# DEVELOPMENT AND VALIDATION OF A SIMULATION METHOD FOR DYNAMIC TENSILE AND FRACTURE TESTS USING EXPLICIT AND IMPLICIT FINITE ELEMENT CODES

K. Kussmaul, A. Klenk, M. Schüle, H. Klamser

JOURNAL DE PHYSIQUE IV C3-821
Colloque C3, suppl. au Journal de Physique III, Vol. 1, octobre 1991

# DEVELOPMENT AND VALIDATION OF A SIMULATION METHOD FOR DYNAMIC TENSILE AND FRACTURE TESTS USING EXPLICIT AND IMPLICIT FINITE ELEMENT CODES

K. KUSSMAUL, A. KLENK, M. SCHÜLE and H. KLAMSER[1]

*Staatliche Materialprüfungsanstalt (MPA), Universität Stuttgart, Pfaffenwaldring 32, D-7000 Stuttgart 80, Germany*

**Résumé** - Ce papier décrit une méthode pour simuler des essais dynamiques de traction et de rupture lors d'une vitesse de traction moyenne. Avec la méthode des éléments finis on a simulé l'événement d'impact entier en effectuant dans une 33 MJ machine rotative aux essais au choc des essais de traction avec des éprouvettes rondes et des essais de rupture avec des éprouvettes plats de traction qui étaient entaillées des deux côtés. On a fait une comparaison des résultats experimentals. On a atteint des informations complémentaires sur les conditions de charge et trouvé une distribution de contrainte homogène sur le long de la jauge de contraintes fixée à l'éprouvette de traction. On a également obtenu des déplacements aux interfaces appropriées des éprouvettes plats de traction entaillés des deux côtés. Ces déplacements ont été utilisés comme des conditions aux limites pour une analyse avec la méthode des éléments finis, à qui appartient un réseau fine pour calculer des paramètres de rupture mécanique et des contraintes et déplacements au bout de l'entaille.

**Abstract** - A method for the simulation of dynamic tensile and fracture tests at intermediate strain rates is described. The whole impact event which takes place during dynamic tensile tests with smooth round bar specimens and fracture tests with double edge notched tensile (DENT) specimens at a 33 MJ-rotating disk impact machine is simulated by means of an explicit finite element code. A comparison with experimental results was made. Additional information about loading conditions could be obtained and a homogeneous stress distribution along the gauge length of the tensile specimen was found. Displacements at suitable interfaces of the DENT-specimens were obtained and used as boundary conditions for a finite element analysis with a fine mesh with an implicit time integration scheme to calculate fracture mechanics parameters as well as stresses, strains and displacements in the vicinity of the notch tip.

## 1 Introduction

In many technical fields such as traffic, production and energy technique, loading rates may occur which call for reliable values to characterize the material behaviour under similar loading conditions. The rotating disk impact machine, Figure 1, is an instrument to investigate characteristic values at strain rates up to $\dot{\varepsilon} = 10^3$ 1/s and fracture mechanics parameters at loading rates up to $\dot{K} = 10^6$ MPa$\sqrt{m}$/s /1/. The parameters investigated in these tests (e.g. yield and ultimate tensile strength, stress-strain curves, dynamic fracture toughness $K_{Id}$

[1] *Present address : Dr.-Ing.h.c. F. Porsche AG, Porschestr., D-7251 Weissach, Germany*

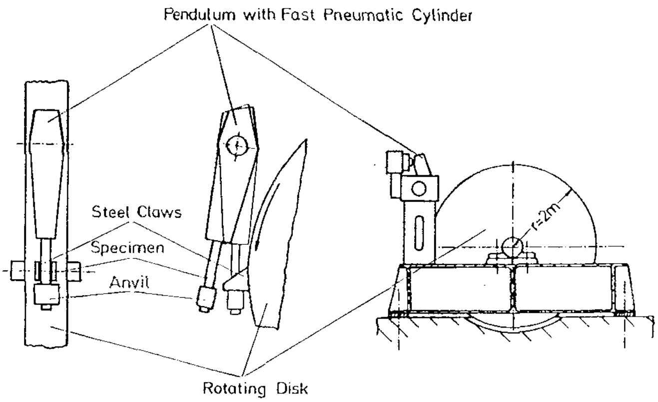


**Fig. 1 :** **33 MJ rotating disk impact machine of MPA Stuttgart**

and critical values of the J-Integral $J_{Id}$) are only reliable if there is an accurate knowledge of the testing conditions. This knowledge is also necessary to adjust specified strain rates or loading rates.

