

CREATING GEOMETRIC SHAPES IN AUTODESK INVENTOR USING OF BORN (BASE ORPHAN REFERENCE NODE) TECHNIQUE

Nebojsa J. Dimitrijevic

College of Applied Professional Studies, Vranje, Serbia, nebojsa.j.dimitrijevic@gmail.com

Abstract: The basic concept of BORN (Base Orphan Reference Node) technique is the use of a Cartesian coordinate system as a first shape, which preceded the creation of any geometrical shape. After the establishment a Cartesian coordinate system, are obtained by three mutually normal working plane (XY, YZ and ZX), which can be used as a sketching plane, three working axes (X, Y, and Z) and a working point (the origin). Three working planes are used as a reference for dimensions and geometric constructions. Autodesk Inventor automatically adjust set of reference geometry, which consists of three working-plane, three working axes and the origin when we begin to create a new part. All of the following geometric shapes can be used by the coordinate system and / or reference geometry as sketching plane.

Keywords: Autodesk Inventor, BORN technique, geometric shape

1. STARTING AUTODESK INVENTOR

Autodesk Inventor can be started in several ways. For most Windows systems Autodesk Inventor can run in the Start menu: All Programs / Autodesk / Autodesk Inventor / Autodesk Inventor Professional. It is also possible to access the program using the icon on the Desktop.

After starting the program appears the standard Autodesk Inventor screen drawing, shown on Figure 1, which contains the Application menu, Quick Access toolbar, tabs Get Started, Tools and Vault, with accompanying toolbars, Browser, Area for drawing and Status bar.

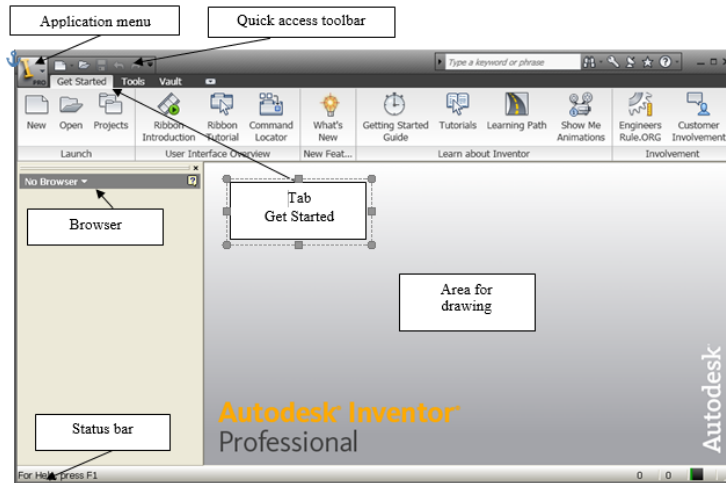


Figure 1. Autodesk Inventor drawing screen

On the tab **Get Started**, on the toolbar **Launch**, select command **New** to activate it. In the dialog box **New File**, select tab **Metric**, and then select the field for selecting the icon with the name **Standard (mm).ipt**. To accept settings click on the button **OK**.

2. SETTING SKETCHING PLANE

On the tab **Tools**, Figure 2, on the tool palette **Options**, select command **Application Options**.

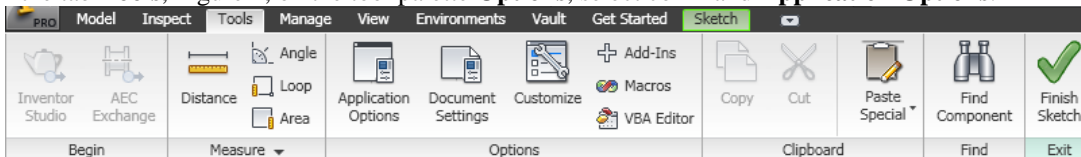


Figure 2. Tools tab

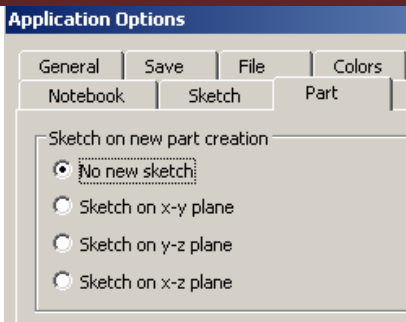


Figure 3. A dialog box Application Options

In the dialog box **Application Options**, Figure 3, select the tab **Part**, in order to modify the default settings of sketching plane. In section **Sketch on new part creation** select the radio button next to option **No new sketch**. To accept the selected settings select the button **Apply**.

