Finite Element Simulation of Manufacturing Process of Chassis Side Members

W. Blom

Supervisor: dr.ir. R.H.J. Peerlings

Eindhoven, 25th April 2005
Table of contents

1 Introduction ................................................................................................................... 2
  1.1 Acknowledgements .................................................................................................... 3
2 Material properties ......................................................................................................... 4
  2.1 Chemical composition ............................................................................................... 4
  2.2 Mechanical properties ............................................................................................ 4
3 Finite element discretisation ......................................................................................... 6
4 C-forming ....................................................................................................................... 7
  4.1 Choices made in Mentat ......................................................................................... 8
5 Bending ......................................................................................................................... 10
  5.1 Choices made in Mentat ......................................................................................... 11
6 Results ........................................................................................................................... 14
  6.1 Results C-forming .................................................................................................. 14
    6.1.1 Normal stress $\sigma_n$ in layer 1 ........................................................................ 14
    6.1.2 Effective plastic strain ....................................................................................... 15
  6.2 Results bending C-profile situation I ....................................................................... 16
    6.2.1 Results layer 1 ................................................................................................. 17
      6.2.1.1 Normal stress $\sigma_n$ in layer 1 ................................................................. 17
      6.2.1.2 Normal stress $\sigma_{\text{pl}}$ in layer 1 ............................................................ 17
      6.2.1.3 Shear stress $\tau_{\text{pl}}$ in layer 1 ................................................................. 18
      6.2.1.4 Equivalent Von Mises stress .................................................................... 19
      6.2.1.5 Effective plastic strain ............................................................................... 19
      6.2.2 Results layer 5 ................................................................................................. 20
        6.2.2.1 Normal stress $\sigma_n$ in layer 5 ................................................................. 20
        6.2.2.2 Normal stress $\sigma_{\text{pl}}$ in layer 5 ............................................................ 21
        6.2.2.3 Shear stress $\tau_{\text{pl}}$ in layer 5 ................................................................. 22
        6.2.2.4 Equivalent Von Mises stress .................................................................... 22
        6.2.2.5 Effective plastic strain ............................................................................... 23
    6.3 Results bending C-profile situation II ................................................................. 24
      6.3.1 Results layer 1 ................................................................................................. 24
        6.3.1.1 Normal stress $\sigma_n$ in layer 1 ................................................................. 24
        6.3.1.2 Normal stress $\sigma_{\text{pl}}$ in layer 1 ............................................................ 25
        6.3.1.3 Shear stress $\tau_{\text{pl}}$ in layer 1 ................................................................. 26
        6.3.1.4 Equivalent Von Mises stress .................................................................... 26
        6.3.1.5 Effective plastic strain in layer 1 .............................................................. 27
      6.3.2 Results layer 5 ................................................................................................. 28
        6.3.2.1 Normal stress $\sigma_n$ in layer 5 ................................................................. 28
        6.3.2.2 Normal stress $\sigma_{\text{pl}}$ in layer 5 ............................................................ 28
        6.3.2.3 Shear stress $\tau_{\text{pl}}$ in layer 5 ................................................................. 29
        6.3.2.4 Equivalent Von Mises stress in layer 5 .................................................... 30
        6.3.2.5 Effective plastic strain in layer 5 .............................................................. 30
  6.4 Discussion of the results ......................................................................................... 31
7 Conclusion ..................................................................................................................... 32
Literature list ................................................................................................................... 33
1 Introduction

Important parts of the chassis of a truck are the chassis side members. These members are formed from metal plate by rolling and bending. The rolling process causes a C-profile. This C-profile is bent twice along an axis perpendicular to the longitudinal direction. The reason for this is that in the front of the truck, an engine is located between the two side members and at the back of the truck, double wheels are placed at the outside of the side members. This is sketched in figure 1.1. Along the length of the truck, many more components are connected to the chassis side members. The manufacturing of the chassis side members results in plastic strains and residual stresses. DAF is interested in predicting these residual stresses, because they want to use them in fatigue simulations. Residual stresses in a product have influence upon whether it will fail or not. Compressive stresses are usually more desirable than tensile stresses, because they can result in a longer fatigue life.

