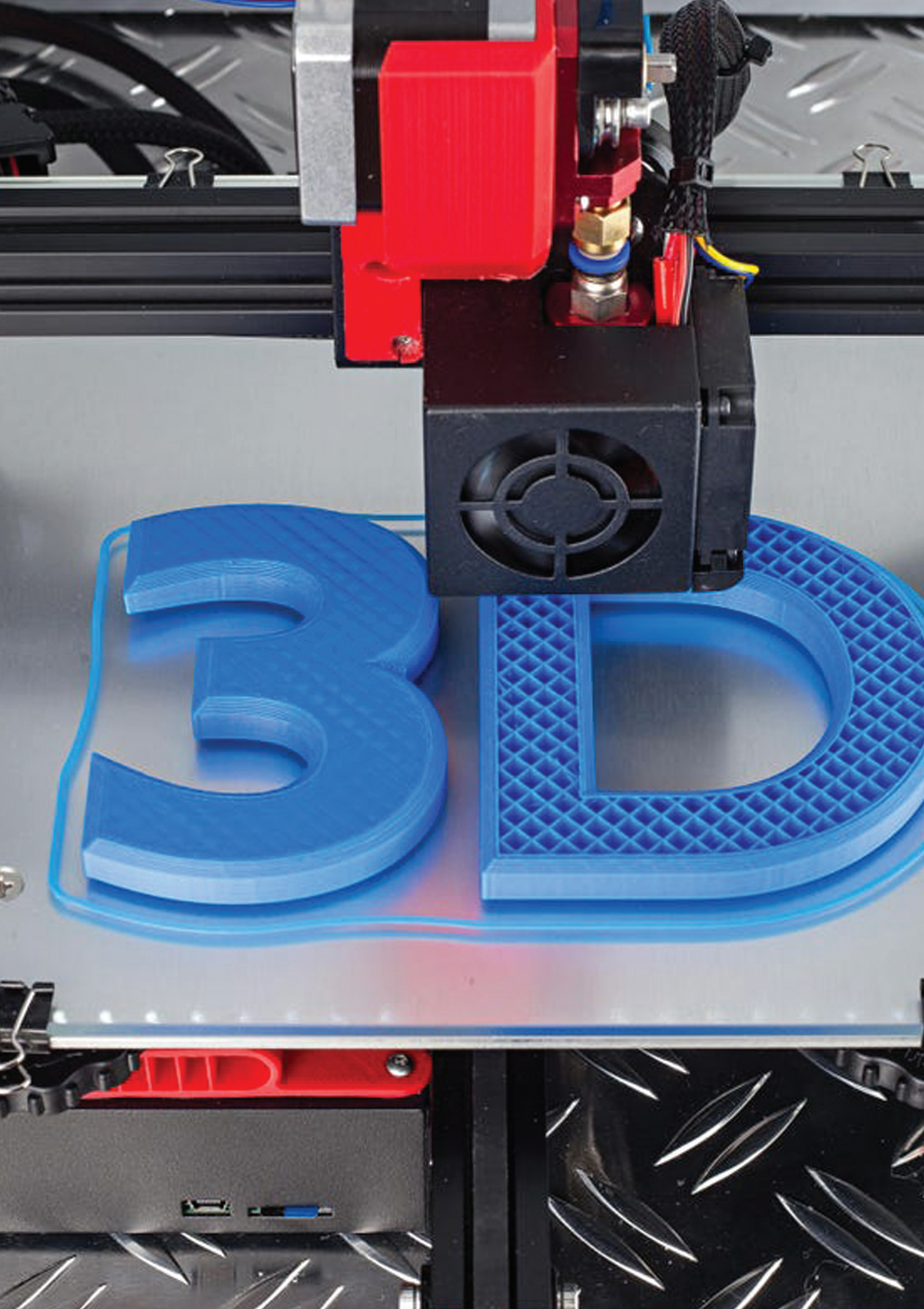
ROBOTROAD
by Olatus Systems Private Limited

VOLUME 5

3-D PRINTING


3D

Contents

CHAPTER 1: INTRODUCTION TO 3D PRINTER

PRINT BED

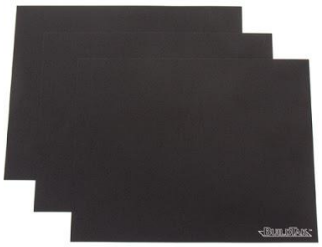
The print bed is the surface that your objects are printed on to. Typically, it will consist of a sheet of glass, a heating element, and some kind of surface on top to help the plastic stick.

HEATED/NON-HEATED

Most print beds are heated in order to prevent the object from warping while it is being printed. Due to thermal contraction, the plastic will shrink slightly as it cools. This causes the object to warp upwards around the edges and peel off the bed. Heated beds keep the bottom of the object warm, in order to prevent this. See also; Enclosure, Bed Surfaces.

Some printers do not have heated beds. This limits them to printing a narrow range of materials including mainly PLA (the material that is least prone to warping) and sometimes PET.

BED SURFACES

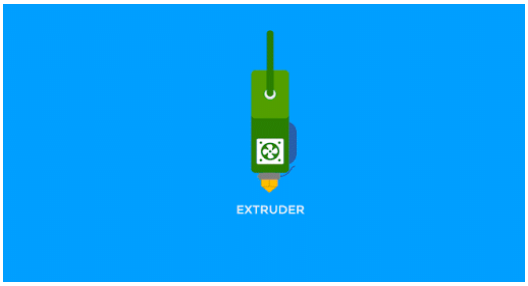


The bed surface helps the plastic stick to the bed during printing but also allows it to be removed easily when printing is done. There are many different kinds of bed surfaces. Most printers will come with some kind of all-purpose surfaces, like BuildTak or PEI film. However, for best results, you will want to use different surfaces depending on the material you are printing. Use this guide for print bed recommendations based on the material.

FILAMENT

This is the plastic that's consumed by the printer. It comes on a spool. Printers use two different sizes of the filament, 1.75 mm and 3 mm. There are a variety of different materials. To learn more about them, check out the Matter Hackers filament guide.

EXTRUDER



The extruder is the core of the printer. It is where the plastic gets drawn in, melted, and pushed out. It is essentially a fancy hot glue gun. It is small, but it is where most of the printer's technology is located. The extruder consists of two parts; the hot end and the cold end. The cold end has a motor that draws the filament in and pushes it through. The hot end is where the filament gets melted and squirted out.

DIRECT DRIVE VS BOWDEN EXTRUDERS

- On **direct drive** printers, the hot end and cold end are connected together, one on top of the other. The filament goes straight down through the cold end and into the hot end.
- With a Bowden setup, the hot end and cold end are separated. The cold end will be stationary and bolted somewhere onto the printer's frame. The filament is pushed through a long tube (called a Bowden tube) to the hot end. This means that the printer has less weight to move around.

HOBBED GEAR

This gear bites the filament and pushes it down through the hot end.

IDLER GEAR

The idler is a spring-loaded wheel that pushes the filament up against the hobbed gear. Most printers have a way to adjust the tension on the idler so that it neither squeezes the filament too hard or too little.

HOT END - ALL METAL VS PEEK/PTFE

By not using any plastic insulators in their construction, all metal hot ends are able to reach much higher temperatures and print a wider range of materials. However, they require active cooling.

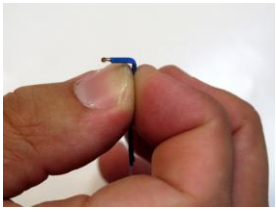
HOT END - HEAT SINK / HOT END FAN

This ensures that heat does not travel up the plastic and melt it prematurely before it reaches the nozzle. This phenomenon is called heat creep and it causes jams, especially with PLA. This fan should be running whenever the hot end is warm.

HEATER CARTRIDGE

The heater cartridge is pretty self-explanatory. It heats the plastic. It is simply a high-power resistor. Almost all modern printers use cartridge heaters, but many older printers used coils of nichrome wire (like the kind in a toaster). If you are replacing your heater cartridge, of even your entire hot end, make sure you know if your system is running 12v or 24v.

THERMISTOR/THERMOCOUPLE/RTD



These are all various types of sensors for determining the temperature of the hot end. They are essentially electronic thermometers. Thermistors are the most common type of sensor, but some printers will use thermocouples for extremely high-temperature printing.

NOZZLE

The nozzle is simply a piece with a small hole for the melted filament to come out of. Nozzles are interchangeable, and come in various sizes; 0.4 mm is normal, while you might use a smaller nozzle for finer detail or a larger nozzle to print faster. Nozzles can also sometimes get clogged. This is one of the most common issues with 3D printers. See this article for advice on unclogging your nozzle.

