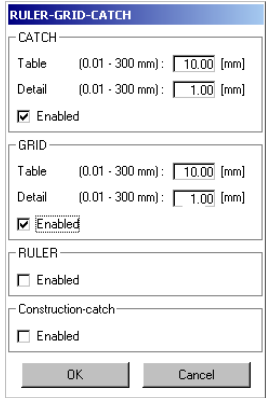
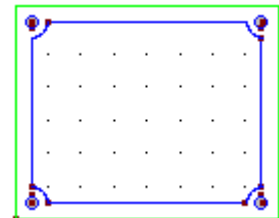


Engraving your nametag with CNC-milling machine

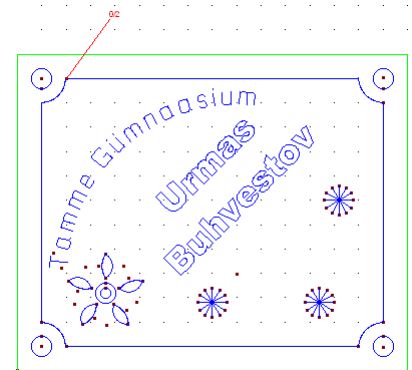
1. Start program *nccad*, click to close advertisement.
2. Start new CAD/CAM milling drawing from File menu.
3. Set work piece zero point (CAM - ) into point (50 , 50) (look CA values on status bar under gridlines). As you can see this will be new coordinate zero point.
4. Set work piece changing position (CAM - ) into point (20 , 150) with height 20 mm. This is the place where the program will finish its work.
5. Draw a rectangle to mark borders of nameplate. This will be guide for us to see area where engraving is allowed.
 - a. Choose rectangle tool (CAD Standard - )
 - b. Click once at zero point.
 - c. Press key K and enter coordinates (85 , 65) (comma is necessary, no brackets) and press enter. This is shortcut for setting point which isn't easily accessible by mouse.
6. Zoom out with Choose detail tool (View - ) . Start zooming area from (-10 , -10) and end it at (100 , 80).
7. Change properties of gridlines with Ruler/Grid/Catch tool (Settings (or right click on grey area) - ) to values shown on picture. Catch means how accurate you can select points by mouse. In our case 10 mm in normal view and 1 mm in zoomed out view. Value of Grid shows how often helping points are drawn on background).
 
8. Change default layer on status bar to second one (Layer 2 ) . This helps us to make difference between lines we need to engrave and which not and choose order of work.
9. Draw four circles with radius 2 mm and midpoint coordinates (5 , 5), (5 , 60), (80 , 5) and (80 , 60). To do that choose circle tool (CAD Standard - ) , click at midpoint (you always can use K to mark them from keyboard) and then click 2 points above (CR value on status bar is (0 , 2)).
10. Draw four lines (CAD Standard - ) parallel to borders on connection line of two circle midpoints so that distance between circle and line endpoint is 3 mm. For example one line has ending points (5 , 10) and (5 , 55).
11. Draw four arcs to connect lines (CAD Standard - ) . Click on the first end point, then on the second one and finally mark the centre of the nearest the circle as centre of the arc. Program draws arc clockwise, so if you choose end points in wrong order then you get different arc. You can always Undo (Edit - ) to cancel LAST action (only one) or Delete Last (Edit - ) to erase last drawing. If you notice a mistake later then you can use Delete tool (Edit - ) clicking on start point of line or arc (brown square). Sometimes it is easier to start over.
12. Add text with one of engraving text tools (CAD Standard -  or )
 - a. Differences of tools
 - i.  adds multi line text in any Windows font. Unfortunately it takes more time on machine to get them ready and program makes outline (two lines), so material breaks easily and result isn't always nice). Best results are with font size more than 16 points.
 - ii.  adds multi-line text in fonts specially designed for milling machine. They are quick fonts and almost won't damage material. But unfortunately these fonts don't have all letters with ascents. If you have one you have to use the first option.



- b. Mark rectangle where the text must fit in. It's important to remember where you marked the first corner of text area, good practice is to use point where isn't any start point of other drawing element.
- c. Choose font, font size, vertical and horizontal align (usually centred is the best) and type text.
- d. To edit text or its parameters use Edit Text tool (Edit - **TEXT**). To start it you must find starting point of text area.

13. Add description of technology (CAM - ). Start line from any starting point of blue line and end it somewhere outside the drawing area, select parameters

- a. Number: 0
- b. Layer: 2
- c. Type: Metabo
- d. Machining: Single piece /counter
- e. Countour correction : none
- f. Feed 100
- g. Tool diameter: 0.8
- h. Entrie depth: 0.15
- i. Depth per step: 0.15
- j. Shift Z-zero point. 0



14. See result by Simulation → OpenGL Milling. No need to change parameters.
15. Add lines, arcs, circles, text etc to make your nametag unique.
16. Save work in teacher chosen folder with your name. Fill in creator information.

How to make a flower?

1. Draw two arcs with same endpoints with opposite directions or just one short line
2. Select tool Circular arrangement (Edit - ) , angle 360 degrees, number on your taste ($3 < n < 20$) and size of the flower, draw rectangle around arcs (two clicks), choose centre of rotation.
3. Draw circles or lines to make it more interesting.
4. Make sure that there is no drawing outside green lines – it would break milling tool.

How to rotate a part of the drawing?

Select tool Rotate (Edit - ) , mark rotating part (once on brown square or rectangle surround it with two clicks), click in centre of rotation and draw line in direction you desire). It's easier to understand if centre of rotation is somewhere in the middle of rotating object.

How to move part of drawing?

Select tool Shift (Edit - ) , mark moving part (once on brown square or rectangle surround it with two clicks), choose one point (no need to be point on line), wait a bit and then click where firstly chosen point should be after moving.

How to make copy of some part of drawing?

Select tool Copy (Edit - ) , mark part (once on brown square or rectangle surround it with two clicks), choose one point (no need to be point on line), wait a bit and then click where firstly chosen point should be in the copy. If you click more then you will get several copies.