MADYMO 5.1 finite element module

Pape, K.W.; Bekkers, F.P.J.

Published: 01/01/1994

Document Version
Publisher's PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

• A submitted manuscript is the author's version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

Citation for published version (APA):

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal

Take down policy
If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

Download date: 30. Dec. 2018
Title: MADYMO 5.1 FINITE ELEMENT MODULE

Practical training report by:
K.W. Pape       id.nr. 310073
F.P.J. Bekkers  id.nr. 264145

Supervisors:
Ir. E. Poeze (TNO-WT)
Ir. R.Y. Schouten (TNO-WT)
Dr.ir. A.A.H.J. Sauren (TUE)

Delft, March 1994
## CONTENTS

1 • NOTATIONS ABOUT THE MADYMO 5.1 FINITE ELEMENT MODULE

1.1 Introduction ........................................... 4
1.2 A Brief History ....................................... 4
1.3 Basic Ideas ........................................... 4
1.4 Fundamentals .......................................... 5
1.5 A system of elements ................................. 7
1.6 Numerical integration ................................. 8

2 • ELEMENT & MATERIAL TYPES IN MADYMO 5.1

2.1 Introduction ........................................... 10
2.2 Truss2 Element ........................................ 10
2.3 Beam2 Element ........................................ 10
2.4 Mem3 Element ......................................... 11
2.5 The Facet6 & Shell6 Element ......................... 11
2.6 The Shell4 Element ................................... 12
2.7 The Solid1 & Solid8 Element ......................... 13
2.8 Linear elastic isotropic material behaviour ......... 13
2.9 Linear elastic orthotropic & anisotropic material behaviour 14
2.10 Linear elastic material behaviour with hysteresis 14
2.11 Elastic plastic material behaviour ................. 14

3 • BETA-SITE TESTS OF MADYMO’S FINITE ELEMENT MODULE

3.1 Introduction ........................................... 15
3.2 Rigid body movements ............................... 15
3.3 Continuum mechanics ................................. 16
3.4 Two dimensional membranes ........................ 18
3.5 Testing material models in MADYMO ............. 19
3.6 Using mixtures of element types ................... 19
3.7 Problems detected ................................... 20

4 • HOURGLASS MODES

4.1 Introduction ........................................... 21
4.2 Shape functions ...................................... 21
4.3 One-point integration of the element volume .... 25
4.4 Hourglass control .................................... 26
5. STRESS ANALYSIS USING MOHR'S CIRCLE

5.1 Introduction

5.2 Problem description

5.3 Examining the stresses in the element

5.4 Summary

6. REFERENCES

APPENDICES
1.1 Introduction

Until recently MADYMO offered a modest finite element module, developed mainly to model airbags in crash simulations. Studying vehicle deformations and detailed human body segment models requires nevertheless a more sophisticated use of finite element techniques. To enlarge the applicabilities and virtues of the code, the finite element module is extended in the 5.1 version. The combination of the multibody module and this new finite element module makes MADYMO a powerful tool in crash safety simulations. The scope of this report will be solely on the new finite element module. This chapter is concerned with a brief introduction to the finite element method as it is implemented in MADYMO.

1.2 A Brief History

The finite element method has passed into common usage in all branches of engineering. At present many computer codes are available, all with their specific purposes. However, it took a long time for the finite element concept to achieve its broad applicability. The idea was stimulated by the work of the mathematician Courant, but the development of the technique was initialised by others at the end of the nineteenth century. Progress in development was slow until 1920 due to practical limitations of solving algebraic equations with multiple unknowns. In the early fifties of this century researchers began to develop a method which relied totally on the digital computer. Almost all of these pioneers had connections with the aircraft industry since this sector derived much benefit from the possibilities of the new technique. Moreover, this industry had the necessary research capital to buy the very expensive early digital computer. In the succeeding decades the finite element method was developed, parallel with the rise of the computer, to its recent status.

1.3 Basic Ideas

Computers can be instructed to describe structures in terms of continuous functions, but this is not a feasible option for complex constructions. Discretization is therefore a keyword in the finite element method. The basic idea is simple. Firstly, a structure is divided into elements which are so small that variations of the displacement and stress fields can be approximated. The problem is then spatially discretised. Secondly, all elements have to
be assembled together to ensure continuity along all element boundaries. The elements are connected to a discrete number of points, the nodes. The identification of these material points is very important in studying the continuum deformation and displacement.

There are two different ways to describe the system kinematics. The Lagrangian method, which is normally used in solid mechanics, regards the entire continuum or part of it. Independent variables in this method are the initial state of the body and the time. The other method which is applied mainly in fluid mechanics is the Euler description. This method considers a part of the space, where the actual state and the time are the independent variables.

MADYMO uses a Lagrangian description, which means that the elements are fixed to the material and move through space. The element formulation is based on displacements because of its simplicity. Motion of points within an element is defined as a function of the motion of the element nodes. This is achieved by choosing a relationship between the displacements at any arbitrary point in the element and the nodal points. This relationship is established via the shape functions and it determines how specific magnitudes vary across the element.

1.4 Fundamentals

In solving structural problems there are three different physical principles that have to be considered. These principles must always be satisfied, whatever the type of loading, be it static or dynamic and whatever the properties of the structure’s material. The principles are:

1. Equilibrium. These laws relate stresses to applied forces. If the structure is excited dynamically, inertia forces are inserted in the equations of equilibrium.

2. Compatibility. Geometrical laws define strains as a function of displacements. These relationships depend on the type of deformation and the geometry of the particular structure.


The equations of equilibrium for a certain structure can be reflected by the principle of virtual displacements, which relates internal to external work using the product of forces and virtual displacements.
The internal virtual work for a 3D structure will be:

\[ W_{\text{int}} = \int_{V} \sigma \varepsilon \, dV \quad (1.1) \]

where \( \sigma \) and \( \varepsilon \) reflect the column of stress and strain components within the structure respectively. The external virtual work done by body and surface forces is:

\[ W_{\text{ext}} = \int_{V} \mathbf{P} : \varepsilon \, dV + \int_{S} \mathbf{P} : \mathbf{u} \, dS \quad (1.2) \]

To satisfy equilibrium the internal work must equal the external work and thus the principle of virtual displacements is:

\[ \int_{V} \sigma \varepsilon \, dV = \int_{V} \mathbf{P} : \varepsilon \, dV + \int_{S} \mathbf{P} : \mathbf{u} \, dS \quad (1.3) \]

In the finite element concept body and surface forces are concentrated in the element nodes. For static as well as for dynamic analysis, both internal and external nodal forces have to be known. By manipulating the internal virtual work a relationship will be derived between nodal displacements and internal nodal forces.

Via the shape functions a relationship can be established between nodal displacements \( d_n \) and displacements of an arbitrary point within the element:

\[ \mathbf{u} = N d_n \quad (1.4) \]

Now the principle of compatibility can be applied. This relationship between strains and displacements depends on the definition of strain:

\[ \varepsilon = \frac{\partial \mathbf{u}}{\partial \mathbf{x}} = \frac{\partial N}{\partial \mathbf{x}} d_n \quad (1.5) \]

\( N \) represents known shape functions and therefore differential operators can be applied to them. The differentiated shape functions can be denoted as \( \frac{\partial N}{\partial \mathbf{x}} = \mathbf{B} \) which results in:

\[ \varepsilon = B d_n \quad (1.6) \]

The constitutive law, which defines stresses as a function of strains, can be given by:

\[ \sigma = E \varepsilon \quad (1.7) \]

where \( E \) is the material stiffness matrix. The first term of equation (1.3), which reflects the internal virtual work can be rewritten using (1.6) & (1.7) as:
\[
\int \sigma^T e dV = \int \epsilon^T E \epsilon dV = d_n^T B^T E B dV d_n = d_n^T K_g d_n \]

(1.8)

where the element stiffness is defined as:

\[
K_g = \int B^T E B dV
\]

(1.9)

The work done by the applied forces in equation (1.3) can be rewritten using \( u = N d_n \), which gives:

\[
\int P_s^N N dV d_n + \int P_s^i N dV d_n = f_{ext}^i d_n
\]

(1.10)

The equations of equilibrium (1.3) now look like:

\[
d_n^T K_g d_n = f_{ext}^i d_n \rightarrow f_{int}^i d_n = f_{ext}^i d_n
\]

(1.11)

The relationship between internal nodal forces and the nodal displacements is given by:

\[
f_{int}^i = K_g d_n
\]

(1.12)

1.5 A system of elements

So far only a single element has been considered. For a system of elements the same theory can be applied. At this point most common finite element codes execute an element summation by constructing a system stiffness matrix. For MADYMO, however, the elementary part of the finite element concept ends with equation (1.12). The code has been developed primarily for dynamic analysis and therefore the systems equations of motion have to be known. These are, in contradistinction to most codes, expressed in index notation, because that is most suitable for the solution process MADYMO uses. This solution process will be described later on in this chapter.

Constitutive material properties which vary through a continuum have been spatially discretised and the continuum is now represented by a collection of nodes. Kinematic and constitutive relationships between these nodes can be stored in a column \( K \) containing all stiffness factors \( K_g \).

In a dynamic analysis the effects of inertia forces are considered in the calculation. In order to define such a problem the minimum information that has to be specified is the stiffness and the inertia of the system. For a response calculation some initial conditions for displacement and velocity are also required. In addition any real system will contain damping that dissipates the vibrational energy and a set of time-varying loads.
The principle of virtual displacements for a dynamic system yields the equations of motion. Internal and external forces have already been transformed to a nodal level. Mass is concentrated in the nodes via a process called lumping, which results in the mass column. Damping is specified as a linear function of mass and stiffness columns, which is known as Rayleigh damping. The resulting equations of motion for a system of finite elements can thus be denoted as:

\[ M\ddot{x} + C\dot{x} + Kx = F_{\text{ext}} \]  

(1.13)

where \( M \), \( C \) and \( K \) are the mass, damping and stiffness columns, \( F_{\text{ext}} \) is the applied load column and \( x \) a column containing the nodal displacements.

### 1.6 Numerical integration

After the equations of motion have been spatially discretised solving the resulting differential equation (1.13) remains to be done. This is done by numerical integration where the quantities are calculated at a number of discrete points in time. Basically there are two different methods for the numerical integration of differential equations; implicit and explicit methods. For explicit methods there is a timestep above which the numerical integration process becomes unstable. The timestep in implicit methods is only bounded by accuracy requirements. An explicit method has, in comparison to implicit methods, the advantage that the costs per timestep during the integration are much lower since no matrix decomposition techniques have to be used. Explicit methods are most suited to crash simulations. In crash simulations, deformations take place in a small time interval and thus the strain rates will be large. The timestep must be small to ensure the correct calculation of stresses. The disadvantages of the necessity of a small timestep turn out to be less apparent.

In MADYMO the central difference scheme with a fixed timestep is used. The relations for this method are given by:

\[ \dot{x}_{n,0.5} = \ddot{x}_{n} + \Delta t\dddot{x}_{n} \]  

(1.14)

\[ x_{n+1} = x_{n} + \Delta t\dot{x}_{n,0.5} \]  

(1.15)

To benefit from the efficiency of the central difference method it is necessary to assume Rayleigh damping. In MADYMO Rayleigh damping only dependent on the mass matrix is chosen.

\[ C = \gamma M \]  

(1.16)

Furthermore it is assumed that the internal forces can be written as:
\[ F_{ad} = \gamma M \ddot{x}_n + Kx_n \]  

(1.17)

Combination of (1.13), (1.14), (1.15) & (1.17) yields:

\[ \ddot{x}_{n,0.5} = \frac{2 - \gamma M}{2 \gamma M} \ddot{x}_{n,0.5} + \frac{2 \gamma M}{2 \gamma M} M^{-1}(F_{ad} - F_{ad}) \]  

(1.18)

\[ x_{n,1} = x_n + \Delta t \ddot{x}_{n,0.5} \]  

(1.19)

Equations (1.18) & (1.19) are fully explicit. In exchange for its simplicity, however, the central difference method is only conditionally stable and the timestep is bounded. It can be deduced that due to stability the timestep must satisfy:

\[ \Delta t \leq \frac{2}{\omega_{max}} \]  

(1.20)

where \( \omega_{max} \) is the maximum eigenfrequency appearing in the mesh. This is the so-called Courant criterion. This condition requires that the timestep is small enough to ensure that a sound wave may not cross the smallest element during one timestep. Since the speed of sound is a function of material properties, the critical timestep depends on the size of the smallest element and the constitutive properties of the material modelled.
2 ELEMENT & MATERIAL TYPES IN MADYMO 5.1

2.1 Introduction

In order to discretise a construction, finite elements are commonly used. In MADYMO 5.1 eight different elements can be used to build a discretised computer model of a continuous system. For the material behaviour, one can choose from of 7 different material properties. However, not all combinations of elements and materials are implemented in version 5.1. Here the elements and material types that can be used in MADYMO are given, with a brief description of their characteristics.

2.2 Truss2 Element

The Truss2 element is the most basic finite element in MADYMO 5.1. It’s a one-dimensional element, connecting 2 nodes. The mass is equally distributed over the two nodes. A Truss2 element can only conduct axial loads, no torsion or bending. So each node has three global translational degrees of freedom.

The element has a constant cross-sectional area which is defined by the user.

![Truss2 Element Diagram]

There are three options for the material behaviour of the Truss2 element at the moment: Isotropic elastic, Elasto-plastic and Hysteresis.

2.3 Beam2 Element

The Beam2 is a one-dimensional element, connecting 2 nodes, like the Truss2. The difference between a Truss2 and a Beam2 element lies in the fact, that a Beam2, besides axial tension and compression, can also handle torsion and bending. Each node has six degrees of freedom.

The geometrical input consists of the area moments of inertia and area of the cross-section of the beam.
There is one option for the material-behaviour of the Beam2 element: Isotropic elastic.

### 2.4 Mem3 Element

The Mem3 element is a two-dimensional element, connecting three nodes. It can carry only in-plane loads. Because the element cannot bend, the angular orientation of the nodes is of no importance. The deformations are fully determined by the three translational degrees of freedom of the nodes. The mass of the element is lumped and distributed over the three nodes conferring element distribution factors. These factors are proportional to the angle enclosed by the two element edges joining in the vertex.

The thickness of the element is presumed to be constant during the simulation and must be defined by the user.

The tension in the element is characterized by Cauchy stresses and is presumed to be constant over the element. The stresses are defined in the local coordinate system of the element.

