Journal of Achievements in Materials and Manufacturing Engineering

VOLUME 24
ISSUE 1
September
2007

Experimental and finite element simulation of formability and failures in multilayered tubular components

M.S.J. Hashmi *
co-operating with
M.D. Islam, A. G. Olabi
School of Mechanical and Manufacturing Engineering,
Dublin City University, Glasnevine, Dublin-9, Ireland
* Corresponding author: E-mail address: din_buet@hotmail.com

Received 06.04.2007; published in revised form 01.09.2007

Analysis and modelling

ABSTRACT

Purpose: Optimization of the operating conditions is one of the most significant studies in the hydroforming process, which affect the forming of successful components.
Design/methodology/approach: Finite Element simulations have been carried out using the commercial finite element package ANSYS in order to predict the most efficient and acceptable operating condition for certain material properties of multilayered blank and initial blank geometry.
Findings: This paper studies the hydroforming process involving combined (axial feed and internal pressure) and multi stage non-linear loading action theoretically and experimentally for a multi layered tubular blank placed in a pre-shaped die block.
Research limitations/implications: Finite Element simulation of a multi layered tube hydroforming and optimisation of the best forming conditions based on a number of simulations were carried out to avoid wrinkling or bursting of the tubular blanks.
Originality/value: Experimental validation for a certain simulation has been presented.
Keywords: Numerical techniques; Multi-layered; Hydroforming; Optimization; Failure

1. Introduction

Hydroforming is one of the unconventional metal forming process which is widely used in order to form complex shapes. Compared to other traditional manufacturing methods (such as stamping), hydroforming offers better part quality (tighter tolerances, increased rigidity) with lower manufacturing costs resulting from reduced number of forming and assembly operations. Moreover, with the advent of new technological capabilities, particularly more complex and improved control systems, hydroforming has become a flexible, reliable, and hence an attractive technology. Now a day, bulge forming is widely used in making tubular parts of different configurations used in automotive and aircraft industries, household appliances, and other applications.

In hydroforming process the material is biaxially stretched by applying internal pressure with the help of a hydraulic medium. In advance hydroforming process axial load is added with the internal pressure to improve the formability of the process. During the forming action the tube blank is placed between two die halves and the tube ends are held by two plungers. The axial load is applied by the plungers as a punch. There are holes through both plungers and hydraulic fluid is pumped through these holes in order to build up the internal pressure inside the tube blank. The plungers are pushed inward to provide the compressive axial load. Both the internal pressure and the axial compressive loads are applied simultaneously, which results a greater deformation.

In the hydroforming process there are two modes of defects which may appear. If the axial feed is too high in comparison with the internal pressure then there is the chance of formation of wrinkle and buckling of the tubes. On the other hand if the internal pressure is too high compared to the end axial load there would be excess thinning of the tube resulting in bursting of the tube. Thus bursting, wrinkling and buckling are the main failure modes for any tube hydroforming process. In order to apply the process successfully an optimum loading condition should be established to avoid all the instabilities and failure modes which will enable proper deformation of the product. Hydroforming process is one of the highly complicated processes to analyse, the analytical methods for plastic deformation in hydroforming have produced limited understanding of the process. Numerical analysis and simulation have the potential to provide a much deeper understanding of the process and hence allow for better design of process equipment and end products. Recently Islam et al [1-2] verified hydroforming of a multi-layered tubular components and describe the mechanical stresses in the forming products. Lang et al. [3] carried out experimental and numerical investigations into the very thin layer in the middle. Ray and McDonald [4] studied finite element analysis of simple X and T-branches using hydroforming processes and experimentally verified the results. McDonald and Hashmi [5-6] studied Three-dimensional finite element simulation of bulge forming. Ahmed and Hashmi [7] verified three-dimensional finite-element simulation of bulge forming. Hwang et al. [8] analyzed tube hydroforming in a square cross-sectional die. Altan et al. [9] carried out application of two dimensional (2D) FEA for the tube hydroforming process.

