FINITE ELEMENT SIMULATION OF THE TESLA-CAVITY HYDROFORMING PROCESS

GONIN I.*, KAISER H., PROCH D., SINGER W.

DESY, *Hamburg, Germany*
INR RAS, *Moscow, Russia

SUMMARY

The fabrication of TESLA-cavity by hydroforming process was analyzed by using of numerical simulation. Finite element code ANSYS [1] is used. Nonlinear elasto-plastic behaviour of stress-strain curve and isotropic hardening rules are taken into account. The analysis gave some information about the effects of applied loads in wide range on the deformation behaviour. The optimal distribution of applied loads in terms of internal pressure - axial displacement was obtained. The choice of outer diameter and thickness of initial tube are discussed.

INTRODUCTION

Hydroforming is one of the techniques, by which a initial tube is deformed in a die cavity by internal pressure and axial compression. Material deformation during the forming process, stress distribution and thickness in the final shape are influenced by the process parameters, such as force applied, amount of axial displacement and hydraulic pressure. The surfaces contact conditions between tube and constrain is influenced also. In this paper the hydroforming process of one cell of the TESLA-cavity is simulated by using finite element code ANSYS.

NUMERICAL MODEL

Rate-independent plasticity theory is used, i.e the plastic strains are assumed to develop instantaneously, that is independent of time, and characterized by the irreversible straining that occurs in a material once a certain level of stress is reached.

All calculations were done under the following conditions:

1. material behavior is characterized by MISO - Multilinear ISOtropic hardening option. Fig. 1 shows the stress-strain curve for Niobium, which was used in the calculations.
2. von Mises yield criterion for multi-component stresses was used.

The yield criterion determines the stress level at which yielding is initiated. The Yield criterion is represented as a function of the individual components $f(\sigma)$, which is interpreted as an equivalent stress σ_{eqv}.

$\sigma_{eqv} = f\{\sigma\}$, where $\{\sigma\}$ is the stress vector.

When the σ_{eqv} is equal to a material yield parameter σ_y, the material will develop plastic strains. Isotropic work hardening option means, that the yield criterion is changed with work hardening as described in [1].

Von Mises or equivalent stress is given by:

$$\sigma_{eqv} = \sqrt{\frac{1}{2} \cdot \{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\}} \qquad (1)$$

where principal stresses $(\sigma_1, \sigma_2, \sigma_3)$ are calculated from the stress components $(\sigma_\varphi, \sigma_r, \sigma_z)$ by the cubic equation:

$$\begin{vmatrix} \sigma_\varphi - \sigma & \sigma_{\varphi r} & \sigma_{\varphi z} \\ \sigma_{\varphi r} & \sigma_r - \sigma & \sigma_{rz} \\ \sigma_{\varphi z} & \sigma_{rz} & \sigma_z - \sigma \end{vmatrix} = 0 \qquad (2)$$

In our case $\sigma_{\varphi r} = \sigma_{\varphi z} = 0$, and $\sigma_{rz}, \sigma_r << \sigma_z, \sigma_\varphi$, therefore we can rewrite (1) as:

$$\sigma_{eqv} \approx \sqrt{\sigma_\varphi^2 + \sigma_z^2 - \sigma_\varphi \cdot \sigma_z} \qquad (3)$$

For each isoline both values σ_φ and σ_z have a maximum:

$$\sigma_\varphi = \frac{\sigma_z}{2} \text{ and } \sigma_z = \frac{\sigma_\varphi}{2}.$$

NUMERICAL SIMULATIONS AND DISCUSSION

Starting from tube with an intermediate diameter R (fig. 2a), two series of metalforming simulations were done.

1. by means of an external force load, the initial tube was deformed to achieve the desired intermediate shape (fig. 2b). Different distributions of applied force F(z) (fig. 2c) were used.

2. a tube with new shape was loaded by internal pressure and axial displacement to receive the final shape, which is defined by constraint (fig. 2d). For this step the standard Coulumb friction with constant coefficient $\mu = 0.01$ was assumed over the contact surfaces.

