under the auspices of
Regione del Veneto

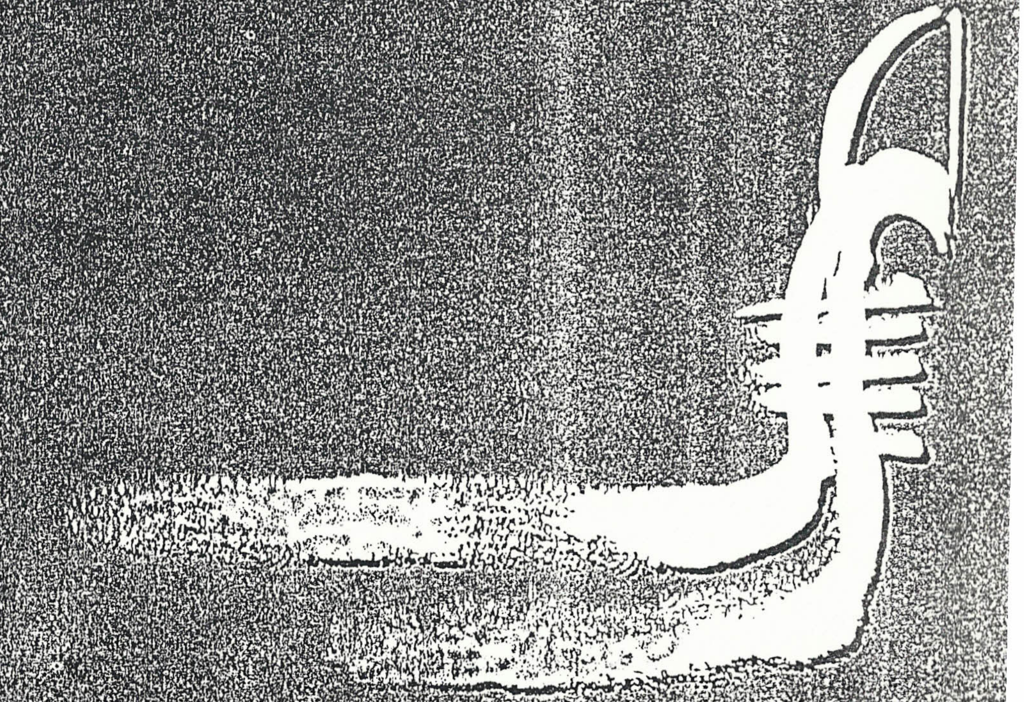
● [illegible]

---

14 th

International Forging Congress

Preliminary Program



Venice
27-29 September 1993
Technical visits:
30 September - 1 October

APPLICATION OF THE FINITE ELEMENT METHOD TO THE FORMING PROCESS OF A CV JOINT

G. Pontini (°), M. Marengo (°), G.A. Berti (*), L. D'Angelo (*)

(°) Teksid S.p.A., via Plava 74, Torino, Italy
(*) DIMEG - University of Padova, via Venezia 1, Padova, Italy

ABSTRACT

The current metal forming practice is largely based on experience and trial and error techniques. Major objectives of analytical modelling of metal forming is to become a source of the necessary information for proper design and control of metal forming processes and to help in reducing the number of expensive trials necessary to set up a new forming operation.

The paper is aimed at presenting the application of the finite element method to forming of a CV joint for the automotive industry. The emphasis is on the application itself rather than on the theoretical developments which make this simulation possible.

The code FORGE2 has been used, which is suitable for analysing 2D geometries, namely long or axisymmetrical products. For this reason, deviations from rotational symmetry have been neglected in the forging process simulation of the component.

1. NUMERICAL SIMULATION IN METAL FORMING

FEM-based codes are used in metal forming to simulate metal flow throughout a forming operation, assisting the forming engineer in establishing and optimising process variables and die design. If the simulation indicates that the selected design does not fill the die or if too much material is wasted, then another design can be selected and the computer simulation repeated.

