Name of unit:
Producing a sprocket

Field of education:
Mechanical engineering

Professional qualification:
CNC operator

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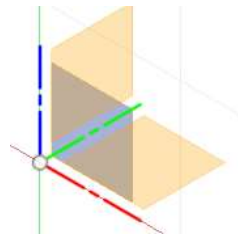**Technical drawing of a sprocket (solution)**

Variant:
Sprocket 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.
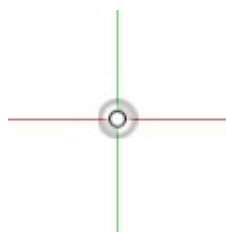
6. With the **Rectangle** option it is possible to draw a rectangle. Setting of dimensions is possible using the keyboard and for change of dimension inserting box to another dimension it is possible to use **Tab** key. Drawing of the rectangle (and finishing of drawing command) is done by pressing the **Enter** key on keyboard.
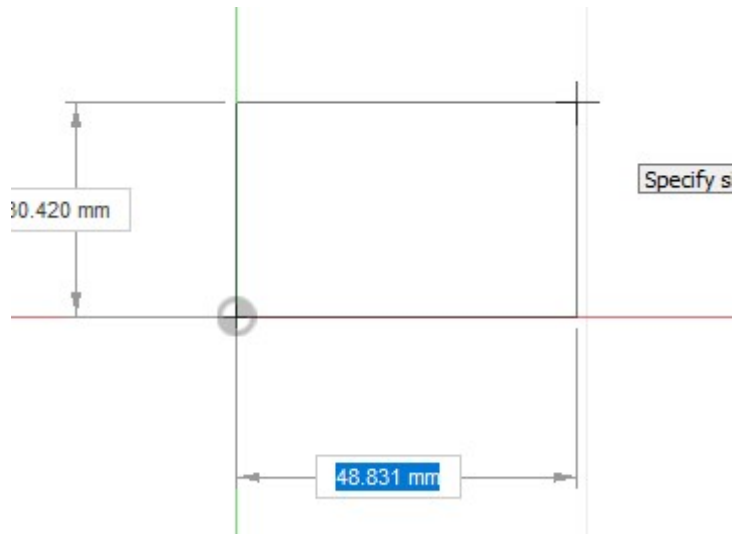
7. By using the **Sketch Dimension** command it is possible do define the length of line or a diameter of the circle and where we want to draw it, as well as make and show the exact dimensions from the coordinate system origin. If there are bisectors in the drawing, we can draw them as auxiliary lines for easier positioning of other lines or circles and delete them later.