For finite element calculations of structures containing cracks or notches a fine mesh around the notch is necessary. In order to reduce the costs of such a calulation a detail model was used for the calculation of stresses, strains and displacements in the notch tip field und for determination of J-integral values.

The developed simulation method is described schematically in Figure 2. First, the whole impact event which takes place during a dynamic tensile test with a smooth round bar specimen in the rotating disk impact machine is simulated. The finite element model is validated by means of comparison with experimental results. Additional information about the loading conditions can be obtained. Second, the smooth round bar specimen in the model is replaced by a double edge notched tensile (DENT) specimen and the impact event simulation is subjected to the same validation procedure. Third, a detail model of the middle part of the DENT-specimen is used to obtain detailled information about the notch tip region. Displacements at suitable interfaces of the specimen are taken from the second part of the procedure and are used as boundary conditions.

**3d Finite element simulation of tensile tests**
- Explicit time integration
- DYNA3D

Validation

**Experimental Results of tensile tests:**
- strain
- displacement and velocity of the anvil

**3d Finite element simulation of fracture tests**
- Explicit time integration
- DYNA3D

Validation

**Experimental Results of fracture tests:**
- strain
- displacement and velocity of the anvil

Displacements at interfaces

**2d Finite element calculation with a fine-meshed detail model**
- Implicit time integration
- ABAQUS

**Numerical Results:**
- notch tip displacement field
- J-Integral

Fig. 2: Simulation method for tensile and fracture tests.

## 2 Experimental set-up and finite element models

The rotating disk impact machine is equipped with fixed steel claws. When the specified testing conditions are established, the specimen and the anvil are slewed into the path of the claws by a fast acting pneumatic cylinder. The impact of the steel claws is damped by aluminium cylinders or plates. This damping elements have a strong influence on the time when the anvil reaches the circumferential velocity of the rotating disk. The dampers are fixed on positioning screws which avoid an asymmetric impact and bending of the specimen.

The tests were performed with smooth round bar specimens, Figure 3, and DENT specimens, Figure 4, with a cross section of 14 x 7 $mm^2$ and 14 x 14 $mm^2$ made of the fine-grained structural steel 20 MnMoNi 5 5 (yield strength = 520 $N/mm^2$, ultimate tensile strength = 706 $N/mm^2$, elongation at fracture $A_5$ = 23 % at ambient temperature). The DENT specimens were notched to a/W = 0.5 using the electric discharge method. Several strain gauges were attached to dynamometer sections which are parts of the specimens with elastic behaviour to determine stresses and forces via quasistatic calibration. Post yield strain gauges were attached to the gauge length of the smooth round bar specimens and to the ligament of the DENT-specimens. An opto-electronic sensor /1/ illuminates and measures the reflected light of a bar code applied at one side of the anvil. The displacement and velocity of the anvil can be deduced from the signal of the sensor. The impact velocities of the tests varied between 1 m/s and 30 m/s.

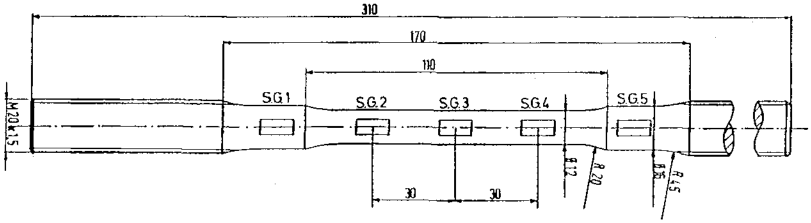


Fig. 3: Smooth round bar specimen.

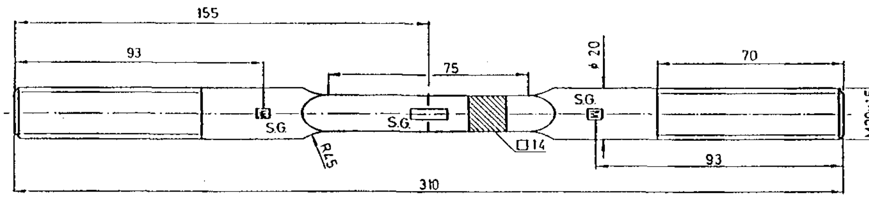


Fig. 4: Double edge notched tensile specimen.

The finite element model for the simulation of the impact event, Figure 5, consists of all the parts which are relevant for the transmission of force onto the specimen. Hexaeder elements were much more efficient and were used for the whole model which consists of about 4200 elements and about 6500 nodes. Two different geometries of damping elements and the above mentioned types of specimens were modeled. With exception of the steel claw only a quarter of the model is meshed which is due to the symmetry. The material behaviour of the anvil, the steel claws and the pendulum is assumed to be elastic. For the specimen and the damping element an elastic-plastic material model was used. The strain rate dependence , Figure 6, given in a point by point representation was considered for the specimen only, because of the low sensitivity of aluminium on strain rate effects. The finite element code DYNA3D /2/ on a CRAY 2 computer was used to perform the calculations for impact velocities in the range of 1 m/s to 30 m/s. In order to simulate a dynamic event with a large model, the use of first order isoparametric elements with a vectorised explicit time integration scheme which is realized in DYNA3D /3/ is advantageous. An hourglass control according to /4/

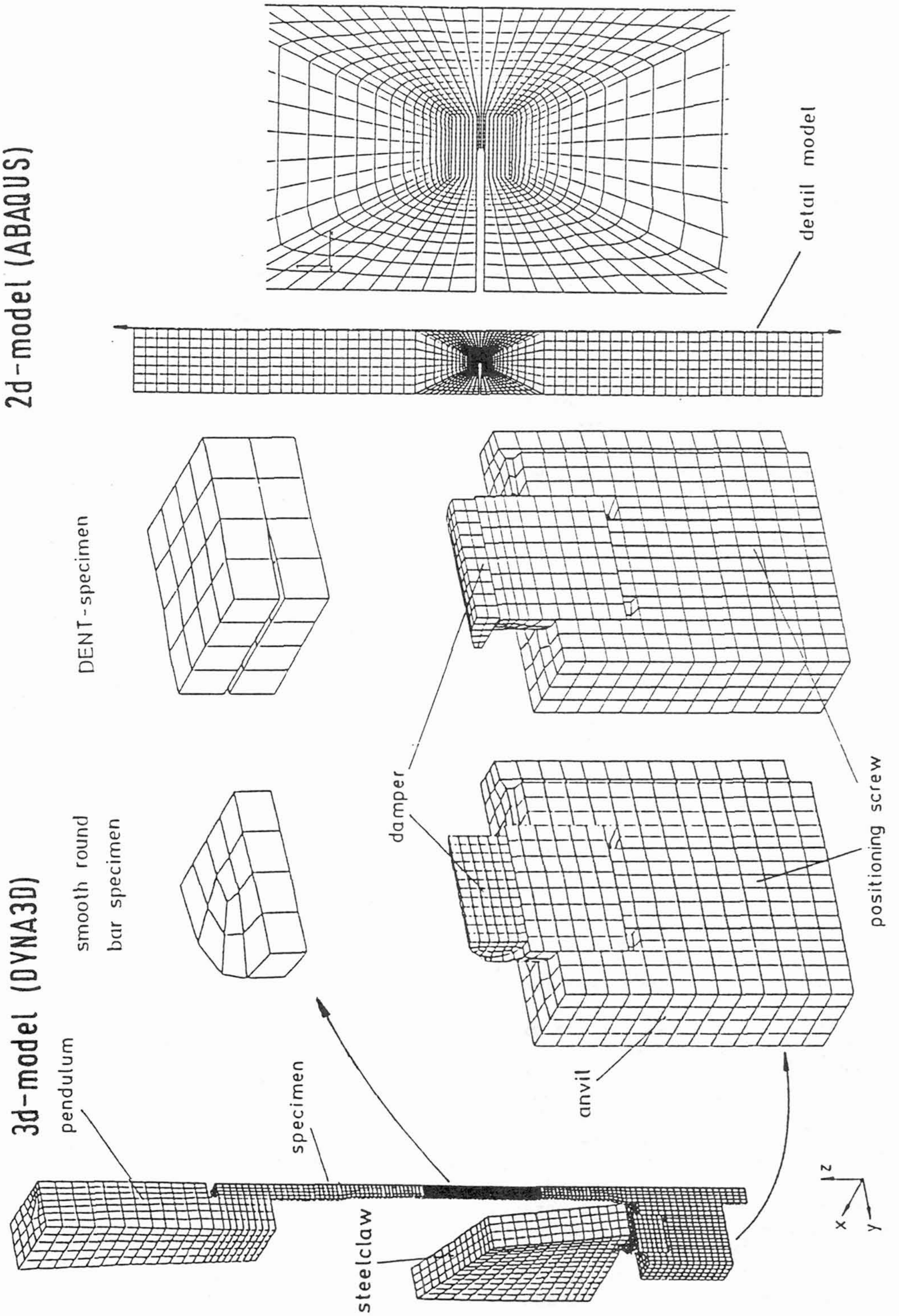


Fig 5: Finite element idealization

was used. Although the minimum time step of the calculations was 62 ns, the CPU-time of the simulations amounted to 367 s for the impact velocity of 30 m/s and to 7156 s for the impact velocity of 1 m/s. A variety of interface definitions can be used in DYNA3D to model the positioning screws and the interfaces between steel claws and dampers and between dampers and anvil. Friction was modeled between steel claws and dampers and between dampers and anvil. Fixed interfaces were used to model the transition of the circular to the rectangular cross section of the DENT specimens.

The calculations for the detail model were performed with the finite element code ABAQUS /5/ using a discretization with 1494 elements (four nodes, bilinear) and 1598 nodes. An implicit operator /6/ is used for time integration with automatically chosen time steps. The calculations were performed under plane stress and plane strain conditions. Special crack tip elements were not used, since the element size in the notch tip region is 0.1 x 0.1 mm$^2$. The elastic plastic material model is given point by point. The strain rate dependence, Figure 6, is described by the equation

