

SOLIDWORKS DRAWING COMMANDS CHEAT SHEET

5 Essential Drawing Commands Every SolidWorks User Should Know

1. Orthographic View

What it does: Shows a 3D model from different 2D directions, such as Front, Top, and Side.

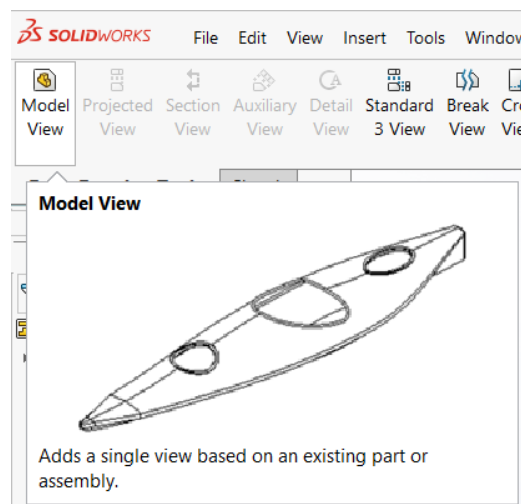
Use it for: Clearly communicating the shape, size, and features of a part in an engineering drawing.

How it works:

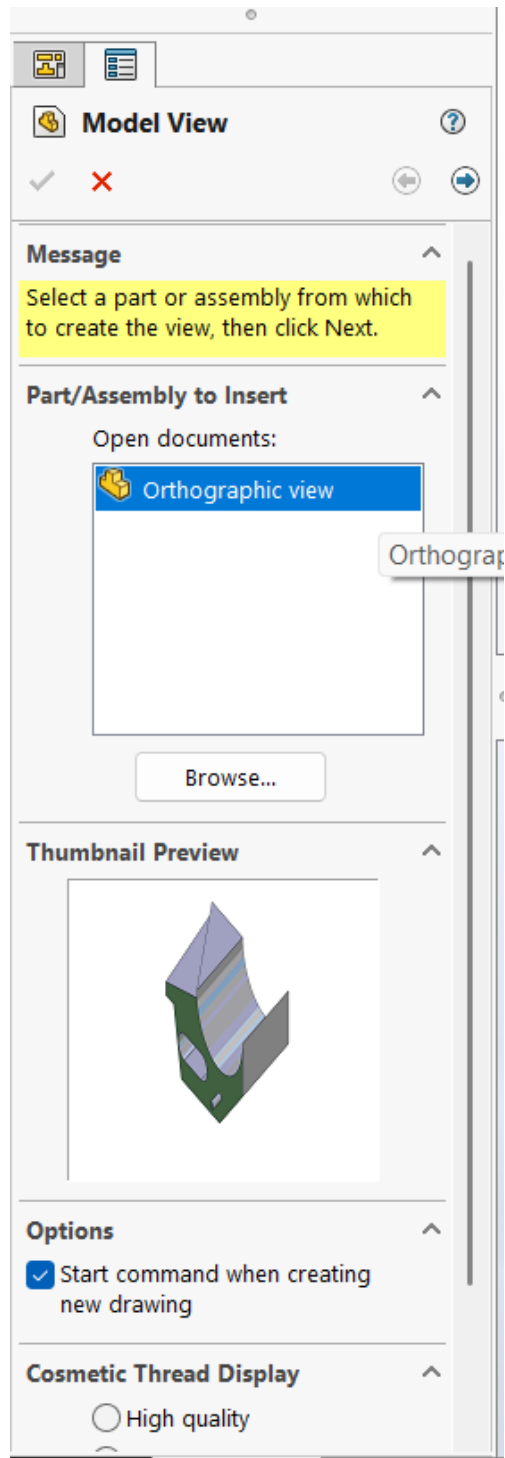
Orthographic views are 2D drawing views created from a 3D model, such as the **Front**, **Top**, **Right**, and **Isometric** views.

These views help clearly show the shape and size of the part from different directions.

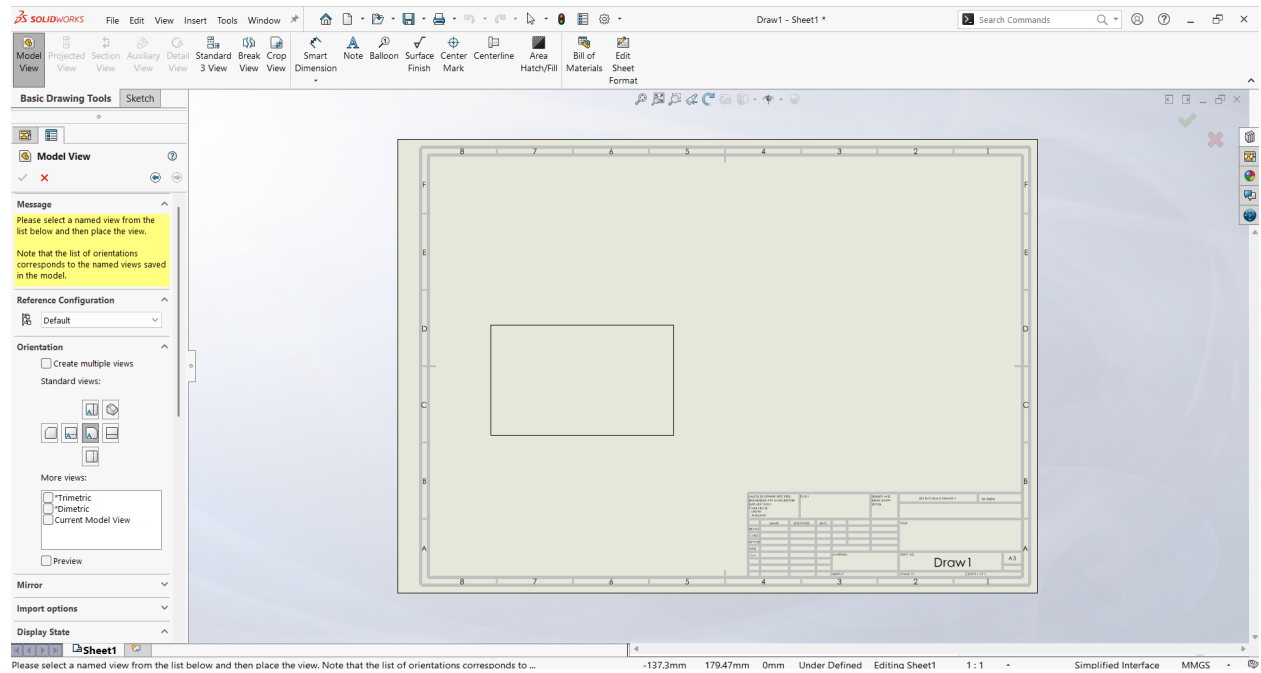
To begin, click the **Model View** tool.

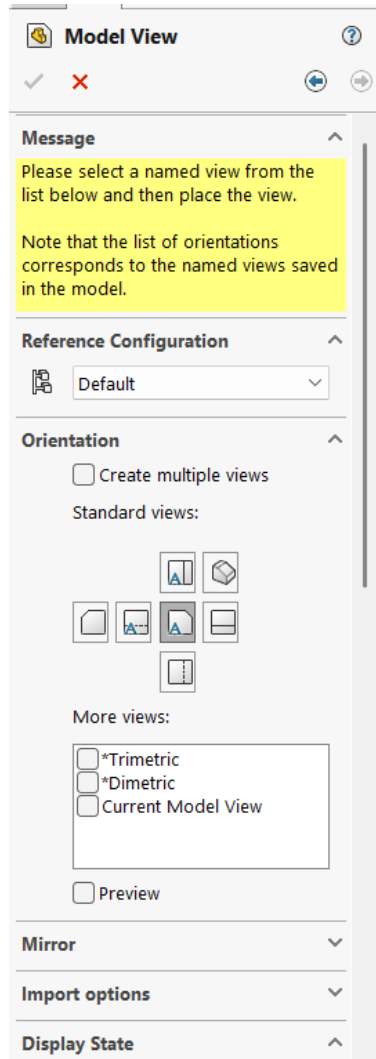


Double-click the part or assembly you want to use. If you do not see it in the list, click **Browse** and select the file manually.

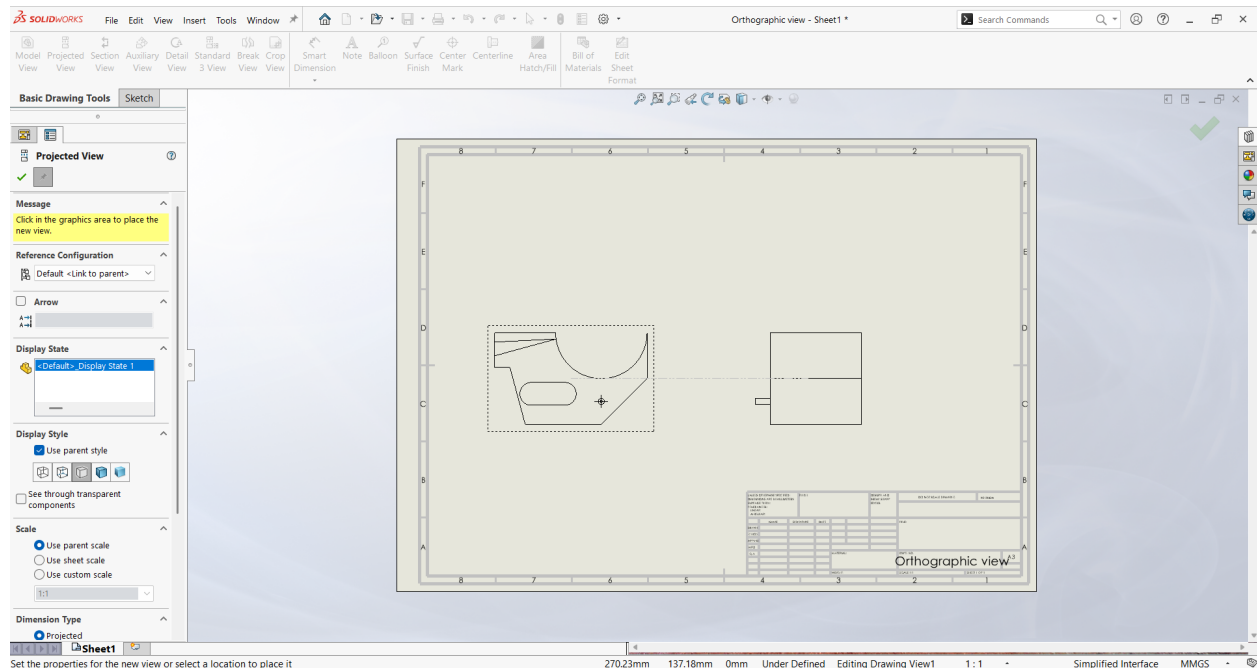


Move the mouse over the drawing sheet. On the left side, in the **PropertyManager**, you can choose which view you want to place under **Orientation**.

