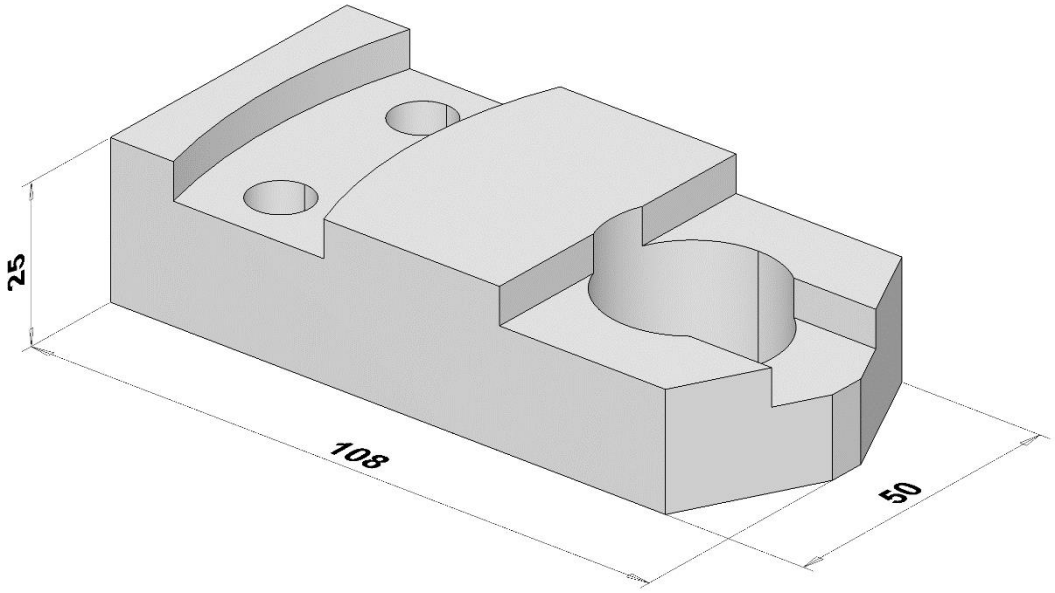
COURSE UNIT

3D



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INTRODUCTION

How to use this course unit

The course unit is a supplement to our manual, in which all functions are described in detail and enables you to learn ELITECAD step by step. Every mouse click and every input is described in detail, so you receive a teaching aid with which you can independently get to know ELITECAD in its basics. In each chapter you have explanations, information and a workshop in which the functions are practiced. So that the workshops can be worked through without any problems, please read the description beforehand and then start working. As the workshops build on one another, please do not change the order of the chapters.
At the end of this course unit, there are plans for another example that you create yourself in order to deepen your skills with ELITECAD. All examples are stored as ELITECAD files on the ELITECAD installation medium. This gives you the opportunity to compare the example you have worked out with our target and thus determine any differences.
We wish you every success in learning ELITECAD.

Requirements for this course unit

Successfully completed COURSE UNIT 2D

START PROGRAM

WORKSHOP

1. Launch the programme.
ELITECAD ME15 3D icon is located on the Desktop.

Double-click on the icon.

2. ELITECAD launches and initializes the different components.

Give the program time to fully start and only then start working.

WORKSHOP END

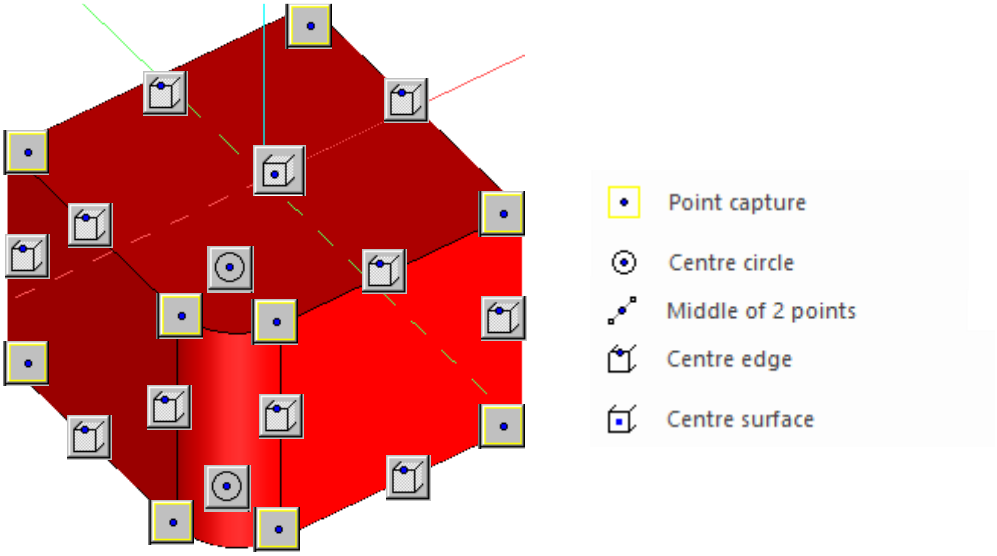
General

Important short cuts

Ctrl +B	Change background colour
Ctrl +D	Solid on/off
Ctrl +E	Elements on/off
Ctrl +K	Edges on/off
Ctrl +T	Transparency on/off
Ctrl +Pos1	Starting position and format
Ctrl +Space	Starting position current zoom

For further functions see manual or online help!

The following snap functions are available in 3D (3D model)



TIP

ATTENTION!!
In 3D, first in the shading, then in the wireframe and finally the 2D lines will be snap on.

ROTATE, ZOOM AND MOVE THE SCREEN

Rotate the screen

If you just keep the middle mouse button pressed and move the mouse, you will turn into 3D.

Zoom screen

Simultaneously hold down [⇧**Shift**] + middle mouse button and move the mouse up and down. If you have a mouse with a scroll wheel, this wheel can be used to zoom.

Move the screen

Simultaneously hold down [**Ctrl**] + middle mouse button and move the mouse or, while holding down the [**Ctrl**] key, use the arrow keys on the keyboard [←→↑↓].

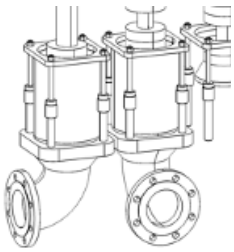
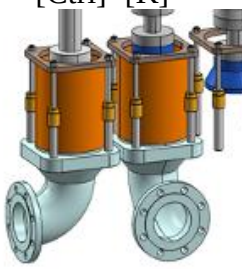
Screen starting position

 oder [**Ctrl**]+[**Space**]
Select this function (Reset camera) to return to the starting position from a rotated screen. Starting position means "view normal to the work plane" and "sheet format centered on screen".

TIP

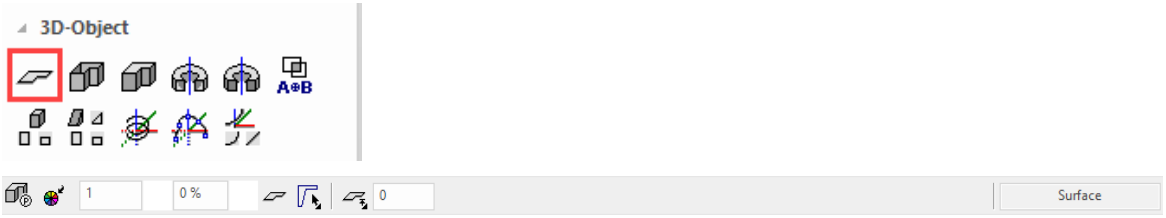
If you want to keep the current working plane and only zoom in on the format, you can do this using the ZOOM ON FORMAT function.

Representations

Wireframe model	Pixel-hiddenline	Solid model	Edges on
[Ctrl]+[D]	[Ctrl]+[H]	[Ctrl]+[D]	[Ctrl]+[K]
			

DEFINE PLANE

As soon as you select the DEFINE PLANE function, the properties bar appears.



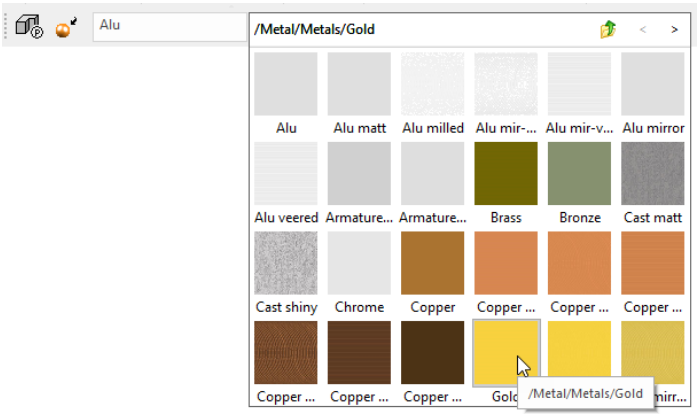
When defining planes, the contour must be closed.

3D-Colour, transparency

The colour and transparency for the 3D display can be selected here.

Material

With the help of this button it is possible to switch between the colour and material mode. If you switch to material mode, a 3D texture can be assigned to the plane.
With the help of the SELECT MATERIAL button, the selection opens and you can choose between different types of material.



TIPP

You can also assign a material to the 3D objects using the material management. The assignment is made very simply by drag and drop.

Type of 3D definition

There are 3 options for creating a plane:
Either you draw the contour immediately after starting the function or you have already drawn a polygon before calling the function and only "declare" it to be a box or it has already been selected.
The function calls itself again and again in order to be able to draw several planes one after the other. When you have finished creating the boxes, you can cancel the function.

Via contour input

If you create a plane with the function, you only have to make sure that the result is a closed polygon. After you come back to the starting point of drawing, the function defines the plane.

Via select contour

If you want to define an existing polygon for the plane, you should make sure that the outline is drawn closed. The plane can be generated by tapping the outline polygon.

About selection

If “VIA SELECT CONTOUR” is activated and a selection has already been made on the screen when selecting the function, a query appears asking whether this should be used.

TIPP

If polygons are inside one another and they are drawn from the outside inwards, tapped or they are in the selection, recesses result.

Type of input

Limiting surface over height

The height can be entered as a numerical value (in the currently set unit) in the input line or in the property bar.

Limiting surface via entry of point

With the help of the SNAP FUNCTIONS, the height can be defined directly by selecting a height point in the 3D model.

Limiting surface over 3 points

The area is defined using 3 points. The height is requested for each point. The definition of the three Z-coordinate points for the inclined plane can be done anywhere on the screen and is not tied to the elements of a pattern!

TIPP

Here, too, it is possible to select points directly in the 3D model. The clicked height is suggested above the input line and can be simply confirmed with **[Enter]**.

Limiting surface over plane

The position is defined by tapping an existing plane.

Change contour

This can be used to subsequently modify the contour of the plane. After you have clicked on the function, the 2D of the plane is automatically displayed. Now change the contour with the normal 2D drawing functions according to your wishes. Make sure that the new contour results in a closed polygon again and terminate the function with the "Contour finished" button, which is visible at the top left.

TIPP

If you click on a handle or gripper of the contour after selecting the plane, manipulation functions are also available directly in a toolbar or by pressing [TAB] on the cursor.

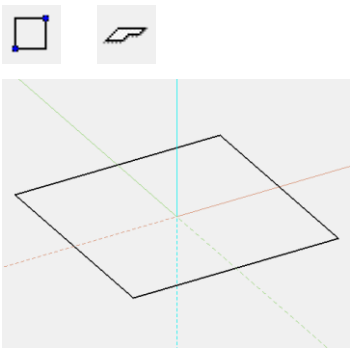
Modify resolution

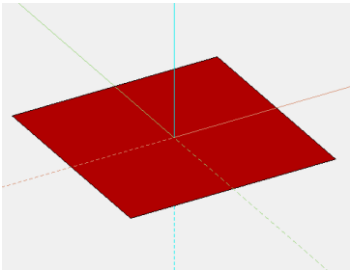
This function is used to set the parameters for the 3D resolution. You can switch between a uniform and a curvature-dependent resolution. The analytical data model with nurbs is an exact description of curves and surfaces. For the graphic output, an approximation must be made using facets, which can, however, be as fine as desired. The higher the fineness, the better the graphics, but the longer the computing time.

WORKSHOP

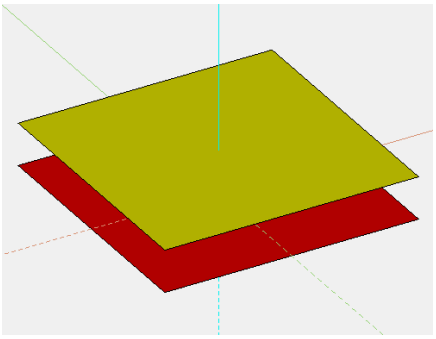
Define a plane

Draw a rectangle, then click on DEFINE PLANE and set the *type of 3D definition* to *select contour*. Set the height to **0**, the colour to **2** and tap the rectangle.





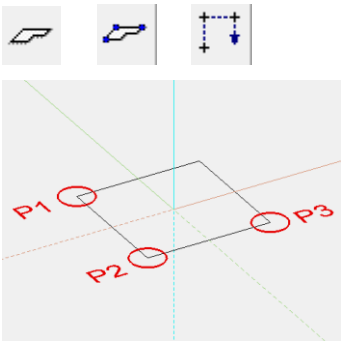
Several definitions can also be stored for the same contour. Now set the height to **50**, the colour to **3** and tap the rectangle again.

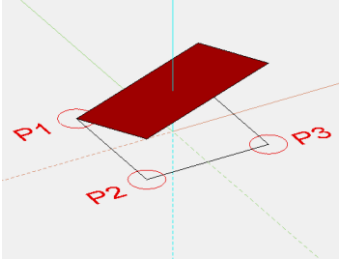


TIP

The individual parameters of the planes can be modified with the part mode.

To create an inclined surface, you must set the *type of input* to *3 points*. After drawing the contour, tap point 1 and enter a height of **0**. Then click on points 2 and 3 and enter the heights **100** and **200**, respectively. This results in an inclined plane.

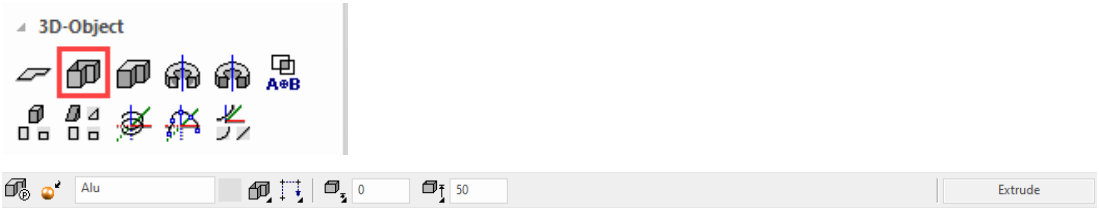




WORKSHOP END

DEFINE EXTRUDE

As soon as you select the DEFINE EXTRUDE function, the properties bar appears.



In the case of an extrude definition, the contour can be closed - but does NOT have to be.

Type of 3D definition

There are 3 options for creating an extrude:
Either you draw the contour immediately after starting the function or you have already drawn a polygon before calling the function and only "declare" it to be a box or it has already been selected.
The function calls itself again and again in order to be able to draw several extrudes one after the other. When you have finished creating the boxes, you can cancel the function.

Via contour input

If you create a plane with the function, you only have to make sure that the result is a closed polygon. After you come back to the starting point of drawing, the function defines the plane.

Via select contour

If you want to define an existing polygon for an extrude, you should make sure that the outline is drawn closed. The plane can be generated by tapping the outline polygon.

About preselection

If"VIA SELECT CONTOUR" is activated and a selection has already been made on the screen when selecting the function, a query appears asking whether this should be used.

Type of input

Lower and upper limiting surfaces above height

The height can be entered as a numerical value (in the currently set unit) in the input line or in the property bar.

Lower and upper limiting surfaces via entry of points

With the help of the SNAP FUNCTIONS, the height can be defined directly by selecting a height point in the 3D model.

Lower and upper limiting surfaces over 3 points

The area is defined using 3 points. The height is requested for each point. The definition of the three Z-coordinate points for the inclined extrude can be done anywhere on the screen and is not tied to the elements of a pattern!

TIPP

Here, too, it is possible to select points directly in the 3D model. The clicked height is suggested above the input line and can be simply confirmed with **[Enter]**.

Lower and upper limiting surfaces over plane

The position is defined by tapping an existing plane.

Change contour

This can be used to subsequently modify the contour of the plane. After you have clicked on the function, the 2D of the extr 8s7ude is automatically displayed. Now change the contour with the normal 2D drawing functions according to your wishes. Make sure that the new contour results in a closed polygon again and terminate the function with the "Contour finished" button, which is visible at the top left.

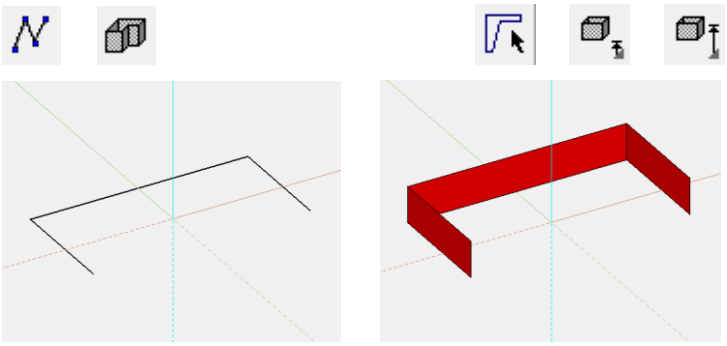
TIPP

If you click on a handle or gripper of the contour after selecting the plane, manipulation functions are also available directly in a toolbar or by pressing [TAB] on the cursor.

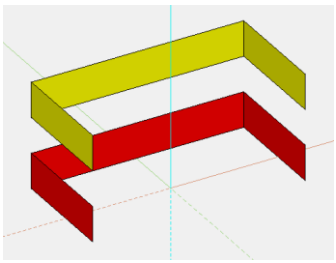
WORKSHOP

Define extrude

Draw a contour, then click on DEFINE EXTRUDE and set the *type of 3D definition* to *Select contour*. Set the lower height to **0**, the upper height to **100**, the colour to **2** and tap the contour.



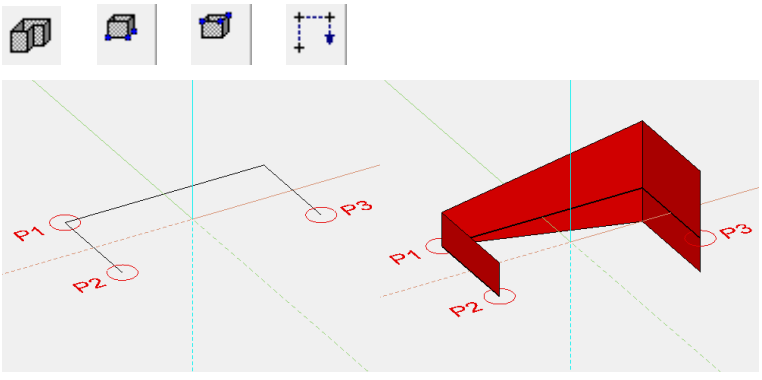
Several definitions can also be stored for the same contour. Now set the lower height to **200**, the upper height to **300**, the colour to **3** and touch the contour again.



TIPP

The individual parameters of the extrudes can be changed with the part mode.

To create an incline extrude you have to set the *input type* to *3 points*. After drawing the contour, tap point 1 and enter a height of **0**. Then click on points 2 and 3 and enter the heights **0** or **-100**. Proceed in exactly the same way for the upper height and enter the heights as **100** and **200**. This results are inclined extrude.



WORKSHOP END

DEFINE BOX

As soon as you select the DEFINE BOX function, the properties bar appears.

In the case of box definitions, the contour must be closed.

Type of 3D definition

There are 3 options for creating a box:
Either you draw the contour immediately after starting the function or you have already drawn a polygon before calling the function and only "declare" it to be a box or it has already been selected.
The function calls itself again and again in order to be able to draw several boxes one after the other. When you have finished creating the boxes, you can cancel the function.

Via contour input

If you create a plane with the function, you only have to make sure that the result is a closed polygon. After you come back to the starting point of drawing, the function defines the plane.

Via select contour

If you want to define an existing polygon for the box, you should make sure that the outline is drawn closed. The plane can be generated by tapping the outline polygon.

About selection

If”VIA SELECT CONTOUR” is activated and a selection has already been made on the screen when selecting the function, a query appears asking whether this should be used.

Type of input

Lower and upper limiting surfaces above height

The height can be entered as a numerical value (in the currently set unit) in the input line or in the property bar.

Lower and upper limiting surfaces via entry of points

With the help of the SNAP FUNCTIONS, the height can be defined directly by selecting a height point in the 3D model.

Lower and upper limiting surfaces over 3 points

The area is defined using 3 points. The height is requested for each point. The definition of the three Z-coordinate points for the inclined extrude can be done anywhere on the screen and is not tied to the elements of a pattern!

TIP

Here, too, it is possible to select points directly in the 3D model. The clicked height is suggested above the input line and can be simply confirmed with **[Enter]**.

Lower and upper limiting surfaces over plane

The position is defined by tapping an existing plane.

Change contour

This can be used to subsequently modify the contour of the box. After you have clicked on the function, the 2D of the box is automatically displayed. Now change the contour with the normal 2D drawing functions according to your wishes. Make sure that the new contour results in a closed polygon again and terminate the function with the "Contour finished" button, which is visible at the top left.

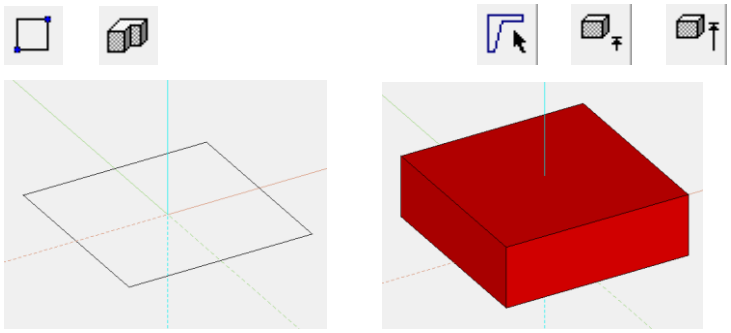
TIP

If you click on a handle or gripper of the contour after selecting the plane, manipulation functions are also available directly in a toolbar or by pressing [TAB] on the cursor.

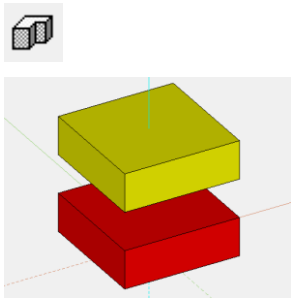
WORKSHOP

Define box

Draw a rectangle, then click on DEFINE BOX and set the *type of 3D definition* to *Select contour*. Set the lower height to **0**, the upper height to **100**, the colour to **2** and tap the contour.



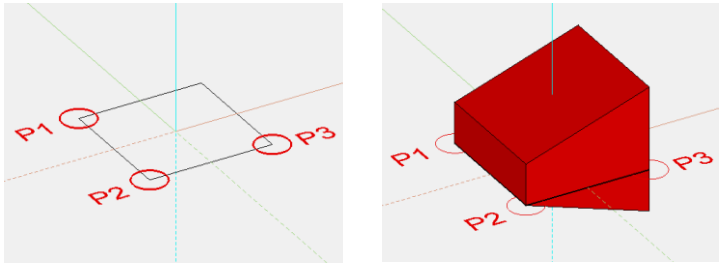
Several definitions can also be stored for the same contour. Now set the lower height to **200**, the upper height to **300**, the colour to **3** and tap the rectangle again.



TIP

The individual parameters of the boxes can be changed with the part mode.
In addition, when the box is activated, the 2D, 3D or all handles and grippers can be switched on using the TAB key.

In order to create an inclined box, you have to set the *type of input* to *3 points*. After drawing the rectangle, click point 1 and enter a height of **0**. Then click on points 2 and 3 and enter the heights **0** or **-100**. Proceed in exactly the same way for the upper height and enter the heights as **100** and **200**. This results is a inclined box.



WORKSHOP END

DEFINE ROTATION EXTRUDE

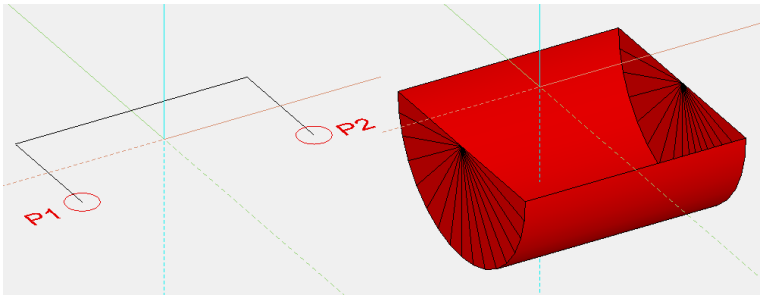
As soon as you select the DEFINE ROTATION EXTRUDE function, the properties bar appears.

In the case of rotation extrude definitions, the contour can also be open, but the contour must be in its own pattern. The axis of rotation must not lie within the contour to be rotated. This is not a problem outside or on the contour.

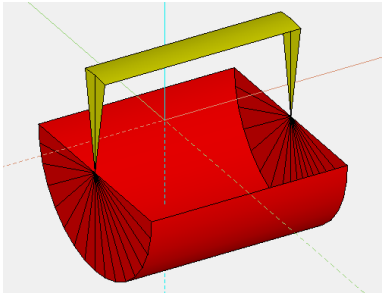
WORKSHOP

Define rotation extrude

Draw a contour, then click on DEFINE ROTATION EXTRUDE and set the start angle to **0**, the end angle to **-180** and the colour to **2**. Click points **P1** and **P2** for the position of the axis and tap on the contour.



Several definitions can also be stored for the same contour. Simply click again on DEFINE ROTATION EXTRUDE and set the start angle to **80**, the end angle to **100** and the colour to **3**. Click again on points **P1** and **P2** and tap on the contour.

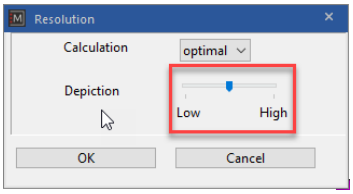


WORKSHOPENDE

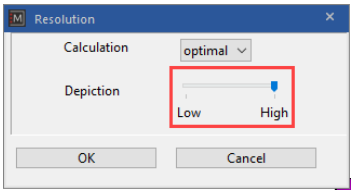
TIPP

In the case of round parts with a large radius, whether defined with a box, extrude, rotation box, rotation extrude, etc., the resolution can be set too coarse. In this case you can adjust the resolution of the object in the property bar of the respective object with the function MODIFY RESOLUTION.

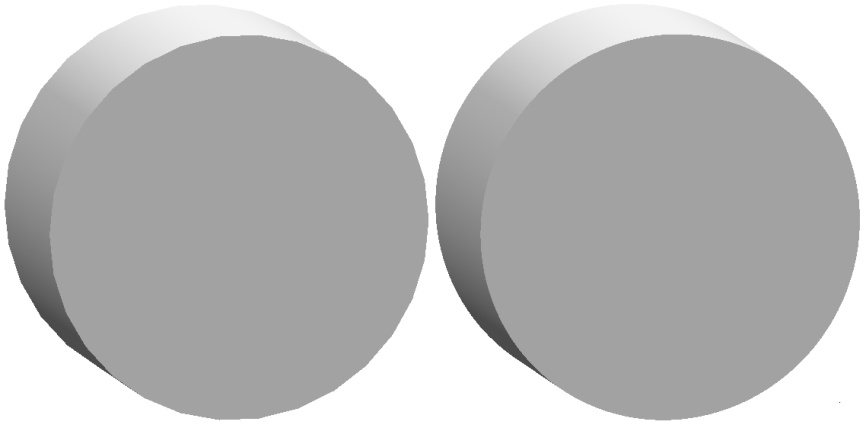
Default setting



Adjusted

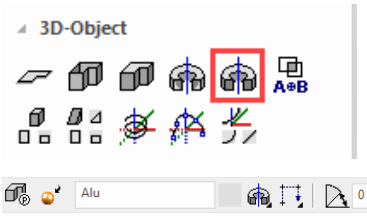


Difference between default setting (left) and fine (right)



DEFINE ROTATION BOX

As soon as you select the DEFINE ROTATION BOX function, the properties bar appears.

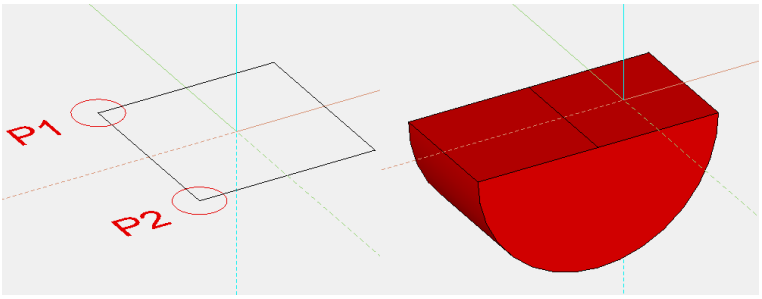


With rotation box definitions, the contour should be closed, the contour must be its own pattern. The axis of rotation must not lie within the contour to be rotated. This is not a problem outside or on the contour.

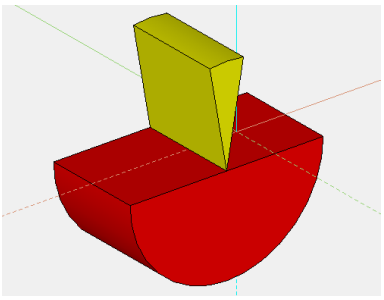
WORKSHOP

Define rotation box

Draw a contour, then click on DEFINE ROTATION BOX and set the start angle to **0**, the end angle to **-180** and the colour to **2**. Click points **P1** and **P2** for the position of the axis and tap on the contour.

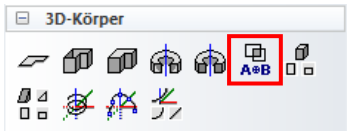


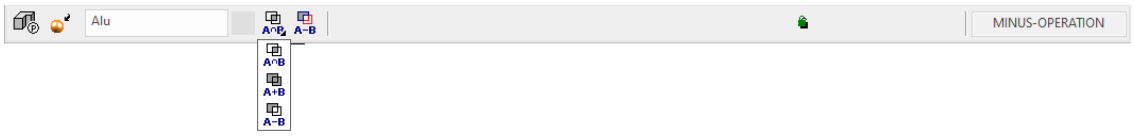
Several definitions can also be stored for the same contour. Simply click again on DEFINE ROTATION BOX and set the start angle to **80**, the end angle to **100** and the colour to **3**. Click again on points **P1** and **P2** and tap on the contour.