The complexity of metalworking processes is such that FEM-based program are mainly used for developing comparative tests aimed at evaluating the influence of parameters on processes. In some cases they are successfully used to try to predict and eliminate flow induced defect in work materials [2].

One major difficulty in developing correct simulations of forming processes is evaluation of the large number of parameters influencing the problem. In particular, when deformation takes place at high temperatures, material properties can vary considerably with temperature. During a metalworking process, heat generation usually occurs, and if the dies are at considerably lower temperatures than the workpiece, the heat losses by conduction to the dies and by radiation and convection to the environment result in severe temperature gradients within the workpiece.

Thus, including temperature effects in the analysis of metal forming problems is very important.

For developing reliable simulations in metalworking it is necessary to have the following information [1]:

(i) flow stress data for materials to be formed at various (a) temperatures, (b) strains and (c) strain rates,
(ii) thermal and physical properties of die and deforming material in function of temperature,
(iii) heat transfer coefficients and friction data in function of temperature and pressure for interface conditions used (i.e. lubricant, coating, surface finish, etc.),
(iv) data on heat generation due to deformations and to slidings between deforming material and dies.

Kobayashi [3] presented an extensive review of a large number of numerical techniques available for the analysis of metal forming processes. Shabaik pointed out the distinction between the various constitutive formulations used to simulate the deformations of metals [4 - 6]. The following four categories can be defined:

1. Rigid Plastic,
2. Rigid Viscoplastic,
3. Elastic Plastic,
4. Elastic Viscoplastic.

In most metal forming processes strains reach large values of many percent, even a few hundred percent is quite common. When compared with less than one percent elastic strains, it becomes apparent that it is very reasonable to neglect the elastic contribution to the total strain. The major drawback is that residual stresses are not obtained at the end, but this is not a main concern in hot forging. The rigid viscoplastic material is an idealisation of an actual one, by neglecting the elastic response. The material shows the dependence of flow stress on strain rate in addition to the total strain and temperature. Also, it can sustain a finite load without deformation.

2. SHORT PRESENTATION OF THE FEM CODE

Based on the possible fields of application, FEM-based programs can be roughly divided into two main categories. On one side are multi-disciplinary programs, containing many material models, that, as a consequence, can be used for simulating many different kinds of process. As alternatives are programs usually including only one constitutive law for material modelling, addressed to solve one or few classes of problems (metal forming *or* injection moulding *or* thermal exchanges *or* dynamic analyses, etc.). Usually this second category of programs are more user-friendly also because fewer input data are needed.

FORGE2 vers. 2.0, developed at CEMEF (France), is the program used for the simulation presented. The chosen constitutive law is rigid viscoplastic. Such constitutive law allows simulation of processes for forming both metals and polymers.

It is assumed that, at the forging temperature, the work material behaves as rigid-viscoplastic and can be described by the following constitutive law [7]:

$$\sigma_S = 3^{\frac{m+1}{2}} \cdot K \cdot \dot{\bar{\varepsilon}}^{m}$$

where:

σ_S is the normal yield stress,
m is the strain rate coefficient,
K is the material consistency.

$$\dot{\bar{\varepsilon}} = \left(\frac{2}{3} \cdot \dot{\varepsilon} : \dot{\varepsilon}\right)^{\frac{1}{2}}$$

<:> is the tensor product operator

The dependence of the material consistency K versus temperature T and strain ε is defined by the following law:

$$K = C \cdot e^{\frac{\beta}{T}} \cdot \bar{\varepsilon}^{n}$$

where:

C is a constant,
β is the coefficient of temperature,
n is the strain hardening exponent,
T is the temperature.

$$\bar{\varepsilon} = \left(\frac{2}{3} \cdot \varepsilon : \varepsilon\right)^{\frac{1}{2}}$$

where <:> is the tensor product operator.

It is assumed that at the interface workpiece-tool the tangential stress is given by the following expression:

$$\tau = \alpha \cdot K \cdot \left|\nabla \vec{V}\right|^{p}$$

where:

$\nabla\vec{V}$ is the relative speed of the two surfaces.

