



Juraj Dobrila
University of Pula



Co-funded by the
Erasmus+ Programme
of the European Union

3D and VR for VET



InSciEd



Industrijsko-obrtnička
škola Pula

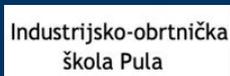
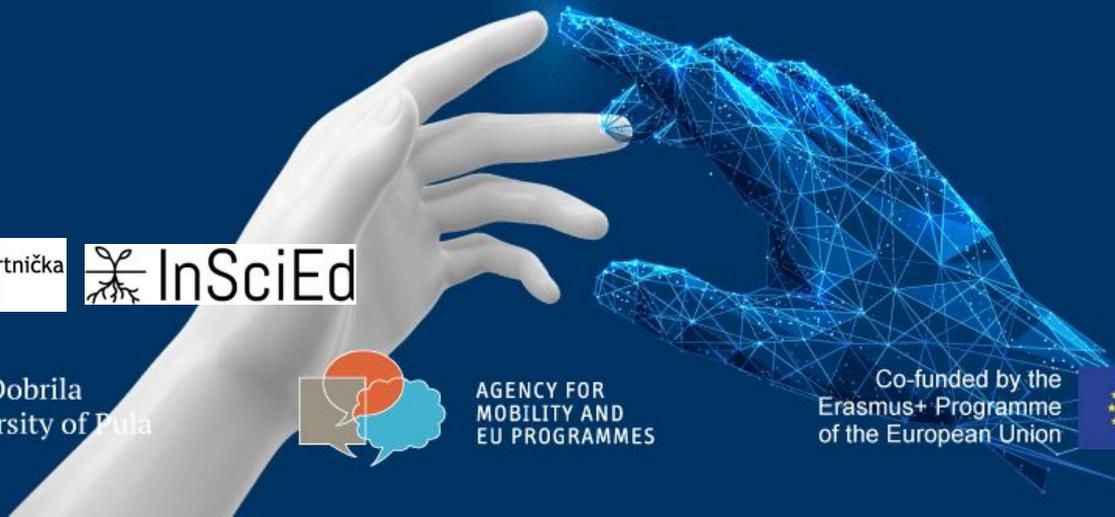
USN Universitetet
i Sørøst-Norge



3D and Virtual Reality Technologies for VET
Project number: 2019-1-HR01-KA202-06100



AUTODESK® FUSION 360™



Juraj Dobrila
University of Pula

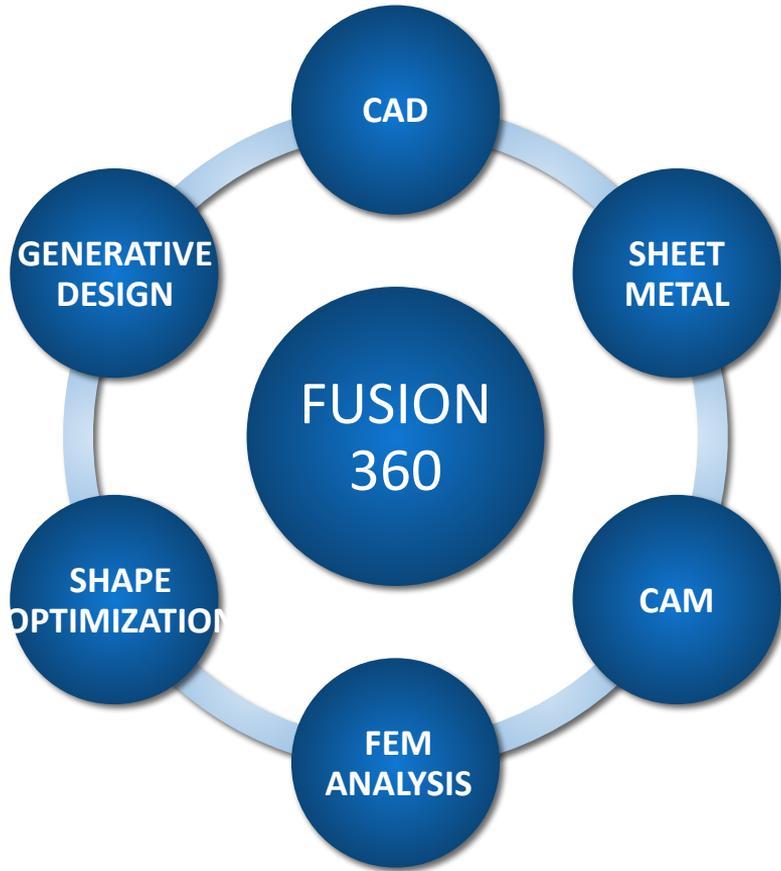


AGENCY FOR
MOBILITY AND
EU PROGRAMMES

Co-funded by the
Erasmus+ Programme
of the European Union



WHAT IS FUSION 360?



CAD MODELING

- Fusion 360 main functionality is as a CAD modeling software. CAD is an acronym for „Computer-aided Design“. With the help of CAD softwares, we can create virtual 3D models, that help us visualize and rapidly prototype new or existing products.

Sheet Metal

- The sheet metal functionality is located under the CAD module, but in a separate section, and allows the 3D modeling of sheet metal components. This is a specific tool, that allows us to create a flat pattern of the model, once created, in order to have it immediately prepared for laser or plasma cutting. The model can be saved in different formats such as dwg. or dxf.

CAM

- CAM is another significant built in functionality of Fusion 360. The acronym CAM stands for „Computer-aided manufacturing“. With this module of work we can simulate CNC machining and other manufacturing process and create a G-code that is compatible and ready to use with our machine.

FEM ANALYSIS

- The “finite element analysis” is used to make different kind of simulations on our model, such as put an imaginary static or dynamic load on the model and see how it will react. With this results we can easily spot the weak points of the model, but also the segments that are maybe over engineered for the required load. In the same way we can test the model with thermal load, fluid mechanics, and even vibrations.

SHAPE OPTIMIZATION

- The shape optimization tool is a good supplement to FEM analysis, and is used to optimize the shape of our model. In other words, with the shape optimization tool, we get rid of the over engineered parts of the model, that add additional unnecessary material and weight to the already created model.

GENERATIVE DESIGN

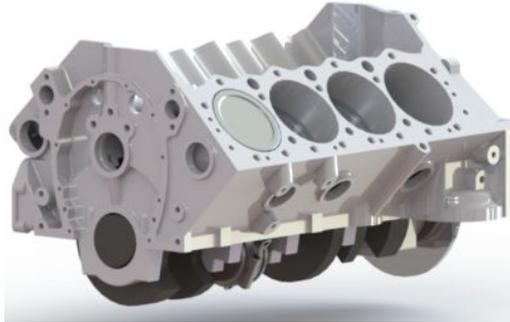
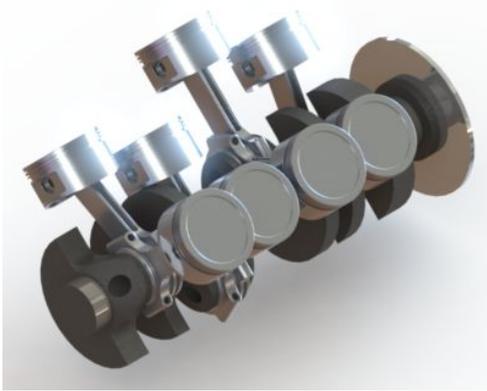
- In a way, generative design is very similar to the shape optimization tool. It is also used to create an optimized model (with the lowest possible mass and highest stiffness for example). The difference is, that in this mode the models are created with the help of AI, following the various input and constraints that we give. Inputs would be, the material, the load, some geometry that needs to be created, machining techniques and so on. The constraints will be the boundaries of the model, or some geometrical constraints that need to be respected. The program automatically creates many possible iterations that can be achieved using different materials and manufacturing techniques.

CAD MODELING

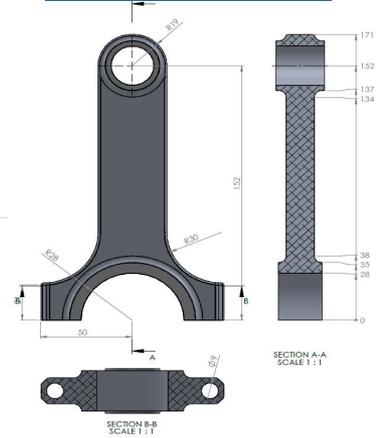
Single parts



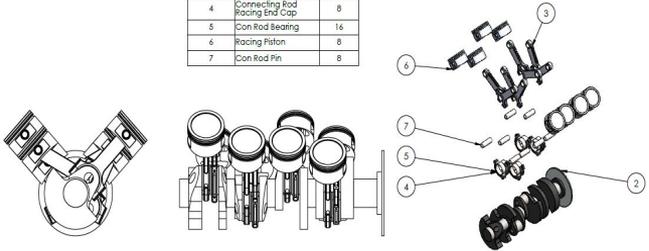
Subassemblies and assemblies



Drawings



ITEM NO.	PART NUMBER	QTY.
1	Block	1
2	Crank Shaft	1
3	Connecting Rod	8
4	Connecting Rod Racing Top End	8
5	Con Rod Bearing	16
6	Racing Piston	8
7	Con Rod Pin	8

