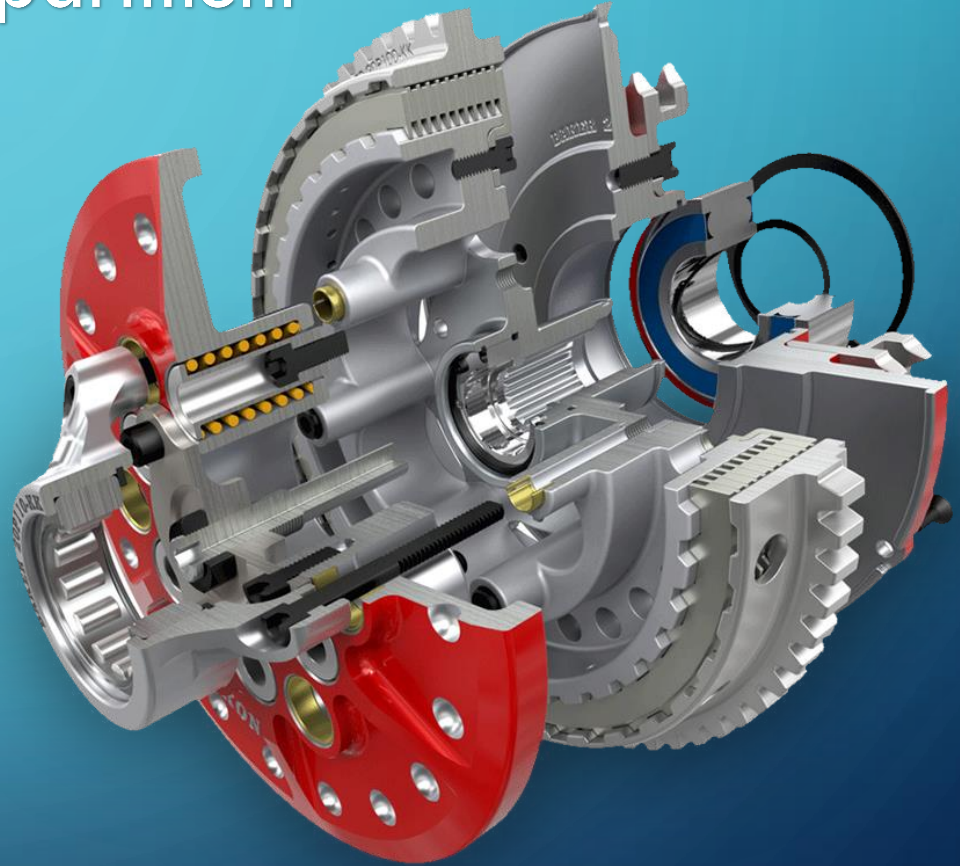
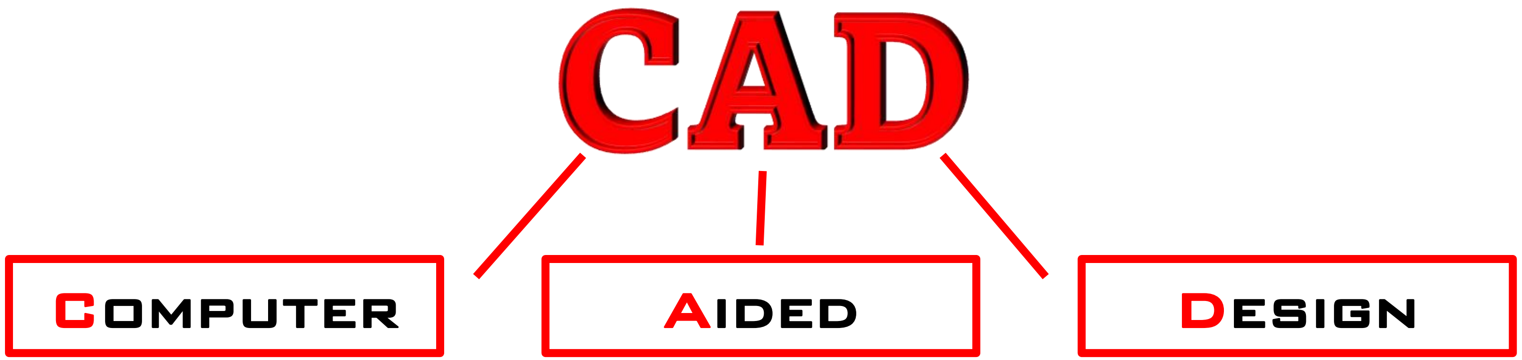
Tikrit University

Mechanical Engineering Department



SOLIDWORKS

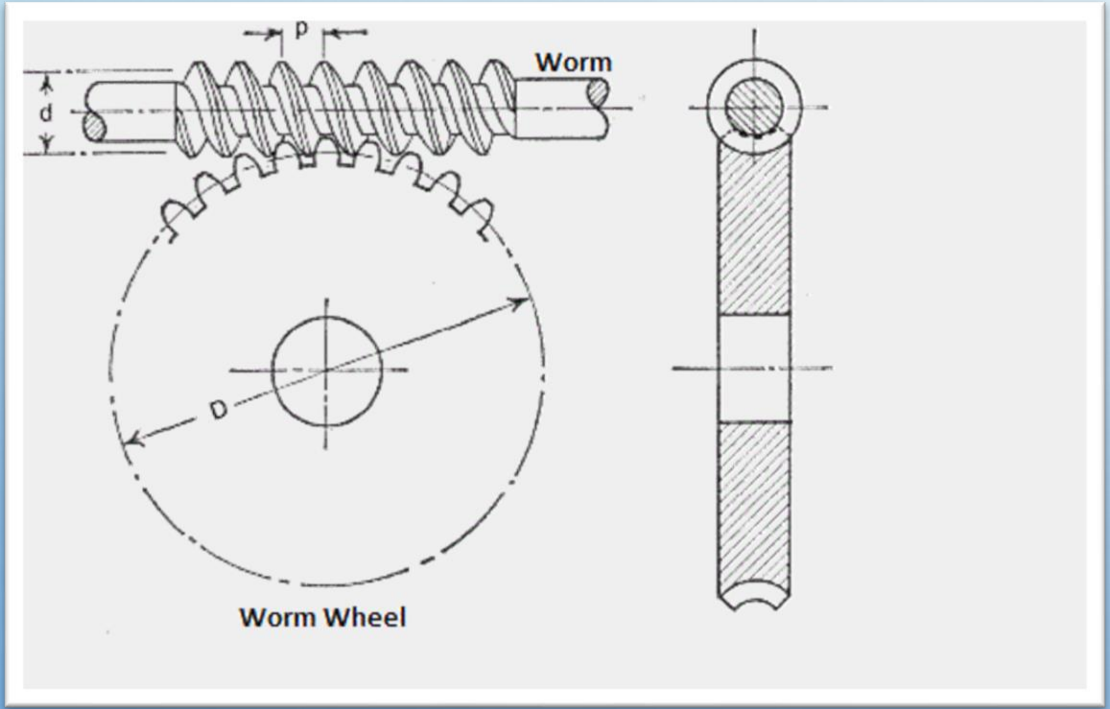
By L.A. Omar Aldaban



- **CAD** is the use of computers to aid in the creation, modification, analysis, or optimization of a design. This software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing.

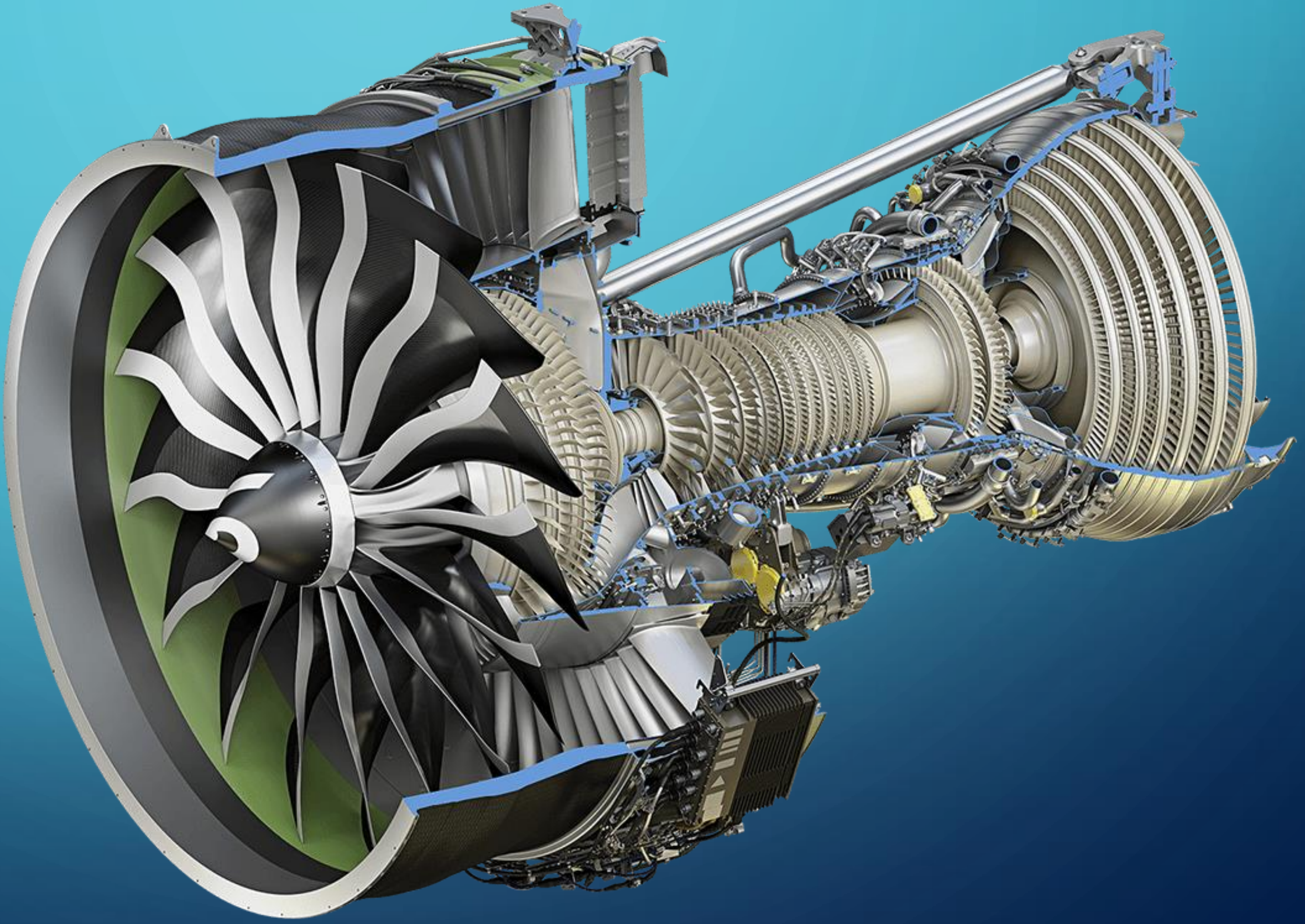
Solid Works

- SOLIDWORKS IS A **THREE-DIMENSIONAL** (**3D**) DESIGN APPLICATION. THIS IS A **COMPUTER-AIDED DESIGN** (**CAD**) SOFTWARE THAT RUNS ON WINDOWS COMPUTER SYSTEMS. IT WAS LAUNCHED IN 1995 AND HAS GROWN TO BE ONE OF THE MOST COMMON SOFTWARE THAT USED GLOBALLY IN RELATION TO ENGINEERING DESIGN



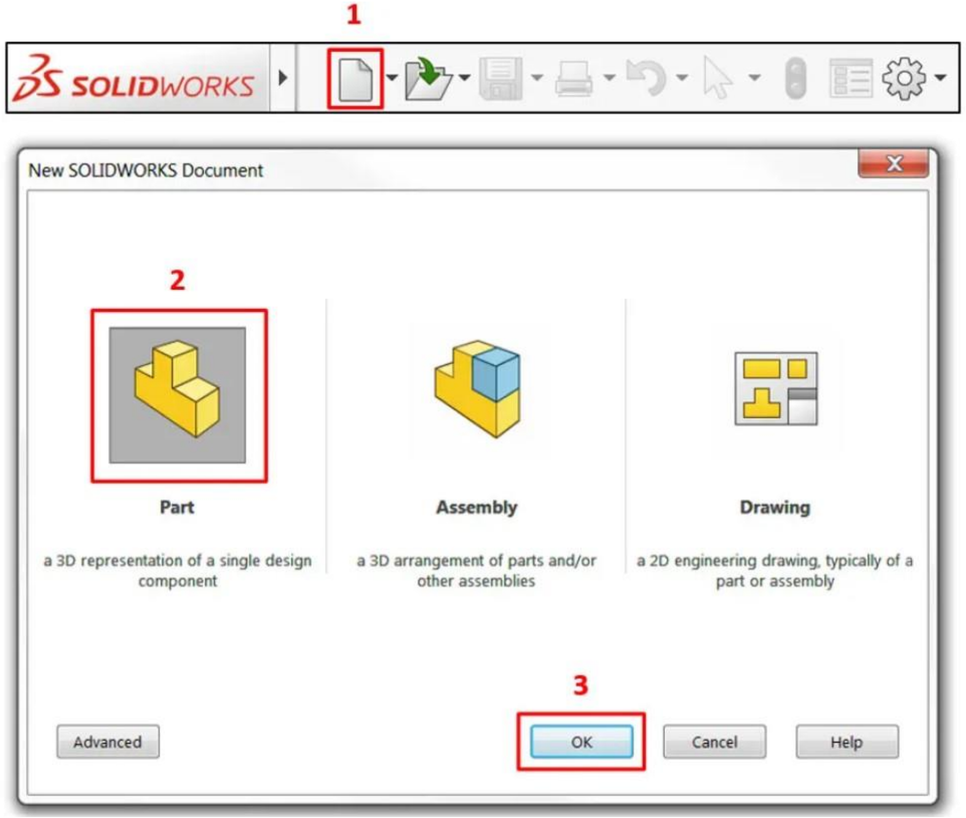
SOLIDWORKS APPLICATIONS

- Aerospace
- Construction
- High-tech electronics
- Medicine
- Oil and gas
- Packaging machinery
- Engineering services
- Furniture design
- Energy
- Automobiles

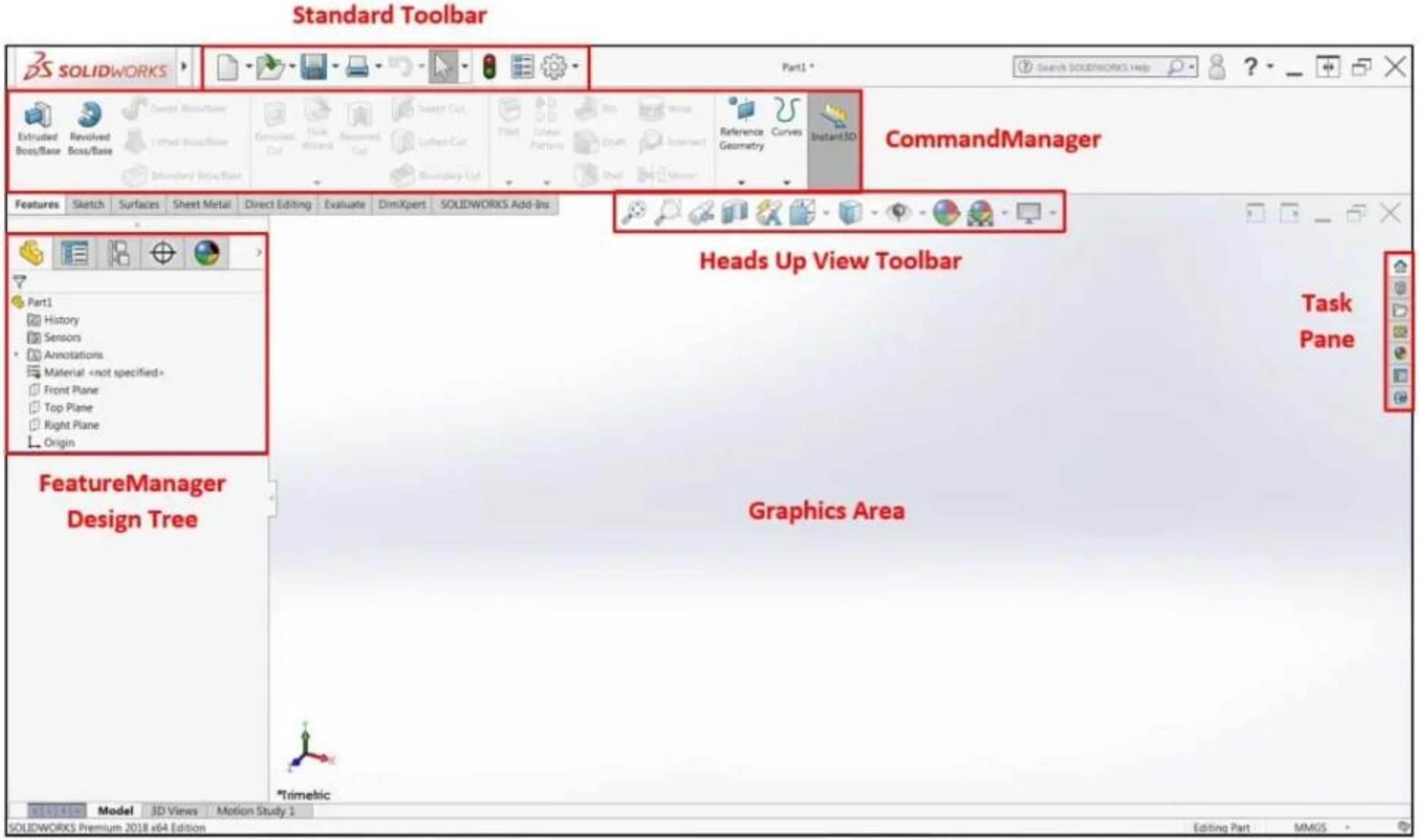


SolidWorks User interface

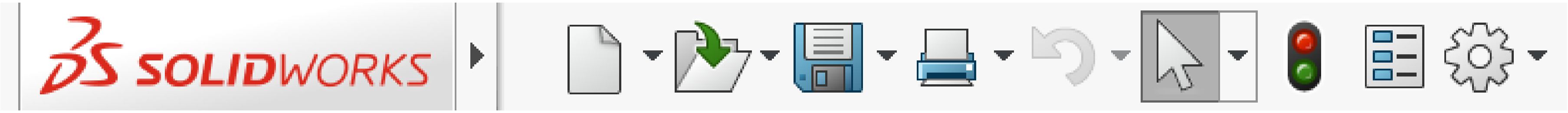
1. Starting a new part document



Once the new part document has been opened, the interface will look like the figure shown below



Standard Toolbar



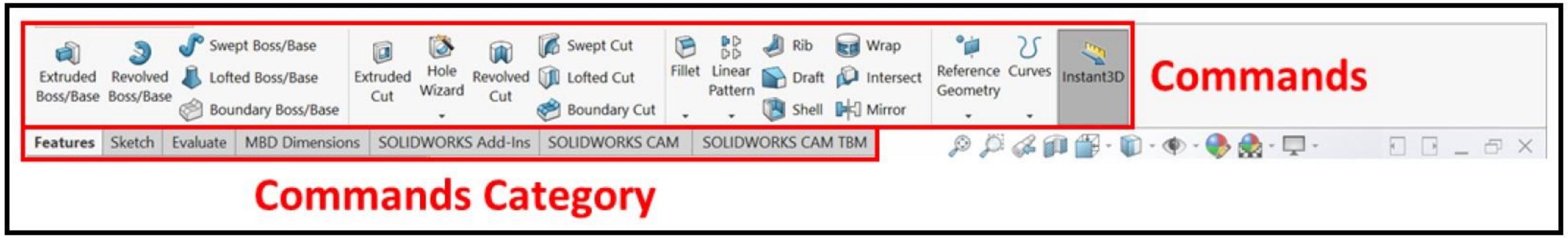
The Standard Toolbar contains basic commands including

- New Document
- Open Document
- Save Document

Additionally, at the right end of this toolbar, a gear icon is available which will open a **System Options** dialog, where a variety of settings can be changed. Hovering the cursor over the SOLIDWORKS icon will reveal several dropdown menus that contain all available commands, including a Help dropdown menu where tutorials may be accessed.

The command bar

The command bar is located at the top of the screen. The Command Manager contains all the SOLIDWORKS commands that are used for building models. It contains different categories of commands, and each category contains a set of different commands.

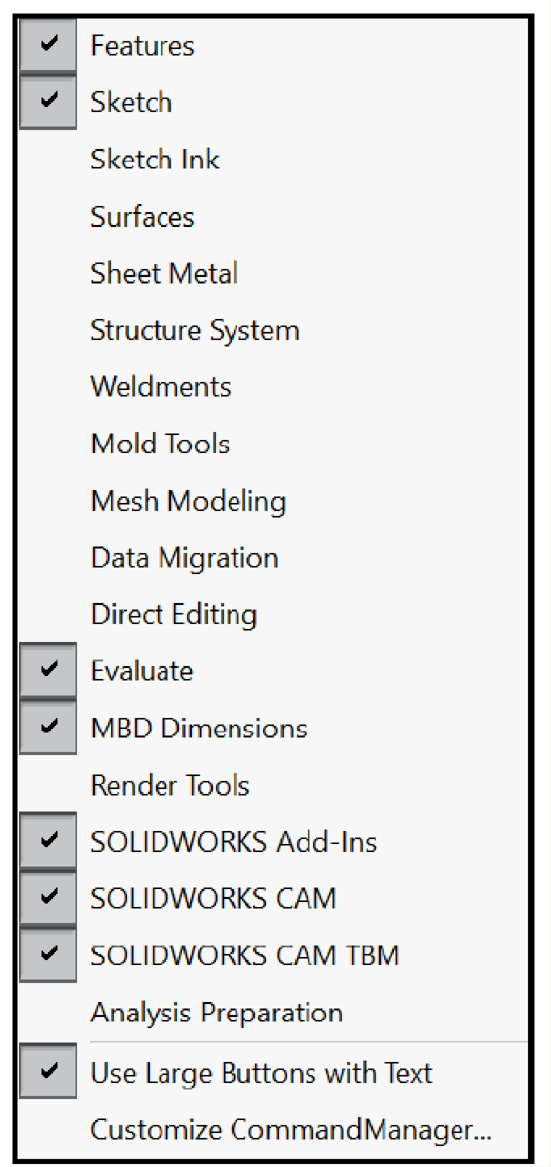


- Different categories (tabs) of commands correspond to different functions. For example, in the **Sketch** category/tab, you will find all the commands that we will need in the sketching phase. In the **Features** category/tab, you will find all the commands that we will need in order to go from the sketching phase and start creating a 3D model. Those categories are not the only ones SOLIDWORKS provides, but they are the most common ones we will use.

To show the hidden **Commands** categories, we can do the following:

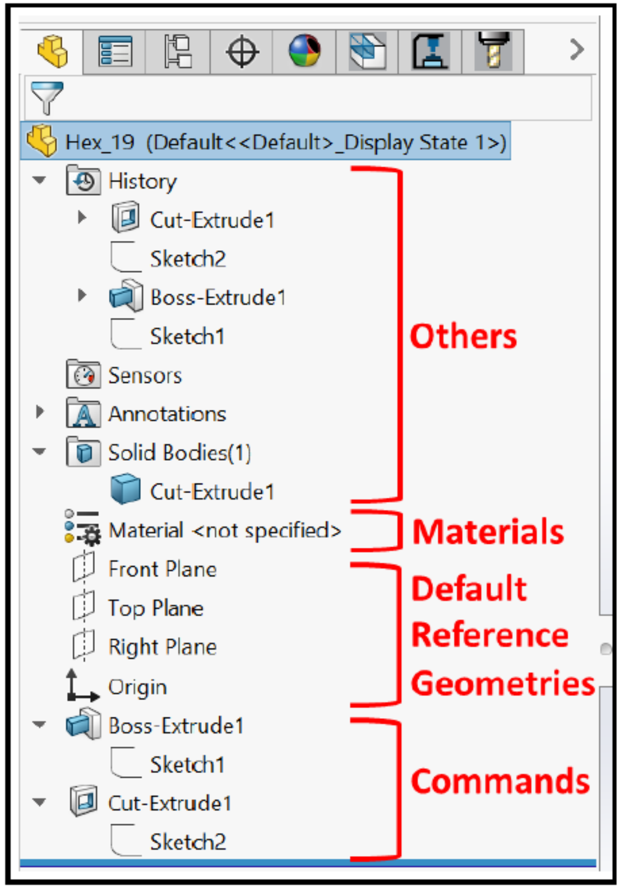
1- Right-click on any of the **Commands** categories. This will make the following menu appear, which contains more **Commands** categories, such as **Surfaces**, **Weldments**, **Mold Tools**, and so on.

2- Select the categories you want to be shown. By doing this, these categories will be added to the command bar:

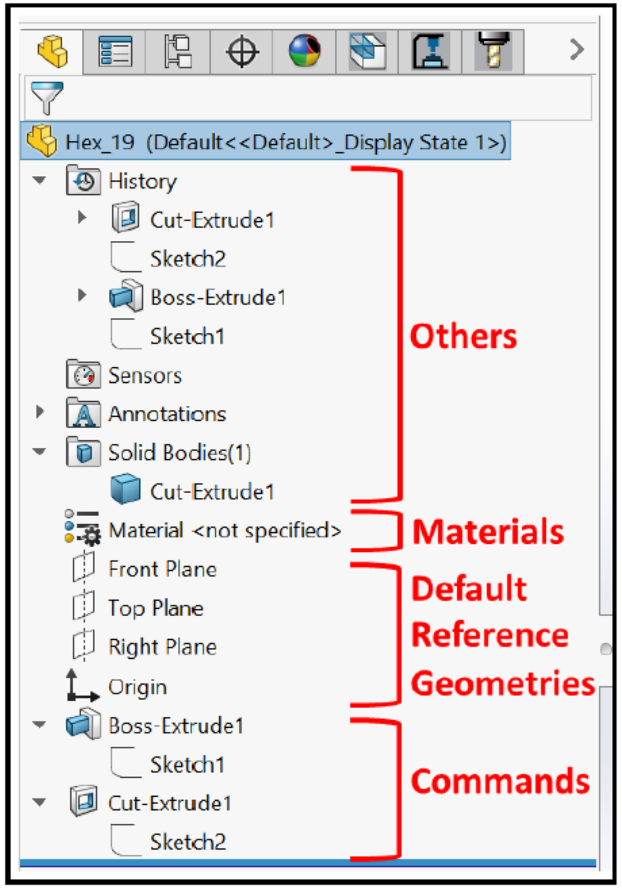


Feature Manager Design Tree

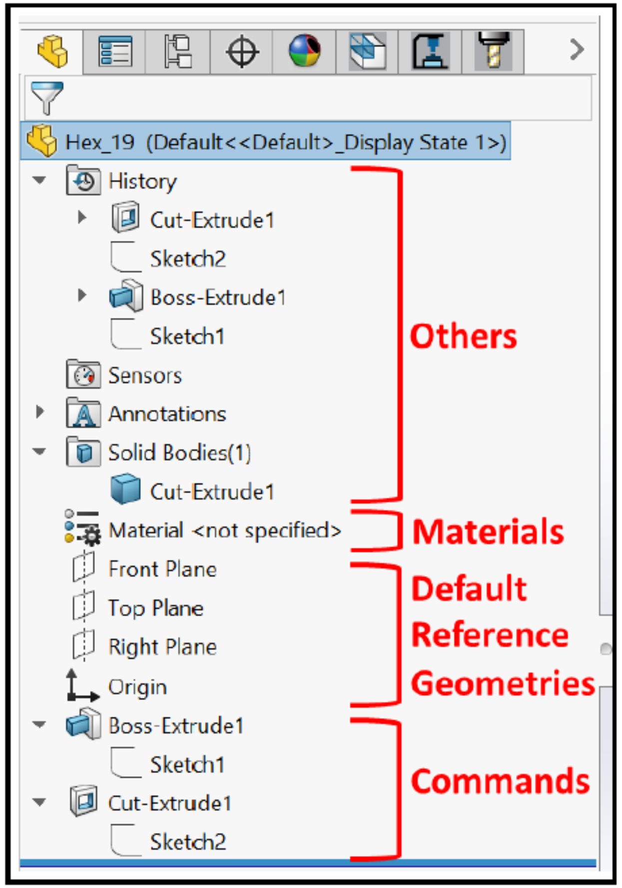
- The four parts of the design tree are as follows:
- **Commands**: These are the commands that are used to build the model. This includes sketches, features, and any other supporting commands that were added during the modeling phase (since we are building a 3D model). In the preceding screenshot, two features were used to create the model, as indicated by **Commands**. The first is **Boss-Extrude1** and the second is **Cut-Extrude1**. Note that these commands are listed in the order of when they were applied.



- **Default Reference Geometries**: The SOLIDWORKS workspace can be understood as endless space. These **Default Reference Geometries** are what can fix our model to a specific point or plane. Without these, our model will be floating in an endless space without any fixtures. Throughout this term, we will start our models from these default references. There are three planes (**Front**, **Right**, and **Top**), in addition to the origin.

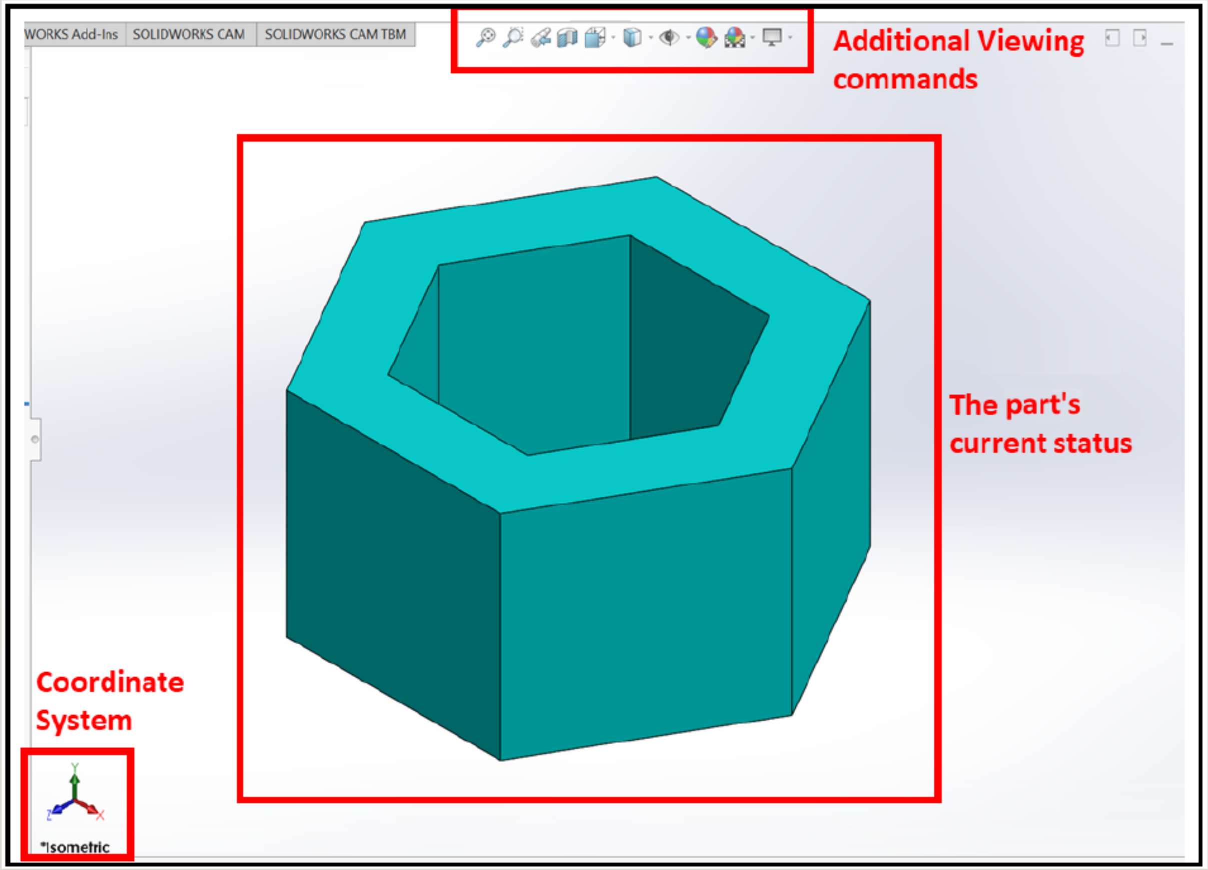


- **Materials**: Realistically, all of the artifacts we have around us are made of a certain material. Some examples of materials include plastic, iron, steel, rubber, and so on. SOLIDWORKS allows us to assign what structural material the part will be made of.
- **Others**: This section includes other aspects of our model's creation, such as **History**, **Sensors**, **Annotations**, and **Solid Bodies**.



