Numerical Simulation on Radial Gear

S.Shankaranarayanan [1], R.Karthikeyan [2] and T. Naresh Kumar [3]

1Department of Mechanical Engineering, Velammal Engineering College, Surappattu, Chennai, Tamil Nadu 600066, India
2Department of Mechanical Engineering, VelTech, Avadi, Chennai 600062, Tamilnadu, India
3Department of Mechanical Engineering, Malla Reddy Engineering College, Hyderabad, Telangana 500100 India

ABSTRACT

This research focuses on numerical and experimental research of the net shape forming in the process of radial gear extrusion in cold state. Experiment was carried out using Al99.5 as a modeling material to record the force-stroke diagram. In the end of forming process, gear geometry was cut in horizontal and vertical planes to measure the hardness with the aim to compare the results and numerical simulation. Numerical simulation was carried out in MSC.Marc Mentat 2003, a commercial program package capable of dealing with finite strain using the Finite Elements Method. Offering a great number of modeling possibilities, numerical simulation was considered as a technology. Therefore the experimental model was needed to asset the optimal FE model, capable of capturing the physical phenomena like finite strain, strain hardening and friction. Comparison between the FE simulation and the experiment was done via numerically and experimentally obtained force-stroke diagram and via comparison of the measured hardness with numerically obtained equivalent strain. Comparative analysis of the results has shown that the actual mesh geometry, friction model and updated Lagrangian formulation of the finite strain, represent a satisfying numerical model of a gear extrusion in cold state. Refined by the experiment, obtained FE model can be applied to the other gear geometries only by making minor, geometrical changes in numerical model.

Keyword : - *Radial gear , Numerical Simulation, FEM.*

---

1. INTRODUCTION

Commercially available FE programs for forming process simulation, enumerated in [1], represent a great selection to choose from. One above the mentioned programs is MARC, nowadays known as MSC.Marc. It was used as an implicit type solver to preform a numerical modeling of the radial gear extrusion as a bulk metal forming process in a cold state.

Modelling included three phases. In first phase, using the MSC.Mentat preprocessor, all the objects were turned into numerical ones and the parameters for numerical simulation were set up. Second phase is represented by numerical crunching in MSC.Marc solver. Performed at the end of simulation, postprocessing in MSC.Mentat can be considered as a last, or a third phase. All the three phases can be separated and each one performed in a separate program package, but than a compatibility on relation preprocessor – solver – postprocessor becomes a topic too large for an end user of a commercially available FE program.

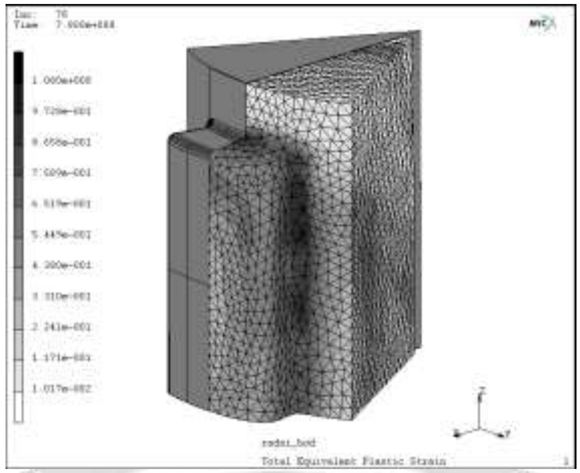

Fig. 1. Distorted body mesh of tetrahedral, four noded elements. Satisfactory fulfilment of gear cavity.

Main advantage in using a commercial elasto-plastic program packages lies in fast creation of a FE model. Therefore in order to create a reliable simulation, an experiment was carried out over the same geometry and material, using 6000 kN hydraulic press. Later comparison of the results improved both, the simulation and the experiment. Since the experiment and simulation can be considered as technologies, it is advisable to investigate the radial gear extrusion using more universal, analytical method known as Upper bound method like authors performed in [2].

The starting geometry of this analysis was Al 99.5 billet that underwent a large deformation in order to become a gear of simplified geometry. One sixth of the gear is shown in figure 1. Since the gear teeth are formed in a cold state by radial material flow, perpendicularly to the punch movement, this process is called the radial gear extrusion in a cold state. This manufacturing process has recently been introduced into the automotive industry to increase productivity.

2. EXPERIMENT

In order to record a force-stroke diagram, special tolling was designed and made as shown in figure 2. Changing the die insert for the same container geometry, results in a different teeth geometry being formed on the same billet radius. Ring stiffness diminishes the dimensions of the container and the die insert making this tooling applicable for radial extrusion of diverse gear geometries.

