New Developments in FEM Based Process Simulation to Predict and Eliminate Material Failure in Cold Extrusion

Manas Shirgaokar, Hyunjoong Cho, Taylan Altan
Engineering Research Center for Net Shape Manufacturing (ERC/NSM), The Ohio State University, 339 Baker Systems, 1971 Neil Avenue, Columbus, Ohio-USA.

Abstract

The use of ductile materials in metal forming is limited by the occurrence of material failure due to cracking (internal or external). To characterize the behavior of the material and predict material failure, it is necessary to determine its flow stress and critical damage value (CDV).This paper deals with the prediction and elimination of defects such as chevron cracks (or central burst defects) during the cold forward extrusion process using Finite Element (FE) based process simulation techniques.

Two major approaches are proposed in predicting ductile fracture: a) macroscopic modeling approach and b) constitutive modeling approach. In this paper, brief insights into the integral damage model and the Model of Effective Stress (MES) model are made and some case studies have been discussed.

1 Introduction

The use of ductile materials in metal forming is limited by the occurrence of material failure due to cracking (internal or external). During a multi-stage forming operation, these cracks can develop early and get closed during later forming stages. This complex material behavior can lead to potential failure in the final product for a given set of process conditions and ultimately lead to rejection.

In the automotive industry, many shaft and shaft-like components, including fasteners, are produced by forward extrusion. Some of these components are considered critical for the safe performance of the vehicle and must be free of defects. These defects could be visible external ones, such as laps or cracks, or non-visible internal defects, such as chevrons. See Figure 1 and Figure 2 for examples of chevron defects.

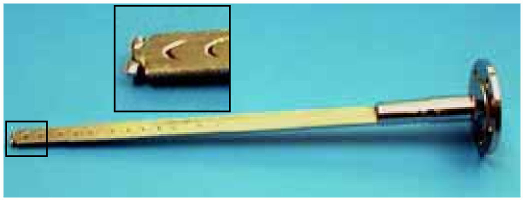


Fig. 1 Automotive axle shaft with chevrons /1/.

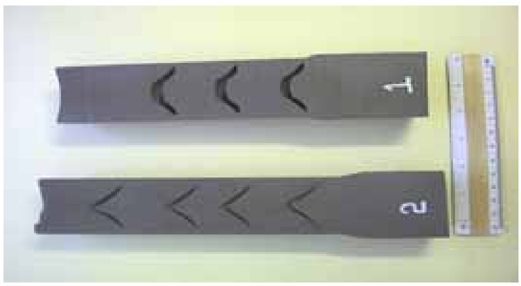


Fig. 2 Illustration of different chevron configurations /1/.

It is highly desirable to predict such defects in the design and development stage, and modify the process conditions accordingly to avoid such failures. With the development of reliable finite element simulation techniques, FEM can be used as a standard tool during the design stage to predict and optimize the process conditions to minimize these forming defects. However, the accuracy of the FEM technique depends on the use of a proper damage model to predict failure.

Two major approaches are proposed in predicting ductile fracture: a) macroscopic modeling approach and b) constitutive modeling approach. Macro-mechanical strain dependent models are functions of stress, strain and material dependent parameters that represent the fraction of the whole forming energy causing the formation of the macroscopic crack. On the other hand, the constitutive models work by accounting for the loss in load bearing capacity. The most widely used constitutive damage model is the Model of Effective Stress (MES).

The main goals of simulation in manufacturing process are to reduce manufacturing/part development time and cost and increase quality and productivity. For instance, in metal forming, process simulation can be used to develop the die design and establish process parameters by a) predicting metal flow and final dimensions of the part, b) preventing flow induced defects such as laps (forging) and excessive thinning and wrinkling (sheet forming), c) predicting temperatures (warming forming operations) so that part properties, friction conditions, and die life can be controlled. Furthermore, process simulation can be very beneficial in predicting and improving grain flow and microstructure, reducing rejects, and optimizing product design /2/.

2 Determination of reliable input parameters for process modeling

The accuracy of FE process simulation depends on reliable input data namely, a) CAD data of the die geometry, b) speed and force characteristics of the press used for forming, c) flow stress of the deforming material as a function of strain, strain rate and temperature in the range relevant to the process being analyzed, and d) friction characteristics at the interface between the deforming material and the die. The tool designers usually provide CAD data of the dies, and press characteristics. Material properties of the deforming material and the friction conditions need to be estimated through tests that emulate similar deformation and friction conditions compared to the process to be analyzed. Various tests and advanced methodologies have been developed at the ERC/NSM to estimate material properties and friction for process simulation in metal forming.

2.1 Determination of flow stress data

The cylinder compression test is widely used to estimate the material properties of the deforming material at room and at elevated temperatures in forging. In cylinder compression test, the upsetting load and the height of the specimen are measured to estimate the flow stress by analytical equations that assume homogeneous deformation of the billet in the test. However, due to the friction at the billet-die interface, barreling is often observed at the center plane of the cylindrical test specimen (Figure 4).

Therefore, in order to accurately determine the material properties, it is necessary to account for the friction involved in the test. In the ERC/NSM, test results namely, a) upset load versus stroke and b) shape of the billet at the end of forming are used in the FE simulation based inverse analysis technique to estimate the flow stress that compensates for friction at the billet – die interface, Figures 4 and 5. The material parameters are determined by minimizing the difference between the experimental and calculated loads. Then the shape of the deformed specimen is compared with the corresponding computed one. If they do not match, the friction factor m_f is adjusted in order to reduce the observed difference at the next iteration. If they match very well, the friction factor that is used in FEM simulation is the identified friction factor, Figure 3 /3/.

