A finite element analysis of backward can extrusion

Eindhoven University of Technology
Faculty of Mechanical Engineering
Production Technology and Automation
Laboratory for Forming Technology

A Finite Element Analysis of Backward Can Extrusion

M.Sc. J. H. Yang

Nov. 1991 WPA 1191

Report on the Course:
"Metal Forming Processes"

A FINITE ELEMENT ANALYSIS OF BACKWARD CAN EXTRUSION

J. H. Yang

Summary

This report deals with a finite element simulation of backward can extrusion in ABAQUS. Based on the introduction of input file, the problem encountered in the course of calculation and analysis of the results, the main advantages and disadvantages are analyzed when this kind of problem is solved in ABAQUS. Especially, it is of important practical significance when the prediction of extrusion force is carried out that the difference between the finite element solution and the experimental solution is explained briefly.

1. Introduction

Since the flow of metal in metal forming operations will have a major influence on the quality and distribution of properties in a finished component, the methods of predicting metal flow would be of value in reducing the experimental and development work and costs involved in producing new components. The finite element method has proved to be a highly valuable tool of analysis in studying metal deformation processes. The problem in simulating backward can extrusion with a finite element program is the deformation of the mesh during the process.

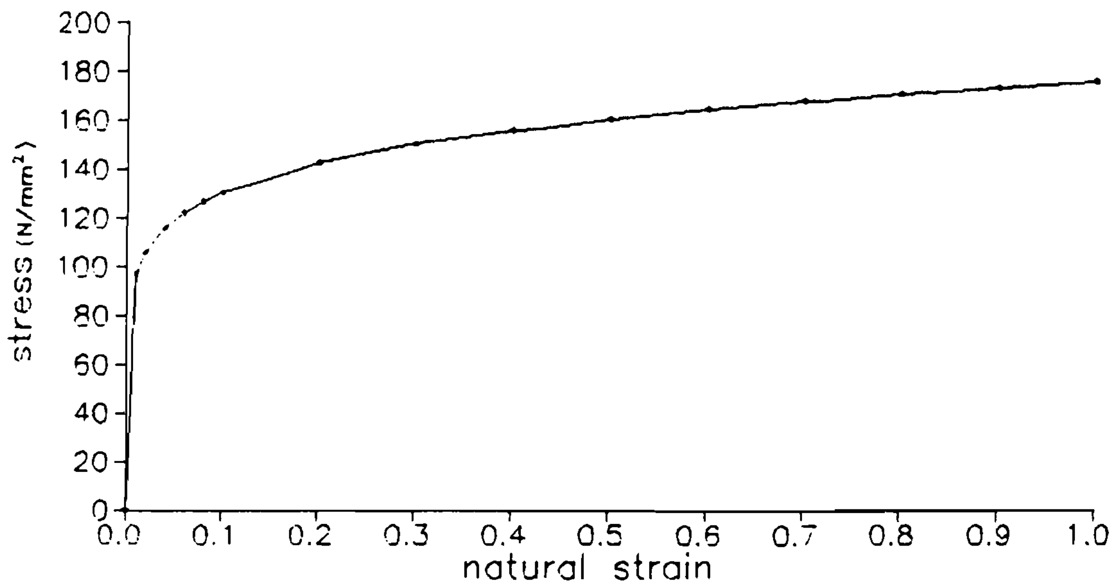
Some authors have used finite elements to simulate the deformation of backward can extrusion in some finite element software such as P. Hartley [1] [2]. he used the program MAFORGE 111 developed by P. Hartley in 1977 and also the program EPFEPM [3] in which the eight node element is used for rezoning the mesh on a microcomputer system. C. J. M. Gelten [4] used the program MARC in which the mesh can be rezoned automatically, etc.. But up to now, the paper which describes the finite element simulation of backward can extrusion in ABAQUS has not been found. In this report, all procedures for simulating backward can extrusion in ABAQUS are introduced and some defects in comparison with other programs are discussed and the results are analysed briefly.

2. Finite-element model

In the finite-element model, the billet has a radius of 6 (mm) and an initial height of 4 (mm); the ram has a radius of 5 (mm). Symmetry of the die about the centre line of the billet means that the solution needs only be carried out over half of the billet. This symmetry determines the boundary conditions on the centre line, where shear traction and normal displacement are both zero. In the initial mesh used for this model (Fig. 2(a)), 600 four-node quadrilateral bilinear elements with 651 nodal points and 2400 integration points are used.

About the material properties some parameters were obtained by means of the tensile test of recycled aluminium [7]. For example, in the constitutive equation $\bar{\sigma}=c\bar{\varepsilon}^n$, where $\bar{\sigma}$ is equivalent stress and $\bar{\varepsilon}$ is equivalent strain, the specific stress c=176 (N/mm^2) and the strain hardening exponent n=0.13. The other parameters are Young's modulus E=7E+04 (N/mm^2) and the Poisson's ratio ν=0.345. The stress-strain curve according to the constitutive equation is shown in Fig. 1.

Fig. 1. The stress-strain curve for recycled aluminium at the temperature of 234 degrees Celsius



The values chosen for the interface friction coefficient μ between the ram, the die and the billet is 0.1. The model is classical Coulomb friction. It is difficult to assess the magnitude of the friction in a certain situation. The friction coefficient, used in the finite-element calculation, is only an estimation of the real friction. The finite-element analysis was carried out until the height of the billet below the ram was reduced by 75%.

3. Description of the input file of the FEM calculation

In this section, a brief discussion of the input file of a FEM calculation will be given. Because to solve this kind of problem the finite-element mesh has to be rezoned several times, for every rezoning, an input file has to be written. There is only a little difference between these input files. Therefore, in this section only one input file is discussed.

In general, an input file can be divided in two parts. In the first part the geometry, the material behavior and the initial boundary conditions are defined and in the second part a certain history of the load or displacement is given.

3.1. Definition of the geometry

```
*HEADING
BACKWARD EXTRUSION PROBLEM
```

The heading block is used to print a title for the analysis.

```
*RESTART,WRITE,FREQ=20
```

This option will cause ABAQUS to create a restart file during this analysis job, after each increment at which the increment number is exactly divisible by 20, and at the end of each step of the analysis, regardless of the value 20 at that time. This file must be saved upon completion of the job for the continuous analysis and output of results.

```
*NODE
```

```
1,0,0
30,5.8E-03,0
501,0,1.05E-03
530,5.8E-03,1.05E-03
531,6E-03,1.05E-03
131,6E-03,2E-04
31,5.985E-03,1.5E-05
8888,6E-03,0
9999,0,7.05E-03
*NODE,NSET=TOP1
4826,5.0000E-03,9.8392E-03
4827,5.2007E-03,9.8502E-03
4828,5.3997E-03,9.8682E-03
4829,5.5993E-03,9.8784E-03
4830,5.7997E-03,9.8854E-03
4831,6.0000E-03,9.8868E-03
*NGEN,NSET=APPEN
131,531,100
*NGEN,NSET=BOT
1,30
*NGEN,NSET=TOP
501,530
*NFILL
BOT,TOP,5,100
*NSET,NSET=MIDDLE,GEN
526,531,1
*NFILL
MIDDLE,TOP1,43,100
*NSET,NSET=SIDE1,GEN
526,4826,100
*NSET,NSET=SIDE2,GEN
531,4831,100
*NSET,NSET=ALL
9999,APPEN,TOP1,SIDE1,SIDE2
*NSET,NSET=AXIS,GEN
1,501,100
```

