

EXERCISE 3.X – ENGINEERING DRAWINGS TOOLS

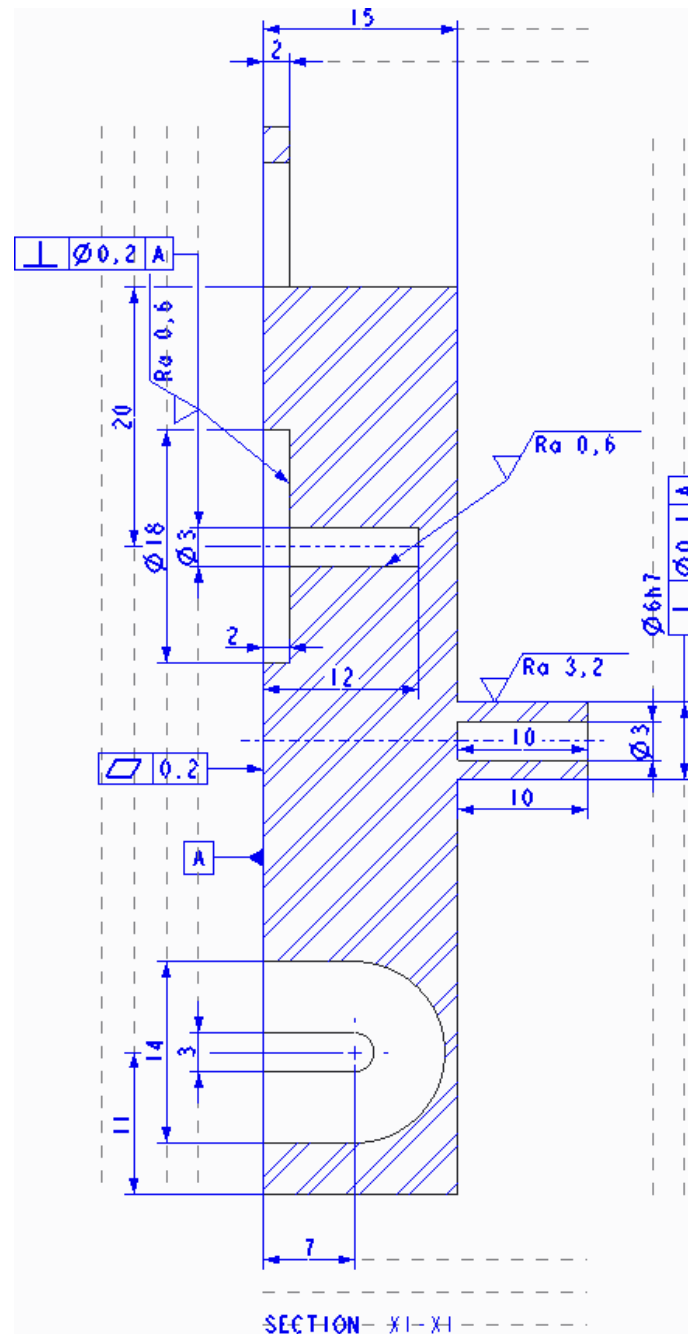


Figure 1: A drawing.

Learning Targets

In this exercise you will learn

- ✓ to use drawing templates
- ✓ to select views and projections
- ✓ to create cross-sections
- ✓ to place dimensions and tolerances
- ✓ to place other drawing symbols
- ✓ to export drawings.

This paper shows how different tools work in the drawing mode. This is created for Creo version 3.0, but majority of the tools should work in same way in Creo 6.0.

A model named *mechanism.prt* is used as a demo part in this paper.

Creo engineering drawing file (.drw) is essentially a set of instructions on how to create a 2D drawing from an existing 3D part model. The drawing file is always dependent of the part file, and does not by itself carry any information about the part geometry. Thus, drawing files automatically update to reflect changes in the part files, but naturally cannot be distributed separated from them.

Layout

Layout tab controls projections and cross-sections.

Creating New Drawing

1. Open the part or assembly file that you want to create the drawings for.
2. Now create a new file, choosing “*Drawing*” as its type. As you can see, the model you opened earlier becomes the default model for the new drawing.
3. When modeling at Aalto, it’s recommended to select “*Empty with format*” for the template, and browse for “*A3_AALTO.FRM*” for your drawing format (Figure 2). This will define the paper size, orientation, background graphics and possible information layout automation for parts lists etc.
4. Click “OK” to accept the format and you will be entering the drawing mode.

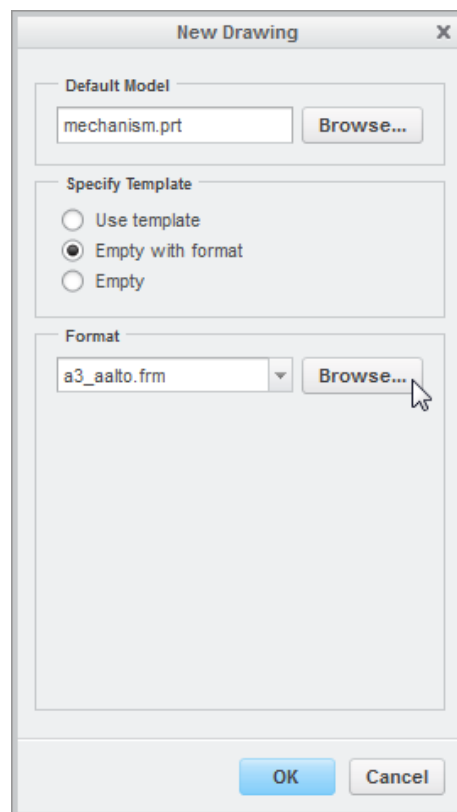



Figure 2: Settings for drawing (mechanism.prt).

Adding basic views

1. Hold **RMB** on the drawing board and select **Insert General View** (the same can be achieved by clicking on the **General**  icon in *Layout* tab). Keep option **No Combined States** (Figure 3):

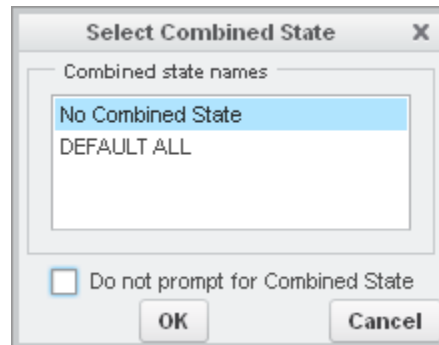


Figure 3: Select Combined States -window.

2. Creo asks for the location of this view, click somewhere on the drawing board.
3. A *Drawing View* window will open, from which you can customize all aspects of your drawing:

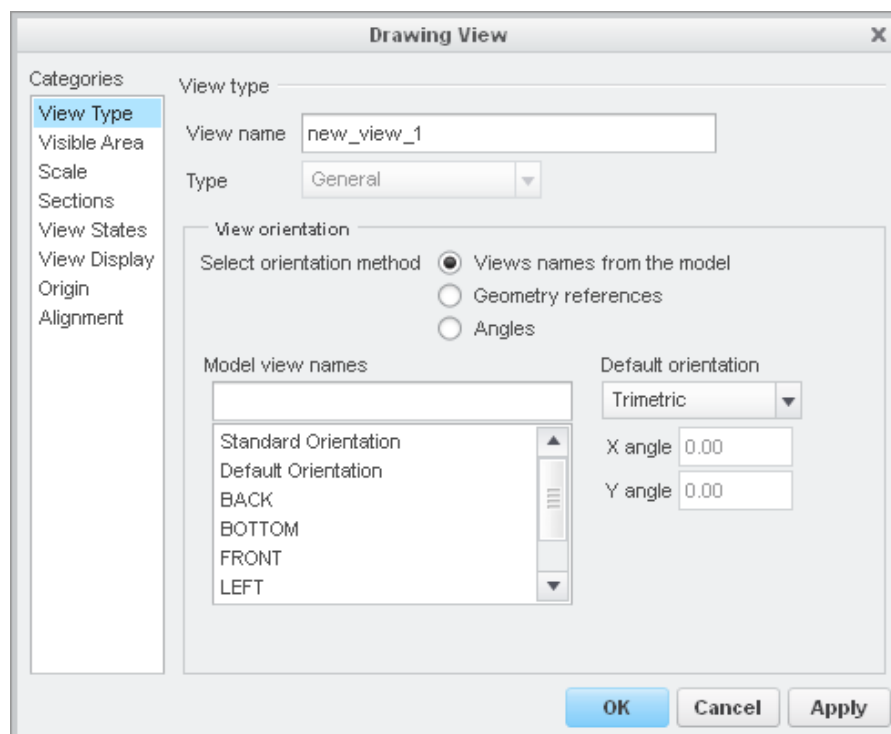







Figure 4: Drawing View -window.