The objective of this project is to predict the stresses and to find out what influence the process parameters have on these stresses. This is done for the most produced tractor of DAF, the XF. To make the predictions, Marc/Mentat is used. This is done by modeling the process in Mentat and simulating it in the finite element program MARC.

The work done is explained in a number of chapters, beginning with the material properties of the chassis side members. After that, the two different element structures which have been used will be discussed. Then the rolling process and the bending process will be described and modeling choices made in Marc/Mentat are discussed. The results of the two models will be shown, followed by the conclusions.

Figure 1.1: Sketch of the position of the chassis side members in a truck.
1.1 Acknowledgements

I am grateful for having the opportunity to work on this project from DAF (department Chassis & Suspension) and to make a contribution towards solving the problem of the chassis side members. I thank Harold Heijnens for arranging various things and his view on the total project. I also want to thank René Liebregts for giving insight in the chain which DAF uses for fatigue evaluation and verification. I owe thanks to Rens Evers in the matter of the material properties and the material test data. Furthermore, thanks to Frank Bekkers and the aforementioned persons for suggestions and comments on the models and the results. I owe special thanks to my supervisor Ron Peerlings for coaching me, giving suggestions and comments on the models, results and this report.
2 Material properties

The production process of the chassis side members starts with metal plate material. The specific product name is Domex600mc. Domex, the producer, describes its product(s) as follows: All Domex cold forming steels are produced in modern plants under carefully controlled, computerized conditions. Low carbon and manganese contents, together with small additives of grain refiners, such as niobium (and/or titanium) and an otherwise very pure steel provide the good metallurgical conditions for the ultimate properties of the steels. Compared to the high strength cold forming steels, these steels have somewhat higher contents of niobium, titanium and/or vanadium micro-alloying elements. A further contributory factor to the good properties of the steels is the thermo-mechanical rolling, with its very carefully controlled heating, rolling and cooling sequences. These conditions give the Domex cold forming steels their characteristic combination of high strength and good formability, weldability and impact strength.

These steels are applied in truck chassis, cranes and earthmoving machines. In these applications the high strength of the steels is used to save weight and/or to increase the pay-load.

2.1 Chemical composition

The Domex600mc steel has a chemical composition as listed in Table 2.1:

<table>
<thead>
<tr>
<th>C % max</th>
<th>Si % max</th>
<th>Mn % max</th>
<th>P % max</th>
<th>S % max</th>
<th>Al % max</th>
<th>Nb % max</th>
<th>V % max</th>
<th>Ti % max</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.12</td>
<td>0.10</td>
<td>1.90</td>
<td>0.025</td>
<td>0.010</td>
<td>0.015</td>
<td>0.09</td>
<td>0.20</td>
<td>0.15</td>
</tr>
</tbody>
</table>

Hereby, the sum of Nb, V and Ti = 0.22% max. Also Mo = 0.50% max. and B = 0.005% max.

2.2 Mechanical properties

In order to determine the forces/stresses during manufacturing, the mechanical properties of Domex600mc are needed.

The Young’s modulus E is 210 GPa and the Poisson’s ratio ν is 0.3. It has an initial yield stress $\sigma_{Y0}$ of 600 MPa.

![True stress - true strain curve in a uniaxial tensile test on Domec600mc.](image-url)
These properties can be seen in the true stress $\sigma_t$ - true strain $\varepsilon$ curve recorded in a uniaxial tensile test, see figure 2.1. This figure also shows an elastic region and a plastic region with hardening from approximately 650 to 820 MPa. In Marc/Mentat the hardening response has been defined as a table of $\sigma_Y/\sigma_{Y0}$ against the plastic strain. To construct this table, the elastic data was deleted and $\sigma_t$ was divided by $\sigma_{Y0}$. After that, the start of the plastic hardening curve was moved to (0,1) by shifting it to the left, see figure 2.2. Note that this procedure introduces a small error because it implicitly assumes the elastic strain to be constant.

