Estimation of Material Model Parameters from Mixed Loading Test for Effective Simulation of Incremental Sheet Forming

Pavan Kumar, and Puneet Tandon, *Member, IAENG*

***Abstract*—The present work focuses on numerical analysis of Incremental Sheet Forming (ISF) process based on Johnson-Cook (JC) model. The objective of the present work is to improve the accuracy of Finite Element Analysis (FEA) simulation by acquiring material parameters from mechanical tests which describes mixed loading conditions. Literature reports that the simulation of ISF process is done with material parameters acquired from uniaxial tensile test, i.e., the test with dog bone specimen. However, ISF process is a combination of tension, bending and shear. For the current research, the material was tested in biaxial direction by fabricating the special shape of the specimen. Two loading condition, i.e., pure tension and combined (tension+shear) were considered. A truncated cone with 65.3° wall angle was used as a case study for carrying out the numerical simulation. The thickness distribution, plastic strain, and von-Mises stresses were analyzed and compared. A comparative analysis based on the two loading conditions of the material was made.**

***Index Terms*—Numerical Simulation, Finite element analysis, Incremental sheet forming (ISF), Mechanical testing of the material**

I. INTRODUCTION

Incremental sheet metal forming (ISF) is a promising technology that produces low volume sheet metal components in a flexible manner due to the lack of requirement of the part specific tooling. Implementation of ISF on a standard Computer Numerical control (CNC) milling machine allows the use of the existing machinery for the task, which aids its applicability and acceptability. The dexterity of the process is further augmented through validations provided by computer controlled pre-fabrication simulations. The finite element analysis (FEA) is one of the most important CAE tool, which helps in analyzing the forming process under any imaginable condition, thus, allowing the forming process to be further refined, prior to any real-time experiment. This reduces the number of experiments and hence, the lead time and cost to manufacture the products.

ISF is performed using a punch which impinges the sheet clamped along its outer periphery along negative Z direction and makes highly localized plastic deformation. Controlled by a CNC controller, the tool follows a path in the XY plane and incrementally moves in negative Z direction and drags deformation along with it. In the due course, this progression of incremental deformation results in the formation of desired contour or geometry on the sheet as shown in Fig. 1.

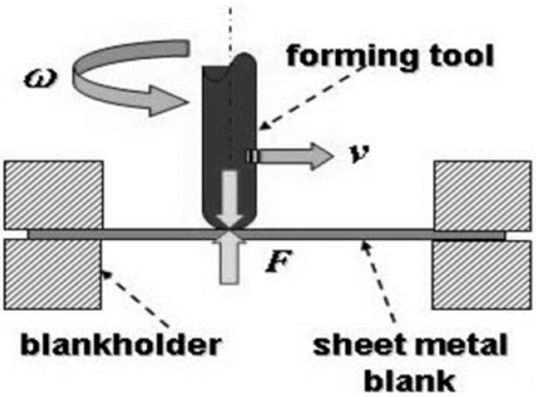


Fig. 1. Incremental sheet forming process

Finite element methods had been utilized by many researchers to analyze the damage evolution, deformation mechanism and forming forces in ISF process. Fang et al. [1] described the localized deformation mechanism of ISF by their analytical and experimental work**.** The results of their analysis suggested that the geometrical accuracy of the formed component would be affected by the deformation in non-contact zone. The material modelling for FEA of ISF is challenging due to the complexity of material behavior in ISF. Sahu and Tandon [2] used the Johnson-Cook model as material model for FEA of ISF because it is suitable for the elastic-plastic behavior of material. Their work also included the investigation of effect of different shell elements to analyze the ISF process. Their results show that for fine meshing Belytschko-Tsay (BT) shell element is appropriate, while the Improved Flanagan-Belytchko (IFB) shell element gives good result for coarse adaptive meshing. The fracture model used to describe the material behavior during forming was proposed by Xue [3]. The fracture criterion proposed by Xue was based on hydrostatic stress and Lode angle and the FEA results show good correlation with the real-time experiments. To predict the thinning, fracture depth and forming forces of ISF process, Malhotra et al. [4] utilized

Manuscript received May 12, 2017; revised June 07, 2017.

