

# Finite element calculations on cold upsetting of anisotropic materials

**Citation for published version (APA):**

Kuiken, J. (1993). *Finite element calculations on cold upsetting of anisotropic materials*. (TH Eindhoven. Afd. Werktuigbouwkunde, Vakgroep Produktietechnologie : WPB; Vol. WPA1473-1). Technische Universiteit Eindhoven.

**Document status and date:**

Published: 01/01/1993

**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 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](#)

**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.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

[www.tue.nl/taverne](http://www.tue.nl/taverne)

**Take down policy**

If you believe that this document breaches copyright please contact us at:

[openaccess@tue.nl](mailto:openaccess@tue.nl)

providing details and we will investigate your claim.

# FINITE ELEMENT CALCULATIONS ON COLD UPSETTING OF ANISOTROPIC MATERIALS

WPA 1473

by: J.Kuiken

WPA 1473

January 1993

Eindhoven University of Technology

# CONTENTS

1 INTRODUCTION	3
2 ABAQUS IN GENERAL	4
3 PROBLEM DEFINITION	8
3.1 Material behavior: strain hardening and anisotropy	8
3.2 Boundary conditions, loading and solution control	10
4 RESULTS AND DISCUSSION	12
REFERENCES	16
APPENDIX 1 THE FINAL ABAQUS FILE	17

# 1 INTRODUCTION

To define the ductility of different aluminium alloys a large number of experiments has to be carried out. As a part of these experiments also upsetting tests were done. In order to evaluate failure criteria it would be useful if these experiments could be carried out with a finite element model. The problem is that the relation between model and experiment is influenced very much by the friction between test piece and die and the anisotropic material properties. In order to get detailed information of the stress system the measured strain-path has to be fitted. A physical meaningful curve fit can be obtained by FEM. In this way no assumptions about the shape of the strain-path (e.g. linear or parabolic) have to be made. The strain-path up to failure at the equatorial free surface was measured in these experiments and the calculated strain-path must be the curve-fit of the measured strain-path. However, the finite element calculations have to be carried out many times to compare the resulting strain-path relation with the experimental values. This can only be done when the amount of CPU time needed for one calculation is no longer than about 4 hours. The standard input data-deck from the ABAQUS problems example, with minimal stress accuracy, demands about 80-100 hours for one calculation. The goal of this work is to reduce the calculation time and to increase the final stress accuracy, allowing a change of the frictional values of the model to find the optimal values to describe a certain set of experiments.

## 2 ABAQUS IN GENERAL.

A finite element model consists of a geometric description, which is given by the elements and their nodes (the "bulk data"), and a set of properties associated with the elements, describing their attributes: material definitions, cross-section definitions in the case of structural elements like beams and shells, and other parameters for interface elements, springs, etc. There may also be constraints that must be included in the model -"multi-point constraints" or "equations": linear or non-linear equations involving several of the fundamental solution variables in the model, or simple "boundary conditions" that are to be imposed at the start of the analysis. Environment properties, such as attributes of the fluid surrounding the model, must also be defined in some cases. Non-zero initial conditions, such as initial stresses temperatures or velocities, may also be required.

All of these are classified in ABAQUS as "model definition data", and are given as the first part of an ABAQUS analysis data deck.

Various "mesh display" options are provided so that the finite element model can be displayed on a graphics or plotting device. The use of such displays is strongly recommended.

All the different features of ABAQUS are described in the USERS-manual, the THEORY-manual, and the EXAMPLE PROBLEMS-manual. Although these manuals provide the necessary information it takes some experience to be able to transform a given problem into an ABAQUS data-deck.

The "bulk data" (elements and nodes) define the finite element model geometrically. The behavior of the elements must also be defined. This can range from the simplest case of a single spring constant, to such complex cases as a laminated composite shell with nonlinear behavior in its laminae. In ABAQUS these definitions of element behavior are provided through the property assignment options, like \*SOLID SECTION, \*SHELL SECTION, etc.

The purpose of an analysis is to predict the response of a model to some form of external loading, or some non-equilibrium initial condition. ABAQUS is based on the concept of analysis steps, each step being a portion or period of the history. Typically, a step is simply a change from one load to another. In nonlinear cases ABAQUS will increment and iterate as necessary to analyze a step, depending on the severity of the nonlinearity. The

state at the end of a step provides the initial conditions for the next step, making it easy to simulate consecutive loadings of a model, such as dynamic responses following a static preload, or the loading of a product during its usage following a simulation of the manufacturing process. A "step" in ABAQUS is any convenient period of the analysis procedure. New steps may be introduced simply to change the output requests, or to change the load, or to change the analysis procedure. There is no limit to the number of steps in an analysis. The number of increments in a step, and the number of iterations in each increment in a nonlinear case are both generally limited by parameters on the \*STEP card.

