

FINITE ELEMENT ANALYSIS OF A FORMING PROCESS, USING STATIC STRUCTURAL (ANSYS) AND ANSYS LS-DYNA

RADU Ioan Eugen¹, GREBENIŞAN Gavril¹

¹University of Oradea

iradu@uoradea.ro, grebe@uoradea.ro

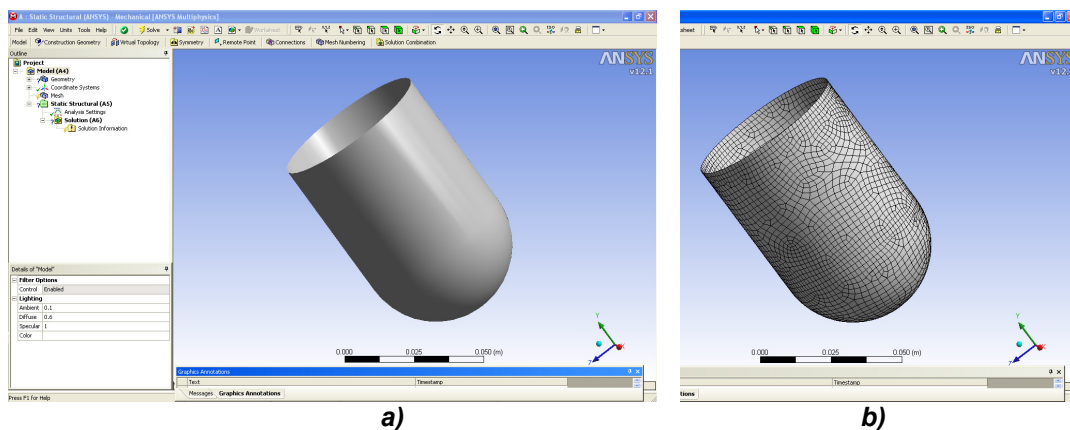
Keywords: Finite Element Analysis, mesh, numerical methods, ANSYS

Abstract: The problem consists on analyse of behaviour of deformed sheet material which got a spatial deformed state, has studied and reported in this work. The analysis method, used here, were Finite Element Analysis, on ANSYS software. We considered that the workpiece was pre-deformed, using a conventional method, by deep forming process (coupled deep drawing- a direct deep drawing combined with an inverse deep spherical forming process). At this stage, the aims were to find most deformed zones, and most exposed (the safety factor magnitude) ones. This work has solved this problems using ANSYS software.

1. INTRODUCTION

A Finite Element Analysis (FEA), a computer aided engineering (CAE) process, request higher computational efforts and abilities, in addition with good proficiency on forming process theory. Nevertheless, a forming process simulation or analysed, using specialized software, the post-processing work should be done on good conditions, such as using theory basis and user expertize.

Starting with a pre-deformed workpiece, an hollow cylinder with a spherical end, the workpiece was deformed by a deep spherical drawing process, on opposite direction, of spherical end. Analysis process starts with geometry model creation on ANSYS Workbench Component System, saved and imported (or connected as standalone system on Project Workwindow), into Static Structural (ANSYS)- Mechanical (ANSYS Multiphysics):



**Figure 1 – Starting workpiece, pre-deformed, a),
and Mesh Geometry, b)**

This model have to be updated, namely mesh generate will done, in order to be used as geometric model on Static Structural system, and imported into ANSYS LS-DYNA for analysis to be performed after. The geometry and mesh model used as start base for analysis were defined using SI units, and the material will be defined into ANSYS LS-DYNA workspace. The refined mesh size no need at this stage, untill the results no request this. The level of mesh accuracy it's devoted by the quality of work parameters, such as wall thinning, spatial deformation state, the convergence of numerical and simulation method, statistical distribution of residuals and calculation errors density.

