

EXERCISE 1.3 – CREATING A CLOCK MECHANISM PART

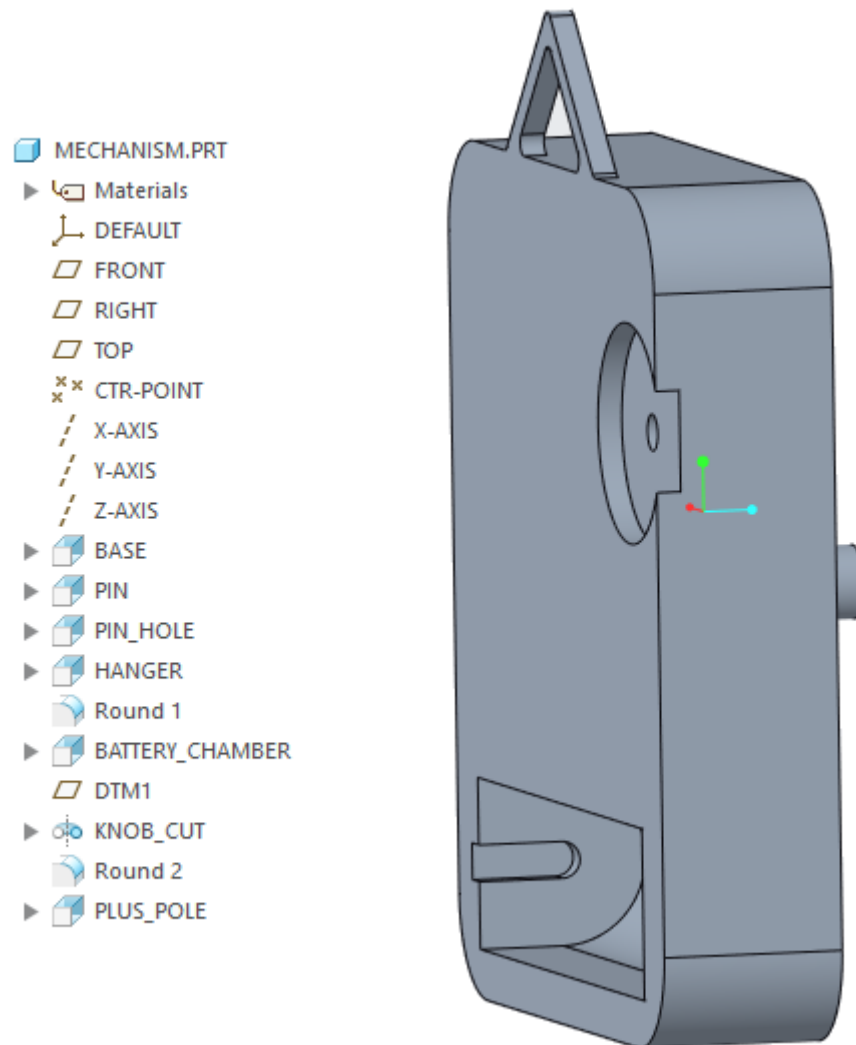


Figure 1: The clock mechanism part and its model tree.

Learning Targets

In this exercise you will learn:

- ✓ to create Datum planes.
- ✓ to choose references
- ✓ to use thicken Extrude
- ✓ to use Revolve to remove material.

In this exercise, we create a mechanism part. Program version is 6.0.2.0.

Design Plan

Let us decide that the model (see Figure 2) is oriented (with respect to the basic datum planes) similarly to the hand model, the positive side of the TOP datum plane pointing to the ceiling and the negative side of the FRON datum plane pointing to the wall, as the clock is hung on it. The intersection axis of TOP and RIGHT datum planes is coaxial with the rotation axis of the hands. The basic shape of the box is also symmetrical about TOP and RIGHT datum planes.

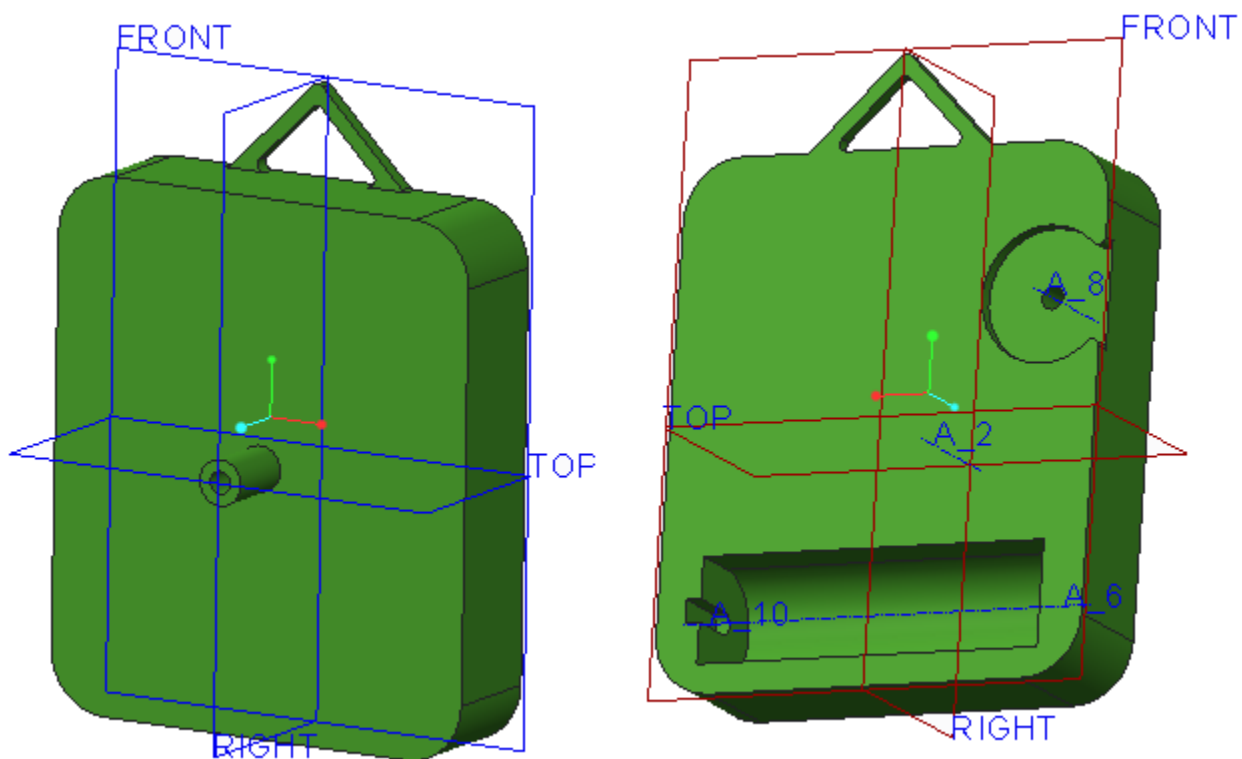


Figure 2: Model from front and back view. Positive plane side is highlighted in blue, negative in red.

The Basic Geometry

Create a new model (part, solid) and name it as **mechanism**.

Select **Extrude** (📐, from *Shapes* group), hold **RMB**, select **Define Internal Sketch** (✎).
Select **FRONT** as *Sketch Plane* (RIGHT orientation Right as reference) and **Sketch**.

Sketch

Our first rectangular sketch should be symmetrical about the default references. Start by sketching a **Corner Rectangle** (☐, *Sketching* group) that is about symmetrical, as in Figure 3. Ignore dimension values!

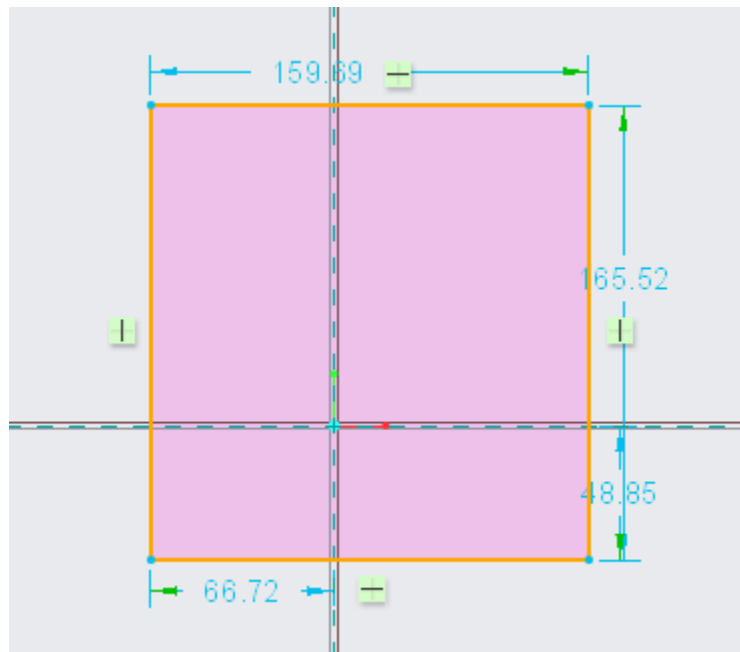


Figure 3: Sketched Rectangle. Ignore dimension values and locations.

