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Possibilities of Applying FreeCAD in Teaching the Technology Subject

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Abstract

The use of CAD systems is currently one of the common educational problems. Their use in the field of technical education can now be considered non-compliant, especially at higher secondary education at vocational technical schools and universities. Their application is also important in primary school and it is necessary to distinguish the difficulty and availability of the program for the school. The FreeCAD is also available for a complex CAD system usable for the needs of technology teachers at lower secondary education. This paper deals with the possibilities of its use in the creation of working ideas.

Keywords: FreeCAD, technical drawing, 3D modelling, sheet metal, technical education

Introduction. CAD systems in the technical education

Teaching of technical subjects is an important pillar of the development of expertise and practical skills in areas such as engineering, architecture, electrical engineering, design and production. This kind of education combines theoretical knowledge with practical applications aimed at solving design, production and technological and procedural problems. The basis is systematic education leading to technical capabilities from the level of primary education, through lower secondary education, higher secondary education at vocational technical schools to university education at universities.

In recent decades, this process has undergone a fundamental transformation thanks to digitization and the wide use of computer tools, where CAD (Computer

Aided Design) also plays a key role (Kučerka, 2025). These systems allow the creation of 2D technical drawings and 3D models that define the exact dimensions and shape of the proposed products and are essential for modern project activity. CAD systems provide a practical tool for students that supports the development of spatial imagination, accuracy and ability to create complex products of products (Damaren et al., 2025). In addition to the design, they allow simulation, analysis and integration with production systems, including CNC and 3D technologies, providing students with a comprehensive view of the entire production process from design to implementation.

The use of the CAD system has become the subject of research of several scientific publications, focusing on the possibilities and effectiveness of its application in teaching. The implementation of research in CAD teaching has shown the possibility of improving the student experience of learning and increasing the academic level. CAD courses taught in coordination with other subjects allow students to develop complex competences related to technical drawing and graphic communication (Redon Santafé, Piquer Vicent, Haba Guerra, García-Domínguez, 2016). Research in the field of CAD/CAE systems for the creation of e-learning courses of technical objects at universities points to the potential of these technologies to modernize the educational process and improve the interactivity of teaching (Kozlov, Harlashkin, 2018).

According to Park, Han, and Lim (2025) The use of CAD technologies in teaching technical subjects significantly contributes to the development of key skills for the 21st century, such as critical thinking, collaboration and computing thinking, which is the basis for success in integrated STEM education. One of the key questions in CAD education is the choice between specialized academic software and commercial solutions. Research points to the need to consider specific needs of the educational process in choosing the right tools (Rubio, 2014).

The use of CAD systems requires some degree of management of work with the program (Koziar, Hubal, Burchak, Botviniev, Saveliev, 2025) and knowledge from other disciplines such as mathematics, technical drawing or design. Their use in lower secondary education in the form of pupils' education has not been the subject of research so far. In our paper, we will focus on the analysis and exploring of one of the freely available Open Source CAD software FreeCAD in terms of preparation for the lesson and the creation of working ideas for the subject of technology at lower secondary education, based on the curriculum of this subject.

The subject matter of the study. FreeCAD software

FreeCAD is freely available open-source CAD software for 3D modelling, designed especially for engineering and technical drawing. Unlike many commercial programs, such as Solidworks or Autodesk Inventor, FreeCAD is

completely free and can be used without any license fees in education. Its main advantage is that it is modular and customizable, which means that it can be extended with new functions using different desks (workbenches).

Working in FreeCAD is based on parameter modelling. This means that every object that is created retains its history and can be changed at any time its parameters.

The user interface is divided into several main parts:

1) Workbar panel: It is located up and is used to switch between different modules (workbenches). Each workbench is specialized in a type of work, such as 2D sketches, 3D modelling, or technical documentation.

2) The model of the model: on the left side there is a tree structure that shows all the objects you created (parts, operations, sketches). Clicking on individual items you can return to and edit them.

3) Working window: The central part is used to visualize the 3D model.

4) Task and features panels: On the right side, panels are displayed to modify the properties of selected objects and to control various operations.

