Teaching materials

Deliverable 2. Teaching materials for students

MISCE project

Mechatronics for Improving and Standardizing Competences in Engineering

Competence: CAD software

Workgroup: RzuT UNICA, UCLM, UNICAS

To be able to analyse and optimise a mechanical de

This document is the Teaching materials for students of the technical competence 'CAD software'. Its briefly contains the experimental platform analysed in MISCE project, to be designed and standardised for improving the acquisition level of this competence on engineering degrees.

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Visit https://misceproject.eu/ for more information.

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1 Exercise Instructions for Students

Exercise: CAD software

Objective: To acquire skills related to the working in CAD software, modelling and modification of solid, making technical drawings.

Required Tools:

- Computer with Widows operating system
- Installed one of the following programs: SolidWorks, Inventor, Catia, Siemens NX or other

Task: Make the model according to the instructions below. The model presented is an example. The shape, dimensions, and order of the features made may vary.

The module consists of two exercises:

1. **Exercise 1: Solid Modeling in CAD- Software comparison**– focused on developing basic skills in 3D modeling and technical drawing.
2. **Exercise 2: Designing a Simplified Mechatronic Device Using Parametric Modeling** – aimed at applying parametric design techniques to create a flexible mechatronic model.

Objectives:

1. **Exercise 1: Solid Modeling in CAD - Software comparison**– focused on developing basic skills in 3D modeling and technical drawing.

The aim of this exercise is to develop students' skills in using CAD software for creating three-dimensional models. During the exercise, students will design a simple solid model by applying fundamental CAD tools such as sketching, extruding, revolving, and adding features like fillets and chamfers. Additionally, they will learn how to define constraints and dimensions in sketches and prepare a technical drawing of the modeled part. This task will allow students to gain practical experience in solid modeling and technical documentation.

2. **Exercise 2: Designing a Simplified Mechatronic Device Using Parametric Modeling** – aimed at applying parametric design techniques to create a flexible mechatronic model.

The purpose of this exercise is to design a simplified mechatronic device model by applying parametric modeling techniques in CAD software. Students will define global variables and equations to control key parameters of the model, such as pressure and flow, and observe how changes in these parameters affect the geometry of the device. The exercise aims to enhance students' understanding of parametric design and the use of CAD tools for creating flexible and adaptable models of mechatronic systems.

1.1 Exercise 1 – Solid Modeling in CAD- Software comaprison

The educational exercise introduces students to CAD environments. Example programs were chosen. The ability to use different programs will make it easier for students to make choices in the future. The selected systems: SolidWorks, Catia, Siemens NX, and Inventor are presented below (Fig. 1). Students can choose any program and should perform similar exercises.

a)

b)

c)

d)

Fig. 1. Selected systems: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

The aim of the exercise is to create a solid similar to the one shown in the Fig. 2.

a) b)

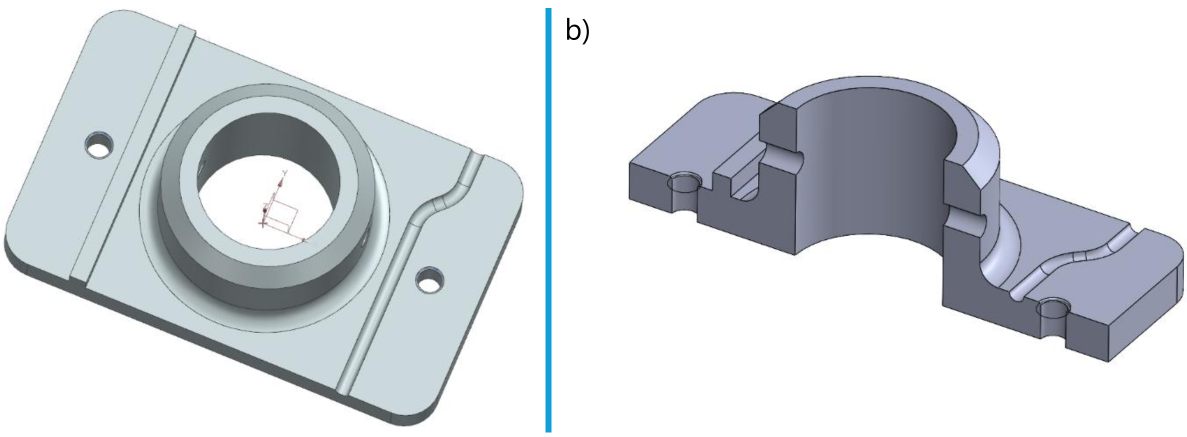


Fig. 2. Example solid: a) model, b) section view of model

1. Select the tool Sketch and the Plane of sketch

Selecting a sketch plane is the first step to creating a solid. In the Cartesian coordinate system, three sketch planes are available (Fig. 3).

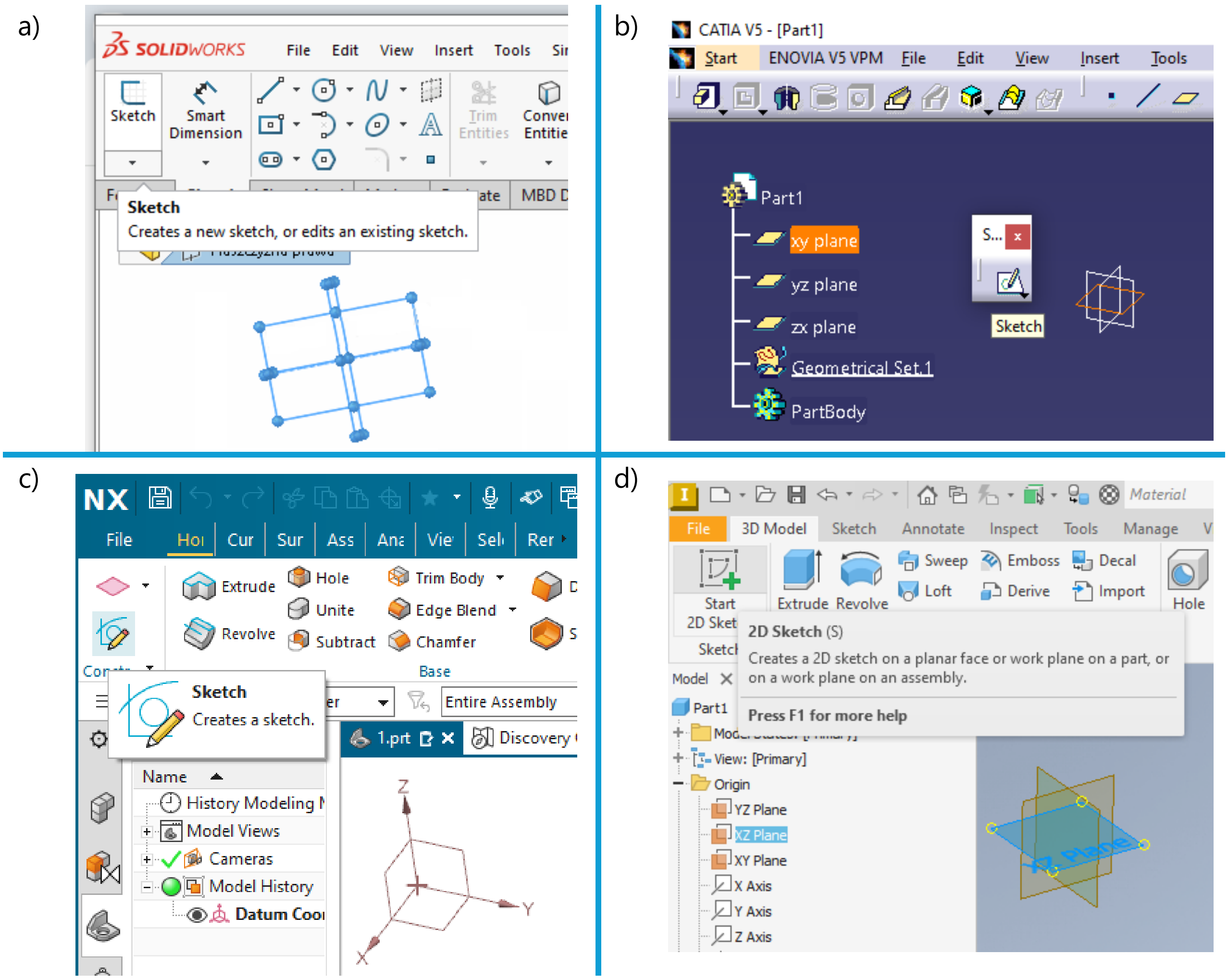


Fig. 3. Sketch in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

2. Create the outline of the solid

An outline of the solid should be made on the selected plane (Fig. 4). This will be the basis for making the solid in three dimensions (3D). It is recommended that simple sketches be drawn and the geometrical features of the solid be given using the modeling function. The basic sketching tools in each program are the following: profile, rectangle, circle, line, arc, helical, spline, point, fillet, chamfer, mirror, pattern, offset, trim, extend, corner, split, text, project geometry, and the following constraints: dimension, coincident, collinear, concentric, horizontal, vertical, tangent, parallel, perpendicular, equal, symmetric, midpoint.

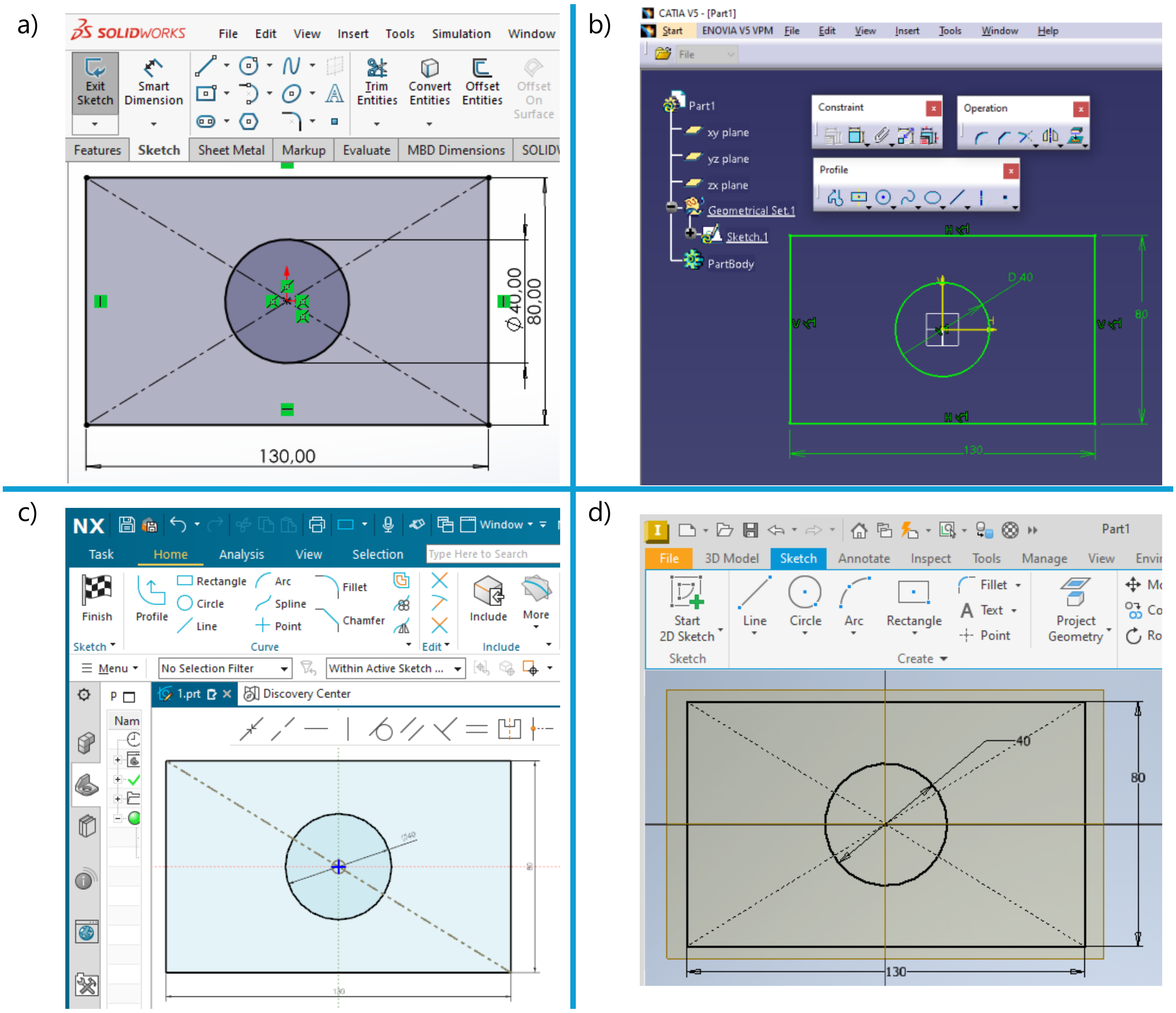


Fig. 4. Outline of the solid in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

3. Create a feature or body by adding depth to the profile.

The first step after drawing the solid outline is to create a three-dimensional solid. One of the functions that can be used for this is “Extrude” (or "Pad" in Catia). The extrude creates a feature or body by extruding a sketch or selected sketch contours in one or two directions, adding depth to a profile (Fig. 5). Closed profiles create solids or surfaces, while open profiles create surfaces. The extruded features are being built for parts. Extrusion adds or removes material from parts or removes material from assemblies. Extrusion can create new solid bodies in a multibody part file. You specify where the extrude starts and define its direction and extent, depth, taper angle, and termination method for the extrusion.

