EXERCISE 1.1 – CREATING A SIMPLE BATTERY
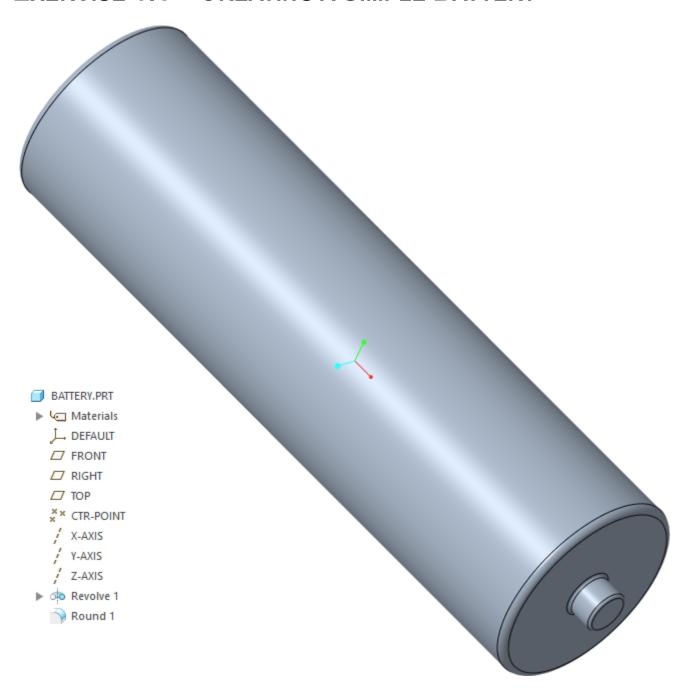

Figure 1: Battery modelled in this exercise and its model tree.

Learning Targets

In this exercise you will learn:

- ✓ to use Revolve to add material
- ✓ to use Round with multiple sets.
- ✓ to define parameters and relations

In this exercise, you will learn how to create revolved shapes.

Program version in this exercise is Creo Parametric 6.0.2.0 (same as in university's computer classes).

Used acronyms:

- − LMB; Left Mouse Button
- − MMB; Middle Mouse Button (press the wheel)
- − RMB; Right Mouse Button

Starting

Start Creo Parametric using shortcut defined in previous exercise (1.0). Select **New** from the *Home* tab (Figure 2). Ensure that *Type* is *Part*, *Sub-type* is *Solid* and give a name (**battery**) for the new part (Figure 3).
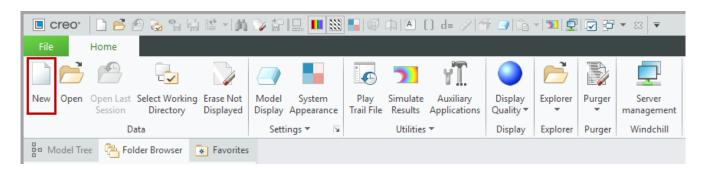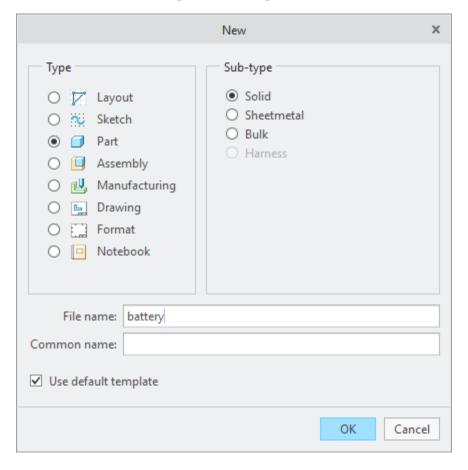

Figure 2: Selecting New.

Figure 3: New window, name to a model given (battery).

Using Revolve

The battery (Figure 1) is almost completely modeled using a single feature, revolve. Select **Revolve** (⊙) from *Shapes* group (Figure 4).
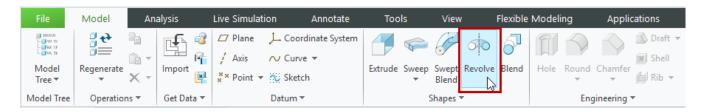

Figure 4: Selecting Revolve feature from Shapes group.

Revolve feature can use both external (created previously) or internal (created when Revolve tools is in use) sketches. In this case we use internal, so from *Revolve* dashboard, select *Placement* and under *Sketch* **Define** (Figure 4).
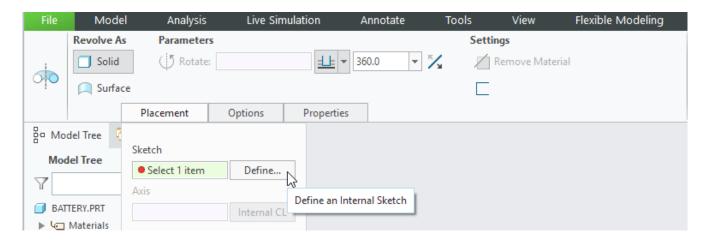

Figure 5: Defining new sketch for revolve feature.

A *Sketch* window appears. Select **FRONT** plane from the graphical area (or from model tree) as a sketching plane (field highlighted in green is showing what is excepted from the user next) as seen in Figure 6. In the Sketch window you can define how the sketch is viewed: *Plane* defines sketching plane (i.e. where you draw needed geometry), *Reference* defines reference plane datum and Orientation to what direction this plane's normal axis points (in our case, the normal of the RIGHT plane points Right). Positive side of the plane is draw in dark brown and negative with light brown. To enter sketching mode, select **Sketch**.
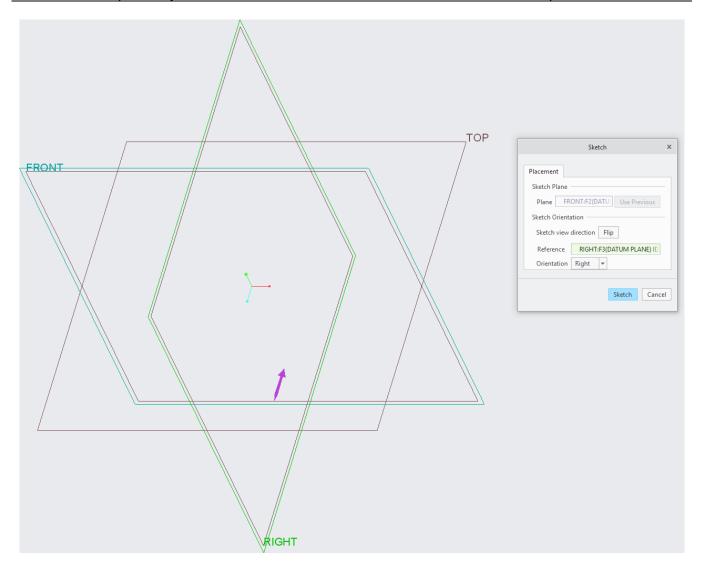
Figure 6: Location for the sketch,

Sketch

In the case of revolved features, the first thing to sketch is a <u>centerline</u>. Centerlines are infinitely long, defined by two points, and they do not produce any 3D geometry. They are used to mark axes of revolution, symmetries and so on. You may sketch as many as is needed, but Creo takes the first one as the axis of revolution by default, if one is required. It can be changed afterwards but learning to sketch the centerline first is probably easiest. To sketch a geometric centerline on the horizontal reference line, select **Centerline** (⁞) from *Datum* group (Figure 7, there are two centerlines, the other from *Sketching* group is visible only in sketch and thus revolve feature will not see it).
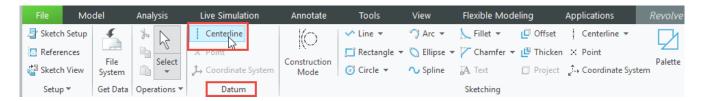
Figure 7: Selecting *Centerline* from <u>Datum</u> group.

Select two points on the horizontal reference line (TOP plane), letting the pointer to snap on it. Check that a collinear constraint symbol (⬚) appears.

Then activate the **Line** tool (⌄ from *Sketching* group) and sketch what you see in Figure 8. Start by clicking on the intersection of the reference lines (1) and then click once on each point (2…6), finally closing the section by clicking again on the first point (7). **MMB** after the last point to end the line and **MMB** again to close the tool.
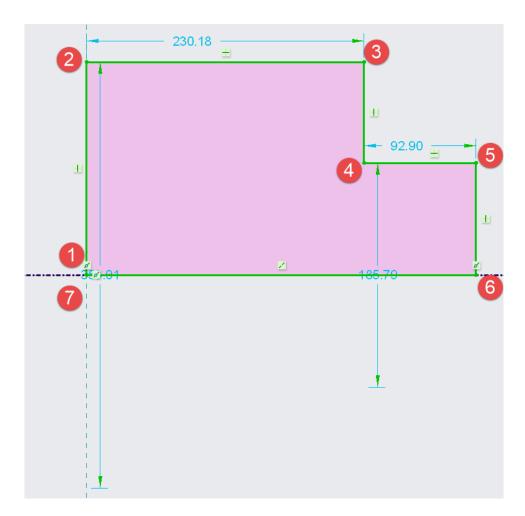

Figure 8: Sketched shape started from 1 and ended in 7. Ignore the dimension values and locations.

Make sure that the proper constraints are created by the sketcher. Having four dimension constraints is an indication of it. You should have — (horizontal), | (vertical) and ⬭ (point on entity) constraints as well. Probably the dimensions are not identically located, don't care about that.

Dimensioning

Regardless on how the existing dimensions are located, you will replace some of them with new dimensions. The blue dimensions are <u>weak</u>, they can be replaced without deleting. Select **Dimension** (⟻from *Dimensioning* group) and select the topmost horizontal line. Place the new dimension by pressing **MMB** on a suitable place above the line (ignore values). Do the same thing with the bottom horizontal line, although you already had at least one of the dimensions. Notice that the new dimensions are black, i.e. strong dimensions.

Then, (although you might know it's not right,) create another dimension to constrain the length of the shortest horizontal line. The constraint solver does not find a solution, because the sketch is over constrained. The *Resolve Sketch* window appears (Figure 9), showing the constraints that together cause the problem. CAD programs usually do not accept over constraining even if the constraints are consistent.
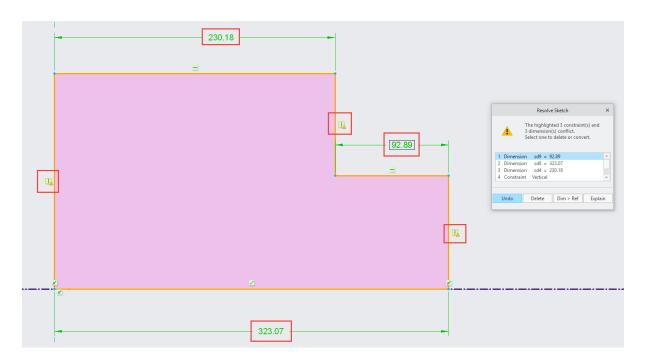

Figure 9: Resolve Sketch and three created dimensions (green).

Highlight the constraints one by one on the list and observe how the corresponding symbols are highlighted in the graphics window. Leave the topmost dimension selected (the third one in Figure 9) and click **Delete**.

The two vertical weak dimensions are OK, so you can close the *Normal* tool (**MMB**). By double-clicking, edit the dimension values as seen in Figure 10. Tip: start with the smallest values! You can drag dimensions to make your sketch clear.

Figure 10: Final dimensions. Notice the dark red locked dimensions.

When ready, accept the sketch ( ). Accept default options: variable and 360° as revolving angle ( ).

Using Round with Sets

Let's finish the battery with a couple round sets. Round in Creo can include more than one reference sets, each having their own radius values. Select **Round** (from *Engineering* group) and select the two edges shown in Figure 11, while holding **Ctrl** when selecting the second edge.
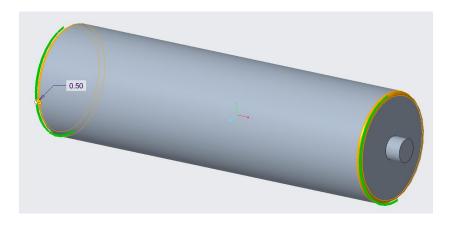
Figure 11: Two edges selected.

