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MECHANICAL DESIGN PROCESS IMPROVEMENT

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Opinnäytetyössä käsitellään Saskenin mekaanisen suunnittelun CAD-sovelluksen vaihtoa. Saskenilla on ollut toista vuosikymmentä käytössä CAD-sovellus nimeltään CATIA V5. Sovellus on varsinaisessa 2D- ja 3D-suunnittelussa tehokas työkalu edelleen, mutta sen automatisointi, liitännäisominaisuudet muihin sovelluksiin sekä osa- ja kokoonpanotietojen käsittely ovat osoittautuneet asioiksi, joita tehostamalla parannetaan työn laatua ja vähennetään inhimillisiä virheitä. Opinnäytetyössä esitetään uusi CAD-sovellus, Autodesk Inventor joka on valittu Saskenin uudeksi CAD-sovelluksesi. Työssä käydään läpi suunnittelun osa-alueet, joissa on havaittu puutteita, ja kuinka uusi CAD-sovellus ehostaa toimintaa näillä alueilla. Osa opinnäytetyötä oli esittää kysely Saskenin mekaniikkatiimille, jossa kysytään kokemusta uudesta CAD-sovelluksesta ja sen käyttöönotosta.

Asiasanat

Autodesk, Inventor, Dassault Systems, CAD, CATIA, PCB, ID, Molding, SLA

ABSTRACT

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This thesis is about Sasken mechanical design process switching CAD-software. For a few decades Sasken has been utilizing CAD-software called CATIA V5. For pure CAD-designing the software is still capable, however the software is lacking options in automatization, add-ons, and compatibility to other softwares, as well as part and assembly information handling have shown to be objects that requires improvements to enhance the quality of designing and minimizing human mistakes. The new CAD-software is chosen to be Autodesk Inventor. In this thesis, the different stages of the mechanical design process are presented and described as to how does the new CAD-software improve the operations on these cases. Part of the thesis was to do an inquiry for the mechanical team of the CAD-software changing. Inquiry will give information as to has the team been satisfied with the options that the Autodesk Inventor has given to these issues that was previously seen with CATIA V5.

Key words

Autodesk, Inventor, Dassault Systems, CAD, CATIA, PCB, ID, Molding, SLA

TERMS

Autodesk

Autodesk is a software house founded on 1982. Autodesk is a developer of the Inventor CAD-software.

CAD

CAD stands for Computer Aided Design. CAD is a designing method describing the 2D or 3D design performed on personal computers.

Dassault Systèmes

Dassault Systèmes is a software house founded in 1981. Dassault Systèmes is a developer of the CATIA V5 CAD-software.

ID

ID stands for Industrial Design. ID is a design phase which consist of products appearance, functionality, and manufacturability.

Molding

Molding is a process which is used for mass-production of the parts.

PCB

PCB stands for Printed Circuit Board. PCB is a board which is inside the product that has different types of components mounted to make the device work with the software.

Sketch

Sketch is a tool inside the CAD-software on which the main shapes are drawn. Sketch objects can be later extruded to 3D model.

Snap Fit

Snap fit is a feature on plastic parts. Snap fits are used to assemble plastic parts together. Both parts will have counter parts that fit together.

SLA

Stereolithography 3D Printing is a rapid prototyping process. SLA provides a mass production quality part with rapid prototype method.

Vault

Vault is a product data management software which is part of Autodesk ecosystem.

TIIVISTELMÄ

ABSTRACT

TERMS

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1 INTRODUCTION

I work at a design house called Sasken Finland as a mechanical design engineer. For a few decades there has been the same CAD-software used for mechanical designing of the products. The current product in use is Dassault Systems CATIA V5. As a pure CAD-design software, it is a powerful tool, but to get the most out of it in mechanical design aspects it has shown quite many limitations. For example, part or assembly information transfer from one software to another has been done manually, which has been shown to be very time consuming and adds steps for human errors.

The mechanical team has been attempting to develop current CAD-software in the occurred limiting factors, but the software does show its age and there simply are cases where it is not possible to enhance the situation. The mechanical team in co-operation has agreed to move forward and implement CAD-software which would have the potential to automate stages, make the workflow smoother and more efficient but also improve design quality by giving new tools and options. The selected software that Sasken mechanical team has decided to go for is Autodesk Inventor.

For this thesis there was utilized a Plan-Do-Act-Check method for detecting the lacking functions of the current CAD-software. These functions were analysed on how the Autodesk Inventor has been able to improve the situation. During thesis process the mechanical team was inquired of the CAD software change on how it has benefitted the design process. Outcome of the inquiry is analysed at end of the thesis.

2 SASKEN FINLAND OY

Sasken Finland Oy is a design house whose headquarters are located at Central Ostrobothnia, Kaustinen. Sasken Finland Oy was previously known as Botnia Hightech from 1989 till 2006. Indian based Sasken Technologies acquired the company during 2006. Sasken Finland Oy has a long history in multiple design departments such as radiotechnology, electronic-, antenna-, audio- and mechanical design. Botnia Hightech was a subcontractor for Nokia in the phone industry and dozens of Nokia phones were designed with Botnia Hightech.

Sasken Finland Oy today has three offices. Its headquarters in Kaustinen, is hardware design based. Tampere site is focused on software development. The Vantaa office specializes in marketing and sales. All-in all, Sasken Finland has roughly 70 employees.

3 SASKEN PRODUCT DEVELOPMENT PROCESS FLOW

In product development projects Sasken utilizes the waterfall process type. It has proven to be the most efficient way to handle the product development projects. Each of the design departments work close-by to make sure project flow is concurrent, information transfer is undisturbed, and it enables rapid decision making in many cases. Sasken waterfall design process is represented in the Figure 1. below.

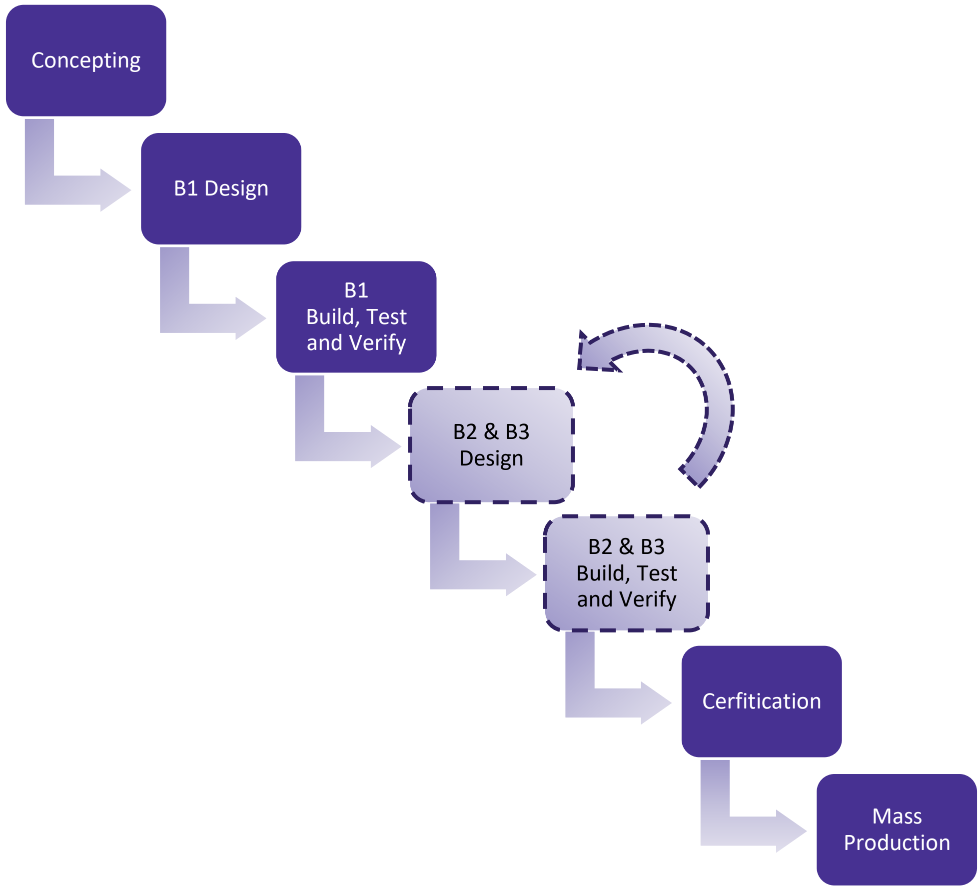


Figure 1 Sasken Product Design Waterfall Process

Product development projects start with the concepting of the product, through that phase begins the detailed designing of the product. There can be multiple builds of the product (B1, B2, B3). Build count is based on the complexity of the product, required testing to verify the product operability and possibility to fine tune product from build-to-build. Simple products may have one build and multiple product families may require two to three builds.

3.1 Concepting

Product development projects starts with concepting phases. During the concept phase the foundation for the actual design phase is created. Concepting includes rough mechanical design (i.e., size, weight, connectors, user interface.), PCB layout, electronical component selection. Once the concept design has been locked and the customer has approved the approach, then can be moved on to the actual Design phase of product.

3.2 Build Design

During build design the product is being designed to have a first working unit within a period of two to four months. Use-case, complexity and variations of the product defines how much of the product will be operatable during the first build. First build tends to have protype mechanics available, functioning PCBs and operatable user interfaces and connections. Products are tested thoroughly in every aspect: mechanics, electronics, RF, audio, and software. Based on the outcome of the testing, the next build will have improvements implemented to any possible findings.

3.3 Certification

Certification of the product is required to launch the product for sale in certain market areas. Sasken has created products for multiple different market areas and those have required a different certification approval for the products. There can be products that have variations which are meant for each market type.

European Economic Area requires for many of the products to be sold in that market to have a CE-Marking. CE-Marking is a statement that the product meets the EU safety, health, and environmental protection requirements. Examples of products that require CE-markings are tablets, smartphones, and diagnostic equipment. (European Commission 2023.)

Another example is a Japan market area that requires JATE-certificate which is required for telecommunication devices that are sold in Japan. JATE-certificate is a marking that the product meets Japanese Telecommunications Business Act requirements. JATE-certification is a required example for Bluetooth devices or satellite phones that are to be sold in the Japanese market. During product development projects Sasken will perform required certification tests with accredited certification laboratories to meet the application requirements. (JATE 2023.)

