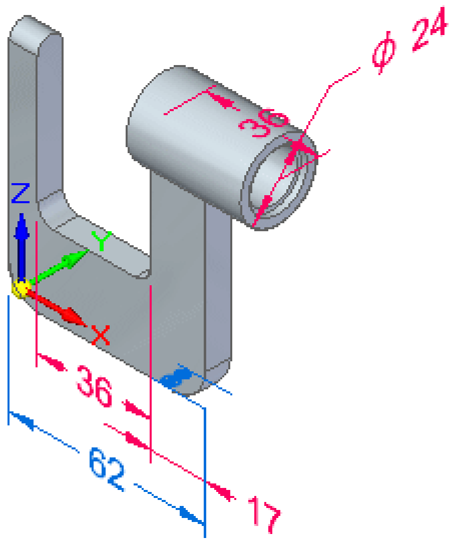
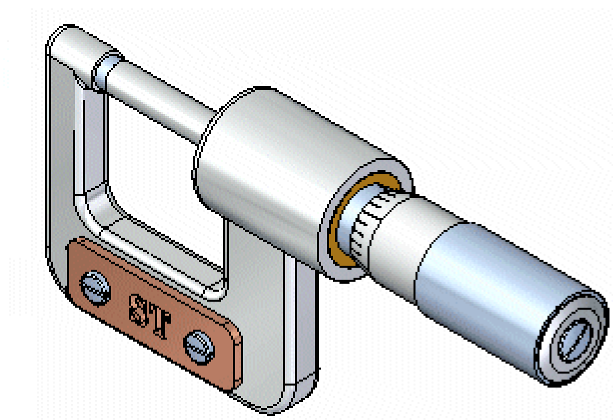
Solid Edge Testdrive

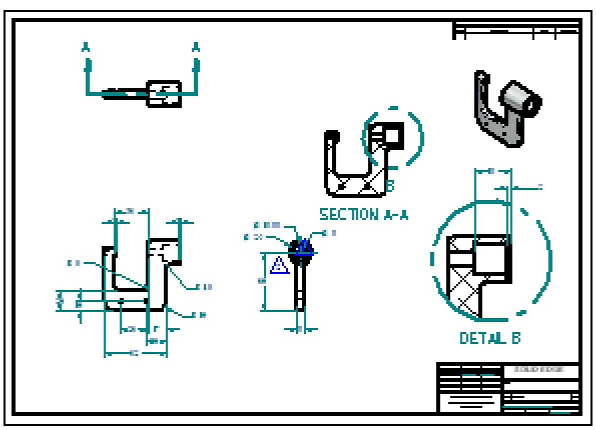
Proprietary and restricted rights notice

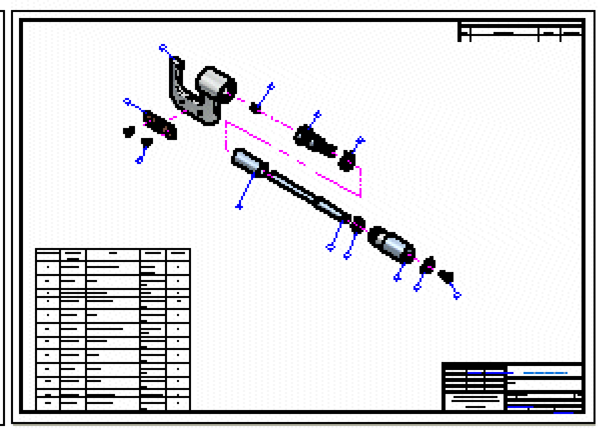
Welcome!

This test drive demonstrates a typical workflow for modeling parts, creating assemblies, creating detailed drawings, and performing finite-element analysis simulations with Solid Edge. Get behind the wheel and discover how easy Solid Edge is to use!





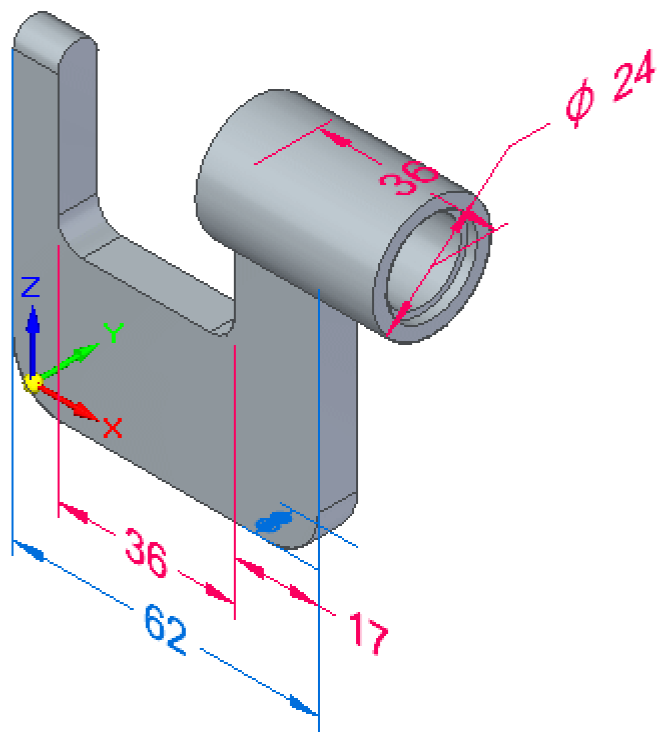




This test drive does not demonstrate everything Solid Edge can do. Its purpose is to show how powerful and intuitive Solid Edge is, and to get started in learning more.

Expect to spend about two hours working through this test drive.

Chapter 1: Introduction to part modeling

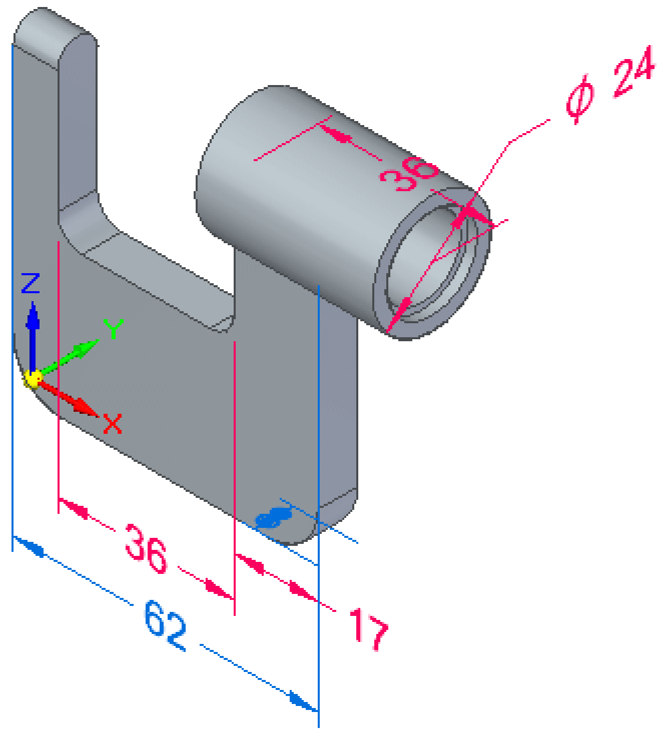


In this test drive, construct the model shown in the illustration above while learning various synchronous modeling techniques, such as:

- Drawing sketches.
- Constructing features.
- Dimensioning model edges.
- Working with PathFinder.
- Editing features.

You will also create a preliminary detail drawing of the 3D model .

Solid Edge Part Modeling Workflow Overview



You model parts in Solid Edge using the following basic workflow:

- Draw a sketch for the first feature.
- Add dimensions to the sketch.
- Extrude or revolve the sketch into a solid feature.
- Add more features.
- Edit the model dimensions and solid geometry to complete the part.
- Create a drawing.

Solid Edge is made up of several components called environments. These environments are tailored for creating individual parts, sheet metal parts, assemblies, and detailed drawings.

In the Solid Edge: Part environment, construct a base feature and then modify that base feature with additional features such as protrusions, cutouts, and holes to construct a finished solid model.

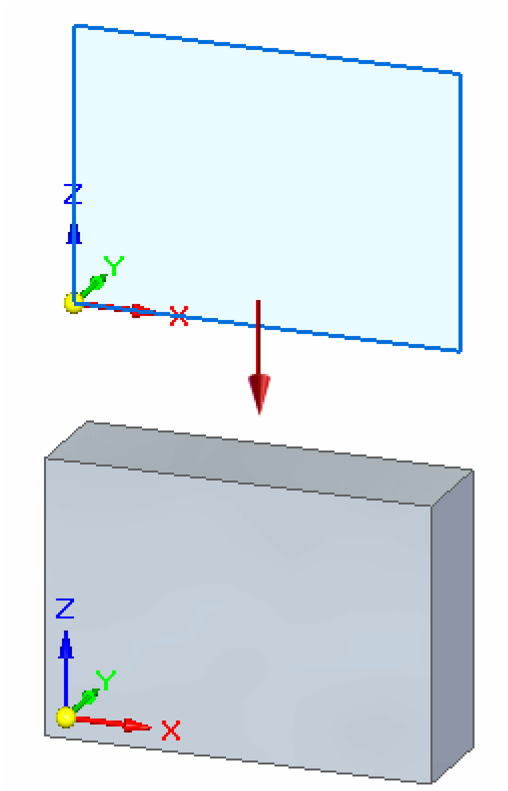
Create a part file

□ On the **Application** menu, click **New→ISO Metric Part**.

The templates listed on the **New** page are the industry standard which is set during the Solid Edge installation. This test drive uses the **ISO Metric** standard. To change the industry standard, click the **Edit List** option on the **New** page. Select the **ISO Metric** standard from the **Standard Templates** list and then click **OK**.

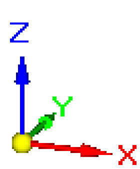
Modeling the base feature

Model the base feature



In the next few steps, sketch a rectangle and then construct the base feature of the model as shown above. Draw the sketch on the XZ principal plane, indicated by the base coordinate system.

Observe the base coordinate system



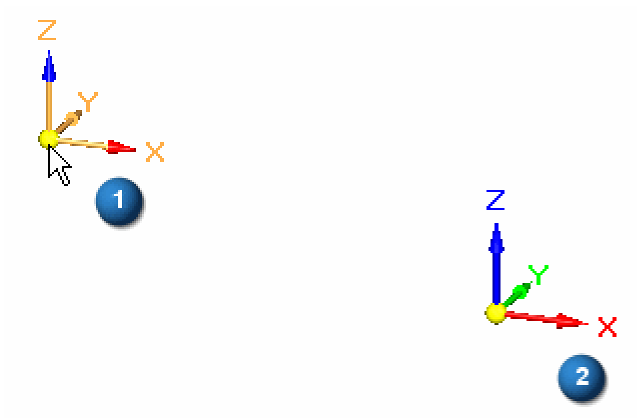
The first step in drawing any new part is drawing the sketch for the base feature. The first sketch defines the basic part shape.

First draw a sketch on one of the principal planes on the base coordinate system, and then extrude the sketch into a solid.

What is the base coordinate system for?

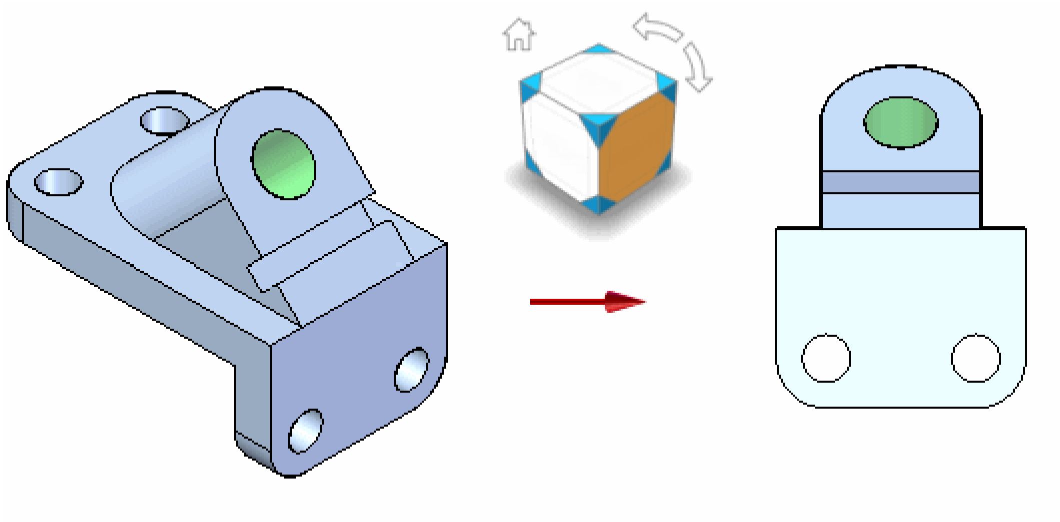
The base coordinate system is located at the origin of the model file, as shown above. It defines the principal X, Y, and Z directions. It also defines the XZ, XY, and YZ planes. These can be used in drawing any sketch-based feature.

> Depending on the computer configuration, there may also be a view orientation triad displayed in the graphics window. If so, the base coordinate system (1) is the selectable element shown highlighted in the illustration below. The view orientation triad (2), which cannot be selected, is for view orientation purposes only. For the remainder of this tutorial, the view orientation triad will not be shown.

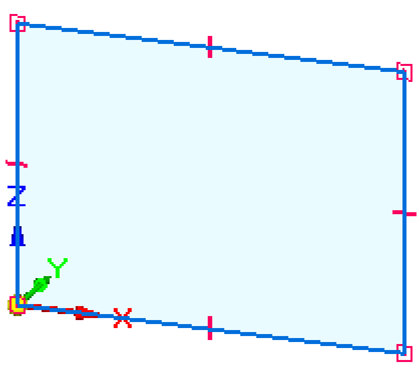


Quick View Cube

A cube (called **Quick View Cube**) may display in the same location as the view orientation triad (2). Use the **Quick View Cube** tool to rotate a view to any principal or isometric orientation of the model geometry. For the remainder of this tutorial, the **Quick View Cube** will not be shown.

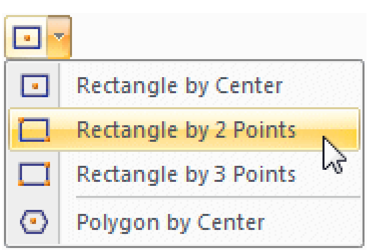


Start the Rectangle command



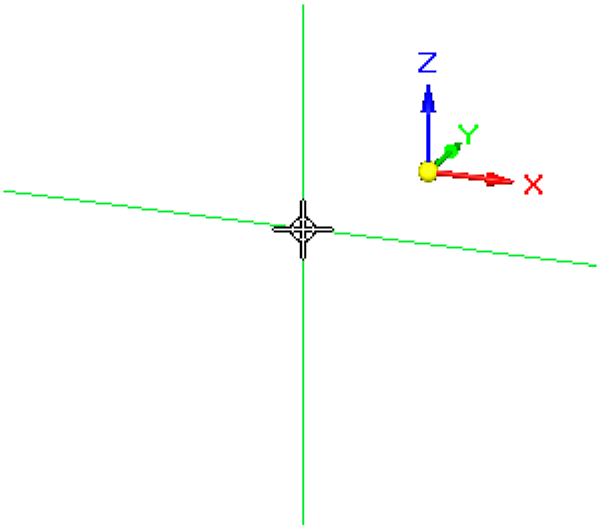
Draw a rectangle on the XZ principal plane, indicated by the base coordinate system.

- ☐ On the command menu, at the top of the Solid Edge application, choose **Home** tab→**Draw** group→**Rectangle by 2 Points** .



The **Rectangle** command bar displays, and the behavior of the cursor changes to display alignment lines.

Observe the alignment lines attached to the cursor

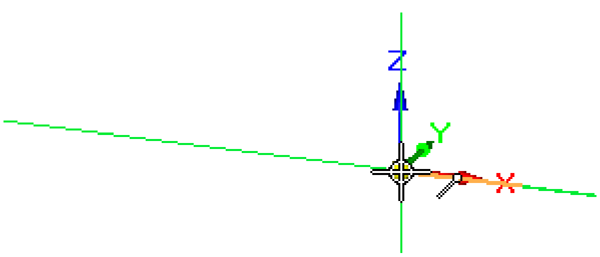


- ☐ Move the cursor around the graphics window and notice that alignment lines extend outward from the cursor.

The alignment lines are oriented to the XZ principal plane on the base coordinate system. The XZ principal plane corresponds to the front view.

The alignment lines indicate the orientation and position in 3D space in which sketch elements will be drawn.

Observe the keypoint indicator



- ☐ Position the cursor over the origin point of the base coordinate system, as shown above, but do not click.

Notice that a keypoint indicator symbol displays adjacent to the cursor.

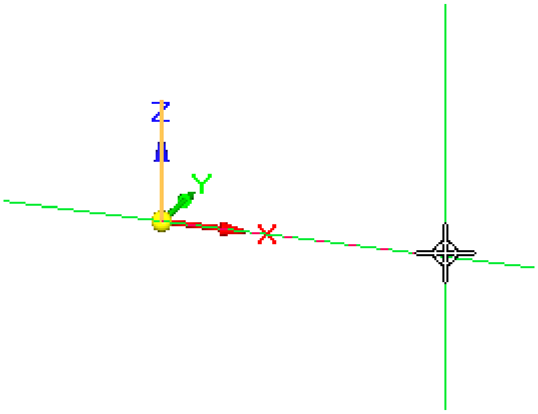
This is **IntelliSketch** in action, which makes it easy to draw precisely, relative to other geometry.

This keypoint indicator is the end point indicator .

It displays when the cursor is over the end point of a line, a model edge, or in this case, the origin of the base coordinate system.

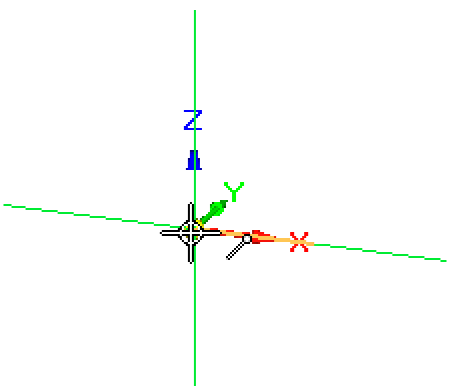
- ☐ Move the cursor to the right slowly and then up and down slowly.

As shown below, notice that the horizontal alignment line extending from the cursor snaps into position and displays dashed when the cursor is horizontally aligned to the origin point of the base coordinate system.



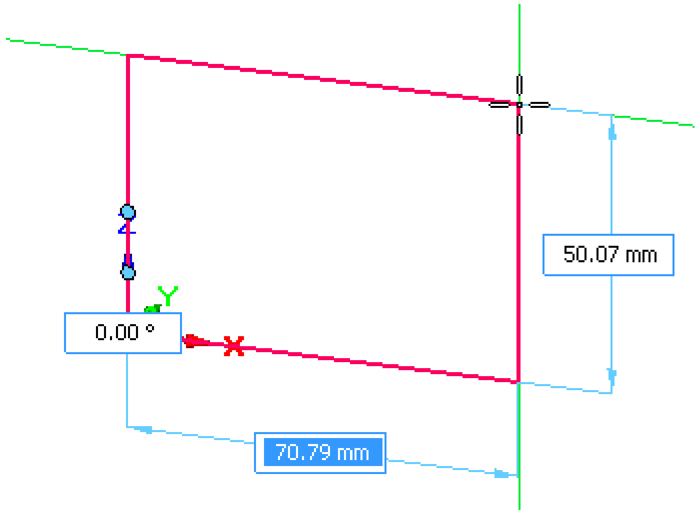
IntelliSketch provides many different types of snap points, such as end point, midpoint, silhouette, and center point. **IntelliSketch** makes it easy to align new geometry with existing geometry.

Draw a rectangle



- ☐ Position the cursor at the origin of the base coordinate system.
- ☐ When the endpoint symbol appears, click to define the start point of the rectangle.

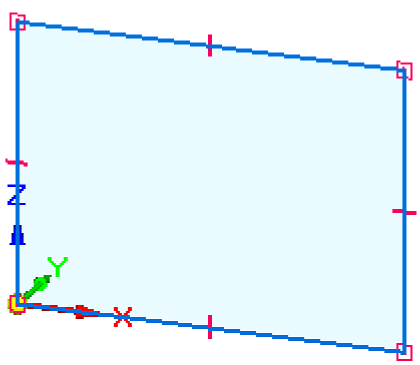
- Move the cursor to the right and up. Notice that the Width and Angle edit boxes update to reflect the current cursor position.



- Position the cursor so that the Width value is near 70 mm and the Height is near 50, then click to define the second point of the rectangle.

- Press Escape to end the **Rectangle** command.

Observe the rectangle

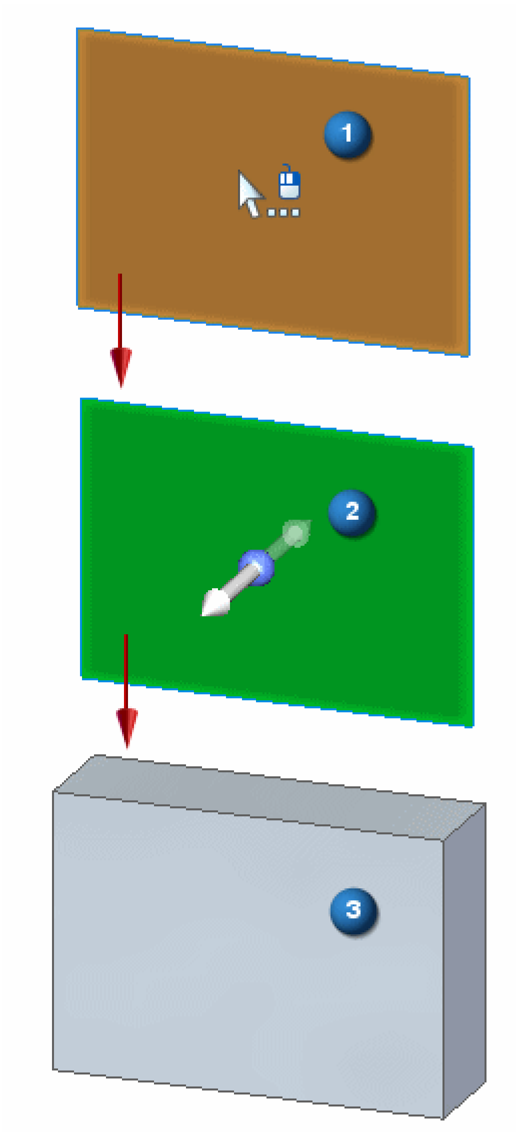


Observe that the rectangle displays as a shaded element.

The rectangle displays as a shaded element because the lines that form the rectangle define a closed region.

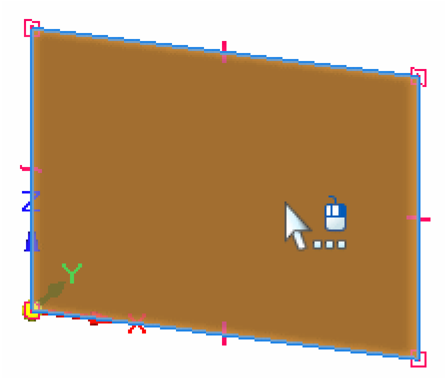
In Solid Edge synchronous modeling, when 2D elements form a closed area, they are called sketch regions.

Prepare to construct the base feature



In Solid Edge synchronous modeling, solid geometry is constructed using the Select tool (1) and grab and go handles, such as the **Extrude** handle (2), to quickly transform 2D sketch geometry into a 3D solid (3).

Select the sketch region using QuickPick



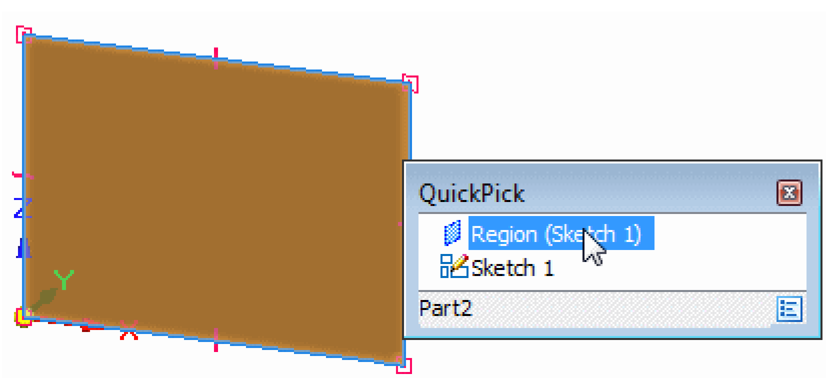
- Press Escape to start the **Select** command or choose the **Home** tab→**Select** group→**Select** button . This command is available on most of the ribbons. But at any time, pressing Escape starts the **Select** command, and this is generally considered the quickest and easiest way to do it.

- Position the cursor over the sketch region as shown, stop moving the mouse, and notice that the cursor image changes to indicate that multiple selections are available.

- Right-click, and the **QuickPick** list displays, as shown below.

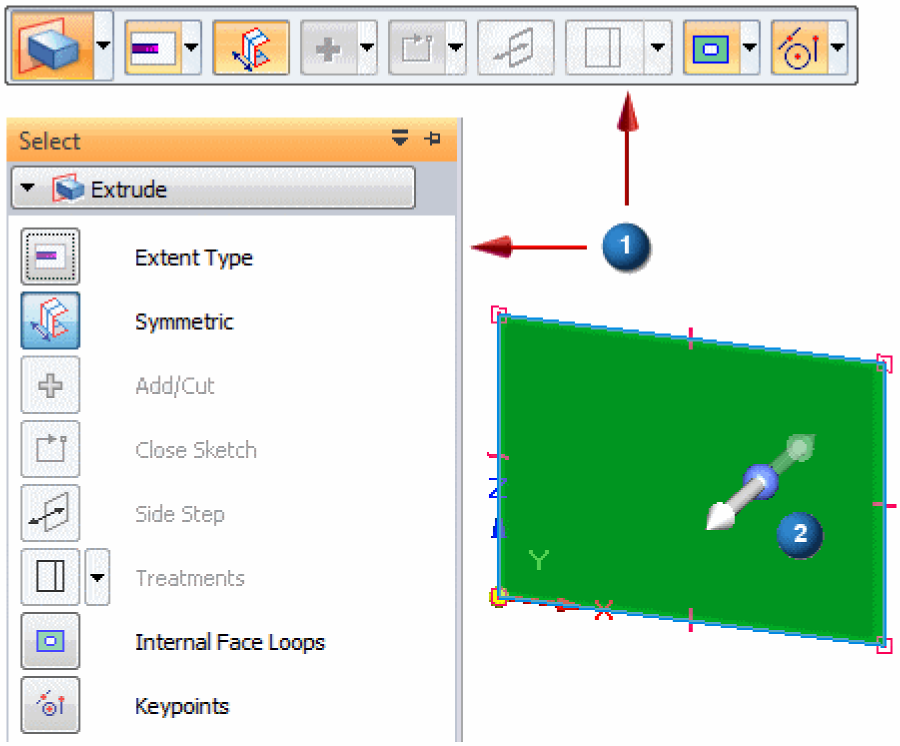
  Move the cursor over the different entries in **QuickPick**, and notice the different elements highlight in the graphics window. Use **QuickPick** to select from a list elements that are near the cursor.

- Position the cursor over the entry in **QuickPick** that highlights the region as shown below, then click to select it.



Depending on the current computer settings, the sketch region may highlight as a shaded element or only the edges may highlight.

Observe the on-screen tools



Notice the following, as shown in the illustration:

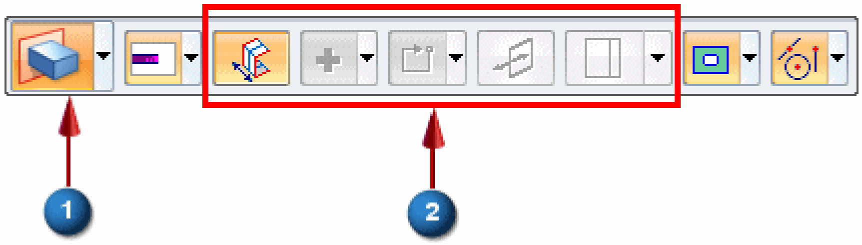
- A command bar (1) displays in the graphics window, either vertically, as shown on the left, or horizontally, as shown above. The appearance of the command bar depends on the user interface theme selected in the Solid Edge **Options**→**Helpers** page. The test drive instructions use the horizontal representation of the command bar, because it is smaller and easier to display; but the two versions of the command bar always contain the same information and the same options.

- An extrude handle (2) displays on the sketch, near where the sketch was selected.

The command bar provides a list of possible actions and the available options for the current action.

Use the extrude handle to construct the feature. Before you construct the feature, you will learn more about the command bar.

Command bar overview



The command bar displays when selecting certain types of elements. Based on the elements selected, the command bar presents a targeted set of actions and options. The horizontal version of the command bar is shown here, though a vertical version may display, depending on the user interface theme in use.

Actions:

The Actions list displays on the left side of the horizontal command bar (1), or the top of the vertical command bar.

For a sketch region, the default action is to construct an extruded feature. Select different actions from the Actions list. For example, a revolved feature can be specified instead of an extruded feature.

Options:

The options available for the current action are displayed on the remainder of the command bar (2). For an extruded feature, specify options such as, whether material is added or removed, the feature extent, whether the feature is constructed symmetrically about the sketch region.

These options will be explored in the testdrive.

Set the command bar options

□ Pass the cursor slowly over the various options on the command bar.

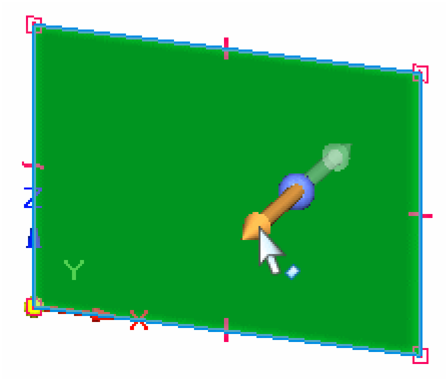
Notice that tool tips display that provide additional information about the options on the command bar.

□ On the command bar, set the options shown above. Notice that the set options display with an orange background.

(1) Extent type=Finite.

(2) Extent direction=Symmetric.

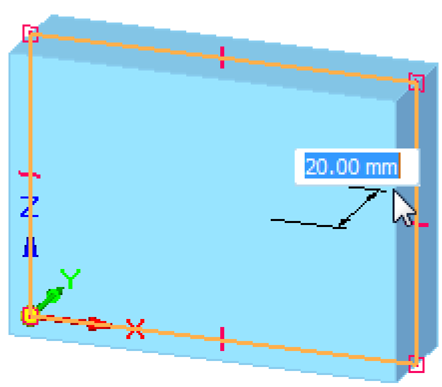
Select the Extrude handle and define the base feature extent



- ☐ Position the cursor over the extrude handle as shown above, and when it highlights, click.
- ☐ Move the cursor slowly and notice that the feature is drawn symmetrically on both sides of the sketch as the cursor moves.

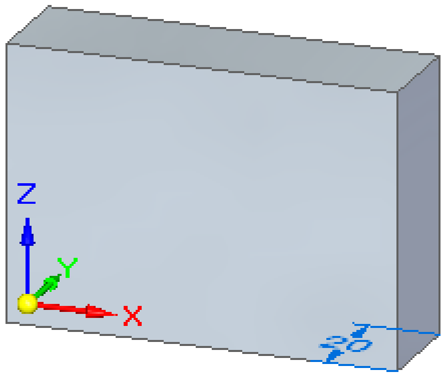
Also notice that a dynamic input box displays in the graphics window.

- ☐ Position the cursor such that the feature extent is approximately 20 millimeters, type **20** in the dynamic input box, then press Enter to define the extent for the feature, as shown below.



Base feature construction is complete.

Observe the results

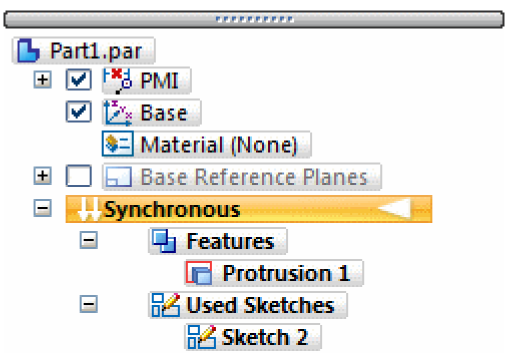


The graphics window should resemble the illustration. Notice that a solid base feature displays without the sketch.

When constructing a feature from a sketch in Solid Edge, the sketch elements used to create the feature are moved to the **Used Sketches** collector in **PathFinder**.

Learn more about **PathFinder** in the next step.

Explore PathFinder



Explore **PathFinder** which is located on the left side of the application window.

PathFinder helps to evaluate, select, and edit the components that comprise the models created in Solid Edge.

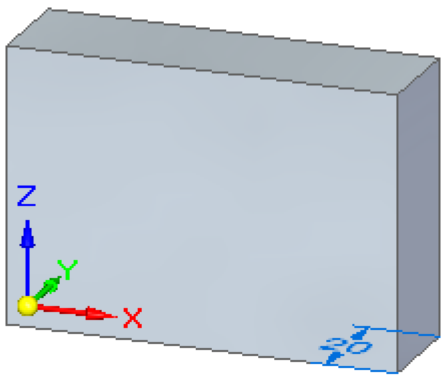
□ Click the ⊞ symbols in **PathFinder** to expand the various headings until the display matches the illustration.

Notice the following in **PathFinder**:

- A **Features** collector that contains a Protrusion entry, which represents the base feature constructed.
- A **Used Sketches** collector that contains a Sketch entry for the sketch used to construct the feature.

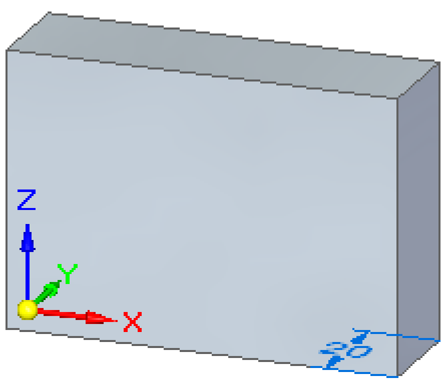
When constructing sketch-based features, only the sketch elements used to create the feature move to the **Used Sketches** collector in **PathFinder**. Use these sketches for subsequent features later. The unused sketch elements remain in the **Sketches** collector.

Save the part



- ☐ Save the file by clicking the **Save** command on the **Quick Access** toolbar.

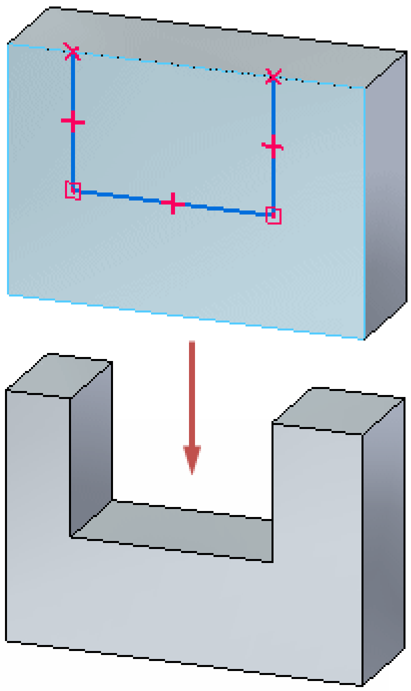
Base feature creation complete



Congratulations. The first step in constructing a part, the base feature, is complete.

Remove material from the base feature

Remove material from the base feature

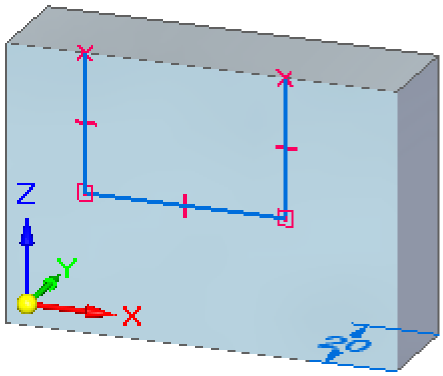


In the next few steps, construct an extruded cutout feature shown in the illustration. Use a workflow similar to the one used to construct the base feature.

Draw a sketch on the front face of the part, then use the **Select** tool to construct the feature.

Also, learn how to control the display of 2D sketch relationships.

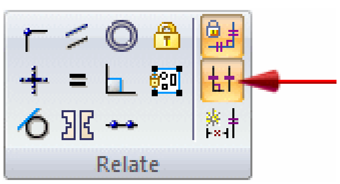
Evaluate the sketch



In the next few steps, draw the sketch for the next feature, as shown in the illustration.

- Learn about the display of the geometric relationships that help define the behavior of the 2D elements of the sketch.
- Lock to the front planar face of the model to draw the sketch on.
- Use the **Line** command to draw the three lines shown.

Display the sketch relationship handles



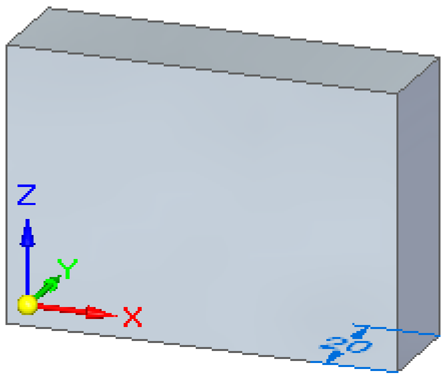
By default, the **Relationship Handles** display is turned on.

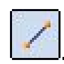
□ You can turn off the display by choosing **Home** tab→**Relate** group→**Relationship Handles**.

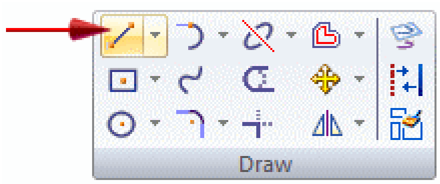
Relationship handles show how sketch elements are geometrically related to each other.

Learn more about this after drawing the sketch.

Start the Line command

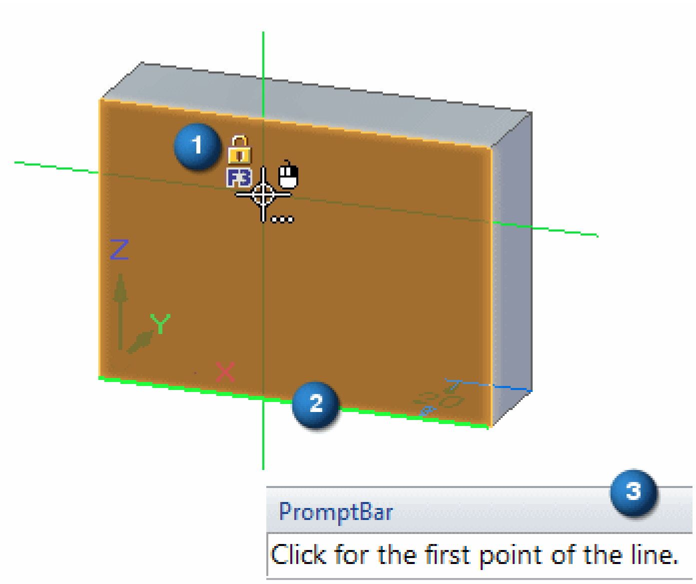


- ☐ Choose **Home** tab→**Draw** group→**Line** command .



Notice that the **Line** command bar (1) displays.

Observe the on-screen tools



□ Position the cursor over the planar face of the model and pause the cursor.

Notice the following on-screen tools:

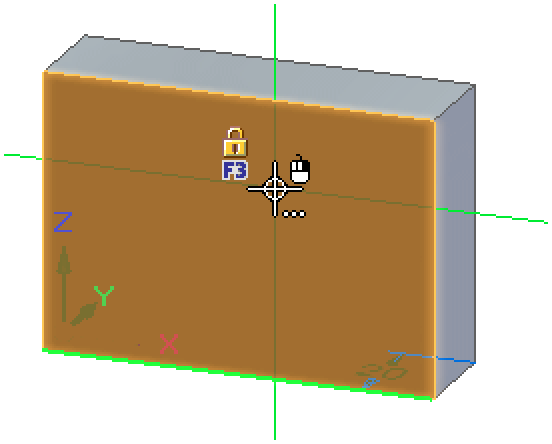
- (1) A lock symbol  displays to the left of the cursor.
- (2) One of the model edges on the face highlights in green.
- (3) The PromptBar, which provides instructional messages while working, displays along the bottom of the application window.

Clicking the lock symbol (or pressing **F3**) locks sketch input to the selected face on the model.

The green edge indicates the X direction on the sketch plane. The green edge on the model may be different than the illustration. Use shortcut keys (n=next, b=back, and p=base plane) to define the sketch plane orientation.

The **PromptBar** gives useful information about the current command and options.

Set the sketch plane orientation and lock sketch input to a model face



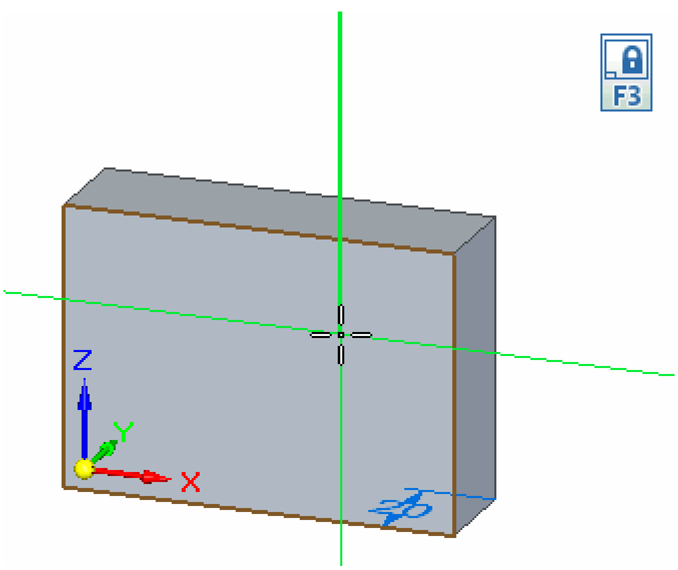
- ◻ Position the cursor over the planar face shown in the illustration.
- ◻ If necessary, press **N** until the bottom edge of the model displays as shown above.

  > If this presents trouble, click the title-bar of the Solid Edge window to ensure it has focus, and ensure that the cursor is inside the Solid Edge window.

- ◻ Press **F3** to lock sketch input to the model face.

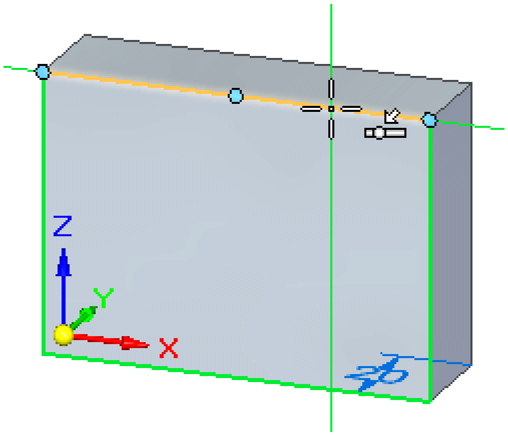
  > If this presents trouble, click the title-bar of the Solid Edge window to ensure it has focus, and ensure that the cursor is inside the Solid Edge window.

Notice that a locked plane indicator displays in the top-right corner of the graphics window, as shown below.



Also notice that when the cursor moves over the other model faces, they no longer highlight. All sketch input is now locked to the selected model face.

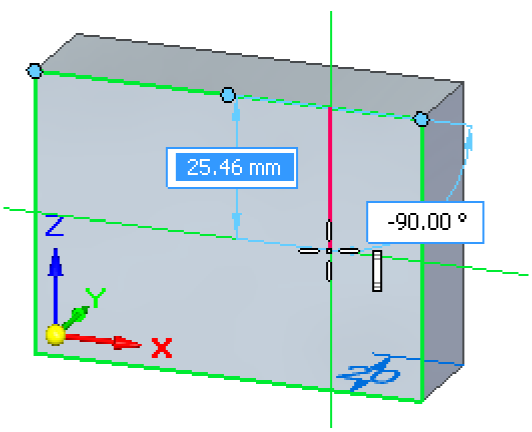
Start the first line



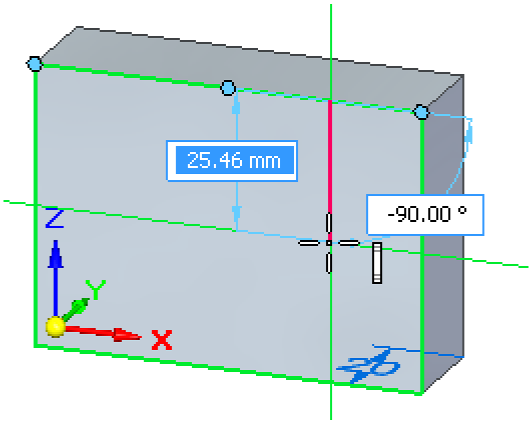
- ☐ Position the cursor as shown in the illustration, and when the point-on relationship indicator displays adjacent to the cursor, click to start the line.

Notice that when locating an element, the end points and midpoint highlight in blue. This makes selecting keypoints easy. When pausing over a located keypoint, it turns orange.

- ☐ Move the cursor downward. Notice the following:
  - A line stretches to follow the cursor wherever it moves. The line also has edit boxes attached to define length and direction.
  - When the line is vertical, a vertical relationship indicator displays adjacent to the cursor.

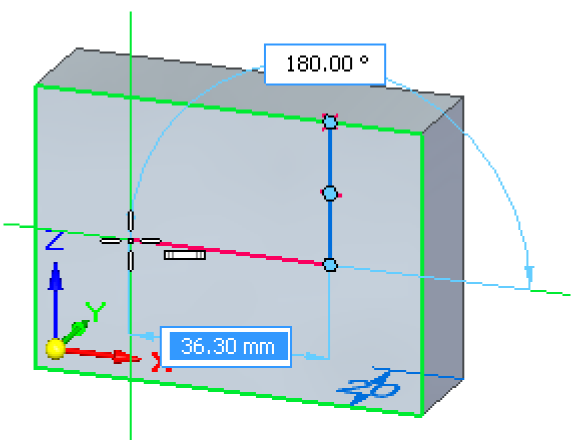


Finish the first line



- ☐ Move the cursor until:
  - The vertical relationship indicator displays at the cursor.
  - The Length in the edit box is approximately 25 mm.
  - The Angle in the edit box is -90 degrees.
- ☐ When the line is vertical, and approximately 25 mm long, click to finish the first line.

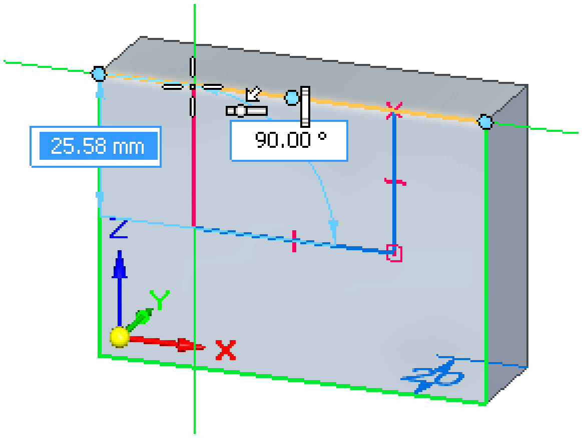
Draw the second line



The **Line** command is still active, ready to draw a line connected to the end point of the previous line.

- ☐ Position the cursor as shown, and when the line is approximately 36 mm and the horizontal relationship indicator displays, click to place the second line.

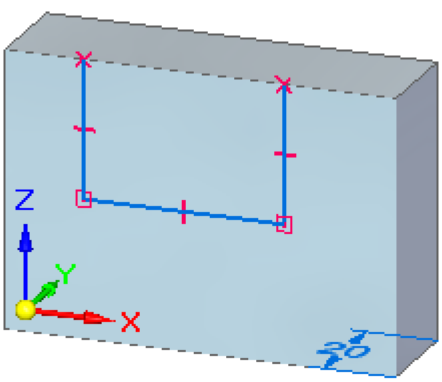
Draw the third line



The **Line** command is still active, ready to draw a line connected to the end point of the previous line.

- □ Position the cursor as shown, and when the point on element and vertical relationship indicators display, click to place the third line.
- □ Right-click to restart the **Line** command.

Observe the results



Observe the finished sketch.

Notice that the sketch elements form a sketch region. Since a region is formed, the sketch is valid for constructing a feature using the **Select** tool. Although this sketch is not closed, it is treated as a region because the model edge at the top of the sketch closes the gap between the three lines on the sketch.

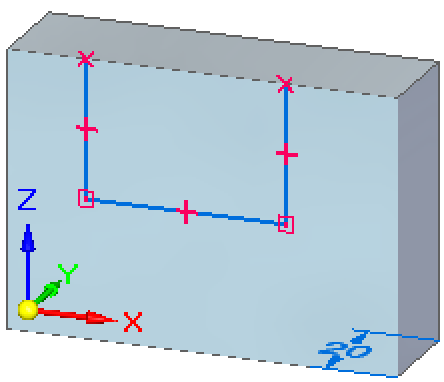
Also notice the relationship symbols where the lines connect to the solid model, at the endpoint of each line, and at the midpoint of each line.

These relationships specify the following:

- The lines remain connected to the model edge.
- The lines remain connected to each other.
- The lines remain vertical or horizontal.

Although the sketch and the relationships are discarded when constructing the next solid feature, building these relationships into the sketch is helpful. When constructing the solid feature, these 2D relationships orient the faces that are constructed from the sketch, and help define the behavior desired when editing the model later.

Unlock the sketch plane

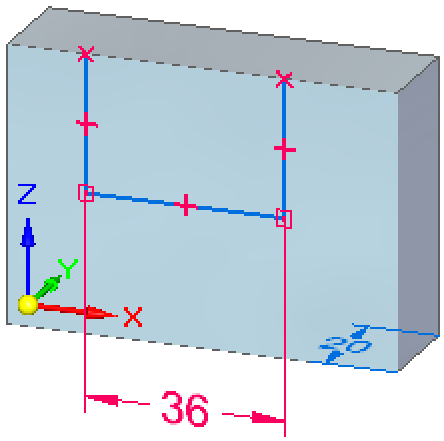


The sketch is complete. Unlock the sketch plane.

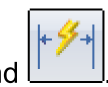
□ Press **F3** to unlock the sketch plane.

Notice that the locked plane symbol no longer displays in the graphics window.

Start the Smart Dimension command

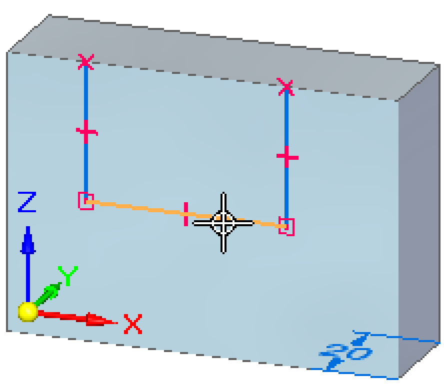


In the next few steps, place a dimension on the sketch using the **Smart Dimension** command.

- Choose **Home** tab→**Dimension** group→**Smart Dimension** command .

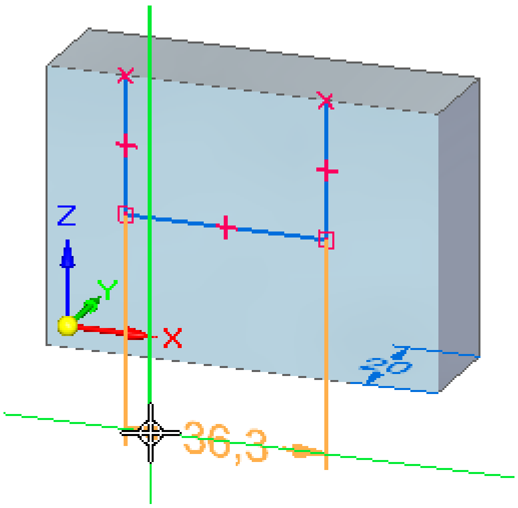
Use this command to place a dimension on one element or between two elements.

Place a dimension on the sketch



- Position the cursor over the sketch element, as shown. When it highlights, click to select it. Notice that dimension elements attach to the cursor.