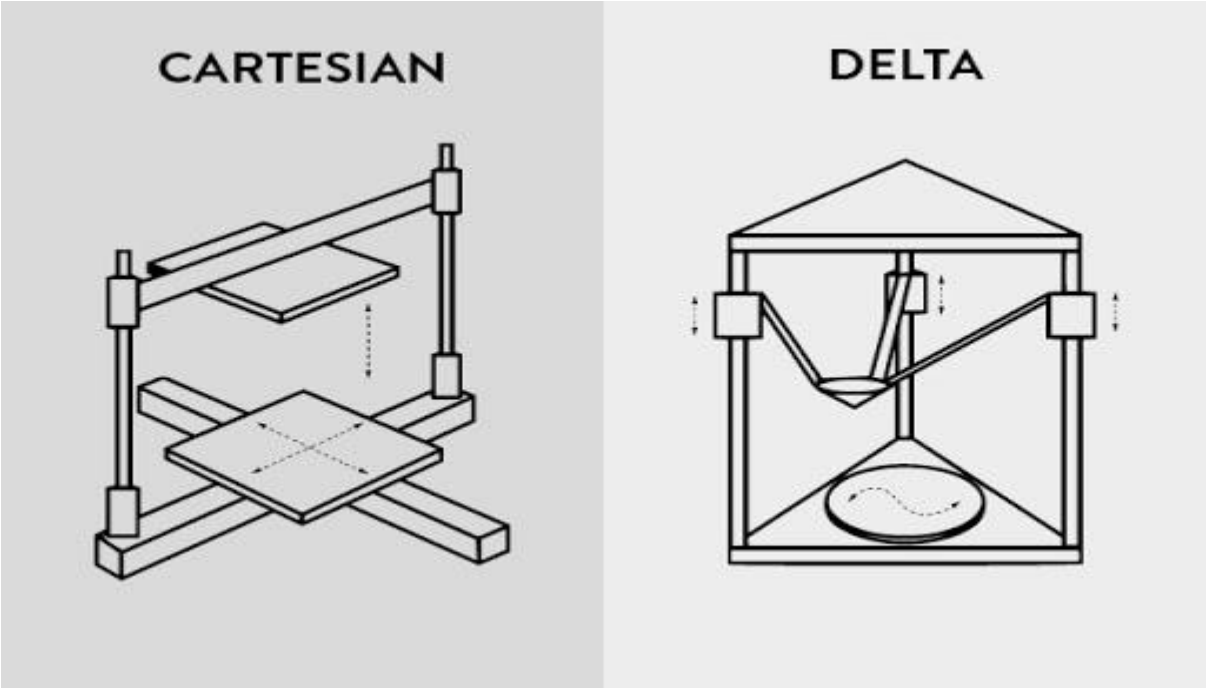
LAYER COOLING FAN

This fan cools off the plastic immediately after it is deposited by the nozzle. It helps the object hold its shape. The slicer will turn this fan on and off under different circumstances, depending on what material you are printing. It is not to be confused with the heat sink fan, which cools the hot end itself and not the printed object.

MOTION CONTROL - X, Y, Z-AXIS:

DELTA VS CARTESIAN

- Cartesian printers move one or two motors along each of the X, Y, and Z axes and the name was derived from the Cartesian coordinates system. They typically have a rectangular build area and the printers themselves tend to have a cube-like shape. The Lulzbot Mini is a fine example of these types of printers.
- Delta printers have three arms that come together in the center to suspend the extruder above the build area. Deltas also use a Cartesian coordinates system to move around in, but instead of moving one motor per axis at a time, all three arms move at different rates or times to precisely move the nozzle with triangulation. The See Me CNC Rostock MAX V2 is a prime example of a delta printer.

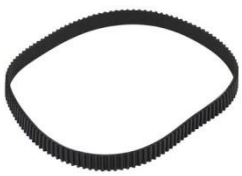


THREADED RODS / LEADSCREWS

These are usually used on the printer's Z axis. They rotate, thus forcing nuts to move up and down. Inexpensive printers will use simple threaded steel rods, which are essentially extra-long bolts. Higher quality printers have smooth chrome plated lead screws designed to minimize backlash.

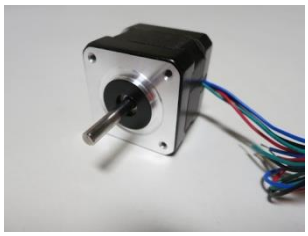
BELTS

Belts move things. The X and Y motors have sprockets that drive the belts. Most printers also have some way of adjusting the tension on the belts.



STEPPER MOTORS

Unlike regular DC motors, which rotate continuously when given power, stepper motors rotate in increments. This gives them precise control over their position. Most printers use NEMA 17 type motors with 200 increments (steps) per revolution.



FRAME

The frame holds everything together. Early printers had frames made out of laser cut plywood. Printers now have frames made of sheet metal, aluminum beams, or plastic. Many parts of the frame are often 3D printed themselves. The more rigid the frame, the more precise the printer's movement will be.

ENCLOSURE

Enclosures for 3D printing are used for safety. There are moving parts and heating elements that users will want to protect themselves from. If your printer does not offer an enclosure it is easy to construct your own. Something as simple as a cardboard box could suffice.

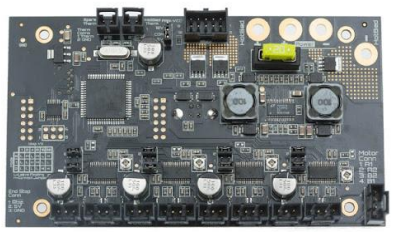
ELECTRICAL COMPONENTS:

POWER SUPPLY

- This takes the 120V AC electricity from the wall and converts it to low voltage DC power for your printer to use
- **ATX Power Supplies**- These are the same power supplies used in desktop computers. They have been repurposed for use in many printers. They are very beefy and efficient and have separate lines that provide power at a variety of voltage (12V, 5V, 3.3V).
- **Voltage** - some machines run 12 volt systems, while others run 24 volt systems. This becomes critical if you are going to replace components - especially your heater cartridge or hot end. Make sure you order the appropriate parts.

MOTHERBOARD

The motherboard is the brain of the printer. It takes the commands given to it by your computer (in the form of G-Code) and orchestrates their execution. The motherboard contains a microcontroller (essentially a tiny, self-contained computer) and all the circuitry needed for running the motors, reading the sensors, and talking to your computer. Here is a comparison of the different motherboards we carry.



STEPPER DRIVERS

These chips are responsible for running the stepper motors. They fire the coils of the motor in sequence, causing it to move in increments. Many motherboards have the stepper drivers built in, but some also have them in modules that can be unplugged. By balancing the power fed to each coil, the driver is also able to divide steps up into further increments. This is called micro stepping and allows more precise control over the motor than is normally possible. The stepper driver also controls how much electrical current is fed to the motor. More power makes the motor stronger, but also makes it run hotter. See this article for more information on adjusting your motor current.

USER INTERFACE

Some printers have an LCD screen so they can be controlled directly without hooking them up to a computer. These can be basic black and white displays like the VIKI 2 or advanced Wi-Fi-enabled touch screens like the Matter Control Touch.

SD CARD SLOT

Some printers also have an SD card slot from which they can load G-Code files. This allows them to run independently without a computer.

END STOPS (ONE FOR EACH AXIS)

The end stops are how the printer knows where it is. They are little switches that get pushed whenever an axis moves to the end. This is how the printer finds it's starting point before printing. Most printers use mechanical switches, but some are known to use optical sensors.

BED LEVELING

Many printers have some kind of a system for automatically making sure that the bed is level with the nozzle. Some do not, though, and must be calibrated by hand. Matter Control also has the ability to account for unevenness in software. For more information, see our wiki article on bed leveling.

CHAPTER 2: INTRODUCTION TO SOLIDWORKS

What is SolidWorks?

SolidWorks mechanical design automation software is a *feature-based*, *parametric solid modelling* design tool which takes advantage of the easy to learn Windows graphical user interface. You can create *fully associative* 3-D solid models with or without *constraints* while utilizing automatic or user defined relations to capture *design intent*.

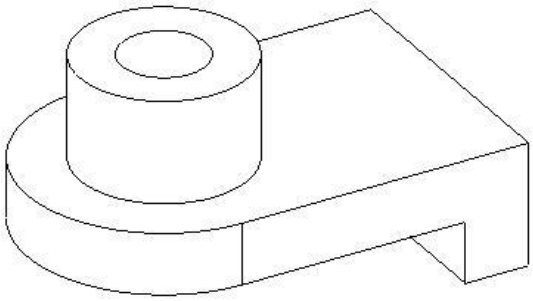
Feature-based

Just as an assembly is made up of a number of individual piece parts, a SolidWorks model is also made up of individual shapes/forms known as features. The SolidWorks interface allows near intuitive creation of geometric features such as bosses, cuts, holes, ribs, fillets, chamfers, and draft. As the features are created they are applied directly to the work piece. Features can be classified as either sketched or applied.

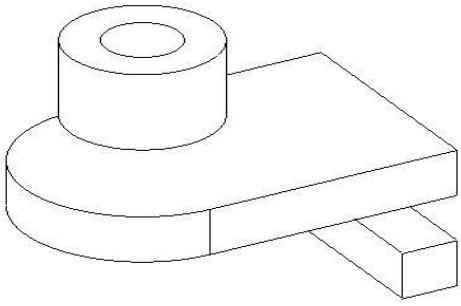
Sketched Features: One that is based upon a 2-D sketch – only this type can be used as a base feature.

Applied Features: Created directly on the solid model. Fillets and chamfers are examples of this type of feature.

To illustrate the concept of feature-based modelling, consider the part shown at the right:



This part can be visualized as a collection of several different features – some of which add material, like the cylindrical boss, and some which remove material, li ke the blind hole.



Parametric

The dimensions and relations used to create a feature are captured and stored in the model. This not only enables you to capture your design intent, it also allows you to quickly and easily make changes to the model.

Driving Dimensions

These are the dimensions used when creating a feature. They include the dimensions associated with the sketch geometry, as well as those associated with the feature itself. A simple example of this would be a feature like a cylindrical boss. The diameter of the boss is controlled by the diameter of the sketched circle. The height of the boss is controlled by the depth to which that circle was extruded when the feature was made.

Relations

These include such information as parallelism, tangency, and concentricity. Historically, this type of information has been communicated on drawings via feature control symbols. By capturing this in the sketch, SolidWorks enables the user to fully capture your design intent up front, in the model.