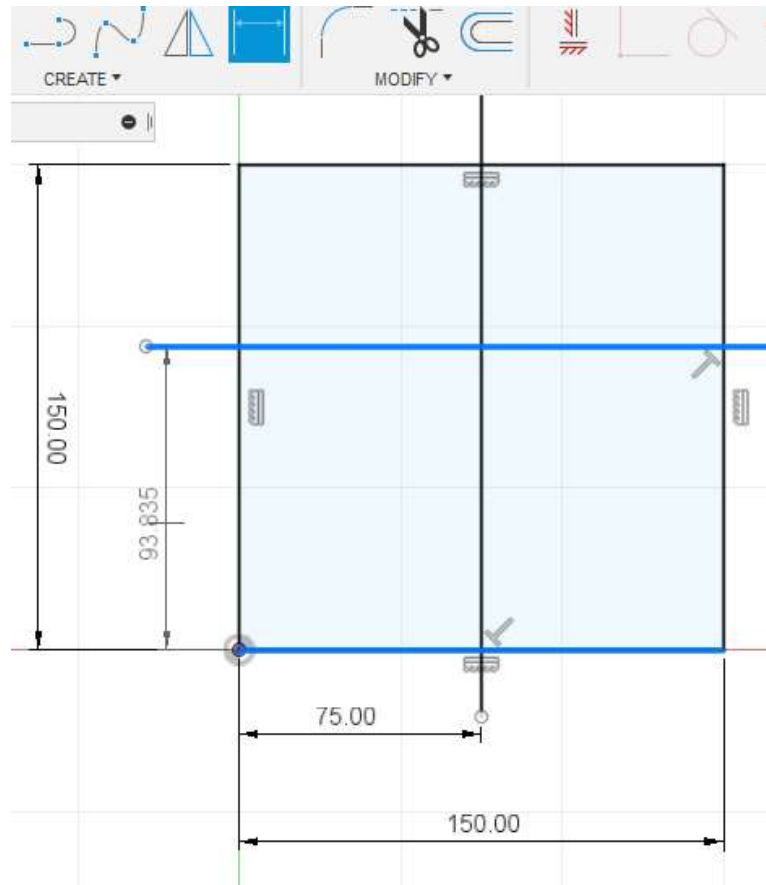
8. With the *Circle* command we can draw the circle and enter it's diameter.
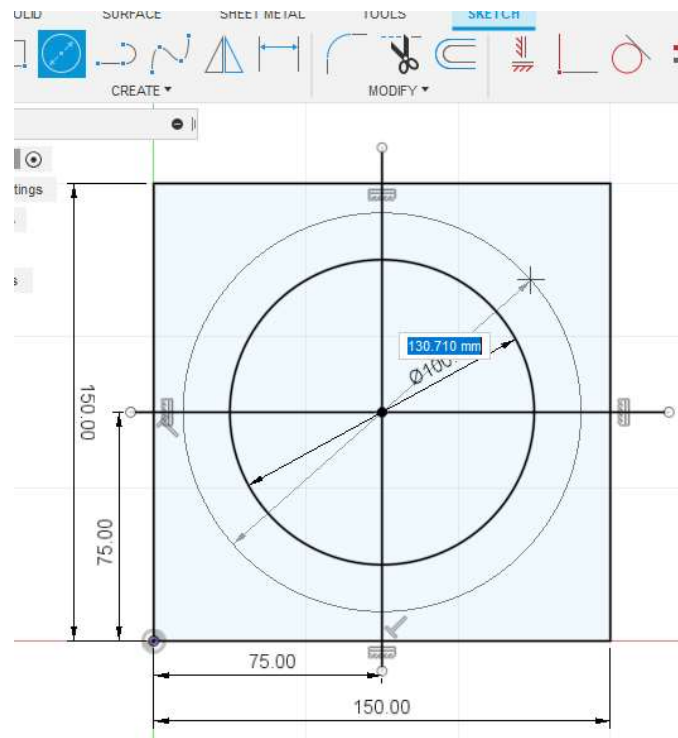
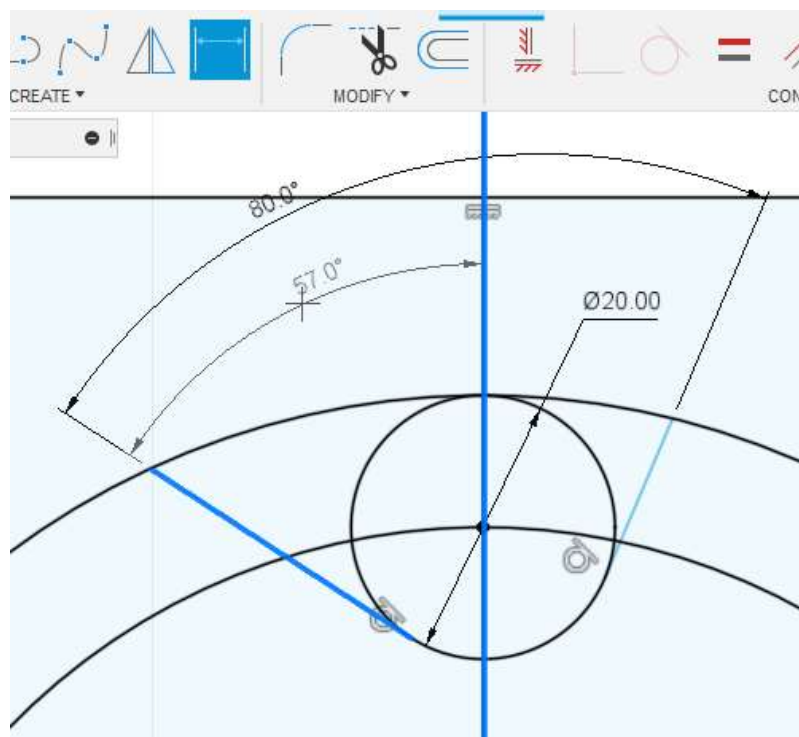9. With the *Line* command it is possible to draw the contour of the model. We can draw all object lines at once. Using just a left mouse click, the command remains active, and we can draw the next desired line. With the *ESC* key we turn off the line command.
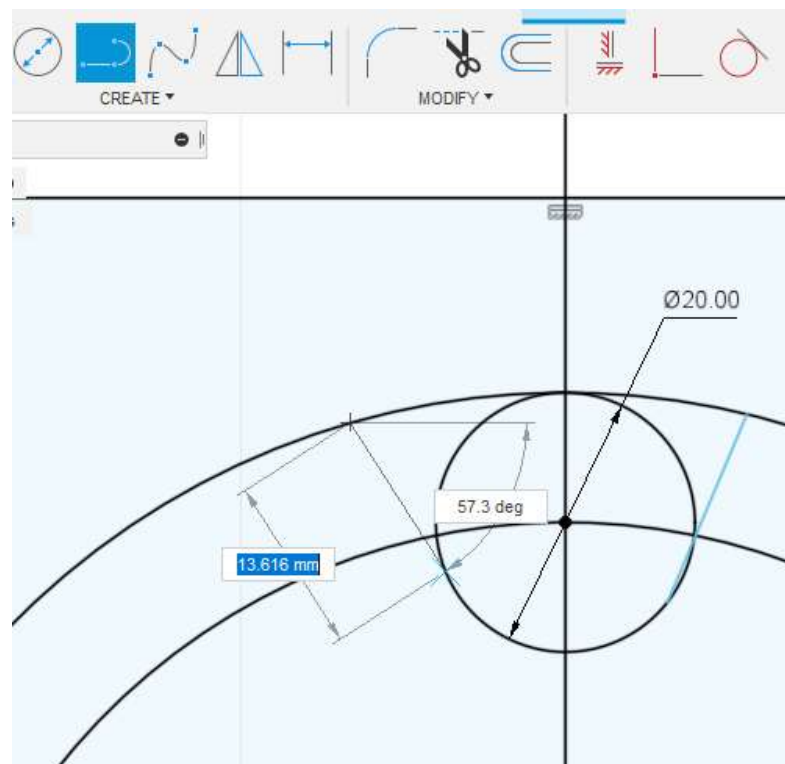
10. Using the *Tangent* command we can place a circle or line tangentially to another circle. First we select the object that needs to stay where we want, later the second object that needs to be moved. In case this doesn't work either, we can use a *Fix* option (padlock) so that our object can't move. Then we repeat the action.
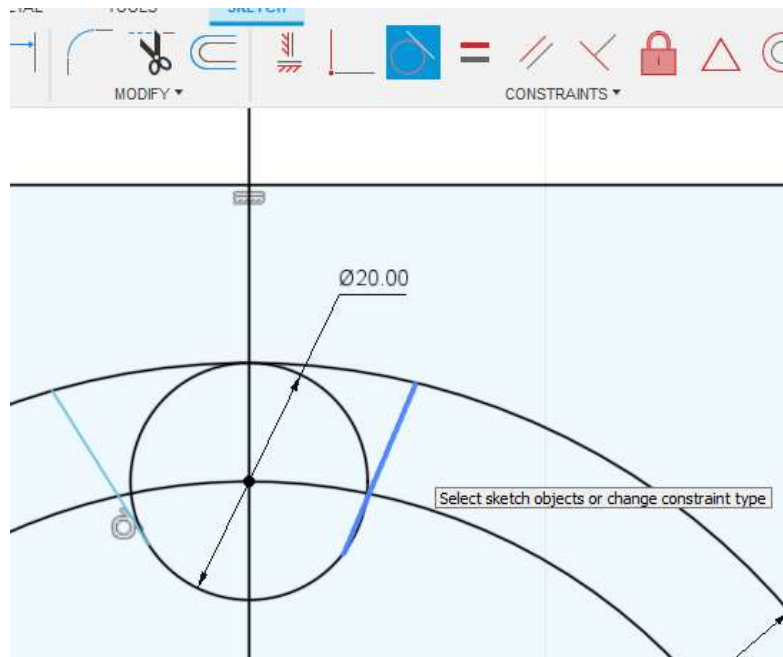

11. After placing all the lines tangently to the circle, we can use the *Dimension* command to determine the angles of inclination.

12. 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).