3.4 Mass Production

Products that are designed to be used by the masses requires mass production process to achieve the demand. Sasken can act as a production support partner for the end-customer or take production responsibility with partners to achieve the mass production quantity required by the customer. Successful mass-production requires overall knowledge and know-how of the product, mass production unit testing and verification of the product quality to match the reference design standard.

Depending on the product type and market situation, the mass production lifetime can be from a few years to a few decades. During this period there may be a need for modification of the product. Modifications can be for example, end-of-life component changes and material changes.

4 IMPROVING MECHANICAL DESIGN PROCESS

Perspective for seeking improvement in the mechanical design process is being laid out by the quality leading, especially from the continuous improvement point of view. For this thesis, it was decided to utilize the PDAC-method. Plan-Do-Act-Check-method is used where issues are detected, that can have a planned remedy action, which are checked and verified, but constantly improved. Below is the PDAC-method visualized in figure format which is a continuous loop. (Nicholas 2018, 23)

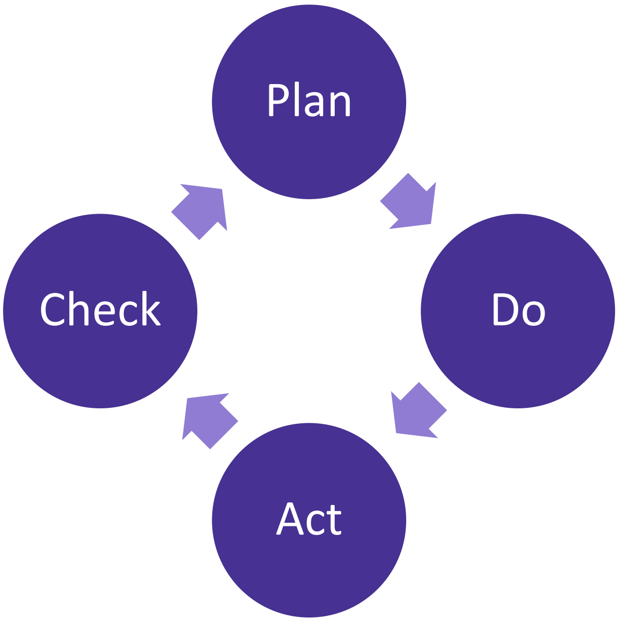


Figure 2 PDAC-Method Visualized

Continuous improvement for mechanical design process, particularly in this thesis is CAD-software orientated, is a way for identifying obstacles or tasks that takes a lot of effort to complete or has a high risk for human error. These tasks can take up resources to identify and report the issues, plan the action to improve the situation, proceed with the recovering actions and verify the situation. As this process can take resource effort, however when the effort is contributed to the value adding portion of the mechanical design process it will improve quality of the product for the customer. (Nicholas 2018, 24)

This thesis will be the baseline of identifying the base tasks on the mechanical design process from the CAD-software point-of-view that are found to be time consuming and has high risk of human mistakes. By identifying these sorts of issues in the design process is the first step to improve the situation. At the end of this thesis there is an inquiry analysis which was given for mechanical team to answer as to how has the CAD-software change improved their mechanical design process on the first iteration.

5 MECHANICAL DESIGN PROCESS FLOW

In this chapter the mechanical design process flow at Sasken is described in step-by-step format. At the end of some design stages, it is also brought to attention the kinds of limitations and difficulties in the process that are caused by the current CAD-software. Improvement of these issues is to be found with more suitable CAD-software.

Figure 3. below visualizes the main steps of the mechanical design process flow. In each of the steps there can be CAD-software being utilized straight on or product information is replicated further on from that. All the steps are represented in the upcoming pages.

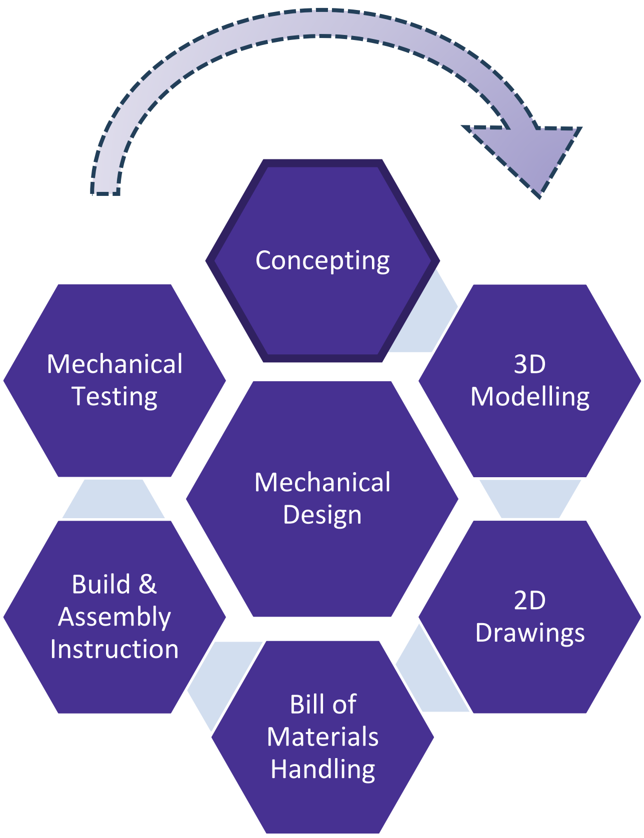


Figure 3 Mechanical Design Process Flow

5.1 Mechanical Concepting

Concepting is the very first step that utilizes CAD-software. During this phase there is a rough 3D modelling done according to the specifications laid out by the customer. Concepting can hold many different design approaches and over time there will be one that is taken ahead further. This phase will give information of potential ID, form factor and weight of the product. As well as quantity of parts, assembly order, layout information of the PCB to the electronic and layout-design departments.

Concepting phase also includes simulations applied to the 3D model. Thermal simulations especially are valuable as electronic components produce heat, which needs to be taken in to account from early stages of the product development. During concepting phase, it is sensible to have any specification changes to be done for the product as they can be put into action most efficiently.

5.1.1 CAD-Software limitations on Mechanical Concepting

During concepting multiple mechanical iterations of the product, it has been found challenging to create different models of parts and assemblies both having similarities but being individual. CATIA V5 has a tool called New-From which creates a new part with a new part code and is not linked to the previous part in anyway. This method is time consuming as the original part or assembly cannot be open at the same time and must be closed to perform the command. It would be an ideal situation to have simultaneously open assemblies and parts that are decided to be modified to another iteration.

Simulations are crucial on the concept phase as the results can change the design dramatically. CATIA V5 does not have built-in simulation capabilities. As for now Sasken has used Autodesk Fusion 360 to perform any simulations necessary. This is not an efficient design method as the model needs to be saved and opened in another software to perform the simulation. The back-and-forth actions are unnecessary and should be minimized.

5.2 Mechanical 3D Modelling

The second phase of the mechanical design process flow is the 3D modelling. This phase is the detail modelling that will be started after the concept design has been approved by the customer. Concept model is a foundation regarding the size of the product and an ID point-of view.

Detailed 3D modelling is about engineering the product with precise manufacturable design guides that are dependable on the materials, material thicknesses, manufacturing methods and use-cases of the product. Main aspects that differentiate concept 3D modelling of the detailed 3D modelling:

- **Draft angle.**
    - High quantity mass-production parts are usually manufactured by molding process.
    - Materials need to have draft angles for the part to be able to be pushed out of the molding tool.
        - Correct draft angle design prevents the part from being stuck in the molding tool.
        - Low angle will cause scratches on the surface of the part as it is being pushed out of the mold.
    - During concepting phase, the external surfaces have the ID based draft angles.
        - Detailed modelling considers internal draft angles also.
    - Figure 4. Below represents how the part is being molded between Mold Core and Mold Cavity.

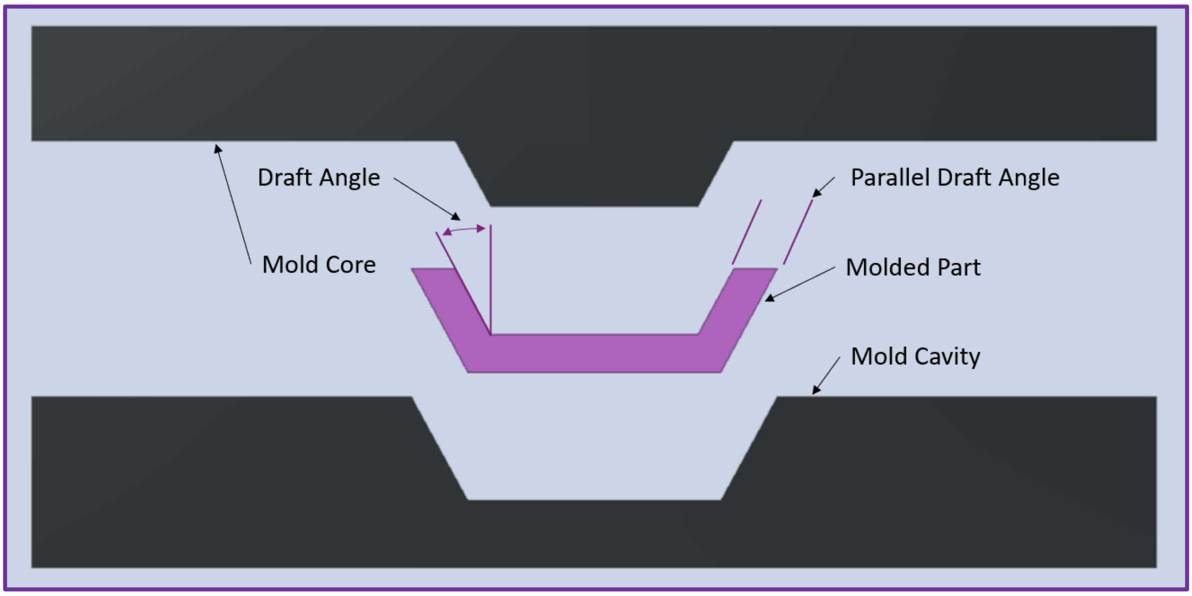


Figure 4 Part Mold Design

- **Part interfaces.**
  - Products with multiple parts.
  - Parts need to be compatible with each other.
    - Suitable gaps with tolerance analysis.
    - Assembly order.
- **Cable design.**
  - During the concept phase there is no actual cabling modelled.
  - Detailed design phase includes all the necessary cabling to be modelled.
    - Cables are routed to the actual places.
    - Cables will be ordered with the measurement data gathered from 3D-model.