The Mem3 Element has eight material options: Isotropic Elastic (tension/pressure, and tension only), Orthotropic Elastic (tension/pressure, and tension only), Anisotropic Elastic (tension/pressure, and tension only), Elasto-Plastic and Hysterisis.

### 2.5 The Facet6 & Shell6 Element

In contradistinction to a Mem3 element, a Facet6 (or Shell6) element can handle bending loads. For this purpose 3 rotational nodes where added to the element. See figure.

The difference between the Facet6 and Shell6 element is that the shell6 accounts for the coupling between membrane and bending stiffness for a curved shell, whereas the facet6 neglects this effect and is therefore expected to require less CPU time.
So the element has three translational nodes and three rotational nodes. The rotational nodes cannot handle forces or translations, just rotations and moments in one direction. The translational nodes cannot handle rotations or moments, just translations and forces. The geometrical properties needed are the thickness of the shell or facet element and the number of integration points over the thickness.

Material types Isotropic Elastic and Elasto-Plastic can be used for the Shell6 and Facet6. For Isotropic Elastic material behaviour only one integration point over the thickness is needed. For Elasto-Plastic material a minimum of two and a maximum of seven integration points can be chosen.

2.6 The Shell4 Element

The Shell4 element is a quadrilateral element that connects 4 nodes and can carry in-plane loads as well as bending loads. The Shell4 element has 20 degrees of freedom. In its local coordinate system, each node of the Shell4 has two rotational degrees of freedom in the element plane and three translational degrees of freedom. So each node can handle translations or forces and rotations or moments in three directions in the global system. The mass of the element is lumped and equally distributed over the nodes. As geometric data, thickness and the number of integration points over the thickness are needed.

The material types that can be used for the Shell4 Element are: Isotropic-Elastic and Elasto-Plastic.

It's possible to use the Shell4 element as a triangular element. This is done by collapsing the fourth node onto the third node. A triangular element results with its rotational degrees of freedom in the corner nodes, in contradistinction to a shell6 or facet6 element where the rotational degrees of freedom are handled in special nodes on the element.
side. Due to the reduced integration scheme for this element hourglassing can occur. This will be described in chapter 4.

2.7 The Solid1 & Solid8 Element

The Solid1 & Solid8 elements are eight noded brick elements that can carry tension, compression and shear loads. The Solid1 uses only one integration point, over the elements volume. This leaves the element with twelve so-called Hourglass modes or zero-energy modes. The advantage of this element however is that it uses less CPU time than a Solid8. To be able to use a solid element that is always stable, the Solid8 was implemented. This element uses eight integration points over the elements volume, and leaves no zero-energy modes. Each node has three translational degrees of freedom. And the elements mass is lumped and equally distributed over the eight nodes.

No geometric data needs to be specified.

![Solid1 & Solid8 Element Diagram]

Three material models can be applied for the Solid elements: Isotropic-Elastic, Hysteresis and Mooney-Rivlin.

The Solid elements can be used with collapsed nodes. This way solid elements with seven, six, five or four nodes can be used.

2.8 Linear elastic isotropic material behaviour

Linear elastic material behaviour can be described using Hooke’s law. Isotropic material has the same material behaviour in all directions in the element. Its behaviour can be fully described by two parameters, sidecontraction coefficient $v$ and elasticity modulus $E$. This material is as ISOLIN realised in MADYMO. A variant on this material is the isotropic tension-only material behaviour. This material-type has no stiffness against pressure and is named ISOTEN in MADYMO.
2.9 Linear elastic orthotropic and anisotropic material behaviour

Linear elastic orthotropic material behaviour can be described in the same way as linear elastic isotropic material behaviour. In ortholinear materials, however, three orthogonal planes can be defined in any point, to which material properties are symmetric. The vectors normal to these planes can be denoted as the material's main directions. Because of the material's symmetry, 9 independent parameters are necessary to describe the material behaviour. These 9 variables can be seen as contraction and stiffness coefficients for every main direction in the material.

A material without any symmetry is said to be anisotropic, which means that the constitutive relationships between strains and stresses are different for every material direction. Since this material type can only be used for membrane elements in MADYMO, 6 parameters have to be defined to describe this material behaviour.

2.10 Linear elastic material behaviour with hysteresis

The hysteresis model can be used to model energy dissipation in a material. This is done by defining a loading curve which the element follows when the stress in an element increases. And when the stress decreases, the stress will go along the unloading curve. A slope parameter defines a linear function between the loading and unloading curve to get from one curve to another. An elastic limit can be defined, so if the strain in the element stays under the elastic limit, the material behaviour will keep following the loading curve, even when it unloads. The hysteresis model assumes incompressible material behaviour. MADYMO has three different hysteresis models available, which we will not fully describe here. Instead, we will suffice to refer to the MADYMO manual (Chapter 2.9).

2.11 Elastic plastic material behaviour

With the ISOPLA material type, elastic plastic material can be modelled. The user can specify the modulus of elasticity, Poisson's ratio, hardening function, and the yield stress. The material behaves as linear elastic isotropic material until the stress (according to the Von Mises yield criterion) exceeds the yield stress. After the stress has become larger than the yield stress, the material deforms plastic, following a user-defined hardening function. If the stress decreases, the material will always unload with the slope of the elasticity module.
3.1 Introduction

In software development alpha- and beta-sites are generally accepted household words to indicate certain objective relationships. ‘Alpha-site’ labels the team that is responsible for the development of the concerned software, while ‘beta-site’ is applied to indicate a cooperating company or institute.

Testing MADYMO’s finite element module without having access to the source code therefore can be denoted as beta-site testing. Of course MADYMO offered a modest but thorough finite element module in earlier versions, but this is no guarantee of the thoroughness of the new expanded element module which is to be released in version 5.1.

To validate the implementation of the new module and the interaction with other modules the MADYMO development team performs tests that can be referred to as alpha-site tests. These tests generally concern global loading situations of more or less complicated constructions, performed mainly to check whether the new module delivers reasonable output. Labour intensive detailed tests concerning well-determined loading situations of specific element configurations are only incidentally carried out by the development team itself. These tests are exclusively suitable to be performed by beta-sites since they are mainly heavy-users and they thus will derive much benefit of a well-functioning finite element module. The tests described in this chapter may be referred to as beta-site tests because of their detailed character.

At first all element types MADYMO offers were individually tested, defining isoinlinear material behaviour. Secondly, the implementation of all material models available in the finite element module was evaluated and finally the interaction of different element types in determined configurations was checked. This chapter is concerned with the global character of all tests performed. A concise but detailed overview of all tests classified per element is given in appendix D.

3.2 Rigid body movements

A logical choice for the tests to start with is checking rigid body movements for all element types. These motions can be subdivided in rigid body translations and rigid body rotations. In both cases the element is assumed to remain stress and strainless.

As shown in chapter 1 shape functions relate nodal displacements to displacements of an arbitrary point within the element. Checking rigid body movements can be considered as evaluating certain coefficients in the shape functions. A rigid body translation requires arbitrary points within the element frame to undergo identical
displacements as the nodes. When looking at equation (1.4) it can be deduced that the shape functions must contain a coefficient which is independent of local coordinates. Otherwise rigid body rotations require arbitrary points within the element to undergo displacements that are a linear combination of the nodal displacements. The shape functions thus must contain coefficients that are linear in the local coordinates. The remaining coefficients appearing in the shape functions are not explicitly checked by rigid body movements.

Rigid body movements can be applied in two different ways. Firstly, displacements can be prescribed after which strains and stresses are checked to be zero in the MADYMO output. Secondly, external loadings will force a non-supported element to accelerate according to Newton's second law. The latter method may be preferred since mass computation is involved according to the former law, which states external forces to be a product of acceleration and element mass. Mass computation is connected with integration of the element volume. Element volume is determined by all nodal coordinates and the implemented shape functions and thus we may conclude again that checking the element mass can be seen a check on the correctness of the shape functions.

3.3 Continuum mechanics

Rigid body movements are undoubtedly very important in the finite element technique and a logical basis for the tests to start with, but of course the concept is especially developed to assess strains and stresses in structures exposed to external loadings. The method therefore can be seen as an instrument for analyzing structures with the use of theories derived from continuum mechanics. Continuum mechanics describes mechanical phenomena like deformation, stress state and relation between deformations and stresses for mathematical idealised material behaviour of continuous structures. An important part of continuum mechanics is concerned with the description of deformation and displacement of structures and is called kinematics. Kinematics describes deformation without paying attention to its origins. At any given moment some physical laws have to be served and this establishes relationships between forces and deformation. Quantities which describe the deformed state of a structure can be expressed by the constitutive relationships.

The aim of all performed tests is to check whether MADYMO presents output that agrees with analytical results obtained with the laws of continuum mechanics. Behaviour of individual elements can be predicted analytically and this can serve as a reliable test for the output MADYMO generates.
For a general three-dimensional body there are six components of stress and strain, three
direct and three shear. For an isotropic material the constitutive relationship is defined by
two properties: Young's modulus $E$ and Poisson's ratio $\nu$. The full stress-strain law is:

$$
\begin{bmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{yz} \\
\sigma_{zx} \\
\end{bmatrix} = \frac{E}{1-2\nu} \begin{bmatrix}
1-\nu & -\nu & 0 & 0 & 0 & 0 \\
-\nu & 1-\nu & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\
0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\
0 & 0 & 0 & 0 & 0 & 2(1+\nu) \\
\end{bmatrix} \begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\varepsilon_{zz} \\
\varepsilon_{xy} \\
\varepsilon_{yz} \\
\varepsilon_{zx} \\
\end{bmatrix}
$$

(3.1)

This gives the material flexibility matrix. For the displacement method the inverse of this
is required:

$$
\begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\varepsilon_{zz} \\
\varepsilon_{xy} \\
\varepsilon_{yz} \\
\varepsilon_{zx} \\
\end{bmatrix} = \frac{1}{E} \frac{1}{(1-2\nu)} \begin{bmatrix}
1 & \nu & \nu & 0 & 0 & 0 \\
\nu & 1 & \nu & 0 & 0 & 0 \\
\nu & \nu & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{(1-2\nu)}{2} & 0 & 0 \\
0 & 0 & 0 & 0 & \frac{(1-2\nu)}{2} & 0 \\
0 & 0 & 0 & 0 & 0 & \frac{(1-2\nu)}{2} \\
\end{bmatrix} \begin{bmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{zz} \\
\sigma_{xy} \\
\sigma_{yz} \\
\sigma_{zx} \\
\end{bmatrix}
$$

(3.2)

which is the material stiffness matrix. In crash simulations deformations take place in a
small time interval and thus the strain rates will be large. MADYMO uses a logarithmic
strain definition to account for severe deformations. This strain is defined as the logarithm
of the elongation factor, while the elongation factor can be defined as:

$$
\lambda = \sqrt{\varepsilon (\sigma F^o F^o \delta)}
$$

(3.3)

where $\varepsilon$ is one of the three unity vectors and $F$ is a second order deformation tensor.
Direct strain can thus be represented by:

$$
\varepsilon_j = ln(\lambda)
$$

(3.4)

The double subscript $(ij)$ refers to the three spatial coordinates $x$, $y$ and $z$. Shear strain is
defined by:

$$
\varepsilon_{ij} = \frac{\sigma F^o F^o \delta_{ij}}{\sqrt{(\sigma F^o F^o \delta) \sqrt{(\sigma F^o F^o \delta)}}}
$$

(3.5)
The above laws can be checked for three dimensional elements only. MADYMO offers two of such element types, the SOLID1 and the SOLID8. Basically, two types of tests are performed: checking stresses as a result of prescribed deformation and the other way round; checking strains as a result of prescribed nodal forces. Both test types can be completely described by equations (3.1) and (3.2).

3.4 Two dimensional membranes

Two-dimensional ‘thinwalled’ structures are very common. These structures are usually designed to be primarily loaded in their plane and to resist loads by membrane action rather than bending. MADYMO offers several two dimensional elements, like the MEM3, SHELL4, SHELL6 and FACET6. Some of these elements can handle bending, but first only in-plane loading situations are considered. When looking at (3.1) and (3.2) we may conclude that for two-dimensional elements the out of plane shear terms $\varepsilon_{xz}$, $\varepsilon_{yz}$, $\sigma_{xz}$ and $\sigma_{yz}$ are all zero. The third direct stress, $\sigma_{zz}$, is assumed to be zero and this leads to the plane stress model. The material stiffness matrix can now be given by:

$$
\begin{bmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{xy}
\end{bmatrix} = \frac{E}{1-\nu^2}
\begin{bmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & \frac{(1-\nu)}{2}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\varepsilon_{xy}
\end{bmatrix}
$$

(3.6)

with the subsidiary equation:

$$
\varepsilon_{zz} = -\frac{\nu}{E}(\sigma_{xx} + \sigma_{yy})
$$

(3.7)

defining the through thickness strain. This assumption is valid for thin sheet materials. The flexibility matrix can be given by:

$$
\begin{bmatrix}
\varepsilon_{xx} \\
\varepsilon_{yy} \\
\varepsilon_{xy}
\end{bmatrix} = \frac{1}{E}
\begin{bmatrix}
1 & -\nu & 0 \\
-\nu & 1 & 0 \\
0 & 0 & 1+\nu
\end{bmatrix}
\begin{bmatrix}
\sigma_{xx} \\
\sigma_{yy} \\
\sigma_{xy}
\end{bmatrix}
$$

(3.8)

Just like in the three dimensional case, two test types were performed: checking deformation as a result of external forces and checking stresses caused by prescribed displacements.

If thin flat plates are loaded normal to their plane, bending takes place. Provided that displacements are small compared with the plate thickness there will be no stretching. Thus membrane behaviour and bending are uncoupled. Unfortunately, this is not true in general. The SHELL4, SHELL6 and FACET6 elements in MADYMO can serve as plate bending elements. Since bending is described in higher order differential equations
verification with analytical results is only possible for very simple situations. Plate bending problems were analytically approximated by beam bending solutions for certain loading situations.

### 3.5 Testing material models in MADYMO

In MADYMO 7 material types are available since modelling vehicle parts and human body segments requires application of several material models. Material behaviour is expressed by constitutive relationships which depend on experimental evidence. These relationships are described in chapter 2.

The kinematic correctness of the shape functions was checked by means of assumed isolinear behaviour. It checking the implementation of constitutive laws remains to be done for all other material models MADYMO offers. These models can be divided in three different groups for which appropriate test procedures were used.

