

GENERATING OF TECHNICAL DRAWING AND ASSOCIATIVE FUNCTIONALITY IN AUTODESK INVENTOR

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Abstract: Autodesk Inventor enables the generating of a 2D technical drawing based on a 3D geometric model, so that a drawing can be printed on paper at a specific scale. The first view that is created is called the Base View. Other views of the model can be obtained by designing a basic view. Projected views can be orthographic projections or isometric projections. Orthographic projections are aligned with the base view and inherit the size and view settings of the base view. In addition to the shape dimensions, which are used to create shapes, additional reference dimensions can be created. Shape dimensions are used to control geometry, while reference dimensions control existing geometry. Therefore, reference dimensions can be added or deleted, while shape dimensions can be shown or hidden. The associative functionality of Autodesk Inventory allows you to change the design of the model at any level, and the program will automatically reflect the changes at all levels.

Keywords: Technical and technological sciences, technical drawing, basic and projected view, associative functionality, Autodesk Inventor.

1. OPEN THE STANDARD DRAWING SHEET

Start **Autodesk Inventor** in the **Start** menu: All Programs / Autodesk / Autodesk Inventor / Autodesk Inventor Professional.

On the **Get Started** tab, Figure 1, on the **Launch toolbar**, select the **Open** command to activate.



Figure 1. Get Started tab

In the **Open** dialog box, Figure 2, in the **Name** area, select **Nosac vratila.ipt** file. Click the **Open** button in the dialog box to open the selected file.

On the **Get Started** tab, on the **Launch toolbar**, select the **New** command. In the **New File** dialog box, Figure 3, select the **Metric** tab, and then select the icon named **Iso.idw** in the selection area.

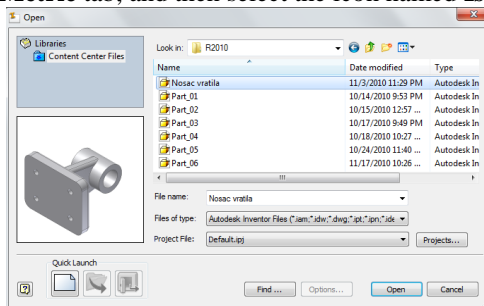


Figure 2. Open dialog box

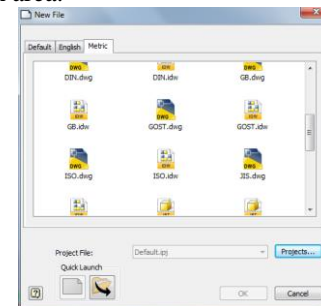


Figure 3. New File dialog box

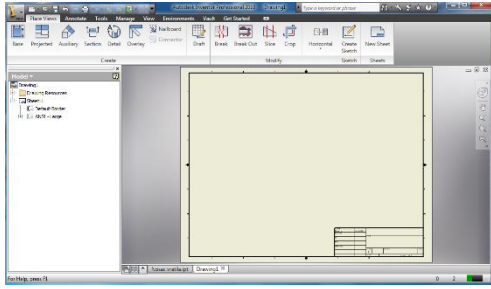


Figure 4. Graphics window

The Autodesk Inventor graphics window with a standard drawing sheet is shown in Figure 4. Switching between a geometric model and a drawing is done by selecting the appropriate minimized graphics window.

The **Drawing1** is drawing name which appears in the browser **Model**, as well as functions: **Drawing resources** and **Sheet: 1**. The first item in the **Model** browser is the drawing name **Drawing1**, which is also the file name.

2. SETTING THE DRAWING STANDARD

On the **Tools** tab, Figure 5, on the **Options** toolbar, select the **Document Settings** tool.

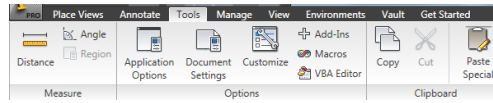


Figure 5 Tools tab

The **Document Settings** dialog box appears, Figure 6.