- **In-house prototype creation.**
  - 3D printing of the 3D designed models throughout the design process is a rapid way to verify certain aspects of the design once needed.
  - During the concepting phase, the main exterior dimensions and assembly process can be witnessed with rapid prototyping methods.

- **Type-label design.**
  - Commercial product needs to have a type-label which includes specific information regarding that product. For the concept, roughly the main dimensions of the possible type-label sticker are known. Final type-label may have information as such:
    - Product manufacturer and product name.
    - Serial number with Bar-code or QR-code.
    - Date of production.
    - Country of origin.
    - Certification markings.

- **Graphic design.**
  - Products can have multiple kinds of graphical content. These can be for example, manufacturer logo, special model logo, connector icons and phone keypad numberings and special icons. These are designed in high level during the concept phase, detailed curvatures and visual identification are specified in the actual modelling phase of the product.

5.2.1 CAD-Software limitations on Mechanical 3D Modelling

CATIA V5 is a very powerful modelling tool which has been used in the automotive and airplane industries for decades. This heritage is still present as the CATIA V5 may have the most advanced surface-modelling tools available which are beneficial for aerodynamics design. Although, the tools in the CATIA V5 are powerful and capable, the user interface in the software is not user friendly. See below Figure 5. the CATIA V5 with 3D part modelled.

Depending on the workspace i.e. (part, assembly, sheet metal or surface modelling) the toolbar content on the right handle changes the available tools. Tools are represented in an icon model and icons may not always identify clearly as to what is the purpose of the tool. Only by using the tools they get familiar, but it takes time to get used to them regarding the icons and what the tool represents.

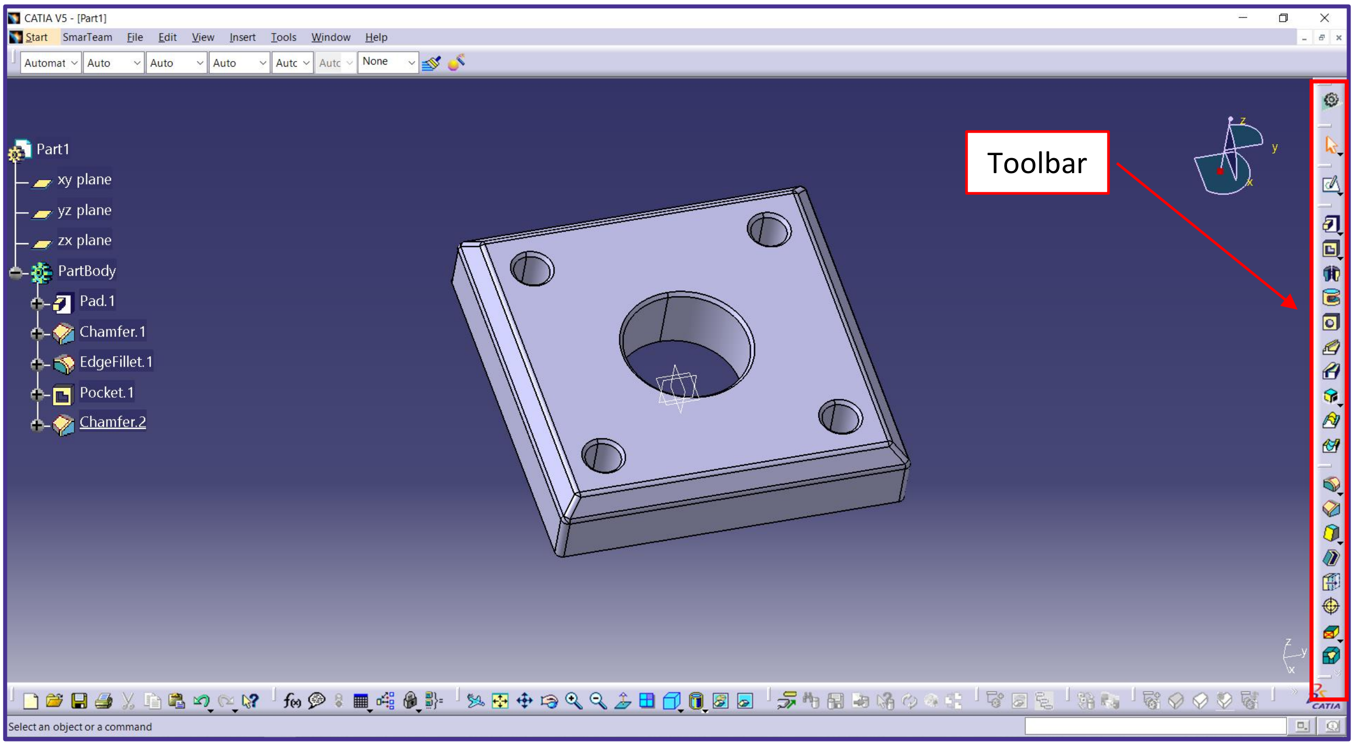


Figure 5 CATIA V5 User Interface

When designing a product which will be produced by molding methods, it requires a draft angle and analysis of the draft angles. On CATIA there is a basic option to add draft angle to a side of part, however there isn't any tools to aid the design process by analysing the draft angle. Manual designing is also required when designing a snap feature, screw tower or lip features. These sorts of aspects are very often designed when the part has plastic materials.

Many products will require a cable designing inside and outside of the unit as well. Current CAD-software lacks capabilities in this aspect. It has shown limiting factors when there are dozens of cables inside the assembly and there is not a tool to efficiently route cables. Cable modelling takes a lot of manual work and there is not constrain tools available to keep the cables connected from end-to-end if there is a need to adjust the components. Therefore, the routing needs to be done manually afterwards.

Overall graphical designing that includes also type-label design is a multiple stage process with CATIA V5. During sketching of the artwork design it is not possible to have text or numbers added straight with the sketch-tool. Those need to be written in a separate 2D-drawing from where these are then copy-pasted to the sketch on the part design. Inside the sketch, the numbers and words need to scale to the correct side to fit the product. During the copy-paste process there are scenarios when the lines are not complete, and this requires time to sort out the sketch.

The supporting functions of the CATIA V5 have been limiting factors on the mechanical design process the most. Supporting functions are information transfer of the parts and assemblies to another design phases, like to the 2D drawing, Bill of Material handling and the Assembly Instructions phases. There are no known property managers to maintain additional information that would be beneficial to be carried forward straight from the 3D modelling phase.

It has shown from time to time that there can be information not updated in some stage that was applied in another. Ideal situation is that part or assembly can have sub data assigned on the 3D modelling stage and that is carried out automatically to the 2D drawing information and as well to the BOM-file. Lack of this kind of function adds additional risks for human mistakes, and all necessary information may not be carried to all the documents. Later, colleagues who encounter this project are put in a difficult position if they are not familiar as to which document holds the upmost correct information.

5.3 2D Drawings

Once the 3D models are in a steady state and project is getting ready for the protype ordering, it is time to create the 2D drawings out of the parts. 2D drawings are necessary for part manufacturing as these documents involve information that supports the part manufacturing processes. Drawings consist of following information which can variate depending on the part type:

- Measurable dimensions
- Tolerances
- Material information
- Surface finish and treatment
- Revision history
- Assembly information
- Welding guide

5.3.1 CAD-Software limitations on 2D Drawings

Each of the part and assembly drawings will have an information table on the right corner of the drawing. All the information is necessary as in the end, the 2D drawing is the go-to document to check information at the manufacturer and supplier end. As for now the information that is visible in the information table must be written by hand. There is not any linkage or automation known that would address the content in place. See below Figure 6. of the information table in the drawing templates.

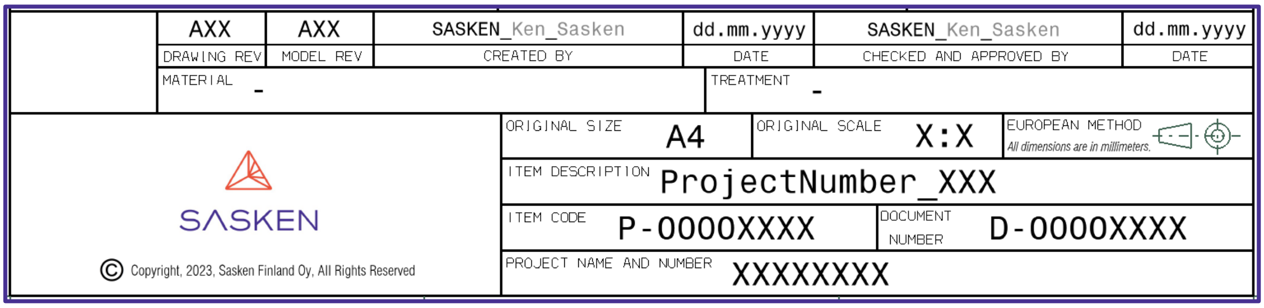


Figure 6 Sasken 2D Drawing Information Table

In assembly drawings there will be an additional table on top of the information table that withholds the parts which are used in that very specific assembly. Information for that table is also added manually. Balloons and corresponding numbering of that part are added manually as well. Automatic part list information and balloon recognition to a part will benefit the design process drastically, especially in the large assemblies.