$$\dot{\varepsilon} = D \; ( \sigma / \sigma_0 )^p$$

where $\sigma$ is the flow stress at the strain rate $\dot{\varepsilon}$ and $\sigma_0$ is the flow stress under quasistatic conditions. Using results of tensile tests a mean value for the quotient $\sigma/\sigma_0$ was determined for each strain rate value $\dot{\varepsilon}$. The fitted values of D and p are shown in Figure 6.

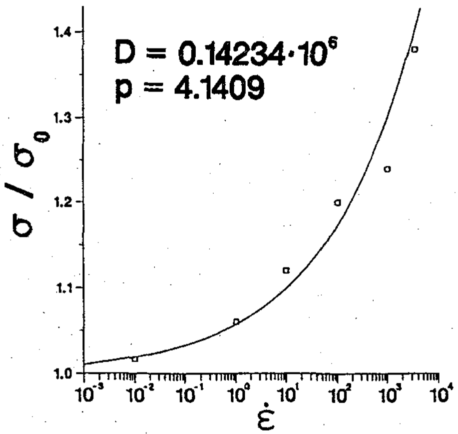


Fig. 6: Strain rate dependence

## 3 Experimental Results

The comparison of the experimental and numerical results was in good agreement. Figure 7 shows the measured and calculated velocity of the anvil for a tensile test with the smooth round bar specimen. The circumferential velocity of the disk was 20 m/s. The oscillations in velocity depend on the geometry and have the same frequency for all impact velocities, that means they are only visible at higher velocities. The rise time of the anvil velocity amounts to 250 µs and is nearly independent of the circumferential velocity of the disk, but it is dependent on the geometry of the damping element and on the resistance of the specimen.

Strain gauge 5 is used for the determination of stresses via quasistatic calibration. Figure 8 shows the strain versus time curve for the same test. It is remarkable that numerical and experimental results show the same oscillations which were caused by the onset of plastic flow. Since the elevation of the upper yield stress is not described in the material model, the numerical results are different from the experimental ones in the range of the upper yield stress. The deviation at the end of the test is due to the necking of the specimen. The material model based on the true stress-true strain curve is not valid anymore in this range.

The analysis of the axial stresses over the gauge length of the specimen showed a homogeneous stress distribution after onset of plastic flow. Wave propagation and reflection effects can be observed only in the elastic part of loading history.

