

EXERCISE 1.2 – CREATING A FAMILY OF CLOCK HANDS

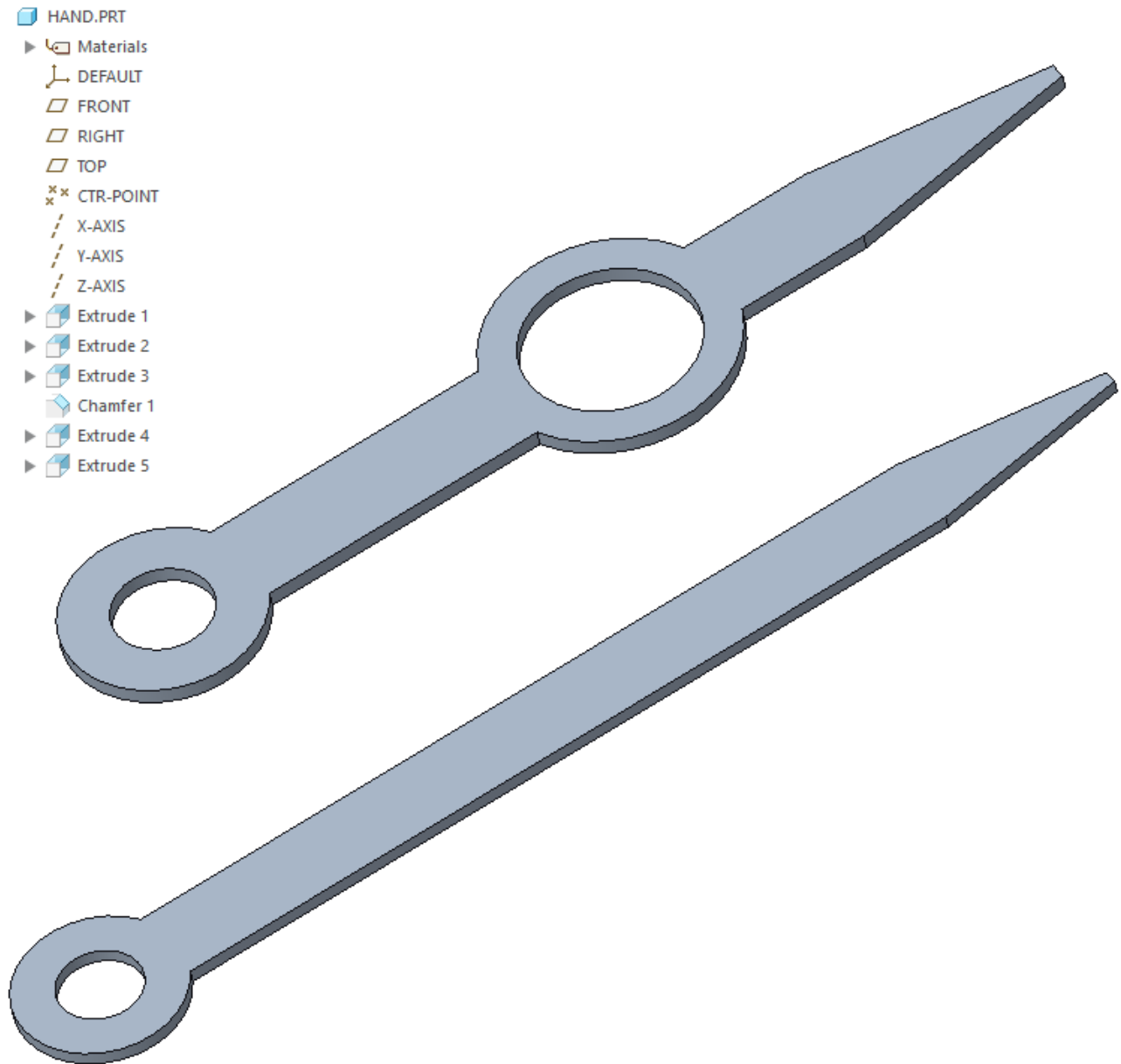


Figure 1: Clock hand and its model tree.

Learning Targets

In this exercise you will learn:

- ✓ to use Extrude to add material
- ✓ to redefine existing feature
- ✓ to use Extrude to remove material
- ✓ to create Chamfers.
- ✓ To use references
- ✓ To use family tables
 - To define varying parameters
 - To define varying features
- ✓ About internal memory

In this exercise you will learn the basics about Creo Parametric interface, to create extruded protrusions, chamfers, and to redefine features. When the basic geometry is created, a family table is defined – This allows us to use one model to create several parts (in this case minute and hour clock hand). Good modeling practices are also considered. Program version is Creo Parametric 6.0.2.0.

Used acronyms:

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

Start

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 (**hand**) for the new part (Figure 3).

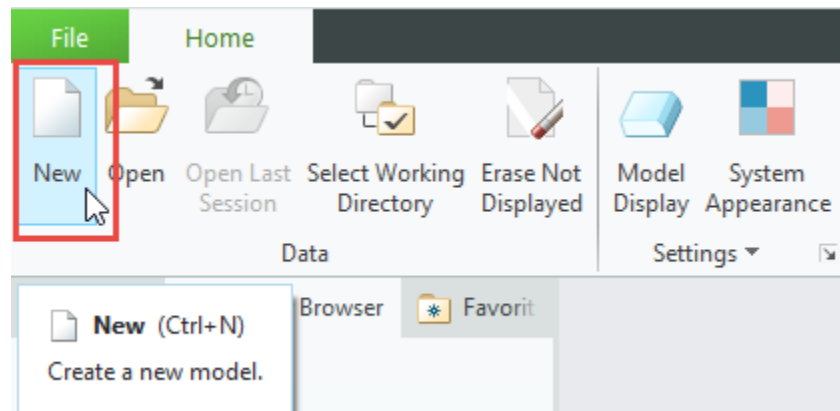


Figure 2: Selecting New.

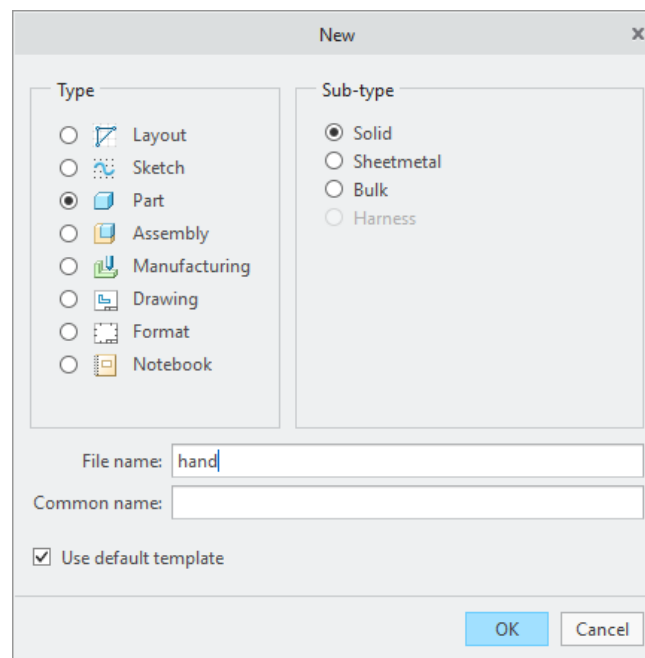



Figure 3: New window, name to a model given (hand).


Click **Plane Display** ( in quick access toolbox) a few times to turn the visibility of datum planes on and off. Leave the datum planes visible. Rotate the view (hold MMB and drag) to see that one side of each datum plane is brown and the other is grey. The brown color marks front or positive side of that plane. Click **CTRL+D** to see default view.

The Plan

The first step of modeling is to make some general level decisions about the model. For example, where “the origin” should be placed. Anyhow, there is an essential “center point” in the clock hand that is to be modeled, namely, the middle of the mounting hole. Let us decide that the center of the hole will be in the intersection point of the three datum planes.

It is also good to make it clear (to oneself), how the model is oriented with respect to the basic datum planes. Let us decide that (when the clock hangs on the wall,) the front datum plane is parallel to the wall, with its front side showing. Further, let the front side of top datum plane be facing upwards. Finally, let the hand lie horizontally, pointing right.

The First Extrude

Let's start modeling. The first feature will be a rectangular prism, defining the main dimensions of the hand. Thus, extrusion is an appropriate feature type. Click **Extrude** ( , from *Shapes* group, Figure 4) to enter the extrusion dashboard (Figure 5, above the graphics area).

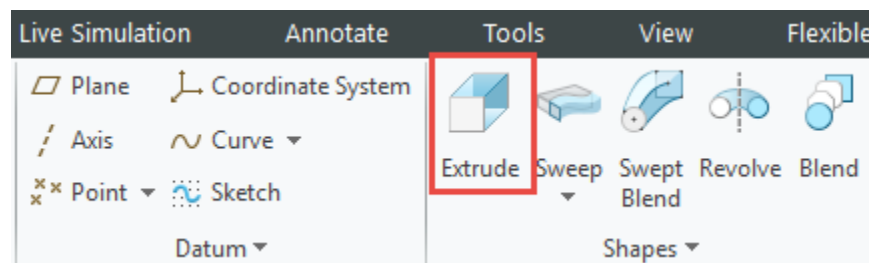


Figure 4: Selecting Extrude from Shapes group.

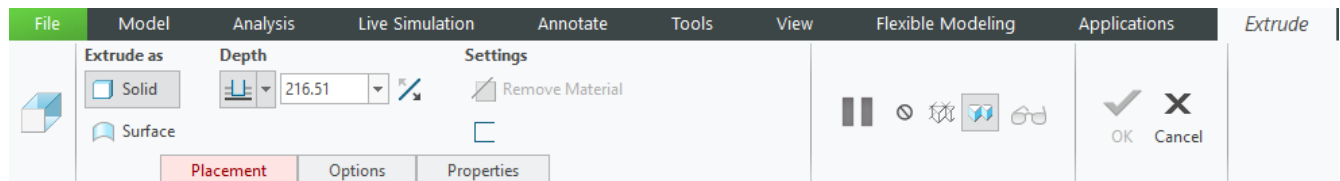


Figure 5: Extrude dashboard.

As you can see, the tab called *Placement* is in red. Select it and select **Define...** A *Sketch* window opens. Select **FRONT** plane (from the graphical area or from the model tree) to be your sketching plane. Notice, that **RIGHT** with *Right* as orientation is selected as *Sketch Orientation*. This means, that the positive side (brown) of the **RIGHT** plane will point to right when sketching. Click **Sketch** (Figure 6).