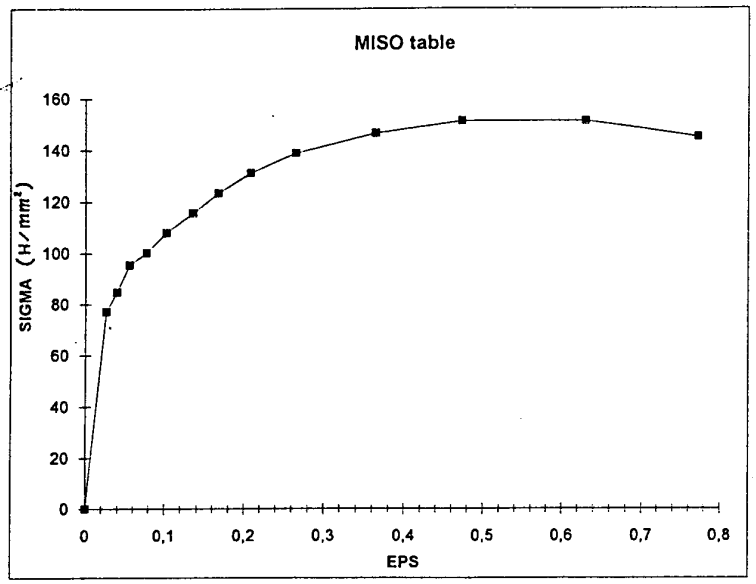

Fig. 1 Stress-strain curve for Niobium.

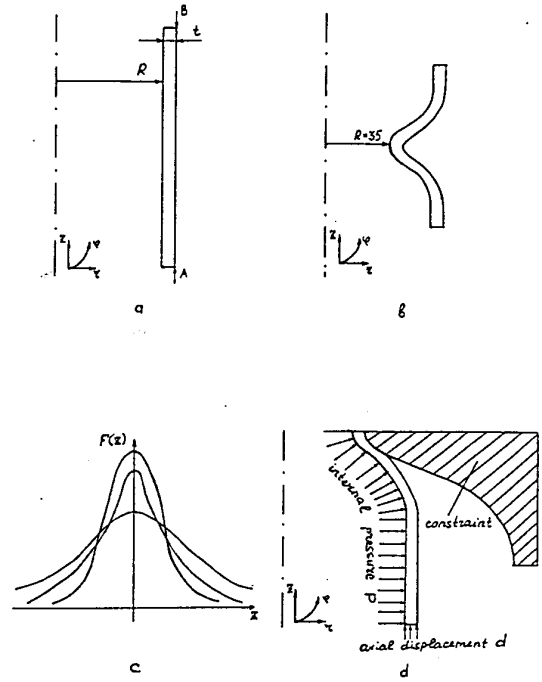

Fig. 2 a. Initial tube with intermediate radius R.
b. Intermediate shape after the first stage of forming process.
c. Distribution of force loads.
d. Final shape, defined by constraint.

For the first step, the tube shape after the metalforming process depends on internal tube radius (R), tube thickness (t) and applied force F(z). Choosing R and t one can find the F(z) distribution to achieve the desired shape. Fig. 3 illustrates the final distribution of the σ_{eqv} for R=65mm, t=3mm.

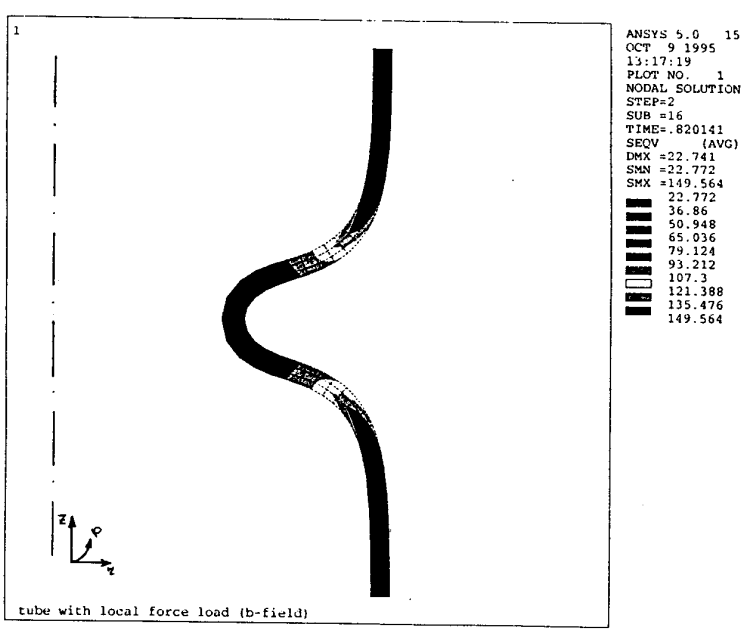

Fig. 3. σ_{eqv} distribution after first stage (R=65mm).

