Determination of Lemaitre damage parameters for DP590 steel using Teacher-Learner based optimization

Ashwani Verma and Ravindra K. Saxena
Department of Mechanical Engineering, Sant Longowal Institute of Engineering and Technology, Longowal-148106, Sangrur, Punjab, INDIA
E-mail: rksaxena04@yahoo.com

Abstract. Dual phase (DP) steel is widely used in high strength applications. The components or parts sometimes damage under the applied load during the production stage or the service stage. The damage behaviour of material under such loading conditions need to be analysed. The finite element analysis of the component and/or material incorporating the damage model is a useful step. The determination of parameters of the damage model envisages the application of such models into the numerical analysis. The present work proposes a model to determine the parameters of the Lemaitre damage model using the “Teacher-Learner based optimization” (TLBO) algorithm and the finite element method. The model simulates the tensile test and predicts the flow curve till fracture to determine the optimum set of parameters for the Lemaitre damage model. The proposed model incorporating uncoupled TLBO algorithm and finite element method is a general model and can be effectively employed to determine the values of the unknown parameters for the model.

Keywords: Dual phase steel; Teacher-Learner based optimization; Lemaitre damage model; Finite element method.

1. Introduction
Automotive sector demands the suitable material for mass reduction. It is becoming an important and significant factor for the large scale and effective production. The newly developed material viz. advanced high strength steel showed excellent properties for use in this sector [1]. Dual phase (DP) steel is one of the materials from the large set of advanced high strength steel. DP steels have excellent mechanical properties suitable for such application viz. high strength along with good ductility. It is found that most of these structural components are prone to failure under different loading conditions. Therefore, the failure initiation and damage mechanisms are considered as important areas to investigate the performance of these structural parts. The modeling of engineering structures using numerical modeling techniques is widely adopted for estimating the performance of component or machinery in actual service conditions. Finite element modeling procedures requires the characterization of the material and mathematical modeling of the deformation behavior under the applied load.

The determination of parameters for the identification of damage or failure in the component during the analysis using finite element methods is a challenging task. The material is generally tested in uniaxial loading case showing the complex deformation behavior upto failure. Manguiera et al. [2] attempted to determine the damage parameters directly from the material properties. They developed the equation for the material deformation in plane-stress and uniaxial tension state using the concept
proposed by Comi and Perego [3]. The damage parameters were identified using the maximum strength of the material observed during the hardening process.

[Flowchart]

- Initialize input parameters viz. class strength, elite size and number of subject
- Extract defined elite solution
- Find the mean of each subject
- Find the best solution and the score of each student
- Modify values of each student based on the best solution
  \[ X_{j,k,i} = X_{j,k,i} + \eta \left( X_{j,k,best,i} - T_{j,M_{j,i}} \right) \]
- Is result corresponding to \( X_{j,k,i} \) better than the result corresponding to \( X_{j,k,i} \)?
  - Yes: Replace the earlier values
  - No: Keep the earlier values
- Randomly select two solutions \( X_{total-P,j} \) and \( X_{total-Q,j} \)
- Is result corresponding to \( X_{total-P,j} \) better than the result corresponding to \( X_{total-Q,j} \)?
  - Yes: Replace the earlier values
  - No: Keep the earlier values
- Replace worst solutions with extracted elite solutions
- Satisfy convergence criterion?
  - Yes: Solution achieved!
  - No: Continue

Figure 1. Detailed flow Chart for elitist-TLBO [12]

West et al. [4] adopted the “Gurson-Tvergaard-Needleman” (GTN) model [5] for the prediction of damage behavior of the deforming material. They performed a number of experimental tests along
with the numerical simulation on representative volume element (RVE) for determining values of parameters for the damage model. The micromechanical approach was employed to determine these parameters for DP steel. They simulated a number of tensile tests using numerical modelling and the GTN model incorporating the identified damage parameters to predict the location of damage point under the influence of stress triaxiality, equivalent plastic strain, and Lode angle parameter. Gruben et al. [6] adopted the optical measurement techniques to determine the damage parameters. They investigated damage behavior within the uniaxial tension to the equibiaxial tension using Marciniak–Kuczynski and Nakajima tests [7] for DP steel. Ruzicka et al. [8] elaborated the calibration process for uncoupled Johnson–Cook damage model and Rice–Tracey damage model [9]. The calibration procedure adopted the tensile tests on notched bar, small punch tests and double curvature (butterfly) tests loaded in different directions. The process was simulated on ABAQUS finite element software for tensile tests to investigate the suitability. Netrasiri et al. [1] adopted “Micro-crack Initiation Criterion”. The criterion is based on “ductile crack initiation locus” due to the damage condition of material. The GTN model [5] was adopted for the determination of parameters of damage model. The damage curve was developed using the results from the tensile tests of different sample geometries using the numerical modelling and experiments for verification. Džugan et al. [10] determined the parameters of damage model using the experimental procedure and comparison with the corresponding finite element simulation. They adopted the damage models of Johnson–Cook, Rice–Tracey [9] and Bai–Wierzbicki [11] for pressure vessel steel for the determination of parameters using the experimental procedures.