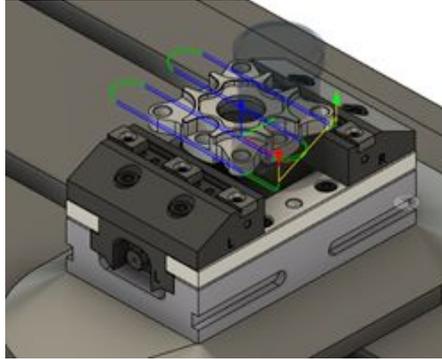


Note: all pictures are taken from the Autodesk Fusion 360 official website: <https://help.autodesk.com/view/fusion360/ENU/>

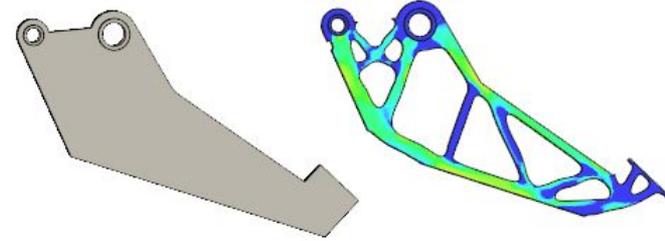
SHEET METAL



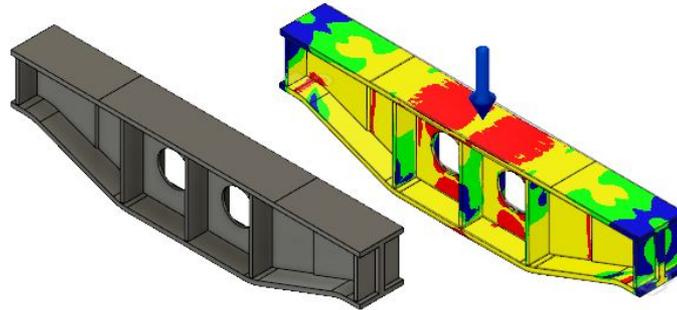
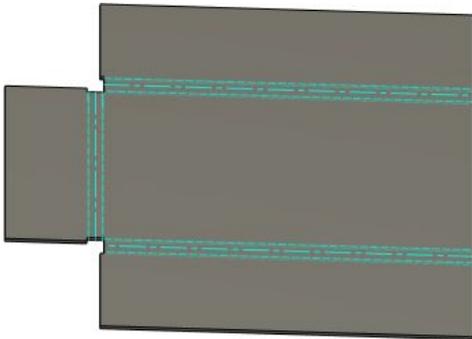
CAM



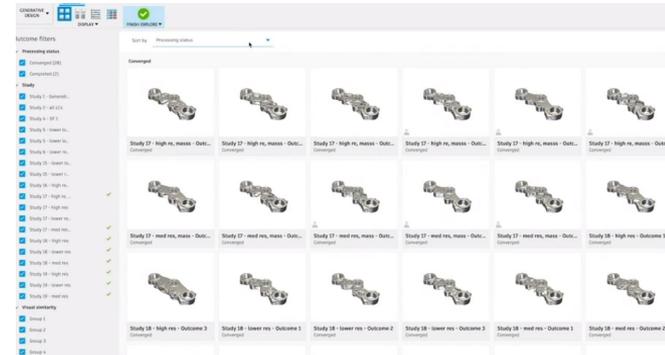
SHAPE OPTIMIZATION



FEM ANALYSIS



GENERATIVE DESIGN



Note: all pictures are taken from the Autodesk Fusion 360 official website: <https://help.autodesk.com/view/fusion360/ENU/>

Advantages of using CAD softwares

Visualization of the final product

Reduced the possibility for errors

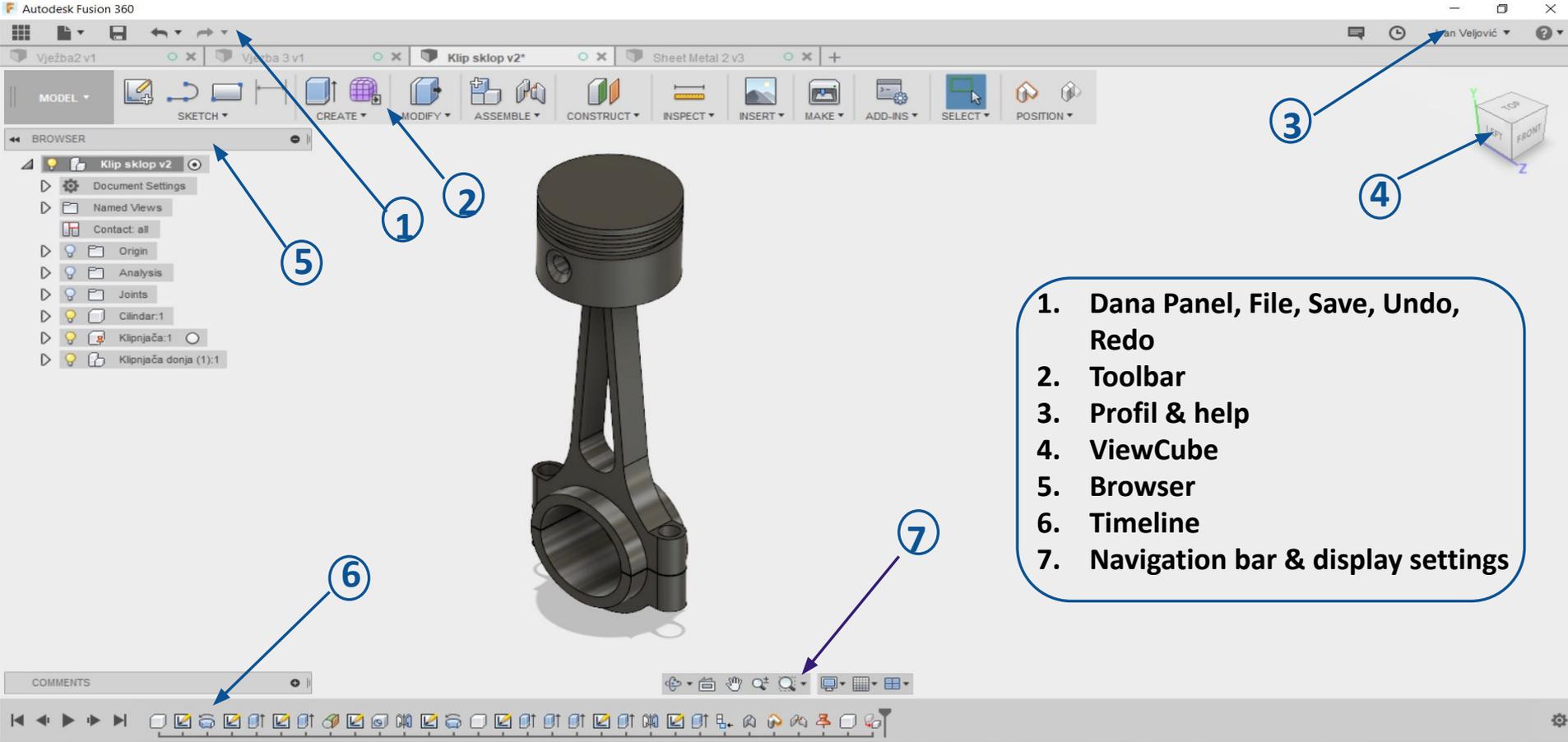
Simulation / G-code generation / 3D printing and manufacturing

Essential tool for rapid prototyping

Possibility of simultaneous work of team members with the help of Cloud

Accelerated production process

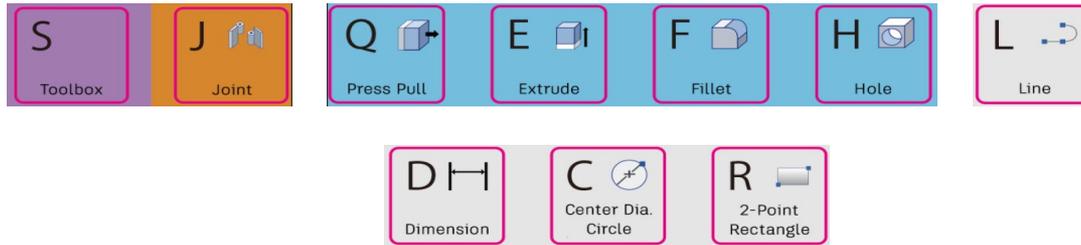
Working interface



1. Dana Panel, File, Save, Undo, Redo
2. Toolbar
3. Profil & help
4. ViewCube
5. Browser
6. Timeline
7. Navigation bar & display settings

Keyboard shortcuts

- Fusion 360 is very flexible. It allows the possibility to access different tools using the tool pallet or with predefined keyboard shortcuts (note that keyboard shortcuts can also be changed to individual needs). The most used shortcuts are represented in the picture bellow.



<http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-F0491540-0324-470A-B651-2238D0EFAC30>

Navigation

- There are 3 main ways of navigating in Fusion 360: using the navigation bar / View Cube / Mouse

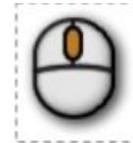
Navigation bar



View Cube



- Click on different corners, edges or faces to select different views of the model.

