Material Modeling for Dynamic Modulus Degradation using Abaqus

Anup V. Kale[1], Senthil Subramanium[2] and Dr. B. S. Suresh[3]

[1&3] Dept. of Mechanical Engineering, BMS College of Engineering, Bangalore, India-560019

[2] Robert Bosch Engineering and Business Solutions, Bellandur, Bangalore, India-560103

[1]kaleanupv@gmail.com, [2]senthil.subramaniam@in.bosch.com and [3]dr_bss@yahoo.com

***Abstract**: Real life mechanical failures can be predominantly attributed to three factors: uncertainty in magnitude and character of loads, insufficient consideration to in-service behavior of material, unreasonable assumptions in design. FEA Simulation can provide valuable inputs to avoid risk of failures. Realistic material modeling has an important role in this regards. In Practical engineering analysis, modeling material damage through degradation of Elastic modulus is one of the effective techniques. Using relevant amongst several theories, constitutive equations and empirical formulations characterizing this phenomenon, creates a plausibly realistic FE Model. However these models are approximate, not generically applicable and are not mathematically well posed. Additionally these models lack dynamic interaction with field variables of simulation. Material variation data from experimental results remains the most reliable resort; however Material modeling for dynamic variations using experimental material data is not practiced widely owing to absence of automated modules in FE packages. This paper proposes a method for material modeling for dynamic modulus degradation with field variables of simulation. A UMAT subroutine for customized material model was developed that interactively modifies the material model for each increment of simulation. A case study is performed demonstrating comparison of results with traditional approach and dynamic material models; establishing that the proposed method provides more realistic and accurate results.*

***Keywords**: Computational Plasticity, Material Modeling, Damage Mechanics, FEA, modulus degradation, Abaqus, UMAT.*

1. Introduction

The design of any mechanical structure or component primarily involves arriving at a solution in terms of Geometry and Material to support a given loading conditions. Avoiding failure of structures/components is always a major concern for designers. Prominent drivers of Mechanical failures are:

- Uncertainty in Magnitude and Character of loading
- Undesired changes in Material properties during service
- Consequent changes in geometry

A designer therefore must analyze the possibility of such conditions. A detailed understanding and characterization of behavior of given structure/component under such circumstances is a key to effective design. Finite Element (FE) Simulation can provide valuable inputs in this regards.

Quality of FE Simulation results depends on the quality of FE model: How real world conditions are translated into the FE Model. FE models mainly consist of Geometry, Material and Loads. Hence for better simulation results designer's attempts should be focused towards more realistic FE Models in terms of Geometry, Materials and Loads.

It has been commonly observed that the material properties vary during service with deformations, stresses, strain energy, temperature, irradiation etc. Thus for realistic FE Modeling such effects should be incorporated the Material model.

1.1 Background

A significant amount of research work has attempted to characterize in-service behavior of the materials. (Krajcinovic, 1989) in his seminal work in the field of Damage Mechanics comments about the practical applicability of engineering research providing better understanding of material property variation. He suggests the use of a comprehensive damage parameter/s to characterize in-service material variation for practical analysis instead of getting into details of several microscopic or macroscopic processes.

In context of mechanical failures, material property of key importance is Elastic Modulus and much research suggests that, modeling material damage through degradation of Elastic modulus is one of the most effective techniques.

(Morestin, 1996) commented about the necessity of taking into account the variation of young's modulus with plastic strains and a quantitative comparison of diminution of Young's modulus with percentage plastic strain. (Marko Vrh, 2008) demonstrated the effect of Young's modulus degradation in spring back calculation of steel drawing process experimentally and numerically. (M. Yang, 2004) evaluated change in material properties due to plastic deformation using both macroscopic and microscopic experiments and showed that the phenomenon affects the spring-back significantly. (Hai Yan Yu,2009) examined variation of elastic modulus of TRIP steels during plastic deformation and its influence on spring-back of U channel forming and proposed an empirical expression for modulus variation analyzing the case study numerically.

