Name of unit:
Producing a journal

Field of education:
Mechanical engineering

Professional qualification:
CNC operator

Exercise:
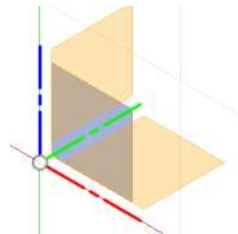**Technical drawing of a journal (solution)**

Variant:
Journal 2

Solution of exercise:

1. Open Autodesk Fusion 360 software.

2. To enter the sketch mode for drawing click on the following icon:

3. Upon showing of coordinate system origin mark, select the plane in which you want to draw - front view, top view, side view (depending on the shape of the object for drawing).

4. Tools for drawing of lines or circles are shown upon choosing of the drawing plane.

5. At the beginning of drawing the first line or circle it is recomended to use the origin of the coordinate system.
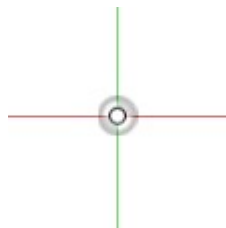
6. With the *Line* command it is possible to draw the contour of the model or the auxiliary lines for the next steps. We can draw all object lines at once. Using just a left mouse click, the command remains active, and we can draw the next desired line. With the *ESC* key we turn off the line command. For the cylindrical models we draw only a half of the model.
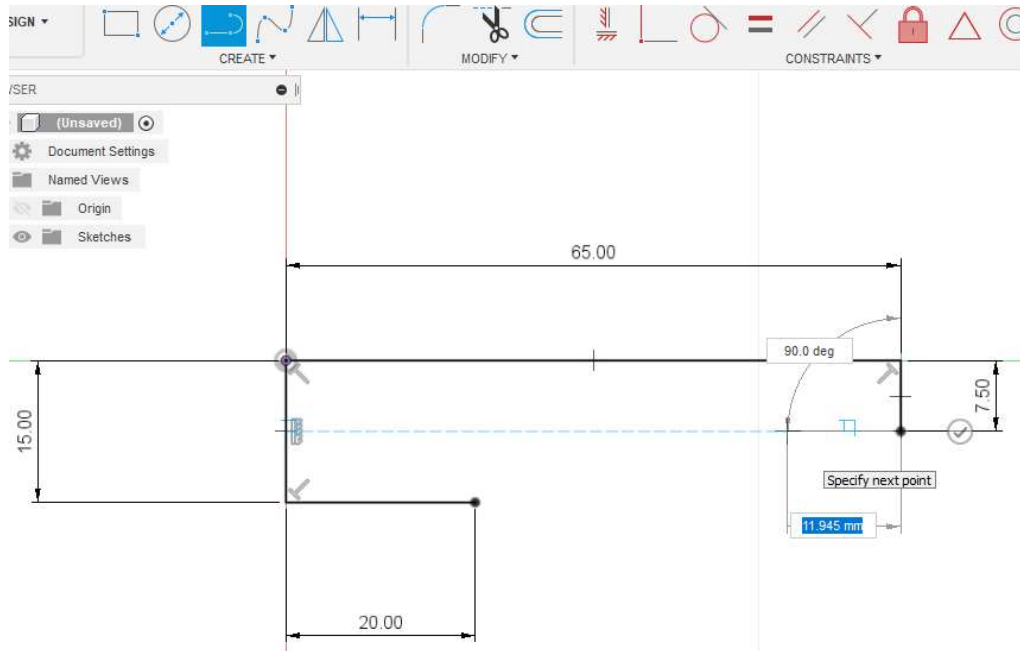
7. After drawing all the lines, we can use the **Dimension** command to determine the dimensions of the lines or circles. If the dimension is already visible, with a double click we have the option to change the dimension.
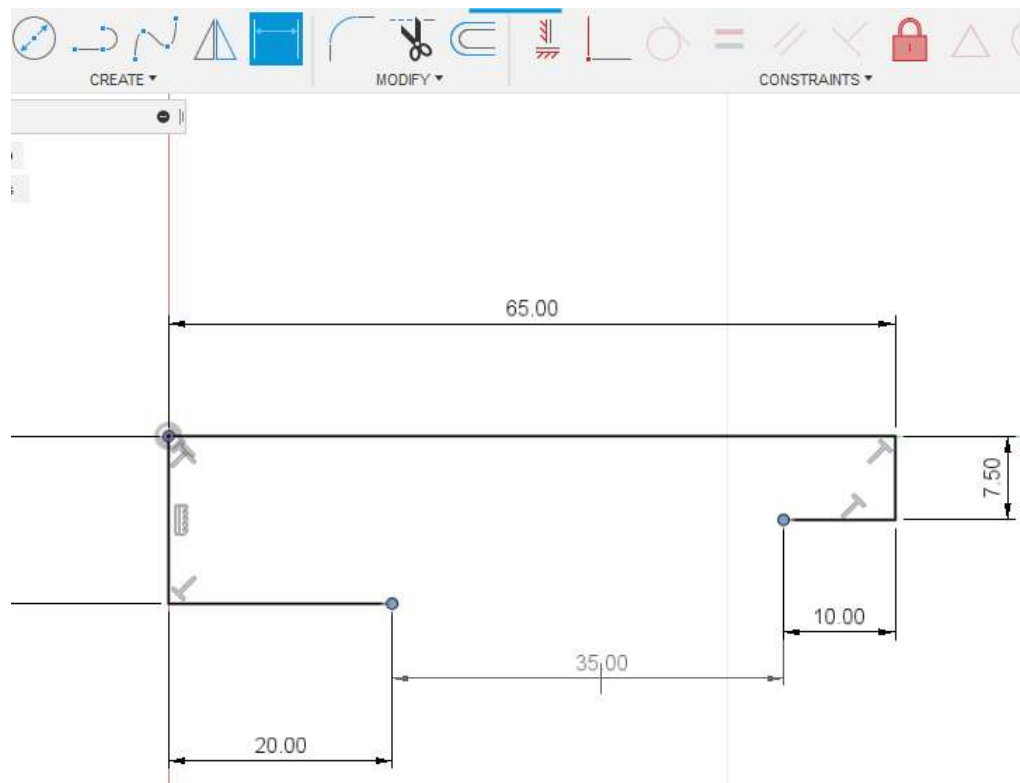
8. If you are bothered with the dimensions on the sketch, in the **Sketch Palette** you can customize user interface by hiding it (remove the check mark beside **Show Dimensions** option).
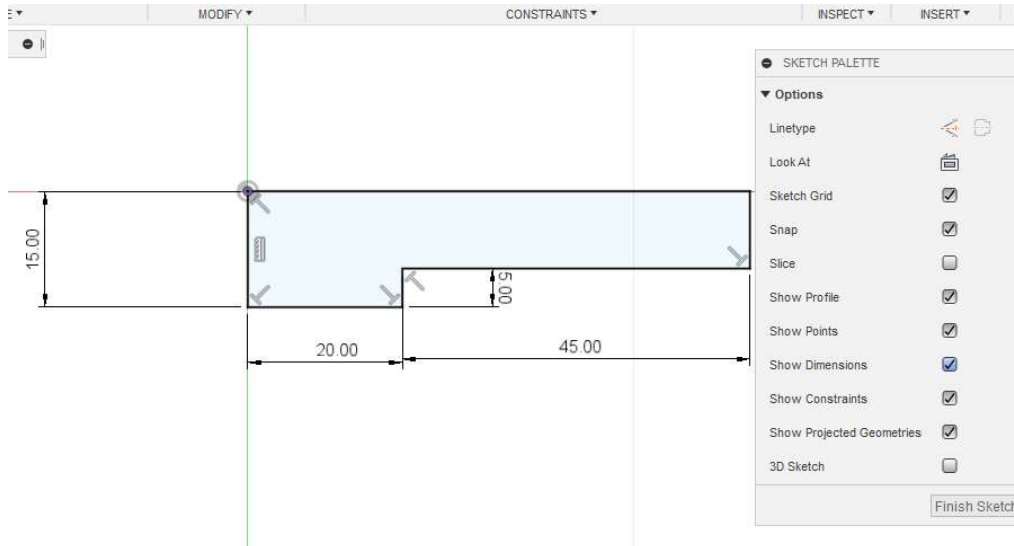