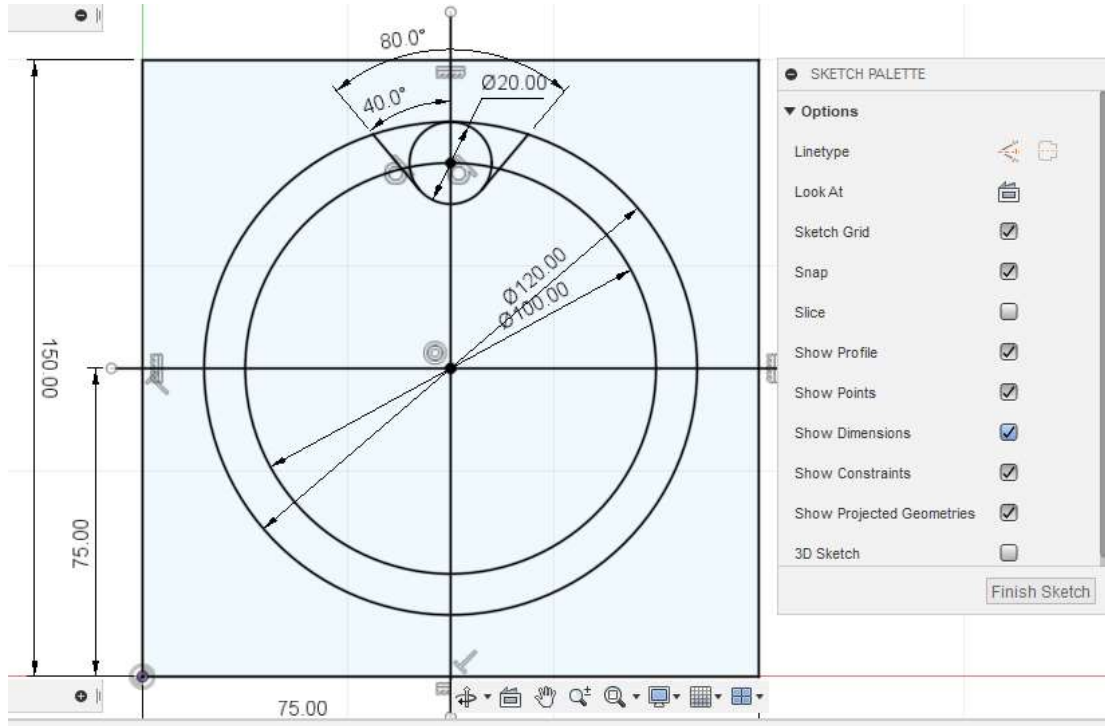

13. By using the *Trim* command it is possible to cut out part of lines or circles that we do not need anymore. If we want to delete an entire auxiliary line