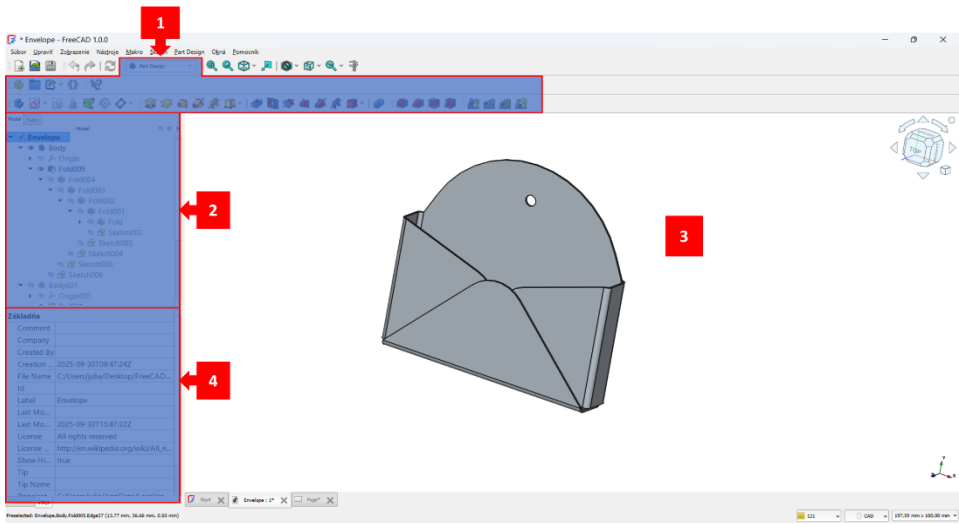

Figure 1. FreeCAD user interface

Research methodologies and tools

The research methodology consists of three main phases:

1. Phase 1 The study of the documentation is focused on official documentation from the FreeCAD in the form of tutorials in order to understand theoretical foundations and functions of the desk.

2. Phase 2. The object is a metal bent envelope and its developed shape of the clip, designed to store paper links. Workbench tools are tested for sketching,

modelling, sheet metal formation and drawings. The output is a file with a working theme model in FreeCAD. The implementation of this section is in 3 steps:

- Step 1: Creation of 2D sketches in Sketcher module and their transformation into 3D models using Part Design.
- Step 2: Design and modelling of plate components in Sheet Metal module.
- Step 3: Creating a technical drawing in the TechDraw module.

3. Phase 3.: Analysis and evaluation to critically evaluate FreeCAD based on the practical experience of the experimental phase. Partial appreciation of modules: Sketcher, Part Design, Sheet Metal, TechDraw.

Development (analysis of research results)

The FreeCAD is a comprehensive CAD System for creating models, assemblies, drawings, specific technical parts such as thin sheets, but also various analyses or programs for CNC or 3D printing. It contains a wide range of modules in a program called workbenches or workbenches. The basic working environment can be considered relatively intuitively responded to by traditional user interfaces that the user can be accustomed to from other regular user programs. Selected workbenches are pre -installed within the basic installation. The user can make other workbenches in the Addon Manager in the offer, to which there is a relatively good description for what they are for. For our needs, it was necessary to finish the workbench Sheetmetal, which was not part of the basic installation. Before working with individual workbenches and the environment, it is necessary to make the input user settings in the Tools menu to adjust the user preferences.

Creation of 2D sketches in Sketcher module and their transformation into 3D model using Part Design: Workbench Sketcher work environment can be considered intuitive, consisting of the necessary tools of elements (point, line, arc, circle …). An important role is played in the creation of sketches, a group of tools by which the user can define all the necessary bonds of the elements (horizontal/vertical binding, parallelism, equality …) and dimensional links (dimensions) that determine the size or location of the individual elements. From the point of view of tool work efficiency as a disadvantage, the unconventional character of the tool, such as the line and the grouping of dimensional bonds into the drop-down menu, can be considered as a disadvantage. If the shape and all the necessary dimensional and positional bonds are comprehensively defined, the sketch will change the colour, and the sketcher can be stacked by the Close command in the TASK panel that the user returns to the part of the Part Design. In this workbench, the user chooses several 3D volume sketches. This volume part can be manipulated in space by sliding, zoom or rotation. Or choose an area in which a new sketch can be created and gradually model more complex shapes by adding or removing volume based on a sketch, or various shape elements such as collision, rounding, normalized hole or thread.