To place rectangle symmetrical, we can use mid-points like in hand exercise (Exercise 1.1). But, we can also use symmetric constrain. To use that, we first need some centerlines. Select **Centerline** (☐, from *Sketching* group) and create centerlines to horizontal and vertical reference lines. Then, select **Symmetric** (↔, from *Constrain* group). Select one point from the top horizontal line, then vertical centerline, and other point from the horizontal line (Figure 4). Notice that arrow symbols appeared (↔). Do the same thing to vertical line and horizontal centerline.

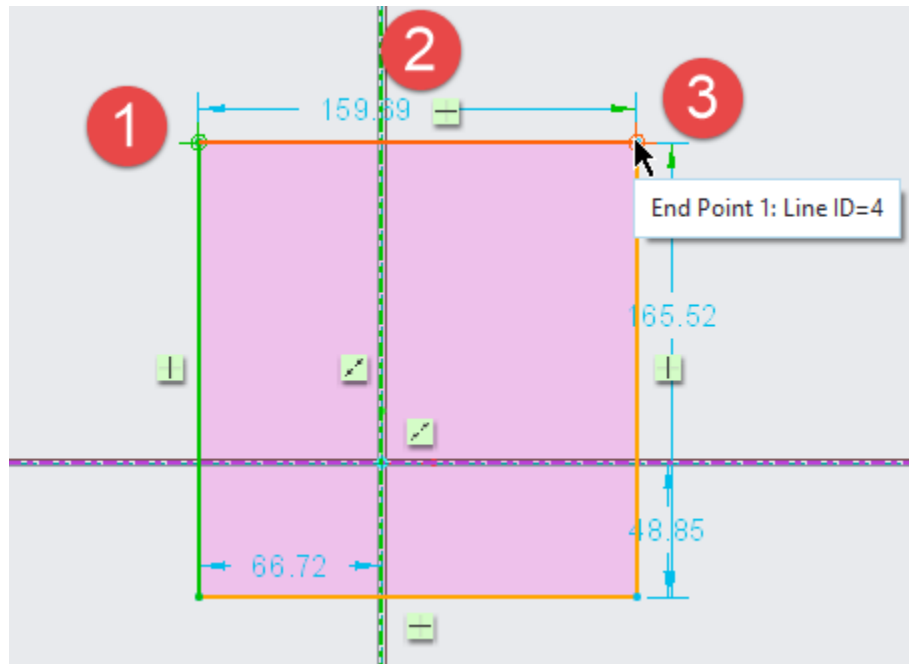



Figure 4: Selecting the last point (3.). Point (1.) and centerline (2.) already selected.

You should only have two dimensions, change them as seen in Figure 5. Accept the sketch ().

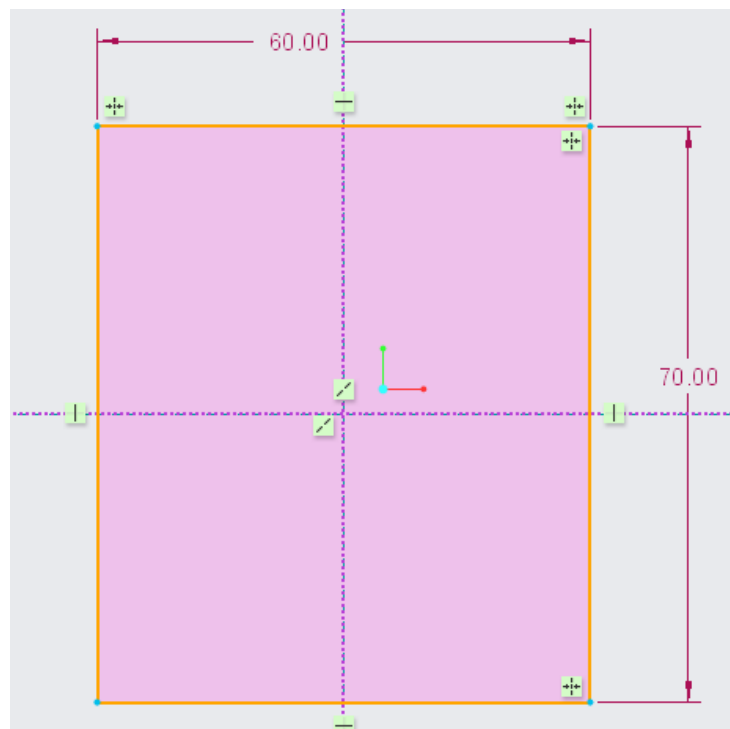


Figure 5: Ready-to-accept sketch.

Finalizing the feature

Enter **15** as the extrusion depth. Open **Properties** tab and give a new name to the feature, for ex. BASE (Figure 6). This is not needed, but the usability of the model is much better when features are smartly named.

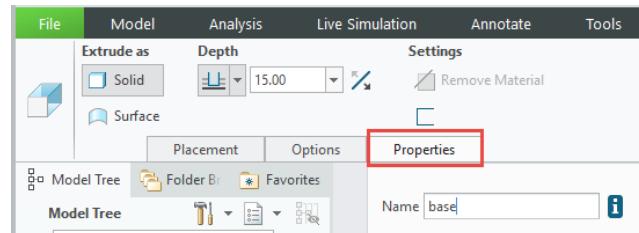


Figure 6: Feature named to BASE. Note: UPPERCASE is not needed, the program automatically changes all user renamed features to UPPERCASE.

When ready, accept the feature (✓) or **MMB**.

The Pin

Next we create a pin. Using **Extrude** (✎), create a sketch on the previously created surface (Figure 7).

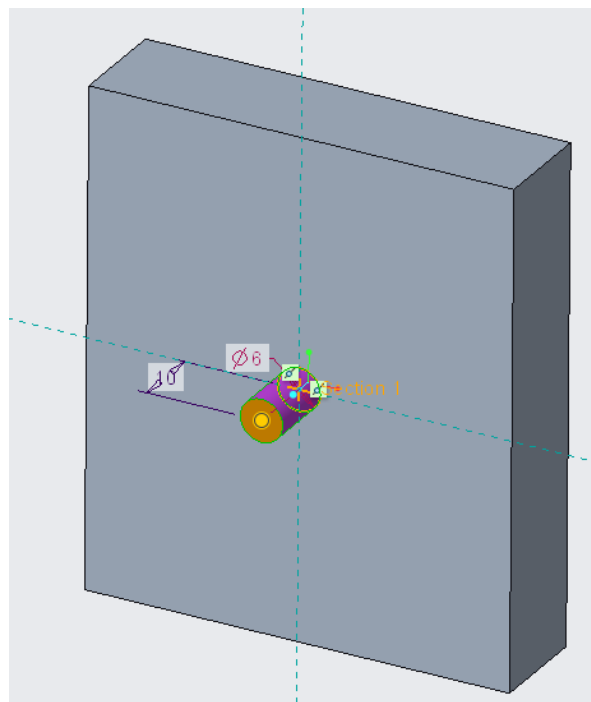




Figure 7: Created circular sketch (diameter of 6).

When the sketch is ready, accept it, extrude blindly **10** and accept the feature. Then select Extrude 1 (previous feature named as BASE) from model tree, click **RMB** and select  **Rename**. Rename feature as PIN. Save your model.

The Pin Hole

References

Although the next feature could be sketched using the default references, the references are selected manually for educational reasons. We are about to cut a hole into the previous feature, making it a tube. Create new **Extrude** () on the top of the previously created PIN feature (Figure 8).