Pavan Kumar is with PDPM Indian Institute of Information Technology, Design and Manufacturing, Jabalpur, M.P. 482005 India (e-mail: pavan.kumar@iiitdmj.ac.in).

Puneet Tandon is with PDPM Indian Institute of Information Technology, Design and Manufacturing, Jabalpur, M.P. 482005 India (corresponding author phone: +91-761-279 4411; fax: +91-761-279 4411; e-mail: ptandon@iiitdmj.ac.in).

the fracture criterion proposed by Xue One of the famous model which is used by researchers for FE analysis of ISF is Gurson-Tvergaard-Needleman (GTN) and it is based on void nucleation, growth and coalescence. Li et al [5] implemented the GTN model integrated with Hill'48 anisotropic yield criterion in ABAQUS through user subroutine to simulate the ISF process. However, this model is limited to capture the effect of first and second invariants of stress on damage only. The effect of third invariant on ductile damage was incorporated by the Xue [6]. The Bao-Xue-Wierzbicki model [7] is a 3D locus fracture criterion having the advantage to model the combined effect of different loading (tension, shear, compression) conditions for effective finite element modelling of ISF.

Literature indicates that at least biaxial stress states is required to exactly estimate the damage and failure properties of the materials and formulating the related constitutive equation. However, due to the complexity of the testing apparatus and difficulty in the fabrication of specimen the experimental work currently employed for this purpose are limited. In the research work of Gning et al. [8], two specimens, Mat31 and Mat32, made of glass fibre reinforced were used to apply biaxial loading. In Mat31 specimen, fibres were parallel to notch-to-notch line and cracked in parallel to the notch-to-notch direction when load was applied. However, in Mat32, fibres were perpendicular to the plane of the specimen and fragmented into three different parts with two opposite cracks tending to propagate obliquely towards the grips when loaded under biaxial loading. Overall, Mat31 specimens were more resistant compared to Mat32 in terms of failure loads.

A modified Arcan fixture was examined for the nonlinear shear response of pultruded composites by El-Hajjar and Haj-Ali [9]. They proposed that this testing method could be used to determine the material shear stiffness and its non-linear stress-strain response up to ultimate stress under different biaxial test conditions. They reported that the designed specimen minimized the stress concentrations at the edges, and allowed for approaching the nominal stress at the center. An extensive three-dimensional parametric finite element analysis had been done to define the optimal shape of a new type of Arcan specimen having non-uniform thickness by Pucillo et al. [10]. The numerical simulation resulted in fracturing of the specimen, where the cross-section area was minimum, under a pure shear stress distribution. The results were more uniform compared to the available Arcan specimens.

Bao and Wierzbicki [11] studied a series of tests including upsetting tests, shear tests and tensile tests on 2024-T351 aluminum alloy to provide the clues on wide range of stress triaxiality. They performed numerical simulations of each test for a special configuration of the specimen. Good correlation was achieved between the experiments and numerical simulations. They concluded that for negative stress triaxialities, fracture is governed by shear mode, for large triaxialities, void growth is the dominant failure mode, while at low stress triaxialities between the two regimes, fracture may develop as a combination of shear and void growth modes.

Literature indicates that most of the simulations of ISF process are done with material parameters acquired from uniaxial tensile test. However, ISF is a combination of tension, bending and shear. Thus, the present work tries to simulate the ISF process in a more realistic manner by acquiring material parameters from mechanical tests which describes mixed loading conditions. The overall objective of the work is to improve the accuracy of FEA simulation.

II. METHODOLOGY

To obtain the material model parameters, the specimens were tested with two loading conditions, i.e., pure tension and combined loading incorporating both tension and shear. Based on the stress-strain curve for both the cases, Johnson-Cook material model parameters were estimated and used for the numerical simulation of ISF process.

A. *Mechanical Testing and Parameter Estimation*

Specimen of two different shapes were used to obtain the characteristic stress-strain curve under pure tension and combined loading (tension+shear). Fig. 2 shows the CAD model of the specimen under both loading conditions. The specimens were cut from the copper sheet of 2 mm thickness using Abrasive Water Jet machine for experiments and analysis.

