CHAPTER 6

FINITE ELEMENT SIMULATION OF COLD UPSETTING PROCESS

6.1 INTRODUCTION

The upsetting of solid cylinders is an important metal forming process and an important stage in the forging sequence of many products. Cold forming process minimizes material wastage, improves mechanical properties such as yield strength and hardness and provides very good surface finish. The tools used are also subjected less thermal fatigue [1]. Metal flow is influenced mainly by various parameters like specimen geometry, friction conditions, characteristics of the stock material, thermal conditions existing in the deformation zone, and strain rate. Metal flow influence the quality and the properties of the formed product; the force and energy requirements of the process. A large segment of industry depends primarily on the predominant applications of this process which includes coining, heading and closed die forming.

When ever a new component is produced there is usually a trial and error stage to tune the progress, this helps in obtaining a component without defects, using required quantity of raw material. Previous experience of designers and manufacturers gives an important aid in reducing these trails. Use of new materials associated with new shapes and designs create the possibility of new behaviour. For better understanding of this behaviour and better knowledge of metal flow free deformation is to be studied. A systematic and detailed study of free deformation can be used to predict the metal flow under various working conditions and with various tool and work piece geometries. This information can then be used to design better forgings, to make improved intermediate dies, and to avoid the defects and failure of materials during plastic working. The finite element simulation helps in analyzing the process and predicting the defects that may occur at the design stage itself. Therefore, modifications can be made easily, before tool manufacturing and part production, reducing the trail and error stage and its associated costs [2-8].

The basic idea in the finite element analysis (FEA) is to find the solution of a complicated problem by replacing it by a simpler one. Since a simpler one to find the solution replaces the actual problem, we will be able to find only an approximate solution rather than the exact solution [1]. The existing mathematical tools are not
sufficient to find the exact solution of most of the practical problems. Thus in the absence of convenient method to find the approximate solution of 3-D problem, we have option for FEA. The FEA basically consists of the following procedure. First, a given physical or mathematical problem is modeled by dividing it into small interconnecting fundamental parts called Finite Elements. Next, analysis of the physics or mathematics of the problem is made on these elements: finally, the elements are re-assembled into the whole with the solution to the original problem obtained through this assembly procedure.

6.2 LITERATURE REVIEW

Computer technology became inevitable in the engineering and manufacturing industry as computing has become cheap and fast enough to simulate metal forming numerically in a useful timescale. The finite element (FE) method has been used to solve equations which describe solid metal flow [9]. In the last 15 years considerable efforts have been made, largely in a research environment, to apply FE modeling to the simulation of forging operations and in evaluating the results. The technique is now being adopted tentatively by the forging industry. FE codes which will analyze the plastic deformation of metals are available but are often not readily useable by non-experts.

Although a considerable amount of research has been done in developing the FEM for metal forming simulation since the pioneering work [10] was presented in 1973, rigorous three-dimensional simulation of metal forming problems still remains a challenging task from the standpoint of computational efficiency, solution accuracy, graphics visualization, mesh generation and automatic re-meshing, and so on. As computer technology and FEM advance, wider and more complicated metal forming processes are being investigated. It is believed that the further development of FEM will be continuously challenged by the need from the industry to make the modeling more accurate, more practical, and more affordable.

To analyze metal forming the FE method requires a mathematical description of the metal's plastic flow behaviour. Two alternative formulations have been used to do this [11, 12]. One model of plasticity is a structural formulation derived from the analysis of solid bodies and the other which is allied to fluid mechanics concepts is a flow formulation. Calculation of elastic effects such as residual stresses and spring-
back can be included in both formulations. Figure 6.1 shows how the mechanical formulations are categorized in four ways which depend on how plasticity is modeled and if elastic deformation of the solid metal is neglected or not [13]. Generally the flow formulation is applicable to hot forging where the mechanical properties of the metal are sensitive to rate of deformation and the structural formulation applies to cold forging where the influence of deformation rate can often be ignored.

![Fig.6.1 Different types of material behaviour](image)

The elasto-plastic constitutive equation together with contact conditions are used for metal forming analysis. The radial return mapping algorithm is used to compute the stress and internal variables (Krieg and Key, 1976) [14], and the consistent tangent operator (Simo and Taylor, 1985) [15], which preserves the quadratic convergence rate of the Newton method is also used.