4. From the topmost category (*View Type*) you can select which of the model's views to use as this view (*Model view names*). Double-click to activate one of the model's views. For a custom view or orientation, you can either define one here using the *Geometry references* option, or go back to normal 3D modeling mode and create new orientations to the model view list using the **View Manager** () tool.
5. From the *Scale* category you can specify a special scale for this view only, but if you want all of your views to have the same scale, you should edit the drawing board's general scale by double-clicking on the very left corner of the screen (by default, *SCALE: 1,000*).
6. From the *View Display* category you can set up the appearance of this view. Default option is *No Hidden* for the *Display Style* in normal projection views.
7. You can apply and close the *Drawing View* window. It can be re-opened by double-clicking on the view.
8. Note that the view's position is by default locked. Activate it by clicking on in (it should get green dotted lines around), then hold **RMB** to open a menu, from which uncheck the () **Lock View Movement**. Now it can be moved around. You can also use **Lock View Movement** () option from *Document* group (in *Layout* tab).

Adding Projection Views

1. Select the main view. Click and hold **RMB** for a menu, and select () **Projection View** (or **Projection View** () from *Model Views* group).
2. Try surfing around the board with your mouse. You can select the projection direction and position. Click on a suitable position (Figure 5).

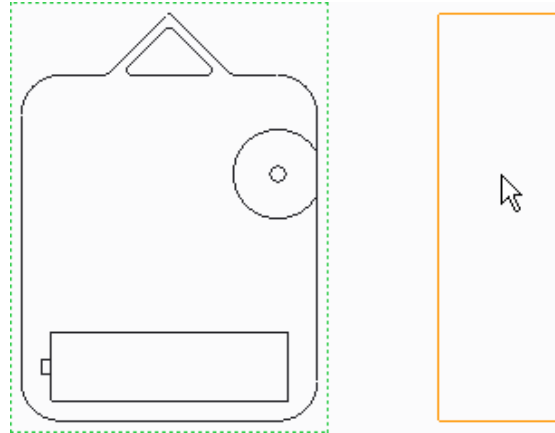


Figure 5: Looking for a position for projection view.

Creating a Detailed (Zoom In) View

1. Select **Layout** tab and then **Detailed** (🔍).
2. Zoom in and select one entity (for an example a line) as your center point of the detailed view. This selection will parametrically control the center point of the view.
3. Select locations to sketch a spline (closed or open) to include the area that you want to include in your zoom-in detailed view. **MMB** to complete the spline. Last, click to position the detailed view on the drawing board (Figure 6).

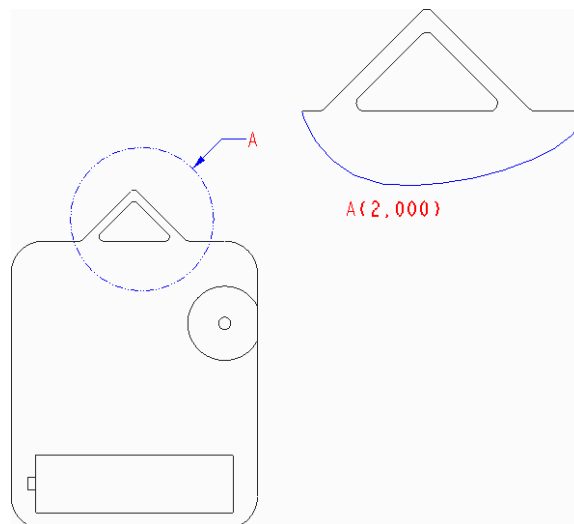


Figure 6: A detail view.

4. You can adjust the scale of the zoomed in view by double-clicking on number value in the detail view. As usual, other options can be adjusted by double-clicking on the view itself.

Adding Section Views

1. It is easiest to first define some section views in the 3D modeling mode. Open the part in normal 3D part mode (). Open **View Manager** (), from *Quick Access Toolbar*. Go to the **Sections** tab (Figure 7).

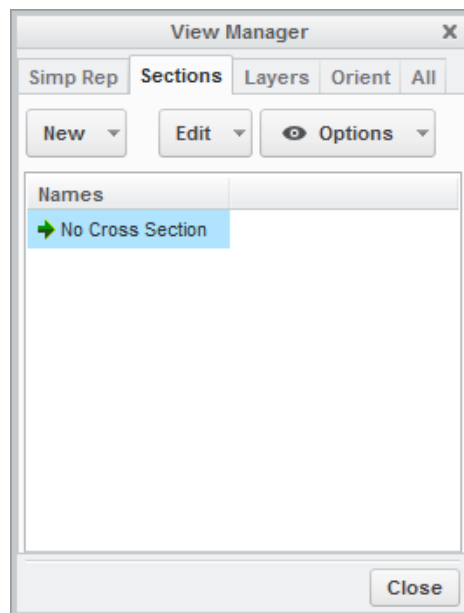


Figure 7: View Manager and Sections tab selected.

2. Create a new section by clicking **New** and select cross-section type:
- Planar**; a simple cross-section defined by a single surface or plane.
 - X/Y/Z Direction**; a planar cross-section using default planes.
 - Offset**; a sketched line on a surface will be extruded infinitely to split the part in order to create the section.
 - Zone**; multiple planes can be used to define the cut-away solid.

Give a name for the new section (for this example X1). When you hit ENTER, a *Section* dashboard will be activated. Options depends on selected cross-section type. In *Offset*-case, select **Sketch**, **Define**.

3. Choose the appropriate type and finish the definition of the section (for ex. in Figure 8, an *Offset* cross-section is defined).

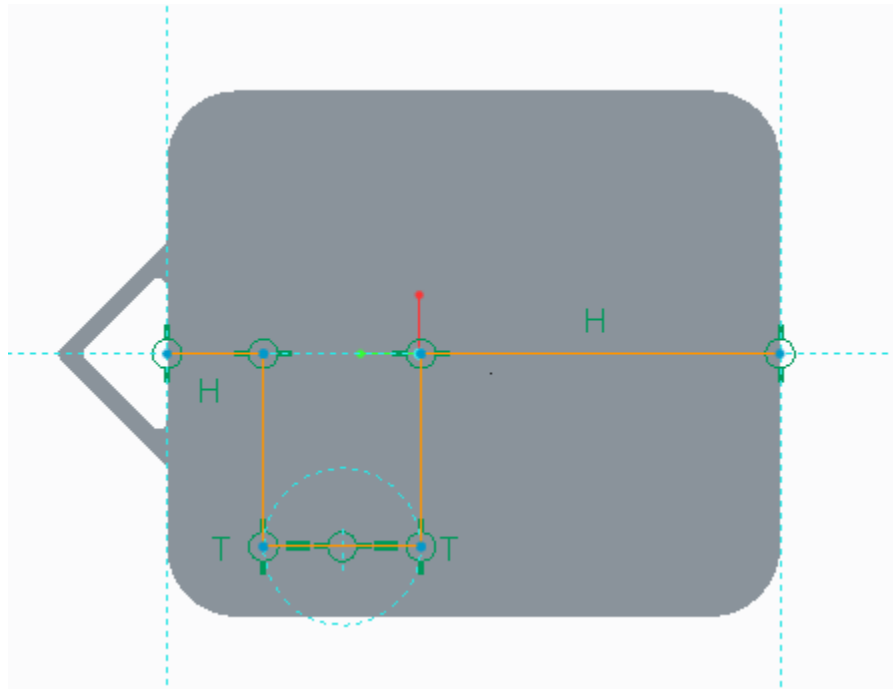


Figure 8: Sketch for Offset-type cross-section.

When ready, accept feature (☒) to get back to the *View Manager*. Close it and return to the drawing mode.

4. If not present already, create a view from a proper direction for this new cross section. This could be done for example by normally adding a projection view.
5. Double-click (or **RMB** menu, **Properties**) to open the settings window.
6. Go to the **Sections** category and select **2D cross-section**. The plus button will allow you to create a new section definition based on cross sections previously defined in the part model. A red cross in front of the section name means that this section cannot be used in this direction; a green check means that it can be used. You can also create completely new section definitions at this point.