(a) (b)

Fig. 2. Shape of the specimen; (a) Pure tension and (b) Tension+shear (combined)

To analyze the material behavior, uniaxial tensile testing machine Tinius Olesn H25KS was used (Fig. 3). Six specimens were tested with 0.45 mm/min strain rate.

Fig. 3. Tinius Olesn H25KS tensile test machine

For the case of pure tension, the specimen shape and size are shown with the help of Fig. 4. Similarly, for tension+shear (combined) condition, Fig. 5 shows the dimension of the specimen.

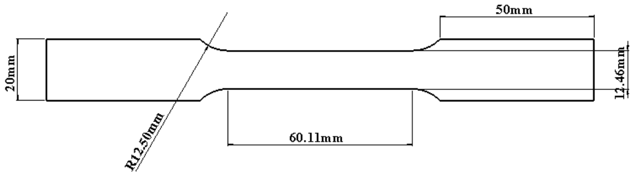


Fig. 4. Dimensions of the pure tension specimen

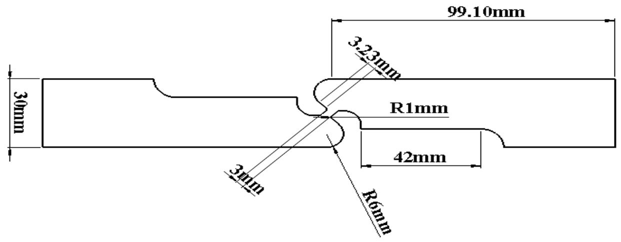


Fig. 5. Dimensions of the tension+shear specimen

The finite element analysis of ISF had been carried out based on Jonson-Cook model. The parameters which are required for material modeling can be classified under plasticity model parameters and damage/fracture model parameters. For the present case, only plasticity model parameters (i.e., strain hardening effect of Johnson-Cook model) were considered for numerical analysis of ISF process. The modulus of elasticity, density and Poisson's ratios for the materials were obtained from the material hand book. The calculation of other three parameters required to complete the plasticity model parameters, i.e., initial yield stress, hardening modulus or hardening coefficient and hardening exponent, involves the following steps:

- Convert experimental data into true stress-true strain form
- Subtract elastic part of the strain to get true stress–true plastic strain values
- Fit the curve from the acquired data
- Obtain hardening constants using nonlinear curve fitting technique

The flowchart for the estimation of plasticity model parameters is shown in Fig. 6. The value of yield stress (A) in ductile materials is not well defined. Usually, the yield stress in ductile material is defined as 0.2% offset strain. The yield stress at 0.2% offset was determined by finding the intersection point of a straight line and stress-strain curve.

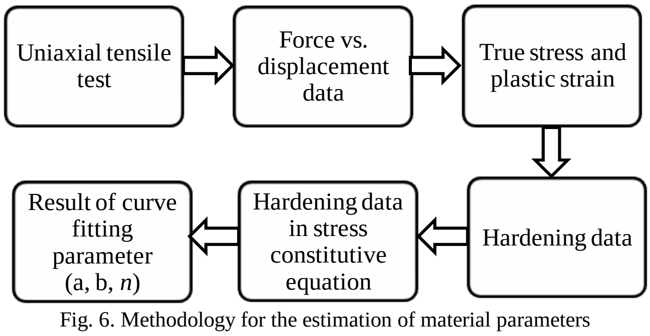


Fig. 6. Methodology for the estimation of material parameters

The true stress-strain curves based on the uniaxial tensile test for both the cases, i.e., pure tension and tension+shear (combined) are shown in Fig. 7 and Fig. 8.