![Figure 2.2: Hardening response $\sigma_Y/\sigma_{Y0}$ versus effective plastic strain as derived from the tensile test data.](image)

There is only one problem left. The strain almost reaches 10%. Unfortunately, in the model, plastic strains are reached of more than 20%. So the hardening is extrapolated linearly up to 40% strain, see figure 2.3. This data is used in Marc/Mentat as plastic strain table.

![Figure 2.3: Extrapolated hardening behavior used in the simulations.](image)
3 Finite element discretisation

In order to model the rolling and bending operations, two different types of element structures can be used: a shell structure or a fully three dimensional structure. The shell structure is also a 3-D structure whereby certain assumptions are made with respect to the strain distribution in thickness direction. In Marc, the shell elements are numerically integrated through the thickness. The fully three dimensional solid structure consists of three dimensional brick elements.

From a long list of elements, which all have their own properties, element 7 (3-D solid structure) and 75 (shell structure) were chosen. Element 7 is described [2] as an eight-node, isoparametric, arbitrary hexahedral. It uses trilinear interpolation functions and the strain tends to be constant throughout the element. This element is preferred over higher-order elements when used in a contact analysis. The stiffness of this element is formed using eight-point Gaussian integration. For nearly incompressible behavior, including plasticity or creep, it is advantageous to use an alternative integration procedure. This constant dilatation method eliminates potential element locking. This element can be used for all constitutive relations.

Element 75 is described [2] as a four-node, thick-shell element with global displacements and rotations as degrees of freedom. Bilinear interpolation is used for the coordinates, displacements and the rotations. The membrane strains are obtained from the displacement field; the curvatures from the rotation field. The transverse shear strains are calculated at the middle of the edges and interpolated to the integration points. In this way, a very efficient and simple element is obtained which exhibits correct behavior in the limiting case of thin shells. The element can be used in curved shell analysis as well as in the analysis of complicated plate structures. Due to its simple formulation when compared to the standard higher order shell elements, it is less expensive and, therefore, very efficient in nonlinear analysis. The element is not very sensitive to distortion, particularly if the corner nodes lie in the same plane. All constitutive relations can be used with this element.
4 C-forming

The first manufacturing process is C-forming. A metal plate is formed to a C-profile. In reality this is done by roll forming, an operation in which sheet-metal is formed into curved sections by means of a series of rolls. As the sheet passes between the rolls, the rolls are brought toward each other to a configuration that achieves the desired radius [1].

In this case, the sheet has to be bent 90 degrees and the desired radius is 11 mm. This has to be done at each side of the sheet. The other dimensions (mm) of the bent plate are shown in figure 4.1.

Figure 4.1: Cross-section dimensions of the C-profile

Modeling the rolling process in Marc/Mentat is quite complicated and results in costly simulations. For this reason, the process is approximated here by a simple bending operation.

The full solid model of the rolling process is shown in figures 4.2 and 4.3. This model has the right dimensions of the plate, except in longitudinal direction (z-direction), because of the length of the plate in that direction. In the z-direction, plane strain boundary conditions are applied. The model exists of 4 (contact) bodies. Contact body 1 is the metal part which is deformed. Contact bodies 2 and 3 are rigid contact bodies and 4 is moving with a certain rotational speed around the z-axis. All contacts are frictionless.

Figure 4.2: Brick model (element 7) with reduced length and plane strain boundary conditions.
Results of the brick model will be compared with those of the more efficient shell model. The only difference between the brick model and the shell model is the deformable contact body, see figure 4.4. The distance between contact bodies 2 and 3 (figure 4.2) is exactly the same and the shell is modeled precisely between contact bodies 2 and 3 and has a thickness which is equal to the solid model.

4.1 Choices made in Mentat:
The numerical stability of the simulations and the sensibility of the results obtained are highly dependent on a number of settings as defined in Mentat. The choices made for these settings are therefore discussed in detail below. This will be done only for the shell model, because that is the model which will be expanded with the second bending, see bricks_cforming_7_80.mud and shells_75_80.mud on the cd-rom.

Mesh generation
With mesh generation, the model was made in the same dimensions as the sheet is in real, with the exception of the z-direction.

