ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering
Tome IV (Year 2011). ISSN 2067-3809

[1]Nazimuddin QURESHI, [2]Muhammad Arsalan FAROOQ,
[3]MD. Mushtafizur REHMAN, S.M. Sohaib TARIQ

ADVANCED SIMULATION OF GAS METER COMPONENTS

ABSTRACT:
Gas meter has been modeled through many different simulation techniques in the recent times, suggesting new solution and giving new ways to minimize the material cost. In this project we will present an advanced simulation of the gas meter components, taking into account the maximum tensile strength and structural integrity. Our design presents a basic gas meter model to predict the behavior of structures while that it can withstand external & internal forces. Those points will be identified which are exceeding the limits of maximum tensile strength of the material. The result will show clear picture of the Von-Mises stresses, Strain, displacement and deformation. There are some inherent problems in simulation like time and memory consumption during analysis is huge. In this paper we shall analyze the gas meter and try to find a suitable solution for the inherent problems.

KEYWORDS:
Simulation, Boundary Conditions, Von Mises Stresses, Structural Integrity, Strain, Deformation

INTRODUCTION

A gas meter is used to measure the volume of fuel gases such as natural gas and propane. Gas meters are used at residential, commercial, and industrial buildings that consume fuel gas supplied by a gas utility. Gases are more difficult to measure than liquids, as measured volumes are highly affected by temperature and pressure. Gas meters measure a defined volume, regardless of the pressurized quantity or quality of the gas flowing through the meter. There are different types of gas meter used, Diaphragm meters, Rotary meters, Turbine meters, orifice meters and ultrasonic Flow meters. We will discuss and perform our simulation on the Diaphragm type only. The countermeasure of permanent deformation of Gas meter demands for detailed knowledge of its structural integrity and the properties of the material. For this reason, the combination of numerical and experimental simulation is a very promising way.

Simulation is the imitation of some real thing, state of affairs, or process. The act of simulating something generally involves representing certain key characteristics or behaviors of a selected physical or abstract system (Davis and Eisenhardt, 2007). Simulation is used in many contexts, including the modeling of natural systems or human systems in order to gain insight into their functioning. Other contexts include simulation of technology for performance optimization, safety engineering, testing, training and education. Simulation can be used to show the eventual real effects of alternative conditions and courses of action. In the future, CAE systems will be major providers of information to help support design teams in decision making (Carrol and Harrison, 1998).

EXPERIMENTAL WORK.

Structural Analysis of Measuring Unit

Structural analysis comprises the set of physical laws and mathematical required to study and predicts the behavior of structures. The subjects of structural analysis are engineering artifacts whose integrity is judged largely based upon their ability to withstand loads; they commonly include buildings, bridges, aircraft, and ships (Fakhir and Muzzaffar, 2009). Structural analysis incorporates the fields of mechanics and dynamics as well as many failure theories. From a theoretical perspective the primary goal of structural analysis is the computation of deformations, internal forces, and stresses. In practice, structural analysis can be viewed more abstractly as a method to drive the engineering design process or prove the soundness of a design without a dependence on directly testing it (Hartmann 1996).

Gas meter consist of three major components, Upper case, Lower case and measuring unit. Measuring unit is a part from where gas flows and the reading is measured. Figure 1 shows the outer surface of measuring unit on which we will do analysis so that it can withstand high external and internal forces.

Figure 1. Isometric view of shell

Pre-Processing

Now to analyze the part on ANSYS Multiphysics 11.0, it is necessary to create a Mesh. Meshing means discretization of the whole part into number of elements. The finite element mesh subdivides the geometry into elements, upon which are found nodes. We need now to define the Material Properties and to apply the Boundary Conditions; we selected SOLID 92 as material type because it has a quadratic displacement behavior and is well suited to model irregular meshes. The element also has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

The material of the measuring unit is Acetal Copolymer F-20-02 and the properties are;

Tensile Modulus (Elastic Modulus) = 28,900 “Kg/cm^2”= 2832.2 “N/mm^2 ”

Poisson’s Ratio = 0.35

Material properties were then assigned to the model.