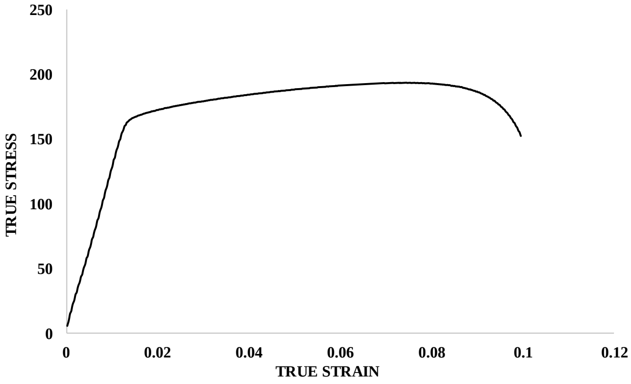


Fig. 7. True stress-strain curve for pure tension

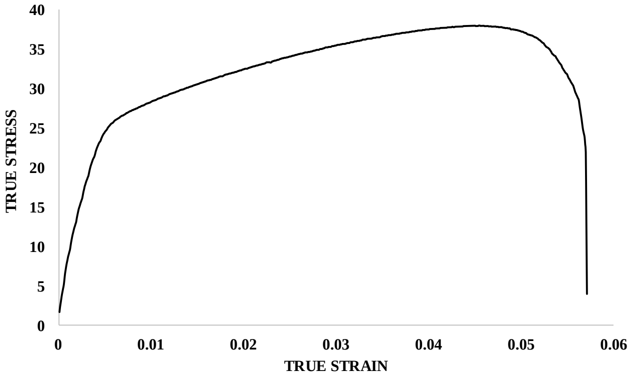


Fig. 8. True stress-strain curve for tension+shear

The other constants (B and n) were obtained by the curve fitting technique (power law equation) in MATLAB R2014a. The final values of plasticity model parameters for the two cases of pure tension and tension+shear (combined) are detailed in Table I.

TABLE I
MATERIAL PARAMETERS FOR PURE TENSION AND TENSION+SHEAR

Symbol	Quantity	Pure Tension	Tension +Shear
A	Yield stress (MPa)	167	28.033
B	Hardening modulus (MPa)	244.4	78.38
N	Hardening exponent	0.087	0.157
ε_p^{max}	Failure plastic strain (%)	9.81	5.88
σ^{max}	Plasticity maximum stress (MPa)	193.04	36.36
P	Density (kg/m^3)	8960	8960
E	Young's modulus (MPa)	117000	117000

B. *Numerical Modelling*

The accuracy of numerical analysis depends on many factors such as boundary conditions, meshing, material model, etc. The detailed finite element modelling of the present work is discussed below and shown in Fig. 9.

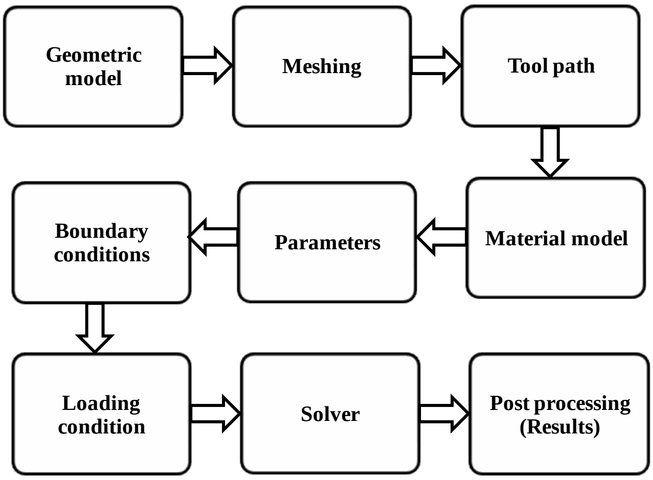


Fig. 9. Steps involved in FEA of ISF

The numerical modelling initiated with the modeling of hemispherical tool (punch) of 6 mm diameter and Cu sheet of 2 mm thickness in SOLIDWORKS 2016. The models were exported to HyperMesh 12.0 for finite element analysis. The sheet had been trimmed into five sections to refine the meshing of sheet at critical sections, as shown in Fig 10.

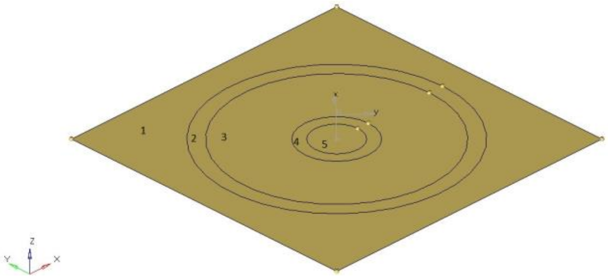


Fig. 10. Division of sheet in sub-sections

Coarse meshing had been done in Sections 1 and 5, because the deformation is negligible in these sections as compared to Section 3. Due to the possibility of large deformations at section 3, fine meshing had been done as shown in Fig. 11. As it has been suggested by Sahu and Tandon [2], the BT shell elements give good results for ISF simulation, this work employs BT shell elements only.

