

## Finite Element Analysis of Sheet Metal Forming Process

**Hakim S. Sultan Aljibori**

*Mechanical Engineering Department, Faculty of Engineering  
University Malaya, 50603, Kuala Lumpur, Malaysia*

E-mail: [hakimss@um.edu.my](mailto:hakimss@um.edu.my)

Tel: 60-37967622; Fax: 603-79675317

**Abdel Magid Hamouda**

*Mechanical and industrial Engineering Department  
Faculty of Engineering, Qatar University, Doha, Qatar*

### Abstract

Minimization of response times and costs and maximization of the efficiency and quality in producing a product are imperative for survival in the competitive manufacturing industry. Sheet metal forming is a widely used and costly manufacturing process, to which these considerations apply. Aluminum sheet becomes favorable compare to steel regards to some improvement at aerodynamic designs, increased engine efficiency and fuel economy. Wide range of aluminum automotive product included doors, fenders, bumpers face bars, seat frames and roof panels have been produced.

This paper was carried out to study the finite element (elastic-plastic) analysis of sheet metal forming process using the finite element software. LUSAS simulation was carried out to gain accurate and critical understanding of sheet forming process. Axis-symmetry element mesh and plain strain element mesh were use incorporated with slide-line features to model and study the sheet metal forming process. Simulation of elastic-plastic behavior of aluminum sheet was carried out under non-linear condition to investigate sheet metal forming process.

**Keywords:** FEA, Sheet metal forming process, LUSAS simulation,

**List of Symbols**

<i>Symbol</i>	<i>ID</i>
FEA	Finite Element Analysis
E	Young's modulus
F	Bending force
K	Numerical constant
L <sub>b</sub>	Bend allowance
L	Contact length
ρ	Material density
R <sub>d</sub>	Die radius
MBR	Minimum bend radius
σ <sub>y</sub>	Uniaxial yield strength
σ <sub>ps</sub>	Yield stress in plain strain tension
ε	Strain
t	Sheet thickness
ν	Poisson ratio
R <sub>f</sub>	Radius of curvature after springback
S	Ultimate tensile strength
α	Bend angle
R <sub>o</sub>	Original radius of curvature
W	Width of die

**1. Introduction**

In the manufacturing process based on numerical standards, simulation is an important tool in applying the experience and know-how gained in the past to the development of new models. A limiting factor in design process is the necessity of producing the desired shape in a single deep drawing for example, with no cracks or wrinkles in the sheet metal. As automotive and aerospace industries are growing rapidly the demand for precise and accurate information concerning parts design and formability of metal sheet becomes essential. Sheet metal forming simulation plays an indispensable role in integrating manufacturing necessities into the product design process at an early stage. In conjunction with concurrent engineering, sheet metal forming simulation is proving to be an important tool in linking design and manufacturing. Since sheet metals are most susceptible to failure under plane strain condition, several plane strain membrane finite element codes for sheet metal forming analysis has been developed by M. J. Saran and Wagoner, W. H. Frey and Wenner, M. P. Sklad [1-3]. These codes, in the absence of shape tooling curvatures, are capable of predicting deformation strains very accurately, but as tooling curvature becomes larger to cause significant bending strains, these codes can no longer capable to predict accurately [4]. The objects of this works will focus on simulating non-linear sheet forming process using finite element technique in order to come up with a clear and better understanding of metal flow of sheet metal forming process. Schematic diagrams of shearing with punch and die are shown in Figure 1.

**2. Bending Allowance and Spring-back Phenomena**

The force required to punch is basically the product of the shear strength of the sheet metal and the area being sheared. Bending allowance is the length of neutral axis and is used to determine the blank length for the bent part. The approximate formula for the bend allowance ( $L_b$ ) is given by

$$L_b = \alpha(R + Kt) \quad (1)$$

Where ( $\alpha$ ) is the bend angle in radians, ( $R$ ) is the bend radius, ( $K$ ) is a constant ( $K=0.33$  for  $R < 2t$  and  $K=0.5$  for  $R > 2t$ ) and ( $t$ ) is the sheet thickness. The minimum bend radius (MBR) is referring to the radius at which the crack appears on the outer surface of the bend. In bending within elastic limit, the outside of the bend is in tension and the inside of the bend is in compression. While the neutral axis half way through the sheet thickness is remain (Figure 2). When the bend becomes plastic, the position of neutral axis changes and moves towards the concave side of the bend [5], thus the tensile strain is

greater than the compressive strain. As the bend radius  $R$  decreases, then  $\epsilon_1$  and  $-\epsilon_3$  increase and thinning become more obvious until finally cracking can occur.

