CHALMERS

Idealization of riveted joints

Degree project in Machine engineer program

THURESSON ADRIAN

VALLENTIN DANIEL

Institution of Applied mechanics

Department of Dynamics

CHALMERS UNIVERSITY COLLEGE

GOTHENBURG, SWEDEN 2012

DEGREE PROJECT 2012:02

DEGREE PROJECT 2012:

Idealization of riveted joints

Degree project in Machine engineer program

THURESSON ADRIAN

VALLENTIN DANIEL

Institution of Applied mechanics
Department of Dynamics
CHALMERS UNIVERISTY COLLEGE
Gothenburg, SwedenGOTHENBURG, SWEDEN 2012

Idealization of riveted joints
Degree project in Mechanical engineer program
THURESSON ADRIAN
VALLENTIN DANIEL

Degree project 2012: ISSN 1652-9901
Institution of Applied mechanics
Department of Dynamics
Chalmers University College
SE-412 96 Gothenburg
Sweden
Phone number: + 46 (0)31-772 1000

Printing/ Institution for Applied mechanics
Gothenburg, SwedenGOTHENBURG, SWEDEN 2012

Abstract

Fasteners play an important role in the aerospace industry. There is a big range of bolt and riveted joint applications, both in aircraft and jet engine structures. The need of accurate joint strength evaluation constantly grows due to higher performance and low weight expectations, thus a wide knowledge spectrum in this range is definitely required to make products more efficient from the performance and cost point of view. Cold structure department at Volvo Aero has a need of knowledge growth in the range of fasteners strength and life evaluation, especially riveted joints.

Advanced FE-models demands a lot of computing power and long simulation runs when performing FE-calculations and can thereby not be run in the complexity you would want it to. Therefore parts are replaced in the model with idealized parts which should behave approximately like the none-idealized part. These idealized parts demands a lot less computing power and simulation time. Riveted joints are extensively used in these big simulation models and finding a working idealized model is considered as a high priority.

The aim and purpose with this degree project is to find a working idealization for riveted joints which should behave like the solid reference model created in Hypermesh. When a working idealized model has been found, more idealized models will be developed in order to maybe find a working general idealization solution to the problem at hand.

Our study shows that there are some relationships that can be established between the working idealized models and the solid models. A working idealization method for every unique riveted joint has been developed. Our final idealized model consists of spring-elements placed on- or off-center between the riveted plates fastened with CERIGS. The first riveted joint idealization led to an angular displacement deviation, a global displacement deviation and a stress deviation seen in table 1.1. These are also compared to the idealized reference model; BEAM-model.

	Maximal tensile stiffness deviation	**Maximal bending stiffness deviation**
BEAM-model	40.1%	37.6%
On-center	0.9 %	16.9%
Off-center	0.3 %	2.5%

Table 1.1. Maximal resulting deviations

This idealization method also worked for five more models that we tried. The analysis time reduced from around 30 hours when performing FE-calculations on our solid reference models to 10 minutes when performing FE-calculation on our 1-D rivet models.

Preface

This degree project was carried out between February to the end of June 2012 at the department of cold structures at Volvo Aero in Trollhättan. This degree project marks the end of the Bachelor of Science in mechanical engineering at Chalmers university college, Gothenburg.

We would especially like to thank our supervisor Robert Rödström for his constant presence, support and help throughout this degree project. We would also like to thank Andreas Rydin, Robert Reimers, Martin Larsson and Raja Visakha for their help and support during this project. We are also very thankful for all the help and support we have been given from everyone here at the Cold structures department and who have made our stay here at Volvo Aero a very pleasant one.

Finally we would like to thank our examiner Ph.D. Peter Bövik at the department of Mechanical engineering for his help and assistance during the project.

Trollhättan, June 2012

Adrian Thuresson

Daniel Vallentin

Table of Contents

Nomenclature/ Symbols

Symbol	Unit	Description
F	N	Tensile force on edge of plate
	MPa	Tensile stress in plate section
	MPa	Shear stress in plate section
	Degrees(°)	Angular displacement
λ	µm	Global displacement in x-direction

Term	Description
BEAM model	Idealized reference model
VAC	Volvo Aero Corporation
Substep/-s	A substep is an increment of load within a step in an FE-calculation
Mechanical continuum	Classic mechanics for continuous bodies
Rigids	Connects two nodes rigidly, typically between a beam node and a solid node
Mesh/-ing/-es	A grid of finite elements
1-D rivet model	The solid rivet in the model is replaced with a beam or spring element, other properties are equivalent.
DOF	Degrees of freedom
FE	Finite Element
FBO	Fan Blade Out
MPa	Mega Pascal
Ti	Titanium
Inco	Inconel 718
Dev.	Deviation

1. Introduction

1.1 Background

Fasteners play an important role in the aerospace industry. There is a big range of bolt and riveted joint applications, both in aircraft and jet engine structures. The need of accurate joint strength evaluation constantly grows due to higher performance and low weight expectations, thus a wide knowledge spectrum in this range is definitely required to make products more efficient from the performance and cost point of view. Cold structure department has a need of knowledge growth in the range of fasteners strength and life evaluation, especially riveted joints.

Advanced FE-models demands a lot of computing power when performing simulations/calculations and can thereby not be run in the complexity you would want it to. Therefore parts are replaced in the model with idealized parts which should behave approximately like the none-idealized part. These idealized parts demands a lot less computing power.

Riveted joints are extensively used in these big simulation models and finding a working idealized model is considered as a high priority. In order to get even more data on how these riveted joints behave in the physical jet engine structure a shear test of different riveted joints will be performed at VAC during the second quarter of 2012.

In jet engines there are often a riveted flange design is often a riveted flange design. VAC wants an improved method for modeling the rivets with higher accuracy in the results. This should be studied using a single riveted joint model, with similar dimensions on rivet and plate thickness as in a realistic riveted flange design.

1.2 Aim of the study

The purpose with this degree project is to perform FE-studies on different riveted joints in order to come up with the best idealized model as possible. The purpose with the results is to make guidelines on how an idealized riveted joint should be modeled. It should also be simple to model.

1.3 Delimitations

No physical tests are going to be executed by us personally, thereby should no part in this degree project be about that. If the project runs smoothly and ahead of schedule a study about how general our solution is will be performed. I.e. how does this idealized model work on a joint with bigger/smaller rivets, thicker/thinner plates, different rivet heads, and so on.

1.4 Clarification of the issue

- How should a 1-D model be modeled in order to get the best possible idealized model which results corresponds well with our solid model?

2. Theory/Models

2.1 Riveted Joints

The most commonly used method when creating a riveted joint is to deform one side of the rivet with some impact tool which leads to plastic deformation of the one side of the rivet, see figure 2.1. This demands an anvil by the rivets main end. Other riveting methods have been created when it is not possible or preferable to use an anvil, these rivets can be blind-or pop rivets, see figure 2.2. The most common rivets can be found in SS (Swedish standard), this standardization includes the rivet shaft diameter and main shape.[1]

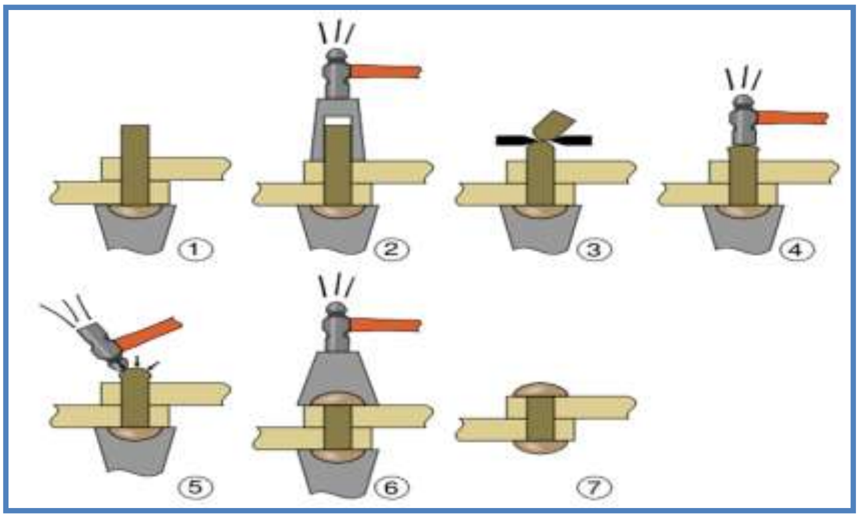


Figure 2.1. Solid riveting

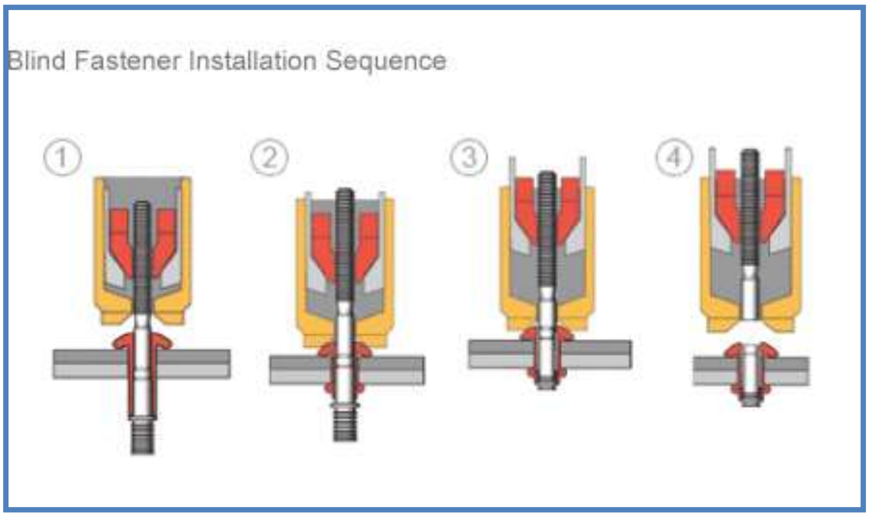


Figure 2.2. Blind fastener installation sequence

The rivets main function is to carry forces along its shear plane, only in exceptional cases can the rivet carry forces orthogonally against its shear plane. Riveted joints can be divided into two main categories, single lap and multiple lap joints. Single lap joints connects two plates, thus it only has one shear plane. Bending torque on the rivet also occurs in these single lap joints. In multiple lap joints the rivet has multiple shear planes, hence, the rivet tends to shear in multiple planes.[1]

The rivet can either be cold drawn (room temperature) or hot drawn (about 500˚C difference between rivet and goods). The riveted joints function is by a big extent decided by which type of method that has been used. When cold drawn it is important to take in consideration that a gap is present between the rivet and the goods due to an elastic relaxation after the rivet has been plastically deformed. This gap can be prevented if the plates are pre-stressed when the rivets are cold drawn, then the rivets fill the plate holes completely. One important property that the riveted joint has when it is cold drawn is that there is nearly no pretension. This method is used for steel-, aluminum-, copper- and soft metal rivets where the diameter on the rivet is ≤8mm. When the riveted joints are hot drawn there are a temperature difference of 500˚C between rivet and goods. Using this method leads to shrinkage of the rivet when it is cooled down. Radial plastic elongation occurs when the rivet is cooled because it cannot shrink axially, the rivets yield strength is often achieved when this occurs. Hence, this leads to a radial gap between rivet and the plate hole. This method is used for steel rivets with a diameter of ≥ 8 mm. Hot drawn rivets are primarily used when the force required to draw the rivets is too big.[1]

Hot drawn rivets in a single lap joint (pre-stressed) transmit force first by friction forces in the shear plane. After all the friction forces have been triggered, more force can be transmitted through shear of the rivet and bending of the rivet head. When the gap is completely gone between the rivet and the plate hole even more force is transmitted through the rivet, i.e. the rivet begins to shear in the shear plane. In cold drawn rivets (none pre-stressed) the rivet fills the rivet hole completely, thus the rivet begins to shear directly, i.e. the force transmission goes through the whole rivet.[1]

2.2 FE-modeling

The finite element method is a useful general numerical method for solving a wide variety of different problems, such as; flow- , acoustic-, electromagnetic fields and dynamic problems. FE calculations can be very useful when trying to understand weaknesses in the construction already in the concept phase. Thereby, FE calculations can be a form of guidance in terms of concept selection and various detail design of the structure or product. The validity of FE analyses depends on how good the boundary conditions are set, how well the mathematical model have converged and how the mesh density is in the model. Analysis of models calculated with FEM demands experience of previous performed FE calculations in order to verify the validity of the results. Some FEM-programs are very easy to use which can result in misinterpretation of FEM as an easy calculation tool. If you lack experience in the specific software, misinterpretation of the results is easy and there is a risk of misusing the program. Handmade calculations are often done before any FE-calculations are made in order to have some guidance within what level of detail is required in the analysis. Handmade calculations can also show that FE-analysis is not needed due to that the loads and displacements are small. They can also act as a reference value to the FE-calculations. Linear FE-analysis is the most commonly used analysis when the displacements are small and the stresses are under the materials yield strength. When failure loads are to be simulated, where the material deforms plastically, nonlinear material data in the analysis of the model have to be used which leads to more complex calculations. Contact is another nonlinear problem that makes the simulation more complex.[2]

By looking at a continuum mechanical problem we can illustrate how the finite element method works in a linear example. The problem can mathematically be described with partial differential equations which has an infinite number of degrees of freedom. A big model can then be described as a number of partial differential equations, these equations are often not solvable analytically and therefore have to be solved e.g. with the finite element method. The number of degrees of freedom in the model reduces when applying FEM by dividing the model into finite elements, this also leads to a simplification of the model. The elements are connected to each other by a number of points (nodes). Every node in the model has known elastic properties. This leads to a linear system of equations that describes force- and displacement relationship which can now be solved.[2]

A balance between required accuracy and reasonable calculation times are always in consideration when modeling with FEM. In order to reach convergence a fair amount of elements and a sufficient number of iterations are needed. The number of elements, iterations and substeps are primarily decided by how large loads or stresses there is. When solving linear problems, i.e. pure elastic, the number of needed iterations is always one. When dealing with non-linear models, e.g. contact models, the number of substeps must be increased in order to divide the problem into shorter easier steps. Important to always have in consideration is that FEM is an approximate solution method, thereby calculations that have converged with high demands on the convergence value may still give incorrect results.[2]

2.3 Models

There are a lot of different dimensions of riveted joints used in different models and sections at the cold structures department of Volvo Aero, so analysis and idealizations of all of them are not possible if not a general solution is made. There are a few higher prioritized riveted joints that are shown in red. These riveted joints are commonly found in riveted jet engine flange designs. The rivets that have been given to idealize come from the AS9319 SAE aerospace standard, see figure 2.2. There are three types of rivets that primarily have been focused on, they are shown in the red box in the specification (rivet 1, rivet 2 and rivet 3).

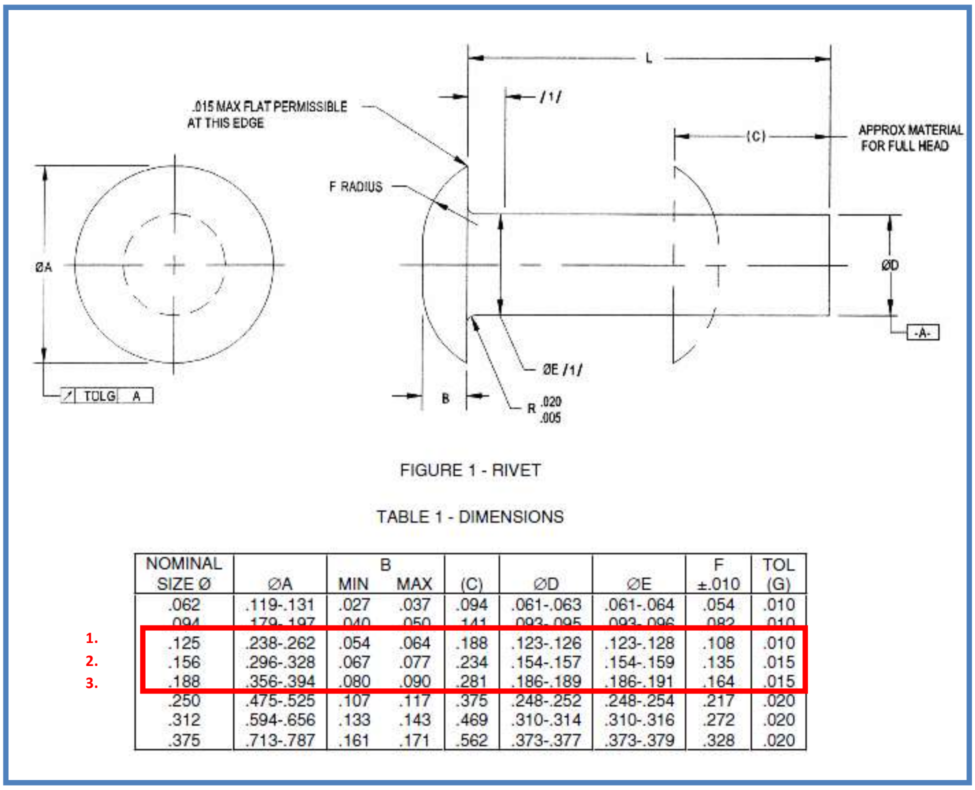


TABLE 1 - DIMENSIONS

	NOMINAL SIZE Ø	ØA	B MIN	B MAX	(C)	ØD	ØE	F ±.010	TOL (G)
	.062	.119-.131	.027	.037	.094	.061-.063	.061-.064	.054	.010
	.094	.179-.197	.040	.050	.141	.093-.095	.093-.096	.082	.010
1.	.125	.238-.262	.054	.064	.188	.123-.126	.123-.128	.108	.010
2.	.156	.296-.328	.067	.077	.234	.154-.157	.154-.159	.135	.015
3.	.188	.356-.394	.080	.090	.281	.186-.189	.186-.191	.164	.015
	.250	.475-.525	.107	.117	.375	.248-.252	.248-.254	.217	.020
	.312	.594-.656	.133	.143	.469	.310-.314	.310-.316	.272	.020
	.375	.713-.787	.161	.171	.562	.373-.377	.373-.379	.328	.020

Figure 2.2. AS9319 SAE aerospace standard

Since the rivets are cold drawn, the rivet fills the rivet hole completely, so the diameter in the specification above is bearing no interest regarding the modeled rivet. The rivet diameter in the specification above was used to calculate the rivets ultimate strength and the fatigue limit of the rivet, see equations 2.1 and 2.2 and table 2.1. Equation 2.2 is just an assumption.

…eq. 2.1

… eq. 2.2

	Rivet 1	Rivet 2	Rivet 3
Ultimate tensile strength	2400 N	3800 N	5600 N
Fatigue limit	1200 N	1900 N	2800 N
Reference load (75 % of ultimate tensile strength)	1800 N	2850 N	4200 N

Table 2.1 Reference loads

There are also different dimensions of the plate thicknesses; 1.0, 1.2, 2.0, 2.5 and 2.54 mm. These plates have three different diameters of the holes, depending on which one of the rivets there is. Rivet 1 is cold drawn into a hole diameter of 3.3 mm, rivet 2 is cold drawn into a diameter of the hole of 4.1 mm and rivet 3 is cold drawn into a hole diameter of 4.5 mm. The first model is Alfa which has the same dimensions as the test specification, see figure 2.3. This test specimen was the base model to this project. The fatigue test of the riveted joint will be carried out sometime during the late second quarter of 2012.

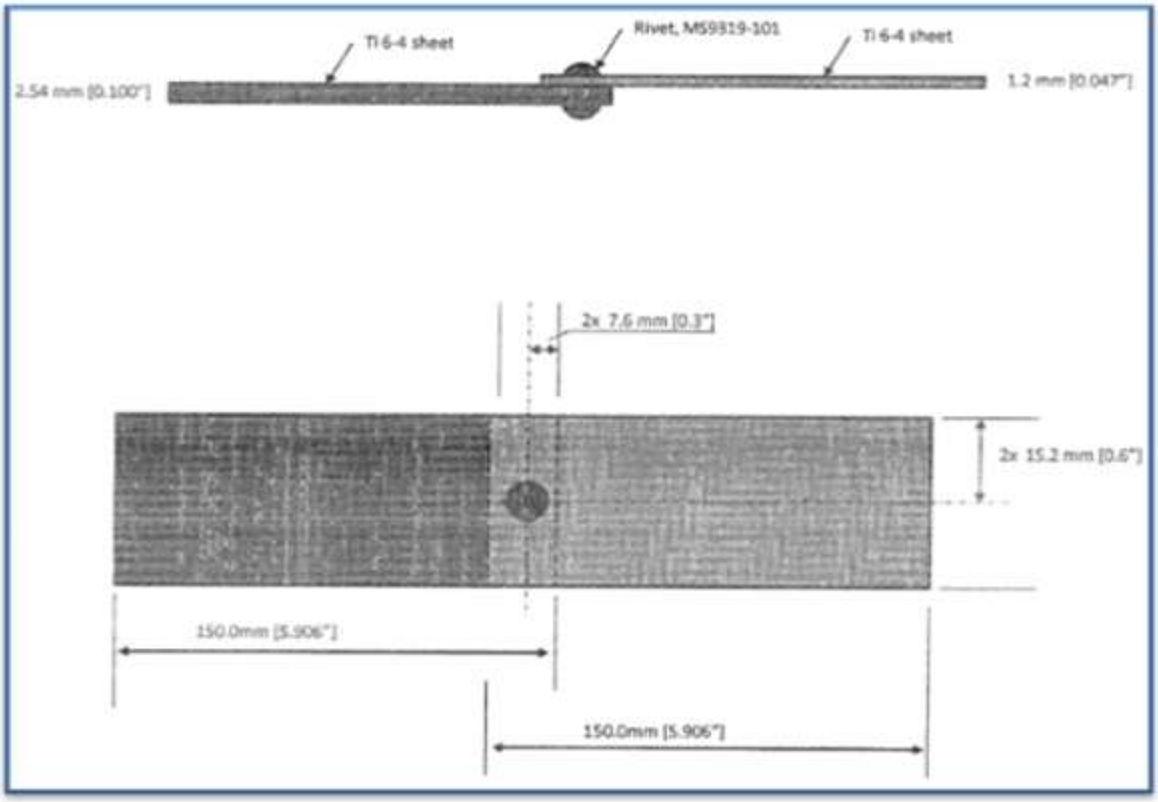


Figure 2.3. Riveted joint test specification

The names of the different riveted joints to be investigate are Alfa, Beta, Gamma, Delta and Epsilon, see table 2.2. One of the models appears with three types of rivets/hole dimensions. All units are in millimeters. The most common riveted joint is the Alfa model. Rivet 2 and 3 have been included in order to find a general solution for different rivet dimensions.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thicknesses [mm]	
			Under plate	Upper plate
Alfa	1/2/3	3.3/4.1/4.5	1.2	2.54
Beta	1	3.3	1.2	2.0
Gamma	1	3.3	1.0	1.2
Delta	1	3.3	2.54	2.54
Epsilon	1	3.3	2.0	2.5

Table 2.2. Riveted joint models

In the analysis model, the length of the plates has been reduced from 150 mm to 40 mm, to be able to run faster analyses. This is done without jeopardizing the results accuracy. No double lap joints have been taken in consideration.

3. Idealization

3.1 Introduction

A large amount of computer capacity is required if big and complex models are to be simulated with FEM. In order to get realistic results when using FE-analysis every subpart of the entire model must work properly. Every subpart cannot be modeled as if they were to be modeled individually, this would give too long simulation times. Therefore it is needed to idealize some subparts as best as possible, without jeopardizing their basic functionality. In some cases, where a specific subpart occurs several times in one model, for example rivets and bolts, an idealization of the specific subpart is required.

In this case, it is about getting the riveted joints which are connected in riveted realistic flange model to work properly under a simulation process. Since these riveted joints are well used in the model, a good idealization is required which behaves similar to a regular rivet. It is vital that the stiffness of every idealized rivet in the big model is equivalent to the reality so that it will not affect any other subpart in the model. It is important when the big complex model is under load in a simulation that every riveted joint experiences the same amount of load it would have experienced in the reality. If the riveted joints absorb more force than they normally do, visualization of the most critical parts that this specifically load simulation was meant for will not show the real life-span of these subparts. If the riveted joints would instead absorb less force than they normally would have, some other subpart will absorb this energy, which leads to an incorrect visualization of which parts are critical and which are not. Therefore it is vital that the idealized riveted joint works and behaves similar to the physical joint.

When discussing the stiffness of the idealized rivet the stiffness becomes an important parameter to address. When the stiffness is linear to the displacement under constant load, see equation 4.1, the initial objective is to look at displacement simulations. I.e. if the displacement is known for a specific load on the solid model an idealized equivalent model can be made with the same stiffness by some iterative solution. A structure that is put together by a number of elastic elements has a stiffness that can be described as equation 4.1. If the stiffness is non-linear a linearization of the stiffness is required to be able to find a perfect idealized model.

…Equation 4.1

Where:

Force)

)

3.2 Solid model

In order to state that a useful and reliable idealized model has been found, a reference model is needed that resembles the reality as good as possible.

- The model shall consist of a solid rivet with correct measurements and material data.
- The model shall consist of two solid plates with correct measurements and material data.
- All contact surfaces must be set and act naturally with friction constants.
- The mesh in the model shall lead to results that converge and resembles the reality.
- In a simulation the displacements and stresses should behave in a natural way.
- The riveted joint shall not have any pretension according to 3.1, as they are cold drawn.

To create the FEM model using the list of requirements above, Ansys 11 classic, and Hyperworks were used. The models geometry and mesh was created using Hypermesh. The solid reference models in this study were made of single lap riveted joints, see figure 3.1. Both plates were set to the material Ti-64Al with a modulus of elasticity of approximately 120 GPa. The rivet in the solid model was set to the material Inconel 718 with an elastic modulus of approximately 200 GPa. The material used for the rivet in the solid model is not the same as the rivets in the specification, but they have similar material data and can therefore be used here. All the thick plate nodes on its free edge were locked in all DOF and all the thin plate nodes on its free edge was experiencing a tensile force only in x-direction, thus the force movement is locked in z- and y-direction. The rivets in this study are cold drawn and thus the rivets are filling the plate hole completely. The cold drawn head has been modeled as a “real” head. Thus not completely reflecting the exact geometry of a rivet as in the reality, where one side of the rivet is plastically deformed and becomes uneven. All the contact surfaces have a friction coefficient of 0.1 in the solid model, i.e. contacts between rivet – plates and between thin and thick plate. The contacts were set to CONTA174 (master contact) on the plates and TARGET170 (slave contact) were set on the rivet.

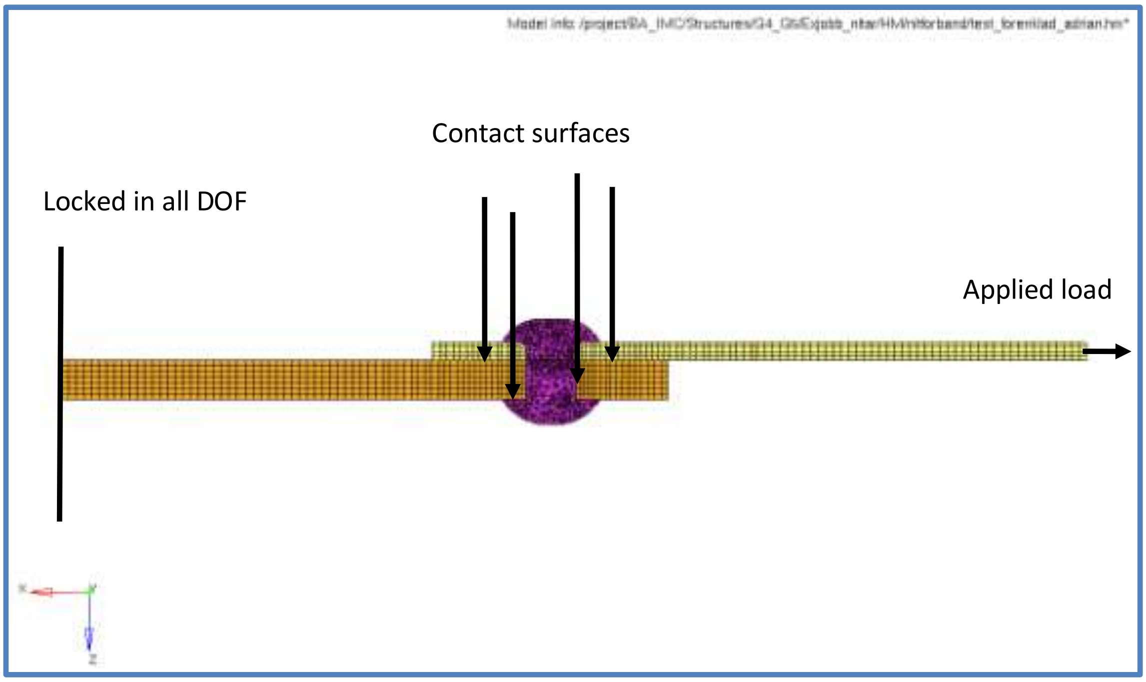
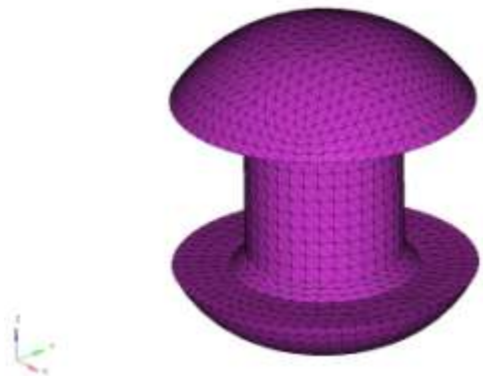


Figure 3.1. Solid models constraints

3.2.1 Solid mesh

All the mesh was created in Hypermesh. The plates were considered to have a simple geometry, thus using Hexa elements with a few penta-elements was preferable here, see figure 3.2. Here the element length was set to 1 mm in size with 0.5 mm elements around the plate holes. The rivet had a more complex geometry and was therefore meshed only with tetra elements, the size of these elements were set to 0.3 mm with some minor adjustments around the edges and in places of interest as in the rivets shear plane. In order to analyze the shear- and tensile stresses in the rivet a plane was created in the rivets shear plane. All the elements in the solid mesh were set to second order elements.



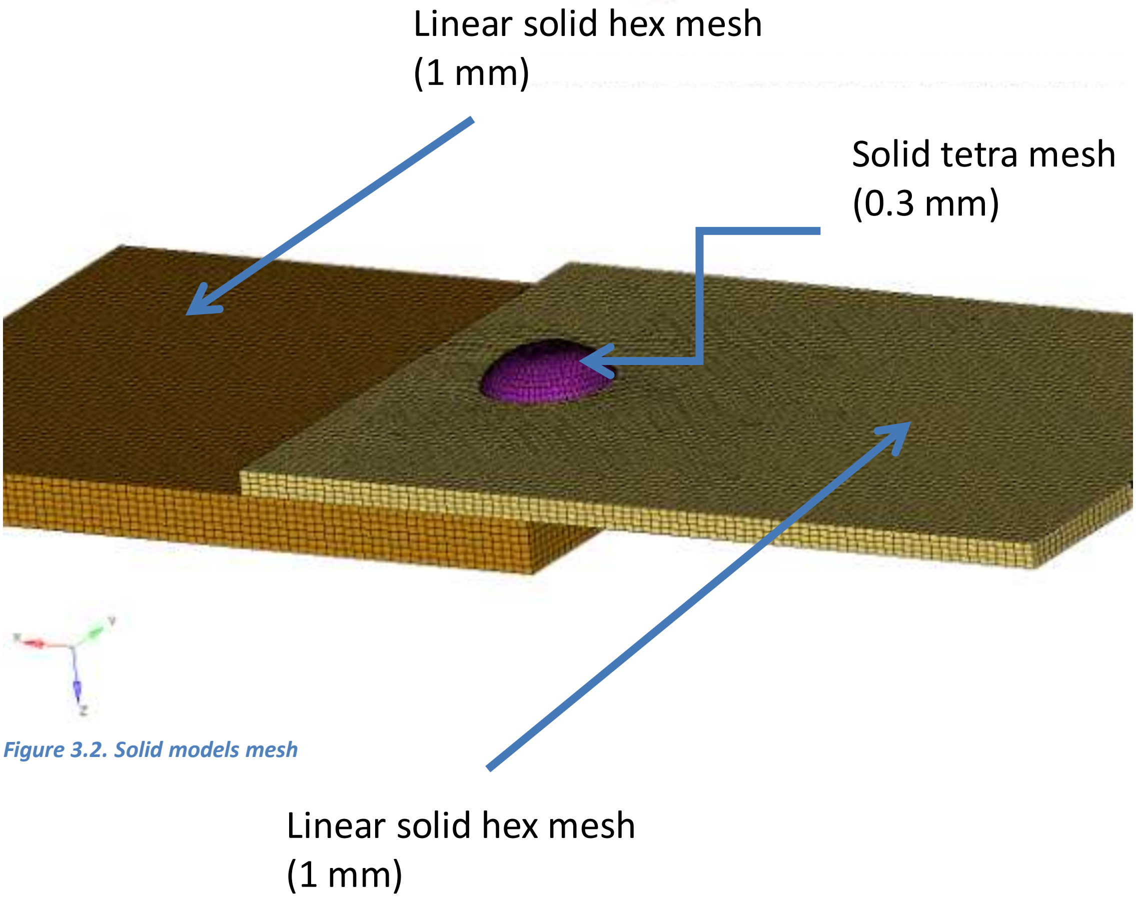


Figure 3.2. Solid models mesh

3.3 1-D rivet modeling

The reference model to all the 1-D rivet studies is the correspondent solid model. The aim and purpose with the 1-D rivet modeling is to find an idealized model of a riveted joint by trading the solid rivet to a beam-, spring- and/or rigids elements, see figure 3.3. Hence, the plates mesh is exactly the same. The idealized model should behave similar to the solid model in terms of stresses, displacements and stiffness. This will lead to shorter calculation times and will save computer memory while still obtaining the same results.

As a starting point for our 1D analysis, a rivet model consisting of BEAM-element and CERIGS will be used. This will be called the BEAM-model. The BEAM-model consists of a BEAM188 element (shown in red) with a circular cross section. CERIGS (shown in blue) connects both of the beam nodes to the whole inner mantle surface of the hole, see figure 3.3. The beam element length is set to 0.1 mm and is centered along the shear plane of the joint. In all the 1-D rivet studies the contacts between rivet and plates and between the plates has been removed. This is chosen to be the reference model for all other types of 1D-rivet models since Beam-elements and CERIGS are a common method to model rivets and bolts. For more information about the BEAM188 element and CERIGS see Appendix A – Ansys 11.0 help manual.

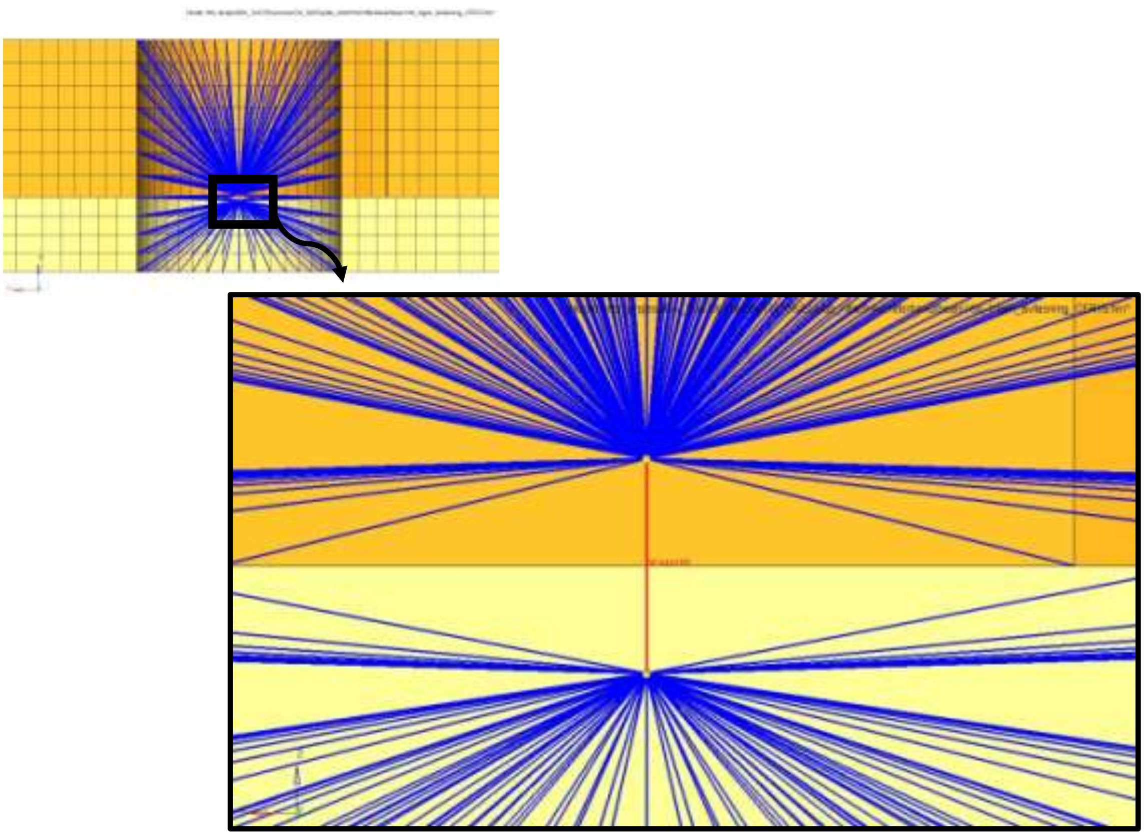


Figure 3.3. BEAM - model

3.3.1 1-D rivet mesh

The mesh on all the 1-D rivet models needs to be similar to the solid mesh (disregarding the rivet) to be able to compare the solid reference model to our 1-D rivet studies. The different rivet replacements used in this study includes BEAM188 and COMBIN14, they are combined in multiple ways with the rigids RBE3 and CERIG elements.

