Comparing SolidWorks and Creo Software on 3D Modelling

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Bachelor's thesis

Degree programme in Mechanical Engineering and Production Technology.

May 2022

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Mechanichal Engineering and Production Technology
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Year 2022
Subject Comparing SolidWorks and Creo Software On 3D Modelling.
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The main objective of this thesis is to compare CAD programs Creo and SolidWorks. This thesis compares the two modelling tools using modelling a jet engine with Creo and SolidWorks. Due to I had the privillage to learn at different Universities, it gaves me the chance to take different CAD program softwares. This situation gave me the chance to ask my self to compare the different Cad softwares that I got the chance to learn at different time at different Universities.

To compare the two softwares, I am going to model Jet engine with Creo software and Gas jet engine with with SolidWorks software and my data will be collecting Models. Based on this, I am going to compare the two softwares with the way file is saved, ease of use, assembling and Certification.

I did not produce the model I did with different softwares in the HAMK workshopes. It is one of the limitations of my thesis.

Keywords Modelling, Creo and SolidWorks
Pages 15 pages

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Foreword

I started to study in Mechanical engineering and production technology program at Savonia University of applied sciences, which is found in the northern part of Finland. I studied there for one year and then transferred to University current Häme University of applied sciences, HAMK. I continued to study in the same program which I started in Savonia. After I studied most of the courses in HAMK, I got the privilege togo study as an exchange student for one semester in Canada at the Confederation College in Thunder Bay. Studying at different Universities and colleges has given me many life experiences and challenges.

One of the privileges was to study different 3D CAD softwares. This opportunity allowed me to compare the CAD software that I have learned at different Universities. These opportunities and challenges inspired me to adopt it as my thesis topic. I hope that my thesis work will contribute to an informed understanding of CAD modelling software that I have learnt during my study. There are some limitations in my thesis work. I hope this could encourage other students to study more CAD modelling software. When I was doing the practical part of developing and modelling with respective software, it would be very difficult to materialize my thesis without my advisor's frequent help and monitoring. I want to express my appreciation to my thesis supervisor Jaakko Vaska for his constructive and valuable suggestions during the writing and development of my thesis.

1 Introduction

1.1 History of CAD

According to the technical Design college, DIGITAL SCHOOL of Canada, until early 1970s, Compter Aided Design (CAD) software was not cable of designing 3D designs, yet the 2D drawing designed by CAD software started to change the way the industry viewed design and designing. The advancement in CAD software enabled engineers to draw 3D designs. By the late 1970s, solid modelling software emerged, enabling users to combine basic geometric shapes to build solid modelling. The advancement in 3D solid modelling continued to grow and introduced new algorithms and theories of 3d designs. In the 1990s, the rendering of 3D models was integrated into the then moderndays 3D modelling software. Currently, the CAD software focuses on designing and manufacturing automation, enabling increased efficiency and speed in manufacturing (Anonymous, 2022).

CAD software is used to create electronic files for printing machining and other manufacturing operations. CAD software enables not only an increase in the productivity of the designers but also improves the quality of design. The benefits of CAD software are supporting both designers and engineers across a range of industries, including automation, architecture and aviation (Geddes, 2020).

Nowadays, in 21 century, it is common to see real-world objects modelled using 3D computer-aided designs. The modelling of real-world objects involves numerous mathematical and geometrical calculations behind the scene yielding virtual reality using 2D and 3D techniques. Currently, modelling has been enhanced with modern modelling techniques easing the designing process through their features such as extruding, revolving, scaling and adding pattern elements. Besides, state-of-the-art designing tools enable the realization of virtual objects having similar properties to the real ones. Thus, virtual objects have the look and feel of natural objects having similar attributes such as size, density, weight, physical properties and appearance. Virtual objecting having real-world object

characteristics has enabled designers and engineers to inspect and test them using computer simulations (Shakurova, 2019, p.7).

1.2 Objectives

This thesis focuses mainly on the users of CAD software for educational purposes or public design projects. The primary purpose of this study is to compare the desktop CAD software, Solidworks and Creo, to identify which software is more functional for users, especially for engineers.

The main objective is achieved by collecting data both from Creo and Solidworks CAD software. This thesis presents a model of similar engine products with the two software, starting from developing the parts up to assembling them with the respective software. Finally, the results of the two software are presented in thesection 4 of comparison of Creo and Solidworks section. The results presented in this thesis compare the two software from modelling the part to assembling the product.