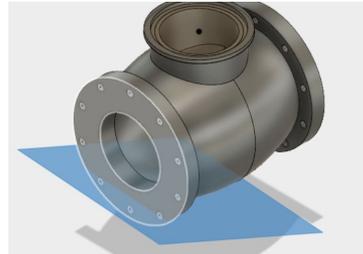
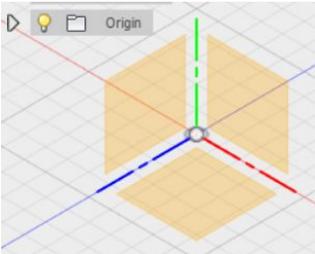
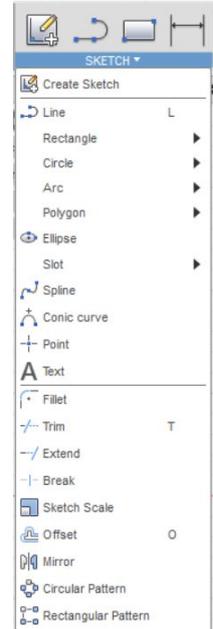
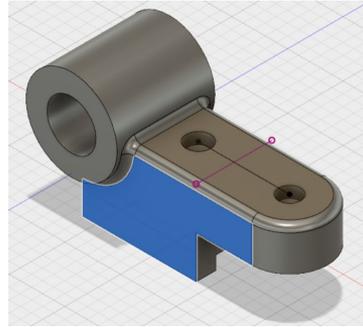


Mouse

- Zoom: Scroll up or down with the middle mouse button
- Pan: Shift + middle mouse button and move the mouse
- Rotate: Middle mouse button and move the mouse

Sketch

- Sketch is used to create a 2D contour from which a 3D model will be created. To start creating a sketch, click on the Create Sketch icon, located on the upper left corner of the tool pallet, then select a plane or a face on which to start the sketch.

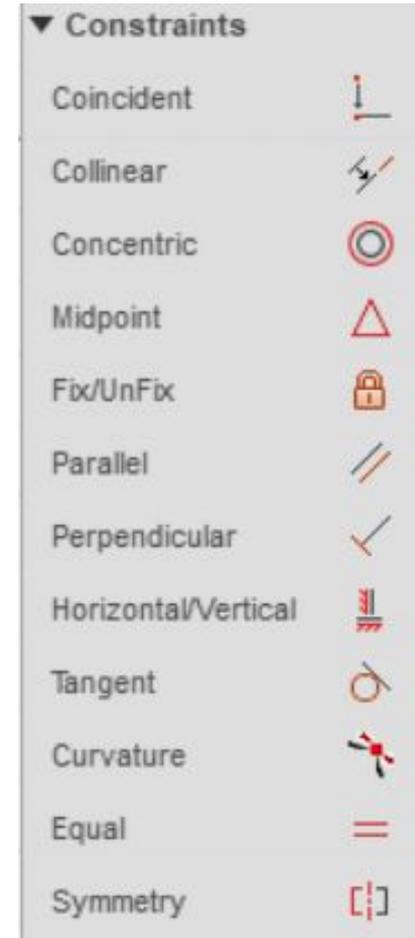


- In the sketch pallet tool we have many different options to use. The most used ones for creating contours are:
Line /Rectangle /Circle /Arc / Ellipse / Spline
Other commands that help us with making our work easier are:
Mirror - used to create a mirror image of a sketch
Pattern - duplicate a specific pattern for a number of times
Offset - create an offset of a sketch, moved for a specified distance
Fillet - round the sharp edges
Trim - brake and delete around an intersecting line

Sketch - constraints

- The constraint commands are used to give us certain relations between points, lines, curves and circles.
Also with these commands we can take away the degrees of freedom of movement, for example we can limit that the line can change only along its length along the horizontal axis.
The most used types of constraints are:

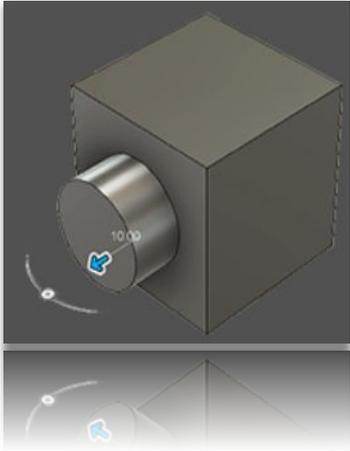
1. **Coincident** - serves to match points, lines or curves
2. **Collinear** - that two or more lines lie in the same direction
3. **Concentric** - that circles or arcs share the same center
4. **Midpoint** - we center the center point of a line with some other parameter (for example with the endpoint of another line)
5. **Fix / UnFix** - with this command we lock a certain contour
6. **Parallel** - makes two or more lines parallel with each other
7. **Perpendicular** - makes the lines perpendicular to each other
8. **Horizontal / Vertical** - sets lines horizontally or vertically with respect to the origin of the coordinate system
9. **Tangent** - assigns a tangent relationship between a line and a curve
10. **Equal** - all lines or curves will be of equal dimensions



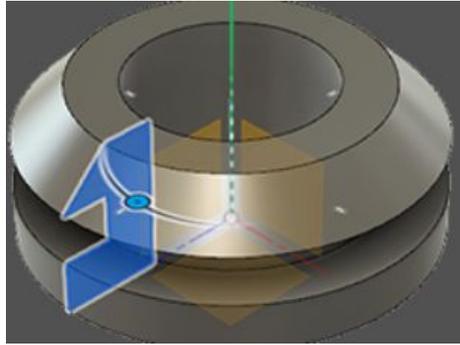
3D modeling tools

- There are 4 main commands used for creating a 3D model, and they are:

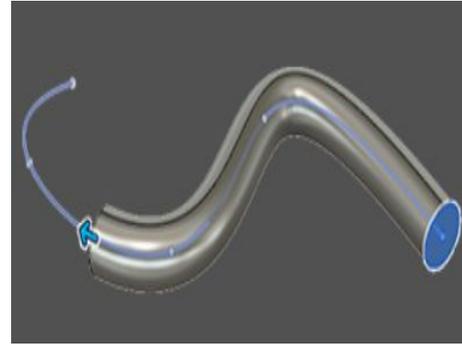
Extrude - gives volume to a closed sketch contour by giving it height.



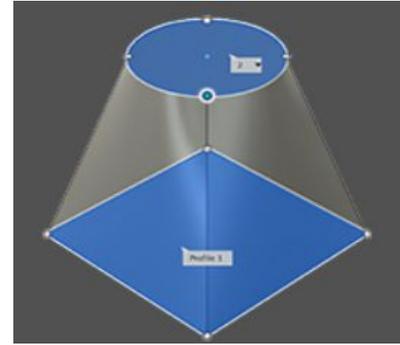
Revolve - gives volume to a closed sketch contour by rotating it around a specified axis.



Sweep - extrudes a contour on a specific path, that was created using another sketch (so two sketches are needed)



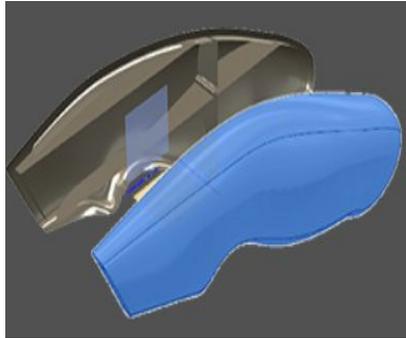
Loft - used to create a 3D model by connecting two contours on two different planes



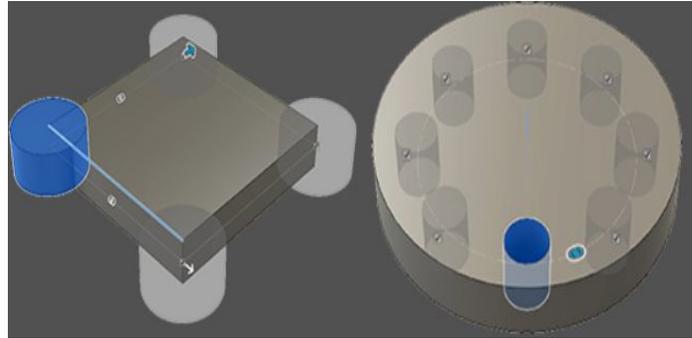
3D modeling tools to increase productivity

- There are many tools that are offered in order to increase the productivity and the flexibility while 3D modeling. Some of the most used ones are:

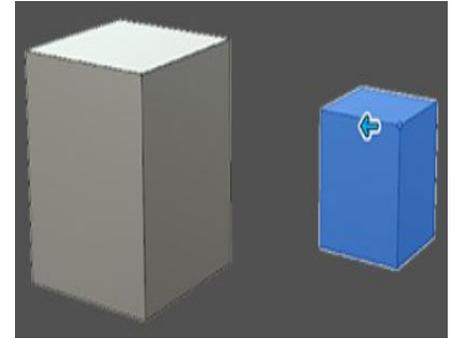
Mirror - we can mirror a sketch, features, face, an entire body or assembly around a plane