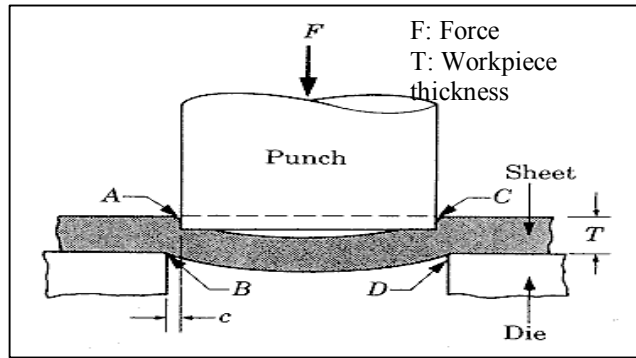
$$\epsilon_1 = -\epsilon_3 = \ln \left( 1 + \frac{1}{\frac{2R}{t} + 1} \right) \tag{2}$$

Springback in simple bending is shown in Figure 3, Hosfold and Caddeell [6] have propose for the simple bending case that

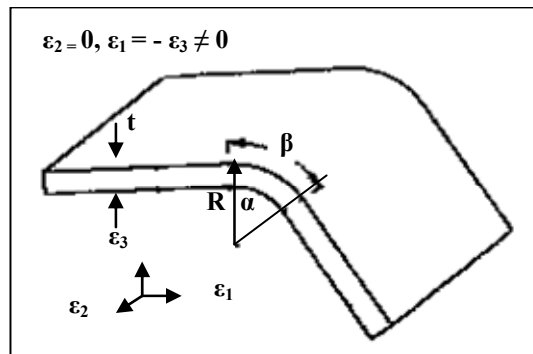
$$\frac{1}{R_0} - \frac{1}{R_f} = \frac{3\sigma_{ps}}{tE} \tag{3}$$

Where,  $R_0$  is the original radius of curvature,  $R_f$  is the radius of curvature after springback,  $\sigma_{ps}$  is the yield stress (in pane-strain tension),  $t$  is the thickness of plate and  $E$  is the elastic modulus.

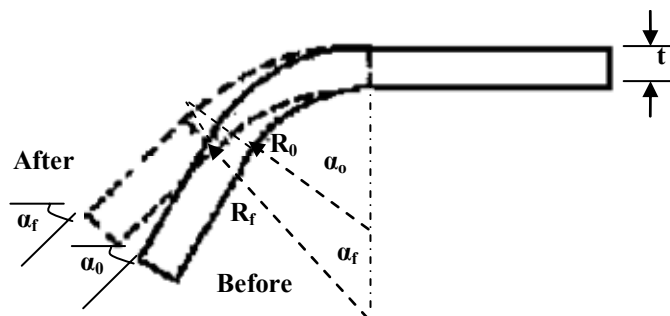
**Figure 1:** Schematic Diagram of Shearing With Punch and Die