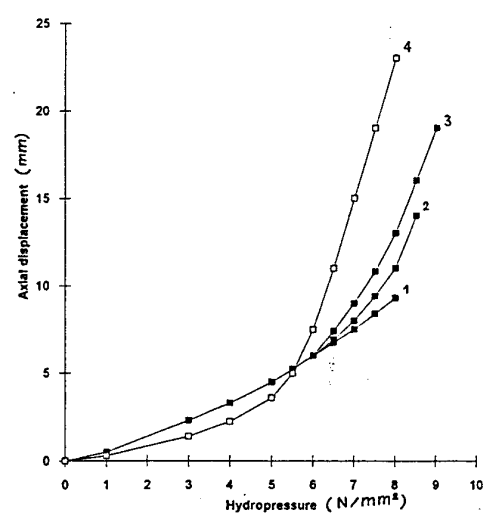

Fig. 4 d(p) paths, which were used in the second stage of simulation.

The simulation described above shows the possibility to achieve the desired tube shape by choice of appropriate force load, but the value of σ_{eqv} is near the limit (fig. 1). Choosing R=60mm and R=55mm one can obtain the desired shape with $\sigma_{eqv} \approx 0.95\sigma_{lim}$ and $\sigma_{eqv} \approx 0.9\sigma_{lim}$.

The second step of simulation is a more complicated process, due to two independent loads applied - internal pressure and axial displacement. Fig. 4 shows different step by step curves in the plane d-p, which were used in the process of simulation, where d are values of axial displacement (mm), p are values of internal pressure (N/mm^2).

The final shape of the tube after the hydroforming process depends on prior of loads applied, e.g. on the path in d-p plane. Instability started, when σ_{eqv} exceeded the stress limit, which depends on the stress-strain curve. Fig. 5 shows the final distribution of the displacement vector for path 2 in fig. 4 before instability was initiated. As shown by the simulation, to increase the displacement in r-direction it is necessary to use the d-p path, which leads to stress components σ_φ and σ_z values near to $\sigma_z \approx 0.5\sigma_\varphi$. Fig. 6 shows the distribution of σ_{eqv} for the last point of path 4 (fig. 4). Using this path one can obtain the desired shape, but the value of σ_{eqv} is near the limit.

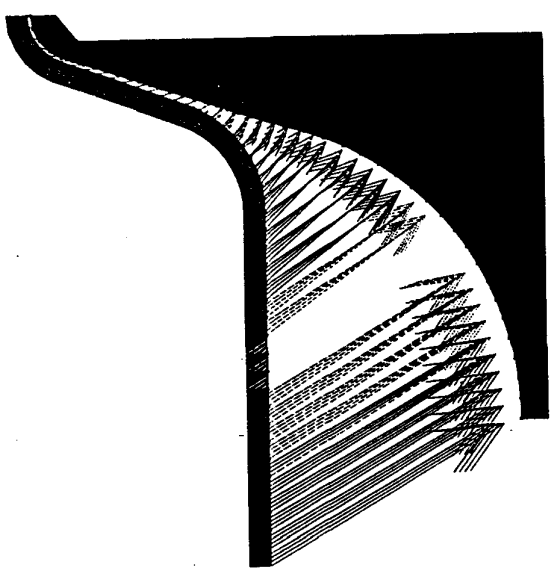

Fig. 5 Final distribution of displacement vector for path 2 (fig.4).

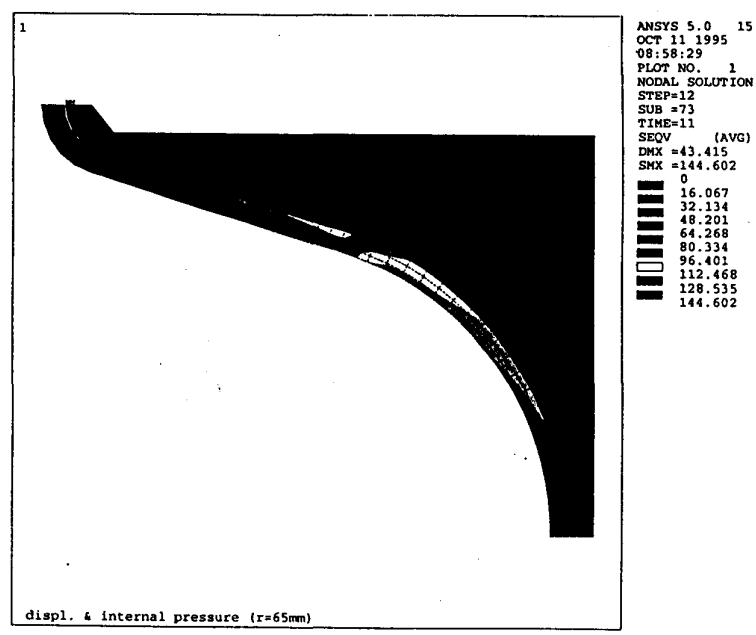

Fig. 6 Distribution of σ_{eqv} for the last point of path 4 (fig.4).