On the tab **Get Started**, on the toolbar **Launch**, select command **New**. In the dialog box **New File**, select tab **Metric**, and then, in the field for selecting, select the icon with the name **Standard (mm).ipt**. Confirm the selection by clicking **OK**.

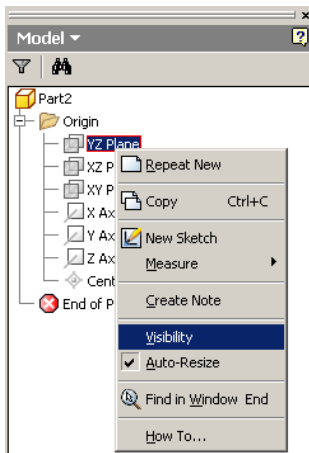


Figure 4. Option Visibility

In the **Model** browser, shown in Figure 4, appear the name of the new part „**Part2**“ and is not set the default sketching plane. Displayed seven feature, three working planes: **YZ Plane**, **XZ Plane** and **XY Plane**, three working axes: **X Axis**, **Y Axis**, and **Z Axis**, and origin **Center Point**.

Move the cursor to the first working plane **YZ Plane** in **Model** browser. To display the options menu click the right mouse button on the **YZ Plane**. In the displayed menu, Figure 4, click the left mouse button on the option **Visibility** to selected working plane was visible in the drawing. Repeat the procedure for activating the visibility for the remaining two working planes, as well as for all three working axes and the origin.

On the tab **Model**, Figure 5, on the palette **Sketch**, single click the left mouse button select the command **Create 2D Sketch**.

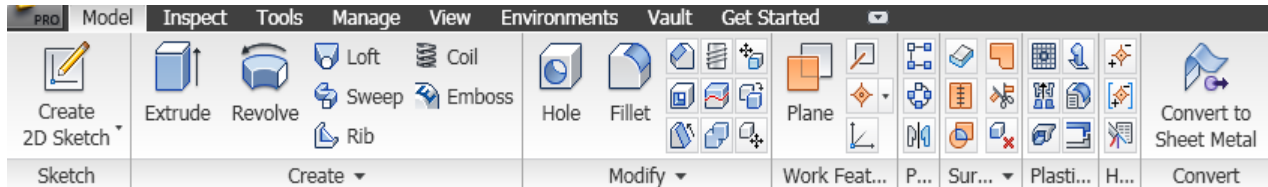


Figure 5. Model tab

In the **Model** browser move the cursor to the work plane **XZ Plane**, it becomes marked in the drawing. In order to select the **XZ plane**, as the drawing plane, click the left mouse button once on it.

On the tab **Sketch**, Figure 6, on the toolbar **Draw**, click once with the left mouse button on command **Project Geometry** to activate it.

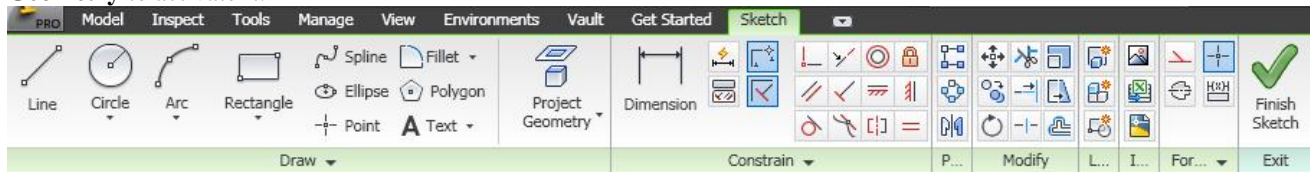


Figure 6. Sketch tab

In the **Model** browser move the cursor to the origin **Center Point**, it becomes marked in the drawing. To project **Center Point** on the sketching plane click the left mouse button once on it.