**Figure 2:** Bending Processing



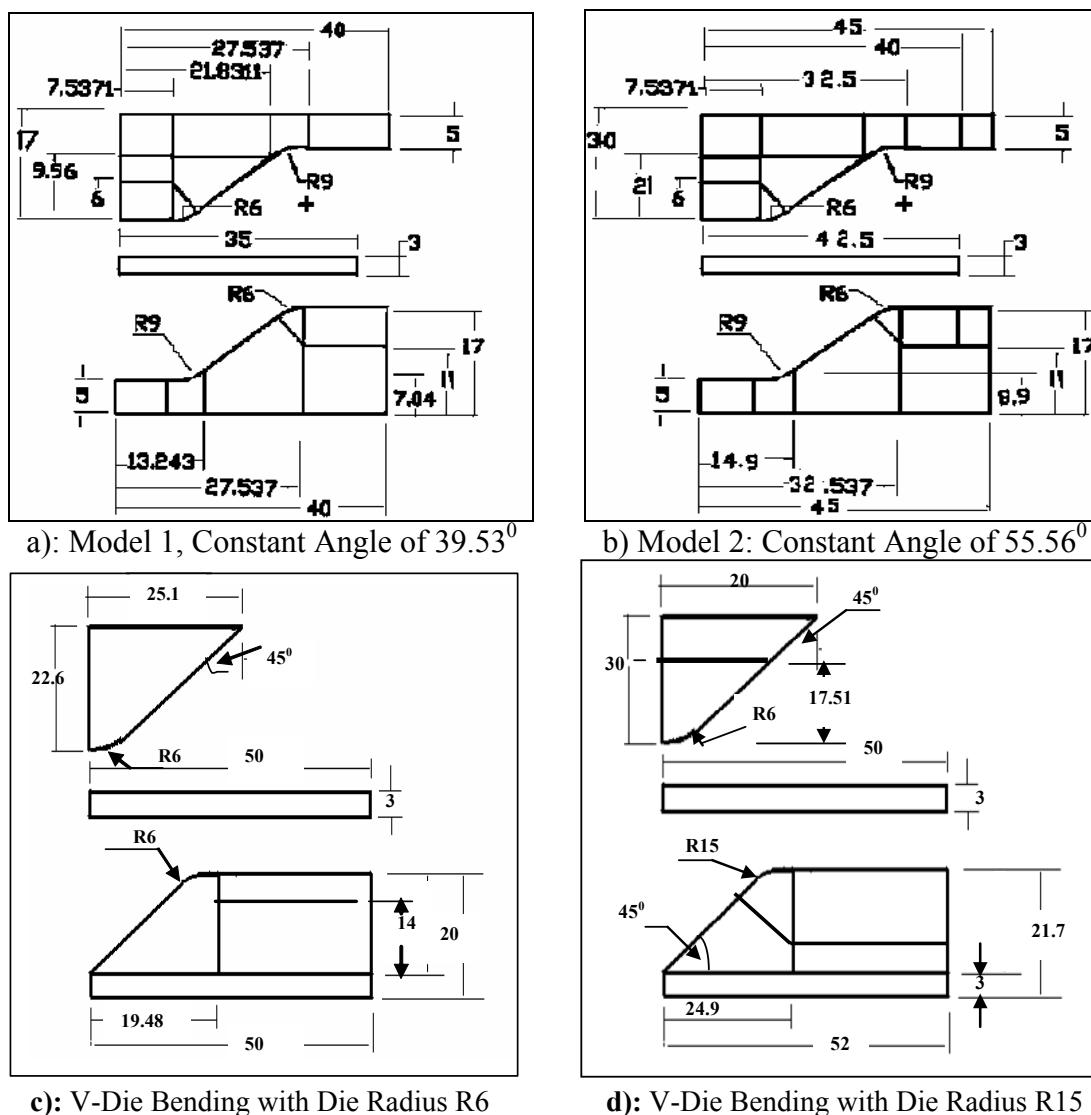
**Figure 3:** Spring-back in Bending



### 3. Definition of Tooling Geometry

Sheet metal forming processes are the complex interaction between specimen (geometry, tolerance, surface topology, etc.), the forming process (tooling, forming machine, force, lubrication, etc.) and the material (ductility, material parameters, microstructure, corrosion resistance, residual stress, etc.) which exist in forming processes. According to Huang and Leu [7] a larger die radius obviously causes springback. Therefore in the first stage of tool design used a smaller radius 6mm as the initial condition for the analysis. The punch stroke suggested by Meguid and Refaat [8] is 35mm. But eventually throughout the modeling exercise it is proved that 35 mm is not suitable for the model in use and some modification has been applied on the model itself including the punch stroke, length between symmetry line to the center of die curvature and draft angle of the die generally ( $1^{\circ}$ – $2^{\circ}$ ) degrees. At the view point of economic consideration the thickness of the die should be optimize to withstand the applied and cyclic load. Standard sheet thickness 3mm is taking as default throughout the computational experiment. After considering the final dimension of the die geometry was produced using Computer Aided Design (CAD) software, the drawing is in 2D since the analysis is carried out in 2D simulation as well. Since there is some inaccuracy while exporting the geometry of the model therefore it was decided to transfer the drawing manually using the coordinate (with the accuracy of 4 decimal points) of the nodes. Figure 4 shows Schematics diagram and dimensions of the models.

Figure 4: Schematics Diagram and Dimensions of the Models



## 4. Finite Element Simulation

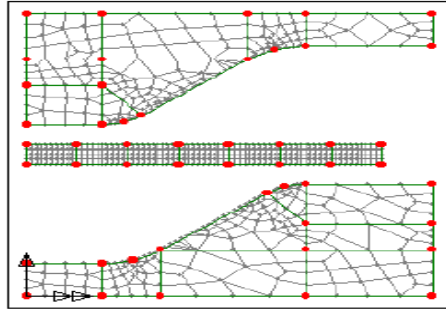
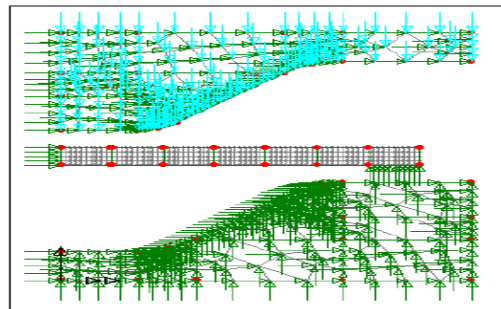
Finite element analysis is a simulation technique which evaluates the behaviour of components, equipments and structures for various loading conditions including applied forces, pressures and temperatures. Finite element analysis is a computerized method for predicting how a real object will react to forces, heat, vibration, etc. by mesh of simpler interlocking structures, the simpler structures or finite elements being amenable to mathematical analysis. The analysis of whole structure is obtained by simultaneously analysis the individual finite elements, having due regard to their individual positions within the mesh and being totally dependent upon the assistance of an automatic computer [9]. Numerical modelling of metal forming processes has now gained the industrial stage, and it became possible to simulate metal deformation and to calculate stress and strain states for complex processes. By using the finite element analysis method, simulation of design can help to predict errors and modification can be done at early stage before the parts was fabricated and tested. Subsequently the labour cost and time lost can be reduced. Therefore finite element methods will gradually replacing manual trial-and-error design iteration with sophisticated numerical simulations in future. It can be seen that the increasing demands of using finite element method in manufacturing process (especially in pre-processing analysis stage) will greatly enhance the efficiency and saving of time and manpower.