J.Babu Rao et.al [16], reported the deformation behaviour of Al-4Cu-2Mg alloy during cold upset forging, and found the relationship between the circumferential strain and axial strain increments is continuously changing with increasing deformation.
J. Appa Rao, J. Babu Rao, Syed Kamaluddin et al [17], reported the cold workability limits of Brass, as a function of friction, aspect ratio and specimen geometry, they also conducted the ring compression test to determine the friction factor, and finally validated the experimental results with the finite elements software.

S. Dikshit, et al [18], have studied and evaluated the effect of homogenization and deformation behaviour of AA2014/SiC/10p composite billets, having dispersions in size range of 20–40 µm, by conducting cold upsetting experiments under unlubricated conditions and made comparison with finite elements software.

6.3 PROCEDURE ADOPTED IN MODELING THE PROBLEM UNDER TAKEN.

The computational modeling of each forming process stage by the finite element method can make the sequence design faster and more efficient, decreasing the use of conventional “trial and error” methods. In this study, the application of commercial general finite element software ANSYS-10 has been applied to model a forming operation.

Finite element analysis of deformation behaviour of cold upsetting process was carried out for all the HSA(P) composites and base alloy (A356) with aspect ratios of 1.0 and 1.5 in dry condition. Due to axisymmetric nature of the geometry only quarter portion was modeled with symmetric boundary conditions. Rigid-flexible contact analysis was performed for the forming process, so that the die appears like a thin plate, and the stress analysis was done on the billet only. The billet geometry was meshed with 8-node Brick elements (solid 185 in ANSYS Library). Element size was selected on the basis of convergence criteria and CPU time. Too coarse mesh may never converge and too fine mesh requires long CPU time without much improvement in accuracy.

6.3.1: Contact Analysis

The contact problem is a kind of geometrically nonlinear problem that arises when different structures or different surfaces of a single structure, either come into contact or separate or slide on one another with friction. Contact forces, either gained or lost, must be determined in order to calculate structural behavior [19]. The location and extent of contact may not be known in advance, and must also be determined.
Contact algorithms in FM analysis allow contact elements to be attached to the surface of one of two FE discretization’s that are expected to come in contact. A contact element is not a conventional finite element. Their functions is to sense contact and then supply a penalty stiffness or activate some other scheme for preventing or limiting interpenetration. Contact analysis is highly complex and nonlinear analysis. Contact problems fall into two general classes. One is rigid-to-flexible and flexible-to-flexible. In rigid-to-flexible contact problems, one or more of the contacting surfaces are treated as rigid, i.e., it has a much higher stiffness relative to the deformable body it contacts.

In general, any time a soft material comes in contact with hard material, the problem is assumed to be rigid-to-flexible, instances like: metal forming problems. The other class, flexible-to-flexible, is the more common type. In this case, both contacting bodies are deformable, i.e. have similar stiffness. Example, bolted flanges. Ansys supports three contact models; node-to-node, node-to-surface and surface-to-surface contact. In problems involving contact between two boundaries, one of the boundaries is conventionally established as the target surface and the other as the contact surface. For rigid-flexible contact, the target surface is always the rigid surface and the contact surface is the deformable surface. For flexible-to-flexible contact, both surfaces are associated with deformable bodies. These two surfaces together comprise the contact pair. Ansys provides special elements for contact pair. Different contact elements are CONTAC12, CONTAC52, CONTAC 26, CONTAC 48, CONTAC 171,172, TARGET 169, CONTAC 173, and TARGET 170. Figure 6.2 shows the contact pair between die and composite model.

Contact elements are constrained against penetrating the target surface. However, target elements can penetrate through the contact surface. In the present problem rigid-to-flexible analysis is performed. For rigid-to-flexible contact, the choice of target was tooling (upper and bottom platens) which need not require meshing. Obviously the soft billet surface was considered as contact
6.3.2 Material Properties and Real Constants

The material models selected were based on the properties of the tooling and billet materials. Due to high structural rigidity of the tooling, only the following elastic properties of tooling (H13 steel) were assigned assuming the material to be isotropic [20].

$$\text{Young’s Modulus } E = 220 \text{ GPa (for steel)}$$
$$\text{Poisson’s ratio } \nu = 0.3$$

