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Design of a 3D configuration model for electric motors

Thesis submitted as a partial fulfillment of the requirements for the degree of Master of Science in Technology.

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Abstract of master's thesis

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Abstract

Synchronous motor is a tailor made product of ABB and its mechanical design can take up to several months of time. A need of simplified layout model of motor in the beginning of design process has emerged in the department of synchronous machines. Design work of synchronous machines is performed by utilizing computer aided design. Development of computer aided design software has enabled automated creation of 3D models. Applications, which create 3D models automatically, can be called also as 3D model configurators.

The main objective of this thesis was to research, how 3D model configuring could be utilized in creation of layout models. 3D model configurator was to be created and implemented for selected synchronous motor types. Main dimension drawing of motor was also needed to be created based on the configured layout model. The available design tools for the task were CAD software NX 8 and I-Deas as well as PDM software Teamcenter.

Different 3D modeling and assembly creation techniques were studied thoroughly. Also available design tools and their features were researched. Knowledge was gained by utilizing existing literature, interviews of experts and by experimenting. Geometric data exchange methods were discovered for utilization of I-Deas models and for ensuring smooth delivery of configured layout models.

As a concrete result of this thesis, two separate 3D layout model configurators were created for selected synchronous motors. Modeling was performed in NX, but also multiple I-Deas models were utilized with the aid of STEP data exchange method. The configurable assembly was created in NX and a skeleton model was utilized to provide more robust method of constraining component models. Different user interfaces were researched and finally Options & Variants tool in Teamcenter was selected for this purpose. As a new configuration is created, user inputs values for options in Teamcenter. These option values define parameters for models in NX and also which component models are included in the configured structure. Created configurators proved to be effective in creating necessary layout models and main dimension drawings.

Keywords CAD, 3D modeling, configuration model, 3D model configurator, parametric modeling, NX, I-Deas, Teamcenter, product configuring, electric motor

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Tiivistelmä

Tahtisähkömoottori on ABB:n asiakasräätälöity tuote, jonka projektikohtainen mekaniikkasuunnittelu voi kestää useiden kuukausien ajan. Tahtikoneet-yksikössä on tullut ilmi tarve yksinkertaistetulle 3D-tilavarausmallille jo suunnitteluprosessin alkuvaiheessa. Tahtimoottoreiden suunnittelu tapahtuu tietokoneavusteista suunnittelua hyödyntämällä. Tietokoneavusteiset suunnitteluohjelmat ovat kehittyneet merkittävästi ja nykyiset ohjelmat tukevat tehokkaasti myös automatisoitua 3D-mallien luontia. Malliautomaatteja kutsutuaan myös 3D-mallikonfiguraattoreiksi.

Tämän diplomityön ensisijainen tarkoitus oli tutkia, kuinka 3D-mallikonfigurointia voitaisiin hyödyntää tarvittavien tilavarausmallien luomiseen. Työn tarkoituksena oli luoda 3D-mallikonfiguraattori valituille tahtisähkömoottorityypeille. Vaatimuksena oli myös, että konfiguraattori tuottaa 3D-malliin pohjautuvan päämittakuvan koneesta automaattisesti. Työn toteuttamiseen oli käytettävissä suunnitteluohjelmat NX 8 sekä I-deas ja lisäksi tuotetiedonhallintajärjestelmä Teamcenter.

Eri 3D-mallinnustekniikat sekä käytettävissä olevat suunnittelutyökalut tutkittiin perusteellisesti. Tietotaitoa kerättiin saatavilla olevaa kirjallisuutta hyödyntäen, asiantuntijoita haastatellen sekä työkaluja itsenäisesti testaamalla. Myös erilaiset malliformaatit tutkittiin, jotta olemassa olevia I-Deas-malleja pystyttiin hyödyntämään konfiguraattorin rakentamisessa ja jotta konfiguroitu tilavarausmalli saataisiin välitettyä eteenpäin sitä tarvitseville tahoille.

Diplomityön konkreettisena tuloksena saatiin aikaiseksi kaksi erillistä 3D-tilavaraus- sekä päämittakuvakonfiguraattoria valituille tahtimoottoreille. Mallinnustyö toteutettiin NX:llä, mutta myös useita I-Deas malleja hyödynnettiin STEP-tiedonsiirtoformaatin avulla. Konfiguroituva kokoonpano luotiin NX:ssä ja osat paikoitettiin luotettavasti skeleton-mallin avulla. Erilaisia käyttöliittymävaihtoehtoja tutkittiin ja lopuksi päädyttiin valitsemaan Teamcenterin Options & Variants -työkalu. Uutta konfigurointia luotaessa käyttäjä asettaa arvot luoduille optioille. Kyseisten optioiden arvot määrittävät mallien parametrit NX:ssä sekä sen, mitkä kokoonpanon osat sisällytetään konfiguroitavaan rakenteeseen. Luodut konfiguraattorit osoittautuivat tehokkaaksi keinoksi tuottaa tarvittuja tilavarausmalleja sekä päämittakuvia.

Avainsanat CAD, 3D-mallinnus, konfigurointimalli, 3D-mallikonfiguraattori, parametrinen mallinnus, NX, I-Deas, Teamcenter, tuotekonfigurointi, sähkömoottori

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I am also grateful for my parents and family, who have pointed me out the importance of education and also for my friends, who have inspired and helped me throughout my whole academic career. Last but not least, I want to thank my better half Raita, who has always supported me, especially during the most difficult times.

Helsinki, Finland, October 3rd, 2014

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Erkka Laukkarinen

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Abbreviations

2D	Two dimensional
3D	Three dimensional
ABB	Asea Brown Boveri – Multinational engineering company
AMZ	Product code of ABB’s synchronous motors
API	Application Program Interface – A code, which allows two software programs to communicate with each other
ASCII	American Standard Code for Information Interchange – Character encoding, which is used to create STEP-files
CAD	Computer Aided Design
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing
CSG	Constructive Solid Geometry – Most common method of constructive CAD modeling
D-END	Drive end. The shaft comes out from the frame in D-end and motor can be connected to drive the selected application.
DOF	Degree of Freedom – Three translational and three rotational directions are available for 3D models in CAD software
EXPRESS	Standard data modeling language for product data, used in STEP standard
FEA	Finite Element Analysis – Computational tool for performing engineering analysis such as structural analysis and pinch analysis. FEA utilizes FEM in its processes.
FEM	Finite Element Method – Is a numerical technique, which is used for finding approximate solutions to boundary value problems for differential equations.
I-Deas	Integrated Design Engineering and Analysis Software – CAD program developed by SDRC and later acquired by Siemens PLM Software
IGES	Initial Graphics Exchange Specification - Data Exchange Standard developed by National Institute of Standards and Technology
ISO	International Organization for Standardization

JT	Data exchange file format of Siemens PLM Software and standardized by ISO organization
MAM	Master Assembly Model – Is used in Top-Down assembly design process to represent the function of parts, the layout, as well as the detailed geometry of parts.
N-END	Non-drive end. Opposite of D-end, the shaft does not come out of the frame in N-end.
NC	Numerical Control – Is used as a programming method of automated machining tools.
NX	CAD-program developed by Siemens PLM Software
PDM	Product Data Management – System, which manages the product data related to engineering of the product
PLM	Product Life Cycle Management – System, which manages the product data in larger scope than PDM
PTS	Product Structure Editor – Tool to create user interfaces for parametric models in NX.
SDAI	Standard Data Access Interface – Can be referred to API. SDAI tells how application data, which is created according to data model given in EXPRESS, can be worked on.
SET	Standard d´Échange et de Tramsfer - Data Exchange Standard developed for purposes of French airplane industry
STEP	Standard for Exchange of Product data
STEP-file	A file type, which is used to exchange data that is consisted according to STEP standard. STEP is coded by using ASCII.
STEP-XML	A file type, which is used to exchange data that is consisted according to STEP standard. STEP-XML is coded by using XML.
STL	Stereolithography file format, which is non-standardized data exchange method and commonly used especially in rapid prototyping.
TC	Teamcenter – Product Life-Cycle Management Software developed by Siemens PLM Software

TDCD	Top-Down Component Design – In top-down assembly modeling the assembly is divided into parts and these parts are designed in more detailed level by using TDCD sub-process.
VDA-IS, VDA-FS	Verbend der Deutchen Automobilindustrie - IGES Subset/Flächen Schnittstelle - Data Exchange Standards developed for German automobile industry.
VSD	Variable Speed Drive – Electric motor type, which driving speed can be varied. Often used in demanding applications in process industries, and applications where variable speed delivers clear benefits.
WAVE-GL	WAVE Geometry Linker – Tool to copy geometry between models in NX.

Terms

3D Configurator	3D Configurator can be used to configure 3D model variants of a configurable product by inputting the needed parameters for model creation.
Bottom-Up method	In bottom-up design process detailed parts are designed first and an assembly is constructed of these parts.
Configurable product	A product family, which is a combination of product variants.
Configuration model	Original parametric and configurable model, which is used to produce configured models (model variants). Configuration model can be referred to model configurator.
Constraint	3D model can be defined exactly with the aid of constraints. Constraints are used to define relations between elements in 3D model or relations between different part models in an assembly.
Expression	In NX the relations between design elements are called expressions. New variant models can be configured by changing expression values of configuration model.
Generic structure	Can be referred to configuration model. From generic structure it is possible to produce variant structures.
Instance	Intelligent duplicate of an object in 3D assembly.
Modeling kernel	Geometric modeling kernel is a 3D solid modeling software component used in computer-aided design systems.
Parametric modeling	In 3D modeling relations between design elements are defined by constraints. In parametric modeling the system is able to save these constraints and model can be modified by modifying the constraint.
Parametric system	Modeling system, which solves constraints by sequentially solving equations. Each placed value is computed as a function of earlier placed values.
Product family	Product family is a group of products derived from a common product platform. Products of certain product family have always some similar features, for example they can be used for the same purpose.

Product variant	Product variant can be considered as different version (configuration) of certain configurable product. It can be a combination of variant, optional and parametric components.
Rapid prototyping	Group of manufacturing techniques, with which a physical object can be manufactured based on a 3D model. For example, 3D printing and other additive layer manufacturing technologies.
Skeleton modeling	Modeling technique, which can be utilized in top-down design process and in other assembly modeling as well. The geometry or position of parts in an assembly is defined by skeleton model.
Skeleton model	Is a layout or a base model of an assembly. Skeleton model may be used as a base geometry for other parts in an assembly or it may just define the position of parts in an assembly.
Subassembly	An assembly model, which is used as a part of a bigger assembly.
Synchronous motor	Alternating current (AC) motor, in which the rotation of shaft is synchronized with the frequency of the supply current.
Top-Down method	In top-down design process a functional sketch of a complete assembly is created first. The sketch is divided into sub-functionalities and parts, which are then designed in more detailed level.
Variant	Is something, which form differs only slightly from the form of something else. Different configurations of configurable product are product variants. Also configured 3D models are variants of original configuration model.
Variational system	Modeling system, which solves constraints by coupling or grouping equations that describe constraints and solves these coupled or grouped equations simultaneously.

1 Introduction

1.1 Background

ABB is one of the largest engineering companies operating mainly in the power and automation technology areas. ABB has a factory in Helsinki as well and one of the main products in Helsinki is synchronous motor. Synchronous motor is a tailor made product, but it has a modular structure and a lot of standard components are used. Every high voltage motor project is designed separately to meet the demanding customer and application requirements.

Usually designer spends from one month up to six months for a mechanical design of one motor project. Mechanical design department of synchronous machines is using I-Deas NX as its main CAD (computer aided design) software at the time this thesis is written. However, the department is going to migrate to Siemens NX CAD software, which enables much wider usage of 3D functions.

Normal sales process proceeds so that a salesman discusses with a customer about the design of a machine. According to what is agreed, a main dimension drawing of the machine is created by the designer of the project. The main dimension drawing is created with 2D design tools and no 3D geometry is used as a base of each drawing. During the actual design work, the designer may create a 3D assembly model by using existing part models or models, which are created for the actual project. This takes a lot of time and the final 3D assembly model can be finished, as the actual design work is completed.

1.2 Research problem and objective

Synchronous motors are used in different kinds of industrial applications. Motors are used for instance in chemical industry, power plants, mining industry, metal industry, textile industry etc. What is common for these applications is that motors are often operating in really constricted spaces and conditions. Because of this, it is really important to explore that the motor can be easily mounted on its place.

Nowadays, 3D models are used for spatial analysis, such as exploring mounting, and that is why customers of ABB want to have a 3D layout model of the final product even before the actual design work is completed. Because the final 3D assembly model is normally produced during the design work, some faster method of creating necessary 3D layout model is needed. For this task the most practical solution is to have a 3D layout model configurator, which produces simplified 3D assembly model for the customer, as all the basic features and dimensions of motor are agreed.

Another purpose for this layout model configurator is that main dimension drawing of motor can be created based on the layout model. Main dimension drawing is a technical drawing, which presents all the important outer dimensions of motor. If main dimension drawing of motor is created by using only 2D methods, as it is nowadays, the revising is really laborious as changes occur to the design. Possibility of making mistakes, while revising the drawing, is significantly higher and some geometry may be difficult to present in correct manner. Because of these reasons, a 3D model is needed as a basis of main dimension drawing.

The main objective of this thesis is to research, how a 3D layout model could be created effectively with the available design tools in the beginning of design process. As a concrete result, a 3D layout model configurator should be constructed for selected VSD (variable speed drive) synchronous motors. Also, it should be possible to configure the main dimension drawings based on the created layout model. The configurator is to be used in early stage of each motor project to offer customer the needed 3D model of machine. Creating models and main dimension drawings with configurator should not cause significantly extra work for the designer. Customers can use configured 3D layout models in their own purposes, for instance in spatial explorations.

1.3 Scope of the study

3D layout model and main dimension drawing configurator will be produced for synchronous motor types, which can benefit the most of this kind of configurator and are most suitable for configuring. The configured models should present all the outer dimensions correctly as well as dimensions, which are important for mounting the machine. The geometry should be presented as it is presented in main dimension drawings at the moment. Also applicability and possible development of layout model configurator for other motor types will be examined.

Different modeling techniques, user interfaces and methods of creating 3D configurators are researched and the best methods are implemented into creation of configurator. Available tools are limited to CAD software I-Deas and NX 8, PDM software Teamcenter and other software, which are used in ABB Synchronous Machines. The future development of created configurators is extremely important and one objective of this thesis is to research and instruct how it should be performed to enable effective usage of configurators.

1.4 Research methods and material

3D modeling and assembly modeling methods are examined by using literature and other available sources of information. The structure of a variable speed drive motor is analyzed for creation of layout model configurator. In analysis, the internal documentation and the knowledge of experts in ABB are utilized. Different data exchange methods of 3D models are researched to find out how existing I-Deas models could be utilized in the process and how the created layout models can be delivered to the customer. Research methods for learning the usage of available tools and configurator creation methods are existing literature, benchmarking, interviews of experts and self-experiment.

Some research about the subject has been done already in the department of Synchronous Machines. Especially knowledge about the usage of tools and methods of creating the best possible configurator can be gained by interviewing specialists and by using the documents inside the company. R&D department of Synchronous Machines has had NX as their main CAD software for several years, so numerous engineers have already developed expert level skills in using NX. Contact with R&D department is utilized for getting knowledge how NX could be efficiently utilized in this project. Retailer of Siemens engineering software in Finland provides also strong support for ABB in usage of NX 8 and Teamcenter and this contact is employed in creation of layout model configurator as well.

2 3D modeling

By the mid-1990s the computational power had expanded sufficient enough to enable creation of three dimensional models. Since that 3D modeling has been the basic tool of mechanical design. 3D design means that products are designed in three dimensional forms with the aid of CAD (computer-aided design) software. Designed product models look like real manufactured products and they can be examined in three dimensional space. In the software, models are given all the physical and mechanical features as the real designed product would have. 3D model makes it easier to perceive the shape of the product and the model can be used for many kind of analysis, for example structural and spatial analysis. (Schoonmaker 2003: 169) (Tuhola 2008: 17-20)

This chapter begins by listing the advantages of 3D modeling. The various methods of creating geometric models throughout the history are presented in chapter about geometric modeling. 3D modeling techniques or 3D model representations can be divided into categories in many different manners, but the most common one is the categorization into boundary/shell modeling and solid modeling. The difference is that boundary models represent only the outer surfaces of the object and do not have volume whereas solid models are more like the objects in reality.

All the modern CAD software use method called feature based modeling, which has added more intelligence into 3D model creation. Feature based modeling is presented in chapter 2.3. Finally, an attribute of CAD software, which enables the creation of configurable models, is presented. This attribute is called parametric and variational modeling.

2.1 Advantages

There are many advantages in 3D modeling compared to traditional 2D modeling. When designer is able to use 3D software effectively, the design work can be done much faster than with traditional 2D methods. The difference is not enormous, if designed parts are really simple, but it grows as they turn more complex. For example design of complete product family is many times faster with the aid of 3D design tools. (Laakko 1998: 32)

Usage of 3D design methods decreases the amount of engineering mistakes and makes visualization easier. 3D model is a full mathematical model, which cannot contain impossible geometry. In 2D design there is nothing that constrains creating impossible geometry. Created geometry is also easier to perceive in 3D model. With the aid of 3D tools, it is simple to check, if two parts can be assembled together or not. Technical 2D drawings can be sometimes hard to understand for other people than engineers, so with the aid of 3D models, all the people participating product development can understand how the final product is going to look like. Also all the 2D views created from 3D model look automatically the same as the model looks from certain view. When only 2D design methods are utilized, all the necessary views have to be drawn separately and possibility of mistake is obvious. (Laakko 1998: 33)

3D design enables a lot of analyzing and calculation potential, which is not easy to execute or is not even possible with 2D design. For example volume, mass, dimensions and center of gravity of designed product can be calculated extremely effortless. FEA (Finite Element Analysis) has taken advantages of 3D design to totally new level. Many 3D design software or software packages include this possibility of FEA. Finite element analysis means that designed 3D model is divided into meshes (finite elements) and certain calculations by using FEM (Finite Element Method) are executed. By adding boundary conditions and loads, providing material properties and specifying the type of analysis many calculations and simulations can be done. For example different structural and thermal phenomena can be discovered as well as fluid flow and electromagnetics. (Schoonmaker 2003: 170-171) (Lee 1999: 214-233)

Intelligent 3D models bring new kind of efficiency into design work. Intelligent models usually include adding parametric relationships, entering equations, constraints, etc. to the model. These features enable changing dimensions or parts so that the base geometry does not change. Generic part model is a simple method to generate different kind of variants from certain model. This can be utilized for example to model different products of a product family. Intelligence can be combined also to the versatile usage of 3D models. Same part model can be used in many assemblies and when the original åart model is changed, these changes can be easily updated to all the assemblies, in which the part model has been used. Compared to 2D design, the workload is considerably smaller, because the changes can be automatically updated to all the drawings as well. (Schoonmaker 2003: 171)

As a conclusion, numerous advantages of 3D design can be listed. Compared to traditional 2D design, 3D design is faster, more effective and more precise method of engineering design. It eases visualization and has plenty of built-in calculation and analyzing properties, which can be utilized in design processes. Intelligent models and possibility of updating all the changes automatically in all the documents saves a lot of time. Also design mistakes decrease with the aid of 3D modeling.

2.2 Geometric modeling

Purpose of geometric modeling is to create a data structure, which presents designed physical shape. In other words geometric model is a description geometrical objects' shape. Geometric models are created by using geometric modelers, for example 3D design software. These systems provide similar environment to the real environment, in which the physical model would be created and naturally manipulated. The designer deforms, adds pieces on and cuts pieces off the visual model just like the model would be designed out of clay in the real world. In practice, the shape of real geometrical object is impossible to present precisely because of its complexity. Geometrical models designed with 3D software are abstracted, so that the shape can be expressed mathematically. Three different levels of geometric modeling can be separated: reality, mathematical model and presentation. These are presented in following figure 1. (Laakko 1998: 39) (Lee 1999: 101-102)

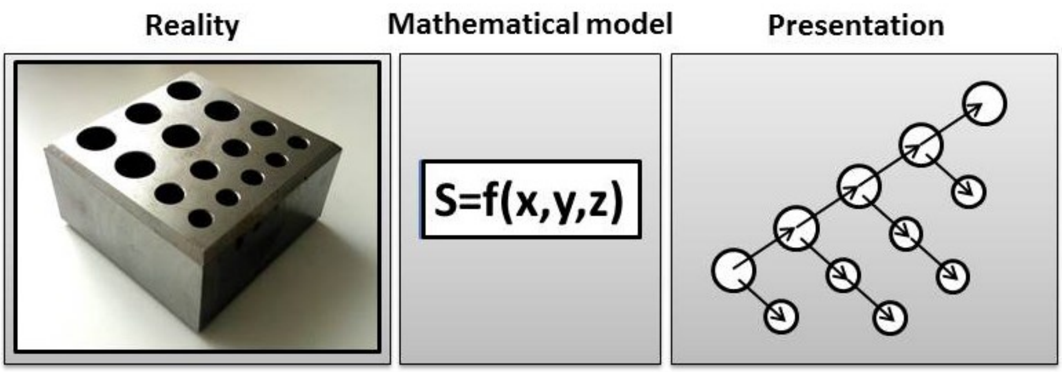


Figure 1. Three different levels of geometric modeling.

Wireframe modeling, which is also known as *graphical line modeling*, is the most primitive way of representing geometric model. This method was widely used, when geometric modeling was first introduced. In wireframe modeling the shape is represented by showing its characteristic lines and end points. In mathematical description there is a list of curve equations, coordinates of the points and connectivity information corresponding the curves and points. The problem of wireframe model is that it can be really ambiguous and the mathematical description does not contain information about the inside and outside boundary surfaces. This problem is demonstrated in figure 2 below. (Laakko 1998: 40) (Lee 1999: 102)

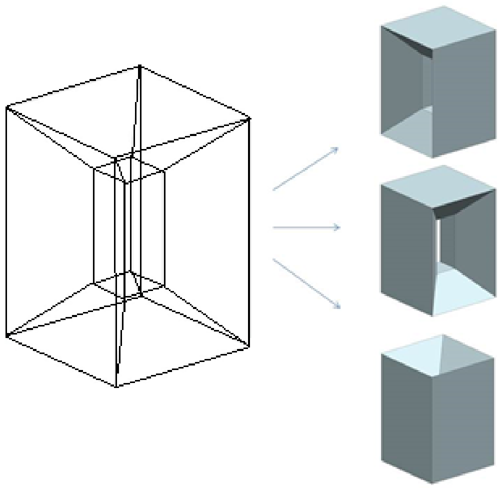


Figure 2. Ambiguous wireframe model and alternative solid models of it.

Surface modeling is more advanced method of geometric modeling than wireframe modeling. In addition to characteristic lines and end points of wireframe model, the mathematical description of model includes also surface information. Although surface model is not totally unambiguous model and it cannot be used for all the calculations or finite element analysis. Surface models are used to present complex surfaces, for example geometry of automobiles, airplanes, ships, castings and forged pieces. Surface modeling is also used by industrial designers to create impressive looking and demonstrative models. The visual model can be used to evaluate aesthetics, and the mathematical description to design tool paths for NC (numerical controlled) machining. Surface models are created by combining surface patches, which are limited by edge curves. Edge curves are usually parametric curves, which can be formed of input points by different methods. In modern CAD software it is possible to create surface models also by sweep presentation method, which is presented below, as solid modeling is presented. An example of surface model is shown in following figure 3. (Laakko 1998: 40-46) (Lee 1999: 103)

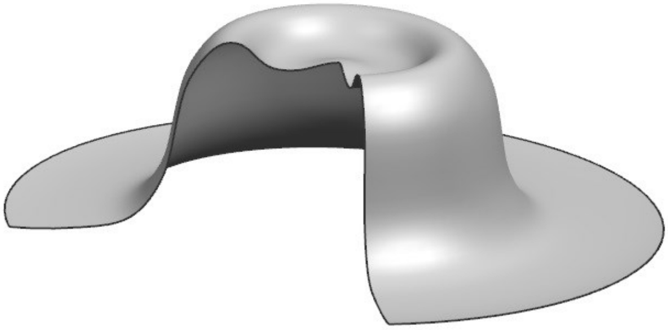


Figure 3. Surface model.

Solid modeling was developed to fulfill the deficiencies of mentioned graphical methods. Solid models have a closed volume and they tend to be "perfect" models of designed objects. As earlier was mentioned, it is impossible to model objects as accurately as they appear in reality. Perfect model means that the model is sufficient to answer algorithmically to all the questions about the geometry of the object. Solid modeling techniques offer advanced methods for creating 3D geometry. Solid models are used for CAM (Computer Aided Manufacturing) and they can offer good visual quality. Solid modeling revolves around a fact that objects can be seen as a subset of three dimensional Euclidean space and described by a mathematical model. Solid model presentation is based on this mathematical model. (Laakko 1998: 46-47) (Lee 1999: 104-105)

There are different categories of presenting or creating geometry in solid modeling: decomposition models, boundary models, constructive models and sweep presentations. *Decomposition model* consist of smaller elementary primitives, which are combined together. These elementary primitives can be different size and shape depending of the used method. In FEM calculations these primitives are used to approximate the geometry in suitable form for analysis. An example of decomposition model can be seen in figure 4 below. (Laakko 1998: 47)

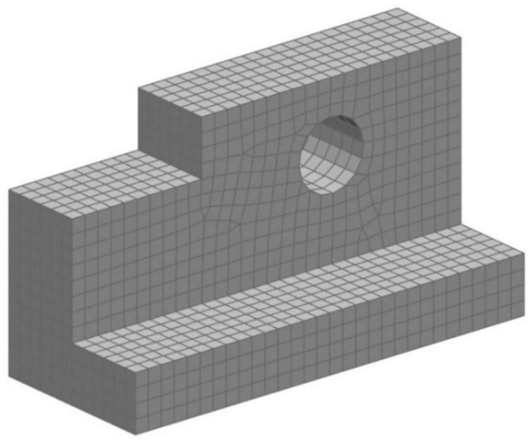


Figure 4. Decomposition model.

Boundary models (boundary representation, B-rep) consist of faces, which together form the surface of presented object. Usually, the shape of each face has a compact mathematical representation. This means that face lies on a single planar, quadratic, toroid or parametric surface. Curves, that limit the faces, are divided into edges, which are presented by their vertices. So, the geometry of an edge is presented as a curve, which has mathematical a presentation. Usually all the information of the shape of entity can be bundled under term geometry, whereas information of their connections can be bundled under term topology. Example of an exploded boundary model and its hierarchic data structure is presented in following figure 5. As can be seen, boundary model consist of faces, which together represent the solid object.

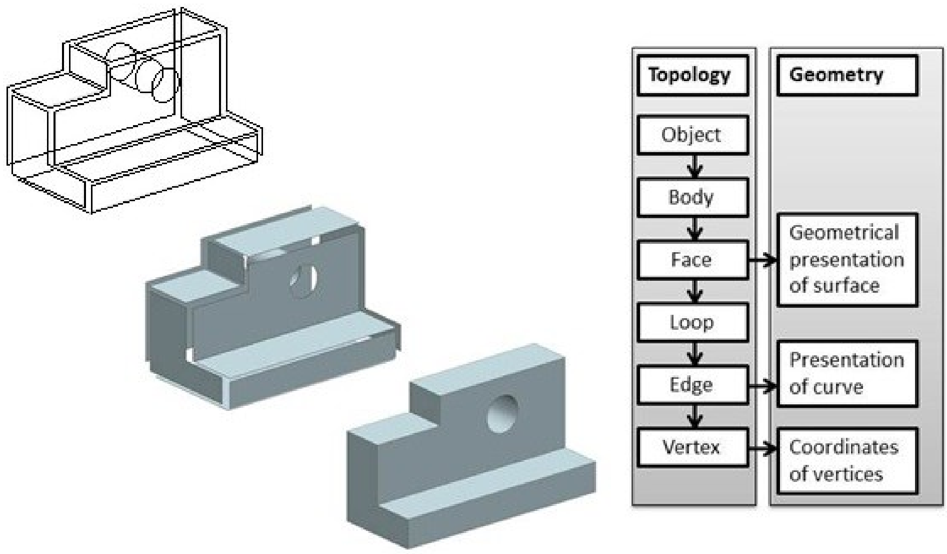


Figure 5. Exploded boundary model and data structure. (Laakko 1998: 51)

Constructive models are based on presenting the object with terms of set theory. Object can be seen as a set of points, which consists of primitive point sets. There are different methods of creating constructive models. The most common one is CSG (constructive solid geometry), which can be called building-block geometry as well. CSG models are based on limited primitives, which are defined as a combination of a half-space. These primitives are usually objects, which have simple shape, for example cuboid, cylinder, sphere, pyramid, cone and torus. These primitives are much bigger than the primitives in decomposition models. It is possible that one primitive forms one part or shape in the complete model. User can modify shape and size of the primitives by specifying parameters. With Boolean operations (union, difference and intersect) the model can be constructed of primitives. This is why models can be called also as Boolean models. The model can be modified by using different transformations (translation, rotation, scaling, symmetry and reflection etc.) as well. CSG models are presented in a form of a tree. Used primitives are the leaves and used operations are the branches of the formed tree, as can be seen in following figure 6. (Laakko 1998: 50) (Mortenson 1985: 461-469)

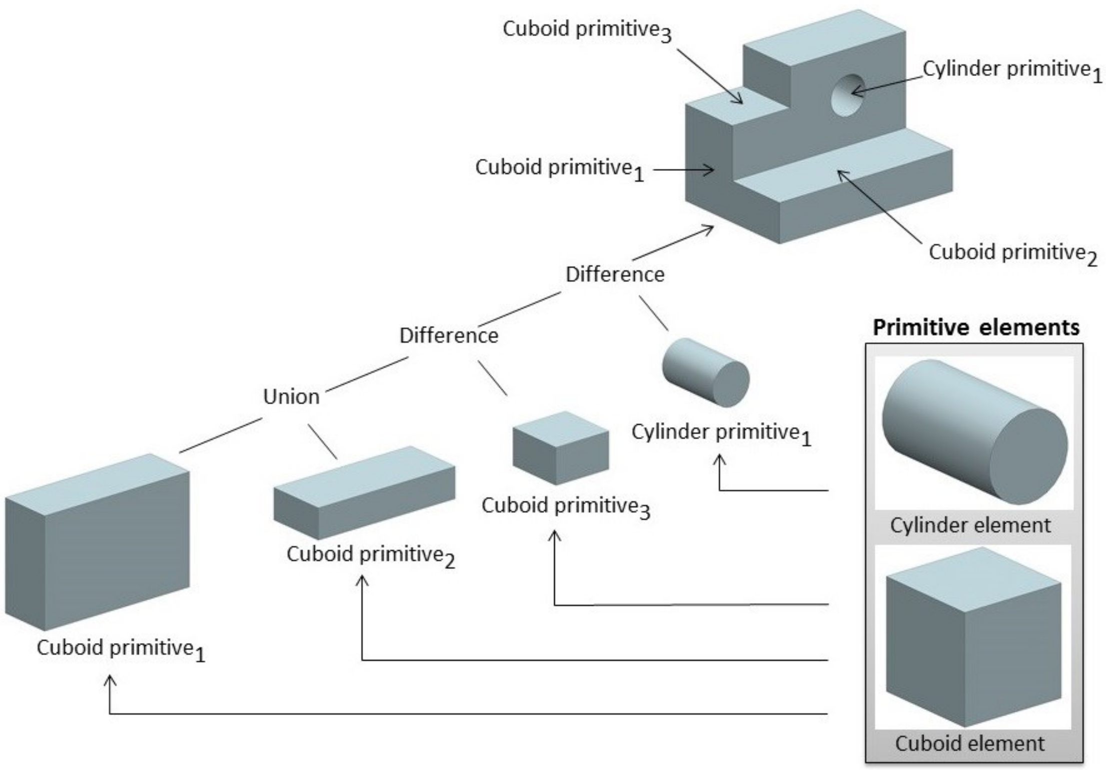


Figure 6. Constructive Solid Geometry - Tree model (Laakko 1998: 50)

Fourth method of creating solid 3D geometry is *sweep representation*. The creation of sweep representation is simple: a two-dimensional boundary is created and swept into third dimension along a vector or certain path. Sweep representation is also called as *"constraint-based solid modeling",* as the created 2D sketch provides constraint to the added dimension. Sweep creation can be divided into three types: linear, nonlinear and hybrid. In linear sweep, the path is linear or circular vector described by a linear. Linear sweep can be also divided into translational and rotational sweep. In nonlinear sweep, the path is described by a higher-order equation. Hybrid sweep combines linear and nonlinear sweeps via set operations and enables creation of complex sweep representations. The following figure 7 presents the different methods of creating sweep representations. The sweep representation is considered more intuitive than CSG and is therefore popular in modern CAD software. (YZU Optimal Design Laboratory)

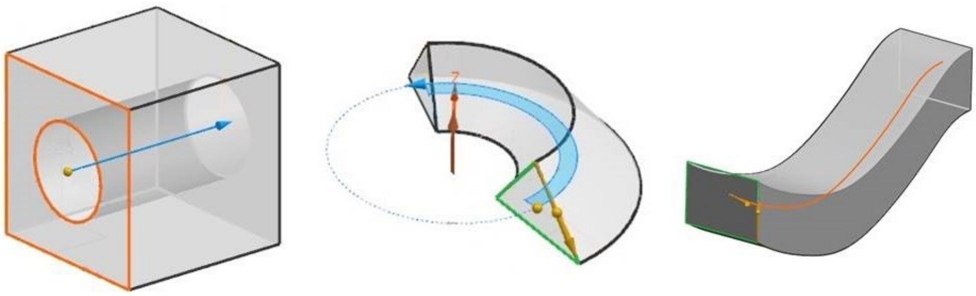


Figure 7. Sweep creation methods: translational, rotational and nonlinear.