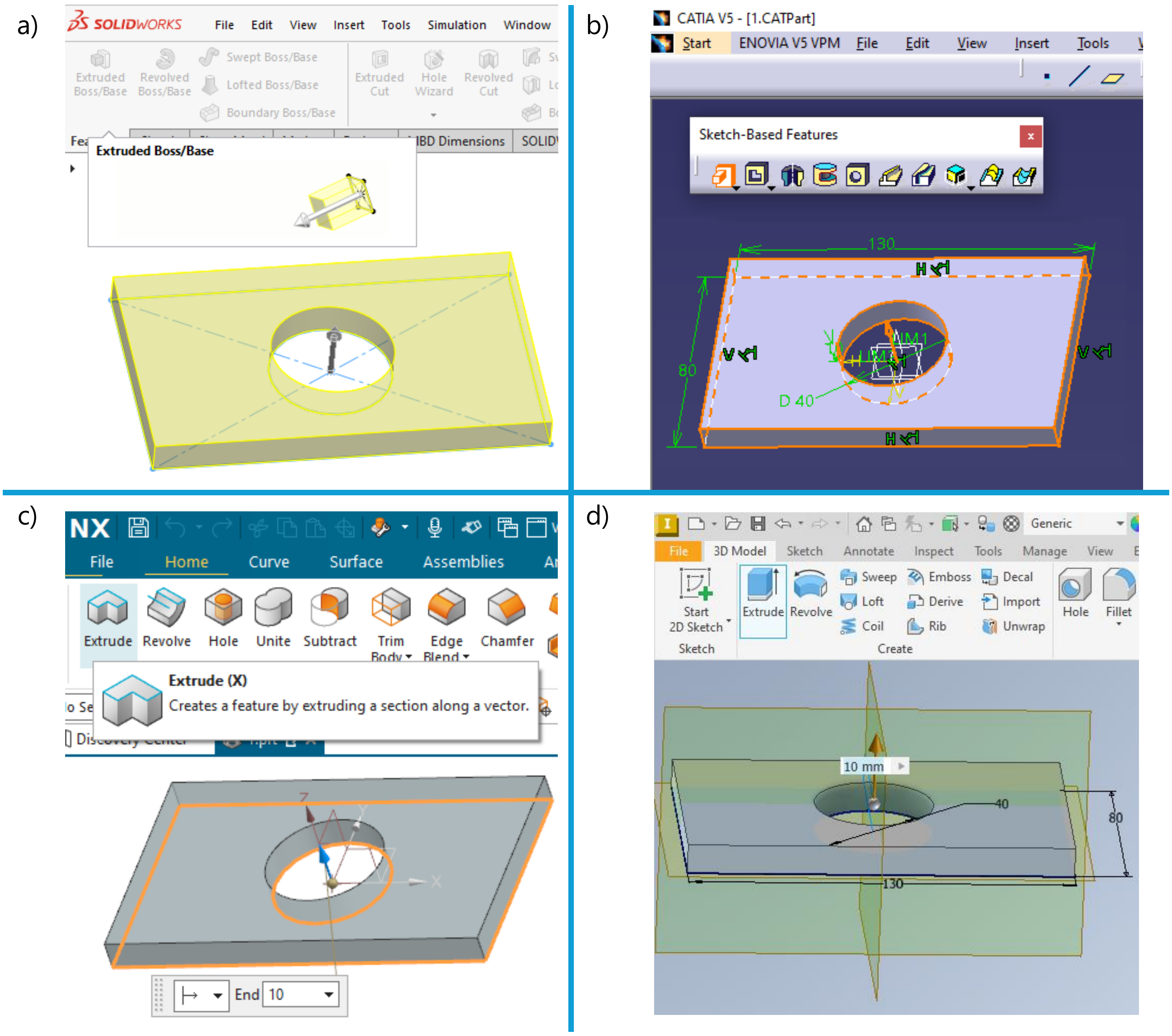


Fig. 5. Extrude of the part in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

4. Create a reference geometry - the work axis.

The work axis (Fig. 6) creates a construction line that is parametrically attached to other objects. Create a reference axis when the current geometry is insufficient to create and position more features. For example, use a work axis to mark symmetry and centerlines or help define a revolve axis location.

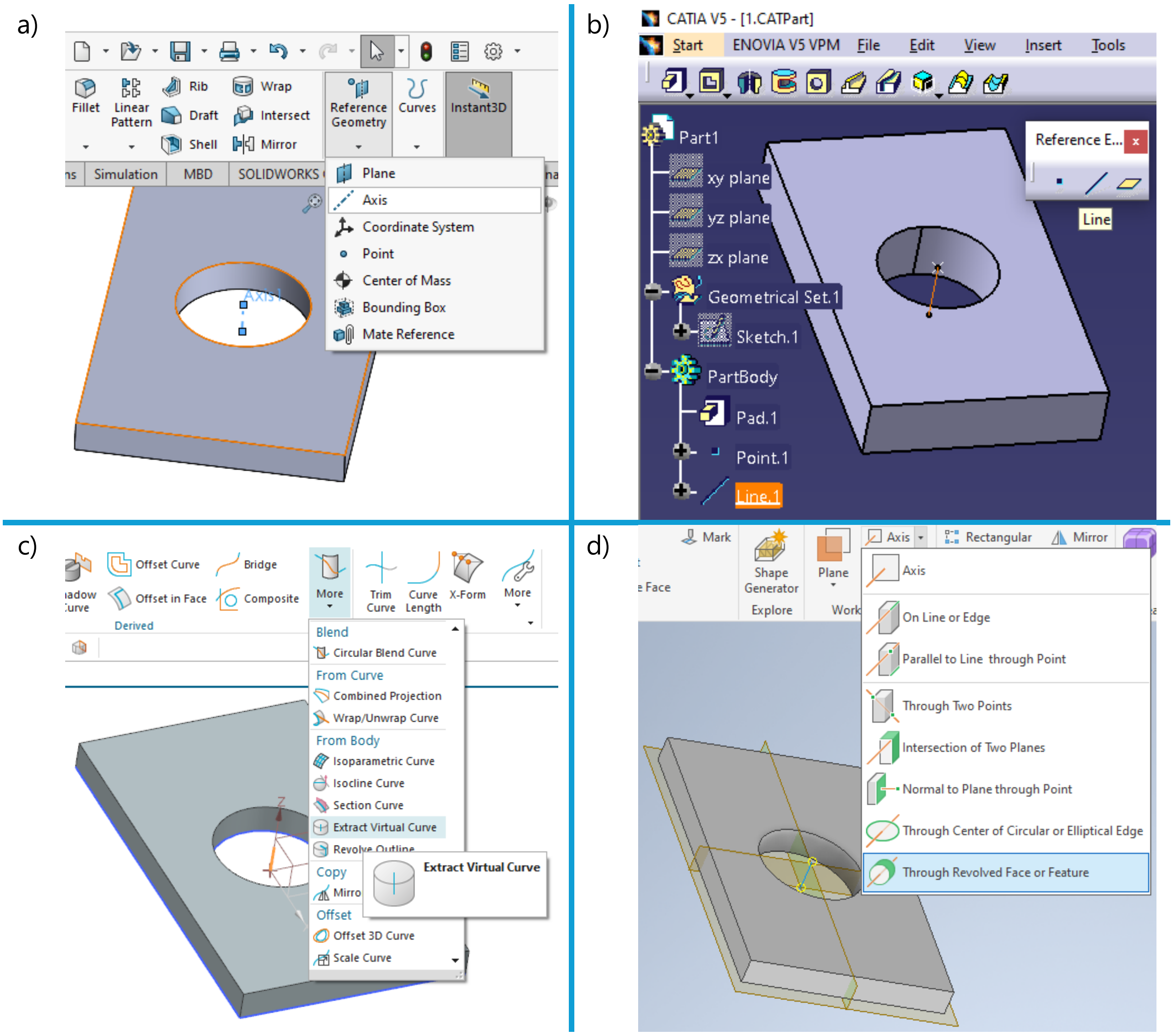
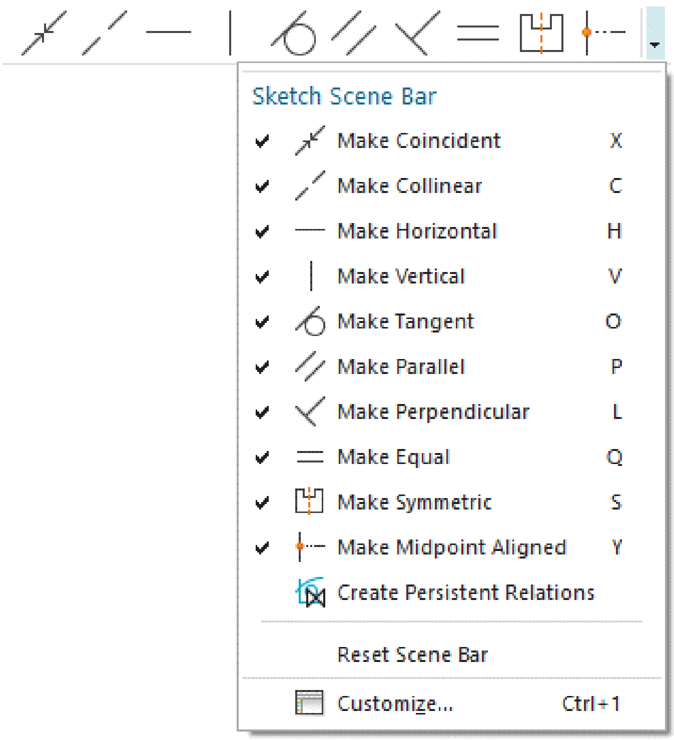


Fig. 6. Create the reference axis in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

5. Create relations of constraints and dimensions in the new sketch, regarding the existing body.

In each sketch, relationships can be inserted between the drawn shapes, e.g. between lines, arcs, points, etc. The most important ones are shown in Figure 6 a). In addition to these, conversion to reference and hide are also used. In sketches, dimensions can be parameterised. Their value can be the result of some function. They can also depend on each other, as shown in Figure 6 b). Dimension p4 is 1/3 of the value of dimension p5.

a)



b)

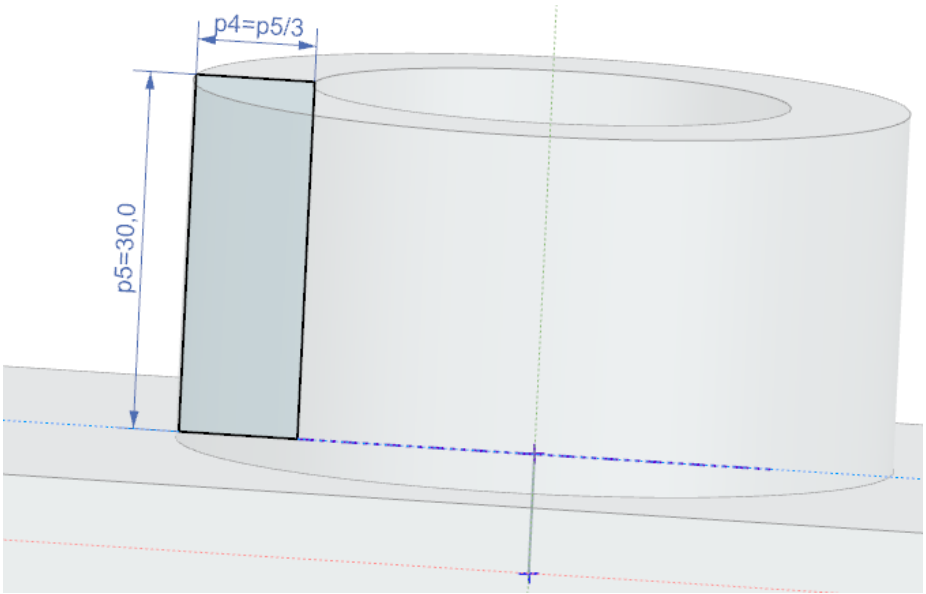


Fig. 7. Relationships in the sketch: a) constraints, b) parameterization of dimensions

6. Create a feature or body by revolving one or more sketched profiles about an axis.

Revolves a new sketch or selected sketch contours around an axis to create a solid feature (Fig. 8). You can revolve profiles through any angle up to 360 degrees. The axis of revolution can be part of the profile or offset from it. The profile and axis must be coplanar. Use the revolve manager to define the axis, direction, and extent of the revolve.