BOOLEAN OPERATION

As soon as you select the BOOLEAN OPERATION function, the properties bar appears.





Associative

If the switch is active, the result is linked with the output definitions in an associative manner. If one of the output definitions is changed, the result is automatically updated.

Average

The result is the average (intersection) of the specified bodies.

Union

The specified bodies are united to one body.

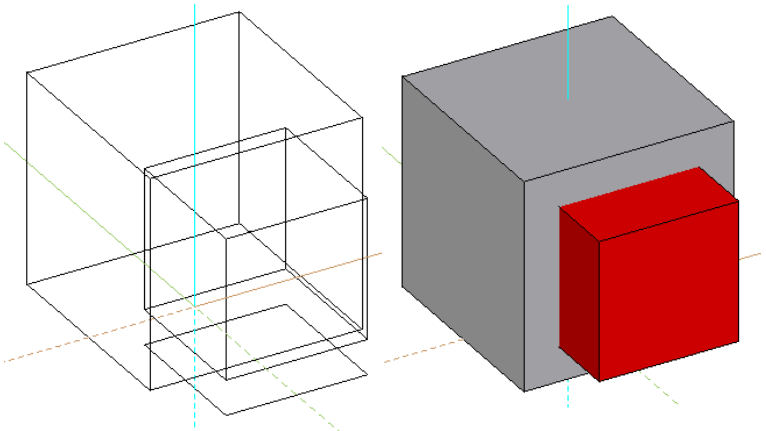
A minus B

The bodies are pulled away from each other. The Boolean operation creates a new body. The new body gets the colour of the A-body and the classification of group, class and level.

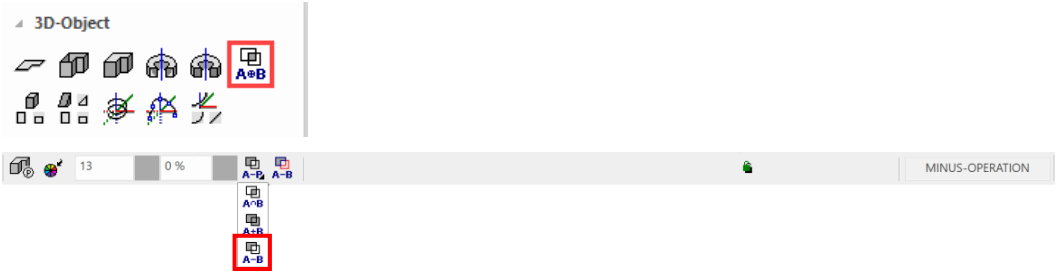
WORKSHOP

Boolean operation

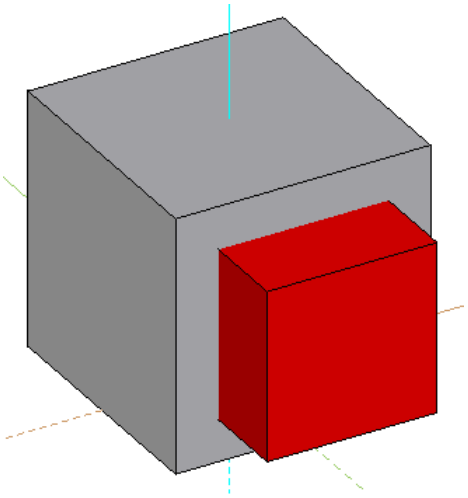
1. The basis in this example are two boxes that interlock. Start a new pattern for both rectangles and define a box from it.

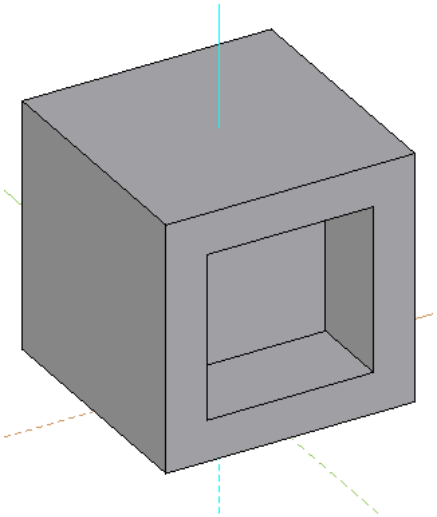


2. Then press the BOOLEAN OPERATION function and make the following settings.

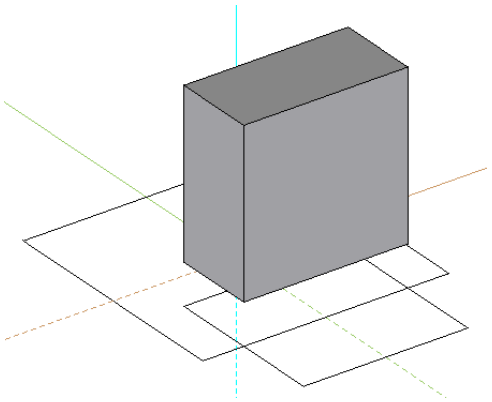


The following message now appears in the info line:
Please click definition A (ENTER = end)
3. Click on the larger of the two boxes with the mouse and then confirm with **[Enter]**.
The following message now appears in the info line:
Please click definition B (ENTER = end)
4. You can now specify the body that is to be deducted. Click on the smaller box and confirm here with **[Enter]**.

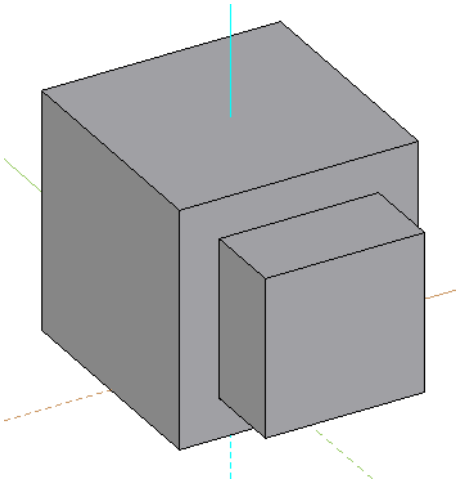




The "Average" option gives the following result. 



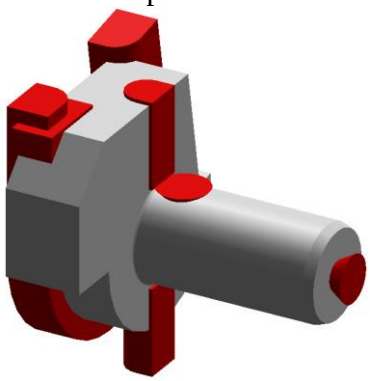
The "Union" option gives the following result. 

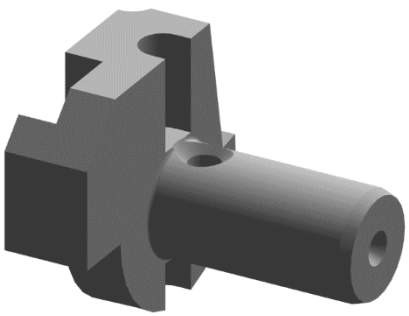


WORKSHOP END

TIP

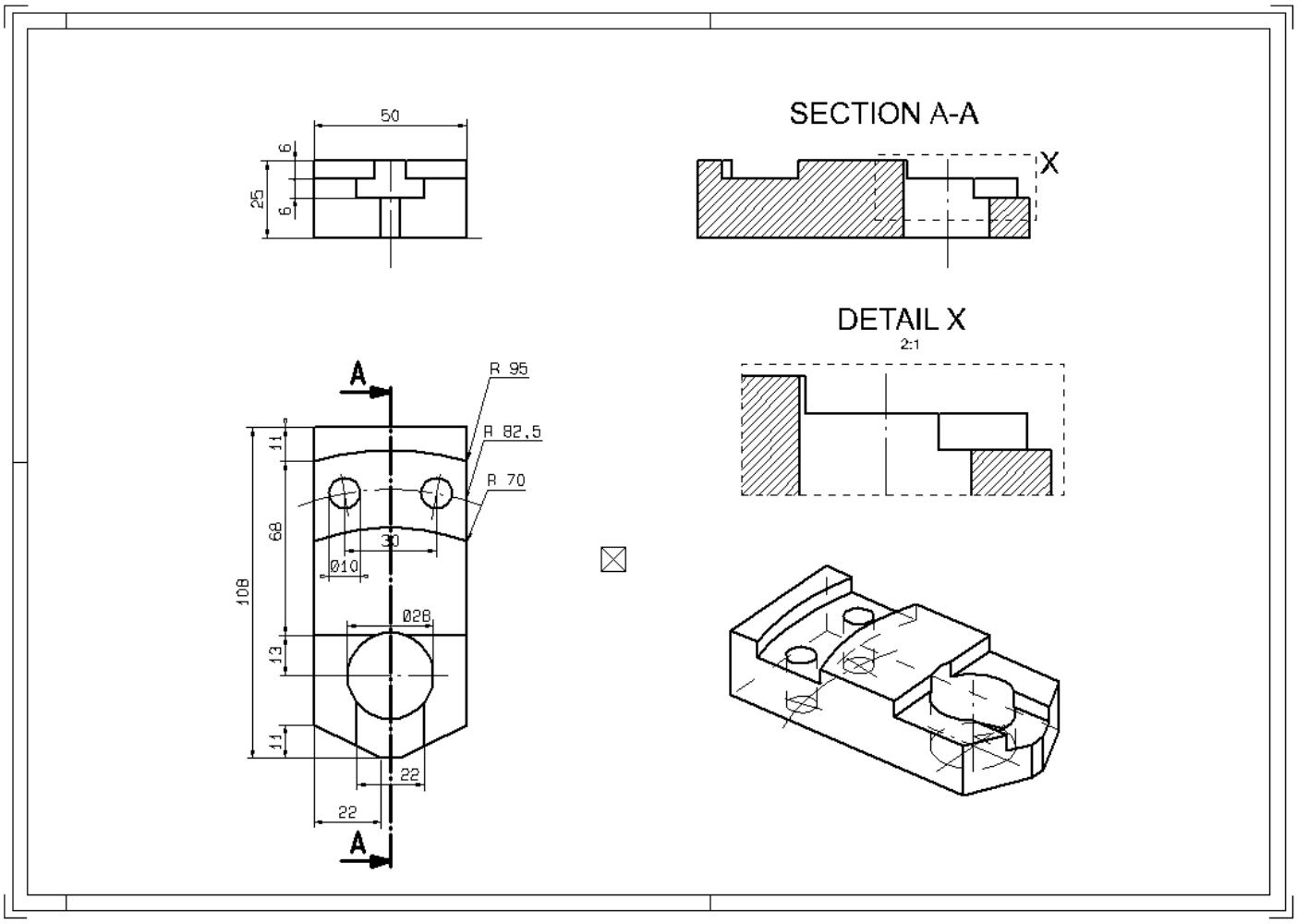
When selecting the various definitions "A" or "B", you can choose as many bodies until the process is ended with **[Enter]**.





EXAMPLE – EXAMPLE 01_3D

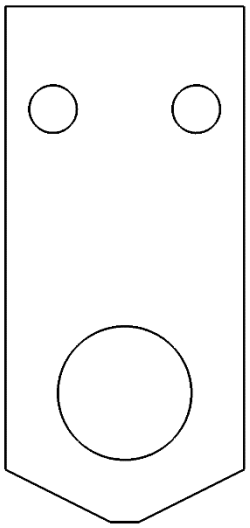
First, let's look at the desired result.



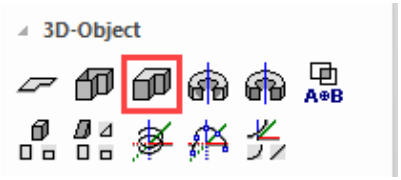
BOX (Define base body)

WORKSHOP

1. Draw the following contour. Here you can already apply the knowledge from the "COURSE UNIT 2D". (Please take the dimensions from the dimension sheet)



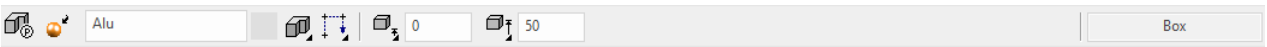
2. Select the DEFINE BOX function.



Please enter height 1 for CURRENT PATTERN {0}
Enter **0 [ENTER]** here.
Please enter height 2 for CURRENT PATTERN {0}
Enter **25 [ENTER]** here.

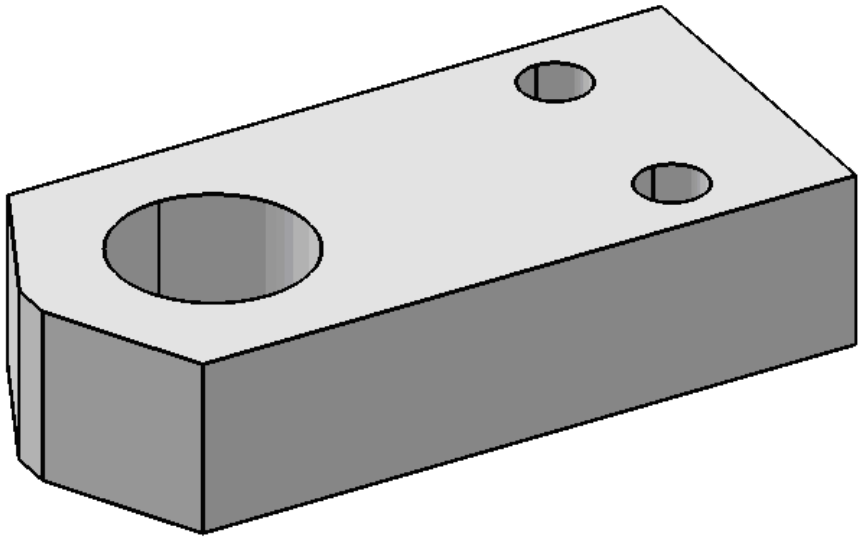
TIP

The heights can also be entered on the property bar.



The colour is also specified here.

3. Now turn into 3D with the middle mouse button pressed and switch on the solid model (Ctrl D) and the edges (Ctrl K).



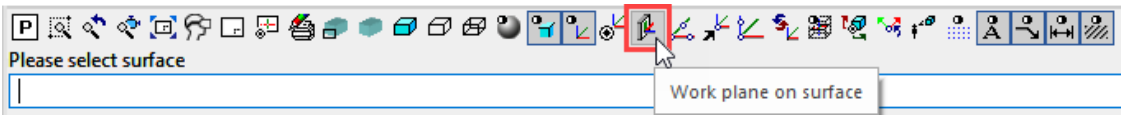
WORKSHOP END

Work plane on surface

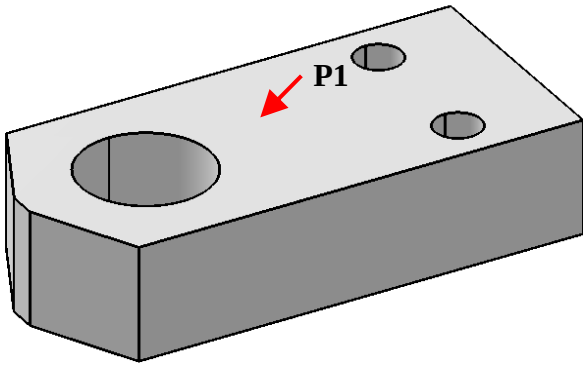
With ELITECAD, work planes can be rotated as required or placed on surfaces. When working in the view, the CLIPPING function is a helpful means of changing the area of visibility.

▼ ▼ ▼ ▼ WORKSHOP

So that the bodies can be drawn for later subtraction (Boolean operation), you must place the work plane on the upper surface.
1. Place the WORK PLANE ON SURFACE

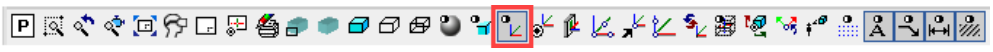


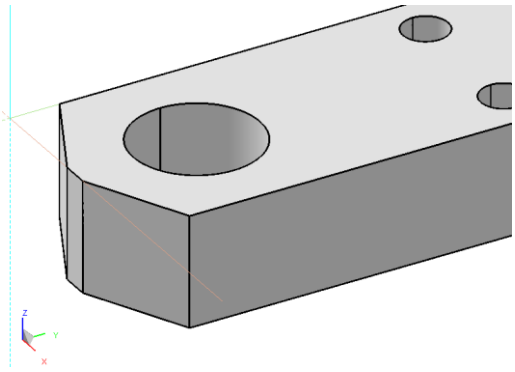
Click point **P1** on the surface.



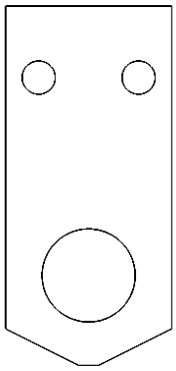
TIP

Switch on the 3D axes so that you have a better overview of the current work plane.

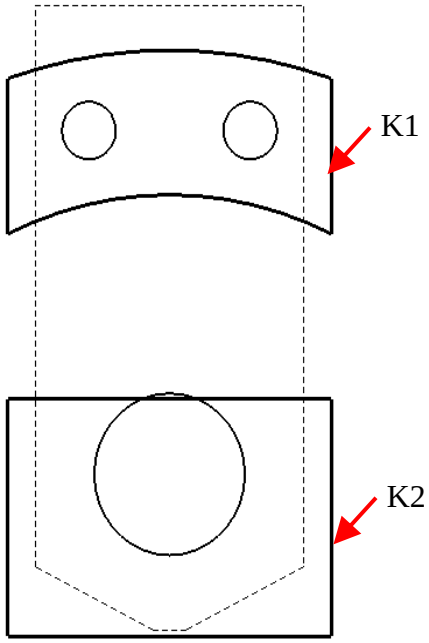




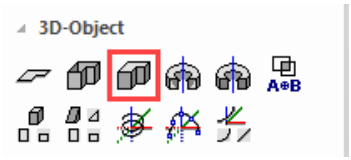
2. Set the working plane perpendicular and switch to wireframe.
[Ctrl] + [Space], [Ctrl] + [D]



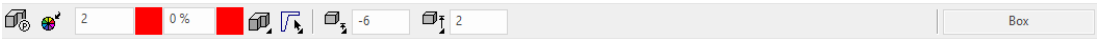
3. Draw the contours **K1** and **K2**. (Please take the dimensions from the dimension sheet)



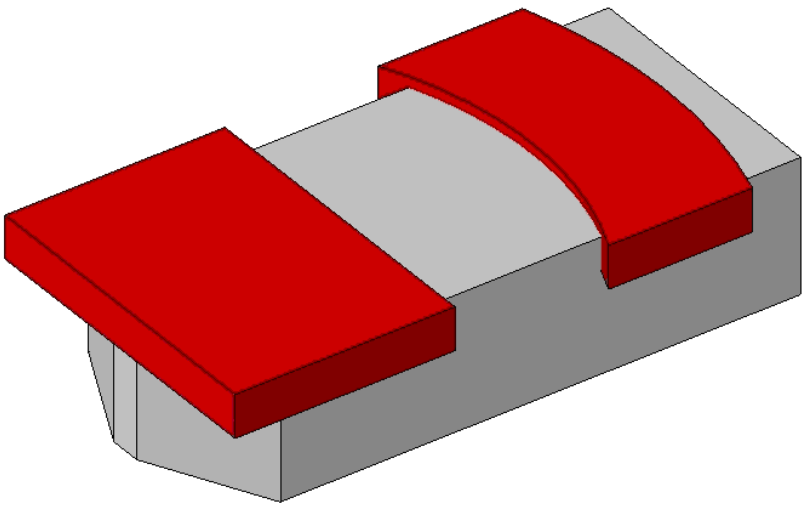
4. Select the DEFINE BOX function.



5. Enter the following values on the property bar.



6. Now turn into 3D with the middle mouse button and switch on the solid model and the edges .



WORKSHOP END

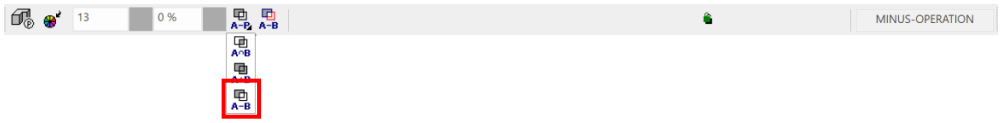
Boolean operation 1

Now you will be remove the two red bodies off the main body. This is done with the Boolean operation.

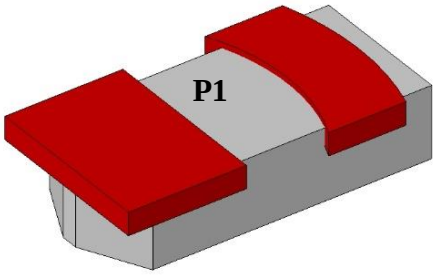
WORKSHOP

1. Start with the BOOLEAN OPERATION function.

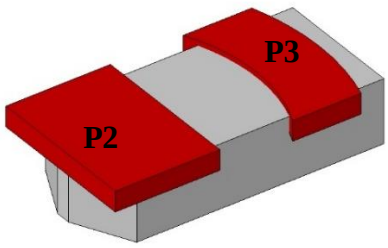
Now select option A minus B.

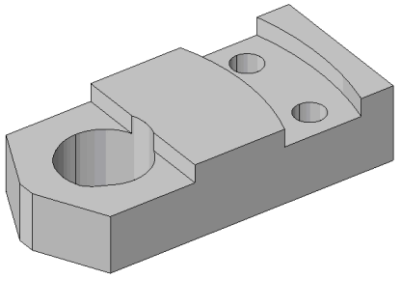


2. Please click definition A (RETURN = end)
Here please select the gray main body point **P1**, and then confirm with **[ENTER]**. (The selected bodies are shown in purple)



3. *Please click definition B (RETURN = end*
Here please select the red bodies point **P2** and **P3**, and then confirm with **[ENTER]**. (The selected bodies are shown in purple)





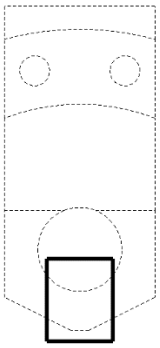
TIP

For the Boolean calculations, the deduction bodies should protrude beyond the base contour (body A) if possible.

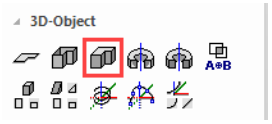
Save work copy

 or with the key combination [Strg]+[W]

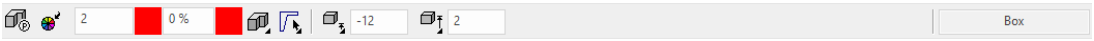
1. Draw the following contour. (Please take the dimensions from the dimension sheet)



2. Select the DEFINE BOX function.



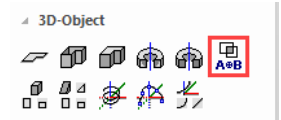
3. Enter the following values on the property bar.



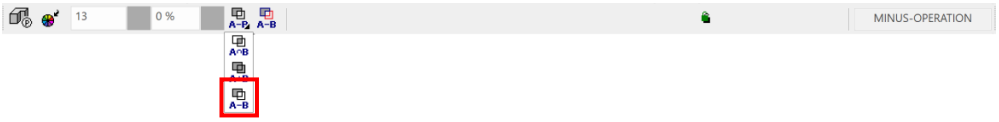
4. Now turn into 3D with the middle mouse button and switch on the solid model  and the edges  .

Boolean operation 2

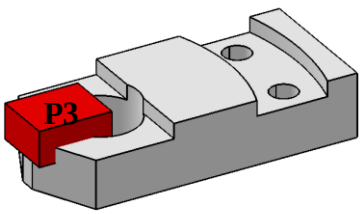
1. Start with the BOOLEAN OPERATION function.

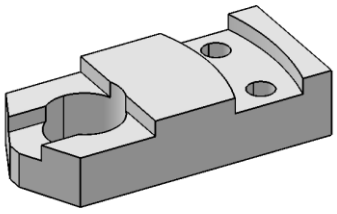


Now select option A minus B and confirm with OK.



2. *Please click definition A (RETURN = end)*
Here please select the gray main body point and then confirm with **[ENTER]**. (The selected bodies are shown in purple)
3. *Please click definition B (RETURN = end)*
Here please select the red body point **P3** and then confirm with **[ENTER]**. (The selected body is shown in purple)

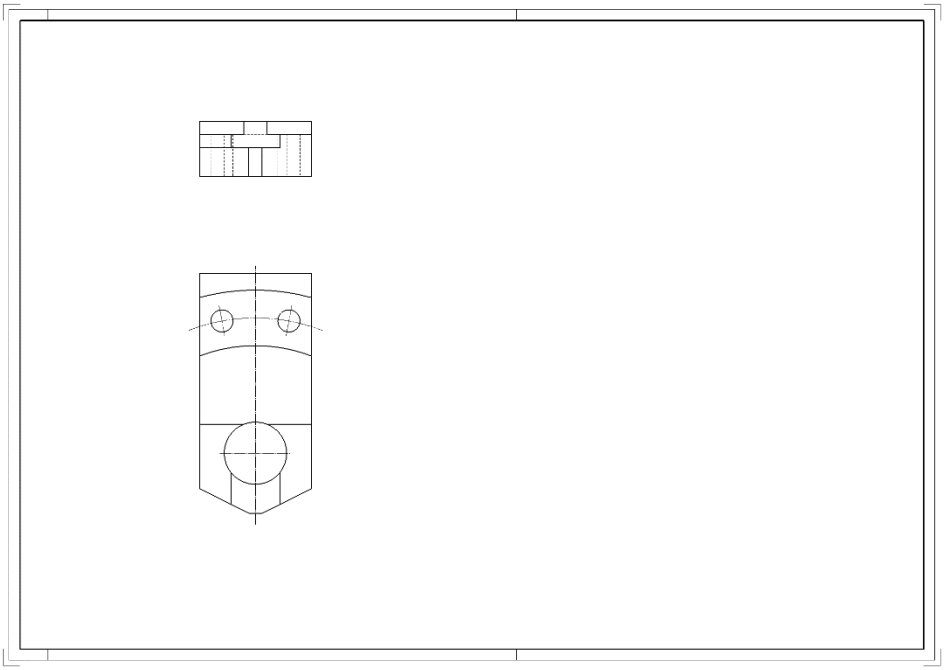




WORKSHOP END

Define views

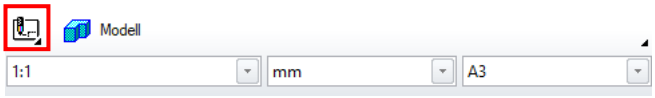
With ELITECAD, it is possible to generate 2D drawings, views and sections at the push of a button.



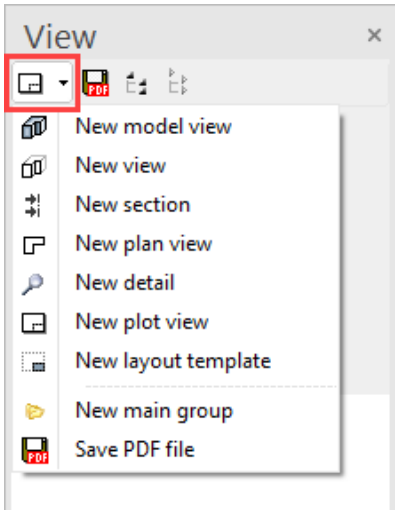
▼▼▼▼ WORKSHOP

Generation of 2D views.

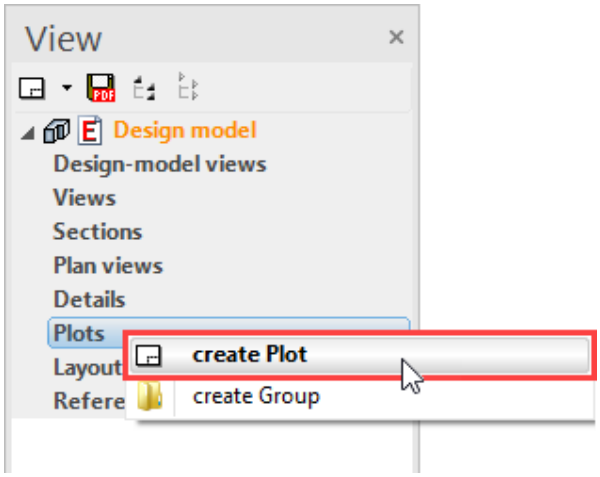
1. Call up view management.



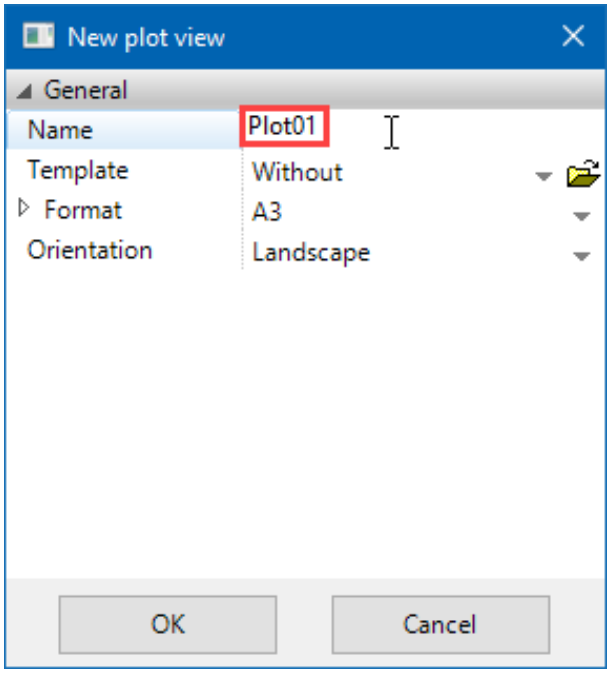
2. Create a new plot.



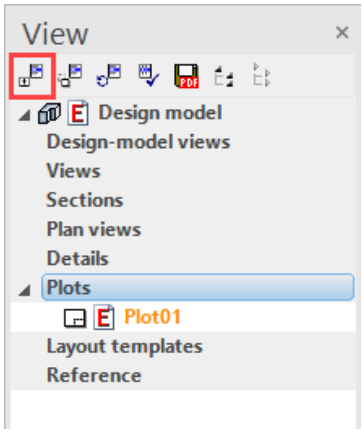
or with the right mouse button



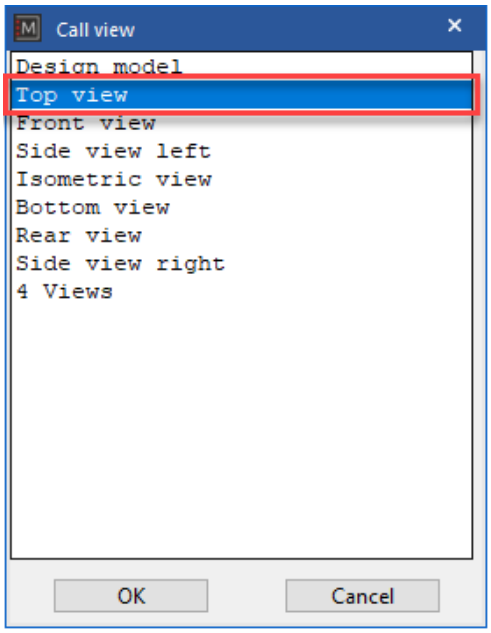
3. Make settings and confirm with OK.



4. Now the desired view is selected with "Insert view".

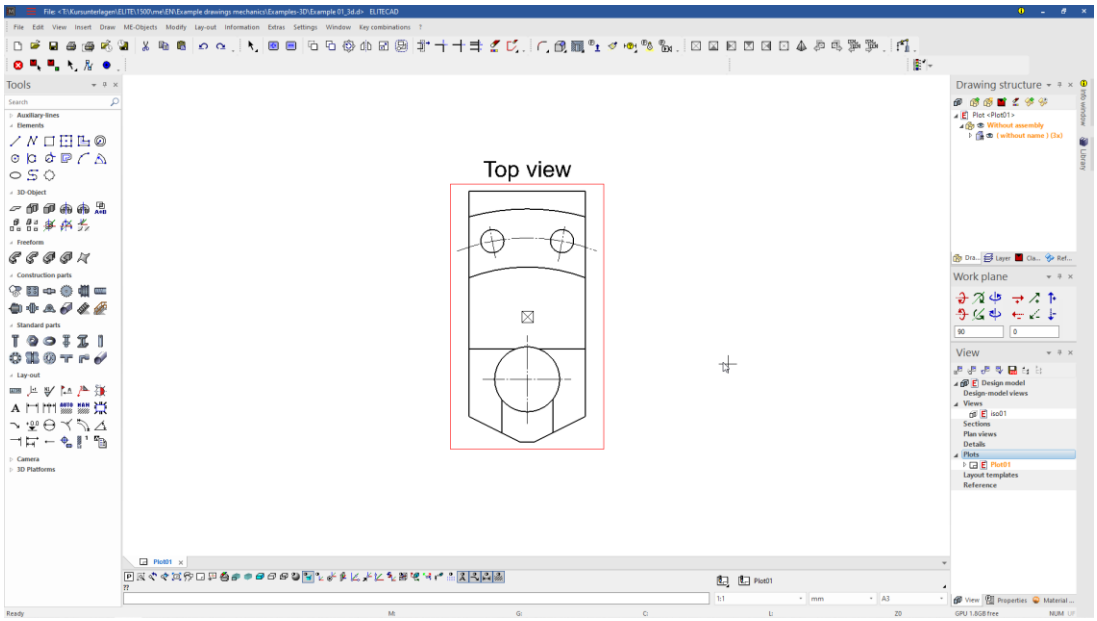


5. Select view and confirm with OK.



IMPORTANT: With this selection, the filter (group, class, level) is taken into account. If, for example, a group is selected and set up before "SELECT PLOT", only those parts are shown in the plot that have this group name.

6. Place the top view on the plot.

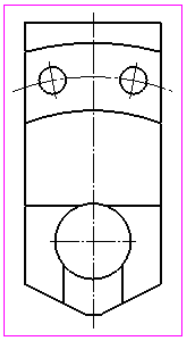


TIP

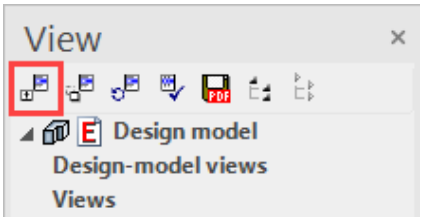
As described in points 4 to 6, all other views could now can also be inserted into the plot.

Another possibility is shown here:

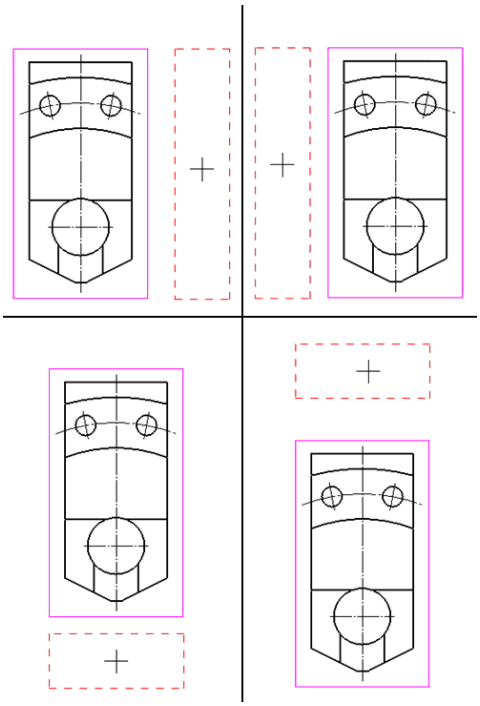
Select (tap once) the top view that has already been inserted. (the frame turns purple)



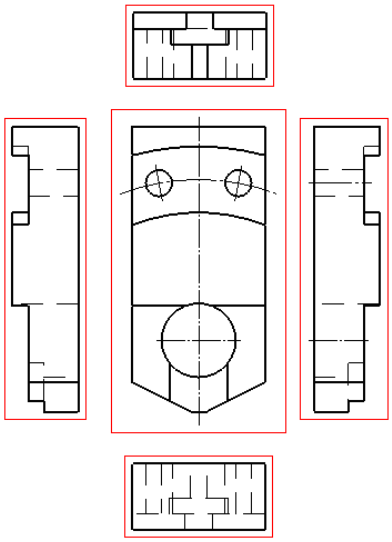
Now click again on "Insert view".



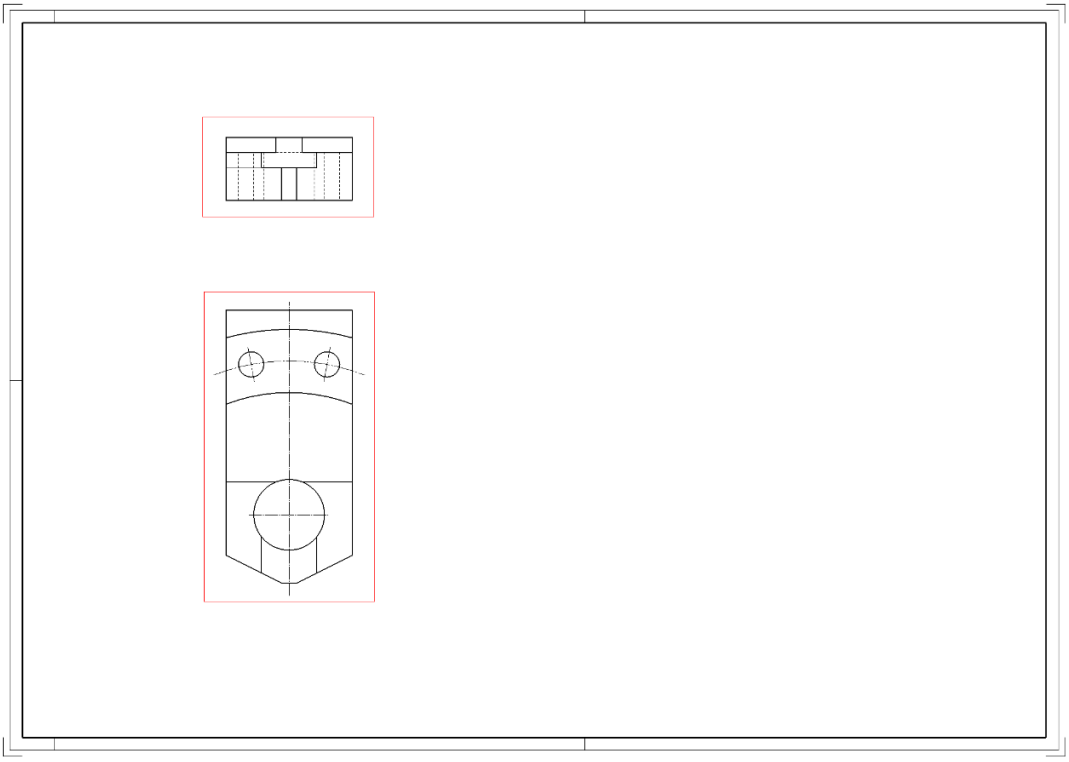
Move the mouse pointer over the "plot" (left, right, up, down). Now all four views can be positioned with a click of the mouse, over the top view, so to speak.



After inserting the views, the result should look like this.



7. Insert the top and front view and position them.

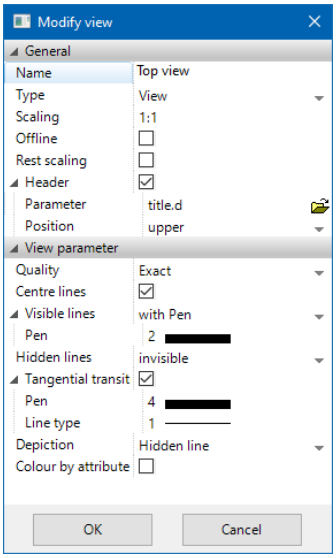


WORKSHOP END

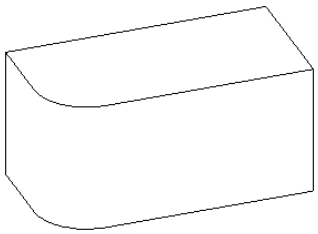
TIP

Individual settings or changes can be made to all views:
1. Select the desired view by double-clicking on the red frame (the frame will be bold)
2. Select "MODIFY VIEW".

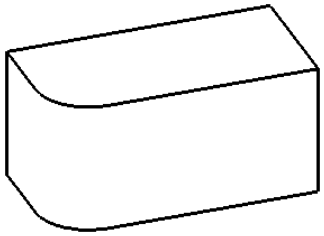
3. All settings / modifications can be made in the following window.



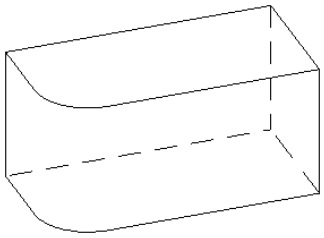
Erklärung der Einstellmöglichkeiten:



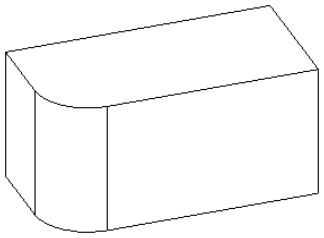
Visible lines — Edge colour



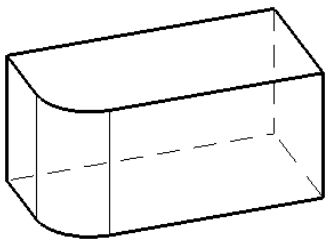
- Visible lines — with Pen
  - Pen — 5



- Hidden lines — with Pen
  - Pen — 6
  - Line type — 3



- Tangential transit — ☑
  - Pen — 4
  - Line type — 1



- Visible lines — with Pen
  - Pen — 2
- Hidden lines — with Pen
  - Pen — 3
  - Line type — 4
- Tangential transit — ☑
  - Pen — 4
  - Line type — 1

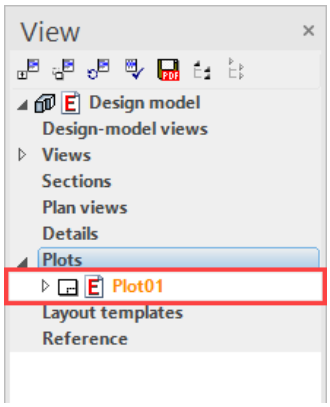
Define section

Sections are calculated 2D views. Changes that you make in these sections are not taken into the model. However, changes that you make in the model and affect the sections can be accepted using the REFRESH VIEW function.
Now you are going to make a section through the model.

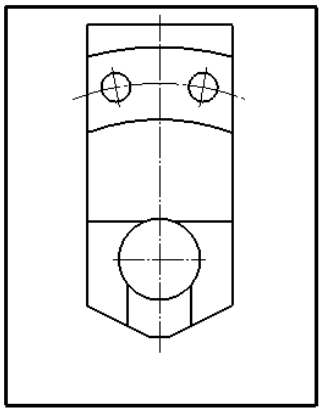
WORKSHOP

Create a simple section.

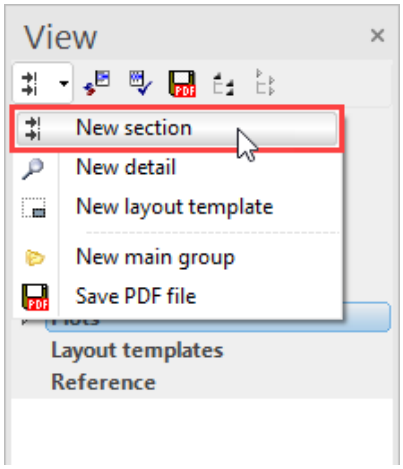
1. Switch to the plot.



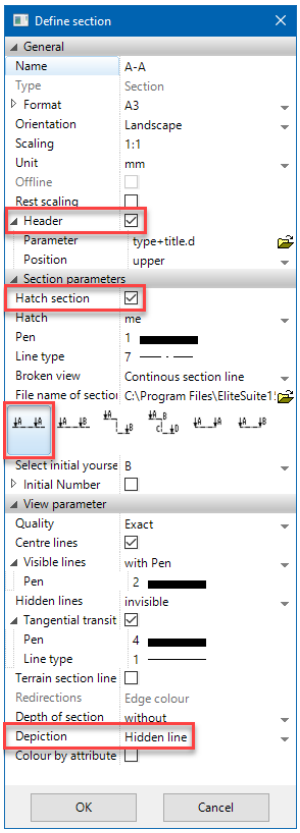
2. Activate the top view. (Double click on its frame)



3. Define a NEW SECTION



4. Make settings and confirm with OK.



Enter point1 for cutting course [identical point -> end]

5. Specify point **P1**.

Enter point2 for cutting course [identical point -> end]

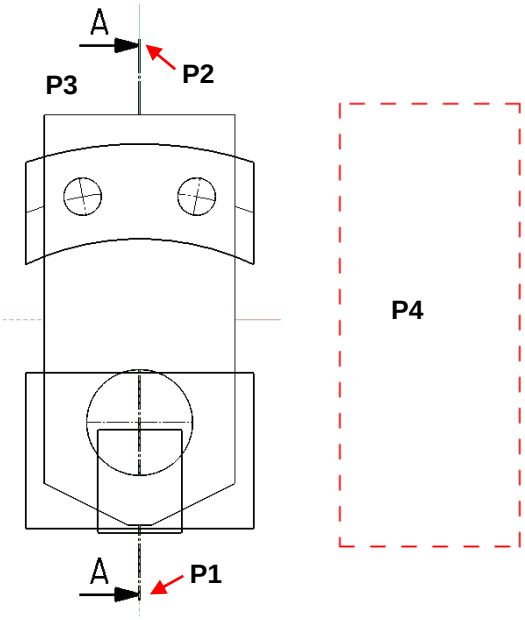
6. Specify point **P2**.

Which direction?

7. Specify point **P3** for the direction.

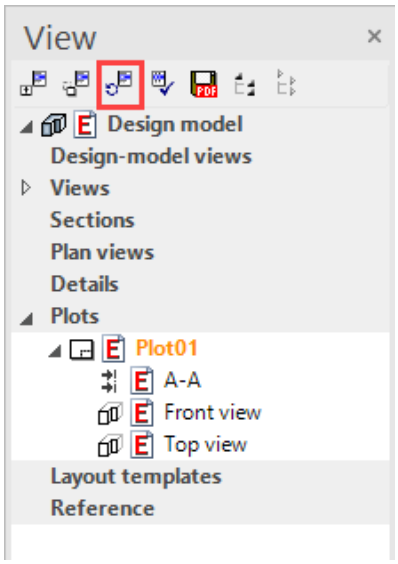
Position where to?

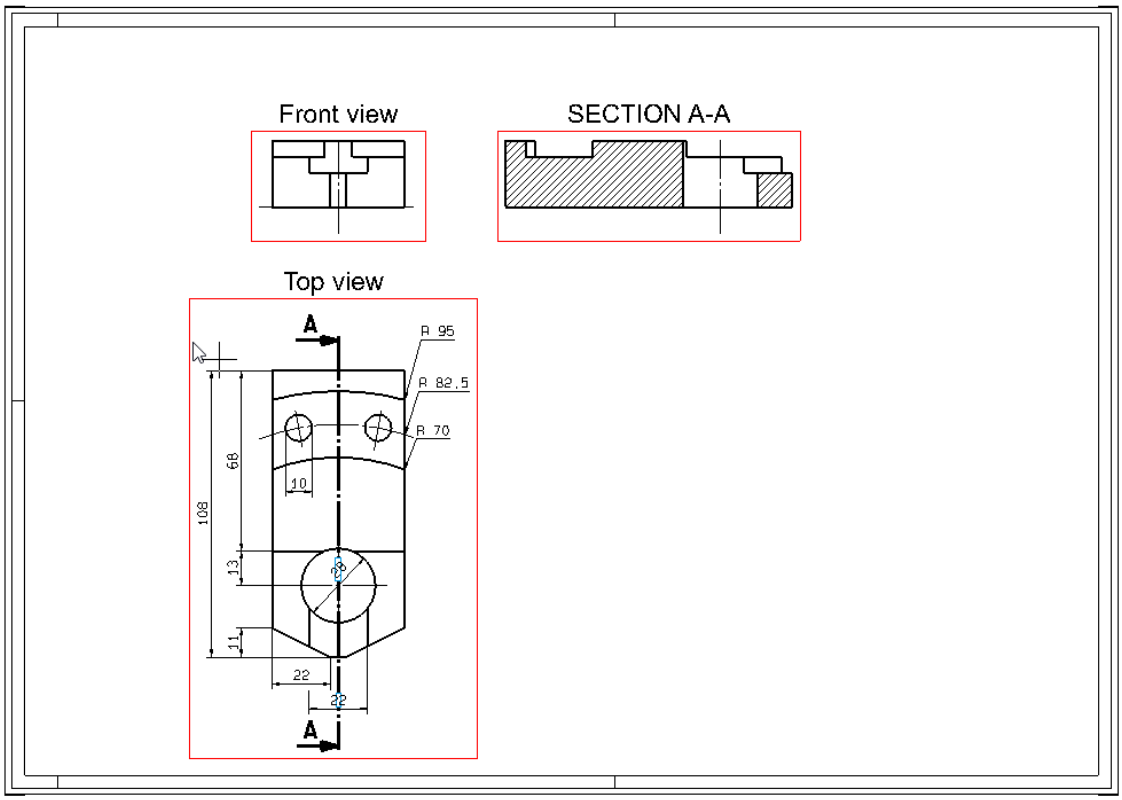
8. Draw the section to **P4**.



9. The section has now been added to the plot.

10. Rotate the section 90 degrees and position it.





WORKSHOP END

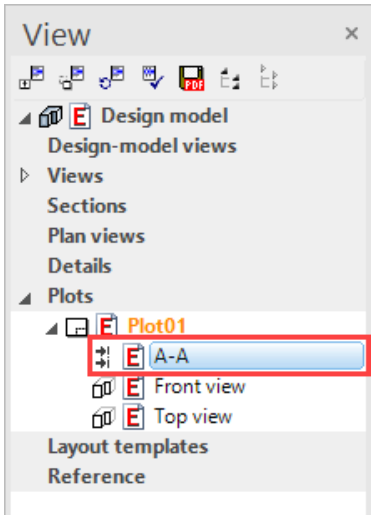
Define detail

The details are calculated 2D plans. Changes that you make in these details are not taken in the model or in the original plan.
However, changes that you make in the model and affect the details can be accepted using the REFRESH VIEW function.

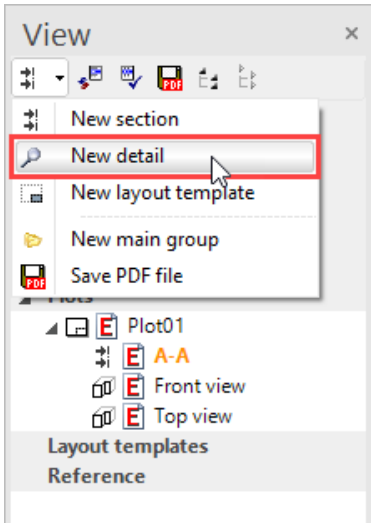
WORKSHOP

Create a detail of the previously created section A-A

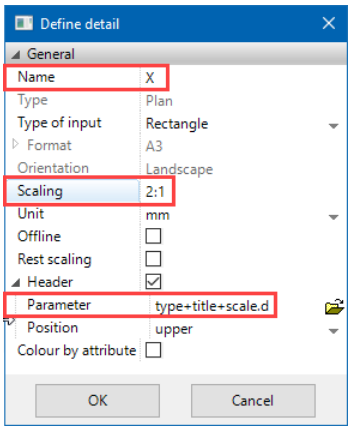
1. Switch to section A-A. By clicking in the view manager or double click on the frame of section A-A.



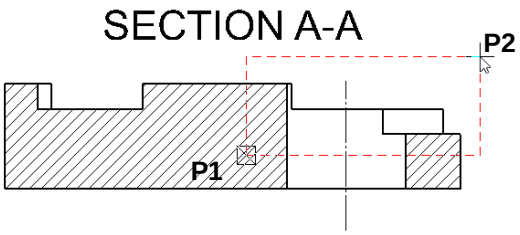
2. Define a NEW DETAIL



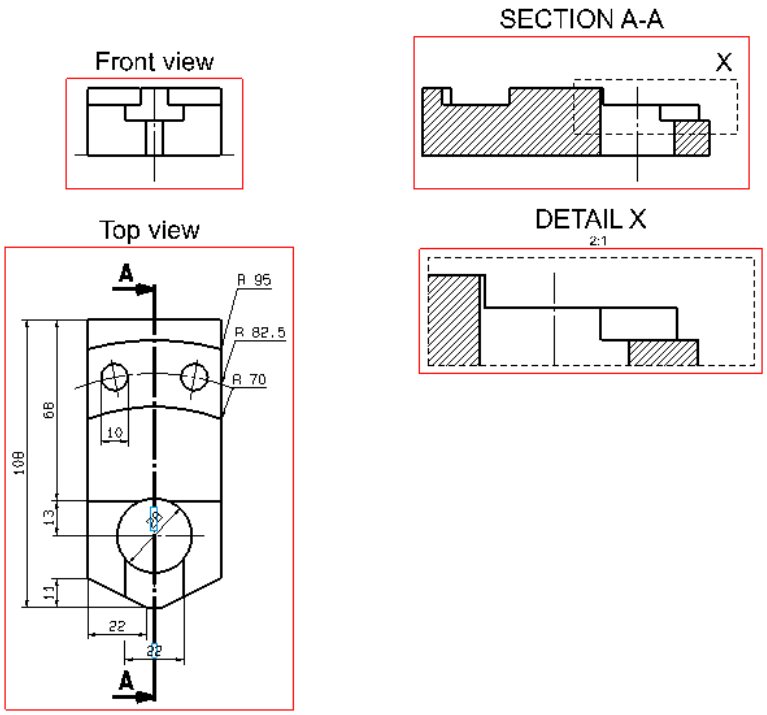
3. Make settings and confirm with OK.



Enter a corner of the rectangle!
select **P1**
Diagonal corner of the rectangle?
select **P2**



5. Place the detail letter X next to the detail outline.
6. Then position this detail in the plot



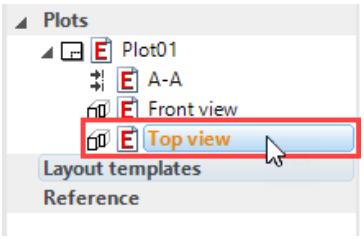
WORKSHOP END

Dimensions

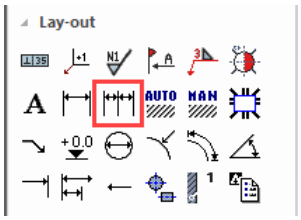
The dimensions are shown using the top view.

WORKSHOP

1. Activate the view in which the measurement should now be carried out. To do this, click on the view in the view management under Plot. The frame of the view now becomes thick and red.



2. Select the CHAIN DIMENSION function.



1. Point of dimension?

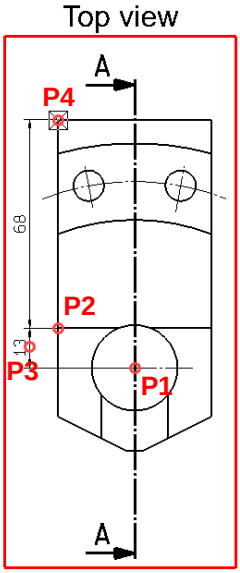
3. Select point **P1**

2. Point of dimension?

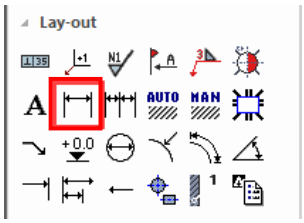
4. Select point **P2**

Position of dimension?

5. Select point **P3**. Since we have selected the CHAIN DIMENSION function, a second measure can now be set immediately. Click on point **P4**.



6. Select the SINGLE DIMENSION function.



1. Point of dimension?

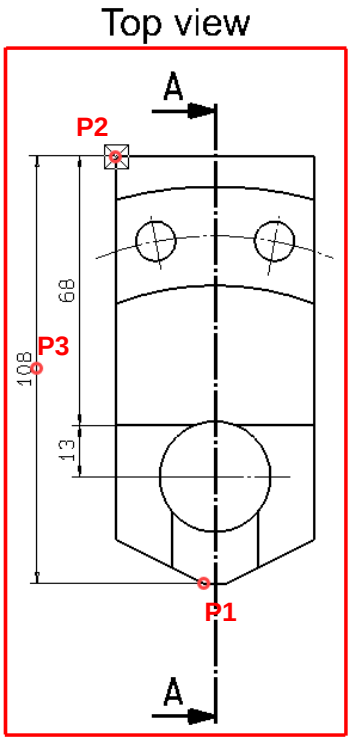
3. Select point **P1**

2. Point of dimension?

4. Select point **P2**

Position of dimension?

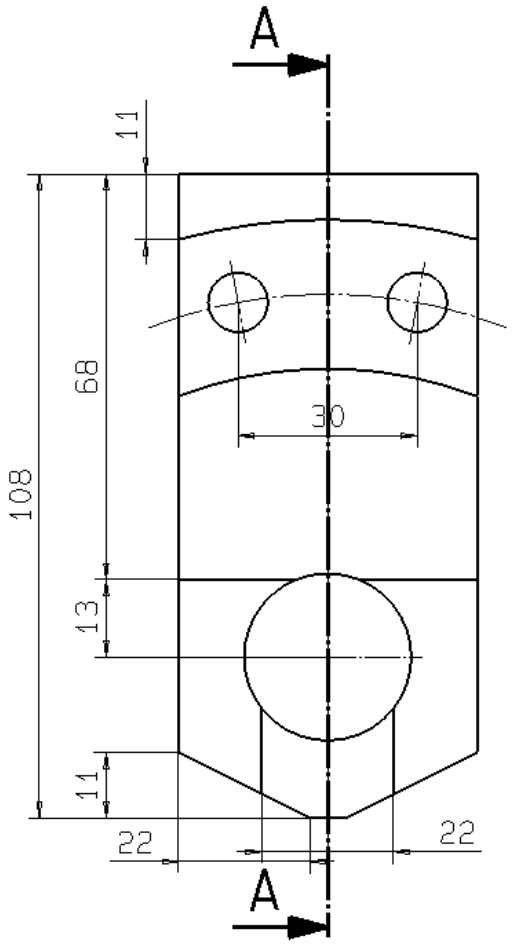
5. Select point **P3**.



TIPP

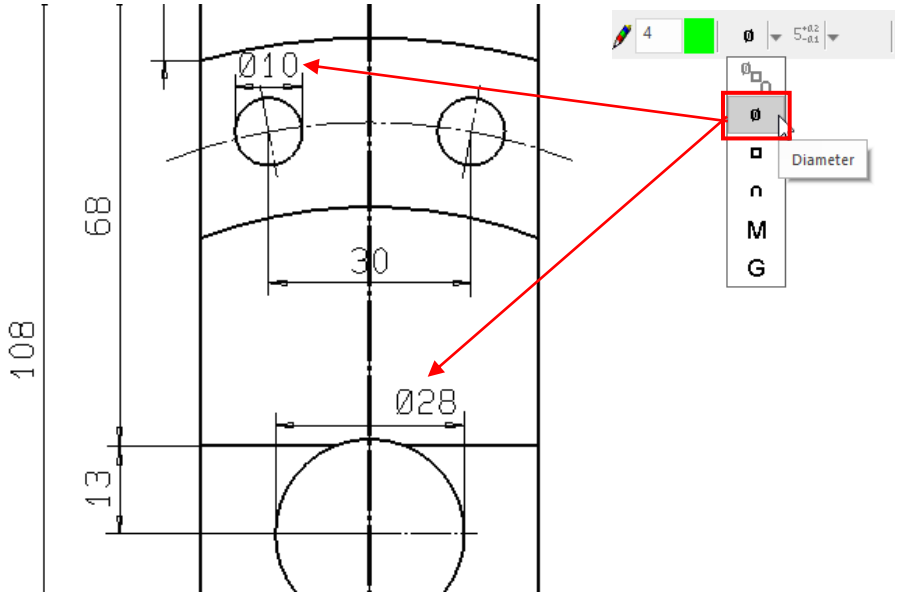
The SINGLE DIMENSION function remains active until you cancel it with [Esc].

