Offset modeling of shell elements

A study in shell element modeling using Nastran

Master thesis by:
David Klarholm

Department of management and engineering
Division of Solid Mechanics
Linköping institute of technology
Linköping University

ISRN:LIU-IEI-TEK-A--16/02456—SE

Linköping, 2016
Title:
Offset modeling of shell elements - A study in shell element modeling using Nastran

Author:
David Klarholm

Supervisors:
Bo Torstenfelt, Linköping University
Tomas Lundgren, Saab AB

Examiner:
Daniel Leidermark, Linköping University

Publication type:
Master thesis in mechanical engineering

ISRN-number: LIU-IEI-TEK-A--16/02456—SE

Linköping Institute of Technology
The Department of Management and Engineering
The Division of Solid Mechanics

Linköping, June 2016

Keywords: Offset shell elements, finite element, Nastran, shell element modeling, Rigid body connection, Rigid links, Kirchhoff shells, Multipoint constraint
Abstract

At Saab Aerostructures they are manufacturing a lot of parts for Airbus and Boeing. When these components are investigated using finite element analysis four-node Kirchhoff shell elements and a very fine mesh is often used. In order to make the pre-processing easier Saab would like to offset the shell midsurface from the nodal plane (the modeling surface) rather than to extract midsurfaces for the entire component. This would also make it easier to model a component which needs a thickness change later on, this since the original modeling surface could be used but with an offset of the elements in order to represent the new geometry.

When offset is used in Nastran multi point constraints are created between the nodes and the shell midsurface points. All loads, which are applied in the nodal plane, are then transformed to the midsurface where the stiffness matrices, displacements and stresses are calculated. In order to be able to use this method more knowledge about its effects are needed, which is the reason for this thesis work.

The offset is studied for two simpler cases, thickness variation and a 90° corner, as well as for a more complicated component called a C-bar. This is a hinge connecting the flaps to the wings of an airplane. The simpler cases are modeled using both midsurface and offset models subject to either a transverse load, an in-plane load or a bending moment. These are compared to a solid model in order to determine which is the most accurate. When midsurface modeling is used for the thickness variation the surfaces are connected using rigid links.

The conclusion made from these simulations is that using offset may give different results if the load is an in-plane load. This kind of load leads to the creation of a bending moment, which is linearly dependent on the amount of offset. The severity of this depends on the overall geometry and how this load is applied.
**Sammanfattning**


När offset används i Nastran bildas "multi point constraints" mellan noder och punkter på skalets mittyta. Alla laster, vilka appliceras i nodplanet, transformeras till mittytan där styvhetsmatriser, förskjutningar och spännings beräknas. För att kunna använda denna metod behövs mer kunskap om dess effekter, vilket är anledningen till detta arbete.


Slutsatsen från dessa simuleringar är att offset kan ge annorlunda resultat om lasten läggs i planet. Denna typ av last ger upphov till ett böjande moment som beror linjärt på hur stor offset som används. Hur stor effekt detta får beror på modellens geometri samt hur lasten appliceras.
Preface

This master thesis has been carried out at Saab AB and The Division of Solid Mechanics at Linköping University during the spring of 2016. I would like to thank my supervisor and associated professor Bo Torstenfelt for the guidance. I would also like to thank my supervisor at Saab AB, Tomas Lundgren, for all help during this work and also Bérénice Bonamy for helping me understand the software.

This thesis ends my studies for the degree of Master of Science in Mechanical Engineering.

David Klarholm
Linköping, Sweden
June 2016
## Contents

1 Introduction .................................................................................. 1  
1.1 Background ............................................................................. 2  
1.2 Problem formulation and goals ................................................. 2  
1.3 Delimitations ........................................................................... 2  
1.4 Other considerations ................................................................. 2  
2 Theory ......................................................................................... 3  
2.1 Shells ...................................................................................... 3  
  2.1.1 Kirchhoff-plates ................................................................. 3  
  2.1.2 Shell theory ....................................................................... 5  
2.2 Shell elements ........................................................................... 6  
2.3 Element offset in Nastran .......................................................... 7  
  2.3.1 Mathematics of ZOFFS ....................................................... 9  
  2.3.2 Rigid links in Nastran (RBE2 elements) .............................. 12  
3 Method ......................................................................................... 15  
3.1 Thickness variation model ......................................................... 16  
3.2 90° corner model ..................................................................... 17  
3.3 C-bar model ............................................................................. 20  
4 Results ........................................................................................ 21  
4.1 Thickness variation .................................................................. 21  
  4.1.1 Overall behavior ............................................................... 22  
  4.1.2 Stresses in the thick part ................................................... 27  
  4.1.3 Stresses in the thin part ..................................................... 29  
4.2 90° corner ............................................................................... 31  
  4.2.1 Same thickness ratio ......................................................... 31  
  4.2.2 Different thickness ratio .................................................... 37  
4.3 C-bar ....................................................................................... 42  
5 Discussion .................................................................................... 47  
5.1 Thickness variation case ............................................................ 47  
5.2 90° corner cases ...................................................................... 50  
  5.2.1 90° corner subject to a transverse force or bending moment 50  
  5.2.2 90° corner subject to an in-plane load ................................. 51  
5.3 C-bar models .......................................................................... 52  
6 Conclusions .................................................................................. 53  
7 Further work ................................................................................ 56  
Appendix A: Overall stress results for the thickness variation ....... 57  
Appendix B: Results from the 90° corner simulations .................. 61  
  B1: Corner with same thickness of the plates .............................. 61  
  B2: Corner with different thickness of the plates ......................... 64  
Appendix C: Results from the C-bar simulations ......................... 70
Nomenclature

\[\cdot\] Matrix representation
\[\mathbf{0}\] A zero matrix
\[\mathbf{I}\] The unity matrix
\[\mathbf{K}\] Stiffness matrix
\[\mathbf{R}\] Constraint coefficient matrix
\[\mathbf{T}\] Coordinate transformation matrix
\{\cdot\} Vector representation
\{\mathbf{P}\} Load vector
\{\mathbf{q}\} Constraint force vector
\{\mathbf{u}\} Displacement vector
\xi, \eta, \zeta Element local coordinate system for isoparametric elements
\[l, m, n\] Direction cosines
\[u, v, w\] Displacements in the \(x\)-, \(y\)- and \(z\)-direction respectively
BC Boundary condition
d.o.f. Degree(s) of freedom
FE Finite element
FEA Finite element analysis
g-set Set containing all nodes
m-set Set containing all dependent degrees of freedom
MPC Multi point constraint
n-set Set containing all independent degrees of freedom
RBE2 Rigid link used in Nastran
RBE3 Interpolation elements used to distribute loads in Nastran
vM von Mises
ZOFFS Offset of the element midsurface from the nodal plane in Nastran
1 Introduction

In many finite element (FE) applications it is common to model thin structures using shell elements, which are defined in the midsurface of the component. If the different parts of the shell has different thickness and a common bottom surface, traditional midsurface modeling may not be used, as illustrated in figure 1. How to handle this is not clear and is, among other things, to be investigated in this thesis.

![Figure 1: Illustration of why traditional midsurface modeling (top picture) does not work if the bottom surface should be the same for the adjacent plates. The bottom picture illustrates how the midsurfaces has to be located in order to represent this case. The red lines represents the midsurfaces and the circles the nodes.](image)

Furthermore it might be the case that the component to be investigated has a change in thickness in one direction, i.e. one surface of the component is fixed during the thickness change. Instead of extracting a new midsurface to account for this change one might want to use the original midsurface and instead offset the elements from this. An example of a component that is only allowed to change thickness in one direction can be seen in figure 2 where the outside measurements are fixed in order for it to fit with other components.

![Figure 2: An example of a geometry one might want to evaluate using shell elements. It has fixed outside measurements and thus, this thickness is only allowed to change in one direction. The geometry is a part of a C-bar.](image)

If the component that is to be modeled is complex it might also be hard work to extract its midsurface, an example of a geometry that might be of interest to model is a C-bar, part of which is seen in figure 2. This is basically a hinge connecting the flaps to the wings of an airplane. If elements are modeled in one of the interface surfaces of the solid component, instead of its midsurface, and elements are offset from this, it might help make the pre-processing of the finite element analysis (FEA) faster. The use and behavior of the offset of shell elements is the main focus of this thesis.
1.1 Background

At Saab AB they are manufacturing parts for both Boeing and Airbus. Airbus would like most components to be modeled using four-node shell elements with a very fine mesh. This model is then solved by use of the FE program Nastran [1].

When shell elements are used Saab would like to offset the elements from an interface surface or midsurface to make the meshing procedure faster, by not having to extract a midsurface for every thickness of the part, or in order to handle changes in the geometry more easily. The use of offset modeling would thus be a way to speed up the time consuming pre-processing in the FEA.

As of now there is little documentation about the effects of using offset shells in Nastran and in order to be able to use this technique more information is needed. This is the reason for this thesis work.

1.2 Problem formulation and goals

The goal is to investigate the effects of using offset shell elements for FEA. Focus is on determining the differences between this and traditional midsurface modeling. Both ways of modeling will be compared to a solid model, which is used as a reference in order to determine which of the shell models that is the more accurate.

The goal of this thesis is to produce guidelines that can be used in order to determine if offset modeling may or may not be used in different situations. Furthermore a real component called a C-bar (figure 2) is to be investigated using a combination of offset modeling and midsurface modeling. This is to be compared to a solid model in order to determine if the shell model will give satisfying results.

1.3 Delimitations

The delimitations made are the following:

- Only flat four-node Kirchhoff shell elements are to be studied.
- Only linear elastic static analyses will be carried out.
- Only isotropic materials are to be used.

1.4 Other considerations

No ethical or gender related questions arise in this work. Nor is it directly linked to issues concerning the environment or sustainable development.
2 Theory

In this section a brief review of the theory of shells and shell elements is given together with the theory of how Nastran deals with the offset of these elements.

2.1 Shells

In order to understand how shell elements work one first has to understand plate and shell theory. As shell theory is based on plate theory this is a good place to start. There are two major theories that describe the behavior of plates, and also shells. One is the Mindlin-theory (thick plates and shells) and the other is the Kirchhoff-theory (thin plates and shells). In this work only Kirchhoff elements are used and the theory will be limited to the description of plates and shells of this type.

2.1.1 Kirchhoff-plates

When dealing with plates it is common to place the coordinate system in the midsurface of the plate, with the $z$-direction in the normal (through thickness) direction of the plate. The midsurface is, for a homogeneous plate, a neutral plane according to plate theory (where there are no in-plane loads considered). Meaning that the strains on this plane is zero \([2]\).

Furthermore, plate theory is based on the assumption that the thickness of the plate is much smaller than its other overall dimensions. Because of this assumption one often also assumes that the stress developed in the through thickness direction is negligible. As a final assumption the deflection in the thickness direction, $w$, is said to be independent of the position in the through thickness direction, i.e. $w = w(x, y)$ \([3]\).

In Kirchhoff plate theory an idealized behavior is adapted by saying that an initially straight line which is normal to the midsurface will remain straight and normal to the midsurface when deformed by a lateral load, see figure 3.

![Figure 3: Illustration of how a point on a line normal to the midsurface moves upon lateral loading of a plate according to Kirchhoff plate theory.](image)

Using figure 3 one will arrive in the following equations for the deflections in the $x$- and $y$-direction ($u$ and $v$), if assuming small deflections and angles \([2]\). Note that the deflection in the $yz$-plane can be thought of exactly as the one in the $xz$-plane.
\[ \begin{align*}
\begin{cases}
u = -z \phi_x = -z \frac{\partial w}{\partial x} \\
v = -z \phi_y = -z \frac{\partial w}{\partial y}
\end{cases}
\end{align*} \]

(1)

Where \( u \) and \( v \) are displacements in the \( x \)- and \( y \)-direction respectively.

Using basic solid mechanics the strains can be derived from the expressions for the displacements, see equation (2). The strain in the \( z \)-direction has been omitted since it is not needed.

\[ \begin{align*}
\varepsilon_x &= \frac{\partial u}{\partial x} = -z \frac{\partial^2 w}{\partial x^2} \\
\varepsilon_y &= \frac{\partial v}{\partial y} = -z \frac{\partial^2 w}{\partial y^2} \\
\gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = -2z \frac{\partial^2 w}{\partial x \partial y} \\
\gamma_{xz} &= \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} = 0 \\
\gamma_{yz} &= \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} = 0
\end{align*} \]

(2)

From equation (2) one can derive expressions for the stresses. This is not done here but how they vary throughout the thickness of the plate is displayed in figure 4a [2].

**Figure 4:** (a) The stresses in a Kirchhoff plate due to an arbitrary lateral load \( q \). (b) The resulting moments per unit length in a Kirchhoff plate due to an arbitrary lateral load \( q \).