The simulations described above show, that one can receive the desired shape, using an intermediate tube diameter by means of two metalforming stages. In both cases the final values of σ_{eqv} are near the limit. In practice one can achieve the value of σ_{eqv} not greater then 0.8-$0.9\sigma_{lim}$. Under this condition, intermediate annealing is necessary.

Fig. 7 show the distribution of σ_{eqv} and tube shape after a few steps of the hydroforming process (initial tube with R=55mm was chosen and was deformed by means of an appropriate load). Let's assume that the annealing process removed all stresses. Using this new shape as initial one for the further process, one can obtain the desired final shape, which is shown in fig. 8. In both stages the maximum of σ_{eqv} is less then $0.9\sigma_{lim}$.

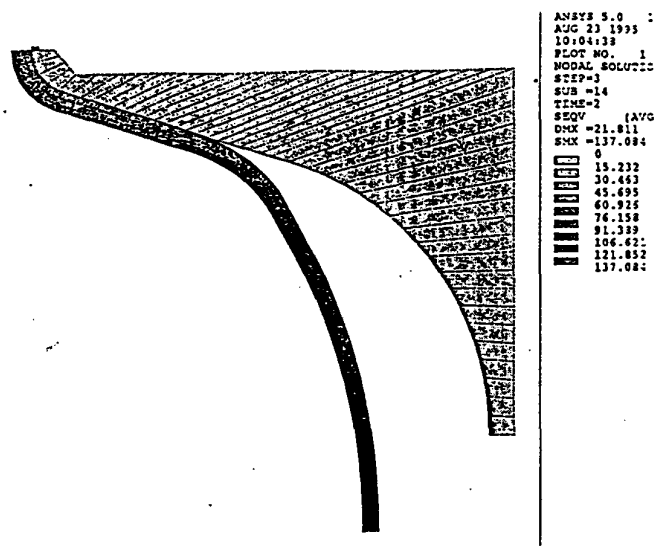

Fig. 7 Distribution of σ_{eqv} before annealing for the initial tube (R=55mm).

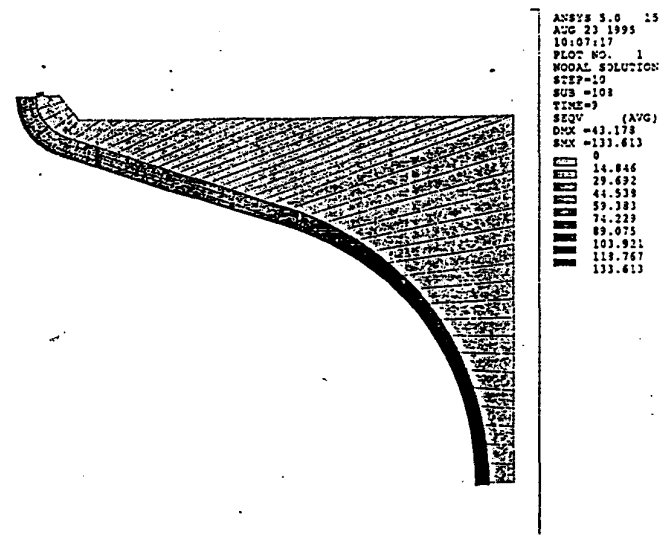

Fig. 8 Final distribution of σ_{eqv} for the initial tube (R=55mm).

CONCLUSION

The finite element approach can be used to simulate a wide range of hydroforming processes with reliability and efficiency. The ANSYS analysis gave some useful information to predict the effects of combination of process parameters, such as internal pressure and axial compression. Using tube with R=65mm and loads according to path 4 (Fig. 4) it's possible to obtain the desired shape without intermediate annealing. In this case the critical values of stresses are achieved. Further calculations will be done to answer such questions as influence of rate dependence of strain on hydroforming process, influence of nonuniform thickness of tube and so on.

REFERENCES

[1] ANSYS User`s Manual, V.1, Swanson Analysis Systems Inc.,1992.