Solid Modelling

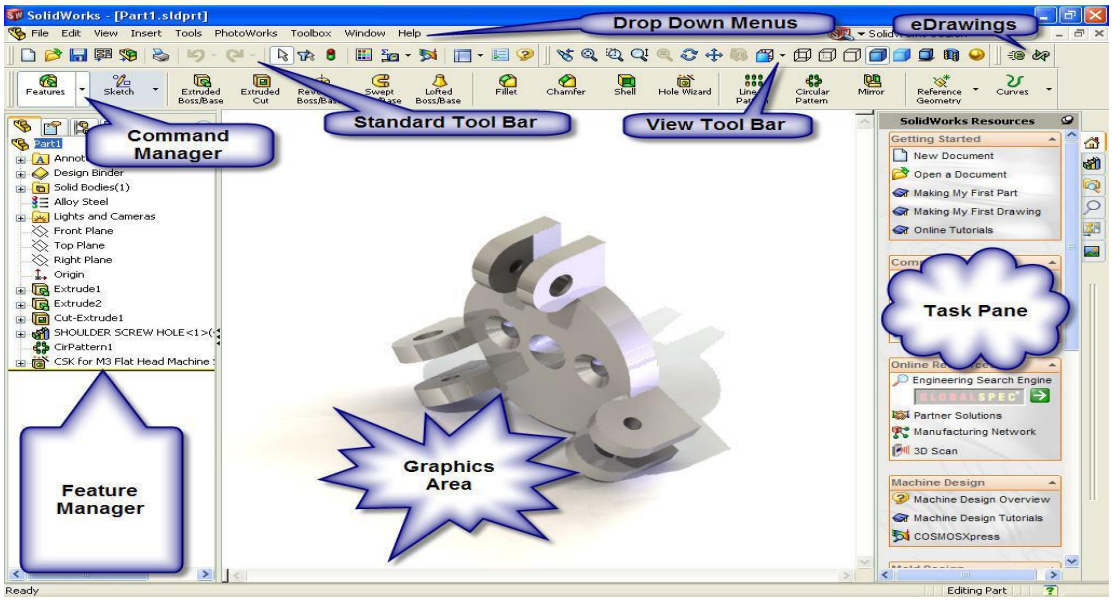
A solid model is the most complete type of geometric model used in CAD systems. It contains all the wire frame and surface geometry necessary to fully describe the edges and faces of the model. In addition to the geometric information, it has the information called topology that relates the geometry together. An example of topology would be which faces (surfaces) meet at which edge (curve). This intelligence makes operations such a filleting as easy as selecting an edge and specifying a radius.

Fully Associative

A SolidWorks model is fully associative to the drawings and assemblies that reference it. Changes to the model are automatically reflected in the associated drawings and assemblies. Likewise, you can make changes in the context of the drawing or assembly and know that those changes will be reflected back in the model.

SOLIDWORKS USER INTERFACE

First, In Solidworks we are going to review the user interface or the SolidWorks window. To begin, using the left mouse button double click on the SolidWorks icon on the PC desktop screen to open the program. If there is no icon visible the program can be started through Start - All Programs – SolidWorks 200X. You can access commands in SolidWorks using menus, toolbars and the mouse. The SolidWorks interface is dynamic in that different toolbars and menus appear depending on the active document type.



Document Windows

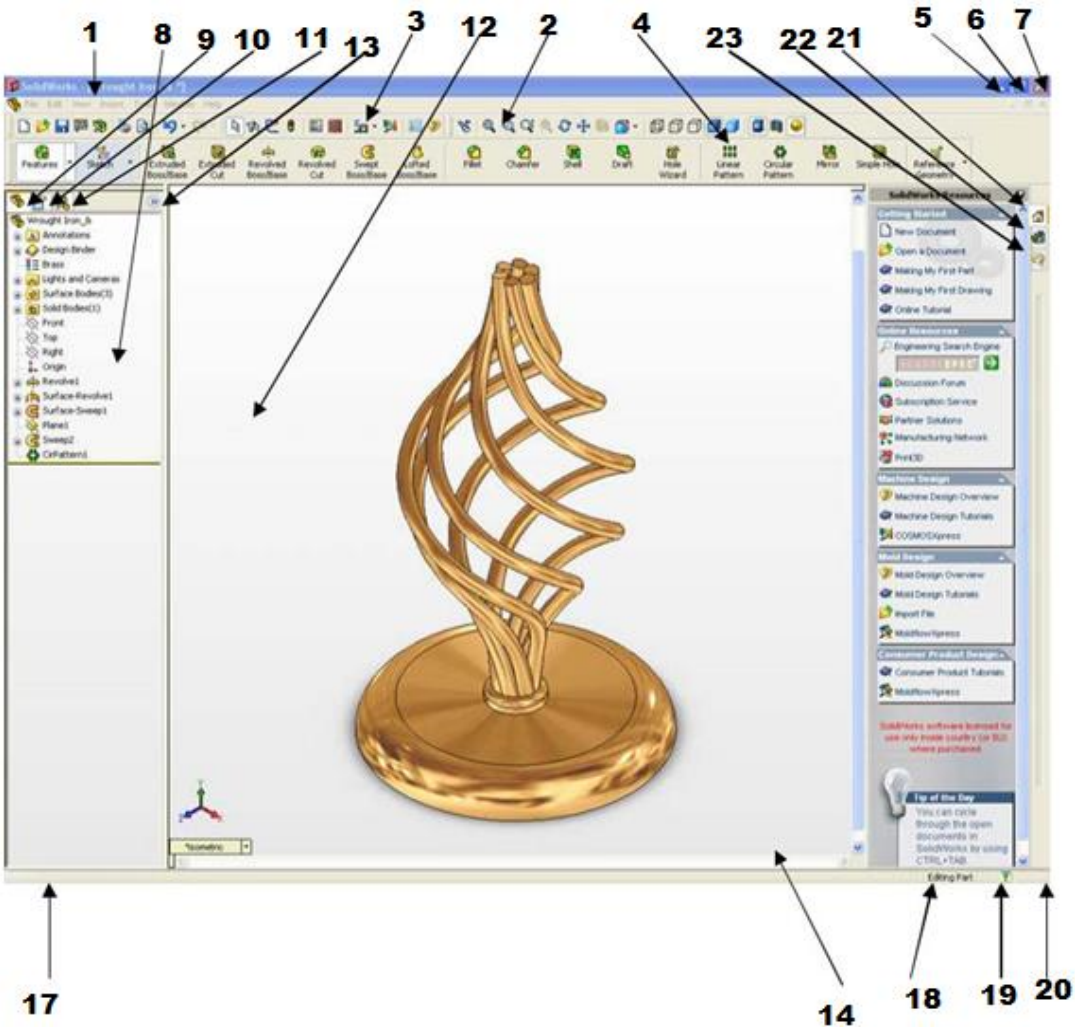
The format of the SolidWorks window reflects that of windows itself. The same is true for any SolidWorks document. Once opened a document appears split into two panels. The right is the graphics window, where your model or drawing appears. You can create and manipulate the document in the graphics window. The left panel contains the following SolidWorks document windows:

FeatureManager design tree: Similar to the windows explorer tree it lists the structure of the part, assembly or drawing.

PropertyManager: Appears in the left panel when you select many of the SolidWorks commands such as sketches, fillet features and so on. The PropertyManager displays selection icons to enter relevant command options and boxes/fields to enable the user to enter relevant design and data parameters.

ConfigurationManager: Appears in place of the FeatureManager design tree. Helps create, select and view multiple configurations (variations of parts and assemblies in a single document).

The Solidworks Window



The Main elements of the SolidWorks user interface are as follows (see Figure above for the corresponding item numbers):

1 Title bar: Displays the name of the active document and active document window with a blue (default color) title bar. Inactive document windows are shown with a grey title bar. If you haven't saved any changes to a document, you see a * after the document name.

2 Main Menu: A set of drop down menus (File, Edit, View, and so on) across the top of the user interface. The menu bar contents are task dependent based on the active document type. SolidWorks toolbars display these functions whereas the menu bar contains the complete set.

3 Standard toolbar: Found just beneath the main menu this toolbar is consists of a set of the most commonly used command buttons.

4 View toolbar: Features a series of commonly used command buttons that allow you to zoom, rotate and view the part in different orientations.

5 Minimize window: Shrinks the document window.

6 Maximize window: Enlarges the viewing window to full-size.

7 Close window: Closes solid works. If you have made any changes to you documents, SolidWorks prompts you to save the document.

8 Command Manager: A dynamic toolbar that lists the command buttons for the type of document you are working on.

9 FeatureManager design tree tab: Displays the FeatureManager design tree.

10 PropertyManager: Appears in the left panel when you select many of the SolidWorks commands such as sketches, fillet features and so on. The PropertyManager displays selection icons to enter relevant command options and boxes/fields to enable the user to enter relevant design and data parameters.

11 ConfigurationManager tab: Appears in place of the FeatureManager design tree. Helps create, select and view multiple configurations (variations of parts and assemblies in a single document).

12 FeatureManager design tree: Similar to the windows explorer tree it lists the structure of the part, assembly or drawing.

13 Show display pane: Expands or collapses the display pane.

14 Graphics area: Displays the part assembly or drawing

15 Pointer: Indicates the position of the mouse and lets you select items within the user interface.

16 Tool tip: A pop up informational message about a feature or function. It appears when you hover the pointer over an object. It disappears after a few seconds.

17 Status bar: Gives more complex explanation of the selected function.

18 Status bar: Indicates whether it is a drawing, part or assembly that you are editing.

19 Quick tips help: Indicates with a question mark button whether Quick Tips is on or off. Click the icon to toggle

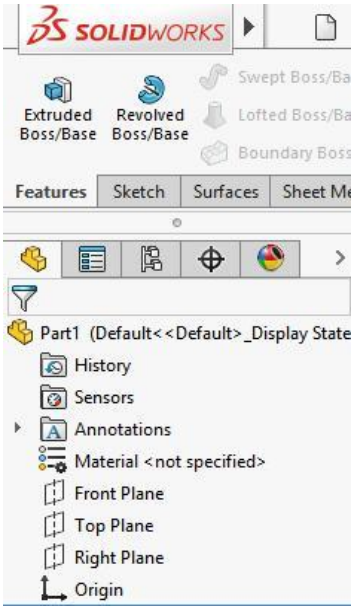
20 Resize window: Enables resizing (by clicking and dragging) the window if it isn't already maximized.