This part of the input file is used to create all nodes by means of the commmands NODE, NGEN (node generation) and NFILL (node fill) and to create all node sets by the command NSET (node set) for outputing results and exerting constraints on some nodes.

```
*ELEMENT,TYPE=CAX4,ELSET=METAL
1,1,2,102,101
626,526,527,627,626
*ELGEN,ELSET=METAL
1,30,1,1,5,100,100
626,5,1,1,43,100,100
*ELSET.ELSET=LITTL,GEN
1,401,100
*ELEMENT,TYPE=IRS21A
```

```
7701,4831,4731,8888
6601,31,30,8888
9901,501,502,9999
9926,526,626,9999
*ELGEN,ELSET=SECOND
7701,48,-100,1
6601,30,-1,1
*ELGEN,ELSET=PUNCH
9901,25,1,1
9926,43,100,1
```

In this part of the input file, the elements between the nodes and the interface elements between the ram, die and the billet are defined by using the commands ELEMENT and ELGEN (element generation). The following types of elements are used:

CAX4: This a four node axisymmetric element with four integration points for output.

IRS21A: This is a three node contact element and is used together with a rigid surface. It determines whether there is contact between the billet and the rigid surface or not. When there is contact, the contact pressure and a possible friction force are calculated. The element consists of 3 nodes. Two of them are the nodes on a free edge of a four node element and the third node is a reference node on the rigid surface.

```
*RIGID SURFACE,ELSET=SECOND,TYPE=SEGMENTS,SMOOTH=0.00005
START,-0.001,0
LINE,0.006,0
LINE,0.006,0
*INTERFACE,ELSET=SECOND
*FRICTION
0.1,1E+11
*RIGID SURFACE,ELSET=PUNCH,TYPE=SEGMENTS,SMOOTH=0.00005
START,0.005,1.005E-02
LINE,0.005,1.05E-03
LINE,-0.001,1.05E-03
*INTERFACE,ELSET=PUNCH
*FRICTION
0.1,1E+11
```

Two rigid surfaces are defined and they are coupled to the groups of contact elements called SECOND and PUNCH respectively. With the use of the INTERFACE card, two friction coefficients are defined between the billet and the corresponding rigid surface. The friction coefficient has to be given with a "stiffness in stick". The "stiffness in stick" is an elastic stiffness which will transmit the shear force across the element as long as these forces are below the friction limit. This stiffness simulates the fact that there should be no relative motion between the surface until slip occurs. Its value can be defined by dividing the expected friction limit force by an acceptable relative displacement before slip occurs.

```
*SOLID SECTION,ELSET=METAL,MATERIAL=EL
*MATERIAL,NAME=EL
*ELASTIC
70E+09,0.345
*PLASTIC
7.1957E+07,0
1.3064E+08,0.099
1.4287E+08,0.199
1.5057E+08,0.299
1.5629E+08,0.399
1.6088E+08,0.499
1.6473E+08,0.599
1.6806E+08,0.699
1.7010E+08,0.799
1.7363E+08,0.899
```

This option "SOLID SECTION" is used to define properties of solid elements. The option must be used to assign a material definition to a set of solid elements. And "MATERIAL" sets the value of this parameter to the name of the material to be used with these elements. The material behavior is defined as a linear elastic behavior in combination with a nonlinear plastic behavior. But the stress-strain curve is defined in piecewise linear segments in the "PLASTIC" option up to a total strain level of 89.9%. The yield stress remains constant for plastic strains exceeding the last value given.

```
*BOUNDARY
AXIS,1
```

The initial boundary conditions are defined. The card "AXIS,1" indicates that all nodes in node set "AXIS" are fixed in x-direction.

```
*PLOT
*DRAW
*DRAW,NODENUM
*DRAW,ELNUM
```

This option "PLOT" is used for sizing and placement of plots onto display device, setting the character for text appearing on the plots, providing a title to appear on each frame in the series, etc. The following three options indicate that a mesh, a mesh with node numbers and a mesh with element numbers will be displayed respectively.

```
*INITIAL CONDITIONS,TYPE=OLD MESH,STEP=2,INC=8
```

This option "INITIAL CONDITION" is used to prescribe initial conditions for analysis and the "TYPE=OLD MESH" option provides the rezoning capability and uses the restart file from the old mesh to obtain the solution data at the time of rezoning.

3.2. Definition of the history of the load

```
*STEP,INC=200,CYCLE=10,SUBMAX,AMP=RAMP,NLGEOM
*STATIC,PTOL=50
*BOUNDARY
```

```
9999,1,2
9999,6
8888,1,2
8888,6
*EL PRINT,ELSET=LITTL
MISES
*NODE PRINT,NSET=ALL
RF
*END STEP
```

When initial stresses are given from the interpolation of an old mesh onto a new mesh-rezoning, the initial stress state may not be an equilibrium state for the finite element model. The options from "STEP" to "END STEP" are used to check equilibrium and iterate under the condition of zero displacement of the rigid surface on which the reference point with the node number 9999 is fixed. The other details will be explained with next step.

```
*STEP,INC=200,CYCLE=10,SUBMAX,AMP=RAMP,NLGEOM
*STATIC,PTOL=50
0.1,1
*BOUNDARY
9999,1
9999,6
9999,2,,-0.00005
8888,1,2
8888,6
*EL PRINT,ELSET=METAL,FREQUENCY=20
S,E
MISES,PEEQ
PRIN
*NODE PRINT,NSET=ALL,FREQUENCY=10
COORD,RF
*EL FILE,FREQUENCY=20
S,E
*NODE FILE,FREQUENCY=20
RF,U
*PLOT,FREQUENCY=10
*DISPLACED
U,1
*PLOT,FREQUENCY=10
*DETAIL,ELSET=METAL
*CONTOUR
PEEQ
MISES
*END STEP
```

