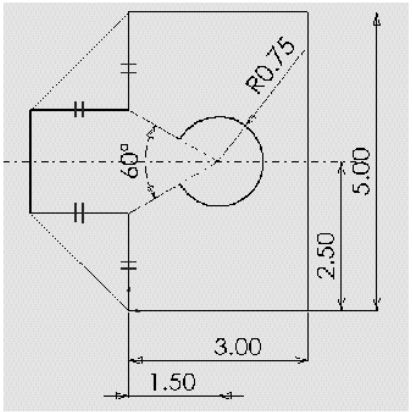
3.2 Why does circle trimming follow the counterclockwise rule?

3.3 Give some examples where the layering concept is useful.

3.4 Give some examples where the grid concept is useful.

3.5 Give some examples where the offsetting concept is useful.

3.6 How can you use the geometric modifiers to create the following sketch without having to calculate any other dimensions explicitly:



Part II: Laboratory

Use your in-house CAD/CAM system to answer the questions in this part.

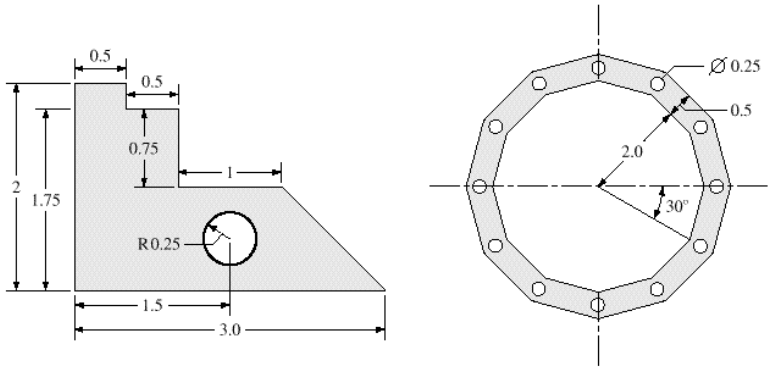
3.7 Find the geometric modifiers available on your CAD/CAM system. What is their syntax? How do you use them? What is the default modifier?

3.8 Find all the layer-related commands on your system, specifically how to select/deselect layers, assign entities to layers, assign layers to entities, assign colors to layers, modify layer colors, and modify layers of existing entities.

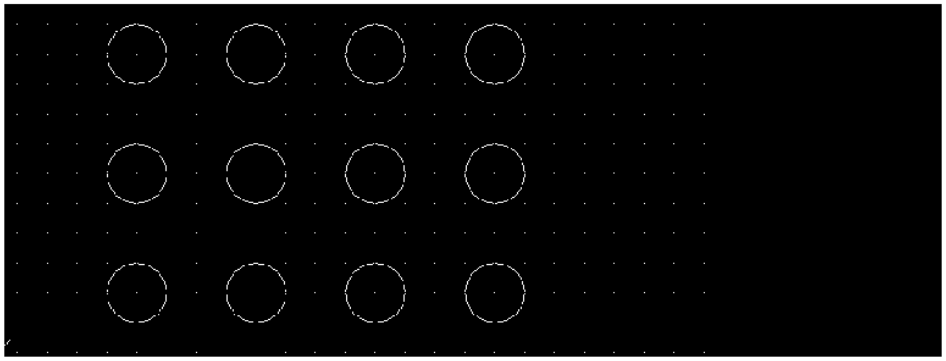
3.9 For the model shown in problem 3.6, create the straight lines on layer 10, the circle on layer 20, and the dimensions on layer 30.

3.10 Find the commands related to rectangular and radial grids. How can you use them? Investigate their related modifiers.

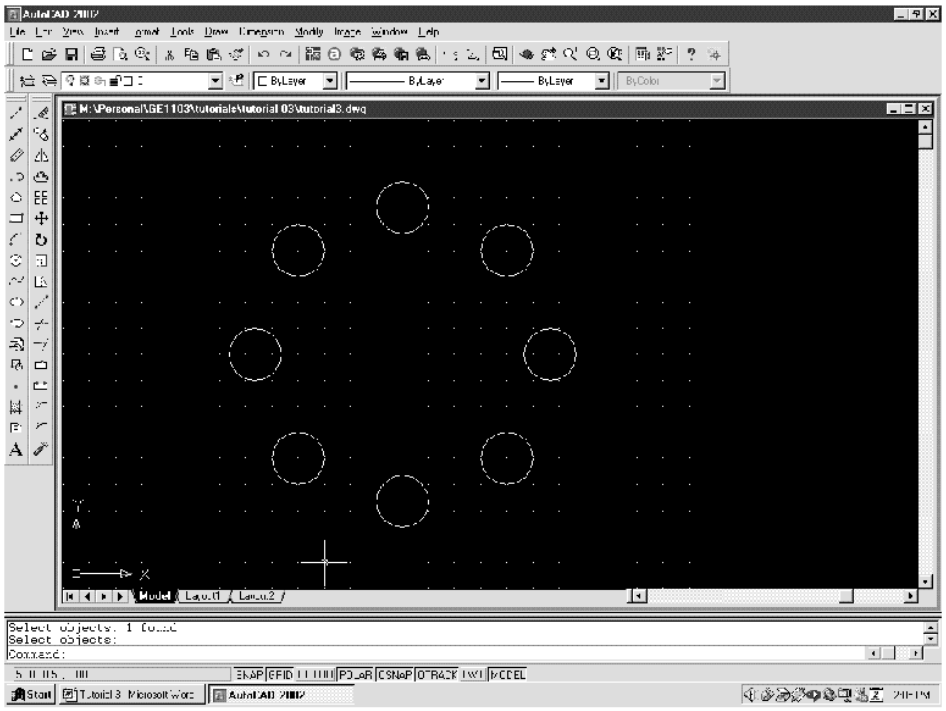
3.11 With the aid of grids, sketch the following geometry:



3.12 Create the following 4 4 rectangular array. Use a spacing of 1 in. in both directions.



3.13 Create the following circular (polar) array that sweeps a 360-degree angle and has a center at · · = 4 2 .



3.14 Modify Tutorial 3.16.1 to create a helical gear. The gear thickness is 25 mm. The gear helix angle is 15 degrees. Use Eqs. (3.1) and (3.2) to generate points on the involute curve. Use $_0 = 0$, · · · = 75, and = 5. Use a total of 16 points on the curve. Use 18 teeth for the gear. Offset the two cross sections of the gear by an angle of 4.08 degrees. The gear dimensions are shown in the screenshots below.