A frustum of cone of 65.30° wall angle had been chosen for numerical analysis of ISF process for both the loading conditions, i.e., pure tension and combined loading, i.e., tension and shear. The test specimen is shown in Fig. 12. For the selected geometry, a spiral toolpath was generated.

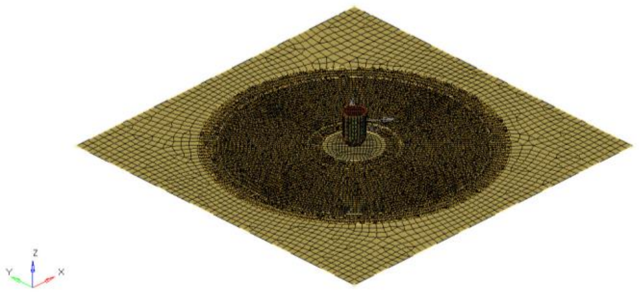


Fig. 11. Meshing of the sheet

The Johnson-Cook plasticity model was used as the material model for the present case. The nodes of the outer periphery of the sheet were restrained (all 6 degrees of freedom) to fix the sheet. However, the tool was only constrained with rotations in X and Y directions. To define contact, master (tool) and slave (sheet) algorithm was used. A master node of the tool was created to assign the tool path with imposed displacement load. The tool was made to revolve at 1000 rpm along Z axis. The boundary condition load collector (BCS) was used to fix the blank and movement of the tool [12]. For the present work, Radioss solver of HyperWorks 12.0 was used to solve the FEA model and for post-processing Hyperview was used.

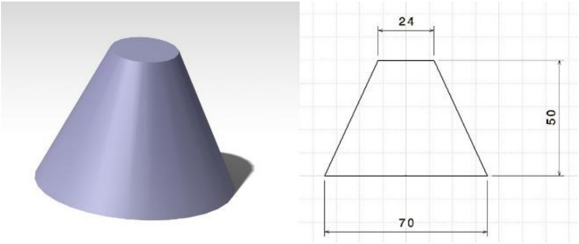


Fig.12. Frustum of cone and its dimension

III. RESULTS AND DISCUSSION

This section present the results of numerical analysis of ISF process based on Johnson-Cook model for the two loading conditions considered as part of this work, i.e., pure tension and tension + shear (combined). The results obtained from numerical analysis helps in predicting the thickness distribution, plastic strain and stress distribution. Further, a detailed comparative analysis of these response parameters helps in understanding the difference between the two cases of pure tension and combined loading condition.

A. Thickness Distribution

In sheet metal forming, product quality is mainly affected by the thickness distribution in the formed parts. In ISF, if the thickness reduction reaches a certain value then the chances of fracture may increase. To avoid fracture in real time experiment and to predict the thickness variation in the formed part, numerical simulation of the process using finite element analysis is carried out. The thickness variation of the truncated cone formed by the ISF process is shown with the help of Fig. 13 for pure tension and combined loading conditions. Pure tension based FE analysis predicts that the minimum thickness of the component wall for the truncated cone would be 0.7499 mm for the sheet having initial thickness of 2 mm. However, the minimum thickness obtained from FE model for the combined loading case came out to be 0.7796 mm for the same sheet.