Figure 3.4 shows an example on how it could look with the solid rivet traded for a BEAM element. Figure 3.5 shows how it could look like when the solid rivet is traded for a COMBIN14 element (the COMBIN14 element has a length of 0 mm in this study). The blue lines in figure 3.3, 3.4 and 3.5 are the CERIGS.

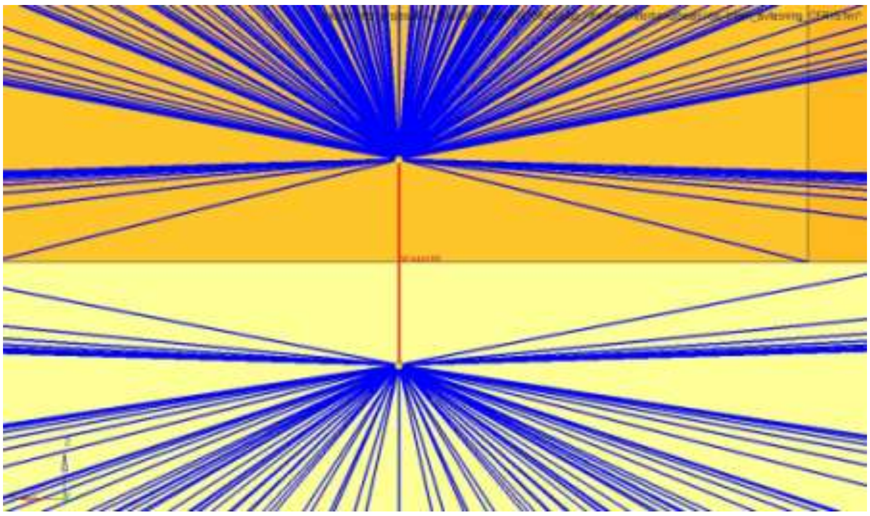


Figure 3.4 – BEAM model

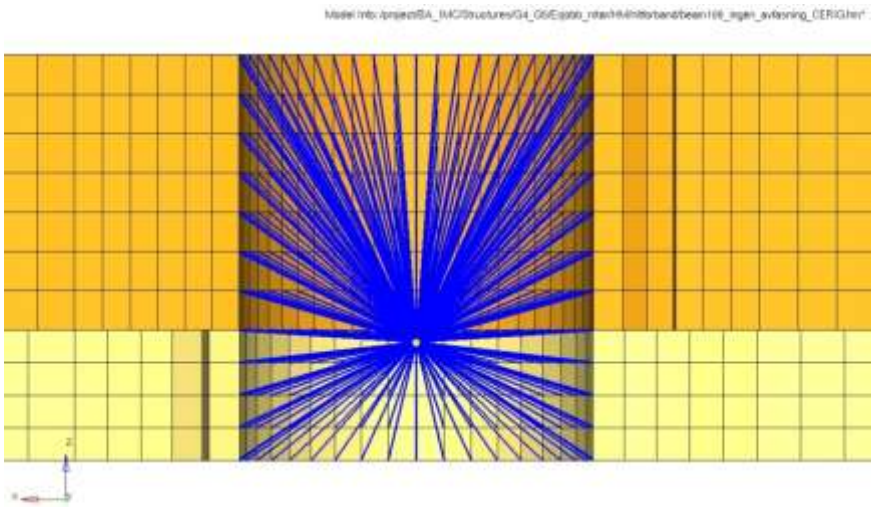


Figure 3.5 – Spring model

The BEAM188 element is a linear 2-node beam with a circular cross section. This element can also simulate a square sectional beam. The COMBIN14 element is a 2-node spring with associated stiffness factors in all six DOF. CERIGS are rigids which are used to rigidly connect one master node to a slave node. The RBE3 element is similar to the CERIG, only a bit weaker. For more information on the different elements see appendix A.

4. Method

All the meshing that is to be performed is done in Hypermesh where full control of the mesh-parameters is available. So, the FE-analysis will be investigated in Hyperview and the FE-modeling will be created using Hypermesh 11.0 and simulated using Ansys 11.0 classic. Drawings of the riveted joints are supplied by VAC. The programs that are to be used are available at VAC, therefore most of the time is going to be spent at Volvo Aero. The first few weeks will be spent learning the different programs that will be used in this degree project.

The first FE-analysis will be performed on the solid model of the riveted joint with the rivet 1, 2 and 3, see models 3.3. It is important that this model resembles the reality as much as possible. This will be done by meshing the solid model well enough and by using contact surfaces in the whole model with the correct dimensions supplied by the drawings, i.e. making it as close to reality as possible.

An idealized 1-D model of the riveted joint is created with as similar properties to the solid reference model as possible, but has a much shorter and less demanding simulation time. Facts and guidance of different idealization techniques in the Ansys 11.0 help file is available as this is a well known problem throughout different industries. Our supervisor here at Volvo Aero will be at our service when help is needed.

In order to decide if the 1-D rivet models are good enough a reference value is needed from the solid model. Investigation of the solid model begins with looking into some specific parameters that must be equivalent to the 1-D model, these are as follows:

- Correct shear stress in both plates, located 10 mm from the hole center (YZ-plane, arises due to bending of the plates under load)
- Correct tensile stress in both plates, located 10 mm from the hole center (X-direction, arises due to displacement of the plates under load)
- Correct bending stiffness, i.e. correct angular displacement (Max. deviation ≤ 10 %)
- Correct tensile stiffness, i.e. correct displacement in x-direction (highly prioritized, max. deviation ≤ 1 %)
- Correct hole deformation (visual inspection)

The most critical area where the idealized model must be the same as the solid model is between 1200 N- and 2400 N tensile force. Because the assumed fatigue limit of rivet 1 is 1200 N and 2400 N is the tensile strength of the rivet. Figure 4.1 shows the joints tensile stiffness. If we get the load case at 1800 N to be equivalent to the 1-D model a good idealized model has been found (See the dashed trend line). That is because the 1-D rivet models stiffness functions is always linear.

The solid models tensile stiffness is not linear. This is because the presence of contacts in the solid model. There is a very small gap between all the contacting surfaces in the model with smaller load cases. It takes some load for full contact between all surfaces, hence the non-linearity.

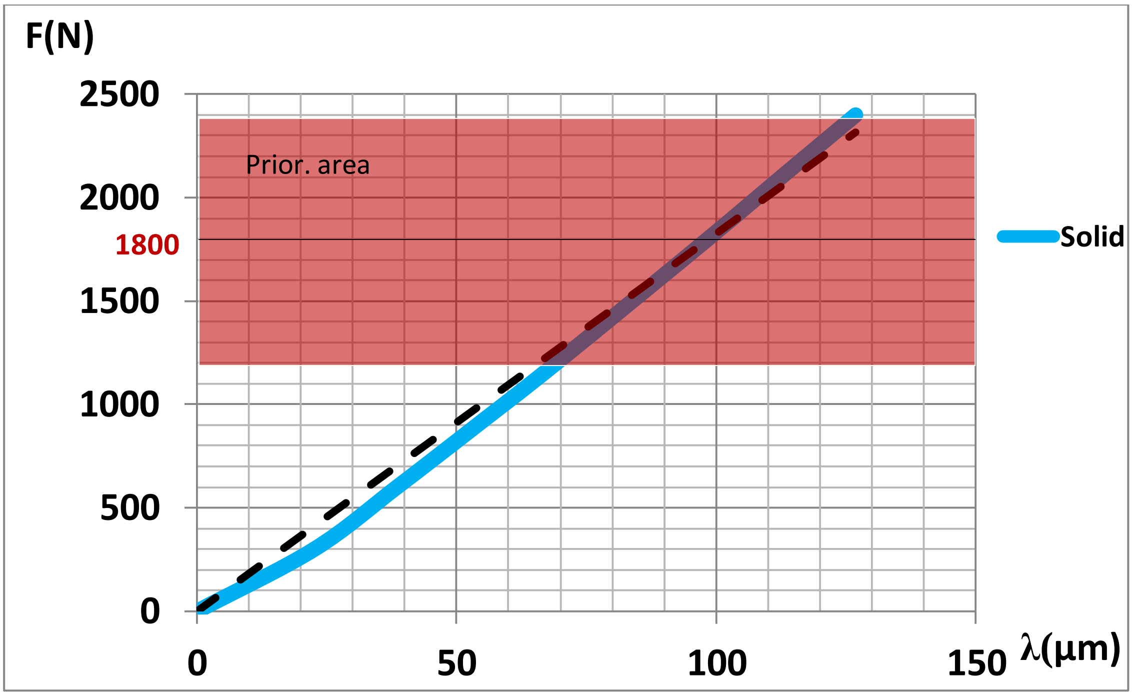


Figure 4.1. Example of a riveted joints tensile stiffness

Figure 4.2 shows the bending stiffness of the solid model. The solid models bending stiffness is linear. An idealized model with equivalent bending stiffness is much easier to determine than a model with equivalent tensile stiffness because of the tensile stiffness non-linearity.

Comparison between the bending-, tensile stiffness and mean stresses in the plates between all the 1-D rivet models with the corresponding BEAM-model will be done in all the upcoming 1-D modeling chapters. This will be done in order to see the deviation between the models.

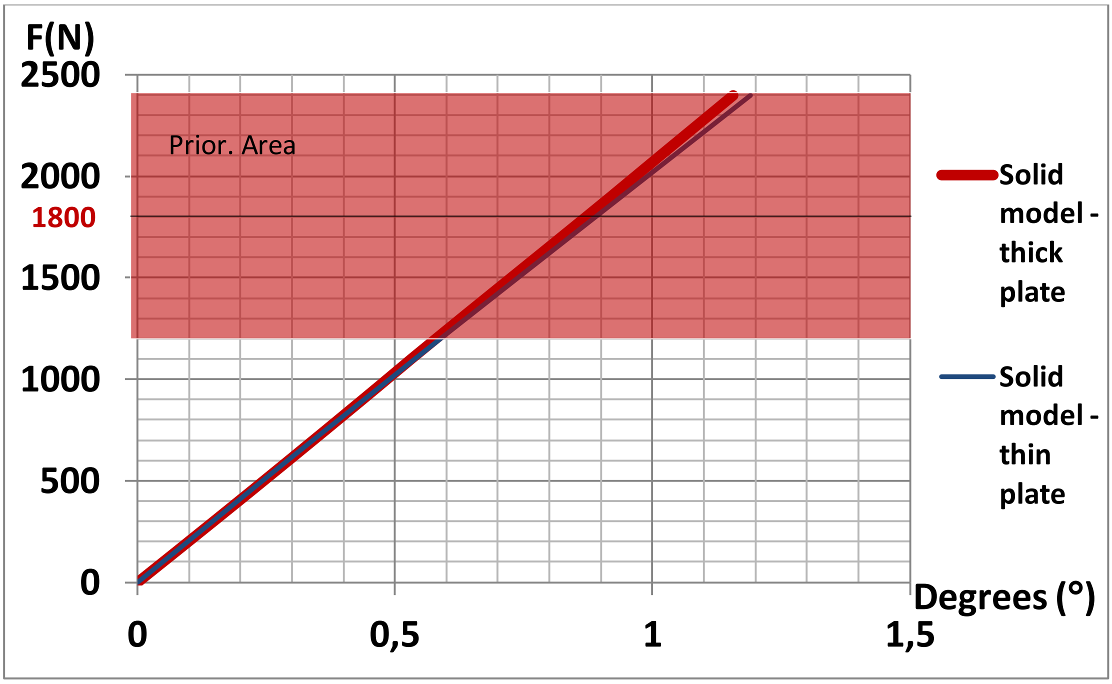


Figure 4.2. Example of a riveted joints bending stiffness

The idealization method used in this study on the 1-D rivet models is divided into 4 steps essentially, see figure 4.3. It is called the “Inside and out-method”, which means that the purpose of this method consists of working from the inside of the riveted joint (the hole) to the very outside of it (global deformation) in the following order:

- **Step 1** is to control the hole deformation so that it is similar to the Solid reference model in order to get similar plate- and hole stresses.
- **Step 2** is to by iterative solution find the correct placement of the springs in the hole in order to get the correct length of the torque arms from the centre of both plates to the position of the springs in order to get the correct torque of both plates. This will lead to a correct angular displacement in the hole i.e. bending stiffness for both plates.
- **Step 3** is to control the mean stresses in the plates 10 mm from the holes center in order to determine that no underlying deviations in the models are present. A significant fact is that this step is only used in order to control the deviations, no deeper evaluations have been done of why the stresses behave like they do.
- **Step 4** is to by iterative solution determine the correct tensile stiffness in the joint by altering the stiffness factor in all the springs DOF.

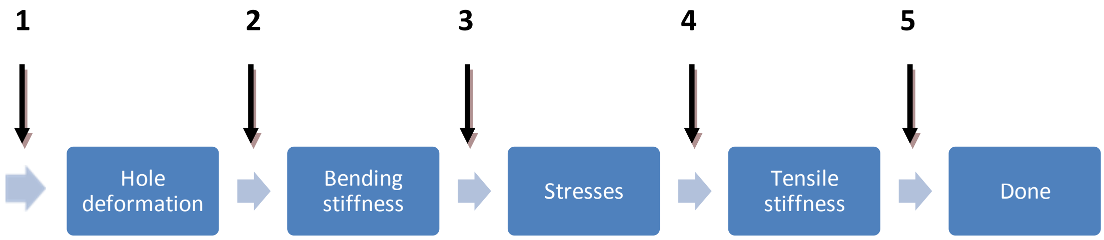


Figure 4.3. The inside – out method

- **Step 5,** the idealization process is completed.

Measurement for obtaining tensile stiffness in x-direction takes place on the free edge of the thin plate, in order to get global tensile stiffness for the whole joint. The bending stiffness is determined by measuring the angular displacement in the rivet hole and the tensile stiffness is determined by measuring the x-direction displacement, see figure 4.4.

Tensile and shear stresses analysis will be measured from a cross section in both plates 10 mm from the hole center and are the mean stress values, see figure 4.5 and 4.6.

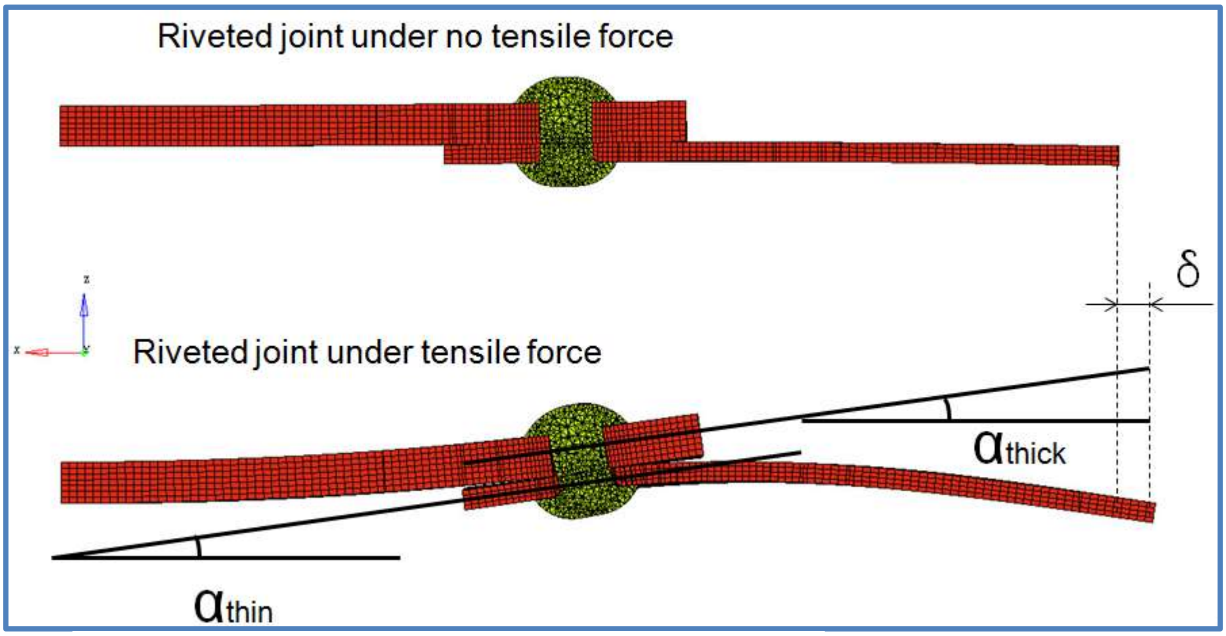


Figure 4.4. Angular displacement measurments

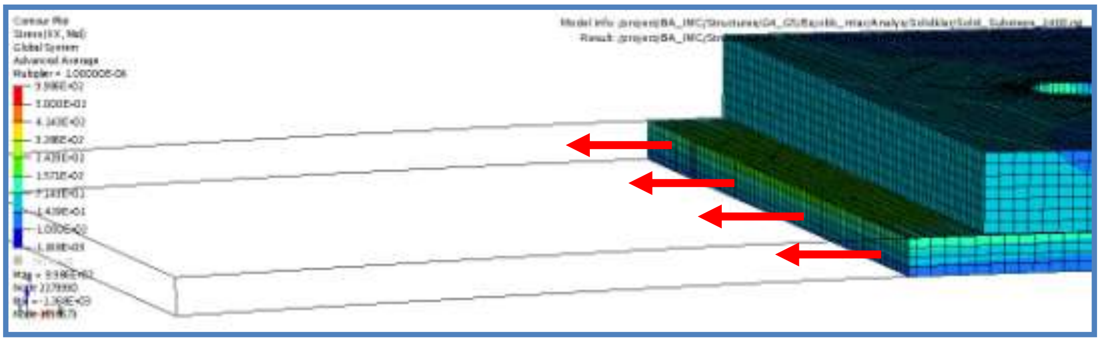


Figure 4.5. Plates cross section, 10 mm from the hole on the thin plate. Tensile stress plotted.

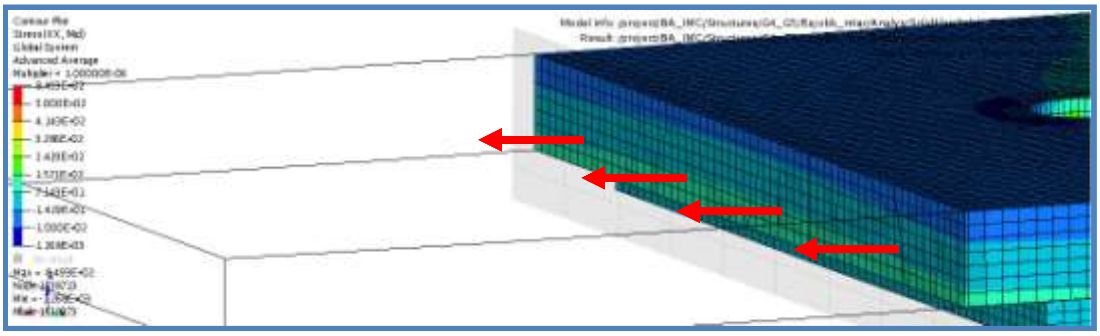


Figure 4.6. Plates cross section, 10 mm from the hole on the thick plate. Tensile stress plotted.

5. Solid model – Results

In this chapter all our results from the solid models will be presented in order to have clear reference values for the future idealized models.