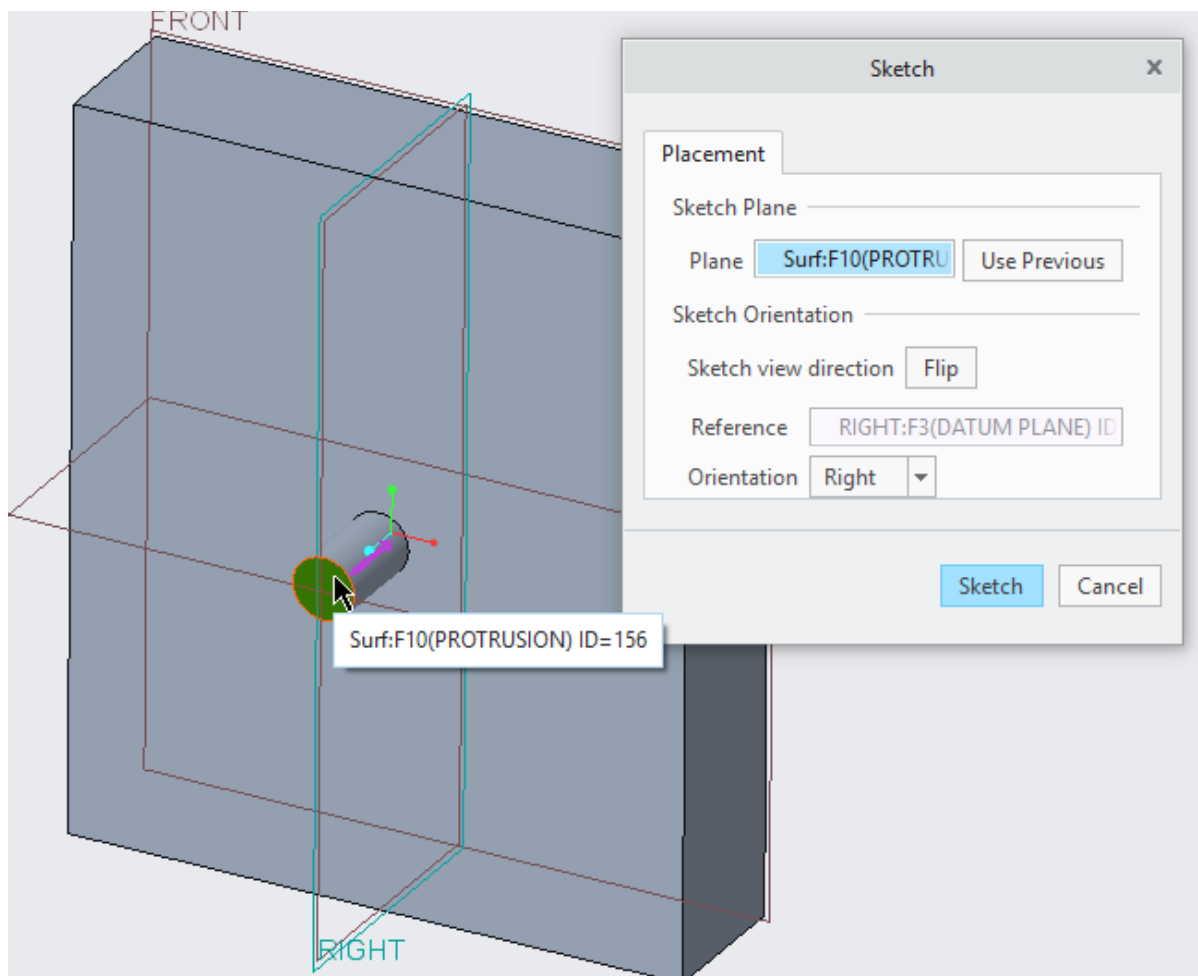


Figure 8: Location for the sketch highlighted in green.

Click **Sketch** to accept the *Sketch Plane*, but do not use the default references! You can access the references by selecting **References** (🔍) from *Setup* group. Select TOP and RIGHT datum planes one by one, and **Delete** them from the *References*.

Try closing the *References* window. Creo warns you about redundant references, answer **No** to *Missing References* window. This was just to show that PTC strongly prefers selecting references manually! Select the surface shown in Figure 9, make sure that the circle center (a cross) is there. Click **Solve** and ensure that *Reference status* is “Fully Placed”. **Close** the references window.

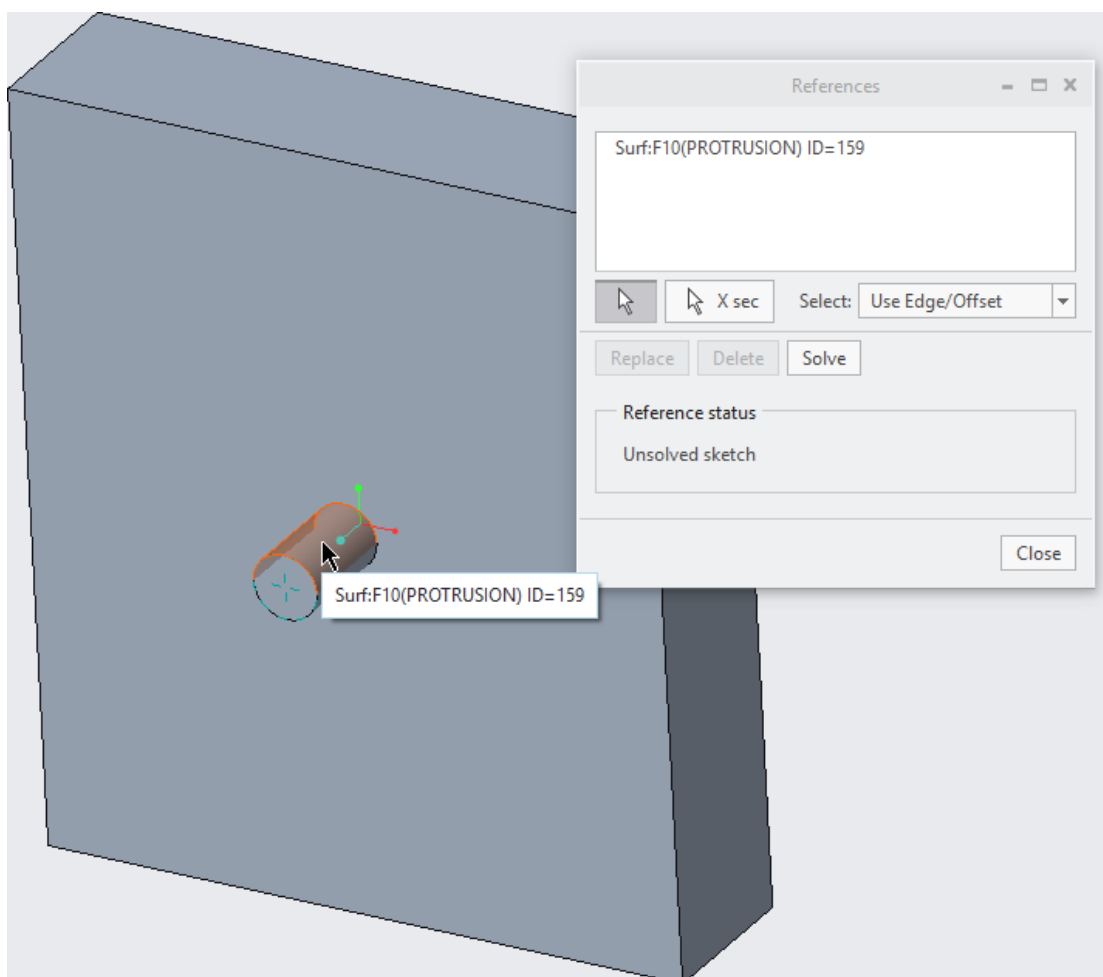


Figure 9: Selecting surface to references.

Sketch a **Circle** (🕒, *Sketching* group), concentric with the reference circle and having diameter of **3**. Accept the sketch. Click on **Remove Material** (🔪). Hold **SHIFT** and drag the extrusion depth handle until the handle snaps to the front face of the mechanism, see Figure 10.