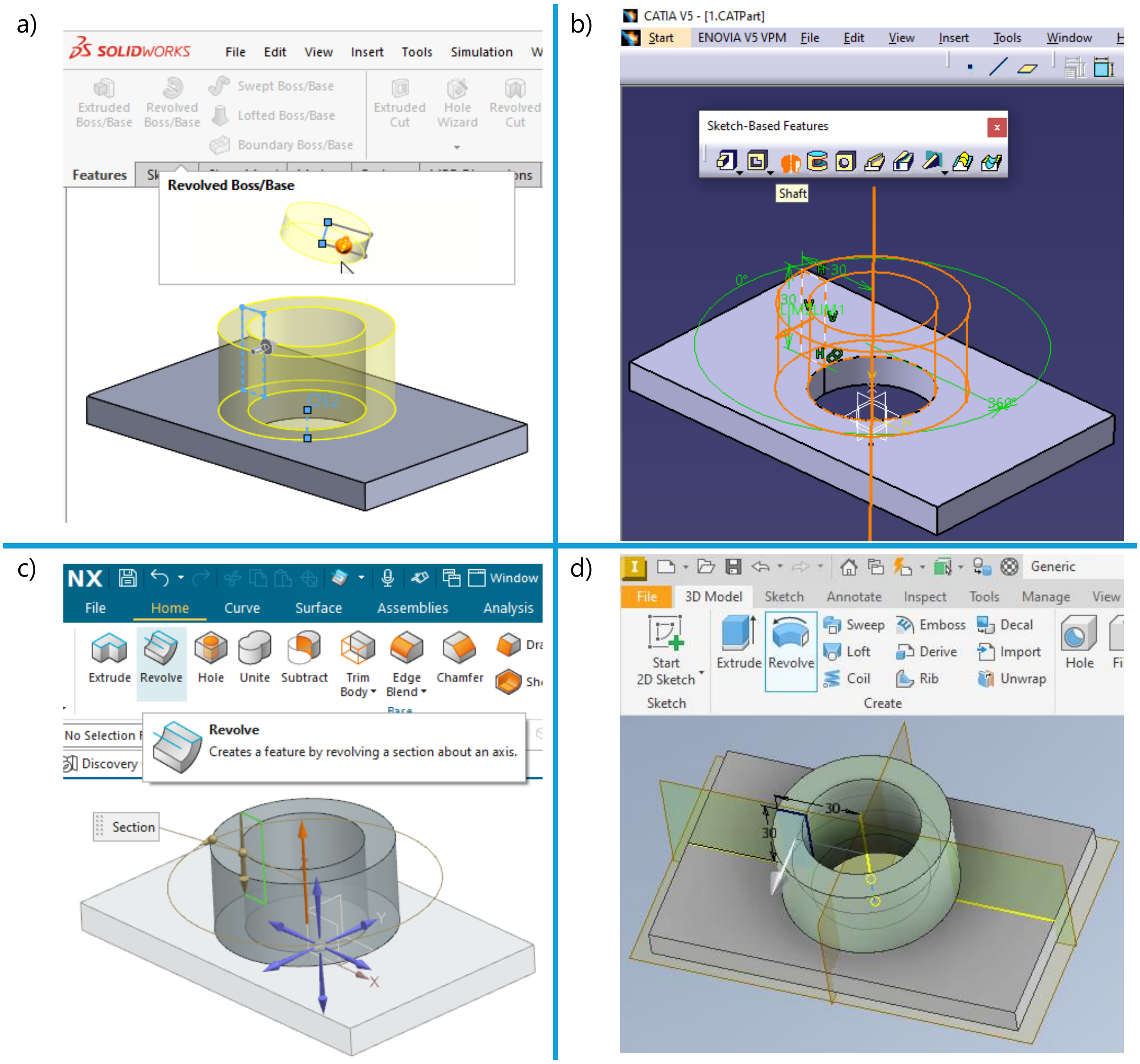


Fig. 8. Revolve feature of the part in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

7. Create a feature at the edge of the body by adding a fillet.

Fillet creates a rounded internal or external face along one or more edges on a solid or surface feature (Fig. 9). Fillets can be of constant or variable radius. You can add fillets as one feature or as separate features. Use the toolbar palette to change between fillet types and selection priority.

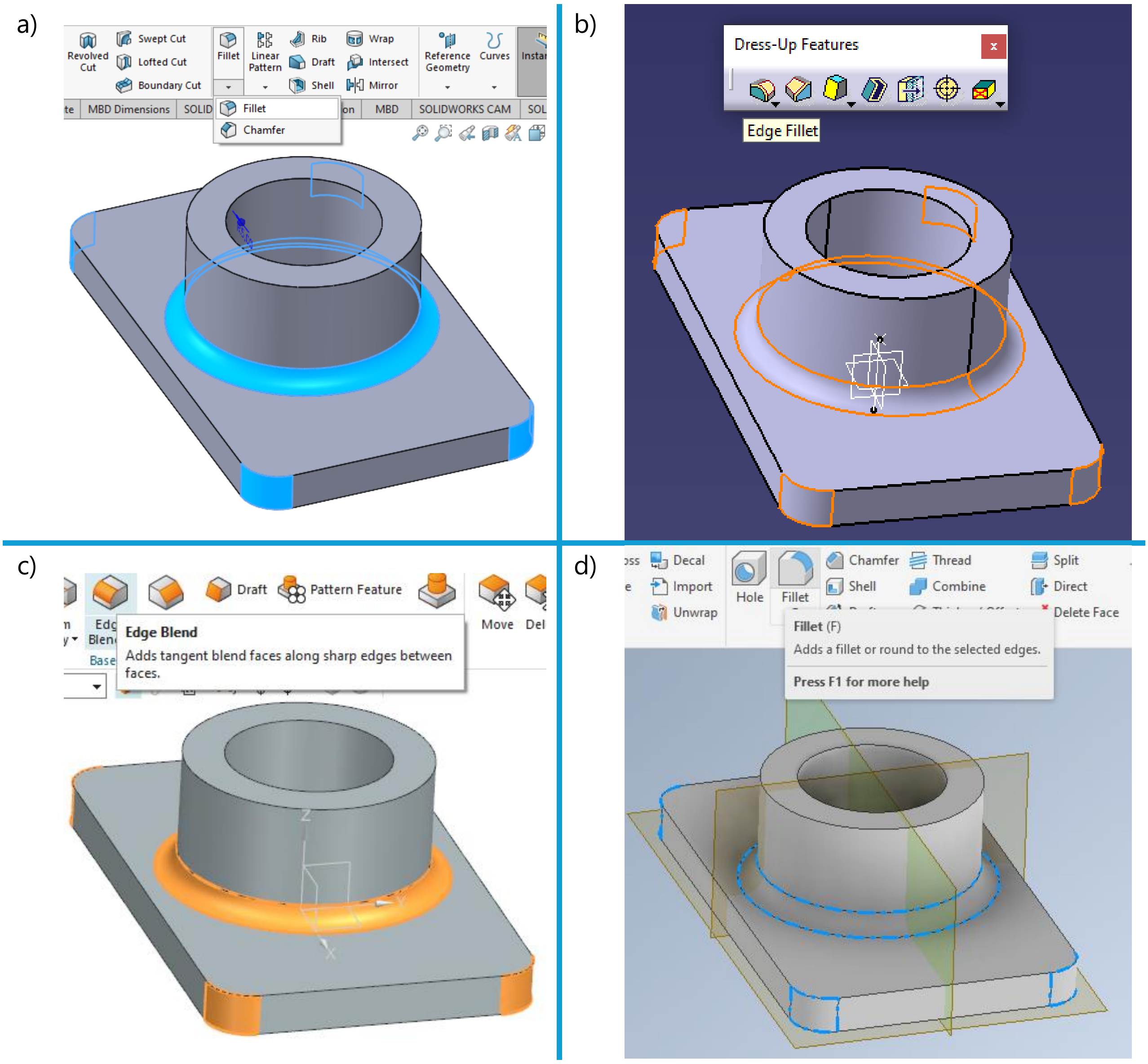


Fig. 9. Fillet feature of the part in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

8. Create a feature at the edge of the body by adding a chamfer.

Chamfers sharp edges between faces (Fig. 10). Applies a bevel to one or more component edges. You can define a chamfer by a single distance, a distance and angle, or two distances.

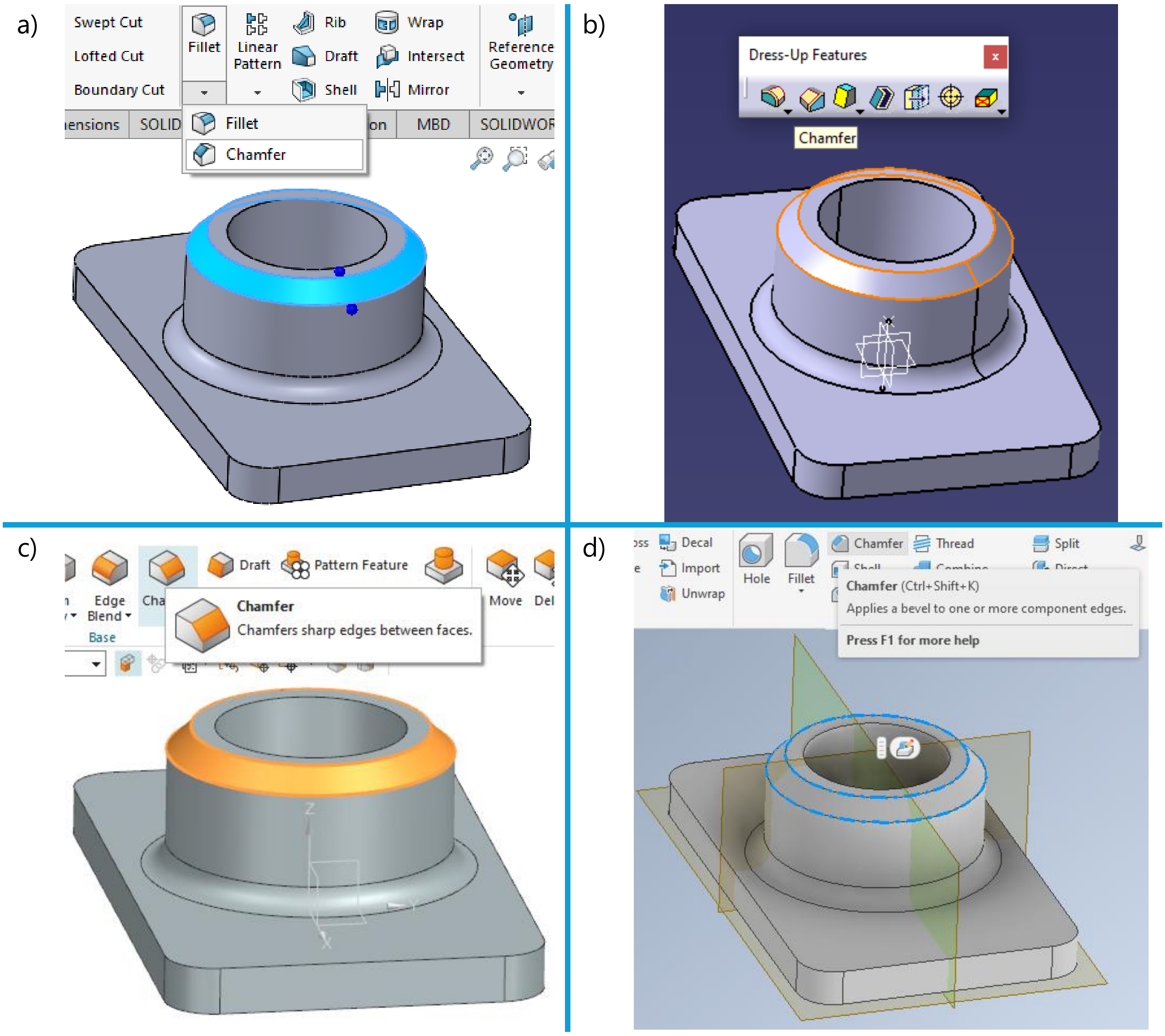


Fig. 10. Chamfer feature of the part in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

9. Create a feature in the body by adding an extruded cut.

Cuts a solid model by extruding a sketched profile in one or two directions. If the cut affects multiple bodies in multibody parts, you can select which bodies to keep. In Catia this function name is called Pocket, but w Siemens NX and Inventor in the Extrude function select the correct option with Boolean operation - subtract or cut (Fig. 11).