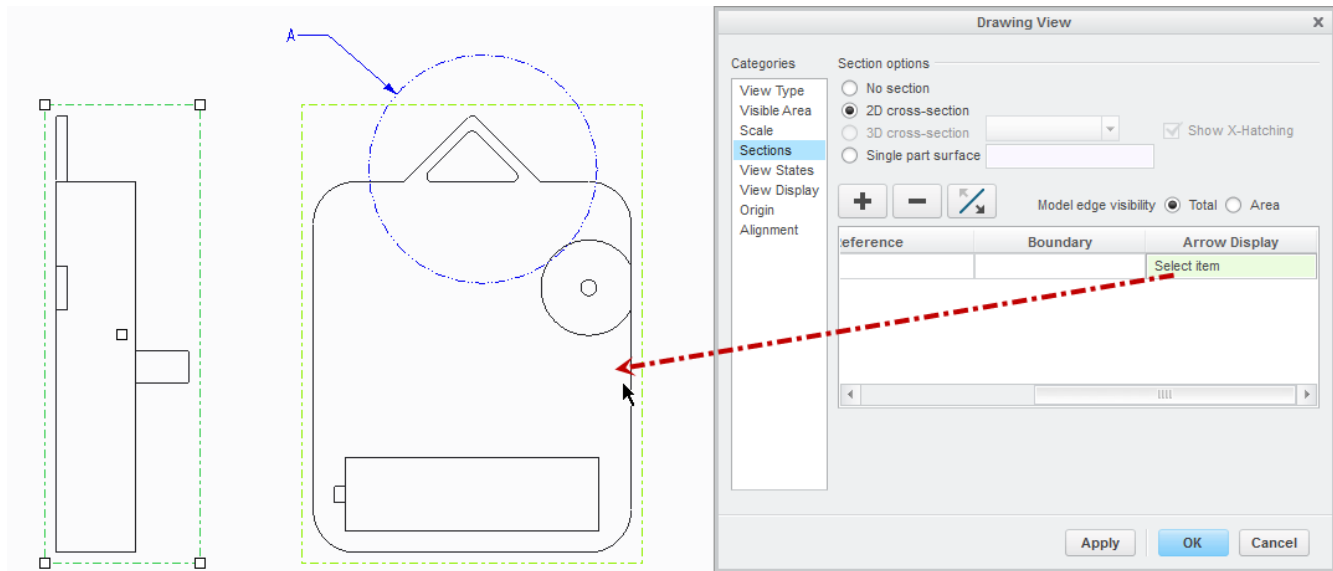


Figure 9: 2D cross-section defined, selecting view for *Arrow Display*.

By clicking on *Arrow Display* column, cross-section arrows can be created (Figure 9). After successfully selecting or creating a section, click **OK**.

7. The section is ready, but you might want to adjust the cross-hatching. Double-click on the hatching to enter a menu, and click **Spacing** -> **Half /Double** until you are satisfied (Figure 10), and finally **Done**.

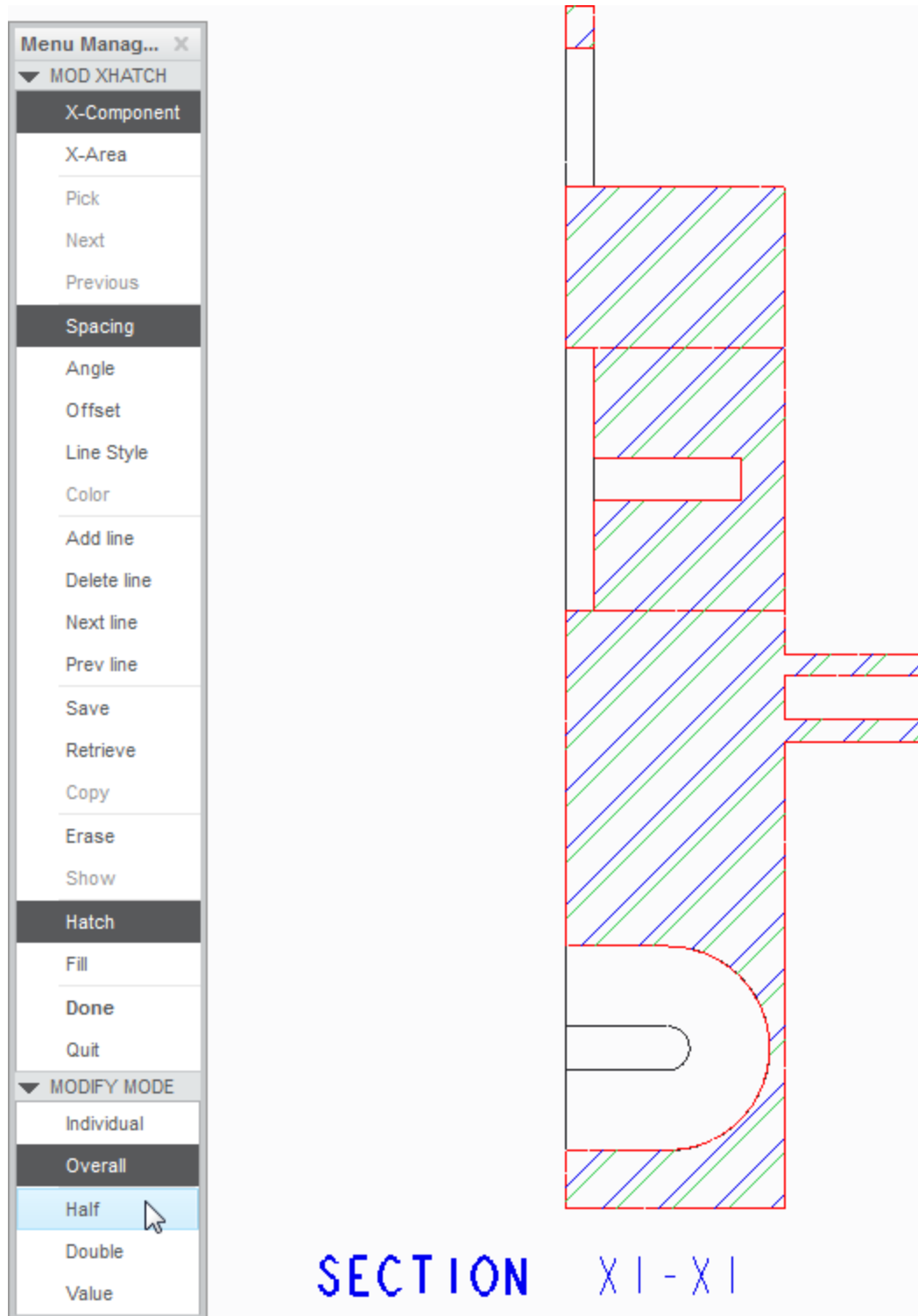


Figure 10: Created cross-section on right and spacing adjustment on left.

Annotate

Adding dimensions, tolerances and other drawing symbols happens in **Annotate** tab.

Adding Axes

1. Click on **Show Model Annotations** (📐) in the drawing toolbar, *Show Model Annotations* window appears.
2. Select the *Model datum* tab (Figure 11).

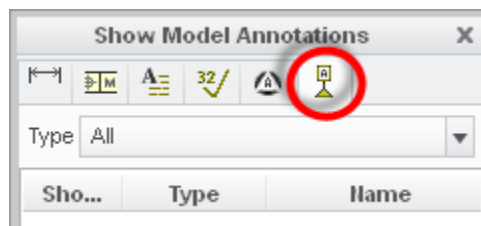


Figure 11: Model datum tab.