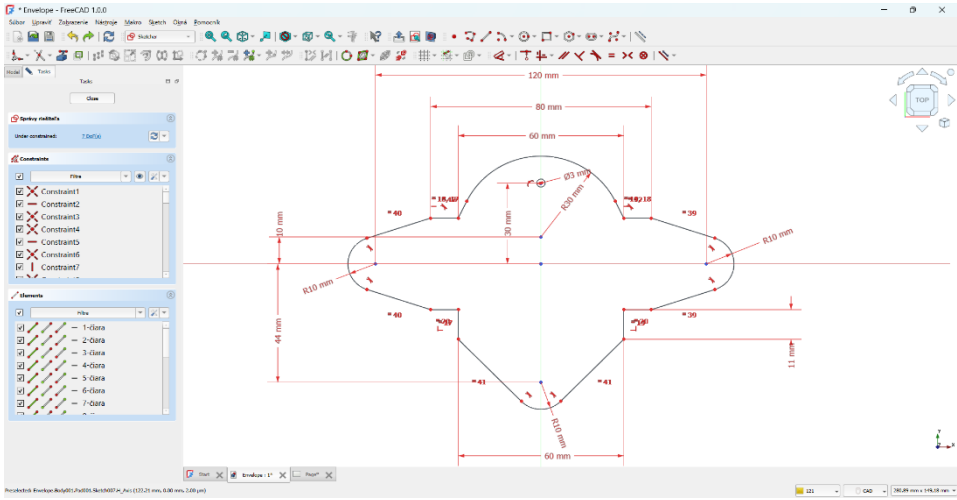

Figure 2. Sketcher – 2D sketch creating

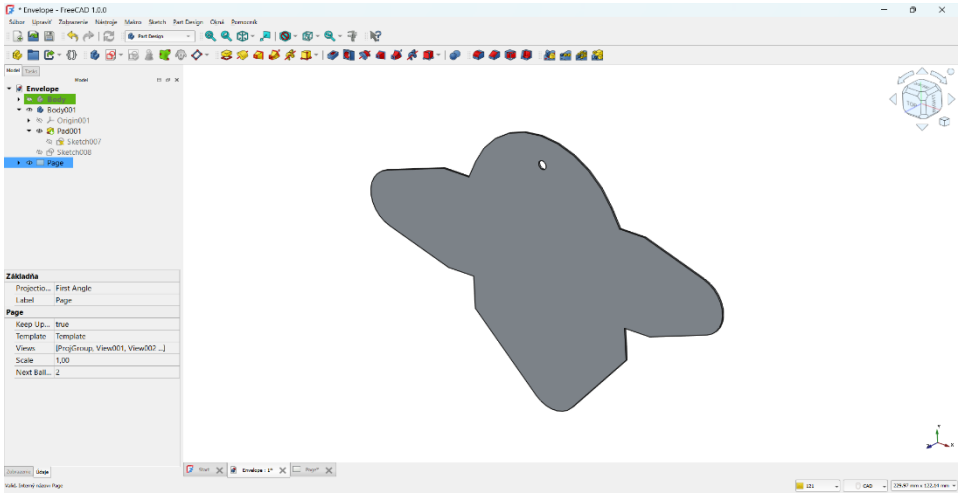

Figure 3. Part Design – 3D body creating

The design and modelling of plate components in the Sheet Metal module: The creation of a bent model from the developed shape cannot be considered in principle. For individual bends, we defined a line on the surface of the inner bend, which corresponds to the bending line and gradually bent the flat shape into the desired shape using the Fold and Wall tool. The disadvantage can be considered that workbench does not have the same Sketcher tool as the line causing repeating steps and switching between Part Design and Sheetmetal tools.

