Performance evaluation of a simplified FEM-tool for designers

Hjalmar Öberg

Department of Mechanical Engineering
Blekinge Institute of Technology
Karlskrona, Sweden
2008

Performance evaluation of a simplified FEM-tool for designers

Hjalmar Öberg

Department of Mechanical Engineering
Blekinge Institute of Technology
Karlskrona
Sweden
2008

Master of Science thesis in Mechanical Engineering

in cooperation with
Abstract

On today’s competitive market it is important to analyse and optimise product performance. The finite element method is a very common tool to do so when dealing with stress analysis. There are many different commercial finite element software available on the market. They range from complex ones needing special skills for proper use to more easy to use aimed at design engineers with little experience of the finite element method.

The latest versions of Autodesk Inventor include a stress analysis tool that falls under the second category (easy to use). The analysis tool of Inventor is only able to perform a few types of analysis, and the user control is limited. The question is how well this tool works. What are its strengths and weaknesses? This thesis aimed to answer those questions.

This was done by analysing a few “benchmarks”. Some of the analysed parts are just made up for this thesis, but most of them are provided by Water Jet Sweden AB, and are typical parts from a water jet cutting machine. These benchmarks has been thoroughly tested using both modal and stress analysis. The results from Inventor have then been compared with analysis results from I-deas, a well known commercial finite element software. The aim of these tests is not to see how close to reality Inventor gets, but if it gets a result close to a more advanced FE-tool. Depending on the benchmark different results is presented. For the larger parts and for the beams both stress and modal analysis are made, and the first six modes are presented. The deflection/deformation is shown for most benchmarks and equivalent/von Mises stress is shown for all.

It is shown that the analysis tool in Inventor works fine when it comes to small and/or simple parts. Furthermore it is very easy to use compared with I-deas, although the solving time sometimes is rather long for big parts. A few problems do however exist. The main problem is that if the difference between the smallest and largest dimension in the part is big, the part cannot be meshed. This problem occurred mainly on benchmarks including long beams. The conclusion of this work is that as long as one can mesh the part fairly good results are attained. It may take a long time to solve, but one can trust the result.

Keywords:
Finite element method, Inventor, I-deas, Water jet.
Acknowledgements

This thesis has been performed at Swedish Waterjet Lab (SWL), in Ronneby, Sweden, under the supervision of Dr Johan Wall.

I wish to express my gratitude to Johan Wall for all his guidance and input during this work. I also want to thank all the personnel at SWL, which in one way or the other made this thesis possible, and made the work enjoyable.

I wish to thank Water jet Sweden AB, and especially Stefan Hansell, for the providing of benchmarks.

Last but not least I want to thank my family and friends for all the support and understanding.

Karlskrona, October 2008

Hjalmar Öberg
Contents

Abstract ...................................................................................................................... 3
Acknowledgements ................................................................................................... 4
Notation ...................................................................................................................... 6

1 Introduction .......................................................................................................... 7
   1.1 Background ...................................................................................................... 7
   1.2 Aim and purpose ............................................................................................. 7
   1.3 Method ............................................................................................................. 7

2 Water jet cutting .................................................................................................... 8
   2.1 History ............................................................................................................. 8
   2.2 How does it work? .......................................................................................... 11
   2.3 Advantages ..................................................................................................... 11

3 Fem in Inventor ..................................................................................................... 12
   3.1 Pre-processor ................................................................................................. 13
   3.2 Post-processor ............................................................................................... 24

4 The benchmarks .................................................................................................... 28
   4.1 Benchmark 1 .................................................................................................. 28
   4.2 Benchmark 2 .................................................................................................. 45
   4.3 Benchmark 3 .................................................................................................. 50
   4.4 Benchmark 4 .................................................................................................. 57
   4.5 Benchmark 5 .................................................................................................. 63
   4.6 Benchmark 6 .................................................................................................. 69
   4.7 Benchmark 7 .................................................................................................. 76
   4.8 Benchmark 8 .................................................................................................. 83
   4.9 Benchmark 9 .................................................................................................. 87
   4.10 Benchmark 10 .............................................................................................. 91
   4.11 Benchmark 11 .............................................................................................. 99

5 Discussions and conclusions ................................................................................ 106

6 References ............................................................................................................. 108
## Notation

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$B$</td>
<td>Width</td>
<td>mm</td>
</tr>
<tr>
<td>$F$</td>
<td>Force</td>
<td>N</td>
</tr>
<tr>
<td>$K_t$</td>
<td>Stress concentration factor</td>
<td>-</td>
</tr>
<tr>
<td>$b$</td>
<td>Width</td>
<td>mm</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius</td>
<td>mm</td>
</tr>
<tr>
<td>$t$</td>
<td>Thickness</td>
<td>mm</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Stress</td>
<td>Pa</td>
</tr>
</tbody>
</table>
1 Introduction

1.1 Background

Swedish Waterjet Lab (SWL) is a research laboratory that deals with water jet cutting technology. The lab is collaboration between Blekinge Institute of Technology and the industry. The purpose of SWL is to make a strong contribution in the development of water jet cutting technology and competence.

This thesis is made at SWL together with Water Jet Sweden AB (WJS), which is one of the companies that SWL is collaborating with. WJS is one of Europe’s biggest manufactures of water jet cutting machines.

1.2 Aim and purpose

The purpose of this work is to evaluate the performance of Autodesk Inventors new finite element solver. The FE-tool in Inventor is a simplified tool that is aimed at designers rather than specialists. How efficient this software is and when it can or cannot be used is to be determined.

1.3 Method

To make the evaluation, 11 different benchmarks are analysed. A few of these benchmarks are made up for this work, but most of them are provided by WJS, and are typical parts from a water jet cutting machine.

Depending on the benchmark, different kinds of tests are made, and the result is then compared with results from the same test using I-deas, a well known FE-tool, instead of Inventor. This will show how well Inventor works.

In this work it is not interesting how close to the real world Inventor gets, but how close it gets to other FE-tools. Normally a “reality check” has to be made to verify the results. An analytical solution is therefore made on benchmark 1.
2 Water jet cutting

2.1 History

It all started in the early 50:s when the forest engineer Norman Franz had an idea to use a thin water stream to cut through lumber [1]. He did manage to create short burst of high pressure by dropping weights on to the water.

The problem was that he did not manage to create at continuous water stream so the technology was very limited. But the idea of using water to cut materials was born.

The next large development step happened in 1979, when Dr. Mohamed Hashish began his research to find a way to use a water jet to cut metal and other hard objects. He started to add an abrasive, a hard type of sand, to the water jet and using this he was able to cut through steel [1].

At the beginning it was the space industries that had an interest in water jet cutting, but over the years many other industries has started to use the technology.

2.2 How does it work?

To be able to cut things using water one needs water at a very high pressure, 140 to 414 MPa [2]. To make this pressure a high pressure pump is used.

2.2.1 Cutting head without abrasive

In the cutting head, see Figure 2.1, the water is forced to pass through a tiny orifice. The orifice is made of a jewel with a tiny hole which is typically 0.1 to 0.3 mm in diameter. This creates a very high velocity jet of water, about 900 m/s.
It is this velocity that actually cuts the material. When the water jet hits the material it loses tiny grains of material and a cut is made.

![Diagram of cutting heads](image)

*Figure 2.1, Cutting head without abrasive. Photo courtesy of Flow International Corporation.*

### 2.2.2 Cutting head with abrasive

This type of cutting head, Figure 2.2, normally consists of the same type of cutting head as on the pure water jet machines. To this cutting head a new part is added, that adds the abrasive to the water. The velocity of the water jet is used to accelerate the abrasive, and in this case it is the abrasive that erodes the material, not the water itself. That is why one is able to cut through very hard material using an abrasive water jet cutting machine.
2.2.3 Motion equipment

There are several types of water jet cutting machines, from 1 to 3-dimentional.

The 1-dimension cutters works in many ways as a band saw, where the material is fed through the water jet.