9. With the **Circle** command we can draw the circle and enter it's diameter.
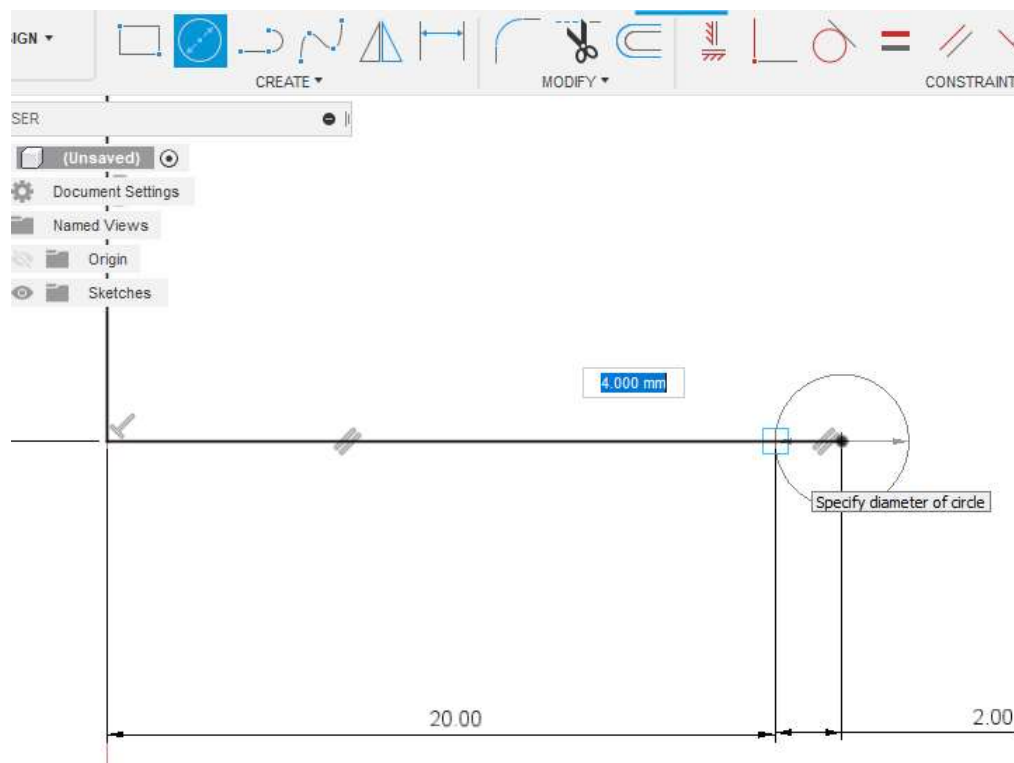
10. By dragging a line toward a circle we can accurately tangently draw a line on a circle, when the tangent constraint is displayed.
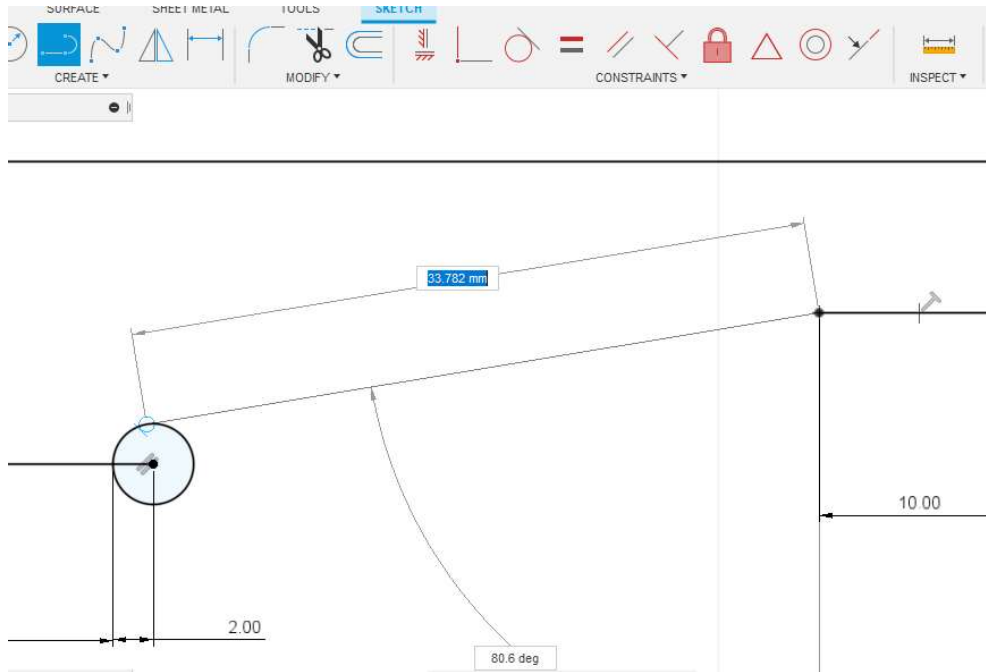
11. By using the **Trim** command it is possible to cut out part of lines or circles that are not needed anymore. If we want to delete an entire auxiliary line or circle, we need to select it first, then use the **Delete** key to delete the entire line or circle.