3. Select the view / feature which axes you want to see (Figure 12).

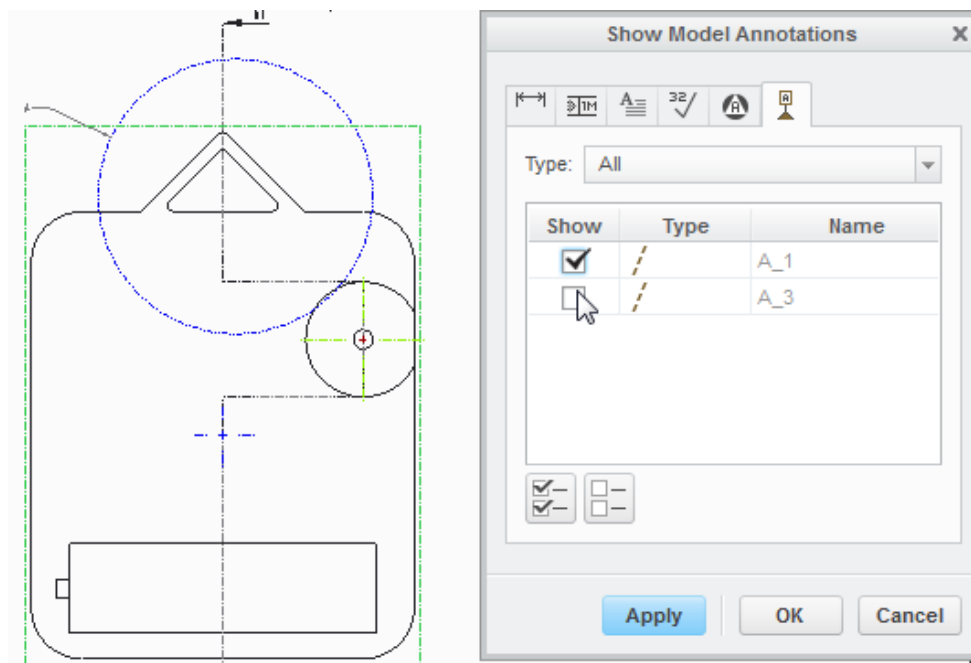


Figure 12: View selected, one axis selected, selection another one (green highlighting)

4. NOTE: If you need an axis and in the model you don't have one, you just need to create a new axis! You can do it in the part mode (using *Axis* (/)) from *Datum* group) or in drawing mode (Figure 13).

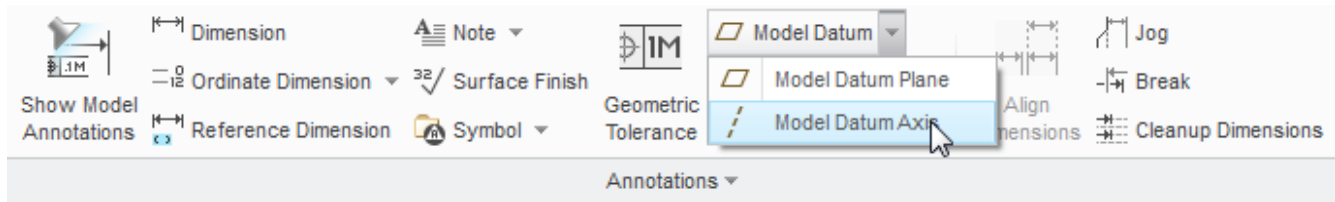


Figure 13: Model Datum Axis.

For example, in Figure 12 battery slots needs to have an axis (Figure 14).

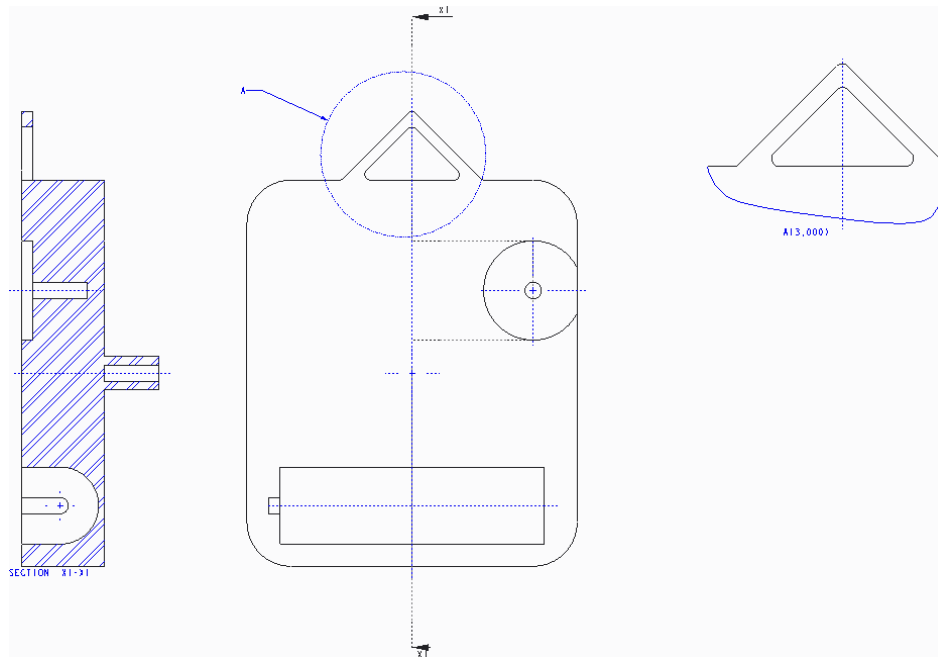



Figure 14: Axis for battery slot created.

Adding Dimensions

Semi-automatic dimensioning

The *Show Model Annotations* tool shows the dimensions that the 3D model has (i.e. sketching dimension, protrusion lengths etc.)!

1. Click on **Show Model Annotations** (IM) in the drawing toolbar, *Show Model Annotations* window appears.

2. Click on **Dimensioning** () tab and select wanted type (in this case *All*)
3. Select the feature or view which dimensions you want to add.
4. In *Show Model Annotations* window choose wanted dimensions by clicking in the box (Figure 15).

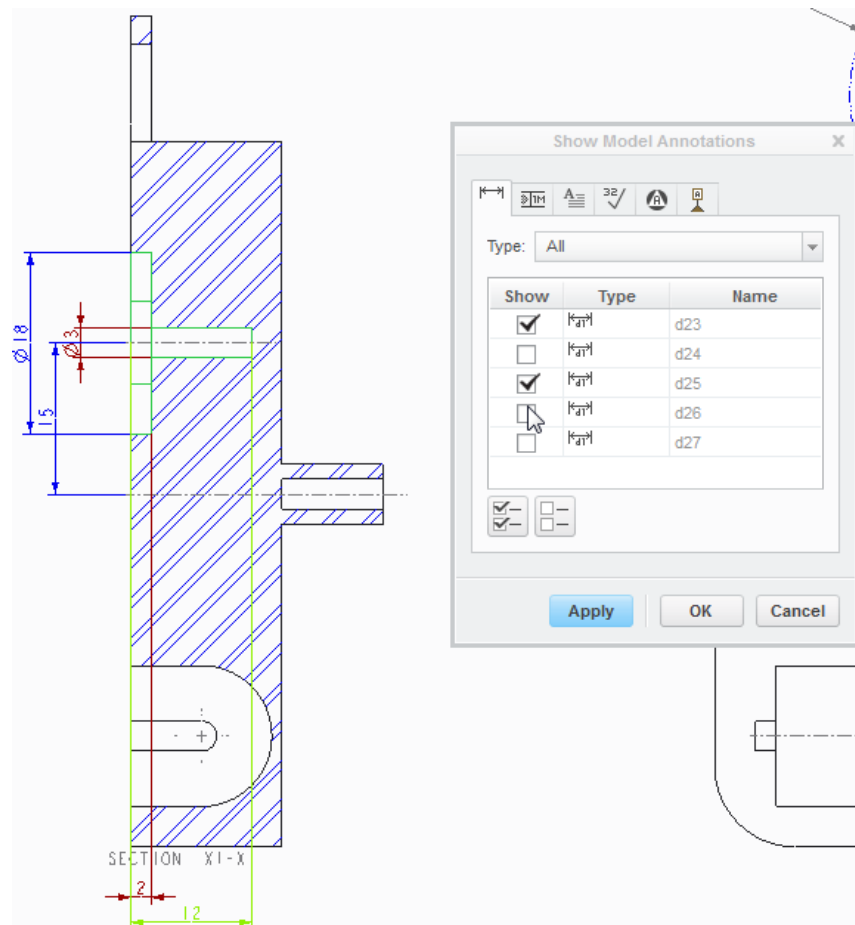




Figure 15: Selecting dimensions. Blue selected, red unselected.

5. Accept with **Apply**. When all dimensions you wanted are placed, close the window (tool).
6. The procedure is ready. Select and drag the dimensions if needed. You can also automatically position the dimension by using the **Cleanup Dimensions** () from *Annotations* group).

Manual dimensioning

1. You can use the **Dimension** tool ( from *Annotations* group) to create dimensions that are not based on dimensions in the model, but rather measurements. When creating dimensions, press CTRL to select additional references (Figure 16). Accept placement with **MMB**.