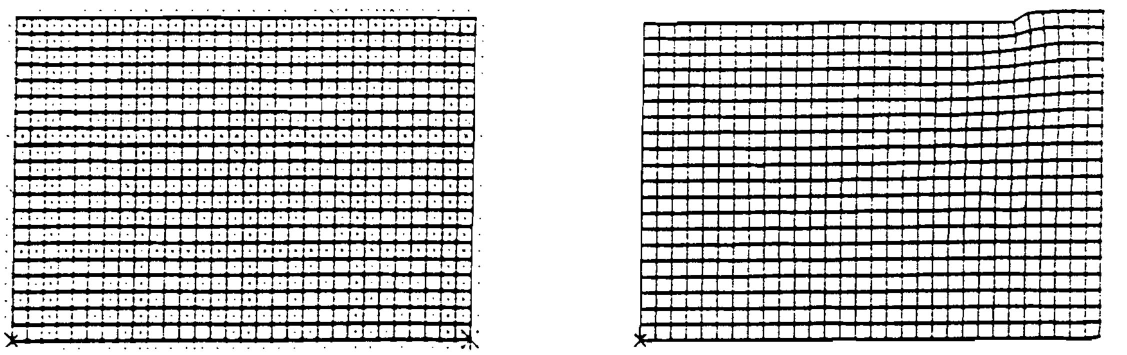
The "STEP" option is used to begin a step. The following parameters are that the "INC" is used to specify the maximum number of increments in a step and the "CYCLE" the maximum number of iterations in a step, the "SUBMAX" is used to suppress subdivision except when convergence is not achieved in the maximum number of iterations allowed, the "NLGEOM" indicates geometric nonlinearity should be accounted for during the step and the "AMP=RAMP" indicates the displacement of the ram varies linearly over the step with the time variation.

The "STATIC" option indicates the step should be analyzed as a static step. The external parameters loading the structure are defined using the loading option and/or the prescribed boundary condition options such as 9999,2,-0.00005 means the ram will move down 0.05 (mm). The "PTOL=50" is the basic tolerance measure for solution of the equilibrium equations at each increment.

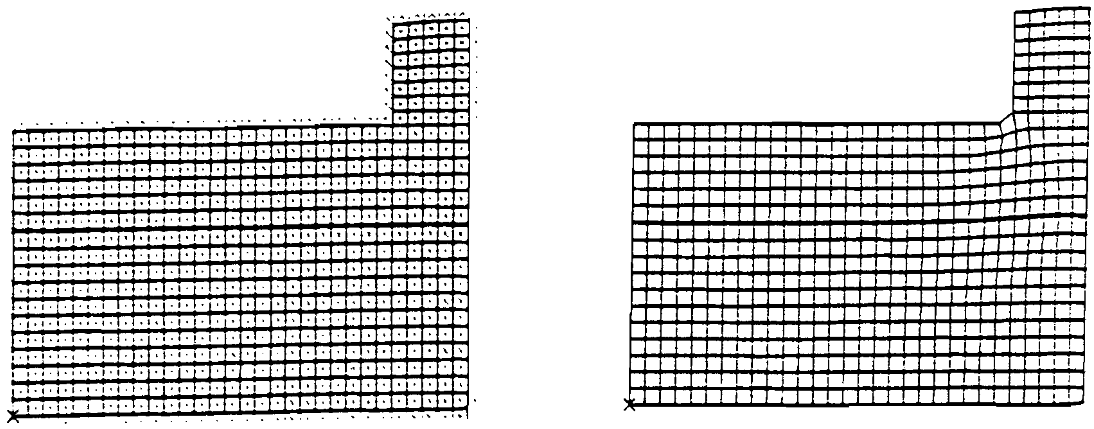
The options "NODE PRINT" and "EL PRINT" provide printed output of tables of nodal variables (displacement, reaction force, etc.) and element variables (stress, strain, etc.) in data file. The options "NODE FILE" and "EL FILE" are used to write the nodal variables and the element variables to the result file.

The "DISPLACED" option is used to draw the displaced shape of model. The "DETAIL" option is used to plot a part of the model. The "CONTOUR" option allows contour plots to be generated during an analysis.

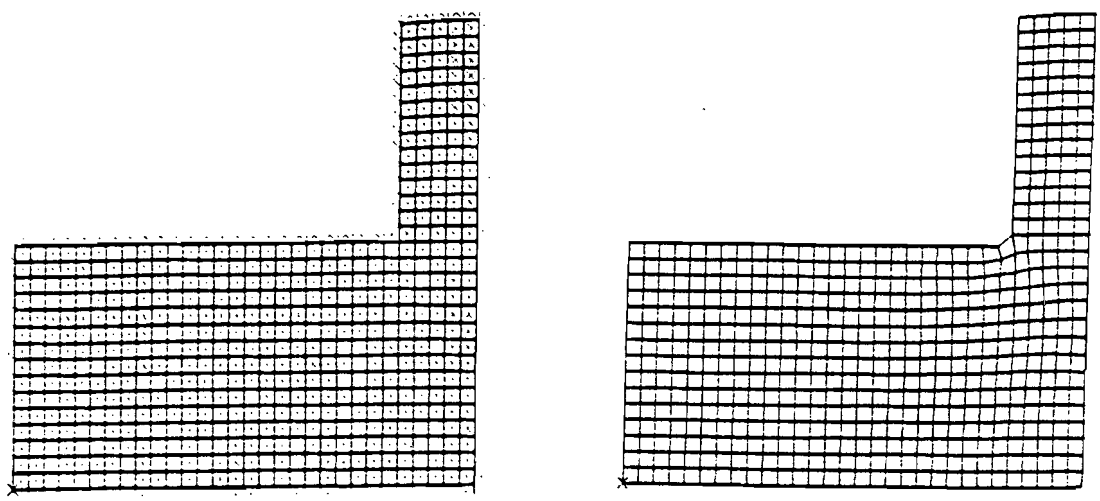
Fig. 2. Rezoned mesh and deformed mesh (The deformation is defined as the percentage of displacement at any instantaneousness and total displacement of the ram)



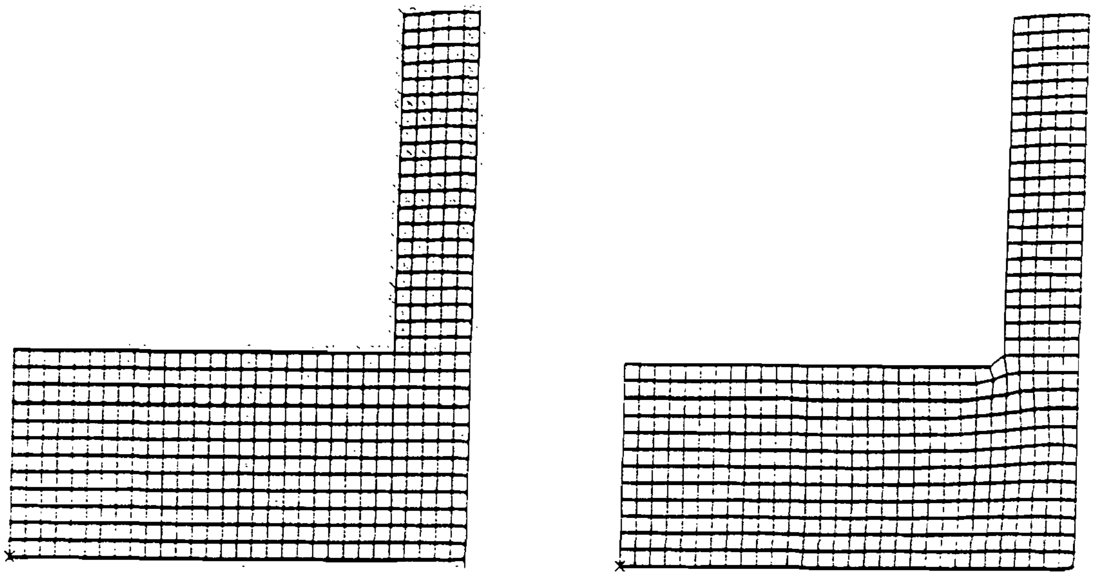
(a) Initial mesh and the first deformed mesh