Applying Boundary Conditions

Boundary Conditions are the loads and constraints that represent the effect of the surrounding environment on the model. Loads can be forces, moments, pressures, temperatures, accelerations etc. Constraints resist the deformations induced by the loads (Law and Kelton, 1991). In our case, the load is applied on the two extrusions of the Shell through which the Measuring Unit is supported on the Lower Case. In figure 2 those areas are highlighted which are experiencing impact force due to direct contact with the lower case of the gas meter.

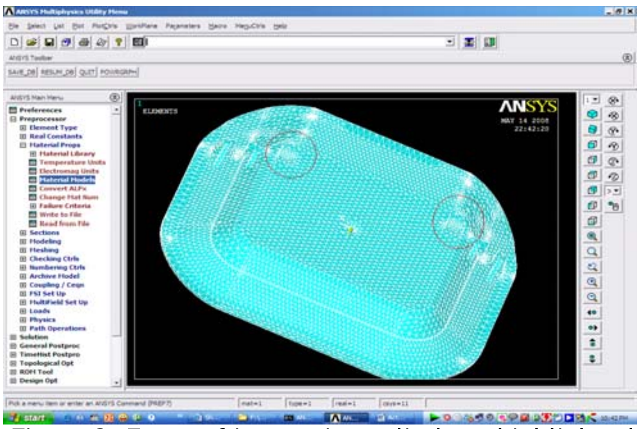

Figure 2. Force of impact is applied are highlighted

The impact pressure was than calculated from the laboratory by direct testing and it was found to be 15.6 “N/mm2”. It should be noted here that since the extrusion are on the supports at half of their lengths, so the pressure applied at half of the area. Figure 3 shows the pressure applied on half of the extrusion of the model.

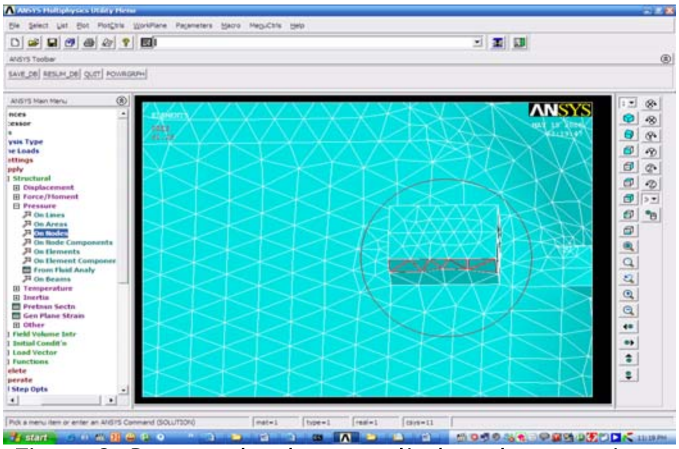

Figure 3. Pressure has been applied on the extrusion

Now, since the Shell is ultrasonically welded with the rest of the Measuring Unit, so the Degree of Freedom constraints were applied on the edges of the part as shown in Figure 4.

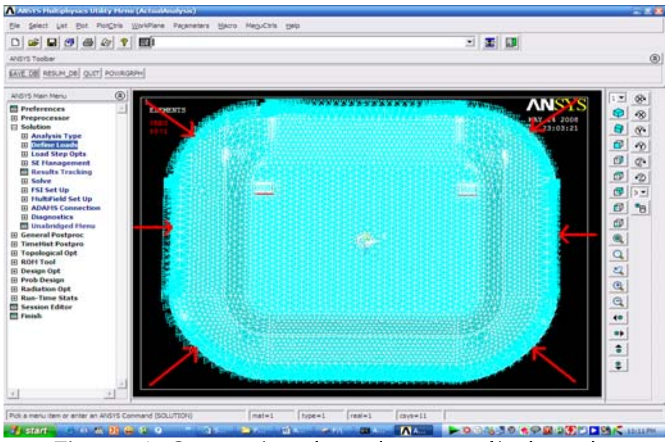

Figure 4. Constraints have been applied on the boundaries

SOLUTION

The solution is often a batch process, and is demanding of computer resource. The governing equations are assembled into matrix form and are solved numerically. The assembly process depends not only on the type of analysis (e.g. static or dynamic), but also on the model's element types and properties, material properties and boundary conditions (Repenning, 2002).