3. CREATE 2D SKETCH OF THE BASIC SHAPE WITH COMMAND RECTANGLE TWO POINT

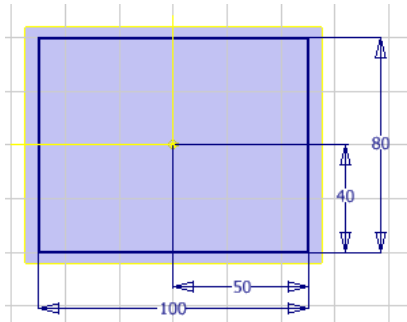


Figure 7. Creating a rectangle

On the tab **Sketch**, on the toolbar **Draw**, click once with the left mouse button on command **Rectangle Two Point** to activate it.

Draw a rectangle of arbitrary size, where the projected origin should be located in the rectangle, see figure 7.

On the tab **Sketch**, on the toolbar **Constrain**, click once with the left mouse button on command **Dimension** to activate it.

Create dimensions, in order to fully constrain the drawing, as shown in Figure 7.

Rounding the vertices of the rectangle with command 2D Fillet

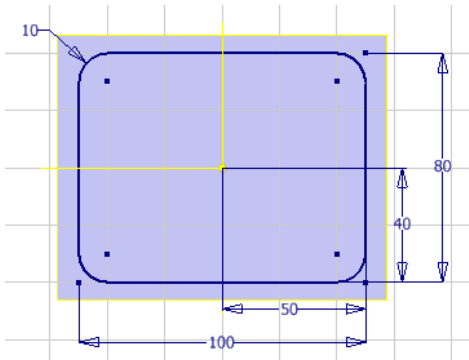


Figure 8. Rounding vertices

On the tab **Sketch**, on the toolbar **Draw**, click once with the left mouse button on command **Fillet** to activate it.

Appears dialog box **2D Fillet**, in which, in the input field, should be defined the numerical value of the radius of rounding.



With the left mouse button select all four vertices of a rectangle. Autodesk Inventor automatically sizing the radius of rounding, see Figure 8.

To complete sketch, with left mouse button select command **Finish Sketch** on the tab **Sketch**.

4. CREATE THE FIRST FORM WITH COMMAND EXTRUDE

In the tab **Model**, on the toolbar **Create**, click once with the left mouse button on command **Extrude** to activate it.

In the dialog box **Extrude**, Figure 9, the card **Shape** is active.

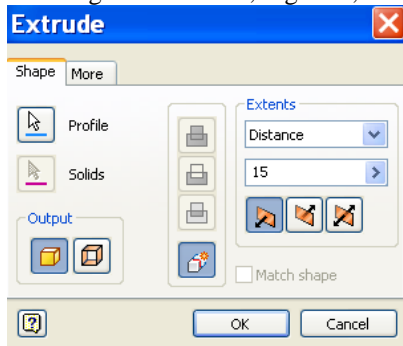


Figure 9. Dialog box Extrude

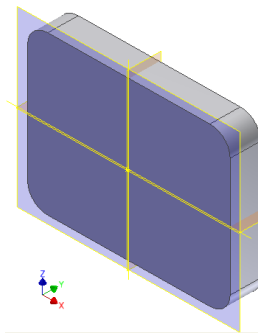


Figure 10. First extrude form

Sketche area is automatically selects as a profile for extrude, because it is the only choice.



Operation  **New solid** is active, which means it will be created a new solid. In the field **Extents**, from the drop-down list, is selected option **Distance**. In the input field enter: **15** for size of the extrude. Select the positive direction of the extrude . To exit from dialog box **Extrude**, select **OK**.

Figure 10 shows a geometric shape obtained by using command **Extrude**.

5. CREATING A HOLE

On the tab **Model**, on the toolbar **Modify**, click once with the left mouse button on command **Hole** to activate it.

In the dialog box **Hole**, shown in Figure 11, in the field **Placement** select, from the drop-down list, option: **Linear**.