The history input consists of the following options:

#### 1) Response type.

Here ABAQUS is told to perform a certain type of analysis: static, dynamic, transient, etc. The type of analysis can be changed from step to step. For example, a static preload may be analyzed first, then the response type may be changed to dynamic transient, and a dynamic analysis may be performed, based on the initial conditions left at the end of the static solution. In most cases the user is given the option to direct solution control, where the subdivision of a step into increments is provided directly, or tolerance based control, where the user inputs tolerances and the solution is generated in order to conform to these tolerances, the program choosing how it subdivides the step.

#### 2) Loading.

Here the external parameters are varied to define the history. The simplest form of loading input is to define the amplitude of a load at the end of each step. ABAQUS will then either assume that the variation is applied instantaneously at the beginning of the step, depending on the response type that was chosen, or the user's choice, and it will analyze the step as a series of increments, choosing the increment size either directly or via tolerance parameters under user control. The loading may also be a prescribed function of time such as a known ground motion in a seismic event, or a pressure or temperature transient in a fluid surrounded structure.

#### 3) Output control.

Four types of output are available:

- a) printed output;
- b) results plotting (which also may be done by post-processing);
- c) storage of the results on a computer-readable file, for subsequent post-analysis use as a data file (this can be used to generate time-history and variable-variable plots, using the ABAQUS post-processor);
- d) restart file output, for subsequent continuation of the analysis, or for later use in obtaining more printed or plotted output.

The history input cards are put together as needed to define the history of the analysis - typically several steps will be used to describe the entire history.

#### 4) The data in the input data-deck.

Two types of cards are used: keyword cards and data cards.

Keyword cards introduce options or sub-options within a particular option block. The keyword itself (such as \*NODE, \*ELEMENT, \*MATERIAL) must be given exactly as defined in the manual, except that blank characters can be omitted. A keyword card always begins with the character \*. Keyword cards often have parameters, which are used to specify values for the data options controlled by the keyword. A parameter may have a value.

Data cards provide the numeric entries. On any data card, free or fixed format may be used. The comma is used as the separator for free format. Free and fixed format data cards may both appear in any option: the user can switch from free to fixed format (or vice versa) from one card to the next, but not on a card: if a comma appears anywhere on a card, it is assumed to be a free format card.

#### 5) Conventions.

Certain conventions are used throughout ABAQUS, as follows:

##### a) Degrees of freedom.

The degrees of freedom in ABAQUS are always referred to as follows:

- 1      x displacement
- 2      y displacement

- 3 z displacement
- 4 rotation about the x-axis
- 5 rotation about the y-axis
- 6 rotation about the z-axis
- 8 pore pressure
- 11 temperature
- 12 second temperature (for shells or beams)
- 13 third temperature (for shells or beams)
- 14 etc

except for axisymmetric elements, where the degrees of freedom are:

- 1 r displacement
- 2 z displacement
- 6 rotation in the (r-z) plane

b) Units.

ABAQUS has no units built into it. Therefore the units chosen must be self-consistent.

Units are indicated for the value to be given on load and flux types as follows:

DIMENSION	INDICATOR	EXAMPLE( S.I.units)
length	L	meter
mass	M	kilogram
time	T	second
temperature	Θ	degree Celsius
force	F	Newton
energy	J	Joule

6) The used input data deck.

From the used input data deck, the bulk data for nodes, elements and boundary conditions is removed in order to compress the information. The exact values for all nodes and elements are not important, when only the used cards are to be discussed. In appendix 1 the used optimal, with respect to minimum computational costs and maximal stress accuracy, input data-deck is given.



# 3 PROBLEM DEFINITION

The geometry is the standard testcase of Lippmann. et al (1979), and the deformed geometry is shown in figure 3. A circular billet, 30, 20 or 15 mm long, with a radius of 10 mm, compressed between flat, rough, rigid dies. Experimental values were obtained with concentrically grooved dies and as a result of this the optimal frictional values must be determined through a change of the friction parameters.

The finite element model is axisymmetric and includes the top half of the billet only, since the middle surface of the billet is a plane of symmetry. Elements of the type CAX4R are initially used. These are four node quadrilaterals with a single integration point and "hourglass control" to control spurious mechanisms caused by the fully reduced integration. This element is, initially, chosen because it is relatively inexpensive for problems involving non-linear constitutive behavior, since the material calculations are only done at one point in each element. The finally used element is element type CAX8R. This is an eight noded biquadratic interpolation, reduced integration element. The eight noded elements produce a higher order approximation of the deformation field and due to the four integration points the stress system is determined with a higher accuracy. The die is modelled with the \*RIGID SURFACE option. There are interface elements of type IRS21A (with element CAX4R) or IRS22A (with element CAX8R) attached to the top and outer surface of the top half of the billet because it is anticipated that part of the outer surface will contact the die. Different element distributions have been used and their influence on the results will be discussed.