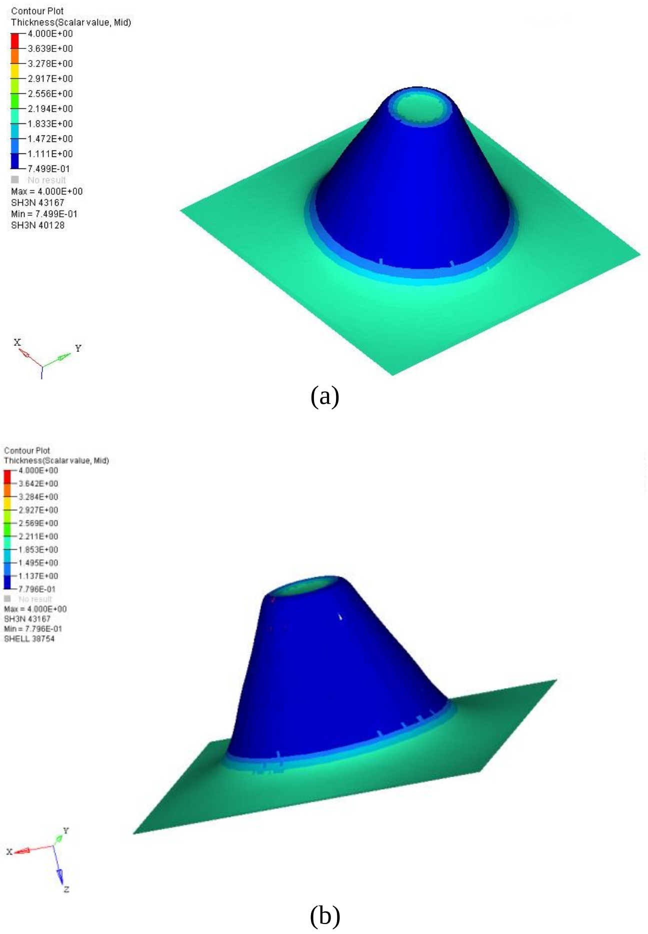


Fig. 13. Contour plot of thickness based on (a) pure tension, and (b) Combined Loading

To check the variation of thickness with the depth, the average thickness of the sheet in terms of elements (obtained as per meshing in FEA) at various depths is plotted for both the cases. The graph shows that both the cases predict similar thickness reduction during forming. Fig.14 shows the variation of thickness with depth for both the loading conditions.

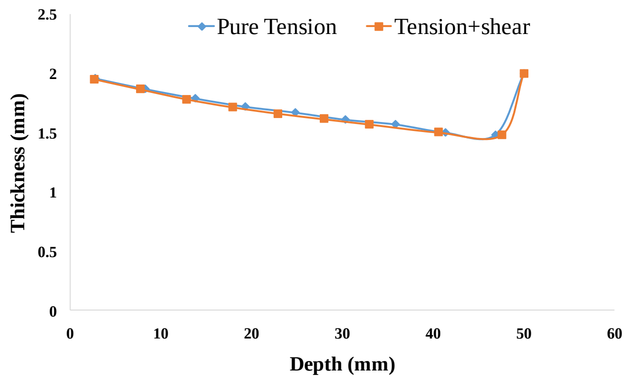


Fig. 14. Comparison of thickness variation with depth

B. Plastic Strain

In ISF process, the deformation being incremental in nature, the strain in the sheet increases with the depth of the formed component. In this work, the numerical value of plastic strain is calculated by taking an average of the finite elements at a depth. Fig. 15 shows the variation of plastic strain with the depth of the formed component. It was observed that the value of plastic strain increases with the increase in depth. The graph shows similar behavior for both the loading conditions but the amount of plastic strain in the case of tension+shear case is more in comparison to pure tension condition.

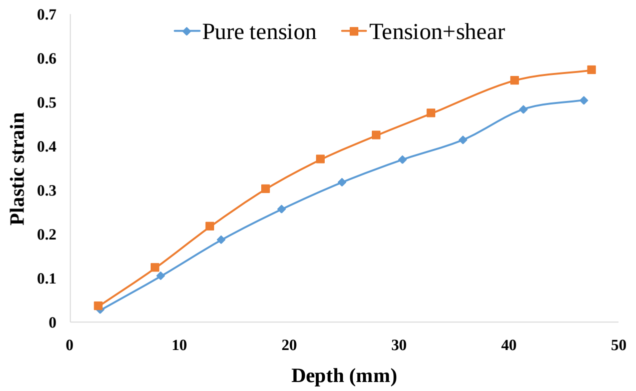


Fig. 15. Variation of plastic strain with depth

Fig. 16 shows the contour plot for plastic strain for pure tension and tension+shear conditions. From the plots, it is evident that the value of plastic strain in the middle of the part is more for both the cases.