5.1 Alfa w. Rivet 1 - results

The solid model Alfa dimensions are shown in table 5.1.

Riveted join model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thick plate	Thin plate
Alfa	1	3.3	1.2	2.54

Table 5.1. Alfa w. Rivet 1

The stresses in both plates 10 mm from the holes center were analyzed, see table 5.2.

Stress table (1800 N)		Solid model
Tensile stress	Thin plate	52 MPa
	Thick plate	24 MPa
Shear stress	Thin plate	39 MPa
	Thick plate	22 MPa

Table 5.2. Stresses in the plates

The tensile stiffness of the solid model is shown in figure 5.1. This stiffness function is non-linear due to the presence of contact surfaces in the Solid model. A trend line (dashed line) is also plotted to the tensile stiffness function. It is clear that the linearized function (dashed line) of the solid model and the non-linearized function of the solid model are both equivalent to the center of our prior load area.

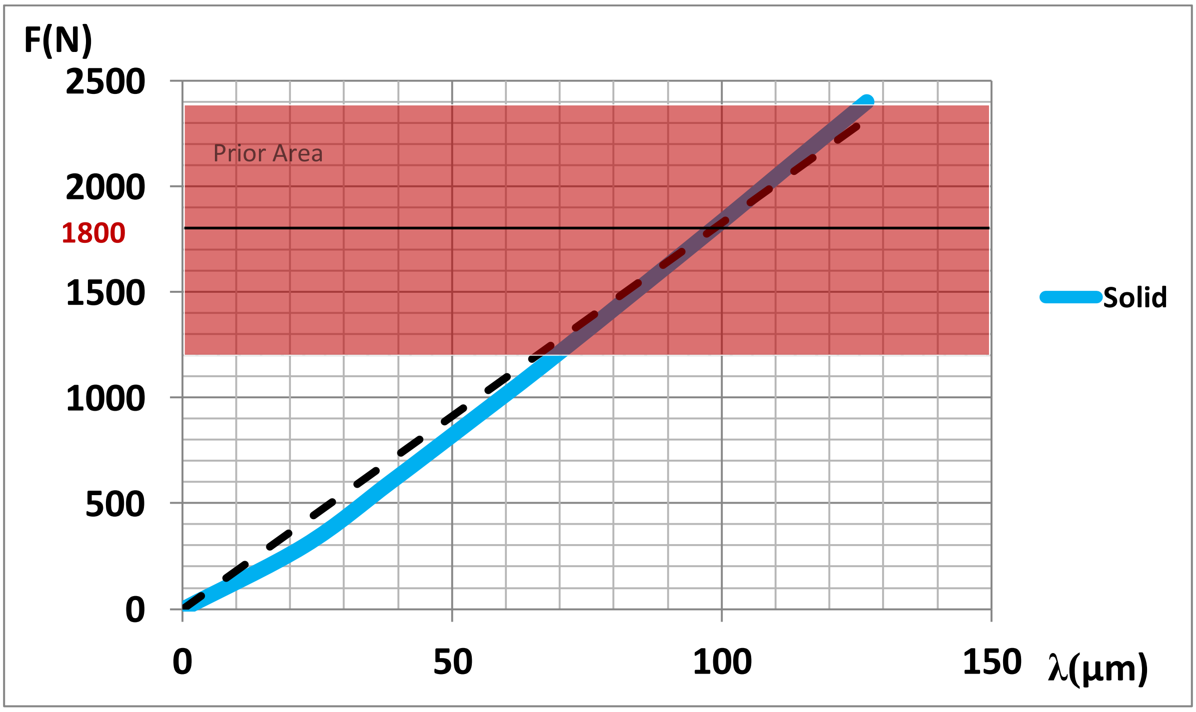


Figure 5.1. Alfa w. Rivet 1, tensile stiffness

The angular displacement in the hole for the thin and thick plate can be seen in figure 5.2 and represents the solid model Alfa w. rivet 1 bending stiffness. The bending stiffness in the solid model is linear.

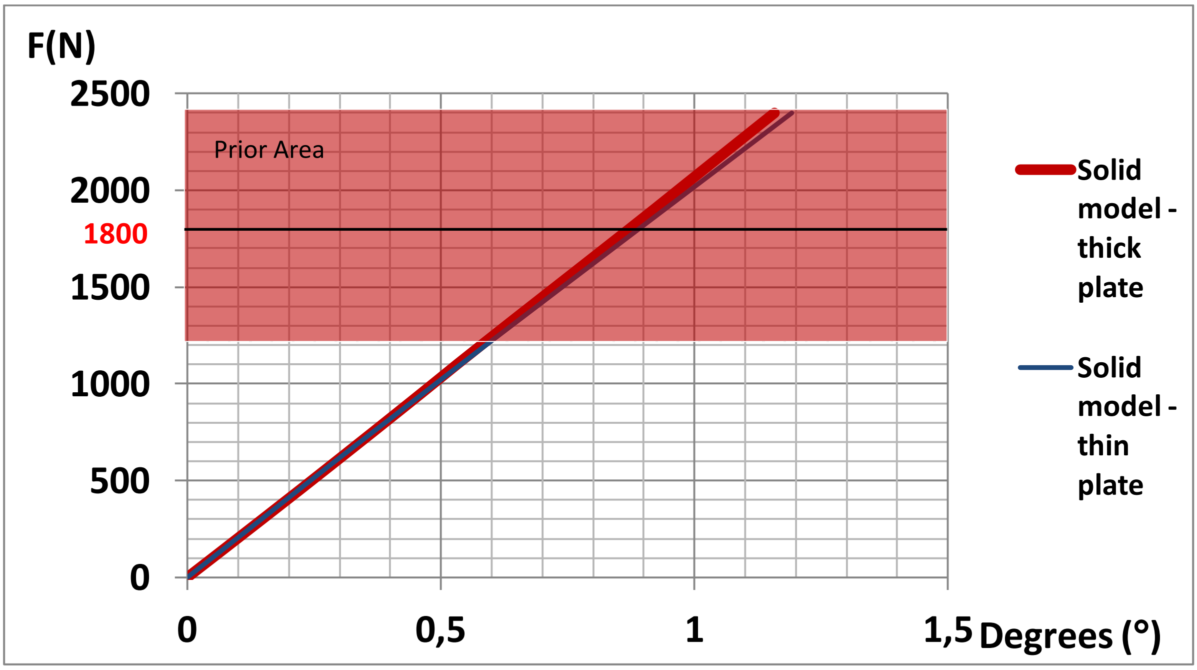


Figure 5.2. Alfa w. Rivet 1, bending stiffness

Table 5.3 represents the Reference values for the solid model Alfa of a load value of 1800N.

Alfa stiffness (1800 N)	Angular displacement	Displacement in x-direction	Stiffness factor [10^7]
Thin plate	0.8894°	98.4 µm	1.8 N/m
Thick plate	0.8712°	0 µm	-

Table 5.3. Alfa w. Rivet 1, reference values at 1800 N

5.2 Alfa w. Rivet 2 - results

Solid model Alfa w. Rivet 2 geometry is the same as solid model Alfa w. Rivet 1, only that rivet 1 has been traded for rivet 2, see table 5.4.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Alfa	2	4.1	1.2	2.54

Table 5.4. Alfa w. Rivet 2

The stresses in both plates 10 mm from the holes center were analyzed, see table 5.5.

Stress table (2850 N)		Solid model
Tensile stress	Thin plate	78 MPa
	Thick plate	37 MPa
Shear stress	Thin plate	66 MPa
	Thick plate	35 MPa

Table 5.5. Stresses in the plates

Figure 5.3 shows the tensile stiffness of Solid model Alfa with rivet 2. A trend line has been plotted to show that a perfect idealized model is equivalent with the solid model if the stiffness is equivalent at 2850 N which lies in the center of the prior. area.

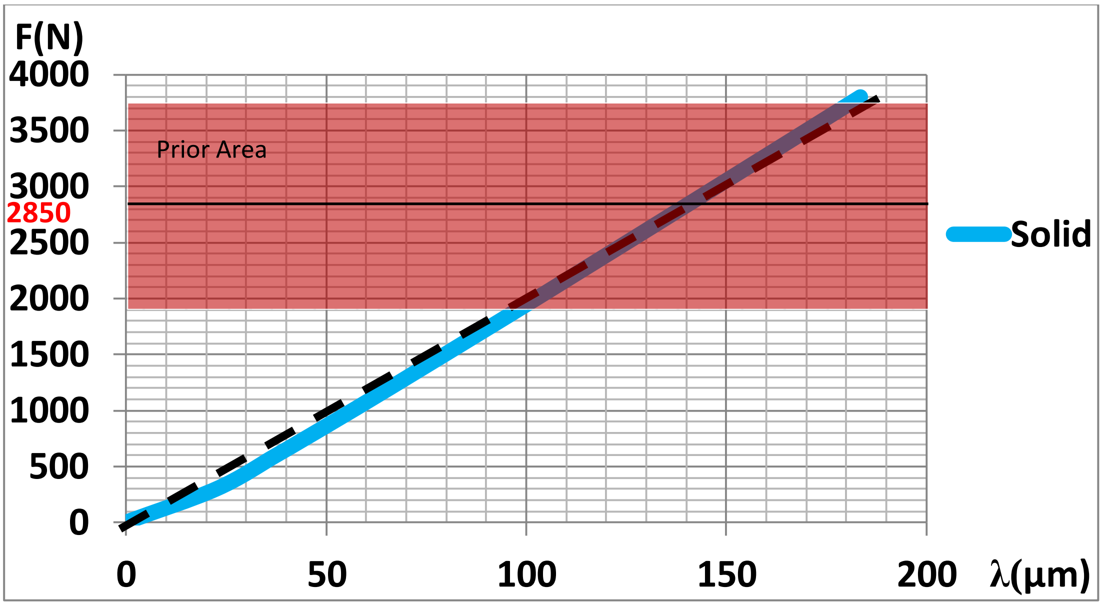


Figure 5.3. Alfa w. Rivet 2, tensile stiffness.

The bending stiffness is linear as in the Alfa model w. Rivet 1, see figure 5.4.

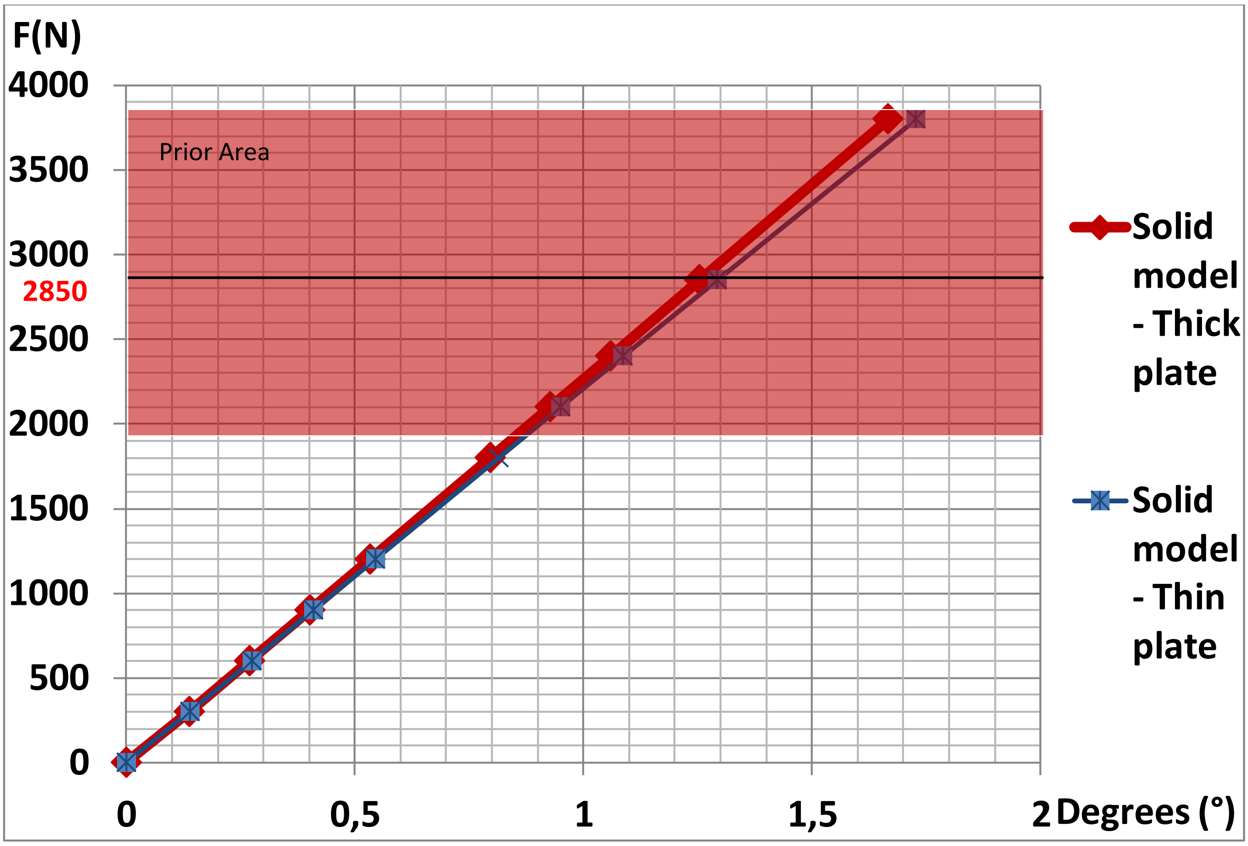


Figure 5.4. Alfa w. Rivet 2, bending stiffness

Table 5.6 shows solid model Alfas w. Rivet 2 reference values.

Alfa stiffness (2850 N)	Angular displacement	Displacement in x-direction	Stiffness factor [10^7]
Thin plate	1.2924˚	140.82 µm	2.0 N/m
Thick plate	1.2536˚	0 µm	0

Table 5.6. Alfa w. Rivet 2, reference values at 2850 N

5.3 Alfa w. Rivet 3 - results

Solid model Alfa with rivet 3 geometry is the same as in Alfa w. Rivet 1 only that the rivet 1 has been traded for rivet 3, see table 5.7.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Alfa	3	4.5	1.2	2.54

Table 5.7. Alfa w. Rivet 3

The stresses in both plates 10 mm from the holes center were analyzed, see table 5.8.

Stress table (4200 N)		Solid model
Tensile stress	Thin plate	124 MPa
	Thick plate	58 MPa
Shear stress	Thin plate	95 MPa
	Thick plate	51 MPa

Table 5.8. Stresses in the plates

Figure 5.8 shows the global tensile stiffness for solid model Alfa with rivet 3. Due to the larger rivet in this model the tensile stiffness function is almost linear, see the dashed trend line.
A good idealized model would be found if the load case at 4200 N is equivalent with this solid model.

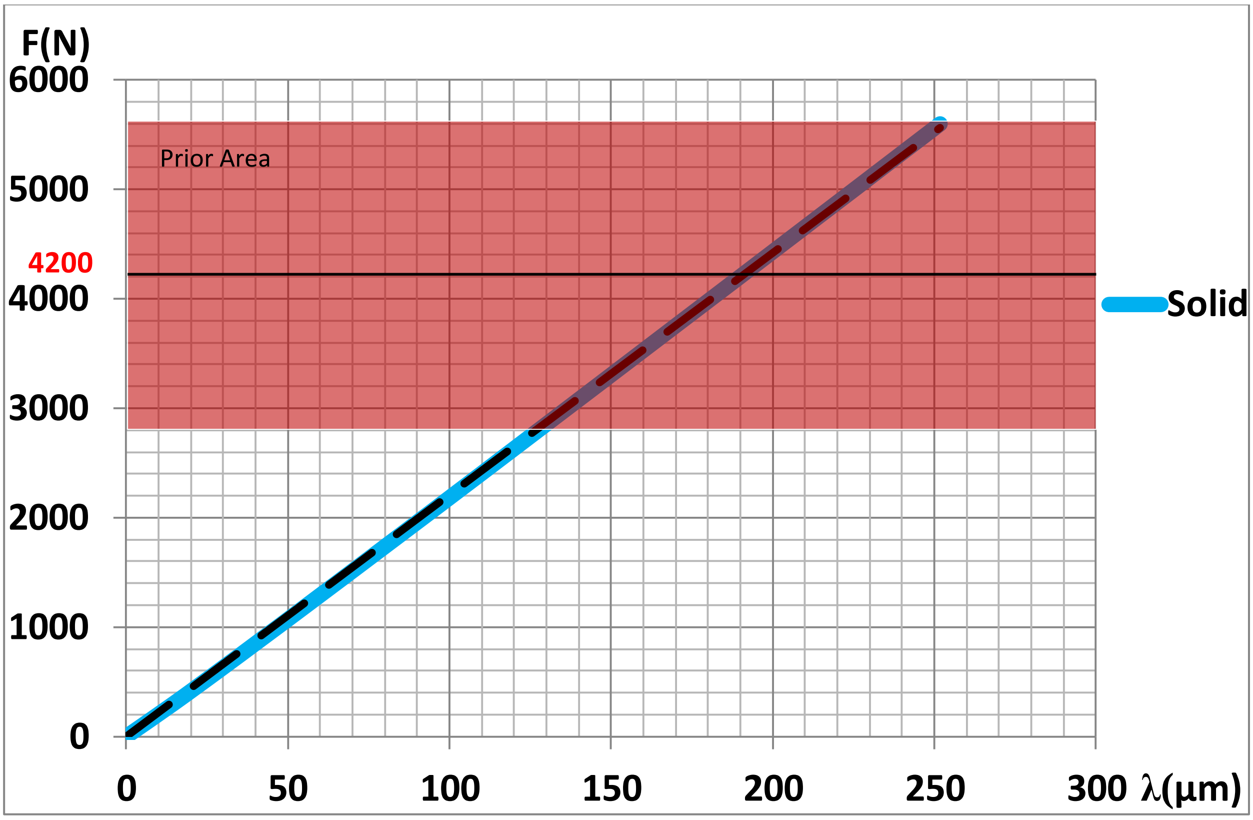


Figure 5.8. Alfa w. Rivet 3, tensile stiffness

Figure 5.9 shows the bending stiffness for solid model Alfa with rivet 3. The bending stiffness function for both thin and thick plate is linear.