### 4.1. Model Development

A complete LUSAS Finite Element Analysis consists of three stages: I) Pre-Processing, II) Finite Element Solver and III) Results-Processing. LUSAS software consists of two parts to perform a full analysis:

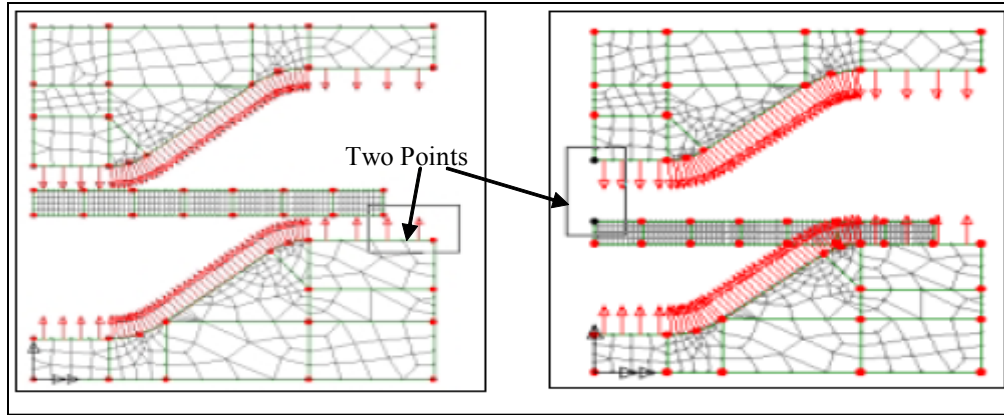
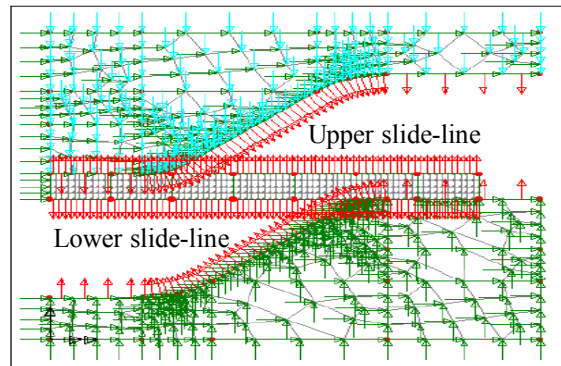
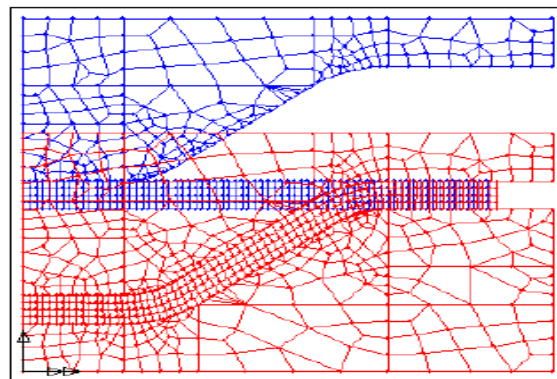
- LUSAS Modeller is fully interactive pre- and post-processing graphical user interface
- LUSAS Solver performs the finite element analysis

LUSAS is a general purpose finite element analysis system which incorporates a variety of analysis facilities. Its products can be used to solve a variety of linear and non-linear stress, dynamics, composite and thermal engineering analysis problems [5]. Corresponding to real engineering problems LUSAS is capable to model and simulate the structure by assembling all of the simple expressions into a set of simultaneous equations with the degree of freedom at each node. The combinations of nodes and lines were finally defined the finite elements models [11]. Sheet metal forming processes are the complex interaction between specimen (geometry, tolerances, surface topology, etc.), the forming process (tooling, forming machine, forces, lubrication, etc.) and the material (ductility, material parameters, micro structure, corrosion resistance, residual stresses etc.), which exist in forming processes. The processes deep drawing, and V-Die bending that are under investigation in this paper. The reason why the interactions in sheet metal forming processes are so complex is that a change in one area creates changes in the other areas and that the interactions are highly non-linear. Initially, the sheet is discretized into uniformly regular finite elements mesh. Applying incremental displacement throughout the iteration steps was carried out along the forming process. Subsequently the parts of the mesh are refined. To determine the contact points between the forming tools and the sheet metal in order to ensure smooth flows of material, a proper consideration has been carried out regarding punch and die radius. Additionally, large plastic strains had to be modelled using specially developed material laws with corresponding strain hardening. Since the model is symmetric therefore only half of the model is generated and simulated for time saving. The mesh generated and boundary conditions of the model were shown in Figures 5 and 6.