Isolinear material behaviour can be seen as a special case of the ortholinear and anisotropic material models. A logical start for testing the latter models is to simulate isolinear behaviour by manipulating parameters and check the results. The second stage is to vary the parameters slightly from isolinear values and check the results again. Finally parameters which have no connection with isolinear values can be used, but it is difficult to verify the output in this case.

Testing the isoplastic material model requires a different approach. Here it is logical to start with ideal elasto-plastic material modelling. Since no hardening will occur the relationship between yield stresses and plastic strains can be checked. This material model can be used only for membrane elements and so merely all tests were concerned with in-plane stress situations, although bending was tested also. After these first tests a hardening curve, which relates yield stresses to strain values, was defined. This necessitates the use of prescribed displacements since only then plastic strains can be determined analytically.

The hysteresis material model was originally tested using only the basic options such as straight loading and unloading curves and prescribed displacement in only one direction. After comparing the stress-strain relationships MADYMO delivered with analytical results the elastic limit and more complex loading and unloading curves were used.

### 3.6 Using mixtures of element types

If only one type of elements is used then there will be complete compatibility throughout the structure. MADYMO contains elements that are compatible with each other and elements within a compatible family can be mixed together freely. Several tests were carried out to check this compatibility, using the isolinear material model. All possible
two-element configurations within the MADYMO element family were exposed to loading situations, be it prescribed deformation or prescribed external forces.

### 3.7 Problems detected

The above-mentioned tests were performed for all element/material combinations. Several problems were detected and it is beyond the scope of this chapter to describe them, especially because they appeared incidentally and are solved at this moment. For the rest all tests are described in appendix D. This may be the place to mention that most of the problems encountered were solved directly, at least when the origins were detected. The outcome of the tests was used to improve the finite element module in detail. Therefore testing and development were parallel processes interacting with each other. Some problems, however, occurred more frequently and they will be described in the next chapters. One of them, so-called hourglassing, is a problem that arises due to the use of reduced integration in certain elements. It cannot be seen as an implementation fault but since it was encountered several times it may be interesting to look at it in more detail.
4. HOURGLASS MODES

4.1 Introduction

Full Gaussian integration of element stresses is very expensive in transient analysis and therefore some elements in MADYMO like the SHELL4 and SOLID1 use reduced integration schemes. Reduced integration may, however, give rise to problems. Higher order strains can unfortunately be zero at the Gauss points in some modes and so the element simulates a lower order model but with the extra degrees of freedom. The stiffness matrix integral mentioned in equation (1.9) of chapter 1 receives its information solely from the Gauss points and this must reflect the true degrees of freedom of the element.

In case of a single SOLID1-element with 6 strain components the work integral has $6 \times 1 = 6$ contributions, whereas the element, regarding 6 rigid body movements has $8 \times 3 - 6 = 18$ degrees of freedom. This exceeds 6 so there are $18 - 6 = 12$ modes not accounted for in one-point integration of a SOLID1-element.

In a solution process these hourglass modes quickly dominate and destroy the solution. Little thought was given in literature to the origin of these modes. An explanation can be that most finite element codes use implicit integration schemes. The nature of application of these codes involves reduced integration of elements to be less interesting. Belytschko and Flanagan (1980) presented a technique for precisely isolating the orthogonal hourglass mode shapes. A part of this technique will be illustrated applied to a SOLID1-element, in the remainder of this chapter.

4.2 Shape functions

In chapter 1 the fundamentals of the finite element technique were described. The basic idea was that the position of any point within the element can be described in terms of the elements nodal coordinates, using so-called shape functions. Isoparametric mapping carries this idea further and describes geometry and displacement field with the same shape functions. These shape functions transform a unit cube in $(\xi, \eta, \zeta)$ space to a general hexahedron in $(x, y, z)$ space. The centre of this cube can be chosen at the origin in $(\xi, \eta, \zeta)$ space so that the shape functions can be expressed in an orthogonal set of base vectors.

Every node of an element has its own shape function: a polynomial with a number of coefficients that is identical to the number of nodes of that element. A SOLID1 has 8 nodes and therefore there are 8 shape functions with 8 coefficients. These shape functions can be denoted as:
\[ \phi_p = \sum_{p=1}^{8} \alpha_p \pi(\xi, \eta, \zeta) \]  

Equation (4.1)

This is in conformity with equation (1.4) by stating:

\[ u(\xi, \eta, \zeta) = d_n N \]  

Equation (4.2)

with:

\[ d_n = \begin{bmatrix} d_{nx} - d_{x0} \\ \vdots \\ d_{nx} - d_{x0} \end{bmatrix}, \quad N = \begin{bmatrix} \phi_1 \\ \vdots \\ \phi_8 \end{bmatrix} \]  

Equation (4.3)

where \( u \) are the displacements of an arbitrary point within the element and \( d_n \) are the nodal displacements. The part of the continuum inside an element must be able to undergo a rigid translation, which means that for all points within the element the displacement components must have the same values. The shape functions therefore must contain a coefficient that is independent of the local coordinates \( \xi \), \( \eta \) and \( \zeta \), so \( \alpha_1 \pi(\xi, \eta, \zeta) \) is constant. Furthermore the part of the continuum inside an element must be able to undergo a rigid rotation. This can be achieved if the shape functions contain coefficients that are linear in \( \xi \), \( \eta \) and \( \zeta \), respectively \( \alpha_2 \xi \), \( \alpha_3 \eta \) and \( \alpha_4 \zeta \). Now four of the eight required coefficients are addressed. The last four coefficients are logically chosen to be bilinear. After normalising this results in shape functions described in terms of an orthogonal set of base vectors, given in table 1, as follows:

\[ \phi_p = \frac{1}{8} \sum_1 \Lambda_{1p} \xi + \frac{1}{2} \Lambda_{2p} \eta + \frac{1}{2} \Lambda_{3p} \zeta + \frac{1}{2} \Gamma_{1p} \eta \xi + \frac{1}{2} \Gamma_{2p} \zeta \eta + \frac{1}{2} \Gamma_{3p} \xi \zeta + \Gamma_{4p} \eta \zeta \]  

Equation (4.4)

### Table 1

<table>
<thead>
<tr>
<th>Node</th>
<th>( \xi )</th>
<th>( \eta )</th>
<th>( \zeta )</th>
<th>( \Sigma_p )</th>
<th>( \Lambda_{1p} )</th>
<th>( \Lambda_{2p} )</th>
<th>( \Lambda_{3p} )</th>
<th>( \Gamma_{1p} )</th>
<th>( \Gamma_{2p} )</th>
<th>( \Gamma_{3p} )</th>
<th>( \Gamma_{4p} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-0.5</td>
<td>-0.5</td>
<td>-0.5</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
<td>-1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-1</td>
</tr>
<tr>
<td>2</td>
<td>0.5</td>
<td>-0.5</td>
<td>-0.5</td>
<td>1</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>0.5</td>
<td>0.5</td>
<td>-0.5</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
<td>1</td>
<td>1</td>
<td>-1</td>
</tr>
<tr>
<td>4</td>
<td>-0.5</td>
<td>0.5</td>
<td>-0.5</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>-1</td>
</tr>
<tr>
<td>5</td>
<td>-0.5</td>
<td>-0.5</td>
<td>0.5</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>6</td>
<td>0.5</td>
<td>-0.5</td>
<td>0.5</td>
<td>1</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
</tr>
<tr>
<td>7</td>
<td>0.5</td>
<td>0.5</td>
<td>0.5</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>8</td>
<td>-0.5</td>
<td>0.5</td>
<td>0.5</td>
<td>1</td>
<td>-1</td>
<td>1</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
<td>-1</td>
<td>-1</td>
</tr>
</tbody>
</table>
The above vectors represent the displacement modes of a unit cube. The \( \Sigma_p \)-column accounts for rigid body translations. The linear base vectors \( \Lambda_1, \Lambda_2 \) and \( \Lambda_3 \) define three normal strain modes, three shear strain modes and three rigid body rotations. They can be called volumetric base vectors since they are the only base vectors which appear in the element volume expression. The last four vectors give rise to strain modes that are neglected by one-point integration and can be called hourglass vectors.

Returning to the displacement field the time derivative of the basic equation (4.2) can be given by:

\[
\dot{u}(\xi, \eta, \zeta) = \dot{d}_n N
\]  

(4.5)

Furthermore the following expression is valid:

\[
\nabla \dot{u} = \nabla(\dot{d}_n N) = C \dot{d}_n^T
\]  

(4.6)

where:

\[
C = \begin{bmatrix}
\frac{\partial \eta}{\partial \xi} & \cdots & \frac{\partial \eta}{\partial \zeta} \\
\vdots & \ddots & \vdots \\
\frac{\partial \eta}{\partial \xi} & \cdots & \frac{\partial \eta}{\partial \zeta}
\end{bmatrix}
\]  

(4.7)

\( \nabla \) is the linear gradient operator. Of course equilibrium must be established between internal and external forces. In chapter 1 the principle of virtual work was used to achieve this. Another possibility is the principle of virtual power:

\[
f_{int} \dot{d}_n^T = \int_V \dot{\tau}_q \nabla \dot{u} dV
\]  

(4.8)

where \( \tau_q \) is Cauchy's stress tensor. In one-point integration this expression is approximated by:

\[
f_{int} \dot{d}_n^T = \nabla \tau_{qmean} \dot{u}_{mean}
\]  

(4.9)

\( \tau_q \) represents the uniform stress field and depends only on mean strains since nonlinear displacements are neglected. The mean velocity gradient is defined by integrating over the element volume as follows:

\[
\nabla \dot{u}_{mean} = \frac{1}{V} \int_V \nabla \dot{u} dV
\]  

(4.10)
The $B$-matrix, mentioned earlier in chapter 1, which relates nodal displacements to element strains, can be defined as:

$$ B = \int_C dV $$

Combining the former equations the mean velocity gradient can be given by:

$$ \nabla u_{\text{mean}} = \frac{1}{V} BD_n^T $$

So the nodal forces can be expressed by:

$$ f_{nl} = r_{\text{linear}} B $$

Computing nodal forces requires evaluation of the $B$-matrix and the element volume $V$, which are related to each other. Isoparametric mapping describes geometry in exactly the same way as the displacement field and therefore it is true that:

$$ x = x_n N $$

where:

$$ x_n = \begin{bmatrix} x_1 & \cdots & x_8 \\ i & \cdots & i \\ z_1 & \cdots & z_8 \end{bmatrix} $$

Equations (4.7), (4.11) and (4.15) yield:

$$ Bx_n^T = VI $$

where $I$ is a (3x3) unity matrix. Now the $B$-matrix may be expressed as:

$$ B = \sqrt{V} x_n (x_n^T x_n)^{-1} $$

$B$ is a matrix with the same dimensions as $x_n$, which are (3x8). This matrix serves a key function in the finite element technique because it is the actual operator which discretises continuous quantities.
4.3 One-point integration of the element volume

An expression for the element volume is needed to obtain the B-matrix. To integrate this volume the jacobian of the isoparametric transformation matrix is used:

\[
V = \iiint_{\Omega} \det(J) \, \partial \xi \partial \eta \partial \zeta
\]  
(4.18)

This transformation matrix looks like:

\[
J = \begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \zeta} \\
\frac{\partial y}{\partial \xi} & \frac{\partial y}{\partial \eta} & \frac{\partial y}{\partial \zeta} \\
\frac{\partial z}{\partial \xi} & \frac{\partial z}{\partial \eta} & \frac{\partial z}{\partial \zeta}
\end{bmatrix}
\]  
(4.19)

With the use of equations (4.3) and (4.14) and with the use of table 1, this can be rewritten as:

\[
J = \begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \zeta} \\
0 & 0 & \frac{1}{2} \\
0 & 0 & \frac{1}{2}
\end{bmatrix}
\]  
(4.20)

The three-by-three determinant of this transformation matrix may be written with the use of the permutation symbol \( \varepsilon_{\ell k} \) which selects the necessary terms and affixes the proper sign to each term:

\[
\begin{vmatrix}
J_{11} & J_{12} & J_{13} \\
J_{21} & J_{22} & J_{23} \\
J_{31} & J_{32} & J_{33}
\end{vmatrix} = \sum_{\ell=1}^{3} \sum_{k=1}^{3} \sum_{m=1}^{3} \varepsilon_{\ell k m} J_{1\ell} J_{2k} J_{3m}
\]  
(4.21)

Looking at the form of equation (4.20) it is evident that the determinant of the transformation matrix will contain polynomials that are at most bi-quadratic in the isoparametric coordinates. Since the determinant is integrated over a symmetric region any term that is linear in the isoparametric coordinates will vanish. The only terms which survive the integration will be the constant, square and double square terms. After integration the resulting expression for V is:

\[
V = \sum_{\rho=1}^{8} \sum_{\sigma=1}^{8} \sum_{\tau=1}^{8} x_{\rho} y_{\sigma} z_{\tau} J_{\rho \sigma \tau}
\]  
(4.22)

where:
So the element volume is, as can be logically expected, a function of the real nodal coordinates $\mathbf{x}_n$ and the coefficients in the shapefunctions. Via equation (4.17) the $B$-matrix can be computed. When evaluating this matrix it becomes apparent that the rows of $B$ are orthogonal to the vectors $\Sigma$ and $\Gamma_n$ which are represented in table 1:

$$B\Sigma = 0 \quad (4.23)$$

$$B\Gamma_n = 0 \quad (4.24)$$

Equations (4.23) and (4.24) imply that the $B$-matrix contains only components of the volumetric base vectors $\Lambda_i$. Therefore, by equations (1.6) and (1.7) only these vectors lead to stresses or nodal forces within the one-point integration approach.

### 4.4 Hourglass control

It has been made clear that, due to reduced integration, some deformation modes can be distinguished to which no constitutive laws are related. If these modes are excited in a solution process they will quickly destroy the solution. This problem can effectively be solved by awarding stiffness or damping to these modes. Therefore the hourglass modes need to be isolated so that they may be treated independently of the rigid body and uniform strain modes.

The determination of the element strain rate considers only a fully linear velocity field. The remaining portion of the nodal velocity field is the hourglass field. This field is orthogonal to all base vectors in table 1 except the hourglass base vectors. The hourglass field can therefore be represented by a linear combination of the hourglass base vectors.

Now there are two methods to control 'hourglassing'. Artificial damping can be applied to the hourglass modes to prevent violent oscillations. Since there is no stiffness this method cannot prevent mesh distortions. The other method uses an artificial stiffness applied to the hourglass modes which effectively prevent severe distortions. Both methods can be combined but there is no evidence that additional damping provides any improvement.