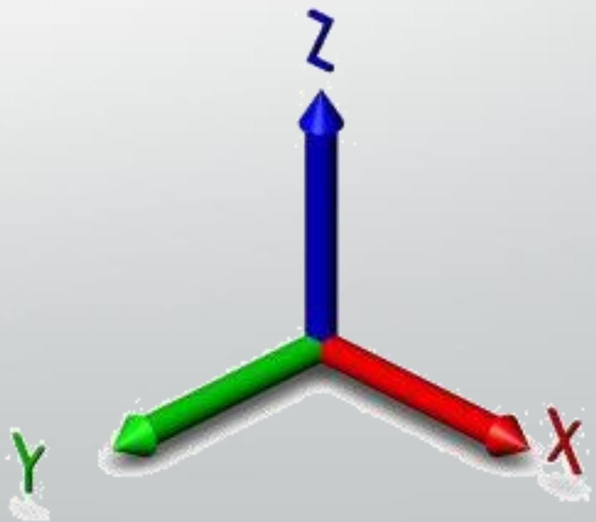
THE CANVAS (WORKSPACE)

The canvas provides a visual representation of the model we have at hand. It contains three main components:



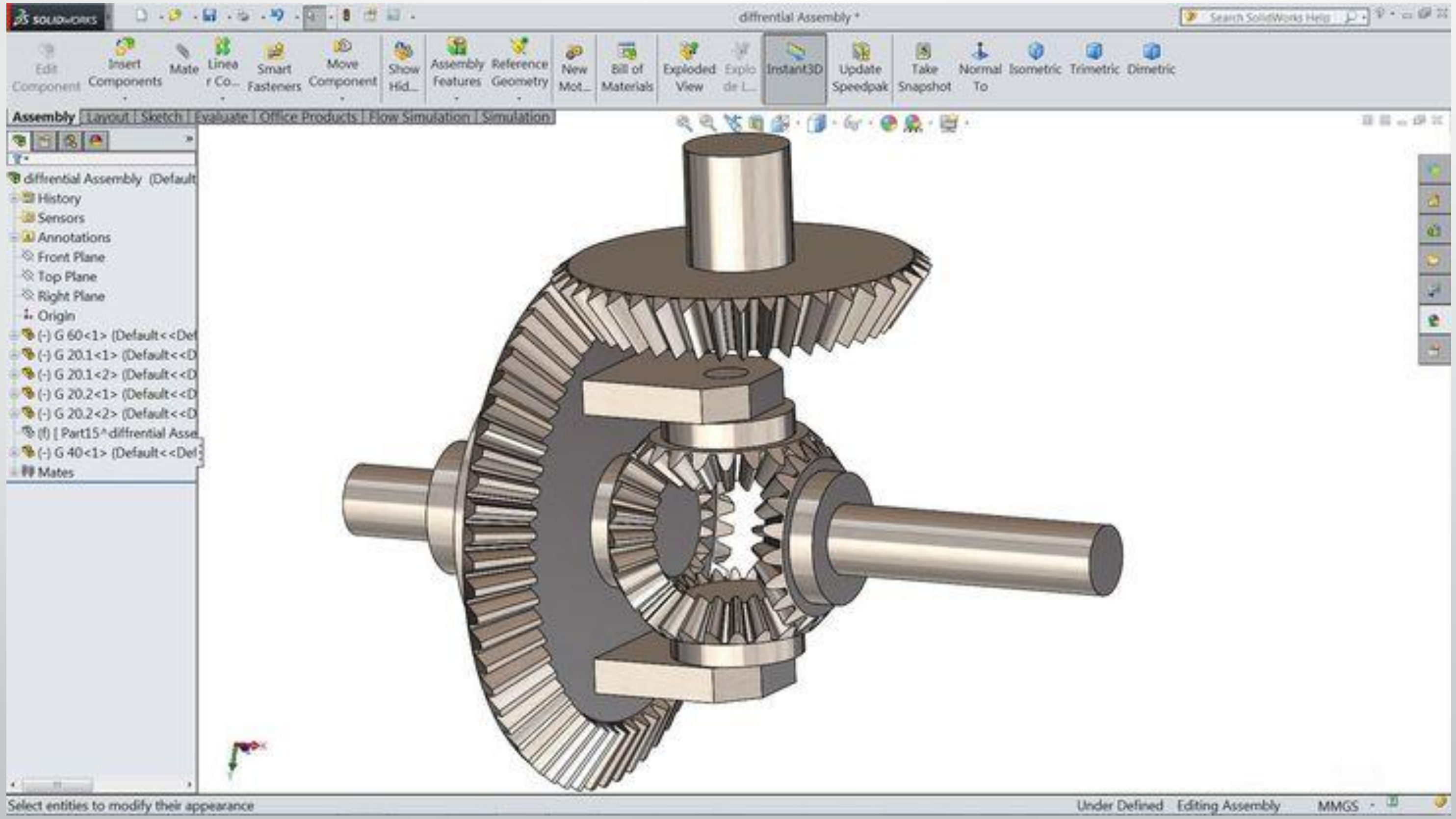
Coordinate System

1. **Coordinate System**: This shows the orientation of the model in relation to the default coordinate system in terms of the x, y, and z axes. They are interactive and can be used to position the viewing angle of the model. By clicking on the various axes, you can arrive at that viewing orientation.



The part's current status

2. **The part's current status**: This shows the current status of the part at work. This is updated with every construction command that's used to build the model.



Additional Viewing commands

3. **Additional Viewing commands**: These contain alternative views of the model, such as the wireframe view and section view. It also provides shortcuts that we can use to modify the scene and the appearance of the model.

THE DOCUMENT'S MEASUREMENT SYSTEM

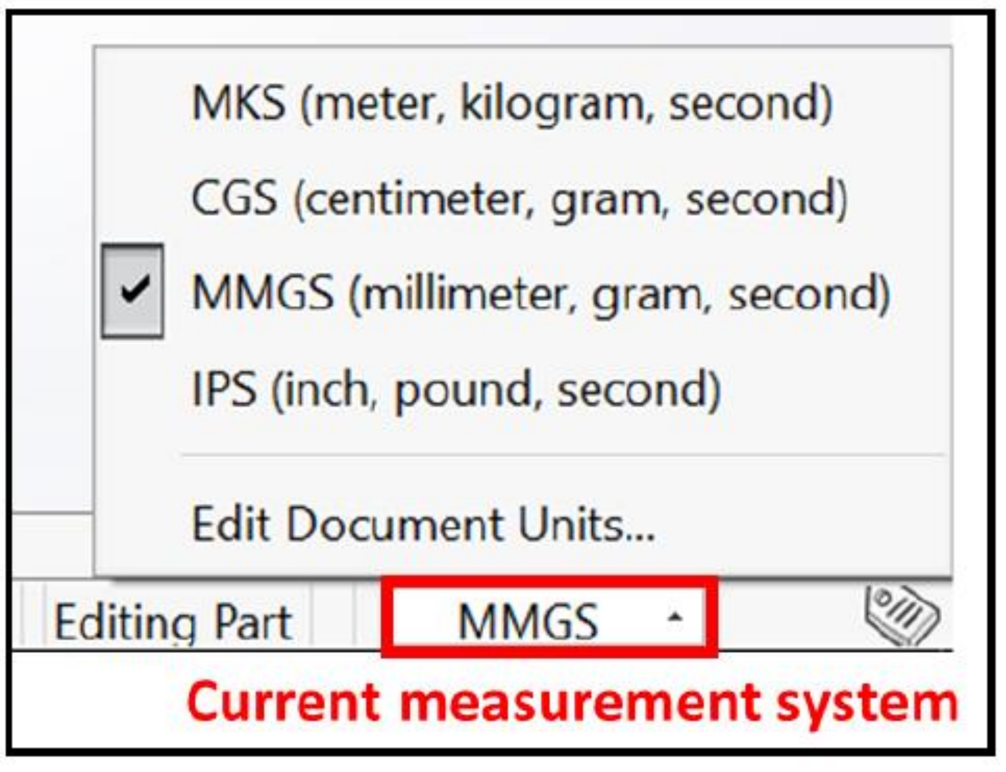
- SINCE SOLIDWORKS IS AN ENGINEERING SOFTWARE, ALL OF THE MODELS ARE CONSTRUCTED IN RELATION TO THE USER-PROVIDED (USER-INPUT) MEASUREMENTS. TO FACILITATE COMMUNICATION, SOLIDWORKS USES STANDARD SYSTEMS THAT ARE CURRENTLY USED IN THE INDUSTRY, INCLUDING THE **INTERNATIONAL SYSTEM OF UNITS (SI)**, THE **IMPERIAL SYSTEM**, AND VARIATIONS OF EACH.

	Imperial unit	**SI unit**
Length	Inches (inch)	Meters (m)
Mass	Pounds (lb)	Kilograms (kg)
Time	Seconds (s)	Seconds (s)

ADJUSTING THE DOCUMENT'S MEASUREMENT SYSTEM

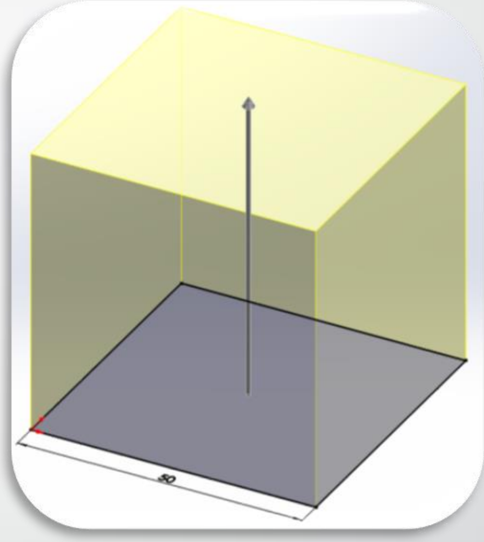
- NOW THAT YOU HAVE DECIDED WHAT SYSTEM TO USE, YOU MUST SET IT UP ON THE SOFTWARE. YOU CAN ADJUST THE UNIT OF MEASUREMENT BY FOLLOWING THESE STEPS:

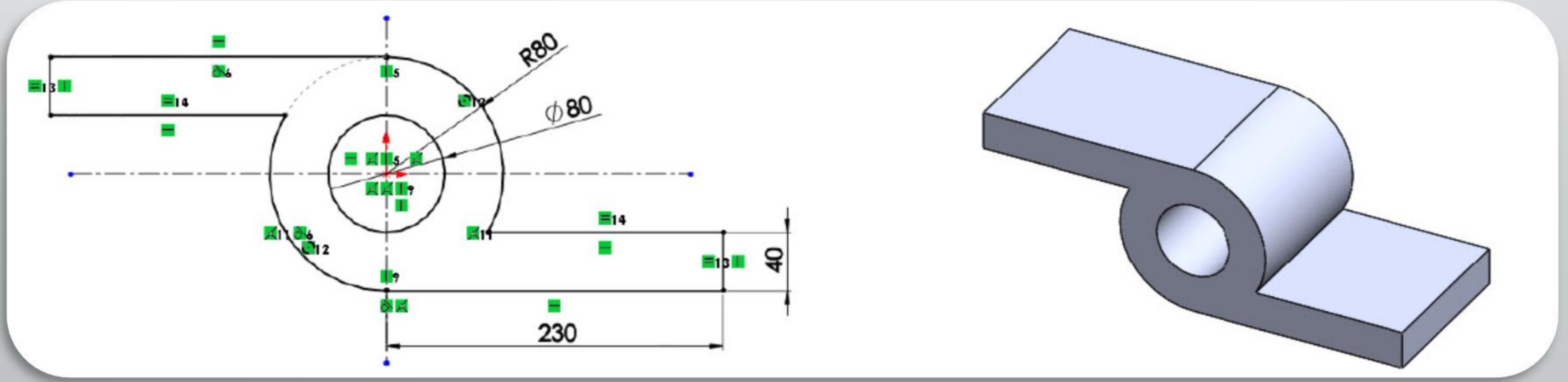
1. OPEN A NEW PART FILE.
2. IN THE BOTTOM-RIGHT CORNER, YOU WILL FIND THE CURRENT/DEFAULT MEASUREMENT SYSTEM, AS SHOWN IN THE FOLLOWING SCREENSHOT:



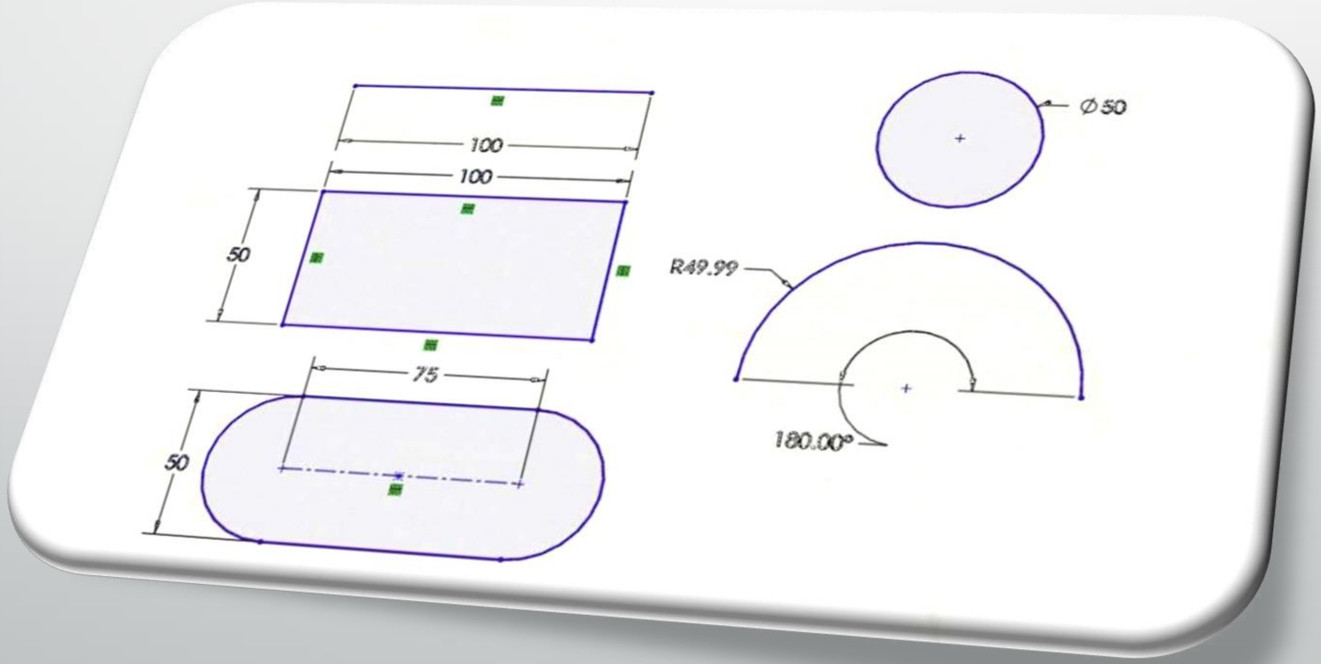
SOLIDWORKS **2D** Sketching

- The foundation of any 3D SOLIDWORKS model is a 2D sketch. This is because SOLIDWORKS builds 3D features based on the guidance of 2D sketches.



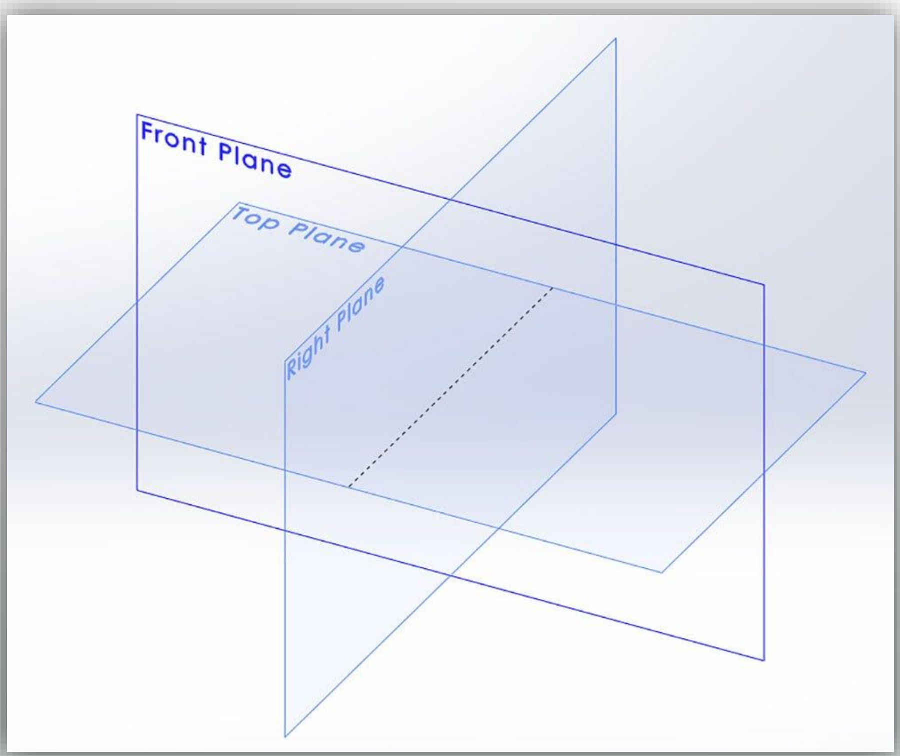


- SOLIDWORKS has many ready-made commands that we can use to create simple sketch shapes such as lines, squares, circles, ellipses, arcs, and so on.

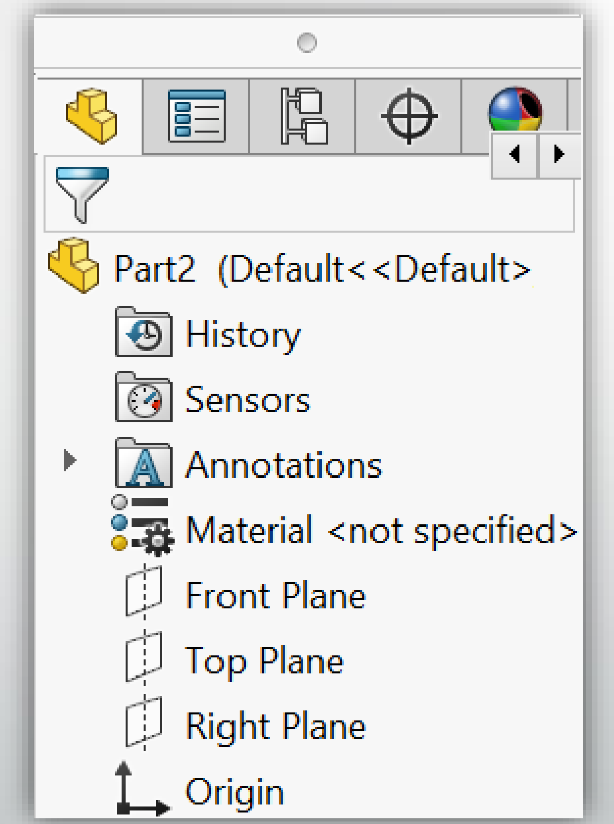


Sketch planes

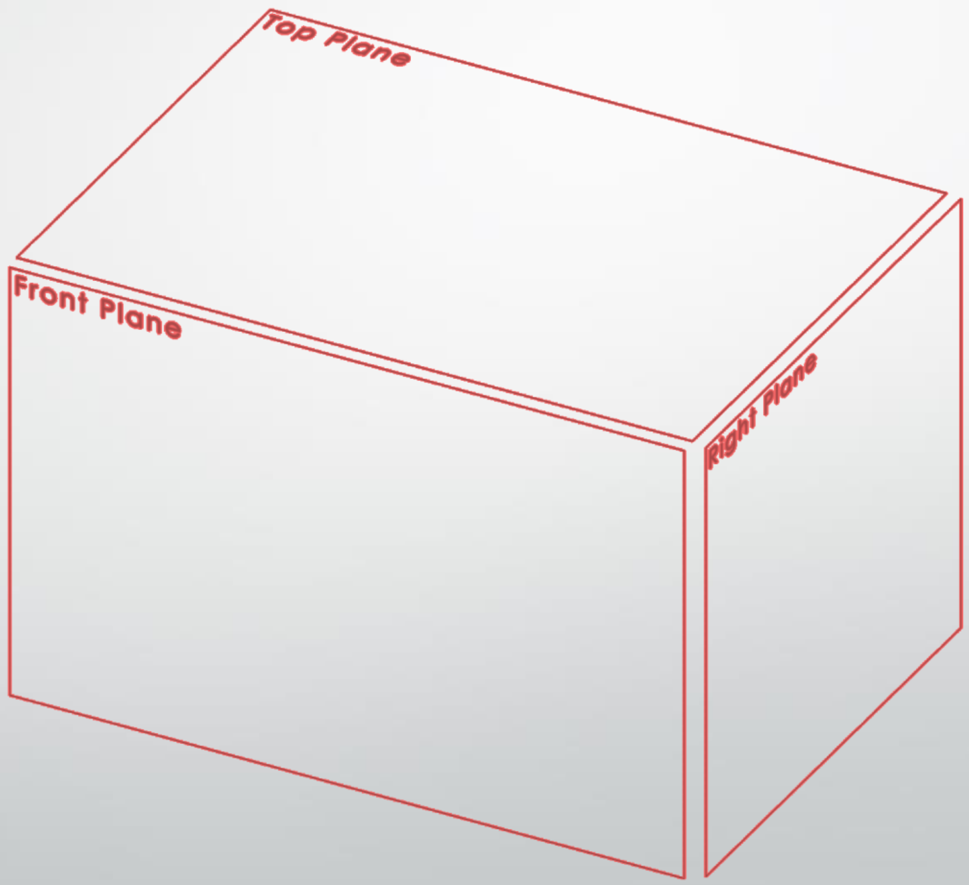
- Sketch planes are flat surfaces that we can use as bases for our sketches. They are important because they give our sketches a solid location. If we didn't have them, our sketches would float undefined in 3D space.



- **SOLIDWORKS** provides us with default sketch planes when we start a new part. We can find them listed in the design tree. The default sketch planes are the **Front Plane**, the **Top Plane**, and the **Right Plane**.



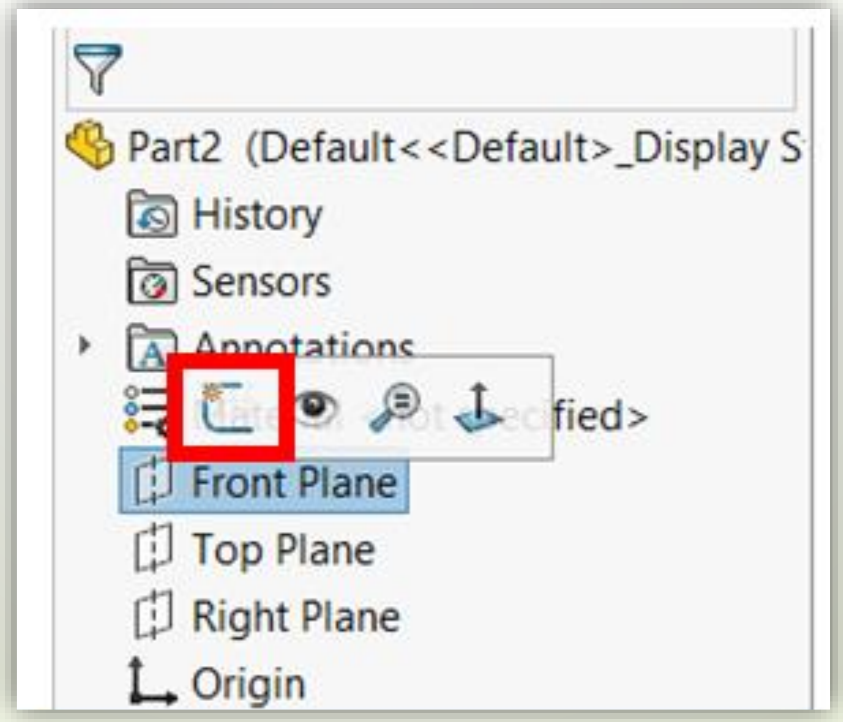
- To help you visualize these default planes, imagine a box and the planes being the top, the front, and the right-hand sides of it.



Getting into the sketching mode

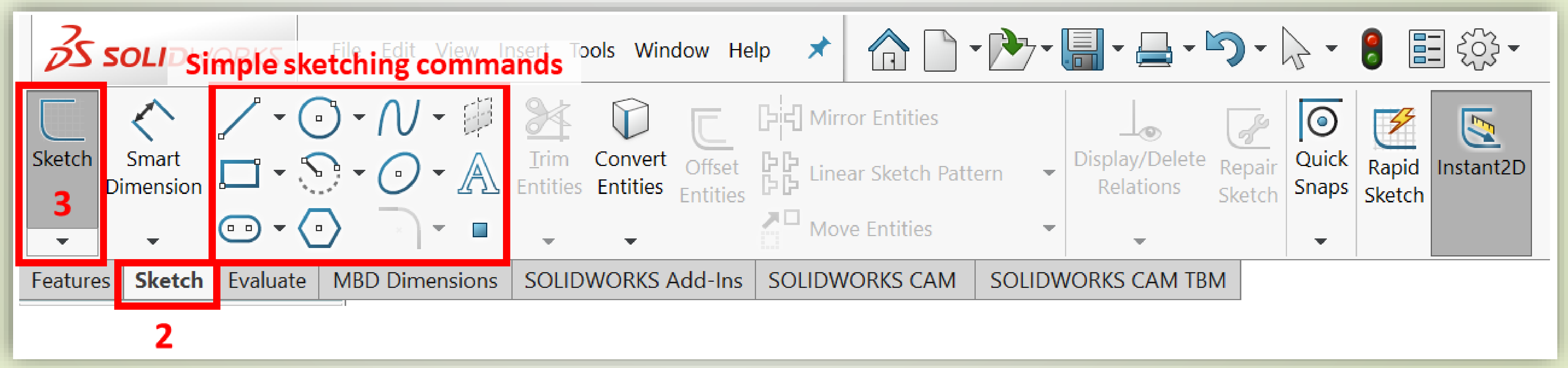
To start a sketch, we need to have a part file open. Then, we can follow these steps to get into the sketching mode:

1. Select one of the default sketch planes: Front, Top, or Right. Then click on the **Sketch** shortcut that appears when you click on one of the planes.



2. In the **Command Manager**, select the **Sketch** option, which is marked as **2** in the following screenshot. This will open up the Sketch commands category, which will show all the commands related to sketching.

3. Select the **Sketch** command, which is marked as **3** in the following screenshot. This will allow us to enter the sketching mode. When we're in the sketching mode, we can apply different sketching commands, such as the marked Simple sketching commands:



Defining sketches

Constructing a sketch will require two elements:

- The first is the sketch entities, such as **lines**, **arcs**, and **circles**.

- The second is the **dimensions** and **relations** that define the sketch entities.

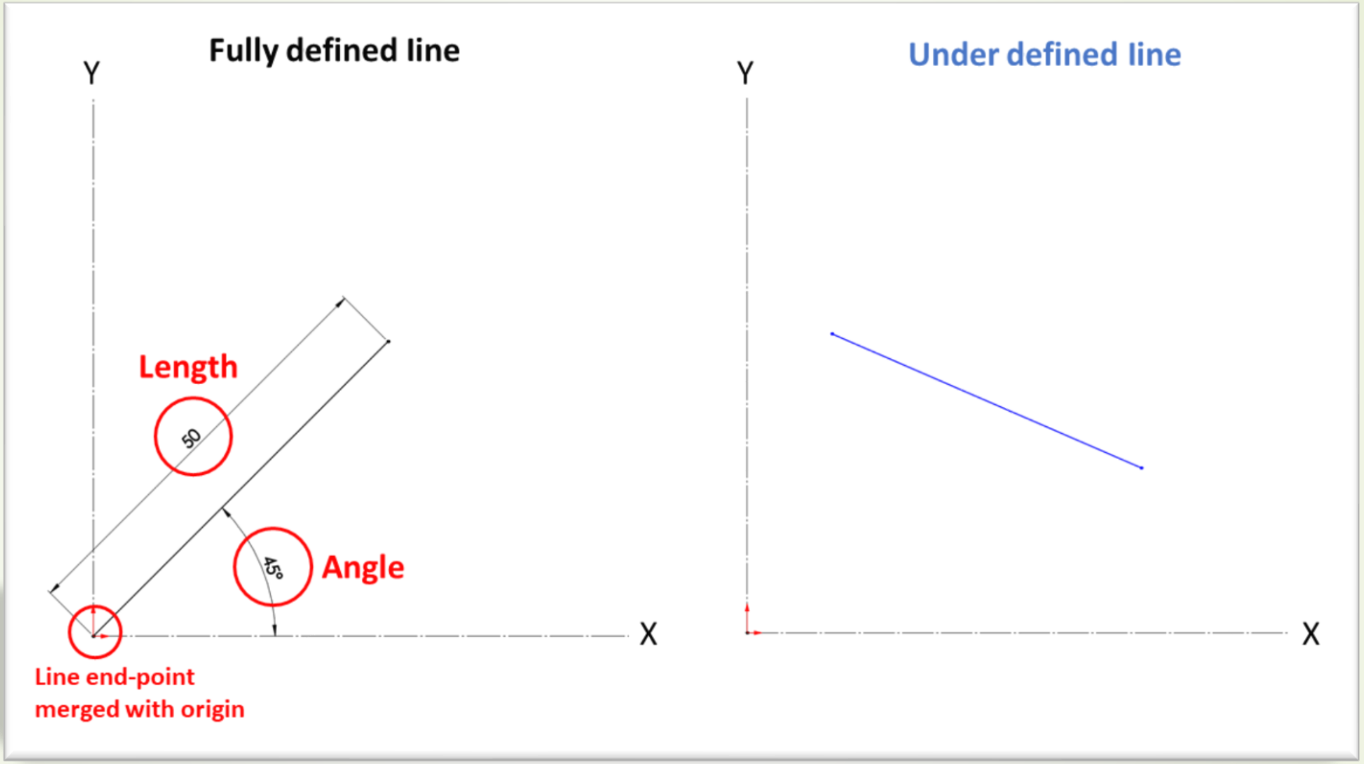
Dimensions: These represent distances and angles that can be defined with a number. Some examples are as follows:

- Lengths of lines
- The diameters and the radius of circles and arcs
- An angle between two lines

Relations: These represent geometric relations between the different parts of a sketch. Some examples are as follows:

- A line can have a *horizontal* relation to the sketch plane
- Two lines are *perpendicular* to each other
- Two circles are *concentric* to each other
- A line is a *tangent* to a curve

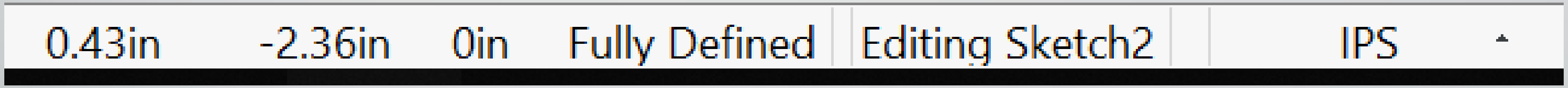
Let's look at a visual comparison between two lines. One is **Fully defined,** and one is **Under defined**:



Of the two preceding lines, the one on the left is fully defined while the one on the right is under-defined. The line on the left is defined as such because of the following:

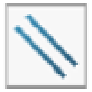
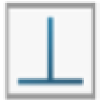
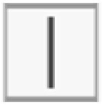
- One end coincides with the origin of the coordinate system
- The length of the line was defined as **50** millimeters
- The angle between the line and the x-axis is **45** degrees

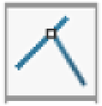
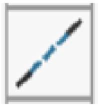
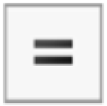
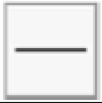
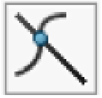
To make it easier to distinguish between fully defined and under defined sketches, SOLIDWORKS color-codes them. ***Black*** parts are fully defined, while ***blue*** parts are under defined. In addition to color-coding, SOLIDWORKS also indicates the status of our sketch below the canvas. The following screenshot shows an indication of a **Fully Defined** sketch. Other classifications include **Under Defined** and **Over Defined**:



Geometrical relations

The following table summarizes most of the geometrical relations we will come across while working with SOLIDWORKS sketching.

Midpoint		This can position a point so that it's in the middle of a line.
Parallel		This can make more than one line parallel to another.
Perpendicular		This can make two lines perpendicular to each other.
Tangent		This relates to circles, arcs, and other curved entities. It can turn a line tangent into a curved entity. It can also make more than one curved entity tangent to each other.
Vertical		This makes lines vertical to the sketch plane. In addition, it can make more than one point lie in a vertical line or in relation to another point.