The cutting tables are mainly used for 2 dimension cutting, but there are models that can lean the cutting head, and therefore cut in 3 dimensions. The cutting tables can cut almost any material and any 2-d shape, and the 3-d versions can cut cones from flat sheets, chamfers and holes in tubes. Often, to increase the capacity, more than one cutting head is mounted on the same machine [3].

Robots can cut in any desired direction, even from underneath the material. This gives a very big flexibility, and the robot can be used to cut very complex geometries. The robot can actually cut anything the cutting table can, but it does not have as good accuracy [3].
2.3 Advantages

One of the biggest advantages is that a water jet cutting machine can cut almost any material and any shape. Thin and/or soft materials like soft rubber, foam and carpet can be cut with pure water and thicker/harder materials like steel, glass and hard wood is cut with abrasive [4].

It is also ideal when it comes to nesting (how close one can place the geometries on the material and optimize the material usage). Since the water jet does not produce any heat there will not be a heat affected zone around the cut. The geometries can therefore be placed very close and less material is wasted. The thin water jet also makes it possible to cut rather difficult geometries [3].

The cutting can start right in the middle of a sheet so there are no needs for a starting hole. The cutter is also very flexible and can e.g. be used to drill holes, to polish the surfaces, as a lathe and as a milling cutter. It can replace many different kinds of machinery. It is also easy to change the cutting program to perform several different operations on the same piece, and this without having to change the working tool [5].
3 Fem in Inventor

As said above the FEM-tool in Inventor is very basic and easy to use. In this chapter the basics in how the FEM-tool works and how to use it is described. This is not a complete guide, but by reading this one should be able to start using the tool. To learn more about this tool, reading the help topics and try some tutorials is suggested.

The FEM-tool consists of two tool-panels (Figure 3.1 and 3.2). The first panel contains the pre-processor and a bit of the post-processor. The second panel contains the rest of the post-processor.

![Stress Analysis panel 1](image.png)

*Figure 3.1, Stress analysis panel 1.*
3.1 Pre-processor

The pre-possessor consists of loads, constraints and the “stress analysis settings” in the first panel.

3.1.1 Loads

To apply a load, click a load tool on the Stress Analysis panel bar and specify the geometry, magnitude, and if applicable, the direction. There are 5 different load tools in Inventor: Force, pressure, bearing load, moment and body loads.

3.1.1.1 Force

One can apply a force to a vertex, edge, or face on a part. One can specify the direction of the force either by entering the X, Y, Z components or by
selecting a plane, face edge or axis that is parallel to the force. If a face or plane is selected to orient the force, the force is aligned normal to the selected geometry. When geometry is selected to apply the force to all selections must be of the same type. For example, one can select four different faces but not two edges and two faces. The total force is evenly divided among the selected geometry. The force tool is shown in Figure 3.3.

![Figure 3.3, The force tool.](image)

### 3.1.1.2 Pressure

A pressure is applied to a face on the geometry. The pressure’s direction is automatically normal to the face it is directed towards the face. Just enter a negative magnitude if you want the pressure to be directed away from the face. The pressure tool is shown in Figure 3.4.

![Figure 3.4, The pressure tool.](image)
3.1.1.3 Bearing load

A bearing load is applied to an inside or outside of a cylindrical face, and the direction is chosen in the same way as with the force. The bearing load tool is shown in Figure 3.5.

![Figure 3.5, The bearing load tool.](image)

A bearing load is applied where a bolt, shaft or pin makes contact with the part. If the pin/shaft/bolt has a tight fit it is assumed that the contact area full surface, but if the fit is loose one might have to split the surface and make a smaller contact area. Using a bearing load, the load is distributed as shown in Figure 3.6.

![Figure 3.6, Load distribution using bearing load.](image)
3.1.1.4 Moment

A moment load is applied to faces, and the direction is specified by entering X, Y, Z components or chose a planer face. The moment tool is shown in Figure 3.7.

![Moment tool](image)

**Figure 3.7, Moment tool.**

3.1.1.5 Body loads

Body loads include loading due to gravity and acceleration and apply to the entire part, not to specific areas of the part. Gravity is applied to include the part's weight into the analysis. The direction is set as the X, Y, or Z part axis. Linear acceleration or rotational velocity is applied to determine the effect of accelerating the entire part. The direction is determined by selecting a planar face, work plane, linear edge, or work axis. The body load tool is shown in Figure 3.8.
3.1.2 Constraints

There are three different constraints that can be applied using Inventor: fixed constraint, pinned constraint and frictionless constraint.

3.1.2.1 Fixed constraints

Fixed constraints prevent translation in one or more directions. By default, all directions are fixed, but one can change that if one wants to allow translation in one direction using the X, Y, Z components. These components can also be used to specify a displacement to the geometry. The fixed constraint tool is shown in Figure 3.9.
3.1.2.2 Pinned constraint

A pin constraint is used to prevent a cylindrical surface on the part from moving radially, tangentially, or axially. Pin constraints are typically used where holes are supported by bearings or pins. Which directions to fix with respect to the cylindrical surface can be chosen. The pinned constraint tool is shown in figure 3.10.
3.1.2.3 Frictionless constraint

A frictionless constraint enables a surface to freely slide along a plane or surface but prevents the surface from moving normal to itself. Frictionless constraints are used to model face-to-face and surface-to-surface contact between parts where one part can slide on the other. Since most surfaces in contact are not entirely frictionless, frictionless constraints give conservative results because the friction's contribution to the overall model stiffness is not included.

Frictionless constraints are also used to model symmetry boundary conditions. When a model's loading and geometry are symmetric, you can analyze a portion of the model to save analysis time. Frictionless constraints are used on all of the symmetry surfaces in the model. The frictionless constraint tool is shown in Figure 3.11.

![Frictionless Constraint Tool](image)

Figure 3.11, Frictionless constraint tool.

3.1.3 Stress analysis settings

The stress analysis setting window is shown in Figure 3.12 and this is where the element mesh is generated and the type of solution is chosen.
The first thing that one needs to do is to choose if one wants to make a stress analysis or a modal analysis (or both). Modal analysis determinates the resonance frequencies and the mode shapes for the 6 first modes, and the stress analysis determines the strength of components and enables you to optimize designs by indicating the stress distribution and potential failure.

Then the type of element is to be chosen. For normal parts only standard solid model can be used, but if sheet metal is used one can chose between standard solid model and optimized thin model. This will make the solving go faster.

When this is done the mesh size has to be set. This is done by using the slider called “Mesh Relevance”. Using this option the average overall mesh size is set. The default setting of zero is a good starting point for most analyses. Use a higher number to get a high density mesh. This provides a more accurate answer but increasing analysis time. Use lower settings only to perform a quick analysis to ensure that the model, loads, and constraints are correctly
applied before you run an analysis with a smaller mesh (3 examples of this is shown in Figure 3.13 – 3.15). Note that this cannot be done when using optimized thin model.

If the result does not converge using the slider and smaller mesh, one can chose the “Result Convergence”. If this is selected, the mesh is automatically refined. This significantly increases analysis time but generally provides a more accurate result. An example of how the mesh will look like after this is shown in Figure 3.16. Note that this cannot be done if modal analysis or optimized thin model is chosen; it only works with stress analysis.

Figure 3.13, Mesh example using mesh relevance -100.
Figure 3.14, Mesh example using mesh relevance 0.

Figure 3.15, Mesh example using mesh relevance 100.
It is important to consider stress singularities when using result converges. If for example a load is applied on an edge or a point, this will not converge, and will therefore take much longer to solve. So to save time, always make sure to place the loads on surfaces when this is possible.

After chosen what to do and how the mesh should look like then the model needs to be solved. In Inventor there are no complex solve module, only press in the standard toolbar. This will solve the model.
3.2 Post-processor

This is where the results are shown and the model is modified if something did not work. First panel 2, (Figure 3.2), is examined:

3.2.1 Loads & constraints

The loads and constraints can be modified if something is wrong. This is done by right-clicking on the parameter under “loads & constraints” that should be changed and chose edit. The direction, the magnitude and the place that the load/constraint is working on can be changed. One can also get the reaction forces from the different constraints by right clicking on them. Loads are stored as user parameters, so their magnitudes can also be edited using the Parameters tool on panel 1 (Figure 3.1). Here all parameters that relates to the part can be modified and checked. Use this to change the value of more loads than one.