3. PRESENTATION OF THE INDUSTRIAL CASE

It was simulated the forming process for one steel component on a Hatebur AMP 70 L press at the TEKSID factory in Turin. The component taken into account is a CV joint for car.

The study of the plastic deformation was aimed at describing the material and thermal flows during the process. Calculations were carried out on a Digital VAXStation 3100 at the Dipartimento di Innovazione Meccanica e Gestionale (DIMEG), University of Padova.

The part material is a C 53 steel. In the simulation the following values were used:

$C=0.85*10^3$ [Kg*s^{m-2}/mm]
$m=0.2286$

work material specific heat $Cp=7.076*10^8\left[\frac{mm^2}{°K \cdot s^2}\right]$

$\beta=4200$
$n=0$

As concerns material characteristics, the following values were used:

workpiece initial temperature $T=1200°C$

thermal conductivity $K=2.805*10^4\left[\frac{Kg \cdot mm \cdot °K}{s^3}\right]$

tooling temperature $T_t=300°C$

In the simulation friction stress is supposed to be the tangential yield stress for the work material, according to Von Mises model.

A mechanical crank press is selected, whose data are the following:

crank radius: $r=160$ mm
crank rotation speed: $\omega=2\cdot\pi\cdot\frac{rad}{s}$
connecting rod length: $l=800$ mm.

4. RESULTS OF SIMULATION OF THE PREFORMING AND FINISHING OPERATIONS

Results are presented of simulations of the forging sequence for a CV joint for car. The analysis is aimed at (i) highlighting how the plastic flow develops during the forging process, (ii) evaluating the strain and (iii) temperature distribution in the work material, and (iv) pressures on dies.

In Figure 1 is presented the mesh at the beginning of the preforming stage. Due to the rotational symmetry of the component, half section is analysed.

Postprocessing facilities of the program are used to show results of the numerical simulation of preforming and finishing operations for the component. Figures 2 to 4 show (i) the strain distribution, (ii) temperatures and (iii) pressures on die surfaces at the end of the preforming stage. In figures 5 and 6 the velocity field of work material is presented at two different stages of the process.

Figures 7 to 9 show (i) the strain distribution, (ii) temperatures and (iii) pressures on die surfaces at the end of the finishing operation. The velocity field of work material is presented in figures 10 and 11. Figure 12 shows the diagram of the forging load versus the upper die stroke in the finishing operation.