Circular edges consist of 180° pieces, but selecting only "half a circle" is enough. Then
without Ctrl, select one of the smaller circles (in the + pole of the battery), and the other one
holding **Ctrl**. Open the **Sets** slide-down panel. You should now have two reference sets, both
containing two edges. Select **Set 1** from the list. Double click the corresponding radius value
in the graphics area or enter **0.5** straight to the *Radius* field in the *Sets* panel (Figure 12).
Similarly change the value of **Set 2** to **0.2**. Accept the *Round* feature (✔).
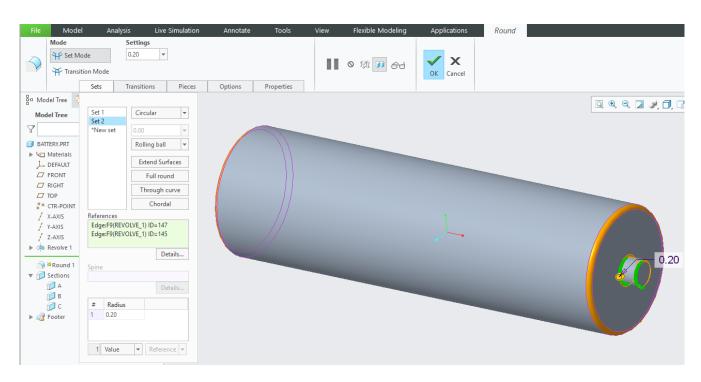

Figure 12: Changing *Set 2 Radius* value.

Check that your battery looks like Figure 1. Did you remember to **save** your model?

Adding Parameters and Relations

The geometry of the part is ready. Next, a set of parameters and relations are added to the model. With parameters, users can edit the geometry without knowing exactly what feature to edit.

Select *Relations* (d=) from *Model Intent* group (Figure 13).

Figure 13: Selecting Relations.

This opens a *Relations* window (Figure 14) where all *Relations* (how dimensions/parameters are affecting each other) and *Local Parameters* (user defined variables, click ▶ to see them) are listed.
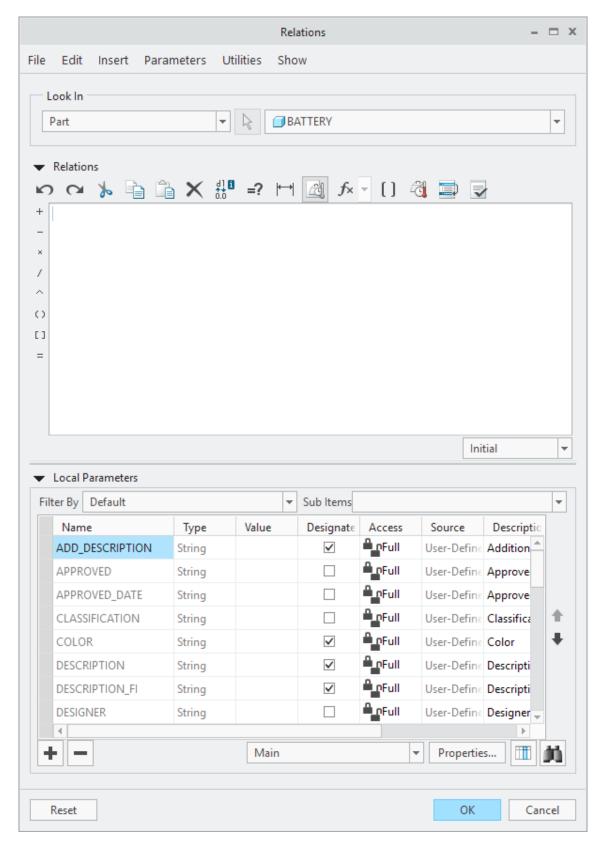
Figure 14: Relations window. Notice the shown Local Parameters list.