1.3 Relevance to working life (Practical use)

SolidWorks is known for its user-freindliness and several features compared to other CAD software in particular CATIA. Thus, it is common in academia as well as industry used by designers and mechanical engineers. Besides, the company SolidWorks bearing the name of the software itself provides certification programs tailored to designers or engineers with scopes rangining from beginners to experts. The certification programs open up opportunities to new graduates to take to the next level in their professional career. The most challenging part of studying SolidWorks is that it demands a powerful computer and SolidWorks may conflict with other software installations in the same computer (Le, 2018).

2 SolisWorks

SolidWorks is the most widely used CAD and CAE software, and it is especially known for its adaptability and its countless benefits, such as its widespread use in the industries, user-friendly interface, automation, and simple learning. SolidWorks employs a process tree, and it follows a parametric based approach to follow the history of the modifications of the part. If one wants to draw a 3D object, she/he needs to draw a 2D sketch using basic geometry as the starting design (Shakurova, 2019).

Primarly, we can create 3 basic things in Solidworks software

1. Part
2. Assembly
3. Drawing

   All the things I mentioned above have their own file format.

2.1 SolidWorks part

The fundamental building block of SolidWorks is the 3D part. SolidWorks design starts with the creation of a new file by choosing the template "part". After the template is created, the workspace will be visible where the "Tree Items" toolbar will be activated, which displays the tools that are necessary to sketch your drawing. Besides, SolidWorks supports the editing of elements through the provision of visual feedback by dragging or entering numeric values. Fillets and Chamfer could be drawn asymmetrically, and they could be gradually adjusted by increasing or decreasing them by dragging their edges (Shakurova, 2019).

2.2 Assembly in solidWorks

2.2.1 Assembly Definition

An assembly is a collection of related parts saved in one SOLIDWORKS document file with a .sldasm extension (Systemes, 2015).

2.2.2 Assemblies:

“Contain anyware between two to more than one thousand components, and components can be parts or other assemblies called subassemblies” (Systemes, 2015)

Display motion between related parts within their degrees of freedom. The components that are found in each assembly are defined using assembly mates with respect. The assembly components are identified using the types of mates found in SolidWorks, such as concentric, coincident, and distance. Furthermore, it is possible to combine multiple parts that are fitting together to make assemblies, and mates define the direction of movements that is allowable (Systemes, 2015).

The positioning of components with respect to each other is done through the utilization of mates. Besides, the positioning process defines how each component rotates and moves with respect to each other. Also, mates define geometric relations such as tangent, coincident, and perpendicular, and each mate is suitable for a combination of geometric shapes such as planes, cylinders cones and extrusion. For instance, you could mate a cone with another one, and the valid types from these mates could include concentric, distance, and coincident. (Systemes, 2015).

You create assemblies using two basic methods, namely bottom-up design and top-down design. It is also possible to combine these two methods. With any of the two methods or a combination of them you could create or mate the components to create the assembly or subassembly (Systemes, 2015)

It is simple to edit features as dragging an element is accompanied by instal visual feedback to enable numerical manipulation. Features are typically adjustable in SolidWorks. Besides, Chamfer and fillets are gradually increased and decreased by changing their sizes by dragging the edges. SolidWorks also supports reusing a sketch to create features and geometries that are reusable (Shakurova, 2019).

SolidWorks is a parametric modelling software with straightforward capabilities. According to SolidWorks Tutorial, SolidWorks has six main areas of interface, including Menu Bar, Status bar, Feature Manager Design tree, Command Manager, Head-up view toolbar, and Graphics area (Le, 2018).

2.3 Drawings

The drawings of 2D documents convey a design for manufacturing. Sheet formats and drawing templates are two distinct entities. The SolidWorks software is shipped with a set of

sheet formats and one drawing template (in English and metric). When you begin a new drawing, SolidWorks prompts for a sheet format, and you can use the default drawing template; the size of the drawing is undefined.

The sheet format controls are:

- Size of the drawing sheet
- Drawing borders
- Title block
- Sheet scale

3 Creo

3D solid models are constructed by subtracting and adding geometric volumes and creating the ultimate design of a product. The use of collective geometric shapes allows the representation of new parts and products (MarkCheli, 2013).

3.1 Parts and solid modeling in Creo

Solid modelling uses volumes to build parts of models. The resulting model having volume will be assigned materials having physical properties such as conductivity, density, etc. (MarkCheli, 2013). Part models are collections of negative and positive features.

Features are volumes, and while building parts of a model, volumes are subtracted or added. Also, 2D sketches are extended to 3D shapes. In PTC Creo parametric, it is possible to create features in four ways. In Creo,the modelling starts with a 2D sketch, yet we need to keep the special requirement to create a 2D sketch. 2D sketches must be on the same plane.

