OULUN YLIOPISTO

FACULTY OF TECHNOLOGY

EFFICIENT USE OF CREO SIMULATE AS A PART OF MECHANICAL PRODUCT DESIGN PROCESS

Terho Iso-Junno

DEGREE PROGRAMME OF MECHANICAL ENGINEERING

Master’s thesis 2017

ABSTRACT

Efficient use of Creo Simulate as a part of mechanical product design process

Terho Iso-Junno

University of Oulu, Degree Programme of Mechanical Engineering

Master’s thesis 2017, 100 p. + 5 p. appendixes

Supervisor: Toni Liedes

In this thesis, how Creo Simulate can be used efficiently during a mechanical product design process is studied. The purpose of this thesis was to produce simulation training material to be used at Nokia. The aim was to create material which would be accessible for all of Nokia’s mechanical designers who could study strength simulation and perform strength simulations on their designs. With strength simulations it is possible to reduce the design time required for the new parts since a simulated part should have the right strength properties already in first prototype. As a result of this thesis five training videos and a short Simulate workflow guide were produced for Nokia.

For this thesis, the *finite element method* and the features available in Creo Simulate from the point of view of the mechanical engineer were studied. The researched information was gathered into this thesis to create a simulation guide for a new Simulate user. This thesis covers static strength analyses and design studies. Two case examples are included in order to clarify the simulation process with Simulate for the new users.

Keywords: finite element method, Creo Simulate, strength simulation

TIIVISTELMÄ

Creo Simulaten tehokas käyttö osana mekaanisen tuotteen suunnitteluprosessia

Terho Iso-Junno

Oulun yliopisto, Konetekniikan koulutusohjelma

Diplomityö 2017, 100 s. + 5 s. liitteitä

Työn ohjaaja: Toni Liedes

Tässä työssä on tutkittu kuinka Creo Simulatea voidaan käyttää tehokkaasti osana mekaanisen tuotteen suunnitteluprosessia. Työn tarkoituksena oli tuottaa Nokialle koulutusmateriaalia Simulaten käytöstä. Tavoitteena oli että kaikki Nokian mekaniikkasuunnittelijat voisivat opiskella lujuussimulointia ja suorittaa niitä tarvittaessa suunnittelemilleen osille. Lujuussimuloinneilla on mahdollista vähentää tarvittavaa suunnitteluaikaa uuden tuotteen kehittelyssä, koska simuloiduilla osilla pitäisi olla oikeanlaiset lujuusominaisuudet jo ensimmäisessä tilatussa prototyypissä. Työn tuloksena tehtiin Nokialle viisi koulutusvideota ja lyhyt pikaopas simulointiprosessin vaiheista.

Työn aikana tutkittiin *elementtimenetelmän* teoriaa ja Creo Simulaten ominaisuuksia mekaniikkasuunnittelijan näkökulmasta. Tutkitusta materiaalista on koostettu tämä diplomityö siten, että se muodostaa aloittelevalle Simulaten käyttäjälle soveltuvan lujuussimulointioppaan. Tässä työssä on käsitelty vain staattinen lujuussimulointi Simulatella. Työhön on sisällytetty myös kaksi esimerkkitapausta selkiyttämään simulointiprosessin vaatimaa kokonaisuutta aloitteleville käyttäjille.

Asiasanat: elementtimenetelmä, Creo Simulate, lujuussimulointi

PREFACE

This Master's Thesis is done for Nokia during my seasonal trainee position at Nokia in the summer of 2016. I want to thank my superior Tarmo Korva who made this work possible and my supervisor Toni Liedes. Thanks for Iikka Finning, Juho Tuovila, Markku Leskelä, Hannu Kärppä and Åvist Matti for reviewing and giving feedback. Thanks also for Annukka Tyni and Veikko Niskasaari for testing the training material.

Special thanks for Lassi Keränen, who has been trusty coffee break mate during the writing process and has provided advice and additional material for my work. Thanks ngfor Mirjami Alanko for helping with linguistic challenges.

Oulu, 04.01.2017

Terho Iso-Junno

TABLE OF CONTENTS

SYMBOLS AND ABBREVIATIONS

C	failure compression stress value
c	independent constant
$\mathbf{D}$	Elasticity matrix
e	error
$\mathbf{F}$	vector of forces
h	characteristic element length
$\mathbf{K}$	stiffness matrix
l	length
p	rate of convergence
T	failure tension stress value
$\mathbf{U}$	displacement matrix
u	displacement

ε	strain
$\boldsymbol{\varepsilon}$	strain matrix
σ	stress
$\boldsymbol{\sigma}$	stress matrix

DF	degree of freedom
FEM	finite element method
R&D	Research & Development

1 INTRODUCTION

The mechanical part design process aims to produce parts that fill given requirements, while maintaining low part costs and efficient usage of material. The process starts with innovating a solution with the needed features for a given task and then modeling that with 3D modeling software. The quality of the created solution is based on designer experience and it will be usually tested by ordering a prototype of the designed part.

Creo Simulate is a tool used to study the quality of a designer solution from view of mechanical strength. It is a simulation tool that uses the *Finite Element Method* (FEM) to calculate stress levels, displacements and strains occurring in a part with a given load. A FEM simulation of a parts mechanical strength helps to avoid both too robust and too fragile solutions. It helps to make better a solution for the first prototype part and leads to shorter R&D process time.

Creo Simulate is one product in Nokia's mechanical design tool software. Simulate is an add-on package to the 3D modeling software Creo Parametric, used also by Nokia. Simulate is available for all mechanical designers at Nokia but most of them do not have the skills and knowledge to use Simulate. In this Master's thesis, made for Nokia, how to use Creo Simulate efficiently as a part of mechanical part design process is studied. The aim is to produce a simple guide material for Nokia's mechanical designers so that they can make strength analysis for their own parts and create better parts even without years of design experience.

There is already available a simulation tutorial for Creo Simulate 3.0 (Toogood 2015), where most features of both structure and thermal simulations of Simulate are widely presented. In this Master's Thesis, the usage of Simulate only from the needs of mechanical designer will be presented. This guide is also created for designers who are unfamiliar with FEM simulations, therefore the basic theory and principles behind FEM calculation will be covered. The whole simulation process is presented step by step and the purpose of each step is explained. Compared to Toogood's tutorial, this thesis gives more detailed information from a narrower area, and the usage of the Simulate is demonstrated with real case examples from Nokia.

In this thesis, what settings and what kind of simulations can be used in the Simulate are studied. The Study is done by examining Creo Simulate's online manual (PTC 2016a) and running a countless amount of test simulations. Some of the performed test simulations are presented in the thesis, to allow the reader to also understand the capabilities of the studied features better. Recommendations for many settings and options based on the studies and test simulations made during the thesis work are given. A couple of ways to ease and speed up the simulation process is also included.

2 INTRODUCTION TO FINITE ELEMENT METHOD

The basic principle behind FEM is dividing complex continuing problems into smaller discrete parts and solving those parts to find approximation of the accurate solution for the problem. FEM includes all the necessary steps from discretization to solving results. It is a procedure in which a complex mathematical problem is represented with multiple simple equations. This procedure can be automatized so FEM suits really well for solving complicate problems with computers.

First applications of the idea behind FEM were used in aircraft structural analysis. In 1941 Hrenikoff introduced the framework method in which the collection of bars and beams were used to model a part of a plane. Another early name behind the idea was Courant in 1943, who used triangular elements in his works. However, the creation of the *finite element method* is formally thought to have been done by Turner, Clough, Martin and Topp in 1956 and Argyris and Kelsey in 1960. The term *"Finite element"* was first time brought up by Clough in 1960. After that the development of the FEM has been done much and there are countless applications and use cases for the method. (Reddy 1993: 5).

2.1 FEM for static structural analysis

The FEM in structural analysis can be summarized into the following steps:

1) Geometry is divided into *finite elements* by lines or surfaces. Elements are assumed to be interconnected only through *nodes* located usually on element corners or edges. Together those connected elements are creating *finite element mesh.*

2) There is selected a set of *shape functions* which are used to approximate displacement within each finite element based on its nodal displacements. Displacement of nodes is the unknown parameter of the structural analysis.

3) All forces affecting in nodes are determined as many equations as there are unknown values are created. As a result a stiffness relationship between nodal forces

and nodal displacements is created for the whole structure (Eq. (1)). When equations are solved, there will be found displacement values for each node.

4) *Shape functions* are now defining also the unique state of strain in each element based on its nodal displacements. That strain can be used to calculate stress through the element when combined with initial strain and material properties. (Taylor & Zienkiewicz 2000: 18; University of Oulu 2014: 14).

2.1.1 Finite elements

Meshing is a procedure in which geometry is divided into *finite elements* by imaginary lines or surfaces. There are different types of elements for different geometries and purposes. Elements are assumed to be interconnected through points called *nodes* located usually on element corners or edges (Taylor & Zienkiewicz 2000: 18). Figure 1 is an example of a solid tetra element mesh. Nodes are located in the corners of the elements.

There are two main reasons for dividing geometry into *finite elements*. First is to represent the geometry of the examined component. Second is to represent the solution over the whole geometry by approximating it over each element of the mesh. (Reddy 1993: 72).

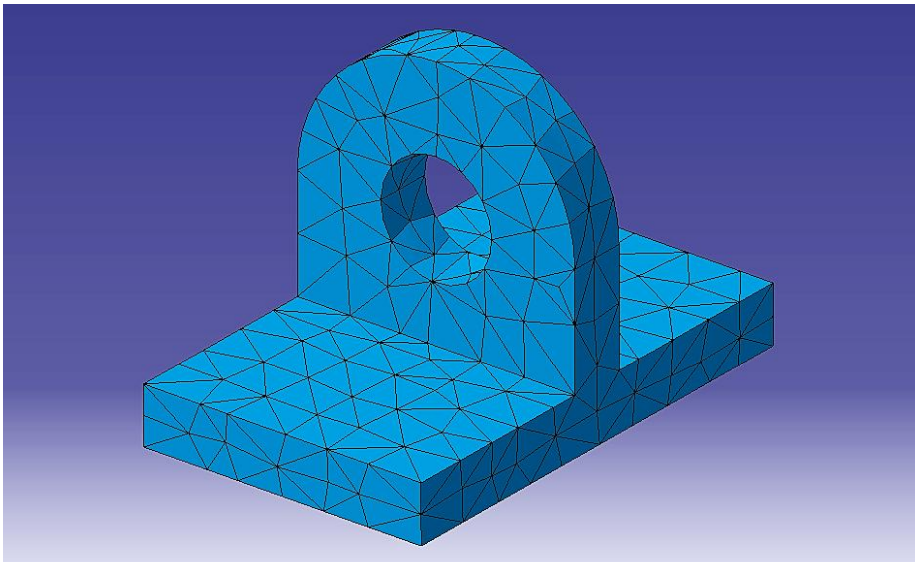

Figure 1. Example of a solid tetra finite element mesh.

Displacement of the *nodes* is the unknown parameter of the structural problem. There are a total of six possible types of displacements called *degrees of freedom* (*DF*) for the single *node*. Those are translational movements along three axis and rotations around same three axis. Different types of elements can have different amount of DF in nodes.

In Figure 2, a couple of different elements used in FEM are shown. There are truss element and beam element on the first line. The truss and beam element both have two nodes and one line connecting those. Difference in elements is in DF defined for the elements. Both have translational freedoms but the beam also has rotational freedom so it can also convey bending loads. The amount of translational freedoms depends on whether the element is defined for 2D or 3D calculation.

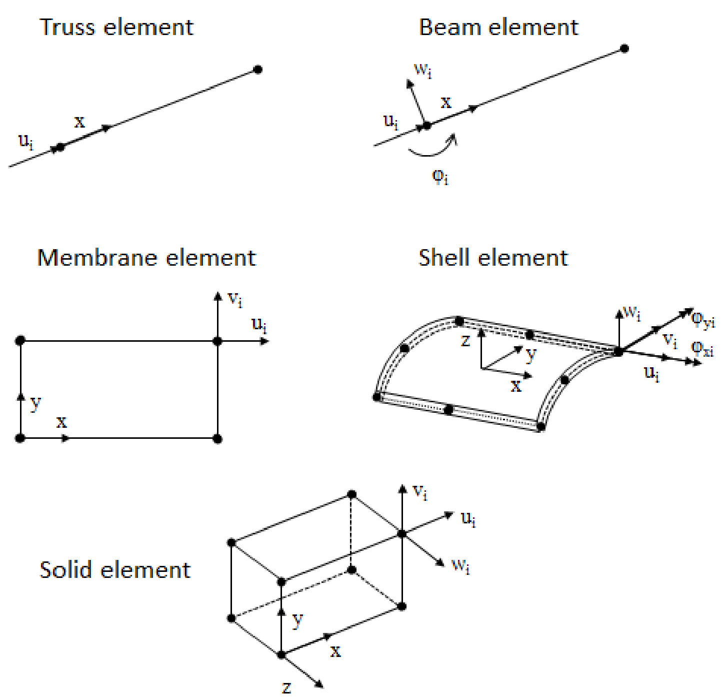

Figure 2. Different type of elements used in FEM (University of Oulu 2014: 170).

Membrane and *shell elements* are shown on the second line of Figure 2. Those are used to calculate thin membranes and plates. The *Membrane element* does not have rotational DF so it cannot convey any bending loads similar to truss element. The s*hell element* has rotational freedoms around the axis along the shell surfaces so it can be used to calculate stiff thin structures. The *Membrane element* in the picture is a so called *linear element*, it has *nodes* only in the corners. The shell element has *nodes* also in the middle of the edges so it is a *quadratic element*.

At the bottom in Figure 2 is a linear solid element. Solid elements have three translational DFs in each node. It is possible to have different shaped solid elements. Commonly used shapes are bricks shown in Figure 2, tetras (Figure 1) and wedges. The solid element differs from the others in the way that it can fill volume. Because of that it suits well when FEM simulating 3D geometries created with 3D design tools.

There are a couple of requirements for the mesh so that it can give accurate results. First, mesh should represent the problem geometry accurately. It also should to be able to represent possible large gradients in the solution. Basically, these two requirements can be ensured by using elements that are small enough. The third requirement is that there should not be used elements with unacceptable geometries, which can distort results. (Reddy 1993: 442)

Element geometry validity can be measured with the angle between two sides. Angle is not allowed to be 0° or 180°. For the element to work well the corner angle should be between 15° and 165° as a general guideline. (Reddy 1993: 441) In Figure 3 examples of elements with too large or too small corner angles are shown. Angles over 180 degrees are also not acceptable.

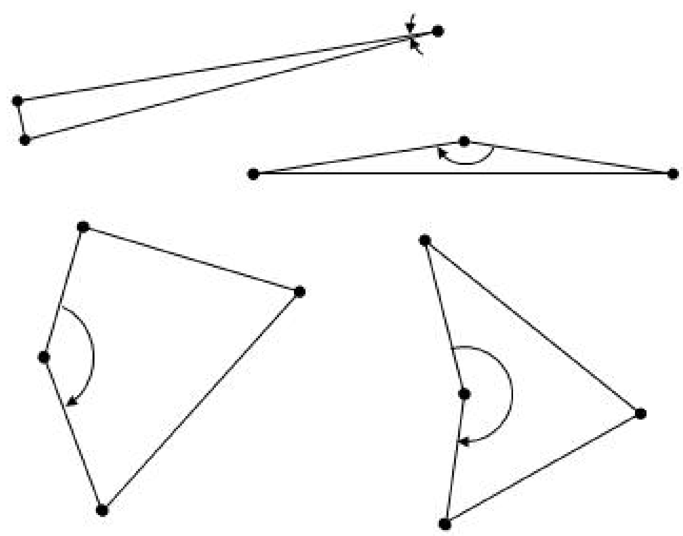

Figure 3. Unacceptable 2D element geometries with too large or small angles in corners. (Reddy 1993: 440).

2.1.2 Shape functions

Exact displacement solutions as exact functions for simple beam load cases have been calculated. Solutions for different kinds of cases have been then collected for example into books of Mäkelä *et al.* (2010: 147 - 150) and Airila *et al.* (2010: 784 - 788). This kind of approach suits only for relatively simple cases, since the more difficult the case is, the more difficult it is to find a continuous function which describes the displacement of the beam correctly.

It is not tried to find one function to describe and solve the whole problem in the FEM. Instead known *shape functions* are used, which are defining displacement only within the element. Those functions are the same for every specific element type. Now when a complex beam load case is divided into multiple elements, it is possible to define complex displacements of the beam with simple s*hape functions*. After displacements in nodes are solved, displacement over element can be solved with interpolation of element *shape functions* (University of Oulu 2014: 53).

An example of shape functions for Euler Bernoulli beam element is shown in Figure 4. *Shape functions* have an important role in the FEM since polynomial order of the *shape function* affects the accuracy of the element. (Taylor & Zienkiewicz 2000: 21). Terms *element order* or *edge order* are also used instead of order of the shape function, all terms meaning the same (Reddy 1993: 72).

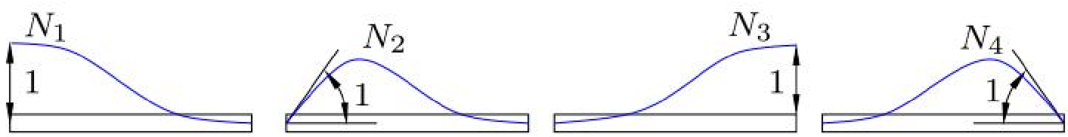

Figure 4. Example shape functions of the Euler–Bernoulli beam element (University of Oulu 2014: 15).

2.1.3 Creating and solving equations

In the FEM calculations are done by using matrix calculation. Matrix is rectangular array of numbers, symbols or expressions arranged in rows and columns. The matrix structure allows combining multiple equations into one equation presented in matrix the form. It is a really useful way to handle large amounts of equations with a computer.

The equation creation process in the FEM is mathematically complex and it is not presented more specifically in this thesis. More information related to that can be found for example from books of Reddy (1993) and Taylor & Zienkiewicz (2000). However as a result stiffness matrix **K** for the whole structure is produced, which will connect displacements and forces in nodes with following equation:

$$\mathbf{KU} = \mathbf{F} \tag{1}$$

where **K** is stiffness matrix of the whole structure,
U is matrix of the node displacements,
F is vector of forces in nodes [N] (University of Oulu 2014: 19).

There are usually different kinds of loads in the structure. The structure load vector **F** includes only the forces affecting straight into nodes. Point loads are added directly into one node so those can be added directly into the load vector. Distributed forces need to be changed into equivalent nodal forces which will then be added into structure load vector **F**. Unknown node displacements can then be solved after the stiffness matrix and **K** and load vector **F** are created. (University of Oulu 2014: 19, 90).

2.1.4 Solving stress results

As a result of the structural *finite element analysis* the displacement over the structure geometry is solved. However, usually in structural analysis, the point of interest is stress value over structure, which has to be calculated from displacements. First strains based on node displacements are calculated. Strain is a relative change in parts length and it can be calculated with the equation

$$\varepsilon = \frac{\Delta l}{l_1} \tag{2}$$

where ε is strain,
Δl is change in length [m],
l_1 is original length [m] (Mäkelä *et al.* 2010: 139).

After strains in structure are solved based on the equation (2), stress results for the structure can be calculated. Linear relationship between strain and stress is

$$\boldsymbol{\sigma} = \mathbf{D}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_0) + \boldsymbol{\sigma}_0 \tag{3}$$

where $\boldsymbol{\sigma}, \boldsymbol{\sigma}_0$ are Stress matrix and initial stress matrix [Pa],
$\mathbf{D}$ is Elasticity matrix containing material properties,
$\boldsymbol{\varepsilon}, \boldsymbol{\varepsilon}_0$ are Strain matrix and initial strain matrix (Taylor & Zienkiewicz 2000: 22).

In 3D calculations, the stress matrix calculated with equation (3) includes stress values in three axis directions with six shear stress values. It is possible to calculate principal stress values, which basically means that the stress state of the point is observed in such directions that the shear stress values become zero. Maximal shear stress can be then calculated from the principal stress values.

2.2 Error, convergence and singularity

2.2.1 Error

According to Reddy (1993: 199) three different basic sources for errors can be found in the finite element solution:

1. Domain approximation error
2. Quadrature and finite arithmetic errors
3. Approximation error

The domain approximation error is caused by elements, when curved geometry is approximated with straight edge elements. That error can be reduced with mesh refinement since smaller elements can approximate geometry more accurately. (Reddy 1993: 200).

The Quadrature and finite arithmetic *errors* are caused by numerical approximations used in calculation with computers. Integrals are calculated with numerical methods and there are also some errors caused by number round-offs. (Reddy 1993: 200).

The approximation error is due to the approximation of the solution. The unknown solution is approximated by the interpolating element shape functions (Taylor & Zienkiewicz 2000: 58). In some cases it can be shown that the approximation error is zero and FEM is giving the exact solution (Reddy 1993: 200).

2.2.2 H-convergence and p-convergence

In the FEM convergence means that *the approximation error* in solution decreases when simulation accuracy is increased. Mathematically stated that can be defined as follows:

$$\|e\| = \|u - u_h\| \leq ch^p, p > 0, \tag{4}$$

where e is error in solution,
u is exact solution,
u_h is solution with FEM,
c is independent constant,
h is characteristic element length,
p is rate of convergence (Reddy 1993: 203).

According to the equation (4) error can be reduced either by decreasing element size h or by increasing rate of convergence p. Achieving a converged solution by decreasing the element size and so adding the total amount of elements is called *h-convergence*. The rate of convergence p can be increased by increasing polynomial degree of *shape functions* and that approach for convergence is called *p-convergence*. (Reddy 1993: 201).

H-elements used in *h-convergence* have always low degree, as used elements are linear or quadratic. In order to get good results with *h-elements*, quite small elements has to be created for the problem geometry. *H-elements* are also required to be quite good shaped to give good results so elements with large aspect ratio or rapid size variation in the mesh are not allowed. (Toogood 2015: 2 - 8).

In the *h-convergence analysis* process multiple simulations are done and after each simulation *mesh refinement* is done. In the refinement process the element size is reduced, which can be done for the whole geometry or only for critical areas based on local error estimations. When the simulated solution changes only a little after the refinement, the solution is converged and the results are accurate. (Toogood 2015: 2 - 8).

P-elements have a different approach for the convergence. A quite a coarse mesh is created for the geometry and convergence is performed by increasing the degree of the shape functions for the element. Advantage from this approach is that it is possible to use more coarse mesh, which will reduce the total amount of elements in the problem. That is possible because there is not as strict a size and shape limits as h-convergence requires. (Toogood 2015: 2 - 10).

In *p-convergence analysis* the same mesh is used in each step so there is not need for mesh refinement, which saves simulation time. Since mesh is not changed in convergence, it can be tied directly into geometry. That makes it possible to perform fast parametric simulations, where parameters of the geometry can change without the need for re-meshing. (Toogood 2015: 2 - 10).

In *convergence analysis,* there are different values that can be used to measure the convergence. Two commonly used values are the maximum Von Mises stress and the total strain energy. A limit for the selected measure is set and when the difference between the two last simulations is smaller than the limit, convergence is achieved. Therefore, the set limit dictates the accuracy of the solution. Limit value is usually given as a percentage, which defines change in the percent compared to the previous simulation. (Toogood 2015: 2 - 11).

It is also possible to use a combination of both *h* and *p-methods*: the *hp-method*. In the *hp*-method *h*-convergence is performed first so that for example a 5 % convergence is achieved. After that *p*-refinement is performed to achieve final accuracy. (Taylor & Zienkiewicz 2000: 415).

To summarize, the main difference in *p-elements* and *h-elements* is the polynomial order of the *shape functions* used for the element. *P*-elements have a variable order for *shape functions* while *h*-elements have a fixed low order for its *shape functions*. (Johnson 2016).

2.2.3 Singularity

Singularity in *finite element method* means that there is a point in geometry where the result value, like stress, becomes mathematically infinite (Taylor & Zienkiewicz 2000: 234). Values at the singularity and in the small area around it are not a valid approximation of physical reality. Although there are singularities in the model, values in the other areas are still valid (PTC 2016a).

Typically singularity is caused by some of the following:

- reentrant corner
- point load or point constraint
- line load or point constrains on solid
- interface between elements of different properties, materials or element types (PTC 2016a)

Another type of source for singularity can be the element with narrow geometries like in Figure 5. In theory, the element's coefficient matrix is singular only if all three end nodes lie on same line. However, if two nodes are really close to each other compared to the third, or all three nodes are almost on same line, the coefficient matrix can become nearly singular and then be numerically noninvertible. (Reddy 1993: 305).

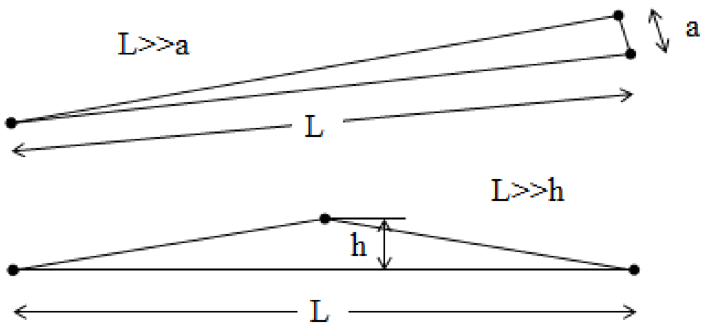

Figure 5. Examples of element shapes that can cause numerical singularity (Reddy 1993: 305).

2.3 Computer implementation

The *Finite element method* converts complex problems into multiple smaller and easier problems. There is a huge amount of unknowns and equations, which would take too much time to solve by hand. Therefore computer implementation is an obvious choice for using FEM efficiently.

According to Reddy (1993: 259) the finite element program consists of three main parts; preprocessor, processor and postprocessor. The preprocessor reads the input data given by the user and generates input data for the processor. This includes inputs like problem geometry, material settings, meshing settings, loads, constraints and selected simulation type. Based on the given inputs, a simulation model is created, which includes all data required to define the problem exactly. Element mesh is also created in the preprocessor. (Reddy 1993: 259).

The processor does the hard and time consuming solving of the problem. First it generates element matrices using numerical integration. Next it assembles element equations and then assigns defined boundary conditions. After that it calculates the created equations to solve primary nodal values. For example, in a case of structural analysis displacements of nodes are calculated. (Reddy 1993: 260).

The postprocessor is responsible of converting the solved data into a form which the program user can understand. It plots results into requested graphs and calculates secondary values like stresses or strains from primary values. It also calculates result values on other points than nodes, if needed. (Reddy 1993: 259).

2.4 Failure criteria

Stress results of the simulation are observed in strength analysis to find if the structure will fail or not. Different kinds of failure criteria have been developed, which can be used for the evaluation of the results. Properties of simulated material are defining which kind of criteria there should be used. All criterions introduced in this chapter are for isotropic materials.

Materials can be divided into ductile and brittle materials. According to Christiansen (2007) whether material is ductile or brittle can be estimated based on its uniaxial failure tension (T) and failure compression (C). When the ratio of T/C is close to one (or T≈C) material is ductile. And the smaller the ratio is the more likely the material is to be brittle.

Ductility of the material can also be defined based on material elongation. Elongation at fracture for the material is measured as a percentage. If elongation is below 5 %, the material is brittle and if over, the material is ductile (Pope 1997: 312). It has to be remembered that temperature and other environmental conditions can affect the ductility of the material (Efunda 2016).

2.4.1 Ductile materials

The maximum shear stress criterion (also known as Tresca’s or Guest’s criterion) can be used to predict the yielding of ductile materials. In ductile materials failure often occurs when crystal planes are slipping along maximum shear stress surface. Therefore, according to the shear stress criterion, there is not a failure when maximum shear stress at the point is less than the material maximal yield shear stress. The maximum shear stress criterion is visualized in the 2D situation in Figure 6. (Efunda 2016).

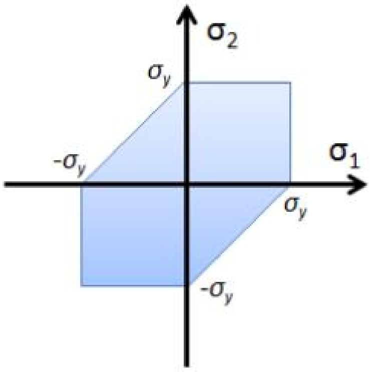

Figure 6. Shear stress failure criterion for 2D stress in graphical form (Efunda 2016).

Von Mises criterion is also known as the maximum distortion energy criterion. In the von Mises criterion the limit for the failure is energy of distortion. The material fails if the energy of the distortion in a multi-axial stress situation is higher than the same energy in the uniaxial tension at the moment of failure. Mathematically, the von Mises criterion is defined according to form (5)

$$\frac{1}{2}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right]\leq \sigma_y{}^2, \tag{5}$$

where σ_1, σ_2, σ_3 are principal stresses [MPa],
σ_y is material yield stress [MPa] (Efunda 2016).

Von Mises criterion can be used for ductile materials. It allows for more stress to be applied to the material before yielding, when comparing to the maximum shear stress criterion. This can be seen in Figure 7 where both von Mises limit and the maximum shear stress limit are visualized in the same graph.

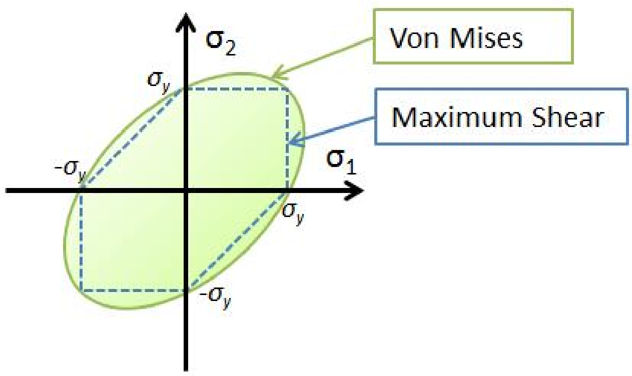

Figure 7. Von Mises failure criterion in 2D case (Efunda 2016).

2.4.2 Brittle materials

There are different stress limits for compression and tension when loading brittle materials. The maximum normal stress criterion states that failure occurs when the largest/smallest principal stress reaches either tensile strength or compression strength limit of material. That is visualized in Figure 8, where it can be seen how brittle materials can handle more compression stress than tension. (Efunda 2016).

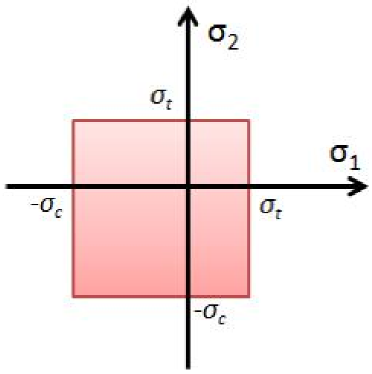

Figure 8. Maximum normal stress criterion visualized in 2D stress state (Efunda 2016).

Mohr's theory of failure criteria (Mohr–Coulomb theory) is based on the famous Mohr Circle. According to Mohr's theory, material fails from critical combination of normal and shear stress. It is more conservative than Maximum normal stress criterion, which can be seen in Figure 9. (Efunda 2016).

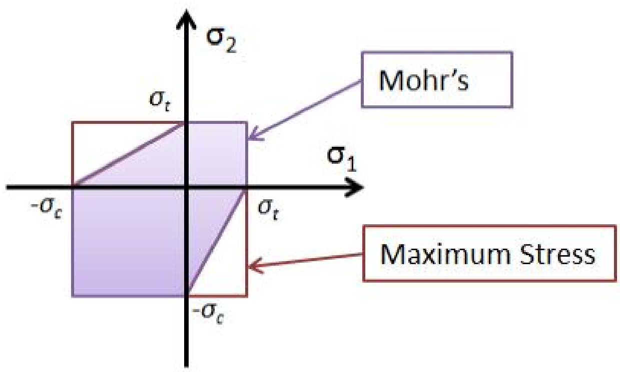

Figure 9. Failure criterion according Mohr's in 2D stress state (Efunda 2016).

2.4.3 Christensen criterion for ductile and brittle materials

Christensen has developed a failure theory which covers both ductile and brittle materials. According to Christensen's (2007) theory there will be failure when the normalized and combined stress result is bigger than one. Mathematically stated that is

$$\begin{aligned}&\left(\frac{1}{T}-\frac{1}{C}\right)(\sigma_{11}+\sigma_{22}+\sigma_{33})\\&+\frac{1}{TC}\left\{\frac{1}{2}\left[(\sigma_{11}-\sigma_{22})^2+(\sigma_{22}-\sigma_{33})^2+(\sigma_{33}-\sigma_{11})^2\right]+3\left(\sigma_{12}^2+\sigma_{23}^2+\sigma_{31}^2\right)\right\}\leq 1\end{aligned}, \tag{6}$$

and if

$$T\leq\frac{C}{2} \tag{7}$$

then also

$$\sigma_1\leq T \tag{8}$$

where T is failure value of uniaxial tension [MPa],
C is failure value of uniaxial compression [MPa],
σ_{11}, σ_{22},… are values of stress state [MPa],
σ_1 is the largest of principal stress [MPa] (Christensen 2007).

The failure limit in Christensen's theory is basically the same with the Von Mises theory when T = C. When T starts to be smaller than C, the area of acceptable stress states is shifted towards the compression stress according to equation (6). If material ductility is below T/C ratio according form (7), then additional cut for failure limit area is done according to form (8). (Figure 10).

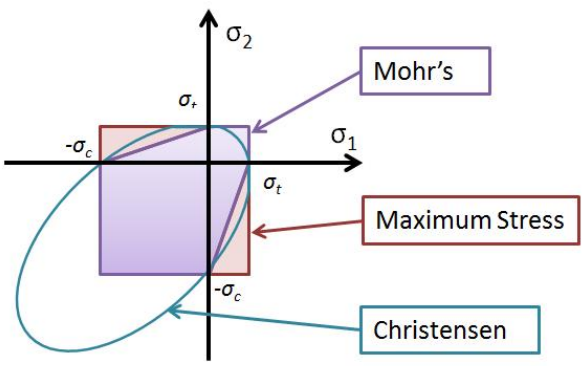

Figure 10. Comparison of brittle material failure criterions in 2D stress (Efunda 2016).

2.5 Introduction to Creo Simulate

Creo Simulate is a closely integrated *Finite Element Analysis* (FEA) solver with Creo Parametric. The model created in Parametric can be opened with the same user interface in Simulate. Integration makes it easy to modify a simulated part and run new simulations for modified geometry without the need to define the whole simulation again.

All work presented in this thesis is based on Creo Simulate 3.0 and computer the used for all the example and test simulations was HP EliteBook 8570w, including Intel i7-3740QM 2.7GHz, 16GB RAM and Nvidia Quadro 1000M with 2GB RAM.

2.5.1 Simulate capabilities

Analysis capabilities in Creo Simulate with the advanced simulation extension (PTC 2016b) are:

- Linear and nonlinear static structural analysis
    - Large displacements and strains
    - Sliding contact
    - Hyper-elastic materials
    - Elasto-plastic materials
    - Nonlinear springs
    - Boundary conditions applied sequentially
    - Snap-through
- Modal and dynamic structural analysis
    - Time response
    - Frequency response
    - Random response
    - Response spectrum
- Linear buckling structural analysis
- Pre-stress structural static and modal analysis
- Linear and nonlinear steady state thermal analysis
    - Temperature dependent convections

  - Gray body radiation
  - Temperature dependent material properties
  - Boundary conditions applied sequentially
- Transient thermal analysis
- FEM mode: use of NASTRAN or ANSYS solver
- Fatigue (optional module)

2.5.2 Features of Creo Simulate studied in this thesis

The scope of this thesis is to give required information for mechanical engineers so that they can perform strength analysis to their own part designs. Findings are presented so that also new FEM users can start to do simple FEM simulations. This scope will narrow down the features that will be presented from Simulate.

This thesis will focus on static structural analysis, which is the most useful simulation for efficient strength analysis of a part or assembly. In this work linear, nonlinear and contact type of analyses, different design studies and static pre-stress analysis will be covered. Those are presented more in chapter 4.

Dynamic and modal analyses are not included in this work because using them requires deeper understanding of FEM and dynamics to create reliable simulations. In the scope of this thesis it is not possible to provide the required information for that.

Creo Simulate also includes thermal analysis mode. It allows the user to apply thermal loads to the structure and solve transient and steady state problems. There is also a possibility to simulate stresses caused by thermal load and use those in structural analysis. Nokia has different programs for simulating heat problems so the thermal mode is not included in this work, although it can be useful for the mechanical engineers.

Creo Simulate also has a FEM mode which allows the user to create a simulation with materials, loads and connections with Creo's interface and then run that simulation with another FEM solver like Ansys or Nastran. The Benefit from this feature is that you can maintain the possibility to change the model without having to recreate simulation between modifications. However, there were no easily available proper manuals for the connectivity between Ansys and Simulate so that feature did not work properly when tried to test with Nokia’s licensed versions of those programs.

Based on testing of FEM-mode it is not most likely beneficial to study it further. If it is necessary to use some other software than Creo Simulate for the simulation, it is probably easier to transfer the model manually as a step file or as an original Creo part file since Ansys can handle also those. The FEM mode could be good feature if the user will get it to work, but it is not studied more in this thesis.

2.5.3 Simulate from point of new FEM software user

Before using Creo Simulate I have been using mostly Abaqus, which is pure FEM simulation software. Although Simulate can be used as standalone simulation software, it is more like additional package to Creo Parametric. For same reason it has different approach to software user interface.

Simulate aims to have really easy-to-set-up interface for simulation. In addition, many technical decisions have been made so that even a new FEM user could build basic simulations easily. All simulation features are added in same type of user interface with Creo Parametric. Understandable feedback in problem cases is also given and the user is guided to select the correct settings by disabling and preselecting certain settings based on added simulation features. There is not too many complicate settings and in most cases the default settings will work fine by giving fairly good results.

There are also negative sides in the Simulate’s user interface decisions. Although it is easy to learn, many of the settings are hid from the user or there are not so many possibilities compared to pure simulation software. A too “easy” set up can also lead to a situation where the user does not know at all about all possible settings, some of which could be essential in some cases. When it is possible to get results without touching any settings, some users can then skip studying of what the purpose of settings is and when those should be changed.

3 CREATING SIMULATION MODEL

In this chapter, the required process steps of creating a simulation model from a 3D model are walked through. That includes all steps from possible geometry modification to the definition of material, mesh, loads, constraints, idealizations and connections. The creation of simulation model is also summarized in Appendix 1, which is short simulation workflow description made for Nokia.

The generated simulation model is only an idealization of a real part or assembly. The accuracy of the model depends on how well all the geometry and computation details of the real part are managed to apply to the simulation model. However, a really accurate model is not equal with a good model, since more accuracy always means a larger model and longer solving time to get results. A good model is therefore a compromise between accuracy and computing resources.

The simulation model includes all the features required by a solver to run the simulation. In Creo Simulate that model is based on a Creo Parametric part. In Simulate simulation features added to the part are saved to a same part file with part geometry which ensures preservation of the simulation model. The Simulation features added to a part are shown also when opening the assembly simulation with the same part included.

3.1 Model simplification

A model simplification is not mandatory to carry out every time when making a simulation, instead it depends on the models geometry and complexity. The purpose of the simplification is to simplify simulation model in order to reduce calculation time. Sometimes with really small features it is necessity to make model possible to run for FEM solver.

Toogood (2015: 2 - 5) presents in his Simulate Tutorial book an example of how a real part should be simplified into the idealized physical model. In the given example all chamfers, fillets and even mounting holes have been removed from the part. With a CAD integrated FEM solver you should never do that kind of oversimplification. Linear static analysis with a single part is usually so fast to compute that removing geometry features from 3D model takes way more time than the reduction achieved in computation time. Most benefits of the simplification are achieved when running nonlinear analyses with many calculation steps or design studies.

3.1.1 Simplification by feature reduction

Simplification is done by suppressing or removing features that are not important from point of model strength or simplified features are outside of an interesting area. Features to suppress or remove are usually small details that do not include anything needed to be simulated but are requiring lots of elements. Simplified feature can be for example a small hole, pointy feature or round in outside corner. Example of simplified feature is shown in Figure 11.

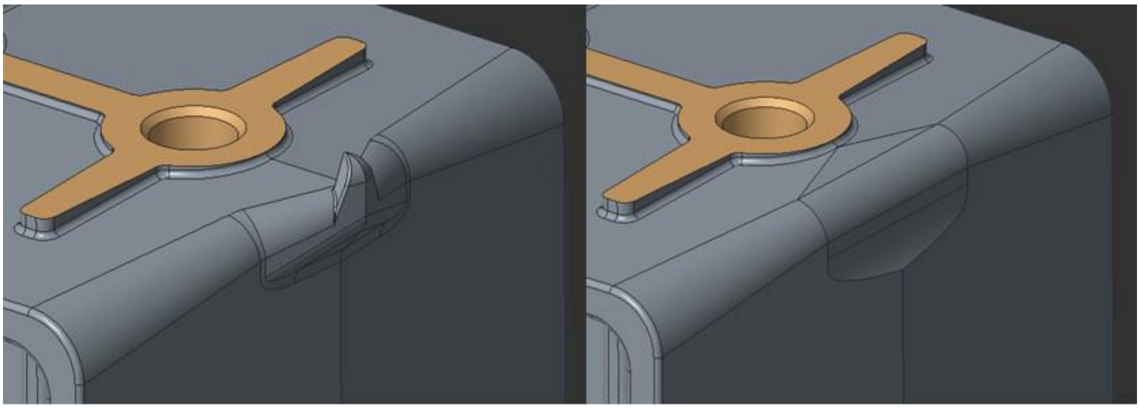

Figure 11. Example of simplification. Pointy feature removed with extrude.

3.1.2 Simplification based on symmetry

A symmetrical model can be simulated including only half of the geometry into the simulation model. Geometry with two symmetry axis can be reduced to one fourth of the original model or even to one eight piece with 3 axis. The usage of symmetric simplification can therefore give great reduction in model size. In order to use symmetry, constraints and loads applied to the model have to be symmetric also.

A symmetrical model is formed simply by cutting away geometry from one side of symmetry plane. In Creo Parametric that can be easily done by first selecting symmetry plane and then using solidify command. The cropped model geometry is then opened in Simulate and a symmetry constraint is added onto the cutting surface.

The usefulness of symmetric reduction has same principles than feature reduction. It is worth using mainly with models that takes long time to solve. Although it can reduce the calculation time more than feature simplification, up to half of the original, half of the 30 seconds is only 15 seconds so no significant benefit can be obtained with a small model.

3.2 Material definition

Every part has to be assigned with properly defined material for simulation. There are different types of material models, depending what is purpose of simulation and what needs to be considered. The material model has to include at least density, Poisson’s ratio and Young’s modulus. The given values define mathematically how the material will behave in the simulation. Poisson’s ratio defines how much a brick of material compressed in one direction will expand to the other two dimensions. Young’s modulus tells how much the material will stretch in relation to the applied stress.

The simplest material model is the linear elastic model in which it is assumed that material will behave only elastically and plastic deformation does not happen. The linear material model works well when yield strength of the material is not exceeded in the simulation.

There are plenty of other material models possible to use in Simulate including multiple elastoplastic and hyperelastic material models. Elastoplastic models are used to model materials with plastic deformation applied to simulation, hyperelastic models are for materials with really big elasticity like rubber. And then there is a possibility to use one’s own model based on tabled test results. (PTC 2016a).

Simulate also supports anisotropic materials. Those have different material properties depending from the load direction related to material orientation. A good example of anisotropic material is wood, it has grains which makes it behave differently depending on the loading direction. (PTC 2016a).

3.3 Constraints

Constrains are features used for fixing the chosen reference of the model to place. The model needs to be constrained so that there is not free body movement in order to run the simulation (expect in modal analysis). Obligatory constraint can be replaced either with the point to ground spring or the *inertia relief* setting. Inertia relief simulates the model as if it floats freely in space and it automatically balances user applied loads with body loads. Inertia relief can be used for the linear cases where the user is simulating stress caused by internal forces or the model is kept in place with two forces cancelling each other instead of a fixed connection.

When creating the constraint, it is possible to choose which *degrees of freedom* (DF) are tied. The amount of DF for the model depends from the element type used in simulation. In Simulate the solid elements have three translational DF to XYZ directions of the coordinate system. Shell elements have 3 additional rotational DF around XYZ axis.

The Constraint is completely rigid, which means that a fully constrained surface or edge cannot deform. Because of that the constrained surface shows no stress levels, but on the edges of the constrained area can be found high stress concentrations. Those small spots of high stress can be usually ignored since they are not real stress levels. Without rigid constraint, stress would be divided on a larger area which would lead into lower stress levels. Constraint applied into an edge or point will cause singularity. In singular points stress value is theoretically infinite (PTC 2016a: Singularities; chapter 2.2.3 page 20).

Constraints are applied to a model so that those correspond with real world situation from the view of functionality. With simulation models it is easy to fail getting correct result simply by using wrong kind of constraints or placing those to wrong places. Therefore before blindly applying just something to the model it is important to carefully go through the models fixing points and understand how those affect the model. Constraints are typically added to real fixing points of a part or assembly. Good choices are for example the inner walls of the fixing bolt hole, the surface to be mounted against a wall or another reference representing a fixed part of the model. Combinations of more than one constraints are also possible and, in some cases, necessary in order to simulate the fixing of a part correctly.

In Simulate, there are some predefined constraint types, in addition to the normal fully manual constrain definition. Those include planar, pin, ball and symmetry constraint. Planar is used to tie movement normal to selected surface and it can be applied only to planar surface. Pin is an option to create bolt or hinge type fixing where reference is cylinder shaped and rotation around feature axis is allowed. Ball is similar to pin but fixed feature is sphere and rotation around its center is allowed in all directions. Symmetry is specially used to add symmetry constrain to a cropped surface of symmetric model.

3.4 Loads

Loads are one of the most important features in simulation and every simulation (expect modal) needs a load. Load is usually applied as a force or pressure but a simulation can also be loaded with forced displacement constraint instead of force. Before applying a load it is important to calculate or estimate the value to for the load. For lifting eyes that can be the mass of the attached object and for the bracket structures it can include for example both mass of the unit and wind load. A safety factor can also be included into the value of the load.

Loads can be applied to points, edges and surfaces. Loads do not cause a reference feature to turn rigid like constraints do. When simulating models with solid elements it is best to use surfaces as reference. When the load is applied to an edge or point it can lead to unrealistic high stress levels near the edge or point (PTC 2016a: Singularities). With the surface, the load is divided to higher number of element nodes, which prevents that problem. The existing surface can be used as a reference for the load. If needed, there is also easy way to cut a specific shaped reference surface from another surface in

Simulate. There is a variety of load types available to use in Creo Simulate. It includes force/moment, pressure, bearing, temperature, gravity, centrifugal and preload. In static analysis it is in many cases better to use equivalent force instead of gravity.

3.5 Meshing

In meshing, the volume of a part is divided to smaller pieces for calculation. Generally in FEM programs mesh is created according to user selections before simulation is solved. In Simulate meshing is done by default without any user needed definitions as a part of running the simulation.

This is possible because Simulate uses *p-type elements* so created mesh does not have to be as good as with traditional *h-type elements* (Toogood 2015: 2 - 10). However, Simulate tends to use really coarse mesh by default and for better results it is recommendable to use some control for mesh creation. Although there is added mesh control, actual mesh is not created until running simulation.

Itis possible to run *AutoGEM*, a meshing tool of Simulate beforehand if needed. *AutoGEM* creates a mesh according to the given mesh controls and displays it to user. There are couple of things to keep in mind when using *AutoGEM*. First, displaying mesh with *AutoGEM* requires much capacity from computer when showing models with great amount of elements. In test simulations, Simulate crashed multiple times because of that. Second, the mesh created by *AutoGEM* in advance is not automatically used for simulation run. The user has to save a mesh and change simulation run settings to use the saved mesh instead of a default setting which always creates a new mesh for the run.

There are a few things the user can modify when defining a mesh to obtain better results. Those are related to the element type, element size and node locations. Mesh control is in a really important role when doing nonlinear, contact or other more challenging simulations. With linear static analysis mesh control is not so critical, and good results can be got also with simple mesh control.

3.5.1 Element type

The most useful element type for solid 3D part is the solid tetrahedron. Tetrahedron is composed of four triangular faces and it has four nodes. For basic linear structure analysis there is not usually a need to change the type of element from the default solid tetrahedron element.

For the sheet metal parts or to other parts with one dimension much smaller than two other can be used shell or thin solid elements. The shell element is basically a 2D element and the thickness of the plate is given as parameter. Shell can be formed in Simulate by midsurface compression which creates a shell surface between top and bottom surface of the plate. Shell elements can be defined to be squares and triangles or only triangles.