Boundary condition
For the nodes on the symmetry plane, all displacements and rotations are set to zero. Furthermore, the z displacement and x and y rotations of all nodes are set to zero to impose a plane strain state.
Material properties
It is assumed that the material behaves the same in all directions, so it is isotropic. Further it will be deformed elastically and plastically. For the plastic strain table see paragraph 2.2 of material properties.

Geometric properties
Because a shell structure is a 3-D structure with a thickness, 3D should be chosen and then shell. After that a thickness has to be given. The thickness of the chassis side members is 6 mm, so the shells have a thickness of 6 mm.

Contact
Here the contact bodies have to be defined. First the deformable contact body and then the others (rigid). The moving body has to be given a rotational speed and a time table. This time table describes the velocity of the moving contact body as function of time. The contact bodies are surfaces. Attention should be paid to the correct definition of the front and the back faces, otherwise contact will not be recognized by Marc.

Load cases
Because the sheet will be deformed mechanically, choose mechanical. The loads are the boundary conditions. Convergence testing has to be absolute, because if one chooses relative, and the contact is moving back without touching the deformed sheet, displacements are divided by zero and that is not allowed. Furthermore, the absolute displacement can be tuned. The tighter this tolerance is set, the longer Marc has to iterate to convergence to a solution, but the more accurate this solution is.

Jobs
Here one defines what job(s) has/have to be done. The present simulation is a mechanical job. When one clicks contact control and then advanced contact control a contact distance tolerance can be defined. By default, Marc uses the thickness of the shell divided by four. In this case that means a distance tolerance of 6/4=1.5 mm. This is far too much. So 0.005 mm is chosen.
Under mechanical analysis options choose large displacements, and under advanced options the updated Lagrange procedure. This is done because it is a large strain plasticity problem. Furthermore, the plasticity procedure is large strain additive. With job result one can choose any result wanted. The number of layers, over the thickness, can be chosen under job parameters. By default, five layers are used and in this case that is enough because the thickness is 6 mm.
Under element types, element 75 must be selected.
5  Bending

The second manufacturing process considered here also results in bending of the work piece. Contrary to the rolling, this process is done by DAF itself. The C-profile is bent along an axis perpendicular to the longitudinal direction. This is done twice per chassis side member. A picture of this process is shown in figure 5.1.

Figure 5.1: A finished chassis side member is coming out of the bending machine.

In figure 5.2, the machine in which the process takes place can be seen. The chassis side member is coming in and will first be bent downwards by stamp 1. After that, the part will be moved further and then stamp 2, which can not be seen in the picture, will bend it upwards.

Figure 5.2: Bending machine with incoming part.

The bending process is modeled by a second load case in Mentat. Per load case, one can define which contact bodies are in contact with the deformable body. A picture of a model can be seen in figure 5.3.
The model consists of 7 contact bodies: one deformable contact body, two moving contact bodies and four rigid contact bodies. Bodies 1, 2, 3 and 4 are used in load case 1 and bodies 1, 5, 6 and 7 in load case 2. This can be defined with contact tables. Because the chassis side member is bent twice and those two situations are different, there are two models. They contain exactly the same C-bending, but they differ at the point of second bending. In one case, the moving contact turns clockwise around the x-axis and in the other case the contact turns counterclockwise around the x-axis. One of these models will be discussed, because the only difference between the two models is a time table which describes the velocity of the moving contact. In the next chapter, the results of both models will be presented.

5.1 Choices made in Mentat:

The translation from manufacturing process to model can be difficult, but has to be done as well as possible to give good results. In the machine, the stamp is a rounded area that will roll over the profile during the bending. As a result, there will be little friction. In Mentat, the stamp is modeled as a flat surface with frictionless contact. Also very important are the boundary conditions. The choices made in Marc/Mentat will be explained in the same order as the Mentat menus.

Mesh generation
The model was made in the same dimensions as the plate is in real, with the exception of the longitudinal direction of the side member (z-direction). The deformable body is 500 mm long, which is 250 mm at each side of the bending line. To reduce calculation time, elements in which the largest strains are expected have a smaller size than where small strains occur, see figure 5.4. Only shell elements have been used in these simulations for reasons of computational efficiency.
Also the contact bodies have to be made here.