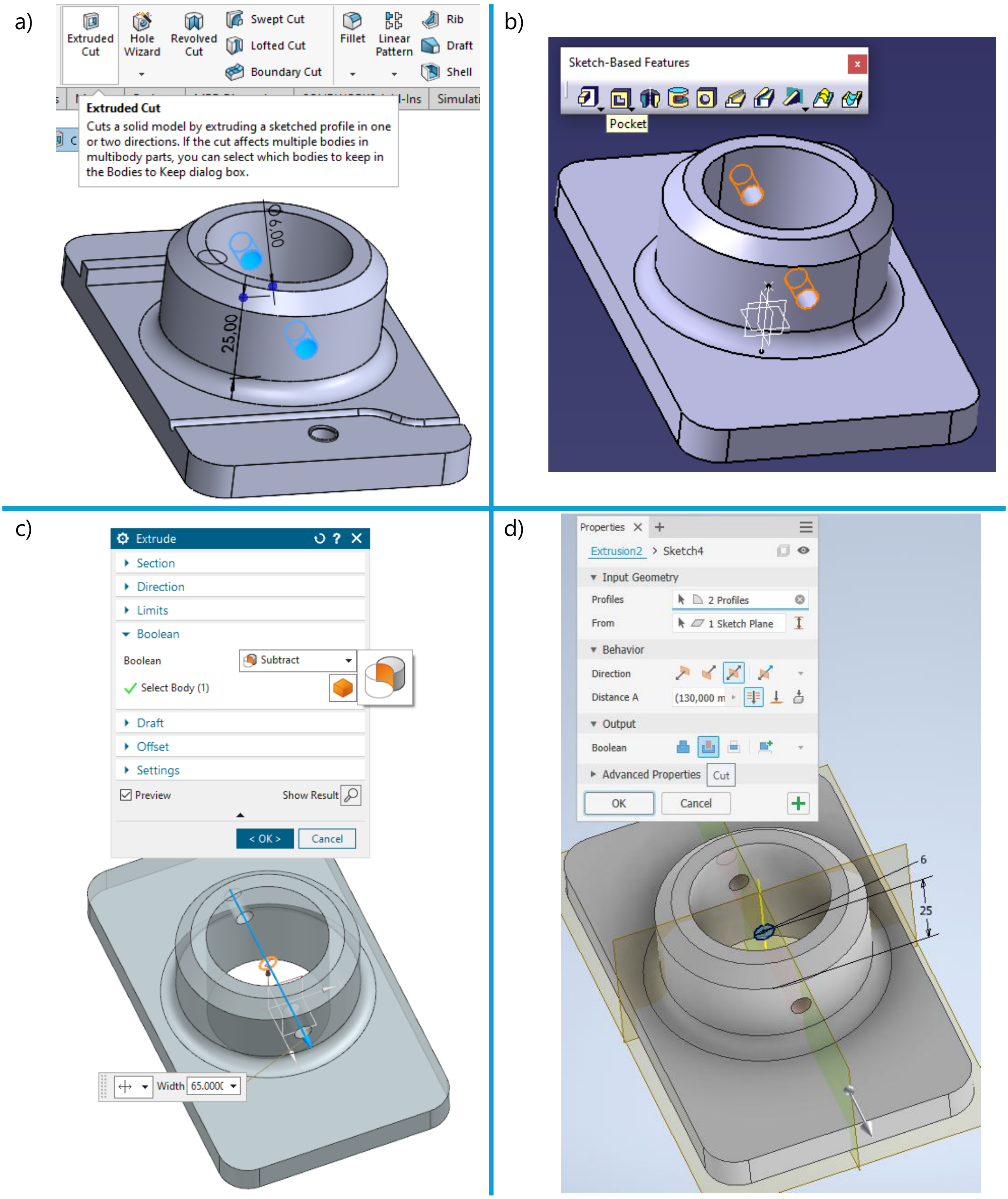


Fig. 11. Extrude cut feature of the part in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

10. Create a feature in the body by adding a hole.

Creates holes based on sketch points or other geometric selections to one or more solid bodies in a part or assembly (Fig. 12). You can create simple, clearance, tapped, or tapered tapped holes, and include thread types from the thread data sheet. You can create counterbore, countersink, thread, or spotface holes. You can specify tolerances for the hole dimensions.

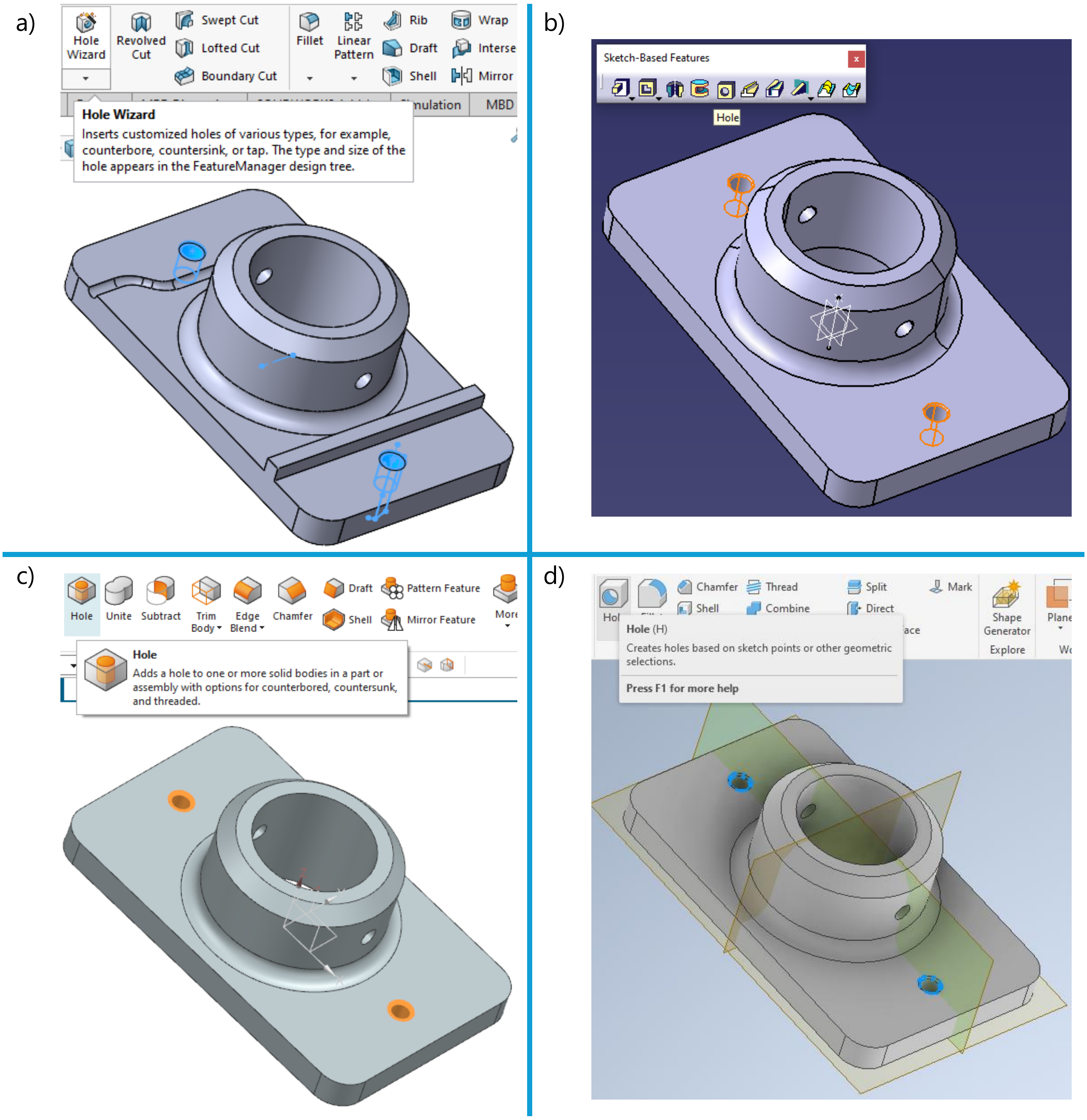


Fig. 12. Hole feature of the part in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

11. Create a reference geometry - the plane.

Creates a construction plane, datum plane, work plane. Different names are used. Creates a datum plane used to construct other features (Fig. 13). This plane is attached parametrically to other objects and is a work plane to sketch on when no planar face exists. You also use datum planes to apply planar assembly constraints on a part where no planar face exists, such as the middle of a part.

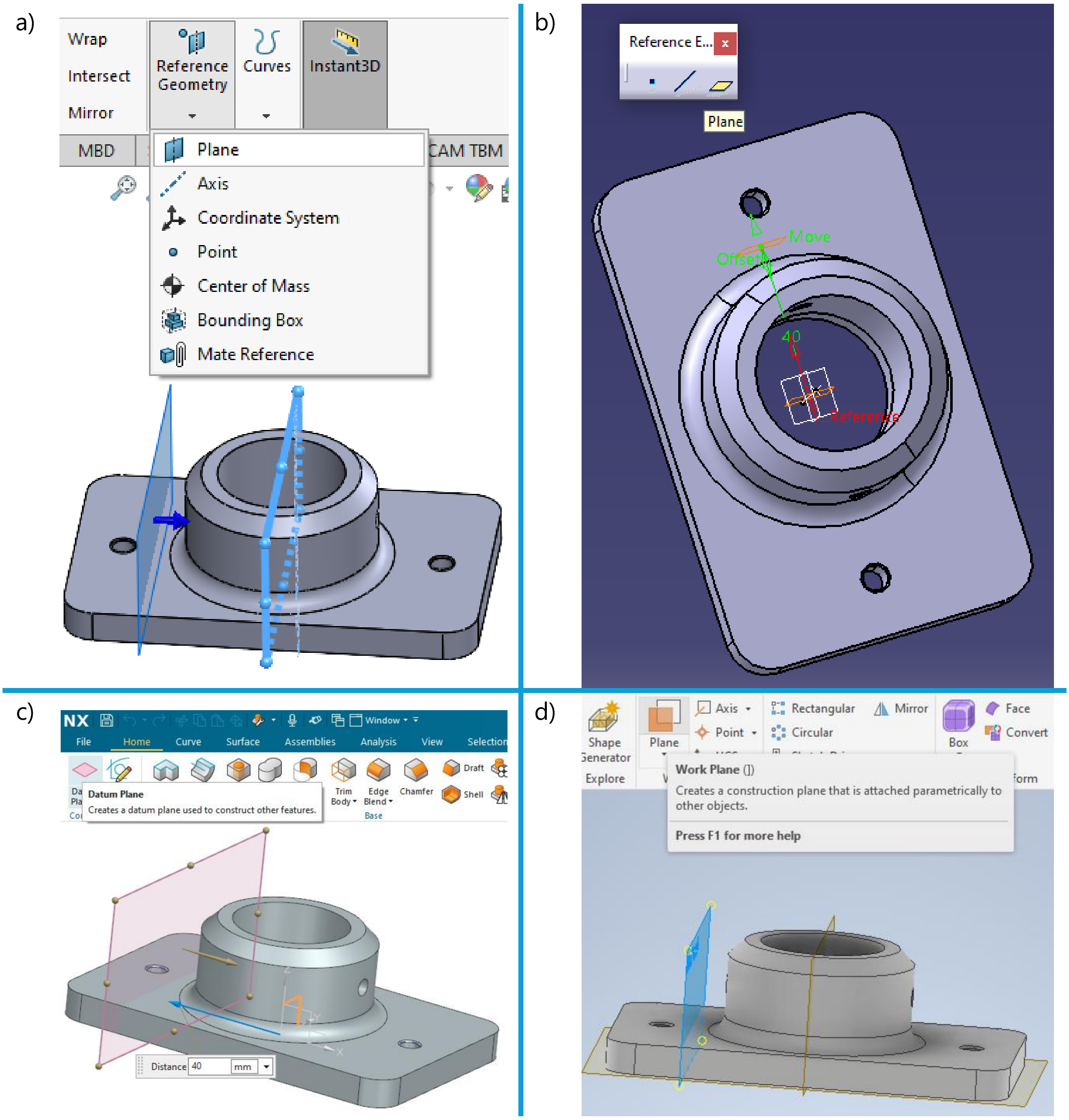
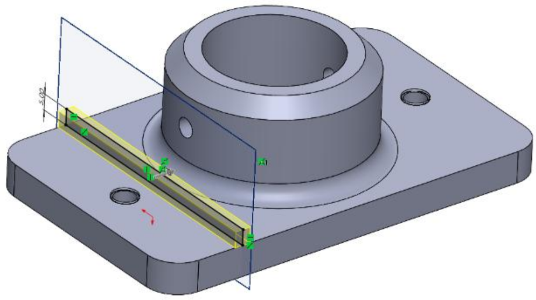