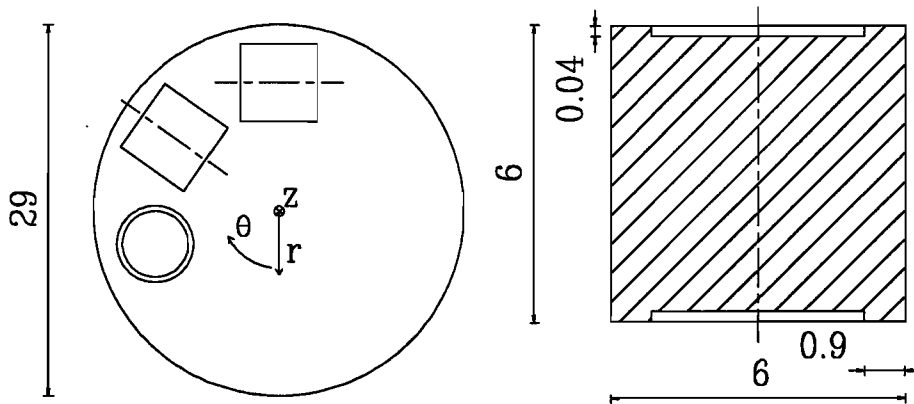
## 3.1 Material behaviour: strain hardening and anisotropy

For the determination of stress-strain curves, tensile and compression specimens with their axis in compression direction were used. Tensile specimens 6 mm in diameter and 60 mm gauge length were tested. Compression specimens with 8 mm diameter and 9 mm height were tested on a laboratory press up to an effective strain of 0.35. No significant differences could be observed between the tensile and compression stress-strain curves. Hollomon strain hardening  $\bar{\sigma} = C \cdot \bar{\epsilon}^n$  (C is the specific stress, and n the strain hardening exponent) was used to fit the results (table 1).

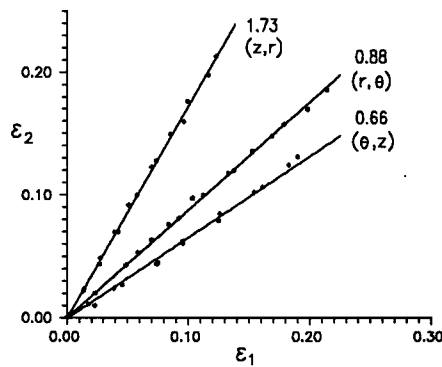
	C [MPa]	n [-]	$R_{p0,2}$ [MPa]*	$R_m$ [MPa]*	$\bar{\epsilon}_F$ [-]*
* tensile test	374	0.157	160	265	0.94

**Table 1:** Tensile- and compression-test results (annealed: 350 °C 45' furnace-cooled)

The plastic anisotropy of the rod was determined by using small-sized compression-specimens, electrical discharge machined from the rod in three orthogonal directions (fig. 1). The specimens are of the Rastegaev-type. These specimens have shallow cavities on the circular faces which contain a solid lubricant (paraffin wax). When properly dimensioned, these specimens do not barrel during testing. Transverse strains are measured at discrete intervals up to an effective strain of ca. 0.35.



**Fig. 1:** Anisotropy-specimens: position in rod cross-section and geometry



**Fig. 2:** Measured transverse strains; directions of  $\epsilon_1$  and  $\epsilon_2$  indicated in parentheses; specimen axis in third direction

Measured transverse strains are plotted in fig. 2., together with a best fit line. The tests are made in duplicate. As can be seen (fig. 2) the anisotropy values are high and fairly

constant over the strain-range measured. The product of the three fitted anisotropy values equals unity within experimental error. This indicates that the Hill yield-criterion for anisotropic materials is an acceptable approximation of the real yield-surface.

The stresses on the equatorial free surface of the upset-specimens were calculated out of measured surface strains using Hill's yield-criterion. For principal directions it states:

$$H (\sigma_{\theta} - \sigma_z)^2 + G (\sigma_z - \sigma_r)^2 + F (\sigma_r - \sigma_{\theta})^2 = 1 \quad (1)$$

in which F, G and H are determined by the anisotropy-values and the effective stress. The stresses in Eq. (1) are principal stresses in usual cylindrical coordinates.