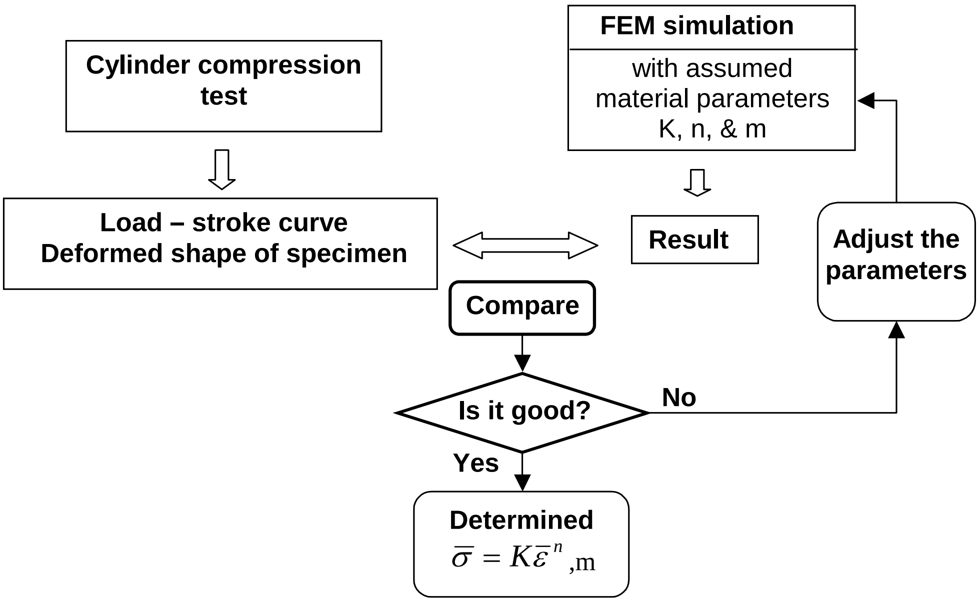


Fig. 3 Flow chart of the inverse analysis methodology to estimate flow stress using FE simulation /3/.

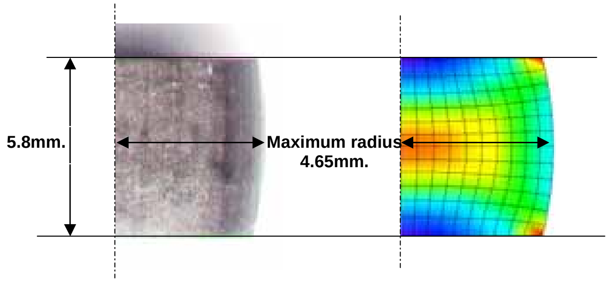


Fig. 4 Comparison of deformed shape in the compression test from FE simulation and experiment for the optimized material parameters ($\sigma = 795.7\varepsilon^{0.15} MPa$) and friction conditions (friction factor m = 0.15) /3/.

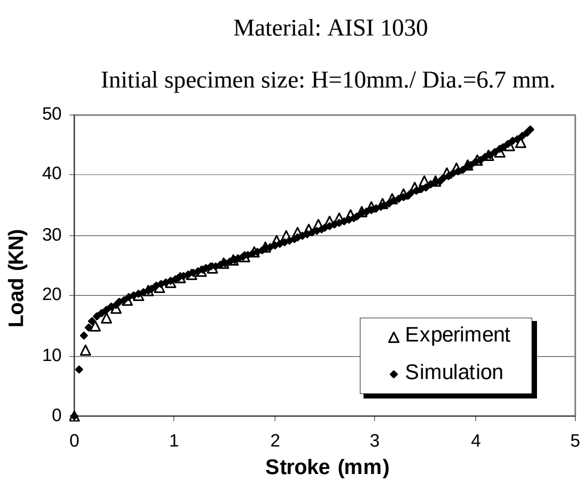
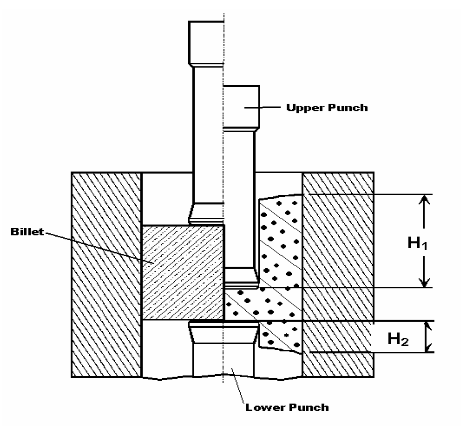


Fig. 5 Comparison of load stroke curve from FE simulation and experiment for the optimized material parameters ($\sigma = 795.7\varepsilon^{0.15} MPa$) and friction conditions (friction factor m = 0.15) /3/.

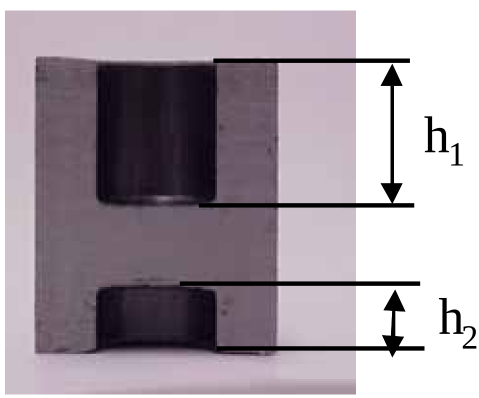
2.2 Determination of interface friction condition

The friction coefficient is another important input in the FE model. The friction value may vary from cold to hot forging processes. Prior to process modeling one needs to know what lubrication system will be used in production.

In forging, the ring compression test and double cup extrusion test are used to evaluate and to estimate friction factor (m) of the lubricants and coatings. The choice of the test depends on the contact pressure and surface expansion in the process that is being studied. Double cup extrusion test is used to quantify lubrication used in processes that involve high contact pressures and surface expansion, Figure 6, while the well known ring compression test is mainly used to test lubricants in cold heading operations. The ERC /NSM has tested several environmentally friendly lubricants for replacement of zinc phosphate coating based lubricant using the double cup extrusion test (Figure 7).



(a) Double cup extrusion test



(b) Cup height ratio = h_1/h_2

Fig. 6 Double cup extrusion test /4/.

The principle behind this test is that the difference in the cup height is an indication of lubricity. If the cup height ratio is equal to 1 then friction is equal to zero. Figure 7 shows the friction values obtained from four tested lubricants. MEC Homat was found to be the best among the four tested lubricants, with a friction factor of about 0.035. MCI and Daido also performed better than zinc phosphate lubricant.

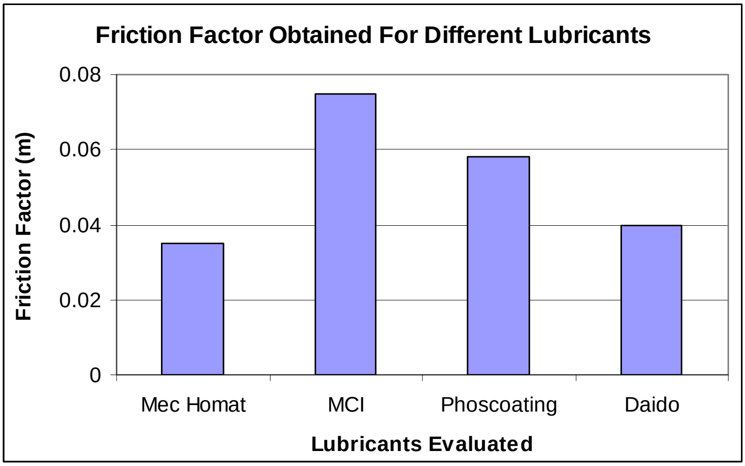


Fig.7 Ranking of the lubricants based on the friction factors obtained /4/.