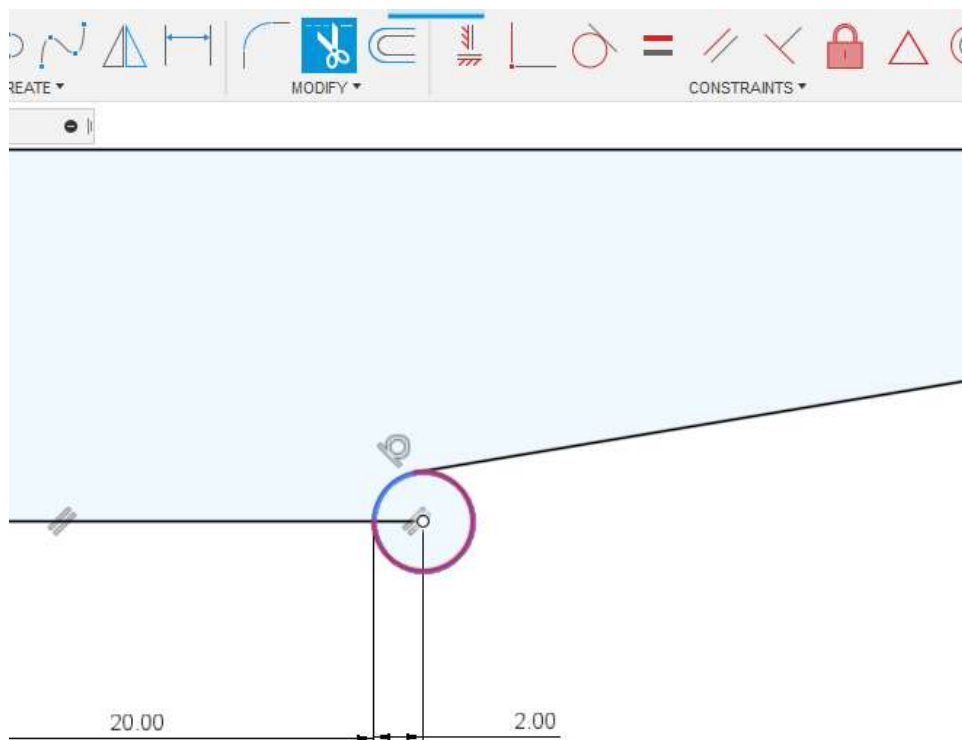
12. With the *Chamfer* command we can draw beveled edges. Then we choose the edge we want to bevel.
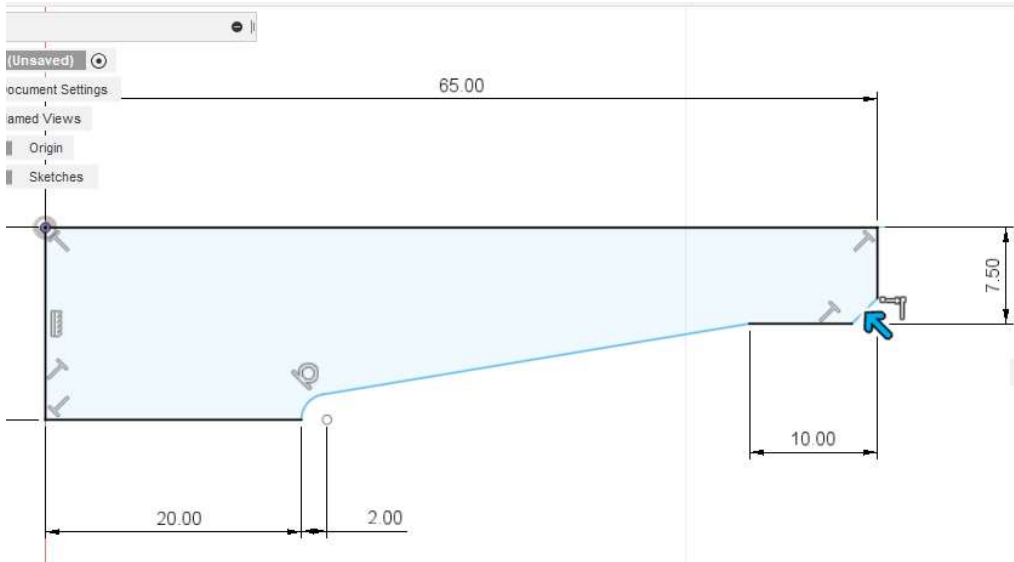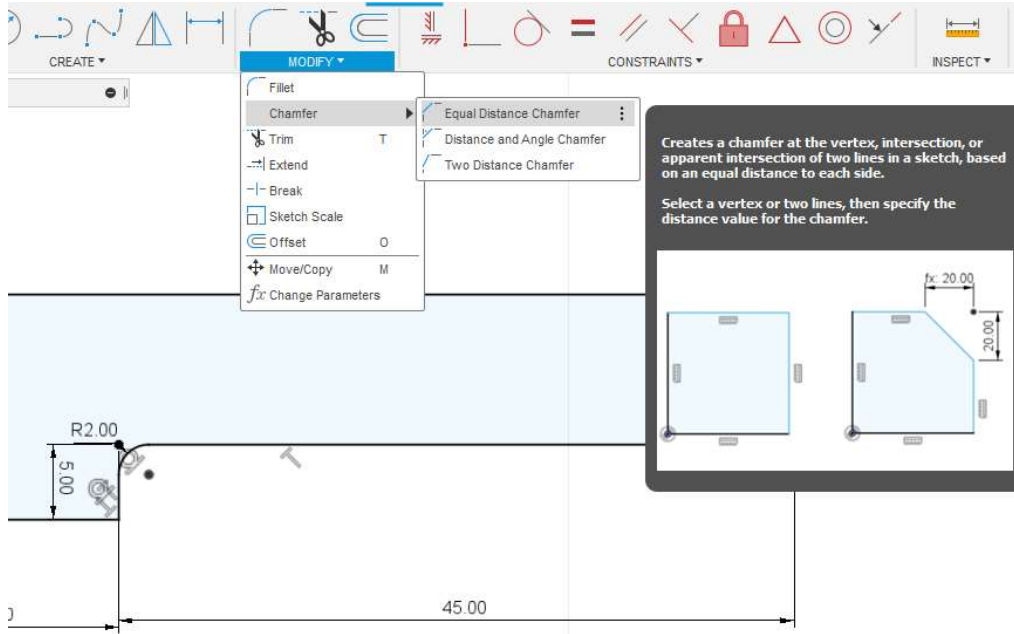
13. If all needed parts of a model are drawn, to create the 3D model it is necessary to exit from the sketch mode (**Finish Sketch** command). To apply **Extrude** or **Revolve** commands all the surfaces have to be closed (all lines connected).
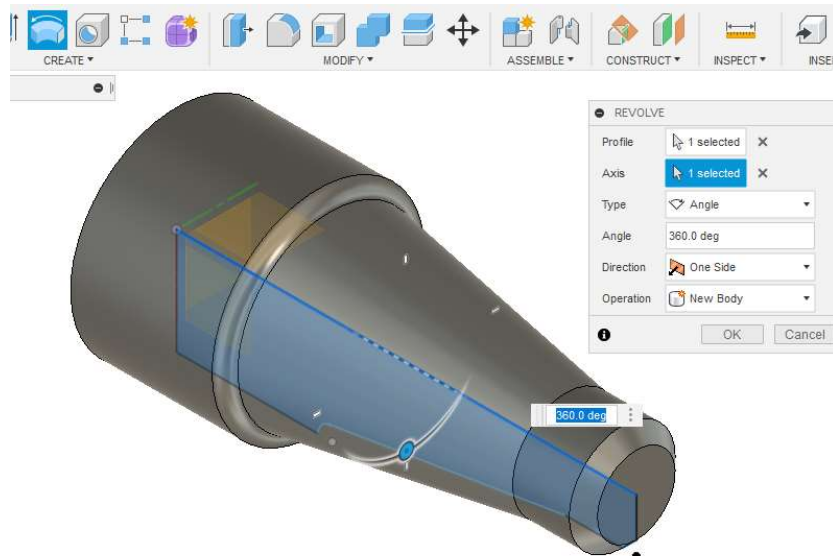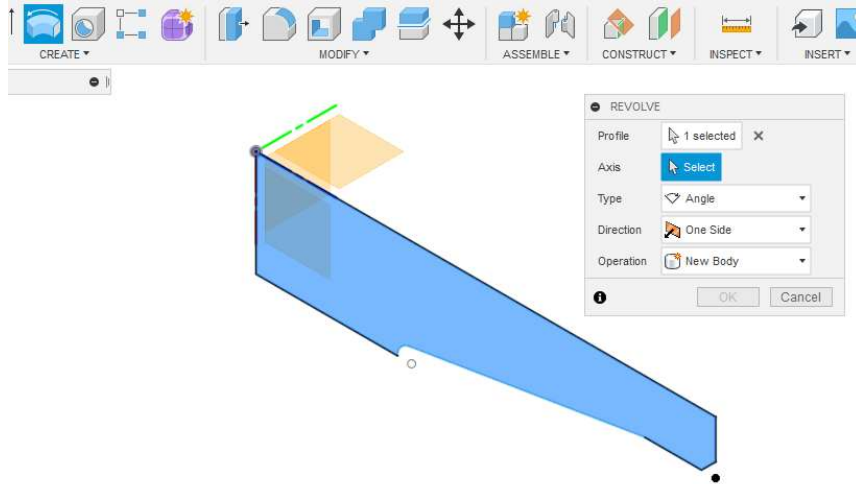