In MADYMO hourglass control is implemented by the so-called hourglass parameter which represents artificial stiffness attached to the hourglass modes. When not used this parameter has a default value.

In the tests we performed on the SHELL4 and SOLID1 elements we have encountered hourglassing several times. Most of these tests were executed with single elements and
It appeared to be very difficult to stabilize the solution by manipulating the hourglass parameter. In most applications, however, these elements will be embedded in a mesh and individual elements will in this way be supported by their neighbours. In this more friendly situation the hourglass control parameter is expected to prevent distortions effectively. Choosing the appropriate value for the hourglass parameter seems to be pragmatic and depends on the structure considered. We are not able to give recommendations about the use of this parameter. MADYMO users that encounter hourglass problems should gain experience in manipulating the parameter themselves, on the other hand the problems can easily be avoided by replacing reduced integration elements in critical zones by full integration elements. Unfortunately, this will result in higher CPU times, but stability will be guaranteed.

Examples of hourglass modes for the SOLID8 and SHELL4 element types.
5. STRESS ANALYSIS USING MOHR’S CIRCLE

5.1 Introduction

During the tests of MADYMO’s finite element module, some problems were discovered regarding the in-plane shear stress handling of the triangular elements. Because this problem appeared several times, during the testing period, we will take a closer look at one particular case in the following paragraphs.

5.2 Problem description

We put the Mem3, Sh3l6 and Facet6 in the following situation (see figure 5.1). To get a better idea about where the element fails, a complete stress analysis of the element will be done.

First the coordinate systems have to be defined. There is a global coordinate system which has its origin at node 1. Besides that, the element has its own local coordinate system. The origin of the local system is node 1. The x-axis is defined from the first node in the direction of the second node. Initially the second node of the element lies on the global x-axis so the local and global axis coincide at $t = 0$. The y-axis is perpendicular to the x-axis, in the plane of the element. The y-direction is chosen so, that the local y-coordinate of node three has a positive value. Orthogonality leaves only one alternative for the position of the z-axis. The direction of the z-coordinate is only of great importance when a surface load acts on the element, for example the pressure in an airbag simulation. The local coordinate system moves along with the element, so the x-axis stays always oriented from node 1 to node 2.
At \( t=0 \) the external force \( F=0 \). Then \( F \) increases to \( 800\sqrt{2}\times10^6 \) N and after the system has become stationary the madymo results can be compared to theoretical results. It's not hard to assume that the main stress directions are oriented in the directions of \( \sigma_1 \) and \( \sigma_2 \).

### 5.3 Examining the stresses in the element

It is possible to calculate the stationary position that the element will reach after some time.

The stress \( \sigma_2 \) in \( \epsilon_2 \) direction equals:

\[
\sigma_2 = \frac{F}{A_2}
\]  
(5.1)

Where:

\[
A_2 = b_2 h = (b_2 \epsilon^0) h
\]  
(5.2)

And from the constitutive relations we know:

\[
\varepsilon_1 = -\frac{v}{E} \sigma_2
\]  
(5.3)

If we combine these relations we get:

\[
\sigma_2 = \frac{F}{b_2 \epsilon^0}
\]  
(5.4)

Rewriting and using the values of the variables yields:

\[
0.05 \times \frac{1}{2} \times \sqrt{2} \sigma_2 e^{\frac{\rho_1}{\sin^2 \phi}} - 8 \times 10^6 \sqrt{2} = 0
\]  
(5.5)

This equation can be numerically solved. This results in: \( \sigma_2 = 34.150765 \times 10^6 \) N/m\(^2\). Once \( \sigma_1 \) and \( \sigma_2 \) have been determined, the main stresses can be deduced using the constitutive relations \( \Rightarrow \varepsilon_2 = \frac{\sigma_2}{E} = 1.626227 \times 10^{-1} \) and \( \varepsilon_1 = -\frac{\sigma_2}{E} = -6.504908 \times 10^{-1} \).

We now have to determine the final location of node 2, in order to know the direction of the local coordinate system of the tested element in its final stationary situation. The displacement of node 2 in \( \epsilon_1 \) direction equals

\[
i_2 = \sqrt{2}(e^{\phi_1}-1)
\]  
(5.6)

And in \( \epsilon_2 \) direction the displacement of node 2 is

\[
j_2 = -\sqrt{2}(e^{\phi_2}-1)
\]  
(5.7)

Using transformation matrix \( A \), the displacement vector \((i,j)^T\) will be transformed to the displacement vector \((u,v)^T\) in the global coordinate system:

\[
A = \begin{bmatrix}
\cos \phi & \sin \phi \\
-\sin \phi & \cos \phi
\end{bmatrix} ; \quad \phi = -\frac{\pi}{4}
\]  
(5.8)

So vector
Now the coordinates of node 2 are determined:

\[
\begin{bmatrix}
V_2 \\
U_2
\end{bmatrix} = \begin{bmatrix} \frac{a}{\sqrt{2}} & -\frac{a}{\sqrt{2}} \\
\frac{a}{\sqrt{2}} & \frac{a}{\sqrt{2}}
\end{bmatrix} \begin{bmatrix} y_2 \\
x_2
\end{bmatrix}
\]

(5.9)

\[
\begin{bmatrix}
U_2 \\
V_2
\end{bmatrix} = \begin{bmatrix} e^\alpha + e^{-\alpha} - 2 \\
e^\alpha - e^{-\alpha}
\end{bmatrix}
\]

(5.10)

Now the coordinates of node 2 are determined:

\[
\begin{bmatrix} p_{2x} \\
p_{2y}
\end{bmatrix} = \begin{bmatrix} 2 \\
0
\end{bmatrix} + \begin{bmatrix} U_2 \\
V_2
\end{bmatrix} = \begin{bmatrix} e^\alpha + e^{-\alpha} \\
e^\alpha - e^{-\alpha}
\end{bmatrix}
\]

(5.11)

\[
\begin{bmatrix} p_{21} \\
p_{22}
\end{bmatrix} = \begin{bmatrix} 2.113614 \\
-0.2395712
\end{bmatrix}
\]

(5.12)

Finally, the angle between the global x-axis and the local x-axis can be derived:

\[
\theta = \arctan \left( \frac{p_{2y}}{p_{2x}} \right)
\]

(5.13)

Mohr's circle can be used to calculate the stresses in the local coordinate system. In order to do this, we need to know the angle between the main stress coordinate system and the local coordinate system. This angle is:

\[
\phi = \varphi + \theta
\]

(5.14)

A Mohr circle can be drawn, using: \(\sigma_1\), \(\sigma_2\) and \(\phi\). (figure 5.2)

![Figure 5.2](image)

The local stresses and main stresses must obey the following relations in this particular situation:
\[
\sigma_x = \frac{1}{2}(1 - \cos(2\phi))\sigma_2 \\
\sigma_y = \frac{1}{2}(1 + \cos(2\phi))\sigma_2 \\
\tau_{xy} = \sigma_{xy} = -\frac{1}{2}\sin(2\phi)\sigma_2
\]

(5.15)

The three triangular elements that are implemented in MADYMO at this moment, were tested in this particular situation. The results shown in table 5.1 are the results such as achieved with the latest MADYMO 5.1 beta-site test version on Friday 3-11-'94.

<table>
<thead>
<tr>
<th>( \sigma_1 )</th>
<th>( \sigma_2 )</th>
<th>( \sigma_x )</th>
<th>( \sigma_y )</th>
<th>( \tau_{xy} )</th>
<th>node 2</th>
<th>node 2</th>
<th>node 3</th>
<th>node 3</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4.56E-2</td>
<td>-1.351E-1</td>
<td>4.6215E-2</td>
<td>-2.1011E-1</td>
<td>2.113614</td>
<td>2.0914</td>
<td>2.1431</td>
<td>2.0914</td>
</tr>
<tr>
<td>( \sigma_2 )</td>
<td>3.41508E10</td>
<td>2.924E10</td>
<td>3.4151E10</td>
<td>2.9571E10</td>
<td>-0.23957</td>
<td>1.18E-11</td>
<td>-0.2691</td>
<td>-0.21333</td>
</tr>
<tr>
<td>( \sigma_x )</td>
<td>2.08972E10</td>
<td>1.462E10</td>
<td>2.129E10</td>
<td>2.9571E10</td>
<td>-0.23957</td>
<td>-0.42667</td>
<td>-0.2103</td>
<td>-0.21333</td>
</tr>
<tr>
<td>( \sigma_y )</td>
<td>1.32526E10</td>
<td>1.462E10</td>
<td>1.285E10</td>
<td>4.6214E-2</td>
<td>2.113614</td>
<td>2.0931</td>
<td>2.0843</td>
<td>2.09142</td>
</tr>
<tr>
<td>( \tau_{xy} )</td>
<td>-1.66422E10</td>
<td>-1.462E10</td>
<td>-1.655E10</td>
<td>3.0031E-6</td>
<td>2.113614</td>
<td>2.0931</td>
<td>2.0843</td>
<td>2.1137</td>
</tr>
</tbody>
</table>

Table 5.1

When the tests were first performed, all elements came up with the same results, which is not surprising taking into account that the plane stress routines for the newly implemented facet6 & shell6 were copied from the Mem3 element. All three elements came up with the results such as printed under 'Mem3 test8' in table 5.1. Having noticed these poor results, the three elements were changed. In fact updating the coordinate system was improved, now that orientation of the local coordinate system was determined every half time step. This resulted in the results such as printed under 'Facet6 & Shell6 test8' in table 5.1.

Other tests, however, showed that the Mem3 element had become more instable after the latest changes, so at this moment the Mem3 element has been set back to its original state, where the local coordinate system is not determined at half time steps. The test results show a rather strange behaviour for the Mem3 element. It almost seems as if node 2 of the element is supported in y-direction, because node 2 tends to follow the y-axis. Its deformation is totally a-symmetric, whereas the Facet6 & Shell6 elements show much
better results on the same test. The main stresses are calculated with great accuracy, while the local stresses show errors in the order of 4%. Let's analyze these values. The main stresses $\sigma_1$ and $\sigma_2$ have obviously been calculated correctly. To determine angle $\phi$, we need angles $\varphi$ and $\theta$. The latter can be determined by using the coordinates of node 2 in the MADYMO output. $\varphi=-\frac{\pi}{4}$, presuming that the main stresses are oriented as expected. Substituting these values in Mohr's circle (figure 5.2), yields the values as MADYMO calculated for $\sigma_x$, $\sigma_y$, and $\sigma_{xy}$. This also confirms that the assumption that the main stresses have been oriented correctly, is right. Furthermore, the relations between the local stresses and strains have been consistently followed. In the same way the relations between the strains and stresses have been consistently followed.

To see whether the results could be improved for this test, the test was performed again but now with a different orientation of the element (see figure 5.3).

![Figure 5.3](image)

The direction from node 1 to node 2 is chosen in the main stress direction. The results of this test are printed in the last two columns of table 5.1 called test8a. The elements show significant improvement. The deformation is now perfectly symmetric. Moreover Facet6 & Shell6 have approached the solution with a deviation of less than 0.2%. Mem3 shows poor results.

### 5.4 Summary

Summarizing we can say that the error in the angle between the main stress coordinate system and the local coordinate system of the element causes the differences between MADYMO's solution and the theoretical solution. The results of a simulation can be orientation dependent.
6. REFERENCES


(5) NAFEMS, A Finite Element Primer., Bell and Bain Ltd., Glasgow, 1992. 3rd Reprint.


FEM MODEL OF A RIB CERTIFICATION TEST

Introduction

The TNO Crash-Safety Research Centre is responsible for the design of some types of crash dummies that are used to assess safety aspects of passenger cars. Recently a new dummy type: the EUROSID-1 has been developed as a result of a diverting emphasis of crash-safety research from frontal to side impacts. The EUROSID-1 has adapted injury assessment capabilities. Since its introduction in 1990 the EUROSID-1 has been used all over the world for vehicle development and optimization.

The eurosid-1 rib module

The rib module of a EUROSID-1 dummy is ideal for comparing modelling techniques with experimental data since well-defined component tests are available. A single rib consists of 3 different components:

- A thin steel strip, bent in the shape of a rib and covered with a layer of polyurethane foam which is held onto the rib by a cotton sleeve covered with a pvc-based clothing. The foam has visco-elastic properties, thus resembling human flesh behaviour.
- A piston/cylinder assembly which is connected to both sides of the rib and guarantees uni-axial deformation.
- A spring/damper combination which is rigidly attached to the cylinder.

Certification tests

The response of a rib module due to a side impact must satisfy some special requirements. Certification is defined as the tuning and checking of the components to ensure that their behaviour are within a predefined corridor. Lateral collision drop tests of the module are mainly of interest to test the stiffness and damping properties of the rib, the springs and the damper.

Fem model of the rib module

There are several MADYMO models that have been used for analyzing the rib in a certification test. Among them are a rigid body model, a flexible body model and a finite element model. The latter has been built up using plane stress elements. In this model only the steel strip is concerned. Ellipsoids are connected to the finite element structure for modelling the contact between the foam and the impactor.
The recent MADYMO-version offers more sophisticated modelling techniques since it is equipped with a more advanced finite element module. The focus of this brief report will be on the modelling of the foam with solid elements. The already existing mesh for the steel strip is used to generate a new 3D mesh for the foam. No thorough analysis is employed since the aim of this model adaptation is merely to construct a demo of MADYMO’s new finite element module.

**Material model for the hoop**

The thin steel strip, bent in the shape of a rib was specified with the following material properties:

- Young’s Modulus: $2.05 \times 10^11$ (N/m$^2$)
- Poisson’s Ratio: 0.27
- Density: 7850 (kg/m$^3$)
- Thickness: $2.5 \times 10^{-3}$ (m)

**Material model for the rib foam**

The foam has visco-elastic properties and a slow state of return from deflection which resembles human flesh behaviour. Since MADYMO offers no visco-elastic material model yet, the HYSISO material type is used to model the behaviour of the foam in the rib.

- Density: 102.4 (kg/m$^3$)
- Thickness: $2.0 \times 10^{-2}$ (m)

The hysteresis curve is modelled with the following input data:

- Loading Function
  - Elastic Slope: $1.5 \times 10^{10}$ (N/m$^2$)
- Unloading Function: $5 \times 10^3$ (N/m$^2$)

**Finite element model of the foam**

At first a SOLID1 mesh was generated using the already existing FACET6 mesh. A mesh refinement appeared to be necessary in the environment of the contact area on account of the extreme deformation in this region.
The first computation run crashed because of a so-called hourglass-mode-deformation of the SOLID1 elements in the contact zone. This is a result of the reduced integration method that is employed for these elements.