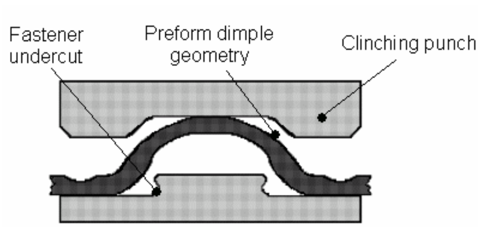
3 Examples of state-of-the-art simulations in forging

The most effective applications of simulation in forging are a) development of forging sequences, b) die stress analysis, and c) solving problems related to an existing manufacturing sequence such as defects. FE process simulations are also used for quotation of new jobs by forging companies.

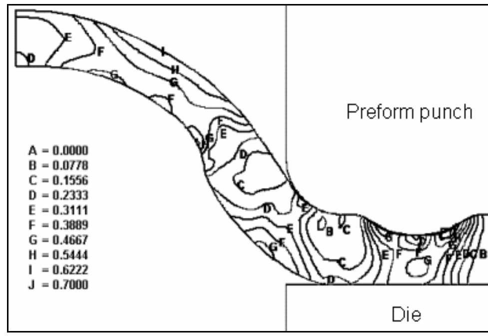
3.1 Clinching Simulation

Clinching is a joining technique for fastening sheet metal components. Sheet metal is pushed into the die cavity by a punch and a leak proof mechanical interlock is formed. There are also many leak-proof joints that require hard fasteners to be attached to a sheet metal part /5/. Depending on the size of the fasteners and the required strength of the joint, the clinching design may be extremely difficult as the process window for clinching may be very narrow.

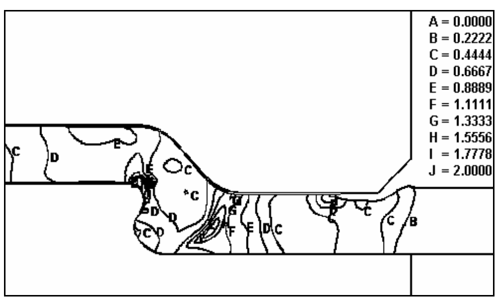
With the aid of FEM, ERC/NSM successfully designed a clinching process for an airtight system for a company. Iterative FE simulations were conducted to obtain a better preform where they finally clinched to the required pull out load (Figure 8).



(a) Preform dimple



(b) Effective strain distribution-final preform.



(c) Effective strain distribution-final joint.

Fig. 8 FE simulation of clinching /5/.

3.2 Advanced orbital forging simulation

Orbital forging is a very unique process with a complicated die movement that can be used to reduce axial load requirements for axisymmetric or near axisymmetric forging operations. Incremental deformation mode and less contact area in orbital forming reduces the axial load necessary for forming a part, compared to a regular forging operation.

At ERC/NSM, FE simulation of the orbital forming process of spindle bearing assembly is being conducted in DEFORM 3D to design a robust assembly process of an automotive spindle and an outer ring (Figure 9). Through FE simulations, influence of various process parameters such as axial feed, tool axis angle, etc., on the residual stress in the bearing inner race of the assembly, deformed geometry of the spindle, and the axial load that the assembly can withstand is being studied. Figure 10 shows the residual stresses predicted by FE simulation in the inner race at the end of the forming process for a given set of process conditions. The load required for forming, deformed geometry of the spindle and the residual stress were found to agree with the experiments /6/.

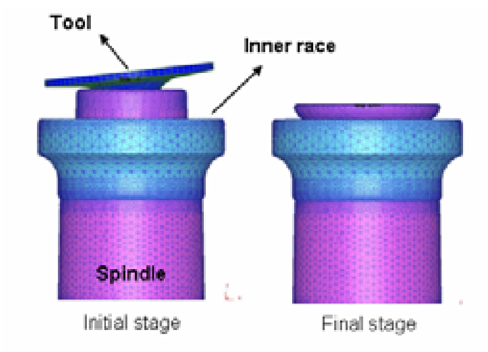


Fig. 9 FE model for orbital forming of spindle bearing assembly /6/.

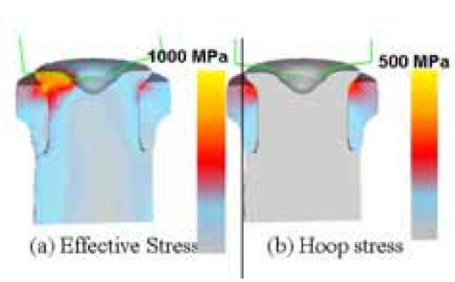


Fig. 10 Residual stresses predicted by FE simulation in the spindle bearing assembly at the end of orbital forming /6/.

3.3 Reducer rolling for preforming in hot forging

Figure 11 shows the 3D simulation of a reducer rolling process used in preforming a billet for hot forging of aluminium upper control arms. Optimum material untilization is a critical issue in hot forging with flash due to the material lost in scrap. The reducer rolling operation in used for achieving optimal material distribution along the length of the billet, thus improving material yield in the forging process.

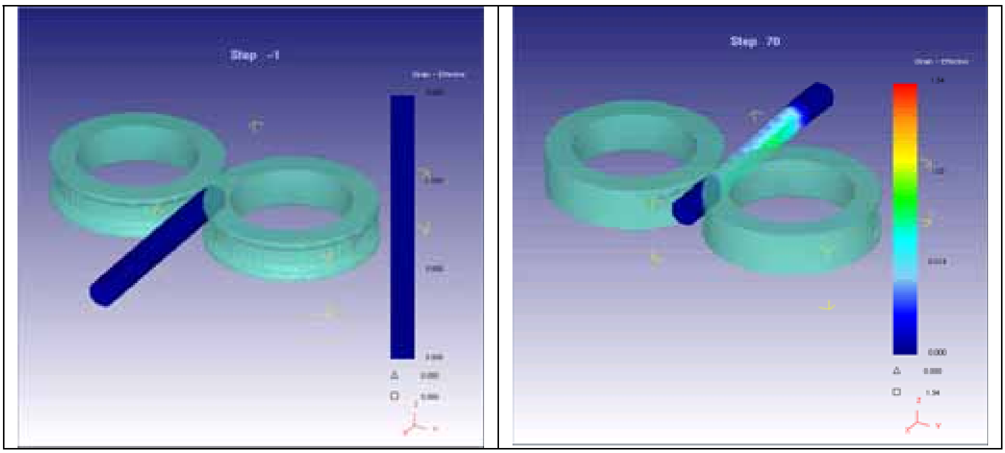


Fig. 11 Three dimensional FE simulation of reducer rolling.

4 Prediction of material failure in cold forging

4.1 Macroscopic approach

Ductile fracture can be defined as a fracture that occurs after a component experiences a significant amount of plastic deformation /7/. Fracture is influenced by numerous parameters, such as, the deformation history of the workpiece material, and the process conditions such as the rate of deformation, lubrication and friction /8/. Other factors that influence fracture include chemical composition, microstructure, surface conditions and homogeneity. There are several ductile fracture criterions, which seem to be reliable for predicting surface or internal cracks, according to experiments. These can be generally represented by:

$$\int F(deformation)\,d\bar{\varepsilon} = C \qquad (1)$$

Equation 1 indicates that ductile fracture is a function of the plastic deformation of the material. This includes the geometry, damage value, C, and strain of the workpiece (or effective strain), $\bar{\varepsilon}$. When the maximum damage value (MDV) of the material exceeds the critical damage value (CDV), crack formation is expected. Different ductile fracture criteria yield different damage values for a given process /9/.

Kim, et al., [1994] developed a methodology to predict ductile fracture by using FE simulations (see Figure 12). Seven different ductile fracture criteria were implemented in the FE code DEFORM™-2D. Five different processes were selected for the experimental investigation: a) compression test with grooved dies, b) notch tensile test, c) collar test, d) pierce upsetting and e) multi-pass forward extrusion /9/.

It was found that the modified Cockroft and Latham's criterion predicts with good agreement the location of the maximum damage value and that fracture occurs when the cumulative energy due to the maximum tensile stress exceeds a certain value (Equation 2) /10/:

$$\int^{\bar{\varepsilon}} \frac{\sigma^*}{\bar{\sigma}}\,d\bar{\varepsilon} = C_b \qquad (2)$$

Cerreti, et al. [1996] further developed the methodology to simulate the fracture of components subject to large amounts of deformation and simulated sucessfully the formation of chevron cracks for the extrusion experiments performed by Kim, et al., [1994].

4.1.1 Methodology to predict ductile fracture

Based on the assumption that CDV is a material constant, several tests should be performed to obtain the characteristic data from the material, such as the flow stress and the critical damage value (CDV) as shown in Figure 12. Once the CDV for a specific material is obtained it is possible to predict through FEM simulations the formation of cracks in forming operations. Figure 13 shows the methodology used to predict and prevent the formation of cracks in forming operations.

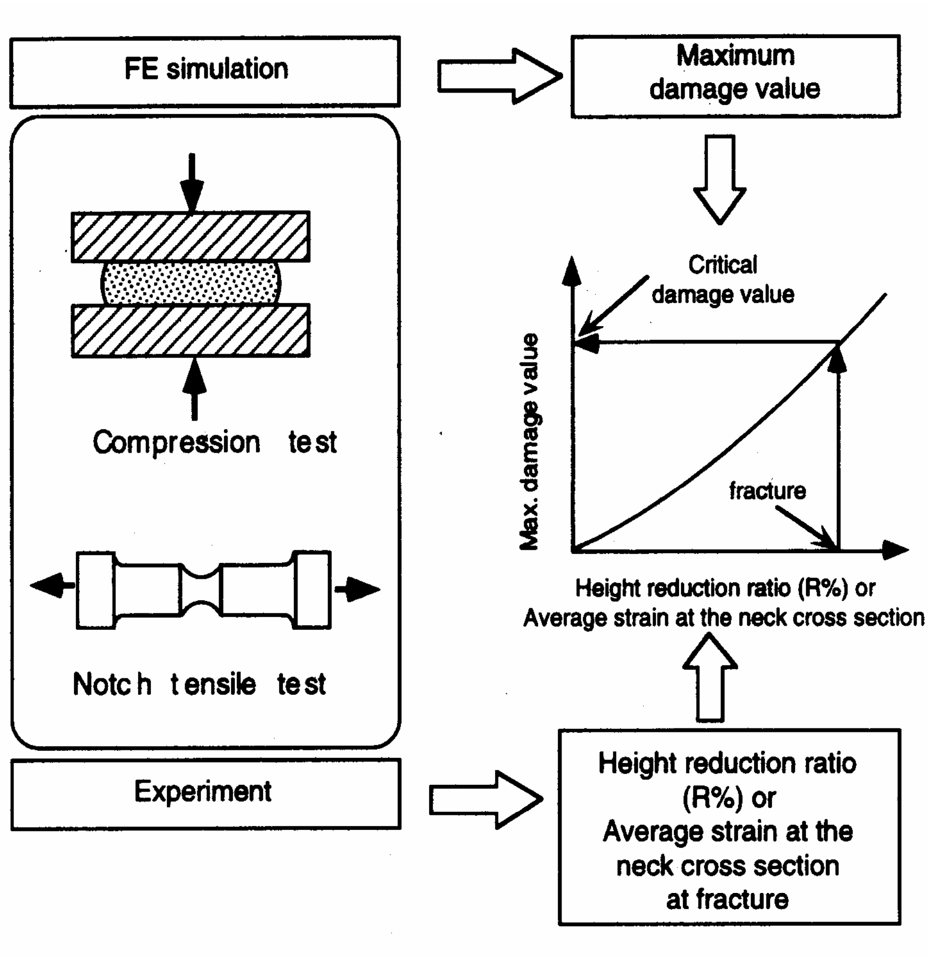


Fig. 12 Proposed approach to obtain critical damage value from compression and notch tensile test in conjunction with FE simulations /9/.