Relation's Name	Relation Icon	Relation Function
Coincident		Coincident relations occur between points and lines, arcs, circles, and so on. This would make a point lie in other sketch entities, such as lines.
Colinear		This makes two or more lines lie in one direction.
Concentric		This can make two or more circles or arcs share the same center.
Coradial		This applies to two or more arcs if the different arcs share the same center and radius.
Equal		This makes two or more lines or arcs equal to each other in terms of length.
Equal Curve Length		This relation occurs between an arc or circle and a line. It makes the perimeter equal to the line in terms of length.
Fix		This fixes the selected sketch entity to where it exists at the time of setting the relation.
Horizontal		This makes lines horizontal to the sketch plane. In addition, it can make more than one point lie in a horizontal line.
Intersection		This will position a point at the intersection point of two lines. This includes the extension of the lines, as well as the lines themselves.
Merge		This can merge more than one point together into one point location.

Sketching lines

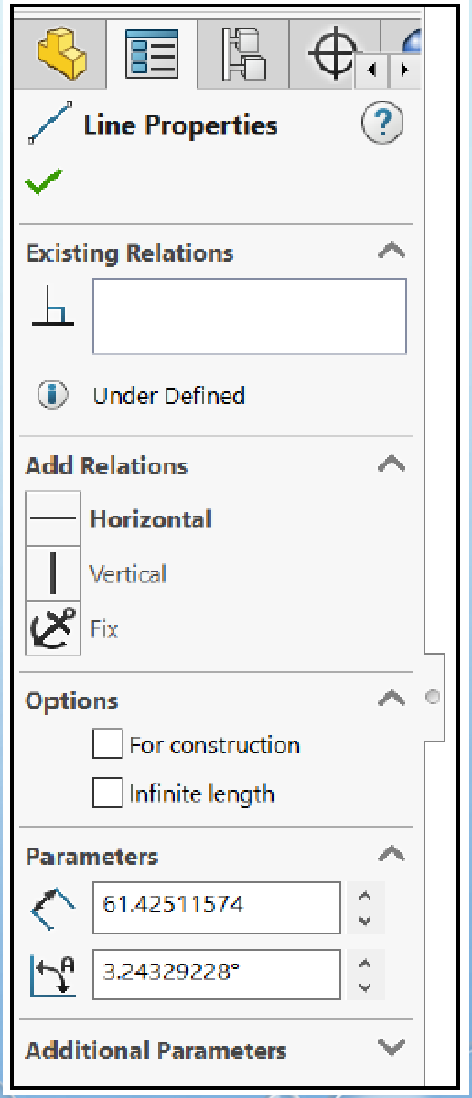
1. Select the **Line** sketch command.
2. Move the mouse cursor into the canvas and click on the first point location (selected by the designer according to the design).
3. Click on the second point location.
4. Exit the **Line** sketching command by pressing *Esc* on the keyboard.

Defining the Line

In the defining stage, we will work on defining our outline with the necessary dimensions and relations to fully define our sketch. Note that some of the relations are set automatically by SOLIDWORKS, according to how we place our lines.

Follow these steps for the defining stage:

1. Click on the line you aim to modify.
2. A new panel will appear on the left, in place of the design tree. It will be titled **Line Properties**, as shown in the following screenshot.

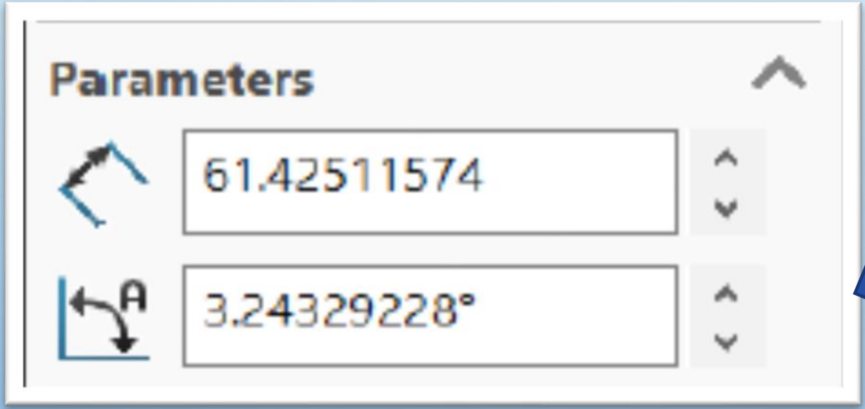


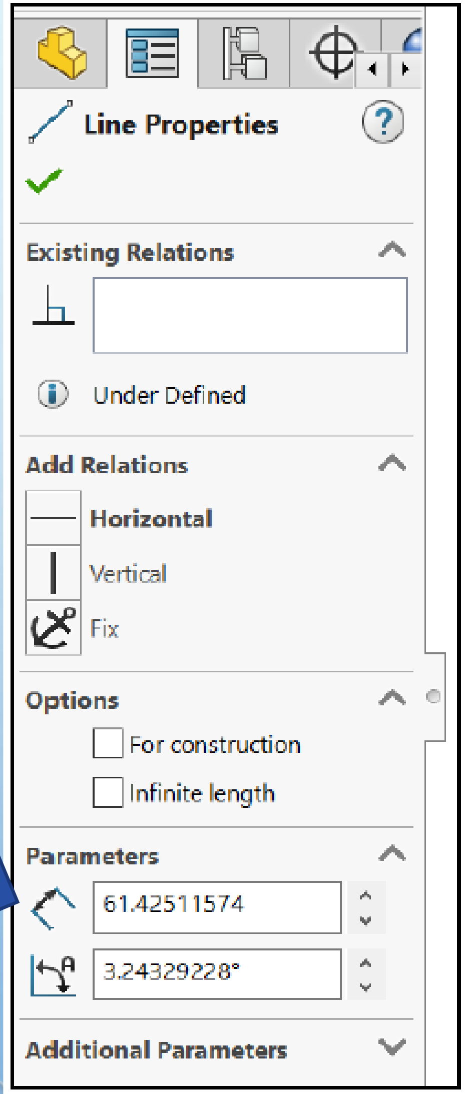
To define the Relations of the line :

Under **Add Relations**, you can choose one of the relations (**Horizontal** or **Vertical**) This will add the relation to the line. You will see a small icon appear next to the line, showing the relation type.

To define the Parameters of the line :

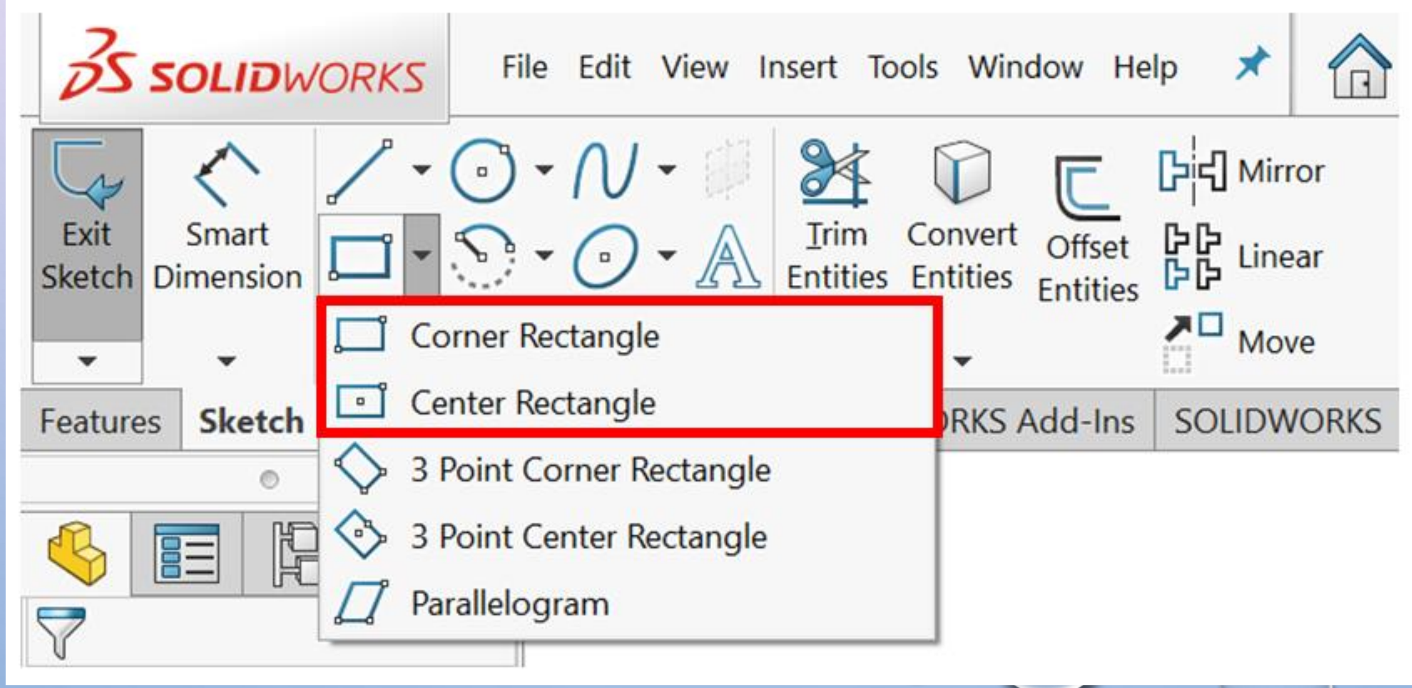
Under **Parameters**, you can set the **length** and the **slope angle** of the line.





Sketching rectangles and squares

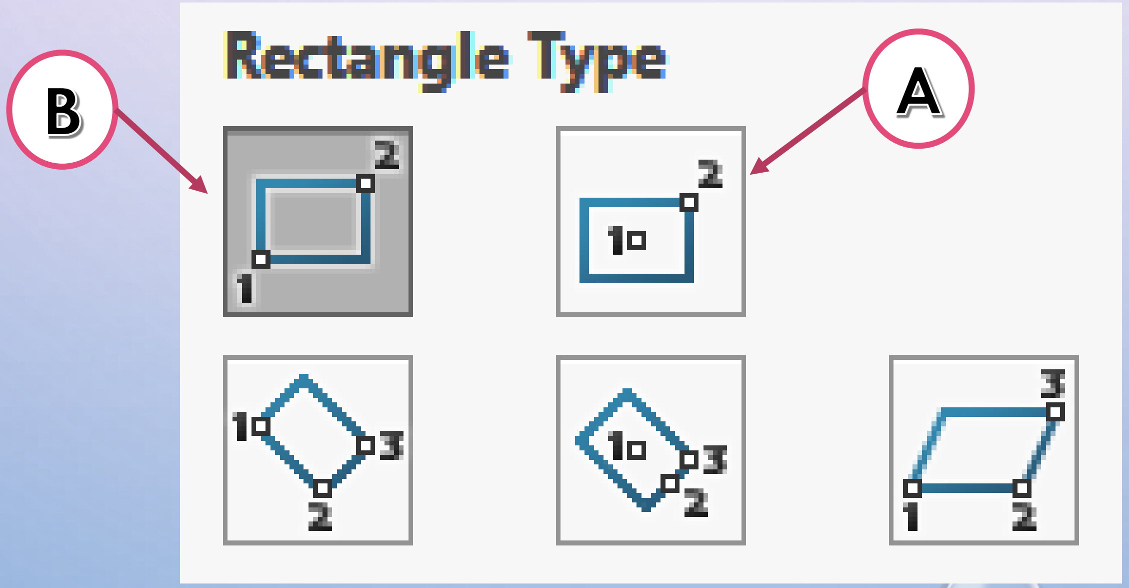
On the command bar, select the drop-down menu next to the rectangle shape we will see five commands, which are shown in the following screenshot: **Center Rectangle, Corner Rectangle, 3 Point Corner Rectangle, 3 Point Center Rectangle** (All commands create rectangles) **And Parallelogram.**



The difference is how those rectangles are created.

A. **Center Rectangle** is created with two clicks: ***one*** indicating the center and the ***other*** indicating a corner.

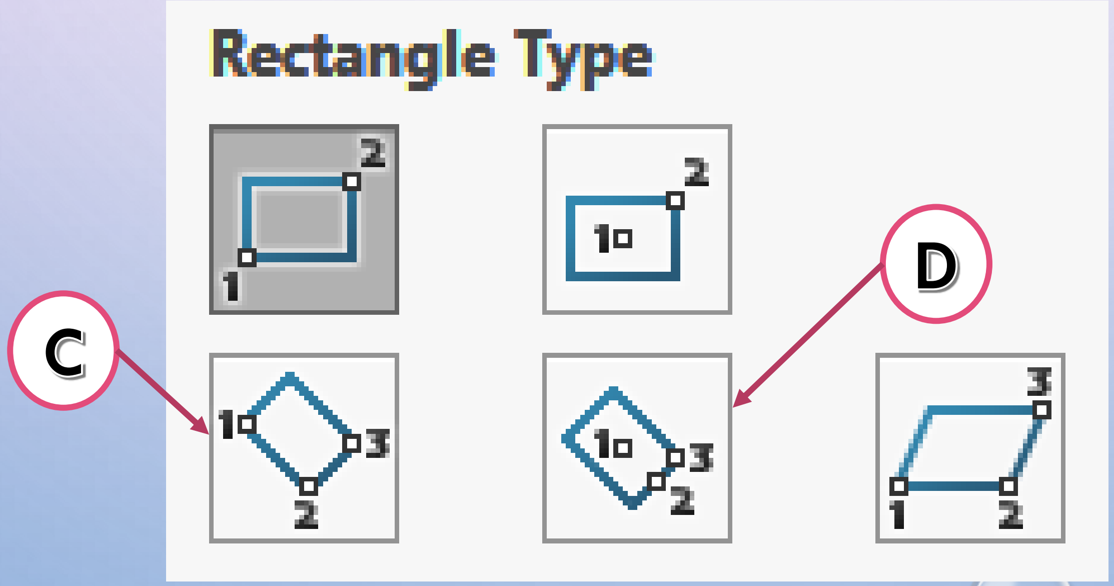
B. **Corner Rectangle** is created by two clicks, indicating the opposing corners.



C. **3 Point Corner Rectangle** is created with three clicks:

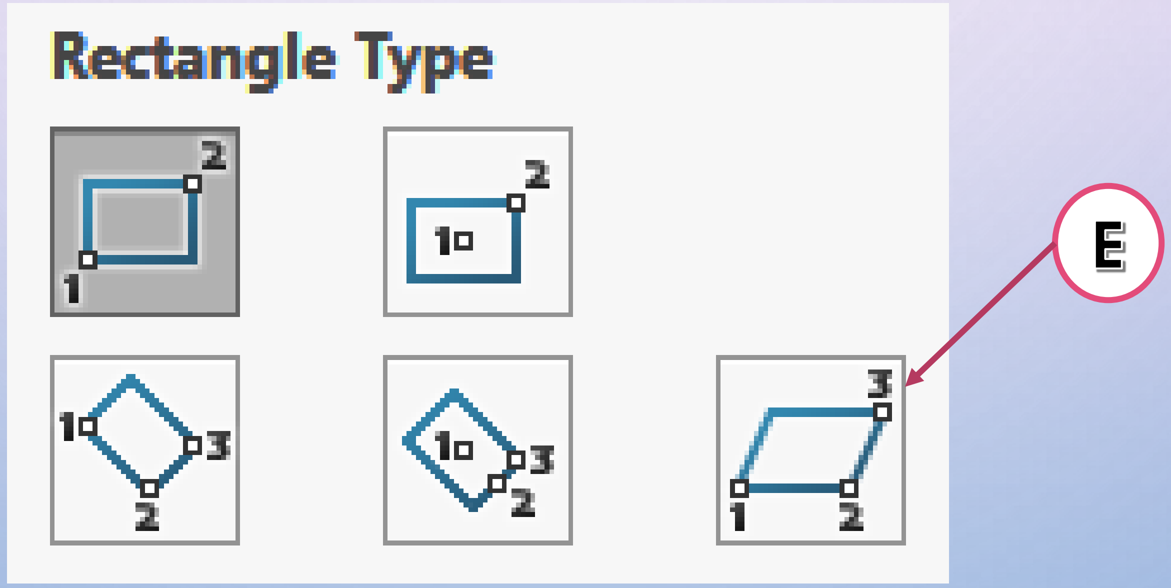
Everyone indicating a corner.

D. **3 Point Center Rectangle** is created with three clicks: ***one*** indicating the center and the ***Second*** indicating the center of one side and the ***other*** indicating a corner.



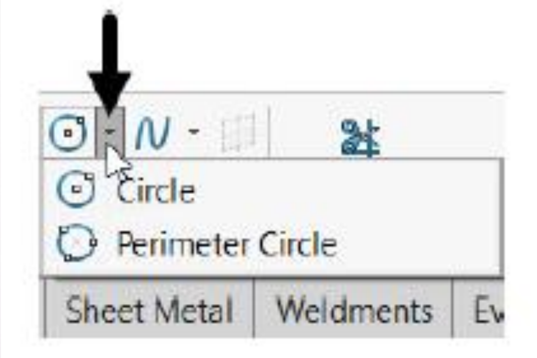
E. Parallelogram

This command creates a parallelogram by using three points



Sketching circles and arcs

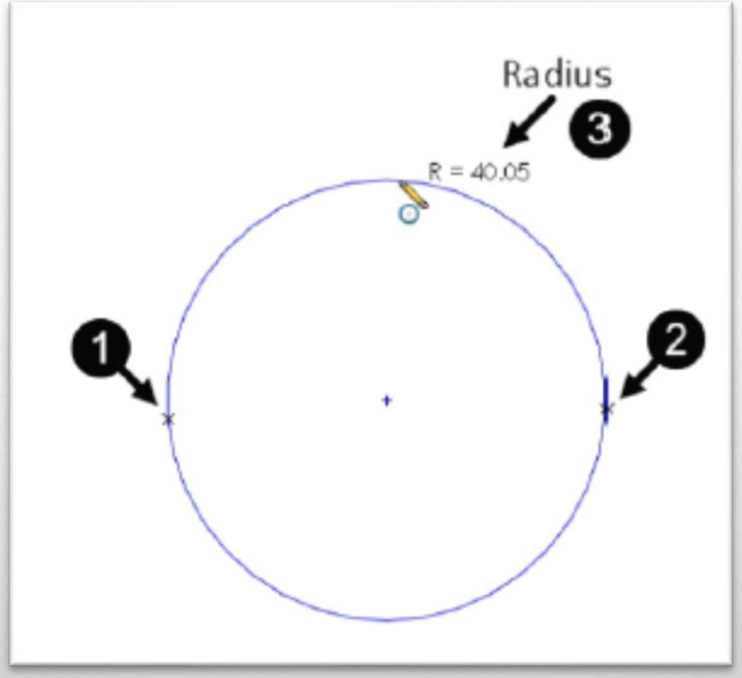
This tool is used to draw a circle during drawing a sketch. There are two options available in the Circle flyout for creating circles:



Circle : This is the most common and easiest way to draw a circle, it's created by two clicks:

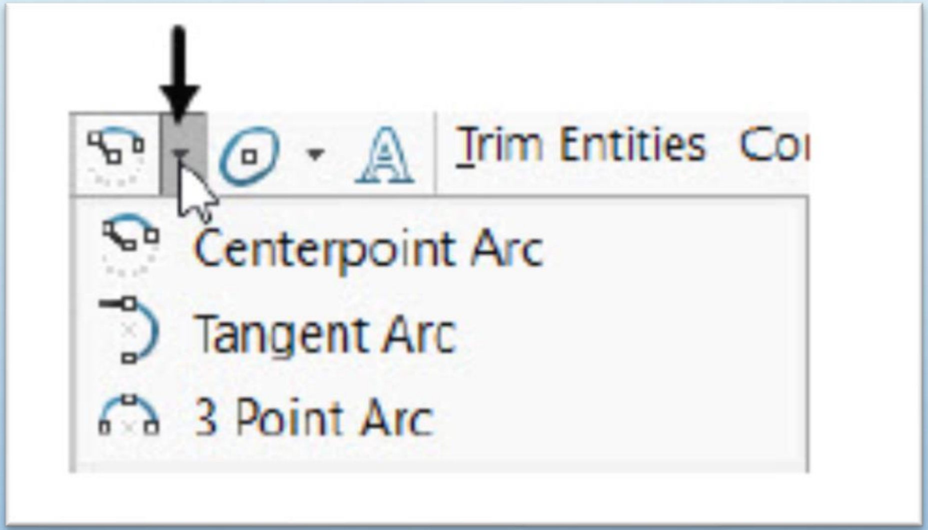
1. Click to define the center point of the circle.
2. Move the cursor up to some distance and click to define the radius of the circle.

Perimeter Circle : This tool is used to create a circle by using three points, The first two points define the location of the circle, and the third point defines its radius.

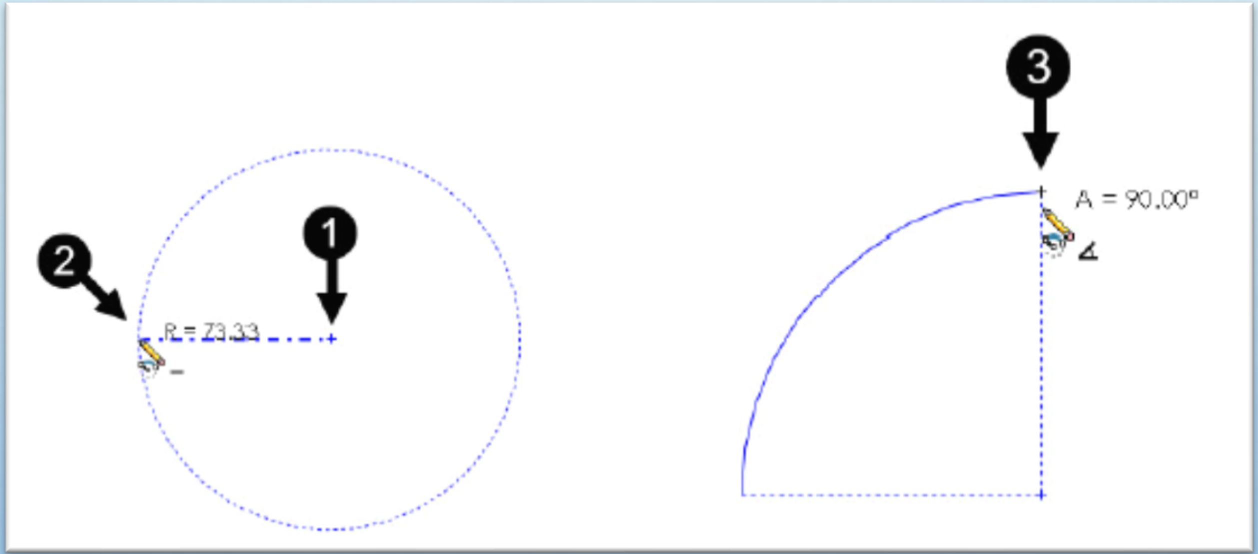


Sketching Arc

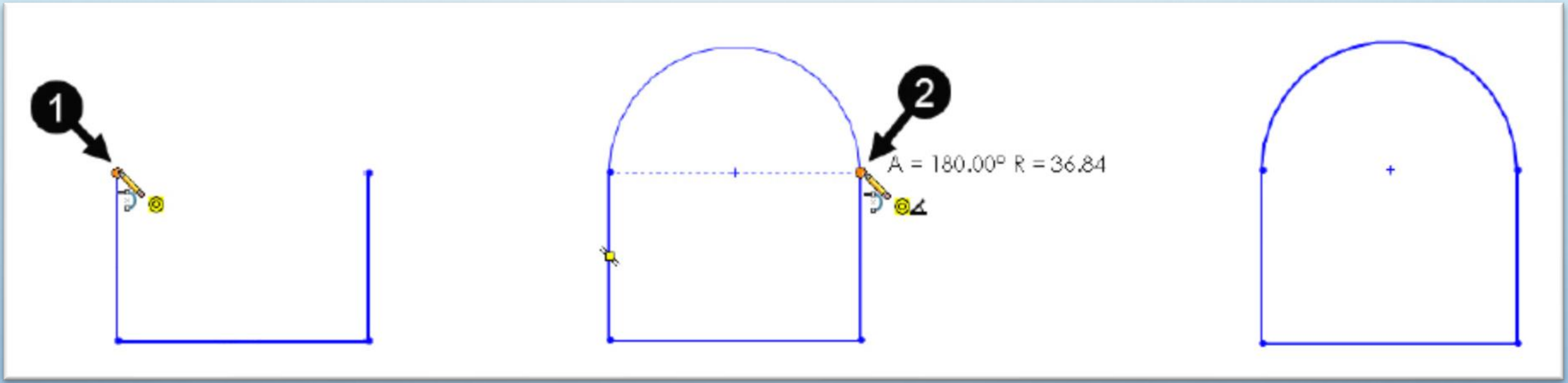
This tool is the used to draw an arc during drawing a sketch. There are three different tools available in the Arc flyout to create three different types of arcs.



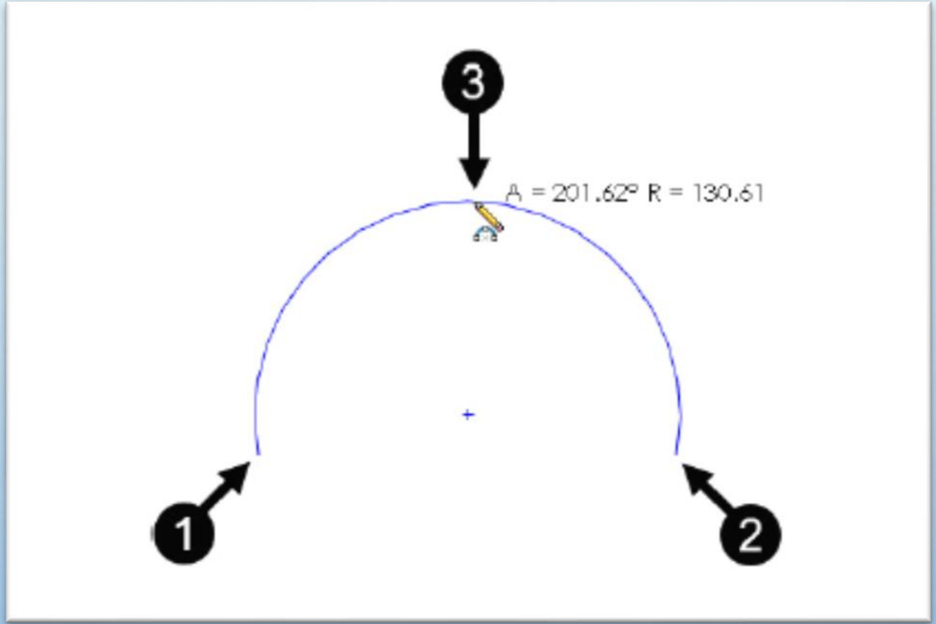
A. Center point Arc Tool : This tool is used to draw an arc by defining center point, start point and end point.



B. Tangent Arc Tool : This tool is used to draw arc tangent to an existing entity.

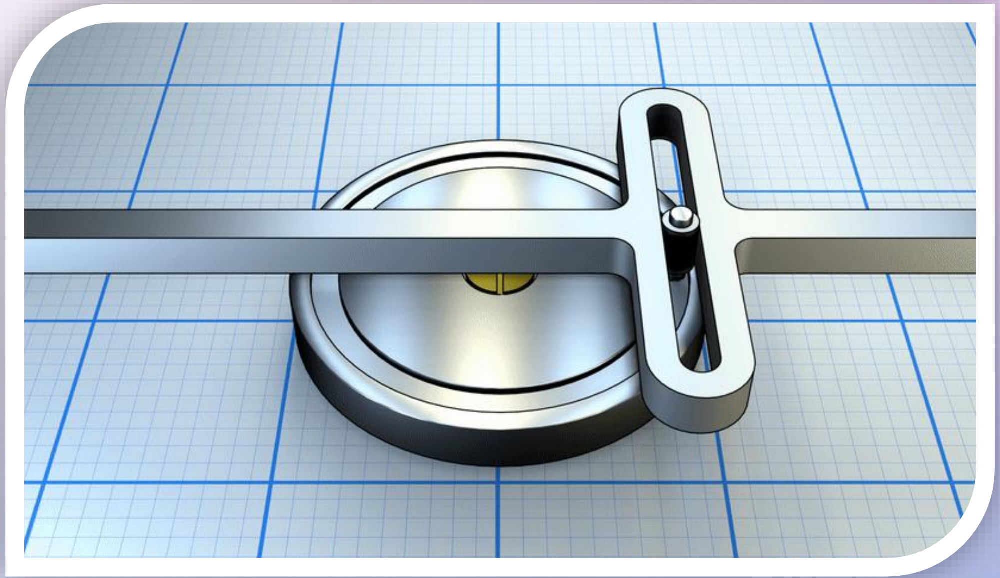


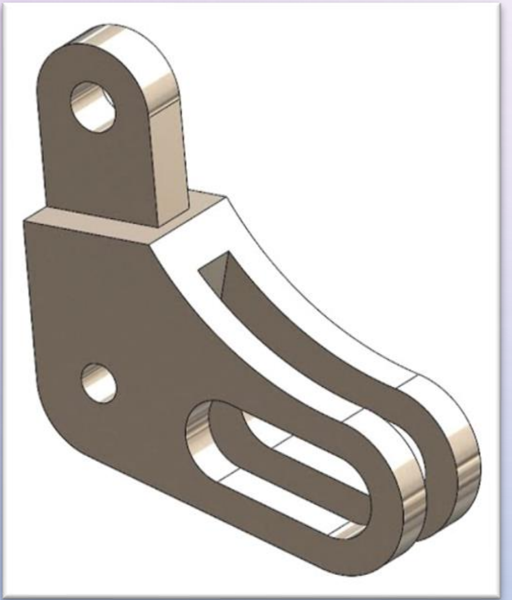
C. 3 Point Arc : This tool is used to create an arc by defining its two endpoints, and a radius.

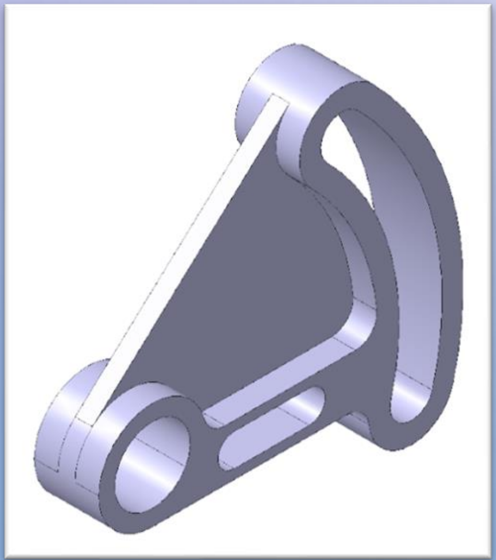


Sketching Slot tool

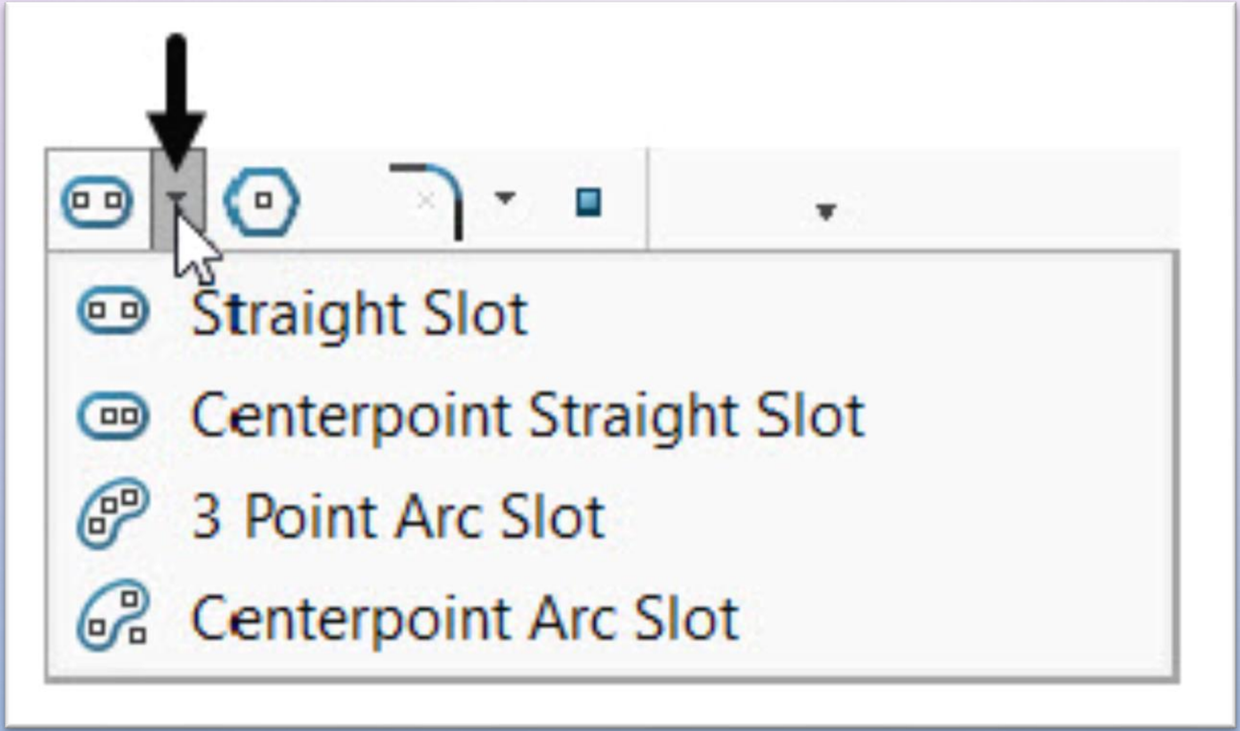
These tools are used to create slots. There are four different type of tool in the Slot flyout to create slots, as shown.