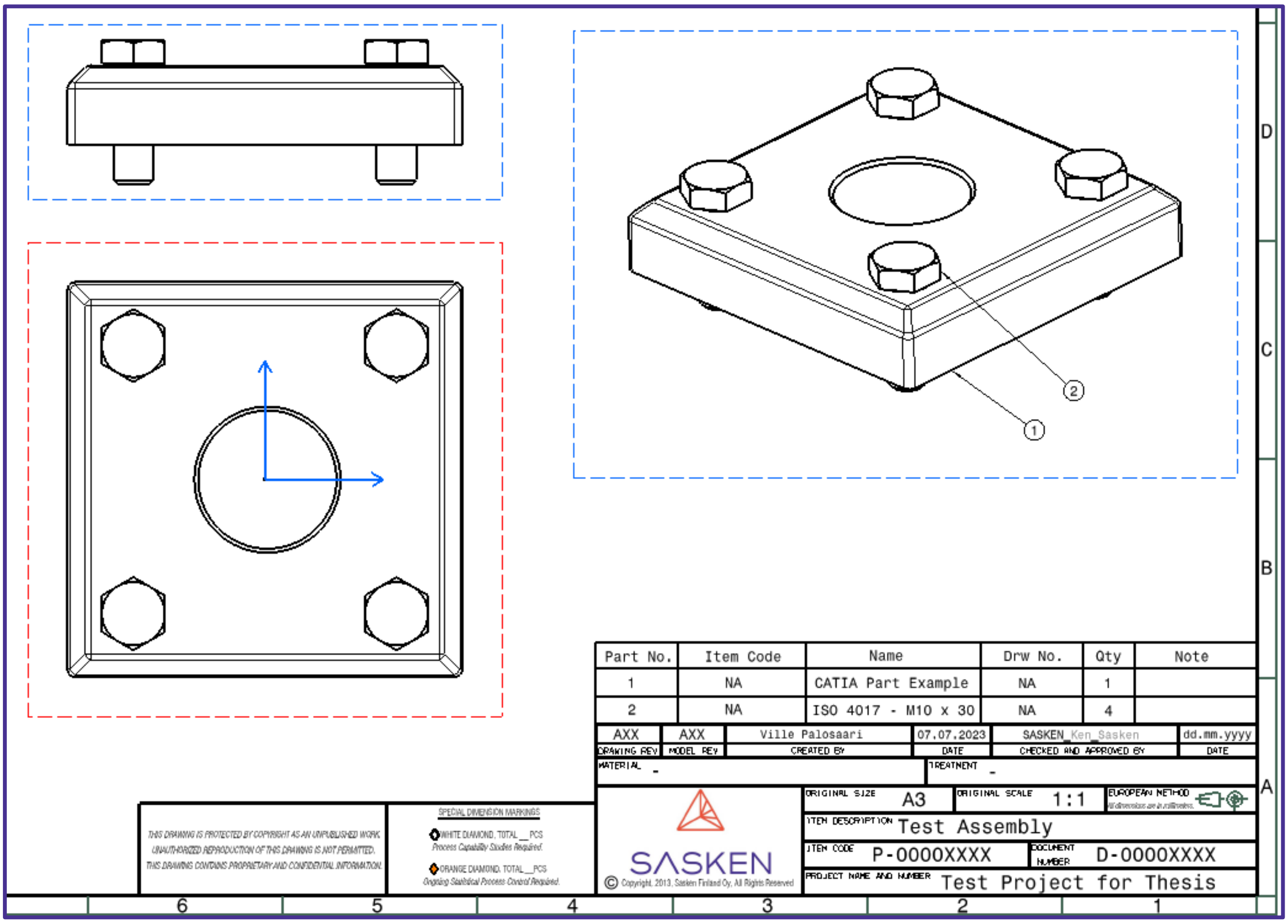


Figure 7 Assembly Drawing Part List Information

5.4 Bill of Materials Handling

Bill of materials of the product is required to maintain and control the product part and assembly information efficiently in one place. Bill of material in Sasken mechanical process includes the following information:

- Part description
    - Part must have a described name of the part function or purpose.
- Part supplier / manufacturer
- Part code and revision
- Part drawing code and revision
- Category information:
    - Category of the part type
        - Injection molded plastic part.
        - Laser cut sheet metal.
        - Purchased standard part.
- Material information
- Colour information
- Treatment information
    - Information if the part is i.e., powder coated or anodized.
- Dimension information
- Project information
    - Project name and number to identify end location.

5.4.1 CAD-Software limitation on Bill of Materials

Bill of materials holds the exact same information that is available in the 2D drawings of the parts and assemblies. The information from the drawings is handed over to the bill of materials excel-file manually. This method produces an extreme amount of manual work and adds another step for human error. There is a method to produce BOM file out of the CATIA V5, but it can't hold the information that is necessary for maintaining the product structure. The ideal situation would be to have parts and assemblies include all the necessary information which can be made visible in the exporting of the BOM excel-file.

5.5 Build & Assembly Instructions

Builds are milestone stages on the product development where there will be usually mechanical enclosure of the product ready with functioning PCBs, connectors, and user-interfaces. Builds in mechanical point-of-view differentiate in following manners regarding the complexity and specifications of the product as well as timeline of the project:

- **Build 1:**
    - **First build can be done with SLA parts**.
        - Assembly process verification.
            - Assembly process documentation.
                - First assembly instructions are created.
                    - Part and assembly information gathered to the document through the BOM.
                    - Every assembly step is documented with images.
        - Visual quality verification.
        - Minor in-house mechanical testing.
        - Molded parts are to be available in B2 and further on.
    - **First build can be done with molded parts.**
        - Assembly process verified with in-house FDMA prototype printing method and/or SLA models before tool ordering.
        - Assembly with tooling parts.
            - Mass production quality parts with tools.
            - Assembly process documentation.
                - First assembly instructions are created.
                    - Part and assembly information gathered to the document through the BOM.
                    - Every assembly step is documented with images.
            - Torque specifications.
        - Thorough in-house mechanical testing, documentation, and verification.
        - Accredited testing house perform the mechanical tests.

- Based on the first build findings, the modifications to the mechanical parts are to be corrected for the next build.
- Depending on the project timeline, the required product quality can be achieved in Build 1 and then actual mechanical design phases are complete.

- **Build 2:**
  - Parts are produced with modified tools with data based on the first build outcome.
    - Assembly instructions are updated if there have occurred changes in the assembly process i.e., either visually in the parts or new parts applied to the assembly.
    - Thorough in-house mechanical testing, documentation, and verification.
    - Accredited testing house perform mechanical tests.
  - Depending on the project timeline the required product quality can be achieved in Build 2 or Build 3, then actual mechanical design phases are complete.

CAD-software itself doesn't limit the build or assembly instruction phases. The information that is gathered manually from the 2D-drawing to the BOM-file is further transferred to the assembly instructions. This is another manual step, and when there are changes in 2D-information this also reflects to the rest of the document. It creates a wave of manual work.

5.6 Mechanical Testing

Mechanical testing of the products assembled during the builds are done according to the specifications assigned by customer. Sasken performs most of the mechanical tests in-house, but external accredited test house partners are also utilized to perform tests. Below tests are often used to verify product quality:

- Sasken In-House Testing:
  - Drop-Test
    - Guided Drop Test
    - Random Free Fall Test
  - Damp Heat Test
  - Thermal Shock Test
  - Cycle Test

        - Moving hinged parts
        - Connector tests
    - Rod-Scratch Test
    - Sharp-Edge Test
    - IK-Test
    - Bend and Twist Test
    - Chemical Resistance Test

- External Test House Tests:
    - IPXX Test
    - Salt Mist Test
    - Solar Radiation Test
    - Vibration Test
    - Wind Tunnel Test

After completion of the tests required for the build, the product can be finished, or the mechanical design process flow continues to another lap for the next build and testing of the product.

6 AUTODESK INVENTOR IN SASKEN

Autodesk Inventor has been used in the company from October 2022 onwards. Overtime, the software has been modified considerably from out-of-the box Inventor to match the mechanical team's requirements. There were certain aspects that were known to be modifiable during the trials and demo sessions, but the actual modifiability of the software was immense as the experience has grown with the software. There have been several projects where Inventor has been utilized from the get-go, and occasionally there appears situations that something in the design perspective can be enhanced with slight modification to the software. This topic will go through personal experience of the Inventor so far and show off the software from a Sasken point-of-view. These are the main issue topics compared to the CATIA which were described on the previous chapters.

6.1 User Interface

First time opening the software gave the impression that the user interface is extremely modern and user friendly. Icons are large and have text to clarify the intention of the tools. Toolbar changes, available tools as well as on the CATIA V5 depending on the modelling type (part, assembly, sheet metal etc.). Below Figure 8. visualizes the user interface of the Autodesk Inventor.

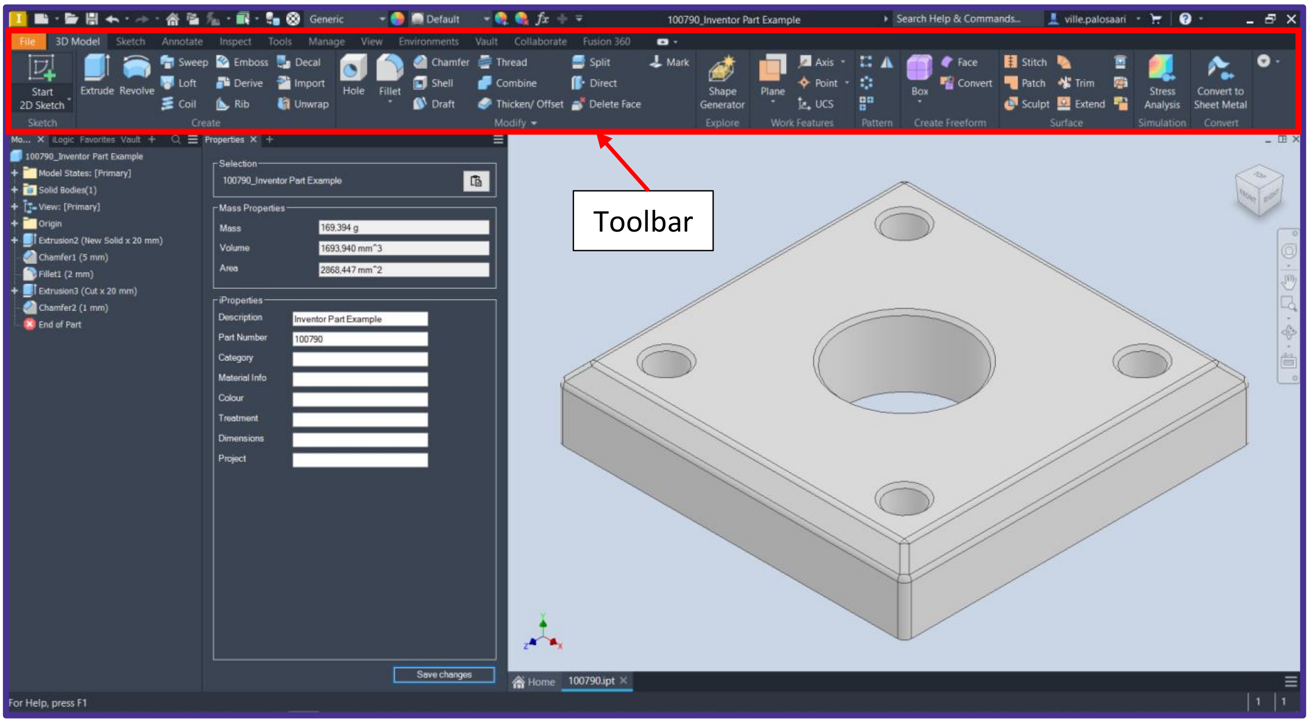


Figure 8 Autodesk Inventor Part Modelling User Interface

6.2 Modelling

Modelling with Inventor is intuitive and rewarding as the progression with the model creation is swift. User interface is easily operated, and the tools which hasn't been yet used are usually found quite fast. There is rarely the situation where the user feels lost as some modelling tool needs to be utilized but can't be found from the UI. There is an integrated search bar on the software which helps to find information with ease.