3.2.2 Results

There are two types of results, stress analysis and modal analysis. If both are chosen, the result will look like in Figure 3.2.

Modal result
By clicking on the different results one will see the mode shapes of the first 6 modes. The frequency of this mode and the mode shape will be presented. To change the number of modes one can right click on the modal result and chose a new one.

Stress result
There are 5 different results one can look at. The first three is different kinds of stresses. The most important of these is the equivalent stress, also known as von Mises. In this both principal and shear stresses are involved and it gives a more accurate result. The other two shows the minimum and maximum principal stress.

The forth result one can look at is the deformation. The deformation shape is shown graphically and the deformation magnitude is presented in mm.
The last result is the safety factor. The safety factor is equal to the yield strength of the material divided by the equivalent stress. If the safety factor is greater than 1, the yield strength is greater than the equivalent stress, and the part should not yield.

3.2.3 Material

Here the material of the part can be changed.

3.2.4 Features

In this option all the part features are stored and can be modified without closing the analysis tool. Features not to be included in the analysis can be suppressed, e.g. chamfers, holes and fillets. The features values e.g. the length of an extrusion can also be changed.

These are the things that can be done using the second panel in Figure 3.2, but there are also bits of the “post processor” in the first panel (Figure 3.1):
3.2.5 Color bar

Here the appearance of the color bar can be changed. The tool is shown in Figure 3.17.

![Color Bar Tool](image)

*Figure 3.17, Color bar tool.*

The color bar relates the colors of contoured results to numerical values. The color bar can be allured to ignore extreme results such as a stress singularity or to give more detail in a specific range of values.

To adjust the color bar, click the Color Bar tool on the Stress Analysis panel bar. In the Color Bar dialog box, one can:

- Clear the Automatic check boxes to specify a range of stress values of interest to you.
- Change the color styles from color to monochrome.
- Orient the position of the color bar in the graphics window in standard or compact form.

Each result item maintains its own color bar, so the changes that one make, for example to the equivalent stress color bar, do not affect the deformation or safety factor color bars.
3.2.6 Report

When an analysis is complete, one can use the Report tool to generate an HTML report of the result. The report includes:

- Loads.
- Constraints.
- Geometry and mesh.
- Material properties.
- Tabular results, such as maximum and minimum stress, deformation, safety factor, and modal frequencies.
- Graphical results, such as equivalent stress, deformation, safety factor, and mode shapes.

3.2.7 Animate

Use the Animate Results tool to see how the part reacts under load by animating the deformed model. The animation uses the current deformation scale. One can view the animation on screen, or save it to an AVI file to include in reports or share with team members. Using standard media player controls one can play, pause, stop, and record the animated results.

3.2.8 Export to ANSYS

This is the last choice in the stress analysis tool. This will export the part to a *.dsdb file, that can be opened with ANSYS. The file includes the geometry and all stress analysis results. This feature can be used if the part is too advanced to be solved in Inventor or when you want to make a more complex analysis.
4 The benchmarks

There are 11 different benchmarks that have been tested during this work. The first one is not only checked by using I-deas, but also verified by an analytical solution. Normally when doing this type of analysis one should always verify the results, but in this case the question is not if the results from Inventor are true, but if the results are the same as another well known FEM-tool.

4.1 Benchmark 1

The first benchmark (Figure 4.1) is chosen because there is an analytical solution. This benchmark is used to test how well the symmetry (frictionless) constraint works, so it will be solved three times: The whole part, one half of the part and one quarter of the part (Figure 4.2). The part is 80 mm long, 40 mm wide and 6 mm thick, and the radius is also 6 mm. The material is steel with a Young’s modulus of 210 GPa and Poisson’s ratio of 0.3.
4.1.1 Modelling in Inventor

The FEM modelling is very easy in Inventor, since normally only solid elements can be used. In this case one of the short sides is fully constraint and a load is placed on the other, pulling the part with 1000 N.

For the half part a frictionless constraint on the “inner” long side is placed and since the surface the load is on is half the original surface, the load is changed to 500 N. On the quarter part a frictionless constraint is placed on the back short side. These loads and constraints are shown in Figure 4.3-4.4.

Number of nodes used to solve the whole part is 14049 and the number of elements is 8539.
4.1.2  Modelling in I-deas

By using export/import step file the part is transferred from Inventor to I-deas, and this is solved using quadratic quadrilateral shell elements, and the same loads and constraints as in Inventor. The part is meshed with free mesh, and the number of elements around the radius is increased a few times so that the result converges. Only the whole part is solved in I-deas.

Number of nodes used are 959 and the number of elements are 302.
4.1.3 Results

On this first benchmark all the results are reported, all 6 modes and all stresses. This is to show how the results are presented in Inventor. For the other benchmarks only the necessary results are presented.

4.1.3.1 Mode shapes

Only the mode shapes (Figure 4.5-4.10) from Inventor and the whole part is presented here.

![Mode shape 1](image)

*Figure 4.5, Mode shape 1.*
Figure 4.6, Mode shape 2.

Figure 4.7, Mode shape 3.
Figure 4.8, Mode shape 4.

Figure 4.9, Mode shape 5.
Figure 4.10, Mode shape 6.

The resonance frequencies from Inventor and I-deas are shown in table 4.1. The difference is comparison between the whole part and I-deas.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Whole part</th>
<th>Half part</th>
<th>Quarter</th>
<th>I-deas</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode 1</td>
<td>792.89</td>
<td>786.33</td>
<td>5.6e-2</td>
<td>780.082</td>
<td>2 %</td>
</tr>
<tr>
<td>Mode 2</td>
<td>3153.1</td>
<td>4730.8</td>
<td>4645.4</td>
<td>3157.88</td>
<td>0.2 %</td>
</tr>
<tr>
<td>Mode 3</td>
<td>4224.3</td>
<td>12586</td>
<td>19174</td>
<td>4189.78</td>
<td>0.8 %</td>
</tr>
<tr>
<td>Mode 4</td>
<td>4738.4</td>
<td>15685</td>
<td>25652</td>
<td>4694.19</td>
<td>1 %</td>
</tr>
<tr>
<td>Mode 5</td>
<td>10587</td>
<td>19998</td>
<td>29152</td>
<td>10638.2</td>
<td>0.5 %</td>
</tr>
<tr>
<td>Mode 6</td>
<td>12593</td>
<td>24854</td>
<td>30827</td>
<td>12524</td>
<td>0.5 %</td>
</tr>
</tbody>
</table>
4.1.3.2 Stresses

**Whole part, Inventor:**
The results are shown in Figure 4.11-4.15.

![Figure 4.11, Equivalent stress.](image)

![Figure 4.12, Maximum principal stress.](image)
Figure 4.13, Minimal principal stress.

Figure 4.14, Deformation.
Figure 4.15, Safety factor.

Half part, Inventor:
The results are shown in figure 4.16-4.19.

Figure 4.16, Equivalent stress.
Figure 4.17, maximum principal stress.

Figure 4.18, Minimal principal stress.
Figure 4.19, Deformation.

*The quarter, Inventor:*
The results are shown in Figure 4.20-4.23.

Figure 4.20, Equivalent stress.
Figure 4.21, Minimal principal stress.

Figure 4.22, Minimal principal stress.
**I-deas:**
Only pictures of the equivalent stress and the deformation (Figure 4.24-4.25) is included here.

**Figure 4.23, Deformation.**

**Figure 4.24, Equivalent stress.**
The other results and comparison between the whole part and I-deas is shown in table 4.2.