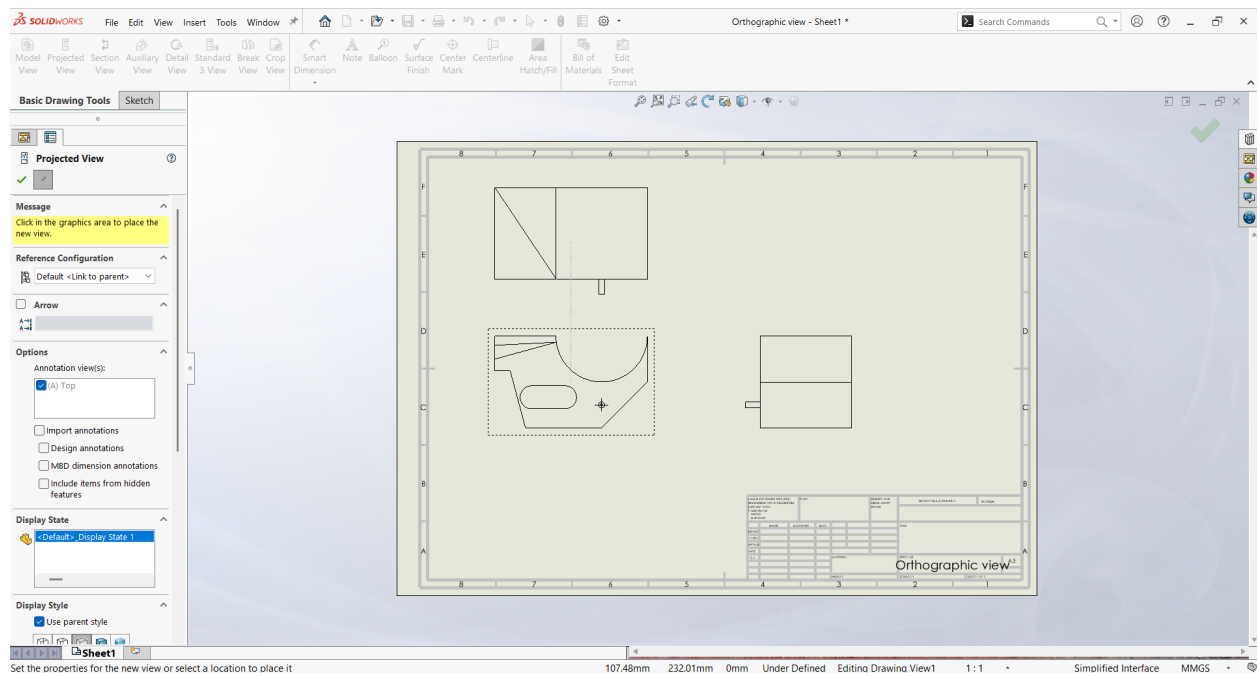




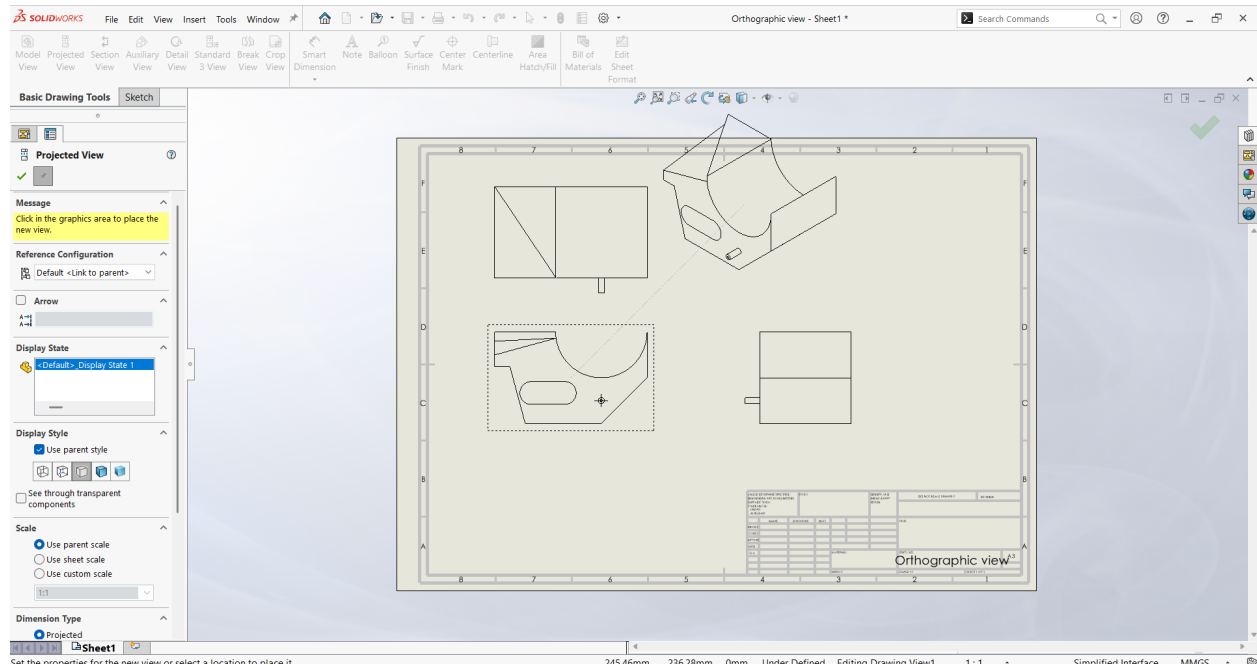
Click on the drawing sheet to place the first view. If you move the mouse to the left, SOLIDWORKS automatically creates another projected view. Click again to place the second view.



When you move the mouse upward, SOLIDWORKS automatically creates another projected view. Click again to place this view on the drawing sheet.

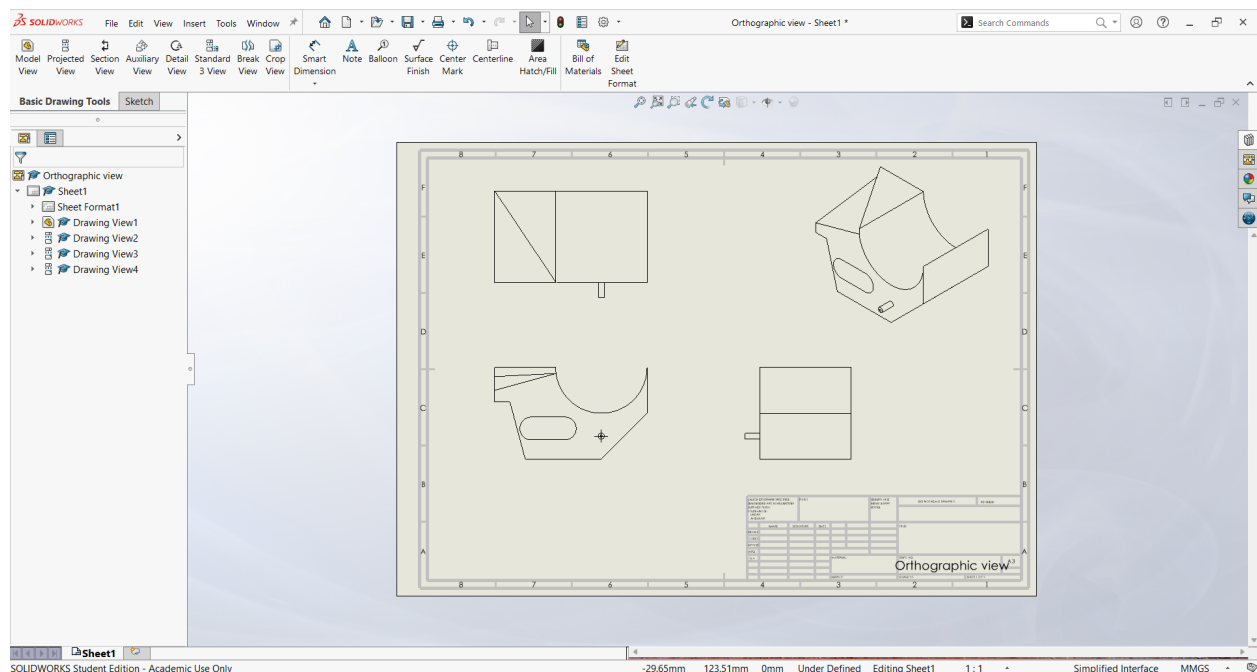


When you move the mouse diagonally to the side, SOLIDWORKS automatically creates an **Isometric View**. Click again to place the view on the drawing sheet.



To finish with view creation just press Esc.

To move a drawing view, click and hold the view, then drag it with the mouse. You can position the view anywhere you want on the drawing sheet.



2. Projected View

What it does: Shows a 3D model from different 2D directions, such as Front, Top, and Side.

Use it for: Clearly communicating the shape, size, and features of a part in an engineering drawing.

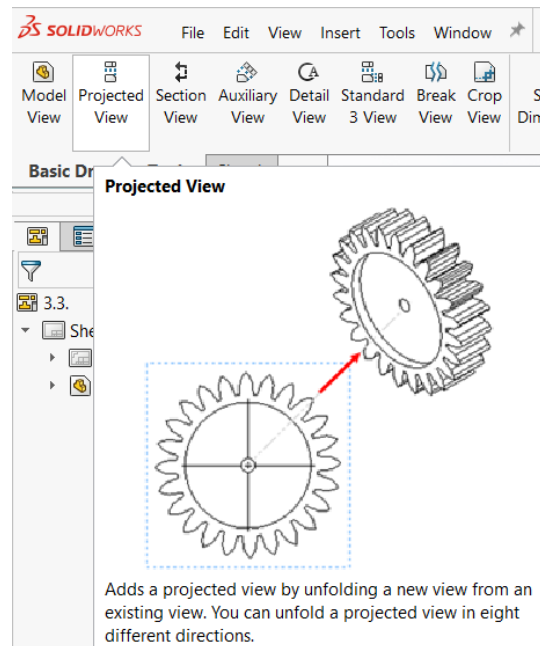
How it works:

A **Projected View** is a drawing view created from an existing view.

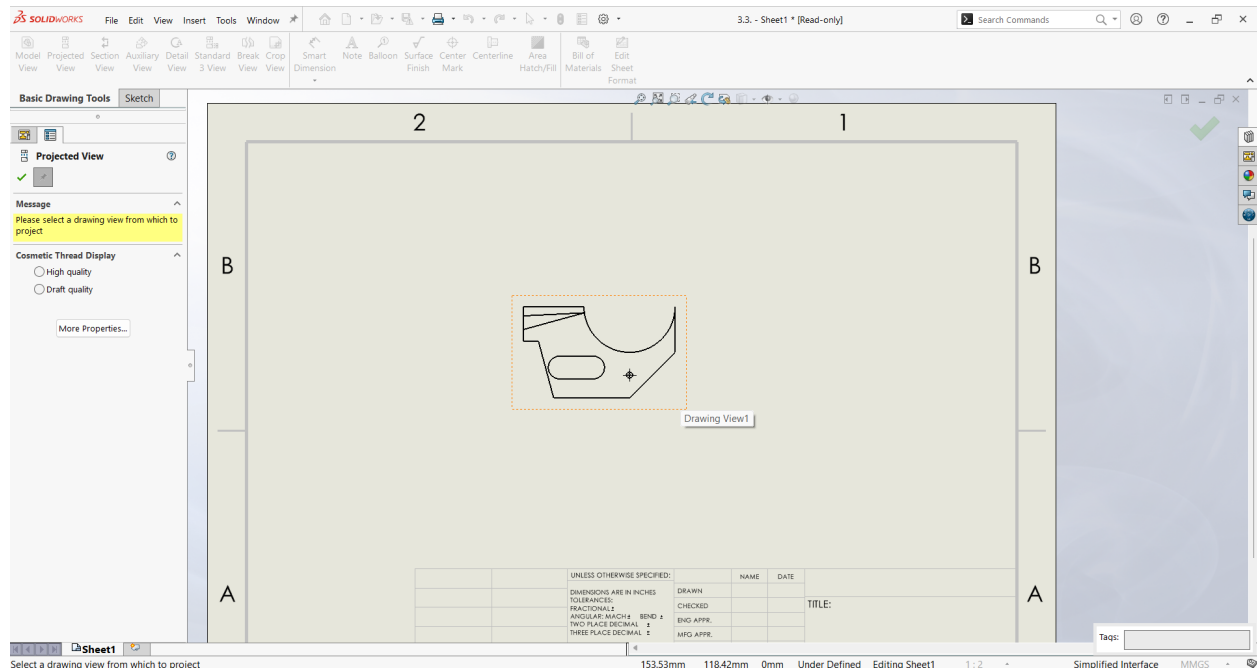
It shows the same part or assembly from another direction, such as the top, right, left, or bottom side.

Projected views are useful because they help show the complete shape of the model in a clear 2D drawing.