As the nature of loading is non-cyclic, Bauschinger effect could be neglected and the non-linear data was approximated to piecewise multi linear with 10 data points. The material was assumed to follow the Isotropic hardening flow rule. Suitable elastic properties were also assigned for the material under taken for analysis. As the experiments were conducted at room temperature, the material behaviour was assumed to be insensitive to rate of deformation.
6.4 RESULTS AND DISCUSSION

Figure 6.3 and 6.4 shows the meshed models of billets and tooling for the aspect ratios of 1.0 and 1.5 respectively. Figure 6.5 and 6.6 shows the 50% deformation specimen with zero friction for the aspect ratios of 1.0 and 1.5 respectively. Since there was no friction at metal-die contact, the deformation can be treated as homogeneous since no bulging was seen. The maximum radial displacement corresponding to 50% for the aspect ratio of 1.0 is shown as 2.435mm in figure 6.7. This means that the diameter after 50% deformation equals to 12 + 2 × 2.435 = 16.87 mm. The value of analytically determined diameter after 50% equals to 12 × √2 = 16.970 mm, (assuming volume constancy) leading to a very little error of 0.59% usually can be discarded in non-linear finite element analysis such as in large deformation / metal forming applications. Hence the analysis procedure adopted is validated.

For the present study the friction factor ‘m’ was found to be 0.36 (explained in chapter 5) and the extent of barreling with this friction at 50% deformation for alloy and composites under investigation was shown in figures 6.9-6.40 respectively. These results were supported by many authors [11-30]. Lower aspect ratio (H₀/D₀ = 1.0) samples has shown more barreling affect compare to higher aspect ratio (H₀/D₀ = 1.5). The above results were experimentally evidenced, as discussed in chapter 5.

The notations used in the analysis were, radial displacements (UX): circumferential stress, σᵧ(SY), axial stress σz(SZ), hydrostatic stress, σ_H(NLHPRE), Von-Mises equivalent stress σ (SEQV).

The variations in FEA results compared to analytical results obtained in chapter 5 were shown in figures 6.41 to 6.44. The obtained FEA results revealed that these values are closely matching with the experimental values with a maximum deviation of less than 5%. Hence the FEA model adopted for solving the present upsetting analysis was validated with the analytical results of chapter 5.
Figure 6.3: Undeformed sample ($H_0/D_0 = 1.0$)

Figure 6.4: Undeformed sample ($H_0/D_0 = 1.5$)
Figure 6.5: Deformed sample ($H_0/D_0 = 1.0$) at 50% deformation for zero friction

Figure 6.6: Deformed sample ($H_0/D_0 = 1.5$) at 50% deformation for zero friction
Figure 6.7: Radial displacement at 50% deformation for zero friction ($H_0/D_0 = 1.0$)

Figure 6.8: Radial displacement at 50% deformation for zero friction ($H_0/D_0 = 1.5$)
Figure 6.9: Axial Stress at 50% deformation of A356 alloy (H_o/D_0 = 1.0)

Figure 6.10: Axial Stress at 50% deformation of A356 alloy (H_o/D_0 = 1.5)
Figure 6.11: Circumferential Stress at 50% deformation of A356 alloy ($H_0/D_0 = 1.0$)

Figure 6.12: Circumferential Stress at 50% deformation of A356 alloy ($H_0/D_0 = 1.5$)
Figure 6.13: Hydrostatic Stress at 50% deformation of A356 alloy (H_0/D_0 = 1.0)

Figure 6.14: Hydrostatic Stress at 50% deformation of A356 alloy (H_0/D_0 = 1.5)
Figure 6.15: Von-Mises Stress at 50% deformation of A356 alloy ($H_0/D_0 = 1.0$)

Figure 6.16: Von-Mises Stress at 50% deformation of A356 alloy ($H_0/D_0 = 1.5$)
Figure 6.17: Axial Stress at 50% deformation of 5% composite \((H_0/D_0 = 1.0)\)

Figure 6.18: Axial Stress at 50% deformation of 5% composite \((H_0/D_0 = 1.5)\)
Figure 6.19: Circumferential Stress at 50% deformation of 5% composite (H_0/D_0 = 1.0)

Figure 6.20: Circumferential Stress at 50% deformation of 5% composite (H_0/D_0 = 1.5)
Figure 6.21: Hydrostatic Stress at 50% deformation of 5% composite ($H_0/D_0 = 1.0$)