**Table 4.2, Stresses and deformation.**

<table>
<thead>
<tr>
<th></th>
<th>Eq. stress</th>
<th>Max princ.</th>
<th>Min princ.</th>
<th>Def. mm</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Whole part</strong></td>
<td>12.735 MPa</td>
<td>13.003 MPa</td>
<td>1.9592 MPa</td>
<td>0.0018 mm</td>
</tr>
<tr>
<td><strong>Half part</strong></td>
<td>12.727 MPa</td>
<td>12.895 MPa</td>
<td>1.9424 MPa</td>
<td>0.0018 mm</td>
</tr>
<tr>
<td><strong>Quarter</strong></td>
<td>12.709 MPa</td>
<td>12.966 MPa</td>
<td>0.3221 MPa</td>
<td>0.0009 mm</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>12.800 MPa</td>
<td>12.800 MPa</td>
<td>1.8500 MPa</td>
<td>0.0018 mm</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>0.5 %</td>
<td>1.5 %</td>
<td>5 %</td>
<td>0 %</td>
</tr>
</tbody>
</table>
4.1.4 Analytical solution

The dimensions of the part are shown in Figure 4.26.

\[ F = 1000 \text{ N} \]
\[ B = 40 \text{ mm} \]
\[ b = 28 \text{ mm} \]
\[ r = 6 \text{ mm} \]
\[ t = 6 \text{ mm} \]

The maximum stress is calculated with the following formula [6]:

\[ \sigma_{max} = K_t \times \sigma_{nom} \quad (4.1) \]

\( K_t \) is the stress concentration factor, and can be looked up in a mechanical handbook, if one knows the ratios B/b and r/b.

\[ \sigma_{nom} = \frac{F}{b \times t} = \frac{1000}{28 \times 6} = 5.95 \text{ MPa} \quad (4.2) \]

\[ \frac{B}{b} = \frac{40}{28} = 1.43 \quad (4.3) \]
\[ \frac{r}{b} = \frac{6}{28} = 0.21 \quad (4.4) \]

(4.3) and (4.4) are used to get \( K_t \).

\[ K_t = 1.92 \]
Putting $K_t$ and (4.2) into (4.1) gives

$$\sigma_{\text{max}} = 11.42 \text{ MPa}$$

This result is about 10% smaller than the values from Inventor and I-deas. This value depends of $K_t$ and $K_t$ can be difficult to get exactly, so the result can differ a bit. But 10% is rather good, and therefore the results from Inventor and I-deas are verified.

### 4.1.5 Comparisons and conclusions

Comparing the symmetric constraint in Inventor, the frequencies are, as expected, completely different. This is because they are three different parts; they are just locked to behave the same when pulling on them. So when it comes to mode shape and resonance frequencies, this constrain will not work. But for the stresses, the result is the same in all three parts. The pictures look pretty much the same and the values are the same except the minimal principal stress on the quarter part. This is expected because on the other two the max value of the minimal principal stress is where the model is fully constraint, on the back short side. This side does not exist on the quarter part, so on this the max value is on the hole edge. This value is the same as the value on the hole of the other two parts.

So it can be said that the symmetrical constraint work well when it comes to stresses. However, it is important to check the model well. If the load where put on the big surface instead, and pushed it down, there will be a ridged body motion on the quarter part. So when solving a part that is only constraint with frictionless constraint one will get the message shown in Figure 4.27:

![Figure 4.27, Message when using only frictionless constraint.](image)
But, as said, in this case this will not be a problem, since the force is working in the plane.

Comparing the results with I-deas shows that I-deas gives the same results on all tests; the biggest difference is 5%. All the stresses, the deformation, mod shapes and frequencies are verified. So, on this very simple part the FEM tool in Inventor works fine. It solves the part fast and gets a good result.

### 4.2 Benchmark 2

Benchmark 2 (Figure 4.28) is a rectangular pipe that is welded to a triangular plate with a hole. The plate is 3 mm thick and the length and width is both 100 mm. The pipe is 190 mm long and 30 mm wide, and the thickness is 2 mm. The hole has a diameter of 30 mm and is placed in the middle of the triangular plate. The material is Aluminum-6061 with a Young’s modulus of 68.9 GPa and Poisson’s ratio of 0.33.

![Figure 4.28, Benchmark 2.](image)
4.2.1 Modelling in Inventor

This is a very straightforward part to model. The back side is fully constrained and a load of 500 N is placed on the free edge of the pipe. But first a fillet is made on the edge the load is placed on. This is because the model will not converge otherwise, and will therefore take much longer to solve. Now the model will converge, but the maximum stress will still be where the load is placed. This is however not a problem because the critical point on this part is very clear; on other parts it can be harder to deal with this, since one may not know if the real max stress will be where the model does not converge. The loads and constraints are shown in Figure 4.29.

Number of nodes used are 170810 and number of elements is 114536.

4.2.2 Modelling in I-deas

This model can be solved in many different ways, e.g. beam elements and shell elements. In this case the model is solved with solid elements, since that is how it is solved in Inventor. The elements used are parabolic tetrahedron, and the model is meshed with free mesh. This is then solved a few times, and the mesh is refined at the critical point until the result has converged.
111036 nodes are used and 59487 elements.

4.2.3 Results

4.2.3.1 Resonance frequencies

The resonance frequencies from both Inventor and I-deas are presented in table 4.3.

<table>
<thead>
<tr>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Mode 4</th>
<th>Mode 5</th>
<th>Mode 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inventor</td>
<td>869.46</td>
<td>1330.8</td>
<td>1958.9</td>
<td>3549</td>
<td>4006.6</td>
</tr>
<tr>
<td>I-deas</td>
<td>865.388</td>
<td>1330.86</td>
<td>1933.59</td>
<td>3525.76</td>
<td>3994.06</td>
</tr>
<tr>
<td>Difference</td>
<td>3.5 %</td>
<td>0 %</td>
<td>1.3 %</td>
<td>0.7 %</td>
<td>0.3 %</td>
</tr>
</tbody>
</table>

4.2.3.2 Stresses

*Inventor:*

The result is shown in Figure 4.30.

*Figure 4.30, Equivalent stress.*
The maximum is on the edge where the load works. This is of course not the real case; the load is not placed like this in the real world. The real maximum is in the place where the pipe and the plate meet. The colour indicates that the max stress is some ware between 168 and 196. By changing the values on the color bar a better result is shown. On Figure 4.31 the value between the two top colours is 173 MPa and the value in Figure 4.32 is 178 MPa.

*Figure 4.31, Critical point, 173 MPa.*

*Figure 4.32, Critical point, 178 MPa.*
The maximum equivalent stress is therefore 173 - 178 MPa. There are no other ways to get the value for different nodes or elements in Inventor. This is the way one has to do it.

**I-deas:**
As in Inventor the maximum stress is where the load works. But by showing only the result of the critical point Figure 4.33 is gotten. It is hard to see on this picture but the maximum stress at the critical point is 148 MPa.

![Figure 4.33, Equivalent stress at the critical point.](image)

The results from this test are also presented in table 4.4 as well as the comparison between Inventor and I-deas.

<table>
<thead>
<tr>
<th>Equivalent stress</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor</strong></td>
<td>173-178 MPa</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>148 MPa</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>15-17 %</td>
</tr>
</tbody>
</table>
4.2.4 Comparisons and conclusions

The frequencies are the same in Inventor and I-deas so the modal test in Inventor is verified. There is, however, a rather big difference in the stress values. Actually, Inventor gave a few different results. Sometimes the stress converged at about 98 MPa, and sometimes at about 178 MPa. And sometimes it did not converge at all. This is of course not good, but the result presented here is the result that occurred the most times, it converged at about 178 MPa. This is as said not very close to the value from I-deas. It is about 15% larger.

Investigating the mesh size at the critical point shows that the mesh is finer in I-deas then Inventor, therefore the I-deas value are more accurate.

This is the only benchmark that has given this kind of problem; all other problems can be explained. So, for some reason, Inventor did not work well on this benchmark.

4.3 Benchmark 3

This benchmark (Figure 4.34) is the X-beam on a water jet cutter.

![Figure 4.34, Benchmark 3.](image-url)
This benchmark is tested by placing a force of 15 000 N on the middle of the beam and then the deformation and stresses at the middle of the beam are checked. Also the resonance frequencies are tested. On this benchmark the weight of the beam itself are also considered. The material is SS1412 with a Young’s modulus of 220 MPa and Poisson’s ratio of 0.275. The mass density is 7860 kg/m³.