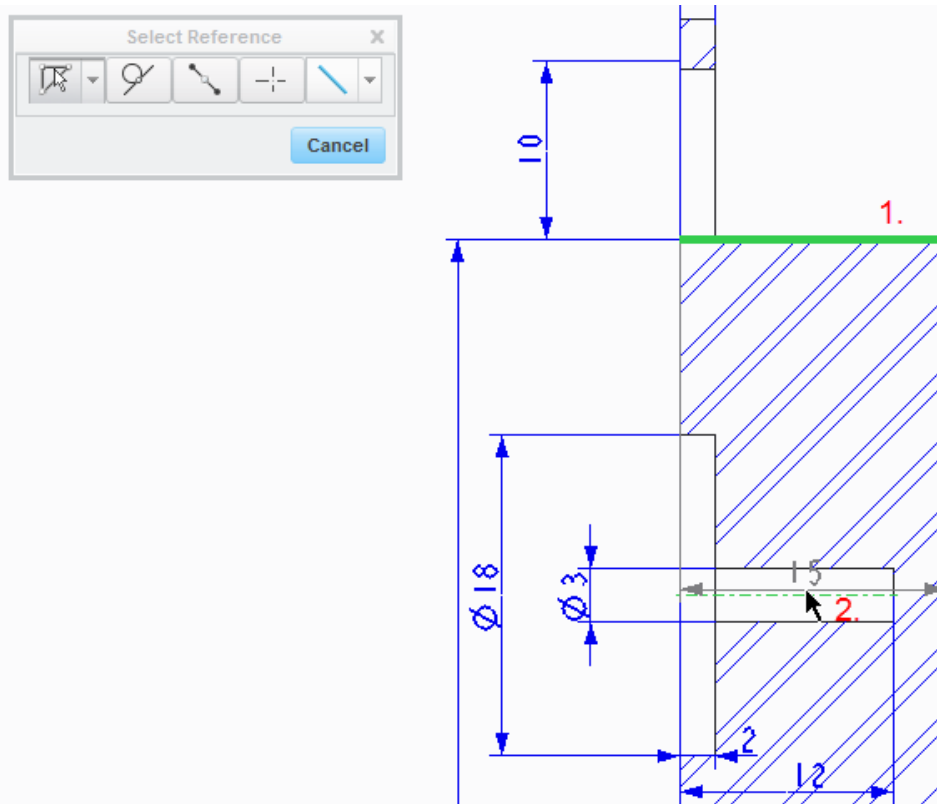


Figure 16: Line selected (1.) and holding CTRL selecting axis (2.).

2. Dimensions created with *Dimension* tool are not visible in part mode (3D mode).

Other notes

- Clicking **RMB** on drawing view and selecting **Show Model Annotations** shows all dimensions in model that are visible in the selected view.
- •Dimensions can be deleted by selecting the dimension and hold **LMB**, **Delete** (or by pressing DEL). **Erase** command just hides the dimension from the view. You can view annotations and other drawing objects from the *Drawing Tree* (Figure 17Figure 17: Drawing tree and right_5 projection expanded.).

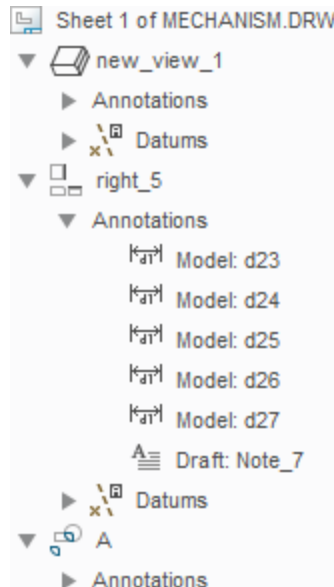


Figure 17: Drawing tree and right_5 projection expanded.

- If you want to change the arrow direction of the dimensions, Select a dimension, click and hold **RMB** for the menu, select **Flip Arrows** (Figure 18).

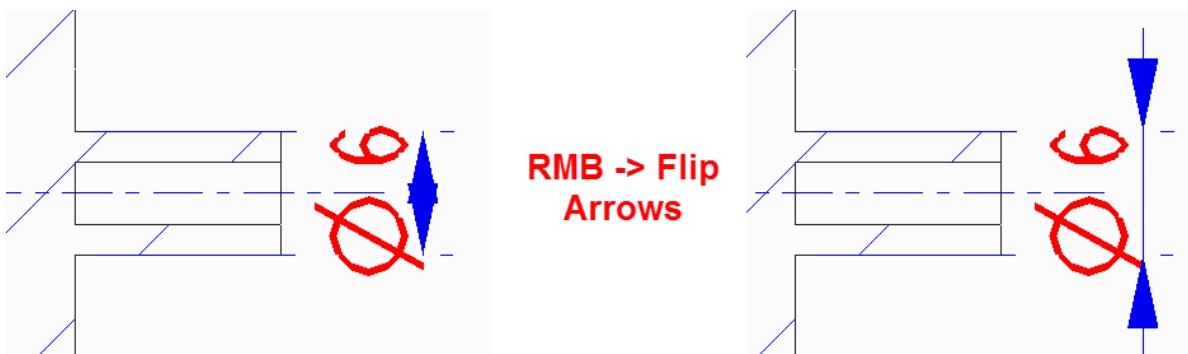


Figure 18: Flipping arrows.

Adding Tolerances to Dimensions

1. Double-click on the dimension baseline (or select it and open **RMB** menu and select **Properties**).
2. Select the wanted *Tolerance Mode* (*Nominal, Limits, Plus-Minus, +-Symmetric*). You can use default *Tolerance table* (*General*) or custom values (*none*). Tolerance table values come from part's general tolerance defined in the part mode.

3. Type in the values below it (Figure 19).

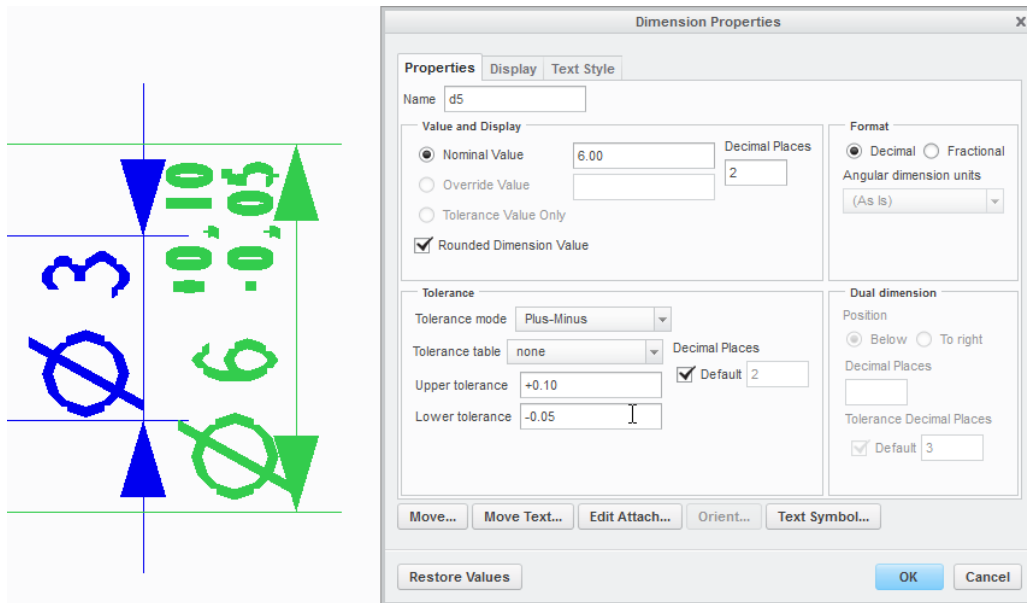


Figure 19: Tolerance mode changed to *Plus-Minus* and Tolerance table to *none*.

4. To add tolerance classes (h7, h6 etc...), select *Display* tab and add correct *Suffix* (Figure 20). You can also add *Prefix* if needed (for ex. for dimensions created by Dimension tool).