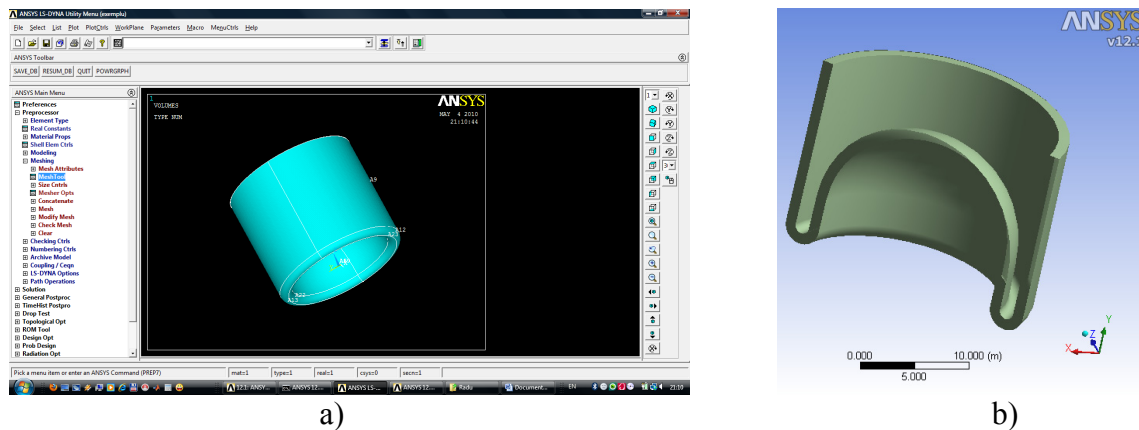


Figure 2 – Geometry model (imported to ANSYS LS-DYNA), a), and cross section of the workpiece, b)

2. ANALYSIS SETTINGS

2.1 General synopsis

Synoptically analysis settings comprises following components which have to be declared as work parameters:

- Material Data
 - Structural Steel
- Units
- Model (B4)
 - Geometry
 - Solid
 - Coordinate Systems
 - Connections
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Loads
 - Solution (B6)
 - Solution Information
 - Results

All data comprises on this synoptic content are reviewed and exposed into the Mechanical Report generated at the final of analysis, if the user wants to. This report comprises also, charts and graphics automatic generated by the software, as additional option.

2.2 Apply loads

As part of FE analysis, applying loads and constraints (although it is named boundary conditions, technically inadequate expression), consists on define where should be applied and which is the magnitude of forces acting on workpiece, also which parts from geometrical model moves (i.e. defining Degree of Freedom (DOF)). Loads apply process, comprise: forces, pressures, temperatures, displacements, velocities, constraints (DOF, moving blocked or free displacements), contact zones, fixed supports:

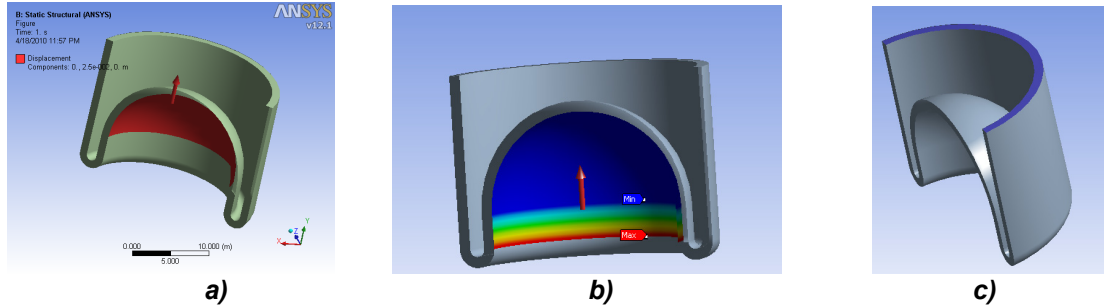


Figure 3 – Apply loads
a)-displacement; b)-hydrostatic pressure; c)-fixed support

Working on ANSYS LS-DYNA software one may be visualize vectors representing forces or displacements as applied loads, on every mesh nodes if desired:

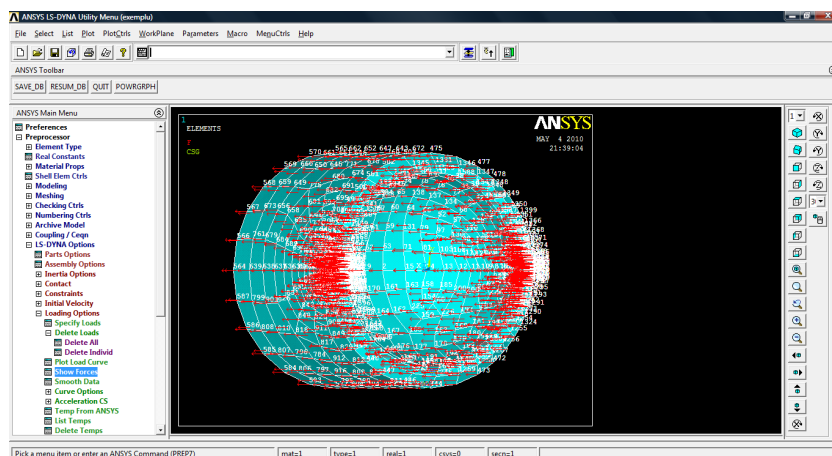


Figure 4 – Show forces on numbered nodes

3. RESULTS

Once settings are done, the problem may be solved explicit, using time control and output controls defined by the user. Solving problem consists on finding a solution, a complete post-processing part of analysis may be undertaken.