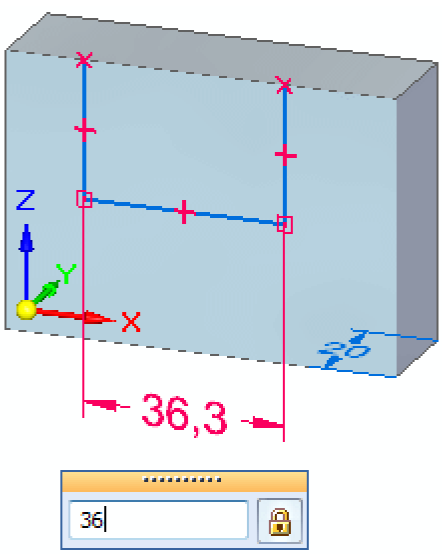
- Position the cursor below the model, and click to place the dimension.



A dynamic input box displays near the cursor so that the dimension value can be changed.

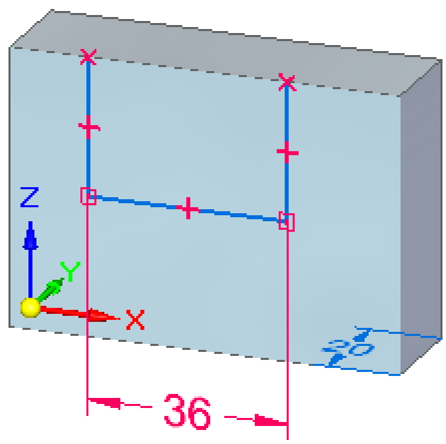
The dimension value on the model may be different than the illustration.

Edit the sketch dimension value



- In the dynamic input box, type **36**, then press Enter.

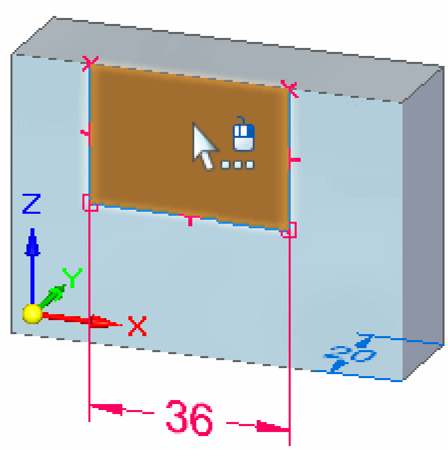
Observe the results



Learn more about dimensions later in this test drive. For now, observe that the dimension color is red, and that the dimension could be edited immediately upon placement.

When placing dimensions on sketch elements, they are placed as locked dimensions, which are red. Locked dimensions maintain their value if other parts of the model change.

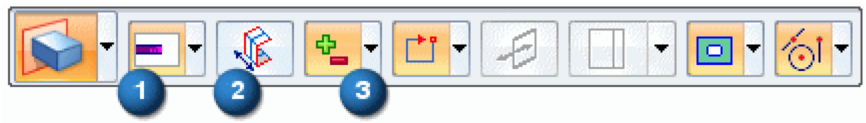
Start the Select command and select the sketch region



- ☐ Press Escape to start the **Select** command.
- ☐ Position the cursor over the sketch region, and when it highlights, click to select it.

The Extrude handle and the command bar display.

Set the command bar options



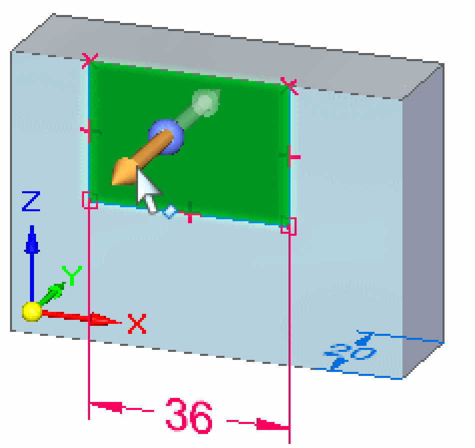
- On the command bar, set the following options:

  (1) Extent Type = Finite.

  (2) Symmetric Extent = Symmetric extent Off.

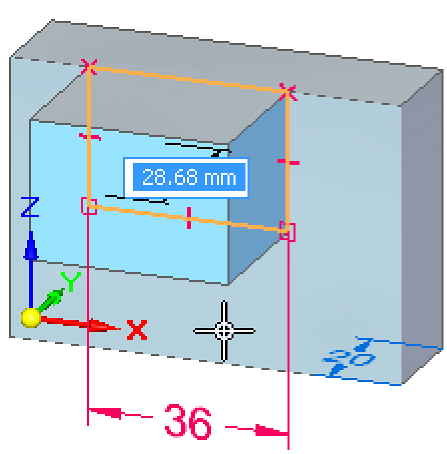
  (3) Add/Cut = Automatic.

Select the Extrude handle and define the feature extent

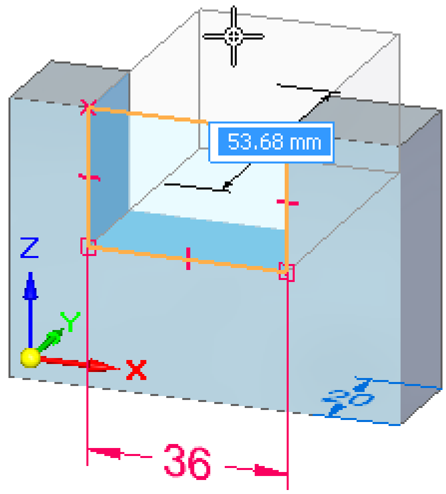


- Position the cursor over the extrude handle and click when it highlights.

- Position the cursor in front of the sketch and notice that material is dynamically added to the part as the cursor moves.

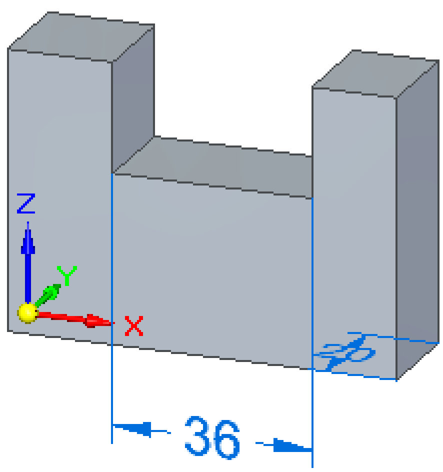


- Position the cursor behind the part and notice that material is dynamically removed from the part.



- Position the cursor so that material is removed all the way through the part, and click to construct the cutout.

Observe the results



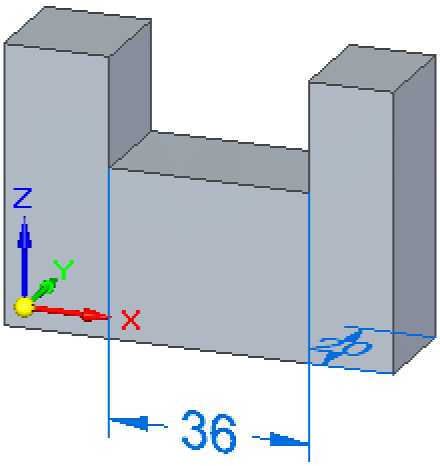
Notice that the dimension color changed to blue when the feature was constructed. When constructing a sketch-based feature, any sketch dimensions are moved to the model and control the model shape, and they change to unlocked dimensions.

Unlocked dimensions are blue and can change value when other parts of the model change.

Locked dimensions are red, and a locked dimension keeps the dimension value from being changed when a connected face moves or changes size.

Learn more about dimensions later in this tutorial.

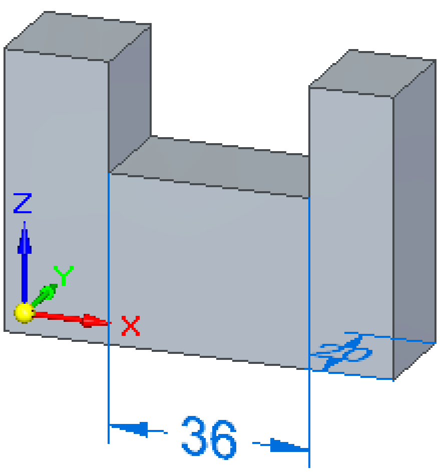
Save the part



- On the **Quick Access** toolbar, click the **Save** button.

> It is good practice to save the model regularly, but it can be easy to forget. Solid Edge has an option to automatically preserve open documents by saving them at user-defined time intervals.

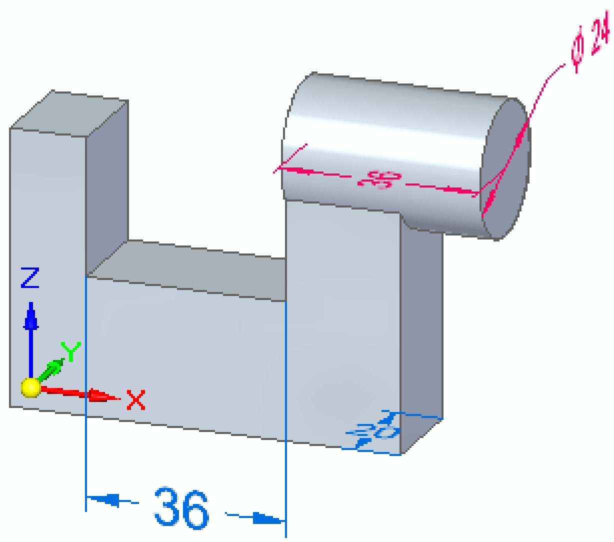
Cut feature creation complete



The second feature of the part is complete.

Create an extruded feature

Create an extruded feature

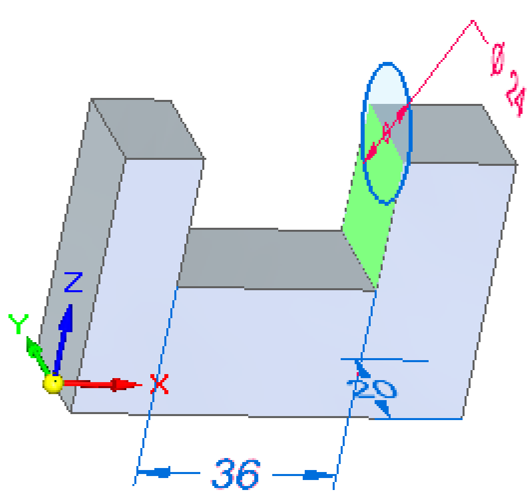


In the next few steps, construct the extruded feature shown in the illustration.

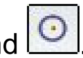
Draw a circular sketch on a face of the model, then use the **Extrude** command to construct the feature.

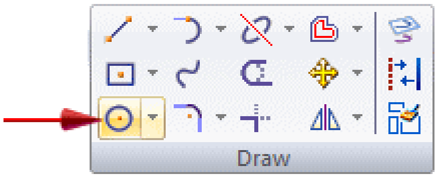
Also learn how to use the **Keypoints** options on the **Extrude** command bar to precisely control the extent of a feature relative to existing geometry.

Start the Circle by Center Point command

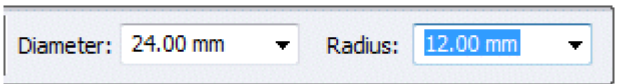


Draw a 24 mm circle on the green model face with the circle center at the midpoint of the top face edge. Place a dimension on the circle.

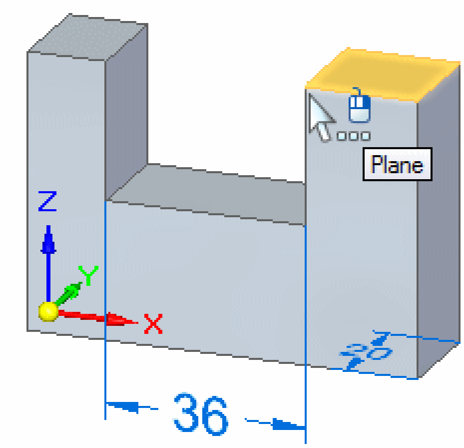
- Choose **Home** tab→**Draw** group→**Circle by Center Point** command .



- On the **Circle by Center Point** command bar, in the Diameter field, type **24**, then press Enter.

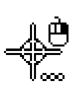


Specify the sketch plane using QuickPick



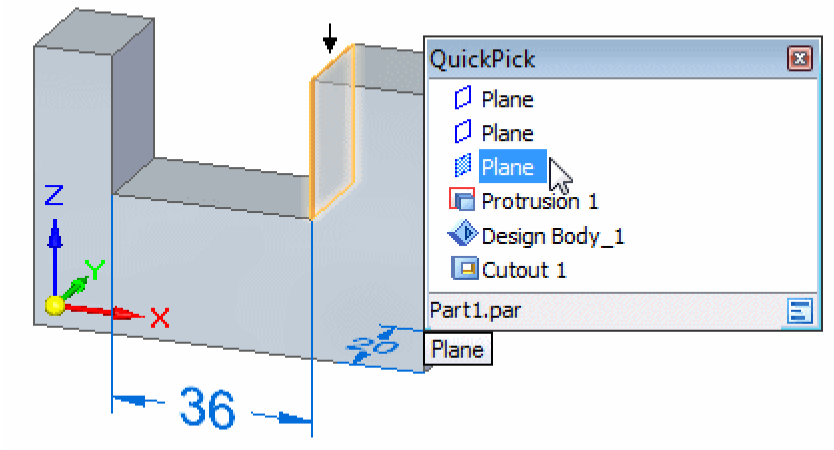
The part could be rotated to make the model face easier to select. However, **QuickPick** will be exposed at this point.

- Position the cursor over the corner of the model as shown above, stop moving the mouse, and notice that the cursor image changes to indicate that multiple selections are available.

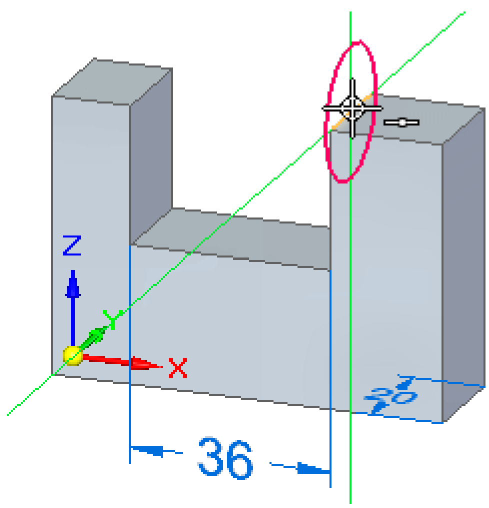


- ☐ Right-click to display **QuickPick**.

- ☐ Position the cursor over the entry in **QuickPick** that highlights the model face shown, then click to select it.



Define the center of the circle

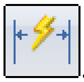


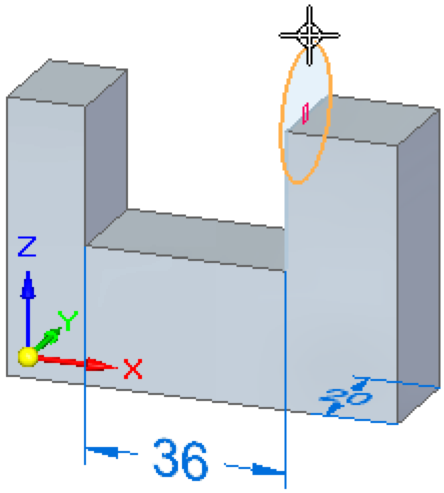
Notice that when defining the sketch plane, the sketch circle is attached to the cursor.

- ☐ Position the cursor over the middle of the top edge, and when the midpoint relationship indicator displays adjacent to the cursor, click to place the circle.

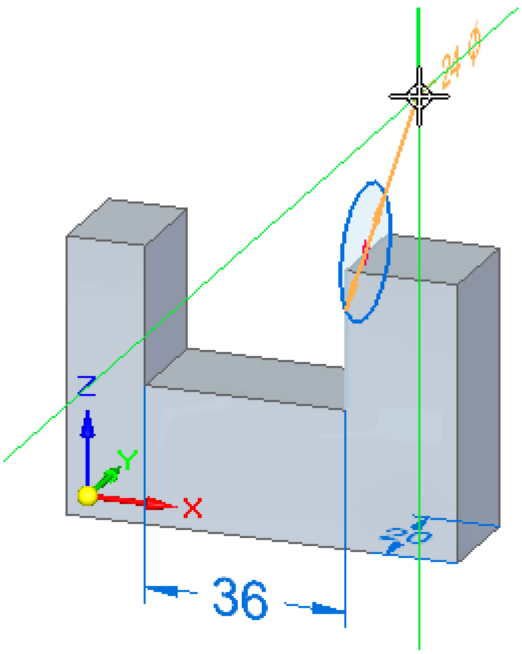
- ☐ Press Escape to end the **Circle** command.

Place a dimension on the circle

- Choose **Home** tab→**Dimension** group→**SmartDimension** command .
- Position the cursor over the sketch circle just placed, then click to select it.

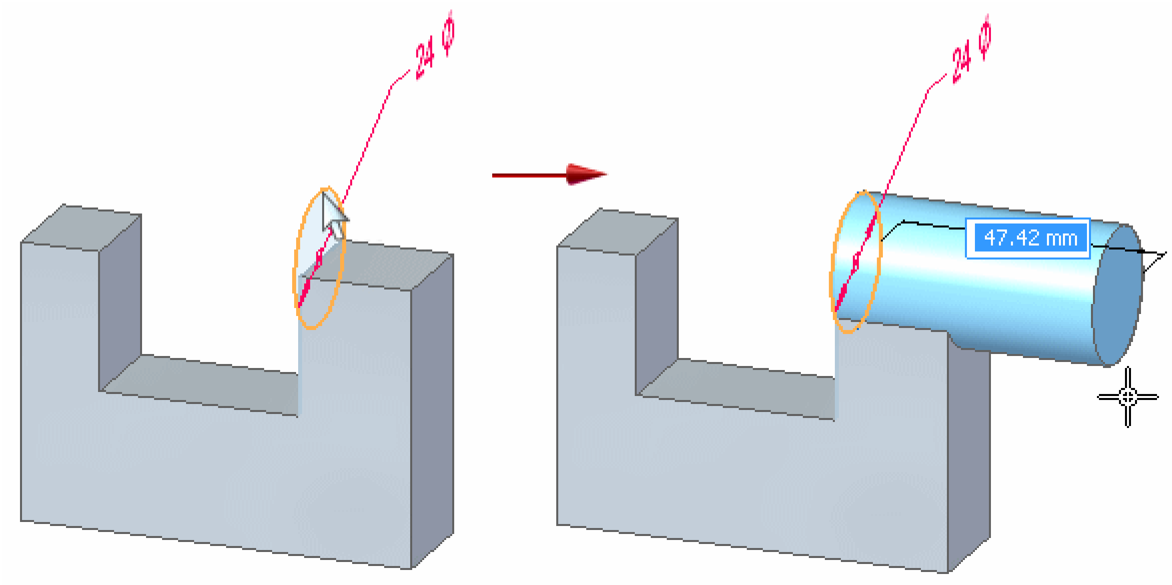


- Position the cursor as shown, then click to place the dimension.



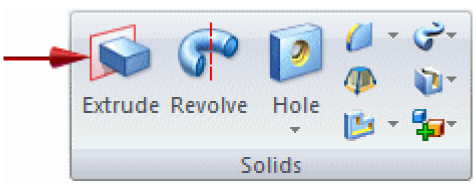
- Because the circle diameter was precisely defined earlier, right-click to accept its current value, and dismiss the dynamic input box.

Start the Extrude command



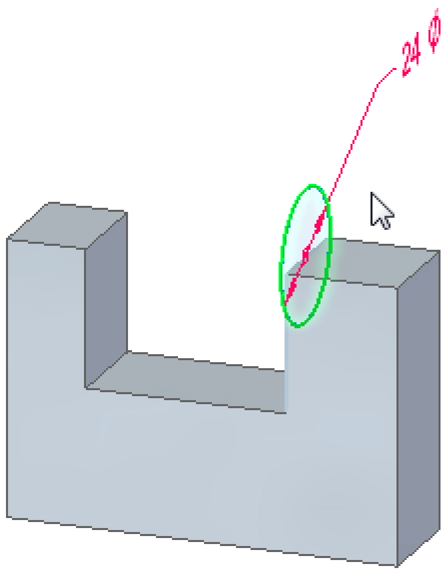
Use the **Extrude** command to construct another feature.

- ☐ Choose the **Home** tab→**Solids** group→**Extrude** command.



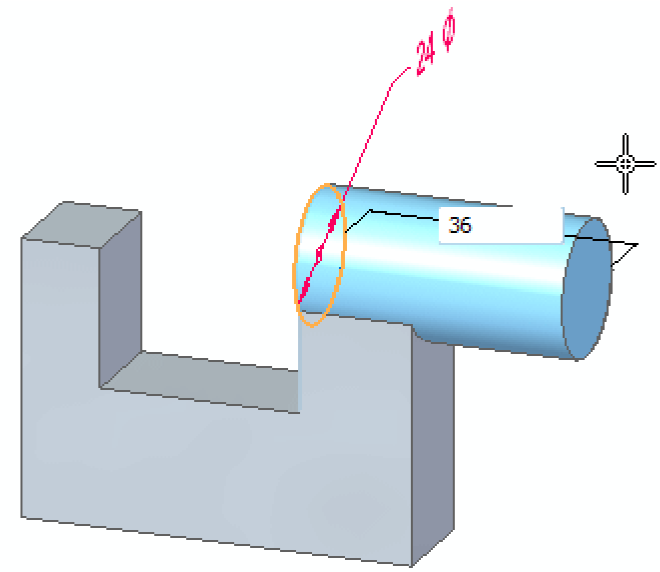
Select the sketch element to extrude

- ◻ On the **Extrude** command bar, make sure the **Single** option is set.
- ◻ Select the sketch circle and then right-click.

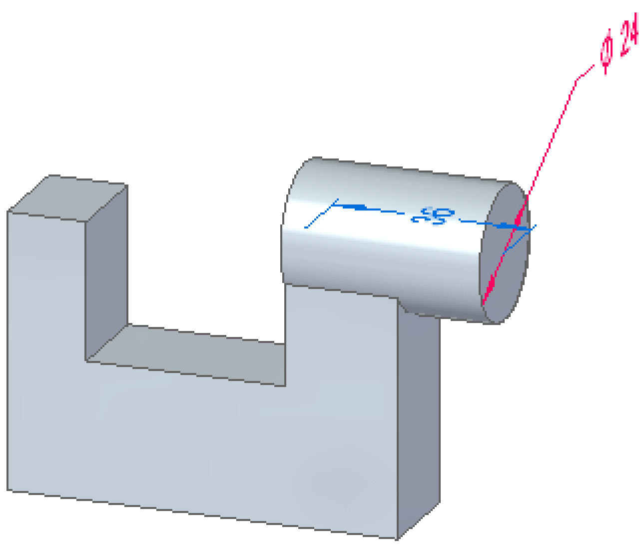


Define the extrusion extent

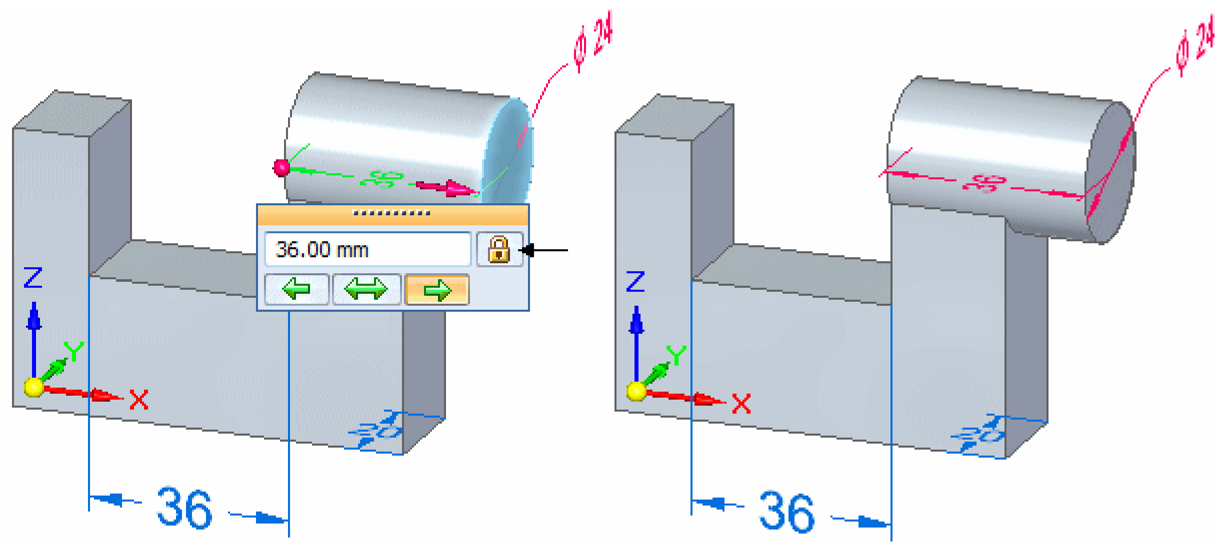
- ☐ The extent distance moves dynamically with the cursor. On the dynamic edit control box, type **3** and then press Enter.



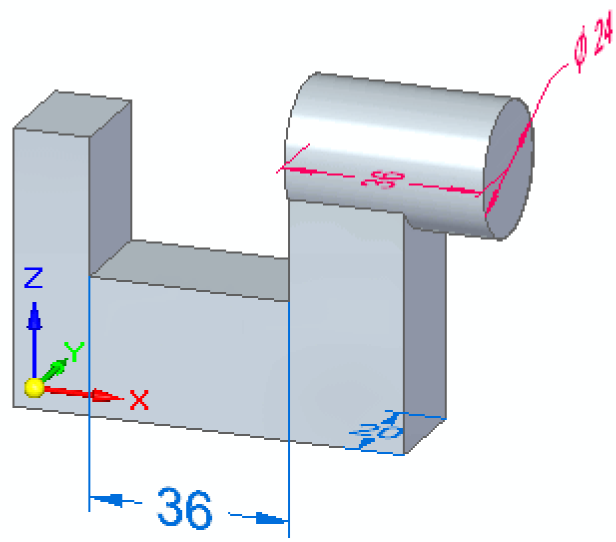
Extrude feature creation complete.



□ The extruded feature is to maintain a length of 36. Click on the dimension text and then click the lock button.



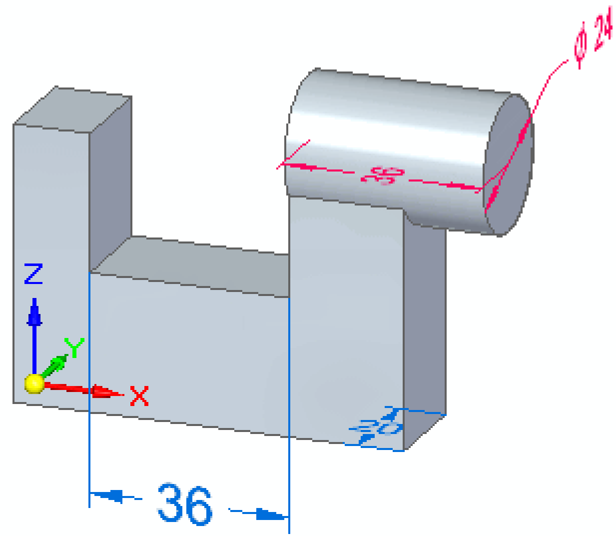
Observe the results



Your extrude feature should resemble the illustration.

Notice that the 24 mm sketch dimension on the circle now controls the extruded feature diameter.

Save the part

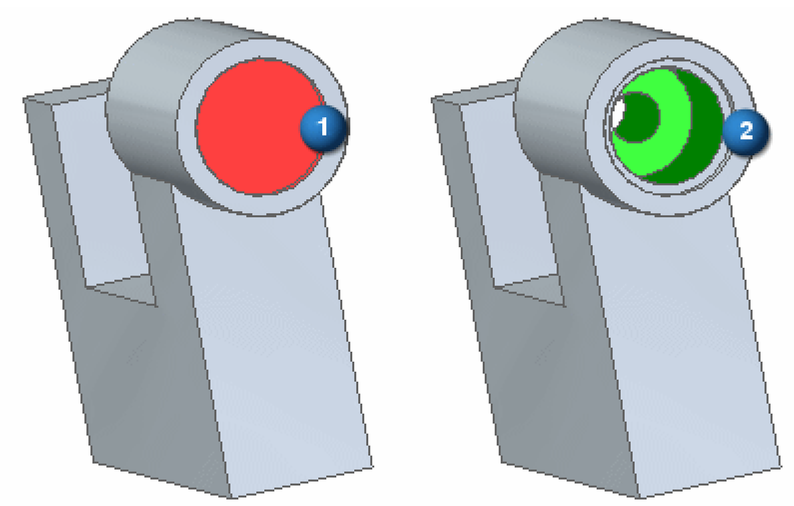


- On the **Quick Access** toolbar, click the **Save** button .

The steps to create the cylindrical protrusion feature are complete.

Adding Hole features

Construct hole features



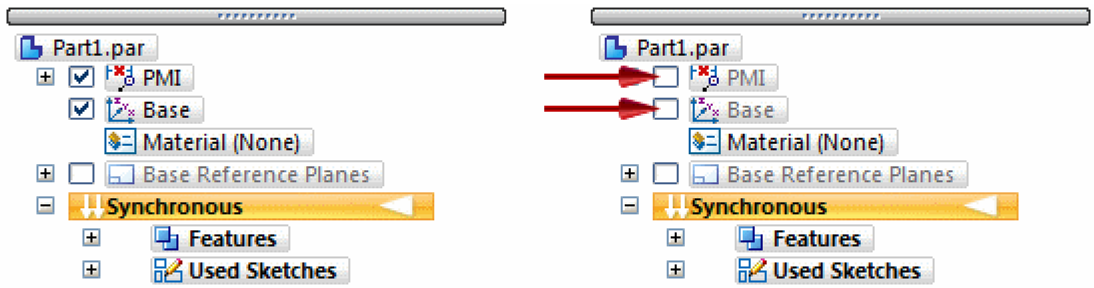
In the next few steps, construct a simple (1) and a counterbore (2) hole feature.

First, hide the base coordinate system and the existing dimensions using **PathFinder**, and then change the view orientation.

Hide the base coordinate system and dimensions using PathFinder

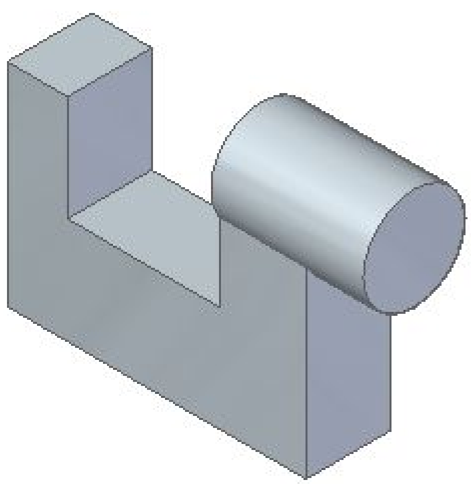
- □ In **PathFinder**, position the cursor over the check box adjacent to Base, then click to hide the Base coordinate system.

- □ Position the cursor over the check box adjacent to PMI, then click to hide the existing dimensions.



Notice that the Base and PMI entries in **PathFinder** change color and that the Base coordinate system and dimensions are hidden in the graphics window.

Rotate the view

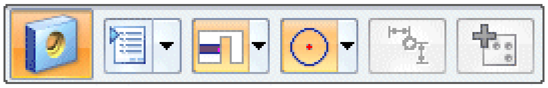


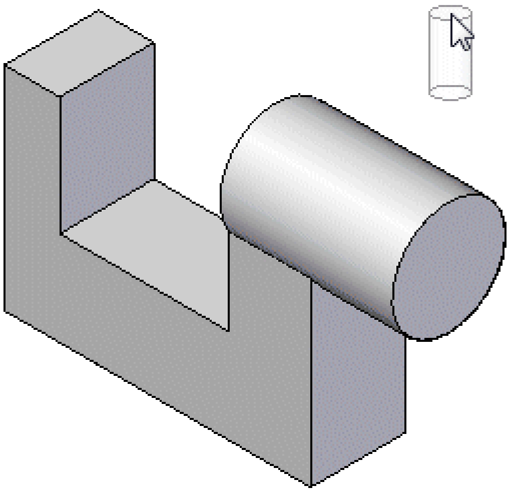
To make it easier to position the hole feature, rotate the view to the isometric orientation.

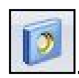
Keyboard shortcuts can be used to rotate the view orientation to align to standard views. For example, holding down the Ctrl key and pressing L, aligns the view to the left side of the part, Ctrl+T aligns the view to the top view, and Ctrl+I aligns the view to the isometric view.

- □ Hold down the Ctrl key , then press **I** to rotate the view to an isometric view.

Start the Hole command



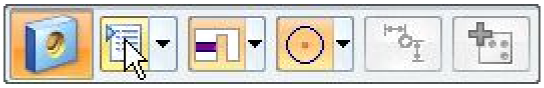


□ Choose the **Home** tab→**Solids** group→**Hole** command .

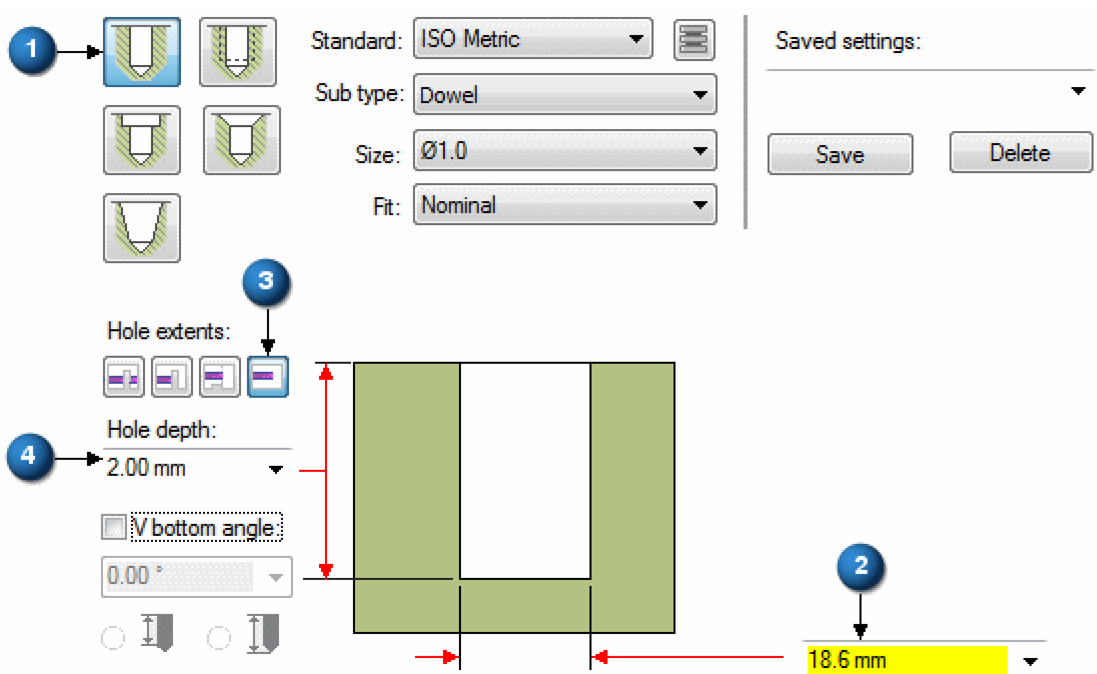
Notice that the **Hole** command bar displays in the graphics window, and that a default hole feature is attached to the cursor.

Define the simple hole parameters

□ On the **Hole** command bar, click the **Hole Options** button. The **Hole Options** dialog box displays.



□ Set the following hole properties for a Simple type hole:



- Set the Type to Simple (1).
- Set the Diameter (2) to **18.6**.

  The diameter field turns yellow to denote that the diameter value for the hole type is not in the hole database.

- Ensure the Extents option is set to Finite (3) and a Hole depth (4) of **2**.

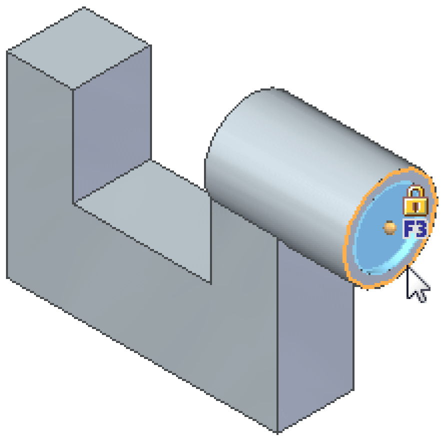
□ On the **Hole Options** dialog box, click **OK**.

Notice that the hole attached to the cursor updates to reflect the hole properties specified.

Position the hole feature

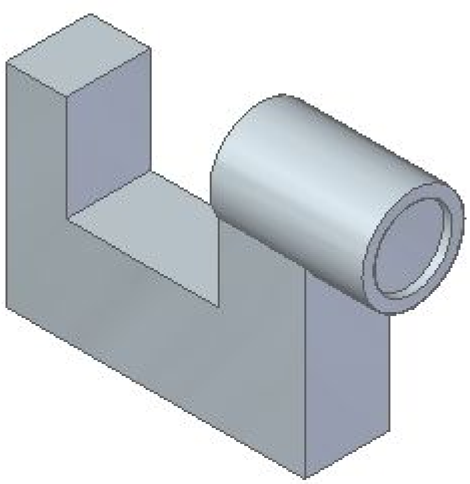
- ☐ Move the cursor over different faces of the model, and notice that a preview image of the results display.

- ☐ Position the cursor over the face shown, but do not click.



- ☐ Move the cursor to the circular edge as shown, and notice that the hole centers itself on the circular face.

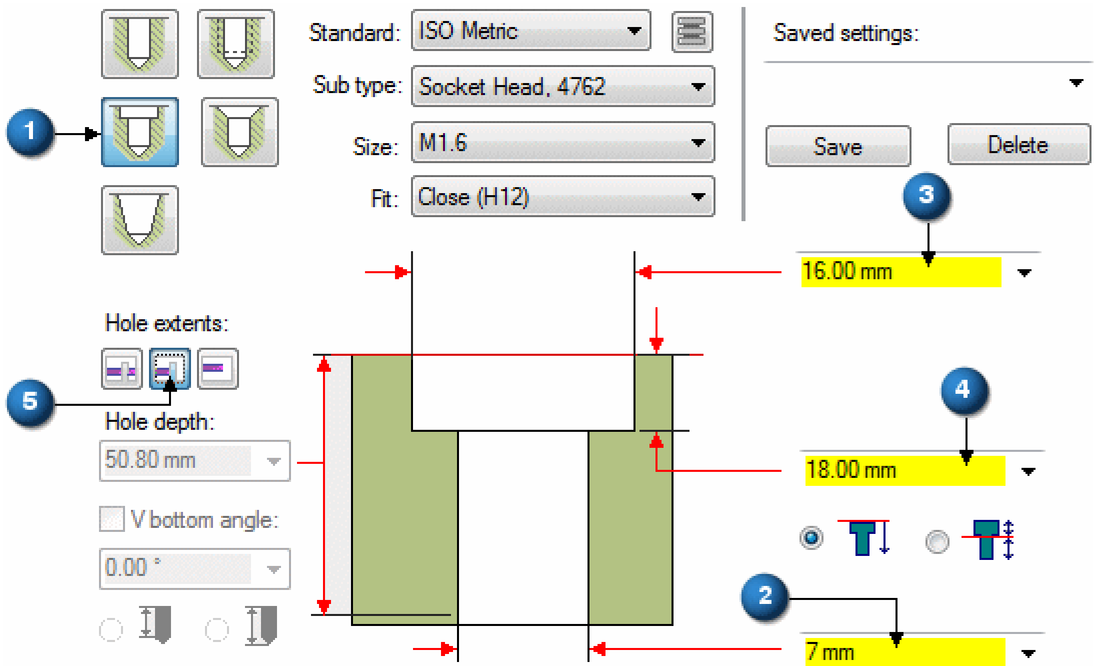
- ☐ Click to place the hole feature.



- ☐ Notice that a hole feature is still attached to the cursor. Since this is the only Simple hole to construct, right-click to finish placing holes.

Construct a counterbore hole

- ◻ Choose the **Hole** command.
- ◻ Set the following hole properties for a Counterbore type hole:



- Set the Type to Counterbore (1).
- Set the Hole diameter (2) to **7**.
- Set the Counterbore diameter (3) to **16**.
- Set the Counterbore depth (4) to **18**.
- Ensure the Hole extents option is set to Through Next (5).

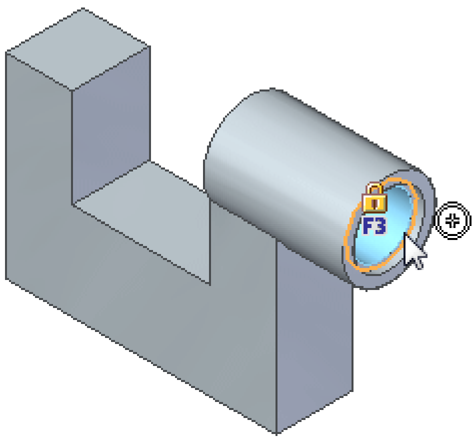
The value fields turn yellow when the values for the hole type are not in the hole database.

- ◻ On the **Hole Options** dialog box, click **OK**.

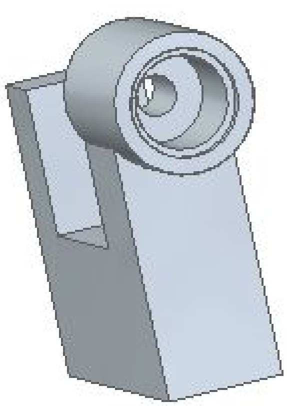
Notice that the hole attached to the cursor updates to reflect the hole properties specified.

Position the counterbore hole

- ☐ Move the cursor over different faces of the model, and notice that a preview image of the results display.
- ☐ Move the cursor to the circular edge as shown, and notice that the hole centers itself on the circular edge.

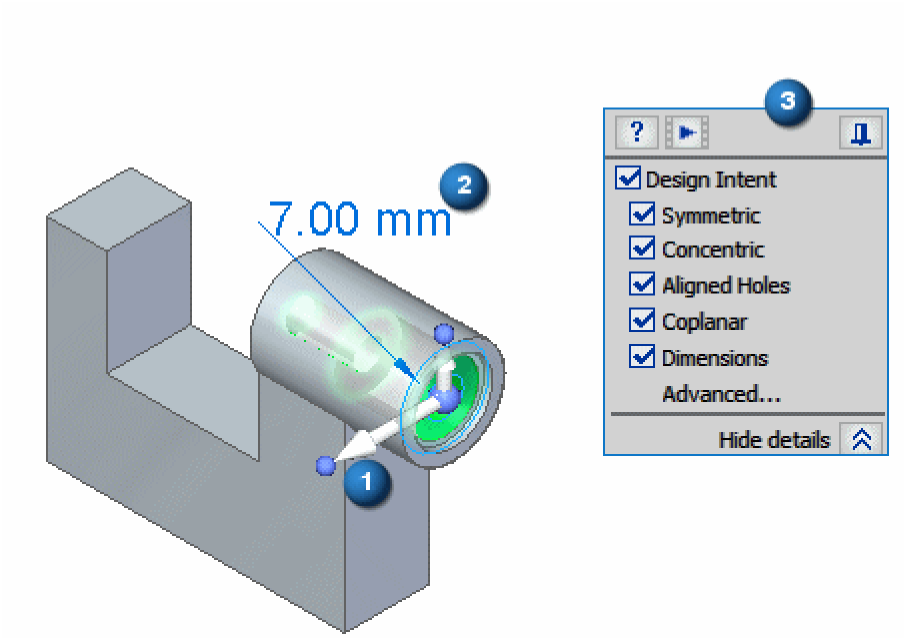


- ☐ Click to place the hole feature.



- ☐ Notice that a hole feature is still attached to the cursor. Since this is the only Counterbore hole to construct, right-click to finish placing holes.

Observe the results



Notice that in addition to the hole feature, the steering wheel (1) and the edit definition handle (2) are displayed.

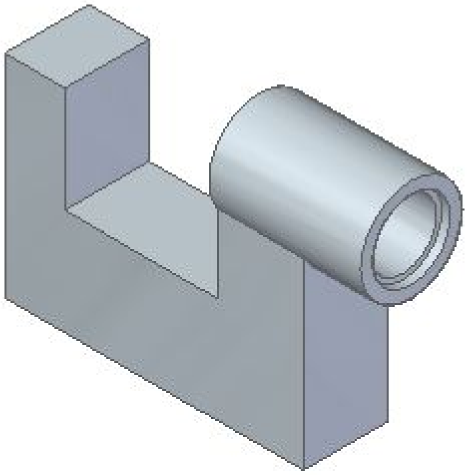
Learn more about the steering wheel later.

The edit definition handle is used to edit procedural features, such as holes. For this test drive, the hole feature will not be edited.

The Design Intent panel (3) also displays in the window.

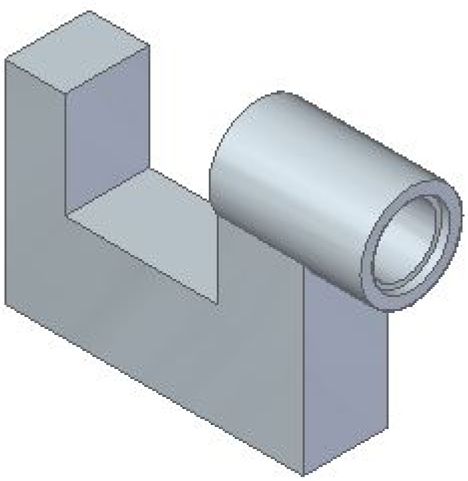
In later steps, learn how Design Intent helps control how a model changes when editing synchronous features.

Save the part



- On the **Quick Access** toolbar, click the **Save** button .

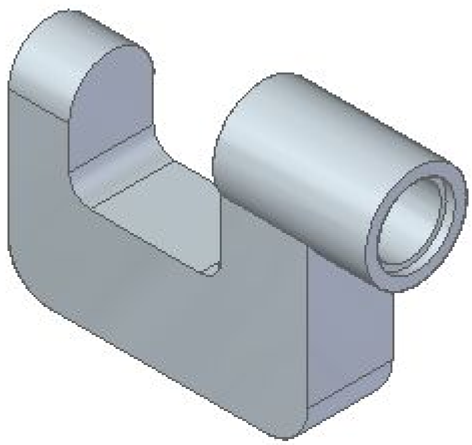
Adding hole features complete



The steps to model the simple and counterbore holes are complete.

Applying round features

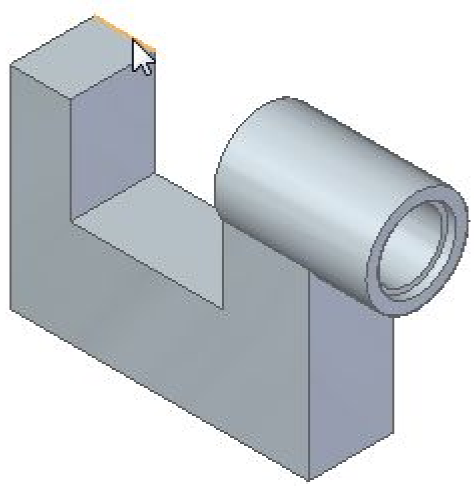
Rounding edges



In the next few steps, use the **Round** command to round several edges on the part.

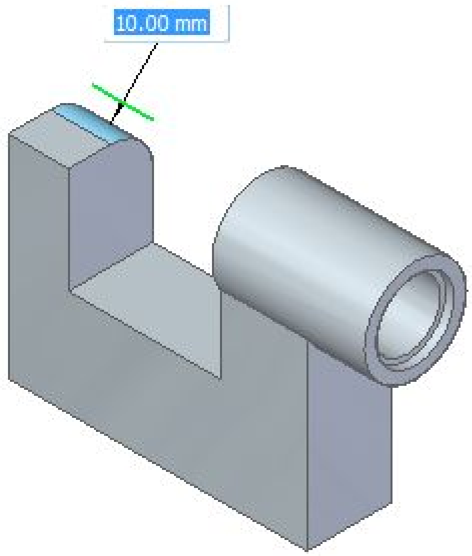
- ◻ Choose **Home** tab→**Solids** group→**Round** command .

Select the first edge to round



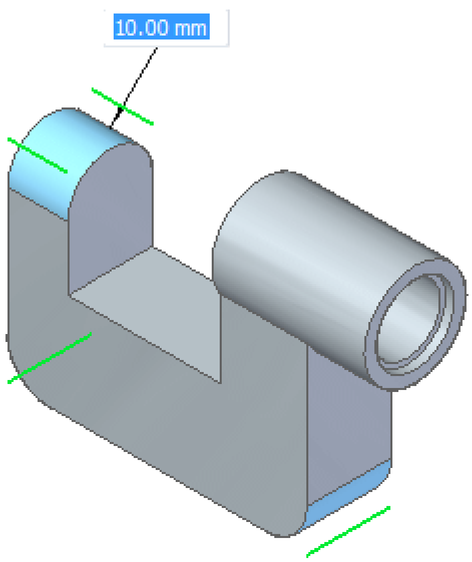
- ◻ Select the edge shown.

- When the dynamic edit box displays, type **10**, and press Tab.



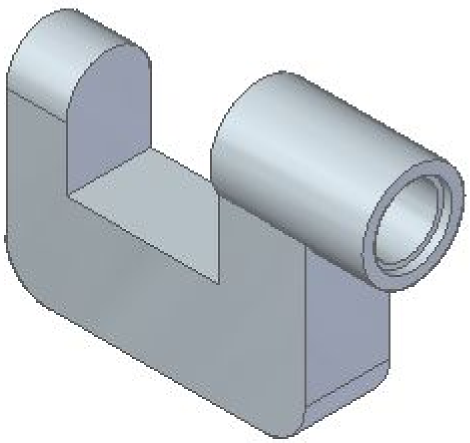
When rounding multiple edges, press Tab to continue to select more edges.

Select the remaining edges to round

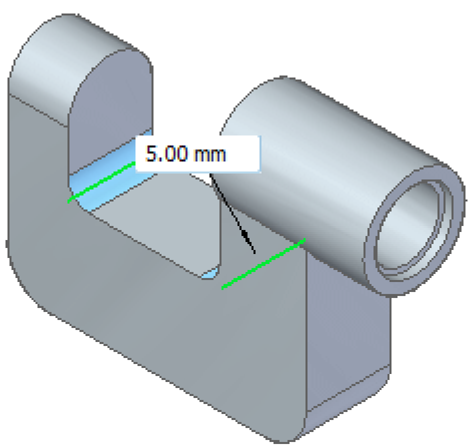


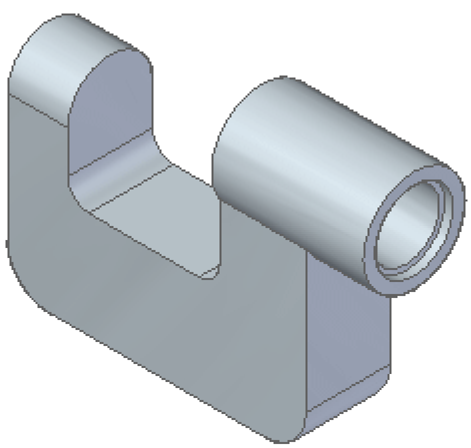
- Select the edges shown.

- ☐ Right-click to finish rounding edges.

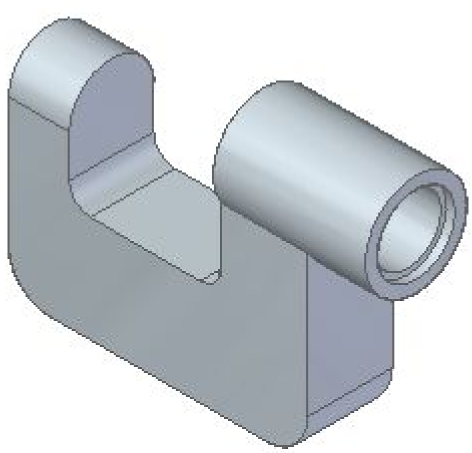


- ☐ Add a 5 mm round to the two edges shown.





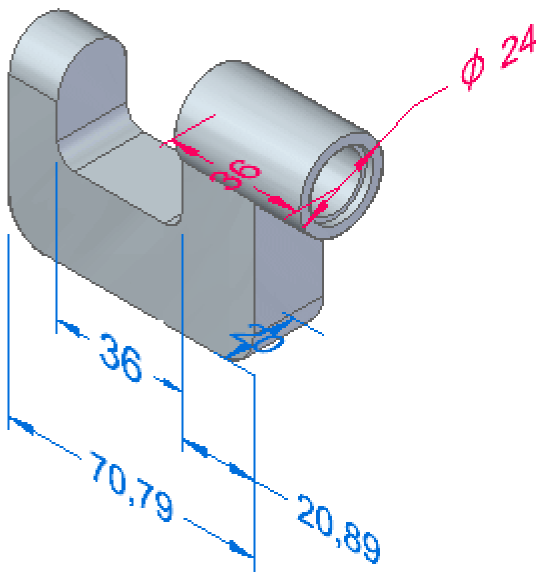
Observe the results



The round features are complete. The model should now resemble the illustration.