2. FE analysis

2.1. Modelling

In metal forming analysis using finite element method good modelling of deformed body is important in order to achieve an accurate solution. In this work 3-D [10] models have been developed using ANSYS pre-processor with structural solid brick elements for analysing the process with ANSYS implicit and explicit finite element codes. Since it was considered that during the deformation the die would not deform, so the die surface was considered as rigid. In this work a mapped mesh has been created for both brass and copper tubes. In order to build a fine mesh the tube was divided by using the mesh tool and line set. Volume meshing command was applied for both tubes. In contact analysis, two points should be considered, first the contact and secondly the target. In ANSYS simulation, it is possible to solve both implicit and explicit contact analysis.

The model consists of both rigid-to-flexible and flexible-to-flexible. Since it was assumed that the die surfaces are rigid and only the tube will deform, so rigid-to-flexible contact element type 174 has been created between die surface and the external surface of the brass tube. Contact surfaces between the tubes are modelled as flexible-to-flexible contact pair. This is one of the common types of contact pair at which both the surfaces would be deformed. An elastic coulomb friction law was assumed and the value of the coefficient of friction was assigned as 0.15. The second contact pair has been created between the inner and outer tube contact surfaces. In this case the inner surface of the outer tube has been considered as the target surface and outer surface of the inner tube was assumed as contact surface. During the simulation both surfaces would be deformed simultaneously. A sticky friction condition has been assumed and the maximum value of the coefficient of friction has been applied 0.577. The pressure was applied on the inner surfaces of the internal copper tube and the axial feed was applied at the tube ends. The die was constrained in all degrees of freedom as the die surfaces have been assumed to be rigid. The tube ends were allowed to move only along the axial direction. Experimentally the axial feed was applied by end punch but in the simulated model the axial feed was applied as displacement along the positive X direction in the global co-ordinate system. As a result of the constraints tube end nodes would not be able to move along the normal direction of the circumference of the blank.

2.2. Simulation

An elasto-plastic (bilinear isotropic hardening) nonlinear material property was used with the following material properties. The outer tube was commercial brass and the inner tube was copper. Mechanical properties of brass tube are as: Ultimate Tensile Strength 540 MPa, Tensile Yield Strength 170 MPa, Breaking strain 40%, Young's Modulus 105 GPa, Poisson's Ratio 0.34, Tangent Modulus 720 MPa, and density 8.44 $g{\cdot}cm^{-3}$. Mechanical properties of copper tube was as : Ultimate Tensile Strength 470 MPa, Tensile Yield Strength 365 MPa, Breaking strain 20%, Young's Modulus 110 GPa, Poisson's Ratio 0.32, Tangent Modulus 552 MPa, and density 8.81 $g{\cdot}cm^{-3}$. The thickness of the brass tube was 1 mm and the internal copper tube thickness was as 0.9 mm. Total thickness of the model was 1.9 mm. A number of simulations have been carried out applying different boundary conditions. First solution was carried out for a X branch by 10 mm axial feed and 50 MPa internal pressure. The solution was completed by 7 sup steps in 0.6275 second. The maximum von Mises stress over the deformed tube is 500 MPa which is well below the ultimate stress of the materials of the tube. The predicted branch height is 2.6 mm. It was clear that the stress over the tube is uniformly distributed and the location of the maximum stress is the bottom of the formed branch.

Second simulation was carried out for 10 mm axial feed and 60 MPa internal pressure. In this case after a time of 0.6875 seconds the von Mises stress developed in the tube exceeds the ultimate stress of the materials. Maximum von mises stress is between 501 to 543 MPa after a time of 0.6875 second. The maximum ultimate stress for the outer tube materials was 540 MPa, for this reason it has been considered that at this stage the solution is within the safe limit just before the failure of the materials. The location of the maximum stress is at the same region as the previous solution. The maximum branch height for this solution is 3.1 mm. Next solution was carried out for 10 mm axial feed and 80 MPa internal pressure. The maximum von Mises stress over the simulated tube is 475 MPa and the branch height for this simulation is 3.7 mm. Figure 1 shows the von Mises stress distribution over the simulated X branch for 100 MPa internal pressure and 10 mm axial feed. It can be seen from the figure that over the branch area there is a higher tensile stress than the undeformed part of the tube. It is because of applying higher internal pressure compared to the axial feed. It is very important in the hydroforming process to balance the combination of internal pressure and the axial feed in order to obtain good products. It is clear that the bulge height is 3.92 mm. Figure 2 shows the von Mises stress distribution for 150 MPa internal pressure and 10 mm axial feed and he maximum von Mises stress in the simulated bilayered tube is 484 MPa.