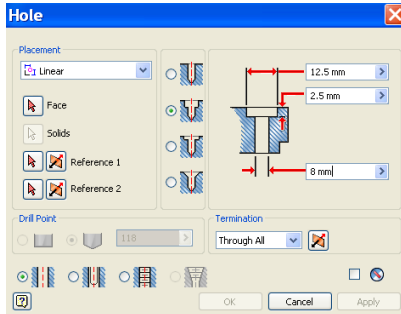


Figure 11. The dialog box Hole

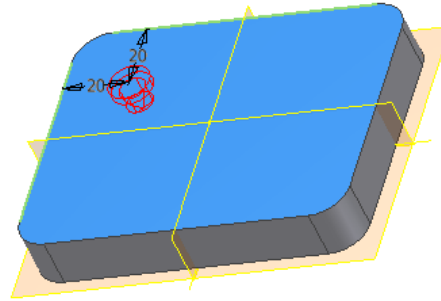







Figure 12. Creating a hole


In the field **Placement**, button  **Face** is selected. Click once with the left mouse button on the upper horizontal surface of the first geometric form, in order to select it.

In the dialog box **Hole**, select button  **Reference 1**. Click the left mouse button once on the long edge on the upper surface of the first horizontal geometrical shapes to select it. In the dialog box **Hole**, select button  **Reference 2**. Click the left mouse button once on the short edge on the upper surface of the first horizontal geometrical shapes to select it, see Figure 12. In the field for the tip hole select radio button in front of the option

Simple Hole , which selects the standard tip hole. In the field **Termination**, from the drop-down list, choose

the option: **Through All**. Select a radio button for option **Counterbored** , which selects the type of hole. Define dimensions for this type of hole. To accept settings and execute the command select the button **Apply**. To exit from the dialog box **Hole** choose the button **Done**.

6. DUPLICATE HOLE WITH COMMAND RECTANGULAR PATTERN

On the tab **Model**, on the toolbar **Pattern**, click once with the left mouse button on the command **Rectangular Pattern** to activate it. In the dialog box **Rectangular Pattern**, shown in Figure 13, the button  **Features** is selected.

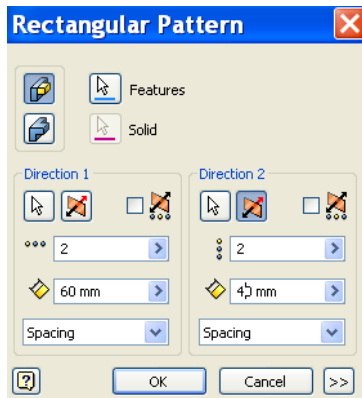


Figure 13. The dialog box Rectangular Pattern

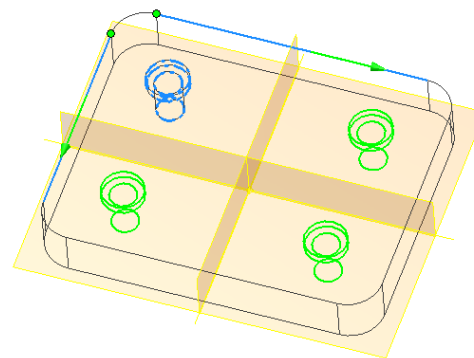










Figure 14. Creating duplicate hole

Click once with the left mouse button on the geometric shape **Hole1** to select it. In the field **Direction 1**, select the icon . Click the left mouse button once on the long edge to the upper horizontal surface of the first geometric shape **Extrusion1**. In the entry field behind the icons **Count**  specify the number of times to copy the selected geometric shapes in rows, enter: 2. In the entry field behind the icons **Length**  specify how much would be the distance between the duplicate geometric shapes in rows, enter: 60. In the field **Direction 2** select the icon . Click the left mouse button once on the short edge on the upper horizontal surface of the first geometric shape **Extrusion1**. In the entry field behind the icons **Count**  specify the number of times to copy the selected geometric shape of the columns, enter: 2. In the entry field behind the icons **Length**  specify how much would be the distance between the duplicate geometric shapes in columns, enter: 40. To accept settings and perform command **Rectangular Pattern** select button **OK**. On Figure 14 are shown the geometrical shapes obtained by duplicating the selected shape.