6. Now set the other dimension lines yourself, as shown in the next figure.

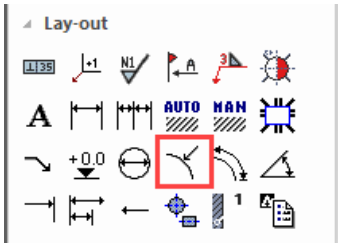


7. The diameter symbol "Ø" is required for the next dimension lines. Select SINGLE DIMENSION and activate it.

8. Now set the two other dimension lines (diameter) yourself.

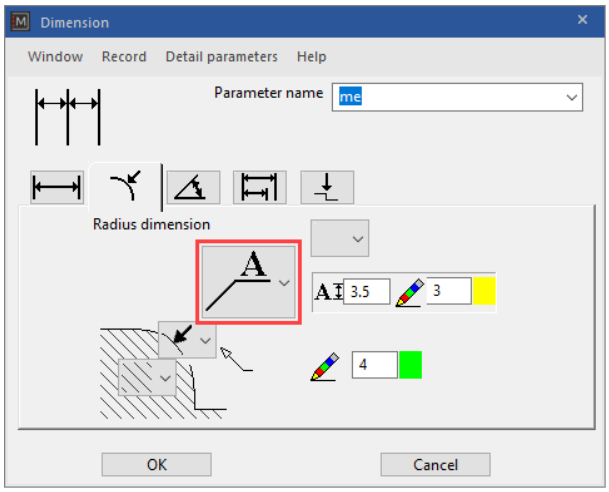


9. Now activate the RADIUS DIMENSION function.



10. Switch to the DIMENSION PARAMETERS

11. Make the following settings and confirm with OK.

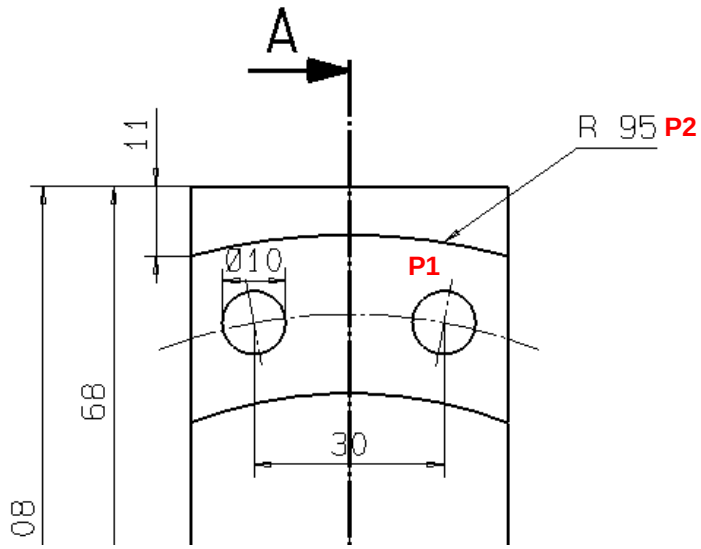


Arc for radius dimensioning?

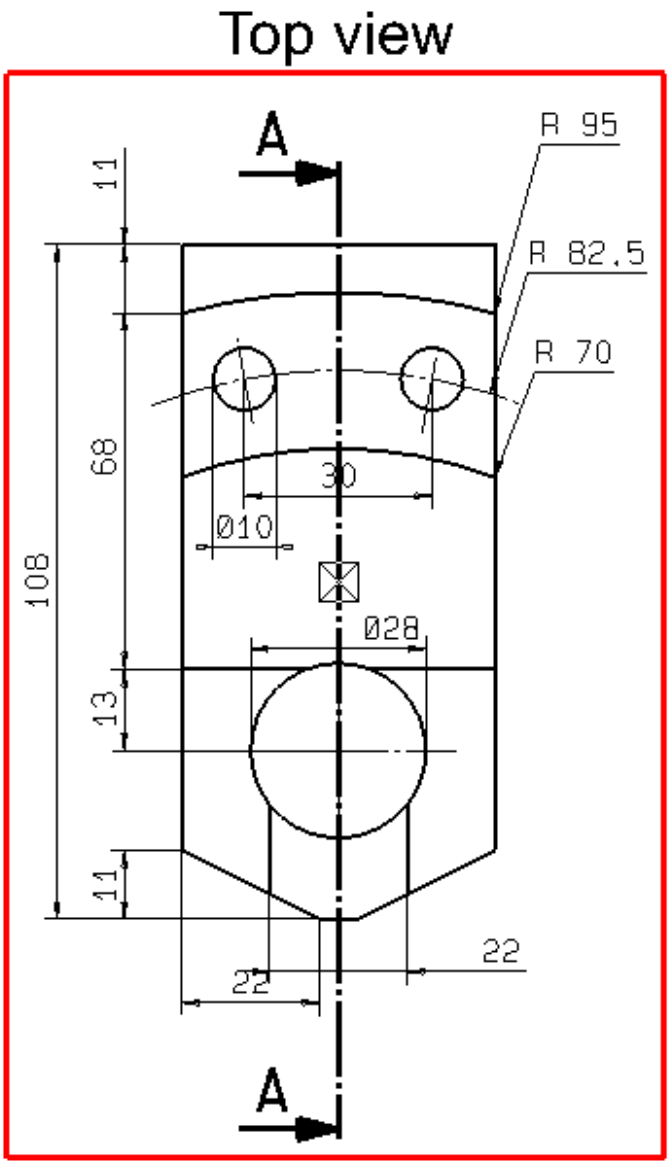
12. Select **P1**

Text position?

13. Select **P2**



14. Now set the other radius dimensions yourself.

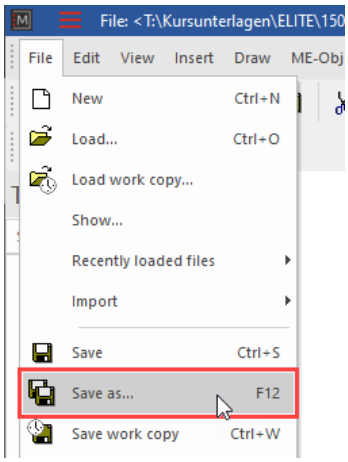


WORKSHOP END

Save your drawing

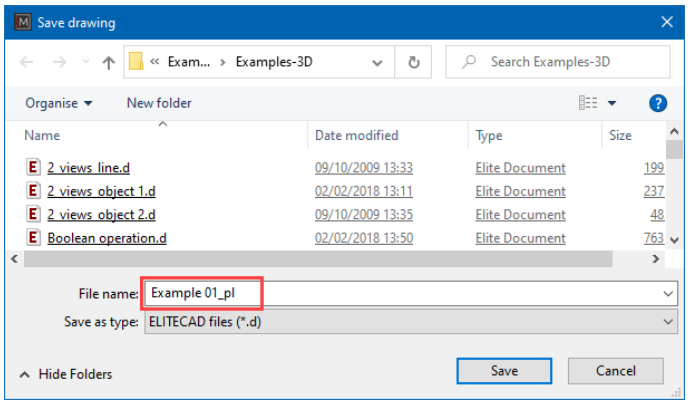
WORKSHOP

We are now saving our drawing under a new file name



or with the key [F12]

Now enter the desired directory and file name



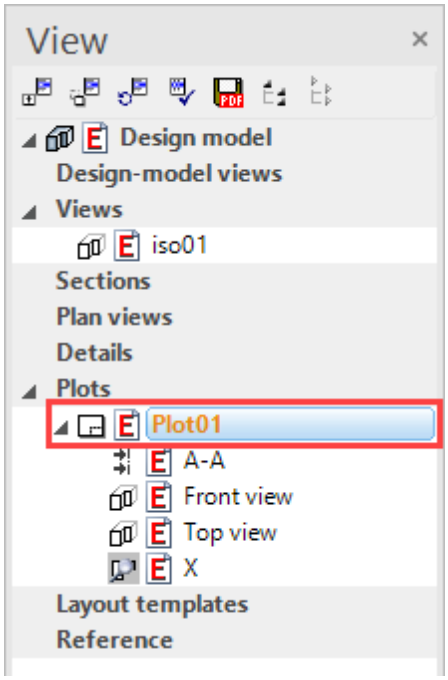
WORKSHOP END

Printing / plotting

Now all you have to do is get the plan down on paper.

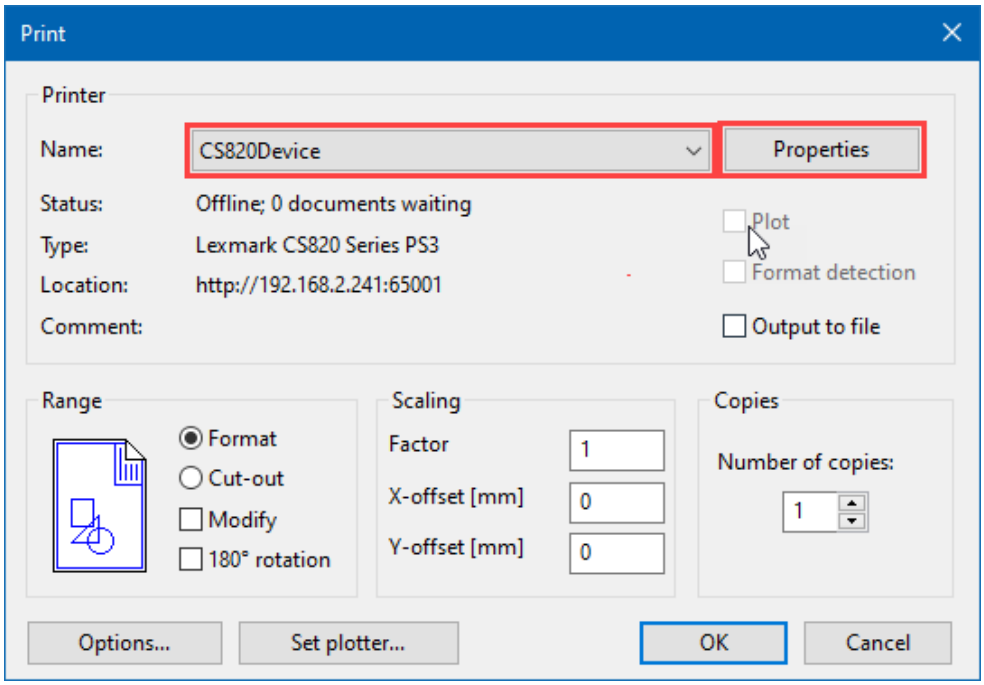
▼▼▼▼ WORKSHOP

1. Load the plot "Plot01" onto the screen.



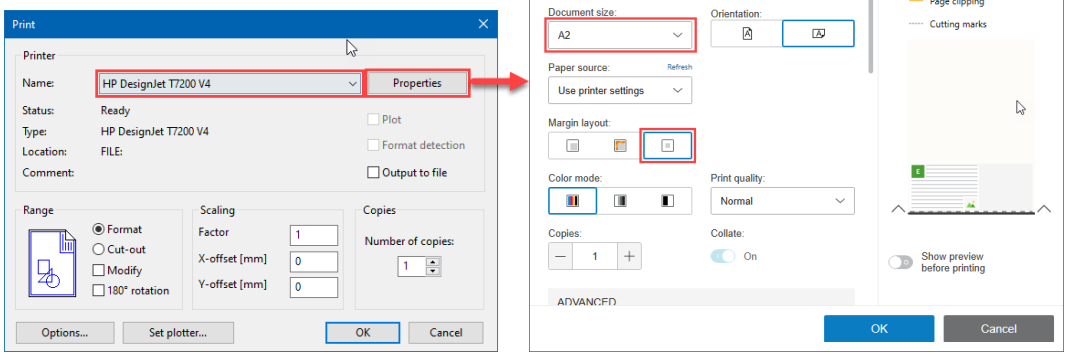
2. Click the PRINT icon.

3. Select the name of the plotter

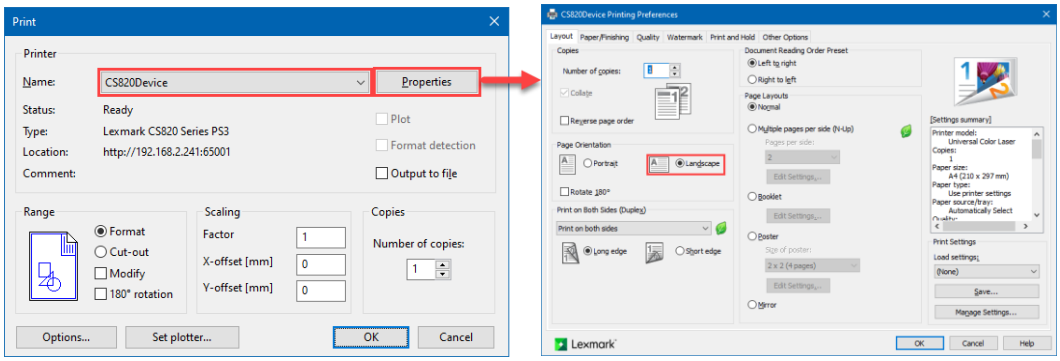


4. More settings
Click on PROPERTIES. The mask looks different depending on the plotter / printer and driver.
If a roll is clamped in the plotter, an oversize must be set for the sheet format. This is the only way to print the edge of the plan.

HP DesignJet T7200



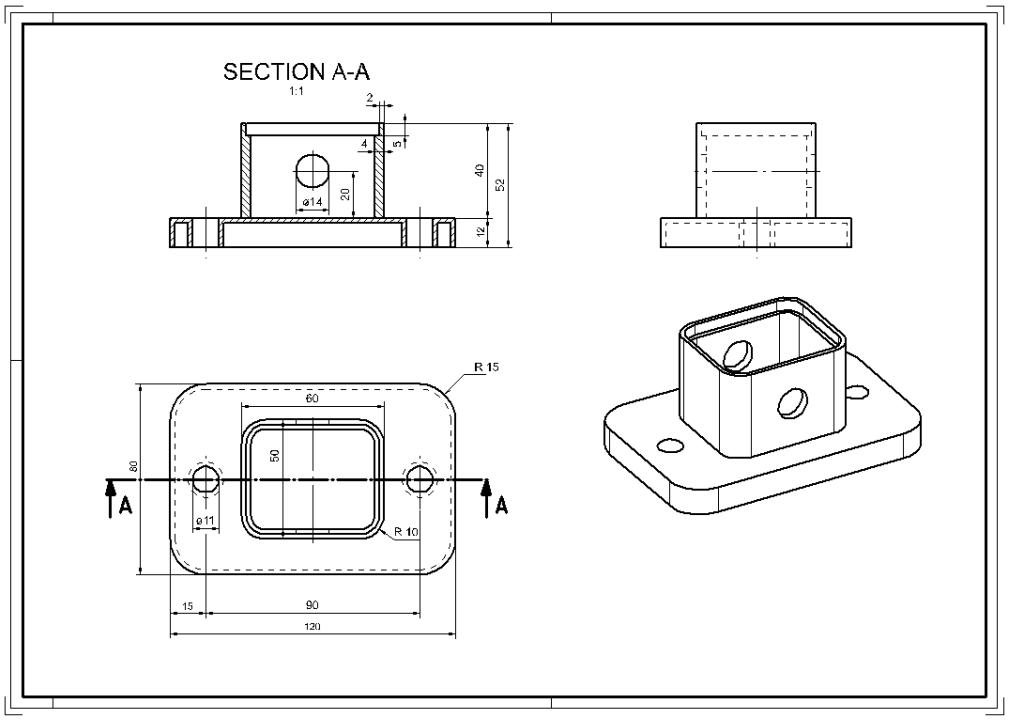
Lexmark CS820



5. Confirm both masks with OK and the plan will be output on the plotter / printer.

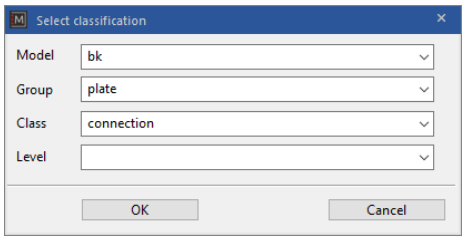
WORKSHOP END

EXAMPLE – CONNECTION 3D

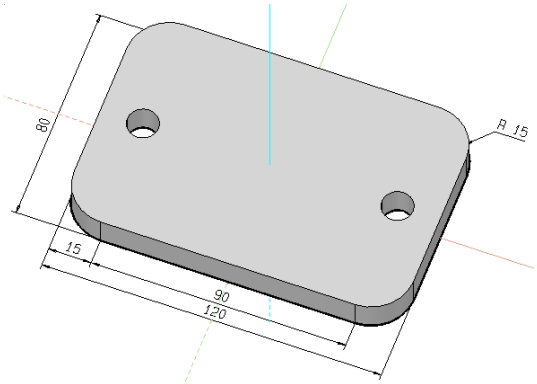


WORKSHOP

1. NEW MODEL 

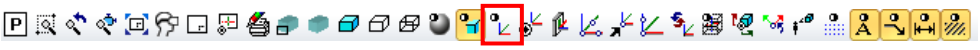


2. RECTANGLE  (120x80) and define it as a BOX.
(*Height 1* = **0**; *height 2* = **12**)
3. Round corners (radius 15mm) and draw circles (holes) (result is immediately visible in 3D). Measure the plate.



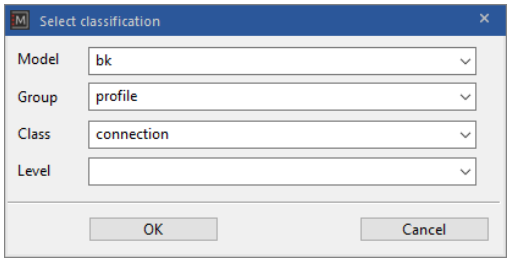
TIP

Switch on the 3D axes so that you have a better overview of the current work plane (WP).

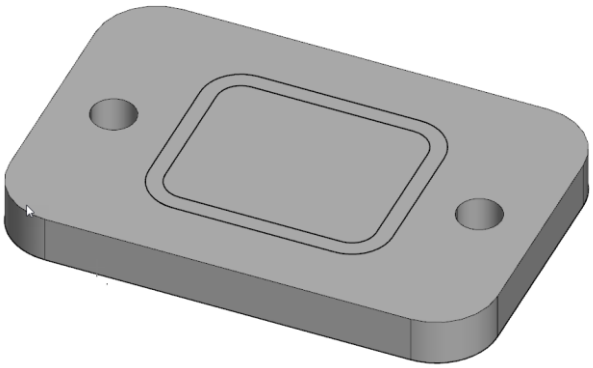


4. SELECT WORK PLANE ORIGIN and place it on the upper surface of the plate (snap mode on center surface).

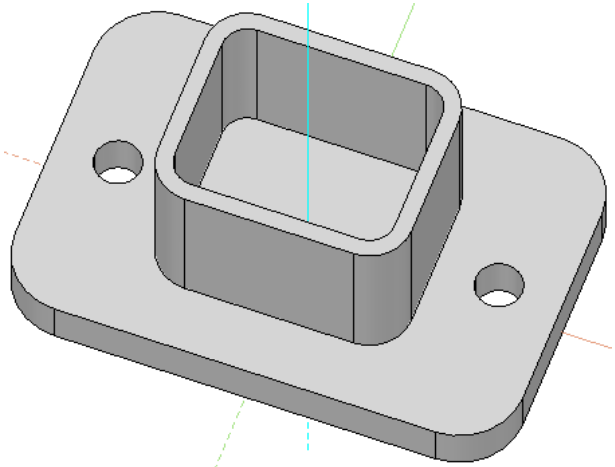
5. MUSTER STARTEN 



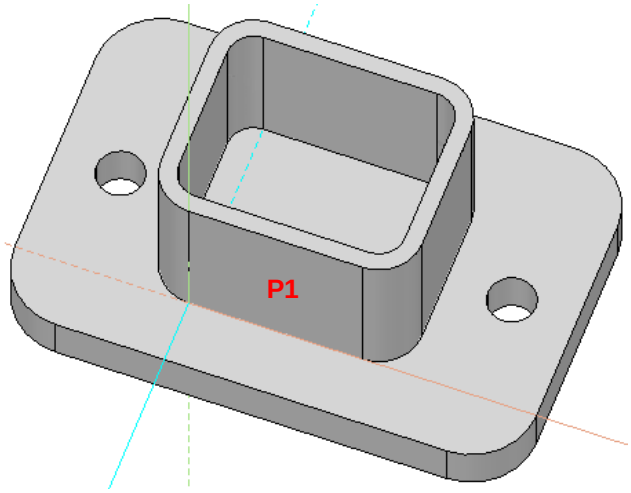
6. Draw the upper rectangle (60mmx50mm), round the corners (radius 10mm) and enter a parallel contour (4mm inside).



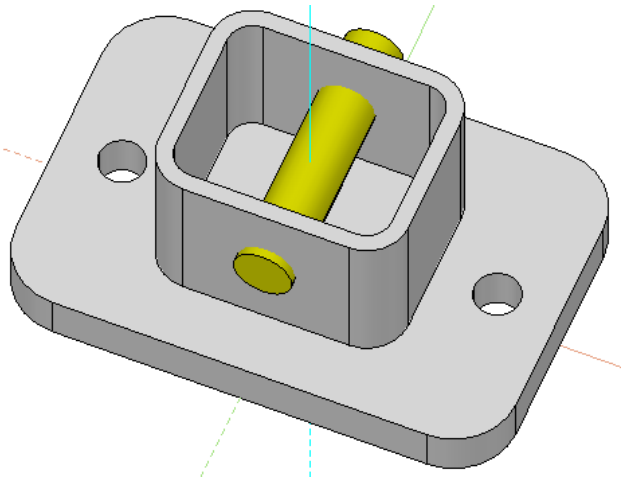
7. Define as BOX (*height 1* = **0**; *height 2* = **40**) and tap both contours.



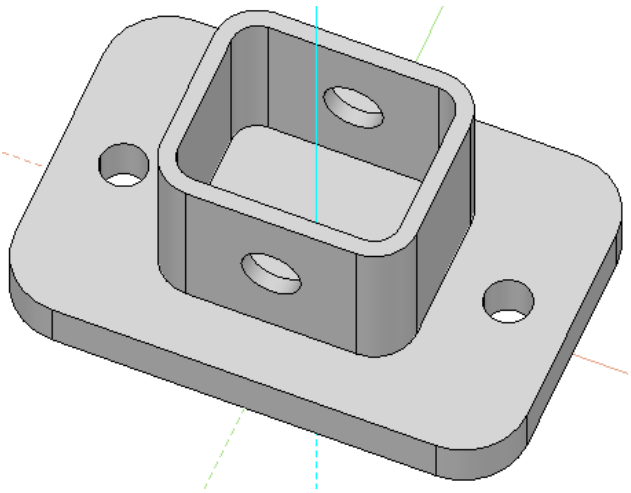
8. Select WORK PLANE ON SURFACE and tap the front of the profile 60x50) in the shaded model. (**P1**)



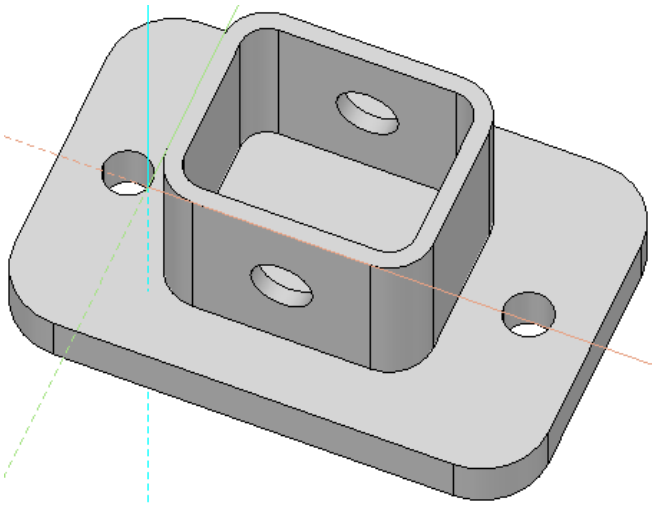
9. Draw the circle (radius 7mm) and define it as a BOX (*height 1* = **2**; *height 2* = **-62**).



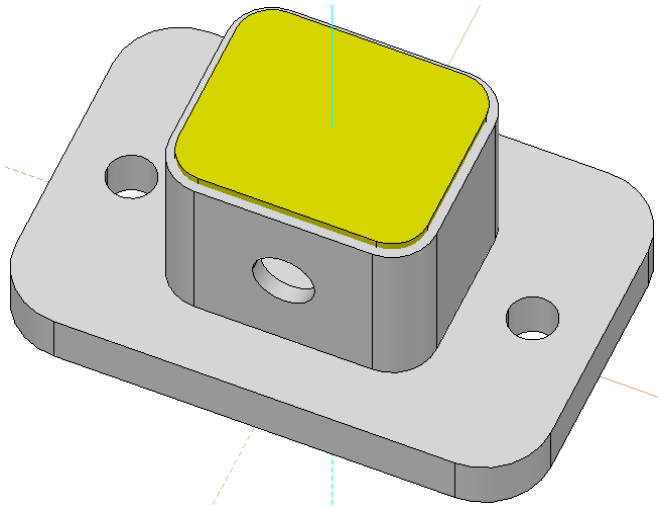
10. Perform the Boolean operation A MINUS B. (A = profile, B = bore).
ATTENTION: A-body remains.



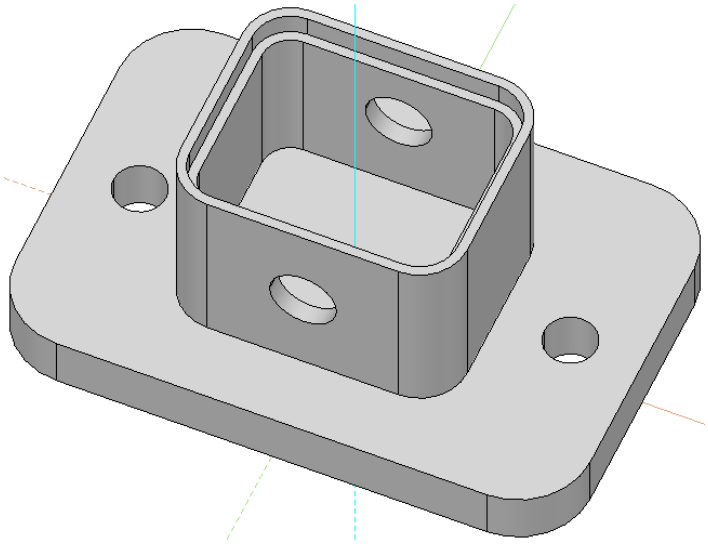
11. Place the working plane on the upper edge of the profile (60x50).



12. Draw the contour of the recess (56x46) and define it as a BOX (*height 1* = **-5**; *height 2* = **2**).
ATTENTION: Parts that are used for the Boolean calculation should always **OVERSIZE** the A-body!