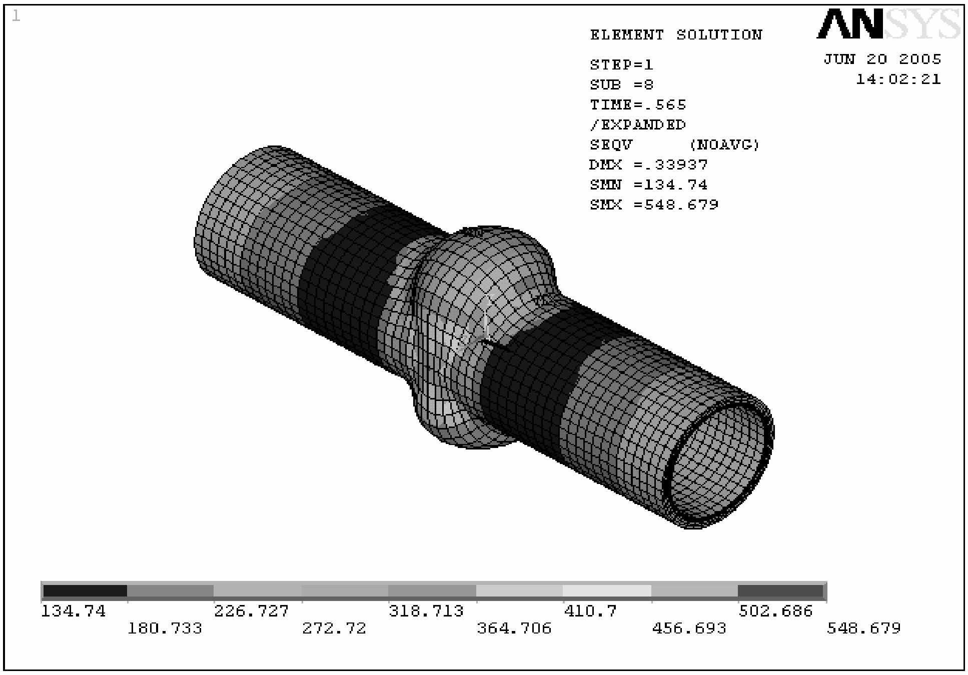


Fig. 1. Von-Mises stress distribution over the simulated X branch by 10mm axial feed and 100 MPa pressure

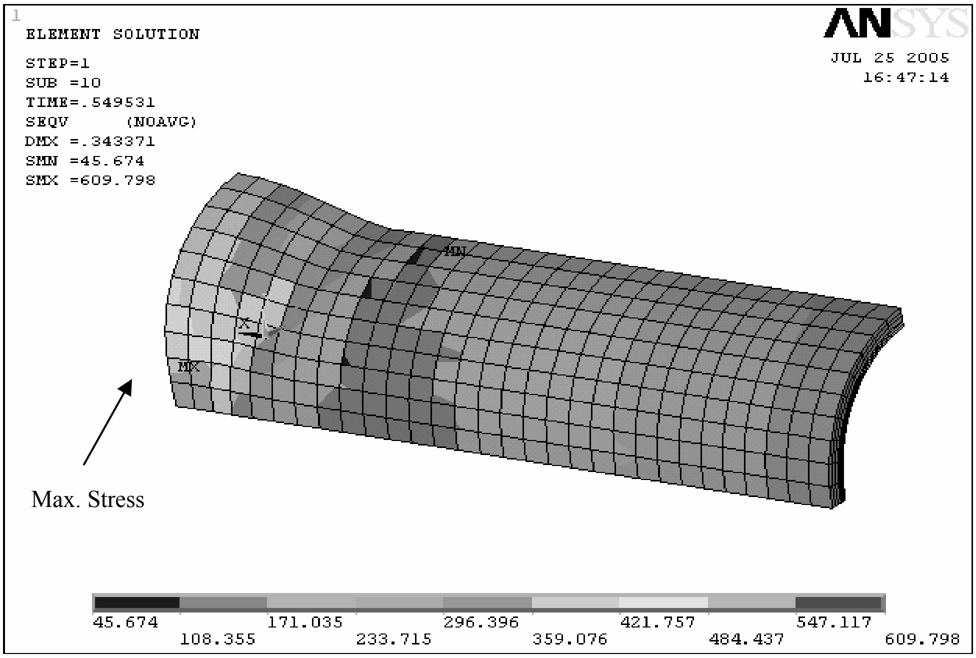


Fig. 2. Von-Mises stress distribution for the simulation of X branch by 10mm axial feed and 150 MPa pressure