A number of hypotheses, theories, and Empirical findings have been established to characterize the Modulus degradation with respect to strains, temperature, deformation, irradiation etc. consequently numerous approaches are available for material modeling which mainly include Constitutive Equations, Power Laws, and Empirical Formulations etc. However these available models lack generic applicability for all materials, are not mathematical well posed for programming and are tedious to implement because several empirical constants/parameters associated. Additionally they may not be reliable in many cases owing to level of approximations involved. As most of these traditional models are approximated characterization of material behaviors observed during laboratory tests, use of material testing data itself in FE Material model would possibly yield best results concurrently avoiding most of the difficulties in implementing traditional approaches.

1.2 Proposed Model

The present work proposes a method of using the experimental material data of uniaxial tensile test to model material variation for FE simulation. Material variation is modeled as modulus degradation with strains. Incremental formulations of material plasticity are rearranged employing analogy with elastic formulations to compose stiffness matrix representing modulus degradation corresponding to given strain state. Using these rearranged formulations degraded stiffness matrix (for a given strain state) is calculated referring material testing data. Stiffness Matrices evaluated for varying strains over the course of simulation represent the dynamic variation in material definition.

The proposed model is implemented using UMAT: User Defined Material Definition Subroutine provided with Abaqus/Standard FE Solver. The subroutine UMAT is essentially a FORTRAN program that interacts with material data of uniaxial tensile test to formulate degraded stiffness matrix at all material calculation points of element during every increment.

In this work the proposed model is critically reviewed by simulating a uniaxial tensile test as case study. The case study is performed with traditional material modeling approach like Isotropic hardening and the proposed model both making use of same material data. The results of simulation are compared with actual experimental results of the material. It is thus demonstrated that the proposed model yields more accurate and realistic simulation results.

2. Methodology

2.1 Theory

From the theory of computational plasticity (Dunne, 2005) incremental stress formulation is given as Equation 1.

$$\sigma = \sigma_0 + d\Delta\sigma$$

Equation 1

Where 'σ_0' is the Stress state at the beginning of increment and 'σ' is the Stress State at the end of increment. '$d\Delta\sigma$' can be given as Incremental form for Hooke's law Equation 2.

$$d\Delta\sigma = \frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\, d\Delta\varepsilon$$

Equation 2

The term $\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)$ is called as Material Jacobian or Tangent Stiffness Matrix in Abaqus and given as Equation 3 for elastic behavior.

$$\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right) = \begin{bmatrix} 2G+\lambda & \lambda & \lambda & & & \\ \lambda & 2G+\lambda & \lambda & & 0 & \\ \lambda & \lambda & 2G+\lambda & & & \\ & & & G & 0 & 0 \\ & 0 & & 0 & G & 0 \\ & & & 0 & 0 & G \end{bmatrix}$$

Equation 3

A generalized expression for Material Jacobian which is applicable for elastic as well as inelastic behavior of material can be expressed as Equation 4.