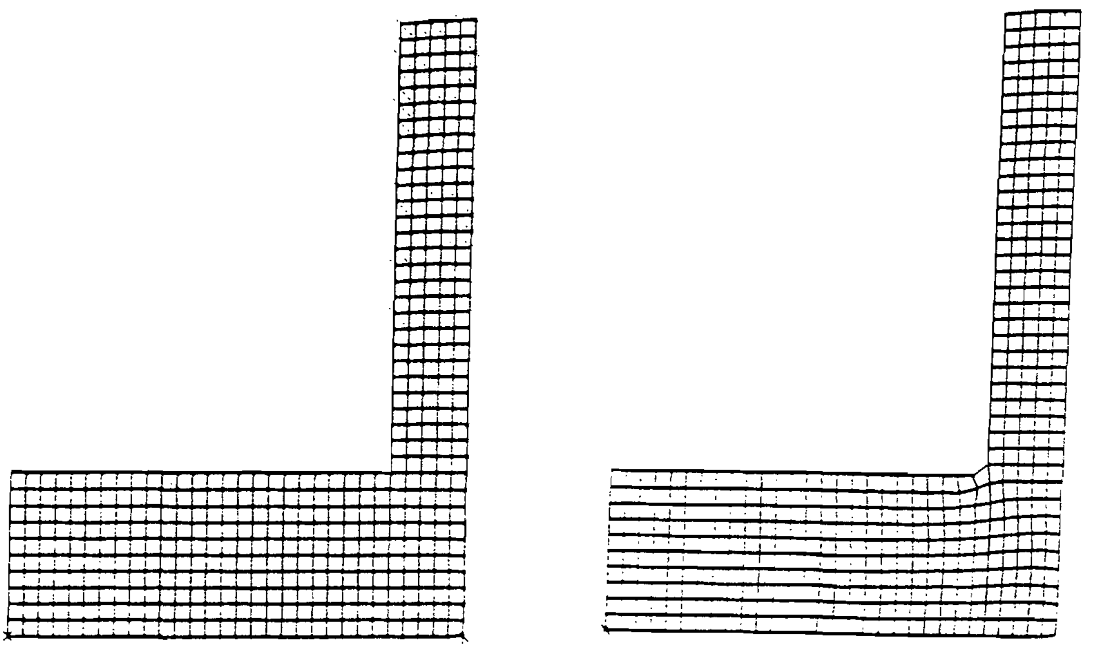
(b) Rezoned mesh and 12.5% deformed mesh