21 SolidWorks resources: Click to open the SolidWorks Resources tab, which contains links to resources, tutorials, tips of the day and also command buttons to open or create SolidWorks documents.

22 Design Library: Click to open the design library. Inside you see the Design Library, Toolbox and 3D Content Central each which contain many standard design elements you can drag and drop into your design.

23 File Explorer: Duplicates Windows Explorer in your computer. Lists recently opened documents and currently opened documents. You can drag documents from here into the graphics area.

The FeatureManager Design Tree



On the left side of the SolidWorks document window the FeatureManager design tree provides Windows Explorer-type selection and editing access to all the entities in your active document. The type of entities the design tree is populated varies depending on the active document type from features sketches for part documents to drawing views for a drawing parts/subassemblies in an assembly.

Key advantages offered by the FeatureManager design tree:

- **Shows the order in which the elements were created with the oldest at the top**
- **Links to the graphics pane**
    - Allows selection/highlighting of an object by clicking/hovering the pointer over the appropriate element in either the design tree or graphics area.
- **Displays graphical feedback that displays feature or component characteristics:**
    - For Example, if the component is suppressed, it appears in grey.
- **Allows you to see the contents of the folders in the tree:**
    - You can click + or – to maximize or minimize the folder.
- **Gives you access to quick functions when you right click:**
    - The functions displayed will depend on the object and document type.

The Command Manager

Located just below the standard tool bar in the top left-hand corner of the document window, is the CommandManager, a smart toolbar that displays the menus you need for the task at hand. The CommandManager is divided into two areas: the control area made up a selection of buttons any of which selected displays a toolbar in the toolbar area (directly to the right of the control area as shown above). For example, if you select the Features button, the Feature commands display in the toolbar area (Extruded base/boss, extruded cut and so on).

The CommandManager makes efficient use of the space onscreen by embedding a number of toolbars in just one. As a result, this minimizes the number of menu picks, mouse movements and overall allows the user to work faster and more efficiently.

Command Manager

VIEW AND ORIENTATION

A set of predefined views can be selected through the Standard view toolbar, a flyout (similar to a drop-down Menu) tool bar that is embedded in the View toolbar. If you click the Standard View button in the View toolbar, you get a pull-down menu of several commands that represent standard engineering views. This enables the user to select views such as Side, Front, Top, Right and Bottom as well as perspectives: Isometric, Trimetric and Dimetric which vary in viewing.

Orientation

The orientation section of the View toolbar offers several tools to enable the user to manually manipulate the orientation of a model. These manipulation tools include the following:


Zoom to Fit: Zooms in or out so the entire model is visible.


Zoom to Area: Zooms in on a portion of the view that you select by dragging a bounding box. The centre of the box is marked with plus (+) sign.


Zoom In/Out: Zooms in as you press and hold the left mouse button and drag the mouse up. Zooms out as you drag the mouse down.


Zoom to Selection: Zooms to the size of a selected entity.


Rotate View: Rotates the view as you press and hold the left mouse button and drag the mouse around the screen.


Pan View: Scrolls the view so the model moves as you drag the mouse.

Display Modes

The third section of the View toolbar offers the following display modes for model and drawing view in drawing documents. These tools include the following:


Shaded


Shaded with Edges

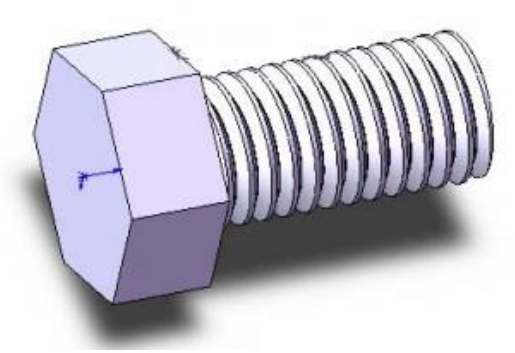

Hidden Lines Removed


Hidden Lines Visible

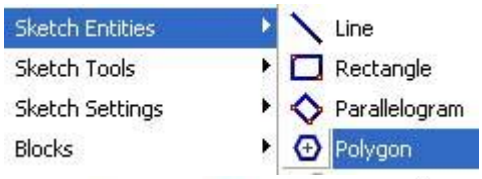

Wireframe (Displays all edges of the model)

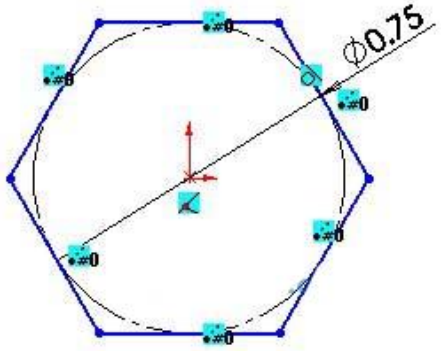

Section View

CHAPTER 3: HOW TO CREATE A HEX BOLT

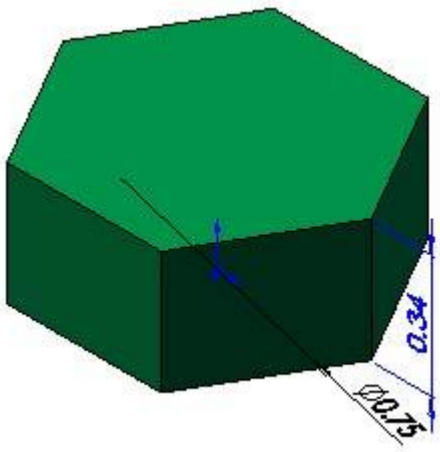


1. Sketch a polygon with 6 sides, Tools>Sketch Entities>Polygonset diameter to 0.75in.

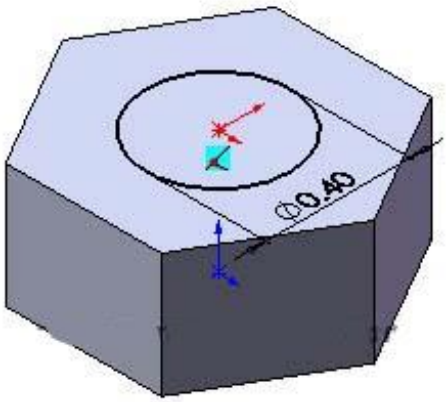




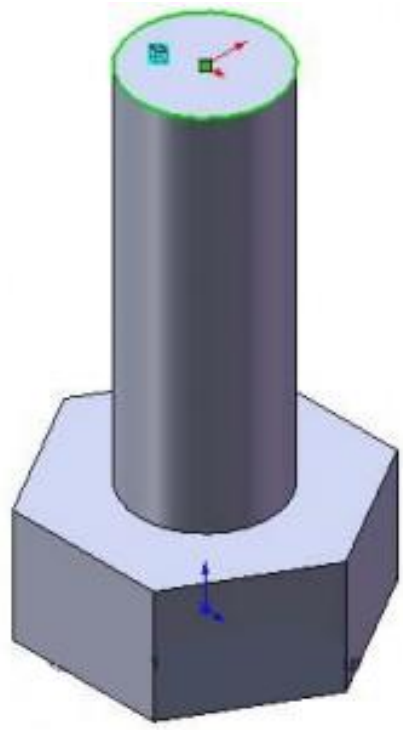
2. Extrude  sketch to 0.34in.



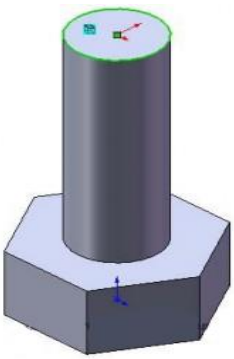
3. Create minor diameter for the thread, sketch circle on the top face, set diameter to 0.4in.



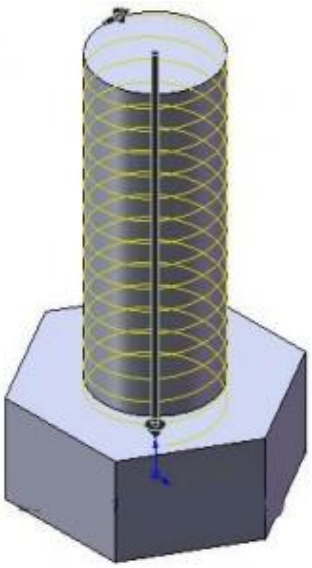
4. Extrude sketch to 1.1in.



5. Click end edge of thread shaft, click convert entities  .

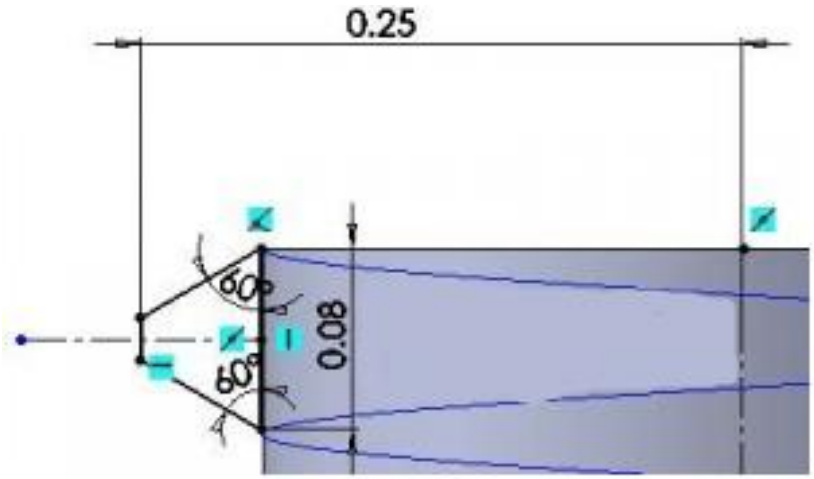


6. Select Helix/Spiral feature set height to 1.2in, thread per inch=pitch 13/1in



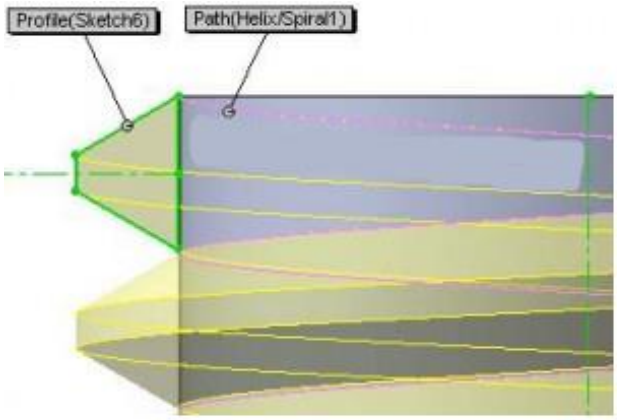
7. Right click on Front plane, Insert sketch 

, sketch thread profile.