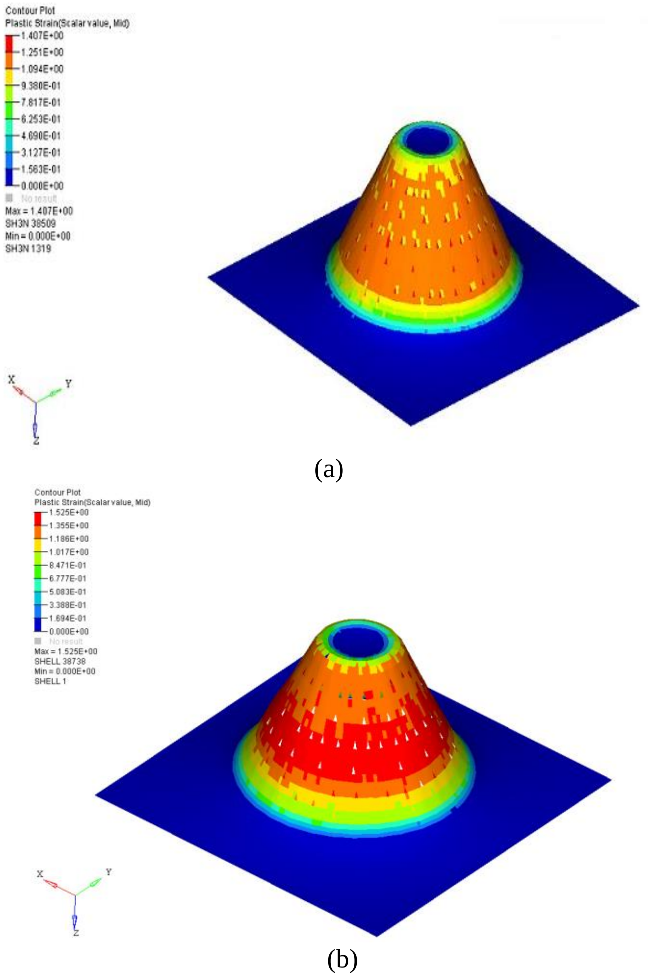
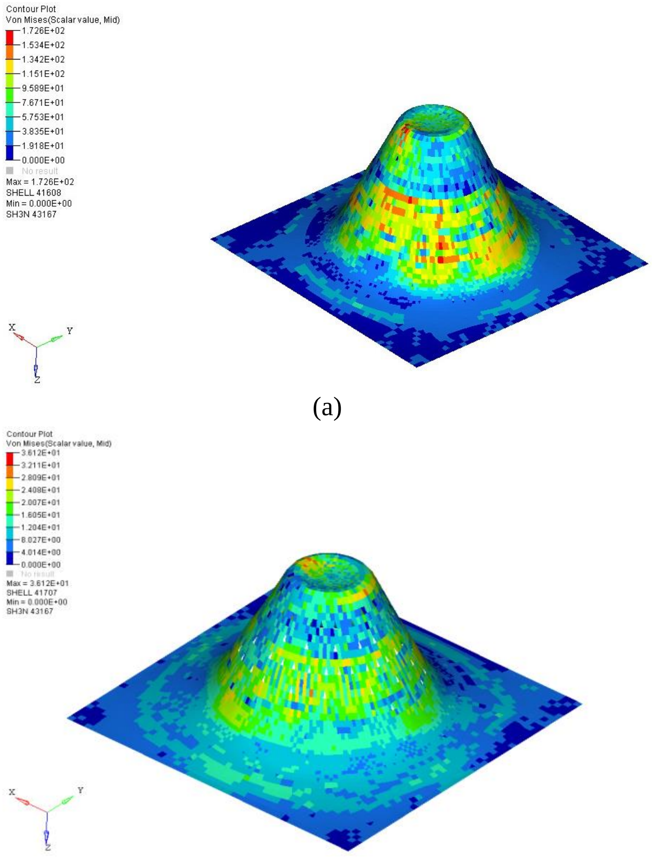


Fig. 16. Contour plot of plastic strain based on (a) pure tension, and (b) tension+shear (combined)

C. Stress Distribution

In a typical incremental sheet metal forming process, the major deformation occurs under plane stress condition, where the third stress, normal to the sheet is negligible. von-Mises yield contour plot has been used to depict the stress distribution in the formed component, which in turn shows the yielding in a material under deformation. Fig. 17 displays the von-Mises stress distribution in the formed component for both the cases. From the plot, it can be observed that the value of von-Mises stress predicted by tension+shear condition is more as compared to pure tension condition.