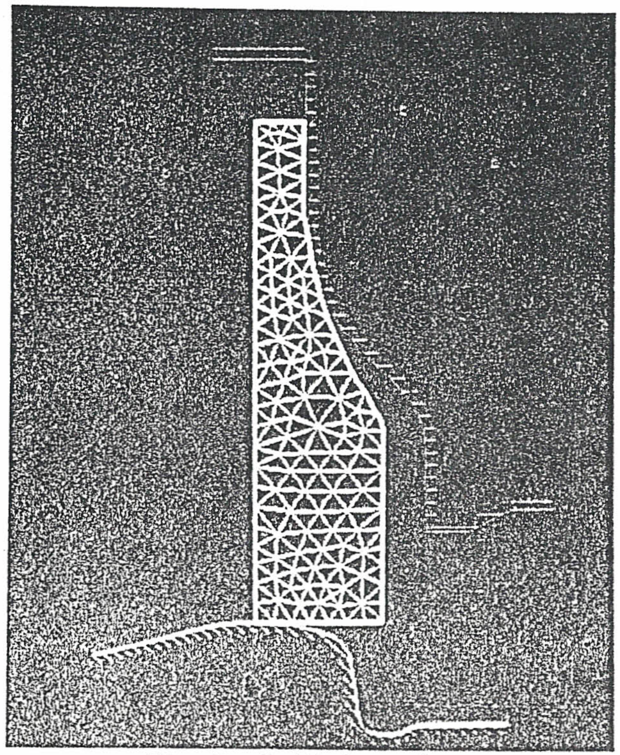


Fig. 1 - Initial mesh at the preforming stage

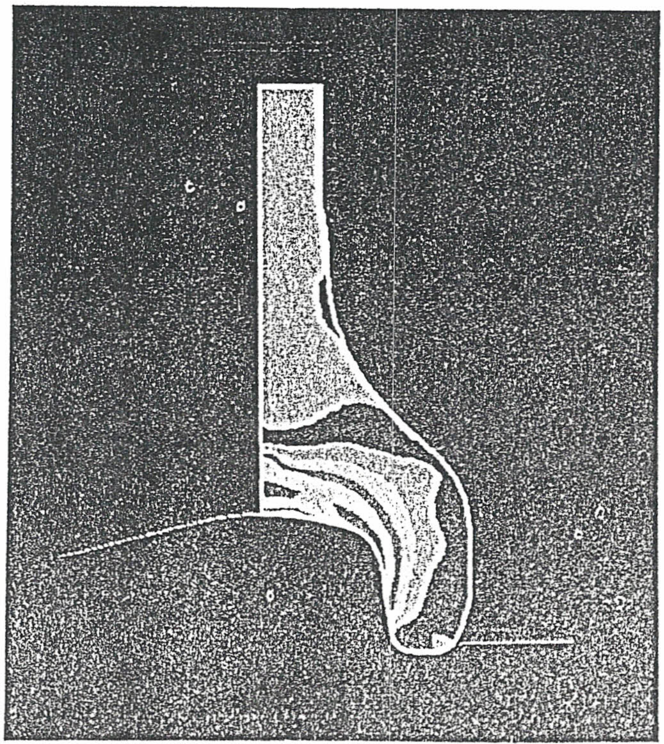


Fig. 2 - Strain distribution at the end of the preforming stage

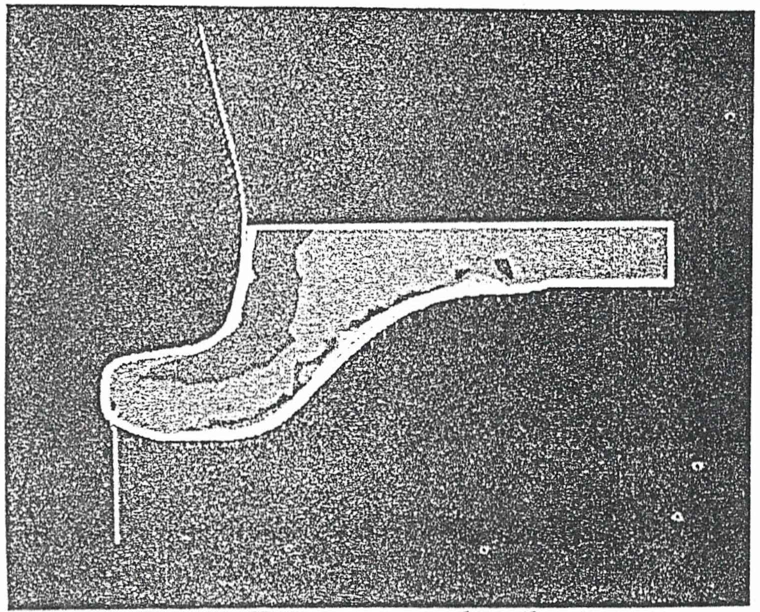


Fig. 3 - Temperature distribution at the end of preforming

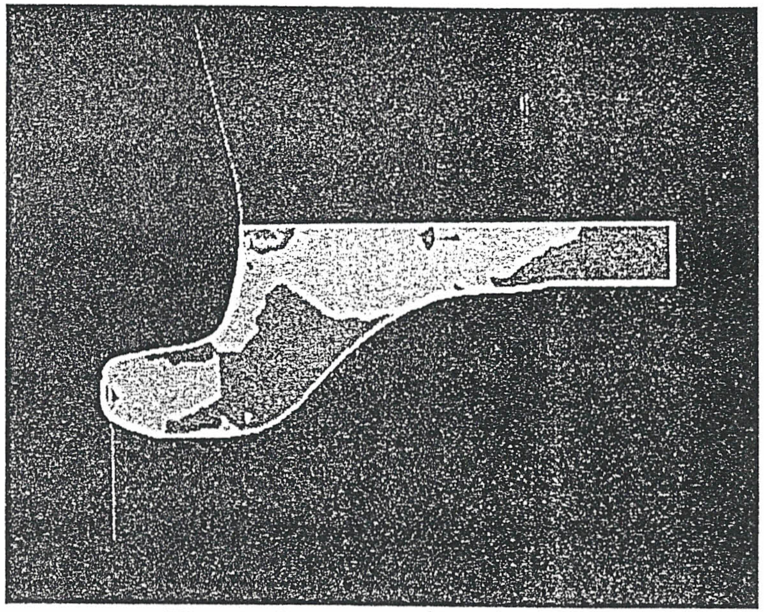


Fig. 4 - Pressure distribution at the end of preforming

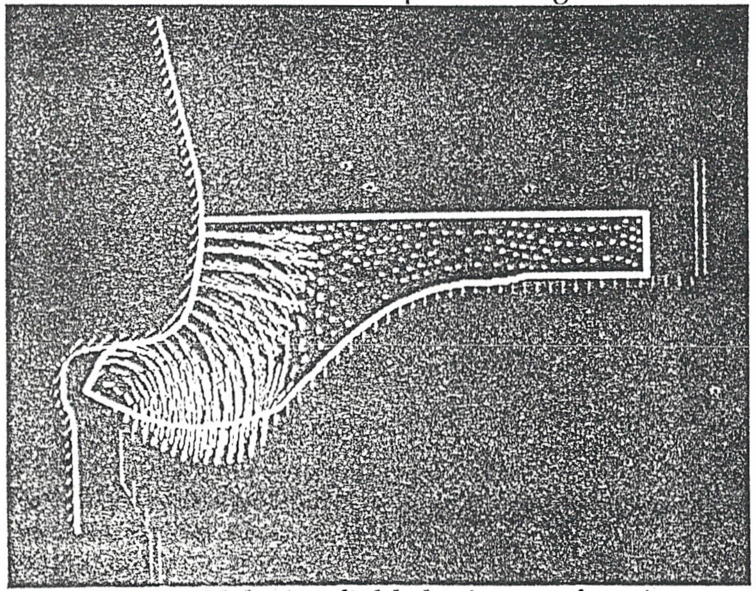


Fig. 5 - Velocity field during preforming

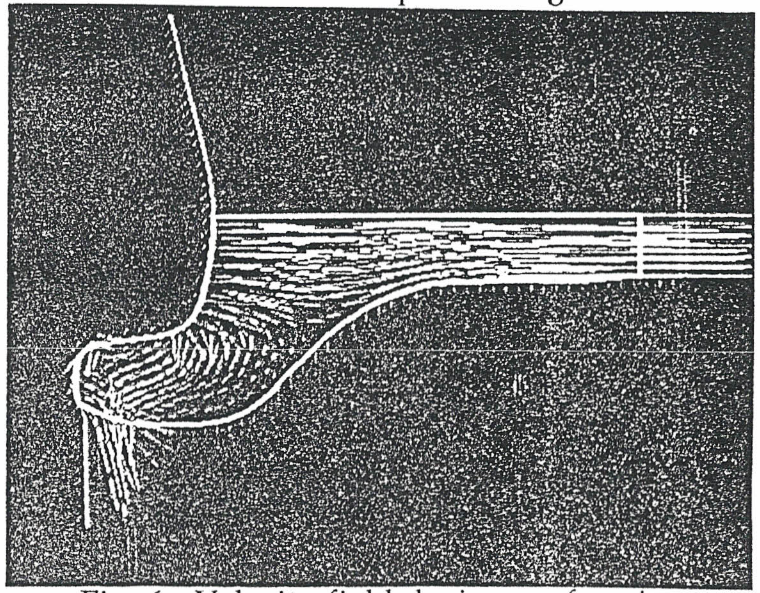


Fig. 6 - Velocity field during preforming

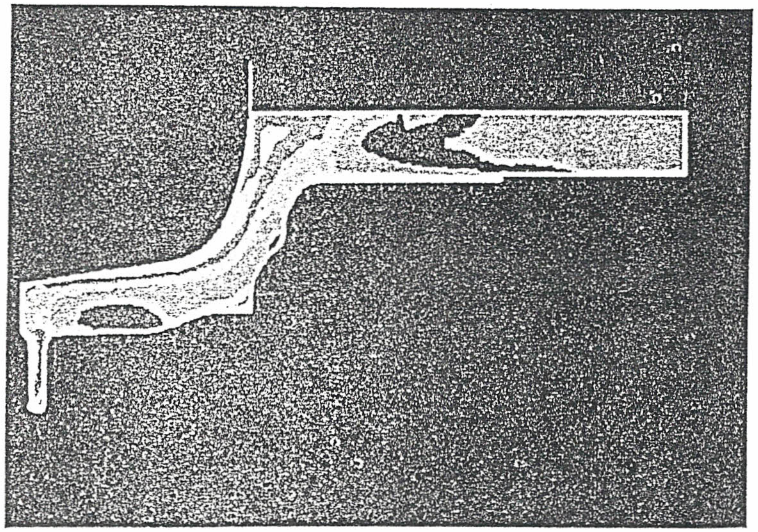


Fig. 7 - Strain distribution
at the end of finishing