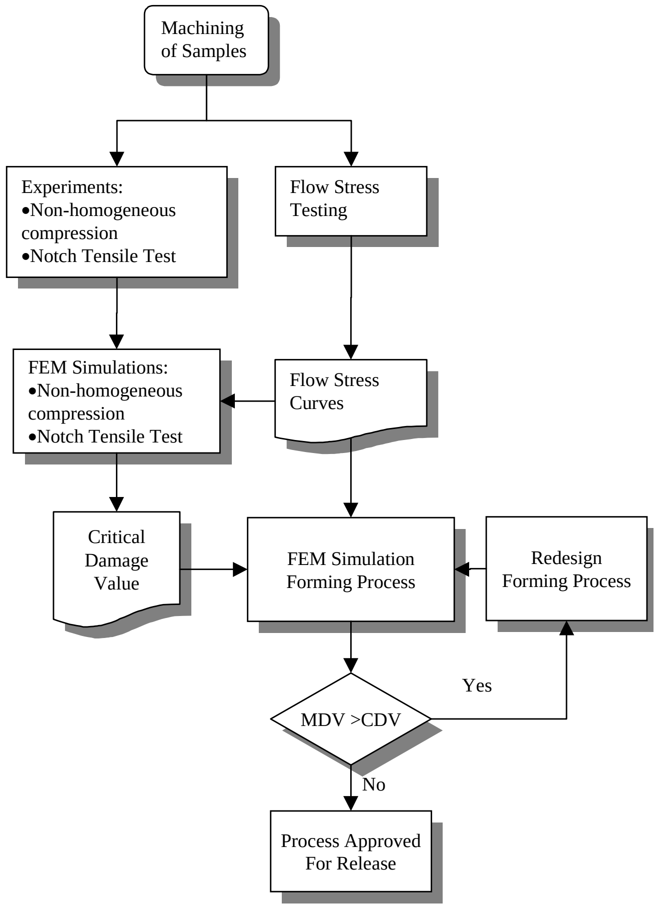


Fig. 13 Methodology to predict and prevent the formation of cracks /9/.

4.1.2 Technique for measurement of critical damage value (CDV)

The critical damage value can be obtained by two tests; the non-uniform compression test with grooved dies and the tensile test with a notched specimen. The non-uniform compression test with grooved dies consists of upsetting a cylinder until a crack, observed near the equatorial surface, is detected. The specimen height at which the crack occurs is known as the fracture height, H_f. Then FE simulations are conducted for the compression test up to the fracture height, H_f to obtain the distribution of the damage value in the work piece at the time of fracture. If the maximum damage value (MDV) coincides with the location of the fracture, then this is the critical damage value of the material tested.

In the tensile test with notched specimens, the neck diameter is measured continuously until the occurrence of fracture (d_f). Then simulations are conducted to calculate the damage distribution at the instant the neck reaches the fracture diameter d_f. The MDV just before fracture is the CDV.

4.1.3 Experimental measurement of critical damage value

As discussed earlier, non-homogeneous compression with grooved dies were conducted with a cylindrical specimen to produce cracks in the free surface or bulging surface. Vertical cracks were seen in the equatorial surface of the specimens as shown in Figure 14.

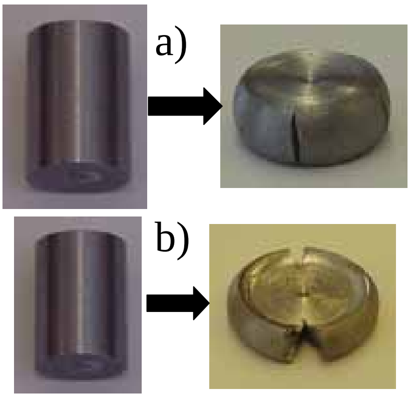


Fig. 14 Specimens for non-uniform compression test, a) SAE 1137 hot rolled, b) SAE 1524 spheroidized annealed /12/.

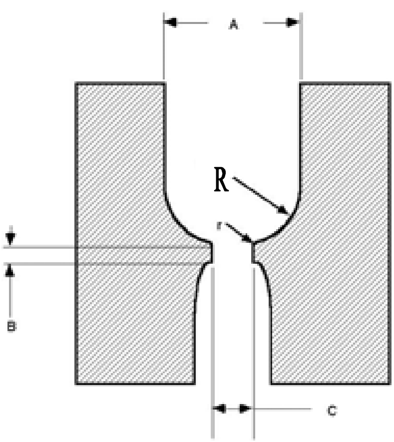
Tensile tests were conducted to provide information on the average fracture strain at the neck cross section obtained from load vs. neck diameter curves. Cracks are expected to start from the center of the notched cross section. The fracture found in the tension specimens is cup-and-cone type, which can be seen in Figure 15. This means that the fracture begins at the center of the notched specimen.

Fig. 15 Notched tensile test specimen before-and- after fracture /11/.

4.1.4 Damage value prediction: single-pass extrusion

It is necessary to correctly confirm the location of the critical damage value at the center of the workpiece because this is where chevron defects may occur. The parameters to be investigated during forward extrusion are shown in Figure 16. Figure 17 shows the variation of the damage value for several reduction ratios for SAE 1137. In all simulations the maximum damage value is located along the center of the work piece.

The highest damage value was 0.33, which does not exceed the selected average CDV of 0.51 for SAE 1137. In fact, it is below the conservative value of 0.44. Therefore, under the selected process conditions, chevron cracks would not be expected during one-pass forward extrusion.



A = entrance (bore) diameter: 1 in.
RR = Reduction ratios: 30%, 45%, and 55%
C = exit (die) diameter: according to reduction ratio
R= die radius: A-C
B = die land length: low: 0.060”, high: 0.150”
r = transition radius: 0.005 in., 0.200 in, 0.300 in.

Fig. 16 Schematic representation of the parameters to be investigated during forward extrusion with spherical die.

Also shown in Figure 17 is that the effect of the die land length (B) is essentially negligible. When the small transition radius (r = 0.005”) is used, there is a higher stress concentration and therefore, the flow of the material will be restricted. The variation of the damage value for the given transition radius is observed in Figure 18. This shows that damage value decreases when the transition radius is increased, especially for reductions in area between 45-55%. The lowest damage values obtained were at the highest reduction in area evaluated. Notice that as the reduction in area increases, the deformation near the center of the work piece turns more compressive and DV decreases (see Figure 19 for the distribution at 30%, 45%, and 55%).