POST-PROCESSING

After a finite element model has been prepared and checked, boundary conditions have been applied, and the model has been solved, it is time to investigate the results of the analysis. This activity is known as the post-processing phase of the finite element method (Gani and Pistikopoulos, 2002).From Figure 5 it can be analyzed that the maximum deformation occurs at the two extrusions which are actually acting as the supports through which the Measuring Unit is mounted on the Lower Case.

2011. Fascicule 3 [July–September]. ©copyright FACULTY of ENGINEERING – HUNEDOARA, ROMANIA

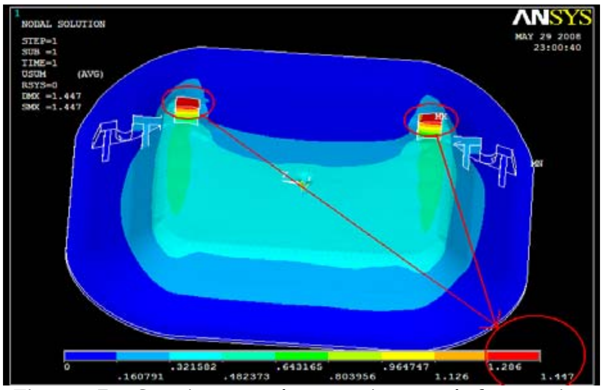

Figure 5. Sections under maximum deformation

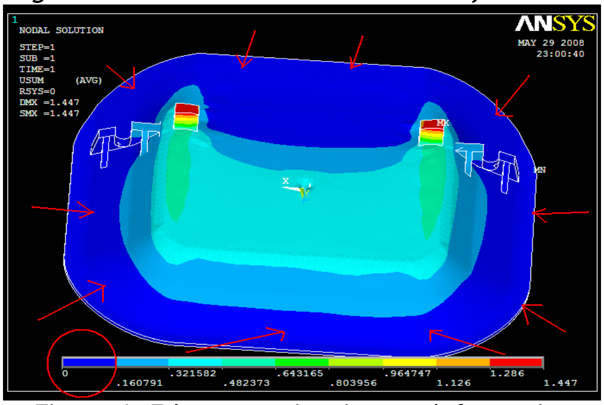

Figure 6. Edges experiencing no deformation

It is also worth noted that since the extrusions were subjected to direct force at half of their lengths, so only half of the length is under maximum deformation. It is observed during analysis that after the application of sudden load, both extruded entities are deformed up to major extent, causing the areas below them to deform, which resulted in the deformation in the middle-portion of the part. The complex contours besides the extrusion entities can be find minimizing and limiting the extent of Deformation.

Now since the corners of the Shell are Ultrasonically Welded with the Measuring Unit, Figure 6 shows that the corners and boundaries did not experience any deformation.

STRESSES

The result of key concern in this case was the stresses generated in the part, after the application of Impact force because these stresses were responsible for the cracks than fracture of the Shell.

From Figure 7, it can be observed that stresses are maximum near the base of extruded part and from then they propagates along the sharp geometry to the upper corner of the part. Now according the Material Specifications from the data sheet, the Tensile Strength of the Acetal Copolymer F20-02 is;

Tensile Strength = 61.25 "N/mm2".

Comparing the Tensile Strength with the stress bands, it can be noted that the stresses in the 'Red' band (SEQV = 55.549 "N/mm2"- 62.492 "N/mm2") are greater than the Tensile Strength of the material (UTS = 61.25 "N/mm2") so the fracture is certain here.

Red band is preceded by the bands in 'Orange' (SEQV = 48.605 "N/mm2"- 55.549 "N/mm2") and 'Yellow' (SEQV = 41.662 "N/mm2"- 48.605 "N/mm2"). The magnitude of these stresses is also enough to generate cracks.