4.3.1 Modelling in Inventor

To be able to solve this many features has been suppressed. All features except the main beam extrusion are suppressed. Then this is solved using fixed constraint on one side and by letting it move in the beam direction on the other side. A load of 15 000 N is placed on the upper surface and the gravity is included. This is shown in Figure 4.35.

This model was solved using 74809 nodes and 45563 elements.

It will be shown in the following benchmarks that there are a few problems when solving beam structures. Using sheet metal and optimized thin model may be an alternative. Therefore this benchmark is also solved as a sheet metal model.

Number of nodes used for the sheet metal is 548 and number of elements 269.

![Figure 4.35, Loads and constraints.](image)
4.3.2 Modelling in I-deas

This is solved using parabolic beam elements and the model has converged at 10 elements. It is very easy to build the beam-elements using the dimensions from the blueprint.

4.3.3 Results

4.3.3.1 Resonance frequencies

The resonance frequencies are presented in table 4.5.

<table>
<thead>
<tr>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Mode 4</th>
<th>Mode 5</th>
<th>Mode 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inventor, solid</td>
<td>15.868</td>
<td>24.509</td>
<td>50.877</td>
<td>78.126</td>
<td>103.84</td>
</tr>
<tr>
<td>Inventor, sheet</td>
<td>16.21</td>
<td>24.12</td>
<td>52.12</td>
<td>77.21</td>
<td>107.53</td>
</tr>
<tr>
<td>I-deas</td>
<td>15.7</td>
<td>23.4</td>
<td>50</td>
<td>74.5</td>
<td>102</td>
</tr>
<tr>
<td>Difference</td>
<td>1.1 %</td>
<td>2.9 %</td>
<td>1.3 %</td>
<td>1.8 %</td>
<td>1.8 %</td>
</tr>
</tbody>
</table>

4.3.3.2 Stresses

On this benchmark, the stresses and deformation at the middle of the beam is examined.

**Inventor:**
The result from solid model is presented in Figure 4.36-4.37, and the result from sheet metal model is shown in Figure 4.38-4.39.
The maximum stress is about 26 MPa, but this is on the constraint. A constraint that is more stiff than in the real life, so the stress will not be that high there. This will however not affect the values in the middle of the beam, where the stress is about 11.5-14.5 MPa and the deformation is 1.76 mm at maximum.
Figure 4.38, Equivalent stress - Sheet metal.

Figure 4.39, Deformation - Sheet metal.

The equivalent stress is about 12.9 – 15.5 MPa and the deformation 1.87 mm.
**I-deas:**
As for the other benchmarks solved with beam elements, it is hard to read the result on the figures, but the maximum equivalent stress (Figure 4.40) at the middle of the beam is about 13 MPa and maximum deformation (Figure 4.41) is 1.77 mm.

*Figure 4.40, Equivalent stress.*

*Figure 4.41, Deformation.*

The results are also presented in table 4.6 together with the comparison.
Table 4.6, Equivalent stress and deformation.

<table>
<thead>
<tr>
<th></th>
<th>Equivalent stress</th>
<th>Deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor, solid</strong></td>
<td>11.5-14.5 MPa</td>
<td>1.76 mm</td>
</tr>
<tr>
<td><strong>Inventor, sheet</strong></td>
<td>12.9-15.5 MPa</td>
<td>1.87 mm</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>13 MPa</td>
<td>1.77 mm</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>10.4 %</td>
<td>0.6 %</td>
</tr>
</tbody>
</table>

4.3.4 Comparisons and conclusions

These 2 models are quite different; one is solved with solid elements and the other with beam elements, so slightly different results are expected. They do give a similar result when it comes to resonance frequencies. The equivalent stress at the middle of the beam is also almost the same, about 13 MPa and the percentage shows that the values are pretty close to each other. But the solving time in Inventor was several minutes and only a few seconds in I-deas, so on this benchmark Inventor was quite inefficient.

Table 4.6 shows that the sheet metal model gives almost the same result as the solid model. The stress is a bit higher and the deformation is a bit larger, but they are still very good values. And the solving times was just seconds. One is however not able to make a convergence test using optimized thin model.

This shows that sheet metal model may be an alternative if the normal solid model is hard to solve.
4.4 Benchmark 4

This benchmark (Figure 4.42) is another beam structure from a water jet cutting machine:

![Figure 4.42, Benchmark 4.](image)

The material is SS 1312 with a Young’s modulus of 220 GPa and Poisson’s ratio of 0.275. The beam length is 6680 mm.

4.4.1 Modelling in Inventor

Like for the last benchmark all the features like small holes is suppressed, so that only the two beams and the two plates is included in the analysis. But it still was not possible to mesh it. A message was shown saying that the differences between the largest and the smallest dimension where to big. In this case the cross section of the beam is too small in comparison with the length. So this benchmark is shortened from the length 6680 mm to 4000 mm. Now it can be meshed. This will of course not give the right result in comparison with the real benchmark, but it is still interesting to solve. A fixed constraint is placed on one of the “plates” and a frictionless constraint is placed on the other. A load of 981 N is placed on each beam on the long surfaces. The loads and constraints are shown in Figure 4.43.

Number of nodes used is 349801 and the number of elements 223427.
4.4.2 Modelling in I-deas

Parabolic beam elements are used to solve this. The mesh is shown in Figure 4.44. As for benchmark 3 this beam is simple and easy to make beam elements of. Each beam is made of 10 beam elements. In this case also the "plates", that are in the shape of a thin beam, are meshed with beam elements.

Figure 4.44, The element mesh.
4.4.3 Results

4.4.3.1 Resonances frequencies

The resonance frequencies from both Inventor and I-deas are presented in table 4.7.

<table>
<thead>
<tr>
<th></th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Mode 4</th>
<th>Mode 5</th>
<th>Mode 6</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor</strong></td>
<td>35.52</td>
<td>35.62</td>
<td>62.25</td>
<td>62.303</td>
<td>96.511</td>
<td>96.815</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>33.4</td>
<td>33.4</td>
<td>57.6</td>
<td>57.6</td>
<td>90.6</td>
<td>90.6</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>6 %</td>
<td>6 %</td>
<td>7.5 %</td>
<td>7.5 %</td>
<td>6.2 %</td>
<td>6.2 %</td>
</tr>
</tbody>
</table>

4.4.3.2 Stresses

**Inventor:**
The result from Inventor is shown in Figure 3.45-4.46.

![Figure 4.45, Equivalent stress.](image)
The maximum deformation is about 0.22 mm and the equivalent stress is about 2-3 MPa in the middle of the beams. These results were made not using result convergence. A result convergence has been done and that took almost 20 minutes to solve (see the number of nodes and elements presented in chap 4.4.1) and the model did not converge where the beams are fastened. Modifications to make it converge has not been done; it would take too much time (hours). Normally this would be done, but on this case it is obvious that Inventor is not so good on this benchmark. However, the result at the middle of the beam has converged, so these values can be used.

**I-deas:**
Once again, the results can be hard to read from the following figures (Figure 4.47-4.48), but in I-deas one are able to get the result from any node/element, and the equivalent stress at the middle of the beam is 2.02 MPa and the deformation is 0.27 mm.
Figure 4.47, Equivalent stress.

Figure 4.48, Deformation.
The comparison between Inventor and I-deas is made in table 4.8.

<table>
<thead>
<tr>
<th>Table 4.8, Equivalent stress and deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equivalent stress</td>
</tr>
<tr>
<td>-------------------</td>
</tr>
<tr>
<td><strong>Inventor</strong></td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
</tr>
<tr>
<td><strong>Difference</strong></td>
</tr>
</tbody>
</table>

### 4.4.4 Comparisons and conclusions

Since it is hard to get the exact value from Inventor, comparison is not that easy. Table 4.8 shows that the difference is some ware between 1-30 % which is quite inconclusive. But since both software gives a result around 2 MPa one can say that the result is ok. The difference in deformation is also ok.

The resonance frequencies are very close to each other, so in this case Inventor gives a good result.

However, the model in Inventor did not converge so the maximum stresses, at the ends, are not verified. And the model had to be shortened to be solved, and still the solving time was about 20 minutes during the convergence test.