Thin solids are basically shell elements extruded to the normal direction to create solid elements. Defining those is a similar process to shell definition. The benefit of thin solids is that the shape of the elements is better and the amount of the elements created is less compared to solid tetrahedral elements (PTC 2016a: Thin solid). Thin solids have, however, one disadvantage in Simulate since they do not support bonded connections between surfaces or weld connections. This makes the usage of those elements more complicated because the connection area has to be either simulated with tetrahedral elements or rigid link has to be used. Therefore those elements work best with one part simulations without any connections.

Shell elements and thin solids are overpowered compared to tetra elements for sheet metal parts, when thickness of the plate is really small related to the other dimensions of the plate. Solid tetra elements have limitations for allowed minimum corner angle so the thickness of the plate limits maximum size for the tetra element. Therefore for example a 0.1x100x100 mm plate with solid tetra requires almost 39 000 elements with Simulate's default meshing settings to fulfill element shape requirements. With thin solids and default meshing settings same plate is modeled with 3 200 elements. With shell elements that amount can be reduced all the way even to 1 element, depending from the needed accuracy. Calculation time depends mostly from the amount of the elements so suitable element type can provide significant reduction in simulation time.

As a rule of thumb it could be said that using thin solids or shell elements is worthwhile when one dimension of the part is less than 1 - 5 % from the other dimensions.

There is also a possibility to make so called mapped mesh which means basically dividing volume into elements with straight lines to produce structured mesh. Mapped mesh often results in much shorter analysis run time compared to unstructured meshes. Mapped mesh is useful when modeling contact surfaces with contact pressure calculation, running large deformation analysis, using elasto-plastic materials or calculating large deformations with thin-walled surfaces. (PTC 2016a: Mapped mesh).

Creating mapped mesh to a 3D model requires either cube shaped volume with planar surfaces or cylinder shaped volume. Part geometry can be divided to cubes and cylinders with *volume region*. Defining mapped mesh takes some extra effort compared to unstructured meshes. Therefore it is pointless to use it for any other than really challenging simulations mentioned above where accuracy of results depends highly from the used element type or reduction in calculation time is significant.

Prismatic elements in Simulate are some kind of combination between thin solids and mapped mesh. It is like thin solid elements stacked to form many layers, and some of layers can have different outline shape. Requirements for prismatic elements are planar and parallel top/bottom surfaces while all other walls have to be orthogonal with bottom surface. Usage of prismatic elements is limited strictly by geometry which makes them useful only for special cases and not so useful for normal quick part design strength simulation. (PTC 2016a: Prismatic elements).

3.5.2 Element size

Element size is one important feature affecting the accuracy of the simulation. The solution of simulation approaches the exact solution when the size of the elements is reduced, if the solution converges (Chapter 2.2.2 page 18). Therefore smaller element size gives more accurate results but requires more calculation time by increasing the amount of elements.

Reducing the element size to get more accurate results is called *h-refinement*. There is also a possibility to locally reduce the element size on interesting areas while keeping element count low by bigger element size on unimportant areas. When refining is done based on local error estimations, term *adaptive meshing* can be used for the used method. (Taylor & Zienkiewicz 2000: 402).

Adaptive p-refinement is used in Simulate (Taylor & Zienkiewicz 2000: 402; Chapter 2.2.2 page 18) for ensuring result convergence. Instead of reducing the element size there is increased edge polynomial order for the element, which will increase the element accuracy. Therefore there is not a need for adaptive meshing tools and aggressive element size control in Simulate. However it is possible to get more accurate results when using suitable mesh control (chapter 3.5.5 page 41).

There are lots of ways to control element size in Simulate. The fastest way is to give general maximum element size for whole part. That is best solution for models consisting only one relatively simple part since it is fast to define and gives good results with reasonable calculation time. The maximum element size can also be assigned to a separate volume, surface, edge or point which is one way to refine element size locally.

3.5.3 Node location

Sometimes it is important to get element nodes to specific places. For example bonded connection between two overlapping surfaces requires that both surfaces have nodes in same places to work well. Simulate makes that automatically so normal static part strength simulation does not require manual node location definition at all.

However, location of nodes can be defined by hard surfaces, hard curves or hard points. Hard point defines location ensuring presence of node on that point. Hard surface or curve means that the element nodes are placed along the reference plane/curve. An edge or a curve can also be assigned with *edge distribution* which places the given amount of nodes divided equally spaced onto the reference. That is most useful of node controls since it can be used also for element size refinement.

3.5.4 P-type element capacity test with coarse mesh versus fine mesh.

Simulate uses p-type elements, which should be capable to give good results also with coarse mesh. In this chapter, the capacity of p-elements is studied by comparing results simulated with really coarse mesh to results with fine mesh. Study is done with a simple model geometry shown in Figure 12. The geometry is selected so that it is possible to model only with three elements and at the same time there is form which causes changeable stress distribution into the structure when loaded. The material used is steel (Poisson’s ratio 0.3, Young’s modulus 200 GPa). The model is constrained from left end surface and 1 kN load is applied on to the right end surface according Figure 13.

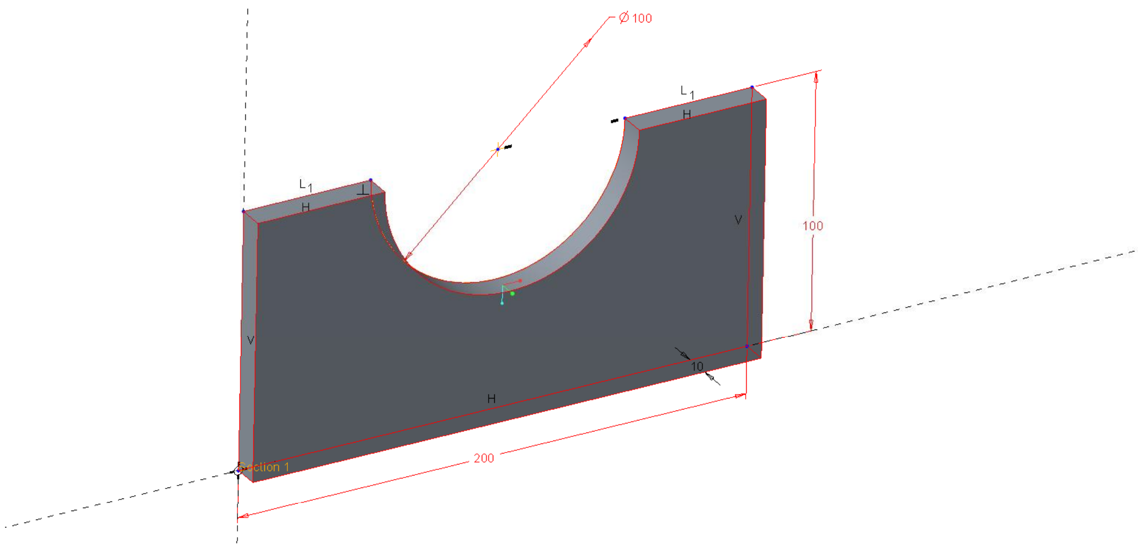

Figure 12. Geometry used for mesh test.

The element type used for in this test is thin solid element, which creates wedge or brick shaped elements. Same element type is used for coarse mesh and fine mesh. Fine mesh is created with 15 mm maximal element size limit added for whole geometry and total amount of elements is 138. Coarse mesh is created without any mesh control and amount of elements is 3. Created mesh can be seen in Figure 13.

Element edge order is increased in p-elements in order to get more accurate results (Toogood 2015: 2 - 10; Chapter 2.2.2 page 18). There are three different control options for edge order which can be used in the simulation. The quick check simulation makes the whole simulation with the edge order of three for each element edge. The single-pass adaptive method runs two simulations; the first simulation is run with the edge order of three. After that Simulate creates local error estimations and based on that increases edge order on needed edges for second simulation. The third multi-pass option runs multiple simulations and increases edge order between simulations until given convergence limit or given maximal edge order is achieved.

In the first comparison the simulation results with fine mesh are compared to coarse mesh with single-pass convergence method. Von Mises stress results from both simulations are shown in Figure 13. According to the results there is not a significant difference between fine and coarse mesh. The difference in the maximal stress value is only 1.2 % and visually observed the stress distribution looks similar.

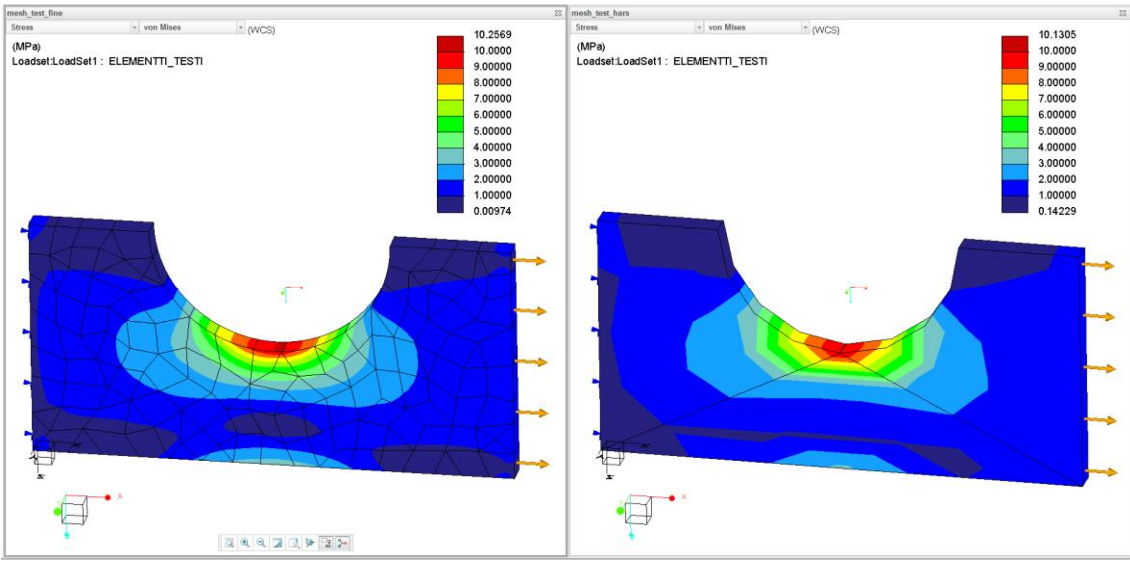

Figure 13. Comparison in results between fine and coarse mesh. Fine mesh on left side of the figure.

Next how used convergence method affects into results is studied. That is done with coarse mesh by simulating the model with both quick check and multi-pass methods. A 3 % convergence limit and maximal edge order of 9 for multi-pass adaptive method is given. Results from both simulations are shown in Figure 14. Results with quick check are significantly lower than with other methods. Stress distribution is similar with other methods but with much lower stress levels. Difference in maximal stress is 50 % compared to fine mesh result. The multi-pass method gives almost the same results with the single-pass method and fine mesh. Maximal von Mises stress value is 1.3 % higher compared to fine mesh results.

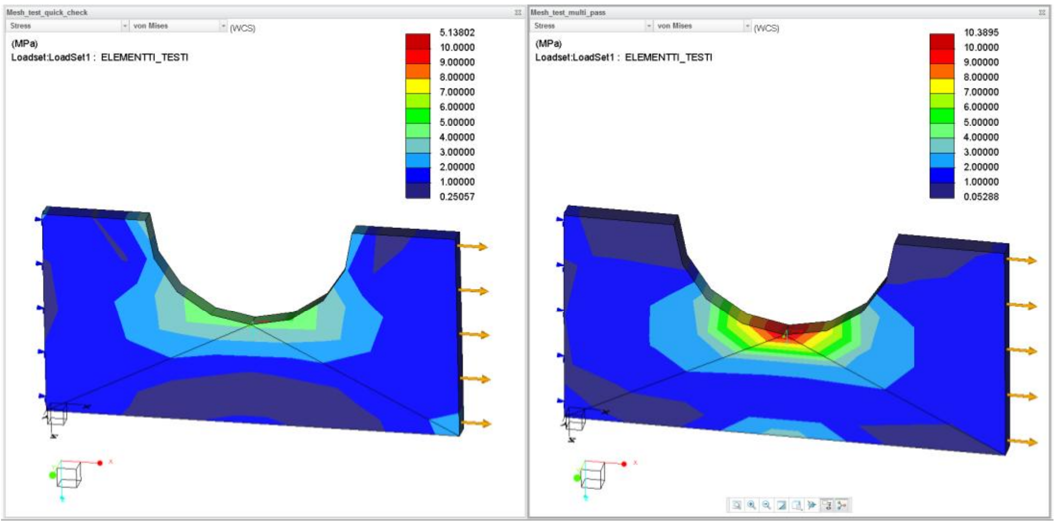

Figure 14. Quick check results on left side and multi-pass results on right.

It is possible to plot the used edge orders in model simulation. Those plots can be used to find areas where maximal edge orders are used. The edge orders of these test simulations are shown in Figure 15. There it can be seen thatthe single-pass method increased p-levels to the maximum value of 7 while multi-pass method used maximal level of 9. Reason for high p-levels in multi-pass method is really low convergence limit of 3 % used for simulation.

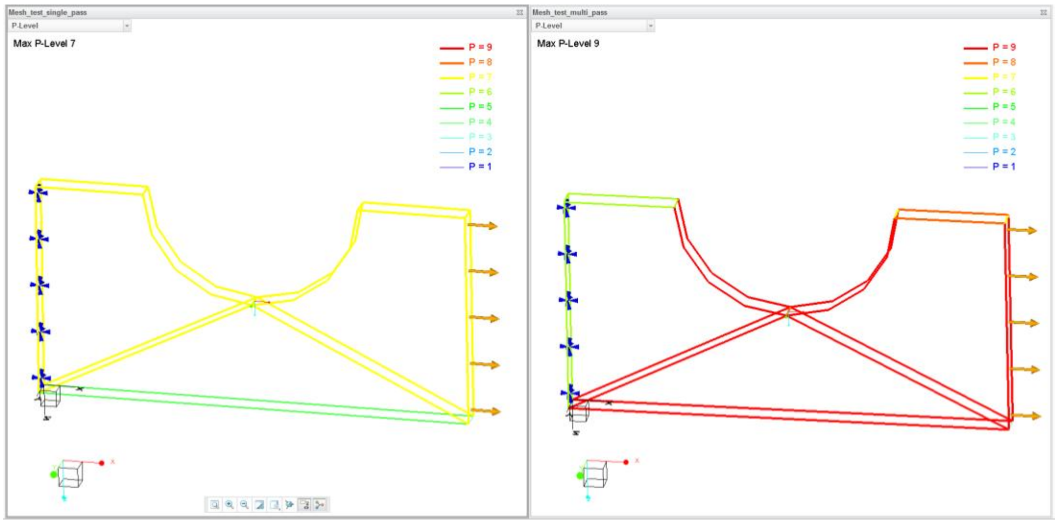

Figure 15. Edge orders of single-pass method on left side and multi-pass convergence method on right.

The conclusion from this mesh test is that it is possible to get quite good results with coarse mesh when using Simulates p-type thin solid elements. Results also show that quick check method should not be used with coarse mesh at all. Single-pass adaptive seems to give already quite accurate results so there is not a reason to use multi-pass adaptive in most cases.

3.5.5 Effect of element size limit to the result accuracy.

In this chapter it is studied how changing mesh size changes results in Simulate. That kind of test is called *convergence analysis*. Maximal von Mises stress or total strain energy can be used as a measure for convergence. In h-type analysis mesh size is reduce while p-type analysis is performed by increasing element edge order. (Toogood 2015: 2 - 11).

Simulate uses automatically p-type convergence in simulations. Purpose of this test is to find out whether additional benefits or accuracy are achieved by also using h-type mesh refinement for the simulation model. The combination of the p and h-type methods can be called hp-convergence (Taylor & Zienkiewicz 2000: 415).

The test is done for the case example model presented in chapter 7.2 at page 79 (improved design geometry). Model is 4 mm thick sheet metal part and it is simulated with default solid tetra element. The first simulation is run without any mesh control. For the other simulations a maximal element size limit is added into the whole part. As a reference for the accuracy is used finest mesh with 3 mm element size limit shown in Figure 16. All simulations are done with single-pass adaptive convergence method.

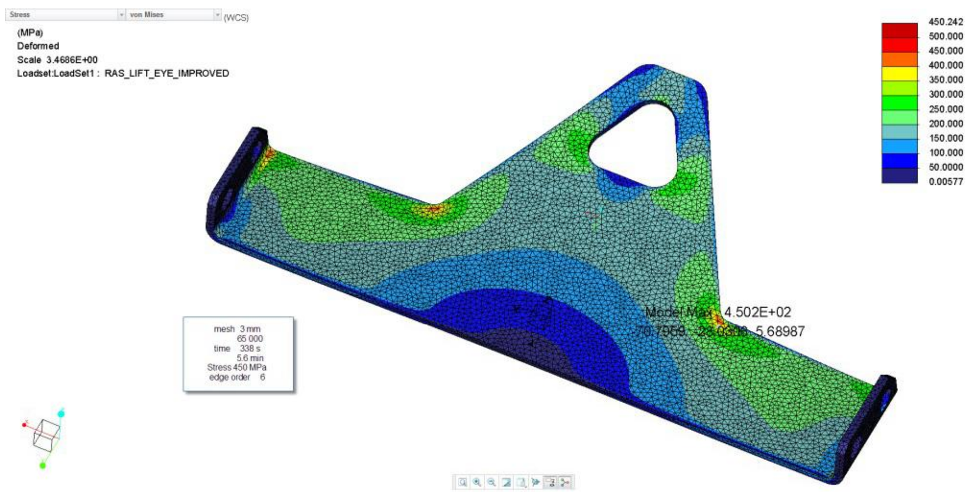

Figure 16. Reference mesh with 3 mm element size limit.

From each result is found maximal von Mises stress value in high stress concentrations near inner corner of the lifting flange. That value is compared to value calculated with smallest 1 mm mesh to calculate result accuracy. Results are shown in Figure 17. The horizontal axis tells the element size limit used for particular simulation. Highest stress found in flange corners is shown with yellow line. The green column shows error in the percent compared to result got with finest mesh. The blue column shows calculation time needed with the used laptop computer (HP EliteBook 8570w including Intel i7-3740QM 2.7GHz, 16GB RAM and Nvidia Quadro 1000M with 2GB RAM).

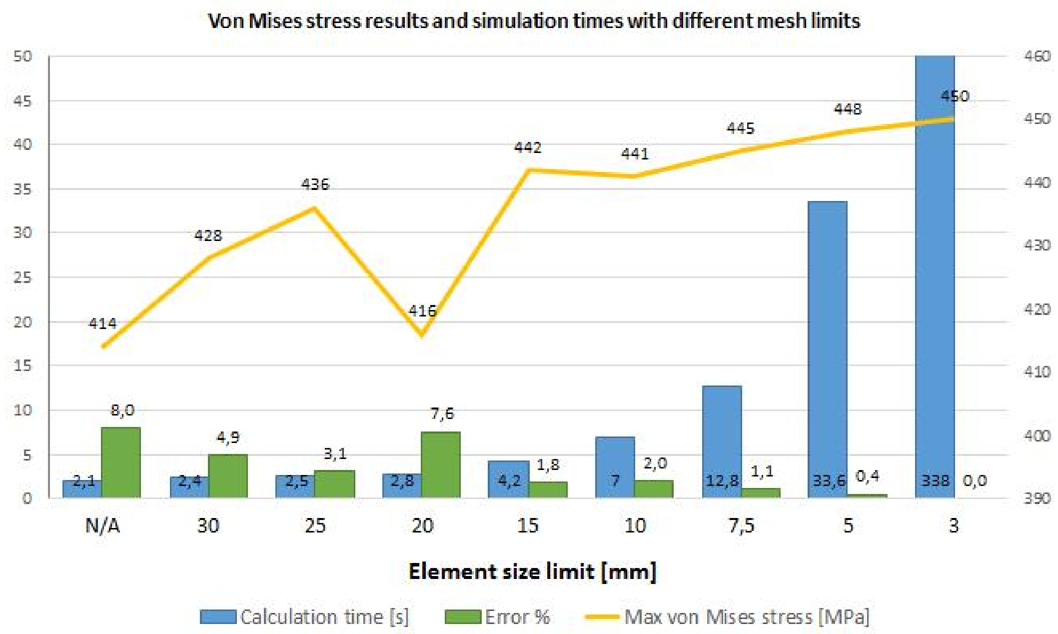

Figure 17. Results from element size limit test simulations.

The yellow maximal von Mises stress result (Figure 17) shows that the simulated maximal stress value is converging. For some reason there were unexpectedly low results when the 20 mm mesh size limit was used. Anyway, according to the test results the element size limit which is 2 - 4 times the thickness the plate seems to work best (4 mm in this case so 8 - 16 mm in element size limit). The results are quite accurate and the simulation time is still reasonable.

Results of this test do not mean that the discovered size limit for solid tetra elements works best with all kinds of model geometries. It shows that results are most likely to be more accurate when using a finer mesh although Simulates p-elements work quite well with default coarse mesh. The reason for more accurate results is probably that limiting tetra element size gives elements with better shape, which are usually giving more accurate results.

Conclusion from the test is that *hp-method* gives more accurate results than Simulates single pass adaptive p-method alone. That can be adapted most easily into practical use by applying systematically suitable mesh size limit for whole geometry for all models and then letting simulates p-convergence to do adaptive convergence based on local error estimations.

3.6 Idealizations and connections

Idealizations and connections are predefined simulation features. Most of those can be simulated by creating corresponding geometry with right material properties and manual node-to-node connections. Using idealizations and connections makes simulation model building process easier and faster.

3.6.1 Idealizations

Idealizations are used to add properties of the real part or assembly like mass and springs to the simulation. Idealization turns the real feature into a simulation feature while trying to keep its behavior as realistic as possible. Creating idealization requires in some cases defining numerical parameters.

Available idealizations in Simulate are beam, spring, mass, shell and crack. A beam is, according its name, used to simulate beam going from one point to another or along a predefined curve. It is useful if a standard beam with known numerical properties is used in a model. It suits best for situations where it is not necessary to produce a 3D model of a product with exact geometry but strength simulation is required.

Mass and spring features are needed more when doing dynamic simulations. Mass can be used to add physical behavior of additional part or assembly into simulation model without adding the part itself. That makes the simulation faster to calculate because there is no need to mesh that new pure idealized component. In static cases mass is better to be replaced with corresponding force.

Springs can be used to connect two points or features with idealized spring. Simulates spring definition supports both extensional and torsional stiffness with constant and curve related stiffness. Springs can also be used instead of constraints for fixing model to ground.

Crack idealization can be used for simulating how the structure will withstand loads if there is a crack in the part. It is a way to simulate how the part will behave if it is not flawless. The same principle of simulating with small errors can be used especially with compression type loads where buckling can happen for the structure. In those cases small errors can be added to the model either with geometry modification or additional geometry changing force.

3.7 Connections

Connections are used to make joints to a part or to connect two parts in assembly. Connection is usually modeled as rough idealization of real connection. More detailed or realistic connection simulation requires lots of work and is not worth the effort unless connection is target of the simulation.

3.7.1 Contacts for overlapping surfaces

Simulate automatically detects overlapping surfaces in assemblies. By default it adds bonded connection between those surfaces. Bonded connection works like the contacted parts are glued together. It creates similar mesh on both surfaces and ties matching nodes together. Other interfaces for between two surfaces are free and contact. Free means that those surfaces do not react at all and can penetrate one another. All surfaces are defined with free by default and free interface needs to be used only if there is two overlapping surfaces that are not meant to be bonded.

Contact interface means that selected surfaces can separate one another but those cannot penetrate. Contact can be defined to be frictionless, with infinite friction or with finite friction. Contact between surfaces turns the simulation into nonlinear simulation which requires more calculation compared to linear analysis. Therefore contact should be used only if the simulation specifically requires it. For example the designer is designing a snap feature and is interested how the male part of structure will take bending. Simulation can be done by modeling both parts, defining contacts between the needed surfaces and then running a lot of time taking nonlinear simulation. Instead it is faster to calculate linear simulation only with the male part and add deformation as forced constraint displacement. Both will give same result but the latter is much faster.

3.7.2 Weld connections

Weld connection can be used to create a bonded like connection between two features if they do not overlap. For example if the sheet metal part is bended to form a closed loop and then welded, there are two ways to simulate that connection. One way is to modify the geometry so that features to be connected will overlap but easier way is to use weld connection. An example of weld connection is shown in Figure 18. End weld connection in the figure extends wall to be welded on to selected surface and then forms bonded connection between overlapping surfaces.