The material behaviour is defined by the cards \*ELASTIC, \*PLASTIC and \*POTENTIAL defining the anisotropic elastic-plastic material behavior. The values of the constants are defined by measurements and ABAQUS interpolates linearly between the values given on the PLASTIC card in Appendix 1.

### 3.2 Boundary conditions, loading and solution control

Kinematic boundary conditions are symmetry on the z-axis (all nodes on  $r=0$  have  $u_r=0$  prescribed), symmetry about  $z=0$  (all nodes at  $z=0$  in node set MIDDLE have  $u_z=0$  prescribed), and the  $u_z$  displacement of the rigid body reference node for the die is prescribed as having a constant velocity in the axial direction so that the total displacement of the die is 9 mm over the history of the upsetting.

The parameter PTOL on the \*STATIC card defines the maximum allowable residual in the equilibrium equations. PTOL has dimensions of force, and so must be small compared to typical nodal forces. Based on a typical yield stress value (300 MPA for an effective strain of about 0.2) and area associated with a node (30 mm<sup>2</sup>) we expect nodal forces to be about  $4.5 \cdot 10^3$ . PTOL is chosen to be 100N- about 2% of the typical nodal force. This is rather a coarse choice for the tolerance: it is made because we are seeking a solution of reasonable accuracy and minimal computational costs. The computational costs are expressed by the total amount of CPU time needed to solve the problem. Additional control parameters have been defined on the \*STEP card (see the input data-deck in appendix 1). The INC parameter specifies the maximum number of increments allowed in the step, and is set to 800. The CYCLE parameter specifies the maximum number of iterations allowed

in an increment. Its default value is 6, and that has here been increased to 10 because our experience has generally been that it is effective to allow more iterations per increment in a problem as non-linear as this case.

## 4 RESULTS AND DISCUSSION

In order to reduce the computational costs of these highly non-linear calculations a number of finite element calculations has been carried out to find the optimal mesh and minimal CPU time given a certain needed stress accuracy. In general the accuracy of the stresses is one order lower as the displacement accuracy. For determination of failure and/or local surface instability the stresses are very important and must be determined very accurate. With the initial input data-deck for an upsetting problem, it took about 80-100 hours of CPU time to obtain reasonable results. For the mesh optimization a PASCAL program has been written which generates an ABAQUS input data-deck for a certain element distribution. The input files are created on a personal computer and by means of a TELNET connection transported to the DEC-5000 station on which the calculations are done.

The conclusions from calculations with different meshes are that both the accuracy of the solution and the amount of CPU time needed are not influenced very much by different element distributions. Of course is the total needed CPU time directly influenced by the number of elements, but the number of elements used is very near the minimum. There was, however, an extremely small increment observed during the total step of these calculations. This small increment caused the very high amounts of CPU time. In order to reduce the computational costs this increment has to be increased during the calculations. This was achieved by adding the MONOTONIC control parameter to the \*STEP definition. This parameter is only usefull for nonlinear cases where it indicates that the proces is essentially monotonic, so that ABAQUS should extrapolate the solution at the start of each increment to begin the non-linear equation solution for the next increment. The parameter is especially beneficial in many large strain forming process simulations with elastic-plastic material models, and is generally recommended for such cases. In the PROBLEM EXAMPLES Manual this card has not been added to the step definition and as a result of this the extremely small increments were obtained, as the default is no extrapolation except in procedures in which the response is usually expected to be monotonic, such as the \*VISCO or the \*SOILS procedure.