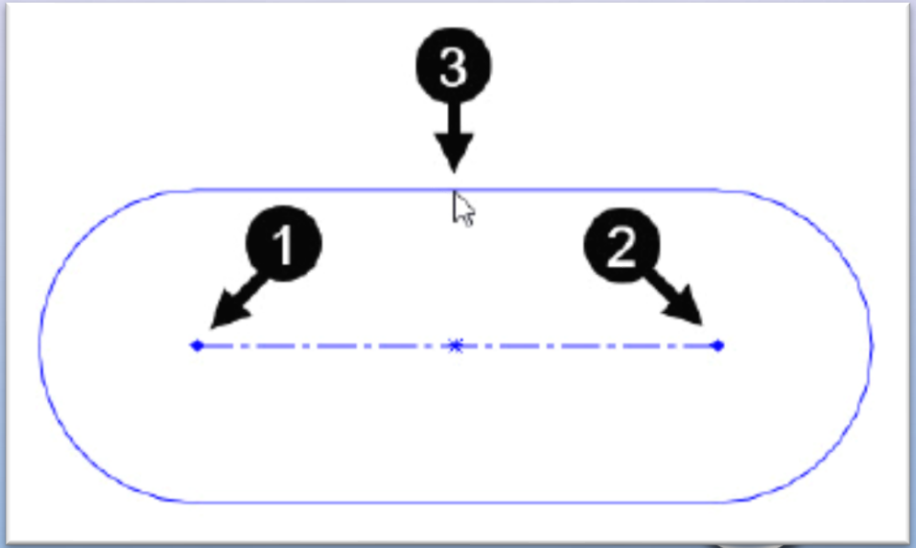


There are four different type of tool in the Slot flyout to create slots, as shown.



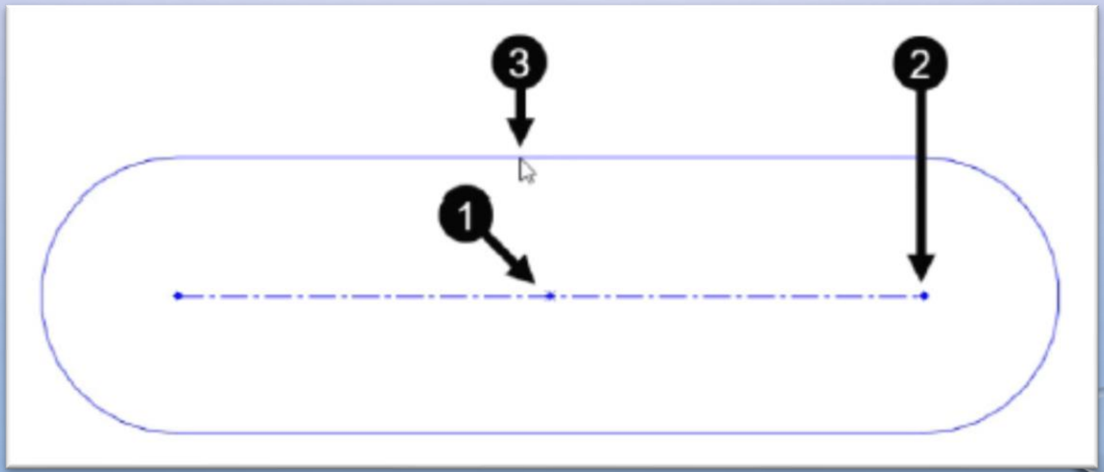
A. Straight Slot Tool : This tool is used to create a straight slot by defining three points – (1) start point, (2) end point, and (3) point that defines its width .

1. Click in the drawing area to define the point (1) as first end/start point of the straight slot.
2. Move the cursor up to some distance and click to define point (2) as second end point of the slot to display the preview of slot.
3. Again, move the cursor up to some distance and click to define point (3) for width of Straight Slot.



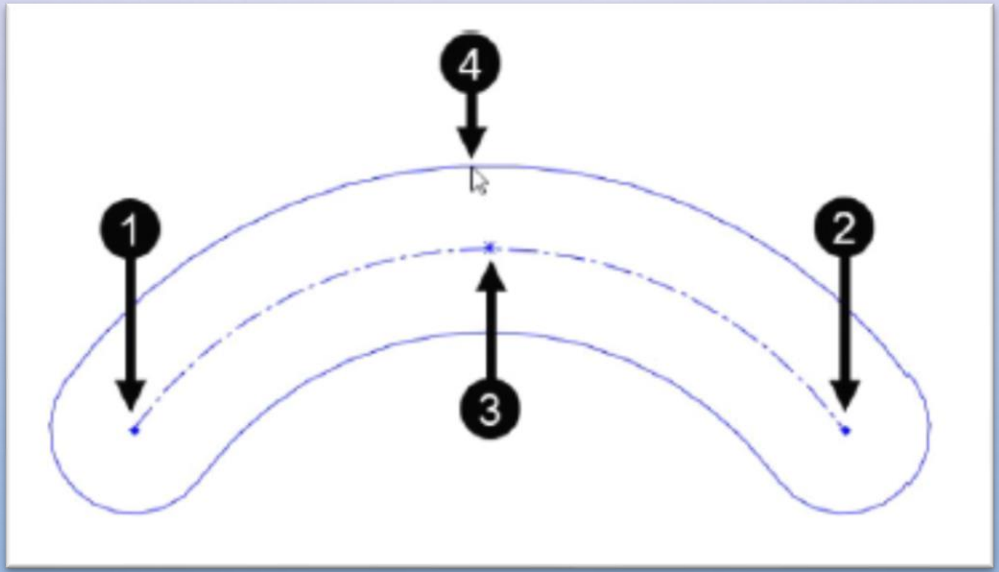
B. Center point Straight Slot Tool : This tool is used to create a slot by defining three points - center point, endpoint, and point for width.

1. Click in the drawing area to define the point (1) as center point of the Center point Straight Slot.
2. Move the cursor up to some distance and click to define point (2) as end point of the slot and display the preview of slot.
3. Again, move the cursor up to some distance and click to define point (3) for width of Center point Straight Slot and display the preview of slot.



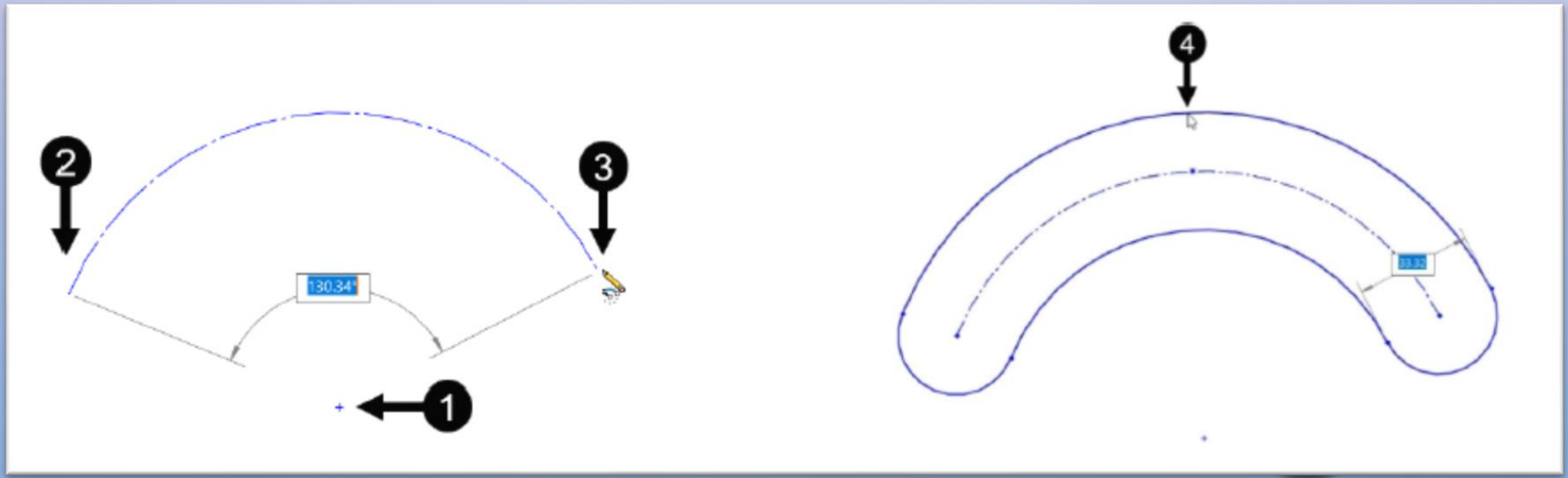
C. 3 Point Arc Slot Tool : This tool is used to create a 3 Point Arc Slot by defining four points.

1. Click in the drawing area to define the point (1) as start point of the 3 Point Arc Slot.
2. Move the cursor up to some distance and click to define point (2) as end point of the slot and display the preview of slot.
3. Again, move the cursor up to some distance and click to define point (3) and radius of Slot.
4. Now, move the cursor up to some distance and click to define point (4) for width of Slot and display the slot.



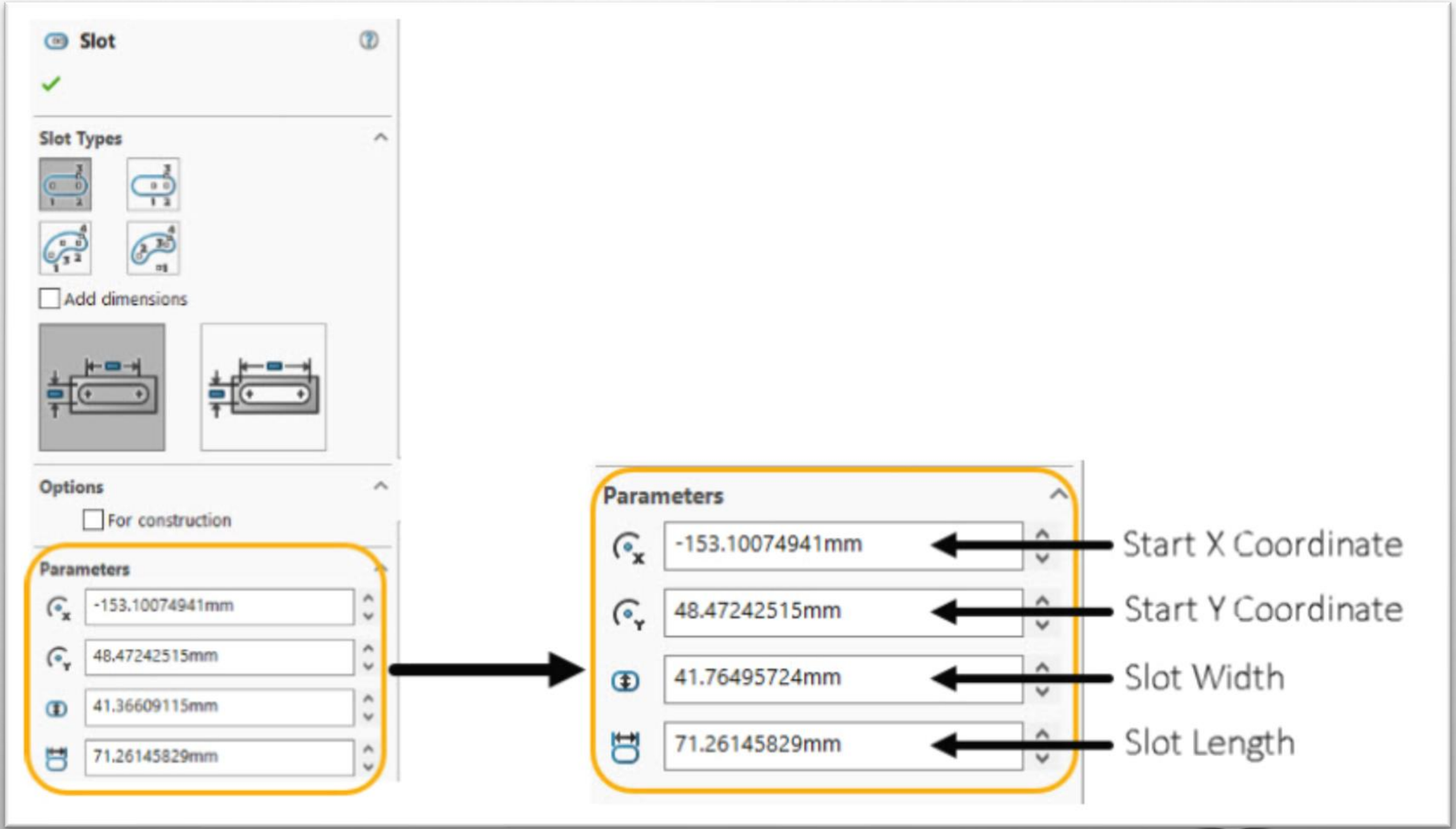
D. Centerpoint Arc Slot Tool : This tool is used to create an arc slot using a center point as a reference.

1. Click in the drawing area to define the point (1) as center point of the Centerpoint Arc Slot.
2. Move the cursor up to some distance and click to define point (2) as start point of the slot.
3. Move the cursor up to some distance and click to define point (3) as end point of the slot and display the preview of slot.
4. Again, move the cursor up to some distance and click to define point (4) for width of Slot to display preview of the slot.



Modify the Slot

The Slot Property Manager with active edit boxes under Parameters rollout get displayed ,you can modify the slot by entering required values in their respective edit boxes if required, as shown.



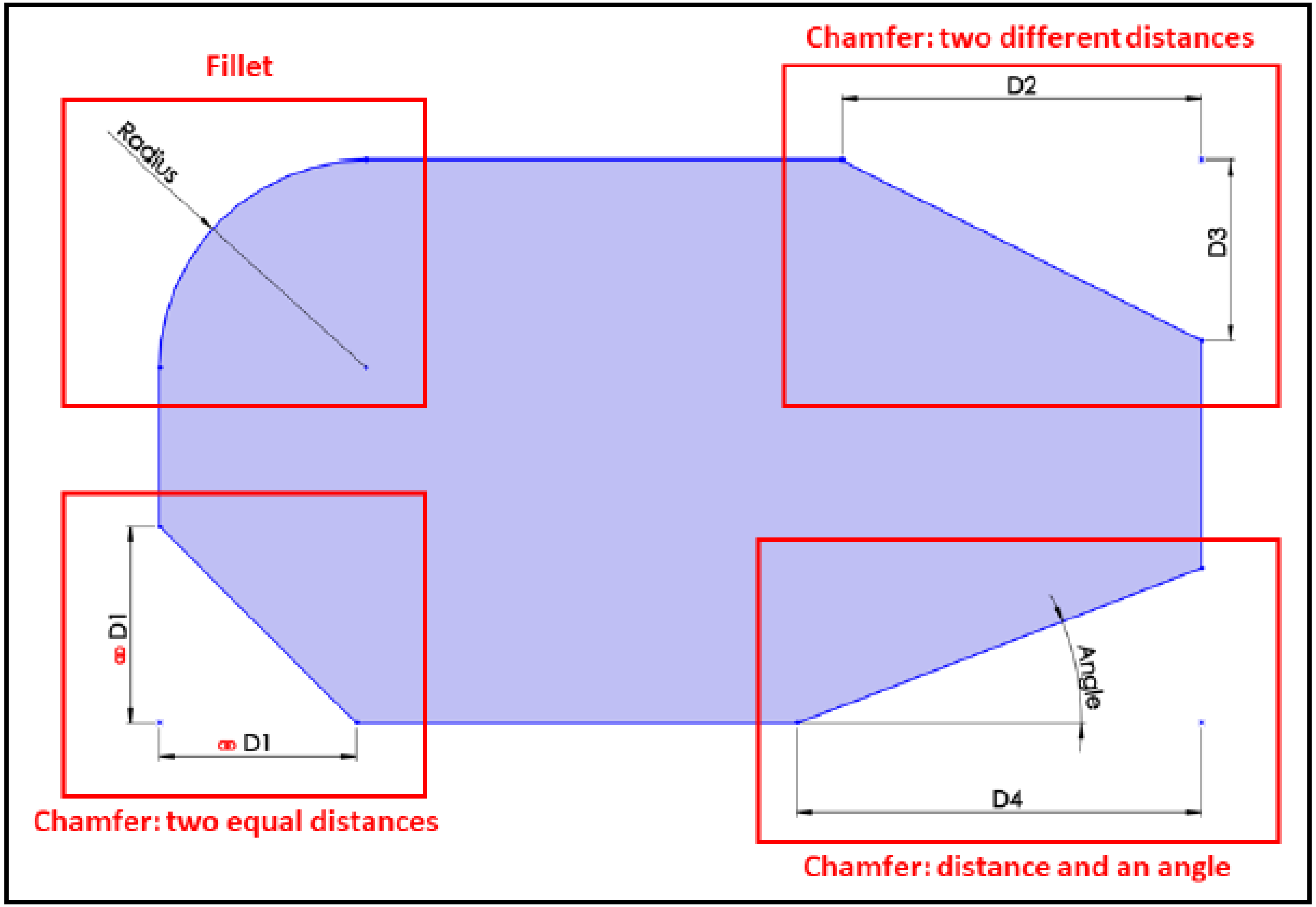
Fillets and chamfers

In this section, we will discuss making fillet and chamfers for our sketches. Fillets and chamfers can be applied between two sketch entities, usually between two lines. They are defined as follows :

Fillets: Fillets can be viewed as a type of arc. Thus, they are defined in the same way, that is, with a center and a radius.

Chamfers: Chamfers can be defined in different ways. These include two equal distances, two different distances, or a distance and an angle.

The following image illustrates the shapes of fillets and chamfers, as well as how they are defined:



Special Sketching Commands

Mastering SOLIDWORKS sketching is not only about sketching shapes such as rectangles and ellipses; it also depends on other supporting functions such as mirroring, patterns and trimming. These special commands will greatly enhance our ability to sketch complex shapes faster.

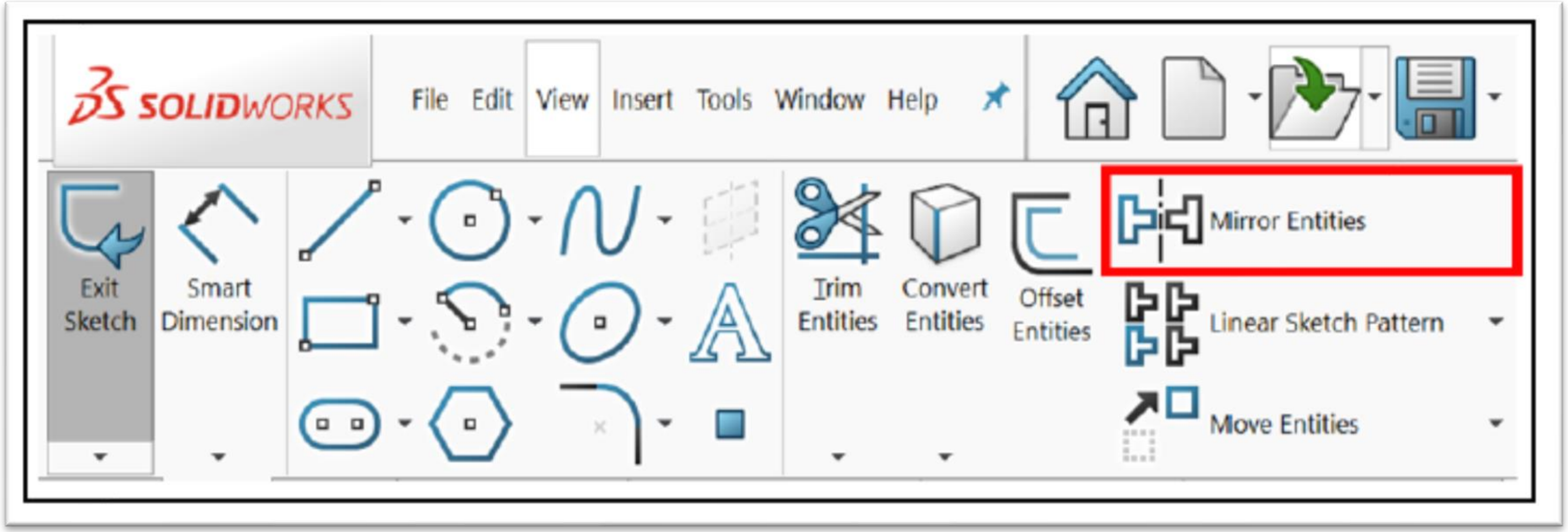
We will cover the following topics:

- Mirroring and offsetting sketches
- Creating sketch patterns
- Trimming in SOLIDWORKS sketching

Mirroring a sketch

As the name suggests, mirroring a sketch means to reflect one or more sketch entities around a mirroring line. It is very similar to reflecting an image in a mirror. The following diagram illustrates the components of mirroring in SOLIDWORKS:

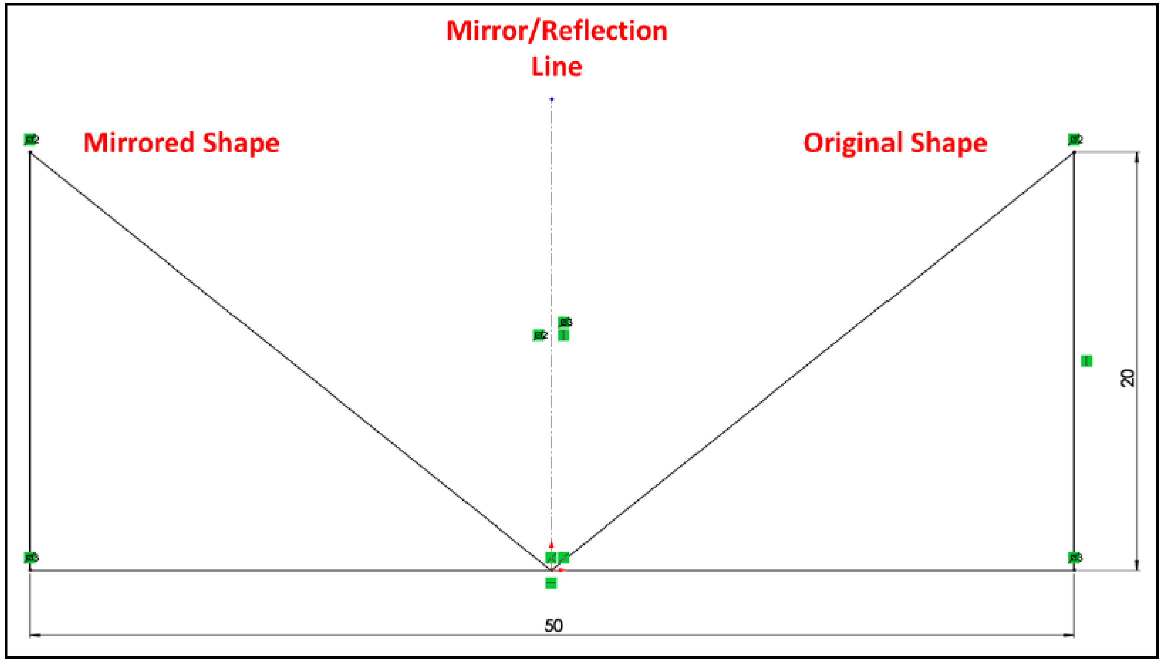
To start applying the sketch Mirror Entities command - Select the Mirror Entities command, as shown in the following screenshot :



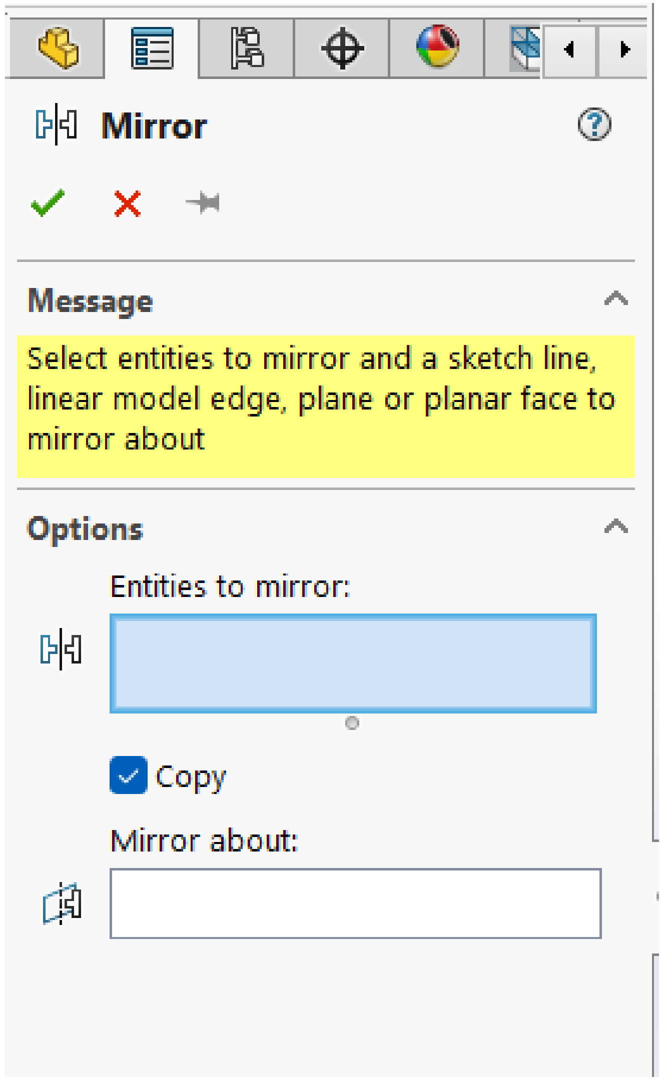
We can see that there are two parts that we need in order to use mirroring:

- Sketch entities to mirror
- A mirroring or reflection line

Since the two shapes are mirror sketch entitles of each other, any changes that happen to one shape will automatically happen to the other.



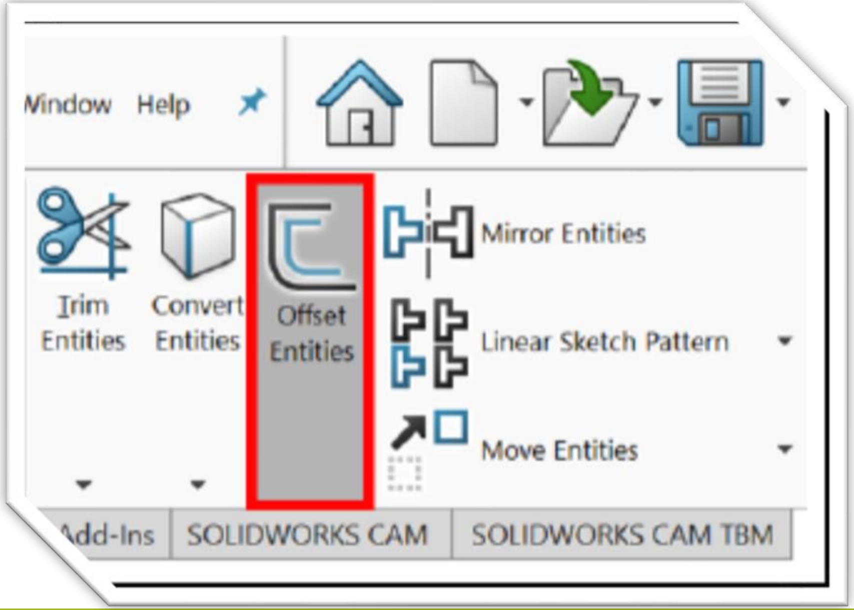
On the left-hand side, we will see the available Mirror options.

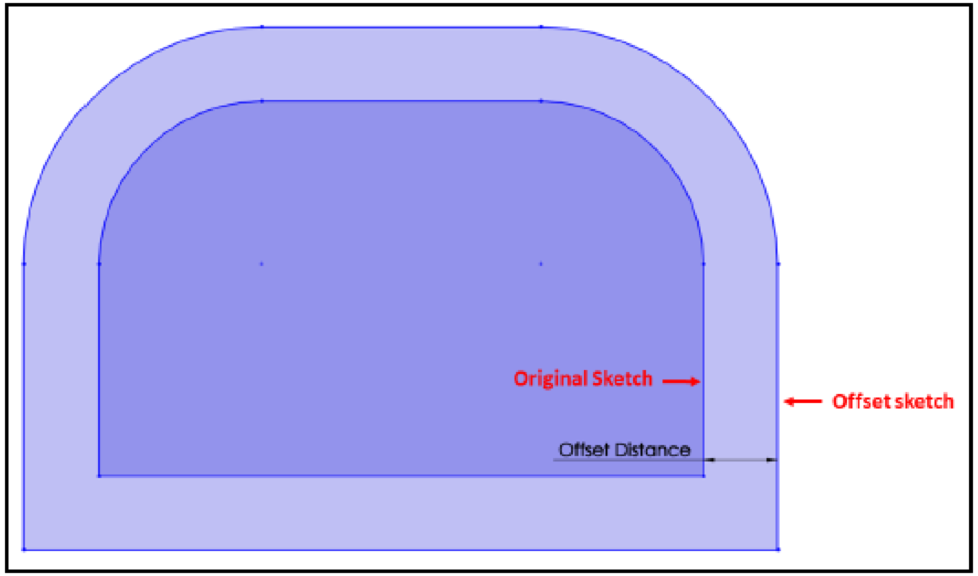


Offsetting a sketch

Offsetting a sketch makes it easier for us to duplicate existing sketch entities at an offset from the original sketch entity.

The following diagram shows an example of a sketch and its Offset Distance from the Original Sketch. Note that the original image is the one we sketch first; after that, the Offset sketch is created by applying the Offset Entities sketch command. The Offset sketch is defined by inputting an Offset Distance:

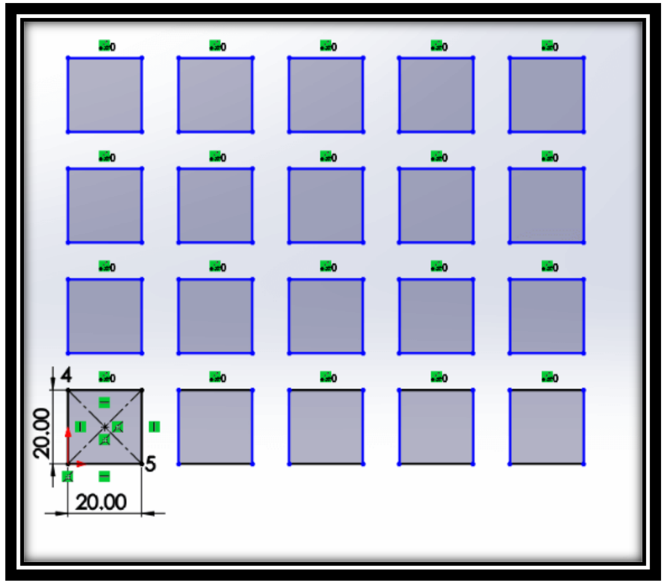


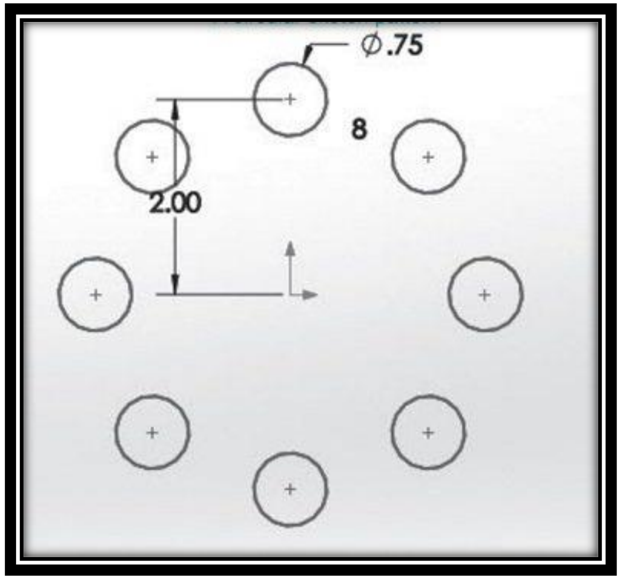


Creating sketch patterns

Patterns are repeated formations that can be commonly found in consumer products, architecture, fabrics, and more.