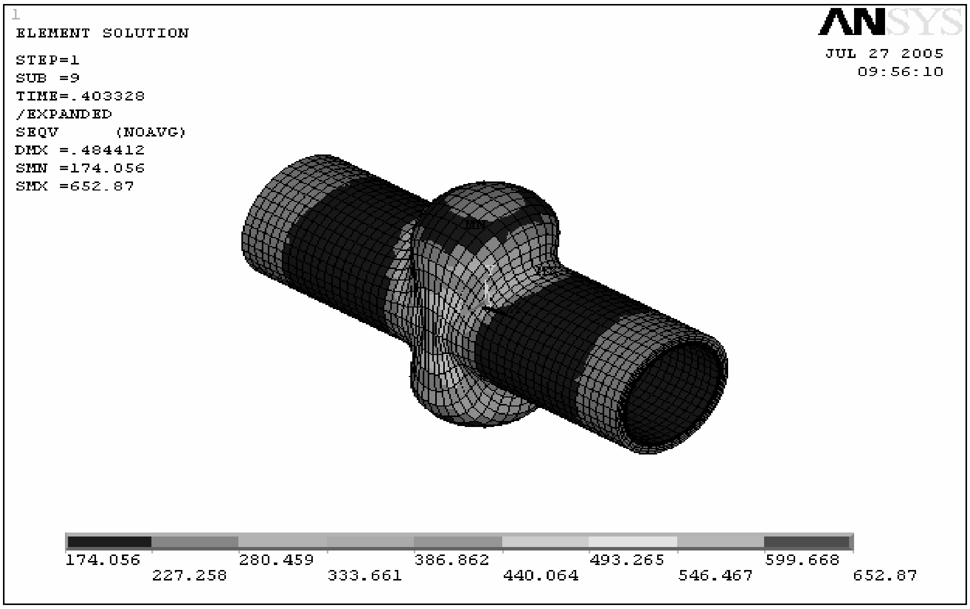


Fig. 3. Von-Mises stress distribution for the simulation carrying out by 20mm axial feed and 150 MPa pressure

The figure shows only the one-eight symmetry of the simulated model and indicated the location of the maximum stress in the tube. To study the effect of the axial feed for a fixed maximum internal pressure, a further solution has been run by applying 150 MPa internal pressure and the axial feed was increased to 20 mm instead of 10 mm. Figure 3 shows the von Mises stress distribution over the branch area of the formed product. A comparison between figure 2 and 3 where the simulation were carried out for the same pressure but different axial loads. It is clear that the stress over the branch area is lower for higher axial feed. From figure 3 it can be stated that bulge top flatter than other solutions with smaller axial feed. It shows that the maximum bulge height is 13 mm.

3. Experimental validation of the process

The hydro bulge forming machine in DCU is capable of producing a maximum internal hydrostatic pressure of 41.7 MPa. This value was obtained through previous tests carried out on the machine on copper specimens. The experimental tests were carried out with a maximum internal pressure of 37.5 MPa. To verify the analytical results experimental tests were carried out with the same boundary condition. Figure 4 depicts a typical formed product by hydroforming machine. It is clear from Figure 4 that a moderate deformation occurred during the hydroforming process. Figure 5 shows the loading pattern used in the hydro bulge forming machine. This loading pattern has been automated by Lab View program with the bulge forming machine. The graph shows the increase of internal pressure and axial force with respect to time. The increase in displacement from 0 to 53 at the start of the process which represents the plunger's movement to make contact with the multi layer tube. As the graph shows there is some displacement of the plungers after this initial contact. A further simulation has been carried out in order to compare between the experimental and analytical results. Simulation has been carried out for the same dimension of an X branch model as obtained experimentally in Hydro Bulge forming machine. The maximum equivalent von Mises stress is 493 MPa and the maximum branch height is 3.56 mm. The comparison between the simulation results and the practical results from the test can be observed in figure 6 below. The left hand side part of the figure represents the simulated result carried out by 10 mm axial feed and 37.5 MPa internal pressure.