On the **Standard** tab, select **Default standard (ISO)** from the drop-down list in the **Active Standard** area.

The drawing will be created using ISO sketching standards.

In addition to the selected standard, the following standards are also available: ANSI, BSI, DIN, GB, GOST and JIS.

Click the **OK** button to confirm the standard selection.

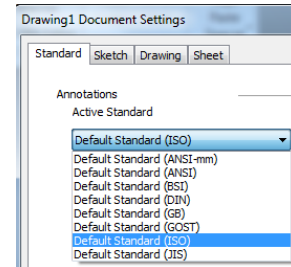


Figure 6. Document Settings dialog box

3. DEFINING THE DRAWING SHAPE FORMAT

Right-click on **Sheet 1**, inside the browser **Model**, to display a submenu with options, Figure 7. Select the **Edit sheet** option to display the **Edit sheet** dialog box, Figure 8. Adjust the size of the sheet of paper and the orientation.

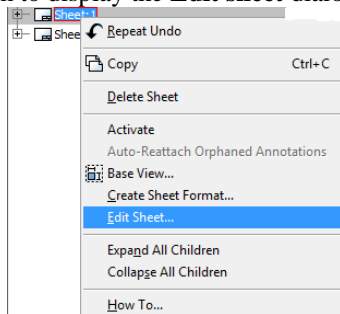


Figure 7. Submenu

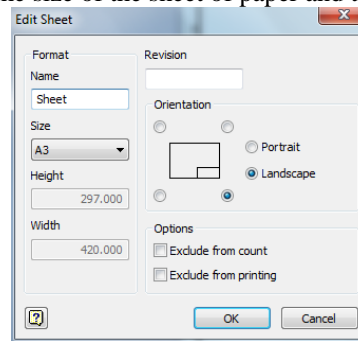


Figure 8. Edit sheet dialog box

4. CREATING A BASIC VIEW

To create a basic view, click on the **Place Views** tab, Figure 9, on the **Create** toolbar, on the **Base** tool.

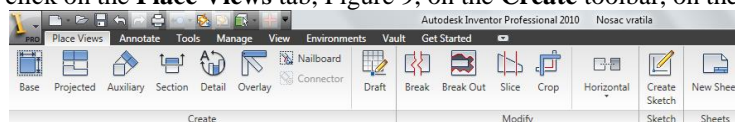


Figure 9. Place Views tab

The **Drawing View** dialog box appears, Figure 10. Select the **Front** option in the **Orientation** area. In the **Style** area, select the **Hidden Line** option.

In the **Scale** area, select the **1:1** option.

Move the **Drawing View** dialog box out of the paper space, and then left-click on the paper space to specify the position of the base view.

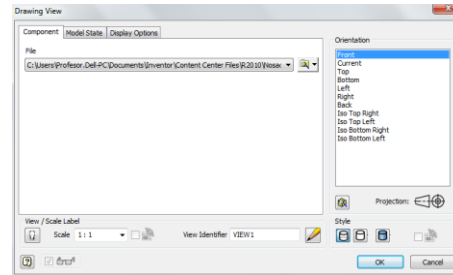


Figure 10. Drawing View dialog box

5. CREATING PROJECTED VIEWS

To create a projected view, click on the **Place Views** tab, on the **Create** toolbar, on the **Projected** tool.

Select the basic view of the model as the view to be projected. Move the mouse cursor below the basic view and left-click on the paper space to set the position of the first projected view.

Move the mouse cursor to the right relative to the base view and left-click on the paper space to set the position of the second projected view. Move the mouse cursor diagonally to the base view and left-click on the paper space to specify the position of the isometric view.

Right-click on the paper space and from the drop-down menu, Figure 11, select the **Create** option to create the projected views, Figure 12.