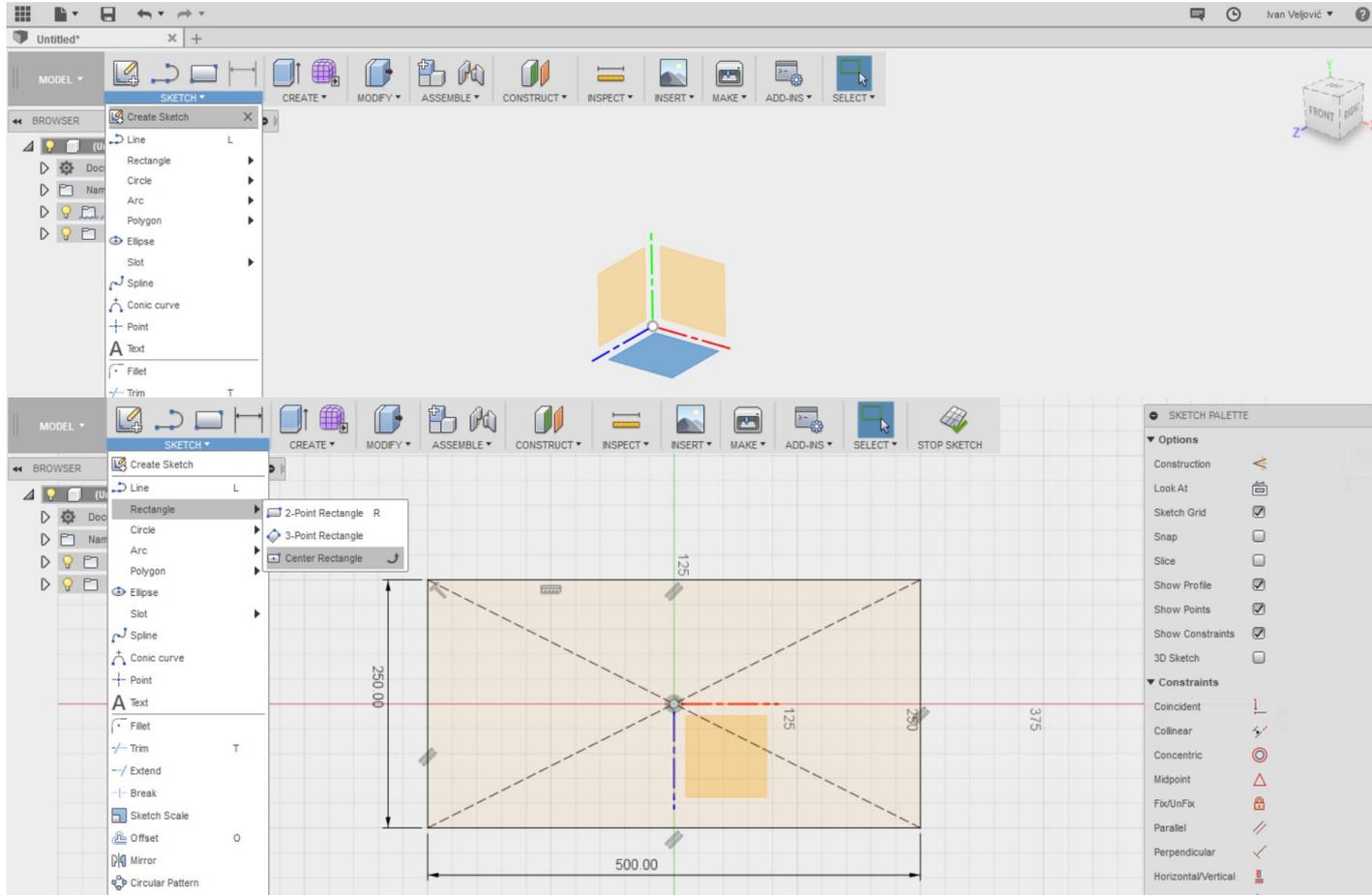
Pattern - can be rectangular or circular. It serves to duplicate a sketch, feature or body a number of times at regular intervals

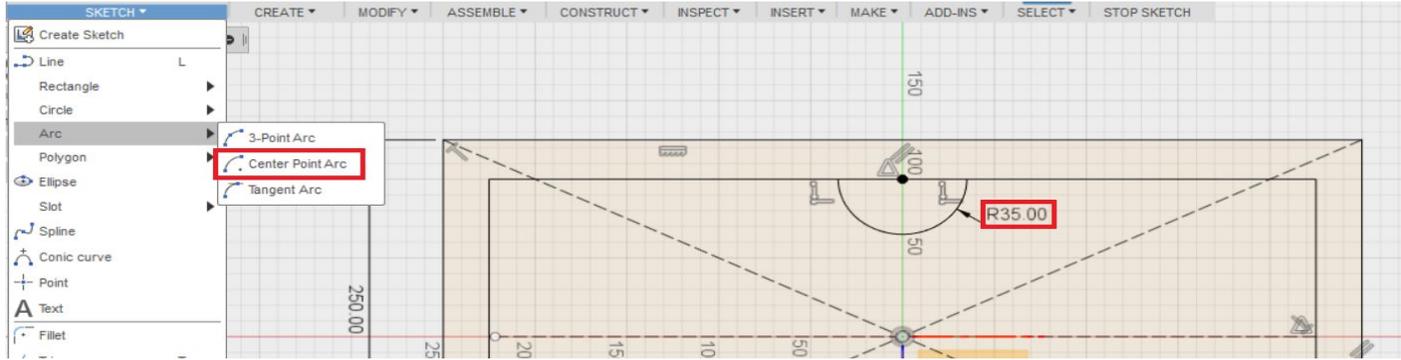
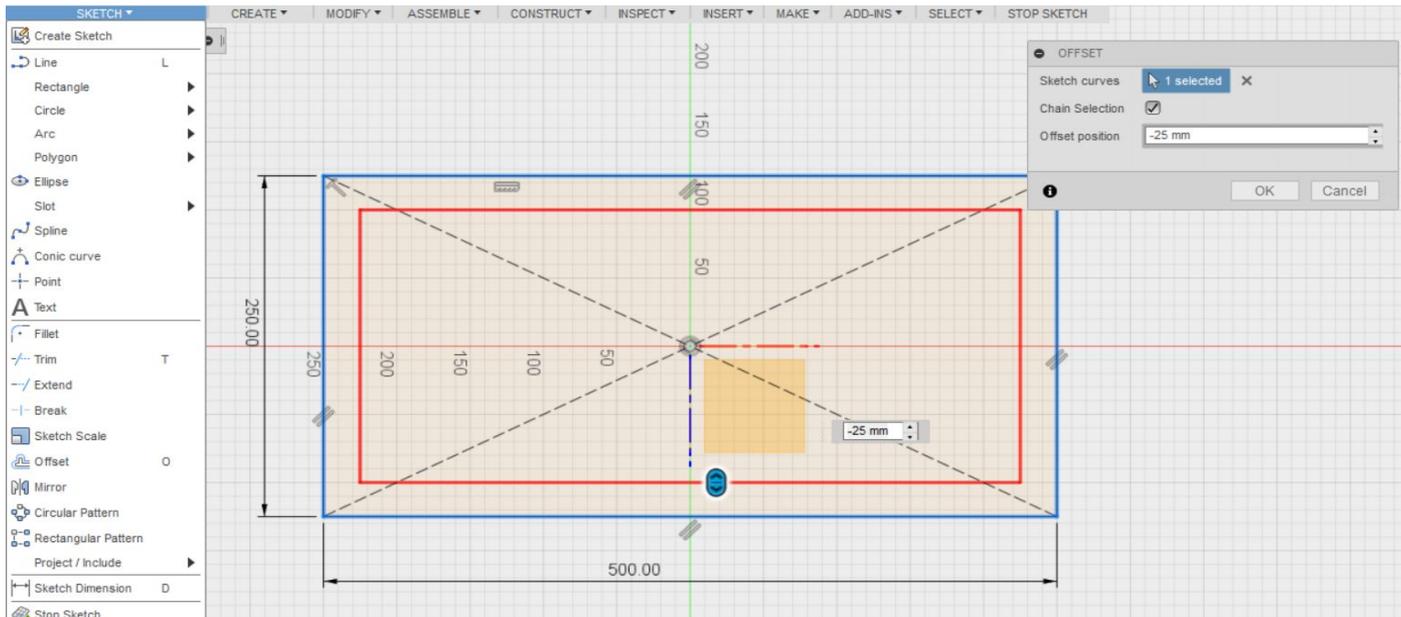