$$\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)=\begin{bmatrix}\frac{\partial\Delta\sigma_{11}}{\partial\Delta\varepsilon_{11}} & \frac{\partial\Delta\sigma_{11}}{\partial\Delta\varepsilon_{22}} & \frac{\partial\Delta\sigma_{11}}{\partial\Delta\varepsilon_{33}} & \frac{\partial\Delta\sigma_{11}}{\partial\Delta\gamma_{12}} & \frac{\partial\Delta\sigma_{11}}{\partial\Delta\gamma_{23}} & \frac{\partial\Delta\sigma_{11}}{\partial\Delta\gamma_{31}} \\ \frac{\partial\Delta\sigma_{22}}{\partial\Delta\varepsilon_{11}} & \frac{\partial\Delta\sigma_{22}}{\partial\Delta\varepsilon_{22}} & \frac{\partial\Delta\sigma_{22}}{\partial\Delta\varepsilon_{33}} & \frac{\partial\Delta\sigma_{22}}{\partial\Delta\gamma_{12}} & \frac{\partial\Delta\sigma_{22}}{\partial\Delta\gamma_{23}} & \frac{\partial\Delta\sigma_{22}}{\partial\Delta\gamma_{31}} \\ \frac{\partial\Delta\sigma_{33}}{\partial\Delta\varepsilon_{11}} & \frac{\partial\Delta\sigma_{33}}{\partial\Delta\varepsilon_{22}} & \frac{\partial\Delta\sigma_{33}}{\partial\Delta\varepsilon_{33}} & \frac{\partial\Delta\sigma_{33}}{\partial\Delta\gamma_{12}} & \frac{\partial\Delta\sigma_{33}}{\partial\Delta\gamma_{23}} & \frac{\partial\Delta\sigma_{33}}{\partial\Delta\gamma_{31}} \\ \frac{\partial\Delta\sigma_{12}}{\partial\Delta\varepsilon_{11}} & \frac{\partial\Delta\sigma_{12}}{\partial\Delta\varepsilon_{22}} & \frac{\partial\Delta\sigma_{12}}{\partial\Delta\varepsilon_{33}} & \frac{\partial\Delta\sigma_{12}}{\partial\Delta\gamma_{12}} & \frac{\partial\Delta\sigma_{12}}{\partial\Delta\gamma_{23}} & \frac{\partial\Delta\sigma_{12}}{\partial\Delta\gamma_{31}} \\ \frac{\partial\Delta\sigma_{23}}{\partial\Delta\varepsilon_{11}} & \frac{\partial\Delta\sigma_{23}}{\partial\Delta\varepsilon_{22}} & \frac{\partial\Delta\sigma_{23}}{\partial\Delta\varepsilon_{33}} & \frac{\partial\Delta\sigma_{23}}{\partial\Delta\gamma_{12}} & \frac{\partial\Delta\sigma_{23}}{\partial\Delta\gamma_{23}} & \frac{\partial\Delta\sigma_{23}}{\partial\Delta\gamma_{31}} \\ \frac{\partial\Delta\sigma_{31}}{\partial\Delta\varepsilon_{11}} & \frac{\partial\Delta\sigma_{31}}{\partial\Delta\varepsilon_{22}} & \frac{\partial\Delta\sigma_{31}}{\partial\Delta\varepsilon_{33}} & \frac{\partial\Delta\sigma_{31}}{\partial\Delta\gamma_{12}} & \frac{\partial\Delta\sigma_{31}}{\partial\Delta\gamma_{23}} & \frac{\partial\Delta\sigma_{31}}{\partial\Delta\gamma_{31}}\end{bmatrix}$$

Equation 4

G and λ are the Lame's Parameters and are expressed in terms of Young's Modulus 'E' and Poisson's ratio 'ν' as Equation 5 and 6.

$$\lambda=\frac{E\nu}{(1+\nu)(1-2\nu)}$$

Equation 5

$$\mu=G=\frac{E}{2(1+\nu)}$$

Equation 6

Material Jacobian for Elastic region can be formulated using the Equation 3 directly using elastic properties E and ν of the material with the help of Equations 5 and 6. However Material Jacobian for plastic region needs to be obtained by using Equation 4. Proposed method makes use of Material data of uniaxial tensile test to evaluate expressions in Equation 4. The direct elements of Material Jacobian matrix expressing gradient of normal stress with corresponding normal strain (viz. $\frac{\partial\Delta\sigma_{11}}{\partial\Delta\varepsilon_{11}}$, $\frac{\partial\Delta\sigma_{22}}{\partial\Delta\varepsilon_{22}}$, $\frac{\partial\Delta\sigma_{33}}{\partial\Delta\varepsilon_{33}}$) are derived directly by numerically processing material test data. Other elements of matrix are determined by employing analogy between elastic stiffness matrix (Equation 3) and generalized form (Equation 4). Initially expressing Lame's parameters in terms of incremental partial derivative (or Tangent Stiffness) at given uniaxial strain and Poisson's ratio as Equations 7 and 8.