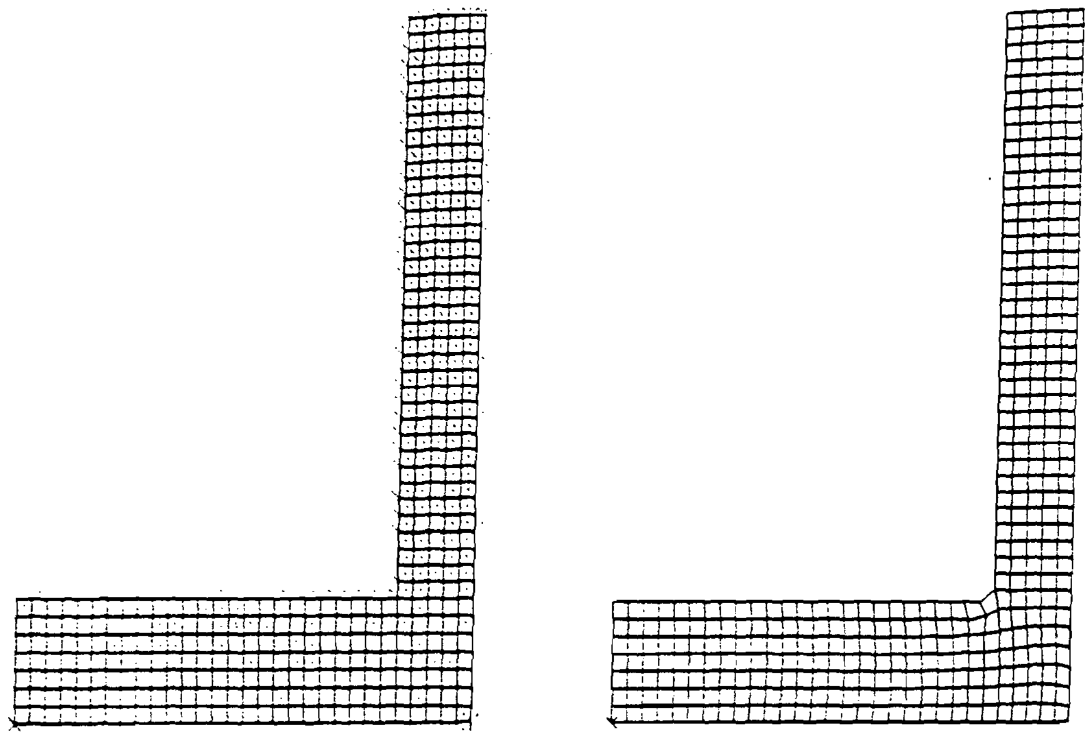
(c) Rezoned mesh and 25% deformed mesh



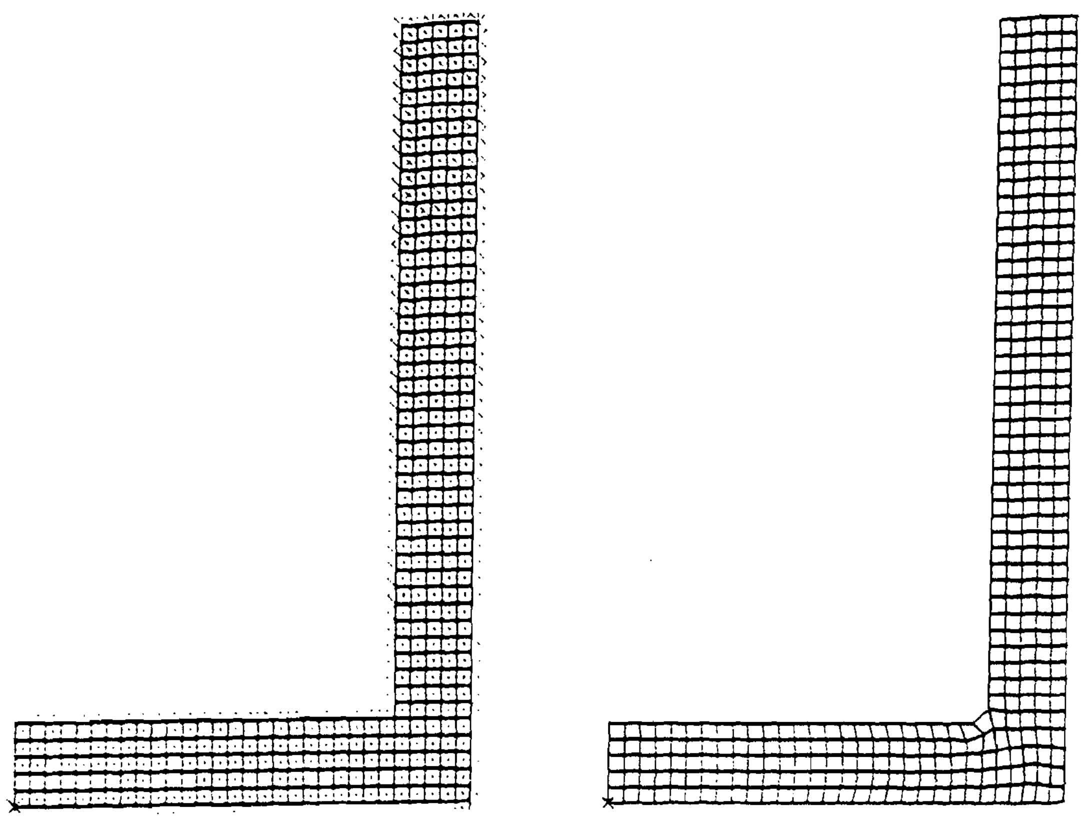
(d) Rezoned mesh and 37.5% deformed mesh



(e) Rezoned mesh and 50% deformed mesh



(f) Rezoned mesh and 62.5% deformed mesh



(g) Rezoned mesh and 75% deformed mesh

4. Results and discussions

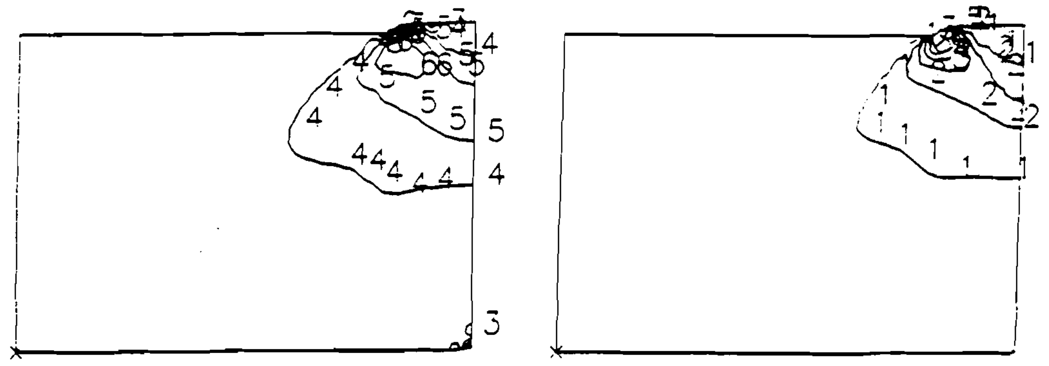
4.1 The deformed mesh

Fig. 2 shows the rezoned mesh and the deformed mesh at each 12.5% deformation up to 75% besides the initial mesh and the first deformed mesh. The mesh was rezoned totally 59 times for protecting the elements from excessive distortion even "inside-out" which makes further analysis impossible because the backward can extrusion problem is a large deformation problem. In order to continue a large deformation analysis with sufficient accuracy it will be necessary to regenerate a new mesh. But ABAQUS has no function by means of which the mesh can be rezoned automatically.Therefore, a mesh has to be rezoned by rewriting the input file. This takes a lot of time. Another defect which has been found is that the element on the sharp corner will cut across the sharp corner of the ram. Therefore, after rezoning the mesh, some material will be lost, because there is just a node of the rezoned mesh on the sharp corner. If the eight-node elements are used to generate the mesh, the defect can be overcome and the accuracy is better than that of four-node elements. But the eight-node element is not supported for rezoning by ABAQUS.

4.2 Equivalent stress and strain

Fig 3 shows the contour plots of equivalent stress and strain at each 12.5% deformation upto 75% besides the first deformed

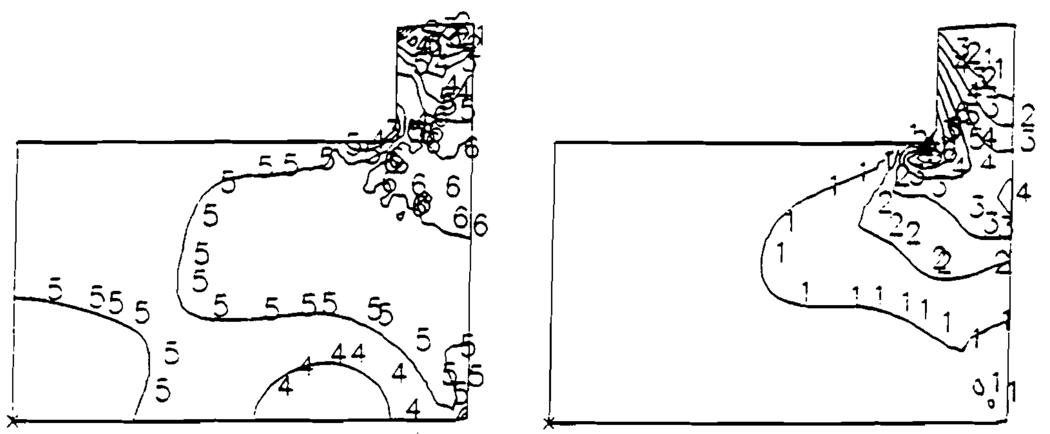
Fig. 3. Contour plots of equivalent stress (N/m^2) (left) and equivalent strain (right)



Equivalent stress
1=4.58E+07
2=5.89E+07
3=7.20E+07
4=8.50E+07
5=9.81E+07
6=1.11E+08

Equivalent strain
1=0.028513
2=0.053896
3=0.079459
4=0.104932
5=0.130406
6=0.155879