Fig. 13. Create the reference plane in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

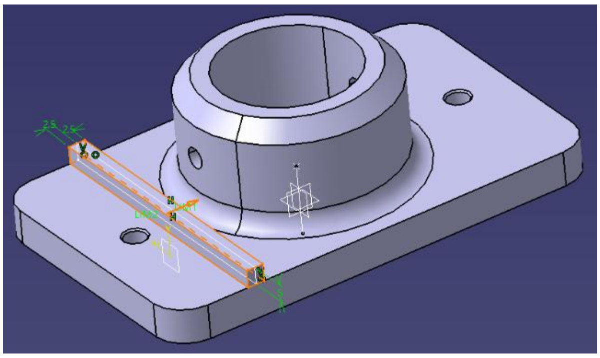
12. Create a feature on the new work plane.

Create any solid feature, e.g. a cuboid on the newly created plane (Fig. 14).

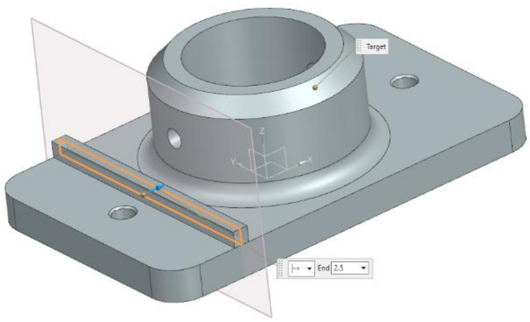
a)



b)



c)



d)

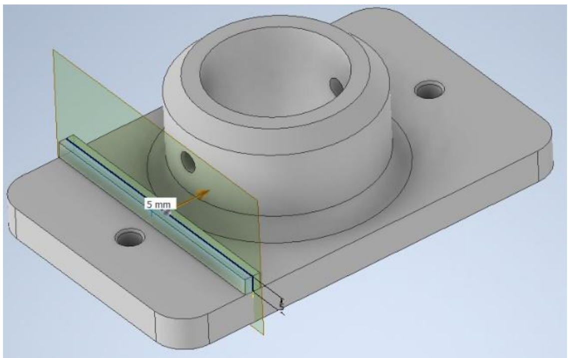


Fig. 14. Create the cuboid on the reference plane in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

13. Create a feature in the body by adding a swept cut.

Cuts a solid model by sweeping a closed profile along an open or closed path (Fig. 15). If you sweep multiple profiles, they must exist in the same sketch. The path can be an open or closed loop, but must pierce the profile plane. If the cut affects multiple bodies in multibody parts, you can select which bodies to keep.

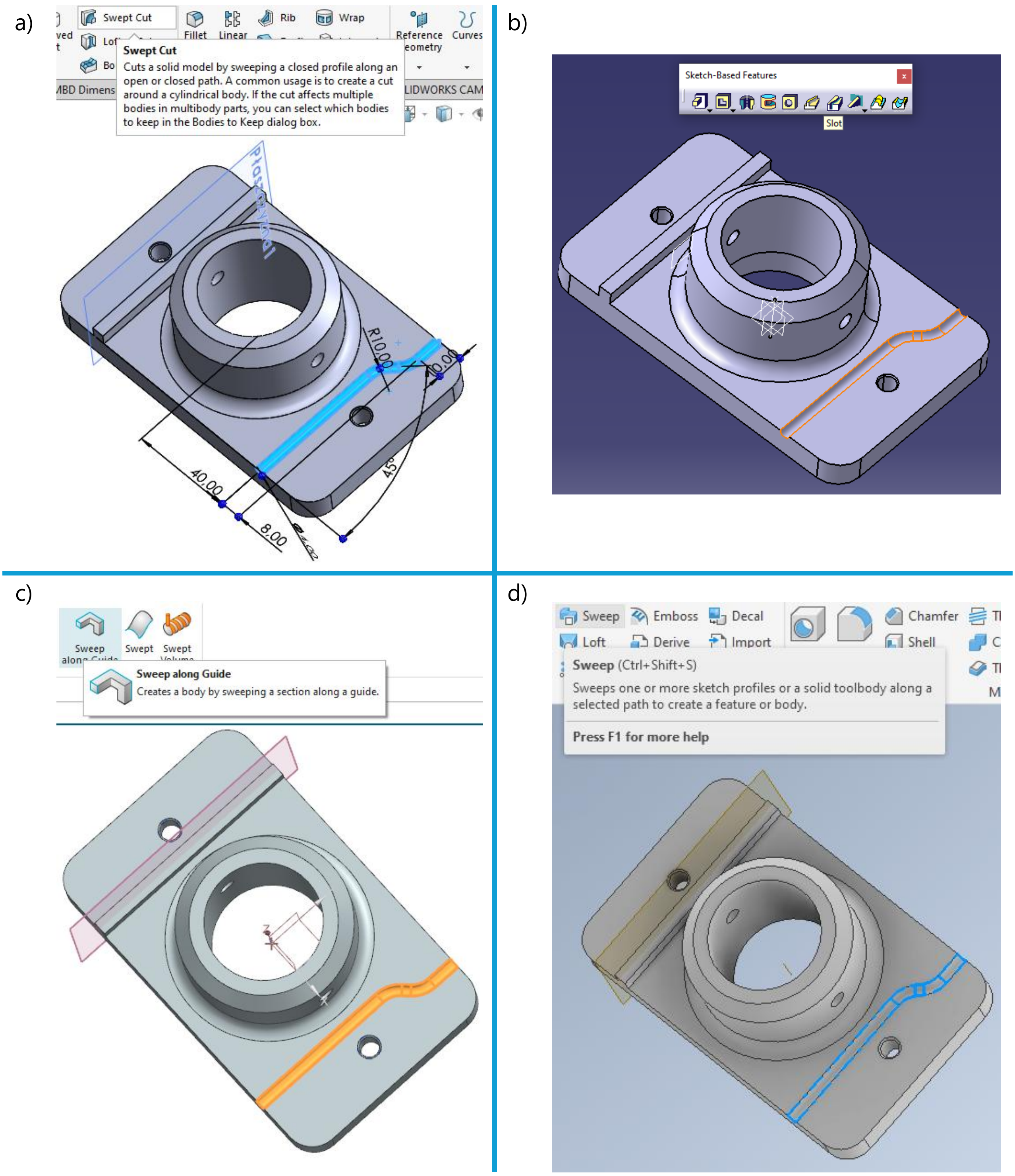


Fig. 15. Sweep feature of the part in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

14. Create a technical drawing.

An example of a technical drawing is shown in Fig. 16.

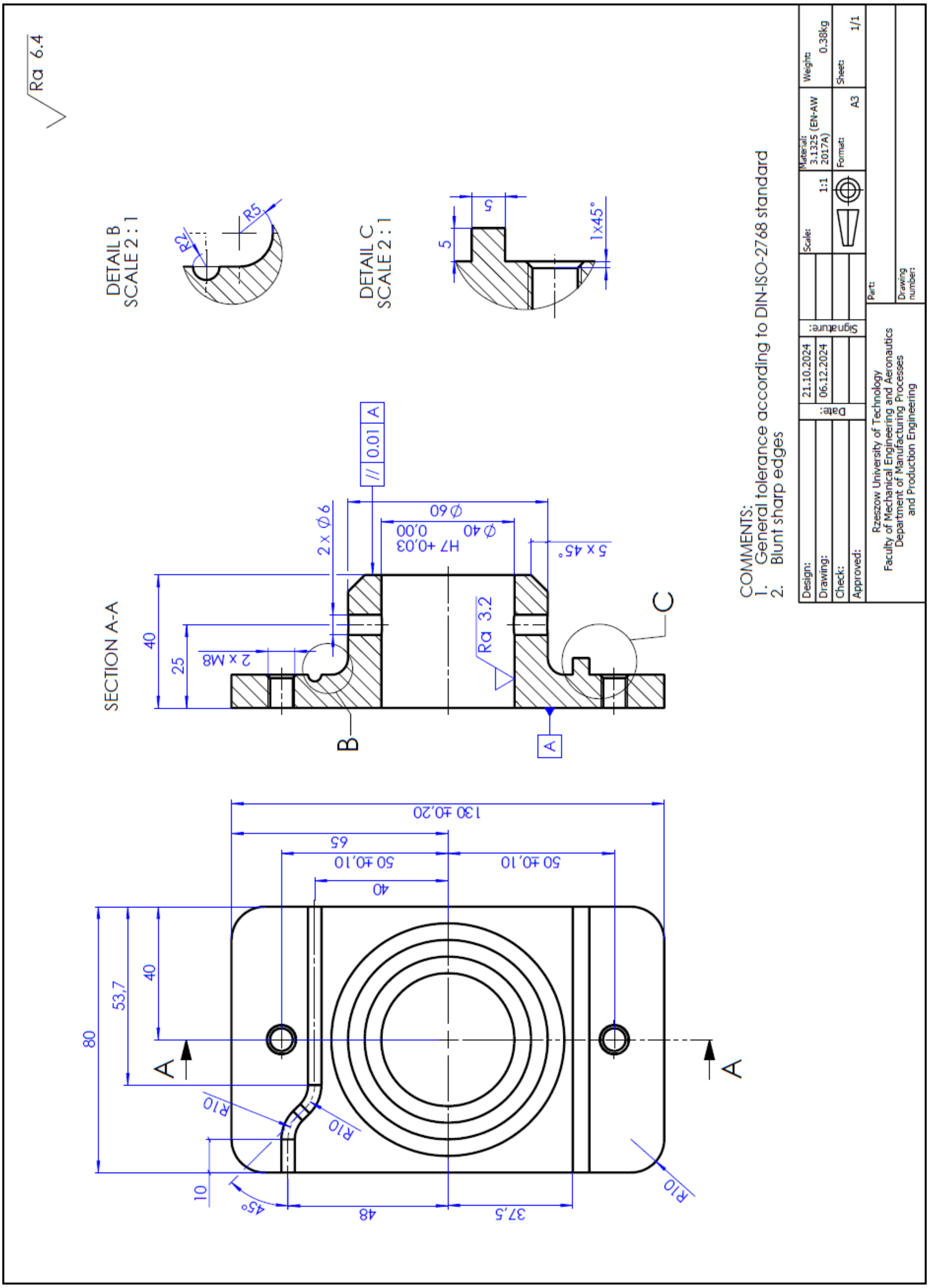


Fig. 16. Example technical drawing

The first step is to move from the modeling module to the drawing module. In each program, this step looks similar, Fig. 17.