The next logical step was to replace the SOLID1 elements in the contact zone by SOLID8 elements. While all degrees of freedom for SOLID8 elements are related to a stiffness factor, 'hourglassing' cannot occur. A computation run confirmed the expectations. This time the computation was executed successfully.

The final full FEM model included 80 FACET6 elements, 44 SOLID1 elements and 16 SOLID8 elements.
4 m/s certification drop test

Rib deflection (m)

Time (ms)

Full finite element model versus experimental data
SIMULATION OF A MINOR SHIP COLLISION

Introduction

The basic tests of the elements and material types available in MADYMO 5.1 have showed that the new finite element module operates with sufficient accuracy. These tests, however, concerned mostly individual elements, subjected to simple loading situations. MADYMO will generally be used for more complex structures and therefore it is necessary to perform physical realistic tests involving a bigger number of elements. An article, written by L. Zhu and D. Faulkner was found in the International Journal of Impact Engineering and it seemed to deliver an interesting test case for Madymo’s finite element module. The article concerned the study of a simplified ship collision which was analysed using the finite difference technique based on energy equations. In the remainder of this report a description of the problem will be given and the output Madymo delivered will be compared with the results given in the article.

Description of experiments

The collision of ships and offshore platforms has been studied extensively. To simulate the ship-ship collision process a lot of work has been done in which the sidewall of the struck ship is subjected to impact from a rigid wedge representing the striking ship. An experimental programme on the collision of plates and small scale ship models was conducted in the Department of Naval Architecture and Ocean Engineering at the University of Glasgow. The experiment consisted of bringing a rigid striker into violent contact with a deformable ship model. As the permanent deformation caused by collision is highly localised, it is reasonable to use a simplified model which uses a fully clamped rectangular plate as shown in the figure. The striking ship is modelled as a rigid knife-edge indentor. Since no rupture occurs this collision can be classified as minor.

Theoretical analysis

Besides the experimental work a theoretical analysis of the collision is performed with the aim of identifying some important parameters involved in ship collisions. This theoretical analysis is restricted to energy equations. To predict the extend of damage of the struck ship plate it is necessary to study the collision dynamically. In modelling the impact force the following assumptions were made:
- The striker hits the centre of the plate with the knife-edge perpendicular to the long edge of the plate.
- The striker keeps in contact with the plate during the collision.
The von Mises yield function is adopted and the hardening curve is chosen to be linear. In the numerical analysis the variational finite difference method was used. This method was developed by Ni and Lee (1973) and is based on a minimum principle in dynamic plasticity. The results obtained by this numerical approach were close to the experimental test data and quite a good correlation was achieved.

**Finite element model in MADYMO**

The paper presents sufficient parameters to perform a MADYMO computation. The problem seems to be suitable for MADYMO because of the typical dynamic aspects. Furthermore the line-impact loading situation is expected to be a very suitable test for the mesh continuity. The input data needed are listed below (See figure): \( L = 150 \text{ mm} \), \( B = 200 \text{ mm} \), \( H = 1.65 \text{ mm} \), \( \rho_0 = 2700 \text{ kg/m}^3 \), \( \gamma = 0.33 \), \( E = 51.5 \times 10^9 \text{ N/m}^2 \), \( E_\| = 310 \times 10^6 \text{ N/m}^2 \), \( \sigma_s = 115 \times 10^6 \text{ N/m}^2 \), \( m_\omega = 23.3 \text{ kg} \), \( V_0 = 2.313 \text{ m/s} \) and \( L_0 = 100 \text{ mm} \).

At first only the clamped plate was built up as a finite element model using SHELL4 elements. The wedge was constructed with two rigid planes attached to a moving body. A mesh refinement was used in the contact area, so that the element edge size in that area was 3 times smaller than in the outer surface. Continuity could be achieved using collapsed elements in the transition zone.
Results

The deflection history at three plate points, strain-time histories for the corresponding elements and impact force histories for both sides of the wedge are given at the next pages. Among these points (indicated in the figure) point 4 has the largest maximum x-strain. The y-strain at this point is also significantly large and this point is thus in a state of bi-axial tension. The deflection-time histories show that point 1 which is positioned exactly in the centre of the plate, undergoes the largest deflection. The deflection of point 4 is considerably smaller which means that the assumption that the striker should be in full contact with the plate can only be met if the wedge undergoes a slight deformation. Since this wedge is modelled to be undeformable these results are not very accurate. The several histories MADYMO delivers have exactly the same form as those represented in the article by Zhu and Faulkner. Therefore, the results can be concluded to be qualitatively correct; Quantitatively, however, there is poor resemblance between the results presented in the article and the output MADYMO has generated. The deflection of point 4 is 8% larger than mentioned in the paper, whereas the strains in x-direction are about 65% smaller. On the other hand the contact forces seem to agree with those found in the article.

Since MADYMO uses a logarithmic strain definition and the article presents no strain definition the detected differences cannot be criticized properly. The other results show a less severe deviation. Apart from modelling errors the MADYMO computation can be considered as being successful.

After this first test another computation was performed. In this second test both the plate and the wedge were constructed as finite element models. However, the computation crashed due to contact interaction problems. At the moment this test was performed these contact interactions between finite element models were not implemented properly. Since no results were obtained a further description of this test will not be given.
Deflection of reference points

Deflection point 4
Deflection point 1
Deflection point 2

Finite element model of minor ship collision
Strain of reference elements

Strain element 4
Strain element 1
Strain element 2

Finite element model of minor ship collision
Contact Forces

Finite element model of minor ship collision
TESTING THE ELEMENTS IN PRACTICE

Introduction

After the testing of the single elements, we tested the elements in a bigger construction. This test was taken from an article by J.M. Kennedy (et al.). The construction is used in a nuclear reactor design. The cylindrical element is a part of the above-core structure and its function is to support the upper internal structure-mass (figure 1). The mass of the upper internal structure (UIS) equals 40,000 (kg), and the bottom surface equals 6.818 (m²). The bottom is loaded with the pressure as shown in table 1. Obviously the information about the time-pressure curve is not very detailed, but from the rest of the article can be deduced that the pressures involved are impulsively. The MADYMO input values about the pressure time curve we used are printed in table 2. MADYMO uses linear interpolation between two defined points. These values were obtained by trying to approach an impulsively pressure course and fitting the MADYMO results to the results of the simulations in the article.

![Figure 1: Upper Internal Structure](image-url)
Four small-radius columns in nuclear reactor design.

In the first test four small-radius columns were used (instead of one large-radius cylinder as shown in figure 1), to support the UIS-mass. The material plastic data of the columns is given in table 3. In order to trigger the buckling mode, a slight imperfection was added to the model. To the y-coordinates a term \( d = \frac{1}{800} \sin(\frac{2\pi x}{l}) \) was added. To reduce the simulation time, only half a column was modeled, with the necessary boundary conditions. Consequently the UIS mass and the bottom area were reduced with factor 0.125. In the article 2 different elements were used to model the pipe:

- Quadrature shell elements, comparable to the MADYMO Shell4 element.
- Triangular shell elements, with their rotational degrees of freedom in the corner nodes, comparable to collapsed Shell4 elements.

Because of lack of detailed information in the used article, it is merely useful to look at the global behaviour of the construction in a MADYMO simulation. The half column was, with three different kinds of elements, tested in five different ways:

- A column consisting of 6x40 = 240 square Shell4 elements.
- A column consisting of 2x6x40 = 480 triangular Facet6 elements.
- A column consisting of 2x6x40 = 480 triangular Shell6 elements.
- A column consisting of 2x6x40 = 480 triangular Facet6 elements (fishbone structure).
- A column consisting of 2x6x40 = 480 triangular Shell6 elements (fishbone structure).

The results were practically identical for the Shell4 element and Facet6 element, they also coincided also very well with the results in the article. The only difference between the Facet6 column and the Shell4 column is that the Facet6 column has sharper buckling edges, which could indicate stronger local plastic deformation. The column build of Shell6 elements behaved less stiff than the column made of Shell4 or Facet6 elements.

This does not unconditionally mean that the Shell6 element is worse, because the article is only based on computational simulations and not on real-life experiments. Although one needs to notice here, that the Shell6 element showed poor results in one of the basic tests performed earlier, (see appendix D 'single shell6: test1.shell6/test 10') the Shell6 element behaved worse than the Facet6 and Mem3. The fishbone structure did not make much difference in the final results.

The Shell4 column in the article showed Hourglassing when no Hourglass suppressing parameter was defined. MADYMO performed somewhat better in this case, because
changing the Hourglass parameter to 0, didn't cause Hourglassing in this particular MADYMO simulation.

**One large cylinder in nuclear reactor design.**

Now one large cylinder (shown in figure 1) will be simulated instead of four columns. Because only half a cylinder is modeled, the UiS-mass equals 20,000 (kg). And the area of the bottom surface was divided by 2. Now a surface pressure is added to the model because this large cylinder has a much smaller thickness/radius ratio, therefore the surface pressure cannot be neglected. Again an imperfection was added to the model, to initiate a realistic buckling mode. The term $w = -5.08E-3 \sin\left(\frac{\pi}{r}\right) \cos(\theta)$ was added to the radius of the modeled cylinder. The tests were performed using Shell4, Facet6, Shell6 and collapsed Shell4 elements. The deformation of the Shell4 cylinder, coincided with the deformation as described in the article. The other elements all behaved differently. To detect where the difference comes from, the same tests were performed only without bottom pressure. This resulted in exactly the same deformation for all elements, except for the collapsed Shell4 elements. The collapsed Shell4 elements failed because of a wrong description of the surface loads, which were used to model the pressure acting on the lateral surface. Because the Facet6, Shell6 and Shell4 all behaved similar in the last test, the difference comes from the axial load. The elements all have a different bending behaviour when an axial load buckles the element. This is an aspect of the elements which was not tested in the basic element tests. It might be interesting to perform these tests in the near future, because these kinds of loads are very common in crash simulations.
### Table 1

<table>
<thead>
<tr>
<th>Bottom surface</th>
<th>Lateral surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>pressure (MPa)</td>
<td>time (ms)</td>
</tr>
<tr>
<td>34.47</td>
<td>0.4</td>
</tr>
<tr>
<td>0.0</td>
<td>3.0</td>
</tr>
<tr>
<td>24.13</td>
<td>4.4</td>
</tr>
<tr>
<td>13.79</td>
<td>5.0</td>
</tr>
<tr>
<td>5.52</td>
<td>18.0</td>
</tr>
<tr>
<td>0.0</td>
<td>36.0</td>
</tr>
<tr>
<td>0.0</td>
<td>100.0</td>
</tr>
</tbody>
</table>

### Table 2

<table>
<thead>
<tr>
<th>Bottom surface</th>
<th>Lateral surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>pressure (MPa)</td>
<td>time (ms)</td>
</tr>
<tr>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
<td>0.38</td>
</tr>
<tr>
<td>34.47</td>
<td>0.4</td>
</tr>
<tr>
<td>0.0</td>
<td>3.0</td>
</tr>
<tr>
<td>0.0</td>
<td>4.38</td>
</tr>
<tr>
<td>24.13</td>
<td>4.4</td>
</tr>
<tr>
<td>13.79</td>
<td>5.0</td>
</tr>
<tr>
<td>5.52</td>
<td>18.0</td>
</tr>
<tr>
<td>0.0</td>
<td>36.0</td>
</tr>
<tr>
<td>0.0</td>
<td>100.0</td>
</tr>
<tr>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

### Table 3

<table>
<thead>
<tr>
<th>Material plastic data</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_p$ (MPa)</td>
</tr>
<tr>
<td>------------</td>
</tr>
<tr>
<td>4596</td>
</tr>
<tr>
<td>3103</td>
</tr>
<tr>
<td>2758</td>
</tr>
<tr>
<td>1896</td>
</tr>
<tr>
<td>1724</td>
</tr>
<tr>
<td>1551</td>
</tr>
<tr>
<td>1379</td>
</tr>
<tr>
<td>1207</td>
</tr>
<tr>
<td>1034</td>
</tr>
<tr>
<td>862</td>
</tr>
</tbody>
</table>
RECENT DEVELOPMENTS IN EXPLICIT FINITE ELEMENT TECHNIQUES
AND THEIR APPLICATION TO REACTOR STRUCTURES

J.M. KENNEDY
Engineering Mechanics Program, Reactor Analysis and Safety Division, Argonne National Laboratory, Argonne, IL 60439, USA

T. BELYTSCHKO and J.I. LIN
Civil Engineering Department, The Technological Institute, Northwestern University, Evanston, IL 60201, USA

Received 28 March 1986

A triangular element which requires only one quadrature point per element is described along with its implementation in the nonlinear, explicit time integration program SAFE/RAS. The implementation of an Ilyushin flow law which eliminates the need for integration through the thickness and simple formulas for stable time steps is also described. The performance of the triangular and quadrilateral elements is compared in large deflection, elastic-plastic problems. Applications to the analysis of above-core structures in breeder reactors are also described.

Example 2: A C3 support column (design A).

In the Clinch River breeder reactor design, four columns with dimensions and material properties given in table 3 were used to support the above core structures. The loading applied to the column in design studies is based on an SRI experiment and is shown in table 4. The finite element model of one of the columns is shown in fig. 3; the deformed configuration is also shown. In order to trigger the lateral buckling mode shown, it was necessary to add an imperfection to the model by displacing all nodes laterally according to the following

\[ d = a \sin \left( \frac{2\pi z}{l} \right), \]  

where \( l \) is the length of the column and \( z \) is the coordinate along the column; the origin is at the bottom; \( a = a/l \) is \( 1.25 \times 10^{-3} \) for this simulation. Note that the column cross section buckles in at the nodes of the beam buckling mode. The column also exhibits slight buckles on the sides which are loaded.

<table>
<thead>
<tr>
<th>Table 1</th>
<th>Material properties and parameters for cylindrical panel (sample problem 1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>( \rho = 2.5 \times 10^{-6} ) lb s(^2)/in.(^4)</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>( E = 1.05 \times 10^{7} ) psi</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>( v = 0.33 )</td>
</tr>
<tr>
<td>Yield stress</td>
<td>( \sigma_y = 4.4 \times 10^{4} ) psi</td>
</tr>
<tr>
<td>Plastic modulus</td>
<td>( E_p = 0 )</td>
</tr>
<tr>
<td>Initial velocity</td>
<td>( v_0 = 5650 ) in./s</td>
</tr>
</tbody>
</table>
compressively at the top and bottom of the columns. The shear reduction factor, eq. (21), was not used in these runs.