(a)

(b)

Fig. 17. Contour plot of von-Mises stress distribution based on; (a) pure tension, and (b) tension+shear (combined)

IV. CONCLUSION

The present work is devoted to the estimation of Johnson-Cook material model parameters and numerical analysis of ISF process. The overall objective is to improve the accuracy of numerical analysis. Experiments had been carried out to acquire the material model parameters and based on these parameters effective simulation of ISF had been performed for the two loading conditions and results were compared. The conclusions drawn from this work are listed below:

- The constitutive equation of Johnson-Cook model consists of three components, namely, strain hardening, strain rate, and temperature effect. Current work tries to analyze the effect of strain hardening for this material model.
- The required material parameters for this strain hardening effect had been identified by carrying out biaxial mechanical testing of specimen on uniaxial tensile testing machine.
- Numerical simulation of ISF had been carried out based on the material parameters for the cases of pure tension and combined (tension+shear) conditions.
- The results obtained through numerical simulation of ISF had been compared for three factors, i.e., thickness distribution, plastic strain and stress distribution.
- The comparison made based on three response parameters indicates that the combined (tension+shear) loading condition predicts higher plastic strain and von-Mises stress as compared to pure tension condition. This is because ISF is the process which includes tension, shear and bending simultaneously.

REFERENCES

[1] Y. Fang, B. Lu, J. Chen, D. K. Xu, H. Ou, "Analytical and experimental investigations on deformation mechanism and fracture behavior in single point incremental forming," *Journal of Materials Processing Technology*, Vol. 214, no. 8, Aug 2014, pp.1503-1515.

[2] G. N. Sahu, and P. Tandon, "Development of Finite Element Analysis Methodology for Incremental Sheet Forming Process," *M.Tech. Thesis*, Mechanical Engineering, PDPM IIITDM, Jabalpur, 2015.

[3] L. Xue, "Damage accumulation and fracture initiation in uncracked ductile solids subject to triaxial loading," *International journal of solids and structures*, vol. 44, no. 16, Aug 2007, pp. 5163-5181.

[4] R. Malhotra, L. Xue, T. Belytschko, J. Cao, "Mechanics of fracture in single point incremental forming," *Journal of Materials Processing Technology*, vol. 212, no. 7, Jul 2012, pp. 1573-1590.

[5] J. Li, S. Li, Z. Xie, W. Wang, "Numerical simulation of incremental sheet forming based on GTN damage model," *The International Journal of Advanced Manufacturing Technology*, vol. 81, Dec 2015, pp. 2053-2065.

[6] L. Xue, "Constitutive modeling of void shearing effect in ductile fracture of porous materials;" *Engineering Fracture Mechanics*, vol. 75, no. 11, Jul 2008, pp. 3343-3366.

[7] RADIOSS Theory Manual, "Large displacement finite element analysis," *HyperWorks*, 2012.

[8] P. B. Gning, D. Delsart, J. M. Mortier, D. Coutellier, "Through-thickness strength measurements using Arcan's method," *Composites Part B: Engineering*, vol. 41, no. 4, Jun 2010, pp. 308-316.

[9] R. El-Hajjar, R. Haj-Ali, "In-plane shear testing of thick-section pultruded FRP composites using a modified Arcan fixture," *Composites Part B: Engineering*, vol. 35, no. 5, Jul 2004, pp. 421-428.

[10] G. P. Pucillo, M. Grasso, F. Penta, P. Pinto, "On the mechanical characterization of materials by Arcan-type specimens," *Engineering Fracture Mechanics*, vol. 78, no. 8, May 2011, pp. 1729-1741.

[11] Y. Bao, and T. Wierzbicki, "On fracture locus in the equivalent strain and stress triaxiality space," *International Journal of Mechanical Sciences*, vol. 46, no. 1, Jan 2004, pp. 81-98.

[12] RADIOSS Reference Guide, "RADIOSS block format reference guide," *HyperWorks*, 2012.