The determination of parameters of the damage model is generally determined from the uniaxial tensile tests. In this work, an approach for determining the parameters of damage model using the uncoupled finite element procedure and optimization algorithm is proposed. The “Teacher-Learner based optimization” (TLBO) algorithm is adopted for the same as it does not require any algorithm specific parameters. The specification of common input parameters viz. population size, elite size and convergence criterion etc. is sufficient to apply the TLBO algorithm for analysis. The objective function for the optimization algorithm is specified to minimize the error between the uniaxial curve obtained from experimental observation from literature and the curve predicted from the finite element tensile test simulation. The simulated curve is obtained from the damage coupled large deformation non-linear finite element analysis. For the applicability of the proposed model, an inhouse uncoupled non-linear FE code is applied in association with the TLBO optimization code.

2. Teacher-Learner based Optimization
The “Teacher–Learner-based optimization” algorithm proposed by Rao [12] is widely adopted for solving engineering problems. The advantage of using TLBO is that it does not require any algorithm specific parameters and requires only the general input parameters viz. population size, elite size, and convergence criterion etc. The details of the formulation and implementation procedure of the algorithm are given in Rao [12]. The detailed flow chart for the algorithm of elitist-TLBO [12] is given in figure 1.

3. Finite element formulation
The finite element (FE) equations are generated using the equilibrium equations [13]. The “Logarithmic strain” is used in the present work which is obtained from the principal values of the “Right Stretch Tensor”. The corresponding principal axes directions are used for stress updation to maintain the “objectivity” of the stress in large deformation problems [14]. For the material undergoing plastic deformation the incremental stress-strain relationship for elastic-plastic stage is expressed as

\[ t\Delta \sigma_{ij} = t'c_{ijkl}^e t\Delta \varepsilon_{kl}^\epsilon \]  

(1)

The tensor \( t'c_{ijkl}^e \) is the fourth order constitutive tensor for elastic-plastic stage at time \( t \) and is given by
\[
C_{ijkl}^{EP} = 2\mu \left( \delta_{ik}\delta_{jl} + \frac{\nu}{1-2\nu} \delta_{ij}\delta_{kl} \right) - \frac{9\mu\sigma_{ij}^\prime\sigma_{kl}^\prime}{2\left(3\mu + H^\prime\right)} \left(1 - D\right) 
\]

(2)

where, \(\mu\) is Lamé’s constant, \(\delta\) is Kronecker delta, \(\nu\) is Poisson’s ratio, \(\sigma_{ij}^\prime\) is the deviatoric part of Cauchy stress, \(\sigma_{eq}\) is the von-Mises equivalent stress, and \(H^\prime\) is the hardening slope obtained from the material flow curve. Further, \(D\) is the damage value proposed by Lemaitre and Desmorat [15]. The total value of damage is updated in each increment as \(D = D + \Delta D\), where the relationship for the incremental damage value \(\Delta D\) is given by [15].

\[
\Delta D = (cD\Delta t) \frac{R_v}{\varepsilon_p - \varepsilon_o} \Delta \varepsilon_{eq}^p 
\]

(3)

where, \((cD)\) is the “critical value of damage”, \(\varepsilon_p\) is the assumed “equivalent plastic strain” at the initiation of fracture, i.e., when the total damage value reaches the critical value, \(\varepsilon_o\) is the assumed “threshold value”, \(\Delta t\) is the “triaxiality function”, and \(\Delta \varepsilon_{eq}^p\) is the “equivalent plastic strain” at time \(t\) [14]. Radial backward return algorithm is used for iterative calculation of the incremental stress tensor in Eq. (1).