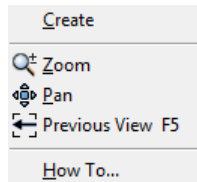


Figure 11. Drop-down menu

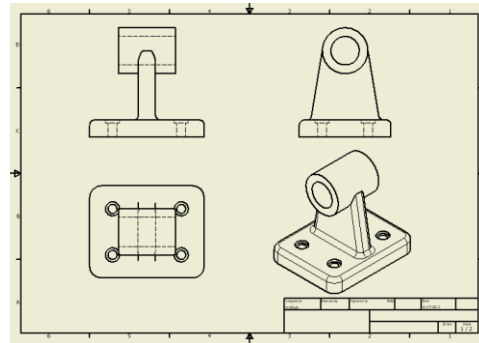


Figure 12. Projected model views

6. DISPLAYING THE DIMENSION ON THE MODEL PROJECTIONS

Place the cursor on the projected view below the basic view until the frame appears around it, then right-click once to open the submenu with options, Figure 13. Select the **Retrieve Dimensions ...** in the submenu.

The **Retrieve Dimensions** dialog box appears, Figure 14, allowing to display the parametric dimensions used to create the model. In the **Select Source** area, check the **Select Parts** option.

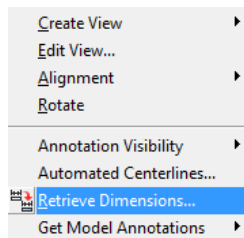


Figure 13. Submenu

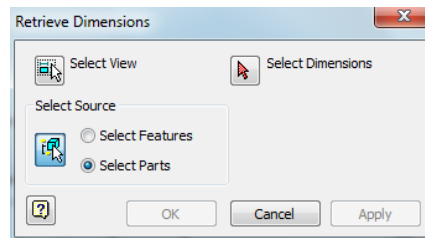


Figure 14. Retrieve Dimensions dialog box

Move the cursor to the projected view, which is below the basic view, and left-click to select a part.

In the **Retrieve Dimensions** dialog box, select the **Select Dimensions** option.

Select the dimensions we want to display with a single left-click on them. Click the **Apply** button in the **Retrieve Dimensions** dialog box. Only the selected dimensions will be displayed, Figure 15, those dimensions that we did not select will simply be removed.

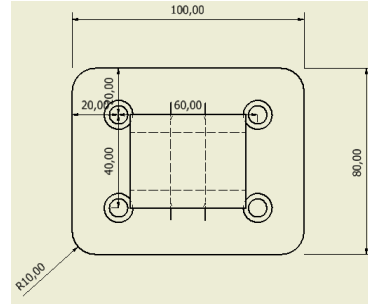


Figure 15. Selected dimensions

7. CREATING ADDITIONAL DIMENSIONS

On the **Annotate** tab, Figure 16, on the **Dimension** toolbar, select the **Dimension** tool.

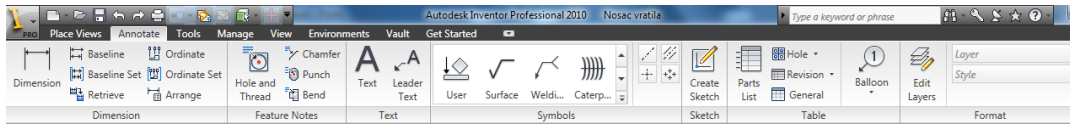


Figure 16. Annotate tab

Select the geometry and then specify the position of the dimension text. The **Edit Dimension** dialog box appears, Figure 17, the **Text** tab is active. Place the cursor in the field in front of the dimension value $\langle\langle\rangle\rangle$, and then from the drop-down list of symbols, Figure 17, select diameter symbol \varnothing .

In the **Edit Dimension** dialog box, select the **Precision and Tolerance** tab, Figure 18, and then set the **Primary Unit** field to **0**.