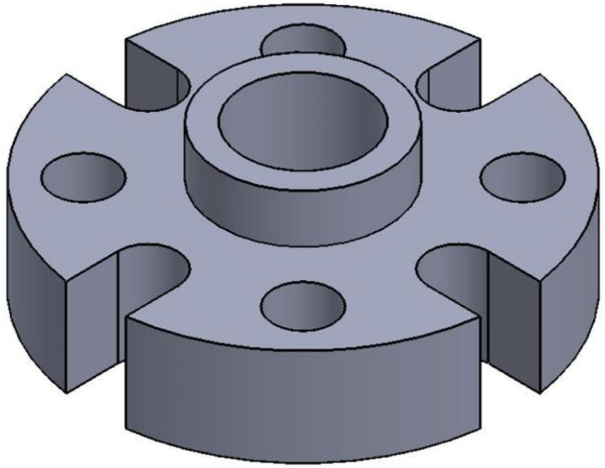
In patterns, we often have a base shape, sometimes called a **base cell** or **patterned entity**, which is created from scratch. Then, the basic shape is repeated multiple times. This is then repeated multiple times to form a bigger piece. There are two common types of patterns: **linear patterns** and **rotational patterns**

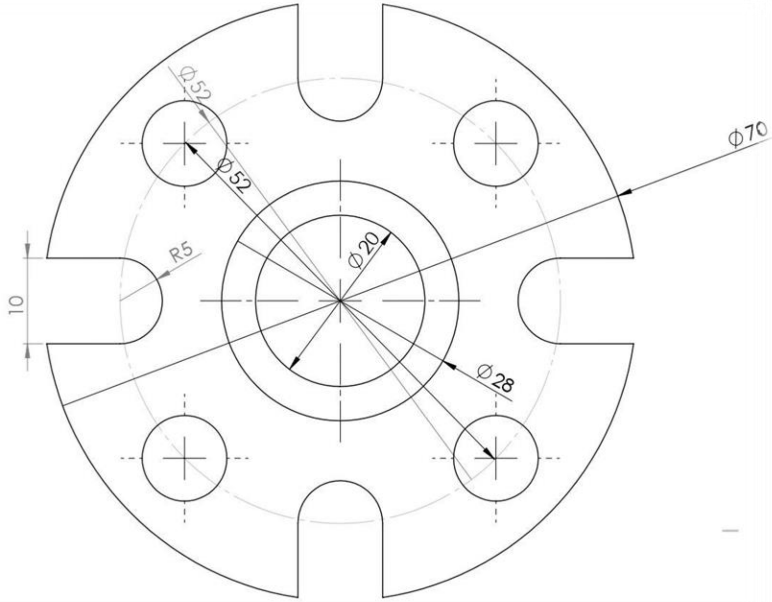


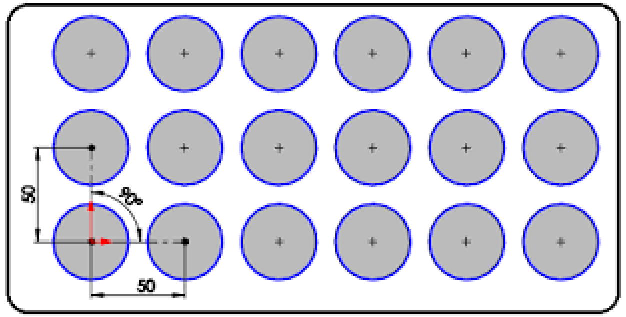


Rotational patterns

Circular patterns

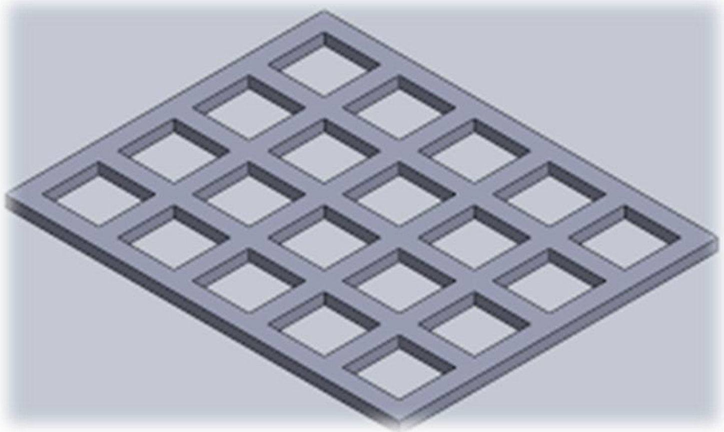


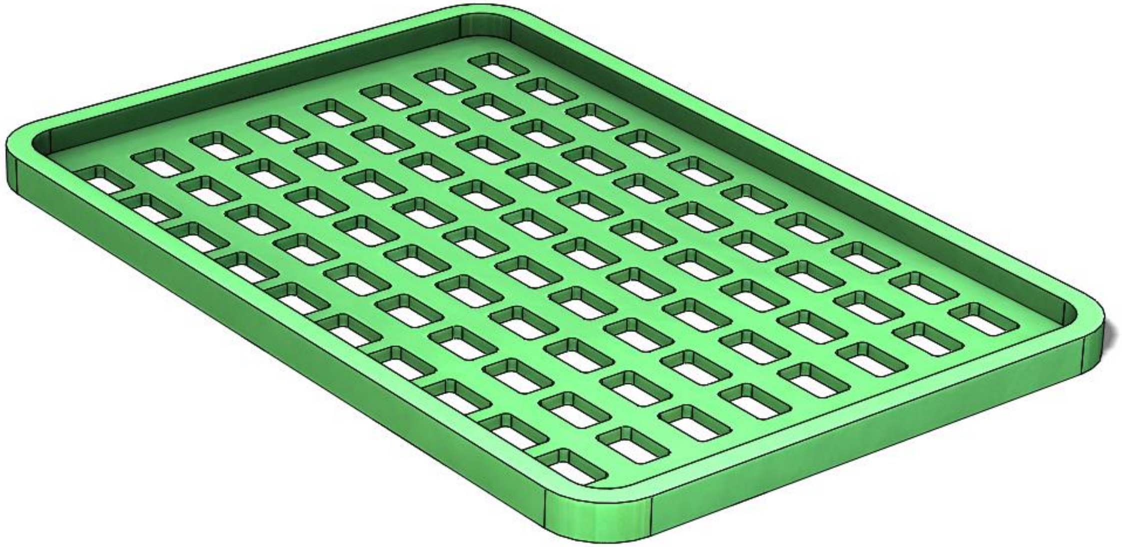




Linear patterns

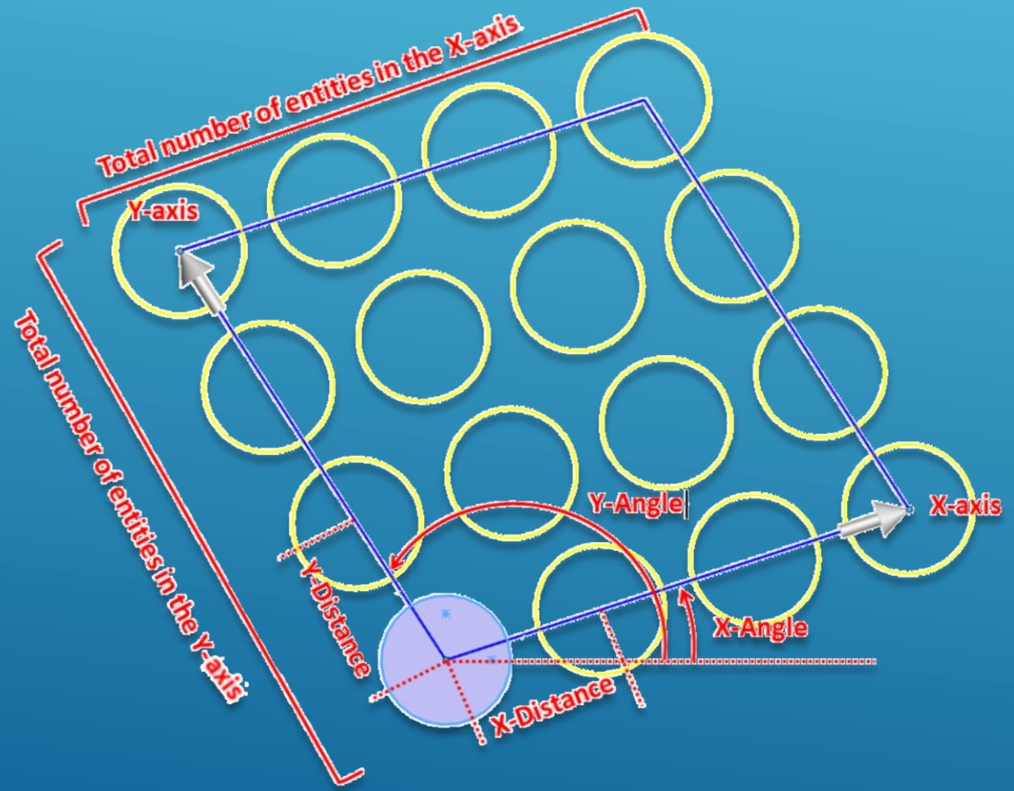
linear patterns





Linear sketch patterns

Linear sketch patterns allow us to pattern sketch entities in a linear direction. The following sketch shows us how we can define linear patterns in SOLIDWORKS sketching



In the preceding sketch, the shaded circle is the base circle, while the other ones are additions to be made with the pattern command. The annotations in writing are the parameters that we need in order to communicate a pattern with SOLIDWORKS sketching. Each of the annotations is repeated twice, one for the **X-axis** and one of the **Y-axis**. They are as follows:

- **Axes**: The **X-axis** and **Y-axis** represent the direction in which our pattern is implemented.

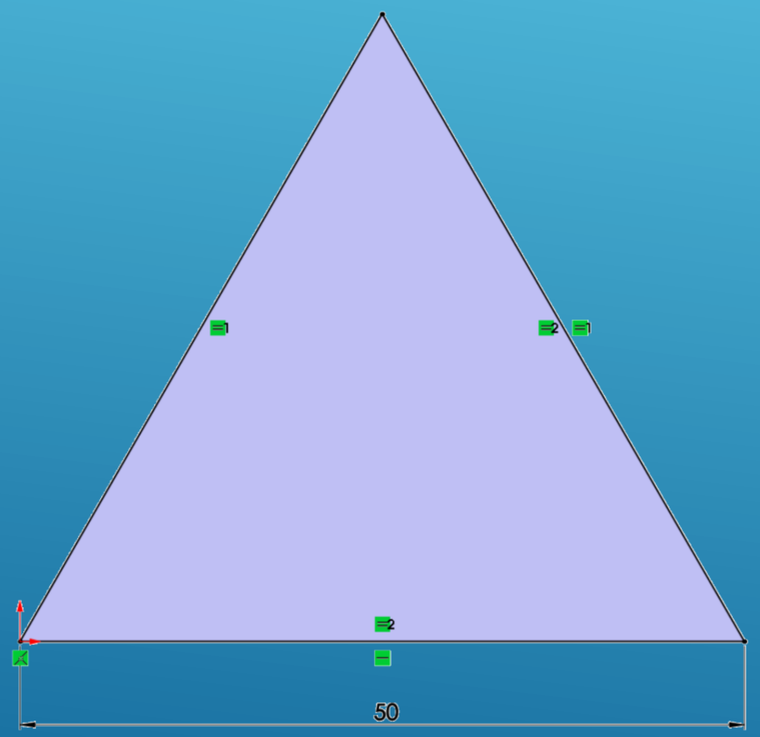
- **Total number of entities**: The number of times we want the entity to be sketched, including the base entity. In the preceding sketch, the number of entities is four for the *X* and *Y* directions.

- **Distance**: This specifies the distance that divides every two entities from each other. In the preceding sketch, the distance between every two patterned entities in the *X* and *Y* directions is 10 mm.

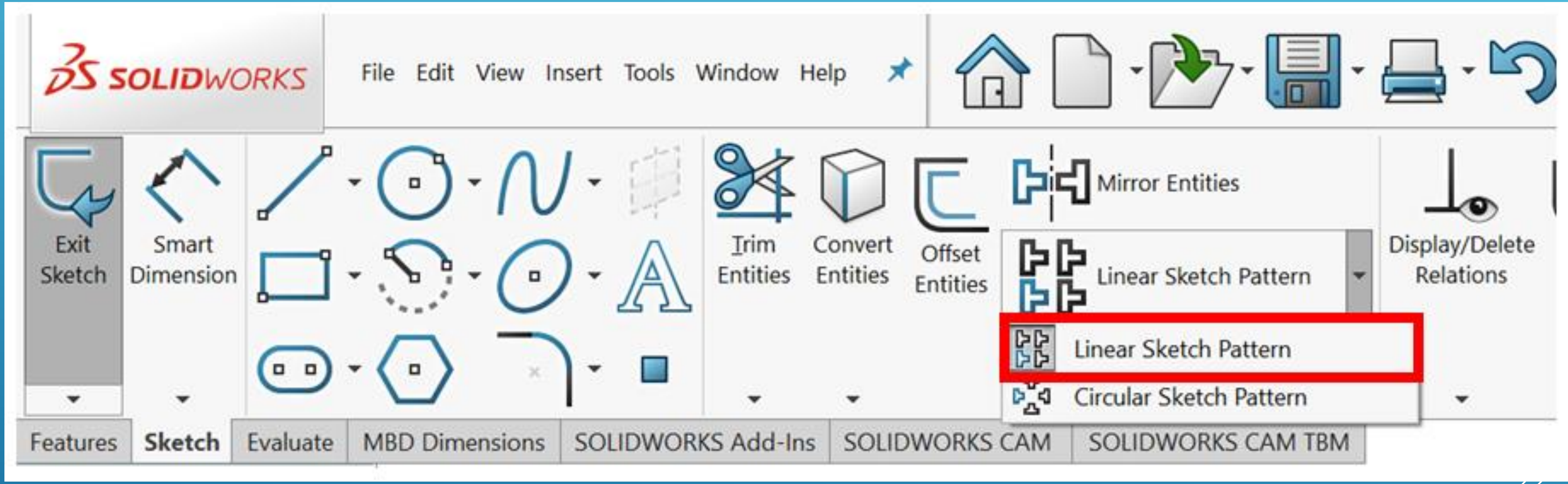
- **Angle**: This specifies how tilted our axes should be since the axes determine the direction of the patterns. Therefore, the whole pattern will shift as we change the direction of an axis. In the preceding sketch, the *X* and *Y* angles are 30 and 120 degrees, respectively. Note that the *X* and *Y* angles start from the same baseline.

To sketch the preceding diagram, follow these steps:

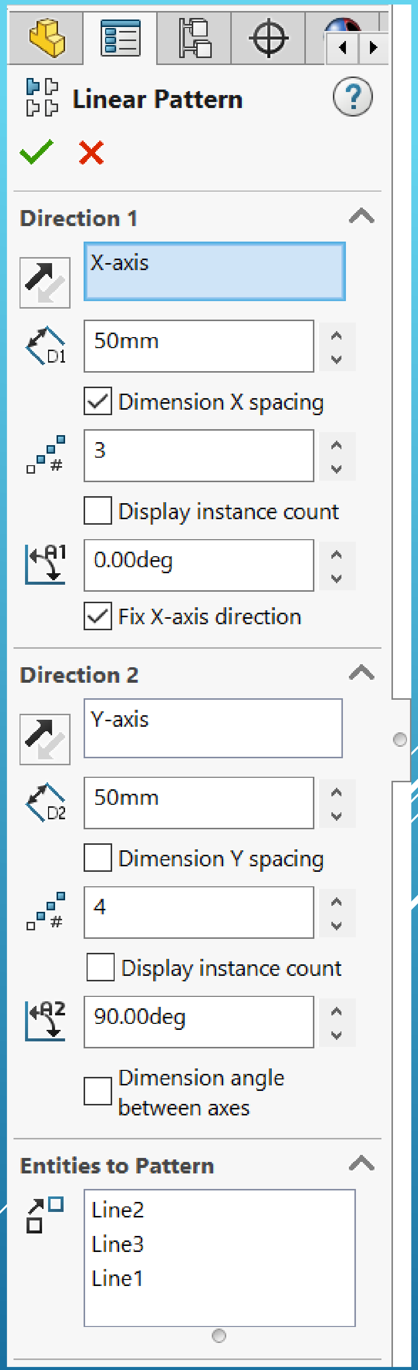
1. Sketch and define the base equilateral triangle, as follows:



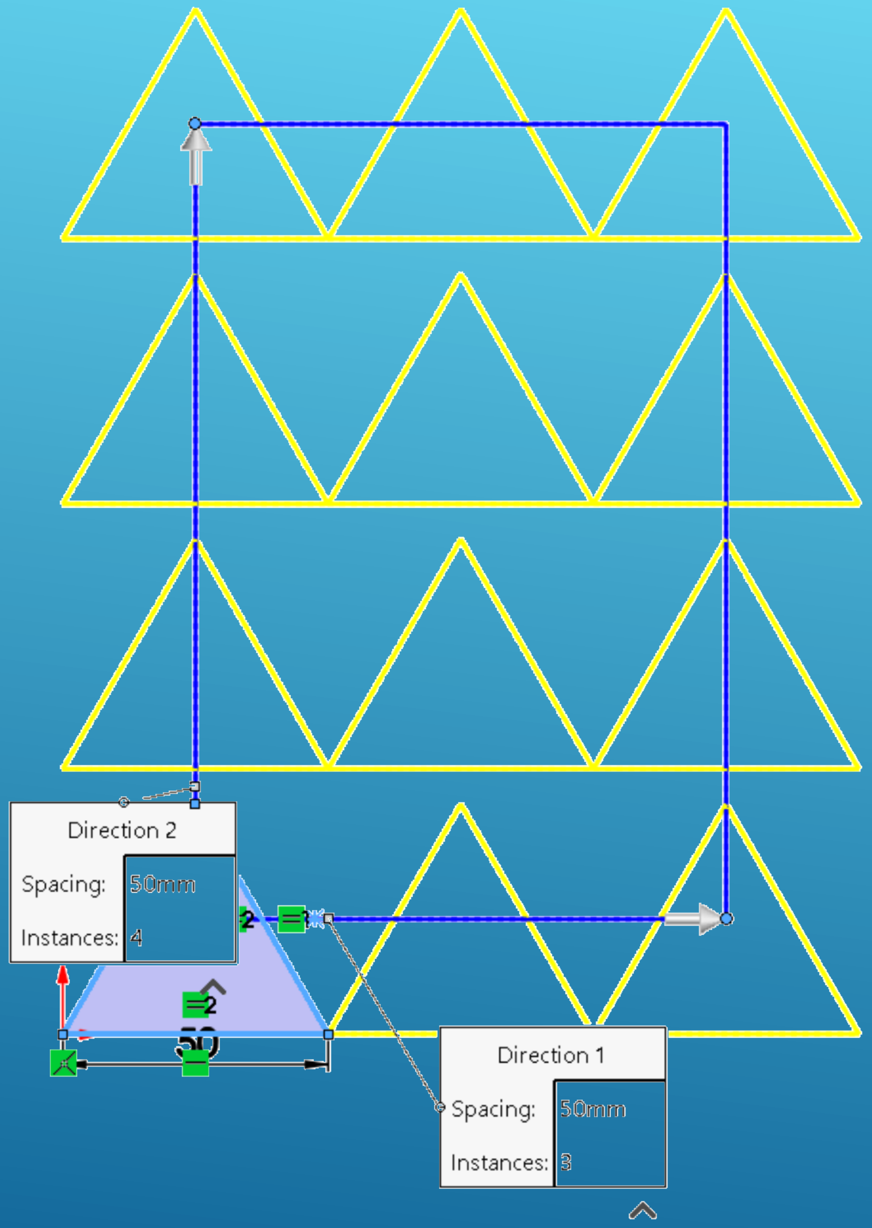
2. Select the **Linear Sketch Pattern** command:



3. Select the three lines that form our base triangle. Now, we can set up our **Linear Sketch Pattern** using the options that appear on the left-hand side of SOLIDWORKS. Set the following options:

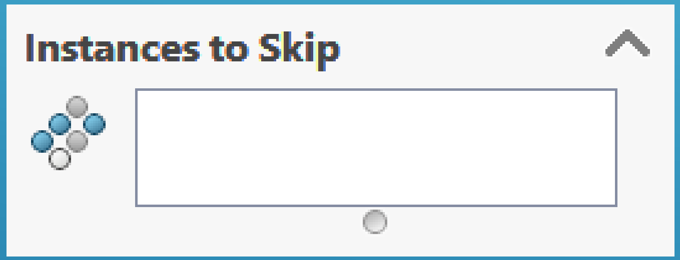


As we are adjusting those options, a preview of the final shape will appear on the canvas.



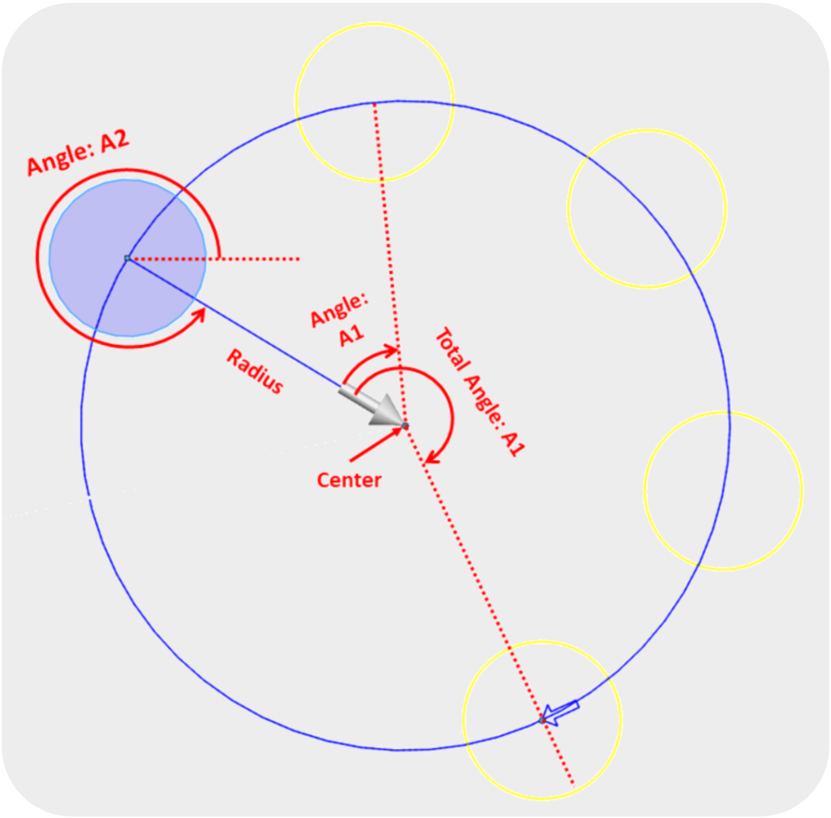
Let's have a look at some related commands:

- **Instances to Skip**: At the bottom of the **Linear Pattern** options, we will find the **Instances to Skip** option. We can use this to skip instances of the pattern. For example, in the preceding exercise, we can remove the two middle triangles by adding them to the **Instances to Skip**. This will exclude the middle triangles from the pattern:



Circular sketch patterns

Circular sketch patterns allow us to pattern sketch entities in a circular direction. The following sketch highlights how we can define a circular pattern in SOLIDWORKS sketching:

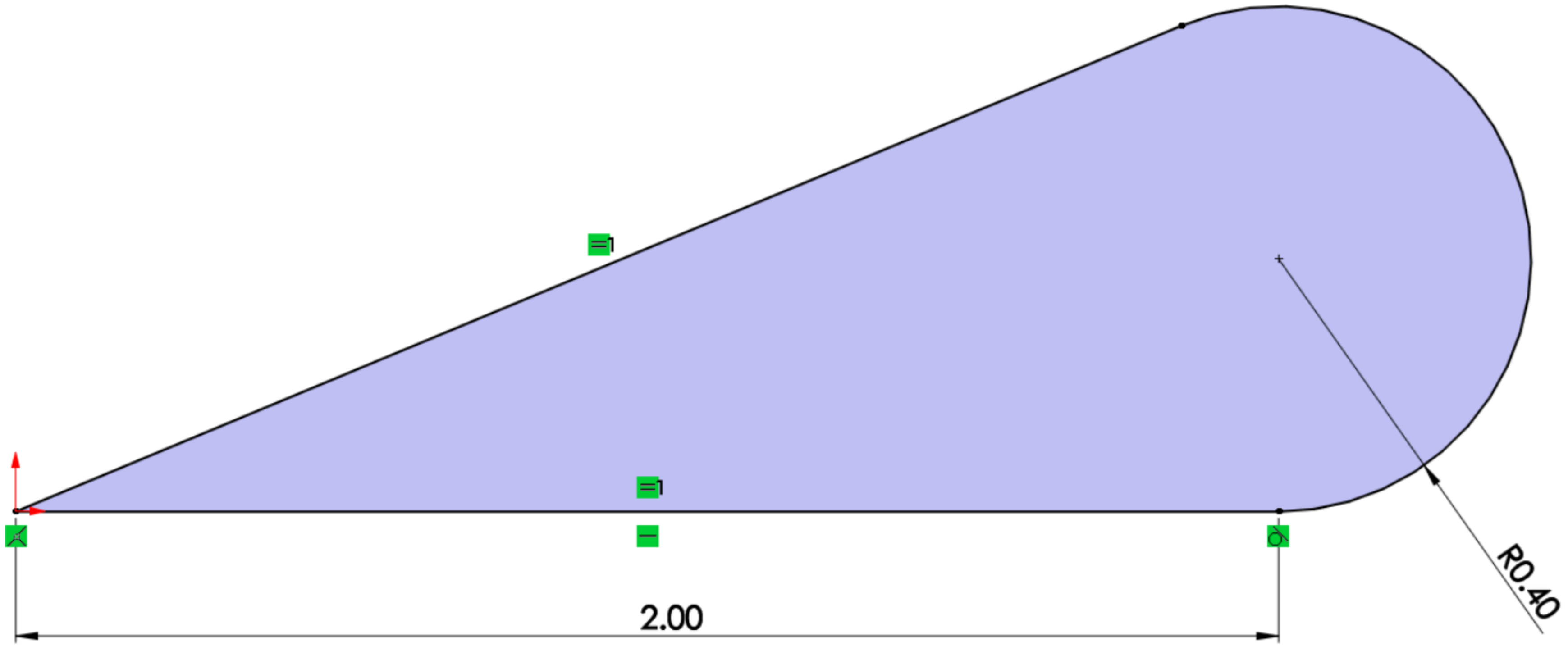


In the preceding sketch, the shaded circle is the base circle, while the others are additions to be made with the pattern command. The annotations in red are the parameters that we need in order to communicate a pattern to SOLIDWORKS sketching. The following is a small description of the different annotations in the preceding sketch:

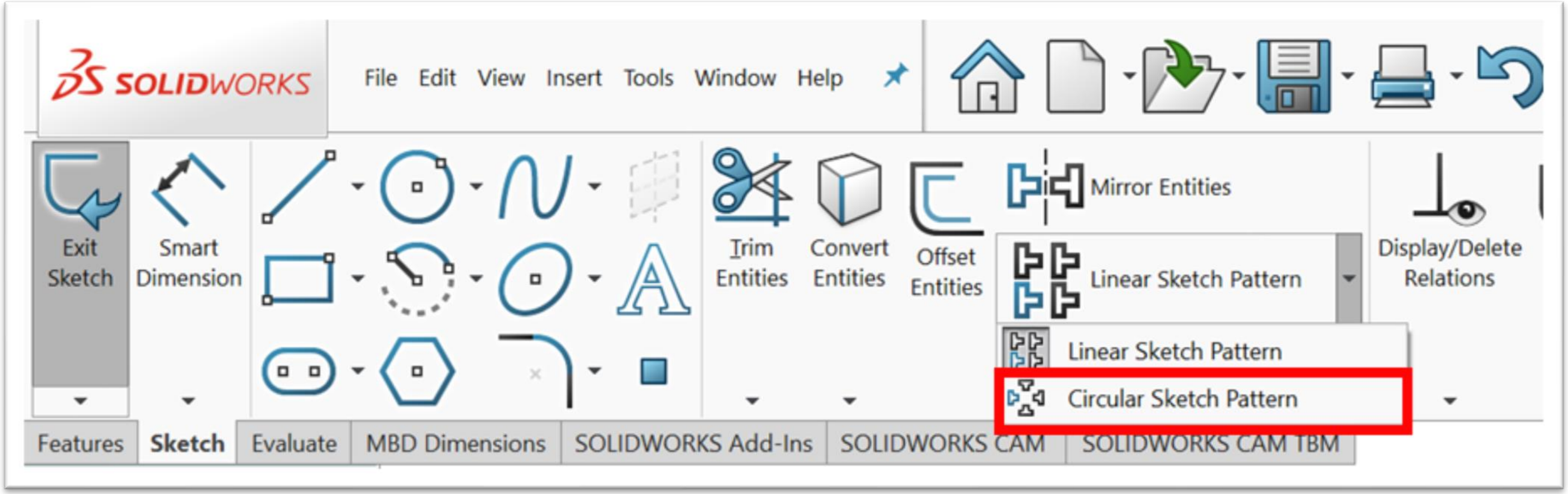
- **Center**: This represents the center of rotation for the circular pattern. This can be determined with specific x and y coordinates or by relating it to another point.
- **Radius**: This is the distance between the original entity and the center of the pattern.
- **Angle: A1**: This is the angle between two adjacent patterned entities.
- **Total Angle**: This is the angle between the original and the last patterned entity.
- **Number of patterned entities**: This shows the total number of patterned entities, including the base sketch.

To sketch the preceding diagram, follow these steps:

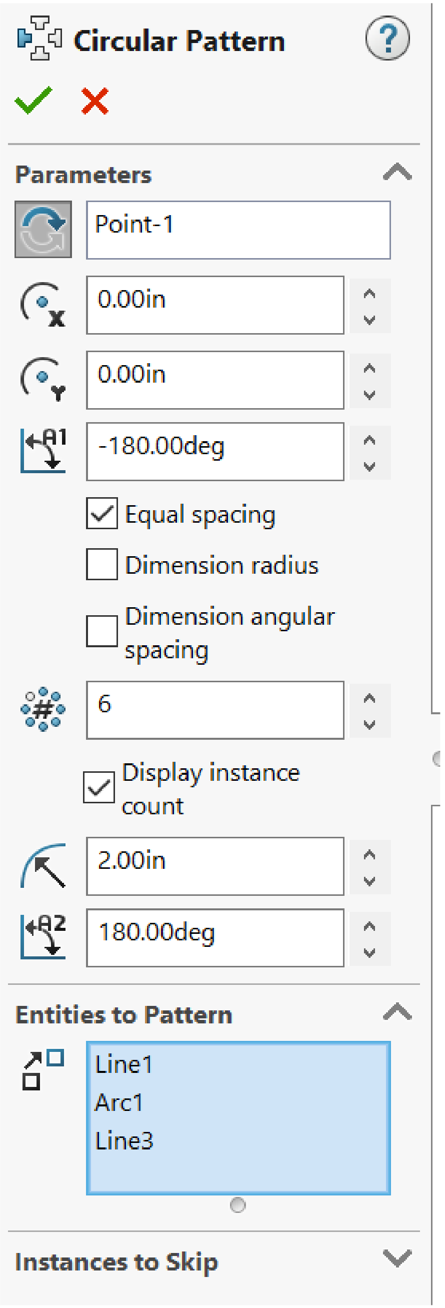
1. Sketch and fully define the base entity, as shown in the following sketch:



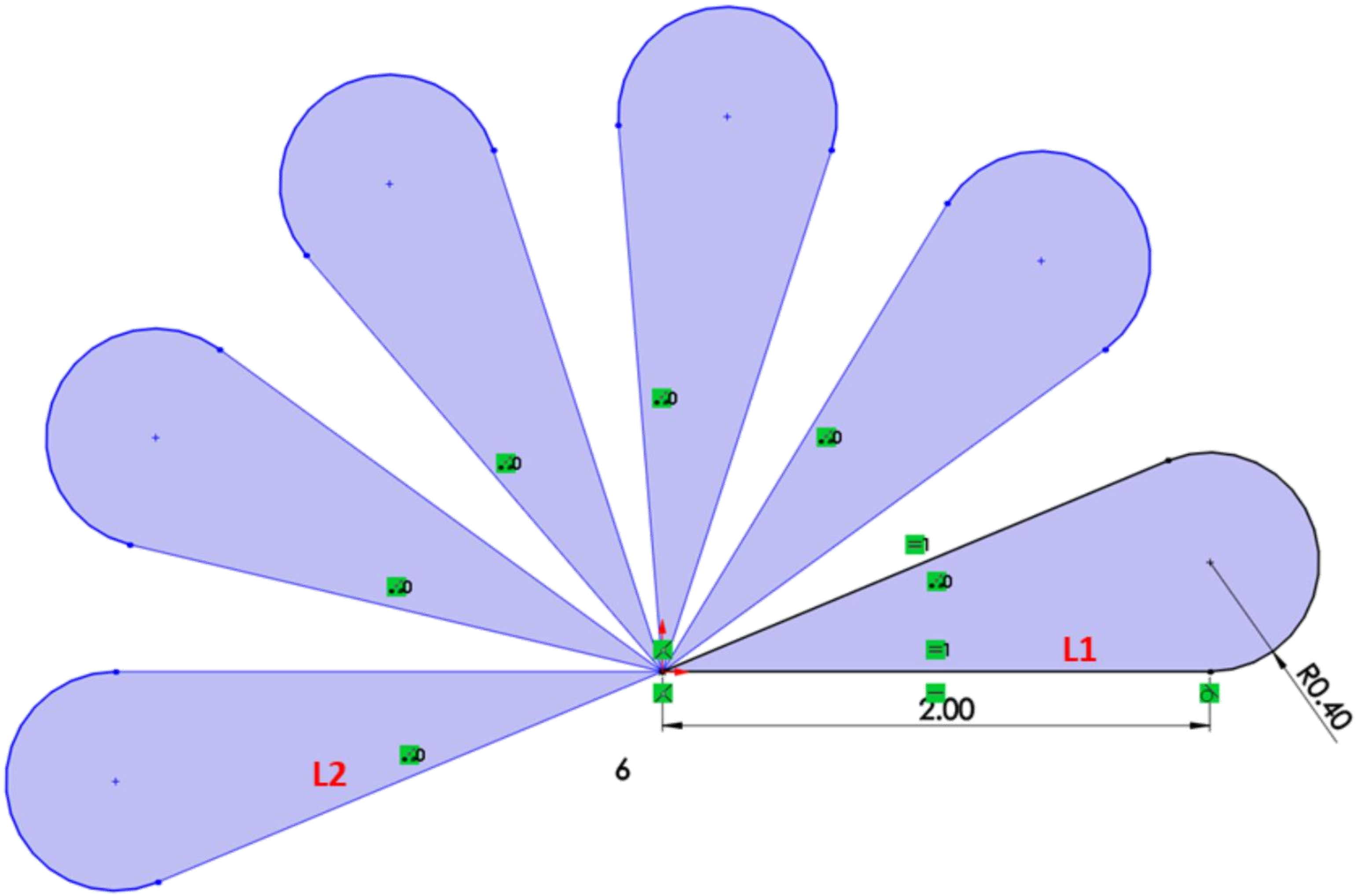
2. Select the **Circular Sketch Pattern** command from the **Sketch** command bar:



3. Select the three lines that form our base sketch. Now, we can set up our **Circular Pattern** using the options that appear on the left-hand side of **SOLIDWORKS**. Set the options that are shown in the following screenshot. As we are adjusting those options, a preview of the final shape will appear on the canvas.