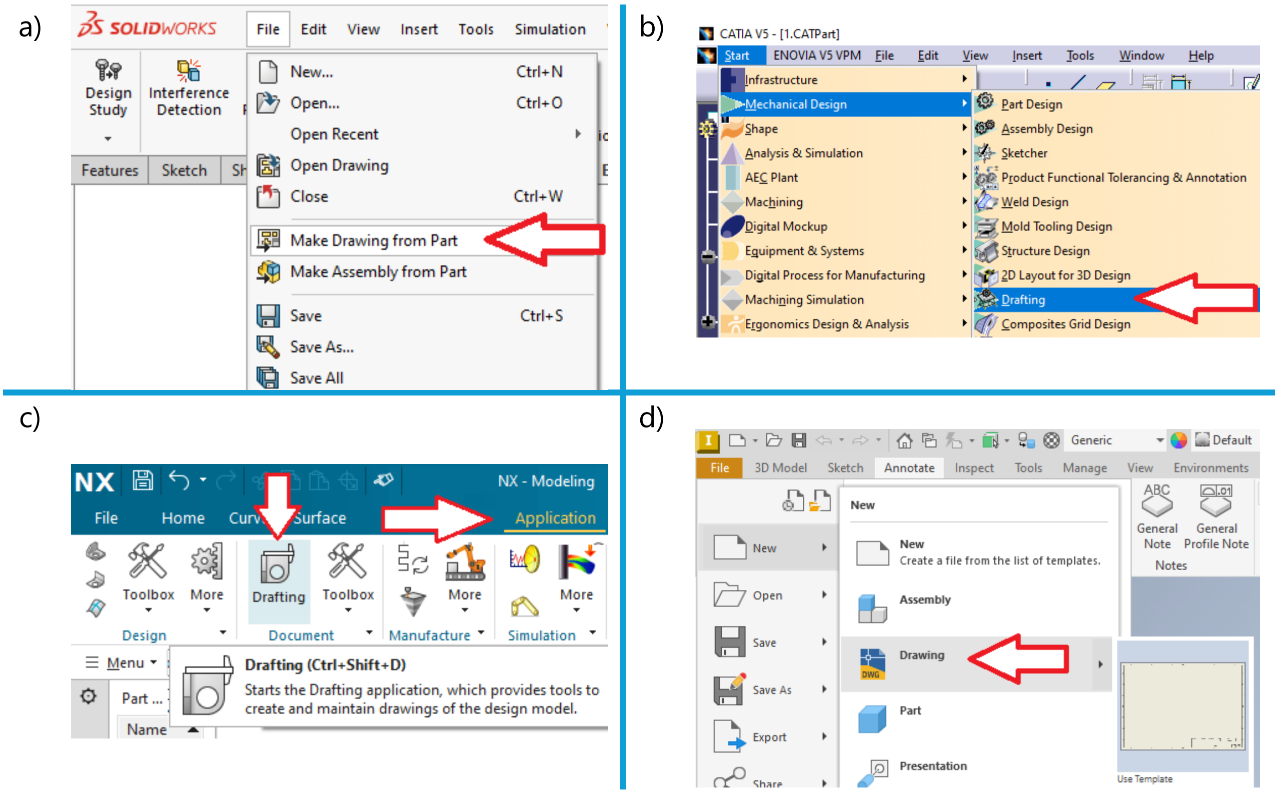
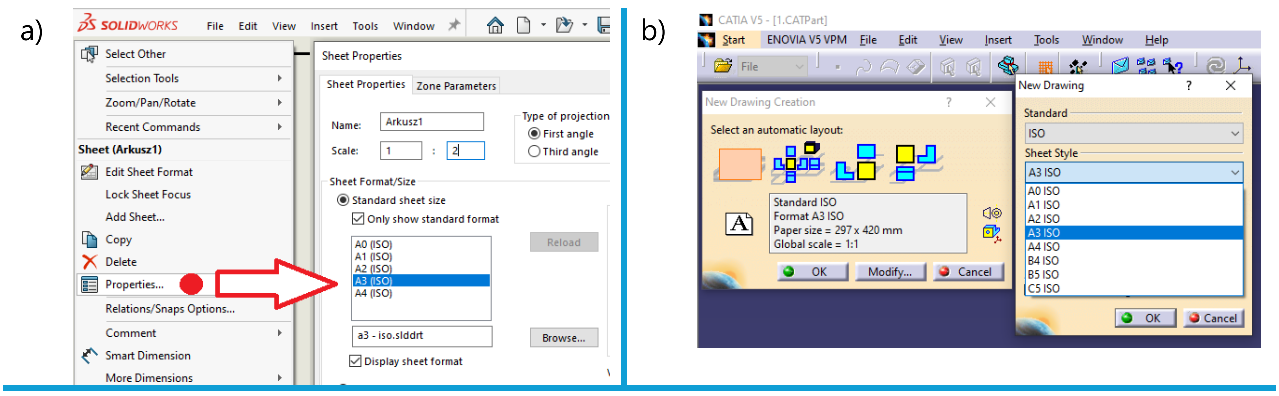
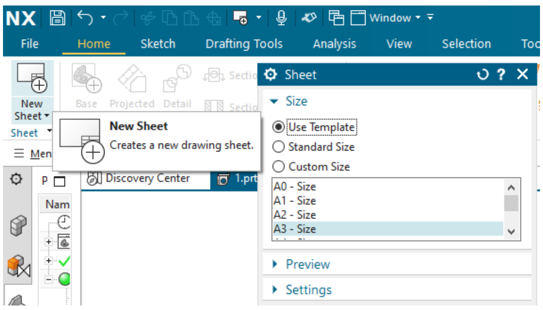


Fig. 17. Move to the drawing module in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

The second step is to choose the format of the sheet, e.g. A4, A3, and scale, e.g. 1:1, 1:2, Fig. 18.



c)



d)

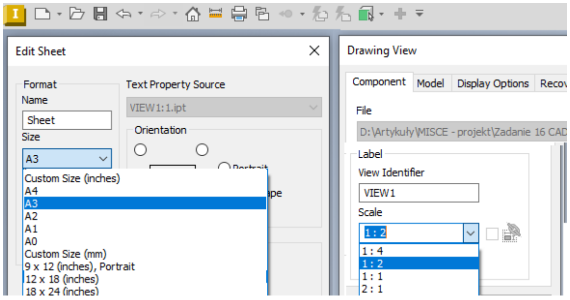


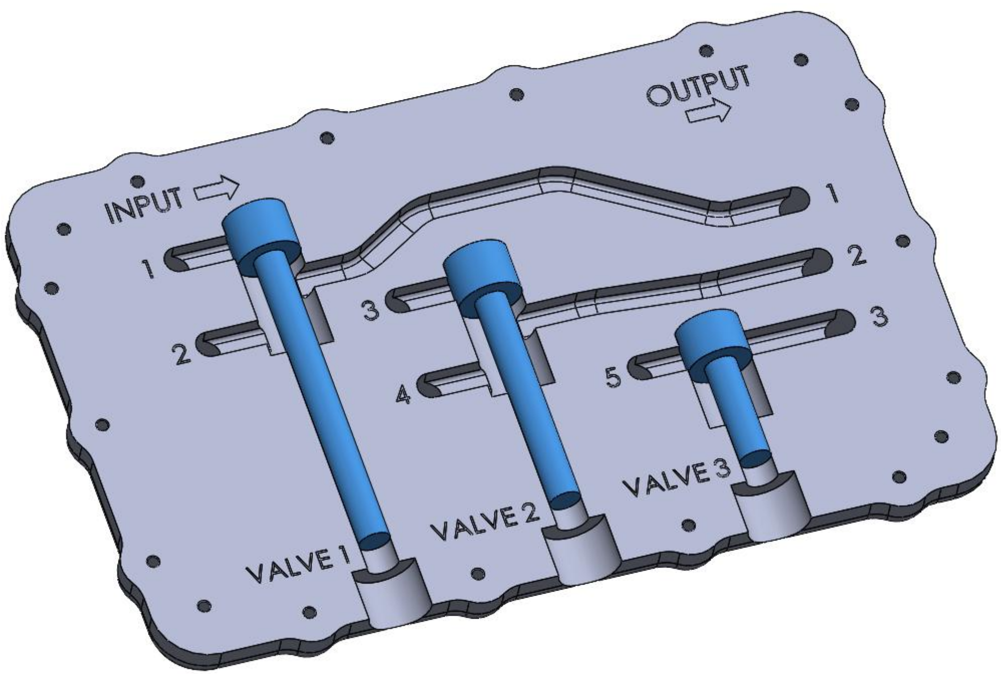
Fig. 18. Choose the format and scale of the sheet in: a) SolidWorks, b) Catia, c) Siemens NX, d) Inventor

Using the available functions, in addition to the main view, you should create section views, projected views, detail views, and add dimensions, tolerances, and other symbols, as shown on example - Fig. 16.

1.2 Exercise 2 – Designing a Simplified Mechatronic Device Using Parametric Modeling

For this competence, the MISCE project proposes the joint use of the prepared model to develop a variant of a mechatronic device using parametric modeling. The model is simplified and simulates the valve system (Fig. 19). For the purposes of the project, the exercise was carried out in SolidWorks, but students can choose any program.

a)



b)

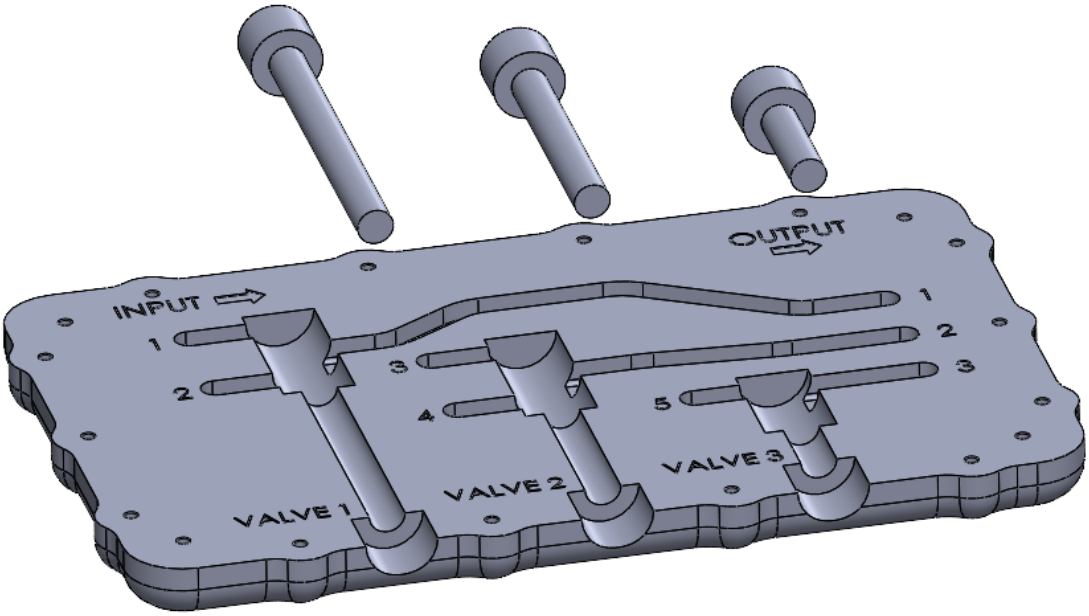


Fig. 19. Valve control system: a) assembly, b) exploded view

Real-life examples of mechatronic valve control systems are shown on the figure Fig. 20.