14. On the orientation cube the view on the drawing model could be set. By clicking on the house symbol it is possible to switch to the isometric view.

15. An example of the isometric view of the drawing model:
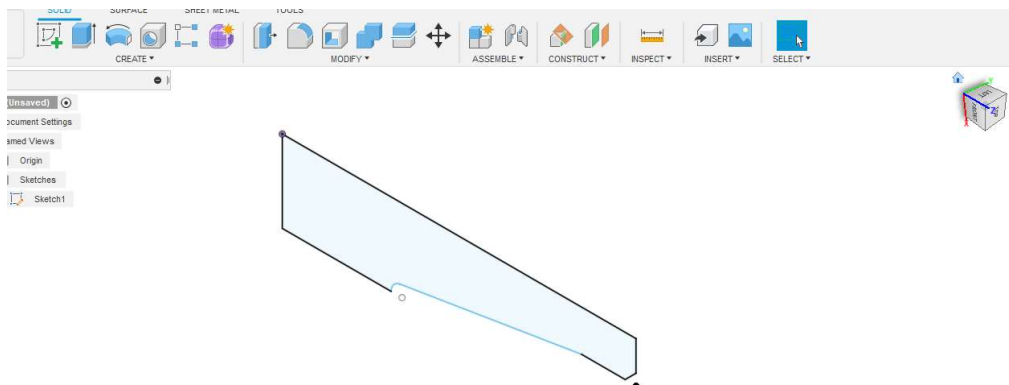
16. By clicking on the **Revolve** command, we select closed surfaces. By selecting the axis of rotation, a 3D model is created.

17. After creating of the 3D model the body of the drawn part is visible, but the sketch is not. To show the sketch again we need to click on the eye icon and everything that is drawn in this sketch will be visible in drawing space. If there is need for change of some part of the sketch by double clicking on the sketch it is possible to enter the sketch editing mode.
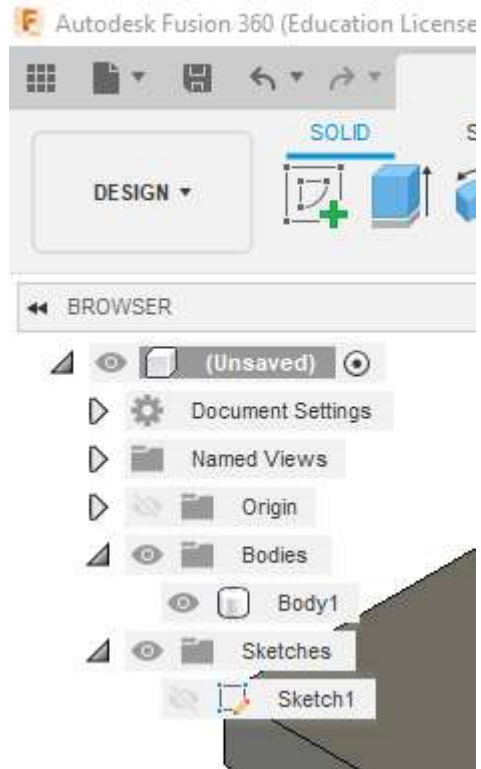
18. From the **Design** menu, by choosing the **Manufacture** option, it is possible to switch to the mode for making of the simulation and generating the NC code.
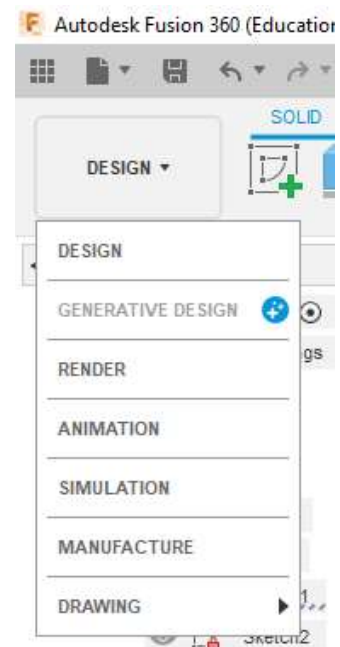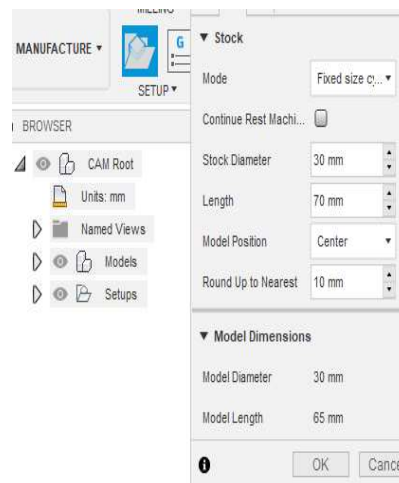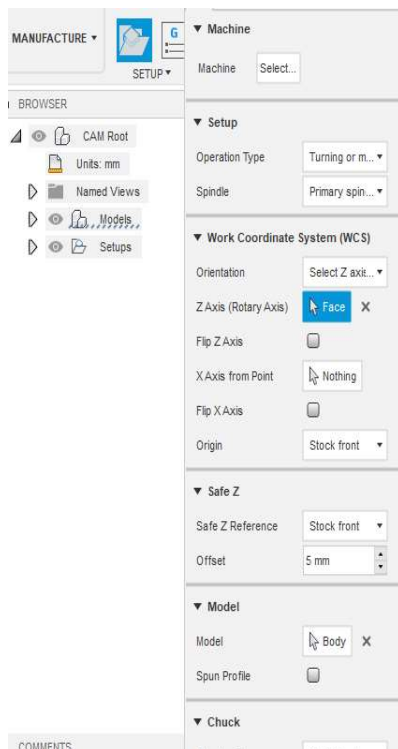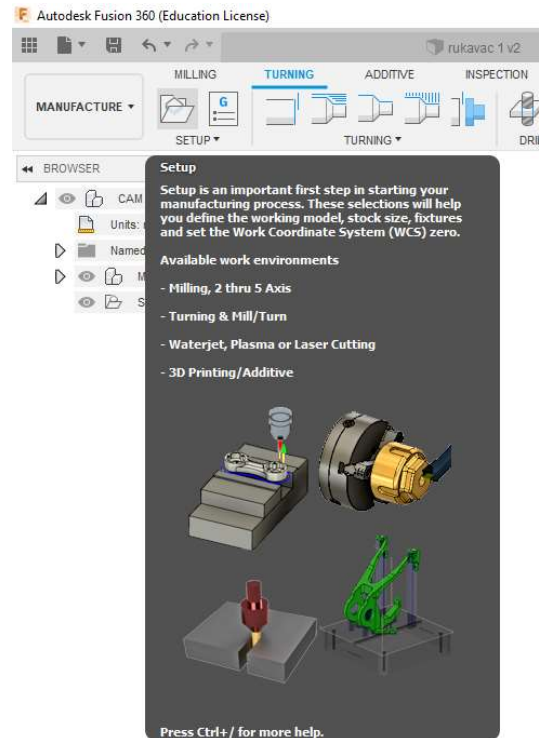
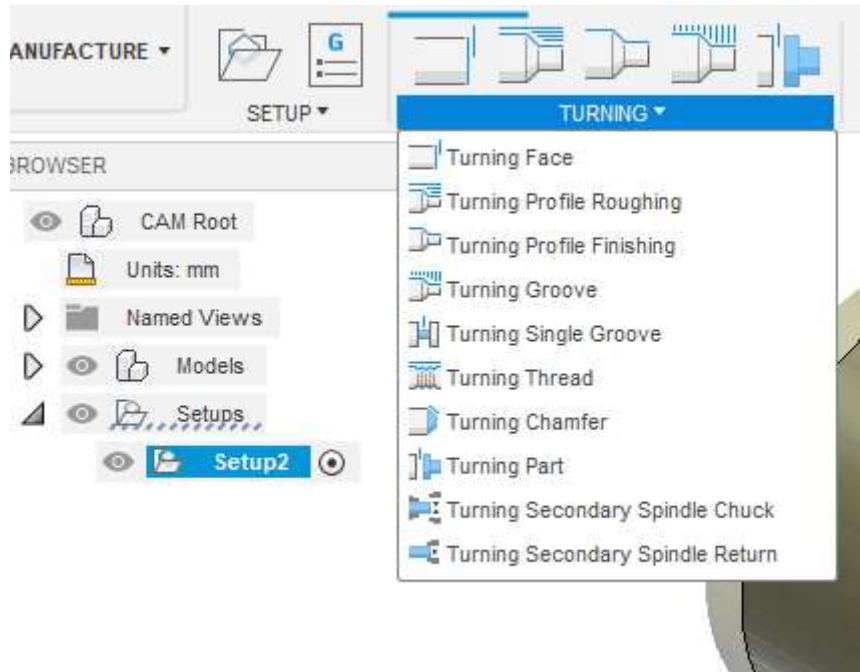19. Upon entering the manufacture mode there are new options available.