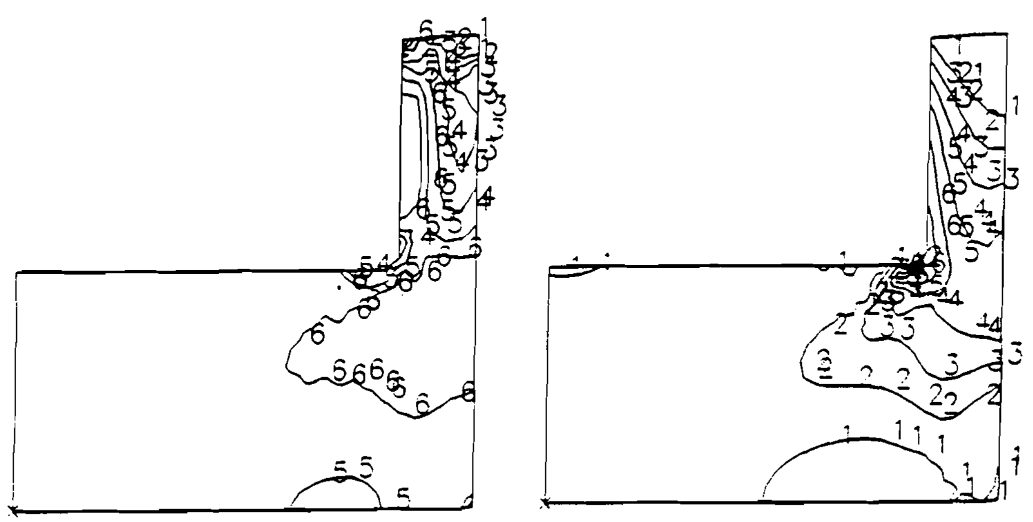
(a) Initial deformation



Equivalent stress
1=5.11E+07
2=7.18E+07
3=9.25E+07
4=1.13E+08
5=1.34E+08
6=1.55E+08

Equivalent strain
1=0.158464
2=0.314381
3=0.470299
4=0.626217
5=0.782134
6=0.938052

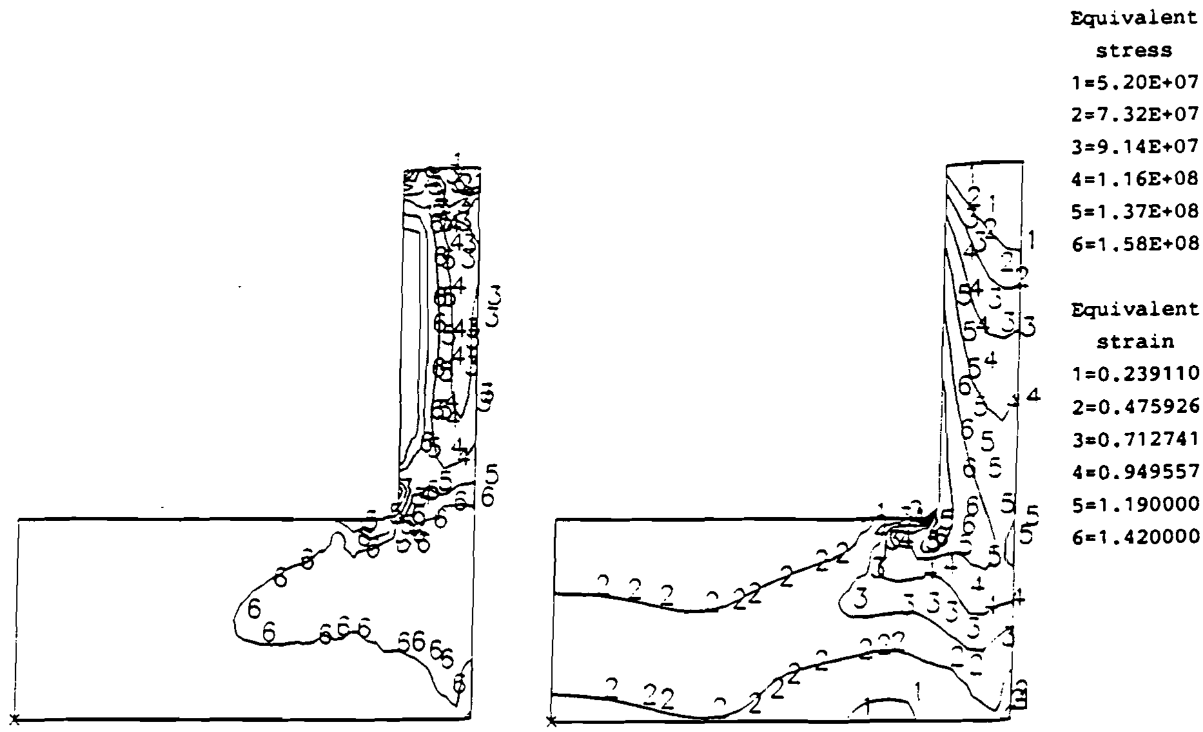
(b) 12.5% deformation



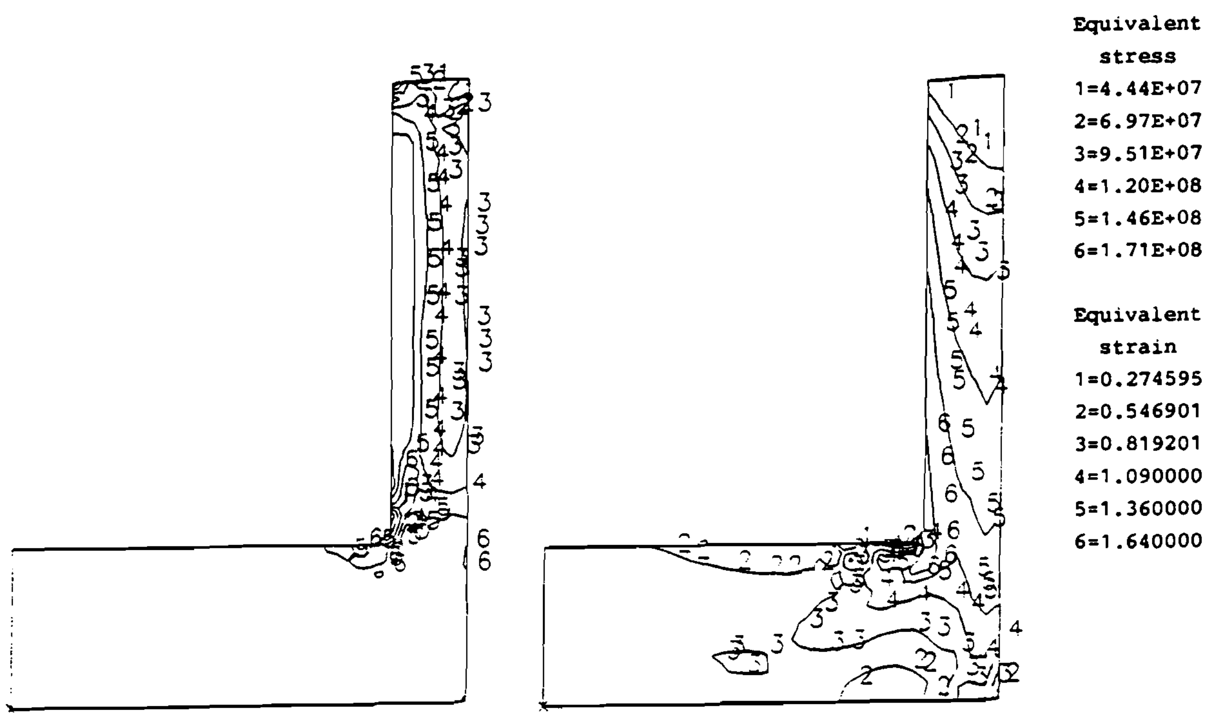
Equivalent stress
1=4.92E+07
2=7.03E+07
3=9.14E+07
4=1.12E+08
5=1.34E+08
6=1.55E+08