Add dimensions

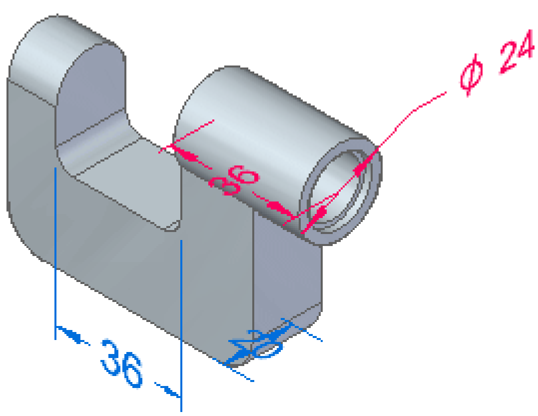
Add dimensions



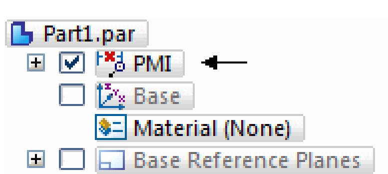
In the next few steps, place two more dimensions on the edges of the model, as shown. Use the **Distance Between** command.

First, display the existing dimensions hidden earlier.

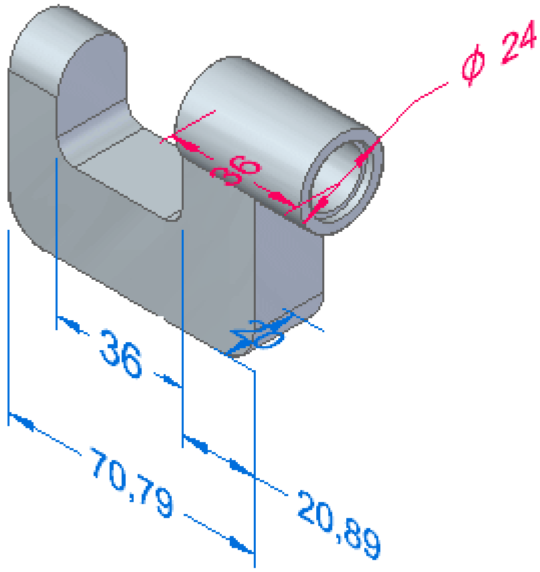
Use PathFinder to display the existing dimensions



- ☐ In **PathFinder**, position the cursor over the PMI check box, then click to display the dimensions in the graphics window.



Start the Distance Between command

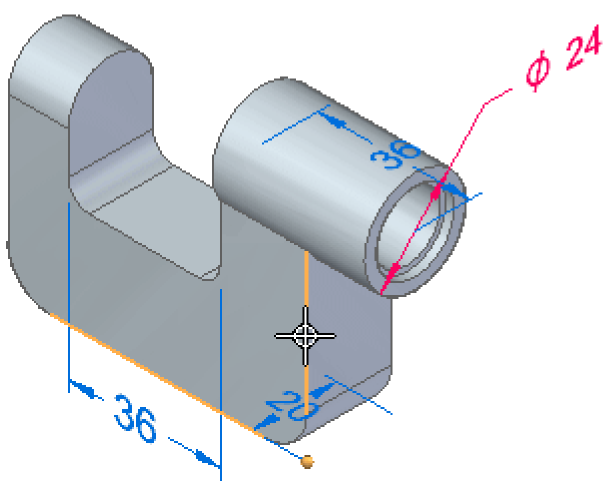


In the next few steps, use the **Distance Between** command to dimension the part.

- Choose **Home** tab→**Dimension** group→**Distance Between** command .

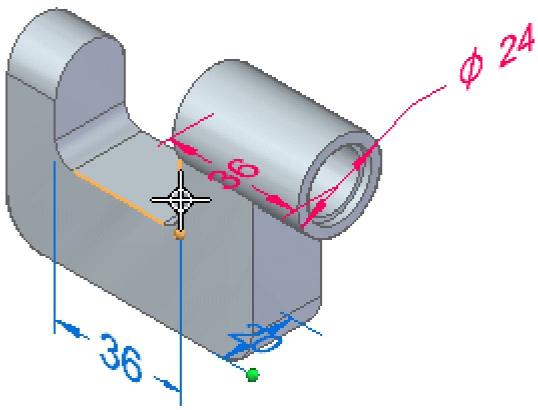
Use this command to dimension between multiple edges or sketch elements. Chained or stacked dimensions can be placed with this command.

Select the first element to dimension

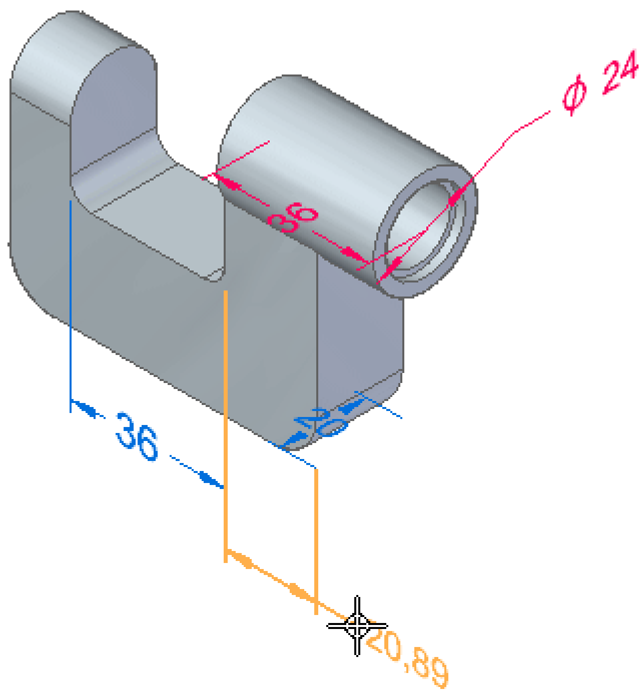


- Position the cursor over the edge of the model shown, then click to select it.

Select the second element to dimension

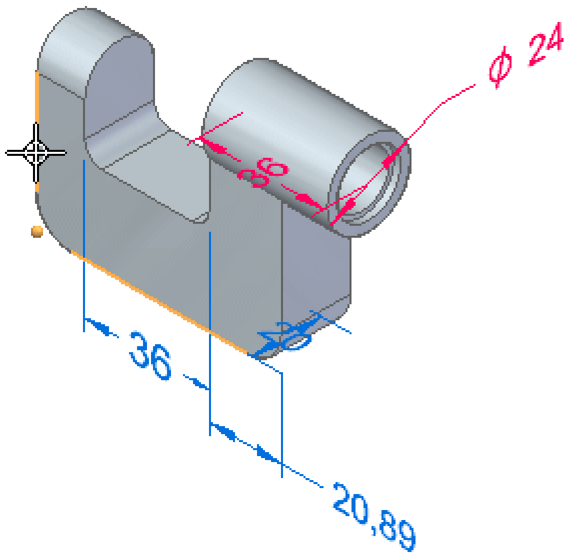


- ◻ Position the cursor over the edge shown above, then click to select it.
- ◻ Move the cursor below the model, then press **N** until the dimension orients as shown below.
- ◻ Click to place the dimension, and then click outside the dimension value box to accept the value.

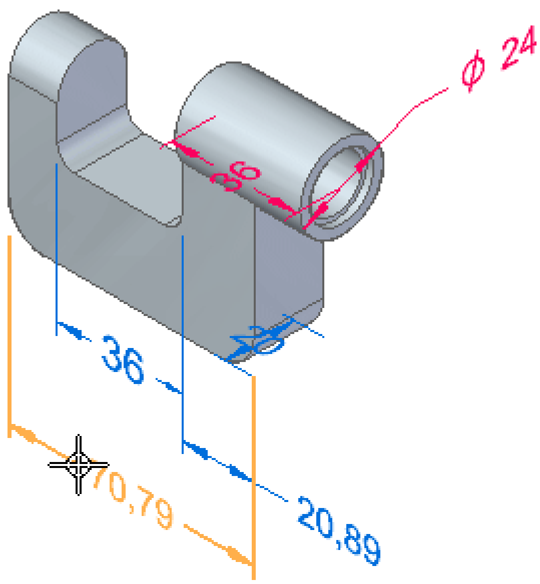


The dimension value for the dimension may be different than the illustration.

Select the third element to dimension



- ☐ Position the cursor over the edge shown above, then click to select it.
- ☐ Position the cursor approximately as shown below. Click to place the dimension, and then click outside the dimension value box to accept the value.

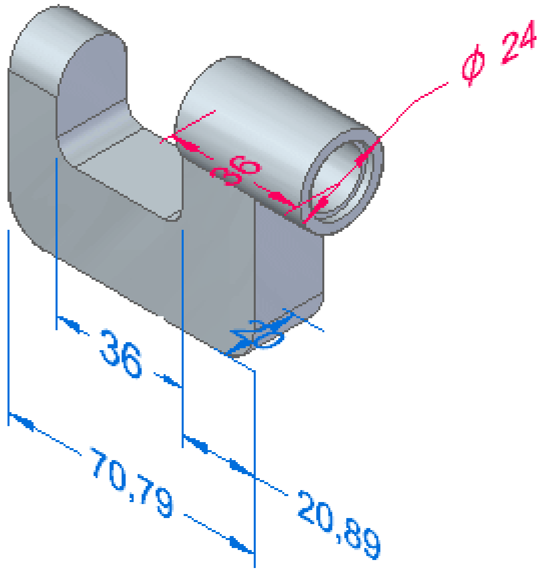


The dimension value for the dimension may be different than the illustration.

Save the part

- ☐ On the **Quick Access** toolbar, click the **Save** button.

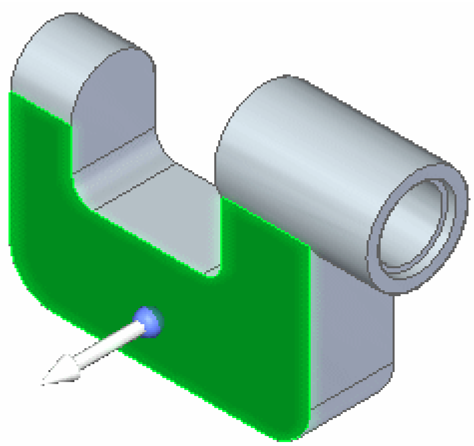
Dimensioning complete

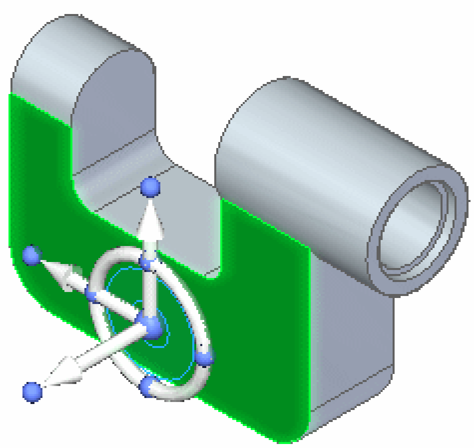


Dimension placement step is finished.

Modify the model

Modify the model



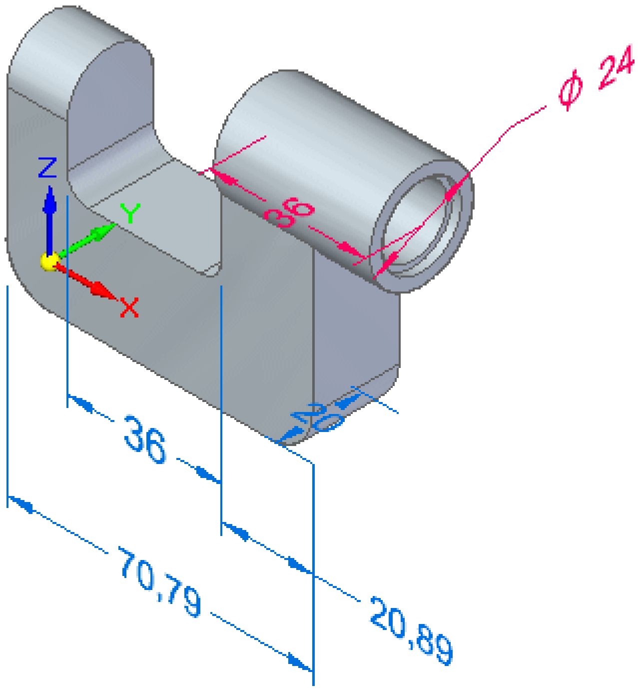


In the next few steps, explore the various methods to modify models in Solid Edge.

First, explore using the grab and go tools, such as the **Select** tool and the steering wheel to interact directly with the faces on the model.

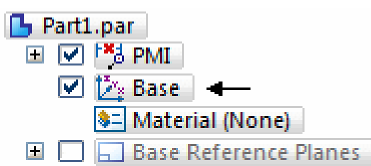
Then edit the value of a dimension to edit the model.

Display the base coordinate system



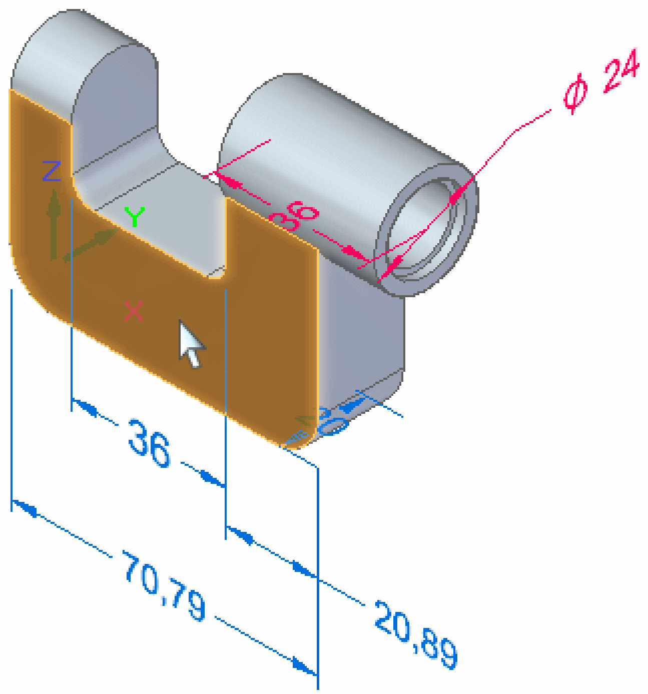
When modifying synchronous models, any model symmetry that exists about the base coordinate system is detected by default. When the base feature was constructed, it was constructed symmetrically about the XZ plane of the base coordinate system.

- ☐ In **PathFinder**, click the Base check box to display the base coordinate system.



The base coordinate system displays as shown in the top illustration.

Select a model face



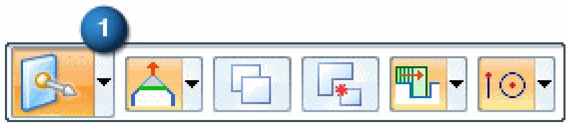
□ The **Select** command should be active. If not, press Escape.

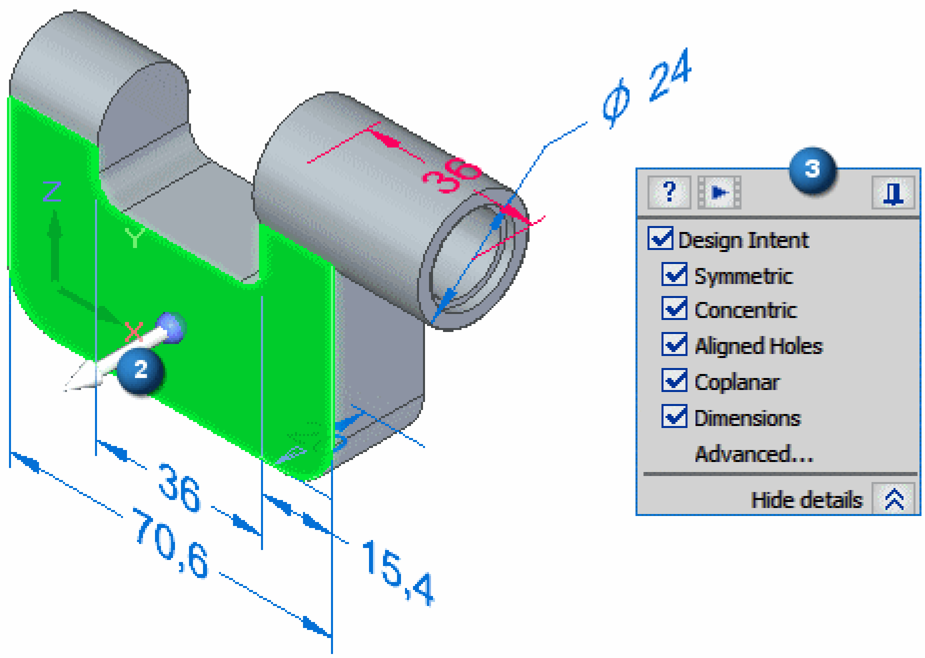
□ Position the cursor over the face shown above. When it highlights, click to select it.

Several tools display that can be used to evaluate and control how the model reacts to the modification:

- Steering wheel
- Command bar
- Design Intent panel

Observe the on-screen tools

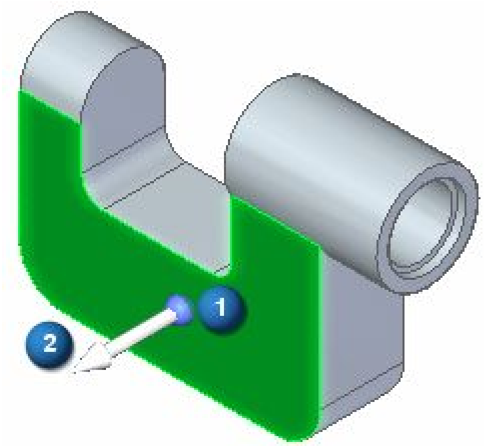




Notice the following, as shown in the illustration:

- A menu, called the command bar, displays in the graphics window (1). The command bar is shown in its horizontal format here, but if it may appear vertically, depending on the user interface theme settings.
- The steering wheel displays at the approximate point the face was selected (2).
- The Design Intent panel (3) also displays.

Steering wheel overview



When selecting a face on a model, the default command bar action is to move the face. Other options can be specified, but for this test drive, focus on the **Move** option.

Use the steering wheel to manipulate model elements, such as to move or rotate one face, or a set of faces.

Use the different controls on the steering wheel to control the manipulation process.

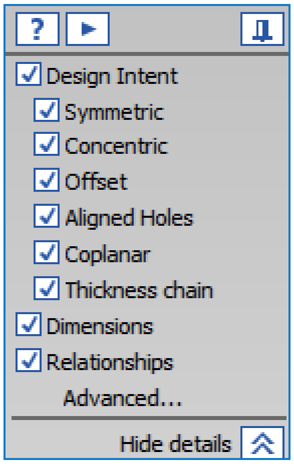
When using the steering wheel to move faces along a linear vector, it has two components:

- (1) Origin knob - Used to define the from-point for from-to moves. Click and drag the origin knob to reposition the steering wheel to another location on the model. This redefines the axis direction and change the point of reference for the move.

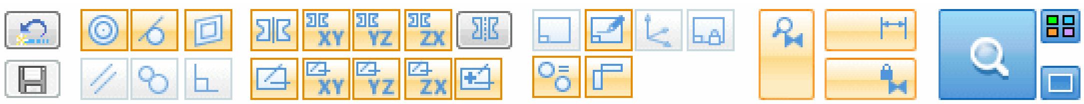
- (2) Primary axis - Click this to move elements along this axis.

Design Intent overview

Click the **Advanced**... option in the Design Intent panel to open the **Advanced** Design Intent panel.



Design Intent Panel



Advanced Design Intent panel

Depending on the current configuration of your computer, the active Design Intent settings on your computer may be different than the illustration.

□ On the **Advanced** Design Intent panel, click the **Restore Defaults** button .

The Design Intent settings should now match the illustration.

Use the Design Intent options to locate and show the relationships between faces in the current select set and the rest of the model. Use this information to control how synchronous modifications are performed.

The Design Intent panel opens when making the following types of synchronous modeling modifications:

- Moving or rotating model faces.
- Defining 3D geometric relationships between model faces using the Relate command.
- Editing the dimensional value of a 3D PMI dimension.

The current settings show:

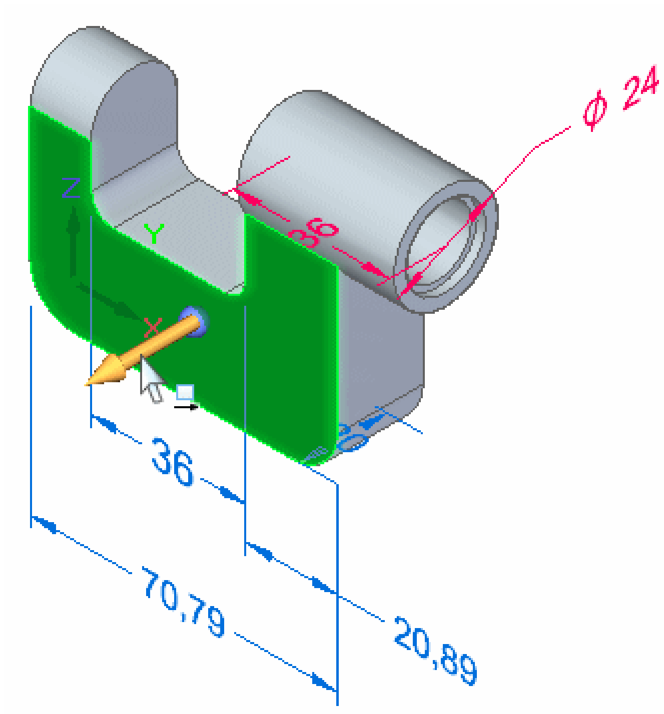
 Concentric faces remain concentric.

 Coplanar faces remain coplanar.

 Tangent edges remain tangent.

 Model symmetry about the base coordinate system is maintained.

Modify the model using the steering wheel



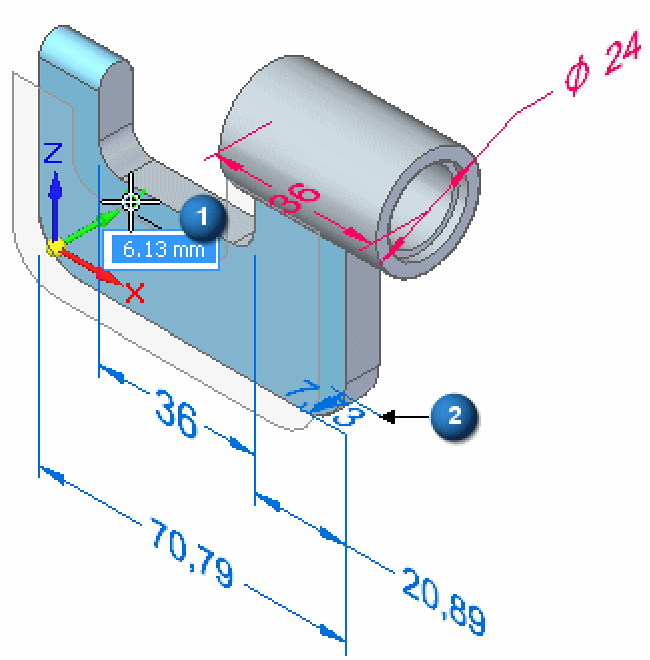
- ◻ Position the cursor over the primary axis on the steering wheel, and when it highlights, click to select it, as shown above.
- ◻ Move the cursor slowly to the right and then left.

Notice the following:

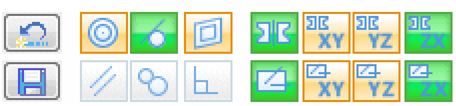
- The dynamic input box displays near the cursor (1) in order to type a precise value for the delta distance of the move.

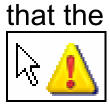
- The dimension value text of the affected PMI dimension updates (2).

- The model updates symmetrically about the base coordinate system XZ plane.

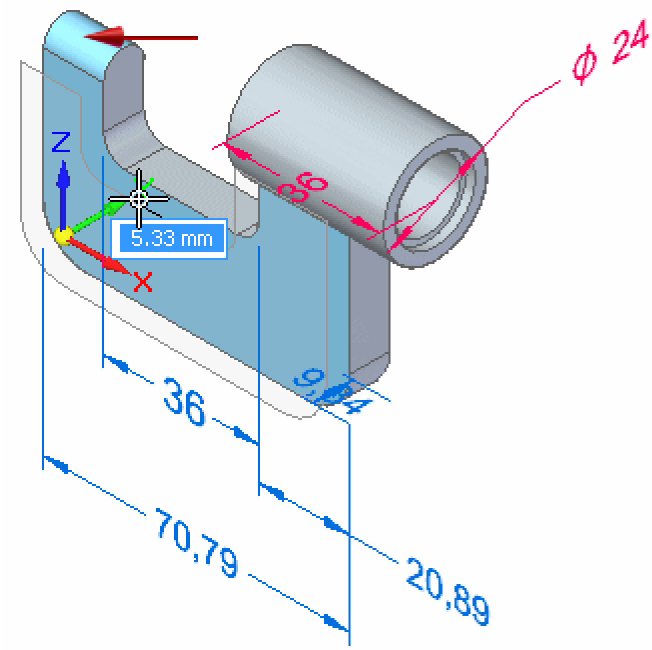


- Design Intent **Advanced** panel options that have an effect on the move operation are shaded green.



- When moving the cursor too far to the left, such that the planar face extends past the cylindrical face at the top-right of the part, an error symbol  displays.

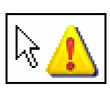
Synchronous technology in action



Options in Design Intent **Advanced** panel are shaded green when non selected model geometry matches a Design Intent setting.

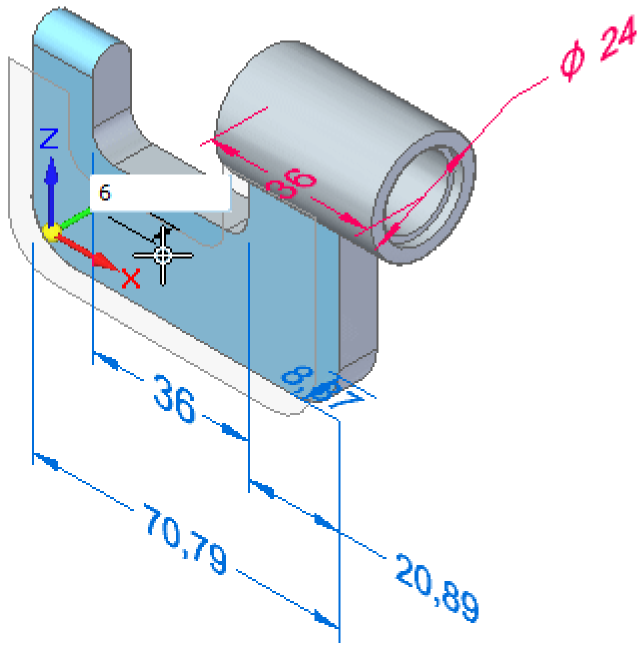
In this example, the Tangent Edges option  ensures that the non selected tangent cylindrical face shown above maintains tangency when moving the planar face.

The Symmetric about Base: (Z)X setting maintained model symmetry about the (Z)X plane of the base coordinate system.

The error symbol  displays when moving the cursor past the point a valid solid model could be produced.

Synchronous technology ensures a notification for these conditions.

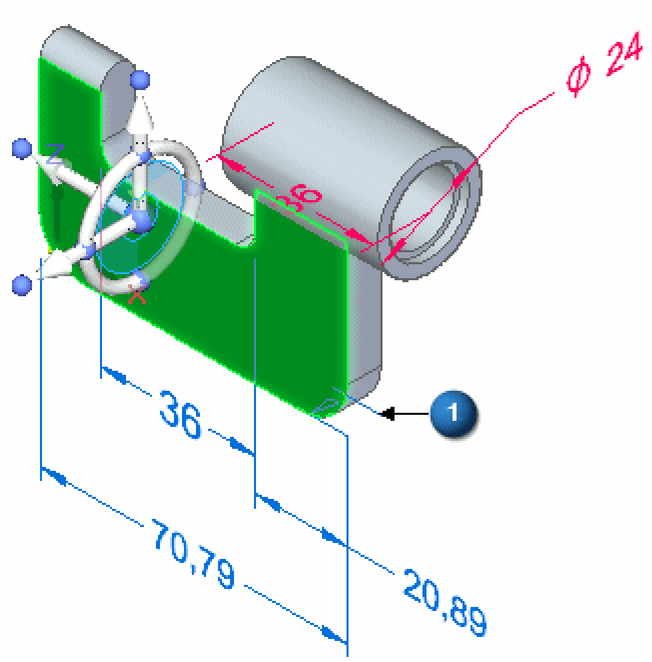
Define the delta distance of the move



- □ Position the cursor such that the model is thinner than the original 20 mm, as shown above.
- □ In the dynamic edit box, type **6**, then press Enter.

Notice that the PMI dimension for the model thickness (1) is now 8 mm, as shown below.

Because model symmetry was maintained, both the front and back faces moved 6 mm.



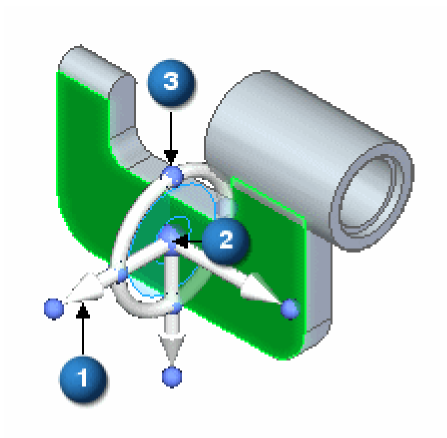
Steering wheel overview, continued

Notice that the steering wheel display has changed.

When first selecting this face on the model, the steering wheel only displayed the primary axis and the origin knob. The display of the steering wheel handle can vary based on the elements in the select set. This display variation is called progressive exposure. This means that in many modeling scenarios, only some of the steering wheel components display when selecting an element.

Now that the move operation on the face is complete, the steering wheel is fully displayed, ready for additional work, if called upon.

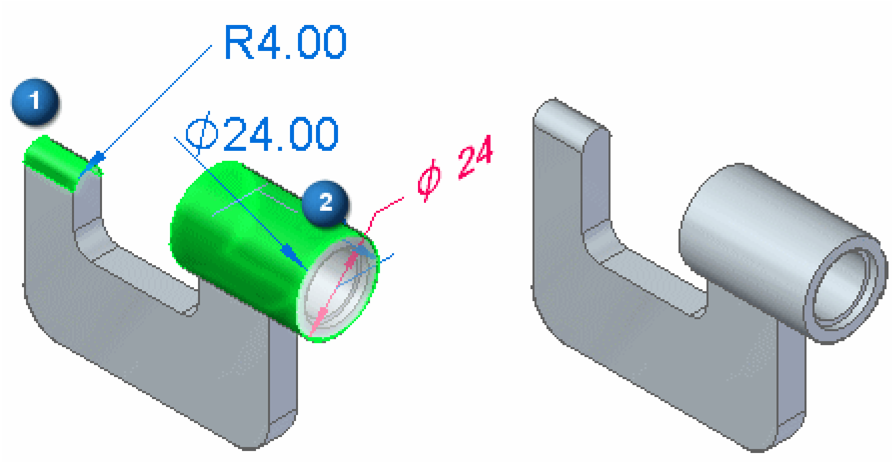
Some of the fundamental features of the steering wheel are:



- (1) Axes - Click one of the three axes to move elements along the axis.
- (2) Origin knob - The origin defines the from-point for from-to moves. Click and drag the origin knob to reposition the steering wheel where needed on the model. This redefines the axis direction and change the point of reference for the move.
- (3) Torus knobs - Click one of the four knobs to reposition the axes in the selected direction.

The steering wheel is a powerful modeling tool, and with these fundamental components, many things can be accomplished.

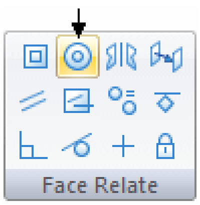
Apply a face relationship



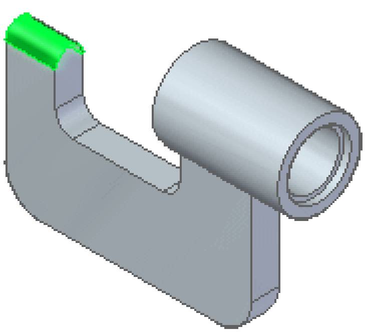
The design intent for this model is for the cylindrical faces (1) and (2) to be concentric.

Select faces to relate

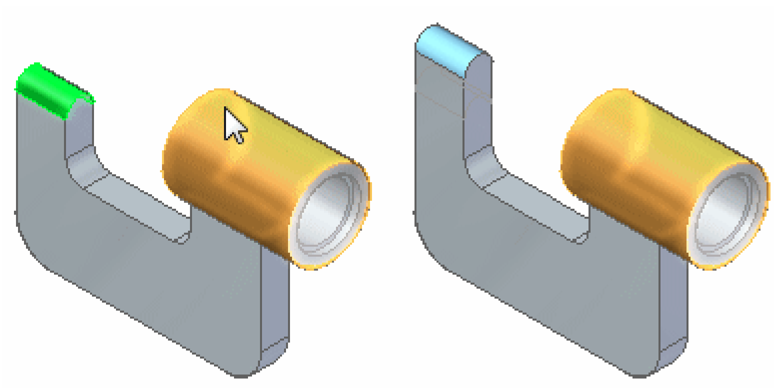
- Choose the **Home** tab→**Face Relate** group→**Concentric** command.



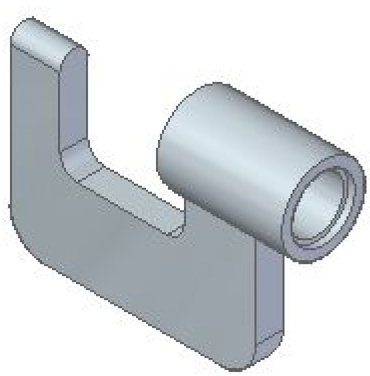
- Select the cylindrical face shown and then right-click or click the check mark on the command bar.



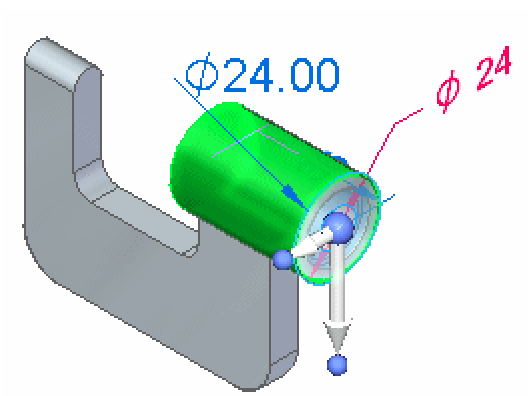
- ☐ Select the cylindrical face shown to be the face to concentrically align to. Right-click or click the check mark to apply the relationship.



Cylindrical faces are now concentrically aligned.



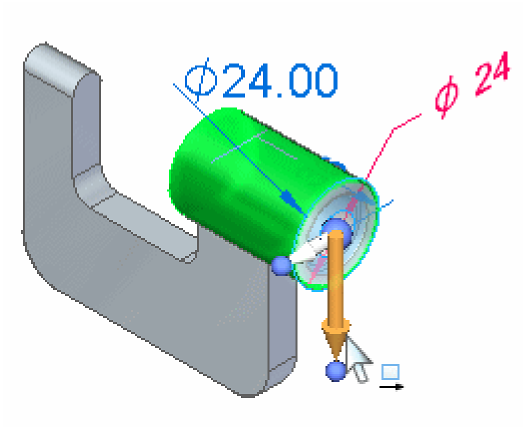
Select more faces to move



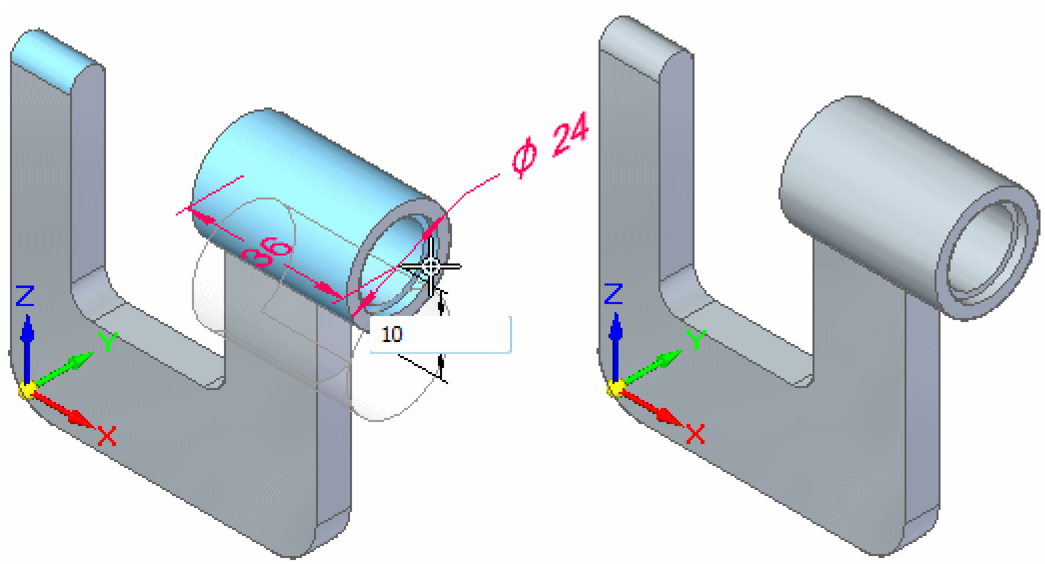
- ☐ Select the cylindrical face shown in the top illustration.

> As this face moves, the two holes and the rear cylindrical face also move to maintain concentricity. Design Intent controls this behavior.

- Click the steering wheel axis shown.

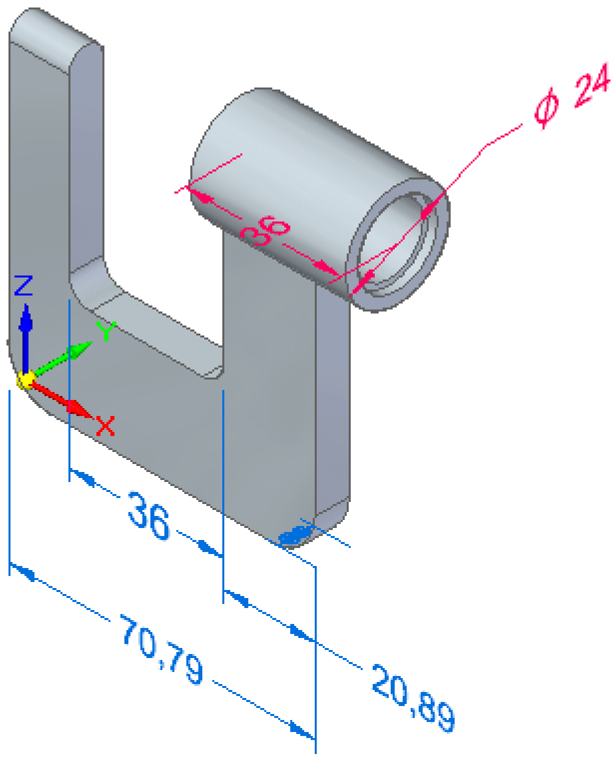


- Move the cursor in an upward direction and type **10** in the dynamic edit box.



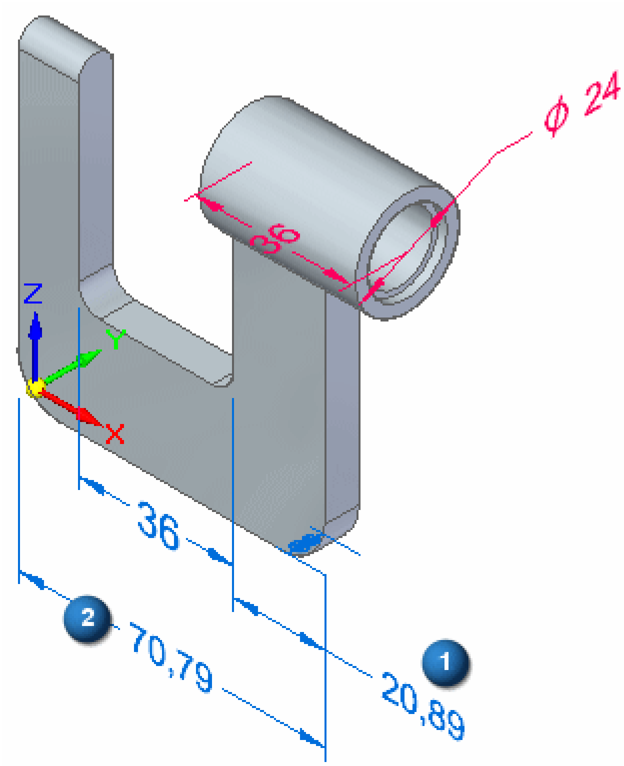
The move faces action is complete.

Save the part



□ On the **Quick Access** toolbar, click the **Save** button .

Edit dimensions

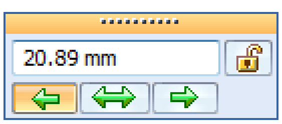


In the next few steps, edit two of the PMI dimensions (1) and (2) on the model.

- ☐ Position the cursor over the dimension text (1) and click.

Notice the dimension display shows a red arrow indicating which end of the model changes when editing the dimension value.

The **Dimension Value** Input dialog box displays as shown below.

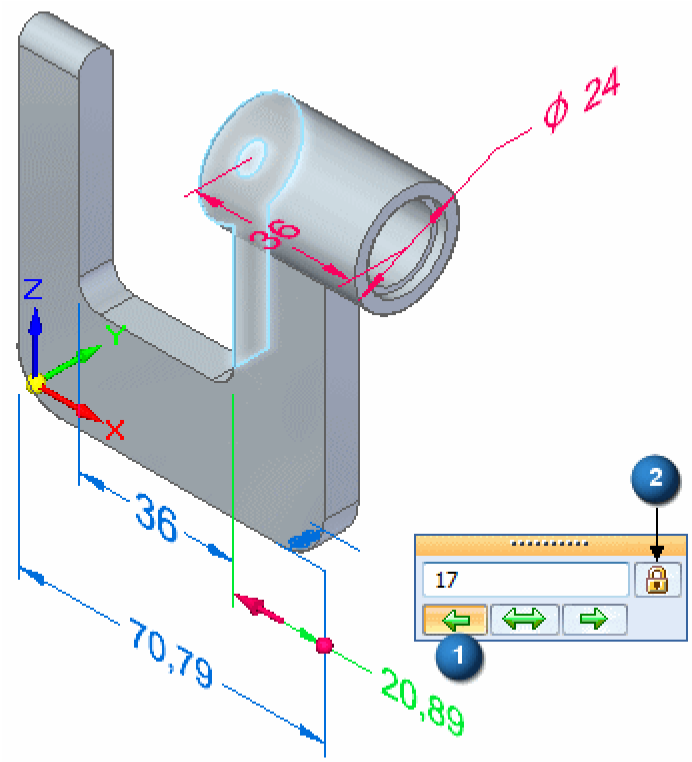


Dimension Value Input overview

Observe the options on the Dimension Value Input dialog box and the display of the selected dimension.

- (1) Edit Direction 1 - Specifies that the model geometry moves from this end when set. The face that moves highlights in light blue.
- (2) Symmetric Edit - Specifies that the model geometry moves symmetrically when set.
- (3) Edit Direction 2 - Specifies that the model geometry moves from this end when set. The face that moves highlights in light blue.
- (4) Locked/Unlocked - Specifies whether the model geometry controlled by the dimension can change when making modifications with the steering wheel. When a PMI dimension is set to unlocked, its turns blue. When it is set to locked, its turns red.
- (5) Dimension Value box - Specifies a precision value for the dimension. Use this box to type new dimension values when editing models.

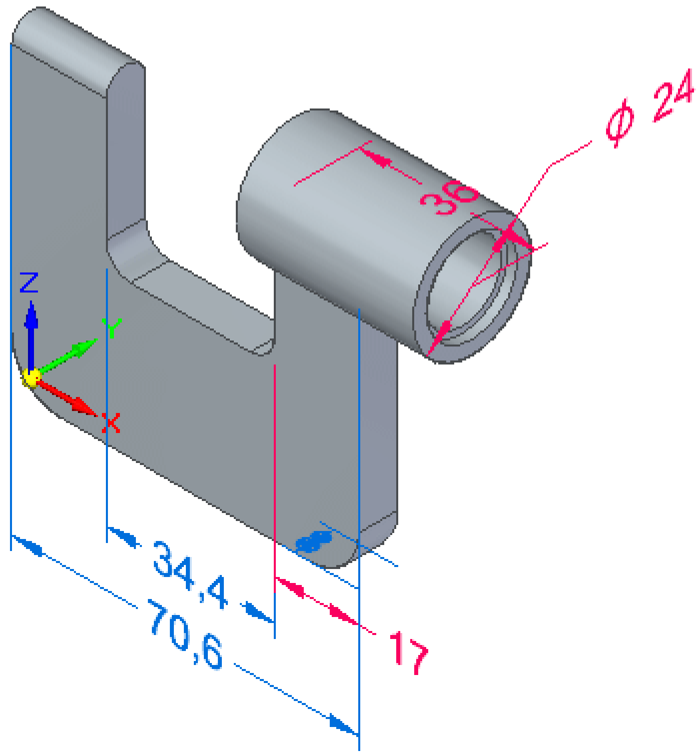
Edit a dimension



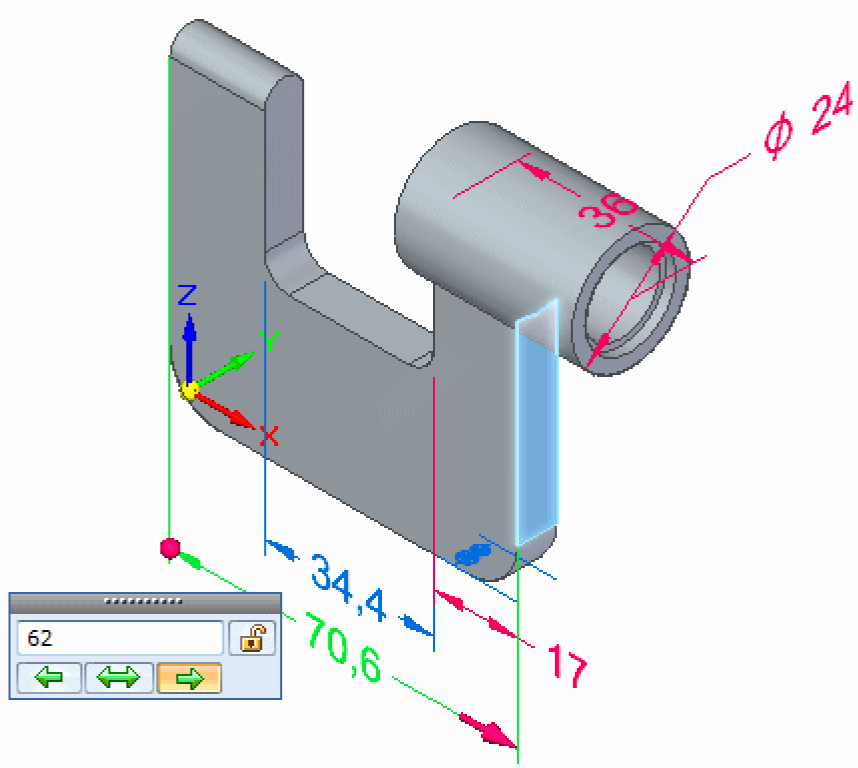
Do the following:

- ☐ Ensure the Edit Direction 1 option is set. (1)
- ☐ Click the lock button to lock the dimension. (2)
- ☐ If the mouse has a scroll wheel, position the cursor in the dimension value box and rotate the scroll wheel back and forth slowly to edit the model. Notice that the dimension value box and the model update dynamically.

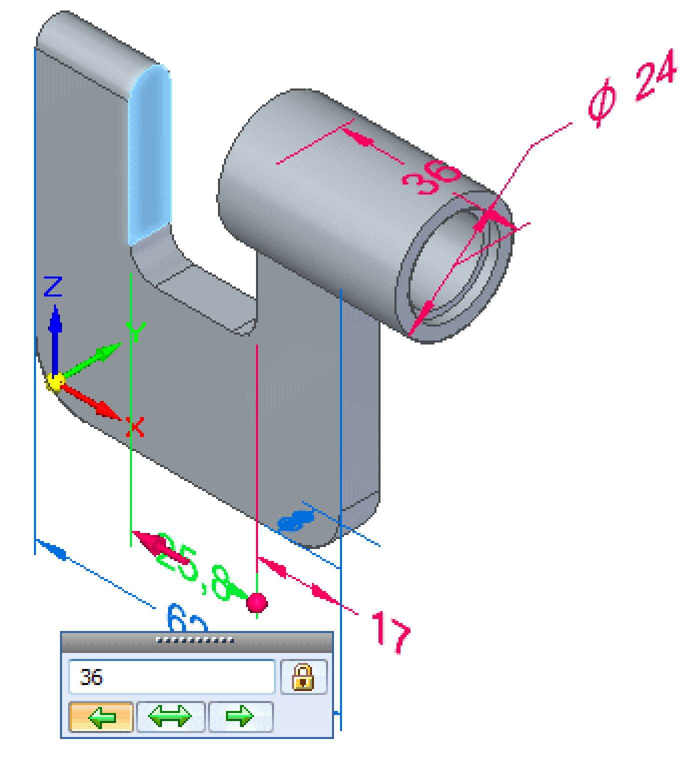
□ Type **17**, then press Enter to define the model shape.



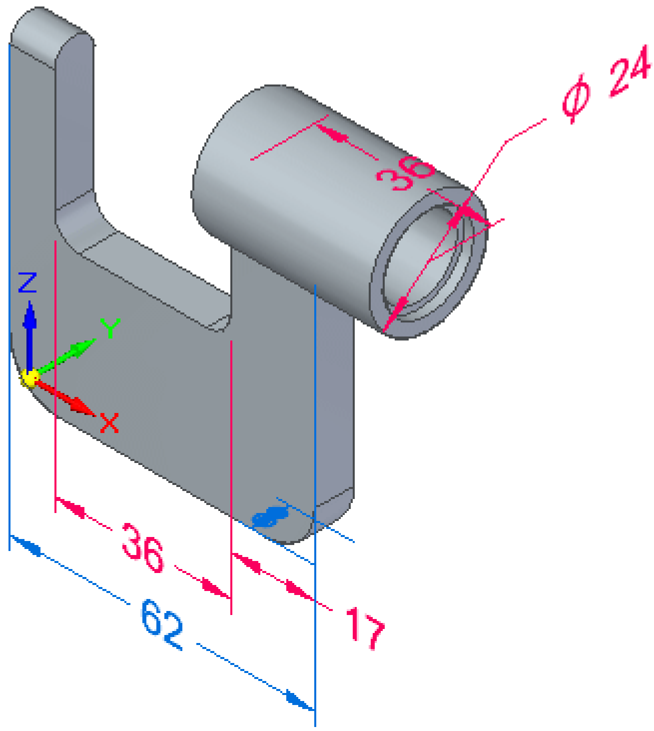
□ Edit the lower dimension. Set the direction shown and type **62**. Press the Enter key.



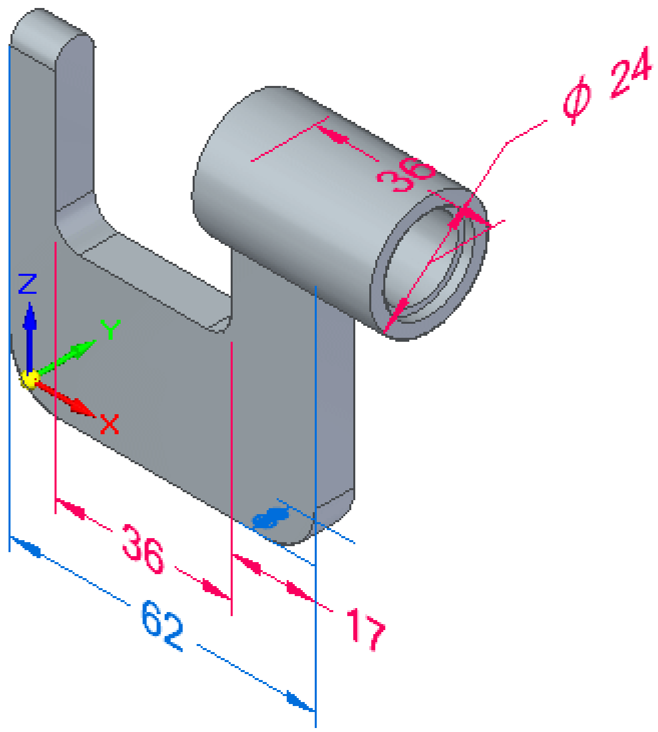
- Edit the dimension shown. Lock the dimension, Set the direction shown and type **36**. Press the Enter key.