7. ROUNDING THE EDGES WITH COMMAND FILLET

On the tab **Model**, on the toolbar **Modify**, click once with the left mouse button on command **Fillet** to activate it. In the dialog box **Fillet**, shown in Figure 15, icon **Edge Fillet**  is active.

On the card **Constant**  **Constant**, in the entry field **Radius** enter the radius of rounding: 4mm. In the field **Select mode**, select the radio button in front of the option **Edge**. Click once with the left mouse button on all the edges on the upper horizontal surface of the first geometric shape **Extrusion 1**. In the field **Edges** is shown the number of selected edges. To accept settings and execute the command select the button **Apply**.

To exit from the dialog box **Fillet**, select button **Cancel**. In Figure 16 is shown a appearance of geometrical form which is prepared using the commands **Fillet**.

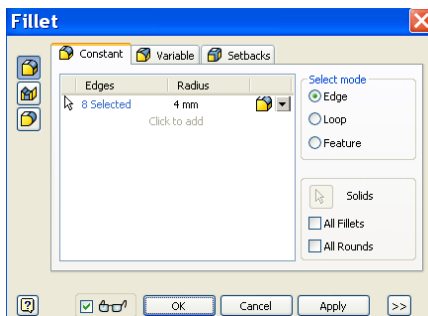
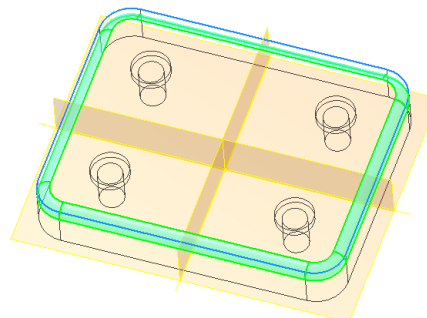


Figure 15. The dialog box Fillet



Slika16. Creating a fillet

8. SELECTION OF WORKING PLANE YZ AS DRAWING PLANE

On the tab **Model**, on the toolbar **Sketch**, a single click the left mouse button select the command **Create 2D Sketch**.

In the **Model** browser, move the cursor to the working plane **YZ Plane**, it becomes marked in the drawing. In order to select the **YZ Plane** as drawing plane one click the left mouse button on it.

Project the origin and the Y-axis to the active drawing plane

On the tab **Sketch**, on the toolbar **Draw**, click once with the left mouse button on the command **Project Geometry** to activate it. In the **Model** browser move the cursor to **Center Point**. To be projected **Center Point** to the sketching plane click the left mouse button once on it. Then move the cursor to the **Y Axis**. In order to be projected **Y Axis** to the sketching plane click the left mouse button once on it.

9. CREATING OF OPENED REGION WITH COMMAND LINE

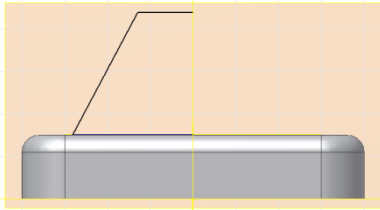


Figure 17. Creating an open region

On the tab **Sketch**, on the toolbar **Draw**, click once with the left mouse button on the command **Line** to activate it.

Complete sketch with command **Line** so as to create open region shown in Figure 17.

10. CREATION OF CLOSED REGION WITH COMMAND MIRROR

On the tab **Sketch**, on the toolbar **Pattern**, click once with the left mouse button on the command **Mirror** to activate it.

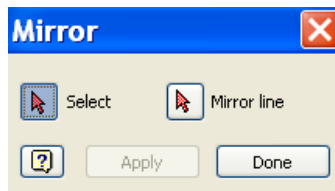




Figure 18. The dialog box Mirror

In the dialog box **Mirror**, shown in Figure 18, the icon  is active. Select the open region created with the command **Line**, Figure 19.

In the dialog box **Mirror**, select the icon . Select the **Y axis**, which had previously been projected to the active sketch plane, Figure 19.

To accept settings and execute the command select the button **Apply**. To exit from the dialog box **Mirror** choose the button **Done**. In Figure 20 is shown closed profile created with commands **Line** and **Mirror**.