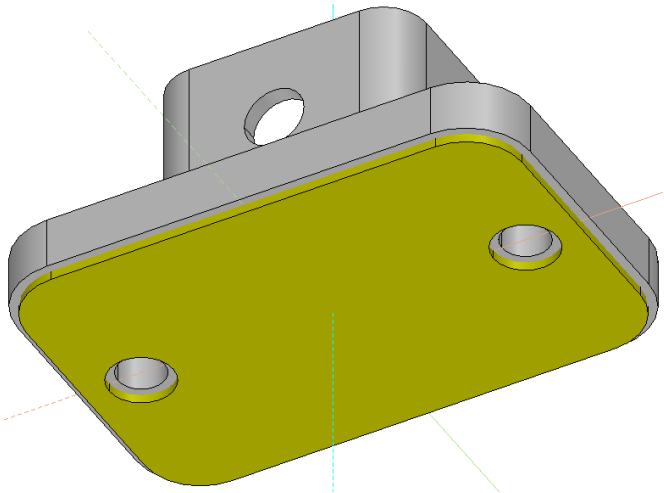


13. Perform Boolean operation A MINUS B.

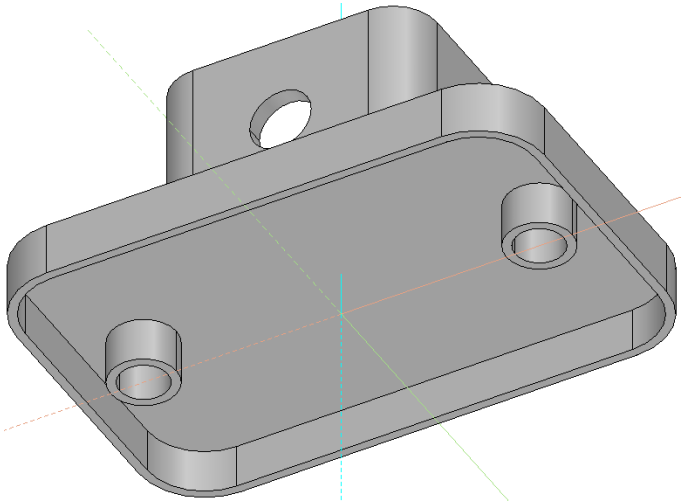


14. Reset the work plane. (RESET WORK PLANE)

15. Parallel contour (2mm INSIDE), draw the outer contour and hole and define it as a BOX (*height 1* = **-2**; *height 2* = **10**)

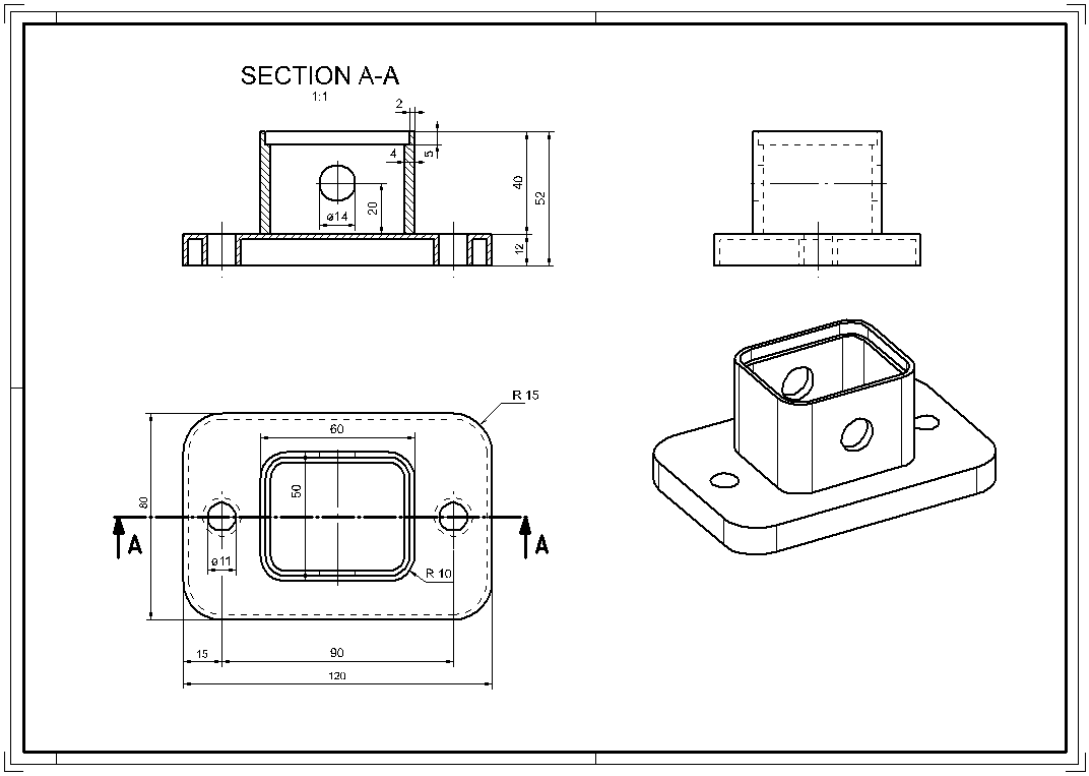


16. Perform Boolean operation A MINUS B.



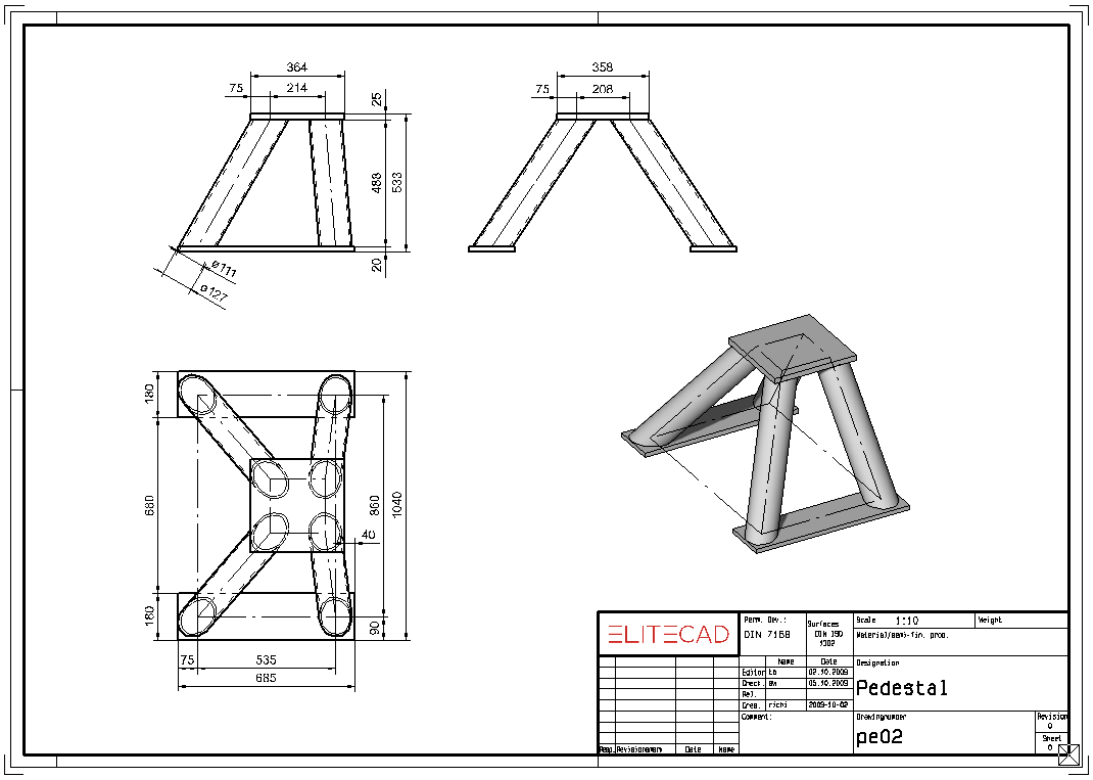
17. Save drawing temporarily.

18. Now create a new plot yourself, which should look similar to the illustration below. Then set dimensions for the top view and the section.



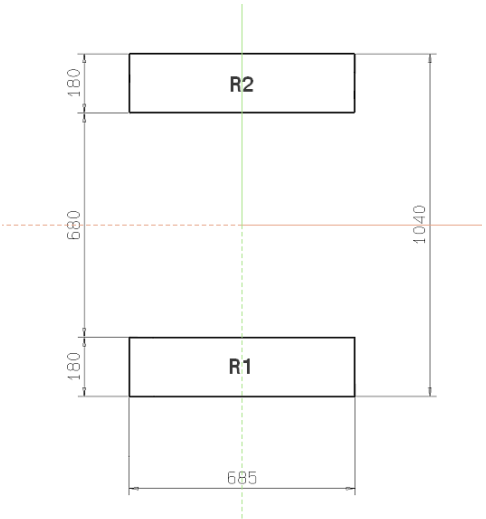
WORKSHOP END

EXAMPLE – PEDESTAL

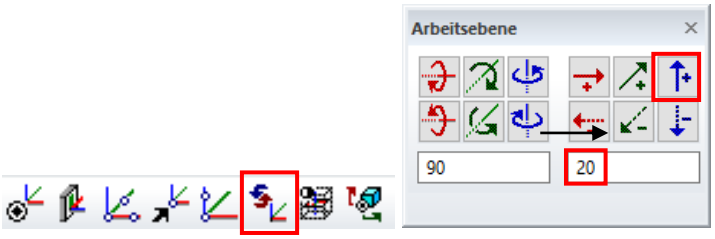


WORKSHOP

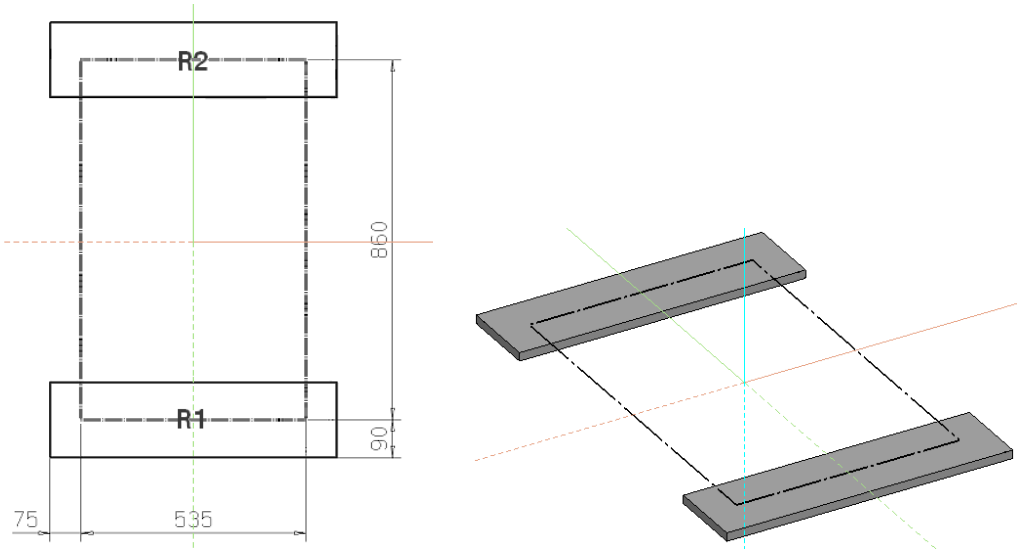
1. Start a new model  (e.g.: bk, plate, pedestal)
2. Draw a rectangle (685 x 180) and put it in the correct position. (**R1**).
3. Define the rectangle as a box with a height of **20**.
4. Copy the rectangle with COPY SELECTION at position **R2**.



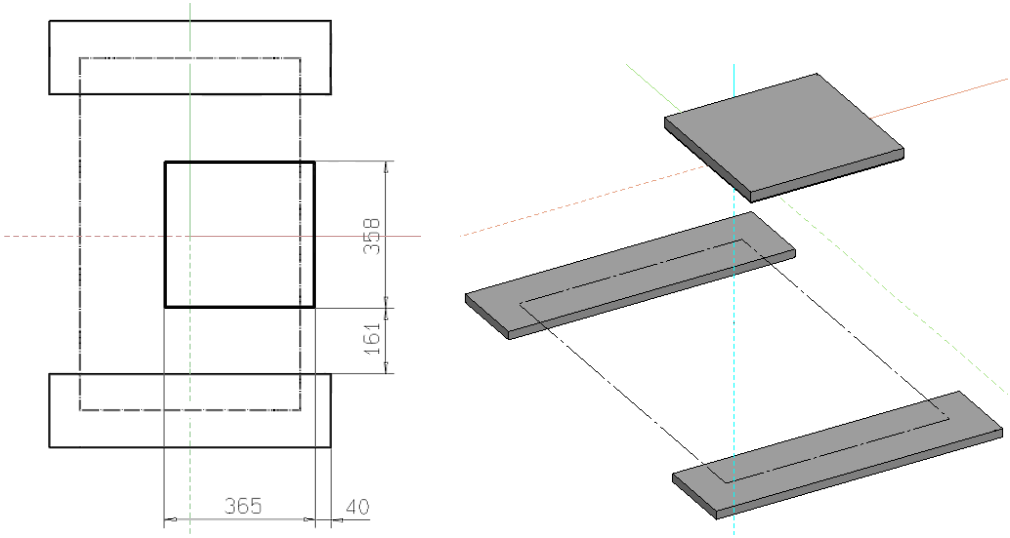
5. Move the work plane (WP) upwards by **20 mm**.



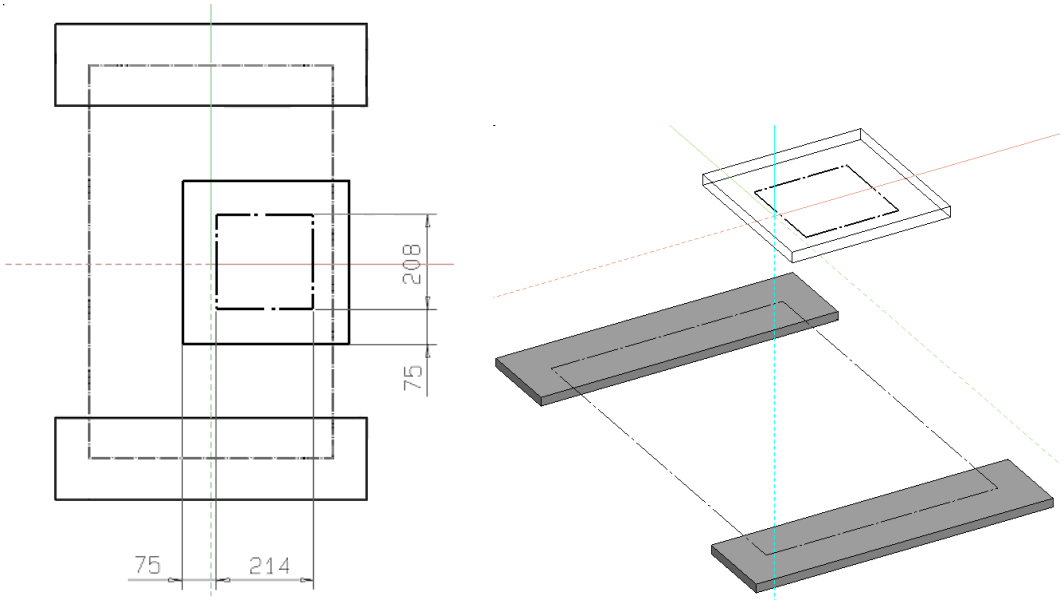
6. Draw the following auxiliary contour, this will be required for the later determination of the WP for the pipes.



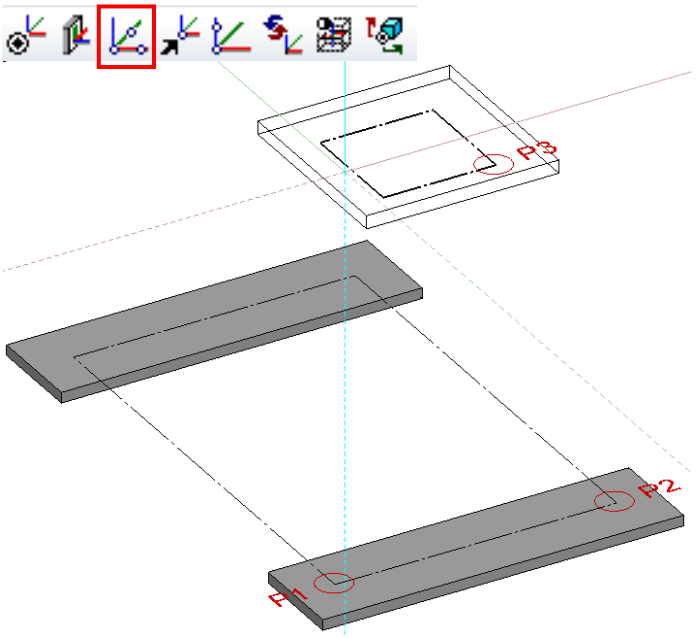
7. Set the working level up again by **488 mm**. Draw the top plate and then define it as a box (*height 1* = **0**; *height 2* = **25**).



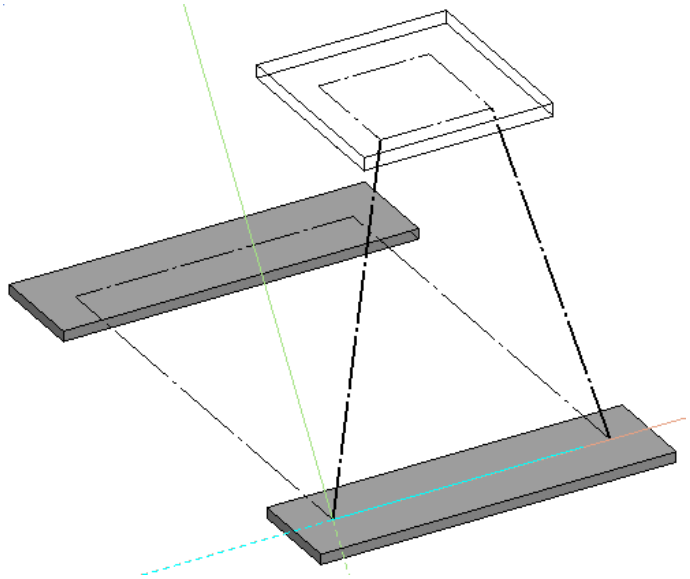
8. Draw the upper auxiliary contour, this will be required for the later determination of the WP for the pipes.



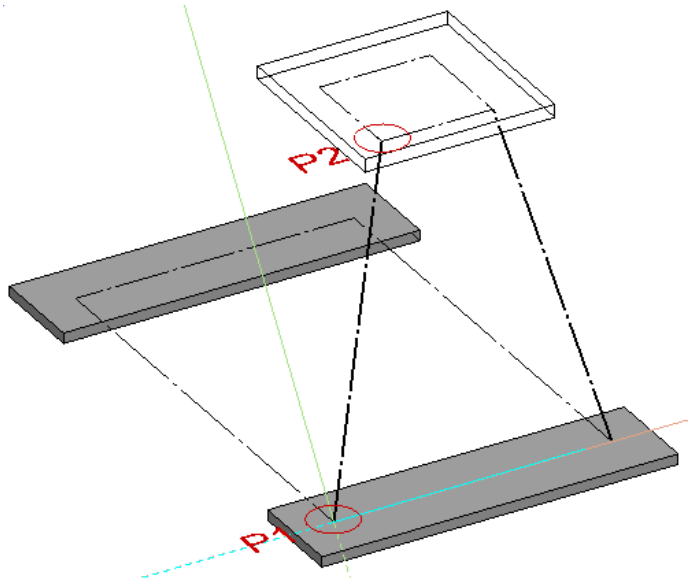
9. Select WORK PLANE BY 3 POINTS and use the snap function to catch the following three points.



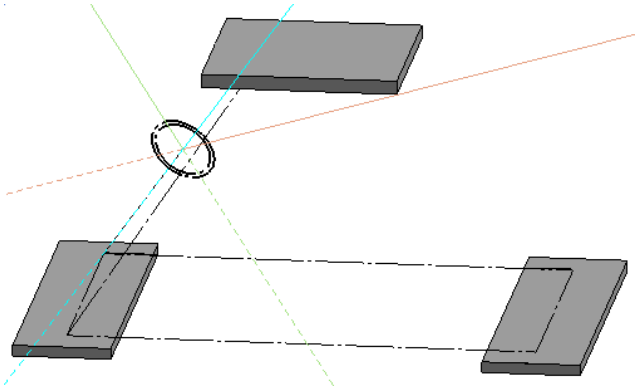
10. Now draw the centre lines for the pipes



11. Switch to WORK PLANE BY NORMAL and set it along the first pipe centre line by clicking on *point 1* and *point 2*.
Then use the SELECT WORK PLANE ORIGIN and the snap function "Centre of 2 points" to place the origin in the centre of the first centre line. Click again on *point 1* and *point 2*.

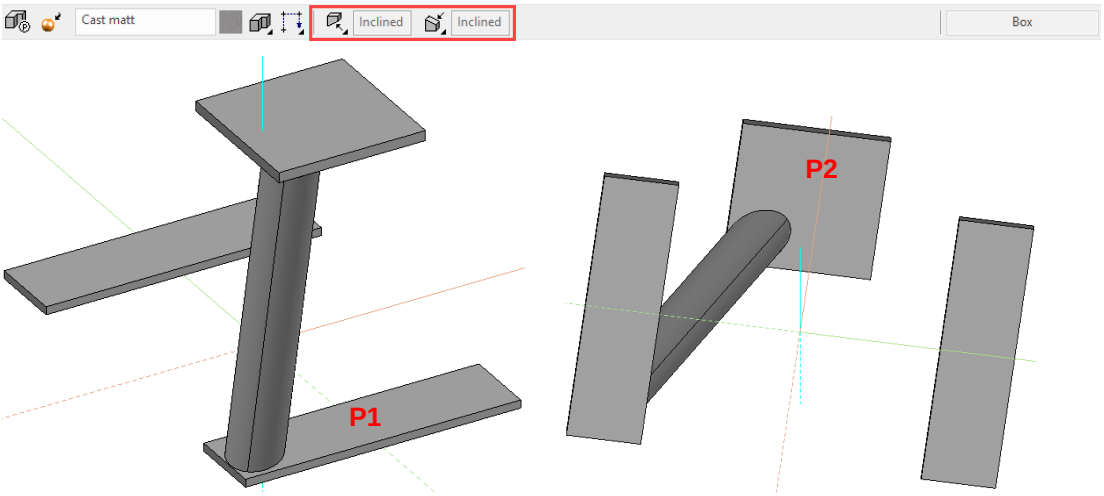


12. Now draw the pipe cross-section. *Circle 1* with **radius 55.5mm** and *circle 2* with **radius 63.5mm**.

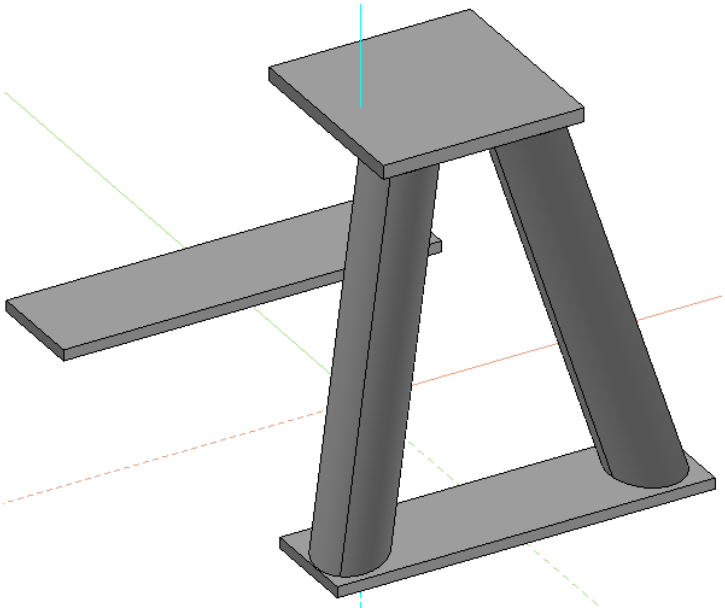


13. Define a box.
14. Activate the shaded model. (**[STRG]** + **D**)

15. Change from BASE PLAN DEFINED BY- HEIGHT to BASE PLAN DEFINED BY- SURFACE for the lower plane and click on the upper surface of the lower plate. (**P1**)
Now change from BASE PLAN DEFINED BY- HEIGHT to BASE PLAN DEFINED BY- SURFACE for the upper plane and click on the lower surface of the upper plate. (**P2**)

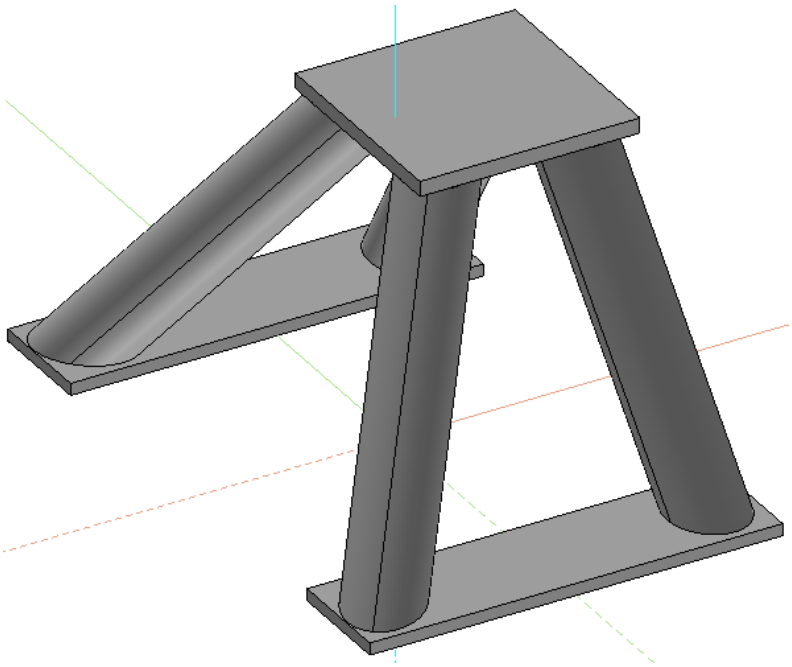


16. Now repeat points 11 to 15 again for the second pipe.

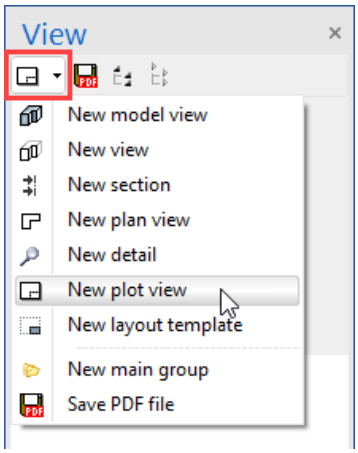


17. Reset the work plane to the origin.

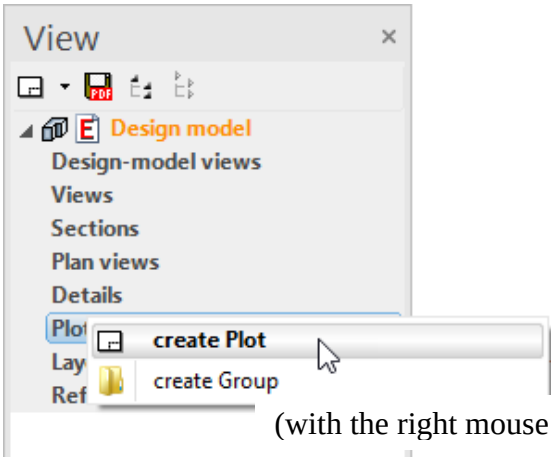
18. Now mirror the two tubes.



19. Create a new plot and then enter the desired plot name. (e.g .: **example 08_pl**)

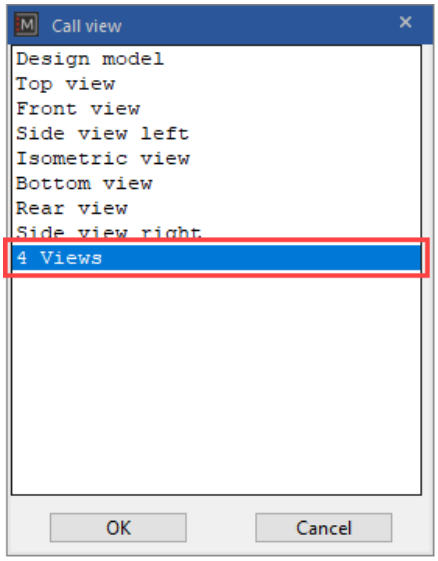


oder

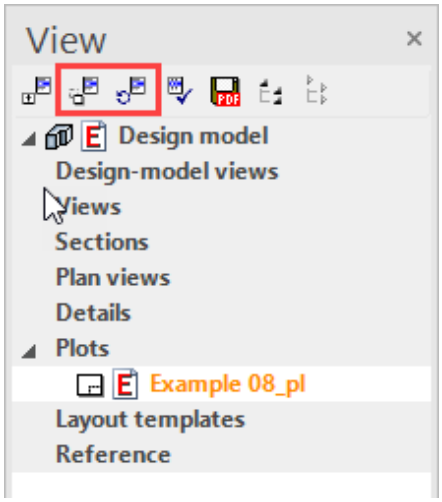


(with the right mouse)

20. Now click on INSERT VIEW and select 4 VIEWS.



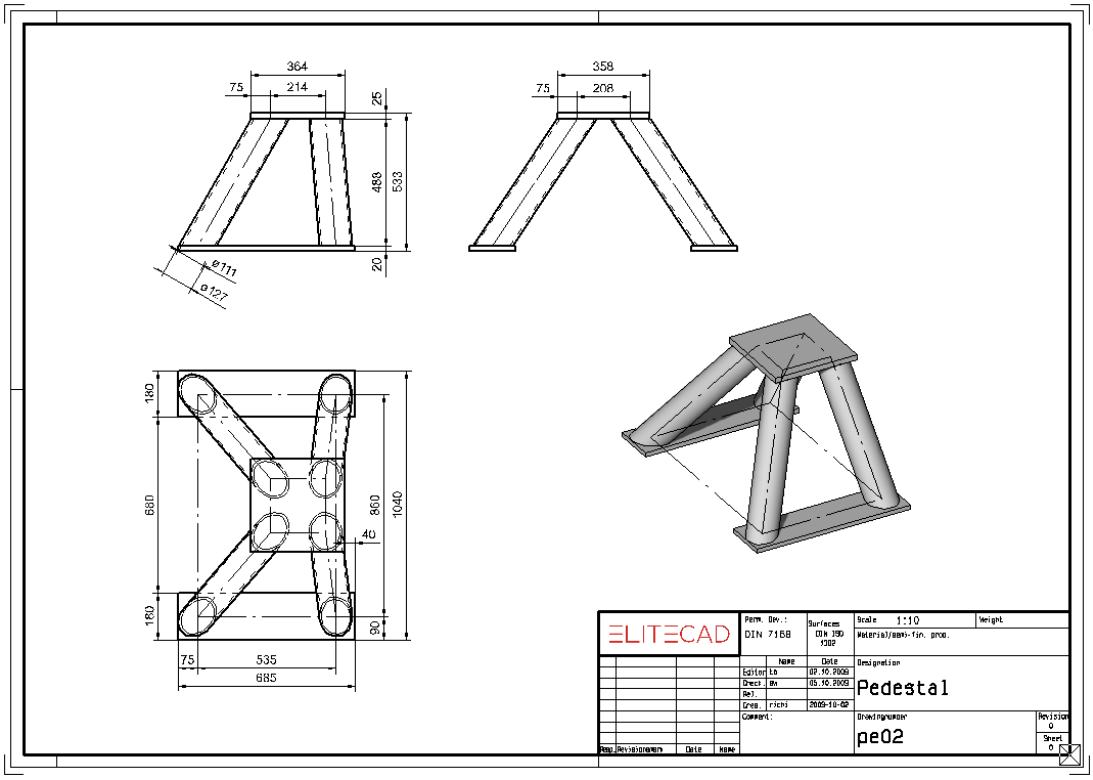
21. The views can now be positioned using MOVE VIEW, ROTATE VIEW.