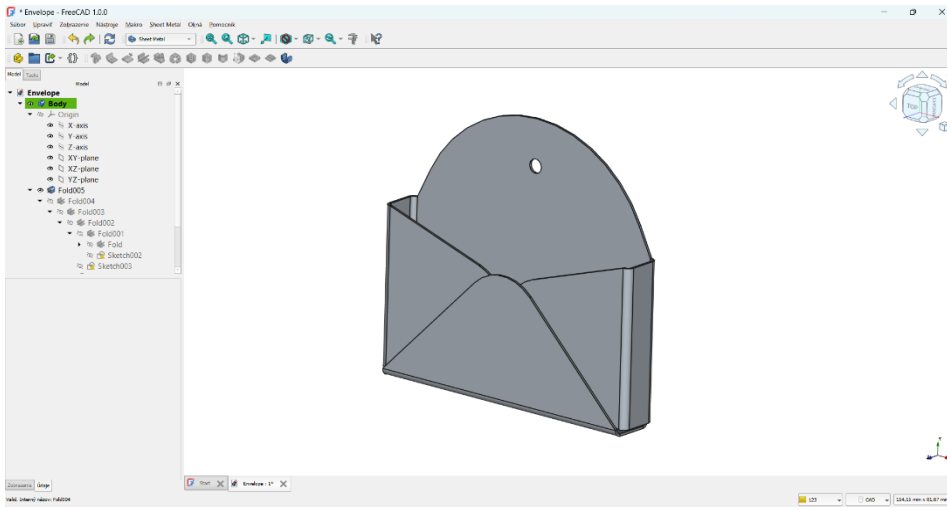

Figure 4. SheetMetal – metal bending

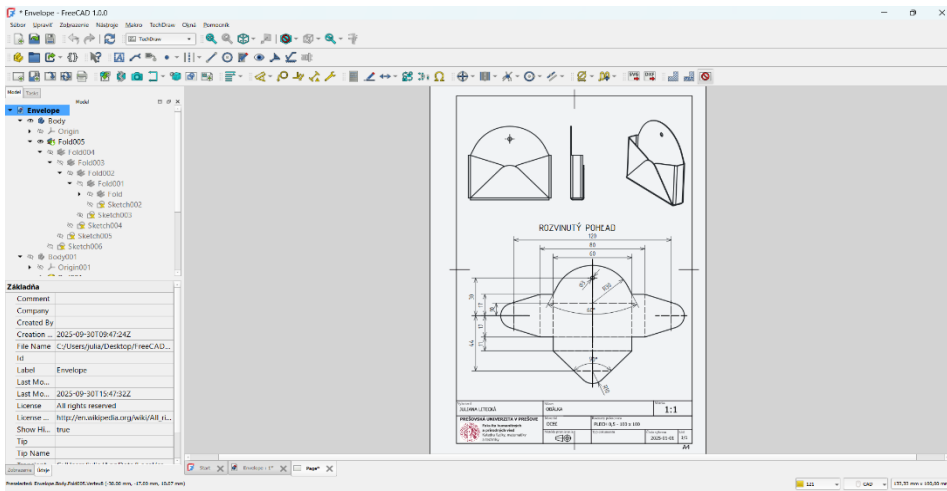

Figure 5. TechDraw – technical dawing

Creating a technical drawing in the TechDraw Module: Module Tech-Draw provides a wide range of tools and options for creating drawings, including the necessary views, dimensional dimensions and drawing elements (axes, centres of circles and arches, notes, link lines or special symbols). For this module, it is necessary to make a relatively wide range of settings before starting, so that the view corresponds to the appropriate technical standard determining the criteria for drawing drawings. Here we have to say that the possibilities are not sufficient and

not intuitive for the user. An important role in the creation of drawings is the template of the modified drawing format according to the user's needs or the institution. Templates can be created through the Inkscape vector program and uploaded to the appropriate folder with predefined templates of drawing formats according to multiple standards. Adding views and their settings is all the necessary options for basic display in the first or third quadrant. There is no possibility to choose a colourful variety or look with invisible edges in the settings window. There is also no possibility to not display lines from rounded edges. When combining the views of several bodies as in this case, the developed shape and bent shape, it is necessary to create two separate files and merge these into one, so that the views can be added gradually. All kinds of necessary code can be added to the menu, it is easy to align them. We consider it insufficient, however, but tools for the creation of technical elements such as symmetry axis, centres of circles and arches, legacy lines. These elements cannot be mass edited in bulk, the thickness and type of line does not answer the requirements of the line type even after the initial settings. There is no possibility of free drawing and editing of lines, such as bending lines. We had to project these after drawing in the sketchbook in a developed view. Overall, this workbench would require a more detailed overwork aimed at automating tools in accordance with the requirements of technical standards.

Conclusions

We consider the CAD System FreeCAD to be a very suitable tool for designing working ideas and creating technical documentation. For developers behind the main, we recommend that you remove key shortcomings in tools and options for settings of individual workbench. For the motivation of teachers to use this program, we recommend training in the form of video tutorials or workshops focused on the FREECAD tools and possibilities.

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