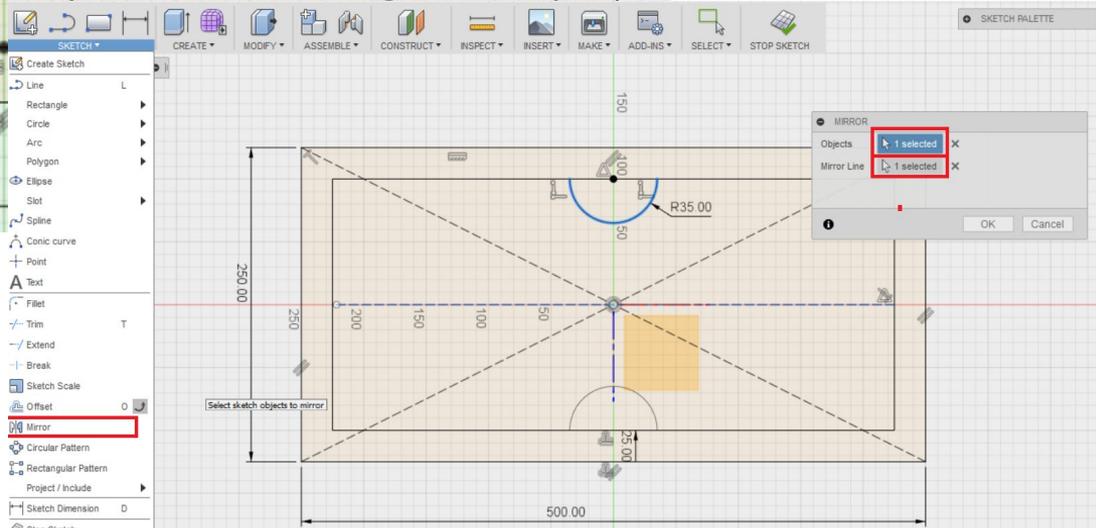
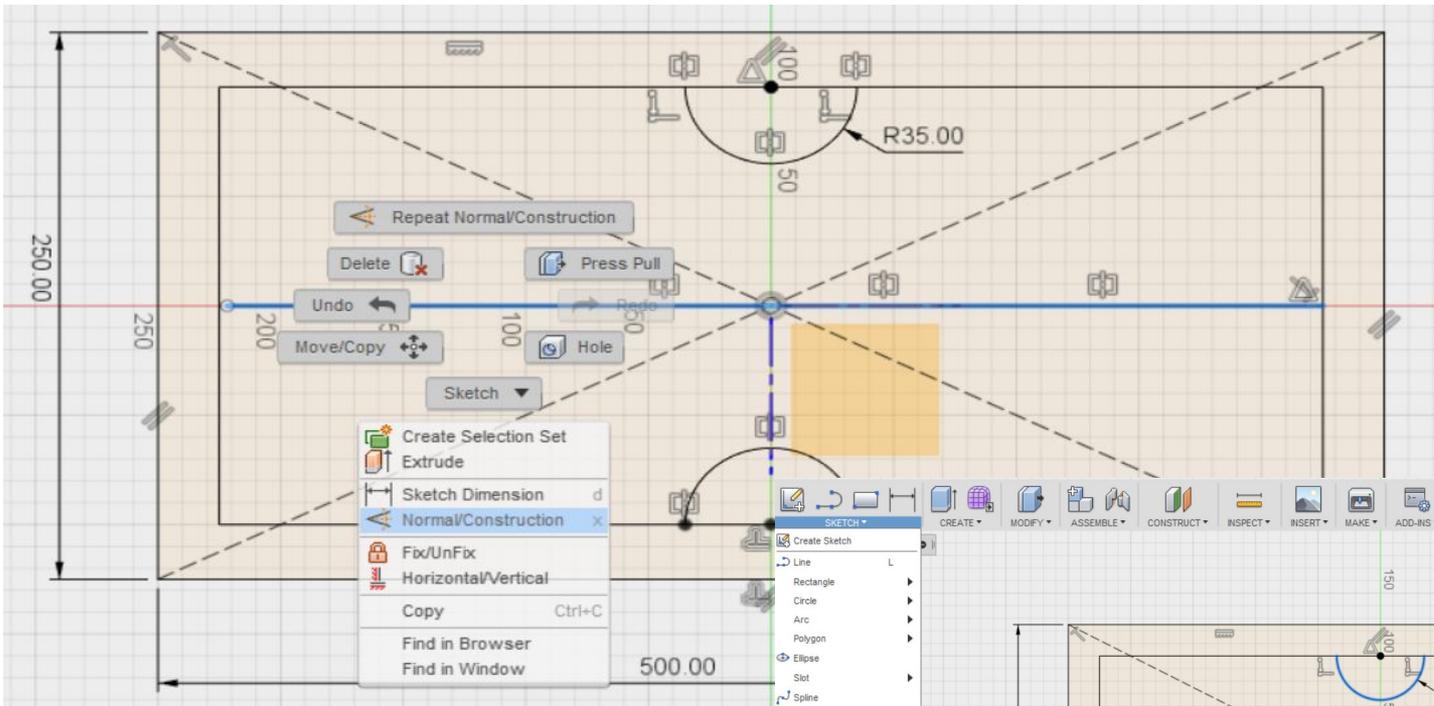
Scale - used to scale an object up or scale it down by a percentage

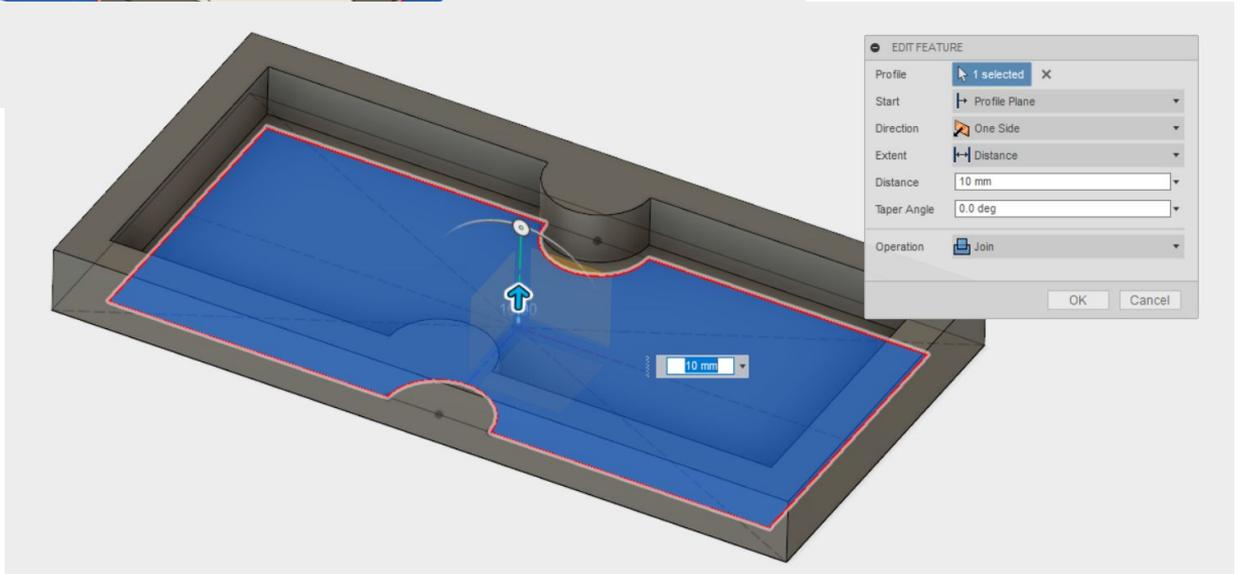
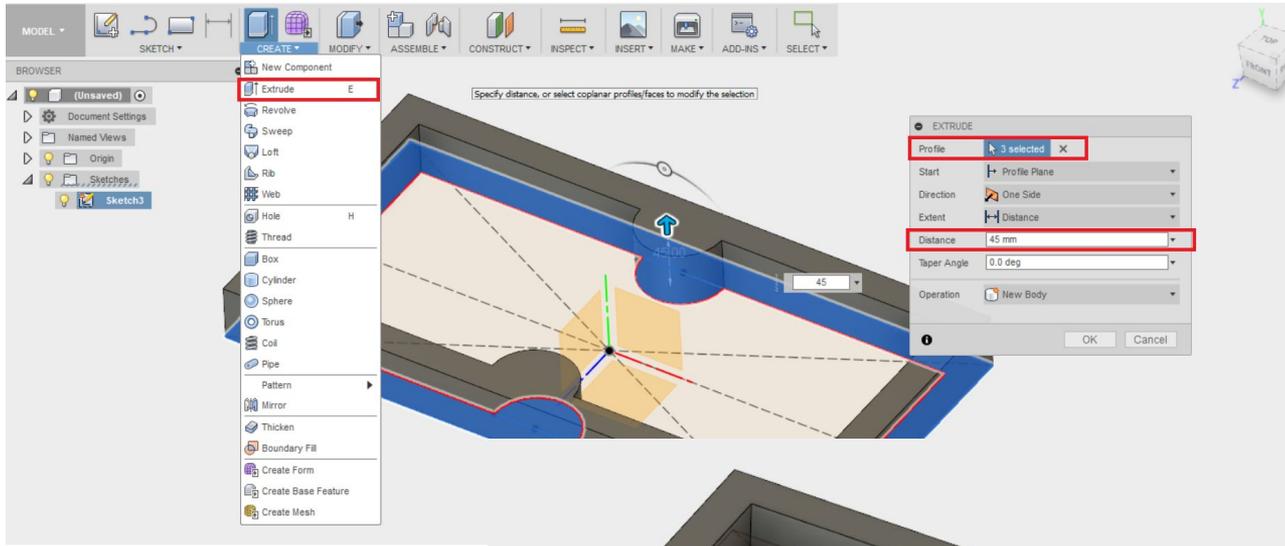