**Figure 5:** Geometry and Mesh Generation of Model**Figure 6:** Load and Boundary Conditions of Model

#### 4.2. Punch, Sheet Metal and Die Bring Into Contact

Before running the analysis, the punch, sheet metal and die are bringing into contact first. Selected two points starting from the sheet and followed by the edge of the die to move it and the same procedure is apply to move the punch where a different sets of points are selected as shown in Figure 7. Only the expected region of contact should be defined as a slide-line surface for tied slide-line analyses and coarse mesh discretisation in the region of contact should be avoided. Slides must be continuous and should not subtend an angle greater than 90 degrees. Sharp corners are best described by two separate slides. No modification is applied for the transformation involve. The final model show in Figure 8 and it is ready for running the analysis. Numerical simulation for the specimen is run until complete of analysis. As the punch moving down further the free end of metal sheet will experience some compression. Under perfect lubrication the penetration still occurs at a very low scale when the punch and die are making contact with the metal sheet. Such problem been reduced by changing the stiffness scale of slideline. The deformed mesh (represent by red color) and original mesh (represent by blue color) shows the deformation of metal sheet as shown in Figure 9.

**Figure 7:** Two Points Selected to Invoke Precise Translation**Figure 8:** Final Model Geometry with All Features Visualize**Figure 9:** Original (Blue) and Deformed Mesh (Red)

### 4.3. Material Properties

This section refers to material types used to model; there are two types of isotropic material involved in this model: linear elastic-plastic (for sheet metal) and linear elastic (for the punch and die). The basic material properties required for this model are given in Table 1

### 4.4. Nonlinear Analysis

Linear finite element analysis assumes that all materials are linear elastic in behavior and that deformations are small enough to not significantly affect the overall behavior of the structure. While nonlinear analysis is much more complicated than simple linear analysis because it is required many

variables such as changes in geometry, permanent deformations, structural cracks and buckling. A fundamental difference between geometrically linear and geometrically nonlinear analysis is that in linear analysis equilibrium is satisfied on the initial un-deformed configuration, whereas in nonlinear analysis equilibrium must be satisfied in the deformed configuration [9]. In this study, two types of nonlinear analysis using LUSAS finite element software: Geometric nonlinearity in which arise from significant changes in the structural configuration during loading and material nonlinearity in which effects arise from a nonlinear constitutive model (that is, progressively disproportionate stresses and strains). Nonlinear geometric and material effects may be incorporated in this analysis. To achieve final equilibrium in a nonlinear analysis, we solve the model many times, constantly adjusting the applied forces based on the current state of equilibrium, and modifying the geometry based on the current displacements. Convergence nonlinear analyses are generally solved by an iterative procedure and the level of convergence reached tells us the likely error in the solution. Not all nonlinear analysis was guaranteed to converge but often convergence can be helped along by modifying some of the parameters. For nonlinear analysis, since it is no longer possible to directly obtain results according to external loads, a solution procedure is usually adopted in which the total required load is applied in a number of increments. The analysis allows you to set any desired load level for each increment or time step of a nonlinear solution. For the analysis of nonlinear problems, the solution procedure adopted may be of significance to the results obtained. In order to reduce this dependence, wherever possible, nonlinear control properties incorporate a series of generally applicable default settings, and automatically activated facilities.

**Table 1:** Material Properties of the Model

<b>Sheet Metal (Aluminum)</b>		
<b>Linear Elastic-Plastic</b>		
<b>Material Property</b>	<b>Symbol</b>	<b>Property Value</b>
Young's Modulus	E	78 GPa
Uniaxial yield strength	$\sigma_y$	550MPa
Poisson's Ratio	$\nu$	0.3
Material density	$\rho$	2700 kg/m <sup>3</sup>
<b>Punch and Die ( Steel)</b>		
<b>Linear Elastic</b>		
Young's Modulus	E	400 GPa
Uniaxial yield strength	$\sigma_y$	4000MPa
Poisson's Ratio	$\nu$	0.3
Material density	$\rho$	7800 kg/m <sup>3</sup>

## 5. Result and Discussion

This section will focus on the results obtained from LUSAS Finite Element to analyze the prescribed model to produce the output files. Deep drawing analysis was carrying out and the effect of geometry towards sheet metal forming process has been studied. V-die bending was preformed and the effect of die radius towards force applied was observed. Several simulation tests were carrying out to obtain the most appropriate value for some of the parameter.