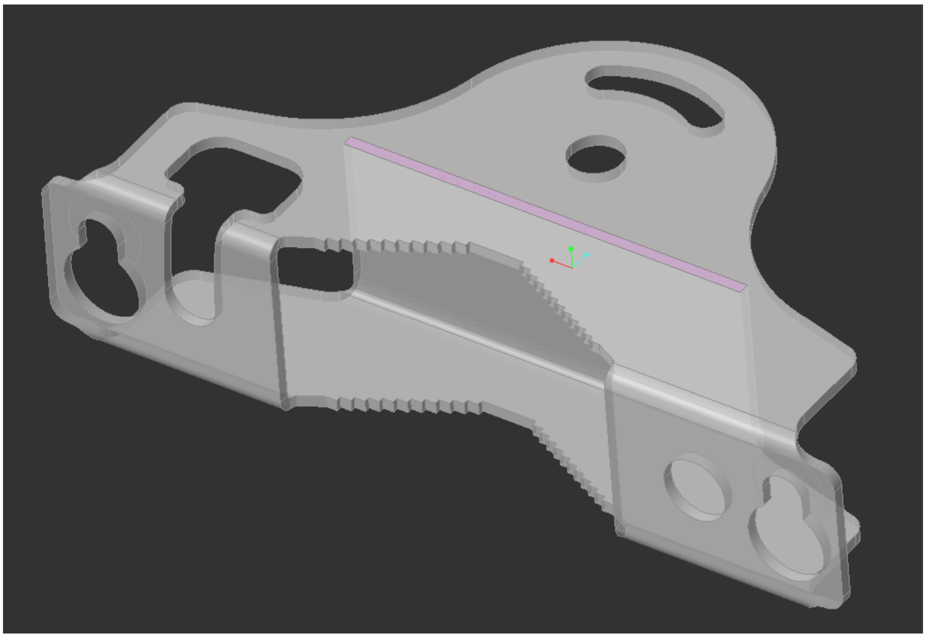

Figure 18. Simulation model with end weld connection (pink color).

Weld connection definition allows creating of spot weld features and perimeter welds. Spot welds can be added if surfaces to be connected are separated with small gap. Spot weld suits well when trying to simulate amount of spots required for connection or just for connecting two separated surface. Perimeter weld connects also surfaces but it does it by extending edges of first selected part to second part. Those surfaces are then used as connecting elements between two parts. Perimeter weld works otherwise well but because it uses edges as connecting features, it causes misleading and incorrect high stress levels near weld connection (singularities).

There is also a feature based weld creation where the weld connection is defined by selecting the weld feature created in welding application. That possibility works similarly to perimeter weld by creating a surface like connection element along the outer surface of the defined weld. In the test simulation the result was that the actual parts were able to penetrate each other because the created fillet weld connects only two edges on weld outer surface, not the whole weld area itself. That kind of weld idealization does not work realistic way so the results obtained with it cannot be good either.

3.7.3 Rigid links

Rigid link is an easy and fast way to connect two features. It suits well for creating connection from one part to another if there is not overlapping surface or weld connection cannot be used. In Simulate rigid link works by connecting all DF of selected element nodes together. That means in practical that if two surfaces are connected both will become rigid and will move like part of common rigid body. The surface with rigid connection behaves in a similar way with the constrained surface. It cannot deform or show any stress which causes incorrect high stress levels near edges of those surfaces.

The rigid link is a really rough idealization of connection. However in most cases that idealization is good enough and in many cases it is only easily working option to simulate needed connection. It has not got usage limits almost at all since the rigid link can be added basically between any features.

The typical use for the rigid link is connection between the screw hole and the part to be attached with screw. Rigid links should not be added onto a really large surface or onto a surface where should be calculated some results. *Surface region* is tool that can be used to cut needed area from bigger surface to be used as reference.

3.7.4 Weighted link

Weighted link is a tool that can be used to connect masses or loads acting at single point to the selected geometries. The weighted link connects a point to multiple geometries so that the point will follow the average movement of the target nodes. Because of that average movement interpolation the weighted link does not stiffen the structure like a rigid link. The weighted link transfers only translational forces and for each link can be selected what directions of XYZ are linked.

3.7.5 Fasteners

There is also idealized *Fastener* connection in Simulate. Fastener definition allows the creating of idealized screw connection between two solids. It is also possible to use for connecting two shells from edge to edge or point to point. Fastener is simulated as a spring between fastened components.

Some comparison between fastener, rigid link and bonded connection were done. There were two identical plates in test simulation which were run three times and the plates were connected differently each time. The results of the simulation are in Figure 19. The simulation with two bolts as fastener was slowest to solve with 85 seconds of calculating time while it gave the result of 1.74412 mm maximal displacement. The next simulation was made with two rigid links between concentric holes. The calculation time was only 8.6 seconds and the displacement differed 0.24 %. The third simulation was run with bonded connection between surfaces. Calculation time was 9.2 seconds and the difference in the result compared to bolts was 2.7 %.

The results of the test simulations show that although fasteners are easy to add and simulate, it is not necessary to always use those for connections. Fasteners can be used if the purpose of the simulation is to simulate bolt or screw itself. It can be used to find how many, where, and which size of connection elements should be used. If the purpose of the simulation is related to the connected part, similar results can be found faster with other connections like bonded constraints or rigid links.

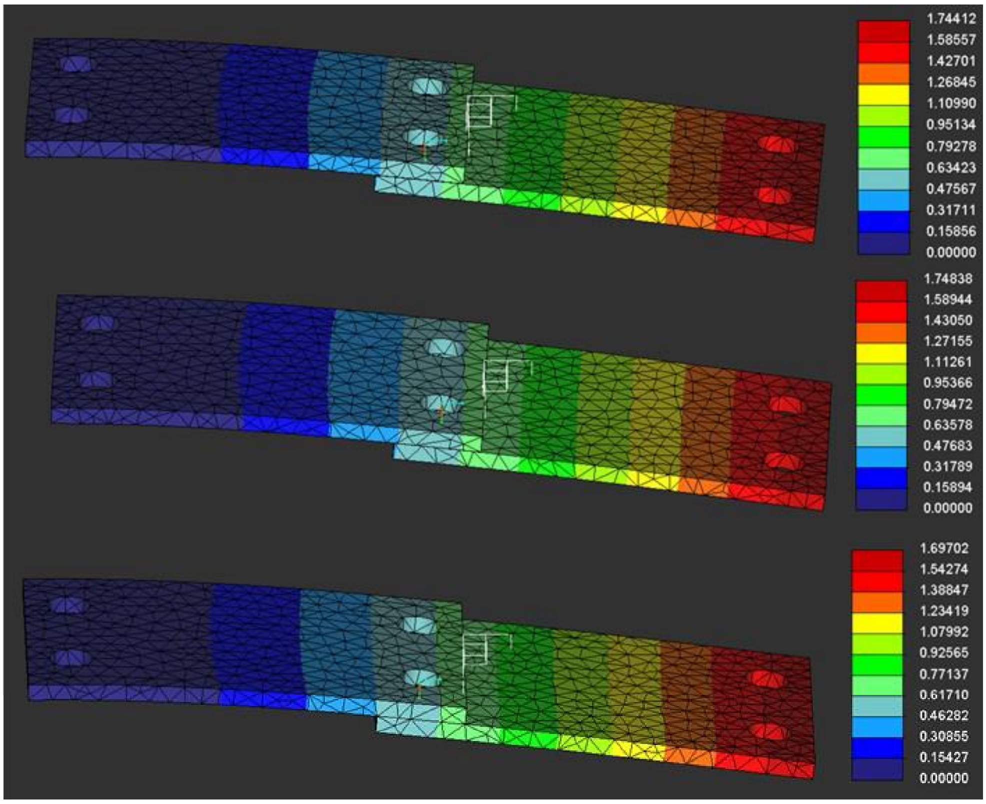

Figure 19. Displacement results [mm] of three different connections. Bolt connection on the top, rigid link in the middle and bonded connection at the bottom.

3.8 Measures

Measures are used to calculate additional information from the model during simulation. Simulate calculates by default stresses, displacements and strains over a model. Those can be viewed as 3D figure plot. Measure collects data as numerical information and can be viewed as a 2D graph. Measure has to be defined before simulation.

Measures are needed for example when finding reaction forces at constraints. The simulated part can be set to deform with forced displacement constraint and force needed to determine can be found with measure. Measure is also useful if the result needs to be in 2D graph or in numerical format.

4 CREATING AND RUNNING SIMULATION

After the simulation model is completed it is time to set up the simulation itself. Setting up the simulation starts by creating a new analysis or design study. It can be set multiple analyses for the same simulation model. Each analysis should be named to preserve the simulation results for later review.

Simulations can be divided into analyses and design studies. Analysis performs single simulation while design study runs multiple simulations. Design study changes given parameters between simulations and find how that will effect to the result.

A new analysis or design study can be created in *Analyses and Design Studies* window. The desired simulation type is selected from *File* drop down list and the corresponding settings window will open. In this chapter, how to define and run static analysis and different types of design studies are presented. A short summarization from that is included in Appendix 1.

4.1 Static analysis definition window

In this thesis, work will be focused on static analysis and other analysis types are left for readers to study on their own. Static analysis finds static solution for the given problem, where all forces and deformations are in balance. Mass of structures does not matter because the parts are not moving (unless gravity is used as a load).

4.1.1 Linear and nonlinear analysis

Static analysis can be linear or nonlinear. In nonlinear analysis, result solving is divided into multiple steps. Load, which can be force or displaced constrain, is applied gradually on the model during steps. The first step has no load at all and the last step has full load, while model is updated between each step. Nonlinear analysis approaches correct solution like an imaginary man walking on the curvy road step by step while updating direction after each step. That way man will be sure not to get lost from the road to the correct solution. Linear analysis instead applies full load at once and it can be used only for cases where way from the start to the solution is straight. It calculates the direction and the distance of the solution and goes right to the result with one step.

Simulation is created as linear by default, if there are not any nonlinear features in the model. Linear analysis is always faster to run than nonlinear so it should be used if possible. Nonlinear analysis is needed in some specific cases so the simulation is possible to be solved.

There are five different reasons for nonlinearity in static analysis. First is large deformation where nonlinear analysis can be used for more accurate results. With the *Calculate Large Deformation* setting the selected result is calculated with multiple steps and mesh (and stiffness matrix of the structure) is updated after every step. Therefore the model's mesh and stiffness will change during model deformation leading to more accurate results. Other reasons for nonlinearity are contacts, plasticity, hyperelasticity and nonlinear springs. If one or more of those is defined in a simulation model, Creo Simulate automatically sets nonlinear analysis as the selected analysis type.

4.1.2 Load and constraint selection

It is possible to create multiple load cases for the same model by creating new load/constraint set for another load case. The desired set can then be selected in the simulation definition window. It is also possible to combine multiple sets to one and use for the simulation. Simulate will ensure the correct selection since it does not allow to create a simulation without sufficient constraints and loads.

If analysis is linear, it is also possible to run the simulation with multiple load sets at once without combining those together. The desired combination of loads for result window can be then selected after the simulation. This is possible because linear analysis uses the principle of superposition (NPTEL 2009). According to the principle a displacement in the structure with all loads affecting simultaneous is sum of displacements caused by loads affecting individually.

For linear analysis it is possible to use *inertia relief* as a constraint. It can be selected in the static analysis definition window.

4.1.3 Convergence method settings

Creo Simulate has three different convergence methods for simulation. Those are called quick check, single-pass adaptive and multi-pass adaptive. These methods are based on the adaptive p-type elements of Simulate. The polynomial degree of element edge shape functions can be increased in order to refine results on certain areas. (Taylor & Zienkiewicz 2000: 402; PTC 2016a; Chapter 2.2.2 page 18)

Quick check uses the polynomial order fixed to three for every element (PTC 2016a: Convergence options for structural analyses). It should not be used for final result calculation. Instead, it can be used to check if the simulation runs without failures and if the whole model is deforming reasonably.

The single-pass adaptive method is a default option for simulation convergence. In practice, it calculates the simulation two times. First it runs the simulation with the polynomial order of three. After that it determines the local stress error estimation and based on that, increases the polynomial order on critical areas. The final pass is then calculated with these increased p-levels. (PTC 2016a: Single-pass adaptive convergence method).

The multi-pass adaptive runs simulations until the given convergence limit or maximum polynomial order is achieved. Analysis convergences in multi-pass method when the percentage difference in results between the current pass and the previous pass is smaller than the defined limit. Default convergence percent is 10 and the Simulate's user's manual recommends it to be set between 1 % and 25 % (PTC 2016a: Multi-pass adaptive convergence method)

According to the PTC user's manual (PTC 2016a), the user should use the single-pass adaptive method since it gives similar results to the multi-pass adaptive method while run time is usually 10 % shorter. That can be also affected by meshing, since reasonably sized and locally refined mesh should give good results even with the quick check method. The multi-pass adaptive is necessary mostly when precise accuracy is required. In many cases, the value for the load is roughly estimated so a small difference in stress accuracy does not actually mean anything.

4.1.4 Result settings

In output tab, it can be selected what results Simulate is calculating during the simulation. The available options are stresses, rotations, reactions and local stress errors. The simulation is solved by calculating displacements so those are always available in the results although not selectable (University of Oulu 2014: 19).

Stresses are the most important values when studying structure strength so that option should always be selected. Rotations are calculated only to elements with rotational DF, not for solid elements even if the option is selected. The reactions option calculates the reaction forces and moments at the constrained points and edges. If constraints are added to a surface, this selection does not calculate anything. One possible way to find reactions at surface constraints is to set up a measure for that. Local stress error calculates estimations for errors in local stress. It can be used to check the estimated errors locally. It shows the estimation on the element level which can be used as a reference for mesh refinement.

Plotting a grid value defines the amount of straight lines on the element edge used to plot results. Adding more plotting points causes increased need for computing resources and increased time for result window generation. Plotting grid value can be changed to a minimum of two when using fine mesh for large models. With really coarse mesh, a low plotting grid leads into a coarse plot (Figure 20). Higher plotting grid values can be used then to increase result plot accuracy, although the default value of 4 gives usually sufficient accuracy.

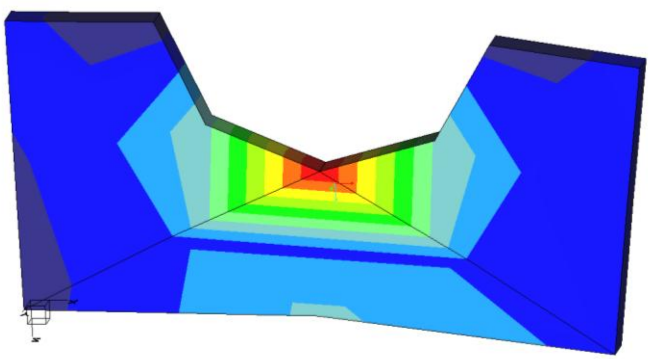

Figure 20. Coarse mesh plot with plotting grid setting of two.

4.1.5 Nonlinear analysis step settings

The amount of steps required to calculate a correct result depends a lot on the simulation type. If only nonlinearity in the model is large deformation, can the correct solution be found approximately in 5 to 50 steps, depending how much deformation there is in the structure. Contact analysis has to have more steps, because if the step is too big, the correct result can be missed. Contact between the surfaces can fail with too big steps, because contact pairs can jump over each other without actually contacting.

There are automatic settings for output step definition in Simulate but those do not work and should not be used at all. From U*ser-Defined Output Steps* the total number of steps can be set manually with *User-defined Steps* button. Every step has a value which corresponds to its calculation time. The desired step configuration can be made easily by defining the time for the first and the last step and then using the *Space Equally* button. The Simulate scales the given load for the steps according to the *Time Dependence* option and the default ramp gives the full load at time of 1. Therefore, with default settings the first step should be always set to a 0 and last to a 1. If bigger time is used, Simulate will continue increasing the load linearly and giving double the load at the time of 2.

4.2 Design studies

Design studies can be used to find the best value for the part geometry dimensions. Design studies will help to find what would be the simulation result, if the value of round or thickness of wall is changed. There are three different types of design studies to use in Simulate.

4.2.1 Standard design study

Standard design study will answer to question “what if I change that dimension?” It changes the selected dimension to a new value and then simulates the new results with the modified geometry. Standard design study is easy to set up and it makes it possible to study the effect of the changes without modifying the original model geometry.

Before it is possible to define a new *standard design study* static analysis has to be defined. When the design study runs a simulation it actually runs the selected standard analysis with modifications defined in design study. *Standard design study* allows the user to select one or more dimension (variable) from the model to be changed for the design study simulation. Current values are shown for selected dimensions and new values can be defined in the *Setting* column.

Whether the actual analysis will be run or just regeneration, can be selected from *standard study definition* window. When *Regenerate* is run nothing is simulated but the geometry is regenerated according to the *Setting* values. The *Regenerate* option can be used for two purposes. Before the actual simulation, it can be used to check if the given dimension change will lead to the correct geometry. After simulations, when the best values are found, it can be used to change the original geometry to a new, optimized design.

4.2.2 Sensitivity design study

Sensitivity design study is used for studying how a change in dimensions affects the predefined or user defined measures. There are two types of sensitivity studies, global and local. In global study, the maximum and minimum values are given to the studied variable. The study will then run multiple simulations changing the selected variables gradually from the minimum value to the maximum. The amount of simulations can be selected with the *Steps* setting. The result from a global sensitivity study is a graph where the simulated variable can be plotted against the selected measure. For example, it is possible to calculate and plot the relation between wall thickness and the total mass of the part.

The Local sensitivity study calculates a slope between the selected variable and a measure at the defined point of the variable. It can be used to find how much each of variables affects the specific measure. The results can be found from the *Summary* tab of the *Run Status* window as a slope number. However, the local sensitivity study is not very helpful feature at all, since the numerical information it gives is not really needed for anything. According to the Simulate manual (PTC 2016a: Local sensitivity study), it can be used to find which dimensions of the model should be optimized, but usually those dimensions can be easily found just by looking at the model structure.

4.2.3 Optimization design study

Optimization design study is a simulation which automatically modifies the given variables to find the best solution in the given limits. There are two different optimization types called feasibility and optimization.

The feasibility study aims to find whether there is a solution that will be inside the given *Design Limits* and the limits given to *Variables*. A design limit can be, for example, maximal displacement or maximal stress and it can be defined for any of the predefined or user defined measure. Each included variable has a minimum and a maximal dimension value which can be adjusted.

The optimization study has the same design and variable limits than feasibility study but it also has a goal. It can be, for example, minimal total mass. The optimization study then tries to find a solution which is closest to the goal while still inside given limits. It suits well for cases where more than one variable is going to be optimized and it is difficult to manually determine the best combination of variable values.

4.3 Running the simulation

Once the simulation model is ready and the simulation definition done, it is time to run the simulation. To ensure a fast simulation, a couple of settings need to be checked before starting the run.

4.3.1 Run settings

There are couple of important settings in the *Run Settings* window, which can be opened by selecting *Run > Settings…* from the *Analyses and Design studies* window. First are the *Directory for output files* and the *Directory for temporary files*. Simulate uses the current working directory for simulation files if nothing else is defined. The output folder can be defined manually for every session or a default folder can be selected with the configuration editor (Chapter 6.4 page 73). Simulate tends to fill the working directory with obscure files so by using separate folder for the simulation results will clarify the handling of the result files. All simulations can be saved to the same location if a unique name is used for every simulation. Using the same name again will overwrite the results from the previous simulation.

Another important setting is *Memory allocation*. It defines amount of RAM to be reserved for the solver and it has great effect on the simulation run time, especially with big models. The default setting is 512 MB, which is way too small considering the typical total capacity of memory nowadays (8 - 16 GB). According to the Simulate manual (PTC 2016a : Guidelines for setting solram), *Memory allocation* value should be set to 0.25 - 0.5 times the equal machine RAM. In test simulations a value of 6144 MB was used on computer with 16 GB of total RAM, which worked well.

However, the *Memory Allocation* value should not exceed the amount of available RAM at the current moment. Defining more RAM than available, will slow down the simulation process to a point where almost nothing happens. If big models are already opened in Creo Parametric, it is possible that most of the total amount of RAM is already in use before the simulation run. Because of that it is good to check the available RAM from *Windows Task Manager* and set *Memory allocation* so that it is at least 1 GB smaller than the available memory at the current moment. If less than 2 GB is available for the simulation, it is better to close all other programs using a lot of RAM or restart Creo and open only the model to be simulated to also release memory. With the correct memory allocation, the simulation will run fast while it is possible to do other things with the computer.

It is also possible to select *iterative solver* to be used instead of the default *direct solver*. Usually the direct solver results to a faster simulation with less RAM and disk memory usage. Iterative solver can be tested if the simulation does not run properly with direct solver, but in the performed test simulations that was not necessary. (PTC 2016a: Use iterative solver).

4.3.2 Launching the simulation run

The simulation can be started by selecting the desired analysis from the *Analyses and Design Studies* window and then clicking the green flag. It is possible to run multiple analyses at the same time but, according to Simulate manual (PTC 2016a), simulations are completed faster when running one after another than simultaneously. Simulation can be stopped by clicking the red flag if necessary.

4.3.3 Simulation progress following

The simulation status can be followed on the *Run Status* window. The status window has three tabs and those all will be updated while the simulation is progressing. The *Log* tab collects information of the time needed for individual parts of the simulation. The *Checkpoints* window is updated most often and it also collects information from the convergence. That is the place to be checked if simulation is not progressing at all or the solution does not converge.

The most important tab is the *Summary* tab. The results of the user defined measures are shown on the *Summary* tab with the other default measures. There is also a lot of other information related to the simulation including element information and error estimation.

4.3.4 Diagnostics window

After the simulation is completed or failed, a diagnostic window will appear. A single part of the analysis is marked with a blue color if it was successful, yellow if something needs to be paid attention to and red if run was a failure. Diagnostics will then try to specify the problem, in most cases successfully.

One common diagnosis with the yellow marker is “One or more measures were evaluated at (or close to) results singularities…” This means that there are some areas in the model where the stress levels are high because of singularities. This will make the calculated maximal stress on the model to be inaccurate. The inaccurate maximal stress should not be then used as a validity measure of the design.

5 VIEWING AND UNDERSTANDING THE SIMULATION RESULTS

In this chapter required settings for viewing results are presented. It is also covered how to understand the results given by the simulation. These are also summarized in Appendix 1.

5.1 Creating a Simulate results window

There is another application called *Simulate Results* for viewing simulation results in Creo Simulate. Results can be opened in that application directly from Simulate. Another possibility is to open the application first and then open selected results. While *Simulate Results* is opened, all other Creo windows are disabled.

It is possible to open multiple views/subwindows to a Simulate Results window, and compare results from two or more different simulations side by side. A result view can be defined manually, with default settings or with a predefined template. Next the settings required to define a result view will be presented.

5.1.1 Display type

There are four different types of result displays. The *Fringe* is the default option and it shows the results as colored 3D model, where different colors mean different values for presented quantity. Stress, displacement, strain, p-level and strain energy can be plotted as fringe.

The *Vector* option creates a plot where value is represented with colored arrows. Arrows are plotted inside the wire-framed model and show also the direction of the represented value. Stress, displacement and strain can be plotted with vectors.

The *Graph* option creates a 2D graph of the defined measure or along a defined curve. The represented value is on the Y axis while the simulation time or curve distance is plotted on to the x axis. Features in the *Graph* option are limited. It for example, does not allow the user to create more than one curve to the result window.

The *Model* option was found basically useless. It can show displacement, which it does only by showing blue wire framed model with maximal displacement in numerical form. Another thing it shows is reactions in point constraints. Since point constraints will cause singularities and are thus not recommended to use, the point constraint reactions are needed only in rare cases.

5.1.2 Display location

The *Display location* setting allows the user to limit where the results are shown in the model. The default option is *all* and it is not necessary to be changed unless the model is so big that Simulate has problems showing it or opening the result window takes lots of time. Limiting the display location removes unselected geometries from the result window.

5.1.3 Display options

Display options include additional selections concerning how results are shown. There are many options but the most important are *Deformed, Show Element Edges* and *Animate* selections.

When the *Deformed* option is checked, the model will be shown as it would be when deformed by the load. The scaling factor defines how much the actual deformation is scaled. Scaling is necessary since usually the deformations are small and without scaling it would be hard to see how the model deforms. By default, scaling is defined so that the maximal displacement is set to be 10 percent of the maximal dimension in the model. Unscaled deformation can be shown by deselecting the % box and setting 1 as the scaling factor.

The *Show element edges* option draws elements used in the simulation to a result window. That option allows the user to see how the results are depended from mesh because all values shown in result window are calculated in mesh nodes. It also can be used to check how good the created mesh is.