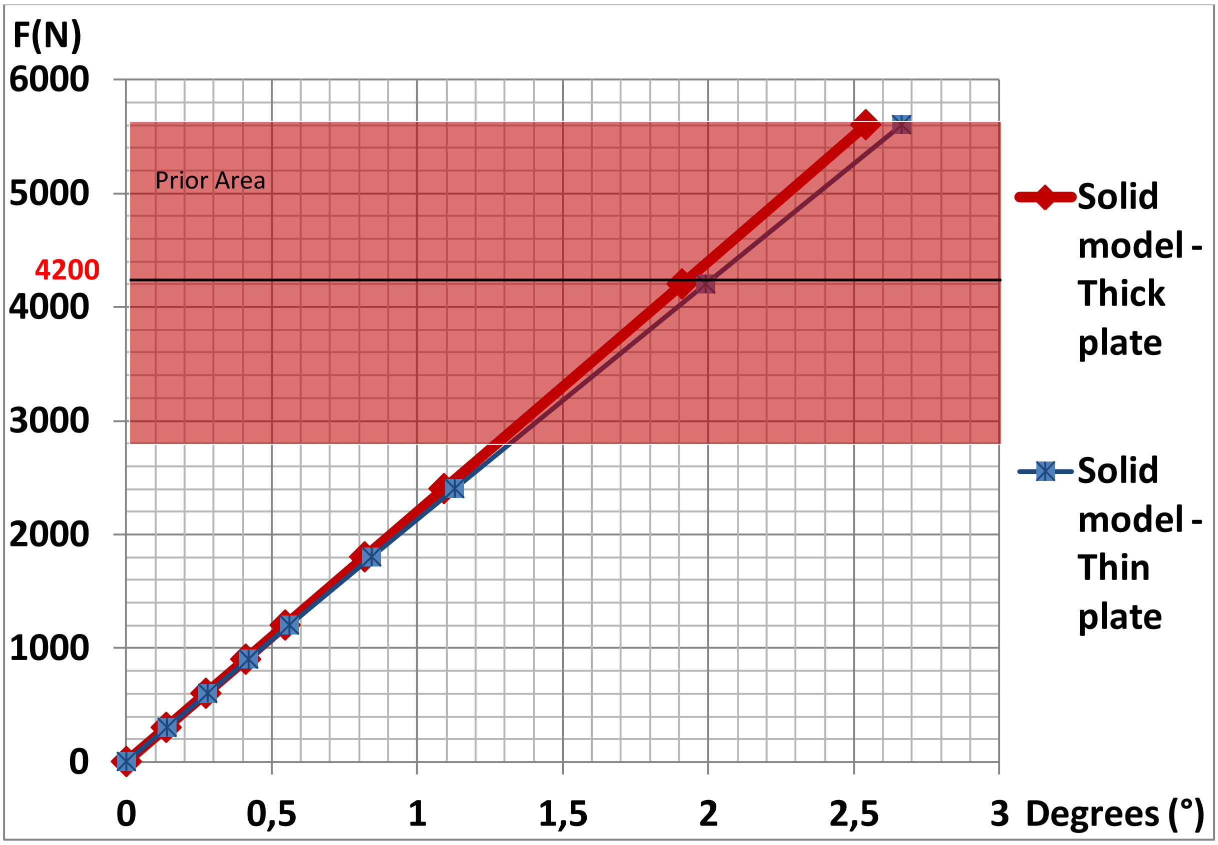


Figure 5.9. Alfa w. Rivet 3, bending stiffness.

Hence, the reference values for model Alfa with rivet 3 are presented in table 5.9.

Alfa stiffness (4200 N)	Angular displacement	Displacement in x-direction	Stiffness factor [10^7]
Thin plate	1.9088°	189.8 µm	2.2 N/m
Thick plate	1.9911°	0 µm	0

Table 5.9. Alfa w. Rivet 3, reference values at 4200 N

5.4 Beta w. Rivet 1 - results

The solid model Beta dimensions can be seen in table 5.10.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Beta	1	3.3	1.2	2.0

Table 5.10. Beta w. Rivet 1

The stresses in both plates 10 mm from the holes center were analyzed, see table 5.11.

Stress table (1800 N)		Solid model
Tensile stress	Thin plate	48 MPa
	Thick plate	31 MPa
Shear stress	Thin plate	41 MPa
	Thick plate	25 MPa

Table 5.11. Stresses in the plates

As seen in appendix B; Beta, the linearized function (dashed line) of the solid model and the non-linearized function of the solid model are both equivalent to the center of our prior load area. As in the Alfa-model 1800 N is the reference load.

Similar to the Alfa-model, appendix B, section Beta, signifies that the bending stiffness for this model is linear too, which makes the bending stiffness idealization less troublesome.

The reference values for angular and x-direction displacement for solid model Beta is presented in table 5.12.

Beta stiffness (1800 N)	Angular displacement	Displacement in x-direction	Stiffness factor [10^7]
Thin plate	1.0716°	92.98 µm	1.9 N/m
Thick plate	1.0388°	0 µm	0

Table 5.12. Beta w. rivet 1, reference values at 1800 N

5.5 Gamma w. Rivet 1 - results

The solid model Gamma dimensions can be seen in table 5.13.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Gamma	1	3.3	1.0	1.2

Table 5.13. Gamma w. Rivet 1

The stresses in both plates 10 mm from the holes center were analyzed, see table 5.14.

Stress table (1800 N)		Solid model
Tensile stress	Thin plate	57 MPa
	Thick plate	48 MPa
Shear stress	Thin plate	54 MPa
	Thick plate	37 MPa

Table 5.14. Stresses in the plates

As seen in appendix B; Gamma, the perfect reference load case is at 1800 N, according to the prior area. In future studies of the solid models, investigations of the different kind of parameters will only take place at the load case of 1800 N when performing studies on rivet 1. If this specific load case matches the associated 1-D rivet model, an idealized model has been found.

Similar to the Alfa- and Beta-model, appendix B, section Gamma, signifies that the bending stiffness for this model is linear too.

The reference values for angular and x-direction displacement for solid model Gamma is presented in table 5.15.

Gamma stiffness (1800 N)	Angular displacement	Displacement in x-direction	Stiffness factor [10^7]
Thin plate	1.7933°	114.6 μm	1.6 N/m
Thick plate	1.7777°	0 μm	0

Table 5.15. Gamma w. rivet 1, reference values at 1800 N

5.6 Delta w. Rivet 1 - results

In this analysis, there was trouble with the convergence values. The final Delta-simulation contained penetration of the contact-surfaces. Therefore these test result are not trustworthy and shouldn't be trusted as a good conclusion of the model. However, results are presented but further work is required.

The solid model Delta dimensions can be seen in table 5.16.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Under plate	Upper plate
Delta	1	3.3	2.54	2.54

Table 5.16. Delta w. Rivet 1

The stresses in both plates 10 mm from the holes center were analyzed, see table 5.17.

Stress table (1800 N)		Solid model
Tensile stress	Under plate	24 MPa
	Upper plate	24 MPa
Shear stress	Under plate	20 MPa
	Upper plate	17 MPa

Table 5.17. Stresses in the plates

The reference values for angular-displacement and x-direction displacement for solid model Delta is presented in table 5.18 at the specific load case of 1800 N tensile force.

Delta stiffness (1800 N)	Angular displacement	Displacement in x-direction	Stiffness factor [10^7]
Under plate	0.3539˚	95.36 µm	1.9 N/m
Upper plate	0.3598˚	0 µm	0

Table 5.18. Delta w. rivet 1, reference values at 1800 N

5.7 Epsilon w. Rivet 1 - results

The solid model Epsilon dimensions can be seen in table 5.19.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Epsilon	1	3.3	2.0	2.5

Table 5.19. Epsilon w. Rivet 1

The stresses in both plates 10 mm from the holes center were analyzed, see table 5.20.

Stress table (1800 N)		Solid model
Tensile stress	Thin plate	31 MPa
	Thick plate	23 MPa
Shear stress	Thin plate	25 MPa
	Thick plate	18 MPa

Table5.20. Stresses in the plates

The reference values for angular-displacement and x-direction displacement for solid model Epsilon is presented in table 5.21 at the specific load case of 1800 N tensile force.

Epsilon stiffness (1800 N)	Angular displacement	Displacement in x-direction	Stiffness factor [10^7]
Thin plate	0.5002°	72.91 µm	2.5 N/m
Thick plate	0.4890°	0 µm	0

Table 5.21. Epsilon w. rivet 1, reference values at 1800 N

6. Idealized modeling techniques

This chapter contains the explanation of some modeling techniques and their advantages and disadvantages.

6.1 BEAM188 using RBE3

The method used here is substituting the solid rivet with BEAM188 and RBE3 elements. The beam elements, the blue ones, were centered on the shear plane, see figure 6.1. The beam element is connected by two nodes, the master nodes of the RBE3, which are placed in the center of the plate hole. These nodes are then connected to all of the nodes in the plate hole by RBE3 elements. The cross section radius of the beam in this study is set to 1.65 mm, which is the correct dimension after the rivet has been cold drawn, hence the beams cross section radius is filling the hole completely.

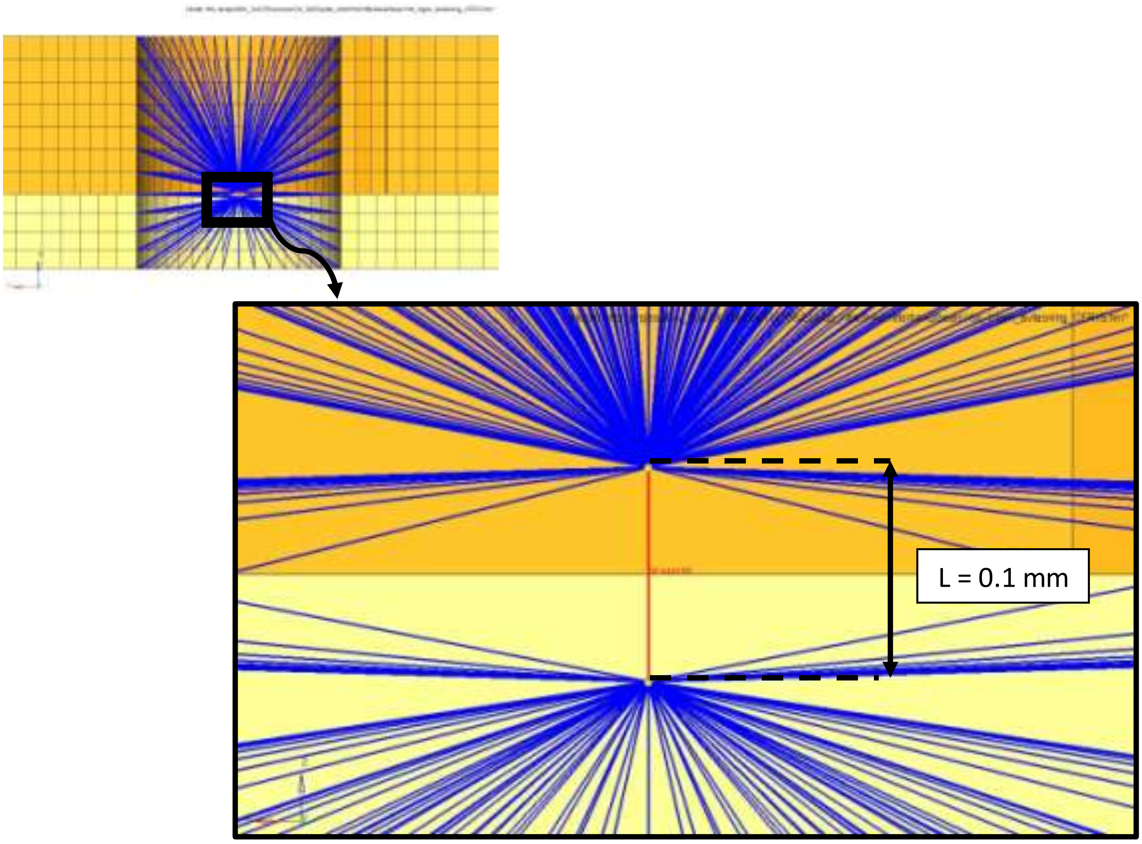


Figure 6.1. BEAM188 using RBE3 element, centered along the shear plane (L=0.1 mm)

A vital factor that must be equivalent is the global deformation of the joint. RBE3 element is in conflict with the required stiffness of the hole deformation. Hence, this method is not a preferable solution to the problem, as the hole deformation must be equivalent in order to create a trustful idealization. Investigation of BEAM188 using CERIGS are therefore presented in chapter 6.2.

6.2 BEAM188 using CERIG

Substituting a solid rivet using a beam with CERIGS will be used as the idealized reference model, see figure 6.2. This method is identical to the method used in 6.1 but the difference is that the RBE3 elements are replaced with CERIG rigids. See appendix A for more info on the RBE3 elements and the CERIGS. The hole deformation in the solid model is similar to the deformation in the 1-D model when using CERIGS.

Similar bending- and tensile stiffness to the solid model were determined by iterative solution. Similar bending- and tensile stiffness to the solid model were accomplished by changing the length of the beam and the radius of the beam.
The radius of the beam element that gave similar tensile stiffness to the solid model was 14% of the initial hole /rivet radius. The length of the beam element that gave similar bending stiffness was 14% of the initial beam length. As in chapter 6.1, the beam element is centered on the shear plane.

Thus, a trustworthy idealized model has been found. But, altering the tensile stiffness using springs is easier because rather than changing the beams cross section, changing the k-factor for every spring gives us the same effect, i.e. changing the k-factor for all DOF in every spring.

To come up with a good idealized model by using a BEAM188-element combined with CERIG is not possible without changing the cross section of the BEAM188-element. A relatively good idealization is not possible by just changing the length and the position of the elements.

This method is not a good solution to the problem because it is neither a general solution nor a good enough modeling method, investigations of using springs instead of a beam element is presented in chapter 6.3.

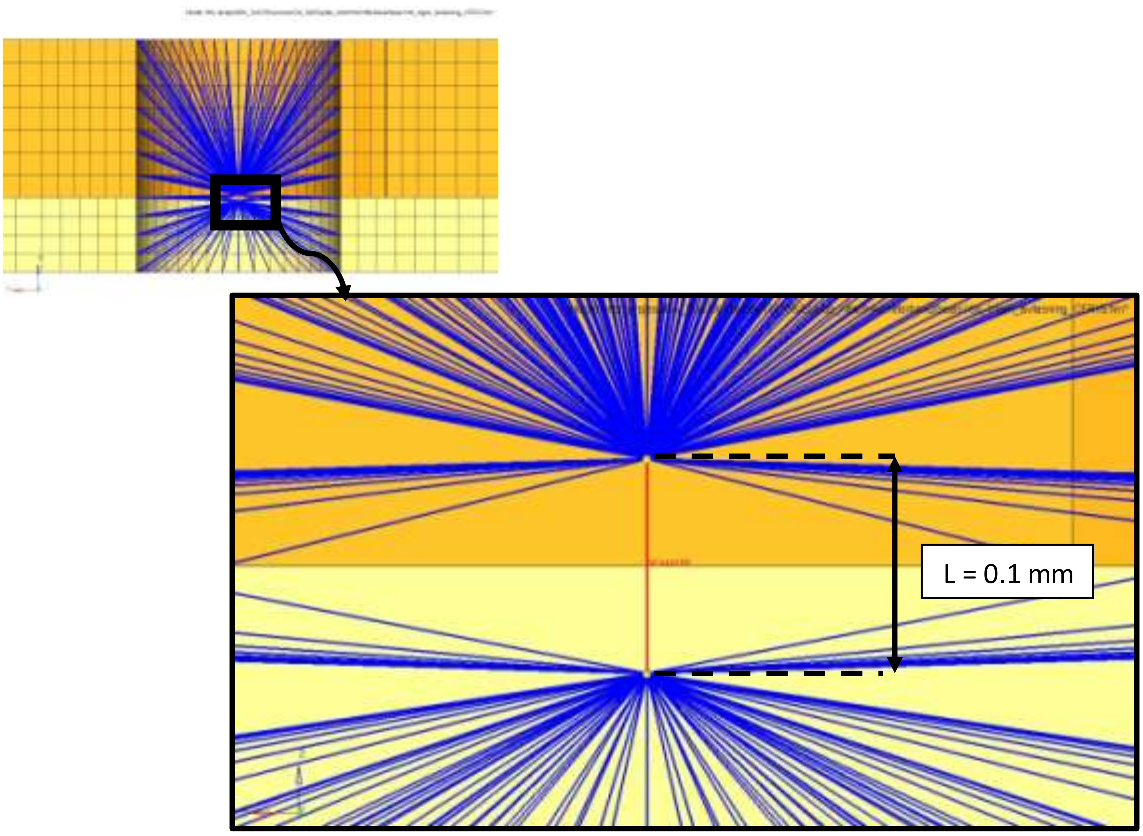


Figure 6.2. BEAM-model centered along the shear plane (L=0.1 mm)

6.3 COMBIN14 using CERIGS

The hole deformations when using COMBIN14-springs and CERIGS is similar to the hole deformation in chapter 6.2. Analogously with 6.1 and 6.2 the constraints for the idealized model were no different here.

At this point, this was the most suitable modeling method of the idealization. The method is very simple and specific. All properties of the joints as stresses in the plates, bending stiffness, tensile stiffness and hole deformations were equivalent to the solid model. A good idealization is easy to make when using springs and CERIGS because bending stiffness and tensile stiffness is not dependent of each other. The "inside-out" method used for all upcoming idealizations is described in chapter 4.

However, some studies were made before this idealization method was used. First off, some different spring rotation constants were tested to ensure that the spring rotation constant (rot. X, rot. Y and rot Z) does not interfere with the bending stiffness. Table 6.1 shows that the angular displacement in the hole does not change much and is negligible when the spring rotation constant in rot. X, rot. Y and rot. Z varies in the interval of — —— .

Angular Displacement

Thin plate	Spring rotation constant = k_{rot} $0 \leq k_{rot} \leq$	0.8697°< v < 0.8987°
Thick plate		0.8475°< v < 0.8580°

Table 6.1. Angular displacement study

The same study was made again, but when only changing the spring stiffness factor in X, Y and Z directions. The difference was that the interval was not as big as in the previous study. Due to an all too weak spring in X, Y, and Z direction results in total loss of stiffness in the joint. But the results were the same, the angular displacement was also here negligible when changing the springs stiffness factor in X, Y and Z directions in a large interval ($10^5 \leq k \leq 10^9$). Hence, according to both of the spring constant experiments, in order to control the bending stiffness of the joint it is only necessary to change the position of the master nodes in the hole. That is why the idealization method presented in chapter 4 works.

At last, confirmation of that the tensile- and bending stiffness in the 1-D rivet model is similar to the Solid reference model model is done by comparing the reference values.

The dimensions for the upcoming 1-D models Alfa, Beta, Gamma, Delta and Epsilon can be found in table 2.2 and in the beginning of every subchapter.

As in chapter 6.1, 6.2 and 6.3 the correct bending stiffness were determined by having the correct amount of torque on the springs. In figure 6.4 the spring-elements is placed off-center from the shear plane, and is placed 0.11 mm from the shear plane and down (Off-center). The off-center measurements in further studies are always measured from the shear plane and down. Figure 6.3 shows the master node placement centered on the shear plane (on-center). Placing the master node on-center of the shear plane is a more preferable solution and that is why it's tested.