Since Model 2 have greater punch stroke compare to Model 1, therefore the induced stress is higher. The stress is not only concentrated at curve region but also at the free end of the axisymmetry model. For plane strain element the stress distribution is steadily spread over the sleeve between two-curve regions. Model 1 does not suffering from the thinning effect and sliding-sticking condition that happens on Model 2. Penetration causes the irregular formation of the final dimension. Higher load is required for greater punch stroke. Equivalent Stress (SE) contours for model 1 and model 2 are shown in Figure 10 to Figure 17 for 2D plain element mesh and 2D axisymmetry element mesh. From the plotted results contours we can see that there are greater stress induced for displacement test compare to loading test since the force is directly applied to the punch. For plain strain model there required



larger load applied to cause the metal sheet to fulfill the die cavity. Meanwhile the stress concentrate at the middle part of the metal sheet for plain strain model that is quite different from the high stress concentration happens at the curvature region of metal sheet for the axisymmetry model.

The results of the V-die models can be obtained from this analysis. The increment of die radius causing decrement in the load needed to deform the same pattern some and at the same time increasing the maximum stress concentration on curve region. Figures 18 and 19 shows the stress of the deformed shape and load-displacement curve was shown in Figure 20.

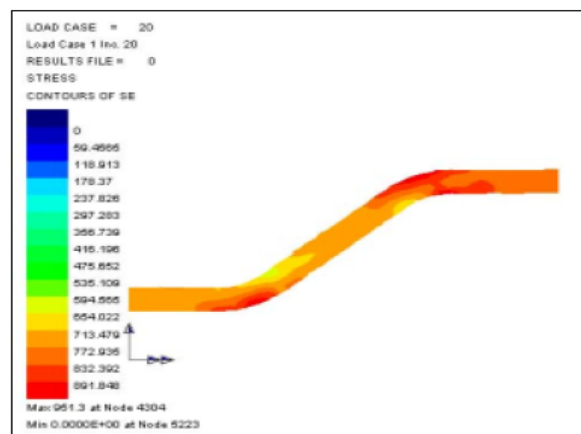
## 6 Conclusions

The use of finite-element simulation in the better understanding of forming operations is becoming more important as it provides a cheap and efficient way to determine important process parameters. This paper presents a 2D finite element simulation study concerning the sheet metal forming process. Deep drawing analysis was carrying out and the effect of geometry towards sheet metal forming process has been studied. V-die bending was preformed and the effect of die radius towards force applied was observed. Several simulation tests were carrying out to obtain the most appropriate value for some of the parameter. An attempt base on computational experiments is made to explore the effect of process variables on stress distribution and punch load. An elastic-plastic finite element computational program was developed to simulate successive deep drawing process and V-die bending process. Satisfactory agreement between the loading test and the displacement test was obtained that verified the program.

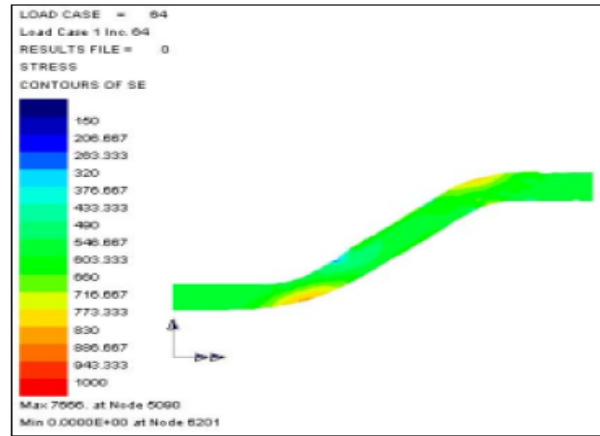
Some findings of the project is summarized as follows:

1. Element mesh for simulation was constraint by the use of slide-line option.
2. Greater punch stroke will induce greater stress especially at the bending region and the stress distribution will spread wider along the metal sheet.
3. Sliding-sticking condition becomes obvious as the forming process induces greater change of sheet dimension. Plasticity elongation of material occurs especially at the contact point of bending zone.
4. Punch and die radius shall be specially designed in order to maintain a smooth flow of material to maintain a good quality of final product.
5. Excessive displacement and excessive load of punch shall be avoid ensuring that the metal sheet is not penetrating the punch and die.
6. As the die radius (for V-die only) increases the punch load decreases.

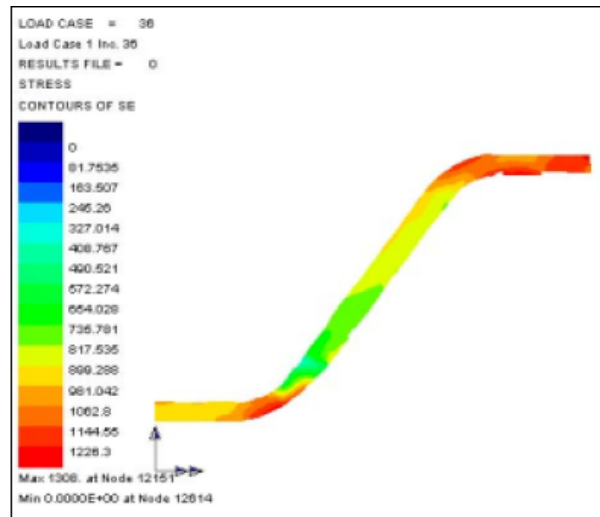
**Figure 10:** Axi-symmetry Model Displacement Test for Model 1