$$\lambda=\frac{\nu}{(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)$$

Equation 7

$$\mu = G = \frac{1-2\nu}{2(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)$$

Equation 8

Then assuming 'ν', the Poisson's ratio of the material to be constant throughout the elastic and plastic range of deformation, using Equation 7 and 8 in Equation 3 the complete Material Jacobian is composed as Equation 9

$$\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right) = \begin{bmatrix} \left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{11}} & \left\{\frac{\nu}{(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{22}}\right\} & \left\{\frac{\nu}{(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{33}}\right\} & & 0 & \\ \left\{\frac{\nu}{(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{11}}\right\} & \left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{22}} & \left\{\frac{\nu}{(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{33}}\right\} & & & \\ \left\{\frac{\nu}{(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{11}}\right\} & \left\{\frac{\nu}{(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{22}}\right\} & \left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\varepsilon_{33}} & & & \\ & & & \frac{1-2\nu}{2(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\gamma_{12}} & 0 & 0 \\ & 0 & & 0 & \frac{1-2\nu}{2(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\gamma_{23}} & 0 \\ & & & 0 & 0 & \frac{1-2\nu}{2(1-\nu)}\left(\frac{\partial\Delta\sigma}{\partial\Delta\varepsilon}\right)_{\gamma_{31}} \end{bmatrix}$$

Equation 9

Using Equation 9 the complete Material Jacobian for inelastic response can be composed using Experimental Material data and Poisson's ratio.

2.2 Implementation using UMAT

UMAT is a subroutine available in Abaqus that is used to model user defined mechanical behavior of material (Ref: Abaqus User Subroutines Reference Guide). The Strain and Stress State tensors are passed to the subroutine as VOIGT vectors designated by arrays 'STRAN' and 'STRESS' respectively. The subroutine is programmed to check the give stress state against yield criteria and calculate DDSDDE using Equation 3 for elastic range or Equation 9 post yielding. When using Equation 3 subroutine makes use of Material properties E and ν, entered through Abaqus/CAE or using INPUT (*.inp) file and stored in array PROPS. When using Equation 9 to construct DDSDDE for plastic region, subroutine interacts with pre-processed material data file containing Incremental partial derivatives of normal Strain with normal strain (Tangent Stiffness) values corresponding to true strain values. Using tangent stiffness value for a given strain and a Poisson's ratio ν, the complete Material Jacobian matrix DDSDDE is composed.

Using DDSDDE, STRESS and an incremental Strain array DSTRAN; a modified STRESS vector is calculated and returned by subroutine UAMT to the main program.

A flow chart elaborating structure of UMAT is depicted as Figure 1.