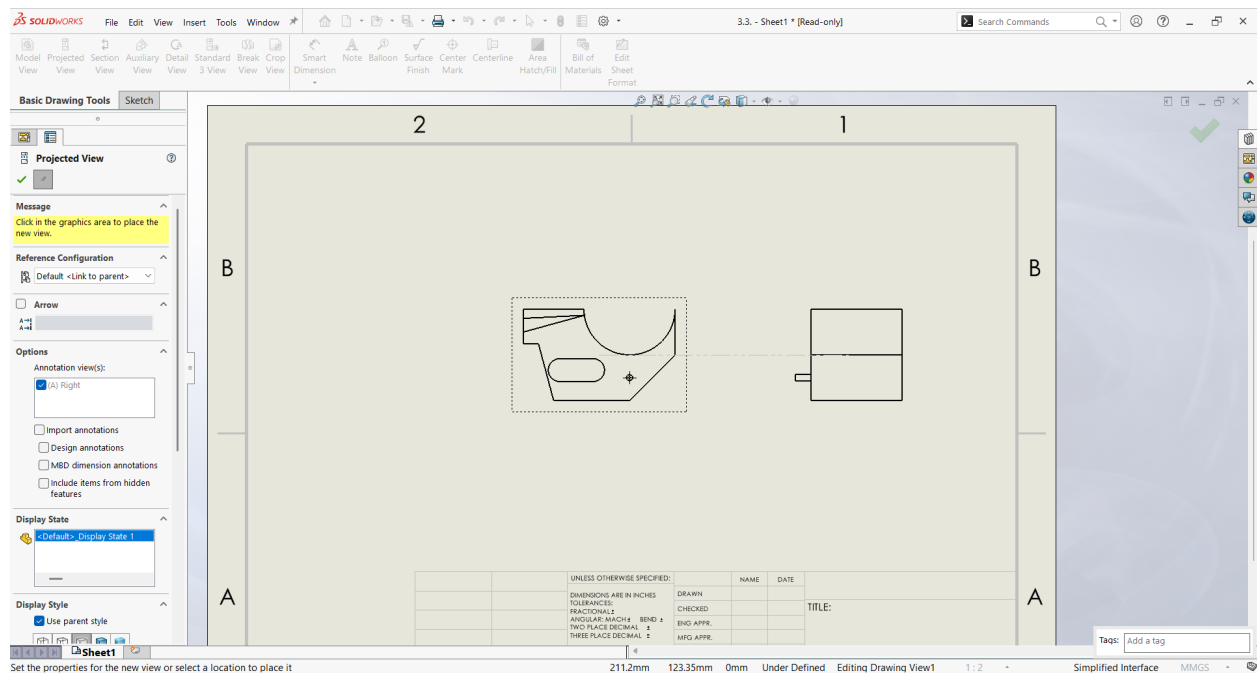
To create a **Projected View**, go to the **Basic Drawing Tools** tab and select **Projected View**.



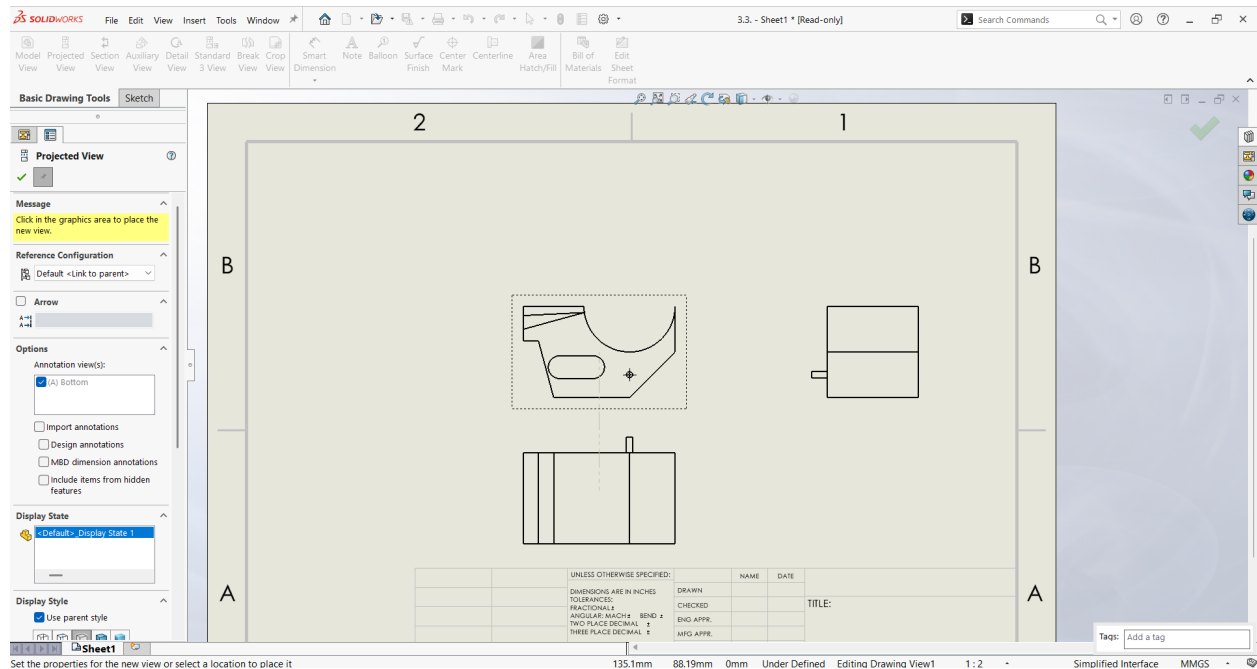
Click on the existing drawing view.



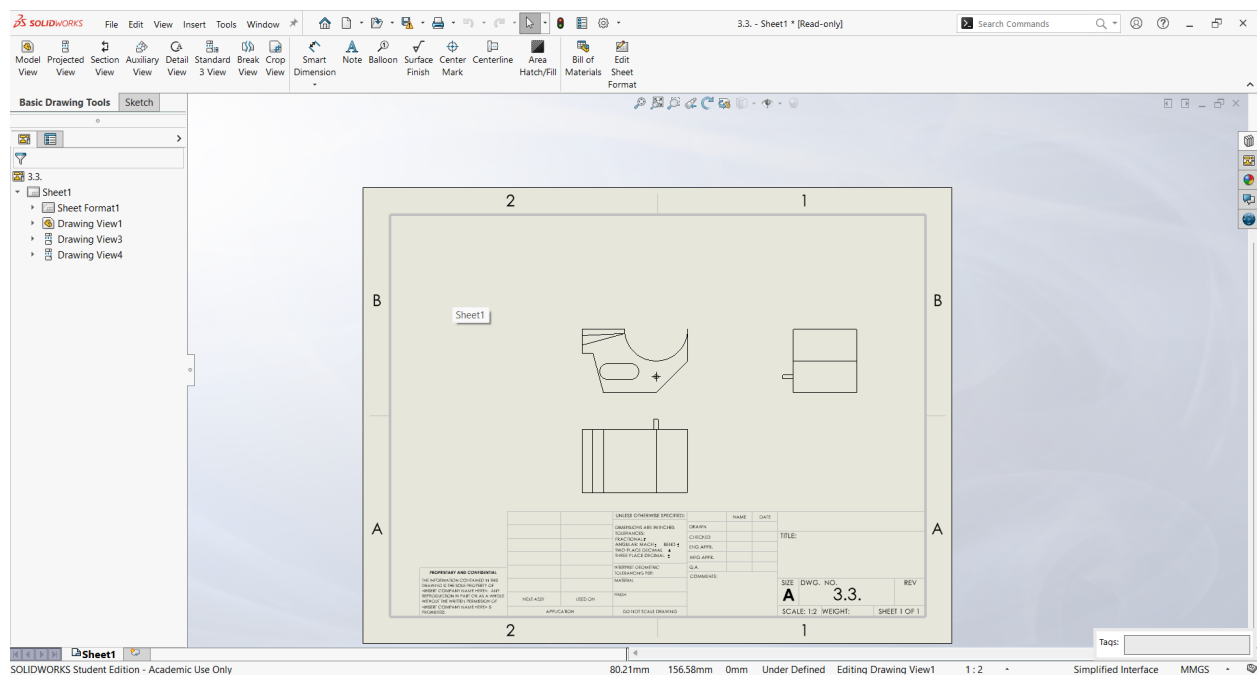
Move the mouse to the right side of the existing view. Then click on the drawing sheet to place the projected view.



Move the mouse downward and click to place the second **Projected View**.



When you are finished, click **OK**.



3. Section View

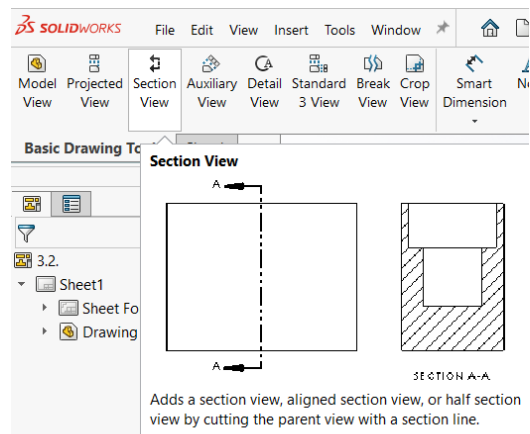
What it does: Cuts through the model virtually to reveal internal geometry.

Use it for: Showing internal features that are difficult to communicate with standard views.

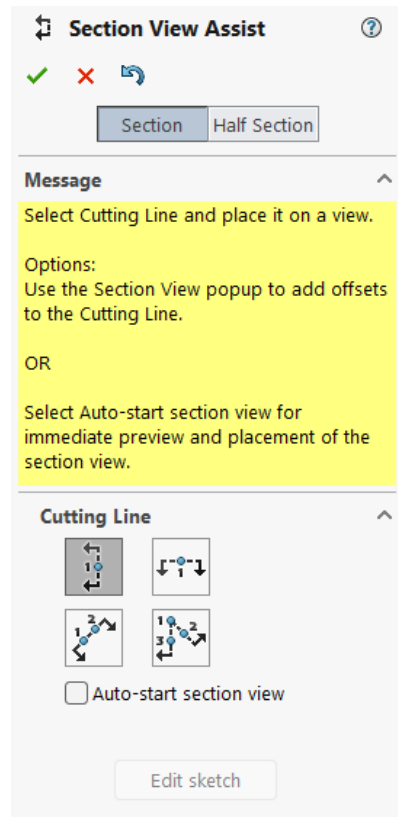
How it works:

A **Section View** is a drawing view that shows the inside of a part or assembly. It is created by cutting through the model with a section line. This allows us to see internal features that are not clearly visible from the outside, such as holes, slots, pockets, or hidden geometry.

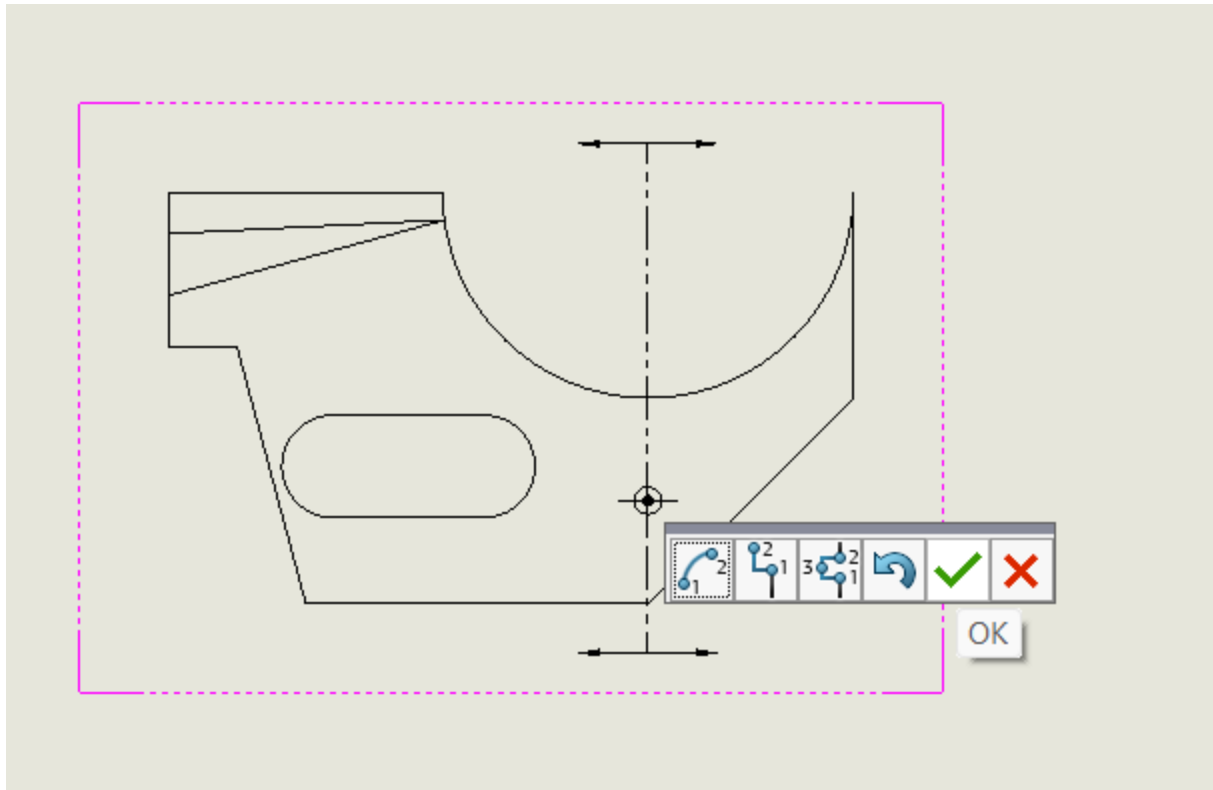
To create a **Section View**, go to the **Basic Drawing Tools** tab and select **Section View**.