TIPP

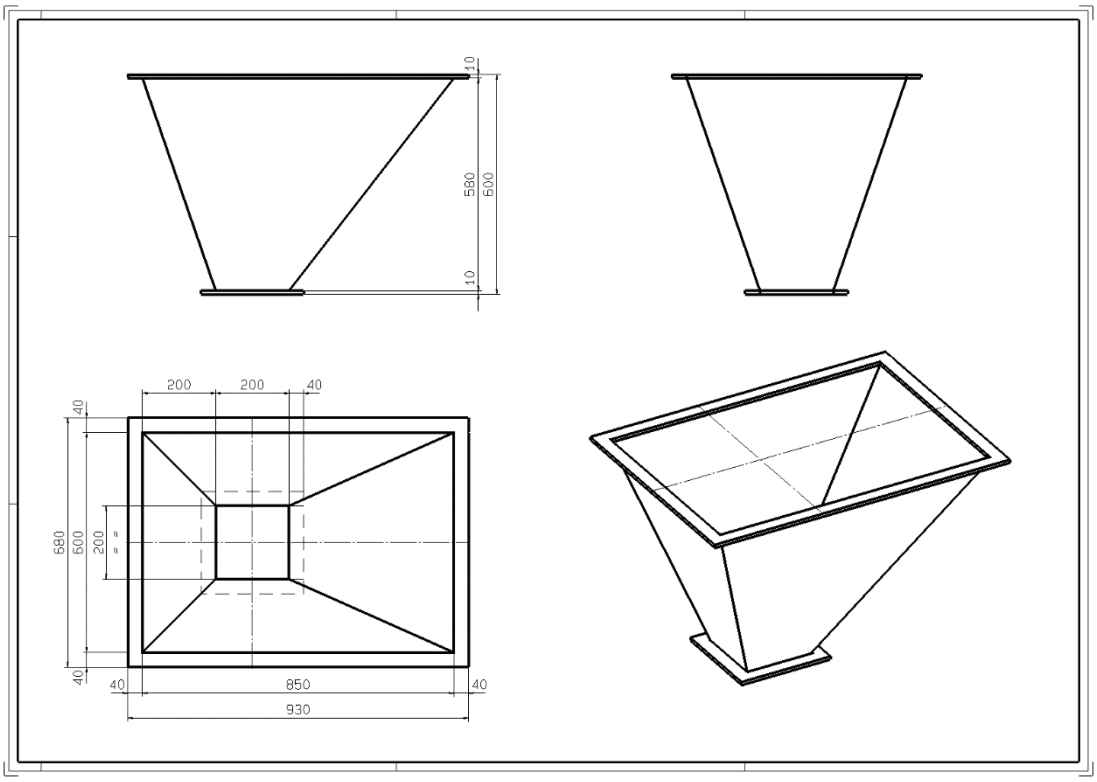
The individual views can also be positioned with MOVE SELECTION and ROTATE SELECTION.

22. Set the main dimensions in the three views.



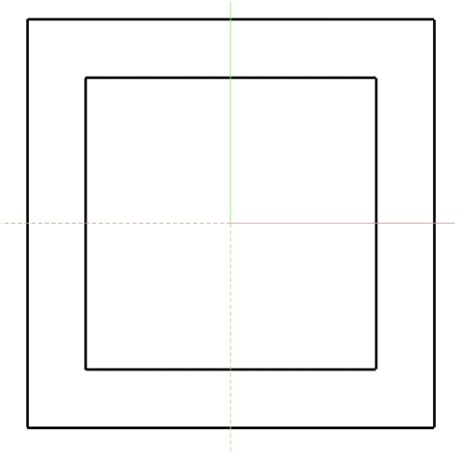
WORKSHOP END

EXAMPLE – SHOUT

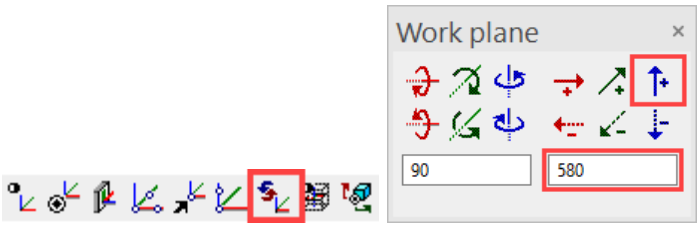


WORKSHOP

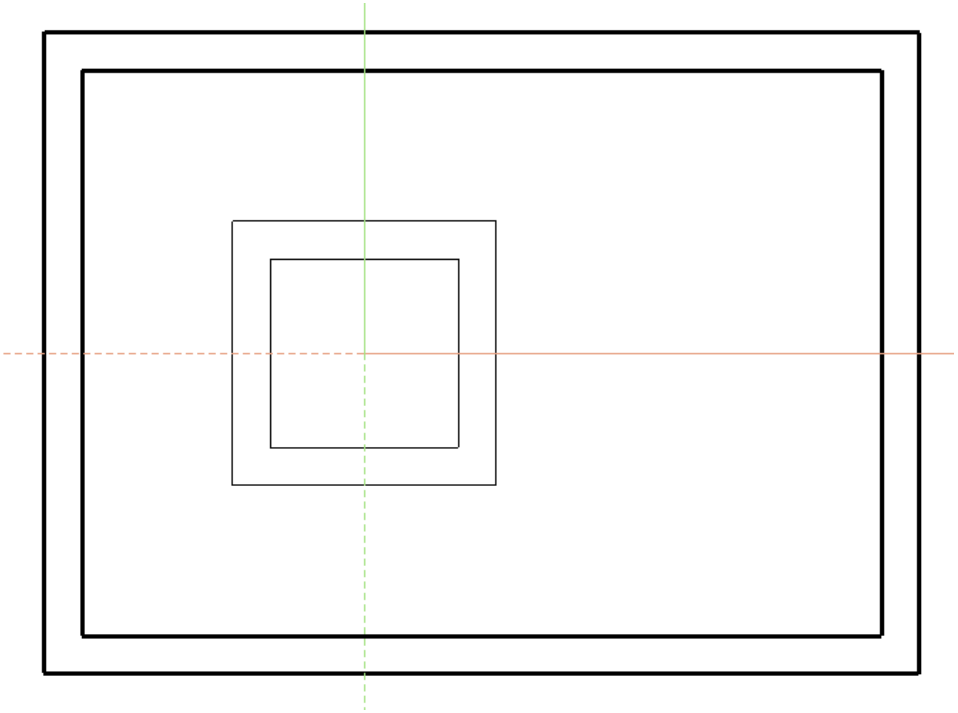
1. Start NEW MODEL (e.g.: "bk, flange1, shout")
2. Draw two rectangles [(200 x 200) and (280 x 280)] and put them in the correct position. Define a box. (*height 1* = **-10**, *height 2* = **0**)



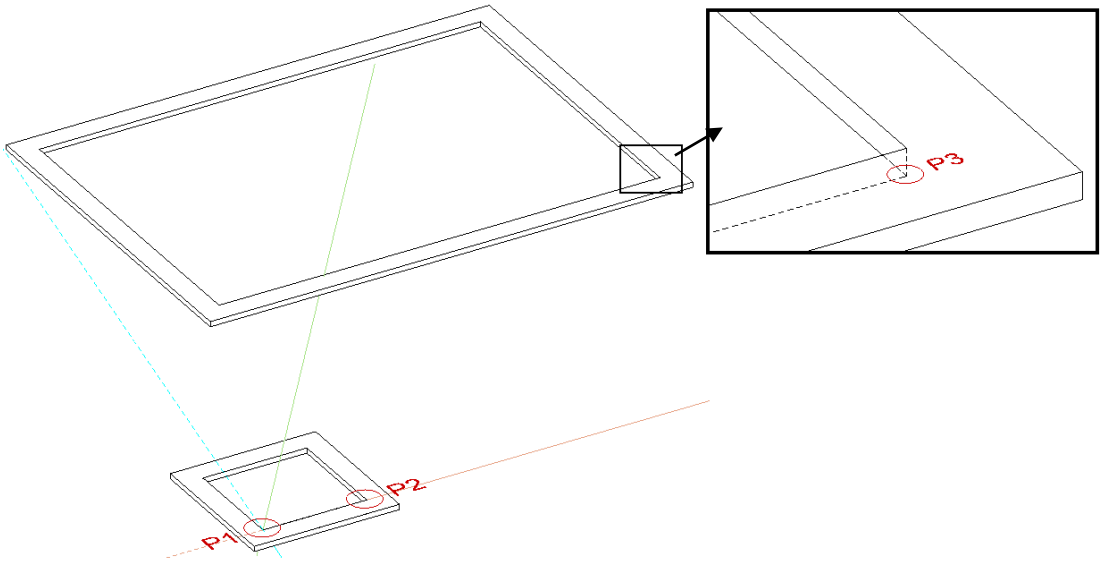
3. Move the working plane **580mm** upwards. (CHANGE WORK PLANE)



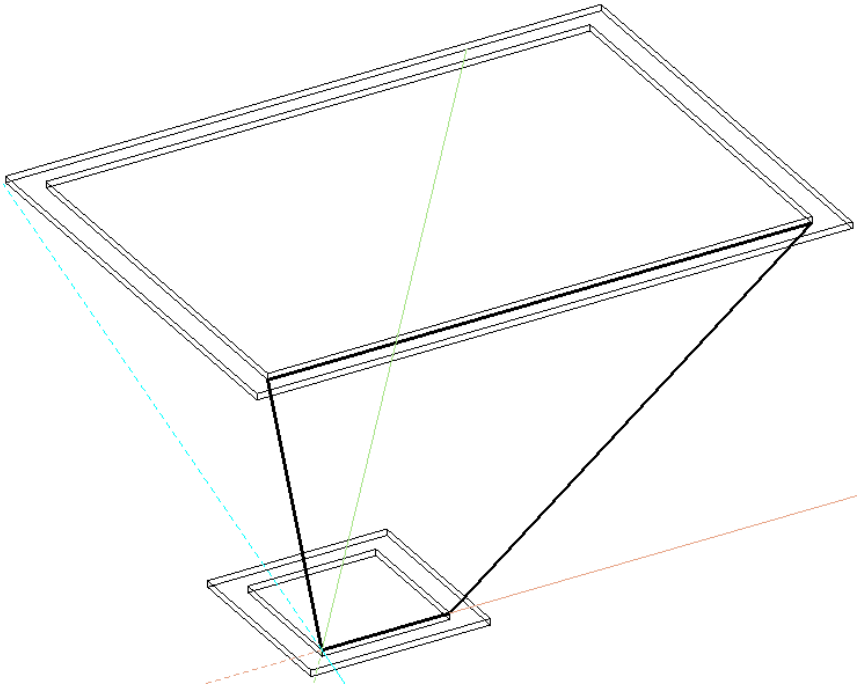
4. Start NEW MODEL (z.B. „bk, flange2, shout“) and draw two more rectangles [(850 x 600) and (930 x 680)]. Define a box (*height 1* = **0**; *height 2* = **10**)



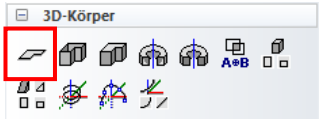
5. Select WORK PLANE BY 3 POINTS and snap to the three points using the snap function.

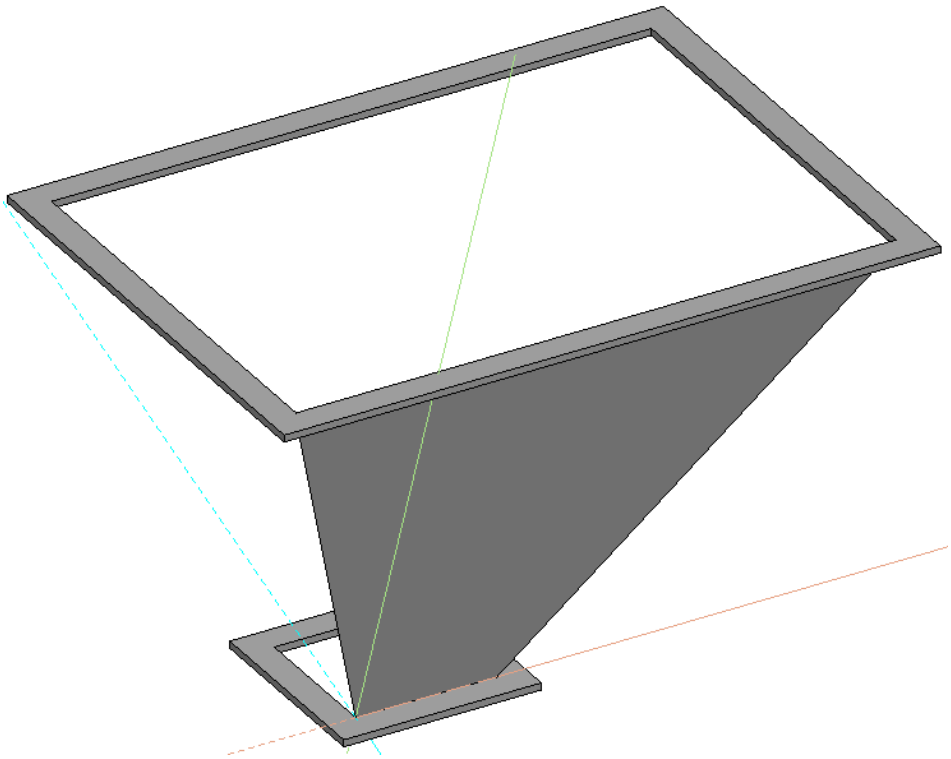


6. Start NEW MODEL (e.g. name: „sheet metal 1“) and draw the first side sheet metal. (must be a closed contour!).



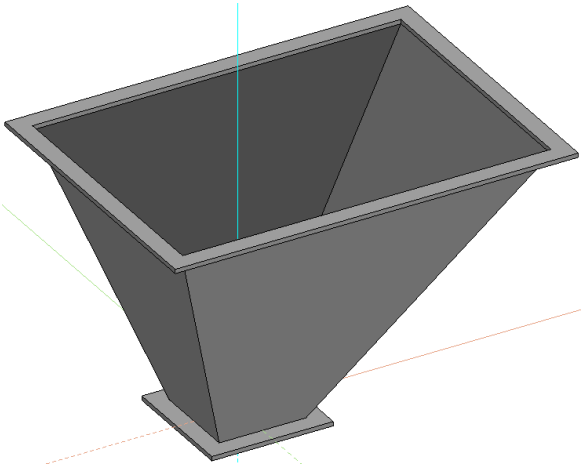
7. Select DEFINE PLANE. (*height* = **0**) / (*colour* = **12**)



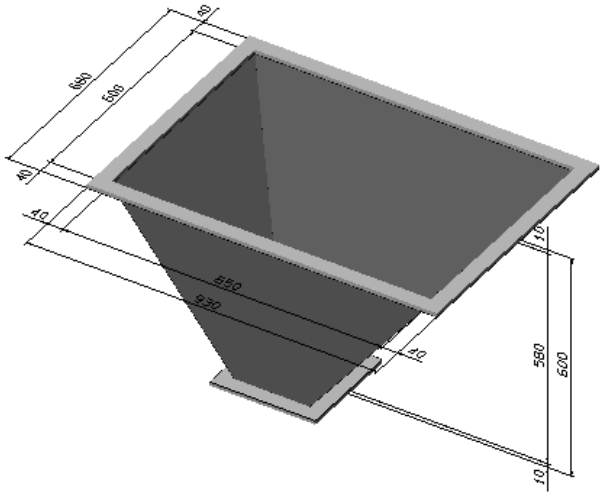


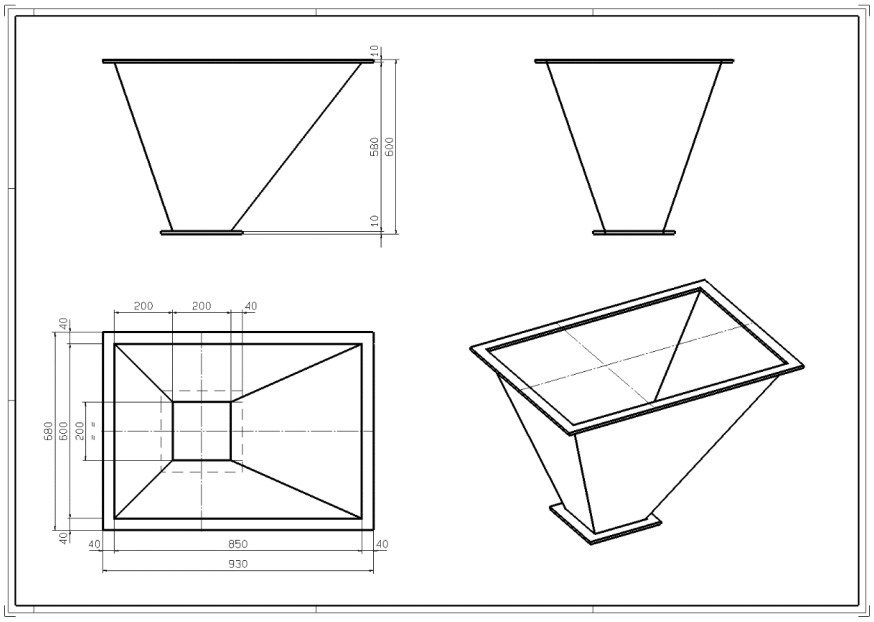
8. This plane can now be mirrored on the opposite side using the MIRROR SELECTION. (first reset the WP). Or repeat points 5 - 7 for the second side.

9. Repeat points 5 - 7 for the third and fourth sides.



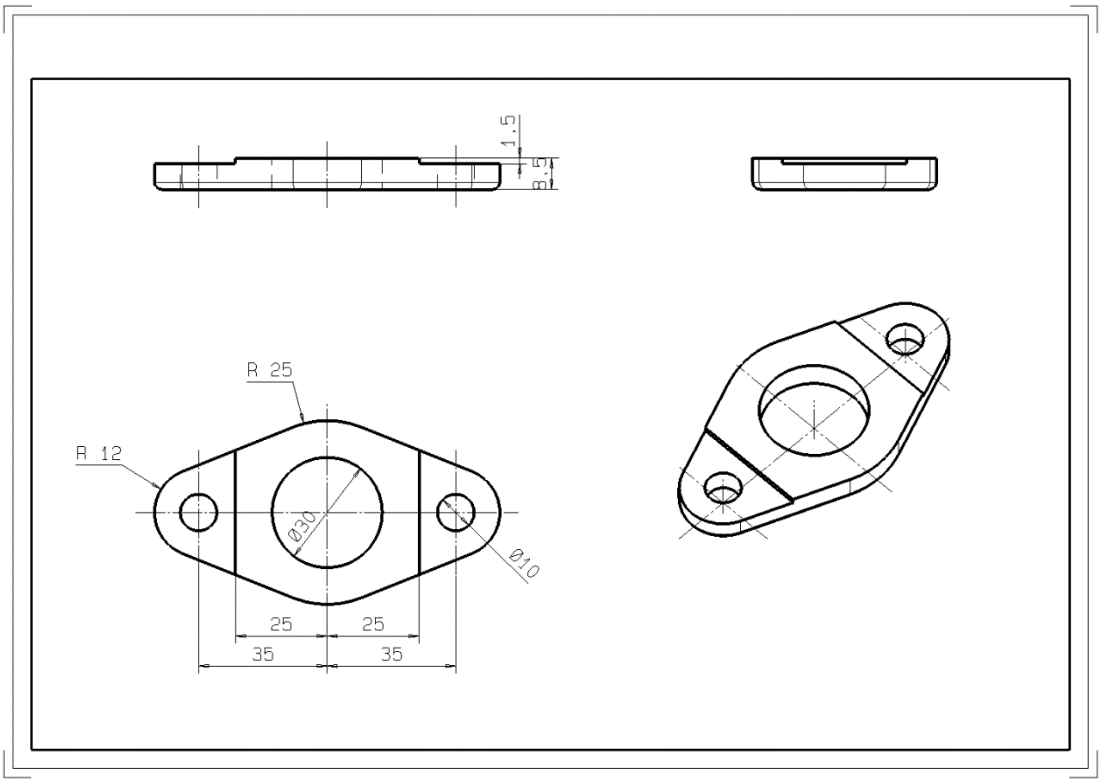
10. Set all main dimensions and then derive the plot (see example PEDESTAL, from point 19).





WORKSHOP END

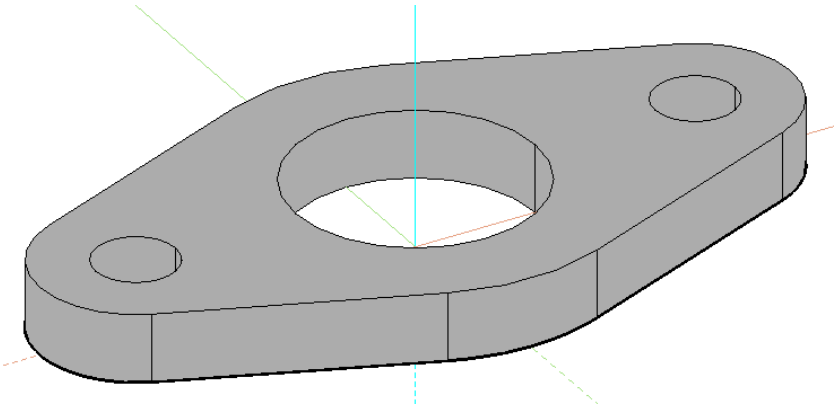
EXAMPLE – FLANGE



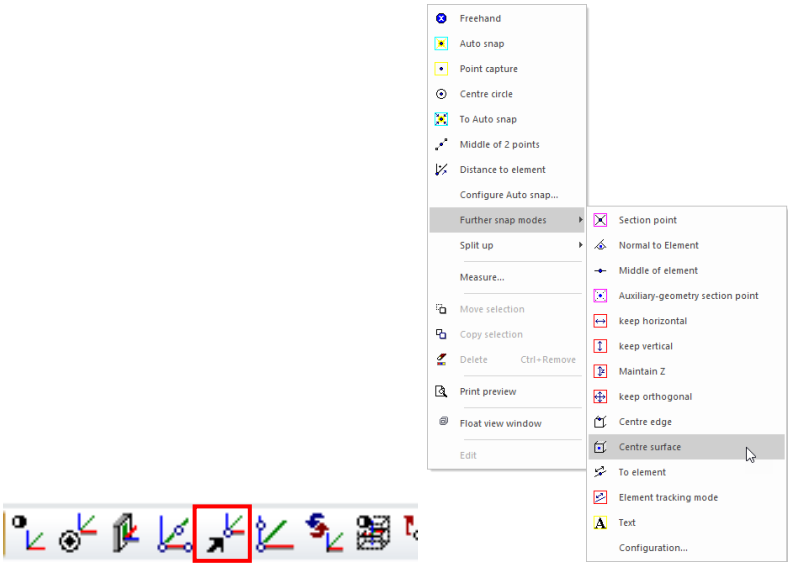
WORKSHOP

1. Start NEW MODEL (e.g.: „bk, flange, bend“)

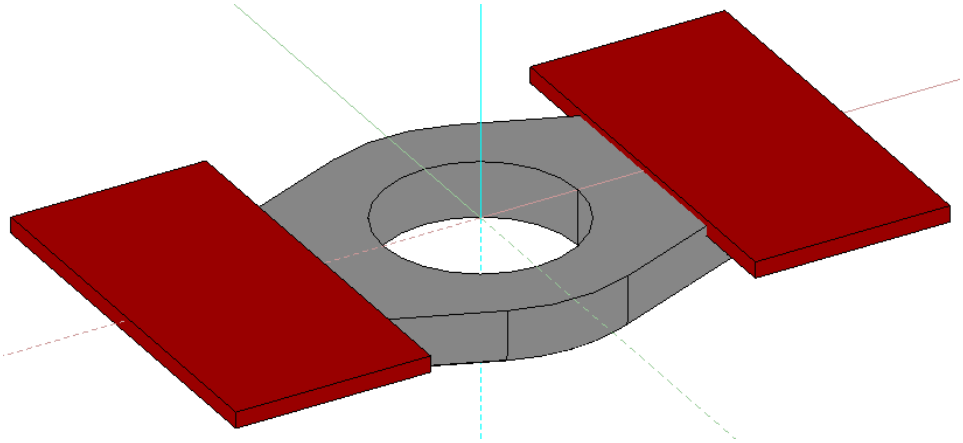
2. Draw the outline of the top view and then define it as a box. (*height 1* = **0**; *height 2* = **8.5**)



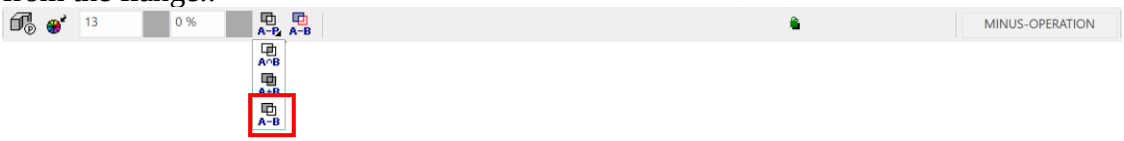
3. Move the work plane up by 8.5mm. Select WORK PLANE ORIGIN, then CENTER SURFACE (RIGHT-CLICK > FURTHER SNAP MODES), now click on the upper surface of the top view contour.

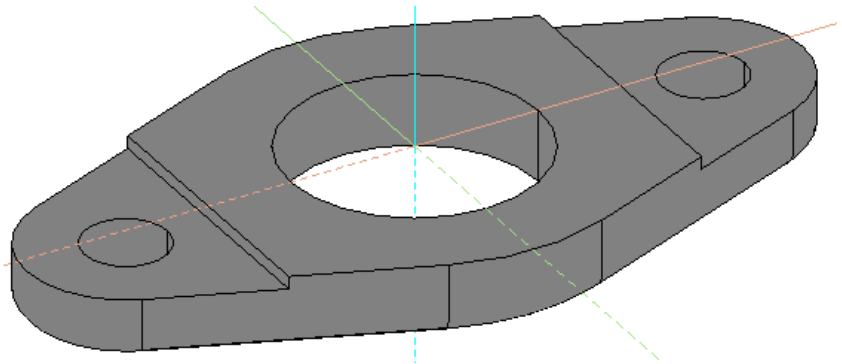


4. Start NEW MODEL (z.B.: „w1, flange“). Then draw the tool contour and define it as a box. (*height 1* = **1**; *height 2* = **-1.5**)



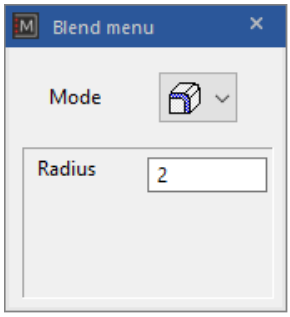
5.Perform the Boolean operation, select A minus B and subtract the tool contour from the flange.:



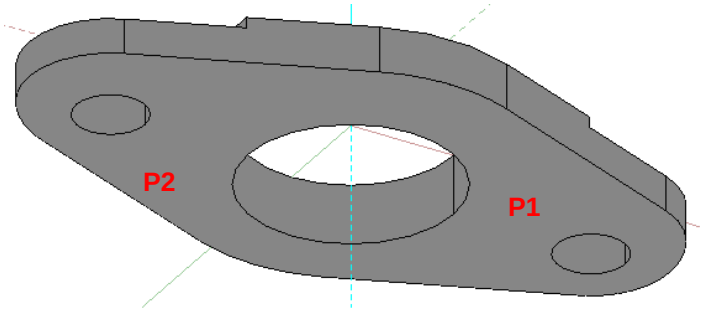


6. Select «EDGE BLEND».

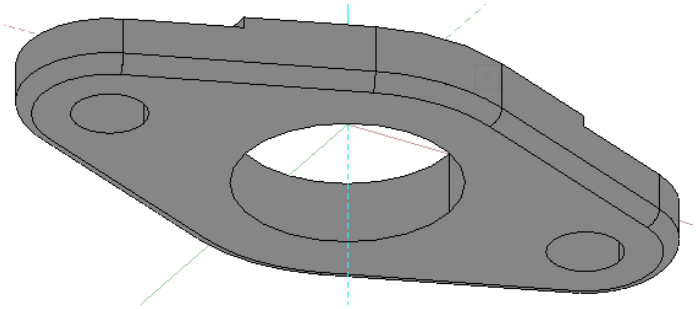
Please select definition
Touch the flange within the 3D body. (**P1**)



Choose a radius of 2mm.



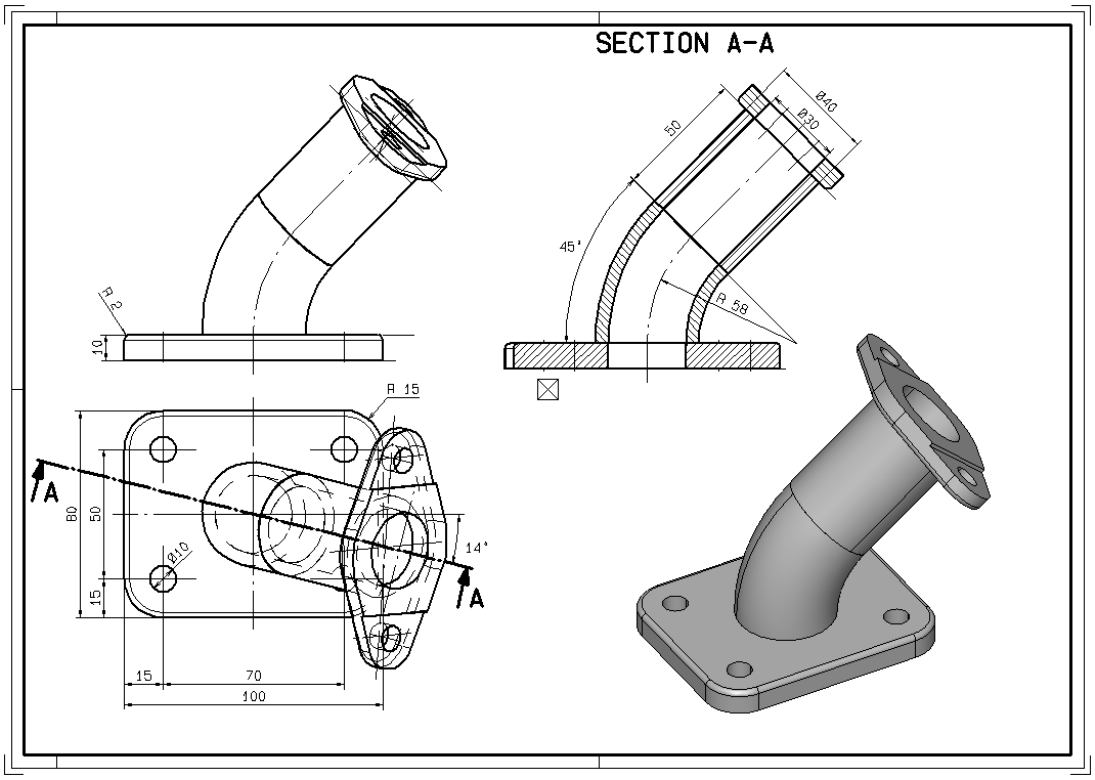
Please make selection [ENTER -> End]
Touch the desired edge (within the 3D body). (**P2**) and the selected edge is marked.
This selection? Y
Confirm (only if the desired edge has been marked). Otherwise you can go back with "N" (No). If the desired edge has been selected and you do not want to select any further edges for rounding, confirm with [**Enter**] and the rounding is created.