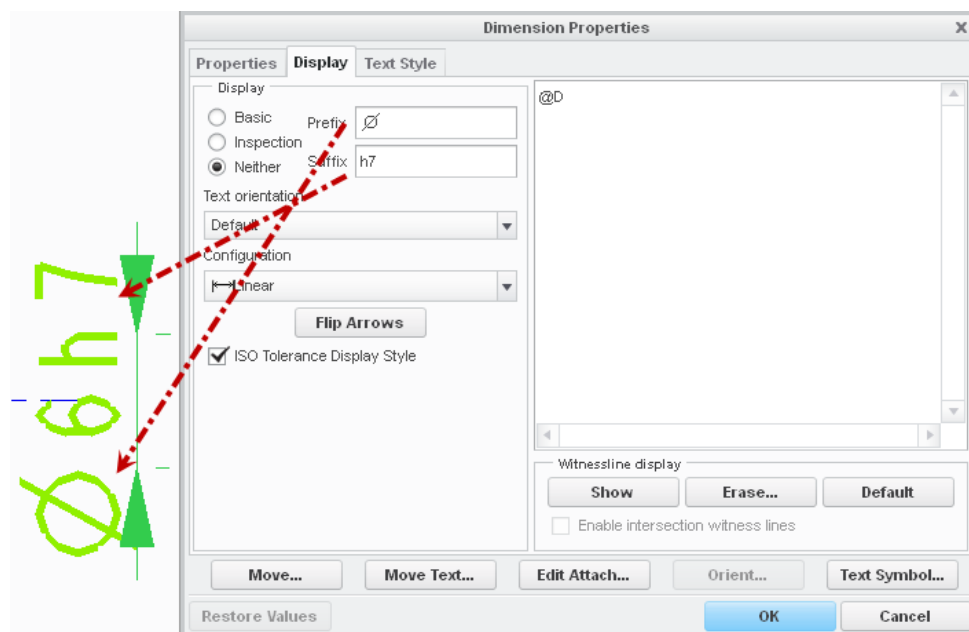




Figure 20: Dimension with Prefix (Ø) and Suffix (h7).

Cleaning-up Drawing

To make engineering drawing readable, several tools can be used.

Cleanup Dimensions

1. Select dimensions you want to clean.
2. Hold **RMB** and select  **Cleanup Dimensions** (or select **Cleanup Dimensions** () from *Annotate* group).
3. Select desired parameters for *Offset* and *Increment*. Check that **Create Snap Lines** option is on.
4. Click **Apply**, change values if needed and select **OK**. Dimensions are cleaned (Figure 21).

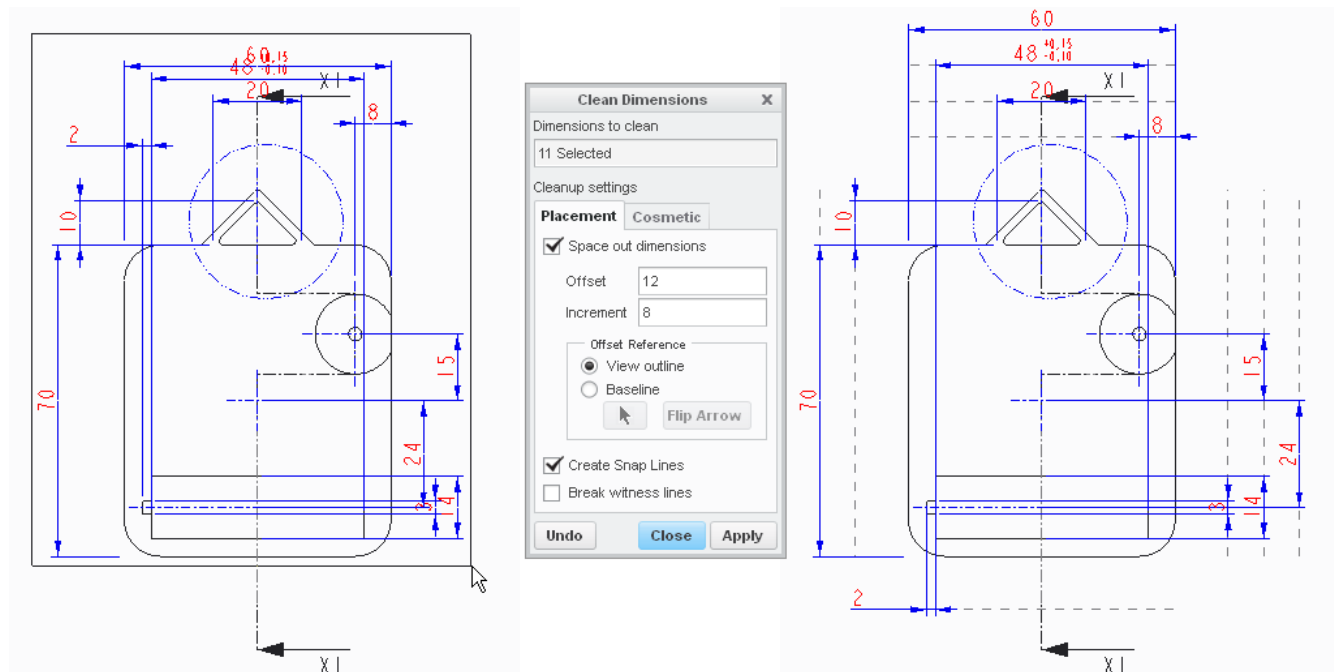


Figure 21: On left: Selecting dimensions. On middle: Cleanup Dimensions window and options. On right: Cleaned dimensions and created Snap Lines.

5. Of course this tool isn't perfect; you still need to manually change locations of dimensions. You can use the created snap lines to rearrange dimensions (snap lines will be automatically hidden when printing).

Witness Lines

1. Select dimension which witness line you want to modify.
2. Hover your mouse to the end of the witness line (black box).
3. Drag and move the witness line to the desired location (you can hold SHIFT to snap witness line to geometry (Figure 22).

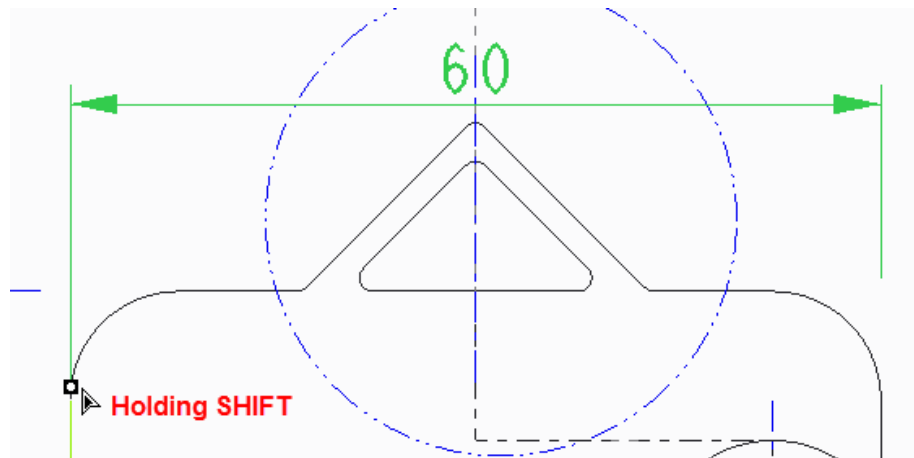



Figure 22: Holding SHIFT and selecting end point of an edge.

4. You can also break witness lines to make your drawing clearer. Select the witness line, hold **RMB** and select  **Insert Break**. Select the start and the end point of the break (Figure 23).

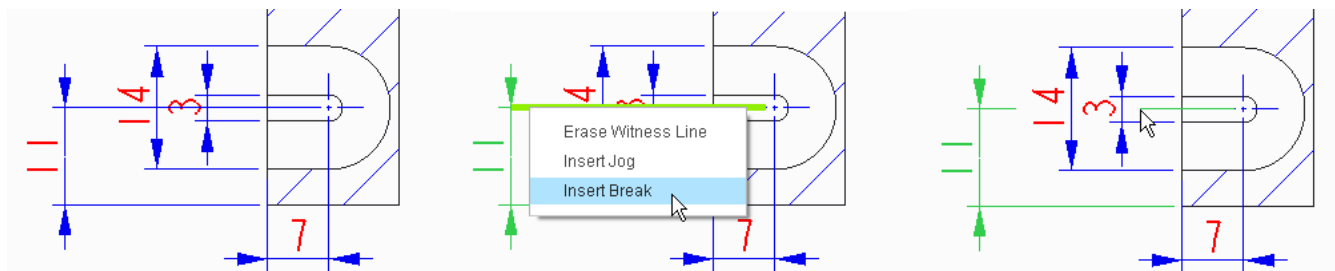




Figure 23: On left: Witness line of dimension 11 overlapping other dimensions. On middle: selecting *Insert Break*. On right: the start and end point of the break selected. Dimensioning is now cleaner.

6. You can remove witness line breaks by selecting from **RMB** menu  **Remove All Breaks**.

Center Lines

1. Sometimes centerlines are overlapping important values. A break is needed.
2. Select **Break** () from *Annotate* group.
3. Select start point and then end point of the break (Figure 24). A brake is added.