The shear stress \( \tau_{xy} \) gives rise to a twisting moment \( M_{xy} \) and the normal stresses \( \sigma_x \) and \( \sigma_y \) will give rise to bending moments \( M_x \) and \( M_y \). These are calculated according to equation (3) and are represented in figure 4b. Note that all the resultant moments are per unit length [2].
\[ M_x = \int_{-t/2}^{t/2} \sigma_x z \, dz \]
\[ M_y = \int_{-t/2}^{t/2} \sigma_y z \, dz \]
\[ M_{xy} = \int_{-t/2}^{t/2} \tau_{xy} z \, dz \]

\[ N_x = \int_{-t/2}^{t/2} \sigma_x d\bar{z} \]
\[ N_y = \int_{-t/2}^{t/2} \sigma_y d\bar{z} \]
\[ N_{xy} = \int_{-t/2}^{t/2} \tau_{xy} d\bar{z} \]

2.1.2 Shell theory

In order to describe a shell one starts with a plate but also considers the effects of in-plane loading. This will give rise to forces tangent to the shell surface, so called membrane forces [2, 3]. If a plate is loaded with a stress according to figure 5, the resultant forces per unit length can be calculated by using force equilibrium. This results in expressions according to equation (4) [2, 3].

Figure 5: Illustration of a plate subjected to in-plane loading by the stresses \( \sigma_x, \sigma_y, \tau_{xy} \) and the resulting forces (per unit length) \( N_x, N_y, N_{xy} \).
Since small displacements are assumed, the total stresses $\sigma_x$, $\sigma_y$ and $\tau_{xy}$ can be calculated by superposition of the stresses from ordinary plate theory and the ones from in-plane loading [2, 3]. This means combining equation (3) with equation (4). It is assumed that the membrane stresses (equation (4)) are independent of z-position and that the flexural stresses (from the moments in equation (3)) vary linearly with z [2]. This statement holds for a thin homogeneous plate made of a linear elastic material. The use of superposition gives expressions for the stresses in the plate according to equation (5) [2]. These are the governing equations for stresses in a Kirchhoff shell and are used to construct shell elements.

$$\sigma_x = \frac{N_x}{t} + \frac{12M_x z}{t^3}$$

$$\sigma_y = \frac{N_y}{l} + \frac{12M_y z}{l^3}$$

$$\tau_{xy} = \frac{N_{xy}}{t} + \frac{12M_{xy} z}{t^3}$$

(5)

2.2 Shell elements

Shell elements are used in FEA in order to analyze thin structures and all are based on shell theory, one of which was discussed in the preceding section.

Shell elements are based on either the Mindlin-theory or the Kirchhoff theory. The CQUAD4 elements in Nastran, which were used in this thesis work, are based on the Kirchhoff-theory [4].

The main difference between a solid element and a shell element is that a solid has nodes on all of its surfaces while a shell only has them on its midsurface, as can be seen in figure 6, and that the nodes have different amounts of d.o.f. associated to them [2].

![Figure 6: (a) Illustration of a eight-node solid element. (b) Illustration of a four-node shell element.](image)

Figure 6: (a) Illustration of a eight-node solid element. (b) Illustration of a four-node shell element.

Shell elements may be thought of as degenerated solid elements. Looking in the plane, as in figure 7, and assuming that the shell and solid should result in the same displacements, one can easily see that the normal displacement in the shell can be thought of as the mean normal displacement from the two corner nodes in the solid. The shell only has one displacement in the through thickness direction for each node, meaning that one must assume that the through thickness displacement is the same for both corner nodes in the solid, i.e. the shell does not change thickness during loading. The solid displays rotational stiffness due to the translational stiffness associated to each corner node. For the shell to behave in a similar way, rotational stiffness (and rotational freedom) must be associated to each node.
The nodes of a solid element have three d.o.f., these are the displacements in the $x$-, $y$- and $z$-direction. For a shell to display rotational stiffness, each node has three rotational d.o.f. in addition. These are described by the node local coordinate system $V_1$, $V_2$ and $V_3$ as is illustrated, for a general curved shell, in figure 8a, where the $V_3$-vector is defined as the through thickness normal of the shell, see figure 8b. The $V_1$- and $V_2$-vectors are orthogonal to each other and also to the $V_3$-vector [2]. Note that the $V_1V_2V_3$-system does not have to coincide with neither the global $xyz$-system nor the element local $\xi\eta\zeta$-system, seen in figure 6.

![Figure 7: How a solid element will be degenerated to a shell element. The view is in the plane, only the nodes of one corner is depicted and the arrows displays d.o.f.](image)

2.3 Element offset in Nastran

Usually when shell elements are modeled their midsurface coincide with the nodal plane, or grid-points. This means that the nodes are placed at the element midsurface, see figure 9a. If one wants to offset the element midsurface from the nodal plane in Nastran one may use ZOFFS. ZOFFS defines the distance that the midsurface is moved away from the nodal plane as can be seen in figure 9b. This is the primary way of offsetting elements with a PSHELL property if the material is still symmetric with respect to the midsurface [4]. The element will be offset in its $z$-direction, which for a flat element coincides with its normal direction [4, 5, 6, 7].
When using ZOFFS, the nodes on the nodal plane act as master nodes to the points on the element midsurface. The relation between them is established using multi point constrains (MPC:s) [7]. The MPC:s can also be thought of as rigid links (called RBE2 elements in Nastran), as illustrated in figure 9c [5]. How these connections are dealt with mathematically in Nastran will be discussed later. But first, there will be a discussion about some of the other general aspects of the ZOFFS offset.

It is important to be aware of a couple of things when using the ZOFFS offset. First of all, one should know that all element matrices, except for the mass matrix, are calculated in the offset position and are connected to the nodal plane using MPC:s (or rigid links) [5, 7]. As stresses and displacements are calculated in the reference plane (the midsurface) this will result in the creation of a load eccentricity, which will lead to a change in stiffness compared to the element with no offset [5]. This can be easily illustrated by applying a point load $F$ in the nodal plane. Since the links between the midsurface points and the nodes are rigid this means that at the midsurface points there are not just a force $F$ but also a moment $M$ which is dependent on the offset, see figure 10. This imposition of a moment $M$ will cause error in displacements and stresses as this is added to the load vector.
In addition to this, the use of ZOFFS has a couple of other limitations. The use of ZOFFS is not included in the computation of thermal loads, gravity loads or pressure loads and neither does it support differential stiffness calculations, meaning that it is not suitable for buckling analysis. Since the MPC:s (or rigid links) impose a linear transformation of the element properties, as will be discussed later, they are not suited for non-linear analysis. All of this is true for what Nastran calls the original ZOFFS method. There is an enhanced method that can be used instead, which does not have these limitations [7]. However, this is beyond the scope of this report.

2.3.1 Mathematics of ZOFFS

First of all a couple of different terms that are used in Nastran need to be explained.

- g-set: Set containing all structural grid points (nodes).
- m-set: Set containing all dependent d.o.f.
- n-set: Set containing all independent d.o.f.

To which set an entity belongs will be indicated by using subscripts, single for vectors and double for matrices. For example, the stiffness matrix belonging to the g-set is written as $K_{gg}$ and the stiffness matrix linking the n- and the m-set is written like $K_{nm}$. All of this is in agreement with the Nastran theoretical manual [8]. If nothing else is stated, this is the main source for all the equations stated in this section.

To start with, the MPC:s need to be explained as these are the foundation for how the ZOFFS are employed in the design, as mentioned before. A MPC works as to make one d.o.f. dependent on one or more d.o.f. in a linear fashion according to equation (6).

$$A_j x_j = \sum_{i=1}^{N} A_i x_i$$

Where $A_j$ and $A_i$ are scale factors, $x_j$ and $x_i$ are d.o.f. and $N$ is the number of independent d.o.f. [4, 8]. Note that a d.o.f. can not be both dependent and independent, i.e. it can not belong to both the n- and the m-set.

By moving all the terms to one side and writing them in matrix form, one arrives at equation (7).

$$[R_{gg}] \{u_g\} = \{0\}$$

Where $[R_{gg}]$ contains the constraint coefficients $A_j$ and $A_i$ and $\{u_g\}$ contains all d.o.f.

Since the dependent and independent nodes are known, both the constraint coefficient matrix and the d.o.f. (displacement) vector may be partitioned into a dependent and an independent part as shown below.
\[ \{ u_g \} = \{ u_n \} \ldots \{ u_m \} \quad [R_{gg}] = [ R_{mn} ; R_{mm} ] \] (8)

Combining this with equation (7) gives the following expression:

\[ [R_{mn}] \{ u_n \} + [R_{mm}] \{ u_m \} = \{ 0 \} \] (9)

The constraint coefficient matrix belonging to the m-set is non-singular, meaning that an expression linking the d.o.f. in the m-set to the ones in the n-set can be obtained from equation (9) as

\[ \{ u_m \} = [G_{mn}] \{ u_n \} \] (10)

where,

\[ [G_{mn}] = -[R_{mm}]^{-1} [R_{nn}] \] (11)

This will now be used in order to modify the stiffness matrix. Worth noting is that the \([G_{mn}]\) matrix is orthogonal.

Before any MPC:s are imposed, the problem to be solved is the standard linear elastic FE-problem, equation (12).

\[ [K_{gg}] \{ u_g \} = \{ P_g \} \] (12)

Where \([K_{gg}]\) is the stiffness matrix and \([P_g]\) is the load vector.

Now, if there are MPC:s present this means that the g-set can be divided into the m- and the n-set, as done in equation (9). Applying the same reasoning to equation (12) gives the following:

\[
\begin{bmatrix}
[K_{nn}] & [K_{nm}] \\
[K_{mn}]^T & [K_{mm}]
\end{bmatrix}
\begin{bmatrix}
\{ u_n \} \\
\{ u_m \}
\end{bmatrix}
=\begin{bmatrix}
\{ P_n \} \\
\{ P_m \}
\end{bmatrix}
\] (13)

Where \([\bar{F}_n]\) is a vector containing the forces acting on all independent d.o.f. and \([P_m]\) contains all the forces acting on the dependent d.o.f. As the MPC:s are enforced they will give rise to constraint forces on the dependent d.o.f. How these are handled will now be shown.

By definition, a constraint can not perform any work on the point it is acting on. This may be written for a constraint according to equation (14).

\[ W_c = \sum_{l=1}^N q_{cl} u_l = 0 \] (14)

Where \(W_c\) is the work carried out due to constraint \(c\) and \(N\) represents the number of d.o.f. The constraint force vector, \(q_{cl}\), contains information about which constraint forces are applied to which d.o.f. (\(l\)) due to constraint \(c\). Writing equation (7) for a constraint \(c\) gives equation (15):

\[ \sum_{l=1}^N R_{cl} u_l = 0 \] (15)

\(^1\)Note that double subscript is used here for a vector. The reason for this is that each constrain corresponds to a row in \([R_{cl}]\). The first subscript is thus fixed for a given constraint, meaning that it is a row matrix (with one row), or a vector, that is investigated.
As \( c \) represents a constraint it corresponds to row \( c \) in the constraint coefficient matrix, and as \( l \) represents a d.o.f. it corresponds to a column in said matrix and a row in the d.o.f. vector.

Since \( u_l \) in equation (14) is the same as in equation (15), the only way to fulfill equation (14) is if the constraint force vector for constraint \( c \), is proportional to the \( c \):th row in the constraint coefficient matrix. Note that \( q_{cl} \) contains forces and these do not have to equal the values in \( R_{cl} \), which in a way prescribe displacements. This means that one may write:

\[
q_{cl} = R_{cl}Q_c
\]

or

\[
\{q_c\} = \{R_c\}Q_c
\]

Where \( Q_c \) is a scale factor, or force constant, for the \( c \):th constraint and \( \{q_c\} \) is the constraint force vector for the \( c \):th constraint. This has the same length as the d.o.f. vector. Note that \( \{R_c\} \) contains all columns for row \( c \) in the constraint coefficient matrix. If there are more than one constraint present, one simply adds the constraint force vectors for the different constraints to get the total constraint force vector, see equation (17).

\[
\{q_c\} = \sum_{k=1}^{c} \{q_k\}
\]

Where \( c \) represents the number of constraints.

Writing this in terms of the \( m \)- and \( n \)-set gives the expression in equation (18). Note that this is done in the same manner as for the displacement vector, see equation (10).

\[
\{q_m\} = [G_{mn}] \{q_n\}
\]

As the constraints give rise to constraint forces, these have to be added to the equilibrium equations for them to hold. This is done using equations (18), (10) and (13) to give the complete equilibrium problem according to equation (19).

\[
\begin{pmatrix}
[K_{nn}] & [G_{mn}]^T & \{u_n\} & \{P_n\} \\
[K_{nm}] & [K_{mm}] & -[I] & \{u_m\} \\
[G_{mn}] & -[I] & [0] & \{q_m\}
\end{pmatrix}
= \begin{pmatrix}
0 \\
0 \\
0
\end{pmatrix}
\]

Remember the minus sign incorporated in the \([G_{mn}]\) matrix from equation (10). \([I]\) and \([0]\) is the unity matrix and zero matrix respectively. The constraint force vector for the dependent d.o.f. \( \{q_m\} \) is a so called Lagrangian multiplier.
Elimination of \{u_m\} and \{q_m\} from the equation system in equation (19) gives the following expression:

\[
\left( [K_{nn}] + [K_{nm}] [G_{mn}] + [G_{mn}]^T [K_{nm}]^T + [G_{mn}]^T [K_{nm}] [G_{mn}] \right) \{u_n\} = \\
\{P_n\} + [G_{mn}]^T \{P_m\}
\]

\[
\iff \\
[K_{nn}] \{u_n\} = \{P_n\}
\]

This new equation has components belonging to the n-set (the independent d.o.f.) only, and this is the equation solved in Nastran. Note that both the stiffness matrix, and the load matrix, have been updated to incorporate the effect of the MPC:s. What has to be defined, before an analysis containing MPC:s can be carried out, is the constraint matrix \([G_{mn}]\). For the ZOFFS offset, this is done automatically by Nastran. Otherwise, this can be done either by the MPC-card or by the use of rigid links.

The use of rigid links is perhaps a more intuitive way of looking at these constraints, and as this will result in MPC:s, the rigid link will be discussed in detail below. This will give further insight into how the ZOFFS constraints are formed.

2.3.2 Rigid links in Nastran (RBE2 elements)

A rigid link is easiest thought of as a rod which has infinite stiffness, no mass, no cross sectional area and one node in each end (in the Nastran manual [4] it is called a rigid RBE2 element). It does not work as to prescribe infinite stiffness to the stiffness matrix of the element (link), but rather imposes constraints on the dependency of connected nodes, just like MPC:s. The advantage of using this approach is that ill-conditioning of the global stiffness matrix is avoided. This is otherwise a problem if very stiff elements are used together with less stiff elements [2, 9].

When using a rigid link, constraints are established between its nodes. The number of d.o.f. that are associated with the rigid link depends on what element it is attached to. If it is attached to solid elements it has three d.o.f. associated to each node and if it is attached to a shell element it has either five or six d.o.f. depending on the configuration of the shell, as discussed previously [4, 8]. This holds for the most common rigid link used in Nastran, called RBE2. There are other rigid links with a slightly different behavior. These are however beyond the scope of this report and the interested reader is referred to the Nastran Theoretical manual [8], or Quick reference guide [4].

Since this report mainly aims at investigating shell elements, the rigid link discussed in this section will be one with six d.o.f. (the most general shell element). This type of rigid link is illustrated in figure 11.
Figure 11: General illustration of a rigid link with its different d.o.f. depicted.

If one assumes small displacements and rotations, while remembering that the element is rigid, one can easily set up the relations between the displacements and angles for the two nodes $A$ and $B$ in figure 11. These are written in matrix form in equation (21).

$$
\begin{bmatrix}
    u_A \\
    v_A \\
    w_A \\
    \theta_{XA} \\
    \theta_{YA} \\
    \theta_{ZA}
\end{bmatrix} =
\begin{bmatrix}
    1 & 0 & 0 & 0 & (Z_B - Z_A) & -(Y_B - Y_A) \\
    0 & 1 & 0 & -(Z_B - Z_A) & 0 & (X_B - X_A) \\
    0 & 0 & 1 & (Y_B - Y_A) & -(X_B - X_A) & 0 \\
    0 & 0 & 0 & 1 & 0 & 0 \\
    0 & 0 & 0 & 0 & 1 & 0 \\
    0 & 0 & 0 & 0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
    u_B \\
    v_B \\
    w_B \\
    \theta_{XB} \\
    \theta_{YB} \\
    \theta_{ZB}
\end{bmatrix}
$$

(21)

This equation is expressed in terms of the global coordinate system. By use of the coordinate transformation matrix, equation (22), this equation can be rewritten in terms of node-local coordinate systems. The relation between the global and node-local coordinate systems may be written according to equation (23) for point $A$ [2].

$$
[T] =
\begin{bmatrix}
    l_1 & m_1 & n_1 \\
    l_2 & m_2 & n_2 \\
    l_3 & m_3 & n_3
\end{bmatrix}
$$

(22)

$$
\begin{bmatrix}
    u_A' \\
    v_A' \\
    w_A' \\
    \theta_{XA}' \\
    \theta_{YA}' \\
    \theta_{ZA}'
\end{bmatrix} =
\begin{bmatrix}
    l_1 & m_1 & n_1 & 0 & 0 & 0 \\
    l_2 & m_2 & n_2 & 0 & 0 & 0 \\
    l_3 & m_3 & n_3 & 0 & 0 & 0 \\
    0 & 0 & 0 & l_1 & m_1 & n_1 \\
    0 & 0 & 0 & l_2 & m_2 & n_2 \\
    0 & 0 & 0 & l_3 & m_3 & n_3
\end{bmatrix}
\begin{bmatrix}
    u_A \\
    v_A \\
    w_A \\
    \theta_{XA} \\
    \theta_{YA} \\
    \theta_{ZA}
\end{bmatrix} =
\begin{bmatrix}
    [T_A] & [0] & [T_A]
\end{bmatrix}
\begin{bmatrix}
    u_A \\
    v_A \\
    w_A \\
    \theta_{XA} \\
    \theta_{YX} \\
    \theta_{ZA}
\end{bmatrix}
$$

(23)

In both equation (22) and (23), $l_i, m_i, n_i$ are direction cosines between the new primed axes and the original axes, these are illustrated in figure 12. The index number indicates which new axis the rotation is measured towards.
Figure 12: General illustration of the rotation of one coordinate system relative another.

Now using equation (23) for point \( B \) as well (note that the direction cosines for point \( A \) generally does not coincide with the ones for point \( B \)) and inserting the two transformations into equation (21), the following expression is obtained:

\[
\begin{bmatrix}
    u'_A \\
    v'_A \\
    w'_A \\
    \theta'_{XA} \\
    \theta'_{YA} \\
    \theta'_{ZA}
\end{bmatrix}_{6 \times 1} =
\begin{bmatrix}
    [T_A] & [0] \\
    [0] & [T_A]
\end{bmatrix}_{6 \times 6}
\begin{bmatrix}
    0 & \pi & -\bar{y} \\
    -\bar{x} & 0 & \pi \\
    \bar{y} & -\bar{x} & 0
\end{bmatrix}_{3 \times 3}
\begin{bmatrix}
    [T_B^T] & [0] \\
    [0] & [T_B^T]
\end{bmatrix}_{6 \times 6}
\begin{bmatrix}
    u'_B \\
    v'_B \\
    w'_B \\
    \theta'_{XB} \\
    \theta'_{YB} \\
    \theta'_{ZB}
\end{bmatrix}_{6 \times 1}
\]

(24)

Where \( \bar{x} = (X_B - X_A), \bar{y} = (Y_B - Y_A) \) and \( \bar{z} = (Z_B - Z_A) \). These three six by six matrices are multiplied into one six by six matrix called \([G_{AB}]\) according to equation (25). In \([G_{AB}]\), each row corresponds to a dependent d.o.f. at node A while each column represents an independent reference d.o.f. at node B [2].

\[
\begin{bmatrix}
    u'_A \\
    v'_A \\
    w'_A \\
    \theta'_{XA} \\
    \theta'_{YA} \\
    \theta'_{ZA}
\end{bmatrix}_{6 \times 1} = [G_{AB}]
\begin{bmatrix}
    u'_B \\
    v'_B \\
    w'_B \\
    \theta'_{XB} \\
    \theta'_{YB} \\
    \theta'_{ZB}
\end{bmatrix}_{6 \times 1}
\]

(25)

Combining this equation for all dependent nodes yield the same amount of linear equations as there are dependent d.o.f. These equations may be written in matrix form according to equation (26).

\[
\begin{bmatrix}
    u \\
    v \\
    w \\
    \theta_{XA} \\
    \theta_{YA} \\
    \theta_{ZA}
\end{bmatrix}_{m \times 1} = [G_{B}]
\begin{bmatrix}
    u'_B \\
    v'_B \\
    w'_B \\
    \theta'_{XB} \\
    \theta'_{YB} \\
    \theta'_{ZB}
\end{bmatrix}_{m \times 6}
\]

(26)

Here \( \{u\} \) is in the global coordinate system and \( \{u'_B\} \) is in a local system. Note that \( [G_{B}] = \begin{bmatrix} [T_A^T] & [0] \\
[0] & [T_A^T]\end{bmatrix}[G_{AB}] \) and \( m \) is the number of dependent d.o.f. The components in \( \{u\} \) belong to the m-set and the components in \( \{u'_B\} \) belong to the n-set [8].
Comparing equation (26) with equation (10), one concludes that they are on the same form and the components belong to the same sets. This means that the two constraint matrices are the same. From here on, the rigid elements will be regarded as MPC:s by the code, meaning that the same technique for altering the stiffness matrix is employed as in the previous section [8]. The advantage of using rigid links, instead of MPC:s, is that the only thing that needs to be specified is the geometry of the link, i.e. where the connected nodes are located. Note that if a shell element with only five d.o.f. is used, the sixth row and column in the matrices in equation (21) and (23) will disappear together with the sixth row in the vectors in said equations.

This is exactly how the ZOFFS work. When elements are offset from the nodal plane, data regarding their new midsurface position is recorded. This is used to form MPC:s (or rigid links) between the element midsurface and the nodes. Using this, the stiffness and load matrices are updated according to the section above. Note that, the moment that arises from offset due to an in-plane load (figure 10), will be incorporated into the load vector by use of the MPC:s.

3 Method

The work carried out can be separated into two main parts. One part aims to get as much theoretical knowledge as possible about shell elements, the effects of offset modeling and also how this is handled mathematically by Nastran, as discussed in the previous section.

The other part is more practical. Here different simulations are to be carried out in order to compare offset modeling with midsurface modeling. This in order to see how big the differences between the two ways of modeling are. From the C-bar in figure 2 one can identify three main cases that are of interest to investigate (in addition to the actual C-bar):

- 90° corners, figure 13a
- Thickness variation with a common bottom surface, figure 13b
- T-junctions, figure 13c

![Figure 13: Illustration of the different cases of interest to model. (a) A part with a 90° corner. (b) An example of a thickness variation. (c) A part with a T-junction.](image)

The T-junction can be seen as a special case of the 90° corner and thus only the latter will be investigated. All pre-processing will be done using HyperMesh and the post-processing is done using HyperView [10]. The solver used for the analyses is MSC Nastran [1].
3.1 Thickness variation model

In order to investigate the thickness variations, a plate with outside measurements of 50 times 50 mm, according to figure 14a, is used in the simulations. The thinner part of the plate has a constant thickness of one millimeter while the thicker part changes from two to eleven millimeters with one millimeter increments between simulations.

This plate is loaded with either a transverse load $T$, an in-plane load $N$ or a bending moment $M$ at either its thick or thin end. The end that is not loaded is fixed, i.e. all degrees of freedom (d.o.f.) are locked, see figure 14b and 16. When the loads are applied at the thick edge the loads in figure 14b are mirrored in the middle of the plate. All loads are equally distributed over the end nodes, see figure 16, and their total magnitudes are: $T = N = 10$ N and $M = 10$ Nmm. The material used for all simulations is steel with Young’s modulus $E = 210000$ MPa and Poisson’s ratio $\nu = 0.3$.

![Figure 14:](image)

Figure 14: (a) Drawing of the geometry used to study the effect of different modeling techniques for thickness variations. The thickness of the thickest part is allowed to vary between two and eleven millimeters. (b) Load cases used for the different models, here they are applied at the thin edge and the thick edge is fixed.

The plate is investigated using three different ways of modeling:

- Individual midsurfaces connected with rigid RBE2 elements (denoted R=Rigid link), figure 15a
- A common midsurface with the elements of the thicker part offset from the midsurface (denoted C-M=Common midsurface), figure 15b
- Elements are modeled in a common interface surface (the bottom of the plate in this case) and offset from this (denoted If=Interface), figure 15c

A comparison with a solid model (denoted S=Solid) is also made in order to see which shell model gives the most accurate results. The solid elements used are eight-node cubic elements.
Figure 15: Model of a thickness variation using (a) rigid links to connect individual midsurfaces (denoted R=Rigid link), (b) offset from a common midsurface (denoted C-M=Common midsurface), and (c) offset from a common interface surface (denoted If=Interface).

The boundary conditions for the plate can be seen together with the mesh for the solid model in figure 16a for an in-plane load $N$ at the thick edge which is two millimeters thick here. At the fixed edge all nodes are locked in all d.o.f. The same principle is applied for all loads applied at either edge. For the shell models the boundary conditions for the end nodes are according to figure 16b. This holds for all load cases applied at either edge. The mesh for the shell is made up of quadratic elements with a side length of 0.25 mm.

Figure 16: (a) The boundary conditions and mesh for a plate loaded with an in-plane load in total of ten Newton at the two millimeter thick edge. It is fixed at all nodes at the thin edge. (b) The boundary conditions for a shell model for the same load case as for the solid in (a).

3.2 90° corner model

For this case the geometries in figure 17a and figure 17b are used. They are to be modeled using traditional midsurface modeling (denoted M=Midsurface), see figure 18a, as well as offset modeling from a common interface surface (denoted IF=Interface), see figure 18b and 18c. Furthermore, these shell models are compared to a solid model (denoted S=Solid).
Two cases will be investigated for the corner, one where the two adjacent plates have the same thickness (figure 18a and 18b) and one where they do not (figure 18c).