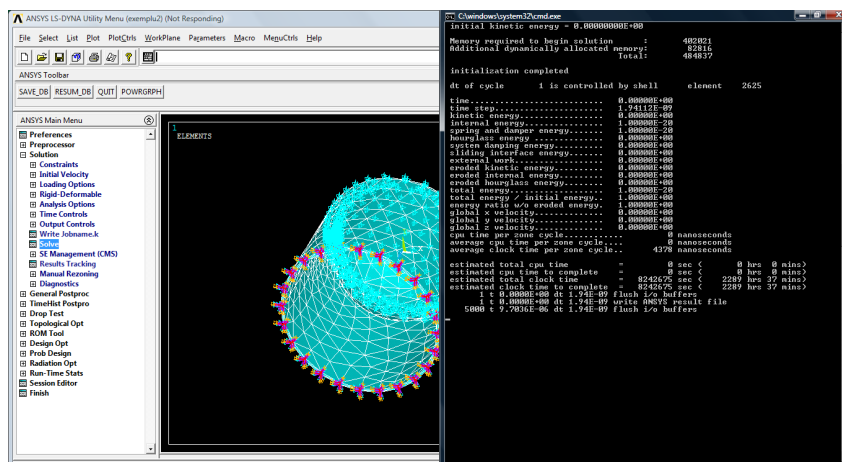


Figure 5 – Solve problem starting

Analysis results implies a good understanding of problem requires. In this case, our aims were to simulate and visualize more exposed zones, spatial strain state, and von Mises equivalent stress disposing, also the remanent stress disposal and magnitude of these stresses. Total deformation, a tool available on Static Structural (ANSYS) for visualize the

results, shows how exposed are surfaces or volumes, and presents the magnitude of displacements, as result of force acting on top of sphere:

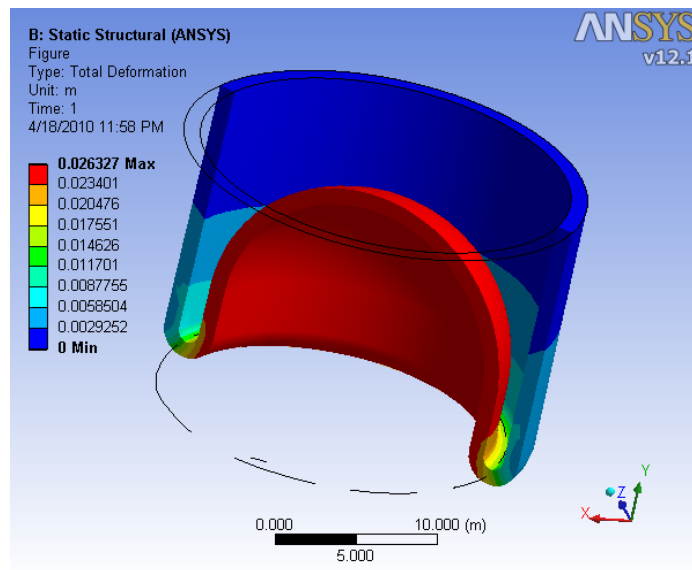


Figure 6 - Total deformation

As may be seen on previous figure (fig. 6), most deformed area it's located on dome, as we expected. Studying the spatial stress state on dome and on the fillet zone, we found that this is situated at the base of dome and cylinder:

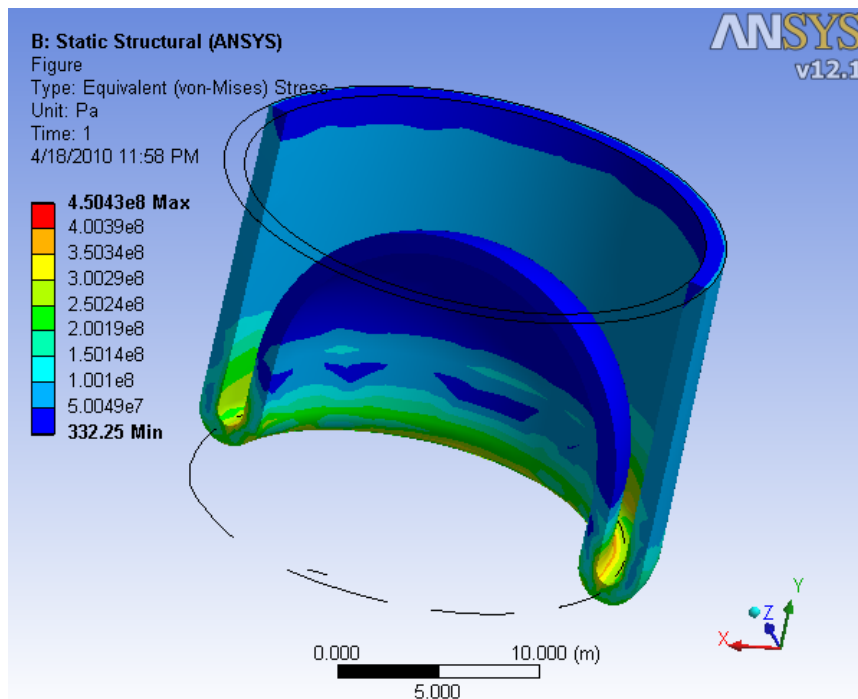


Figure 7 – Von Mises Equivalent Stress

Another attempt, which may be defining, for this study, consists on loading by a focussed force, on top of dome. Results shows that, at this dimensions of dome and fillet radius one could be expect that, the stress state also the strain state, the stress state will generate a

rupture on neighbour zone of fillet zone, as shown in next two figures. It is easy to see from these simulation analysis, at dome height which go beyond over certain values, bulking of workpiece results, when the force, or pressure, are applied on top of the dome:

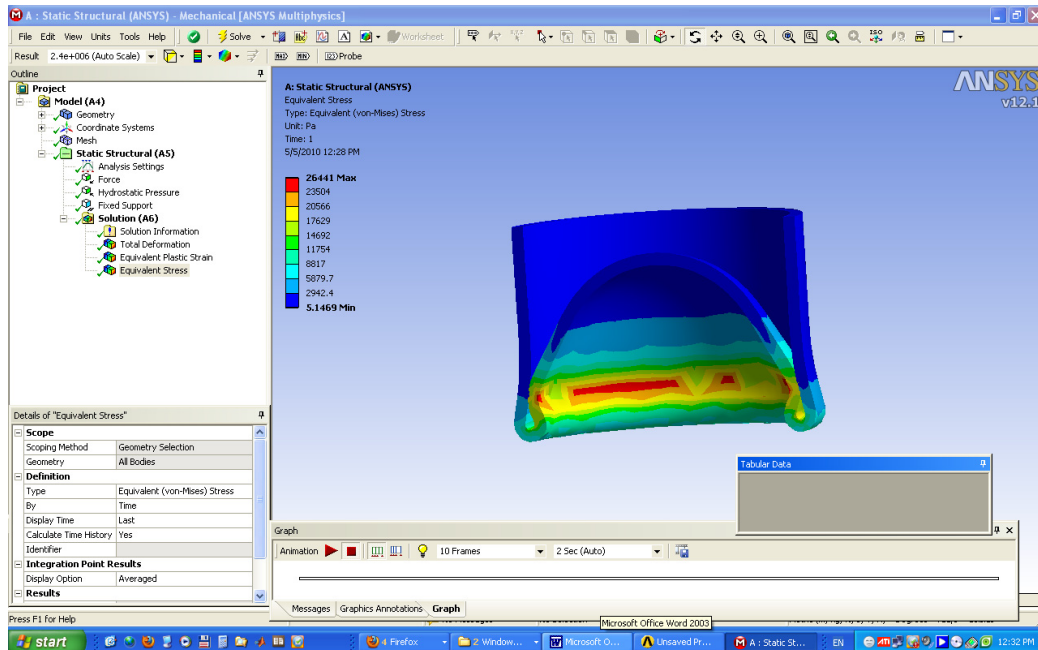


Figure 8 – Equivalent (von Mises) Stress
(the force acting on top)