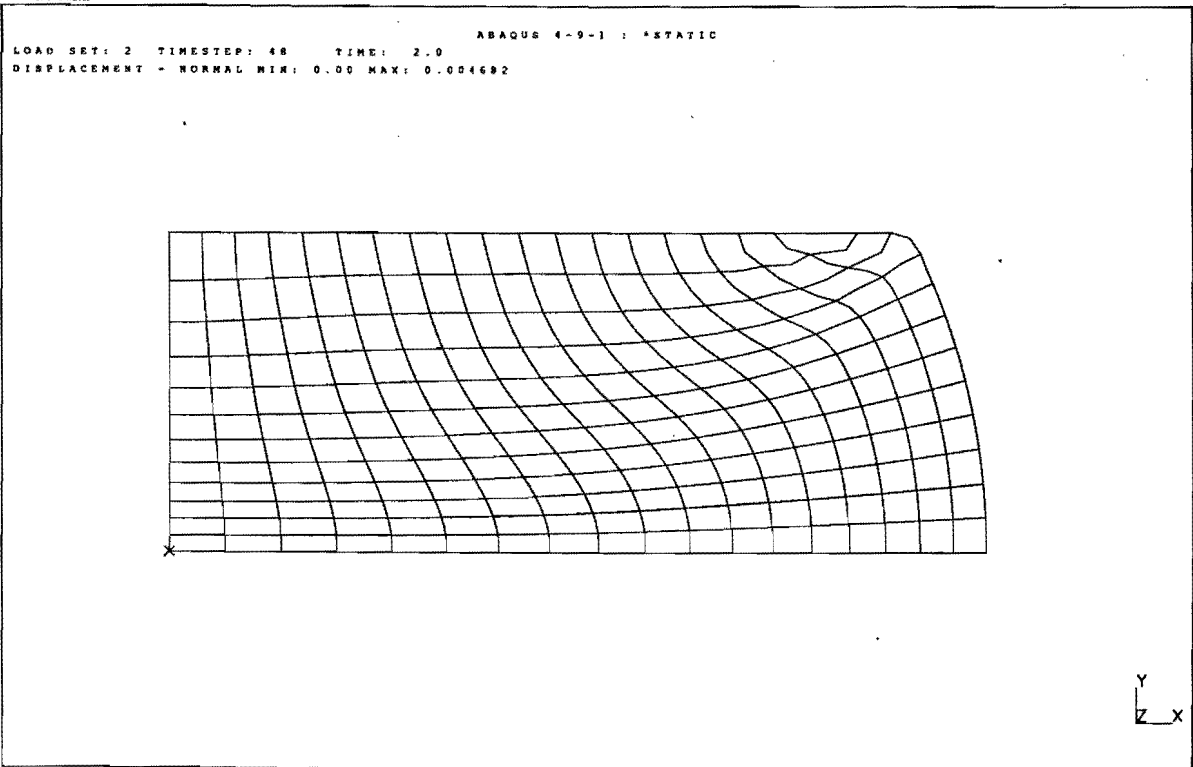
The MONOTONIC control parameter is especially usefull when the problem is fully plastic. This is why the upsetting problem is split into two steps. The first step, being in the elastic-plastic deformation area, without the MONOTONIC parameter activated and generates a fully plastic problem. This initial step is, because of the elastic-plastic, material behavior

carried out with very small increments (despite a higher PTOL value), but as it is only a very small part of the history the total amount of CPU time is not influenced very much anymore by it. The second step, being in the fully plastic regime and the MONOTONIC parameter activated, introducing a very economic incrementation for a large part of the upsetting history. The total amount of CPU time needed was reduced from approximately 100 to 1 hours allowing more and higher order elements in the model to come to a final CPU time of around 2 hours. These times were acceptable to find the frictional values that related the experiments with the finite element calculations. Also the non-linear geometry, NLGEOM, parameter on the \*STEP definition is activated because it is a large deformation problem.

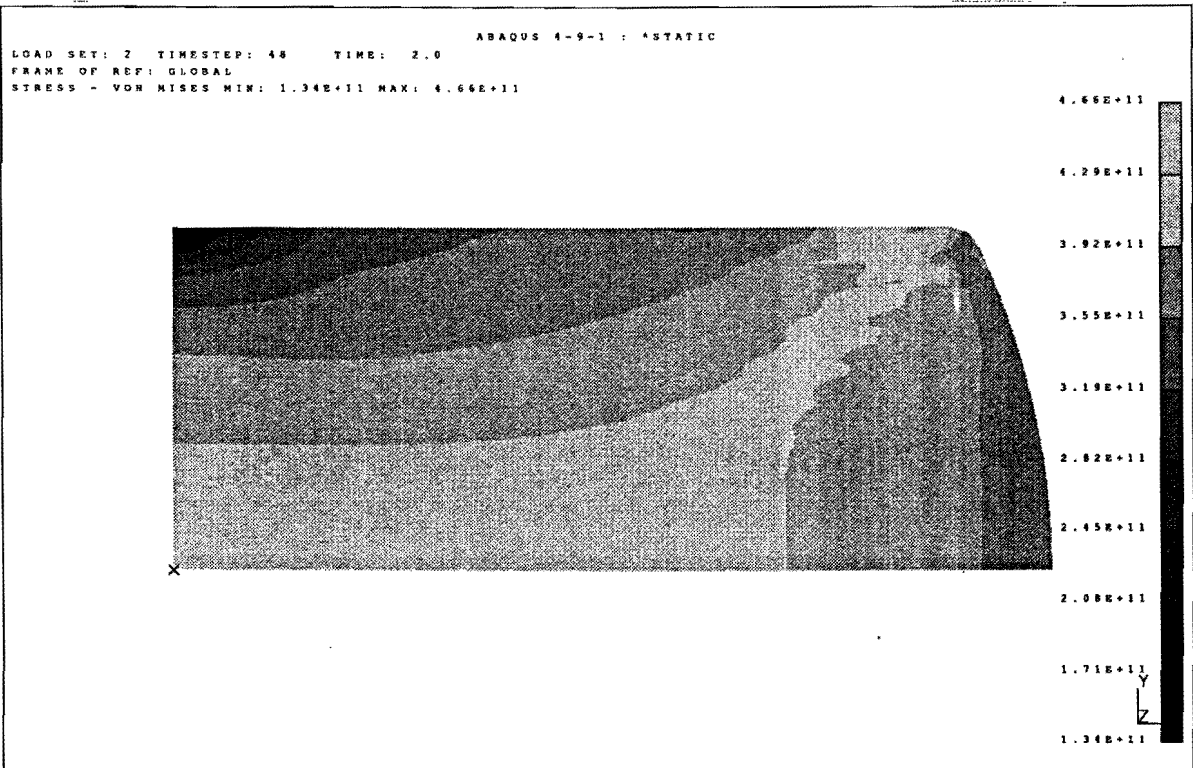
The conclusion is that the equation solver of ABAQUS uses the elastic strain of the preceding increment to find a solution for the next increment, when the MONOTONIC parameter is not activated. In case of fully plastic deformation this method is very ineffective, resulting in extreme small increments and high values of CPU time. With the MONOTONIC parameter the incremental strain direction follows the expected strain-path resulting in large increments and usable values for the needed CPU time.

