

FINITE ELEMENT ANALYSIS OF SUPERPLASTIC FORMING PROCESS, USING ANSYS

GREBENIŞAN Gavril¹, ROMOCEA Sanda²

¹University of Oradea, ²-Drumuri Judeţene Bihor
grebe@uoradea.ro

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

Abstract: An analysis of superplastic forming using ANSYS software is presented here. This work consists on a generalized procedure used commonly in order to establish performances and sizes of strains and stresses that are developed during the process occurs. No any procedure were developed for a gasostatic forming process. In this kind of process, to the workpiece active surface a pressure will be acting on one of free surface alternatively, most of commonly superplastic forming processes. This work presents only one operation, meaningful one direction and one sense acting force (the pressure).

1. INTRODUCTION

A Finite Element Analysis (FEA), consists on an entire and large time consumer process. Also, this computer aided engineering (CAE) process request more than medium level of engineering knowledges. In order to setup a complete analysis, using FE numerical methods, there needed Computer Aided Design (CAD) for geometric model creation, after the problem to be solved were established. If the geometric model was saved on the computational system meshing process may be setup, based on parts, surfaces and solids components. This subsystem, mesh process named here, needs sometime a large knowledge efforts also a consistent computational cost effective. Suppose these processes well done solved, the FE analysis cannot be started before the boundary conditions (BC), constraints, loads and FE formulation will be setting up. No FEA can be started without an element type were stated. The FE formulation, [1], consists, commonly on governing equations which argued the finite element deformation state, finally. Not only Euler, Lagrange, Arbitrary Lagrangian Eulerian formulation are available on ANSYS software. Depends on process analysis, such forming, metal forming, roller heming, many of element formulation are available: Flanagan-Belytschko, co-rotational Technique, Hughes-Liu beam element formulation, Belytschko-Lin-Tsai shell element, triangular shell element, Marchertas-Belytschko shell element, Hughes-Liu shell element, membrane element formulation and more others.

2. SOLVING PROBLEM- USUAL STEPS

2.1 Geometry Import

Technical problem which should be solved is the setting analysis parameters and steps to be followed in order to study the behaviour of a forming die and workpiece shell under gas pressure.

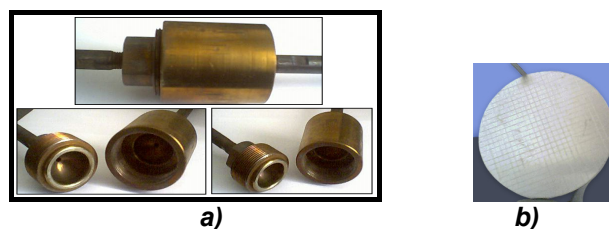


Figure 1 – The forming die a), and workpiece, b)

The analyse starts with the creation of the geometry model. This geometry may be created on any of CAD software, accepted format files of ANSYS, such as CATIA, PROENG, ACAD, etc., and on ANSYS Workbench, obviously. This paper suppose that one create the geometry on ANSYS Workbench:

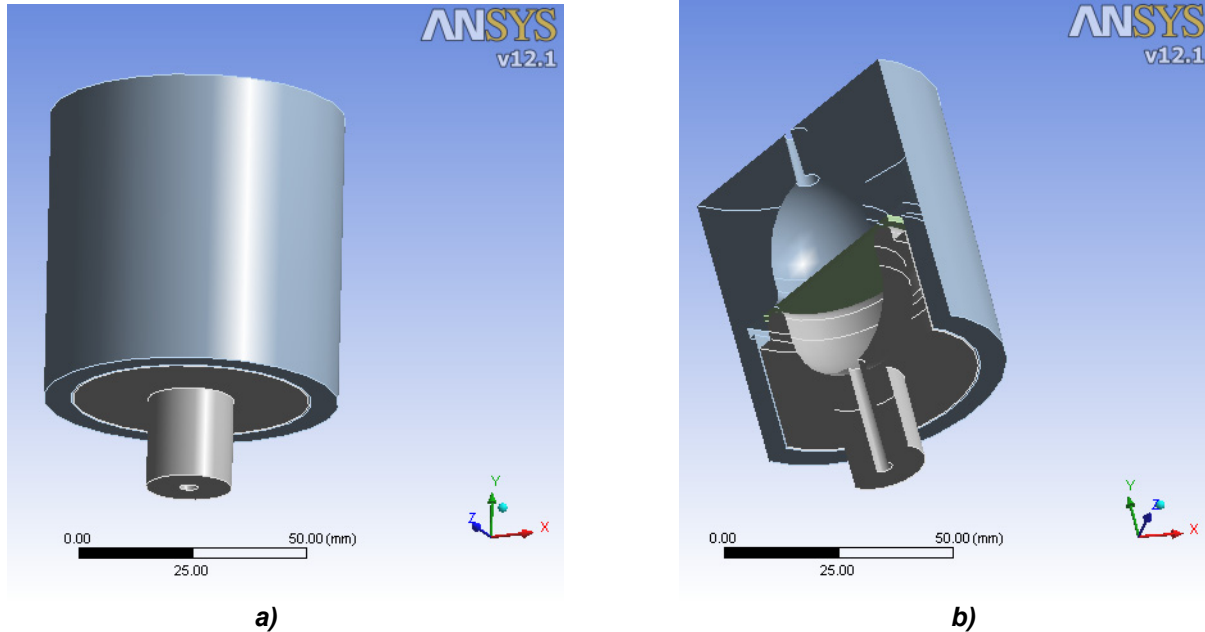


Figure 2 – The forming die assembly, a), and section, b)

The geometry model will be used as base of the applied forces (namely pressure in this case) and further considerations on analysis process. The geometry model will be processed by meshing, load constraints and boundary conditions:

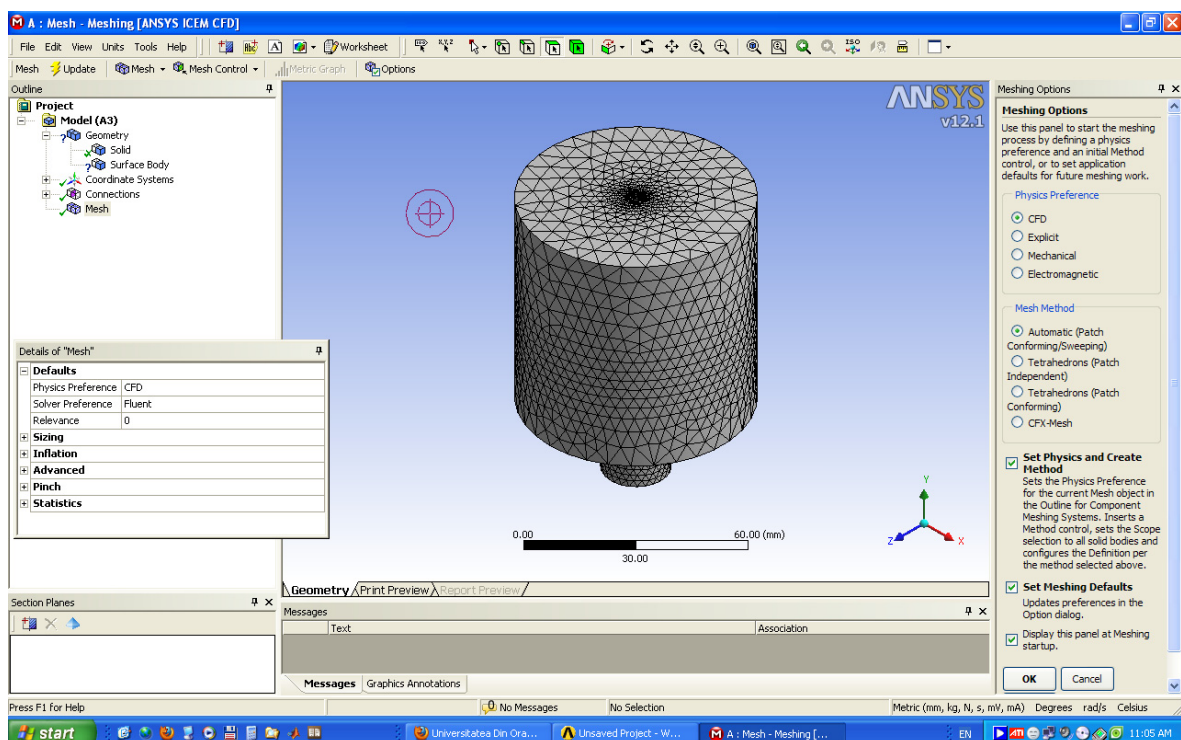


Figure 3 – Meshing of the die

The meshing process may be carried on using a Meshing Options Wizard, by defining the physics preferences, mesh method, default method and so on.

Meshing process may be carried on using such as user mesh details, on tab named Details of “Mesh”, where the Defaults settings may be confirmed and Sizing, Inflation, Advanced parameters, Pinch and Statistics values of parameters may be declared before the process starts.