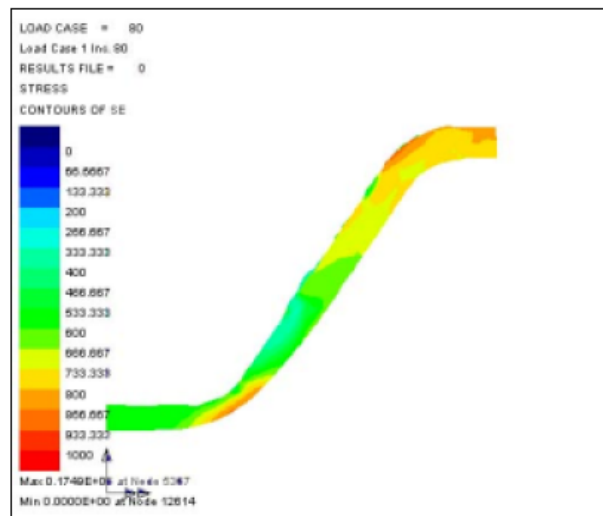
**Figure 11:** Axi-symmetry Model Loading Test for Model 1

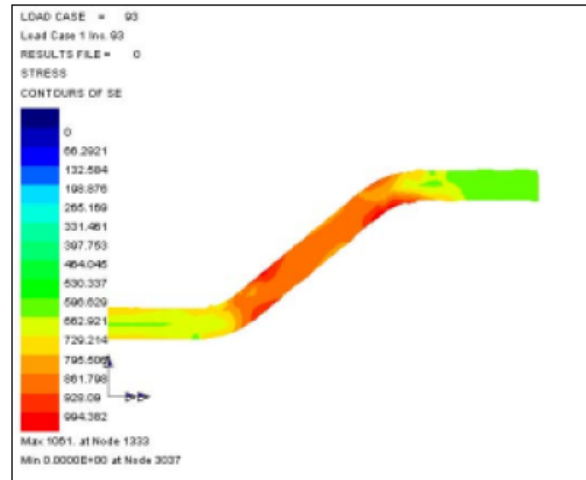
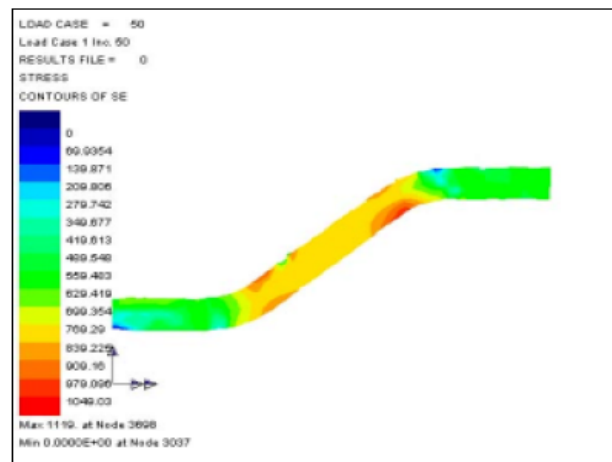
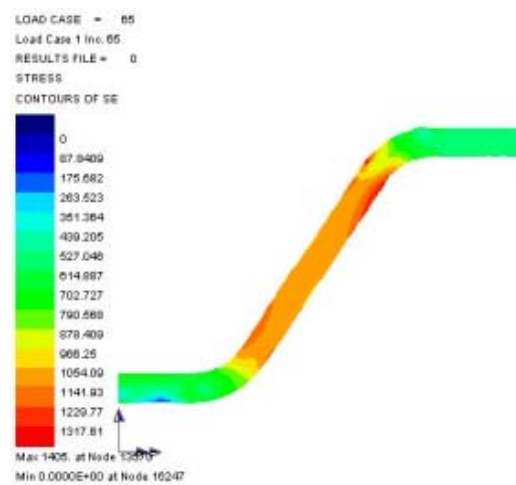