Observe the results



Save the part

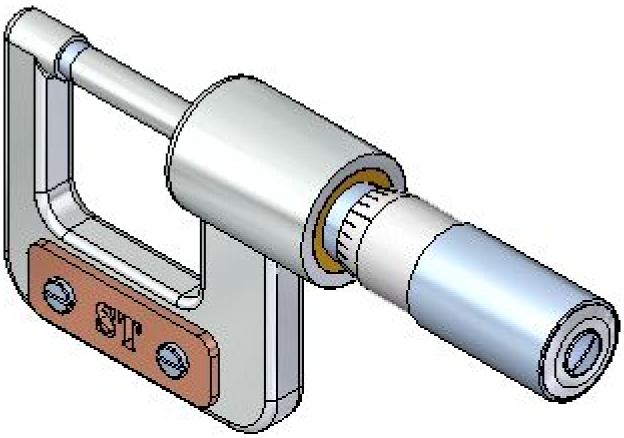


□ On the **Quick Access** toolbar, click the **Save** button .

Congratulations!

The modeling portion of this tutorial is complete. Although there are more features and dimensions that could be added to this part, the basic concepts required to construct a 3D solid model were learned.

Chapter 2: Introduction to creating assemblies

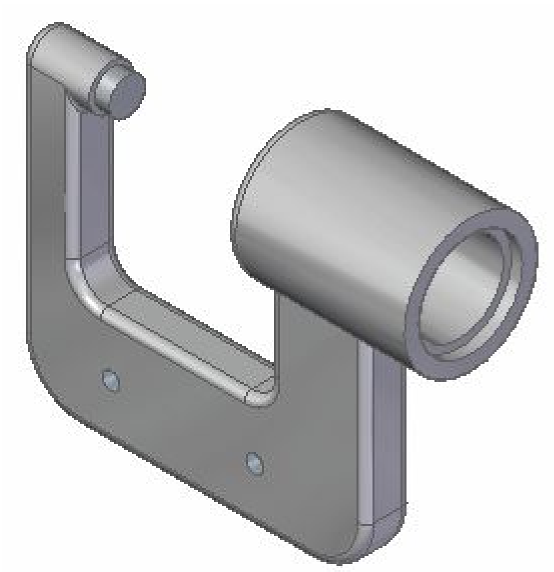


This activity provides step-by-step instructions for building the assembly shown in the illustration above. Building this assembly provides learning techniques such as:

- Using **PathFinder** to manage the display of parts in the assembly.
- Applying assembly relationships between parts.
- Editing parts in the context of the assembly.
- Creating exploded views of an assembly.

Assembly environment overview

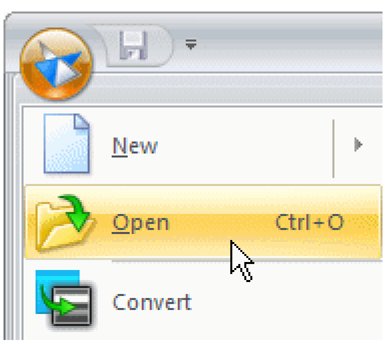
Observe the current assembly



In the next few steps, familiarize yourself with the assembly document.

Learn how to highlight and select assembly components using **PathFinder**.

Open an assembly



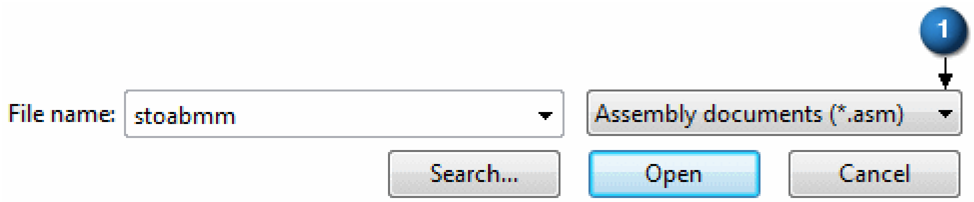
- ☐ On the **Application** menu, click **Open**.

The **Open File** dialog box displays.

- Navigate to the Solid Edge training folder. The default location is:

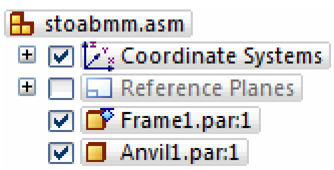
  C:\Program Files\Solid Edge ST9\Training

- Set the File name field to **stoabmm** and choose Assembly documents from the file type list (1).



- Click **Open** to open the assembly.

Display PathFinder



PathFinder should already be displayed. If not, follow these steps to display it.

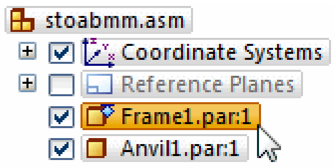
- On the **Applications** menu, click **Settings→Options**.

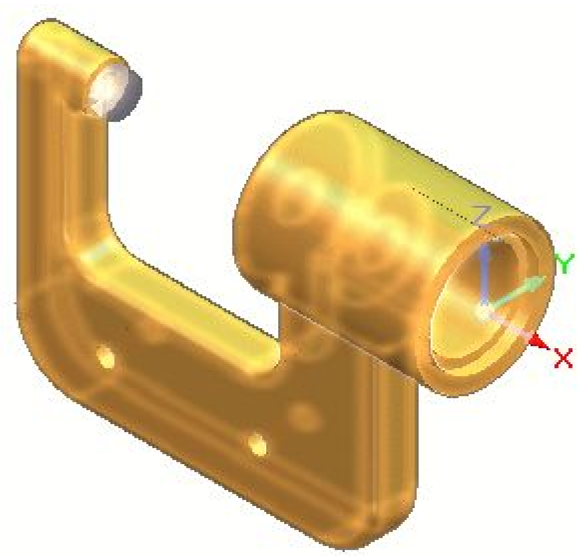
The Solid Edge Options dialog box displays.

- Click the **Helpers** tab, and ensure the *Show PathFinder in the document view* option is set.

Use the **PathFinder** tab to review and edit the assembly structure, hide and display assembly components, such as parts, subassemblies, coordinate systems, and reference planes.

Highlight the frame part



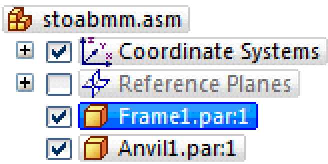


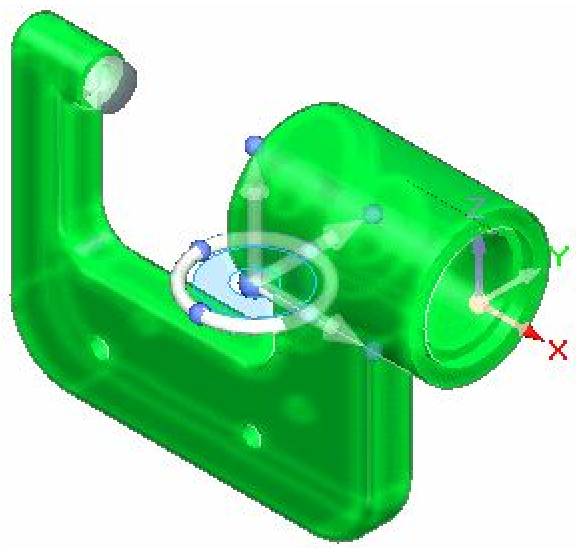
- ☐ In the top pane of **PathFinder**, position the cursor over the **Frame1.par** entry, but do not click.

Notice that the frame part display changes color in the assembly window.

Move the cursor away and notice that the display returns to the previous color.

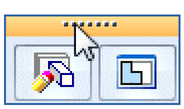
Select the frame part





- In **PathFinder**, position the cursor over the frame part again, then click, and move the cursor away.

- If a command bar with edit commands obscures the view, drag the command bar to the side, out of the way.

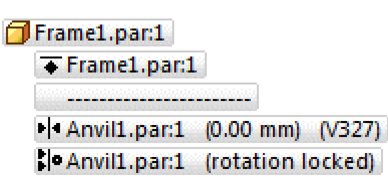


Notice that the part color in the graphics window changes to a different color than in the previous step.

Also notice that when selecting the part, the bottom pane of **PathFinder** displays the assembly relationships used to position the part, as shown below.

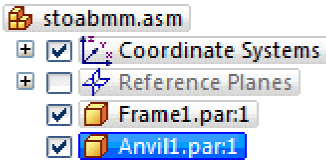
If the relationships cannot be seen, use the scroll bar to move them, and drag the top of this display area higher, to make more room for the display.

Since this was the first part placed in the assembly, the relationship symbol that displays is the ground relationship.



When working in assemblies, temporarily highlight components using **PathFinder**, and also select them.

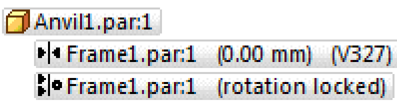
Select the anvil part



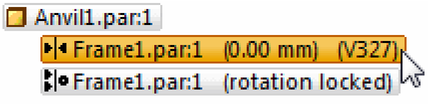
- □ In **PathFinder**, position the cursor over the anvil part, then click, and move the cursor away.

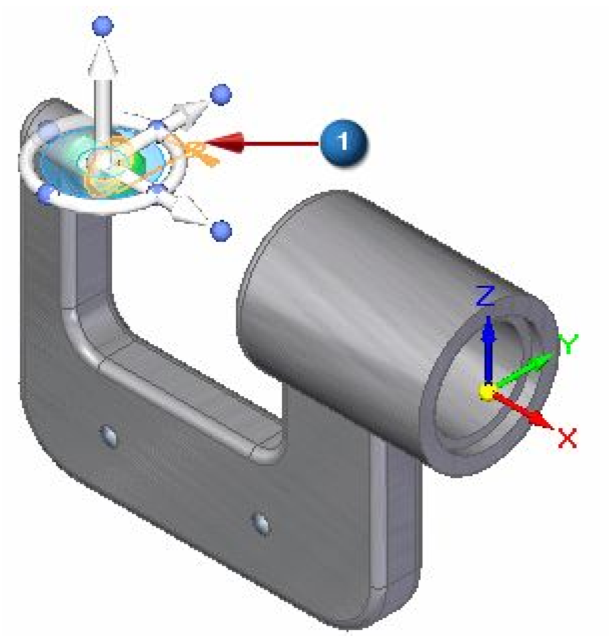
Notice that when selecting the anvil part, the bottom pane of PathFinder displays the two assembly relationships used to position the part, as shown below.

A mate relationship and an axial align relationship were used to position the part with respect to the frame part. Learn more about relationships when placing more parts into the assembly later.



Highlight the relationships in PathFinder



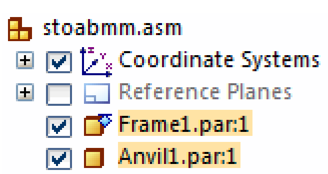


- ☐ Position the cursor over the first assembly relationship in the bottom pane of **PathFinder** as shown in the top illustration, but do not click.

Notice that the two faces used to position the anvil part highlight in the graphics window, along with a symbol (1) representing the relationship between them.

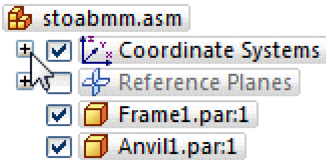
Also notice that shaded boxes display around the two parts in the top pane of PathFinder, as shown below.

These display clues make it easy to evaluate how an assembly was constructed later.

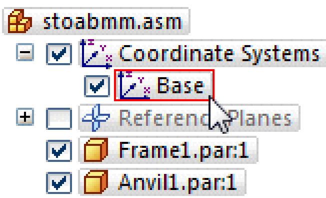


Display the coordinate systems collection

- ☐ In the graphics window, click in free space to deselect the anvil part.
- ☐ In **PathFinder**, position the cursor over the "+" symbol adjacent to the **Coordinate Systems** collection, and click.

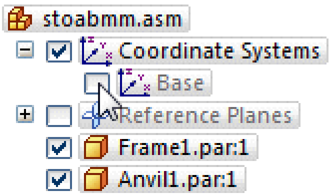


Notice that an entry for the Base coordinate system displays.



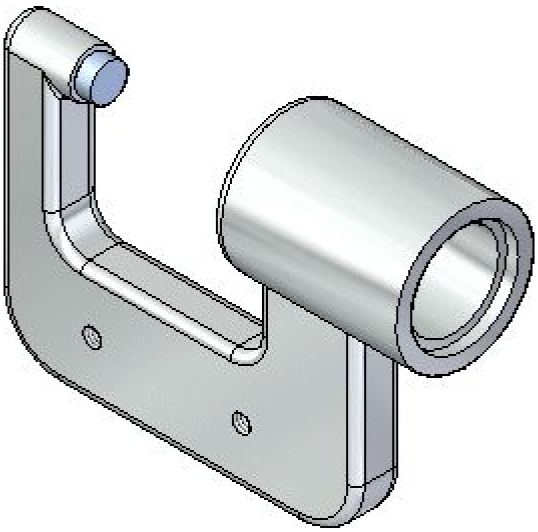
There is one Base coordinate system in an assembly document, located at the center of the design space. Any additional coordinate systems defined are added to the **Coordinate Systems** collection in **PathFinder**.

Hide the coordinate system



- ☐ In **PathFinder**, position the cursor over the check mark adjacent to the Base entry, then click to hide the coordinate system.

The coordinate system is hidden in the graphics window, as shown below.

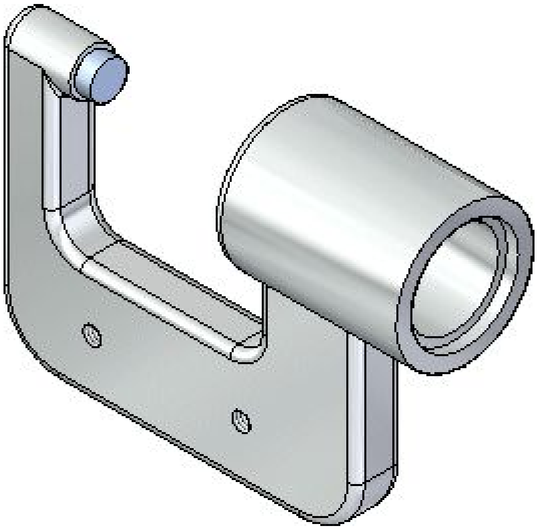


Notice that the text in **PathFinder** for the Base entry has changed color.

Use the check boxes in **PathFinder** to display and hide assembly components.

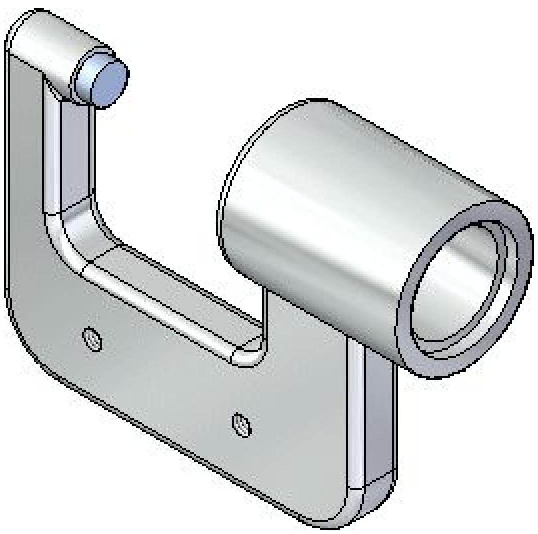
The component entries in PathFinder also change color to indicate the current status of the assembly components.

Fit the view



□ Choose **Fit**  to fit the contents of the view to the graphics window.

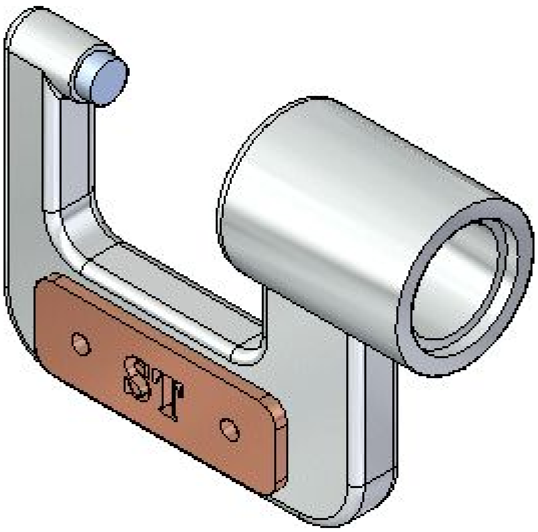
Assembly overview complete



Congratulations, the first step in this activity is complete: getting familiari with the current state of the assembly and learning about **PathFinder** options in an assembly document.

Placing a part in the assembly model

Position a part in the assembly

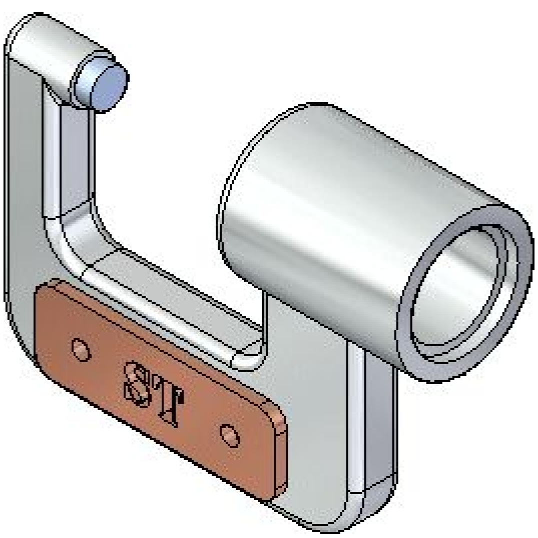


In the next few steps, place a name plate onto the micrometer frame.

Learn how to find and select parts using **Parts Library**.

Learn how to position parts using assembly relationships and the **Assemble** command bar.

Part Positioning Overview



To position parts in an assembly, a variety of assembly relationships are applied.

For the name plate part, apply a mate relationship and two axial align relationships to fully position the name plate with respect to the frame.

Solid Edge provides a tool called **FlashFit** to apply each of these relationships, without having to specify the exact relationship type to use.

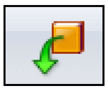
In the next few steps, use **FlashFit** to fully position the name plate as shown above.

Display the Parts Library pane

Both the **PathFinder** and **Parts Library** panes will be used to select and position parts in the assembly.

To make it easier to see the contents of the **Parts Library** and the **PathFinder** panes, maximize their size.

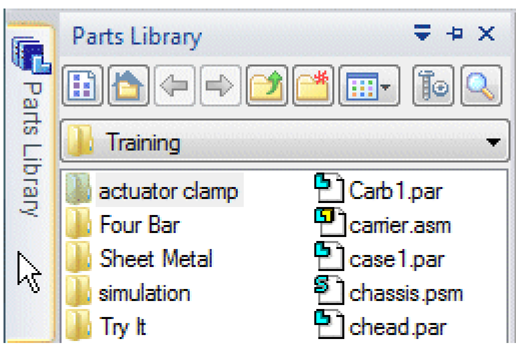
- Choose **Home** tab→**Assemble** group→**Insert Component** command. The **Parts Library** displays.

  The **Parts Library** tab remains displayed as long as the **Insert Component** command is active.

- Press Escape to end the command.

- On the left or right side of the Solid Edge window, depending on the user interface theme selected, move the cursor over the **Parts Library** tab, but do not click.

The **Parts Library** tab displays.

- Move the cursor into the graphics window, away from the **Parts Library** pane, and notice that the **Parts Library** pane closes. As long as the **Parts Library** pane displays and the cursor is over it, it will continue to display.

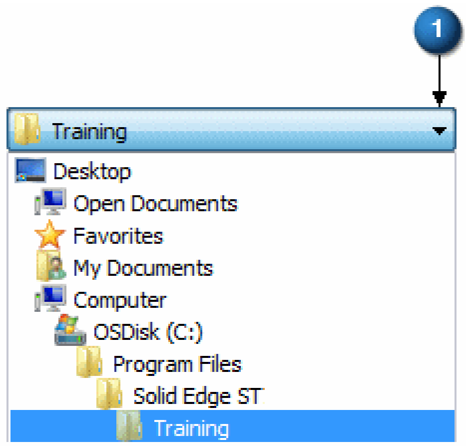
- Now move the cursor over the **Parts Library** tab and click. The **Parts Library** tab displays again.

- Move the cursor away from the **Parts Library** pane, and notice that the pane remains displayed.

- Now move the cursor into the graphics window, away from the **Parts Library** pane, and click.The **Parts Library** tab closes.

In the instructions that follow, work with information in the **Parts Library** pane. Use whichever method of displaying the pane is more comfortable.

Set the Parts Library folder



If the working folder on the **Parts Library** tab is not the Solid Edge Training folder, do the following:

- □ On the **Parts Library** tab, click the arrow (1) on the right side of the Look In control and then browse to the Solid Edge Training folder.

The default location of the Solid Edge Training folder is:

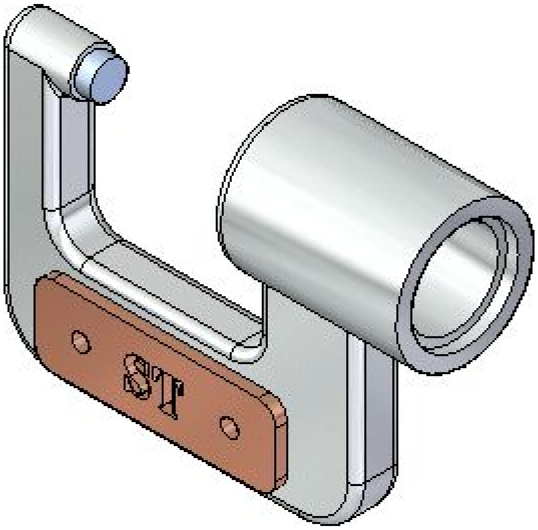
C:\Program Files\Solid Edge ST9\Training

However, the system administrator may have chosen a different location.

Similar to Windows Explorer, define how to view the files listed in the Parts Library: Large Icons, Small Icons, List, and Details.

- □ On the **Parts Library** tab, click the **Views** button, and then set the **Details** option.

If You Have Trouble Placing Parts



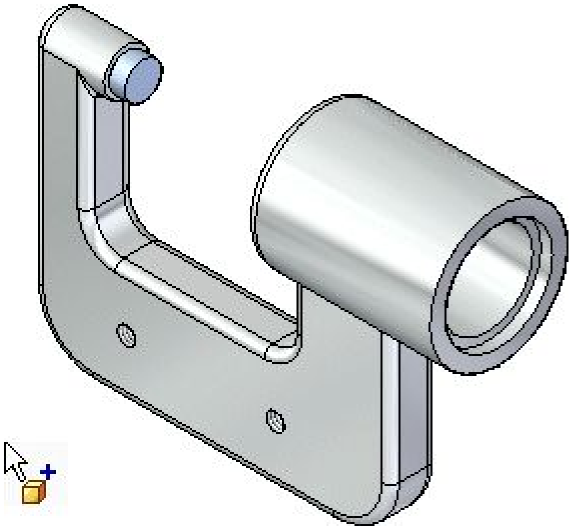
In the next few steps, place and position the nameplate part as shown.

For the remainder of this test drive, if a part is positioned incorrectly or confusion in the steps occurs while positioning a part, press Escape.

Then use the **Select** tool command on the **Home** tab to select the part, and press Delete to delete the part.

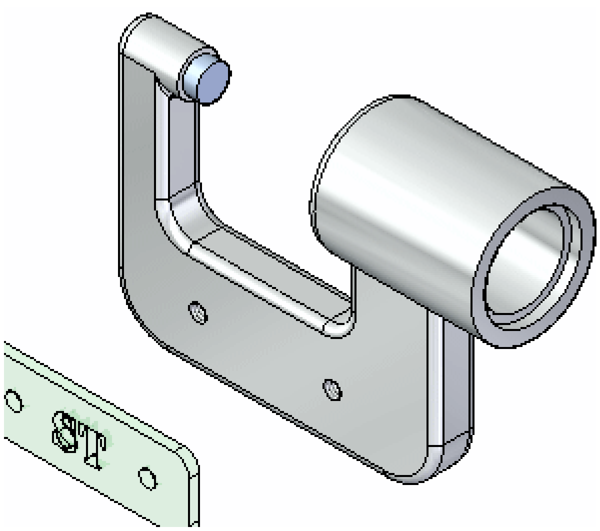
Back up to the step where part placement begins, and try again.

Place the name plate part

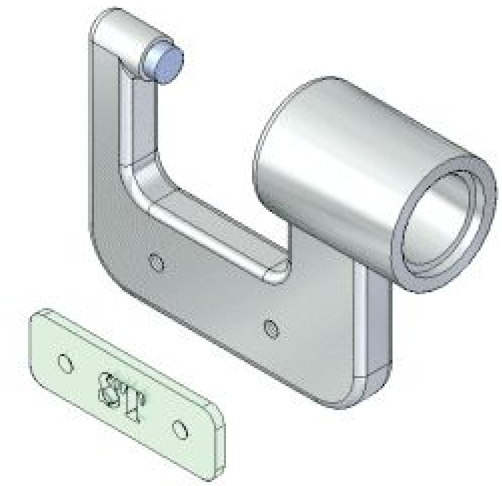


- ◻ Display the **Parts Library**.
- ◻ In the file list area on the **Parts Library** tab, select the file named **NamePlate1.par**, hold down the left mouse button, drag the file into the assembly window, and then release the mouse button at the approximate position shown above.

The nameplate is placed in the assembly, as shown below.

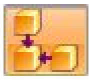


Examine the command bar



When placing the name plate part into the assembly, the command bar displays. It is shown horizontally here, but it may display vertically, depending on the user interface theme selected. In either case, the command bar displays the same information and options.

- Beginning at the left or top, examine the command bar, and notice the options:

 The **Assemble** command is active using the **FlashFit** placement option.

 The **Occurrence Properties** button displays the **Occurrence Properties** dialog box. Use this dialog box to define whether the part is displayed in higher level assemblies, counted in parts lists, and so forth.

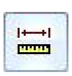
 The **Construction Display** button displays or hides elements for the part being placed, such as reference planes, sketches, and construction surfaces. This can make it easier to position certain types of parts.

The **Relationship List** displays the relationships used to position a part. When editing the position of a part after placement, select the relationship to redefine from the list.

 Use the **Relationship Types** option to select which assembly relationship option to use for positioning a part.

 The **Options** button displays the **Options** dialog box. Use this dialog box to set the **FlashFit** options, **Reduced Steps** option, and so forth.

 Use the **Activate Part** button to select a part and activate it. When placing a subassembly using **FlashFit** or the **Reduced Steps** mode, the parts in the subassembly must be active before selecting a face. If the subassembly is not already active, use the **Activate Part** button on the **Assemble** command bar to activate the placement part in the subassembly which contains the face to select.

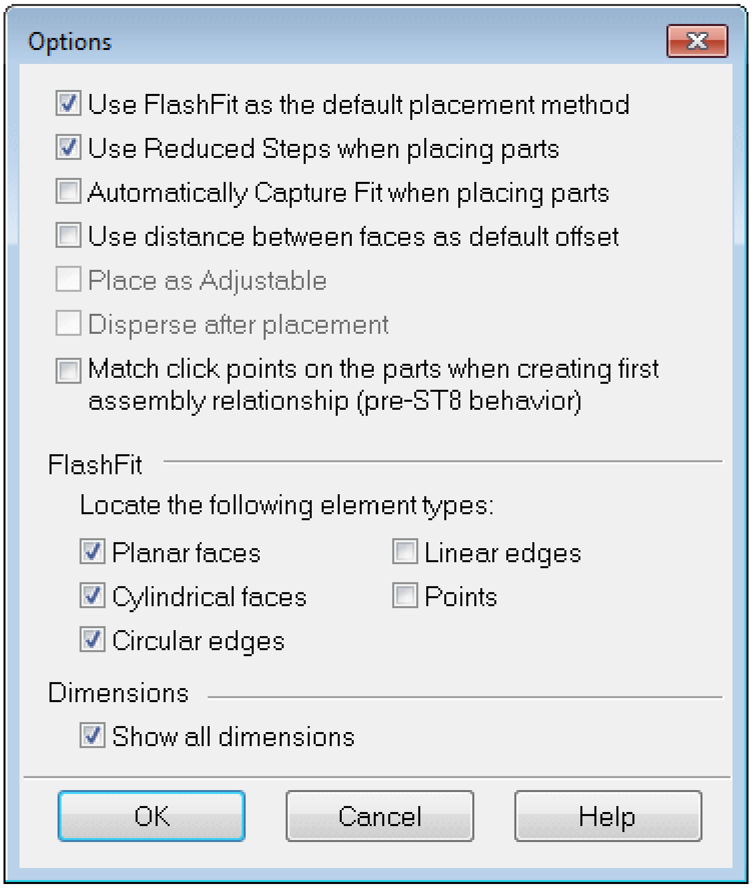
 Use the **Fixed Offset** button to define a fixed numeric offset value based on the relationship being defined.

Use the **Offset Value** box to enter the fixed offset value.

 The **Unlock Rotation** is set. With this option, use another assembly relationship to define the rotational orientation of the part. For example, apply an angle relationship. The **Lock Rotation** option fixes the rotational orientation of the part. This option is useful when the rotational orientation of the part is not important, such as for a bolt being positioned in a hole.

The **Flip** button repositions a part to the opposite side of a face, changing a mate relationship to a planar align relationship.

Review the part placement options



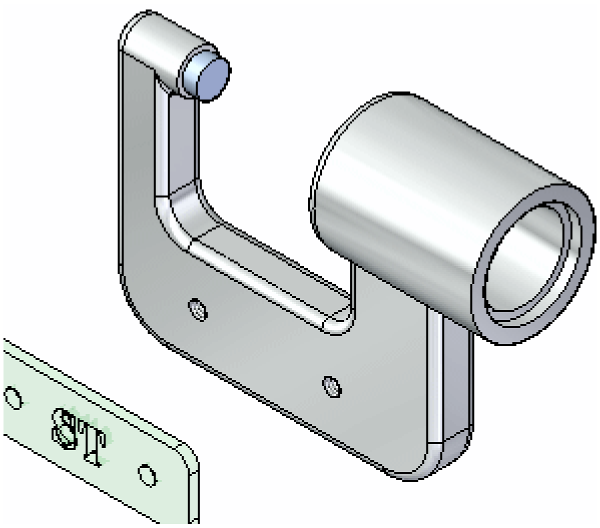
- On the **Assemble** command bar, click the **Options** button. 

- On the **Options** dialog box, ensure that the options match the illustration, and click **OK** to dismiss the dialog box.

Notice that the **FlashFit** option specifies what types of faces **FlashFit** recognizes.

For this test drive, and most part positioning scenarios, the **FlashFit** settings shown work well.

Mate the name plate to the frame

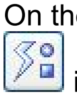


When selecting faces for the first assembly relationship, Solid Edge repositions the part being placed based on the approximate positions on the faces selected on the placement part and the part in the assembly.

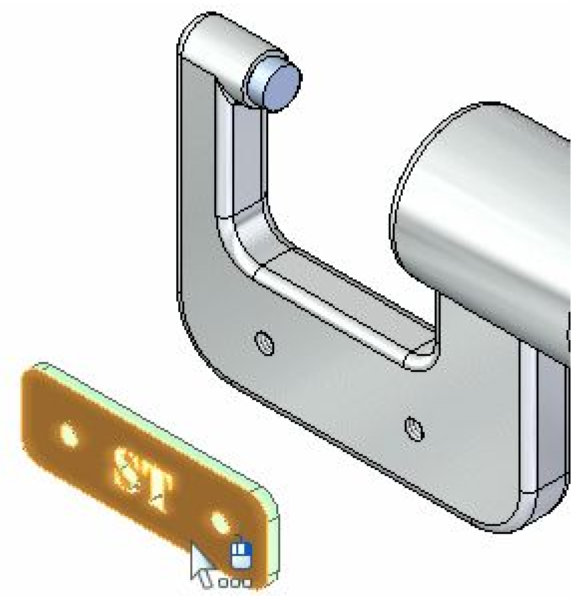
Use **FlashFit** to apply the first mate relationship.

A mate relationship positions a part by orienting two planar faces so that they face each other.

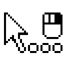
Mated faces can touch or be offset from each other. For this part, the default offset value of zero, where the parts touch, is the appropriate option.

- ☐ On the **Assemble** command bar, in the **Relationship Types** list, ensure that the **FlashFit** option  is active.

Use QuickPick to select the planar face on the name plate

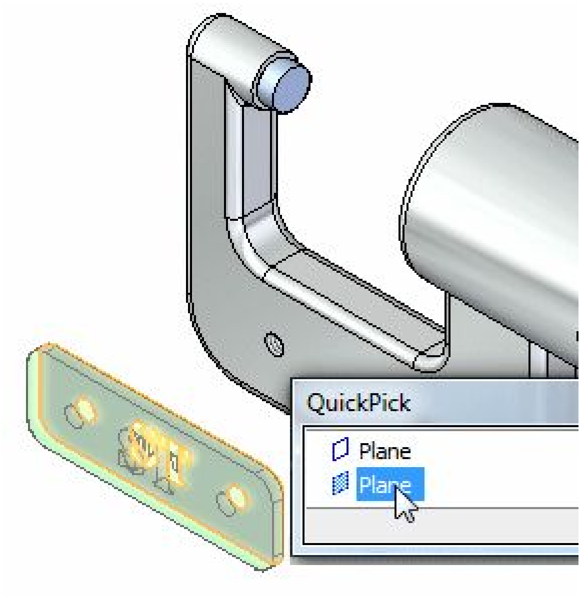


- ◻ Position the cursor over the face shown highlighted in the top illustration, pause the mouse, and notice that the cursor image changes to indicate that multiple selections are available. Also notice that the cursor image indicates which button to click to display the **QuickPick** list. The default is to right-click to display **QuickPick**.

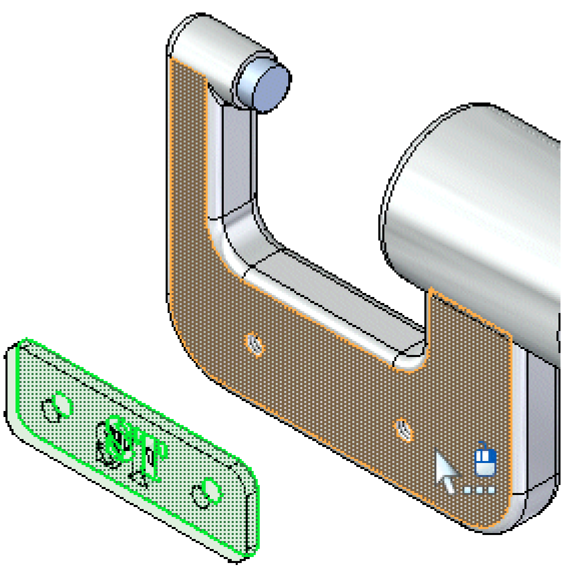


- Right-click, and the **QuickPick** list displays. Move the cursor over the different entries in **QuickPick**, and notice that different elements of the model highlight. Use **QuickPick** to select the exact element wanted, the first time, without having to reject unwanted elements.

- Use **QuickPick** to highlight the planar face shown in the bottom illustration, and then left-click to select it.



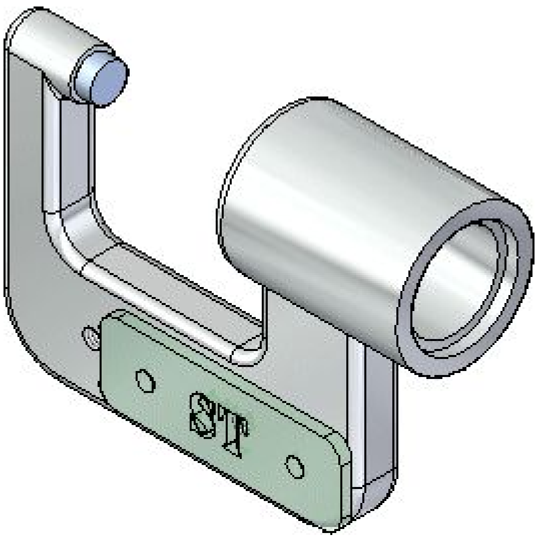
Select the mating face on the frame



If the **QuickPick** cursor displays, but the proper face highlights, bypass **QuickPick** by left-clicking.

- Select the front face of the frame part, as shown in the illustration.

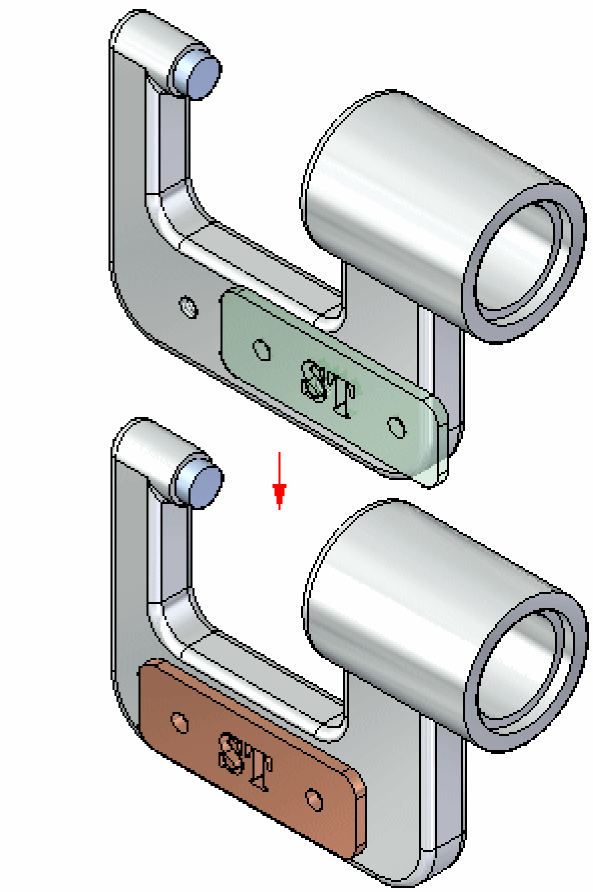
Observe the result



The mate relationship repositions the name plate in the assembly.

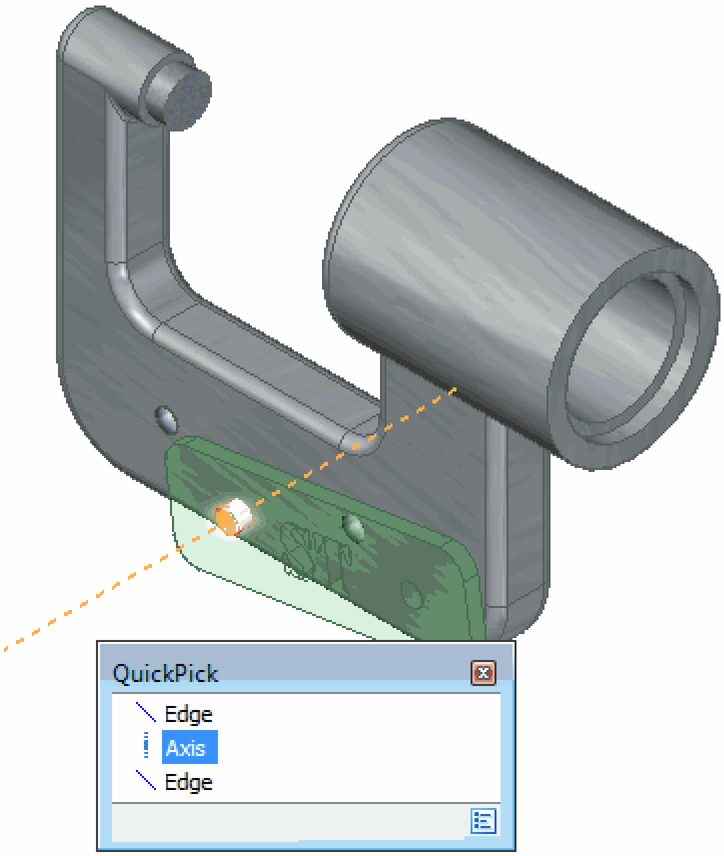
Because only one assembly relationship is applied, the position of the name plate might be different than the illustration.

Axially align the name plate with the frame



In the next few steps, use **FlashFit** to apply axial align relationships between the bolt holes on the name plate with the bolt holes on the frame.

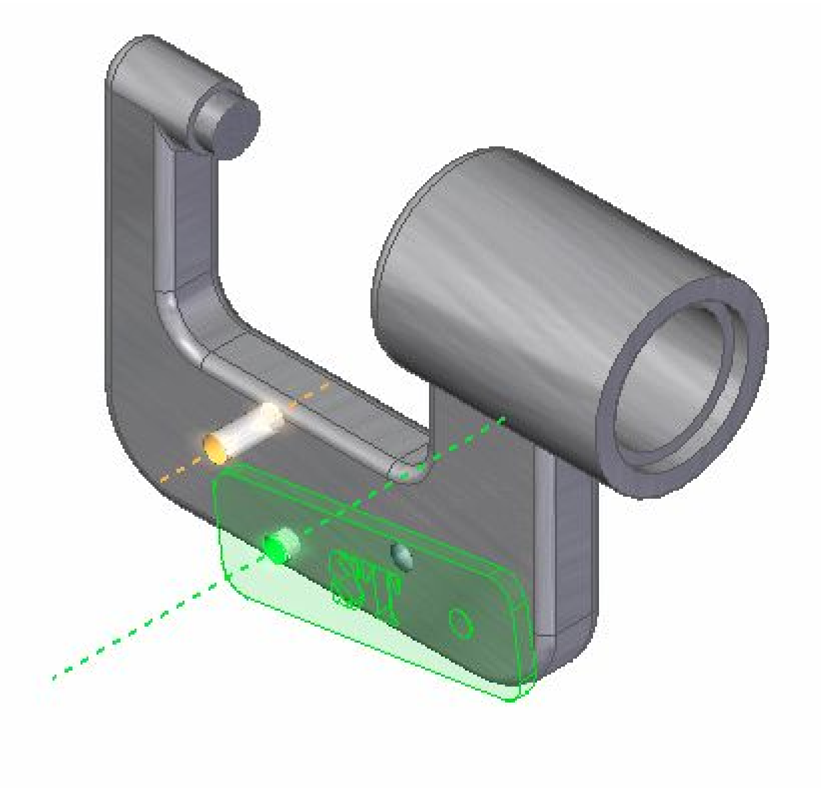
Select the cylindrical axis to align



- ☐ Use **QuickPick** to select the axis shown in the illustration.

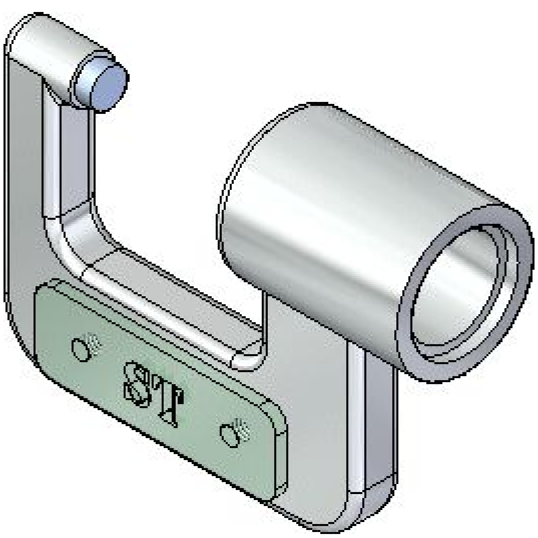
Align this cylindrical axis with the cylindrical axis on the frame.

Select the cylindrical axis on the frame



- Select the cylindrical axis on the frame part as shown in the illustration.

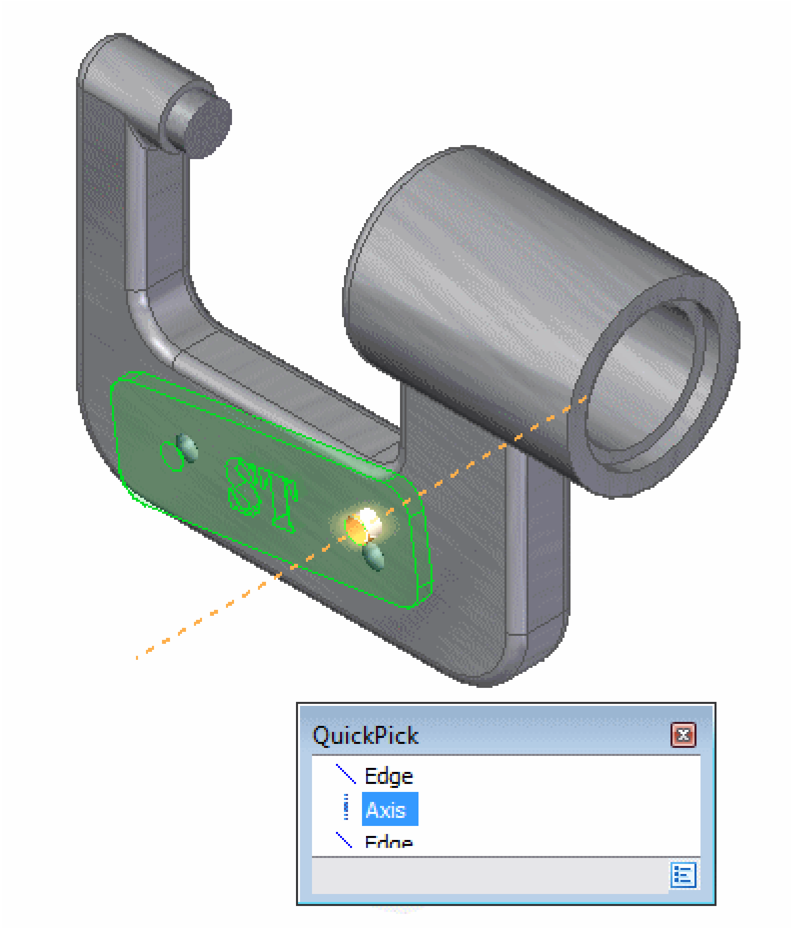
Observe the result



Although the name plate appears correctly positioned on the frame, no relationship prohibits the name plate from pivoting about the cylindrical axes just aligned.

In the next steps, apply another axial align relationship to fully position the name plate.

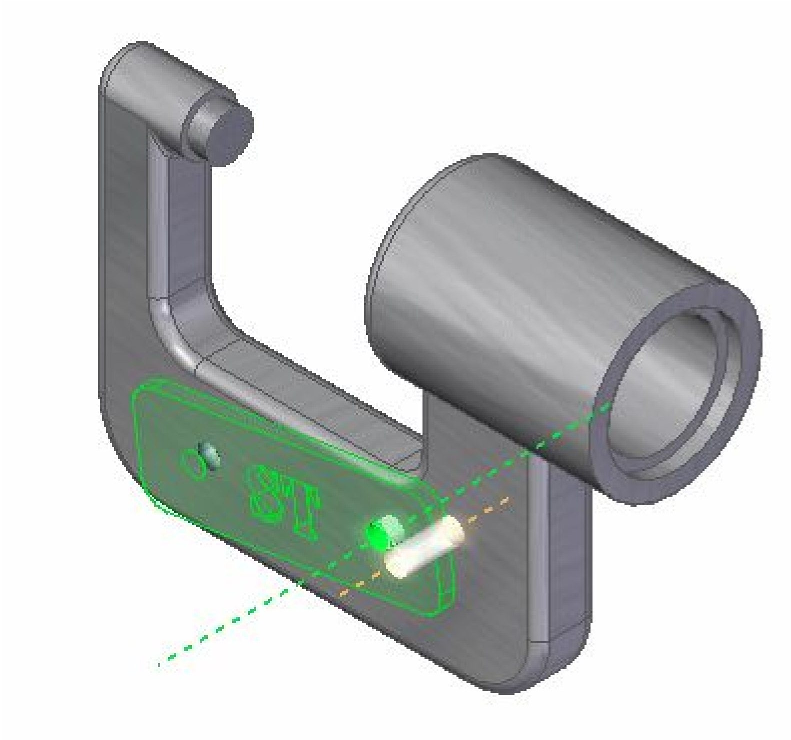
Select the cylindrical axis on the name plate



- ☐ Select the cylindrical axis on the name plate shown in the illustration.

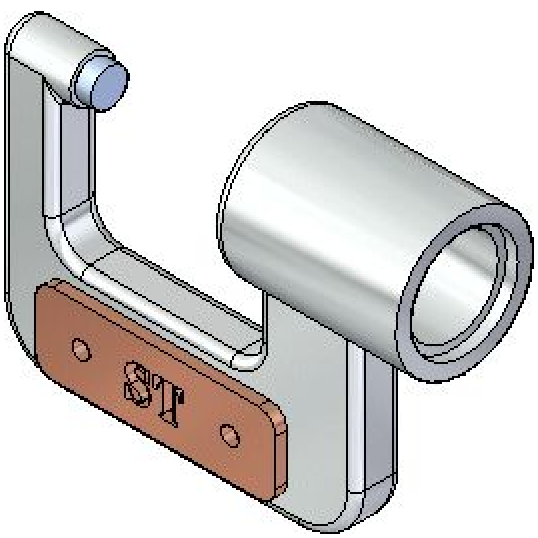
Depending on where the cursor is positioned, **QuickPick** may be needed to select the proper axis.

Select the cylindrical axis on the frame



- ☐ Select the cylindrical axis on the frame shown in the illustration.

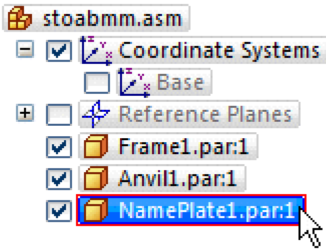
Observe the result



The name plate part is now fully positioned in the assembly.

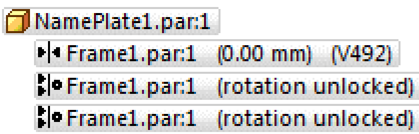
Notice that the **Assemble** command bar is dismissed, and the **Select** command bar displays instead.

Use PathFinder to review the assembly relationships

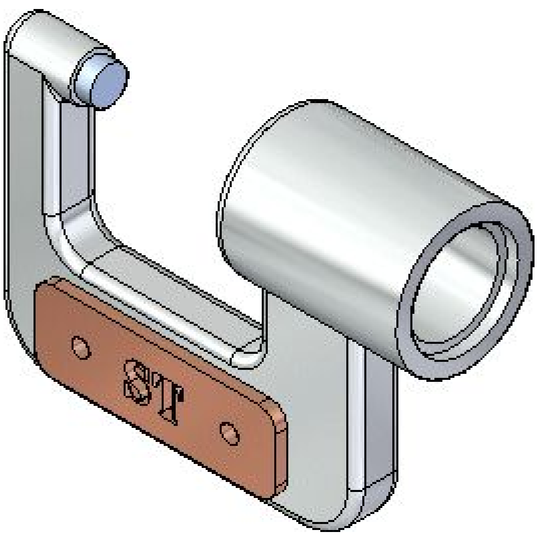


- ◻ In the top pane of **PathFinder**, click the NamePlate.par:1 entry.

Notice that the applied relationships display in the bottom pane of **PathFinder**.

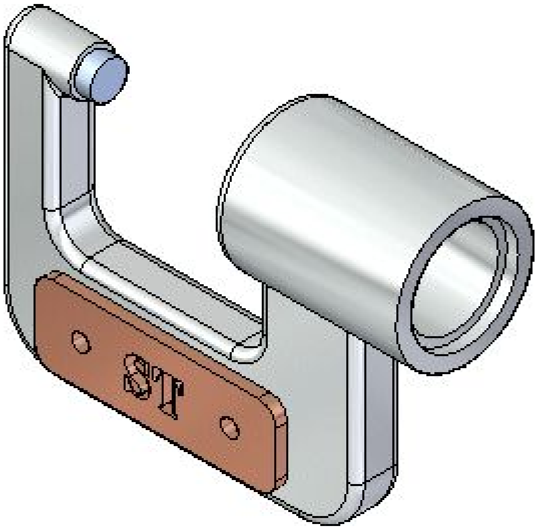


Save the assembly



- ◻ On the **Application** menu, click **Save As→Save As**.
- ◻ On the **Save As** dialog box, in the File box, save the assembly to a new name or location so that other users can complete this tutorial using the original assembly file.
- ◻ In the **Save As** dialog box, click the **Save** button.