20. By opening the *Turning* tab we come to turning operations.

21. By clicking on the *Setup* command new dialog window opens, in which we can set up the initial material and define the coordinate system.
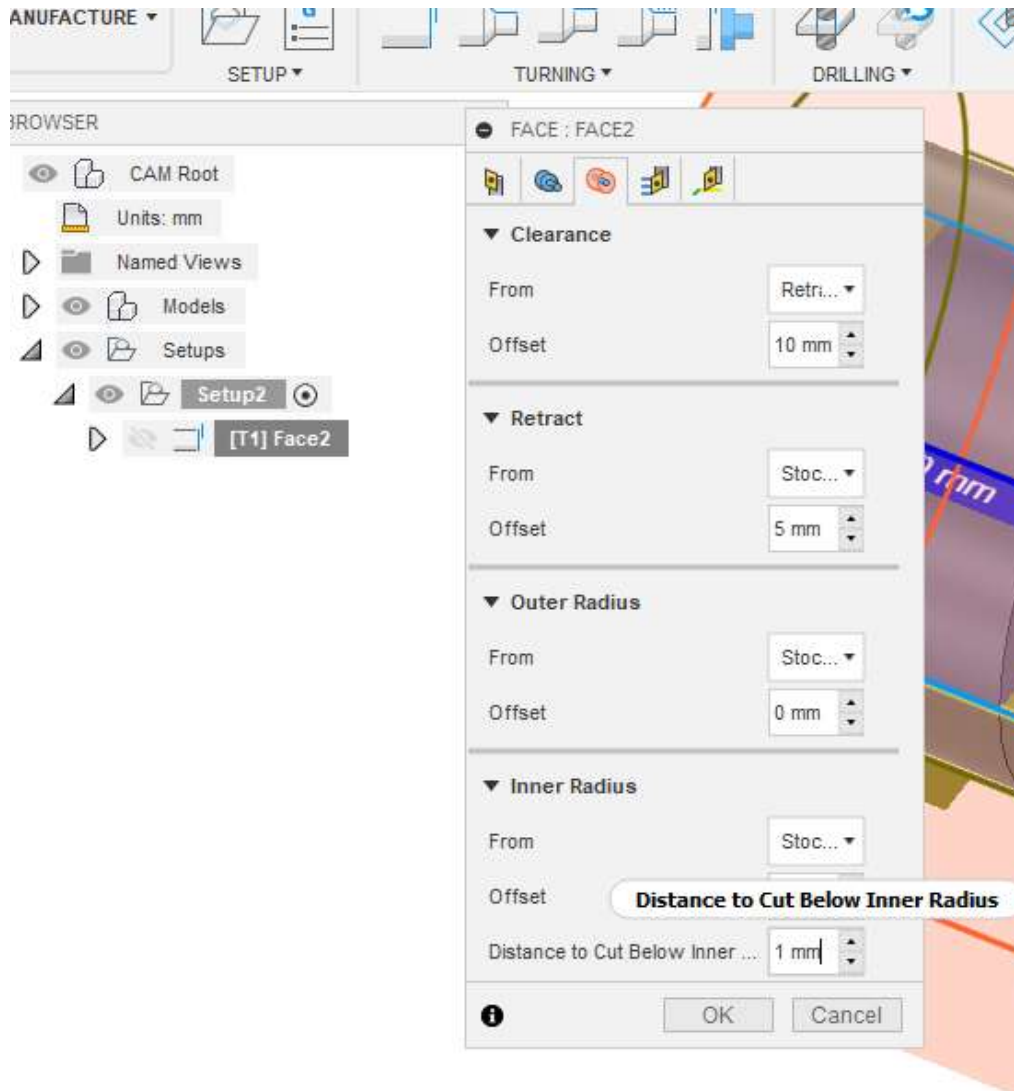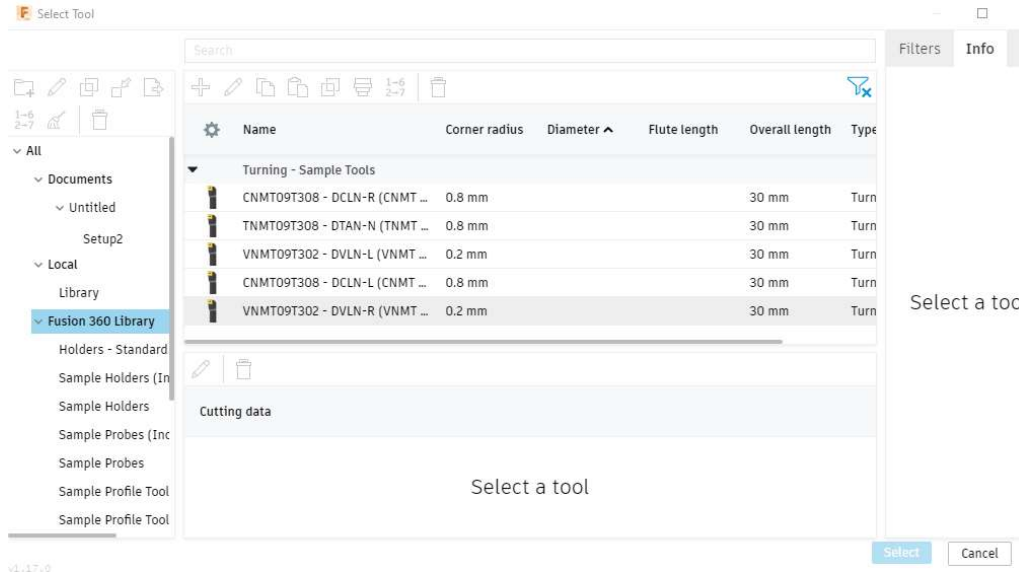
22. A turning operations menu:

23. With **Face** option we can align the forehead of the material. A window opens where we can adjust the depth of cut, ways of approaching and moving the tool away from the workpiece.
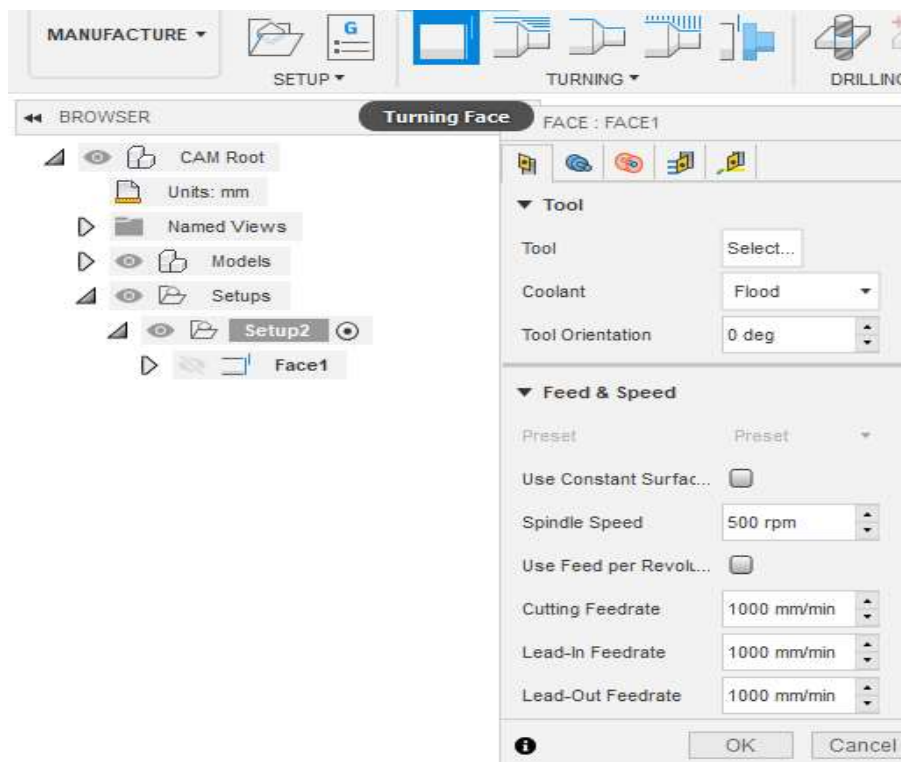
24. An example of adjusting the passage of the tool blade over the axis of the workpiece during the forehead alignment operation:

25. In the **Select Tool** window there is option to choose one of the tool from the list. By using filters it is possible to choose the tool and the tool holder. The tools can be adjusted if needed (e.g. angle of the tool blade).
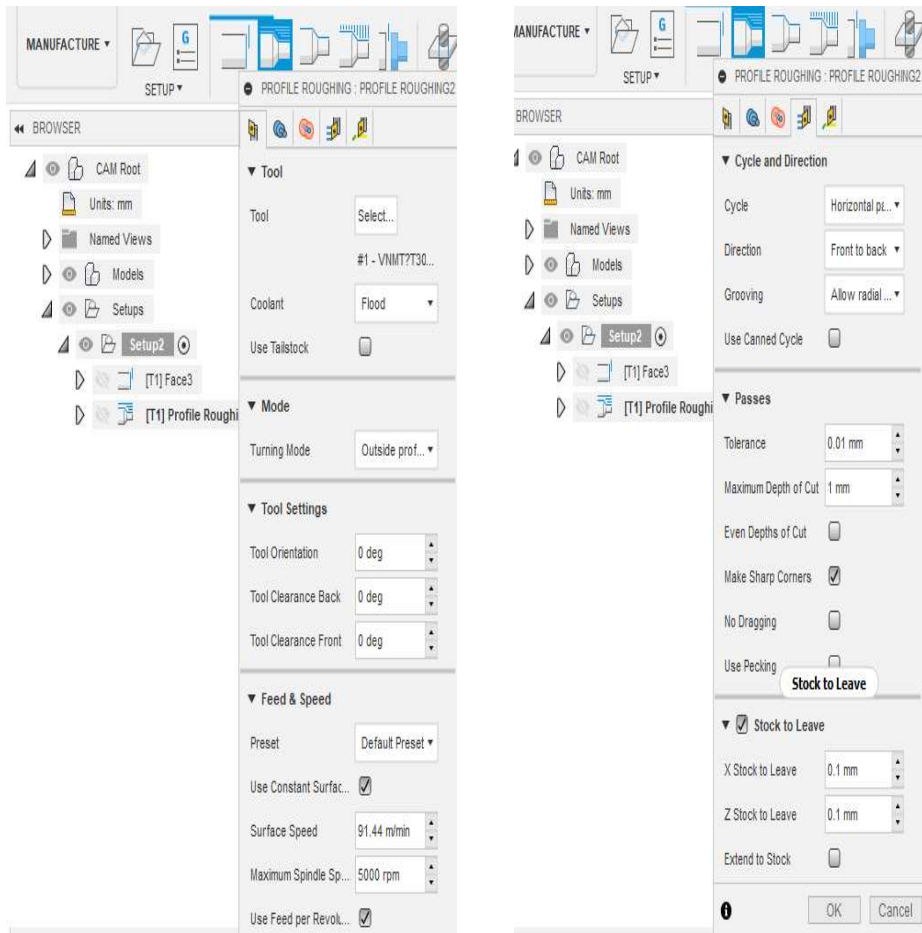