An interesting aspect of this problem is that when the shear reduction factor was activated without

Table 2
Cylindrical panel results

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Element</th>
<th>Mass factor</th>
<th>Quadrature points</th>
<th>Shear correction</th>
<th>Maximum disp (in.) at</th>
<th>Simulation time (ms)</th>
<th>CPU time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>T-Tri</td>
<td>a1, a2</td>
<td>Along 2 1</td>
<td></td>
<td>y = 6.28</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6 x 16</td>
<td>T</td>
<td>-</td>
<td>1</td>
<td>no</td>
<td>0.917</td>
<td>0.401</td>
<td>1.0</td>
</tr>
<tr>
<td>6 x 16</td>
<td>Q</td>
<td>a1</td>
<td>5</td>
<td>yes</td>
<td>0.926</td>
<td>0.412</td>
<td>1.0</td>
</tr>
<tr>
<td>6 x 16</td>
<td>Q</td>
<td>a1</td>
<td>5</td>
<td>yes</td>
<td>0.986</td>
<td>0.480</td>
<td>1.0</td>
</tr>
<tr>
<td>6 x 16</td>
<td>Q</td>
<td>a2</td>
<td>5</td>
<td>yes</td>
<td>1.001</td>
<td>0.494</td>
<td>1.0</td>
</tr>
<tr>
<td>8 x 16</td>
<td>T</td>
<td>-</td>
<td>5</td>
<td>no</td>
<td>1.052</td>
<td>0.479</td>
<td>1.0</td>
</tr>
<tr>
<td>8 x 16</td>
<td>T</td>
<td>1</td>
<td>-</td>
<td>no</td>
<td>1.037</td>
<td>0.458</td>
<td>1.0</td>
</tr>
<tr>
<td>8 x 16</td>
<td>Q</td>
<td>a1</td>
<td>5</td>
<td>no</td>
<td>1.034</td>
<td>0.469</td>
<td>1.0</td>
</tr>
<tr>
<td>10 x 20</td>
<td>T</td>
<td>5</td>
<td>-</td>
<td>yes</td>
<td>1.108</td>
<td>0.510</td>
<td>1.0</td>
</tr>
<tr>
<td>10 x 20</td>
<td>T</td>
<td>1</td>
<td>-</td>
<td>yes</td>
<td>1.106</td>
<td>0.526</td>
<td>1.0</td>
</tr>
<tr>
<td>10 x 20</td>
<td>Q</td>
<td>a1</td>
<td>1</td>
<td>no</td>
<td>1.106</td>
<td>0.536</td>
<td>1.0</td>
</tr>
<tr>
<td>10 x 20</td>
<td>T</td>
<td>5</td>
<td>-</td>
<td>yes</td>
<td>1.092</td>
<td>0.502</td>
<td>0.6</td>
</tr>
<tr>
<td>10 x 20</td>
<td>Q</td>
<td>a2</td>
<td>5</td>
<td>yes</td>
<td>1.108</td>
<td>0.462</td>
<td>0.6</td>
</tr>
<tr>
<td>12 x 24</td>
<td>T</td>
<td>-</td>
<td>5</td>
<td>yes</td>
<td>1.116</td>
<td>0.527</td>
<td>0.6</td>
</tr>
<tr>
<td>12 x 24</td>
<td>Q</td>
<td>a1</td>
<td>5</td>
<td>yes</td>
<td>1.124</td>
<td>0.473</td>
<td>0.6</td>
</tr>
<tr>
<td>12 x 24</td>
<td>Q</td>
<td>a2</td>
<td>5</td>
<td>yes</td>
<td>1.146</td>
<td>0.551</td>
<td>0.6</td>
</tr>
<tr>
<td>16 x 32</td>
<td>T</td>
<td>-</td>
<td>5</td>
<td>yes</td>
<td>1.163</td>
<td>0.550</td>
<td>0.6</td>
</tr>
<tr>
<td>16 x 32</td>
<td>Q</td>
<td>a1</td>
<td>5</td>
<td>yes</td>
<td>1.196</td>
<td>0.585</td>
<td>0.6</td>
</tr>
<tr>
<td>16 x 32</td>
<td>Q</td>
<td>a2</td>
<td>5</td>
<td>yes</td>
<td>1.194</td>
<td>0.585</td>
<td>0.6</td>
</tr>
</tbody>
</table>

Notes: 1) a1 = A/8, a2 = (h'2 + A)/12. 2) One quadrature point indicates Ilyushin approach from Appendix A.

Table 3
Dimensions and properties of above-core structures

<table>
<thead>
<tr>
<th>Design A - Column support</th>
<th>Design B - Cylindrical support</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall thickness, cm</td>
<td>Wall thickness, cm</td>
</tr>
<tr>
<td>2.540</td>
<td>1.651</td>
</tr>
<tr>
<td>Column diameter, cm</td>
<td>Cylinder diameter, cm</td>
</tr>
<tr>
<td>33.02</td>
<td>203.2</td>
</tr>
<tr>
<td>Cross-sectional area, cm²</td>
<td>Cross-sectional area, cm²</td>
</tr>
<tr>
<td>263.5</td>
<td>1054.0</td>
</tr>
<tr>
<td>Column length, cm</td>
<td>Cylinder length, cm</td>
</tr>
<tr>
<td>406.4</td>
<td>406.4</td>
</tr>
<tr>
<td>Added node mass, kg</td>
<td></td>
</tr>
<tr>
<td>3.28</td>
<td></td>
</tr>
</tbody>
</table>

Material properties

| Young's modulus | 1.93 x 10^5 MPa | 4596 | 327.5 |
| Density | 8888 kg/m³ | 3105 | 389.6 |
| Poisson's ratio | 0.25 | 2758 | 444.7 |
| Yield stress | 241.3 MPa | 1896 | 482.6 |
| UIS mass | 1551 | 1724 | 517.1 |
| Diameter | 294.6 cm | 1379 | 575.7 |
| Height | 281.9 cm | 1207 | 599.8 |
| Young's modulus | 7.17 x 10^4 MPa | 1034 | 620.5 |
| Mass | 40,000 kg | 862 | 792.9 |
hourglass control, a w-hourglass mode appeared, as shown in fig. 4. This hourglass mode does not occur in the absence of the shear factor because of the constraints of the support conditions. Note that the hourglass mode is a very low-frequency mode with a period of about 25 ms.

Fig. 5 shows the same simulation with the triangular elements. The overall response of the columns is quite similar but locally the triangles appear to be somewhat stiffer. Note that at the nodes, the cross section does not buckle in as far and that at the top and bottom of the columns the compression buckles are very slight.

Remark. In these problems the Il'yushin results obtained with the material law differ substantially from the Von Mises (integrated through the thickness). It predicts much stiffer behavior; this may be due to

Fig. 3. Undeformed and deformed meshes for axially loaded hollow column modeled with quadrilateral elements.
Fig. 4. Hourglass mode occurrence for axially loaded hollow column.

Fig. 5. Undeformed and deformed meshes for axially loaded hollow column modeled with triangular elements.
Fig. 6. LMR design with cylindrical ACS.

Fig. 7. Finite element model of cylindrical ACS.
errors in tracking shell behavior when one side unloads elastically while the other side continuously loads plastically.

Example 3: Thin shell ACS support structure (design B).

A new concept in the LMR design for the ACS is to use a single, large radius cylinder, as shown in fig. 6, where it is identified as the upper internal structure. The dimensions, material properties, and other aspects of the finite element model are given in table 3. The finite element model is shown in fig. 7. Note that although this is not shown in fig. 7, the bottom row of nodes are all rigidly interconnected to a single node and a mass of 40 000 kg is added to this node. To reflect the mass of the surrounding fluid, a mass of 3.28 kg was added to each shell node. Half of the cylinder is modeled, and a plane of symmetry is assumed. The same force was applied to the bottom of the model as for design A (although multiplied by 4 because this shell replaces four columns). In addition, a pressure load, with time history shown in table 4, was applied to the entire side of the cylinder; this load was not applied to design A for in view of the much greater thickness/radius ratio, it was not significant in the response of the columns.

As in the previous case, the initiation of a realistic buckling mode requires the introduction of an imperfection in the model. Three imperfections were tested:

\[ w = -\varepsilon \sin \frac{\pi z}{l} \sin 2\theta, \]
\[ w = -\varepsilon \sin \frac{\pi z}{l} \sin 6\theta, \quad (23) \]
\[ w = -\varepsilon \sin \frac{\pi z}{l} |\sin 2\theta|, \]

with \( \varepsilon = 5.08 \times 10^{-3} \) cm (2 \times 10^{-3} in.).

Imperfection (a) did not exhibit any lobe patterns in its buckling, whereas for imperfections (b) and (c), six and eight lobes, respectively, appeared in the post-buckled shape of the cylinder. Several deformed
meshes for imperfections (b) and (c) are shown in Figs. 8 and 9, respectively. As can be seen, the overall pattern of deformation for the two imperfections is quite similar except for the difference in the number of lobes.

Fig. 10 compares the time history of the vertical deflection of the rigid mass at the bottom of the cylinder. The response for the 4-column support is also shown. The important result is that the response is independent of the number of lobes in the post-buckling pattern. Thus, although the character of the
MADYMO Shell4 Cylinder

MADYMO Facet6 Column

MADYMO Shell4 Column
<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td></td>
<td>Large rotation about z-axis</td>
<td>None</td>
<td></td>
<td>Truss2/test1</td>
</tr>
<tr>
<td>Mass computation</td>
<td></td>
<td>Pointload on all nodes in x-dir.</td>
<td>None</td>
<td></td>
<td>Truss2/test2</td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e8, beamlength 3e-2</td>
<td>Pointload on 1 nodes in x-dir.</td>
<td>1 node, in x-dir.</td>
<td></td>
<td>Truss2/test3</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, beamlength 5e-2</td>
<td>Prescribed DOF for 1 nodes in x- &amp; y-dir.</td>
<td>1 nodes, in x-dir.</td>
<td></td>
<td>Truss2/test4</td>
</tr>
</tbody>
</table>
**SINGLE BEAM2**
MATERIALTYPE ISOLIN

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of a prescribed torque</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| MATERIALTYPE ISOLIN                          |                  |                              |                |                                             |                   |

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of a prescribed torque</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

| MATERIALTYPE ISOLIN                          |                  |                              |                |                                             |                   |

**MULTIPLE BEAM2 ELEMENTS**
MATERIALTYPE ISOLIN

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending of Beam2 elements combined in a beam, by prescribed force</td>
<td>E=2.1e11, beam dimensions = 0.6 * 0.1 * 10</td>
<td>Prescribed force in z-direction on last node</td>
<td>First node</td>
<td></td>
<td>Test.beam2/test4</td>
</tr>
<tr>
<td>Deformation of Beam2 elements combined in a beam, by prescribed rotation</td>
<td>E=2.1e11, beam dimensions = 0.6 * 0.1 * 10</td>
<td>Prescribed rotation on last node, in R1 direction</td>
<td>First node</td>
<td></td>
<td>Test.beam2/test6</td>
</tr>
<tr>
<td>Deformation of Beam2 elements combined in a beam, by prescribed rotation + prescribed force</td>
<td>E=2.1e11, beam dimensions = 0.6 * 0.1 * 10</td>
<td>Prescribed force in z-direction on last node &amp; prescribed rotation on last node, in R1 direction</td>
<td>First node</td>
<td>Instable solution process</td>
<td>Test.beam2/test7</td>
</tr>
</tbody>
</table>
## SINGLE MEM3
**MATERIALTYPE ISOLIN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td></td>
<td>Rotation, around z-axis, prescribed by</td>
<td>2 nodes, 1 supported by</td>
<td>There appear some stresses in the material</td>
<td>Test1.mem3/test17</td>
</tr>
<tr>
<td></td>
<td></td>
<td>moving null system</td>
<td>a moving null system</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>( E=2.1\times 10^7, \ Nu=0.4 )</td>
<td>Rotation, around z-axis, due to prescribed force</td>
<td>1 node</td>
<td></td>
<td>Test1.mem3/test11</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>( E=2.1\times 10^7, \ Nu=0.4 )</td>
<td>Rotation, around y-axis, due to prescribed force</td>
<td>2 nodes</td>
<td>The element is not able to handle large rotations</td>
<td>Test1.mem3/test20</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mass computation</td>
<td></td>
<td>Pointload on all nodes in 3 dir.</td>
<td>None</td>
<td></td>
<td>Test1.mem3/test18</td>
</tr>
<tr>
<td>Deformation as a result of an external force Poissons ratio = 0.4</td>
<td>( E=2.1\times 10^7, \ Nu=0.4 )</td>
<td>Pointload on 1 node in x-dir.</td>
<td>1 node</td>
<td></td>
<td>Test1.mem3/test5</td>
</tr>
<tr>
<td>Deformation as a result of an external force Poissons ratio = 0.4</td>
<td>( E=2.1\times 10^7, \ Nu=0.4 )</td>
<td>Pointload on 1 node in y-dir.</td>
<td>1 node</td>
<td></td>
<td>Test1.mem3/test6</td>
</tr>
<tr>
<td>Deformation as a result of an external force Poissons ratio = 0.4</td>
<td>( E=2.1\times 10^7, \ Nu=0.4 )</td>
<td>Pointload on 1 node in x-direction and on another in y-direction</td>
<td>1 node</td>
<td></td>
<td>Test1.mem3/test7</td>
</tr>
<tr>
<td>Shear-stress as a result of an external force Poissons ratio = 0.4</td>
<td>( E=2.1\times 10^7, \ Nu=0.4 )</td>
<td>Pointload on 1 node in direction(-1,1) and on another in direction(1,-1)</td>
<td>1 node</td>
<td></td>
<td>Test1.mem3/test8</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>( E=2.1\times 10^7, \ Nu=0.0 )</td>
<td>Pointload on 1 node in x-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.mem3/test1</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>( E=2.1\times 10^7, \ Nu=0.0 )</td>
<td>Pointload on 1 node in y-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.mem3/test3</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>( E=2.1\times 10^7, \ Nu=0.4 )</td>
<td>Pointload on 1 node in x-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.mem3/test2</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
**SINGLE MEM3**
**MATERIALTYPE ISOLIN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation as a result of a prescribed displacement Poissons ratio = 0.4</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Pointload on 1 node in y-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.mem3/test4</td>
</tr>
<tr>
<td>Shear-stress as a result of a prescribed displacement</td>
<td>E=2.1e11, Nu=0.0 thickness = 0.05</td>
<td>Displacements by prescr. DOF. 1 node in direction (-1,1) another in direction (1,-1)</td>
<td>1 node</td>
<td>The shear-stresses aren't calculated correctly</td>
<td>Test1.mem3/test9</td>
</tr>
<tr>
<td>Shear-stress as a result of a prescribed displacement out of the x-y plane</td>
<td>E=2.1e11, Nu=0.0 thickness = 0.05</td>
<td>2 nodes connected to a moving null system</td>
<td>1 node to inertial space</td>
<td>The shear-stresses aren't calculated correctly</td>
<td>Test1.mem3/test10</td>
</tr>
<tr>
<td>Stability for small elements</td>
<td>Size = 2mm * 2mm thickness = 0.05mm</td>
<td>Equal to test10 only a factor 1000 smaller</td>
<td></td>
<td>None (of course the shear-stresses are still wrong)</td>
<td>Test1.mem3/test19</td>
</tr>
</tbody>
</table>