Selecting *Animate* will create an animation from the model deformation. Nonlinear analysis is calculated in multiple steps and animating will show all those steps in the result window. The results are possible to be viewed as an automatic animation or steps one by one. *Animate* is really useful for nonlinear analysis, especially for contact analysis. The *Animate* option can also be used for linear analysis but it does not offer much additional information, unless the user is interested in stress levels with lower load levels.

5.2 Result window features

5.2.1 Legend

Legend is the most important setting in the result window. Simulate will create its own legend for every new result window with maximal and minimal stress and all other values linearly divided between those (Figure 21). This random legend value distribution makes it hard to compare results between two simulations so it is good to change the legend values into a more general distribution. That has to be done for every new result window, but it is possible to define a *mapkey* to speed up the process (Chapter 6.5.2 page 77).

The default setting for legend levels is 9, which can be modified manually or changed with the configuration editor (Chapter 6.4 page 73). A good selection for legend values is such that when the design/stress limit is exceeded, the color of that area turns into red. That way the user can easily see and understand the results.

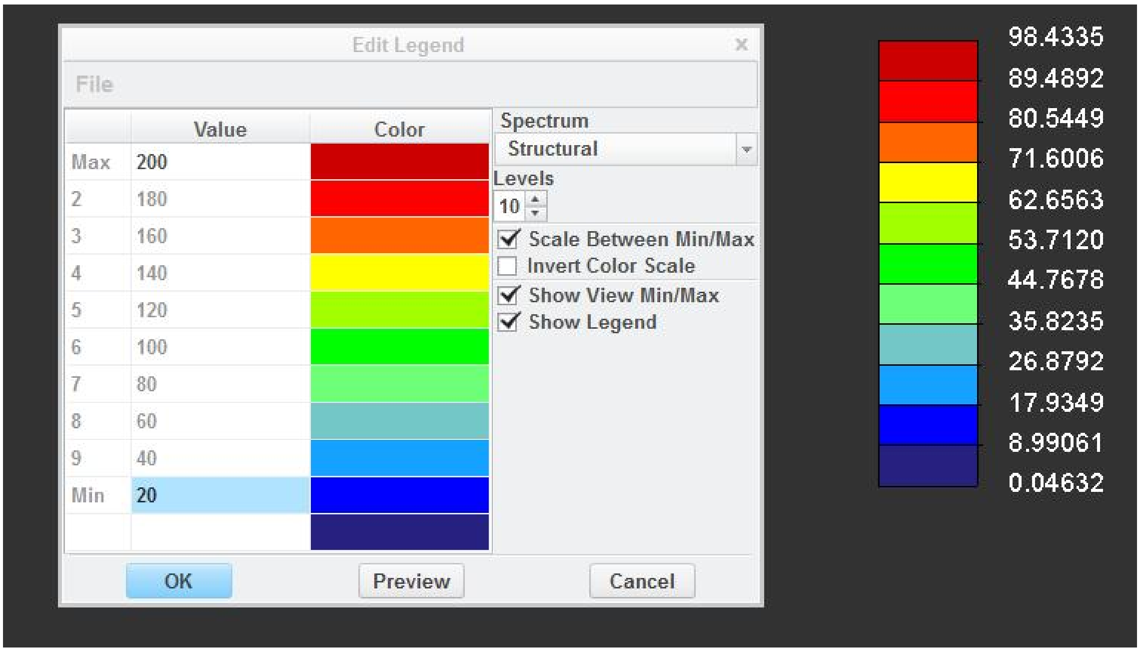

Figure 21. Example of legend settings.

5.2.2 Result query and reports

The approximate result can be defined with legend color. Accurate results are found with *Dynamic Query* which will show the exact result value in the selected location. There are also buttons to find location for smallest and largest value of results in *Query* setting group of *Home* tab.

The *Measures* button shows values for the all user defined and default measures in the model. Selecting a measure will show its location in the model with a gray dot. The *Linearized Stress* option creates a 2D graph from the stress level on line between two defined points. It is possible to select which of the stress components is used to draw graph.

5.2.3 View options

There are many settings that can be used to define how part will be shown in the result window. In this chapter a couple of the most useful settings are explained.

On the appearance tab there are settings for *shaded, continuous tone, transparent* and *Exploded*. *Shaded* will add effect of light on view, which helps to perceive the shape of the model. For flat surfaces it can cause problems to distinguish different colors from certain angles. Turning it off will remove that problem. *Continuous tone* will change the coloring to a rainbow-like smooth change from color to another. Although it looks good, it also makes it really hard to see actual results contour values so it is rather useless.

The *Exploded* setting is really useful; it makes an automatic exploded view for simulated assembly. That makes it possible to see results on parts that are otherwise hidden. *Transparent* allows to see results in parts inside the assembly, but it tends to be heavy to rotate model with it.

It is also possible to create *cutting* and *capping surfaces* in Simulate. Those make it possible to see what results are inside the part while the normal view only shows results on outside surfaces. *Cutting surface* shows one straight and thin result surface from the model. It is possible to create multiple *Cutting surfaces* into one view. *Dynamic* button allows the user to move created surface with mouse and see how results change inside the part volume.

Capping surface differs from cutting surface so that it cuts away volume above or below the surface while cutting removes both. The cutting view can be done also with *isosurface* selection. *Isosurface* creates a cutting surface along a defined result value in the result plot. *Isosurface* can be used for example with capping surface definition to plot all volumes that exceed certain stress level. An example of a capping surface plot is shown in Figure 22.

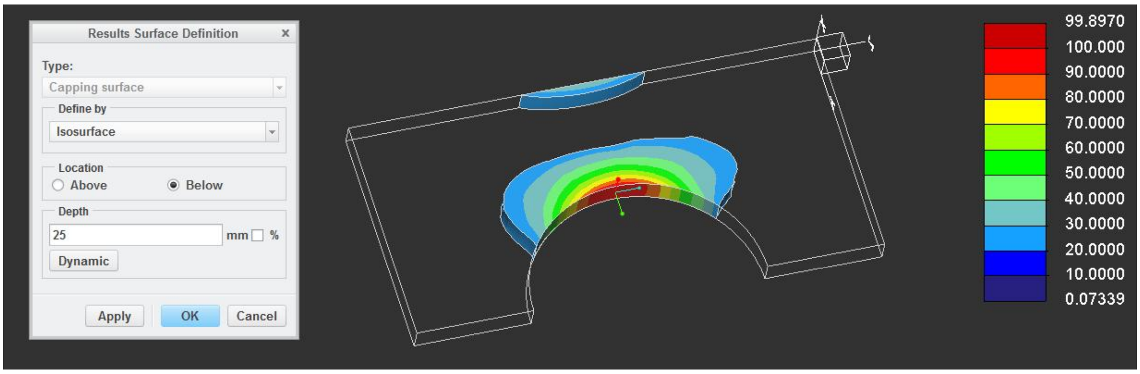

Figure 22. Example of capping surface plot.

5.2.4 Result window template

It is possible to save the defined result window as a template in Simulate Results. Theoretically the template is a universal definition type and possible to use for all different models, but that is not how it works in Simulate. When saving a template, a couple of features are saved from the simulation to the template, including maximal and minimal stress values for the model. When the next model is opened with the created template, those saved values are shown instead of correct values of the current model. Instead of the poor templates the user can define a *mapkey*, which will define similar result windows to all models (Chapter 6.5.1 page 75).

5.3 Result validity verification

Before the user can use simulation results it is good to check that the results can be trusted. A couple of things could reveal whether the simulation results are incorrect or inaccurate. The user has to remember that although the simulation can give good results, it is always possible to get completely false results.

5.3.1 Convergence analysis

Convergence analysis can be performed on the model in order to reveal possible failures in the model. H-convergence analysis is presented in chapter 3.5.5 on page 41. However, Simulate uses p-adaptive convergence by default so the results should be converged without separate analysis by the user.

5.3.2 The magnitude of the results

The first thing to be checked from the result plot is the magnitude of results. If the values given by the result window are unusual or do not feel to be even close to expected, it is highly possible that something has gone wrong.

The user can do couple of things if a simulation gives suspicious results. All defined parameter values should be checked that they are given in the right units and correctly. Element mesh can be refined or element type changed. If results are still the same after different simulations, it is possible to confirm results with another program or rough manual calculation.

5.3.3 Deformed state

The simulated model should be checked also in deformed state. Deformation can reveal problems especially related to failed connections and links. Those problems can be detected as penetrating or separating parts.

Although connection is defined properly, it is possible that Simulate still fails it. Failed connection can be tried to fix in a couple of ways. The connection can be defined manually, if the automatic bonding does not work. In some cases the bonded connect can be replaced with the weld connection. Also recreating the assembly can help in some cases.

5.3.4 Mesh quality

Poor mesh with narrow shaped elements does not give really good results. There are p-type elements used in Simulate which allow more loose shape restriction for elements compared to traditional FEM programs. However, the user should still check that there are no failed elements in the geometry and that the created elements are representing the geometry without errors.

5.3.5 Error estimate

Simulate makes an error estimate after simulation, which can be found from the *Summary* tab of the *Run Status* dialog box. Error estimate excludes regions of possible singularities from the estimation. If the error estimate is too large, results can be refined with a finer mesh or with using multi pass adaptive method instead of single pass.

The error estimate is a measure which tells how accurate the results are from the view of the simulation program. The user should understand that it does not tell of errors relative to the actual stress in the real part. In other words, it is possible that error estimates show that results are good while results can be totally unrealistic and false.

5.4 Understanding the results

5.4.1 Displacement results

Displacement results simply are showing the amount of displacement in the structure caused by the load. Displacement results are useful in many cases. It is easily understandable unit and so gives for the user a clear idea right away how much the simulated part is deforming.

In some cases, a part has to be designed so that it does not feel too bouncy or too flexible. Displacement result is only of the available results which can be used to estimate flexibility of the structure. Multiple simulations can be run either with different load values or with different structure thickness. The suitable geometry for structure can then be evaluated by comparing maximal displacements between simulations.

5.4.2 Stress results

Multiple different kind of stress values can be plotted from the results. Most common are the von Mises stress results which can be used to estimate failure of ductile materials. In Chapter 2.4 (page 9) is presented different failure criterions which can be used to estimate if the material is going to fail.

Stress peaks caused by singularities (Chapter 2.2.3 page 20) can be ignored when viewing stress results. Otherwise the basic idea of viewing results is simple. Based on the material of the part which criteria is going to be used to estimate failure is selected. Corresponding stress result with selected criteria is then plotted and stress levels in the structure are compared into limits of the selected criteria. If the limit is exceeded, there will be failure or yield on exceeding areas.

When linear material model and linear analysis are used, stress results which are above material yield stress are not accurate. Therefore with linear analysis it is impossible to find the exact load which is going to break the structure. Also in the cases, where some areas of the part are above the material yield stress limit and some areas below that, it is hard to say if there is going to be a visible deformation in the part or not.

5.5 Saving and exporting results

The simulation model itself will be automatically saved to an original Creo part or assembly file. That makes it possible to recalculate the created simulation afterwards. Results can be also saved, which removes the need for recalculation and saves time if the results have to be checked later.

5.5.1 Saving results

If results are needed later, the most important thing to do is to rename the created analysis or design study with a unique name. A unique name will ensure that results are not overwritten by another simulation. When the simulation run is done, result files are saved to a folder defined in *Run Settings* as *Directory for output files*. Directory can be defined manually every time or a new default folder can be defined with configuration editor (Chapter 6.4 page 73). The model geometry is also saved to the same folder with results, which makes it possible to view the results even if the original part is not available.

The result window definition can also be saved as *.rwd file. That file contains information defining which model should be opened, what results are shown and what settings are used for them.

The results are also possible to save into Windchill. The results are compressed into *.mrs file, which can be loaded into Windchill. The file is created in *Analyses and Design Studies* window with *Vault Results* option in the *Run* dropdown list. The created file will be found from the current workspace. The result file can be opened with *Simulate Results* application and *New* option.

5.5.2 Exporting results

The results can be exported to different forms and reports. It is possible to create a web page report (*.html file). Animations can be saved as different video formats. The whole model geometry with result colors can be saved as Creo view file (*.ol), which can be viewed with PTC Creo View Express.

Exported results are made with *Save As* option in *Simulate Results* tool while the model to be exported is opened. The desired output type is selected from the *Type* dropdown list as file format. A Creo view file is easy to create and does not require any extra settings. If the saved model is animated, the frame that is currently displayed in the view will be saved.

Web page report exporting opens settings window, where what results are shown on the report can be selected. Exported image/movie size is defined from the *Preferences…* button. A setting of 920x1920 pixels worked well, but best option depends on the monitor resolution. The image format can be selected between jpeg and bmp and it is saved from current frame displayed on the view. Movie speed depends from the amount of frames defined in *Result window definition*, something between 16 - 32 gives usually suitable speed.

Movie export feature has not so far been updated to support current video resolutions and it even lacks the option to keep the original aspect ratio automatically. Better quality videos can be made with web page export, but the duration cannot be changed then. However, movie exporting starts by selecting *TLMPEG* (*.mpeg) or *Movie* (*.avi) and giving a name for the video. The mpeg file format creates a more packed video with smaller file size. It is possible to change video image size by selecting *Custom* from *Output Setting* in *Movie Export* window. The image size has to be defined right in order to keep original aspect ratio. Best video resolution is 352x720 pixels with 16:10 aspect ratio monitor. Animation speed in exported video can be changed with two settings. The *FPS* setting defines how many frames will be shown in a second so a lower FPS will produce slower animation. Speed can be adjusted more also by changing the amount of frames defined in the *Result window definition.*

6 EXAMPLES AND TIPS FOR USING SIMULATE

In this chapter a couple of different type of simulations that Simulate is capable to perform is presented. Guides are not step-by-step for these simulations, since the purpose is only to introduce these advanced simulation types. In the chapter methods that the user can use to ease simulation process are also presented. Those include modifying configuration settings in order to change certain default settings and creating mapkeys for repetitive tasks. With the given tips it is possible to speed up simulation process and make using Simulate more comfortable.

6.1 Simulation with forced deformation

In some cases, the designer is interested in the force needed to deform a part with a specific amount of displacement. For example, a part is used as a spring and compressed during assembly. The designer needs to find the material thickness for the part so that force needed in the assembly is suitable. One example of that kind problem is in Figure 23. There is a wall with spring features on both ends compressed 1 mm with the lid. The wall is kept in place with supports on both ends, allowing vertical movement for the wall.

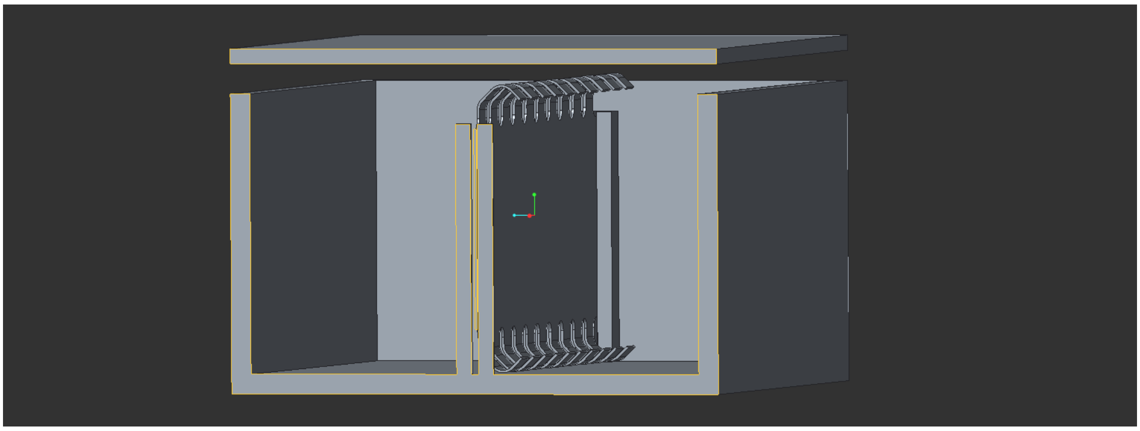

Figure 23. Spring simulation problem.

One possible simulation model for that problem is in Figure 24. The model is constrained from both side surfaces in z direction and in x direction only from one side. The surface contacts between the box and spring features are idealized with constraint applied into the edge. This will allow spring features to turn while the part is deforming. If there is no suitable edge for the constraint in right place already, it can be created with surface region. On the bottom side of the plate a rigid constraint in y direction is added and a forced displacement of -1 mm is added on the top according to Figure 24. Force measurement is added for moving constraint.

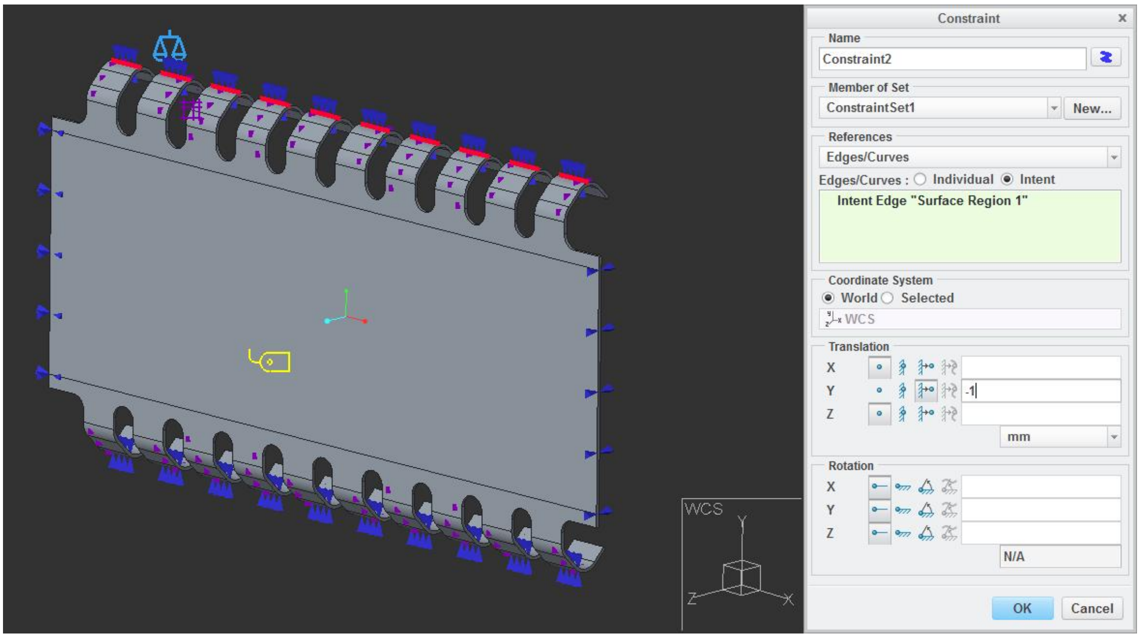

Figure 24. Simulation model of the spring wall.

If the simulation includes relatively large deformations, nonlinear simulation gives more accurate results. The situation can also be simulated with sensitivity design study using linear analysis. Thickness of the wall can be added into the design variable and after the simulation it is possible to draw a relation curve between wall thickness and reaction force. More accurate results can then be calculated with nonlinear analysis if needed.

With thin sheet metal parts faster calculation can be achieved with shell elements or thin solids. For example this simulation model is 4 - 20 times faster to solve with shell elements than solid tetras. Reduction in calculation time is more when the thickness of the plate decreases. A shorter simulation time is advantagious especially when calculating design studies or nonlinear analysis. It is not possible to do nonlinear analysis using shell elements but thin solids can be used instead.

6.2 Snap feature with contact

Snap feature is locking feature that can be used for attaching a part into an assembly or making a joint between two parts. An assembly with snap joints is fast to assemble and usually possible to be unassembled. However, a snap has to be designed with care to ensure proper operation. A FEM simulation can be used to check validity of the design. A snap feature is simulated in following example (Figure 25).

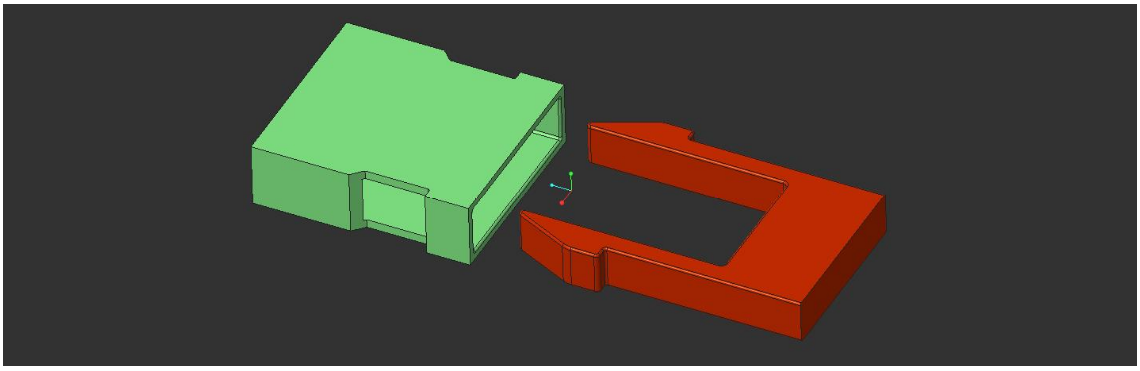

Figure 25. Assembly of the snap feature.

It is possible to simulate a couple of different things from the snap feature. Strength of the male part can be simulated with forced displacement applied onto the tip of the snap feature. Displacement can be easily calculated based on the snap geometry assuming that the female part does not deform at all.

The whole snapping process can be simulated with static nonlinear analysis, which gives stress distribution in parts during assembly process. This is done in simulation by adding contact interface between contacting surfaces. The female part is then constrained rigidly while the male part is given forced displacement.

In many cases it is also good to know the force required for pressing the snap feature together. Force is possible to get by adding force measure with correct direction component into the moving constraint. Default contact interface is frictionless, but it is possible to add friction into the interface to get more realistic results.

Full snap simulation with finite friction contacts is relatively heavy to calculate so the simulation model has to be created carefully. All unnecessary features are better to be removed from the geometry. Rounds are applied only to functional edges or edges on high stress areas. Mesh refinement can be done to the contact surfaces and high stress areas, otherwise mesh is best to keep relatively coarse. If the model is symmetric, the calculation time can be reduced significantly with a half model.

The simulation model created for this problem is shown in Figure 26. The model is cut into half in assembly mode with solidify (chapter 3.1.2 page 30). The female part is constrained from the left end surface rigidly. To reduce unnecessary calculation, the male part is placed really close to the female part but without penetration. The required movement to connect the snap joint is measured and applied as forced displacement into right end surface of the male part. The cutting surface for the female part is constrained with symmetry constraint and for male part with fixing in X direction.

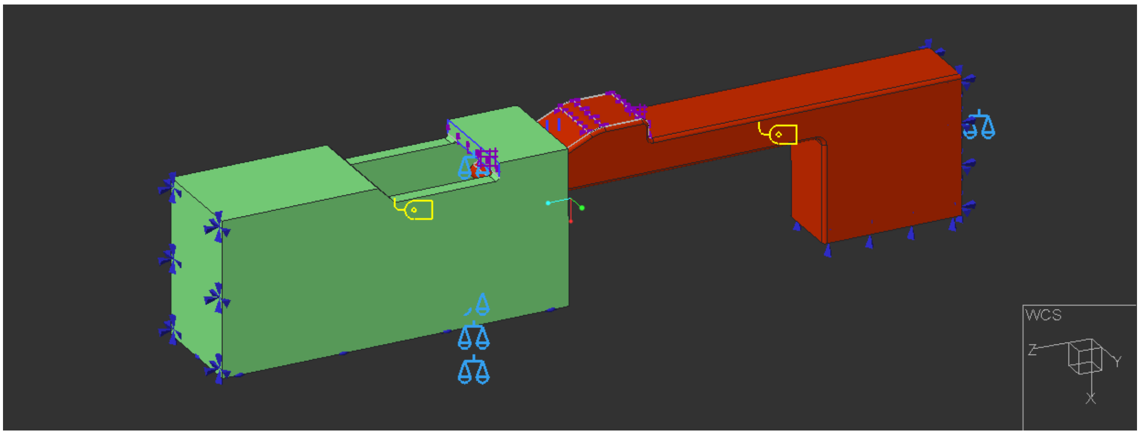

Figure 26. Simulation model of the snap feature.

Contact has been created between two intent surfaces which removes the need to define contact multiple times. The intent surfaces can be created with *Reference* tool (Chapter 6.6, page 78). *Maximum Element Size* limit of 2 mm was given to contact surfaces and no other mesh controls were added. Simulation was run with 24 steps and 81 steps while displacement was 24 mm. With a lower step amount Simulate gave a warning that better convergence of the nonlinear solution may be achieved if the analysis is redefined with smaller time steps. However, the difference in results for measured pushing force was below 3 % so the amount of steps does not seem to affect result accuracy significantly as long as the simulation still runs correctly.