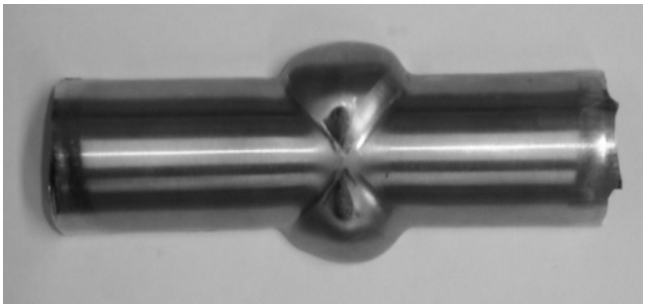


Fig. 4. Multi-layer Material Deformation at 37.5 MPa internal pressures and 10 mm axial feed

The branch height for this simulation is 3.56 mm where the developed equivalent stress of the formed tube was well below the ultimate stress of the materials. The right hand side part of the figure was formed in the hydroforming machine by applying the same axial feed and internal pressure. The branch height of the formed X branch in the hydroforming machine was 3.61 mm which is slightly greater than the simulated height.

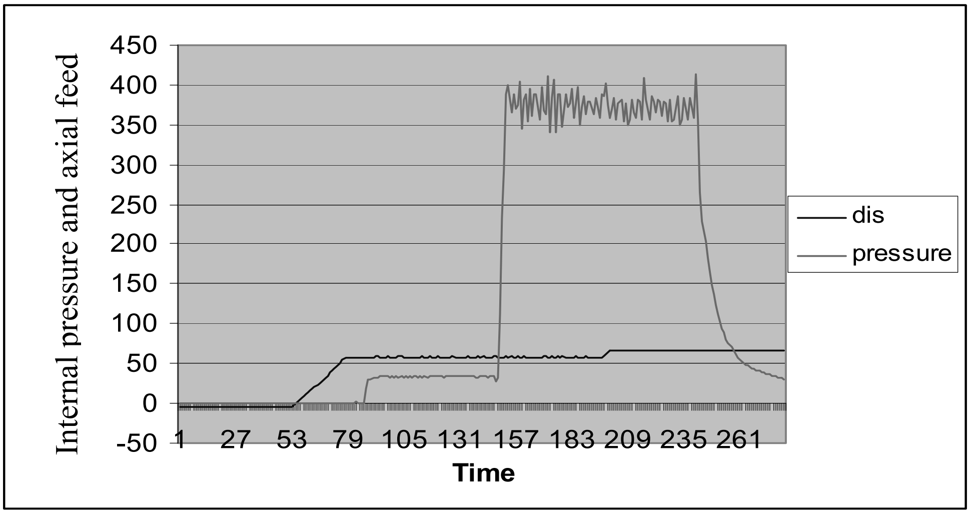


Fig. 5. Loading profile on the hydro bulge forming machine at 37.5 MPa internal pressure and 10 mm axial feed

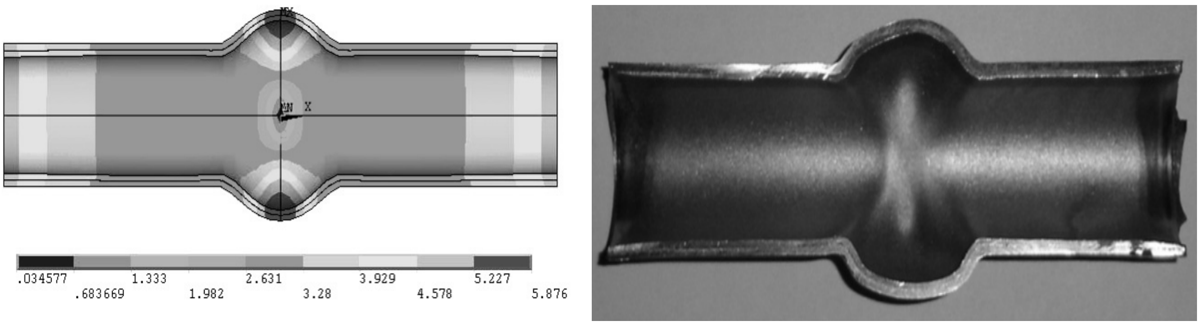


Fig. 6. Comparison between simulation and experimental results

4. Failures in hydroforming process

One of the significant purposes of the hydroforming process simulation is to get marketable good products for a certain loading condition. In order to get acceptable components, the load path should be controlled. The main failures of the hydroforming process which involves axial feed and internal pressure, are wrinkling, buckling or bursting, thinning and thickening. Due to an excessive axial feed in comparison to the internal pressure the tube could fail by buckling or wrinkling. Alternatively, if the pressure is too high with respect to axial feed there would be a chance of bursting the tube. Thus the forming load has to be adjusted properly in order to obtain a successful formed component with the hydroforming process.