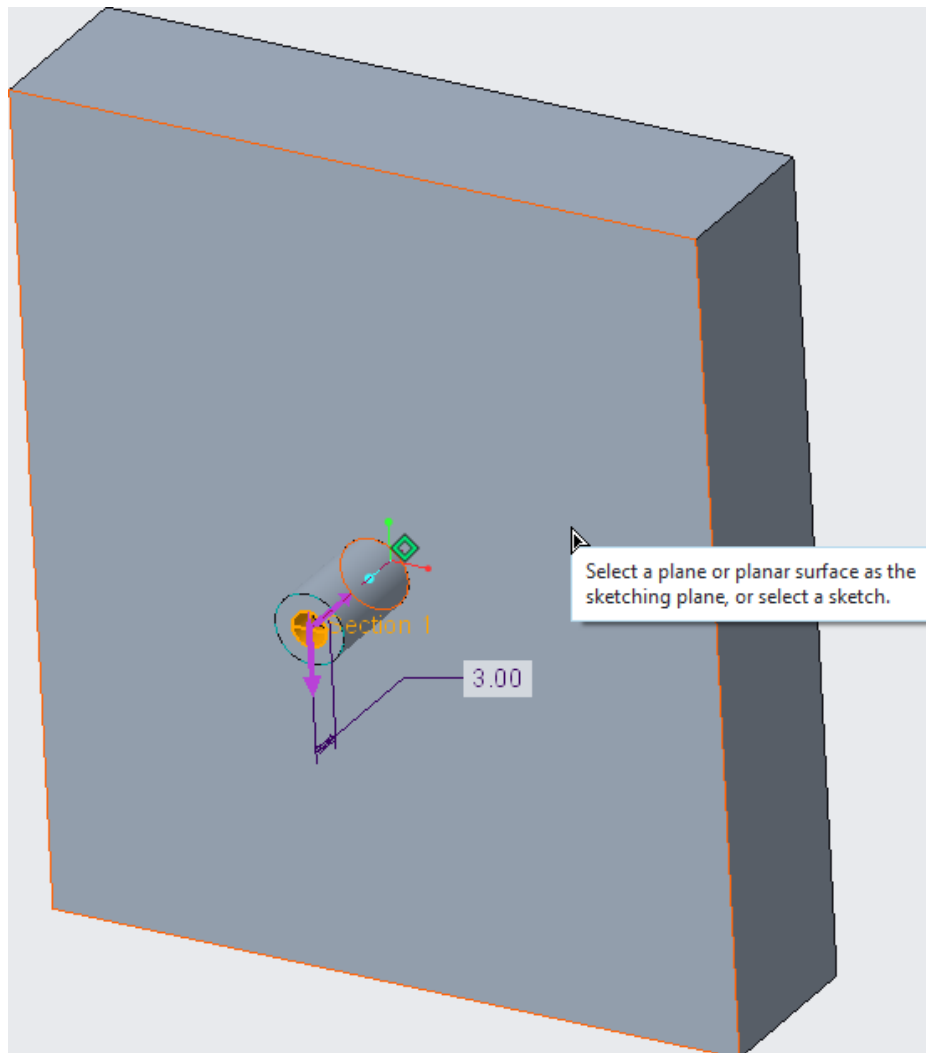


Figure 10: To Selected selected, selected face highlighted in turquoise.

The *Blind* method is automatically changed to *To Selected* in the dashboard. Enter the Verify mode (🔍 from the dashboard) to see a preview of the cut feature. If everything seems OK, accept the feature (**MMB**). Otherwise, exit the Verify mode (▶) and try flipping the arrows. Rename the feature as PIN_HOLE.

The Hanger

The next feature, an extrusion using thicken sketch option, will form a hanger. Start a new **Extrude** (📄), using FRONT as the sketch plane. Delete TOP from the reference list and select the top surface of the solid model as replacement (see Figure 11). (Our “top” face is pointing upwards on your monitor, since a minute ago you defined, probably paying no attention it, that the top datum plane will point to direction “top”. More about this later...).

Sketch two lines as in Figure 11. You may use different kind of constraints to achieve the given extents. In the case of this example, a centerline (⌀) was created on the vertical reference line, and a symmetry constraint (↔-symbol ↔) was created between the centerline and the two endpoints.

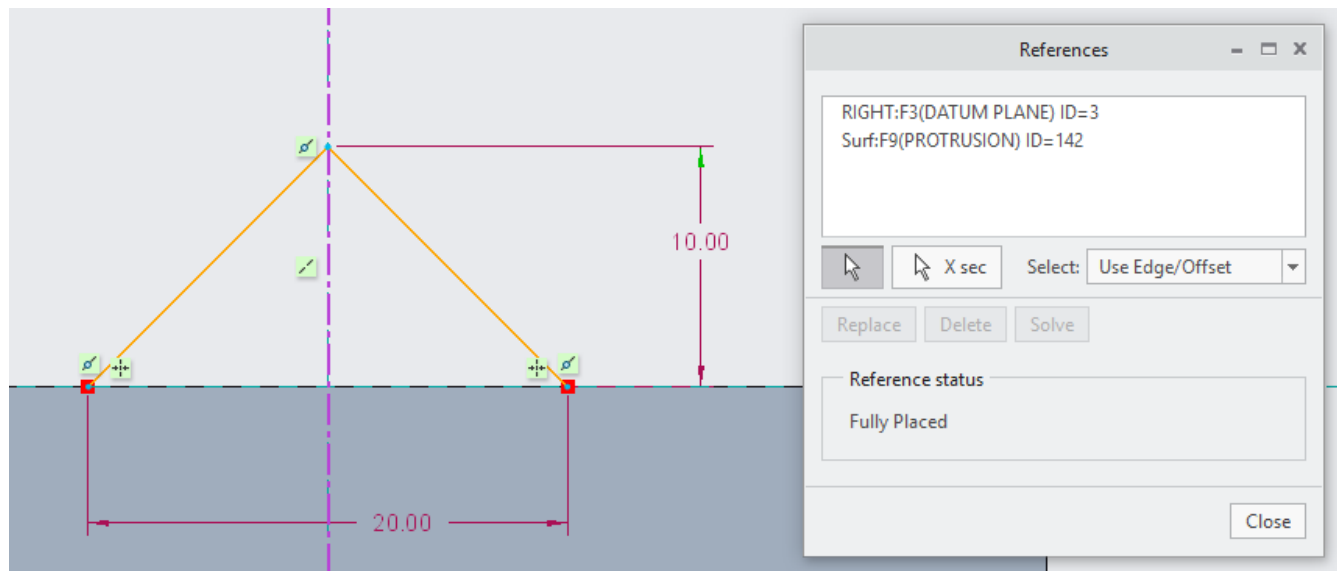


Figure 11: Sketch for the hanger. Notice the chosen references.

Accept the sketch (✓). In the dashboard, set **Thicken Sketch** (📄) option on and give **2** as a value. Give **2** also as a value for the length of the protrusion. Your model should look like in Figure 12.

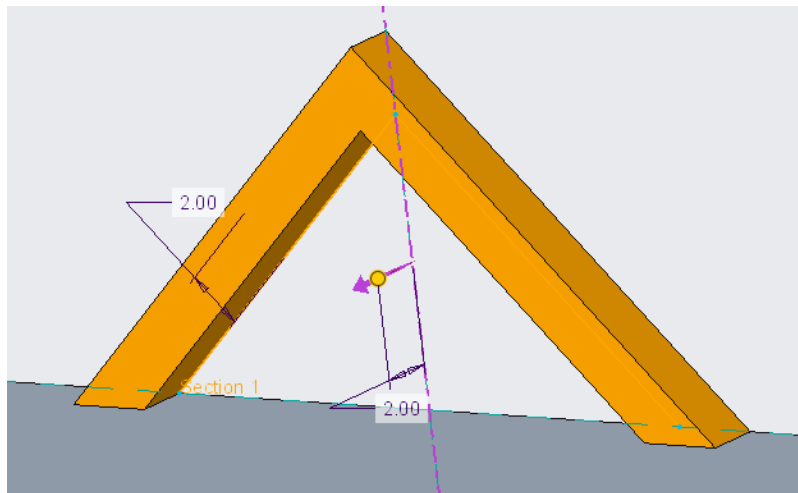


Figure 12: Ready-to-accept feature. Notice that the material is created outside the sketched geometry.

Notice, that material is added outside the used sketch. Click **Change Direction** (↔), the rightmost one in the dashboard) couple of times to see its affect. Leave it as seen in Figure 12. When ready, accept the feature (**MMB**). Rename feature as HANGER.