The integral form of equilibrium equation is given by the following virtual work expression [13]:

\[
\int_{t}^{t+\Delta t} \sigma_{ij} \delta \left( t + \Delta \varepsilon_{ij} \right) d\tau = \int_{t}^{t+\Delta t} R_v d\tau = \int_{t}^{t+\Delta t} R_v d\varepsilon_{eq}^p 
\]

(4)

Here, \(\int_{t}^{t+\Delta t} R_v d\varepsilon_{eq}^p\) is the virtual work of the external forces and \(\int_{t}^{t+\Delta t} \sigma_{ij} \delta \left( t + \Delta \varepsilon_{ij} \right) d\tau\) is the virtual linear strain tensor corresponding to the virtual displacement vector \(\delta^t u_j\) at time \(t + \Delta t\). The finite element equations are developed from Eq. (4) [14] after suitable approximation. The substitution of elemental representation in terms of shape function for the assumed finite element and the assembly over all the finite elements, leads to the following form of algebraic equation in terms of global values [14]:

\[
[K] \{\Delta u\} + \{f\} = \{F\} 
\]

(5)

Here, \(\{\Delta u\}\) is the “Displacement Vector” and \(\{f\} \) is the “Coefficient Matrix”, \(\{F\}\) is “Internal Force Vector” and \(\{F\}\) is “External Force Vector”. The solution of Eq. (5) represents only an approximate solution to the governing equations in Eq. (4), because of the linearization and approximation involved [14]. To minimize the error of the approximating solution, the modified Newton-Raphson algorithm [13] is used. The details of the mathematical formulation for the finite element analysis are given in Saxena and Dixit [14].

4. Mathematical formulation

The general optimization procedure requires the objective function depicted in a clear mathematical form derived from the problem statement. The material subjected to uniaxial loading case follows a typical “load-displacement” curve. The “load-displacement” curve is converted to engineering stress and engineering strain curve using the procedures given in the standard book [16]. The curve shows the decrease in trend for engineering stress after the maximum value during the plastic deformation and follows a decreasing trend till fracture. It is generally assumed that the cavities formed after this maximum stress and necking is observed. These cavities coalesce and ultimately the part separates into two pieces at the fracture point. The objective of the finite element simulation of the process is to get the respective flow curve for the deforming material under uniaxial tensile loading. The “load-
displacement” curve is obtained from the simulation. It is assumed that the “load-displacement” or the resulting engineering stress-engineering strain curve so obtained should follow the same trend till fracture. In that view, the objective function for the present case is depicted as

$$Obj = \min \left[ \text{abs}\left( (t^\Delta F_{FE} - t^\Delta F_{Exp}) \left( t^\Delta X - t^X \right) \right) \right]$$  \hspace{1cm} (6)$$

where, $t^\Delta F_{FE}$ is the value of load value obtained from FE simulation, $t^\Delta F_{Exp}$ is the value of load value from experiment, $t^\Delta X$ is displacement of the tensile specimen at time $t + \Delta t$ and $t^X$ is the displacement of the specimen at time $t$. The non-linear large-deformation finite element code is operated to run in coordination with the TLBO algorithm. The uncoupled numerical program to simultaneously run the code for TLBO algorithm and the code for the finite element analysis is executed on MATLAB [17].

![Dog Bone tensile specimen for finite element simulation as per ASTM E8](image1.png)

**Figure 2. Dog Bone tensile specimen for finite element simulation as per ASTM E8**

![Finite element mesh (color picture is available in online version)](image2.png)

**Figure 3. Finite element mesh (color picture is available in online version)**

### 5. Results and Discussion

The parameters in equation (3) need to be determined using the proposed model. The parameters $(cD), \varepsilon_p, \varepsilon_o$ are the unknown parameters. The present work, gives an outline of a proposed model to determine these parameters using uncoupled TLBO algorithm and finite element analysis of the tensile test. The experimental tensile test flow curve for DP590 steel is given in Paul [18]. The respective material properties of DP590 steel are taken from Paul [18]. The simulation is performed on Dog-Bone type tensile specimen as per ASTM E8 standard [19]. The schematic of the Dog bone tensile specimen is given in figure 2. The analysis is performed on $1/8^{th}$ of the sample and the respective FE mesh is given in figure 3. The problem is solved as a displacement-based problem with a small incremental displacement specified on one side of the finite element mesh. The load value is derived from the load end of the mesh and is converted into the engineering stress and incremental displacement value is converted into engineering strain. The finite element model solves the tensile test problem in each of TLBO step and the algorithm of TLBO evaluates the objective function given
in equation (6) for minimization and to get the optimum values for the unknown damage parameters viz. \((cD), \varepsilon_p, \varepsilon_o\).