or circle, we need to select it first, then use the *Delete* key to delete it.
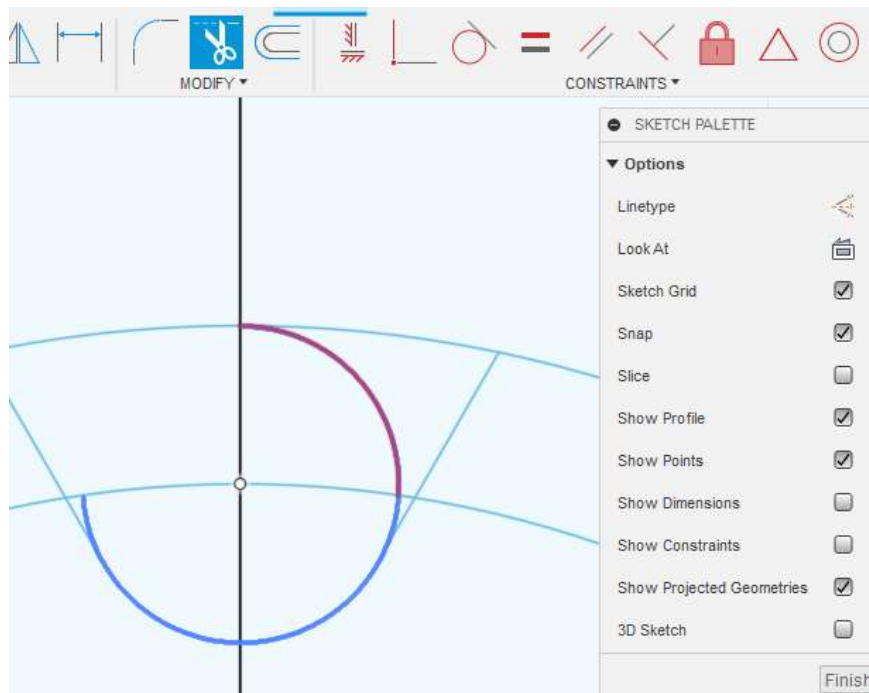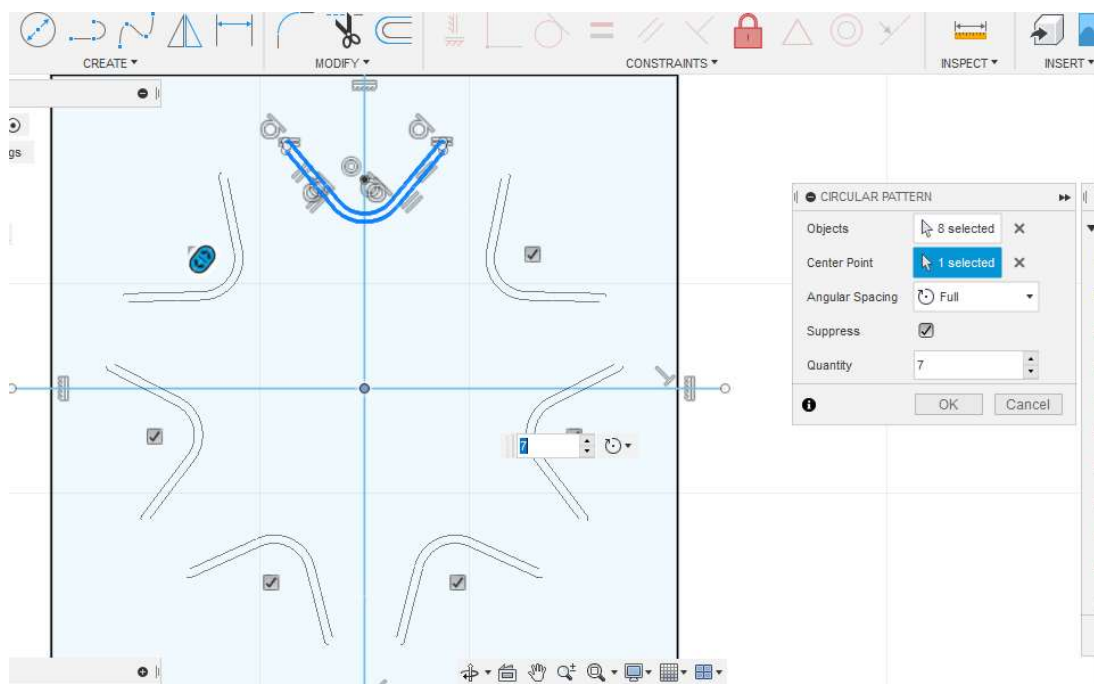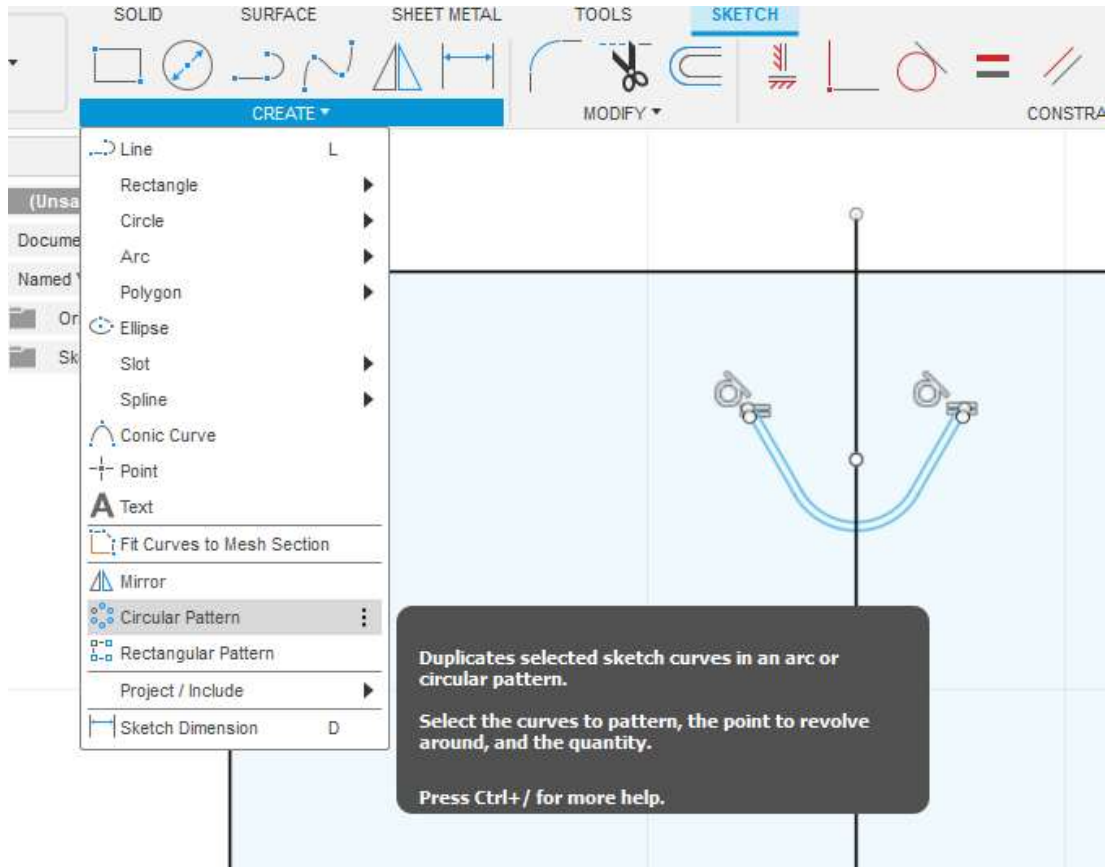