The most of the mentioned 3D modeling methods are not used similarly in modern CAD software as they were at the time of their discovery. In modern CAD software the models are created mostly by sweep creation method. Also some design features maybe added with methods similar to CSG creation. The created models can be though presented in earlier 3D model representation formats. For example, it is still possible to represent created models as wireframe, but wireframe modeling is not used as model creation method anymore. Surface modeling and boundary representations are used for certain purposes, but they are also not created as they were earlier. Models can be also turned into decomposition models for FEA purposes, but the model is not constructed in that manner.

In modern 3D modeling, the steps of model creation are relatively similar each time. In the first step designer selects or creates a sketch plane. Sketch plane can be an existing face of a part or an imaginary flat plane, which is shown on the screen. The second step is to create 2D geometry on the selected sketch plane. In the third step the created 2D geometry is expanded into third dimension with the aid of selected 3D method. Expansion can happen with earlier mentioned translational, rotational or nonlinear method. First three steps of model creation are shown on the right in figure 8. The procedure can be used also for removing volume from existing model. Many CAD software offer also design elements, which are similar to primitive elements in CSG model creation. For instance holes, bosses, fillets and chamfers can be added to the geometry created with sweep creation methods and their dimensions manipulated as wanted. In figure 8 holes and chamfers are added to the model as design elements. The geometry creation can begin also by adding certain design element as a base shape, such as cylinder or cuboid, and by modifying it with other tools. (Schoonmaker 2003: 189-191) (Tuhola 2008: 26)

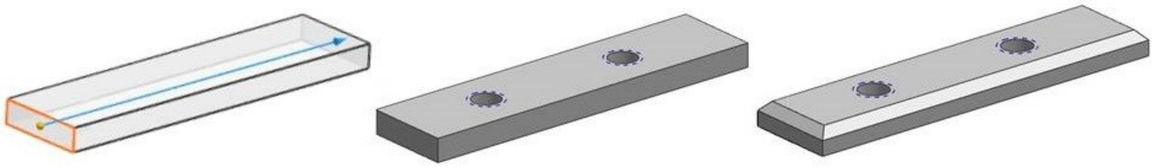


Figure 8. Modeling with modern CAD software.

However, the presented geometric modeling techniques contain a lot of deficiencies in aspect of product modeling, if they are only able to describe the geometric shape. Information related to the geometric shape varies during the design process and it should be possible to modify the geometric model throughout the process. In the beginning of the design process, when functional design is constructed, the designer is only concerned about the geometry, which has significant impact on the functionality of the product. Yet, there is no need to be concerned about minor details. Later when the basic geometry is designed, so that it fulfills the requirements of functionality, design work becomes more detailed. For example, manufacturing aspects have to be considered and geometry has to be changed so that manufacturing is possible. During the whole process geometric information is constantly changed and this is not taken into consideration in simple geometric model creation. (Shah 1995: 92-93)

The data of geometric models is at microscopic level. The problem is that decision making and reasoning of engineering process needs macroscopic entities as well. In other words, simple geometric models are not suitable to be used as design tools throughout the whole engineering process, because of their detailed structure. Geometric models consist of precisely dimensioned geometric entities, so the geometry of the part being designed has to be known accurately in advance. (Shah 1995: 93-96)

Another problem, which is related to microscopic data, is that geometric models can not differentiate design intent information. For example, geometric model does not understand, if geometry is there to satisfy interface constraints or to satisfy functional requirements. Design rational representation, which uses higher-level entities, is needed to capture this type of information. (Shah 1995: 94-96)

Geometric model, which has no design intent information, is not able to provide support for editing either. Following figure 9 demonstrates this problem. Designer should grow the thickness of the plate. The depths of recesses should remain the same and the hole should look the same as it does now, but with new dimensions. With simple geometric modeling tools, the designer should fill the hole and recesses, then thicken the plate and re-create all the attached geometric elements. Instead of specifying microscopic entities with rigid dimensions, but creating geometric constraints between high-level entities, such as holes and recesses, the structure could be changed easily just by giving the new plate thickness. (Shah 1995: 94-96)

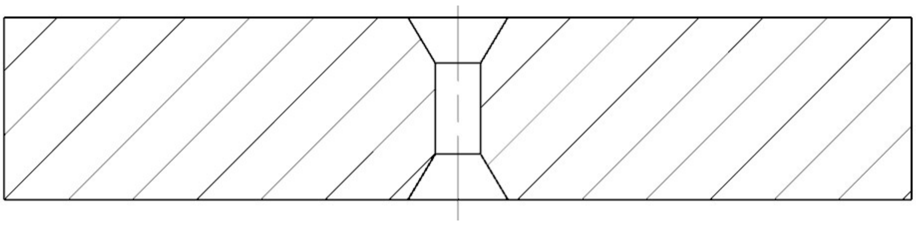


Figure 9. Rigid definition in geometric model. (Shah 1995: 95)

The high-level entities such as “hole”, “distance between holes” and “sheet thickness” may be vital for manufacturing, but they are not available in simple geometric models. Also some other product information, which is needed in certain phases of product design process, may not be included in model consisting of only microscopic primitives. For instance, tolerances are needed for manufacturing, but to utilize them in models, the higher-level entities are needed. In conclusion, using simple geometric models leads to underspecification. (Shah 1995: 92-93)

All the above mentioned problems lead to a fact that traditional geometric models were not separately the most suitable ones for supporting the engineering work. Because of this, better methods have been developed. The following chapter about feature based modeling presents how it has been possible to include the design intent in models, make them easier to be modified and all in all make them match better the purposes of engineering.

2.3 Feature based modeling

Feature based modeling is a method, which has turned 3D modeling more useful for design work. Feature based modeling uses high-level modeling entities, which are called features, instead of simple geometric primitives. These high-level entities correspond better the real life design work than mathematical primitives. Nowadays, all the most commonly used CAD software are founded on feature based modeling. Features can contain a lot of additional information attached to the feature taxonomy than just geometric shape: dimensional intelligence, manufacturing requirements, tolerances, material information etc. This information supports the whole engineering process. (Laakko 1998: 83)

Feature itself can have different definitions or meanings. As it describes the geometry of a model, it can be seen as a set of geometric primitives or other features, which are needed to be handled as a set. According to Shah & al. in the book of "Parametric and Feature-Based CAD" geometric feature is characterized as following (Shah 1995: 97):

- A Feature is a physical constituent of a part
- A feature is mappable to a generic shape
- A feature has engineering significance
- A feature has predictable properties

In object-oriented programming, features are objects, which are described by different kinds of attributes and relations. Attributes are like characteristics or properties of feature, which define the feature itself. These attributes are for example geometric parameters, dimensions, tolerances and manufacturing information. Relations are also like attributes, but they affect between features. These relation attributes can have information about relative positioning, geometric constraints or compatibility. (Laakko 1998: 83) (Shah 1995: 98)

A feature model is a data structure, which consists of different features. CAD software is able to save the data structure of features as a design history, which is usually presented in a tree form and is useful for many purposes. It is possible to return back in design history and design the structure differently, if that is wanted. Changes can be made to the attributes of earlier created features or some features can be even deleted, if they are not related to later created features. The design history can be also used for analyzing, why the created design did not work. With the aid of design history, it is possible to save different configurations and versions of created designs. The following figure 10 shows 3D model, which is created by using feature based modeling. In the design history it can be seen that the model consists mostly of features, which are created using earlier mentioned sweeping method. Extrudes are translations sweeps, in which certain 2D sketch is protruded along selected vector. Also other features, such as additional datum planes, chamfers and blended edges, are used to construct the model. (Laakko 1995: 83-97) (Stroud & Nagy 2011: 423-432)

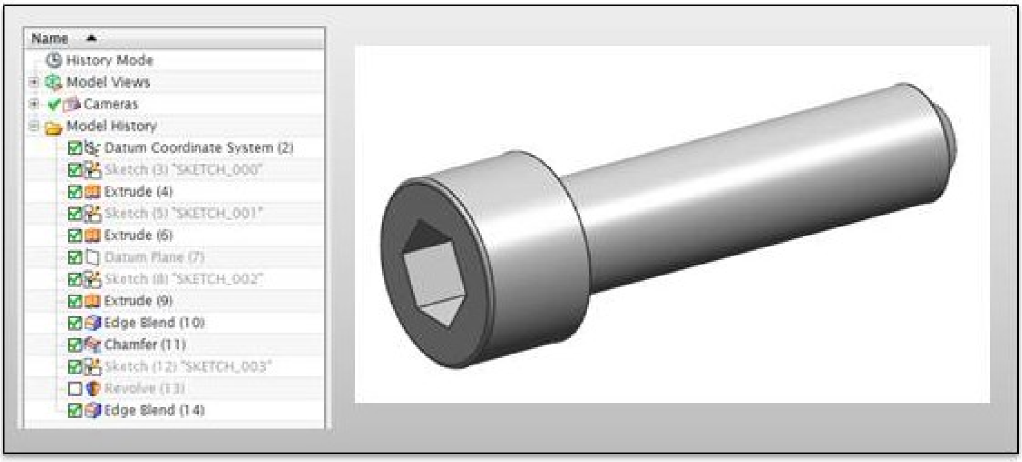


Figure 10. Feature based model and its design history.

2.4 Parametric and variational modeling

All the modern CAD systems save the relations, which are defined during the modeling work. In this context, relation means the dependency between two details in a product model. In product models these details can be for example primitives, attributes or features. Basically, relations of geometric model describe the constraints between details in the model. Because of this feature of CAD systems, it is much easier to edit created models. Editing can be done easily by changing the relation. Two examples of relations or geometric constraints in 3D model are presented in following figure 11. On the left side there is a constraint between line-primitives and on the right side there is a constraint between two features. (Laakko 1998:56)

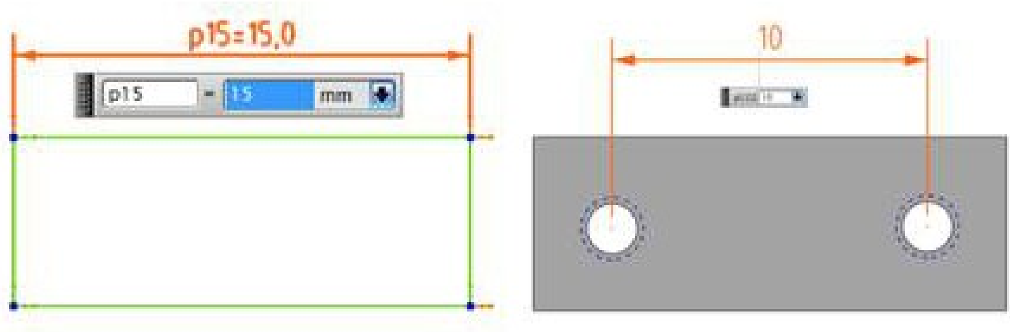


Figure 11. Created geometric constraints (relations) in a 3D model.

The techniques that modeling kernels use for solving geometric constraints in 3D models are divided into two groups: parametric and variational. Terms "parametric" and "variational" have been used almost interchangeably in technical and commercial contexts. For the end user it does not make difference, which technique the modeling kernel is using. It is impossible to define from the outside, which method the system is using. The overall design processes are similar: (Shah 1995: 79) (Laakko 1998:56-57)

1. User creates a model by using ordinary geometric or solid modeling operations. The result is a model, which contains desired geometric elements and connectivity between them, but does not contain dimensions.
2. User creates relations between model entities, by giving them geometric constraints. The constraints specify the desired mathematical relationships

between the numerical variables of model entities. For two dimensional designs, the typical geometric constraints are presented in figure 12 below.

3. Modeling system evaluates the constraints and as an outcome there should be evaluated model, in which the given constraints are satisfied. If the system of constraints cannot be satisfied, the user should be informed with a warning.
4. It is possible to create new kind of variant by changing the values of the constrained variables. A new instance of the model will be created after each change and after re-executing the constraint solution procedure. User can also add new constraints or remove existing constraints when creating new variants.

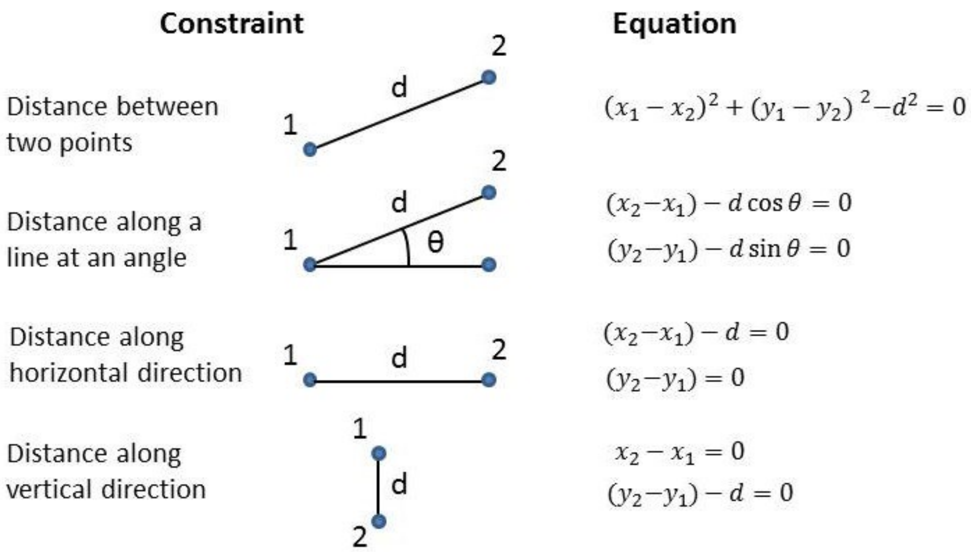


Figure 12. Typical geometric constraints for 2D designs. (Shah 1995:80)

Many systems use hybrid system, which means that both, parametric and variational methods, are used and that is why the difference between them is also blurred from the technical point of view. In the book of "Parametric and Feature-Based CAD" from Sami J. Shah and Martti Mäntylä, the terms parametric and variational are clearly summarized as following:

"*Parametric systems* solve constraints by applying sequentially assignments to model variables, where each assigned value is computed as a function of the previously assigned values. Unlike procedural systems, the order of the assignments is flexible, determined by a constraint propagation algorithm." (Shah 1995: 83)

"*Variational systems* solve constraints by constructing a system of equations representing the constraints, and solving all constraints of the system simultaneously on the basis of a numerical equation-solving procedure or some equivalent method." (Shah 1995: 83)

A simplifying example is also given in the book by using quadratic equation:

$$x = \frac{-b \pm \sqrt{(b^2 - 4ac)}}{2a} \qquad\qquad ax^2 + bx + c = 0$$

The form of the equation on the left side can be used to compute the value of variable x directly by giving the parameter a, b and c. Equation is explicit and parametric systems work by scanning the constraints and using predefined solution methods, such as this equation. (Shah 1995: 83)

The form of the equation on the right side cannot be used directly to compute the value of variable *x*, because it should be "solved" first for the x. This form is called implicit. Multiple solutions are possible for the equation. For instance, if variable *a* should be computed instead of *x*, the form of the equation on the left side would be useless, but the form on the right side, could be used for the task. The implicit equation could be actually used to compute any of the variables *a*, *b*, *c* or *x*. Variational method creates system of equations by using these kinds of implicit equations and solving them for the right variable. This way it can solve all constraints of the system simultaneously. (Shah 1995: 83)

Both have their advantages and disadvantages. Parametric models can be evaluated rapidly, because of the more straight-forward solving of explicit equations. But the disadvantage is that parametric method cannot deal with mutually coupled constraints. Due to the usage of implicit equations, variational models can deal with mutually coupled constraints. But this makes them to work slower, and they may have difficulties in detecting inconsistent models. In the case of inconsistent model, the solver may try to use variational method to work out unsolvable equations, which may cause the computer to crash. (Shah 1995: 83)

As was mentioned earlier, modern CAD systems use both methods, parametric and variational, to solve the equations related to created relations. This so called hybrid system tries to decouple constraint equations in subsets, which can be processed sequentially. It is also possible, that hybrid system uses parametric and variational modeling for different purposes. For instance parametric modeling can be used to define geometric shapes and variational modeling to define design constraints. Because of the common interchangeable usage of terms parametric and variational modeling, the *parametric modeling* is used in this thesis as a term for modeling, in which relations are created and used for effective editing of 3D model, as all the modern CAD systems do. (Shah 1995: 83)

The possibility of utilizing created relations in CAD systems makes them much more powerful tools for the engineering design work. It has been estimated that 80% of all the design tasks are variational, which means that the goal of the design task can be reached by editing existing basic design to match the new requirements. Parametric modeling offers a huge advantage in a case of configurable products, because models for a whole product family can be easily created just by changing the input parameters. This is discussed more in chapter about configuration models. (Shah 1995: 79)

3 Design of 3D structures

Most of the mechanical products compose of multiple parts. CAD systems are forced to response this fact to provide efficient and precise modeling for product design. Assembly modeling is basic feature for modern CAD systems. In assembly modeling, the modeled 3D parts can be combined to a mechanical structure. Modeled assemblies can be used for tests and calculations, but the most common test is to experiment, how the whole mechanical structure looks like and how the modeled 3D parts fit together. In addition to these possibilities, different kinds of simulations and structural-analysis are done with 3D assemblies.

This chapter goes through the basics of assembly modeling, how assemblies are created and what are different assembly creation methods. The concept of configurable 3D model is presented. Configurable 3D models are the base of 3D product configurators, which for instance provide efficient method of designing and visualizing product families. Skeleton modeling is a technique, which makes especially configurable 3D assemblies more robust and easier to use. Skeleton modeling is presented in the last subchapter.

3.1 Assembly modeling

An assembly consists of components, which together execute the purpose of the whole system. To create a working assembly model there are two main issues to be considered. First of all, components of the assembly should be chosen and defined correctly. Secondly, the relations between components in the assembly should be defined.

One common characteristic to all the assemblies is that they have a structure. *The assembly structure* is basically a list of part models or sub-assemblies, which form the assembly itself. This list is also called as *assembly hierarchy* or *product structure*. In brief, the assembly modeling concentrates on creating working system by utilizing correct components and subassemblies and creating working relations between them. (Laakko 1998: 68) (Schoonmaker 2003: 242-244)

Another essential characteristic for assembly modeling is that it has to have some mechanism to track and store *positional information* of 3D part models and subassemblies. In every modeled part and also in assembly models, there is a master or global origin, which defines how the model is located in the modeling space. Location coordinates X, Y and Z are all zero in this origin. The location of model in modeling space can be defined as a distance from global origin according the X, Y and Z directions. Besides the positional information of parts in an assembly, also rotational information has to be tracked and stored. As all the parts and assemblies have their own coordinate systems the rotational information can be expressed as angle between coordinate systems. The parts in an assembly can be rotated to all three, X, Y and Z rotation angles. The possibility of tracking and storing translational and rotational location enables creation of assemblies (Schoonmaker 2003: 245-246)

Third critical characteristic is that assembly model has to be able to use *instancing*. Instance can be expressed as an intelligent duplicate of an object. In assembly modeling, each part model in the assembly structure should be considered as an instance. In other words, parts in assembly are instances of created part models. Same part model can be used many times in one assembly. If this was not possible, assembly modeling would be extremely laborious. An assembly can contain dozens of similar parts, for example screws, washers, springs etc. and if every single one of these should be modeled separately, it would take a lot of effort. (Schoonmaker 2003: 248-249)

These three essential characteristics, assembly structure, positional information and instancing, should be present in every assembly modeling system. Otherwise, it can be questioned, if the system is doing real assembly modeling at all, or is it just showing multiple part models simultaneously. An assembly model has to keep track of many types of data. For the user, it is important to understand, how the CAD system deals with the data, so that the user knows whether to work on the assembly model or the part models that form the assembly. Following table 1 shows where different types of data should be located in the assembly modeling. (Schoonmaker 2003: 249)

Table 1. Assembly model information types and locations. (Schoonmaker 2003: 250)

Type of information	Positional data	Type of information
Part geometry	Primarily part data	The **geometry of 3D part** (surfaces, volumes, features etc.) will definitely be in **part data**. **In some cases** features maybe added **in the assembly context**, for example in a case, where hole should be drilled through two parts as they are aligned together.
Positional data	Instance data	The location (translation and rotation) of parts and subassemblies in an assembly will be **in instance data**.
Cosmetic data	Part or instance data	The appearance (color, transfluency, texture mapping etc.) can be **in part data or in instance data**.
Attributes	Part data	Attribute data (such as material, manufacturing process etc.) **needs to be in part data**. If other attributes should be used for similar part (for example another material), typically another part should be made.

As was mentioned in previous chapter, constraints in 3D models refer to geometric or mathematical rules or restrictions, which are created during modeling. These model constraints enable effective modeling and editing of the model. A*ssembly constraints* are used to control or restrict the location of the part instances in the assembly model. (Schoonmaker 2003: 251)

The part instances have six DOFs (degrees of freedom) in assembly models. They can translate in X-, Y-, and Z-directions and also rotate about X-, Y-, and Z-axes. With assembly constraints these DOFs can be removed and part instances tied to origin, sketches, planes or other parts in the assembly. This makes it easier to edit the created assembly, as part instances follow the geometry they are tied to. It is also possible to create assemblies just by moving part instances to correct positions in an assembly model and not constraining them to anything. But in these cases as changes occur, the assembly will “explode”, which means that the part instances will not be in correct positions in the assembly. (Schoonmaker 2003: 251)

An example is shown in following figure 13, in which a parametric plate model has two holes for bolts. The length of this parametric plate is changed. In failed constraining, the bolt part instances were not constrained to the plate model and the model “exploded”, as the length was changed. In successful constraining, the screw part instances were constrained to the holes of the plate model and they followed the changes in plate model correctly. (Schoonmaker 2003: 251)

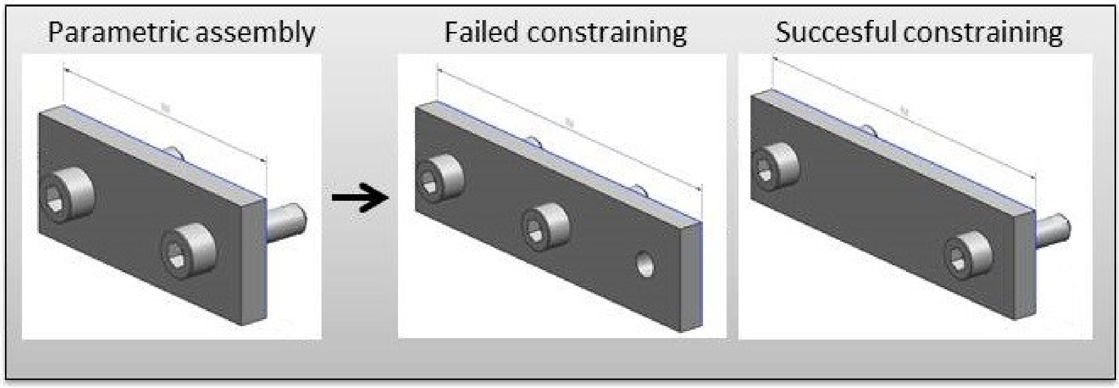


Figure 13. Assembly constraints in parametric model.

Also, all the issues that are present in constraining part models apply for assembly modeling as well. First problem is *underconstraining*, which means that all the DOFs are not constrained. In underconstrained model the part instance can still translate or rotate into some direction. In some cases, this will be a problem and the assembly may explode as changes are made. However, sometimes underconstraining may be even purposeful. For example crankshaft of an engine should be able to rotate around its axis and if some kinds of simulations or movement analysis are performed, this DOF should be left underconstrained. (Schoonmaker 2003: 251)

Another problem is *overconstraining*, which means that some DOF is constrained more than once. For instance, a facet of a cube may be constrained so that it cannot move in X-direction and another constraint is added to prevent rotation about Y-axis. However, rotation about Y-axis was already prevented by the first constraint, so adding this constraint again leads to overconstraining. In this particular case the added constraint does not violate the existing constraint. It depends of the CAD system, how it solves the simultaneous equations of constraints, and if this type of overconstraining is acceptable or not. Completely different situation is, if constraints violate each other. For instance, if a bolt is constrained to be aligned with a hole and a dimension is added to constraint its distance from plate end. The distance is different than the distance to the hole. Now the constraint solver tries to locate the bolt with respect to the hole and with respect to the dimension. Presumably, this does not work and another one of set constraints has to be removed. This kind of violating overconstraining causes problems to the solver and it may be possible that the model cannot be opened, because the solver cannot locate the overconstrained bolt. (Schoonmaker 2003: 252)

There are many methods to constrain parts to an assembly. It is recommendable that users, who are working with same models, have certain rules or standards of using constraints. For instance, dimensioning between two planes or faces could be used whenever it is possible. Many constraint methods use existing geometry, such as faces, edges vertices etc., but there are also methods, which use so called extra geometric entities. These entities are contained in the model, but would not be seen in the physical part. For instance datum planes, reference geometry, coordinate systems, additional lines and points. Most typical assembly constraints are listed in following table 2. (Schoonmaker 2003:253)

Table 2. Most typical assembly constraints. (Schoonmaker 2003: 253)

Constraint type	Description
Dimension **between planes** (plane of assembly and plane of part instance)	Really useful constraint, because it is the most similar to what 2D-dimension implies in a drawing. Constrains **one translational** and **two rotational DOFs.**
Dimension **between plane and straight edge**	Can be misleading, because the part instance that owns the edge can still rotate about the edge. Constrains **one translational** and **one rotational DOFs.**
Dimension **between plane and point**	Not so effective constraining method. The part instance with the point is ebale to rotate in any direction, so this method constrains only **one translational DOF.**
Parallelism **between two planes**	Keeps two planar faces aligned, but they can be at any distance apart. Constrains **two rotational DOFs.**
Parallelism **between two straight edges**	Constrains **one rotational DOFs.**
Perpedincularity **between two planes**	Constrains **two rotational DOFs.**
Coincident and colinear **straight edges**	Allows constrained part instance to slide along the share edge. Constrains **two translational** and **two rotational DOFs**
Tangency between **a curved face and a plane**	Allows the part instance with curved surface to roll along the flat plane. Constrains **one translational** and **one rotational DOFs**.
Fixing (”grounding”) part to global origin	Makes part instance immovable. The part instance will stay at the place, where it was, when fixed constraint was given. Constrains **all the DOFs**.
Fixing to another part instance	Makes part instance immovable related to another part instance. Instances cannot translate or rotate relative to each other, but together in the assembly they can. **Constrains all the DOFs between two instances**.

One typical feature for CAD software is that they are usually focusing on design of individual parts and the assembly is then created of designed parts. This method is also known as bottom-up design process. However, this is a little contrary to the real life product design process, which usually starts by defining the requirements and sketches of the whole working system. Modern CAD software are trying to offer tools also for this kind of design work, which is known as top-down design process. (Laakko 1998: 68)

In *bottom-up assembly design process* single parts of an assembly are fully designed and defined, before the assembly itself can be constructed. The process is shown in figure 14 below. Assembly modeling is used for positioning the parts in the final assembly and also for defining the relations between parts in the assembly. Because of the technical properties of CAD software, this is the most common method of creating assemblies in CAD. Especially, in older software there is no other possibility than to use bottom-up method in assembly design. In real life design work, bottom-up method is particularly useful, when existing components are used instead of designing completely new structures and components. (Laakko 1998: 70)

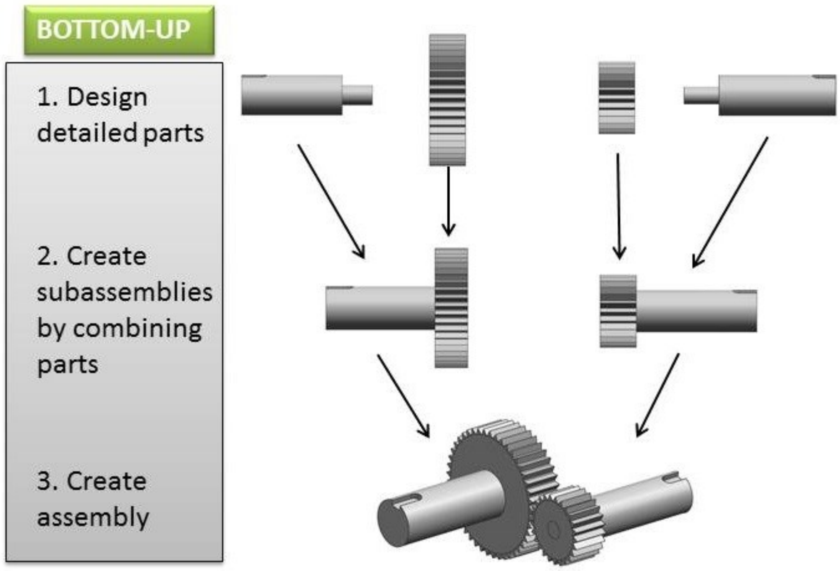


Figure 14. Bottom-up assembly design method. (Laakko 1998: 69)

Although most of the CAD software support only bottom-up process, the usual product design is performed according to *top-down assembly design process*. In top-down design process the design work starts by gathering requirements for the whole system and creating sketches of it. Created sketch of system can be then divided into sketches of components. Detail design work begins, as the assembly is divided into lowest level components and sketches are created for these components. In detail design phase the designer creates exact geometry for the parts, designs how they are manufactured, tests their strength and calculates the costs. Usually, in top-down design process multiple iterations are needed and the designer has to refine the design of part models as the design of the whole structure becomes more accurate. The principle of top-down method is shown in figure 15 below. (Laakko 1998: 70-72)

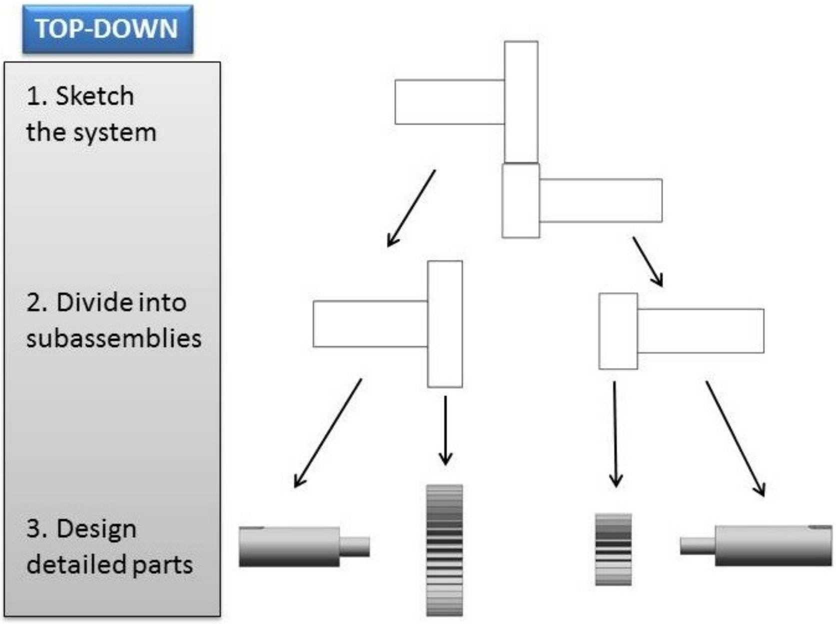


Figure 15. Top-down assembly design method. (Laakko 1998: 69)

Even though the top-down assembly design process is more natural for product designers, it is not yet so successfully adopted into CAD software. Usual product design process can be divided into five phases: functional design, conceptual design, embodiment design, detail design and engineering analysis. These design phases are usually not sequential, but iterative, recursive and mixed together. In order to support top-down product design effectively, CAD tools should support design phases as they are processed in real life. This means that from sketched geometry of an assembly, the model should be possible to divide into pieces, which then can be designed in more detailed level. Chinese researchers Cheng et al. have suggested one solution for this problem in their article: "Multi-level assembly model for top-down design of mechanical products. (Chen et al. 2012)

In the top-down assembly design process, an elemental sub-process *top-down component design* (TDCD) is executed on each component, as they occur along the expansion of an assembly tree. TDCD is first executed for the sketch of the whole assembly, which is decomposed into several sub-components. Then TDCD can be carried out to sub-components and sub-sub-components, until the whole product structure is formed of loosely coupled components. Loosely coupled system means that components have, or make use of, little or no knowledge of the definition of other components in system. (Chen et al. 2012)

For execution of top-down design on a component, the overall function and a shape skeleton should be defined. In this context, the shape skeleton means a vague and incomplete shape of a component. The skeleton modeling, which is different method, is discussed more in chapter 3.3. Other expressions for the shape skeleton are for example "design space" or "base shape". The overall function and shape skeleton for the product root are defined based on the product requirements and for the components based on their requirements. (Chen et al. 2012)

TDCD process on assembly is executed in three steps (Chen et al. 2012):

1. In *abstract design* the overall function is decomposed to establish a function structure and an engineering structure, which fulfill the requirements of the whole product. The result of abstract design is a concept of the product at a specific abstract level.
2. *Skeleton design* is carried out based on the results of abstract design. Skeleton design mainly considers information about shape and spatial arrangement, but it also describes the mating rules and relative motions between two or more components. The result of skeleton design is a "3D layout". This 3D layout describes spatial configuration of the assembly elements and kinematics behavior between them.
3. *Top-Down Component Design (TDCD)* is executed on each sub-component, as abstract design and skeleton design are finished. As the object of TDCD is a part, the shape skeleton is refined to the detailed shape and design aspects such as manufacturing costs and ergonomics are considered.

The following figure 16 shows how the multi-level assembly modeling works in CAD system. Among the shape skeleton, a lay-out skeleton is used for assemblies to design, how the parts are located in the assembly. In the top-down assembly design it is necessary to switch among abstract design, skeleton design and detail design until detailed models of all the parts are generated. The efficient usage of multiple levels of model representations needs fluent transfer of design knowledge and information. This data transfer is called "inheritance" and to provide efficient top-down modeling, the CAD system should allow the designer to choose which data he wants to transfer to the following phase. Examples of this "inheritance" are also shown in figure 16. From the shape skeleton of an engine, it is really important to transfer the data for length of crank shaft. (Chen et al. 2012)

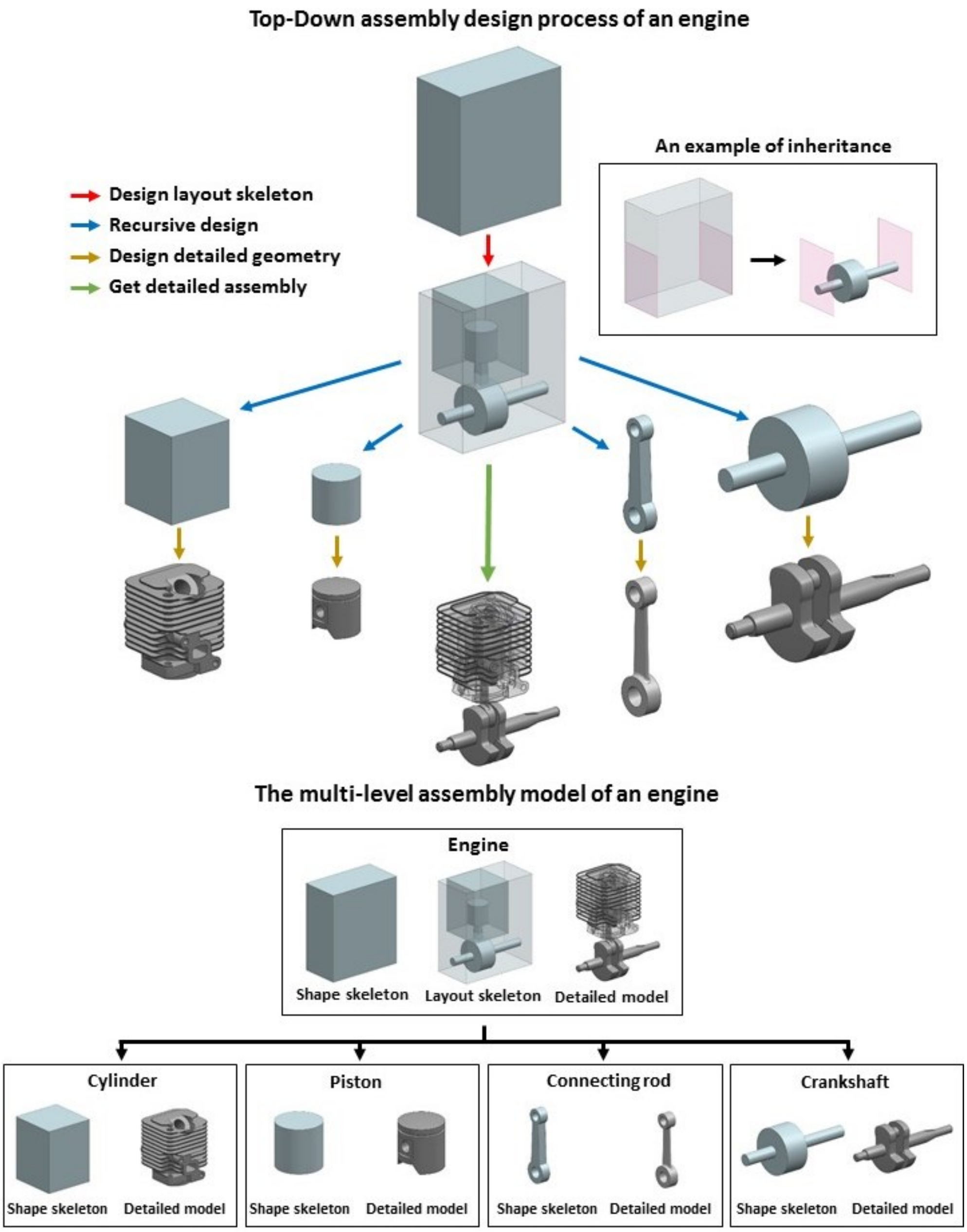


Figure 16. Top-Down assembly design and multi-level assembly model (Chen et al. 2012)

There are of course some special requirements for CAD system to support the top-down assembly design. First of all the system should support the usage of multi-level assembly model, which captures the information in different levels of abstraction during the top-down assembly design. The multi-level assembly model should represent not only the detailed geometry, but also the more abstract information needed for top-down modeling, such as function and layout. The system should be able to provide various inheritance mechanisms as well. This would ensure the transition and association of design information between different design phases. The design intent of certain parts should be possible to document during the process as well. The system should support effective usage of existing models and also enable simultaneous creation of different end result options. (Chen et al. 2012) (Laakko 1995: 70-71)

The modern CAD software are aiming to support top-down assembly modeling and they already manage to do it at some level. All in all, assembly modeling is vital for creating precise product models. The assembly modeling itself has three essential characteristics: assembly has to have a structure, parts have to have positional information and assembly has to be able to utilize instancing. Constraining parts correctly is really important, especially in the case of parametric models. To create 3D model configurator, this has to be taken into consideration.

3.2 Configuration models

Configuring can have different meaning in different context, but *configurable product* means a product, which is modified according to customer requirements. Configurable product is not tailor made product, because the customer cannot list all the requirements arbitrary, but is offered different kinds of options to choose from. For instance, customer can choose his size and favorite color of all the available sizes and colors for certain T-shirt, which is sold in an online store. Still customer cannot decide all the exact dimensions of the T-shirt. This is one simple example of configurable product. In the book of “PDM –Tuotetiedon hallinta” (in English: PDM – Product Data Management”) Peltonen & al. have defined that configurable product has following features: (Peltonen 2002: 79)

1. Every individual product is produced according to customer requirements.
2. The product is designed to fulfill a group of similar customer needs.
3. Individual product is produced by combining predesigned components.
4. Individual products are based on predesigned structure.
5. Only routine and systematic work is needed to modify individual product.

In comparison to products, which are produced by mass production, configurable products offer more options for the customer, but are also a little more expensive because of that. Then again, in comparison to completely tailor made products, configurable products do not offer as vast range of options, but are also not as expensive as tailor made products. The following figure 17, shows the relation of mass produced, configurable and tailor made products. Configurable products can be sometimes called as mass customized products, but this is usually in cases, where production volume is high. (Peltonen 2002: 79-80)

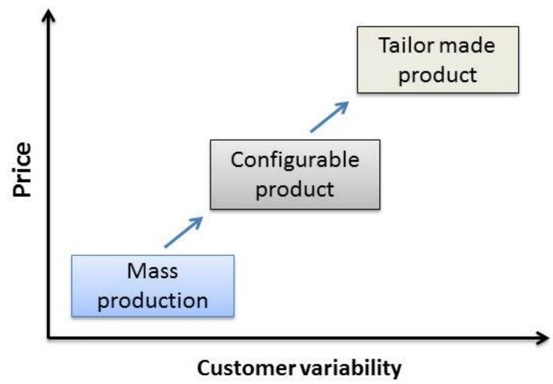


Figure 17. Comparison of product categories (Peltonen 2002: 80)

Group of configurable products can be thought as *a product family*, which contains different kinds of *product variants*. Usually, these product variants are a combination of variant components, optional components and parametric components. Variant component is certain compulsory option, which customer has to choose from a group of options. For example, rims of a car can be variant component, as customer has to choose certain kind of rims from a variety of different rims. In any case, some rims have to be chosen for the car. Optional components are components, which can be chosen or left out. For instance, wheel covers can be chosen for the rims, but they are not needed to be included. Parametric component is certain feature of variant or optional component. For example the color of rims or wheel covers can be a parametric component, as it is possible to decide from a variety of color options. Because the amount of different product variants increases significantly in relation to components, it is usually impossible to define all the product variants separately. This is why a *product family structure* has to be created. In product family structure all the product variants are defined with the aid of variant components, optional components, parametric components and relation rules, which define how these components can be combined. Product family structure is also called as "configuration model" or "generic product structure". (Peltonen 2002: 81)

Configuration process can be executed in many phases. Especially in a case of complex products, the real customer requirements will sharpen during the process. The requirements affect significantly to the whole structure of product. To fulfill the customer needs as well as possible; the requirements should be defined clearly. Following figure 18 presents an example order-delivery process chart of a configurable product. Green boxes present different actions in the process and white boxes the information, which is delivered between these actions. Usually *configuration process is divided into two configuration phases*: sales configuring and product configuring.

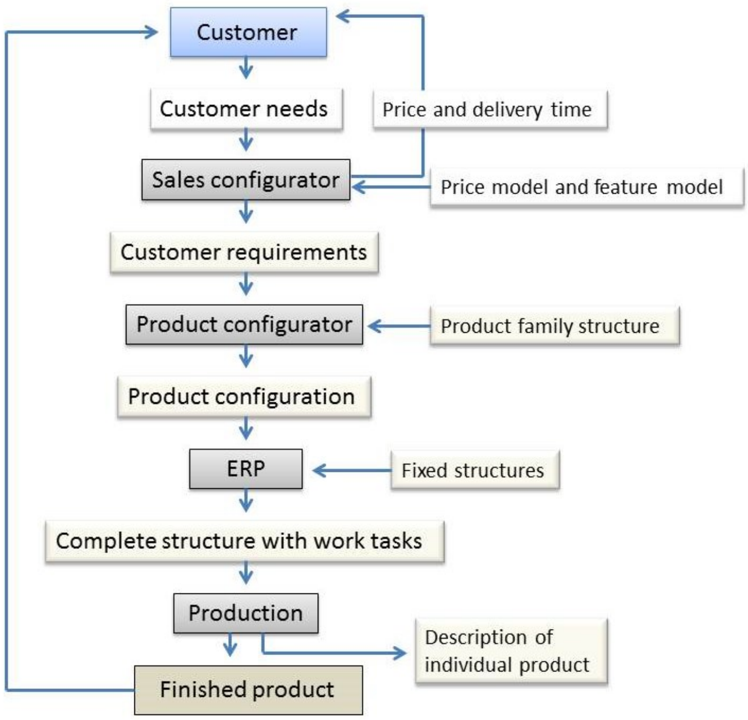


Figure 18. Order-Delivery Process of configurable products. (Peltonen 2002: 83)

In *sales configuring* it is all about the customer. Customer is interested in features, price and delivery time of the product. It is essential to discover the customer needs and solve, if the product is suitable to fulfill these needs. An output of sales configurator is agreed list of customer requirements, which can be used for the product configuring.

In *product configuring* the existing product family structure is utilized. In other words, components are chosen according to customer requirements and required product variant is consisted. Result of this phase is a product configuration, which contains all the information needed for manufacturing the product. Next action in process is ERP (enterprise resource planning), in which it is planned, how resources (time, materials, labor, machines etc.) are used to manufacture the product. Final phase is the production. As on outcome of production is the product itself, but sometimes it is useful to document some kind of description of individual product as well. This description of individual product is another outcome of production phase. For instance, many products can be produced according to a same product configuration, but in the quality of manufacturing there may be some variance, which can have effect on features of product. In configuring some nominal value may be given to certain feature of the product and this feature can be measured, as the product is produced. The test data is one example of data, which can be included in the description of individual product and probably used for service purposes etc. later on. (Peltonen 2002: 81-85)

Configuration model basically describes the configuration of a product in certain phase. As was mentioned earlier, in sales process it is all about the customer and his needs. Customer is interested in features of the product instead of how the product can be manufactured for example. *The configuration model in sales process* does not have to describe all the details of the product, but the details, which the customer is interested in. Also, needs and requirements of customer can be clarified with the aid of visual sales configurator. For instance, many car manufacturers use sales configurators, which allow the customer to design the appearance of the vehicle as well as choose accessories from a variety of options.

The purpose of *the configuration model in product configuring* is to offer all the necessary documentation for manufacturing the product. Product configuration model presents the geometry and other features of the product as they should appear in reality. The manufacturing drawings are created based on the product configuration model. The details, which were not so interesting in sales configuration model, become really important now. Sometimes the sales configuration model contains already precise enough description of geometry for manufacturing, but it is also possible that the product configuration model is created separately.

The configuration models can be created by utilizing parametric modeling features of CAD software. Created parametric model should be able to present all the product variants of certain product family. Model may contain parts, which have parametric dimensions, and components, which are either included or excluded in the configuration model. Model configuring which means giving the needed parameters for the model, can happen in two different methods. First method is that parameters are given inside the design software. All the CAD software, which support parametric modeling, have certain kind of user interface for collective input of design parameters. Another method is to use external configuring program to input the parameters for configuration model. External configuring program can be called as configurator. External configuring programs usually gather also some other information than just the parameters for configuration model, as they are used. For example, the program may calculate the costs of the product. (Laakko 1995: 116-117)

As a summary, configurable products are created according to customer requirements. However, customer cannot arbitrarily define requirements for the product, but it is possible to combine the most suitable solution from available options. Configuration model presents the configured product. Sales configuration model may present only the data, which is important for sales configuring, but the product configuration model should present the exact geometry of manufactured product as they are used for manufacturing purposes. Configuration models are based on parametric modeling and features of CAD systems, which allow the user either include or exclude certain components in the model. The configuring itself can be performed in CAD software or with some external configuring program. The following figure 19 shows an example of a configurable product and configured variant models of it.

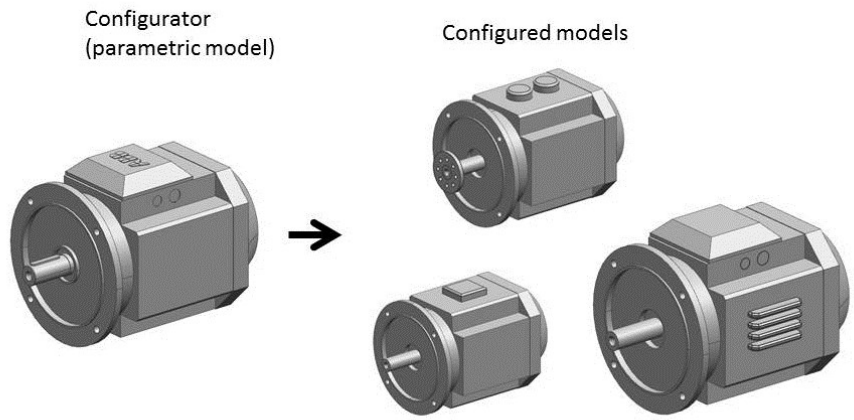


Figure 19. Different configurations (product variants) created by configurator.

3.3 Skeleton modeling

Skeleton modeling (or skeletal modeling) means that some kind of simple base geometry or frame is used in design of an assembly model. Especially, when an assembly of configurable product or product family is modeled, skeleton technique provides more robust, safer and design friendlier method of creating assembly. Skeleton makes assembly parts independent of each other, which means that any part can be removed or changed in assembly without causing harm to constraints of other parts in the assembly. In usual assembly design parts are positioned in relation to each other. Constraints are created between parts and if some part is removed or changed, it may turn the created constraints impossible, which causes the assembly file to crash. In skeleton modeling parts are constrained only to the skeleton, so these kinds of problems do not occur. (Laakko 1995: 121)

Skeleton modeling is also one method to support top-down assembly design. The idea of skeleton is to create necessary base geometry first, before modeling detailed parts. Usually, the skeleton consists of sketches (points, lines etc.) and planes. However, nothing forbids using solid or surface geometry either. As modern CAD software support copying of geometry from a part to another, the geometry created in skeleton model, can be utilized for designing component models. For instance, the space for certain part can be limited by planes and when these planes are copied to the part model, designer automatically knows the outer dimensions for the part. Usage of parametric assemblies becomes easier, as all the parametric features can be created into the skeleton and then copied into the parts. For instance, new parameter value is given for the distance between planes in skeleton. The changes will occur also in the part model, into which the planes were copied from the skeleton. As the geometry of part is related to a distance between planes, changes will occur in the geometry. Skeleton modeling enables dynamic modeling of assembly parts and makes it easier to control dependencies in assemblies, as parts are designed based on the geometry of a skeleton. In following figure 20 an example of skeleton model is presented.

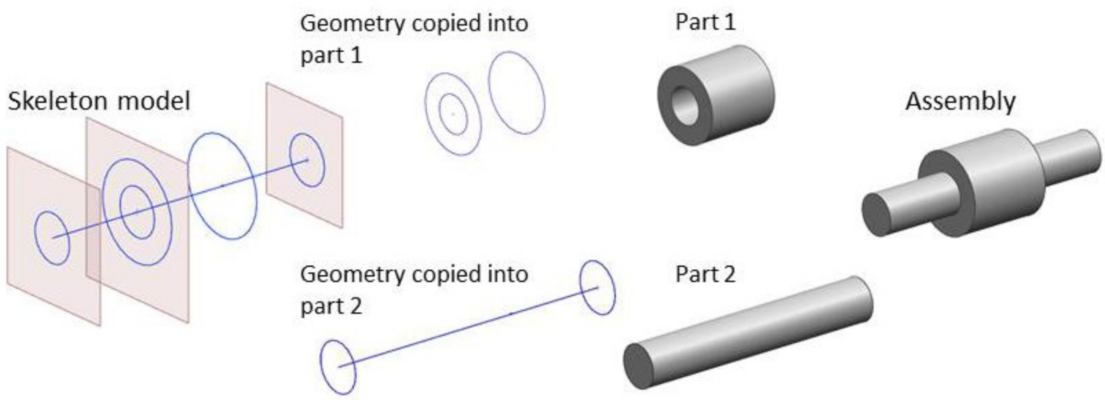


Figure 20. Skeleton model and parts, which are created based on the geometry of skeleton.

Clear advantages of skeleton modeling can be listed as following (IGETIT: Skeleton):

1. It makes assembly modeling more robust and enables quick changes to the geometry.
2. It allows the designer to work with temporary data in smart manner until the data for exact geometry is known.
3. It enables concurrent design, because a small amount of geometry can be used as a guide for much larger design.
4. It simplifies relational design by imposing the parts automatically according to skeleton.

Also few disadvantages for skeleton modeling can be listed (IGETIT: Skeleton):

1. It usually takes more time and requires more knowledge about functionalities of CAD system to create the skeleton.
2. When skeleton modeling is utilized, the modeling technique, which is deliberate and robust, has to be used for part modeling.

3. It turns the assembly modeling more complex and in shared design work, all the designers should understand and manage the usage of skeleton.

All in all, skeleton modeling offers useful and robust method for assembly modeling, especially for creation of configurable models. It may require a little more time than usual assembly design or turn the modeling a little more complex, but when the designers are used to it, it becomes more natural and proficient. For the most simple assembly models, there may not be advantage of using skeleton modeling, but as the models turn more complex, the advantages of skeleton stand out. Skeleton modeling enables also simple method for top-down assembly design.

4 Design tools

In this chapter the design tools, which were available to be used for creating the 3D model configurator, are presented. Since ABB is a large company, it is using various engineering tools globally. The department of synchronous machines has chosen to use mostly products, which are owned by Siemens Corporation. NX I-Deas 6.2 is used as a main 3D design tool and Siemens Teamcenter as PLM software of mechanical design department. However, Siemens NX 8 is used for CAD purposes of research and development department and in the near future it will be adapted to the use of mechanical design department as well. As NX I-Deas 6.2 is already old software and does not support parametric modeling or creation of configurable models as effectively as NX 8, the NX was chosen as main CAD tool in this project.

The use and potential of NX 8 were examined thoroughly and most important features of software are presented in this chapter. Because NX I-Deas has been used for several years as main CAD software of design department, it was important to find methods to utilize existing 3D models of designed motors. The data exchange between CAD software was researched and the results of research are presented in this chapter as well. For the user interface there were many options and the most suitable one was needed to be found. Options were to use the configurator in NX, in Teamcenter or in some totally different software, for example in Microsoft Excel. The advantages and disadvantages of different user interfaces were examined and are also presented in this chapter.

4.1 Siemens NX 8

NX 8 is one of the leading CAD software on the market. It is a product of Siemens Corporation as is the PLM (product life cycle management) software Teamcenter. NX 8 is not only solid modeling tool, which unites 3D parametric features with 2D tools. It is able to guide the user through the whole design-manufacturing process. Cooperation of 3D modeling and 2D drafting is based on bidirectionally associative nature of the software, which means that modifications made in the model are updated in the drawing views and vice versa. NX contains different modeling environments for different purposes. Modeling Environment, Shape Studio Environment, Assembly Environment and Drafting Environment are discussed below. NX 8 has also Manufacturing

Environment, but this feature is not useful for creating the required configurator. (Tickoo 2012: 1-3)

In *the Modeling Environment* it is possible to create solid parametric models by feature based modeling technique. Models are created normally by utilizing sweep creation method. 2D sketches are drawn in the Sketch Task Environment or straight in the Modeling Environment. Many kinds of drawing tools are provided for creating the geometry of a sketch and also constraints and dimensions can be set automatically or manually. After drawing the sketch, it has to be converted into a feature to create three-dimensional geometry. These tools, such as extrude and revolve, are available in the Modeling Environment. It is also possible to create placed features to the model in this environment. These features are for example fillets, chamfers, tapers, holes, bosses etc. (Tickoo 2012: 1-2)

The Shape Studio Environment is useful for conceptual and industrial design, because it is intended for creating complex surface models. It is parametric and feature-based environment as is the Modeling environment and the tools are relatively similar as well. But the tools in the Shape Studio Environment are used to create basic and advanced surfaces and this environment contains also editing tools, which can be used to manipulate created surfaces. (Tickoo 2012: 1-3)

Components can be assembled together by using constraints in *the Assembly Environment*. In Assembly Environment there are two options for designing an assembly. In bottom-up approach components are designed first and then assembled together in Assembly Environment. In top-down approach the assembly is designed straight in Assembly Environment. A skeleton model or certain sketches for assembly can be created first. User is able to choose any created geometry and copy it into a part model to design the detailed component. (Tickoo 2012: 1-3, 9-4, 10-2)

The Drafting Environment is used for creating documentation of parts and assemblies, which are modeled. NX has two types of drafting techniques: generative drafting and interactive drafting. In generative drafting technique drawing views are generated automatically from parts and assemblies. The parametric dimensions can also be automatically generated and displayed in the drawing views. This technique has also the earlier mentioned bidirectional feature, which means that changes made in drawing will appear to the model and vice-versa. In interactive drafting mode drawings can be created by sketching them with normal sketching tools and adding dimensions manually. (Tickoo 2012: 1-3)

Basic modeling in NX is really similar to all the other major CAD software on market. In normal modeling work solid geometry is used, but also surface models and boundary models can be created with NX. NX has FEA features as well and created models can be turned into decomposition models for purposes of FEM analysis. However, this thesis will focus only on the most interesting features of NX 8, which are needed for creating the configurator or which can significantly ease the work of designer. Some of these features are described in following.

In the Modeling Environment, it is extremely important to fully *constrain the created geometry*. Otherwise, there is a possibility of geometric changes, as new parameter values are given to a parametric model. Creating fully constrained geometry, is a robust method of modeling and makes it easier to change already created shapes. As speaking about constraining the geometry, it basically means constraining the sketches, which define the geometry. There are two methods of constraining sketches: geometric constraints and dimensions. (Samuel 2012: 80)

In brief, creating *geometric constraints* means that geometry is given some rules to follow. For example, diameter of created circles can be set to be the same, lines can be set to be horizontal, vertical, parallel, equal lengths or aligned with each other and some geometry can be fixed in relation to origin. Smart usage of geometric constraints makes the modeling a lot easier and also eases the editing of already created geometry. For instance, diameter of multiple holes can be changed just by changing one parameter, if all of the holes are given the "equal diameter" constraint. (Samuel 2012: 80-89)

Geometry can be constrained also by adding dimensions. In NX 8 dimensions can be added automatically while sketching, but it is recommended to add dimensions manually, so that automatic dimensioning does not constrain the geometry in unwanted manner. NX

8 presents fully constrained sketch by coloring it red as is shown in figure 21 below. (Samuel 2012: 81)

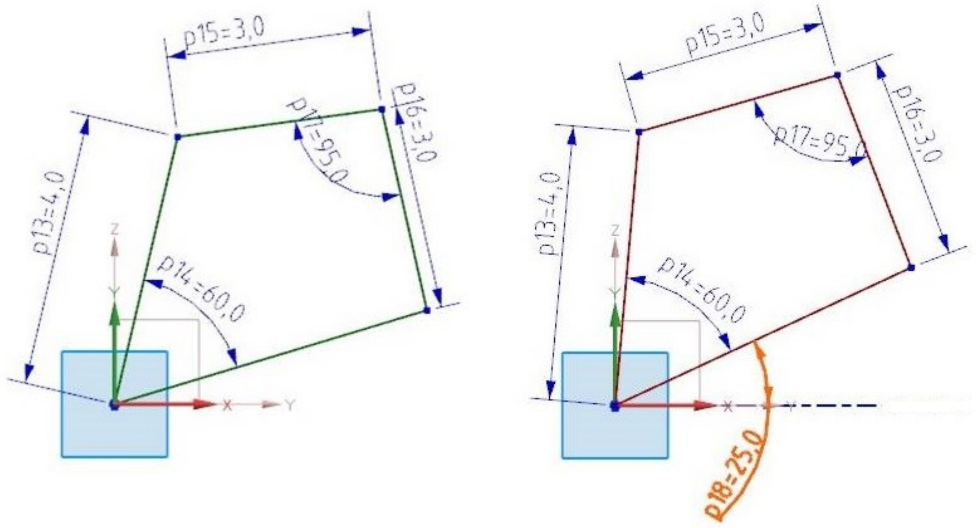


Figure 21. Fully constrained sketch marked by red color in NX 8.

Because NX 8 supports parametric modeling, adding dimensions or design features creates relations between design entities and these relations are automatically stored in the system. An example of this kind of relation is distance between lines in a sketch, which is shown in following figure 21. Distance between these lines defines also the length of the plate model. Dimensioning the distance constrains the relation into certain value. The relation can be changed by inputting new value for the dimension. The input value is commonly called as parameter. In NX parameters are known as "Expressions" and they can be written as a single numerical value, a mathematical operation or a combination of other parameters. All of the created feature expressions can be seen in the Expressions dialog box, which can be opened by choosing: Tools/Expressions. The software names created expressions systematically starting from p0. The values of expressions can be changed either in graphical view or in expression dialog box. Also, names of expressions can be changed and comments added into the comment column. (Samuel 2012: 293-307)

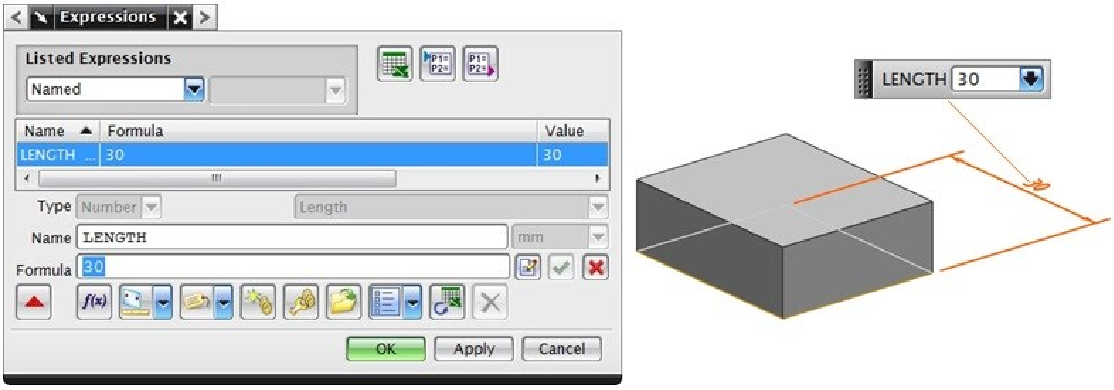


Figure 22. Expressions in graphical view and expression dialog box.

As configurable models are created, it is also necessary, that certain features in parts or parts in assemblies can be hidden. Different variant and optional components can be either shown or hidden in configured model with this method. In NX this is happens by using function called “Suppress by Expression”. Any feature or part can be selected to be suppressed by expression. Usage of this function creates an expression, which can be seen in Expressions dialog box. The expressions, which control suppression, can be named in similar method as the expressions related to dimensions. These expressions can get either value 1 or 0. Value 1 means that suppression is not performed (feature/part can be seen in model) and value 0 means that suppression is performed (feature/part is hidden from the model).

Expressions are created on part level, but they can be transferred into assembly level by using *Interpart Reference tool*. Transferring expressions into assembly level, enables input of all expression values on assembly level instead of inputting values separately on each part level. For instance, in the assembly presented in figure 22 there are two parts: test_part_2, which is the shaft and test_part_3, which is the disc. The diameter of shaft, as well as the diameter of the hole in the disc, can be controlled by expressions created on part level.

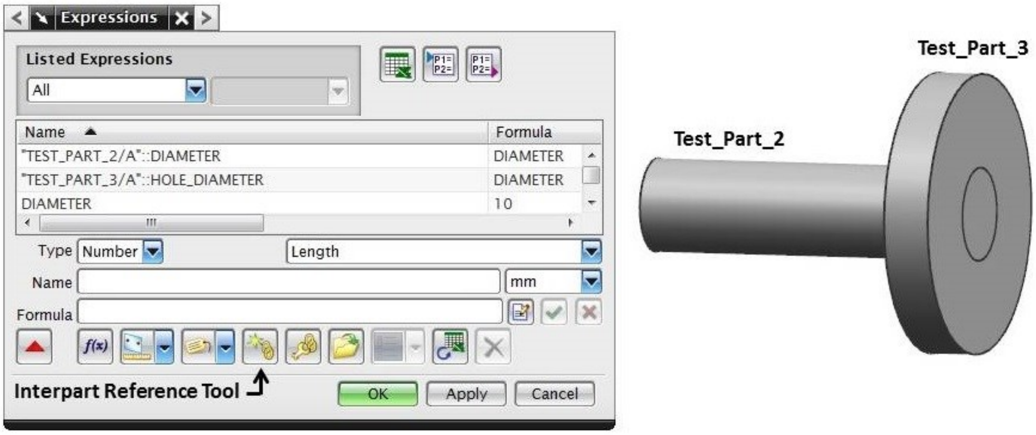


Figure 23. Using Interpart Reference tool to control Expressions on assembly level.

By opening the Interpart Reference tool on assembly level, it can be selected, from which part expressions are transferred to assembly level. The following step is to select the expression to be transferred. When selecting the expression, it is necessary to thick the "Override Expression in Component Part", so that it is possible to control the Expression on assembly level. As expressions from both component parts are transferred on assembly level it is possible to input values for them or create new expressions, which control these transferred expressions. In figure 22 a new expression "DIAMETER" is created to control both diameters by one input value. However, overriding expressions on assembly level means that expression values cannot be controlled on part level as the assembly is open in NX. Small lock figure next to expression in dialog box of component part indicates that Interpart Reference is used and it is not possible to change the value on part level. If assembly is closed and only part model is opened, it is possible to manipulate expression value on part level again. (Liira 2014)

The usage of expressions enables creation of parametric models as well as configurable product models with the aid of NX. As was mentioned, parametric model can be manipulated just by changing expression values in Expressions dialog box or in graphical view. But *it is also possible to transfer parameters into Microsoft Excel* for inputting. NX is linked to MS Excel and by clicking "Spreadsheet edit" icon; expressions can be exported into an Excel file format for inputting. As the Expression values are input in Excel it is possible to transfer them back to NX to create the desired configuration. (Samuel 2012: 293)

Another great method to input parameter values and modify created parametric models is *Product Template Studio (PTS)*. PTS is an add-on program for NX and with the aid of PTS it is possible to create simple and easy-to-use user interfaces for parametric models. However, to create user interfaces with PTS, so called "Author" license is required. Author license is not included in normal license. With the normal license, which is also called as "Consumer" license, it is still possible to use PTS for configuring models, if user interface has been already created. An example of PTS user interface is shown in following figure 23.

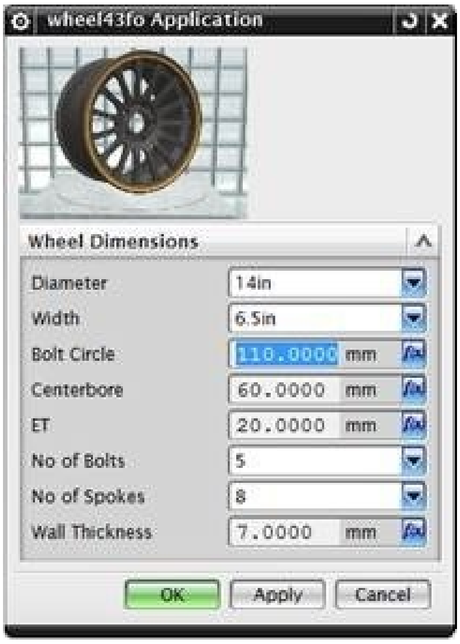


Figure 24. An example of user interface created with PTS. (Svärd, Tjurnev 2014)

PTS does not only input the given parameter values into NX, but it is also able to duplicate or copy models in generic structure. Duplicated models do not have connection to original models, but still have the geometry of them. Possibility of duplicating models enables modification of part models without "harming" the original parametric structure. For example, in case of tailor made products, which are built on certain product platform, this is really useful. The basic structure can be configured and new identification numbers and names changed for all the parts, which probably are in a need of customization. Customizing changes can be made for the parts and no changes to the original parametric structure will be made. This duplicating feature of PTS utilizes actually another operation of NX, which is called as Clone operation. (Svärd, Tjurnev 2014)

Clone operation is an easy method to duplicate created models and assemblies in NX. In the Clone operation it can be chosen, which models of an assembly are needed to duplicate. The duplicated models can be renamed and will have new ID numbers in PDM system, if such a system is used to store the data from CAD system. The duplicated structure, which is created by the Clone operation, is mostly independent of the original structure. Still all the geometry, relations, constrains, annotations and 2D drawings remain the same in duplicated models, as they appear in original models. With the aid of the Clone operation, it is really easy to create structures for customizing purposes. All necessary modifications can be made to duplicated (cloned) items without harming original models. (Svärd, Tjurnev 2014)

As an assembly is cloned, expression values, which were transferred from component parts by using Interpart Reference tool, remain in the Expressions dialog box of new cloned assembly. Expression values can be controlled on assembly level of cloned item. If both assemblies, original and cloned one, are opened in NX at the same, the expression values are locked for the assembly, which was opened first. If changes are made to values, they will appear in another assembly as well.

A useful tool for copying geometry from model to another in NX is *WAVE Geometry Linker (WAVE-GL)*. WAVE-GL is especially helpful in creation of parametric assemblies and in top-down assembly design. First, an assembly needs to be created and some geometry added into it. The geometry, which is needed to be copied to another part, can be selected from the assembly view. Both parts have to be in same assembly, to copy geometry from one to another. If geometry is changed in the original model, the changes will appear to another model as well. It is also possible to break the link between models, so that copied geometry will not change, if original geometry is changed. (Samuel 2012: 510-515)

WAVE Geometry Linker is a tool, which enables the usage of skeleton modeling for product design purposes in NX. A parametric skeleton can be created in the Assembly

Environment. Layout geometry for part models can be copied with WAVE-GL from skeleton model. Detailed geometry can be added to the part, but it has to be constraint to the geometry copied from skeleton, so that it follows the parameters of skeleton correctly. As all the parts are modeled and assembly is finished, it is possible to modify the geometry just by changing the parameter values in skeleton model.

4.2 Data exchange between design software

In design and manufacturing different systems are used to produce technical data, which is needed for producing the final product. For example, CAD software is used for mechanical design, but different software may be used for PDM, FEM, industrial design or CAM. In collaboration of companies different software are often used and it is also possible that inside one company, many different software are used even for a very same purpose. Systems have multiple different data formats and usually they are not directly compatible with each other. This makes things complex and laborious. (Shah 1995: 232) (Laakko 1998: 254)

In many cases there is a need to combine together models, which are designed with different software. For instance, parts of final model are created with different software and there is a need to create assembly of these models. 3D model created with certain CAD software has to be manufactured by using other CAM software. Because of these reasons, CAD systems are required to work with other systems and application programs as well. The software needs two translators for usage of different CAD file formats: one that reads and another that writes.

Some software developers have included these kinds of translators into their software, but the problem is that there should be plenty of translators to ensure the vast usage of various file formats. Much better idea is to use neutral file formats, because then CAD software needs only one reader translator and one writer translator for cooperation. This can be assimilated to usage of languages. Different nations have different languages, but by learning one common language (for example English), they are able to communicate together without need of learning multiple languages. Following figure 24 demonstrates how neutral file format can make data exchange a lot easier. (Laakko 1998: 254) (Shah 1995: 232)

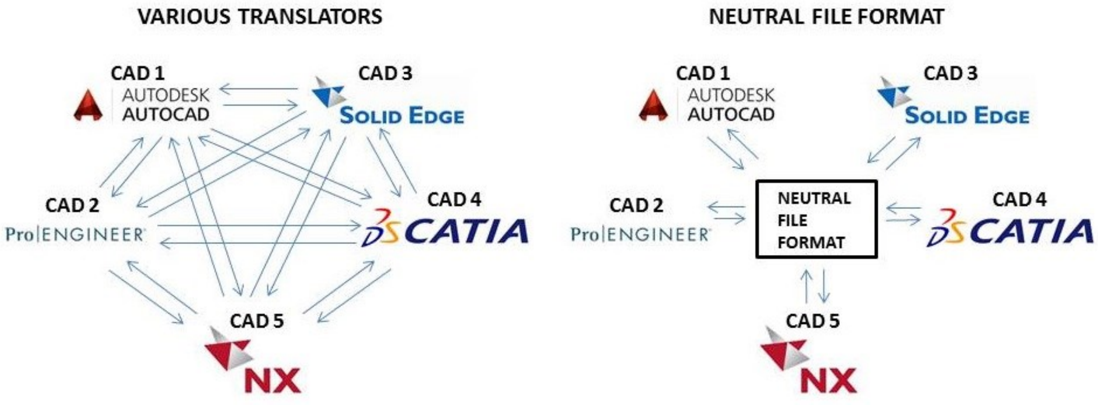


Figure 25. Advantage of neutral file format. (Laakko 1998: 254)

4.2.1 National data exchange standards

The most successful efforts of neutral file format have been the standards for data exchange. The first standards for data exchange were national and focused on geometric data exchange. They included IGES (the Initial Graphics Exchange Specification) in the USA, SET (Standard d'Échange et de Transfer) in France and VDA-IS/FS (Verband der Deutschen Automobilenindutrie - IGES Subset/Flächen Schnittstelle) in Germany. (STEP Tools inc.)

IGES (the Initial Graphics Exchange Specification) was the first standard, which defined the neutral CAD data exchange. The development work started already in the late 1970's by National Institute of Standards and Technology in the United States. Nowadays IGES is supported by almost all the software on the market and has probably spread the widest of all the data exchange standards. IGES defines following elements: 2D and 3D wireframe models, surface models, boundary models, CSG models, annotations, trimmed surface geometry, dimensioned geometry associativity, structures, attributes and features. The structure of IGES file is relatively complex, which sometimes may cause problems. (Laakko 1998: 257)

VDA-IS (Verband der Deutschen Automobilenindutrie IGES Subset) was developed by German automobile industry organization. IGES had some reliability issues during first years of its existence so Germans wanted to develop more reliable and safer method to exchange 2D geometry data and 3D wireframe and surface model data. *VDA-FS (Verband* *der Deutschen Automobilenindustrie Flächen Schnittstelle)* was developed by the same organization to exchange the data of plastic and sheet metal parts. VDA-FS defines the presentation of three dimensional surfaces and curves. VDA standards present the models as surface models, which are divided into triangles. This makes the file sizes relatively large. However, VDA-FS works in many cases better than IGES. Even though the development work of VDA standards was ended in the late 80's, many CAD systems still support VDA data exchange. (Laakko 1998: 257)

SET (Standard d'Échange et de Transfer) was developed by French airplane industry and is almost as comprehensive as IGES. Purpose of SET was to create more reliable method for data exchange than IGES is. Main goals were to develop unambiguous system, which creates small file sizes. Nowadays SET supports wireframe, surface and solid models including boundary representations and CSG models. Also drawings and FEM models are supported. Although files are more compact and SET supports data exchange of solid models better than IGES, SET is not so commonly used. (Laakko 1998: 261)

4.2.2 International data exchange standard: STEP

An international effort in developing product data exchange standards has been *STEP (Standard for Exchange of Product data)*, which is developed by ISO (International Organization for Standardization). STEP standard is also known as ISO standard 10303. The purpose of STEP was to offer neutral file format, which can be read and written by all software. STEP includes representation of product data and the mechanisms, which are needed for exchange of product data. Actually STEP is not only one standard. It contains a set of separate ISO standards, which define different methods of presenting product data on various fields of industry. Most commonly used part of STEP is the presentation format of geometric data. This part of STEP is included in all the modern CAD software. (Peltonen 2002: 93)

STEP is organized in series of standards. Some standards in these series are called AP's (application protocol), which define the product data for an application or a consistent set of applications. For example, AP's can define industry or product-specific data exchange standards, such as sheet metal die planning or design of composite structures. In many of these application protocols it is necessary to represent same kind of data, for example geometric shape. Because of this, STEP defines IR's (Integrated Resources), which are used as building blocks for different AP's. Examples of IR's are geometry, finite elements, drafting resources and configuration management. However, each application protocol defines how IR's are used in that certain AP and the methods may differ a lot from each other. (Peltonen 2002: 94) (Datakit) (Shah 1995: 233)

All the data models are presented by product description language EXRESS (can be called also as standard data modeling language for product data), which is defined in ISO standard 10303-11 (11^{th} part of standard series in STEP). EXPRESS is in a computer sensible lexical form, but it is not a programming language. It is used to support precise definition of STEP information models. (Shah 1995: 232-234) (Datakit)

The application data, which is consisted according to given data model, is exchanged either as STEP-file, STEP-XML or via SDAI (Standard Data Access Interface) in shared database access. Of these three, the STEP-file is the most common one. The STEP-file format is defined in ISO 10303-21. The standard defines how the data model created by EXPRESS language should be encoded. STEP files themselves are coded by using ASCII (American Standard Code for Information Interchange), which makes them easy to read. The file extensions .stp and .step indicate that the model is created according to some application protocol. Extension .p21 is used for all the other purposes. (Peltonen 2002: 90-95)

STEP-XML format is defined in ISO 10303-28 and it is an alternative method to STEP-file for exchanging data. XML (extensible markup language) is markup language, which is originally designed to describe structured documents, but is used more widely to describe every kind of structural data. The standard defines how EXPRESS schemas and data, which are governed by the EXPRESS schemas, can be represented by XML. (Peltonen 2002: 90-95)

SDAI (Standard Data Access Interface) is third implementation method of STEP. SDAI can be referred to API (Application Programming Interface), which defines how some software components should interact with each other in computer programming. Basically, SDAI tells how the application data, which is created according to data model given in EXPRESS, can be worked on. Functionality of SDAI is defined quite strictly in ISO 10303-22, but nowadays term SDAI is used for all kinds of API's, which support STEP. SDAI itself is not independent of any specific programming language and language bindings can be found at least in C++, C and Java programming languages. (Peltonen 2002: 90-95)

The data models, which are formed according to STEP standards, are presented by using EXPRESS description language. How EXPRESS defines the model, is determined by Integrated Resources. Usage of IR's is then again determined by Application Protocol. The AP's are the top parts. In this manner the final data model for data exchange is constructed and it is called AIM (Application Interpreted Model). Following table 3 presents the most important parts of STEP in domain of mechanical CAD. (Shah 1995: 232-234) (STEP tools inc.) (Peltonen 2002: 93-95)

Table 3. STEP standard series (Datakit)

Group	Series	Part	Description
Environment	1		Overview of STEP
	...		...
	1x		Description methods (EXPRESS documents)
	...		...
	2x		Implementaion methods
	...		...
Integrated data models	4x		Integrated Generic Resources
		41	Fundamentals of product description and support
		42	Geometric and topological representation
		43	Representation structures
		...	
	1xx		Integrated Application Resources
		101	Draughting
		...	
		109	Assembly model for products
Top parts	2xx		Application Protocols (AP)
		AP 203	Configuration controlled 3D design
		...	
		AP214	Core data for automotive mechanical design processes
		...	
		AP242	Managed model based 3D engineering
Integrated data models	5xx		Application Interpreted Constructs (AIC)
	...		
	1xxx		Application Modules

As can be seen in table 3, STEP contains several hundreds of parts and every year some parts are added or revised. STEP is actually the biggest of all the ISO standards. The STEP application protocols AP203 and AP214 are the most common ones in the domain of CAD data. As was mentioned earlier, application protocols use integrated resources as building blocks. There are differences how AP's apply IR's, but in case of AP203 and AP214, they share the same definition for three dimensional geometry, assembly data and basic product information. The scope of AP214 is roughly the same as the scope of AP203 and CAD vendors can support these both with one piece of code. The AP242, which is under development, is planned to replace AP203, AP214 and other mechanical design AP's in future. First version of AP242 should be launched in 2014. (Datakit) (STEP tools inc.) (STEP AP242 Project)

4.2.3 Other data exchange methods

There are also multiple other exchange methods than the previously presented ones, which were originally designed as a standard. Other exchange methods are often called as *non-standard exchange methods*. Some of these methods have been in so wide use that they have gained an own standard. When using these non-standard exchange methods, the user should be extremely careful to avoid all kinds of mistakes in exchange of geometric data. All in all, the standardized methods maybe more stable, but non-standardized methods can be as useful as or even more useful than the standardized methods in some cases. This chapter presents two exchange methods, which were not originally designed as a standard. (Stroud & Nagy 2011: 413)

STL (Stereolithography) format was developed originally for rapid prototyping purposes. STL uses simple triangulation method to represent geometry. Each triangle is formed of three curves and three vertexes. These curves, vertexes and normal of the surface can be presented by seven lines of code. STL is a simple and unambiguous method, but also relatively verbose method to describe the geometry, which makes the size of STL files rather large. It is widely used non-standardized method, but also criticized of its inaccuracy because of triangulation. Triangles cannot present curved surfaces precisely, but for rapid prototyping and other NC tools, this is not a problem, because as they are programmed, the edges of triangles can be programmed to be made of smooth curves. In rapid prototyping the model has to be intersected by a plane. Intersecting can be done easily for STL models and because of this they are really common in rapid prototyping use. According to some sources this is kind of a heritage from the past and nowadays the computational power has grown enough so that it would be possible to use exact models instead of STL. However, STL is still widely used in rapid prototyping and also as a non-standardized exchange method of geometric data. (Stroud & Nagy 2011: 413-415) (Laakko 1995: 259)

Another interesting data exchange method is *JT file format*, which was designed as an industry focused, high-performance, compact and flexible file format for 3D product data. JT file format was originally developed by Engineering Animation Inc. (EAI), but later Siemens PLM Software (at that time UGS Corporation) purchased EAI and became the owner of JT file format. Development of JT file format has been exceptional, because the process has been organized partly as on open source development project in a form of JT Open Program. The program has enabled wide collaboration of automotive, aerospace and consumer products members as well as other interested developers.

JT is mainly used for product visualization, collaboration and CAD data exchange. In the September 2009, it was accepted as an ISO Publically Available Specification (PAS) and in the end of 2012 as an ISO/DIS standard: "ISO 14306:2012 Industrial automation systems and integration — JT file format specification for 3D visualization". According to the standard: "JT can contain any combination of 3D approximate (faceted) data, 3D exact boundary representation surfaces (NURBS, B-Spline), Product and Manufacturing Information (PMI), and Metadata (textual attributes) either exported from the native CAD system or inserted by a PDM system. JT contains three possible exact 3D representations (JT B-Rep, XT B-Rep and XT-Parasolid LWPA)." The JT file format can be opened at least with following CAD software: Siemens NX, Siemens I-DEAS, Siemens Solid Edge, Dassault Systems CATIA, PTC Pro/ENGINEER and Autodesk Inventor.
(Siemens PLM Software 2014: JT Open) (ASD: ISO 14306 JT v1)

4.3 Siemens Teamcenter

Teamcenter is a PLM (product life-cycle management) system created by Siemens PLM Software. The terms PLM and PDM (product data management) are often used interchangeably, but the difference between them can be expressed so that PLM is more developed system of PDM. PDM software focuses only on engineering tasks and data, whereas PLM software can contain more comprehensive view of the information that is managed as a part of product life cycle. PLM may contain data such as marketing requirements, sales brochures and post-sales documentation, which are not considered as a part of PDM. As this thesis focuses more on the engineering side of products, the term PDM system will be used of Teamcenter, even though it has more capabilities than just regular engineering PDM system. (Product Life-Cycle Management 2011) (Peltonen 2002: 9-10)

In this thesis the research is focused on the design engineering features of TC, which may be useful in creation of main model configurator. As was mentioned in JT file format chapter, the JT file format is supported by all the major CAD systems. This is why TC utilizes JT file format in its actions and is suitable for these major CAD systems. There is no need to exchange CAD data with other methods as TC and CAD systems both support JT file format. This has also great advantages for collaborative design work, because different departments can have different CAD systems, but still work on the same models in shared Teamcenter PDM system. (Siemens PLM Software: Teamcenter 2014)

Usually the integration between CAD and PDM software can be either unidirectional or bidirectional. In unidirectional integration the data is transferred only from CAD software into PDM system. In bidirectional integration it is possible to transfer data also from PDM system into CAD software. For example, the product structures (assemblies) can be modified in PDM and then sent to CAD, where the modifications can be seen in the structure as well. Same thing works into other direction, so if models are removed from an assembly in CAD and the structure is saved into PDM system the removed parts will not be shown in the structure in PDM either. Teamcenter is bidirectional system. (Peltonen 2002: 109)

As new models are created in CAD, they can be saved as items in TC. All the items are named and they get identification numbers (ID) from the system. Items can have also a lot of other data in TC, for example material, manufacturing, or supplier data, which may be needed in the engineering process. This data may have to be input for the item or it may come from some other data source, for example from CAD system. In TC it is possible to search created items with different methods, for instance by the name of item, ID, material or name of creator. All the TC users are tracked, so that it is possible to see, who has created a model or modified it. As a newly created assembly is saved from CAD system into TC, the TC will automatically create hierarchical structure of it. These structures can be modified in TC and new data can be given to the assembly itself, or to the parts of it. The feature of TC, which focuses on structure management, is called Structure Manager.

Structure Manager offers plenty of interesting features, which could be useful for the creation of layout model configurator as well. Of course, TC is important in the role of storing CAD data and enabling all the designers to use it, but it is possible to make the configuring itself also in TC. In Structure Manager there is a tool called Options and Variants (O&V), which can be used for product configuring. (Liira 2014)

Options are variables, which control the whole configuring process in TC. As new items are configured, the user inputs values only for created options. Relations created in CAD system can be exported into TC and made dependent of options by option constraints. (Liira 2014)

Variants are parts, which are either included in the configured structure, or excluded from it. With variant conditions the variants are made dependent of option values. O&V tool has its own user interface for configuring, in which values for options are input. It is also possible to include different kind of questions and selection tables in the user interface to make it more understandable. Basically, any user without knowledge of using Teamcenter, should be able to configure new variant with O&V tool. (Liira 2014)

5 Specification of configurator

The main objective of the configurator is to produce layout models and main dimension drawings for selected VSD motors, as was explained already in the introduction chapter. The layout model can be delivered to customer for spatial analysis, design of installation and to clarify the agreed requirements in demonstrative manner. The main dimension drawing is an important document for the customer, but also for the whole order-delivery process. Many different things can be verified from the main dimension drawing during the design and manufacturing process.

This chapter defines the specification for the 3D configurator. In the first subchapter it is explained, in which part of the process the configurator should be utilized and how it can improve the whole process. Because synchronous motors are not fully configurable products, but have also a lot of customer variance, it is discussed, which features should be taken into consideration before starting to create the 3D model configurator. Second subchapter deals with functional requirements for the specific configurator, which is the outcome of this thesis. In the last subchapter the basic structure of VSD motor is described from the point of view, which serves the creation of 3D model configurator.

Based on the literature research it is decided that the layout configuration model will be consisted of solid models. NX uses feature-based and parametric modeling, which are efficient in creation of configurable models. Modeling happens mainly by utilizing sweep creation, but features are added also with CSG modeling methods. Skeleton modeling will be utilized in assembly creation. All the component models will be constrained to the created skeleton, so that assembly becomes more robust and any of the components can be removed or switched to another component.

Assembly creation happens with bottom-up assembly design method, as all the parts have already been designed in earlier motor projects. Basically, the configured layout model is still a sketch of final design, as changes may occur in components according to customer requirements. Also, most of the models cannot be used for creating manufacturing drawings as they only present correctly the geometry, which is shown in main dimension drawing. Because the configured layout model presents the outer dimensions and space for different parts in the assembly, it can be considered as a base model, which is normally used in top-down design process. The detail design of components continues still after configuring the layout model.

The available design tools are limited to I-Deas and NX, as was defined earlier. It is decided that all the modeling happens in NX, but already created I-Deas models are also utilized. The STEP file format will be used for data exchange. Also JT-model and IGES format were tested, but STEP worked the best for this specific purpose. The final model will consist of parametric and variant models. For user interface there were many options, but it is decided that Options & Variants tool in Teamcenter will be selected for this purpose. Some basic research was done also for other options and reasoning for choosing O&V tool is provided in chapter 6.3.

Different data exchange formats were also tested for delivering the created layout model to customer. It seems that STEP 203 is the most compact format and worked the best for this purpose, so it is recommended that this format will be used for data exchange with customer, if the customer accepts that. The following figure 26 combines all the decided configurator creation methods.

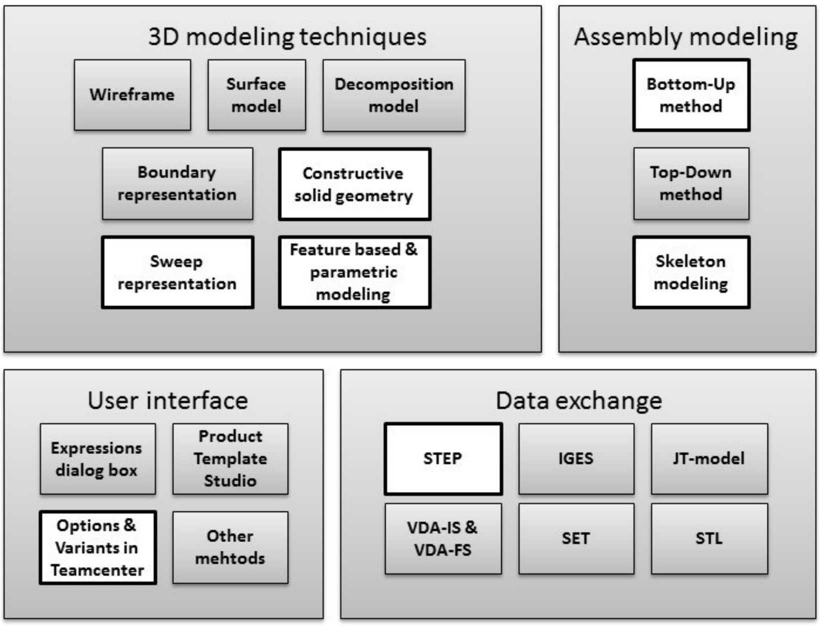


Figure 26. Selected configurator creation methods.

5.1 Advantages of configurator

In chapter 3.2 about configuration models it was discovered that configurable products are an easy method to add some customer variance into a product, but still maintain the advantages of using standard parts. Usually, two kinds of configurators are used in order-delivery processes. Sales configurator is used in sales process and product configurator in product design process. Also, these configurators are already used at ABB's synchronous machines department. The layout model and main dimension drawing configurator locates somewhere between these two configurators in the process chart as can be seen in following figure 27.

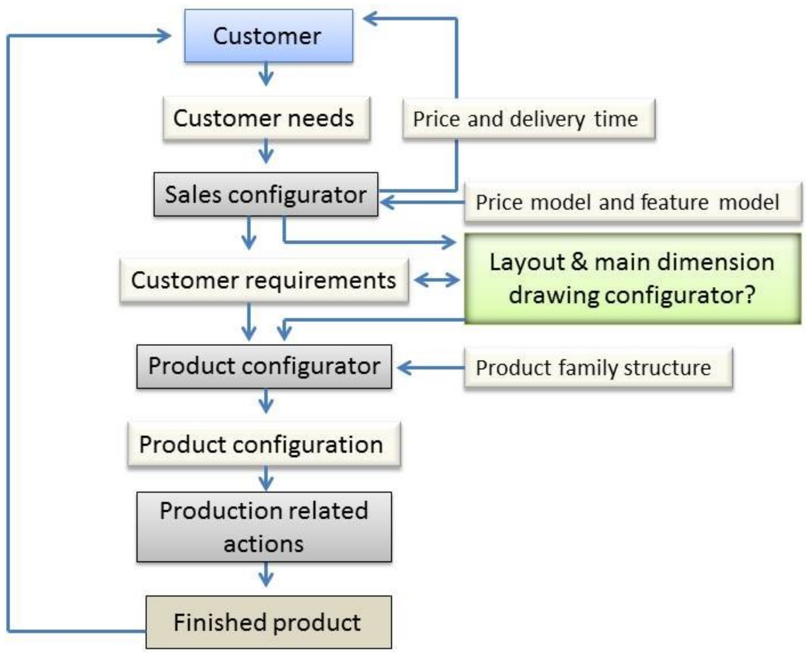


Figure 27. Layout model configurator in relation to other configurators.

The layout model is configured, when the sales process is already closed. However, it is still a little unsure, if some changes to the structure are needed. With the aid of configured layout model, it is possible to ensure that dimensions of motor are suitable for customer requirements. For instance, it is possible to test, how well the motor fits on its mounting place. If the motor does not fit, it is still possible to change the design.

The created configurator makes the mechanical design process easier as well. The main dimension drawing can be created automatically and it is based on 3D geometry, which makes the process faster and more robust. Another advantage is that created layout model can be used as a sketch for detail design as in top-down assembly design process. Even though the actual design work would be still performed in I-Deas, the configured layout model makes it easier for the designer to perceive the whole project and it is possible to measure spatial limitations from the layout model.

As was studied in chapter 2.4 about parametric and variational modeling, creating parametric models is much more time consuming than creating simple product models. Sometimes it may not be useful to create configurable models to fulfill the whole variety of products. Instead of configuring parametric models, it may be faster to model simple 3D models for different variations. There are few important factors, which affect, if it is wise to create configurable model or not.

The first factor to be considered is the sales volume. Synchronous motors are not sold in enormous amounts annually. The biggest advantage can be achieved probably by creating the configurator for the most sold group of products.

The most important factor is the configurable nature of products. Some synchronous motors are more tailor made products, whereas others consist more of standard parts. This has to be taken into consideration and the configurator has to be able to fulfill as large amount of product variants as possible. The more configurable the product is, the easier it is to create the configurator for it.

During the study the above mentioned factors were considered and it was decided that this thesis will focus on creating the layout model and main dimension drawing configurator for VSD (variable speed drive) motors AMZ 0710 and AMZ 0900. The structure of these products is modular and mainly fulfills the specification of configurable product. Because of this, layout models can be effectively created by configurator. These products have had the highest total demand of all the VSD motors during the last four fiscal years and it can be assumed that they are the most sold ones also in the near future. That is why the biggest advantage can be reached by creating the configurator for these products.

The main components in these two motor types are similar, but have many differences in shape and size. Because of this it was decided that two separate configurators will be created. The structure of motors is discussed in more detailed level in chapter 5.3 and especially the structural features, which have effect on creation of configurator, are taken into consideration. Even though there are many differences in structures of AMZ 0710 and AMZ 0900, they also share many components. This eases the modeling work significantly, because some of the component models can be used in both configurators.

5.2 Functional requirements

Because the configurators are used only for creating layout model and main dimension drawing of purchased motor, it is not necessary to include detailed geometric data in the models. In some cases, it could be even harmful to show detailed geometry. Layout models are delivered to customers of ABB and it is not possible to control what kind of purposes customers use the layout models for. Models could end up to usage of competitors as well. In this case, clear competitive advantage would be given to competitors and this is not purposeful in creation of layout model configurator. Geometry of model should present all the data of motor in a form, which is informative enough for the customer, and for the other purposes of main dimension drawing. The geometry has to possible to deliver to the customer by using suitable data exchange method.

The geometric requirements for the configurator are the following:

1. Presents the outer geometry as it is shown in main dimension drawings at the moment (some additional data may be included compared to older drawings, which do not include all the necessary data)
2. Geometry may not contain technical innovations or solutions, which are not supposed to be revealed to the competitors of ABB.
3. The geometric data has to be possible to deliver in suitable data exchange format.

The configurator should be easy to use and no considerable extra work should be needed in configuring process in addition to inputting the configuring parameters. The configuring process should be possible to execute significantly faster than creating the layout model with some other methods. Because every motor project is designed separately, the projects have individual differences. Own layout model should be possible to create for each motor project and the model should contain the individual features of each project. Because changes may occur in design after the sales process, the configured layout model should be easy to revise. These requirements are taken into consideration as the configurators are constructed.

5.3 Structure of VSD motor

Variable speed drive motors are used for many kinds of purposes, such as power source of compressors, pumps and extruders in industry. Each application has its own specific requirements and that is why the most suitable solution is discovered together in collaboration with the customer. Each delivery is its own project and designed separately according to customer requirements. Even though VSD motors are tailor made products, the structure is modular, which makes the product easier to manufacture and shortens the delivery time. Also many standard components are used. Typical AMZ 0900 VSD motor is presented in following figure 28. (ABB 2014)

Figure 28. AMZ 0900 Synchronous Variable Speed Drive motor. (ABB 2014)

The main components of VSD motor can be considered as following:

1. Central frame. Inside the frame there are stator and rotor.
2. D-End Frame (drive end)
3. N-End Frame (non-drive end)
4. Cooling system
5. Main terminal box
6. D-End shield
7. D-End bearing
8. N-End shield (not shown in figure 28)
9. N-End bearing (not shown in figure 28)
10. Exciter (not shown in figure 28)
11. Auxiliary terminal boxes (not shown in figure 28)
12. Oil pipes and lubrication unit (not shown in figure 28)
13. Rotor shaft (drive end of the shaft)
14. Foundation plates

Presented main components consist of smaller components and in total several hundreds of items are used to construct the whole motor. However, as only a layout model configurator is created, it is not necessary to present every component as a separate item. Multiple structures of motor projects were researched and compared mostly by browsing through different main dimension drawings and structures in Teamcenter. Based on this research, it was possible to decide, which variants of motors are included in designed configurators.

Basically, parametric components of motors are the central frame, cooling system, oil pipes, foundation plates and shaft end. AMZ 0710 is offered in three and AMZ 0900 in six different frame lengths. The length of cooling system, oil pipes and foundation plates are of course dependent of the frame length. In different shaft ends there is a lot of variation and customer can decide for example the length and diameter of shaft end. Also the geometry of shaft end can be modified. It is decided that these components will be modeled as parametric models. All the other components are regular, variant or optional components, which dimensions cannot be changed by parameter input. The following chapter explains the creation process of configurators.

6 Creating configurator

This chapter presents how the configurators were created by utilizing the existing design tools. The whole process began by modeling the needed 3D models. Modeling was executed as effectively as possible. Most concentration was needed to create the models for parametric parts and they were all modeled in NX. For regular, variant and optional components, the needed models were produced with various methods. It was always researched first, if the model could be sourced from subcontractor, from some other department or find it from PDM system in some usable format. For many components this was successful, but most of them needed to be modeled in NX.

The second phase in the process was to create parametric assemblies of created models. In this phase the most useful 3D structure creation methods learned in chapter 3 were utilized. Skeleton models were created for the assemblies and component models were constrained to the created skeletons. The functioning of created parametric parts was tested as the assemblies were constructed.

In third phase the user interface was created for the configurator. As was discussed earlier, the Options & Variants tool in Teamcenter was chosen for this purpose. Other options were also tested shortly and in subchapter 6.3 the advantages and disadvantages of different user interfaces are listed first. The needed options, option constraints and variant conditions were set so that it was possible to use configurators via O&V tool.

The last phase was to create 2D drawings also known as main dimension drawings for the configurators. In creation of main dimension drawings some special methods were utilized as well. These are presented in subchapter 6.4.

6.1 Modeling parts

Because NX is not used as a main CAD software in the mechanical design department of synchronous machines, most of the parts for configurators had to be modeled in NX. However, as synchronous motors have been designed with I-Deas CAD software for several years, it was possible to find some useful models in I-Deas format. With the aid of data exchange methods it was possible to utilize these existing models in creation of configurator by NX. Some of the needed parts were already modeled in NX by R&D department of synchronous machines, which has been using NX for couple of years already. It was possible to find these models by asking the experts of R&D department or by using search feature in Teamcenter, in which the models have been stored.

During the project it was decided that step file format will be used to utilize already created models, which are modeled with different CAD software. Step file format can be used in all the modern CAD software as was researched in chapter 4.2.2. In this case the most employed application protocol 203 was utilized. The edition 1 of AP203 which is also known as "Configuration controlled design", exchanges geometrical data, such as manifold surfaces and boundary representations. However, at least this first edition of AP203 does not cover constructive solid geometry and more generally, it does not cover construction history. This means that imported step models cannot be used for purposes of parametric modeling. Though, imported step models present geometry as it is designed, so for the most of the standard components it was possible to utilize this method of data exchange between CAD software. (Datakit)

For all the components, which did not need any parametric features, the possibility of step model utilization was researched. Most commonly needed parts existed already as I-Deas models in Teamcenter and it was possible to export them straight into step files. For example, models for frame ends, terminal boxes, cover parts and mounting brackets were exchanged in this manner. Because some I-Deas models are corrupted during the years of revising and migrating, necessary data may have lost from these models. In many cases it was possible to export models into step file format from I-Deas and import them into NX as well. The problems occurred in some cases as the models were once again exported into step files in NX. As is known, the purpose of the configurator is to create layout model, which can be exported into step file and then delivered to the customer. To ensure that configured models can be delivered without problems, it was necessary to test this step export with component models beforehand.

An example of one problem that occurred in these tests is shown in following figure 29. On the left side in the figure there is a step model, which was exported from I-Deas and imported into NX. On the right side the same model is exported again into step file in NX and opened in the very same software. As can be seen, some of the features were not presented correctly. In this case, it was necessary to model the frame end in NX.

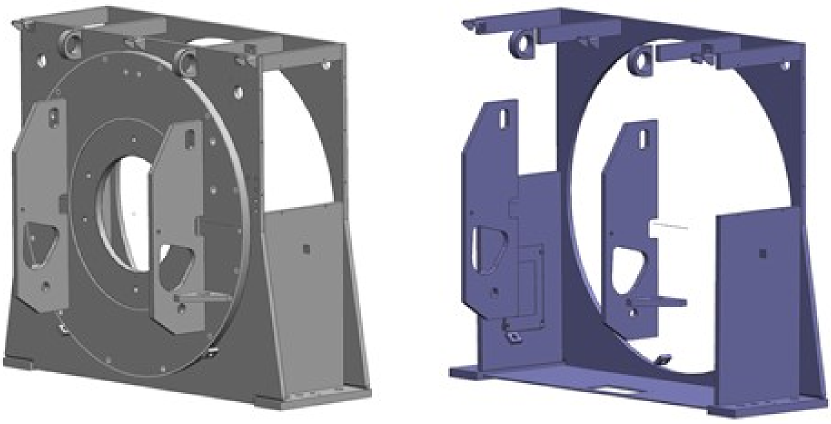


Figure 29. Failed step creation.

The usage of STEP models in configurators may actually ease the revising work of components, at least as long as I-Deas is being used as main CAD software. Changes can be updated to the manufacturing models in I-Deas. New revision of model can be then exported into STEP file and this STEP file used as a new revision of component in structure of configurator. In this manner, there is no need to repeat the modeling work in NX. Because skeleton model is used for constraining the parts in the assembly, the revised model can be easily constrained again in its place in the assembly. Losses of constraints should neither happen in other parts, as all the parts are constrained to the skeleton model.

Many components of synchronous motors are manufactured by subcontractors, so it was possible to source the models of these components straight from subcontractors. Also STEP file format was utilized in data exchange of these component models. For instance the models of sleeve bearings were delivered by subcontractor as well as the models of lubrication units. STEP file formats of models are shown in following figure 30.

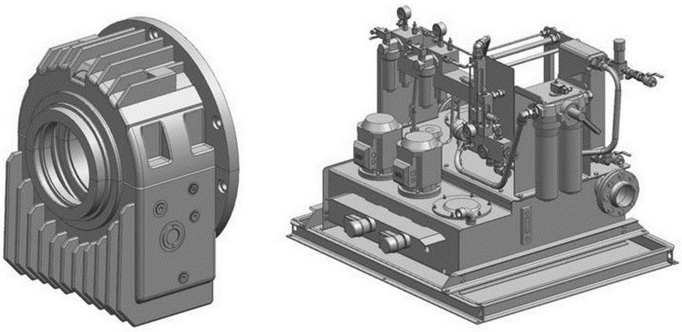


Figure 30. STEP model files from subcontractors.

All the parametric parts were modeled in NX, as well as the parts, which did not have available models in other formats. The parametric parts were designed carefully with fully constrained structures. Expressions were written on part level, but it was tested that they work also on assembly level. Expressions were imported to the assembly level for testing with interpart reference tool, which was discovered in chapter 4.1. All of the main component models of AMZ 0710 can be seen in following figure 31.

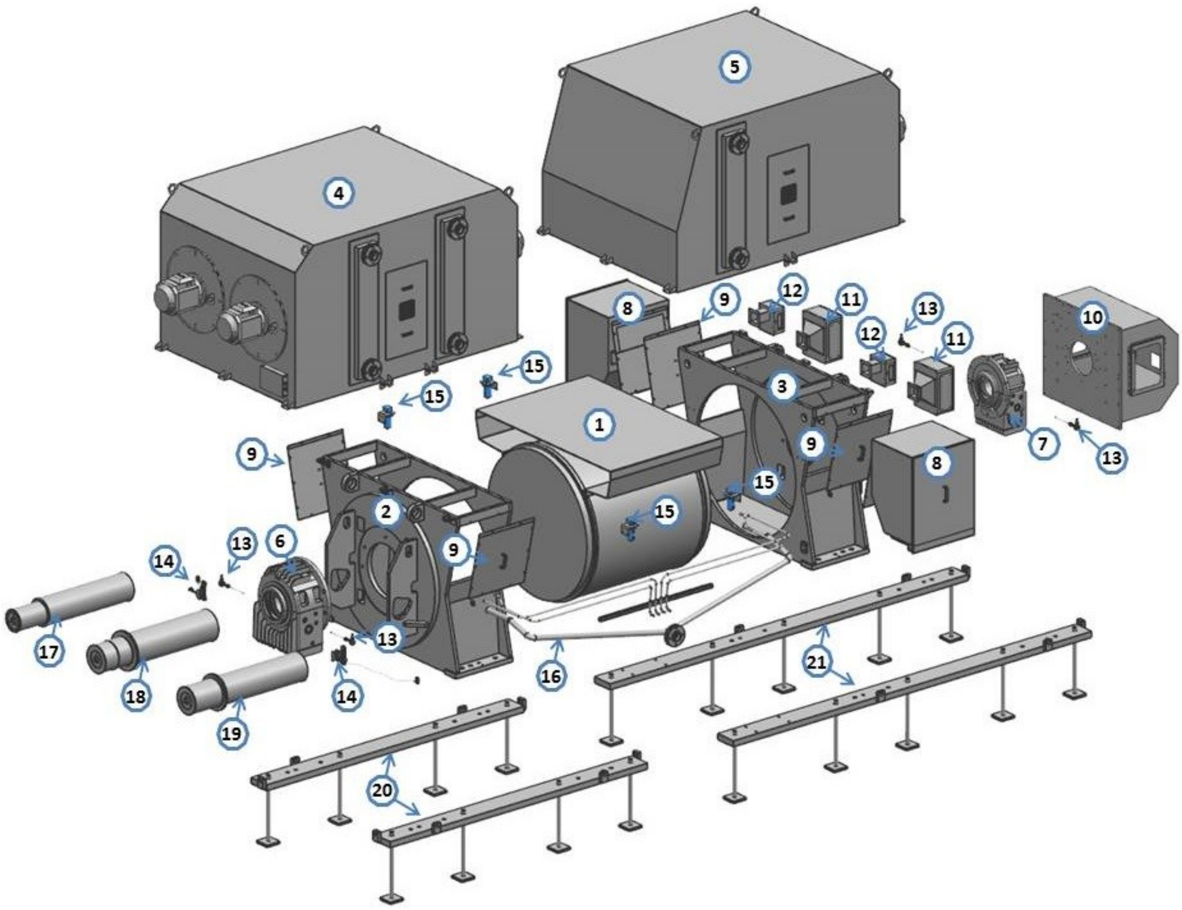


Figure 31. All the part models of AMZ 0710 configurator.

The structure of AMZ 0900 is really similar to the structure of AMZ 0710 and many of the component models can be used interchangeably. The biggest difference is in the frame, which had to be modeled separately for AMZ 0900. Also bearings and shaft ends are bigger in AMZ 0900. In following all the main component models and their modeling types are listed:

1. Central frame – Parametric NX model (varying length)
2. D-End frame with end shield – Regular NX model
3. N-End frame with end shield – Regular NX model
4. Symmetric cooling system – Parametric NX assembly (varying length)
5. Asymmetric cooling system – Parametric NX assembly (varying length)
6. D-End bearing – Imported STEP model
7. N-End bearing – Imported STEP model
8. Main terminal box – Imported STEP model
9. Cover plate – Imported STEP model
10. Exciter cover box – Imported STEP model
11. Auxiliary terminal box B3A – Imported STEP model
12. Auxiliary terminal box B3B – Imported STEP model
13. Temperature sensor – Imported STEP
14. Earthing brush system – Regular NX model
15. Leakage water detector – Imported STEP
16. Oil pipes – Parametric NX model
17. Shaft end (Option 1) – Parametric NX model (varying length, diameter & shape)
18. Shaft end (Option 2) – Parametric NX model (varying length, diameter & shape)
19. Shaft end (Option 3) – Parametric NX model (varying length, diameter & shape)
20. Short foundation plates – Parametric NX assembly (varying length)
21. Long foundation plates – Parametric NX assembly (varying length)

Many of these main components are assemblies, so the total number of created models was a lot bigger. As is known, all the models created in NX are parametric models, but in this context, parametric model or assembly is especially designed so that its dimensions can be changed by inputting different expression value. As all the models were created, it was possible to create the configurable assembly.

6.2 Creating configurable assembly

A skeleton model was utilized to constrain the parts in the final assemblies. As was studied in chapter 3.3, skeleton modeling provides more robust, safer and design friendlier method of creating an assembly. In usual product design a skeleton model is designed first and it can be utilized as a layout of the final product. Features of skeleton can be copied into part models and utilized in detail modeling. In this case the skeleton was utilized mostly for constraining the created parts in robust method. Constraining part models to skeleton model enables that any part model can be modified or removed from the assembly and no harm will be caused to constraints of other parts. If skeleton model is not used in assembly modeling, parts are usually constrained to features of other parts. If these features are changed, there is a huge probability of losing constraints. The plain skeleton model of AMZ 0710 configurator is shown in following figure 32.

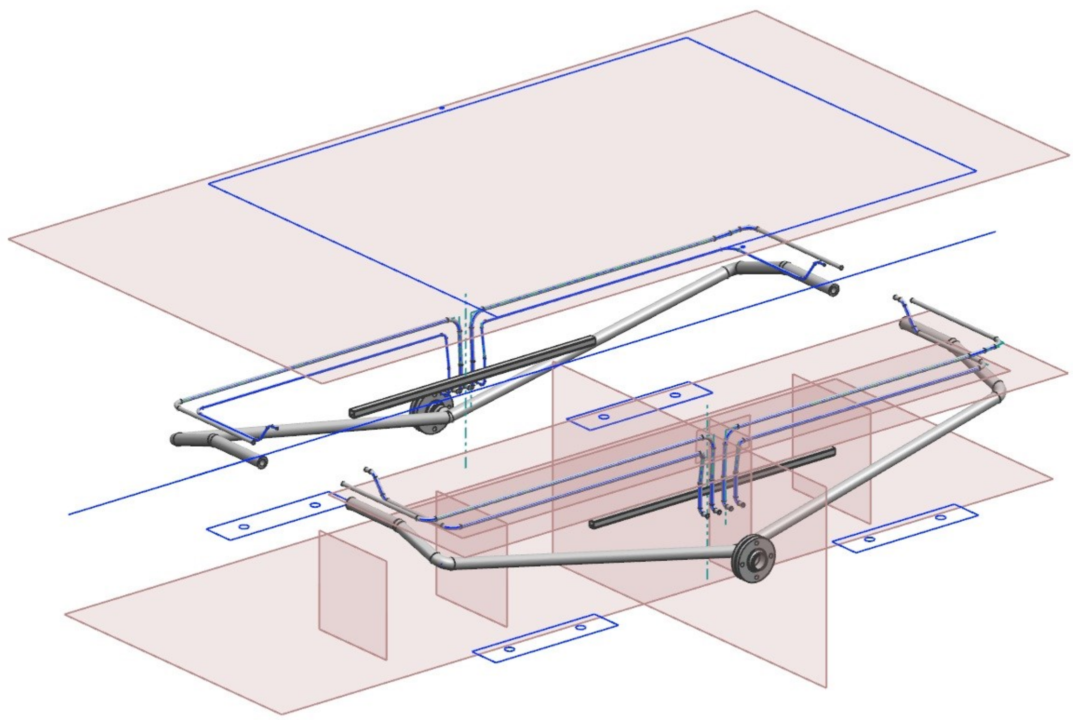


Figure 32. Skeleton model of AMZ 0710 configurator.

As can be seen the skeleton model consists mostly of planes, sketches and curves. Because of simplicity, some of the planes and sketches of skeleton model are hidden in figure 32. The skeleton model has to be parametric as well, so that the model acts correctly, when for instance the length of motor is changed. The sketches, which are used to constrain the frame ends and the cooling system, can be seen clearly in figure 32. The middle line of the shaft is also presented. In this case the skeleton was created in similar manner as a separate part model, but it could have been modeled straight in the assembly model as well.

The skeleton model contains some solid geometry as well, because the oil pipes were modeled to the skeleton model. The oil pipes are a little different component compared to other components of motor. They are not included in usual product structure of motor projects as separate model and because of that no exact information of their geometry was available. Oil pipes are installed manually as the whole motor is assembled and installation is done according to separate instructions. Because oil pipes are shown in main dimension drawing, it was necessary to include them as a separate model into the layout model.

The easiest method to model oil pipes was to add the frame of motor in the assembly model and then create the layout sketches for oil pipes in the skeleton model. First, created layout sketches were copied into a separate oil pipe part model with WAVE geometry linker tool. Some problems occurred in configuring process later and it was necessary to model the oil pipes straight to the skeleton model. The oil pipes are rather special part in configurator from many perspectives. First specialty is that there are no detailed drawings of them. Secondly, the shape of oil pipes is relatively challenging to model compared to other components. The third deviant feature of oil pipes is that their appearing in the configured structure is not controlled in similar manner as the appearing of other variant components. Oil pipes are controlled by suppression expression as other components are controlled by variant conditions created in Teamcenter. This means also that after configuring a new variant item, appearing of oil pipes on can be easily changed.

After refining the skeleton model, all the parts were added into the assembly model and constrained to the skeleton model. The expressions were tested once more by using interpart reference tool and importing all the expressions to the assembly level. For instance, changing the length of motor has effects on many parts in the assembly: central frame, oil pipes, cooling system as well as to the skeleton model. When all the expressions were on assembly level, it was possible to relate them to one length parameter. Changes could be seen in all the models by inputting different values for this single length parameter. The final assembly of AMZ 0710 configurator is shown in following figure 33. The assembly may seem a little odd by first sight. This is because the configuration model contains all the available variant component models. For instance, asymmetric cooling system and symmetric cooling system are both shown in figure 33. When anew variant is configured, it is defined which ones of the models are included into the configured structure.

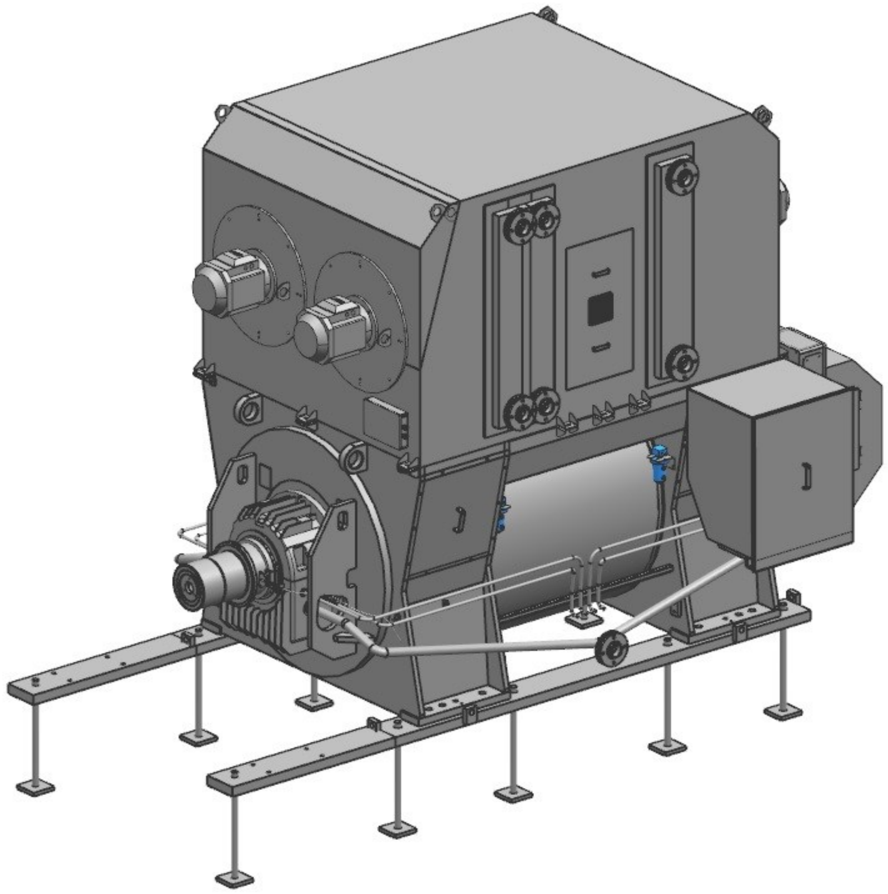


Figure 33. AMZ 0710 configuration model.

6.3 User interface

This chapter reasons the decision of choosing Options & Variants tool as a user interface and explains how the user interface was created. Basically, the user interface was not decided before the configuration model was finalized. Different options for user interface were examined, as the modeling work was performed. The first subchapter defines briefly the different user interface options and reasons the final solution. The second subchapter explains how the parametric relations were coded to function in the user interface. In 3D model configurators, it is possible to manipulate, which part models appear in the configured structure and which not. The final subchapter describes how this was done by selected user interface.

6.3.1 Selecting the user interface

Few different options for user interface were studied during the research process. The possible user interfaces were: CAD software NX 8 itself, Options & Variants tool in Teamcenter or some other user interfaces, for example Microsoft Excel. Basic level research was done for all of these user interfaces.

In NX there are few different options to use the configurator. Every time parametric or configurable models are created, it is possible to input parameter values in Expressions dialog box. This is the simplest method to control the parametric model and the most useful, when there is no need to create more visual, easier to use or more informative user interface. Comments can be added to comment raw after each expression and these can guide the user through the configuring process. Created configuration could be cloned and new ID given to it for each customer project.

To use the configurator in this manner, the user should know exactly how the Expressions dialog box and Clone operation work. Also, it is not possible to add logical questions or phrases in expressions dialog box to guide the user. For instance, in case of variant components, the user can only input value 1 or 0 to show or to suppress certain part model. No logical selection table can be made. Another problem related to variant and optional components is that suppression does not remove component from the structure. For example, three different main terminal box types can be offered for certain motor type. One of these main terminal boxes is chosen to be included into the configured structure and if configuring happens in NX, this is managed by suppressing the other two main terminal boxes from the structure. As the configured structure is saved into Teamcenter with new ID, all three main terminal boxes would be shown in structure of configured item. This may be a little confusing, at least if the configured structure is observed in TC.

With a tool called Visual editor it is possible to add graphical instructions to help the input of parameter values in Expressions dialog box. The problem of a little confusing appearance of Expressions dialog box could be improved by using Visual editor. Also, Microsoft Excel is useful for inputting expression values in more simplified manner. In Excel it is possible to create really easy to use and visual selection tables and transfer input values into NX. Even though the usage of Visual Editor or MS Excel would guide the user in inputting the parameter values correctly, the problem related to usage of suppression status would not be removed. Another problem is that the original configuration model should be available for chances, if the configuring was performed in this manner. The chance of losing some crucial information from the model would be significant. Because of these reasons, it was decided not to use the configurator in NX, even with the help of Visual editor or MS Excel.

Another option as user interface is Product Template Studio, which was presented already in chapter 4.1. With the aid of PTS, the usage would be easier than in Expressions dialog box and the cloning of parts could happen automatically. The problem in using PTS is the lack of author licenses. The configurator could be used via PTS with consumer license, but to create the user interface and also to revise it, the author license is needed. Because author licenses were not available, this option was discarded.

The last discovered option for the user interface was O&V (Options & Variants) tool in Teamcenter. As was presented in chapter 4.3 there are two methods, with which O&V tool enables product configuring. Expressions created in NX can be transferred into TC and their actions controlled by O&V tool. This happens by creating option constraints in TC and is presented thoroughly in following subchapter. Another method to control configuring process is to create variant conditions by O&V tool. Variant conditions are used either to include parts in the structure or to exclude them. In the end, all the actions are dependent of option values, which are input in the user interface of O&V tool.

Of all the discovered methods, the O&V tool offered the simplest user interface for configurator in this case. Because O&V has not been used in mechanical design department of synchronous motors, there was a great interest to discover its functions. The Options & Variants tool can be used for product configuring as well and one reason in decision of choosing O&V as user interface, was to research, what kind of possibilities O&V could offer for product configuring. The following two subchapters explain in detailed level, how the Options & Variants tool was used to enable layout model configuring.

6.3.2 Creating options and option constraints

Before it is possible to create option constraints, the options themselves have to be created. Options are variables, which receive certain value in configuring process. Parametric relations in 3D model are associated to options, so that option values define the dimensions of model.

In NX the relations are called expressions as was discovered in chapter 4.1. To create options, Variants tab has to be opened in Data Panel of Structure Manager. In the bottom column there is a button called "Create a new option". Teamcenter guides the user through the upcoming creation process. If option is supposed to be related to certain expression in NX, it has to be named exactly with the same name as the expression in NX. In following figure 34 an example of creating option is provided. The exact name "B_DIMENSION_FOR_SOUND_PROOFING_PLATES" is given to the option in TC as is the name of expression in NX.

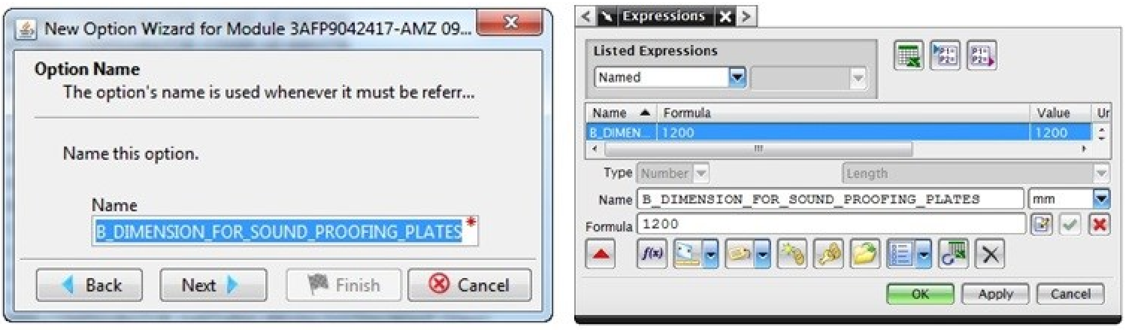
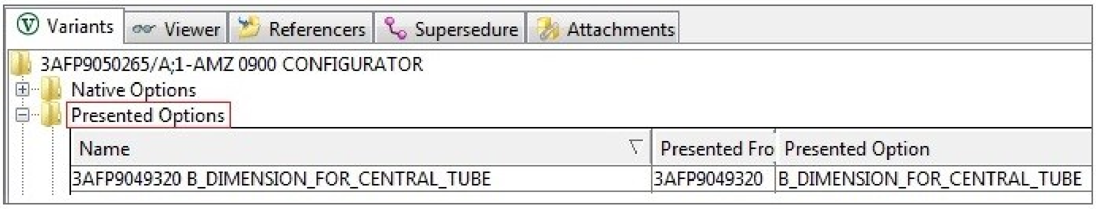


Figure 34. Creating relations between options and expressions.

Now, if a new configuration is created in TC and certain value is given to the option, the related expression in NX receives the same value as the option in TC. The related option is created in TC to the same object, which has the expression in NX. So for example the option in figure 33 above is created for item "SOUND PROOFING PLATES". It appears to the Native Options list of item "SOUND PROOFING PLATES" item.

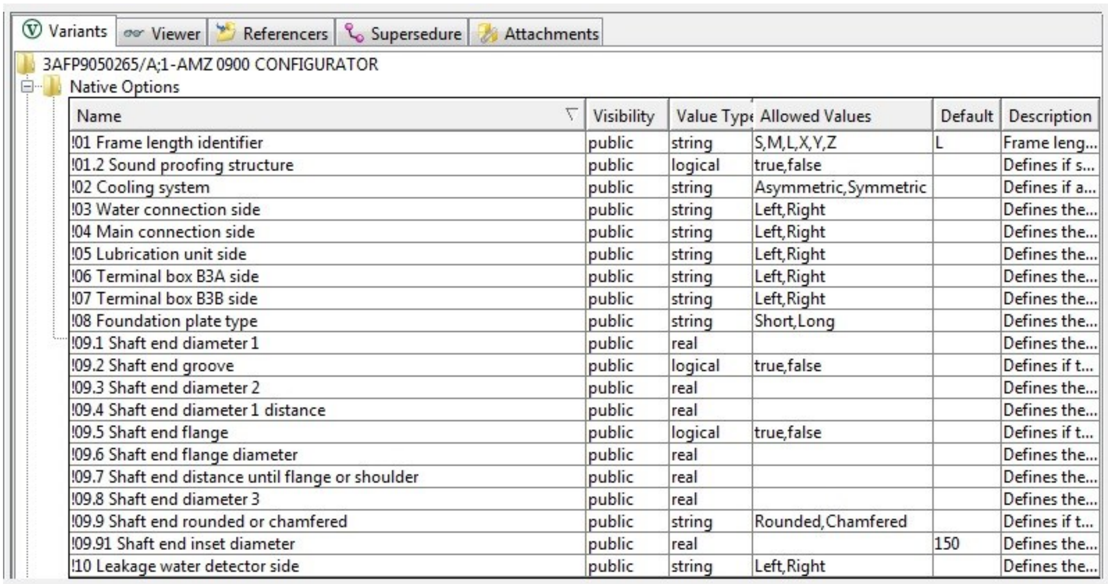
Options can be then presented from lower level items to the top level item in TC, so that all of the options are listed on the top level item. Options, which are presented from lower level items, appear in the Presented Options list of upper level item. Presented Options list is shown in following figure 35.



Name	Presented Fro	Presented Option
3AFP9049320 B_DIMENSION_FOR_CENTRAL_TUBE	3AFP9049320	B_DIMENSION_FOR_CENTRAL_TUBE

Figure 35. Presented option.

As options are created, it is not necessary to directly relate them with expressions in NX. They can be related to each other as well. In following figure 36 the native options created for AMZ 0900 Configurator are presented. Native options are the options, which are created on certain item in TC. It is possible to define different features for created options. For instance, the value type of an option can be defined as a real number, integer, logical value (true/false) or string (writing available options for user). Also allowed values can be limited, default values set and descriptions written.



Name	Visibility	Value Typ	Allowed Values	Default	Description
!01 Frame length identifier	public	string	S,M,L,X,Y,Z	L	Frame leng...
!01.2 Sound proofing structure	public	logical	true,false		Defines if s...
!02 Cooling system	public	string	Asymmetric,Symmetric		Defines if a...
!03 Water connection side	public	string	Left,Right		Defines the...
!04 Main connection side	public	string	Left,Right		Defines the...
!05 Lubrication unit side	public	string	Left,Right		Defines the...
!06 Terminal box B3A side	public	string	Left,Right		Defines the...
!07 Terminal box B3B side	public	string	Left,Right		Defines the...
!08 Foundation plate type	public	string	Short,Long		Defines the...
!09.1 Shaft end diameter 1	public	real			Defines the...
!09.2 Shaft end groove	public	logical	true,false		Defines if t...
!09.3 Shaft end diameter 2	public	real			Defines the...
!09.4 Shaft end diameter 1 distance	public	real			Defines the...
!09.5 Shaft end flange	public	logical	true,false		Defines if t...
!09.6 Shaft end flange diameter	public	real			Defines the...
!09.7 Shaft end distance until flange or shoulder	public	real			Defines the...
!09.8 Shaft end diameter 3	public	real			Defines the...
!09.9 Shaft end rounded or chamfered	public	string	Rounded,Chamfered		Defines if t...
!09.91 Shaft end inset diameter	public	real		150	Defines the...
!10 Leakage water detector side	public	string	Left,Right		Defines the...

Figure 36. Native options of AMZ 0900 Configurator.

In configuring process some value has to be given for each native option as well as for each presented option. Though, if presented options are related to native options by option constraints, it is not necessary to separately input configuring parameters for them. In following the methods of creating option constraints are presented.

Option constraints can be created with three different tools: internal module constraints tool, child module constraints tool or MVL (modular variants language) editor. Internal module constraints tool can be used to create constraints for native and presented options. Child module constraint tool is intended for creating option constraints for lower level items, without presenting them to the top level item. The third possible tool, MVL (modular variants language) editor actually includes the code of all the created constraints. So, the constraints created with two earlier mentioned graphical tools, appear in the MVL code. However, it is possible to create constraints also by coding them straight in MVL editor.

With *internal module constraints tool* user can set options to get certain values, if created conditions are fulfilled. Only options, which are either created on the item (Native options) or presented (Presented options) to it, can be used. In following figure 37 an example of setting internal module constraint is shown. The selected option "B_DIMENSION_FOR_SOUND_PROOFING_PLATES" is presented from lower level item: Sound proofing plates. The option is related to NX expression, which controls the length of plates. In internal module constraints tool the option can be constrained to receive certain value or the value of another option. In this case the option is constrained to get the value of 1000. Also, different conditions can be added. Typical coding statements: if, and & or are available in creation of conditions. In this case the condition is that native option "!01 Frame length identifier" has to get value M in configuring. So, the created internal module constraint defines now, that selected option will get value of 1000, as the frame length identifier option gets value M in configuring.

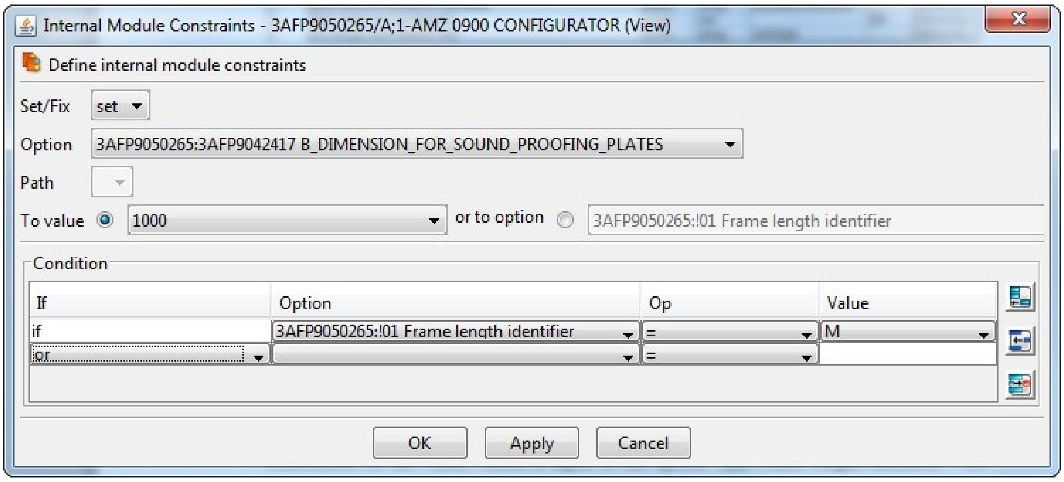


Figure 37. Creating option constraints

Option constraints are created in similar manner by using the *child module constraints tool* as they are created with internal module constraints tool. Difference between them is that in child module constraints tool it is possible to create constraints to the options of lower level items without presenting them to the top level item. So, if it is not necessary to list all the options on the top level item, this tool could be used. The constraint creation window looks exactly the same as the window of internal module constraints tool, but now only the options of lower level items (child modules) can be constrained.

Third method of creating option constraints is to use *MVL (modular variants language) editor*. Constraints can be added by simply writing them in the code. Following figure 38 presents how the code lines look like in MVL editor. The option constraints presented in figure 38 were created by using internal module constraints tool. They could have been added also by writing them in MVL editor, as they are expressed now. When creating new option constraints in MVL editor, the software helps designer by suggesting options to be used. Teamcenter does not always approve all kinds of option constraints created by earlier mentioned methods, but by using MVL editor, these constraints can be created. MVL editor was also utilized, as configurators were created

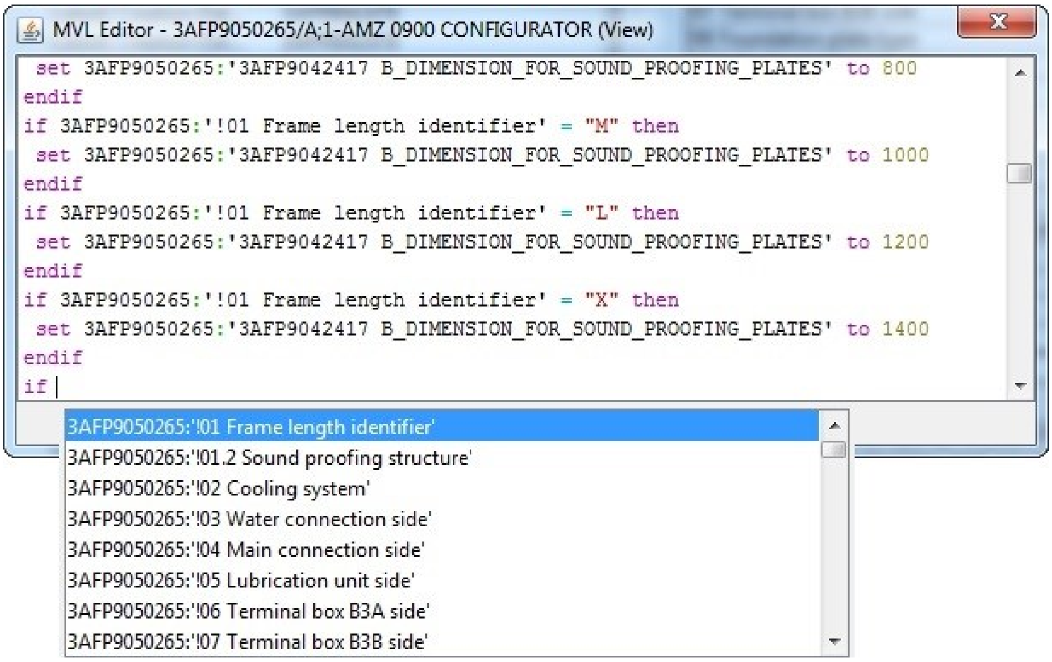


Figure 38. Creating option constraints by Modular Variants Language editor.

First, all the NX expressions were related to options on part level in Teamcenter. This was performed for all the parametric parts, which are listed on page 65. The second step was that native options of top level item (presented in figure 36 on page 72) were created. Option constraints were utilized to relate the native options to the lower level options, which define the expression values in NX. In this manner it was possible to create the configurator so that user has to input configuring parameters only for the native options. As parametric parts were in control by options, it was time to utilize the other tool of creating configurable model: setting variant items.

6.3.3 Creating variant conditions

In Teamcenter variant conditions define, which parts are included in configured structure and which are not. Variant conditions are expressed with the aid of options, similarly to the creation of conditions for option constraints. Configurable nature of VSD motors is mainly based on variant components, so possibility of setting variant conditions turned out to be really useful, as the layout model configurators were created. For instance terminal boxes, cooling systems and foundation plates are variant components in VSD motors.

The variant condition tool was also used for "locating" parts correctly according to configuring parameters. As was shown in figure 33 on page 68, the configuration model included certain components multiple times in the structure. With the aid of variant conditions, it was possible to control, which ones of the components are included in configured structure. For example, the configuration model contains main terminal boxes on both sides of the machine. With variant conditions it can be set that only other one of these boxes ends up to the configured structure.

Variant conditions are set in Structure Manager of Teamcenter. Setting variant conditions happens by selecting the component in the structure and then choosing: Edit/Variant Conditions. In following figure 39 the variant conditions are edited for asymmetric cooling system of AMZ 0900 configurator. As was mentioned, variant conditions are created with the aid of options. Once again, typical coding statements (if, or & and) are available in creating conditions. A simple "if" condition is set in figure 39. Now, as a new variant is configured, if option "!02 Cooling system" gets value "Asymmetric", the asymmetric cooling system part will be included in the structure.

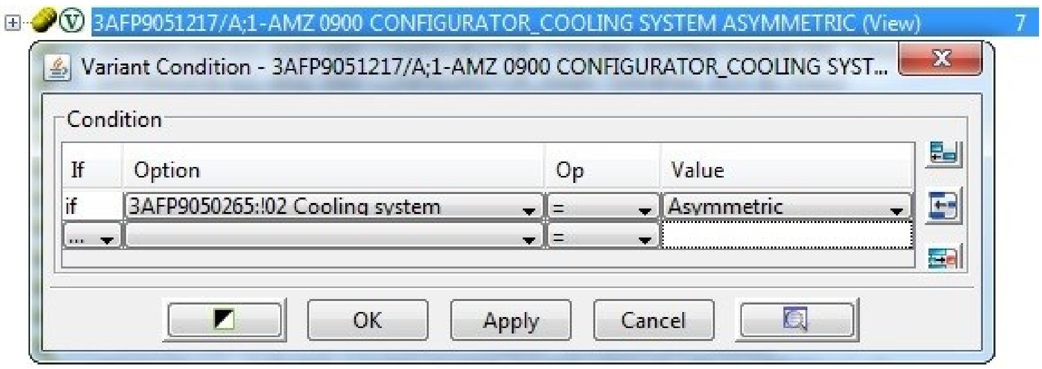


Figure 39. Editing variant conditions.

As the variant condition is set, if the default value for option "!02 Cooling system" was something else than "Asymmetric", the asymmetric cooling system part would automatically disappear from the structure view in Structure Manager. This happens, even before any configuring actions are performed. To avoid this, no default value should be set for "!02 Cooling system" option.

Because the structure contains the same models multiple times, it would be extremely difficult to select the correct model in Structure Manager without any tool. Because of this, the Structure Manager offers viewer tool for detecting the location of parts in the assembly model. Viewer can be used by opening the data panel and selecting the viewer tab. Small windows appear next to the parts in Structure Manager and by ticking those windows it is possible to show the selected parts in the viewer window. The parts can be highlighted by clicking the part in the viewer window or in the structure. In the following figure 40 the whole structure of AMZ 0900 configurator is shown in viewer window. The D-end bearing is selected and it can be seen as highlighted.

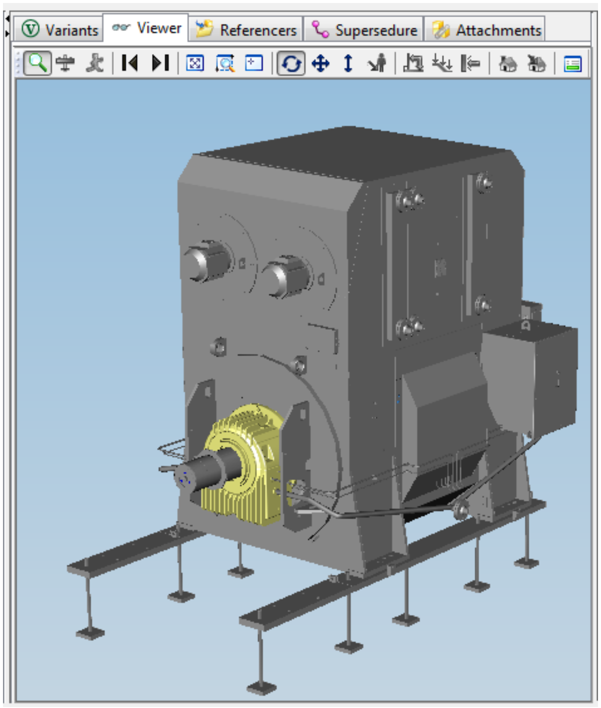
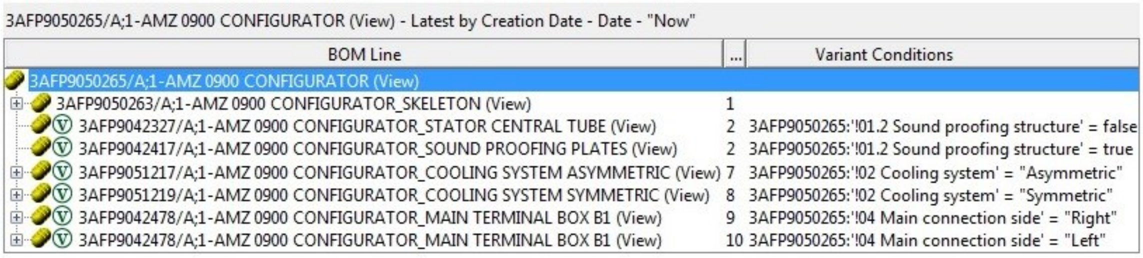


Figure 39. Highlighted D-end bearing in viewer window.

Sometimes opening complicated structures in the viewer window takes a lot of time. Because of this, it is recommendable to add the location of certain part to the additional info field in Structure Manager, as the location is once detected. For example, in the additional info field of main terminal box it was written, if the specific terminal box locates on the right or on the left side of motor from D-end view. Another useful thing is to show the variant conditions column in Structure Manager. All the created variant conditions can be seen at the same time in this column. For all the variant and optional components in the structure, variant conditions were set to be related to the native options with the aid of these methods. In following figure 41 some of these components and their variant conditions are presented as an example. The items, which have variant conditions, are marked by green V mark by Teamcenter.


3AFP9050265/A;1-AMZ 0900 CONFIGURATOR (View) - Latest by Creation Date - Date - "Now"

BOM Line	...	Variant Conditions
3AFP9050265/A;1-AMZ 0900 CONFIGURATOR (View)		
3AFP9050263/A;1-AMZ 0900 CONFIGURATOR_SKELETON (View)	1	
3AFP9042327/A;1-AMZ 0900 CONFIGURATOR_STATOR CENTRAL TUBE (View)	2	3AFP9050265:'!01.2 Sound proofing structure' = false
3AFP9042417/A;1-AMZ 0900 CONFIGURATOR_SOUND PROOFING PLATES (View)	2	3AFP9050265:'!01.2 Sound proofing structure' = true
3AFP9051217/A;1-AMZ 0900 CONFIGURATOR_COOLING SYSTEM ASYMMETRIC (View)	7	3AFP9050265:'!02 Cooling system' = "Asymmetric"
3AFP9051219/A;1-AMZ 0900 CONFIGURATOR_COOLING SYSTEM SYMMETRIC (View)	8	3AFP9050265:'!02 Cooling system' = "Symmetric"
3AFP9042478/A;1-AMZ 0900 CONFIGURATOR_MAIN TERMINAL BOX B1 (View)	9	3AFP9050265:'!04 Main connection side' = "Right"
3AFP9042478/A;1-AMZ 0900 CONFIGURATOR_MAIN TERMINAL BOX B1 (View)	10	3AFP9050265:'!04 Main connection side' = "Left"

Figure 41. Variant components and their variant conditions.

If the configuring would have been performed straight in NX, similar method of creating variant conditions as in Teamcenter would not have been possible. Usually, when configurable models are configured in CAD software, certain features or models can be hidden or suppressed from the structure. This means that the model cannot be seen in graphical window, but it is still included in the structure. Because the oil pipes were modeled in the skeleton model and it is not possible to control certain features with variant conditions tool of Teamcenter, an alternative solution had to be found. The solution was to use “Suppress by Expression” function, which was discovered in chapter 4.1. The suppression status of oil pipe features in skeleton model was set to be controlled by expressions. Options were created in Teamcenter for these expressions of right and left side oil pipes. The options were constraint to get value 1, when certain oil pipe should be shown in the structure and value 0, when it should not be shown.

6.4 Creating main dimension drawing

After creating the configuration model and selecting suitable user interface, it was time to create the main dimension drawing. Main dimension drawing is technical drawing, which shows all the important outer dimensions of motor, accessories and some other essential information. The main dimension drawing is used for many purposes. First of all it shows the customer, what the customer is actually purchasing and confirms agreed details. It is also important document of design engineering and manufacturing.

The main dimension drawings of configuration models were created with normal 2D design capabilities of NX. Drawings were created in drafting mode of NX. In NX, user can easily create new drawing and move into drafting mode by clicking the "new" button in top left corner and selecting "drawing". Normal A2 sized drawing base was selected for the main dimension drawing as well as ABB's title block. The views were placed as they normally occur in main dimension drawings. Also the section view, which shows the drillings of foundation plates, was created. The texts and dimensions were added by using common tools of NX drafting mode.

A lot of effort and concentration was needed to place all the dimensions in main dimension drawing. Because the configuration model contains parametric parts, which dimensions are varied during the configuring, and variant parts, which are either included in or excluded from the configured structure, adding all the dimensions makes the drawing look a little confusing. In following figure 42 the main dimension drawing of AMZ 0710 configurator is presented.

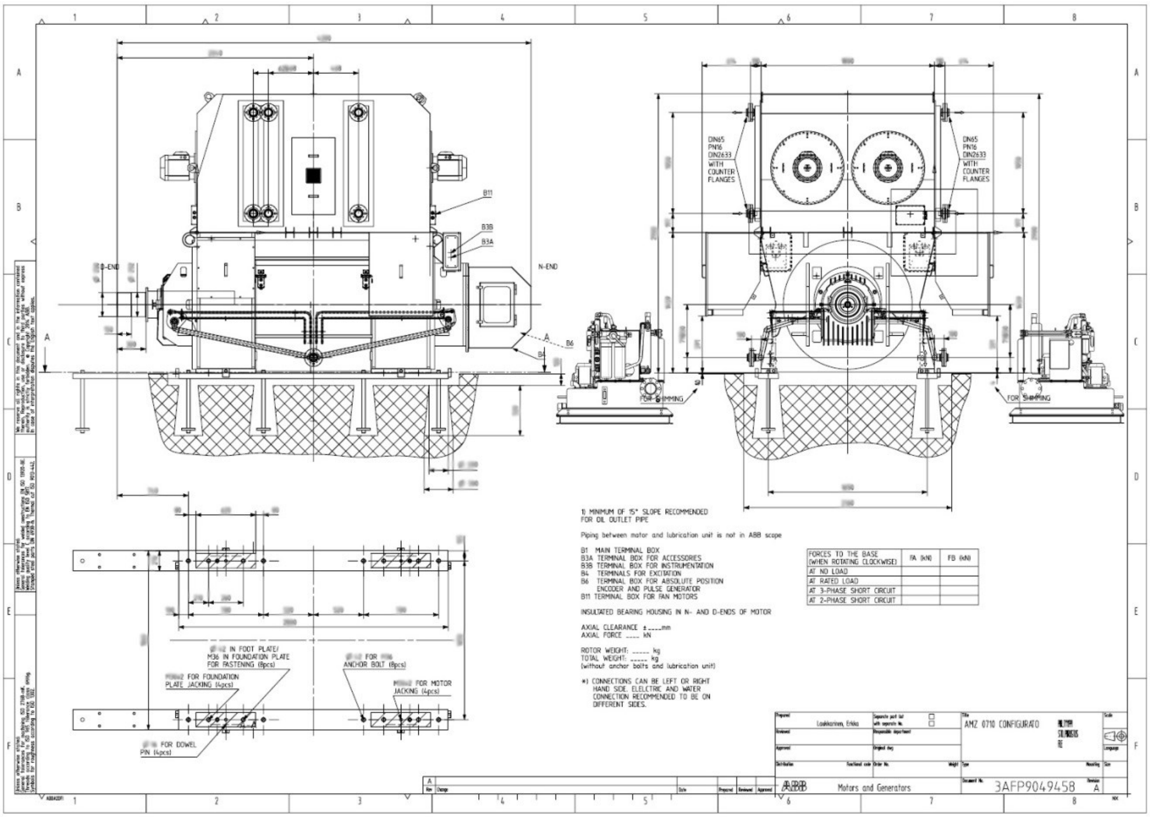


Figure 42. The main dimension drawing of AMZ 0710 configurator.

After configuring suitable variant item with configurator, unnecessary dimensions and annotations can be removed from drawing. This needs some manual work from the designer, but most of the unnecessary drawing marks can be removed automatically in NX. This is discussed more in chapter 7.2.

When dimensions were added to the drawing, extra concentration was also needed to find suitable reference edges and points for the dimensions. Otherwise, it could be possible that references would be lost during the configuring. One example is the length of cooling system. Configurator contains two different cooling systems, symmetric and asymmetric, which both are included in the assembly of configurator. During the configuring process it is decided, which one of the cooling systems is included into the configured structure. If the dimension expressing the length of cooling system would have been referenced to the end faces of cooling system, two separate dimensions would have been needed: one for symmetric and another one for asymmetric. Instead of adding two dimensions, the faces of frame ends can be used as reference and with this method, only one dimension is needed. Dimensions were added in intelligent manner, so that they should not lose their references in configuring process.

As all the parts are included into the structure of configurator, they are shown on top of each other in the drawing. This makes it more complicated to select the needed faces or edges as reference, because there may be numerous of them on top of each other. One good way is not to display all the parts in the drawing, as dimensions are added. For instance, when the short foundation plates were dimensioned, it was really useful to hide the long foundation plates from drawing. This could be done easily by hiding the part in the structure and updating the drawing.

Another useful method to ensure that dimensions will find their references in configured drawing was to use additional points in 3D model. These points were added to skeleton model, which is always present in configured structure. Additional points obeyed all the configuring rules of parametric parts. For instance, the center marks of holes in foundation plates are referenced to the points of skeleton model. The annotations, which describe drillings in foundation plates, are referenced to these points, instead of referring them to geometry of foundation plates. In this manner, it was necessary to add annotations only once, otherwise they should have been added twice; for both, short and long foundation plates. Same kind of additional points were used also to add the centerline of motor and annotations for terminal boxes.

Also, the foundation pit, in which the motor should be mounted, is shown in main dimension drawing. Because the length of foundation plates varies according to the length of motor, the pit has to be parametric as well. However, the pit should not be included in the 3D layout model, which is sent to the customer. This challenge was solved by sketching the pit only to the main dimension drawing. The sketch was constrained by dimensions so that it follows the changes in length of foundation plate correctly. Because two different sets of foundation plates are offered (short ones or long ones), it was necessary to sketch two different foundation pits as well. The dimensions constraining sketched foundation pits should not be shown in the main dimension drawing. These dimensions were moved into layers, which can be easily shown from the layer settings, if needed.

Dimensions for both, short and long foundation plates, were added into the drawing of configuration model. Naturally, only the dimensions of selected foundation plate should be shown in the final main dimension drawing. The finalizing work, which is required from the end user of configurator, was eased by grouping these dimensions. Also the foundation pit sketches were grouped with dimensions. User can select in Part Navigator, which group of dimensions should be shown in the final drawing. This is demonstrated in following figure 43 by showing short foundation group.

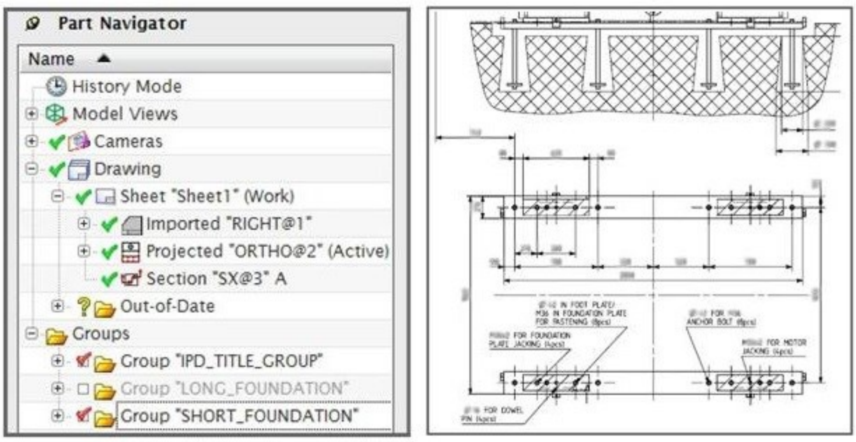


Figure 43. Showing foundation and dimensions of foundation plates.

Basically, layers could have been used also for this task, but usage of groups is perhaps a little easier for the end user and now there is a clear difference between usage of layers and groups. In the layers there are only dimensions, which are needed for modifications of drawing, but should not be shown in final drawing. The grouped dimensions are meant to be shown in main dimension drawing.

Lubrication oil units are also shown in main dimension drawing, but they are not included in the final layout model, which is sent to the customer. Separate model of lubrication unit can be delivered to the customer, if needed. The installation of oil lubrication units is not in the scope of ABB and basically lubrication unit can locate anywhere nearby the motor, as the customer wants.

Two different methods to add the lubrication unit on main dimension drawing without including it into 3D structure were discovered. First method was to create "2D block" of lubrication unit. 2D block is kind of a sign or mark, which can be created of existing lines in 2D drafting mode and added to a drawing. Another method was to add lubrication units as figure. The problem of adding figures in the drawing is that figures are formed of pixels and they cannot be zoomed as the other drawing. Because the drawing is delivered also in DXF format to the customer, it was decided that lubrication units will be added as 2D blocks. The blocks were first created in separate drawing of lubrication unit and then they were added to the main dimension drawing as is shown in following figure 44.

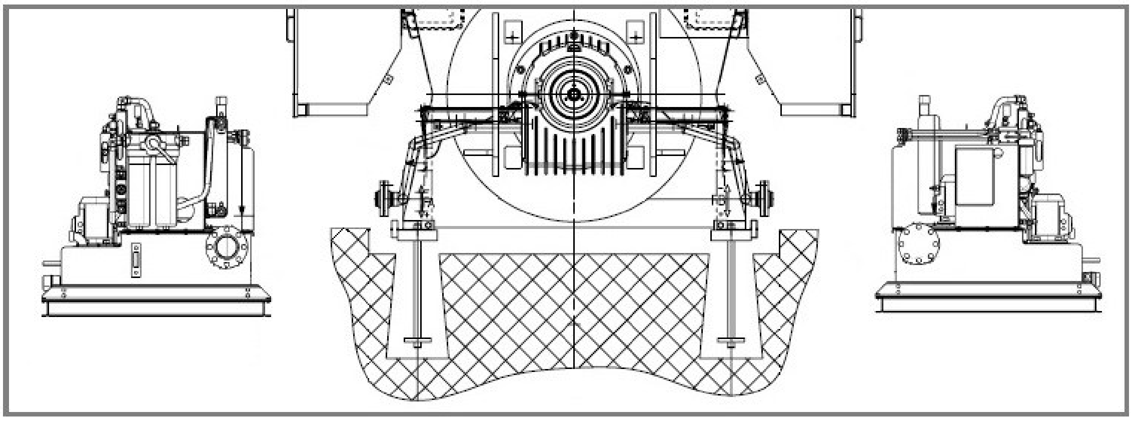


Figure 44. Presenting lubrication oil units in 2D drawing.

In main dimension drawing, certain parts are presented by showing so called hidden geometry. This means the geometry, which locates behind other parts in the drawing, is presented with dashed lines. For instance, oil pipes, water connections and terminal boxes are usually shown in this manner. User can create “Render sets” to control, which parts are shown in view as hidden geometry. This happens by selecting: Preferences / Drafting / Define Render Sets. Targeted parts have to be selected from the structure or in the drawing view, and set has to be named. As the render set is created, the parts gathered into it can be shown in view, by selecting the view and clicking: Edit / Define Render sets in view. Set can be selected from a list of all the created sets and shown in that specific view. In following figure 45 the flanges of cooling system and the oil pipes are gathered into a render set and shown in selected view.

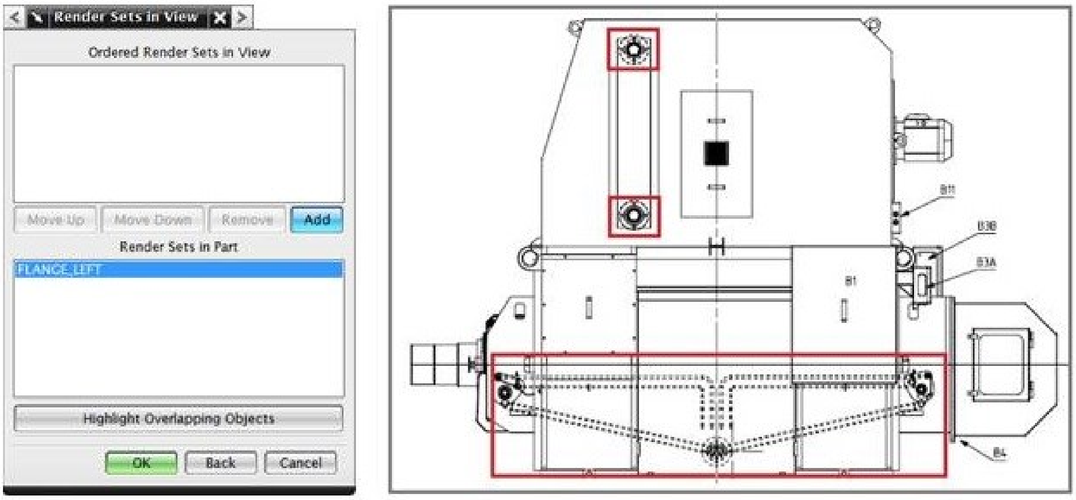


Figure 45. Showing hidden geometry with the aid of render set.

The configurator was ready for usage, as the main dimension drawing was finalized. Both of the configurators were created according to presented creation process. As both of the configurators were finalized, all the items related to configurators were approved in Teamcenter and the specific user instruction was created. The following chapter presents the outlines of using and updating the created configurators.

7 Usage of configurator

This chapter explains in general, how the configurators can be used to create layout models and how the layout model should be delivered to the customer. Configuring of main dimension drawings is discussed in second subchapter. The configurators should be easy to update and revise to ensure that they work correctly as changes occur in the configured products. In the last chapter it is discussed, how revising and updating should be performed. Detailed user instruction was produced for usage of ABB, but the instruction is not included in this thesis. It is extremely important that the configurators can be used in correct manner and the one, who is revising them, knows exactly how to do it.

7.1 Configuring layout model

As is known, the Options & Variants tool in Teamcenter was selected to the role of user interface for configurators. The configuring starts by opening the structure of configurator in Structure Manager. In following figure 46 part of the structure is presented. As the structure is opened and the top level item selected, it is possible to start configuring process by clicking the “Set option values for selected module” icon in the top column. The icon is shown in figure 46 as well. This opens the configuring window, in which the parameters for configuring can be input.

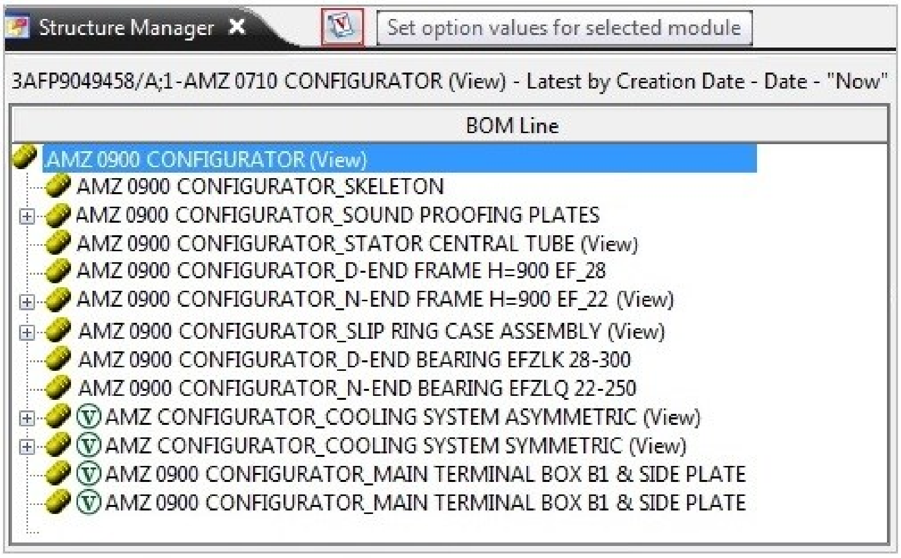


Figure 46. Structure of configurator (partly presented).

The configuring window of AMZ 0900 configurator and the information to be filled for configuring is shown in following figure 47. As can be seen, the configuring window consists of five columns. First column presents the item (item number), from which the option is. In this case, all the options that are shown in figure 47 are native options of configurator item. The second column defines the name of the option. Third column presents the selected value for certain option. As was discussed in chapter 6.3.2, the value type of option defines what kind of values the option can get. There can be also certain restrictions in values. For instance, the option “!10 Leakage water detector side” has only values left or right available. The fourth column shows how the selected value has been set. The fifth and last column shows the description written for the option. Descriptions can be used for guiding the user in configuring process. As new variant is created, certain value has to be selected for each of the native options. Specific meaning of each option is described in user instruction, which was created for internal use.

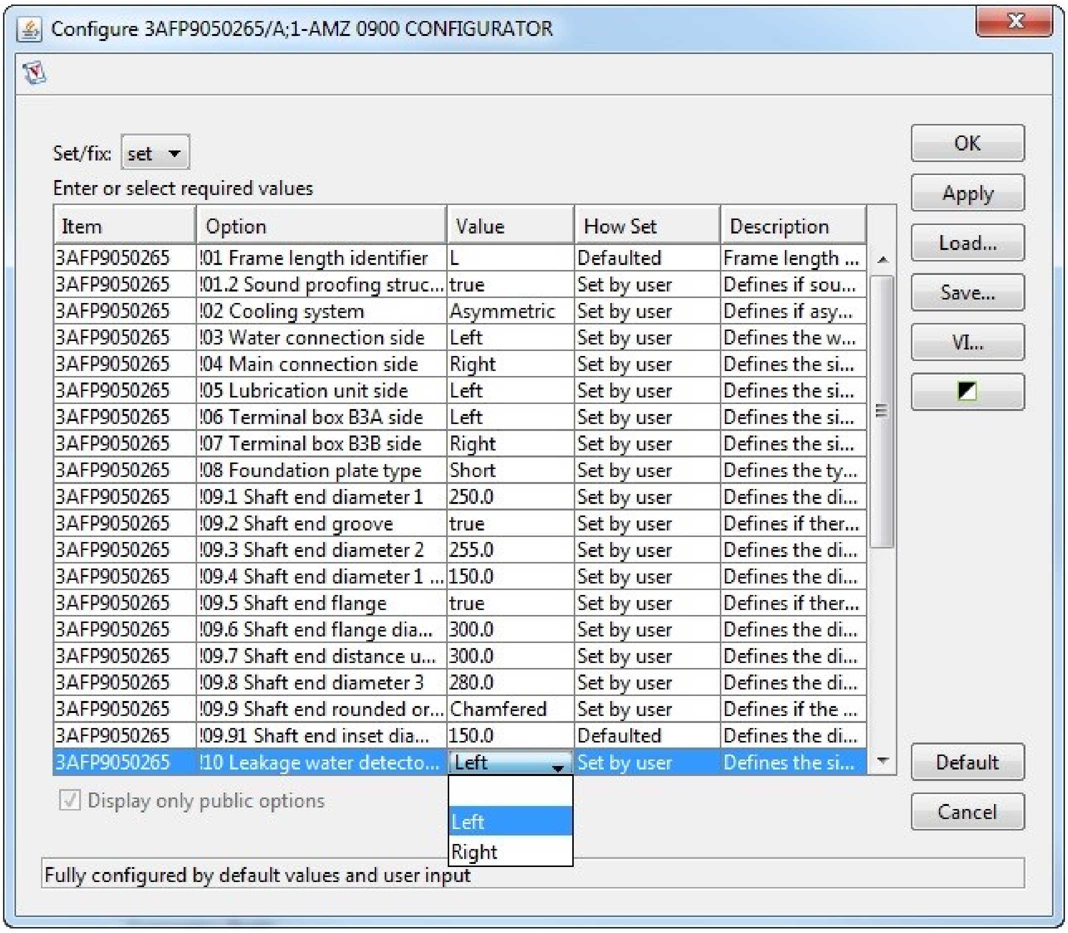


Figure 47. Configuring options in Teamcenter.

When every native option has certain value selected, it is possible to configure a new variant. The text: “Fully configured by default values and user input” in bottom column informs about this fact. User has to click the “VI” button to start the process. The first step in configuring process is to search, if variant items with same option values already exist. Teamcenter is able to link all the configured variant items to the configurator and store their configuring options. In this case, it is supposed to create new product related item, so earlier configured variants cannot be utilized.

System searches also for parametric parts, which are configured with similar option values and could be utilized as a component of a new variant, which is to be configured. For instance, if the frame has been already configured with value “L”, the search feature identifies this configured variant and tries to utilize that in new configuration. The problem with the search feature is that the original generic structure cannot be changed, if it has linked variants. Because of this, it may be beneficial to unlink the configured parametric parts, so that the search feature cannot find them at the time the new configuration is created. Unlinking is discussed more in chapter 7.3. After searching for suitable variants to be used in the substructure, the configuring itself may start by pressing the “Auto VI” button. The text shown in figure 48 appears to the screen.

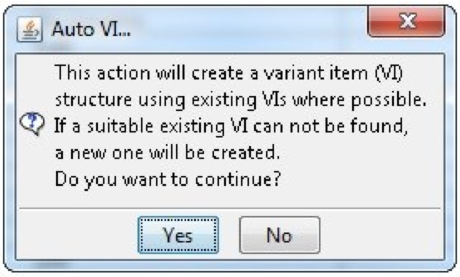


Figure 48. Message about creating new variant item by configurator.

“Auto create of VI structures” window opens and it is possible to begin the creation process by clicking the “Start” button. Teamcenter starts to create new items for parametric parts, which did not already have configured variants to be utilized in this configuration. The following figure 49 presents “Auto create of VI structures” window of AMZ 0900 configurator, in which the creation process is shown.

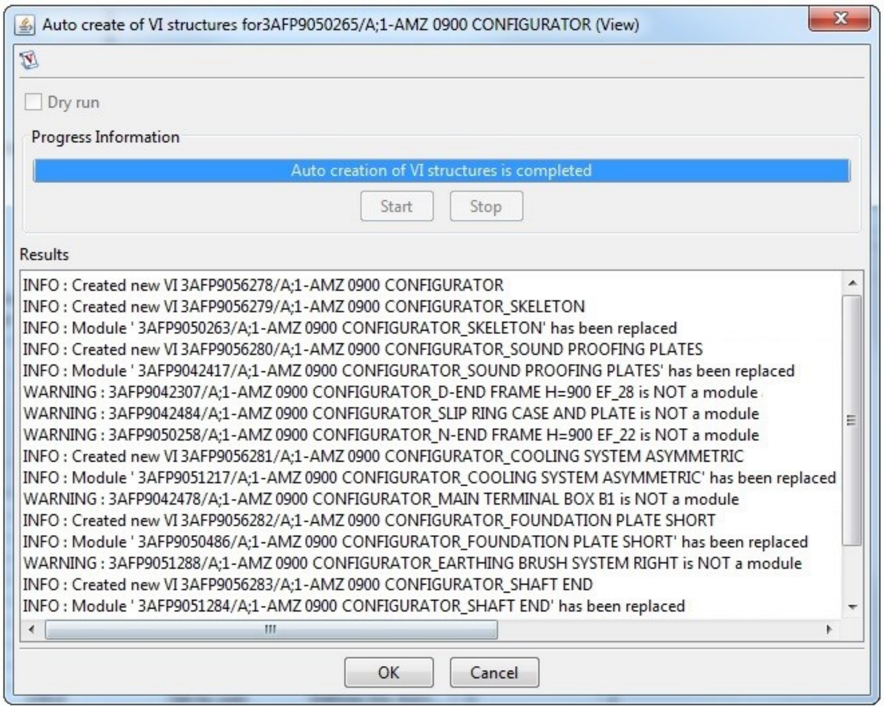


Figure 49. "Auto create of VI structures" window of AMZ 0900 configurator.

TC informs user about newly created items in the window. In this case, new items were created for all the parametric parts. The original parts in generic structure are replaced with these newly created items. The warnings shown in window are related to regular parts in the structure. The regular parts are not constrained with option constraints or variant conditions. In other words, they are parts, which appear always in configured structure in the very same form. These warnings do not need any actions from the user.

The configured structure can be opened in Structure Manager of TC by clicking the "Open" button. Following figure 50 shows configured structure, as it appears in Structure Manager. The items marked by red check mark are the parametric parts, which were configured with new parameter values and received new ID numbers. If unlinking is not done, the search tool is able to find these items next time the configuring is executed. Unlinking can be done by selecting the item to be unlinked and clicking Tools / Variants / Unlink Variant Item from the top column. It should be noticed that unlinking cannot be canceled, once it has been executed. As can be seen in figure 50 the structure does not contain all the variant items, which were in original structure (in figure 46 on page 84). According to configuring options, some of the variant components were included in the structure and others were excluded from it. None of the components has green mark of variant item anymore.

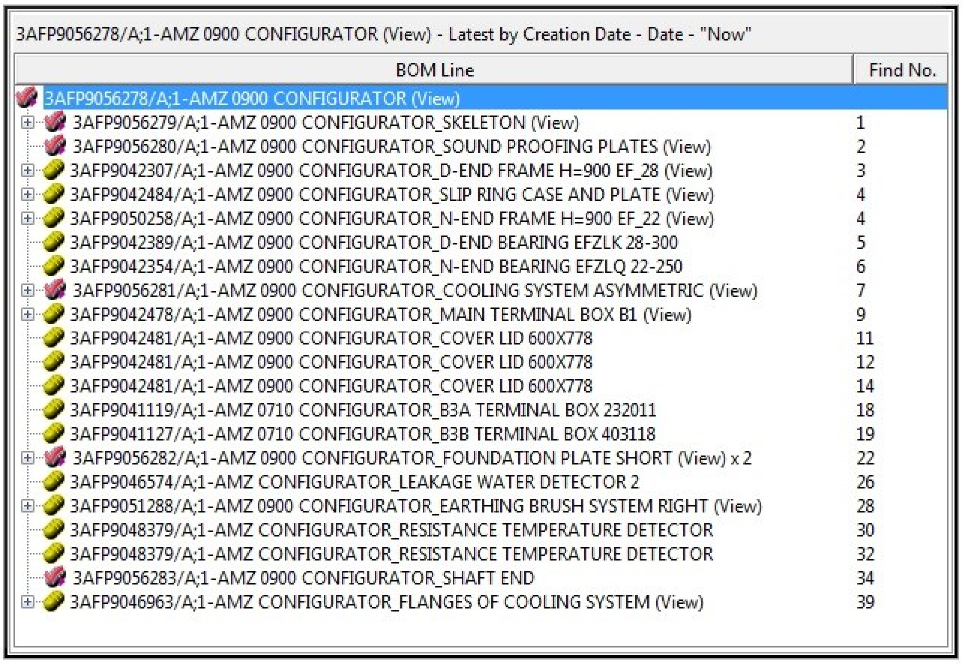


Figure 50. Configured structure in Structure Manager.

The configured layout model can be opened in NX for examination. In following figure 51 the configured layout model is shown. If the configuring process worked as it is supposed to, all the dimensions and appearance of parts should be as was selected in configuring options. If there were some mistakes in parameter inputs, or if the configuration did not appear as it was supposed to for some reason, it is still possible to modify the model. Basically, the parts, which get new ID in every configuration process, can be modified freely. If appearance of other parts is needed to be modified, the part should be cloned before any modifications to ensure that other configured models will not be harmed. One really common mistake in configuring process is that some parts, for example main terminal box or water connection flanges are located on the wrong side of motor. The parts can be simply repositioned or reconstrained to fix this problem. More information about revising the configured models is in chapter 7.3.

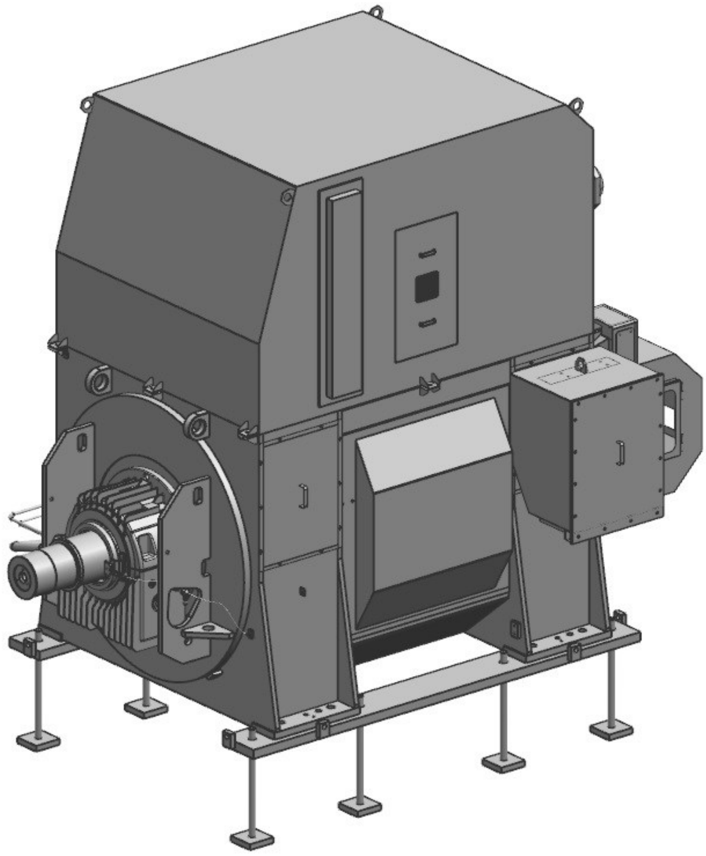


Figure 51. Configured layout model of AMZ 0900

The layout model is ready to be delivered to the customer. At first it should be exported into a suitable data exchange format. The exporting can be done easily by selecting File / Export, and then the suitable exchange method. Layout models are delivered mainly by using STEP. NX supports STEP standards 203 and 214, which are the most common ones for exchanging geometric data. Also IGES and JT-model are supported.

According to study in chapter 4.2.2 about STEP data exchange method, standards 203 and 214 have roughly the same scope. As configured model was exported into STEP 203 file format and STEP 214 file format, it was discovered that STEP 214 has a little smaller file size than STEP 203. In case of configured layout model of AMZ 0900 the file sizes were 18 930 KB and 20 319 KB. Supposedly this was because of the fact that STEP 214 did not present the sketches, curves and etc., which are included in the model or actually in the skeleton part. STEP 214 presents only the geometry as boundary representation whereas STEP 203 presents the additional sketches and curves as well. Both of the STEP models are presented in following figure 52. The numbers of features in STEP 214 file was 391 and in STEP 203 file 476. The difference is not significant, but it would be recommendable to use STEP 214 every time it is possible. In layout model, which is delivered to the customer, there is no need to have the additional sketches and curves. The smaller file size is always easier to send and maximum file size of e-mail attachment is often only 20 000 KB. Also, the fewer amount of features enables CAD software to handle the model faster and better.

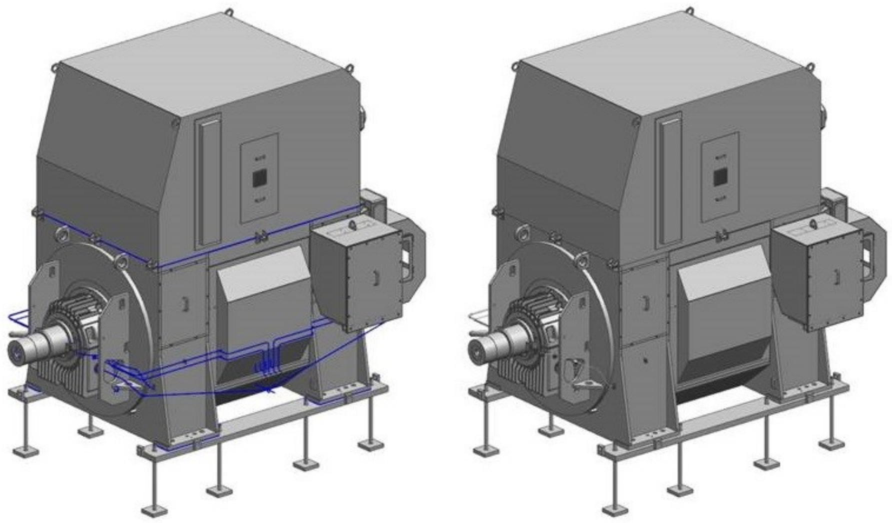


Figure 52. STEP 203 file compared to STEP 214 file.

NX was able to export the configured model into IGES file format without any problems. However, the processing took much longer than in case of STEP files. File size of created IGES file was 55 158 KB and number of features was 11 012. IGES presents every edge and surface, which is limited by edges, as separate feature. STEP method is able to combine parts or other manifolds as one feature and this makes the file size much smaller and the whole model easier to handle. Also exporting JT-model was tested in NX. To export the model as JT-file some changes in settings would have been needed. With the aid of experts from R&D department, the needed changes were made, but still for some reason it was not possible to export the JT-file. As a result of these tests it can be said that STEP 214 file format should be used as a data exchange method every time it is suitable for the customer. Also, STEP standard 203 can be used. These are the most compressed and reliable methods to exchange the geometric data of layout model.

7.2 Configuring main dimension drawing

Main dimension drawing is configured automatically as the layout model is configured. The drawing appears under the configured item in Teamcenter and it can be opened in NX, by clicking the “open in NX” command. Opening the main dimension drawing opens also the layout model in NX. The drawing needs always some manual work, before it is in suitable form. The following figure 53 presents the D-end view in main dimension drawing before it is cleaned up from unnecessary dimensions and annotations.

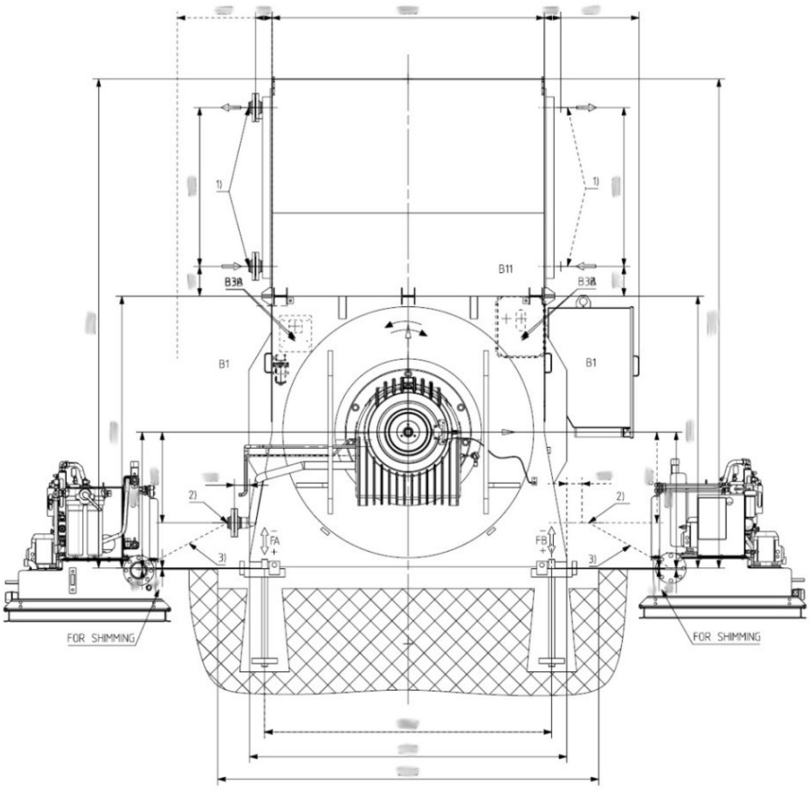


Figure 53. D-end view in configured main dimension drawing.

The first thing to do is to update all the views in the drawing. If there are some unnecessary sketches shown in the drawing, the user should hide them in 3D model and update the views after that. Next task is to remove unnecessary dimensions. The drawing contains many of the dimensions twice, because locations of terminal boxes, lubrication unit and water connections effect on which side of the motor the dimensions should be shown. As was discussed in chapter 6, NX is able to show the dimensions, which have lost their references and also delete them by one command. In figure 53 these kinds of dimensions can be seen marked with dashed line. For example, the dimension related to left side main terminal box, which is not included in this configured structure. The dimensions marked by dashed line can be removed by selecting from top column: Preferences / Drafting / Annotation and changing the retained annotations to invisible. All the other excessive dimensions in the drawing can be deleted manually.

It is also possible that some dimensions need to be repositioned and this can be done simply by dragging dimensions to their correct position. Additional lubrication units should be removed as well. The D-end view after cleaning excessive dimensions and annotations is shown in following figure 54.

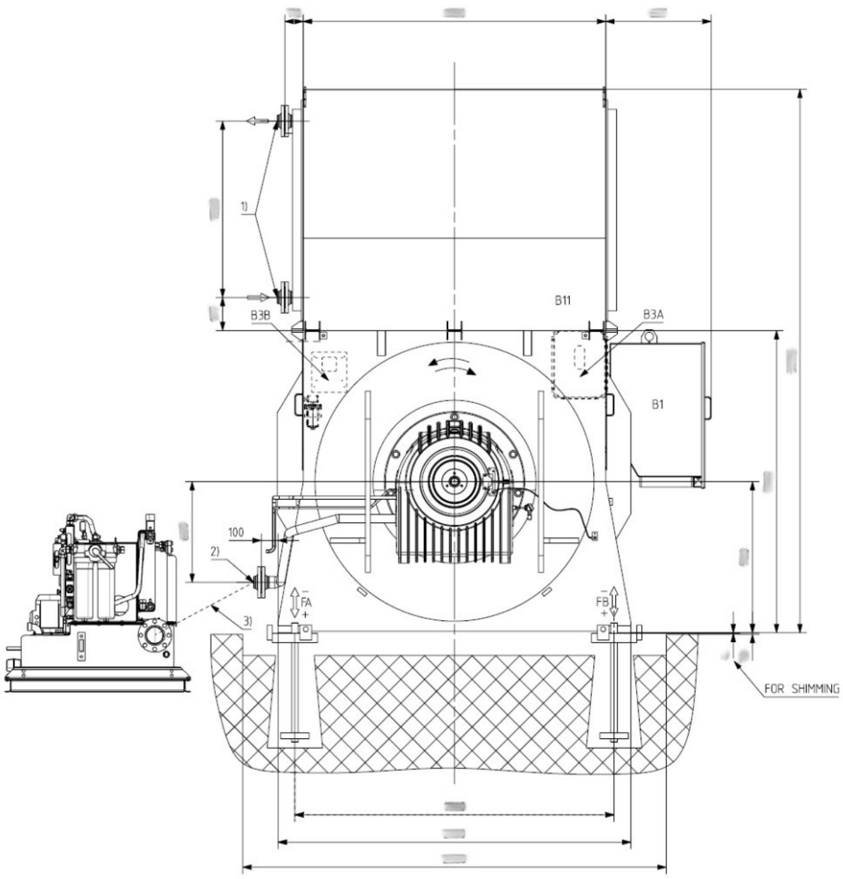


Figure 54. D-end view in configured drawing after clean up.

All the views presented in main dimension drawing are cleaned up from unnecessary dimensions and annotations. The suitable group of annotations and dimensions should be shown to present the foundation correctly in view on the left side. The dimensions will appear to the view presenting the foundation plates in down left corner of the drawing. Also, it should be checked that the other group of annotations and dimensions for foundation is not shown. The Part Navigator is presented in figure 55 below. In part navigator it is possible to select the group to be shown or hidden. In this case the group "short_foundation" is shown and the group "long_foundation" is hidden, as they should be.

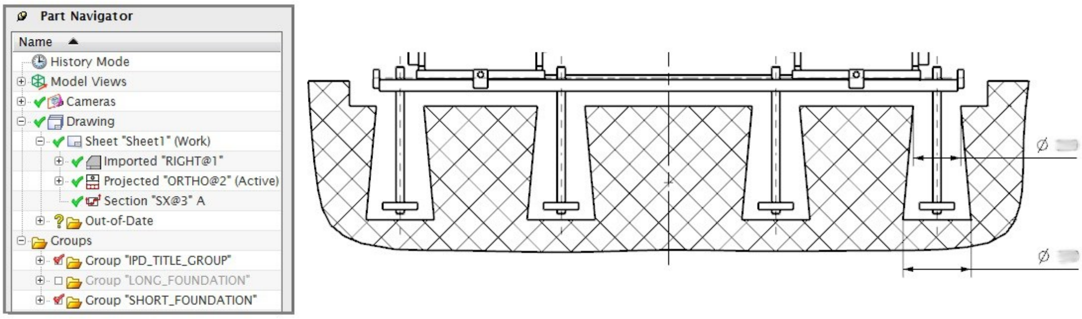


Figure 55. Selecting correct group of annotations & dimensions for foundation.

When the annotations and dimensions of whole drawing are fixed, there is still one thing to do. The title block should be filled in. This happens by adding the information to the main data sheet of item in TC. The filled in information can be updated to the drawing in NX by selecting ABB / Update Drawing Title and Frame. These are found in top column of NX. As the drawing is complete, it can be saved to Teamcenter. Also, PDF file can be created. To ensure that the model and drawing will not be harmed by other users, they can be "Approved" in Teamcenter. Approving items means that changes cannot be made to them without first creating new revisions. Revising the configured models and drawings is discussed in following chapter.

7.3 Revising configured models and drawings

Many times there is a need to revise the approved layout model or more likely there is a need to revise the main dimension drawing. Some changes may occur to the structure and these changes should be shown in the main dimension drawing or in the layout model. Usual changes to the main dimension drawing are changes in informative texts of the drawing. Also, the side of main connection, water connections or lubrication oil connections may change during the project. When customer gets the layout model delivered by ABB, it is possible to notice, if these kinds of changes are needed. Also, some special parts or modifications may be needed in certain project and these should be shown in the layout model and main dimension drawing. These have to be added to the configured model.

The configuring can be performed only once for certain item. In other words, it is not possible to create new revision of item by configuring it. The changes occurred in structure have to be updated to the model and drawing by revising process. Revising means that new version of approved item is created, so that the item is not in approved state anymore and is available for modifications.

Basically, all of the modifications can be done by revising only the main item. For instance, it is possible to add, remove or relocate parts, update changes to the main dimension drawing and also modify the texts, annotations and dimensions in the drawing. Revising process starts by creating new revision of the item in Teamcenter. This happens simply by selecting the item and clicking: File / Revise. New revision letter can be given to the revised item and new revision appears under the main item in TC. Now it is possible to open the model and the drawing in NX and modify the structure. New parts can be added into the structure as they normally are added in NX. Constraints can be deleted and new ones added to relocate the parts in the structure. The changes made in the 3D model, can be updated to the drawing by updating the views. Annotations, texts and dimensions can be added, removed or changed and the title block is available for updating as well.

As the configured variant is modified, it should be kept in mind that all the lower level items are used in the main configurator and in all the configured structures as well. This means that, if a lower level item is revised and modified, the changes will appear to the main configurator and configured variants, as they are updated. For this reason, if some project specific customization must be done to the parts that exist already in the structure, the part should be copied first and the copy of it added to the structure. The old part can be then removed from the structure. By this it will be ensured that no harm is done to the original configurator. Most likely these kinds of actions should be performed, when additional drillings to the foundation plates or to the frame are needed or the geometry of the shaft end should be changed.

8 Results

Different 3D modeling methods and available tools for mechanical design were studied thoroughly. The most important discoveries and observations are presented in this thesis. Also, the requirements for 3D layout model configurator, which were highly dependent of varying structure of synchronous motor, were discovered carefully. Different motor types were compared and research was targeted on types, which can benefit the most of created configurator. As a concrete result of this thesis, two separate layout model configurators were created for AMZ 0710 and AMZ 0900 VSD motors. Configurators were created by utilizing Siemens NX 8 CAD software and Siemens Teamcenter PDM software. Also, detailed user instruction, which enable first time user to manage through the configuring process, was produced.

The configurators enable creation of layout model for mentioned motor types by inputting parameter values in Teamcenter. The layout model is configured in NX according the input parameters. The main dimension drawing is configured automatically at the same time and only minor actions are needed from the designer to refine the drawing. The first subchapter describes, how well the configurators match their requirements and what are the advantages and disadvantages of creating layout model configurators by selected methods. Usage of configurators can be beneficial in many cases and their benefits can be measured from different points of view. The second subchapter measures the benefits of layout model configurator compared to old layout model creation method.

8.1 Configurators

The original scope of this thesis was to research different methods for creating layout model and main dimension drawing configurator with available design tools. The target product for configurator was synchronous variable speed drive motors. During the research, it was decided that two separate layout model configurators will be created. The first configurator was constructed for AMZ 0710 VSD motors and the second one for AMZ 0900 VSD motors. These were the products, which can benefit the most of layout model configurator, mainly because of the high modularization and standardization level of products. Also, the amount of delivered motors during last fiscal years was a significant factor.

After considering different methods for constructing configurators, the most suitable methods were utilized. Some of the component models and all of the parametric models were created by using NX. Existing I-Deas models were utilized as much as possible. Many of them were exported into STEP file format in I-Deas and then imported into NX. Different user interfaces were researched and finally the Options & Variants tool was selected for this purpose. O&V tool is originally designated for product configuring, but it was suitable for layout model configuring as well. The knowledge gained about O&V tool is valuable, if certain products or product components need to be configured in Teamcenter in the future.

The created configurators fulfill their requirements mostly well. In chapter 5.2 the most important functional requirements were listed. Layout models created by configurators fulfill the most of the listed geometric requirements. Configured models present the geometry as it is presented in old main dimension drawings, but more precisely and with some additions. Some components are presented in more detailed level. Still the model does not reveal technical innovations and is simplified enough, so that it can be delivered to the customer. Data exchange works really well, as the layout model can be exported into STEP form easily and delivered to the customer for example by e-mail.

One disadvantage in created configurators is that they cannot cover all the variants of AMZ 0710 and 0900 VSD motors. In test phase, dozens of main dimension drawings of delivered motor projects were printed. It was tested, if the motors presented in these main dimension drawings could be configured by using the layout model configurator. The results were really satisfying, because it was possible to configure almost every motor, as it appears in the drawing. In some cases minor modifications were needed after configuring. The tested projects were not the most exceptional ones and it has to be taken into account that configurator cannot present all the motor variants. However, the test proved that the configurator is able to configure the most delivered motor variants. In the future, the scope of configurator can be widened, by including more variant components in it.

Most of the other requirements listed in chapter 5.2 were achieved as well. The usage of configurator does not need considerable extra work in addition to parameter input. In some cases the layout model needs minor modifications. The dimensions and annotations in the main dimension drawing need to be fixed every time a new configuration is created, but this is really simple task and can be performed without extreme effort.

Other listed requirements were related to unique nature of each motor project and easiness of revising the configured model and drawing, as changes occur. The configurator creates new ID for variant item automatically, as the configuring is executed, so each project can get own unique model and drawing. The revising is relatively easy and for the most of the cases only the main item has to be revised for modifying the layout model or the drawing. All in all, the configurators work as they are supposed to and fulfill their requirements. The configuring process is much faster and more robust method of creating layout model and main dimension drawing than the old methods. This is discussed more in following subchapter.

8.2 Improvements to layout model and main dimension drawing creation

The improvements in layout model and main dimension drawing creation can be examined from different points of view. The most important and measurable properties are probably savings in time and money. Simple comparisons between old method and new method were performed to research how the process was improved. Many different factors had to be taken into consideration in these comparisons. Following paragraphs explain how the layout model and main dimension drawing creation process was improved.

At the moment, if customer wants to have the layout model of motor in the beginning of design process, the designer has to create it manually by using I-Deas CAD software. Designer can use models which are created in earlier projects as a base, but normally full assembly of delivered motor is not composed in design process. The layout model creation was tested with I-Deas in a case, in which most of the parts were already modeled. However, the frame and foundation plates had to be modified. Also, the cooling system, lubrication oil pipes, bearings and several other components did not exist as 3D models. Modifying old models was really challenging, because I-Deas does not support parametric modeling extremely well and old dimensions cannot be changed in easy manner. Modeling of oil pipes was extremely difficult by using I-Deas, because of their complicated geometry and actually no proper pipes were modeled. As modifying the parts is really difficult, the chance of mistake is obvious. In test case, it was really hard to get the dimensions to match the desired once and many rounds of modifications were needed. The whole task took two full working days before it was finished. As the model was created, major problems were faced in exporting the model into STEP file format for some reason.

Compared to creating layout model by configurator, the difference in time consumption is significant. Creating similar model by configurator took only ten minutes. Even though some modifications should have been performed to the configured model, the difference measured in time consumption is multiple hours or even days.

The creation of configurator took a lot of time and that has to be taken into consideration. It could be assumed that person, who knows how to use the tools for creation, could build the configurator in about two or three weeks. So, if more than five different layout models of certain product type would be delivered to customer, it would be easier to create the configurator than to compose the needed models in I-Deas. Of course, if the layout model would be composed by using NX in similar method, as it is done by using I-Deas, the difference in time would not be so significant. NX supports parametric modeling fully and created models can be modified easily. Still, it would take several hours or even days depending of the complicity of product type.

This can be converted into monetary value as well. We can assume that cost of one design hour is 25 €. If designer uses two weeks to create the configurator, the cost of first version of configurator would be 80*25€=2000€. When creating layout models by configurator, the cost of one created model would be 6.25€ (15 minutes work for designer). If designer composes the model by using I-Deas the cost of the model would be approximately 16*25€=400€.

For instance, if it is assumed that during one year six different layout models would be delivered to customer, the cost of one configured layout model would be about 339,60€. This is already much cheaper than the cost of layout model composed in I-Deas (400€). In the long run the difference is much more significant and for example after five years of usage (30 layout models delivered), the cost of one configured layout model would be 72,90€. Layout model is something, which cannot be easily charged from the customer during the project, because it is already assumed that the model is included in the delivery. However, in this case the value of customership is extremely high and providing layout model is part of offering the best possible service. Still the costs have to be cut down and as is proved, the layout model configurator is able to create models much more cost effectively.

The main dimension drawing creation is also really different between old and new method. Old method is based on 2D geometry only. Process starts by looking for as similar as possible main dimension drawing of delivered project from the system. When such a drawing is found, modifications are done simply by using 2D modeling tools. Some components may be added as blocks from other drawings and lines are drawn to match the current project. As can be imagined, the method is really vulnerable and often some simple drawing mistakes appear in the drawings. Creating main dimension drawing in this method can be laborious and take time up to few hours.

With the aid of configurator, the main dimension drawing is configured automatically at the same time with the layout model. Created main dimension drawing is related to the layout model and all the changes made in model will appear to the drawing. Because drawing of configuration model has to contain some of the dimensions twice, manual work is needed to delete the excessive ones. Some dimensions and annotations have to be repositioned as well. For user, who has basic skills of using NX, this should not take more than fifteen minutes.

Improvements enabled by 3D layout and main dimension drawing configurator are presented in following table 4. All in all, the configurator offers many kinds of advantages compared to old method of composing layout model in I-Deas and creating the drawing with 2D drafting methods. The creation of configurator needs resources and also advanced skills in using design tools. On the other hand, if configurator can cover the most of the variants in certain product family, it will pay itself back after few times of usage. The using is easy and manageable with basic NX skills.

Table 4. Improvements enabled by created configurators compared to old method.

Category	Configurator	Composing model in I-Deas
Time consumption	Creation of configurator ≈ 2 weeks (80h) Configuring layout model ≈ 15 mins Modifying drawing ≈ 15 mins	Composing layout model ≈ 2 days (16h) Creating main dimension drawing ≈ 30 mins -2h
Cost	Creation of configurator ≈ 80h*25€=2000€ Creating layout model ≈ 0,25h*25€=6,25€	Composing layout model ≈ 16h*25€=400€
Data exchange (converting into STEP or IGES format)	Works everytime	Does not work everytime. Requires modifications to the model, which may take several hours of extra work.
Precision of layout model	All the important outer dimensions and mounting dimensions are precise. Most of the parts are presented exactly as they appear in reality.	Contains variation in outer dimensions and possibly in mounting dimensions. Many of the models do not present reality.
Creating layout model	User instruction can guide the first time user through the process. Requires a little knowledge of NX to inspect the model. If layout model requires modifying, the user should have basic skills in using NX.	Requires most probably advanced skills in using I-Deas. Some of the geometry, for instance oil pipes, needs really deep knowledge of using I-Deas. Composing layout model of old models, which need modifications, is not an easy task and I-Deas is not easy to use for this purpose.
Creating main dimension drawing	Drawing is created automatically as the layout model is configured. Needs some manual work for deleting and repositioning dimensions and annotations.	Drawing is not created based on 3D model. Old drawing of reference project is modified with 2D design tools to match the current project.

The results that can be achieved with these configurators are prominent. The process becomes much faster, it improves the quality of both, layout models and main dimension drawings, and it makes the process easier to be performed. There are two main problems related to configurators. The first one is how to create them effectively so that they fulfill the requirements as well as possible. The first one is how to create them effectively so that they fulfill the requirements as well as possible. The creation should not take too much effort compared to constructing several layout models with old methods. The second problem is how to maintain configurators effectively so that they fulfill the requirements also in the future. This is discussed more in following chapter 9.

9 Recommendations

There are many methods to get more advantage of using configurators. It is possible to add new features into them and with this approach produce more information in configuring process. The process itself can be developed more fluent and one option is that configuring parameters are transferred straight from the sales configurator for layout model creation. There are also different manners to utilize created layout models and configurators in general. In this thesis the tools for configurator creation were examined. These tools can be extremely beneficially for many other purposes as well than just for layout model configurators.

One of the secondary objectives of this thesis was to research, how a layout model configurator could be constructed for other product families as well. This is discussed in first subchapter. Created configurators should be developed in the future to ensure that they are able to provide up to date models. This is actually really important for the usage of configurators. User should have at least basic knowledge of using NX and O&V tool in Teamcenter to update the configurator. The knowledge is partly provided in this thesis as well and with little practice, it should be possible to revise the configurators. The second subchapter explains how the configurators should be revised in the future. Third subchapter is reserved for listing future visions of developing configurator. Some ideas and options are provided to enhance the usage of configurators.

9.1 Creating configurators for other product families

It is obvious that configurators can improve the process by making the layout model and main dimension drawing creation easier. On the other hand, the development of configurator needs resources and advanced skills in using design tools. The development of configurator may be too laborious compared to work load, which is needed to create layout models and main dimension drawings in old manner by modifying models and drawings of reference projects. If the sales volume is low, it is possible that the effort used to develop the configurator is not repaid.

The most important factors of making decisions about creating configurator were presented in chapter 5.1, as the most suitable product types for this project were chosen.

The first factor is the variability of product. How modularized and standardized the structure is or on the contrary how tailor made the structure is. If the configurator cannot be constructed with reasonable work amount, so that it covers most of the product variants that are delivered, it may not be wise to use resources for constructing it. Another factor is sales volume. If only few variants are sold annually, it may not be reasonable to create configurator, which is able to configure hundreds of models for different product variants. Creating needed models by modifying existing models uses much less resources than creation of configurator in this case.

The variability of synchronous motors usually increases and the sales volume decreases as the size of motor type grows. From this point of view, it may not be advantageous to create separate layout model configurator for bigger motor. However, if it is possible to provide layout model for smaller motors, it should be possible also for the bigger ones. Anyway, customers need to perform similar layout tests for the bigger motors as well. Different approach for creating layout model for bigger motors could be considered.

In case of AMZ 0710 and 0900 VSD motors, it was possible to create configurators, which provide extremely precise layout models of motors for most of the product variants. This was mainly possible because of the standardized structure of these motors. In the beginning of configurator design project it is not necessary to try to create extremely precise configurator. This means that the configurator does not have to be able to create all the product variants straight in the beginning. Also, all the components do not need to be presented with precise geometry, if the geometry is not known. The configurator can be developed more precise during usage.

In bigger motors there are many components, which are designed during the process according to customer specifications. In layout model configuring phase, these components could be presented with more simplified models. As the design process proceeds, the components in layout model could be replaced with more accurate models, to present the geometry as it appears in finished product. The spatial tests as well as the tests, which ensure that the motor can be mounted correctly, could be performed with less precise layout model. The configured layout model can be referred to design layout skeleton, which is used in top-down assembly design process (presented in chapter 3.1.).

The layout model shows only outer dimensions and important mounting dimensions. The final design can be constructed within these limitations. The objective of this kind of configurator is not to produce finished model, but the base frame for the design work.

For every case it should be considered separately, what kind of layout model configurator is it wise to construct. The first version of layout model configurator does not have to be completely accurate. The idea is that skillful users can develop the configurator as they are using it. If the designer notices that certain component or more accurate model should be added to the configurator, he is able to add it himself, and next time it can be configured straight away. In this manner the configurators are developed over the time and all the users are aware of using and modifying configurators. If it is too difficult to train every designer to use and develop configurators, it is possible to name certain key user, who is responsible for all the development actions.

9.2 Updating existing configurators

As was discussed earlier, in every case the created configurators need to be developed further. Structures of motors and their components change all the time. These changes should be updated to the configurators as well. To ensure that configurators are able to produce up to date models and drawings it is necessary to continuously update them.

Most of the updating actions can be categorized into three different categories:

- Revising and modifying existing components
- Adding new components and creating suitable option constraints & variant conditions
- Modifying the configured main dimension drawing

Changes may occur to the components, which already exist in the structure of configurator. In this case, the component has to be revised and changes made to the component model. The new revisions of components will be automatically updated into the structure of configurator, even though the configurator (top level item) would be in approved state. In figure 56 an example structure is presented. In the first picture on the right, all of the items in the structure are in approved state. The second lower level item, “TEST_PART_1”, is revised so that it can be modified. As can be seen in the second picture in middle, the revised item (B-revision) is updated into the structure even though the top level item (TEST_ASM_2) is still in approved state. Actually the new revision of TEST_PART_1 item will be updated to all the structures, in which it is included.

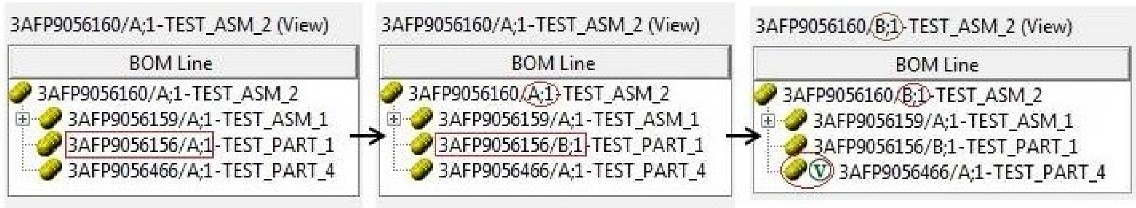


Figure 56. Revising configurator and its parts.

New components cannot be added into the configurator before the main item is revised (not in approved state). Also, new option constraints or variant conditions cannot be created. In figure 56 the picture on the right presents situation, in which new revision of TEST_ASM_2 is created. The structure is no more in approved state and new components, option constraints and variant conditions can be added. In figure 56 a new variant condition is added for TEST_PART_4. V-sign in front of the item number informs about this action. As new components are added into the structure, new variant conditions should be written for them, unless the item is supposed to be included in the structure every time a new configuration is produced. Also, option constraints can be written for added components. Creation of option constraints and variant conditions happens in similar manner, as was presented in chapters 6.3.2 and 6.3.3.

Third category of updating actions is targeted to the main dimension drawing. To update changes into the drawing the main item has to be revised. Geometric changes in component models can be updated to the drawing in this method. Also, if some other changes have to be implemented into configured drawings, the main item has to be revised before modifications can be performed.

During the process some problems in updating created configurators were faced. Many of these problems were related to the options and option constraints created by O&V tool. The modification of created options was extremely difficult or sometimes even impossible. As configurator is already used and new variants have been created, Teamcenter does not allow changes in options or option constraints. For example, if the designer tries to change the type or default value of certain option, the error message presented in figure 57 appears to the screen. This message appears also, if new options are added into a generic structure or child options presented from lower level items.

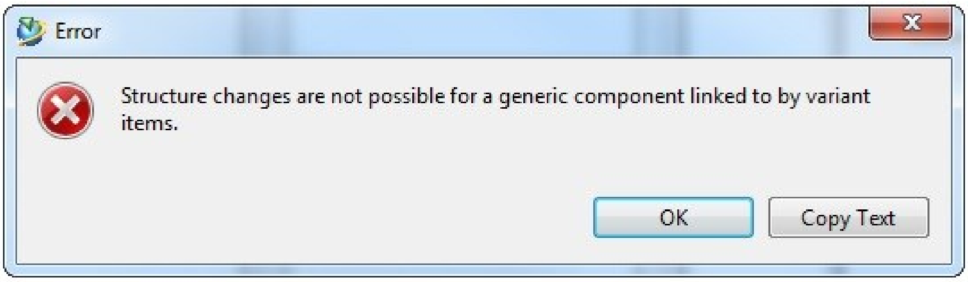


Figure 57. Error message informing about linked variants.

This error can be avoided by unlinking the created variant items, closing the generic structure in Structure Manager and reopening it. Unlinking happens by selecting the variant and clicking: Tools / Variants / Unlink Variant Item. It may be difficult to find the variant, which is linked to the generic structure, because TC does not show it with any external sign. All of the created variants can be seen in Data Panel of generic structure. Select the References tab and double click the item of generic structure, as is presented in following figure 58. After this created variants will be shown.

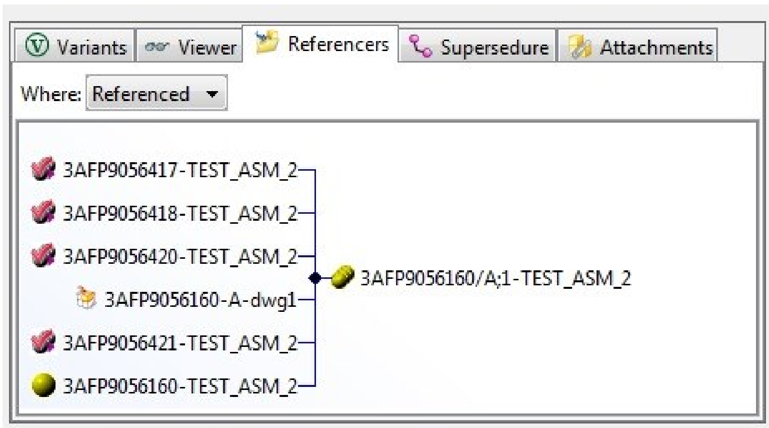


Figure 58. Created variants on References tab.

As all the created variants are unlinked, Teamcenter allows the user to create new options, present them from child modules or change default values of options, but still it is not possible to change the names of already created options or delete them. The error presented in following figure 59 shows up, if user tries to perform these disallowed actions.

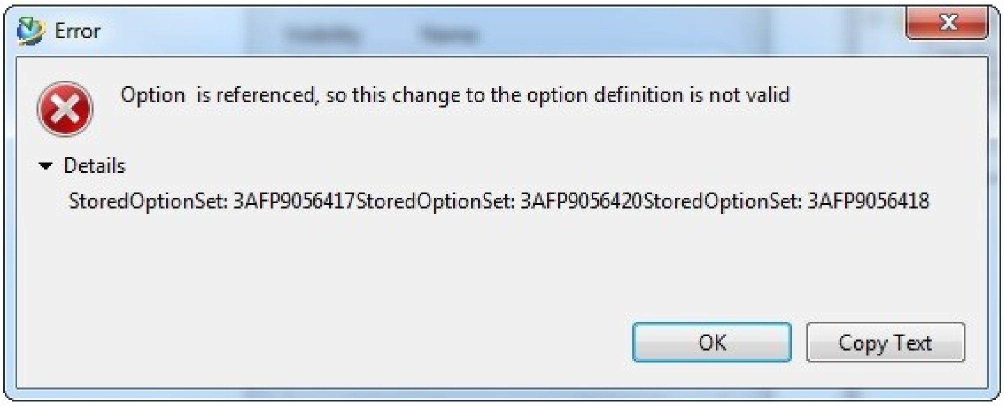


Figure 59. Error message informing about Stored Option Set (SOS)

During this research it was not discovered, if it is possible to unlink the connection to Stored Option Sets, so that options could be fully modified after creating variants. However, the problem with SOS is not that significant. Names of options just have to be defined in correct manner, before creating option constraints and especially before configuring variants. As was presented in chapter 6.3.2, the names of options should begin with certain number or prefix so they can be organized in correct order. All the options, native and presented ones, are listed in configuring window. If they are not named correctly, it is difficult to know, which the essential ones for configuring are.

It is possible to create new options and they can be named as wanted. The fact that created options cannot be deleted anymore, may turn the list of options a little confusing. Especially, if a lot of changes happen to the structure of configurator. It is possible to delete the option constraints related to certain option and turn the option meaningless in this manner. Still the option will be shown in the list of native options. Also, certain value has to be chosen for it, as new variant is configured, even though the option does not affect to the structure at all. These are examples of downsides of Options & Variants tool.

9.3 Future visions

The first step in implementing the layout model configurators to usage of mechanical design department is to fully switch to use NX as main CAD software. NX is powerful tool for 3D design and offers many other possibilities as well than just usage of parametric or configurable models. Regular 3D modeling is much faster and more proficient with NX than with I-Deas. As designers are trained to use NX, they are able to effectively utilize the layout model configurators. Effective utilization means that designers are able to develop and update the configurators, as they are using them. Another option is that users report about changes, which should be updated to the configurators, and key user does the updating work.

Similar configurators could be developed for other products of synchronous machines, if the layout models will be offered for all the products. Before the products have been designed by using NX for certain time, the most efficient method to create layout models is to create the configurator first. Otherwise, the models should be modeled separately in each layout model creation process. As was mentioned in chapter 9.1 the creation of configurator should be considered separately for each product type, but if layout models are needed for all the products, the configurator is the most efficient method in matter of time.

In future, new features can be added into configurators. Few ideas for such features were gathered during this project. Usually in main dimension drawing the total mass of motor is presented as well as the center of gravity. The total mass can be calculated precisely, as the project has been designed, but the best possible approximation could be presented in drawing already after creating the configuration. Also, the center of gravity could be presented automatically in main dimension drawing by utilizing this data.

As Options & Variants tool is supposed to be used for product configuring purposes, the second step in layout model configuring could be to combine the product configuring and the layout model creation. At the moment, separate product configurator is used to create the base structure of designed motor project in Teamcenter. The structure contains most of the parts for motor, but it does not create 3D assembly of them. If product configurator would be based on 3D assembly in similar manner, as the layout model configurator, the assembly model of whole motor could be created while configuring the product structure. In this case the created assembly would contain precise models, which are used for manufacturing as well. If the exact geometry is not supposed to be sent to the customer, it is easy to remove certain parts from the STEP file. The assembly is also extremely useful for mechanical design work, because it makes it easier to perceive the whole structure and provides layout for detail design.

There are a lot of issues to be clarified before the product configuring could be performed by using combination of NX and Teamcenter. Anyway, it is possible and automatic creation of motor assembly would be beneficial for the whole process. If there is not enough resources for researching this possibility or for some other reason the Options & Variants tool is discarded for product configuring, there is another method to develop layout model creation. The layout model could be still used as separate model among manufacturing models of motor, but it could be configured by information transferred from sales configuring. As sales configuring is performed, the information could be straight exported into layout model configurator. In this manner the designer would not need to input the parameters and the model would be configured automatically by using information from sales configurator.

The tools and methods examined during this study can be utilized in many different methods to improve the mechanical design work of synchronous machines. Layout model is useful for the customer as well as for the mechanical design department. It is strongly recommendable that 3D design, layout model creation and product configuring are developed continuously in the future to improve the mechanical design of synchronous machines.

10 Conclusions

A need of providing 3D layout model of motor in the beginning of design process had occurred in the mechanical design department of synchronous motors at ABB. Customers wanted to have the layout model for spatial experimentations, before the mechanical design was fully performed. Usually only a main dimension drawing of motor was offered to the customer for this kind of purposes, but it does not serve as well as 3D model in experiments performed by customer. Another problem is that because the drawing is not based on 3D geometry, it is more vulnerable for design mistakes. A suitable method had to be found to effectively produce 3D layout models and main dimension drawings with available design tools.

The main objective of this thesis was to research how 3D layout models could be created for synchronous motors. It was decided that a layout model configurator would be constructed for motor types, which can benefit the most of created configurator. The main dimension drawing was also required to be created with the aid of configurator.

At first, the existing 3D modeling techniques were studied thoroughly. Different methods are used for different purposes, but nowadays solid 3D geometry is created mainly with two methods: sweep representation and constructive solid geometry. All the modern CAD software use also feature based modeling. Feature based modeling means that high-level entities, which contain more information than simple geometric primitives, are used to construct the model. These entities are called features. Also NX uses feature based modeling.

Other important terms learned, while studying 3D modeling, were parametric and variational modeling. Parametric modeling and variational modeling are often used interchangeably. As modeling is performed in modern CAD software, relations are defined between details in the model. The CAD system is able to save these relations. This property enables easier editing of created models. Modeling kernels use parametric and variational methods to solve the geometric constraints in models. For the end user, it does not matter, which kind of technique the modeling kernel is using. What matters is that the user is able to change the relation values, which are also known as parameters, so that the model retains its shape and does not fall apart. Creation of configurable models is based on this feature of CAD software.

Also different methods of creating 3D structures were examined. There are usually two different approaches for assembly creation in product design, bottom-up and top-down methods. The bottom-up method means that components of an assembly are designed first in detailed level and the final assembly is then constructed of these components. The top-down method is contrary to this. It starts by sketching the whole assembly construction and then dividing it into smaller sketches, which are the components of assembly. The detailed design starts, as sketches of components are separated from the whole assembly. It was understood that the layout model configurator is constructed with bottom-up method, as components, which are designed in earlier projects, are used in the assembly. However, the configured layout model could be used as a shape skeleton or layout skeleton of top-down design process. For instance, if main terminal box has to be redesigned, the box in the layout model works as a layout skeleton, which shows the outer dimensions and the mounting surface. These can be utilized while designing new box in detailed level.

Different types of products from configuring point of view were studied as well. ABB's synchronous motors are often categorized into tailor made products group, but they have also configurable product features. Configurable product is a product, which is consisted according to customer's requirements. The customer is offered certain options to choose from and the product is consisted according to these decisions. In tailor made products, customer can have also arbitrary decisions, which are implemented if possible. Configuration model describes the configuration of the product. Different kinds of configuration models can be used in different phases of process and the layout model configuration is one example of these. The layout model is extremely demonstrative method, because the configuration can be seen in three dimensional view. Configurator is a tool for creating different kinds of configurations, which are also known as product variants. It is important to understand that only configurable product features of synchronous motors can be included into the configurator.

With the aid of skeleton modeling, it is possible to make the assembly models more robust, especially in case of parametric models. Skeleton modeling means that simple base geometry, which can consist of sketches, planes, curves etc., is used as a frame of the whole assembly. Instead of constraining parts to each other, the parts are constrained to the skeleton frame. This enables that any of the parts can be changed to another part without losing the created constraints. Also in parametric models, as the relation values are changed, it may happen that the software cannot define created assembly constraints anymore. With skeleton modeling this can be avoided and the functioning of model secured in every case.

One of the most important objectives of this thesis was to discover, how the layout model configurator can be created with the existing design tools. Mechanical design department of synchronous machines is using I-Deas NX as its main CAD software. I-Deas is already relatively old CAD software and because of that, the design department is going to migrate using newer CAD software from Siemens, NX. The configurator was designed by using NX, but the models created in projects designed by using I-Deas, were also utilized.

In design work, PDM system is important tool, because all the design data is stored in it and it enables systematic revision control for created items. Siemens Teamcenter is used as PDM software in synchronous machines. Teamcenter offers also tools for product configuring and these were utilized in creation of layout model configurator. The modeling tools of NX, as well as the possibilities of utilizing Teamcenter, were examined thoroughly, before the configurator was created. Data exchange methods between different CAD and PDM systems were researched, so that effective method of utilizing I-Deas models in NX could be found. Data exchange methods had to be known also to deliver the created layout model in correct form to the customer.

The creation of configurator was reasoned by presenting the advantages of model configurators. The biggest advantage is that model configurator is the most effective method of producing 3D models, but it is possible to construct effectively only for products, which have as much features of configurable products as possible. In case of synchronous motors this means that the structure is modular and standard components are used. This was taken into consideration, as it was decided, for which product type the layout model configurator will be constructed in this project. Another important aspect was the sales volume, because the most delivered machines can also benefit the most of created configurator. It was decided that the configurator will be created for AMZ 0710 and AMZ 0900 VSD motors, which have relatively standardized structure and are the most delivered motors of this product family.

The requirements for configurator were listed. The main requirements were that the layout model should present geometry as it appears in main dimension drawings, but it should not reveal too detailed data of the structure. The model should be possible to export into suitable data exchange format. Usage of configurator should be easy and designer should not need to use a lot of effort to create the model. The main dimension drawing should be created automatically as the layout model is configured.

The structure of VSD motors was examined to understand properly how the configurator can be constructed for selected motor types. Most of the parts were either variant components or optional components. Variant component means that there are different variants to choose from, but in any case, one of these variant components has to be chosen into the structure. Optional components can be either included in or excluded from the structure. For these components it was possible to use earlier created I-Deas models, which were imported into NX in STEP file format. Few of the motor components needed parametric features. For instance, the frame, cooling systems, oil pipes, foundation plates and shaft end were this kind of components and they could not be imported from I-Deas. These models were created in NX by utilizing parametric modeling. Also parametric skeleton models were created.

The assembly was constructed first for AMZ 0710. As the assembly was finished, it was possible to decide the suitable user interface for configurator. Few available options were examined and finally it was decided that the user interface will be created by utilizing Options & Variants tool in Teamcenter. The tool is mostly supposed for product configuring, but it worked for layout model configuring as well. The suitable option constraints and variant conditions were created, so that the user inputs control model creation in correct manner. After fixing the user interface it was possible to create the main dimension drawing. Configurator was tested by producing layout models of earlier delivered projects. Same process was later repeated for AMZ 0900 motor.

As a concrete result of this thesis, two separate 3D layout model and main dimension drawing configurators were created for AMZ 0710 and AMZ 0900 VSD motors. Configurators are able to create layout models for most of the product variants and fulfill the listed requirements really well. The user interface is easy to use and a lot of effort is not needed for creating layout models and main dimension drawings. Created models and drawings can be also revised relatively effortless.

In the future, it is important that the created configurators will be updated and developed further. By updating it is ensured that the configurators are able to produce up to date layout models, as changes in mechanical design occur. The tools researched in this thesis, especially NX and Options & Variants tool, offer plenty of opportunities to improve the design process of synchronous machines.

First step is to find a reasonable method to implement NX as main CAD software. Implementation of NX will definitely improve motivation of designers to explore the possibilities of CAD. Also the recommendations presented in this thesis can be implemented as NX is in use. It should be considered, if similar layout model configurators will be created for other products of synchronous machines as well. It could be possible to combine the layout model creation with the sales configurator or with the product configurator. All in all, 3D model configurators are an efficient method to create 3D models, if the product can be at least mostly assimilated to configurable product and available tools support creation of configurator rationally.

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