2D sketches must be closed, with no openings in the sketch. 2D sketches should not have any dangling edges or freefloating geometry.

The four basic operations of Creo are used to create features or volumes. The four basic operations are: extrude, revolve, sweep, and blend. Extrude uses a cross-section to create volume by extending in following a straight-line path, while revolve uses a cross-section and revolving it around an axis. Sweep uses a cross-section and sweeps it along a curve. Finally, blend takes two or more cross-sections and blends them between a using a curve (MarkCheli, 2013).

3.2 Aseembly in Creo

Product development life-cycle includes assembling parts and subassemblies into a complete model of a product. Assembling parts is the integration of all the pieces and parts that make up a product. Assembling requires the use of degrees of freedom, also known as constraints, to define parts together. Creo uses six degrees of freedom for parts, of which three of the degrees follow the translation axes side to side, forward to back, and up and down movements. The other three degrees of freedom follow uses rotation axes and rotate around axes X, Y, and Z. Also, constraints enable designers to reduce degrees of freedom to orient parts in an appropriate way (MarkCheli, 2013).

3.3 Drawing in Creo

It is explaind in the how to model every thing that A product brief is a one-page document that shows different views of the product with notes that highlight the important aspects. You can use notes and arrows to identify and highlight features of your design. You can also use color to highlight. Different views also help identify the important aspects of the product (MarkCheli**,** 2013).

4 Compartion of Creo and SolidWorks

This thesis compares two CAD software Creo and SolidWork using four points presented in Sections 4.1, 4.2, 4.3, and 4.4. The modelling of a Jet engine with the two CAD software, figure 1 shows a model which is done by Creo and figure 2 shows the model which is done by SolidWorks, is also presented in this chapter.