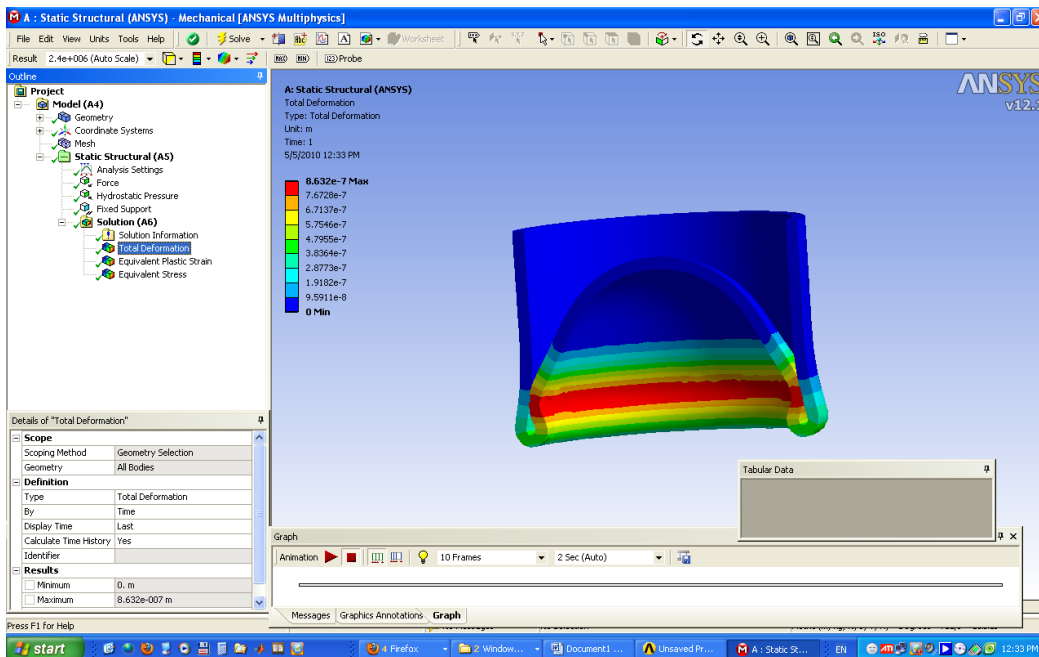
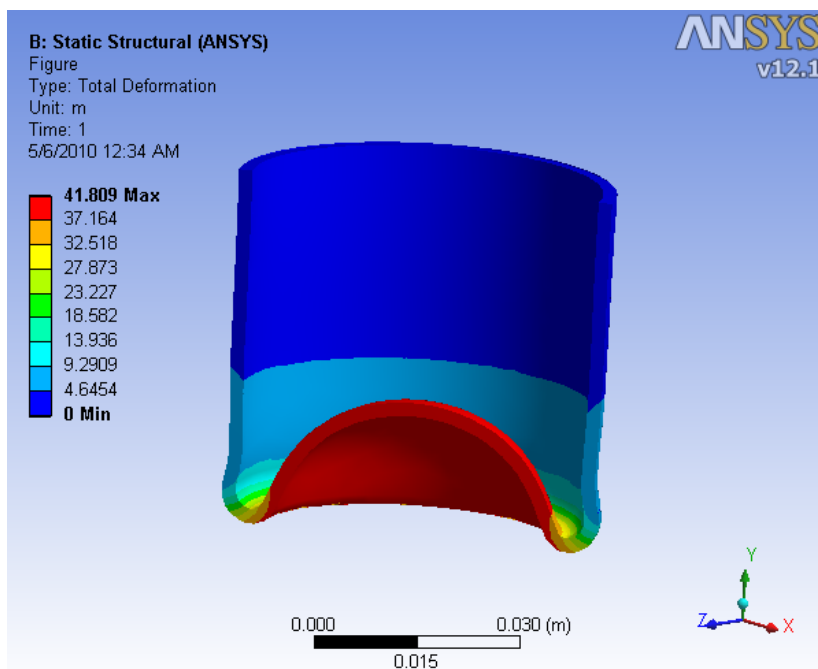