Equivalent strain
1=0.214530
2=0.426723
3=0.638917
4=0.851110
5=1.060000
6=1.280000

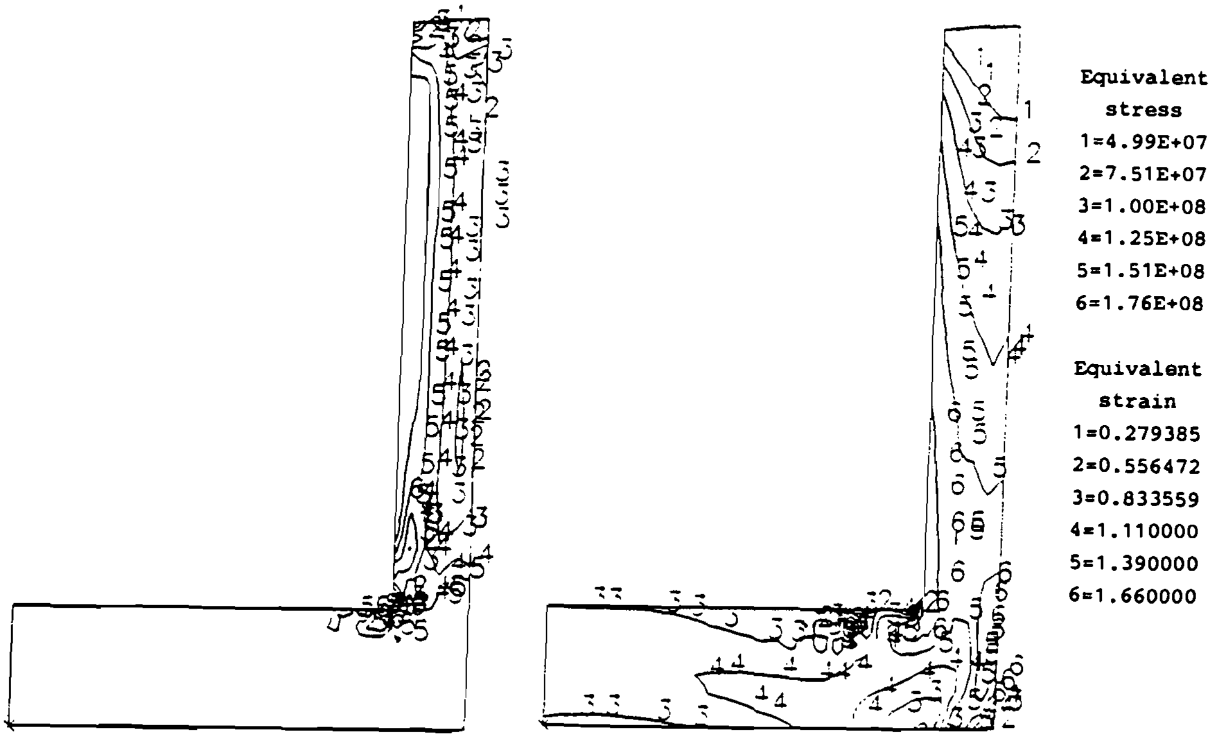
(c) 25% deformation



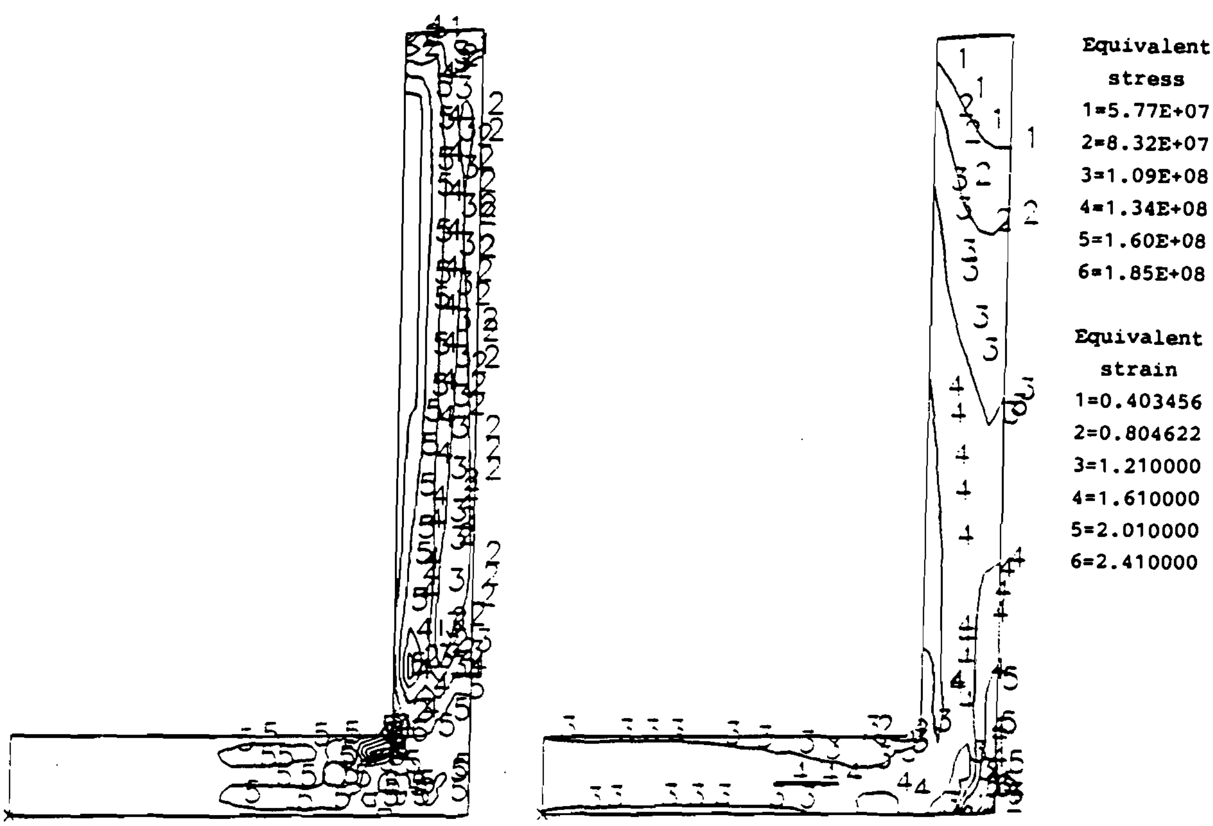
(d) 37.5% deformation



(e) 50% deformation



(f) 62.5% deformation



(g) 75% deformation

mesh.