For the case of both plates having the same thickness the geometry of the plate is according to figure 17a and it is loaded according to figure 17c. One of the loads is applied at a time and the non loaded edge is fixed in all d.o.f. Just as for the thickness variation models the load is equally distributed over all end nodes, see figure 19, and their magnitudes are: $T = N = 10\,\text{N}$ and $M = 10\,\text{Nmm}$. The thickness of the plates is allowed to change from one to six millimeters with one millimeter increments throughout the simulations.
Figure 18: Model of a 90° corner using (a) midsurface modeling, (b) offset from a common interface surface with the same thickness for both plates, and (c) offset from a common interface surface with different thickness for the two plates.

For the case of the plates having different thicknesses one is held constant ($t_1$ in figure 17d) at one millimeter while the other ($t_2$ in figure 17d) is allowed to change from two to seven millimeters with one millimeter increments throughout the simulations. The geometry of the plates can be seen in figure 17b. In this case two ways of loading are of interest. One when loading the thicker plate and one when loading the thinner plate. The edge that is not loaded is fixed in all d.o.f. and the loads used are the same three as in the case of the plates having the same thickness. The thick edge is loaded according to figure 17c and the thin edge according to figure 17d. Note that all loads are applied (distributed) in the same manner as in figure 19 for both edges.

The solid model make use of eight-node cubic elements and the mesh can be seen in figure 19a. The shell models have a mesh consisting of quadratic elements with a side length of 0.25 mm. The material used is steel with Young’s modulus $E = 210000$ Mpa and Poisson’s ratio $\nu = 0.3$.

Figure 19: (a) The mesh and boundary conditions for the corner with different plate thicknesses (one and five millimeters) when loaded in the plane at the thick edge. (b) The boundary conditions for the shell model. The numbers indicate which d.o.f. that are locked.
3.3 C-bar model

As a final confirmation of the conclusions made for the simpler geometries a real component is to be investigated using a combination of midsurface and offset modeling. Meaning that both offset and non-offset elements will be present. This is then to be compared with a solid model.

The component to be investigated is a part of a C-bar, henceforth referred to as C-bar, which is depicted in figure 2. This is, as mentioned previously, a hinge connecting the flaps to the rest of the wing of an airplane. The exact measurements is not allowed to be presented but the component is about 1500 mm long, 650 mm wide and 75 mm thick.

The real boundary conditions (BC:s) for this component are quite complicated and irrelevant for this investigation, which is why a simplified set of BC:s will be used.

The component is modeled using four-node quadratic Kirchhoff shell elements with a size of five millimeters. The reinforcements (blue parts in figure 20) are modeled using the midsurface, i.e. no offset, while the rest of the structure (red parts in figure 20) is modeled using an offset from an interface surface (the outside surface of the solid in this case).

The solid model make use of ten-node tetrahedral elements which also have a size of five millimeters. For both models the material used is steel with a Young’s modulus $E = 210000 \text{ MPa}$ and Poisson’s ratio $\nu = 0.3$.

![Figure 20: The shell model of the C-bar. The blue parts are midsurfaces and have no element offset. The red parts are modeled using an interface surface and the elements are offset half of their thickness towards the inside of the component.](image)

For both the solid and shell model a force of 150000 N is applied at the center of the hole in the front of the C-bar, see figure 21a. The force is applied parallel to the top of the yellow reinforcement seen in this figure and is transferred to the surface of the surrounding hole by connecting the surface nodes to the loaded node (in the absolute center of the hole) with RBE3 elements. These are interpolation elements used to distribute loads without introducing stiffness to the structure [4].

The component is fixed at the far end at ten nodes equally distributed ten millimeters from the surface, see figure 21b. The area over which the nodes are distributed is 403 mm times 34 mm and is centered over the back surface of the C-bar. Each node is completely fixed (all d.o.f. are locked) and is connected to either nine quadratic shell elements (figure 22a), or the base of 18 tetrahedral solid elements with rigid RBE2 elements. Only the corner nodes of the tetrahedral elements are connected to the fixed node, as can be seen in figure 22, meaning that 16 nodes are locked in both the shell and solid model. The reason that these BC:s are chosen is because they represent roughly how the entire C-bar is fixed in reality. However, one would not use RBE2 elements in a real case but rather CBUSH elements (springs). This is however beyond the scope of this report and thus RBE2 elements are used.

---

2The tetrahedral elements have ten nodes in total.
Figure 21: (a) The force (blue arrow) that is applied to the C-bar and the RBE3 elements (green) used to transfer the force to the structure. (b) The location of the nodes which are locked and connected to the component with rigid RBE2 elements.

Figure 22: How each locked node is connected to (a) nine quadratic shell elements or (b) the solid tetrahedral mesh using RBE2 elements.

4 Results

In this section the results from the thickness variation models, the 90° corner models and the C-bar models will be presented. For all models a mesh convergence study has been carried out, however this is not presented in this report.

4.1 Thickness variation

The stresses are evaluated in the bottom of the plate for three different areas. These are the entire bottom of the plate and two smaller areas. These are located in the middle of each part of the bottom of the plate and are depicted in figure 23. For both of these areas, the stress field and the magnitude of the in-plane stress ($\sigma_y$) is investigated. For the overall stress behavior, focus is on the maximum von Mises (vM) stress ($\sigma_{vM}$).
4.1.1 Overall behavior

The overall stress behavior for the four different modeling cases (S=Solid, R=Rigid link, C-M=Common midsurface and If=Interface) can be seen in figure 24. This shows the vM stress at the bottom of the plate for the case of the thicker part being two millimeters thick. The loads are applied at the thick edge of the plate while the thin one is fixed. The applied loads can be seen in figure 14b. This overall stress behavior is similar to the one that arises when the loading is made at the edge of the thinner plate, as can be seen in Appendix A section A1, here one also finds the stress field for the case of the thick plate being six millimeters thick and loaded. One difference worth noting is that none of the shell models give the same results when loaded with an in-plane load at the thick edge, see figure 24. But both the rigid link model and the common midsurface model do if they are loaded at the thinner edge, as can be seen in figure 61 in Appendix A section A1.

Furthermore, it is noticeable that the stress field is the same for all shell models when not loaded with an in-plane load. Additionally, the position of the maximum and minimum stresses change compared to the solid model as the thickness ratio increases, which is to be expected as the shell assumptions agree worse with the solid then.
Figure 24: $\sigma_{vM}$ at the bottom of the plate when loaded with (a) a transverse force $T$, (b) a bending moment $M$ and (c) an in-plane load $N$. All loads are applied at the thick edge of the plate which is two millimeters thick. The applied loads can be seen in figure 14b.
As differences between the shell models only occur when loaded in the plane, this is the case that will be the main focus. For this case, the overall maximum displacement magnitude is extracted and can be found in tables 1 and 2. The maximum displacement is found at the loaded end, see figure 25. The deformation of the plate, where the thicker part is eleven millimeters thick, when loaded with an in-plane load at the thick edge can be seen in figure 26.

Figure 25: The displacement magnitude when the plate is loaded with a tensile in-plane load at the thick end, which is eleven millimeters thick.

Figure 26: Deformation of the plate when subject to an in-plane tensile load of ten Newton at the eleven millimeter thick edge. The deformation has been scaled up by a factor 100. The upper left picture is of the solid (S) and the upper right of the rigid link model (RL). The bottom left depicts the common mid-surface model (C-M) and the bottom right the interface model (If).
Table 1: The maximum displacement values for the solid and shell models and the ratio between them when loaded at the thin edge. \( t \)=thickness, \( S \)=Solid, \( R \)=Rigid link, \( C\text{-}M \)=Common midsurface and \( If \)=Interface.

<table>
<thead>
<tr>
<th>( t ) [mm]</th>
<th>( S ) [mm]</th>
<th>( R ) [mm]</th>
<th>( C\text{-}M ) [mm]</th>
<th>( If ) [mm]</th>
<th>( R/S )</th>
<th>( C\text{-}M/S )</th>
<th>( If/S )</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>( 5.023E^{-3} )</td>
<td>( 4.981E^{-3} )</td>
<td>( 2.626E^{-4} )</td>
<td>( 5.499E^{-5} )</td>
<td>0.992</td>
<td>0.052</td>
<td>1.095</td>
</tr>
<tr>
<td>3</td>
<td>( 1.005E^{-2} )</td>
<td>( 9.934E^{-3} )</td>
<td>( 1.663E^{-4} )</td>
<td>( 5.203E^{-3} )</td>
<td>0.988</td>
<td>0.016</td>
<td>0.518</td>
</tr>
<tr>
<td>4</td>
<td>( 1.511E^{-2} )</td>
<td>( 1.489E^{-2} )</td>
<td>( 1.085E^{-4} )</td>
<td>( 5.096E^{-5} )</td>
<td>0.985</td>
<td>0.007</td>
<td>0.337</td>
</tr>
<tr>
<td>5</td>
<td>( 2.021E^{-2} )</td>
<td>( 1.986E^{-2} )</td>
<td>( 7.885E^{-5} )</td>
<td>( 5.047E^{-3} )</td>
<td>0.983</td>
<td>0.004</td>
<td>0.250</td>
</tr>
<tr>
<td>6</td>
<td>( 2.535E^{-2} )</td>
<td>( 2.485E^{-2} )</td>
<td>( 6.175E^{-5} )</td>
<td>( 5.020E^{-3} )</td>
<td>0.980</td>
<td>0.002</td>
<td>0.198</td>
</tr>
<tr>
<td>7</td>
<td>( 3.056E^{-2} )</td>
<td>( 2.984E^{-2} )</td>
<td>( 5.131E^{-5} )</td>
<td>( 5.003E^{-5} )</td>
<td>0.976</td>
<td>0.002</td>
<td>0.164</td>
</tr>
<tr>
<td>8</td>
<td>( 3.583E^{-2} )</td>
<td>( 3.485E^{-2} )</td>
<td>( 4.463E^{-5} )</td>
<td>( 4.993E^{-3} )</td>
<td>0.973</td>
<td>0.001</td>
<td>0.139</td>
</tr>
<tr>
<td>9</td>
<td>( 4.118E^{-2} )</td>
<td>( 3.989E^{-2} )</td>
<td>( 4.016E^{-5} )</td>
<td>( 4.985E^{-3} )</td>
<td>0.969</td>
<td>0.001</td>
<td>0.121</td>
</tr>
<tr>
<td>10</td>
<td>( 4.661E^{-2} )</td>
<td>( 4.494E^{-2} )</td>
<td>( 3.707E^{-5} )</td>
<td>( 4.980E^{-3} )</td>
<td>0.964</td>
<td>0.001</td>
<td>0.107</td>
</tr>
<tr>
<td>11</td>
<td>( 5.213E^{-2} )</td>
<td>( 5.001E^{-2} )</td>
<td>( 3.486E^{-5} )</td>
<td>( 4.976E^{-3} )</td>
<td>0.959</td>
<td>0.001</td>
<td>0.095</td>
</tr>
</tbody>
</table>

Table 2: The maximum displacement values for the solid and shell models and the ratio between them when loaded at the thin edge. \( t \)=thickness, \( S \)=Solid, \( R \)=Rigid link, \( C\text{-}M \)=Common midsurface and \( If \)=Interface.

<table>
<thead>
<tr>
<th>( t ) [mm]</th>
<th>( S ) [mm]</th>
<th>( R ) [mm]</th>
<th>( C\text{-}M ) [mm]</th>
<th>( If ) [mm]</th>
<th>( R/S )</th>
<th>( C\text{-}M/S )</th>
<th>( If/S )</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>( 6.381E^{-4} )</td>
<td>( 6.423E^{-4} )</td>
<td>( 6.423E^{-4} )</td>
<td>( 3.046E^{-3} )</td>
<td>1.007</td>
<td>1.007</td>
<td>4.774</td>
</tr>
<tr>
<td>3</td>
<td>( 3.894E^{-4} )</td>
<td>( 3.843E^{-4} )</td>
<td>( 3.843E^{-4} )</td>
<td>( 2.310E^{-3} )</td>
<td>0.987</td>
<td>0.987</td>
<td>5.932</td>
</tr>
<tr>
<td>4</td>
<td>( 2.581E^{-4} )</td>
<td>( 2.458E^{-4} )</td>
<td>( 2.458E^{-4} )</td>
<td>( 2.052E^{-3} )</td>
<td>0.952</td>
<td>0.952</td>
<td>7.950</td>
</tr>
<tr>
<td>5</td>
<td>( 1.875E^{-4} )</td>
<td>( 1.702E^{-4} )</td>
<td>( 1.702E^{-4} )</td>
<td>( 1.933E^{-3} )</td>
<td>0.908</td>
<td>0.908</td>
<td>10.309</td>
</tr>
<tr>
<td>6</td>
<td>( 1.463E^{-4} )</td>
<td>( 1.255E^{-4} )</td>
<td>( 1.255E^{-4} )</td>
<td>( 1.869E^{-3} )</td>
<td>0.858</td>
<td>0.858</td>
<td>12.775</td>
</tr>
<tr>
<td>7</td>
<td>( 1.206E^{-4} )</td>
<td>( 9.730E^{-5} )</td>
<td>( 9.730E^{-5} )</td>
<td>( 1.830E^{-3} )</td>
<td>0.807</td>
<td>0.807</td>
<td>15.174</td>
</tr>
<tr>
<td>8</td>
<td>( 1.045E^{-4} )</td>
<td>( 7.856E^{-5} )</td>
<td>( 7.856E^{-5} )</td>
<td>( 1.804E^{-3} )</td>
<td>0.752</td>
<td>0.752</td>
<td>17.263</td>
</tr>
<tr>
<td>9</td>
<td>( 9.344E^{-5} )</td>
<td>( 6.609E^{-5} )</td>
<td>( 6.609E^{-5} )</td>
<td>( 1.787E^{-3} )</td>
<td>0.707</td>
<td>0.707</td>
<td>19.125</td>
</tr>
<tr>
<td>10</td>
<td>( 8.561E^{-5} )</td>
<td>( 5.756E^{-5} )</td>
<td>( 5.756E^{-5} )</td>
<td>( 1.775E^{-3} )</td>
<td>0.672</td>
<td>0.672</td>
<td>20.734</td>
</tr>
<tr>
<td>11</td>
<td>( 7.898E^{-5} )</td>
<td>( 5.142E^{-5} )</td>
<td>( 5.142E^{-5} )</td>
<td>( 1.765E^{-3} )</td>
<td>0.651</td>
<td>0.651</td>
<td>22.347</td>
</tr>
</tbody>
</table>
One may also want to apply the load in the same point in space (in the midsurface) for all models and connect it to the nodes using RBE3 elements, according to figure 27. This is done for the case of the loading being a tensile in-plane load of 10 N applied at the thick edge of the plate, which is eleven millimeters thick. In addition, the thin plate is fixed in the midsurface and connected to the nodal plane using RBE2 elements. The deformation for this is found in figure 28 and the maximum displacement magnitude is found in table 3. The case of the thicker plate being eleven millimeters thick is deemed to be the worst, see table 1, why this is chosen for this test.