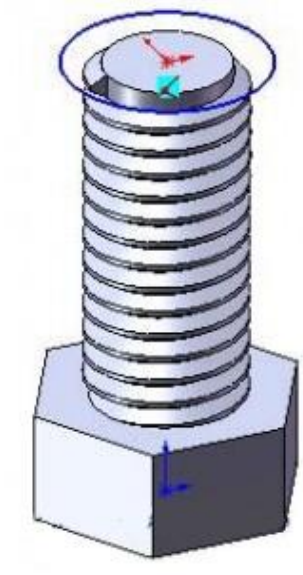


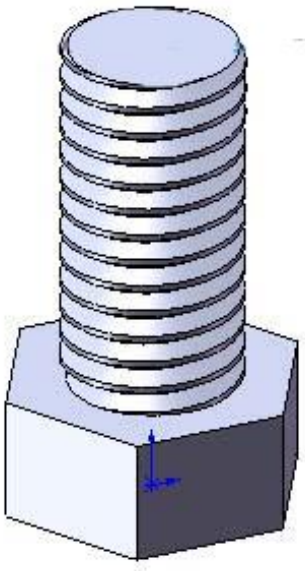
8. Click sweep feature  , 

select sketch profile as sketch and helix as a path, OK.

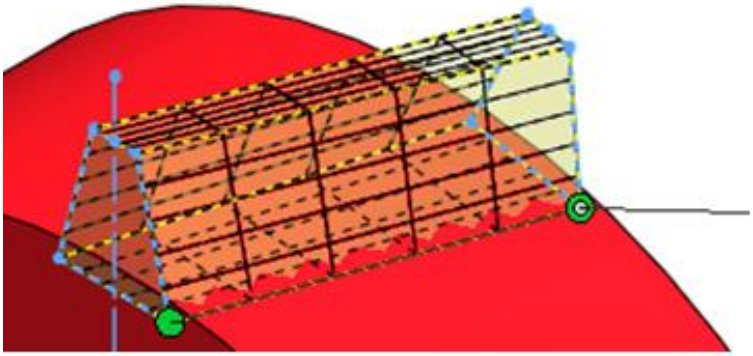


9. Create a sketch of a circle on the end shaft, extrude cut 0.1in  .





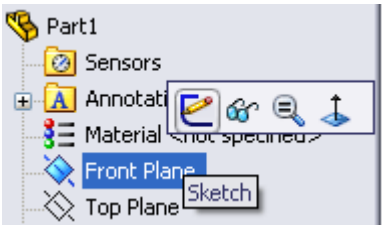
CHAPTER 4:HOW TO CREATE A HELICAL GEAR



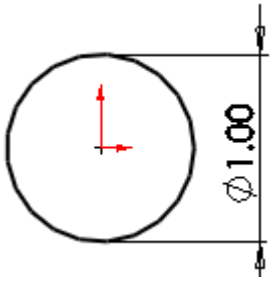
In this chapter, you will create helical gear.

1. Click **New**.  Click **Part**,  **OK**.

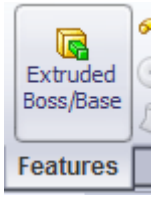
2. Click **Front Plane** and click on **Sketch**.

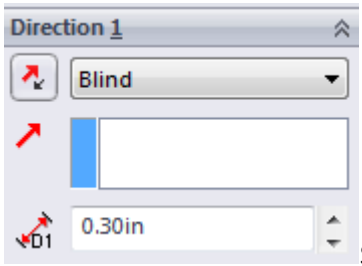


3. Click **Circle**  and sketch a circle centre at origin. Click **Smart Dimension**,  click sketched circle and set it diameter to **1.0in**.

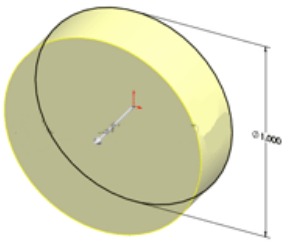


4. You just completed your sketch, let's build feature from it. Click **Features>Extruded Boss/Base**.



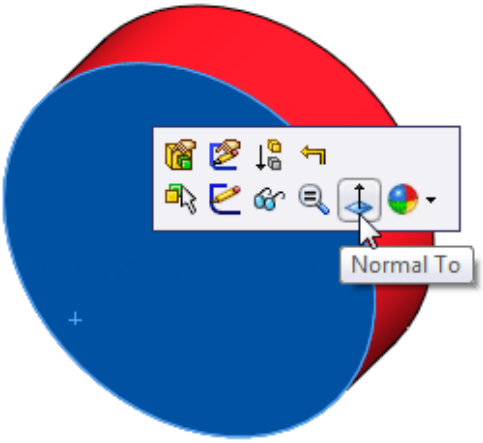


Set **D1** to **0.3in**

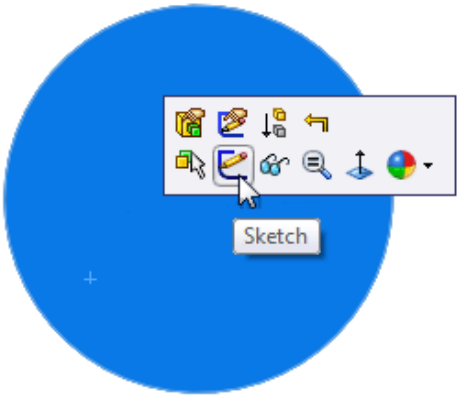


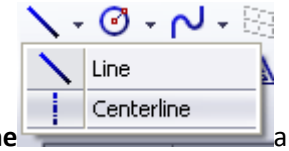
and .

5. Click on front face and click **Normal To**.

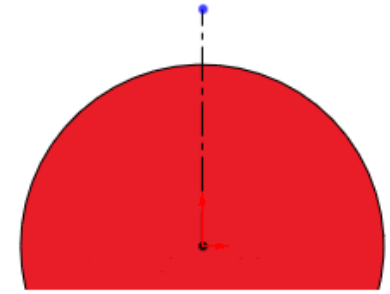


6. Click on front face and click **Sketch**.

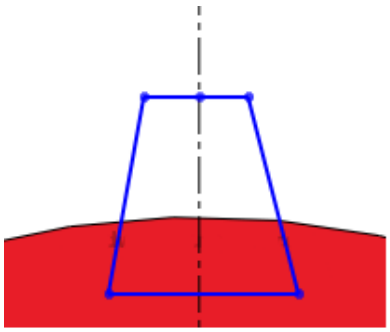


7. Click on **Centreline** 

and sketch vertical Centreline.

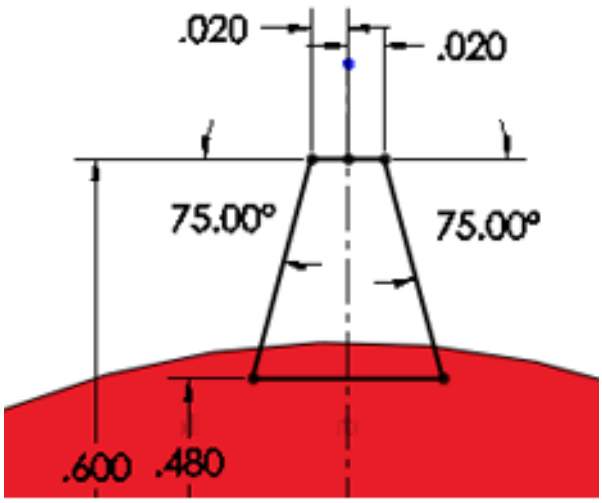


8. Click **Line**  and sketch gear teeth profile.

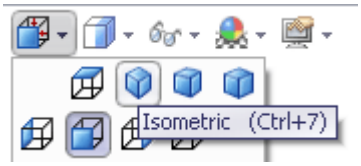


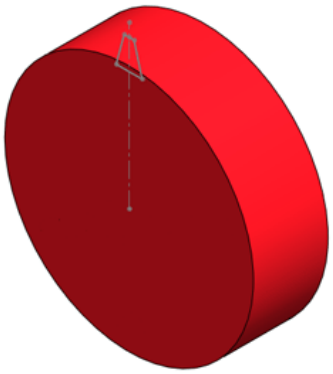
9. Click **Smart Dimension**, 

dimension sketch as sketched below.