Examples and exercises 1



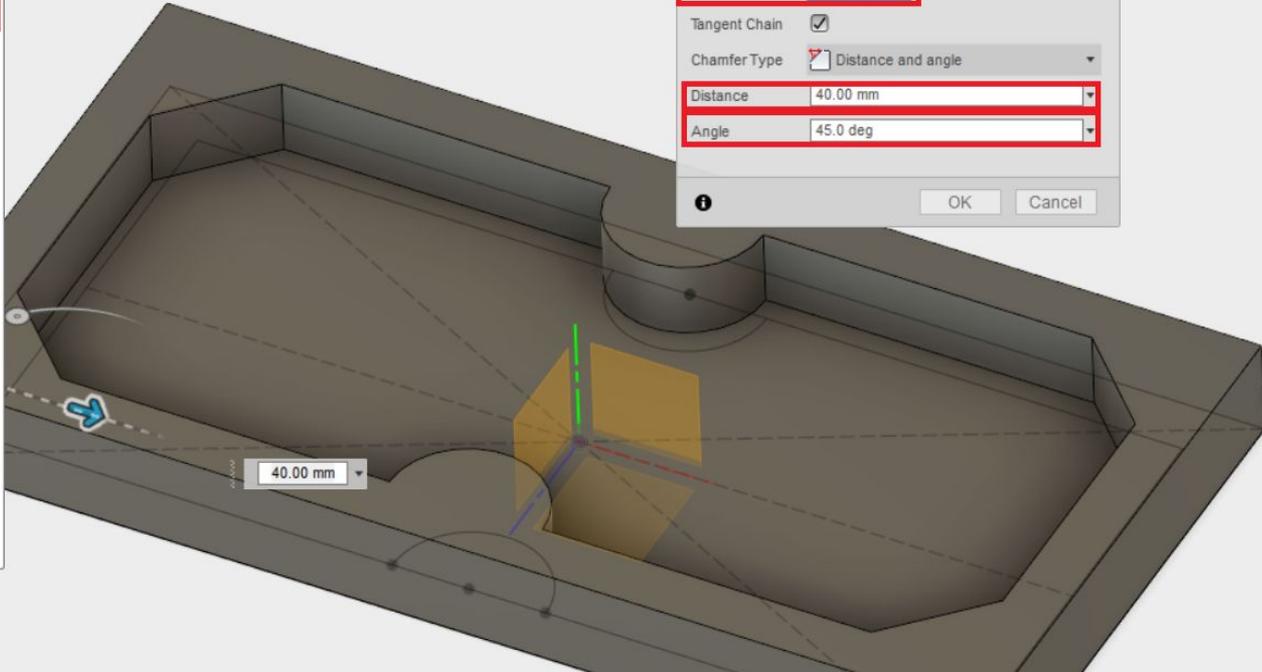








- Press Pull Q
- Fillet F
- Rule Fillet
- Chamfer**
- Shell
- Draft
- Scale
- Combine
- Replace Face
- Split Face
- Split Body
- Silhouette Split
- Move/Copy M
- Align
- Physical Material
- Appearance A
- Manage Materials
- Delete Del
- Compute All Ctrl+B
- Change Parameters



CHAMFER

Edges 4 selected

Tangent Chain

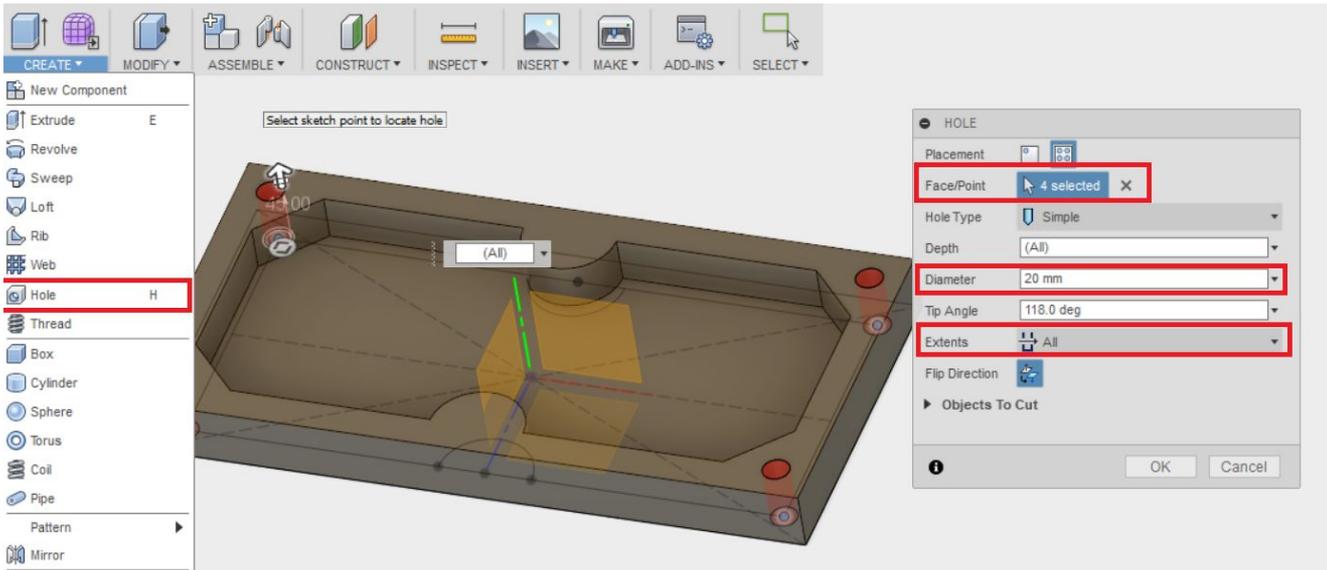
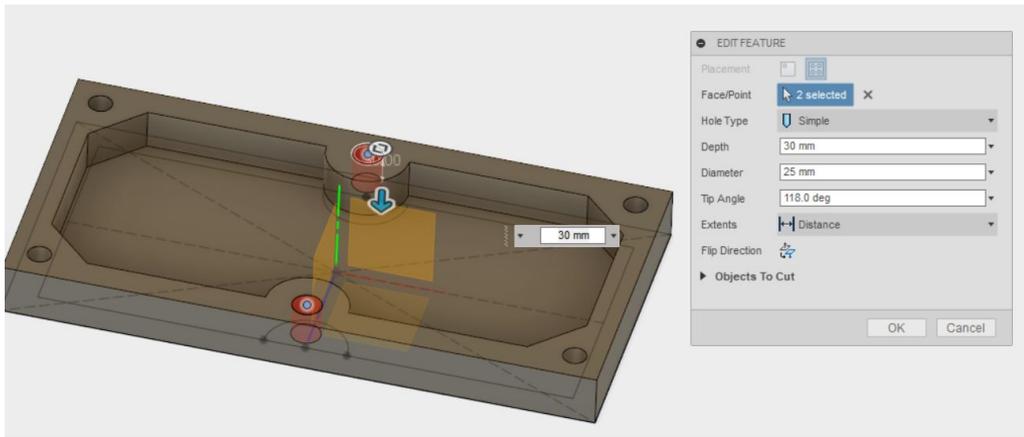
Chamfer Type Distance and angle