After geometry were meshed, the system of the analysis may be created as standalone system. In this case one chosed the Fluid Flow (FLUENT) system:

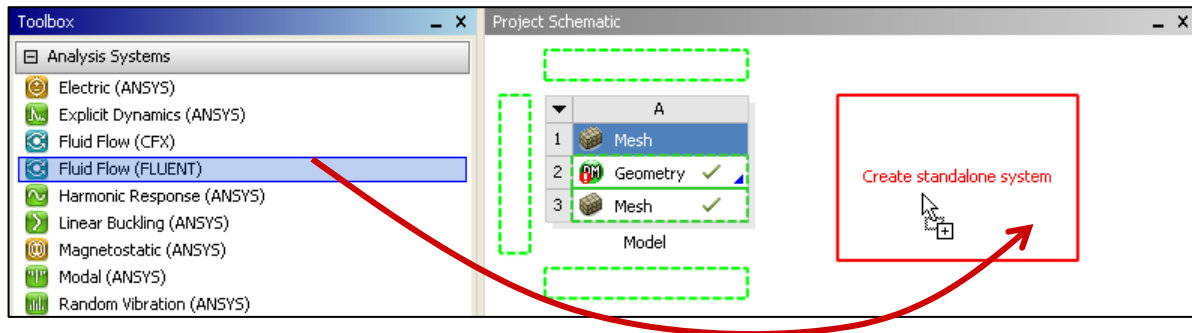


Figure 4 – Creating standalone system

New standalone system created will be connected on Mesh Component System such as generator system which comprises of Geometry model and Mesh model. Effective connection will be realised by drag and overlap on Setup area of FLUENT box:

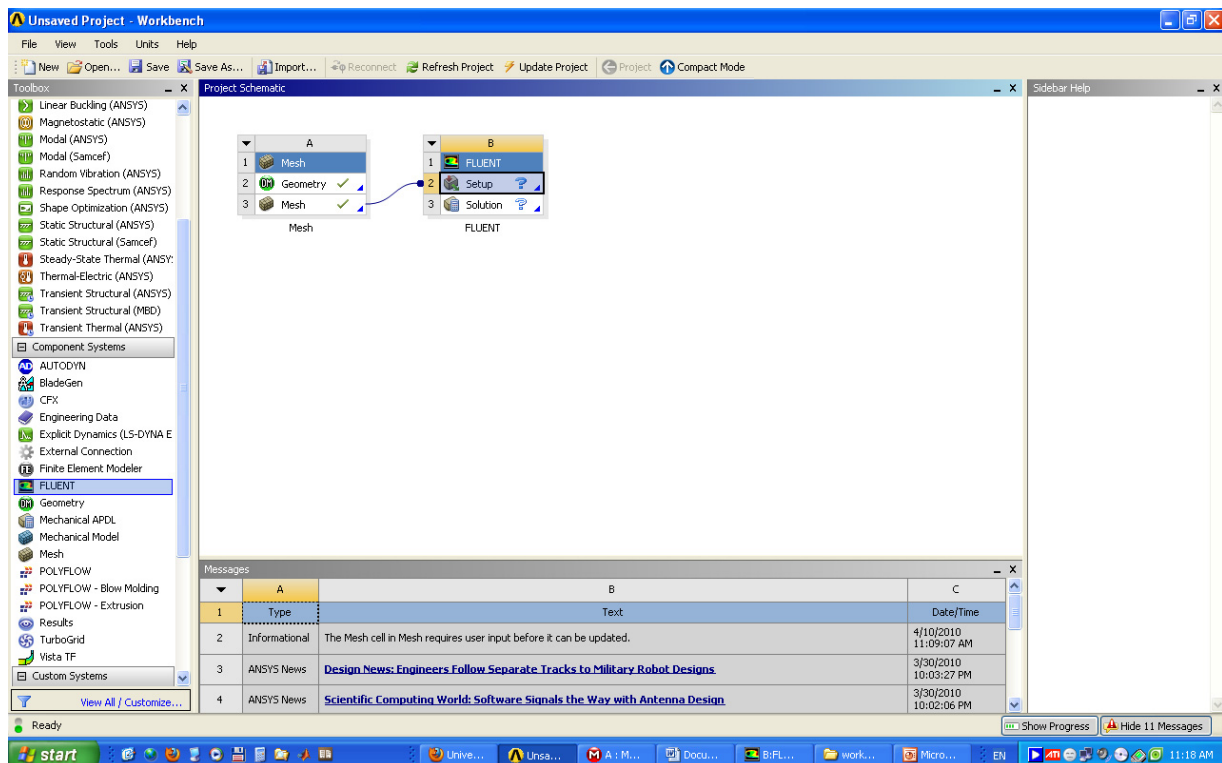


Figure 5 – Connection creating between two components system

This system, new created, should be adapted on meshing parameters, Units System used by the user and the geometry system components. In fact, the FLUENT system should be