The Rounds

Create a **Round** feature (↻, *Engineering* group), using the edges shown in Figure 13 as references. Be sure to hold **CTRL** down while selecting, so that only one reference set is created! (Except if you want to have different radius values on different edges, of course.) Drag the handle to verify different radius values, 1 mm is OK.

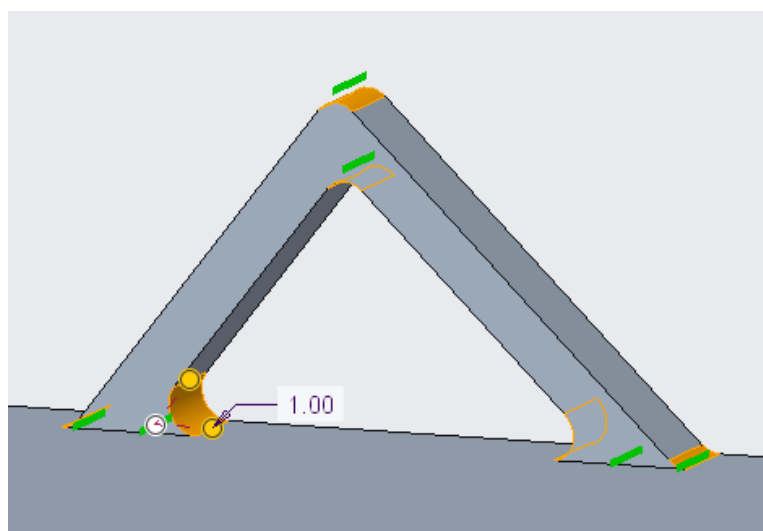


Figure 13: Selected edges for the rounding (highlighted in green).

The Battery Chamber

Next, we will cut a battery chamber. Start a new **Extrude** (📐) using RIGHT datum plane as sketching plane, accepting the default settings and references. What if one wanted the existing geometry to be oriented differently on the screen? For example, it would be nice to have the “back” face of our mechanism box to point upwards. (Make a note on how the model is oriented right now) The “up” direction on the screen can be set in the sketch plane definition window, but we already accepted the “wrong” default settings. To fix them, select **Sketch Setup** (🔧, Setup group). This opens the *Sketch Setup* window.

You probably had FRONT selected as the default reference in the *Sketch Orientation* section. Again, the *Orientation* option probably was “Left”. This means that the positive side of the reference, FRONT datum plane, will be pointing leftwards. Now, select a proper pair of *Reference Plane* and *Orientation* option, to make the “back” face point upwards. The alternatives are TOP + “Left” (as seen in Figure 14) and FRONT + “Bottom”. (To change reference, select the text field and select a new one). Click **Sketch**.

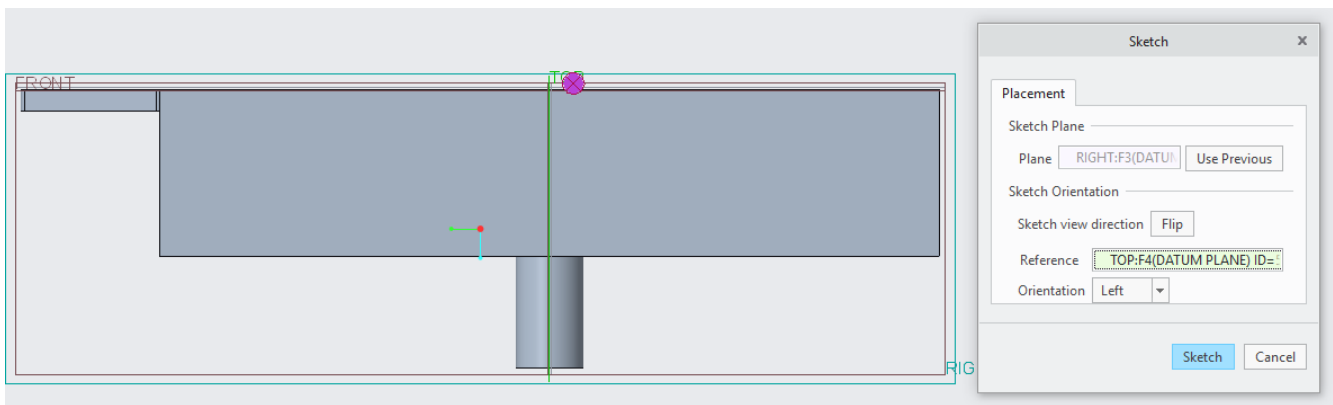


Figure 14: TOP as *Reference* and Left as *Orientation*.

Sketch three lines as shown in Figure 15. Ignore the values.

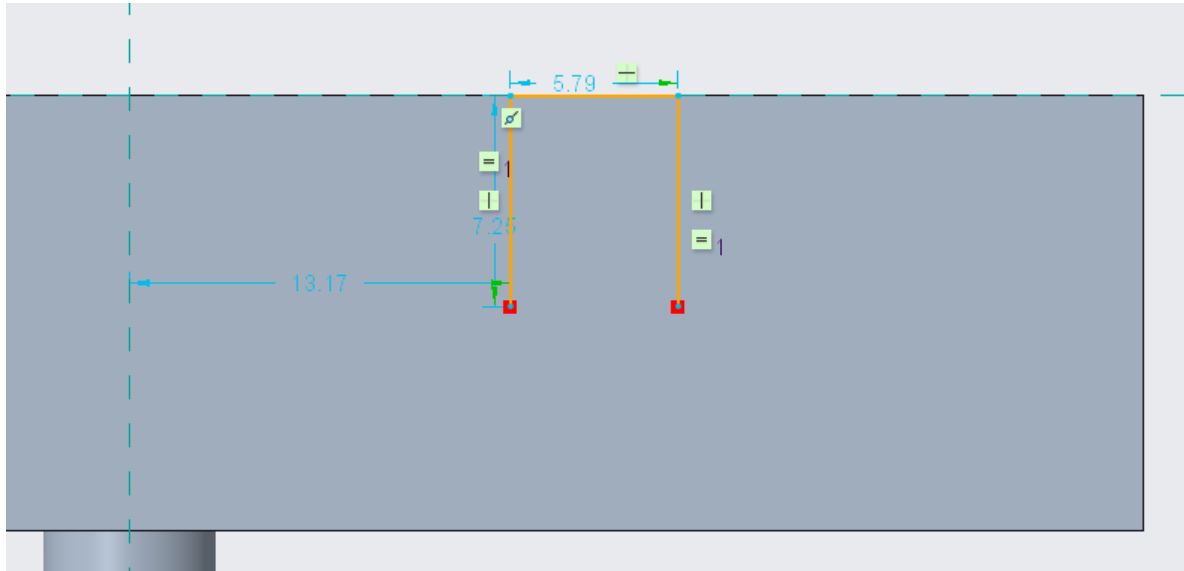


Figure 15: Three lines sketched. Notice the Equal (=) constraint.

Create an Arc (, *Sketching* group) between the loose end points (marked with red squares); click one end, then the other, then move the pointer so that the arc snaps tangential to both lines (Figure 16).