14. By using the *Offset* command it is possible to copy curves, which can speed up work during the drawing process.
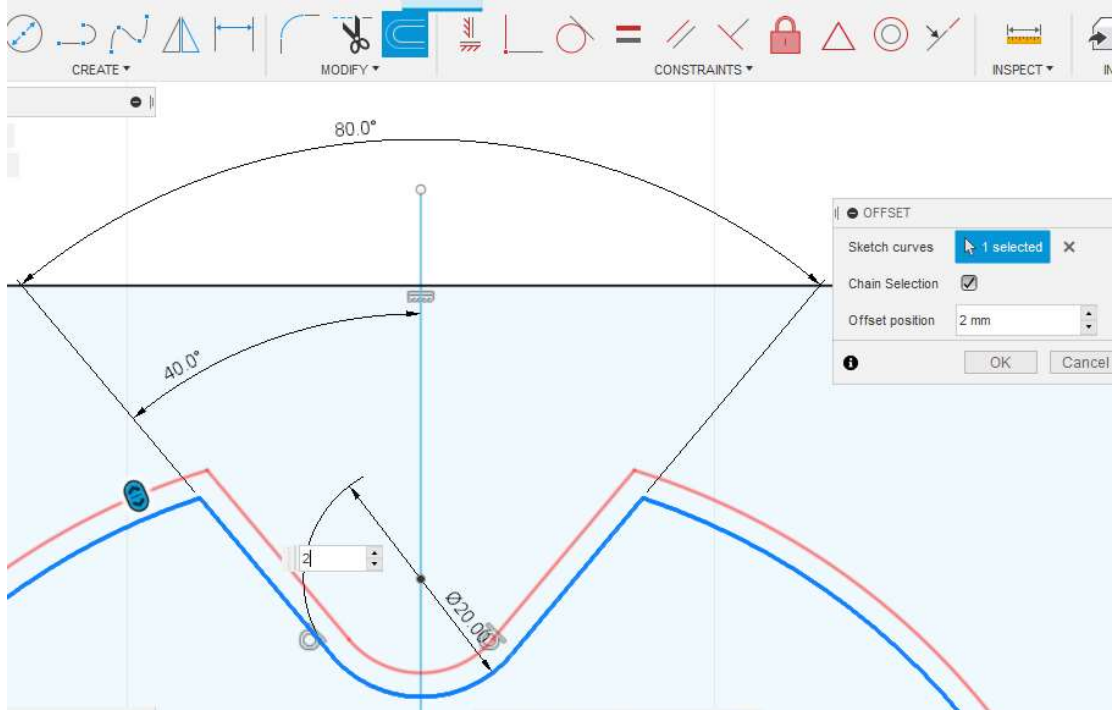
15. The *Fillet* command gives us the possibility to draw the radius between two lines.
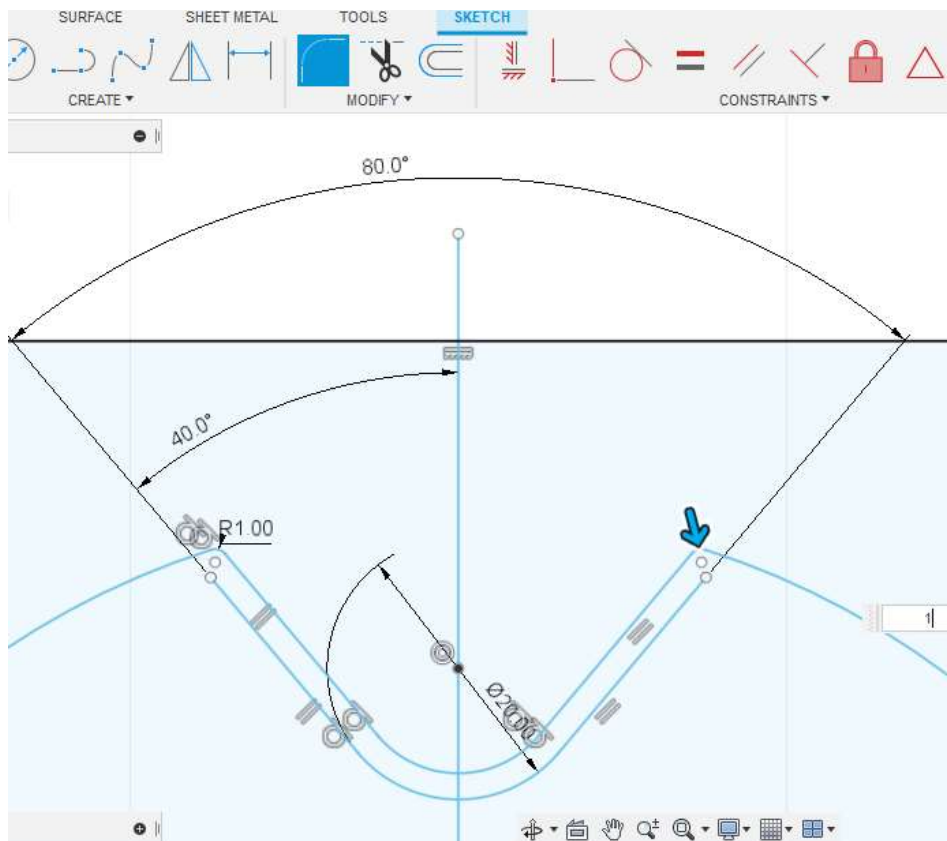
16. With the *Circular pattern* command we can copy an object multiple times around an axis or pivot point.

17. 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).


18. 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.


19. An example of the isometric view of the drawing model:
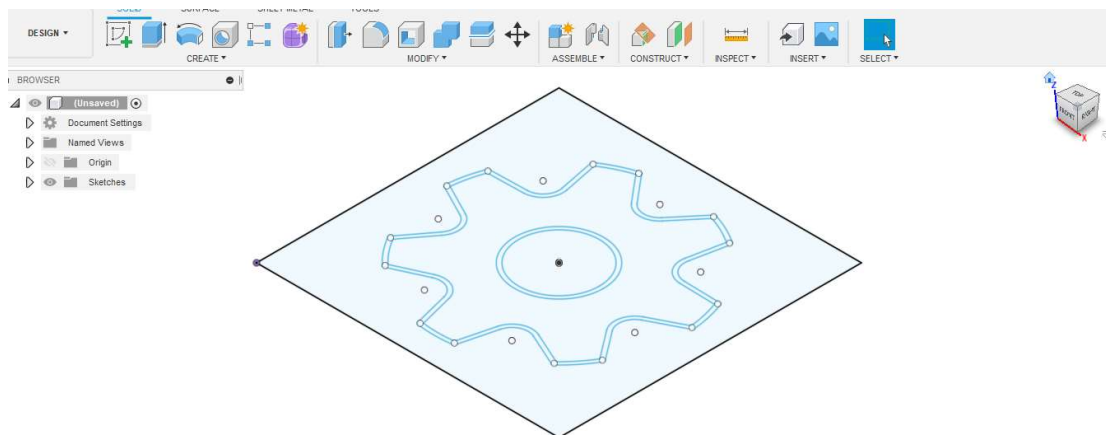
20. With the *Extrude* command we build the 3D model (pull the surface into space). It is necessary to choose closed surfaces. By entering the values or by pulling the blue arrow we define the thickness of the border.