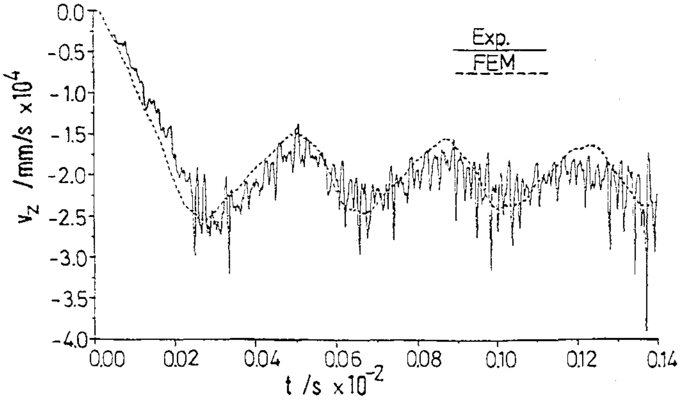


**Fig. 7: Velocity of the anvil, tensile test with an impact velocity of 20 m/s.**

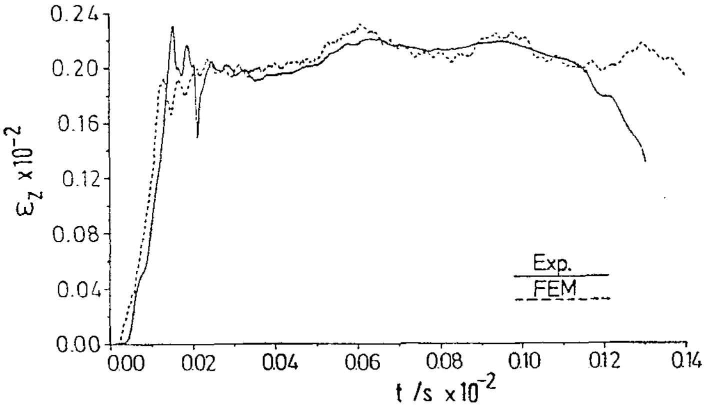


Fig. 8: Strain at the lower dynamometer section of the tensile specimen, impact velocity 20 m/s.

A good agreement between experimental and numerial results could also be found for the simulation of the fracture tests. The strain at the lower dynamometer section of a DENT-specimen with a cross section of 14 x 7 mm$^2$ is shown in Figure 9 for a test with an impact velocity of 20 m/s. The similarity of the oscillations is also clearly visible.

The comparison of experimental and numerical results of the tests with a circumferential velocity of 1 m/s showed that there is not enough kinetic energy stored in the rotating disk. A constant impact velocity could only be found for the DENT specimen with a cross section of 14 x 7 mm$^2$. Hence, a minimum velocity of 3 m/s for tests under constant impact velocity is necessary.

Since the mesh in the area of the notch is coarse, it could not be expected that the strain in this area is calculated correctly. Although a comparison of the measured strain, which is determined by a strain gauge with a gauge length of 10 mm, with the calculated strain shows a good agreement up to a strain of 4 %, Figure 10.

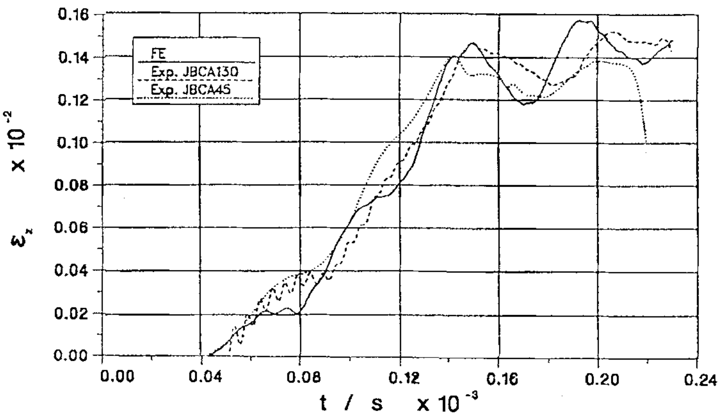


Fig. 9: Strain at the lower dynamometer section of a DENT-specimen with a cross section of 14 x 7 $mm^2$, impact velocity 15 m/s.

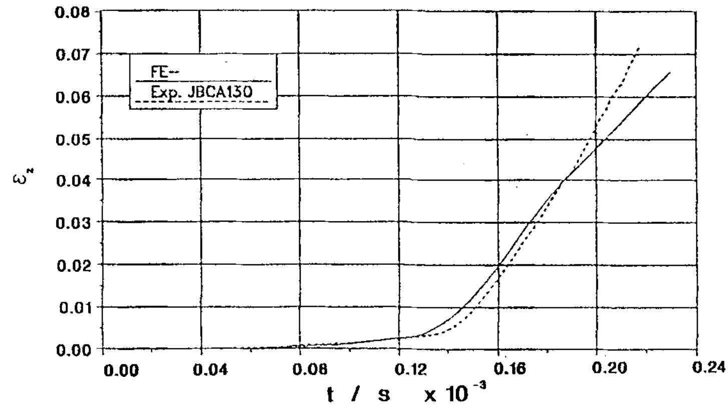


Fig. 10: Strain at the ligament of a DENT-specimen with a cross section of 14 x 7 $mm^2$, impact velocity 15 m/s.