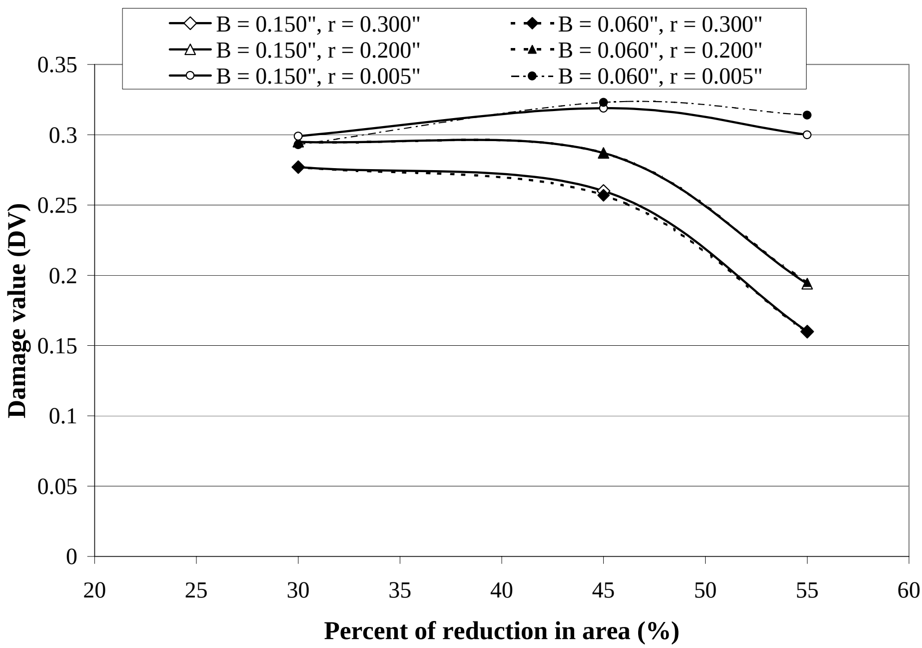


Fig. 17 Effect of reduction ratio on damage value in single-pass forward extrusion /12/.

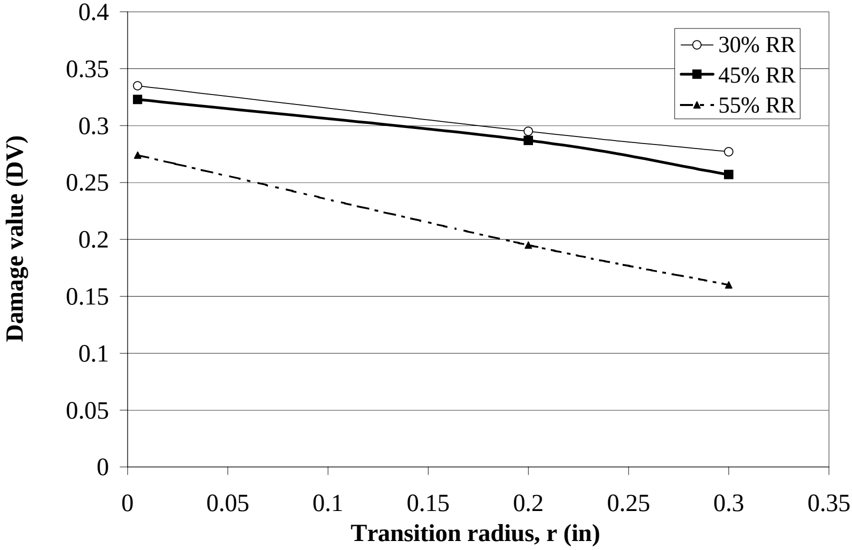


Fig. 18 Effect of transition radius for several reduction ratios (RR) /12/.

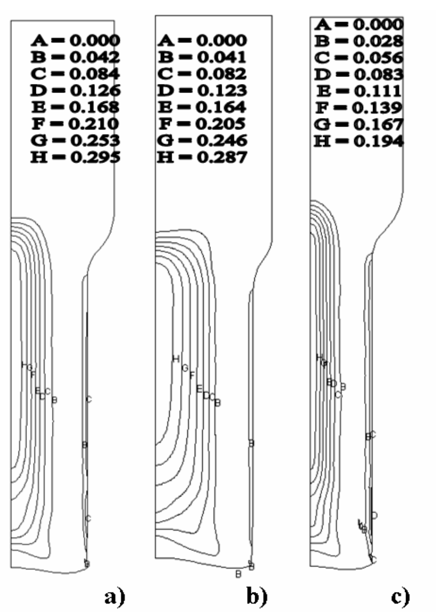


Fig. 19 Distribution of the damage value during several reductions in area: (a) 30%, (b) 45% and (c) 55% (B = 0.150” and r = 0.200”) /13/.

4.1.5 Damage value prediction: multi-pass extrusion

In order to validate the premise that the critical damage value can predict the formation of chevron cracks, simulations for conical dies under the same conditions proposed by [Kim, 1994] were performed for SAE 1137, hot rolled. The main dimensions for the three inserts are shown in Table 1.

Table 1: Die design parameters for forward extrusion with conical dies /12/.

Die insert number	Bore Diameter (in)	Exit diameter (in)	Die land length (in)	Die angle
1	1.04	.896	0.125	22
2	1.05	.826	0.143	22
3	1.06	.742	0.146	22

This sequence was selected because the same area reduction ratios and half die angle were used in a forging company and developed chevron cracks, generating critical damage values over 0.5. For the simulations, the average CDV used was 0.51 (corresponding to SAE 1137).

After the first pass of this forward extrusion simulation with conical dies the damage value obtained is 0.25. Figure 20 illustrates the distribution of the damage value and the formation of chevron cracks along the centerline of an SAE 1137 specimen. These multi-pass forward extrusion simulations show that the damage value at the center of the work piece is much higher than in single-pass extrusion, which leads to higher possibility of chevron cracks.

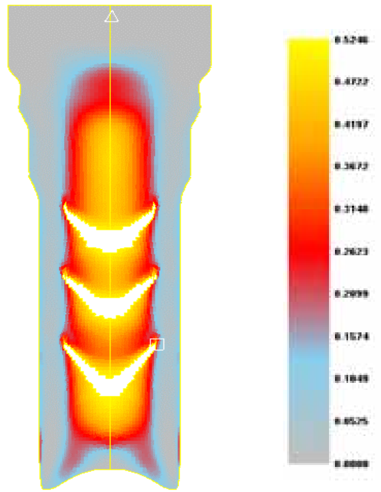


Fig. 20 Formation of chevron cracks during the third stage of three-pass forward extrusion using first pass as spherical die with material SAE 1137 (hot rolled) and its correspondent damage value distribution /12/.

4.2 Constitutive approach

The constitutive models work by accounting for the loss in load bearing capacity during void coalescence by incorporating the effects of void nucleation, growth and coalescence. Among the constitutive models, Model of Effective Stress [MES] proposed by Kachanov and Lemaitre is most widely used /13/. The MES is based on the principle that all values of the material parameters (stress, elastic modulus) that are affected by ductile fracture should be substituted by their respective equivalent or effective values.