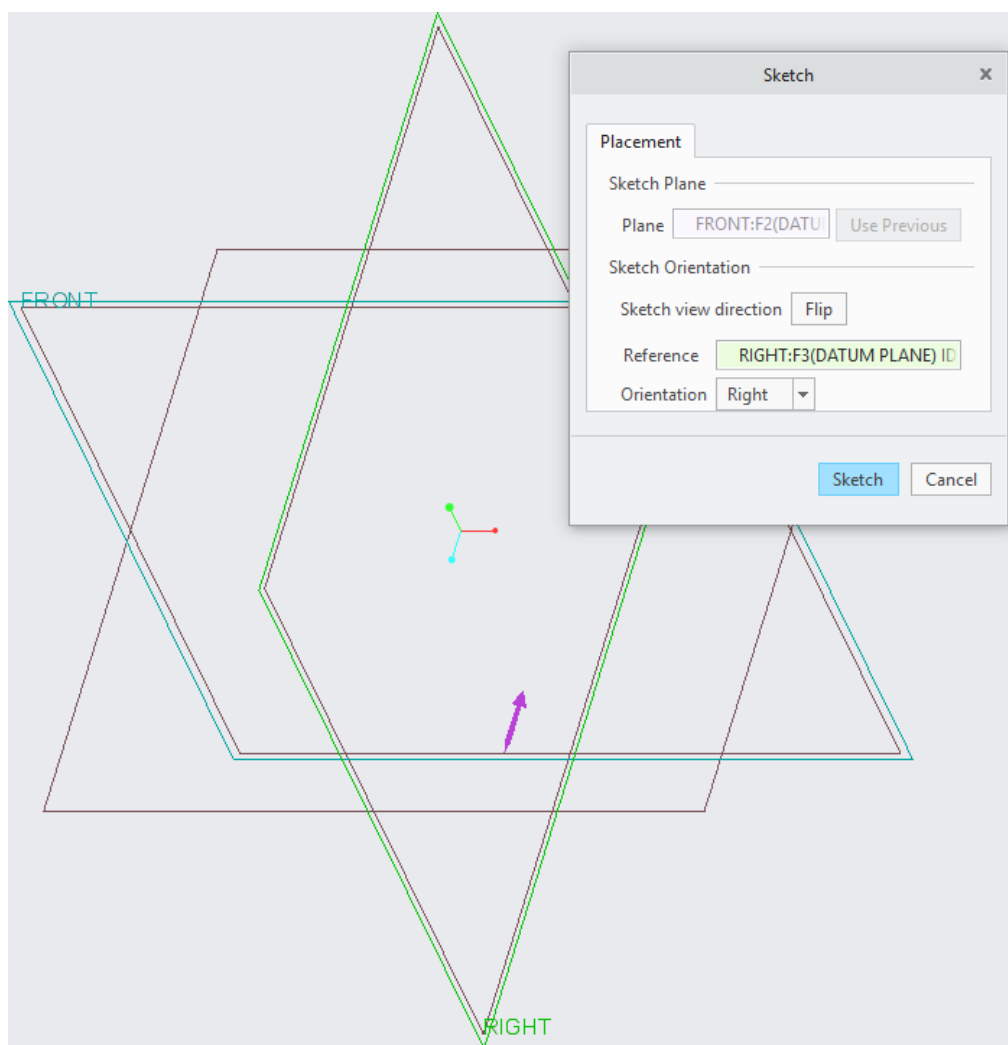


Figure 6: **FRONT** as *Sketch Plane*, **RIGHT** (Orientation *Right*) as *Reference plane*.

Sketching mode

You are now in the sketching mode. Click **Plane Display** (🔍) to hide the datum planes. Observe the turquoise, dash lines in the graphics area, and note that they lie where TOP and RIGHT datum planes intersect with the sketching plane. The dash lines are the sketcher reference geometry for this feature.

Select rectangle tool (📐, Figure 7).

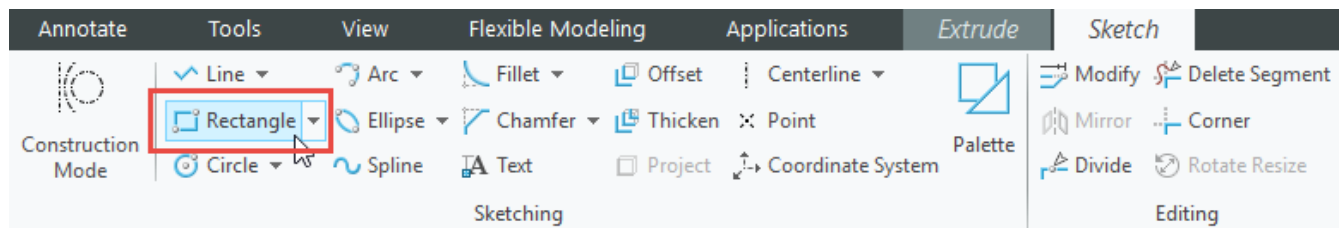


Figure 7: Rectangle tool in the *Sketching* group.

Move the mouse around to notice that it “snaps” on the reference lines. Let it snap on the vertical one, above the horizontal one (1 in Figure 8), and click to place the upper left corner. Click again to place the lower right corner (2 in figure) so that the result looks something like Figure 8. Press MMB to accept rectangle and to close the tool. Ignore the dimension values (blue ones).

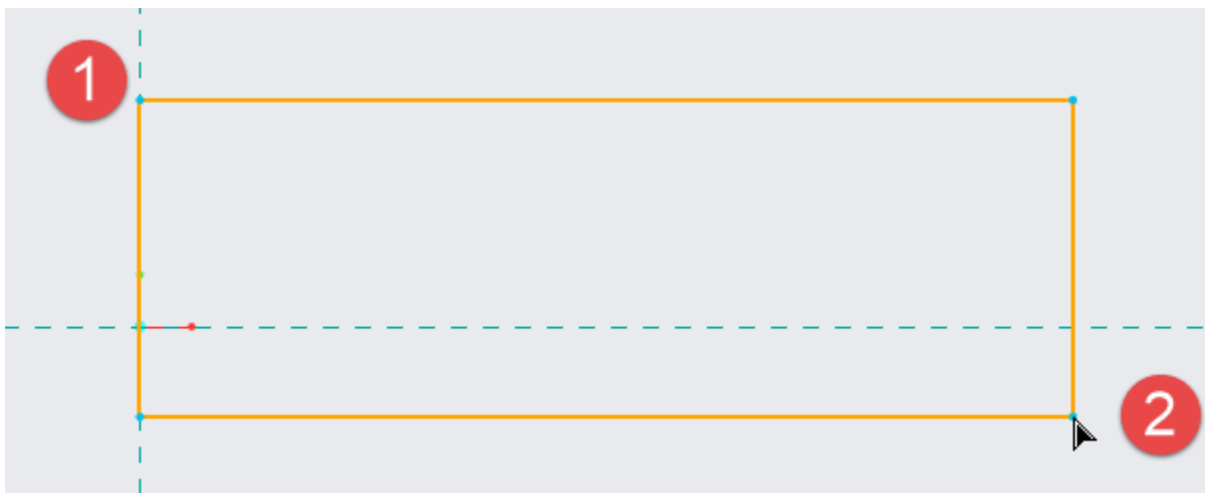







Figure 8: Rectangle, started from 1 and moment before selecting 2.

Notes

Notice the symbols , , , and the linear dimensions. All of them, including the dimensions, are constraint symbols (*rajoite* in Finnish). Constraints are one of the most important things about modern parametric CAD. They essentially are NOT sketching aid tools, although they do that job well, too. Their primary purpose is to maintain geometric relations between sketch entities, no matter what changes are made to the rest of the sketch or to the sketcher references. The constraints are realized simultaneously, actually by solving a set of equations (of second degree). Creo automatically applies a sufficient amount of constraints to make the equation set solvable. Some of them are created by the intent manager, taking user input (while sketching) into account. In this case, the intent manager guessed (correctly) that you wanted to constrain the first point of the rectangle to lie on the vertical reference line, thus the “point on entity” constraint (). The constraints are explained in Exercise 1.0.

Double click the dimensions (pointing on the value) and enter the values shown in Figure 9. Accept the sketch by selecting **OK** () from *Close* group.

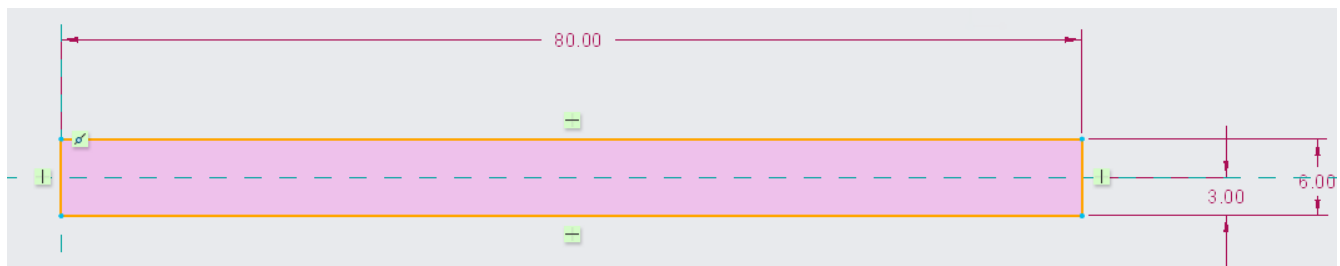



Figure 9: Changed values (green).

Finalizing the feature

Rotate view to see preview of the feature geometry better. Drag the white square-shaped handle to change extrusion depth. Click the pink arrow for a few times to change the extrusion direction. Leave the arrow pointing upwards, to the positive direction of FRONT datum plane (make datum planes visible ). Double click the depth dimension value and enter 1 (or enter the value in the dashboard, above graphics area). See Figure 10.

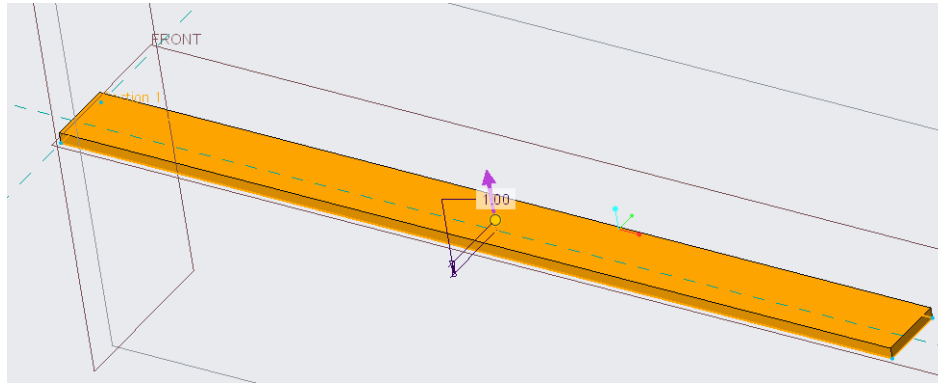



Figure 10: Feature preview, arrow pointing upwards from sketching plane (FRONT).