**Boundary condition**

There are four boundary conditions. Boundary conditions 1 and 2 are used in load case 1. Boundary conditions 3 and 4 are used in load case 2.

1. The first boundary condition defines symmetry. On the symmetry plane (in figure 5.4 the lower edge) all displacements and rotations of these nodes are set to zero.
2. The second boundary condition is applied to all nodes and sets the $z$ displacement to zero and $x$ and $y$ rotations to zero as well.
3. The third boundary condition is applied to the nodes on the left edge in figure 5.4. It fixes all translations and rotations. This is done because in $z$-direction, the part is very long.
4. The fourth boundary condition defines the same symmetry as boundary condition 1. Displacement in $x$-direction and $z$-rotation are set to zero.

**Material properties**

See material properties paragraph 4.1.

**Geometric properties**

See geometric properties paragraph 4.1.

**Contact**

Contact bodies:

See contact paragraph 4.1.

Contact Tables:

Because there is a complex situation with seven contact bodies, which are not all the same time in contact with the deformable body, contact tables have to be made, one for each load case. Contact bodies 2, 3 and 4 are in contact with deformable contact body 1 during load case 1. Contact bodies 5, 6 and 7 are in contact with deformable contact body 1 during load case 2.

Also a contact distance can be defined. By default, Marc uses the thickness of the shells divided by four. In this case that means a distance tolerance of $6/4=1.5$ mm. This is way too much. So 0.005 and 0.007 are chosen, 0.005 mm for the moving body and 0.007 for the rigid bodies. Furthermore, the separation force is set to 0.01 to ensure a proper separation of the workpiece from the tools when these are retracted.

**Load cases**

See load cases paragraph 4.1.
Jobs
Here one defines what job(s) have to be done, i.e. a mechanical job. When one clicks contact control and then advanced contact control one can give a contact distance tolerance. Under mechanical analysis options choose large displacements, and under advanced options the updated Lagrange procedure. This is done because it is a large strain plasticity problem. In the Lagrangian method, the finite element mesh is attached to the material and moves through space along with the material. In the updated Lagrangian approach, the current configuration acts as the reference state. True or Cauchy stress and true strain are used in the constitutive relationship. Furthermore, the plasticity procedure is large strain additive. This is a hypoelastic, rate-based formulation where the strain rate is decomposed into a sum of elastic and plastic terms. The Jaumann rate of Cauchy stress is used with this formulation. With job result one can choose any result wanted. The number of layers can be chosen under job parameters. Under element types an element type can be chosen. In this case, thick shell element 75 is chosen, see also chapter 3.
6 Results

In this chapter, the results of C-forming and the results of bending the C-profile will be shown. The results of C-forming consist of results of the brick model and results of the shell model. The results of bending the C-profile contain two different situations. The first situation in Marc/Mentat is the one with bending of the C-profile clockwise around the x-axis, resulting in extension of the C-profile’s flange. The second is the one with bending counter clockwise around the x-axis resulting in compression of the flange. The results are the final stresses after the manufacturing processes (i.e. after release of the tools). DAF will use these stresses in fatigue simulations. For completeness, also the Von Mises stresses and the effective plastic strain will be shown. All results will be shown for the upper and lower layer of the shell elements.

6.1 Results C-forming

The computed stress components for shell elements are given by Marc/Mentat in terms of local axes, attached to the elements, which rotate as the workpiece is deformed. These axes will be denoted as indicated in figure 6.1, where \( \sigma_{nn} \) is the normal stress in axial direction, \( \sigma_{tt} \) is in the tangential direction and \( \sigma_{nn} \) in thickness direction. The shear stress \( \sigma_{at} \) tangential to the member will also be considered for the loading case which models the bending of the C-profile. For the loading case which models the rolling, however, only the \( \sigma_{tt} \) stress is relevant.

![Figure 6.1: Local axis directions of the normal stresses.](image)

In the shell model, layer 1 is on the outside of the C-profile and layer 5 on the inside.