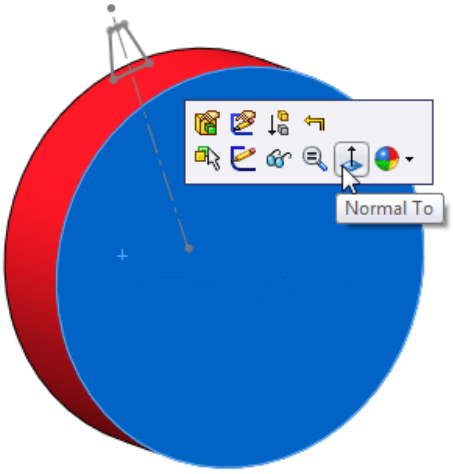


10. Click **Exit Sketch**, change view to **Isometric**.

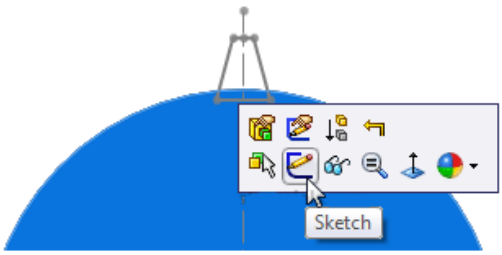




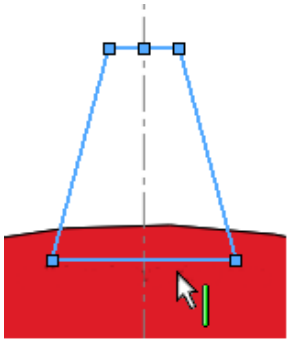
11. Click scroll mouse button and rotate the part to back side.

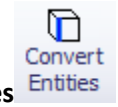


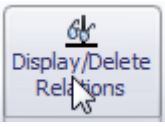
Click the back face and select **Normal To.** Click on this face again and click **Sketch.**



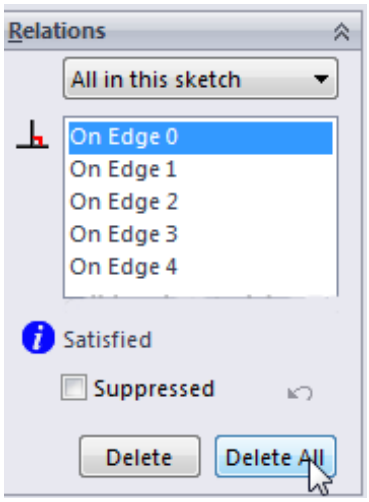
12. We will trace last sketch to this face, while holding **CTRL** click **all sketched line**



and click **Convert Entities** 

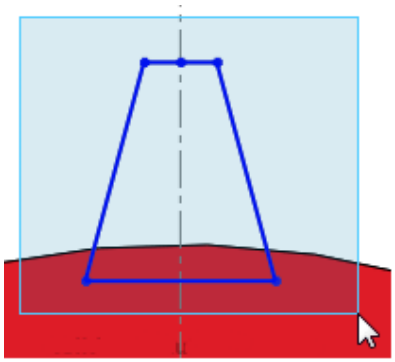
. Now we need removed all relation between this sketch and the other sketch, click **Display/Delete Relations** 

click **Delete All**

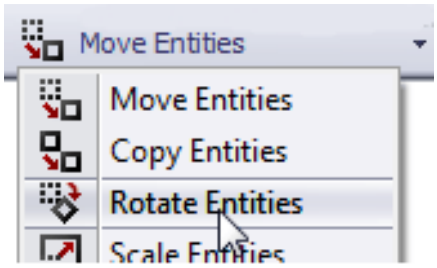


and  .

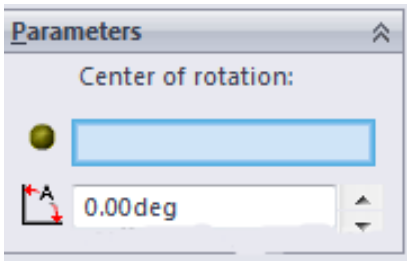
13. Click and drag select all the sketch line.



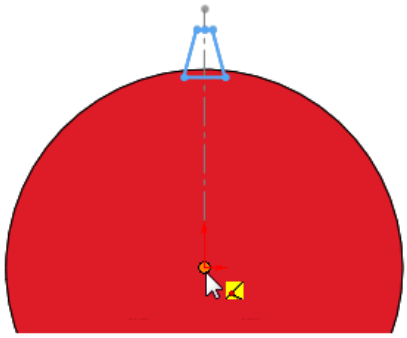
Click on **Rotate Entities,**



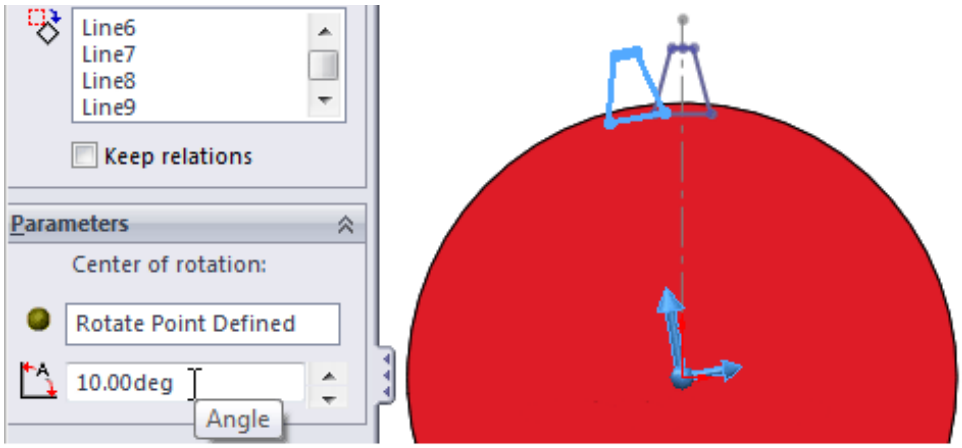
Click **Centre of Rotation box**



and click **origin (centre part).**

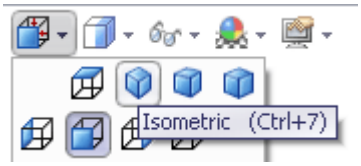


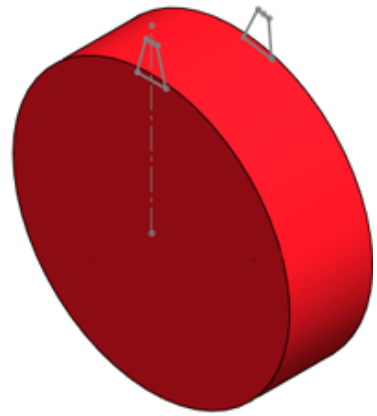
On Parameter option enter **10 deg** rotation.



and  .

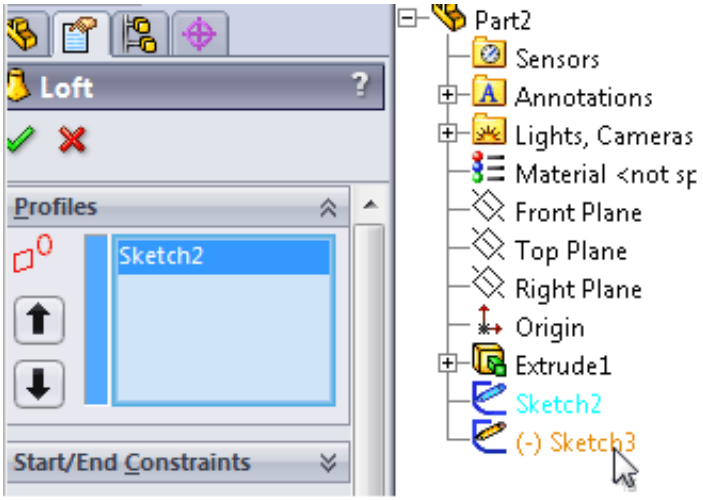
14. Click **Exit Sketch**,  change view to **Isometric**.

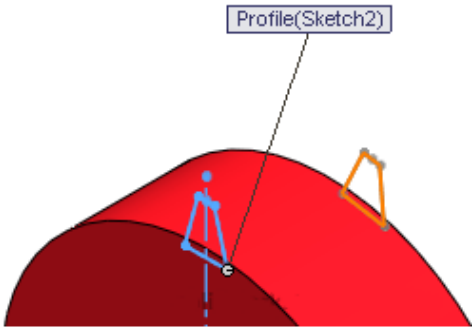




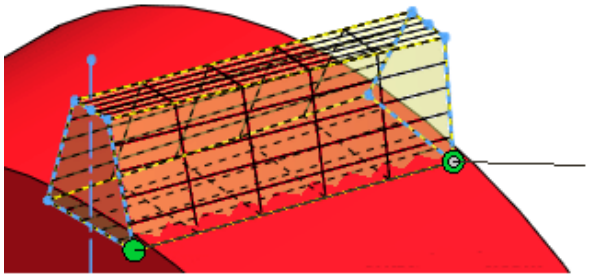
15. Click **Features>Lofted Boss/Base,**

open up part tree and double click **Sketch2** and **Sketch3** to add for lofted features.

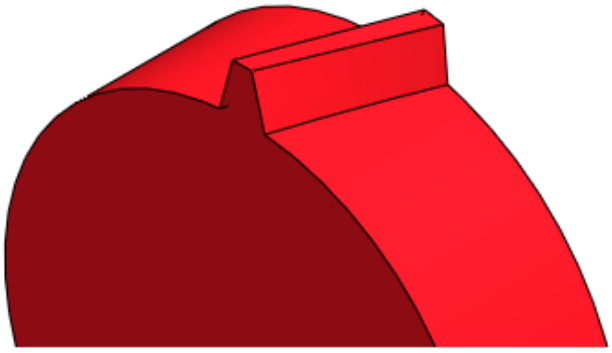


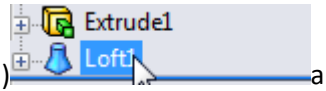


Make sure that the two green points are on the same edge on sketch, if not drag and relocate it.

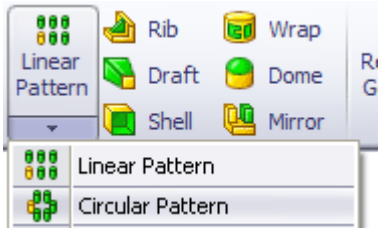


and  .