Using the MONOTONIC parameter in the first step leads to excessively small increments, because some elements pass into the plastic regime, which causes a sudden change of the strain-path. This is the main reason why the total step had to be split up into two different steps.

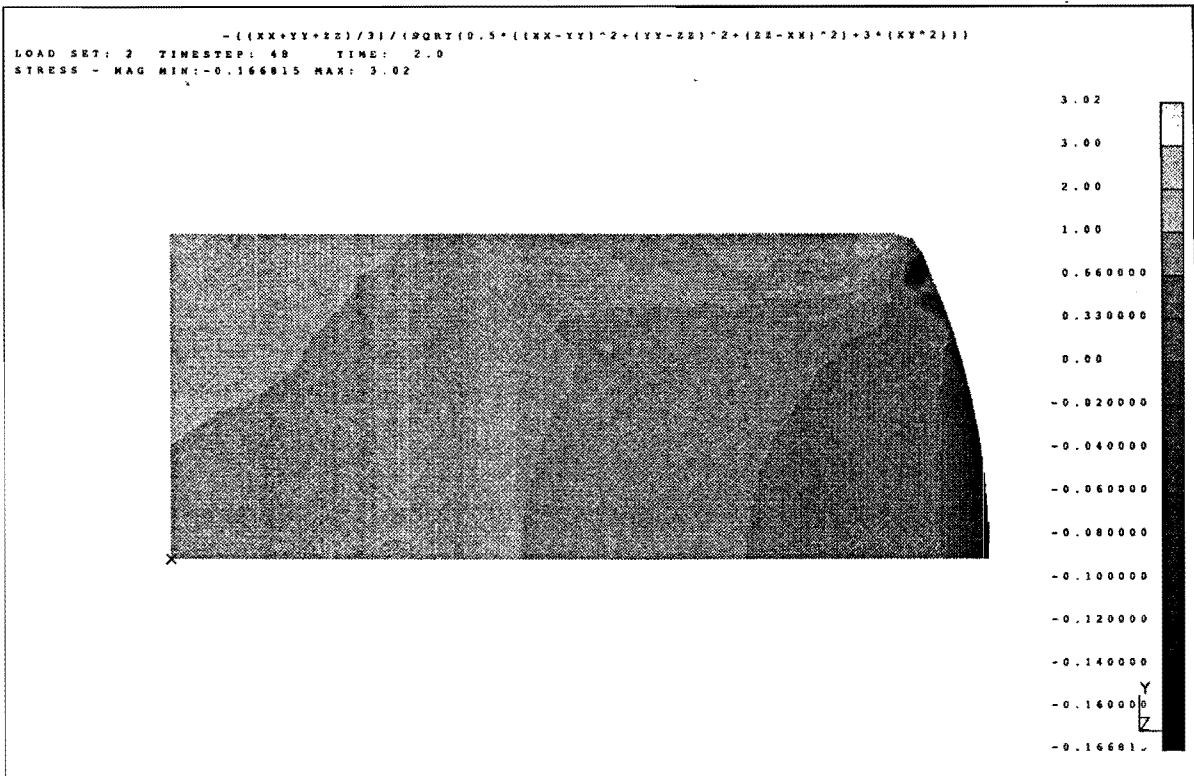
In figure 3 the displaced geometry is given, in figure 4 the Von Mises equivalent stress and in figure 5 the hydrostatic pressure are presented.