Accept the feature,  or **MMB**. The green highlight in the graphics area, and blue one in the model tree, indicate that the new feature is selected (Figure 11). Make a mental note of what a selected feature looks like! Click on the background to deselect.

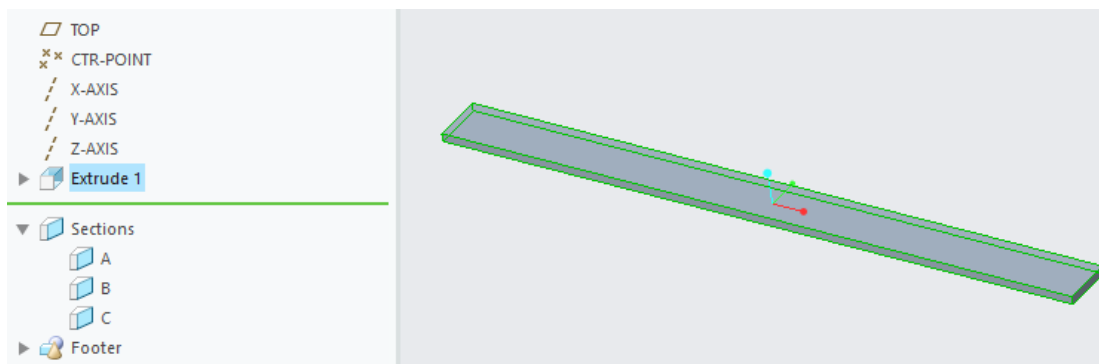



Figure 11: Feature selected in model tree (blue highlight) and in graphics area (green highlight).

Notes

After all this trouble, we still cannot be satisfied with the first feature. What's wrong with the sketch, Figure 9? One should always reduce the number of dimension constraints to minimum, leaving only the essential dimensions. In this case, the dimension with value "3" is redundant, as far as our design intent is considered. Mathematically, with regard to constraint solving, it is not; if we want to get rid of it, we have to replace it with another constraint. Our design intent (derived from our "first decisions") was to make the extrusion symmetric about the horizontal reference. "3" just happens to be the right value for the redundant dimension in this special case, but we would like the symmetry to maintain with any values of the essential dimensions (height and width).

Redefining feature

Redefine the feature to capture the design intent: **RMB** on the feature in the model tree, select **Edit Definition**  (Figure 12). From the *Extrude* dashboard, select **Placement** and **Edit....**

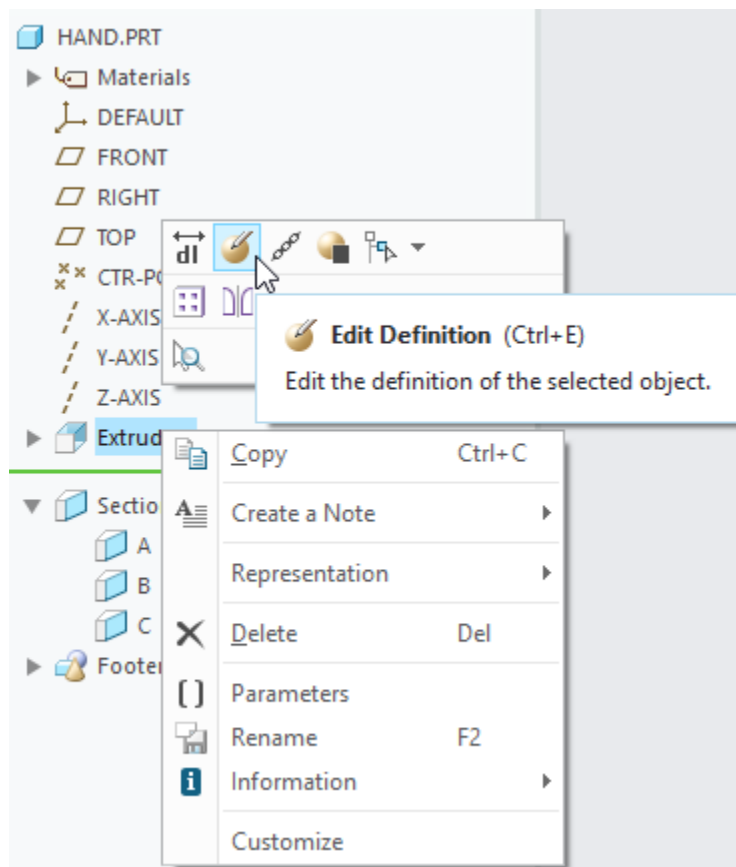






Figure 12: Selecting Edit Definition.

In the sketcher, delete the redundant dimension with value “3” (select it, hit **delete**). Note that a blue colored version of that dimensions appears. The blue color indicates weak constraint, one that is created by Creo and that can be replaced with a user-defined one, without deleting it first. We are going to replace it with geometrical constraints.

First, create a **Point** (✕, *Sketching* group) on the middle of the rightmost line (the line that has a length of 6 mm). You will notice, that Creo's intent manager automatically assigns a middle point -constraint for the point (it will displayed as  symbol). The second new constraint will tie this point to the horizontal reference axis in the middle. We have to do this one manually by selecting **Coincident** () from *Constrain* group.

Instructions will appear in the bottom of the screen (*Select two entities or vertices to align*). Basically, you need to first click on the point, and then on the line that you want it to be aligned with (the horizontal dotted reference line). After this, the point should have two symbols visualizing the geometrical constraints:  and . There should be no weak (blue) dimensions left, and the sketch should look like in Figure 13.

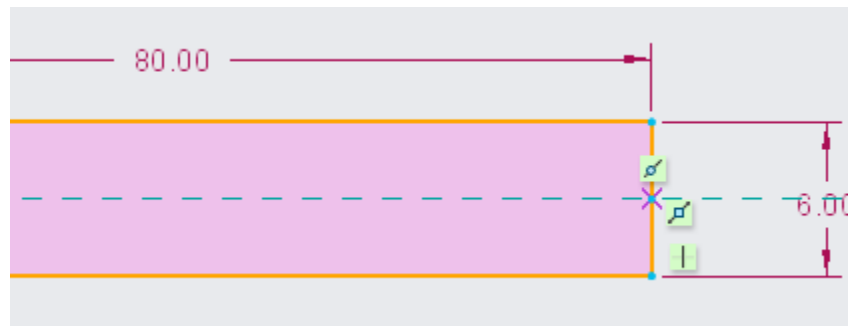





Figure 13: Redefined sketch; notice the midpoint and coincident symbols.

When ready, accept the modified sketch and feature:  and .

Save your model (CTRL+S and OK).

The Second Extrude

The next feature forms the circular portion of the clock hand. Besides the sketch geometry, this extrusion will have a definition little different from the first feature. Start creating an

Extrude (, *Shapes* group), hold **RMB** on the graphics area and select **Define Internal Sketch....** (Figure 14)

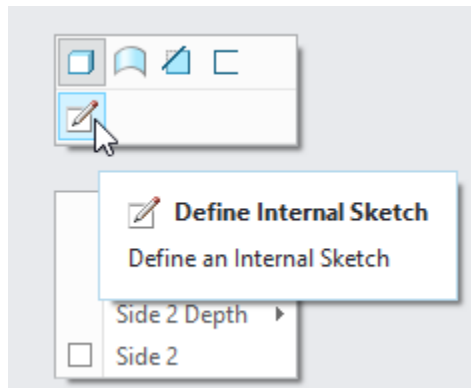


Figure 14: Selecting Define Internal Sketch from the RMB menu.

The sketch plane will again be FRONT; click **Use Previous** to select it (Figure 15).

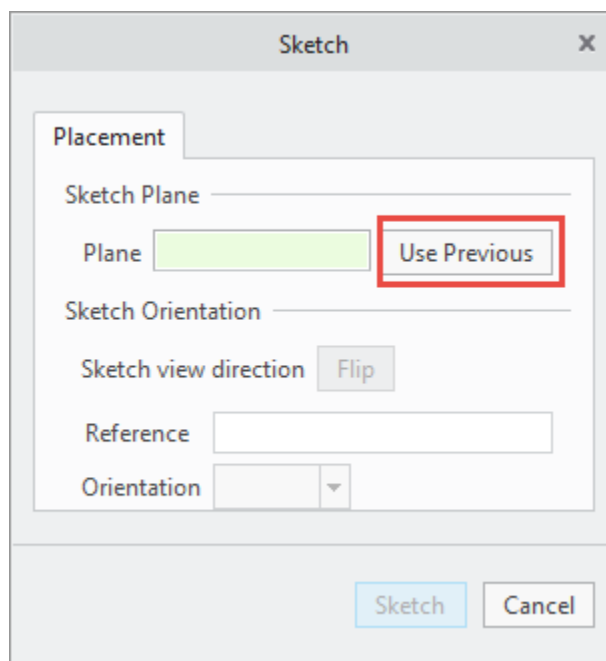


Figure 15: Selecting Use Previous.

Sketch

Sketch a **Circle** (🕒, *Sketching* group) as shown in Figure 16. Notice how wise positioning of the features with respect to the reference planes makes things easier! The default references were just where they should. Modify the diameter dimension value to **12**. Finish the sketch (✅).

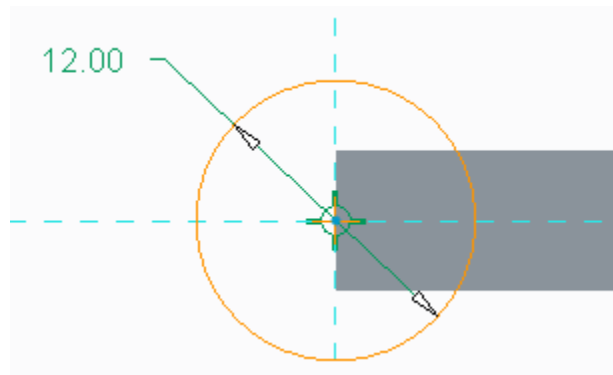




Figure 16: Sketched circle.

Finalizing the feature

Do we enter “1” as extrusion depth? As before, think which dimensions are essential. In this case, we already have defined the hand thickness, which is 1 mm. Depth of this extrusion should actually not be 1 mm but the **same** as the previous one, to be exact. Our **design intent** included that the hand has a uniform thickness.

Let's use the first extrusion as reference, making the new one take the depth value from it. With the first extrusion, the *Blind* option () was used. From the drop-down menu, change it to **To Selected** (, Figure 17).