This shows that Inventor is not very good to solve beam structures this long, and I would not recommend people to solve this type of structure in Inventor.
4.5 Benchmark 5

This benchmark (Figure 4.49) is yet another beam structure from a water jet cutting machine:

![Figure 4.49, Benchmark 5.](image)

The material is SS 1312 with a Young’s modulus of 220 GPa and Poisson’s ratio of 0.275.

4.5.1 Modelling in Inventor

This part is cleaned as much as possible by suppressing as all uninteresting features that do not affect the end result, like chamfers and small holes located far from any critical points. The bottom of all the legs and the outer side of the 2 legs at the ends of the structure are then fully constrained. A load of 1000 N is placed on the side surface of the upper beam. This is shown in Figure 4.50.

This part did not work to do a result convergence, when doing this the software stopped working and shut itself down, (this benchmark is tested several times on 2 different computers with the same result). But it is solved using as fine mesh as possible.

Number of nodes that been used is 102411 and number of elements 52854.
4.5.2 Modelling in I-deas

Once again parabolic beam elements have been used to solve the structure. It consists of 2 different beam sections, and the resulting mesh is shown in Figure 4.51. Total number of elements is 55.
4.5.3 Results

4.5.3.1 Resonance frequencies

Also on this benchmark a modal analysis is made. The resonance frequencies are presented in table 4.9.

<table>
<thead>
<tr>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Mode 4</th>
<th>Mode 5</th>
<th>Mode 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inventor</td>
<td>43.217</td>
<td>64.34</td>
<td>76.582</td>
<td>111.48</td>
<td>120.69</td>
</tr>
<tr>
<td>I-deas</td>
<td>44.4</td>
<td>65.9</td>
<td>77</td>
<td>114</td>
<td>127</td>
</tr>
<tr>
<td>Difference</td>
<td>2.7%</td>
<td>2.4%</td>
<td>0.5%</td>
<td>2.3%</td>
<td>5%</td>
</tr>
</tbody>
</table>

4.5.3.2 Stresses

On this benchmark, the result of the middle of the structure is examined.

*Inventor:*

The result is shown in Figure 4.52-4.53.

*Figure 4.52, Equivalent stress of the middle of the structure.*
Figure 4.53, Deformation of the middle of the beam.

The figures show that the equivalent stress on the leg is about 1 MPa and the maximum deformation is 0.017 mm.
**I-deas:**
The result is shown in figure 4.54-4.55.

**Figure 4.54,** Equivalent stress at one of the legs.

**Figure 4.55,** Deformation.
It is hard to see on these figures but the equivalent stress at the leg is about 1.4 MPa and the maximum deformation is 0.03 mm. The comparison is made in table 4.10.

<table>
<thead>
<tr>
<th>Table 4.10, Equivalent stress and deformation.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Equivalent stress</strong></td>
</tr>
<tr>
<td>Inventor</td>
</tr>
<tr>
<td>I-deas</td>
</tr>
<tr>
<td>Difference</td>
</tr>
</tbody>
</table>

4.5.4 Comparisons and conclusions

On this benchmark there were problems with the result convergence. It stopped during the first refinement and crashed the program so the results that are shown here are not tested for convergence. For the resonance frequencies this is not a problem, and the results are very close to I-deas. As for the stress and deformation, it is hard to say. Both stress and deformation is rather close to I-deas (but still 29 & 44 % off), but since they are not convergence tested they are not trustable. This is however probably just a bug in the software and may be fixed in future versions.

This shows that Inventor is not the best program when it comes to large, in this case 9 m, beam structures, when looking for the stresses. It does however work fine when it comes to the mode shapes and frequencies. It did take some time to solve, but the result is good.
4.6 Benchmark 6

This benchmark (Figure 4.56) is one of the rails that are located on the X-beam. Once again the material SS 1312 is used, with a Young’s modulus of 220 GPa and Poisson’s ratio of 0.275.

![Figure 4.56, Benchmark 6.](image)

4.6.1 Modelling in Inventor

The holes are suppressed; this is the only thing that can be done to simplify the part. When meshing it the same message as in Benchmark 4 was presented. The cross section is too complicated for the length. It had to be shortened to 1 m before it was possible to mesh it.

One side is fully constraint and then two different solutions are made. One using a moment of 10000 N mm on the other side, and one using a force of 100 N. This is shown in Figure 4.57.

The result convergence did not work on this part either, same problem as in Benchmark 5. But this is solved using as fine mesh as possible.

Number of nodes is 22588 and number of elements 4640.
4.6.2 Modelling in I-deas

This is the last benchmark where parabolic beam elements are used to solve the part. By cutting off a bit of the part and using import/export step-file, the shape was transferred to I-deas. By using wireframe the shape of the part is used to make a new beam cross section. The part is then solved with 10 elements.
4.6.3 Results

4.6.3.1 The moment test

*Inventor:*
The result is shown in Figure 4.58-4.59.

*Figure 4.58, Equivalent stress using moment.*

*Figure 4.59, Deformation using moment.*
The figure shows that the maximum equivalent stress is 2.74 MPa and the deformation is 0.35 mm.

**I-deas:**
The results are shown in figure 4.60-4.61.

**Figure 4.60, Deformation using moment.**

Maximum deformation is 0.35 mm and max equivalent stress is 2.68 MPa.
The comparison is made in table 4.11
Table 4.11, Equivalent stress and deformation for the moment test.

<table>
<thead>
<tr>
<th>Equivalent stress</th>
<th>Deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor</strong></td>
<td>2.74 MPa</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>2.68 MPa</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>2.8 %</td>
</tr>
</tbody>
</table>

4.6.3.2 The load test

*Inventor:*

The result is shown in figure 4.62-4.63.

*Figure 4.62, equivalent stress using force.*
Figure 4.63, Deformation using force.

Maximum equivalent stress is 19.79 MPa and the deformation is 1.8 mm.

I-deas:
The result is shown in figure 4.64-4.65.

Figure 4.64, Deformation using force.
Maximum equivalent is about 19.7 and the deformation is about 1.8. The comparison is made in table 4.12.

<table>
<thead>
<tr>
<th>Equivalent stress</th>
<th>Deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor</strong></td>
<td>19.79 MPa</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>19.7 MPa</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>0.5%</td>
</tr>
</tbody>
</table>

### 4.6.4 Comparisons and conclusions

Once again Inventor showed that this type of parts (long beam structure with a bit complicated cross section) is not possible to solve. This benchmark was over 4 m and it was not possible to mesh it. It was possible to mesh it when it was shortened to 1 m but result convergence did not work (same bug as in benchmark 5).

But a comparison with I-deas on this shortened part shows that the results are correct. The values are fine and the stress distribution over the cross section looks the same in Inventor and Ideas. This shows that even when it comes to
beam structures, Inventor will give fine results as long as the part is small and rather simple.

But Inventor did not manage to solve this benchmark, and are therefore still not ideal when it comes to beams.

### 4.7 Benchmark 7

The next two benchmarks are very much alike; they are the 2 carriers of a water jet cutting machine. Different test are made to them even though it would be possible to do all tests on just one of them. The benchmark is shown in Figure 4.66.

The material is SS1312, with a Young’s modulus of 220 GPa and Poisson’s ratio of 0.275.

![Figure 4.66, Benchmark 7.](image)
4.7.1 Modelling in Inventor

To save time solving this part, uninteresting features, like holes and chamfers has been suppressed. One hole is left because it makes the benchmark a bit more interesting; it is not clear where the critical point is. The two wedges are also deleted because they do not affect the result much, and it is unnecessary to fill these with solid element. Finally a fillet of 8 mm is made where the expected maximum stress will be, to take care of the stress singularity that will occur otherwise. The result of the modifications is shown in Figure 4.67.

The model is then fully constraint on the bottom right surface, and a bearing force of 981 N is placed in the holes of each the two holders, pushing the part. These loads and constraints are also shown in Figure 4.67.

Number of nodes used is 90787 and number of elements 60596.