**SINGLE MEM3**
**MATERIALTYPE ISOPLA**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plastic strain and yield stress in 1-dimensional deformation</td>
<td>E=2.1e9, ( \sigma_{02} = 250e6 ) ( \epsilon_{02} = 0.11905 ) The hardening curve contains 2 slopes</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td>The stresses are significantly larger then theoretically expected</td>
<td>Mem3/t6</td>
</tr>
<tr>
<td>Elastoplastic material-behaviour in shear deformations</td>
<td>E=2.1e9, ( \sigma_{02} = 250e6 ) ( \epsilon_{02} = 0.11905 )</td>
<td>Prescribed DOF for 3 nodes in x- &amp; y-dir.</td>
<td>1 node in all dir. and 3 only in z-dir.</td>
<td>The element coordinate-system introduces fictive stresses</td>
<td>Mem3/t7</td>
</tr>
</tbody>
</table>
### SINGLE MEM3
**MATERIALTYPE ANISO**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Validation of materialtype by modelling isolinear materialbehaviour</td>
<td>E11=E22=2.1e9, Nu=0.0, E33=0.5*E=1.05e9 edgelsize 3e-2</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td>None</td>
<td>Mem3/t1</td>
</tr>
<tr>
<td>Validation of materialtype by modelling isolinear materialbehaviour</td>
<td>E11=E22=2.3e9, Nu=0.3, E33=1.6e9 edgelsize 3e-2</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td>None</td>
<td>Mem3/t2</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E11=2.3e9, E22=2.31e9, E33=1.6e9, E12=0.69e9</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td>None</td>
<td>Mem3/t4</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E11=2.4e9, E22=2.24e9, E33=1.6e9, E12=0.65e9</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>2 nodes in all dir. and 2 nodes only in y-dir.</td>
<td>None</td>
<td>Mem3/t5</td>
</tr>
<tr>
<td>Shear deformation as a result of a prescribed displacement</td>
<td>E11=2.3e9, E22=2.31e9, E33=1.6e9, E12=0.69e9</td>
<td>Prescribed DOF for 2 nodes in both x- &amp; y-dir.</td>
<td>2 nodes in all dir.</td>
<td>The element-coordinate-system introduces deviations</td>
<td>Mem3/t3</td>
</tr>
</tbody>
</table>

### SINGLE MEM3
**MATERIALTYPE ISOTEN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e9, Nu=0.3 edgelsize 3e-2</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in y-dir.</td>
<td>This materialtype does not work well</td>
<td>Mem3/t8</td>
</tr>
</tbody>
</table>

64
Hysteresis material behaviour is not accurately modelled. It will be completely revised in future versions of Madymo.

Note. For these tests 2 MEM3 elements are combined in a square.
## SINGLE SHELL6
**MATERIALTYPE ISOLIN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Rotation, around z-axis, due to prescribed force</td>
<td>1 node</td>
<td>There appear some stresses in the material</td>
<td>Test1.shell6/test11</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Rotation, around y-axis, due to prescribed force</td>
<td>2 nodes</td>
<td>The element is not able to handle large rotations</td>
<td>Test1.shell6/test20</td>
</tr>
<tr>
<td>Mass computation</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Pointload on 1 node in x-dir.</td>
<td>None</td>
<td>Deformation as a result of an external force</td>
<td>Test1.shell6/test5</td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Pointload on 1 node in y-dir.</td>
<td>1 node</td>
<td>Poissons ratio = 0.4</td>
<td>Test1.shell6/test6</td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Pointload on 1 node in x-direction and on another in y-direction</td>
<td>1 node</td>
<td>Poissons ratio = 0.4</td>
<td>Test1.shell6/test7</td>
</tr>
<tr>
<td>Shear-stress as a result of an external force</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Pointload on 1 node in direction(-1,1) and on another in direction(1,-1)</td>
<td>1 node</td>
<td>Poissons ratio = 0.4</td>
<td>Test1.shell6/test8</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed</td>
<td>E=2.1e11, Nu=0.0 thickness = 0.05</td>
<td>Pointload on 1 node in x-dir.</td>
<td>2 nodes</td>
<td>Deformation as a result of a prescribed displacement</td>
<td>Test1.shell6/test1</td>
</tr>
<tr>
<td>displacement</td>
<td>E=2.1e11, Nu=0.0 thickness = 0.05</td>
<td>Pointload on 1 node in y-dir.</td>
<td>2 nodes</td>
<td>Deformation as a result of a prescribed displacement</td>
<td>Test1.shell6/test3</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed</td>
<td>E=2.1e11, Nu=0.4 thickness = 0.05</td>
<td>Pointload on 1 node in x-dir.</td>
<td>2 nodes</td>
<td>Deformation as a result of a prescribed displacement</td>
<td>Test1.shell6/test2</td>
</tr>
</tbody>
</table>
### SINGLE SHELL6
#### MATERIALTYPE ISOLIN

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation as a result of a prescribed displacement Poissons ratio = 0.4</td>
<td>E=2.1e11, Nu=0.4 thickness =0.05</td>
<td>Pointload on 1 node in y-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.shell6/test4</td>
</tr>
<tr>
<td>Shear-stress as a result of a prescribed displacement</td>
<td>E=2.1e11, Nu=0.0 thickness = 0.05</td>
<td>Displacements by prescr. DOF. 1 node in direction (-1,1) another in direction (1,-1)</td>
<td>1 node</td>
<td>The shear-stresses aren't calculated correctly</td>
<td>Test1.shell6/test9</td>
</tr>
<tr>
<td>Shear-stress as a result of a prescribed displacement out of the x-y plane</td>
<td>E=2.1e11, Nu=0.0 thickness = 0.05</td>
<td>2 nodes connected to a moving null system</td>
<td>1 node to inertial space</td>
<td>The shear-stresses aren't calculated correctly</td>
<td>Test1.shell6/test10</td>
</tr>
<tr>
<td>Stability for small elements</td>
<td>Size = 2mm * 2mm thickness = 0.05mm</td>
<td>Equal to test10 only a factor 1000 smaller</td>
<td></td>
<td>None (of course the shear-stresses are still wrong)</td>
<td>Test1.shell6/test19</td>
</tr>
</tbody>
</table>

### SINGLE SHELL6
#### MATERIALTYPE ISOPLA

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastoplastic material behaviour when a hardening function is defined</td>
<td>E=2.1e9, ( \sigma_{e2} = 250 e6 ) edgewise 3e-2 thickness 5e-3</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td>No convergence to yield surface</td>
<td>Shell6/t1</td>
</tr>
<tr>
<td>Elastoplastic material behaviour in shear deformations</td>
<td>E=2.1e9, ( \sigma_{e2} = 250 e6 ) edgewise 3e-2 thickness 5e-3</td>
<td>Prescribed DOF for 3 nodes in x &amp; y-dir.</td>
<td>1 node in all dir. and 3 only in z-dir.</td>
<td>The element coordinate system introduces fictive stresses</td>
<td>Shell6/t2</td>
</tr>
</tbody>
</table>
### MULTIPLE SHELL6
#### MATERIALTYPE ISOLIN

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending of shell6 elements combined in a beam, by predescribed force</td>
<td>$E=2.1\times10^1$, $Nu=0.0$</td>
<td>Force in negative z-direction at the end of the beam</td>
<td>2 transl. nodes and 1 rot. node are supported</td>
<td></td>
<td>Test.shell6/strip1</td>
</tr>
<tr>
<td>Bending of small shell6 elements combined in a beam, by predescribed force</td>
<td>$E=2.1\times10^1$, $Nu=0.0$</td>
<td>Force in negative z-direction at the end of the beam</td>
<td>2 transl. nodes and 1 rot. node are supported</td>
<td></td>
<td>Test.shell6/strip6</td>
</tr>
<tr>
<td>Bending of shell6 elements combined in a beam, by predescribed torque</td>
<td>$E=2.1\times10^1$, $Nu=0.0$</td>
<td>Torque about y-axis of the last element</td>
<td>2 transl. nodes and 1 rot. node are supported</td>
<td></td>
<td>Test.shell6/strip2</td>
</tr>
<tr>
<td>Stretching and rotation out of the xy-plane of shell6 elements combined in a beam, by predescribed force</td>
<td>$E=2.1\times10^1$, thickness=1</td>
<td>Force in z-direction at the end of the beam</td>
<td>2 transl. nodes are supported</td>
<td>Fault ≈ 1%</td>
<td>Test.shell6/strip3</td>
</tr>
<tr>
<td>Stretching and rotation in the xy-plane of shell6 elements combined in a beam, by predescribed force</td>
<td>$E=2.1\times10^1$</td>
<td>Force in y-direction at the end of the beam</td>
<td>1 transl. node is supported</td>
<td>Fault = 5%</td>
<td>Test.shell6/strip4 Test.shell6/strip5</td>
</tr>
<tr>
<td>3 dimensional rotation and translation of shell6 elements combined in a beam, by prescribed DOF</td>
<td>1 rotational node prescibed rotation.</td>
<td></td>
<td>2 transl. nodes supported by rotating and transl. null system</td>
<td>Fault = 1%</td>
<td>Test.shell6/strip7</td>
</tr>
<tr>
<td>TEST-AIM</td>
<td>PARAMETERS</td>
<td>LOADS &amp; DISPLACEMENTS &amp; DIR.</td>
<td>SUPPORTS</td>
<td>PROBLEMS</td>
<td>TESTNAME</td>
</tr>
<tr>
<td>-------------------------------</td>
<td>--------------------</td>
<td>------------------------------------------------------------------</td>
<td>---------------------------</td>
<td>---------------------------------------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td></td>
<td>Rotation, around z-axis, prescribed by moving null system</td>
<td>2 nodes, 1 supported by a moving null system</td>
<td>There appear some stresses in the material</td>
<td>Test1.facet6/test17</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>$E=2.1e11$, $Nu=0.4$ thickness = 0.05</td>
<td>Rotation, around z-axis, due to prescribed force</td>
<td>1 node</td>
<td></td>
<td>Test1.facet6/test11</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>$E=2.1e11$, $Nu=0.4$ thickness = 0.05</td>
<td>Rotation, around y-axis, due to prescribed force</td>
<td>2 nodes are supported</td>
<td>The element is not able to handle large rotations</td>
<td>Test1.facet6/test20</td>
</tr>
<tr>
<td>Mass computation</td>
<td></td>
<td>Pointload on all nodes in 3 dir.</td>
<td>None</td>
<td></td>
<td>Test1.facet6/test18</td>
</tr>
<tr>
<td>Deformation as a result of an external force Poissons ratio = 0.4</td>
<td>$E=2.1e11$, $Nu=0.4$ thickness = 0.05</td>
<td>Pointload on 1 node in x-dir.</td>
<td>1 node</td>
<td></td>
<td>Test1.facet6/test5</td>
</tr>
<tr>
<td>Deformation as a result of an external force Poissons ratio = 0.4</td>
<td>$E=2.1e11$, $Nu=0.4$ thickness = 0.05</td>
<td>Pointload on 1 node in y-dir.</td>
<td>1 node</td>
<td></td>
<td>Test1.facet6/test6</td>
</tr>
<tr>
<td>Deformation as a result of an external force Poissons ratio = 0.4</td>
<td>$E=2.1e11$, $Nu=0.4$ thickness = 0.05</td>
<td>Pointload on 1 node in x-direction and on another in y-direction</td>
<td>1 node</td>
<td></td>
<td>Test1.facet6/test7</td>
</tr>
<tr>
<td>Shear-stress as a result of an external force Poissons ratio = 0.4</td>
<td>$E=2.1e11$, $Nu=0.4$ thickness = 0.05</td>
<td>Pointload on 1 node in direction(-1,1) and on another in direction(1,-1)</td>
<td>1 node</td>
<td></td>
<td>Test1.facet6/test8</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>$E=2.1e11$, $Nu=0.0$ thickness = 0.05</td>
<td>Pointload on 1 node in x-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.facet6/test1</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>$E=2.1e11$, $Nu=0.0$ thickness = 0.05</td>
<td>Pointload on 1 node in y-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.facet6/test3</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement Poissons ratio = 0.4</td>
<td>$E=2.1e11$, $Nu=0.4$ thickness = 0.05</td>
<td>Pointload on 1 node in x-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.facet6/test2</td>
</tr>
</tbody>
</table>
### SINGLE FACET6
**MATERIALTYPE ISOLIN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e11, Nu=0.4, thickness =0.05</td>
<td>Pointload on 1 node in y-dir.</td>
<td>2 nodes</td>
<td></td>
<td>Test1.facet6/test4</td>
</tr>
<tr>
<td>Shear-stress as a result of a prescribed displacement</td>
<td>E=2.1e11, Nu=0.0, thickness = 0.05</td>
<td>Displacements by prescr. DOF, 1 node in direction (-1,1) another in direction (1,-1)</td>
<td>1 node</td>
<td>The shear-stresses aren't calculated correctly</td>
<td>Test1.facet6/test9</td>
</tr>
<tr>
<td>Shear-stress as a result of a prescribed displacement out of the x-y plane</td>
<td>E=2.1e11, Nu=0.0, thickness = 0.05</td>
<td>2 nodes connected to a moving null system</td>
<td>1 node to inertial space</td>
<td>The shear-stresses aren't calculated correctly</td>
<td>Test1.facet6/test10</td>
</tr>
<tr>
<td>Stability for small elements</td>
<td>Size = 2mm * 2mm, thickness = 0.05mm</td>
<td>Equal to test10 only a factor 1000 smaller</td>
<td></td>
<td>None (of course the shear-stresses are still wrong)</td>
<td>Test1.facet6/test19</td>
</tr>
</tbody>
</table>