To define a new parameter, click ⊞ from *Local Parameters* field. Name parameter as DIAMETER (Creo will change the name to be UPPERCASE), ensure that *Type* is **Real Number** (decimal number, float type) and give value of **14** to it as seen in Figure 15.
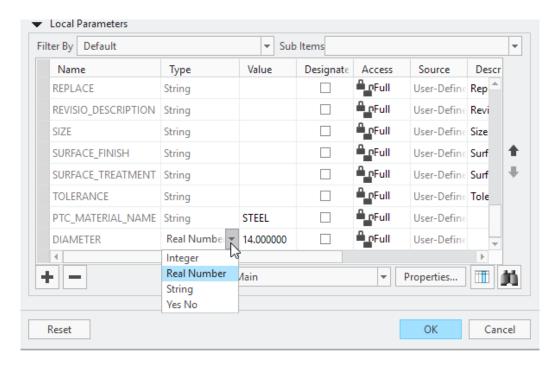

Figure 15: New parameter called DIAMETER is created, type defined and value given.

Next, in addition to DIAMETER parameter, create also the following parameters seen in Table 1.

Table 1: Parameters to create.

Parameter Name	Type	Value
DIAMETER	Real Number	14
LENGTH	Real Number	50
POLE_DIA	Real Number	3
POLE_LENGTH	Real Number	2
MAIN_ROUND	Real Number	0.5
POLE_ROUND	Real Number	0.2

When parameters are created, it is needed to tell the program, how these parameters are utilized. Click anywhere in the *Relations* text field and click on the first user created feature in the model tree (by default names as **Revolve 1**). Now the program shows all dimensions related to the selected feature (e.g. dimensions from the sketch, feature values such as rotation) as seen in Figure 16.
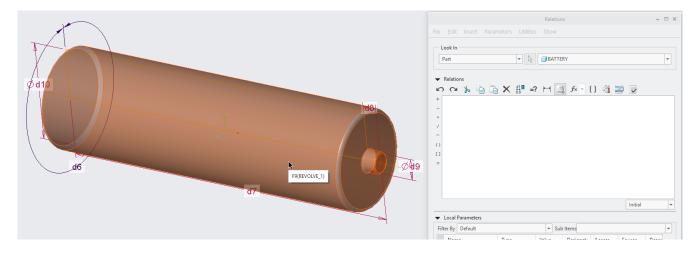

Figure 16: First feature selected and showing all dimensions related. NOTICE: you may have different dimension names (in format d#, where # can be any number).

Click on the dimension defining the main diameter of battery ($\varnothing$#d), in this case **d10**. **Notice:** Creo names dimensions based on the creation order, so you may have different names (no problem about it). Ensure that you select the equivalent dimension from your model. As you can notice, the selected dimension name appears on the relations field. Next, add **=DIAMETER** after the dimension name. Similarly, do the same with all other dimensions (Figure 17). TIP: **/*** marks a comment line and by RMB parameter name *Insert to Relations* can be selected.

Be free to test what happens if you change parameters. You can access only parameters from **Model Intent → Parameters** (**[ ]**).
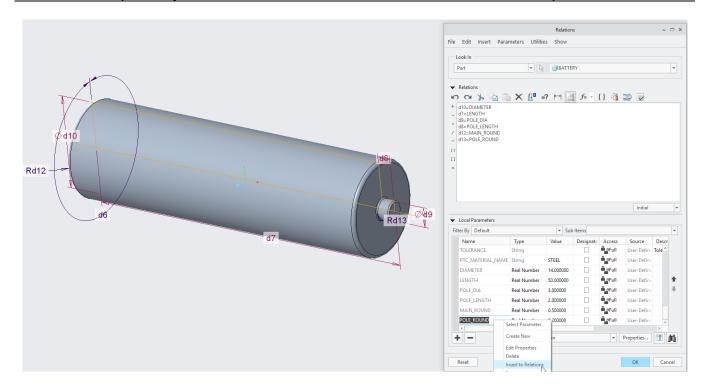
Figure 17: Dimensions defined by parameters.

Last note: If you have more or less dimensions than shown in Figure 17, you may have some errors in your model. To ensure that automatic assessment works, you need to have the parameters defined in Table 1.

This concludes the exercise. Remember to save your model!