A simulation is carried out for a high internal pressure (200 MPa) and a low axial displacement (10 mm) compared to the applied pressure. At this pressure the tube burst due to the excessive thinning in the top region of the branch. Figure 7 depicts the failure of the multi-layered tube by bursting due to high internal pressure.

By applying a high internal pressure compared to the axial feed, the material is not able to flow uniformly over the length of the tube, because the internal pressure is neutralized by the reaction force of the rigid die surface in the tube, except the die hole area. A high internal pressure can deform the tube near the hole of the die branch; as a result materials in this region experience more stretch than any other area of the tube. Due to the continuous action of the internal pressure the elongation increases continuously as long as the pressure exists at the internal surfaces of the tube. Because of the application of a relatively small axial feed the material supply is not sufficient towards the die hole, where the top area of the branch became thinner and at the end the tube burst.

Figure 8a and 8b shows a typical wrinkling failure of the hydroforming process. This solution has been carried out in order to observe the wrinkling failure by applying a lower internal pressure (40 MPa) and a high axial displacement (30 mm) compared to the internal pressure. In hydroforming processes which involve end axial and internal pressure, it is desirable that the end axial feed over the process time should be as high as possible, to allow as much material as possible to be fed into the forming zone. This excess material pushed into the deformation zone helps in maintaining the wall thickness near the highly strained areas within the desired limits. However, the actual value of the axial feeding in the process is limited by the occurrence of wrinkles or buckling. Excess material pushed into the forming zone may result either in buckling or excess wall thickening in certain zones.

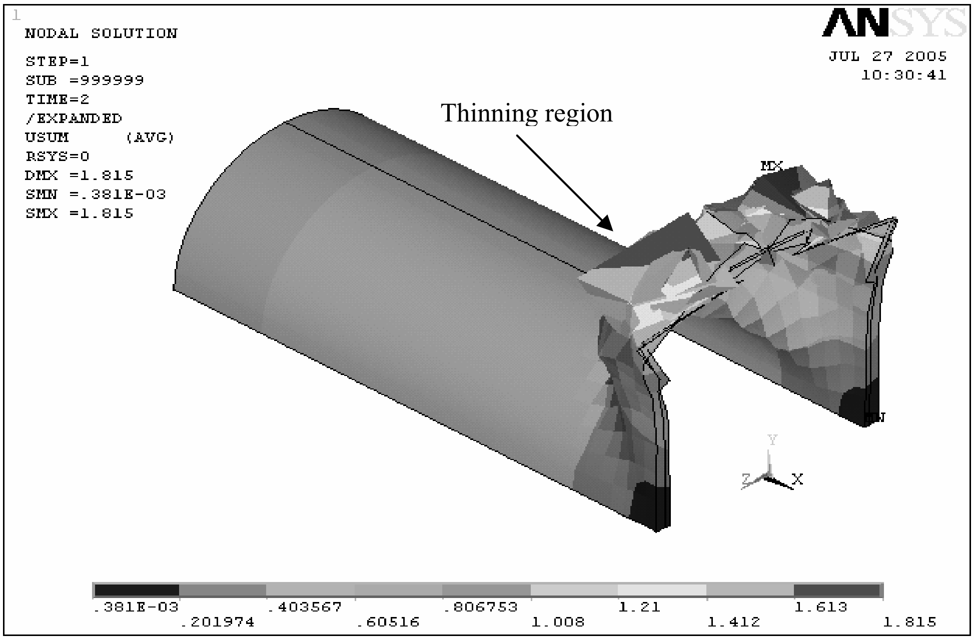
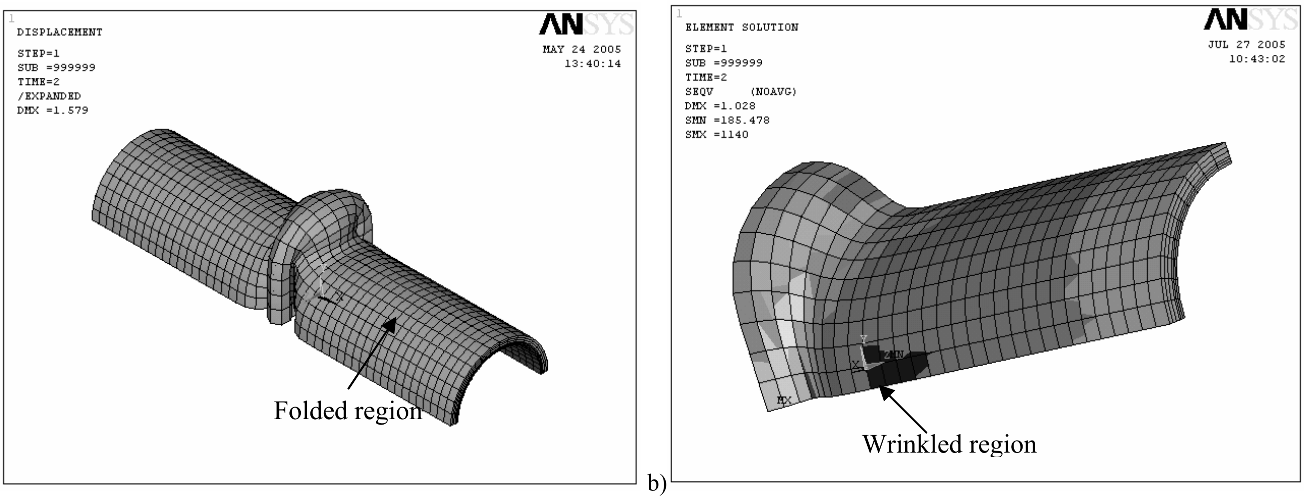