**Figure 27:** Illustration of how the load is applied in the midsurface and connected to the nodal plane using RBE3 elements. The fixed nodes are connected to the midsurface using RBE2 elements. The numbers indicates which d.o.f. that are fixed.

**Figure 28:** The deformation for the different models when the load is applied in the midsurface of the eleven millimeter thick plate by use of RBE3 elements. The constraints are also located in the midsurface and connected to the nodes with RBE2 elements (rigid links). The upper left picture is of the solid and the upper right is of the rigid link model. The bottom left is of the common midsurface model and the bottom right of the interface model.
Table 3: The maximum displacement values for the solid and shell models and the ratio between them when loaded at the thick edge with the load applied in the midsurface of an eleven millimeter thick plate. t=thickness, S=Solid, R=Rigid link, C-M=Common midsurface and If=Interface.

<table>
<thead>
<tr>
<th>t [mm]</th>
<th>S [mm]</th>
<th>R [mm]</th>
<th>C-M [mm]</th>
<th>If [mm]</th>
<th>R/S</th>
<th>C-M/S</th>
<th>If/S</th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>$5.214E^{-2}$</td>
<td>$5.001E^{-2}$</td>
<td>$4.461E^{-2}$</td>
<td>$4.958E^{-2}$</td>
<td>0.959</td>
<td>0.856</td>
<td>0.951</td>
</tr>
</tbody>
</table>

In addition, it might be interesting to know how the different shell models preforms computationally, i.e. how long computational time does each way of modeling need. This is the same for all thicknesses of the shell and is found in table 4, for the loads $T, M$ and $N$ being applied one at a time in the same analysis\(^3\).

Table 4: The computation times for the different shell models when the three loads $T, M, N$ are applied one at a time in the same run. The time for the computation is given in seconds.

<table>
<thead>
<tr>
<th>Total computation time for the different shell models [sec]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid link</td>
</tr>
<tr>
<td>0.13</td>
</tr>
</tbody>
</table>

4.1.2 Stresses in the thick part

In order to get a better view of the changes of the stresses in the thick part of the plate a smaller section is investigated. This is located in the middle of the bottom of the thick plate. It spans five millimeters in the y-direction and stretches across the entire width of the plate, see figure 23. The outside measurements of the plate are, as said before, 50 times 50 mm. The stress field for the in-plane stress ($\sigma_y$) for loading at both the thin and the thick edge with an in-plane load $N$ is depicted in figure 29. The thickness of the plate in this figure is two millimeters and it illustrates the differences in the stress field for all thicknesses rather well.

\(^3\)Three analyses are performed at once using three different load cases in Hypermesh and Nastran.
Figure 29: The in-plane stress ($\sigma_y$) in the thick plate when (a) loaded at the thin edge and (b) loaded at the thick edge. All cases are subject to a tensile in-plane load of ten Newton and the thickness of the thicker part of the plate is two millimeters\(^4\).

\(^4\)The scales are set to be the same. But only the second largest/smallest values can be set in HyperView, which is why the scale is only the same in-between these values.
The biggest in-plane stress experienced in this area of the plate (considering the magnitude) can be plotted for all models and is displayed in figure 30.

**Figure 30:** Biggest in-plane stress ($\sigma_y$) in the middle of the thick plate, when subjected to a tensile in-plane load $N$ of ten Newton at (a) the thin edge and (b) the thick edge.

### 4.1.3 Stresses in the thin part

Just as in the previous section, the in-plane stress in the center of the thinner plate is investigated, both for loading at the thin and the thick edge. The dimensions of the slit which is investigated are the same as for the thick plate, and can be seen in figure 23. The stress field for the in-plane stress ($\sigma_y$) is according to figure 31 when the thickest part of the plate is two millimeters thick. These stress fields once again describe the behavior for the models with thicker plates as well.
Figure 31: The in-plane stress ($\sigma_y$) in the thin plate when (a) loaded at the thin edge and (b) loaded at the thick edge. All cases are subject to a tensile in-plane load of ten Newton and the thickness of the thicker plate is two millimeters.\footnote{The scales are set to be the same. But only the second largest/smallest values can be set in HyperView, which is why the scale is only the same in-between these values.}
As before, the biggest in-plane stresses (considering the magnitude) are plotted for the case of a tensile in-plane load and can be seen in figure 32.

![Max YY thin slice, load on thin edge](image)

**Figure 32:** Biggest in-plane stress ($\sigma_y$) in the thin plate when subjected to a tensile in-plane load $N$ of ten Newton at (a) the thin edge and (b) the thick edge.

### 4.2 90° corner

The 90° corner case can be divided into two sub-cases as mentioned before. One where the thickness of the adjacent parts is the same and one where it is not. For both of these cases there are two major areas that are of interest to investigate, in addition to the overall behavior. These are two slits in the middle of each plate, see figure 33.

![Figure 33: The two major areas to be investigated for the 90° corner case. The areas are the same for both the case of the thicknesses of the adjacent plates being the same and different.](image)

### 4.2.1 Same thickness ratio

To start with the overall displacement behavior for the plates has to be mentioned. The largest displacement magnitude for both a transverse load $T$ and an in-plane load $N$ can be found in tables 5 and 6. This is found at the loaded end of the structure, see figure 34. A deformation plot for the corner when loaded in the plane can be seen in figure 35. Note that there are little difference in displacement for the bending moment $M$ (depicted in figure 17c) why this omitted.
Figure 34: The displacement magnitude for the corner when loaded with an in-plane force. The adjacent plates are both five millimeters thick.

Table 5: The maximum displacement values for the solid and shell models and the ratio between them when loaded with a transverse load $T$. The thickness ratio between the two parts of the corner is constant, i.e. $t_1 = t_2 = t$ which can be seen in figure 17c. $t=$thickness, S=Solid, M=Midsurface and If=Interface.

<table>
<thead>
<tr>
<th>$t$ [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.857</td>
<td>1.860</td>
<td>1.893</td>
<td>1.002</td>
<td>1.019</td>
</tr>
<tr>
<td>2</td>
<td>$2.257E^{-1}$</td>
<td>$2.260E^{-1}$</td>
<td>$2.340E^{-1}$</td>
<td>1.001</td>
<td>1.037</td>
</tr>
<tr>
<td>3</td>
<td>$6.498E^{-2}$</td>
<td>$6.509E^{-2}$</td>
<td>$6.862E^{-2}$</td>
<td>1.002</td>
<td>1.056</td>
</tr>
<tr>
<td>4</td>
<td>$2.665E^{-2}$</td>
<td>$2.670E^{-2}$</td>
<td>$2.866E^{-2}$</td>
<td>1.002</td>
<td>1.075</td>
</tr>
<tr>
<td>5</td>
<td>$1.320E^{-2}$</td>
<td>$1.329E^{-2}$</td>
<td>$1.453E^{-2}$</td>
<td>1.007</td>
<td>1.101</td>
</tr>
<tr>
<td>6</td>
<td>$7.424E^{-3}$</td>
<td>$7.476E^{-3}$</td>
<td>$8.329E^{-3}$</td>
<td>1.007</td>
<td>1.122</td>
</tr>
</tbody>
</table>
Table 6: The maximum displacement values for the solid and shell models and the ratio between them when loaded with a transverse load $N$. The thickness ratio between the two parts of the corner is constant, i.e. $t_1 = t_2 = t$ which can be seen in figure 17c. $t$=thickness, S=Solid, M=Midsurface and If=Interface.

<table>
<thead>
<tr>
<th>t [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$7.852E^{-1}$</td>
<td>$7.827E^{-1}$</td>
<td>$8.068E^{-1}$</td>
<td>0.997</td>
<td>1.028</td>
</tr>
<tr>
<td>2</td>
<td>$9.579E^{-2}$</td>
<td>$9.505E^{-2}$</td>
<td>$1.011E^{-1}$</td>
<td>0.992</td>
<td>1.055</td>
</tr>
<tr>
<td>3</td>
<td>$2.771E^{-2}$</td>
<td>$2.737E^{-2}$</td>
<td>$3.005E^{-2}$</td>
<td>0.988</td>
<td>1.084</td>
</tr>
<tr>
<td>4</td>
<td>$1.142E^{-2}$</td>
<td>$1.122E^{-2}$</td>
<td>$1.272E^{-2}$</td>
<td>0.982</td>
<td>1.114</td>
</tr>
<tr>
<td>5</td>
<td>$5.705E^{-3}$</td>
<td>$5.584E^{-3}$</td>
<td>$6.541E^{-3}$</td>
<td>0.979</td>
<td>1.147</td>
</tr>
<tr>
<td>6</td>
<td>$3.227E^{-3}$</td>
<td>$3.141E^{-3}$</td>
<td>$3.803E^{-3}$</td>
<td>0.973</td>
<td>1.178</td>
</tr>
</tbody>
</table>

Figure 35: Deformation for the corner when the plates have the same thickness (five millimeters) and are loaded in the plane. The deformation is scaled with a factor 1000.

The stress field for $\sigma_{V_M}$ is approximately the same for all models looking at all load cases, as shown in figures 36a to 36c. This holds for all thicknesses and the only big difference between the models occur at the loaded part of the plate when loaded with an in-plane load, see figure 36d. The loads that are applied to the model can be found in figure 17c and it is the stresses for these three loads $(T, M, N)$ that are depicted below.
Figure 36: Overall $\sigma_{vM}$ for (a) a transverse load $T$ and (b) a bending moment $M$. The thickness of the plates is one millimeter and the loads are: $T = N = 10$ N and $M = 10$ Nmm.
Figure 36: (c) Overall $\sigma_{vM}$ for an in-plane load $N$. (d) $vM$ stress in the middle of the loaded plate when loaded with a tensile in-plane load $N^6$. The thickness of the plates is one millimeter and the loads are: $T = N = 10$ N and $M = 10$ Nmm.

Since the stresses are very similar, it is of most interest to look at how much bigger (or smaller) the values from the shell models are compared to the solid. This comparison is done for the largest in-plane stresses (considering the magnitude) $\sigma_y$ and $\sigma_z$ and is presented in figure 37 for a transverse load $T$. These are very similar to the results for the loading being a bending moment. The stresses are extracted from the slit in the middle of either the loaded or the fixed plate, see figure 33. These slits are five millimeters wide, located in the middle of the inside surface of each plate and stretches over the entire plate width.

---

$^6$The scales are set to be the same. But only the second largest/smallest values can be set in HyperView, which is why the scale is only the same in-between these values.
Figure 37: Error in stress relative to the solid model. The load is a transverse load $T$ of ten Newton. Error for the (a) max $\sigma_y$ value in the loaded plate. (b) max $\sigma_z$ value in the fixed plate.

There are more differences between the models of the corner when the loading is an in-plane load $N$. This is depicted, for both the fixed and loaded plate, in figure 38 for the biggest in-plane stresses (considering the magnitude) $\sigma_y$ and $\sigma_z$. The rest of the relative stress errors, including the ones from a bending moment, can be found in Appendix B section B1.

Figure 38: Stress errors relative to the solid model in (a) $\sigma_y$ in the loaded plate and (b) $\sigma_z$ in the fixed plate. The load is a tensile in-plane load $N$ of ten Newton.

Considering the errors for the fixed plate (figure 38b), one might also want to investigate how the difference between the error curves for the interface and the midsurface models changes with increasing thickness. This is shown for the minimum value of $\sigma_z$ in figure 39, and the tendencies are the same for both $\sigma_x$ and $\sigma_{yM}$.