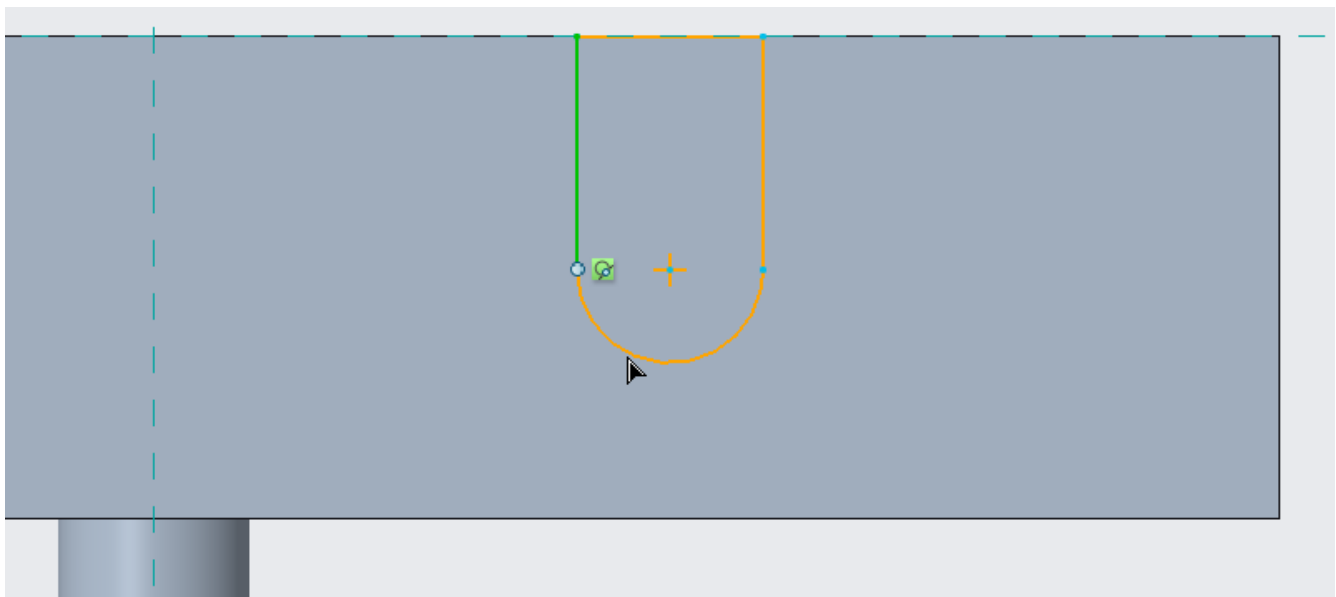


Figure 16: Arc snapping tangentially ().

Enter dimensions as in Figure 17 (24, 14, 7) and accept (✓) the sketch.

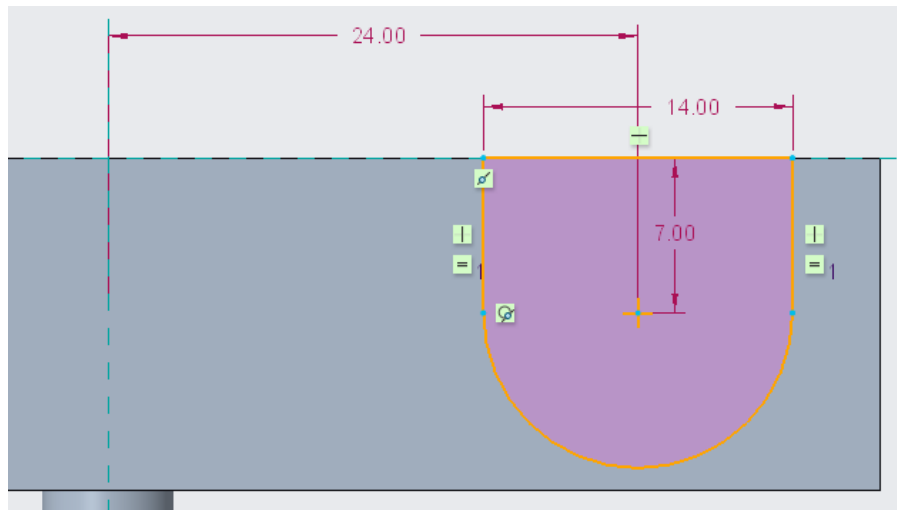


Figure 17: Ready-to-accept sketch. Notice the dimension of 24.

Make two changes to the default settings in the dashboard; click **Remove Material** (🗑️, if needed) on and select **Symmetric** (📐) as extrusion depth option. Set extrusion depth to **48** mm. Your model should look like in Figure 18. Rename feature as BATTERY_CHAMBER (Properties) and accept feature (✅).

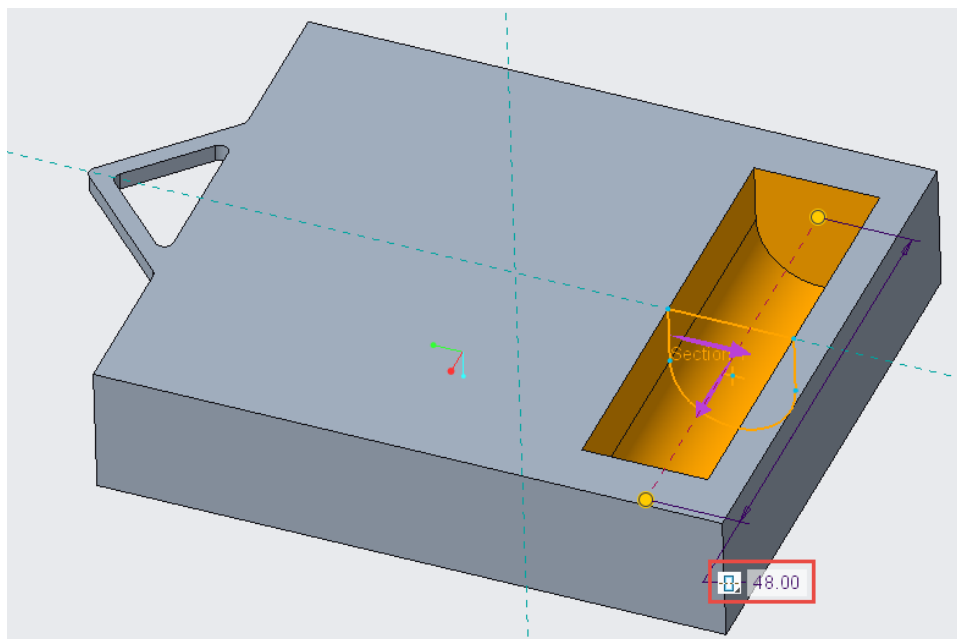



Figure 18: Symmetric cut with the depth of 48.

NOTE! The topmost line segment actually is redundant; Creo is able to create solid cuts with open sections.

The Knob Cut

Plane

An extra datum plane is needed for the next feature (revolved hole for an adjust knob). Click **Plane** (, *Datum* group) and select one of the side faces as the reference (Figure 19). With only have one reference selected, offset is the default and only possible creation type. (In the future, to select multiple references to a single collector, use CTRL as usual!) Set **8** as offset value and check that the preview looks like in Figure 19. To create plane to the other side, give -8 as a value or use drag handle to switch sides. Click **OK** when ready.

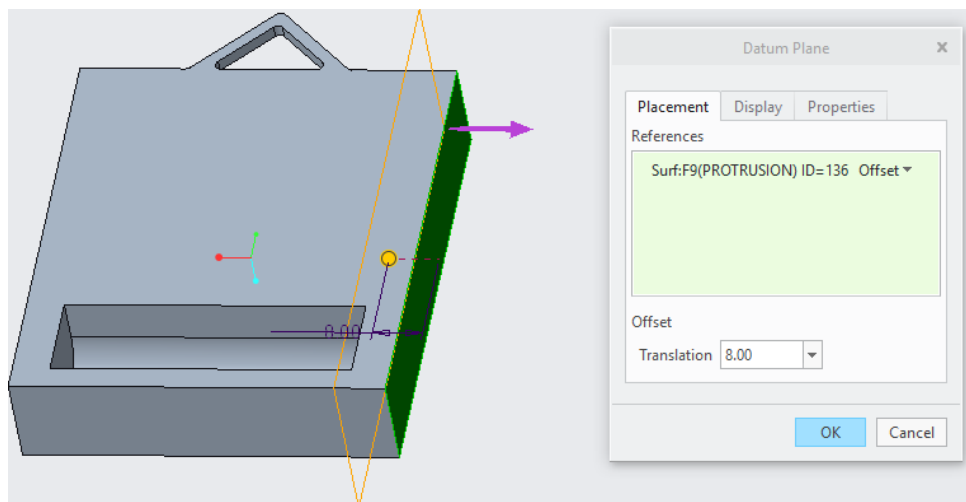





Figure 19: Location of the new plane.

Revolve

With the new datum plane still selected (highlighted in green), start a **Revolve** (, *Shapes* group). The datum plane is automatically selected as sketch plane. (This was just a demonstration of object-action ideology, which is very efficient in daily use; the sketch plane could as well have been selected separately.) Select the sketch direction reference and direction option so that the back face is on the top (using **Sketch Setup** , *Setup* group), if needed.

Sketch and dimension a **Centerline** ( from *Datum* group!, notice the black dashed line) and the six line segments as in Figure 20. If you forgot how to create diameter dimensions, check the Getting Started (Exercise 1.0).