Fig. 7. bursting failure due to high internal pressure

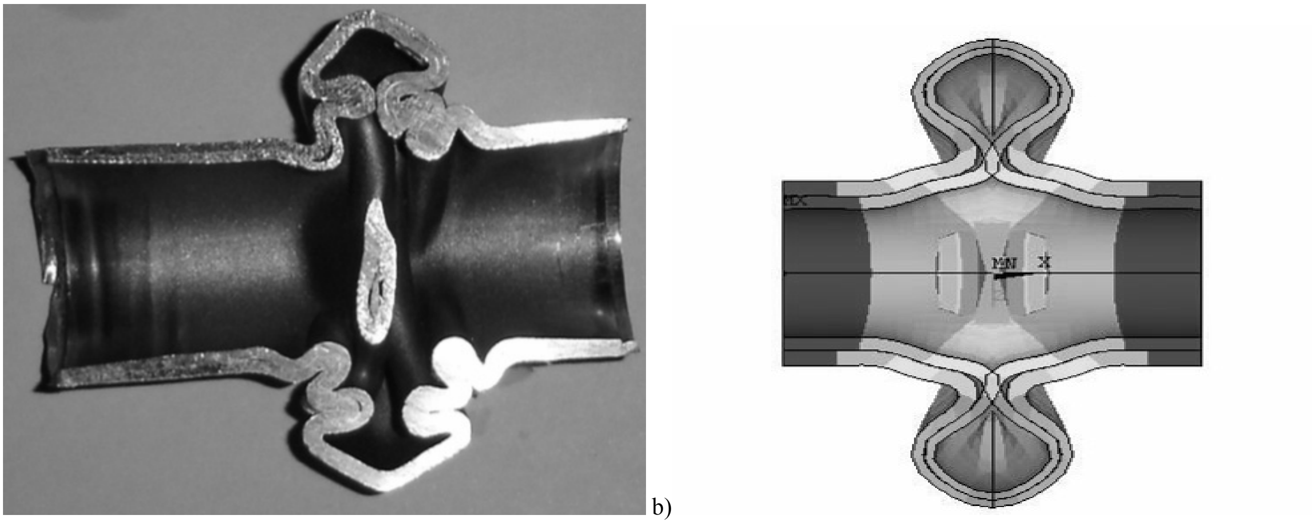


a) b)

Fig. 8. a) Failures due to high axial displacement compare to internal pressure – folded region, b) failures due to high axial displacement compare to internal pressure – wrinkled region

Figure 8 represents one of these ideal conditions. In this solution less internal pressure has been applied compared to the axial feed because the material could not flow along the die hole due to the low internal pressure. This would result in the tube being wrinkled or buckled in the certain region.