Figure 39: The difference in relative error for the minimum $\sigma_z$ in the fixed plate for a tensile in-plane load $N$ of ten Newton.
4.2.2 Different thickness ratio

The overall displacement magnitudes for this case are found in table 7 for a transverse load $T$ and table 8 for an in-plane load $N$. These are retrieved in the same manner as for the case of the plates having the same thickness. The deformation for the case of the thin plate being loaded can be found in figure 40 for a transverse load $T$ and figure 41 for an in-plane load $N$. The thicker plate is seven millimeters thick in both of these figures.

Table 7: The maximum displacement values for the solid and shell models and the ratio between them when loaded with a transverse load $T$. The thickness ratio between the two parts of the corner is not constant. $t_1=$thickness of the thicker plate, $S=$Solid, $M=$Midsurface and If=Interface. The thickness that does not vary is $t_2 = 1 \text{ mm}$ and both thicknesses can be found in figure 17d.

<table>
<thead>
<tr>
<th>$t_1$ [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2</td>
<td>1.483</td>
<td>1.489</td>
<td>1.527</td>
<td>1.004</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>1.425</td>
<td>1.436</td>
<td>1.491</td>
<td>1.008</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>1.393</td>
<td>1.409</td>
<td>1.482</td>
<td>1.011</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>1.365</td>
<td>1.388</td>
<td>1.479</td>
<td>1.017</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>1.338</td>
<td>1.369</td>
<td>1.477</td>
<td>1.023</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>1.312</td>
<td>1.351</td>
<td>1.477</td>
<td>1.030</td>
</tr>
</tbody>
</table>

Maximum displacement magnitude when loaded at the thick edge. $t_2=1 \text{ mm}$.

<table>
<thead>
<tr>
<th>$t_1$ [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2</td>
<td>$5.758E^{-1}$</td>
<td>$5.912E^{-1}$</td>
<td>$6.205E^{-1}$</td>
<td>1.027</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>$4.335E^{-1}$</td>
<td>$4.598E^{-1}$</td>
<td>$5.003E^{-1}$</td>
<td>1.061</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>$3.882E^{-1}$</td>
<td>$4.193E^{-1}$</td>
<td>$4.721E^{-1}$</td>
<td>1.097</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>$3.497E^{-1}$</td>
<td>$3.972E^{-1}$</td>
<td>$4.622E^{-1}$</td>
<td>1.136</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>$3.236E^{-1}$</td>
<td>$3.809E^{-1}$</td>
<td>$4.579E^{-1}$</td>
<td>1.177</td>
</tr>
<tr>
<td></td>
<td>7</td>
<td>$3.006E^{-1}$</td>
<td>$3.668E^{-1}$</td>
<td>$4.558E^{-1}$</td>
<td>1.220</td>
</tr>
</tbody>
</table>

37
Figure 40: Deformation for the corner when the adjacent plates have different thickness (one and seven millimeters) and the thin one is loaded with a transverse load. The deformation is scaled with a factor 20.

Table 8: The maximum displacement values for the solid and shell models and the ratio between them when loaded with an in-plane load $N$. The thickness ratio between the two parts of the corner is not constant. $t_1=$thickness of the thicker plate, S=Solid, M=Midsurface and If=Interface. The thickness that does not vary is $t_2 = 1 \text{ mm}$ and both thicknesses can be found in figure 17d.

<table>
<thead>
<tr>
<th>$t_1$ [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>7.688E-1</td>
<td>7.639E-1</td>
<td>8.022E-1</td>
<td>0.994</td>
<td>1.043</td>
</tr>
<tr>
<td>3</td>
<td>7.536E-1</td>
<td>7.458E-1</td>
<td>8.014E-1</td>
<td>0.990</td>
<td>1.063</td>
</tr>
<tr>
<td>4</td>
<td>7.387E-1</td>
<td>7.280E-1</td>
<td>8.011E-1</td>
<td>0.986</td>
<td>1.084</td>
</tr>
<tr>
<td>5</td>
<td>7.237E-1</td>
<td>7.105E-1</td>
<td>8.010E-1</td>
<td>0.982</td>
<td>1.107</td>
</tr>
<tr>
<td>6</td>
<td>7.084E-1</td>
<td>6.933E-1</td>
<td>8.009E-1</td>
<td>0.979</td>
<td>1.131</td>
</tr>
<tr>
<td>7</td>
<td>6.934E-1</td>
<td>6.764E-1</td>
<td>8.009E-1</td>
<td>0.975</td>
<td>1.155</td>
</tr>
</tbody>
</table>

Maximum displacement magnitude when loaded at the thick edge. $t_2=1 \text{ mm}$.

<table>
<thead>
<tr>
<th>$t_1$ [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>9.763E-2</td>
<td>9.741E-2</td>
<td>1.056E-1</td>
<td>0.998</td>
<td>1.082</td>
</tr>
<tr>
<td>3</td>
<td>2.884E-2</td>
<td>2.875E-2</td>
<td>3.543E-2</td>
<td>0.997</td>
<td>1.229</td>
</tr>
<tr>
<td>4</td>
<td>1.215E-2</td>
<td>1.210E-2</td>
<td>1.856E-2</td>
<td>0.996</td>
<td>1.528</td>
</tr>
<tr>
<td>5</td>
<td>6.233E-3</td>
<td>6.182E-3</td>
<td>1.269E-2</td>
<td>0.992</td>
<td>2.036</td>
</tr>
<tr>
<td>6</td>
<td>3.625E-3</td>
<td>3.575E-3</td>
<td>1.017E-2</td>
<td>0.986</td>
<td>2.806</td>
</tr>
<tr>
<td>7</td>
<td>2.302E-3</td>
<td>2.253E-3</td>
<td>8.925E-3</td>
<td>0.979</td>
<td>3.877</td>
</tr>
</tbody>
</table>
Figure 41: Deformation of the corner when the adjacent plates have different thickness (one and seven millimeters) and the thin edge is loaded in the plane. The deformation is scaled with a factor 1000.

The load may also be applied in the midsurface and connected to the nodal plane using RBE3 elements. In addition, the fixed end is fixed in the midsurface, and connected to the nodal plane using RBE2 elements. This is done for the load being applied at the thin edge and the thicker plate being seven millimeters thick for both a transverse load $T$ and an in-plane load $N$. An illustration is found for the in-plane load in figure 42. The results are found in table 9. Here one also finds the ratio between the results from loading in the midsurface and in the nodal plane for the two shell models. The deformation plot for the in-plane load applied in the midsurface is found in figure 43.

Figure 42: Illustration of how the loads are applied in the midsurface using RBE3 elements. The fixed nodes are connected to the midsurface using RBE2 elements.
Table 9: The maximum displacement values for the solid and shell models and the ratio between them when loaded with either a transverse load $T$ or an in-plane load $N$ at the thin edge ($t_2 = 1$ mm). The thicker plate is seven millimeters thick ($t_1 = 7$ mm) and the loads are applied in the midsurface. $t_1$=thickness of the thicker plate, S=Solid, M=Midsurface and If=Interface. The thicknesses and the loads can be found in figure 17d. The ratio between these displacements and the ones from when the plate is loaded in the nodal plane, found in tables 7 and 8 here here denoted with the subscript nodal, is also presented.

<table>
<thead>
<tr>
<th>$t_1$ [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
<th>M/M$_{nodal}$</th>
<th>If/If$_{nodal}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>3.010E$^{-1}$</td>
<td>3.778E$^{-1}$</td>
<td>4.565E$^{-1}$</td>
<td>1.255</td>
<td>1.517</td>
<td>1.030</td>
<td>1.002</td>
</tr>
</tbody>
</table>

Maximum displacement magnitude when loaded in the midsurface with $N$, $t_2 = 1$ mm.

<table>
<thead>
<tr>
<th>$t_1$ [mm]</th>
<th>S [mm]</th>
<th>M [mm]</th>
<th>If [mm]</th>
<th>M/S</th>
<th>If/S</th>
<th>M/M$_{nodal}$</th>
<th>If/If$_{nodal}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>2.302E$^{-3}$</td>
<td>2.253E$^{-3}$</td>
<td>2.269E$^{-3}$</td>
<td>0.979</td>
<td>0.986</td>
<td>1.000</td>
<td>0.254</td>
</tr>
</tbody>
</table>

Figure 43: The deformation for the corner when loaded in the plane at the thin edge. The load is applied in the midsurface and connected to the nodes using RBE3 elements. The fixed nodes are in the midsurface and connected to the nodes using RBE2 elements. The thick plate is seven millimeters thick and the deformation is scaled with a factor 1000.

The stress fields for this case do behave as the ones for the case of the same thickness ratio. The errors relative to the solid model, in the middle of the plates, are once again plotted. For the case of the load acting on the thick edge, the errors for the in-plane stresses ($\sigma_x$ and $\sigma_z$) are depicted in figure 44 for a transverse load $T$ and figure 45 for an in-plane load $N$. The rest of the relative stress errors, including the ones from the bending moment, are found in Appendix B section B2.
Figure 44: Error in stress relative to the solid model. The load is a transverse load $T$ of ten Newton applied at the thick edge. (a) Error in the max $\sigma_y$ in the thick plate. (b) Error in the max $\sigma_z$ in the thin plate.

Figure 45: Error in stress relative to the solid model. The load is an in-plane load $N$ of ten Newton applied at the thick edge. Error for the (a) max $\sigma_y$ value in the thick plate, (b) min $\sigma_z$ value in the thin plate.

The relative stress errors may also be plotted for the case of the loading being applied at the thin edge. These errors are depicted for the in-plane stresses ($\sigma_y$ and $\sigma_z$) in figure 46 for a transverse load $T$ and figure 47 for an in-plane load $N$. The rest of the relative stress errors can be found in Appendix B section B2. Note that the relative errors for the transverse load $T$ is about the same regardless of which plate that is loaded.

Figure 46: Error in stress relative to the solid model. The load is a transverse load $T$ of ten Newton applied at the thin edge. Error for the (a) max $\sigma_y$ value in the thick plate, (b) max $\sigma_z$ value in the thin plate.
4.3 C-bar

For the C-bar a local coordinate system is used when the data is evaluated. This is a Cartesian system and it is located and oriented according to figure 48. The nodal stresses are calculated by use of the advanced averaging method within HyperView.

Figure 48: The local coordinate system used when evaluating displacements and stresses in the C-bar.

Table 10: The maximum displacement values for the solid and shell model and the ratio between them.

<table>
<thead>
<tr>
<th>Displacement</th>
<th>Solid [mm]</th>
<th>Shell [mm]</th>
<th>Shell/Solid</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-direction</td>
<td>1.254</td>
<td>1.785</td>
<td>1.42</td>
</tr>
<tr>
<td>Y-direction</td>
<td>2.569</td>
<td>1.837</td>
<td>0.72</td>
</tr>
<tr>
<td>Z-direction</td>
<td>-6.024</td>
<td>-7.891</td>
<td>1.31</td>
</tr>
<tr>
<td>Mag $\sqrt{X^2 + Y^2 + Z^2}$</td>
<td>6.362</td>
<td>7.969</td>
<td>1.25</td>
</tr>
</tbody>
</table>
Figure 49: The displacement magnitudes and the deformed C-bar. The deformation is scaled with a factor 15. The upper picture is of the solid model and the lower is of the shell model. The scale of the displacement is the same for both models.

Figure 50: The vM stress field for the entire C-bar. The scales are the same for the two figures\(^7\).

\(^7\)The scales are set to be the same. But only the second largest/smallest values can be set in HyperView, which is why the scale is only the same in-between these values.
In order to get a better view of the stress field in the structure, it is plotted some distance away from the stress concentrations. In order to be able to capture the eventual sign differences, \( \sigma_x \) is the stress that is plotted. This is depicted in figure 51. Here two areas where the stress differences between the models are investigated are also displayed. The stresses in the yellow area are depicted in figure 72 and the ones from the red area in figure 73, which can both be found in Appendix C. These stresses are also found in table 11.

**Figure 51:** The \( \sigma_x \) stress field for the C-bar some distance away from the stress concentrations that arise due to the fixed nodes. The red and yellow areas are selected for investigating the difference between the two models.

**Table 11:** Stresses at different points in the C-bar for \( \sigma_x \) in both the solid and shell model. The ratio between the stresses and the thickness of the part where they are extracted are also listed.