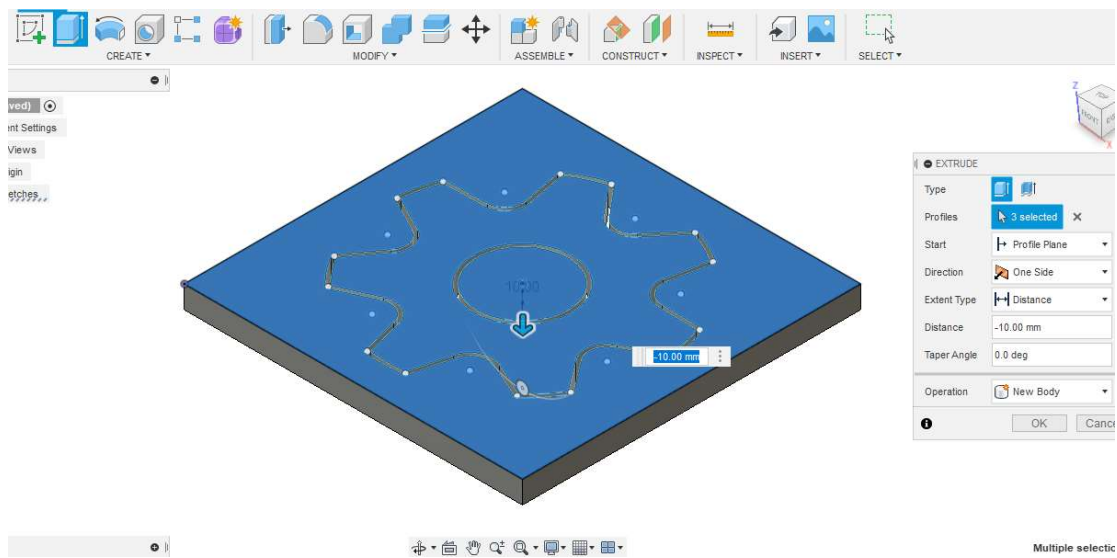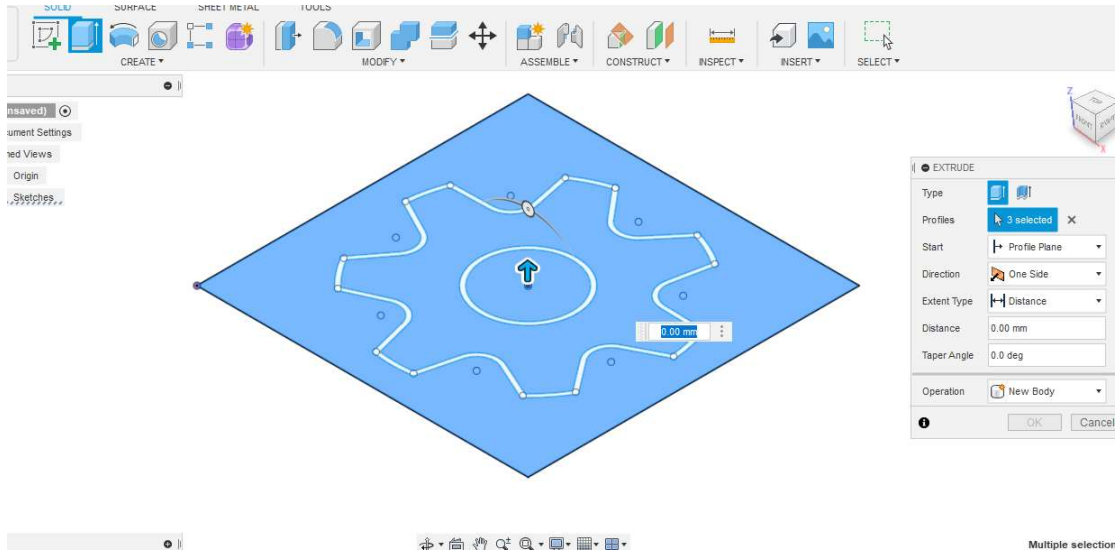
21. By using the *Point* command we define starting and ending point for simulation and generation of NC code.
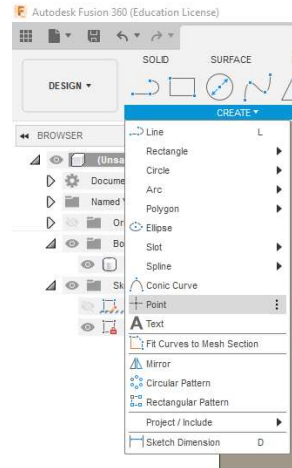
22. After creating of the 3D model the body of the drawn part is visible, but the sketch is not. To show the sketch again we need to click on the eye icon and everything that is drawn in this sketch will be visible in drawing space. If there is need for change of some part of the sketch by double clicking on the sketch it is possible to enter the sketch editing mode.
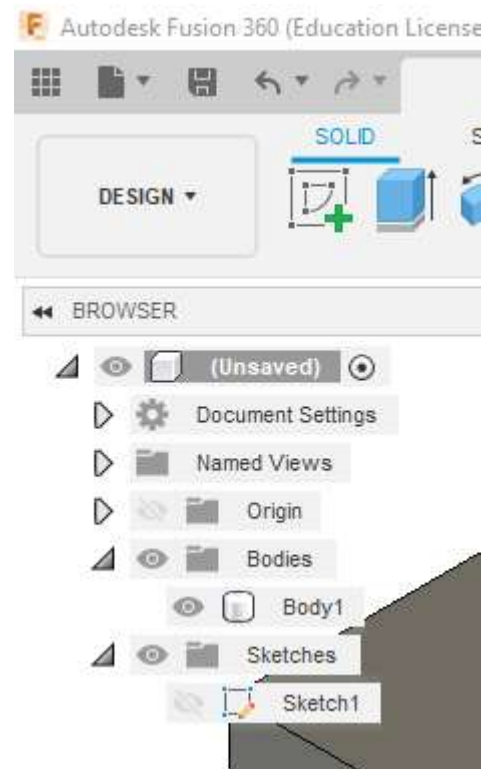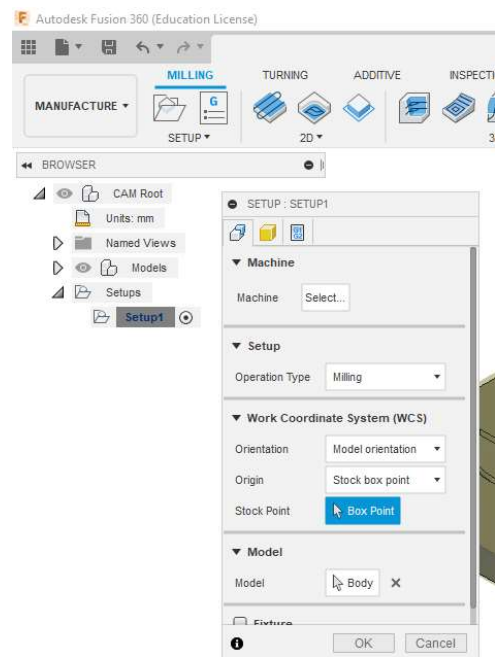
23. From the **Design** menu, by choosing the **Manufacture** option, it is possible to switch to the mode for making of the simulation and generating the NC code.