Results of this kinds of simulations should be viewed critically. Contact with friction is a challenging feature to simulate so results cannot be trusted without doubt. There can be things that simulation does not take into account, for example male part can stick into the sharp edge of the female part instead of sliding. Also coefficient of friction can be hard to define correctly. However, after a couple of successful designs where simulations has been used, the designer is able to see how simulations relate into real parts. That makes it possible for the designer to trust the simulation results and make new designs easier.

6.3 Assembly with press fit

It is also possible to do simulations for press fit joints. In press fit, a joint is made by assembling an axle into a smaller hole than diameter of the axle. The joint is kept locked only by friction caused by the compression in the joint.

An example of the simulation model is shown in Figure 27. Contact interface is added between the axle and the hole surfaces in the Simulation model. Contact interface is created without friction. Movement of the hub is prevented with spring to ground connections. Springs are not needed if the contact is created with finite or infinite friction.

Constraint and loads needed depend on purpose of the simulation. In this simulation the purpose is to find the contact pressure. Since the model with contact interface is nonlinear, the unconstrained model with inertia relief cannot be created (chapter 3.3 page 32). The stress into the model is actually created with initial interpenetration but Simulate does not recognize that as an obligatory load so a nominal load has to be added into the model. In this model the rigid constraint is added on to the left end of the axel and small 0.01 N force is added on to the right end.

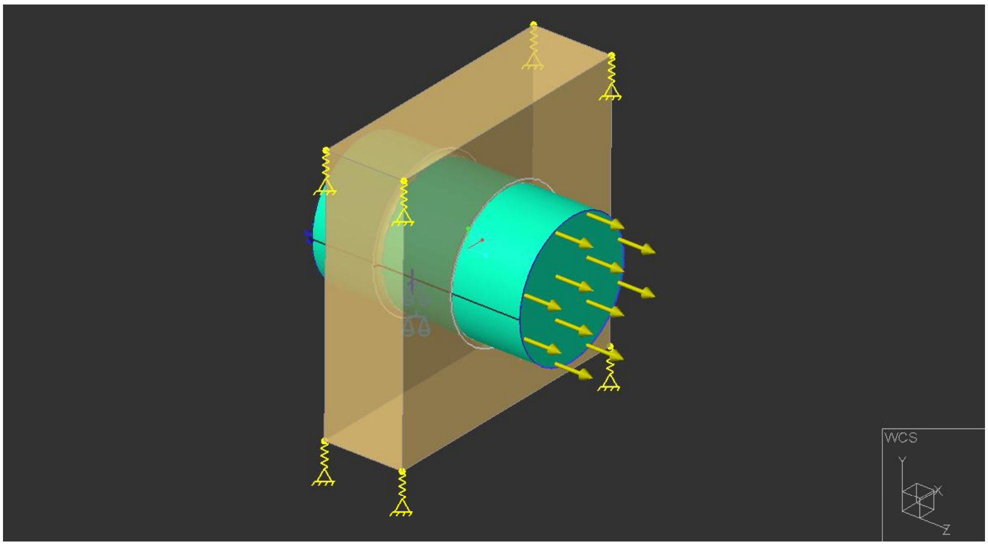

Figure 27. Simulation model of press fit assembly.

Contact interface requires nonlinear analysis, which is selected automatically in analysis definition. To activate initial interpenetration, the *Press fit* option has to be selected in analysis definition. Setting for maximum initial interpenetration has to be set according to the simulation model geometry. With the default value of 0 all interpenetrations are ignored. Output steps can be set to a minimum of two, since stress caused by press-fit is calculated already in step 1.

6.4 Configuration file

Default settings for different parameters are defined with the configuration file in Creo Parametric. The user can create their own config.pro file and save personal settings and mapkeys there. Configuration file has to be saved into the default working directory of Creo. The user defined configurations will be loaded only from there during startup of the program. The default directory of can be defined from properties of Creo shortcut by changing *Start in* option into path of the new directory.

There are a couple of useful configuration settings in Simulate. Defining those to correct setting by default removes the need to modify those manually all the time. Configurations can be changed from *File > Options > Configuration Editor*. The desired configuration option can be found easiest by selecting *All Options* and *By Category* from *Sort* and *Show*.

Sim_solver_memory_allocation is most useful configuration option for Simulate. It defines the amount of RAM given for the simulation solver. Setting this value to higher value than default 512 MB removes the need to manually check and change the value after every program startup. Rules for choosing sufficient value are looked over in chapter 4.3.1 at page 55.

Sim_pp_legend_levels defines the default amount of legend levels for the result window. The standard value is 9 while 10 is preferable because it allows to create uniformly distributed legend levels with 10 > 100, 20 > 200 etc. minimum and maximum settings. Legend level definition is discussed in chapter 5.2.1 on page 60.

Sim_run_out_dir and *sim_run_tmp_dir* can be used to define default directories for simulation output files. Those settings are looked through in chapter 4.3.1 at page 55.

6.5 Creating mapkey

A *Mapkey* can be used to speed up the repetitive process which can include clicking selection windows and giving keyboard inputs. In Simulate, a mapkey can be used at least for result window creation and for defining legend levels. The mapkey itself is a key sequence and when it is typed with keyboard, Creo automatically performs a defined macro operation. The mapkey definition can be found with tool search of Creo or from *File* > *Options* > *Environment* > *Mapkeys Settings*. The *Mapkeys* window shows all mapkeys currently in session, and a new one is created with the *New*... button.

First the defined key sequency for mapkey is to be recorded. The name and description are helping to identify what the particular mapkey does. The mapkey definition starts with *Record* and ends with *Stop*. *Pause* can be used to collect input from the user during macro definition.

For some reason the mapkey definition window itself tends to disappear. Opening it again does not help, because the window goes behind the other windows. It can be found again with the ALT + TAB key combination or with minimizing the current Simulate window and opening it again from the Windows taskbar.

6.5.1 Mapkey for result window definition

Next an example of mapkey creation which will create result window with two result views is presented. First window shows the von Mises stress results and second the displacement results.

Before the mapkey creation the *Analyses and Design Studies* window is opened and the desired analysis is selected. Next, mapkey definition with search tool is opened and creating a new mapkey is begun. Then, a sequence, name and description according to Figure 28 is given (step 1). *Record* is clicked (step 2) and the result window definition begins (step 3).

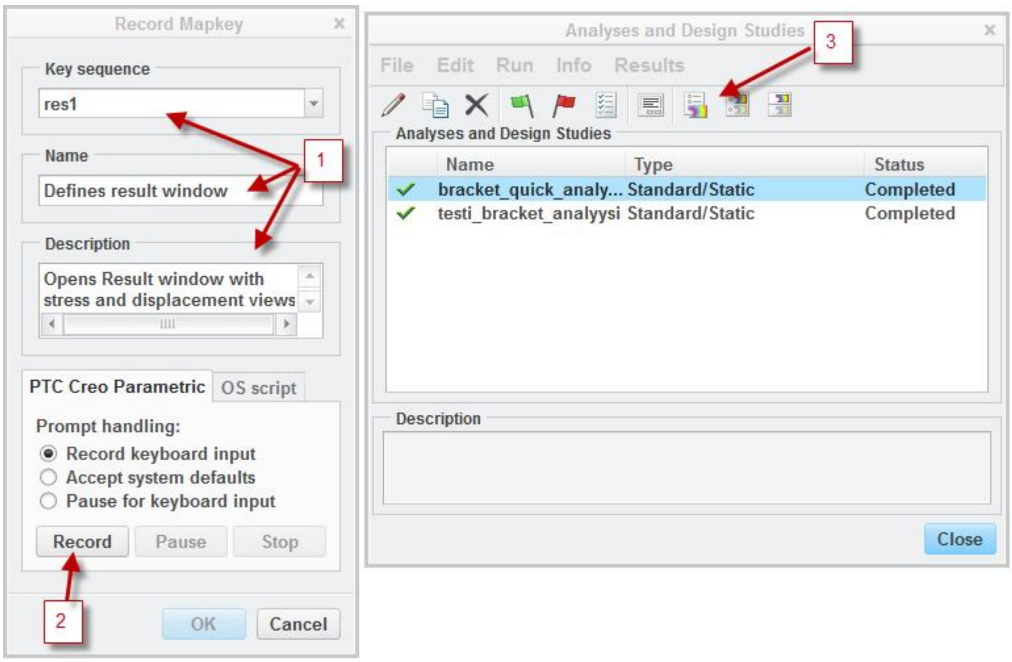

Figure 28. Result window mapkey definition part 1.

In the result window definition the units are changed to MPa (Figure 29, step 4) and the *Display Options* is selected (step 5). In the step 6 *Deformed* and *Show Element Edges* are selected and *Legend Levels* is changed to 10 if needed. After that the *OK and Show* is selected (step 7) which will open the first result window. The second result window is created as a copy of the first one (step 8).

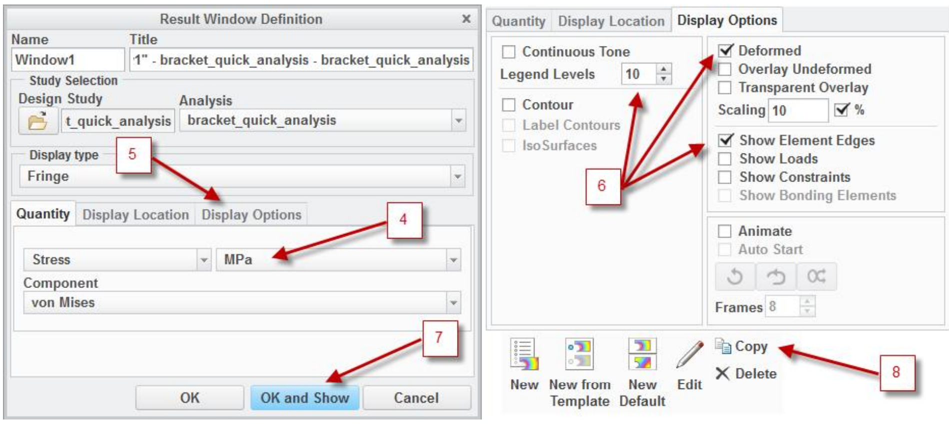

Figure 29. Result window mapkey definition part 2.

To define the second result window *Show element edges* and *Animate* with 12 *Frames* are selected (Figure 30, step 9). Next, *Quantity* tab is clicked (step 10) and *Displacement* with *mm* is selected (step 11). After that the result window definition is accepted with *OK and Show* button (step 12). The mapkey definition is then finished by clicking first *Stop* (step 13) and then *OK* (step 14).

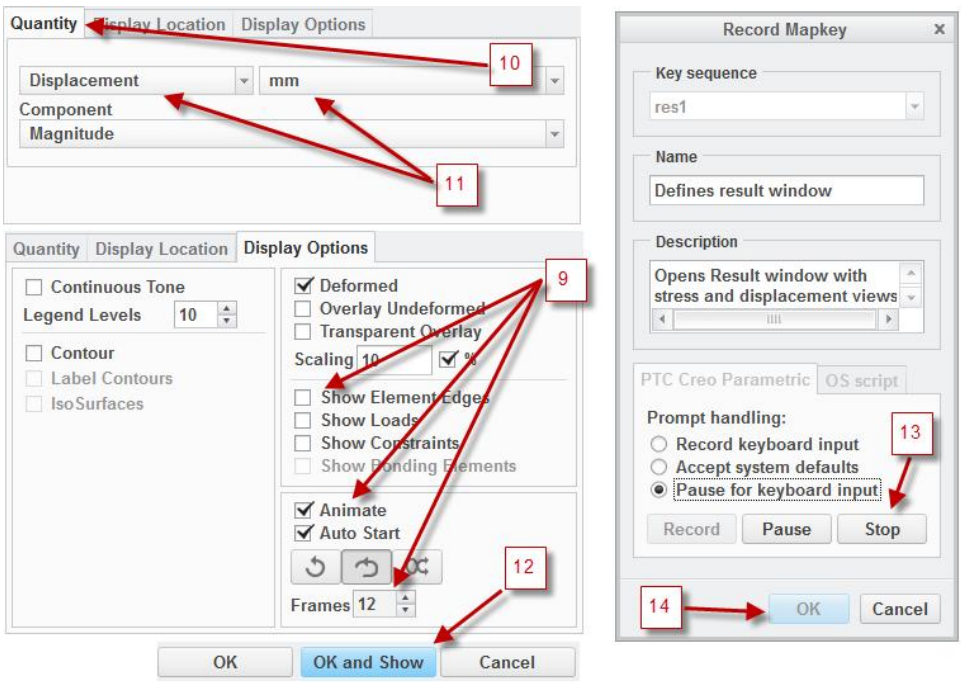

Figure 30. Result window mapkey definition part 3.

After the mapkey is completed, it has to be saved to the configure file. That can be done with *Save Changed* button in the *Mapkeys* window. If the configuration file is already defined, new mapkeys can be saved to the same file. Saving to the same file does not clear older settings but adds a new mapkey definition to the end of the existing file.

The defined mapkey can be used simply by typing its key sequence. This example macro is defined so that before the usage, *Analyses and Design Studies* window has to be opened and the desired analysis has to be selected (highlighted with blue). This mapkey guide will work for linear analysis models, but with little modification it can be applied for all kinds of models with different result window needs.

6.5.2 Mapkey for legend values.

Legend values are defined with linear distribution between maximum and minimum values by default. That makes it difficult to compare the results between simulations unless the legend values are manually set to the same for each simulation. This legend value changing procedure can also be done with macros.

In the next example how to create a mapkey for 0 to 200 MPa legend values is presented. The creation is started in the result window, where the mapkey definition is opened with search tool.

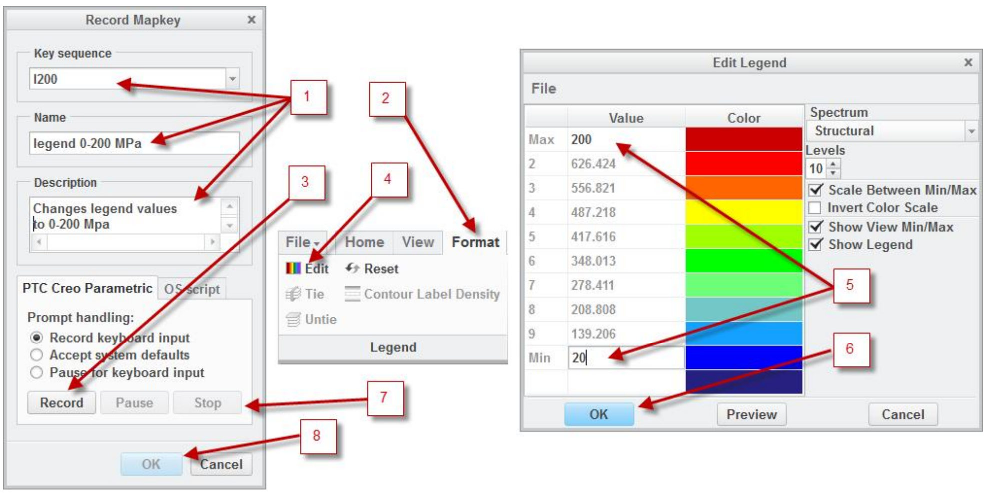

Figure 31. Mapkey definition for legend values.

A new mapkey creation is begun with *New…* button. The required information for mapkey is filled according to Figure 31 (step 1) and *Format* tab is selected before macro recording (step 2). Recording the macro is begun with *Record* button (step 3). The *Edit* is clicked (step 4) to open *Edit legend* window. In step 5 *Min* and *Max* values are changed to 20 and 200 which will produce a legend with 20 MPa intervals if there are 10 levels in the legend. The change can be done by clicking the value once and then typing the new value with the keyboard. *OK* is clicked (step 6) and the macro is finished with steps 7 and 8. The created macro has to be saved into configuration file with *Save Changed* button.

6.6 Creating a reference

A reference is feature which can be selected and constraints, loads etc. can be added into the reference. Planes, lines and points in the geometry can be used as a reference. With the *reference* tool it is possible to create *intent references*, which allow the user to combine multiple planes, lines or points into one reference.

This is useful especially when creating a contact between two components. The *reference,* which includes all contacting surfaces for both components, can be created. This allows the user to create the contact definition between the components with only one feature. Otherwise the individual contact feature should be added for all the surface areas which could be in contact with each other. In Figure 32 is shown addition of *intent reference* for snap feature contact surfaces.

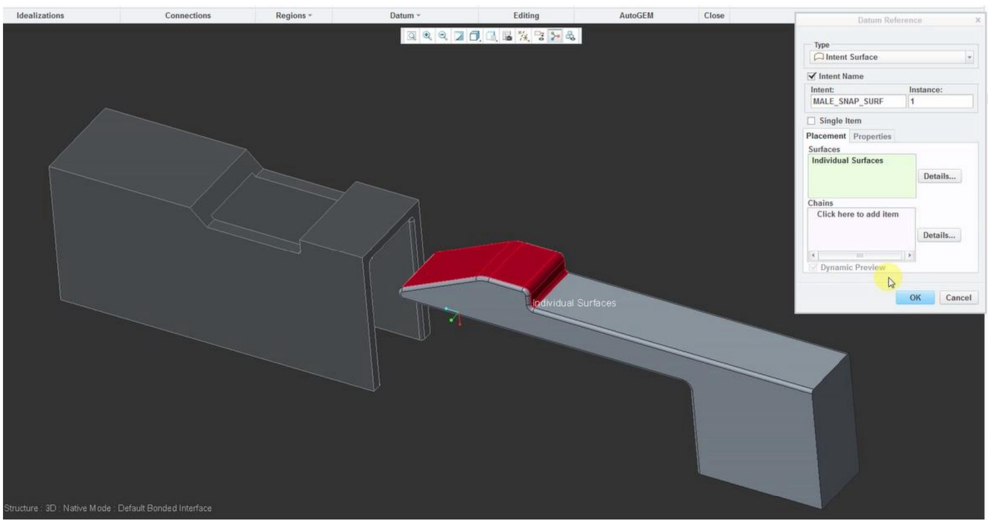

Figure 32. Intent reference addition for contact analysis.

7 CASE EXAMPLES

In this chapter, two different case examples where Simulate has been used successfully to create better design for parts of Nokia products are presented. In both cases the strength of the parts was simulated by static structural analysis. The final design evaluated by simulations was then ordered and tested by Nokia.

7.1 General requirements for Nokia products

General requirements defined by Telcordia are used for Nokia products. Telcordia produces multiple documents, in which are defined the general requirements for telecommunication products. Documents are available only for EL license purchasers. For these case examples requirements defined in the document GR-487-CORE: Generic Requirements for Electronic Equipment Cabinets will be used (Telcordia 2009).

7.2 Lifting eye

Lifting eye is a sheet metal part used to lift heavy radio antenna system (RAS) into the mast. The lifting eye was found to bend and deform plastically in the first prototype when lifting the RAS from horizontal ground position according to Figure 33. The lifting eye deformed so much that it also deformed the plastic cover of the RAS unacceptably. In this case example how Simulate can be used to improve the design of the lifting eye in the horizontal lift position in the ground is introduced. The lifting situation in which the RAS is vertically positioned and in the air is not studied in this example.

There is a requirement in GR-487-CORE (Telcordia 2009) for lifting attachments that visible deformations are not allowed with a load of 3 times the equipment's maximal weight. In addition, a fatal failure is not allowed with 6 times the load. These requirements are not applied for this case, since the RAS is still on the ground during the most critical load. The most critical load is going to be when the lifting is begun and the RAS jerks off the ground. Instead it is required that no visible deformation is allowed after the test lift in the lifting eye.

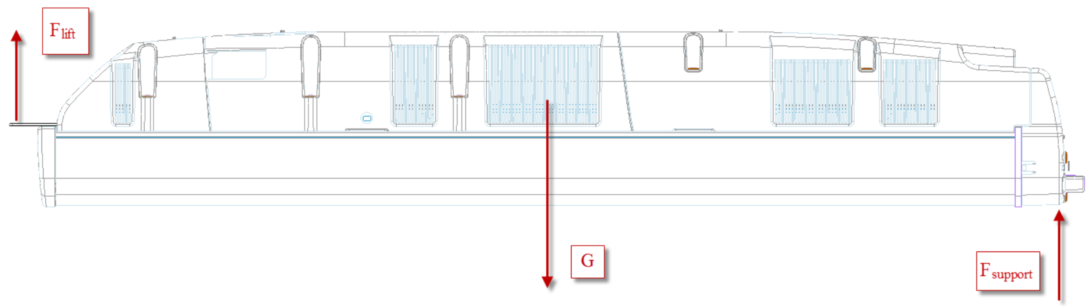

Figure 33. Forces during lifting of RAS.

First has to be determined what are the loads affecting the part during lifting. In Figure 33 are all forces at the moment when lifting from the ground starts. Lifting eye is on the left end of the RAS. In that position bending force affecting in the eye is the biggest. Center of the mass is estimated to be in the middle of the RAS so F_{lift} is half of the gravity force G in static case. Acceleration causes additional force, maximal acceleration in the lifting point during lifting is around 2,5 m/s^2 according to an experimental test (Incerti 2014). Total lifting force F is then calculated as following: F = 0,5mg + 0,5ma = 0,5 * 100 kg * (9.81 + 2.5) m/s^2 = 615 N.

In this case the simulations were not done before the first prototype part, which was then found to be too weak. Before creating a new design it is best to simulate the original design which will then serve as a reference. Simulations done for new design can be then compared with original to see if there has been improvement.

7.2.1 Simulating the original geometry

The first step in simulation is to prepare the part geometry for the simulation. In this case geometry simplification is not needed, because the geometry does not include any small features and the simulation can be done with the linear analysis. Next, the material is created and added to the geometry. In this case the part is made from AISI 304/304L stainless steel. The material is added in the current model material selection with *Materials* from the material library of Nokia and then assigned to the geometry with

Material Assignment. AISI 304 has there a 0.3 Poisson's ratio and 200 GPa Young's modulus. The material assignment appears into model with a yellow tag logo, which can be seen in Figure 34.

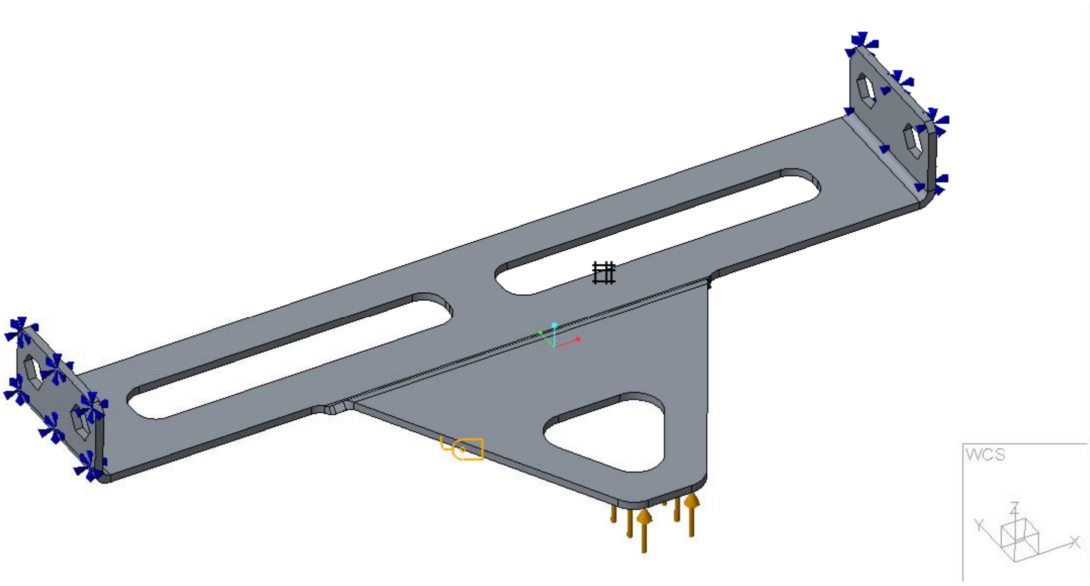

Figure 34. Simulation model of the lifting eye.

Next, constraints are added into the model, which is done with the *Displacement* tool. The lifting eye is fastened with four bolts from rivet nuts pressed into the part. In the simulation this can be idealized with constraints added into both end surfaces of the part according to Figure 34. If we would be interested in the stress near the fixing holes, the constraint could be added into the inner surfaces of the mounting holes. It was also studied how the constraint location (contact surface versus inner surface of the hole) affects the maximal von Mises stress result. The difference between the constraints was below 0.3 % so selected constraint idealization doesn't distort the result.