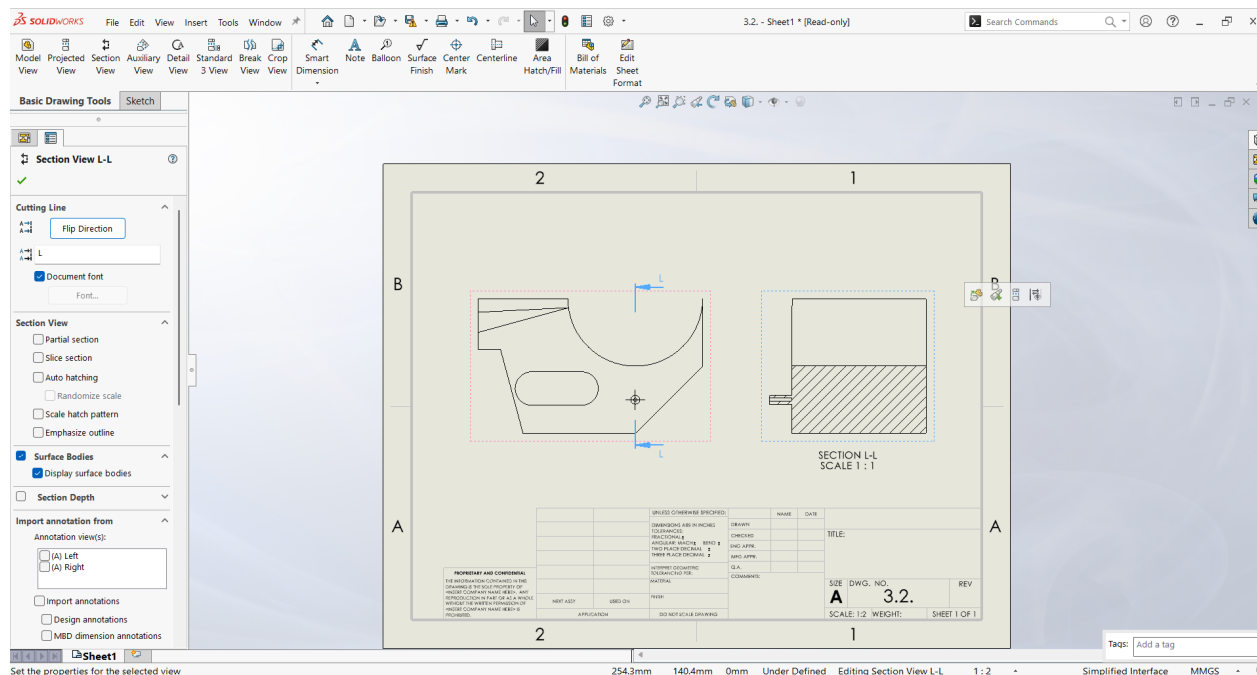
First, choose the cutting line direction. In this example, select a **Vertical** cutting line.



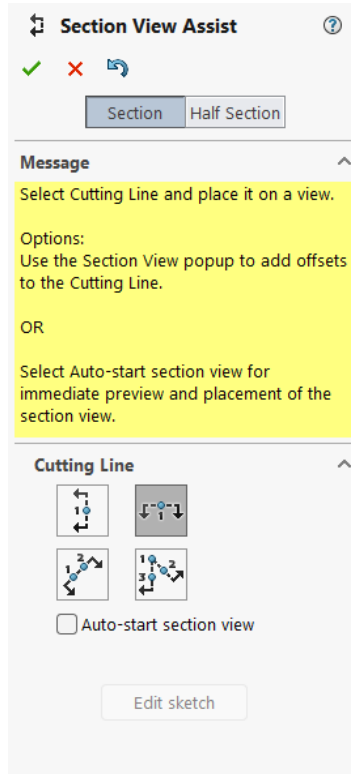
Click on the drawing view to place the vertical cutting line, as shown.



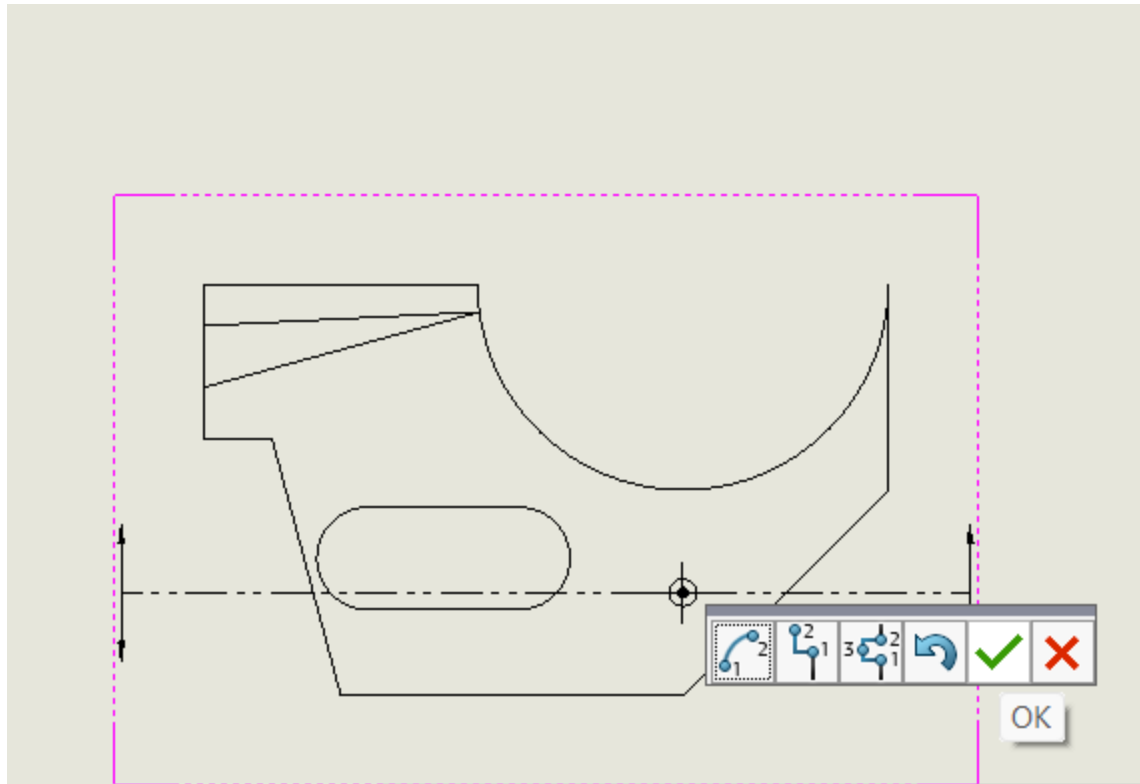
Click on the green check mark to confirm it. Move the mouse to the right side of the original view. Click on the drawing sheet to place the **Section View**. The view properties appear on the left side in the **PropertyManager**. Here, you can change options such as the view name.



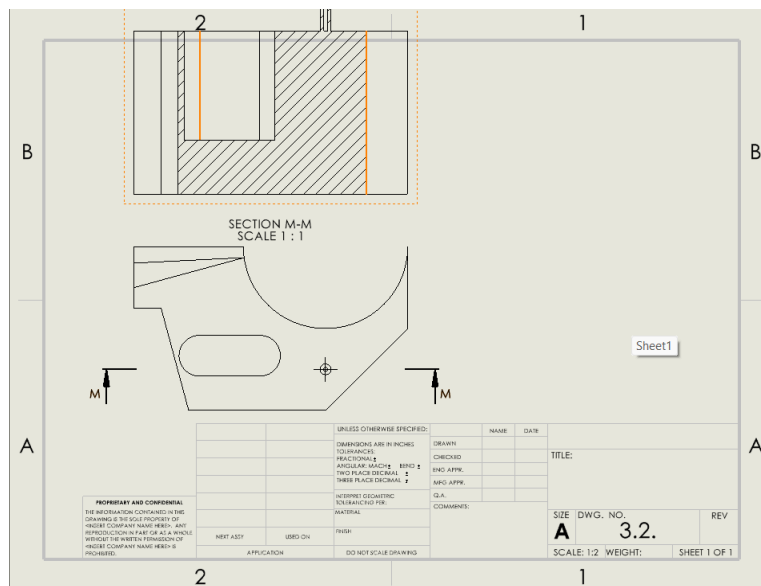
Select the **Section View** tool again. This time, choose a **Horizontal** cutting line.



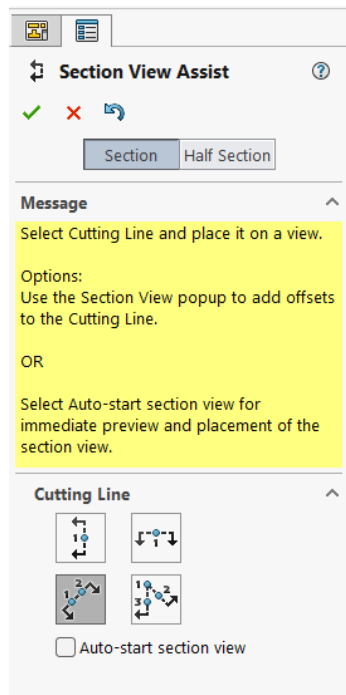
Position the cutting line as shown and click on it.



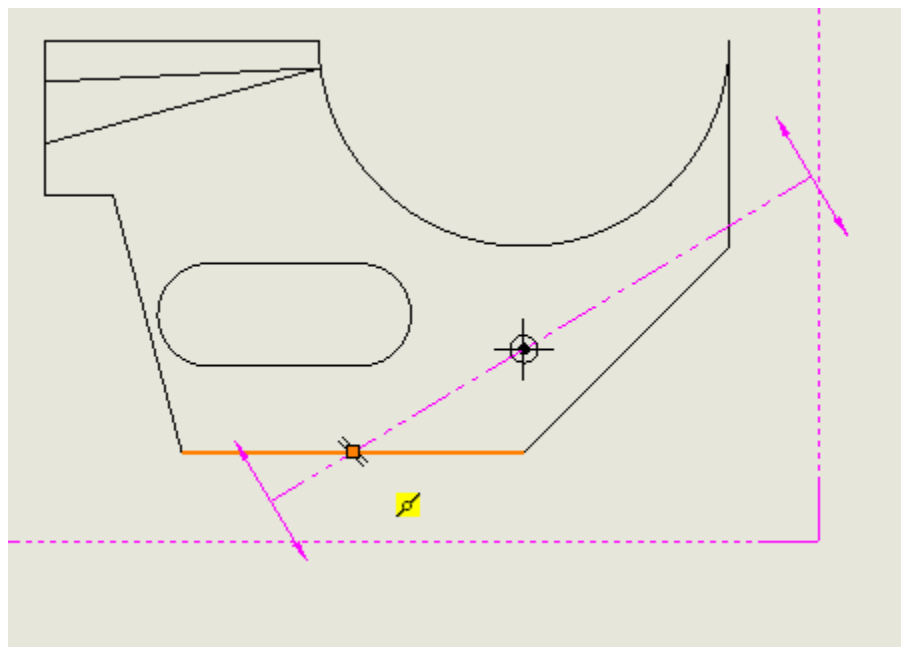
Click on the green check mark to confirm it. Move the mouse upward to create the **Section View**. Click on the drawing sheet to place the view.



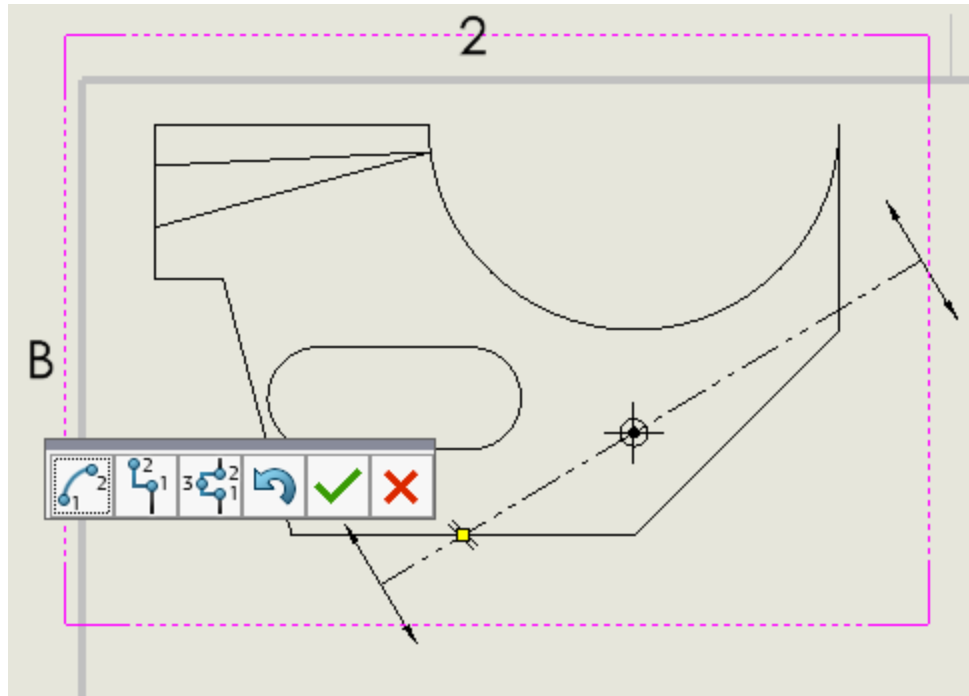
Go to the **Section View** tool again. This time, choose an **Aligned** cutting line. Move the mouse over the drawing view to position the cutting line.



