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Definition of CAD standard that improves and enhances data transfer to CAE

Master’s thesis in Product Development

ALICIA ALINGSJÖ

Department of Product and Production Development
CHALMERS UNIVERSITY OF TECHNOLOGY
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Department of Product and Production Development

Chalmers University of Technology

Division of Product Development

SE-412 96 Gothenburg

Sweden

Telephone + 46 (0)31-772 1000

Cover: CEVT.

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ABSTRACT

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ALICIA ALINGSJÖ
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The purpose of this project has been to investigate how to simplify the transfer from the CAD to the CAE department. Depending on how the work is conducted in CAD, the process in FE can be problematical, especially for plastic parts. The research and studies that has been conducted has suggested that this can be avoided by following the CAD standard for the design of plastic parts, which has been proposed in this thesis. This has been accomplished by studying the A-pillar trim, which is a typical injection moulded part in the automotive industry. The conclusion of why the transfer is problematic depends upon multiple causes and conditions. These reasons are rooted within the geometry itself, and the modelling techniques that have been used during CAD modelling.

The project has followed the methodology of Product Design and Development (Ulrich & Eppinger, 2009). The work has been conducted at CEVT and its aim has been to study the development process between the CAD and CAE departments. The current CAD guidelines at CEVT have been benchmarked together with other companies. Several interviews have been conducted with a total of 14 participants from CEVT, Volvo Cars, ANSA and ÅF.

The modelling technique that has been suggested in the CAD standard proposal is following the practice of using non sub element and creating a logical work flow. The reason for this is to make it easy to modify and replace an element within the model. Also to make the process easier for next person who will work with the CAD model. Surface modelling is most suited for creating the A-surface from the styling surface. The A-surface should be analysed before starting the modelling work so it do not contain holes, overlaps or radiuses that cannot be offset, since the quality is translated to the solid body. The rest of the part should be constructed in solid modelling. These bodies should thereafter be combined with Boolean operations such as add and remove. Very small surface patches, due to dress up features or unintentional design, should be avoided or be removed before reaching FE modelling.

Key words: CAD, CAE, FE, PDM, mid surface mesh, plastic, automotive, product development

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Gothenburg, February 2015

TERMINOLOGY

2D	Two dimensional
3D	Three dimensional
ANSA	ANSA Pre-processor
BIW	Body In White
B-REPS	Boundary Representation
CAD	Computer Aided Design
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing
CAS	Computer Aided Styling
CATIA	Computer Aided Three-dimensional Interactive Application v5
CEVT	China Euro Vehicle Technology AB
CSG	Constructive Solid Geometry
FE	Finite Element
FMH	Free Motion Head
Geely	Geely Automobile
GM	General Motors Company
Iges	The Initial Graphics Exchange Specification
ISO	International Organization for Standardization
PDM	Product Data Management
PID	Property Identification
Saab	Saab Automobile AB
SI	The International System of Units
Step	Standard for the Exchange of Product Data
Volvo Cars	Volvo Cars Corporation

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1 INTRODUCTION

The following section is a description of what is required to be achieved during the master project “Definition of CAD standard that improves and enhances data transfer to CAE”. A background to the problem is given. The aim and purpose are presented together with the scope and limitation of the project. Three research questions are defined to cover three problem areas that will be solved in the end of the project. The method that has been used during the project is also introduced.

1.1 Background

CEVT was founded in 2013 and is a subsidiary company to Volvo Cars and is owned by Geely Holding Group, see figure 1 (CEVT, 2014). CEVT is developing the new modular architecture for Volvo Cars and Geely Auto, such as for the C-segment cars. CEVT is also introducing top hat development, shared component development and complete vehicle design.

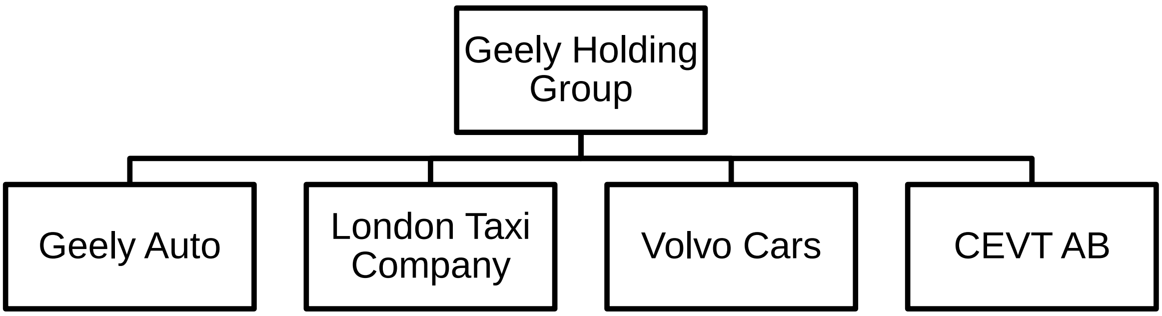


Figure 1: Company structure

As a rapidly developing and fast-growing company CEVT is in need of consistently defined CAD standards and methods for how to build CAD models for a variety of different articles. The CAD standard for thin sheet metal is reasonably stable and well defined, but other types of articles such as injection moulded plastics, extruded, forged, etc., are less well defined. This results in CAD models being built in different manners depending on the designer. The stability and result of this data when transferred to CAE is often dependant on the way the CAD model was constructed.

1.2 Aim and Purpose

The purpose of the project is to develop a CAD standard to make the process from CAD to CAE departments more efficient. Today there are different ways to build up the CAD model, which may complicate the work in CAE depending on the way it is done.

An understanding of the features and functions of the CAD software CATIA and the CAE pre-processor ANSA is expected after the project. Benchmarking of various modular designs will be achieved. The project will validate the proposed designs and a documentation of CAD standards will be set.

The objectives of the project are to elicit the needs, opportunities and limitations. The working method within CAD and CAE departments at CEVT will be explored to identify the needs and which problems that exist today. The project will result in a CAD standard proposal that is to be used by CEVT in the future. The solution will primarily be established for the design department. The methodology will be simple, and easy, to ensure that it will be utilised. The aim with the CAD standard is that the quality of the models should be consistent for all designers, which will make the process for the CAE department easier. This will also provide a faster response from CAE department to CAD. In the case that the outcome results in several solutions, the one which is best suited to the needs of FE modelling will be selected. The purpose of all this is to make the process more efficient and suitable to the rapid working process at CEVT.

1.3 Scope and Limitation

The interior plastic A-pillar trim will be studied during this project. Since the interior design often changes for every new car, there is a need to get a CAD standard of this particularly. It is also a typical automotive plastic part, and the work can therefore be implemented to other injection moulded plastic parts. The scope is to build up a CATIA V5 CAD model of a plastic part. The same CAD model should be made by utilising several different modelling techniques. These models will be transferred to CAE and FE meshes will be generated. Evaluation will be made based on the effort to mesh the article and will form a base for written documentation about the best way to construct the CAD model.

In addition to above, different FE meshing strategies should be evaluated. The CAE is limited to only cover FE modelling. The FE Models should be prepared for FMH, crash, and solidity. In general, mesh generation of a plastic part is created on the mid surface, and this will be covered in this project. Limitations in the project are to only work with CATIA and ANSA.

The work will only be related to the CAD and CAE departments at CEVT. The aim is not to include economic and manufacturing considerations. The solution will not follow the methodology of knowledge-based modelling.

1.4 Research questions

The project will result in answering three research questions, which can be seen in table 1.

Table 1: Research questions

Research question	Definition
RQ1	How does the CAD model influence the FE model?
RQ2	What are the requirements that need to be fulfilled?
RQ3	What common construction principle can be utilised for different features?

1.5 Method

The project will be divided into three major interacting stages; data collection, analysis, and development and validation, see figure 2. In addition, a fourth stage is added, to cover when the project will come to a close and the final results will be unveiled and concluded.

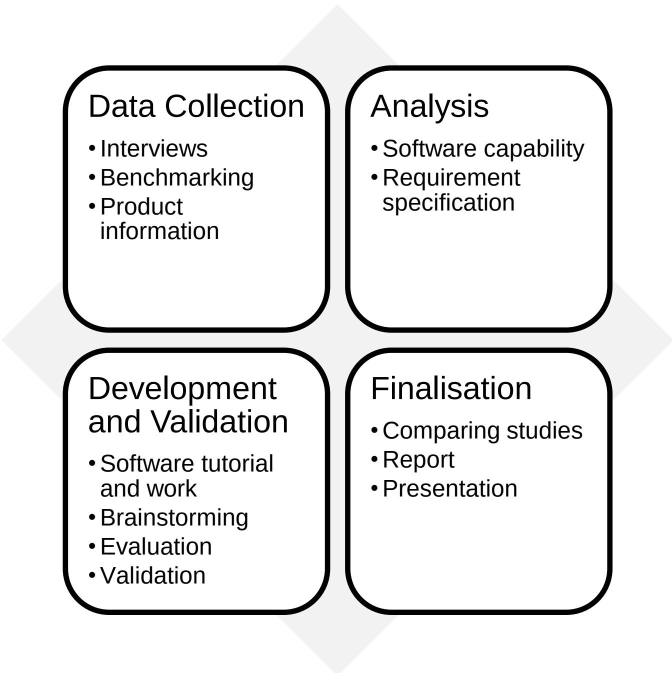


Figure 2: Method

The development process will follow the methodology described in *Product Design and Development* (Ulrich & Eppinger, 2009). The iterative five-step concept generation method structures the complex problem into sub problems to be able to select a final solution, see figure 3.

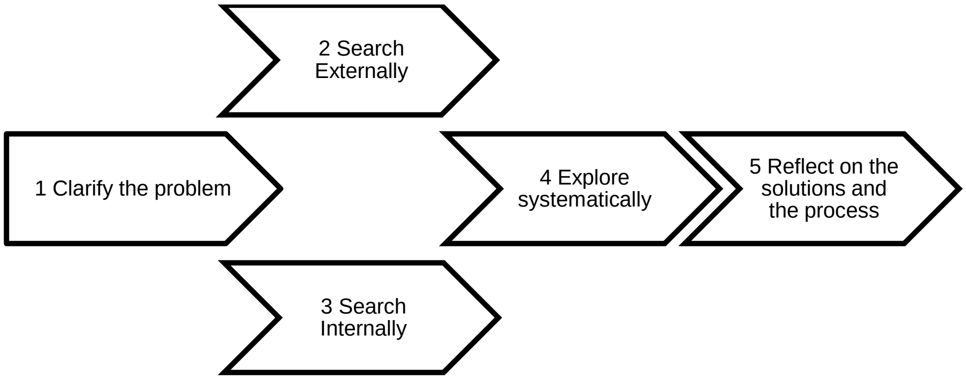


Figure 3: Five-Step concept generation method

The *first* step in the concept generation method is about understanding the problem and to focus on the critical sub problem by identifying customers' needs and product specification. A function diagram or the so called "black box" can be used to decompose the complex problem.

The *second* step is an external search where data is collected in three ways. The first one is through research studies. This will include searching published literature and benchmark related products, which will be relevant and current to today's technology. This will be done by starting the study from general research to more specific. The data will be collected from articles, literatures, companies' web pages and different data bases. Information will also be collected from previous and present courses at Chalmers University of Technology and software documentation. In addition, other companies' processes will be benchmarked. The second one is qualitative interviews; performing semi-structured interviews with lead users and consulting experts to get the knowledge about the current work processes, areas that can be improved and answers to the research questions. The interview guide can be seen in appendix A. The third one is observation; which is an unstructured study of the current process. Knowledge from the company will be elicited, such as working experience, and already working design process.

The *third* step is to search internally, which is usually defined as brainstorming. It entails of discovering ideas, which will be created of the gathered knowledge. The internal research can be divided into different sub problems, such as analogies; how are related problems currently being solved.

The *fourth* step is to explore systematically which is mainly to organise the solutions and categorise them. A classification tree, a concept combination table or solution matrix can be used to divide the solution into which problem they solve and eliminate the least promising one and refine the solutions. Analysing the data cluster will be accomplished and a conclusion of CAD and FE users' problems, needs and requirements will be established. During the whole project this specification will be updated with deeper knowledge.

The last and *fifth* step of the concept generation method is to reflect on the solution and the process. The output can be used for the next process iteration or for future project.

1.6 Thesis outline

Section two in this thesis presents the basic theory relevant to the project, by covering the theory of design and analysis. Additionally, a presentation of former research will be introduced. Data will be gained from both the internal and external research.

Section three gives an overview of the process that has been used and presents the benchmark analysis. The main part of the external search is presented together with knowledge gained from the course CAD Advance at Volvo Cars (Volvo Cars Corporation, 2014)

In section four the result of different studies are presented. The studies has together with the knowledge gained from the internal and external search resulted in three case studies, which defines three different ways to construct the CAD model. The most promising case study, dependant on both the modelling procedure in CAD and the result in FE modelling, is presented as the final CAD standard proposal.

Furthermore, section five will concentrate on considering how these results can be used and identifying any possible complications. A critical observation will be conducted in order to analyse the effectiveness and accuracy of the proposal offered.

In section six, a proposal of future work within this subject is given.

Finally, in section seven, a conclusion of the project is presented.

2 FRAME OF REFERENCES

The following section will cover aspects that are necessary for the project. It will give a brief introduction of the product that will be studied and the purpose it has. It will give a description of the production method and how it is applied at CEVT. Later it will describe CAD and CAE, together with the basic in PDM.

2.1 Product development process in automobile industry

The product life cycle at an automotive company is a complex and an iterative procedure. In figure 4 the general life cycle of an automotive industry can be seen. The phases that will be discussed in this thesis are mainly regarding Design and Calculation. The CAD methodology will be implemented at the Design phase and will improve the work both for Design and Calculation. Typically in the automotive product development process there are constant changes. The dynamic process requires good communication and the ability to always adapt the changes to the work. For example, in the design process it is normal to perform changes on another users CAD model. It is therefore important to create user friendly models.

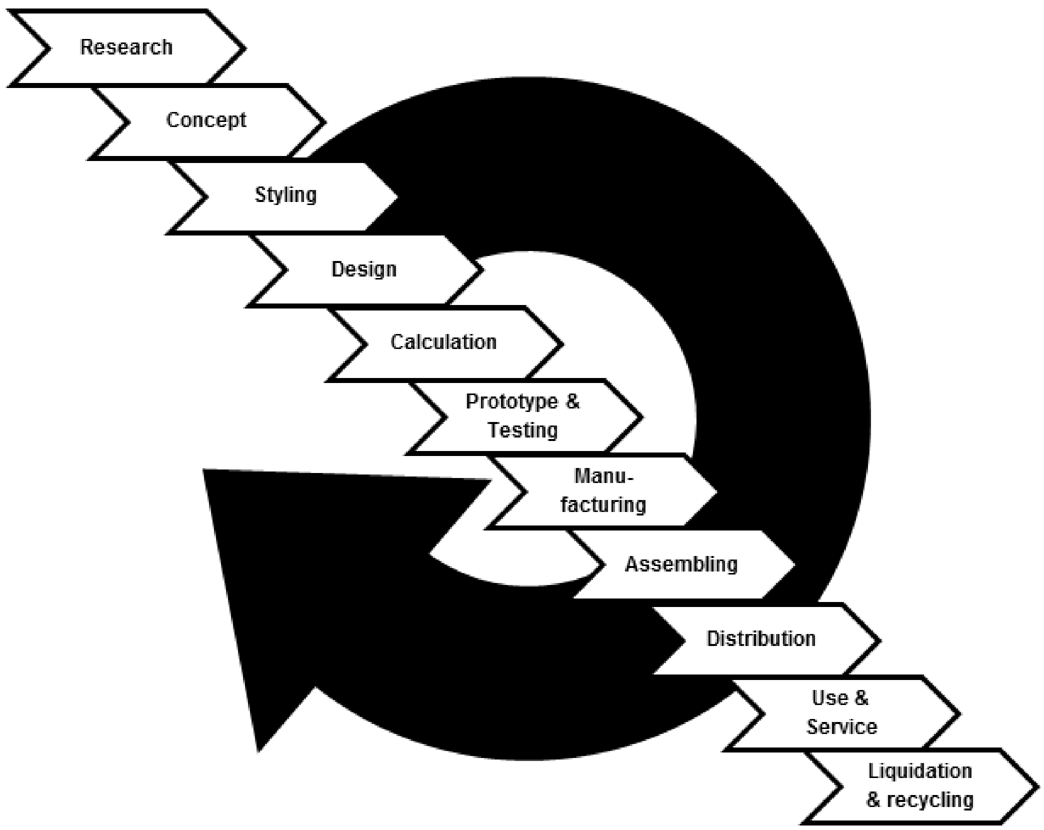


Figure 4: Product lifecycle in the automotive industry (Hirz, Dietrich, Gfrerrer, & Johann, 2013)

2.2 Automobile interior plastic A-pillar trim part

A car consists of near vertical structural-supporting pillars, which are called ABC- and sometimes even D-pillars (Mallens, 2005). The ones that are placed between the front windshield and the first side-window are the A-pillars (Mallens, 2005). In figure 5, a simplified definition of the ABC-pillars is shown.

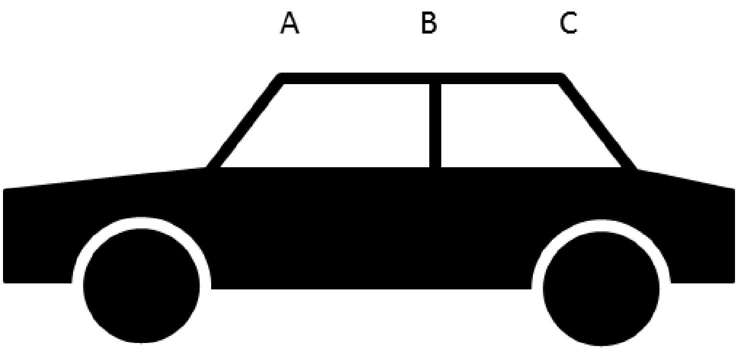


Figure 5: ABC-pillars

These pillars are covered with injection moulded plastic covers, and are sometimes also covered with fabrics, in order to hide the Body In White, BIW, constructions, and to give a more luxurious interior appearance (Mallens, 2005). These covers are called A-pillar trim. The trims purposes are in both a safety aspect as well as aesthetic satisfaction. The A-pillar trim is divided into two parts; A-pillar lower and A-pillar upper, where A-pillar lower is connected to the dashboard (Lygner, 2014). The A-pillar is often integrated with the side airbag (Mallens, 2005). Furthermore they work as impact resistance of the head by absorbing energy (Kim & Kang, 2001). This is done with the rib structure of the A-pillar and the trim offset, which is the distance between the trim and the body. This is a major concern, especially in those countries where there is no law regarding wearing a seatbelt, such as some states in the United States (Lygner, 2014). Besides the ribs the A-pillar trim consists of clip houses and clips. The clips enable to mount the trim cover on the BIW construction. The clip houses creates a clearance between the cover and the BIW (Balasubramanyam, 1999).

The A-pillar is limited in both weight and dimension. There is always a need to have the car as light as possible to keep the efficiency high and improve the engineering characteristics (Lorin, 2012). Thermoplastic is commonly used for this reason, since it has high ductility and impact resistance (Mallick, 2010). Furthermore, the dimensions of the A-pillar also need to be restricted regarding obstructed visibility. A generic shape of the interior part of the A-pillar can be seen in figure 6. The ribs are meant to absorb the energy at a collision. A FMH, Free Motion Head, simulation is performed to validate this.

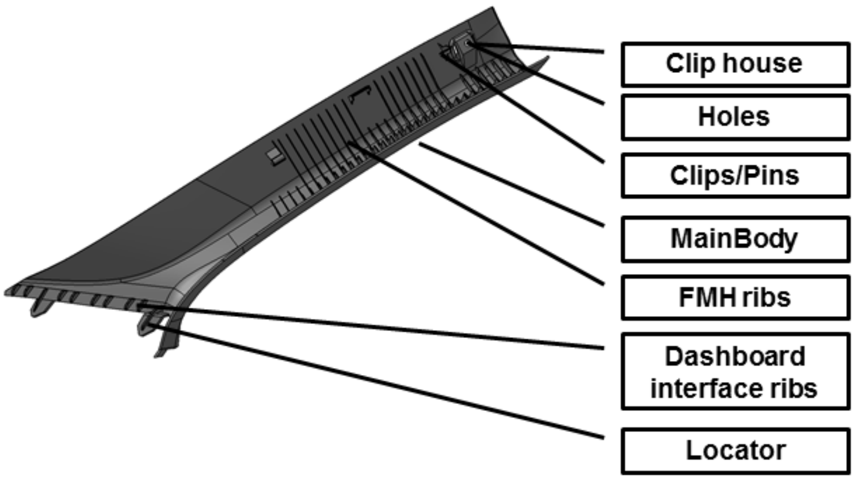


Figure 6: A-pillar trim

A general guideline for the trim to follow is to let the ribs run all the way and to not be abrupt, to get higher efficiency of the absorbing of energy during impact (Volvo Cars Safety Centre, 2011). It is recommended that the gap between the ribs and BIW should be less than 2mm in order to absorb energy by permanent deformation (Volvo Cars Safety Centre, 2011). The ribs are usually 8-10 mm over the B-surface (Lygner, 2014). The ribs locations are defined due to previous knowledge of the designer, and are later confirmed by analysis at CAE. Then the placement or the amount of ribs can be changed. If the impact area requires, the ribs can be crossing each other to absorb the energy more efficiently (Lygner, 2014). The ribs should be placed at the impact area to create stiffness and avoid gaps for sliders (Volvo Cars Safety Centre, 2011). The dimension is usually set to be less than needed when building real prototypes, since it is easier to carve out material than add material to the tool.

The clips and the clip houses are positioned on the A-pillar trim to create stability of the part (Volvo Cars Safety Centre, 2011). They are not positioned at the impact area since it can prevent rotation of the head (Volvo Cars Safety Centre, 2011).

The BIW pillar should support the pillar trim. The A-pillar trim will have its interface against the headliner, the instrument panel, the front door seal, and the windscreen. The different criteria that the part usually should fulfill can be seen in table 2.

Table 2: Example of Part Criteria info needed (Geely, 2014)

Tooling direction	Draft angles
Tooling split lines	Min radius
Legal min radius	Paint lines
Graining	Colour
Surface treatment	Inject, mould flow and weld lines
Material thickness and tolerance	Flange length
Matching surfaces	PQ- Gap and Flush requirements
Design execution sections	

2.3 Plastic injection moulding

Injection moulding is a production method where a certain type of material, such as plastic, is injected into a mould to achieve the desired shape. The tool is divided into separate parts to be able to open up and take out the part. The section in which the tool is divided is identified as the split. This can be seen on the part as a split line. A regular injection mould is open in one direction and the design of the part has to be planned for the tooling direction. The plastic part that is considered in this project has ribs and clip houses and the designer must have the pulling direction in mind for being able to be produced. There is a necessity of eject material from different angels and these are done in separate operations. First the plastic is injected into the tool and then the slider removes the unwanted material from another angel, in particular from the clip houses, before the tool has opened. The sliders need to have space to move and therefore it is not possible to have any ribs positioned next to the clip houses. The injection mould tool can be seen in figure 7.

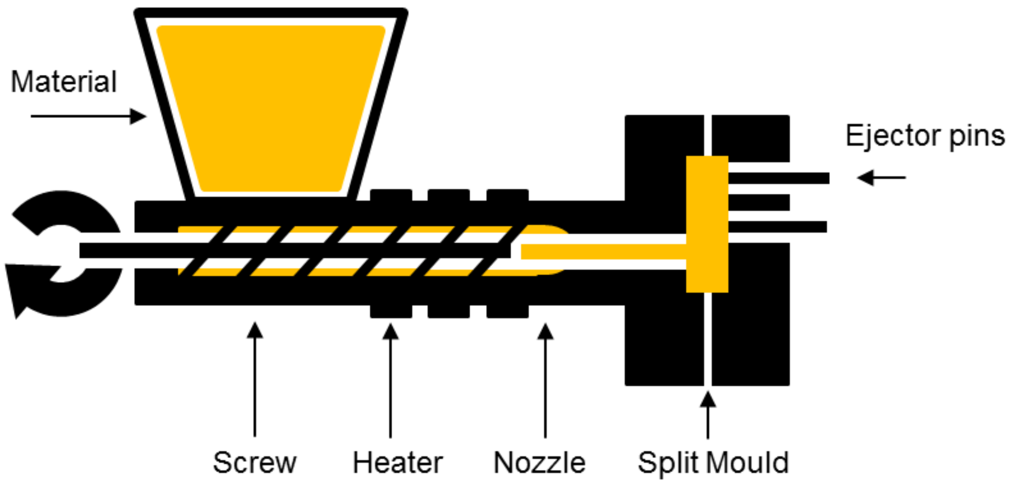


Figure 7: Injection moulding

In general, for injection moulding the visual A-surface is on the die side of the tool, and the B-surface is on the punch side. The A-surface origin from the styling surface and the B-surface is defined as the engineering thickness surface.

In order to be able to pull out the tool the part must ensure to have drafted angles otherwise there is a risk of breakage. Depending on how the structure is desired the size of the angle is different. A larger angle gives surfaces with texture, and is common for visible surfaces (Lygner, 2014). Due to shrinking, during the injection moulding process, the part does not achieve the same shape as the mould. If the process and the design are not planned correctly there is also risk for sink marks, which is caused by the unequal and uneven, cooling process. Recommended design of the rib can be seen in figure 8.

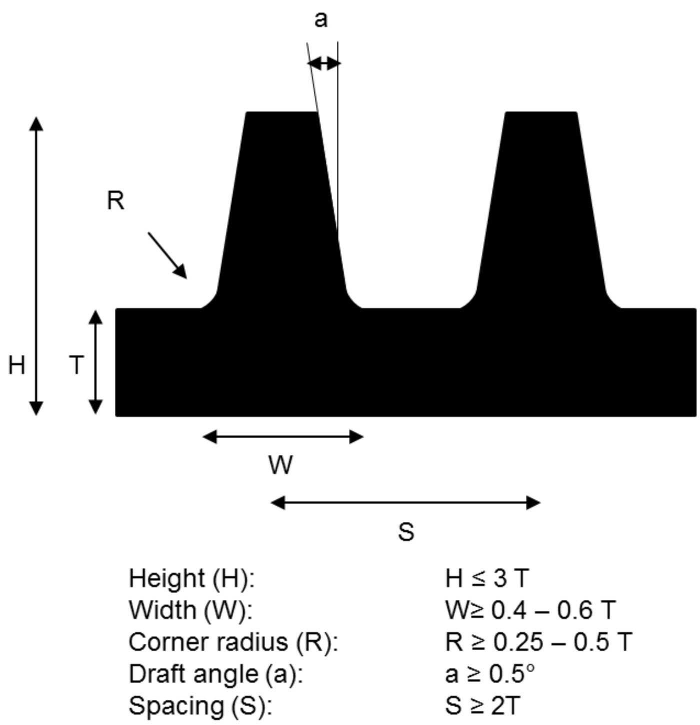


Figure 8: Recommended rib design (Volvo Cars Corporation, 2011)

2.4 Design

Computer-aided design, CAD is a common tool in the product development process and is used in industries such as automotive, aero and consumer products. CAD is becoming more and more of an important tool. Today some prototype tests are replaced by virtual tests, and these can be made already in CAD. It is a good tool to create products designs and to plan for manufacturing. Another benefit of the usage of the CAD software is that it can also perform simulations. Modifications and verifications can therefore be done before the model has reached the CAE department. Benefits within computer technology are as follows (Rosato & Rosato, 2003):

1. Makes the developing process easier and the possibility to adapt innovative ideas to the design
2. Makes drawings more accurate with minimal errors
3. Improves communication with the assembler
4. Better manufacturing precision
5. Able to response to market demand faster

CATIA is a multidisciplinary software that is well suited for the automotive industry and is the tool that is used at CEVT. It can handle CAD, CAM, and CAE in a range of designs, manufacturing process, electrical and fluid systems designs, and systems engineering (Systèmes, 2014).

CAD modelling can be performed in four different ways (Rosato & Rosato, 2003). The first one is *wireframe* modelling. This is a skeletal description where the model consists of points, lines and curves, and does not include any surfaces. This describes the geometric boundaries of the object. Wireframe modelling is the simplest of all of the techniques.

The second one is *surface* modelling. It defines the edge and the surface of the object. It can also include mass-related properties for the 3D model, such as volume, surface area, and moment of inertia. Within surface modelling there are two methods to generate surfaces, which are facets and true curve surfaces (Rosato & Rosato, 2003).

The third CAD modelling method is *solid* modelling, which defines bodies of volume and mass (Rosato & Rosato, 2003). This method ensures that the model is valid and achievable, which surface modelling is not directly achieving. Solid modelling consists of two types; constructive solid geometry (CSG) and boundary representation (B-rep). CSG uses Boolean operation as union, subtraction and intersection, while B-rep is defined by surface boundaries such as faces, edges and vertices. The fourth method is *hybrid solid* modelling, which is a combination of wireframe, surface modelling and solid geometry (Rosato & Rosato, 2003).

Parametric modelling helps to optimise the size and shape of the CAD model more efficient (Rosato & Rosato, 2003). As a guideline it is advisable to use as few parameters as possible (Rosato & Rosato, 2003).

The CAD modelling of the interior design is usually done when the exterior is set. The development process at CEVT starts with the CAS department creating the interior surface that will be visible (Lygner, 2014). This is called the styling surface and is the same as the A-surface, which is created by the studio engineers. In general, the A-surface cannot be changed. The A-surface is usually done in surface modelling and the designers complete the CAD model by either continue working with surfaces, or adding thickness and working on the inside of it by adding ribs, clip houses and clips. The challenge is to know what the constraints are and identify the best way to construct the model so that any necessary modifications can be easily and efficiently done.

A surface with good isoparametric curve flow is when different surface patches have the same flow and there are no angles between these curves. This is visualised in figure 9. A good isoparametric curve flow is often requested for the A- and B-surfaces, due to the isoparametric curve flow will be transferred to the solid. The A-surface is developed from the styling surface by the studio engineers. The intention is to have as good alignment as possible. The challenge is to make the isoparametric lines aligned for the whole model, not to mention the whole vehicle.

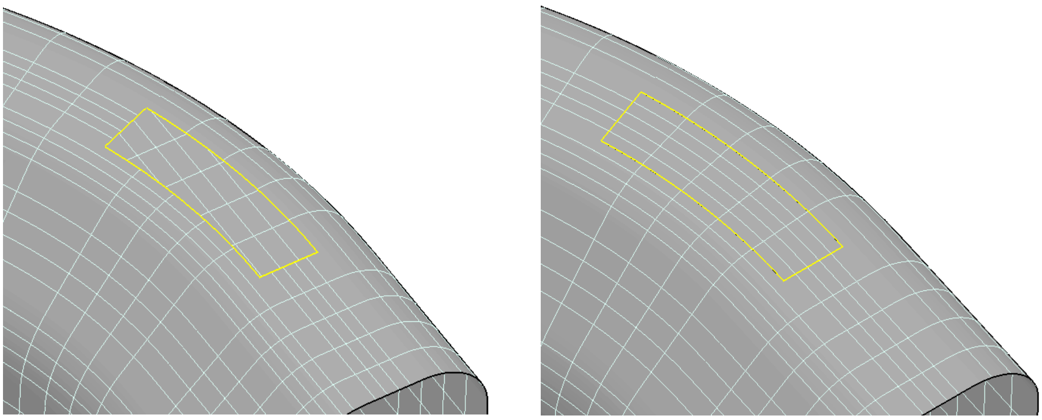


Figure 9: Bad respective good isoparametric curve flow (Volvo Cars Corporation, 2014)

To get a good isoparametric curve flow the surfaces can be aligned with a common spine. This spine is difficult to select and requires a lot of considerations. The curves of the surfaces will end up normal to the spine, and can therefore affect the curvature of the surface so it no longer aligns to the styling surfaces. The spines can be selected from the design cross section and the spine will indicate the direction.

2.5 Analysis

Computer aided engineering, CAE, is to simulate a scenario to test whether the product is fulfilling all the requirements and is capable of the right performance (CAE / Computer-Aided Engineering, 2014). The prototype building stage is pushed later on in the development process since CAE simulations have taken over much of the prototypes purpose. CAE is not intended to replace real prototypes, since these are still important to prove the design. The prototypes today are to clarify once again that everything is correct. Changes and iterations are easily performed with CAE and both time and money can be saved. The process encloses pre-processing, solving and post-processing step. The pre-processing phase includes modelling the geometry and defining physical properties of the model, applying load and/or constraints, and may include the mesh generation depending on the software. At the solving phase a mathematical formulation is applied to the model. The post-processing is the phase where the results are obtained and analysed.

At CEVT a mesh consisting of mixed elements is usually used (Larsson, 2014). A model with a complex geometry can be challenging to receive a result from if it only has quad elements. On the other hand, only triangle elements give a result that is too stiff. Target size of the elements at CEVT is within the range of 3-5 mm for plastic parts.

FE models of plastic parts are usually defined by the mid surface and it is in this way that it is performed at CEVT in order to get the accurate engineering properties (Larsson, 2014). The mid surface is elicited from the middle of the top and bottom faces (Suresh, 2003).A demonstration of the mid surface can be seen in figure 10. This can be achieved in several ways and one of the ways is through a medial axis transform or a so called skeletal representation (Suresh, 2003). These consist of a skeleton and a radius function, where the skeleton follows the shape of the solid, and the radius function seize the local thickness (Suresh, 2003).

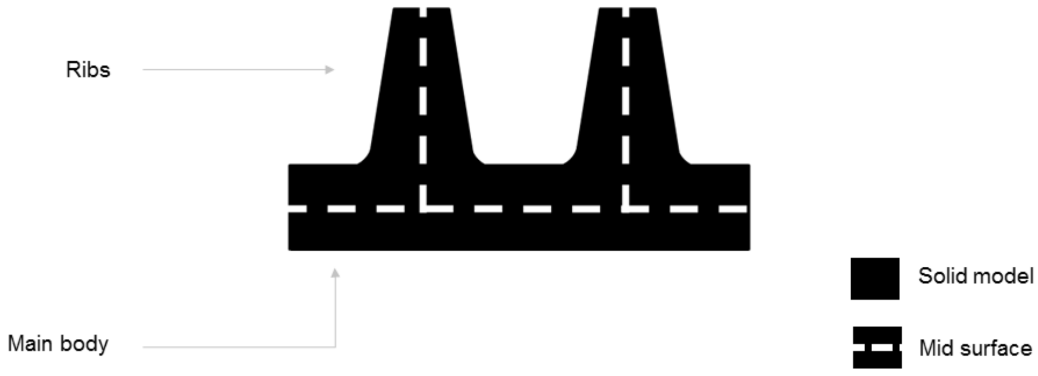


Figure 10: Mid surface definition

It can be difficult to define the mid surfaces on injection moulded plastic parts, due to that the parts can have different thicknesses. The ribs may cause trouble especially if they are located on both sides of another rib with a small offset. Generally, the transition is not done correctly when choosing the automatic tool in ANSA and there is therefore a need of manual work.This might become a problem if the ribs are crossing each other or are placed between the ribs and the clip houses. A demonstration of some geometry for plastic parts that is problematic for FE modelling can be seen in figure 11.

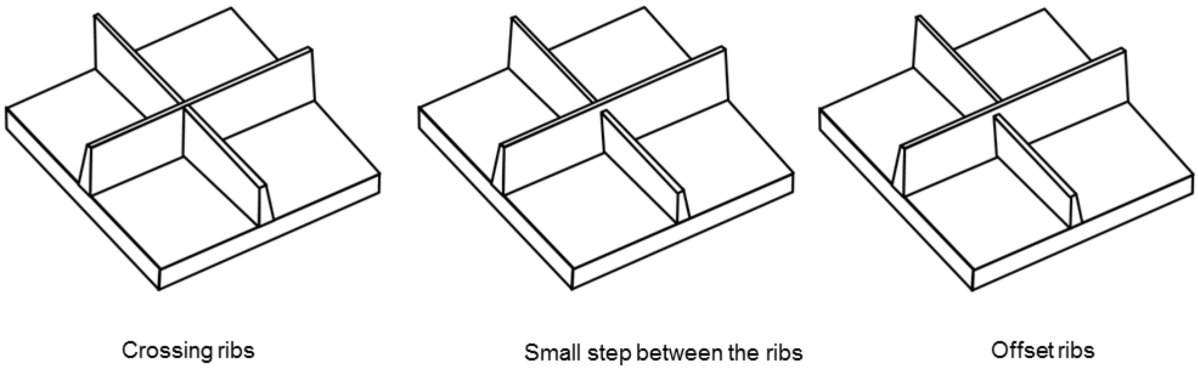


Figure 11: Problematic geometry for plastic in FE modelling

ANSA is used as a pre-processor tool and is capable of maintaining the CAD geometry instead of rebuilding one. The CATIA file can be opened directly in ANSA and the format is transferred into an ANSA file. The mid surface mesh technique in ANSA can be done in different ways (BETA CAE Systems S.A.). The goal is to be able to use ANSA's automatic mid surface tool and to minimise the manual work as much as possible. The automatic mid surface tool offers two options; skin and casting. Skin is suitable for sheet metals and other non-complex parts. It creates a mid surface geometry and applies a mesh on the geometry.

The other option, casting, is more accurate for plastic parts where there is in general a lot of ribs. The casting function does not create a mid surface geometry, only a mid surface mesh.

ANSA can either accept a solid and apply a mesh on that, or it import the mid surface if it is created in CATIA with defined thicknesses (BETA CAE Systems S.A.). Another option is to insert both the solid and the mid surface. ANSA can create a mesh on the mid surface and then use the solid to define the thickness. The thickness can be defined in each node or structured in different property identification's, PID's (BETA CAE Systems S.A.). To define the thickness in each node is a more accurate technique, but it can be easier to handle PID's for a larger model.

In general, there is always a need of clean-up after usage of the automatic mid surface meshing tool. After this clean-up it is necessary to redefine the thickness again to update the geometry for the correct result. Clean-up is usually deleting elements, pasting new ones, and moving elements and it is therefore a need to redefine the thickness. Factors that may affect the mesh result might be that the CAD model is not stitched properly, that there is a poor geometry which is dependent on the designer or the input data for the mesh generation of the FE model.

2.6 From design to analysis

When moving forward from CAD to CAE a new FE model is generated which is based on the CAD model (Rosato & Rosato, 2003). The FE model is constructed of several elements. The elements are connected with nodes where the data result is stored after simulation (Rosato & Rosato, 2003). Generally, more elements represent a more accurate model. To keep it cost and time effective the highest amount of elements are used for areas that are most important, areas where the stress level is the largest (Rosato & Rosato, 2003).

The process of dividing the model into elements is not straightforward (Rosato & Rosato, 2003). When CAD and CAE are isolated modules there can be many data errors as well as duplicated work (Li, Dai, Huang, & Zhou, 2011). CAD systems use relative large tolerances to create a robust geometry (Beall, Walsh, & Shephard, 2004). The model consists therefore of gaps and overlap (Beall, Walsh, & Shephard, 2004). These cannot be seen in the native model, but when translating it to another format this so called "dirty" geometry may occur. (Beall, Walsh, & Shephard, 2004). In the CAD model it is possible to hide geometries that are not being used, such as lines and solids (Schoonmaker, 2003). During the transferring process these are included in the CAD data file, and may become included into the FE model due to incorrect translation (Schoonmaker, 2003). There is a necessity of mesh repair and remodelling in pre-processing for the FE model to become valid. Another reason for the problematic transfer of geometry access for mesh generation is that the analysis is conducted late in the developing phase. The CAD models are therefore far more detailed than what is appropriate (Beall, Walsh, & Shephard, 2004). Whilst, conducting analysis earlier will not solve the problem it will marginally reduce it (Beall, Walsh, & Shephard, 2004).

Furthermore, when designers make changes in their CAD model, the FE model does not update synchronously. The dataflow translation from design to analysis is only a one-way direction and important information such as constraints, non-geometrical entities, and geometric intersection details are generally lost. A way to improve the efficiency is to establish an iterative design-analysis optimisation process (Li, Dai, Huang, & Zhou, 2011). The data result after analysis cannot always be optimised and directly used to enhance the CAD model as they no longer match (Li, Dai, Huang, & Zhou, 2011).

There can also be discontinuities in the part, for example in the meeting point of several surfaces if all of them are not connected properly. There might also be problem when choosing the mesh size and this can be larger than the geometries in the model. There is always a choice between selecting an automatic or a manual mid surface creator. It is rarely ever easy to use an automatic tool and many times is more simple and effective to do it all manually from the start.

The translation problem might be due to translation to another file format, but it might also be due to different tolerance settings in the software (Adolfsson, 2014). The tolerances used in ANSA are not always compatible with the ones in the CAD file (BETA CAE Systems S.A., 2014). To match the tolerances in the CAD file can be controlled by *nodes matching definition* and *curves matching tolerances* (BETA CAE Systems S.A., 2014). Figure 12 shows these matching distance tolerances.

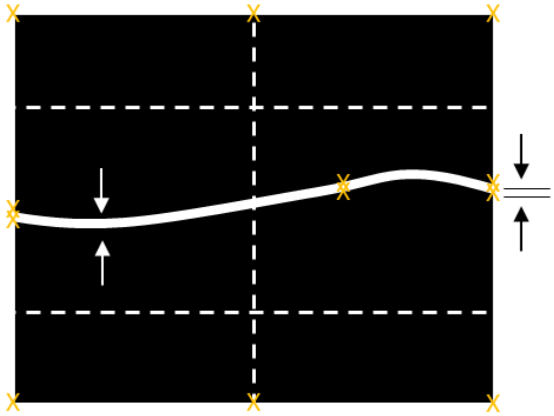


Figure 12: Tolerance settings ANSA (BETA CAE Systems S.A., 2014)

ANSA tolerances may not play a significant role in the translation process and influence its accuracy (BETA CAE Systems S.A.). The tolerance settings in ANSA are only used during actions within the software and the translation result should therefore not be degraded (BETA CAE Systems S.A.). At CEVT and Volvo Cars the geometry scale is set to Standard scale in CATIA which gives the precision of the characteristic value 0.001 mm (Karlsson, 2014). ANSA can be defined by default values or changed depending on if it is plastic or steel (Larsson, 2014).

2.7 Mesh verification

A common misunderstanding while analysing the mesh result according to Shengwei Ma can be concluded in five points (Ma, 2014);

1. A good mesh is when it follows the geometry of the CAD model well
2. A good mesh is adaptable in every situation
3. The mesh type quad is always a better choice than a triangle mesh
4. A good mesh can only be generated by manual work and not by using automatic tools
5. A good mesh is the same as a fine mesh with large elements/node amount

According to the first point; a good mesh is depending more than only on the geometry (Ma, 2014). The physics plays a significant role. The nodes should be placed in areas depending on the stress, strain, displacement, pressure, and velocity that are acting on the model. The geometry does not need to be in detail; often a simplification of the geometry gives a better

FE result. Based on the impact of the physics, the geometry can be simplified and therefore give a good mesh.

The second point is that the mesh is only good for the defined boundary condition, load condition and the analysis type (Ma, 2014). If any of these conditions are changed the mesh might not be accurate anymore. As Ma pointed out, the physics has a major impact.

Historically quadratic meshes have been proved to be better than triangle meshes, as the third point explains (Ma, 2014). As technology has moved forward, the gap between these types is not as large as it has been. Still, due to knowledge of behaviour, in some industries, such as in aerospace, the quad is preferred. Otherwise, triangle can be used for complex areas where quad has a tough time to be implicated and is often time consuming. A triangle mesh needs more elements/nodes than the quad and it gives a longer computational time.

According to Shengwei Ma's fourth point, the mesh that is done with manual work is only as good as the person behind it (Ma, 2014). This requires a lot of knowledge and the person doing it might not take into account everything that is necessary for the simulations. Instead, the automatic tools have the technology to identify curvatures, gaps, small features, and sharp edges and angles. The automatic tool might not achieve the perfect result, but with a combination of automatic and manual work the result may be better, rather than a non-experienced user creates manual mesh without deep understanding of the physics behind it.

The last point indicates that a fine mesh might only affect the computer resource rather than proving a good result (Ma, 2014). The mesh can bring forth an invalid model with unphysical results. Instead it is better to define a finer mesh in impact areas of physics and parts that are going to be analysed. Simplification can always be done, but it is necessary to have the understanding of what it will bring; a 2D model may often give a more accurate result than a 3D model.

The opinion of when the mesh is good or not can vary. An easy way to observe the mesh is to study how, or if, it affects the result (Connector, 2012). A bad mesh can also be defined with no calculation at all can be conducted. Another point of view focuses on the time that is consumed of performing the mesh or running the result. The question might not be about the mesh itself, since the accuracy might lie in the grid (Connector, 2012)

It has been proved that a result of an irregular element mesh with element skewness and element size variation, as having less discretisation error than a model with perfect squared element mesh (Connector, 2012). This shows that a good indicator of mesh quality should not be based on the element geometry alone (Connector, 2012). The result of the study can be seen in figure 13.

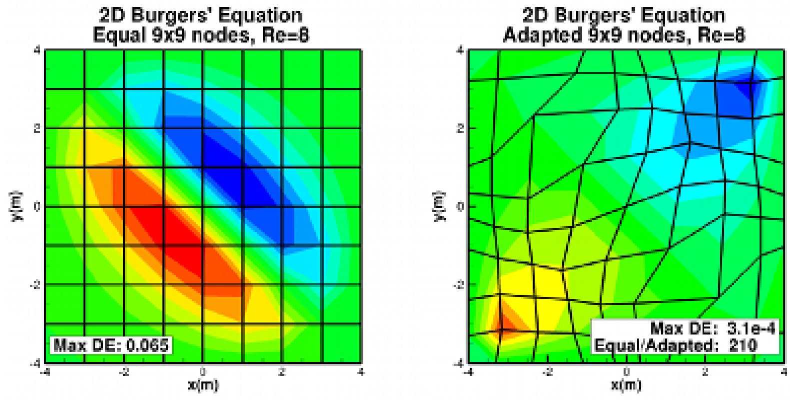


Figure 13: a) "Perfect" elements b) poor mesh (Connector, 2012)

Element skewness and element size variation may bring forth quality errors (Connector, 2012). The issues with this are due to convergence and metric computations.

Mesh refinement should intend to be uniform and consistent (Oberkampf & Roy, 2010). This requires not the mesh itself but the ratio of the grid refinement which should not vary in the domain. The quality of the grid should be consistent in terms of element skewness aspect ratio, stretching factors and so on.

According to the User guide of ANSA (BETA CAE Systems S.A., 2014) a model can seem to have good quality, i.e. continuous and no raptures, but can in fact possess the opposite. In order to check the mesh boundaries for gaps or discontinuities, this can be prevented. These gaps and discontinuities can exist due to improper clean-up. It can also be due to inadequate connection between geometry and the shell elements. It may also occur at the element penetration which can decrease the quality of the mesh. This is due to coarse mesh resolution, incompatible mesh, and due to the CAD model. In case of improper CAD model; dislocated geometry entities can exist, or intersection of parts.

In order to ensure convergence to the exact solution, the mesh is checked if it meets all the requirements as stated by ANSA. The requirements that are selected are aspect ratio, skewness, warpage, and jacobian. This has been selected due to common procedure within automotive industry and by CEVT. The definitions of these are set by the toughest requirements according to the user guide from ANSA (BETA CAE Systems S.A., 2014).

Aspect ratio is the ratio between the largest and smallest element side, which is visualised in figure 14. It is important that the ratio is the same all over the domain, not for the dimensions.

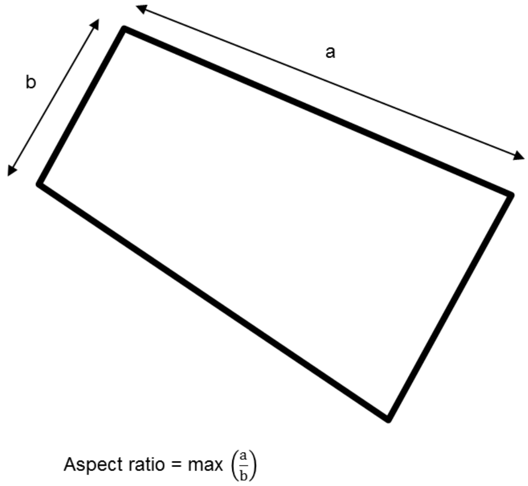


Figure 14: Definition of Aspect ratio according to NASTRAN

Skewness are controlled by the elements skew angle, see figure 15. The perfect angle for a triangular element should be 60° for each angle. Any deviation from this is called skewness.

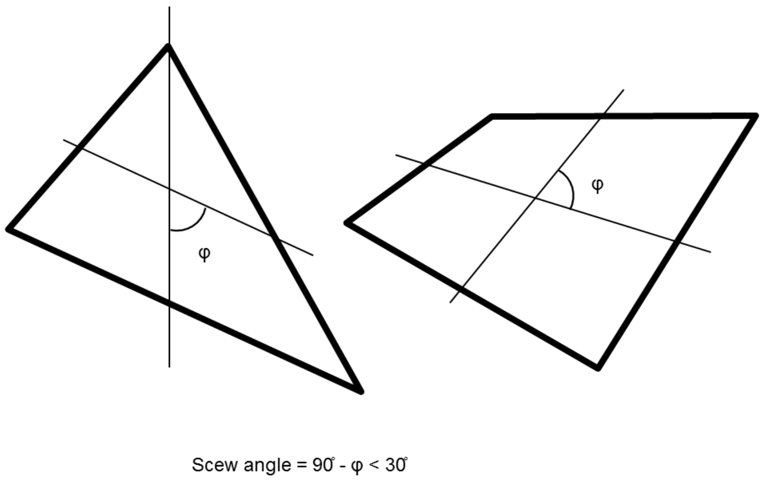


Figure 15: Definition of Skewness according to PATRAN

Warping is only suited to the first order quadratic elements and it controls how far these quadratic elements are from being defined as planar. The warp angle of an element is the angle between two planes, see figure 16. By default ANSA is using I-DEAS definition.

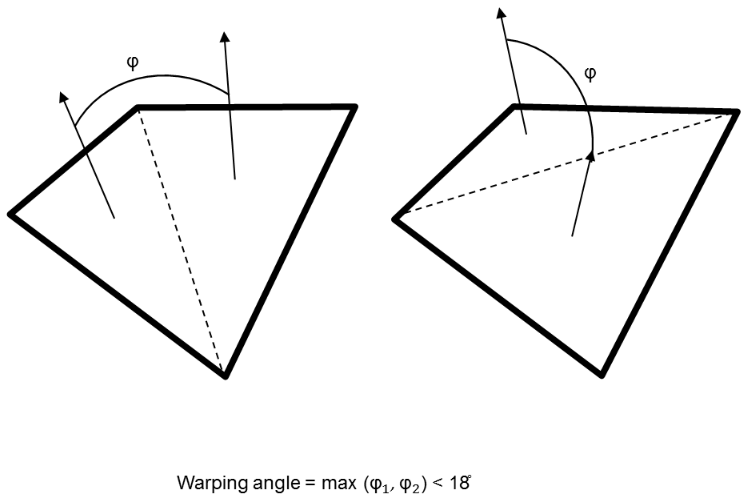


Figure 16: Definition of Warping according to I-DEAS

Jacobian controls the ratio of the smallest value over the largest by calculating the determinant of the Jacobian matrix for each integration point of the element. An ideal result is a deviation of 1, which defines a perfectly shaped element.

2.8 Data transfer and PDM

Product data management, PDM, is a system that contains data and information for the product development process. It keeps information related to design, manufacture, and maintenance (Philpotts, 1996). The PDM system is often an intermediate system where CAD models are stored and managed before being transferred to CAE. Different PDM system attributes and settings also influence the success rate of automatic transferring of the CAD data to the CAE environment. PDM also helps to reduce the cost in manufacturing, shortens the time to market, and improves the quality of the product (Philpotts, 1996). PDM helps to provide the correct data in the appropriate format to the right person on time (Liu & Xu, 2001). This is already implemented at CEVT and the PDM system Teamcenter is used to share CAD models and provides the transfer between CAD and CAE.

2.9 CAD Standards

Industrial companies usually have a formal standard that includes guidelines for file naming structure, folder structure, software products and version, drawing set-up, dimensions, amendment and revisions (Middlebrook, 2000). In addition, it is common that companies have part specific standards, for so called “best practice”. These practices contain more specific guidelines on how to model.

3 METHODOLOGY

This section start with a requirement identification to understand the plastic part that is studied during the project. This is achieved by a black box and a functional tree. Then the result of the external research is presented. Further on, the requests from the stakeholders are presented in the specification of requirements.

3.1 Requirement identification

To understand the process of design and analysis it is first necessary to define them. They can be defined as a system in a functional diagram, a so called black box. Figure 17 of the black box shows how the design and analysis phases are dependent on each other together with requirements from outside areas. As they are inputs into the system, the A-surface from CAS is an important parameter together with technical requirements, legal regulation, restriction, aesthetic, appearance, dimensions, performance, cost, durability, and supplier limitation. The design phase consists of the creation of the CAD model and then the preparation for the analysis with suppressing and removing redundant features. In analysis the mesh is generated at the mid surface. After running simulation and analysing the result, new data and parameters are once again passing the system, together with updated inputs.

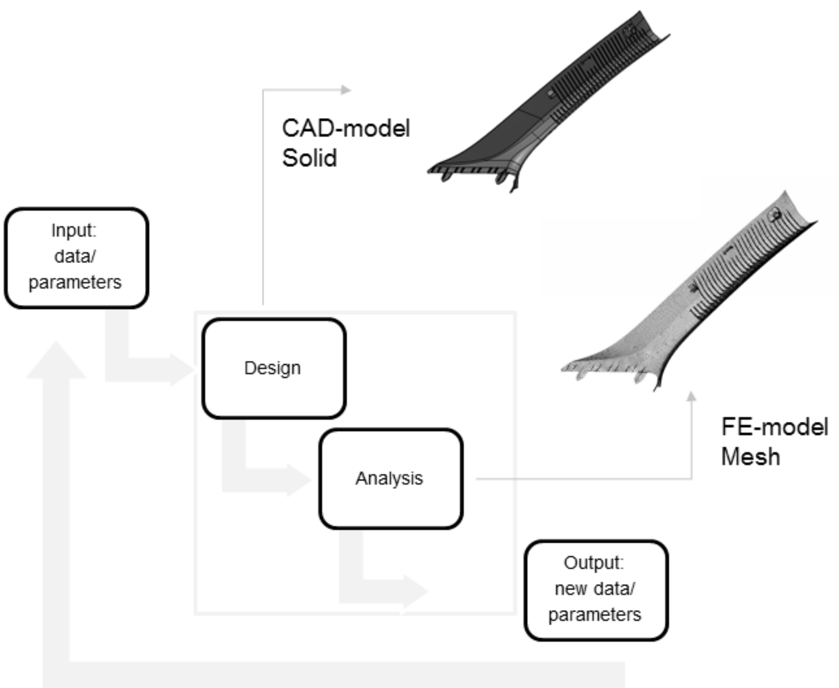


Figure 17: Black box

In addition to the black box, a functional tree has been produced to clarify the product that is being studied for the creation of the CAD standard. The functional tree, which can be seen in figure 18, shows the main function that the part consists of and what is required from design aspect to develop. The first stage in CAD is to set the limitations and to know that everything will work. The technical requirements must be met and the design might have to be changed if the target points are not located properly. The consequence might be that the A-surface needs to be changed, the ribs and clip houses height or placement need to be modified. The A-pillar trim should also include the interaction with mating parts; such as packaging, door, window, dashboard and sometimes the lower A-pillar.

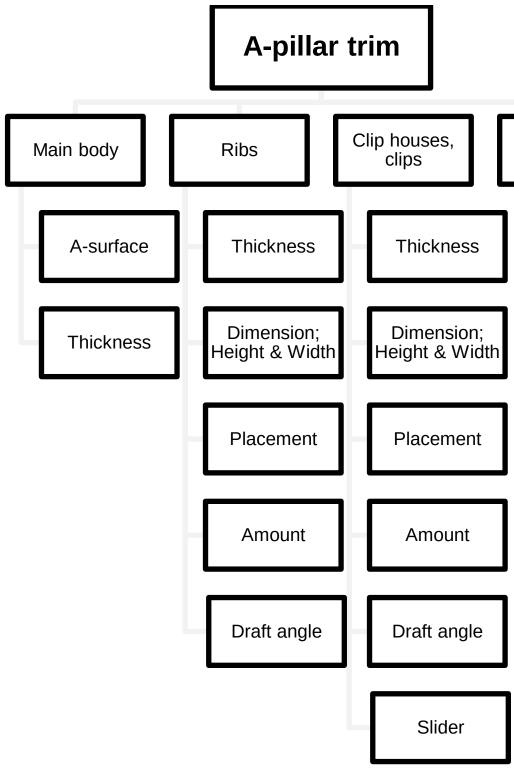


Figure 18: Functional tree

3.2 External search

The benchmark study of the external research has been summarised in appendix B. The external search has also given information regarding different ways to construct the models. The main part of the following section is from the course CAD Advance at Volvo Cars (Volvo Cars Corporation, 2014). One of the major ways is conducted of surfaces in generative shape design and is only added with thickness in part design. Another major way is to add thickness to the styling surface, the A-surface, as the first procedure, and then create the rest of the model with solids. It is more common for sheet metal to work with surfaces and it is rarer for plastic part.

For solid modelling there are different techniques. It is possible to create the model with solid features, such as the ribs from the main body. Or alternatively, another one, which is based on the CAD standard that existed at Saab/GM, is to start with a block and create the shape by removing parts from it.

The styling surface is always under working progress during the whole developing phase and the design phase needs to be started before the actual styling surface is released. This is one reason why the CAD model needs to be created for modification. The Volvo way is to create *flexible* and stable models. The models should be able to be reused, updated and generate an expected result. Functions should not be dependent of each other and should be joined and trimmed as late as possible. Volvo Cars use a tool called *Q-checker*, which is a tool that checks both the work and the tree structure. There should not be anything more and nothing less than stated in the requirements in the CAD standard.

The general guidelines for creating flexible surface models is to work with oversized geometry, avoid sub elements, use join, avoid sketches or keep them to a minimum, and have a reference geometry based on wireframe elements. Oversized geometry goes in hand with having a robust geometry with clean splits and intersections. The model is then able to handle geometrical changes.

Sub elements can be completely avoided at surface modelling, such as lines, points, and faces. It might be hard to replace something based on sub elements, since it is not a one to one replacement and the links are broken. In order to check if the chosen element is a sub element, the selected element always has a backslash in the text field within the applied command in CATIA. When selecting the element in the model there is a possibility to select sub elements, a stable way to do so is to instead select the element in the tree structure. To avoid sub element comes with a cost; to achieve an easier update process there are in general more operation steps in CATIA. Within solid modelling it is difficult to avoid using sub elements; for example draft has to be done by selecting faces of the element.

The CAD standard that exists at CEVT guides to create a model that is parametric with selected thickness, material, driving geometry and geometry controlled by skeleton features. Depending on whom you ask at the company the build-up principle is different. Some designers say that models that consist of a lot of sketches, is equal to an unstable model. Where others designers claims that you can create sketches, but keeping them simple. Instead of having a complex geometry with constraint within and to other features, they states that it is actually better to control it with parameters and skeleton, such as planes. This is also a benefit when modification has to be done and instead of entering the sketch, the changes can be done more easily by just altering the parameter in the specification tree. A general guideline is to have as few parameters as possible and to think them through. It is essential to always link from the bottom to the top in the tree structure, never in the opposite direction. Otherwise it is harder to organise the links and to have control of the work.

To have a stable and flexible model that can be modified, one should chose functions that are not dependent on each other. For example, the designer should select points instead of vertices, boundaries instead of edges, surface features instead of faces. Airbus 3D modelling rules, claims that the reason for this is that when replacing one part with another, for example a surface, the surface is rebuild and has new vertices, edges, and faces. This is not a problem when just changing some parameters. The model is then only modified and not re-built; thereby B-rep is avoided.

It is also important both in CAD and CAE environment to have a good model with good geometry. This can be achieved by not having duplicate lines. Also by having the surfaces trimmed and joined correctly to avoid discontinuousness in meeting points. Moreover, not having single surfaces without thickness, and so on.

3.2.1 Logical tree structure

Another guideline is to maintain the *tree structure* structured and keep the *relations* between features as simple as possible, see figure 19.

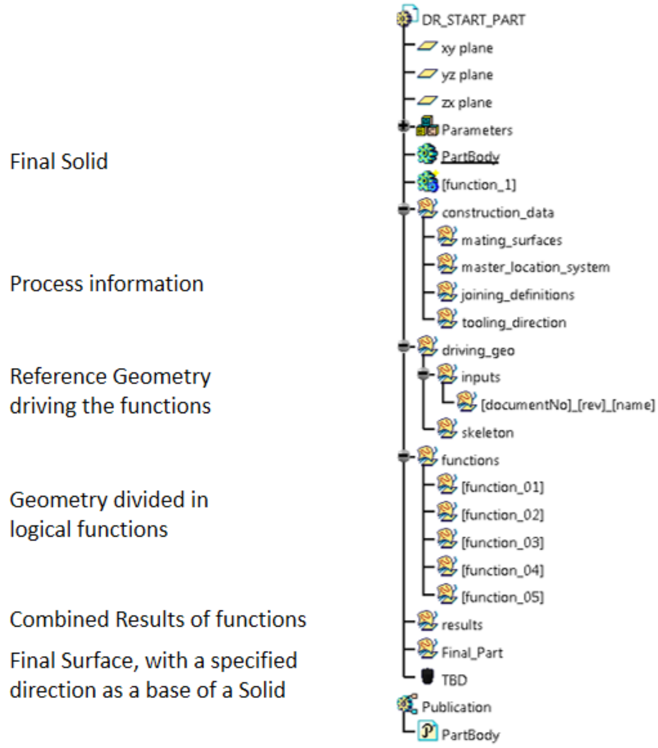


Figure 19: Start-up file

The relations should not be depending on too many features. Creating short and compact relation chains make it possible to have shorter update times, faster troubleshooting, together with a more logical tree structure. To be able to keep track of the relation it is also necessary to have *descriptive names* of key functions, features and elements. It is good to add *colour* to the part model, as well as different line types and point types, to visualise key features and elements in the part. Colours to avoid are red and orange since these are already used in CATIA by default. When assigning a colour it is advisable to not select faces, since it will result in sub elements. The selection should always be the elements when assigning colours.

There is no upper limit of how many elements that can be selected in a command, but it is better to select fewer to avoid update error. They can be divided into logical groups, such as horizontal and vertical.

Isoparametric can be toggled on in order to check the definition of surfaces. This is recommended for A- and B-surfaces. The surfaces have a mesh and there should not be an angle between the surfaces mesh. The surfaces are considered as of good quality if they are aligned in same direction. This can be controlled by using splines. Using common splines for several operations will solve a lot of problems with the surfaces.

Further on, when surfaces are based on curves with bad quality this will affect the surfaces quality as well. Discontinuities and several unnecessary surface patches is the result of the bad quality.

3.2.1.1 Parameters

Parameters can be defined in sub sets to order the tree structure. The formula operation uses SI units by default, so even if modelling in millimetres, the functions are defined in meters if no unit is included. To be able to add a formula to a circle the dimension most be defined on the radius and not the diameter. A quick way to select a parameter is to write an equal sign and then click on which parameters that should be applied in the tree structure.

3.2.1.2 Construction Data

The construction data contains geometrical sets about the process information.

Mating surfaces

The geometrical set contains the contacting areas of other parts, such as joined curves.

Master location system

The geometrical set contains six or more master location points with the correct naming conventions and order. All points should be visible and published.

Tooling direction

The geometrical set contains a line with a descriptive name for each tooling direction. Each feature that will need a slider should also have a descriptive line, even if they have the same tooling direction. For a simple tool, the aim of the selection of tool direction should be to provide the minimum height of the mould tool, and also to include all the drafted components.

3.2.1.3 Driving Geometry

The geometrical set of the driving geometry contains the reference geometry that is driving the function.

Inputs

Reference elements must be dead (isolated), which means that they should be pasted as a result with no links so they do not have any historical reference. It is important to name this correctly so the reference can be found easily in the PDM system. A common reference element is the styling surface, and it is good to tell which revision and date this was retrieved.

Skeleton

Other aspects that need to be followed in the tree structure are the *skeletons* being used as driving geometry. This geometrical set should contain lines, planes etc., that includes parameters that can either subtract or expand the model. Skeleton as well as features should be *reused* to keep the model lighter. Wireframes that are useful for several features should be placed in the skeleton folder; otherwise it should be placed in their function set.

3.2.1.4 Functions

The function contains geometrical sets that divide the geometry into logical functions of areas or purpose; this will set the base for the function bodies of the PartBody. The geometrical set *functions* should be independent and separable. It should be possible to remove this without causing the CAD model to collapse. Another option is to base the functions on each other, in the same flow direction so it becomes heavier further down in the tree structure. The function groups should be done in a flexible manner since these are often changed during the development process.

3.2.1.5 Results

The geometrical set of the *result* combines the functions and a common result is a join or shape fillet.

3.2.1.6 Final Part

The *final part* is a published join with the name DSM (design shape model). It is usually a repetition of the result.

3.2.1.7 PartBody

Solid modelling can be defined as *single body*, where all solid features are placed in one body. This is only suitable for basic models, since it can be hard to track or make features as independent. Another modelling technique is *multi body* modelling, where solid features are located in different bodies and are combined with Boolean operation. This is suitable for a complex model.

3.2.1.8 Publication

Publication of parts or features needs to have the same name as the publication to be able to copy between different models. Publication should only be done at part level, not assembly level.

No deactivated or unnecessary elements should be in the CAD model when *releasing* the part in Teamcenter. It is recommended that the spelling should be controlled, if changes are done after the model is released, it will affect the downstream processes. If the spelling has to be changed, other users' models will not be updated. When releasing a CAD model everyone can use the publication. It is therefore important to execute the model properly.

A published element cannot be replaced easily. This can be overcome by publishing a join of the element instead of only the element.

3.2.2 Guidelines of operations

A *positioned sketch* makes it possible to choose both origin and orientation. Always use positioned sketch, since a normal sketch can become a positioned sketch by changing the sketch support to a positioned sketch. Everything inside the sketch can be described as sub elements, it is therefore necessary to keep it simple.

There are different opinions whether to use the H and V axis or not when constraining the geometry. Volvo Cars recommends using this whilst constraining against flexible geometry to make the sketch more independent. On the contrary, the German automotive industry says that no dimension constraints should be defined against the H and V axis.

The sketch can be checked with the 2D analysis tool bar. All sketches should be ISO constrained and the tool should check whether it fulfils this or whether it is *under* or *over constrained*.

As a rule, it is advisable to avoid sketches, it is better to model in 3D, since this is more stable than sketches. Further on, it is advisable to use a minimum of 3D references in a sketch, since all elements in the sketch are sub elements. *Relation*s should only be referenced to stable and flexible geometry. Referenced elements might be difficult to identify and should be used carefully. Projection and intersection should be used to a minimum within the sketch. The drawback with projection is that it is using sub elements. If projection is used it should be applied on the whole feature not only a sub element. A feature *replace* will not be a one to one replacement if the new object does not have the same amount of sub elements.

It is important to plan the CAD modelling. It is necessary to break down the product to small units and make separate parts and complete the assembly later. Packaging the *Boolean operation* into bodies instead of selecting feature based selections, such as pad, gives a clean tree structure and makes it easier to replace.

The *assembly* Boolean operation should not be used. The solids go into each other and keep both solids, which results in wrong mass etc. *Union trim* provides a decent result after operation but it can be questioned if it is a stable operation. Recommended procedure is to select the face most unlikely to change in the feature. Another guideline is to choose what needs fewer objects to be selected. Union trim needs more manual work and the same result can be given by other Boolean operations. Union trim is not recommended for the ribs due to slow update processes.

Instead of selecting features, as edges of faces, when performing fillets it is possible to select Boolean bodies and create *intersection fillets*. Everything that intersects with the Boolean body gets a fillet. It is a benefit to use the intersect edge fillet since other parts that added to that Boolean later on will also get the same fillet. It is also a benefit if there is need of replacing a body and the number of edges between the bodies is not the same. If fillets should be at other places on the part that intersect with the Boolean, these have to be added inside that Boolean body instead.

Order of operations is important both within CATIA, but especially for downstream processes. Always apply *draft* first and thereafter *edge fillets*. The opposite way gives a variable fillet, which may require an expensive tool when manufactured. Create large fillets first and then smaller ones, otherwise the result will be triangular shapes which can be problematic to manufacture. Fillets created in the correct order also leads to fewer elements that need to be selected when creating fillets. The largest *offset* is the smallest radius, otherwise the radiuses will disappear. This can be controlled by an analysis of the surface (surfacic curvature analysis - type limited).

Join can be seen as a folder where surfaces or lines can be merged and CATIA will handle the elements as one. Checking connexity defines whether they fit together. Tangency checks can be used to make sure nothing is missed. With join it is possible to change direction so an offset will follow the direction of the surface. The direction of the join depends on the first element selected in join. If the surfaces are not connected properly, there will not be a direction of the join.

3.3 Specification of requirements

According to the internal and external search, the requirements for the CAD standard, in manner of both CAD and the affect regarding FE, have been summarised in table 3. An important requirement to fulfil is that the part should not only be dependent on experience in designing and analysing since this can differ a lot. Instead of requiring previous knowledge of the user, it should be assumed that every designer may not know. It should be easy to understand direction combined with very detailed explanations. CEVT are delivering against Volvo Cars, and are therefore required to follow the general guidelines of how to structure and design the model. Those requirements are therefore also on the CAD standard. To generate a CAD methodology the working procedure from Volvo Cars, which is implemented at CEVT, will be followed. The models will be created in the company's start-up files which have a specification tree that is recommended to be followed.

Table 3: Recommendation and what to avoid during CAD modelling

Recommendations	**Avoid**
Positioned sketch	Avoid B-rep/Sub elements (points instead of vertices etc.)
Rename and delete those from start-up part not being used	Unnessesary features
Functional areas joined as late as possible	Cleaning model during construction work, cleaning should only be done on input models
Use of geometrical set to structure	Over-constained products
Simple modelling	Infinite lines, curves, and faces
Common structure	No duplicated lines, curves or surfaces should exist
All elements need to be fully contrained	Avoid project since this causes mathematical errors
Geometries should be created with sharp edges and fillets should be left to the end	Avoid defining up to face
	Splitting exactly on elements boundaries

It should be easy to do changes in the CAD model and to be able to change the engineering properties of the object. It should also be possible to compare the material and dimensions, such as changes of thickness, changes of size and changes of material. This can be achieved through shape optimisation by using parameters. However, the guideline is to use as few parameters as possible. Moreover, standard parts of specific parts, such as clip houses, are needed.

The CAD standard should help to produce a model which can be re-shaped. It should be easy to make changes of mating parts or reference objects, such as replacing the A-surface. A way to detect changes, before the model is published, is to use dead (isolated) bodies or surfaces as a reference object. The benefit with this is when another body is updated the model will not be updated until the reference object, the dead object, has been replaced with the new version. When very large or small changes are made, it is easier to detect if the model has planned for this modification or to study how it will change the model.

The CAE department does not have any specific requirements on the CAD standard. Usually there is a lot of pressure to show results, and a mid surface creation should not be too time consuming since deadlines has to be held. The CAE department want a model that is easier to mesh, and might not even necessary to repair and remodel. In other words, it should be easy to create a mesh on a mid surface. This is easier when the features are not too small. If the small features, such as small steps, holes, chamfers and edge fillets, are done late in the CAD modelling process, then they can easily be suppressed. This is in conflict with current CAD methodology when a guideline is to place these close to associated parent feature. Another way to make the process easier for FE modelling is to have a model of good quality. This might be a problem when different designers or suppliers work with different parts of the model and it creates conflicts if they are not done in the same manner. Another reason is that it is not stitched correctly or double lines exist. A geometry check should be done before passing over the model. Volvo Cars has introduced the tool Q-checker, which makes the design follow the right procedure otherwise the CAD models cannot be published.

When the model enters ANSA, it is possible to have it divided into different objects. The benefit is that the different objects can be meshed separately, otherwise the mesh may not be distinguished into separate parts and would create a mixture of what the software thinks is the mid surface, for example a rib and a wall. If there would be a possibility, changes in CATIA should be easy for the old ANSA model to be able to detect.

Designers want the CAD standard to be similar to the procedure as what they already do. The common opinion is that the most important factor is not FE modelling, the most important factor is in fact that it should be able to be manufactured. So the solution should not be too complex or affect the work too much. If the mid surface should already be created in CATIA, this should not cause more problems or be too time consuming. If the mid surfaces were elicited already in CATIA, the process should be much more effective and simple in ANSA. The conclusion of the possibilities of transferring the CAD model to the CAE environment has been demonstrated into figure 20.

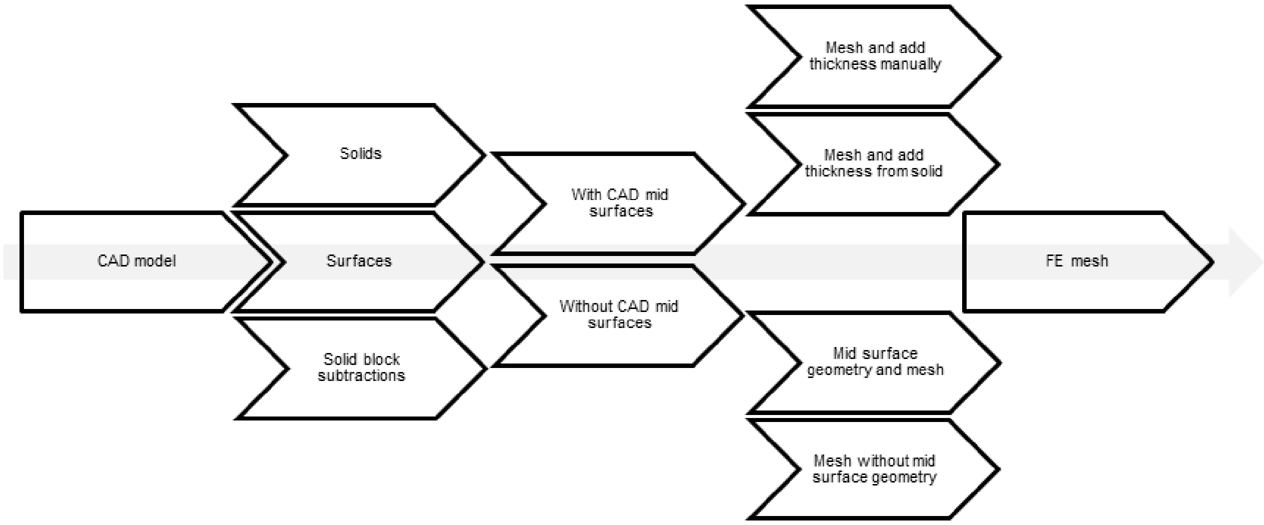


Figure 20: Transferring possibilities from CAD to CAE environment

4 RESULT

The following section presents the developing phase of the project and is divided in three sections. The first is concept generation where the result of pre-study models is presented. In addition to this, other studies are conducted to identify why the problem exists and how this can be solved. The previous researches together with the conducted studies are concluded as three case studies in development and validation. These will be evaluated and validated by different matrices. In concept design and development the most promising case study is presented as the CAD standard proposal.

4.1 Concept generation

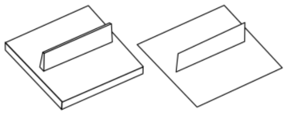
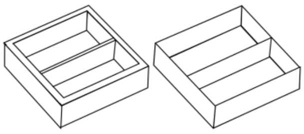
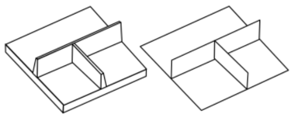
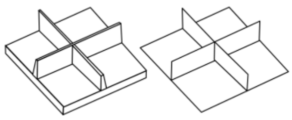
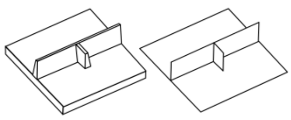
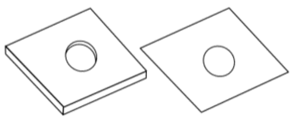
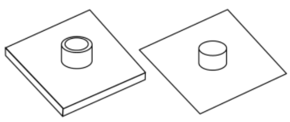
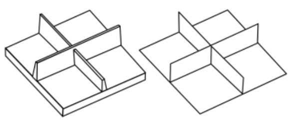
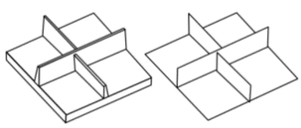
Three study categories has been chosen, in order to observe the problem of why plastic parts creates difficulties for the FE model depending on the way the CAD model was constructed. The categories are presented in this section; the small geometry study, the advanced modelling study and the geometrical requirements study.

4.1.1 Small geometry study

To study how different geometries affect the mid surface generation in ANSA some basic models have been chosen. The interviews resulted in some geometry that could be difficult to create mid surface for. The different geometries are therefore based on the result from the interviews together with previous research. These are shown in table 4 and rated on a scale of one to five, where 5 is good. These are varying in their complexity, components and also in the way they are built in CATIA. The purpose of the study is to observe which way to model would be more suitable in CATIA. The current modelling procedure is following the Volvo Cars guideline as well as the start-up file. The study will therefore also be following these. The plate also originates from a surface to represent the styling surface. The procedure will also follow the over-dimensioned geometries, to be trimmed in different manners.

The rib can be constructed with different techniques as well; by sketching the profile and using the pad function, creating a line and using the thick pad function, rib function, drafted filleted pad, or stiffener function. The over-dimensioned rib can then be trimmed. Thereafter draft angles and fillets are applied. Another way to create the design is to work in generative shape modelling, for the whole model. All the models can be seen in appendix C, with a brief description of what technique that has been used.

Table 4: Small geometry based study. On a scale of 1-5 (x=unsuccessful result)

GEOMETRY	Version 1 Profile sketch	Version 2 Line sketch	Version 3 Rib function	Version 4 Drafted pad function	Version 5 Stiffener function	Version 6 Trimmed surfaces	Version 7 Surfaces with trim solid	Version 8 Boolean operations
A Basic rib 	4	5	4	2	x	4	5	5
B Closed rib 	5	5	5	4	4	5	5	5
C T-junction 	4	5	5	x	x	4	4	4
D X-junction 	3	4	4	x	x	4	5	4
E Buttress 	4	2	4	x	x	4	2	4
F Hole 	2	2	x	x	x	5	x	3
G Boss 	5	2	x	x	x	5	5	3
H Step 	3	4	3	x	x	2	2	3
I Offset ribs 	3	4	3	x	x	3	5	3

The conclusion after studying the geometries in CATIA is that there are a lot of methods in which a model can be constructed; from defining the profile or creating a line, or which function to use. They are all relatively easy to create and there may be an interest in being able to create different parts of the A-pillar trim product with the same technique. The simple function as just using pad is creating a stable geometry and can almost always be performed. The customised functions in CATIA; such as the drafted pad, rib or stiffener function requires a specific environment to be able to be performed and it not as easy to do changes.

In CATIA, the version 5 could not be adapted for all the models since it need to be closed by walls, such as B and E. The drafted pad function did not fulfil the satisfaction and the other methods were more suitable. The B geometry models were all more or less successful.

The mesh in the FE models has been achieved by the casting mid surface function. Selected examples to demonstrate the difference of the CAD model and the result in FE model can be seen in figures 21, 22 and 23. The analysis of these was, despite of what was mentioned in section 2, based on how well the mesh follows the geometry. The purpose was to achieve an understanding of the connection between the CAD geometry and either the FE geometry or FE mesh. The least successful one was A4 of the A geometry study, as it does not follow the geometry in the important area to be analysed. Comparison of A and B it is easier to get a better result of a closed rib then the basic one. It is not as many open sides. The C model does not bring any complications either. The D model has some irregularities. For the last models major differences compared to the other models could be observed. E2 and E7 cannot mesh the outer line properly. This is the same for F1, F2 and G2. H1, H2, and H3 have some irregularities in their mesh, whilst H6 and H7 show bigger differences. H6 creates another shape in the interaction point and H7 does not follow the geometry correctly. For the geometry model I, the offset ribs have a diagonal bridge between them except I7 that follows the correct shape.

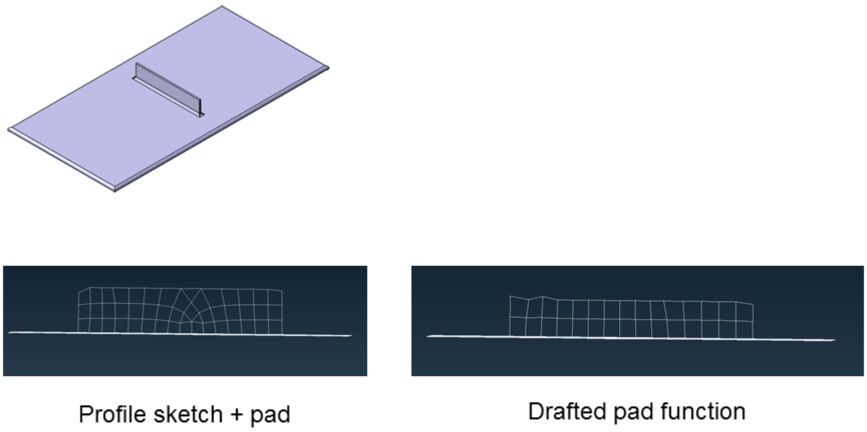


Figure 21: Comparison of different modelling techniques for basic rib (A)

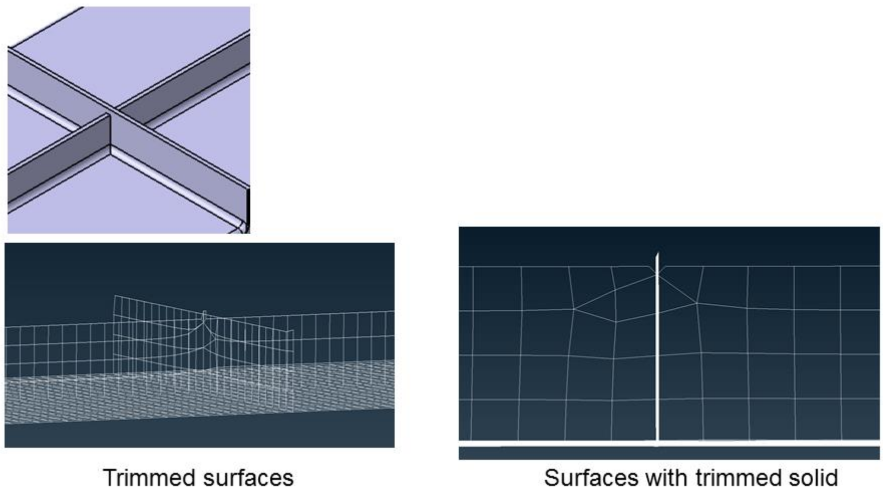


Figure 22: Comparison of different modelling techniques for small step (H)

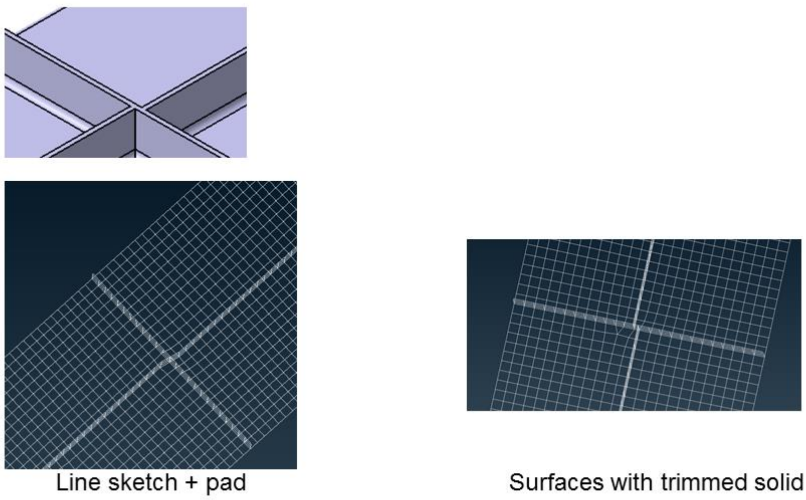


Figure 23: Comparison of different modelling techniques for offset ribs (I)

The mesh quality was also analysed by the tool in ANSA, called Mesh Quality. The distortion measures which was analysed are discussed in section 2. Overall the models failed mainly on jacobian and this occurred at the rib, edge, side of place, buttress, interface, and connection point. Failure of skewness and aspect ratio did also occur and this was mainly at the ribs at some connection or interface point. Model E and H were not successful. Version 1 was the least successful one. Despite the fact that B, C and D did not show complete failures, all the versions did demonstrate part failures at one point or another.

Depending on how certain functions and features are placed in order of their priority, changes can be observed in the FE results. The result may also differ depending on the mesh size and other input values. Changes in geometry which are made by modifying to larger geometry, edges and angles will work well in CATIA. The visual result in ANSA, however, gives a bad representation of the mid surface. Moreover, this gives no mesh quality failures. Adding an edge fillet on the ribs does not make any difference. Removing all the edge fillets in CATIA gives similar result in ANSA, the edges are still soften, and it gives no failures. Deactivating also the draft, the result is the same.

The H model proved to be the most problematic geometry to mesh. When H1 was reformatted with larger geometry, the results improved. However, when only the edges and draft were supressed, this still gave a bad visual result and gave jacobian mesh quality error in the interface. For the H7, the visual results were better with deactivated edge fillets and drafts, but still jacobian error exist at a mesh quality check.

Some observations of how changes of orders and functions have been accomplished. If the rib is split by the bottom surface and no union trim is cleaning the overlapping solids, the result in ANSA will as well be overlapping with mid surfaces. When instead using trim with the upper surface, and without union trim, the result in ANSA is a gap between the rib mid surface and the plates mid surface. In this case the rib and plate were done in different functions groups (bodies). If the union trim is added after the split, the mid surfaces are completed without overlaps or gaps.

If the edge fillets are done by selecting edges, the result in ANSA is good, for both the visual and the mesh quality of the model. Comparing them with selecting faces when conducting the edge fillets, it results in jacobian error in the edges.

The surface based model gave a decent visual result. An attempt to implement this in solid was made in order to use a reference line when performing the ribs with the pad function in CATIA for the problem model H version 2. The result still gave a mesh quality error, but with slightly better visual results.

The study has given information about how some functions and orders are possibly better than other. The rib which was created by just a line and controlled by a plane performs easily, and can be modified without difficulty. They are suitable for many situations. The solid modelling is easily performed in CATIA and gives a good result in ANSA.

In figure 24, the results of different levels of modelling complexity have been visualised, in both CATIA and how it affects the results in ANSA. The first two do not show many visual failures or any failures of their mesh quality. When adding a small step and offsetting the ribs, ANSA changes the geometry and failures do occur. The ones based on surfaces gave a better result, even though the surfaces were not loaded in ANSA. In this case, the surfaces were the actual mid surfaces. Overall the models failed mainly on jacobian error and this occurred at the rib, edge, side of plate, buttress, interface, and connection point. Failure of skewness and aspect ratio did also occur and this was mainly at the ribs at a connection point or at interface point.

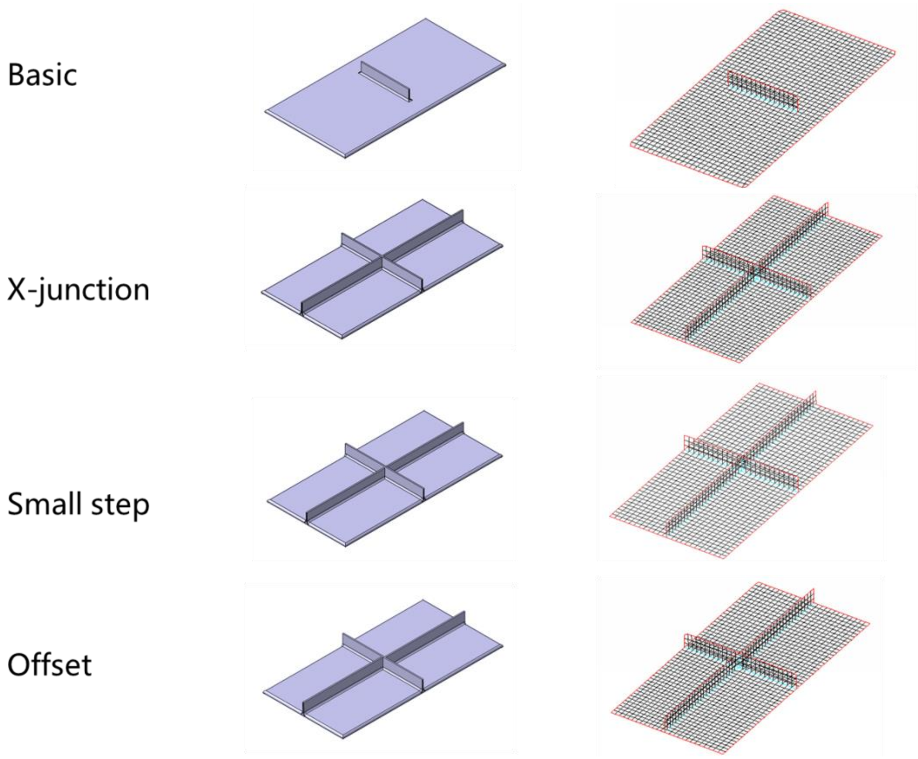


Figure 24: Complexity of the rib

ANSA can accept a solid and define a mesh on that. It can also import the mid surface if it is created in CATIA with defined thicknesses. Another option is to implement both the solid and the mid surface, and ANSA can create mesh on the mid surface and then use the solid to define the thickness. By extracting the faces of the solids and offsetting them to half the thickness the mid surfaces can be elicited from solids in CATIA. For the surface based models the surfaces was chosen to be in the middle to later add thickness from both sides of the surfaces. The result of the basic rib can be seen in figure 25.

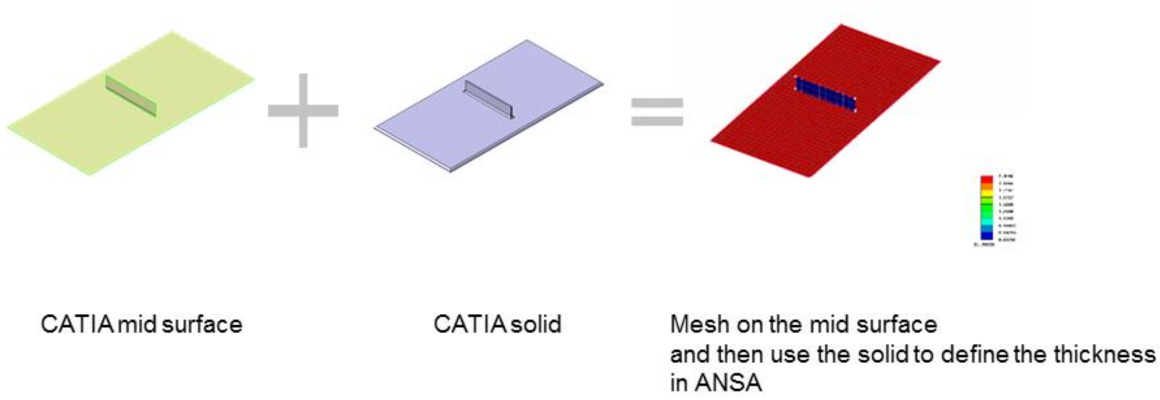


Figure 25: Mid surfaces and solids created in CATIA and loaded in ANSA

The results show a model that is trimmed and joined to the actual size, which is stable and accurate. It is necessary to be careful though the right choice should be done in CATIA, especially when it comes to a more complex part. It is also important to study the correctness of the result in ANSA, when adding the thickness from the solid to the mesh, to make sure that the actual thickness is being adapted.

In order to study the possibility of creating mid surface already in CATIA, alternatives have been elicited. One way is to first clean up and simplify the geometry; such as the edge fillets, holes and drafts. Then by using the extract operation of the solid and creating an offset to the middle of the solid. The extract from the solids can be selected in different stages of the modelling process; such as before edge fillets, before draft, or before different functions group are trimmed together. In figure 26, an example of the procedure is presented. The results of the different procedures depends on how the mid surfaces want to be received in ANSA; if the mid surfaces are already trimmed and joined to the actual size, or if the different parts need to be separated with a visual gap. A visual gap can be a benefit since it can easily be projected to the main body surface in ANSA.

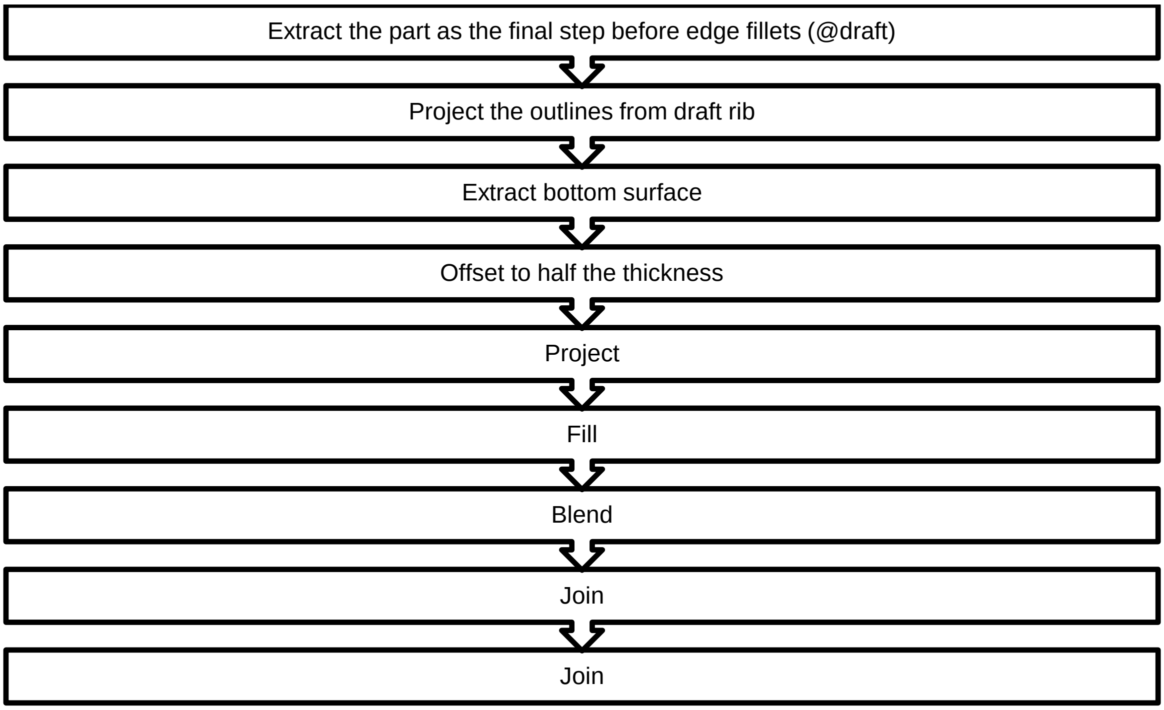


Figure 26: The procedure to extract mid surfaces

4.1.2 Advanced modelling study

In addition to the small geometry based study, a more advanced study has been done. This study has been done to observe the existing A-pillar trim that exists today and also other plastic components that exist at CEVT. These have been chosen to represent the different ways the designers may construct their models.