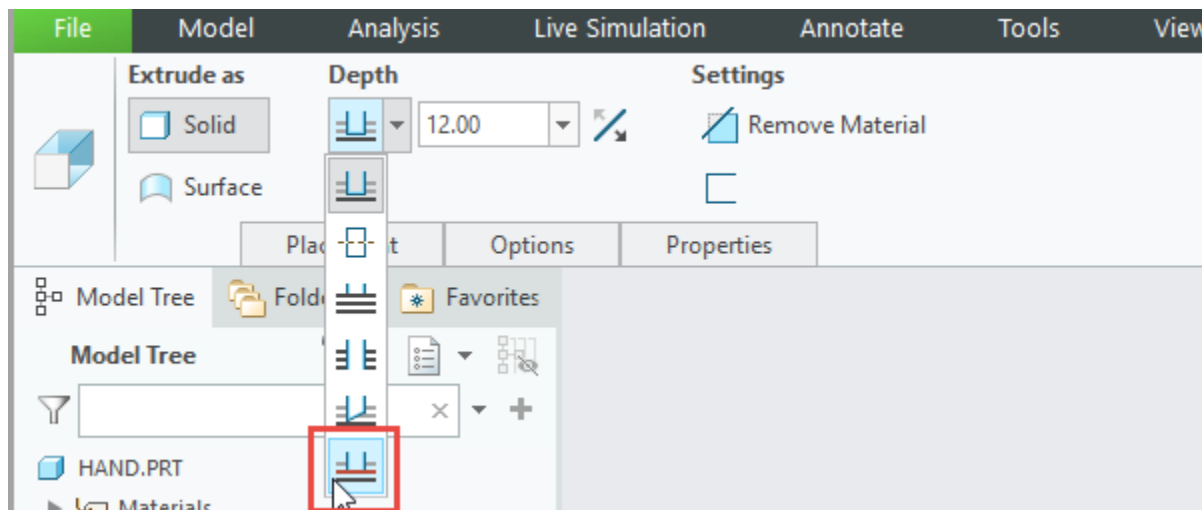


Figure 17: Selecting *To Selected* from drop-down menu.

Select the top surface of the first extrusion (Figure 18). When ready, accept the feature (✅). Remember to save your model.

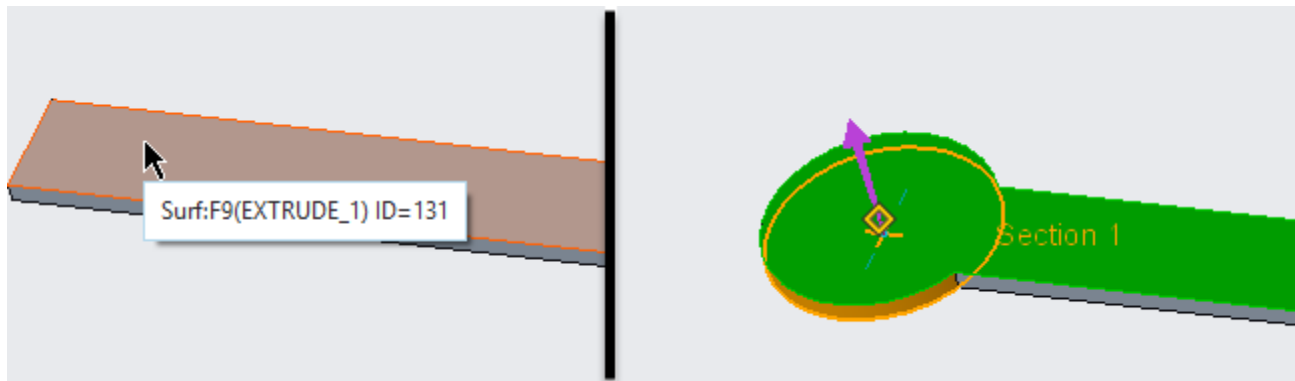


Figure 18: On left: selecting the top surface of the first extrude. On right: surface selected.

Making a Cut

The next feature removes material, creating the mounting hole. In Creo, extrudes add material (we have created two extrusions already), whereas cuts remove material. Let's create an extruded cut, then. The workflow is identical to the preceding feature until exiting the sketcher and with the exception of different circle diameter, **6 mm**. Do it.

After exiting the sketcher (or before sketching), we need to tell that we want to create a cut instead of default protrusion. *Remove Material* (🔪) should be selected in the dashboard by the program, if not, select it. How about the extrusion depth? *To Selected* could be used, of course, but *Through All* is better for cuts. Select it (≡ E). Flip the direction of the pink arrows in the graphics area to see where they affect. When preview looks like in Figure 19, accept the feature (✅).

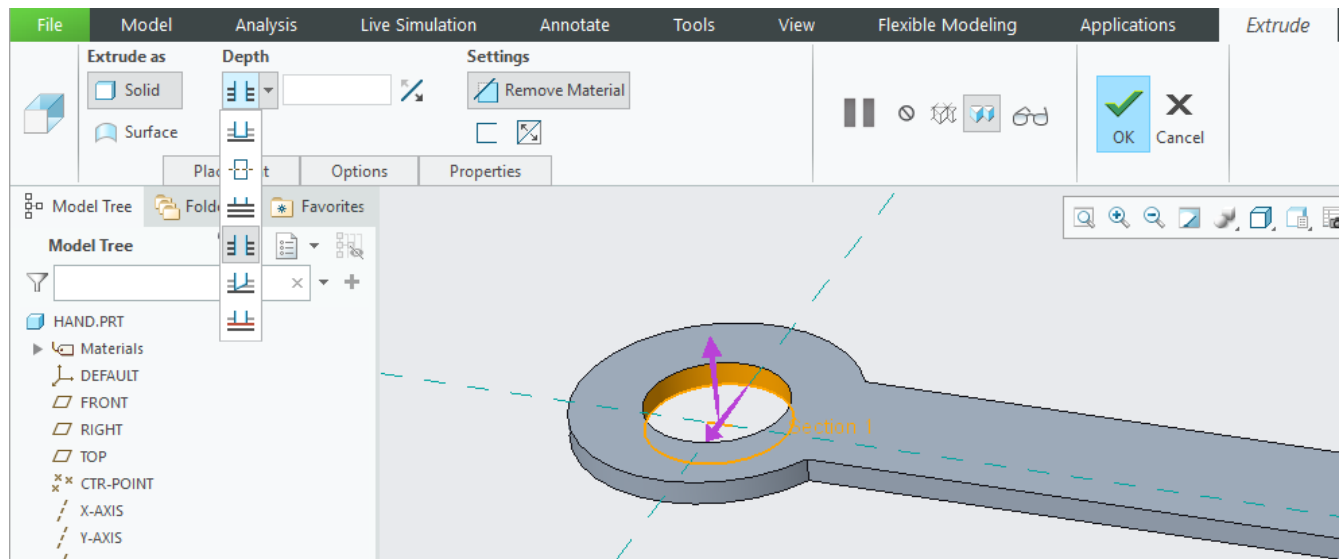


Figure 19: Ready-to-accept feature.

Chamfer

One feature left, namely the cuts that form the pointy end of the hand. An extruded cut could of course be used again, but a chamfer (*viiste* in Finnish) is chosen this time, for educational reasons.

Select **Chamfer** () from *Engineering* group. Select the edges shown in Figure 20.

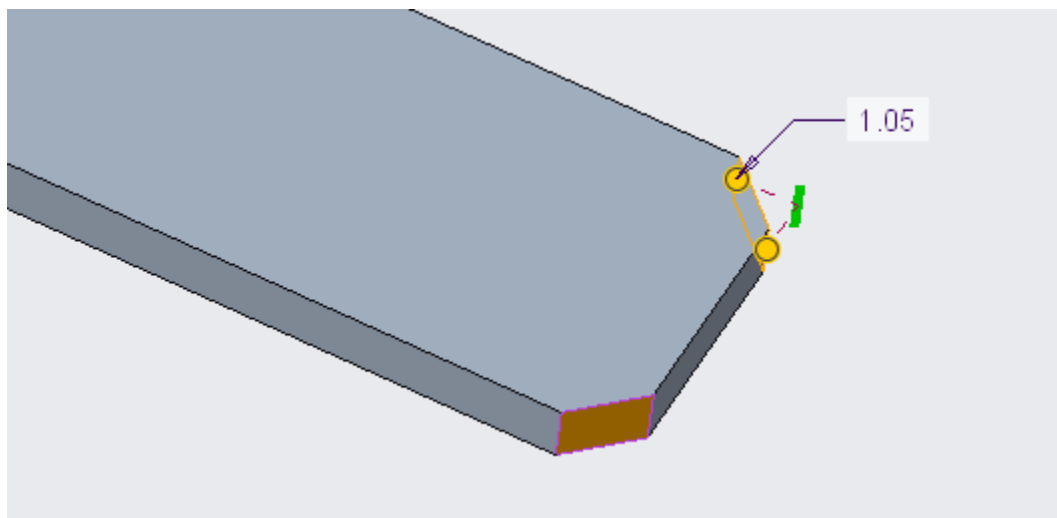


Figure 20: Edges to be selected. Ignore the offered value.

In the dashboard (under the graphics area), open the **Sets** slide-down panel. There are two reference sets, one for each edge you selected. Highlight them one by one to see that each set has its own dimensions. Our design intent included symmetry, so the chamfers of the edges should have common dimensions. Click **RMB** on “Set 2” and select **Delete**. Now hold CTRL while selecting the missing edge, and see how it is added to “Set 1”. CTRL is a common method of selecting multiple entities into one selection set in Creo. You may drag the handle in the graphics area to check that the chamfers do have common dimensions.

$D \times D$, the default chamfer type, is not what we need. Change it to **D1 x D2** (Figure 21).

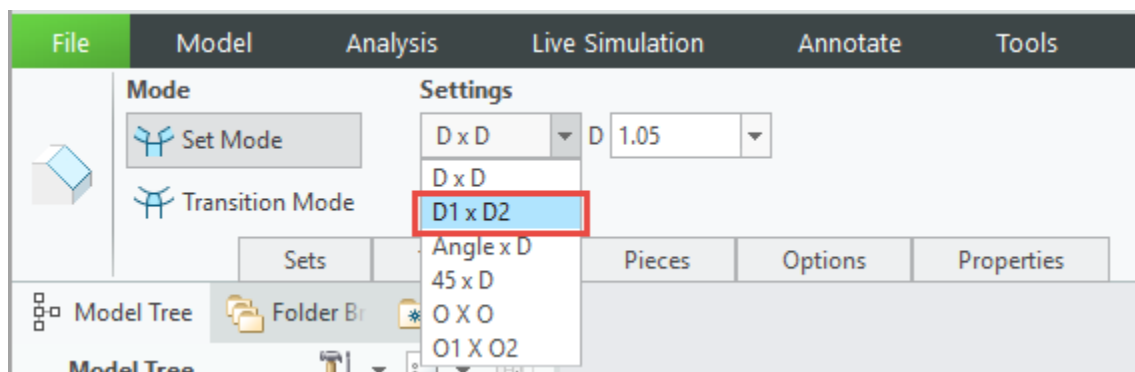


Figure 21: Changing type to D1 x D2.