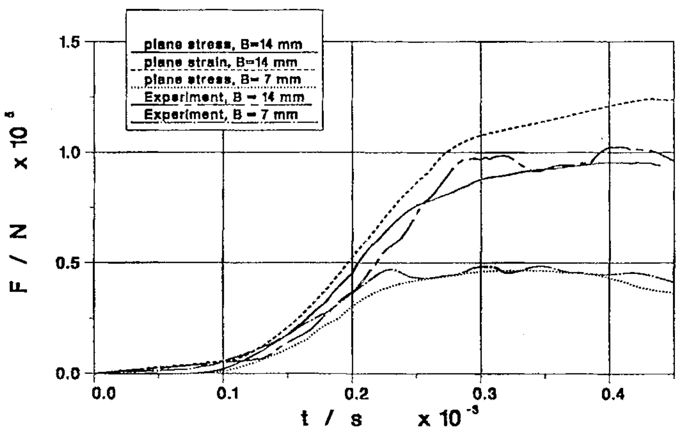


Fig 11: Force versus time diagrams for DENT specimens with different cross sections, impact velocity 5 m/s.

This agreement shows that the calculations are a correct simulation not only of the global behaviour of the specimen and therefore the requirements for the following procedure are met. Displacements at both ends of the rectangular cross sections of the speciemins are determined and used as boundary conditions for the detail model. The comparison of the reaction forces at the boundaries of the detail model, where displacements are prescribed, with the experimental load versus time curve for specimens with cross sections of 14 x 14 $mm^2$ and 14 x 7 $mm^2$ is shown in Figure 11. The 2d idealization shows a fairly good agreement, the experimental results deviate only slightly from either the plane stress or plane strain idealization.

The fine-meshed detail model can be used for the calculation of displacements in the notch tip region. Experimental results determined by an etched microgrid /7/ allowed a comparison for the displacements of a line 200 µm above the ligament for a test with a DENT specimen with a cross section of 14 x 7 $mm^2$ and a impact velocity of 5 m/s, Figure 12.

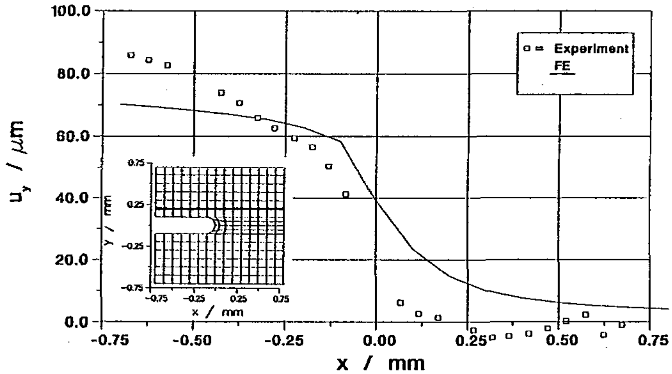


Fig. 12: Displacements in the notch tip region in y-direction on a line 200 µm above the ligament of a DENT specimen with a cross section 14 x 7 $mm^2$, impact velocity 5 m/s.

## 4 Conclusions

The described simulation method could be validated for tensile tests and fracture tests. The additional data which could be obtained from the finite element calculation was useful for the description and analysis of the tests. A detail model which uses the results of the fracture test simulation as boundary conditions allows the calculation of stresses and strains at notch tips and fracture mechanics parameters. The method can be adapted to various other problems, the whole procedure and parts of it can be used to analyze testing conditions and behaviour of components.

## References

/1/ KUSSMAUL, K., DEMLER, T., KLENK, A., Int. Phys. Conf. Ser. No 102: Sess. 4
/2/ HALLQUIST, J. O. , Lawrence Livermore Nat. Lab. , Rept. UCID-19592 1986
/3/ HALLQUIST, J. O. , Lawrence Livermore Nat. Lab. , Rept. UCID-19401 1982
/4/ FLANAGAN, D.P., BELYTSCHKO, T., Int.J.Num.Meth.Eng. 17, 1981, pp. 673-706
/5/ ABAQUS 4.8, Hibbitt,Karlsson & Sorensen Inc. 1989
/6/ HILBER, H. M. , HUGHES, T. J. R. , TAYLOR, R. L. , Earthquake Eng. Struct. Dyn. 6, pp. 99-117, 1978
/7/ KUSSMAUL, K. , DEMLER, T. , KLENK, A. , J. de Phys. Coll. C3, Suppl. au N° 9, Tome 49, 1988, pp. C3-307 - C3-312