**Fig.3:** The deformed billet geometry.

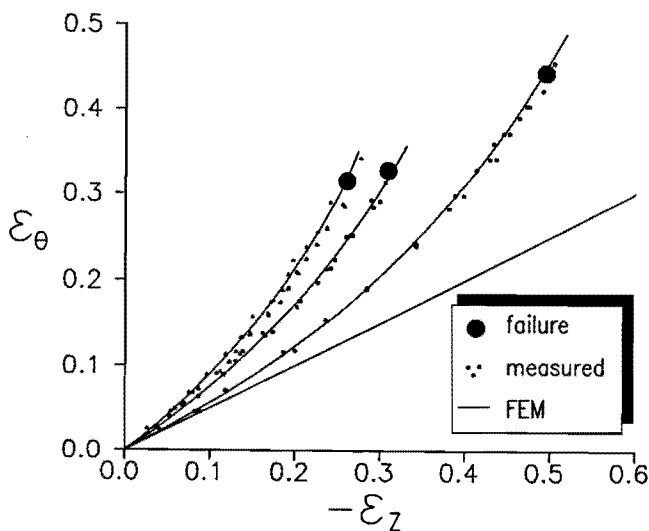


**Fig.4:** The equivalent Von Mises stress in the deformed billet.



**Fig.5:** The hydrostatic pressure in the deformed billet (negative values are positive and vice versa, this strange definition was needed to improve the contrast of this picture).

In figure 6 the results of the finite element calculations and the experiments are compared and it can be concluded that the finite element calculations can be used very well for a curve fit of the strain-path of an upsetting experiment with the in this report discussed modifications of the input data-deck of the PROBLEM EXAMPLES manual.



**Fig.6:** Comparison of calculated and measured strain path of a cold upsetting experiment.



# REFERENCES

ABAQUS, USERS Manual, Hibbitt, Karlsson and Sorensen, Inc., version 4.7, 1988

ABAQUS, THEORY Manual, Hibbitt, Karlsson and Sorensen, Inc., version 4.7, 1988

ABAQUS, EXAMPLE PROBLEMS Manual, Hibbitt, Karlsson and Sorensen, Inc., version 4.7, 1988

Lippmann, H., Metal Forming Plasticity, Springer-Verlag, Berlin, 1979

Taylor, L.M., A Finite Element Analysis for Large Deformation Metal Forming Problems Involving Contact and Friction, PhD Thesis, University of Texas at Austin, 1981.

# APPENDIX 1 THE FINAL ABAQUS FILE

\*HEADING

AXISYMMETRIC UPSETTING PROBLEM.

FOR F.H.A.M. HABRAKEN T.U.E.

Date:

Time:\*\* rond 20 \* 20.(met twee,aal zoveel elementen in de y richting.

\*\* Deze invoerfile rekt de vervormingen van een

\*\* axi-symmetrische stuikproef uit voor een lichaam

\*\* met een straal van 10 mm en een lengte van 20 mm

\*\* De mesh is axi symmetrisch en beschrijft slechts de

\*\* helft van het lichaam omdat het middenvlak toch een

\*\* symmetrie-vlak is.