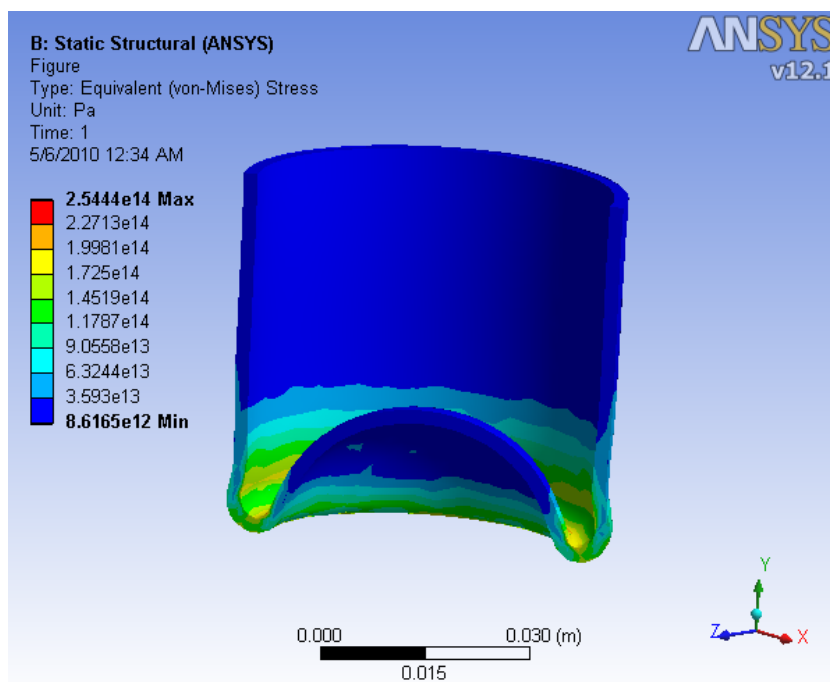
Figure 9 – Total Deformation (bukling)
(the force acting on top)

Second attempt, which allows to find technical solution in order to avoid bukling, gives results when an hydrostatic pressure were applied on top of the dome, instead of a concentrated force applied on the same conditions. From theory it's known that the hardening of material, on deformed zone is present, and this depends on structure of material, shapes of deformed workpiece (namely minor fillet zones), velocity of forming process. So, in this idea, we'll have to expect that at this "rounded corner", also, on

hydrostatic pressure, a more complex stress and strain state occurs on these especially areas. From next figures, we found that a minor buckling occurs if the hydrostatic pressure is applied, instead of focused force, even the cylinder part of workpiece it's free (no fixed support). This phenomenon arise from distributed forces on entire surface of the dome, coupled with space stress state located, as one may see, on fillet zone:



**Figure 10 – Total Deformation (buckling)
 (the hydrostatic pressure acting on top)**



**Figure 11 – Equivalent (von Mises) Stress
 (the hydrostatic pressure acting on top)**

The space strain state it's captured on next figure where one can see the equivalent plastic strain (darker grey central zone of the dome). Addiacent zone (light grey, fillet zone) exhibits other strain state, more precisely, an elastic, or elastic plastic zone, which is

prohibitory for a solely plastic behaviour of material. This is confirmed, also, by the buckling of cylinder zone, into immediately neighbourhood of the elastic-plastic zone:

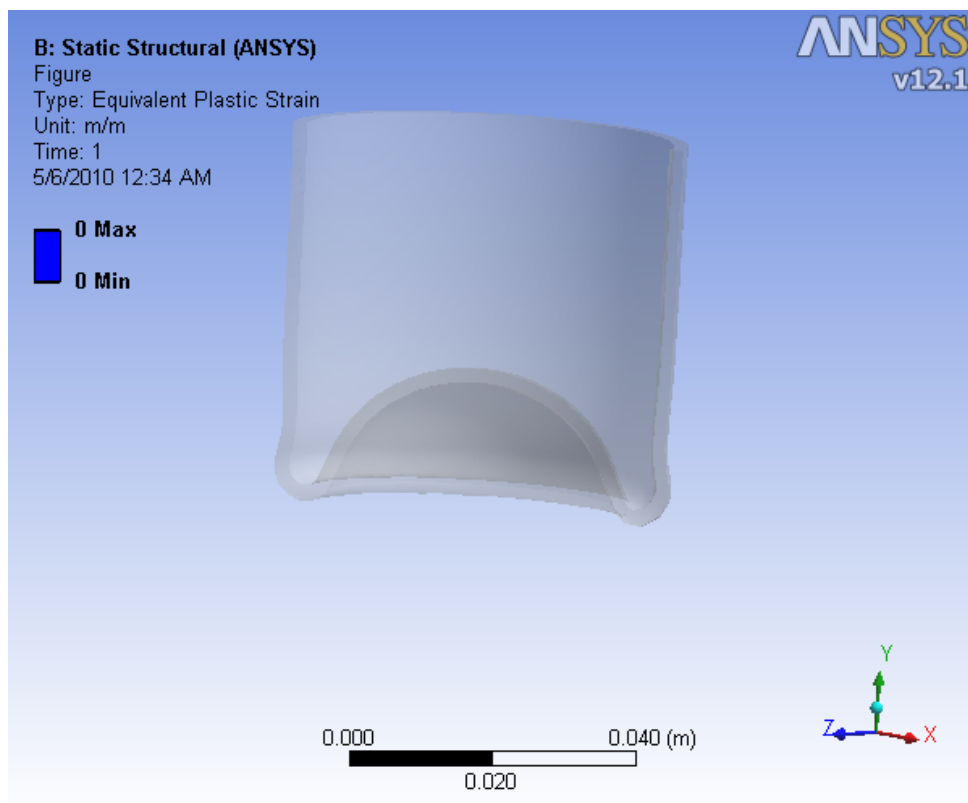


Figure 12 – Equivalent plastic strain

4. CONCLUSIONS

Then and there's more important to underline following aspects, revealed on this work. First of all, using a specialised software, one can be save time and other costs on research work. These simulation and analyses, presented here, suggests that, a more expensive way to find materials behaviour's, consists on attempt to design forming processes, which includes costs referring on die and experimental works. These costs may be avoided using specialised softwares. It should be recognize that, a verification work, have to be unfolded, in order to analyse, on practice, other concealed acuses and effects. On third place, as revealed conclusion, we conclude that referring on space state of stress and strain, the location of affected zones were established, as well. These zones, have to count on, when the forming process will be designed, such as die design, fixed support (which is inhibitory for bukling expansion), the magnitude of fillet radius and type of forces or pressure which will be used.

References

- [1] KONRAD, A.-"Integrodifferential Finite Element Formulation of Two-Dimensional Steady-State Skin Effect Problems", IEEE Trans. Magnetics, Vol. MAG-18 No. 1, January 1982, pg. 284-292.
- [2] AHMED, H. & PEARCE, R.- "Superplasticity in Aerospace – Alumium", Ed. R. Pearce&L. Kelly, publ. Cranfield Institute of Technology, Bedford, 1986, pag. 146-159.
- [3] ARGYRIS, J., DOLTSINIS, J. S. - An Aperçu of Superplastic Forming, in "Plasticity today . Modelling, Methods and Applications", ed. A.Sawczuk, Elsevier Applied Science Publishers, London, 1993.
- [4] ARGYRIS, J., DOLTSINIS, J. S. - A Primer on Superplasticity in Natural Formulation, "Computer Methods in Applied Mechanics and engineering ", vol.46, pag.83-131, 1992
- [5] BEER, G.,ş.a.- Introduction to finite and boundary element methods for engineering, John Wiley & Sons, 1975.

- [6] BEJU, I.,ș.a. - Tehnici de calcul tensorial euclidian cu aplicații, Editura Tehnică, București, 1977.
- [7] BONET, J. - Numerical Simulation of the Superplastic Forming of Thin Sheet Components Using the Finite Element method, "International Journal for Numerical Methods in Engineering", vol. 30, pag. 1719-1737, 1990.
- [8] CHANDRA, N. - Membrane element analysis of axisymmetric and non - axisymmetric superplastic metal forming processes, "Superplasticity and Superplastic Forming" ,ed. C.H. Hamilton & N. E. Paton, Warendale, 1989 .
- [9] GÂRBEA, D - Analiză cu elemente finite, Editura Tehnică, București, 1990.
- [10] GREBENIȘAN, G. - Finite element analysis of superplastic forming process, using ANSYS, Annals of University of Oradea, Fascicle of Management and Technological Engineering, 2010.
- [11] JOVANE, F. – An approximate analysis of the superplastic forming of thin circular diaphragm: theory and experiment, International Journal of mechanical Science, vol. 6, 1968, pag. 403-428.
- [12] KOBAYASHI, S. - Metal Forming and the Finite Element Method, Oxford University Press, New York, 1989.
- [13] MOAVENI, S. – Finite Element Analysis, Theory and Application with ANSYS, Prentice Hall, New Jersey, 1999.
- [14] www.ansys.com
- [15] Theory Reference for the Mechanical APDL and Mechanical Applications (ANSYS)