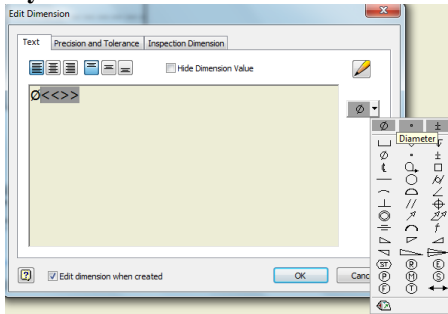


Figure 17. Edit Dimension dialog box, Text tab

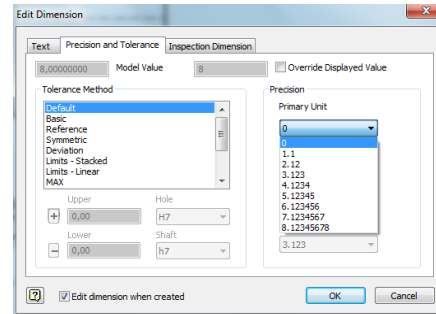


Figure 18. Edit Dimension dialog box, Precision and Tolerance tab

Click the **OK** button in the **Edit Dimension** dialog box to accept the settings and create the $\varnothing 8$ dimension, shown in Figure 19.

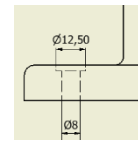


Figure 19. Creating an additional dimension

8. ADDING CENTRAL MARK AND CENTER LINE

On the **Annotate** tab, on the **Symbols** toolbar, select the **Center Mark** tool \oplus .

Left-click on a circle of diameter $\varnothing 40$.

Right-click once, inside the graphics window, to open a submenu with options.

Select the **Done** option in the submenu to end the **Center Mark** command.

Figure 20 shows the added center mark on a circle of diameter $\varnothing 40$.

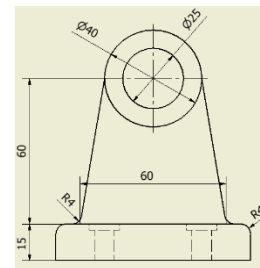


Figure 20. Creating a Center Mark

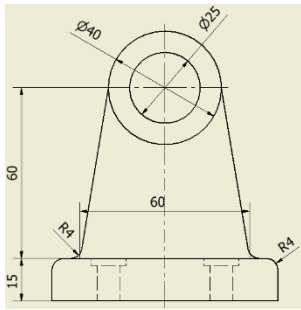




Figure 21. Creating a Centerline

On the **Annotate** tab, on the **Symbols** toolbar, select the **Centerline** tool .

Left-click on the center point of the lower horizontal line on the projected view to the right of the basic view, to specify the first point of the center line. Then left-click on the center of a circle of diameter Ø40, to set the second point of the center line.

Right-click once, inside the graphics window, to open a submenu with options. Select the **Create** option in the submenu to create a centerline.

Autodesk Inventor creates a centerline as shown in Figure 21.

On the **Annotate** tab, on the **Symbols** toolbar, select the **Centerline Bisector** tool .

In the basic view, left-click on the left hidden edge of the Ø8 hole.

Then left-click on the right hidden edge of the Ø8 hole.

Right-click once, inside the graphics window, to open a submenu with options. Select the **Done** option in the submenu to end the **Centerline Bisector** command. Figure 22 shows the created Centerline Bisector of the hole.

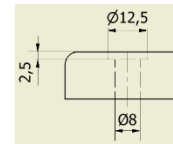



Figure 22. Creating a Centerline Bisector

9. ADD TEXT

On the **Annotate** tab, on the **Text** toolbar, select the **Text** tool .

Select the location inside the header. The **Format Text** dialog box appears, Figure 23.

In the **Format Text** dialog box, in the **Font** area, select **Font** from the drop-down list: **Times New Roman**.

In the **Size** area, enter the font size: **3.00 mm**. Then enter the text as shown in the figure 23. Click the **OK** button to close the **Format Text** dialog box.