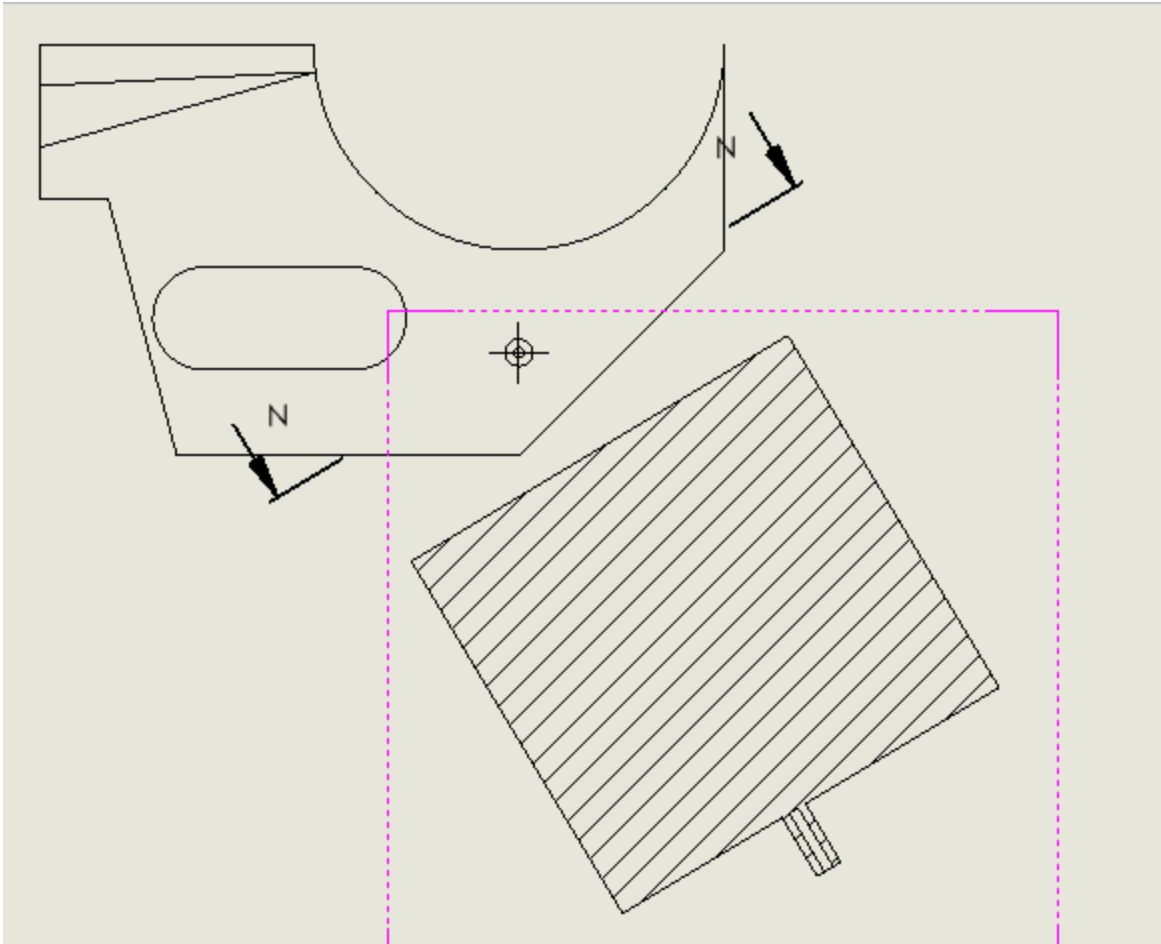
To position the cutting line, you need to select two points. Click to select the first point in the center of the small circle.



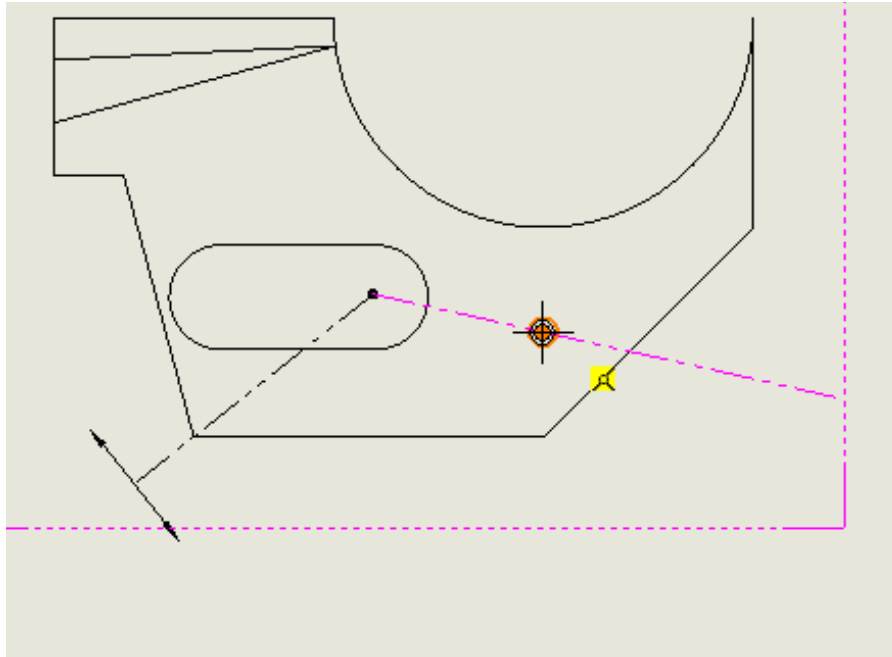
Click again to select the second point on the horizontal line.



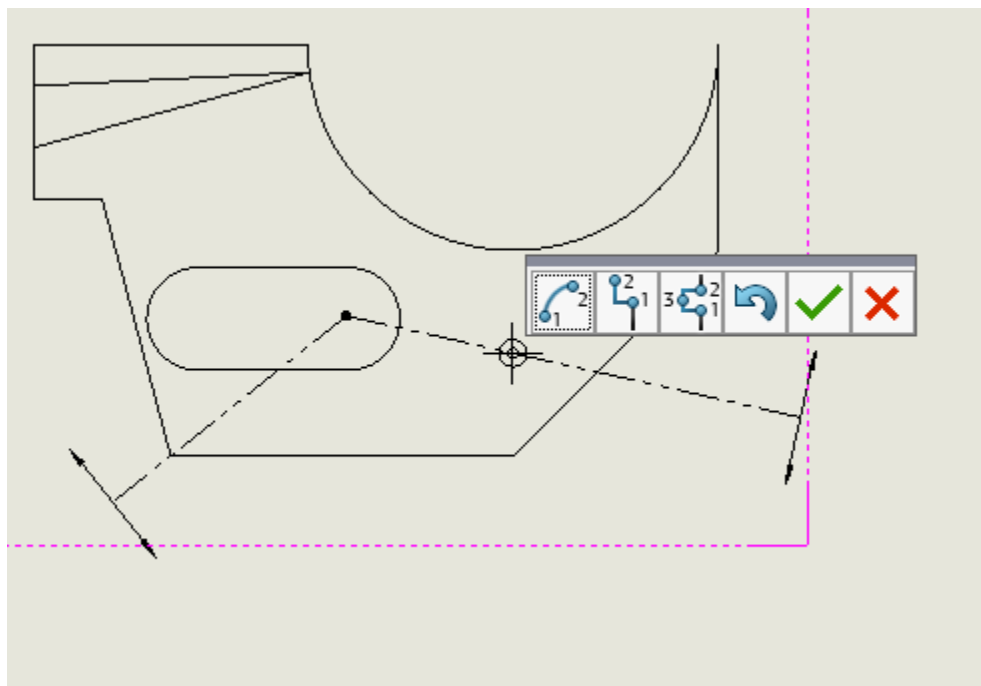
Click on the green check mark to confirm it. Move the mouse to create the section view. Click on the drawing sheet to place the view.



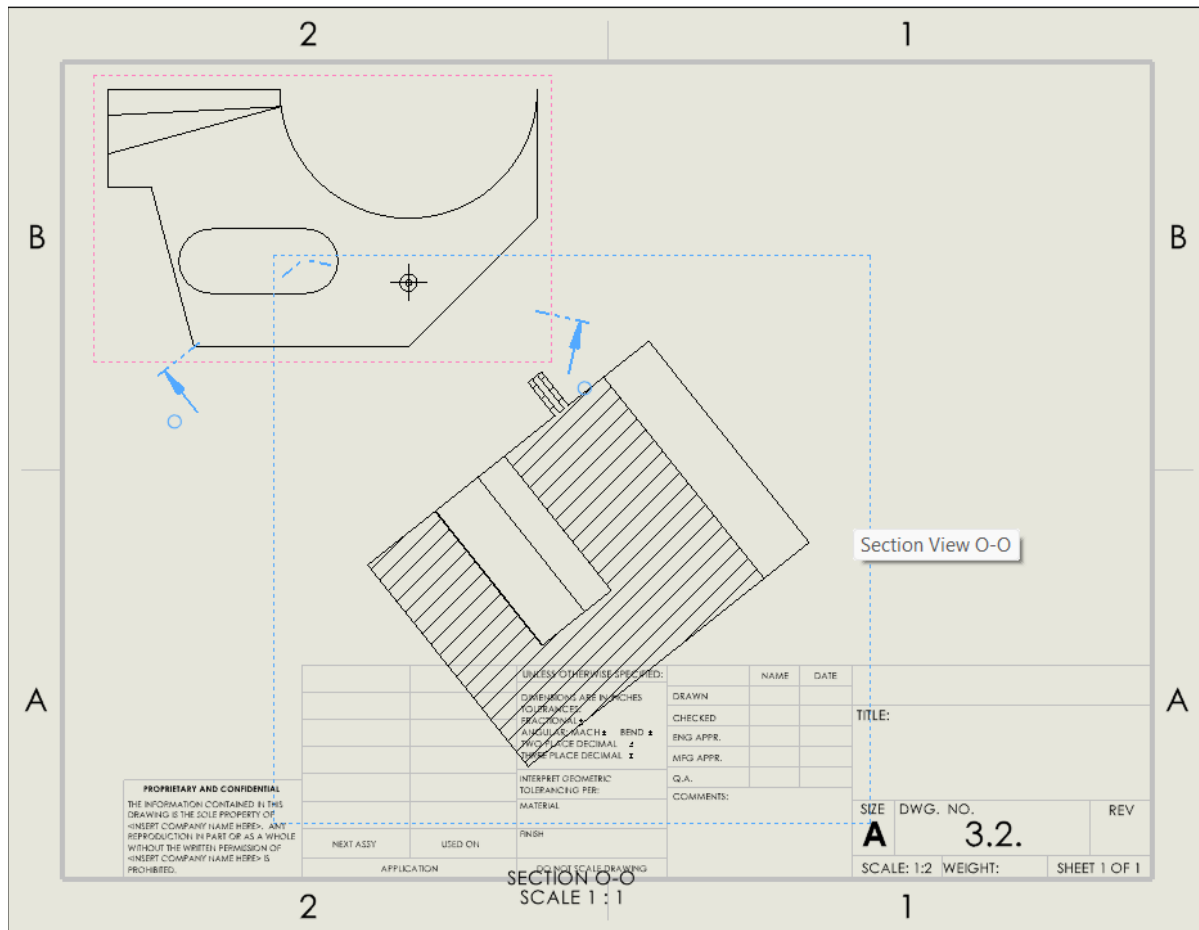
The last option is **Offset Cutting Line**. Go to the drawing view and define three points. First, select the center point.