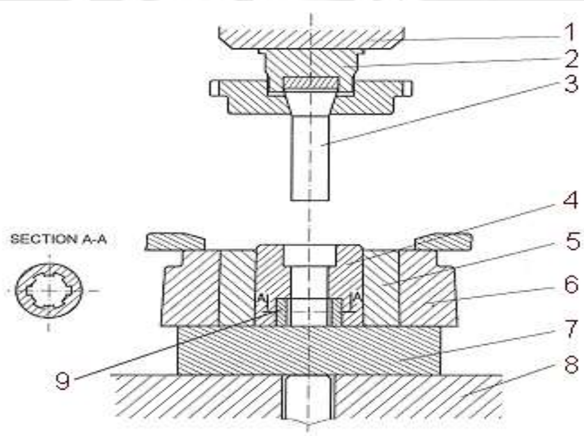

Fig. 2. Experimental tooling

1-upper plate, 2-punch holder, 3-punch, 4-container, 5-ring, 6-ring, 7-lower plate, 8-press, 9-replacable die insert

Fig. 3. Al billet cut in the horizontal and vertical planes in order to measure the microhardness using HV5.

Once the radial material flow turned the billet into the gear, it was cut in horizontal and vertical planes, as shown in figure 3, to measure the microhardness using HV5.

3. SIMULATION

A) Object geometry

Due to the periodical axial symmetry of the gear, FE simulation is carried out over one twelfth of the cylinder. Tetrahedral body mesh is created using Patran surface and body mesher.

Coarsening was used in body meshing to reduce the overall number of elements from 30 kE (30 000 elements) to 15 kE. Since the body mesh is created from the surface mesh, coarsening parameter influences only the inner elements and the number of elements forming the surface remains the same for both 15 kE and 30 kE model. This reduction leaves the friction model unaltered while reducing the calculation time to 3 hours.

B) Boundary conditions

Since a remeshing method is used, boundary conditions are set up using the axisymmetric planes. The planes are used instead of nodal boundary conditions to make the definition of boundary conditions more general and adequate for remeshing. An extra, arbitrary positioned plane is added to the set of boundary conditions, to cut off the sharp corner near the axis of symmetry. This is done in order to eliminate a bug, where nodes penetrate the axisymmetric planes while being captured inside of a sharp corner.

C) Contact bodies

From all the geometrical entities present in the simulation of radial gear extrusion, only the Al billet was modeled as a FE body. All the other, rigid contact bodies; punch, lower plate and the container, were considered as surfaces. Greater interest in physical behavior of rigid bodies, imposes an idea of considering them also as a FE bodies. A method proposed in [3] can be used to diminish the overall number of finite elements in simulation when FE modelling of the rigid bodies is required.

D) Material properties

The material was modeled as time independent rigid plastic material, with strain hardening. Stress - strain flowchart was assigned to Al material model after [4] and used via Von Misses criterion to determine the rigid-plastic flow of material.

E) Element type

After creation of the body mesh a 3D tetrahedral, low order element with a linear shape function was assigned to the 15 kE mesh. Possible locking of the low order elements is disabled using perturbed Lagrangian variational principle [5], while the incompressibility is acquired via Lagrangian multipliers. Applicable for the simulation of incompressible rigid plastic material flow, chosen element type has satisfactory fulfilled volume constancy hypothesis giving a total volume loss of 0.03%.

F) Friction model

A shear based model of friction has been used. This model calculates the friction force as a fraction of equivalent stress acting in the material near the touching surfaces in relative motion. As a result of a friction model, the resulting punch force shown in figure 4, is 5% greater than the force acting on the lower plate.

G) Analysis options

The radial gear extrusion was modeled as a small displacement finite strain analysis using Updated Lagrangian approach. As shown in [6], the equilibrium is expressed using the principle of virtual work in linearized form (1).

In stiffness equation (1) $\boldsymbol{K}$ is the stiffness matrix, $\boldsymbol{v}$ represents a nodal displacement vector and $\boldsymbol{f}$ is the nodal load vector.

$$\boldsymbol{K}\,\boldsymbol{v} = -\boldsymbol{f} \qquad (1)$$

Since continuum after being divided into FE mesh undergoes a large permanent deformations, the current state has little in common with initial coordinate frame. Therefore Lagrangian frame of reference is updated at the beginning of each increment, together with the assembly of the stiffness matrix $\boldsymbol{K}$. Integration of the (1) is then performed over the current deformed FE geometry using an iterative approach described in [5].

4. DISCUSSION

A) Force-stroke diagram

Comparison of the experimentally obtained and simulated force-stroke diagram, gives a satisfactory result with explainable differences coming out from the numerical background of the simulation and the technological background of the experiment.