Distance 40.00 mm

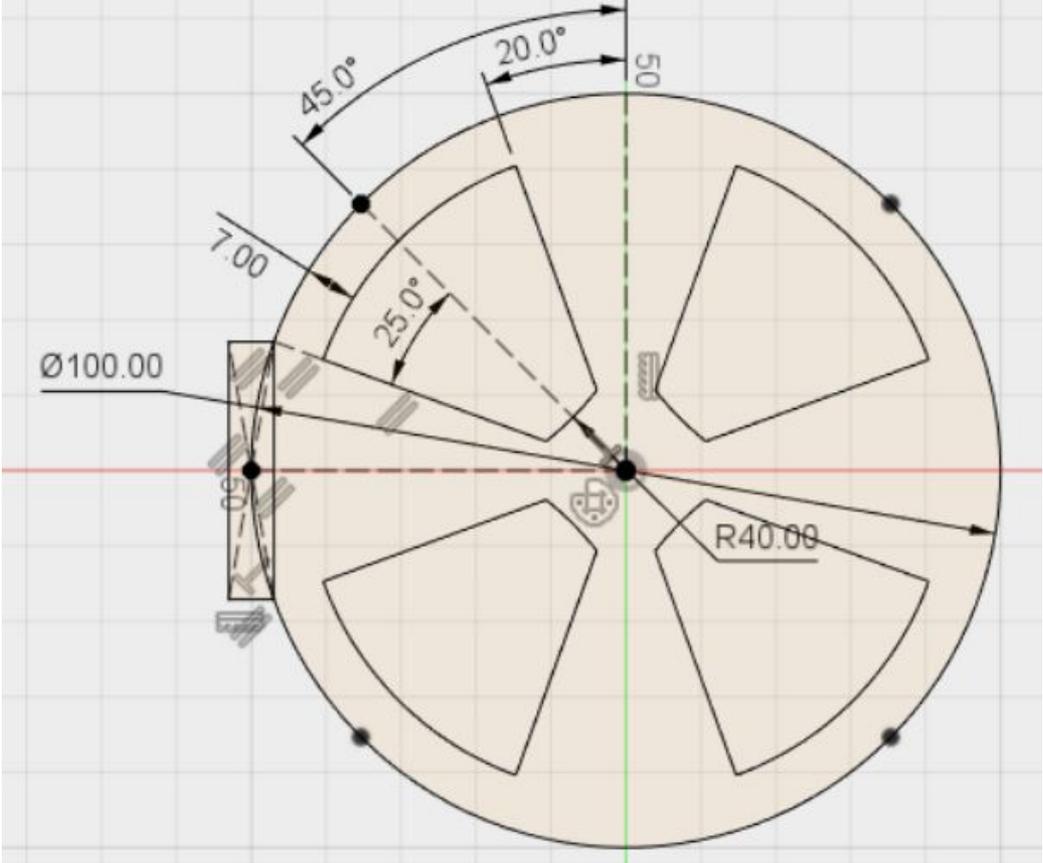
Angle 45.0 deg

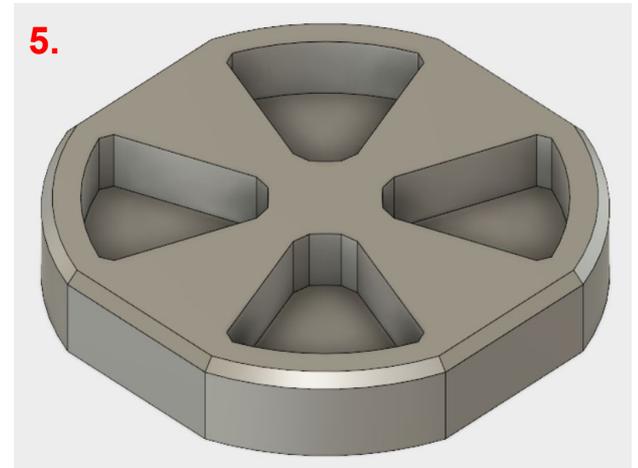
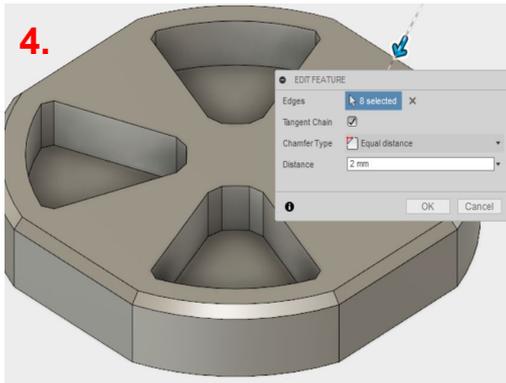
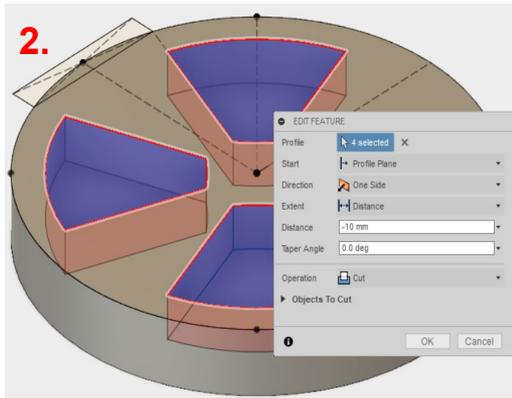
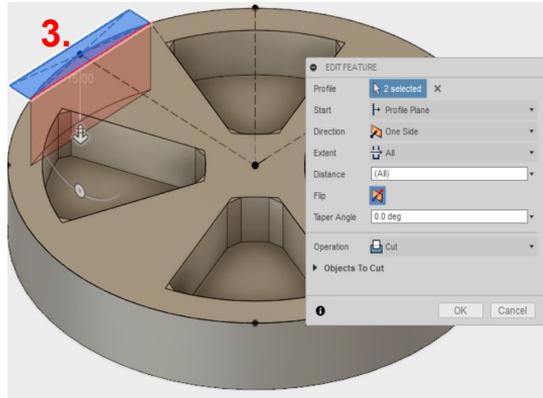
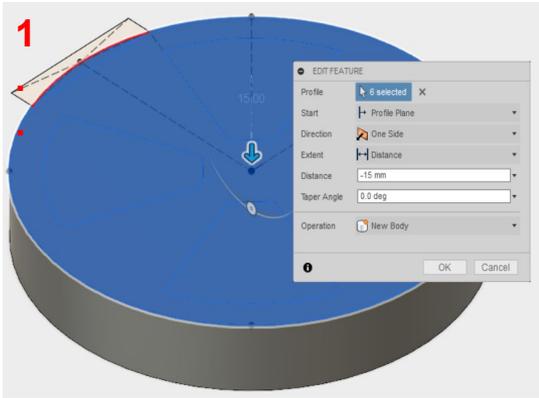
OK Cancel

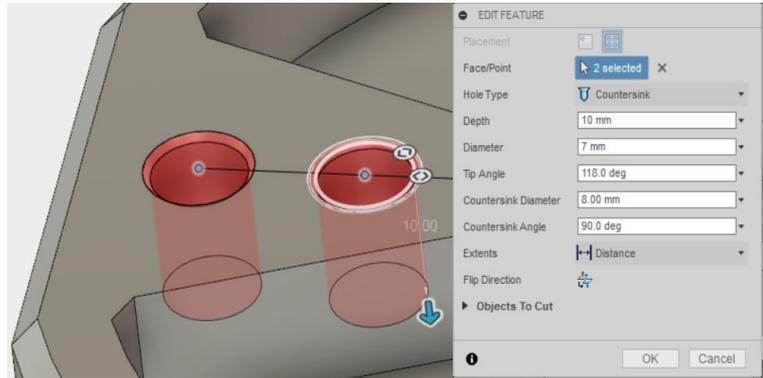
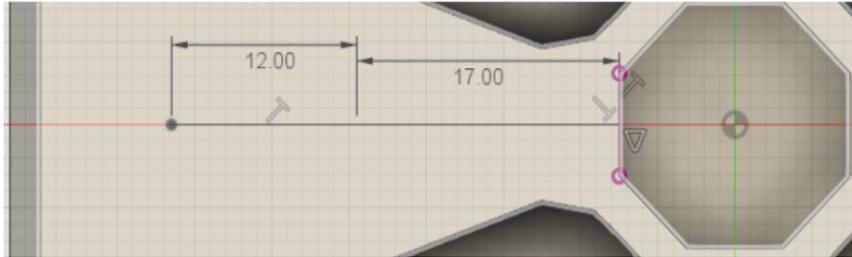
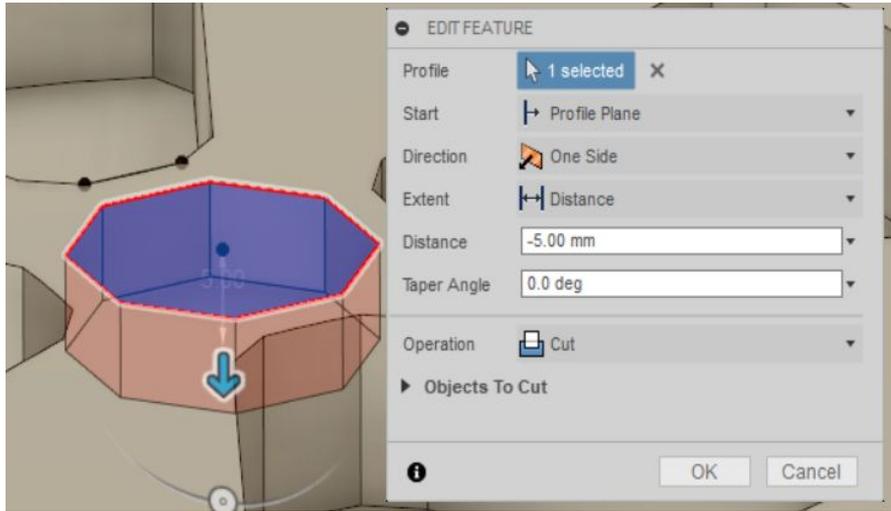


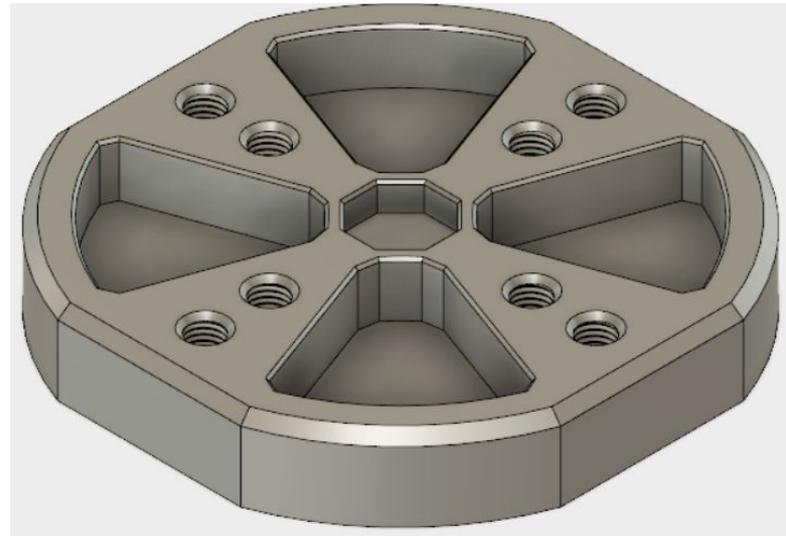
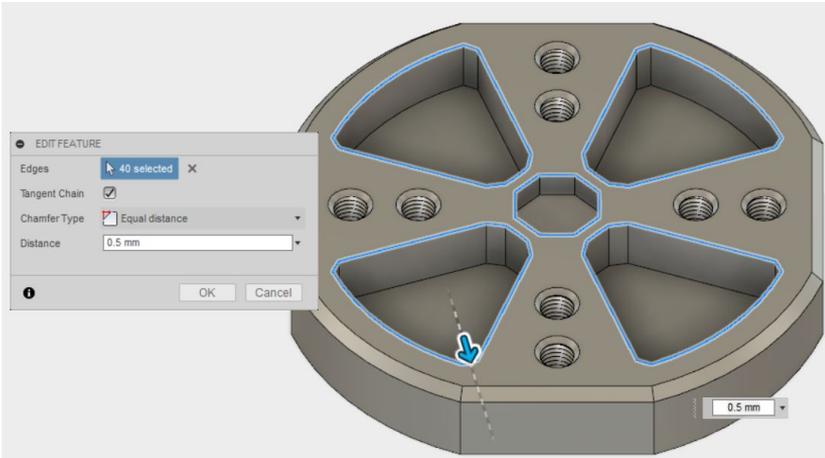
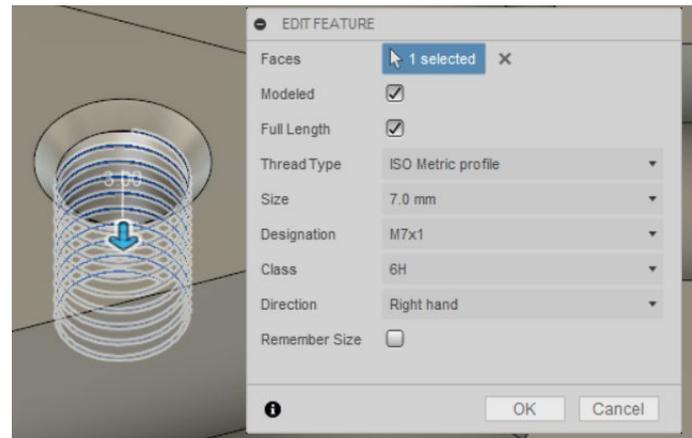
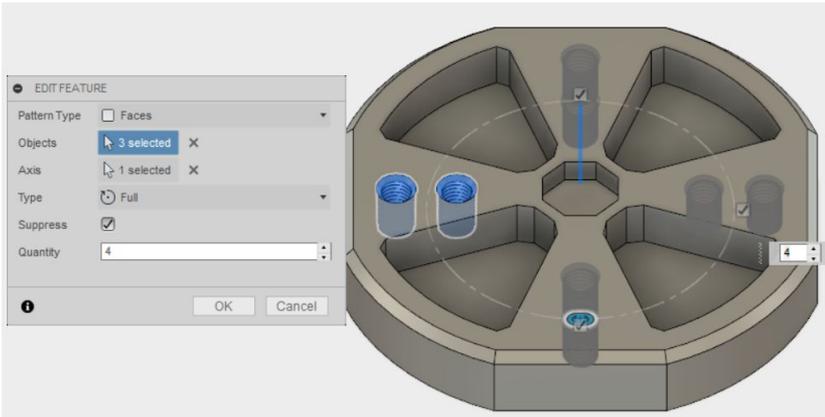