The remaining two points define the angle of the cutting line.



Click on the green check mark to confirm it. Move the mouse to drag out the view. Click on the drawing sheet to place it.



4. Detail View

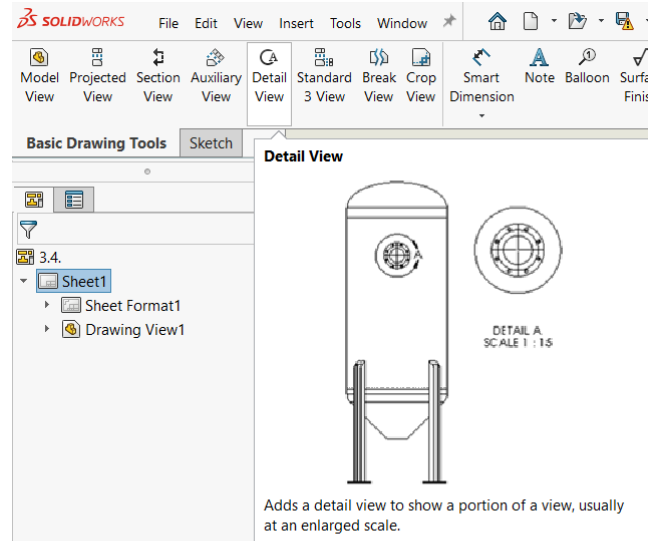
What it does: Enlarges a selected area of a drawing.

Use it for: Small features that require additional clarity.

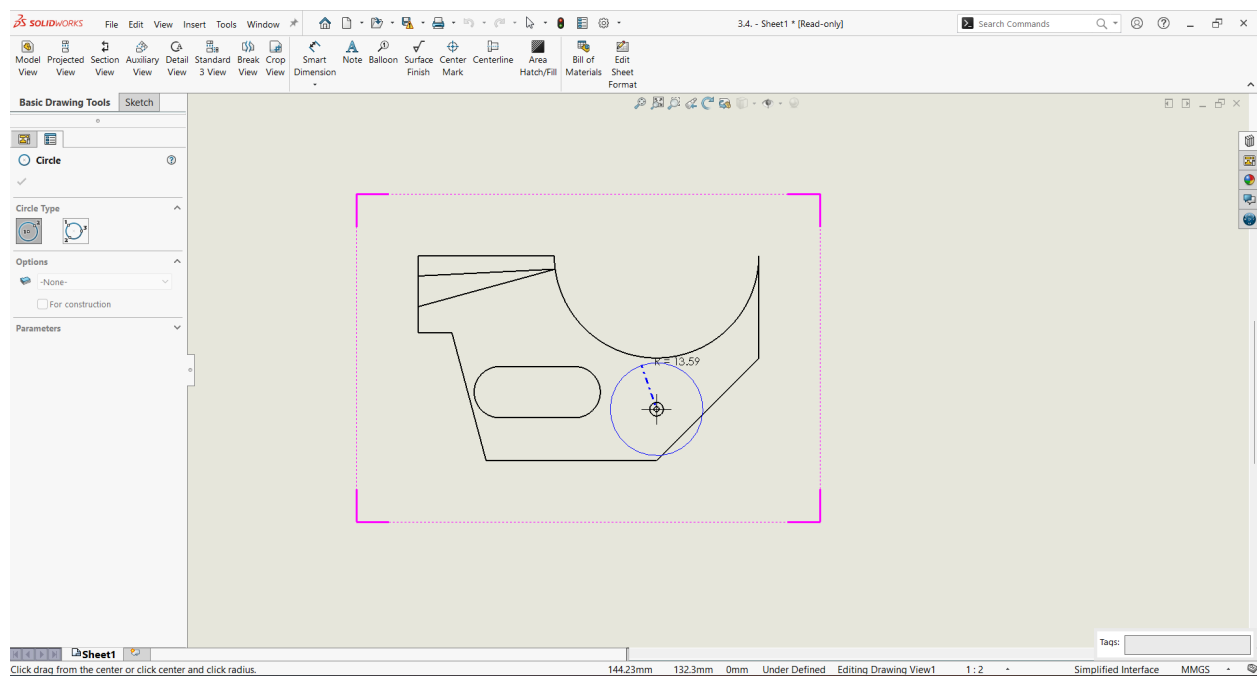
How it works:

To enlarge small details in a drawing, we use **Detail View**. A **Detail View** creates a magnified view of a selected area of the model. This is useful when small features are hard to see clearly in the main drawing view, such as small holes, slots, fillets, or fine geometry.

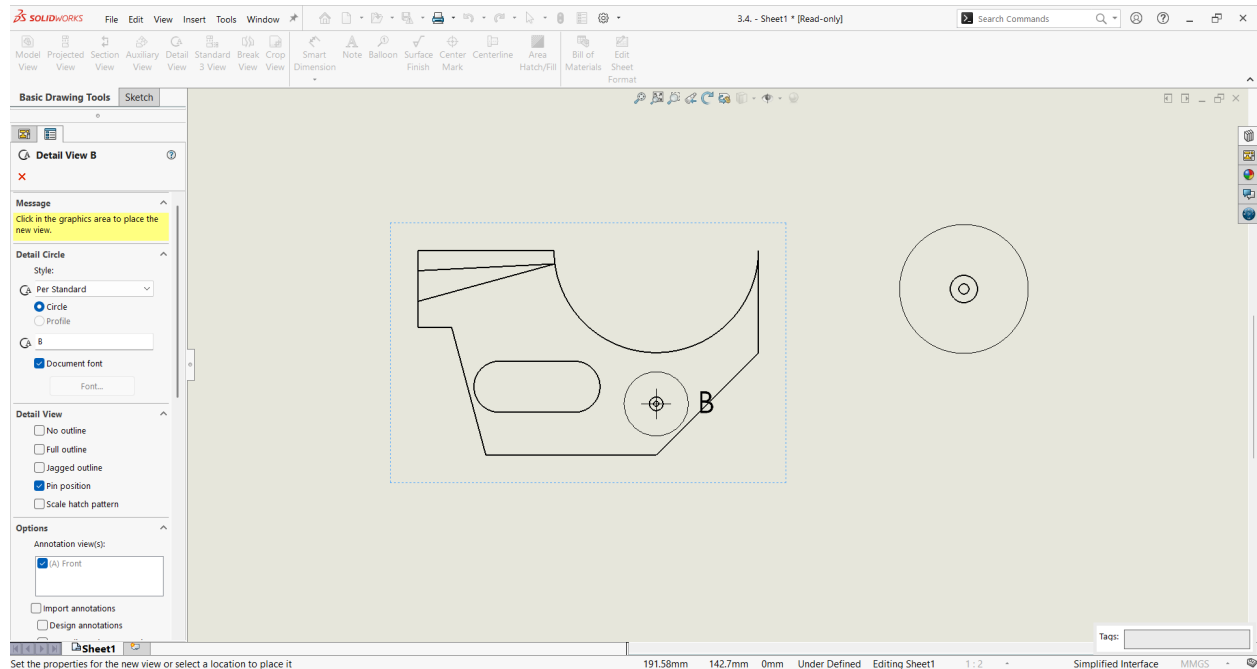
To create a **Detail View**, go to the **Basic Drawing Tools** tab and select **Detail View**.



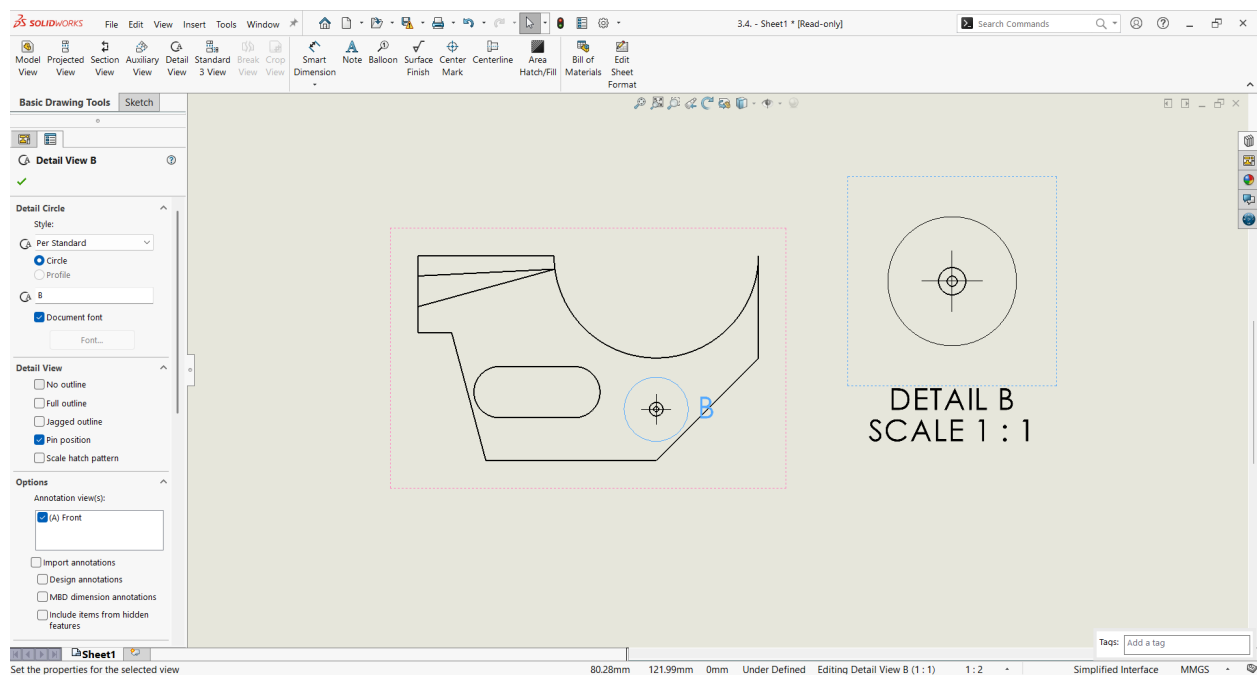
The next step is to draw a circle around the area you want to enlarge. Make sure the detail you want to show is completely inside the circle.



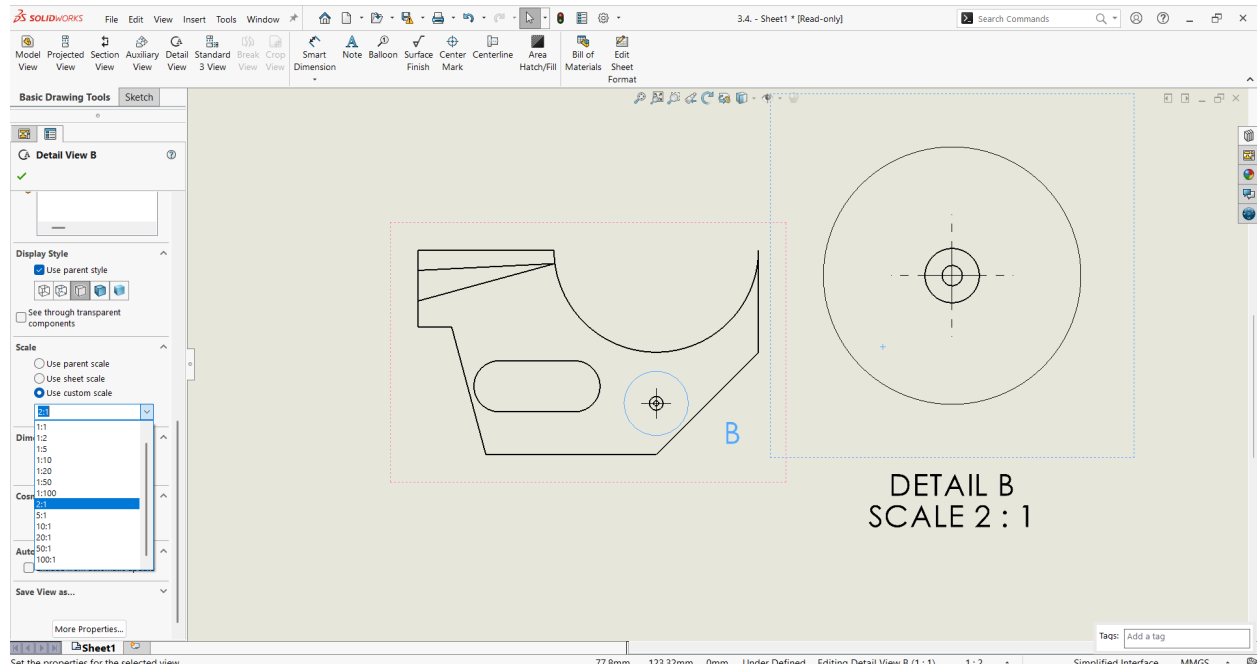
Move the mouse to the right side of the original view. Click on the drawing sheet to place the **Detail View**.



