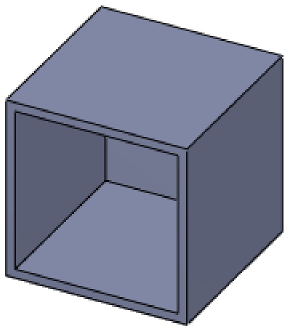
Face<1> appears in the PropertyManager under **Faces to Remove** .

4. Click .
   The box is hollow with walls that are 5mm thick.



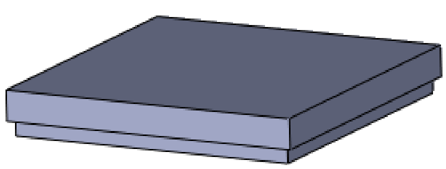
The box is complete.

Saving the Part

1. Click **Save** (Standard toolbar) or **File** > **Save**.
2. In the Save As dialog box:
   a) Browse to the location where you want to save the document.
   b) For **File name**, type `box`.
   c) Click **Save**.
   The part is saved as `box.sldprt`.
3. Keep the part open.

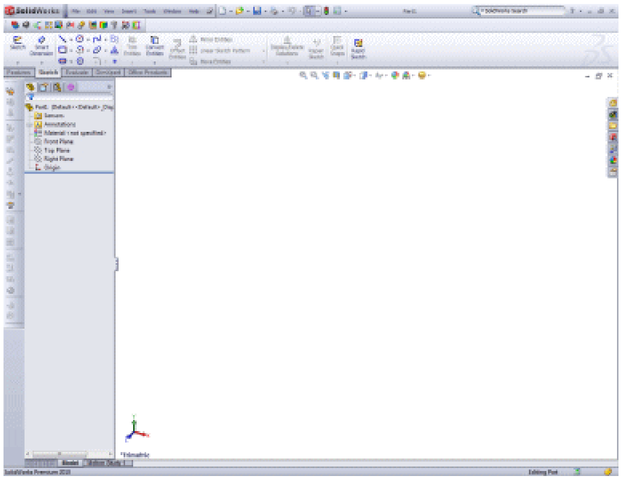
Creating a Lid for the Box

You created the first part, a box. Now you need to create a second part to make a lid for the box.



Opening a New Part

1. Click **New** (Standard toolbar) or **File** > **New**.
2. In the New SOLIDWORKS Document dialog box, click **Part** and click **OK**.
   A new part document opens.



Setting the Drafting Standard and Units

Before you begin modeling, you set the drafting standard and unit of measurement for the part.

1. Click **Options**  (Standard Toolbar) or **Tools** > **Options**.
2. In the System Options - General dialog box, select the Document Properties tab.
3. In **Overall drafting standard**, select **ISO**.
4. On the left pane, click **Units**.
5. Under **Unit system**, select **MMGS** to set the unit of measurement to millimeter, gram, second.
6. Click **OK**.

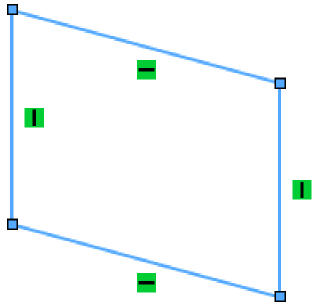
Sketching a Rectangle

The lid for the box is shaped like a square. In this procedure, you sketch a rectangle. Later you can dimension it to fit the box.

1. Click **Corner Rectangle**  (Sketch toolbar) or **Tools** > **Sketch Entities** > **Rectangle**.

   The PropertyManager prompts you to select a plane on which to sketch the rectangle.
2. Click the **Front** plane.