Examples and exercises 2

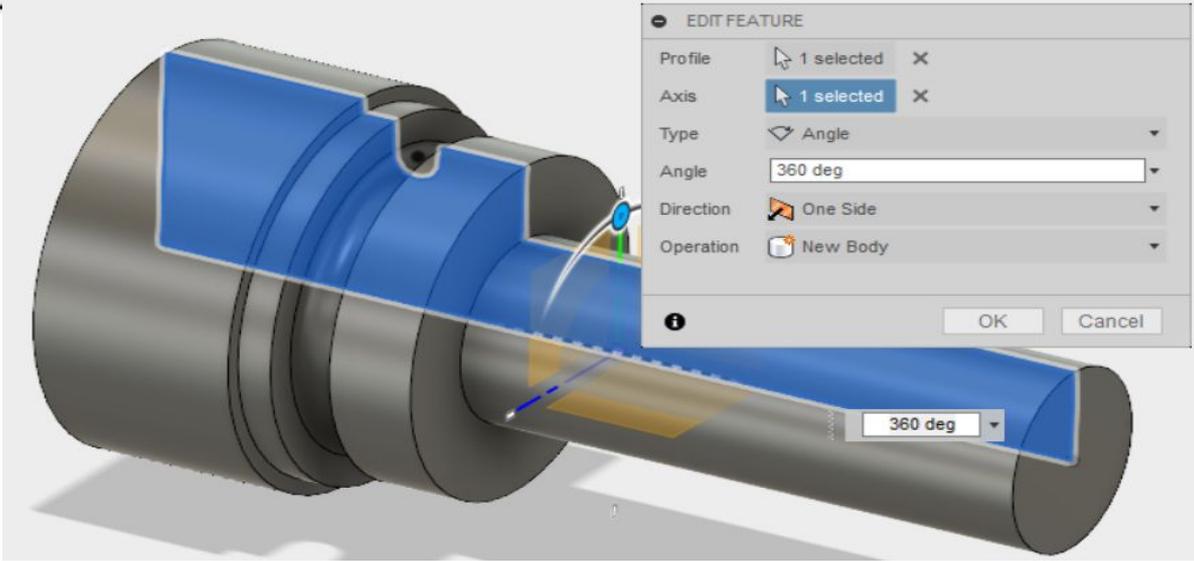
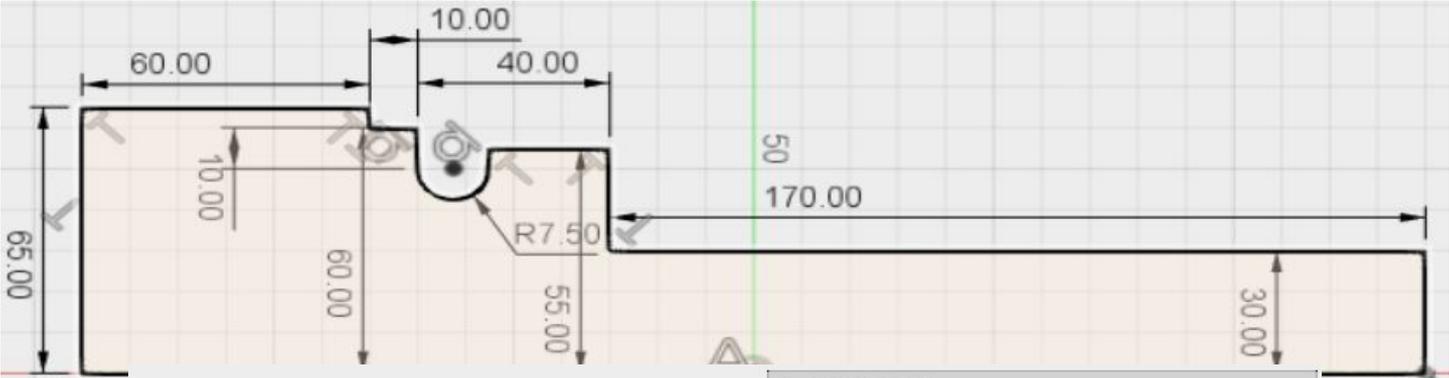


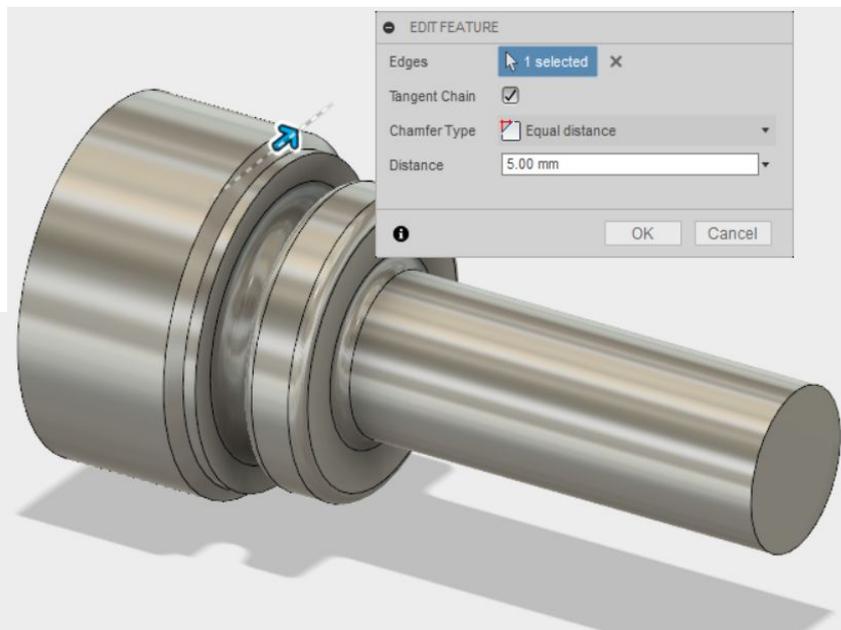
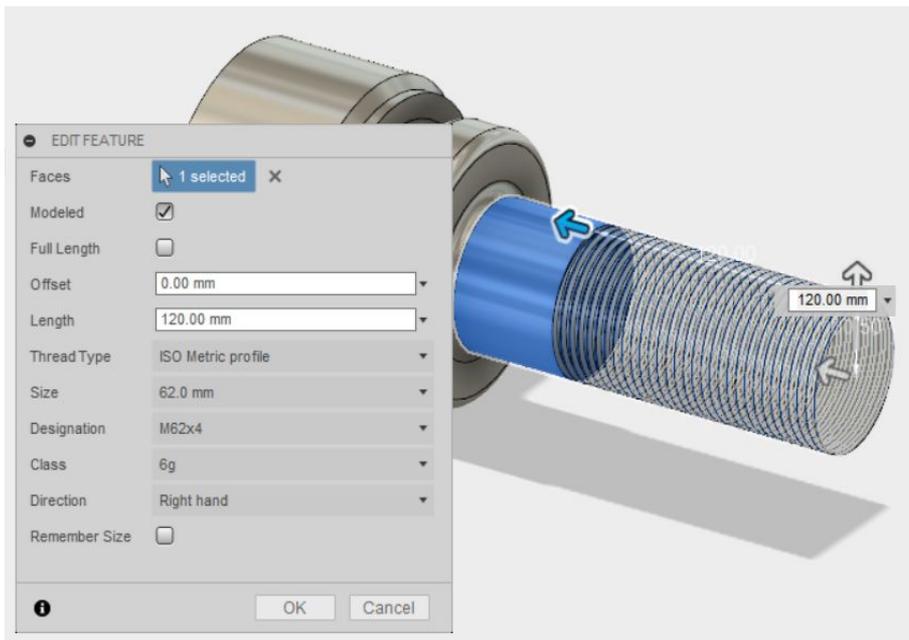






Examples and exercises 3







Juraj Dobrila
University of Pula



Co-funded by the
Erasmus+ Programme
of the European Union

3D and VR for VET



InSciEd



Industrijsko-obrtnička
škola Pula

USN Universitetet
i Sørøst-Norge



3D and Virtual Reality Technologies for VET
Project number: 2019-1-HR01-KA202-06100