The sketching functions were surprisingly much more refined than on the CATIA. Powerful designing aid that was found on Inventor is a support of adding functions when assigning dimensions in the sketch tool. Dimension tool is suitable for calculating complex functions, and the function will remain to be seen inside the sketch, which makes it easy to return and check the values. Dimensions tool is also operatable with parametric functions, which adds value to the design process when there are cases of multiple similar designs with slight dimensional differences.

For plastic part designing, the Inventor has many tools built in. Example tools are a snap-fit designing tool, tool for boss and lip features. Snap fit tool is a quick way to design snap fits for plastic counter parts. The figure on the next page has examples of the outcomes of each of the plastic designing tools. At the left of the figure are shown two examples of snap designs that can be applied to the parts. Middle part of the figure visualizes screw towers produced by boss feature tool. Last part of the figure represents a lip that is designed to be opposite shaped on both parts which aligns them perfectly together. Previously these sorts of designs and shapes were done completely manually, but the Inventor has built-in tools that will produce these functions with much less effort. Inventor also has multiple analysis and simulation tools built-in, and one of those tools is a draft analysis which analyses the draft angles of the part.

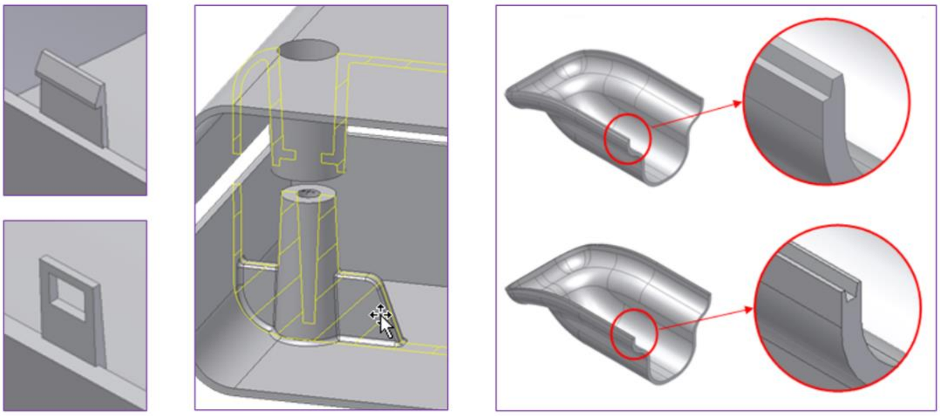


Figure 9 Inventor Plastic Part Design Tools

Surprisingly, working with assemblies is where most of the differences were encountered during the modelling compared to CATIA. Managing assemblies and constraining parts in CATIA was never thought of as an issue as there hasn't been anything to benchmark with. In Inventor there are dozens of ways to constrain parts and assemblies with even variable input methods. Working with projects that have moving parts in the product is now more efficient than ever, and capability to witness each part moving independently gives a major trust to the design in the early phase of the project. Comparing to CATIA there wasn't capability to handle these sorts of moving components inside the assembly.

6.3 Property Manager

There is a possibility to install plugins and add-ons to the Inventor software from the Autodesk App Store. Sasken utilizes external add-on for the information management of the parts and assemblies. Add-on shows up in a separate window to the UI that display information of the part or assembly that is selected. Information fields can be defined by the user on what sort of information is wanted. Figure 10 on the next page shows the UI of the add-on. (Autodesk 2023.)

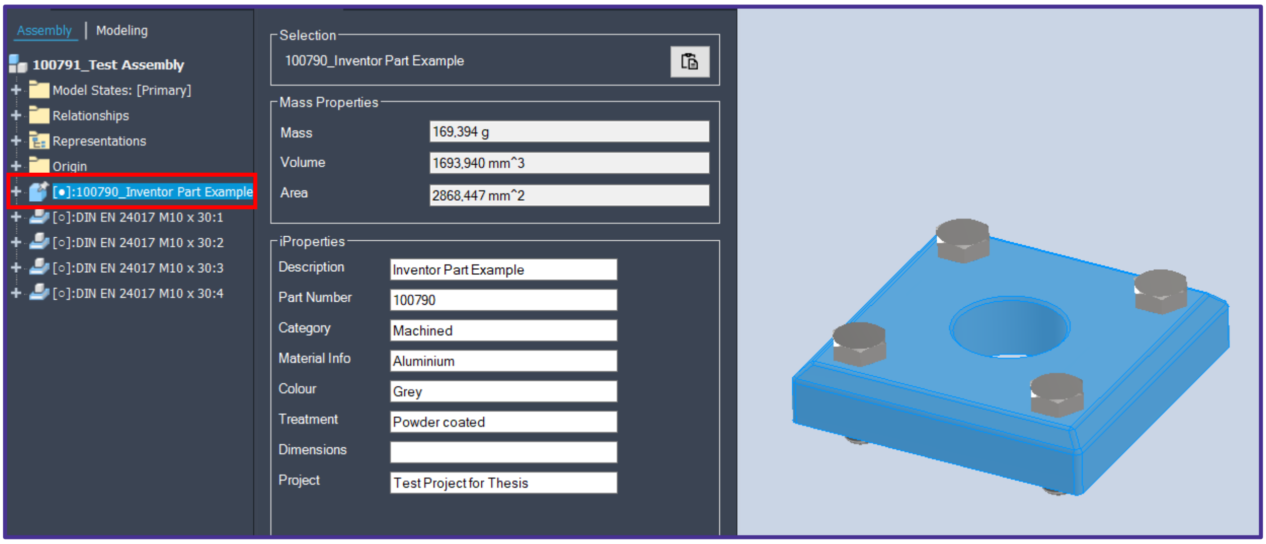


Figure 10 Property Manager Information

6.4 Bill of Materials

Inventor has a great built-in bill of materials tool. With this tool it is easy to check out all the information of the parts and assemblies in single page. Sasken has customized tool which exports the bill of materials to two types of excel files.

1. Part List Excel
   a. This file includes all the parts that used in the main assembly.
   b. This file doesn't include any assemblies, only independent parts.
2. Structed Part List Excel
   a. This file includes all the sub-assemblies that are used in the main assembly.
   b. Sub-assemblies include all the parts that are used in that assembly.

Exportation tool uses an Autodesk Inventor iLogic rule system. iLogic is a VB.net based coding language which is very similar to that of Excel macro coding. iLogic can be used to code parametrical models, find information from the Vault of the part or in Sasken's case to export a excel file. (Autodesk 2023.)

The exported excel file has integrated macros which automatically modifies the excel file to look visually as shown on the below figure. The same information that is also shown in the property manager of the part or assembly will also be exported with the excel file. The figure below shows the high-level part list view of the main assembly.

Project
Test Project for Thesis
Date
13.07.2023

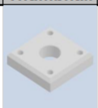
Item	Part Number	REV	Thumbnail	QTY	Description	Category	Material Info	Colour	Treatment	Dimensions	Additional Information
1	100790	NA		1	Inventor Part Example	Machined	Aluminium	Grey	Powder coated		
2	ISO 4017 - M10 x 30			4	Hex-Head Bolt						

Figure 11 Bill of Materials Excel

6.5 Cable Design

There have been multiple projects done with Inventor where there was a need for cables and routing to the be defined between PCBs. Inventor has a separate wire & harness tool which was used to efficiently define the correct cable size, material, and routing between components. To successfully create the cable between two connectors, there needs to be each corresponding pin designated on both connectors. This way the software understands from where to where the cable needs to be routed. Cable can have fine-tuning done by adjusting with additional points the route of the cable if necessary. Inventor automatically calculates the cable length if the connectors are moved to another position. Below figure represents the cable created between two connectors.