“learn” how the geometry and mesh models can be manipulated, in order to be analysed, respects the analyse settings. So, the standalone system, new created here, must imports all general data regarding on the geometry model. This operation, named Patch Conforming Method, will provide all this needed data, such as material, geometry components (lines, surfaces, solids, parts):

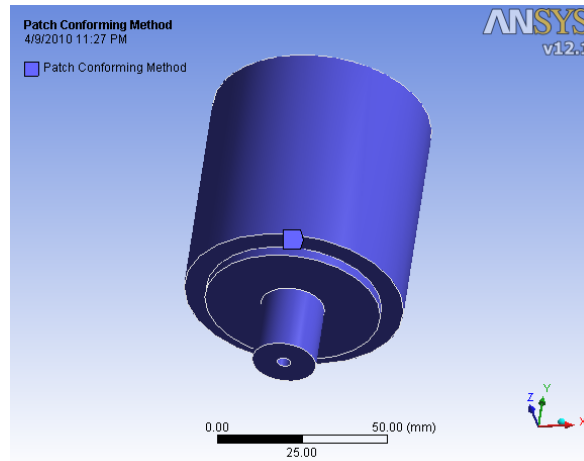


Figure 6 -Patch Conforming Method

Consider the geometry creation operation carried out the analyse parameters may be setting up, as next step of the work.

Opening the FLUENT system, the analysis may be started. Note that the user should be aware of the complexity of the process. No any analysis may be started, before the problem will be detailed and structured on every posible direction. Thus, the superplasticity, as process, involve the air flow process, which lies the gasostatic forming process on the flow dynamics and computational flow dynamics (CFD), as numerical method analysis. In addition, such a complex process involve more than one material “deformed” here: air, superplastic material of the workpiece and the thirth one the die material. All of those materials acting as system assembly, because the process occurs on the medium to high temperature environment (about 560 °C). So, next step in this analysis comprises on reading and checking of the mesh geometry in order to find the conformity to the request and requirements of the FLUENT system:

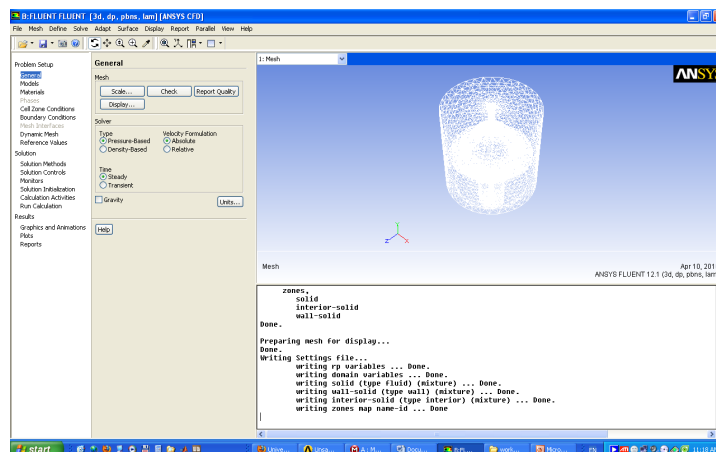


Figure 7 –Reading the geometry mesh general settings

Follow the procedure, scaling of the mesh consists as next step in order to start the FEA settings. In this operation, one may verify the fluence of the surfaces, as mesh occurrence. This operation starts and occurs on few sub-steps which request to the user some additional information, such as the units of mesh created, in order to be converted by the FLUENT system:

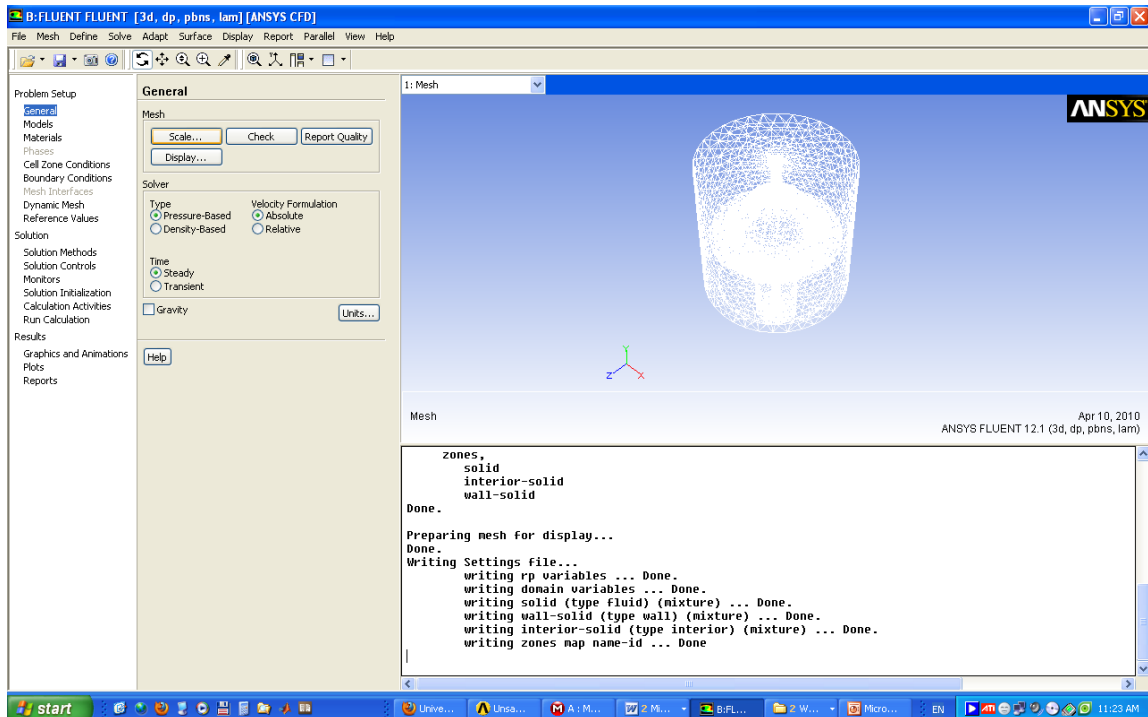


Figure 8 – Reading the geometry and mesh settings

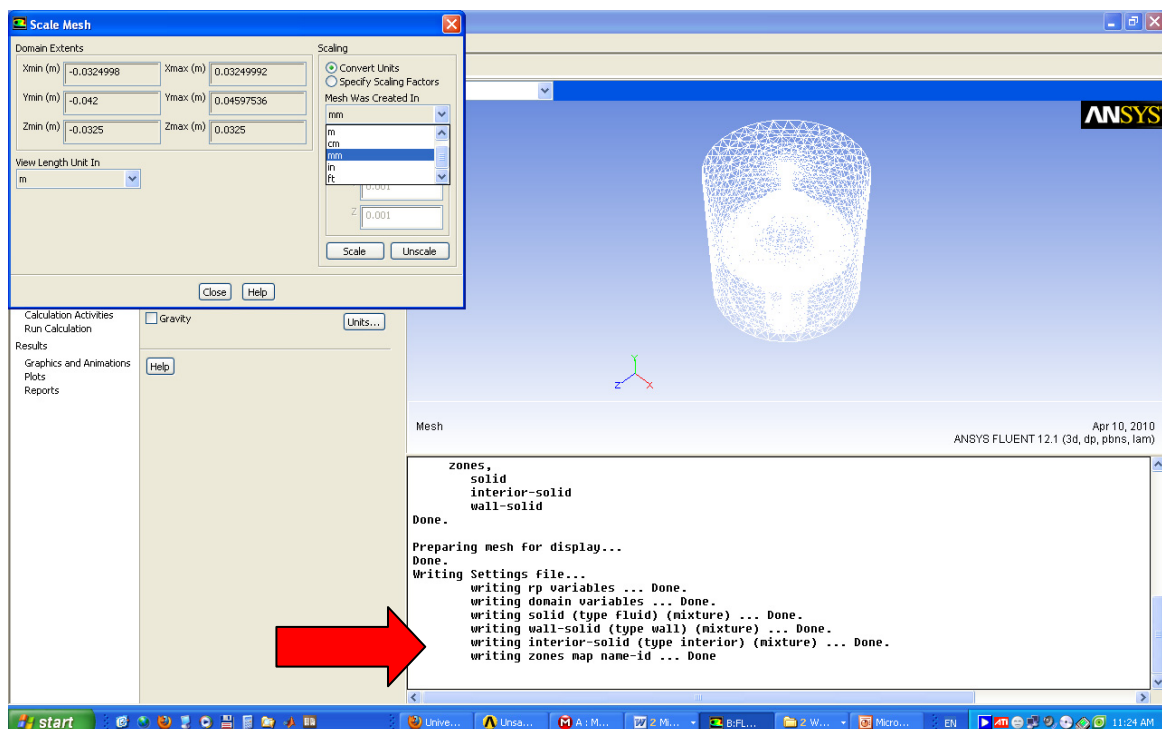


Figure 9 – Scaling mesh geometry

Scaling procedure going to be completed after checking this on different step. This operation include the Report Quality which stands on analysis of conformed mesh geometry settings and parameters. At this level, after checking process ends one may verify the parameters of mesh quality: "Maximum cell squish", "Maximum cell squeness", "Maximum aspect ratio", all of this parameters are established applying quality criteria for tetrahedra/mixed cells:

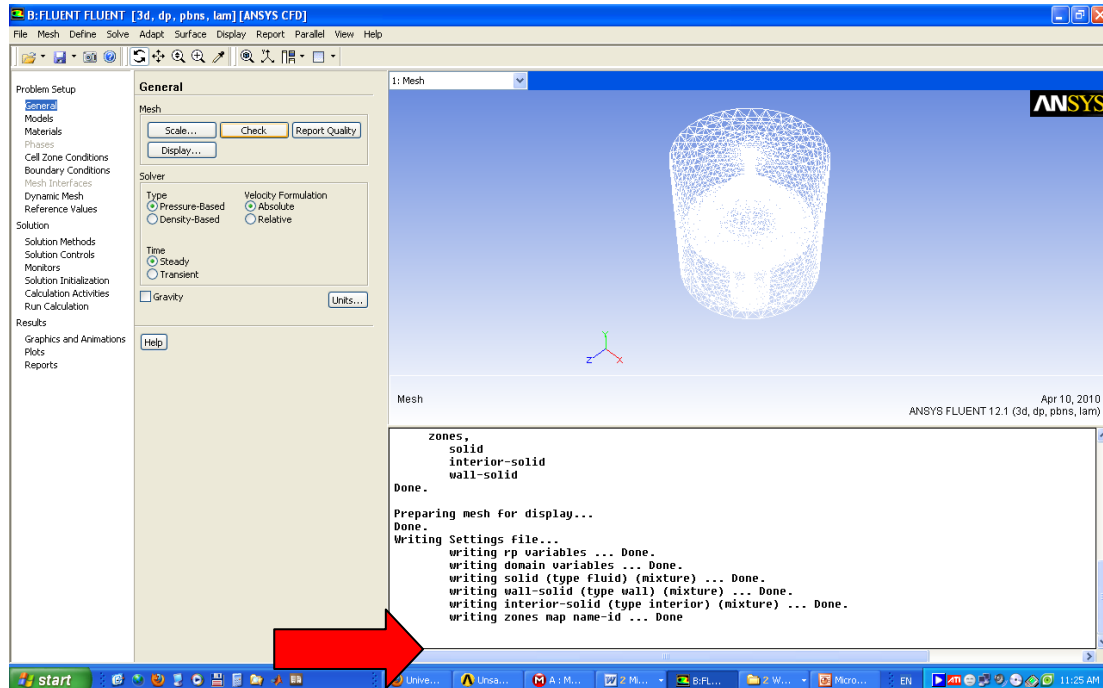


Figure 10 – Start of checking mesh

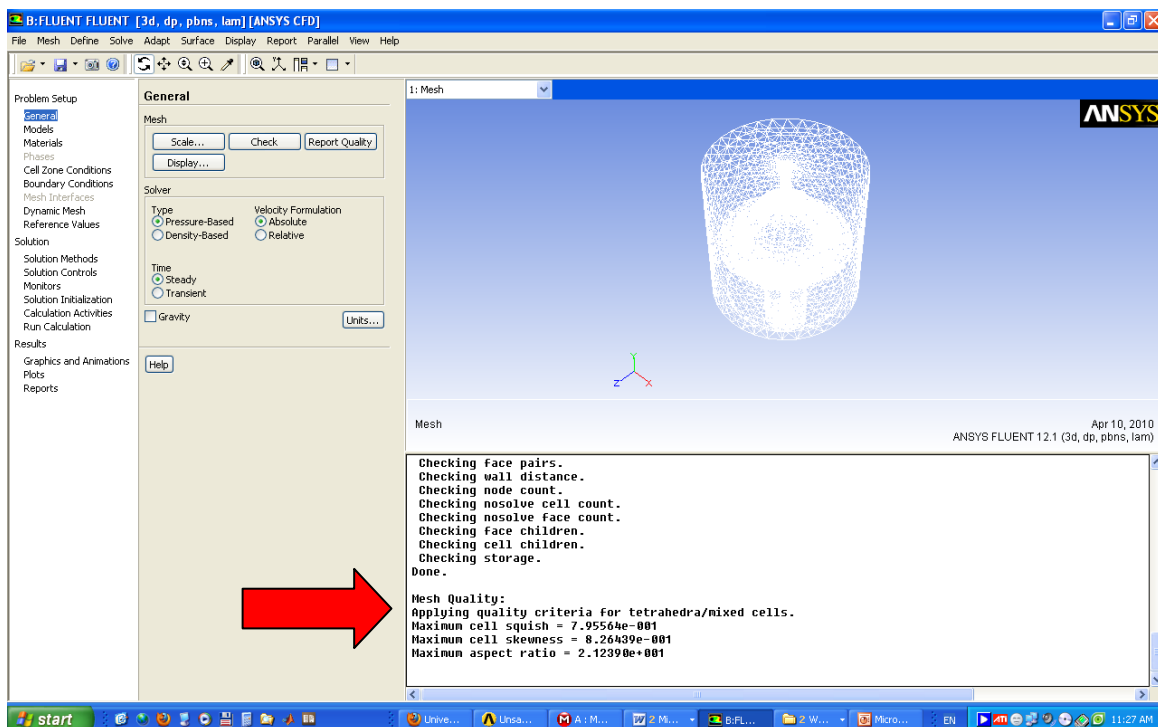


Figure 11 – Results of mesh cecking

Checking the mesh geometry one may finish the geometry status by decide which is the displaying settings:

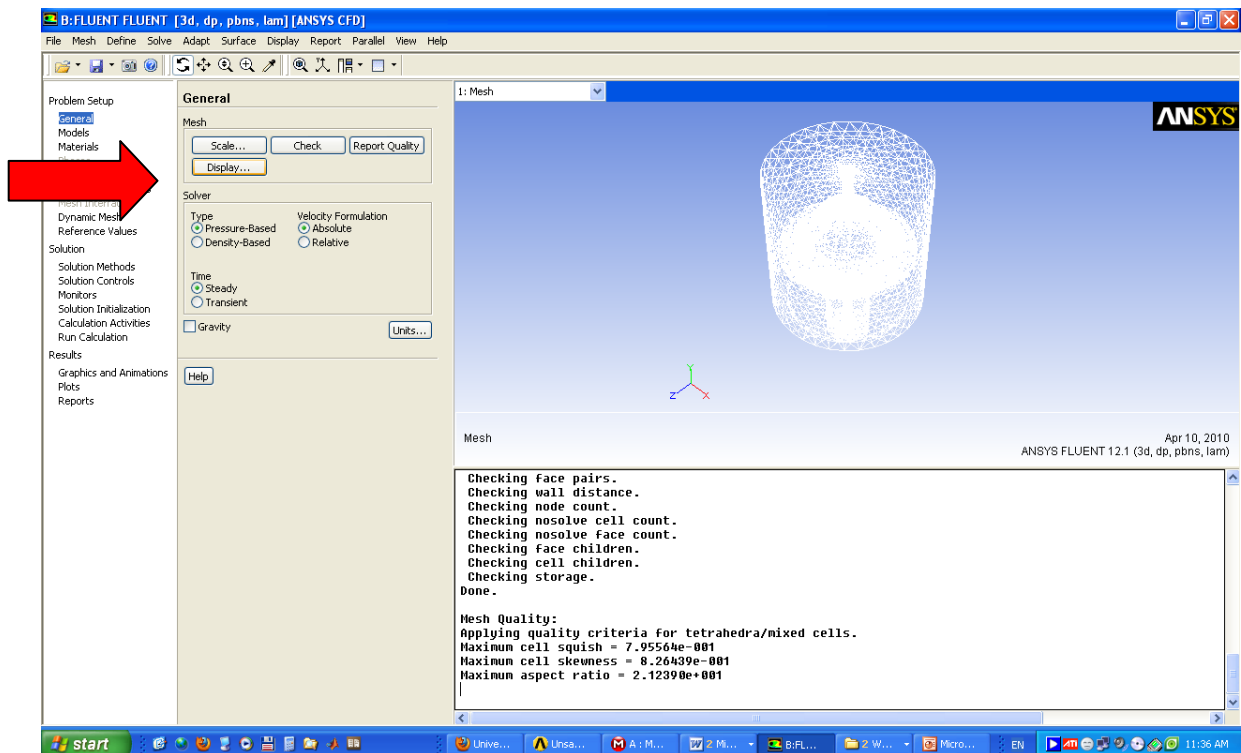


Figure 12 – Displaying the mesh parameters settings

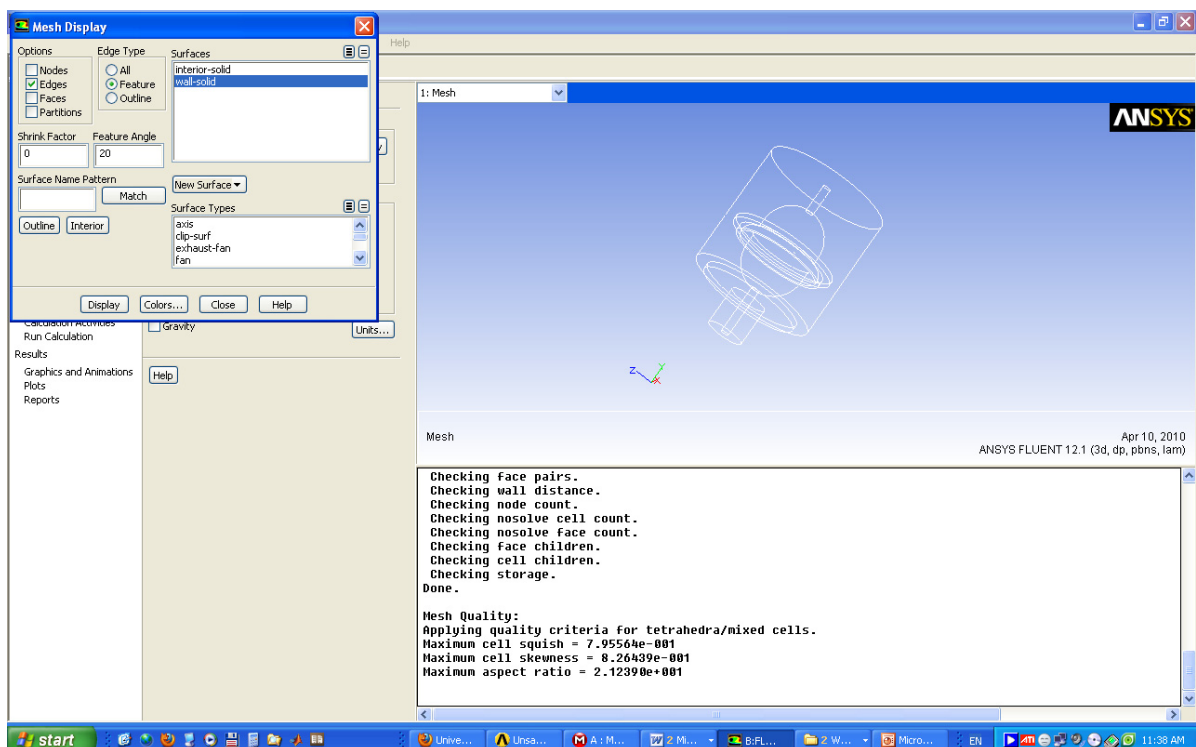


Figure 13 – Displaying parameters settings

Last step before starts the modeling analysis procedure comprise on adjusting units of the geometry, physics and environment:

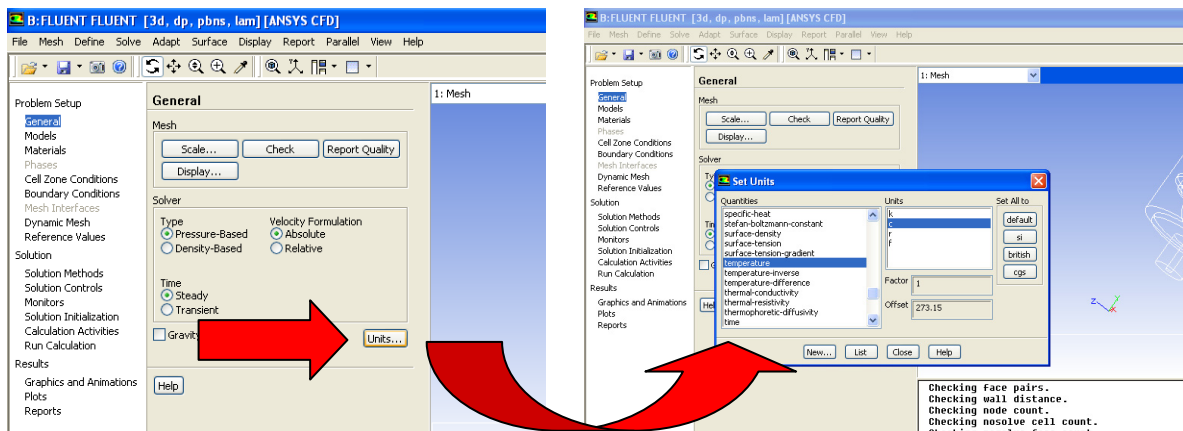


Figure 14 – Adjusting units

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 – Aluminium", 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., ş.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. -Analiza prin metoda elementului finit la deformarea superplastică, Referat II-doctorat, Cluj- Napoca, 1997.
- [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)