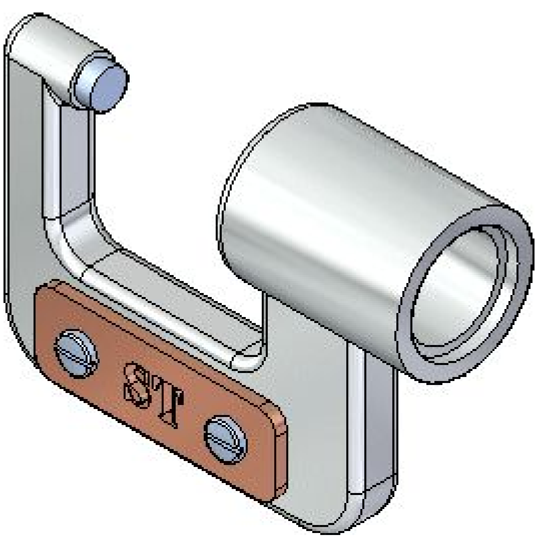
Positioning a part in an assembly complete



The adding a part to the assembly and positioning it with relationships portion of the testdrive is finished.

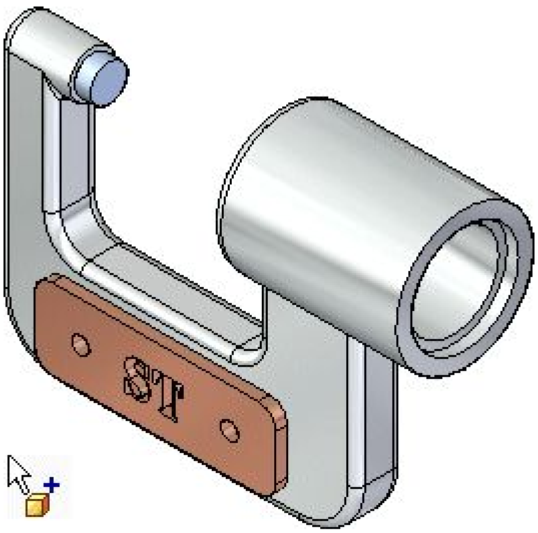
Place fastener components

Place fasteners



In the next few steps, place two screws that fasten the name plate to the frame into the assembly.

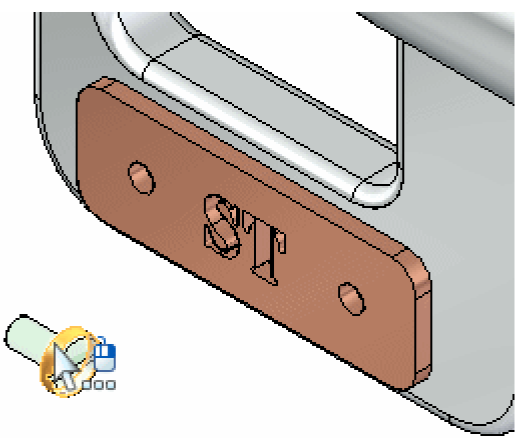
Place the first fastener



- Display the **Parts Library**.
- Drag **Screw2.par** from the **Parts Library** tab and drop it into the assembly.

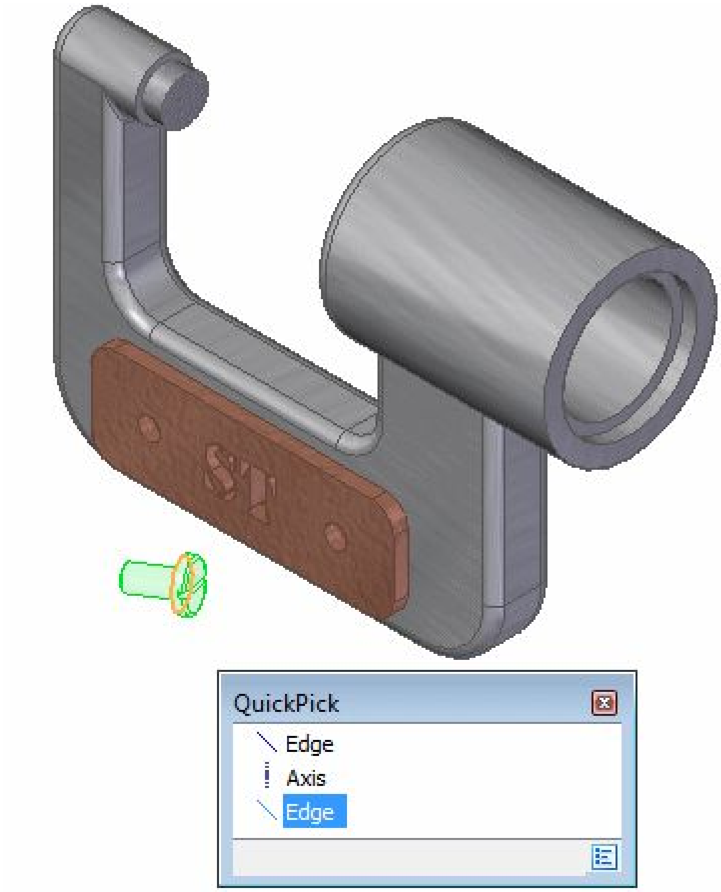
Because the axial orientation of fasteners is typically not important, use **FlashFit** to fully position the part by selecting a cylindrical edge on the fastener and the name plate.

Select the cylindrical edge on the fastener

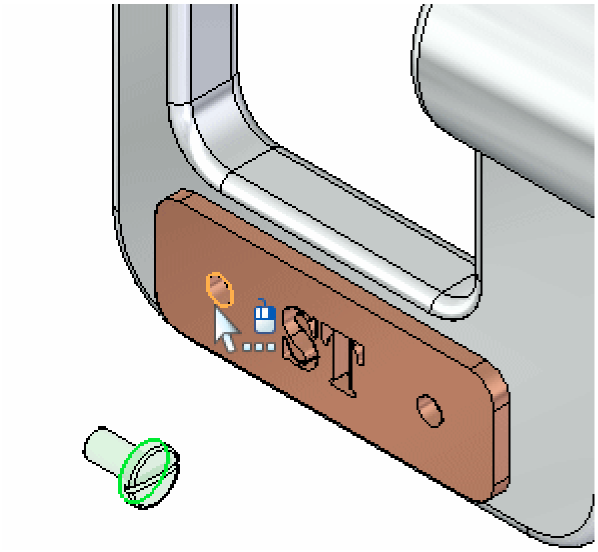


- Position the cursor over the fastener as shown in the top illustration, and wait for the **QuickPick** cursor to display.

□ Right-click, then use **QuickPick** to select the cylindrical edge on the fastener shown in the bottom illustration.



Select the cylindrical edge on the name plate

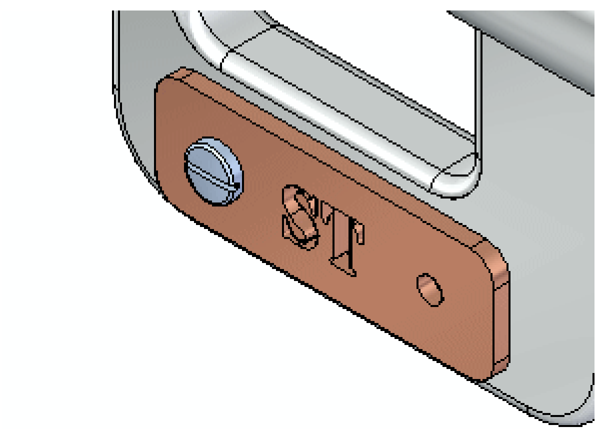


Because a cylindrical edge was selected in the previous step, Solid Edge filters the possible selections in this step to only cylindrical edges.

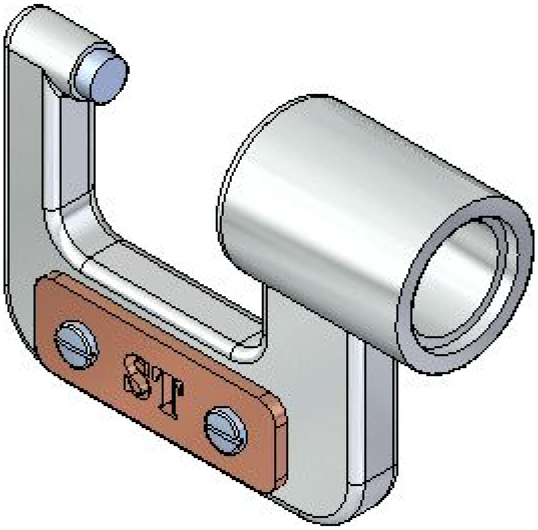
□ Select the cylindrical edge on the name plate shown in the illustration.

Based on how the cursor is positioned, **QuickPick** may or may not be available.

The fastener is fully positioned in the assembly, as shown below.



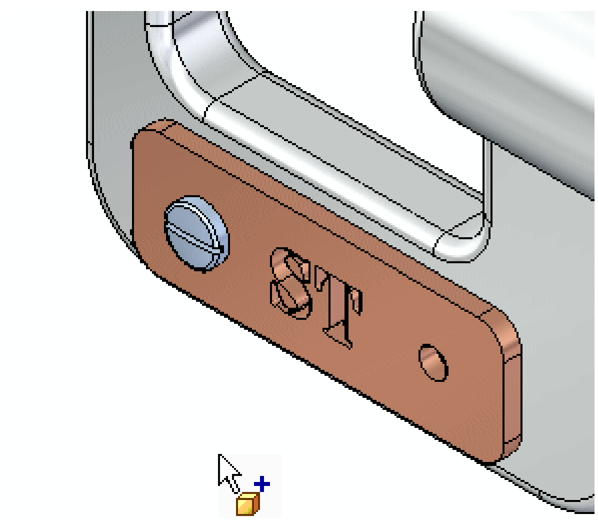
Prepare to place another fastener part



In the next few steps, place another fastener part on the other side of the nameplate.

Use the same steps you used to place the first fastener part.

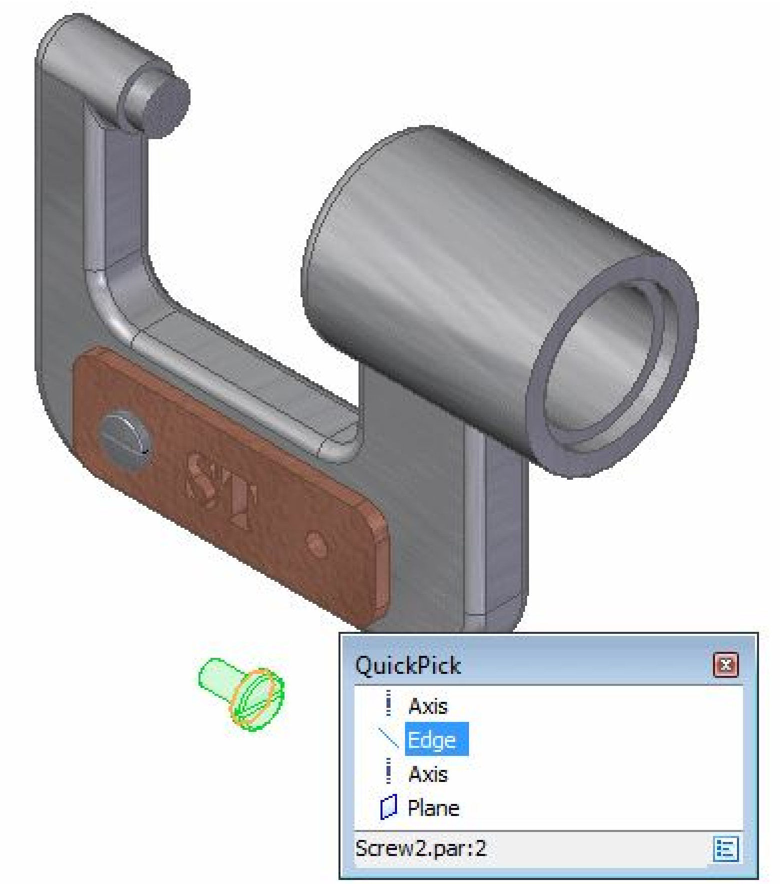
Place another fastener part in the assembly



- □ Drag another **Screw2.par** part from the **Parts Library** tab and drop it into the assembly at the approximate location shown.

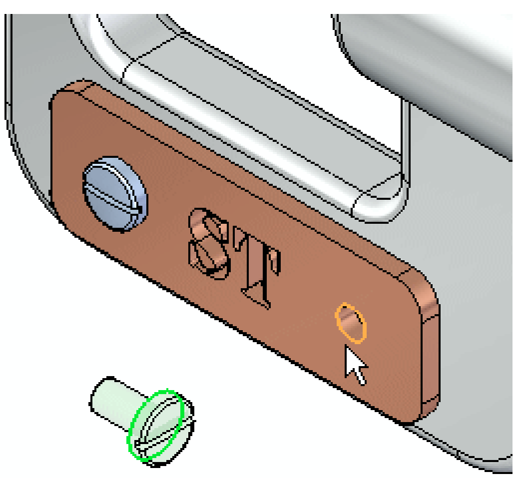
Use **FlashFit** again to fully position the fastener using cylindrical edges.

Select the cylindrical edge on the fastener



- ☐ Use QuickPick to select the cylindrical edge on the fastener shown in the illustration.

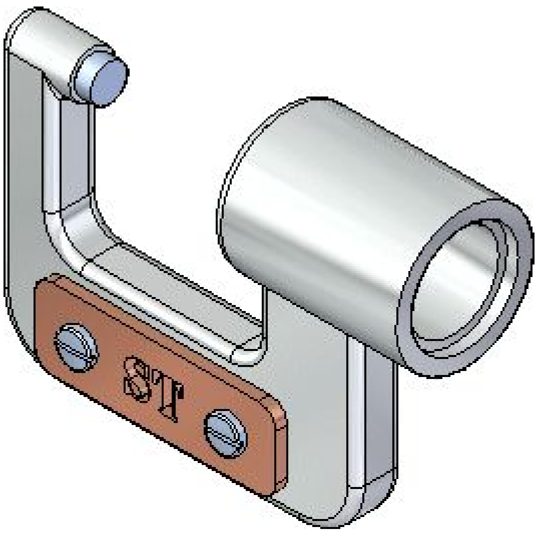
Select the cylindrical edge on the name plate



- ☐ Select the cylindrical edge on the name plate.

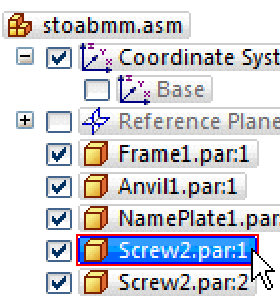
The fastener is fully positioned with respect to the name plate.

Observe the result



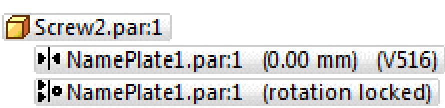
The second fastener is positioned in the assembly.

Use PathFinder to review the assembly relationships



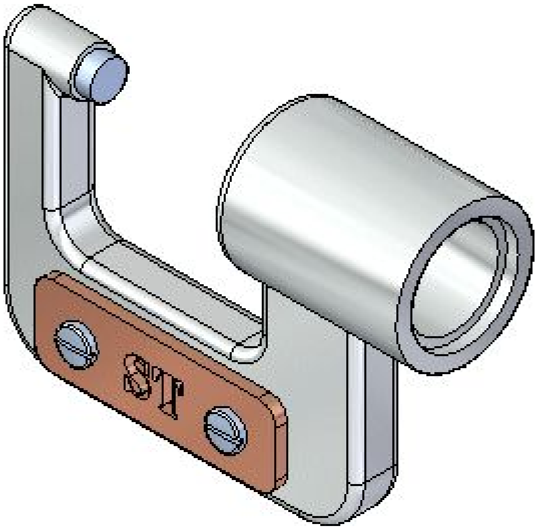
□ In the top pane of **PathFinder**, click the Screw2.par:1 entry.

Notice that the relationships applied display in the bottom pane of PathFinder.



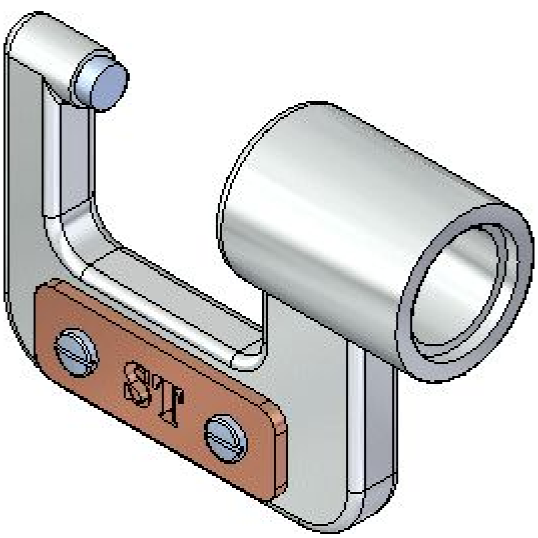
Notice that mate and align relationships were applied, similar to the name plate part. Although edges were selected rather than faces, Solid Edge determined which faces the edges belonged to, and applied the appropriate relationships.

Save the assembly



- On the **Quick Access** toolbar, choose **Save** .

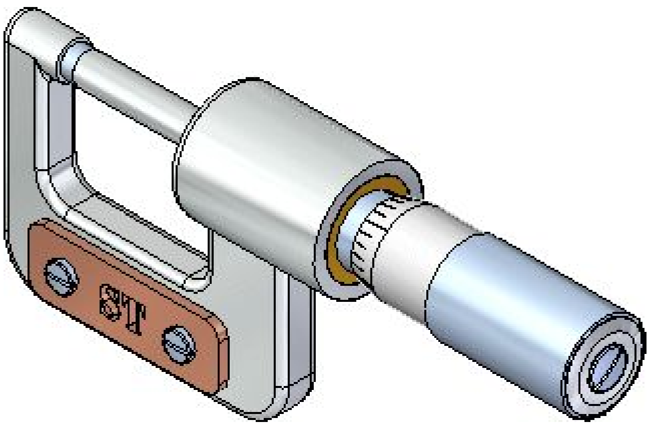
Fastener placement complete



You have finished placing the fasteners in the assembly.

Place a subassembly

Place the spindle subassembly



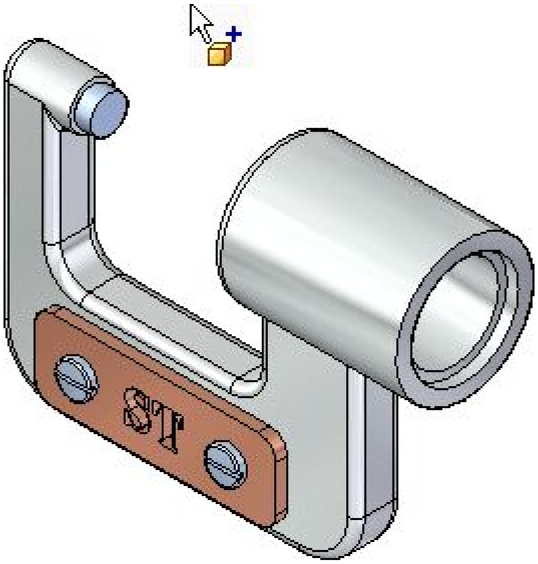
In the next few steps, complete the micrometer assembly by placing the spindle subassembly as shown.

When placing subassemblies, activate the parts in the subassembly to use to position the subassembly.

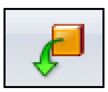
For this subassembly, use a mate relationship and an axial align relationship.

Also use an option available with the axial align relationship to eliminate the need for a third relationship.

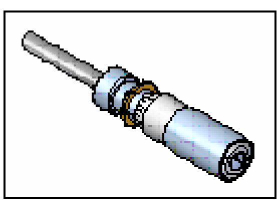
Place the spindle subassembly in the assembly



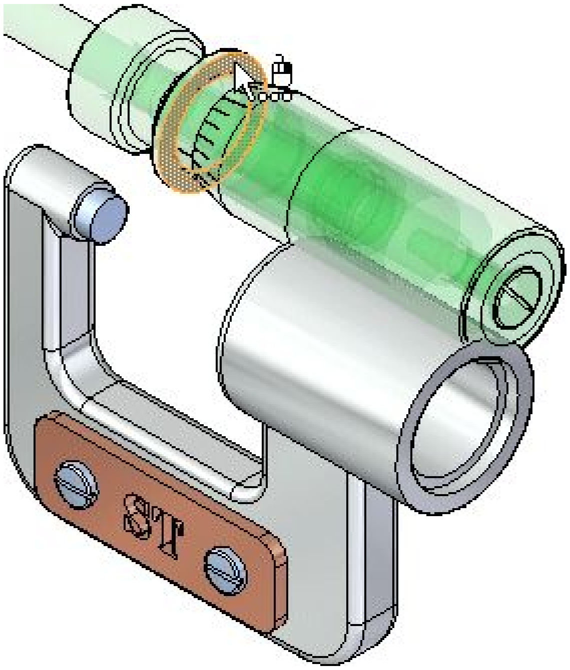
- ◻ Choose **Home** tab→**Assemble** group→**Insert Component** command.



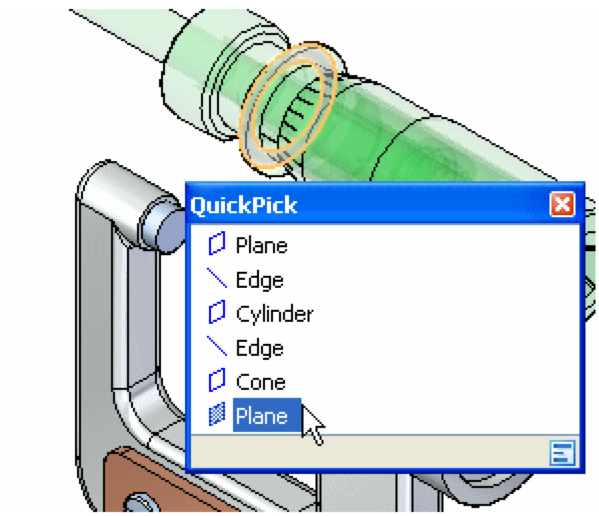
- ◻ In the file list area on the **Parts Library** tab, select the file named **SpindleSub1.asm**, hold down the left mouse button, drag the file into the assembly window, then release the mouse button.



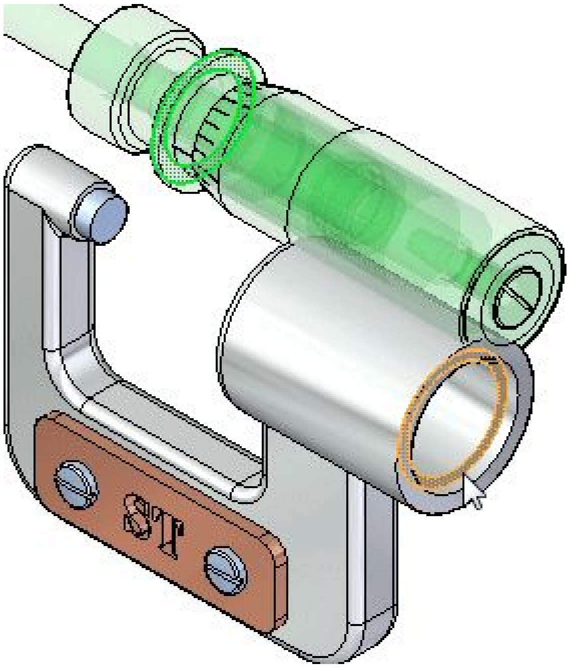
Select the face to mate on the plate part



- ☐ Position the cursor over the face shown in the top illustration, wait for the **QuickPick** cursor, then click to display **QuickPick**.
- ☐ Use **QuickPick** to select the planar face on the far side of the plate, as shown in the illustration below.

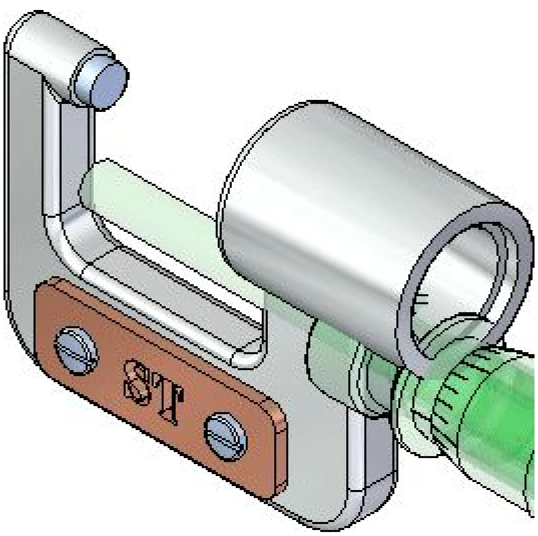


Select the mating face on the frame part



- ☐ Select the planar face on the frame, as shown in the illustration.

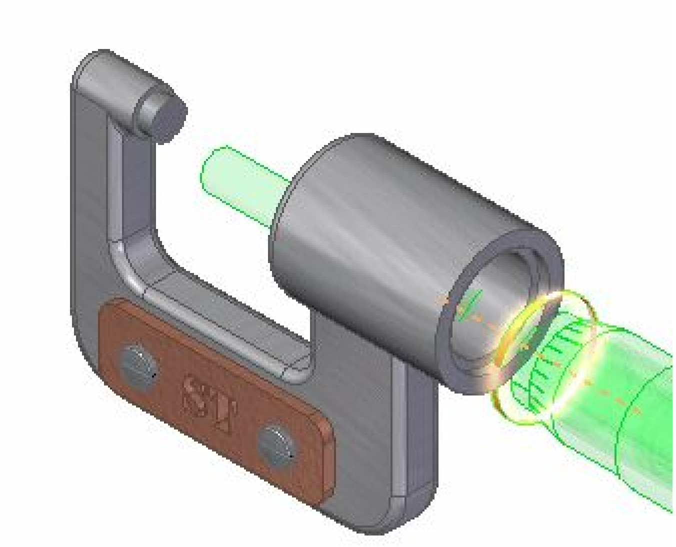
Observe the result



The mate relationship repositions the spindle subassembly in the assembly.

Because only one assembly relationship is applied, the position of the spindle subassembly might be different than the illustration.

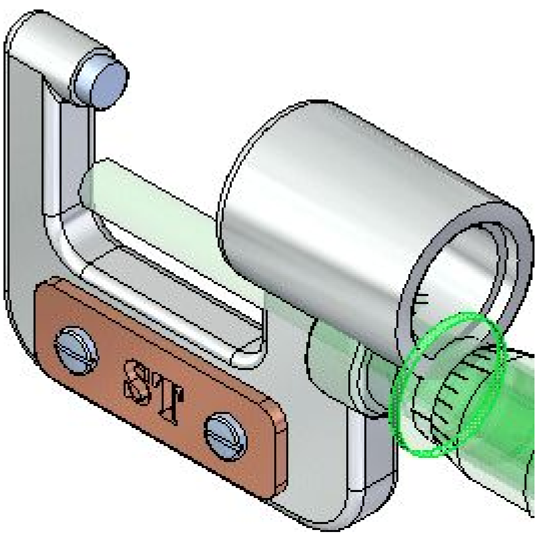
Select the cylindrical face to align on the plate part



□ Select the cylindrical axis on the plate.

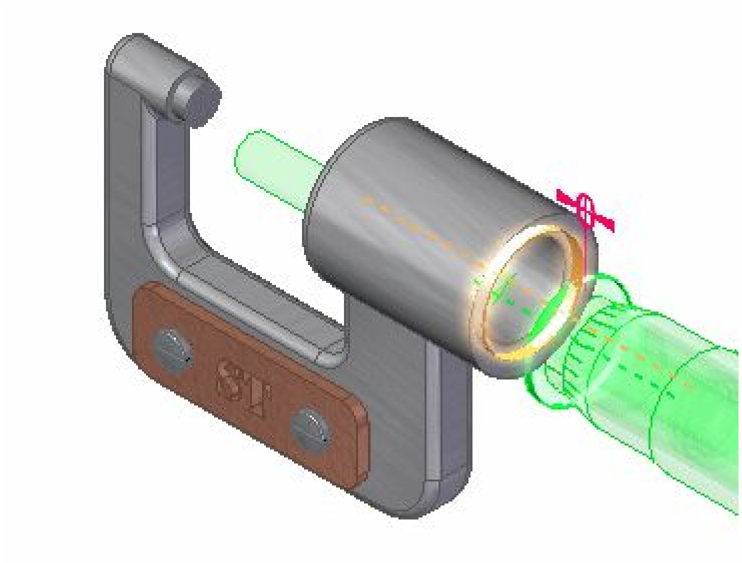
Depending on the position of the cursor, **QuickPick** may be needed.

Set the Lock Rotation option



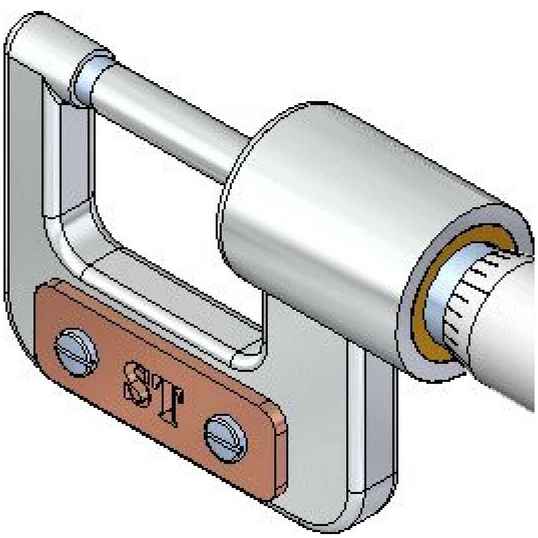
□ On the **Assemble** command bar, set the **Lock Rotation** option .

Select the cylindrical face on the frame part



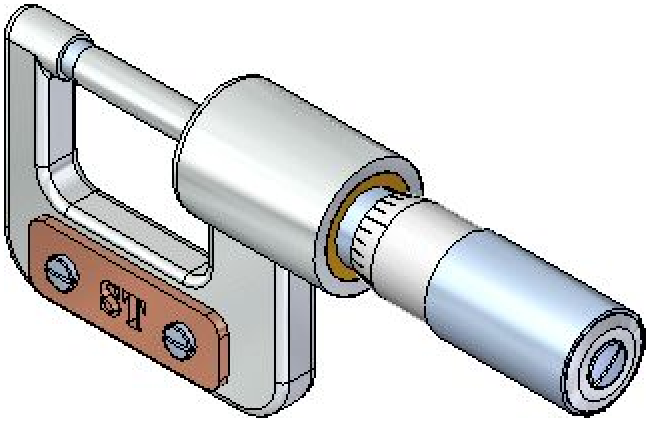
- ☐ Select the cylindrical axis on the frame shown in the illustration.

Observe the result



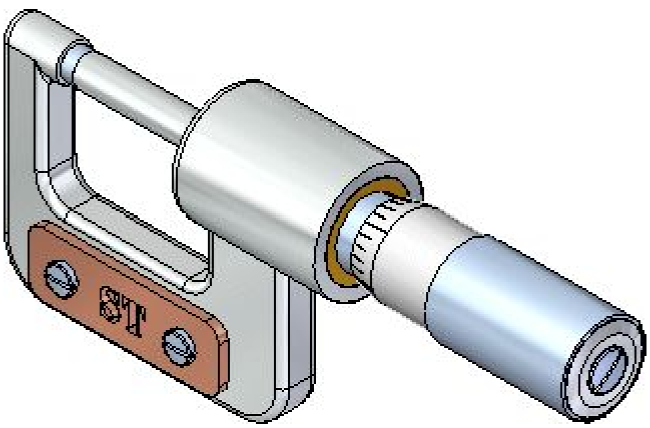
The spindle subassembly is now fully positioned in the assembly.

Fit the view



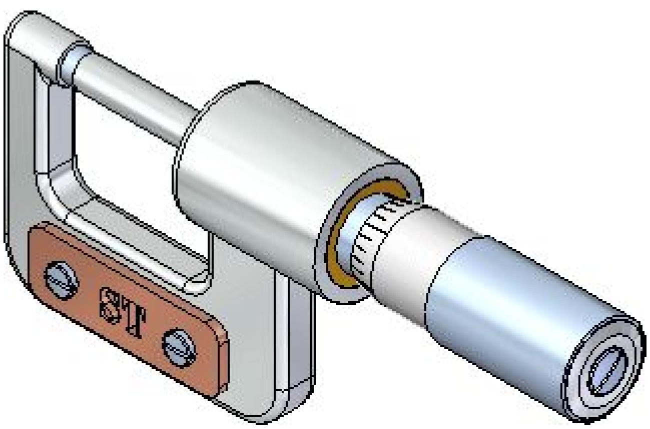
- □ Choose **Fit** to fit the contents of the view to the graphics window.

Save the assembly



- □ On the **Quick Access** toolbar, choose **Save**.

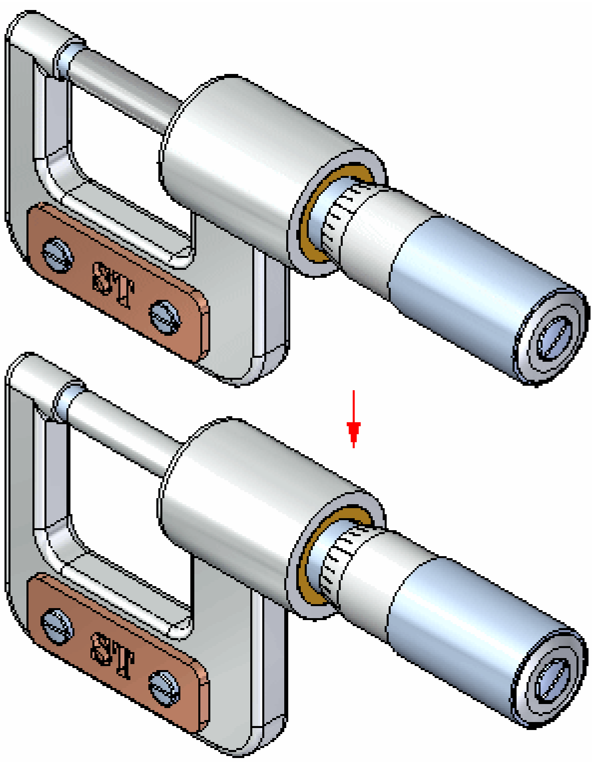
Subassembly placement complete



Placing the spindle subassembly in the caliper assembly is finished.

Edit a component in the assembly

Edit the frame in the context of the assembly



In the next few steps, edit the anvil end of the frame in the context of the assembly, as shown.

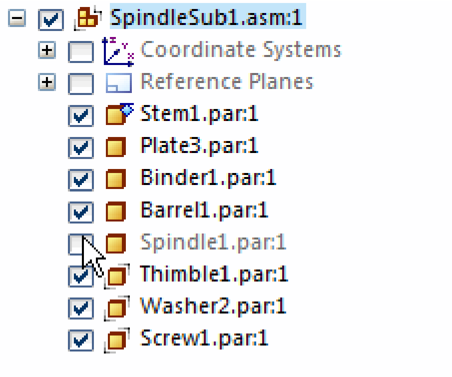
Also learn how to show and hide parts using **PathFinder**.

Hide the spindle part

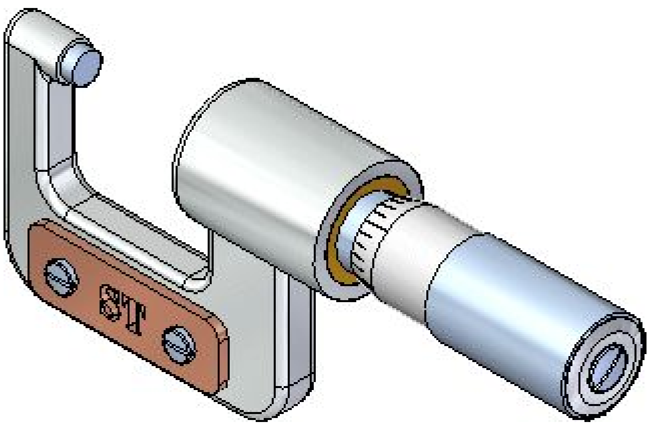
□ The **Select** command should be active. If not, press Escape.

□ In **PathFinder**, click the + symbol to expand the **SpindleSub1.asm** entry to display the list of parts in the subassembly.

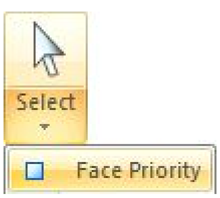
- ☐ Position the cursor over the check box adjacent to the **Spindle1.par** entry, then click to clear the check box.



In the graphics window, notice that the spindle part is hidden.



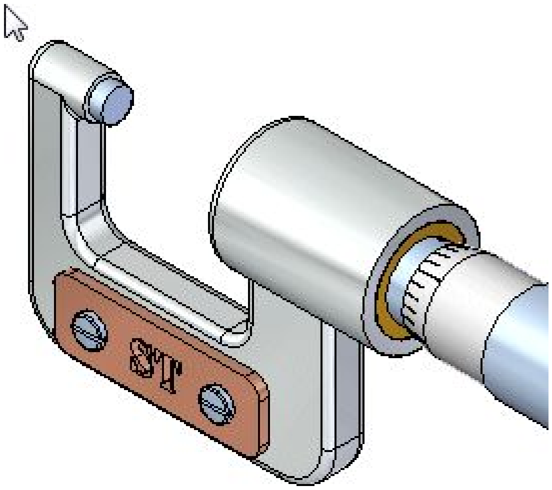
Change the Selection priority options



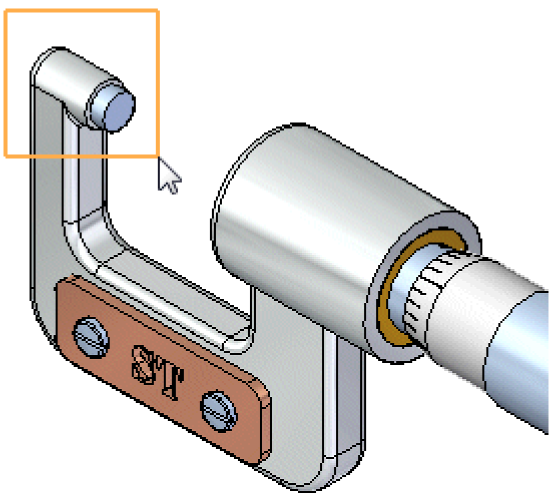
- ☐ Choose **Home** tab→**Select**→**Select Options** (the arrow pointing down, below the Select button)→**Face Priority** option.

The **Face Priority** option makes it possible to select faces before parts. This is a useful option when editing a model by moving faces with the steering wheel.

Select a set of faces by dragging a fence

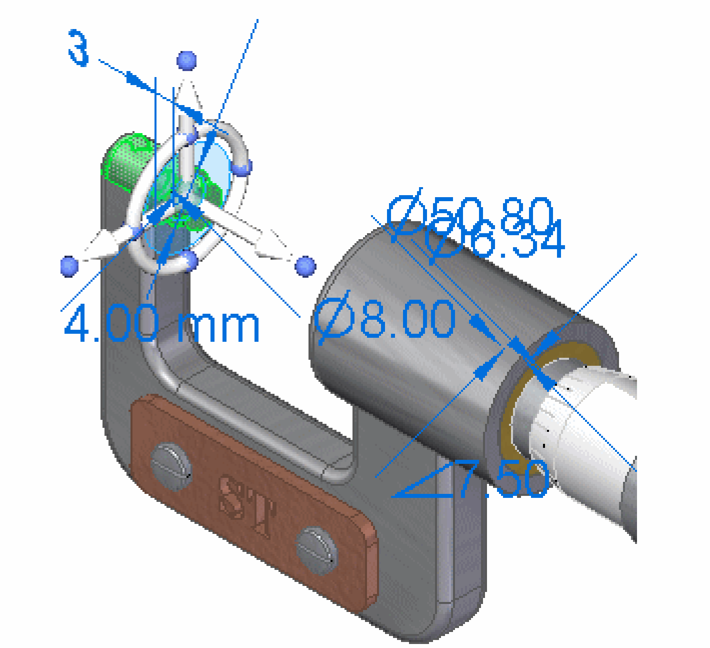


- ◻ Position the cursor at the approximate location shown in the top illustration.
- ◻ Press and hold the left mouse button down, then drag the cursor to the approximate location as shown below.



The steering wheel, **Move command bar**, and the Design Intent panel display.

Observe the steering wheel, command bar, and Design Intent



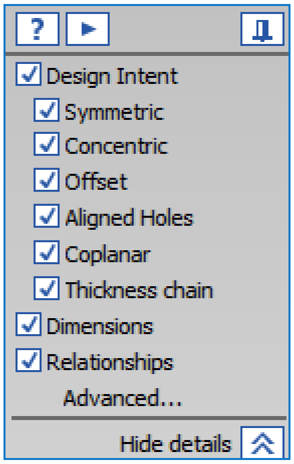
Notice the new tools that display when selecting the face:

- The steering wheel displays at the location on which the face was selected.
- The **Move** command bar displays.
- The Design Intent panel displays.

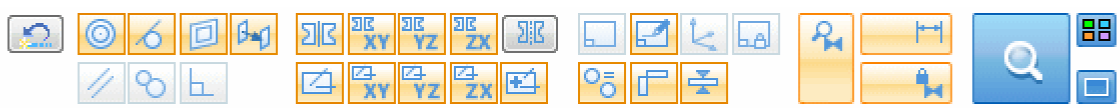
Learn more about these tools in the next few steps.

Design Intent overview

The Design Intent panel automatically appears when moving faces, defining 3D relationships, or editing dimensions in a synchronous model. The active options in the Design Intent panel and **Advanced** Design Intent panel are used to control how much of the design intent that has been built into the model to preserve or ignore when you move a face using the steering wheel.



Design Intent panel



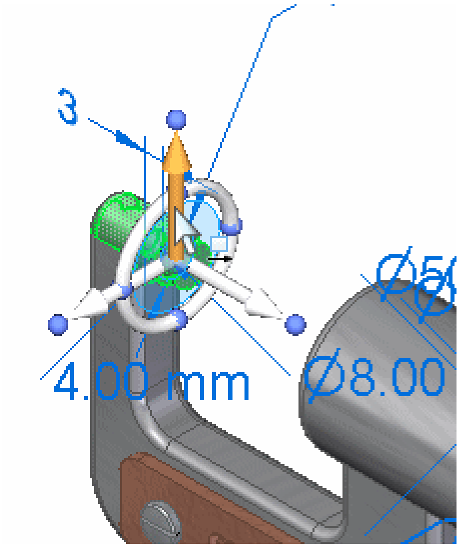
Advanced Design Intent panel

Depending on the current computer configuration, the Design Intent relationships settings may be different than the illustration.

- In the **Advanced** Design Intent panel, click the **Restore Defaults** button .

Design Intent relationship settings should now match the illustration.

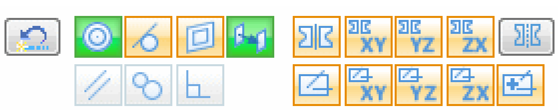
Select the vertical axis on the steering wheel



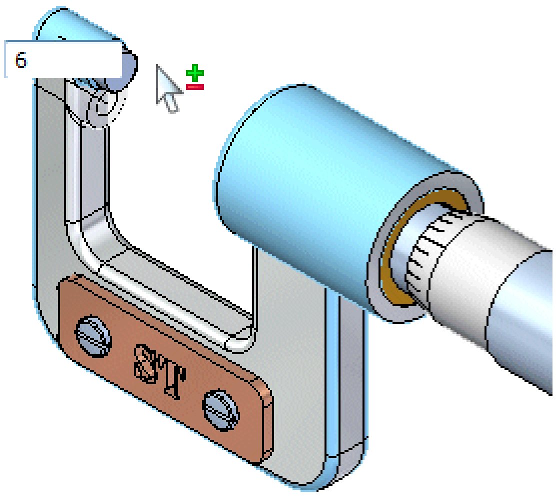
- ☐ Position the cursor ove6r the vertical axis on the steering wheel, and when it highlights, click to select it, as shown above.
- ☐ Move the cursor above the model vertically.

Notice the following:

- The dynamic input box displays near the cursor in order to enter a precise value.
- The right end of the frame part moves vertically along with the selected faces.
- The unselected parts in the assembly also move vertically.
- Several Design Intent Advanced options now glow green, as shown below.

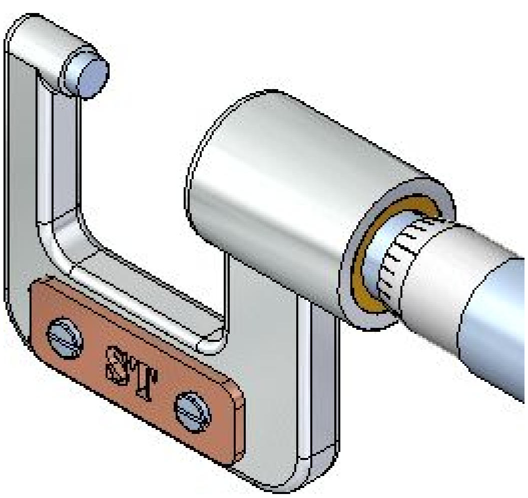


Finish moving the faces

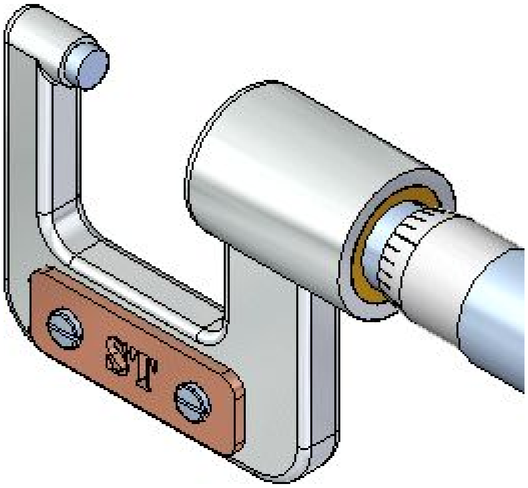


- Move the cursor above the selected face set until the value in the dynamic input box is about 6 mm.
- In the dynamic input box, type **6**, then press Enter.
- Press Escape to deselect the faces.

The assembly updates as shown below.

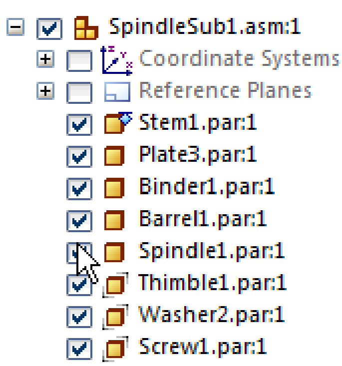


Observe the result



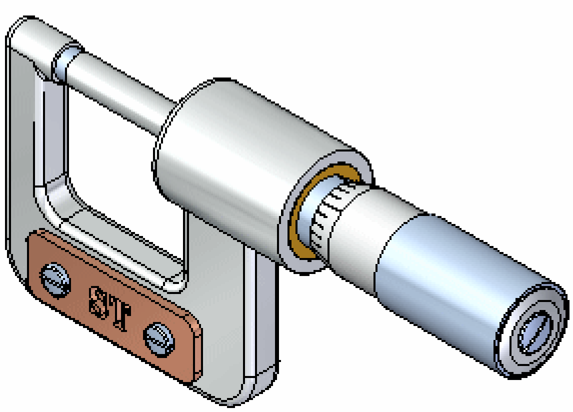
Notice that the selected faces on the frame, the anvil part, and the unselected parts all update their positions.

Display the spindle part

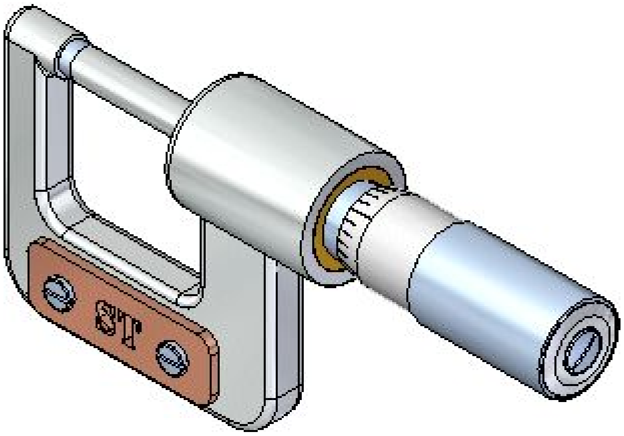


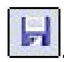
- □ In **PathFinder**, position the cursor over the check box adjacent to the **Spindle1.par** entry, then click to redisplay the spindle part.

In the graphics window, notice that the spindle part displays again.

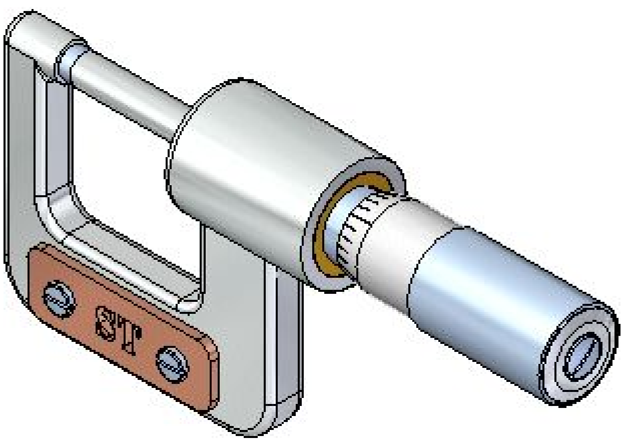


Save the assembly



- On the **Quick Access** toolbar, choose **Save** .

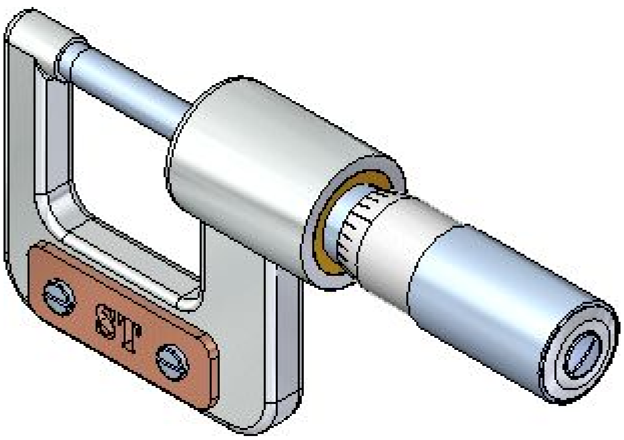
Component edit complete



The modifying the assembly by moving faces within a part in the assembly portion of the testdrive is finished.

Edit a component properties

Edit the properties for a part in the assembly



In the next few steps, update the material properties for a part in the assembly using Property Manager. Because material properties are tied to color display in Solid Edge, the color of the part will also update.

Use **Property Manager** to modify the existing properties or create new properties for one or more Solid Edge documents. Use **Property Manager** to edit the properties for the active document, a group of documents defined, or all the documents used in an assembly or assembly drawing.

Display the Property Manager dialog box

- □ On the **Application** menu, click **Info→Property Manager**.

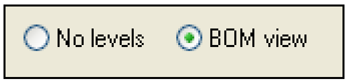
The **Property Manager** dialog box displays.

Observe Property Manager

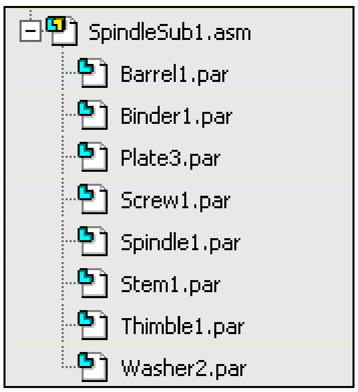
- ☐ Observe the various property categories.

Depending on the current computer settings, **Property Manager** may display using the *No levels* or *BOM view* options.

- ☐ In **Property Manager**, set the **BOM view** option.



- ☐ In **Property Manager**, click the + symbol adjacent to the **SpindleSub1.asm** entry to display the list of parts in this subassembly.



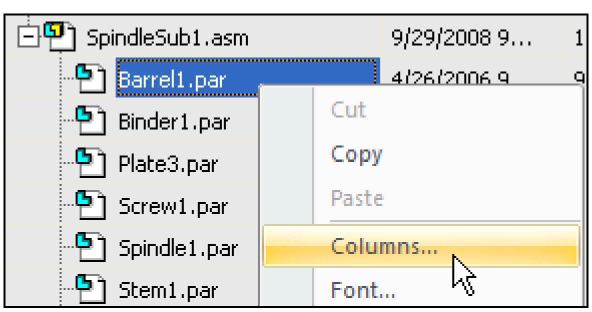
The display should now match the illustration above.