The MES damage model is based on the assumption that ductile fracture occurs in three distinct stages:

- The occurrence of micro-cavities at inclusions in the material
- The growth of voids and
- The final rupture due to macroscopic coalescence of the pores.

To emulate complex material behavior described by this model, the material volume is divided into small Representative Volume Elements (RVE) [Figure 21]. The basic assumption being that each RVE contributes to the whole deformability of the body by carrying a small amount of the load F.

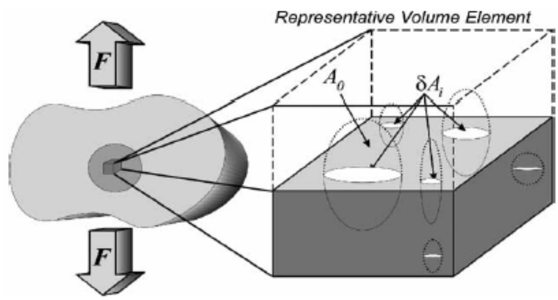


Fig. 21 Definition of Representative Volume Elements (RVE) and absolute damage (F=load acting on the body, A_o is the load carrying area, ∂A_i is the total cavity area) /13/.

As shown in Figure 21, the cavities in the RVE that intersect the load carrying area (A_o) increase the effective or equivalent stress that the workpiece has to endure. The absolute damage, D, can be defined as the ratio between the intersecting cavity areas and the original plane area. Hence, by substitution, the effective stress, $\tilde{\sigma}$ acting on the material can be obtained. After the calculation of the absolute damage value and the damage evolution, the flow potential after Von Mises yield criterion is modified to account for damage. This is achieved by reducing the flow stress of the material in the surrounding area of calculated damage. Like most damage models, the MES also compares the absolute damage predicted against a critical damage value to determine the onset and location of ductile failure. The critical damage value is obtained from tensile and compressive tests performed with materials from the same batch and equal diameter.

4.2.1 Damage value prediction: cold formed double cup

In the production sequence of a five stage cold forged double cup, damage was observed in the fourth stage. This production sequence of the double cup [Figure 22] was simulated using MSC/Superform /14/. The first, second and third stages are axisymmetric backward extrusion processes, while the two subsequent stages were extended to 3D modeling in the FE analysis. It is critical to note that, the annealing stage in between the first and second stage was emulated by resetting the strain and the damage distribution in the workpiece after the simulation of the first stage. In the fourth stage of the production sequence, the two inner cups are formed by two combined backward extrusion processes. A hexagonal outer shape is applied to the lower half of the work piece, which makes the three-dimensional modeling necessary. The FE model was created with 60° symmetry for the simulations. Smaller angles of symmetry caused problems with remeshing of the hexahedral mesh due to rapid distortion. The values from the axisymmetric simulation [3rd stage] were applied to the 3D mesh by an incremental rotation of the 2D mesh with subsequent rezoning. In the fifth stage only a calibration takes place in order to meet the desired tolerances /14/.

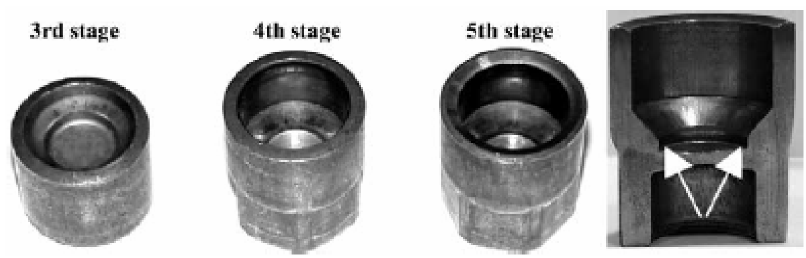


Fig. 22 Production sequence of the double cup and observed damage locations in the fourth stage /14/.

The production sequence was carried out on a multi-stage crank press with a stroking rate of 60 strokes/min. The workpiece was made from plain carbon steel QSt-38. A high amount of scrap (up to 10%) arose in this production sequence in the fourth stage, in the form of surface fissures at the bottom of the upper cup above the outer hexagonal surface [Figure 22]. The material properties for the FE simulation and the MES parameters were determined from the same batch of materials /14/.

The FE simulation of the fourth stage verified the high susceptibility to damage of the work piece at the location observed in reality [Figure 23]. At the bottom of the upper cup the values for absolute and relative damage attain D=1 and D_{rel}=1. Further investigations of the damaged region concentrated on the principal stress distribution, as they were known from the experiments to influence the fracture evolution in ductile materials.

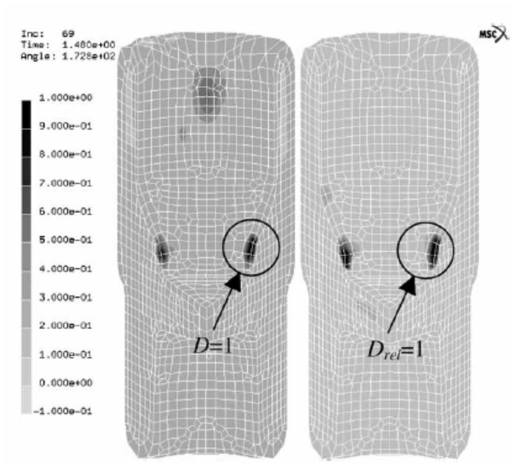


Fig. 23 Absolute and relative damage distribution of the MES in the fourth stage with original tools at 173° crank angle /14/.

All maximum principal stresses in the critical region were positive and much higher than the maximum stress the material can resist [Table 2]. The first step in modifying the design was to smoothen the tool edges, as it was known from practice that less damage occurred with worn tools. The tool radius was only slightly enlarged to about 15%. The result [Table 2] shows the much reduced maximum stresses predicted by FE simulation.

Table 2: Maximum principal stresses with the original and modified punch geometry in the critical workpiece region of the fourth stage /14/.

Principal Stress [N/mm^2]	**Original Punch Geometry**	**Modified Punch Geometry**
σ_{11}	1167.84	595.60
σ_{22}	1168.13	600.33
σ_{33}	1172.01	733.20

The FE prediction of the distribution of the damage value D using the modified tool is shown in Figure 24. The maximum value of $D_{rel} = 0.35$ coincided with the results obtained from production as it indicated no susceptibility to cracking. In Figure 24, it can be observed that the maximum damage value D is now located at the upper rim of the workpiece. It coincides with the value from Figure 23 and can be regarded as not critical. As illustrated in the study conducted by Behrens and Just, the MES model shows good agreement with the production and was used successfully as an index to improve and modify the production sequence in this study /14/.