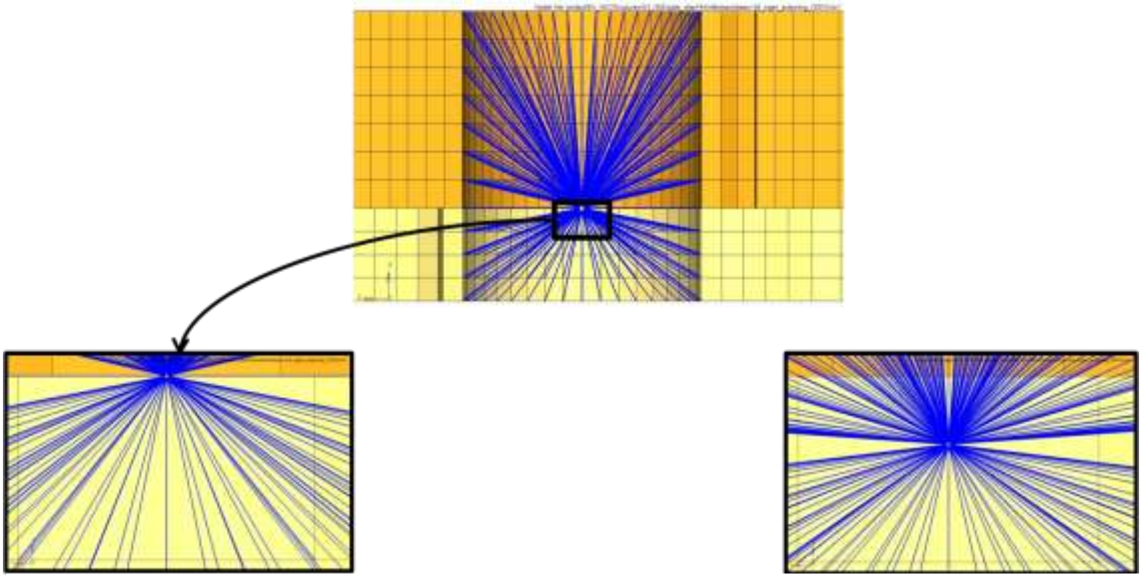


Figure 6.3. Master node on-center.

Figure 6.4. Master node off-center, -0.11 mm.

A key factor that must be equivalent is the bending- and tensile stiffness. When using springs instead of a beam element, the k-factor which is the stiffness factor, is more easily altered. Altering the tensile stiffness using springs is done by changing the k-factor for every spring.
Thus changing the k-factor for all spring-elements (see Appendix A), rather than changing the beams cross section as in chapter 6.1 and 6.2. Each spring has a specific degree of freedom, that's why we use six spring-elements. The bending stiffness was determined by iterative solution and by changing the placement of the master nodes. The bending stiffness is correct when the angular displacements in the hole of the idealized models are equivalent to the angular displacement in the hole of the solid model, see figure 6.5. The solid model is on top and the 1-D model is below, x-displacement is plotted.

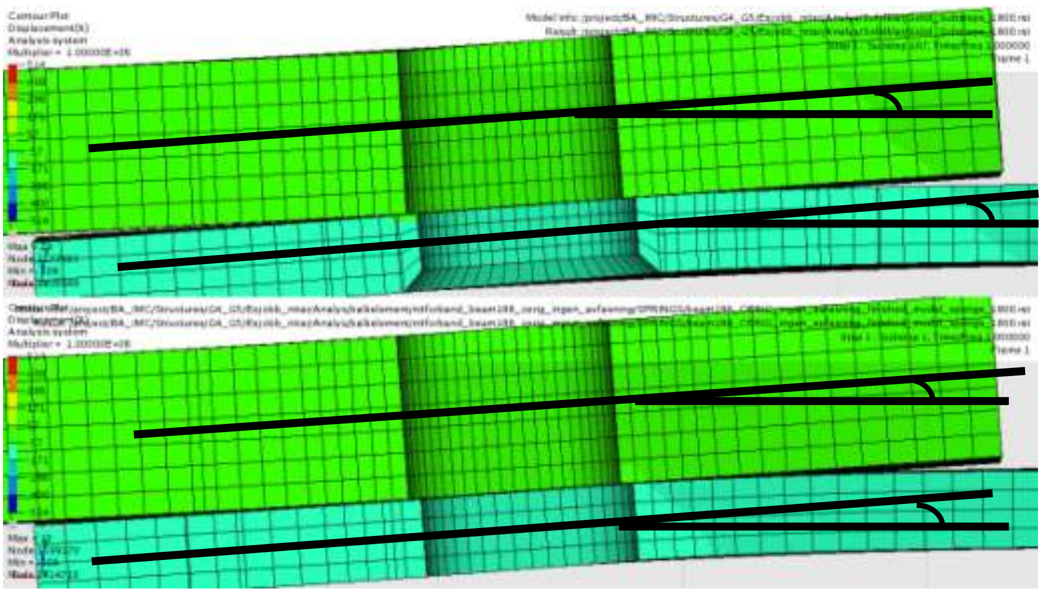


Figure 6.5. Angular displacement measurements. Solid model on top, and 1-D rivet model

7. Idealized model – Results

When using COMBIN14 with CERIGS it is simple to change the joints tensile stiffness just by altering the spring k-factor, and bending stiffness is controlled by placing the master node at the correct place in order to get correct amount of torque around it. This idealization method also led to correct global deformation and accurate tensile- and shear stresses.

7.1 Alfa w. Rivet 1

The dimensions for the 1-D rivet model Alfa with rivet 1 is presented in table 7.1.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Alfa	1	3.3	1.2	2.54

Table 7.1. 1-D rivet model Alfa w. Rivet 1

The angular displacement is presented in table 7.2 for the BEAM-model and when the master node is placed off- and on-center for the spring models. Deviations are also presented for both the 1-D models and the BEAM-model. The angular displacement for the idealized model (off-center) is almost equivalent to the solid model. The BEAM model experiences too large bending stiffness on the thin plate due to a larger torque arm than the 1-D models. The thick plate has a smaller torque arm than the 1-D model and thus has a lower bending stiffness. The most pleasant solution method is by placing both the spring nodes in the shear plane of the idealized model (on-center) at all times. When the master nodes are placed on center the bending stiffness deviation becomes too large compared to having the correct amount of torque around the master nodes. This is because the torque arm for the thin plate increases by 0.14 mm compared to having the master node off-center. Hence, the thick plate obtains a smaller torque arm and thereby less bending stiffness.

Angular displacement (1800 N)	Solid model	1-D model (off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	0.8894˚	0.8737	-1.8 %	1.0399˚	+16.9 %	1.224˚	+37.6 %
Thick plate	0.8712˚	0.8491	-2.5 %	0.8194˚	-5.9 %	0.701˚	-19.5 %

Table 7.2. Angular displacement table for 1800 N

After obtaining the correct bending stiffness to the plate, we used a spring constant of

— —— for both the on- and off-center 1-D rivet models. Table 7.3 shows the joints global tensile stiffness along with deviation estimates.

Displacement (1800 N)	Solid model	1-D model (off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	98.4 µm	98.7 µm	+0.3 %	99.3 µm	+0.9 %	58.9 µm	-40.1 %
Thick plate	0 µm	0 µm	0 %	0 µm	0 %	0 µm	0 %

Table 7.3. Global displacement in x-direction

It is easy to obtain the correct tensile stiffness when using springs due to easy manipulation of the springs k-factor in all DOF. The BEAM model suffers from incorrect tensile stiffness due to the beams too high stiffness.

Table 7.4 shows the plate stresses 10 mm from the holes center when the master node is placed both off- and on-center. The stresses in table 7.4 are based on a tensile force of 1800 N on the thin plate. The plate stresses and the tensile stiffness are the same whether the master node is placed on- or off-center. The only difference between the two methods is the bending stiffness deviation, which corresponds well to chapter 6.3.

Stress table (1800 N)		**Solid model**	**1-D model (off-center) (-0.11 mm)**	**Dev.**	**1-D model (on-center)**	**Dev.**	**BEAM model**	**Dev.**
	Thin plate	52 MPa	50 MPa	-3.8 %	50 MPa	-3.8 %	54 MPa	+3.8 %
	Thick plate	24 MPa	24 MPa	0 %	25 MPa	+4.2 %	23 MPa	-4.2 %
	Thin plate	39 MPa	36 MPa	-7.7 %	39 MPa	0 %	48 MPa	+23.1 %
	Thick plate	22 MPa	24 MPa	+9.1 %	23 MPa	+4.5 %	21 MPa	-4.5%

Table 7.4. Stresses in the plates at 1800 N.

Seen in figure 7.1 and 7.2, there are a little difference between the von Mises stresses in the solid model and our springs model. Figure 7.1 illustrates the stress areas of the thick Alfa plate. Seen in the right picture of figure 7.1 the solid rivet takes all the pressure from the plate. That is why the major stress area is behind the hole. Looking at the left picture in figure 7.1, the CERIGs contribute to both tension and pressure. That is why the stress distribution is both behind and in front of the rivet hole. The arrows indicate in which direction the load is applied in figure 7.1 and 7.2.

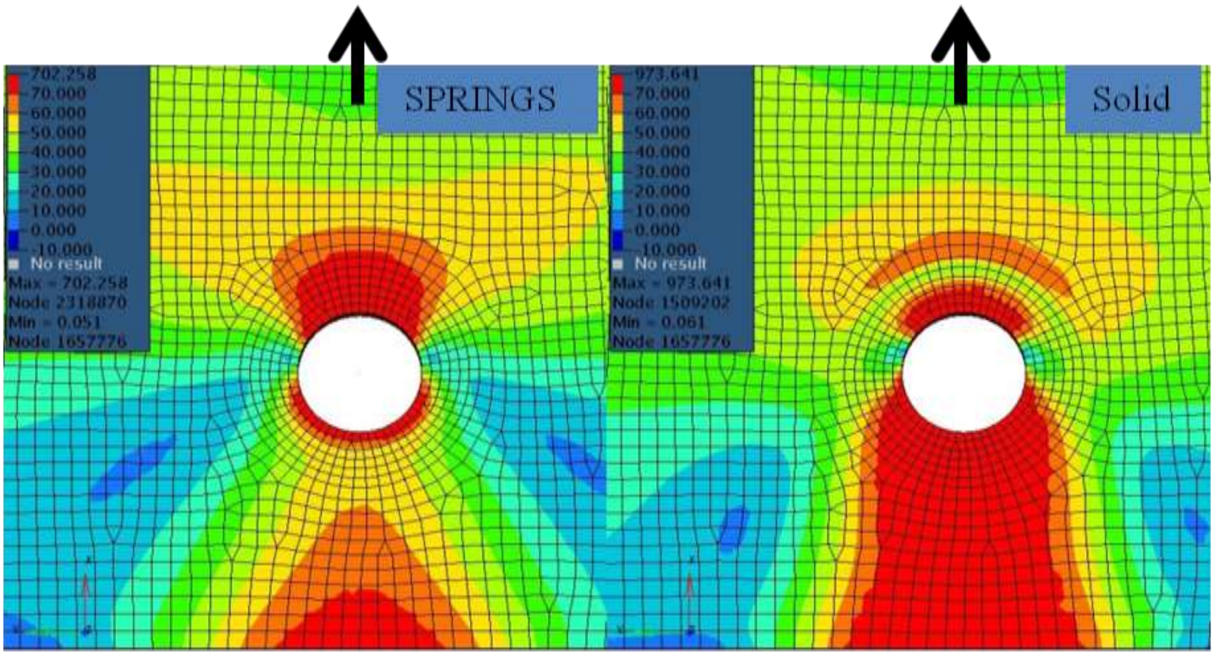


Figure .7.1. vM-stresses, Thick plate, Master node placement on-center.

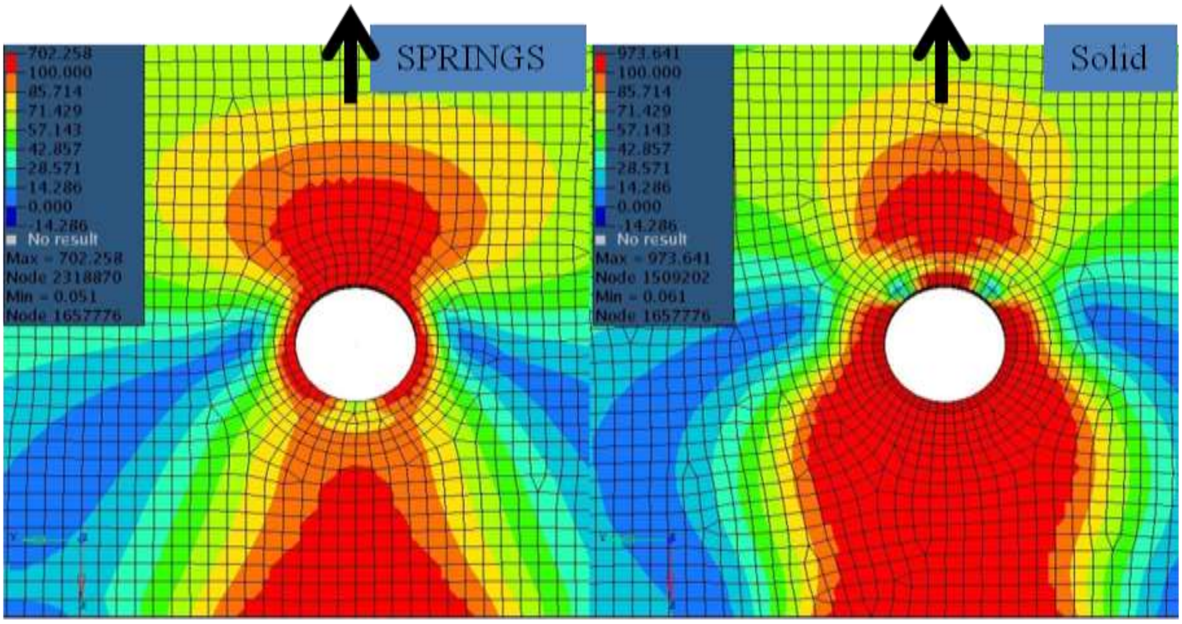


Figure .7.2. vM-stresses, Thin plate, Master node placement on-center.

The global tensile stiffness obtained when using springs with a k factor for all springs of 4.6*10^7 — —— can be seen in figure 7.3. The BEAM-models global tensile stiffness is also presented.

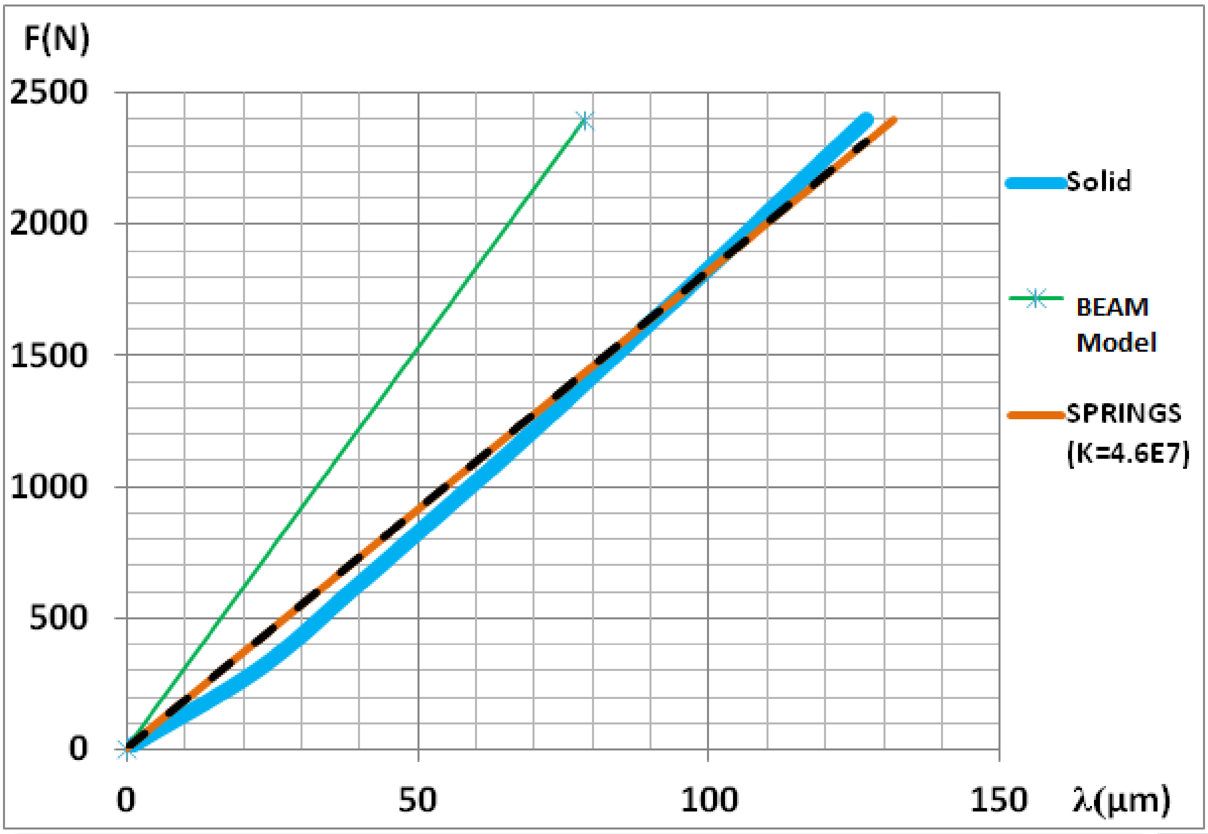


Figure 7.3. Global tensile stiffness.

7.2 Alfa w. Rivet 2

The dimensions for the 1-D rivet model Alfa with rivet 2 is presented in table 7.5.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Alfa	2	4.1	1.2	2.54

Table 7.5. 1-D rivet model Alfa w. Rivet 2.

The angular displacement in the hole for this riveted joint model is shown in table 7.6.

Angular displacement table (2850 N)	Solid model	1-D model (Off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	1.2924°	1.2240°	-5.3 %	1.3734°	+6.3 %	0.9362°	-27.6 %
Thick plate	1.2536°	1.2840°	+2.4 %	1.1765°	-6.1 %	0.6609°	-47.3 %

Table 7.6. Angular displacement table for 2850 N.

The correct tensile stiffness resulted in a k-factor of 5.2*10^7 — —— .

This gives the x-direction displacement shown in table 7.7.

Displacement table (2850 N)	Solid model	1-D model (off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	140.8 µm	140.3 µm	-0.4 %	140.7 µm	-0.1 %	82.24 µm	-41.6 %
Thick plate	0 µm	0 µm	0 %	0 µm	0 %	0 µm	0 %

Table 7.7. Displacement in x-direction for 2850 N

The tensile- and shear-mean stresses in the plates 10 mm from the holes center for both master node placements are presented in table 7.8.

Stress table (2850 N)		Solid model	1-D model (off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
	Thin plate	78 MPa	72 MPa	-7.7 %	75 MPa	-3.8 %	69 MPa	-11.5 %
	Thick plate	37 MPa	35 MPa	-5.4 %	38 MPa	+2.7 %	39 MPa	+5.4 %
	Thin plate	66 MPa	61 MPa	-7.6 %	63 MPa	-4.5 %	63 MPa	-4.5 %
	Thick plate	35 MPa	36 MPa	+2.8 %	35 MPa	0 %	33 MPa	-5.7 %

Table 7.8. Stresses in the plates for 2850 N

7.3 Alfa w. Rivet 3

The dimensions for the 1-D rivet model Alfa with rivet 3 is presented in table 7.9.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Alfa	3	4.5	1.2	2.54

Table 7.9. 1-D rivet model Alfa w. rivet 3

The angular displacement in the hole is presented in table 7.10.