7. Save the drawing under "**flange_k.d**" for later use.

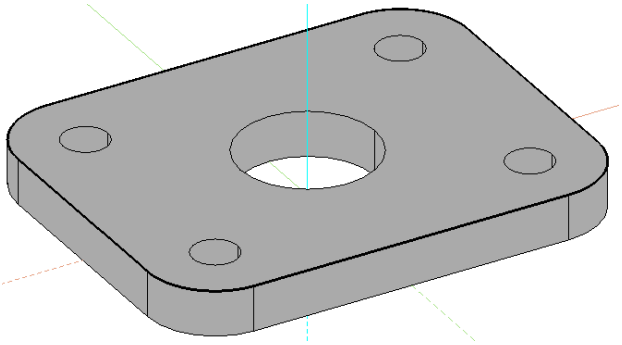
WORKSHOP END

EXAMPLE – BEND



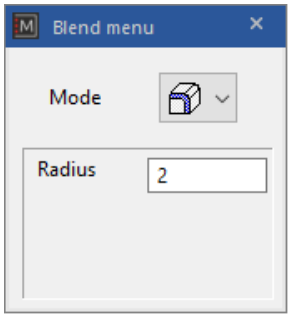
WORKSHOP

1. Start NEW MODEL  (e.g.: „bk, plate, bend“)
2. Draw the plate contour (including circles for holes) and define it as a box.
(*height 1* = **0**; *height 2* = **10**)

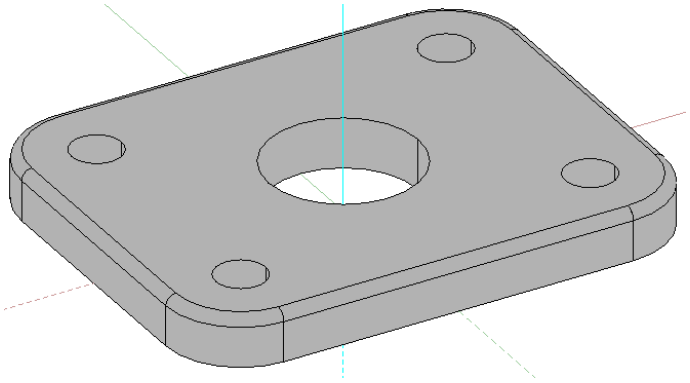


3. Select „EDGE BLEND“.

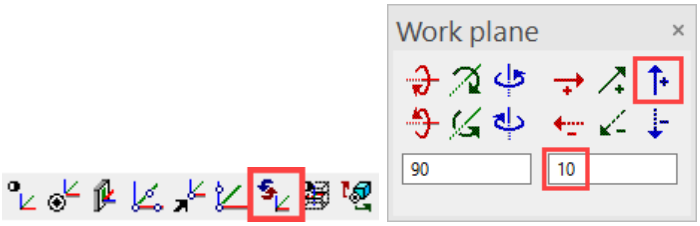
Please select definition
Click on the top of the panel, near the outside edge.



Fillet the lower edge with a radius of **2mm**.



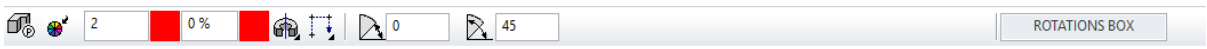
4. Move the work plane upwards by **10mm**. (*CHANGE WORK PLANE*)



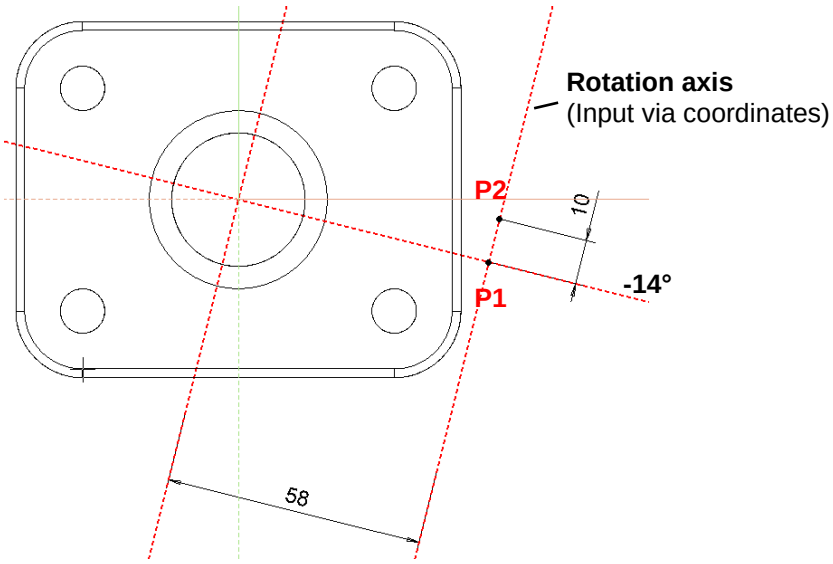
5. Start NEW MODEL  (z.B.: „bk, pipe1, bend“).

6. Draw two circles with a radius of **15mm** and **20mm** in the centre (origin) and then define them as a rotation box. (DEFINE ROTATION BOX)

Set the *start angle* to **0** and the *end angle* to **45**.



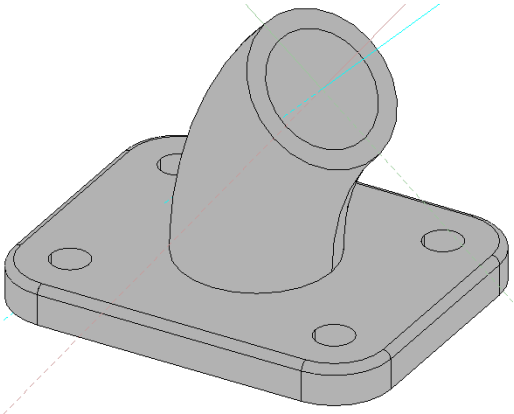
Please click point 1 of rotation axis for new contour.
input **a-14,d58**. (**P1**)
Please click point 2 of rotation axis for new contour
input **a76,d10**. (**P2**)



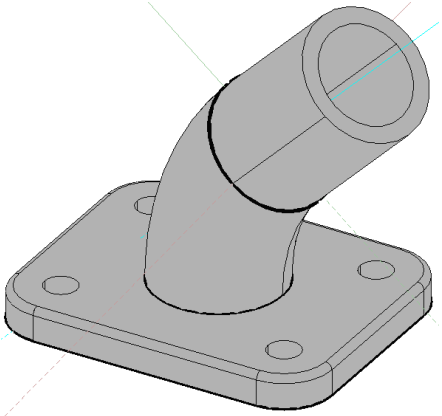
TIP

The axis of rotation can also be indicated by pre-drawn geometry auxiliary lines.

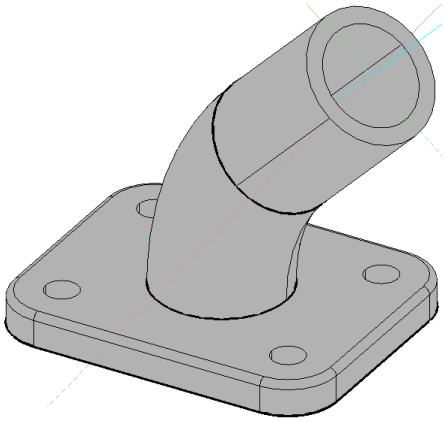
7. Select WORK PLANE ON SURFACE and tap the top surface of the pipe. Then SELECT WORK PLANE ORIGIN and use the CIRCLE CENTER snap mode to set the origin in the pipe centre.



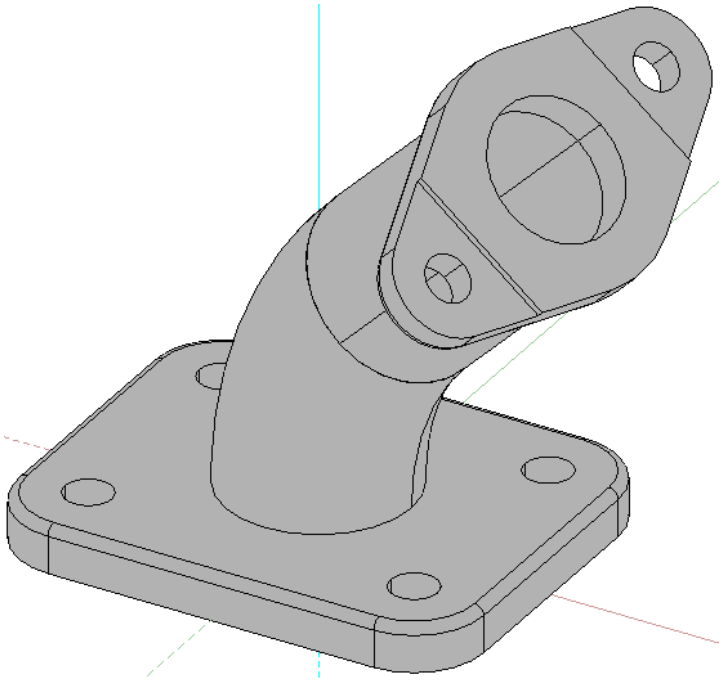
8. Start NEW MODEL  (z.B.: „bk, pipe2, bend“).
9. Draw two circles with a radius of **15mm** and **20mm** in the centre (origin) and then define them as a box. (DEFINE BOX *height 1* = **0**; *height* 2 = **50**)



10. Select SELECT WORK PLANE ORIGIN and set the origin in the upper pipe centre using the CIRCLE CENTER snap mode.



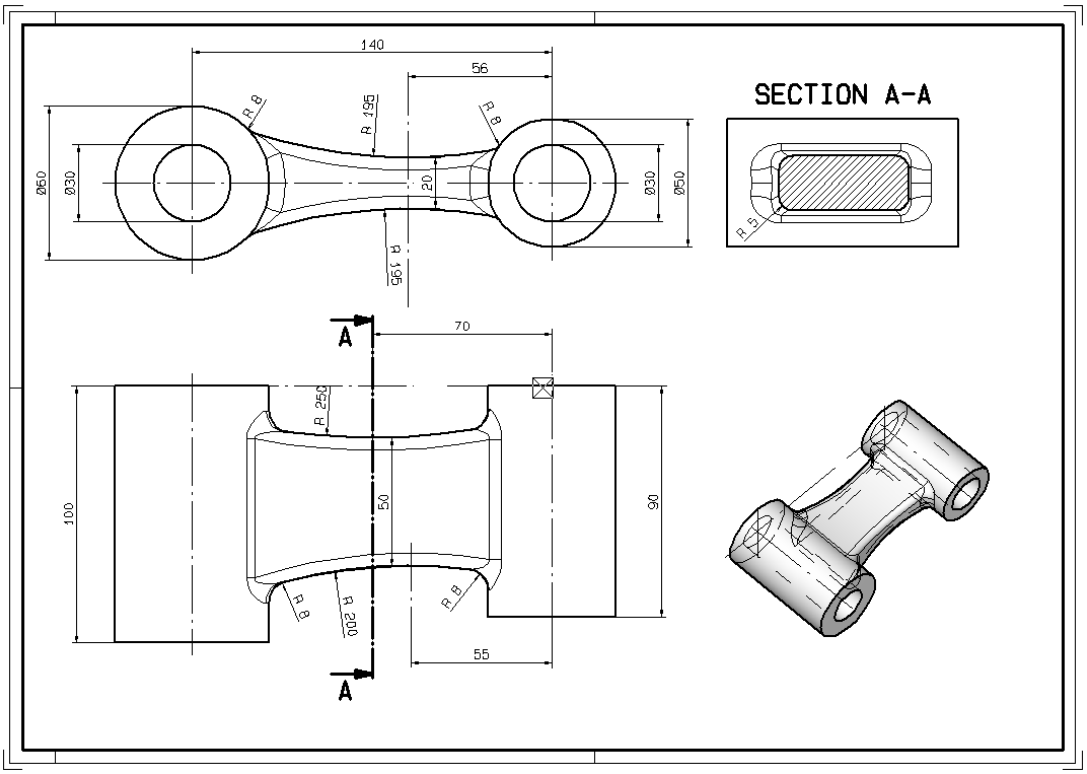
10. Use INSERT and POSITION… to load the previously created drawing **"flansch_k.d"** and position it at the origin.
(**[CTRL] + 1**) = (X0, Y0)



The entire assembly can now be saved.

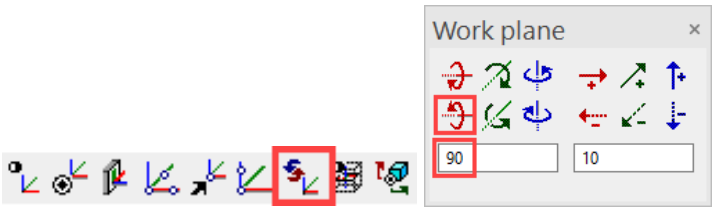
WORKSHOP END

EXAMPLE – BRACKET

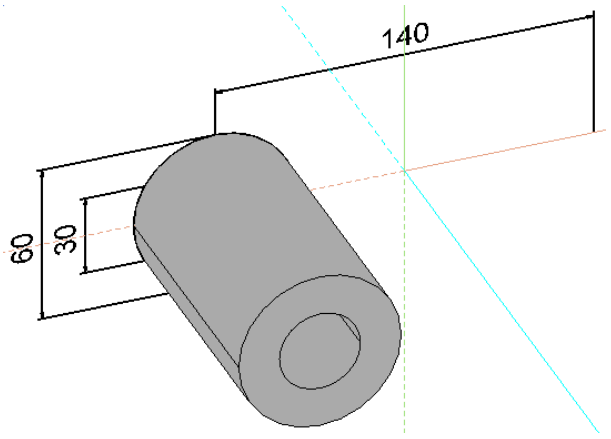


WORKSHOP

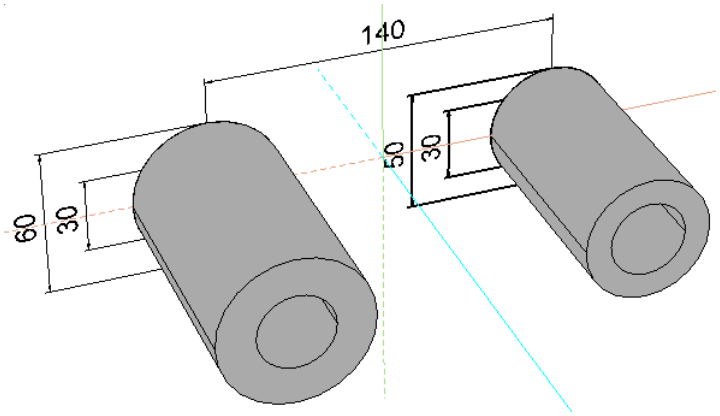
1. Rotate the work plane in the X-axis by 90 °.



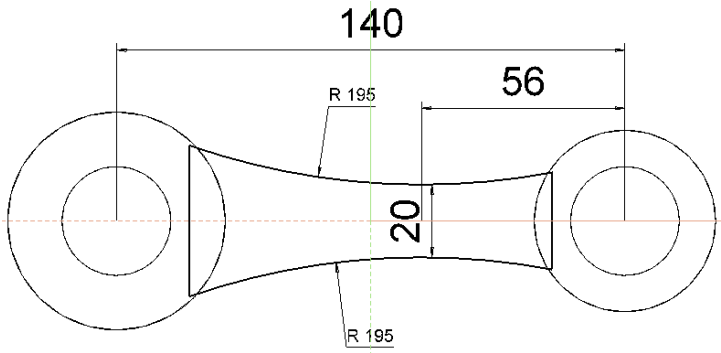
2. Start NEW MODEL (e.g.: „bk1, bracket“)
3. Draw the first pipe contour. Then define this as a box. (*height 1* = **0**; *height 2* = **100**)



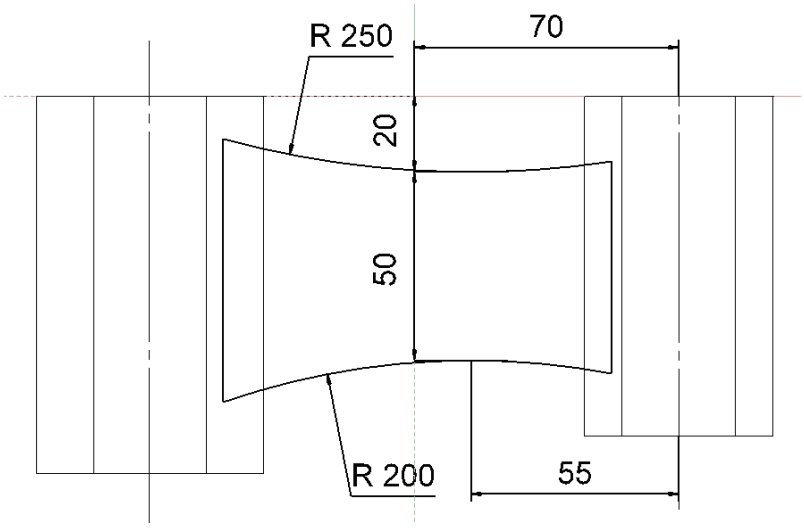
4. Start NEW MODEL (z.B.: „bk2, bracket“)
5. Draw the second pipe contour. Define this as a box as well.
(*height 1* = **0**; *height 2* = **90**)



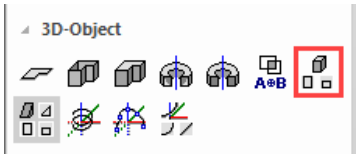
6. Start NEW MODEL (e.g.: „bk3, bracket“)
7. Draw the first contour of the body. The contour should protrude into the two tubes.

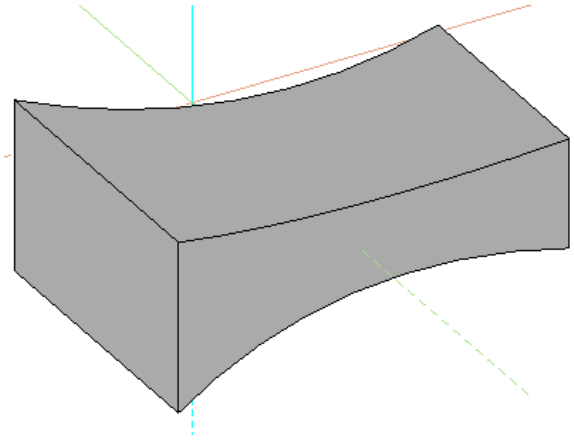


8. Reset the work surface (RESET WORK PLANE) and draw the second body contour. The contour should protrude into the two tubes.

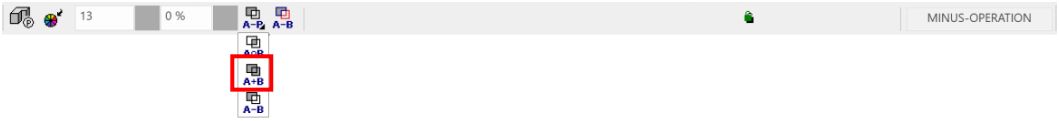


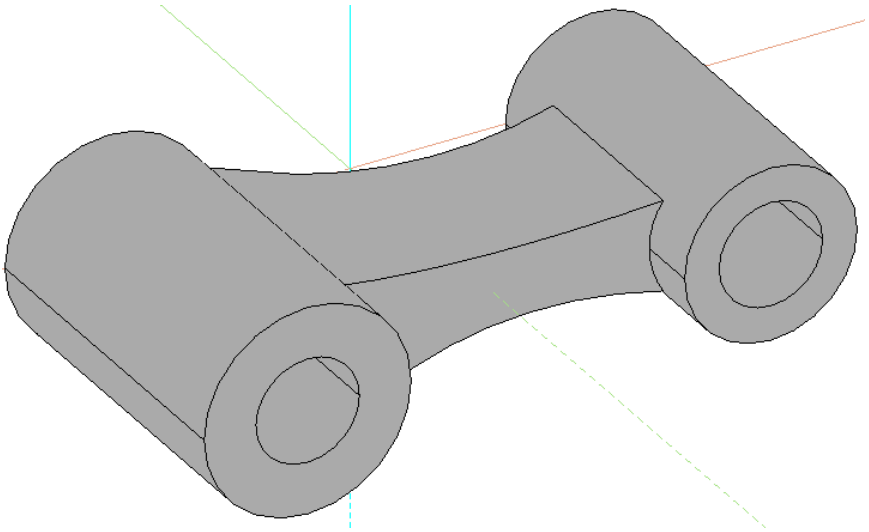
9. Select DEFINE 2-VIEWS OBJECT and click on the two outlines drawn previously. The following 3D body is generated!



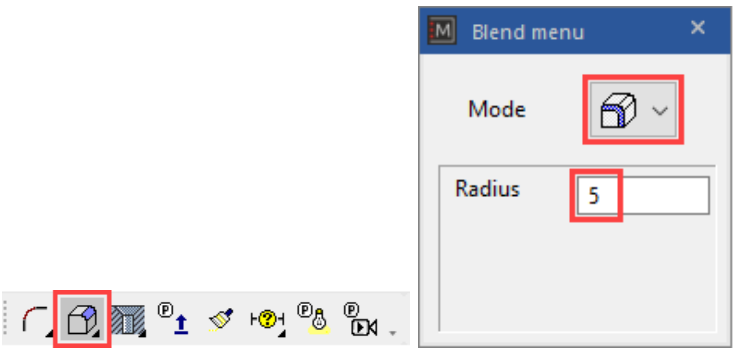


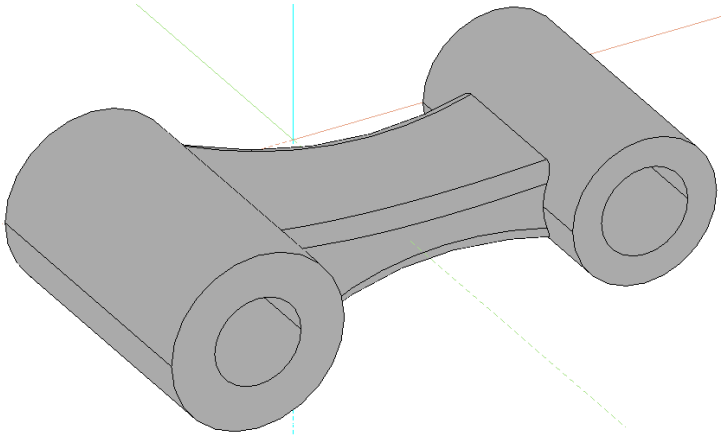
9. Union the three models with the Boolean calculation UNION.



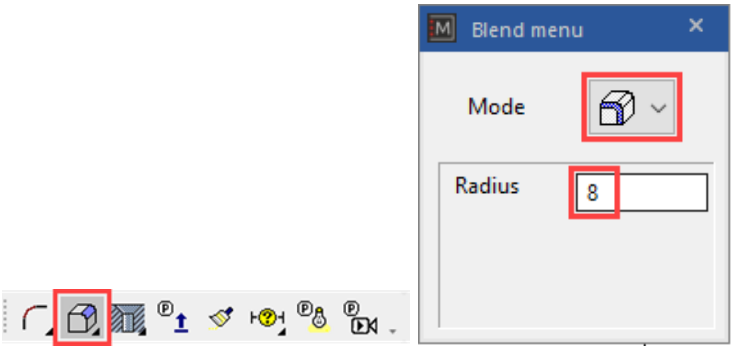


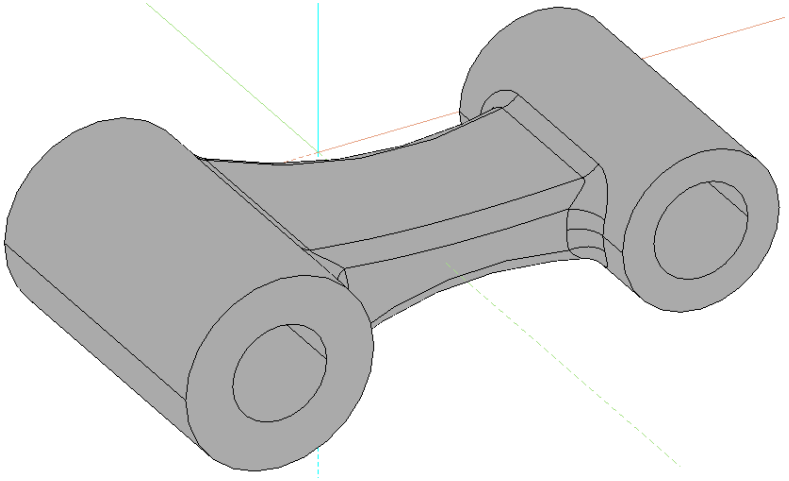
11. Select EDGE BLEND, click on the body. Select the BLEND SINGLE EDGE mode, a radius of **5mm** and round the 4 long edges.



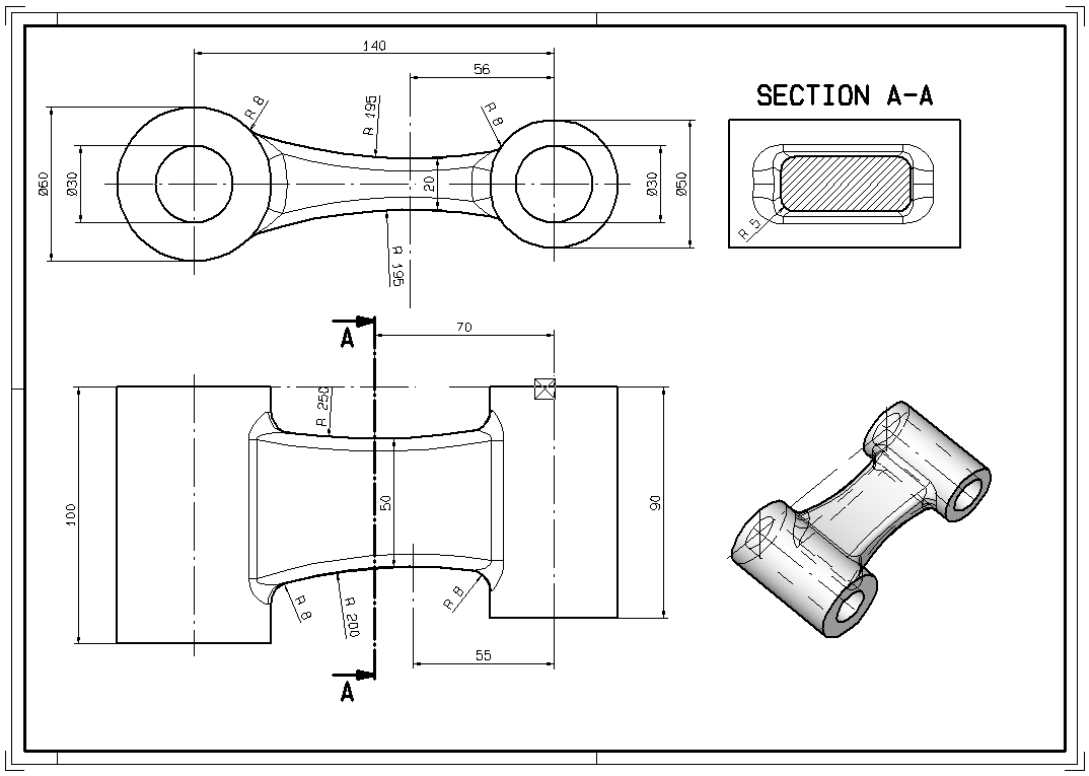


12. Select EDGE BEND, click on the body. Select the mode FILLET SUCCESSIVE EDGES, a radius of **8mm** and round the two tubes with the middle part.





13. Set the desired dimension and create a new plot.



WORKSHOP END

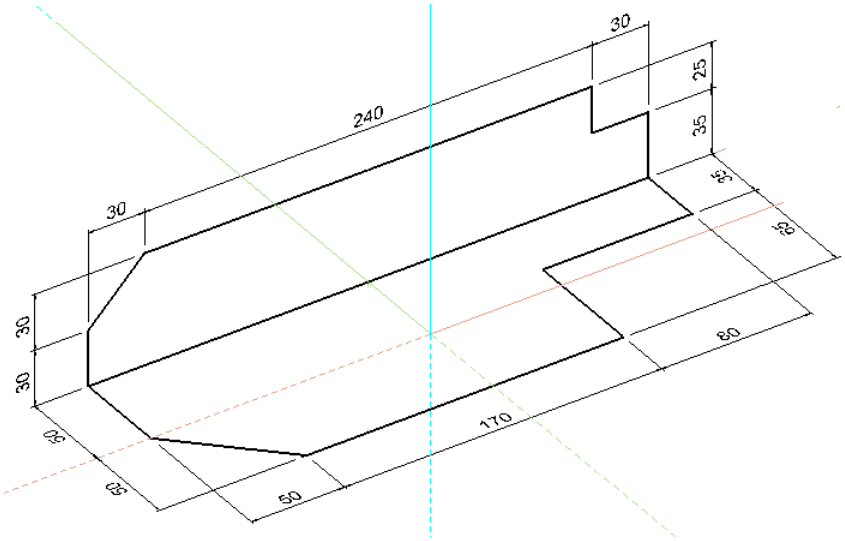
EXAMPLE – 2-VIEWS OBJECT

TIP

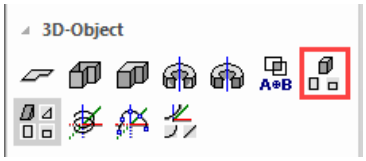
With 2-view objects (2VO), the contours must be closed and drawn in different work planes.