6.1.1 Normal stress \( \sigma_{tt} \)

In figure 6.2, the major principal stress of the brick model is shown. The major principal stress is the largest principal stress in the absolute sense. This stress is shown because when using brick elements, the computed stress components are given in terms of the global axes. So in fact, for bending around the a-axis this is the normal stress in t-direction. During C-forming, first there occur compressive stresses on the inside and tensile stresses on the outside of the profile. But when the tool is retracted, at the surface of the profile tensile stresses become compressive and compressive stresses become tensile.
Figures 6.3 and 6.4 show the inside layer (5) and the outside layer (1) of the shell model. On the inside of the contour tensile stresses occur and on the outside compressive stresses. This result is the same as seen in the brick model. The only difference is the values of the stresses. The maximum tensile stress, on the inside of the profile, in the shell model is 430 MPa and in the brick model 469 MPa. The maximum compressive stress, on the outside of the profile, in the shell model is -396 MPa and in the brick model -233 MPa.

6.1.2 Effective plastic strain

For completeness, also the effective plastic strain of the brick model and the shell model will be shown. In figure 6.5, the effective plastic strain of the brick model is given. The highest plastic strain is about 29 percent on the inside of the profile.
In figures 6.6 and 6.7, the effective plastic strains of the inside (layer 5) and outside (layer 1) layers of the shell model are shown. The highest value lies also on the inside of the profile and reaches a strain of 20 percent, but is almost the same as on the outside of the contour.

6.2 Results bending C-profile situation I

As described above, the first situation is the one with bending clockwise around the x-axis. In the real situation, the chassis side member lies upside down, with its ears downwards and in Marc/Mentat with its ears upwards; see picture 5.3 for the situation in Marc/Mentat. The final stresses are the stresses in t-direction (in Marc/Mentat $\sigma_{11}$),
a-direction ($\sigma_{22}$) and the shear stresses in the plane that is spanned by those two vectors ($\sigma_{12}$).

### 6.2.1 Results layer 1

In this paragraph, all results of layer 1 will be shown, which for this model is on the outside of the profile.

#### 6.2.1.1 Normal stress $\sigma_{tt}$ in layer 1

As can be seen in figure 6.8, the highest tensile stress in t-direction occurs in region 1 with values around 800 MPa. With the t-direction, the in-plane normal stress is meant which goes along with the contour. The highest compressive stresses in t-direction have values of around 1100 MPa and occur in region 2. These are very high values and are caused by rigid contact body 7 (see figure 5.3). In the machine, the chassis side member is not held by such a contact but is clamped on the flanges. So when that should be modeled, perhaps those stresses will not be that high. In region 3, there are also compressive stresses. This is very likely, since the bending is clockwise. So material on the lower side of the profile experiences compression.

![Figure 6.8: Normal stress layer 1 (lower layer) in t-direction.](image)

#### 6.2.1.2 Normal stress $\sigma_{aa}$ in layer 1

In figure 6.9, the normal stresses in a-direction are shown. With a-direction, the in-plane normal stress in axial direction is meant. In region 1, there is a small spot with tensile stresses around the 900-1200 MPa. In region 2, compressive stresses with values around -300 - -600 MPa occur.
6.2.1.3 Shear stress $\sigma_{\text{at}}$ in layer 1

As can be seen in figure 6.10, there are no spots with high in-plane shear stresses. There is a big region with low negative shear stresses (-100 - -10 MPa) and also a big region with low positive shear stresses (10 - 150 MPa).
6.2.1.4 Equivalent Von Mises stress

The equivalent Von Mises stress is often used as a yield criterion. This stress can be compared with the material properties in chapter 2, because it has all principal stresses in it. Figure 6.11 shows the equivalent Von Mises stress. One can see that there are two spots with high stresses. Values are found about 900-1100 MPa. In the rest of the side member, the equivalent Von Mises stress varies from 20 to 500 MPa.

![Equivalent Von Mises Stress Layer 1](image)

Figure 6.11: Equivalent Von Mises Stress in layer 1 (lower layer).