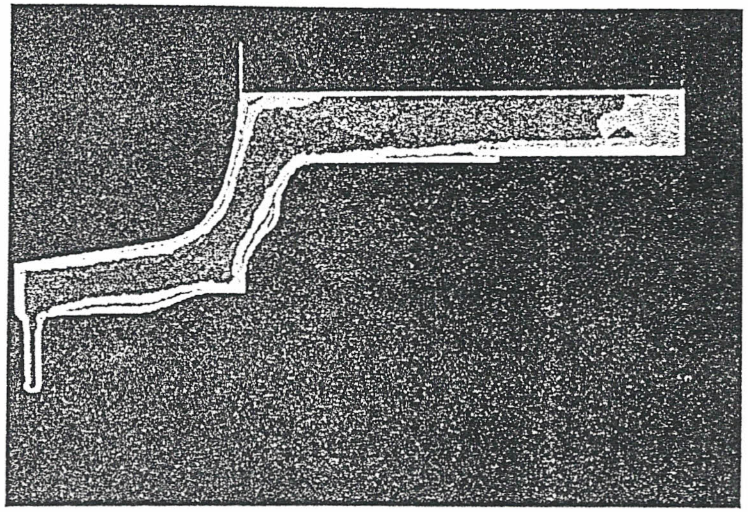


Fig. 8 - Temperature distribution
at the end of finishing

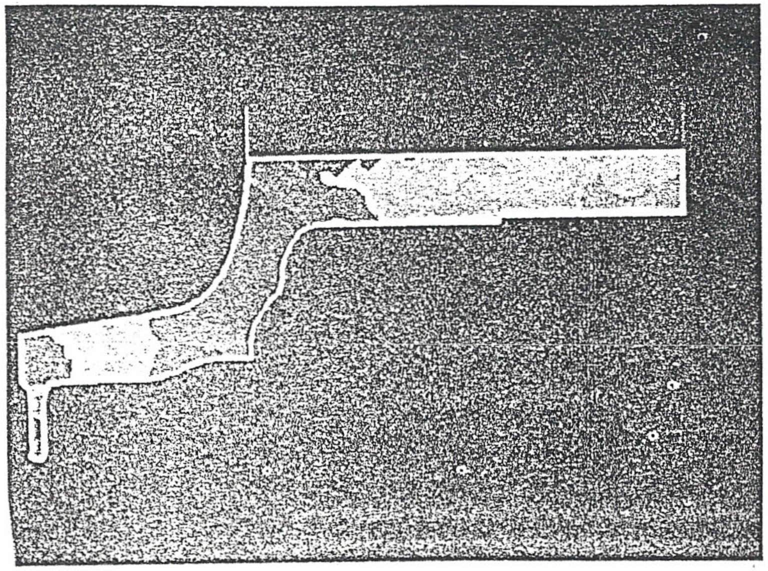


Fig. 9 - Pressure distribution at the end of finishing

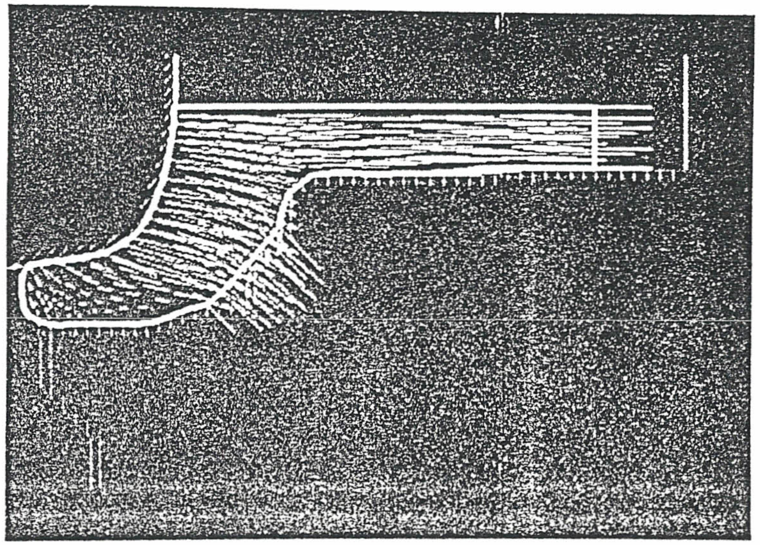


Fig. 10 - Velocity field during finishing

5. CONCLUSIONS

Results have been presented of simulations of the forging sequence of a CV joint for cars. The analysis was mainly focused on (i) highlighting how the plastic flow develops during the forging process, (ii) evaluating the strain and (iii) temperature distribution in the work material. Besides, pressures on dies and velocity of material points were analysed as