<table>
<thead>
<tr>
<th>Stress</th>
<th>Solid</th>
<th>Shell</th>
<th>Shell/Solid</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>-182.272 MPa</td>
<td>-186.080 MPa</td>
<td>1.02</td>
<td>5 mm</td>
</tr>
<tr>
<td>9</td>
<td>-105.915 MPa</td>
<td>-107.397 MPa</td>
<td>1.01</td>
<td>9 mm</td>
</tr>
<tr>
<td>11</td>
<td>-113.524 MPa</td>
<td>-115.145 MPa</td>
<td>1.01</td>
<td>11 mm</td>
</tr>
<tr>
<td>14</td>
<td>-82.0952 MPa</td>
<td>-82.2222 MPa</td>
<td>1.00</td>
<td>14 mm</td>
</tr>
<tr>
<td>9_1</td>
<td>-123.990 MPa</td>
<td>-127.353 MPa</td>
<td>1.03</td>
<td>9 mm</td>
</tr>
<tr>
<td>9_2</td>
<td>-149.939 MPa</td>
<td>-155.164 MPa</td>
<td>1.03</td>
<td>9 mm</td>
</tr>
<tr>
<td>10</td>
<td>-154.683 MPa</td>
<td>-141.335 MPa</td>
<td>0.91</td>
<td>10 mm</td>
</tr>
</tbody>
</table>
As the stresses are in great agreement between the both models away from the constrained edge, and the deformation is very large there (figure 49) it is reasonable to assume that the big difference in deflection is due to the constraints. It might be the case that to few nodes are fixed in the shell model in order for it to behave as the solid. To check this all nodes along the back end of the C-bar were fixed, see figure 52. Note that this does not bear any resemblance to the actual BC:s used by Saab for the entire C-bar. The displacement and deformation is once again plotted and is found in figures 53. The displacements and ratios between them are found in table 12.

Figure 52: Illustration of how all nodes are locked at the back end when investigating the effects of the constraints on the overall behavior of the C-bar.

Figure 53: The deformation for the C-bar when completely fixed at the back end. The top picture is of the solid model and the bottom one is of the shell model.
Table 12: Maximum displacements for the completely fixed C-bar (both solid and shell model) and the ratio between them.

<table>
<thead>
<tr>
<th>Displacement</th>
<th>Solid [mm]</th>
<th>Shell [mm]</th>
<th>Shell/Solid</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-direction</td>
<td>−7.403E−1</td>
<td>−7.011E−1</td>
<td>0.95</td>
</tr>
<tr>
<td>Y-direction</td>
<td>3.438</td>
<td>3.572</td>
<td>1.04</td>
</tr>
<tr>
<td>Z-direction</td>
<td>−3.524</td>
<td>−3.705</td>
<td>1.05</td>
</tr>
<tr>
<td>Mag $\sqrt{X^2 + Y^2 + Z^2}$</td>
<td>4.786</td>
<td>4.936</td>
<td>1.03</td>
</tr>
</tbody>
</table>

This indicates that the error is due to the fastening of the C-bar rather than from the use of offset modeling.

The vM stress field may also be plotted and this is found in figure 54. Furthermore, the stresses in the X-direction are evaluated in the same points as before, and the results from this is found in table 13. These points can be seen in figures 74 and 75 in Appendix C.

![Figure 54: The vM stress field for the entire C-bar, when the back end is completely fixed. The scales are the same for the two figures.](image)

8The scales are set to be the same. But only the second largest/smallest values can be set in HyperView, which is why the scale is only the same in-between these values.
Table 13: Stresses at different points in the C-bar for $\sigma_x$ in both the solid and shell model, when the back end is fixed. The ratio between the stresses and the thickness of the part where they are extracted are also listed.

| Stress 5 | -178.406 MPa | -179.595 MPa | 1.01 | 5 mm |
| Stress 9 | -107.048 MPa | -104.212 MPa | 0.97 | 9 mm |
| Stress 11 | -109.888 MPa | -110.84 MPa | 1.01 | 11 mm |
| Stress 14 | -80.4275 MPa | -79.8649 MPa | 0.99 | 14 mm |
| Stress 9_1 | -133.922 MPa | -134.904 MPa | 1.01 | 9 mm |
| Stress 9_2 | -165.831 MPa | -167.268 MPa | 1.01 | 9 mm |
| Stress 10 | -179.532 MPa | -171.841 MPa | 0.96 | 10 mm |

5 Discussion

In this section the results will be discussed. These will act as a base on which the conclusions in section 6 are made.

5.1 Thickness variation case

Investigating the overall stress behavior, one concludes that the stress fields are exactly the same for all shell models subject to either a transverse load $T$, or a bending moment $M$. This holds for both the case of the load acting on the thin and the thick edge, and the stress fields are also in fairly good agreement with that of the solid model. The reason for this is that a transformation of these loads, which is done when offset is used, will not result in any load eccentricity. This as there is no lever between the force and the midsurface.

However, the case of an in-plane load results in differences in the stress fields for all shell models, when loaded at the thick edge. When the load is applied at the thin edge, the model using a common midsurface, and the model using individual midsurfaces connected by rigid links, give the same result. However, the one with an offset from the common interface surface (the bottom surface) differs. The reason for this is the load eccentricity created by using the offset, see figure 10.

Also noticeable is that the magnitude of the in-plane stress ($\sigma_y$) for the models using offset is about the same when loaded at the thin edge as they are when loaded at the thick edge, see e.g. figures 30 and 32. This indicates that the equilibrium for the plate might be thought of as being performed in the nodal plane. These nodal forces are then transformed to the midsurface by use of offset, illustrated in figure 55a. Meaning that the stresses will be approximately the same for both these load cases. That the offset works as in figure 55 is supported by looking at the deformation when the plate is loaded at the thick plate (figure 26). This also explains why there are sign differences in the stresses in the thin plate when the thick one is loaded. The offset models does not take into account the moment that occurs due to the thickness change (figure 55b), but just the moment that occurs due to the offset (figure 55a). This indicates that the models will deform differently which is the reason for the difference in stress. This will be discussed further below.
Figure 55: (a) Illustration of how the loads are handled when using offset models, in this case an offset from a common interface surface. (b) Illustration of how the load is increased at the transition due to the presence of rigid RBE2 elements.

The load eccentricity will cause quite severe errors. As the in-plane force is transformed to the mid-surface, a bending moment occurs. Consequently, the different models are not subject to the same load case. This results in that the top and bottom surface of the offset shells will not both be in tension/compression. This is supported by figure 31b, where the offset models are in tension and the others are in compression. This sign difference in the offset shells can be shown mathematically by using equation (5) and the forces in the shells offset from an interface surface, depicted in figure 55a. From equation (5), one has:

\[ \sigma_y = \frac{N_y}{t} + \frac{12M_y z}{t^3} \]  

(27)

And from figure 55a, one has:

\[ M_y = -\frac{N_y t}{2} \]  

(28)

Since \( z \) indicates where in the thickness direction of the plate the stress is evaluated, the extreme values are \( z = \pm t/2 \). A positive value of \( z \) corresponds to the top half of the shell and a negative value means the bottom half of the shell. Combining this with equation (27) and (28) gives:

\[ \sigma_y = \frac{N_y}{t} \pm \frac{3N_y}{t} \]  

(29)

or

\[
\begin{align*}
\sigma_{y Top} &= -\frac{2N_y}{t} \\
\sigma_{y Bottom} &= \frac{4N_y}{t}
\end{align*}
\]  

(30)
There is however one case where all models agree well in the bottom of the plate, which is when the in-plane load is applied at the thin edge and the stress is evaluated in the thick plate, figure 30a. An explanation can be found in figure 56, which is easily derived from figure 55. This illustrates, for each plate, if it is subject to tension (T) or tension and bending (T+B) for the midsurface (and solid) model and the fully offset model (modeled in the interface surface). It also shows how the plates will bend, if they do. Note that, how the model with offset from a common midsurface behaves can be determined by combining the two cases found in figure 56. That this is the case is supported by figure 26.

![Figure 56: Illustration of how each part of the plate is subject to tension (T) or tension and bending (T+B). Also the direction of the bending is illustrated. (a) Midsurface (or solid) model loaded at the thick edge. (b) Midsurface (or solid) model loaded at the thin edge. (c) Offset model (from interface surface) loaded at the thick edge. (d) Offset model (from interface surface) loaded at the thin edge.](image)

This explains, together with figure 55, why the results from the common midsurface model and the rigid link model are in agreement for the thick plate, when loaded at the thin edge. For this case, the force and moment will be exactly the same in both models, as the length of the rigid links are the same as the offset value.

This also explains the big difference in the overall stiffness of the structure one notice if investigating the displacement magnitudes. The biggest differences occur due to a significant difference in the bending moment that acts on the thinner plate. This since the bending moment will give rise to greater deformations than the tension load. This can be fixed by applying the load at the midsurface, using RBE3 elements, as concluded from table 3 and figure 28. By doing this, the moment from the offset is canceled out by the moment from the applied load, as illustrated in figure 57.

![Figure 57: Moments that arise due to both RBE3 elements and offset of shell elements.](image)
Studying figure 30a, it is clear that the error in stress between the offset and the solid model decreases as the thickness of the plate increases. This can be explained by equation (30) stated above. Here it is shown that the stress behaves like $1/t$ at the bottom of the plate (if $N_y$ is constant). Consequently, as the thickness increases the stress will converge towards this value. Thus, as the thickness increases the difference in stress between the offset shell model and the solid decreases. This can also be seen in figure 30b. However, one must keep in mind that the sign of the stress is only correct here since the stress is evaluated at the bottom of the plate. Additionally, the overall displacement behavior does not converge towards a common value.

Investigating the computational aspects of the different shell models (table 4), one finds that the offset models are more demanding than the midsurface model using rigid links. The reason for this is that MPC:s are created for all elements with offset. This means that for the interface model every node has a MPC associated to it. In the common midsurface only the nodes in the thick plate has these MPC:s and in the rigid link model they are only present at the transition nodes. That is, the nodes where the rigid links, which works as MPC:s, are found. This in term means that offset modeling might not be to prefer if one has a fine mesh, i.e. many MPC:s are created. But one should also keep in mind that the use of offset will speed up the pre-processing, and the time one makes up in this stage might be greater than the time lost during computations.

5.2 90° corner cases

To begin with, one can conclude that the stress fields for the different shell models agree rather well with the stress field for the solid, except for the in-plane stress ($\sigma_y$ or $\sigma_z$) in the case of the in-plane loaded offset plate, c.f. figures 36, 38, 45 and 47. This is due to the bending moment that occur when using an offset, which was discussed in the previous section.

Observing the loads that give little error, one may conclude that both the bending moment $M$ and the transverse load $T$ give approximately the same error relative to the solid model for both shell models. Some minor differences in the magnitude of the error exists but the behavior remain the same.

For the case with the plate with changing thickness ratio, and where the thinner part of the plate was loaded, there were relatively big differences in the overall stiffness. This can be observed in the displacement results for this case, table 7 and 8. But if the plates have the same thickness there is not much difference between the offset and solid model in the overall displacement.

5.2.1 90° corner subject to a transverse force or bending moment

The errors relative to the solid model are about the same for both shell models, when considering the in-plane stresses ($\sigma_y$ and $\sigma_z$) and the vM stress, as indicated (for example) in figures 37, 63 and 64.

This can seem peculiar, at least for the transverse load $T$. This since one expects the load to be lower in the non-loaded part, when looking at the inside of the plate, as a result of the offset, see figure 58. The reason that this is not seen is most likely due to the fact that the moment from the transverse force, $M_T$ in figure 58, is much greater than the moment from the offset, $M_{ZOFFS}$ in figure 58. If the length of the plates were made shorter, the effect from the offset would, with most certainty, be more pronounced.
The moment that occurs due to the offset does however have a greater influence on the overall stiffness of the structure, when considering the displacement magnitude. This extra bending moment is likely the cause for the displacements being larger for the offset model. This also explains the increased difference between the offset and solid model with increasing thickness, since the moment will increase with thickness. Therefore, the increase in stiffness, due to the thickness increase, is less pronounced in the offset model than in the solid model.

Considering $\sigma_x$, one notices an increase in stiffness, which is greater for the offset model, both for the case of the plates having the same and different thickness. Worth noting is that this increase in stiffness seems to be greater when one plate is thinner than the other. The greatest stiffening effect in the X direction occurs in the offset model in the thick plate, when this is loaded with a transverse force $T$. This case is depicted in figure 66c, and it shows a maximum stress decrease of about 45% relative to the solid model.

![Figure 58](image)

For the shear stress it is not so clear what to expect since both loads may give rise to both increase and decrease in stiffness. However, the midsurface model is closer to the solid model, considering all load cases, if one seeks to minimize the underestimation of the stress values.

### 5.2.2 90° corner subject to an in-plane load

This load case gives much greater errors than the previous two, which is to be expected from the investigation of the thickness variation. The greatest difference is noted for the in-plane stress ($\sigma_y$ or $\sigma_z$) in the loaded plate for the offset model. Here the stress has the wrong sign compared to the solid and this is due to the load eccentricity that occurs when offsets are used, see previous sections. This also means that these simulations do not represent the same problem for the different models. Furthermore, it is worth noting that the midsurface model agrees well with the solid model for all stresses.

Investigating the stresses in the non loaded plate, one notices a linear increase in stress for all stresses except the shear stress. This indicates that it is due to the load increase from the offsets, as illustrated in figure 59. This is also the reason for the plate appearing weaker when modeled using offset. The increases in load will lead to increased deformation, see figure 41. However, the deformation can be reduced by applying the load in the midsurface of the shell, as shown in table 9 and figure 43.
Figure 59: The moments that arise due to offset from the interface surface and how they are transmitted in the corner, when an in-plane force is applied at one end.

For the same thickness, the loaded plate modeled with offset, shows a noticeable increase in $\sigma_x$ and $\tau_{xy}$ relative to the solid model. However, this is not the case when the thickness differs in the corner. Here, the stress decreases relative to the solid model, for both the midsurface and the offset model. The offset model does however give a little bit higher estimation of the stress than the midsurface model.