4.1.2.1 Comparison of different modelling techniques

To observe how some different modelling techniques works in both studies in CATIA and ANSA, three arbitrarily chosen models has been used. They are all plastic components and consist of ribs.

The first one is the A-pillar trim. This one has created geometrical sets with most steps renamed which make the process easy to follow. The geometrical sets consist of trimming parts and sketches that will be used to build up solids in part design. The last step in geometrical shape modelling is the oversized A-surface. This is created of so called slabs, which are done in the free style mode. The work in the part design workbench starts with adding thickness of the A-surface and then trimming it to the required size. After that different independent function groups are created and define different locations on ribs, clearing space for the sliders, and clips etc. These function groups are combined with Boolean operations. The ribs are created by several profiles in one sketch controlled by equations. A pad is then applied on the sketch. The rib clearing space is done with solid cubes and it removes material from the ribs. Edge fillets are done on top of the ribs and exist between the interface between one of the clip houses and the interface. When transferring to ANSA, geometrical failures occur as cracks and single coons. There also exist face errors, depending on problematic areas in the model. These exist next to the ribs and clearing space, as well as in the fillet of the bottom of the A-surface. In figure 27 some of these visual failures can be seen. Both failures are next to the ribs and cannot predict the correct mid surface of the main body.

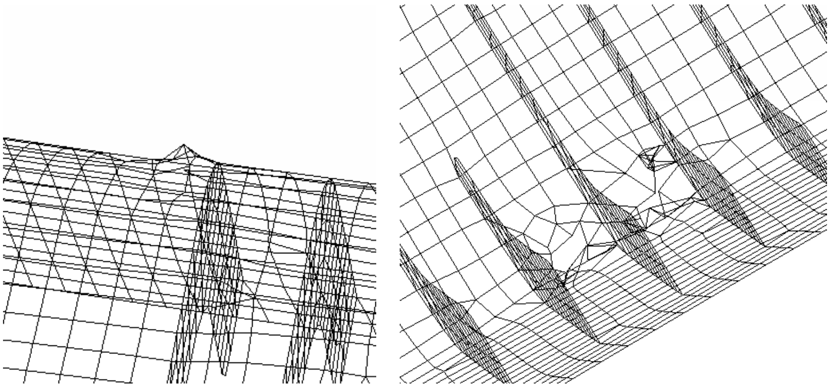


Figure 27: Visual of A-pillar trim

The next model that has been studied is the B-pillar trim. Compared to the A-pillar trim it is much more unstructured. This model has been created by more than one designer. First everything was conducted in only one body, and has later been structured so it should not have so many features dependent on each other. The model consists of some structuring geometrical sets. Ribs are created from dead surfaces to be filled and later multiple extract. In the part design workbench, the command closed surface has been used to create thickness. In part design different functions groups are combined with Boolean operations as union trim and assembly. Fillets are done on the main body. There are no fillets on the ribs but some exist on the top of some of the clip houses. In ANSA the visual is not good. There exist geometrical failures, needle faces, cracks, and single cons. There also exists faces errors in the FE model in some problematic areas. Compared with the A-pillar trim the failures are more major. In figure 28 some of the visual failures are shown. The mesh failures are found in the interface between the main body and one of the attachments. The attachment has been done by a precise sketch fixed to the main body. Failures in mesh are also seen where ribs and a lot of fillets are.

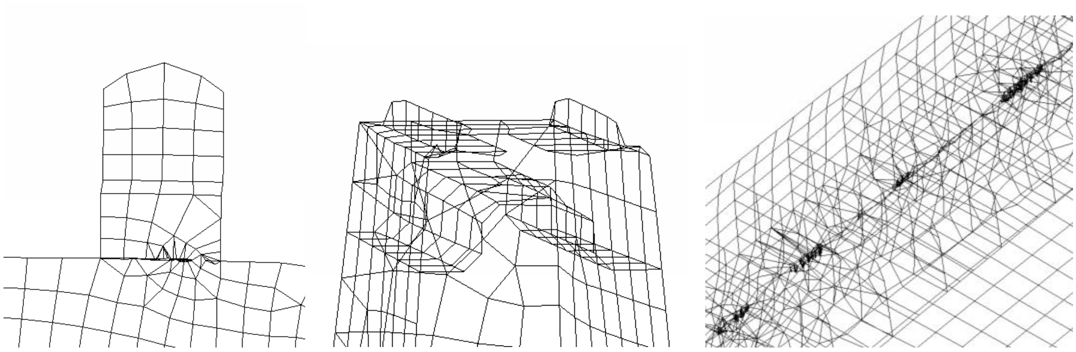


Figure 28: Visual of B-pillar trim

The third model to be studied is a bumper. The body is limited by geometrical steps consisting of surfaces, and is well renamed. In the PartBody large solid boxes are created in different function groups. These are trimmed against geometrical sets and create features by removing other Boolean bodies. Different function groups are, together with Boolean operations, combined with the assembly for the main body. Union trim is used to trim holes etc. Only half of the model is done and then mirrored. The ribs are created by only a line in a sketch and is then finalised with applying the pad command. The ribs are union trimmed with the main body. Fillets only exist on the main body and at the interface between the main body and some attachment points. The visual in ANSA is good. Geometrical errors consist in the form of needle faces. In figure 29, a good visual representation can be seen.

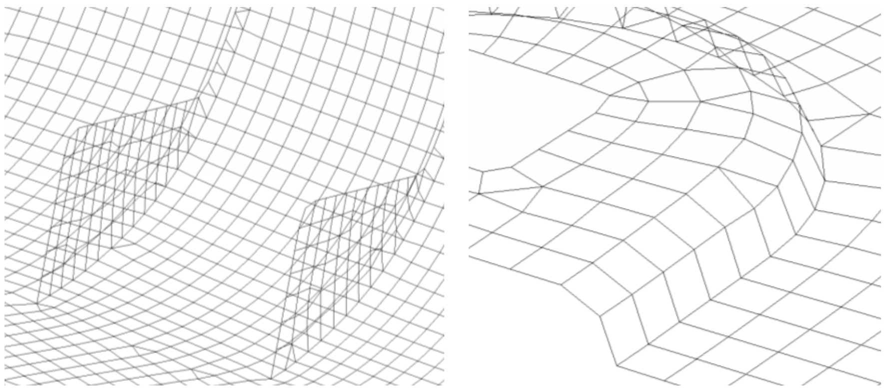


Figure 29: Visual of bumper

In figure 30, the results of the study can be seen. Represent the ratio between the numbers of failed shells versus all shells, for each mesh quality failures. Concluding this study, a well-planned model gives a good result in ANSA. The modelling procedure should as much as possible be planned through. With a good plan for how the finished part will look, unnecessary actions can be avoided, such as too many add-ons in the geometry. Complex areas where ANSA has problem to solve are the areas close to the ribs and where there is a curved geometry. The Bumper gives good results, both on a visual level as in the mesh quality.

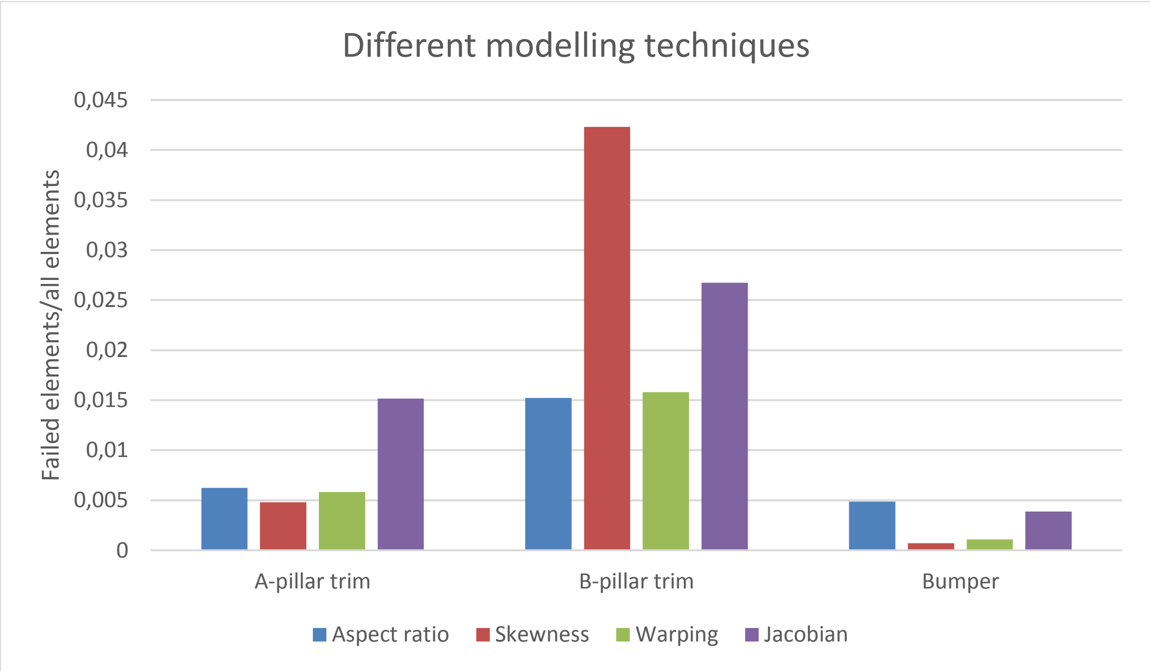


Figure 30: Comparison of result of different modelling techniques

4.1.2.2 Comparison of solid with and without historical reference

A comparison of the differences between a dead CAD model and a real one has been studied in ANSA. The purpose of this is to observe how much this will affect the work. The model that has been observed is the A-pillar trim. The dead model has been created by copying the part body with no link, so that it is created with no historical reference to the original feature. Both of the models gave failures in the same area. The average number of jacobian errors is larger for real solid, but the other mesh quality failures were much less for the real solid. In figure 31 the results of the mesh quality are shown. The quality is less good for the dead solid but the difference is not too big so any conclusions can be drawn.

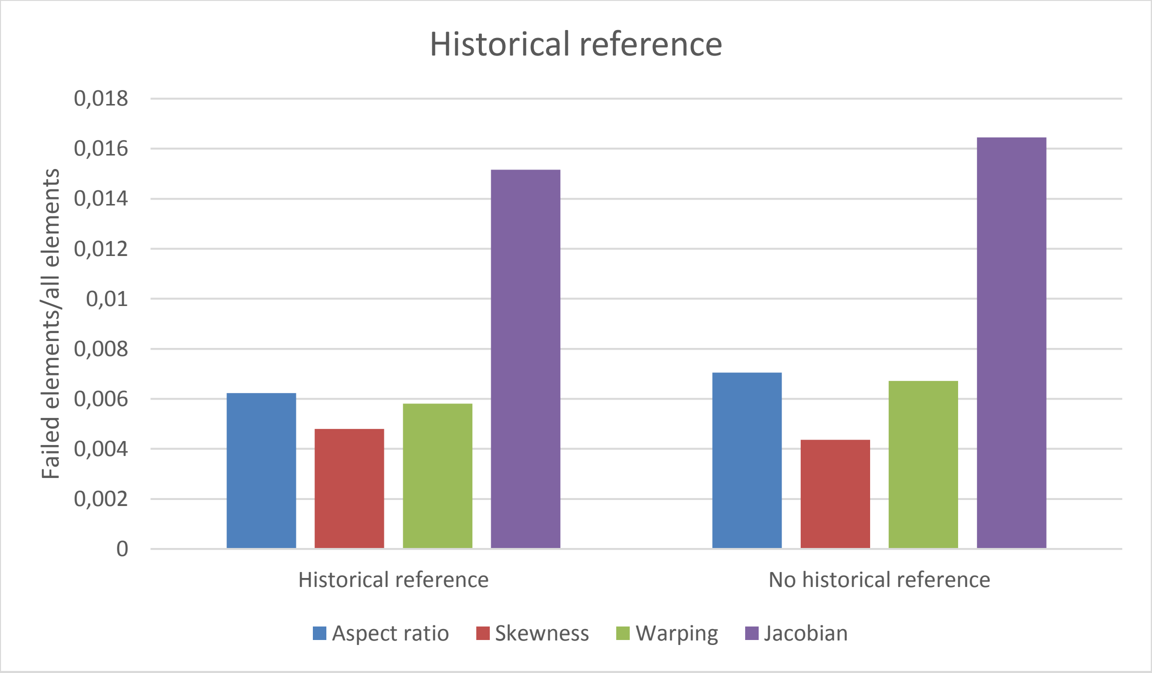


Figure 31: Comparison of result of solid without historical reference and solid with

The visual results can be seen in figure 32 with not too large changes compared with historical reference.

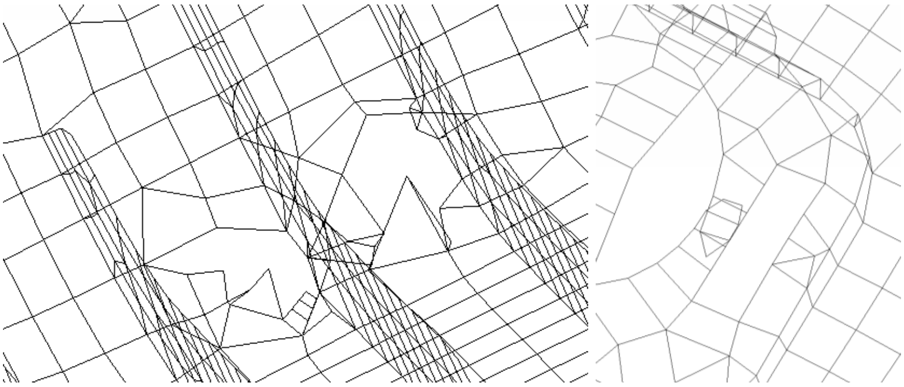


Figure 32: Visual result of mesh with no historical reference

4.1.2.3 Different translation format

A comparison of how different translation formats affects the results in ANSA has been done. The comparison has been accomplished between loading a model as CATIA, Step, and loading the Iges format. This was done to explore if using another translation format and more processes actual causes more error, or if a neutral format can make a more understandable model for ANSA.

For the Step translation, there were no faces or geometry failures as with the CATIA format. Errors did however exist in the mesh, but much less than the other. The mesh is following the geometry better than in the other models. This can be explained simply because some data has been lost. The step file is a so called a dumb format. Normally in the CAD software, when creating for example a hole, the hole is created by a fictional round object and the fictional object is subtracting the other object (Stevenson, 2014). In the CAD software, a mathematical face has been created. The Step file jumps over the procedure with the fictional object, and define instead the hole by saying that within that area there is no material.

In addition, the Iges format has been observed. The result is not accepted in ANSA due to the fact that the result is transferred as single coons and consequently ANSA is unable to read the file properly. This can be explained by the fact that Iges files are mainly developed for surface and wireframes models. For solid models it is much weaker.

In figure 33 the result of the observation can be seen. The conclusion of the study is that the Iges translation is not suited for transfer from CAD to a CAE environment. The step file may be used but should be done with cautiousness. Still, despite the result from this small observation, the recommendation is to go with the CATIA format since it minimises the translation steps.

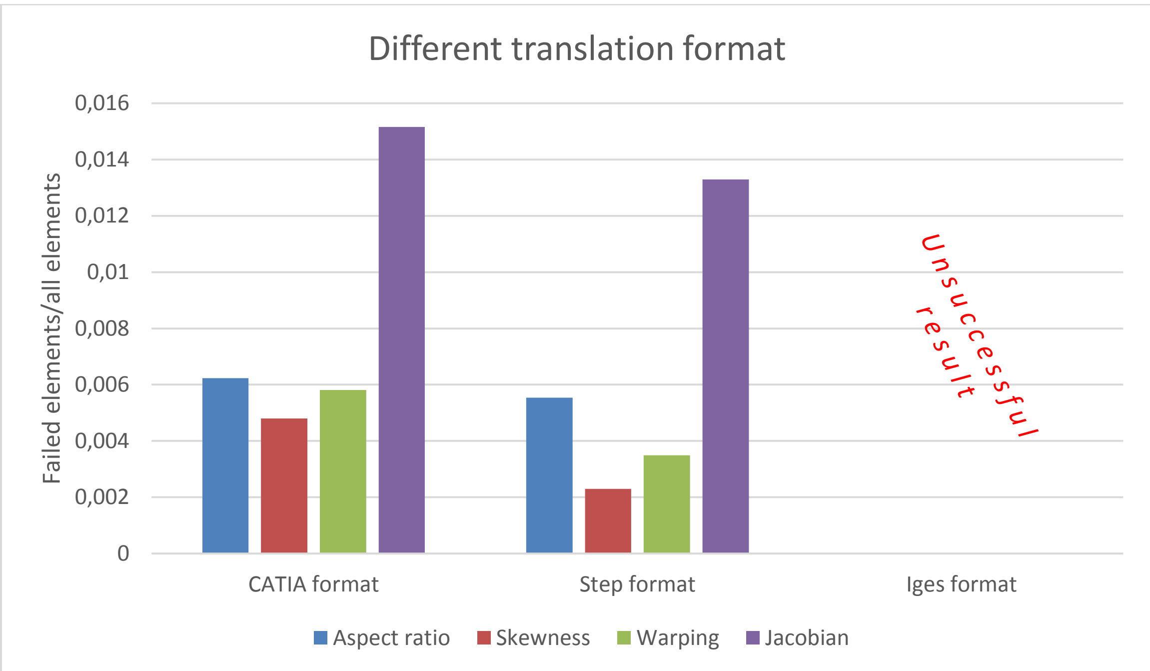


Figure 33: Comparison of result of different translation format

4.1.2.4 Different ribs creations

To be able to observe the problem, the A-pillar has been observed in different built-up stadiums by modelling. The models can be seen in figure 34. It can be observed that the failures of the mesh quality were larger for a more complex model. The curved shape of the trim made the mesh bad for the profile itself and the ribs. Part of this is due to the fact that ANSA have problems separating which lines are construction lines and which are aesthetic surface lines. If possible, planning for lines would make the procedure in CAE much easier. Models without edge fillet are better. It does not make any difference if draft and edge fillets are done before or after union trim.

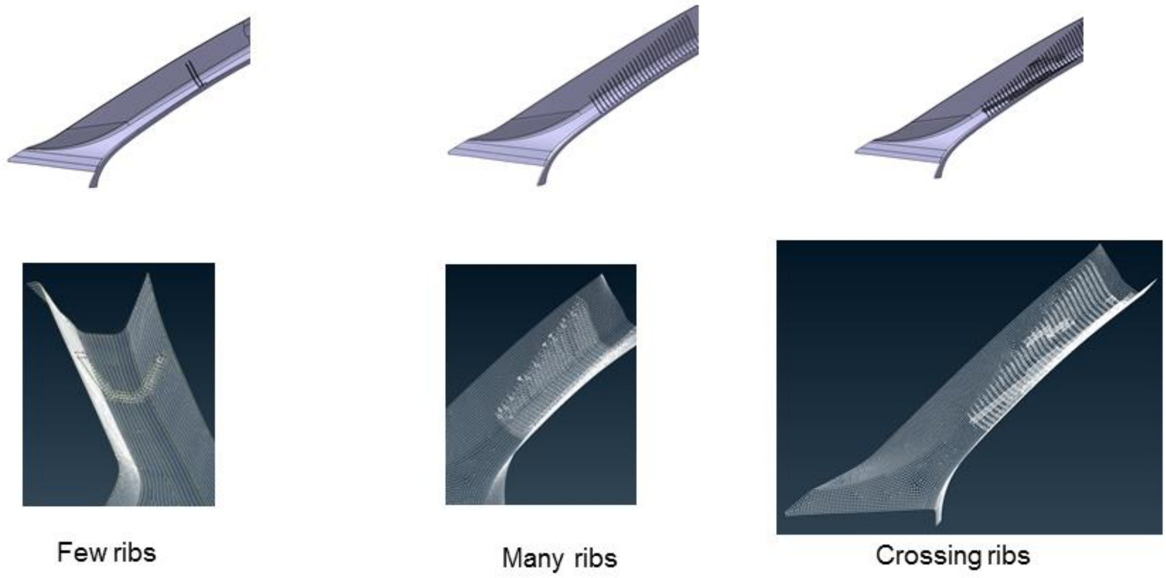


Figure 34: Different ribs creation

Some functions that made the process fail were the CATIA command fill and blend. The reason for this is because the model gets an unstable geometry due to both sharp and fading edges (IBM Support Portal, 2014). This should be avoided, since it gives the model gaps.

As with the basic geometry study, the surface based modelling technique helps the mid surface mesh to have a higher quality. The result in ANSA, with only a few ribs, gives no errors on ribs. However, failures do still exist at the complex area of the main body.

4.1.2.5 Different types of fillet

To be able to study the plastic component further, different types of fillets of the solid body have been performed, and are presented in figure 35. The first test was to try an A-pillar without any ribs. Despite to small geometry based study; this gave few errors in the mesh quality in comparison with the one with fillets. The second test was the A-pillar with ribs. This gave larger failure for jacobian. The third test was ribs with fillets by selecting the faces of the ribs and this increased the failures even more and also gave failures of skewness for the shells. Ribs with just edge fillets gave a better result with fewer failures. The last test was ribs with intersection fillet. This gives a fillet at the interface between Boolean operation bodies. The result had higher skewness and warpage, but lower jacobian failure ratio.

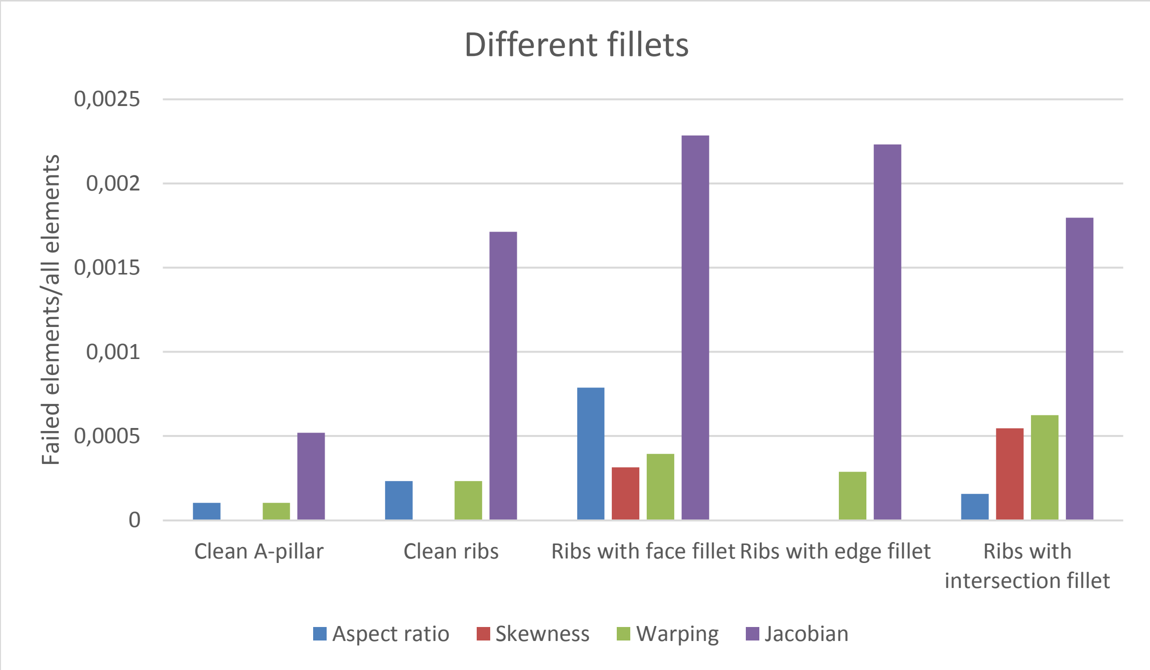


Figure 35: Comparison of result of different fillets

4.1.2.6 Comparison of different isoparametric curve flow

A study of the influence of the isoparametric curve flow has been performed. What is known is that it affects the building of the solid within CAD, but the question is if it also affects the FE model. For a part with bad isoparametric curve flow, the result can be seen in figure 36 and figure 37. Different curve flow can be seen between the surface patches. In areas where this is observed, were particularly closed to the ribs and in the clearance areas for the sliders. The largest problem areas for the mesh are where the isoparametric curve flow is also bad.

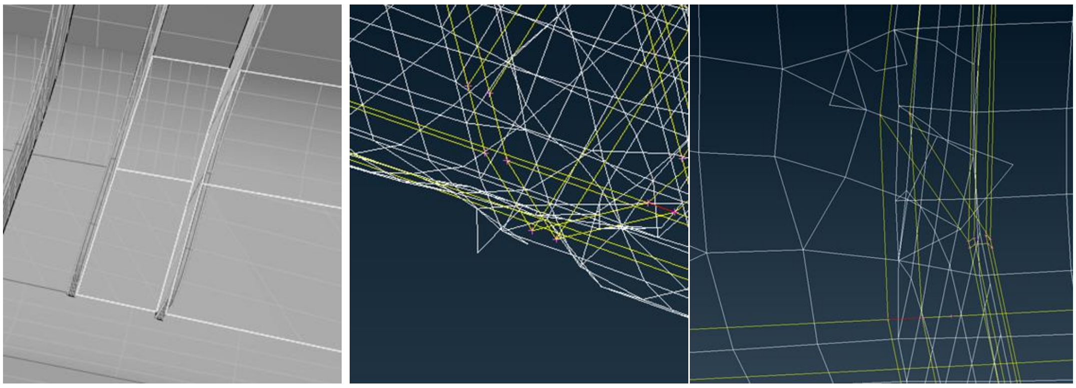


Figure 36: Result in ANSA from different angles from part of a CAD model with bad isoparametric curve

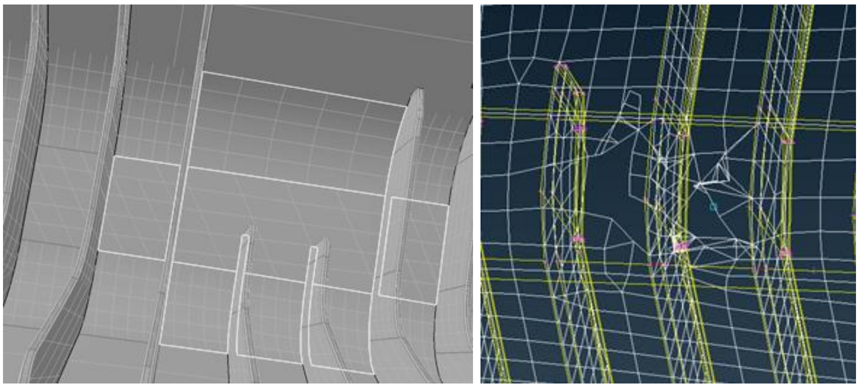


Figure 37: Another result in ANSA from part of a CAD model with bad isoparametric curve

4.1.2.7 Tolerance difference between the CAD and CAE environment

An observation study has been performed in order to investigate whether the difference in the tolerance setting can have something to do with the results, and is presented in figure 38. The reason to study this is that it is a known fact that gaps exist in CATIA and other CAD software but this does not cause any problems until they are translating to another format, such as to ANSA. The translation of the solid does not show any problem. But if other tolerance settings are defined in the pre-processor software, then the work of fixing and healing the geometry and the mesh creation will be done in that tolerance. The tolerance setting in CATIA has been within the requirement that both Volvo Cars and CEVT are following.

Two different tests showed different result. The first test between changing tolerance setting in ANSA to be similar to the ones in CATIA demonstrated improvements in the quality of the mesh. Whether this is due to that the tolerance between the two environments are matching each other or not, requires further investigation. However, they are not the focus of this project. The second test did not show any difference.

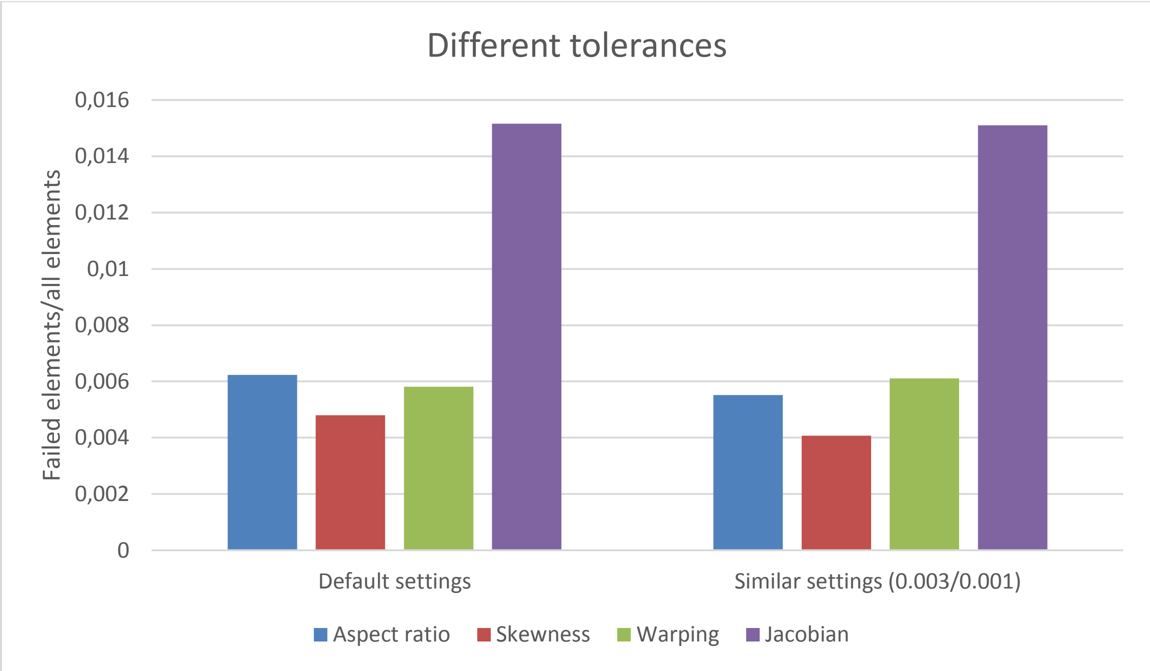


Figure 38: Comparison of result of different software tolerances

A visual result is presented in figure 39 and do not show any improvements.

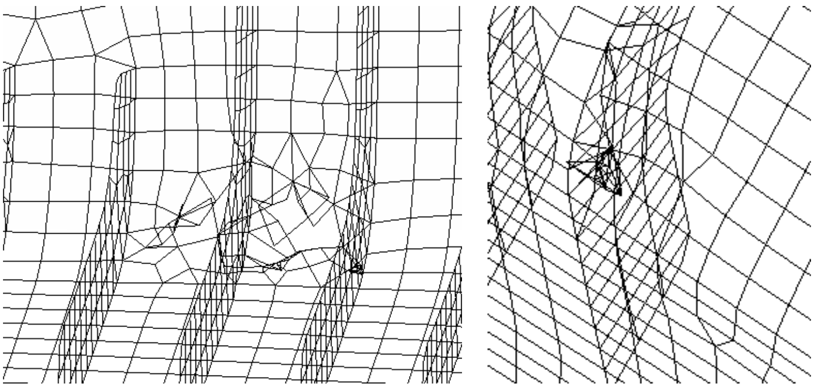


Figure 39: Visual result of mesh with another tolerance setting

4.1.2.8 Different mesh data

To observe how much the mesh data affects the results, different mesh settings have been applied. This has been done by comparing different target sizes of the mesh. In figure 40 a comparison of the target data for the lower range has been done with the upper range. What can be seen is that mesh data in the lower range gives a better quality of the mesh.

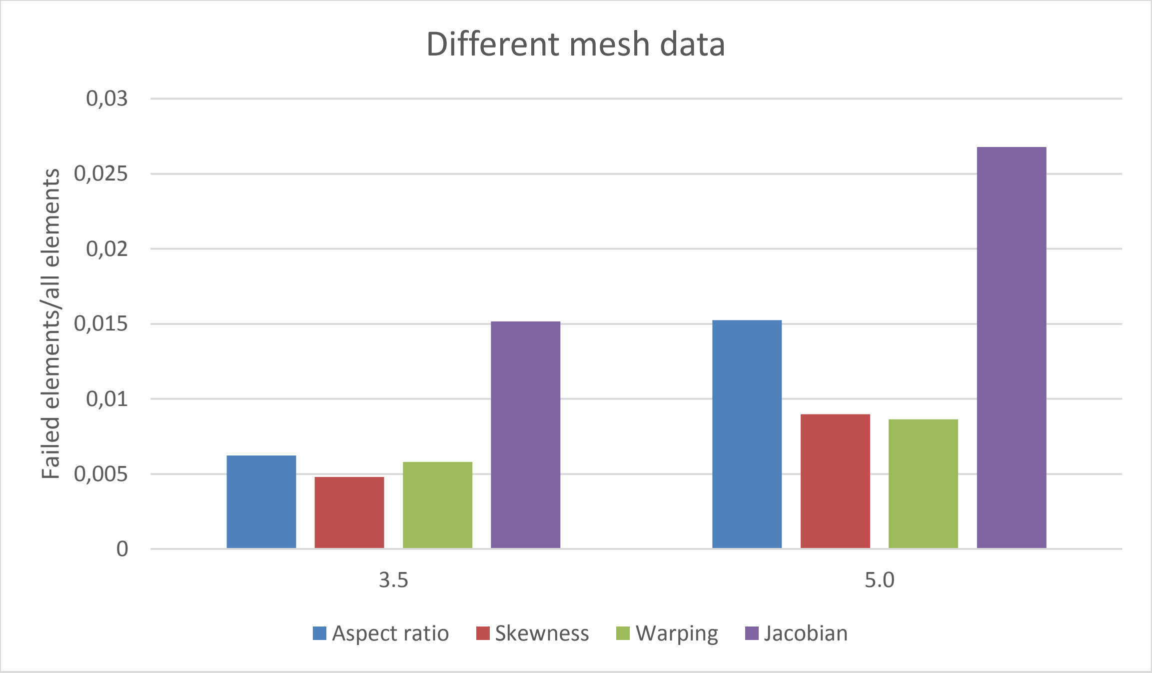


Figure 40: Comparison of result of different mesh data

4.1.3 Geometrical requirements study

To study the best ways to build the CAD model, one must not only study the tools that are used in CATIA, or how the translation is done. Indeed, the aesthetic of the geometry must be explored as well. The study is visualised in figure 41. During this study, the A-pillar trim has been studied through making different sizes of surface patches on the ribs.

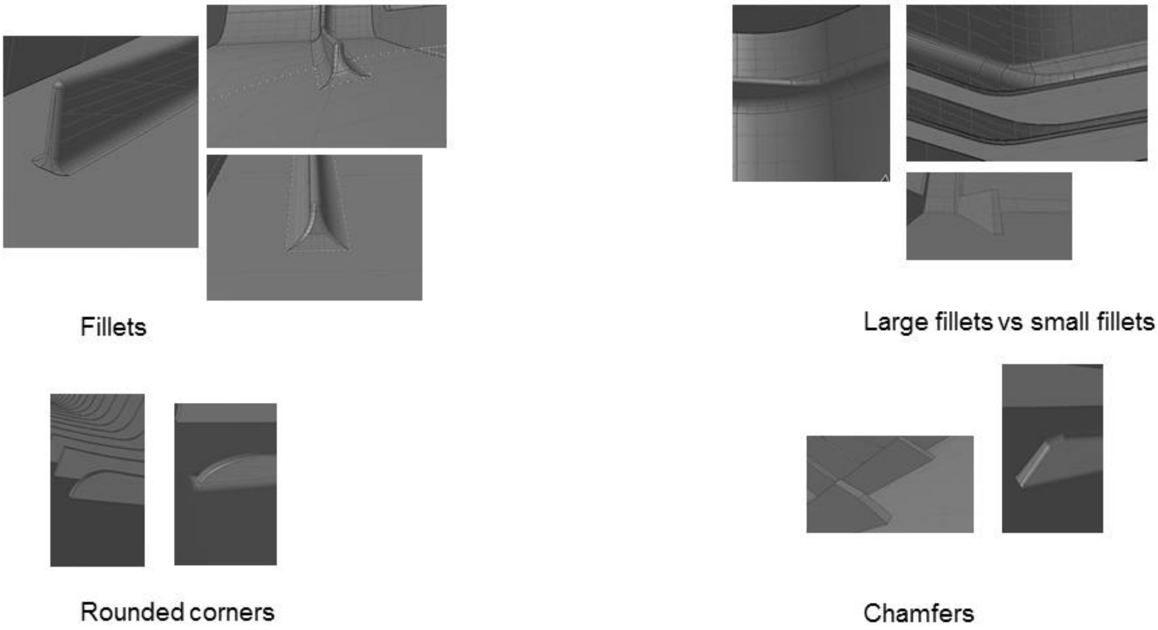


Figure 41: Comparison of result of different features

This study was executed to observe how the model can be done to suit FE modelling in best way. The result of the mesh quality can be seen in figure 42.

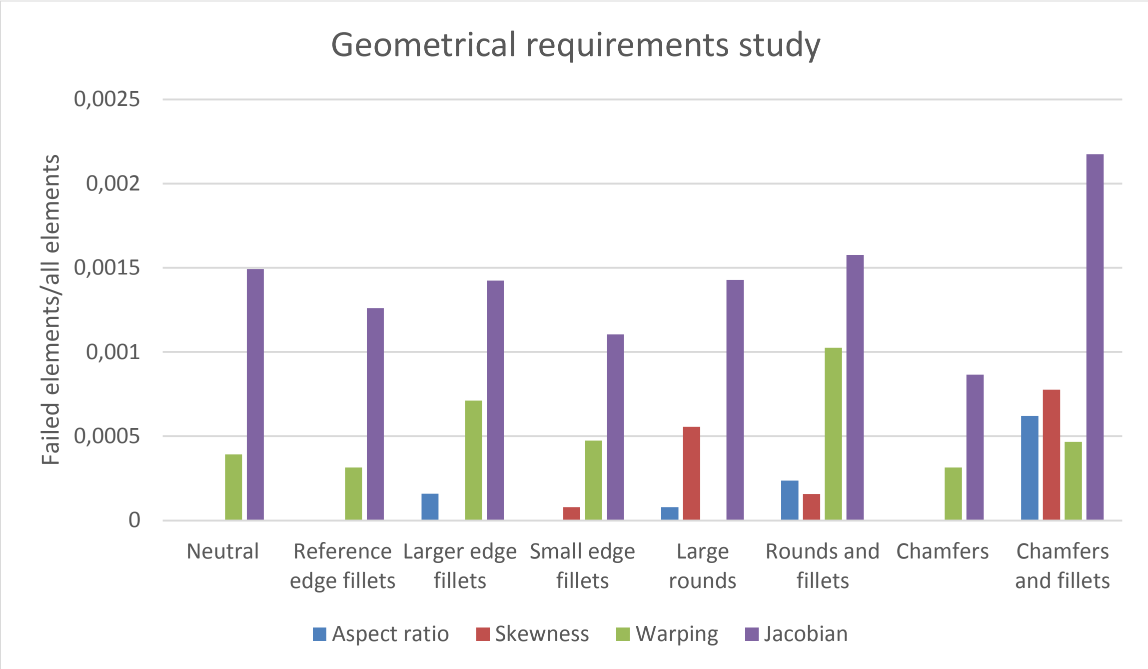


Figure 42: Comparison of result of different features

The conclusion after the geometrical requirements study is that the fillets on ribs cause problem with the mesh when they are not symmetrical on both side. The fillets are defined with the same parameter, but can become asymmetrical depending on the underlying shape, angle between the split on the ribs and the direction of the ribs. What can be seen is that triangular shapes, as a consequence of fillets, cause problems for the mesh. Larger rounds do not give any issues in the mesh itself, but when combined with the edge fillets, it creates difficulties. Chambers do not give any problem for the mesh. The fillets can be controlled by the designer to assure that the model will work for downstream processes. The patches should be observed to not be too small, sharp or short so it is possible to manufacture and to be manageable for FE modelling and other downstream processes.

4.2 Development and Validation

The conducted studies, together with the knowledge and inputs from the interviews, as well as the benchmarking and literature studies, have resolved in three case studies that will form the later on CAD standard. They are; Case 1 – Solid modelling (Part design), Case 2 – Block subtraction modelling (Boolean operations in part design), and Case 3 – Surface modelling (Generative shape modelling). Each case will either have the mid surfaces created within ANSA or they will be implemented in the design phase.

4.2.1 Case 1 – Solid modelling (part design)

After the development, the solid model will mainly consist of building the model of solid features created by pad and Boolean operations. The solid body will be trimmed by other surfaces of mating parts and other shaping parts. If working with oversized geometries, the model is easily modified and update errors can be avoided if functions are selected carefully. The modelling technique can be seen in figure 43.

Figure 43: Case study 1 modelling technique

4.2.2 Case 2 – Block subtraction modelling (Boolean operations in part design)

The solid block subtraction modelling is based on the CAD methodology that exists at Volvo Cars, called *Solid Advanced Inside Outside*, was also used at Saab. The technique minimises the error of gaps and is a safe way of building the model. An oversized block is first created and trough Boolean operations, an opposite tool is created that will subtract unwanted material. The modelling technique is demonstrated in figure 44.

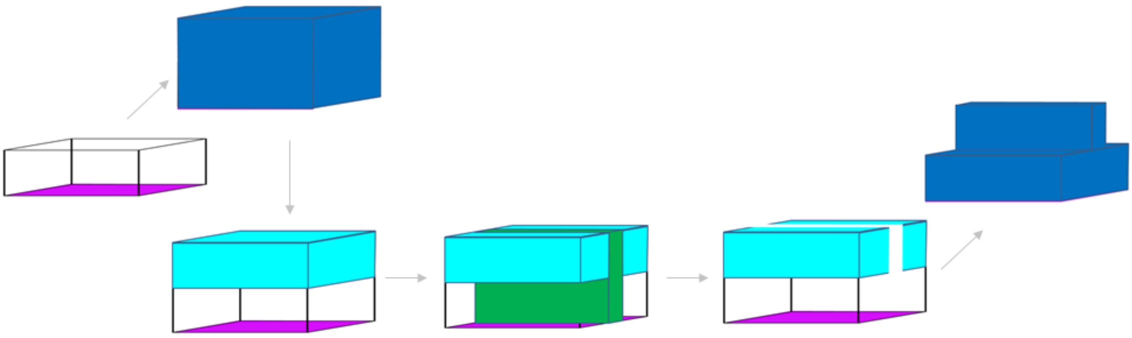


Figure 44: Case study 2 modelling technique

4.2.3 Case 3 – Surface modelling (generative shape modelling)

The model is created of a surface skeleton and thickness is added in the part design workbench. The surface based model is good while creating complex surfaces but is less effective at modifications. Modifying surfaces can create gaps that are difficult to see.

Depending on how the trim and join function is beeing used, the result can differ. In figure 45, the modelling technique for case study 3 can be seen.

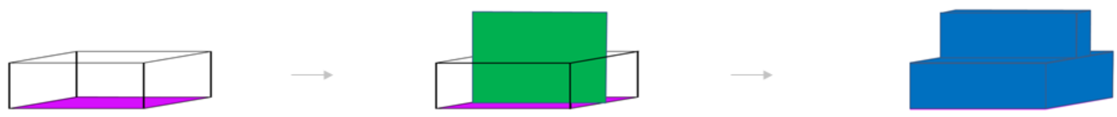


Figure 45: Case study 3 modelling technique

4.2.4 Case studies comparison

The three case studies has been evaluated and taking into the account the result from previous investigations. The result of the mesh quality for the different cases can be seen in figure 46.

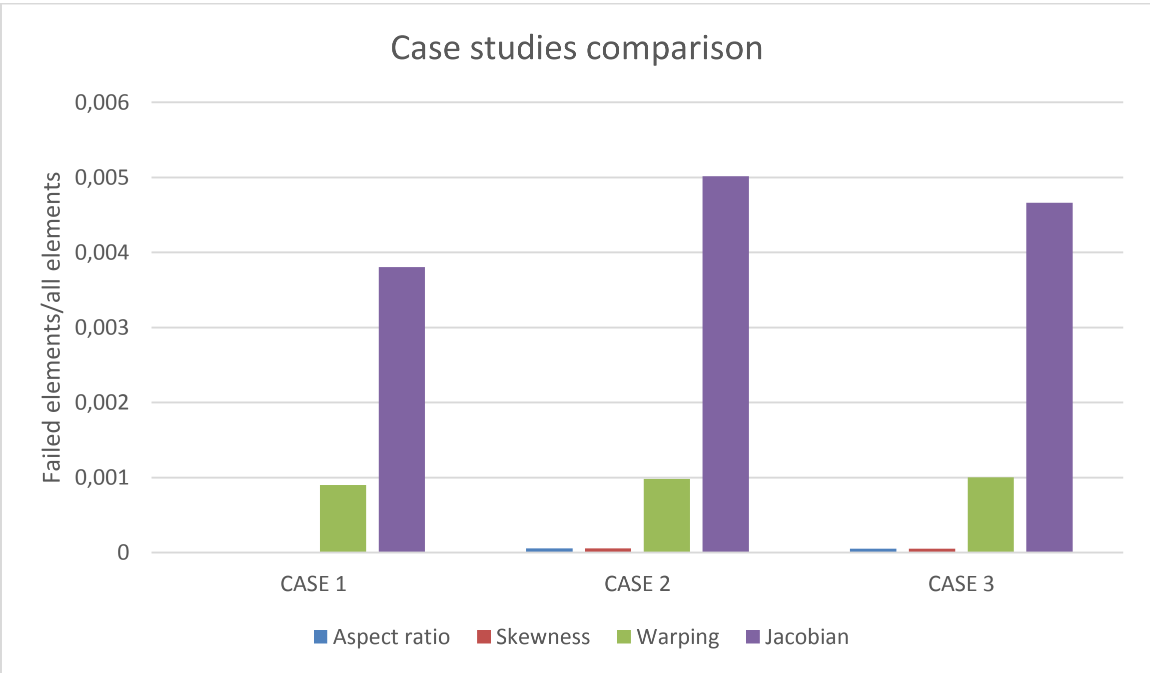


Figure 46: Comparison of result of the case studies

A solution matrix has been created to demonstrate the possibilities for how different features of the part can be done, and can be seen in table 5.

Table 5: The solution matrix of the case studies. On a scale of 1-5 (x=unsuccessful result)

FEATURES	CASE STUDY 1	CASE STUDY 2	CASE STUDY 3
A-surface	x	x	5
Main body	5	4	3
Ribs	5	5	2
Clips-house	5	4	2
Clips	5	4	2
Interface	5	3	2
Total summary	25	20	16
Continue	Y	Y	N

The conclusion after this comparison is that the solution is to begin the work in surface modelling and prepare the A-surface. The rest of the work is best done in solid modelling. The result in FE shows a clear benefit for case 1. Case 2 is unable to provide a proper visualisation of the geometry in the FE model. Case study 2 does also take longer time in CATIA to be conducted. Case study 3 gives a decent result in ANSA but the modelling procedure is less suited for a complex plastic part than the other cases. Drafted angles are easiest performed in solid modelling.

Why the results were shown better for case study 1 instead of 2 can be dependent on that in case study 1 the model consist of different features. Case study 2 is on the other hand; only consisting of one solid, since it is the tool side that has been created of different features.

A Pugh matrix has been conducted using case study 1 as reference and case study 2 as comparison, which can be seen in table 6. What has been taken into account is the modelling technique in CATIA, the similarity of the current working procedure, and how accurate the result is in ANSA.

Table 6: Pugh Matrix of the case studies

CRITERIA	CASE STUDY 1 (REFERENCE)	CASE STUDY 2
Personal opinion	0	0
Understandable	0	-
Complexity	0	-
Usefulness	0	+
How time consuming from CAD to FE	0	-
Easy to do modifications	0	0
Reflects on the calculation results	0	-
Sum +	0	1
Sum -	0	4
Sum 0	0	2
Total summary	0 (ref)	-3
Continue	Y	N

Case study 1 is easier to perform than case study 2. The model is effortlessly accomplished with CATIA user friendly commands. Commands such as thick surface and pad give an accurate result in ANSA. Case study 2 takes a longer time in CATIA to be conducted and do not give as accurate result in ANSA. The ribs are not as well defined as for case study 1. This can be dependent on that in case study 1 the model consist of different features. Case study 2 is on the other hand; only consisting of one solid, since it is the tool side that has been created of different features.

4.3 Concept design and development

After the concept synergy method observation, the choice has been to investigate further case study 1 and to conduct the CAD standard for this one. The CAD standard proposal can be seen in appendix D.

The methodology is an already existing technique. Therefore it fulfils the requirements to be easy to introduce and not affecting the current process too much. The difference is that it has been proved to be one of the most suitable modelling techniques for later work in FE modelling. The methodology described in the proposal is a combination of different designers way to work. The modelling technique that has been suggested in the CAD standard is following the practice of using non sub element and creating a logical work flow. The reason for this is to make it easy to modify and replace an element within the model. As requested it is also following the existing CAD standard. The guideline is to divide the work into surface modelling and solid modelling. What have been observed during the conducted studies is that surface modelling is most suited to work with the A-surface, whilst solid modelling is suited to create the rest of the part. To construct a model with oversized features both in solid and surfaces it overcomes the problem with gaps and cracks. This is due to the fact that any gaps or discontinuities are translated to the solid.

The proposal recommends constructing the model in the start-up file. Here the designer is guided to create a good skeleton, parameterise and create independent function groups. The designer is also guided to create the right construction in the right folder. These are done by identifying certain functions areas of the model and to define these in different folders, such as ribs, clips and clip houses. In surface modelling these folders are called geometrical sets and sketches should be created in these. These sketches should be positioned against a point, line and a plane. The sketches should only consist of simple geometry. For example for the ribs, it is recommended to only create a line. These lines are then recommended to be constrained to planes. The designer is then able to modify how close these ribs should be by changing the parameter which is controlling the planes.

The modelling techniques that have been selected to be compared have been translated to not only work for the main body and the ribs, but will be implemented for the clips, clip houses, and interface attachments within reason. Requested is that these attachments can follow the tool direction as much as possible. To be able to withdraw material from the inside of these fixtures, sliders are included during the manufacturing process. Therefore these additional bodies should be preceded in the direction of the slider.

Depending on the simulation in FE modelling that is going to be applied the geometrical requirements may differ. It is recommended to always have a good dialog between the departments. The CAD standard is a proposal and since it has been developed for the A-pillar trim it may, in some aspects, be different for other plastic parts. These guidelines are dependent of its geometry and also which application it has. The geometrical requirements are based on the previous study during the project, knowledge gained from the company, and literature research. In complement, the geometrical requirement study has been done to confirm this.

5 DISCUSSION

The process selected in the beginning of the project has been followed. The aim of the first stage was to find out what the problems within the subject consisted of. Implementations of the research together with the basic study were thereafter conducted to iterate against a more realistic solution. The research that has been studied has come from a wide perspective and can be seen as accurate. The sources used are from literature, articles, software webpages, and interviews with employees, consultants, adjacent companies, and pre-processing software company. The studies that has been conducted has been to prove what theoretically causes errors and actually do cause errors. The studies were also executed in order to explore the differences between various modelling techniques, transfer options, and pre-processor implementations.

The selection method endeavoured to compare the procedures in modelling in CAD and to observe how it affects the result in pre-processing. What haves been observed is that for plastic parts it is preferred to model with solids. The result is both safe in CATIA and other downstream processes.

The work has been confirmed during the process with people within these areas. Due to the strong relation between CEVT and Volvo Cars, not only people at CEVT have been able to give their feedback but also people at Volvo Cars. The work intends to be implemented into the process at both CEVT and Volvo Cars. To understand the requirement from Volvo Cars better, a course in CAD Advance was conducted; which covered techniques within solid and surface modelling, and the Volvo way.

The results depend on how well the A-surface is performed. The continuous development of the model which has the surface as a foundation and the solid will also be affected. The isoparametric curves are recommended to be toggled on. It is strongly recommended to do an analysis of both the surface and the final solid. The model should be clean and flexible modelled. The geometry should follow the restrictions that have been set in the CAD standard which will give better results in the downstream processes. During the design phase, all geometries are not strongly defined and could therefore be selected by the user. When that option occurs; the geometry should follow these FE modelling requirements. In the FE modelling requirements the CAD model is required to not consist of too small features and surface patches. The result relies on the translation process as well as the data defined for the mesh in FE modelling.

A critical study of the result shows that many of the studies during the project have been to change much of the designers work. The process to change this routine is not simple, and will with large possibilities not be accomplished for all of the designers. To have an automatic solution that controls the geometry of the CAD model before the transfer to FE modelling is a better option.

The correctness of the study can be questioned. The observation that has been conducted has been achieved together with facts from the industry but also from previous research. Even though, these specific observations are not done on the level that it actually prove that this is the only answer. For different environments, criteria, and components another result might be given.

The main focus has been on the procedure in the CAD environment. The study of the FE environment focused on finding out what the problems are and to find the corresponding reason in CAD. The focus has therefore not been to create advanced models in FE, only in CAD.

The project has resulted in a modelling procedure that does not include mid surface creation in CATIA. This is due to the fact that the result from the conducted case studies proved to be an accurate result. If mid surfaces, however, were elicited in CATIA, the transfer would be easier since a lot of difficulties would already been dealt with. Moreover, the designer does not possess the knowledge of which features are non-critical and should be suppressed, or which are critical features and should not be suppressed. The meshing will be improved when removing small features and small dimensioned surfaces. In general these do not affect FE analysis for given load and boundary condition. The problem with the mid surface creation is not supposed to move from one department to another.

6 CONCLUSION

The research questions that were defined in the beginning of the project have been answered. The CAD model influences the FE model by the geometry itself and the modelling technique that has been used.

6.1 Research question 1: How does the CAD model influence the FE model?

The conclusion for the first research question that can be drawn from the transfer between CAD and CAE environment, is that the CAD model is too complex, poorly defined and consist of corrupt geometry. Even if some features are functional in CATIA, problems may occur in the downstream processes. These features are fillets, smaller holes, and small edge patches. They can affect the quality of the mesh and also constrain it from being able to produce. This can be solved by suppressing smaller edge fillets and sliver faces. Holes and smaller steps can also be removed to improve the mesh. To be able to detect unstitched solids and surfaces and make them clean will also improve the mesh. A clean specification tree will remove unnecessary features that may not affect the CAD model visually, but may affect the FE model.

6.2 Research question 2: What are the requirements that need to be fulfilled?

An answer to question two is that the requirements that need to be fulfilled are to have tougher controls during the whole development process. Communication in the form of feedback should be saved and implemented in future processes. Moreover, the A-surface should consist of good quality since it will affect later work. Before the CAD model is published, it is recommended to follow the CAD standard and the geometrical requirements in order to insure a good quality to the downstream processes. When following these requirements, flexible construction needs to be taken into account, so the model does not need to be reconstructed when changes have to be done.

The transfer to the CAE department should be of an actual CATIA file and nothing else since it adds a translation step. The elements should be on the smaller side of the range and should be controlled so it follows the geometry in certain critical areas; such as the ribs.

6.3 Research question 3: What common construction principle can be utilised for different features?

The research question three explored which construction principles in CAD that could be the same and which needs to be different. The basic solid commands are both easy to create in

CATIA, and give a good result in the FE model. With a good surface that form the main body the rest of the part is best accomplished with modelling with solid features. The surface should be analysed before starting the modelling work so it does not contain holes, overlaps or radius that cannot be offset. The surface should also have a good isoparametric curve flow since this is translated to the future work with solids and may also affect the transfer to the FE model. In FE modelling it is a benefit to have as much of a seamless interface as possible. Lines created of different surface patches and from solid operations (pad, edge fillets, chamber etc.) will be seen as different areas and may disturb the meshability.

Solids are recommended to be created with the command pad and to be based on a simple and robust positioned sketch. The sub elements in the sketch should be constraint to a skeleton that makes it easier to control from the main 3D view in CATIA. The sketch should be positioned against a plane, point and line so it can easily be modified. The part should also be done in function groups and combined with Boolean operations. The fillets and draft should be placed outside these groups so the solid body can be replaced.

The differences are that diverse techniques should be applied for surface and solid modelling as just described. Another difference is that it is recommended for the company to create a library for the clips so the designers do not need to spend too much time creating these.

6.4 Conclusion of the most important results

The modelling technique that has been suggested in the CAD standard is following the practice of using non sub element and creating a logical work flow. The reason for this is to make it easy to modify and replace an element within the model. Also to make the process easier for next person who will work with the CAD model.

Surface modelling is most suited for creating the A-surface from the styling surface. The A-surface should be analysed before starting the modelling work so it do not contain holes, overlaps or radius that cannot be offset, since the quality is translated to the solid body. Skeleton, such as points, curves and planes should be defined with help of parameters that can be used to subtract or expand the model. Sketches should be conducted in different independent geometrical sets in the generative shape modelling workbench. These geometrical sets should be describing a certain function or area of the part, such as ribs, clips and clip houses. The sketch should be positioned against a plane, point and line so it can easily be modified.

The rest of the part should be constructed in solid modelling. The sketches created in the geometrical sets should be used to create solid features (for example with pad), and should be constructed in different bodies depending the geometrical set. These bodies should thereafter be combined with Boolean operations such as add and remove. The fillets and draft should be placed outside these groups so the solid body can be replaced. Very small surface patches due to dress up features, such as fillets, should be avoided. Unintentional design, such as creation of small steps or offset between intersecting object should also be avoided.

The CAD model should be cleaned up and unnecessary features should be removed before releasing to downstream processes. The CAE department should catch the CAD model and deactivating certain features, such as fillets, smaller holes, and small edge patches before entering the work in FE modelling.

The conclusion of the most important results that have been achieved during this project have been summarised in table 7.

Table 7: Conclusion of the most important results

Surface modelling is only suited for the creating the A-surface from the styling surface	
	The A-surface is important to be of good quality
	Skeleton, such as points, curves and planes should be defined with the help of parameters
	Sketches, such as for the design of ribs, clips and clip houses, should be conducted in geometrical sets within generative shape modelling
Solid modelling is suited to create the rest of the part	
	Sketches that were conducted in geometrical sets should be used to create solids
	Skeleton and surfaces should be used to control the size of the solid model in part design
Improvement of the CAD quality can be achieved	
	A clean specification tree will remove unnecessary features that may not affect the CAD model visual, but will create problems for the FE model.
	A model with a seamless interface is a benefit for ANSA; since the aesthetic lines are creating conflicts with lines of a feature
	Working with oversized solid features and planning the model and not try to reconstruct when it might be easier to do it right from start.
	Features of the part should be done in different bodies combined with Boolean operations, such as add and remove.
	Fillets and other dress up features can cause problem for the FE model if they are asymmetrical on an object, such as the walls of the ribs.
	The patches should be observed to not be too small, sharp or short so it is possible to manufacture and to be manageable for FE modelling and other downstream processes.
	Objects that are crossing each other with a small step or offset cause problem and should be avoided
	The fillets and draft should be placed outside these groups so the solid body can be replaced
The translation to FE modelling can be improved	
	To improve the identification of different objects in FE modelling the CAD modelling should not consist of add-on.
	Fillets, smaller holes, and small edge patches should be suppressed before the meshing starts
	Detect unstitched solids and surfaces and make them clean will also approve the mesh.
	The CAE department can be able to catch the CAD model, and make the model simpler, with deactivating certain features, before entering the work in FE modelling.
General recommendations	
	It is recommended to add a communication portal regarding the transaction between the CAD and the CAE departments. To follow up these subjects the work can be improved for both teams, and do not have to be reworked for each time.
	To be able to implement the work correctly it is important to introduce training session to assure that the CAD modelling is done in the same manner and in the correct way.

7 FUTURE WORK

It is recommended for CEVT to include a check that controls the model after certain specifications. The naming should be right, the tree specification should be followed, and features that should be hidden should be hidden to mention a few. This can be done with the Q-check which Volvo Cars has already implemented. Since CEVT should be delivering models according to Volvo Cars requirements it is good to implement at least the same criteria. In addition, it is also recommended that within the quality check perform a geometric requirement check. The user can have the option to either accept, remove, or modify a detected design feature such as a small step, faces, gaps, slivers that can cause difficulties in the downstream processes; CAE, manufacturing, rapid prototyping etc.

It is recommended to add a communication portal regarding the transaction between the CAD and CAE departments. To follow up these subjects the work can be improved for both teams, and do not have to be reworked for each time.

Another recommendation is to add a script that controls the renaming of Boolean operation depending on the name of the function body that is being added. This makes it easier for the one that is creating the model. If possible, the renaming should be added to within the function groups as well, so that all points, planes, curves or solids are included in the name of the function groups. This makes it easier to track.

It is recommended that different parameters should be transferred to CAE; material, general thickness, coating. It is also recommended to include the thickness of the ribs, and also the smallest thickness of the whole part.

Further on, exporting the tree structure makes it easy for FE modelling to get the solids, surfaces and so on with the correct name instead of the default.

Some further recommendation is to create a library with power copies of clips and clip houses. With this already defined, it is easier to make smaller adjustments to make them suitable to the part. With these created they can not only be adapted for the A-pillar trim but also for other plastic components. The power copies need to be stable and robustly built to not interfere with any update cycles.

To be able to implement the work correctly it is important to introduce training sessions to assure that the CAD modelling is done in the same manner and in the correct way. Different industries and companies have different views of what is the correct way and therefore designers need to be guided to follow the CEVT way.

Further recommendation is to continue the study of how isoparametric curve flow affects the solid and how much it actually affects the result in FE. Also continue the study with how different translation format may affect the result in FE. Further work is a to implement a method of how the CAE department can be able to catch the CAD model, and make the model simpler, with deactivating certain features, before entering the work in FE modelling.

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APPENDIX A
INTERVIEW GUIDE
Introduction
- A presentation of me and reason to interview– master student in product development that are doing the master thesis at CEVT to make the step from CAD to CAE easier. I'm studying the injection moulded a-pillar interior trim part and will set up CAD standards for this.
- Reason to interview – Master thesis Chalmers, Chalmers.
- My aim needs mapping to find critical driving forces – to find out the current working procedure and to find out areas that can be improved.
- The interview will be used in the report to set a base for requirement specification, to start the concept generation and be used as a research source.
- *Ask if the person*
    Have enough time
    If it is ok to record
    If it is ok to cite/quote the person as a source (ask if the person would like to confirm that the source is correct in the thesis)

Knowledge about the company and the work
- Can you describe your role/position in your company?
    - How long have you worked with this?
- Describe your development process from early concept stages through production
    - How did you come up with how to do the development process?
    - How many design solutions are under process at the same time?
    - How is the part developed?
    - Can you name some problems that may occur

Experience with the process
- Describe your development process from early concept stages through production
    - How did you come up with how to do the development process?
    - How many design solutions are under process at the same time?
    - How is the part developed?
    - Can you name some problems that may occur
- How is the process developed
- What is the most important to think of
- How do you plan product specific
    - I.e. plastic products
- How do you get everyone involved
- Are you planning for avoiding complication and errors
- Are you planning to improve quality of the model
- How are standards being verified?

CAD/Designers
- Can you explain how you are building your plastic part model
- Surface or solid modelling?
    - Surface common for automotive interior?
- How do you avoid errors?
- are you also designing for production
- How does supplier affect your work

        - Constrains from them
- What are the technical properties/parameters you consider for you in the process?
    - Can you indicate the most important one/ones?
- Describe what constraints the cad models and how do you solve that
    - Interface for adjacent parts
    - Optimum trim offset (meet the head impact requirement – FMH
    - Maximum interior space – to not obstruct the view
    - Side airbags? – in case of side impact or rollover impact situations
    - Fastening
- How do you plan for changes and modifications in the part?
    How much knowledge do you possess of why the part exists and why it looks like it does?
    - In terms of knowledge of the part when modifying something such as changing geometry etc.
- What is driving the part? Visual/functionality/CAE/Performance/Legal requirements
    - How are the cad models driven by CAE-related functionalities
        - (… to enable the efficient generation of simplified product models for subsequent FE simulation)
- How much are you aware of how your CAD models affect in CAE?
- When you get feedback from CAE, how do you implement that?
- What do you expect from the cad process?
    - How should it perform according to you? {…}
- What do you think is good about the cad process?
- What would you like to improve with the cad process?

CAE/FE:

- How are you using the CATIA models in ANSA
    - How do you build you model in ANSA
    - Simplify – neglect some parts
    - Etc.
- What type of problems arise when elicit mid surfaces from a part
    - What specifics problems are rising regarding a plastic part, A-pillar trim
        - Gaps, overlaps of mid surfaces
        - Problem with ribs?
        - Problem with different thickness?
    - What specifics problems can arise in other area (area as car and/or other parts then the a/pillar)
- How do you avoid errors?
- What simulations are usually performed on plastic parts in cars
    - What specifics are done on A-pillar
    - What can arise in other area (area as car and/or other parts then the a/pillar)
- How much are you aware of that the modelling in CAD affects you work in CAE?
    - Do you have any explanations of how it can be avoid?
- What do you CAE expect from the process?

    - How should it perform according to you? {…}
- What do you think is good about the CAE process?
- What would you like to improve with the CAE process?

About CAD standards

- About cad standards
    - What would you like to find in those standards
        - Easy to/or not necessary to mesh repair and remodelling
        - Changes in cad should be easy for the old CAE model to detect
        - Would also like it to be easy to do changes and be able to change the engineering properties of the object. Such as be able to comparing material and dimension. such as
            - Changes of thickness
            - Changes of size
            - Changes of material
        - Not only depending on experience in designing and analysing though this can differ between worker a to b
        - Shape optimisation using parameters, guideline to have as few as possible
        - Standards of how to design specific parts such as, clips torn
        - Basic – easy to understand directions.
        - Very detailed explanations.
    - Have does your own guidelines look like?
    - Have you seen other part specific guidelines?

Comparison with colleagues & competitors

- How does a colleague do?
- How do competitors do?

Future requirements/market trends

- Have the process/software changed in recent years?
  {New functions, more data needed, more time consuming, more detailed}
- Have your requirements on process/software changed in recent years?
- Do you think your requirements on process/software will change in coming years?

APPENDIX B
BENCHMARK ANALYSIS

Table B1: Benchmark analysis

DESCRIPTION	Volvo Cars/ CEVT	Airbus	Saab	German automotive group	The automotive body: Volume I, component design	Integrated computer aided design in automotive development
Recommendation	Basic geometries should be as generative as possible. Untrimmed, oversized surfaces should be used to create stable geometries. Geometrical sets should be renamed. Sketches should be fully constrained. Only positioned sketches. Dress up features (such as edge fillets, drafts, and chamfers) should be located next to parent. Larger fillets should be done before smaller fillets. Only large fillets should be included in the design (<15 mm).	Create feature based on selecting other feature, and not selecting sub elements. Create stable geometry and by that making models that allows to be modified and to be re-built	Recommended first step is to start with making the profile by skeletons. Important with good structure and to be able to modify. In early design phase the model do not need to follow the design rules but before it will be release the model must fulfil the defined requirements. For thin-walled design; create oversized blocks and sculpture the blocks by using surfaces to cut pieces off. Continue the work by making flanges (either select standard or own design) by sculpturing the tool block by intersection and combine it with the main body by Boolean operation.	Dependence between components/assemblies should always be in the same direction.	Start by breaking down structure definition. Define main section by skeleton, and make them partly independent of the styling surface.	Create stable models that can handle geometrical and structural changes.
Avoidance	Avoid sub elements. Unwanted construction geometry should be removed. No over constrained products. No duplicated versions of a line. No infinite lines, curves, or surfaces. Healing curve smooth or tolerant modelling operators should not be used in construction work (if necessary it is allowed to be used to clean other input geometries).			External faces/corners/edges should not be used as reference.		Avoid using complex geometries as references.

APPENDIX C

Collection of the geometry based study models in CATIA for model A-I

Table C1: Data for small geometry based study

Plate: 50x100mm
Thickness: 2mm
Height of rib: 4*2=8mm
Thickness of rib: 0.4*2=0.8mm
Width of rib: 30 mm
Draft angle: 0.5
Fillet 1 (edge of plate): 1mm
Fillet 2 (bottom of rib): 0.8mm
Fillet 3 (Side of rib): 0.2mm

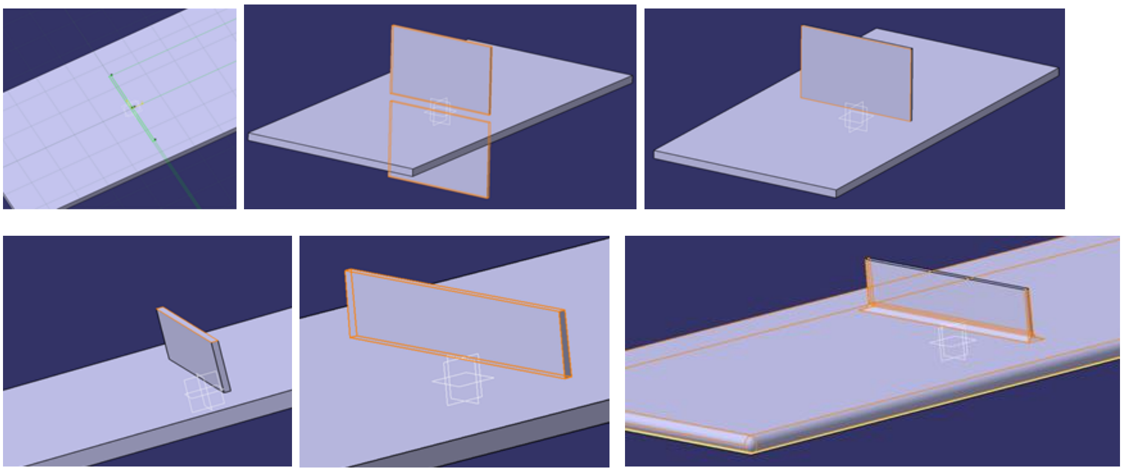
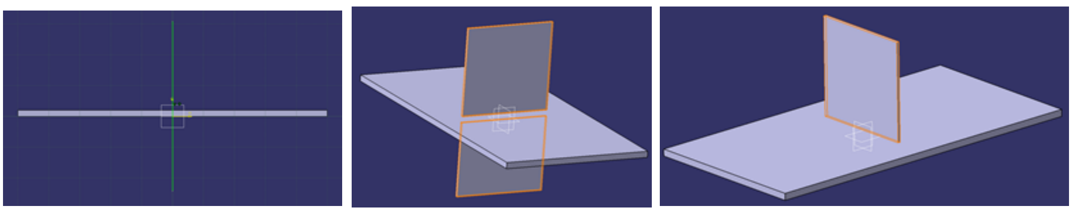


Figure C1: A1 CATIA - Creating CAD models of basic ribs in part design by sketching bottom profile, mirror extended pad. Then using the function union trim between rib and main body. Trim by offset A-surface to define Rib limit. Using the draft function and applying edge fillets in three sizes with largest selected first.



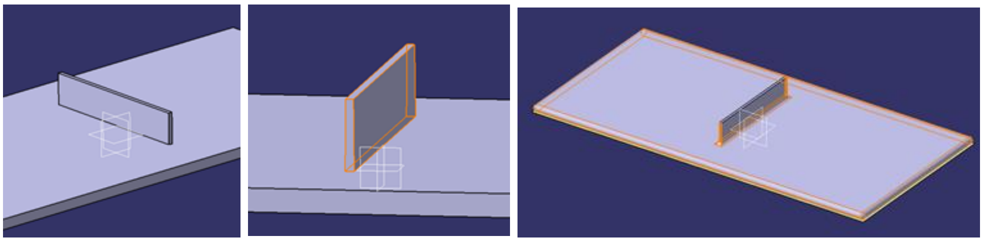


Figure C2: A2 CATIA – Creating CAD models of basic ribs in part design by creating a line on a plane, thick mirror extended pad. Then using the function union trim between rib and main body. Trim by offset A-surface to define Rib limit. Using the draft function and applying edge fillets in three sizes with largest selected first.

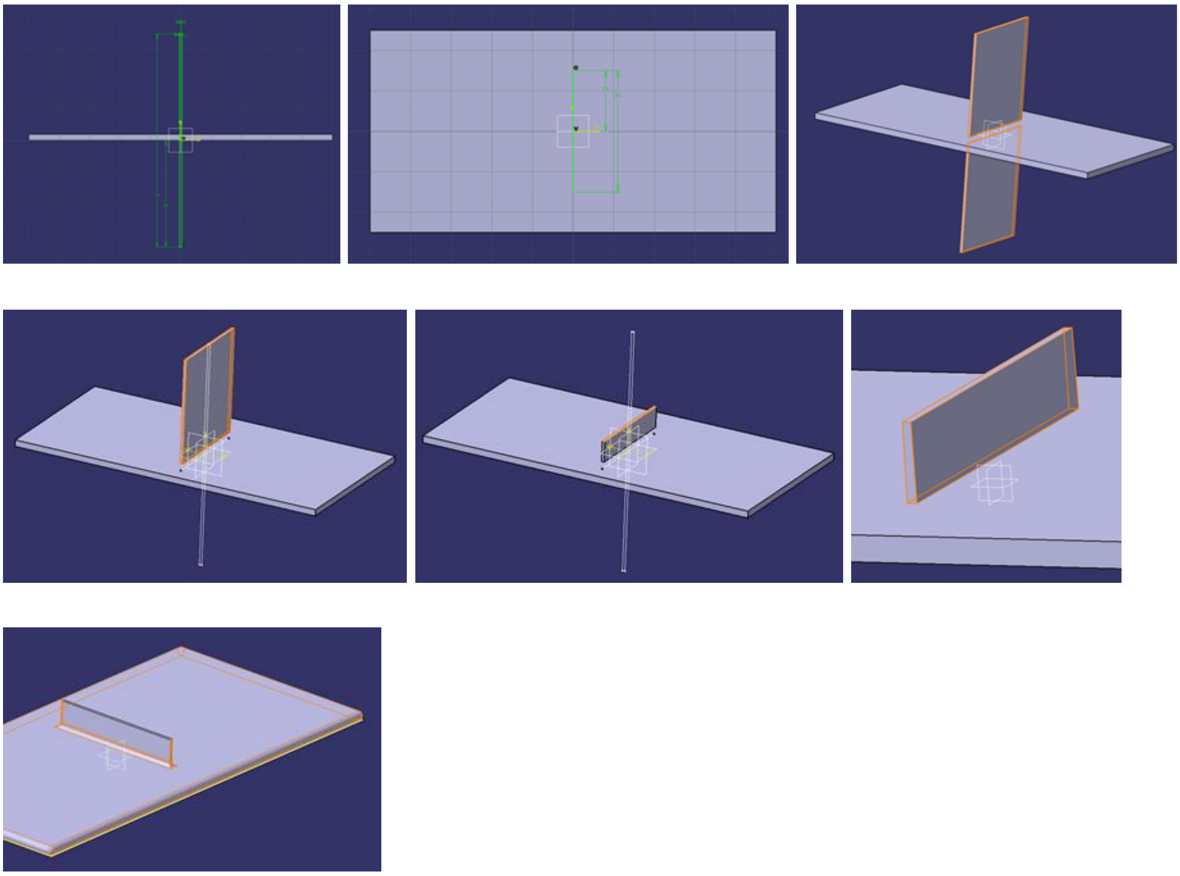


Figure C3: A3 CATIA – Creating CAD models of basic ribs by rib function in part design by creating a profile and (centre curve) in two different planes. Then using the function union trim between rib and main body. Trim by offset A-surface to define Rib limit. Using the draft function and applying edge fillets in three sizes with largest selected first.

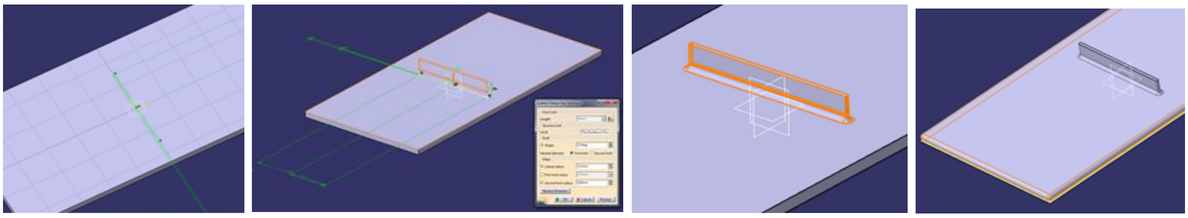


Figure C4: A4 CATIA – Creating CAD models of basic ribs by drafted filleted pad function in part design by creating a bottom profile sketch and defining draft angle and fillets. Then applying edge fillets for the main body.

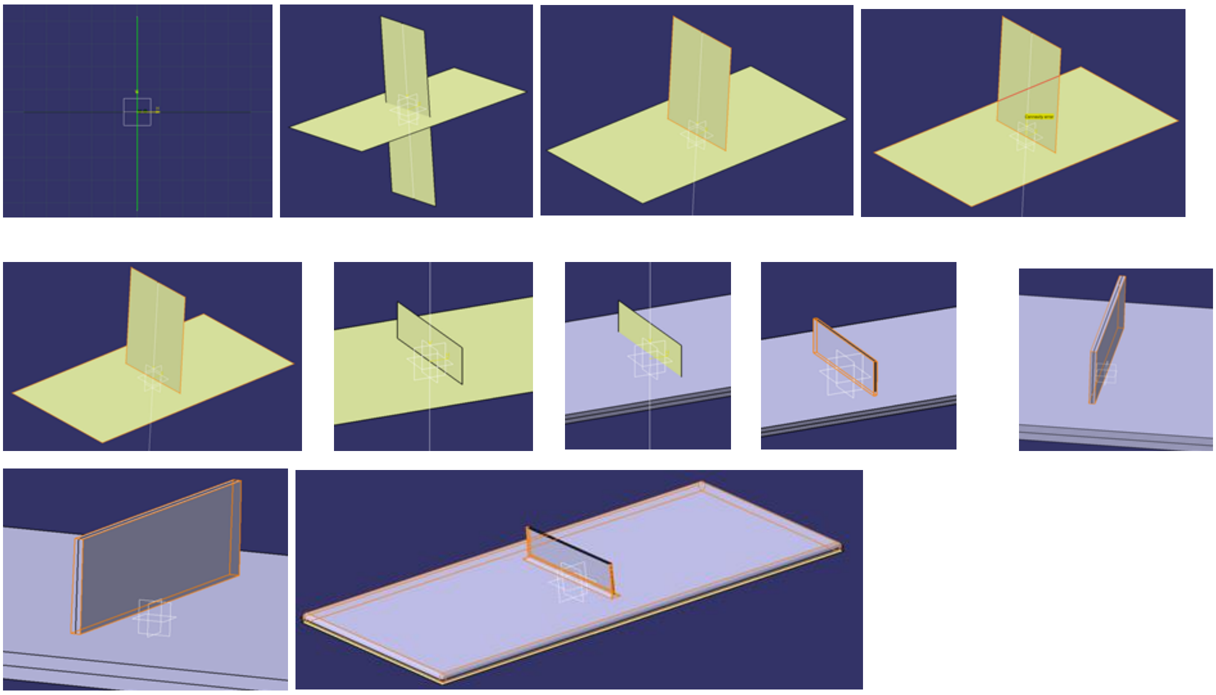


Figure C5: A6 CATIA - Creating CAD models of basic ribs in generative shape design by creating a line on a plane, and extrude. Then using the function split between rib and main body and then by offset A-surface and rib to define Rib limit. These cannot be joined without connexity error, and if joined they have to have the same thickness. Instead moving to part design without join and adding thickness to main body and rib in separate steps. Create union trim between rib and main body. Then using the draft function and applying edge fillets in three sizes with largest selected first.

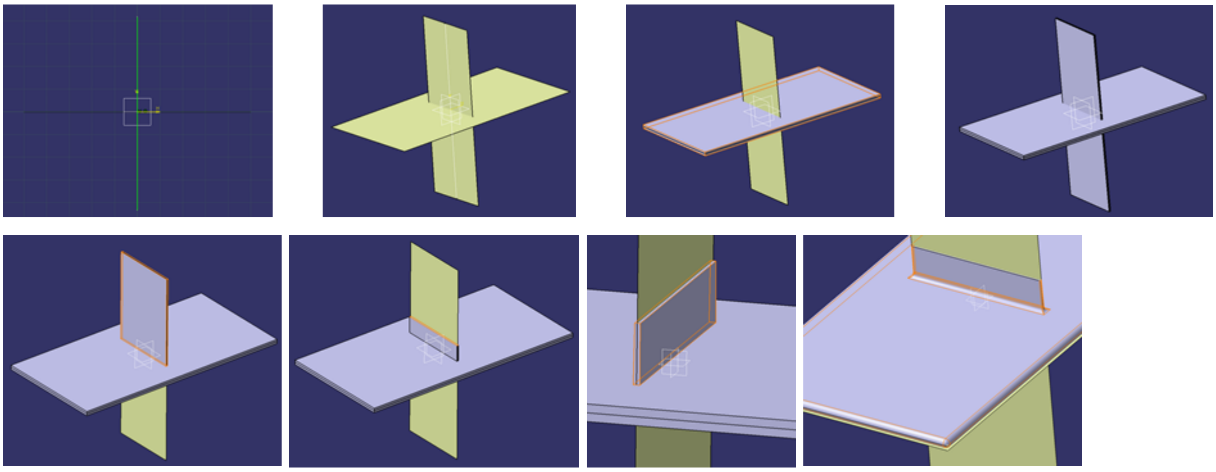


Figure C6: A7 CATIA - Creating CAD models of basic ribs in generative shape design by creating a line on a plane, and extrude. Moving directly to part design without join and adding thickness to main body and rib in separate steps. Create union trim between rib and main body and trim between rib and rib height limit offset surface. Then using the draft function and applying edge fillets in three sizes with largest selected first.

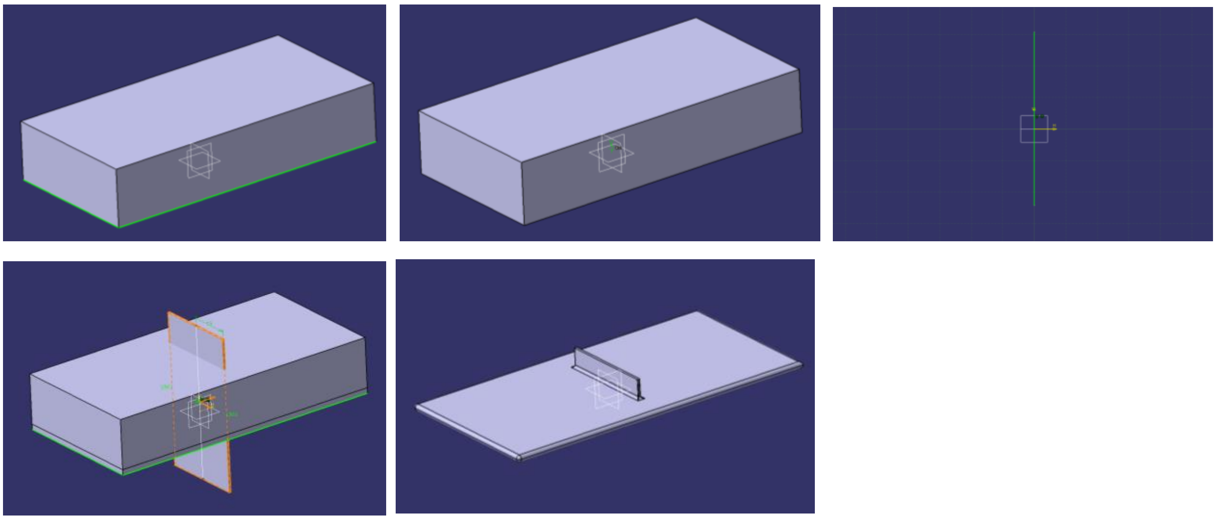


Figure C7: A8 CATIA – Creating CAD models of Boolean operation by first adding over-dimensioned thickness of the A-surface in a body. Copying the body into another body and shrink with minus thickness to create required thickness for the main body. Create another body and create basic ribs in part design by creating a line on a plane in a positioned sketch, thick mirror extended pad. Trim by offset A-surface to define Rib limit, and possible extrapolate to make the surface reach. Then using the Boolean operations and remove the ribs from the second body. Then remove the second body from the first body. Using the draft function and applying edge fillets in three sizes with largest selected first.

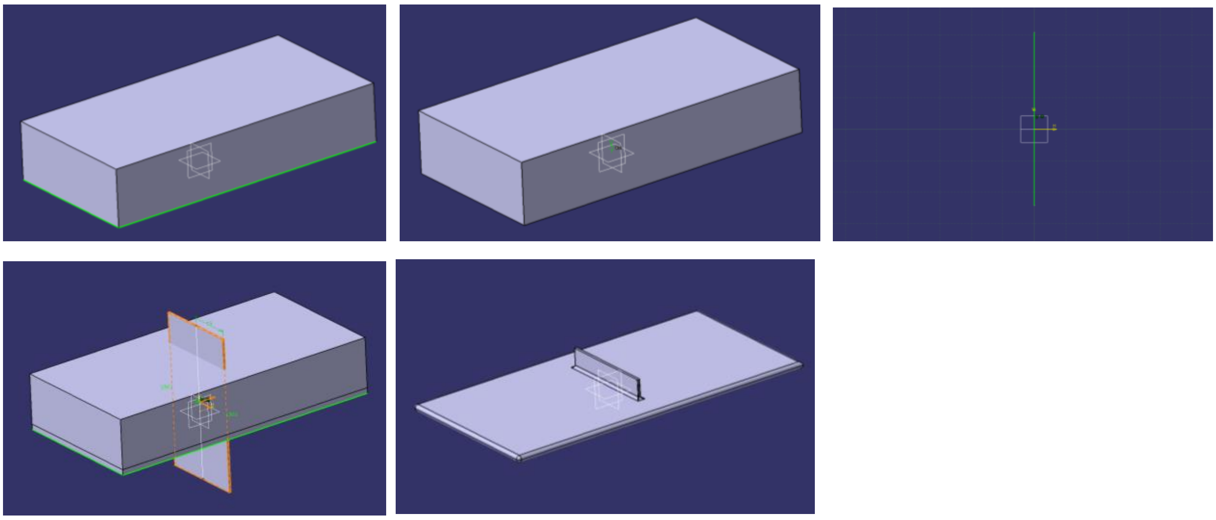


Figure C8: A9 CATIA – Creating CAD models of Boolean operation by first adding over-dimensioned thickness of the A-surface in a body. Copying the body into another body and shrink with minus thickness to create required thickness for the main body. Create another body and create basic ribs in part design by creating a line on a plane in a positioned sketch, thick mirror extended pad. Trim by offset A-surface to define Rib limit, and possible extrapolate to make the surface reach. Then using the Boolean operations and remove the ribs from the second body. Then remove the second body from the first body. Then use intersection edge fillet to apply thickness between the main body and ribs.

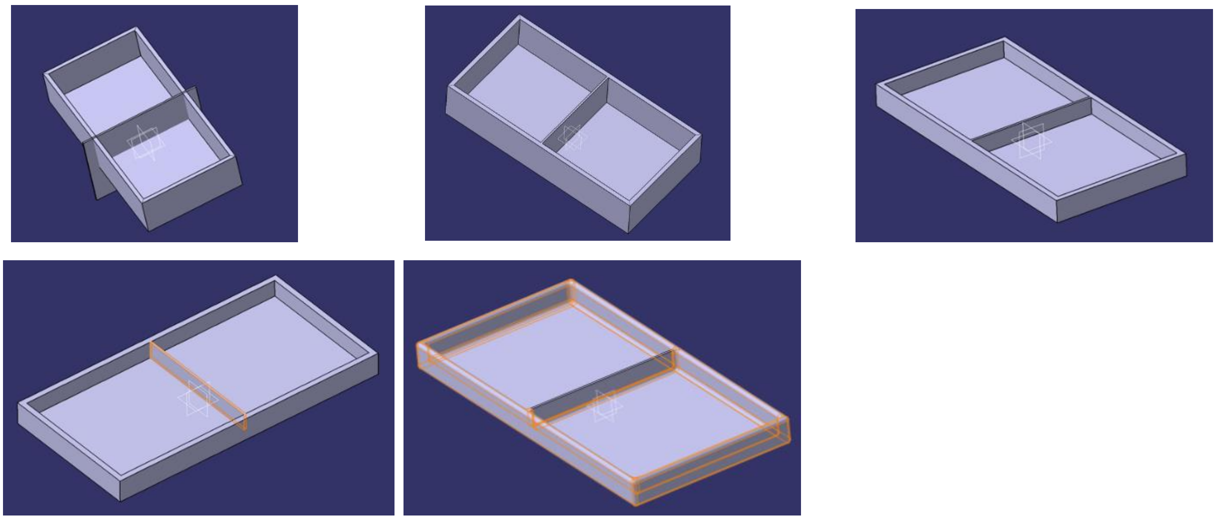


Figure C9: B1 CATIA - Creating CAD models of closed ribs in part design by sketching bottom profile oversized, and mirror extended pad. Then using the function union trim between rib and main body. Trim by offset A-surface to define Rib limit. Using the draft function and applying edge fillets in three sizes with largest selected first.

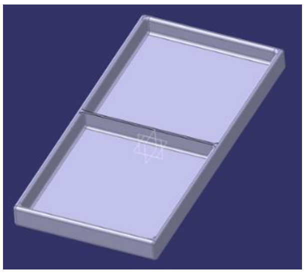


Figure C10: B2 CATIA – Creating CAD models of closed ribs in part design by creating a line on a plane, thick mirror extended pad. Then using the function union trim between rib and main body. Trim by offset A-surface to define Rib limit. Using the draft function and applying edge fillets in three sizes with largest selected first.

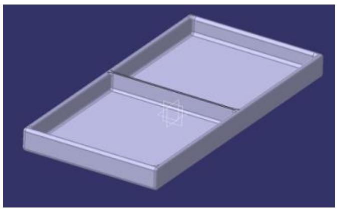


Figure C11: B3 CATIA – Creating CAD models of closed ribs by rib function in part design by creating a profile and (centre curve) in two different planes. Then using the function union trim between rib and main body. Trim by offset A-surface to define Rib limit. Using the draft function and applying edge fillets in three sizes with largest selected first.

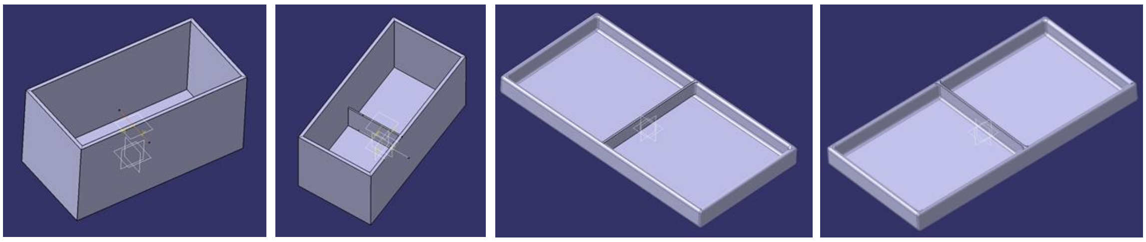


Figure C12: B5 CATIA– Creating CAD models of closed ribs by stiffener function in part design by creating a sketch Split from the rib height limit. Using the draft function and applying edge fillets in two sizes with largest selected first.

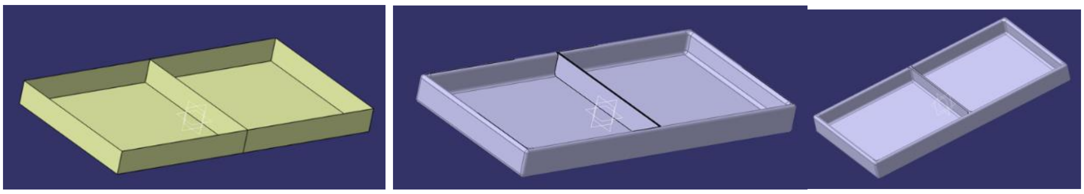


Figure C13: B6 CATIA - Creating CAD models of basic ribs in generative shape design by creating a line on a plane, and extrude. Then using the function split between rib and main *body and then by offset A-surface and rib to define Rib limit. These cannot be joined without connexity error, and if joined they have to have the same thickness. Instead moving to part design without join and adding thickness to main body and rib in separate steps. Create union trim between rib and main body. Then using the draft function and applying edge fillets in two sizes with largest selected first.*

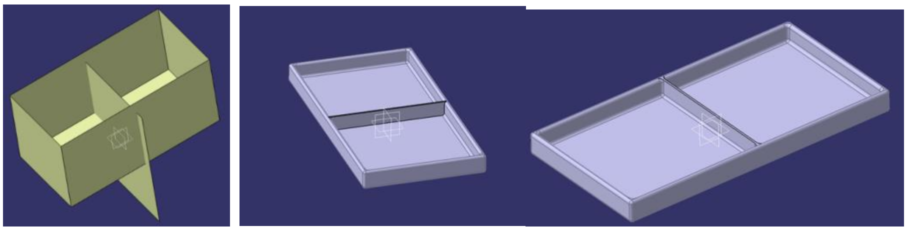


Figure C14: B7 CATIA - Creating CAD models of basic ribs in generative shape design by creating a line on a plane, and extrude. Moving directly to part design without join and adding thickness to main body and rib in separate steps. Create union trim between rib and main body and trim between rib and rib height limit offset surface. Then using the draft function and applying edge fillets in three sizes with largest selected first.

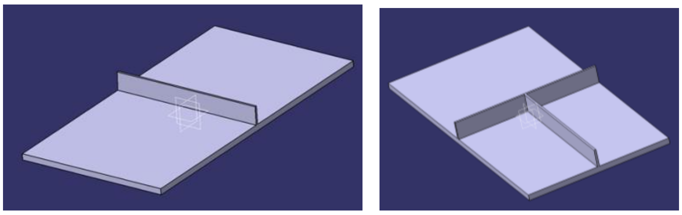


Figure C15: C1

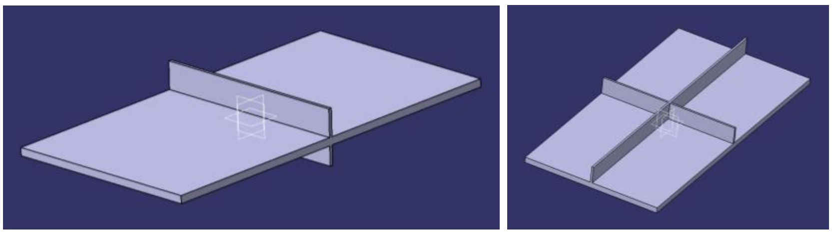


Figure C16: D

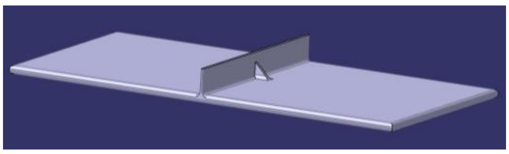


Figure C17: E

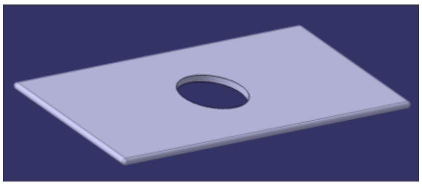


Figure C18: F

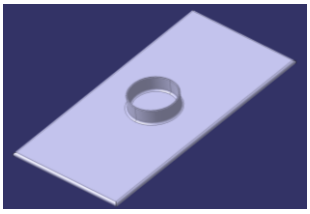


Figure C19: G

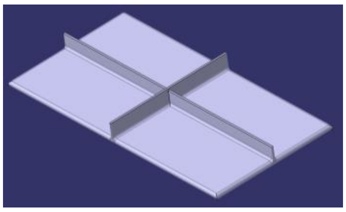


Figure C20: H

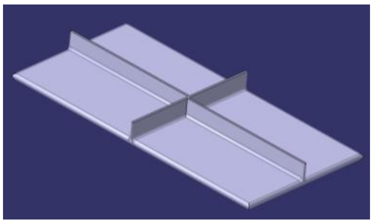


Figure C21: I

APPENDIX D
CAD Standard proposal

DOCUMENTATION RULES

Product Computer-Aided Design Standards

CATIA V5

Basic Guideline for the Design of Plastic Parts

Orientation

Issue 1 is the first proposal of the China Euro Vehicle Technology CAD Standard for the design of plastic parts.

In complement to

CEVT CAD Standard

CONTENTS

1 Scope and field of application

The purpose of this standard proposal is to provide CAD-specific directions to model plastic components, with the A-pillar trim as example, to CEVT. The information in this document is gathered from CEVT CAD standard, CAD Advance at Volvo Cars, and the research and studies that were conducted for the master thesis “Definition of CAD standard that improves and enhances data transfer to CAE”.

This documentation will first, in section 2, present some general guidelines when modelling in CATIA. Then in section 3 it will provide some modelling strategies for plastic part in specific. In section 4, a modelling strategy within generative shape design is provided. This is followed up with a modelling strategy for part design, in section 5. In section 6, a reference list is documented. Finally, in section 7, a record of revision is documented.

2 General Guidelines

The modelling technique that is suggested in this CAD standard consist of only modelling with surfaces for the A-surface and the rest of the body should be constructed with solid modelling. The modelling technique can be seen in figure 1.

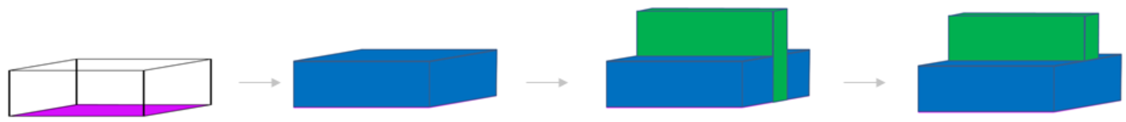
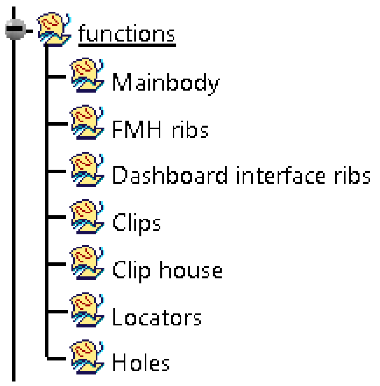


Figure 1: Modelling technique

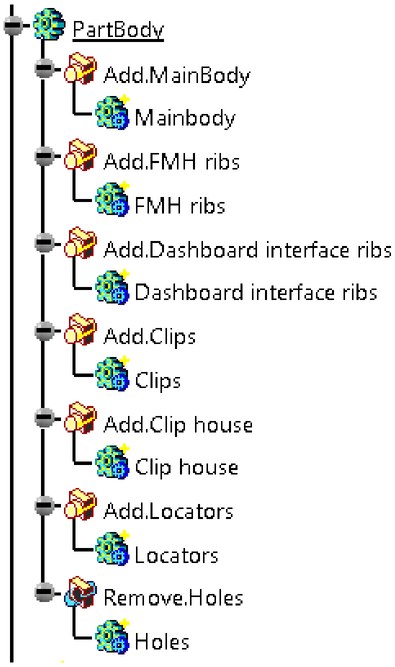
The modelling technique that has been suggested in the CAD standard is only following the practice of using non sub element and creating a logical work flow. The reason for this is to make it easy to modify and replace an element within the model. Also to make the process easier for the next person who will work with the CAD model.

Surface modelling is most suited for creating the A-surface from the styling surface. The A-surface should be analysed before starting the modelling work so it do not contain holes, overlaps or radius that cannot be offset, since the quality is translated to the solid body and can affect other downstream processes.

Skeleton, such as points, curves and planes should be defined with help of parameters that can be used to subtract or expand the model. Sketches should be conducted in different independent geometrical sets in the generative shape modelling workbench. These geometrical sets should be describing a certain function or area of the part, such as ribs, clips and clip houses. The sketch should be positioned against a plane, point and line so it can easily be modified.



The rest of the part should be constructed in solid modelling. The sketches created in the geometrical sets should be used to create object, and should be constructed in different bodies depending the geometrical set. The solid body will be trimmed by other surfaces of mating parts and other shaping parts. If working with oversized geometries, both surfaces and solid, the model is easily modified and update errors can be avoided if operations are selected carefully. The model minimises the errors of gaps that is not only a problem for the CAD model itself but also other downstream processes. These bodies should thereafter be combined with Boolean operations such as add and remove. The fillets and draft should be placed outside these groups so the solid body can be replaced. This also makes it easier to detect since these can be critical features for downstream processes, Very small surfaces patches due to dress up features, such as fillets, should be avoided. Unintentional design, such as creation of small steps or offset between intersecting object should also be avoided. Solid and surfaces should be observed to not be unstitched.



The CAD model should be cleaned up and unnecessary features should be removed before releasing to downstream processes. The CAE department should catch the CAD model and deactivating certain features, such as fillets, smaller holes, and small edge patches before entering the work in FE modelling.

The conclusion of how to model for suit downstream processes as FE modelling can be seen in the following table.

Table 1: Modelling strategies for FE modelling and other downstream processes

Surface modelling is only suited for the creating the A-surface from the styling surface

The A-surface is important to be of good quality

Skeleton, such as points, curves and planes should be defined with the help of parameters

Sketches, such as for the design of ribs, clips and clip houses, should be conducted in geometrical sets within generative shape modelling

Solid modelling is suited to create the rest of the part

Sketches that were conducted in geometrical sets should be used to create solids

Skeleton and surfaces should be used to control the size of the solid model in part design

Improvement of the CAD quality can be achieved

A clean specification tree will remove unnecessary features that may not affect the CAD model visual, but will create problems for the FE model.

A model with a seamless interface is a benefit for ANSA; since the aesthetic lines are creating conflicts with lines of a feature

Working with oversized solid features and planning the model and not try to reconstruct when it might be easier to do it right from start.

Features of the part should be done in different bodies combined with Boolean operations, such as add and remove.

Fillets and other dress up features can cause problem for the FE model if they are asymmetrical on an object, such as the walls of the ribs.

The patches should be observed to not be too small, sharp or short so it is possible to manufacture and to be manageable for FE modelling and other downstream processes.

Objects that are crossing each other with a small step or offset cause problem and should be avoided

The fillets and draft should be placed outside these groups so the solid body can be replaced

The translation to FE modelling can be improved

To improve the identification of different objects in FE modelling the CAD modelling should not consist of add-on.

Fillets, smaller holes, and small edge patches should be suppressed before the meshing starts

Detect unstitched solids and surfaces and make them clean will also approve the mesh.

The CAE department can be able to catch the CAD model, and make the model simpler, with deactivating certain features, before entering the work in FE modelling.

General recommendations

It is recommended to add a communication portal regarding the transaction between the CAD and the CAE departments. To follow up these subjects the work can be improved for both teams, and do not have to be reworked for each time.

To be able to implement the work correctly it is important to introduce training sessions to assure that the CAD modelling is done in the same manner and in the correct way.

Basic guidelines when creating a flexible model is to make sure to avoid using sub elements (B-rep). It might be hard to replace something based on sub elements, since it is not a one to one replacement and the links are broken.

To have a stable and flexible model that can be modified, one should chose functions that are not dependent on each other. For example, the designer should select points instead of vertices, boundaries instead of edges, surface features instead of faces. The reason for this is when replacing one part with another, say a surface, the surface is rebuild and have new vertices, edges, and faces. This is not a problem when just changing some parameters. The model is then only modified, and not re-built, and sub elements are avoided.

To check if the picked element is a sub element, the selected element always has a backslash in the text field within the applied command in CATIA. If selecting the element with the mouse there is a possibility to select sub elements, a stable way is to instead select in the tree structure. To avoid sub element it comes with a cost, since to have an easier update there are in general more operation steps. Within generative shape design it is possible to avoid sub elements completely. Within solid modelling it is difficult to avoid using sub elements; for example draft has to be done by selecting faces of the element.

It is also important both for CAD and FE modelling to have a good model with good geometry. This can be achieved by not having duplicate lines, have trimmed and joined the surfaces correctly to avoid discontinuousness in meeting points, not having single surfaces without thickness, and so on.

Another guideline is to keep the tree structure structured and keep the relations between features as simple as possible. A clean tree structure helps to remove unnecessary features that can cause problem when translating to FE model.

The relations should not be depending on too many features. A short and compact relation chains make it possible to have shorter update times, faster troubleshooting, together with a more logical tree structure. To be able to keep track of the relation it is also necessary to have descriptive names of key functions, features and elements.

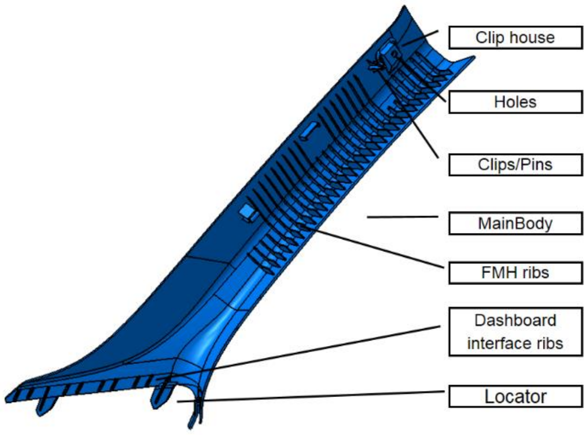
It is good to add colour to the part, as well as different line types and point types, to visualize key features and elements in the part. Colours to avoid are red and orange since these are already used in CATIA by default. When assigning a colour do not select faces, since it will result in sub elements. Always select the elements.

There is no upper limit of how many elements that can be selected, but it is better to select fewer to avoid update error. They can be divided into logical groups, such as horizontal, and vertical.

Some functions that was observed that made the process fail for FE modelling were fill and blend in CATIA. Reason of this may depend on that the model gets an unstable geometry due to both sharp and fading edges. This should therefore be used with consideration, since the model may consist of gaps when translating to another format.

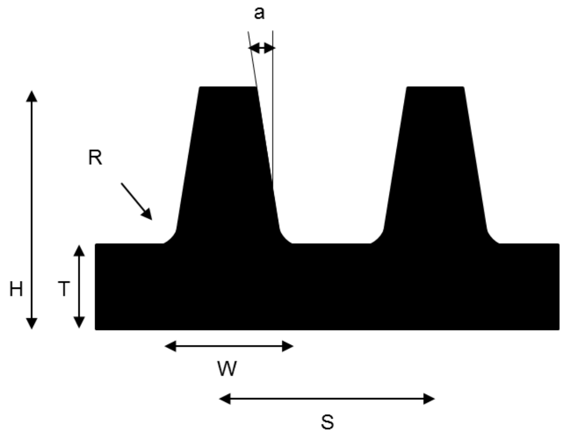
3 Plastic Part modelling strategy

In this section an introduction of modelling strategies for plastic parts is given. In the figure below three descriptive models are shown of the A-pillar trim. The first one shows some necessary features that a plastic part consists of. The other two visualize the over dimensioned A-surface and slabs surfaces.



- Design will be confirmed by CAE analysis
  - Try a design with few ribs and add more ribs if necessary rather than trying with too many first.
- Design will be confirmed of real prototype
  - The design should be done to not cause sink marks and warping on the A-surface during manufacturing.
  - The ribs design should be thinner than required. If the ribs need to be larger it is easier to carve out material of the tool, rather than add material to the tool (if the ribs would happen to be too thick).
  - The design of the clips, clip houses and locators are often required to be thinner close to the MainBody due to sink marks and warping on the A-surface. If this is the case plan for this.

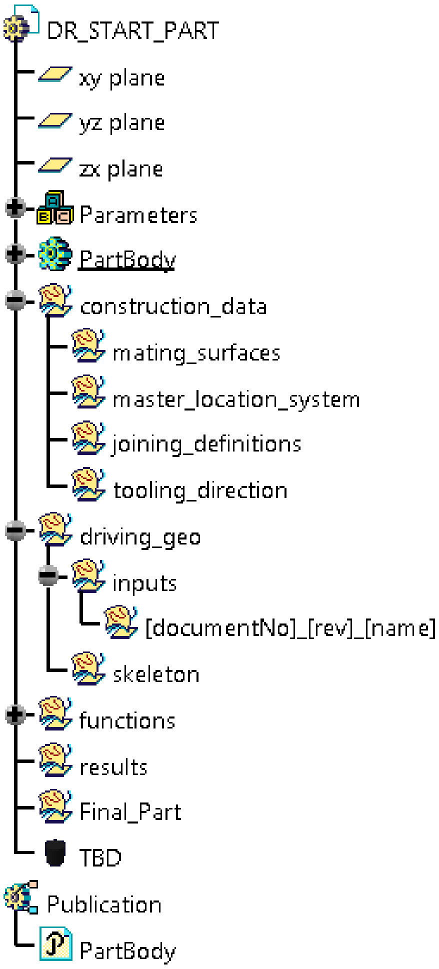
Note: If other requirements are specified, those should be followed.



Height (H):	H ≤ 3 T
Width (W):	W≥ 0.4 – 0.6 T
Corner radius (R):	R ≥ 0.25 – 0.5 T
Draft angle (a):	a ≥ 0.5°
Spacing (S):	S ≥ 2T

4 Modelling strategy within generative shape design

In this section the procedure of the modelling strategies within generative shape design for the design of plastic parts is presented. The structure is presented in a way that is following the tree structure of the start-up files.



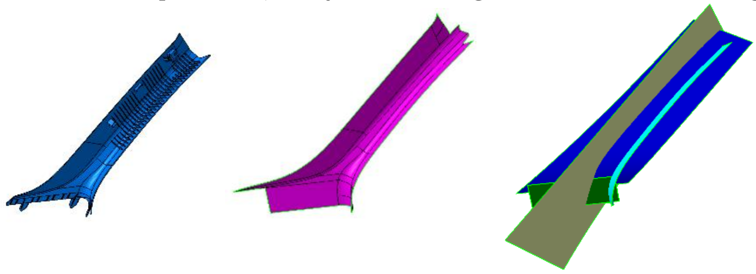
Surface modelling is most suited for creating the A-surface from the styling surface, the rest of the part should be constructed in solid modelling. The A-surface should be analysed before starting the modelling work so it do not contain holes, overlaps or radius that cannot be offset, since the quality is translated to the solid body. The A-surface should also be oversized. The studio engineers should create the A-surface and it should not be changed. If the A-surface does not fulfil mentioned requirement or lack of quality, it should be remade of the studio engineers to ensure the design of the surface.

Skeleton, such as points, curves and planes should be defined with help of parameters that can be used to subtract or expand the model. Sketches should be conducted in different independent geometrical sets in the generative shape modelling workbench. These geometrical sets should be describing a certain function or area of the part, such as ribs, clips and clip houses. The sketch should be positioned against a plane, point and line so it can easily be modified.

- ✓ Use the start-up files.
- ✓ Follow the specification tree.
- ✓ Avoid sub elements.
- ✓ Make the work logical and easy for other to follow. This is done by separating the part into different function groups and renaming the features.
- ✓ Check the model for unintentional design, such as creation of small steps or offset between intersecting object which should be avoided.

If there is no existing styling surface, then create a flexible surface that should be replaced for the real A-surface when available;

- Do not use sub element.
- Make it over-dimensioned.
- Avoid complicated sketches.
- Use shape fillets (and by that avoiding sub elements and making the model stable).



4.1 Start Up Files

The part should be structured by the available Start-up files.

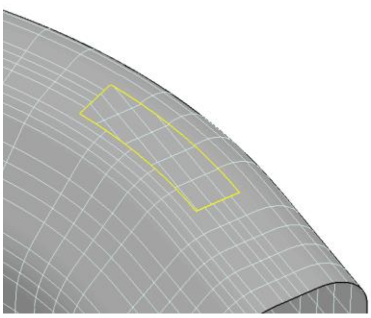
See requirements in CEVT CAD standard.

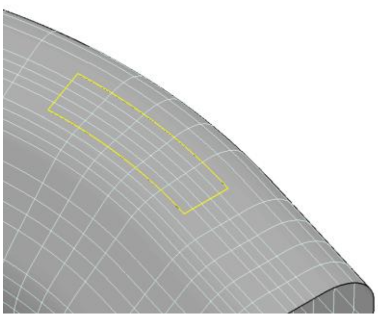
4.2 Identify design inputs

In this section the process of identifying the quality of the styling surface is covered by different types of surface analysis. What is required is to identify the isoparametric curve flow, since this affect the quality of the surface, the solid, and may affect FE modelling. A connect checker analysis should be connected to make sure the surfaces are connected and also the quality. This is important for further work as well with the model. A surface curvature analysis should also be conducted to make sure it can be offset to required distance.

4.2.1 Isoparametric

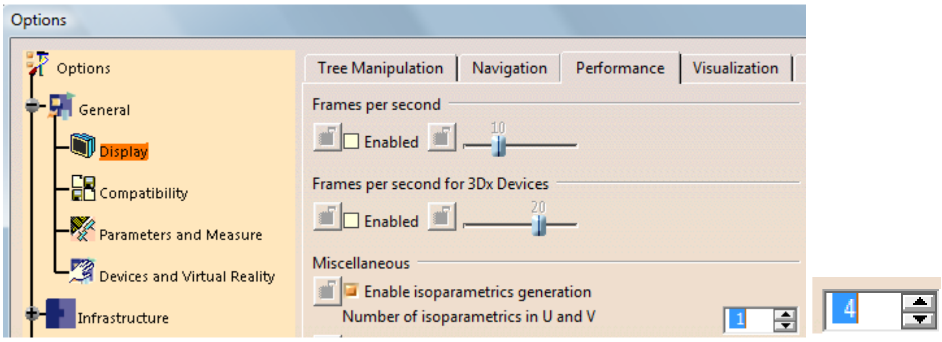
Toggle on isoparametric in customised view mode. It is required to have this when working with A and B surfaces. There should not be any angles between the patches.



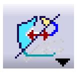


To fulfil this; try to use the same spine and toggle off "Canonical portion detection" when creating surfaces, for example with multi-section surface definition. When surfaces are based on curves with bad quality this will affect the surfaces quality as well. Discontinuities and several unnecessary surface patches is the result of the bad quality.

NOTE! If isoparametric has never been used before go to Tools -> Option -> Display. Guideline to define it to 4. Restart the CATIA session to take effect.

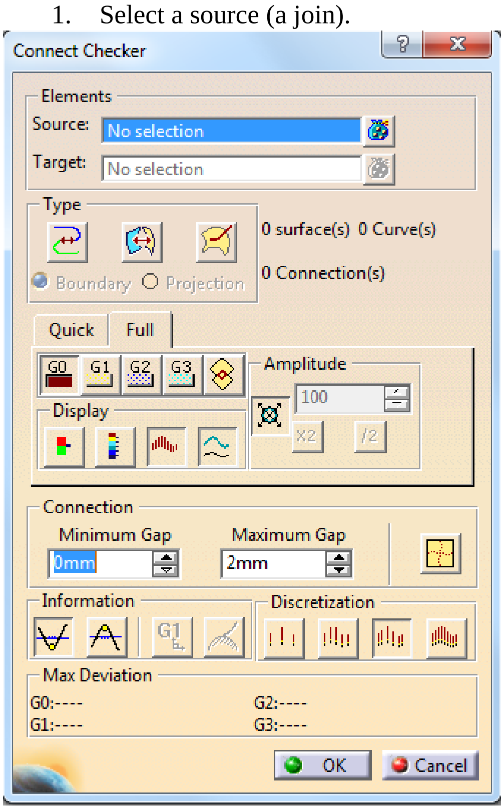


4.2.2 Connect checker Analysis



Make a Connect checker Analysis. To check if surfaces are connected and the quality of the surface etc.

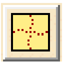
1. Select a source (a join).



2. Toggle on which connection type that should be observed. For example surface-surface connection.

3. At display; select the colour scale to auto min/max.

4. At connection; define gaps to be analysed (recommended range 0-2 mm).
5. Toggle on the internal edge box (internal edges for both surfaces and curves).

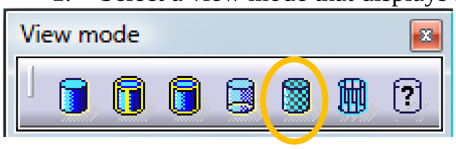


6. Analyse the boundaries by changing the G's.
   - G0 Point continuities [mm] (checks that it is attached without being broken).
   - G1 Tangency continuities [deg] (smoothness, concerned when manufactured).
   - G2 Curvature continuities [% change].
   - G3 Curvature flow continuities [deg].

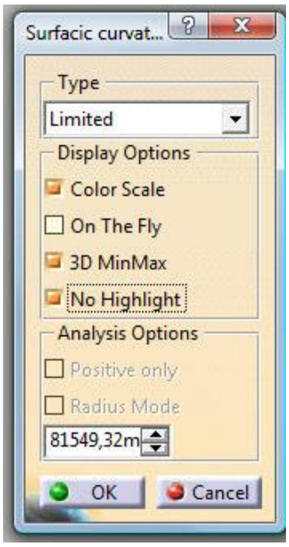
4.2.3 Surfacic Curvature Analysis

Make a *Surfacic Curvature analysis* – type limited. This is a check for radius values. The smallest radius value indicates the largest offset. If any radius is lower than required offset, these surfaces have to be excluded of the main surface and to be added later.

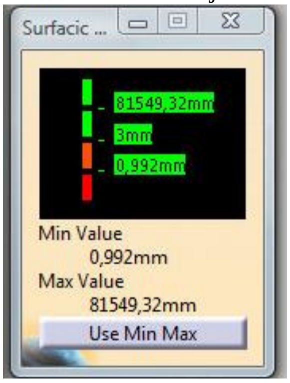
1. Select a view mode that displays material.



2. Select type limited.



3. Select display option colour scale and no highlight.
4. Select the join.



Surfacic Curvature analysis does also check inflections and discontinuities to obtain surfaces and wireframes with required quality.

For plastic parts it is possible to check the A-surface, including holes, that it can be offset to the material thickness. If the minimum radius is lower than the material thickness, these have to be excluded from the A-surface and to be added to the solid, for example with sew.

Note: The quality of the A-surface should be in the condition that it is possible to offset to its material thickness. Otherwise the A-surface should be remade by the studio engineers.

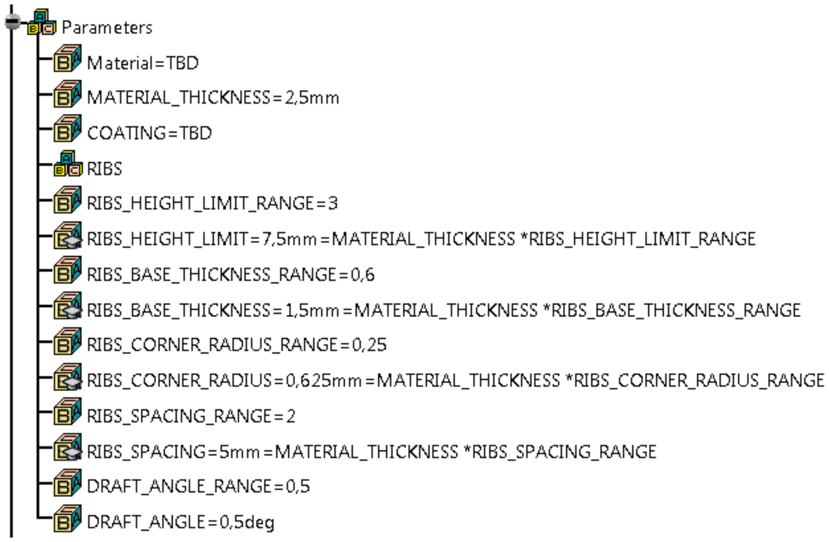
4.3 Identify part position

Parts should be modelled in the right position after the car's coordinate system.

See requirements in CEVT CAD standard.

4.4 Identify important parameters

In this section an introduction of necessary parameters for plastic parts are described.



Parameters can be defined in sub sets to order the tree structure. A general guideline is to have as few parameters as possible and to think them through. When defining the objects in the specification tree with the parameters; always link from the bottom to the top, never the opposite direction. Otherwise there is tougher to organise the links and to have control of the work. With a complex model with many driving parameters it is recommended to order the parameters into sets.

The formula operation uses SI units by default, so even if modelling in millimetres, the functions are defined in meters if no unit is included. The way to avoid this is to type in mm after a value in the formula box.

Note: It is recommended to both model and write formulas in millimetres.

To be able to add a formula to a circle the dimension must be defined on the radius instead of the diameter. A quick way to select a parameter is to write an equal sign, and then click on which parameters that should be applied in the tree structure.

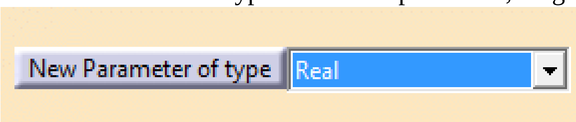
See requirements in CEVT CAD standard.

4.4.1 How to create a new parameter

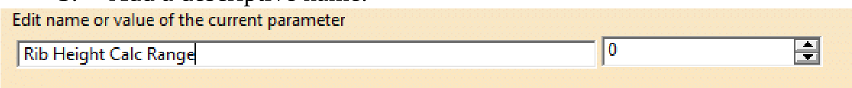
In this section the process of creating parameters is presented.

1. Select formula, f(x), in knowledge toolbar.

2. Define the type of the new parameter; length, degree, range.



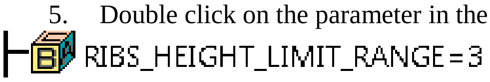
3. Add a descriptive name.



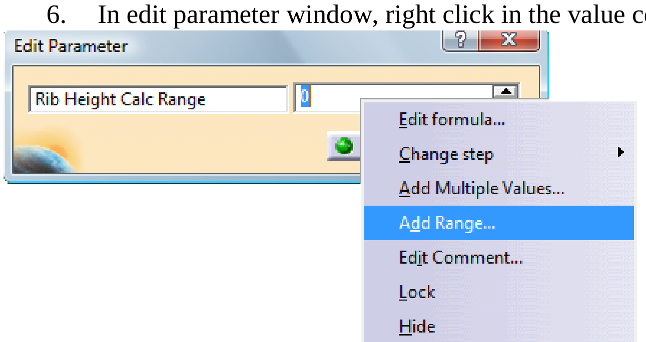
4. Confirm the parameter creation with ok.

If range was selected as type:

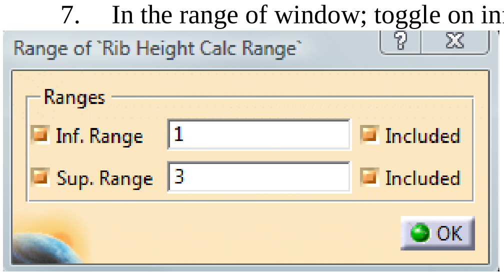
5. Double click on the parameter in the specification tree.



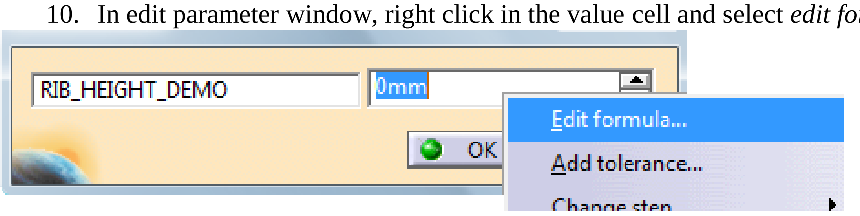
6. In edit parameter window, right click in the value cell and select *add range* from the menu.



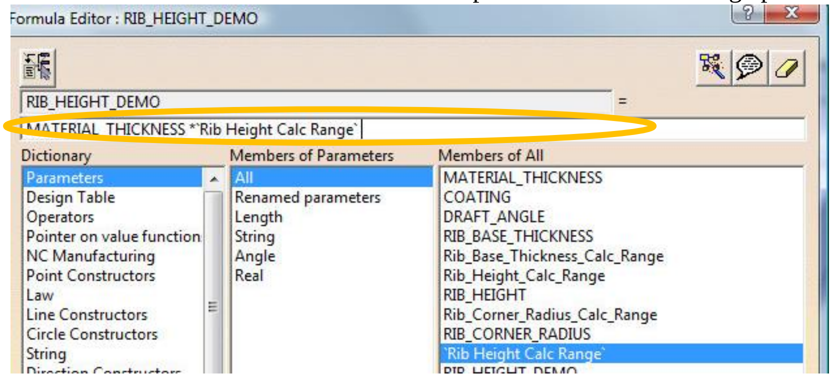
7. In the range of window; toggle on inf. range and sup. range and type in the range that should be used.



8. Create a parameter for what the range parameter should be applied for (step 1-4).
9. Double click on the new parameter in the specification tree.
10. In edit parameter window, right click in the value cell and select *edit formula* from the menu.



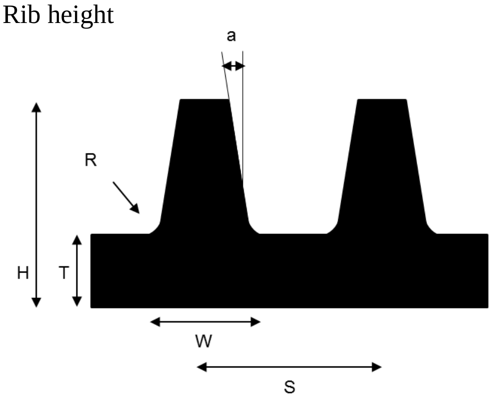
11. In the formula editor write the expression and use the range parameter.



4.4.2 Necessary parameter for plastic parts

In this section some necessary features are described.

- Material type,
- Coating,
- Material thickness,
- Draft angle.
- Thickness for rib
- Range for ribs
- Rib height



Height (H):	H ≤ 3 T
Width (W):	W≥ 0.4 – 0.6 T
Corner radius (R):	R ≥ 0.25 – 0.5 T
Draft angle (a):	a ≥ 0.5°
Spacing (S):	S ≥ 2T

Recommendation to follows (data can be changed if other is specified and is dependent of material type).

Even though, it is recommended to make the ribs even thinner due to real prototype test might show other result. And it is easier to cut of material from the tool than to add material.

4.4.2.1 Material type

The parameter of material type is already defined in the start-up file. Change material in the material library.

See requirements in CEVT CAD standard.

4.4.2.2 Coating

The parameter of coating is already defined in the start-up file. Change coating.

See requirements in CEVT CAD standard.

4.4.2.3 Material thickness (T)

Perform step 1-4 of how to create a new parameter and create a parameter of type length.

See requirements in CEVT CAD standard.

4.4.2.4 Draft angle (a)

Perform the whole how to create a new parameter and create a parameter of type degree and control it with a range.

4.4.2.5 Thickness for rib (W), Range for ribs (S), Rib height (H)

Perform the whole how to create a new parameter and create a parameter of type length and control it with a range.

4.4.2.6 Clips, clips house and locators

If necessary, conduct the whole *how to create a new parameter* and create a parameter of type length

4.4.2.7 Create holes

If necessary, conduct step 1-4 of how to create a new parameter and create a parameter of type radius.

4.5 Identify construction data

The construction data contains geometrical sets about the process information.

4.5.1 Mating surfaces

The geometrical set contains the contacting areas of other parts, such as joined curves. These are recommended to be isolated, so called dead, to avoid problem within the model when other parts are updated. Instead the mating surfaces should be manually replaced when a new revision exists.

See requirements in CEVT CAD standard.

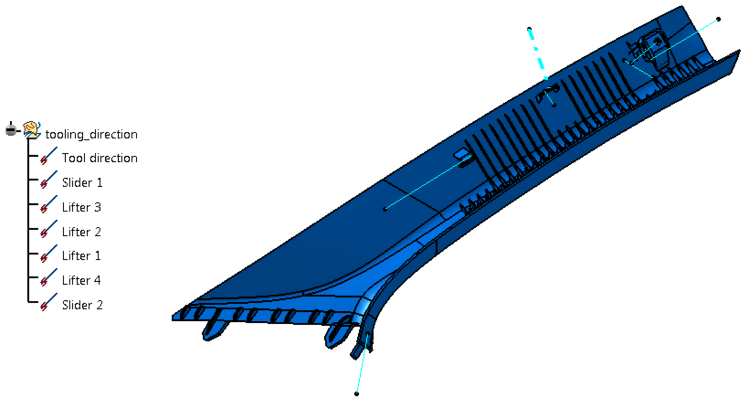
4.5.2 Master location system

The geometrical set contains six or more master location points with correct naming convention and order. All points should be visible and published. Should be done according to the 3-2-1 rule.

See requirements in CEVT CAD standard.

4.5.3 Tooling direction

The geometrical set contains a line with a descriptive name for each tooling direction. Tooling direction of sliders should be added for each one, even if they are in the same direction. For a simple tool, the tooling direction should be selected in the way that make the height of the actual tool as low as possible and be able to include all objects with drafts.



See requirements in CEVT CAD standard.

4.5.4 Punching direction

The geometrical set contains a line with a descriptive name for each punching direction (if several).

See requirements in CEVT CAD standard.

4.5.5 Split line

The geometrical set contains a curve with a descriptive name defining the tooling split line.

See requirements in CEVT CAD standard.

4.5.6 Gating

The geometrical set contains a point and a line.

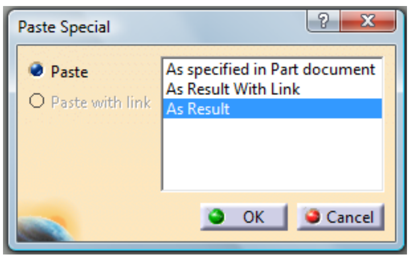
See requirements in CEVT CAD standard.

4.6 Identify part driving geometry

The geometrical set of the driving geometry contains the reference geometry that is driving the function.

4.6.1 Inputs

Reference elements should be dead (isolated), which means that they should be pasted as a result with no links so they do not have any historical reference. It is important to name this correctly so the reference can be found easily in Teamcenter. A common reference element is the styling surface, and it is good to give it a descriptive name and add which revision and date this was retrieved.



4.6.2 Skeleton

Recommendation is to have a good skeleton that can control the model shape. This geometrical set should contain points, lines, surfaces, planes etc., which includes parameters that can either subtract or expand the model. Skeleton as well as features should be *reused* to keep the model lighter. Wireframes that is useful for several features should be placed in the skeleton folder; otherwise it should be placed in their function set.

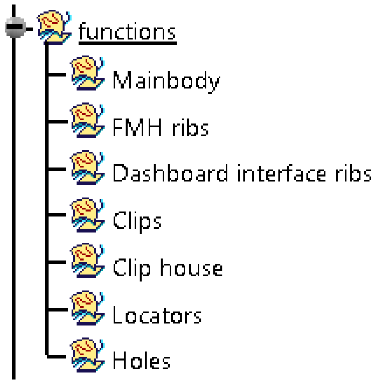
Desired skeletons for the plastic parts

- Interfacing parts.
- Reference plane, points and lines.

See requirements in CEVT CAD standard.

4.7 Identify functional areas of the part

It is important to plan the CAD modelling. Break down the product to small units and make separate parts and assembly later. All functional areas should be created in separated bodies. It should be able to replace features in the model as much as possible. Avoid sub elements as much as possible. In generative shape modelling it is possible to avoid using sub elements completely. Some operation in part design do require sub elements, such as draft.



The function is containing geometrical sets to divide the geometry into logical functions of areas or purpose. It is recommended to create a logical tree that have the same name for the geometrical sets function groups as for the PartBody. In this case it is easy to track which ones belong to each other. The geometrical sets in *functions* should be independent and separable. They should be able to be removed without the CAD-model collapsing. The function groups should be done in a flexible manner since these are often changed during the development process (VM1233 Modelling strategy).

General guidelines for creating flexible surface models is to work over-sized, avoid sub elements, use join, avoid sketches (or keep them to a minimum and use positioned sketches), and has a reference geometry based on wireframe elements.

Oversized goes in hand with having a robust geometry with clean splits and intersection. The model is then able to handle geometrical changes.

4.7.1 Structuring function groups

Within the geometrical sets other sets can be created which makes the selection process much easier. It is recommended to separate the function geometrical sets into;

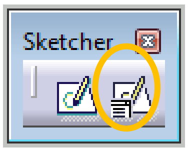
- Input
- Process
- Outputs

The *input* can consists of function specific skeleton planes, points and lines.

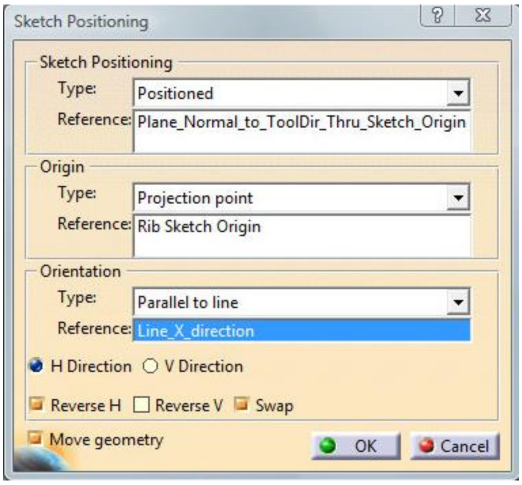
The *process* can include necessary features for the outputs.

The *output* will be used in part design when creating solids. The line, point, and plane of the outputs is what is used to create the positioned sketch.

4.7.2 Positioned sketch



A positioned sketch makes it possible to choose both origin and orientation. Always use positioned sketch. A normal sketch can become a positioned sketch by changing sketch support to a positioned sketch. Positioned sketches are recommended to be positioned against point, line and a plane. By controlling against these, it is easy to change the sketch position. The benefit of placing the sketch in the geometrical set is that it makes it possible in to see the solid in the background when entering a sketch.



Everything inside the sketch can be described as sub elements, it is therefore necessary to keep it simple. Relation should only be referenced to stable and flexible geometry. Referenced elements might be difficult to identify and should be used with carefulness. Use a minimum of 3D references, such as projection and intersection, in sketch. This is due to all elements in the sketch are sub elements. It is also hard to control the offset values and direction. The drawback with projection is that it is using sub elements. If projection is used it should be applied on the whole feature not only a sub element. Otherwise the replace will not be a one to one replacement if the new object does not have the same amount of sub elements.

A guideline is to try to avoid sketches, rather modelling in 3D since this is more stable than sketches.

The sketch can be checked with the 2D analysis tool bar. All sketches should be ISO constrained and the tool checks whether it fulfil the requirements, and by that is not under or over constrained.

4.7.3 MainBody

1. Use the A-surface and untrim to make it oversize. If the styling surface is a join, start with dissemble command and follow with untrim to reveal the base surface.
2. Make sure it can be offset by performing a radius check. Otherwise some surfaces may have to be excluded and to be later on sewed on to the solid.
3. Extrapolate the base surfaces if needed.

Note: Styling surface/A-surface is important to be kept. The B-surfaces can be modified, and do not have to follow the A-surface exactly (if not specified otherwise).

Input:

- A-surface (the styling surface from input driving geometry).
- Or slabs (free styled surfaces).

Process:

- Untrim, Disassemble, extrapolate.

Outputs:

- Positioned sketch and its plane, point, and line.
- Surfaces to sew.

4.7.4 Ribs

When creating the ribs for plastic parts it is a benefit with creating only lines in a positioned sketch. The lines should be over-dimensioned. There is no need to have to enter the sketch and change anything from there. The size of the lines/ribs can be controlled in 3D view.

Note: These features should be planned for the tooling direction.

Note: Make them oversized with a height of the ribs crossing the MainBody to be able to trim later on.

Input:

- Planes controlled by the parameter of rib spacing.

Process:

- Construction work to create the surfaces that controls the ribs height and width.

Outputs:

- Positioned sketch and its plane, point, and line.
- The work within the positioned sketch should only consist of lines that are constrained to the planes.
- Rib width limit – offset the planar slab surface to desired limit, which is recommended to be controlled with a driving parameter of the rib width limit.
- Rib height limit – offset the planar slab surface to desired limit, which is recommended to be controlled with the driving parameter for the rib height limit.

4.7.5 Create clips, clips house and locators

Note. If possible, use an already existing power copy/standard part or create an own if this will be used in several places.

Note: These features should be planned for both the tooling direction and sliders.

Note: Create a profile in a positioned sketch with direction of the slider. Make the height oversized ("the legs"). This to achieve that the clips, clip houses and locators crossing the MainBody, to be able to trim later on.

Note: The design close to the MainBody is often required to be thinner due to sink marks and warping on the A-surface. If this is the case plan for this.

Input:

- Planes controlled by the related parameters

Process:

- Construction work to create the sliders clearance zones for the clips, clips house and locators.

Outputs:

- Positioned sketch and its plane, point, and line.
- The work within the positioned sketch should in first hand consist of lines, otherwise a complete profile of the clips, clips house and locators should be done.
- Surfaces/or positioned sketches that defines the sliders clearance zones.

4.7.6 Create holes

Input:

- Planes controlled by the parameter related parameters.

Process:

- Construction work.

Outputs:

- Positioned sketch and its plane, point, and line.
- A circle controlled with radius parameter (the dimension should be defined as radius to be achievable)

4.8 Combine the functional areas to result

The *results* set combines the functions and a common result is a join or shape fillet surface. All the result from the functions should be placed in the result set.

See requirements in CEVT CAD standard.

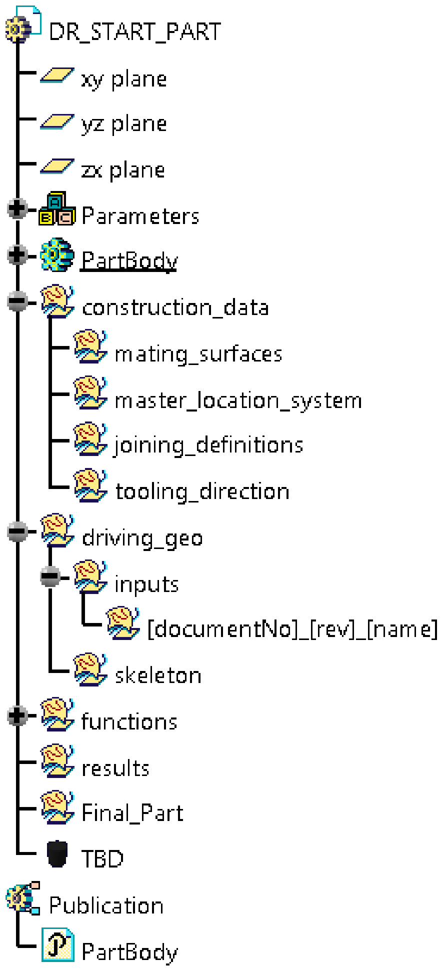
4.9 Create the final part

The *final part* is a surface. It should be a published join with the name DSM (design shape model). It is usually a repetition of the result.

See requirements in CEVT CAD standard.

5 Modelling strategy within Part design

In this section the procedure of the modelling strategies within part design for the design of plastic parts is presented. The structure is presented in a way that is following the tree structure of the start-up files.

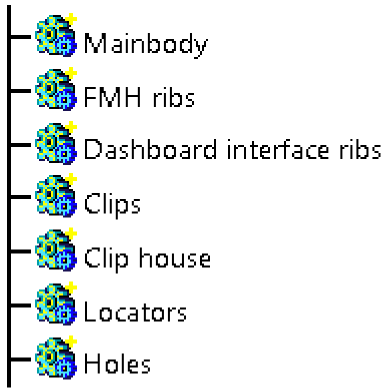


Surface modelling is most suited for creating the A-surface from the styling surface, the rest of the part should be constructed in solid modelling. When continuing to the part design workbench, the sketches created in the geometrical sets should be used to create object, and should be constructed in different bodies depending the geometrical set. The solid body will be trimmed by other surfaces of mating parts and other shaping parts. If working with oversized geometries, the model is easily modified and update errors can be avoided if operations are selected carefully. These bodies should thereafter be combined with Boolean operations such as add and remove. The fillets and draft should be placed outside these groups so the solid body can be replaced. Very small surfaces patches due to dress up features, such as fillets, should be avoided. Unintentional design, such as creation of small steps or offset between intersecting objects should also be avoided.

- ✓ Select positioned sketches, lines, points, planes in defined geometrical sets.
- ✓ When possible create intersect edge fillets between Boolean bodies.
- ✓ If possible keep edge fillets and drafts outside the Boolean body.

5.1 Create functional areas bodies

Solid modelling can be defined as *single body*, where all solid features are placed in one body. This is only suitable for basic models, since it can be hard to track or make features as independent. Another modelling technique is *multi body* modelling, where solid features are located in different bodies and are combined with Boolean operations. This is suitable for a complex model such as plastic parts.



5.1.1 MainBody

1. Add thickness to the A-surface of the geometrical set of the MainBody
2. Split against mating parts etc.
    - Dashboard
    - Window
    - Door
    - Roof
    - Additional lower
    - Additional upper

The thick surface command adds material normal to the surface, which may lead to an overly complicated tool split. If discontinuities on surface edges exist, these will be transferred to the solid. The edge problems have to be corrected manually (time consuming process and should therefore be avoided). Use with carefulness or consider another operation.

5.1.2 Creating ribs (FMH and Interface)

1. Create a thick pad in direction of the tooling direction to the lines of positioned sketches of the geometrical set of the Ribs.
2. Split against mating parts etc.
    - Clearance zone for clips and clip houses - base this on the skeleton lines. Make a line that goes through the MainBody and may thereafter be able to control the size of the clearance zone by the pad, and do not have to enter the sketch.
    - Rib length limit – offset the planar slab surface to desired limit, which is recommended to be controlled with a driving parameter.
    - Rib height limit – offset the planar slab surface to desired limit, which is recommended to be controlled with a driving parameter.
    - A-surface.

5.1.3 Create clips, clips house and locators

Note. If possible, use an already existing power copy or create an own if this will be used in several places.

Note: Create an outer profile in a positioned sketch with direction of the slider. Make it with longer leg than the MainBody. Make a pad.

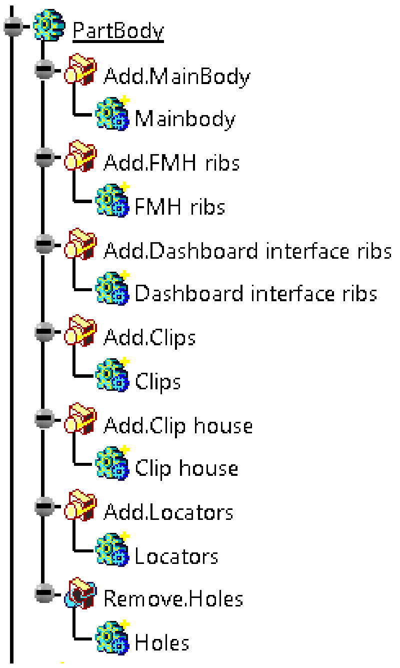
1. Create a thick pad in direction of the tooling direction to the lines of positioned sketches of the geometrical set.
2. Split against mating parts etc.
    - A-surface.

5.1.4 Create holes

1. Create a pad and select the sketch that was created in the geometrical set for holes.

5.2 Create Boolean Operation

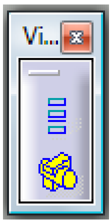
Packaging the *Boolean operation* into bodies instead of selecting feature based selections, such as pad, gives a clean tree structure and easier to replace.



- ✓ In first hand, use *Add* and *Remove* to combine the bodies and rename Boolean operations.
- ✓ Do not use the Boolean operation *assembly*, that gives double mass.
- ✓ *Union trim*. Recommended procedure is to select the face most unlikely to change in the feature. Another guideline is to choose what needs fewer objects to be selected. Union trim needs more manual work and the same result can be given by other Boolean operations. Union trim may not be appropriate for the ribs, due to slow update processes.

When recreating the solid structure use *insert in new*. The features are placed inside a new Body and ordered under an assemble operation.

There are different view modes for Boolean operation. The *solid view mode* is found in the part design workbench and there are two view modes to choose between; the *only current body* and *only current operated*.



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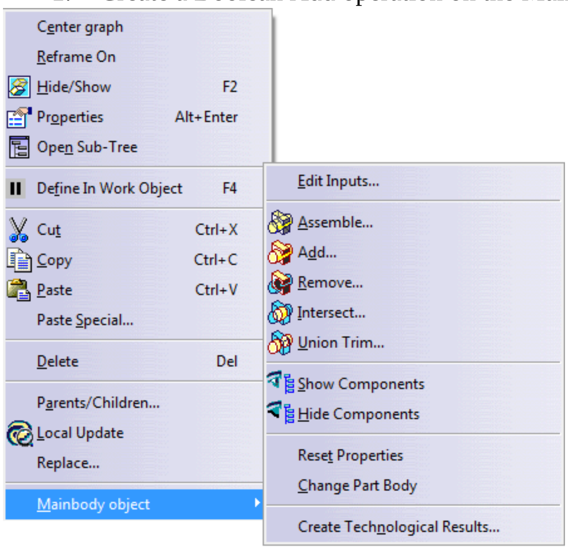
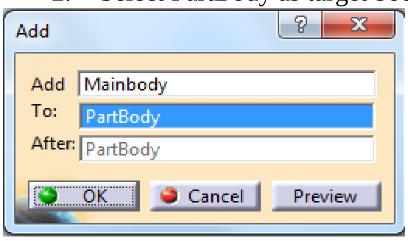
5.2.1 MainBody

1. Create a Boolean Add operation on the MainBody.



2. Select PartBody as target body.



3. Rename the Add operation to a more descriptive name.

5.2.2 Ribs

1. Create a Boolean Add operation on the ribs bodies (FMH/Interface).
2. Select PartBody as target body
3. Rename the Add operation to a more descriptive name

5.2.3 Create clips, clips house and locators

1. Create a Boolean Add operation on the body (FMH/Interface).
2. Select PartBody as target body
3. Rename the Add operation to a more descriptive name

5.2.4 Create holes

1. Create a Boolean Remove operation on the ribs bodies.
2. Select PartBody as target body
3. Rename the Remove operation to a more descriptive name

5.2.5 Draft

In CATIA there exist a function named draft, by selecting faces that should be drafted by defining it in relation to a parameter; it gives a result that is simple and easy to modify. The neutral element could either be the A-surface or the surface that is trimming the height of the object. The draft should be in the same direction as the tool direction.

Order of operations is important both within CATIA but especially for downstream processes. Always apply *draft* first and thereafter *edge fillets*. The opposite way gives a variable fillet, which may in some cases lead to a much more expensive tool if not planned for.

Create a draft

- Chose the draft parameter and control the direction of the draft
- Neutral element: A-surface or a height limit surface
- Pulling direction: Tooling direction

5.2.6 Fillets

Fillets should be done in a logical manner and not too many selections should be done at the same time. To group it the features, such as the ribs, into horizontal and vertical is a good start. Fillets require selecting sub elements and they can be many for a complex part.

Intersection edge fillets should be chosen before edge fillets. Instead of selecting features, as edges of faces, when performing fillets it is possible to select Boolean bodies and create intersection edge fillets. It is a benefit to use the intersect edge fillet since other parts that are added to that Boolean later on will also get the same fillet. This is a more robust operation and will add the fillet to all objects that is added to the Boolean body. This is a benefit if there is a need of replacing a body and the numbers of edges between the bodies are not the same.

If fillets should be at other places on the part that intersect with the Boolean, these have to be added inside that Boolean body instead.

Fillets on other places than the intersection should be done within the function group, which is the same as within the Boolean body that will intersect the other Boolean body.

Order of operations is important both within CATIA but especially for downstream processes. Create large fillets first and then smaller ones, otherwise the result will be triangular shapes which can be problematic to manufacture. Fillets created in the correct order also leads to fewer elements that need to be selected when creating fillets. The largest offset is the smallest radius, otherwise the radiuses will disappear. This can be controlled by an analysis of the surface (surfacic curvature analysis - type limited).

5.3 Clean up the model and remove unnecessary features

- ✓ No duplicated lines etc.
- ✓ Remove unused geometry
- ✓ Check that everything is renamed

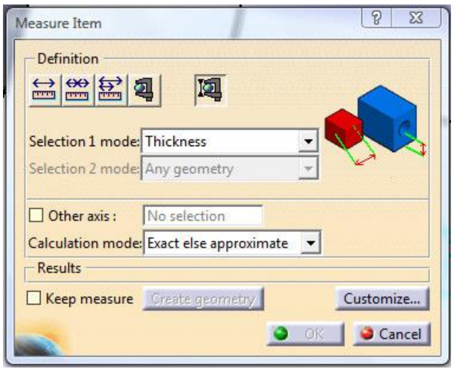
5.4 Analyse result

The complete model should then be analysed after required specification. If nothing else is required a measure of the thickness should be conducted, draft analysis and a sketch analysis. An example of the procedures are provided.

5.4.1 Measure & Thickness Analysis

Perform an analysis of the thickness to avoid warping and sink marks on the A-surface. Plastic must have an even wall thickness. To check this use the sectioning and measure tools to analyse the wall thickness in selected areas.

A simple way to perform this analysis is to use the thickness tool inside the measure item.



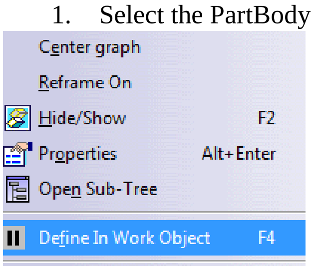
1. Select Measure the Thickness.
2. Hold the cursor over the area to measure.

NOTE! The thickness value can sometimes be displayed incorrectly depending on cursor position.

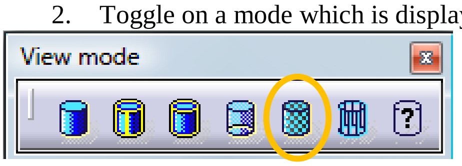
5.4.2 Draft Analysis

Another analysis that needs to be performed is the draft analysis to verify that the plastic part can be taken out of the mould.

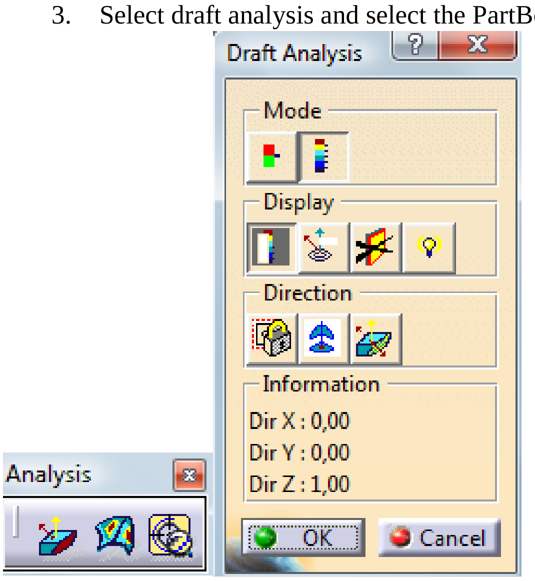
1. Select the PartBody as define in work object.



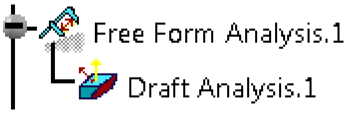
2. Toggle on a mode which is displaying material in view mode.



3. Select draft analysis and select the PartBody.



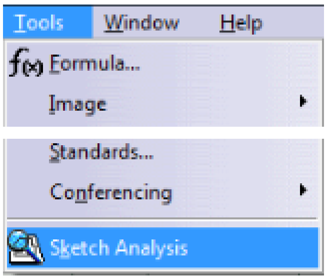
4. Select the compass under direction (by default in CATIA the tooling direction is set to Z). Drag and drop the compass onto the created tooling direction line. Analyse the results.
5. Confirm with OK in the Draft Analysis window. The results are saved as a node in the specification tree.



5.4.3 Sketch analysis

All Sketches should have the status Iso-Constrained and this can be confirmed with the sketch analysis command.

1. Select the sketch analysis under the Tools drop-down menu or the Tools – 2D Analysis sub-toolbar when inside a sketch.



or

1. A quick verification of the sketch geometry can be performed with the sketch solving status command. The Sketch Solving Status dialog box displays whether the sketch is Under-, Over- or Iso-Constrained. Sketch Analysis can be accessed directly from this dialog box.

5.5 Publish required object

The purpose of publications is to act as a folder for geometrical features and making geometrical features available to other users. Publication should only be done on part level, not assembly level. When releasing the CAD model to Teamcenter everyone can use the publication. When using a published feature an external link is created. The external links between publications are visible in Teamcenter. External references can be structured in the tree structure if there are many.

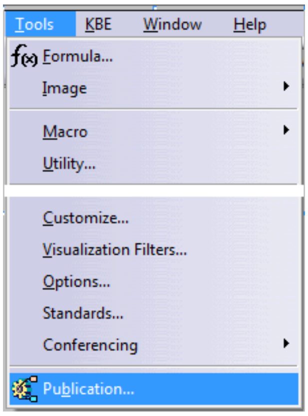
The name of a published feature will have the same name as in the specification tree when created. If the name of the feature in the specification tree is changed, the name of the published element will not be changed. Publication of parts or features needs to have the same name as the publication to be able to copy between different models. It is recommended to control the spelling, if changes is done after released it going to affect downstream process and if the spelling has to be changed, other users models will not be updated.

A published element cannot be replaced easily. This can be overcome by publishing a join of the element instead of only the element.

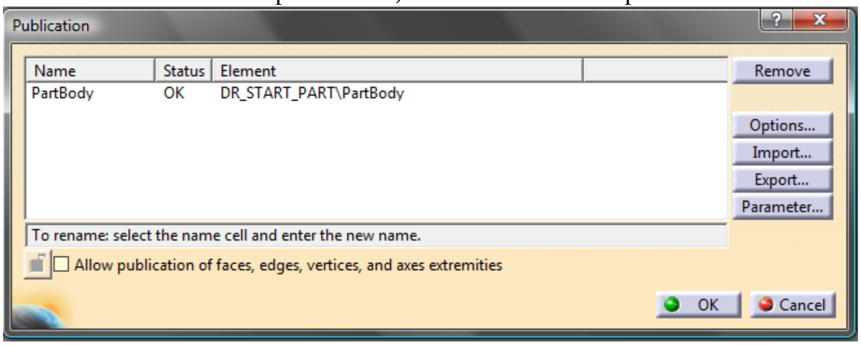
See requirements in CEVT CAD standard.

5.5.1 How to publish a element

1. Publication can be accessed through Tools->Publication



2. To create a new publication, select the feature or parameter when the Publication dialog box is open.



Note: Only published element should be copied with linked to another model.

5.6 Upload to Teamcenter

No deactivated element or useless element should be in the CAD model when *releasing* the part in Teamcenter.

See requirements in CEVT CAD standard.

5.7 Prepare for downstream process

To make a robust model not only for CAD but also CAE it is recommended to make following restriction in the design when possible.

A complex curvature shape can cause problem for CAE it is therefore strongly recommended to control the surface that is building the model. If there is holes, overlaps, gaps, or not connected as they should, it does not only affect the solid itself but also the transfer to CAE. This can be solved by analysing the surfaces, and also do joins in the right way and having common spines as much as possible. This will minimize smaller surfaces patches that otherwise can be created.

Another requirement to be followed is to avoid having a too small distance between features. Features such as crossing ribs, smaller steps and radius should be following this. The mesh is in general defined within the range of 3-5 mm as minimum element and features in the CAD model smaller than this may cause problem and should be avoided when possible.

General guideline when removing small features is that it should be the same limit geometry parameter value for small faces, short edges and sliver faces.

In the end of the work a model suitable for CAE can be created. This is in general accomplished with suppressed features such as edge fillets, chamfers and sometimes also drafts. General recommendation of procedure is to keep fillets outside sketches to simplify detection, changes, removal, and deactivation.

Triangular shapes should in general not exist, neither in CAD nor in any downstream process. This occurs when two or more elements are intersecting with each other. Using commands as edge fillets, chamfers, and splits can create these small faces, short edges, slivers etc. as consequential outcome.

In surface modelling; try to avoid command edge fillets, instead proceed with both trim and fillets simultaneously by shape fillet.

For solid modelling; try to use intersection edge fillets, edge fillets by selecting faces, or edge fillets by selecting edges.

It is recommended to create a positioned sketch in the related function groups geometrical set. It should be positioned against a plane, a line, and a point. The radius should be defined against a parameter. The sketch should then be selected in part design and a pad should be performed and placed within the function group.

Holes should be done independently so it is possible to replace, modify, or suppress.

It is recommended to observe the geometry to make sure it is possible to manufacture and to be used for CAE.

For transfer do only use the CATIA file. Otherwise it adds an extra translation step which may cause data loss. Avoid usage of dead solid.

6 References

CEVT CAD Standard

VM1233 Modelling strategy

CTM 0029/VM1563 Driving Geometry

CTM 0032/VM1565 Solid Structured Modeling

CTM 0036/VM1564 Surface Structured Modeling

VM1566 Solid Advanced Inside Outside

7 Record of revision