this two factors heavily influence tool wear. Pictures show that the process develops regularly and no defects are highlighted, as also confirmed in the real process.

6. REFERENCES

[1] T. Altan, "Application of CAD/CAM in Precision Forming", *Steel Research*, Vol. 53, N°. 5, pag. 212 (1988).

[2] T. Altan, B. Jenkins, S. I. Oh, "Investigation on Defect Formation in a 3-Station Closed Die Forging Operation", *Annals of the CIRP*, Vol. 38, N°. 1, pag. 243.

[3] S. Kobayashi, "The Mechanics of Plastic Deformation in Metal Forming", *Proc. of the U.S. Natl. Congr. of Appl. Mech.*, pag. 115 (1978).

[4] A. H. Shabaik, "Mechanics of Plastic Deformation in Metal Forming Processes: Experimental and Numerical Methods", Current Advances in Mechanical Design and Prod., Ed. G.S.A. Shawki and S.M. Metwalli, (Proc. Conf.), pag. 269 (1981).

[5] A. H. Shabaik, "Analysis of Forming Processes: Experimental and Numerical Methods", ASME (Applied Mechanics Division) Vol. 28, *Applications of Numerical Methods to Forming Processes*, presented at ASME Winter Annual Meeting, pag. 15 (1978).

[6] A. H. Shabaik, "Plane Strain Extrusion and Forging", Numerical Methods in Industrial Forming Processes", (Proc.) Ed. J.F.T. Pittman, R.D. Wood, J.M. Alexander, O.C. Zienkiewicz, pag. 321 (1982).

[7] "FORGE2 Product Description" Manual, pag. 13.