Load is added into the *Surface Region* created into bottom surface of the eye (Figure 35). The calculated value of 615 N is added into the Z direction, which is the direction of the blue axis in Figure 35.

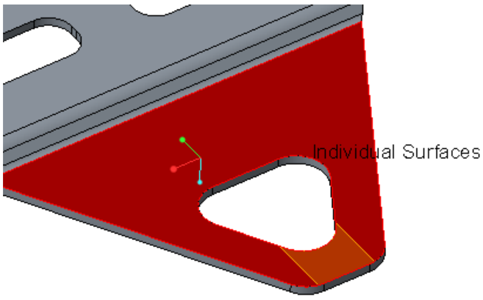

Figure 35. Surface region of the lifting eye.

After adding a load, mesh controls are added. The default mesh created by Simulate gives adequate results, but usually it is good to use some mesh control. With small models like this, the easiest option is to add *Maximum element size* control with *Component* selection into the whole geometry. For this model with 3 mm thick plate, a 6 mm maximal size for the mesh element is used. Mesh control finishes this simulation model and analysis can be defined for the model.

The used analysis type is *static analysis* with the default *single-pass adaptive* convergence method (chapter 4.1.3 page 51). The simulation is run two times with different mesh controls to see how the mesh affects results. Results with default mesh and with 6 mm maximal mesh size are compared in Figure 36. There is a 5 % difference in the maximum von Mises stress but visually viewed difference in the results is not significant.

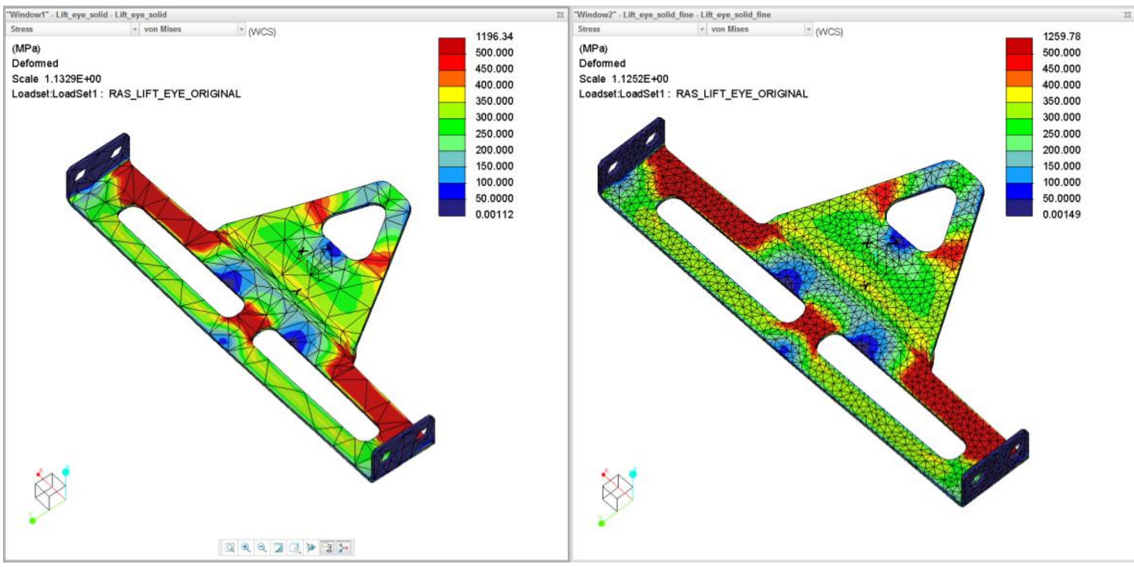

Figure 36. Results of the simulations with two different meshes.

Results in Figure 36 show that the stress levels according to the simulation are much higher than the material properties allow. AISI 304L has a yield strength of 241 MPa and an ultimate strength of 586 MPa (AK Steel 2007). There are lots of areas where the stress exceeds the 241 MPa yield strength limit in the geometry. With linear analysis, the stress results are not accurate when the yield limit is exceeded. Therefore, although the ultimate strength of 586 MPa is exceeded in the simulation, there were not any other failures but permanent deformation in the actual part after lifting.

Displacement results in Figure 37 show that there is around 10 mm of displacement on the area where the plate goes through the plastic cover. The maximal allowance is 2 - 3 mm so that the cover does not break.

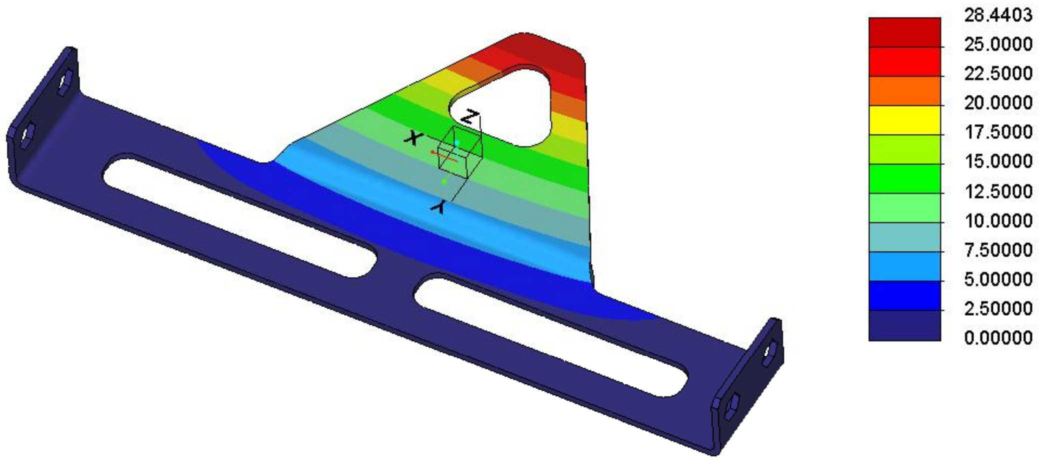

Figure 37. Simulated displacement in the original part geometry.

The simulation results are in line with the findings in the test lifting. This gives the signal that the used simulation method is probably reliable and can be used to design the new version of the lifting eye.

7.2.2 Simulation results of the new design

Finding the best solution for the part is a process where changing the geometry, running the simulation and viewing the simulation results follow each other again and again. This process has multiple repeating settings definitions, which takes unnecessary time from the actual simulation. The process can be helped with the methods shown in chapter 6.

In this case, multiple different design options were simulated. Quite soon it was clear that the material thickness of 3 mm had to be changed to 4 mm. The weight reduction holes had to be removed so that the stress in the part was divided more evenly. The simulation was performed with the same settings as for the previous part. Results for the simulation of the final design are shown in Figure 38. A tutorial video of the strength simulation of the new design for lifting eye was also created for the simulation training purposes of Nokia.

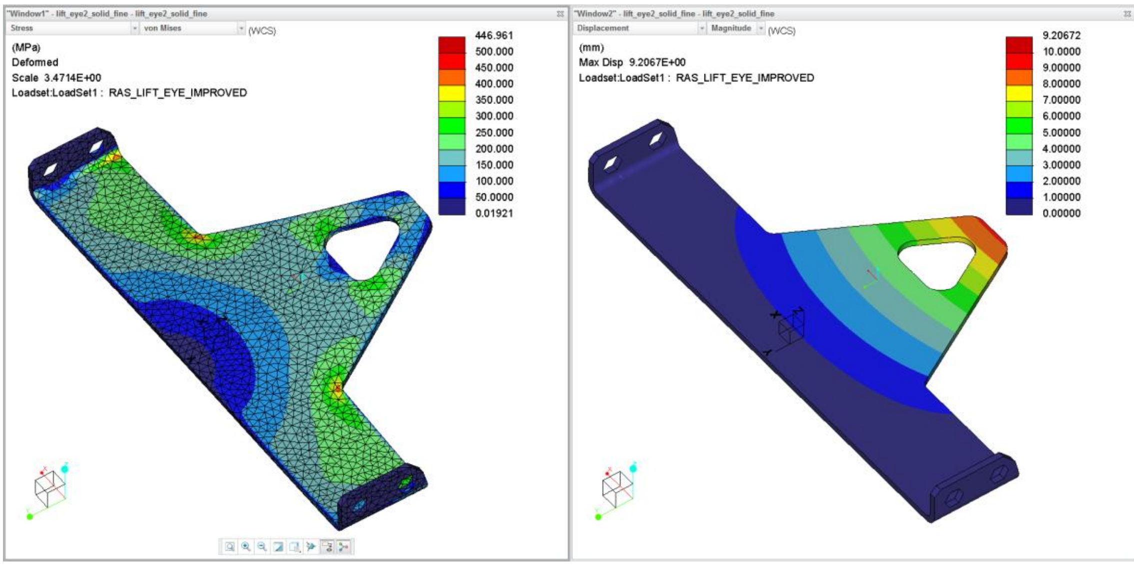

Figure 38. Simulation results of the improved design.

According to the simulations, the stress levels dropped significantly and are below the material's ultimate strength. There are still areas where the stress is over the yield strength of the material (marked with green colors in Figure 38). Since those areas do not cover a significantly large area, there should not be any permanent deformation visible in the part after lift. The displacement near the plastic cover has also decreased into an acceptable value of 2 - 3 mm.

The new part was ordered according to the simulated new geometry. It was fixed in place into the antenna product and tested by lifting the product from vertical position. The part with the new design did not show any visible deformations after the lift test. The requirement for the performed test was that there was no visual deformation allowed in the part after the lift. The test result was that the new part design can be accepted for the product and the simulated new part design is now in production.

This case example shows that if the original design would have been simulated, it probably would not have been created at all. Instead the better design with adequate strength properties would have been made for the first prototype. With aid of FEM simulation it is possible to design a functioning part at once and avoid too weak prototype parts.

7.3 Bracket assembly for antenna system

In this case example the strength of the bracket assembly for the antenna system shown in Figure 39 will be studied. It contains 5 sheet metal parts, an aluminum die cast part and a couple of other fixing elements. The scope of this study is to find out whether the sheet metal and cast parts will withstand the load caused by wind. Results will then be verified by testing.

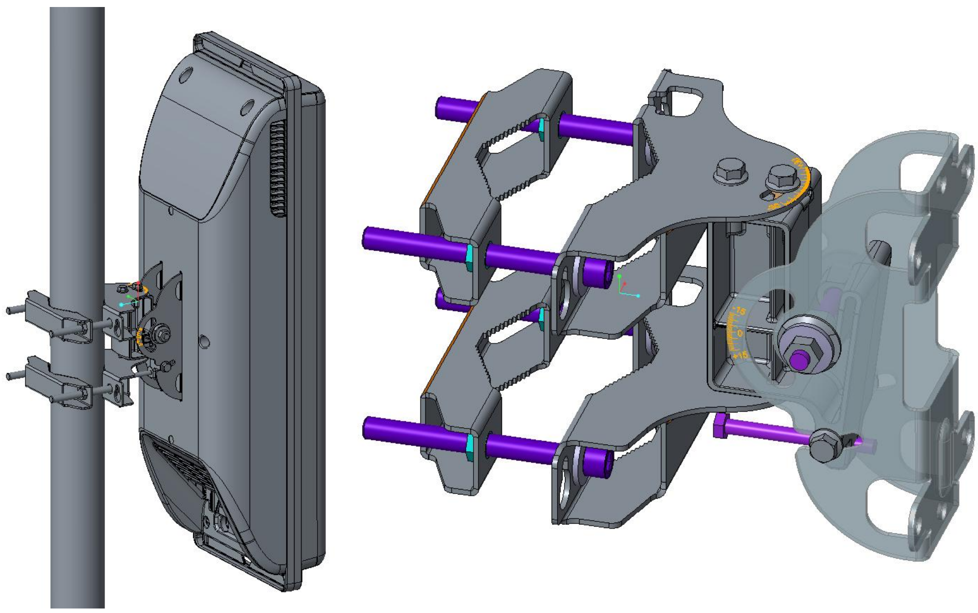

Figure 39. Bracket assembly for antenna system.

7.3.1 Defining the load case

The wind load simulation (Åvist 2016) has been made for the antenna product and results of that simulation are used to determine the load for this strength simulation. A wind load simulation is done for front, rear and side direction wind so the strength simulation is also done with the same directions. The wind simulations were done with

67, 55.5 and 42 m/s wind speeds but the results with 67 m/s wind speed are used for this simulation according to the GR-487-CORE requirements (Telcordia 2009). The antenna unit weight is also applied as a force into each load case of the simulation. An error estimation of 10 % for wind simulation accuracy is added into the load case values (Table 1).

Table 1. Load cases for bracket strength simulation (Åvist 2016).

	Force [N]		
Load case	X (side)	Z (front/rear)	Y (vertical)
Rear	0	349	-210
Front	0	-671	-210
Side	575	0	-210

7.3.2 Creating a simulation assembly

In this case a simulation needs to be done for the assembly. Assembly includes both the structural load supporting parts and the fastening elements like bolts. The results of the simulations are needed only for the sheet metal and aluminum parts, so the other parts can be removed from the simulation model. Unnecessary parts can be suppressed from the original assembly during the simulation or a separate assembly can be created, which will be used only for the simulation. In this case example, a separate simulation assembly is used.

7.3.3 Simplifying of the simulation model

Next step, after the simulation assembly is created, is simplification. Simplification helps to reduce the calculation time for the simulation. A couple of simplifications are recommended to be done for this simulation model.

The wind load can be estimated to affect the geometrical center of the antenna product. In order to distribute the wind load into the actual bracket parts, a simple part is modeled into the simulation model instead of the whole antenna geometry. That part has been measured so that the end surface, where the load is applied, will be in the geometric center of the antenna system. The created simulation part representing the antenna is the blue part in Figure 41.

The sheet metal parts do not require any simplification in this case. There are no such small features, which could cause a failure in the meshing or during the simulation. There are some features where simplification would give some reduction in calculation time, but for linear analysis used in this case the benefits are not worth the effort.

The aluminum die cast part has a couple of small features, which could possible cause problems during the simulation so those are better to be removed. The easiest way for removal is to suppress the added feature. In cases where that is not possible, simple extrudes to remove an unnecessary geometry can be used. If the designer knows during modeling process that the model is going to be simulated, it is possible to create a model, where all small features can be easily suppressed for simulation. From the die cast part all angle pointers and marking for product item label location are removed. The part before and after the simplification process is shown in Figure 40.

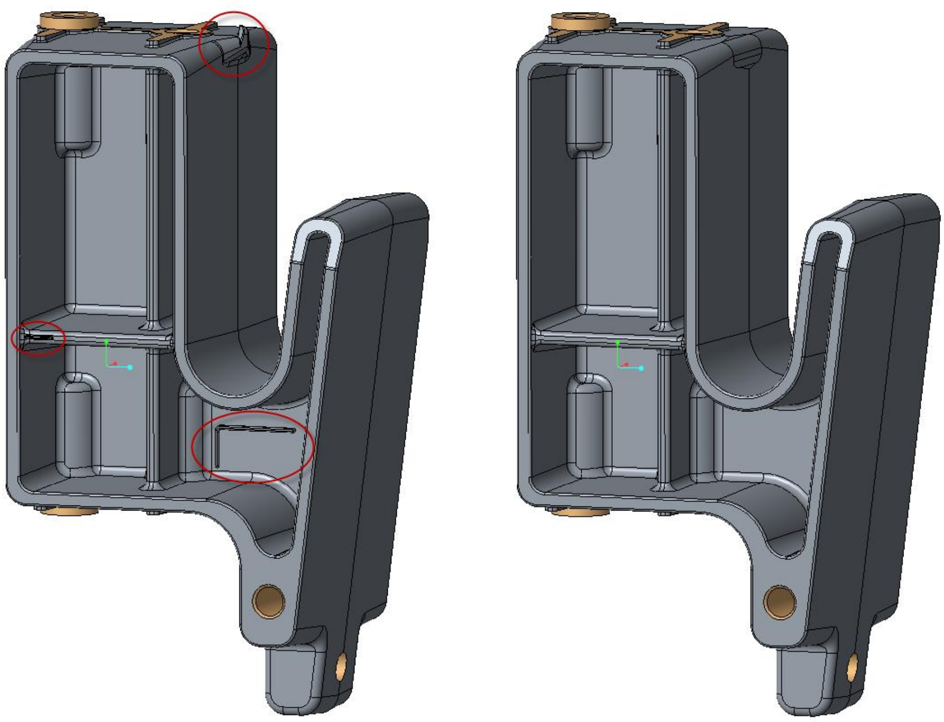

Figure 40. Small features removed from die cast part marked with red circles.

7.3.4 Adding loads, constraints and materials

In this case the bracket assembly is simulated in the pole mounting situation. In the pole mounting there are tooth features that are in contact with the pole. Those teeth can be estimated to stay in place so the displacement constraint is added into estimated pole contacting tooth surfaces according to Figure 41.

Four M12 bolts are used to tighten the bracket around the pole. With 20 Nm tightening moment each bolt can be roughly estimated to generate a 8 kN force. That force is applied onto the surface regions which are created around the bolt mounting holes. Added constraint and bolt loads should simulate stress caused by tightening the bracket around the pole sufficiently. (Figure 41).

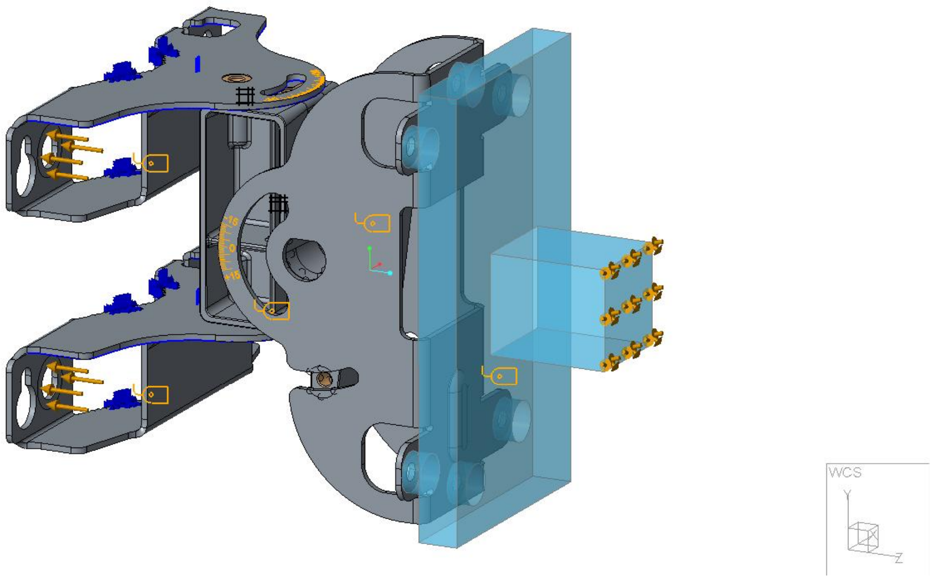

Figure 41. Loads and constraints in the simulation model.

Forces caused by the wind load are applied to the end surface of the blue simulation antenna part according to Figure 41. All three load cases are applied to the same model but each of them has its own *load set*. Bolt load force is also assigned into own load set. In linear analysis, the simulation can be run with multiple load sets at the same time. In the results window definition can be selected which of those are shown in the result window to be created. That saves time since is not needed to define and run the same simulation three times.

Materials are added from material library into the selection of model materials. AlSi10Mg is used for aluminum die cast part and AISI 304 for stainless steel parts and the antenna part. These material definitions are from Nokia's material library. AlSi10Mg has a 0.33 Poisson's ratio and 71 GPa Young's modulus there while AISI 304 has 0.3 and 200 GPa.

7.3.5 Adding connection idealizations

In assembly simulation, each part has to be constrained or connected to the other constrained part. In this case also the end weld connections have to be added into both pole bracket parts. The end weld interface connects the bended structure into a loop and ensures proper behavior in the simulation. All the created connections can be seen in Figure 42.

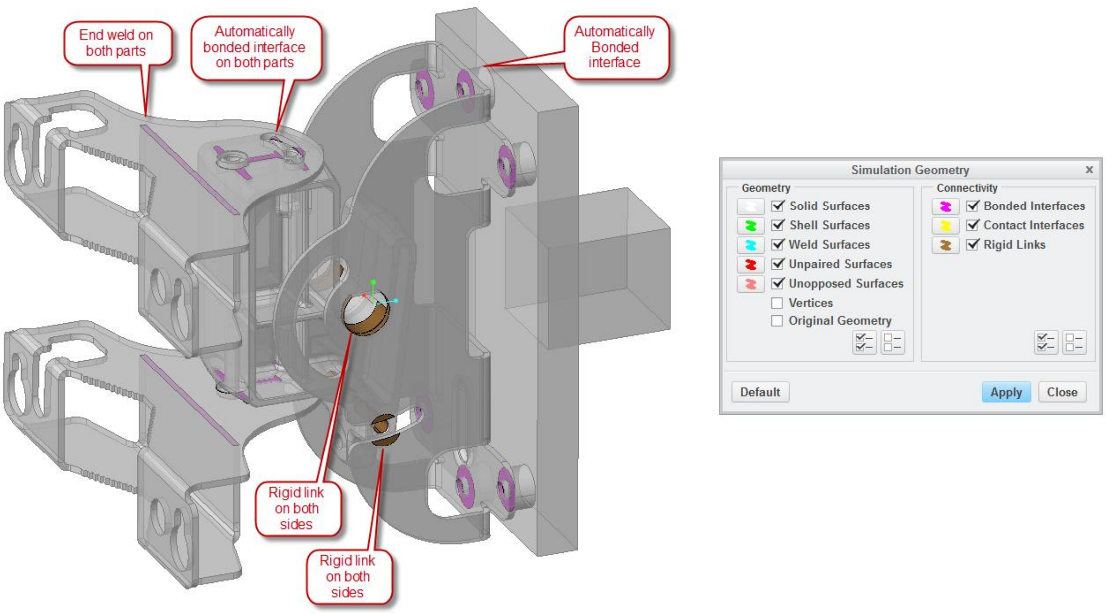

Figure 42. Connections in the simulation model.

The Simulate will automatically detect overlapping surfaces and creates a predefined interface between those surfaces. In this case the bonded interface is defined to be default so Simulate will automatically create interfaces for surface contacts shown in Figure 42. Those bonded connects are idealizing actual screw connections between the contacted parts. The idealization should work well enough since the connected surface areas are compressed together with screws and those should not be separated during the loading of the bracket assembly.

The aluminum die cast part and sheet metal mounting plate do not have any overlapping surfaces since the sheet metal part has been designed to be loose. The bonded connection cannot be done without overlapping surfaces, so the connection between those is done with rigid links, according to Figure 42. The surface regions are created for both parts to limit the connected area and prevent structure of becoming too stiff. The created idealizations do not take into account that the connected parts are actually compressed together with screws and that the upper connection is actually like hinge. However, the created links are on actual mounting points and will convey an added load from a part to another so the results should still be close to reality. Different methods can be used to achieve better accuracy, but that would require more work to create the simulation model.

After all the connections are created, they can be checked with the *Review Geometry*. That tool will create a summary of the simulation geometry as shown in Figure 42. It is an easy way to confirm that connections are created in the way that the user wants. However, there can sometimes be situations, where the connection does not work although it is created correctly and appears to be correct also in the geometry review. Those cases can be tried to solve either with different connection definition or by recreating the assembly and simulation model.

7.3.6 Mesh control

In this example, the whole model is meshed with default solid tetra elements. The sheet metal parts could be simulated with shell elements, but with the linear analysis type the achieved reduction in calculation time is less than time needed to create a shell idealization. The thickness of the plates in relation to the plate size is neither large enough that using shells could reduce the amount of needed elements significantly.

Only maximal element size control is added into the components. The sheet metal parts will give good results with reasonable calculating time when the maximal element size is 2 - 4 times the thickness of the plate (Chapter 3.5.5 page 41). The plate thickness in this model is 3 mm so a 10 mm maximal element size limit is added to all the sheet metal parts. For the die cast part, a 8 mm limit is added. The antenna part used for force distribution does not require any limits, because no results are needed from that part.

With the given element mesh limits, Simulate can calculate this model in couple of minutes depending on the computation capacity. For more accurate results, the mesh limits could be decreased or local mesh refinement could be added. However, Simulate uses p-type elements so mesh is mathematically “refined” near estimated error areas when using single-pass or multi-pass adaptive calculation. Therefore, a combination in which the overall mesh size is given manually and “refinement” is left to the adaptive p-method gives accurate results with reasonable simulation time.