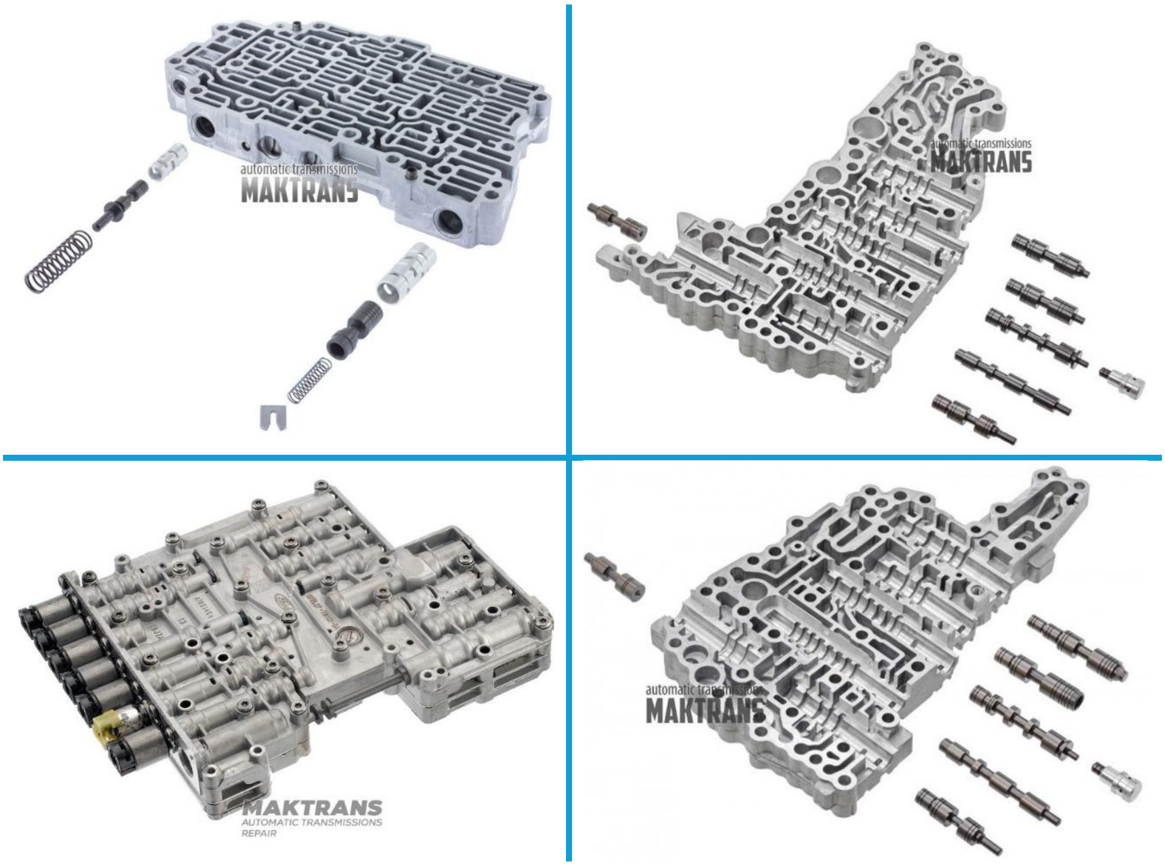


Fig. 20. Real-life examples of mechatronic valve control systems [1, 2, 3, 4]

The ranges of valve movements are derived from the length of the chamber. In the model, this has been limited by relations in the assembly (Fig. 21). There are two types of chambers. The first changes the state of work for one of the two output channels – valves 1 and 2. The second have closed or open status – valve 3.

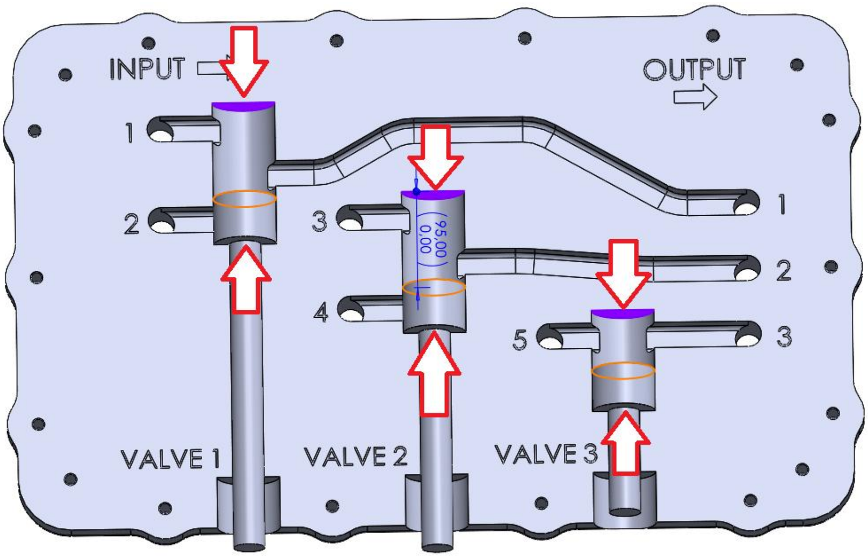


Fig. 21. Ranges of valve movements

Figure Fig. 22 shows the input and output channels. These features are parameterized.

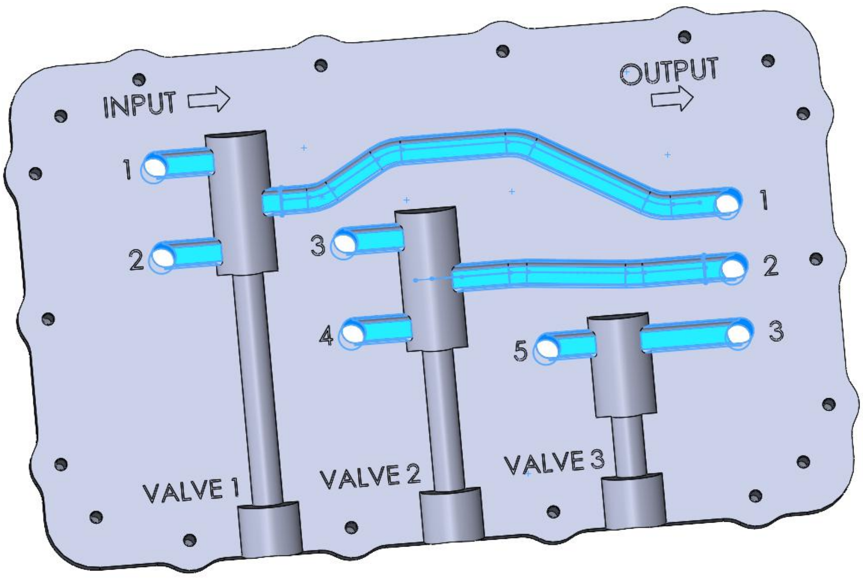


Fig. 22. Parameterised features

Any features of the model were made by using simple tools for modeling, e.g. extrude ("Pad" in Catia), extruded cut ("Pocket" in Catia), fillet, hole, or swept cut (Fig. 23). The tools in the first exercise were presented.

a)

b)

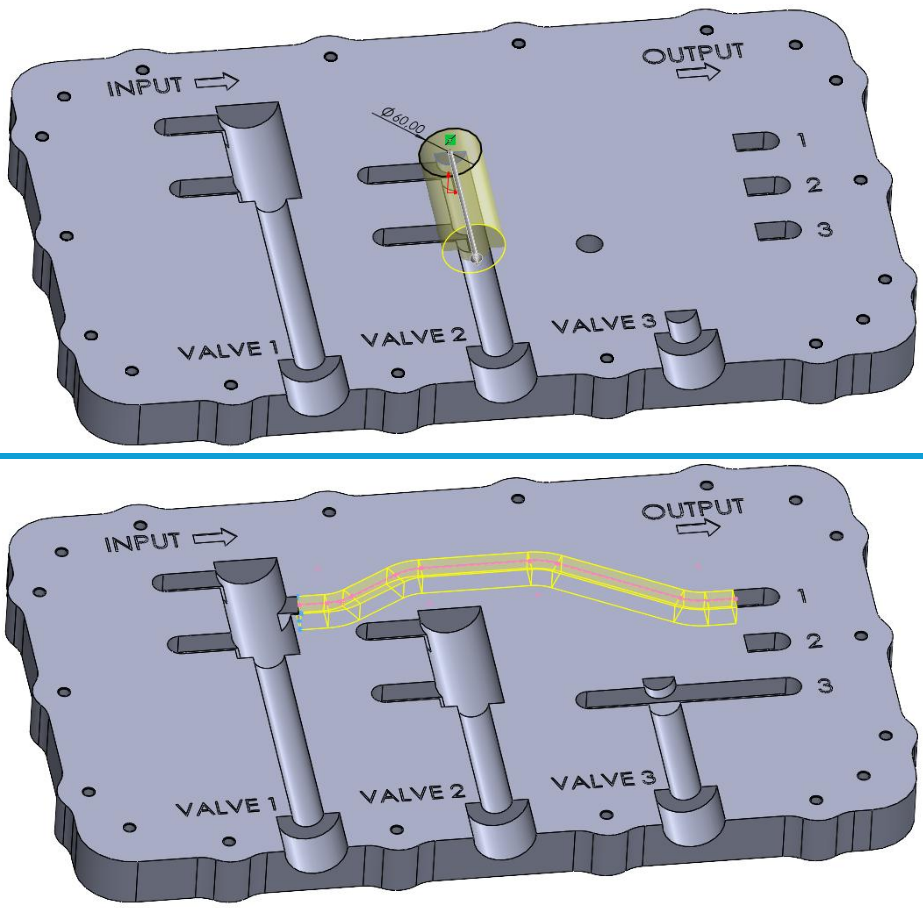


Fig. 23. Example tools used in this model: a) extruded cut, b) swept cut

Parameterisation of selected model elements requires the preparation of global variables to define the values of the relevant dimensions. These include the following:

P – pressure (bar) – variable parameter

G – flow (kg/s) – variable parameter

q – density of oil – constant parameter

b – depth of channel [mm] – constant parameter

Ppasc – pressure (Pa) – pressure converted from bar to Pascal

$$\text{Ppasc} = \frac{P}{100\ 0000} \tag{1}$$

v – flow rate [m/s]

$$\text{v} = \sqrt{\frac{2 \cdot Ppasc}{q}} \tag{2}$$

a – width of channel [mm] max 46 mm

$$\text{a} = \frac{G}{v \cdot q \cdot b} \tag{3}$$

Global variables are defined in the “Equations” folder. Right-click on it and select "Manage Equations" (Fig. 24).

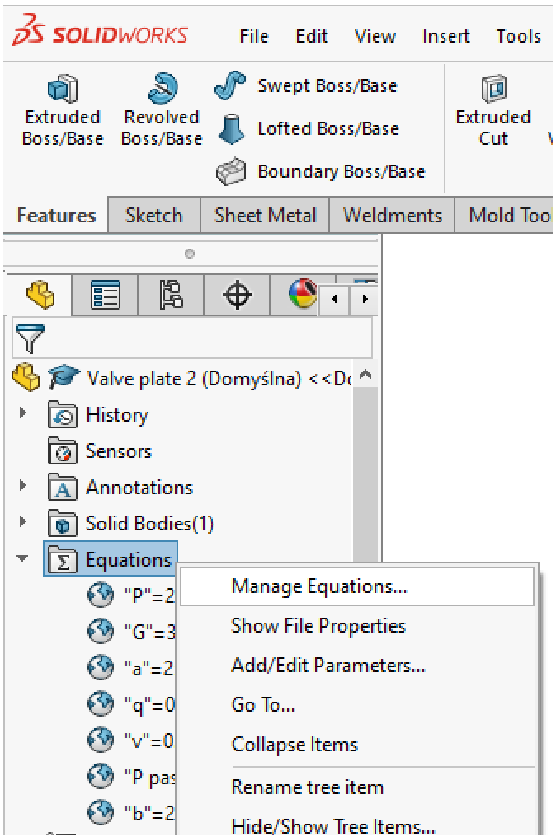


Fig. 24. Manage equations

The chosen values and equations for the global variables are presented below and on figure Fig. 25.

variable parameters	constant parameters	equations
P = 2 G = 3	q = 0.90 b = 20	Ppasc = ("P") / 100000 v = ((2 * "P pasc") / "q") ^ (1 / 2) a = ("G" / ("v" * "q")) / ("b")

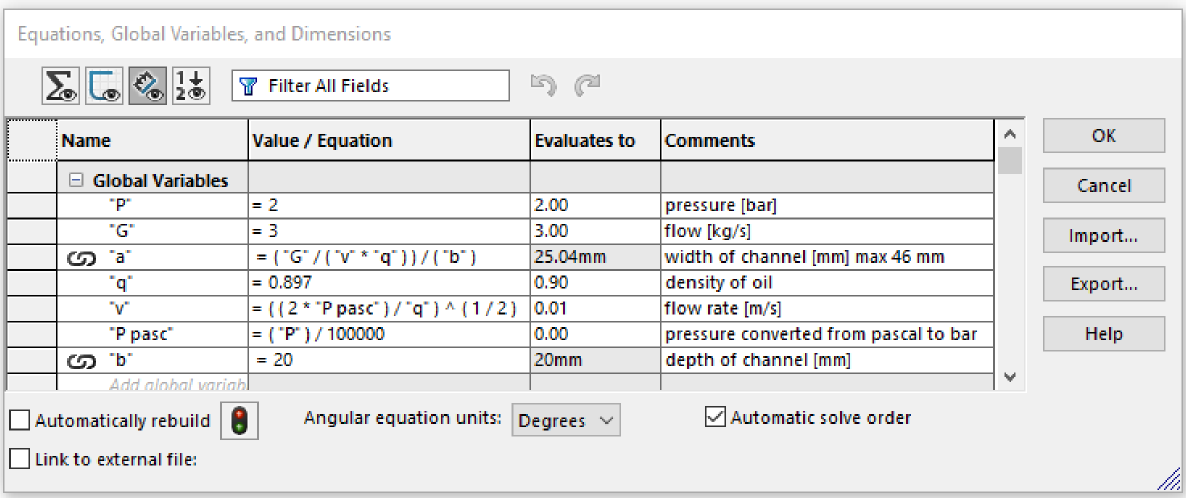


Fig. 25. Global variables, dimensions, and equations

Right-click on dimension and select "Link Values" (Fig. 26).