The final value of the parameters obtained from the given optimization run as per the proposed model are \((cD) = 0.55, \varepsilon_p = 0.85, \text{and} \varepsilon_o = 0.159\) and the respective plot of the engineering stress and engineering strain from the present finite element work is given in figure 4. The experimental stress-strain curve [18] for DP590 steel is also given in figure 4 for comparison. It is observed that there is minimal difference in the curves upto fracture. The deformed mesh for the Dog-Bone specimen from the present FE work at the point of fracture is given in figure 5. It is found that the proposed model can effectively predict the value of the parameters for the damage model effectively.
6. Conclusions
The components or parts subjected to the large deformation loading conditions exhibits the phenomena of ductile damage. The numerical models require the incorporation of suitable damage evolution model into the analysis. Lemaitre damage evolution model is widely used for the prediction of fracture of ductile materials under applied loading. The determination of the parameters for the damage model is discussed in the present work. An uncoupled “Teacher-Learner based optimization” is adopted to find the values of the damage parameters in-coordination with the large deformation elasto-plastic FE method. The model is able to find the values of the parameters with a reasonable accuracy based on the stated objective function. The proposed model is general in nature and can be effectively adopted for such parameters for the other widely used models.

References
[1] Netrasiri C, Suranantachai S, Uthaisangsu V and Jirathearanat S 2015 Damage Curve Determination of Dual-Phase Steel Based on Micromechanical Damage by Using Numerical Analysis and Experimental Investigation. *Int Conference on Advances in Mechanical, Material & Automobile Engineering (ICAMMAE’2015)* June 18-19, Pattaya (Thailand).
[2] Manguereira FB, Oliveira Neto LD and Azambuja MD 2012 Determination of the Numerical Parameters of a Continuous Damage Model for the Structural Analysis of Clay Brick Masonry. *Materials Research 15(6)* 1013-21.
[3] Comi C and Perego U 2001 Fracture energy based bi-dissipative damage model for concrete. *Int. J. Solids and Structures 38(36-37)* 6427-54.
[4] West O, Lian J, Münstermann S and Bleck W 2012 Numerical Determination of the Damage Parameters of a Dual-Phase Sheet Steel. *ISIJ international 52(4)* 743-52.
[5] Needleman A and Tvergaard V 1987. An Analysis of Ductile Rupture Modes at a Crack Tip. *J. Mech. Phys. Solids 35(2)* 151-83.
[6] Gruben G, Vysochinskii D, Coudert T, Reyes A and Lademo OG 2013 Determination of Ductile Fracture Parameters of a Dual-Phase Steel by Optical Measurements. *Strain, 49(3)* 221-32.
[7] Marciniak Z and Kuczyński K 1967 Limit Strains in the Processes of Stretch-Forming Sheet Metal. *Int. J. Mech. Sci.* 9 609–20.
[8] Ruzicka J, Spaniel M, Prantl A, Dzugan J, Kuzelka J and Moravec M 2012 Identification of Ductile Damage Parameters in the Abaqus, *Bulletin of Applied Mechanics* 8 (32) 89-92.
[9] Rice JR and Tracey DM, 1969 On the ductile enlargement of voids in triaxial stress fields, *J. Mech. Phys. Solids 17* 201-17.
[10] Dzugan J, Španiel M, Prantl A, Konopik P, Růžička J and Kuželka J 2018 Identification of ductile damage parameters for pressure vessel steel. *Nuclear Engineering and Design* 328 370-80.
[11] Bai Y and Wierzbicki T 2010 Application of Extended Mohr–Coulomb Criterion to Ductile Fracture. *Int. J. Fracture* 161(1) 1-20.
[12] Rao RV 2016 *Teaching Learning Based Optimization Algorithm and Its Engineering Applications*. Springer. Switzerland.
[13] Bathe KJ 1996 *Finite Element Procedures*, Prentice Hall. USA.
[14] Saxena RK and Dixit PM 2010 Prediction of Flange Wrinkling in Deep Drawing Process Using Bifurcation Criterion. *J.Manuf Process.* 12 19-29.
[15] Lemaitre J and Desmorat R 2005 Engineering Damage Mechanics. Springer-Verlag, Berlin Heidelberg.
[16] Dixit PM and Dixit US 2008 Modeling of Metal Forming and Machining Processes. Springer-Verlag, London.
[17] MATLAB and Statistics Toolbox Release 2017a, The MathWorks, Inc., Natick, Massachusetts, United States.
[18] Paul SK 2013 Real Microstructure Based Micromechanical Model to Simulate Microstructural
Level Deformation Behavior and Failure Initiation in DP 590 Steel. *Materials & Design* **44** 397-406.

[19] ASTM Standard E8 2003 Standard Test Methods for Tension Testing of Metallic Materials
ASTM International. *West Conshohocken PA.*