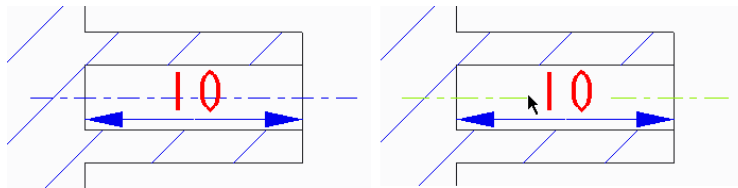


Figure 24: On left: Centerline overlapping with dimension value (10). On right: The end point of the break just selected.

Hatching

Sometimes dimension is needed to place on top of cross-section attaching.

1. Select dimension you want to change (**RMB**→**Properties**).
2. Select Text Style tab and check the option **Break crosshatching**. In *Margin* you can define how widely the hatching will be break.
3. **OK** to accept changes.

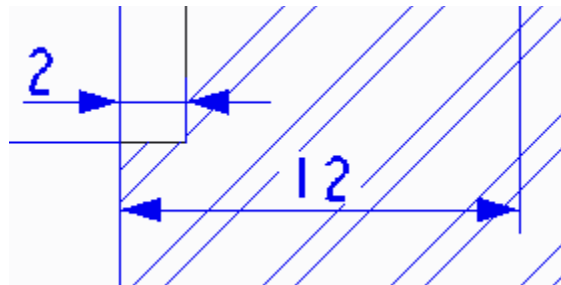




Figure 25: Break cross-section hatching.

Geometric Tolerances

1. If we need a datum (=reference) for our geometric tolerance, we shall create it first. It will be a simple datum plane in this example.

2. From *Annotations* group, select **Model Datum Plane** () .
3. Set the datum plane type as  and enter a name for it (for example A).
4. Click **On Surface...** to simply create the datum aligned to an existing surface of the model (Figure 26). If you need more complex positioning, click **Define**.

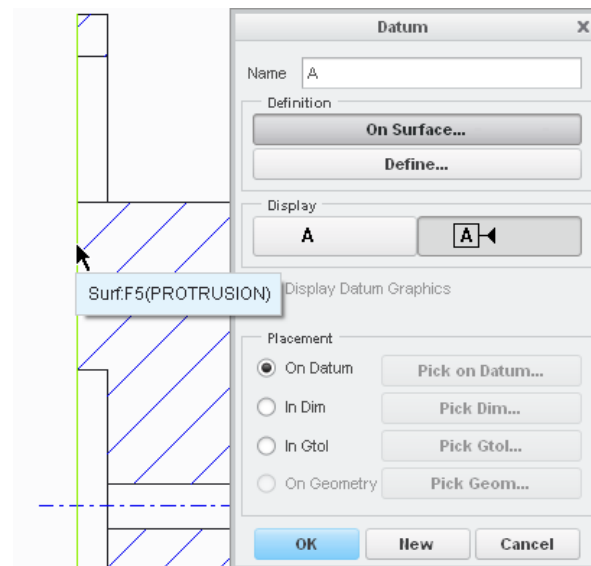




Figure 26: Creating a *Reference Datum* plane to the back of the example part.

5. If reference datum axis is needed, **Model Datum Axis** () tool can be used. You can also transform current planes and axis to a reference object by selecting it, holding **RMB** and selecting **Properties**.
6. Select **Geometric Tolerance** () from *Annotations* group.
7. Select the primary *Reference* (the reference to which we want to point with an arrow) by choosing a proper *Type* and clicking **Select Entity** (Figure 27).

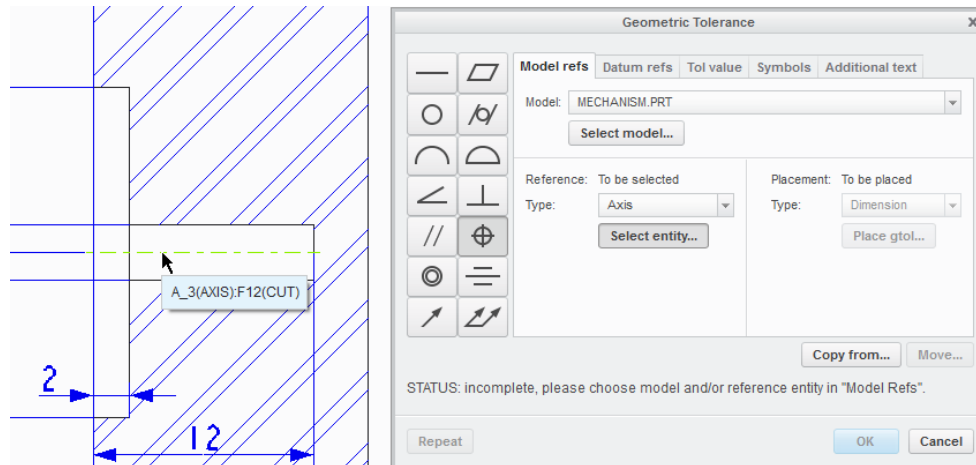


Figure 27: Selecting Axis as a Reference.

8. Select the *Placement* of the geometric tolerance symbol by choosing *With Leader* as *Type*.
9. A new menu will open, waiting for you to select the exact starting location for the arrow (this must be a real entity on the model). Select **Arrow Head**, click on an entity and select **Done**.
10. The bottom dashboard will prompt “*Select placement location*”. Now you will locate the initial symbol location by clicking somewhere on the drawing board.
11. The geometric tolerance symbol should now show up. You can change the tolerance type if needed.
12. Go to the **Datum Refs** -tab. Here you can select previously created datum references as the references of this tolerance definition. We shall select the previously created A-datum plane as the *Primary Basic* reference.
13. If needed, from **Symbols** tab additional symbols can be selected. Tolerance value can be changed from **Tol Value** tab.
14. The tolerance symbols are ready (Figure 28). You can modify geometric tolerance symbol by activating it and selecting **Properties** from the **RMB** menu (or by double-clicking it).

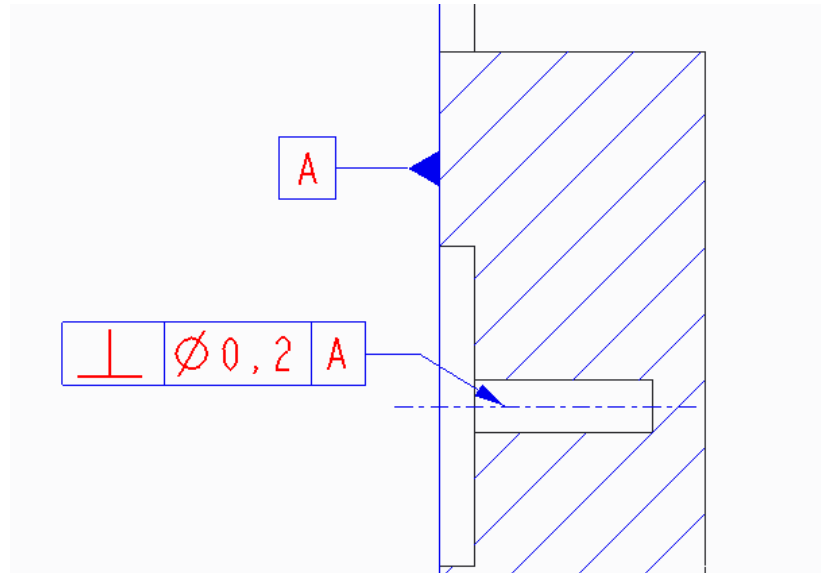


Figure 28: An example geometric tolerance: Pointed axis must be within value of 0.2 perpendicular to surface A.

Geometric Tolerance to Dimension

Sometimes it is useful to add geometric tolerance to existing dimension.