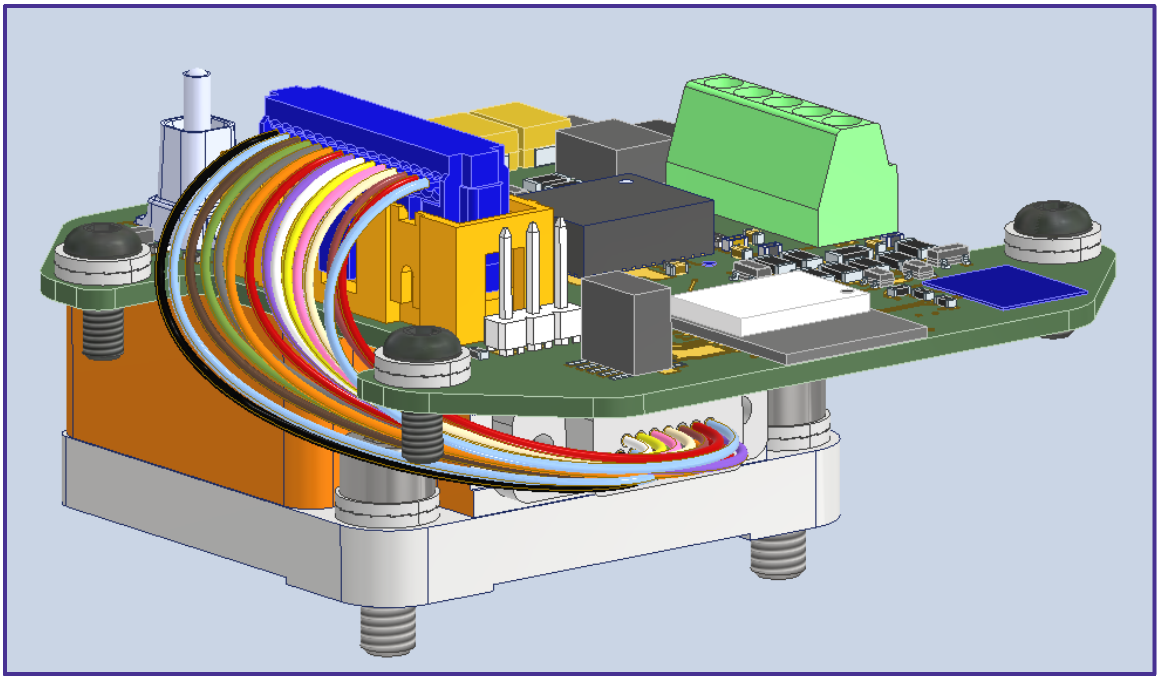


Figure 12 Cabling Created with Wire & Harness Tool

Cable is designed between two connectors and matching pins. The software has automatically routed the cable with the cable bending radius that the used cable has been specified with. The software understands that there is a PCB outline that must be avoided to prevent any collision. This wire & harness tool has turned out to be very efficient to use because the ease of operability and the automatic re-calculation of the cabling if the either of the connector is moved makes a lot of confidence in the minimization of the human errors.

When exporting the bill of materials excel files, the other one is the part list which represents the high-level components that are used in the main assembly. The more in-depth bill of materials includes structured view under the main assembly. The figure below is a structured view snapshot of the cable which was presented on the previous page. The cable is the number nine component of the main assembly. Cable assembly includes 15 different cables. All the cables have the exact length informed with matching cable type information and colouring.

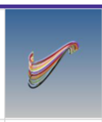
9	100655			Purchased	1	CABLE_ASSEMBLY
9.1	.Wire1		(NULL)	Purchased	63,443 mm	26AWG-BLK
9.2	.Wire2		(NULL)	Purchased	63,174 mm	26AWG-BLU
9.3	.Wire3		(NULL)	Purchased	60,959 mm	26AWG-BRO
9.4	.Wire4		(NULL)	Purchased	60,475 mm	26AWG-GRN
9.5	.Wire5		(NULL)	Purchased	60,627 mm	26AWG-GRY
9.6	.Wire6		(NULL)	Purchased	61,732 mm	26AWG-ORG
9.7	.Wire7		(NULL)	Purchased	60,847 mm	26AWG-RED
9.8	.Wire8		(NULL)	Purchased	60,967 mm	26AWG-VIO
9.9	.Wire9		(NULL)	Purchased	51,132 mm	26AWG-WHT
9.10	.Wire10		(NULL)	Purchased	52,009 mm	26AWG-YEL
9.11	.Wire11		(NULL)	Purchased	52,937 mm	26AWG-PNK
9.12	.Wire12		(NULL)	Purchased	53,945 mm	26AWG-BEI
9.13	.Wire13		(NULL)	Purchased	52,623 mm	26AWG-BRN
9.14	.Wire14		(NULL)	Purchased	53,846 mm	26AWG-RED
9.15	.Wire15		(NULL)	Purchased	60,071 mm	26AWG-BLU

Figure 13 Cable Bill of Materials Structured View

6.6 Artwork Design

On CATIA there was additional steps to create writing on the part. This operation required a 2D drawing which will then have the text written. This text is copied to the sketch on the part design. In Inventor there is a tool for adding text to the sketch. The figure below has the tool shown on the sketching section and text written used by the tool. (Tickoo 2021, 4-90.)

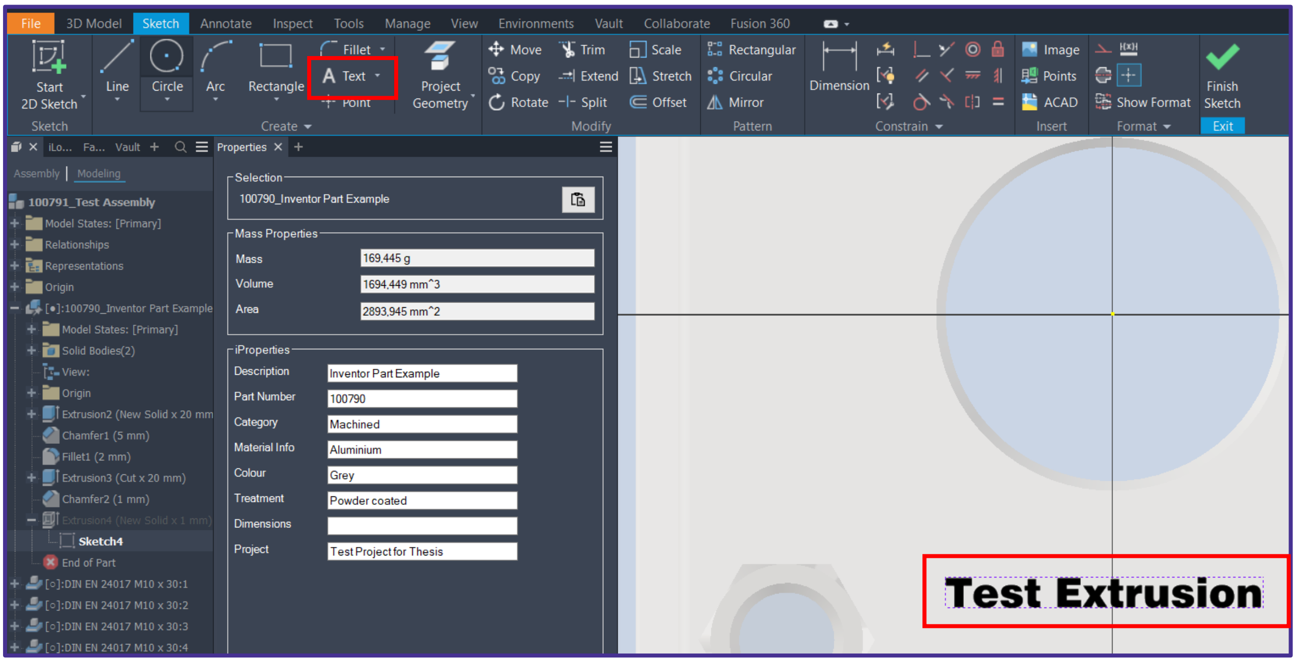


Figure 14 Text Tool

Next figure represents the same text extruded to the model which was written in the sketch tool. This easily operatable tool will increase productivity as the text, sizing and constraining can be done within the same operation. If the text needs any modification that can be easily done in the sketch. Rather than needing to generate new text in drawing and the copy, paste and re-constrain the text to the model.

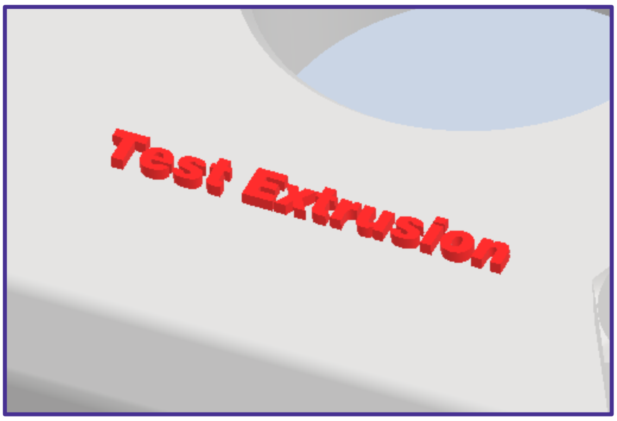


Figure 15 Extruded Text

6.7 2D-Drawing

2D-drawing creation with Inventor has been found to be exceptionally efficient. Especially with the automation aspect of the fill-out the information table and as well the part list information on the assembly drawings. The information that is put on to the part or assembly is automatically carried out to the 2D drawing information table. Figure below is from the 2D drawing that was created out of the example part which was shown on the previous chapter with property manager information. Information automatically fills the corresponding field in the 2D drawing. Information updates if there are changes done in the property manager.

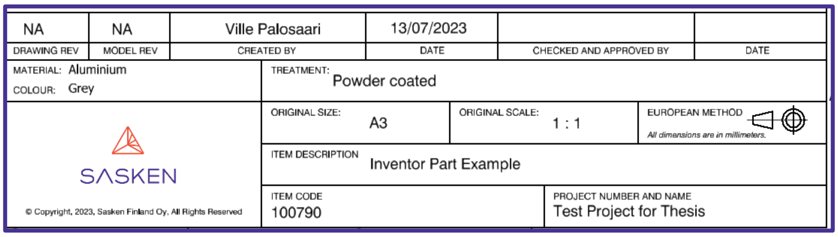


Figure 16 2D-Drawing Information Table from the Part

On assembly drawing perspective the automation methods apply to the part list as well. The information that is put to the property manager appears on the part list; the items are numbered in the list with quantity information. Items can be ballooned to the drawing and the balloon automatically identifies the item number from the list. Below figure 17. shows the part list for the assembly drawing and the ballooned item numbers as well.