After adjusting these options, we can click on the green checkmark. This will give us the following shape.

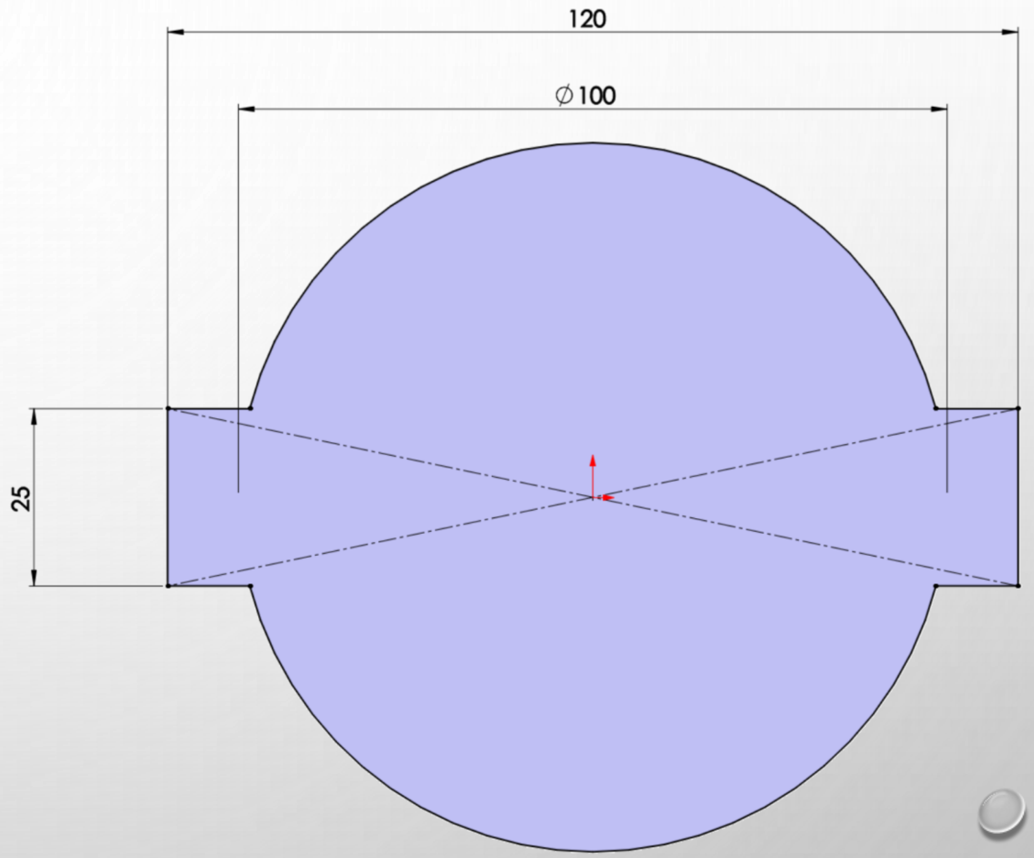


Trimming
in SOLIDWORKS sketching

Trimming in SOLIDWORKS allows us to easily remove unwanted sketch entities or unwanted parts of sketch entities. This makes it easier for us to create complex sketches. In this section, we will cover what trimming is, why we use trimming, and how to use trimming within SOLIDWORKS.

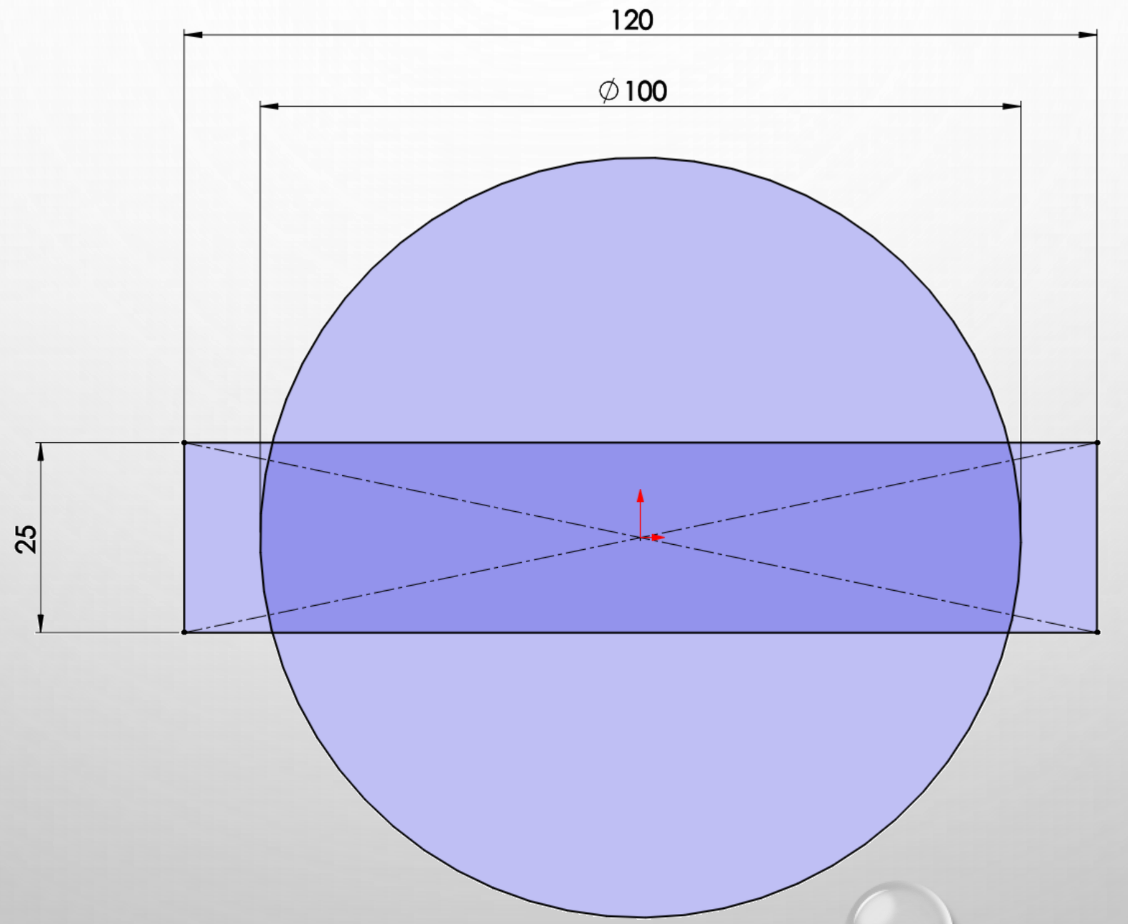
Using power trimming

To show you how we can use the trimming tool in SOLIDWORKS, we will create the following sketch, which consists of a circle and a rectangle. We will use the MMGS measurement system for this exercise:

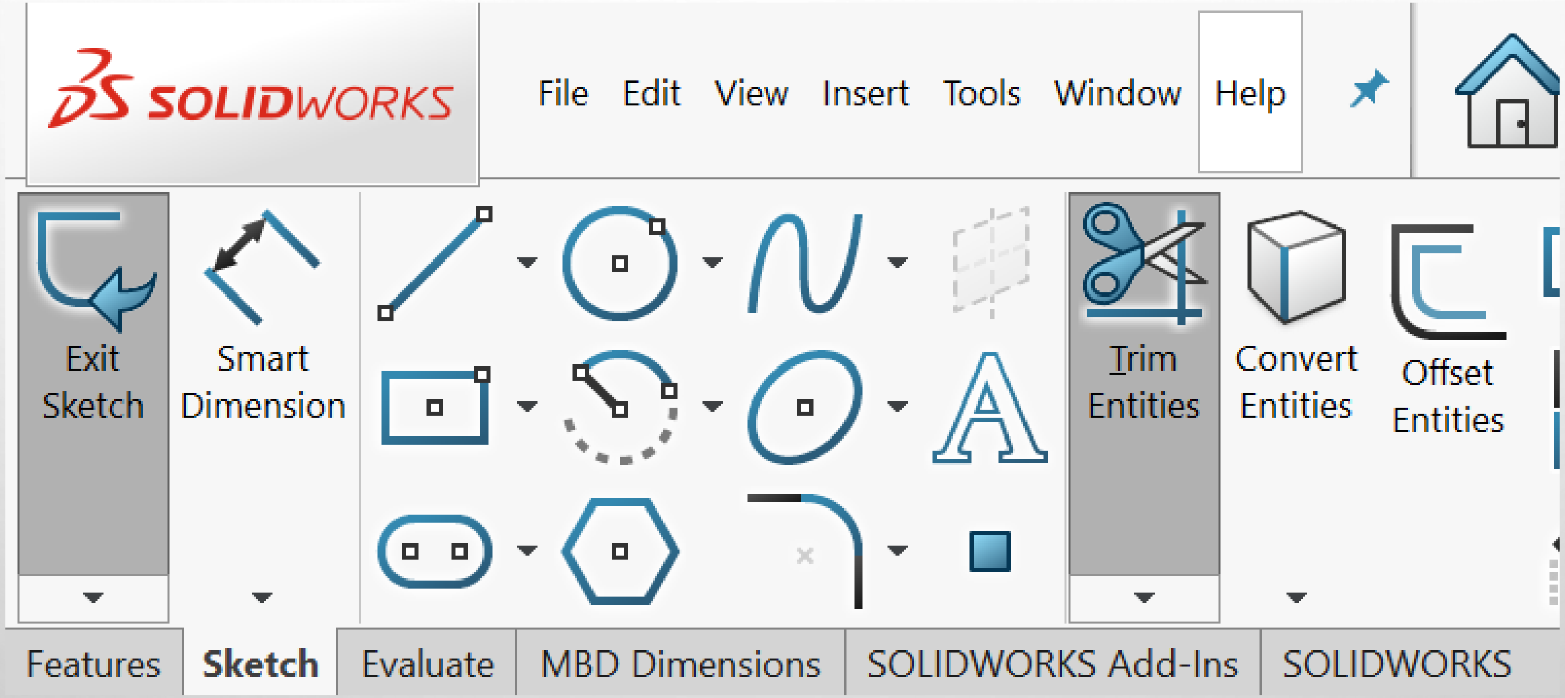


To create the given sketch, follow these steps:

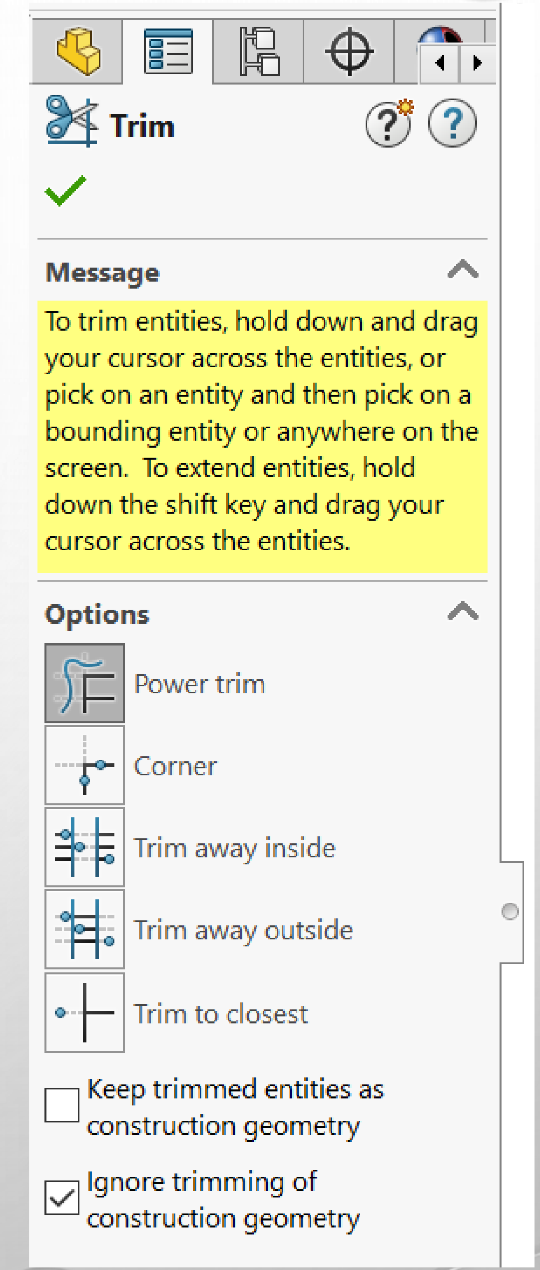
1. Sketch and fully define the base shapes of a circle and a rectangle, as follows:



2. Select the **Trim Entitles** command from the command bar, as shown in the following screenshot:



3. After selecting this command, the Property Manager for it will appear on the left of the interface. This will show different types of trimming tools. In this exercise, we will pick the first one, that is, **Power trim**, as shown in the following screenshot. Here is a brief description of the different trimming tools that are available:

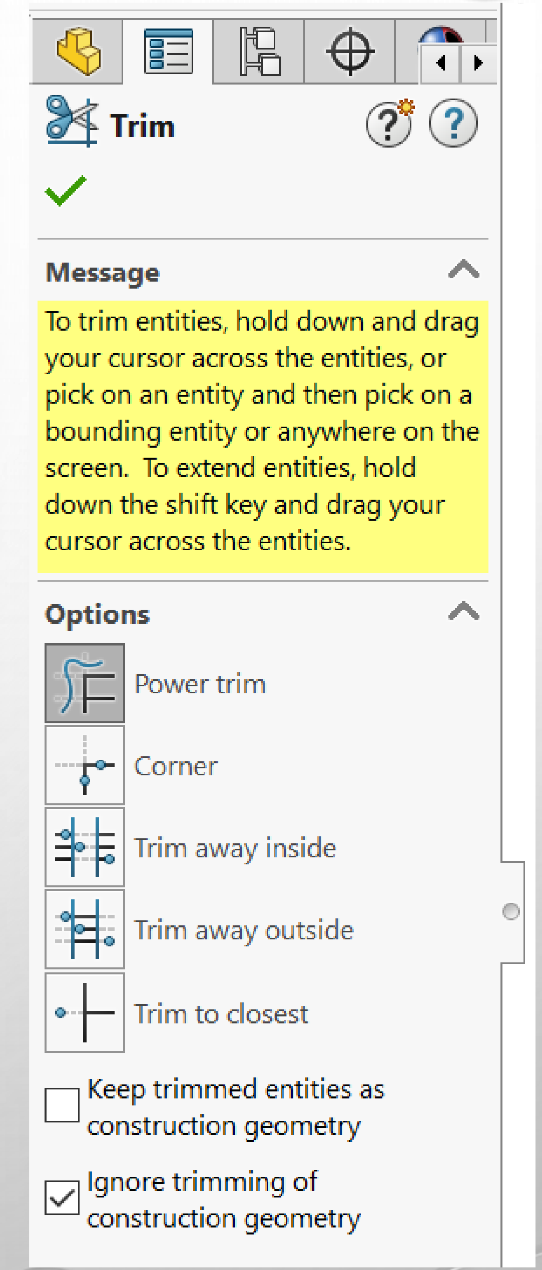


Power trim: This is the easiest method for trimming entities. **Power trim** is a multipurpose trimming tool that allows us to cut entities by going over them in the canvas. In this section of this book, we will only use the **Power trim**.

Corner, Trim away inside, Trim away outside, Trim to closest: These are the different ways we can trim.

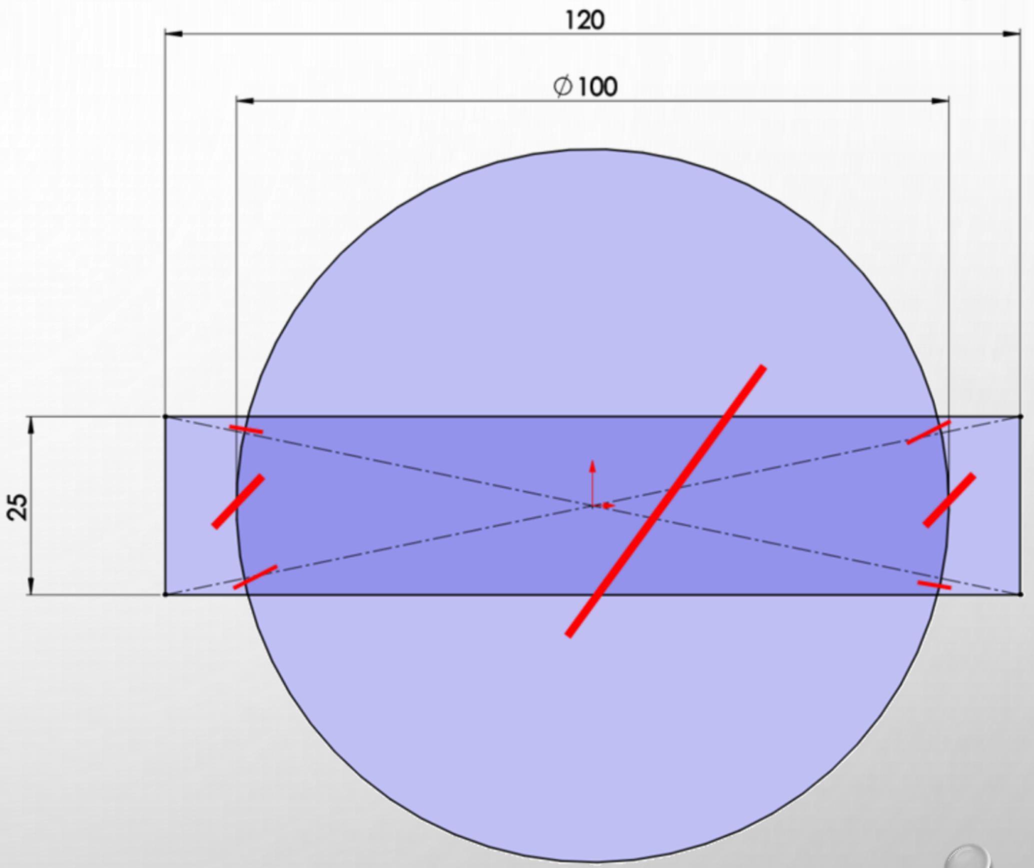
Keep trimmed entities as construction geometry: When checked, trimming will not remove entities from the canvas. Instead, they will be switched to construction lines.

Ignore trimming of construction geometry: When checked, trimming will not function with construction lines. In this example, we didn't want to revoke the construction lines that define our rectangle, so we checked this option:



4. Go back to the canvas and start trimming. To trim unwanted parts, we can click, hold, and move the mouse, as illustrated by the red lines in the following diagram. As we cross each of these lines, we will notice that the line will disappear. By doing this, we will only remove the lines that are not desirable.

If we trim the wrong line, we can simply undo this with *Ctrl* + *Z*. After doing this, we will end up with the final shape, which was shown at the beginning of this section:



Solid Modeling Tools

In SOLIDWORKS, features are what can turn a 2D sketch into a 3D model. we will move on from 2D sketches and start creating 3D models. We will explore the most basic features, such as extruded boss and extruded cut, and revolved boss and revolved cut.

Understanding and applying extruded boss and cut

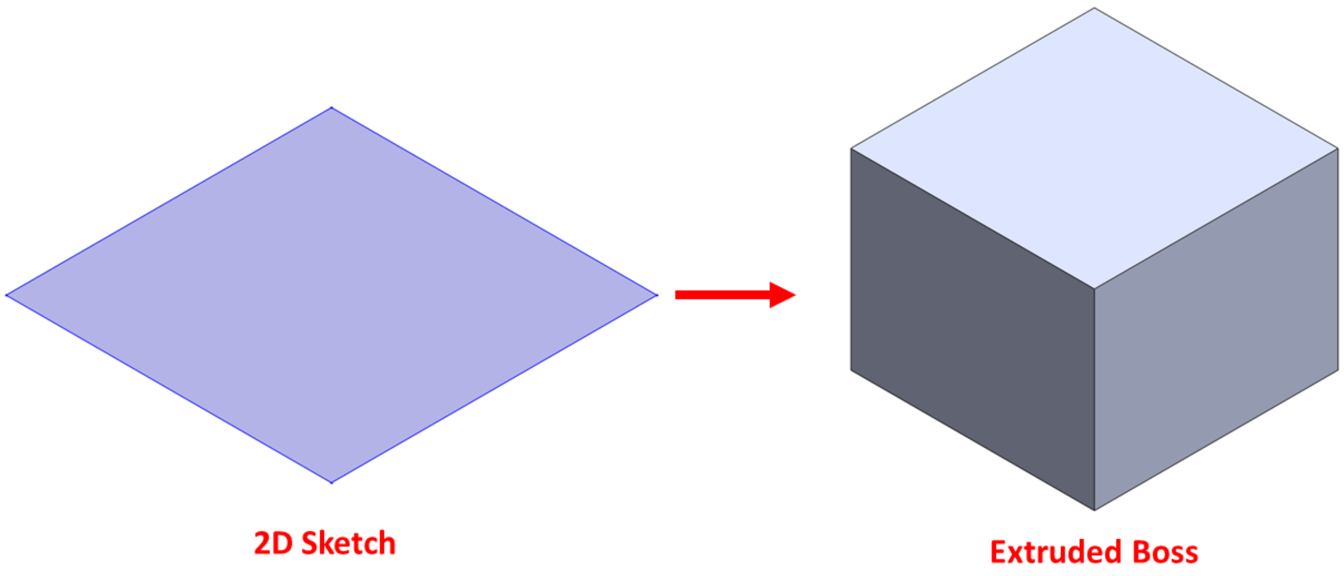
Extruded boss and extruded cut are the most basic and easiest features to apply. They are direct extensions of a sketch and push it into the third dimension. In this section, we will cover what extruded boss and extruded cut are, how to apply them, how to edit them, and how to delete them.

Extruded boss & Extruded cut

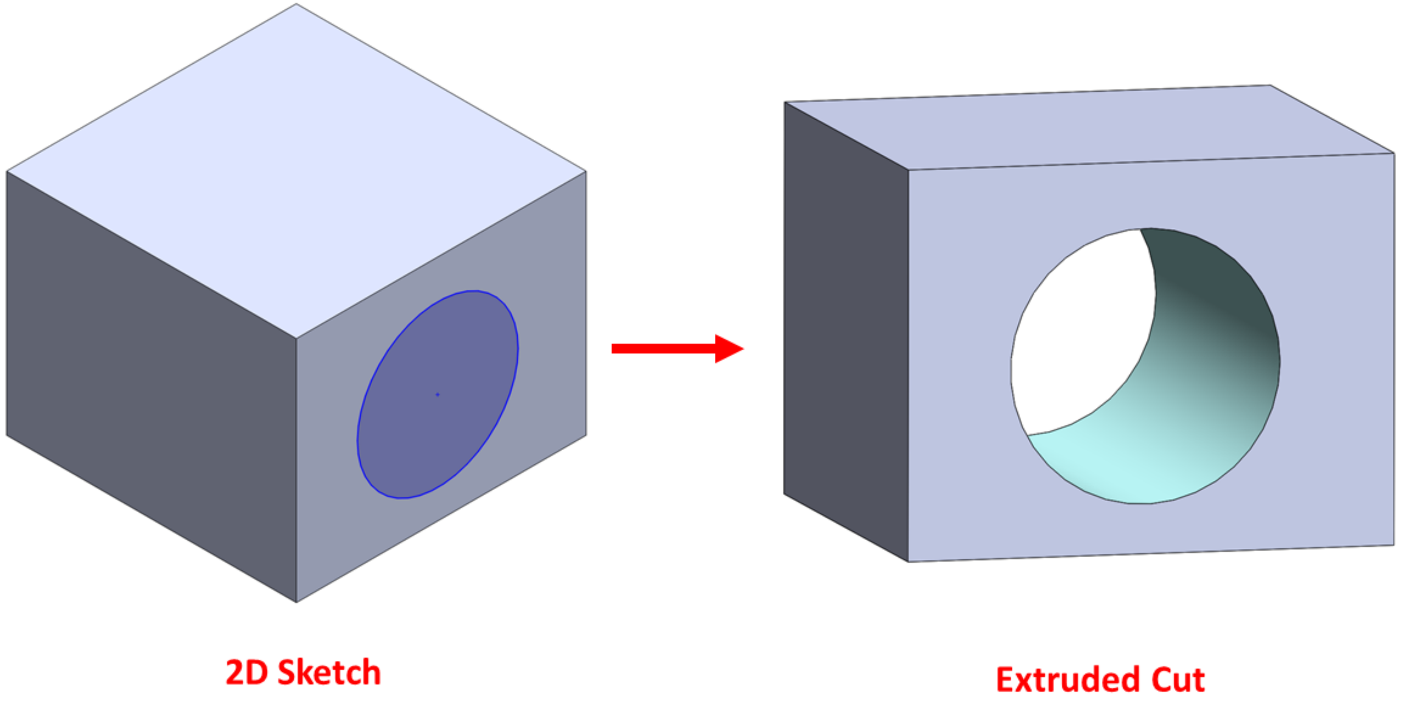
- **Extruded boss**: This is direct extension of a sketch that pushes it into the third dimension, resulting in adding materials.

- **Extruded cut**: This is direct extension of a sketch that pushes it into the third dimension, resulting in removing/subtracting materials.

From these definitions, you can see that extruded boss and extruded cut are quite similar, but they have opposite effects. Extruded boss adds materials, while extruded cut removes material. The following image illustrates the effect of the extruded boss. Note that we were able to go from a 2D sketch to adding materials to form a cube:

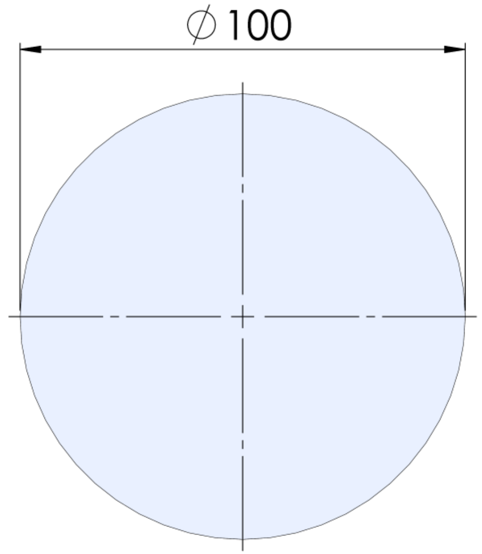


The following image illustrates the effect of extruded cut. Note that we were able to use a sketch to remove materials:



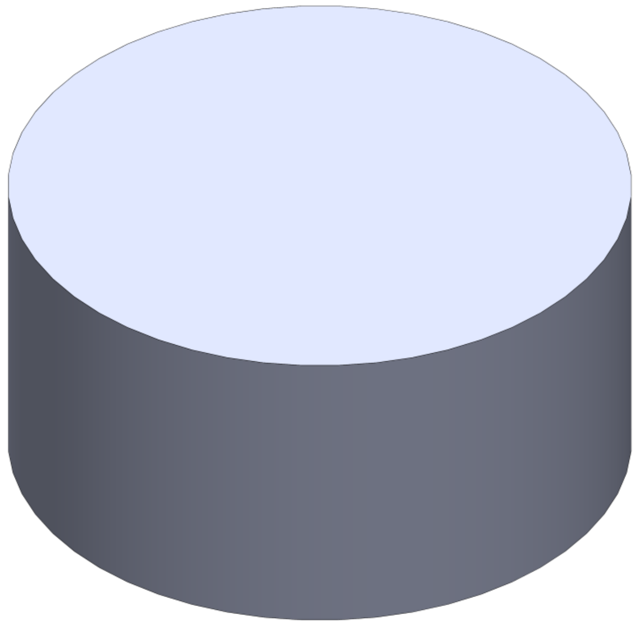
Applying extruded boss

When applying the extruded boss and extruded cut features, we will always start with a 2D sketch, and then apply that feature based on that 2D sketch. Thus, for this exercise, we will split each feature application into two stages: sketching features and applying features.



View (1): Top View

View (2): Side View



View (3): Isometric View

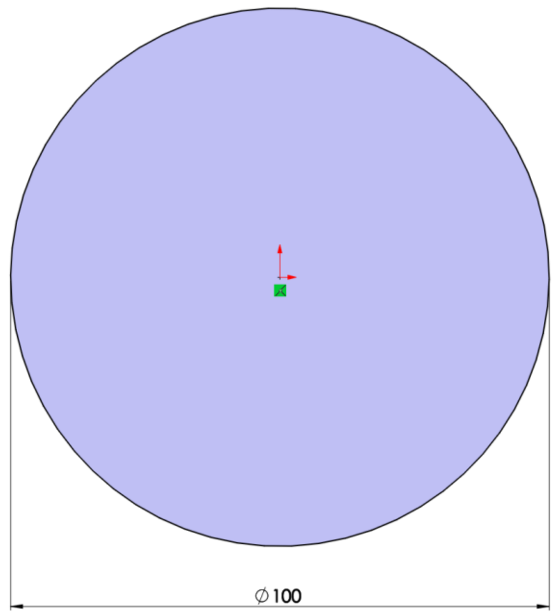
Measurement System: MMGS

One important aspect to keep in mind is that, as we continue modeling, we will need to plan a strategy when it comes to how to model the targeted object. There is no right or wrong way to create a model. Thus, different people will have different plans for making the same model. It is always good to plan ahead when it comes to creating a model. We can do that by either sketching or writing down our ideas. Since we are taking our first steps toward 3D modeling.

we will need to have a brief written plan before we start modeling:

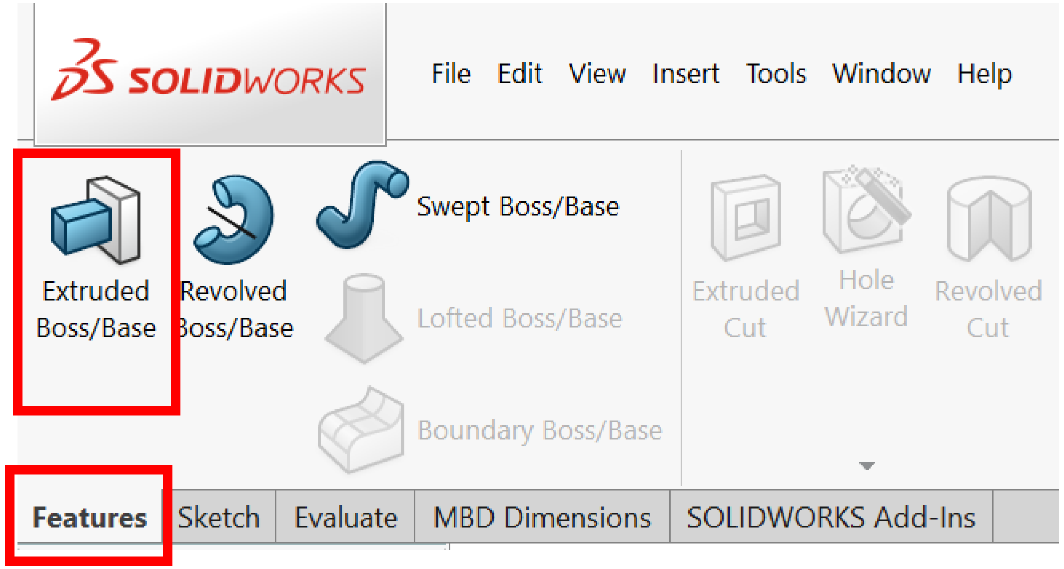
- Planning: We start by creating a circle and then extruding that into a cylinder.
- Sketching: Next, we sketch and fully define a circle with a diameter of 100 mm.