Figure 4.67, Simplified benchmark with loads and constraints.
4.7.2 Modelling in I-deas

Step is used to export/import the simplified model. By using partition the two holders and the “bottom part” are separated from the big “plate”. Solid parabolic tetrahedronal elements are used for the bottom piece and quadratic triangular shell element for the plate and the two holders are used. By refining the elements the model is tested and it is converging. The final mesh is shown in Figure 4.68.

Number of nodes used is 64851 and number of elements 40109.

Figure 4.68, Element mesh at the stress maximum.
4.7.3  Results

4.7.3.1  Resonance frequencies

For this part the resonance frequencies where calculated, and the result is presented in table 4.13.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Mode 1</th>
<th>Mode 2</th>
<th>Mode 3</th>
<th>Mode 4</th>
<th>Mode 5</th>
<th>Mode 6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inventor</td>
<td>44.17</td>
<td>140.77</td>
<td>272.82</td>
<td>284.51</td>
<td>560.26</td>
<td>862.81</td>
</tr>
<tr>
<td>I-deas</td>
<td>44.63</td>
<td>140</td>
<td>270</td>
<td>285</td>
<td>560</td>
<td>854</td>
</tr>
<tr>
<td>Difference</td>
<td>1.1 %</td>
<td>0.5 %</td>
<td>0.9 %</td>
<td>0.2 %</td>
<td>0 %</td>
<td>1 %</td>
</tr>
</tbody>
</table>

Actually these frequencies are in I-deas calculated using parabolic tetrahedronal solid elements on the whole part. The frequencies from the model with shell elements differed a bit from these values. Not much but to be on the safe side a solution with solid elements was made. These results were a bit better and are therefore presented here. The solution time was because of this longer then for the stress test, but still much shorter then Inventor.

4.7.3.2  Stresses

Inventor:

The result is shown in Figure 4.69-4.70.

On this model it actually looked like the stresses converged without using a result convergence. By trying different mesh sizes between 0 and 100 it seemed that the model converged and the maximum equivalent stress was about 22 MPa at the edge of the small hole. But a result convergence was made anyway, and this showed that the stress was 27 MPa at the hole. The reason to this is that even though the mesh size is changed the number of elements did not change much around the hole; there is a big difference between the “converged” mesh and the other.
The figure shows that the maximum equivalent stress is at the hole and not at the "bend". But the maximum principal stress is there as expected:
Maximum equivalent stress is 27.6 MPa and maximum principal stress 30.7 MPa.

**I-deas:**
The result is shown in Figure 4.71-5.72.

*Figure 4.71, Max Equivalent stress at the hole.*
Maximum equivalent stress is 30 MPa and maximum principal stress 29.1 MPa.

The comparison is made in tale 4.14.

<table>
<thead>
<tr>
<th></th>
<th>Equivalent stress</th>
<th>Max principal stress</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor</strong></td>
<td>27.6 MPa</td>
<td>30.7 MPa</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>30 MPa</td>
<td>29.1 MPa</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>8 %</td>
<td>5.3 %</td>
</tr>
</tbody>
</table>

4.7.4 Comparisons and conclusions

The frequencies are very close in Inventor and I-deas. As said, solid elements are used to calculate the frequencies in I-deas. The result is not very far from using shell elements but still a little different. The important thing is that Inventor and I-deas gave the same result.

Also the stresses are verified. The difference between the two software was 8 % or less on both equivalent and principal stress. This is a good result.
On this benchmark Inventor worked very well, and was much easier to use than I-deas. The solution time was a bit longer, but it was as said very easy to use and gave a very good result.

### 4.8 Benchmark 8

This is the other carrier (Figure 4.73):

![Image](https://example.com/image.png)

*Figure 4.73, Benchmark 8.*

The material is SS1312, with a Young’s modulus of 220 GPa and Poisson’s ratio of 0.275.

#### 4.8.1 Modelling in Inventor

The same simplifications as on benchmark 7 are made, suppressing all details except the small hole close to the critical point, and on this one the two large holes are also still included. The same fillet is also made and a new fillet of 2 mm around the upper holder, to make the model converge. This is constrained
the same way as the former benchmark, but on this one the loads are put vertical on top of the two holders instated of horizontal. This is shown in Figure 4.74.

Number of nodes used is 91524 and number of elements 60823.

4.8.2 Modelling in I-deas

This is modelled in the same way as the former benchmark except that parabolic tetrahedronal solid elements is used on the two holders instead of shell elements. Number of nodes used is 65341 and number of elements 40412.
4.8.3 Results

Inventor:
The result is shown in Figure 4.75-4.76.

**Figure 4.75, Equivalent stress.**

**Figure 4.76, Maximum equivalent stress around the upper holder.**
The maximum equivalent stress is located at the upper holder and is 11.77 MPa.

**I-deas:**
The result is shown in Figure 4.77.

![Figure 4.77, Max equivalent stress.](image)

The maximum stress is 11.2. The comparison is made in table 4.15.

<table>
<thead>
<tr>
<th>Equivalent stress</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor</strong></td>
<td>11.77 MPa</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>11.2 MPa</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>4.9 %</td>
</tr>
</tbody>
</table>

### 4.8.4 Comparisons and conclusions

As for benchmark 7, Inventor gave a very good result on this one. The difference is less than 5 %, which is great. The solution time was a bit longer but still Inventor was very convenient to use.
4.9 Benchmark 9

This (Figure 4.78) is one of the holders of the X-beam:

![Figure 4.78, Benchmark 9.](image)

The material is SS2172 with a Young’s modulus of 200 GPa and Poisson’s ratio 0.287.

4.9.1 Modelling in Inventor

As on the other benchmarks the small holes and other features are suppressed to get a simplified model. The model is then constrained as shown in Figure 4.79. On the upper surface a pressure of 10 MPa is placed.

Number of nodes used is 13969 and number of elements 8199.
4.9.2 Modelling in I-deas

This is a rather small part that at the same time is a bit complicated. Therefore is parabolic tetrahedronal solid elements used to solve this in I-deas. Number of nodes is 39021 and number of elements 23581.
4.9.3 Results

*Inventor:*
For this benchmark only the equivalent stress (Figure 4.80) is examined:

![Figure 4.80, Equivalent stress.](image)

The maximum equivalent stress is 87.013 MPa
**I-deas:**
The result is shown in Figure 4.81.

![Figure 4.81, Equivalent stress.](image)

Maximum equivalent stress is 95.1 MPa. The comparison is made in table 4.16.

| Equivalent stress | Inventor 87.013 MPa | I-deas 95.1 MPa | Difference 8.6 % |

### 4.9.4 Comparisons and conclusions

The maximum stress value differs a bit between the two softwares on this benchmark. Both use solid elements but there are still a little different. This can have several reasons. In Inventor the result convergences concede a value to converge if it does not change more than 10 percent. The value from Inventor is less than 10 percent smaller than the value from I-deas. If Inventor made a refinement more it may have gotten a value even closer to I-deas.
All this said, this solution is verified. Inventor gives a fairly good result. It took long time to solve though. Almost 10 minutes compared with about 3 minutes in I-deas.

### 4.10 Benchmark 10

This benchmark (Figure 4.82) is the part that the carriers in benchmark 7 and 8 are fastened to and that travels along the rails:

![Figure 4.82, Benchmark 10.](image)

The material is chosen to stainless steel with a Young’s modulus of 193 GPa and a Poisson’s ratio 0.3.

#### 4.10.1 Modelling in Inventor

As usual the holes and other features that are not interesting are deleted. The simplified model is shown in Figure 4.83.
Two different tests are made on this benchmark. One where one of the two contact surfaces are constraint using fixed constraint and a load of 100 N is applied on the other, and one where both of the contact surfaces are fully constraint and a pressure of 50 MPa is applied on the large middle surface.

Number of nodes used is 292132 and number of elements 204080 for the load test and 98197 nodes and 64111 elements are used for the pressure test.

### 4.10.2 Modelling in I-deas

As in benchmark 9 this is solved with parabolic tetrahedronal solid elements. Number of nodes used is 59519 and number of elements 38193.
4.10.3 Results

4.10.3.1 The load test

*Inventor:*
The interesting results for this benchmark are the equivalent stress (Figure 4.84) and the deformation (Figure 4.85).