3. Click, then drag the pointer to create a rectangle.
4. Click to complete the rectangle.

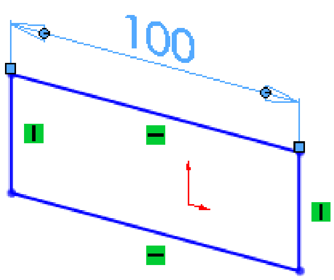


Dimensioning the Sketch

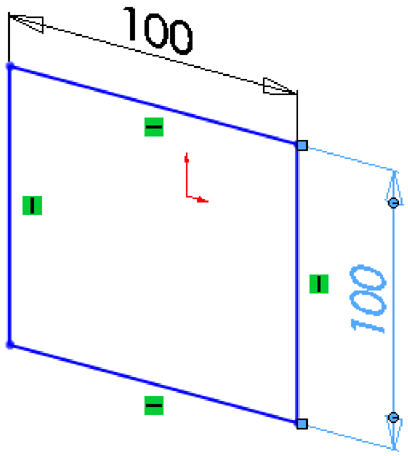
Now that you have a sketched rectangle, you need to dimension it so it has the proper measurements.

1. Click **Smart Dimension** (Dimensions/Relations toolbar) or **Tools** > **Dimensions** > **Smart**.

   The pointer changes to 
2. Select the top horizontal line in the rectangle.
   A dimension appears.
3. Drag the dimension upwards and click to place it.
4. In the Modify dialog box, type `100` and click .



5. Repeat steps 2 through 4 for the right vertical line in the rectangle.



6. In the upper right corner of the window in the Confirmation Corner, click the sketch icon .

   Sketch mode is turned off.

Extruding the Sketch

After dimensioning the 2D sketch, you can extrude it to make a 3D solid model.

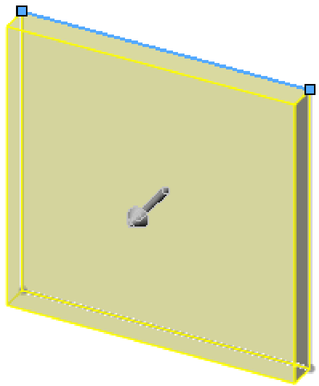
1. Click **Extruded Boss/Base**  (Features toolbar) or **Insert** > **Boss/Base** > **Extrude**.

   Depending on what is selected in the graphics area, the following occurs:

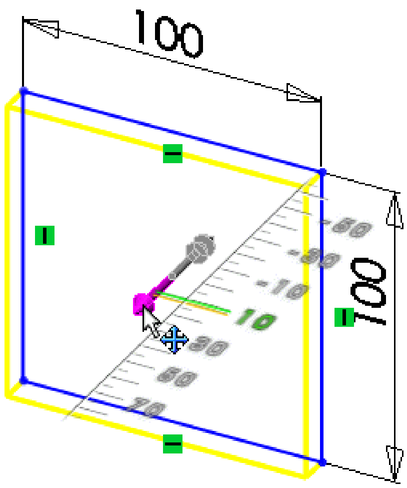
   - If the sketch is selected, the Boss-Extrude PropertyManager appears and a preview of the extrude appears.
   - If the sketch is not selected, the Extrude PropertyManager appears and indicates that you need to select a sketch.

2. If the Extrude PropertyManager appears, select the sketch by clicking any line in the square. Otherwise, go to the next step.

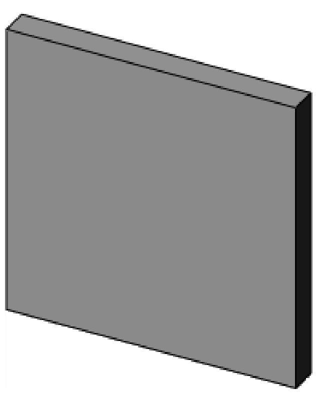
   A preview of the extrude appears.



3. In the graphics area, click the handle (arrow) and drag it until you reach 10 on the scale, then click in the PropertyManager.



The 2D sketch changes to a 3D model.

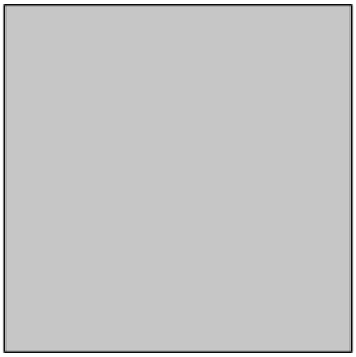


Creating a Lip on the Cover

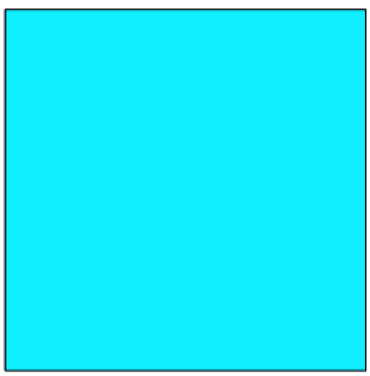
To ensure that the cover fits tightly on the box, you create a lip on the cover using another extrude.

1. Press the spacebar or click **View** > **Modify** > **Orientation**.
2. In the Orientation dialog box, double-click ***Front**.

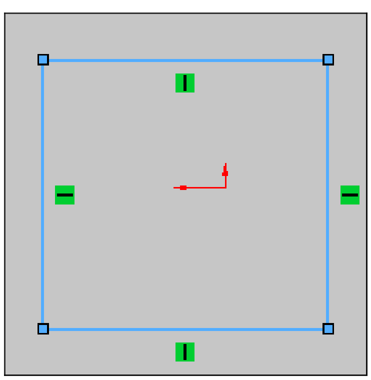
   The cover is rotated so the front is visible.

3. Click **Corner Rectangle**  (Sketch toolbar) or **Tools** > **Sketch Entities** > **Rectangle**.
4. In the graphics area, select the face as shown:

5. Sketch a rectangle on the face. It does not matter what size you make the rectangle; you can dimension it later.

Dimensioning the Sketch

You need to dimension the rectangle so it has the proper measurements.

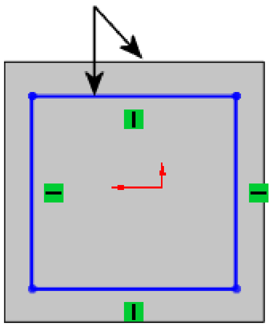
1. Click **Smart Dimension** (Dimensions/Relations toolbar) or **Tools** > **Dimensions** > **Smart**.

   The pointer changes to .

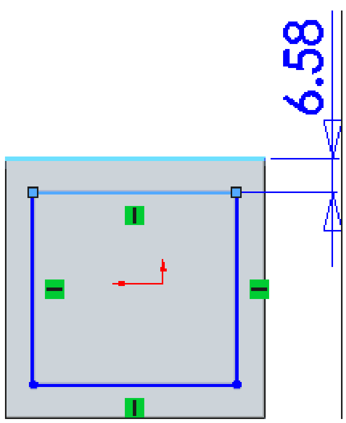
2. In the graphics area:

   a) Select the top horizontal line in the rectangle.

   b) Select the top edge of the extrude.

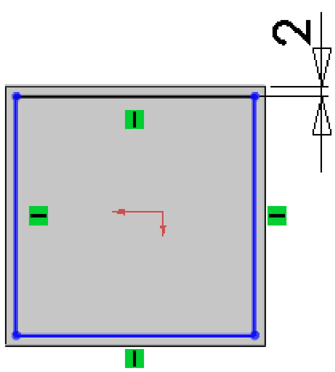


A dimension appears.

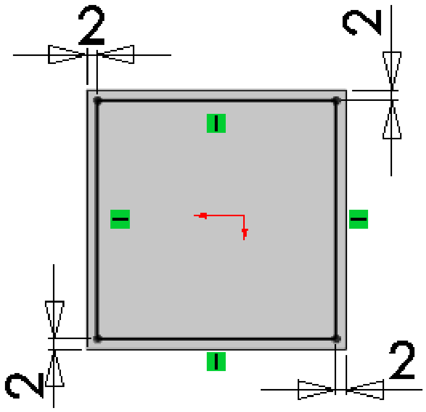


3. Drag the dimension upwards and click to place it.

4. In the Modify dialog box, type 2 and click .



5. Repeat steps 2 through 4 for the rest of the sketch:

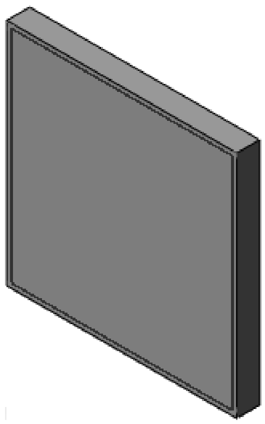


6. In the upper right corner of the window in the Confirmation Corner, click the sketch icon .
   Sketch mode is turned off.

Extruding the Sketch

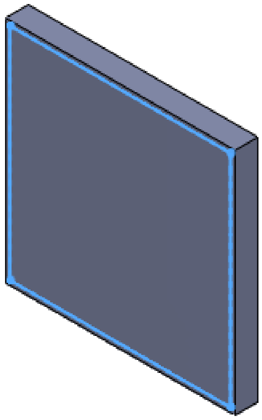
After dimensioning the 2D sketch, you can extrude it to make a lip for the lid.

1. Press the spacebar or click **View** > **Modify** > **Orientation**.
2. In the Orientation dialog box, double-click ***Isometric**.
   The cover is rotated.

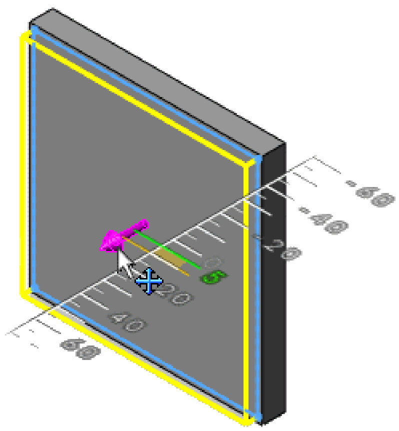


3. Click **Extruded Boss/Base**  (Features toolbar) or **Insert** > **Boss/Base** > **Extrude**.

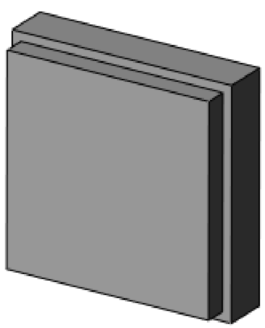
4. In the graphics area, select the sketch by clicking any line in the square.



5. In the graphics area, click the handle (arrow) and drag it until you reach 5 on the scale, then click in the PropertyManager.



The 2D sketch changes to 3D.



The lid is complete.

Saving the Part

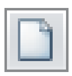
1. Click **Save**  (Standard toolbar) or **File** > **Save**.
2. In the Save As dialog box:
   a) Browse to the location where you want to save the document.
   b) For **File name**, type `lid`.
   c) Click **Save**.
   The part is saved as `lid.sldprt`.
3. Keep the part open.

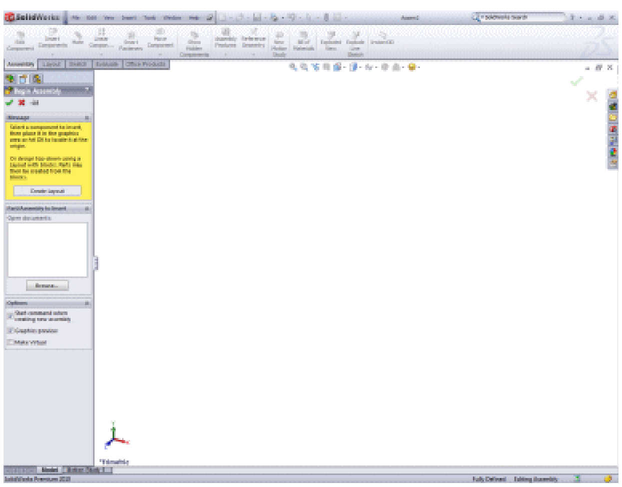
Putting the Box and Lid Together

An assembly is a collection of part documents. The part documents become “components” in the assembly, in this case, the box and lid.

Opening a New Assembly

In this procedure, you open a new assembly document where you will insert the box and lid models.

1. Click **New**  (Standard toolbar) or **File** > **New**.
2. In the New SOLIDWORKS Document dialog box, click **Assembly** and click **OK**.
   A new assembly document opens, and the Begin Assembly PropertyManager appears.



Inserting Parts into the Assembly

An assembly is a collection of parts. In this procedure, you insert the box and lid into the assembly, where they become components in the assembly.