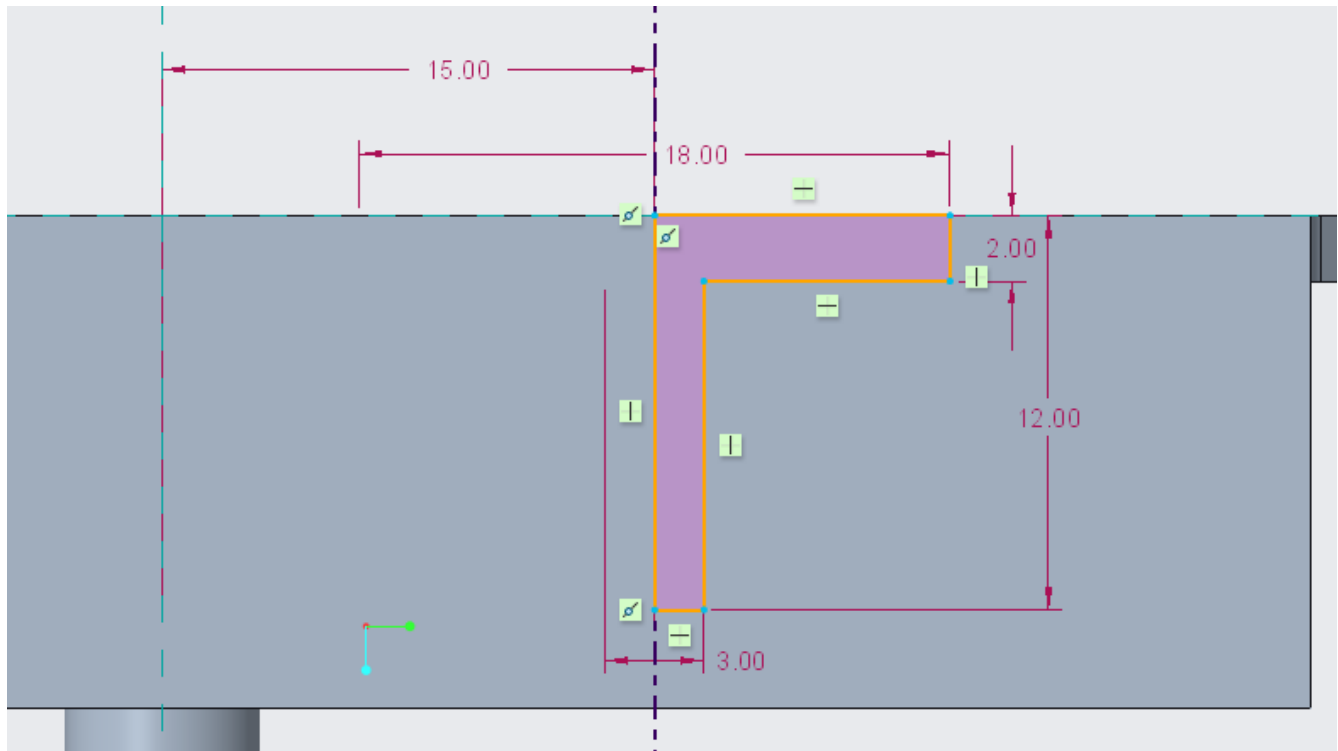



Figure 20: Ready-to-accept sketch.

Accept the sketch and the feature (put **Remove Material**  on). Rename the feature as KNOB_CUT.

Corner Rounds

Select **Round** () and hover your mouse on the top of one big corner edges (Figure 21, leftmost). Then click **LMB** a couple of times to shuttle between different choices (don't hold LMB and don't move your mouse!) When it shows *IntetEdg...*, click **RMB** to select all four edges (Figure 21, rightmost).

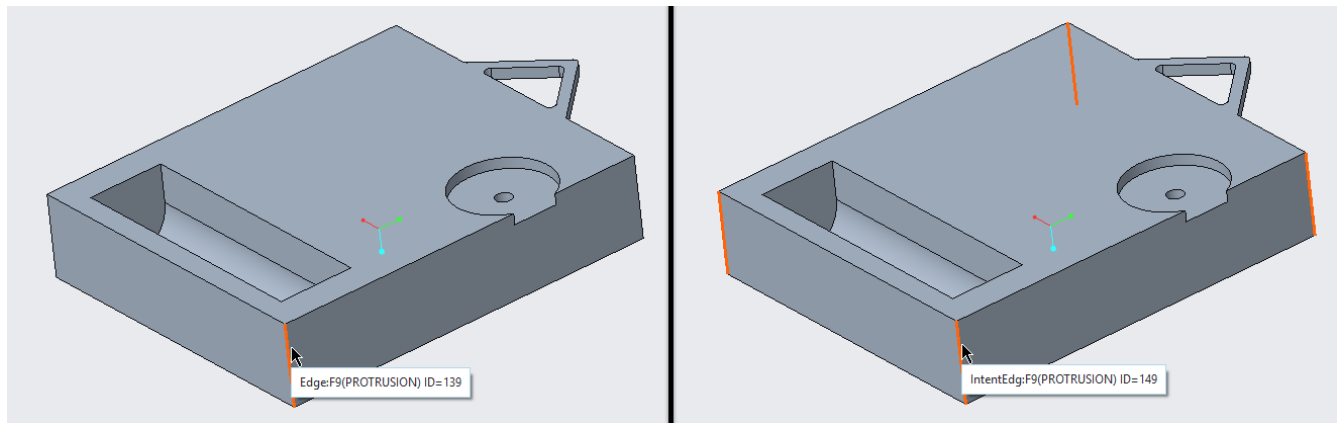


Figure 21: On left: Cursor over one edge. On right: clicked LMB couple of times and all edges selected.

Give a rounding value of **8** () and accept the feature.

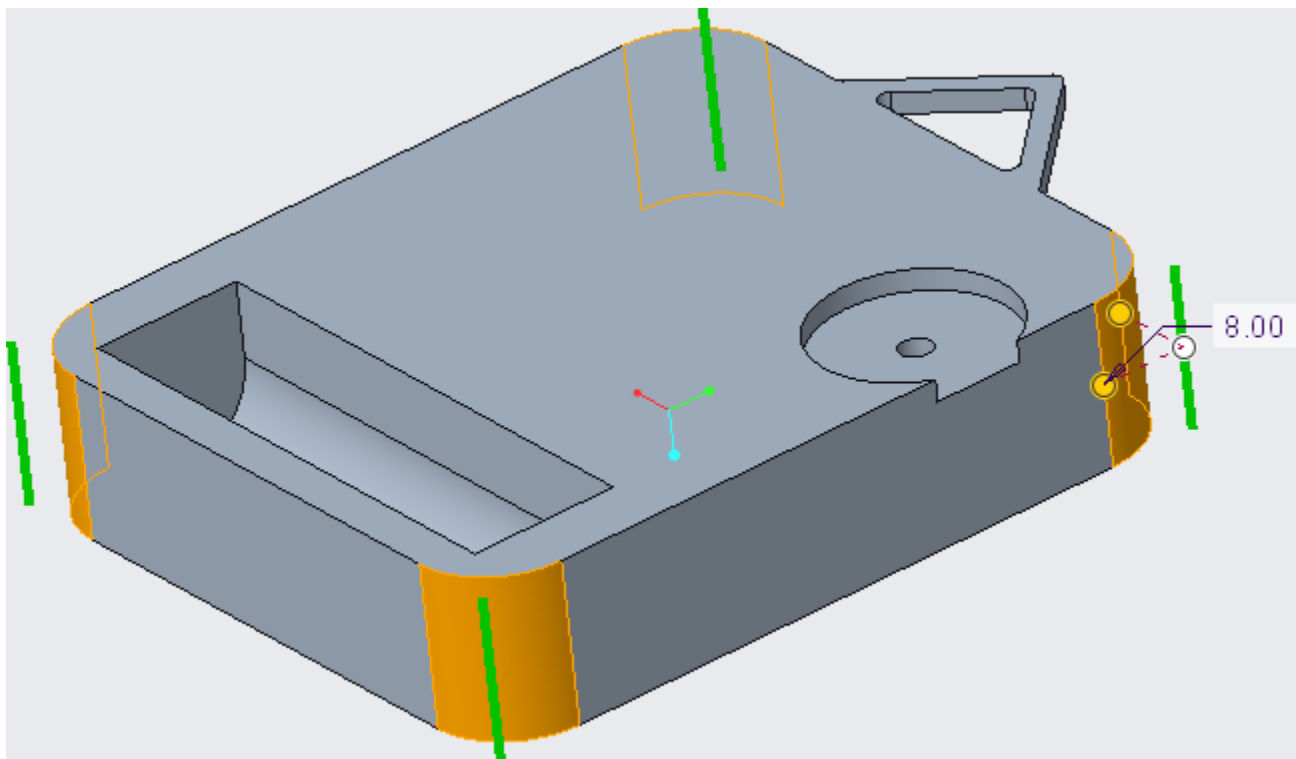


Figure 22: Ready-to-accept round feature.