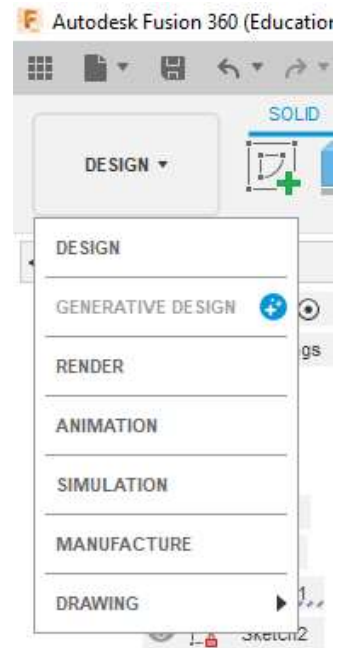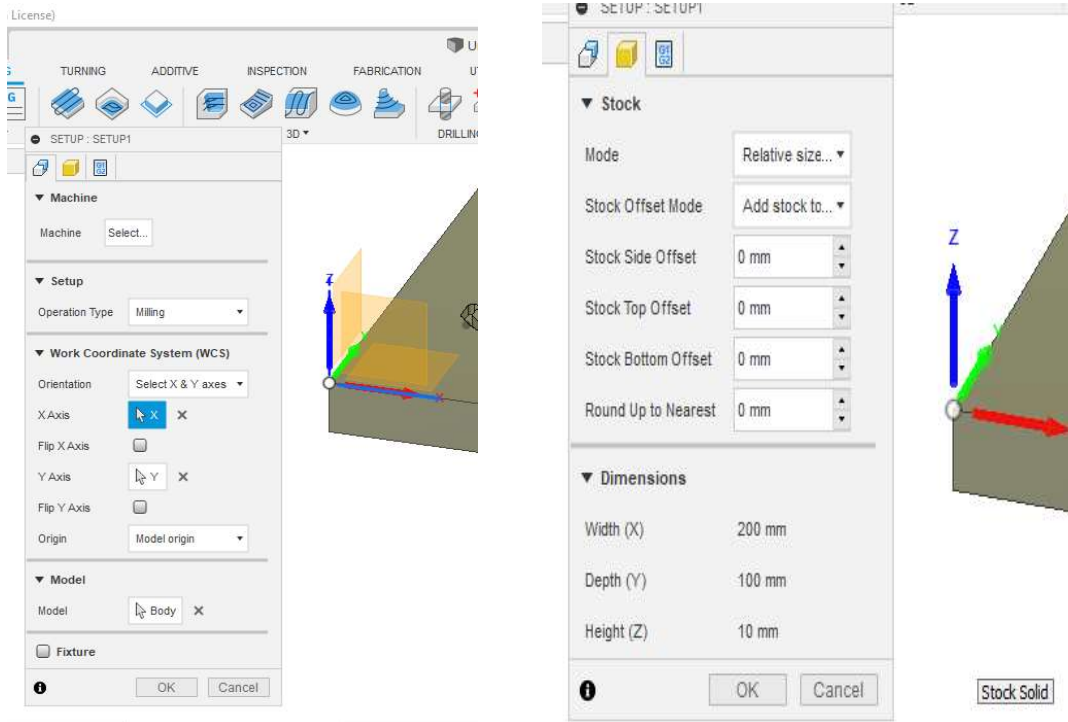
24. Upon entering the manufacture mode there are new options available.
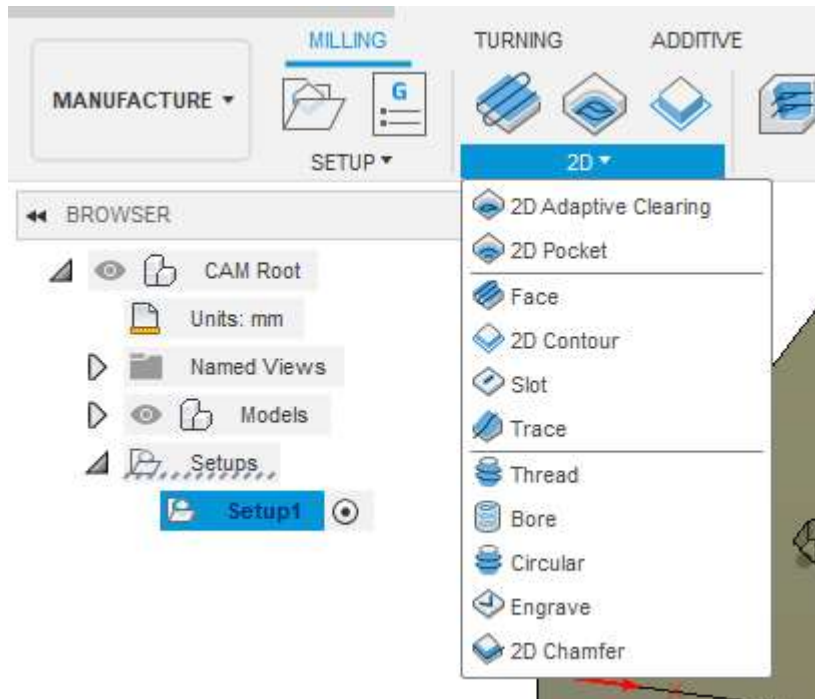
25. By clicking on the **Setup** command new dialog window opens, in which we can set up the initial material and define the coordinate system.
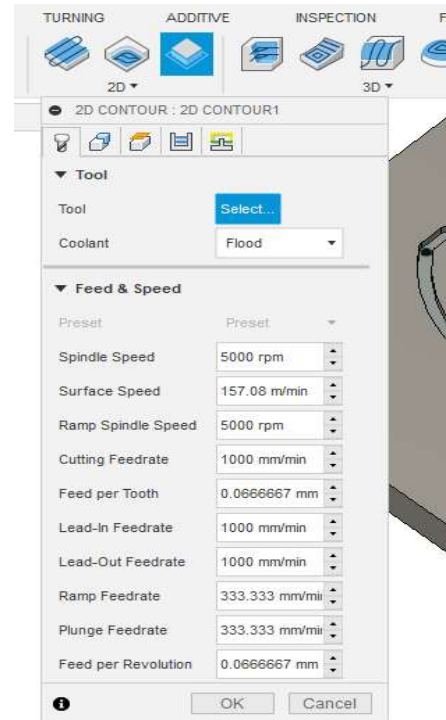
26. A milling operations menu:

27. Choosing the **2D contour** milling option opens the window for setting up the cutting speed and ways of entering the cutting tool into the material. It is necessary to mark the contour on the 3D model. To finish the operation we need to choose the cutting tool.

28. 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. different diameters of cutting tool).