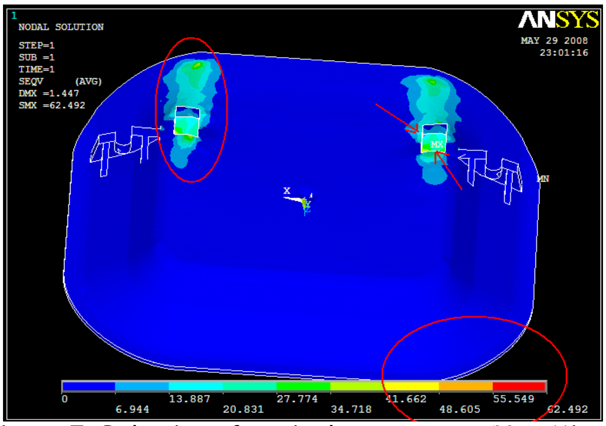

Figure 7: Behavior of equivalent stresses (Von-Mises) after the impact force has been applied

SOLUTIONS FOR THE PROBLEM

The problem of part fracture due to high forces can be solved from different alternative solutions, while modifying the design, we have to keep the modifications as simple as possible, so that the mold may require least amendments and the design changes may be implemented in shortest possible span of time. One solution is to increase the width of the extrusion but this need major changes in the design of the mold which require redesigning of the mold in the software, next step is to analyze the resigned mold on a software to justify the changes, all these steps consume much resources.

Another solution can be increasing the thickness of the extrusion which again requires amendments in the mold and the design changes may lead to the increase in cost and time. The third solution can be to support the extrusion by ribs which require great concentration in the design work because after seeing the current design of the Gas meter it is very difficult to design a rib but research has been done on ribs. So, assistance can be taken from previous research regarding the design of ribs on a complicated mold.

The fourth solution can be to fill the hole of the extruded part completely; this countermeasure needs to just machine the mold from the surface of the extrusion which will just need the cost of designing and machining. As computers become faster and computationally more powerful, the range of application for computer simulation and modeling has also expanded. These benefits should be utilized in the solution of the current problem.

CONCLUSION AND FUTURE WORK

We presented here a basic model of Gas meter and simulation results with Ansys Multiphysics. The problem with the current model is that it can not bear high external and internal forces.

The actual model was then physically tested in laboratory using Tensile Testing machine, the readings was than taken and utilized as input parameters on Ansys Multiphysics. Naturally the accuracy of computer simulation greatly depends on the quality of the input parameters, so laboratory testing was the important phase of the project. The results clearly show that the stresses (62.492 "N/mm^2") generate on the measuring unit was greater than its tensile stress (61.25 "N/mm^2"). Summarizing the four solutions presented, the last solution, to fill the hole of the extruded part completely is less cheap and achievable as it consumes less resources. In the future the reader can analyze the four designs presented or present his own design which should be efficient to design and implement.

The future is very bright for Simulation as it includes more efficient utilization of resource and cost saving from its increased efficiency. Indeed, in many fields, computer simulation is integral and therefore essential to business and research. Computer simulation provides the capability to enter fields that are inaccessible to traditional experimentation. It uses mathematical models and numerical solution techniques and using computers to analyze and solve scientific and engineering problems. By using the advanced simulation techniques in the future we can be able to design aircraft which has light weight, no need to build tunnel for physical testing of the wings of airplane. It can assist Fluid mechanics and heat transfer problems, the flow of material inside a mold can easily be controlled. This paper can also assist other engineering fields.

ACKNOWLEDGEMENTS

This research has been assisted by the Professors of Industrial & Manufacturing Engineering Department of NED University of Engineering & Technology and Deputy Chief Engineer of Sui Southern Gas Company.

REFERENCES

[1.] Carroll, G., & Harrison, J. R. 1998. Organizational Demography and Culture: Insights from a Formal Model and Simulation. Administrative Science Quarterly, 43: 637-667

[2.] Davis, J.P., Eisenhardt, K.M. & Bingham, C.B. 2007. Developing Theory with Simulation Methods. Academy of Management Review, 32(2), 580-599

[3.] Dr. Fakhir Hasani and Dr. Muzzaffar Mahmood, (2009). "Computational Fluid Dynamics Modeling in Narrow Obstructed Channels of Membrane Modules". 4th International Mechanical Engineering Congress, pp. 03.

[4.] Gani R., Pistikopoulos E.N., "Property Modeling and Simulation for Product and Process Design", Fluid Phase Equilib., 194-197, 43-59, 2002.

[5.] Law, A. M., & Kelton, D. W. 1991. