FACULTY OF TECHNOLOGY

CONNECTING ABAQUS AND CATIA V5 IN PRODUCT DEVELOPMENT

Ari-Pekka Jokinen

DEGREE PROGRAMME OF MECHANICAL ENGINEERING

Master's thesis

September 2022

ABSTRACT

Connecting Abaqus and CATIA V5 in Product Development

Ari-Pekka Jokinen

University of Oulu, Degree Programme of Mechanical Engineering

Master's thesis, 2022, 65 pp. + 2 Appendixes

Supervisor at the university: Professor Emil Kurvinen

In this work Abaqus software, which is used for doing finite element analysis, and the associative interface of CATIA V5 computer-aided design software and Abaqus were researched. From the associative interface, the usage with Abaqus and the usage's requirements were researched. From Abaqus it's usage and requirements were studied. The software was also explored by a case study. In the case study there was no need to utilize the associativity of the associative interface. The methods of research in this work were computer-aided design, finite element analysis and the connection of those through associative interface.

In this work it was clarified how the associative interface works with Abaqus by the features, and how Abaqus works by its modules. This work included a case study, in which squeezing of a cavity seal between two plastic shells was simulated. In the case study nonlinear material model for the cavity seal was used. The case study was challenging due high nonlinearity and included especially calculation time challenges. In the case study a situation occurred where the simulation did not pass every step. Because of that a decision was made to make a second finite element analysis model, where the cavity seal model was replaced by gasket model that was simpler in terms of the calculations. The old model was named as model A and the new model as model B. In case study's results models A and B were compared, and represented shell's strains and stress of screw towers were calculated by model B.

In the work's discussion the benefits of associative interface with Abaqus were discussed together with an export problem which appeared during the work, a need of expert for making simulations and the change of gasket model in the case study. As conclusions it was stated of the associative interface that its very useful with Abaqus, and it does not need special skills or experience for the use. However, as a conclusion it was stated that training is needed for use of Abaqus.

Keywords: CAD, FEA, computer-aided design, finite element analysis, associative interface

TIIVISTELMÄ

Abaquksen ja CATIA V5:n yhdistäminen tuotekehityksessä

Ari-Pekka Jokinen

Oulun yliopisto, Konetekniikan tutkinto-ohjelma

Diplomityö 2022, 65 s. + 2 liitettä.

Työn ohjaaja yliopistolla: professori Emil Kurvinen

Tässä työssä tutkittiin Dassault Systémesin Abaqus-ohjelmaa, jolla tehdään elementtianalyyseja sekä CATIA V5 -tietokoneavusteisen suunnitteluohjelman ja Abaquksen välistä assosiatiivista käyttöliittymää. Assosiatiivisesta käyttöliittymästä tutkittiin yhteiskäyttöä Abaquksen kanssa, ja yhteiskäytön vaatimuksia. Abaquksesta tutkittiin käyttöä ja käytön vaatimuksia. Ohjelmistoon tutustuttiin myös esimerkkitapauksen avulla. Tässä esimerkkitapauksessa ei tarvinnut hyödyntää assosiatiivisen käyttöliittymän assosiatiivisuutta. Käytettävät tutkimusmenetelmät työssä olivat tietokoneavusteinen suunnittelu, elementtianalyysi, ja tietokoneavusteisen suunnittelun ja elementtianalyysin yhdistäminen.

Työssä selvitettiin, miten assosiatiivinen käyttöliittymä toimii Abaquksen kanssa ominaisuuksittain ja miten Abaqus toimii moduuleittain. Lisäksi työ sisälsi esimerkkitapauksen, jossa simuloitiin ontelotiivisteen puristusta kahden muovikuoren väliin. Esimerkkitapauksessa käytettiin epälineaarista materiaalimallia ontelotiivisteelle. Tämä esimerkkitapaus oli vaativa ja sisälsi erityisesti laskenta-aikahaasteita. Esimerkkitapauksessa päädyttiin tilanteeseen, jossa simulaatio ei mennyt kaikkein vaiheiden läpi. Tämän takia päädyttiin tekemään toinenkin elementtianalyysimalli, jossa ontelotiivistemalli oli korvattu laskennan kannalta yksinkertaisemmalla tiivistemallilla. Vanha malli nimettiin malli A:ksi, ja uusi malli nimettiin malli B:ksi. Esimerkkitapauksen tuloksissa vertailtiin malleja A ja B, ja otettiin esiin mallilla B laskettuja kuorten venymiä ja ruuvitornien jännitystiloja.

Työn käsittelyosuudessa käsiteltiin assosiatiivisen käyttöliittymän hyötyjä Abaquksen kanssa, työtä tehdessä ilmenneeseen tiedonvienti ongelmaan, asiantuntijan tarvetta esimerkkitapauksen simulaatiota tehdessä ja tiivistemallin vaihtoa työssä. Johtopäätöksinä assosiatiivisesta käyttöliittymästä todettiin, että se on Abaquksen kanssa hyvin käyttökelpoinen, eikä tarvitse erikoisosaamista tai -kokemusta käyttöä varten. Johtopäätöksenä Abaquksesta todettiin, että se tarvitsee käyttöä varten kouluttautumista.

Asiasanat: tietokoneavusteinen suunnittelu, elementtimenetelmä, elementtianalyysi, assosiatiivinen käyttöliittymä

PREFACE

Ths master's thesis about connecting CATIA V5 and Abaqus was done for Bittium company from March to September in 2022. I want to thank Bittium for the opportunity to do the master's thesis for the company.

For a big support for doing the work I want to thank Bittium's Jukka Väisänen, RAND Finland's Kimmo Jämsä and University of Oulu's professor Emil Kurvinen. Their help was crucial through out the whole work. Finally, I want thank my father Hannu for good conversations and other support for doing the work.

Oulu, 6.10.2022

Ari-Pekka Jokinen

Ari-Pekka Jokinen

TABLE OF CONTENTS

ABSTRACT
TIIVISTELMÄ
PREFACE
TABLE OF CONTENTS
SYMBOLS AND ABBREVIATIONS
1 INTRODUCTION 7
1.1 Research Questions 8
1.2 Motivation of Work 8
1.3 Contents and Defining of Work 9
2 METHODS 10
2.1 Computer-Aided Design (CAD) 10
2.2 Finite Element Analysis (FEA) 10
2.2.1 FEA Process 11
2.2.2 Element Types 13
2.2.3 A 3D FEA Example 13
2.3 CAD and FEA combined 15
2.3.1 Combination of CAD and FEA in Product Development 16
3 INTRODUCTION AND USAGE OF CATIA V5 - ABAQUS ASSOCIATIVE INTERFACE 18
3.1 Preparation of FEA-Simulation in CATIA V5 18
3.2 Export Methods from Associative Interface 18
3.2.1 Automatic Export with Associative Interface 18
3.2.2 Manual Export with Associative Interface 20
3.3 Exporting CATIA Publications as Sets in Abaqus 21
3.4 Activating/Deactivating CATIA V5 Components and Elements 23
3.5 CAD Parameters in Associative Interface and Abaqus Software 24
3.6 Units 27
4 INTRODUCTION AND USAGE OF ABAQUS 28
4.1 Importing Models from CATIA V5 28
4.2 Sets and Surfaces 29
4.3 Part Module 29
4.4 Property Module 32
4.5 Assembly Module 32

4.6 Step Module 33
4.7 Interaction Module 35
4.8 Load Module 36
4.9 Mesh Module...37
4.10 Job Module 37
4.10.1 Creating a Job 38
4.10.2 Datacheck 38
4.10.3 Submitting and Monitoring a Job 38
4.11 Using Command Line Window..39
4.12 Visualization Module 40
4.13 Compressing MDB..40
5 CASE STUDY: Squeezed cavity seal..41
5.1 Preparing Simulation in CATIA V5 42
5.2 Symmetry in FEA Model 42
5.3 Assigning Material Property 43
5.3.1 Material Property for Model B -Gasket...43
5.4 Assembly and Step Module...45
5.5 Constraints..46
5.6 Creating Screw Modeling 46
5.7 Assigning Interactions...47
5.8 Assigning Loads and Boundary Conditions...47
5.9 Creating Meshes 48
5.9.1 Model A's Gasket 48
5.9.2 Model B's Gasket 49
5.10 Running the FEA-Simulation..50
5.11 Reviewing Results in Abaqus 51
5.11.1 Comparison of Models A and B in Fourth Step 51
5.11.2 Stresses in Screw Towers 54
5.11.3 Deformations of Plastic Shells...55
6 DISCUSSION 58
7 CONCLUSIONS AND RECOMMENDATIONS 60
8 SUMMARY 62
REFERENCES..63
APPENDIXES

SYMBOLS AND ABBREVIATIONS

CAD	computer-aided design
CAE	computer-aided engineering
FEA	finite element analysis
FEM	finite element method
R&D	research and development
V	version
VPN	virtual private network
vs.	versus

1 INTRODUCTION

In product development the first step is research and development (R&D) (Kenton 2022). R&D in corporations means innovative activities that are taken for developing new services and products and improving existing services and products (Holstein 2022). The R&D is usually taken care of by technical specialists who can be for example engineers (Liam 2021). R&D is used in companies because it allows them to get a competitive edge in the market (Liam 2021).

Nowadays different kind of simulations are widely used in product development and R&D. When in classic product development it was common way to do simulations in different points of product development, in nowadays' product development and R&D simulations are done in parallel when developing a product. The simulations integrated to R&D produce shorter time of design iteration. Some other benefits of the integration are benefits in avoiding early prototyping, cost reduction by reduced tests and material needs, and reduced time to market (Imaginationeering 2021). In this work finite element analysis (FEA) is integrated to computer-aided design (CAD).

This master's thesis is about using Dassault Systémes' CATIA V5 as a CAD software and Dassault Systémes' Abaqus FEA software for simulations in product development. This work is done for Bittium Corporation, which is a company providing solutions for connectivity and communication, medical products and security solutions for mobile devices and laptops (Bittium 2022). In Figure 1 there is a result image of loaded 3D beam in Abaqus/CAE.

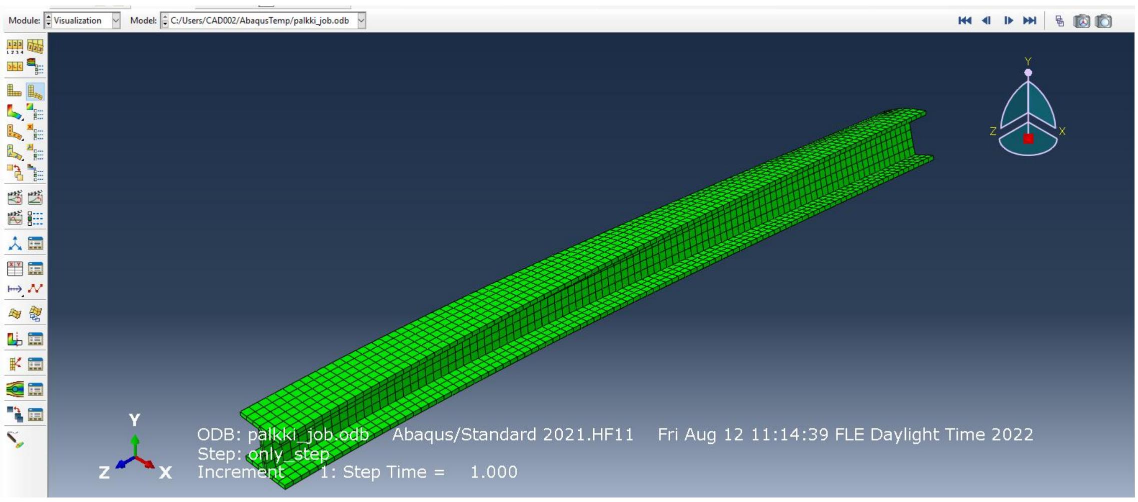

Figure 1: An example of loaded 3D beam in Abaqus/CAE software application.

1.1 Research Questions

The research questions in this master's thesis are:

- How to use CATIA V5 – Abaqus associative interface and Abaqus software in product development?
- How usable CATIA V5 - Abaqus associative interface and Abaqus software are?
- What kind of requirements the use of the associative interface and Abaqus software sets for the user?
- What kind of requirements the use of the associative interface and Abaqus sets otherwise?

1.2 Motivation of Work

In today's general trends of product development, simulations are part of R&D and an integration of simulation to R&D process reduces the iteration time of product design. In many companies FE simulation is bought as an external service if the company does not have the capacity for FE simulations. Rarely used capacity is also expensive. For external service CAD and other material are sent to a service provider. The CAD files must be usually converted to, for example STEP files, which are not as precise as original CAD files. Because of that, the models need to be checked, estimated, and possibly corrected in detail. Those things make the use of external usage of FEA quite time consuming, and small simulations not profitable to do.

FEA consulting is done in companies which provide simulation services as their business. Engineering offices do not always need FEA in engineering, and talented personnel are needed for the use of software. Those are examples of the reasons why engineering offices avoid purchasing FEA software.

FEA is a suitable and relatively fast way to analyse complicated models. For example, analytical formulas are used with Microsoft Excel calculations something that design engineers can use without consultation, but analytical formulations have their limitations in analysis of complex structures.

Bittium Corporation has CATIA V5 as their main 3D CAD software. CATIA V5 and Abaqus come from the same corporation, Dassault Systèmes. Also, the associative interface with CATIA V5 and Abaqus is delivered by Dassault Systèmes.

Because of mentioned reasons, Bittium Corporation wants to research if Abaqus is suitable for their usage with CATIA V5. This would connect CAD designing and FE simulation and therefore improve their product design with shorter time of engineering.

1.3 Contents and Defining of Work

The use of CAD and FEA commercial software programs together are researched in this work. The work's case is Bittium company which has CATIA V5 as a CAD software. The company organized also Abaqus and the associative interface to use for this work. The software vendor's representative gave valuable support in this work.

2 METHODS

This work’s research methods are CAD and FEA. This chapter introduces the use of CAD and FEA.

2.1 Computer-Aided Design (CAD)

Computer-aided design (CAD) means designing and drawing with the help of computer (Krebs 2007, p. 8). With CAD it is possible to create different kind of 2D and 3D models and drawings. Some other features of CAD are:

- detail design
- programming of computer numerical control (CNC) machines
- files for 3D-printing
- automatic produce of material and component lists.

CAD is used for many kinds of drawing and designing, like for example engineering drawings and mechanical designing. When compared to pen and paper CAD is remarkably easier and faster, having an option for 3D modelling, and CAD data is faster and easier to transfer over long distances. Also, the design of different versions is an easy task with CAD.

In this work CAD is used for creating 3D parts and assemblies. Doing a 3D CAD part starts by doing a 2D sketch as a 2D drawing. 2D sketch is done by using drawing elements like points, lines and circles. When the sketch is ready, it is possible to do a 3D design by extruding. Also, doing more sketches for 3D features and many 3D features for a 3D model is possible. Some examples of 3D features are holes, fillets and threads. Part can be added to an assembly and a part drawing can be done in any time during the product development.

2.2 Finite Element Analysis (FEA)

Finite element analysis (FEA) is an analysis process and finite element method (FEM) is a numerical calculation technique used in FEA. With FEM it is possible to calculate strains, displacements, and stresses in complex structures. The models in FEM are divided to small elements, and results obtained with equations which are solved by calculations based on matrix calculus. FEA's results are approximations of situations. With FEA it is also possible to calculate for example, different kinds of vibration, acoustic, fluid dynamic and heat conduction problems.

FEA can be done for static and dynamic cases. This work's FEA is done with Abaqus. This section introduces the FEA process, meshing elements of FEM and a 3D example of using FEA. Due the complexity of geometry, example is impossible to solve with exact equations.

2.2.1 FEA Process

The FEA process can be divided to eight steps. The steps from two to six can be called pre-processing, and they can be done in any order. The step eight can be called post-processing. The steps of finite element analysis are following (Hietikko 2021, p. 171-172):

1. Idealization of the structure for analysis
2. Applying material properties and cross section information
3. Creating structure's mesh
4. Creating boundary conditions of the structure
5. Creating loads of the structure
6. Creating interactions of the structure
7. Solving
8. Interpretation and review of the results

Idealization of the structure is done by deleting details that do not affect the calculation but are using some additional computing capacity. Those details can be also symmetries that can be deleted and replaced by symmetry boundary conditions in FEA model.

Material properties in FEA can be linear or nonlinear and materials can be isotropic or anisotropic (Gallagher 1975, p. 99 -100). Cross section information is needed in cases, which can be modelled with 1D elements. One such case is a beam that has homogenous cross section.

Structure's mesh is done by choosing element types and their sizes. Element types and sizes can vary within the structure. For example, it is possible to do a smaller close mesh in a spot where you want a more detailed and precise analysis (Hietikko 2021, chapter 15). Element types are represented in detail in chapter 2.2.2.

Boundary conditions in FE simulation are in Abaqus for example symmetry and forced displacement properties for idealized models and rigid fixation. Loads in FE simulation are in Abaqus for example concentrated forces and pressures. Without boundary conditions and loads getting a unique solution from simulation would be impossible (Madier 2020, p. 215). In simulation process boundary conditions and loads are doing three things:

- Fixing the value of displacements in regions of simulation model
- Applying representative loads in regions of simulation model
- Replacing not modelled part in simulation model for simplicity (Madier 2020, p. 216).

Interactions define how the models connect to each other. Interactions in the model are mainly surface contacts. Contacts are needed for researching the behaviour and load transfer of parts in an assembly. Contacts are not assumed by default in FEA software. That is because they take a lot of computation power and time. Some contact types in FEA are for example point-to-point contacts, general contacts, and glued contacts (Madier 2020, p. 307-315). Some other interactions than contacts are model change in simulation and pressure penetration.

In the solving phase the numerical solution is computed. Numerical solution is calculated by linear or nonlinear equations. Values for equations are obtained by defining the FEM model in pre-processing (Madier 2020, p. 17, 23).

Results in FEA contains displacements and additional results which are defined in pre-processing. Additional results can be for example strains, forces, and stresses. In post-processing there is also data available from solving the simulation's equations. That data can be for example the log files (Madier 2020, p. 22).

2.2.2 Element Types

Finite element method contains 1D, 2D, axisymmetric and 3D element shapes. 1D elements are always line segments. 1D elements are used in situations where the component pieces are possible to describe as line segment and forces and boundary conditions are in elements' longitudinal direction.

2D elements are basically triangles and quadrilaterals. 2D elements are used in situations, where pieces, boundary conditions and forces can be described in one plane. There can be also isoparametric 2D elements, which can have curved boundaries.

Axisymmetric elements are type of elements that are symmetric around an axis. Axisymmetric elements are 3D elements, but they are described by 2D shapes.

Basic 3D elements are tetrahedrons, right prisms and general hexahedrons. 3D-elements are needed in situations, that are typically in complex shaped objects. There can be also isoparametric 3D elements, which can have curved boundaries.

Situations that are modelled for FEA are usually modelled as three-dimensional (3D) objects. Because elements affect the solutions' accuracy and complexity in analysis, they also affect to the computing time. If it is necessary to reduce computing time, 3D elements can sometimes be substituted by 3D shell elements or 3D beam elements. 3D shell elements are defined in 3D space and are abstracted to 2D elements and have the thickness as a physical property (Kuusisto 2017). 3D Beam elements are also defined in 3D space. The Beam elements have only one dimension, and their other dimensions are given as cross-sectional data (Dassault Systèmes 2010).

2.2.3 A 3D FEA Example

In this chapter there is a FEA example of a loaded beam structure (Salmi and Kuula 2012, p. 403). The structure is shown in Figure 2. The structure is too difficult or impossible to analyse by analytical formulas. However, the example is easy to calculate with FEM because of its numerical calculation method. The wanted results of the analysis are Von Mises stresses of the structure, and displacements in reverse direction to the load.

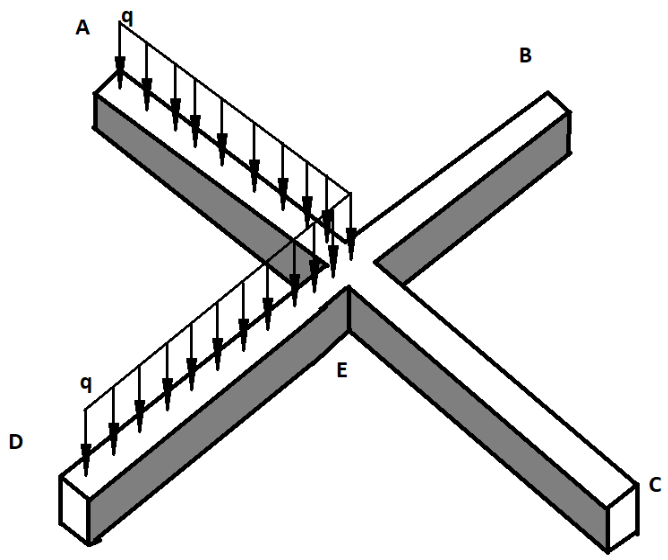

Figure 2: 3D FEA example. Ends A, B, C and D are fixed.

The dimensions in this example are following:

- Distance AE = 100mm
- Distance BE = 110mm
- Distance CE = 120mm
- Distance DE = 130mm
- The beams' thickness in both directions is 40mm

Load q is 40 kN/m. The FEA was done with Abaqus software. Analysis results are shown in Figure 3. The maximum Von Mises stress of the structure is 2.15 MPa, and the maximum displacement is $9.6 \cdot 10^{-4}$ mm.

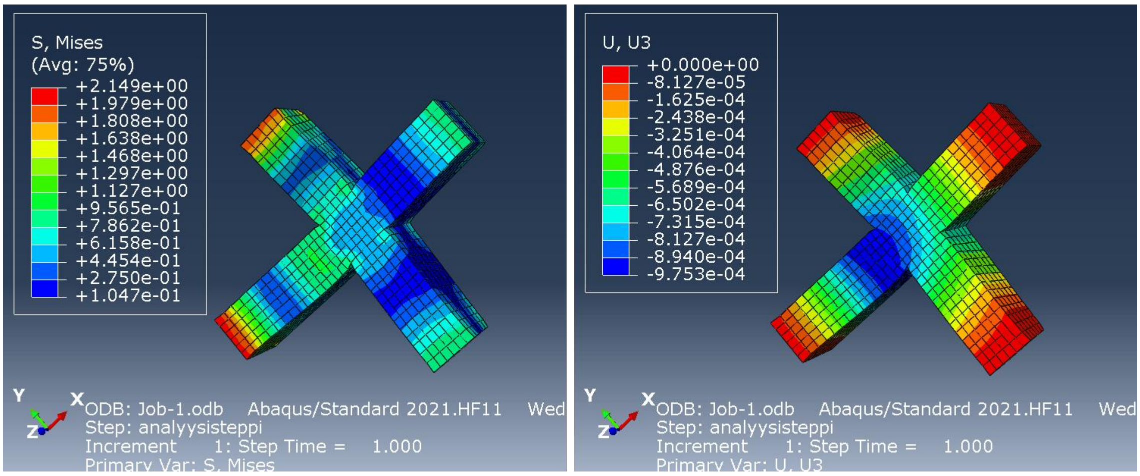

Figure 3: The analysis of 3D example. The end A is in the left top corner and end B in the right top corner. Units of stresses (left picture) are megapascals, and units of displacements (right picture) are millimeters.

2.3 CAD and FEA combined

One of the main themes in this work is combining CAD and FEA in product development. In Figure 4 there is a process diagram of using CAD and FEA separately and in Figure 5 there is a process diagram of using CAD and FEA combined.

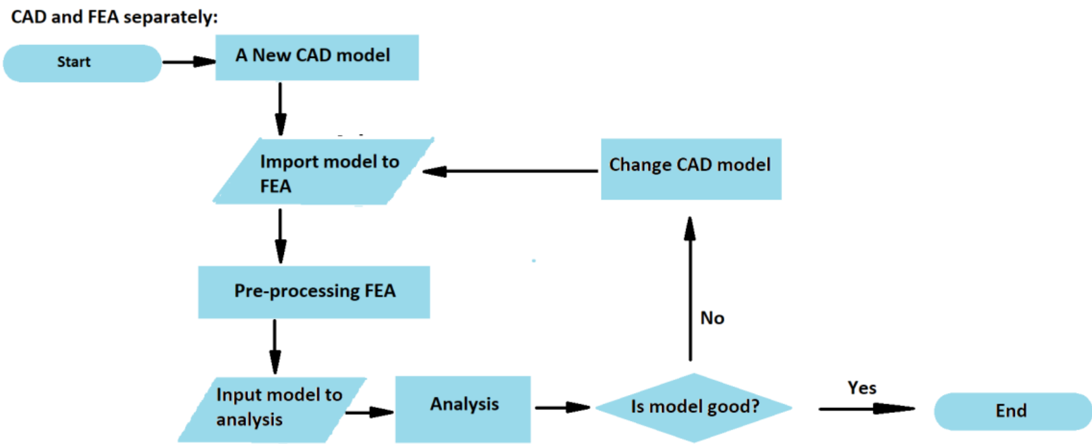

Figure 4: Using CAD and FEA separately.

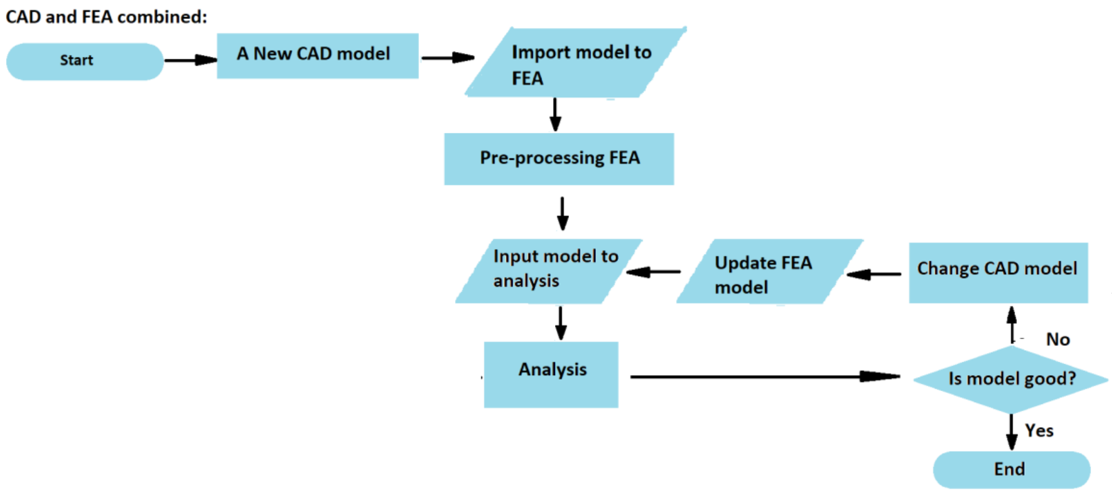

Figure 5: Using CAD and FEA combined.

By combining CAD and FEA the needed iterations are done faster and lead time of the design process is shorter than when using CAD and FEA separately. That is for example the case when the CAD model is not good, and the CAD model must be changed. In this case only the FEA model must be updated, and not to be done completely again. Also, when importing a model to FEA for the first time the import should be faster compared to a case where CAD and FEA are used separately.

The combining of CAD and FEA in this work is done by using CATIA V5 – Abaqus associative interface. The associative interface is introduced and represented in chapter 3.

2.3.1 Combination of CAD and FEA in Product Development

Combination of CAD and FEA is useful in product development because of fast iterations and short lead time. In product development process (Ulrich and Eppinger 2008, p. 9) there are integrative methods that the combination of CAD and FEA can help. Those methods are:

- Concept testing
- Industrial design
- Product architecture
- Design for manufacturing (DFM)
- Prototyping

In concept testing CAD and FEA can help in testing product functions (Hietikko 2015, p. 195). In industrial design they can help in generating more detailed concepts quickly (Ulrich and Eppinger p. 200). In product architecture with CAD and FEA, alternative architectures can be generated quickly. In DFM, designing can be done with CAD and FEA. In prototyping CAD and FEA can replace a physical prototype (Ulrich and Eppinger p. 257).

3 INTRODUCTION AND USAGE OF CATIA V5 - ABAQUS ASSOCIATIVE INTERFACE

CATIA V5 – Abaqus associative interface is basically a CATIA V5 interface which has an option to export a model to Abaqus automatically or manually straight from the interface. There are two ways to move part or assembly information to Abaqus from CATIA V5. The first way is to use the interface's export and the second way is to save a STEP file (.stp, or .step named file), CATIA product file (.CATProduct) or CATIA part file (.CATPart) to a computer and load it in Abaqus. This chapter introduces CATIA V5 - Abaqus associative interface by its usage with Abaqus.

3.1 Preparation of FEA-Simulation in CATIA V5

The preparation of FEA-simulation in CATIA V5 means making a CATIA model ready for FEA processing. That is basically making the model as simple as possible in CATIA without losing important geometries. The benefit of making the model simpler is reduced calculation time in FEA software. On the other hand, by making the model simpler some of the model's realism is lost. One way to prepare FEA in CATIA V5 is cutting a symmetrical part along its symmetry plane if symmetry boundary condition is possible to give for the model in the FEA software.

3.2 Export Methods from Associative Interface

An export method in the associative interface is a way to export CATIA part or assembly to Abaqus software. An export is needed for sending CAD data from the interface to Abaqus software for FEA processing. There are two options for exporting CATIA V5 part or assembly by associative interface. The options are automatic export and manual export.

3.2.1 Automatic Export with Associative Interface

The automatic export with the associative interface means an exporting method in which the CAD model is transferred directly from the associative interface to Abaqus/CAE software application. This method should be used when the CAD and FEA software are used simultaneously on the same computer.

Automatic export is done by a port connection between the associative interface and Abaqus software. Besides model exporting, it is also possible to send parameter data by port connection from CATIA V5 to Abaqus software and vice versa. Parameters in the associative interface and Abaqus software are discussed in chapter 3.4.

Usage of CATIA V5 - Abaqus associative interface's automatic exporting starts by opening the associative export from its desktop icon and Abaqus from its icon. Importing from CATIA is started by choosing module Assembly from Abaqus and from **Menu bar - Tools - CAD Interfaces - CATIA V5(…)**. The path is represented in Figure 6. The port used in importing from new CATIA V5 -window (represented in Figure 7) can be auto-assigned or chosen manually.

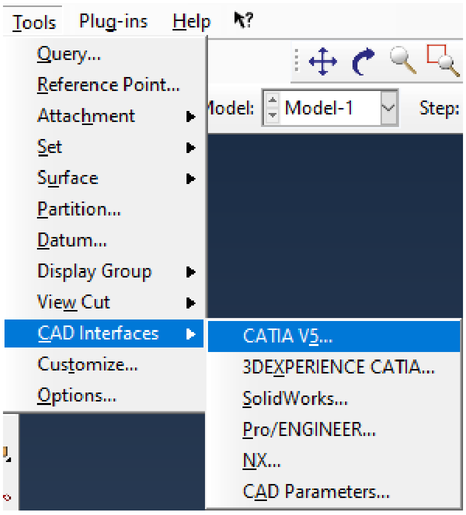

Figure 6: The path of starting the connection between CATIA and Abaqus.

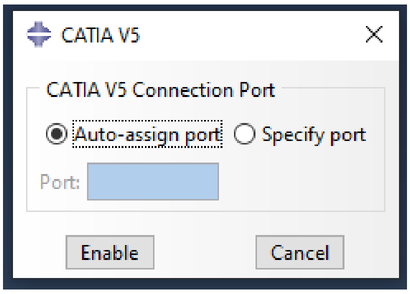

Figure 7: CATIA V5 -window of Abaqus

When having a CAD model for exporting from CATIA to Abaqus, click **Menu bar – Abaqus - Export to Abaqus/CAE(…)**. The export method can be chosen from new Export to Abaqus/CAE -window. Publication and material options are chosen from the same window. The window is represented in Figure 8.

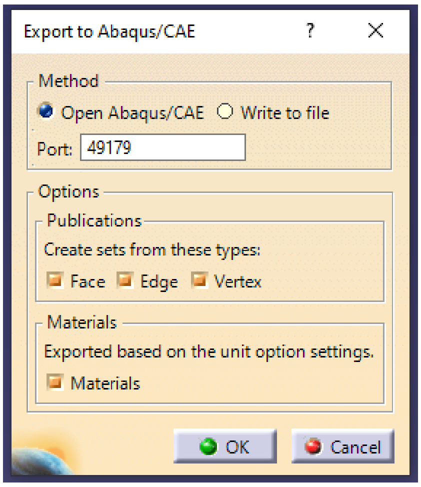

Figure 8: Export to Abaqus/CAE -window.

After the automatic export is done once it can be done by using Quick Export. Quick Export is used by clicking Quick Export -icon which is shown in Figure 9.

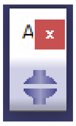

Figure 9: Quick Export -icon.

3.2.2 Manual Export with Associative Interface

In the manual export with the associative interface, the part or assembly is written to an EAF file in the export to a computer. This method should be used if the CAD and FEA tools are located on different computers. In this case it also requires some way of sharing the export between the computers. The path of the import in Abaqus/CAE software application is **File - Import - Assembly (...).** The path is shown in Figure 10.

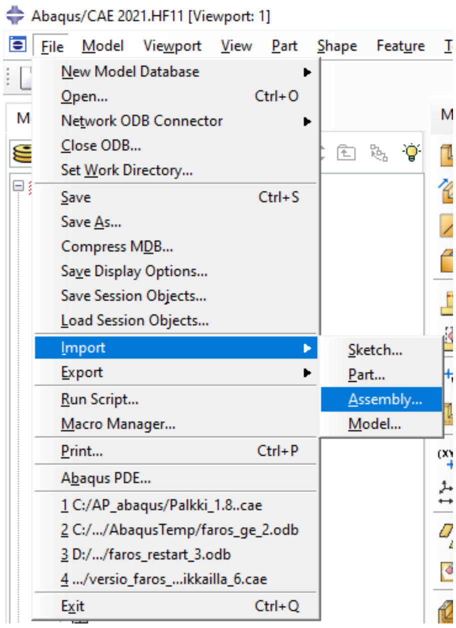

Figure 10: Importing assembly from Abaqus

3.3 Exporting CATIA Publications as Sets in Abaqus

In CATIA V5 there is a possibility to create sets for a FEA model by making publications. Published geometrical features are available in specification tree (Gade 2020). When exporting a model from the associative interface to Abaqus software, face, edge, and vertex publications can be exported as sets in Abaqus software's FEA model.

Doing sets by exporting publications in the associative interface is sometimes easier than in Abaqus. That is because in CATIA it is possible to choose a single geometrical feature straight from model tree, but in Abaqus that must be done always from the 3D view, which is challenging sometimes. Wanted publications are selected in Export to Abaqus/CAE -window, which is shown in Figure 8. Sets in Abaqus software are discussed in chapter 4.2.

Publications are set in CATIA V5 from **Tools - Publications**. That is shown in Figrue 11. When Publication -window is open, wanted publications can be chosen from part or assembly. Publication window is represented in Figure 12. Finally, after choosing publications and exporting a model to Abaqus, publications are shown as part's sets in Abaqus. One exported set is shown in Figure 13.

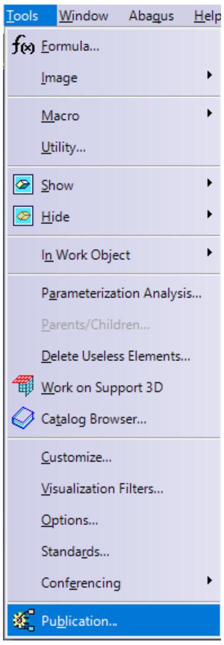

Figure 11: The path of publications in CATIA V5.

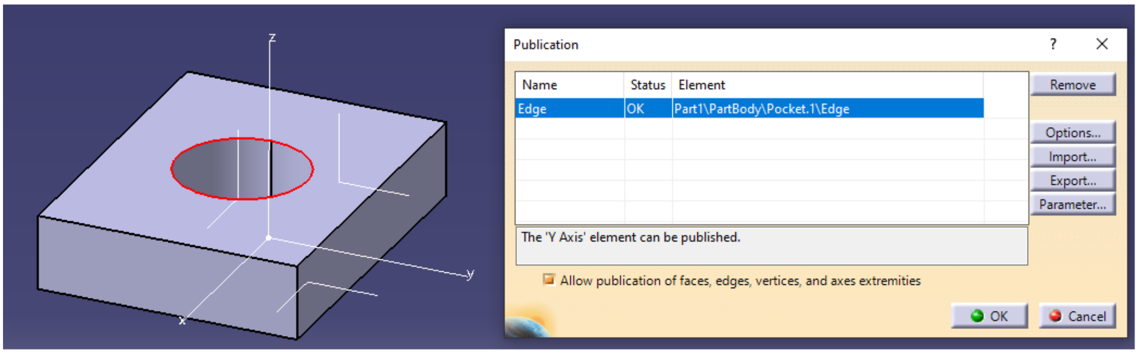

Figure 12: Selecting a publication for a part.

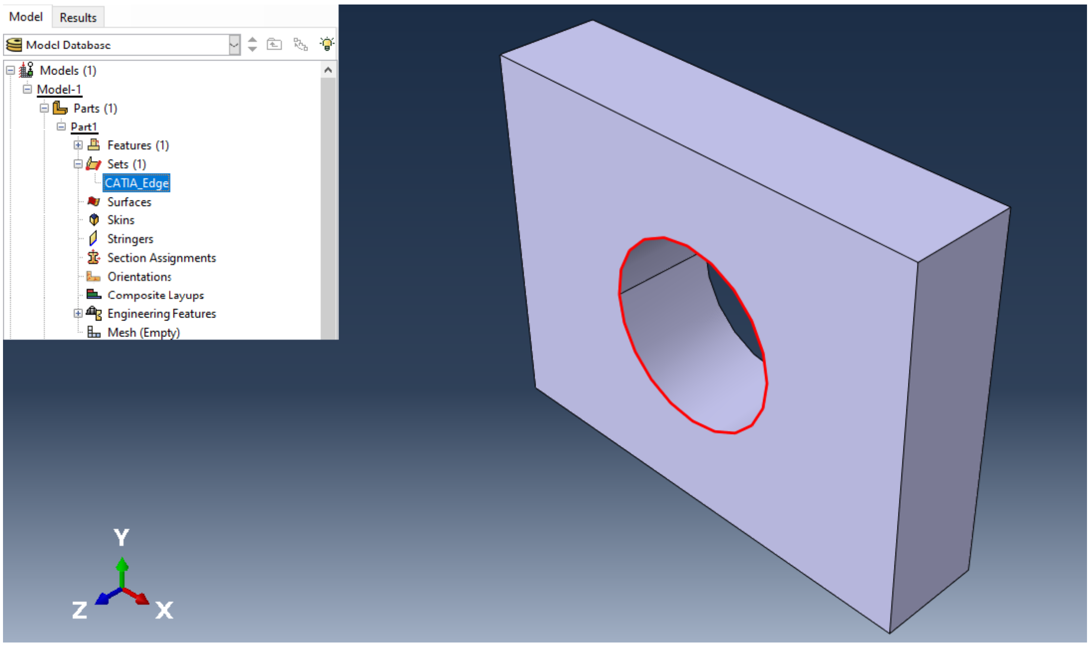

Figure 13: A publication in Abaqus as a set. In top left corner the set is in model tree, and in the big picture there is a part, and the set in red.

If materials are chosen in Export to Abaqus/CAE -window for an export, CATIA's material data is exported to CATIA. The imported material data has in Abaqus/CAE those values which are directly in units chosen in CATIA.

3.4 Activating/Deactivating CATIA V5 Components and Elements

Activation and deactivation are a way to turn on or turn off features of the model without deleting or remaking them. After activation or deactivation and export, the FEM model can be refreshed. Deactivating components and features make them not existing and activating components and features make them exist again in CATIA and Abaqus. Activation and deactivation are done in CATIA by clicking component or feature in model tree or in part view with mouse right button and choosing deactivation or activation from feature's object item from the list. Deactivation of a hole feature in CATIA is shown in Figure 14. The activation of the same feature is shown in Figure 15.

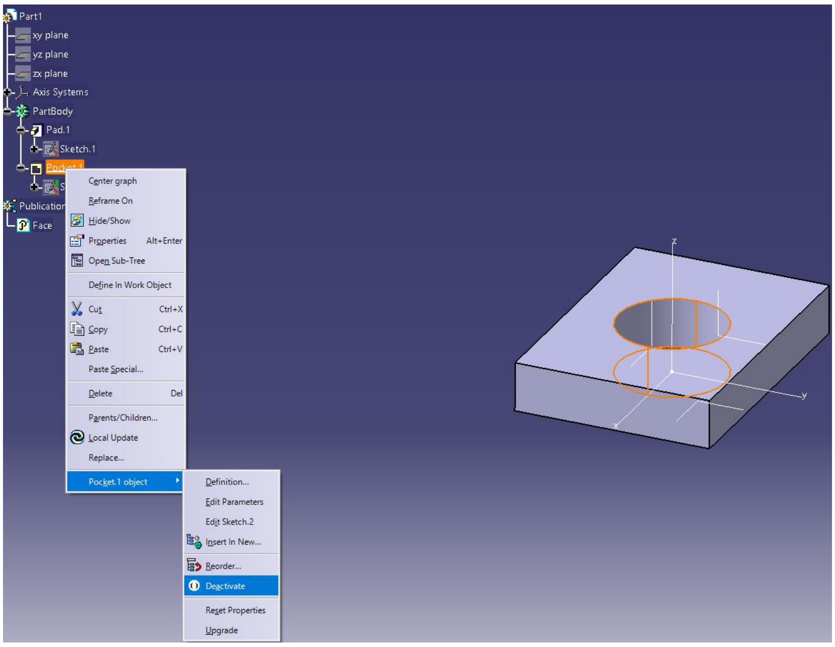

Figure 14: Deactivation of CATIA V5 part's hole feature.

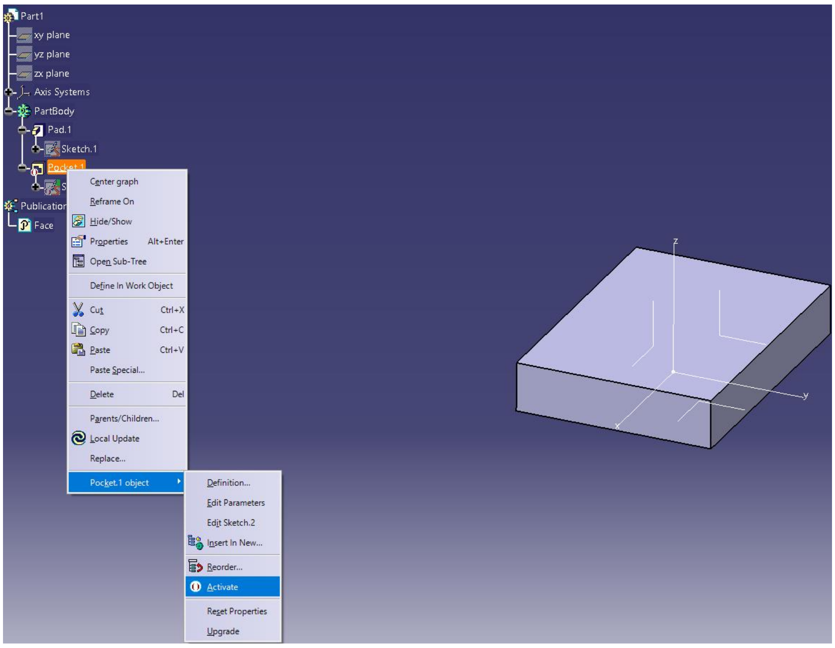

Figure 15: Activation of CATIA V5 part's hole feature.

3.5 CAD Parameters in Associative Interface and Abaqus Software

CAD parameters in the associative interface and Abaqus software are parameters that are set as models' dimensions and can be changed. Those parameters can be for example lengths, angles or formulas set to dimensions. By using the associative interface parameters, it is possible to drive the CATIA model geometry from Abaqus, although usually the CAD tool is used as a master.

The required parameters for an export to Abaqus must have a prefix ABQ_. That way the system recognizes the parameters to be exported. Parameters can be either defined in part level or product level in CATIA. If parameters are defined in part level in CATIA, they are shown and modified in Abaqus from part module in path **Tools - CAD Parameters (…)**. If parameters are defined in product level, they are shown and modified in assembly module of Abaqus in path **Tools - CAD Interfaces - CAD parameters (…).** The values of parameters can be updated in both CATIA and Abaqus. If an exported parameter is based on formula, the formula is not shown in Abaqus, only its value is exported to Abaqus.

In Figure 16, there is a CATIA part model of a brick with a hole in its centre. The dimensions of sides as parameters, are seen on top of the brick. The hole's radius is also as parameter. Model' parameters in Abaqus are shown in Figure 17.

In Figure 18 there is a CATIA product model with the brick, and a shaft in the brick's hole. In Figure 19 there are CAD parameters of the product in Abaqus/CAE application.

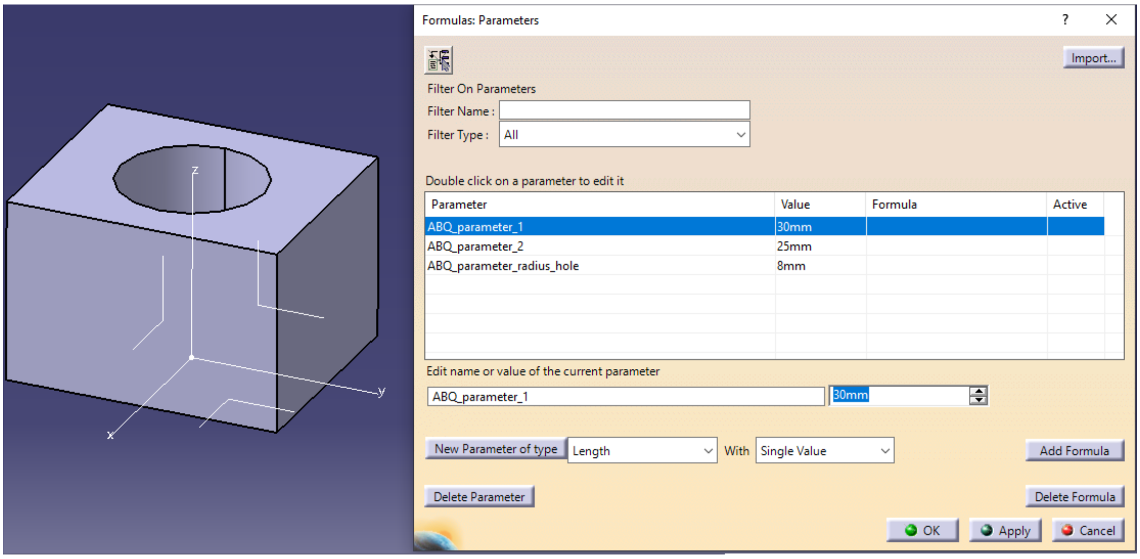

Figure 16: Parameters of a CATIA V5 part.

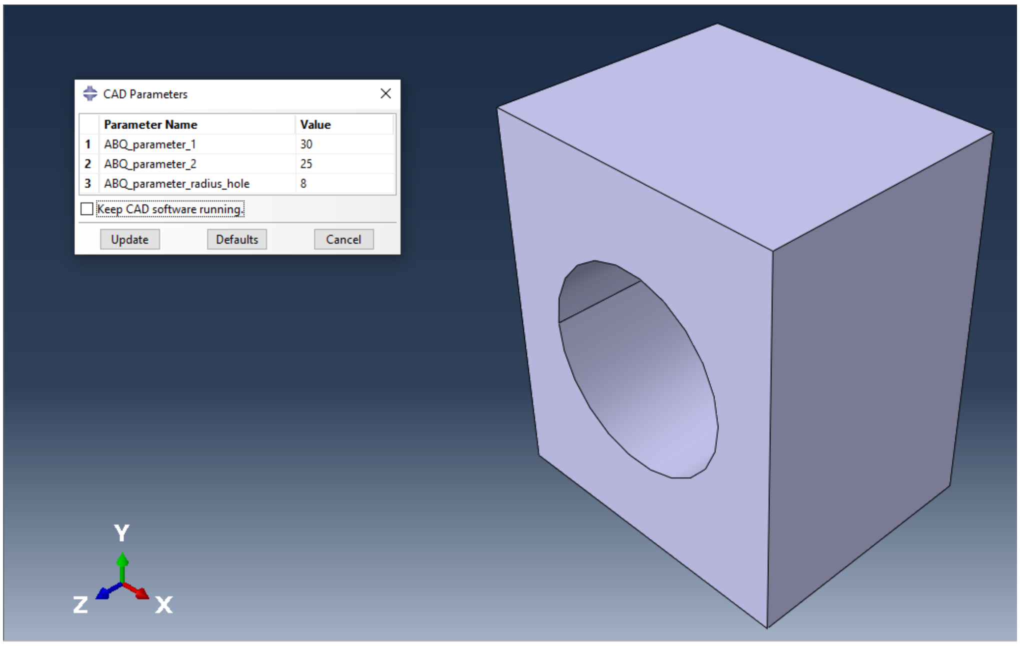

Figure 17: CAD Parameters of the part in Abaqus/CAE application.

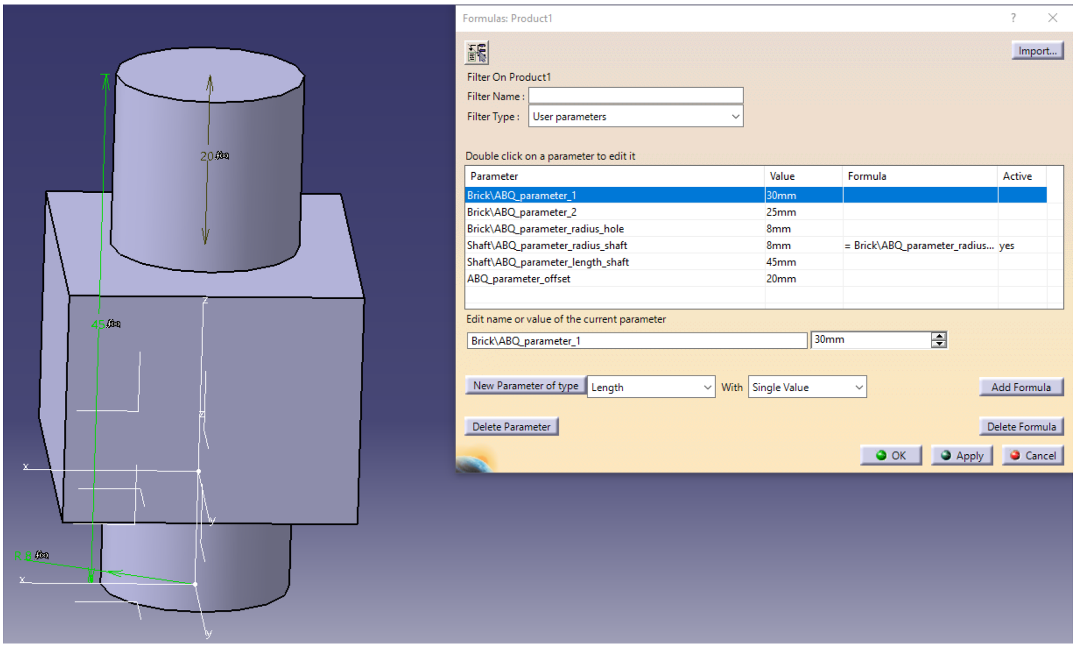

Figure 18: Parameters of a product in CATIA V5.

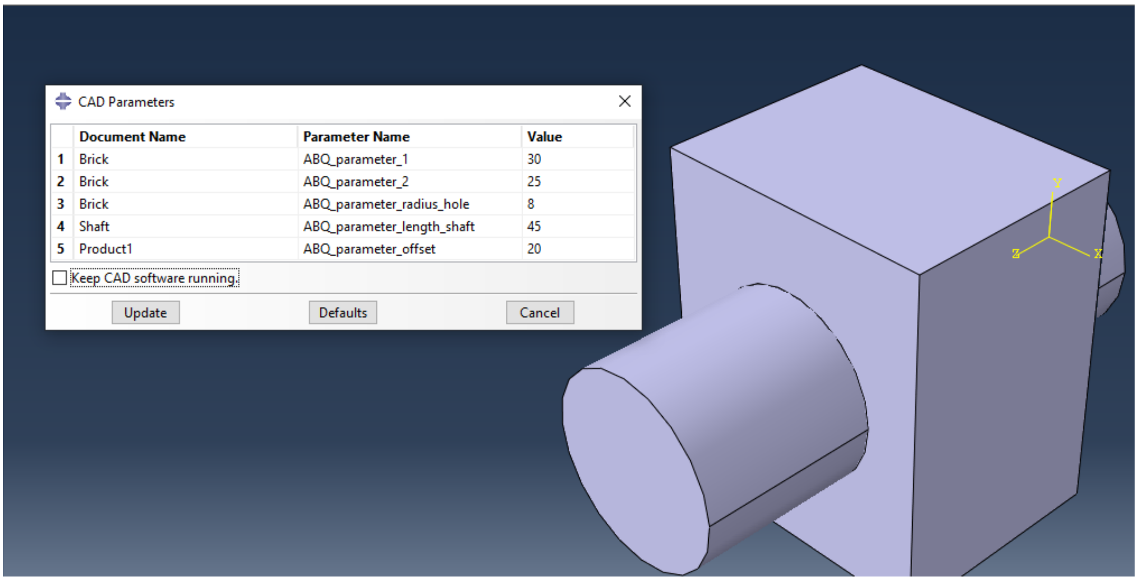

Figure 19: CAD Parameters of the product.

3.6 Units

There are no specified units in Abaqus. Results of the simulations are shown, and quantities are given by numbers without units. Therefore, the user is responsible for the compatibility of units. In structural analysis it is possible to use SI base units or as more typically, mm, N, s, Ton units. With units mentioned later the stress unit is MPa. CATIA exports numeral dimensions always in millimeters.

4 INTRODUCTION AND USAGE OF ABAQUS

Abaqus can be used by Abaqus/CAE or by Abaqus's own command line window. Abaqus/CAE has the following modules with different features:

- Part
- Property
- Assembly
- Step
- Interaction
- Load
- Mesh
- Job
- Visualization
- Sketch

This chapter introduces the most important features of Abaqus usage of importing model from CATIA V5, sets and surfaces, modules without sketch module, command line window usage and Compressing Model Database -feature.

4.1 Importing Models from CATIA V5

Importing models from CATIA V5 can be done in following ways:

- automatically by exporting model from CATIA – Abaqus associative interface with selected port (explained in chapter 3.2), or
- manually by importing with
    - EAF file in the associative interface (explained in chapter 3.2)
    - CATproduct file (CATIA products)
    - STEP file (CATIA products)
    - CATpart file (CATIA parts)
    - IGS file (CATIA parts and products)

Automatic export from CATIA and manual way of importing with an EAF file are the fastest way to import a model from CATIA to Abaqus. Automatic export and manual export in associative interface always update the existing parts. With the other import ways, new parts are added to model, and they must always be pre-processed again after import.

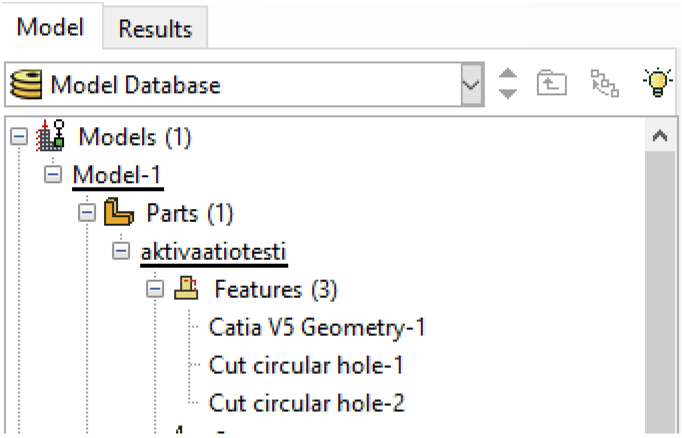

Figure 20: Picture of Abaqus/CAE's features of parts in model tree. The first feature is imported from CATIA V5, and the cuts are done in Abaqus.

4.2 Sets and Surfaces

Set means in Abaqus a defined collection of entities (ABAQUS, inc. 2006b). Sets can be defined for a part or for an assembly. Sets are assigned in creating of many features. For example, you can define a set from part geometry and assign material properties for it.

Surfaces in Abaqus are exactly defined collections of surfaces. Surfaces can be defined for a part or for an assembly. Surfaces must be assigned for example for Surface-to-Surface contacts, where the contact is defined by one surface and node region or by two surfaces.

4.3 Part Module

FE simulation's parts can be created and edited in part module. If CATIA V5 is used for part's creating and editing, part module's typical use is to create partitions, which can help with creating reasonable meshes for parts.

Partitions in Abaqus are a way to control meshing (discussed in chapter 4.9) of assembly's (discussed in chapter 4.5) instances. Meshing is done later in FEA process. Partitions can be done for example by drawing them to a model. Elements are formed up to shapes of the model and partitions according to seeding (discussed in chapter 4.9) for a part for meshing.

In Figure 21 there is a round 2D plane with a center hole without meshing and partitions and in Figure 22 there is the plane is meshed with quads. In Figure 23 there is the Figure 21's and Figure 22's plane with partitions and in Figure 24 the plane is meshed with quads.

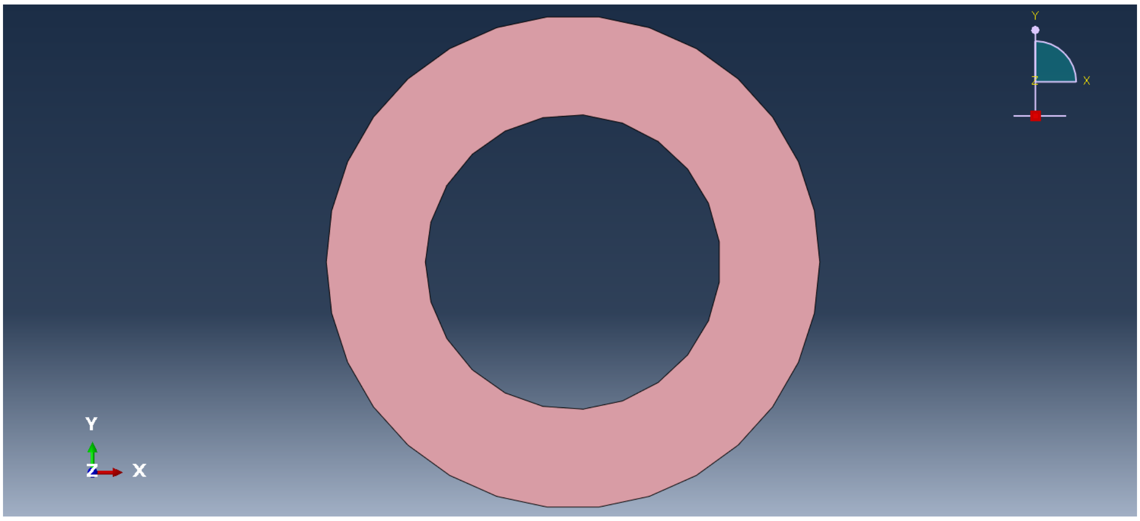

Figure 21: A 2D plane without partitions.

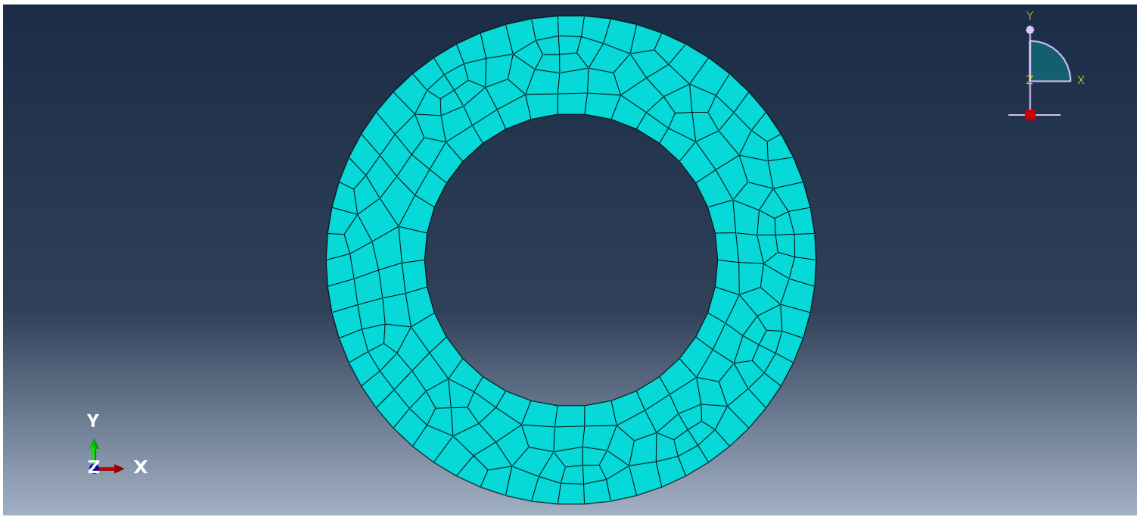

Figure 22: The 2D plane meshed by quads in Abaqus without partitions.

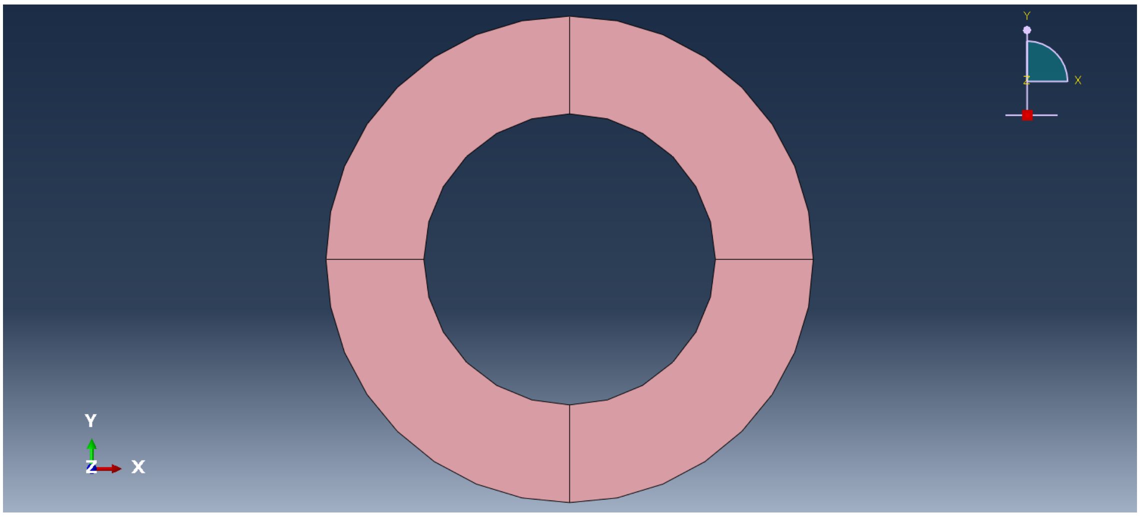

Figure 23: The plane with partitions.

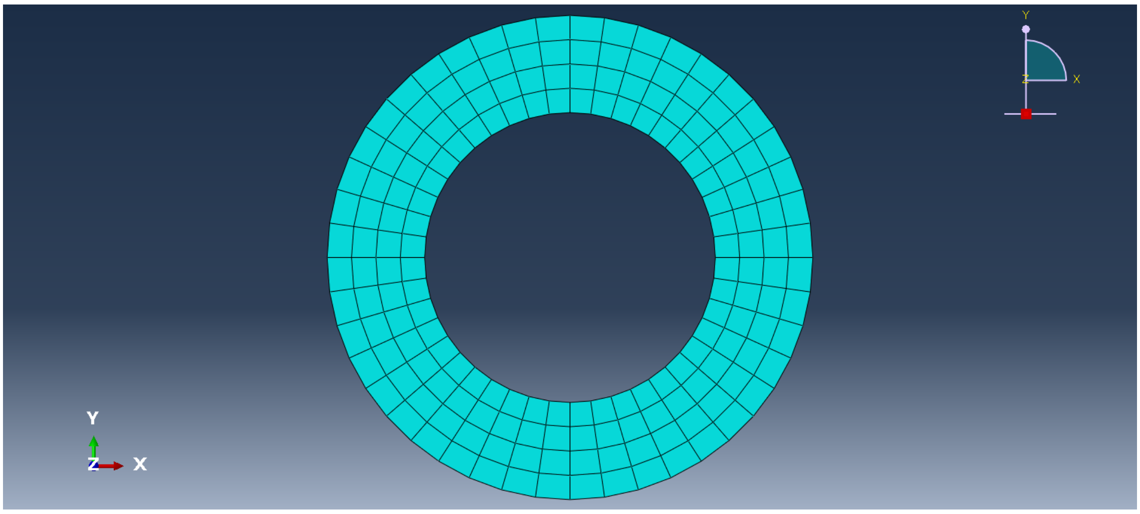

Figure 24: The meshed plane with partitions.

4.4 Property Module

The parts' material properties are assigned in property module. In this work assigning material properties are focused on 3D cases. Material parameters are set in Create Material -feature and in Edit Material -window. The Edit Material -window is in appendix 1 Figure 48.

When material parameters are set, next phase is to create a section for material. A section contains material property, and it is assigned for a part or a region of a part (Dassault Systèmes 2014a). Normally, solid and homogenous selection is the needed option in sections of 3D cases. The Create Section -window is in appendix 1, Figure 49 and the Edit Section -window is in appendix 1, Figure 50. After assigning a section, the last phase of assigning the material properties is to assign sections to the parts.

4.5 Assembly Module

Part instances for the analysis are positioned in assembly module. Abaqus analysis considers only those parts that are brought to an assembly. If a part is not brought to an assembly, it will not affect the analysis at all. Instances can be meshed individually, if needed, and the loads and boundary conditions can be applied on them. In assembly module it is also possible to make linear or radial pattern from instances, translate instances and make a new part and to make new instances by merging and cutting the existing ones. Connections with CAD interfaces, for example with CATIA V5 - Abaqus associative interface (introduced in chapter 3.2), are set in assembly module.

4.6 Step Module

Steps for FE simulation run are defined in step module. The steps define the sequence of how and when the loads and boundary conditions are applied to the model during the simulation. For a simple example a FEA model can contain three steps, where in first step there is a cantilever beam having a load of 10 kN, in second step 15 kN and in last step there is no load at all. By those three steps it is possible to see how the beam reacts to the variating load by time.

A step has a certain time, which is called step time. In analysis job's run (jobs are discussed in chapter 4.10), the goal is to get through the steps. With non-linear models, the step time is divided to smaller increments. The required number of increments depends on the complexity of the model. With purely linear analyses, only one increment is needed. In complicated models it is good to start with smaller sizes of increments than the full time of the step and with a smaller size of minimum increments. The increment size must be small enough to capture the events that have an effect to the next increment. Abaqus automatically decreases the increment size within given limits if the solution doesn't converge.

Abaqus has an initial step, where the software starts the simulation. Initial step sets for example the boundary conditions of the structure at the beginning of the analysis.

The initial step always exists, there is no need to do it separately. Other steps must always be created separately for the model. Easiest way to do the management of steps is to use the step manager, which is shown in Figure 25.

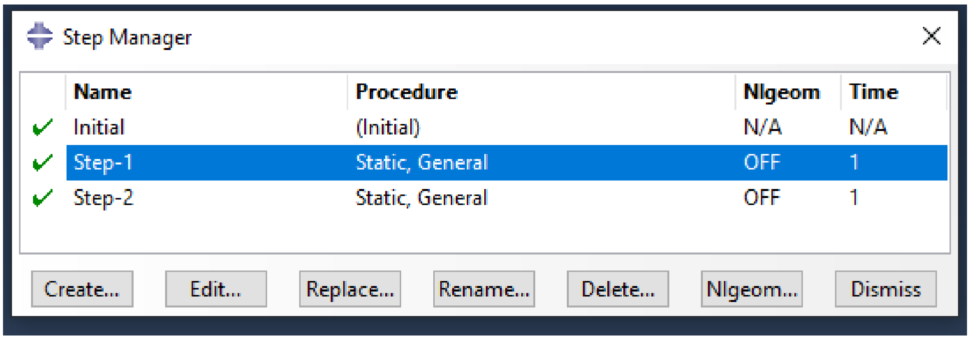

Figure 25: The step manager.

Other important features in the step module are the Field and History Outputs for runs, which give the wanted results later in output database. Field Output results are presented with a graphical display which can have color map and parts colored by represented quantity. The quantities are set in step module's field output -feature before calculations.

History Outputs are typically values for one point during the steps. History outputs are set in step module. With history output, it is possible to create plots, and to research them for example by making a plot of their relations.

For outputs it is possible to set time points, which define the outputs' time intervals. If needed, it is possible to make restart requests and to change general solution controls in this module.

When creating a step, you can choose a step procedure. Some procedures are for example Static, General; Dynamic, Implicit; and Dynamic, Explicit.

Some other setups that can be set in step module are Nlgeom on or off, and time and incrementation setups. Nlgeom must be set on, when Abaqus model has nonlinear-behaving instances in a model. Some examples of nonlinearity with instances are nonlinear materials, variating contact, and big displacements. When editing steps, it is possible to set the time for step and for example in automatic increment sizing initial, minimum, and maximum increments.

4.7 Interaction Module

There is a set of following important features in interaction module:

- interactions
- interference fits for interactions
- interaction properties
- constraints
- reference points

In Abaqus, interactions can be defined between several instances or within one instance. Interactions can be for example different kind of contacts, which are general contacts, surface-to-surface contacts, and self-contacts. For contacts, contacting surfaces must be defined. Interactions can be managed with interaction manager. Defining general contacts is the way to set every possible contact in FE simulation. General contacts are set ready in initial steps in Static, General and Dynamic, Implicit type of analyses; and they cannot be set in other steps. In Dynamic, Explicit type of analyses general contacts are available to set in every step.

Defining Surface-to-Surface contacts is the way to define individual contact pairs. Self-contacts are needed when a surface is contacting with itself, like for example cavity seal from inside when compressed. Surface-to-surface contacts and self-contacts can be set in any step.

In interference fit, basically a weaker instance fit to a stronger one. The weaker instance gets stretched or compressed, which leads to stress in both instances in the step where the interference fit happens. The interference fit is typically done in beginning steps of simulations. The interference fit can be assigned to a contact with General Contact's setup or by Surface-to-Surface contact's setup. In Figure 26 there are pictures of a starting and an ending point of the interference fit simulation.

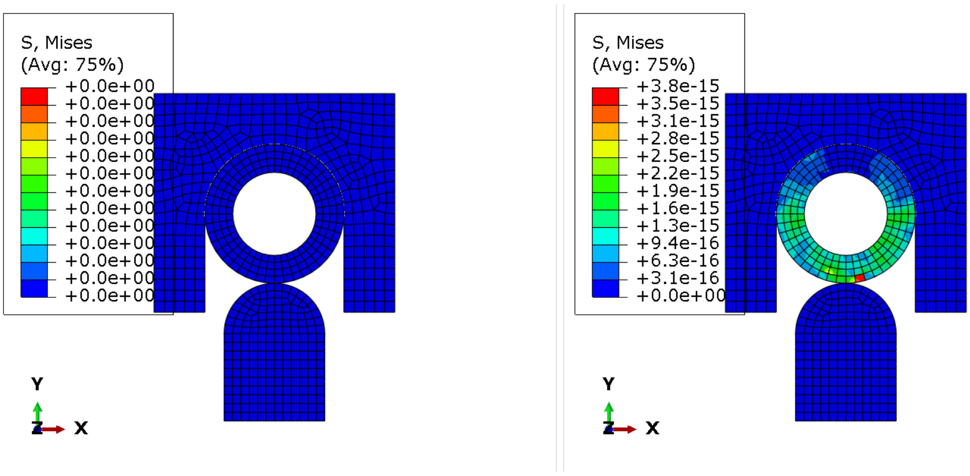

Figure 26: 2D pictures of interference fit simulation. The left picture is the starting point, and the right picture is the ending point of an interference fit simulation.

There are many kinds of interaction properties in Abaqus. In product development, the contact properties are typically needed. In contact properties it is possible to set contact friction and contact's pressure-closure method.

Constraints are a way to affect instances' positions and degrees of freedom in simulations. In constraints the two most needed ones are tie and coupling. Tie constraint ties two surfaces, two node regions, or a surface and a node region together. Coupling constraint couples control points with the selected surfaces. After coupling, the point and the surface are connected, and for example a concentrated force can be set for a surface by coupled point. Control point can be any node of the model.

Reference point is set by coordinates or by an existing node or vertice. For example, it is possible to set a concentrated force in the load module to a reference point.

Interactions can be controlled with a manager window. The Interaction Manager -window is shown in appendix 1 Figure 51.

4.8 Load Module

Loads and boundary conditions for FEA model are given in load module. When adding loads and boundary conditions, it is possible to manage them with manager windows. Loads can be for example concentrated forces, moments, pressures and gravity.

Boundary conditions can be for example displacements, rotations, symmetries and rigid fixations. Create Load -window is shown in appendix 1, Figure 52.

4.9 Mesh Module

FEM-mesh is made in the mesh module. The mesh is a part that consists of only FEA nodes and elements having types defined for meshing. In Abaqus seeds are done before meshing. Seeds are points, where the meshing algorithm tries to place the element nodes.

Seeds can be global or local in Abaqus. When defining the global seeds in Abaqus, an approximate seed size is given for a part or for an assembly instance. The portion is controlled by the user. Element density can be controlled more precisely by setting local seeds to individual edges. The selected regions can be earlier defined sets or surfaces.

It is also necessary to decide the element types for doing mesh. Element types of Abaqus' 3D elements are hexahedral elements, hex-dominating elements, tetrahedral elements, and wedge elements. Hexahedral elements mean a polyhedron element that has six faces, like for example a cube. Hex-dominating elements mean elements that are primarily hexahedral elements but allows sometimes triangular prism elements. Tetrahedral elements mean elements that have four triangular faces and six straight edges. Wedge element means an element shaped as a prism (ABAQUS, inc 2006a).

Global seeds are given in selection "Seed part". The global seeds -window is presented in appendix 1, Figure 53. Local seeds are given in selection "Seed edges". The local seeds -window is shown in appendix 1, Figure 54. Elements are controlled by selecting from **Menu Bar - Mesh – Controls (...)**, and by selecting from **Menu Bar - Mesh - Element type (...)**. The Mesh Controls are shown in appendix 1, Figure 55.

4.10 Job Module

In a job module an analysis job is created, simulation is written as an INP file, and FE simulation run is started. In this chapter creating a job, a datacheck and submitting and monitoring job are introduced. Features creating an input and creating a restart are introduced in appendix 2.

4.10.1 Creating a Job

A job is needed before starting to solve a simulation. One important option in creating a job is choosing the job type, which can be full analysis, recover (dynamic, explicit steps only) or restart. Also, the number of used processors and the use of a graphics card in simulations' calculations can be set for a job.

4.10.2 Datacheck

A datacheck checks the model for syntax errors or missing mandatory information etc. A datacheck create an INP file and a STT file before checking model (Dassault Systèmes, 2014b). Every submitted analysis begins with a datacheck and progresses to actual solving if no errors are found. If a datacheck was done separately, it can be used as a starting point for solving stage using the continue command.

4.10.3 Submitting and Monitoring a Job

The calculations are started by submitting a job. Submitting a job is done by clicking a required job wanted to start by mouse right button from the model tree. By submitting a job Abaqus creates an input, makes a possible data check, and runs calculations.

It is possible to follow the progression of the job by using a monitor in Abaqus/CAE, or by checking the files Abaqus creates during the calculations. Monitoring a job is done by clicking the created from the model tree with mouse right button and choosing to monitor. With Abaqus/CAE's monitor it is possible to observe the following data in the simulation run:

- Log
- Errors
- Warnings
- Output

- Data file
- Message file
- Status file

The files are in the set working folder. With the log it is possible to see simulation's start and end time. Besides the Abaqus/CAE monitor, the log can be found as a file in the set working folder. An error stops the analysis. Warnings are informative messages for possible problems. Output shows the steps of an output that run has done for the model.

Data file, message file and status file can be found besides the monitor from the temporary file folder. Data file contains typically information about the analysis model. Message file contains information related to the progress of the analysis. Status file shows how many iterations were needed for each analysis step and if there were any convergence problems during the run. Data, message, and status files are in the same set working folder where the log file is also found.

4.11 Using Command Line Window

It is possible to run Abaqus jobs by using the command line window. The advantage compared to run in Abaqus/CAE is that computer's RAM and some of processing capacity are saved if the Abaqus/CAE -window is closed before starting the analysis. That is why the analyses launched from command line run typically faster. Also, the analysis status observation is avoided if the analysis is started via command line. In this work runs were started with two commands:

- abaqus job=”xx” cpus=”number” int ask_delete=OFF
- abaqus oldjob=”yy” job=”xx”

The first command is for normal running, and second one for restart. In commands

- “xx” is your job and INP file name.
- “number” is number of cpus you want to use in run
- “yy” is the job and INP file name which is restarted.

4.12 Visualization Module

The results of simulation are shown in visualization module. There are two types of result outputs: Field Outputs and History Outputs. Both Field and History Outputs are introduced in chapter 4.6. Examples of the results are shown in chapter 5.11.

In visualization module it is also possible to do view cuts for models and display them graphically. By doing view cuts, it is possible to investigate Field Output results inside the model.

4.13 Compressing MDB

Model file size grows every time you save a modified model. The file size decreases by compressing model database (MDB) and using the database gets quicker. The compression is done by choosing File from **Menu Bar - Compress MDB(…)**. The location of the selection is shown in
Figure 27.

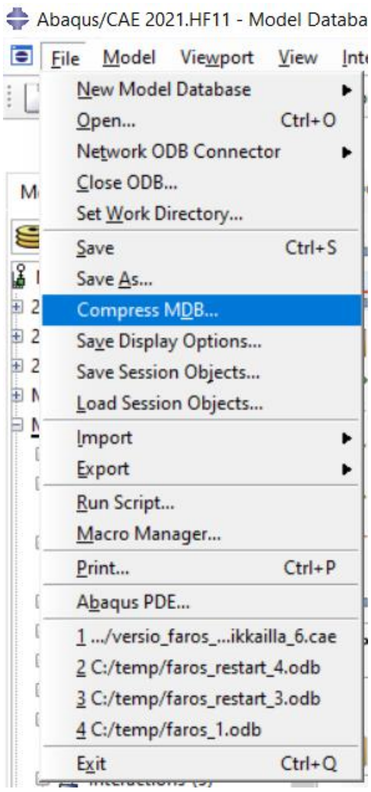

Figure 27: Compress MDB(…) option in menu.

5 CASE STUDY: SQUEEZED CAVITY SEAL

This master's thesis' case study is a situation that has a housing consisting of two plastic shells, which have a cavity seal between them. The cavity seal is made from hyperelastic material and has variating contact between the shells and the cavity seal. By those, it is a good example of nonlinear case for Abaqus. Case study's FE simulation has four steps:

1. Interference fit step
2. Forced displacement step
3. Screw tightening step
4. Relaxing forced displacement step.

In interference fit step the cavity seal shrinks to the sealing groove of the front shell and the seal's compression stress grows. In forced displacement step the cavity seal is squeezed between the shells by forced displacement of the back shell. In screw tightening step, the screw modeling made by wire feature is tightened and is fastening the shells. In the last, fourth step the displacement force set in second step, is relaxed.

The information required from FE simulation is basically this:

- Stresses in screw towers of the plastic shells, and
- Deformations of the plastic shells.

This case study includes two Abaqus models: model A with plastic shells and a seal as original models from CATIA V5, and model B, where the shells are the same, but the seal is modeled by simple rectangular prism gasket-elements. Model B was needed because model A's job run went through the three first steps but stopped in the fourth step. Because of that, it is necessary to know if model B is suitable for representing the simulation case. That is why a comparison of the models was needed. The comparison is done in the step time of 0.6 in the fourth step. Every step's full step time is 1.0.

5.1 Preparing Simulation in CATIA V5

Models A and B contain three parts: front shell, back shell, and cavity seal. The models are differing by their cavity seal models: cross section of model A's cavity seal is round and hollow, and model B's cavity seal is replaced by a gasket part that has a solid and rectangular cross section. Plastic shells' and model A's cavity seal's CAD-parts were used as they were, and model B's gasket part was done in CATIA by replacing model A's cavity seal model with a gasket model that has a cross section as described earlier. Before last export of the models from CATIA to Abaqus, shell models had to be "cleaned" from small details like sharp corners and gaps.

For same reasons, CATIA-parts were halved to utilize symmetry boundary condition. Front shell was halved in CATIA, and back shell and gaskets in Abaqus. The symmetry option was taken in use after it was found. By that long analysis runs of the forced displacement step was reduced.

5.2 Symmetry in FEA Model

For making an Abaqus run in the case study faster Abaqus model's parts were halved, and parts were given a symmetry boundary condition in Abaqus. Front shell was halved in CATIA, and other parts were halved in Abaqus. Halving in Abaqus was done in the following way:

- Every part in the FEA model have origin in the center of the part by their x-direction, which was exploited.
- In part module, as a first thing an additional plane was done as an offset outside of the yz-plane. The offset value was 20.0.
- From part module's icons Create Cut: Extrude was chosen.
- The plane that was done was chosen.
- When a text "select an edge or axis that will appear" came, the part from the side of the done plane was chosen.
- In the sketch mode a rectangle that cover was drawn to cover the whole part.
- Option blind was chosen in the Edit Cut Extrusion -window. As a depth value a 20 was given, which is the same value with plane's offset from part's yz-plane.

- After editing the cut extrusion halving was done.

After removing the other half of the model to use symmetry, a symmetry boundary condition was given for the model's cutting plane. Assigning symmetry boundary conditions to the model's instances is discussed in chapter 5.8.

5.3 Assigning Material Property

The material properties for plastic shells and cavity seal were as given values for case study. In model A material property was given straight in Create Material -feature, and in model B material property for gasket-element was researched within 2D situation.

Young's modulus of 2270 MPa, and Poisson's ratio of 0.36 were the material properties of the shells. For cavity seal hyperelastic Mooney-Rivlin constants were:

- C10: 0.200752
- C01: 0.050188
- D1: 0.039983

5.3.1 Material Property for Model B's Gasket

For model B, finding out material behavior had to be done for gasket element. That was done by doing a 2D FE simulation, which was basically cross section of squeezing the cavity seal and it had three parts: cavity seal, plastic slot for cavity seal and plastic punch, which compressed the cavity seal into the slot.

The simulation was done by three static, general steps: interference fit and two forced displacements in vertical and transversal directions. In interference fit step, the cavity seal fitted into its slot by interference fit. In forced displacement step, the plastic punch compresses cavity seal into the slot further. In transversal directions step, the punch is moving to transversal direction while keeping the vertical displacement constant. Figure 28 to Figure 30 show different phases of the simulation.

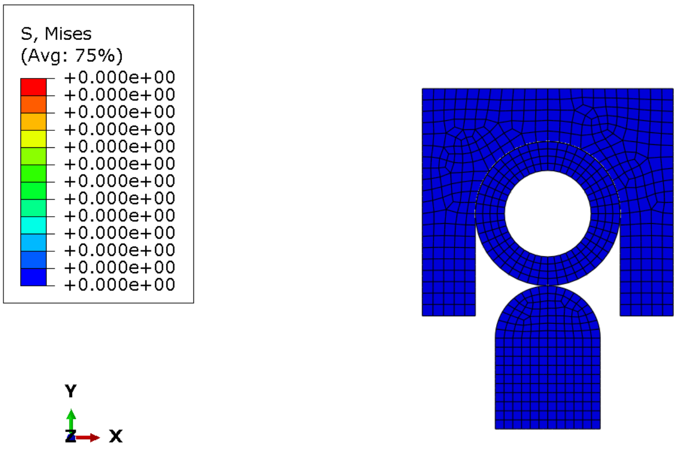

Figure 28: The 2D simulation in starting point.

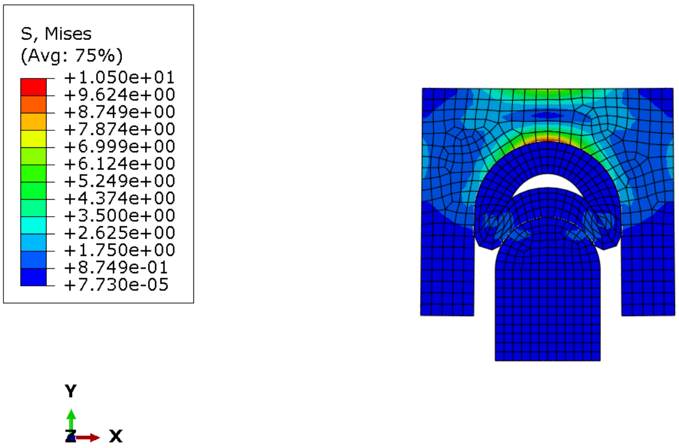

Figure 29: The 2D simulation in the end of the forced displacement step.

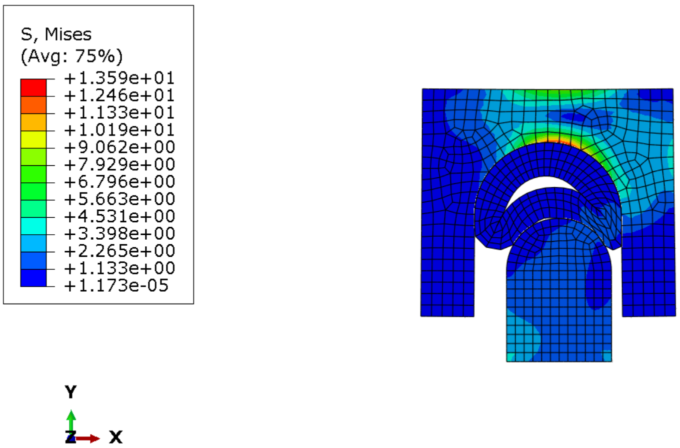

Figure 30: The 2D simulation in the end of the transversal directions step.

By using History Output in the model in the displacement step, plots for force and displacement were stored related to time. Using the operate on XY Data Tools, the results were combined to get a force-displacement curve. The plot is in Figure 31.

By choosing Edit from XY Data Manager -window, a table with columns of the force and the corresponding displacement was got. Because the gasket elements require pressure-compression data, the data was transformed with Microsoft Excel to pressure data for gasket element by dividing the forces with 0.1, which was the width of the gasket in millimeters. Finally, pressure and closure data were written in gasket thickness behavior as material behavior.

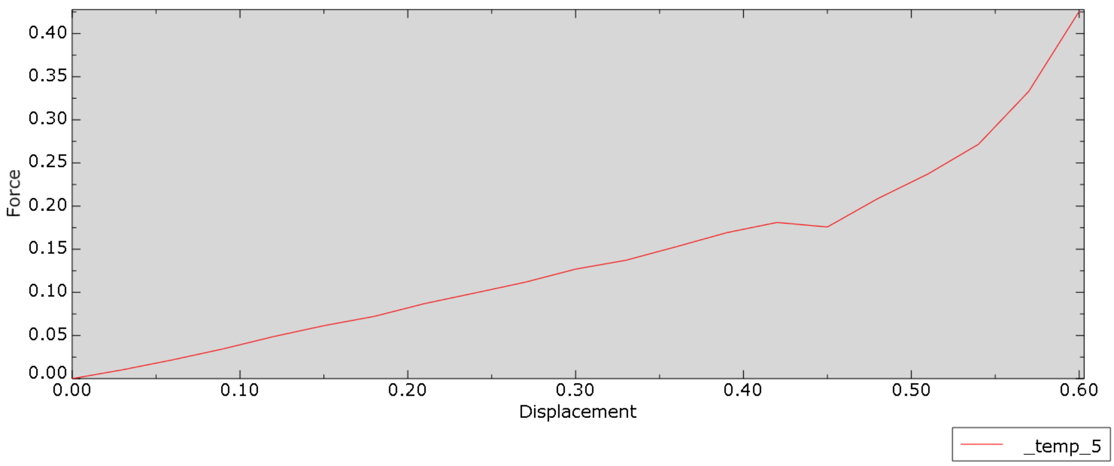

Figure 31: The force-displacement plot of the 2D situation.

From transversal direction step, a plot was also got for force and displacement. With plot of those two, linear factor was got for force and small displacements. That factor was converted to stress and displacement factor by dividing it with 0.1. After that, the result factor was given as a gasket transverse shear elastic material behavior factor. Finally, when the material parameters were given, the gasket section was made and assigned for the gasket model.

5.4 Assembly and Step Module

In the assembly module the back shell was translated 0.52mm out of the cavity seal so that the shell's punch was touching the cavity seal tangentially. There was no other remarkable feature used in assembly module in the case study.

In this case study's FEA model had four Static, General -steps: interference fit, forced displacement, tightening screws and relaxing forces caused by forced displacement. Every step has Nlgeom setup on because of hyperelastic material behavior of cavity seal. Energy-based damping was defined on step level to make simulations easier.

The steps have the field output requests for stresses and strains. The history output requests were not needed. The results were saved from 10 equally distributed points during the analysis step. The restart request was done for model A from end of the interference step. By doing a restart to model A, it was faster to run model A's analysis jobs' again.

5.5 Constraints

In this case study the needed constraints were couplings and ties between the gasket and the shells. For both models A and B reference points were given for couplings. Those were needed for wires of screw modeling and for the last step's rigid fixation of a small area, that prevent the front shell from moving in the end of simulation.

The front shell's screw towers were set to coupling's reference points in the centers of the screw holes in back shells. The used couplings were structural distributing, which means they had no degrees of freedom with surfaces they were set to use with.

There were two tie constraints used in model B. One tie constraint was set between the front shell's gasket groove and the gasket model's corresponding surface. Another tie constraint was set between the compressing geometry of the back shell and the gasket model's corresponding surface. By tie constraints the model B's gasket does not move when compressing it in simulation, and therefore the gasket can be compressed in the right way between the shells.

5.6 Creating Screw Model

The simulations' screw modelling was done with the Wire feature in the interaction module. The first reference points were done in the middle of every three back shell's screw hole and inside and in the middle of every three front shell's screw tower. Then in the Create Wire -feature, a window with disjoint wires method, three pieces of point pairs were chosen. Every point pair has one point from one shell and corresponding point from the other shell. After that, resulting three wires were given a translator section and a datum coordinate system having x-coordinate in direction of wires in global z-coordinate direction. Connector force for wires that model the screw forces, is 10 N.

5.7 Assigning Interactions

In this case study four surface-to-surface contacts are used: two for screw tower pairs of plastic shells, one for seal and front shell's seal groove and one for seal and back shell's punch. There is also a model change, where wires modeling the screws (introduced in chapter 5.6) are set to start existing in screw tightening step.

A contact control -feature is used for the surface-to-surface contacts in this case study. Automatic stabilization with a small factor of 0.01 was done for helping the simulations run through. Also, the interaction properties had to be given for interactions. For interactions "hard" contact -setup and friction coefficient of 0.05 were given. Also, in the model A the interference fit was assigned to the cavity seal model.

5.8 Assigning Loads and Boundary Conditions

The case study's simulation has a connector force for screw simulating wires, that were introduced in chapter 5.6. The case study's analysis has no other loads. Connector force for every wire was 10 N.

This simulation has following boundary conditions:

- Symmetries for cavity seal and shell halves in x-direction
- Back shell's compressing displacement of 0.52 mm for steps from displacement step to screw tightening step's end
- Rigid fixation for front shell's center of gravity, that makes it not moving in three first step. The center of gravity is coupled with an inside surface of the front shell.

- Small rigid fixation for front shell, that makes it not moving but also not making the front shell stiff in the last step.

5.9 Creating Meshes

Meshes were created for both shells and gaskets of both models. Shells were meshed by tetrahedron-elements sized in approximate global size of 0.9. In this chapter also model A's and model B's element sizing are introduced.

5.9.1 Model A's Gasket

The element size of the model A's gasket model should have been very small for passing the simulation. That was not done, because the simulation would have taken unreasonably long time with the available laptop computer.

With a bigger global seed size of 0.01045 and element type of tetrahedrons, simulation was got from the beginning to step time of 0.6 of the fourth step. The sizing was determined by doing iterative calculation with a 2D situation of squeezed cavity seal. In

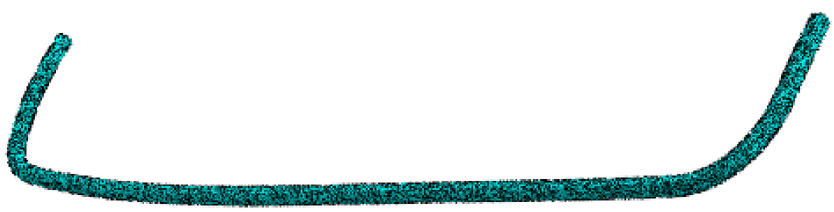

Figure 32 and Figure 33 there are pictures of the model A's meshed cavity seal model.

Figure 32: Symmetry using model A's gasket meshed with tetrahedron elements.

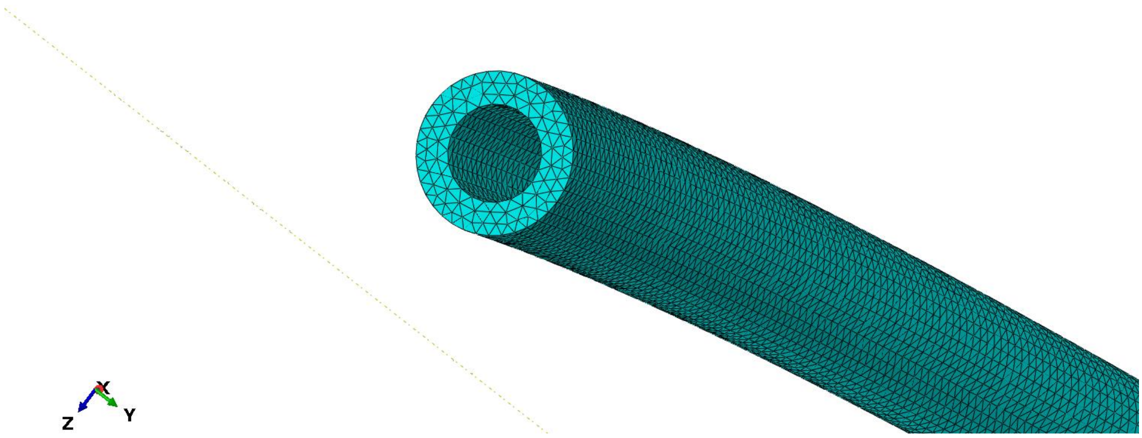

Figure 33: Closer picture of the model A's gasket and it's meshing.

5.9.2 Model B's Gasket

Model B's gasket that replaced the original cavity seal part, is meshed with gasket elements. Gasket elements are special element type of Abaqus. Gasket elements are made for gasket models and are used for squeezing in simulation only in its thickness direction. A material property of pressures and corresponding closures must be given for the gasket element. The material property to the gasket element was given in chapter 5.3.1.

With gasket element there can be only one element in the thickness direction. That thickness direction is set by stack direction in mesh module's controls and clicking assign stack direction button.

For the element, the size of global seeds was one. That value was approximately the smallest one, which did not create more than one element in the thickness direction in the meshing. The model B's gasket is in Figure 34 and a close-up in Figure 35.

Figure 34: Symmetry using model B's gasket meshed with Abaqus' gasket elements.

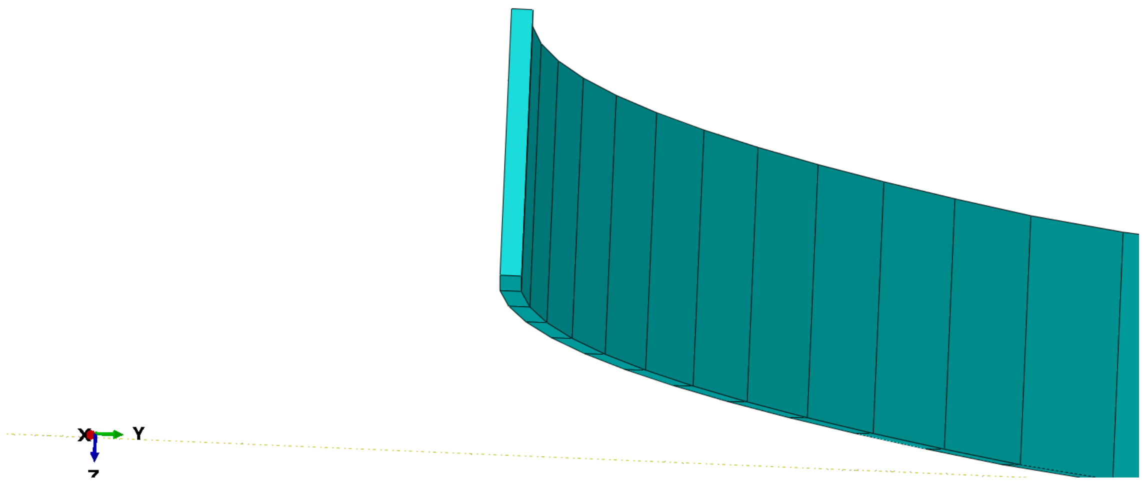

Figure 35: A close-up picture of the model B's gasket model and it's meshing.

5.10 Running FE Simulation

In this work running analysis was done in both Abaqus/CAE and Abaqus' command line window. For model A it was necessary to use restarts to avoid the analysis to start the run from the beginning for each try. The full running time with model A was approximately 9 hours. Model B was remarkably quicker to process even with a computer with lower processor capacity. Model B was done in 30 minutes from start to end.

5.11 Reviewing Results in Abaqus

Results of the case study's analysis are shown in this chapter. For results, models A and B are compared in the middle of the last, the fourth step. Models are compared in that point because model A's job run stopped during that step time. Model B's stresses in screw towers and deformations of plastic shells are shown.

5.11.1 Comparison of Models A and B in Fourth Step

When comparing models, A and B in the beginning of the fourth step, there are two clear differences that separates models and their results. The first clear difference is the longitudinal tension in the models caused by squeezing the seal models. In model A that exists and in model B it does not, because the model B's gasket element is affected only by compression in the direction of thickness.

The second clear difference is that the element shapes of the seal models are different. In model A the meshing elements are tetrahedron elements, and in model B there are gasket elements which have material properties given by calculating the squeeze of 2D model of the seal with quad-elements.

Because of earlier mentioned reasons, model A's and model B's results in a step time of 0.6 in the fourth step are different. Model's A and B shells are compared from Figure 36 to Figure 41. The comparison is done by Von Mises stresses of the shells. Compared halves from inside or outside of the shell are shown in the comparison figures. The units of the figures are megapascals (MPa).

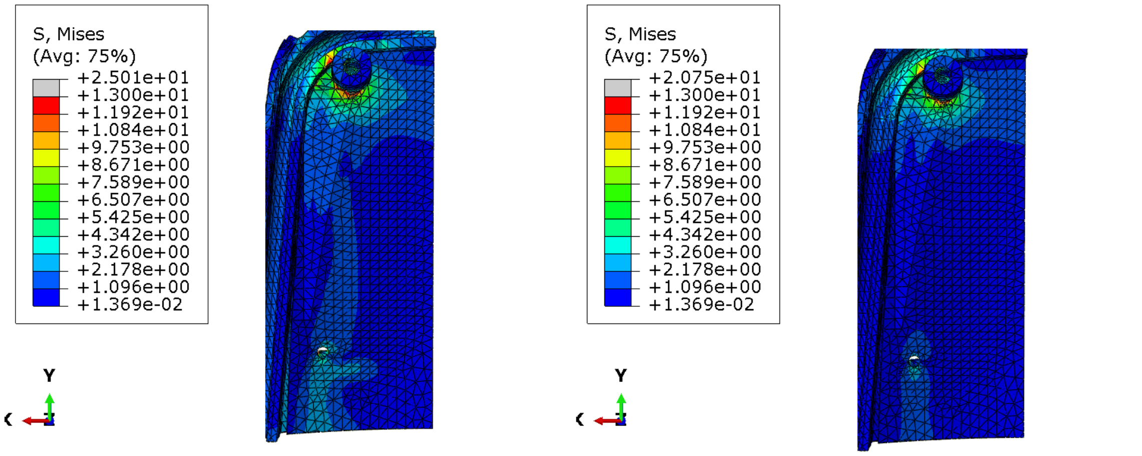

Figure 36: Front shell's first half's inside Von Mises stress graph in models A and B in the fourth step with step time of 0.6.

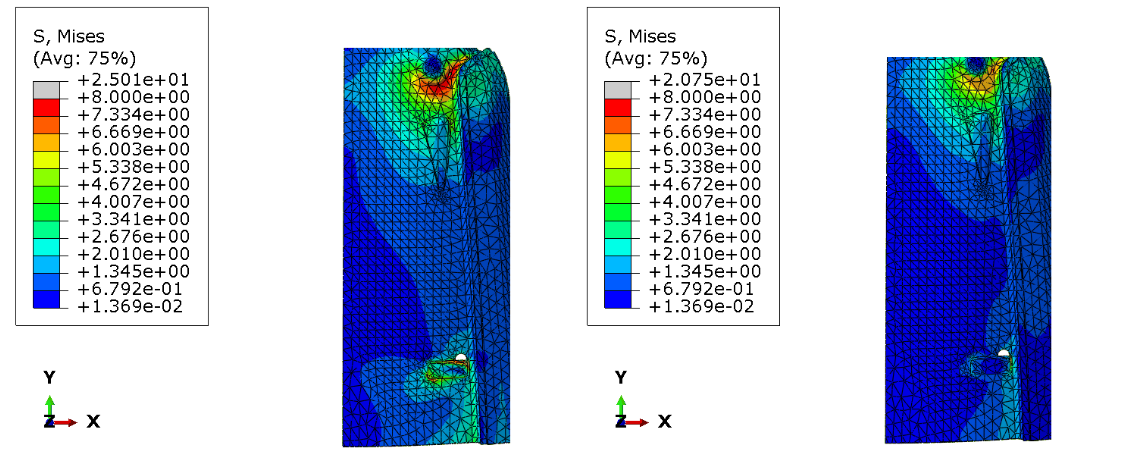

Figure 37: Front shell's first half's outside Von Mises stress graph in models A and B in the fourth step with step time of 0.6.

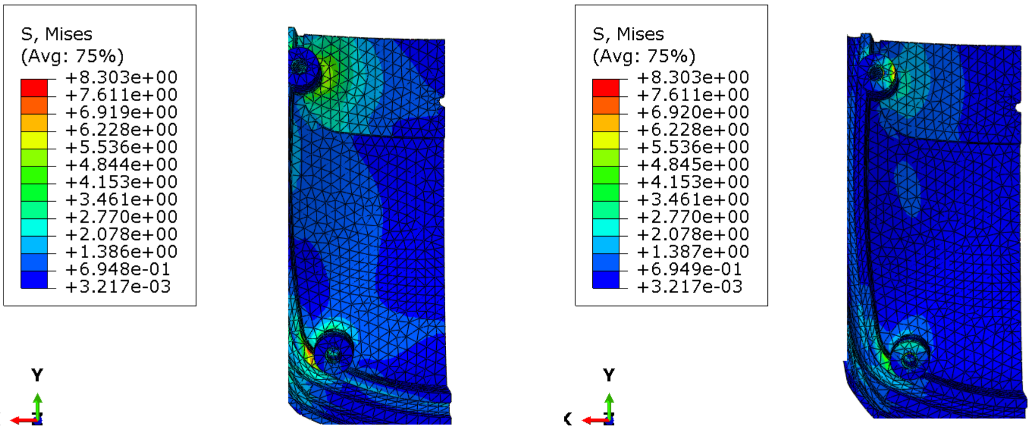

Figure 38: Front shell's second half's inside Von Mises stress graph in models A and B in the fourth step with step time of 0.6.

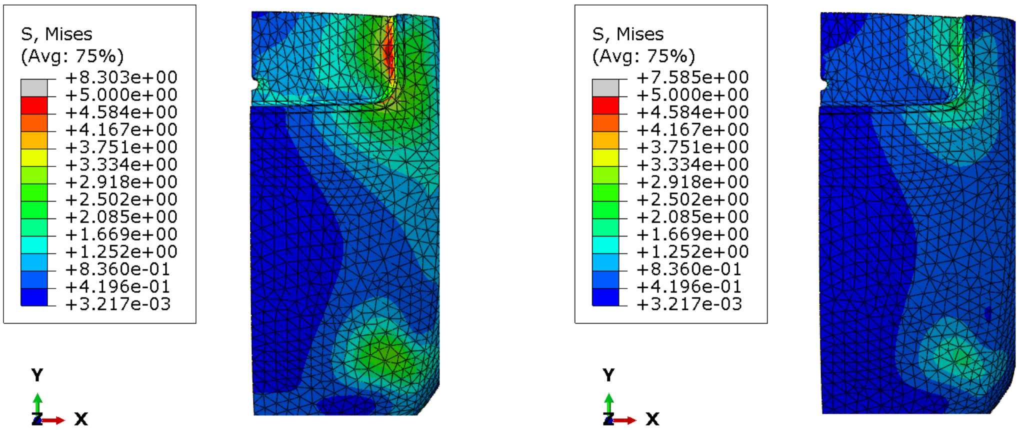

Figure 39: Front shell's second half's outside Von Mises stress graph in models A and B in the fourth step with step time of 0.6.

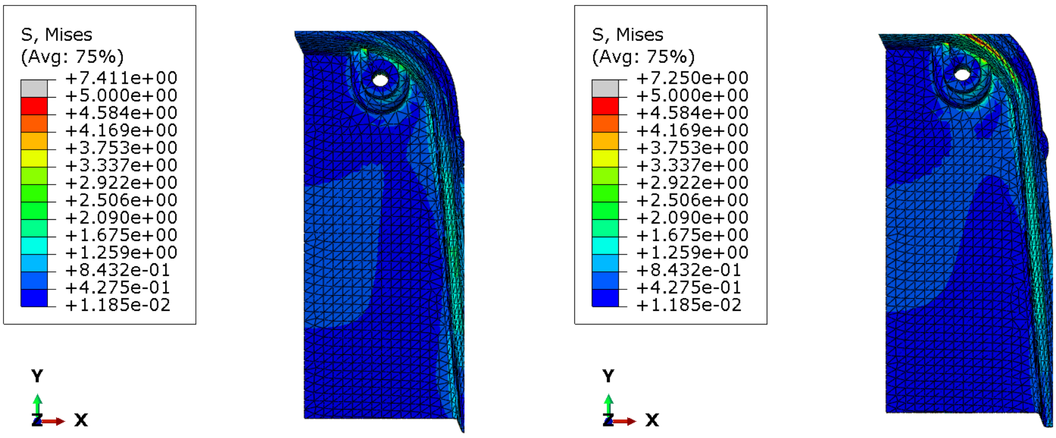

Figure 40: Back shell's first half's inside Von Mises stress graph in models A and B in the fourth step with step time of 0.6.

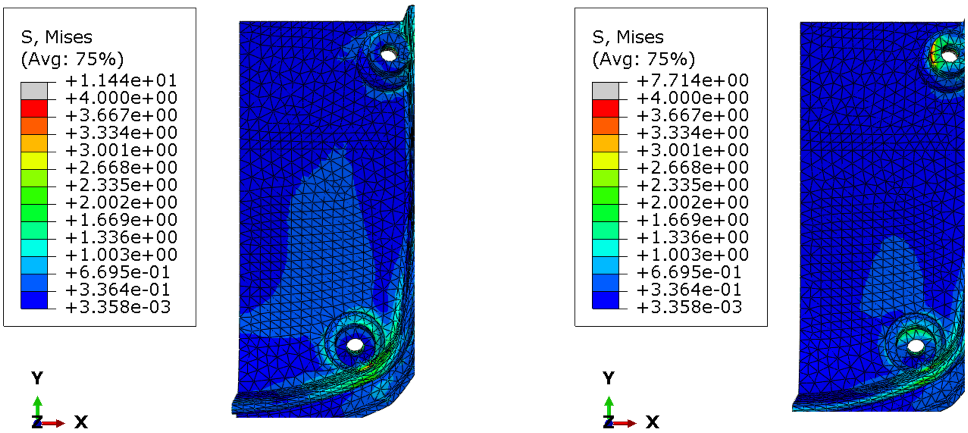

Figure 41: Back shell’s second half’s inside Von Mises stress graph in models A and B in the fourth step with step time of 0.6.

Von Mises stresses of the models are somewhat different. From Figure 38 and Figure 41 it is also possible to see that the middle screw towers are touching each other in model B and not in model A. Still, it is possible to see from this chapter’s figures that model B is good enough for analyzing model’s stresses in screw towers and deformations of plastic shells.

5.11.2 Stresses in Screw Towers

The second part of wanted results (beginning of chapter 5.11) were stresses in screw towers in the end of the simulation. In Figure 42 there are Von Mises stress graphs of the front shell’s graphs, and in Figure 43 there are Von Mises stress graphs of back shell’s screw towers.

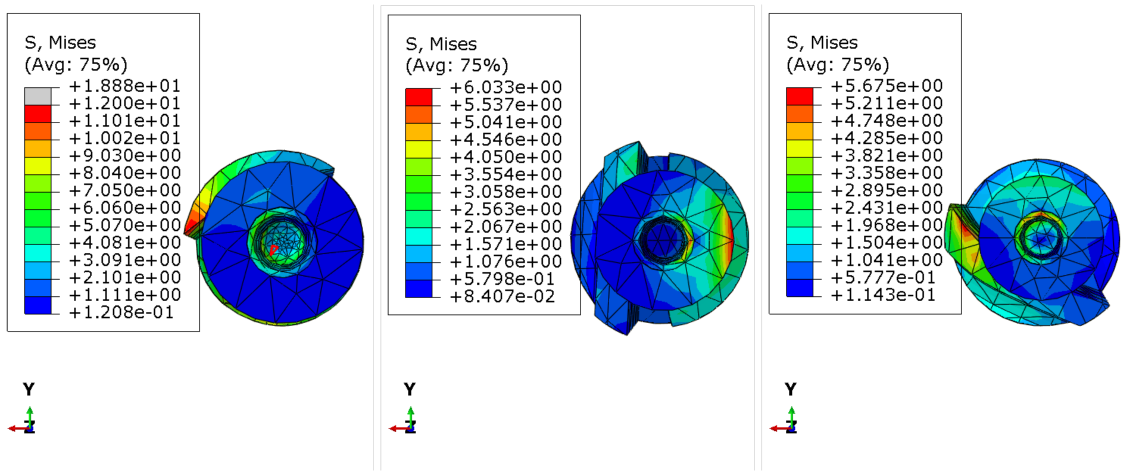

Figure 42: Front shell's screw towers' Von Mises graphs.

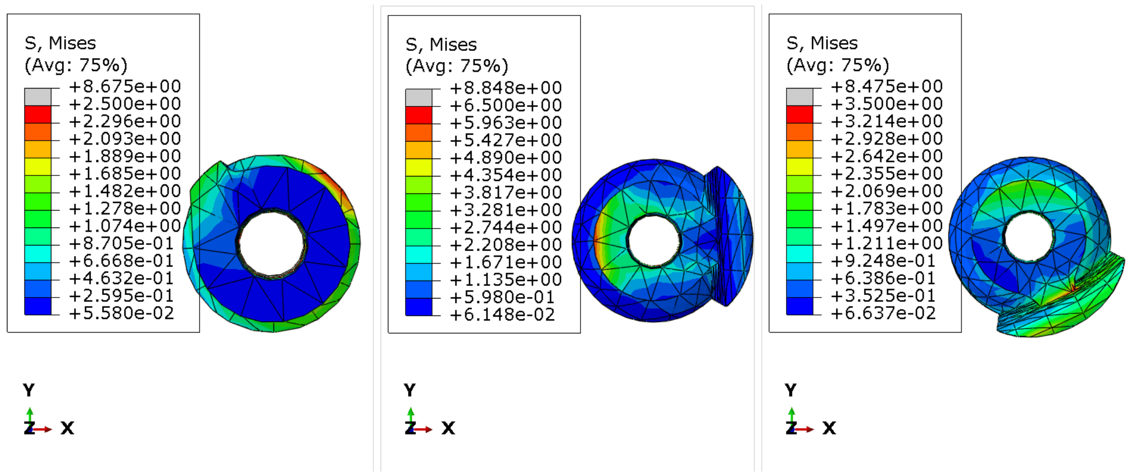

Figure 43: Back shell's screw towers' Von Mises graphs.

5.11.3 Deformations of Plastic Shells

In the figures of this chapter the deformations of the plastic shells are given in logarithmic strains. Strains are in logarithmic form because of the Nlgeom setup. That is because when using Nlgeom setup in Abaqus, step's total strain result is not available. Total strain result gives the strain as an engineering strain. The engineering strain's formula is

$$\epsilon = \frac{L - L_0}{L_0},$$

where L is a new dimension of strain and L_0 is the original dimension of strain. The logarithmic strain's formula is

$$\epsilon_{log} = ln(1+\epsilon).$$

Maximum principal logarithmic strains of the front shell's halves are in the Figure 44 and Figure 45. Maximum principal logarithmic strains of the back shell's halves are in the Figure 46 and Figure 47.

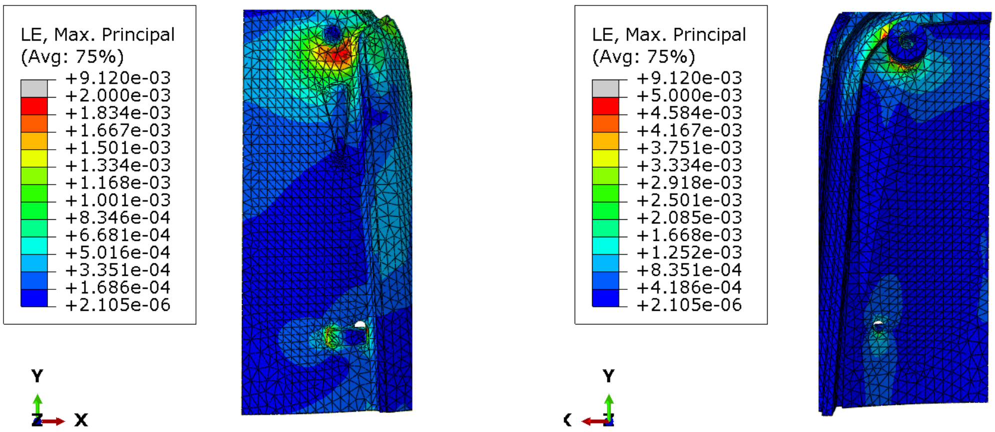

Figure 44: Front shell's first half's maximum principal logarithmic strains.

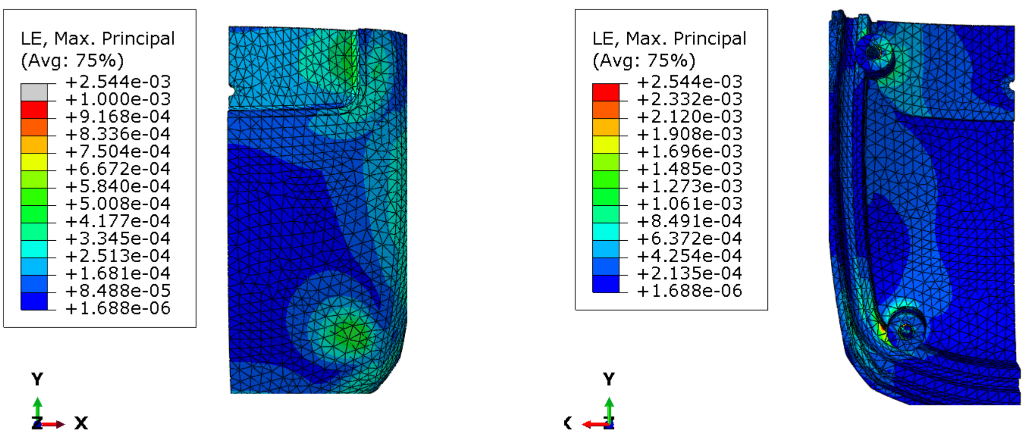

Figure 45: Front shell's second half's maximum principal logarithmic strains.

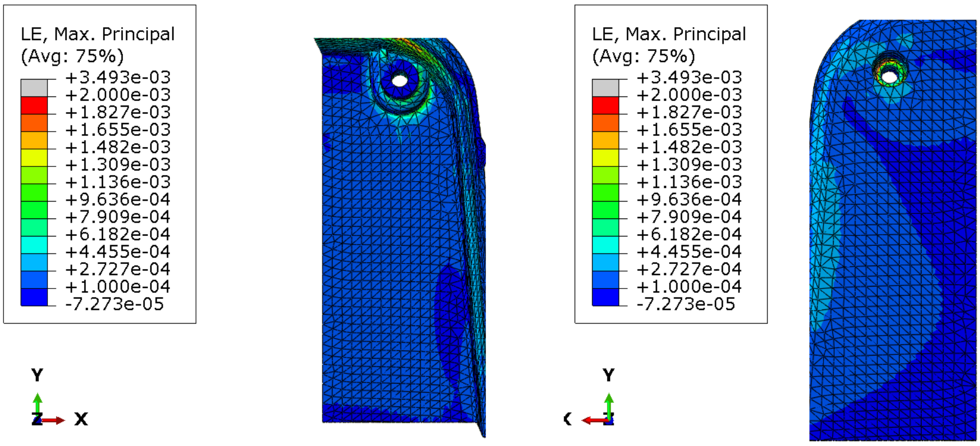

Figure 46: Back shell's first half's maximum principal logarithmic strains.

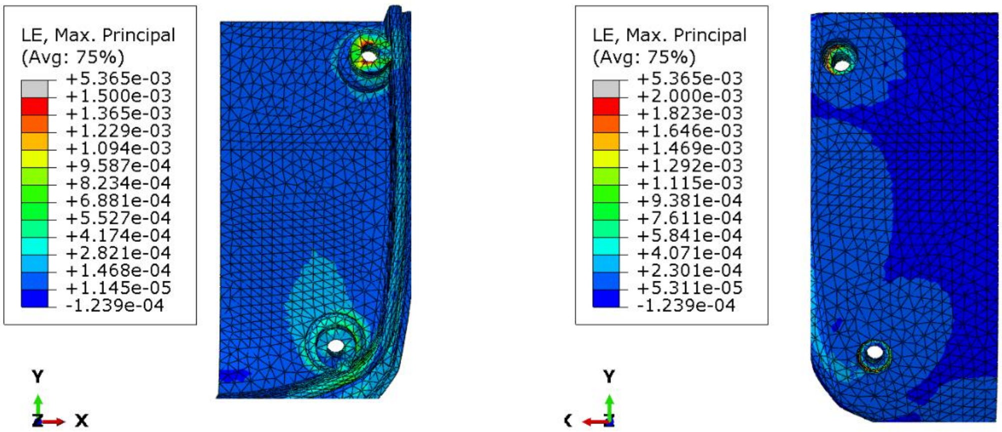

Figure 47: Back shell's second half's maximum principal logarithmic strains.

6 DISCUSSION

The introduction and usage of Abaqus was discussed in chapter 3. In chapter 3.2. the automatic and the manual export with the associative interface were introduced. One big benefit of the associative interface is that CAD tool can update the FEA model directly without breaking the further operations defined in FEA tool. That way, the FEA pre-processing in Abaqus does not need to be done again to the new model. Some of those FEA features that would have to be done again with the other export ways are:

- Assigning material sections
- Defining of parts as assembly instances
- Interactions
- Constraints
- Loads
- Boundary conditions

Some other nice benefits of the associative interface are the possibilities to activate and deactivate components and elements in CATIA and in Abaqus, and to make parameters in CATIA and to update them automatically in both CATIA and Abaqus. Because of the mentioned benefits, the associative feature is very useful in product development where FEA and CAD models had to be developed simultaneously. Also exporting CATIA publications as sets in Abaqus is useful sometimes, because in CATIA it is possible to use model tree for geometries, which is not possible in Abaqus which does not have a model tree for geometries. Without a geometry model tree, sets must always be selected from graphical display in Abaqus.

There was problem with exporting model with the associative interface performing the case study's simulations. By using associative interface, the automatic export did not work, and manual export was slow, and an error occurred before updating the model in Abaqus. There were many attempts to solve the problem. Finally, the problem was in the incompatibility between the associative interface and VPN (virtual private network) tools used in the company. The computer had to be changed because of the incompatibility and the associativity between CATIA V5 and Abaqus had to be researched with the changed computer. As said by the software vendor's representative, the problem should not be generalized, because it is more about incompatibility with the associative interface and data transfer through the VPN used in this work.

In doing the case study many analysis jobs were run in Abaqus. That was because for a start the FEA model's plastic shell models were too detailed with small features for analysis run, cavity seal's geometry was difficult for calculations and the writer of this work did not have much experience with Abaqus before doing the case study. Those reasons lead to many trials and errors with analysis jobs.

The plastic shells got simpler by "cleaning" them in CATIA (chapter 5.1) and halving the shell models as well as the seal models in CATIA and Abaqus (chapters 5.1 and 5.2) for using a symmetry boundary condition. Because of the problem with the cavity seal's geometry, analysis jobs were slow and did not pass the full simulation. Because of that, model B was done with different kind of seal model. Again, because of the writer's poor experience with Abaqus, writer had to be advised many times by the software vendor's representative, and for example ideas of "cleaning" the plastic shell models and replacing the cavity seal with different model were the representative's ideas. Without the representative's help, the case study's simulation would not have passed.

One benefit of Abaqus and the case study was the possibility to use nonlinear material model. In the case study a nonlinear material was set for the cavity seal. Also, the possibility of using the gasket element was helping in doing the case study. A Cavity seal has a geometry that is hard to mesh in a way that simulations would pass with. This is because the cavity seal's wall thickness is very small compared to seal's full size. The element size needed for simulation to pass with the original seal model would have been very small. The number of elements would have been very large. By that, the run time of the analysis job would have been unreasonable long with the computer that was used in simulations.

By changing the original gasket model to Abaqus' gasket element model, the simulation passed. The run time of the simulation with the last model was about half an hour, which was a remarkably short simulation time compared to model A's simulation run time.

7 CONCLUSIONS AND RECOMMENDATIONS

This work had four research questions which were defined in chapter 1.1. Those questions were:

- How to use CATIA V5 – Abaqus associative interface and Abaqus in product development?
- How usable CATIA V5 - Abaqus associative interface and Abaqus are?
- What requirements using the interface and Abaqus sets for the user?
- What requirements using the interface and Abaqus sets otherwise?

The use of the associative interface with Abaqus is discussed in chapter 3. Based on chapter 3 and the discussion (chapter 6), the associative interface can be mentioned to be very useful with Abaqus. There were three remarkable advantages of using the associative interface compared to the using of CATIA and Abaqus without this associativity. The first remarkable advantage was the possibility to export a CATIA part or a CATIA product during the FEA model's pre-processing in Abaqus, without having to do whole pre-processing again after the export. This means work and time savings. The second remarkable advantage was the possibility to use CAD parameters and to update them in both associative interface and Abaqus. The third remarkable advantage was the possibility to activate and deactivate geometric features from the associative interface and update the features from the associative interface to Abaqus. There are no remarkable requirements with the use of the associative interface. Anyway, it can be used easily without special skills or an expert.

In discussion it was mentioned that one nice benefit of Abaqus is the possibility of using nonlinear material model. That feature is needed when modelling for example hyperelastic material like the cavity seal material. Similar situation will occur with any nonlinear parameter or with statically not defined structures.

As mentioned in discussion, the case study would not have got passed the simulations without help of software vendor's representative. With help from the representative and comprehension got by that, the use of Abaqus got much easier. Based on that, Abaqus can be thought to be more an expert level program than a general level program. Because of the amount of necessary comprehension for doing the case study, the user can be said to need training for using Abaqus. Comprehension was needed that much basically because the program was not self-guided in other ways but by a datacheck feature, and the pre-processing had many phases. Also, in complicated cases a powerful computer is needed.

An exporting problem with the associative interface's automatic export and VPN was also mentioned in discussion. The problem of incompatible VPN, that occurred, was a special problem in this case and it cannot be generalized according to the software vendor's representative.

8 SUMMARY

The subject of this master's thesis was connecting CATIA V5 and Abaqus in product development. At the beginning four research questions related to using of CATIA V5 – Abaqus associative interface and Abaqus were defined. Also, the motivation of the work was told to be about Bittium company wanting to research if the associative interface and Abaqus are suitable for their use. Contents and definition of the work was said to use CAD and FEA together, and the case of this work to use CATIA V5 and Abaqus together.

The methods of this work were CAD, FEA and CAD and FEA combined. CAD was introduced, and some features were given. FEA was also introduced, the use of it in companies was discussed. FEA process was discussed, and 3D FEA example was given. CAD and FEA combined was compared to a separated use of CAD and FEA, and CAD and FEA combined in product development were discussed.

CATIA V5 – Abaqus associative interface and Abaqus were introduced, and their usage was handled. After that, there was a case study in this work. The case study was about a situation with a housing that consisted of two plastic shells and a cavity seal squeezed between them. Because the case study did not pass the FEA process with the first model, the second model was created. The first model was named as model A and the second model as model B. When the model A's cavity seal model was realistic, the model B's cavity seal was replaced by a simplified gasket with rectangle cross section and consisted of Abaqus' own gasket elements. Model B passed the simulations. In results review the models were compared and the results of model B were analyzed.

In discussion the benefits of the associative interface with Abaqus, an export problem with the associative interface, the need of an expert for the use of Abaqus and some benefits of Abaqus in the case study were analyzed. As conclusions in chapter 7, the associative interface does not need special skills or experience for use, and Abaqus can be recommended for the company if the designers have comprehension and training for its use.

REFERENCES

ABAQUS, inc, 2006a. ABAQUS/CAE User's Manual, 17.16.2 Choosing an element shape. [web document]. Rising Sun Mills: ABAQUS, inc. Available: https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usi/default.htm?startat=pt03ch17s16s02.html [referred 19.9.2022].

ABAQUS, inc, 2006b. ABAQUS/CAE User's Manual, 48.2.1 What is a set? [web document]. Rising Sun Mills: ABAQUS, inc. Available: https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usi/default.htm?startat=pt06ch48s03hlb02.html [referred 9.9.2022].

Bittium Company, 2022. About Bittium, Facts & Figures, Company Overview [web document]. Oulu: Bittium Company. Available: https://www.bittium.com/about-bittium/facts-figures/company-overview [referred 16.3.2022].

Dassault Systèmes, 2010. Abaqus analysis user's manual, Beam modelling: overview. [web document]. Vélizy-Villacoublay: Dassault Systèmes. Available: http://130.149.89.49:2080/v6.10ef/books/usb/default.htm?startat=pt06ch26s03abo24.html [Referred 14.9.2022]

Dassault Systèmes, 2014a. DS Simulia Abaqus 2016, Defining sections [web document]. Vélizy-Villacoublay: Dassault Systèmes. Available: http://130.149.89.49:2080/v2016/books/usi/default.htm?startat=pt03ch12s02s03.html [Referred 19.9.2022]

Dassault Systèmes, 2014b. DS Simulia Abaqus 2016, File extensions used by Abaqus [web document]. Vélizy-Villacoublay: Dassault Systèmes. Available: https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usb/default.htm?startat=pt01ch03s05aus29.html [referred 25.8.2022].

Gade, S., 2020. Blog, Using Publications in CATIA V5. [web document]. Bengaluru: EDS Technologies. Available: https://edstechnologies.com/blog/making-assemblies-easier-with-publications-in-catia-v5/ [Referred: 13.9.2022]

Gallagher, R. H., 1975. Finite element analysis – Fundamentals. New Jersey: Prentice-hall, Inc., 420 p. ISBN 0-13-317248-1.

Hietikko, E., 2021. Palkki – Lujuuslaskennan perusteet. 4. Painos. Helsinki: BoD - Books on Demand, 218 p. ISBN 9789528043591.

Hietikko, E., 2015. Tuotekehitystoiminta. 3. Painos. Helsinki: BoD - Books on Demand, 203p. ISBN 978-952-330-065-1.

Huebner, K. H., Thornton, E. A. & Byrom, T. G., 1995. The Finite Element Method For Engineers. 3. Edition. New York: John Wiley & Sons, Inc., 627 p. ISBN 0-471-54742-5.

Holstein, W, K., 2022. Britannica, research and development [web document]. Chicago: Encyclopædia Britannica, Inc. Available: https://www.britannica.com/topic/research-and-development [Referred: 10.9.2022]

Imaginationeering, 2021. Blog, Benefits of Simulation-Driven Product Development [web document]. Houston: Imaginationeering, LLC. Available: https://www.imaginationeering.com/benefits-of-simulation-driven-product-development/ [Referred: 10.9.2022]

Kenton, W., 2022. Investopedia, Research and Development (R&D) Definition, Types, and Importance [web document]. New York: Dotdash meredith. Available: https://www.investopedia.com/terms/r/randd.asp [Referred: 10.9.2022]

Krebs, J., 2007. Basics CAD. Basel: Birkhauser. 92 p. ISBN 978-3-0356-1274-5

Kuusisto, E., 2017. SHELLS vs. SOLIDS | Finite Element Analysis Quick Review. [web document] Sunny vale, CA: Linkedin. Available: https://www.linkedin.com/pulse/shells-vs-solids-finite-element-analysis-quick-review-kuusisto-p-e-# [Referred 14.9.2022].

Liam, T., 2021. Grindstone, R&D For Product Development – How Does It Work? [web document]. London: Grindstone Capital. Available:

https://grindstonecapital.co.uk/rd-for-product-development-how-does-it-work/
[Referred: 10.9.2022]

Madier, D., 2020. Practical Finite Element Analysis For Mechanical Engineers. 1. Edition. Montreal: FEA Academy., 639p. ISBN 978-1-9990475-0-4.

Obbink-Huizer, C., 2019. Simuleon FEA Blog, How to restart an Abaqus analysis. [web document]. PP's-Hertogenbosch: Simuleon B.V.. Available: https://info.simuleon.com/blog/how-to-restart-an-abaqus-analysis [Referred: 13.9.2022]

Salmi, T. & Kuula K., 2012. Rakenteiden Mekaniikka. Tampere: Presseus OY., 464p. ISBN 978-952-9835-83-6.

Ulrich, K. T. & Eppinger S. D., 2008. Product Design and Development. 4. Edition. New York: The McGraw-Hill Companies, Inc., 368p. ISBN 978-007-125947-7.

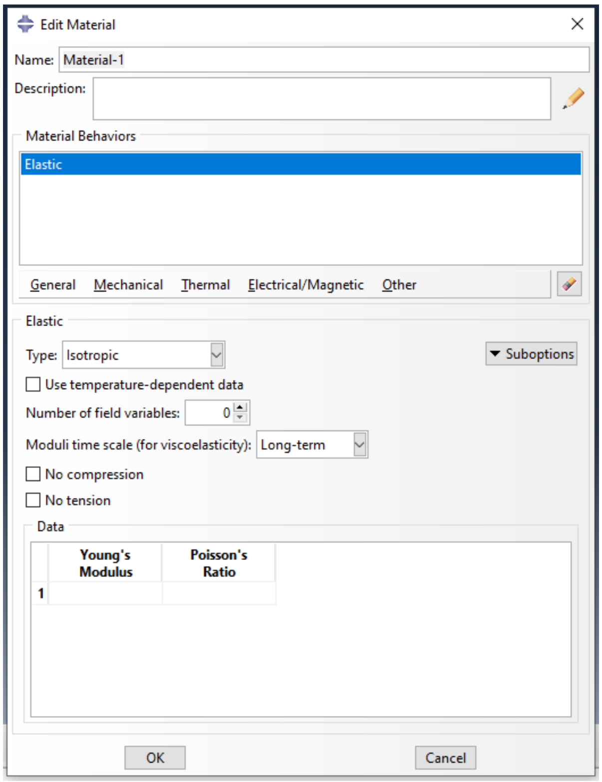

Figure 48: Edit Material -window.

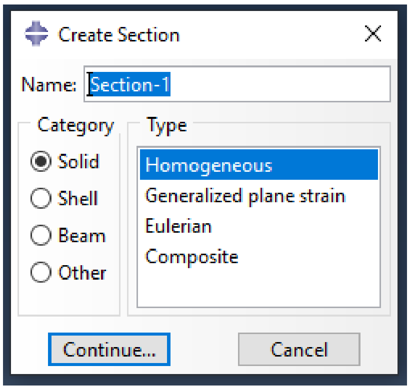

Figure 49: Create section -window.

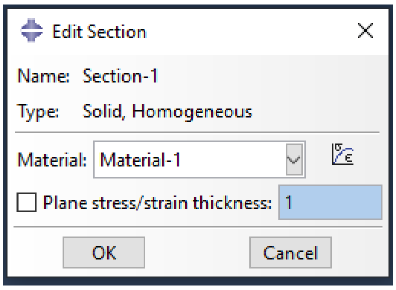

Figure 50: Edit section -window.

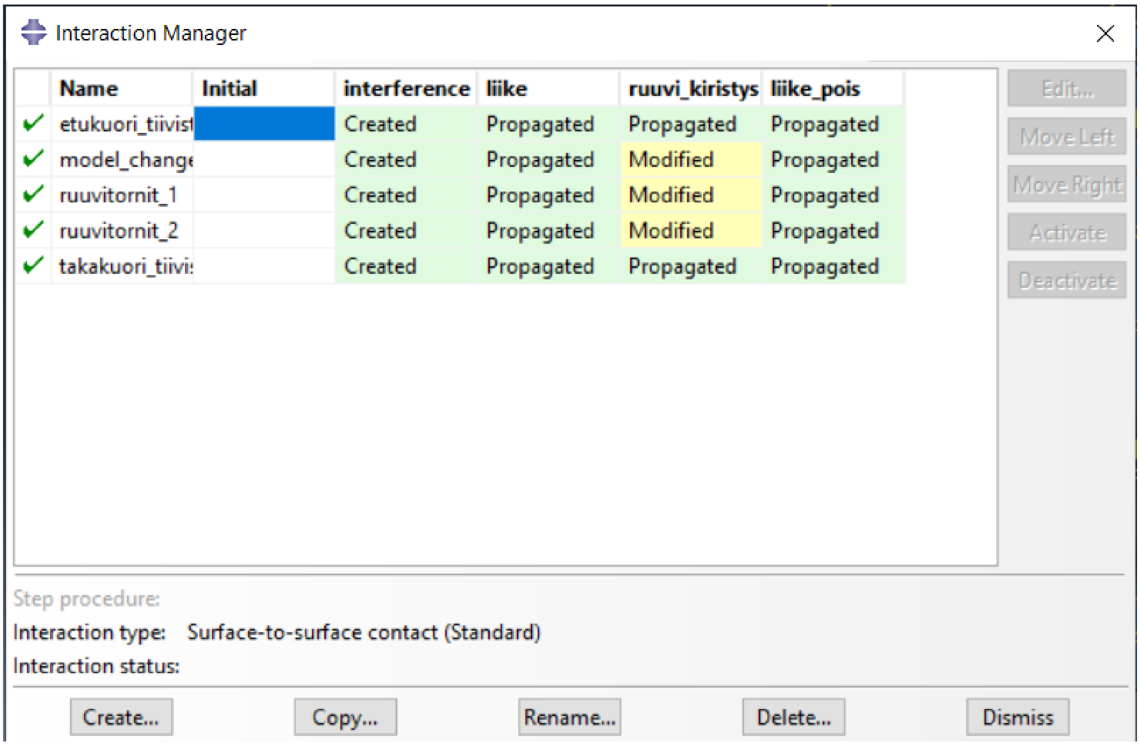

Figure 51: Interaction Manager.

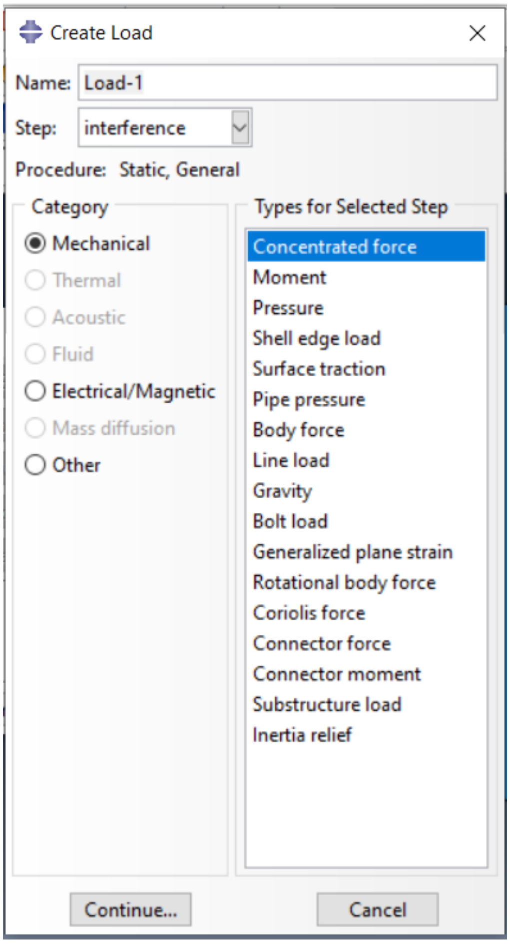

Figure 52: Create Load -window.

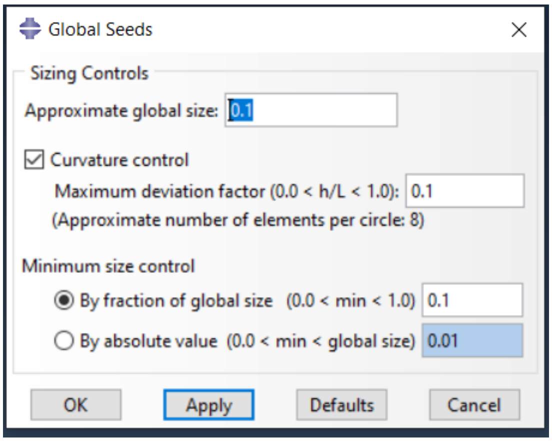

Figure 53: The global seeds -window.

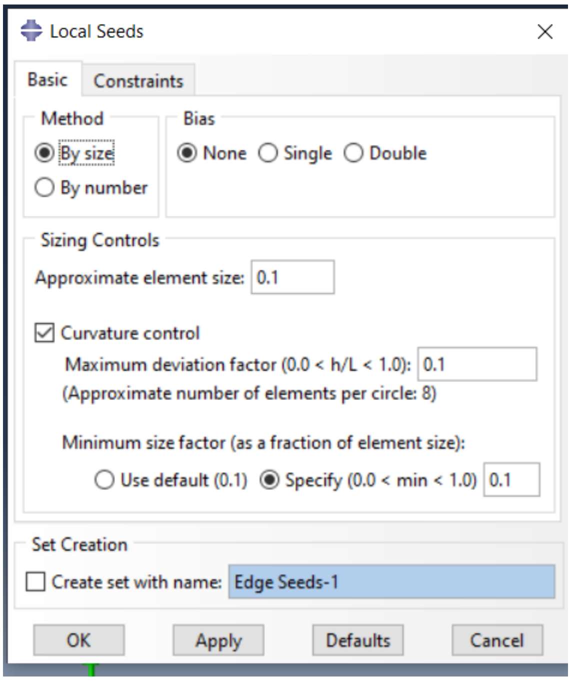

Figure 54: Local Seeds -window.

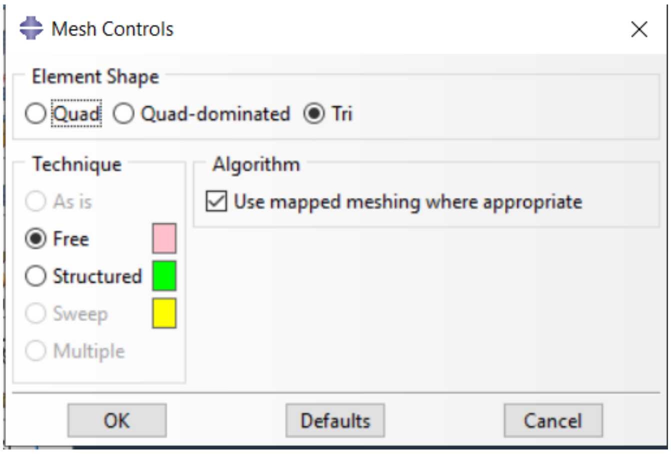

Figure 55: Mesh Controls -window.

Creating an input

Input, or INP file, is a file which contains all data for calculations. Input file contains all information of the analysis and is processed by the solver. Input fle is done always before simulation in Abaqus. INP file can be done in Abaqus/CAE by data check, submitting job or separately. Some benefits for doing INP file separately are the possibility to do revisions by changing model quickly from INP file in text editor, and the possibility to launch simulation by using command line (Dassault Systémes, 2014).

Creating a restart

In Abaqus restart, there is a way to continue an FEA from any step or frame without having to start it from beginning. For example, with an analysis which has 4 steps, it is possible to continue the job in the second run from the middle of third step with Abaqus's restart. By doing that, the first run's results remain in the start of new results, and results from the second run continue the starting results. For a restart following steps are needed:

- Original job done with restart requests
- New model which has setups for restart from model's attributes, and
- A job which has restart as a job type.

Restart requests for original job are set from the step module via path **Output** and **Restart requests(…)**. From there it is possible to set restart times by frequencies or intervals. For example, frequency of two in restart request means that restarting points are in every second increment of the step, and intervals of ten means there are ten restart points in the step that are spaced equally by same time. If switching overlay option on, the new data will overwrite the old data. If switching time marks on, the output will be written in exact time corresponding the given intervals. The Edit Restart Requests -window is in Figure 56 (Obbink-Huizer, 2022).

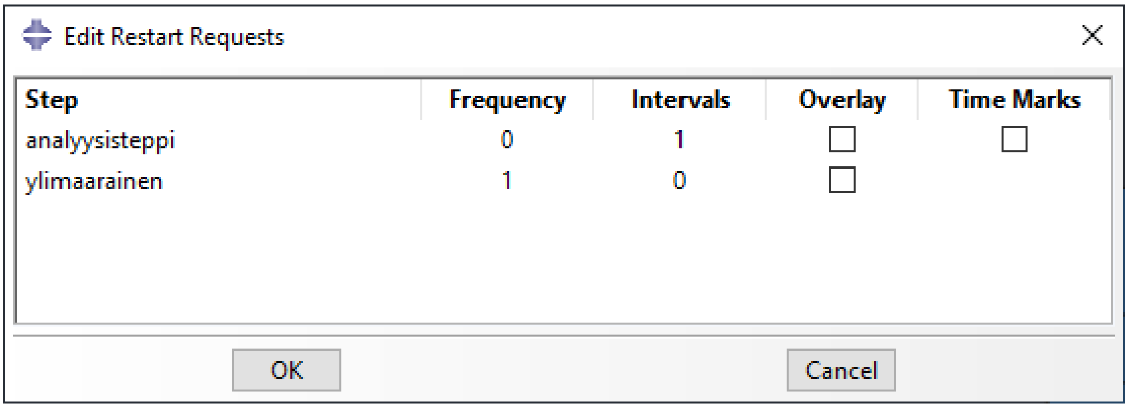

Figure 56: Edit Restart Requests -window.

A new model's setups are done from path **Model**, **Edit attributes** and **model's name**. The read data from job option must be switched on and the original job's name has to be given to the field following the option. Also, the name of restart's starting step and the starting point of the step must be given. The Edit Model Attributes -window is in Figure 57. Finally, a job type must be set as a restart when creating a job.

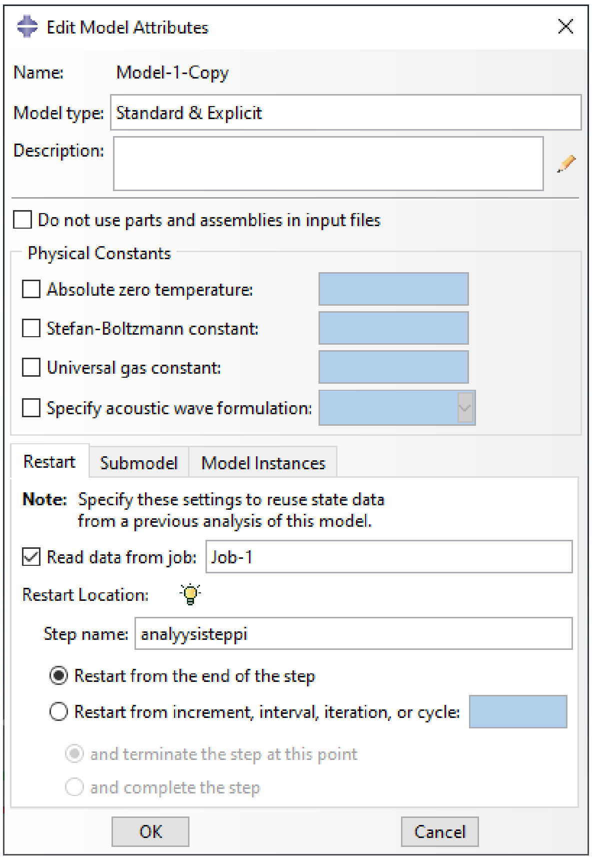

Figure 57: Edit Model Attributes -window.