Angular displacement table (4200 N)	Solid model	1-D model (off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	1.9088˚	1.7530˚	-8.1 %	1.9630˚	+2.8 %	2.120˚	+11.1 %
Thick plate	1.9911˚	1.7998˚	-9.6 %	1.7020˚	-14.5 %	1.530˚	-23.2 %

Table 7.10. Angular displacement table for 4200 N.

The global displacement in x-direction for the joint is presented in table 7.11. Iterative solving resulted in a stiffness factor of 6.4*10^7 — —— .

Displacement table (4200 N)	Solid model	1-D model (off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	189.8 µm	189.2 µm	+0.3 %	189.5 µm	+0.2 %	118.6 µm	-37.5 %
Thick plate	0 µm	0 µm	0 %	0 µm	0 %	0 µm	0 %

Table 7.11. Displacement in x-direction for 4200 N.

The tensile- and shear-mean stresses in the plates 10 mm from the holes center for both master node placements are presented in table 7.12.

Stress table (4200 N)		Solid model	1-D model (off-center) (-0.11 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
	Thin plate	124 MPa	116 MPa	-6.4 %	121 MPa	-2.4 %	117 MPa	-5.6 %
	Thick plate	58 MPa	57 MPa	+1.7 %	51 MPa	-12.1 %	58 MPa	0 %
	Thin plate	95 MPa	93 MPa	-2.1 %	96 MPa	+1.1 %	94 MPa	-1.1 %
	Thick plate	51 MPa	52 MPa	+1.9 %	52 MPa	+2.0 %	46 MPa	-9.8 %

Table 7.12. Stresses in the plates for 4200 N

The first idealization relationship has now been found. For the bending stiffness iteration, the best idealized models for the models with same plate thicknesses and different rivet sizes (all Alfa-models), there was equal nodal placement for correct bending stiffness (-0.11 mm).

7.4 Beta w. Rivet 1

The dimensions for the 1-D rivet model Beta with rivet 1 are presented in table 7.13.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Beta	1	3.3	1.2	2.0

Table 7.13. 1-D rivet model Beta w. Rivet 1

Placing the master node on-center was a very successful placement, see table 7.14. Therefore placing the master node off-center is not presented here. The BEAM model is presented in all three tables below. The results of the on-center placed idealized Beta-model is quite equivalent to the solid model. The BEAM model suffers here from a loss in bending stiffness both in the thin and thick plate due to decreased torque arm compared to both 1-D rivet models.

Angular displacement (1800 N)		Solid model	1-D model	Dev.	BEAM model	Dev.
Master node on-center	Thin plate	1.0716˚	1.0086˚	-5.9 %	0.887˚	-17.2 %
	Thick plate	1.0388˚	1.0017˚	-3.6 %	0.977˚	-5.9 %

Table 7.14. Angular displacement table for 1800 N.

The correct tensile stiffness resulted in a k-factor of — —. This gives the x-direction displacement shown in table 7.15. Here, the BEAM model has a too stiff beam element which leads to an incorrect tensile stiffness.

Displacement table (1800 N)		Solid model	1-D model	Dev.	BEAM model	Dev.
Master node on-center	Thin plate	92.98 µm	92.92 µm	-0.1 %	57.16 µm	-38.5 %
	Thick plate	0 µm	0 µm	0 %	0 µm	0 %

Table 7.15. Displacement in x-direction for 1800 N

The tensile- and shear-mean stresses in the plates 10 mm from the holes center are presented in table 7.16. The stresses deviation estimates for the 1-D model are satisfying. There is no need to investigate further by e.g. placing the master node off-center because of the low deviation values obtained when placing the master node on-center.

Stress table (1800 N)			Solid model	1-D model	Dev.	BEAM model	Dev.
Master node on-center		Thin plate	48 MPa	47 MPa	-2.1 %	63 MPa	+31.3 %
		Thick plate	31MPa	32 MPa	+3.2 %	29 MPa	-6.5 %
		Thin plate	41 MPa	41 MPa	0 %	47 MPa	+14.6 %
		Thick plate	25 MPa	26 MPa	+4.0 %	17 MPa	-32.0 %

Table 7.16. Stresses in the plates for 1800 N

7.5 Gamma w. Rivet 1

The dimensions for the 1-D rivet model Gamma with rivet 1 is presented in table 7.17.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Gamma	1	3.3	1.0	1.2

Table 7.17. 1-D rivet model Gamma w. Rivet 1

The idealized Gamma models angular displacement is presented in table 7.18. for when the master node is placed on-center. The results for the on-center placed idealized Gamma-model is quite equivalent to the solid model. The off-center iteration is thereby not taken care of here.

Angular displacement table (1800 N)		Solid model	1-D model	Dev.	BEAM model	Dev.
Master node on-center	Thin plate	1.7933˚	1.6967˚	-5.4 %	1,017˚	-43.3 %
	Thick plate	1.7777˚	1.7998˚	+1.2 %	2,166˚	+21.8 %

Table 7.18. Angular displacement table for 1800 N.

The k-factor for every spring was found by iterative solution and resulted in a stiffness factor of K= — —— . The tensile stiffness for master node placement on-center is presented in table 7.19.

Displacement table (1800 N)		Solid model	1-D model	Dev.	BEAM model	Dev.
Master node on-center	Thin plate	114.6 µm	114.3 µm	-0.3 %	73.23µm	-36.1 %
	Thick plate	0 µm	0 µm	0 %	0 µm	0 %

Table 7.19. Displacement in x-direction for 1800 N

The tensile- and shear-mean stresses in the plates 10 mm from the holes center are presented in table 7.20.

Stress table (1800 N)			Solid model	1-D model	Dev.	BEAM model	Dev.
Master node on-center		Thin plate	57 MPa	63 MPa	+10.5 %	60 MPa	+5.3 %
		Thick plate	48 MPa	53 MPa	+10.4 %	55 MPa	+14.6 %
		Thin plate	54 MPa	55 MPa	+1.9 %	46 MPa	-14.8 %
		Thick plate	37 MPa	40 MPa	-8.1 %	40 MPa	+8.1 %

Table 7.20. Stresses in the plates for 1800 N.

7.6 Delta w. Rivet 1

As mentioned before, the results of the solid Delta mode simulation is not reliable. Therefore this idealization model results is not reliable either. The deviation values for all Delta models are also suspiciously high. However, results are presented but further work is required.

The dimensions for the 1-D rivet model Delta is presented in table 7.21.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Under plate	Upper plate
Delta	1	3.3	2.54	2.54

Table 7.21. 1-D rivet model Delta w. Rivet 1

The obtained angular displacements for 1-D rivet model Delta are presented in table 7.22 for when the spring nodes are placed on- and off-center. The calculated angular displacement deviation was too large when the master node was placed on-center. When the node was placed off-center (+0.14 mm) from the upper edge of the upper plate the angular displacement became more satisfactory.

The angular displacement for the off-center model is both negative. That means that both plates needs more torque around the master nodes in order to bend more, i.e. larger angular displacement. It is not possible to increase the angular displacement on both plates with this solution method where both master nodes are placed on each other. If increasing the torque on one plate and thereby increasing the angular displacement, the other plate suffers a loss of torque and the angular displacement decreases even further. Hence, the best possible idealized model if placing the master node off-center by +0.14 mm from the shear plane and up.

Angular displacement table (1800N)	**Solid model**	**1-D model (off-center) (+0.14 mm)**	**Dev.**	**1-D model (on-center)**	**Dev.**	**BEAM model**	**Dev.**
Upper plate	0.3598°	0.3232°	-10.2 %	0.5580°	-55.0 %	2.0300°	+464.2 %
Under plate	0.3539°	0.31290°	-11.6 %	0.3170°	-10.4 %	0.4529°	+27.9 %

Table 7.22. Angular displacement table for 1800 N.

The k factor were found here to be k=2.9*10^7 — —— for 1-D rivet model Delta. The global tensile stiffness for the joint is presented in table 7.23. The tensile stiffness deviation the BEAM model has comes from that the beam element is too stiff. That is why it does not deform as much as the solid model, as seen in table 7.23 the deviation for the BEAM-model is negative.

Displacement table (1800 N)	**Solid model**	**1-D model (off-center) (+0.14 mm)**	**Dev.**	**1-D model (on-center)**	**Dev.**	**BEAM model**	**Dev.**
Upper plate	0 µm	0 µm	0 %	0 µm	0 %	0 µm	0 %
Under plate	95.36 µm	95.06 µm	0.3 %	95.53 µm	0.2 %	32.83 µm	-65.6 %

Table 7.23. Displacement in x-direction for 1800 N

The tensile- and shear-mean stresses in the plates 10 mm from the holes center for both master node placements are presented in table 7.24.

Stress table (1800 N)		**Solid model**	**1-D model (off-center) (+0.14 mm)**	**Dev.**	**1-D model (on-center)**	**Dev.**	**BEAM model**	**Dev.**
	Thin plate	24 MPa	27 MPa	+12.5 %	24 MPa	0 %	28 MPa	+16.7 %
	Thick plate	25 MPa	25 MPa	0 %	26 MPa	+4 %	26 MPa	+4 %
	Thin plate	19 MPa	21 MPa	+10.5 %	20 MPa	+5.3 %	19 MPa	0 %
	Thick plate	16 MPa	20 MPa	+25 %	19 MPa	+18.8 %	19 MPa	+18.8 %

Table 7.24. Stresses in the plates for 1800 N.

7.7 Epsilon w. Rivet 1

The dimensions for the 1-D rivet model Epsilon is presented in table 7.25.

Riveted joint model	Rivet type [Nr.]	Hole dimension [mm]	Plate thickness [mm]	
			Thin plate	Thick plate
Epsilon	1	3.3	2.0	2.5

Table 7.25. 1-D rivet model Epsilon w. Rivet 1

The angular displacements for when the spring nodes are placed on- and off-center is presented in table 7.26.

Angular displacement table (1800 N)	Solid model	1-D model (off-center) (+0.35 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	0.5002˚	0.4672˚	-6.6 %	0,3840˚	-23.2 %	0.2970˚	-40.6 %
Thick plate	0.4890˚	0.4554˚	-6.9 %	0,5101˚	+4.3 %	0.5530˚	+13.1 %

Table 7.26. Angular displacement table for 1800 N.

The global x-direction displacement for 1-D rivet model Epsilon is presented in table 7.27 for when using a stiffness factor of — — .

Displacement table (1800 N)	Solid model	1-D model (off-center) (+0.35 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
Thin plate	72.91 µm	72.59 µm	-0.4 %	72.83 µm	-0.1 %	38.56 µm	-47.1 %
Thick plate	0 µm	0 µm	0 %	0 µm	0 %	0 µm	0 %

Table 7.27. Displacement in x-direction for 1800 N

The stresses in both plates 10 mm from the hole center is presented in table 7.28.

Stress table (1800 N)		Solid model	1-D model (off-center) (+0.35 mm)	Dev.	1-D model (on-center)	Dev.	BEAM model	Dev.
	Thin plate	31 MPa	32 MPa	+3.2 %	32 MPa	+3.2 %	30 MPa	-3.2 %
	Thick plate	23 MPa	25 MPa	+8.7 %	24 MPa	+4.3 %	23 MPa	0 %
	Thin plate	25 MPa	25 MPa	0 %	26 MPa	+4 %	27 MPa	+8 %
	Thick plate	18 MPa	19 MPa	+5.6 %	19 MPa	+5.6 %	19 MPa	+5.6 %

Table 7.28. Stresses in the plates for 1800 N.

8. Conclusions

The conclusions to this degree project are that we have come up with a working idealization method. It is relatively simple to model and is in good agreement with the Solid reference model. We have also decreased the simulation time in the FE analysis from around 30 hours with the solid model to 10 minutes with the idealized model.

If something about this idealization project could be better, it is the hole deformations and the hole stresses. If the hole deformations becomes more equivalent to the solid model, the stresses around the hole becomes better.

The simplest and absolutely most ideal modeling method is to place both master nodes on-center when using springs. Placing both master nodes on-center or off-center results in a maximal angular displacement deviation and a stiffness factor k for all springs DOF according to table 8.1. The table also shows all the obtained solid models stiffness factors.

Riveted joint model	Dimensions (Thin plate / Thick plate) [mm]	Spring models stiffness factor [N/m] and [Nm/rad] (10^7)	Solid models stiffness factor [N/m] (10^7)	On-center Max. Angular displacement deviation [%]	Off – center Max. Angular displacement deviation [%]
ALFA 1/2/3	1.2/2.54	4.6/5.2/6.4	1.8/2.0/2.2	16.9/6.27/14.5	2.5/5.3/9.6
BETA	1.2/2.0	5.5	1.9	5.9	-
GAMMA	1.0/1.2	4.9	1.6	5.4	-
DELTA	2.54/2.54	2.9	1.9	55.0	11.6
EPSILON	2.0/2.5	5.4	2.5	23.2	6.9

Table 8.1. Stiffness factor summary for all riveted joint models w. max. angular displacement.

Table 8.1 only shows the maximal angular displacement deviation for on-center and off-center 1-D rivet models. They are presented in order to show the difference between placing the master nodes off- and on-center. The two models Beta and Gamma maximal angular displacement for off-center is not presented because of the low deviations obtained with on-center master node placement. The maximal displacement deviation in x-direction is indifferent between the on- and off-center models and has <1% global displacement deviation compared to the solid model, hence they are not presented here.

We noticed that in order to get the correct bending stiffness in the three Alfa models the master node placements had to be in the exact same place. The two larger rivets; rivet 2 and rivet 3 only primarily increased the global tensile stiffness in the riveted joint.

As mentioned before, the Delta model is not reliable. When comparing the Delta model and the Epsilon model the stiffness deviation should not be that high as it is, due to the similar plate sizes.

9. Discussion

As recently presented, some of the finished idealized models are created by the on-center-method, and some are not. Some of the on-center models have a so high bending stiffness deviation value, that it is not even close to the solid Solid reference model model. The question to ask then is what difference the specific bending stiffness deviation of each idealized rivet, affects the stiffness of the whole Solid reference model-model. Even though the springs are placed on- or off-center, the specific tensile stiffness for each model is very matching to the solid Reference model. Therefore it would be interesting to see how different results the on-center models give compared to the off-center models, when using them in the Solid reference-model. A typical riveted flange design model is much stiffer than a single lap riveted joint, so maybe the bigger bending stiffness deviation doesn't matter as much as in the single-lap simulation. It reduces the modelling time quite a bit by choosing the on-center modelling technique instead of keep controlling the right element placement, so of course that is preferable. Unfortunately there was no time to investigate the result difference between the both modeling techniques applied in a realistic riveted flange model.

An important part of this project has been to follow all the analysis to make sure we have successfully reached convergence. Another important part of the analysis process, was to check the models for defects and unnatural movements after load. With the Delta model we noticed some unnatural movement like penetration of contact surfaces, and therefore we could not use those result. You can tell by only looking at the stiffness factor, that something is wrong with the delta models.

10. Further work

There are essentially three things that we think require further work after this thesis.

- Further work after this degree project could be to find a general solution to the problem at hand. We have come up with an idealization method that can be applied to every riveted joint, but it requires that every stiffness factor (tensile- and bending stiffness) is iterated. A general solution could solve this if one is found, although some relationships can already be established.
- It would have been interesting to see what results we would get if we used our idealized models with on-center master node placements in realistic flange model instead of the off-center idealized models.
- Investigate the stiffness factor for the solid model Delta further.

References

[1]Mägi, M.M. and Gerbert, G.G. (2003) Maskinelement del A. Gothenburg, Chalmers reproservice

[2]Persson, G.P. (-) FEM-Modellering med Pro/Mechanica Wildfire V4.0, -

Appendix A – Ansys 11.0 help manual

BEAM188 element

The BEAM188 element is suitable for analyzing slender to moderately stubby/thick beam structures. This element is based on Timoshenko beam theory. Shear deformation effects are included.

BEAM188 is a linear (2-node) beam element in 3-D with six degrees of freedom at each node. The degrees of freedom at each node include translations in x,y, and z directions, and rotations about the x,y, and z directions. Warping of cross sections is assumed to be unrestrained.

The beam elements are well-suited for linear, large rotation, and/or large strain nonlinear applications.

BEAM188 includes stress stiffness terms, by default, in any analysis. The stress stiffness terms provided enable the elements to analyze flexural, lateral and torsional stability problems (using eigenvalue buckling or collapse studies with arc length methods).

BEAM188 can be used with any cross section defined using different commands. Elasticity and isotropic hardening plasticity models are supported (irrespective of cross section subtype).

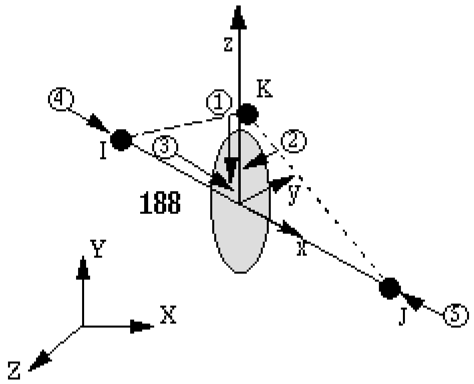


Figure A.1 BEAM188 3-D linear finite strain beam

The geometry, node locations, and the coordinate system for this element are shown in Figure A1. BEAM188 is defined by nodes I and J in the global coordinate system. Node K is always required to define the orientation of the element.
The beam elements are one-dimensional line elements in space. The cross section details are provided separately.

The beam elements are based on Timoshenko beam theory, which is a first order shear deformation theory: transverse shear strain is constant through the cross section, i.e., cross sections remain plane and undistorted after deformation. BEAM188 and BEAM189 elements can be used for slender or stout beams. Due to the limitations of first order shear deformation theory, only moderately "thick" beams may be analyzed.

COMBIN14 element

COMBIN14 has longitudinal or torsional capability in 1-D, 2-D, or 3-D applications. The longitudinal spring-damper option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered. The torsional spring-damper option is a purely rotational element with three degrees of freedom at each node: rotations about the nodal x, y, and z axes. No bending or axial loads are considered.

The spring-damper element has no mass. Masses can be added by using the appropriate mass element. The spring or the damping capability may be removed from the element.

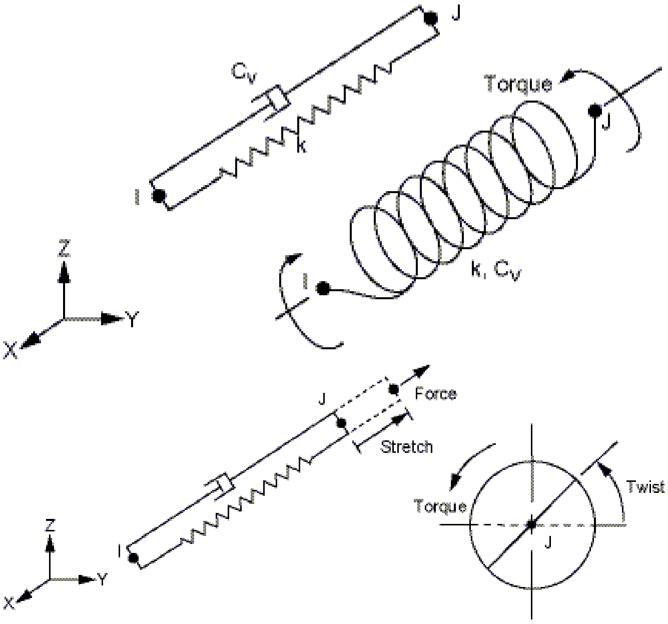


Figure A.2 COMBIN14 geometry

2-D elements must lie in a constant plane. The geometry, node locations, and the coordinate system for this element are shown in Figure A.2 "COMBIN14 Geometry". The element is defined by two nodes, a spring constant (k) and damping coefficients $(c_v)_1$ and $(c_v)_2$. The damping capability is not used for static or undamped modal analyses. The longitudinal spring constant should have units of Force/Length, the damping coefficient units are Force*Time/Length. The torsional spring constant and damping coefficient have units of Force*Length/Radian and Force*Length*Time/Radian, respectively. For a 2-D axisymmetric analysis, these values should be on a full 360° basis.