A steep load rise at the very beginning of the simulation comes from the fact that the modelled surfaces come in immediate contact with all the top end nodes. Sudden change of the contact status of all top end nodes induces the first force jump visible in figure 4. Numerically speaking, change of the contact status implies the change in nodal force $\boldsymbol{f}$ and nodal displacement vector $\boldsymbol{v}$, complicating the integration of the equation (1).

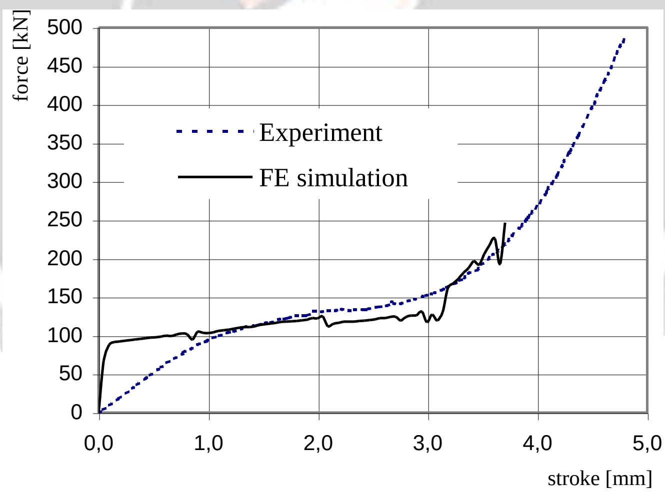

Fig. 4. Force-stroke diagram. Experiment (solid line), FE simulation (dashed line)

Opposed to the simulation, gradual load rise of the experimentally recorded force-stroke curve comes from the elasticity of the tooling and from the gradual fulfillment of vacancies between the tool and the Al 99.5 billet.

Second force jump at the 0.8 mm of simulated stroke is a consequence of global remeshing. During remeshing, nodal load vector $\boldsymbol{v}$ and nodal force vector $\boldsymbol{f}$ are being interpolated from the previous nodal positions in the FE mesh, numerically altering the equation (1).

A similar situation repeats itself to the greater extent at 2.0 mm stroke. Due to action of remeshing, a larger nodal vector values induce greater numerical error resulting in the larger force-stroke jump.

Taking place at 2.7 mm, third remeshing repeats the described situation just before the numerical ‘problem inducing’ phenomenon of nodes coming in contact, takes place at the 2.9 mm stroke. Start of a contact inside gear cavity indicates the final step of teeth forming, analogous to the second inflexion point on the experimentally recorded curve.

Remeshing at the 3.6 mm induces last force jump in simulation. Steep load rise for both, experiment and simulation, represents a final gear cavity filling. Simulation stopped before the expected stroke due to finite geometry of elements, capable of simulating corner filling phenomenon only up to the certain extent for the given element size.

B) Comparison of Measured hardnesses and simulated strain

The measured microhardness using HV5 were translated into the effective strain and compared to the effective strain obtained from the simulation. The results of the comparison are shown in figure 5. Used stress based friction model, increased the differences between simulated and measured strains along the lower plate.

5. CONCLUSION

Good solution for the use of commercial FE programs lies in parallel analysis of the simulation and the experiment, bearing in mind that the both of them are technologies. Their complexity requires a compromise in using and improving them in parallel. Once the satisfactory simulation has been reached, it can be adopted to the other gear geometries only by making a minor, geometrical changes in the simulation. A large number of available commercial FE program packages, represent a big selection to choose from. Users should be very careful if there is a complete set documentation accompanying the commercial FE program, that does not contain a problem of interest. Usually the numerical modeling of a brand new metal forming process leads into the brand new problems including undiscovered program bugs. If the simplicity of the numerical modelling is of the great importance to the user, one should stick to the commercially available FE programs where similar problems have already been solved.

6. REFERENCES

[1] H. J. Haepp, K. Roll, Mathematical modelling of metal forming processes in automotive industry, Advanced Technology of Plasticity, Vol. I Proceedings of the 6th ICTP, Sept. 19-24, 1999.

[2] J.C. Choi, Y. Choi, A study on the forging of external spur gears: upper bound analyses and experiments, Machine tools and manufacture ,38,1193-1208,1998.

[3] N.V. Hattangady, Coarsening of mesh models for representation of rigid objects in finite element analysis, Int. J. Numer. Meth. Engng. 44, 313-326, 1999

[4] MSC.Marc – Volume A Theory and user information (Version 2003) 2003 MSC.Software Corporation, 2 MacArthur Place, Santa Ana, CA 92707

[5] MARC – Volume A (Version K7), MARC Analysis Research Corporation, Palo Alto, California, Printed in USA, 1997.