The Battery Plus Pole Cut

Try on your own and create **a cut** so that plus pole of the battery fits in (Figure 23). There should be only two independent dimensions, **diameter of the plus pole** (3) and **depth of the extrusion** (2). Everything else should be dependent on the existing geometry, using references and suitable sketcher constraints. When ready, your model should look like in Figure 1 and Figure 24. Remember to save your model!

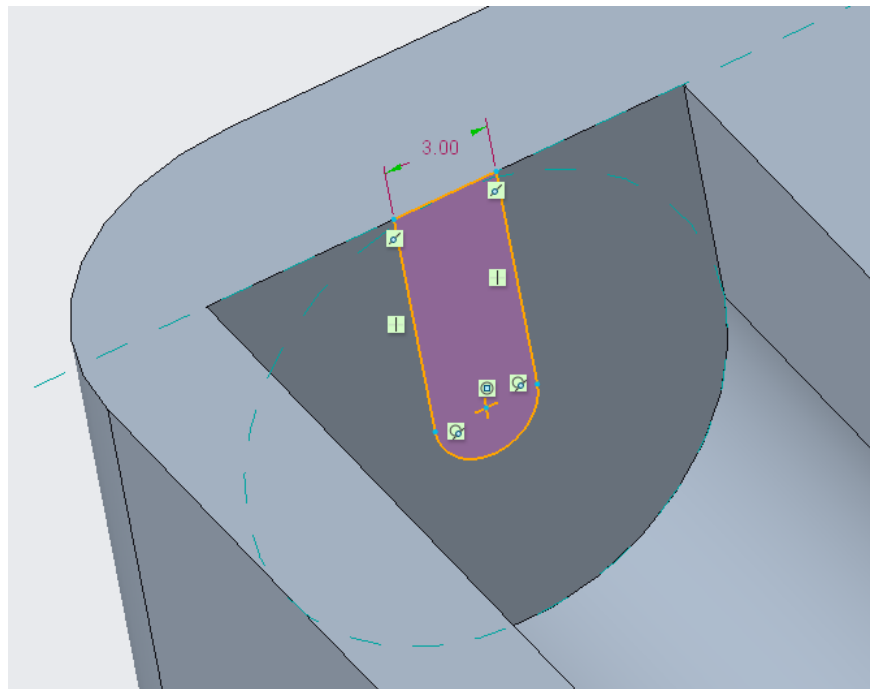


Figure 23: Sketch for the plus pole.

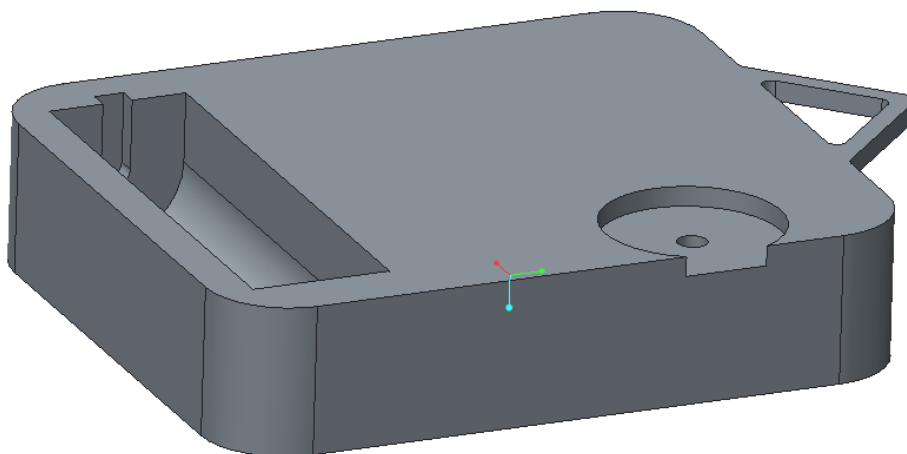


Figure 24: Ready model.

Parameters and Relations

Add the following parameters (Table 1) and relations (Figure 25) into your model:

Table 1: Parameters to be created.

Parameter Name	Type	Value
HANGER_WIDTH	Real Number	20
HANGER_HEIGHT	Real Number	10
BATTERY_LOCATION	Real Number	24
BATTERY_WIDTH	Real Number	48
KNOB_LOCATION	Real Number	15
KNOB_DIA	Real Number	18
KNOB_DATUM	Real Number	8

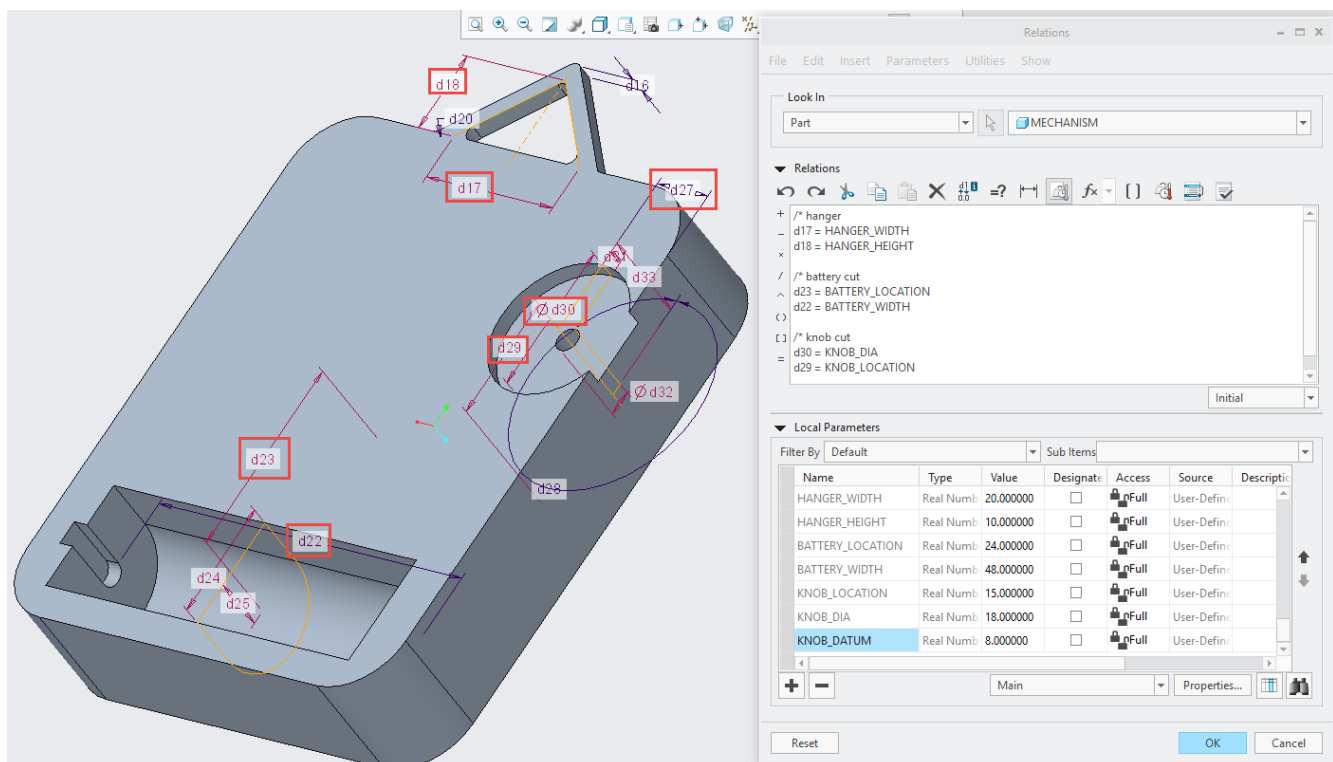


Figure 25: Relations to add.