6.2.1.5 Effective plastic strain

In figure 6.12, the effective or equivalent plastic strain can be seen. One can see that the biggest effective plastic strain is generated by the first bending, which models the rolling of the profile. Strains are reached of 19 percent. The second, real bending process results in 5 percent effective plastic strain at the second bending line and also in the flange.
6.2.2 Results layer 5

In this paragraph, all results of layer 5, on the inside of the profile, will be shown.

6.2.2.1 Normal stress $\sigma_{tt}$ in layer 5

As can be seen in figure 6.13, the highest compressive stress in t-direction occurs in region 1 with values from -700 to -1000 MPa. The highest tensile stresses in t-direction have values around 1100 MPa and occur in region 2. These are very high values and are caused again by rigid contact body 7 (see figure 5.3).
6.2.2.2 Normal stress $\sigma_{aa}$ in layer 5

In figure 6.14, the normal stresses in a-direction are shown. In region 1, there is a small spot with tensile stresses from 800 to 1000 MPa. In region 2, compressive stresses with values around 300-600 MPa occur. This is probably caused by the boundary conditions applied to the nodes on that edge (boundary condition 3).
6.2.2.3 Shear stress $\sigma_{at}$ in layer 5
As can be seen in figure 6.15, there are no spots with high stresses. There is a big region with low negative shear stresses (-200 - -10 MPa) and also a big region with low positive shear stresses (10 - 90 MPa).

![Figure 6.15: Shear stress layer 5 (upper layer).](image)

6.2.2.4 Equivalent Von Mises stress
Figure 6.16 shows the equivalent Von Mises stress. One can see that there are two spots with high stresses. Region 1 consists of three spots with values about 900-1100 MPa. In spot 2, stresses are reached from 900 to 1000 MPa. In the rest of the side member, the equivalent Von Mises stress varies from 20 to 500 MPa.
6.2.2.5 Effective plastic strain

In figure 6.17, the effective or equivalent plastic strain can be seen. One can see that the biggest effective plastic strain is caused by the first bending process (C-forming). Strains are reached of 18 percent. The second bending process results in 4 percent effective plastic strain at the second bending line and also in the ear. These results are almost the same as for layer 1.
6.3 Results bending C-profile situation II

As described above, the second situation is the one with bending counter clockwise around the x-axis, such that the flanges of the profile are in compression.

6.3.1 Results layer 1

In this paragraph, all results of layer 1 will be shown.

6.3.1.1 Normal stress $\sigma_{tt}$ in layer 1

As can be seen in figure 6.18, the highest tensile stress in t-direction occurs in region 1 with values around 700 MPa. The highest compressive stresses in t-direction have values around -1100 MPa and occur in region 3. These are very high values and are caused by rigid contact body 7 (see figure 5.3). This problem appeared also in situation I. In region 2, there are also compressive stresses, with values around -800 MPa.
6.3.1.2 Normal stress $\sigma_{aa}$ in layer 1

In figure 6.19, the normal stresses in a-direction are shown. In region 1, there are tensile stresses with highest values around the 800 MPa. In region 2, compressive stresses with values around -600 - -700 MPa occur. In region 3 are low tensile stresses but these are not very reliable, because the boundary conditions on that edge can be improved.
6.3.1.3 Shear stress $\sigma_{at}$ in layer 1
As can be seen in figure 6.20, there are no spots with high stresses. All stresses are between plus and minus 100 MPa.

![Figure 6.20: Shear stress layer 1 (lower layer).](image)

6.3.1.4 Equivalent Von Mises stress in layer 1
Figure 6.21 shows the equivalent Von Mises stress. One can see that there are two spots with high stresses (1 and 3). Values are found about 800-900 MPa. In spot 2, tensile stresses of 700 MPa. In the rest of the side member, the equivalent Von Mises stress varies from 50 to 200 MPa.
6.3.1.5 Effective plastic strain in layer 1

In figure 6.22, the effective or equivalent plastic strain can be seen. One can see that the biggest effective plastic strain is caused by the first bending process (C-forming). Strains are reached of 19 percent. The second bending process results in 7 percent effective plastic strain at the second bending line and also in the ear.
6.3.2 Results layer 5
In this paragraph, all results of layer 5 will be shown.