Add the Material property column

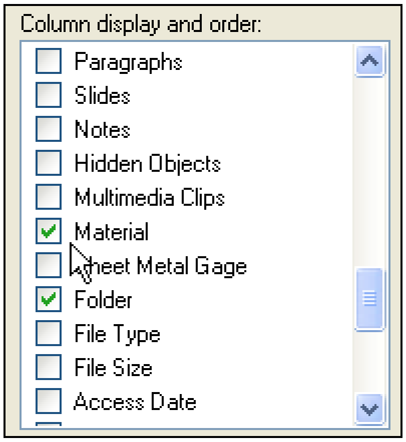
By default, the **Material** property column does not display. Customize **Property Manager** to display the property columns to use. The column order can also be customized.

- ☐ Position the cursor over the **Property Manager** dialog box, then right-click to display the shortcut menu.

- ☐ On the shortcut menu, click **Columns** to display the **Format Columns** dialog box.

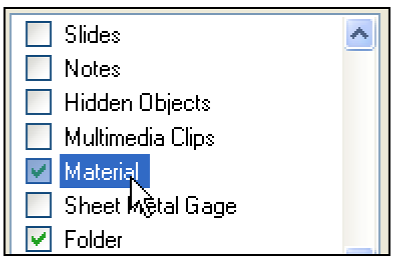


- In the *Column display and order* list, scroll down, then click the check box adjacent to the **Material** property entry.



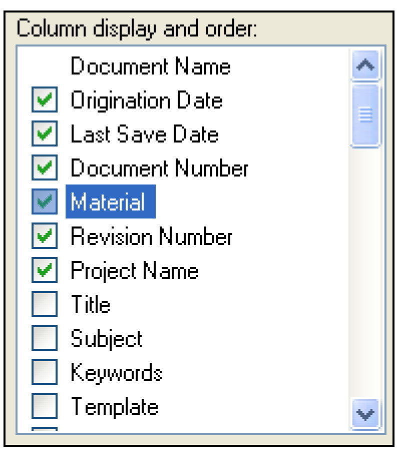
Change the display order for the Material property column

- Position the cursor over the text for the **Material** property, then click to select it.



- Click the **Up** button 

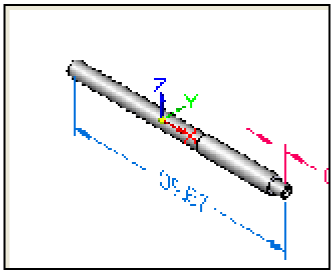
 and use the scroll bar to move the **Material** property up the list until it is near the top of the list, just below the **Document Number** column entry as shown below.



- On the **Format Columns** dialog box, click **OK** to save the changes.

Modify the Material properties for a part

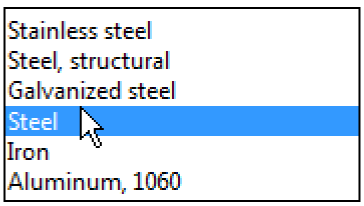
- In **Property Manager**, click the Material cell for the **Spindle1.par** part. The material for the spindle part is currently set to aluminum.



Notice that when selecting the **Material** cell, a preview window of the part automatically displays.

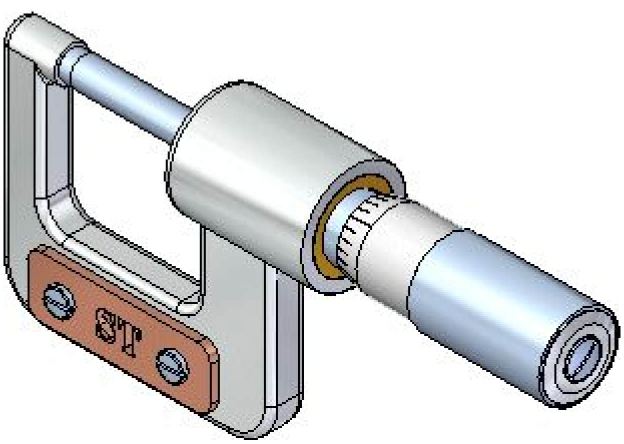
- In the **Material** cell, click the arrow to display the list, then click the **Steel** option.

- On the **Property Manager** dialog box, click **OK**.



Notice that the material color for the part updated. Later in the tutorial, place a parts list for the assembly where the material property will be used.

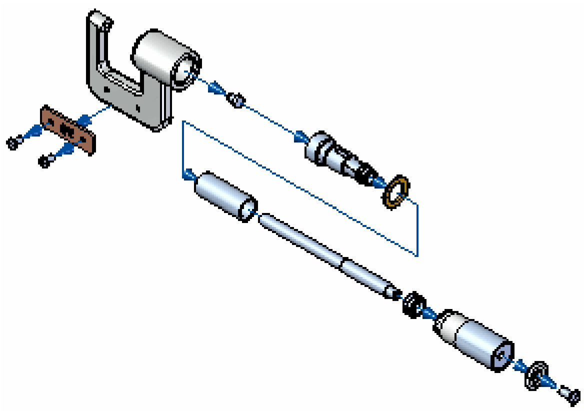
Edit component properties complete



Editing the properties for a part in the assembly is finished.

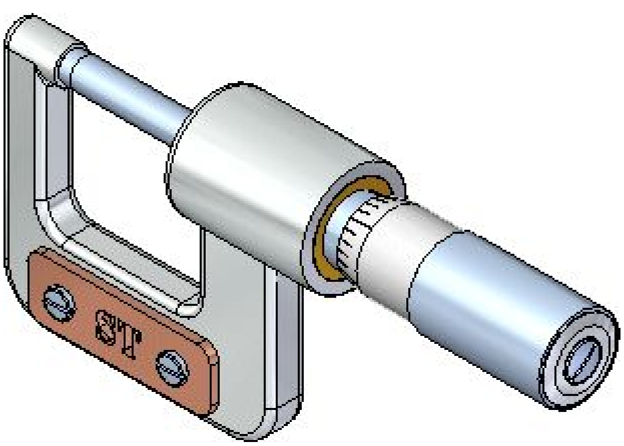
Create an exploded assembly view

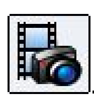
Create an exploded view of the assembly



In the next few steps, create an exploded view of the assembly.

Prepare to create an exploded view of the assembly

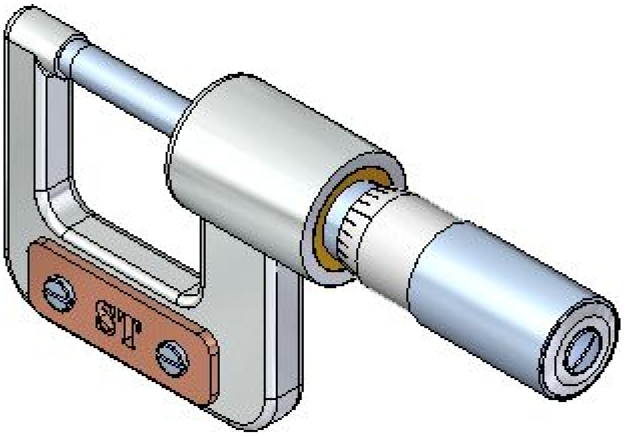


□ Choose **Tools** tab→**Environs** group→**ERA** .

The system displays menus and commands specifically tailored for creating exploded views, renderings, and animations of an assembly.

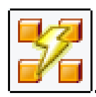
> If the **Explode Pathfinder** pane obscures the view, close it. It is not needed for this test drive.

Start the Automatic Explode command



Use the **Automatic Explode** command to begin defining the exploded assembly view.

The **Automatic Explode** command explodes assemblies based on the relationships applied between parts. In assemblies where the components are positioned using mate or axial align relationships, the Automatic Explode command quickly gives excellent results.

- ◻ Choose **Home** tab→**Explode** group→**Auto Explode** .
- ◻ On the **Automatic Explode** command bar, ensure that the **Select** option is set to **Top-Level Assembly**.

- ◻ On the command bar, click Accept .

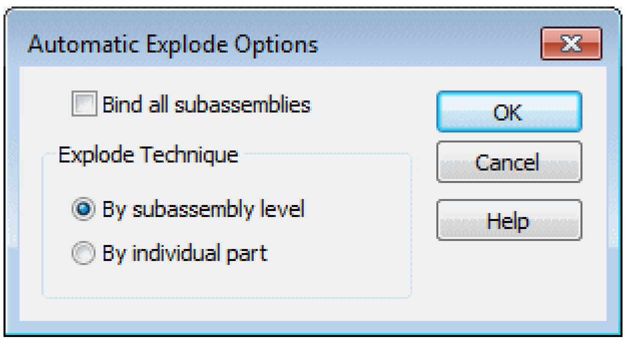
Set the Automatic Explode Options

When using the **Automatic Explode** command on an assembly that contains subassemblies, specify whether the parts in subassemblies are exploded or grouped together as a single unit.

For this assembly, explode the subassemblies also.

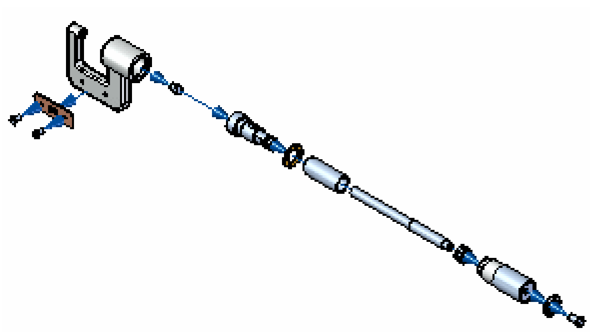
- ☐ On the **Automatic Explode** command bar, click the **Automatic Explode Options** button .

- ☐ On the **Automatic Explode Options** dialog box, clear the **Bind All Subassemblies** option. When clearing this option, the parts in subassemblies explode.



- ☐ Ensure the **By subassembly level** option is set as well, then click **OK**.

Create the automatic explosion



- ☐ On the **Auto Explode** command bar, click the **Explode** button.

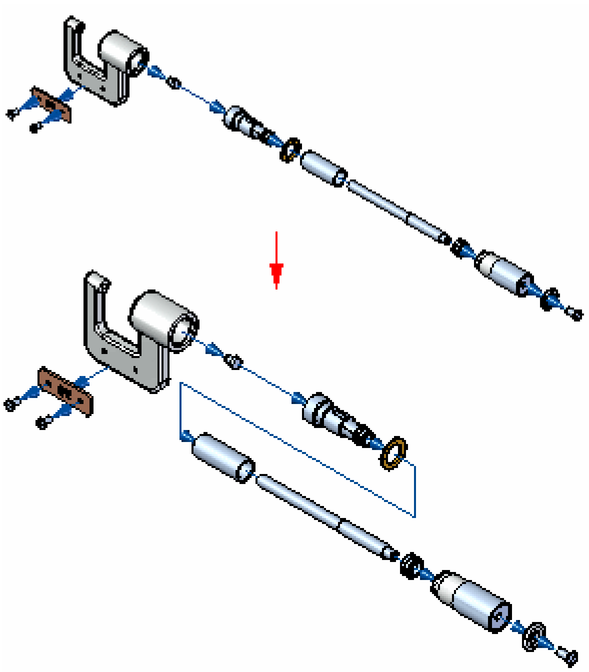
The system processes, and then displays the exploded view.

- ☐ On the **Auto Explode** command bar, click the **Finish** button.

Notice that the display is good, but that would be easier to visualize the exploded view on a drawing sheet if the exploded view was more compact.

In the following steps, use other commands to adjust the positions of the parts within the exploded view.

Prepare to move the parts

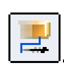


In the next few steps, use the **Move Part** command to move the parts shown to reduce the space requirements of the exploded view.

First move the set of parts downward along the Z axis, then move the set of parts to the left along the X axis.

Start the Drag Component command

Use the **Drag Component** command to move one or more parts in an exploded view along a specified axis.

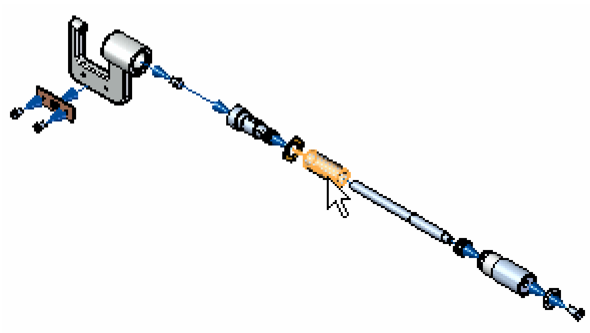
- Choose **Home** tab→**Modify** group→**Drag Component** .
- Examine the options on the **Drag Component** command bar.

Use the **Drag Component** command bar to specify whether to move only the selected part, or the selected part and all its dependent parts.

Also specify whether to move the parts linearly, rotate the parts, or move the parts about a plane.

This command cannot be used to move a part past an adjacent part.

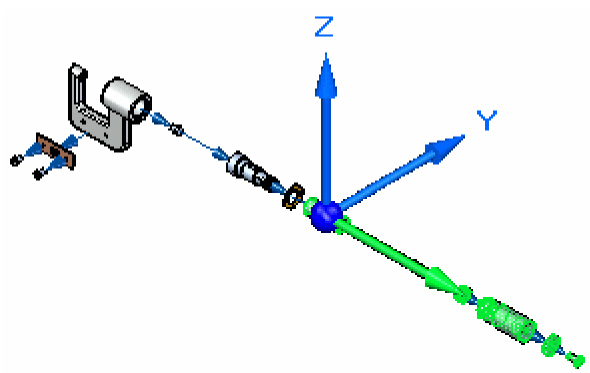
Select the parts to move



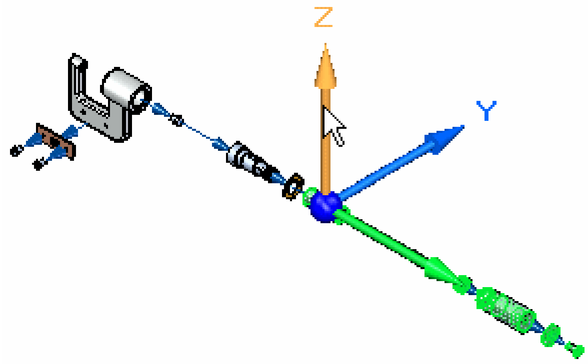
For this operation, move the parts as shown above.

- □ On the **Drag Component** command bar, ensure the **Move Dependent Parts** option is set.
- □ On the **Drag Component** command bar, ensure the **Move** option is set.
- □ Position the cursor over the **Barrel1.par** part, as shown above, then select it. Notice that all the parts to the right of the barrel are also selected.
- □ On the **Drag Component** command bar, click Accept, or right-click.

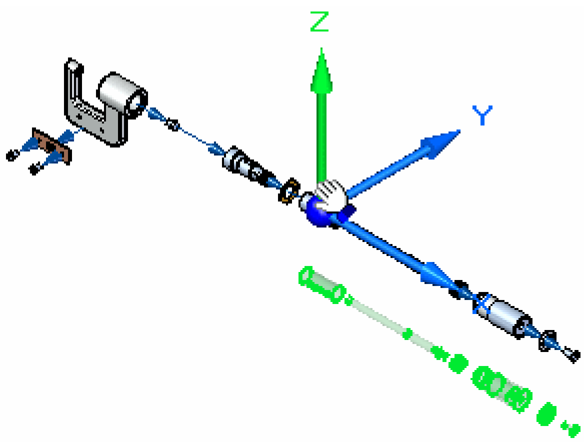
As shown below, notice that a triad displays, with the X axis highlighted. When moving parts, the X axis is automatically aligned with the original explode vector for the selected parts.

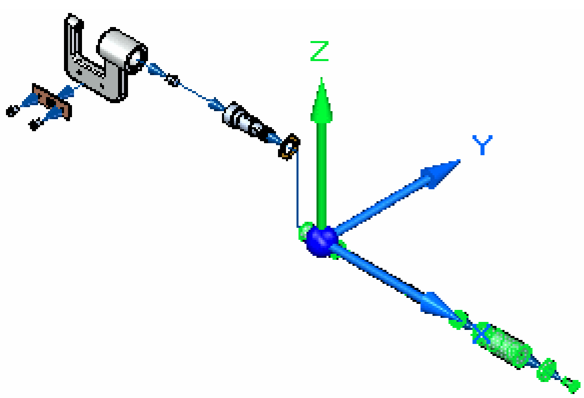


Move the parts along the Z axis

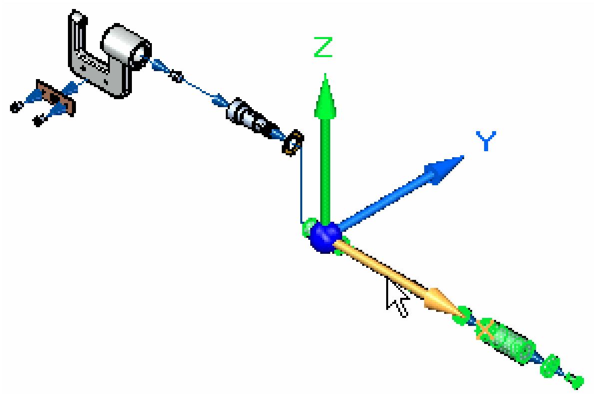


- ◻ Position the cursor over the Z axis, then select the Z axis.
- ◻ Press and hold the left mouse button down, then drag the cursor down as shown below, then release the mouse button to move the parts.

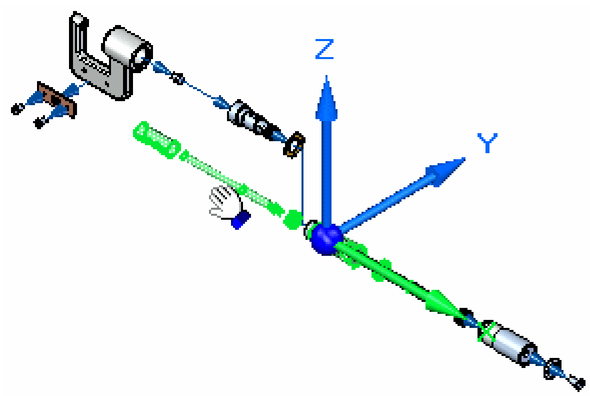


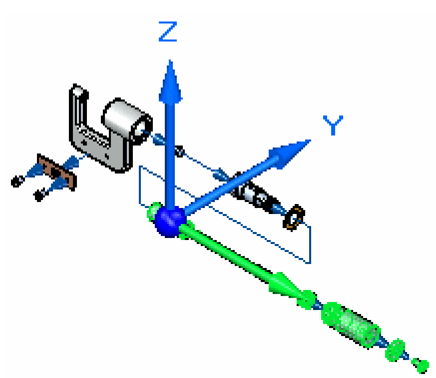


Move the parts along the X axis

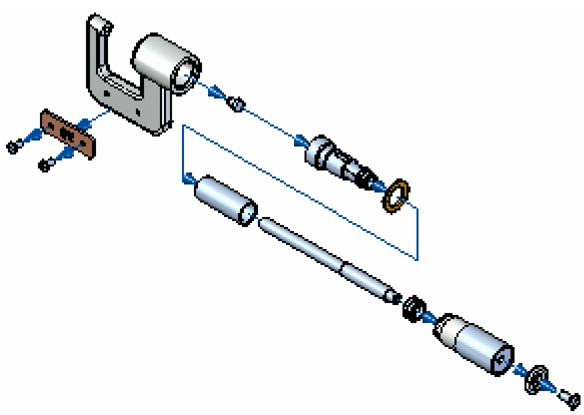


- Position the cursor over the X axis, then select the X axis.
- Press and hold the left mouse button down, then drag the cursor to the left as shown below, then release the mouse button to move the parts.
- Right-click to clear the selection and restart the command.





Fit the view

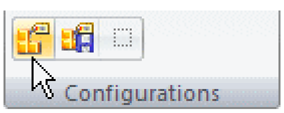


- Choose **Fit**  to fit the contents of the view to the graphics window.

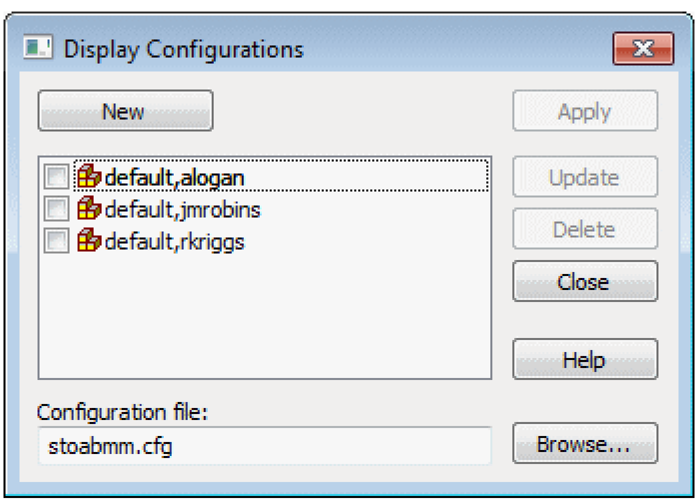
Save the display configuration

To use this exploded view in an assembly animation or in an assembly drawing, save the display configuration.

- Choose **Home** tab→**Configurations** group→**Display Configurations**.



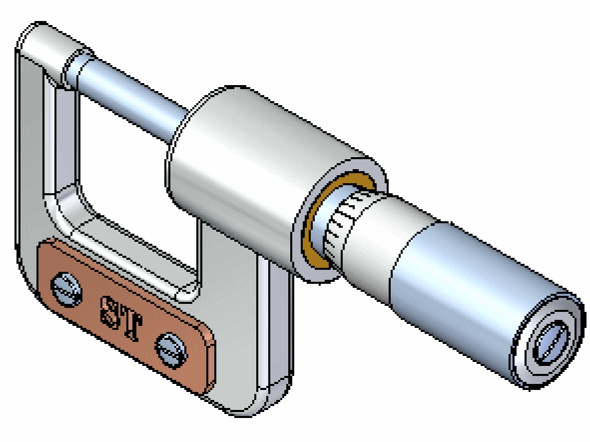
The **Display Configurations** dialog box displays.



- On the **Display Configurations** dialog box, click the **New** button, and type **EXPLODE1** as the name of the display configuration, and click **OK**.

- Click the **Close** button to dismiss the dialog box.

Unexplode the assembly display



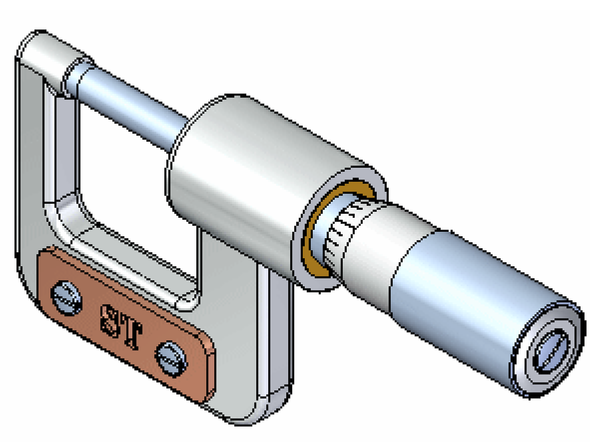
Because an exploded view configuration was saved, use the **Unexplode** command to return the assembly display to the assembled condition. This command is also useful when creating several exploded view configurations.

□ Choose **Home** tab→**Modify** group→**Unexplode** .

A dialog box displays warning that the exploded view will be deleted if no display configuration of the exploded view is saved.

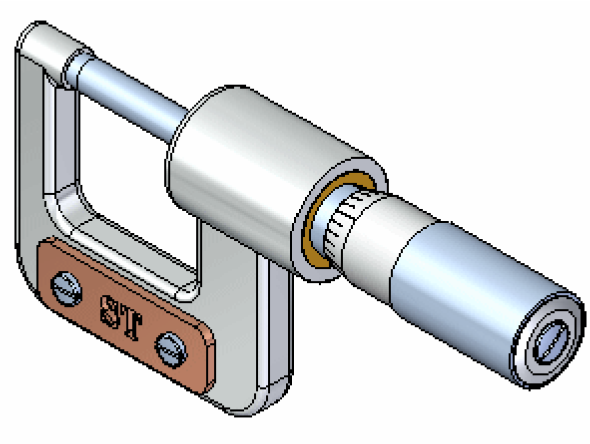
□ On the dialog box, click **Yes**.

Fit the view



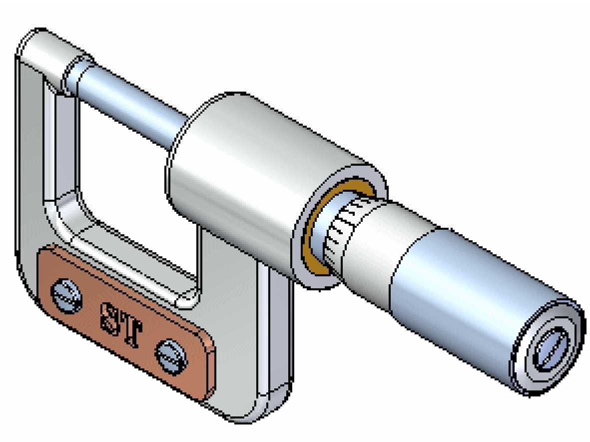
□ Choose **Fit**  to fit the contents of the view to the graphics window.

Close the Explode - Render - Animate application



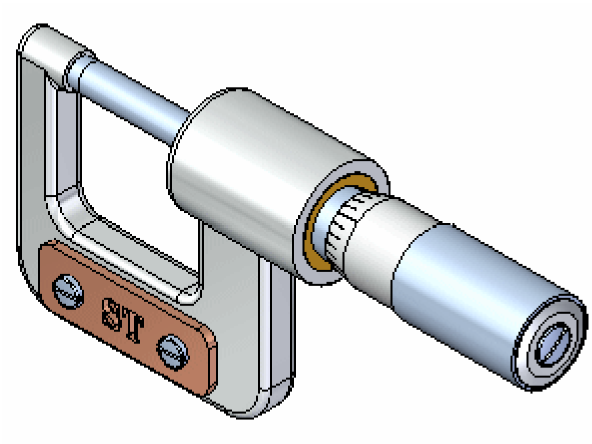
- Choose **Home** tab→**Close** group→**Close ERA** to return to the main Assembly environment.

Save the assembly



- On the **Quick Access** toolbar, choose **Save** .

Exploded view creation complete

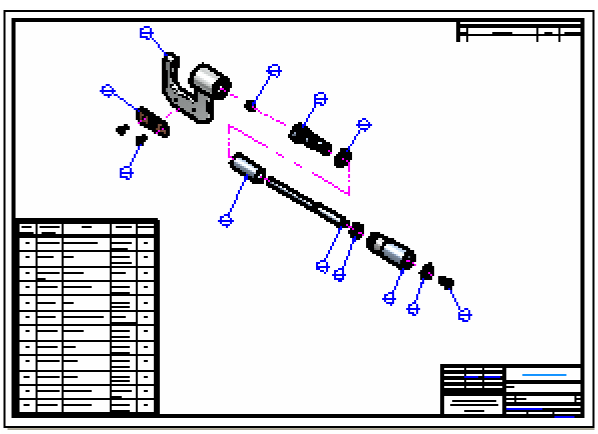


The assembly modeling portion of this test drive is complete.

Although there are many more options available in the Assembly environment, the basic concepts required to build and explode assemblies in Solid Edge has been demonstrated.

Create an assembly drawing

Create an assembly drawing



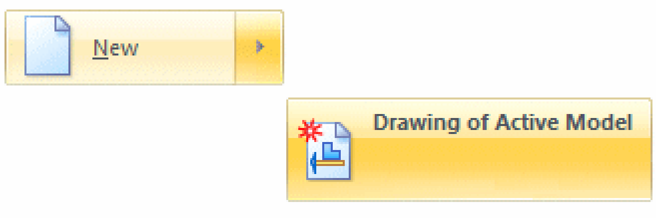
In the next few steps, you will create an exploded view drawing of the assembly.

You will also place a parts list on the drawing.

Start the Create Drawing command

- At the top-left side of the application window, click the Application button  to display the Application menu.

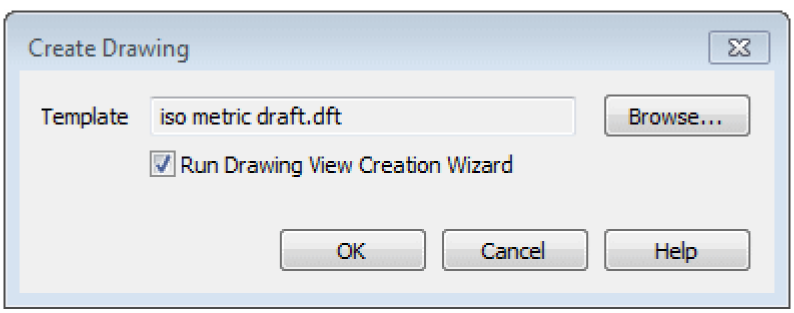
- On the Application menu, point to New, the click Create Drawing of Active Model.



The Create Drawing dialog box displays.

Set the Create Drawing options

- On the Create Drawing dialog box, ensure the Run Drawing View Creation Wizard option is set, then click OK.



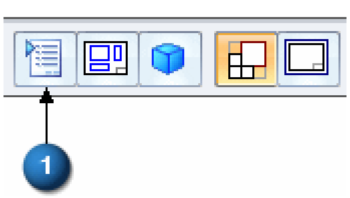
A new drawing document is created, with drawing views of the part created and positioned.

Set the Drawing View Creation Wizard options

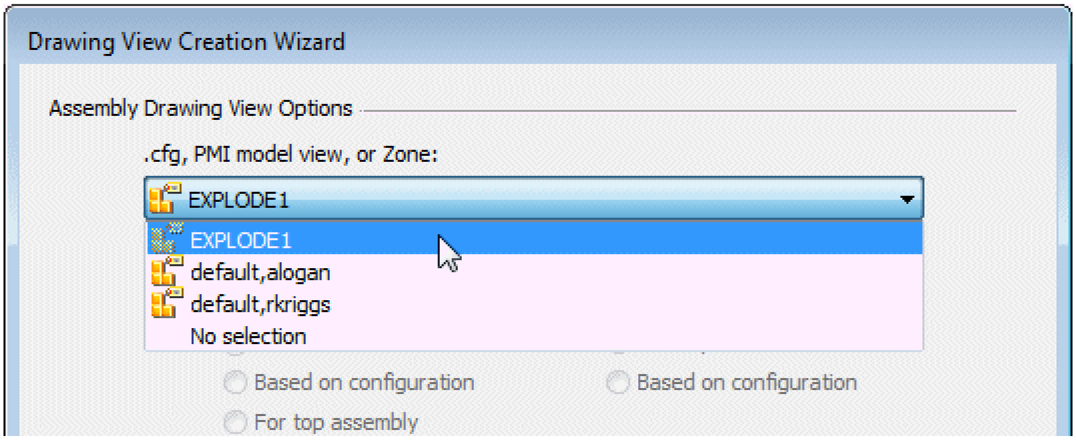
Solid Edge transitions to the Draft environment and creates a new Draft document, using the template specified in the Set the Create Drawing options dialog box. Solid Edge also displays the Drawing View Creation Wizard, which helps streamline the process of placing views of 3D models on a drawing.

The view of the assembly that you will place on the drawing is the exploded view of the assembly that you saved a few steps ago.

- On the Drawing View command bar, click the Drawing View Wizard options (1).



- On the Drawing View Creation Wizard options dialog box, in the field labeled *.cfg, PMI model view, or Zone*, click the list and select the EXPLODE1 view that you saved earlier.



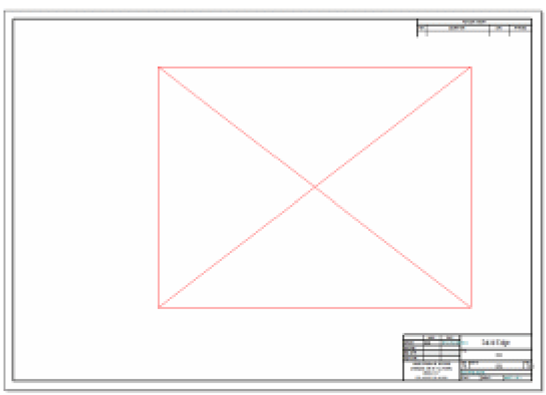
- On the Drawing View Creation Wizard, click Finish.

- On the command bar, set the scale to 1:1.

Position the view on the sheet

When the Drawing View Creation Wizard closes, a rectangle representing the view that will be created is attached to the cursor. The view will be placed where ever you click.

- Position the view approximately as shown, and click to place it.

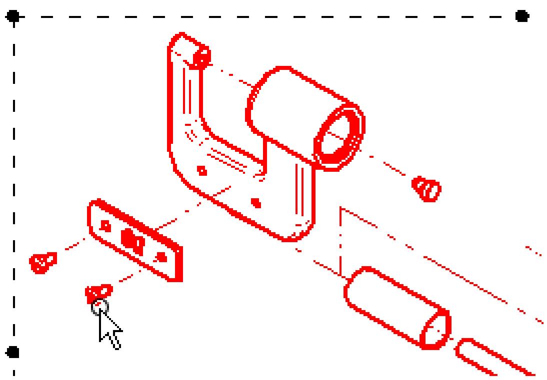


Set the view properties

By default, the view has been created with a wireframe display. Let's make this a shaded display using the model colors.

□ Choose Home tab→Select command or press the Escape key.

□ Position the cursor over the view, as shown below, and right-click. On the short cut menu, choose Properties.



□ On the High Quality View Properties dialog box, on the Shading and Color tab, set the *Show shading in drawing views* option and set the *Use model colors* option.

At the scale of this drawing, the image will appear cleaner if you clear the Apply Part Base Colors to Edge Styles option.

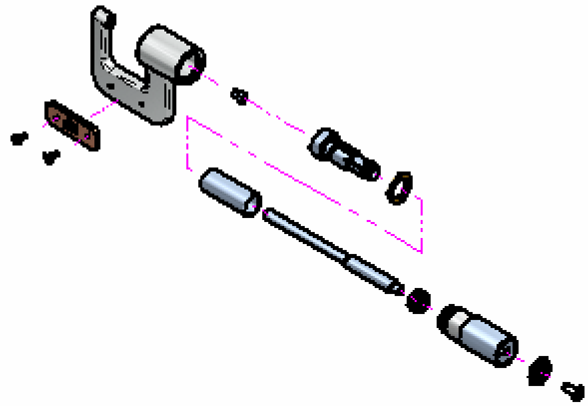
□ On the High Quality View Properties dialog box, click OK.

To apply the options you have set, Solid Edge needs to get more information from the assembly file.

□ Choose Home tab→Drawing Views group→Update Views .

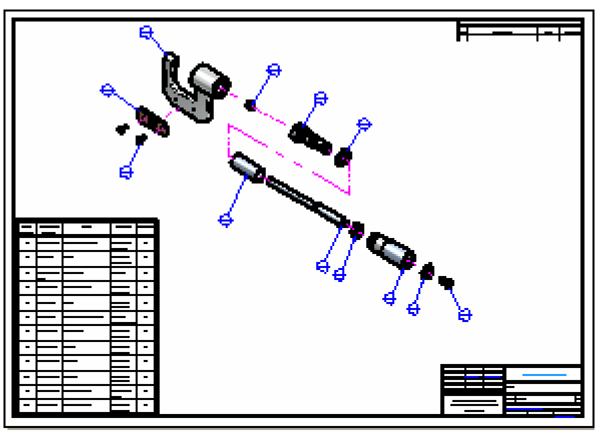
□ Click on the drawing sheet, away from the view, to deselect the view and show its final state.

Observe the result



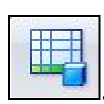
Notice that an exploded view of the assembly is placed in the new drawing document and that the display of the drawing view is shaded with visible edges displayed.

Start the Part List command

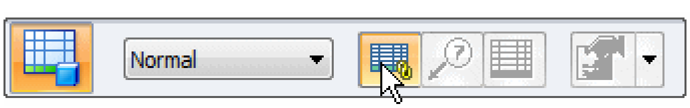


In the next few steps you will place a parts list of the assembly on the drawing sheet.

You will also place balloons on the exploded view automatically using the Parts List command.

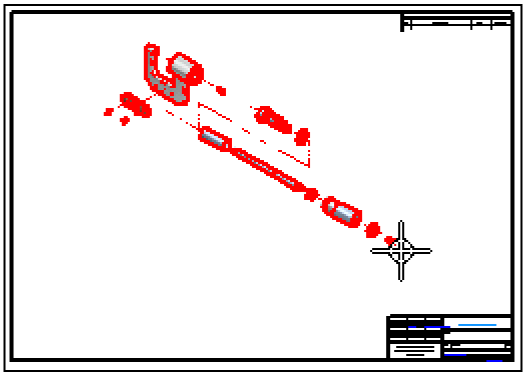
□ Choose Home tab→Tables group→Parts List command .

Set the command bar options



□ On the Parts List command bar, ensure that the Link to Active option is set.

Select the drawing view



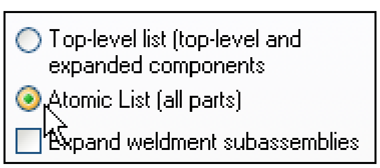
- ☐ In the graphics window, position the cursor over the drawing view, then click to select it.

Notice that additional options in command bar are activated after you select the drawing view.

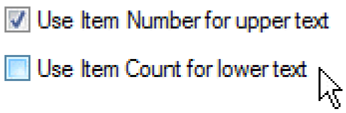
Set the parts list and balloon properties

- ☐ On the Parts List command bar, click the Properties button .

- ☐ On the Parts List Properties dialog box, click the List Control tab, then set the Atomic List (all parts) option.



□ Click the Balloon tab, then clear the Item Count option.



□ Click the Columns tab.

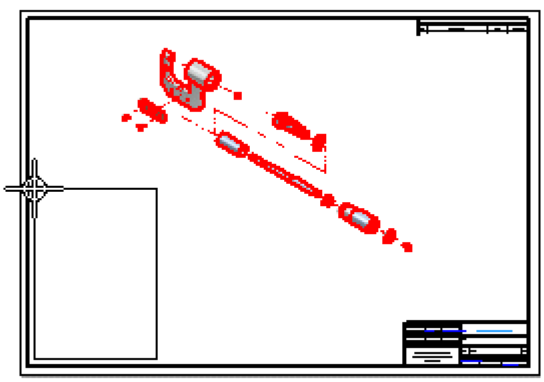
Notice that you can click a property in the lower list, then click the Add Column button to add a property to the Columns list. These are the properties that will display in the properties table. You can use the Move Up and Move Down buttons to change the order of the columns, and you can use the Delete Column to remove columns.

□ Use these controls to define these columns:

Item Number
Document Number
Title
Material
Quantity

□ Click OK.

Finish the parts list



□ A rectangle representing the parts list table is attached to the cursor. Position the table as shown, and click to place it.

Notice that the parts list and balloons are placed on the drawing sheet, and an alignment shape is displayed around the view, with the balloons connected to the shape. You can manipulate the shape to reposition the balloons.

Observe the parts list

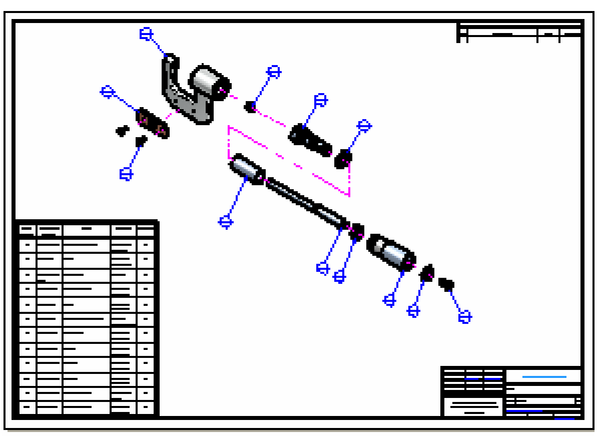
Item Number	Document Number	Title	Material	Quantity
1	FRAME-A10	FRAME, MICROMETER	Aluminum, 7075-T6	1
2	ANVIL-101	ANVIL	Stainless Steel, 420	1
3	NAMEPLATE-101	PLATE, NAME	Copper	1
4	SCREW-201	SCREW, SLOTTED	Stainless Steel, 303	2
5	STEM-102	STEM	Stainless Steel, 303	1
6	PLATE-101	PLATE, BARREL SUPPORT	Brass, yellow brass	1
7	LOCK-101	LOCK RING	Stainless Steel, 303	1
8	BARREL-101	BARREL	Stainless Steel, 303	1
9	STEM-101	SPINDLE	Stainless Steel, 303	1
10	THIMBLE-101	THIMBLE	Stainless Steel, 304	1
11	WASHER-101	WASHER, SPINDLE	Aluminum, 1060	1
12	SCREW-101	SCREW, SLOTTED	Stainless Steel, 303	1

- On the Viewing commands toolbar, choose Zoom Area , then zoom in as shown in the illustration.
- After you have resized the view area, right-click to exit the Zoom Area command.

Notice that the parts list contains rows for each part in the assembly and columns for the Item Number, Document Number, Title, Material, and Quantity.

You can configure parts lists to meet your company's requirements.

Save the drawing



- On the Viewing Commands toolbar, choose Fit  to fit the drawing sheet to the view.
- On the Quick Access toolbar, click the Save button .
- On the Save As dialog box, accept the default filename of **stoabmm.dft**, then click the Save button to save the draft document.

Congratulations!

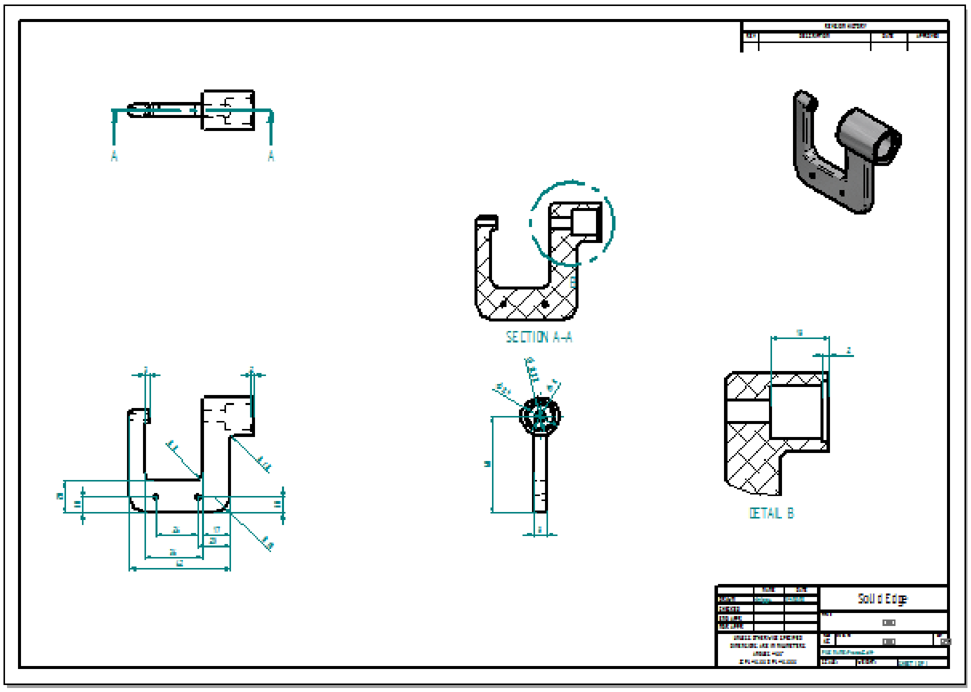
You have completed your first assembly and assembly drawing in Solid Edge.

To learn more about Solid Edge with Synchronous Modeling Technology, you can do the following:

- Place additional PMI dimensions on the model geometry and edit the model to view the results.
- Use the steering wheel to edit different features of the model until you understand more of the options available.
- Select Solid Edge Help from the Help menu, and explore topics that are related to the subjects described in this Test Drive.

In the next section of this Test Drive you will build and create a detailed drawing of the frame of the micrometer and learn more about creating 2D drawings.

Chapter 3: Introduction to detailed drawings

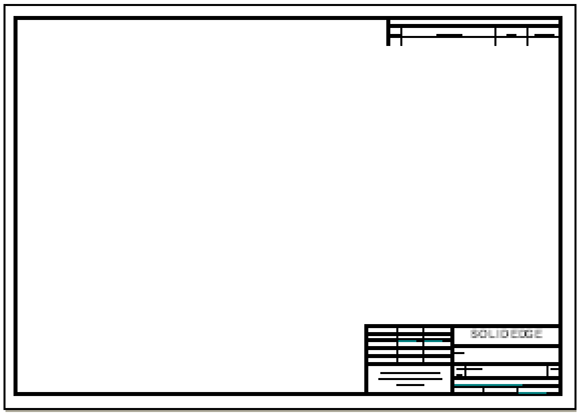


This tutorial provides step-by-step instructions for creating a 2D drawing from a 3D part model. Create this drawing will help to learn techniques such as:

- Defining the size of the drawing sheet you want to use.
- Controlling the projection angle.
- Placing different types of drawing views.
- Placing dimensions and annotations.
- Editing the part model in the context of the drawing.
- Updating the revised drawing.

Set the drawing sheet size

Set the drawing sheet size and projection angle



In the next few steps, specify the drawing sheet size and projection angle to use.

Solid Edge includes a wide range of sample drawing sheets to customize to meet company requirements.

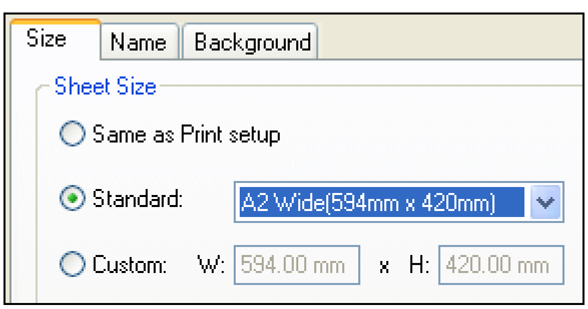
Create a draft document

- Choose **Application** menu→**New**→**ISO Metric Draft**.

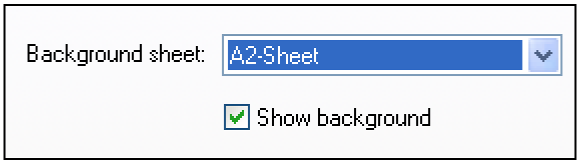
Specify the dimensions of the drawing sheet

The first step in beginning a new drawing is to set up the drawing sheet.

- On the **Application** menu, choose **Settings**→**Sheet Setup**.
- In the **Sheet Setup** dialog box, on the **Size** page, set the Sheet Size option to A2 Wide (594 mm x 420 mm).

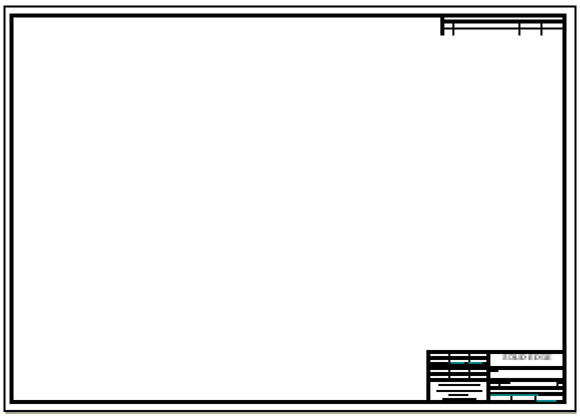


- ◻ Click the **Background** tab, and then set the Background Sheet option to **A2-Sheet**.



- ◻ Click **OK**.

Fit the drawing sheet to the window



- ◻ On the status bar at the bottom of the application window, choose **Fit**  to fit the drawing sheet to the application window.

Observe the projection angle and drawing standards options

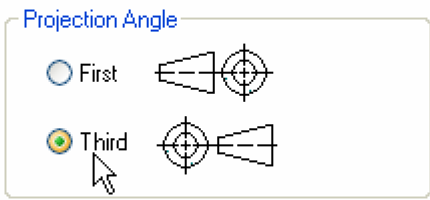
Mechanical drafting standards use either first-angle projection or third-angle projection for creating multi-view projections of a part on a drawing sheet.

- The first-angle method is predominantly used by engineers and designers who follow ISO and DIN standards.
- The third-angle method is predominantly used by engineers and designers who follow ANSI standards.

Solid Edge provides document templates for both projection-angle standards.

For this test drive, use third-angle projection. Set the projection angle on the **Drawing Standards** page of the Solid Edge **Options** dialog box.

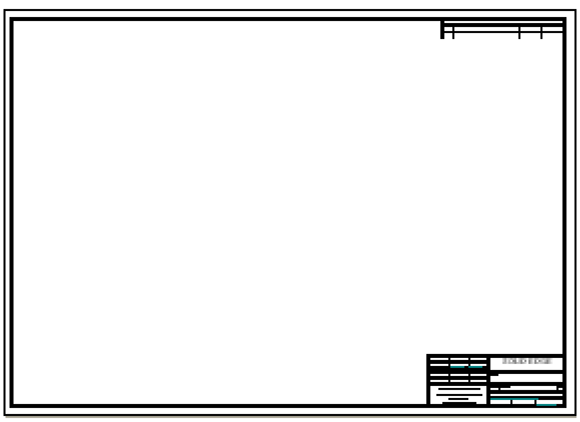
- ◻ Click **Application** menu→**Settings**→**Options** to open the Solid Edge Options dialog box
- ◻ In the Solid Edge **Options** dialog box, click **Drawing Standards**.
- ◻ On the **Drawing Standards** page, under Projection Angle, set the projection angle to **Third**.



Observe the other options on the **Drawing Standards** page.

- ◻ Click **OK** to close the dialog box.

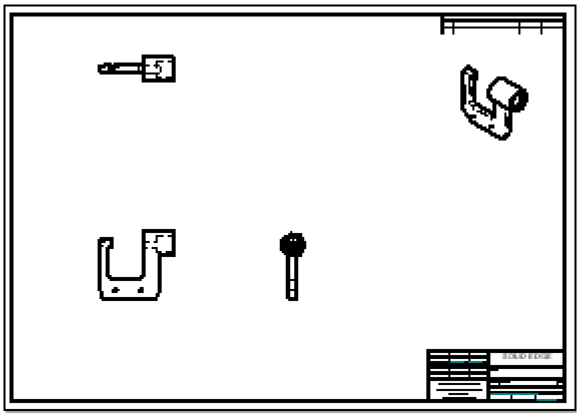
Save the file



- ◻ On the **Quick Access** toolbar, choose **Save** .

Define part model and drawing views

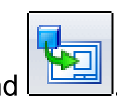
Choose the part model and place the initial drawing views



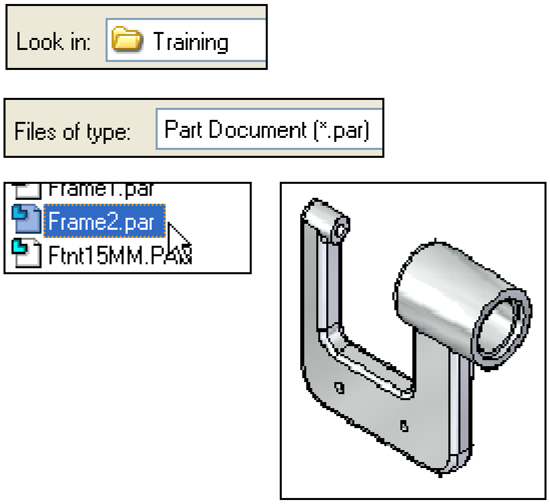
In the next few steps, choose the 3D part model and place the initial drawing views on the drawing sheet using the **Drawing View Wizard**.

Also learn how to move drawing views on the drawing sheet.

Choose a part model to place on the drawing sheet

□ Choose **Home** tab→**Drawing Views** group→**View Wizard** command .

□ In the **Select Model** dialog box, do the following:



- Set the Look In location to the Solid Edge ST9 Training folder.