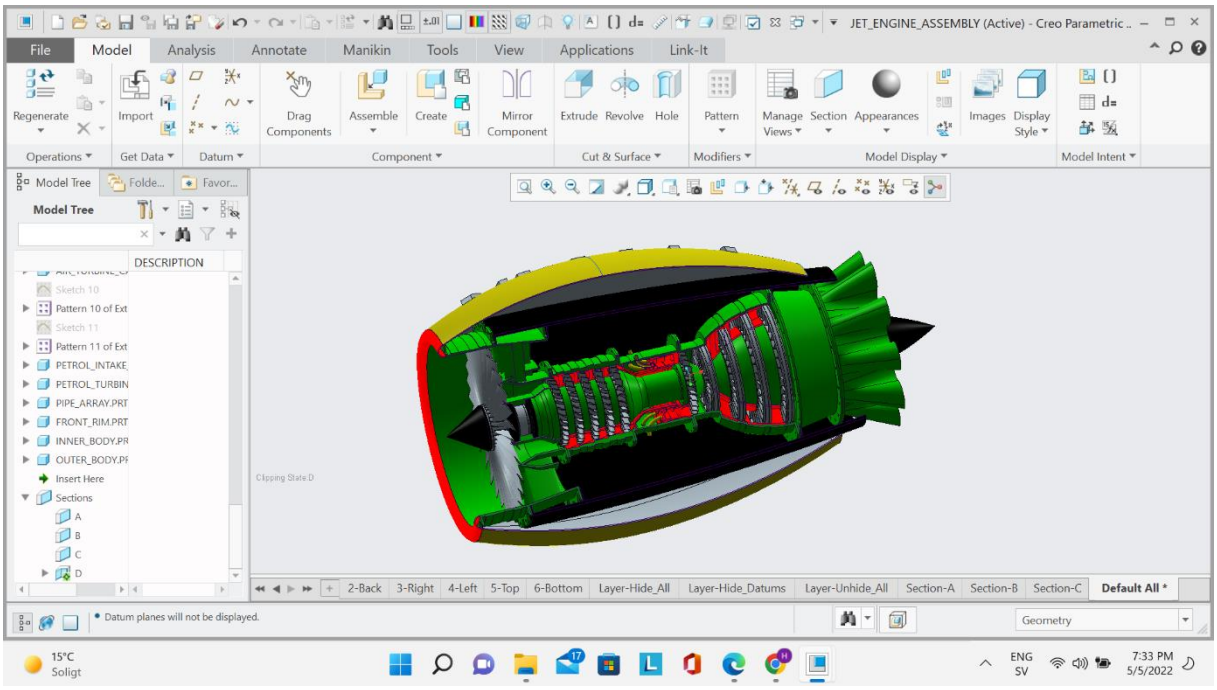

Figure 1 - Jet Engine modelling

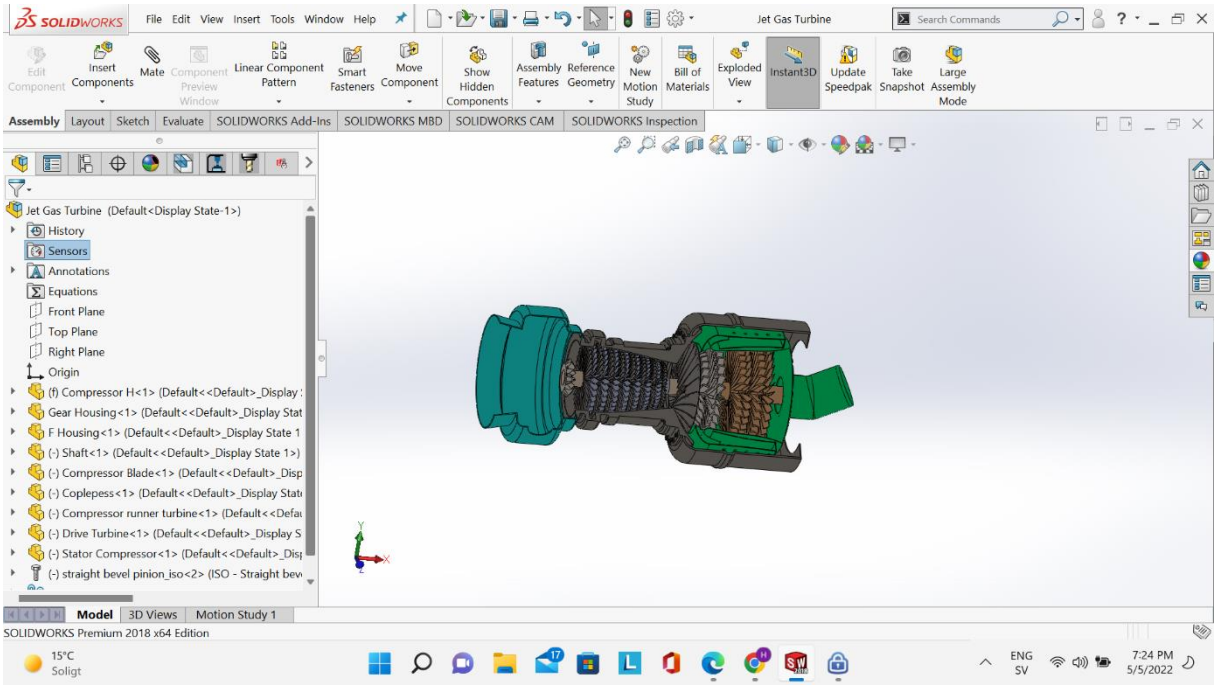

Figure 2 - Jet Gas turbine Engine modelling with SolidWorks

4.1 The way file is saved

In Creo CAD software, before I start modelling a given part and assembling the product, it is a must to set the file folder where the file is to be saved. It is done by “selecting working directory”. But when I use SolidWorks, there is no such kind of restriction. I can use the way I use Microsoft word. In SolidWorks software, I can creat the file folder for software at any stage of the modelling work. Creo software does not allow to do what I can do in SolidWorks.

4.2 Ease of use

After learning and working with the two software, SolidWorks and Creo, I realize that SolidWorks has better ease of use. SolidWorks software is very intuitive. But when it comes to Creo, I have to know the rules how it is going to be done and I have to follow it.

4.3 Assembling

When I assembled the Jet engine with the two software, it was almost the same type of procedure. Both of the softwares follow similar patterns. But, I have seen a clear difference in processing the modelled product. Whenever I open what I have assembled in SolidWorks, it takes a long time. I have read article which compares the speed of the two softwares the spead of processing different complex modelles. In my thesis, I could not use the HAMK Laboratory to measure and compare it.

4.4 Certification

SolidWorks CAD software has one interesting feature when it comes to learning software when we compare it with other software. It can give certification at a different level. At the moment, Creo software does not give certification when someone learns the software. SolidWorks Software gives a better opportunity for someone who learns the software. After all, all this efort to learn is to get the job after education. This certification is a good evedence for those who look after to get a job. In this aspect, SolidWorks softer is better from other CAD softwares.

5 Conclusion

It is not easy to come to a concrete conclusion and say this software is better than the other. I think it is necessary to do additional and in-depth research concerning the two software. I recommend Creo software for educational institutes like HAMK. I have observed that Creo software is profound in its application and significantly stable in many aspects. I did the modelling of the Jet engine with Creo software and Gas Jet engine with SolidWorks software. I did produce it in the HAMk workshope or did not print it with 3D printer. It is one of the limitation of my thesis. It needs to be done more research on the topics.

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