On the tab **Sketch**, on the toolbar **Constrain**, click once with the left mouse button on the command **Dimension** to activate it. Create dimensions on the closed profile as shown in Figure 20.

To complete the sketch select left-click on command **Finish Sketch**.

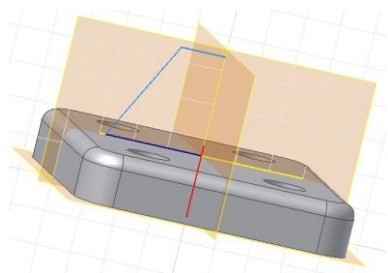


Figure 19. Selecting the region and the axis

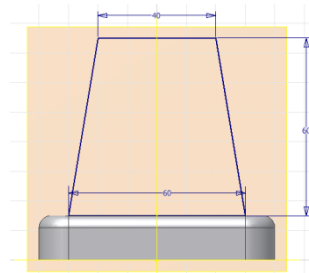


Figure 20 Closed region with dimensions

11. CREATING THE SECOND EXTRUDE FORM

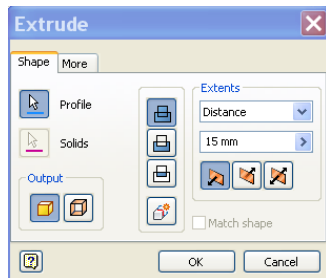





Figure 21. The dialog box Extrude

On the tab **Model**, on the toolbar **Create**, click once with the left mouse button on the command **Extrude** to activate it.

In the dialog box **Extrude**, shown in Figure 21, the card **Shape** is active, while the button  is active.

Select a shape as shown on Figure 22.

On the palette of operation, button  **Join** is active, so that it will get shape to be connected with an existing geometric shape. In the field **Extents**, is selected, from the drop-down list, options **Distance**. In the input field enter: 15 for size of the extrude.

Select the icon **Mid plane** , extrude of equal value on both sides of the selected profile. To exit the dialog box **Extrude**, select **OK**. In Figure 23 shows the geometry shape after command **Extrude**.

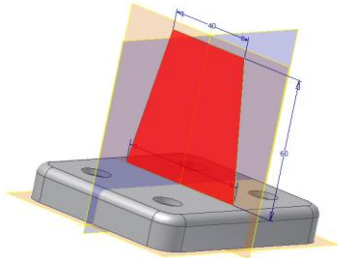


Figure 22. Selecting a profile for extrude

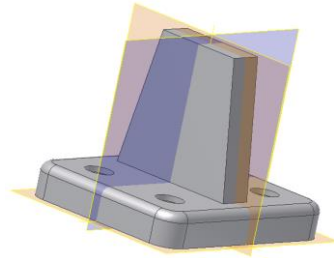


Figure 23. Second extrude form

12. SELECTION OF YZ PLANE AS PLANE OF THIRD EXTRUDE FORM

On the tab **Model**, on the toolbar **Sketch**, a single click the left mouse button select the command **Create 2D Sketch**. In the **Model** browser move the cursor to the **YZ Plane**. In order to select the **YZ Plane** as drawing plane one click the left mouse button on it.

On the tab **Sketch**, on the toolbar **Draw**, click once with the left mouse button on the command **Project Geometry** to activate it. In the **Model** browser move the cursor to **Center Point**. To be projected **Center Point** on the sketching plane click the left mouse button once on it. Then move the cursor to the **Y Axis**. In order to be projected **Y Axis** on the sketching plane click the left mouse button once on it.

13. CREATING THE CIRCULAR SHAPE WITH COMMAND CIRCLE

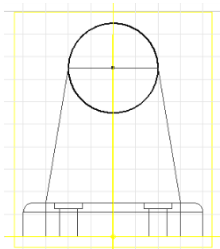



Figure 24. The circular shape

On the tab **Sketch**, on the toolbar **Draw**, click once with the left mouse button on the command **Circle Center Point** to activate it. Click once with the left mouse button on the middle of the upper horizontal line of the second extrude shape to define the center of the circle, Figure 24. Click the left mouse button once at the start point of the upper horizontal line of the second extrude shape to define the point on the circle, Figure 24.