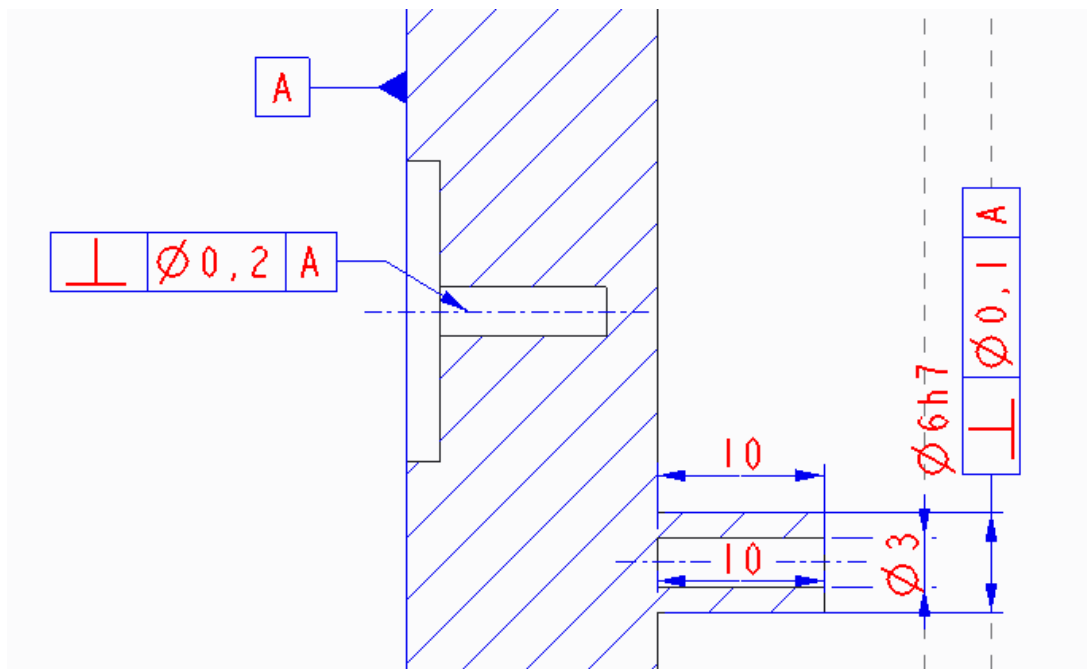


Figure 29: A new geometric tolerance created using *Dimension* as a *Placement*.

Note: To change geometric tolerance attached to a dimension, select gtol “under” the dimension by using **RMB** and **Pick From List** (Figure 30).

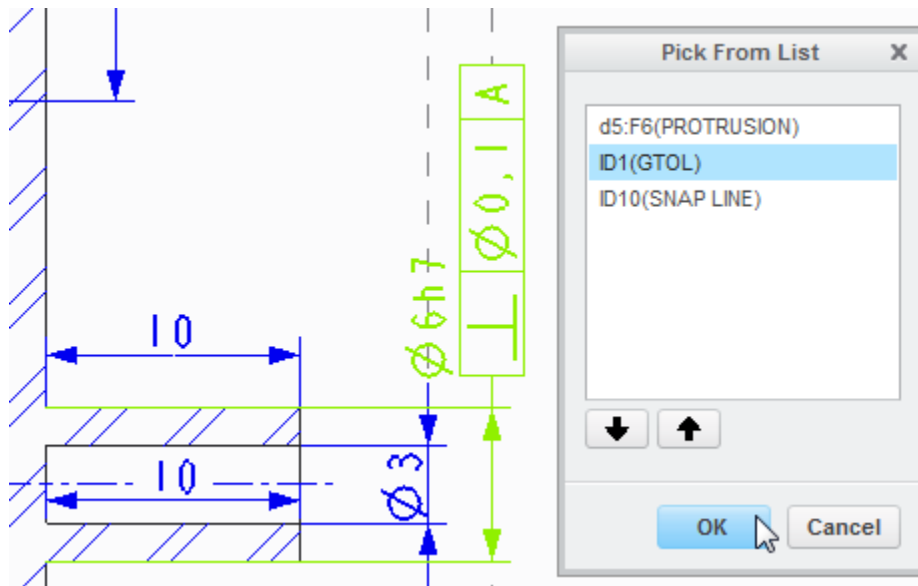


Figure 30: RMB and *Pick From List* selected, selecting GTOL.

Surface Finish Symbols

1. From *Annotations* group, select **Surface Finish** (32/).
2. An *Open* window appears pops up, select **Retrieve**.
3. Browse for an appropriate surface style (for example **machined** -> **standard_iso.sym**) and select **Open**.
4. Back in the menu, select the wanted *Placement Type* (**With Leader** = arrow pointing to a surface, **On Entity** = symbol sitting on top of the surface). If *Leader*, select also arrow type (mostly **Arrow Head**).
5. Set value for the surface roughness in **Variable Text** tab.
6. Select the location and accept with **MMB**.
7. Accept by **OK**. Surface Finish symbol is ready (Figure 31). If needed to modify, just double-click the surface finish symbol.

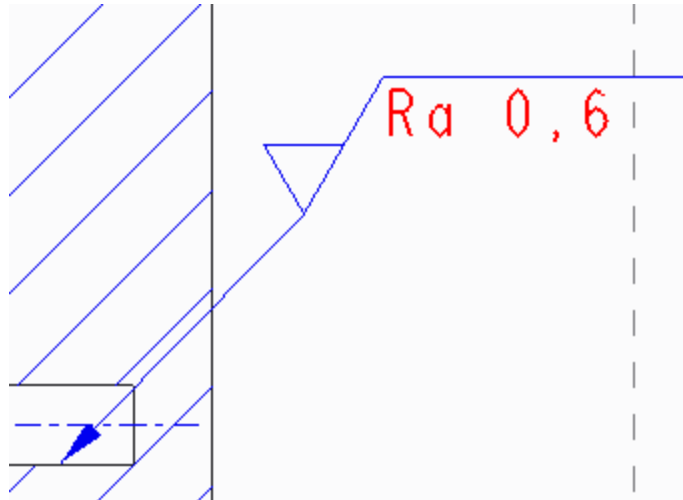



Figure 31: An example *Surface Finish Symbol* (side of the cylinder must have max roughness of 0,6).

Notes

1. To add text to your drawing (for ex. to fill drawing fields), select **Note** () from *Annotation* group.
2. Select location to your note and write text there. Notice the additional dashboard and tools there.

Osa Part	Nimi ID-Code	Nimitys Description	
General to.		Scale	Product
ISO 2768-mK		2.000	Mechanism

Figure 32: Some notes.

Exporting/Printing

To print a drawing, it's recommended to first create a pdf-file.

- 1) Select **File, Save As** and **Export** (Figure 33).

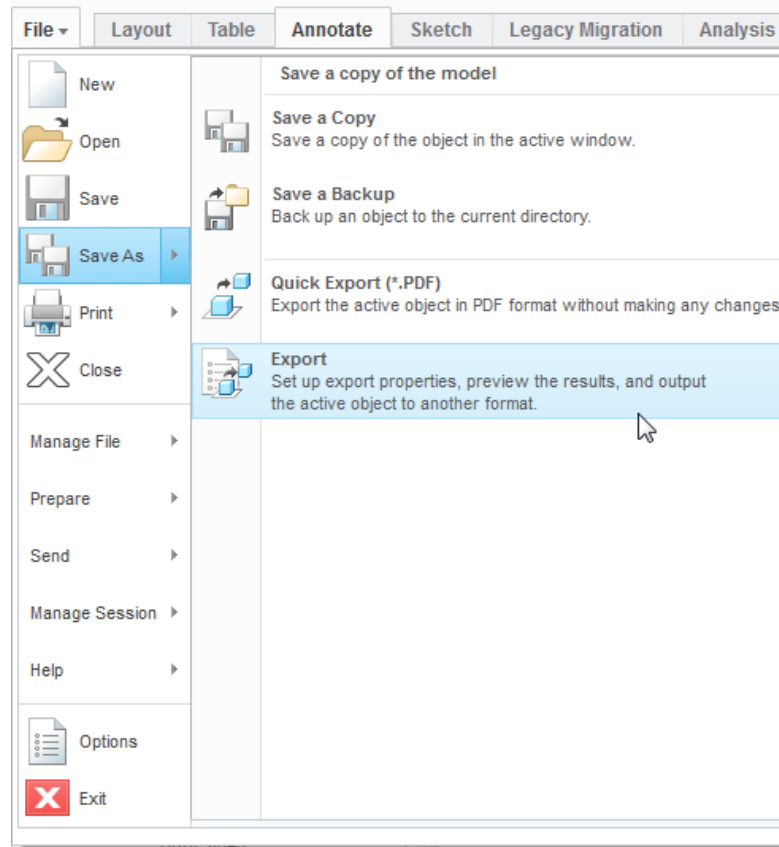


Figure 33: Export.

- 2) Select **PDF** as type in *Configure* and click **Settings** (📄).
- 3) In **General** tab, change *Color* to *Monochrome* (all entities will be black). Other settings should be fine. OK.
- 4) From **Content** tab, select *Stroke All Fonts*. Close settings (**OK**).
- 5) Click **Export** (📄) to create a PDF file.

Note: When exporting, ensure that planes, axis, point and csys displays are off! If not, put them off and press CTRL+R to repaint the drawing area.

Note: When printing a drawing in A3 or A4 size, use setting **Actual Size**. This cuts the extra marks from the drawing and prints in the right size!