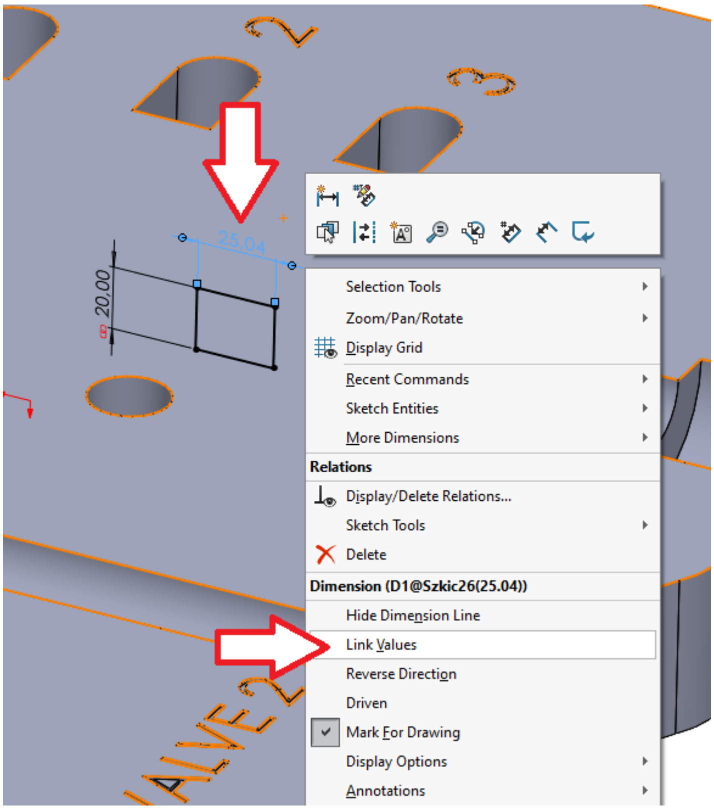


Fig. 26. Linking a dimension to a global variable

Select the appropriate global variable from the list (Fig. 27).

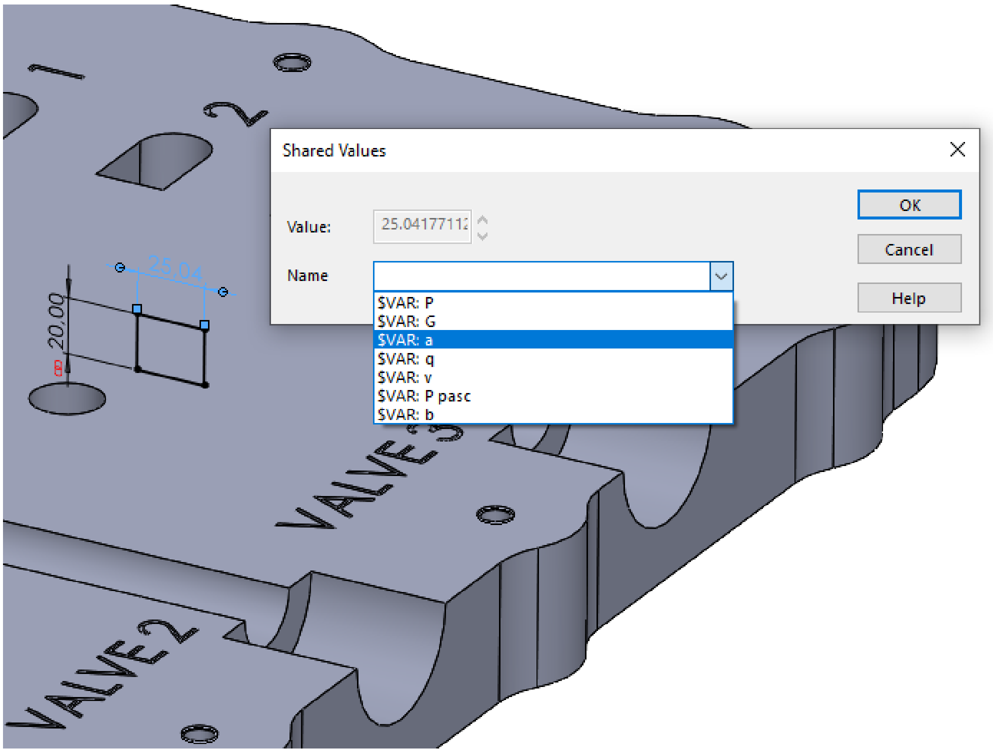


Fig. 27. Selection of the global variable

Dimensions linked to global variables are marked with a symbol as shown in the figure Fig. 28.

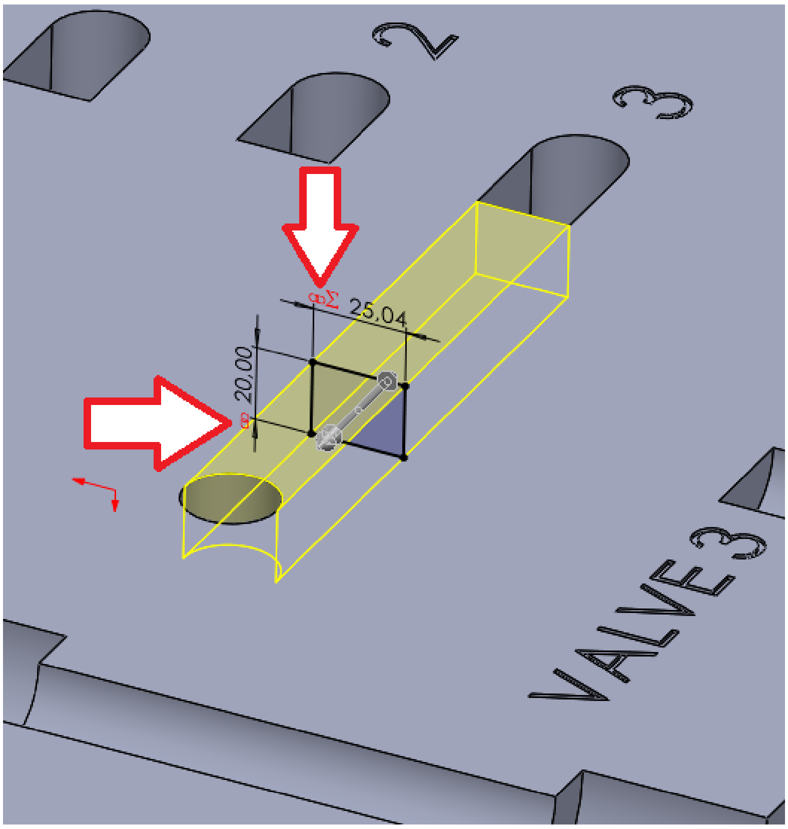


Fig. 28. Dimensions defined by global variables

Changing the pressure (P) and/or flow (G) in the global variables will change the width of the channel. The change will occur automatically, without the designer interfering with the sketches and functions that create the model feature. Examples are shown in the figure Fig. 29.

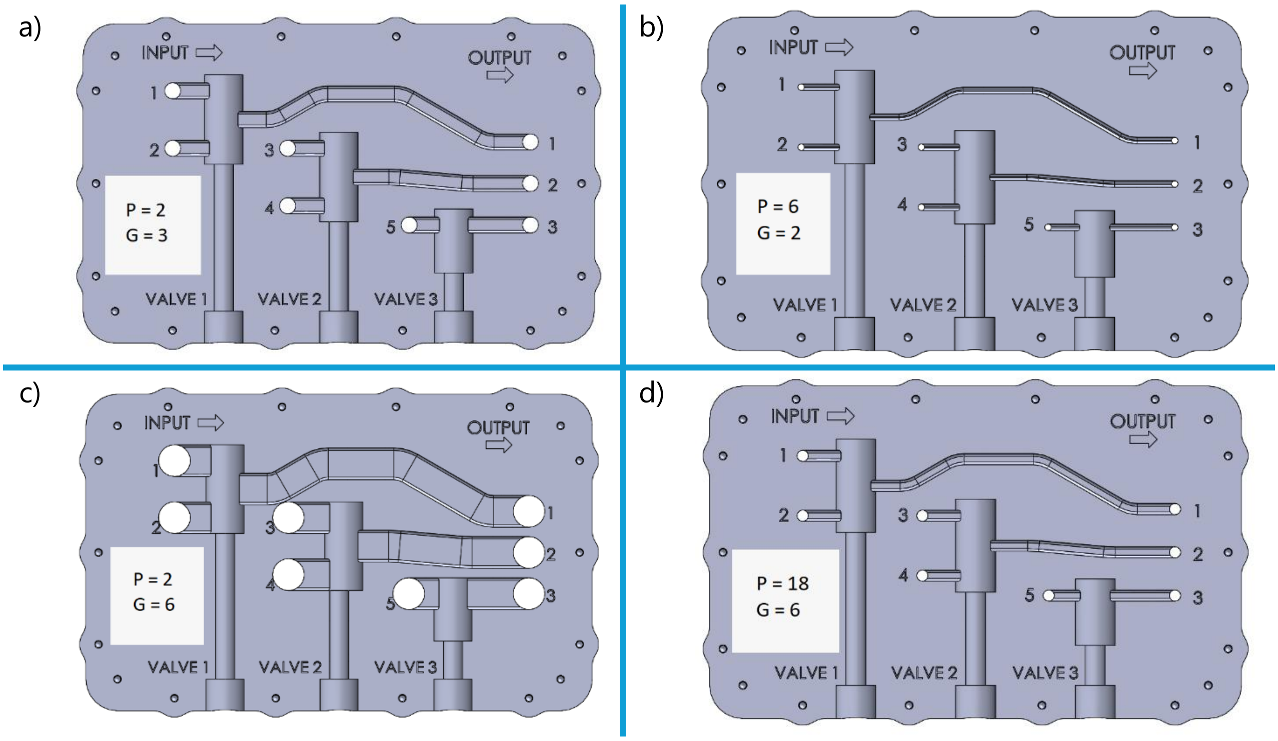


Fig. 29. Channel width configurations for different P and G parameter values:
a) P = 2; G = 3, b) P = 6; G = 2, c) P = 2; G = 6, d) P = 18; G = 6

In this exercise, make your own model. Propose different inlet and outlet channel layouts and valve chamber locations. Use different chamber types. Prepare global variables and define at least one model feature parametrically. You can use the model from the example. In this case, change the positions of the valves and chambers. Change the number of inputs and outputs and route the channels differently.

References

1. https://akpp.pl/remontg6T30E_PL
2. https://akpp.pl/remontgJF010E_PL
3. https://maktrans.net/GBNR-6R-6L2P
4. https://akpp.pl/remontgJF011E_PL