Enter the dimensions shown in Figure 22 (**8** and **2.5**). Notice that as an American product, Creo only accepts period (.) as decimal separator! Accept the chamfer feature (✅).

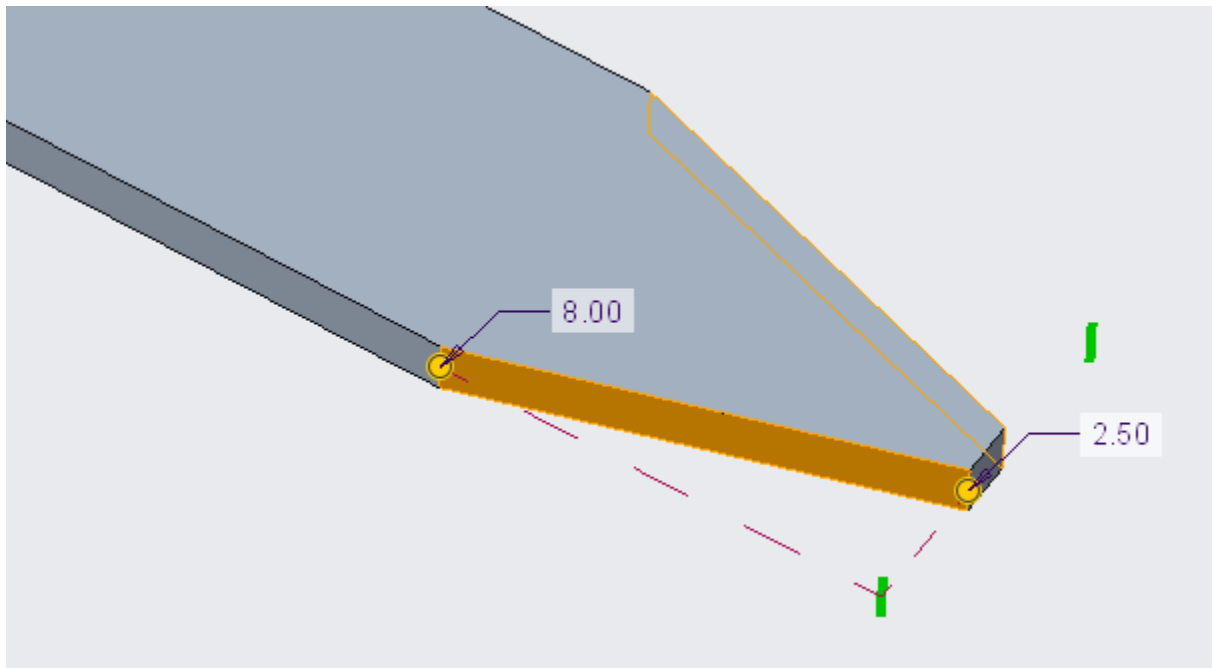


Figure 22: Ready-to-accept chamfer.

Using Edit

You learned earlier that editing feature definition allows changing sketch geometry and making other significant modifications. There is also a “less thorough” way of modifying features. **RMB** on the Chamfer feature in the model tree and select **Edit** (or double click the feature in the graphics area). The related dimensions are shown, and you can change them by double clicking and entering a new value. Do that, change 8 to **15** (Figure 23). The changes are updated in real time. To accept changes, click couple of times on the background.

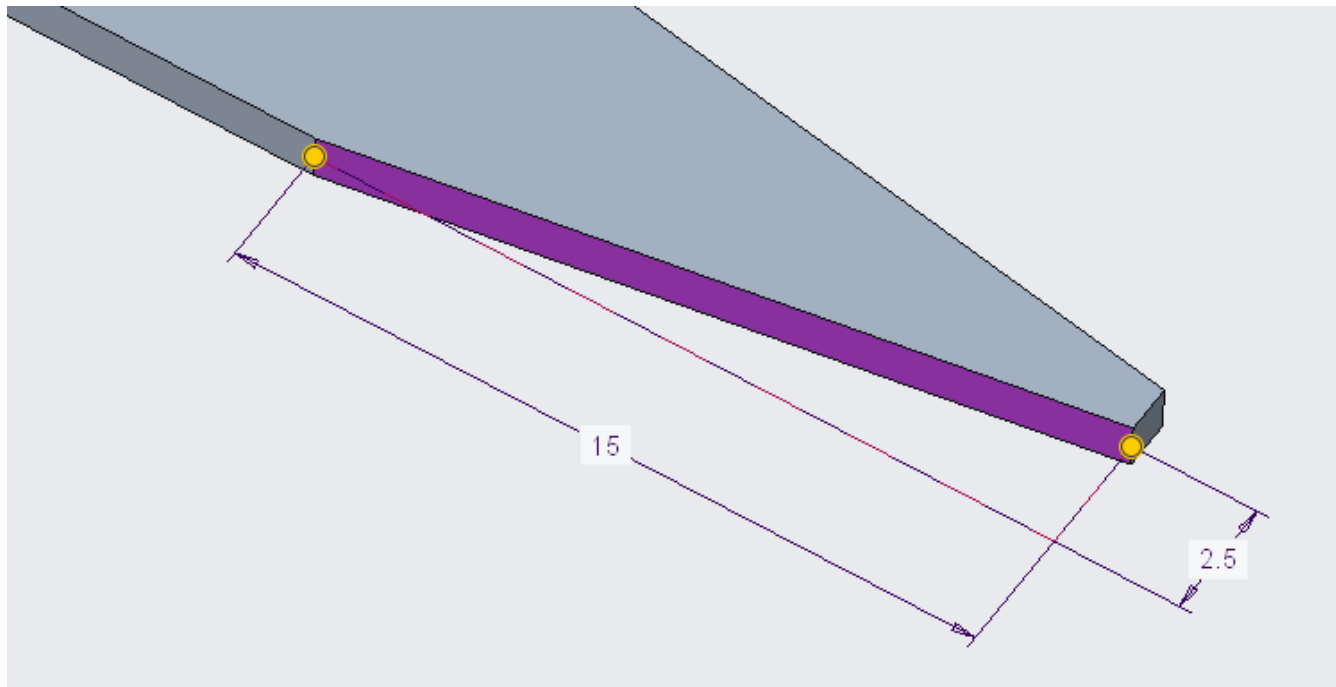


Figure 23: Edited chamfer feature. Notice the purple highlighting of affected geometry.

Notes

*One may claim that the whole hand, including the hole, could have been modeled with one extruded protrusion. It is true. The reason to use multiple features is not solely educational. Always keep your sketches simple! Although very clever and sophisticated, the constraint solvers may fail on very complex sketches. Or a correct solution to the equation set may be found, but not the one that the user needs. (There are actually two solutions per one equation of second degree, and we are talking about **combinations** of solutions!) Anyway, an extra feature is better than a complicated sketch.*

Hopefully this exercise introduced you to Creo interface, basics of feature based modeling and good modeling practices. The following exercises will not be as completely documented as this one, so you should now try on your own, maybe repeat the exercise without this document.

Save your model.

About Family Tables

In the first part of this exercise, a clock hand was modeled. A typical clock has at least two hands, maybe three. What should we do to create more hands, having different lengths?

Some (bad!) suggestions:

- 1) Change the length dimension of one hand to whatever needed.
- 2) Start a new model with different name; redo the modeling identically with the exception of different length.
- 3) Save a copy as minute_hand, change dimensions, rename to hour_hand.

What is wrong with these methods? In 1), there will be only one model (and possibly several identical instances of it). CAD models are often referenced by another models, for example, a part may belong to one or more assemblies. There may also be several copies of a part in a single assembly. Such copies are called instances (“ilmentymä” olisi hyvää suomea, mutta “instanssia” käytetään). It is characteristic to instances that the changes made to the original model are updated to them. This means that all instances of the hand model will be identical in a clock assembly.

The end results of 2) and 3) are similar. In these cases, there are two different models, which of course have different instances. 3) avoids the extra work of creating another model. A problem arises if clock hands in general need modifications. For example, if the attachment hole diameter needs to be changed, the modifications must be made to both two (or all three) models. This does not only cause extra work, but higher possibility of errors. In addition, sometimes the changes are not done manually but by a configurator program.

The bottom line is that sometimes it is useful to define a family of products (or parts), sharing most attributes but having a few differences. In this case, for example, family of clock hands has everything in common except length and a decorative detail. There are two important ways of enabling different instances of a model in Creo. This exercise introduces *Family Tables*. The other one is using flexible components in assemblies.

Modeling

(If the name of the part is something else, rename it to hand.prt and save, Figure 24).

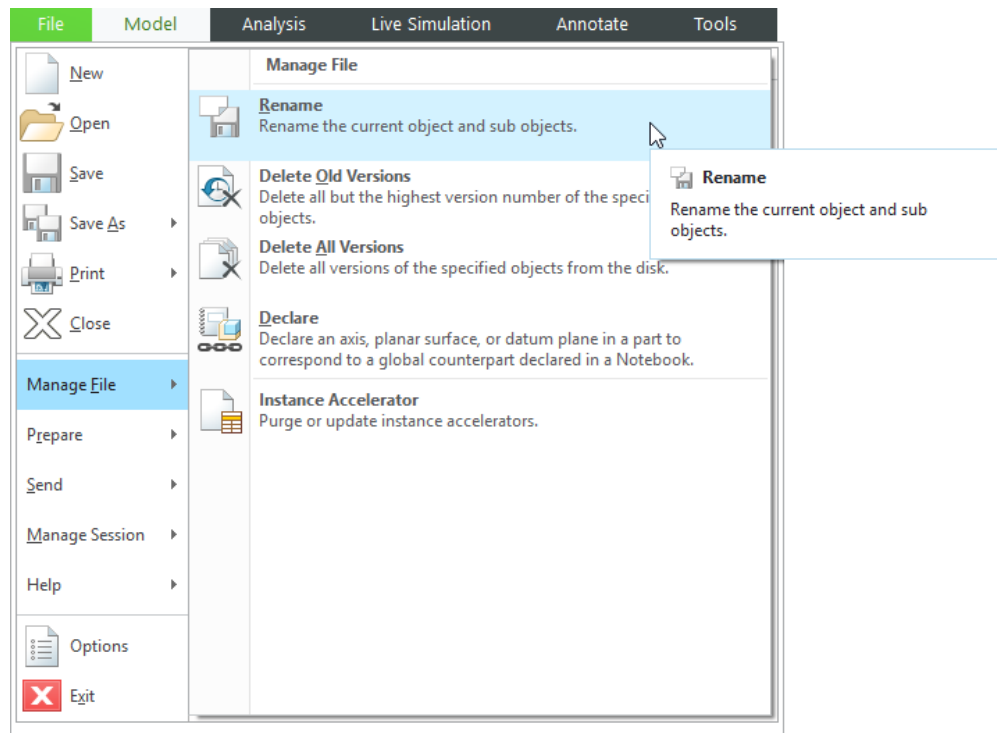


Figure 24: Renaming part.