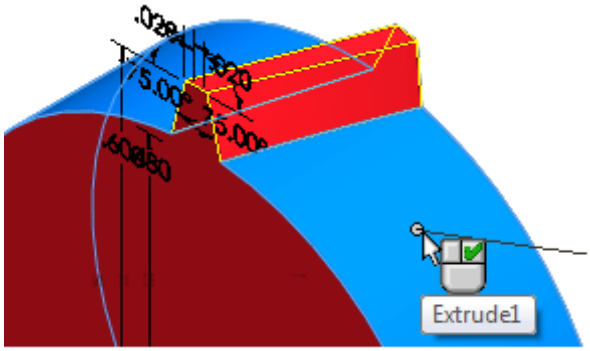


12. Click on **Loft1** (gear teeth) 

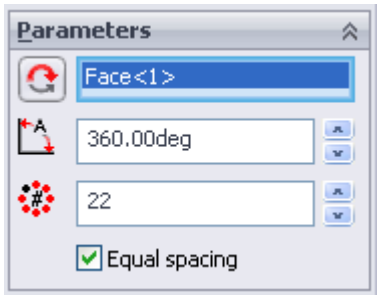
and click **Circular Pattern**.

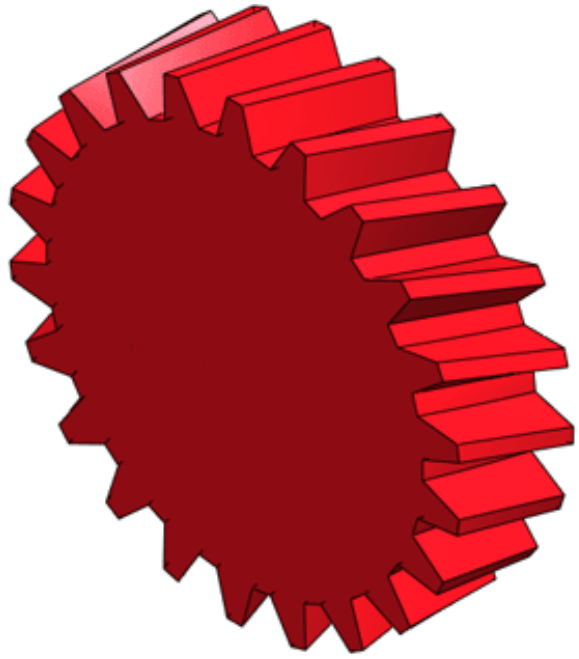


Click on the cylinder face as axis of rotation (or click on View>Temporary Axes select the temporary axis as axis of rotation).

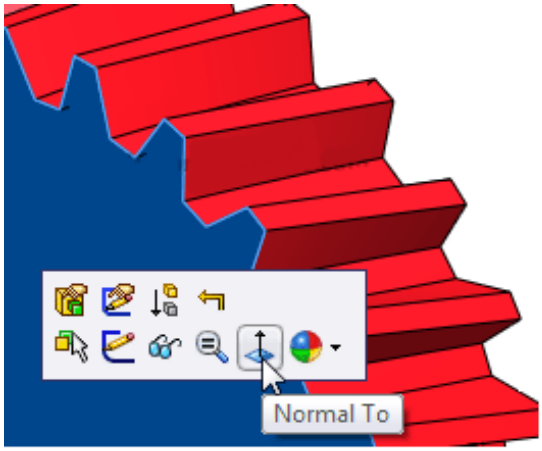


Set **Instances** to **22** and  .

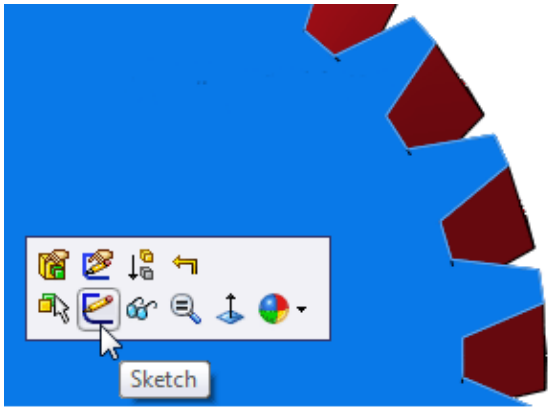




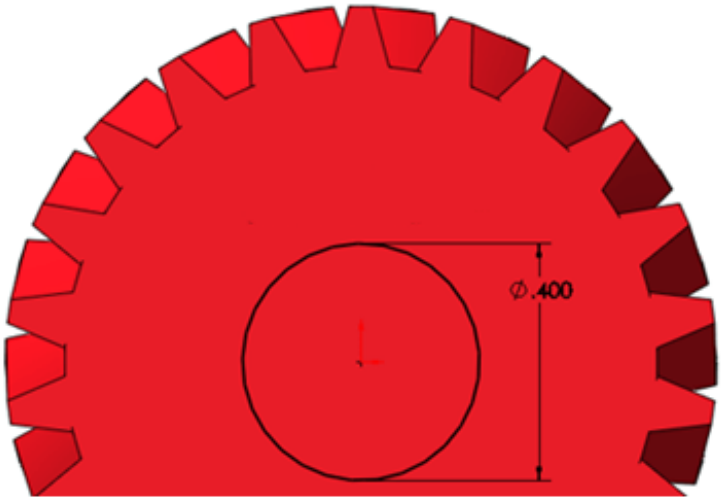
13. Click on Front face and select **Normal To**.



14. Click on front face and select **Sketch**.

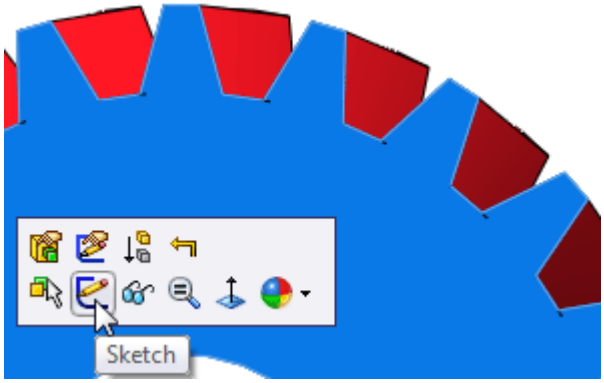


15. Sketch a **Circle** and sketch a circle centred at origin. Click **Smart Dimension**, dimension sketch as **0.40in** circle.

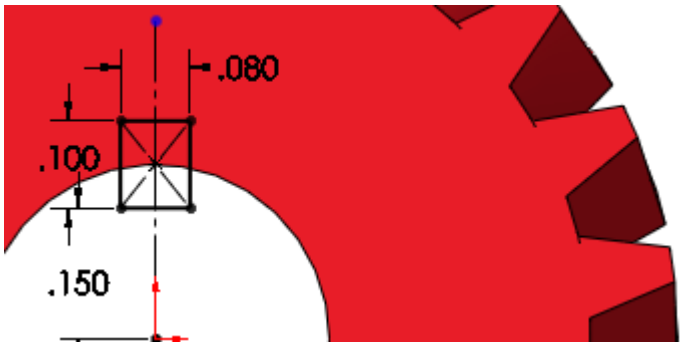


16. Click **Features>Extruded Cut** and set **Direction** to **Through All** and .

17. Click on front face and select **Sketch**.

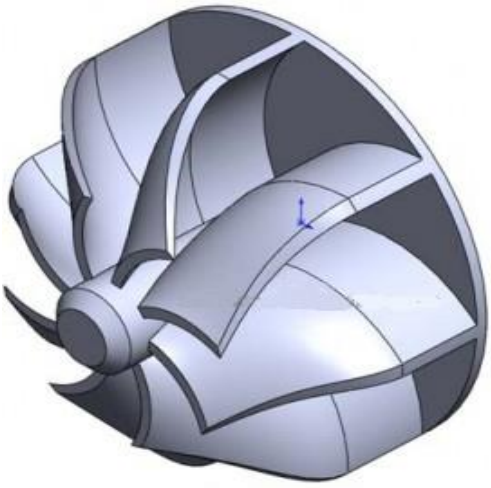


18. Click **Rectangle** and sketch a rectangle as sketched. Click **Smart Dimension**, dimension rectangle as sketched below.

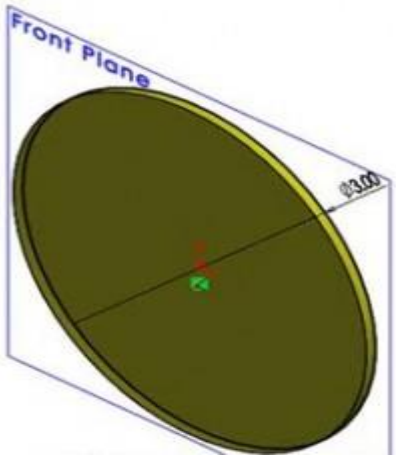


20. Click **Features>Extruded Cut** and set **Direction** to **Through All** and .

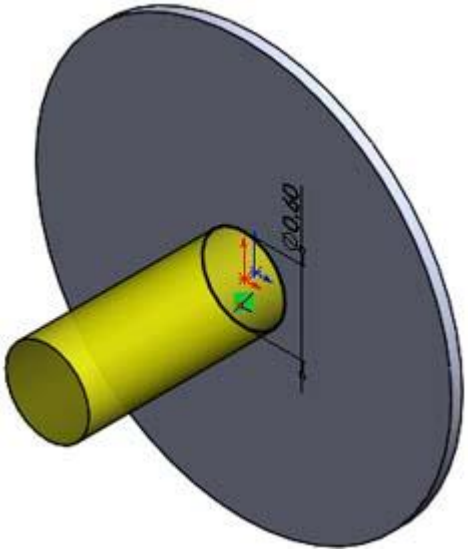
CHAPTER 5: HOW TO CREATE TURBO FINS



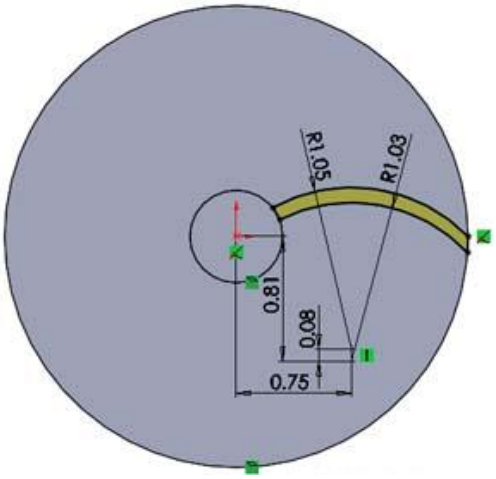
1. Sketch 3in circle and extrude to 0.08in on front plane.