For the shear stress it is harder to find any pattern. But as before, the midsurface model seems to agree best, all cases considered. Keeping in mind that the model should not give an underestimation of the stresses.

An interesting thing to notice is that the moment from the offsets here are transferred from one plate to the other. This was not the case for the thickness variation case and this is due to the fact that in the corner, the sixth d.o.f. is present. This acts as to give the correct stiffness in the corner by coupling d.o.f. from the two adjacent plates. This enables the bending moment to be transferred.

5.3 C-bar models

By observing the overall behavior, one notices that the solid model is 20-30% stiffer than the shell model (considering the displacement magnitude). This is not strange considering that the shell model is subject to larger deformations at the supports, as indicated by the 87% larger vM stress there (figure 50), and larger local deformations in the shell model (figure 49). This is not due to the use of offset but rather the result from the shells being a lot weaker than the solids for these supports (figure 22). It is the shell assumption rather than the offset that is the problem. This is supported by the fact that the far field stresses are in good agreement between the two models and that they will give approximately the same deflection (and stresses as seen in table 13) if the entire back end of the C-bar is locked (figure 53). This makes the entire structure stiffer, which solves the problem with locally large deflections at the fixed nodes. Which in turn means that the source of the error in the original model is eliminated. A better way to handle the fact that the shells are not good for the case of just a few points being fixed, might be to use solid elements at the back end of the C-bar and shells for the rest. This is something that might be worth looking in to in the future.

Investigating the $\sigma_x$ stress field some distance away from the supports, it is concluded that the shell model and the solid model correlate well. It is clear, by examining table 11, that the differences between the solid and shell model decrease, with increasing shell thickness. However, what is a bit unexpected is that the difference seems to be even smaller than for the simpler cases (thickness variation and 90° corner). There are two possible explanations for this.
One is that the stress from the applied bending force is so big that the extra contribution from the offset gives very little effect on the overall stress. This can also be seen for the 90° corner case, when investigating the non-loaded plate. This would be more true farther away from the force, as the lever from the force is larger then. The other reason is that the force give rise to a bending moment, since it is applied a distance of eleven millimeters from the bottom surface of the C-bar, which is in the midsurface of the adjacent shell. This moment would counteract the moment that occurs due to the offset, as discussed previously. A combination of these two explanations is probably the reason for the small difference.

6 Conclusions

Offset shell elements could very well be used in the modeling of components, if one carefully considers how the force is applied. Whether there is no in-plane force or there is an in-plane force applied at a distance equal to the offset from the nodal plane, I would say that it would give little difference compared to a solid model. This can be seen in figures 28, 51 and 53, the last two of which are also given again below. From this, one may draw the conclusion that if the load is applied to a fixed point in space the error from the offset would be small. This would mean that a solid, midsurface shell and offset shell model would be subject to the same loading, which may not be the case if the load is applied directly in the nodal plane.

Figure 53: The deformation for the C-bar when completely fixed in the back end. The top picture is of the solid model and the bottom one is of the shell model.

If the load is an in-plane bending load applied in the nodal plane, I would expect the stress errors to be significant up to a distance a couple of times larger than the offset value away from the load. After that the bending due to the lever from the applied load will result in the creation of a bigger moment than that from the offset. Meaning that the deformation from the offset is little in comparison to the stress from the applied load. This would mean that the stresses would be good for the offset shell model. If the load is an in-plane compressive/tensile load applied in the nodal plane however, the bending moment from the offset would give stress and displacement errors throughout the structure as can be seen in figure 26, which is given again below.
Figure 51: The $\sigma_x$ stress field for the C-bar a bit away from the stress concentrations that arise due to the fixed nodes. The red and yellow areas are selected for investigating the difference between the two models.

Figure 26: Deformation of the plate when subject to an in-plane tensile load of ten Newton at the eleven millimeter thick edge. The deformation has been scaled up by a factor 100 in order to see a difference. The upper left picture is of the solid, the upper right of midsurface shell. The bottom left has an offset from a common midsurface and the bottom right one has an offset from the bottom of the plate.
The thing to remember is that offset moves the midsurface of the element. This means that loads applied in the nodal plane is not in the same geometrical place for an offset shell and a non-offset shell. A shell offset half of its thickness would be loaded in the top or bottom rather than in the middle when applying the load in the nodal plane. To avoid this complication it is good to define a point in space where the load is applied. In that case it would always be in the same position relative to the shell no matter how this is modeled.

Worth keeping in mind is that, the use of offset modeling will cause longer computational times compared to midsurface modeling (even if using RBE2 elements to connect the midsurfaces). This is due to the fact that a lot of MPC:s are created when using offset. This might be a problem if a very fine mesh is used. Although, the time lost during computations will most certainly be made up by a lot faster pre-processing, especially if the geometry is complex.

Recommendations for the use of the offset function can be summarized according to table 14. Note that, an experienced user is here defined as one who understands the complications that arise due to the offset function, and a novice user is anyone that knows how to use a FE-program. Following these recommendations will result in little error due to the use of offset. It should be noted that if the loads are always defined in a point in space relative to the solid model (CAD model), as is mostly done at Saab, even a novice user should be able to use offset without any major complications.

**Table 14:** Recommendations for when to use different modeling techniques. An experienced user knows what happens when the offset function is used, and a novice user does not. The midsurface modeling includes connecting non adjacent nodes at thickness variations with RBE2 elements. If offset is used it is recommended that the elements be modeled in an existing interface surface, as this saves time.

<table>
<thead>
<tr>
<th></th>
<th>Experienced user</th>
<th>Novice user</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Midsurface</td>
<td>Offset</td>
</tr>
<tr>
<td>Fast computation (many elements)</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Fast pre-processing (complex geometry)</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>Linear analysis</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Non-linear analysis</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Buckling analysis</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Dynamic analysis</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Pressure loads</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Thermal loads</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Gravitational loads</td>
<td>Yes</td>
<td>No</td>
</tr>
</tbody>
</table>
7 Further work

As a continuation of this investigation of the offset function, I would recommend investigating the effects of fasteners. This in order to learn if they will cause other complications than the ones that arise from just the offset, as investigated here. This is crucial in order to be able to use offset modeling in a real application, where fasteners are often present.

Another thing that might be interesting to look at is the effect of using offset modeling in combination with the use of solid elements. This could be helpful in order to just use solids at places where shells does not work satisfactory, for instance at the back of the C-bar which is fixed at ten points (figure 21b).

Furthermore it might be interesting to investigate how offset modeling will affect composites rather than isotropic materials (which was the focus in this work).
Appendix A: Overall stress results for the thickness variation
Figure 60: $\sigma_{vM}$ at the bottom of the plate when loaded with (a) a transverse force $T$, (b) a bending moment $M$ and (c) an in-plane load $N$. All loads are applied at the thick edge of the plate which is six millimeters thick. The loads are ten Newton and the moment is ten Newton-millimeter.
Figure 61: $\sigma_{vM}$ at the bottom of the plate when loaded with (a) a transverse force $T$, (b) a bending moment $M$, (c) an in-plane load $N$. All loads are applied at the thin edge of the plate and the thicker part is two millimeters thick. The loads are ten Newton and the moment is ten Newton-millimeter.

59
Figure 62: $\sigma_{vM}$ at the bottom of the plate when loaded with (a) a transverse force $T$, (b) a bending moment $M$, (c) an in-plane load $N$. All loads are applied at the thin edge of the plate and the thicker part is six millimeters thick. The loads are ten Newton and the moment is ten Newton-millimeter.
Appendix B: Results from the 90° corner simulations

B1: Corner with same thickness of the plates

Figure 63: Error in stress relative to the solid model for the case of the plates having the same thickness. The load is a transverse load $T$ of ten Newton. (a) Error in the max $\sigma_{vM}$ in the loaded plate. (b) Error in the max $\sigma_{vM}$ in the fixed plate. (c) Error in the max $\sigma_x$ in the loaded plate. (d) Error in max $\sigma_x$ in the fixed plate. (e) Error in the max $\tau_{xy}$ in the loaded plate. (f) Error in max $\tau_{zx}$ in the fixed plate.
Figure 64: Error in stress relative to the solid model for the case of the plates having the same thickness. The load is a bending moment $M$ of ten Newton-millimeter. (a) Error in the max $\sigma_{vM}$ in the loaded plate. (b) Error in the max $\sigma_{vM}$ in the fixed plate. (c) Error in the max $\sigma_x$ in the loaded plate. (d) Error in max $\sigma_x$ in the fixed plate. (e) Error in max $\sigma_y$ in the loaded plate. (f) Error in max $\sigma_z$ in the fixed plate. (g) Error in max $\tau_{xy}$ in the loaded plate. (h) Error in max $\tau_{zx}$ in the fixed plate.
Figure 65: Error in stress relative to the solid model for the case of the plates having the same thickness. The load is an in-plane load $N$ of ten Newton. (a) Error in the max $\sigma_{vM}$ in the loaded plate. (b) Error in the max $\sigma_{vM}$ in the fixed plate. (c) Error in the min $\sigma_x$ in the loaded plate. (d) Error in min $\sigma_x$ in the fixed plate. (e) Error in max $\tau_{xy}$ in the loaded plate. (f) Error in max $\tau_{zx}$ in the fixed plate.
Figure 66: Error in stress relative to the solid model for the case of the plates having different thickness. The load is a transverse load $T$ of ten Newton applied at the thick edge. (a) Error in the max $\sigma_{vM}$ in the thick plate. (b) Error in the max $\sigma_{vM}$ in the thin plate. (c) Error in the max $\sigma_x$ in the thick plate. (d) Error in max $\sigma_z$ in the thin plate. (e) Error in the max $\tau_{xy}$ in the thick plate. (f) Error in max $\tau_{zx}$ in the thin plate.
Figure 67: Error in stress relative to the solid model for the case of the plates having different thickness. The load is a bending moment $M$ of ten Newton-millimeter applied at the thick edge. (a) Error in the max $\sigma_{\nu M}$ in the thick plate. (b) Error in the max $\sigma_{\nu M}$ in the thin plate. (c) Error in the max $\sigma_y$ in the thick plate. (d) Error in max $\sigma_z$ in the thin plate. (e) Error in max $\sigma_x$ in thick plate. (f) Error in max $\sigma_x$ in thin plate. (g) Error in max $\tau_{xy}$ in thick plate. (h) Error in max $\tau_{zx}$ in thin plate.
Figure 68: Error in stress relative to the solid model for the case of the plates having different thickness. The load is an in-plane load $N$ of ten Newton applied at the thick edge. (a) Error in the max $\sigma_{vM}$ in the thick plate. (b) Error in the max $\sigma_{vM}$ in the thin plate. (c) Error in the min $\sigma_x$ in the thick plate. (d) Error in min $\sigma_x$ in the thin plate. (e) Error in the max $\tau_{xy}$ in the thick plate. (f) Error in max $\tau_{zx}$ in the thin plate.
Figure 69: Error in stress relative to the solid model for the case of the plates having different thickness. The load is a transverse load $T$ of ten Newton applied at the thin edge. (a) Error in the max $\sigma_{vM}$ in the thick plate. (b) Error in the max $\sigma_{vM}$ in the thin plate. (c) Error in the max $\sigma_x$ in the thick plate. (d) Error in max $\sigma_x$ in the thin plate. (e) Error in the max $\tau_{xy}$ in the thick plate. (f) Error in max $\tau_{zx}$ in the thin plate.
Figure 70: Error in stress relative to the solid model for the case of the plates having different thickness. The load is a bending moment $M$ of ten Newton-millimeter applied at the thin edge. (a) Error in the max $\sigma_{vM}$ in the thick plate. (b) Error in the max $\sigma_{vM}$ in the thin plate. (c) Error in the max $\sigma_{y}$ in the thick plate. (d) Error in max $\sigma_{z}$ in the thin plate. (e) Error in max $\sigma_{x}$ in the thick plate. (f) Error in max $\sigma_{x}$ in the thin plate. (g) Error in max $\tau_{xy}$ in the thick plate. (h) Error in max $\tau_{zx}$ in the thin plate.
Figure 71: Error in stress relative to the solid model for the case of the plates having different thickness. The load is an in-plane load \( N \) of ten Newton applied at the thin edge. (a) Error in the max \( \sigma_{vM} \) in the thick plate. (b) Error in the max \( \sigma_{vM} \) in the thin plate. (c) Error in the min \( \sigma_x \) in the thick plate. (d) Error in min \( \sigma_x \) in the thin plate. (e) Error in the max \( \tau_{xy} \) in the thick plate. (f) Error in max \( \tau_{zx} \) in the thin plate.
Appendix C: Results from the C-bar simulations

**Figure 72:** Stresses in the X direction ($\sigma_x$) in the bottom of the C-bar far away from stress concentrations.

**Figure 73:** Stresses in the X direction ($\sigma_x$) in the side of the C-bar far away from stress concentrations.
Figure 74: Stresses in the X direction ($\sigma_x$) in the bottom of the C-bar when the back end is completely fixed.

Figure 75: Stresses in the X direction ($\sigma_x$) in the side of the C-bar when the back end is completely fixed.
Bibliography