Change the width of the first extrude to **60** (select the first extrude feature from the model tree, **RMB** and select **d1**). Then create **Extrude** (📐, *Shapes* group) on the **FRONT** plane as seen in Figure 25 using **Circle** (🕒) tool while in *Sketch* mode.

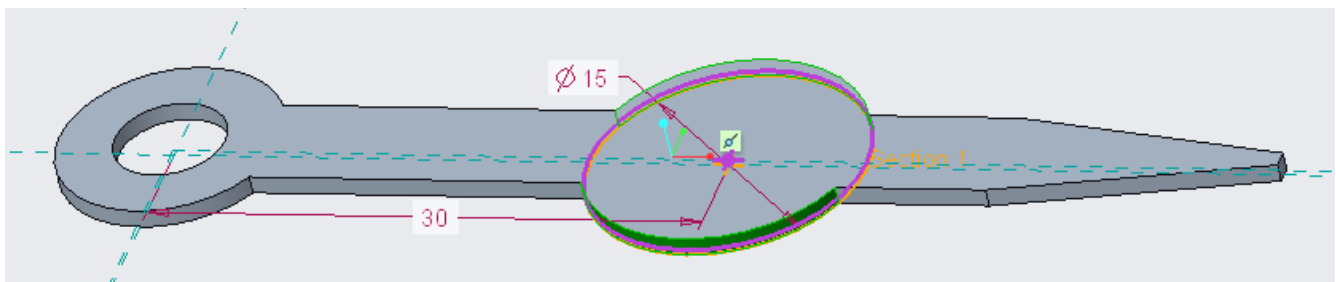


Figure 25: Created feature and its dimensions.

Next, we create a hole in the middle of previously created feature. Select **Extrude** (📐) and use FRONT as a sketching plane. In the sketching mode, select **References** (📐) from the *Setup* group. This opens a window, where all references (geometries where the sketch can snap) are presented. Select the edge of the previously created feature as a reference (Figure 26). Click **Close**.

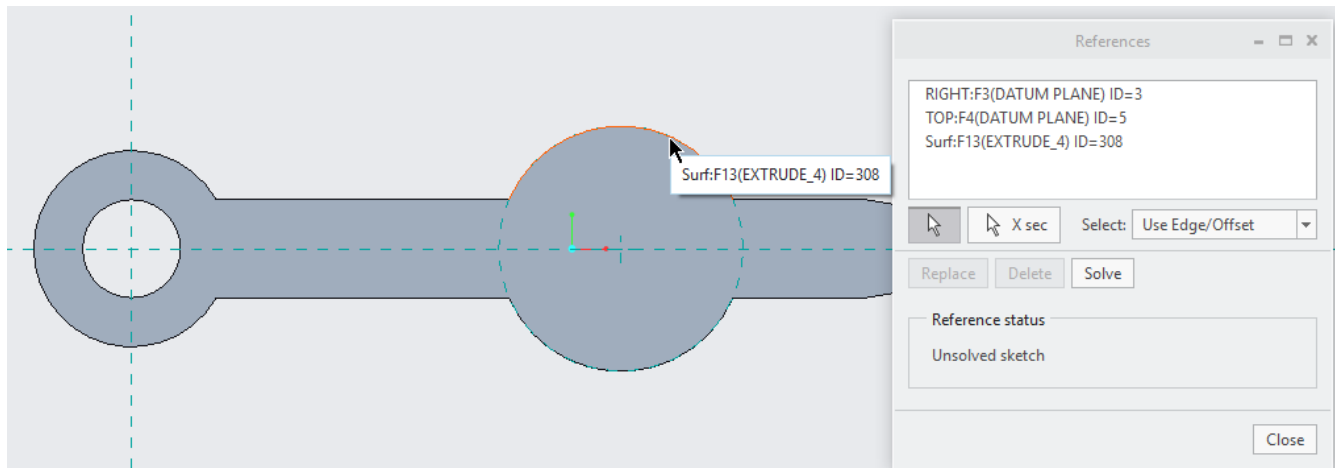


Figure 26: Selecting edge (actually surface) from the previous feature.

Selecting an edge (actually a surface) gives us a center point of the previous cylindrical shape. Using that center point, create a **Circle** (📐, *Sketching* group). Using **Dimension** (📐, *Dimension* group), select the reference circle, then created circle and press MMB in the middle of these two (Figure 27) to create a distance dimension. Change its value to **2**. Now we have a dimension that defines the thickness of the cut in the middle.

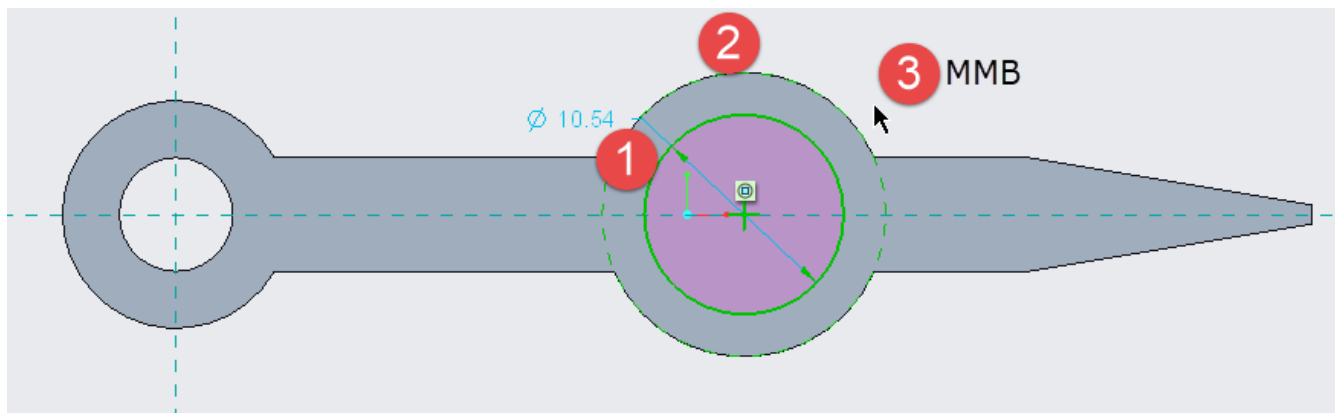


Figure 27: Reference (1.) selected, selecting circle (2.) and then press MMB in the middle of these two.

Accept the sketch (✓). Creo probably has put *Remove Material* (✂) option on. Change the depth of the cut to **Thru All** (≡) and accept the feature (✓ or **MMB**).

Parameters

First we create parameters to our model and associate those to geometry thru relations. Select **Relations** (≡) from *Model Intent* group. Like in Exercise 1.1 (Battery), create parameters to the *Local Parameters* field as seen in Table 1.

Table 1: Parameters for this model.

Parameter Name	Type	Value
LENGTH	Real Number	60
WIDTH	Real Number	6
DEC_LENGTH	Real Number	30
THICKNESS	Real Number	1
DEC_DIA	Real Number	15

Next parameters are associated to the features. In the relations field, define the following relations as seen in Figure 28. (NOTICE! You may have different dimension names, use the corresponding ones!) When ready, accept *Relations* (**OK**).

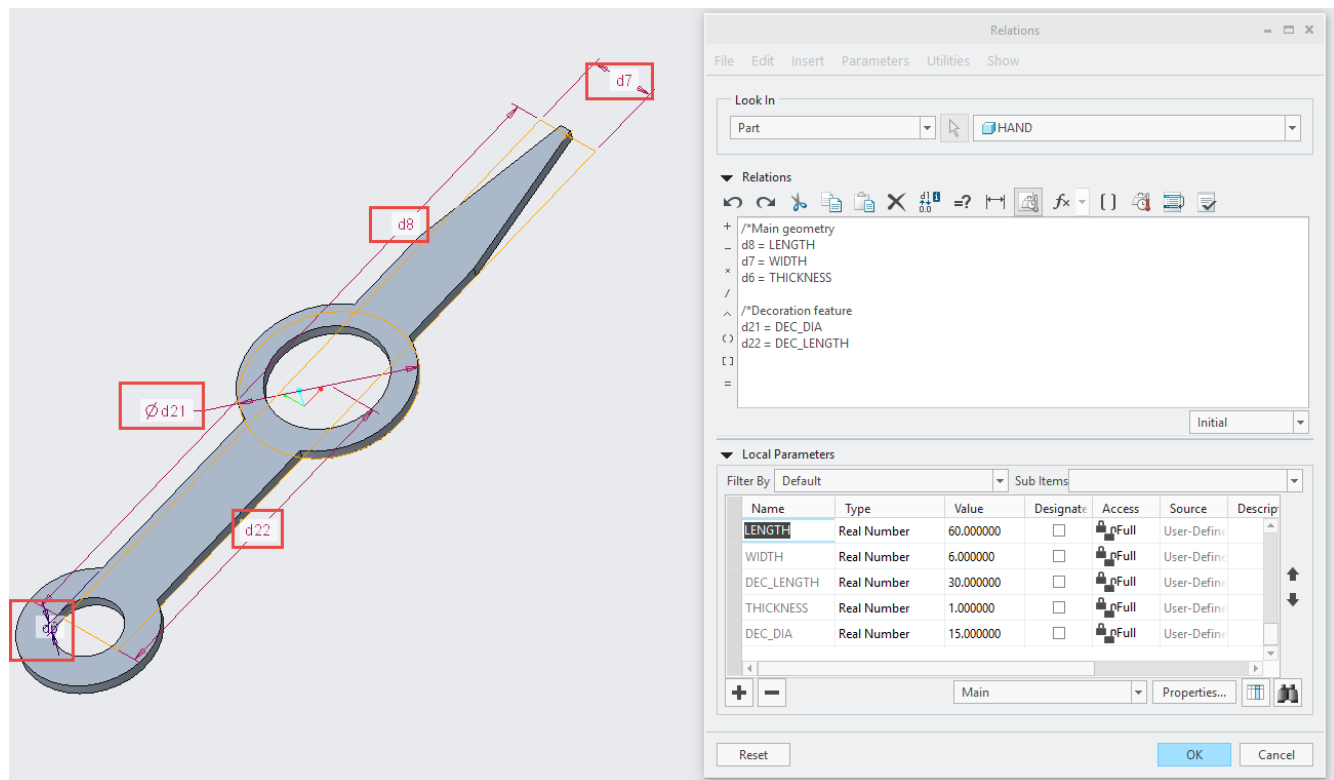


Figure 28: Parameters associated to the relations. NOTICE: You may have different dimension names (d#)!