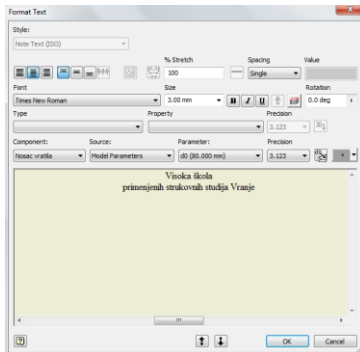


Figure 23. Format Text dialog box

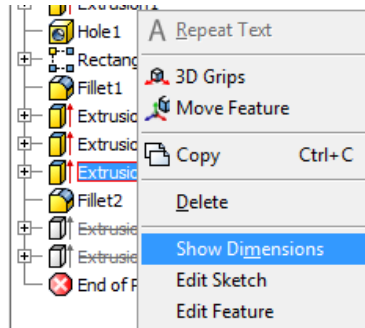



Figure 24 Submenu

10. ASSOCIATIVE FUNCTIONALITY

Double-click with the left mouse button on the Ø25 diameter dimension. The **Edit Dimension** dialog box appears. Changing the model design is automatically reflected in the model drawing. Click on the **Nosac vratila.ipt** window to switch to part modeling mode. In the **Model** browser, right-click on **Extrusion4** to open the submenu with options, Figure 24. Select the **Show Dimensions** option in the drop-down menu. In the **Edit Dimension** dialog box, in the input field, specify **22 mm** for the new diameter dimension, as shown in Figure 25, and then confirm the entry with the icon.

Click the **Local Update**  button on the **Quick access** toolbar. In the graphics window, click **Nosac vratila.idw** to switch to the technical drawing mode, that is, to the paper space. In Figure 26 it can be seen that the dimension of diameter Ø25 is automatically updated, in all views, and the new dimension value is Ø22.

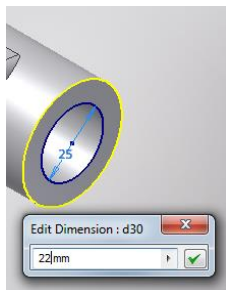


Figure 25. Change dimension value

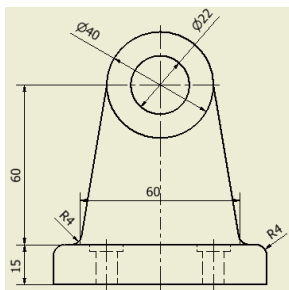


Figure 26. Dimension update

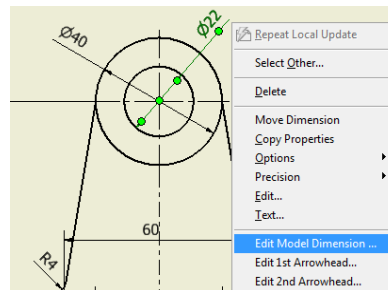



Figure 27. Submenu

Changes to the model's technical drawing will also be automatically updated to the model's design. Click once with the left mouse button on the dimension $\text{Ø}22$ to select it. Then right-click once on the selected dimension to bring up the submenu with options, shown in Figure 27. Select the **Edit Model Dimension** option in the submenu. The **Edit Dimension** dialog box appears. In the input field, set **25mm** for the new dimension of the diameter, and then confirm the entry with the icon.

Click the **Update All Sheets** button  on the **Quick access** toolbar. Click on the **Nosac vratila.ipt** window to switch to part modeling mode. We can make sure that the design of the model is automatically updated.

11. CONCLUSION

This paper presents the procedure for generating a 2D technical drawing based on a 3D geometric model in Autodesk Inventor. A technical drawing was created based on the previously created model. The process consists of the following stages: opening a standard drawing sheet, setting drawing standards, formatting drawing paper, creating (positioning) a basic, projected and isometric view, displaying the shape dimension, creating reference dimension, adding centerline marks and center line, and adding text.

The procedure of creating a basic view is presented, as well as the creation of orthographic and isometric projections, which were obtained by projecting the basic view.

It is pointed out that there are two types of dimensions: shape dimensions and reference dimensions, and the procedure for displaying shape dimensions and creating reference dimensions is explained.

It has been shown that a change in model design is automatically reflected in the model drawing. It has also been shown that changes in the technical drawing of the model are automatically updated to the design of the model, which confirms that the associative functionality allows great flexibility either in changing the design of the model or in changes in the technical drawing.

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