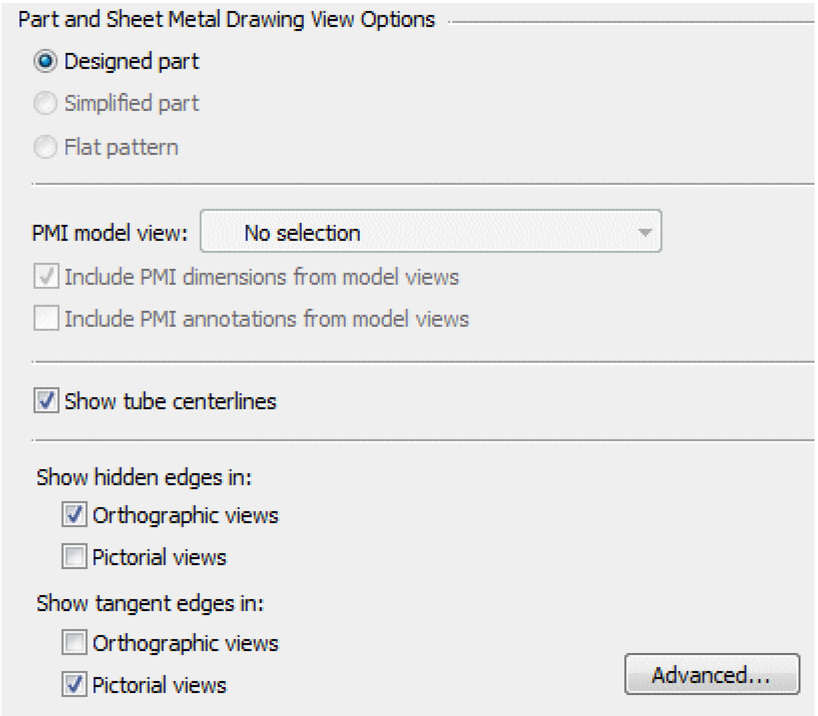
  > The default location of the Training folder is **C:\Program Files\Solid Edge ST9\Training**. However, the system administrator may have chosen a different location.

- Set the Files Of Type option to Part Document (*.par).
- Select the file named: **Frame2.par**.
- Click **Open**.

The **View Wizard** command bar displays.

□ On the **View Wizard** command bar, click the **Drawing View Wizard Options** button.

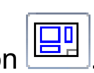
Observe the options on the Drawing View Creation Wizard



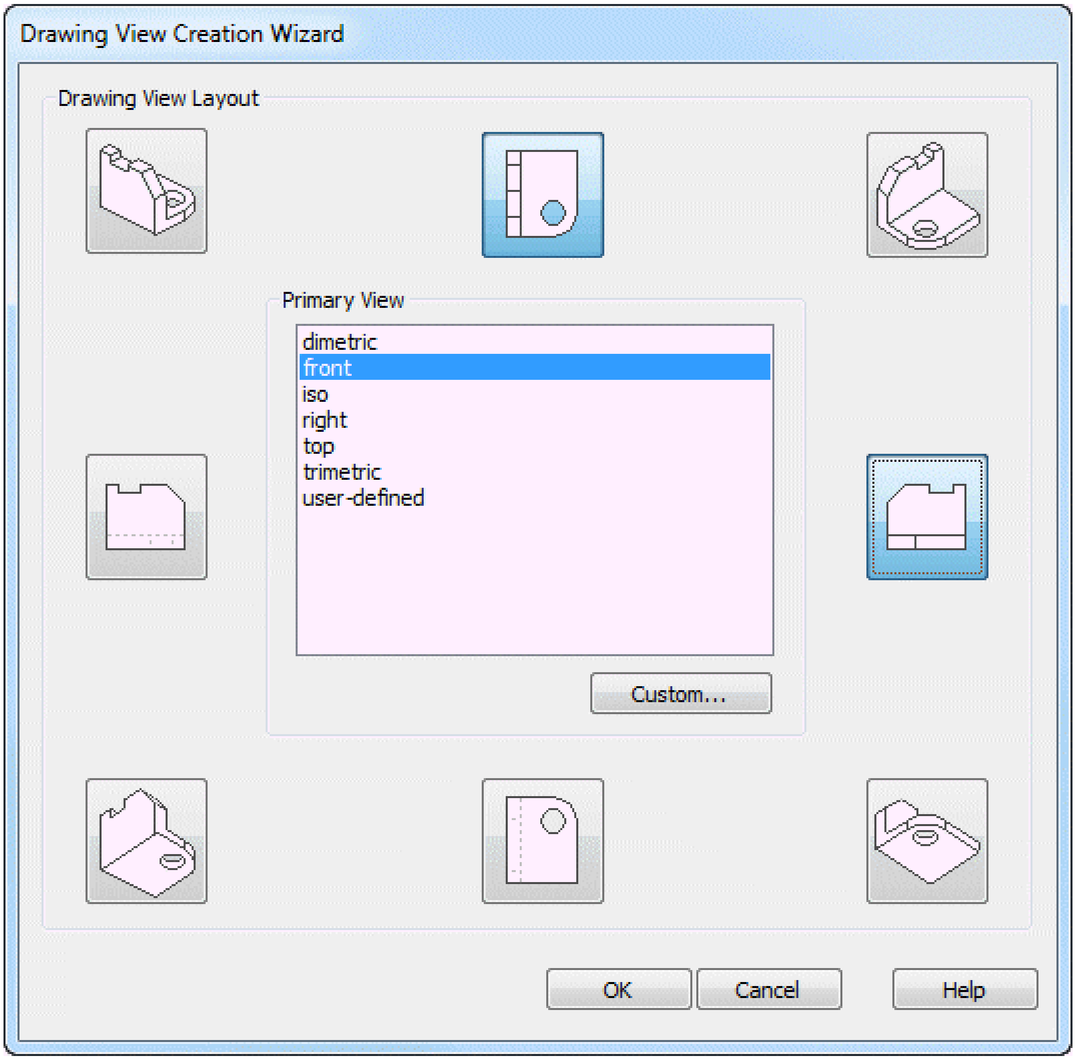
Observe the options on the first page of the **Drawing View Wizard**. For this drawing, the default options shown above work well.

- ◻ Ensure that the part view options on the first page of the **Drawing View Wizard** match the illustration above.

- ◻ Click the **OK** button to continue.

- ◻ On the **View Wizard** command bar, click the **Drawing View Layout** button .

Specify the drawing view layout



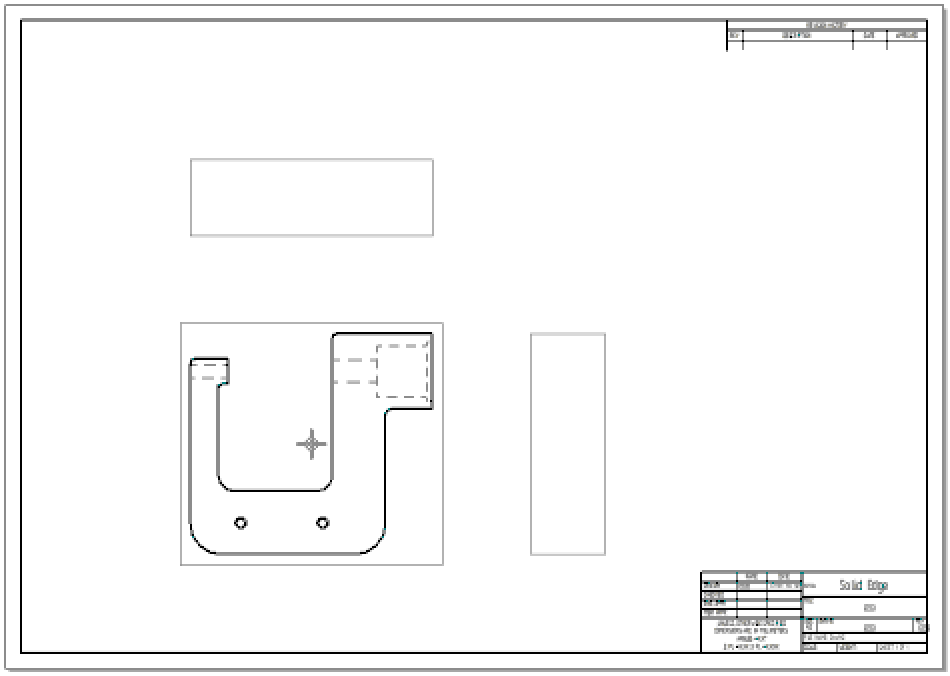
Use the **Drawing View Layout** page to specify the primary drawing view and additional drawing views. The primary view is shown in the center.

All drawing views do not have to be specified in this step. Add them later using commands on the **Home** tab→**Drawing Views** group.

- ◻ On the **Drawing View Layout** page, set the Primary View to **front**.
- ◻ Select the **top** and **right** side views, as shown above.
- ◻ Click the **OK** button to close the **Drawing View Wizard**.

  Do not click the drawing sheet yet.

Place the views on the drawing sheet

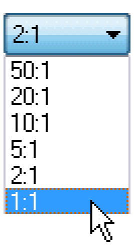


- □ Observe the **View Wizard** command bar, which may display horizontally at the top of the application window or vertically along one side, depending on the user interface theme selected. Notice that the cursor display has changed in the graphics window.

The command bar contains options for controlling the drawing view scale, view display properties and so forth.

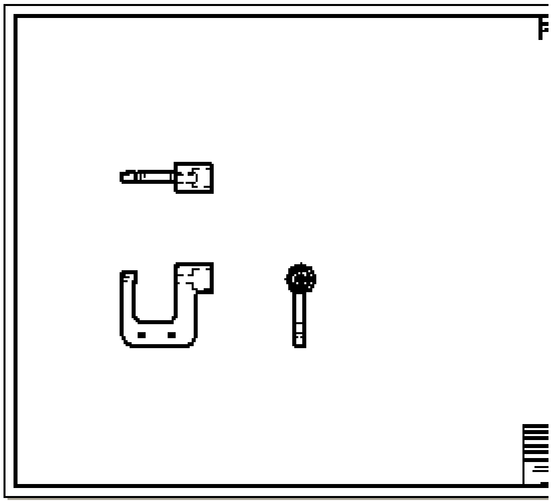
A ghosted image of the drawing views defined in the wizard are attached to the cursor, ready to position them on the drawing sheet. Do not click to place the views yet.

- □ In the **View Wizard** command bar, set the Scale option to 1:1.



- □ Position the cursor approximately as shown above, then click to place the drawing views.

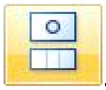
Observe the drawing views



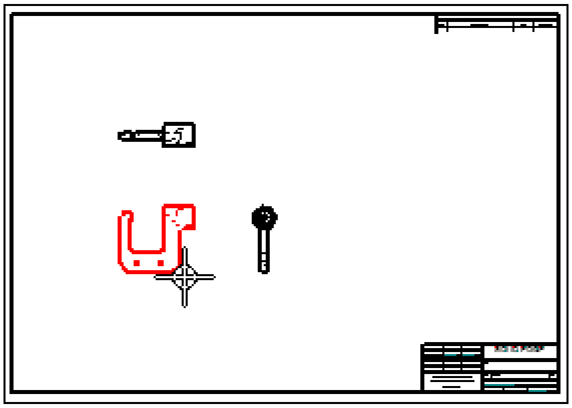
The drawing views should display similar to the illustration.

Place another drawing view

Place another drawing view using the **Principal View** command. Use this command to fold out new drawing views from an existing drawing view.

□ Choose **Home** tab→**Drawing Views** group→**Principal View** .

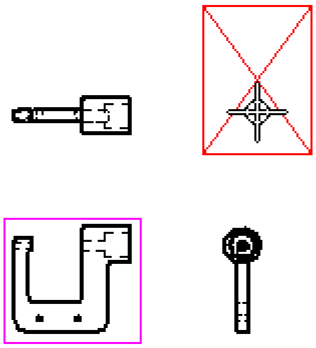
□ Position the cursor over the drawing view shown in the illustration below, then click to select it.



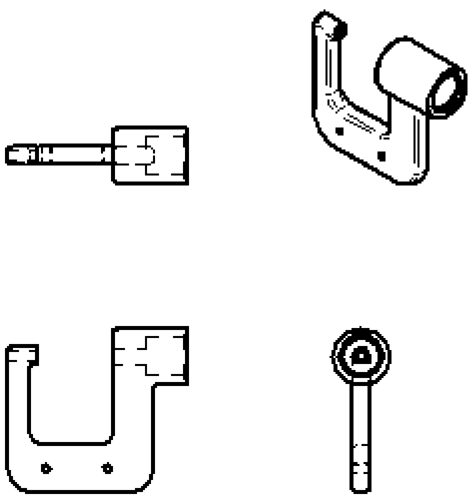
□ Move the cursor around the selected view and notice that a rectangular box is attached to the cursor. Notice that the box size changes as moving into different positions. The size change indicates a different view orientation.

- Position the cursor as shown below, then click to place an isometric drawing view.

  Optional step: Place additional drawing views to observe the results.



Observe the results



Observe the new isometric drawing view.

This drawing view could have placed with the other views using the **Drawing View Wizard**.

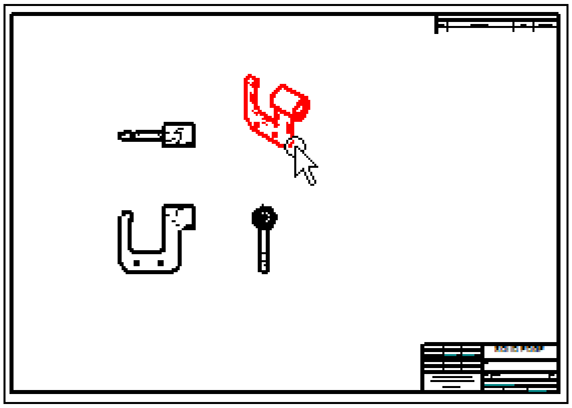
If extra drawing views were placed, select the extra drawing views, then press Delete to delete them.

Reposition a drawing view on the drawing sheet

Adjust the position of a drawing view by dragging it to a new position.

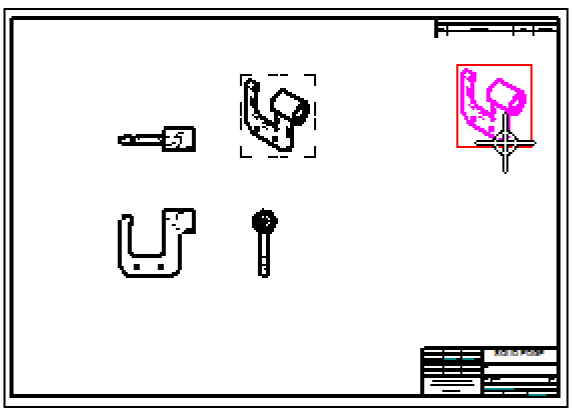
- Press Escape to start the **Select** command .

□ Position the cursor over the isometric drawing view, as shown below:



When the view highlights, drag the cursor to reposition the drawing view to the top right corner of the drawing sheet, as shown below.

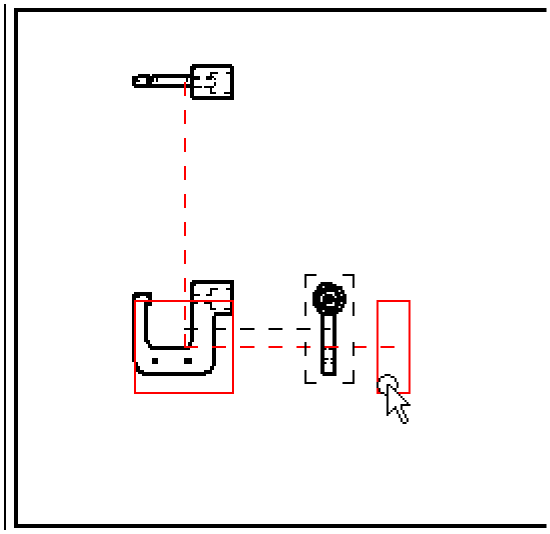
- Hold the left mouse button while dragging the view.
- Release the mouse button to position the view.



Reposition more drawing views

When moving an orthographic drawing view, such as a top, front, or right view, other drawing views may move to maintain proper drawing view alignment. Drawing view alignment lines also display to indicate to which views it is aligned.

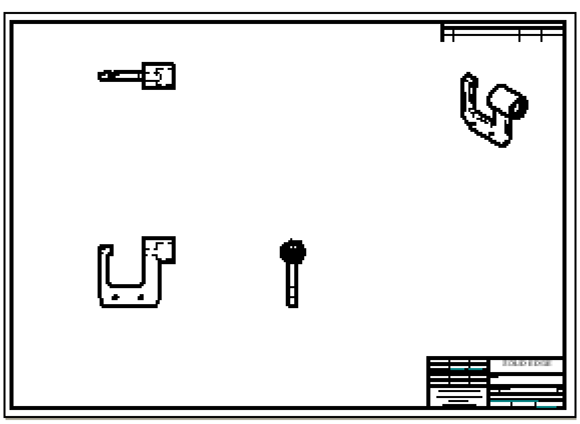
□ Position the cursor over each of the three orthographic drawing views, then drag them to new positions to observe how they remain orthographically aligned.



When moving an orthographic drawing view, orthographic alignment is maintained. If dimensions and annotations were placed on the drawing, these also move.

This makes it easy to adjust the position of the views on the sheet.

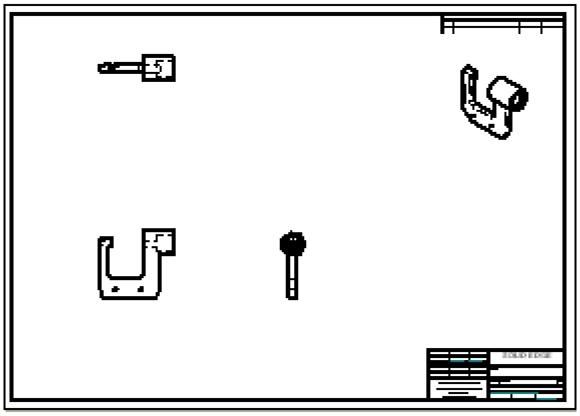
Save the file



The display should now approximately match the illustration.

□ On the **Quick Access** toolbar, choose **Save** .

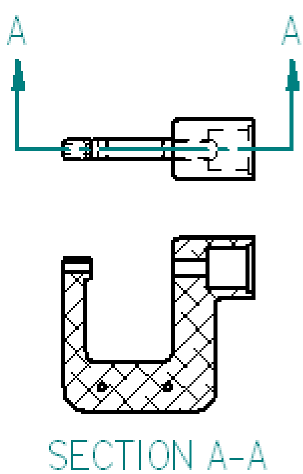
Principal view placement complete



Placing of principal views of the part on the drawing sheet is finished.

Create a section view

Create a section view



In the next few steps, create a section view, as shown above.

To create section views, first define a cutting plane on an existing drawing view using the **Cutting Plane** command.

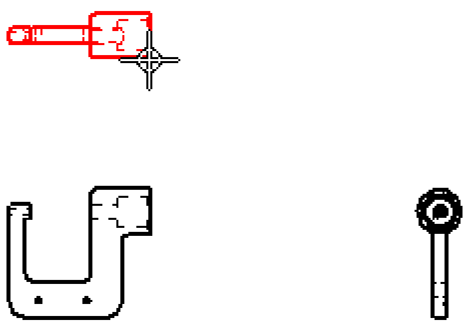
Then use the **Section** command to select the cutting plane and place the section view.

Select the drawing view for the cutting plane

- Choose **Home** tab→**Drawing Views** group→**Cutting Plane** command 

.

- On the drawing sheet, click the drawing view shown below.

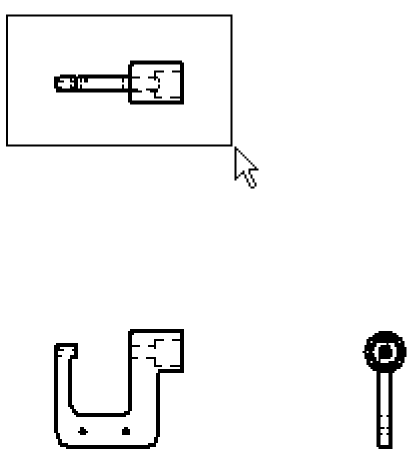


The command ribbon changes to display commands for drawing 2D elements. The **Line** command is active.

Use the Zoom Area command

- At the bottom-right side of the Solid Edge application window, click **Zoom Area** .

- Click above and to the left of the drawing view, then click below and to the right to zoom in around the drawing view.

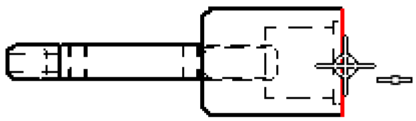


- After resizing the view area, right-click to exit the **Zoom Area** command.

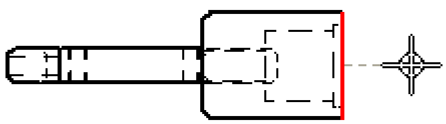
Draw the cutting plane

- ◻ Ensure that the **Home** tab→**Draw** group→**Line** command  is running.

- ◻ Position the cursor over the midpoint of the edge shown in the illustration below, but do not click.



- ◻ When the midpoint relationship indicator displays, move the cursor to the right as shown below.

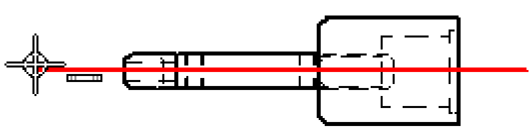


Notice that a dashed line displays between the highlighted edge and the cursor. This indicates that the start point of the line is aligned to the midpoint of the edge.

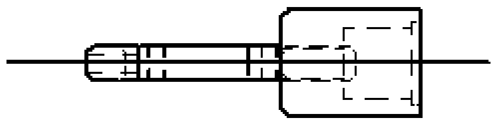
- ◻ Click to place the start point of the line.

- ◻ Move the cursor to the left as shown below, and when the horizontal relationship indicator displays, click to place the end point of the line.

- ◻ Right-click to restart the **Line** command.



Close the cutting plane mode

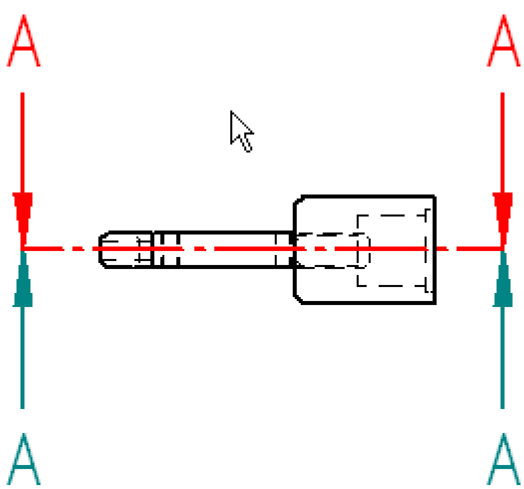


The cutting plane line should display similar to the illustration.

- □ Choose **Home** tab→**Close Cutting Plane** .

The cutting plane options are hidden.

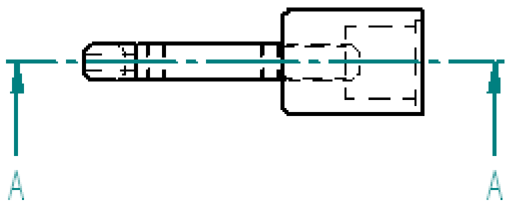
Specify the cutting plane direction



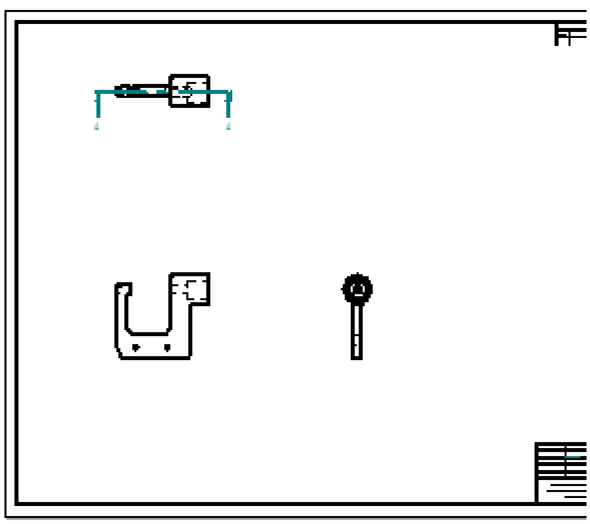
- □ Move the cursor above and below the cutting plane line, and notice that the view direction arrows flip as the cursor crosses the cutting plane line.

□ Position the cursor above the cutting plane line, as shown in the illustration, then click to define the cutting plane direction.

The cutting plane should display as shown below. Depending on the template used to create the draft file, the cutting plane direction arrows might display with a different symbology.



Fit the view



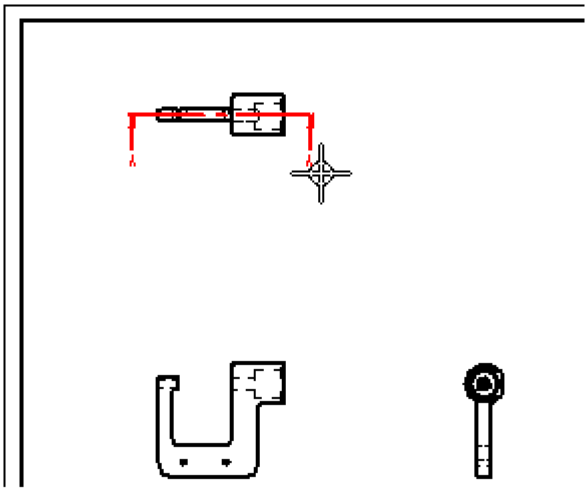
□ On the status bar at the bottom of the application window, choose **Fit** to fit the drawing sheet to the application window.

Create a section view

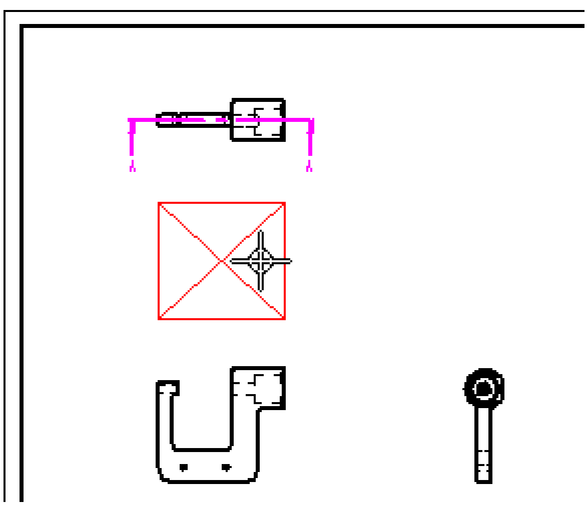
- Choose **Home** tab→**Drawing Views** group→**Section** 

.

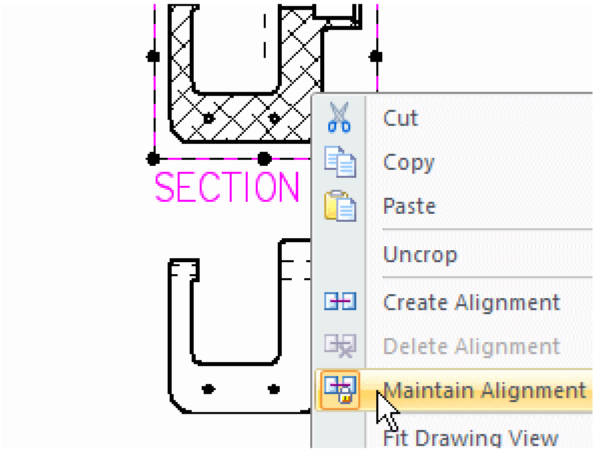
- Click the cutting plane line created previously, as shown below.



- Position the cursor as shown below, then click to place the section view.

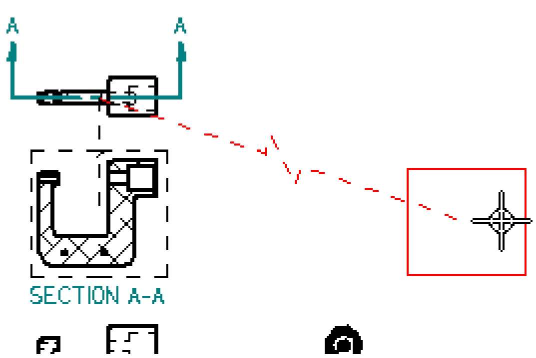


Move the section view



When creating a section view, by default it aligns with its source view. To move the section view independently of its source view, clear the **Maintain Alignment** option first.

- Press Escape to start the **Select** command .
- Position the cursor over the section view, then right-click to display the shortcut menu.
- On the shortcut menu, click **Maintain Alignment** to clear the **Maintain Alignment** option, as shown above.
- Drag the cursor to move the section view to the location shown below.

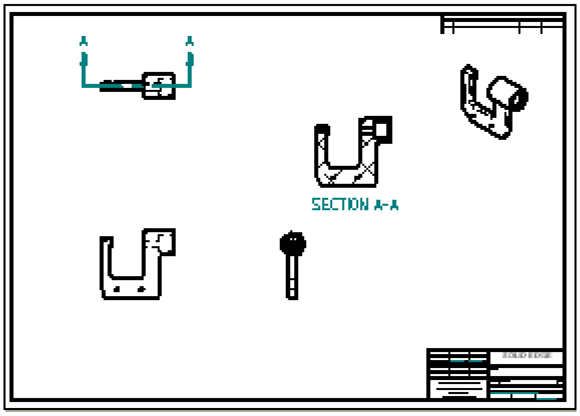


Notice that a dashed line connects the selected section view to its source view.

This indicator makes it easy to determine the source view for the section view later.

- Click in empty space to clear the selection of the section view.

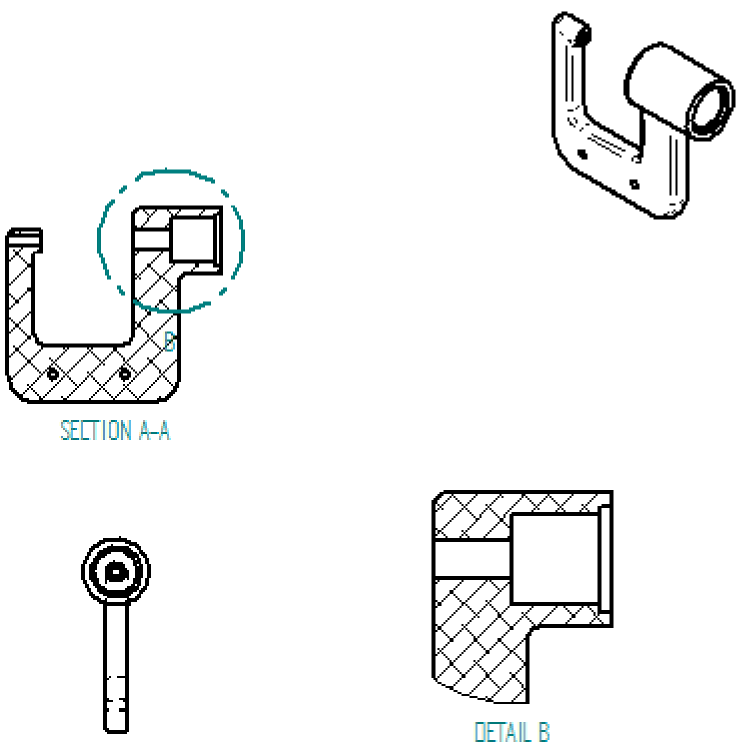
Save the file



- ☐ On the **Quick Access** toolbar, choose **Save** .

Create a detail view

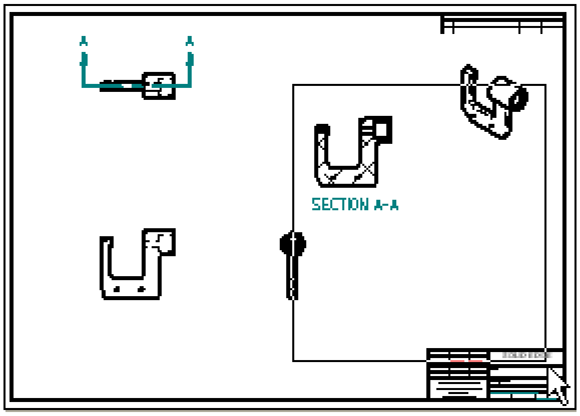
Create a detail view



In the next few steps, create a detail view as shown above.

Use detail views to show magnified areas on a drawing view. Specify the scale of the detail view. A detail view is dynamic. The detail view updates automatically when modifying the source view or moving the detail view circle on the source view.

Use the Zoom Area command



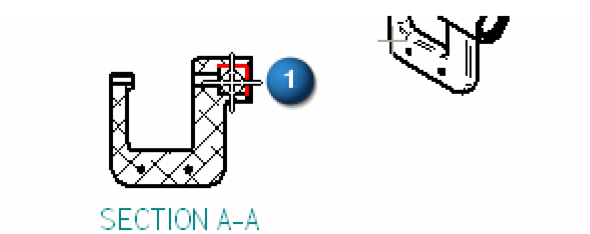
- At the bottom-right side of the Solid Edge application window, choose **Zoom Area** .
- Zoom into the area shown in the illustration above.
- After resizing the view area, right-click to exit the **Zoom Area** command.

Create a detail view

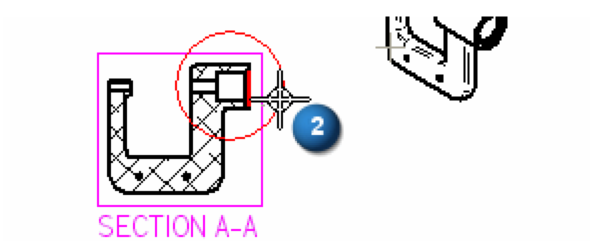
- Choose **Home** tab→**Drawing Views** group→**Detail** command .

Review the options on the **Detail** command bar. Define the detail view scale, whether to use a circular detail view or draw a custom shape for the detail view. For the detail view, use the default options.

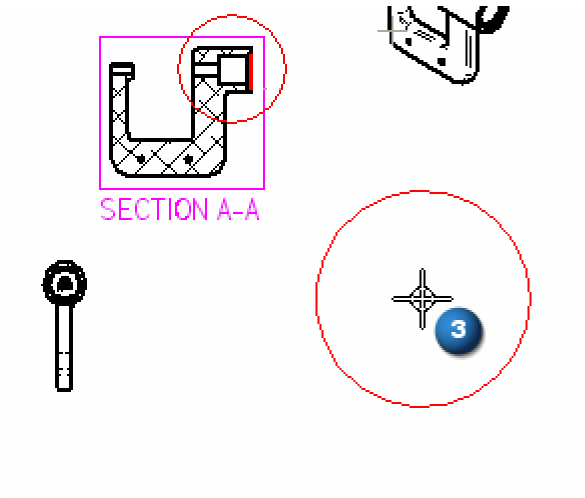
- Position the cursor over the section view as shown in the top illustration, then click to define the center of the detail view circle (1).



- Move the cursor to the side, and then click to specify the diameter of the detail view envelope (2), as shown in the second illustration.

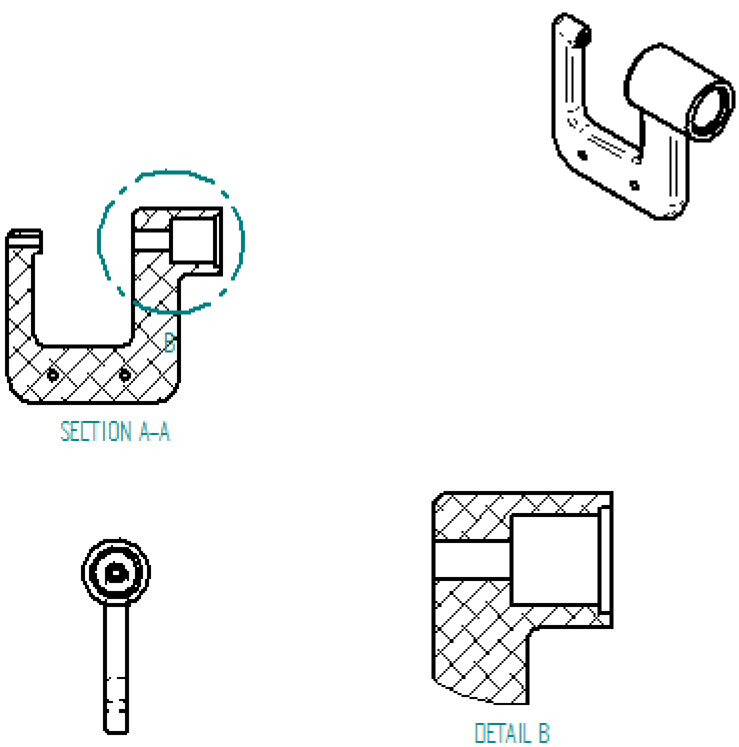


- Move the cursor to position the detail view on the drawing, and then click (3), as shown below.



A custom shape can be used for the detail view area using options on the **Detail** command bar.

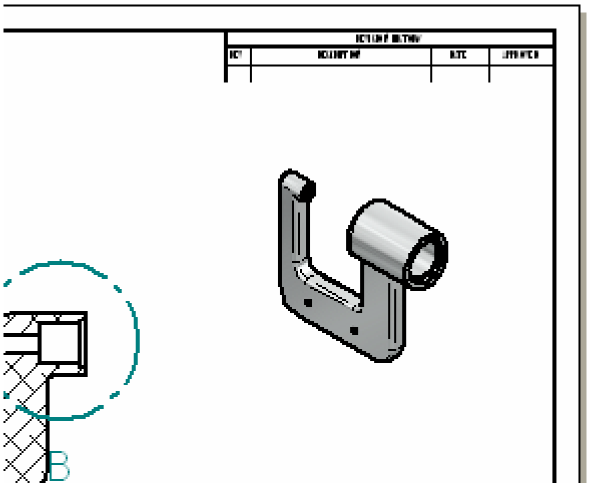
Save the file



- On the **Quick Access** toolbar, choose **Save** .

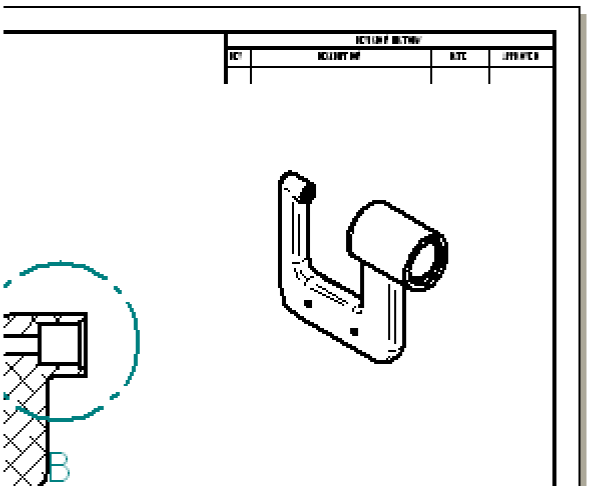
Change drawing view properties

Change drawing view properties



In the next few steps, change the display properties for the isometric drawing view.

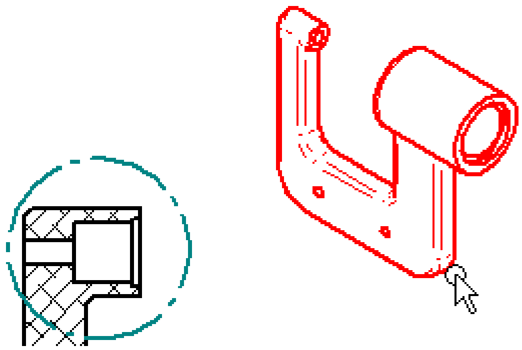
Adjust the view area



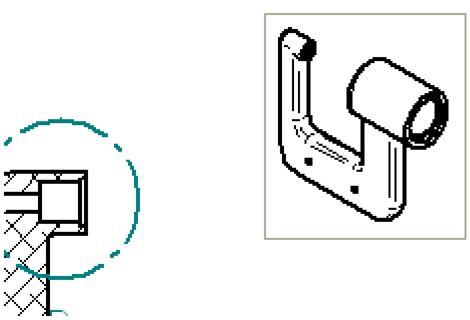
- Use the **Fit**  and **Zoom Area**  commands to adjust the view area to display the isometric view as shown above.

Change the display properties for the isometric view

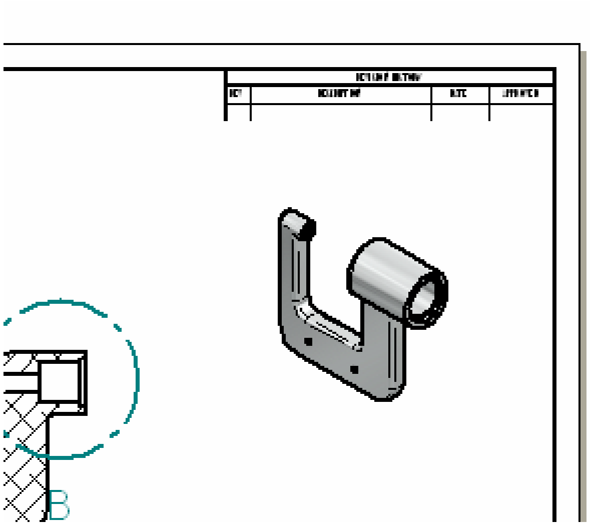
- Ensure that the **Home** tab→**Select** group→**Select** command  is active.
- Select the isometric drawing view.



- On the **Select** command bar, click the **Shading Options** control and set the **Shaded with Edges** option .
- Position the cursor in free space, then click.
- Notice that a grey box displays around the isometric view, and that the display has not updated, as shown below.



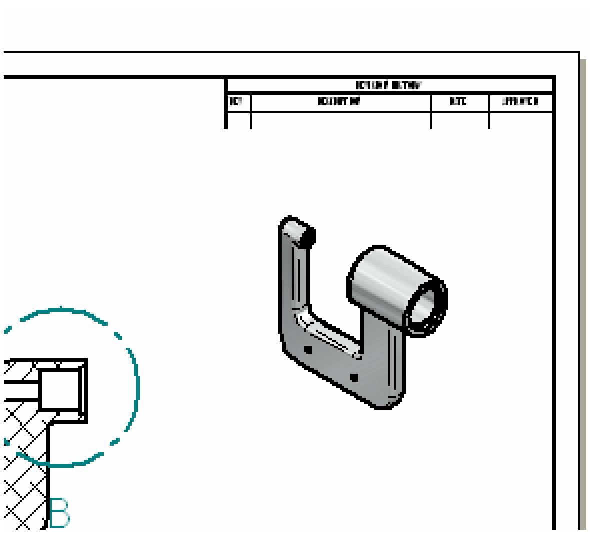
Update the isometric drawing view



- Choose **Home** tab→**Drawing Views** group→**Update Views** .

 An **Update View** shortcut menu command is also available to update a single drawing view. This command is useful when working with complex drawings where updating all the drawing views at once could be time consuming.

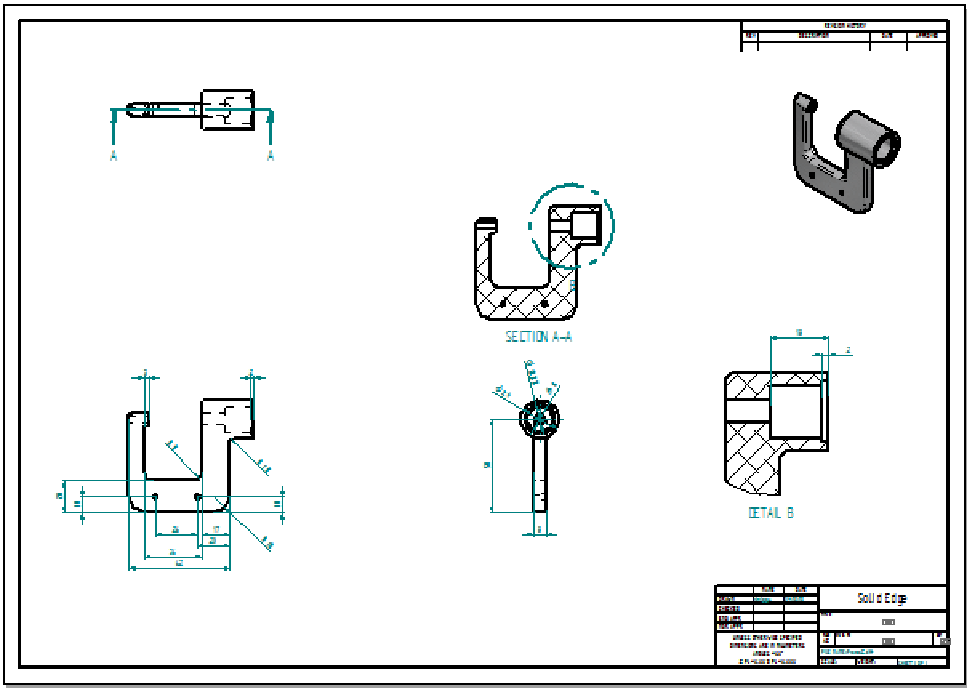
Save the file



- On the **Quick Access** toolbar, choose **Save** .

Add dimensions and annotations to the drawing

Add dimensions and annotations to the drawing



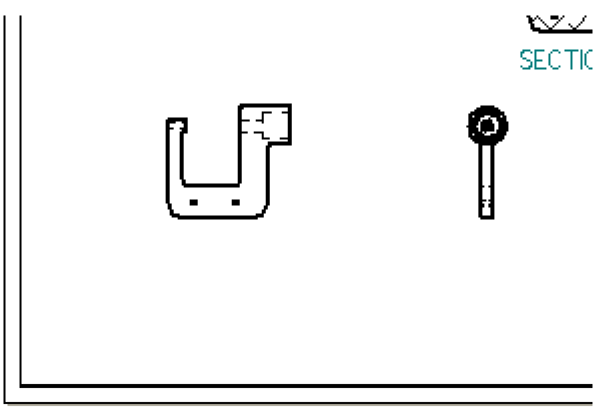
In the next few steps, add dimensions and annotations to the drawing.

Use the **Retrieve Dimensions** command to retrieve model dimensions into the drawing.

Also use some of the other dimensioning and annotation commands to manually add dimensions and annotations.

Adjust the view area

□ Use the **Fit** and **Zoom Area** commands to adjust the view area to display the drawing views shown below.

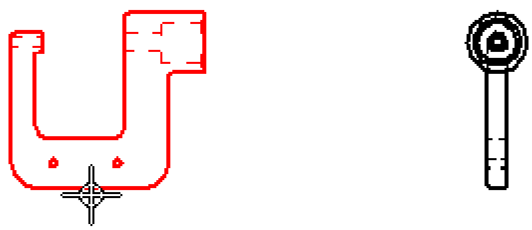


Retrieve dimensions from the model

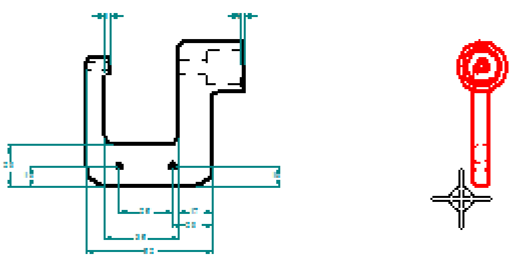
The fastest way to add dimensions to a drawing is to retrieve the dimensions from the model. Use the **Retrieve Dimensions** command to do this.

- ◻ Choose **Home** tab→**Dimension** group→**Retrieve Dimensions** .

- ◻ Click the drawing view shown below.

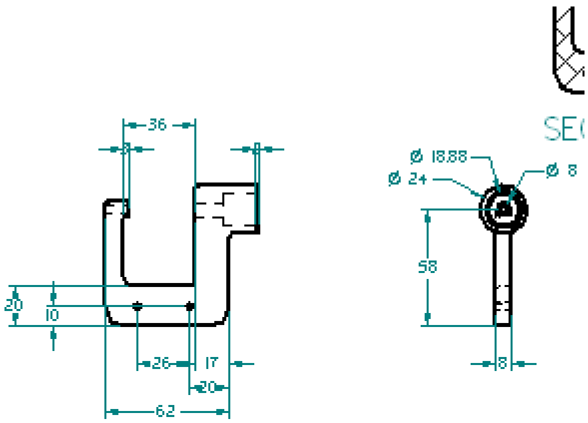


- ◻ Click the drawing view shown in the bottom illustration.



The retrieved dimensions are added to the drawing according to the current settings on the **Retrieve Dimensions** command bar.

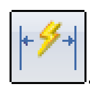
Observe the results

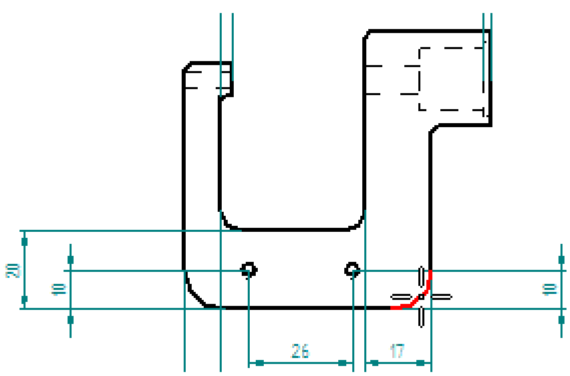


Observe the retrieved dimensions.

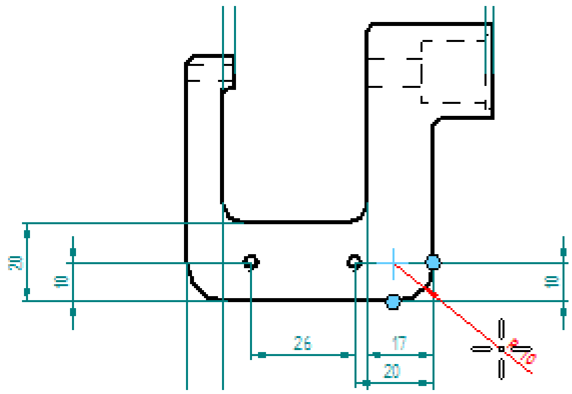
Place dimensions using Smart Dimension

A robust set of dimensioning commands is available for adding dimensions to drawings.

- □ Use the **Zoom Area** command  to zoom in around the side drawing view.
- □ Choose **Home** tab→**Dimension** group→**Smart Dimension** .
- □ Click the radius edge shown below.



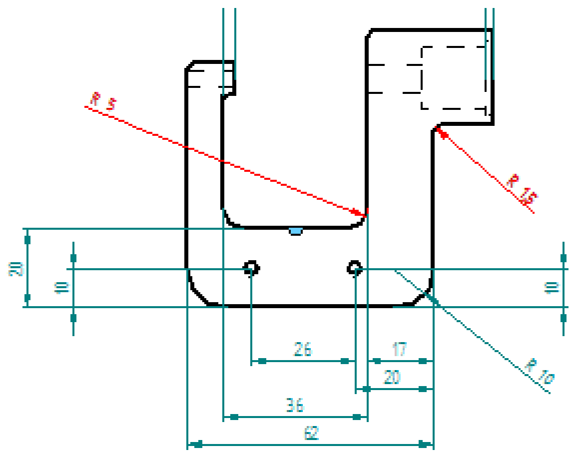
- ◻ Click again to place the dimension, as shown below.



Place more dimensions using Smart Dimension

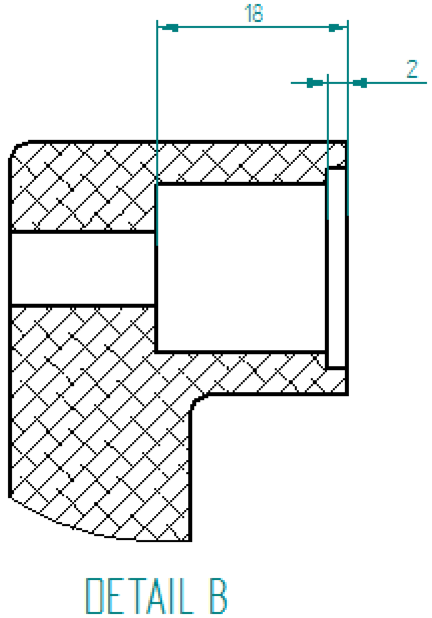
The **Smart Dimension** command should still be active.

- ◻ Place the additional dimensions shown below.



Add dimensions using the Distance Between command

In the next few steps, use the **Distance Between** command to add more dimensions to the part.



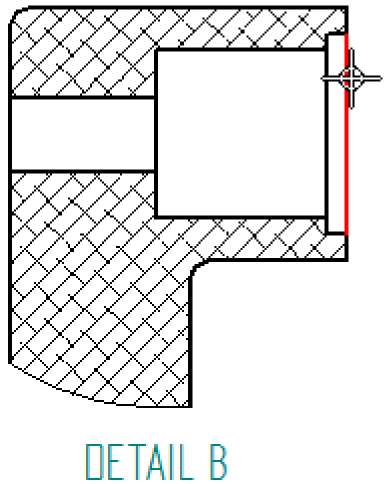
- Use the **Fit** and **Zoom Area** commands to zoom in on the detail view.