26. By clicking on the right mouse button over the option and selecting **Edit Tool** command we can change cutting tool and tool holder dimensions.
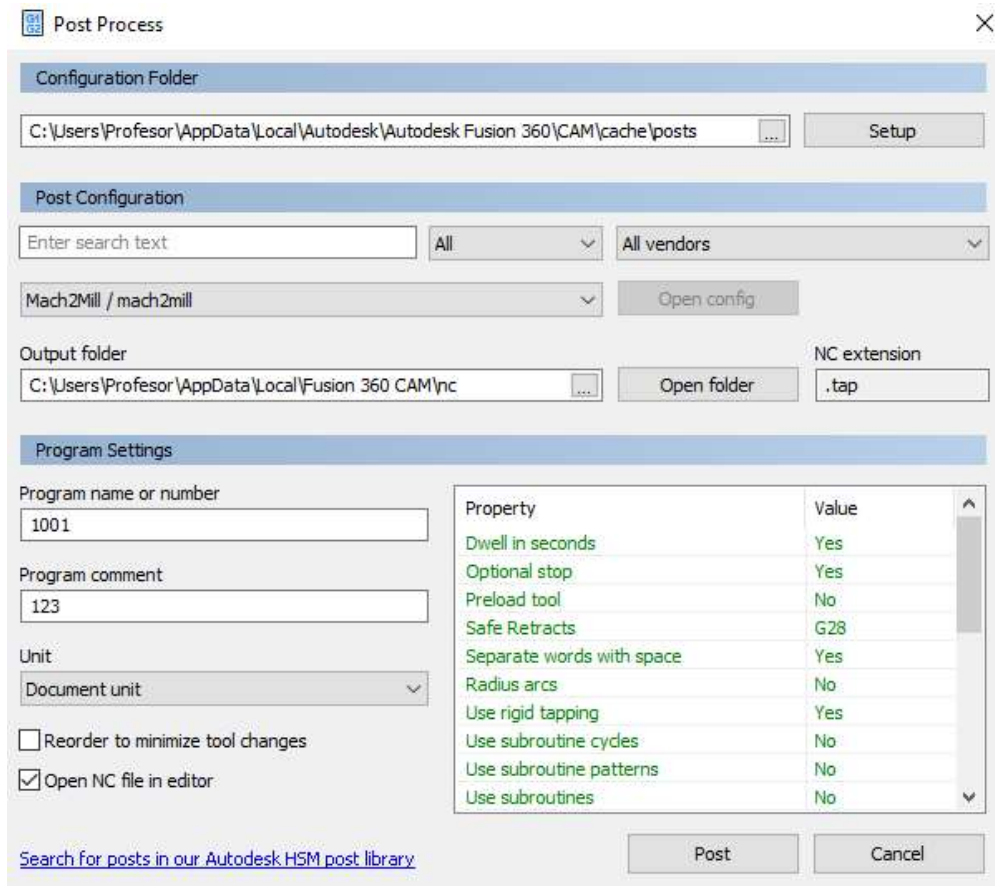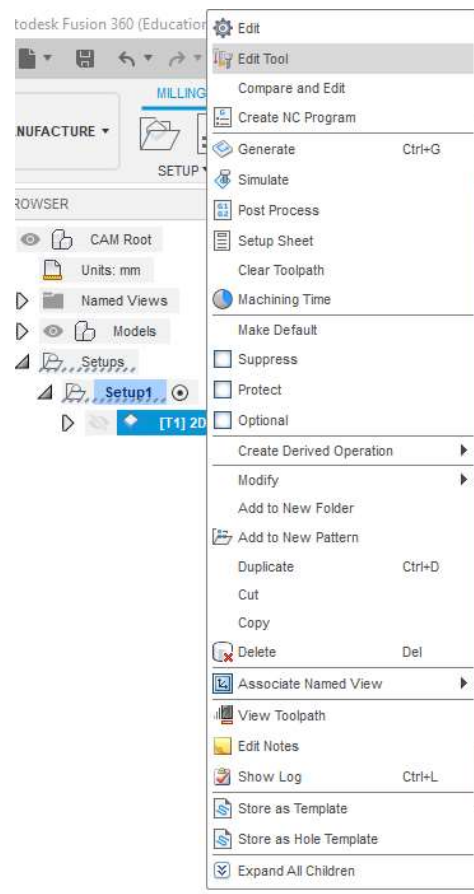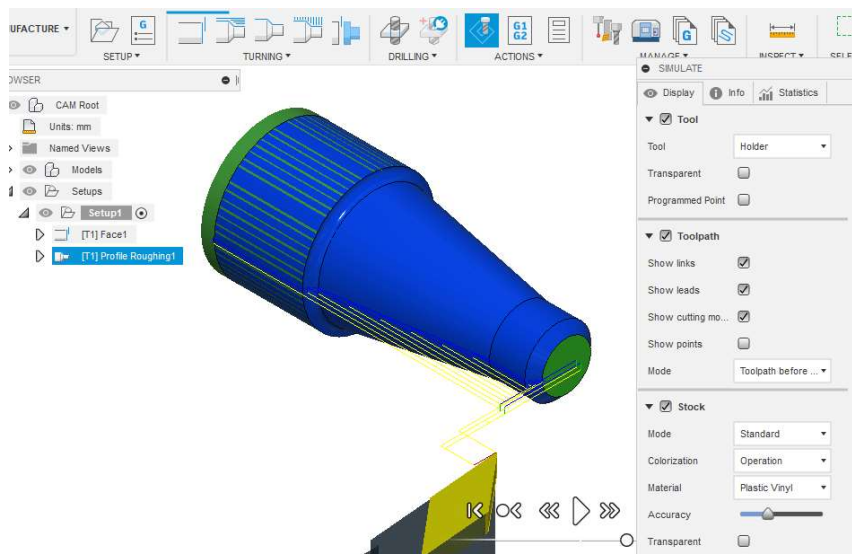
27. In the process of selecting the longitudinal turning operation, we can adjust the cutting depth. If we do not have a fine finishing process, we can adjust not to leave the material for fine finishing.

28. After setting all of the necessary options the simulation can be started by choosing the *Simulate* command from the *Actions* menu.

29. After reviewing the simulation, if no errors are shown, we can proceed to the generation of NC code through the **Post Process** option from the **Actions** menu. A window opens where we can choose the type of machine and the type of program on the computer.

30. After all the settings, clicking on **Post** button opens a new window with the generated NC code, which we have to clean before we transfer it to the machine controller.