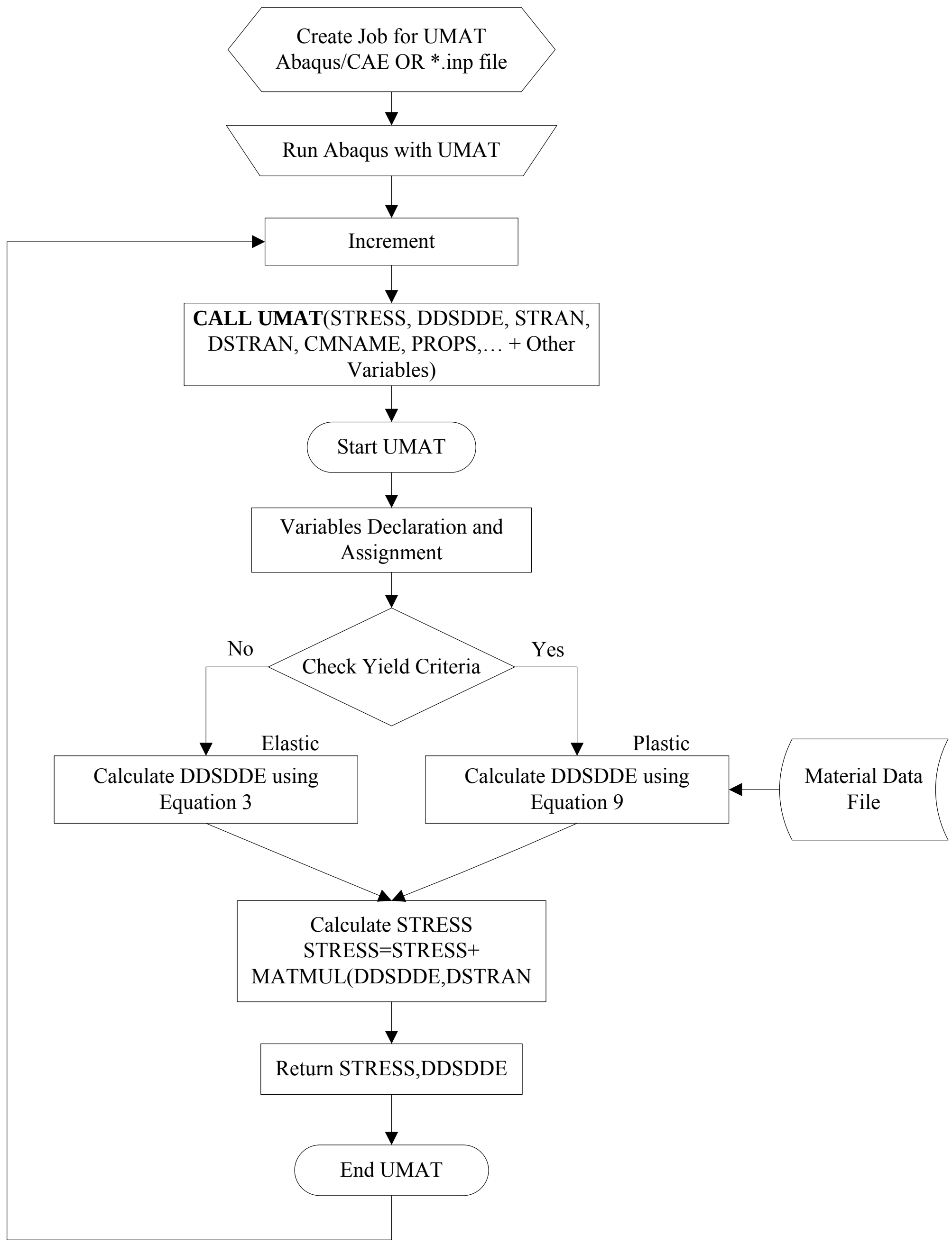

Figure 1. UMAT Flow Chart

3. Case Study

A Uniaxial tensile test is simulation is carried out using traditional isotropic hardening model available in Abaqus and proposed UMAT model both making use of same material data. The simulation results are compared with original Experimental results.

3.1 Experimental Material Data:

The material used in case study is characterized by True Stress-Strain Curve given by Figure 1.