- Choose **Home** tab→**Dimension** group→**Distance Between**.

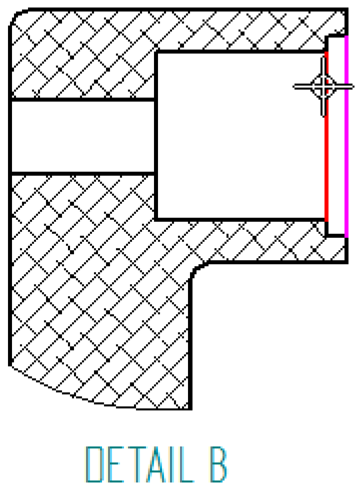
Use this command to dimension between multiple edges or sketch elements. Place chain or stacked dimensions with this command.

Select the first elements to dimension

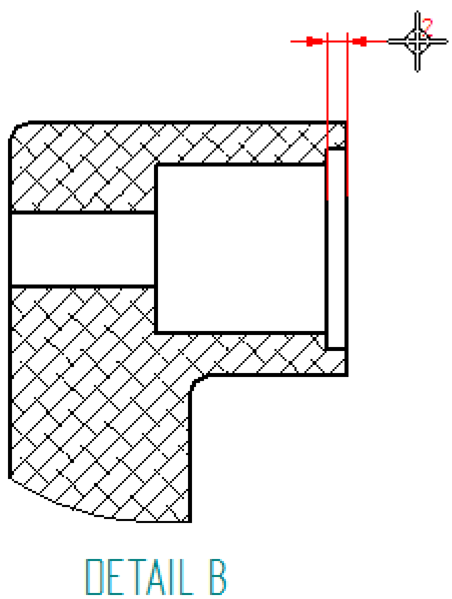
- Position the cursor over the edge shown below, then click to select it.



- ◻ Position the cursor over the edge shown below, then click to select it.

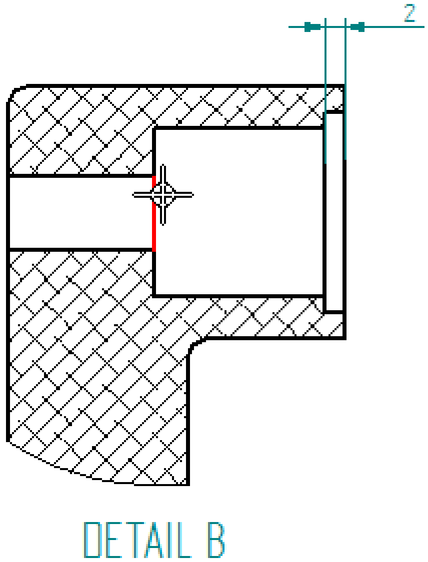


- ◻ Position the cursor above the detail view, then click to position the dimension, as shown below.



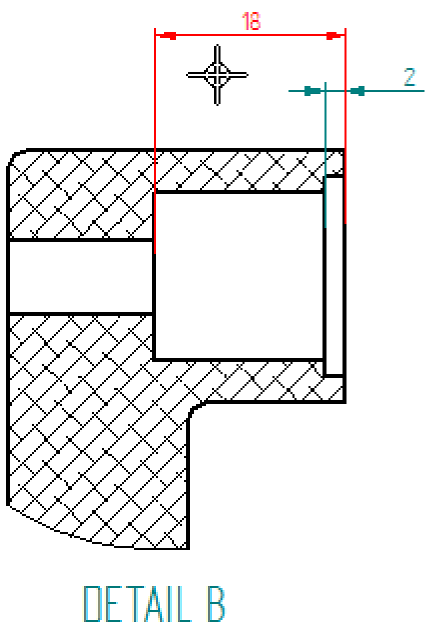
Select the next element to dimension

- Position the cursor over the edge shown below, then click to select it.



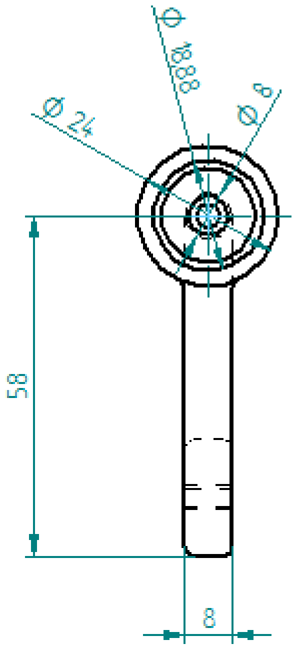
- Move the cursor up and down. Notice that a string or stacked dimension is created by the position of the cursor.

- Position the cursor such that a stacked dimension displays as shown below, then click to place the dimension.

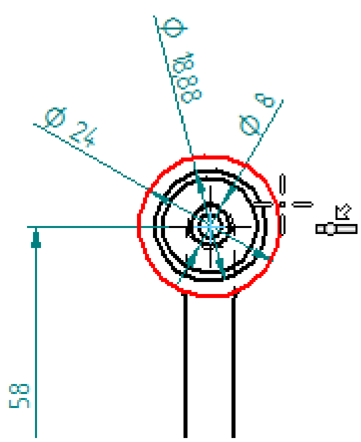


Place a center mark on a drawing view

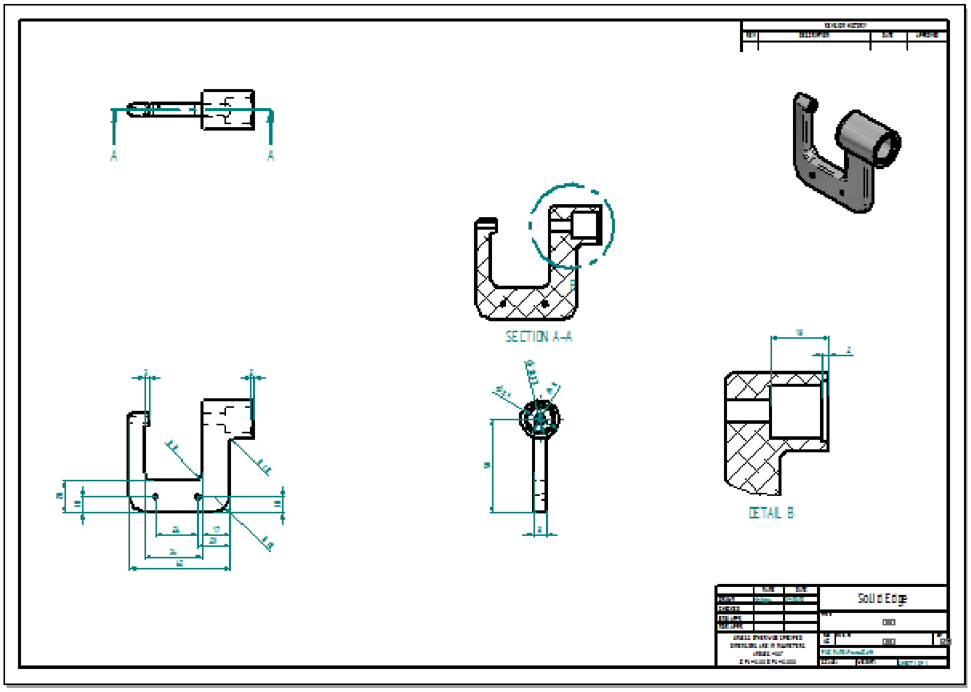
Add center line annotations to circular features, such as holes and cutouts, using the **Center Mark** command.



- Use the **Fit** and **Zoom Area** commands to display the drawing view displayed above.
- Choose **Home** tab→**Annotation** group→**Center Mark** .
- On the **Center Mark** command bar, ensure that the **Projection Lines** option is set.
- Position the cursor to locate the outer-most circular edge, as shown below, then click to place the center mark.

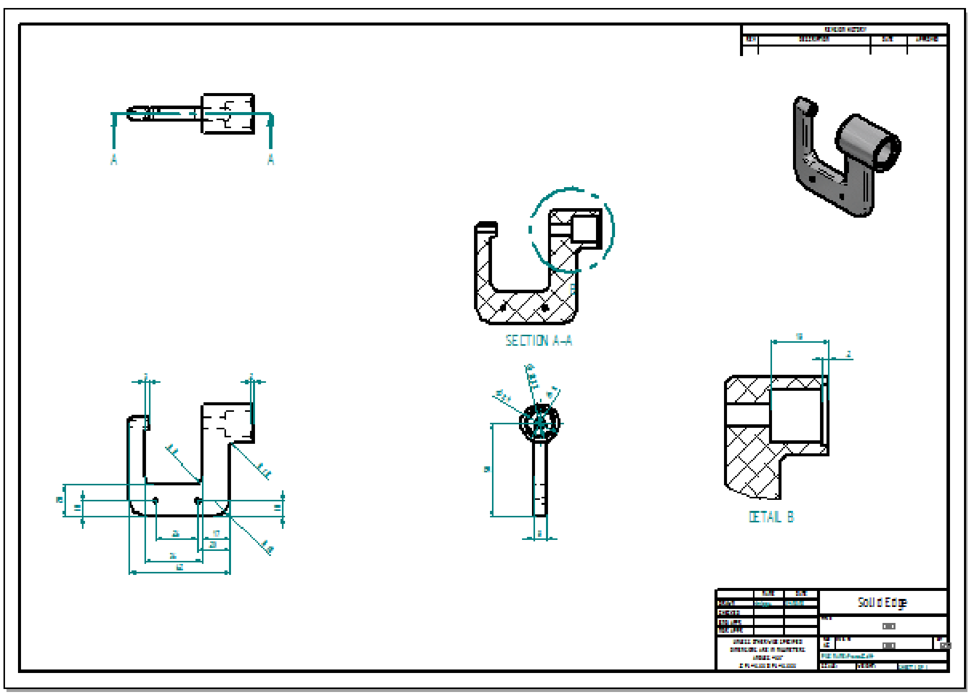


Fit the view



- On the status bar, choose **Fit** to fit the drawing sheet to the application window.

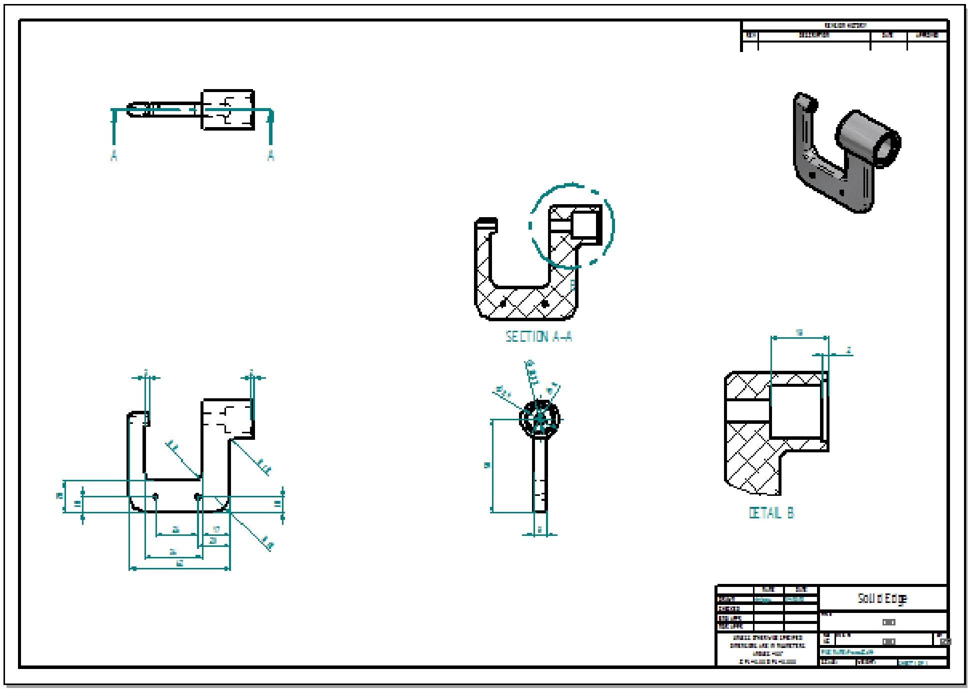
Save the file



- On the **Quick Access** toolbar, choose **Save**.

Edit the model and update the drawing

Edit the model and update the drawing



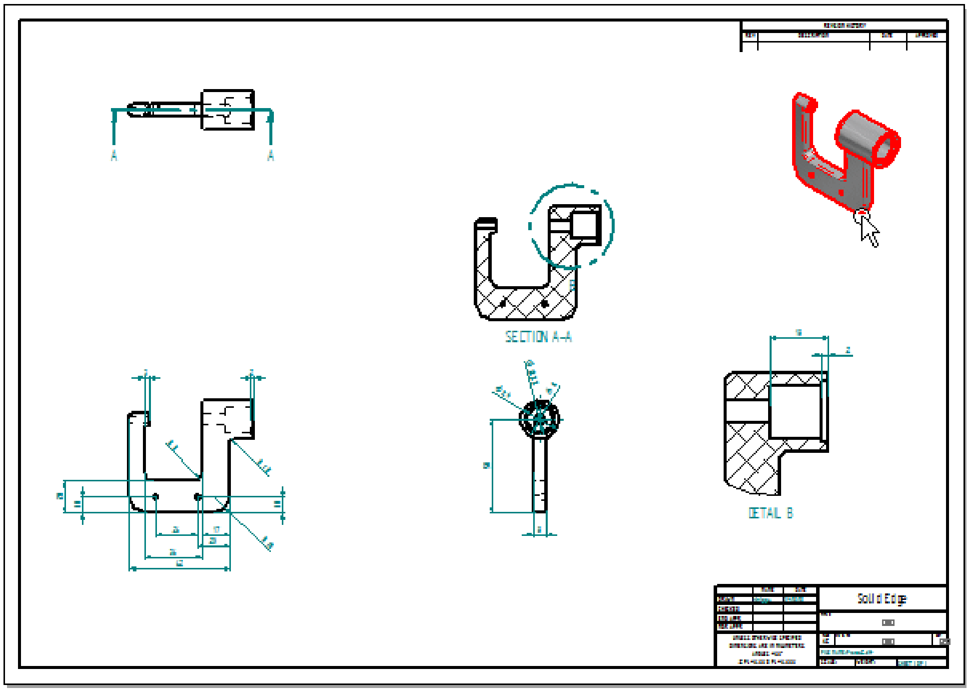
In the next few steps. open the part model to make a design change. Then return to the drawing and update the drawing views.

Open the part model

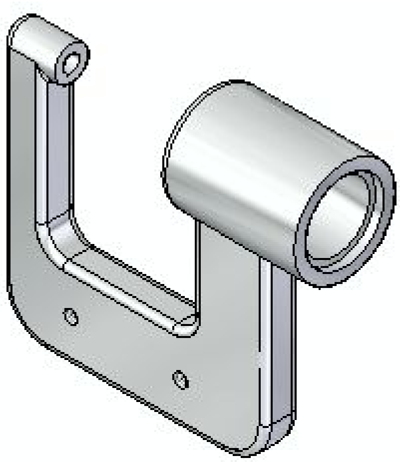
When working in a drawing derived from a 3D part or assembly, open the part or assembly document directly from the drawing by double-clicking any drawing view.

- ☐ Ensure the **Select** command is active .

- Position the cursor over the isometric drawing view shown in the illustration, then double-click to open the part model.

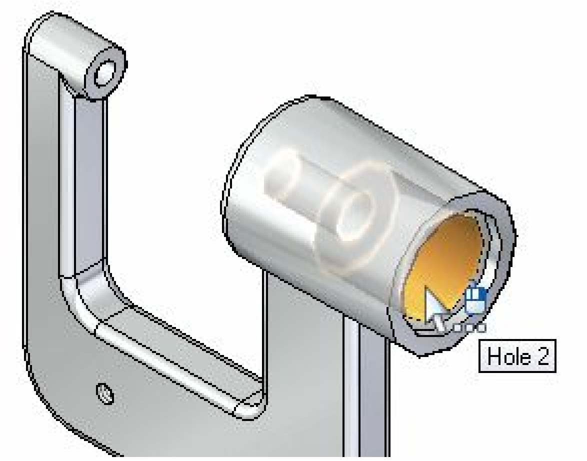


The part model document opens for editing.



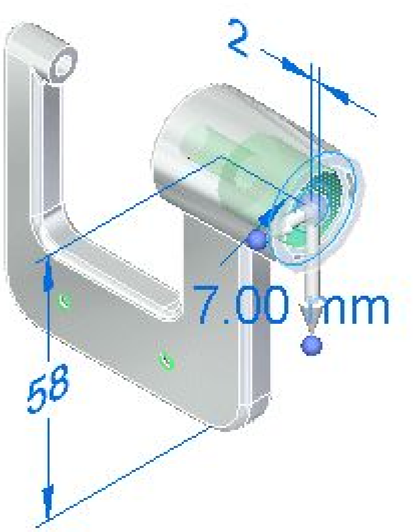
Select a feature to edit

□ Position the cursor over the hole feature shown below. When it highlights, click to select it.



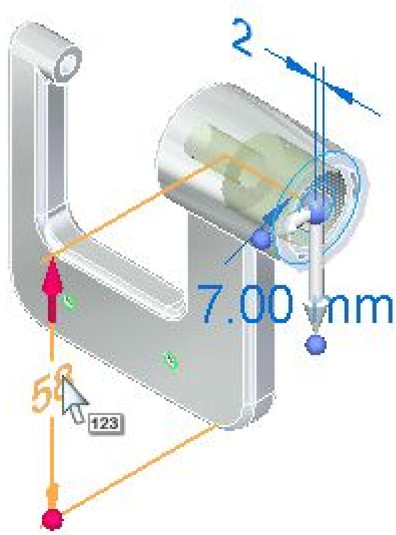
Notice that the dimensions associated with the hole feature automatically display, as shown below.

□ If necessary, on the viewing commands toolbar, choose **Fit**  to fit the dimensions to the view.



Select the dimension to edit

□ Position the cursor over the dimension text as shown below, but do not click.



□ Move the cursor slowly up and down over the dimension text.

Notice that the display of the dimension updates to show a large red arrow on one end of the dimension, and then the other as the cursor moves.

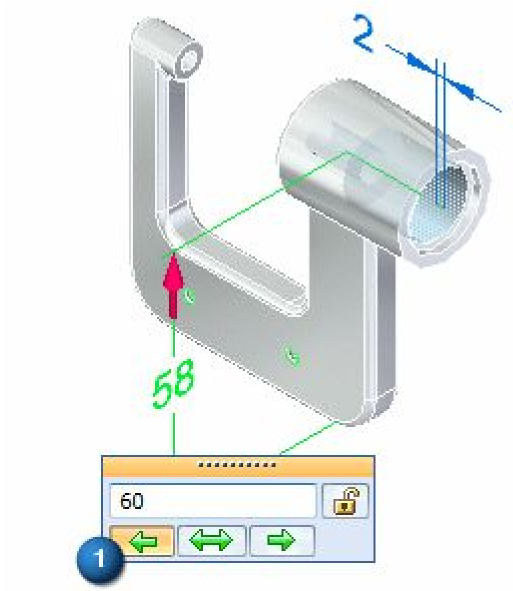
Also notice that the model face on the end of the dimension with the red arrow highlights. These display changes indicate which end of the model will change when editing the dimension value.

□ Position the cursor such that the red arrow displays at the top end of the dimension as shown above, then click to select the dimension.

The Edit Dimension Value box displays.

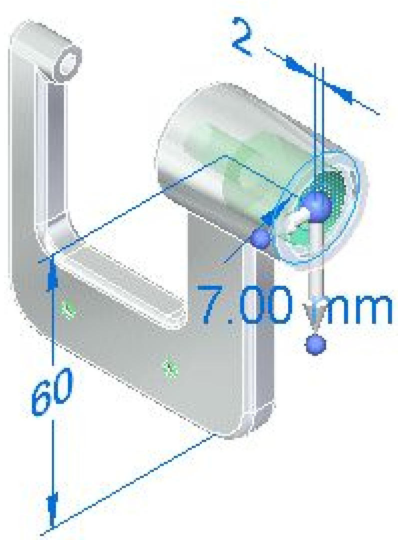
Edit the dimension

- ☐ Ensure the Edit Direction option (1) is set, as shown below.



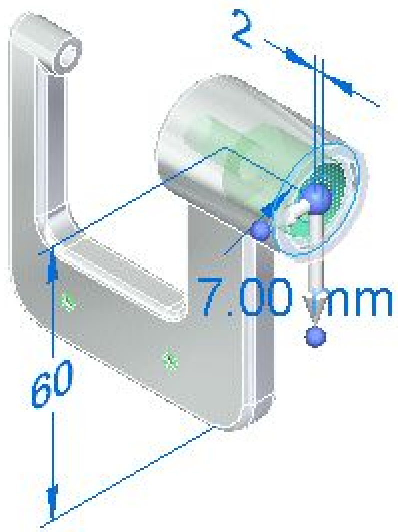
- ☐ Type **60**, then press Enter to define the new location of the hole.

Observe the results



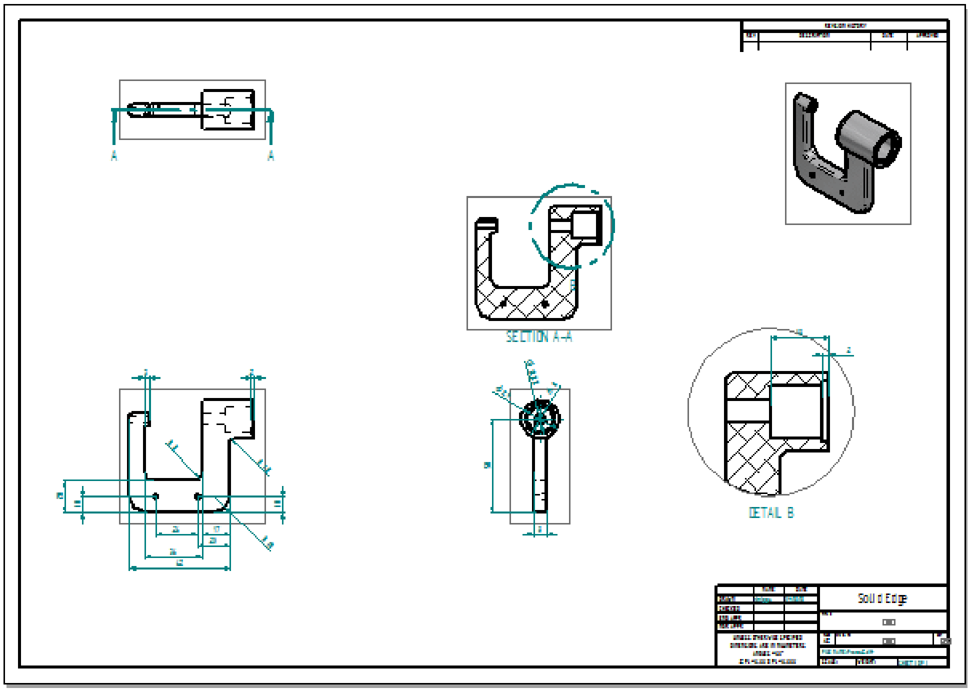
Notice that the top end of the model changed in response to the dimension value edit.

Save and close the part document



- On the **Quick Access** toolbar, choose **Save** to save the edited part.
- Close the part document by clicking the X next to the filename on the bar above the view.

Observe the drawing views



When returning to the drawing, the system prompts that drawing views are out of date. Click **OK** to dismiss this message.

Notice that a gray outline displays around each drawing view. The gray outline around each drawing view means the views are out of date with respect to the part model.

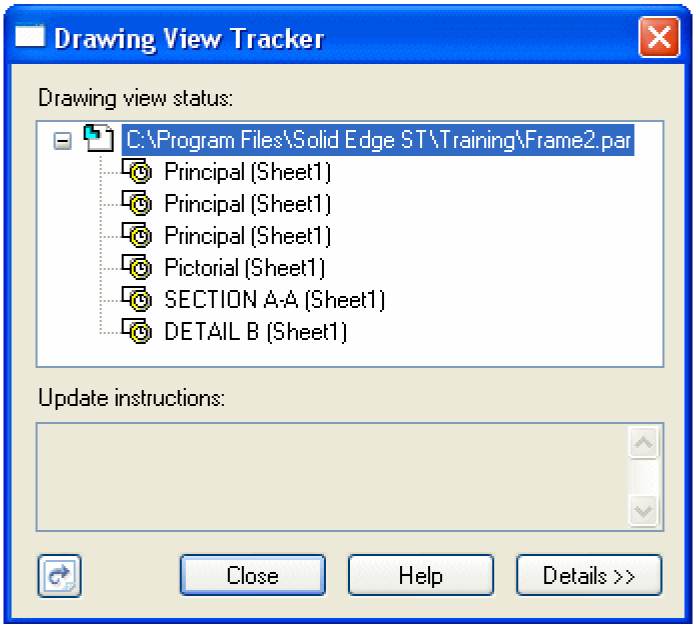
A change to the model caused the drawing views to go out of date.

In the next few steps, learn about the tools available for tracking drawing view and dimension changes in a drawing.

Update the drawing views

□ Choose **Tools** tab→**Assistants** group→**Drawing View Tracker** .

The **Drawing View Tracker** dialog box displays, listing all of the drawing views on the drawing.



The icon to the left of each drawing view entry indicates that a view is out of date.

Use the cursor to select an entry in the list, and the view highlights on the drawing sheet.

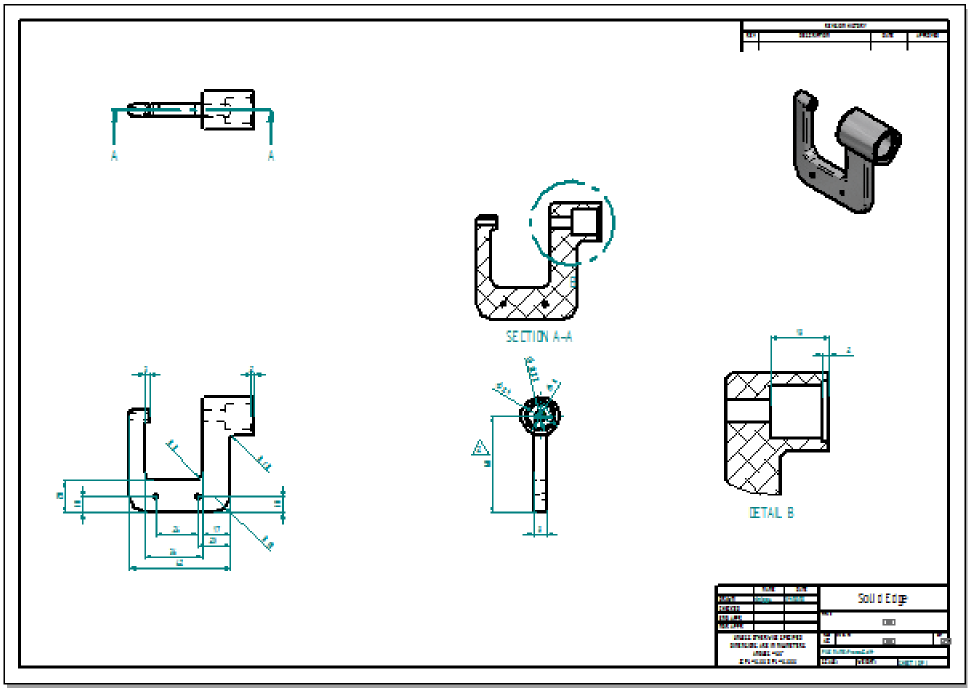
- □ At the bottom of the **Drawing View Tracker** dialog box, click the **Update Views** button to update all of the views at once.

Notice that the out-of-date icon in front of each drawing view name has been replaced by a new symbol , which indicates that the drawing views are up to date.

- □ On the **Drawing View Tracker** dialog box, click **Close**.

The **Dimension Tracker** dialog box is automatically displayed. Learn more about this in the next few steps.

Observe the drawing views and Dimension Tracker

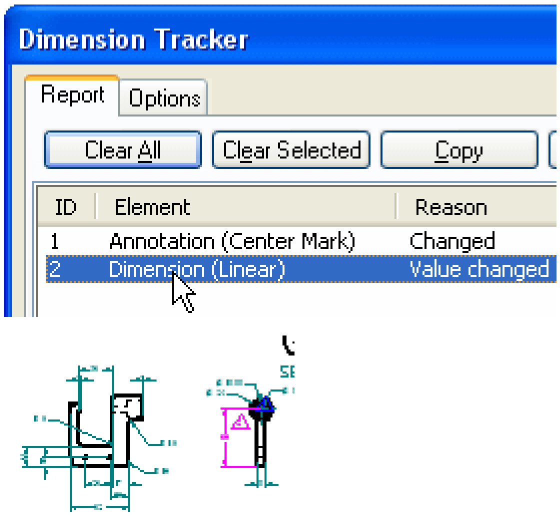


Notice that the gray boxes around the drawing views no longer display, as shown above. This indicates the drawing views are up-to-date.

- □ Observe the **Dimension Tracker** dialog box. Notice that there is an entry for the drawing dimension that corresponds to the model dimension edited.
- □ If required, move **Dimension Tracker** to see all the drawing views.
- □ Click the dimension entry in **Dimension Tracker** and notice that the changed dimension highlights, and that a revision triangle displays adjacent to the dimension.

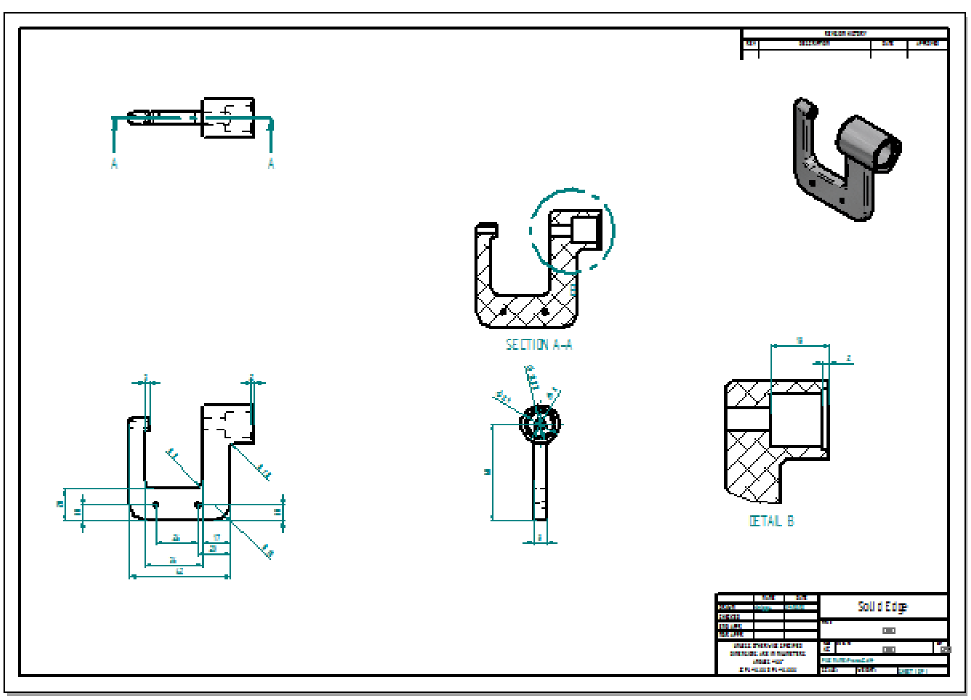
Leave the revision triangle displayed, click **Clear Selected** to clear the triangle for the selected dimension, or click **Clear All** to clear all of the revision triangles.

□ On the **Dimension Tracker** dialog box, click **Close**.



Dimension Tracker ensures awareness of even the smallest design change during drawing updates. Choose to discard the revision marks after reviewing the drawing or save them as revision notes. The shape of the revision balloon can be changed using the **Options** tab.

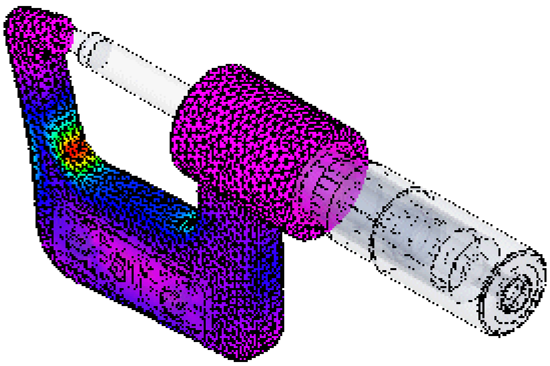
Save the drawing



□ On the **Quick Access** toolbar, click the **Save**  button to save the completed drawing.

□ Close the file.

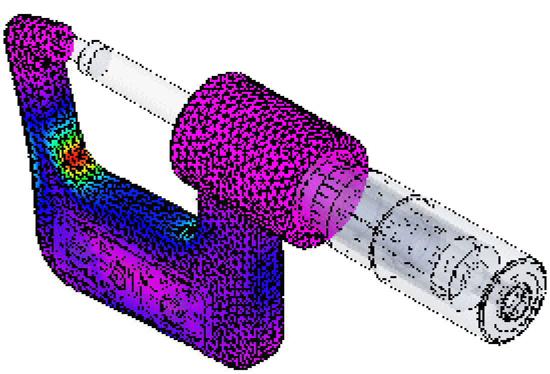
Chapter 4: Introduction to simulation



This tutorial provides step-by-step instructions for simulating typical loading of different configurations of the micrometer model, viewing results, and generating a report. While creating this simulation, learn techniques such as:

- Creating a study and selecting the model geometry to analyze.
- Applying forces and constraints.
- Meshing and solving an analysis.
- Generating analysis reports.

Simulation workflow overview



Solid Edge Simulation is a finite element analysis (FEA) application designed specifically for engineers and designer-analysts who use Solid Edge. In a structural simulation, FEA helps to visualize where structures bend or twist, and it indicates the distribution of stresses and displacements. This information helps to minimize weight, materials, and costs.

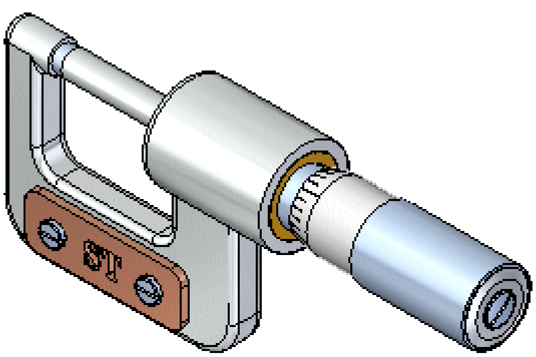
FEA enables entire designs to be constructed, refined, and optimized before the design is manufactured. This reduces the time-to-market and the costs incurred in rework.

The Simulation process works by analyzing a model, reviewing and analyzing results, and then modifying and reusing analysis studies. This tutorial shows enough of the overall workflow to help build confidence in learning the rest without assistance:

Analyzing a model

- Open the part, sheet metal, or assembly model to analyze.
- Create a study and select the model geometry to analyze.
- Optionally add or remove geometry from the study.
- Define loads to apply to the model.
- Define constraints on the model.
- For an assembly model, add connections between part faces.
- Mesh the model.
- Optionally refine the mesh by applying mesh sizing controls to edges, surfaces, or bodies.
- Solve the analysis.
- Review the analysis results.

Open an assembly

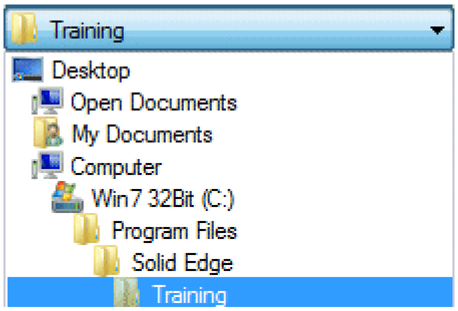


□ On the **Application** menu, click **Open**.

The **Open File** dialog box displays.

□ Set the Look In field to the Solid Edge training folder. The default location is:

C:\Program Files\Solid Edge ST9\Training



□ Set the File Name field to **stoadmm.asm**.

□ Click **Open** to open the file.

Create a new study

In the part and assembly tutorials, by changing the shape of the micrometer frame, it was seen how easily you can modify synchronous features. Now simulate the effect of forces on two different shapes of the micrometer frame.

The force to simulate is that of the spindle on the frame during the use of the device to measure distance.

A simulation study ties together all of the things that go into an analysis.

□ Choose the **Simulation** tab→**Study** group→**New Study** command .

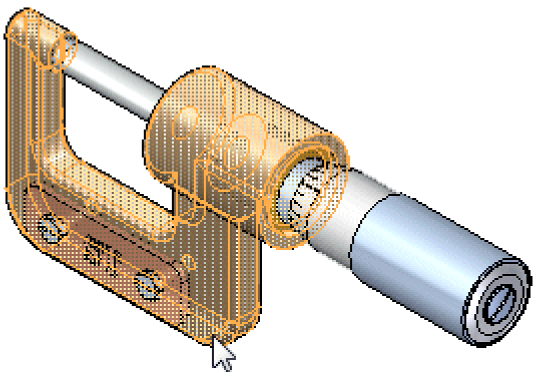
□ In the **Create Study** dialog box, set the Study type to **Linear Static**, set the **Mesh** type to **Tetrahedral**, and then click the **OK** button.

A Linear Static analysis is appropriate to calculate displacements, strains, stresses, and reaction forces under the effect of applied loads.

Tetrahedral meshing breaks the part into a number of smaller volumes for analysis. For most models, tetrahedral meshing is appropriate.

The **Define** command bar displays to select the geometry to which the analysis will be applied.

- ◻ Select the frame of the micrometer, **Frame3.par**.

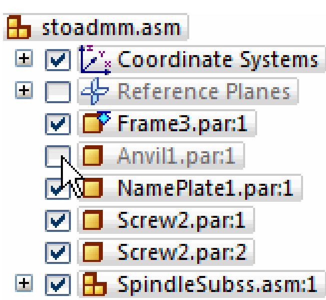


- ◻ On the **Define** command bar, click Accept .

Place a force

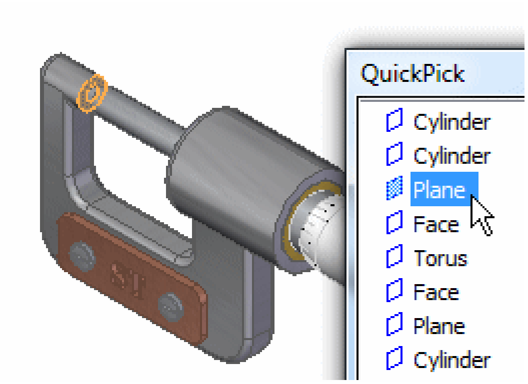
As the spindle is tightened while measuring an object, a force is applied to the left side of the frame. To make this easier to see, hide the anvil that the spindle bears upon.

- ◻ In **PathFinder**, hide the anvil by clearing the check box next to the Anvil1.par file.



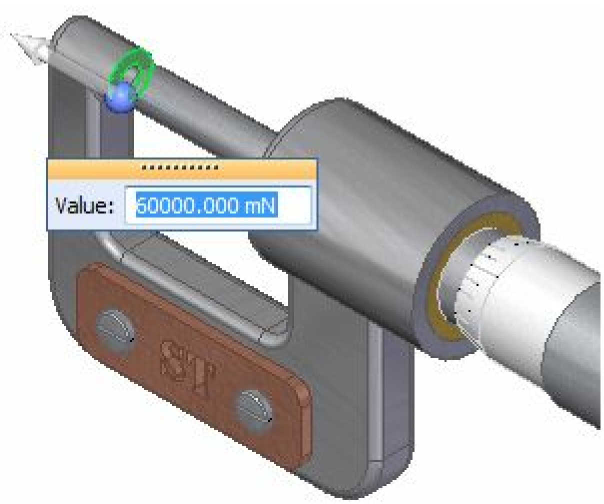
- ◻ Choose **Simulation** tab→**Structural Loads** group→**Force** .

- Use **QuickPick** to select the face of the frame shown in the illustration.

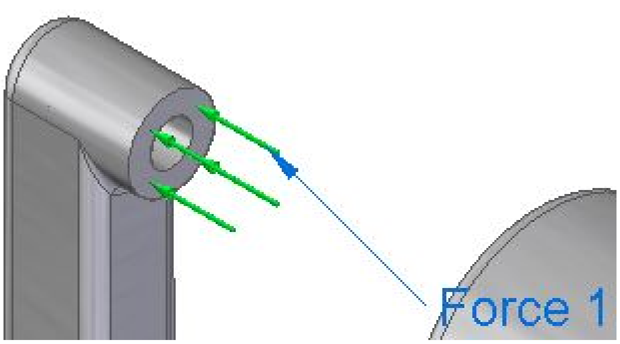


- On the **Force** command bar, click the **Flip** button  to set the direction of the force against the face.

- In the **Value** dialog box, type **6000**, and then press Enter

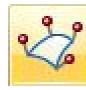


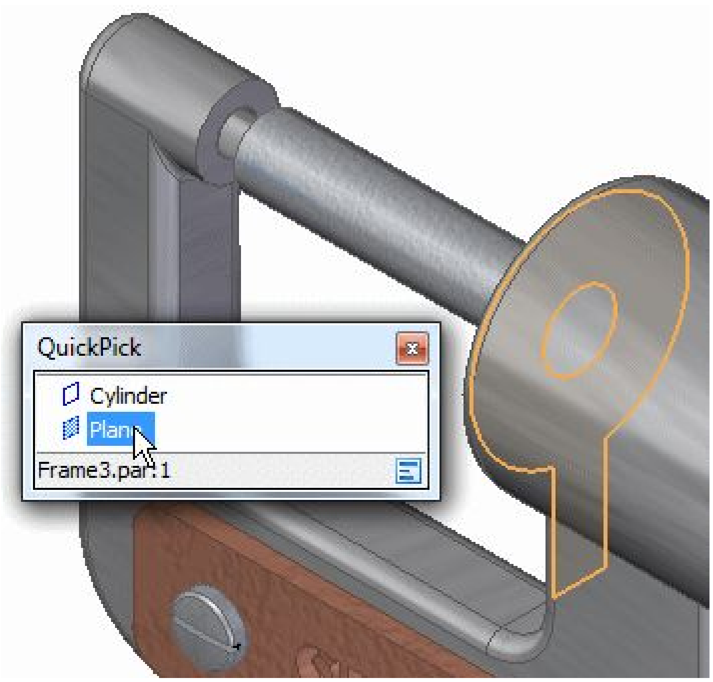
Notice that force symbols display on the face selected.



Place a constraint

Constraints identify portions of the model that are fixed. In this case, the right face of the frame is held in place by the person using the micrometer.

- Choose **Simulation** tab→**Constraints** group→**Fixed** .
- Use **QuickPick** to select the face shown in the illustration below.



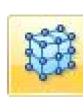
- Right-click to accept the selected face.

Notice that a fixed constraint has been applied to the face.

- Click anywhere in the graphic window to complete the command.

Mesh and solve the study

Apply a mesh, a system of grid points that overlay the model geometry, and solve the study.

- Choose **Simulation** tab→**Mesh** group→**Mesh** .

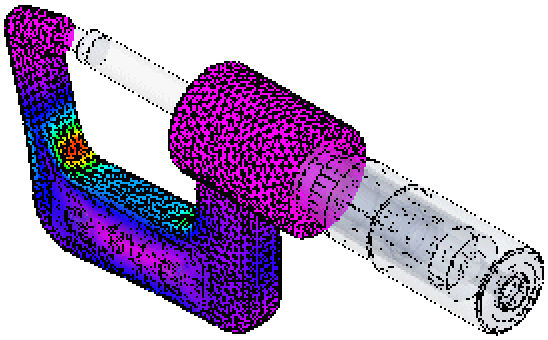
The **Tetrahedral Mesh** dialog box displays, which contains controls characteristics of the mesh. The default values are appropriate for this study.

- On the **Tetrahedral Mesh** dialog box, click the **Mesh & Solve** button.

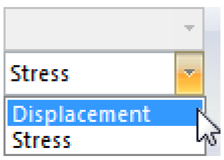
The system processes the information defined and solves the study.

When the solve is complete, results display on the model, and the **Simulation Results** ribbon displays.

Notice that a graph of Von Mises stresses display adjacent to the model.



□ Choose **Home** tab→**Data Selection** group→**Displacement**.



Notice that the graph of stresses is replaced with a graph of normalized displacement values for the simulation.

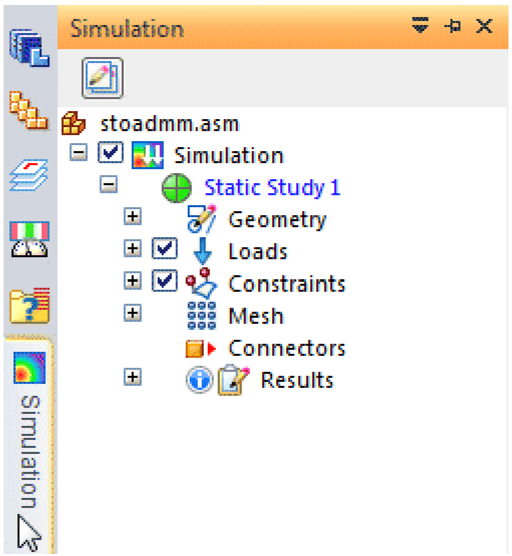
□ After reviewing the results, choose **Home** tab→**Close** group→**Close Simulation Results** .

Hide the force and constraint

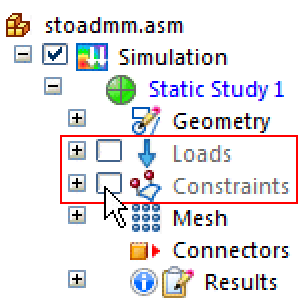
The force and constraint applied to the frame still display. Hide them to make the display clearer.

The window that now displays the **PathFinder** pane can display other panes of information as well.

- On the side (depending on the user interface theme selected, it might be the left or right side) of the Solid Edge window, click the **Simulation** button to display the Simulation pane.

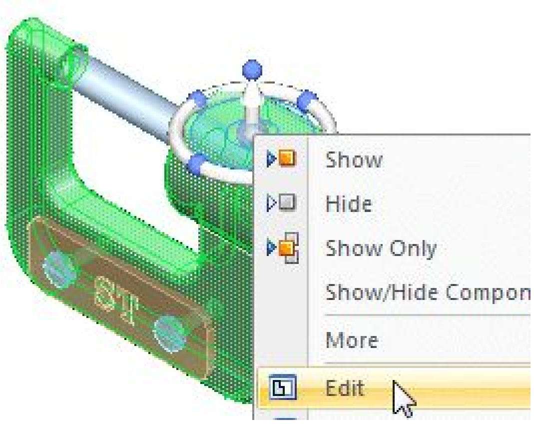


- On the **Simulation** pane, clear the check boxes that control the display of loads and constraints.



Edit the frame part

- ☐ Right-click **Frame3.par**, and on the short cut menu, click **Edit**.

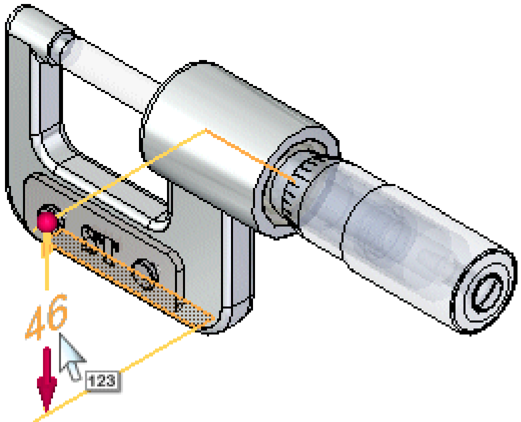


Select a dimension

Edit the height of the vertical frame members.

- ☐ On the **Home** tab, the Select tool should already be running.

- ☐ Position the cursor near the 46 mm dimension such that the red arrow displays at the bottom end of the dimension as shown below, then click to select the dimension.

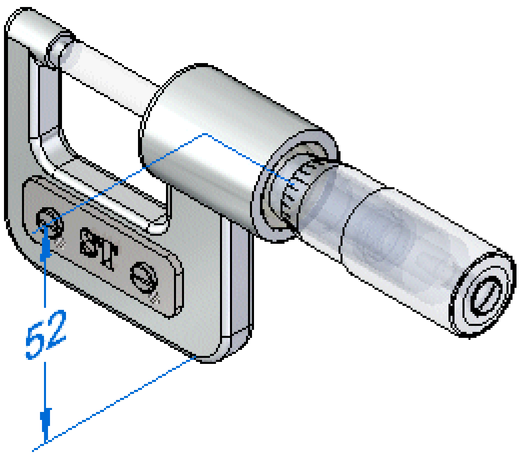


The **Edit Dimension** value box displays.

Edit the dimension value

- □ In the **Edit Dimension** value box, type **52** and press Enter.

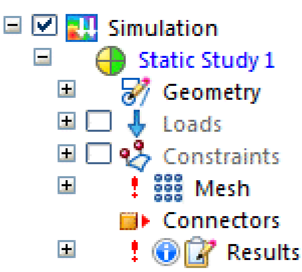
Notice that the height of the frame changes.



- □ Choose **Home** tab→**Close** group→Select→**Close and Return** to close the part document and return to the assembly.

Update the mesh and solution

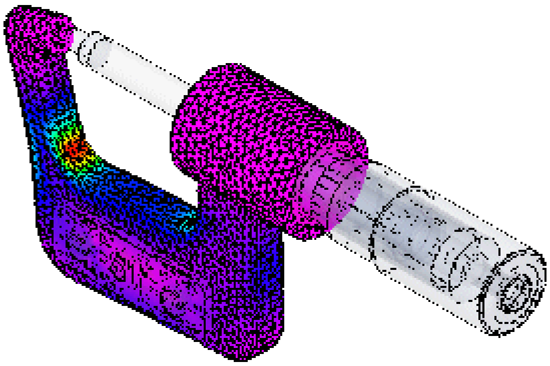
- □ Display the **Simulation** pane, and notice that an out of date symbol **!** displays next to the mesh and results.



This symbol is an alert to the fact that the change made to the geometry of the frame makes the mesh and the results of the first study out of date.

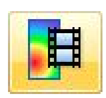
□ In the **Simulation** pane, right-click the mesh and choose **Mesh**, then, on the **Tetrahedral Mesh** dialog box, click **Mesh & Solve**.

The system processes the information again that is defined and solves the study. When the solve is complete, results display on the model, and the **Simulation Results** ribbon displays.



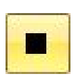
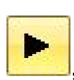
□ Review the Von Mises stress and the displacement graphs to evaluate the impact of the geometry change made to the frame.

Animate the study

□ Choose **Home** tab→**Animate** group→**Animate**  to display the part deflection under the loading and constraint conditions defined.

As the animation displays stress results over time, notice the options available on the command bar for controlling the characteristics of the display.

□ On the command bar, do the following:

- Click the **Stop** button  to stop the display.
- Set the Frames per Second to 2.
- Set the Cycle length to 10 seconds.
- Click the **Play** button , and observe how the animation changes.

□ On the command bar, click the **Close** button to end the animation.

Create a report of the study

- ☐ Choose **Simulation** tab→**Results** group→**Results**.

- ☐ Choose **Home** tab→**Output Results** group→**Create Report**  button.

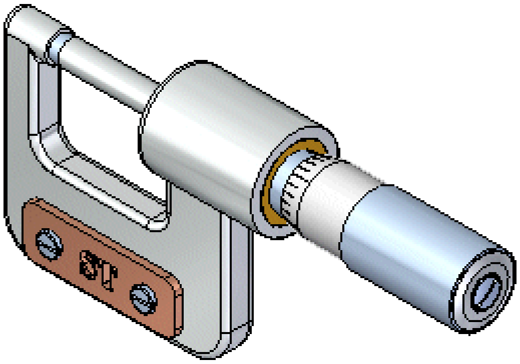
The **Create Report** dialog box displays.

- ☐ On the **Create Report** dialog box, type values for the Report title, Introduction, and Conclusion, and notice the Report Location; then choose the **Create Report** button.

The system processes the simulation results and report contents, and then displays the report in a browser window.

- ☐ Review the contents of the report, and then in the Solid Edge window, choose **Home** tab→**Close** group→**Close Simulation Results** .

Save the assembly



- ☐ On the **Quick Access** toolbar, click the **Save** button to save the assembly.

Congratulations!

This completes the first simulation study in Solid Edge.

To learn more about Solid Edge, do the following:

- Experiment with editing parts in the assembly model and performing additional simulation studies.

- Experiment with simulation studies of other models in the Solid Edge training folder.

- From the Solid Edge Help pane, select Solid Edge Simulation Help, and explore the practice activities to learn more about Simulation capabilities.

Siemens Industry Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 7 million licensed seats and 71,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.