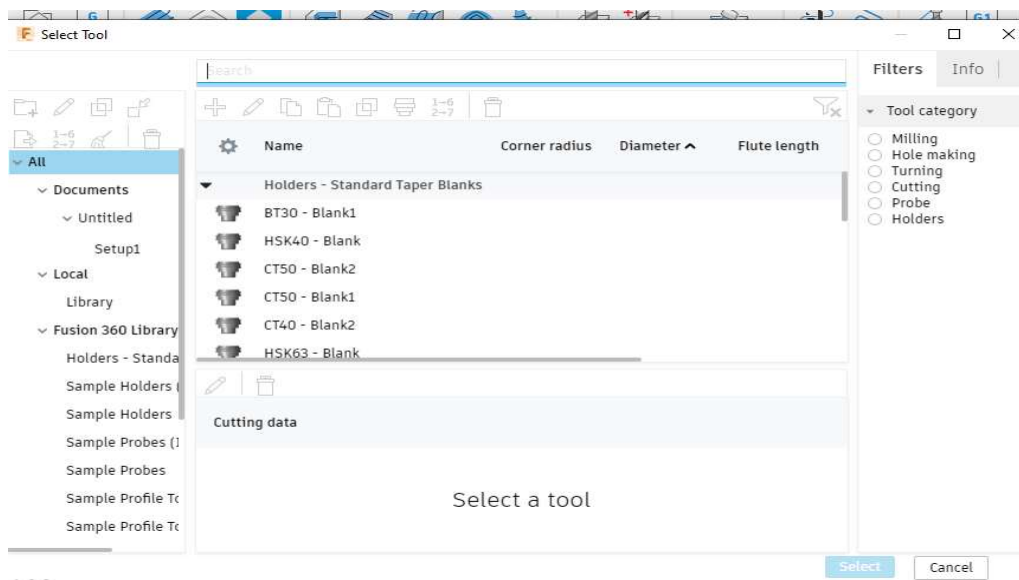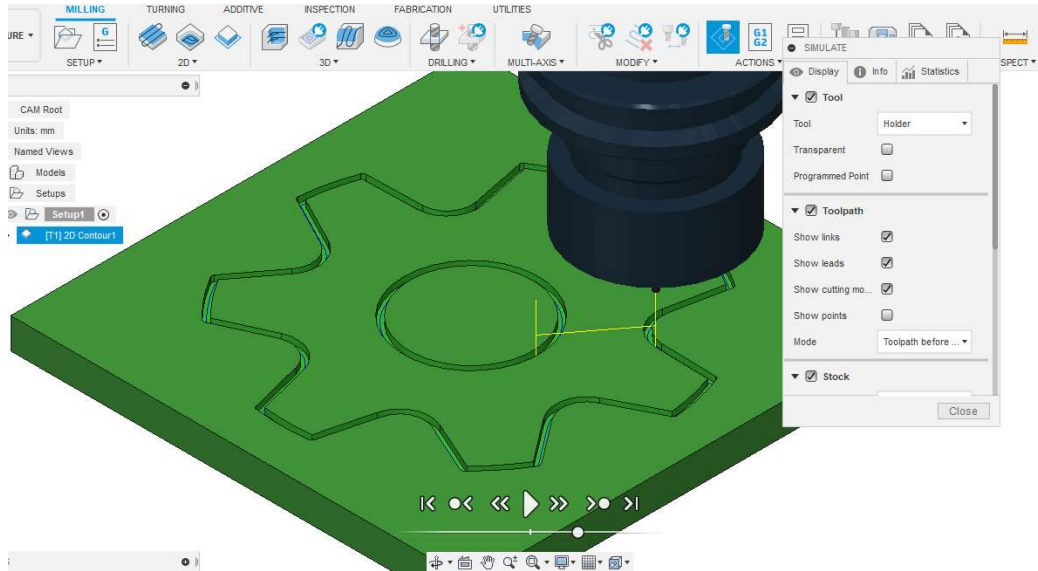
29. By clicking on the right mouse button over the option and selecting **Edit Tool** command we can change cutting tool and tool holder dimensions.
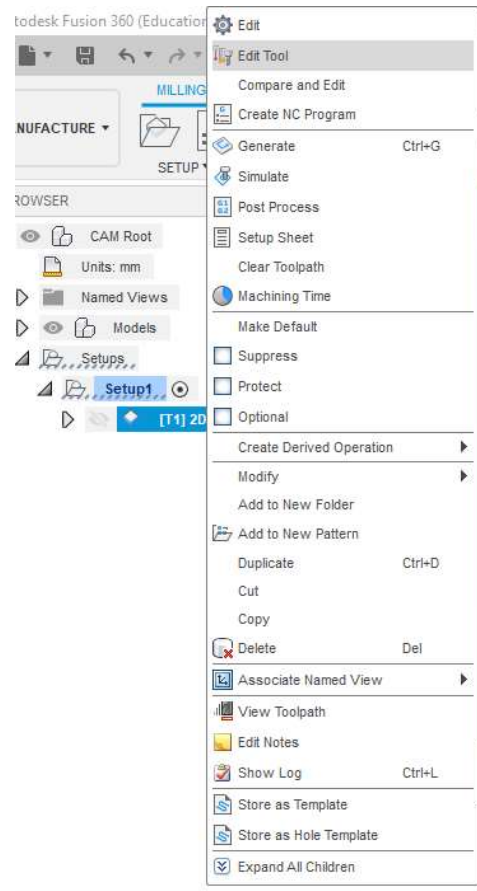
30. After setting all of the necessary options the simulation can be started by choosing the *Simulate* command from the *Actions* menu.

31. 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.
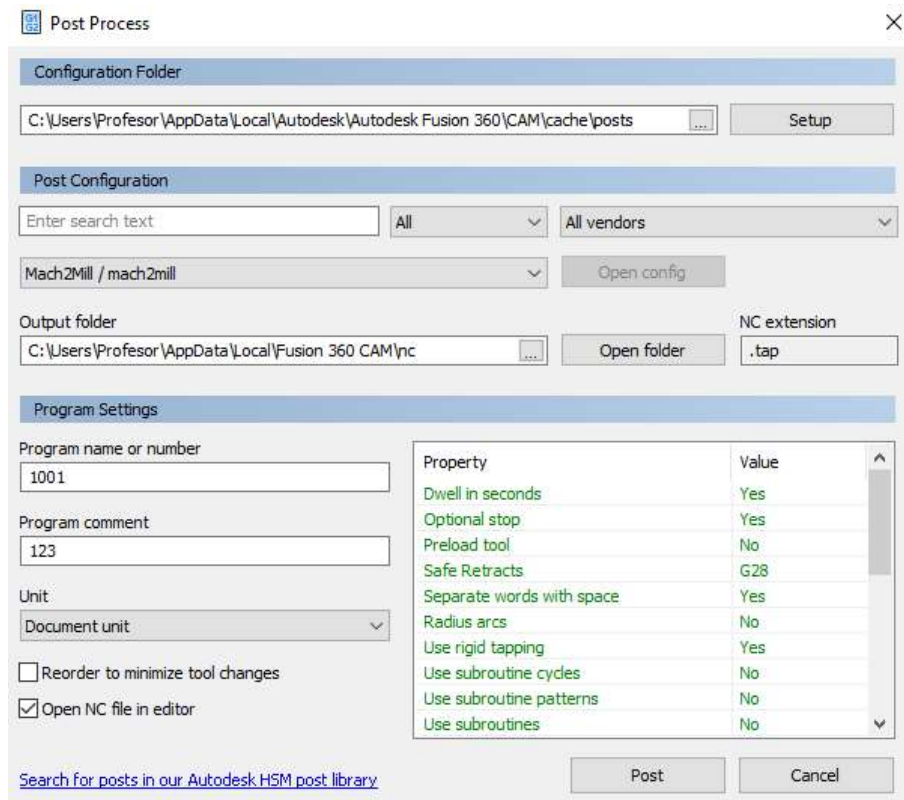
32. After all the settings, clicking on *Post* button opens a new window with the generated NC code, which we have to clean before we transfer it to the machine controller.