Creating Family Table

We will add a family table into this part model, enabling simultaneous existence of a long hand instance without the decorative detail, and a short one with the detail.

Select **Family Table** (📊) from *Model Intent* group (Figure 29).

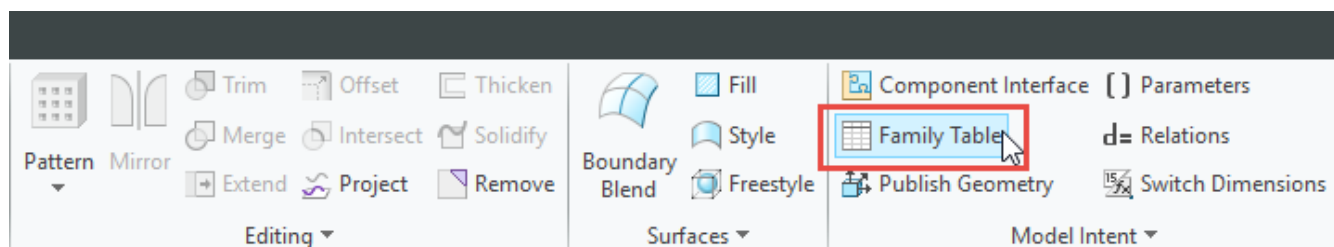


Figure 29: Selecting Family Table from Model Intent group.

As the family table editor appears, click 📊 two times to insert two rows into the table. The rows represent different model instances, so enter “Minute_hand” and “Hour_hand” to the cells in the *Instance Name* column (Figure 30).

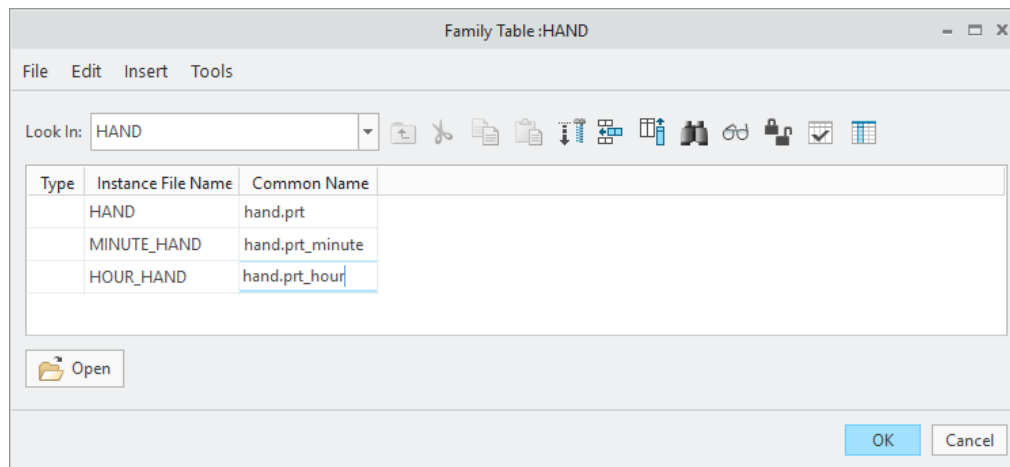



Figure 30: Family Table window with two new instances (rows).

The columns (except *Type*, *Instance Name* and *Common Name*) represent dimensions, features etc. that differ between the instances. Let us first define that the length dimensions differs, and enter its values for both instances. Click  to insert a column. The *Family Items* window appears. Select **Parameter** from the *Add item* field. From the *Select Parameter* list, select **LENGTH** and **DEC_LENGTH** parameters to be inserted (**Ctrl** to select multiple, then **Insert Selected**) and **Close**. (Other three parameters are used by the auto-assessment system.) In the *Items* list, you should have now two parameters listed (Figure 31).

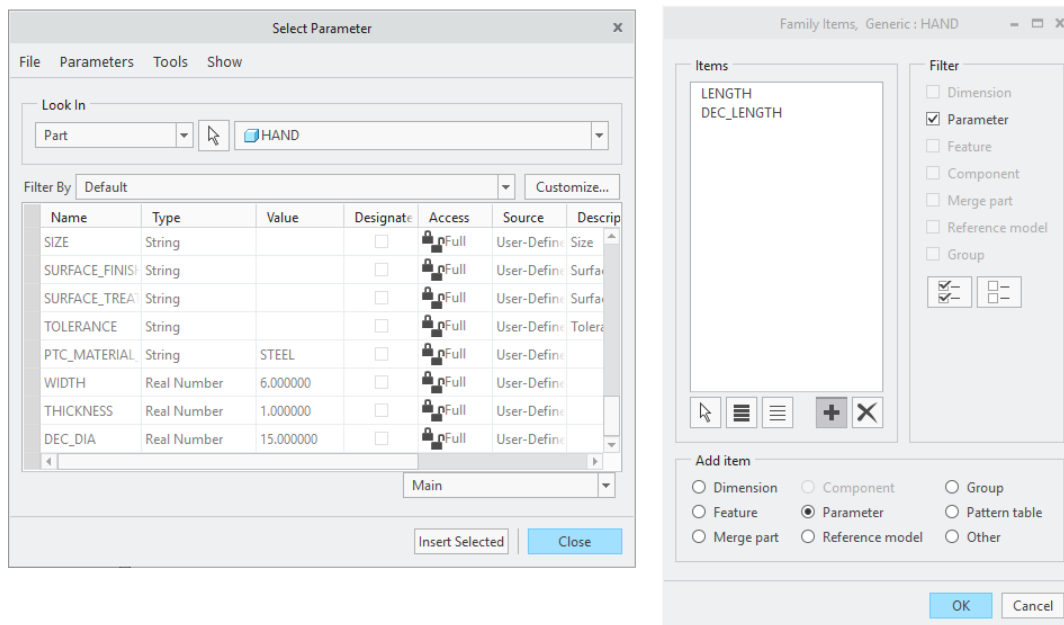
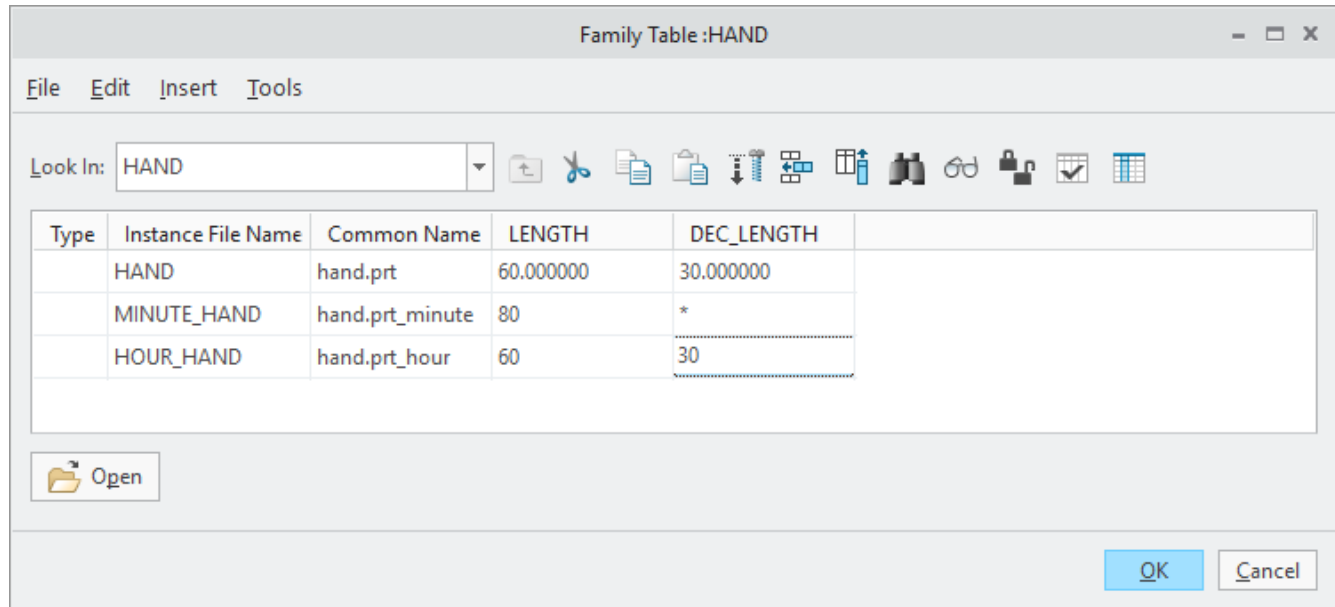


Figure 31: Parameters added to the Items list.

Enter length values for minute and hour hand instances, **80 and *** and **60 and 30**, respectively (Figure 32). Click **OK**. (* means that the instance is using the value from the HAND parent part.)



Type	Instance File Name	Common Name	LENGTH	DEC_LENGTH
	HAND	hand.prt	60.000000	30.000000
	MINUTE_HAND	hand.prt_minute	80	*
	HOUR_HAND	hand.prt_hour	60	30

Figure 32: Updated Family Table.

Note about Instances

Notice text “Instance: GENERIC” in the graphics area. It indicates that you have the original, generic hand model (instance to be exact) opened. To open one of the “non-generic” instances, Click **File, Open...**, select hand.prt. Use the *Select Instance* window to choose minute_hand. See how instance name is shown in the graphics area (Instance:MINUTE_HAND).

Did you notice that the family table seemed already to be applied to the model on hard disk? You selected hand.prt in your working directory, and got the *Select Instance* window without saving after creating the family table? Well, the model with family table actually was not opened from the hard disk. Creo always first checks the memory (for models with the entered name) as *Open* command is given. No error messages are displayed, since no errors are encountered.

Adding Features to Family Table

We will add items that are different between the instances to the family table. The decorative item will be made to appear only in the hour_hand instance. Close the window of the minute_hand instance: **File, Close** (⌘Z). Erase it from memory: **File, Manage Session, Erase Not Displayed, OK**.