**SINGLE FACET6**
**MATERIALTYPE ISOPLA**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plastic strain and yield stress in 1-dimensional deformation</td>
<td>E=2.1e9, σ₀₂ =250e6, ε₀₂ =0.11905 The hardening curve contains 2 slopes</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td>The stresses are significantly larger than theoretically expected</td>
<td>Facet6/t1</td>
</tr>
<tr>
<td>Elastoplastic material-behaviour in shear deformations</td>
<td>E=2.1e9, σ₀₂ =250e6, ε₀₂ =0.11905</td>
<td>Prescribed DOF for 3 nodes in x- &amp; y-dir.</td>
<td>1 node in all dir. and 3 only in z-dir.</td>
<td>The element coordinate-system introduces fictive stresses</td>
<td>Facet6/t2</td>
</tr>
</tbody>
</table>
## MULTIPLE FACET6
### MATERIALTYPE ISOLIN

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending of facet6 elements combined in a beam, by predescribed force</td>
<td>E=2.1e11, Nu=0.0 thickness=1</td>
<td>Force in negative z-direction at the end of the beam</td>
<td>2 transl. nodes and 1 rot. node are supported</td>
<td>Fault = 5%</td>
<td>Test.facet6/strip1</td>
</tr>
<tr>
<td>Bending of small facet6 elements combined in a beam, by predescribed force</td>
<td>E=2.1e11, Nu=0.0 thickness=0.01</td>
<td>Force in negative z-direction at the end of the beam</td>
<td>2 transl. nodes and 1 rot. node are supported</td>
<td>Fault = 5%</td>
<td>Test.facet6/strip6</td>
</tr>
<tr>
<td>Bending of facet6 elements combined in a beam, by predescribed torque</td>
<td>E=2.1e11, Nu=0.0 thickness=1</td>
<td>Torque about y-axis of the last element</td>
<td>2 transl. nodes and 1 rot. node are supported</td>
<td></td>
<td>Test.facet6/strip2</td>
</tr>
<tr>
<td>Stretching and rotation out of the xy-plane of facet6 elements combined in a beam, by predescribed force</td>
<td>E=2.1e11 thickness=1</td>
<td>Force in z-direction at the end of the beam</td>
<td>2 transl. nodes are supported</td>
<td></td>
<td>Test.facet6/strip3</td>
</tr>
<tr>
<td>Stretching and rotation in the xy-plane of facet6 elements combined in a beam, by predescribed force</td>
<td>E=2.1e11</td>
<td>Force in y-direction at the end of the beam</td>
<td>1 transl. node is supported</td>
<td>Fault = 1%</td>
<td>Test.facet6/strip4 Test.facet6/strip5</td>
</tr>
<tr>
<td>3 dimensional rotation and translation of facet6 elements combined in a beam, by prescribed DOF</td>
<td>1 rotational node prescribed rotation.</td>
<td>2 transl. nodes supported by rotating and transl. null system</td>
<td>Fault = 1%</td>
<td></td>
<td>Test.facet6/strip7</td>
</tr>
</tbody>
</table>
## SINGLE SHELL4
### MATERIALTYPE ISOLIN

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td>Prescribed DOF for all nodes in x-dir.</td>
<td>None</td>
<td>Shell4/test1</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>Prescribed rotation around x-, y- &amp; z-axis</td>
<td>1 node is supported</td>
<td>Hourglass-mode</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotation and stretching</td>
<td>Point load on 2 nodes</td>
<td>2 nodes are supported</td>
<td>Shell4/test14</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mass Computation</td>
<td>Prescribed load, acceleration-type on 4 nodes in x-, y- &amp; z-dir.</td>
<td>None</td>
<td>Shell4/test13</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e8, Nu=0.0 edgsize 3e-3 thickness 5e-4</td>
<td>Pointload on 2 nodes in x-dir.</td>
<td>Shell4/test2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e8, Nu=0.3 edgsize 9.46e-3 thickness 5e-4</td>
<td>Pointload on 2 nodes in x-dir.</td>
<td>Shell4/test5</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of a transversal external force (Bending-problem)</td>
<td>E=2.1e8, Nu=0.0 edgsize 1e-1 thickness 2e-3</td>
<td>Distributed force (Stepfunction in time) Strip of 5 elements</td>
<td>Shell4/test7</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, Nu=0.0 edgsize 9.46e-3 thickness 5e-4</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>Shell4/test3</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement Poissons ratio 0.3</td>
<td>E=2.1e8, Nu=0.3 edgsize 9.46e-3 thickness 5e-4</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>Shell4/test4</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Shear-deformation as a result of prescribed displacements</td>
<td>E=2.1e8, Nu=0.0 edgsize 1e-2 thickness 5e-4</td>
<td>Prescribed DOF for 3 nodes in x- &amp; y-dir.</td>
<td>Shell4/test6</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of prescribed transversal displacements (Bending)</td>
<td>E=2.1e8, Nu=0.0 edgsize 1e-2 thickness 2e-3</td>
<td>Prescribed DOF for 2 nodes in z-dir. Strip of 5 elements</td>
<td>Shell4/test8</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
# SINGLE SHELL4

**MATERIALTYPE ISOPLA**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ideal elastoplastic material-behaviour</td>
<td>E=2.1e9, σ_e2 =250e6</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td></td>
<td>Shell4/test9</td>
</tr>
<tr>
<td></td>
<td>edgewise 3e-2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>thickness 5e-3</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Elastoplastic material-behaviour</td>
<td>E=2.1e9, σ_e2 =250e6</td>
<td>Prescribed DOF for 2 nodes in x-dir.</td>
<td>1 node in all dir. and 1 only in x-dir.</td>
<td></td>
<td>Shell4/test10</td>
</tr>
<tr>
<td>when a hardening-function is defined</td>
<td>edgewise 3e-2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>thickness 5e-3</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Elastoplastic material-behaviour</td>
<td>E=2.1e9, σ_e2 =250e6</td>
<td>Prescribed DOF for 3 nodes in x- &amp; y-dir.</td>
<td>1 node in all dir. and 3 only in z-dir.</td>
<td>The element-coordinatesystem introduces fictive stresses</td>
<td>Shell4/test11</td>
</tr>
<tr>
<td>in shear-deformations</td>
<td>edgewise 3e-2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>thickness 5e-3</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Elastoplastic material-behaviour</td>
<td>E=2.1e9, σ_e2 =250e6</td>
<td>Prescribed Momentum for 2 nodes Strip of 5 elements</td>
<td>2 nodes in all dir. all other nodes only in y-dir</td>
<td></td>
<td>Shell4/test12</td>
</tr>
<tr>
<td>in bending problems</td>
<td>edgewise 1e-2</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>thickness 5e-4</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### SINGLE SOLID1

**MATERIALTYPE ISOLIN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td>Prescribed DOF for all nodes</td>
<td>None</td>
<td>The element is not able to handle large rotations</td>
<td>Solid1/test1</td>
<td></td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>Large rotation around x-axis</td>
<td>2 nodes are supported</td>
<td></td>
<td>Solid1/test6</td>
<td></td>
</tr>
<tr>
<td>Mass computation</td>
<td>Pointload on all nodes in 3 dir.</td>
<td>None</td>
<td></td>
<td>Solid1/test7</td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e8, Nu=0.0 edgewise 1.32e-2</td>
<td>Pointload on 4 nodes in x-dir.</td>
<td>4 nodes, all in x-dir.</td>
<td>Solid1/test2</td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, Nu=0.0 edgewise 1.52e-2</td>
<td>Prescribed DOF for 4 nodes in x-dir.</td>
<td>4 nodes, all in x-dir.</td>
<td>Solid1/test3</td>
<td></td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement Poissons ratio = 0.3</td>
<td>E=2.1e8, Nu=0.3 edgewise 1.63e-2</td>
<td>Prescribed DOF for 4 nodes in x-dir.</td>
<td>3 nodes in x-dir and 1 in all dir.</td>
<td>Instable solution-process</td>
<td>Solid1/test4</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement Poissons ratio = 0.3</td>
<td>E=2.1e8, Nu=0.3 edgewise 1.63e-1</td>
<td>Prescribed DOF for 4 nodes in z-dir.</td>
<td>3 nodes in x-dir. and 1 in all dir.</td>
<td>Instable solution-process</td>
<td>Solid1/test5</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement Poissons ratio = 0.3</td>
<td>E=2.1e8, Nu=0.3 edgewise 1.63e-2</td>
<td>Prescribed DOF for 4 nodes in z-dir.</td>
<td>3 nodes in x-dir. and 1 in all dir.</td>
<td></td>
<td>Solid1/test8</td>
</tr>
<tr>
<td>Shear-deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, Nu=0.0 edgewise 3e-2</td>
<td>Prescribed DOF for 4 nodes in x- &amp; y-dir.</td>
<td>4 nodes in all dir.</td>
<td></td>
<td>Solid1/test9</td>
</tr>
</tbody>
</table>

Main problem in testing single Solid1-elements is to avoid the so-called hourglass-modes. Supports can be helpful to effort this but may introduce some deviations between theoretical and numerical results. The Solid1-element appeared to be not well functioning in some trivial loading cases. Until these problems are solved no further tests will be performed.

All the tests mentioned in the table have been carried out with the ISOLIN-materialtype. We have also tried to test hysteresis for single Solid1-elements with the HYSISO-materialtype. This materialtype appeared to be not well modelled for Solid1-elements and therefore these tests are not inserted here. The HYSISO-materialtype will be revised.

**Note.** Tests 6 & 7 are not inserted here. These tests are respectively described in test18 & test19 in the Solid8-chapter.
# SINGLE SOLID8

**MATERIALTYPE ISOLIN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body-movement</td>
<td>Prescribed DOF in x-dir.</td>
<td></td>
<td></td>
<td></td>
<td>Solid8/test1</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>Gravitation, z-dir.</td>
<td></td>
<td></td>
<td></td>
<td>Solid8/test16</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>Rotation, around z-axis</td>
<td></td>
<td></td>
<td></td>
<td>Solid8/test17</td>
</tr>
<tr>
<td>Rigid body-movement</td>
<td>Large rotation around x-axis</td>
<td>2 nodes are supported</td>
<td></td>
<td>The element is not able to handle large</td>
<td>Solid8/test18</td>
</tr>
<tr>
<td>Mass computation</td>
<td>Pointload on all nodes in 3 dir.</td>
<td>None</td>
<td></td>
<td></td>
<td>Solid8/test19</td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e8, Nu=0.0 edgsize 1.32e-2</td>
<td>Pointload on 4 nodes in x-dir.</td>
<td>4 nodes</td>
<td>Instable solution-process</td>
<td>Solid8/test2</td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e8, Nu=0.0 edgsize 1.52e-2</td>
<td>Pointload on 4 nodes in y-dir.</td>
<td>4 nodes</td>
<td></td>
<td>Solid8/test4</td>
</tr>
<tr>
<td>Deformation as a result of an external force</td>
<td>E=2.1e8, Nu=0.3 edgsize 1.63e-1</td>
<td>Pointload on 4 nodes in z-dir.</td>
<td>4 nodes</td>
<td>only 1 in all dir.</td>
<td>Solid8/test8</td>
</tr>
<tr>
<td>Shear-deformation as a result of an external force</td>
<td>E=2.1e8, Nu=0.0 edgsize 3e-2</td>
<td>Pointload on 4 nodes in x- and y-dir.</td>
<td>4 nodes</td>
<td>in all dir.</td>
<td>Solid8/test12</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, Nu=0.0 edgsize 1.52e-2</td>
<td>Prescribed DOF for 4 nodes in x-dir.</td>
<td>4 nodes</td>
<td></td>
<td>Solid8/test3</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, Nu=0.3 edgsize 1.63e-2</td>
<td>Prescribed DOF for 4 nodes in z-dir.</td>
<td>4 nodes</td>
<td>only 1 in all dir.</td>
<td>Solid8/test6</td>
</tr>
<tr>
<td>Deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, Nu=0.3 edgsize 1.63e-1</td>
<td>Prescribed DOF for 4 nodes in z-dir.</td>
<td>4 nodes</td>
<td>only 1 in all dir.</td>
<td>Solid8/test7</td>
</tr>
</tbody>
</table>
### SINGLE SOLID8

**MATERIAL TYPE ISOLIN**

<table>
<thead>
<tr>
<th>TEST-AIM</th>
<th>PARAMETERS</th>
<th>LOADS &amp; DISPLACEMENTS &amp; DIR.</th>
<th>SUPPORTS</th>
<th>PROBLEMS</th>
<th>TESTNAME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear-deformation as a result of a prescribed displacement</td>
<td>E=2.1e8, Nu=0.0 edgelsize 3e-2</td>
<td>Prescribed DOF for 4 nodes in x- &amp; y-dir.</td>
<td>4 nodes in all dir.</td>
<td></td>
<td>Solid8/test9</td>
</tr>
<tr>
<td>Shear-deformation as a result of a large prescribed displacement</td>
<td>E=2.1e8, Nu=0.0 edgelsize 3e-2</td>
<td>Prescribed DOF for 4 nodes in x- &amp; y-dir.</td>
<td>4 nodes in all dir.</td>
<td>Unsatisfactory accuracy</td>
<td>Solid8/test11</td>
</tr>
<tr>
<td>Shear-deformation as a result of a prescribed displacement for a longdrawn element</td>
<td>E=2.1e8, Nu=0.0 edgelsize x: 9e-2 y: 2e-2 z: 1e-2</td>
<td>Prescribed DOF for 4 nodes in z-dir.</td>
<td>4 nodes in all dir.</td>
<td>Unsatisfactory accuracy</td>
<td>Solid8/test13</td>
</tr>
<tr>
<td>Shear deformation as a result of a prescribed displacement for a longdrawn element</td>
<td>E=2.1e8, Nu=0.0 edgelsize x: 5e-2 y: 2e-2 z: 1e-2</td>
<td>Prescribed DOF for 4 nodes in z-dir.</td>
<td>4 nodes in all dir.</td>
<td>Unsatisfactory accuracy</td>
<td>Solid8/test14</td>
</tr>
</tbody>
</table>

All the tests mentioned in the table have been carried out with the ISOLIN-materialtype. We have also tried to test hysteresis for single Solid8-elements, with the HYSISO-materialtype. This materialtype appeared to be not well modelled for Solid8-elements and therefore these tests are not inserted. The HYSISO-materialtype will be revised.