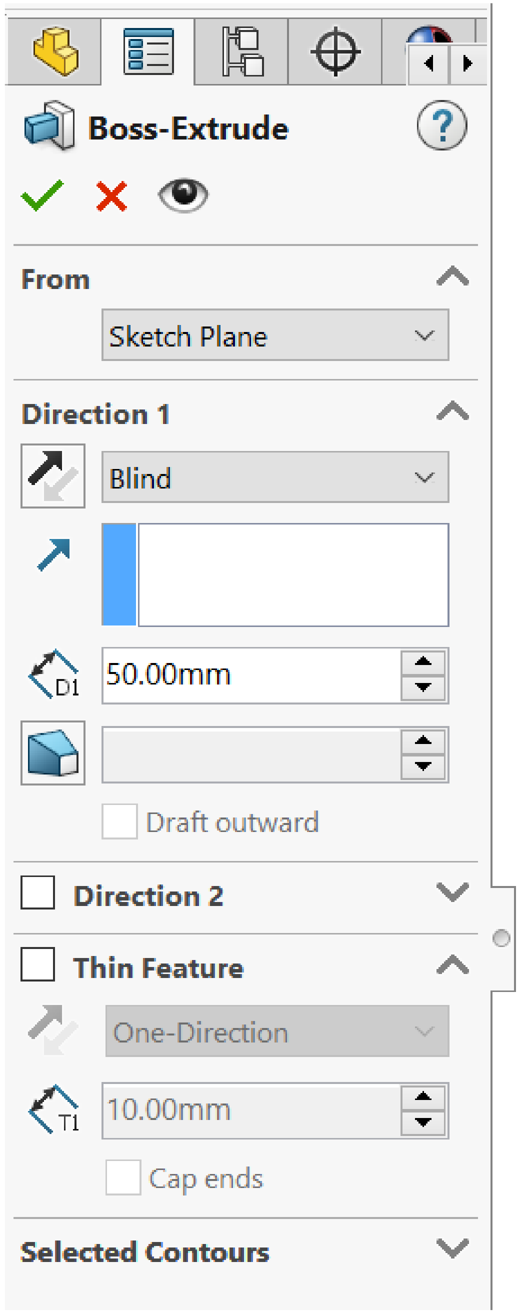
We can see this in the preceding diagram, where it says View (1): Top View. The circle will look as follows:



Applying the feature: In this example, we will apply the extruded boss feature. To apply it, perform the following steps:

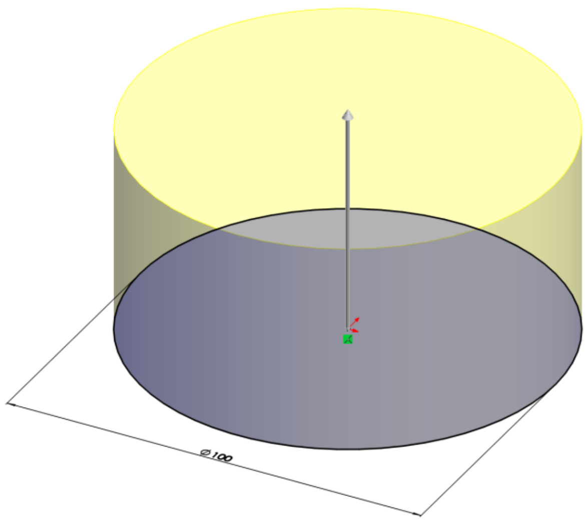
1. Click on the **Features** tab and select the **Extruded Boss/Base** command, as shown in the following screenshot. We don't need to exit sketch mode. As soon as we select this command, SOLIDWORKS will understand that we want to apply this feature to the active sketch:



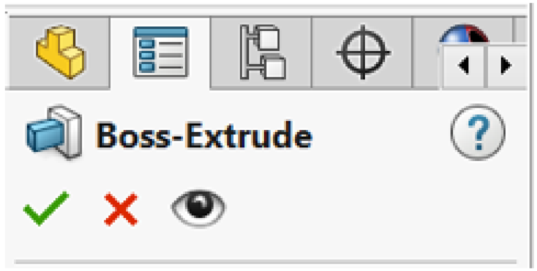


2. Once we click on the **Extruded Boss/Base** command, we will notice that an options panel appears on the left-hand side. The extrusion preview will also appear on the canvas.
3. Fill out the options in the **Property Manager**, as shown in the following screenshot. Fill in the height as 50 mm. The **Property Manager** will appear on the left-hand side of the interface:

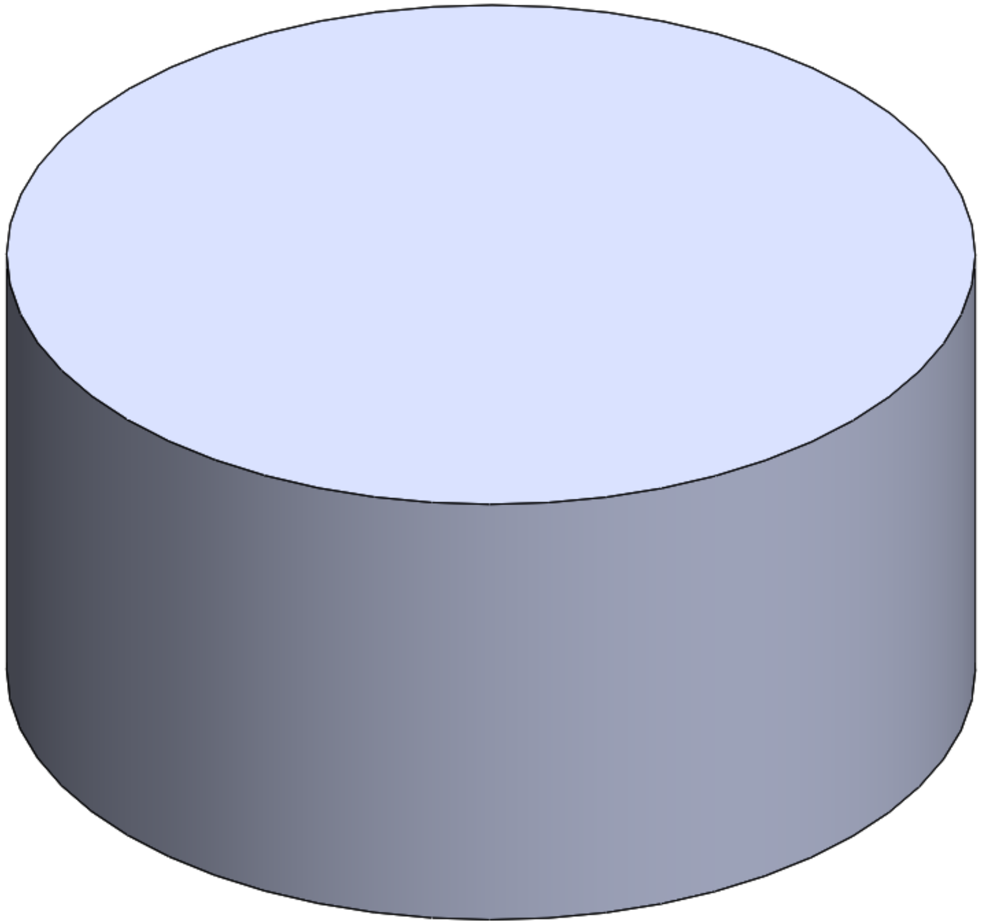
4. Once we've filled in these options, we will see the following preview:

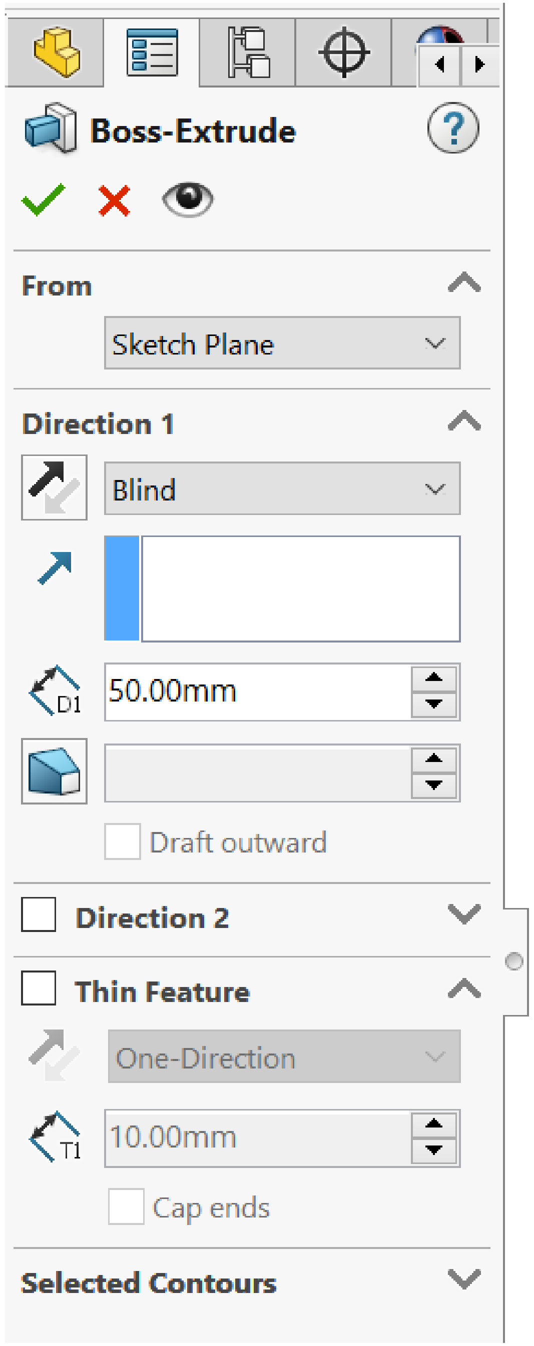


5. After adjusting the options for our extrusion, we can click on the green checkmark at the top of the **Property Manager** panel to apply the extrusion:

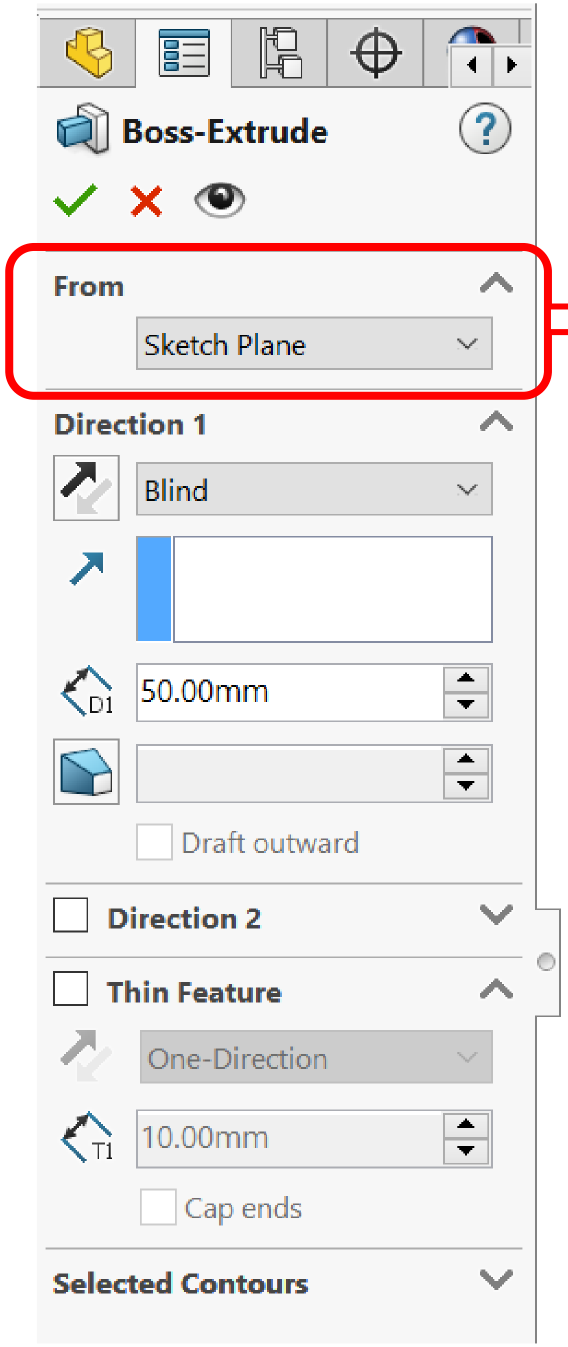


The result will be the following model:

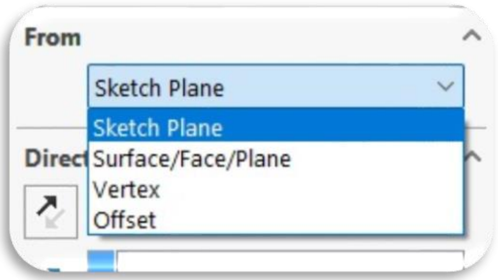




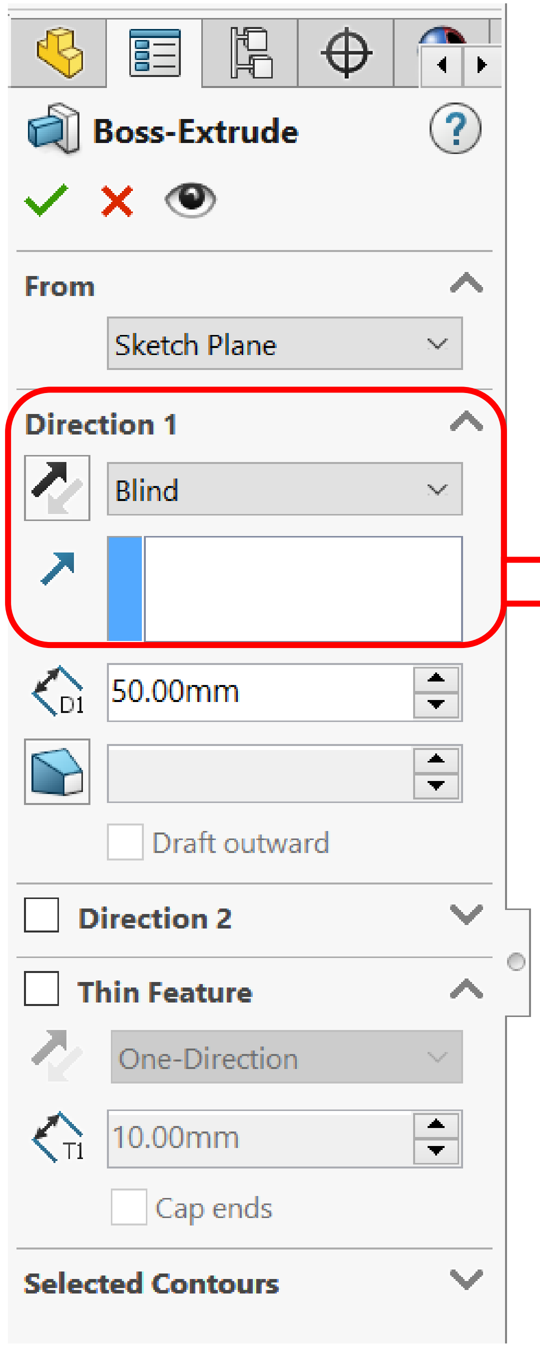
Before we finish looking at the extruded boss feature, let's take a look at options in the **Property Manager**. We will look at them based on their listing order, that is, from top to bottom, as shown in the following screenshot. We will start with the options we used in this exercise and then move on and look at the options we didn't use:



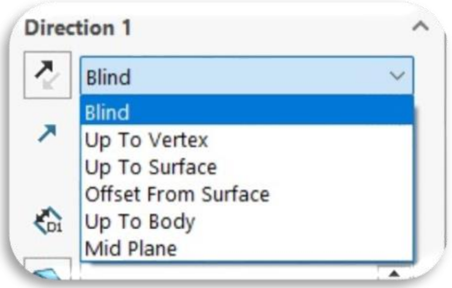
The following options are available for the extruded boss feature:



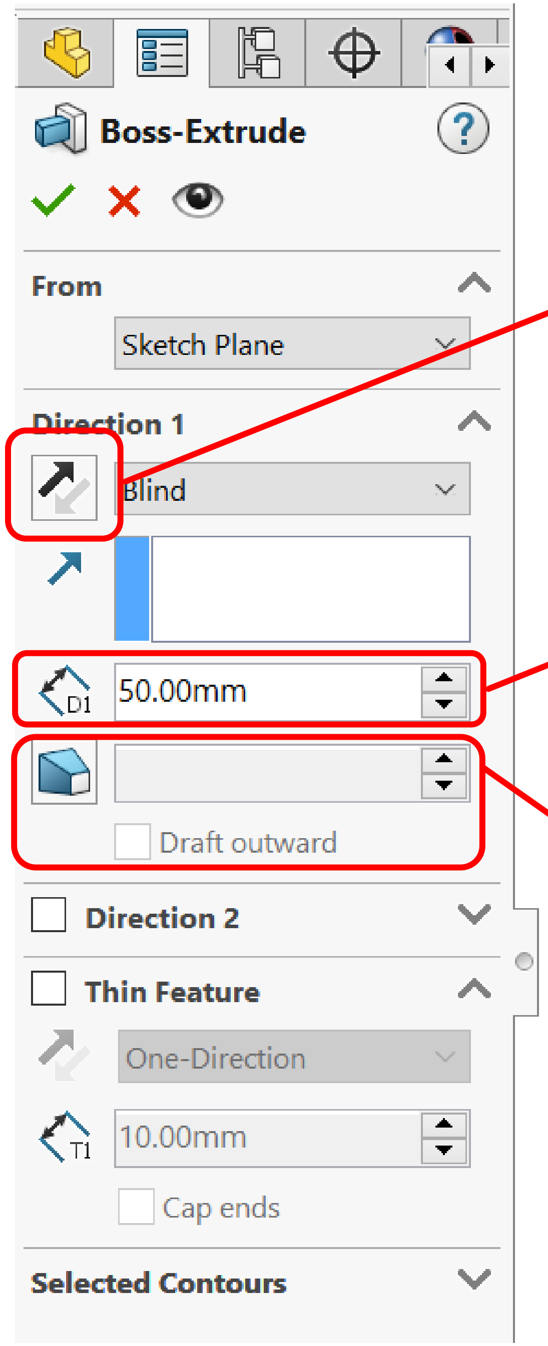
- **From**: This determines where the extrusion features should start. We will mostly use the **Sketch Plane**. This means that the extrusion will start from the sketch that was used to create it. Other options include starting from the **Surface**, **Vertex**, and **Offset**. The first two can't be used in this case since our model doesn't have multiple surfaces and vertices. The last option—**Offset**—can be used to offset the whole extrusion by a certain distance.



- **Direction 1:** This is active by default. Under this heading, we can customize the previewed extrusion that's shown in the canvas. We can hover the mouse over the options to see their full names. The options under **Direction 1** are as follows:



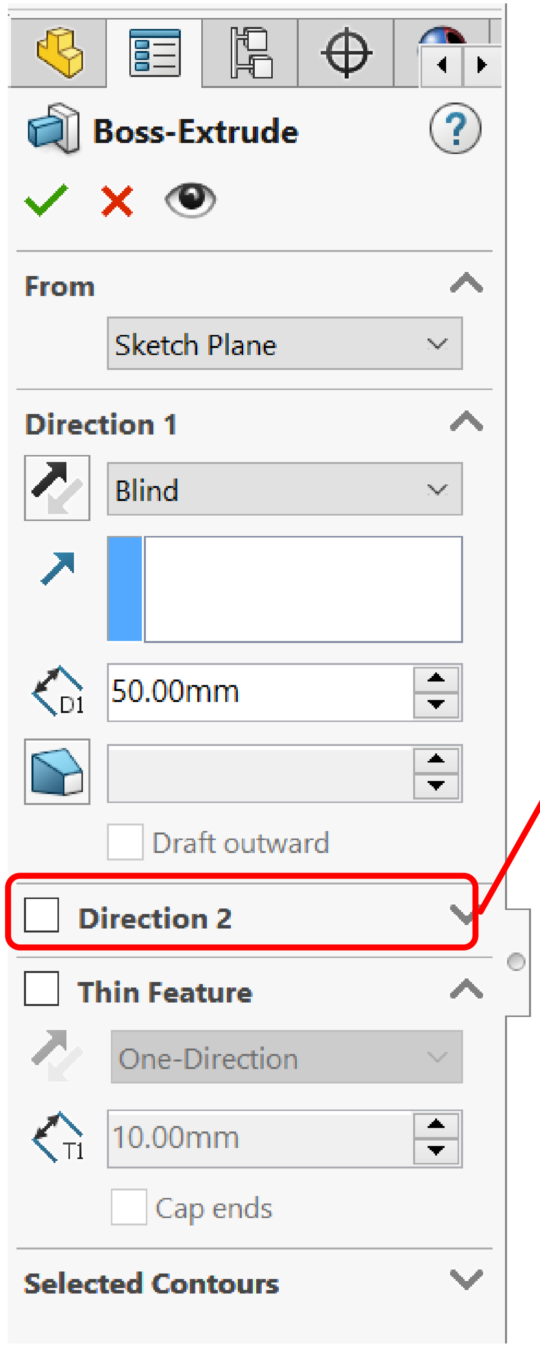
  - **End Condition**: This determines how the extrusion stops. At this stage, we will only be using the **Blind** option, which is selected by default. This means that the extrusion will be extended by the dimensions that we indicate. We will talk about end conditions at a more advanced level later in this book.



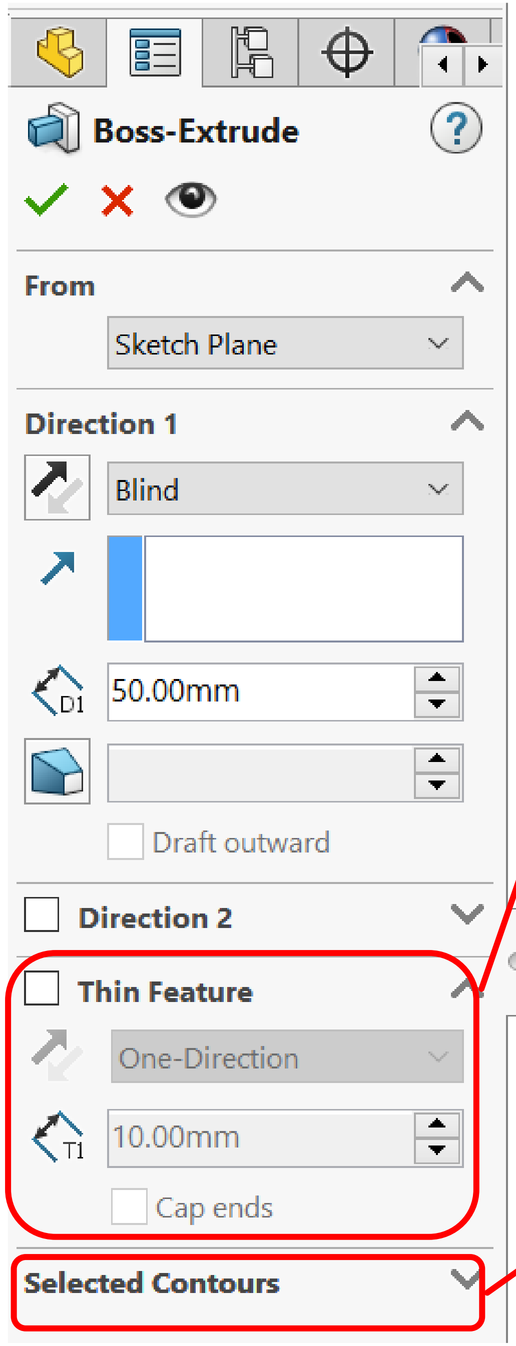
- **Reverse Direction**: This is the arrow to the left of **End Condition**. This can easily reverse the direction of the extrusion from up to down and vice versa.

- **Depth** (**D1**): This determines the depth of the extrusion. In our case, we want the extrusion to be 50 mm deep, so we will input 50 mm.

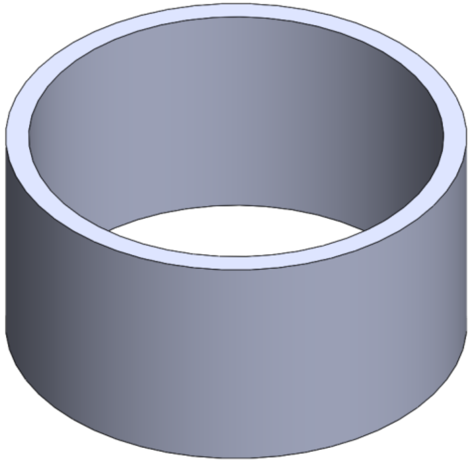
- **Draft**: The icon below **Depth** is used to draft the extrusion. We can activate drafting by clicking on the icon..



- **Direction 2:** This is very similar to **Direction 1**; however, it applies the extrusion in the opposite direction as well. We can use this if we ever want to have different length extrusions in two directions on the sketch. We can simply check this box if we require the second direction. This wasn't needed in our example.



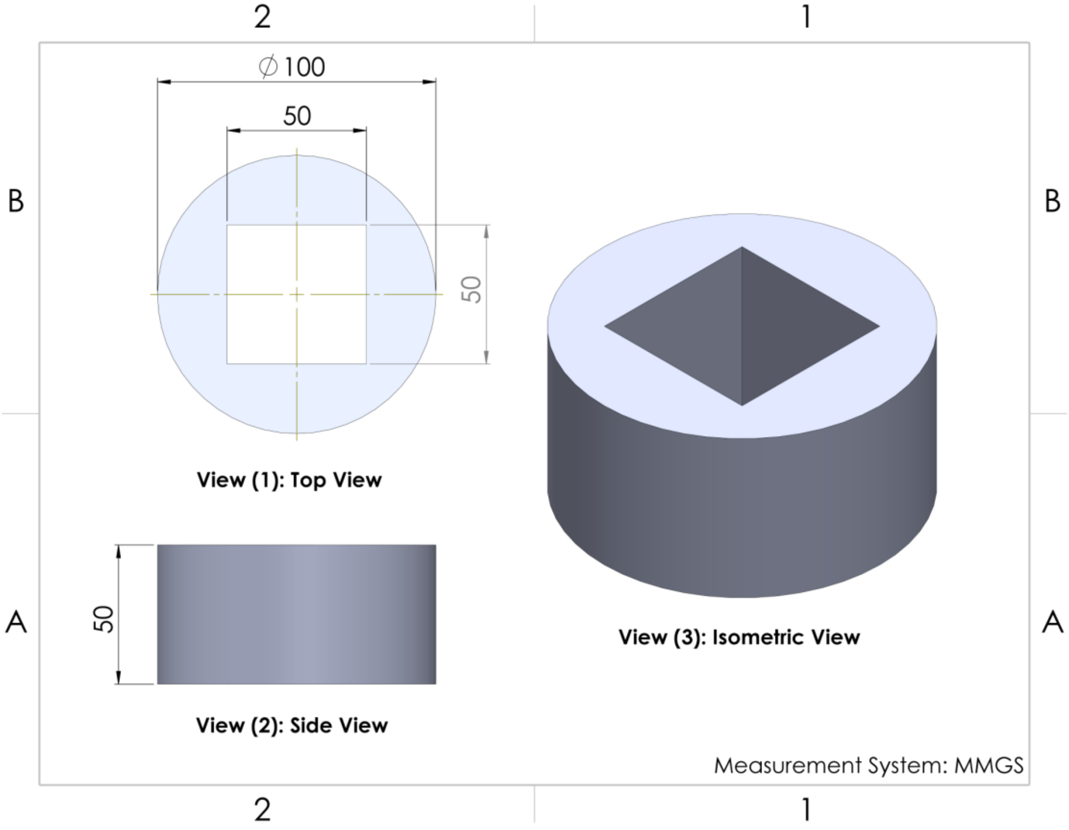
- **Thin Feature:** This applies an extrusion based on the thin borders of the sketch rather than the enclosed shape. It can be activated by checking the box next to it. If we apply a **Thin Feature** to our circle, we will get a result similar to the one shown in the following image.



- **Selected Contours:** This can be used if we have more than one enclosed area. Then, we can select which ones we want to apply the extrusion to.

Applying extruded cut

The extruded cut feature is very similar to the extruded boss feature in terms of the options that are available to us.



Note that, in the preceding model, we are only applying an extra cut over the cylinder that we created with the extruded boss feature. Thus, we will start from the cylinder we created earlier and create an extra extruded cut. We will go through the following phases to do so:

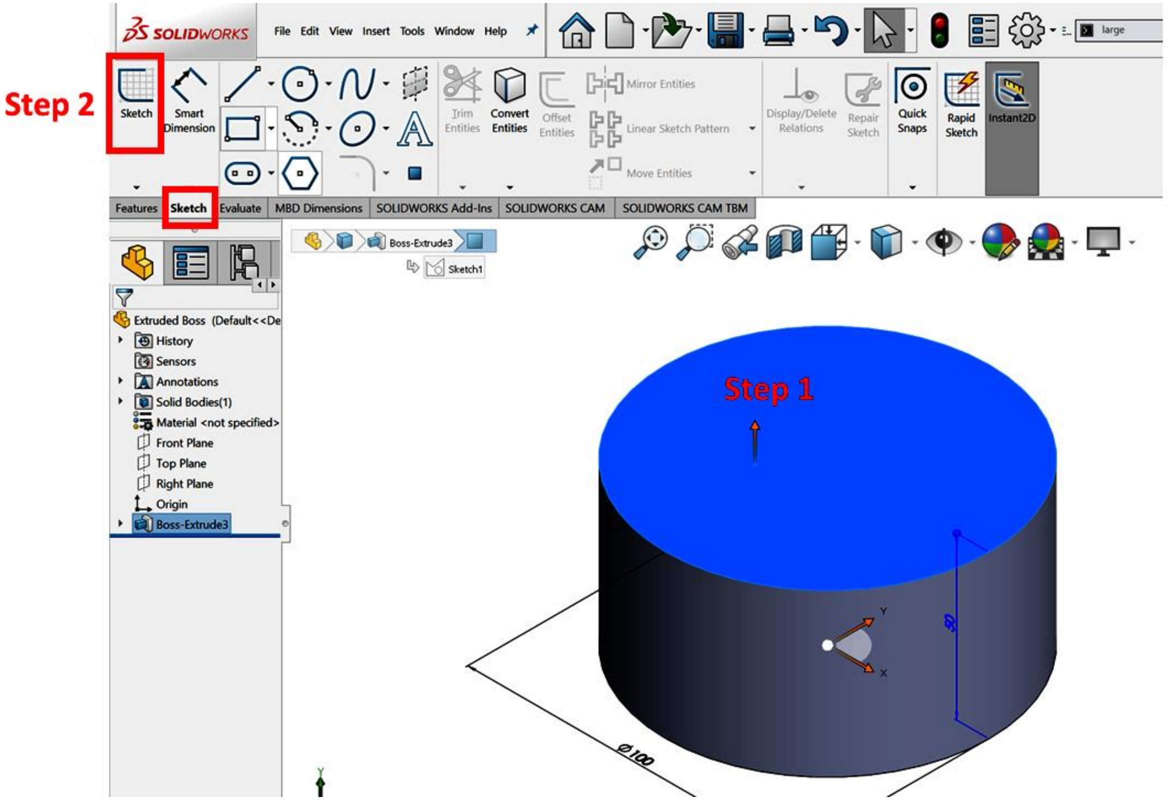
Planning: We will draw a square on top of the cylinder and apply it using the extruded cut feature.

Sketching: The cutoff shape is a square, and so we will sketch a square on the top surface of the cylinder. Note that the top surface is not a default sketch plane. However, it is a straight surface, which means we can use it as a sketch plane.