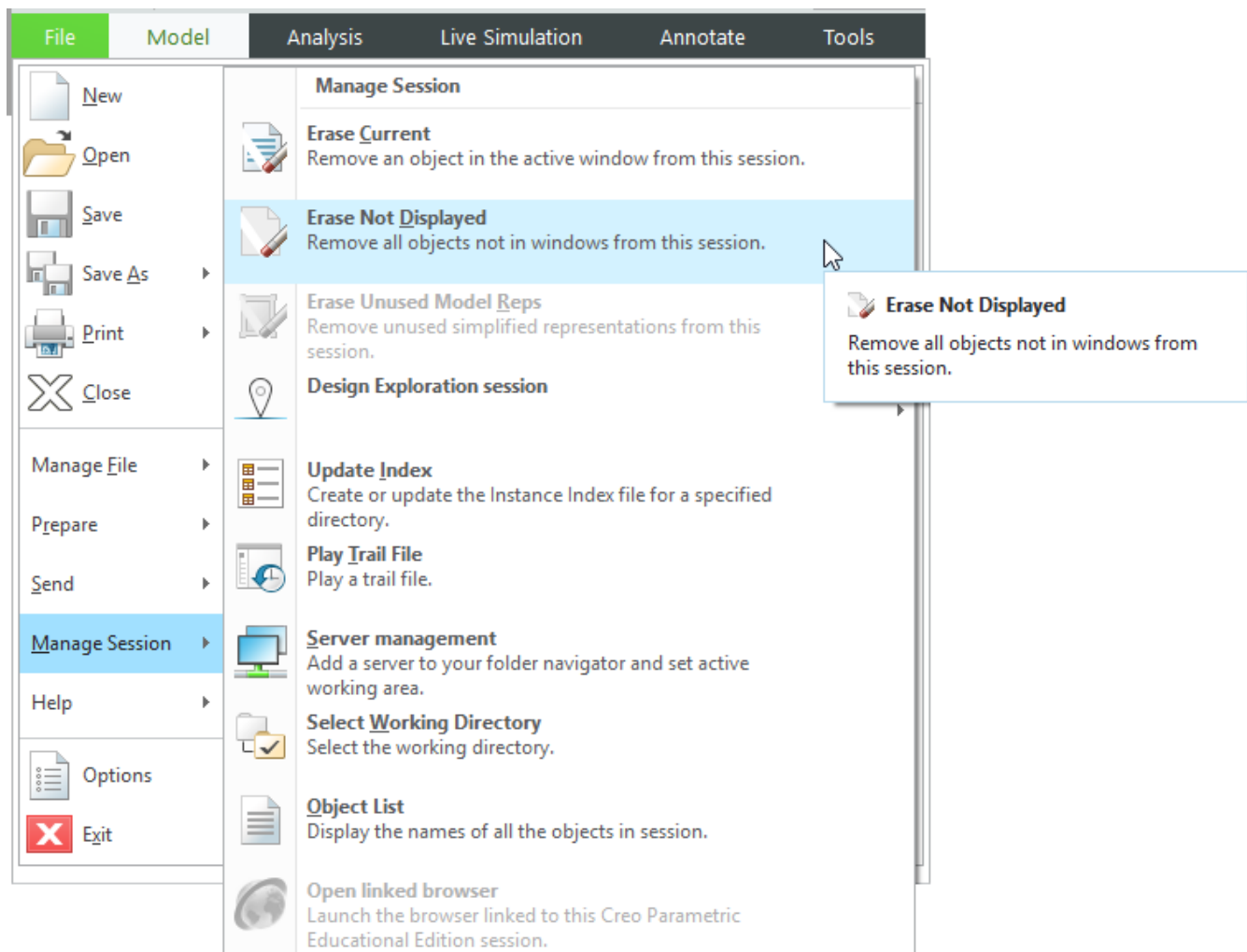


Figure 33: Erasing memory.

As the generic instance is active, select **Family Table** (📊). Click 📌. This time, select *Feature* in the *Add Item* section. Select the first of the two extrusions of your decorative detail from model (Figure 34, feature name may be different), **Done**, **OK**.

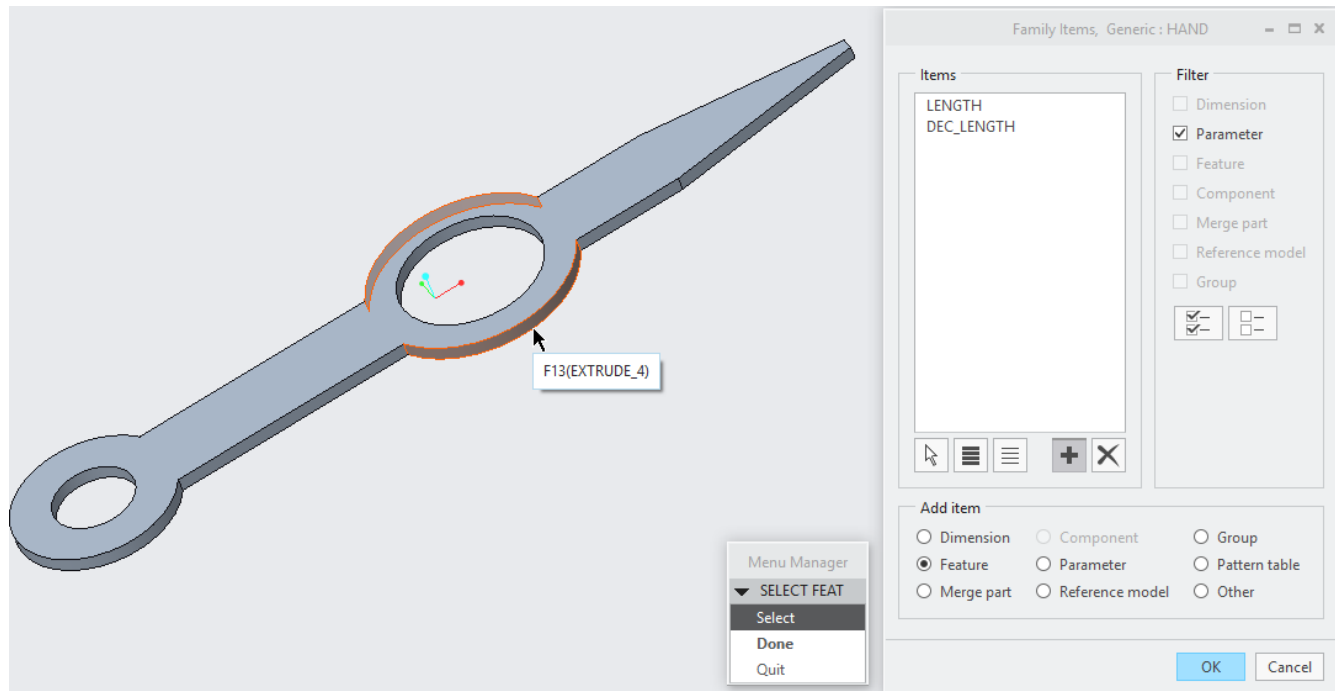


Figure 34: First of the two added features selected.

The options for feature items in family table are Y (yes) and N (no). Y means that the feature exists in the instance, N that it is suppressed. Set the values as in Figure 35; **Y** for hour_hand and **N** for minute_hand. Click **OK**.

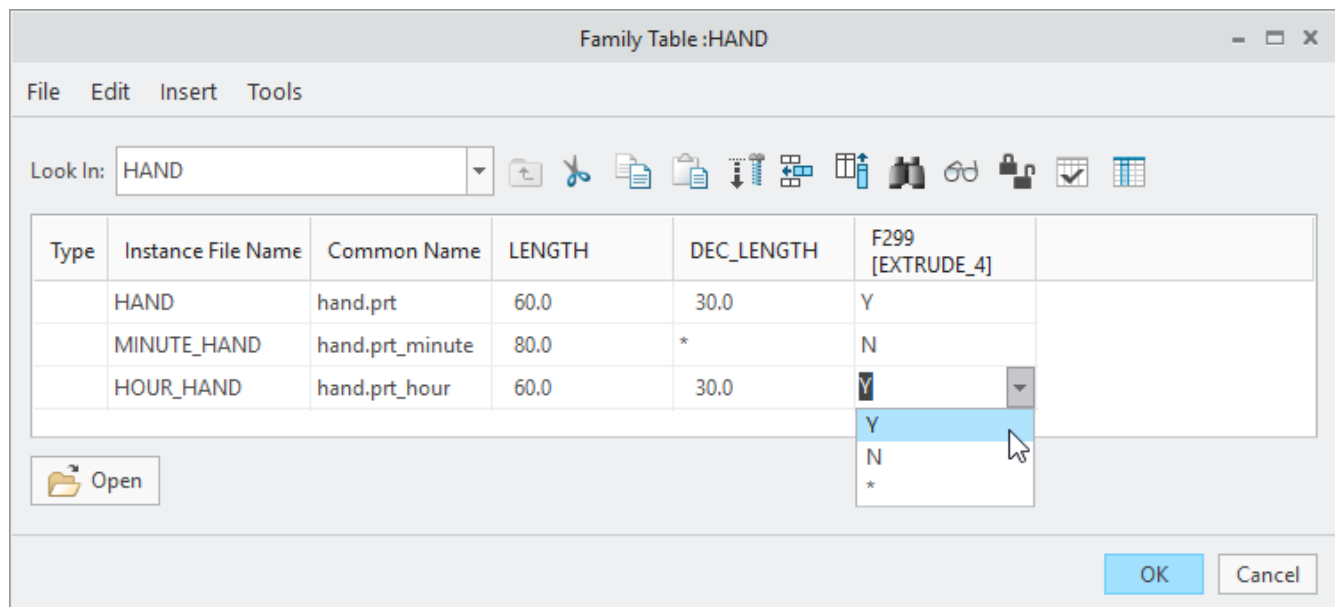






Figure 35: Feature added to Family Table.

Now it is really important that you erased minute_hand instance from memory (Do it if you did not!). Otherwise, when opening it now, the older version (without the changes to the family table) would be retrieved from memory. Select **Family Table** () , select MINUTE_HAND and click (). A *Preview* window will open. Click **Close**. You can also see minute_hand by selecting **Open** (). This will open minute_hand in the normal part mode (notice Instance: MINUTE_HAND). **Close** it ().

Why was it sufficient to suppress the first extrusion only? (Was it in your case, did you use it as sketcher reference when sketching for the cut?) This is because the following cut feature is a child of the selected feature, and child features are suppressed (or deleted) along with their parents. In this case, the position of the cut is dependent on the position of the parent feature. Sketching planes, sketch orientation reference geometry and sketcher reference geometry are the most common causes of parent-child relationship in part modeling.

This concludes our exercise. Did you remember to **save** your model?