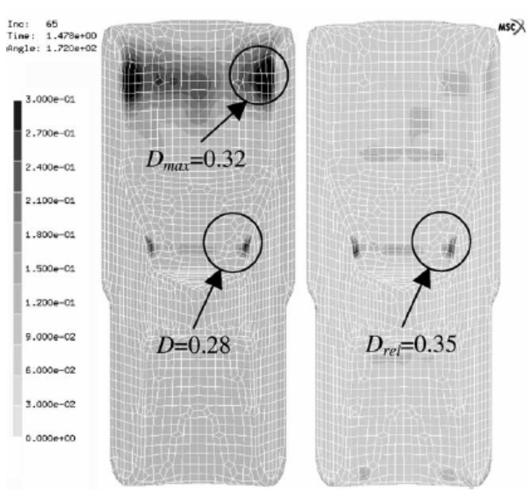


Fig. 24 Absolute and relative damage distribution of the MES in the fourth stage with modified tools at 172° crank angle /14/.

4.2.2 Advantages and limitation of the MES model

The Model of Effective Stress (MES) developed by Lemaitre is capable of:

- Predicting the onset and location of ductile fracture in common cold and warm forging applications.

- Predicting the evolution of damage or damage development in a work piece after the initiation of ductile fracture.

- Adapting to the nature of the deformation process by varying the critical damage value depending on current stress state of the material. This phenomenon increases the accuracy in the prediction of the ductile fracture.

The limitations of the MES model are:

- The material properties required in the MES model are determined under a uniaxial state of stress and are then transformed to a multiaxial state of stress using different triaxiality functions. The accuracy of these triaxiality functions in each load case has to be validated before a detailed study of the process can be carried out.

- For accurate representation of forming limits using the MES model under warm forging conditions (elevated temperature), the material properties used in the MES model should be evaluated at elevated temperature. Information regarding the determination of material properties at elevated temperatures is not clearly mentioned in the literature.

5 Summary and Future Developments

- Finite element analysis can be used in order to develop design guidelines to avoid the formation of chevron cracks during the forward extrusion process. Commercial FE codes such as DEFORM can be used for this purpose by implementing ductile fracture criteria in the software.

- Physical tests such as cylinder compression test and double cup extrusion tests, along with FE based inverse analysis can be used to determine the material properties and interface friction conditions reliably.

- Critical damage values (CDV) were determined, first by conducting experiments of non-uniform compression tests with grooved dies and notched tensile tests and later these results were calibrated by means of DEFORM 2D.

- Additional refinement of the physical testing procedure for CDV will improve the accuracy of predicting defects. These refinements could be better, by developing a more accurate detection of crack initiation.

- The Model of Effective Stress (MES) criteria by Lemaitre et.al. was chosen from various criteria for prediction of ductile damage failure of materials in cold and warm forging. The primary reason for selecting this model was its ability to emulate accurately most of the real world cold and warm forging applications and its ease of integration into commercial FE code.

- In this paper, several case studies demonstrating the capability of implementing damage models in commercial FE codes and accurately predicting defects were discussed. The state-of-the-art indicates that it is possible to predict and eliminate crack formation in cold and warm forging operations.

References

/1/ Vazquez, V.
Altan, T.
"Prediction of Ductile Fracture in Forward Extrusion with Spherical Dies" NAMRC 2000, May 2000, p. 57.

/2/ Ngaile, G.
Altan, T.
"Computer Aided Engineering in Forging", Proceedings of the 3rd JSTP International Seminar on Precision Forging, Nagoya, Japan, March 14-18, 2004.

/3/ Cho, H.
Ngaile, G.
Altan, T.
"Simultaneous Determination of Flow Stress and Interface Friction by Finite Element Based Inverse Analysis Technique, annals of CIRP, v.1, 2003, p.221

/4/	Ngaile, G. Schumacher, R Gariety, M. Altan, T. Kolodziej, J.	“Development of Replacements for Phoscoating Used in Forging Extrusion and Metal Forming Processes,” 2002, Report No. PF/ERC/NSM-02-R-85.
/5/	P. Wenning, G. Ngaile, T. Altan,	“Design of a fastener clinching process using FEM“, Proceedings of NAMRC, May 2003, Hamilton, Ontario, Canada, p.17.
/6/	Cho, H. Ngaile, G. Altan, T.	"3D FEA of Orbital Forming Process and Inverse Analysis for the Determination of Reliable Material Property", NUMIFORM 2004, June 2004
/7/	Hosford, W. Caddell, R	“Metal Forming: Mechanics and Metallurgy”. Second Edition. PTR Prentice Hall, Englewood Cliffs, N.J. 07632
/8/	Cerretti, E. Taupin, E. Altan, T.	“Simulation of metal flow and fracture applications in orthogonal cutting, blanking and cold extrusion.” Annals of the CIRP, Vol. 46/1/1997.
/9/	Kim, H., Yamanaka, M. Altan, T.	“Prediction of ductile fracture in cold forging by FE simulations.” Engineering Research Center for Net Shape Manufacturing. Report No. ERC/NSM-94-42. August 1994.
/10/	Oh, S.I. Chen, C. Kobayashi, S.	“Ductile fracture in axisymmetric extrusion and drawing: Part 2 Workability in Extrusion and Drawing”, Transactions of the ASME, 101, Feb 1979. pp. 36-44.

/11/ Vazquez, V.
Altan, T.
"Measurement of Flow Stress and Critical Damage Value in Cold Forging", International Journal of Forming Processes, vol.4,, Nov. 1-2, 2001, pp. 167-175

/12/ Santiago-Vega, C.
Vazquez, V.
Altan, T.
"Development of Design Guidelines to Avoid Chevron Crack Formation in Forward Extrusion with Spherical Dies", Engineering Research Center for Net Shape Manufacturing. Report No. ERC/NSM-99-R-27-B, 1999.

/13/ Lemaitre, J.
"A Course on Damage Mechanics," 2nd. Edition (1992), Springer, Munich.

/14/ Behrens, A.
Just, H.
"Extension of the forming limits in cold and semi-hot forging by the FE based fracture analysis with the integrated damage model of effective stresses," Metal Forming (2002), Birmingham, UK.