![Figure 4.84, Equivalent stress.](image)

![Figure 4.85, Deformation.](image)
The maximum equivalents stress is 0.61 MPa and maximum deformation is 0.0015 mm.

**I-deas:**
The result is shown in Figure 4.86-4.87.

I-deas found a maximum stress of 0.75 MPa where the part is constraint, but looking at the same place is where Inventor found the maximum, I-deas gets an equivalent stress of 0.59 MPa. The deformation is 0.0013 mm.

*Figure 4.86, Equivalents stress.*
The comparison is presented in table 4.17.

Table 4.17, Equivalent stress and deformation for the moment test.

<table>
<thead>
<tr>
<th></th>
<th>Equivalent stress</th>
<th>Deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inventor</td>
<td>0.62 MPa</td>
<td>0.0015 mm</td>
</tr>
<tr>
<td>I-deas</td>
<td>0.59 MPa</td>
<td>0.0013 mm</td>
</tr>
<tr>
<td>Difference</td>
<td>4.9 %</td>
<td>13.4 %</td>
</tr>
</tbody>
</table>
4.10.3.2 The pressure test

*Inventor:*
The result is shown in Figure 4.88-4.89.

*Figure 4.88, Equivalent stress.*

*Figure 4.89, Deformation.*
Maximum equivalents stress is 1261.5 MPa and maximum deformation is 0.2 mm.

*I-deas:*  
The result is shown in Figure 4.90-4.91.

*Figure 4.90, Equivalent stress.*

Maximum equivalent stress is 1370 MPa and maximum deformation is 0.24 mm.

*Figure 4.91, Deformation.*
The comparison is presented in table 4.18.

<table>
<thead>
<tr>
<th>Equivalent stress</th>
<th>Deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor</strong></td>
<td>1261.5 MPa</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>1370 MPa</td>
</tr>
<tr>
<td><strong>Difference</strong></td>
<td>8 %</td>
</tr>
</tbody>
</table>

4.10.4 Comparisons and conclusions

Equivalent stress for the moment is only off by 8 %, which is a good result. The deformation is off by 16.7 %. This is a bit too much but it still is only 0.04 mm. For the load test the deformation is only off by 0.0002 mm or 13 %. All these values are verified; Inventor gives a fairly good result for this benchmark.

The solving time in Inventor was very long for this benchmark, almost 15 minutes compared with 2-3 minutes in I-deas. But still, it gave a good result.
4.11 Benchmark 11

On this last benchmark (Figure 4.92) the optimized thin model mesh is tested using sheet metal. The part tested is a very simple sheet metal part that could for example be a holder for a wooden beam.

![Figure 4.92, Benchmark 11.](image)

The thickness of the sheet is 0.5 mm and the material is stainless steel with a Young’s modulus of 193 GPa and a Poisson’s ratio 0.3.

4.11.1 Modelling in Inventor

A fixed constraint is put on the two upper surfaces and a load of 100 N is placed on the middle surface (see Figure 4.93). This model is then solved with solid element and result convergence (it would not converge otherwise), and after that solved with optimized thin model.

Using solid elements the number of nodes were 127872 and the number of elements were 73311 and with thin model the number of nodes were 1098 and the number of element were 500.
4.11.2 Modelling in I-deas

Quadratic triangular shell elements are used to solve this in I-deas. Number of nodes used is 34987 and number of elements are 17370. This is quite a large number of nodes and elements, and it is possible to solve this with considerably less number. This is solved with these numbers just to be “on the safe side”. On this small and simple part, this number will not cause any trouble.

4.11.3 Results

Solid elements, Inventor:
This solution took very long time to solve, about 5-10 minutes. The result is shown in Figure 4.94-4.95.
Figure 4.94, Equivalent stress.

Figure 4.95, Deformation.

Maximum equivalent stress is 126 MPa and maximum deformation 0.56 mm.
A close-up of the max eq. stress (Figure 4.96) shows that the no of elements across the cross section, that is 0.5 mm thick, is about 10. This is very much for such a thin part.

![Figure 4.96, Close up of number of elements.](image)

**Optimized thin model, Inventor:**

This solution was very fast, about 30 sec. The result is shown in Figure 4.97-4.98.

![Figure 4.97, Equivalent stress.](image)
Maximum equivalent stress is 122 MPa and maximum deformation is 0.55 mm.

A close-up (Figure 4.99) of the max stress shows the no of elements across the thickness. In this case there is only one element.
**I-deas:**
The result is shown in Figure 4.100-4.101.

*Figure 4.100, Equivalent stress.*

*Figure 4.101, Deformation.*
Maximum equivalent stress is 125 MPa and maximum deformation is 0.47 mm. The comparison between the optimized model and solid as well as the comparison between Inventor and I-deas is made in table 4.19.

<table>
<thead>
<tr>
<th></th>
<th>Equivalent stress</th>
<th>Deformation</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Inventor, solid</strong></td>
<td>126 MPa</td>
<td>0.56 mm</td>
</tr>
<tr>
<td><strong>Inventor, optimized</strong></td>
<td>122 MPa</td>
<td>0.55 mm</td>
</tr>
<tr>
<td><strong>I-deas</strong></td>
<td>125 MPa</td>
<td>0.47 mm</td>
</tr>
<tr>
<td><strong>Difference, optimized/solid</strong></td>
<td>3.2 %</td>
<td>1.8 %</td>
</tr>
<tr>
<td><strong>Difference, I-deas/solid</strong></td>
<td>0.8 %</td>
<td>16.1 %</td>
</tr>
</tbody>
</table>

### 4.11.4 Comparisons and conclusions

First of all, the results from the two softwares are very close, so the solution is verified. There is a slightly large difference in the percentage of the deformation, but it is still a fairly good result.

But the interesting thing here is the two different tests in Inventor. The difference between optimizes thin model and the normal solid mesh are not very large, only 3.2 %. And the thin model was much faster; it was even faster then I-deas. So at least for simple sheet metal parts this option is much more efficient and should be used when possible.
5 Discussions and conclusions

The stress analysis tool in Inventor is claimed to be aimed to designers and not specialists. It is supposed to help during the modelling work and not to be used as a final calculations tool. This thesis work shows that this is the case.

The main advantage in this software is that it is very easy to use. The modelling is very fast, although the solving time can be very long, especially when using result convergence. To get the exact values of the stresses it is shown that one need to use the result convergence on almost any part (all the benchmarks). However, the difference between the converged values and the values gotten without result convergence are not that far apart, often just about 10-25%. During the early modelling work it is therefore not always necessary to use the result convergences. This is because of the large safety factors that often are used when designing. When a designer use the analysis tool to get a hint if the model works, the exact stress values are not so interesting. If 120 MPa is too close to the safety factor than 130 is too close too. So the solving time can be faster if one is not after the exact values.

It is shown that Inventor gives very good results when it comes to modal analysis. The problems that has occurred during this thesis work, has all been connected to stress analysis, often when using result convergence.

There is one big problem with this software. One cannot control the real mesh size, only the relative size. This was a problem in benchmark 4 and 6 where it was impossible to mesh the part because the difference between the longest and shortest dimension was too large. This is a very big problem, because the usability of the software is limited.

But as long as one can mesh the parts on gets a fairly good result. Depending on how exact values one wants the solving time varies. It is possible to get very good result but it takes time.

There are a few things to consider when using this software:

- The part cannot be too complicated e.g. long and thin. If this occurs one may be able to solve it using sheet metal and optimized thin model.
• To save solving time when using result convergence it is important to consider the places where stress singularities will occur.
• If the model is converged without using result convergence, it is important to check the mesh at the stress maximum. Sometimes the mesh is not refined at the maximum when changing the mesh size. See benchmark 7.
• If using sheet metal use the optimized thin model when possible. This saves much time, just bear in mind that one is not able to make a convergence test.
• Always suppress as many features as possible to get a more efficient and faster solution.

If one considers these few points, then Inventor will give a fairly good result, and will work as a help during designing work.
6 References