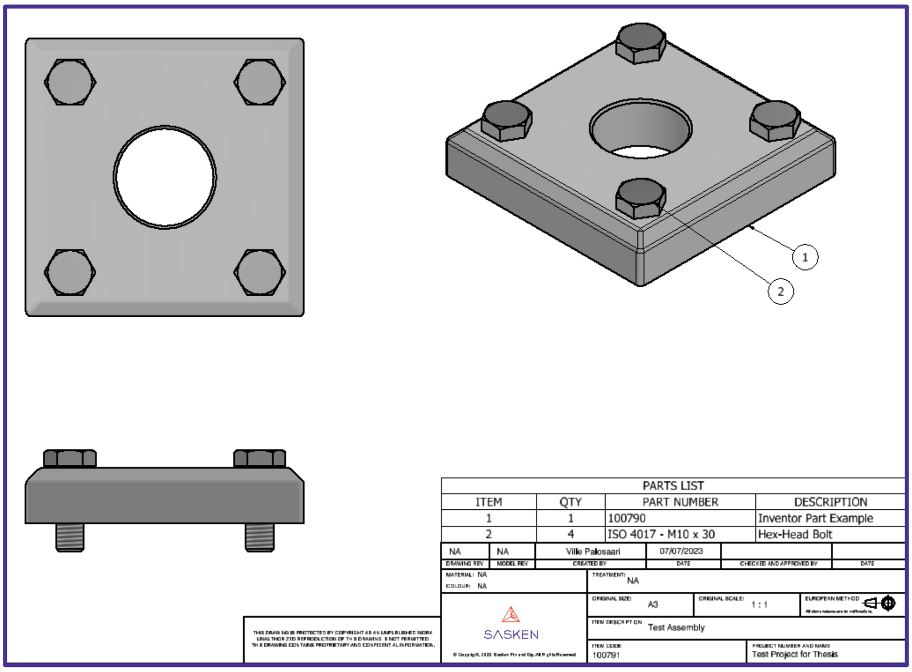


Figure 17 Inventor Assembly 2D-Drawing

With CATIA there was a 2D-drawing template for each paper size used from A0 to A4. Each of the sizes had multiple different templates which variated from information that was inside the template, i.e., different tolerance tables for different materials. This means that plastic parts have a separate template and so has aluminium parts. There were cases when some part was started to be worked on with A3 sized paper, and the paper size didn't have enough space for all the information. On CATIA this paper size can't be adjusted if there isn't enough room, if it can be adjusted, the information tables may not align anymore with the edges of the drawing. The same operation must be started over with the larger paper template to have everything in correct place.

On Inventor on the other hand there is a single template which has options to import different tolerance tables which have been assigned and designed by Sasken's mechanical team to that template. Paper size can be changed during the drawing creation and all the information tables are matched to the correct position and move along with the paper size.

6.8 Sheet Metal Design

There were multiple benefits of the Inventor already known from the demo beforehand and with the team having personal experience on the software. However, during the thesis creation was found out that the sheet metal design portion of the Inventor is a major step forward compared to the CATIA V5. CATIA V5 in sheet metal design partition is a very manually operated process and the user must be sure that all the applied parameters are correct. Inventor however aids the design process by having standardized parameters applied to the software and the user can manually insert or modify everything.

Example improvement on the sheet metal design is the flanged side usability. During sheet metal part designing, there is commonly a flange on the sides on the parts. The flange requires a relief form in the corner to ease the bending process. Inventor gives an option of multiple different shaped relief forms to choose from. Previously when using CATIA these relief forms have been designed manually and cut through the part in a flat pattern view. Below is an image of the multiple different relief forms to a corner of the sheet metal part. (Tickoo 2021, 14-8.)

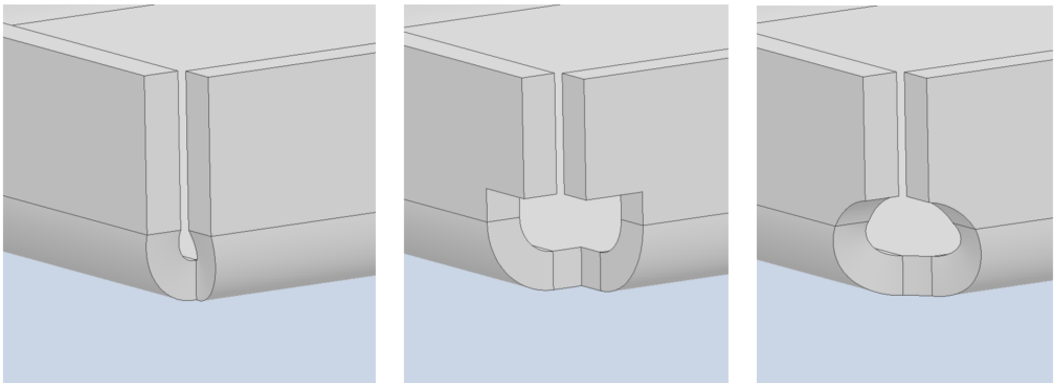


Figure 18 Sheet Metal Design Corner Relief

7 TEAM INQUIRY

To achieve the understanding of new CAD software implementation successfulness, the mechanical team was inquired regarding the software change-up. The inquiry consisted of eight questions which were based on experience so far of the new CAD software in comparison to the previous one. These questions had different types of options to choose an answer from. Nineth question was an improvement suggestion on which the attendee could answer verbally on where they find room for improvement on CAD-design processes.

Inquiry was done by using the Microsoft Forms-application. Inquiry was laid out for seven mechanical designer and every person were able to answer the inquiry. Following chapters goes through the questions that was presented during the inquiry. Each of the question are in a separate heading and the results are analysed as well.

7.1.1 How was the Inventor & Vault Installation process?

First question was about the installation process of the Autodesk Inventor & Vault, how was the Inventor & Vault Installation process? (Was it straight forward, difficult or time consuming?). Answer had three options to answer from: Good, Neutral and Poor. The team had exclusively a good experience of the overall installation process. Installation process has been put up to be straightforward and requires from operator to assign installation location, server, and Autodesk account information.

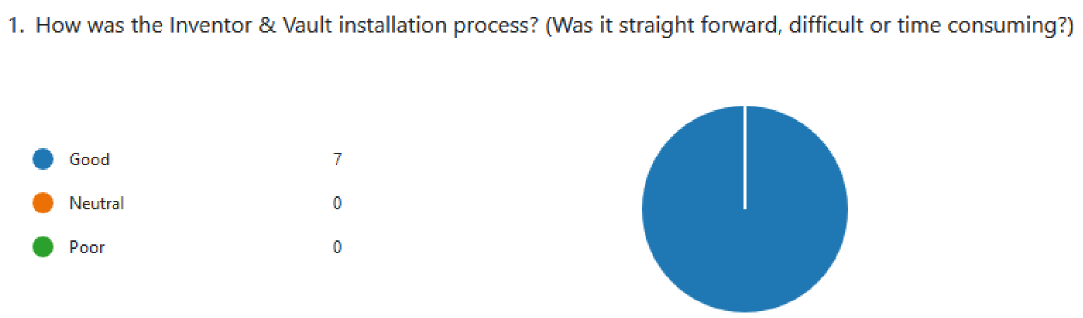


Figure 19 Installation Process– Distribution of Answers

7.1.2 What was your first impression of the UI with Inventor & Vault?

Second question was regarding the first impression of the Inventor & Vault user interface. Question had three options to choose answer from: Intuitive and easy to learn, Neutral, Unintuitive and difficult to learn. The team's answers were distributed between intuitive and neutral options. It seems that the user interface is mainly intuitive, but there might be experience differences between the Inventor CAD-software and Vault-database software. This question didn't separate experience of those software's but were combined.

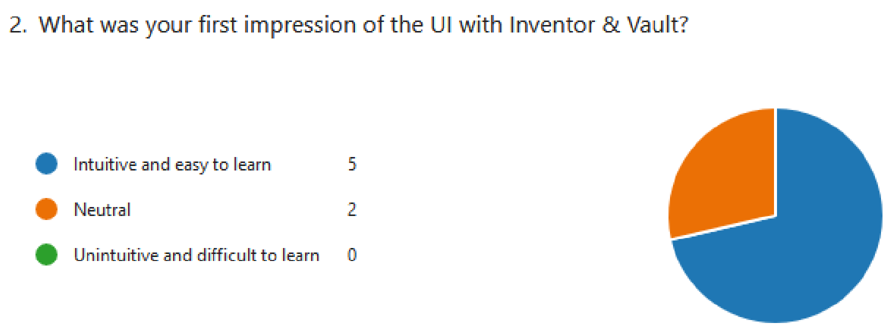


Figure 20 UI First Impressions - Distribution of Answers

7.1.3 How do you find the 3D-modelling compared to previous CAD-software?

Next question is based for the actual 3D-modelling with the software. 3D-modelling includes basically the sketching where the main profiles are drawn, extrusion of the part and overall 3D modelling functionalities. The team had basically found the new software to be better than the previous one.

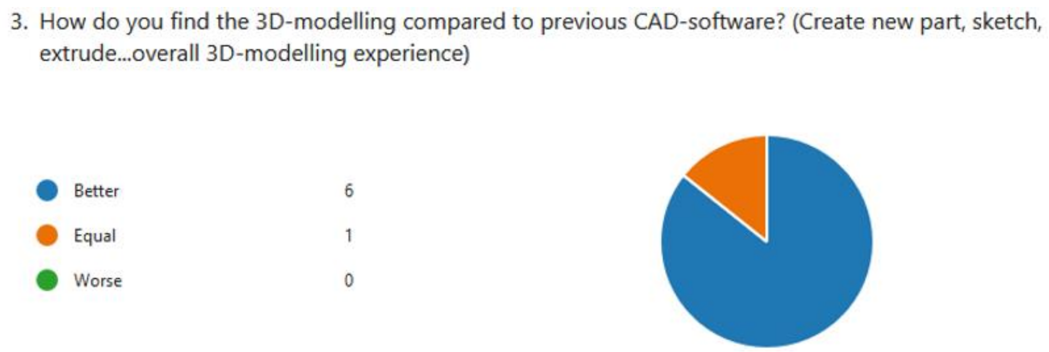


Figure 21 3D-Modelling Experience – Distribution of Answers

7.1.4 How do you feel about the information input possibilities compared to the previous CAD-software?

Fourth question was to gather experience of information input possibilities compared to CATIA V5. There is a separate plugin installed to the software which works as a separate tab and has the possibilities to insert information for the part of the assembly. Distribution was between five answers of having better option for information input and two for having equal input option as on previous CAD-software.