To finish sketch select left-click on command **Finish Sketch**.

14. CREATING THE THIRD EXTRUDE FORM

On the tab **Model**, on the toolbar **Create**, click once with the left mouse button on the command **Extrude** to activate it.

In the dialog box **Extrude**, shown in Figure 25, the card **Shape** is active, while the button  **Profile** is pressed. Select a geometric shape as shown in Figure 26.

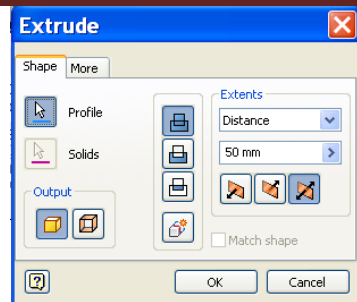




Figure 25. The dialog box Extrude

In the field of operation, button  **Join** is active, so you will get extrude shape to be connected to the existing geometric shape.

In the field **Extents**, is selected, from the drop-down list, option **Distance**.

In the input field enter: 50 for size of the extrude.

Select the icon , extrude of equal value on both sides of the selected profile.

To exit from the dialog box **Extrude**, select the button **OK**.
Obtained geometric shape is shown in Figure 27.

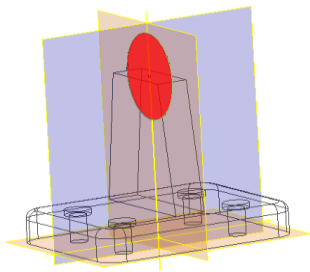
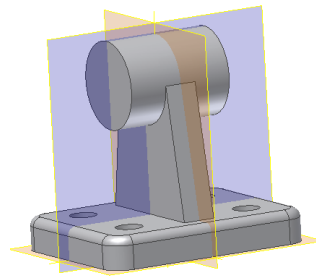


Figure 26. Selecting a profile for extrude



Slika 27. Third extrude form

15. CONCLUSION

This paper presents BORN (Base Orphan Reference Node) technique for creating geometric shapes in the software package Autodesk Inventor.

It has been shown that the basic geometric shape is not the only option for selecting the sketching plane of geometric shapes.

The three working plane (XY, YZ and ZX plane) can be used as sketching plane.

This approach offers more options for creating parametric models, as well as greater flexibility in the modification of parts and changes of design.

REFERENCES

- [1] N. J. Dimitrijević, Z. V. Janjić, N. V. Janić, “Zbirka zadataka iz CAD-a (Autodesk Inventor)”, Visoka škola primenjenih strukovnih studija, Vranje, 2012.
- [2] N. Dimitrijević, Z. Janjić, N. Janjić, D. Nikolić, “Modelovanje simetričnih geometrijskih oblika i konstrukciona geometrija u Autodesk Inventoru”, XXXV konferencija “Poslovnih komunikacija i proizvodnog inženjerstva“, Jun 01–02, Vrnjačka Banja, Srbija, str. 49–57, 2012.
- [3] N. Dimitrijević, N. Janjić, Z. Janjić, D. Nikolić, “Kreiranje šablona crteža i radioničkog crteža, definisanje okvira i zaglavlja, asocijativna funkcionalnost u Autodesk Inventoru”, XXXV konferencija “Poslovnih komunikacija i proizvodnog inženjerstva“, Jun 01–02, Vrnjačka Banja, Srbija, str. 39–48, 2012.
- [4] N. Dimitrijević, Janjić N., Janjić Z., D. Nikolić, “Modelovanje sklopa, primena ograničenja na delove sklopa, promena dizajna komponenata sklopa i eksplozivni prikaz sklopa u Autodesk Inventoru”, Tehnička dijagnostika 1–2, str. 94–99, 2012.
- [5] N. Dimitrijević, D. Nikolić, N. Janjić, “Generisanje radioničkog crteža i asocijativna funkcionalnost u Autodesk Inventoru”, Menadžment znanja 3–4, str. 10–17, 2011.
- [6] Shih R., “Autodesk Inventor – Parametrijsko modeliranje”, Svetlost Čačak, 2007.