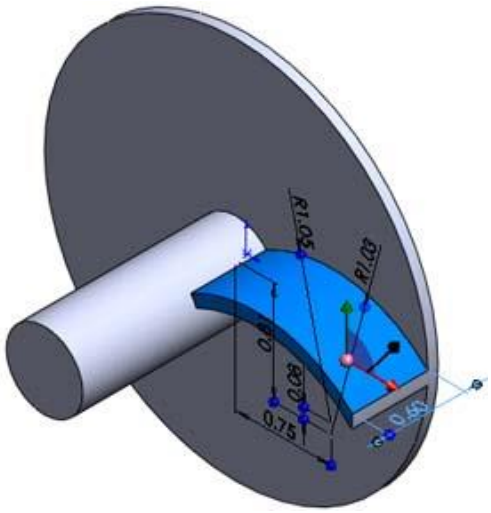


2. Sketch 0.6in circle on top of the extruded face and extrude it to 1.5in.

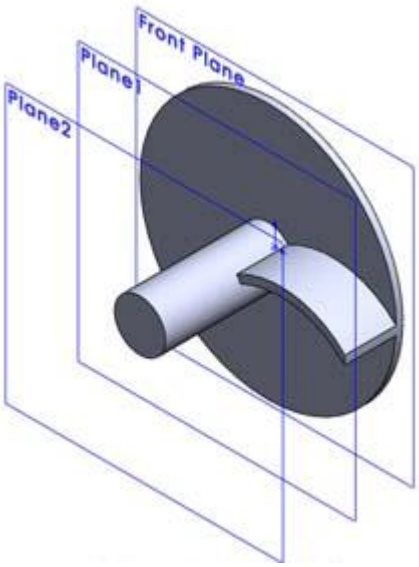


3. Sketch fin profile on the extruded face as shown and extrude it to 0.6in.

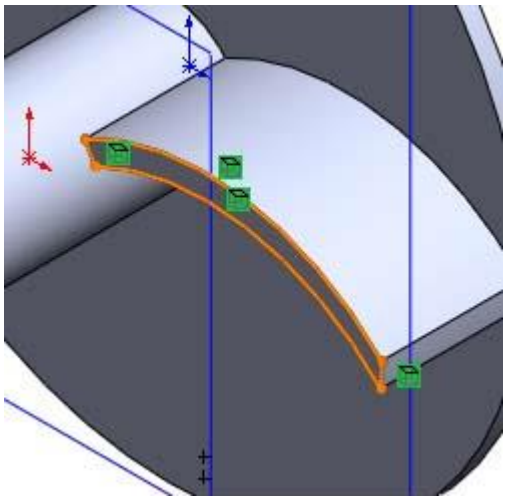




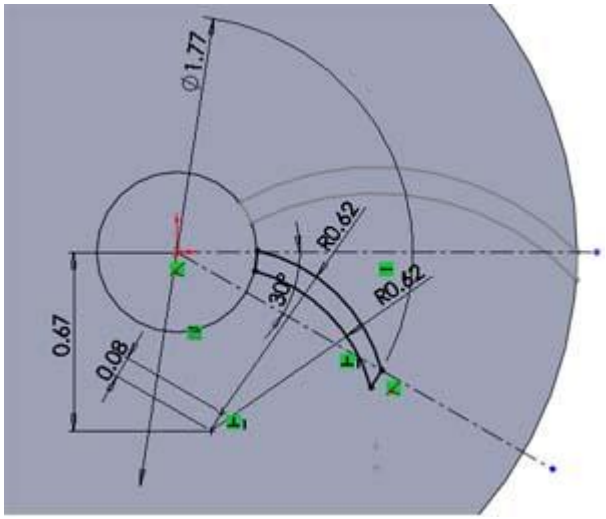
4. Add Plane1 with 0.68in offset from the Front plane and Plane 2 with 0.85in from Plane 1.



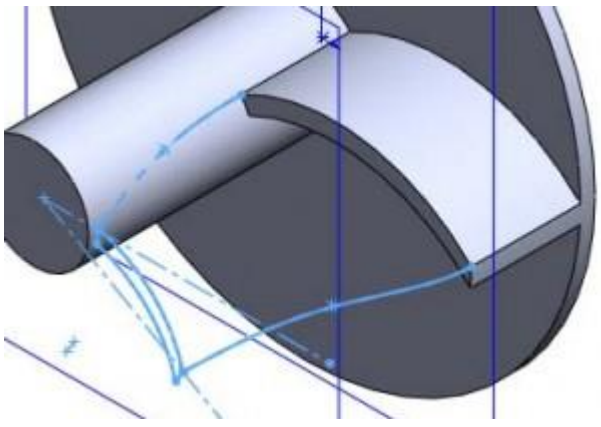
5. Insert sketch on Plane 1, select all edges to the extruded fin and convert it to entities.



6. Insert another sketch on Plane 2, as shown.

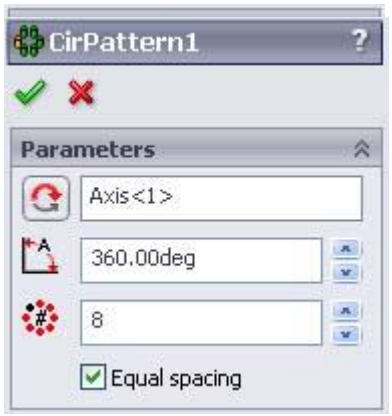


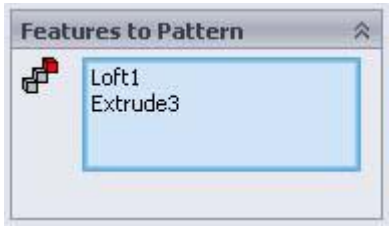
7. Sketch two curve line using 3D sketch tool, as shown.

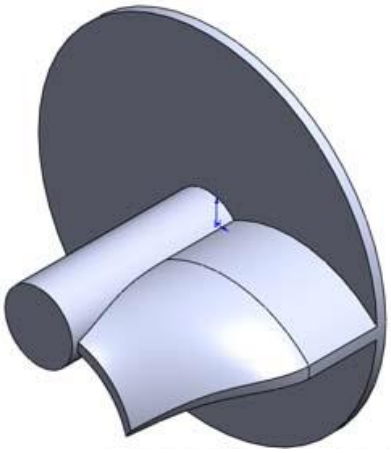


8. Click Lofted Boss/Base 

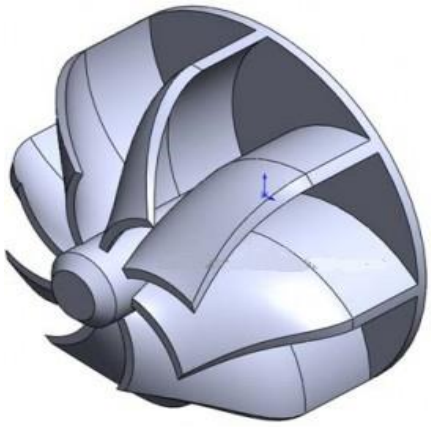
, select profile Sketch 5 and sketch 6and for guide curves select 3DSketch1 and 3DSketch2, OK.



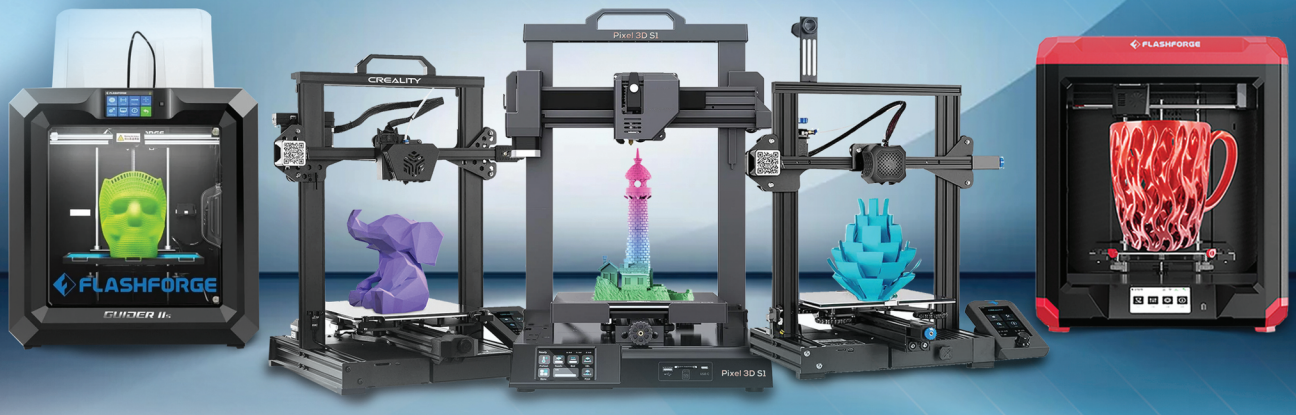




9. Click Circular Pattern, view temporary axis Tools>Temporary Axes. Select centre axis, 360 degrees, #8, Equal Spacing.




TOWER PRO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