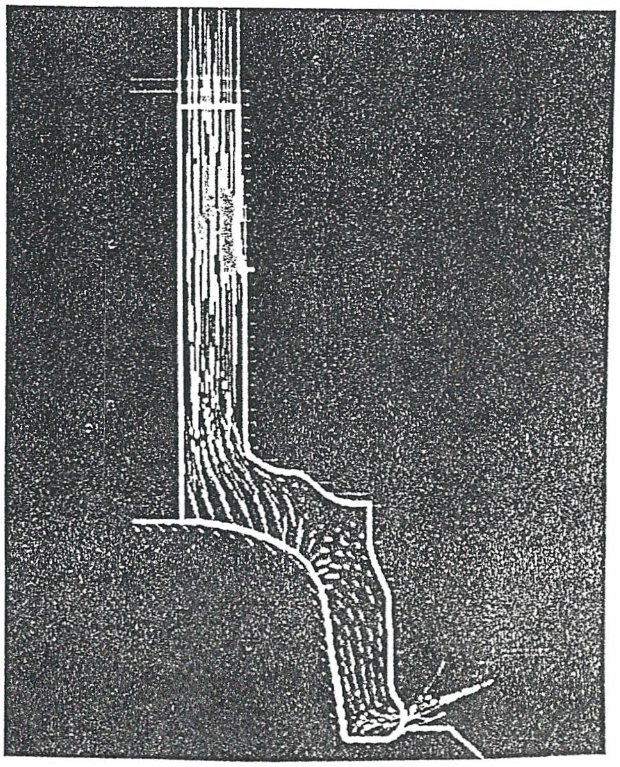


Fig. 11 - Velocity field during the finishing operation

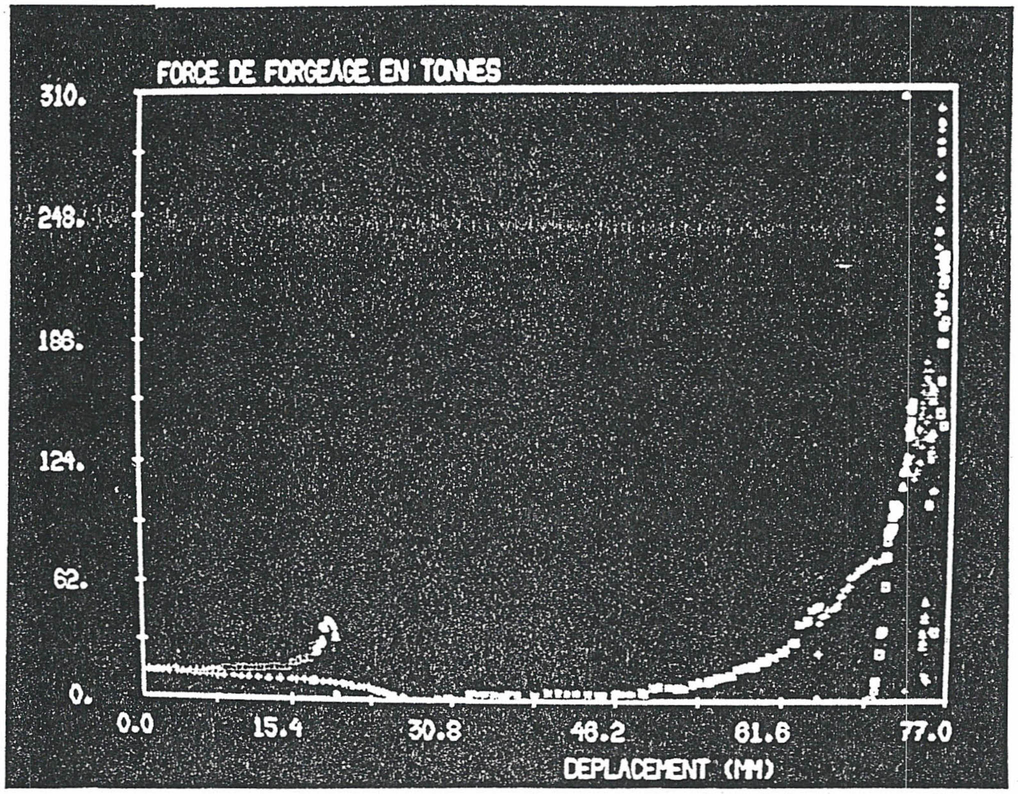


Fig. 12 - Diagram of the forging load in the finishing operation