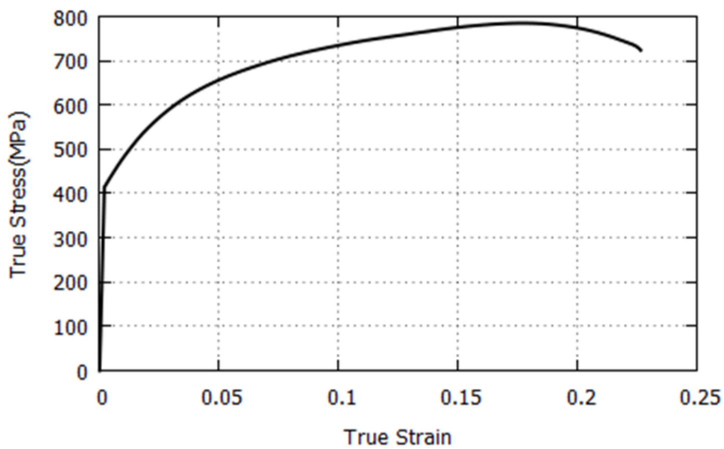

Figure 2. Stress-Strain Curve

3.2 Isotropic Hardening Model:

The Experimental Stress Strain Data is converted to True Stress Strain Data and is processed to obtain input data for Isotropic Hardening Model. Plastic Strain corresponding to each stress value post yield is calculated using Equation 10.

$$\varepsilon_{pl} = \varepsilon_{tr} - \frac{\sigma_{tr}}{E}$$

Equation 10

Where ε_{pl}= Plastic Strain, ε_{tr}= True Strain and σ_{tr}=True Stress

In-built isotropic hardening model for plasticity accepts yield stress v/s plastic strain data as tabular input. A Uniaxial tensile test simulated for this material model.

3.3 User-Defined Material Model (UMAT):

The Experimental Stress Strain Data converted to True Stress Strain Data. True Stress Strain data is processed to obtain $\Delta\sigma$ v/s $\Delta\varepsilon$ data for each true strain value. Tangent stiffness $\frac{\partial \Delta\sigma_{11}}{\partial \Delta\varepsilon_{11}}$ corresponding to true strain ε_{11} is evaluated by numerically differentiating $\Delta\sigma$ v/s $\Delta\varepsilon$ data for ε_{11}.

Forward finite divided difference formula rearranged in terms of σ and ε given by Equation 11 is used for Numerical differentiation.

$$\left(\frac{\partial \Delta\sigma}{\partial \Delta\varepsilon}\right)_{at\ \varepsilon=\varepsilon_{i-1}} = \frac{(\sigma_{i+1} - \sigma_i)}{(\varepsilon_{i+1} - \varepsilon_i)}$$

Equation 11

The available data is fully processed to obtain tangent stiffness $\frac{\partial \Delta\sigma_{11}}{\partial \Delta\varepsilon_{11}}$ for each uniaxial true strain value (ε_{11}). The variation of Tangent Stiffness with true strain for plastic region is characterized by Figure 3.

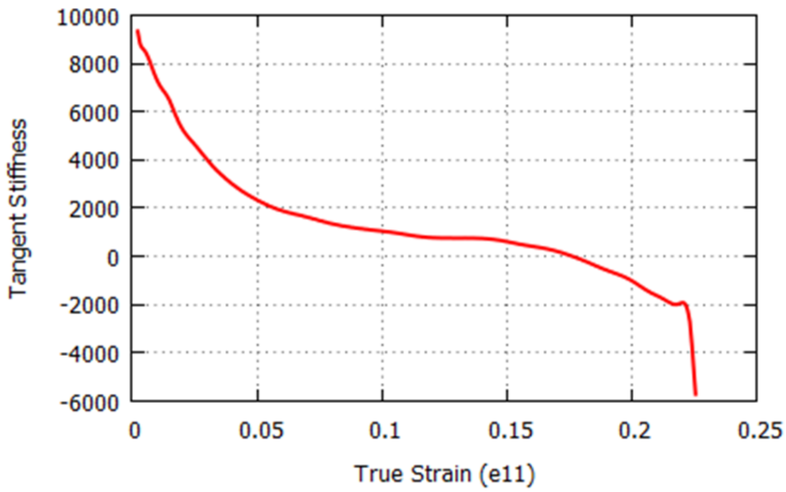

Figure 3. Variation of Tangent Stiffness with Strain

UMAT composes DDSDDE using Equation 3 for Elastic region and this preprocessed data is used to formulate DDSDDE using Equation 9 for simulation post yielding as described in section 2.2. A uniaxial tensile test is simulated using UMAT material model.