Figure 6.22: Hydrostatic Stress at 50% deformation of 5% composite ($H_0/D_0 = 1.5$)
Figure 6.23: Von-Mises Stress at 50% deformation of 5% composite (H_0/D_0 = 1.0)

Figure 6.24: Von-Mises Stress at 50% deformation of 5% composite (H_0/D_0 = 1.5)
Figure 6.25: Axial Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.0$)

Figure 6.26: Axial Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.5$)
Figure 6.27: Circumferential Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.0$)

Figure 6.28: Circumferential Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.5$)
Figure 6.29: Hydrostatic Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.0$)

Figure 6.30: Hydrostatic Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.5$)
Figure 6.31: Von-Mises Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.0$)

Figure 6.32: Von-Mises Stress at 50% deformation of 10% composite ($H_0/D_0 = 1.5$)
Figure 6.33: Axial Stress at 50% deformation of 15% composite ($H_0/D_0 = 1.0$)

Figure 6.34: Axial Stress at 50% deformation of 15% composite ($H_0/D_0 = 1.5$)
Figure 6.35: Circumferential Stress at 50% deformation of 15% composite 
\( \frac{H_0}{D_0} = 1.0 \)

Figure 6.36: Circumferential Stress at 50% deformation of 15% composite 
\( \frac{H_0}{D_0} = 1.0 \)
Figure 6.37: Hydrostatic Stress at 50% deformation of 15% composite ($H_0/D_0 = 1.0$)

Figure 6.38: Hydrostatic Stress at 50% deformation of 15% composite ($H_0/D_0 = 1.5$)
Figure 6.39: Von-Mises Stress at 50% deformation of 15% composite (H_0/D_0 = 1.0)

Figure 6.40: Von-Mises Stress at 50% deformation of 15% composite (H_0/D_0 = 1.5)
Figure 6.41: Comparative graphs between experimental and FEA values of Effective stress $\bar{\sigma}$, stress components $\sigma_\theta$, $\sigma_z$ and $\sigma_H$ as a function of effective strain $\bar{\varepsilon}$ for A356 alloy up to 50% deformed in dry condition with aspect ratio: (a) $H_0/D_0 = 1.0$, (b) $H_0/D_0 = 1.5$. 
Figure 6.42 Comparative graphs between experimental and FEA values of Effective stress $\sigma$, stress components $\sigma_\theta$, $\sigma_z$, and $\sigma_H$ as a function of effective strain $\bar{\varepsilon}$ for 5% composite up to 50% deformed in dry condition with aspect ratio: (a) $H_0/D_0 = 1.0$, (b) $H_0/D_0 = 1.5$. 

128
Figure 6.43 Comparative graphs between experimental and FEA values of Effective stress $\bar{\sigma}$, stress components $\sigma_\theta$, $\sigma_z$ and $\sigma_H$ as a function of effective strain $\bar{\varepsilon}$ for 10 % composite up to 50% deformed in dry condition with aspect ratio: (a) $H_0/D_0 = 1.0$, (b) $H_0/D_0 = 1.5$. 
Figure 6.44 Comparative graphs between experimental and FEA values of Effective stress $\sigma$, stress components $\sigma_\theta$, $\sigma_z$ and $\sigma_H$ as a function of effective strain $\varepsilon$ for 15% composite up to 50% deformed in dry condition with aspect ratio: (a) $H_0/D_0 = 1.0$, (b) $H_0/D_0 = 1.5$. 
6.5 CONCLUSIONS

Detailed comparisons of the experimental variables with the finite element method (FEM) results were carried out to ascertain the accuracy with which the deformation process can be modelled. Predictions from the simulation results were found to be in good agreement with the actual experimentation

1. The cold upsetting process was modeled, simulated and analyzed with a sufficient accuracy.
2. The accuracy of results depends on the accuracy of the input data (true stress-true strain behaviour and friction factor obtained from the experiments) and friction model used in the analyses.
3. The analysis is useful in reducing the lead time of design cycle.
4. The machine down time can be reduced at production stage.
REFERENCES


13. Forging Modelling Project, School of Mechanical Engineering, University of Bath, Report FMP-3, 12-12.


19. ANSYS 10.0 Help manual documentation.