WORKSHOP

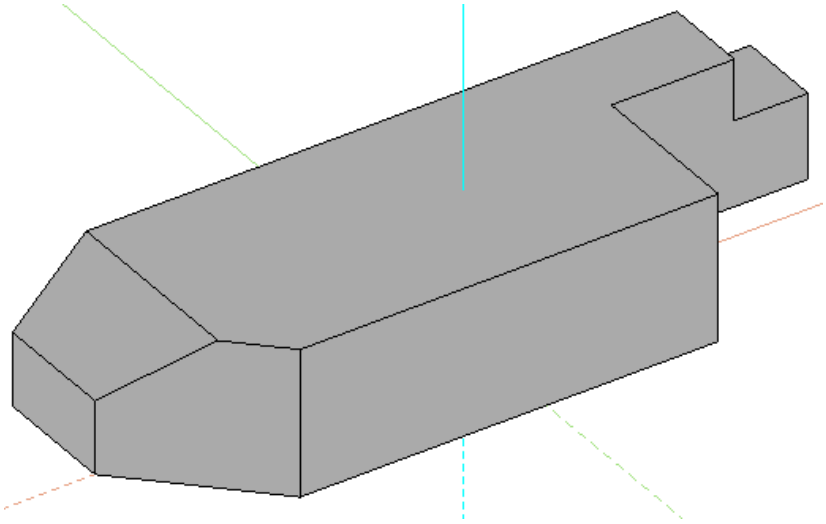
1. Draw the contours as a separate pattern in different work planes.



2. Select DEFINE 2-VIEWS OBJECTS.

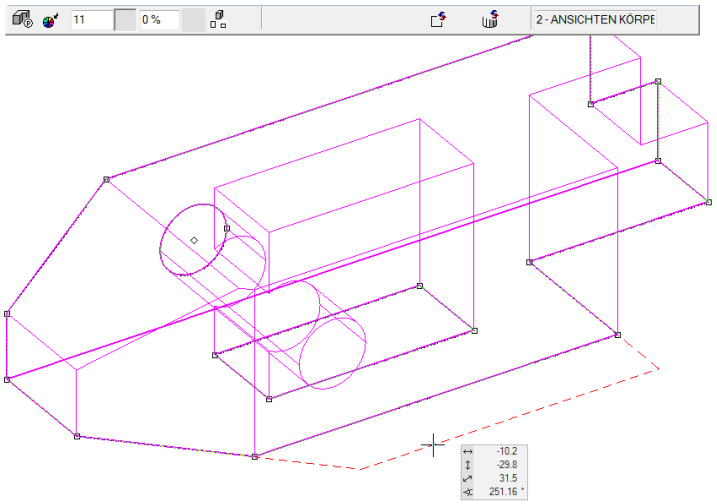


Please enter view 1 (ENTER = end)
Select the first contour
Please enter view 2 (ENTER = end)
Select the second contour

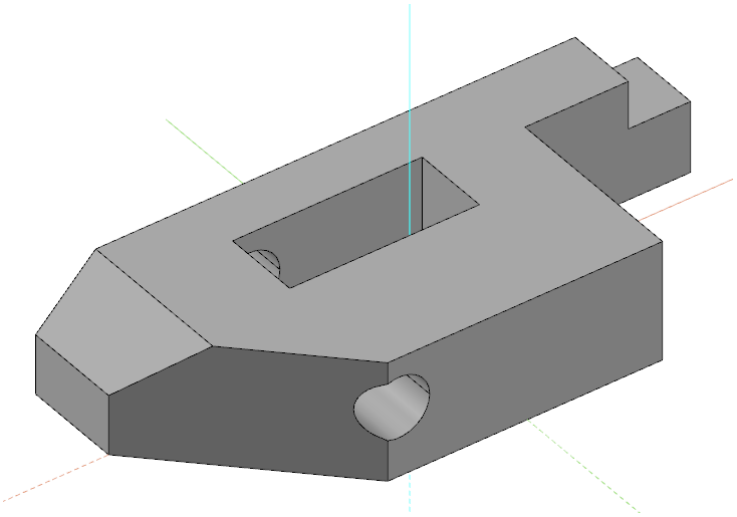


TIP

Modifications can be drawn directly into the corresponding polygon.

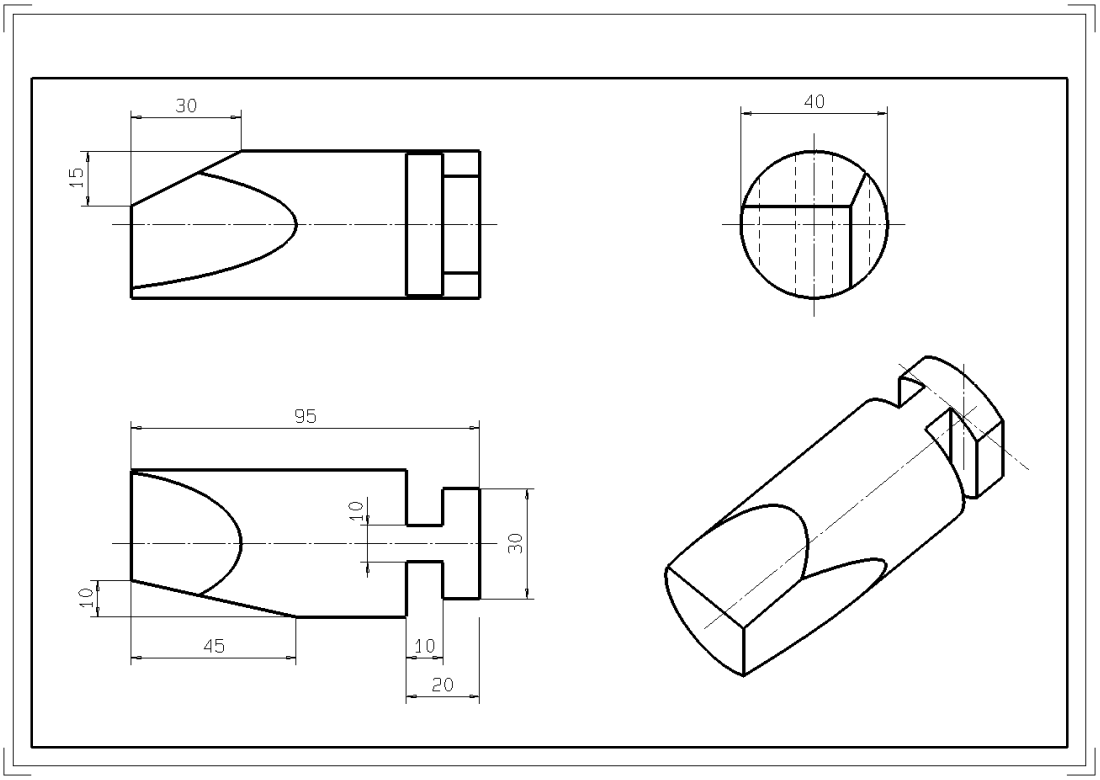


This is also displayed immediately in the 3D model.



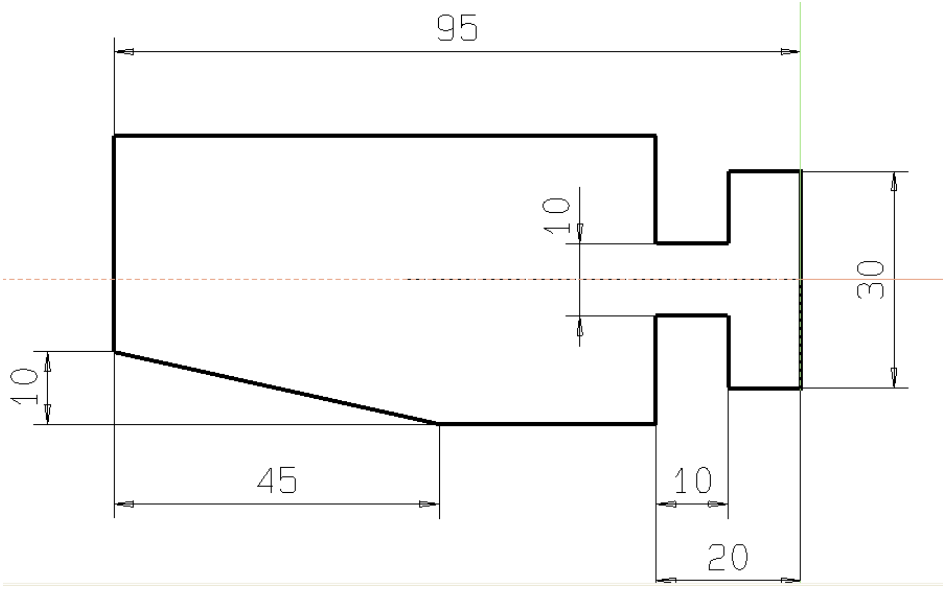
WORKSHOP END

EXAMPLE – BOLT

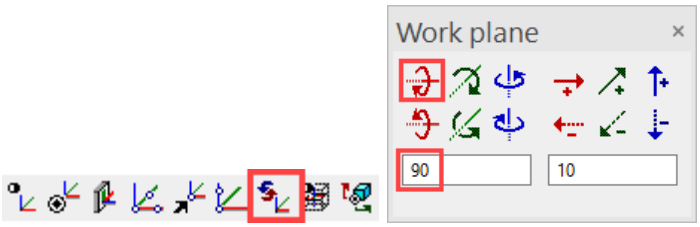


WORKSHOP

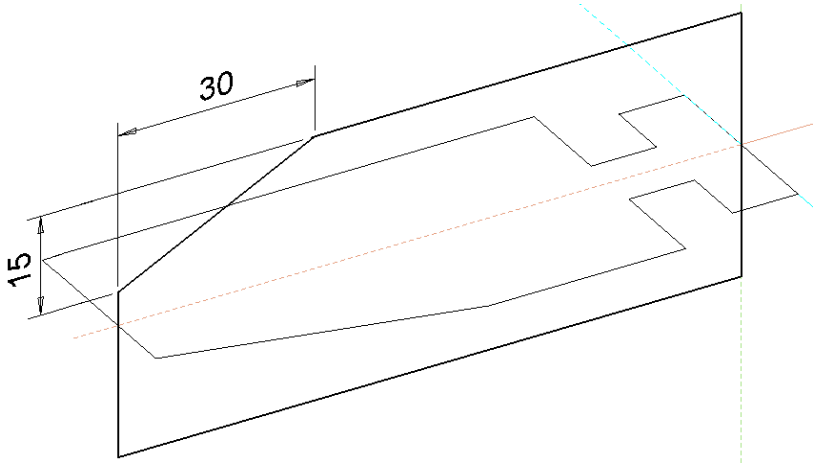
1. Start NEW MODEL (e.g.: „bk1, bolt“)
2. Draw the following contour in the top view



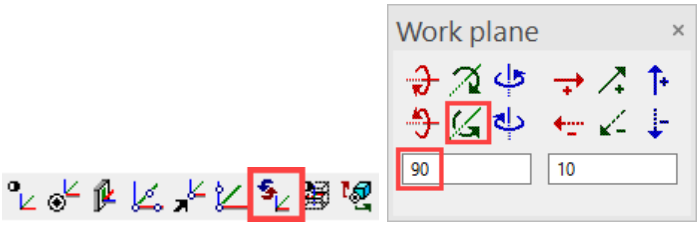
3. Rotate the work plane about the X-axis with 90°.



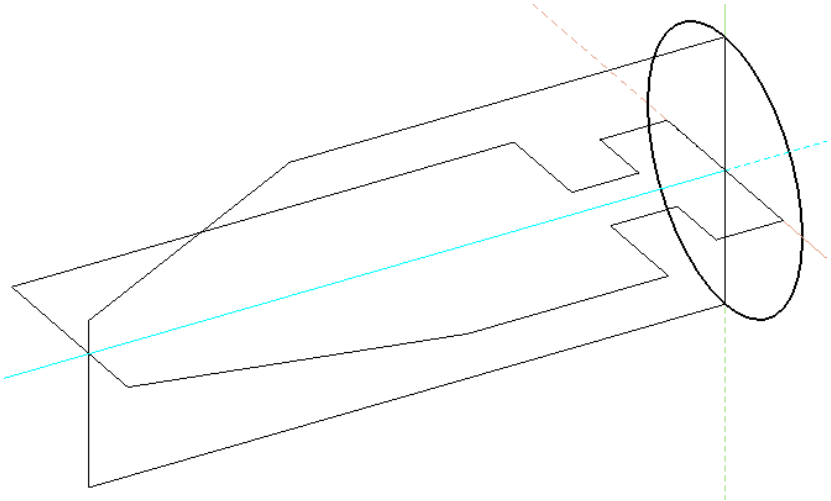
4. Now draw the following second contour.



5. Rotate the work plane about the Y-axis with 90°.



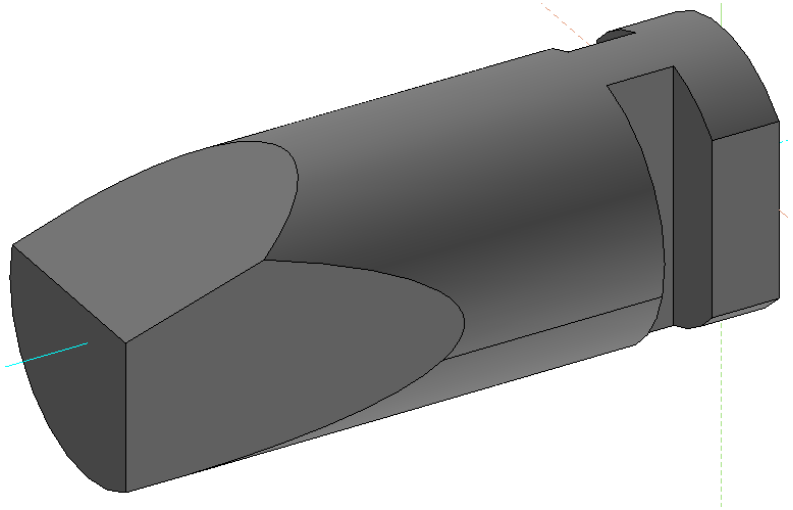
6. Now draw a circle with a radius of **20mm**.



7. Now create a 3D volume with DEFINE 3-VIEWS OBJECT.

TIP

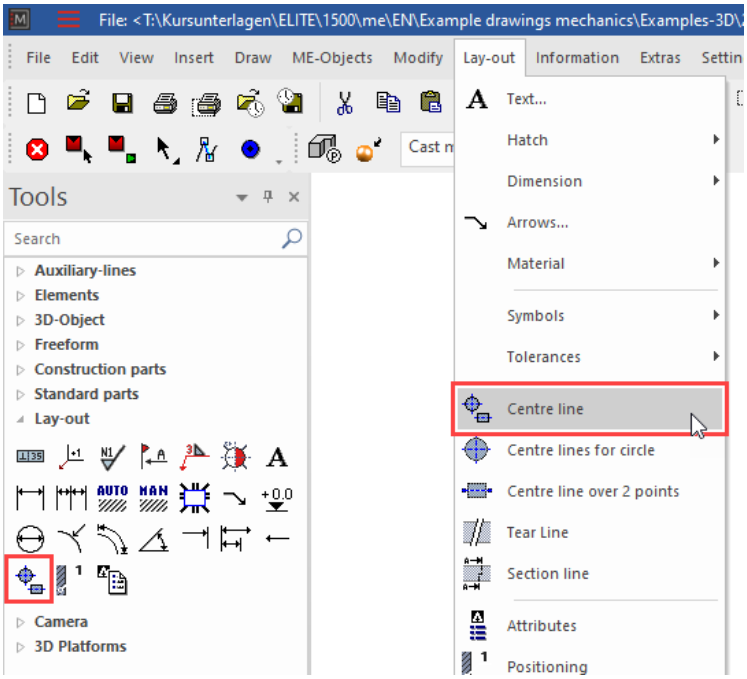
With 3-view object (3VO), the contours must be closed and drawn in different work planes.
The centre line must be its own pattern!
The centre lines can be created via LAY OUT> CENTER LINE or CENTER LINES FOR CIRCLE.



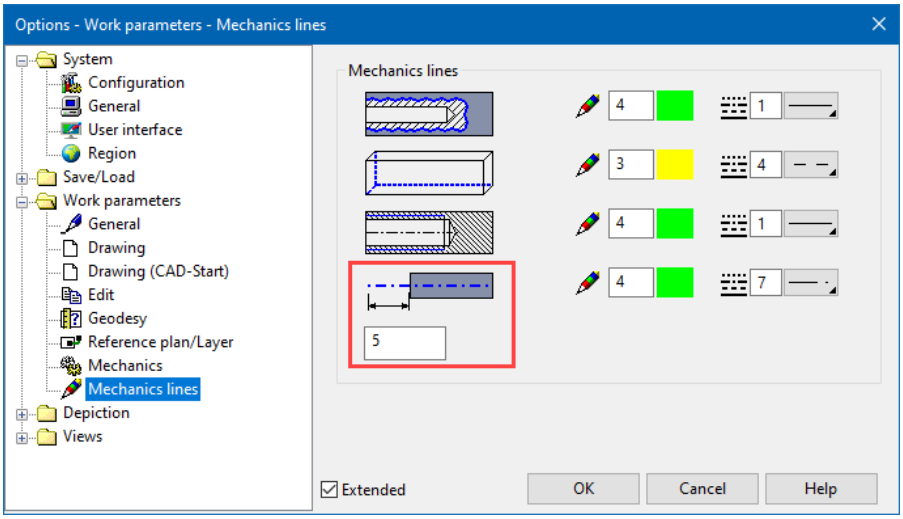
WORKSHOP END

TIP

If the center lines were created with the mechanical functions (LAY OUT > CENTRE LINE or CENTRE LINES FOR CIRCLE), they appear automatically in the 2D plan.

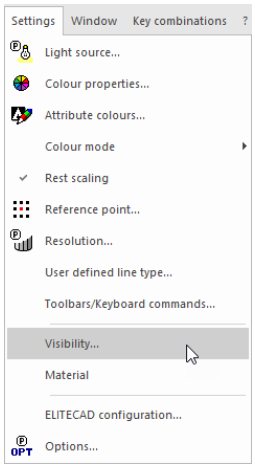


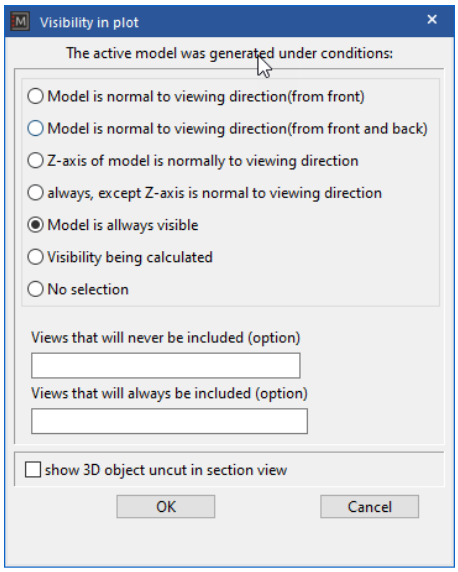
The stick out of the centre lines can be configured via SETTINGS> OPTIONS> WORK PARAMETERS> MECHANICAL LINES.



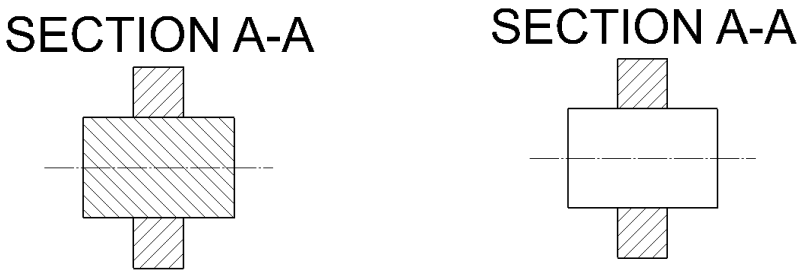
TIP

If centre lines or other contours in the 3D model are not entered with the mechanical functions, they must still be edited via SETTINGS> VISIBILITY.

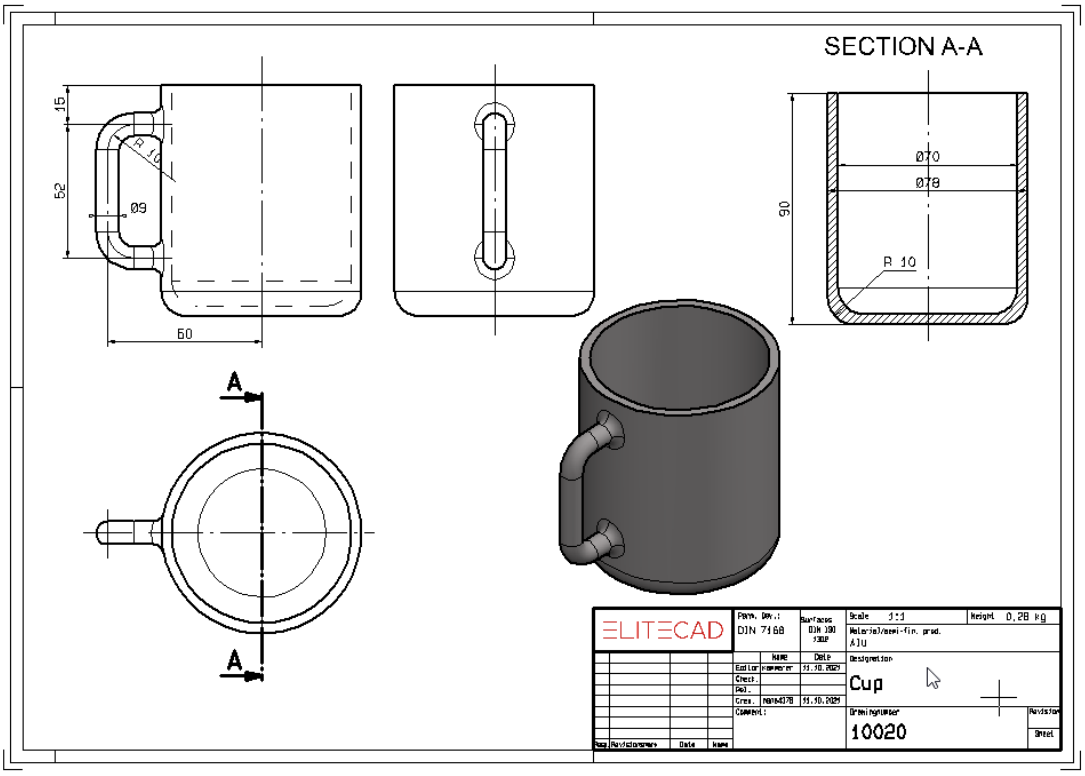




e.g .: start NEW MODEL and draw 2D centre line. If "MODEL IS ALWAYS DISPLAYED" is selected, this centre line appears in all views in the plot.
With the SHOW 3D OBJECT UNCUT IN SECTION VIEW, it can be determined which part (bolts, screws, ...) is shown uncut.
(Screws that were created with a DIN module from ELITECAD already have this attribute)

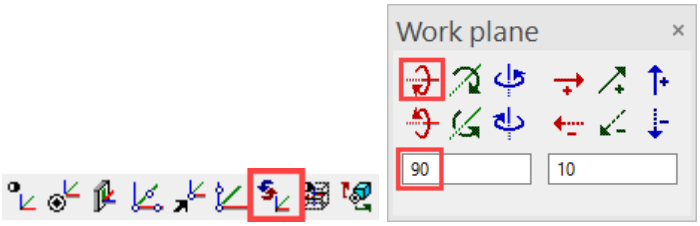


EXAMPLE – CUP

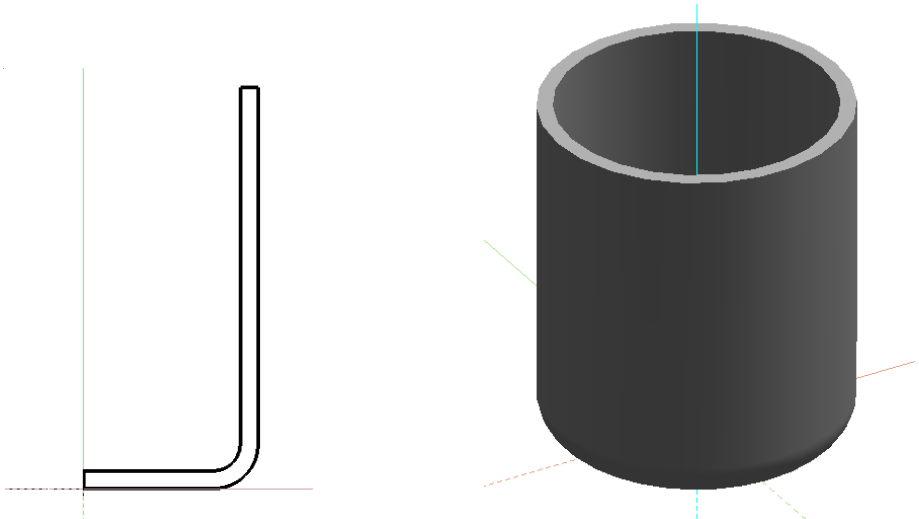


WORKSHOP

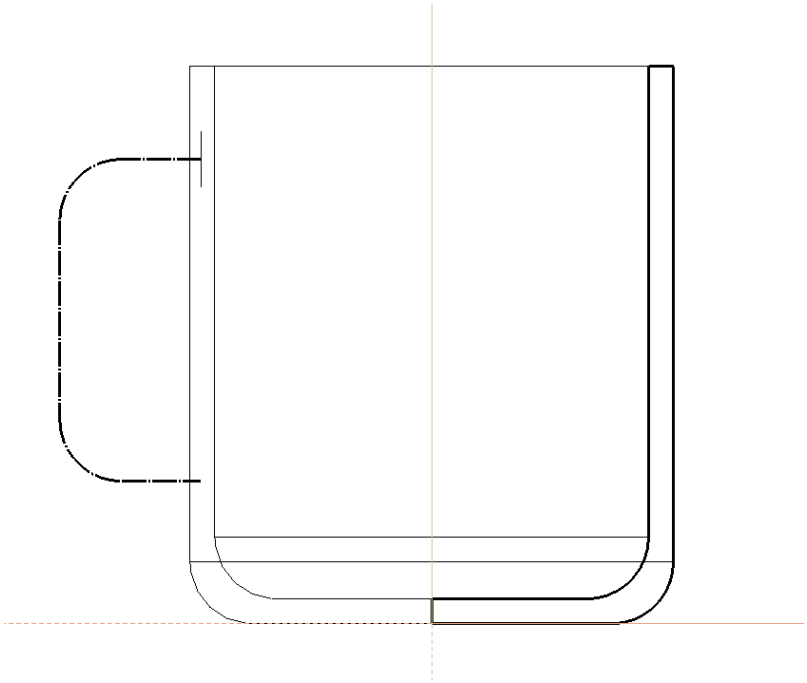
1. Rotate the work plane about the X-axis with 90°.



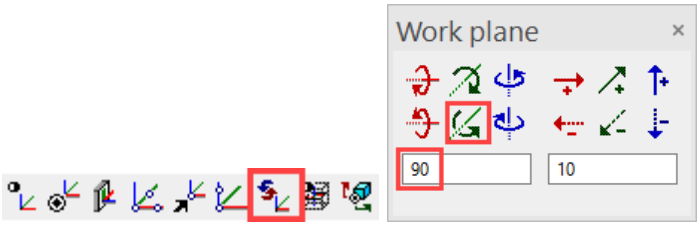
2. Draw half the contour of the cup and define it as a "rotation box".



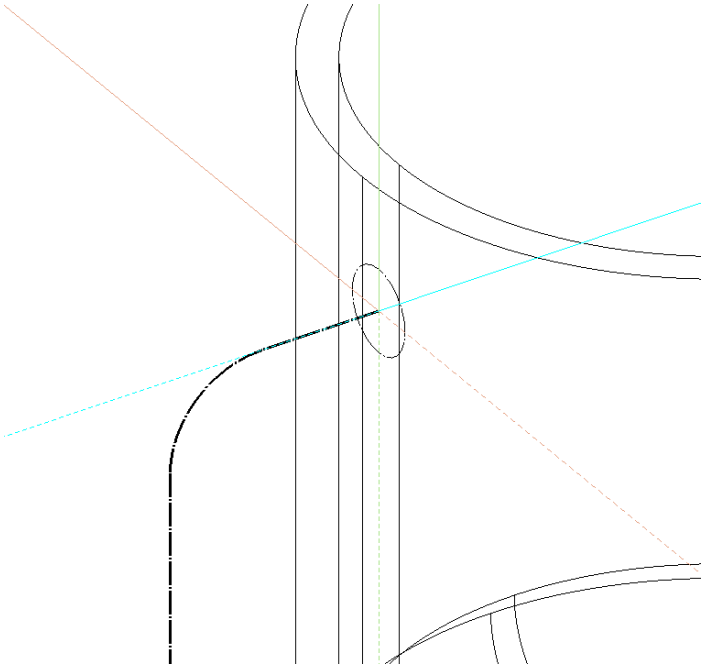
3. Draw way for handle. (in a new pattern)



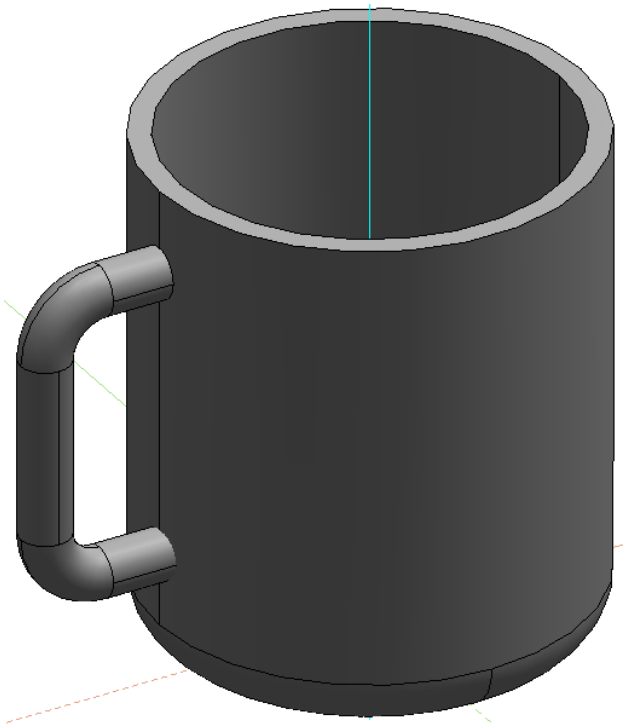
4. Set the work plane at the beginning of the handle contour. (Rotate the work plane by 90 ° in the Y-axis and select WORK PLANE ORIGIN.



5. Draw the cross-section of the handle.



6. Now define a transbox with path and cross section.



7. Perform the Boolean calculation UNION (A + B) and unit the handle with the cup.
8. Round the handle with the surface of the cup. (Radius **4mm**)

WORKSHOP END