4. Results and Conclusion

The results of simulation with isotropic hardening model (3.2) and UMAT implementation (3.3) obtained in terms of stress-strain curve. The simulation results of 3.2 and 3.3 are overlapped on the results of experimental material data as shown in Figure 4. It can be clearly observed that the results of UMAT Simulation very closely follow the experimental trend than Isotropic hardening model.

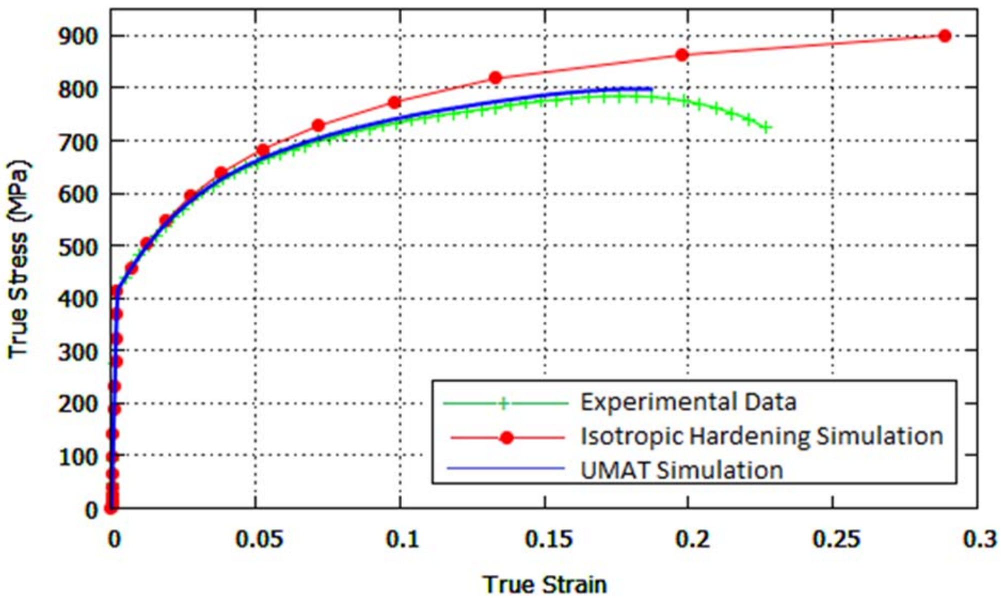

Figure 4. Comparison of the Simulation results with Experimental data

The present work concludes that the proposed method provides more realistic and accurate simulation results than existing traditional material modeling techniques.

5. References

1. Fionn Dunne and Nik Petrinic, “Introduction to Computational Plasticity”, Oxford University Press, 2005.
2. Dusan Krajcinovic, “Damage Mechanics”, Mechanics of Materials, 1989.
3. Fabrice Morestin and Maurice Boivin, “On the necessity of taking into account the variation in the Young modulus with plastic strain in elastic-plastic software”, Nuclear Engineering and Design, 162, 107-116, 1996.
4. Hai Yan Yu, “Variation of elastic modulus during plastic deformation and its influence on springback”, Materials and Design, 30, 846–850, 2009.
5. M. Yang, Y. Akiyama and T. Sasaki, “Evaluation of change in material properties due to plastic deformation”, Journal of Materials Processing Technology, 151, 232–236, 2004.
6. Marko Vrh, Miraslav Halilovic and Boris Stok, “Impact of Young's Modulus Degradation on Springback Calculation in Steel Sheet drawing”, Journal of Mechanical Engineering 54,4, 288-296, 2008.

7. Mohammad Abdel-Karim, “Effect of elastic modulus variation during plastic deformation on uniaxial and multiaxial ratcheting simulations”, European Journal of Mechanics A/Solids, 30 11-21, 2011.
8. Abaqus User Subroutines Reference Guide, Abaqus Documentation, Version 6.13, Dassault Systèmes Simulia Corp., Providence, RI.