The view properties appear on the left side in the **PropertyManager**. Here, you can change options such as the view name and scale.



Set the scale to **2:1**.



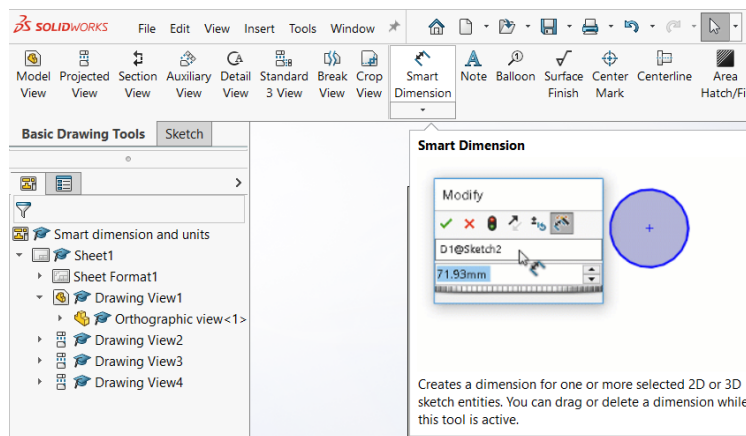
5. Smart Dimension

What it does: Adds dimensions to drawing geometry.

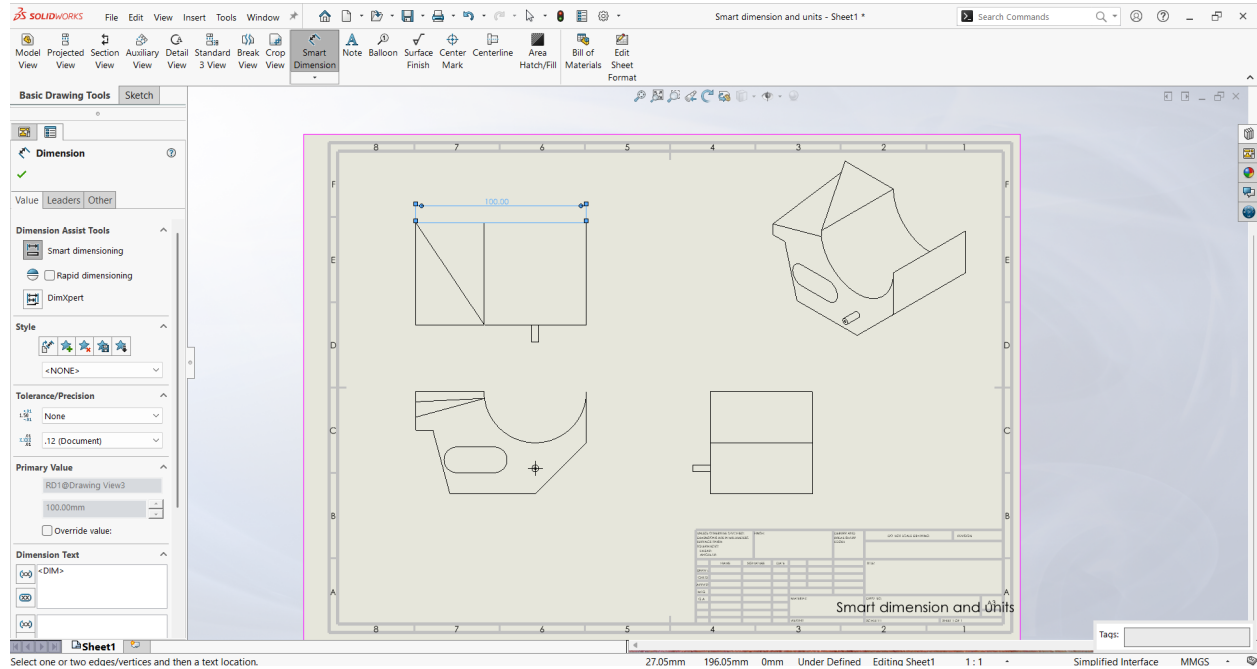
Use it for: Communicating sizes and locations required to manufacture or inspect the component.

How it works:

To add dimensions to a drawing view, we use the **Smart Dimension** tool. This works the same way as when adding dimensions in a sketch.

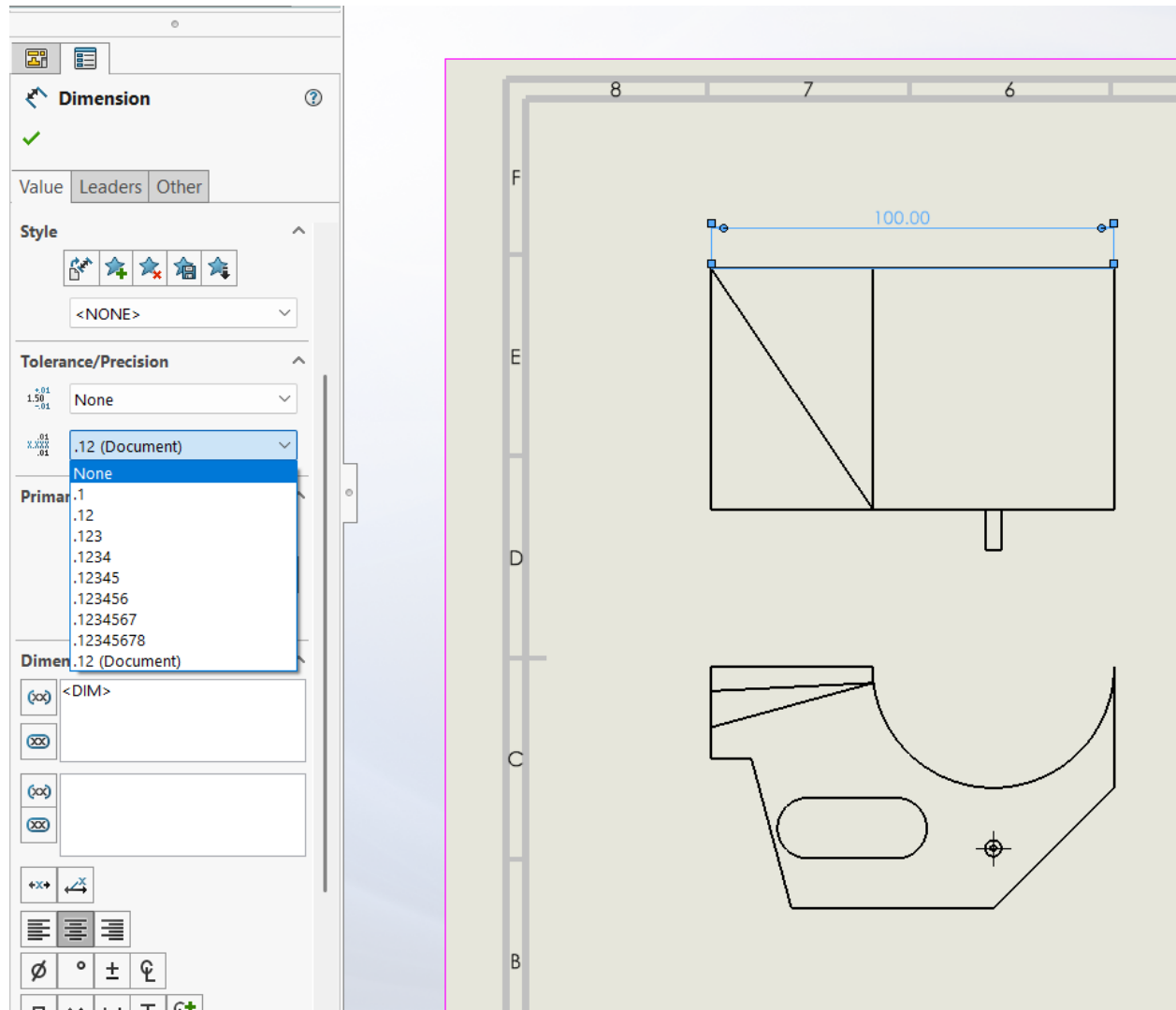


Go to the **Basic Drawing Tools** tab and select **Smart Dimension**.

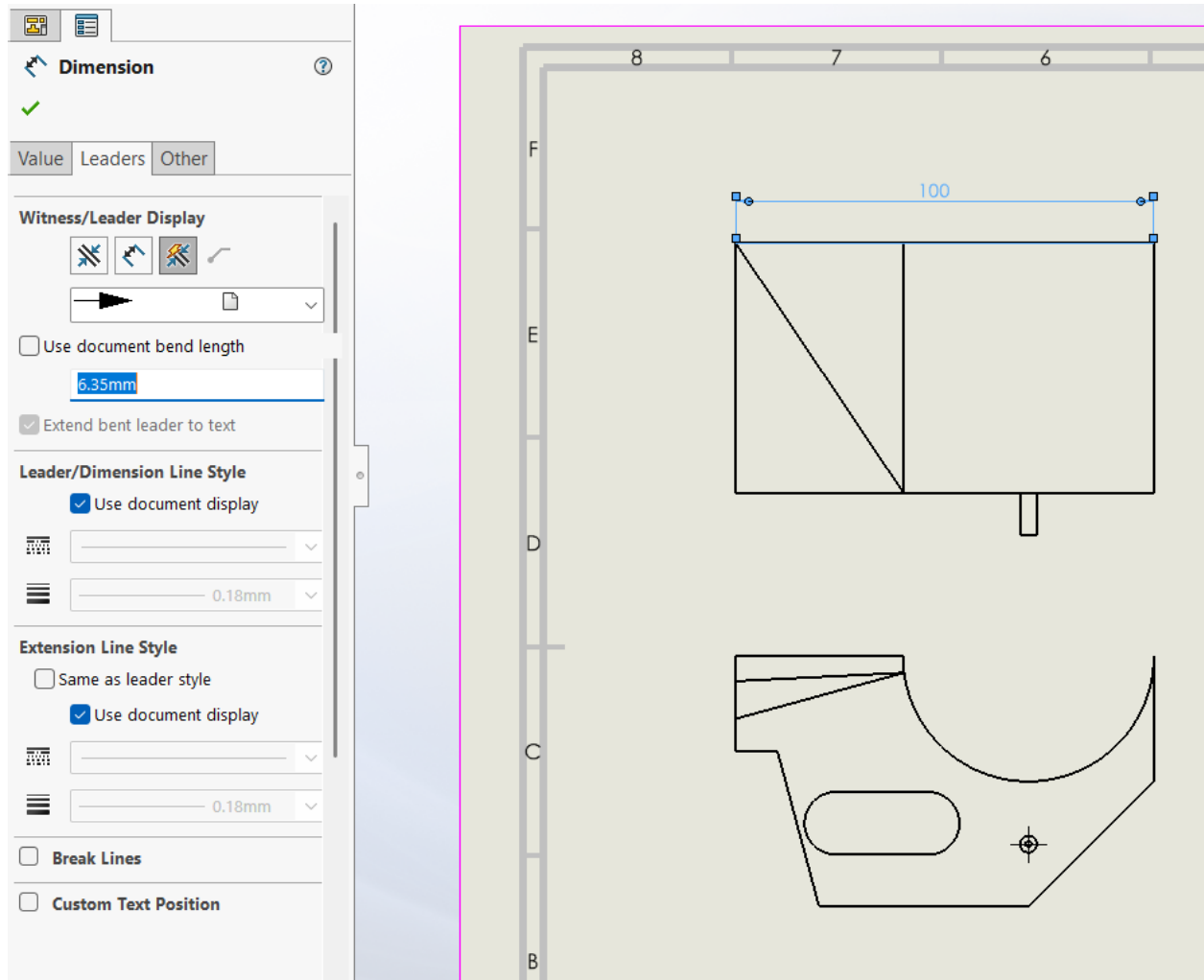


Adding dimensions works the same way as in sketching. If you want to add the length of a line, you can select the line directly. You can also select the two endpoints of the line, or select two vertical lines that represent the start and end of the distance.