The comparison is drawn between the simulation results and the practical experimental test, which is carried out in the bulge forming machine. This test is performed for a high axial displacement (maximum displacement of the machine) and a lower internal pressure compared to axial feed. Figure 9 (a) shows the test result in the bulge forming machine. It is clear from the figure that the multi layer tube failed through wrinkling, folding and squeezing. To represent this failure by simulation, a further solution is undertaken by applying the same boundary conditions as were applied in the practical test. Figure 9 (b) depicts the simulation result. It demonstrates that the failure patterns are apparently the same but in real life they occur in different manner. In the practical case, when the tube fails due to over loading, wrinkling and folding, as soon as a leakage appears in the system, the internal pressure drops instantly. But in the case of simulation, the internal pressure keeps acting on the surfaces until the last moment of the solution even though the structure has already failed. Due to these



a) b)

Fig. 9. a) Failure in bulge forming machine for high axial feed, b) failure in FE simulation by ANSYS

difficulties the failure results differ from practical testing to simulation. Although the branch is severely buckled, the total displacement is only 17mm. The actual displacement of the failed specimen is approximately 15 mm. Therefore it can be concluded that both the practical and simulation results are in good agreement. It can be seen that a large amount of buckling occurs and both materials were under very high stress. The buckling occurred as there wasn't a high enough internal pressure to withstand the high axial displacement or axial force.

5. Conclusions

A number of simulations have been carried out for X branch forming. It can be concluded from the simulation results that the loading condition of 20 mm axial feed and 150 MPa internal pressure gives good X branch. In this case low stress is evident in the component and the maximum bulge height is achieved. Although the experimental result of the hydro forming process validates the numerical simulations reasonably well. Both simulation and experimental results are very close on the basis of deformation profile and the value of the branch height. Significantly the failure parameters in the hydroforming process, is focused in the analysis elaborately. Two different types of failures are presenting for bi-layer hydroforming process. One kind of failure happens due to application of a high internal pressure and the other type of failure may occur due to the application of a large axial displacement compared to the internal pressure.

References

[1] M.D. Islam, A.G. Olabi, M.S.J. Hashmi, Feasibility of multi-layered tubular components forming by hydroforming and finite element simulation, Journal of Materials Processing Technology 174 (2006) 394-398.

[2] M.D. Islam, A.G. Olabi, M.S.J. Hashmi, Mechanical stresses in the multi layered T-branch hydroforming: Numerical Simulation, Proceedings of the 23rd International Manufacturing Conference 2006, University of Ulster, Belfast, UK.

[3] L. Lang, J. Danckert, K.B. Nielsen, Multi-layer sheet hydroforming: Experimental and numerical investigation into the very thin layer in the middle, Journal of Materials Processing Technology 170 (2005) 524-535.

[4] P. Ray, B.J. Mac Donald, Experimental study and finite element analysis of simple X-and T-branch tube hydroforming processes, International Journal of Mechanical Sciences 47 (2005) 1498-1518.

[5] B.J. Mac Donald and Professor M.S.J. Hashmi, Three-dimensional finite element simulation of bulge forming using a solid bulging medium, Finite Elements in Analysis and Design 37 (2001) 107-116.

[6] B.J. Mac Donald, M.S.J. Hashmi, Analysis of die behavior during bulge forming operations using the finite element method, finite elements in Analysis and Design 2 (2002) 137-151.

[7] M. Ahmed, M.S.J. Hashmi, Three-dimensional finite-element simulation of bulge forming, Journal of Materials Processing Technology 119/1-3 (2001) 387-392.

[8] Yeong-Maw Hwang, Wen-Chih Chen, Analysis of tube hydroforming in a square cross-sectional die, International Journal of Plasticity 21 (2005) 1815-1833.

[9] T. Altan, M. Koc, Application of two dimensional (2D) FEA for the tube hydroforming process, International Journal of Machine Tools & Manufacture 42 (2002) 1285-1295.

[10] ANSYS Inc., Theory Reference 9.1.