In the contour plots of equivalent stress, some facts can be paid attention to. The highest stress gradient is concentrated around the sharp corner of the ram. As the ram is moved down, there exist always is the stress in the moving up elements. The reason is because after rezoning the mesh each time, the stress in the old mesh will be interpolated to the new mesh. The contour plots of equivalent stress have shown this sense clearly. It is can also be observed that the highest equivalent stress value on some contour plots is higher than yield stress given in the input file. This is unnormal. The reason is because the contour plots were obtained in IDEAS, a kind of software which can generate finite elmenet mesh and carry out finite element post processing, and then the stress error emerged when the data were processed in IDEAS.

From the contour plots of equivalent strain it can also be observed that the highest strain gradient and the highest equivalent strain are also concentrated around the sharp corner, but with the movement of the ram they move above the lower right hand corner gradually.

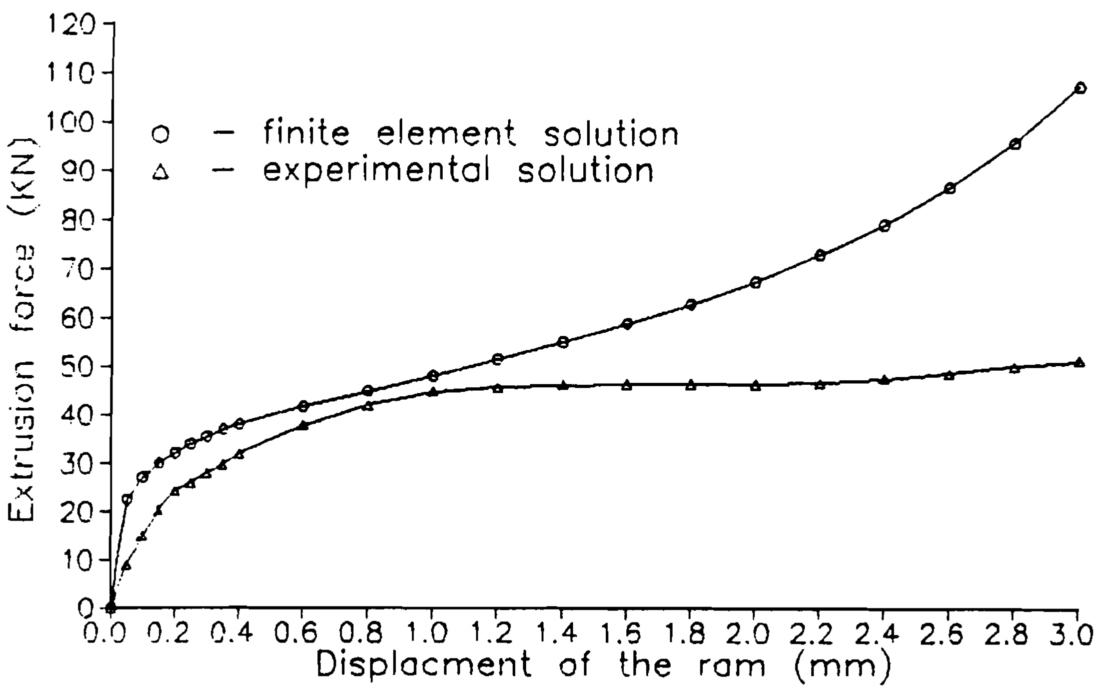
4.3. Extrusion force

Fig. 4 shows the finite element solution and experimental solution of the extrusion force. From these curves it can be seen that good agreement is obtained between the finite element solution and the experimental solution when the deformation is less than or equal to 30%. With the increase of the deformation, the difference between the finite element solution and the experimental solution increases gradually. The reason is (1) because when the mesh was rezoned each time, the stress in old mesh was always interpolated to the new mesh, the upper stress of extruded can could not be released (2) because the elements moving up always touch the rigid surface and there exist was the normal stress in the radial direction of the ram in these elements, there exist was friction force between the rigid surface and elements. As a matter of fact the friction force does not exist after the extruded material moves up to a certain height. The extra friction force was exerted to the extruded material as a force constraint. Therefore, the extrusion force increase rapidly with the increase of the friction force or the rise of the material.

The selection of friction coefficients has an important influence on the extrusion force and the convergence of solution. If a higher value of the friction coefficient is selected, the extrusion force will be much higher than the experimental one. If the friction coeffecient is very small, the excessive distortion of some local elements will occur and result in poor convergence.

After the mesh is rezoned each time, normally, the discontinuity of the extrusion force will occur (not shown in Fig. 4). Because in a step the ram was only moved down 0.05 (mm) and then the mesh was rezoned again, the discontinuity is very small. The deviation is within 5%. If the displacement of the ram increases in a step, the force discontinuity will also increase. Therefore, in each step the selected displacement of ram should not be too large to maintain the necessary accuracy.

Fig. 4. Displacement-extrusion force curve



5. Conclusions

According to above-mentioned results and analysis as well as difficulties encountered during the calculation, the following conclusions are obtained:

(1) The finite element analysis of backward can extrusion can be carried out completely in ABAQUS.

(2) Two defects of ABAQUS have been encountered in solving this

kind of problem. One is that the mesh can't be rezoned automatically and the mesh can only be rezoned by rewriting the input file. It takes a lot of time. Another is that the eight-node element is not supported for rezoning by ABAQUS. If four-node element was used after rezoning the mesh some material will be lost because the element on the sharp corner of ram will cut across the sharp corner of ram.

(3) Friction coeffecient must properly be selected. Otherwise, poor convergence will occur.

(4) Friction force between the ram, die and the billet has a significant influence on the extrusion force. How can the influence of extra friction force be eliminated? Further research work needs to be done.

Acknowledgement

The author wishes to thank Mr. W. H. Sillekens for his help during the project and writing the report.

References

[1]. P. Hartley, C. Z. N. Sturgess and G. W. Rowe, "A finite element analysis of extrusion-forging", Proc. North American Metalworking Research Conf., 1978, pp. 212-219.

[2]. P. Hartley, C. Z. N. Sturgess and G. W. Rowe, "Prediction of Deformation and Homogeneity in Rim-disk Forging", Jour. of Mechanical Technology, 4, 1980, pp. 145-154.

[3]. A. A. M. Hussin and P. Hartley, "Elastic Plastic Finite element Modelling of a Cold Extrusion Process Using a Microcomputer Based System", 16, 1988, pp. 7-19

[4]. C. J. M. Gelten and A. W. A. Konter, "Application of Mesh-rezoning in the Updated Langrangian Method to Metal Forming Analyses", Numerical Method of Industrial Forming Processes, Proc. of the International Conference, 1982, pp. 511-521.

[5] J. S. Park and S. M. Hwang, "Automatic Remeshing in Finite Element simulation of Metal Forming Processes by Guide Grid Method", Journal of Material Processing Technology, 27, 1991, pp. 73-89.

[6] Hibbitt, Karlsson and Sorensen, Inc., "ABAQUS User's Mamual" and "ABAQUS Example Problems", Version 4.7, 1988.

[7] W. H. Sillekens, J. H. Dautzenberg and J. A. G. Kals, "Formability of Recycled Aluminium-Advantages of a Rapid Solidification Process", Annals of the CIRP Vol. 39/1/1990, pp. 287-290.