**Figure 12:** Axisymmetry Model Displacement Test for Model 2

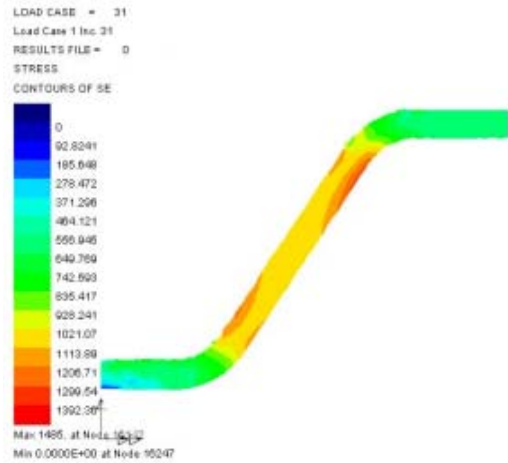


**Figure 13:** Axisymmetry Model Loading Test for Model 2

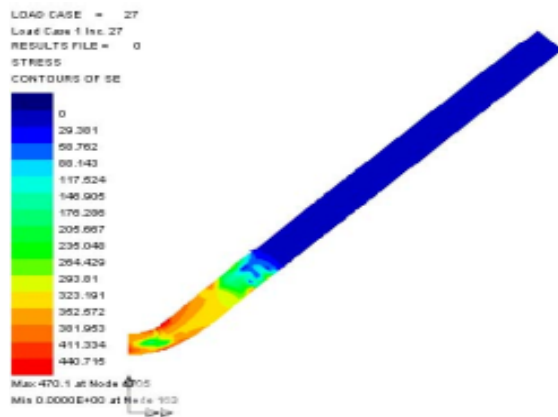


**Figure 14:** Plain Strain Model Displacement Test for Model 1**Figure 15:** Plain strain Model Loading Test for Model 1**Figure 16:** Plain Strain Model Displacement Test for Model 2

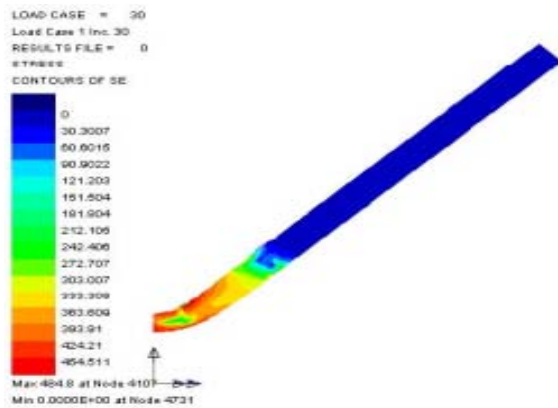
**Figure 17:** Plain Strain Model Loading Test for Model 2

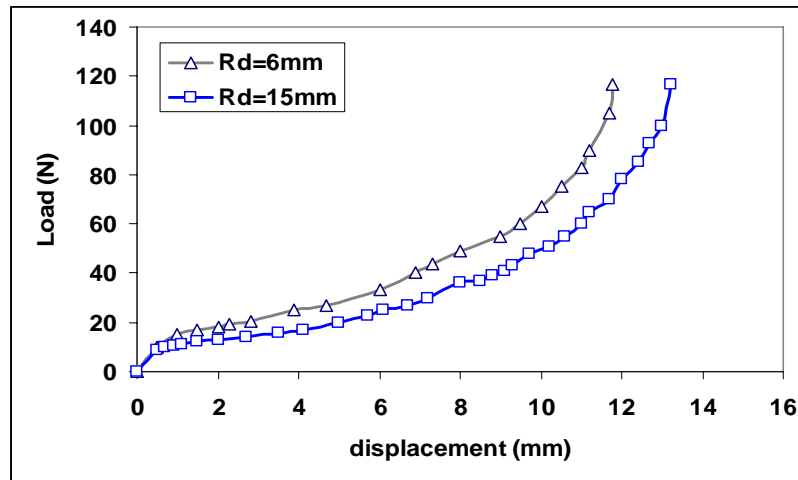


**Figure 18:** Plane strain V-die Model (Rd 6) for Loading Test



**Figure 19:** Plane Strain V-die Model (Rd15) for Loading Test



**Figure 20:** Load-Displacement Curves and Effect of Die Radius for V-Die

## References

- [1] M. J. Saran and R. Wagoner. (1991), A user's manual and verification examples, sheet stamping. Report no. ERC/NSM-S-91-17
- [2] W. H. Frey (1987). Development and application for a one dimensional finite element code for sheet metal forming. Materials Processing, V.1 pp.307-320, Boston
- [3] M. P. Sklad, (1990), 16th IDDRG Congress, ASM International, Sweden. pp. 295
- [4] F. Pourboghrat (1992) Spring-back calculation for plain strain sheet forming using finite element membrane solution, ASME 1556
- [5] Roger Pearce (General, International Deep Drawing Research Group) Sheet Metal Forming. New York, pp81-88
- [6] Hosford W F and Caddell R M (1998), Metal Forming (Englewood Cliffs NJ: Prentice Hall)
- [7] You-Ming Huang and Daw-Kwei, Effects of process variables on V-die bending process of steel sheet. Int. J. Mech. Sci. Vol.40 No. 7, pp 631-650
- [8] S.A. Meguid and M.H. Refaat (1997), Finite element analysis of the deep drawing process using variation inequalities Elsevier Science
- [9] Hakim S. Sultan, (2004), Experimental Investigation and Finite Element Analysis of Composite Conical Structures" PhD thesis, University Putra Malaysia, Malaysia
- [10] LUSAS/standard User's Manual, LUSAS theory manual, version 13.5. FEA United Kingdom