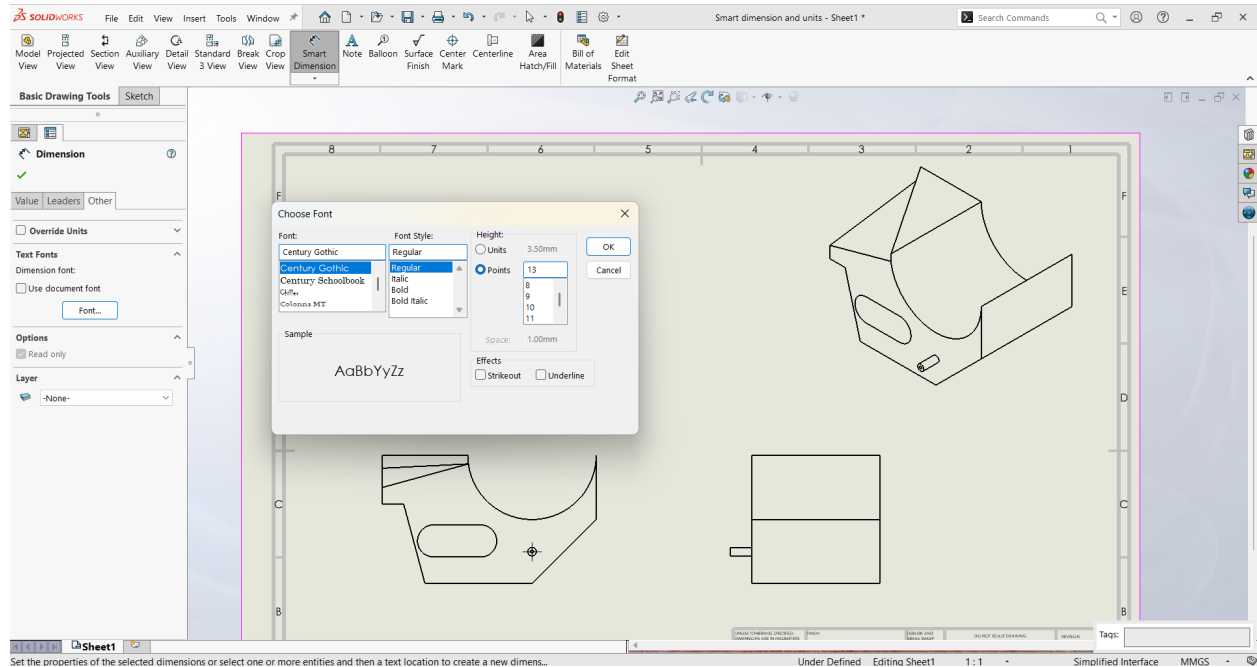
To change the number of decimal places, click on the dimension. Then go to the **Tolerance/Precision** and select how many decimal places you want to display.



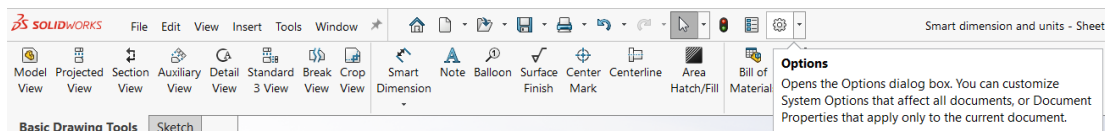
To change the size of the leader, click on the dimension. Then go to the **PropertyManager** on the left side and select the **Leaders** tab. Under **Witness/Leader Display**, define the leader length.



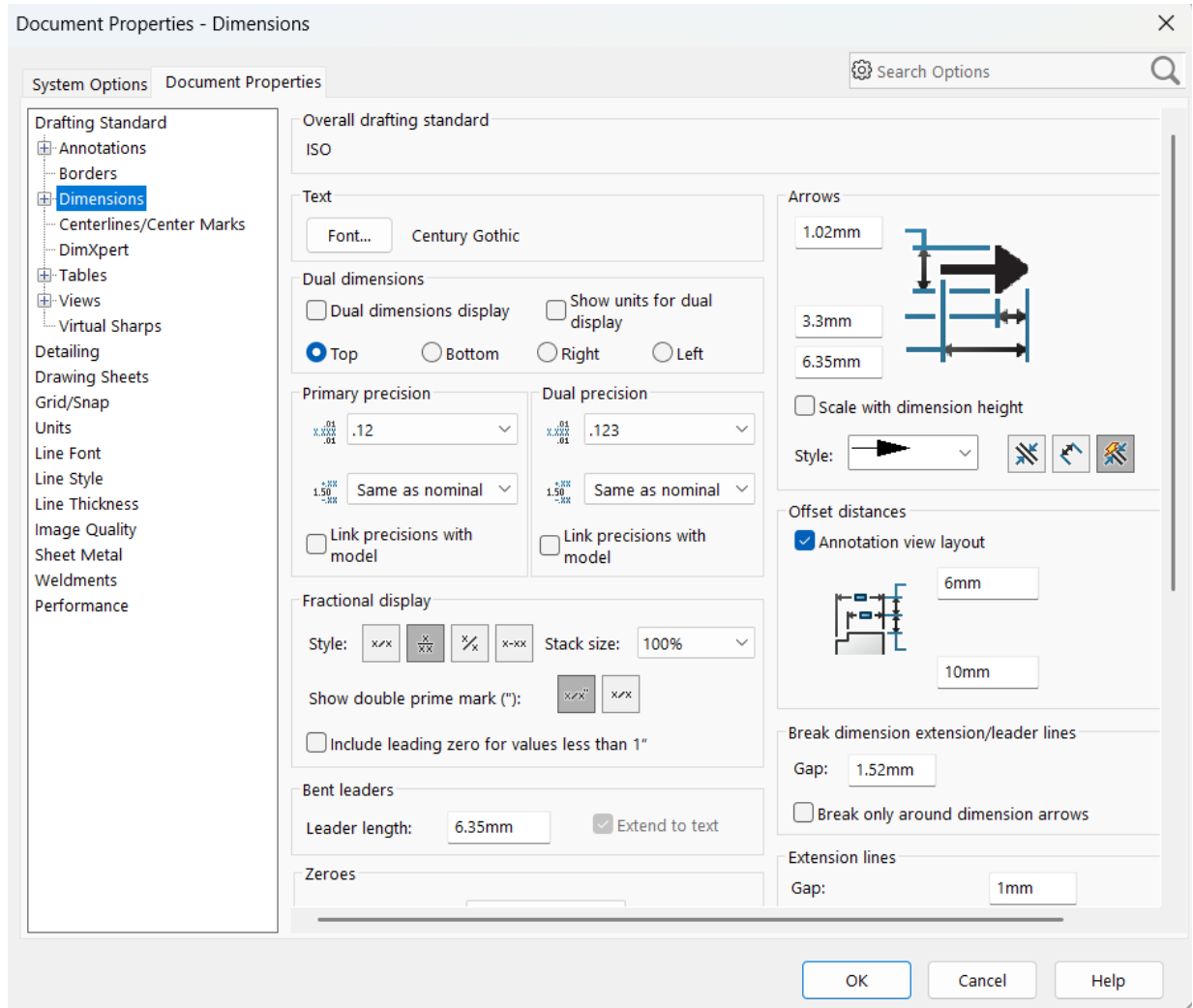
To change the font, go to the **Other** tab in the dimension properties. Then select **Font**. A new dialog box opens. Under **Height**, choose **Points** and define the font size.



To change the properties for the entire drawing sheet, go to **Options**.

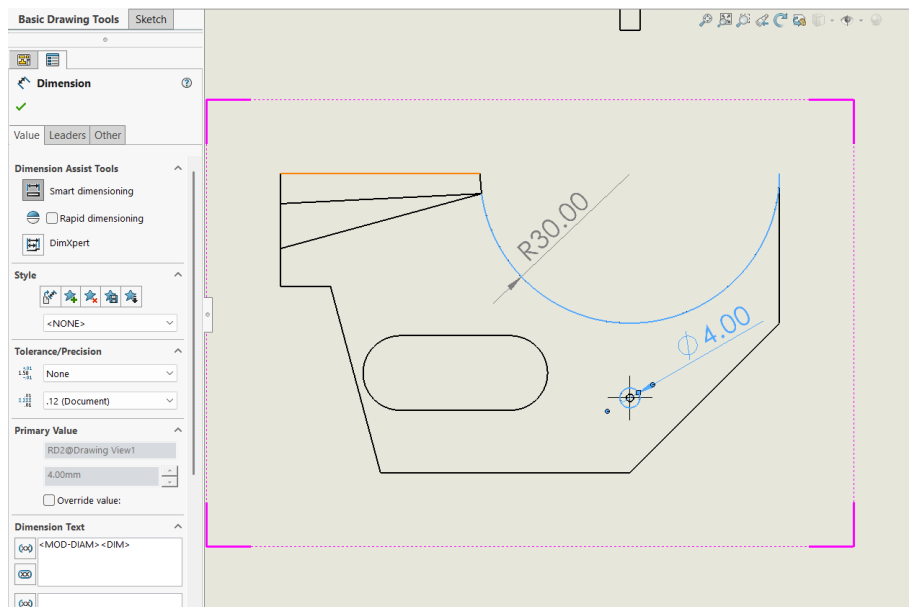
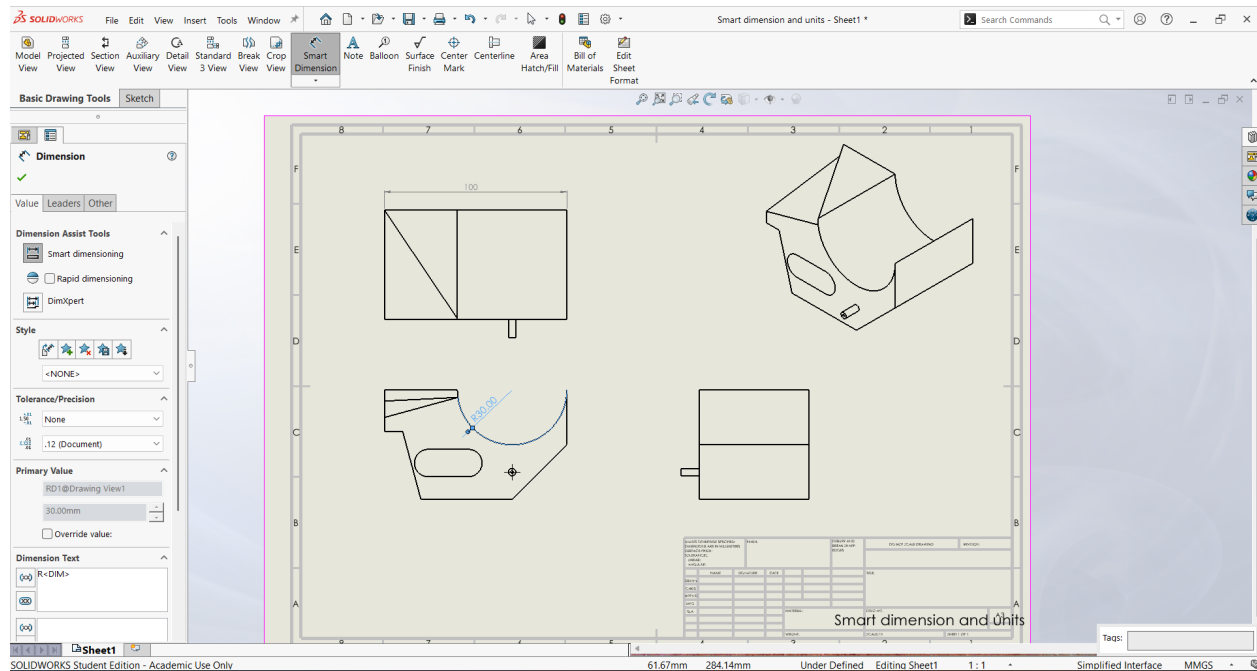


A new dialog box opens. Select the **Document Properties** tab, then select **Dimensions**. Here, you can change the font of the dimension text and adjust the arrow size for the entire drawing sheet.



We can also add a radius dimension using the **Smart Dimension** tool.

Select **Smart Dimension**, then click on the arc.



Want to go beyond individual commands and learn how to actually use **SOLIDWORKS** from beginner to advanced level?

Check out the **SOLIDWORKS Launchpad Course**.

Inside, you'll learn the complete SOLIDWORKS workflow step by step:

- ✓ **Sketching** – Build accurate and fully defined sketches
- ✓ **Part Design** – Create simple to advanced 3D models
- ✓ **Assembly Design** – Combine parts, apply mates, and build complete assemblies
- ✓ **Engineering Drawings** – Turn your 3D models into professional technical drawings

Whether you're completely new to SOLIDWORKS or want to take your existing skills to the next level, the course gives you a **structured path from the fundamentals to advanced modeling**.

Ready to master **SOLIDWORKS**?

[**Start the SOLIDWORKS Launchpad Course**](#)