The damping portion of the element contributes only damping coefficients to the structural damping matrix. The damping force (F) or torque (T) is computed as:

$F_x = - c_v du_x/dt$ or $T_\theta = - c_v d\,\theta/dt$

where c_v is the damping coefficient given by $c_v = (c_v)_1 + (c_v)_2 v$. v is the velocity calculated in the previous substep. The second damping coefficient $(c_v)_2$ is available to produce a nonlinear damping effect characteristic of some fluid environments.

A summary of the element input is given in "COMBIN14 Input Summary". A general description of element input is given in Element Input.

COMBIN14 Input Summary

Nodes:

I, J

Degrees of Freedom(with different keyoptions):

UX, UY, UZ

ROTX, ROTY, ROTZ

UX, UY

Real Constants:

K - Spring constant

CV1 - Damping coefficient

CV2 - Damping coefficient (KEYOPT(1) must be set to 1)

Material Properties:

DAMP
Surface Loads:
None
Body Loads:
None

Special Features:

Nonlinear

Stress stiffening

Large deflection

Birth and death

RBE3 element

Distributes the force/torque applied at the master node to a set of slave nodes, taking into account the geometry of the slave nodes as well as weighting factors.

Master:

Node at which the force/torque to be distributed will be applied. This node must be associated with an element for the master node to be included in the DOF solution.

DOF:

Refers to the master node degrees of freedom to be used in constraint equations, UX, UY, UZ, ROTX, ROTY, ROTZ, UXYZ, RXYZ.

Slaves:

The name of an array parameter that contains a list of slave nodes. Must specify the starting index number. The slave nodes may not be colinear, that is, not be all located on the same straight line (see Notes below).

Wtfact:

The name of an array parameter that contains a list of weighting factors corresponding to each slave node above. Must have the starting index number. If not specified, the weighting factor for each slave node defaults to 1.

Notes:

The force is distributed to the slave nodes proportional to the weighting factors. The torque is distributed as forces to the slaves; these forces are proportional to the distance from the center of gravity of the slave nodes times the weighting factors. Only the translational degrees of freedom of the slave nodes are used for constructing the constraint equations. Constraint equations are converted to distributed forces/torques on the slave nodes during solution.

RBE3 creates constraint equations such that the motion of the master is the average of the slaves. For the rotations, a least-squares approach is used to define the "average rotation" at the master from the translations of the slaves. If the slave nodes are colinear, then one of the master rotations that is parallel to the colinear direction can not be determined in terms of the translations of the slave nodes. Therefore, the associated torque component on the master node in that direction can not be transmitted.

Applying this command to a large number of slave nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of slave nodes.

CERIG

Defines a rigid region.

MASTER:

Retained (or master) node for this rigid region. Graphical picking of the master and slave nodes, first node picked will be the master node, and subsequent nodes picked will be slave nodes, and subsequent fields are ignored.

SLAVE:

Removed (or slave) node for this rigid region. If ALL, slave nodes are all selected nodes.

L-DOF:

Degrees of freedom associated with equations:

- ALL — All applicable degrees of freedom. If 3-D, generate 6 equations based on UX, UY, UZ, ROTX, ROTY, ROTZ; if 2-D, generate 3 equations based on UX, UY, ROTZ.
- UXYZ — Translational degrees of freedom. If 3-D, generate 3 equations based on the slave nodes' UX, UY, and UZ DOFs and the master node's UX, UY, UZ, ROTX, ROTY, and ROTZ DOFs; if 2-D, generate 2 equations based on the slave nodes UX and UY DOFs and the master nodes UX, UY, and ROTZ DOFs. No equations are generated for the rotational coupling.
- RXYZ — Rotational degrees of freedom. If 3-D, generate 3 equations based on ROTX, ROTY, ROTZ; if 2-D, generate 1 equation based on ROTZ. No equations are generated for the translational coupling.
- UX — Slave translational UX degree of freedom only.
- UY — Slave translational UY degree of freedom only.
- UZ — Slave translational UZ degree of freedom only.
- ROTX — Slave rotational ROTX degree of freedom only.
- ROTY — Slave rotational ROTY degree of freedom only.
- ROTZ — Slave rotational ROTZ degree of freedom only.

L-DOF2, L-DOF3, L-DOF4, L-DOF5:

Additional degrees of freedom. Used only if more than one degree of freedom required and L-DOF is not ALL, UXYZ, or RXYZ.

Notes:

Defines a rigid region (link, area or volume) by automatically generating constraint equations to relate nodes in the region. Nodes in the rigid region must be assigned a geometric location. Also, nodes must be connected to elements having the required degree of freedom set (see L-DOF above). Generated constraint equations are based on small deflection theory. Generated constraint equations are numbered beginning from the highest previously defined equation number.

This command will generate the constraint equations needed for defining rigid lines in 2-D or 3-D space. Multiple rigid lines relative to a common point are used to define a rigid area or a rigid volume. In 2-D space, three equations are generated for each pair of constrained nodes. These equations define the three rigid body motions in global Cartesian space, i.e., two in-plane translations and one in-plane rotation. These equations assume the X-Y plane to be the active plane with UX, UY, and ROTZ degrees of freedom available at each node. Other types of equations can be generated with the appropriate Ldof labels.

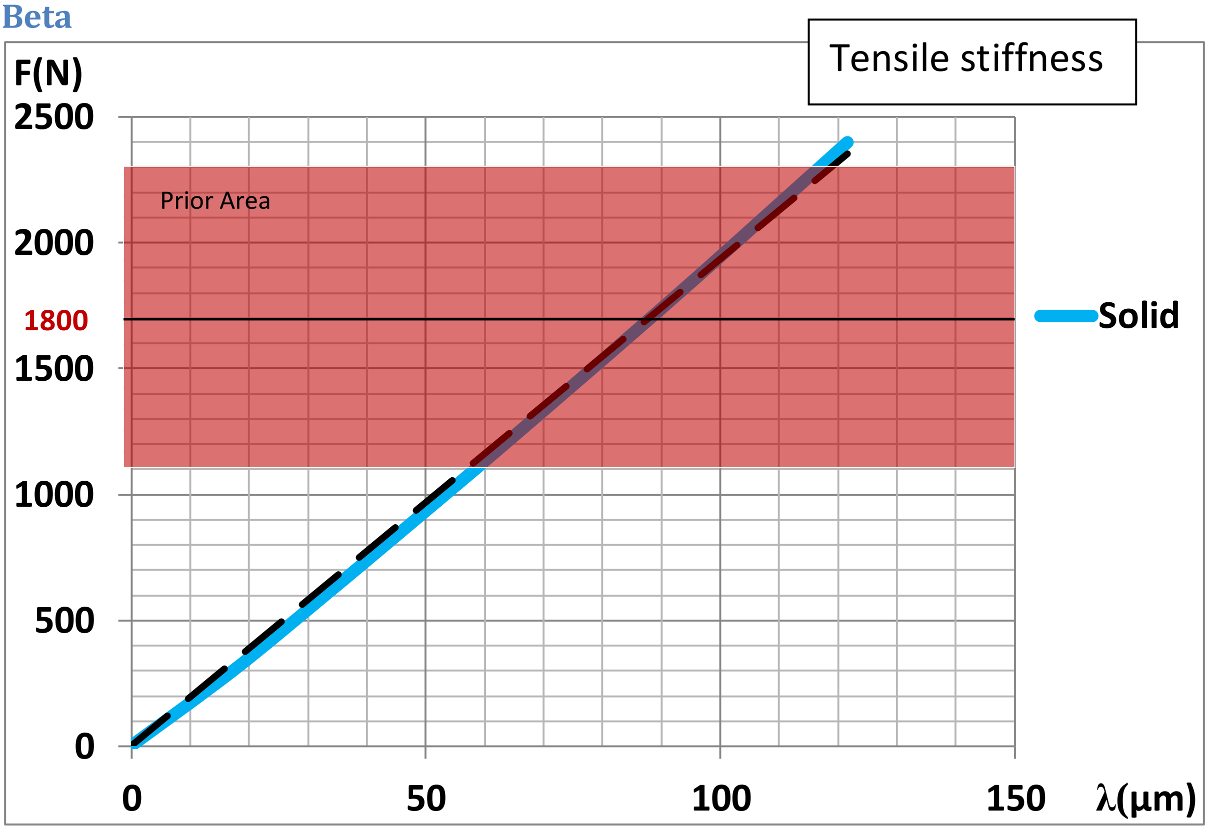
Six equations are generated for each pair of constrained nodes in 3-D space. These equations define the six rigid body motions in global Cartesian space. These equations assume that UX, UY, UZ, ROTX, ROTY, and ROTZ degrees of freedom are available at each node.

The UXYZ label allows generating a partial set of rigid region equations. This option is useful for transmitting the bending torque between elements having different degrees of freedom at a node. With this option only two of the three equations are generated for each pair of constrained nodes in 2-D space. In 3-D space, only three of the six equations are generated. In each case the rotational coupling equations are not generated. Similarly, the RXYZ label allows generating a partial set of equations with the translational coupling equations omitted.

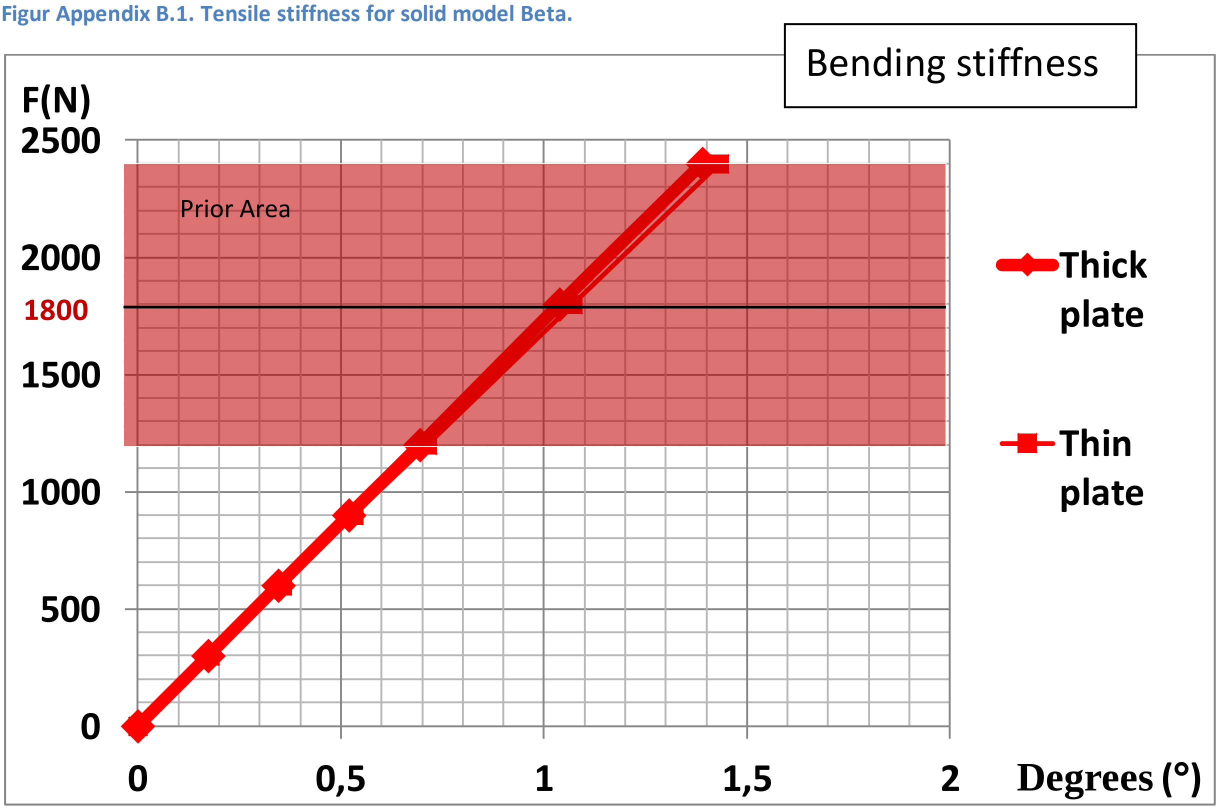
Applying this command to a large number of slave nodes may result in constraint equations with a large number of coefficients.

Appendix B – Stiffness figures

Beta

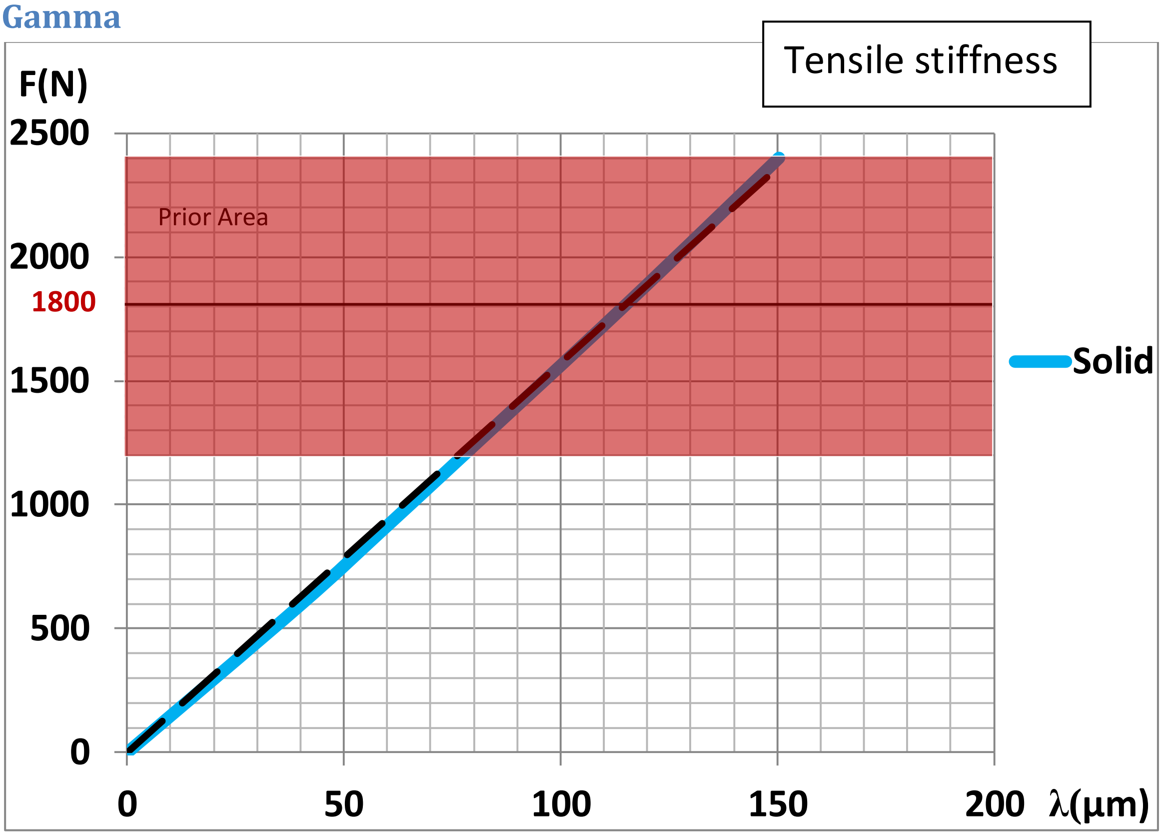


Figur Appendix B.1. Tensile stiffness for solid model Beta.

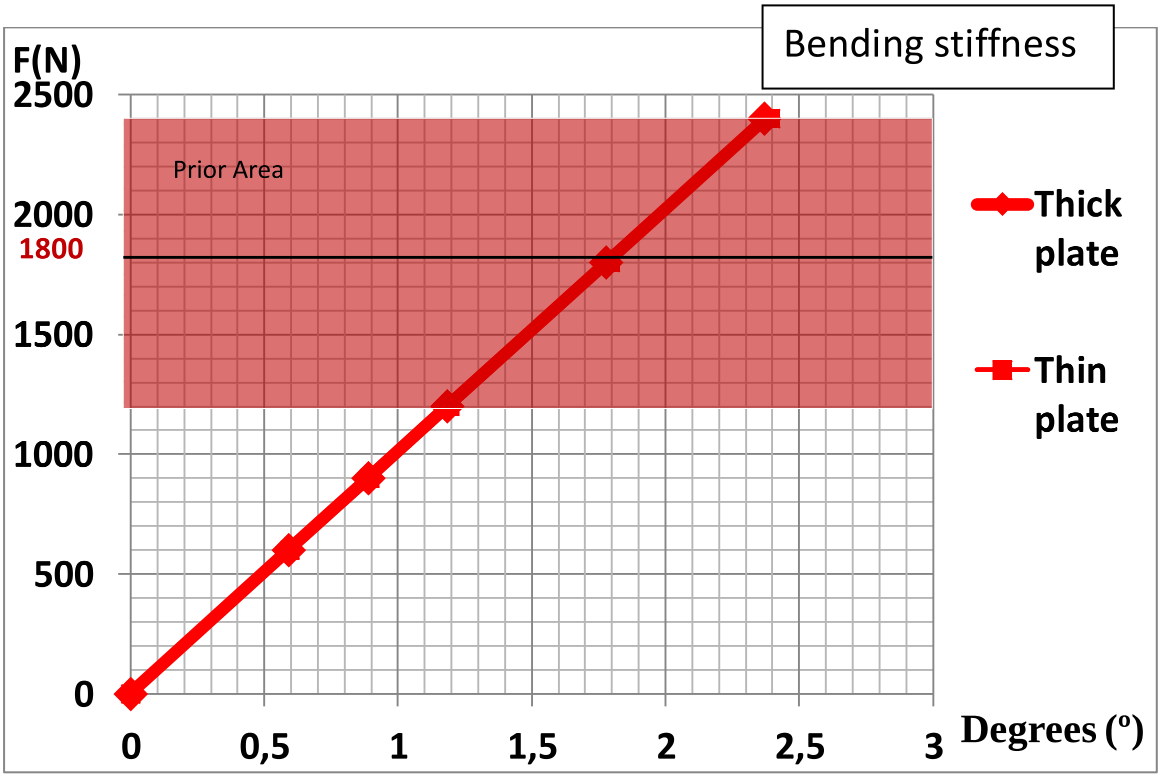


Figur Appendix B.2. Bending stiffness for solid model Beta

Gamma



Figur Appendix B.3. Tensile stiffness for solid model Gamma



Figur Appendix B.4. Bending stiffness for solid model Gamma