\*PRE PRINT, ECHO=NO, MODEL=NO, HISTORY=NO

\*RESTART,WRITE,FREQ=400

\*NODE,NSET=RSNODE

9999,0.,.03

\*\*\*\*\* HIERTUSSEN INVOEGEN\*\*\*\*\*

\*NODE

1,0.00000000,0.00000000

2,0.00000000,0.000417000

3,0.00000000,0.000833000

3623,0.01000000,0.009167000

3624,0.01000000,0.009583000

3625,0.01000000,0.010000000

\*ELEMENT,TYPE=CAX8R,ELSET=METAL

1,1,201,203,3,101,202,103,2

2,3,203,205,5,103,204,105,4

3,5,205,207,7,105,206,107,6

214,3419,3619,3621,3421,3519,3620,3521,3420

215,3421,3621,3623,3423,3521,3622,3523,3422

216,3423,3623,3625,3425,3523,3624,3525,3424

\*ELEMENT,TYPE=IRS22A,ELSET=CONTACT

217,225,125,25,9999

218,425,325,225,9999

219,625,525,425,9999

220,825,725,625,9999

221,1025,925,825,9999

222,1225,1125,1025,9999

223,1425,1325,1225,9999

224,1625,1525,1425,9999

225,1825,1725,1625,9999

226,2025,1925,1825,9999

227,2225,2125,2025,9999

228,2425,2325,2225,9999

229,2625,2525,2425,9999

230,2825,2725,2625,9999

231,3025,2925,2825,9999

232,3225,3125,3025,9999

233,3425,3325,3225,9999

234,3625,3525,3425,9999  
 235,3623,3624,3625,9999  
 236,3621,3622,3623,9999  
 237,3619,3620,3621,9999  
 238,3617,3618,3619,9999  
 239,3615,3616,3617,9999  
 240,3613,3614,3615,9999  
 241,3611,3612,3613,9999  
 242,3609,3610,3611,9999  
 243,3607,3608,3609,9999  
 244,3605,3606,3607,9999  
 245,3603,3604,3605,9999  
 246,3601,3602,3603,9999  
 \*ELSET,ELSET=MID  
 205  
 \*NSET,NSET=MIDN  
 3601,3602,3603  
 \*BOUNDARY  
 1,1  
 2,1  
 3,1  
 3401,2  
 3501,2  
 3601,2  
 \*\*\*\*\*HIERTUSSEN INVOEGEN\*\*\*\*\*  
 \*RIGID SURFACE,ELSET=CONTACT,TYPE=SEGMENTS  
 START,.030,.01  
 LINE,-.001,.01  
 \*INTERFACE,ELSET=CONTACT  
 \*FRICTION  
 0.23,1.E15,1.20E8  
 \*SOLID SECTION,ELSET=METAL,MATERIAL=EL  
 \*MATERIAL,NAME=EL  
 \*ELASTIC  
 7.06E10,.345  
 \*PLASTIC  
 1.20E8,0.00000  
 1.8167E8,0.01  
 2.0250E8,0.02  
 2.1578E8,0.03  
 2.2572E8,0.04  
 2.3375E8,0.05  
 2.4052E8,0.06  
 2.4639E8,0.07  
 2.5160E8,0.08  
 2.5628E8,0.09  
 2.6055E8,0.10  
 2.6809E8,0.12  
 2.7465E8,0.14  
 2.8045E8,0.16  
 2.8567E8,0.18

2.9042E8,0.20  
2.9479E8,0.22  
2.9883E8,0.24  
3.0260E8,0.26  
3.0614E8,0.28  
3.0946E8,0.30  
3.1260E8,0.32  
3.1559E8,0.34  
3.1842E8,0.36  
3.2113E8,0.38  
3.2372E8,0.40  
3.2620E8,0.42  
3.2859E8,0.44  
3.3088E8,0.46  
3.3310E8,0.48  
3.3523E8,0.50  
3.3730E8,0.52  
3.3930E8,0.54  
3.4124E8,0.56  
3.4495E8,0.60  
3.4929E8,0.65  
3.5337E8,0.70  
3.5721E8,0.75  
3.6084E8,0.80  
3.6428E8,0.85  
3.6756E8,0.90  
3.7068E8,0.95  
3.7367E8,1.00  
3.7928E8,1.10  
3.8449E8,1.20  
3.8934E8,1.30  
3.9388E8,1.40  
3.9816E8,1.50  
4.0221E8,1.60  
4.0605E8,1.70  
4.0970E8,1.80  
4.1318E8,1.90  
4.1651E8,2.00  
4.3902E8,2.799  
\*POTENTIAL  
0.9483,1.09744,0.97238,1.04,0.92,1.04  
\*PLOT  
\*DRAW  
\*DRAW,NODENUM  
\*DRAW,ELNUM  
\*STEP,INC=800,CYCLE=45,SUBMAX,AMP=RAMP,NLGEOM  
\*STATIC,PTOL=1000  
0.0015,1.,1.E-8,0.5  
\*BOUNDARY  
9999,1  
9999,6

```
9999,2,,-.00030
*EL PRINT,ELSET=MID,SUMMARY=NO,FREQ=400
S
*EL PRINT,ELSET=MID,SUMMARY=NO,FREQ=400
E
*EL PRINT,ELSET=MID,SUMMARY=NO,FREQ=400
PE
*NODE PRINT,NSET=MIDN,SUMMARY=NO,FREQ=400
COORD
*NODE FILE,NSET=RSNODE
RF
*END STEP
*STEP,INC=800,CYCLE=15,SUBMAX,AMP=RAMP,NLGEOM,MONOTONIC
*STATIC,PTOL=100
0.15,1.,1.E-8,0.5
*BOUNDARY
9999,1
9999,6
9999,2,,-.00060
*EL PRINT,ELSET=MID,SUMMARY=NO,FREQ=400
S
*EL PRINT,ELSET=MID,SUMMARY=NO,FREQ=400
E
*EL PRINT,ELSET=MID,SUMMARY=NO,FREQ=400
PE
*NODE PRINT,NSET=MIDN,SUMMARY=NO,FREQ=400
COORD
*NODE FILE,NSET=RSNODE
RF
*END STEP
```