If assembly simulation is used for optimizing single component of the simulation, it is possible to remove the mesh limits from the other components. This will speed up the optimizing process since the calculation capacity is not used for the other components.

7.3.7 Creating and running the analysis

The static analysis is created for the model. Analysis is defined with the default settings and the single-pass adaptive convergence method. From the load sets all but the *Sum load sets* option are selected. It is good to check the *Memory allocation* from the *Run Settings* and the value should be set to around 0.25 - 0.5 times the machine’s total RAM (chapter 4.3.1, page 55). After that the simulation is ready to be run.

The simulation run is started with green flag. If there are some problems in the simulation model, Simulate will show those in the *Diagnostics* window. In this case two kinds of errors were marked with the yellow dots. The yellow dot means that there is something in the model, which can affect the results but does not prevent the running of the simulation. The errors in this case example are shown in Figure 43. First there are some small curves, which Simulate informs to have been removed from the model and second there are some elements with invalid shape. The location of the errors is shown by clicking the error in the list.

Errors are located on areas marked afterwards to the Figure 43. All of errors are on such areas that accurate results are not needed from there so no actions are required for these errors. Small edge removal could be prevented either by changing the geometry or changing the limit for edge removal. Invalid shape elements can be prevented by reducing the maximal element size on that area or sometimes just by running the simulation again removes that problem. Sometimes errors can be in the CAD geometry, which has to be fixed first to fix meshing problems.

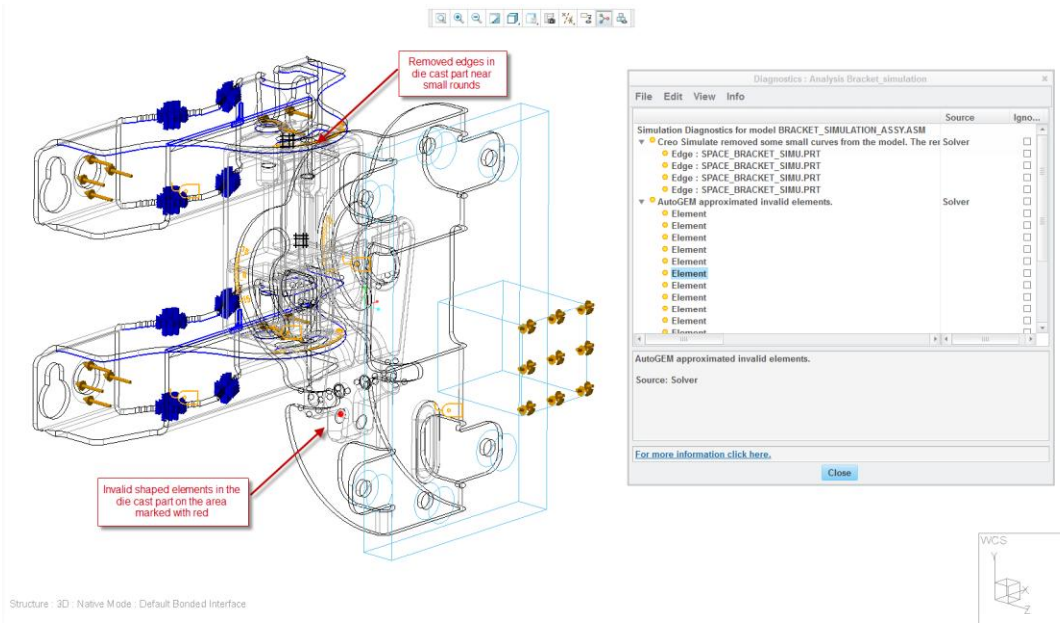

Figure 43. Errors in the case example simulation.

7.3.8 Viewing and understanding results

When opening a result window to a linear analysis with a multiple load set, it is possible to select which load sets are used for the result window. All three load cases can be easily checked to see the stress caused by each of them. The load set containing screw forces is added into all cases. With ductile materials the von Mises stress is a good measure in telling how structure will take a load (Efunda 2016). Another good result to check is displacement. It will give good feedback of actual deformation of the part. In Figure 44, there is an example of two result windows. The left window is defined with the von Mises stress, deformed with scale of 1, element edges are displayed, assembly is exploded and legend is set to 50 - 500 MPa. Displacement is plotted in the right window and deformation is scaled with the default 10 % setting.

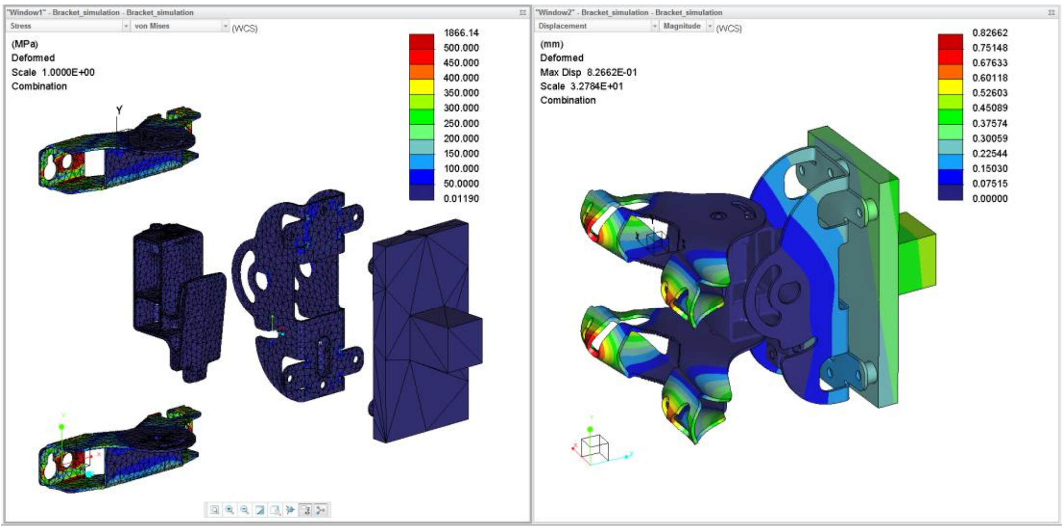

Figure 44. Results of the simulation.

There are a couple of things that should be checked in the results. First is to check model with default (10 % max displacement of parts maximal dimension) deformed state. The deformed state can reveal if there are problems with the created connections in the model. It also shows how the structure bends according to the simulation so it is possible to estimate if the simulation model is behaving in the logical way.

Another thing to check is the element meshing by showing the element edges in the result window. Showing the element edges at the same time with the von Mises stress in the results window also helps to see if there are high stress concentrations caused by mesh idealization and if there are areas where the mesh should be refined in the model.

Exploring the results reveals that highest stresses occur in the side wind case. The highest stress is appearing in pole bracket part and caused mostly by screw tensions. There are quite large areas where the von Mises stress value goes over material’s yield stress of 215 MPa and also some areas where the ultimate tensile strength of 505 MPa is exceeded. The highest displacement is still only 0.8 mm. The linear analysis is not accurate when the yield limit is exceeded in the simulation. However, these results mean most likely that part is going to deform plastically during tightening but it should not break.

In other parts, the von Mises stress values seems to be on much lower levels. In the aluminum die cast part the highest stress is around 50 MPa when the stress concentrations caused by the connection idealizations are ignored. AlSi10Mg has a yield strength of 80 - 140 MPa after cast, depending on used casting method, and with heat treatments can be improved to 200 - 280 MPa (Hydro 2016). The die cast part should therefore last the wind load but adding heat treatment could improve the safety factor and allow the usage of the part also for heavier unit mounting.

The radio bracket has the highest stress concentrations near the mounting flanges relief area. There are peak stresses of 250 - 300 MPa which are over material yield stress of 215 MPa. On larger areas the stress level stays mainly below 70 MPa so there should not be visual deformation in the radio bracket. The ultimate stress limit of 505 MPa is not exceeded anywhere either.

7.3.9 Result verification

A sand bag test was performed for a bracket assembly to verify the required strength of the parts. The sand bag test is used by Nokia to test if a real structure can take a simulated wind load without failure. In the test, the simulated wind load value is converted into equivalent amount of mass. A unit to be tested is then attached into pole and sand bags are loaded on to the unit. The unit is oriented so that mass of sand bags and mass of the unit are loading the structure equivalent with the simulated wind load.

In the performed test, the antenna unit was mounted horizontally to the pole. Only the side load case was tested, since it was most critical situation of the three load directions according the simulations. The mass was added as sand bags on to the antenna product so that the mass was evenly distributed on to the product. The calculated mass for the load was 60 kg. The test setup is shown in Figure 45.

The test was begun by measuring the unit's distance from the floor when only the unit's own weight was as a load. That value was used as a reference to see if the applied load was going to cause permanent displacement into the structure. The calculated mass was then added on to the unit and the distance from ground was measured again. It was decided that the test was to be continued by adding extra load to see if it is possible to find a fatal load for the structure. The mass was increased all the way up to the value of 123 kg. The measured displacements are as a graph in Figure 46.

Figure 45. Sand bag test setup.

After load was removed from the unit, displacement from the floor was measured and compared to the reference value. Permanent deformation was not found according to the measurement. Visual deformation was not found from the parts either. According to the test, the bracket structure can last wind load without breaking. Testing with additional mass showed that there is also some safety strength in the bracket, which is necessary so that parts with nonvisual defects will also last without fail.

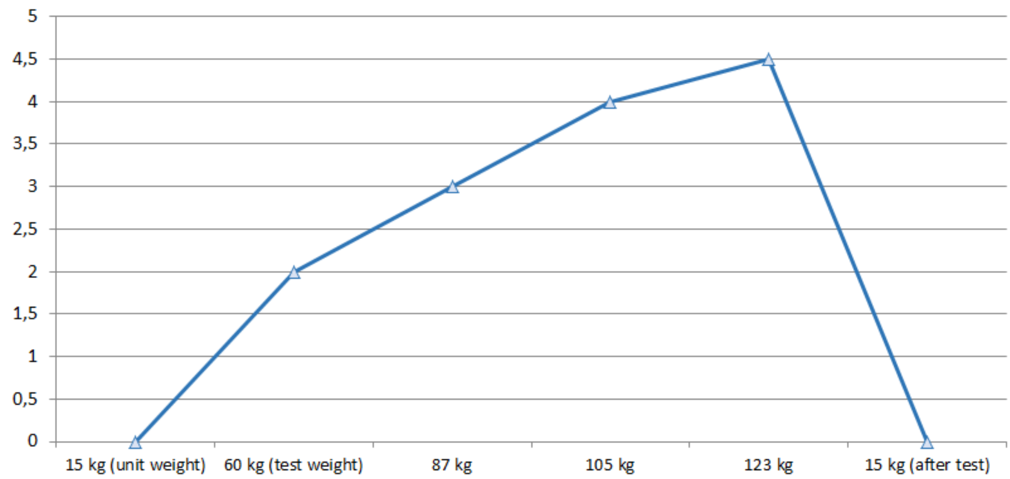

Figure 46. Graph of unit displacement in the sand bag test.

Test verifies the results got with Simulate and the bracket did not fail in the test. According to this case example with the used method it is possible to design a structure which will fulfill the strength requirements already in first prototype part.

8 CONCLUSIONS

Nokia has Creo Parametric as the primary 3D modeling software. They also have a license for Creo Simulate, a *finite element* simulation software, which can be used for strength simulations. Most of the mechanical engineers working for Nokia do not have the knowledge and skills to use Simulate. The purpose of this Master's Thesis was to produce training material for Creo Simulate to be used at Nokia.

During the work period for this thesis the theory of the *finite element method* and the simulation features of Creo Simulate were studied. Countless amounts of test simulations were ran when studying how to use Simulate efficiently. The scope of this work was limited to static strength analyses, although Simulate is capable to perform also dynamic, thermal, buckling and fatigue simulations. Simulate was also used for real product cases at Nokia.

As a result of this work five training videos were produced for Nokia. Also a short description of the simulation workflow was created, which can be used as a quick guide if the user has forgotten how to use Simulate. The workflow description is included in Appendix 1. The studied theory and features of Simulate are combined into this thesis so that it forms a guide for a beginner in the usage of Simulate and strength simulations. It includes a short introduction into the theory of FEM and then the process of simulation is covered step by step.

The simulation process starts from the creation of the simulation geometry. In Simulate, the existing Creo's 3D geometry can be used. If there are small features in the geometry, some simplification may be required. After that simulation features can be added into the model. Constraints, loads and material properties are the features that almost every simulation model requires. Idealizations and connections are required for more advanced simulations and assembly simulations. The user interface is the same in Simulate and Creo so setting up the simulation is easy for the user familiar with Creo.

The meshing in Creo Simulate is different than in the other FEM softwares. Simulate uses p-type elements, which differ from the traditional h-type element by their convergence methods. With h-type elements, convergence is achieved by reducing element size, while p-type elements are converged by increasing the order of the element shape function. During this work it was found that the best results and efficiency in Simulate can be achieved by giving a suitable element size limit for the whole part geometry and then convergence is ensured with Simulate's single-pass adaptive convergence method.

Setting up and running simulation can be done easily, since Simulate automatically selects settings that usually will work. It also prevents the user from creating an analysis with incorrect settings and gives feedback on what settings should be changed and why. The simulation process can be followed from the monitoring window.

Viewing the results is done with a separate application called *Simulate Results*. Viewing the results in a 3D contour plot works well and there are many good settings to simplify the use of the software. However, the user created templates for the result windows did not work properly. Also the features in 2D graph creation are really limited, there is not even a possibility to plot more than one curve into the graph.

Multiple ways to speed up the simulation process were found during the work. The most useful tool for that is the mapkey, which can be used to perform repetitive tasks with simple short keyboard commands. The user can also modify the configuration file to remove the need to change certain settings every time after a new start-up of the Creo Simulate.

The simulation model is saved into the same file which is used in Creo. Thus the created simulation model does not disappear unless deleted. The simulated results are saved according to the simulation name, so with unique naming can be ensured that those are preserved. An especially good feature is that the geometry of the simulation model is saved with the simulation results, so no original part file is required to view the results later.

Simulate was used for multiple real cases at Nokia but two of those are presented in this thesis. In the first case, the existing lifting eye design, which was found to be too weak, was simulated. A new design was created according to the simulations and that passed the tests without failure. In the second case, the wind load case was simulated for bracket assembly. The bracket made according to the simulated design was tested and found to have sufficient strength already in the first prototype.

From these cases it can be figured that Creo Simulate can be used to reduce product design time by preventing design fails of the parts, which are caused by insufficient strength properties. It can also help to reduce the weights of products by preventing too solid structures. As a continuum for this work Simulate’s dynamic analyses could be studied. Earthquake tests are performed for Nokia’s products and dynamic simulations could be used to find out whether the structure can take the load of an earthquake.

9 REFERENCES

Airila M, Ekman K, Hautala P, Kivioja S, Kleimola M, Martikka H, Miettinen J, Niemi E, Ranta A, Rinkinen J, Salonen P, Verho A, Vilenius M. & Välimaa V. (2010). Koneenosien suunnittelu. 4–5th edition. Helsinki: WSOYpro Oy.

AK Steel (2007). Product Data Sheet, 304/304L Stainless Steel. http://www.aksteel.com/pdf/markets_products/stainless/austenitic/304_304l_data_sheet.pdf [25.11.2016].

Christensen R. M. (2007). II. Yield and Failure Criteria for Isotropic Materials. http://www.failurecriteria.com/Media/Yield_and_Failure_Criteria_for_Isotropic_Materials.pdf [9.12.2016].

Efunda (2016). Failure Criteria. http://www.efunda.com/formulae/solid_mechanics/failure_criteria/failure_criteria.cfm [9.12.2016].

Hydro (2016). Primary Foundry Alloys, Technical Data Sheet, AlSi10Mg(a). http://www.hydro.com/upload/Documents/Products/AlSi10Mga.pdf [28.11.2016].

Incerti G, Petrogalli C. & Solazzi L. (2014). Estimation of the Dynamic Effect in the Lifting Operations of a Boom Crane. http://www.scs-europe.net/dlib/2014/ecms14papers/mct_ECMS2014_0124.pdf [25.11.2016].

Johnson D. (2016). Lecture Notes: P-Elements. http://engr.bd.psu.edu/davej/classes/lec_P-elem-Summary.html [25.11.2016].

Mäkelä M, Soininen L, Tuomola S. & Öistämö J. (2010). Tekniikan kaavasto. Matematiikan, fysiikan, kemian ja lujuusopin peruskaavoja sekä SI -järjestelmä. 9th.edition. Tampere: Tammertekniikka.

NPTEL (2009). Civil Engineering, Structural Analysis II (Web), Principle of Superposition, Strain Energy. http://nptel.ac.in/courses/105105109/2 [24.11.2016].

Pope J. E. (1996). Rules of Thumb for Mechanical Engineers. USA: Gulf Publishing Company.

PTC (2016 a). Creo Simulate Help Center. http://support.ptc.com/help/creo/creo_sim/usascii/ [20.11.2016].

PTC (2016 b). Data sheet, PTC Creo Advanced Simulation Extension. http://www.ptc.com/~/media/Files/PDFs/CAD/3D%20CAD/Elements%20Direct/PTC_Creo_Advanced_Simulation_Ext.ashx?la=en [20.11.2016].

Reddy J. N. (1993). An Introduction to the Finite Element Method. 2nd edition. USA: McGraw-Hill, Inc.

Taylor R. L. & Zienkiewicz O. C. (2000). The Finite Element Method – Volume 1: The Basis. 5th edition. Butterworth-Heinemann.

Telcordia (2009). Generic Requirements for Electric Equipment Cabinets, Telcordia Technologies Generic Requirements, GR-487-CORE. Issue 3, April 2009. USA: Telcordia Technologies, Inc.

Toogood R. (2015). Creo Simulate 3.0 Tutorial. USA: SDC Publications.

University of Oulu, Mechanical engineering (2014). Elementtimenetelmät I, Elementtimenetelmät II. Lecture handout.

Åvist M. (2016). CRAS wind load simulation. Internal document of Nokia.

WORKFLOW DESCRIPTION OF SIMULATION PROCESS WITH CREO SIMULATE

1 PREPARE MODEL GEOMETRY FOR THE SIMULATION

1.1 Create the simulation assembly/part geometry

Use existing Creo part file or step file as a base for the simulation geometry. An assembly for the simulation can be done by suppressing all unnecessary parts from the original assembly. Alternative and better method for complicate assemblies is to create a new separate assembly to be used for the simulation. If simulation is done for single part, the original part file can be used. If lot of geometry modification is needed, it can be easier to copy the original part and modify the copy into the simulation part.

1.2 Simplify the geometry

Remove the unnecessary and small features from the geometry. It is good idea to create models so that all the unnecessary features in geometry are collected into the end of the model tree. The simplification is then done easily by suppressing those features. If the feature cannot be suppressed, it can be removed with extrude or some other possible method.

In the assembly simulations some complicate parts, which are not necessary to be simulated, can be replaced with pure simulation part. Usually purpose of those is to distribute load or constraint into actual parts with realistic way.

2 CREATE THE SIMULATION MODEL

2.1 Add material properties

Add all needed materials into the model material selection. Assign correct material for each component in the model.

2.2 Add constraints

Find which surfaces or edges are used to mount the simulation part or are attached to a stiffer structure. Add constraints to those surfaces. The model geometry has to be constrained with one or more constraints so that free body movement is not possible for the part. If needed, use the surface region tool to create a suitable reference surface for the constraint. Use surfaces as a reference instead of edges or points, if possible.

2.3 Add loads

Find the direction and magnitude for the load. Determine where the load is acting in the model. Add the loads to surfaces if possible and create a surface region to limit the area where the load is affecting, if needed. If there is multiple load cases to be run, create own load set for each load case. With linear analysis all the load cases can be simulated at once. With nonlinear analysis correct load set for the particular case can be selected from the analysis definition window.

2.4 Create idealizations and connections

Overlapping surfaces can be connected with the bonded interface. If there is a gap between the surfaces needed to bond, a weld connection can be used instead. Complicate connections can be idealized with a rigid link. Keep the reference surfaces for the rigid links as small, since the rigid link turns the reference surfaces into rigid stiffening the part structure. Be sure that all the parts in the assembly are tied in place either with a constraint or with connection to the other constrained part.

2.5 Add the mesh control

2.5.1 Linear static analysis, nonlinear static analysis with less than 10 steps.

Add maximal element size limit into the parts from which it is necessary to get accurate results. For sheet metal parts overall element size limit, which is 2 - 4 times the plate thickness, gives usually accurate results. For solid parts it is harder to give a good overall limit, but value which is 5 - 10 % of the parts average size can be used as a rough guideline for the element size limit.

Element limit should be set so that the mesh is as fine as possible while the simulation time stays reasonable. That way it is possible to get quite accurate results without wasting design time for unnecessary waiting. If total element count stays below 10 000, linear static analysis with single-pass adaptive method is solved with decent laptop in less than minute and with

30 000 elements calculating takes couple of minutes. (HP EliteBook 8570w, Intel i7-3740QM 2.7GHz, 16GB RAM and Nvidia Quadro 1000M, 2GB RAM)

Default element type is the solid tetra element. With static linear analysis change of the element type is not usually necessary. Shell elements or thin solid elements can be used to reduce the calculation time especially if the plate thickness of the sheet metal part is less than 1 - 2 % of the part's average size.

2.5.2 Nonlinear analysis with 10+ steps, dynamic or otherwise heavy analysis

Run a quick linear static analysis without mesh control to see where the high stress areas are and how good is the default mesh. After that add local mesh control onto areas, where high stresses are going to be or accurate results are needed. Add a fairly large element size limit to the whole part if the default mesh is too coarse. For sheet metal parts use the shell elements or thin solids to reduce the total amount of the elements. Shell elements cannot be used with nonlinear analysis.

2.6 Add measures

If some specific value is necessary to be found, for example a reaction force in the specific constraint, create a measure for it. The results for the measures can be viewed from the run status window, from the results window with the *measures* tool and as a graph with the result view window definition.

2.7 Review Geometry

Run the *review geometry* tool to check all created idealizations and connections. The same tool also shows if shell or solid element definition is added for the geometry. If something is missing in the review, fix the model before running the analysis.

3 CREATE AND RUN THE ANALYSIS

3.1 Create the analysis

Create the analysis and remember to name it with unique name. Unique name ensures that the created analysis is not going to be overwritten with another analysis and results can be viewed later. Select correct constraint sets and load sets to be used for analysis. Use single-pass adaptive method unless you want especially accurate results, which can probably be achieved with multi-pass adaptive convergence method.

3.2 Check run settings

From the run settings check the memory setting for the solver. Value which is 0.25-0.5 times the machine total RAM should be used. Value should not still exceed the amount of the available memory at the current moment. That can be checked from the task manager with the Windows computers.

The directory for the simulation files can be defined also in the run settings. If nothing is specified, the current working directory is used. In the Simulate version used with Creo3.0 M040 there was found a bug which causes that **the simulation files for the design studies has to be saved into the current working directory** so that the simulation will run correctly.

3.3 Run the analysis

Run the analysis and follow analysis progress with the run status window. If the simulation does not progress, it can be terminated with the red flag button. If there is warnings in the interactive diagnostics after the simulation, check the cause of the warnings and do changes into the model, if needed.

4 CHECK THE RESULTS

4.1 Verify the model validity

Create a result window with the deformed scale of 10 % and with the element edges shown. Check that the magnitude of the results is reasonable. Check form of the deformation to see if the structure bends to logic directions and that all the constraints and connections are working. Check the mesh that there are no failed geometry or unusually small or large elements on critical areas.

4.2 View results and do conclusions

Check von Mises stress levels and displacement results from the model. With linear material model and analysis the results are not accurate, when the stress levels are exceeding the yield stress of the used material. Von Mises stress value works well as failure criteria for ductile materials. Plastic deformation happens on the areas, where simulated von Mises stress exceeds the yield stress of the material. If those areas are covering too large area, there will be visible permanent deformation in the part after the load is released. Exact value for the load which will cause visible permanent deformation in the part is hard to find with linear analysis.

View the displacement results with unscaled deformation (1 to scaling value and clear % -setting) so that the actual deformation caused by the load can be seen. The displacement value can be used also to optimize the structure so that too big displacements are avoided in the part.

4.3 Save the results

Results for the analysis with the unique name are saved automatically to the directory named with the analysis name. If needed, it is possible to use *vault results* tool to save the results into Windchill as one file. The simulation model geometry is included in the result files, so no original geometry is needed to view the simulation results later.