6.3.2.1 Normal stress $\sigma_{tt}$ in layer 5
As can be seen in figure 6.23, in a small region (1), tensile stresses and compressive stresses occur. There is a small spot with tensile stresses (800 MPa) and 3 spots around it with compressive stresses (-600 MPa). The highest tensile stresses in t-direction have values around 1100 MPa and lay in region 2.

![Figure 6.23: Normal stress layer 5 (upper layer) in t-direction.](image)

6.3.2.2 Normal stress $\sigma_{aa}$ in layer 5
In figure 6.24, the normal stresses in a-direction are shown. There are three regions with compressive stresses around -700 MPa. In the biggest part of the chassis side member, stresses lay between 10 and 100 MPa.
6.3.2.3 Shear stress $\sigma_{at}$ in layer 5

As can be seen in figure 6.25, there are no spots with high stresses. There is a big region with low negative shear stresses (-300 - -10 MPa) and also a big region with low positive shear stresses (10 - 200 MPa).
6.3.2.4 Equivalent Von Mises stress in layer 5

Figure 6.26 shows the equivalent Von Mises stress. One can see that there are two regions with high stresses. In region 1, stresses of 700 MPa occur and in region 2 stresses of 1100 MPa. In the rest of the side member, the equivalent Von Mises stress varies from 20 to 600 MPa.

![Equivalent Von Mises Stress Layer 5](image)

Figure 6.26: Equivalent Von Mises Stress in layer 5 (upper layer).

6.3.2.5 Effective plastic strain in layer 5

In figure 6.27, the effective or equivalent plastic strain can be seen. One can see that the biggest effective plastic strain is affected by the first bending process (C/U-forming). Strains are reached of 19 and 20 percent. The second bending process affects 3 percent effective plastic strain at the second bending line and 4 percent in the ear. These results are almost the same as for layer 1.
6.4 Discussion of the results

The results of the C-bending seem to be acceptable, but when they are compared with the brick model, the conclusion is that the tensile and compressive stresses in the brick model are not as high as the tensile and compressive stresses in the shell model. Analyzing the results of bending the C-profile is more difficult than analyzing the results of C-bending, because the manufacturing process is more complicated. Although conclusions can be made.

Situation I, bending clockwise around the x-axis, causes tensile stresses in a-direction in the flange, as expected. Furthermore this bending causes tensile stresses in region 1 in layer 1 (figure 6.8) and compressive stresses in region 1 in layer 5 (figure 6.13). The Von Mises stresses are different in region 1: in layer 1 there is on peak (figure 6.11) and in layer 5 there are three peak spots in the same region (6.16). Bending clockwise causes compressive stresses in a-direction in the flange (see figure 6.9 and 6.14).

Bending the C-profile counterclockwise causes compressive stresses in a-direction in region 2 in layer 5 (figure 6.24) and in layer 1 in the same region tensile stresses (figure 6.19). In the flange in layer 1, tensile stresses occur in region 1 and in layer 5, compressive stresses occur in the same region. This is probably caused by the fact that the flange shows a considerable buckle (10 mm). This is not seen in the side members in reality.
7 Conclusion

The objective of this project was to predict the residual stresses in the chassis side members and to find out what influence the process parameters have on these stresses. After a lot of modeling in Marc/Mentat, two final models are released to achieve this objective, but there has to be done more work.

3-D models were made using brick elements and shell elements. Shell elements were chosen because of calculation time. Then, the model was expanded with the bending of the C-profile. The results make clear where tensile stresses occur and where compressive stresses occur.

These results are in general sensible. There are also a few points where the final shell models can be improved:

- Boundary conditions in relation to the bending machine
- Contact bodies (flat surface or rounded stamp, contact body at one side of the flange)
- Contact distances of the contact bodies
- Separation force of each contact body
- Element sizes

Furthermore, DAF wanted to know the effect of variations in bending and maybe find an optimum process. This would be a good subject for a next TU student at DAF who could employ the knowledge and modeling developed in the pres cut project.
Literature list