One of the main concerns previously had been information transfer from software all the way to the BOM. This new method was hoped to be significantly better option and to make the data input easier and more efficient. This aspect is something that needs to be further investigated, have team input and ideas put into action.

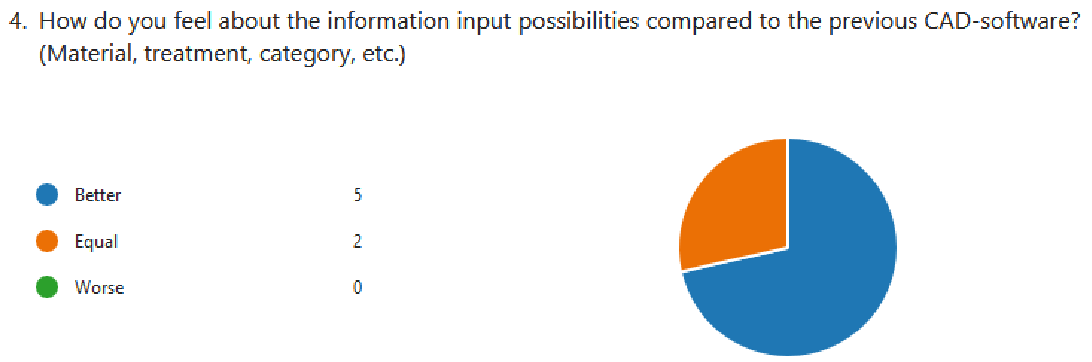


Figure 22 Information Input Experience – Answer Distribution

7.1.5 How does the assembly handling and part constraining compare to the previous CAD-software?

Question was laid out for the team to define how Inventor compares to the CATIA V5 regarding the perspective of the assembly handling processes, constraining of the parts and object. The team solely answered that the Inventor is better in that case. Inventor does have high variety of different kind of constraining options and especially on the assemblies which must have objects that are moving, for example, along the sliding guides have proven to be effective to manage the movement. Next page has the distribution of the answer figure.

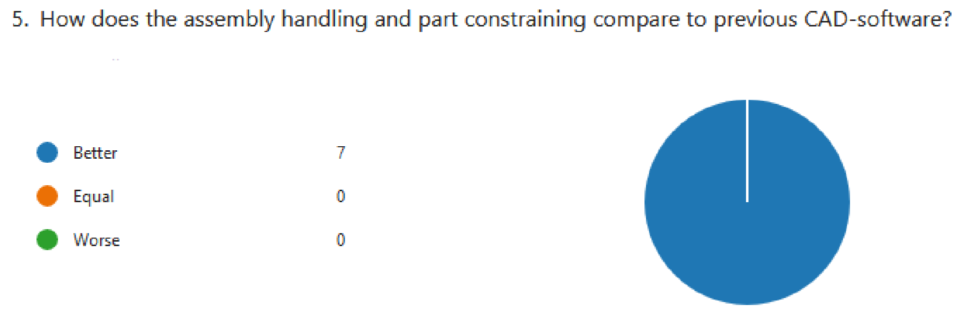


Figure 23 Assembly and Constraining Experience – Distribution of Answers

7.1.6 How do you find the 2D-drawing creation to the previous CAD-software?

Sixth question is to validate how the 2D-drawing creation compares to the CATIA. 2D-drawing section holds aspects like adding dimensions, tolerance table handling and overall information input to the drawing template. Team answers were distributed for five as better and two for being equal to CATIA.

This is another aspect that needs to be investigated further with the team. There is a lot of automation added to the Inventor part and assembly creation which is then mirrored to the 2D drawing information table. Overall result is positive, but 2D drawing needs to be more refined to have a better suitability to the whole team.

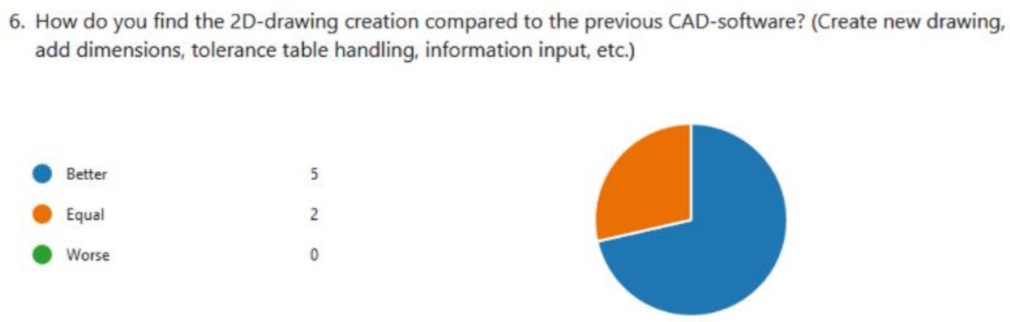


Figure 24 2D Drawing Experience – Distribution of Answers

7.1.7 How do you feel about Bill of Materials handling compared to previous methods?

Bill of Materials handling was one of the most critical aspects of CAD-software changing. Manual information input to the excel template has been extremely time consuming and the possibility for human error is high. Automated BOM excel printout through the CAD-software was a necessity that Sasken mechanical team seek out for.

Based on the team responses, the new Bill of Materials handling process is significantly better option than the manual input to the excel template. The result is a delightful to see that the team has taken the new process into practice and have found it to be better and a more useful option.

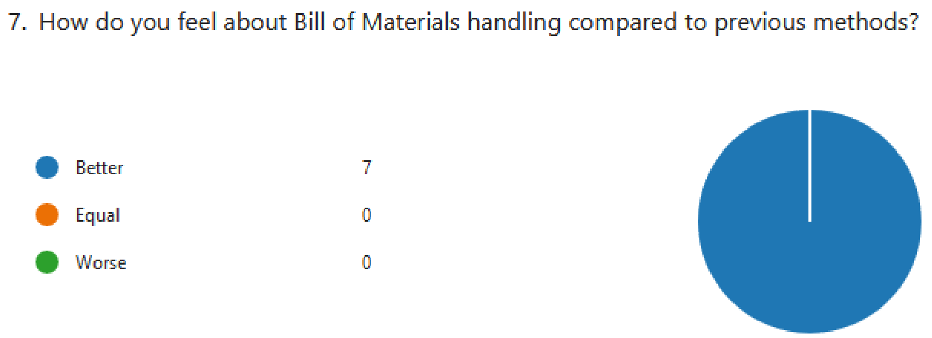


Figure 25 Bill of Materials Experience- Distribution of Answers

7.1.8 Do you feel the mechanical design process has improved with the CAD software change that has been made?

The last question with the options was to gather information as to how the mechanical team found there had been any improvements of the mechanical design process via the change of the CAD-software. This question is an overall summary was that has the experience been positive or has the change not being effective. The team answered that the change has been positive, and the overall design process is improved when using the Autodesk Inventor.

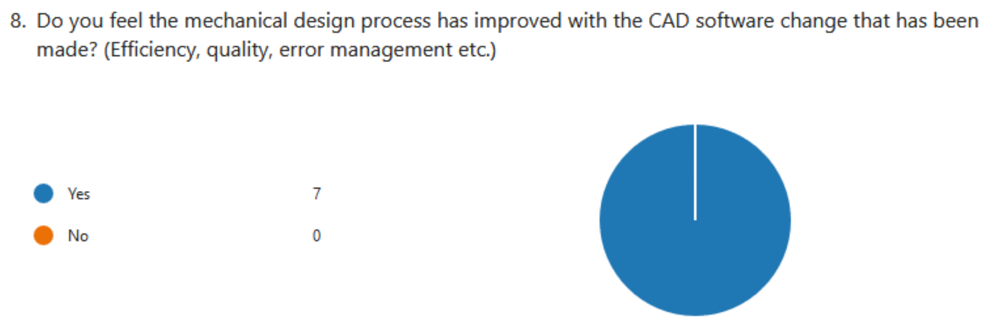


Figure 26 CAD-Software Change on Mechanical Design Process – Distribution of Answers

7.1.9 Do you have any improvement proposals regarding the efficiency of CAD processes in Sasken?

The very last question of the inquiry was not mandatory to answer. There was a free field to give comment on what would be the processes that could be refined and improved on the CAD design perspective. Three persons from the team gave comments in this section.

Answers were:

1. Clear unambiguous ways of working.
2. Minor improvements to the BOM excel template.
3. Create design process documentation to unify practices with all designers.

First and third answer falls to the same category of having a uniformed design processes which would be documented, and all the designers would be following those basic rules. This will be the next step after the thesis. There are certain objects that need to be captured and verified on the Inventor and Vault before making the actual document. During this thesis process there was dozens of screenshots captured throughout the installation and ramp up of the software. Screenshots and information will be utilized to the design process documentation.

Second answer was the improvement request to the BOM excel template. For this thesis there was the very first iteration presented and found to be the baseline. During this thesis process there has been multiple versions created of the BOM. Every now and then there are new ideas coming up which would be useful to be added to the BOM. Additional information columns are easy to add and new version of BOM template is quickly put to the process.

8 CONCLUSION

The thesis was interesting to work on, as these very tasks that were presented in the thesis are part of my daily work. While reading the thesis, seeing the baseline of the previous CAD-software, and witnessing the improvement what the tasks have gone through, is pleasant information. By personal experience when the new CAD-software was introduced, the actual mechanical design has been improved tremendously, especially the data and information gathering and transfer to the Bill of Materials file. There are still room for improvement in every task, but having this software, which is capable of automatization and has support for wide variety add-ons, which can be utilized to aid the design process. Sasken mechanical team is committed to report and study if there are any items or issues occurring during CAD-software usage which could be improved.

The writing process itself turned out to be the more challenging than anticipated beforehand. As the subject was very Sasken based for information from the mechanical design perspective tasks. It was difficult to contribute to the task study with a certain theoretical point-of-view, but the PDAC method turned to be quite suitable for this case. The thesis subject was chosen as it was known that there is room for improvement in the mechanical design process, both from my personal experience and the rest of team. In the end, the study that was utilized for the thesis benefits the company, as those were actual subjects that were lacking and are now improved based on the findings by the inquiry. There are still areas to be improved with the mechanical design process, as based on the inquiry results. These are the areas that will be taken to further investigation with the team and how those items could be improved.

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