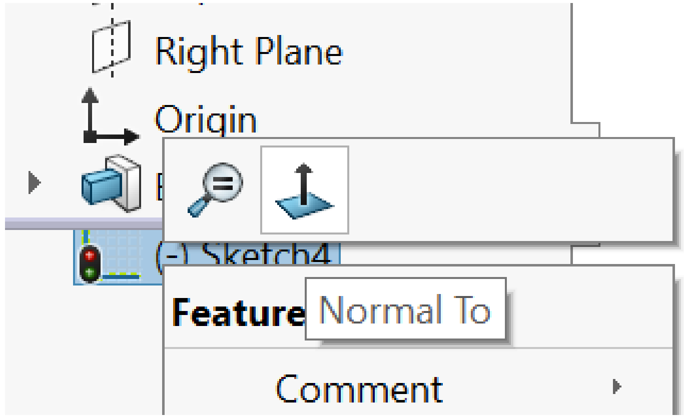
Applying the feature: When we have our sketch, we can apply our feature; in this case, this is extruded cut.

Follow these steps to create the sketch:

1. Select the top surface of the cylinder and click on the **Sketch** command, as shown in the following screenshot. We can also do this the other way around, that is, select the **Sketch** command first and then select the surface we want to sketch on:

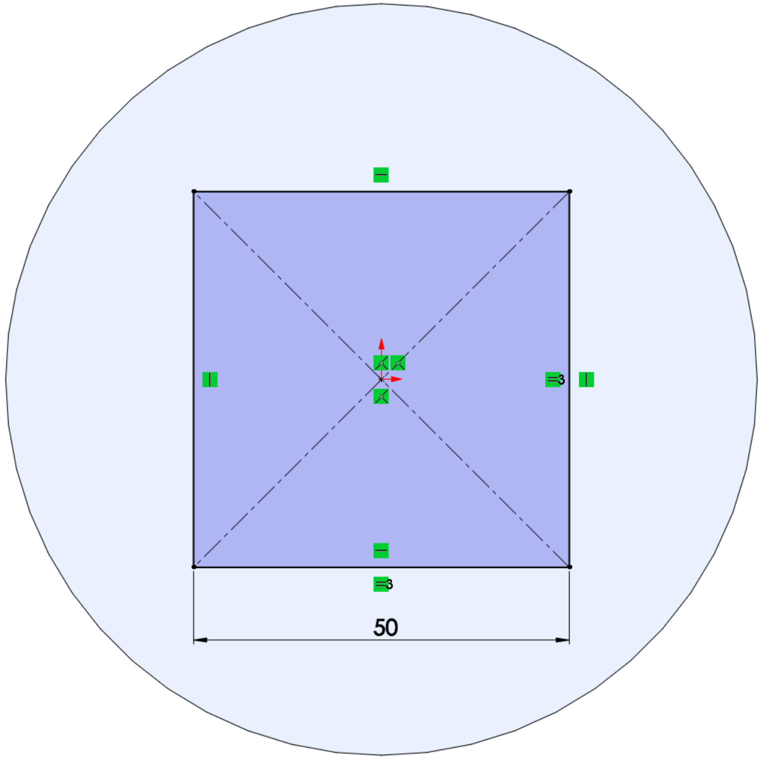


Now we need to start sketching. However, we may have a titled view of our new sketch surface, which will make it harder for us to sketch. We can adjust our view so that it's normal to the sketch surface to make it easier to sketch. To do that, we can right-click on the new sketch at the bottom of the design tree and select Normal To, as shown in the following screenshot:



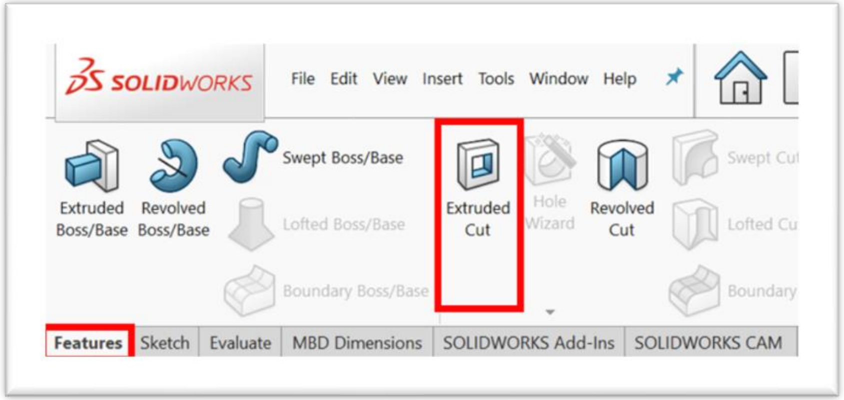
This will change our view of the canvas so that it's facing the sketch surface. If we select Normal To again, the model will flip 180 degrees.

2. Sketch and fully define the required square, as shown in the following image. The side of the square is 50 mm in length, as shown by **View (1): Top View** in the drawing at the beginning of this section:

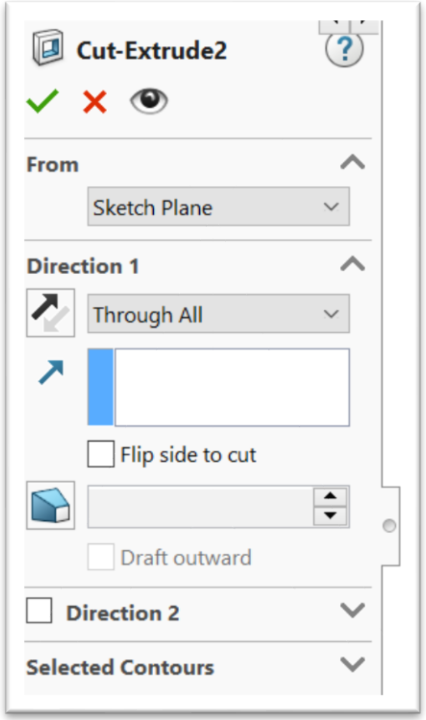


Now that we have our sketch, we can apply our extruded cut feature.

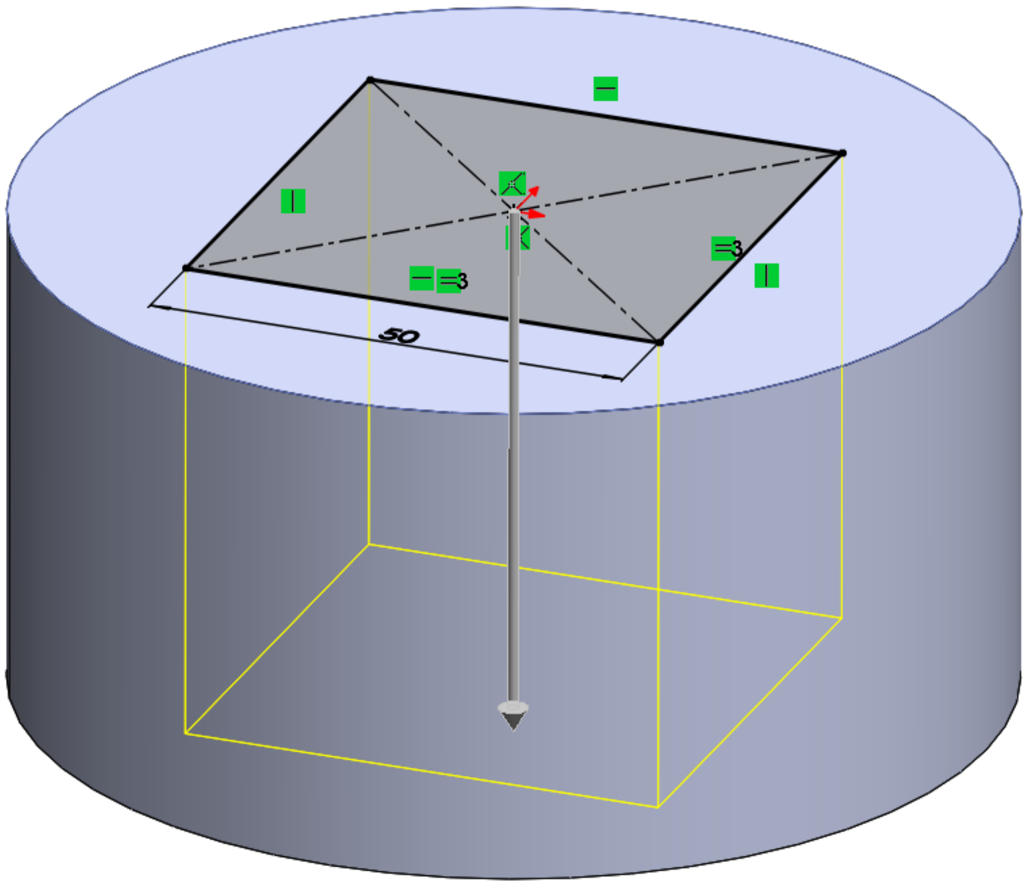
3. Select the **Features** tab and select the **Extruded Cut** command, as shown in the following screenshot. (As with the extruded boss feature, we don't need to exit sketch mode.) As soon as we select the command, SOLIDWORKS will interpret that we want to apply the feature to the active sketch:

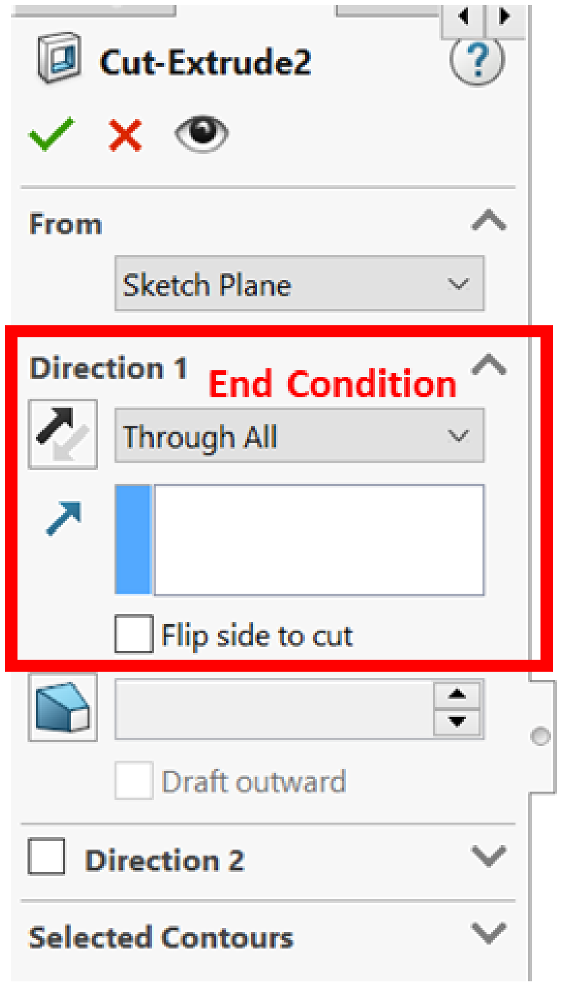


4. Set the options in the Property Manager as follows:

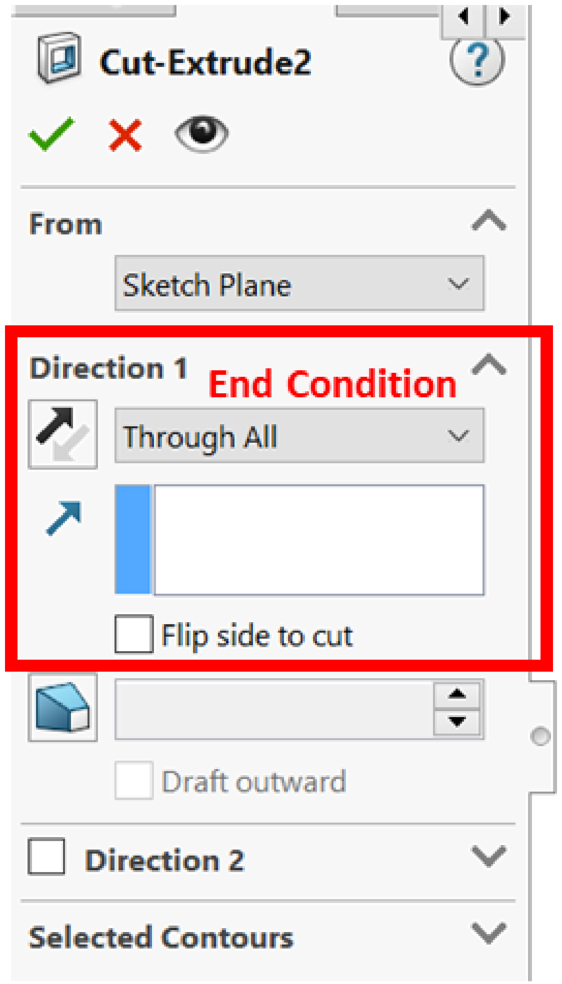


5. As with the extruded boss feature, we will also see a preview of the cut appear on the canvas. This preview is shown in the following image:



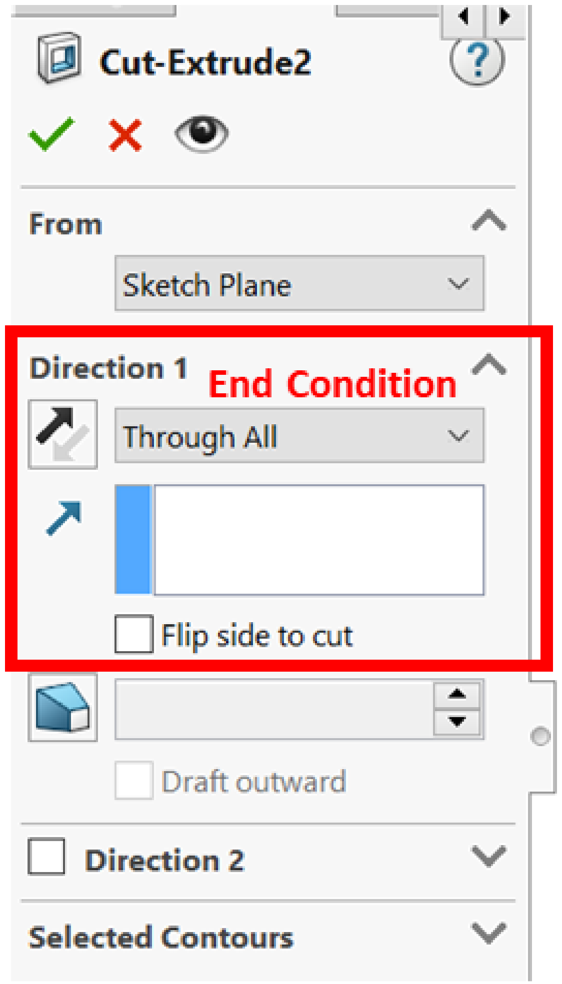


Note that the options that are shown in the Property Manager for the extruded cut feature are almost the same as the options for the extruded boss feature. We will only elaborate on those that are highlighted in the following screenshot:



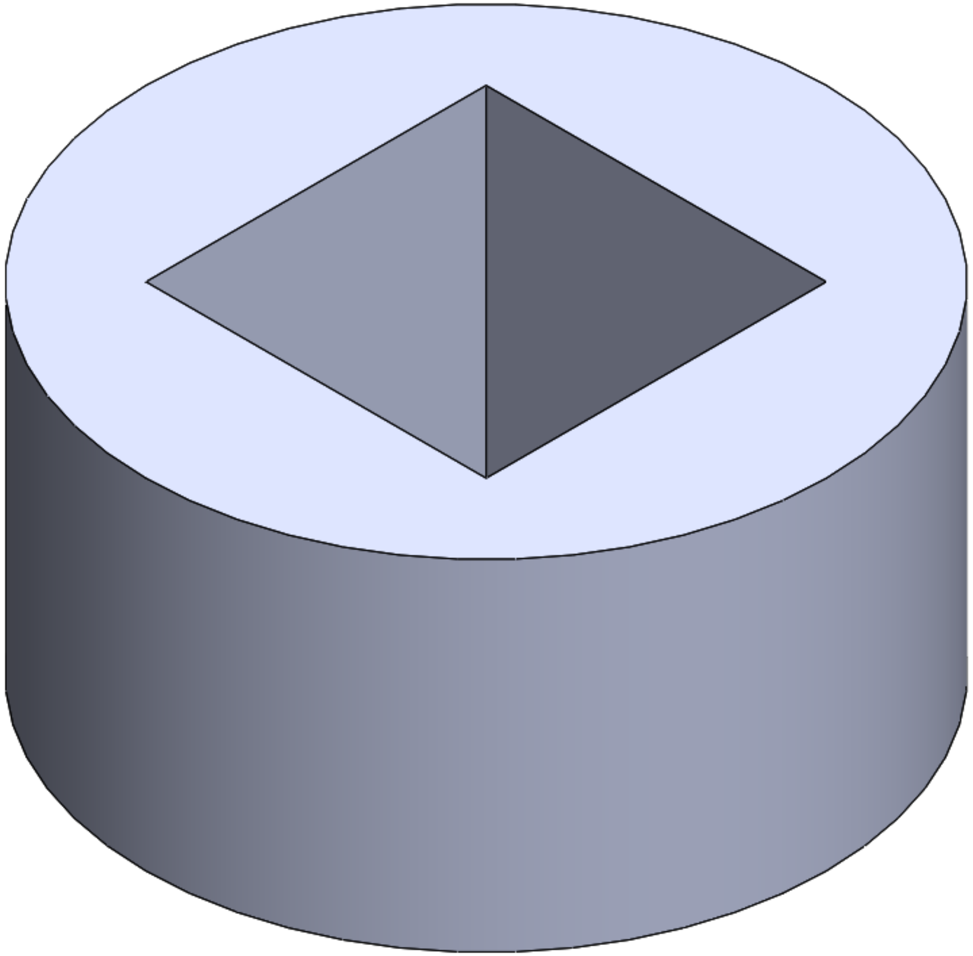
Let's take a look at these options in more detail:

- **End Condition**: Many end conditions are the same as for the extruded boss feature. By default, the end condition is usually **Blind**. However, in our case, we changed it to **Through All**. The **Through All** end condition means that the cut will extend to the end of the model, which is what we want. We can also get the same result by using the **Blind** end condition and setting the depth of the cut to 50 mm or more. To change the end condition, click on the drop-down menu and select the desired condition.



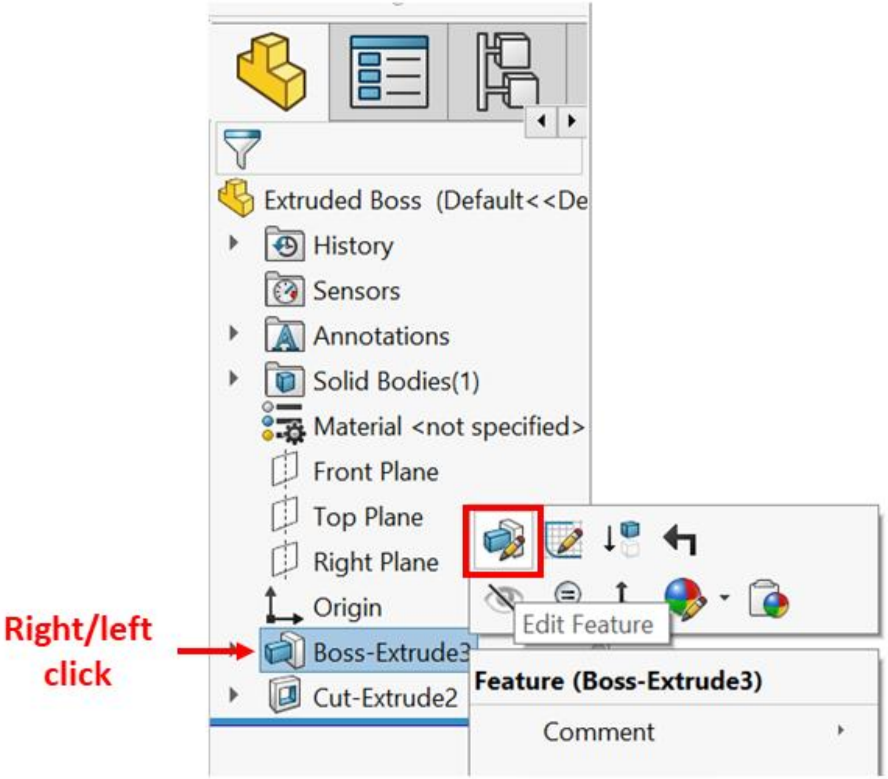
- **Flip side to cut**: Checking this option will turn the cut part around. In our example, if we have this option checked, we will keep the contained square and delete everything else. Having this option unchecked will delete the contained square and keep everything else. Experiment with this option to understand what it does.

6. Click the green checkmark at the top of the Property Manager to apply the extruded cut feature. We will end up with the following model:



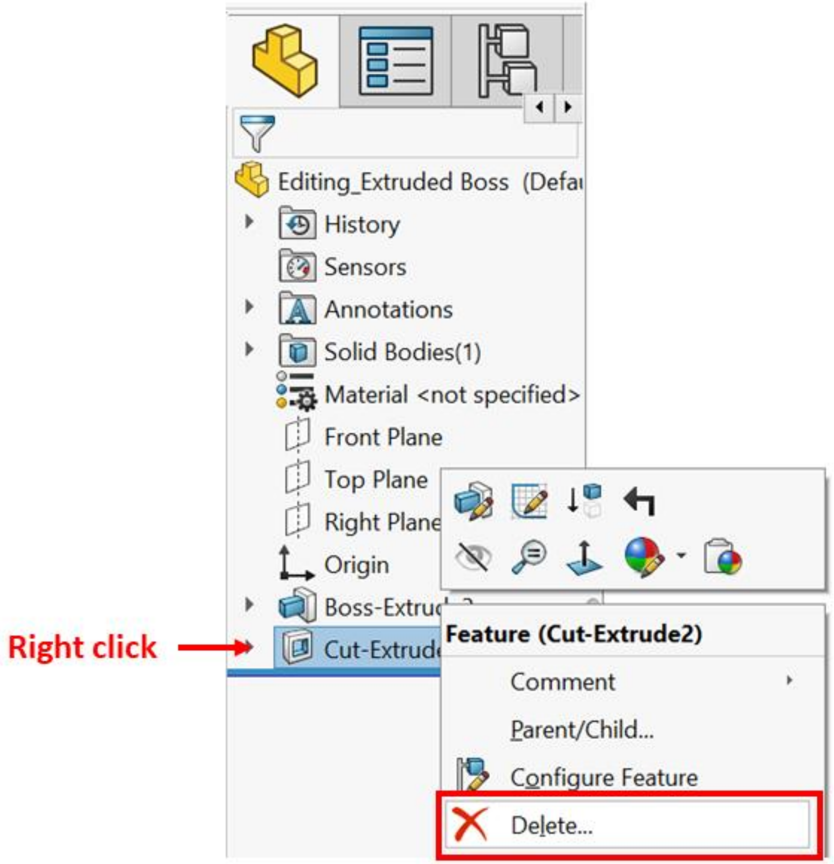
Editing a feature

To edit an implemented feature, we can right-click (or left-click) on it on the design tree and select the **Edit Feature** option.



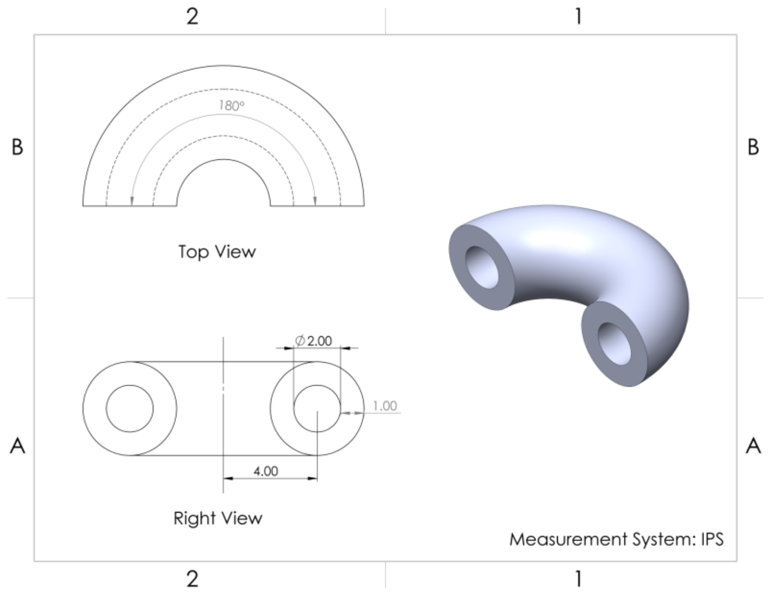
Deleting a feature

To delete a feature, right-click on a feature in the design tree and select the **Delete** option.



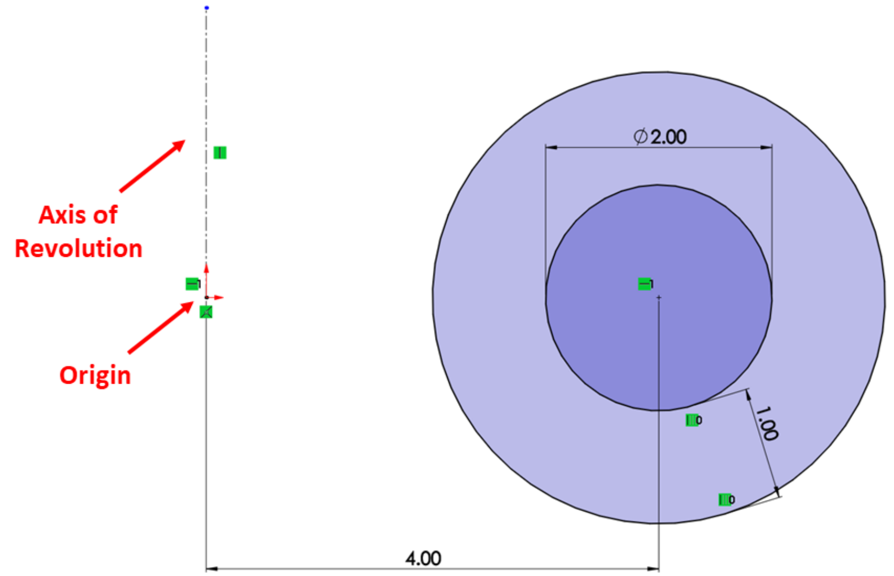
Applying revolved boss

The revolved boss feature adds materials by revolving a sketch around an axis of revolution. To show you how to apply the revolved boss feature, we will create the following model. Now that we are using more and more features, we will start to notice that we use the same options repeatedly. For example, when applying the revolved boss, we notice that most options are the same ones that we use while applying extruded boss. Therefore, we won't explain these again here:



As usual, we will follow our standard procedure planning, sketching, and applying features:

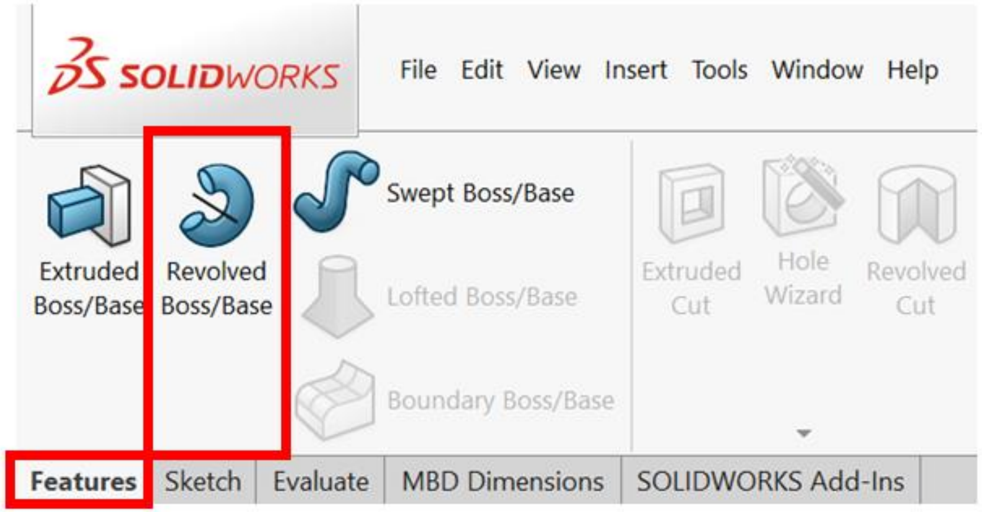
- **Planning**: Here, we'll draw the profile shown in the right view of the preceding drawing, and then we'll rotate that by **180** degrees.
- **Sketching**: Here, we're going to sketch the profile highlighted in the **Right View** using the right plane. This includes using the axis of revolution. Our sketch should look as follows:



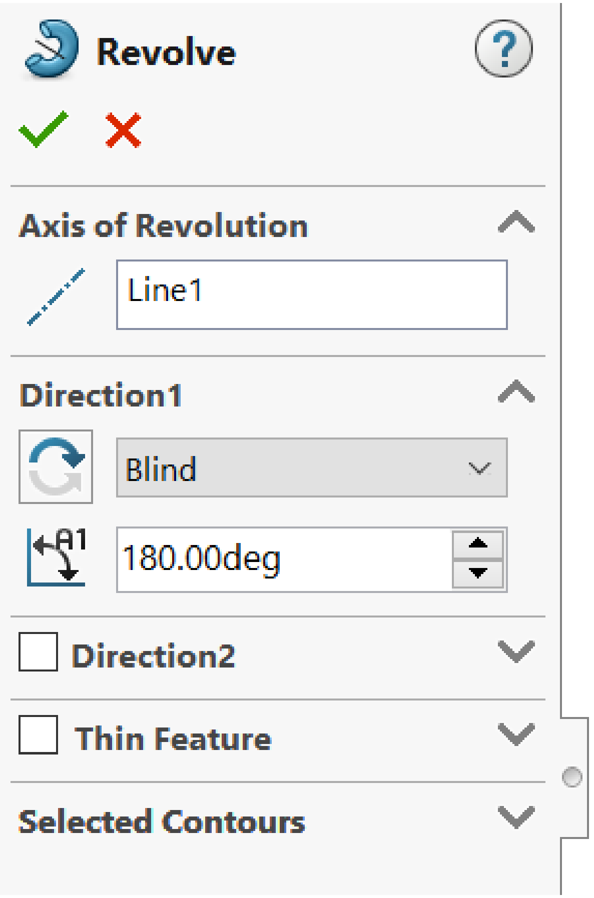
- **Applying the feature**: Here, we are going to apply our revolved boss feature.

To apply the revolved boss feature, follow these steps:

1. Select the **Revolved Boss/Base** command from the **Features** command bar. We don't need to exit the sketch to select the command. However, if we do exit the sketch for whatever reason, we can select the **Revolved Boss/Base** command and then select the sketch we want to apply it to:



2. Adjust the options in the PropertyManager, as shown in the following screenshot:



Here is a brief description of the unique options that can be used with the revolved feature

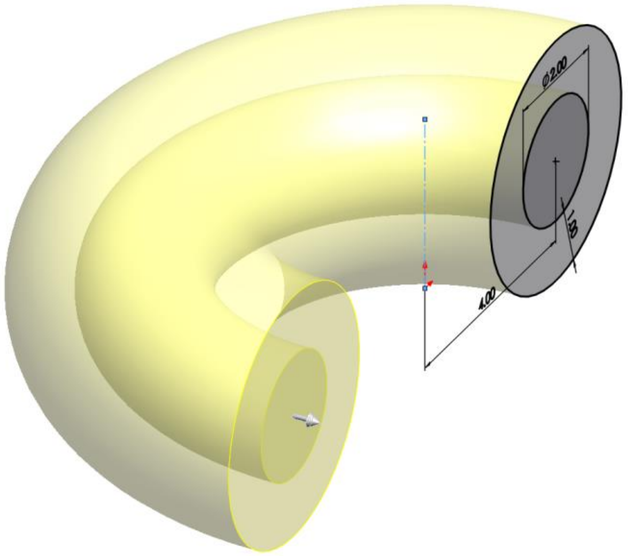
- **Axis of Revolution**: This can be any straight line located on the canvas. This axis will be used to apply the feature by rotating the sketch around that axis. In our exercise, we will select the centerline indicated in the preceding diagram as the axis of revolution.

- **End Condition**: This provides a few options regarding how the revolution will stop. In this exercise, we will use the **Blind** condition. By doing this, we will decide on the end of the revolution by determining the angle of rotation.

- **Reverse Direction**: This is the two curved arrows next to the end condition field. Clicking on this icon will reverse the rotation direction of the revolved boss.

- **Angle (A1)**: This determines when the revolution stops. In this exercise, the revolution will stop at 180 degrees.

3. Since we're setting the feature's options, we will be able to see a preview of the feature on the canvas. This will look as follows:



The rest of the options (**Direction 2**, **Thin Feature**, and **Selected Contours**) were explained when we look at the extruded boss feature. They have the same functionality.

4. To apply the revolved boss feature, we can click on the green checkmark at the top of the **Options** panel. The resulting model should look as follows: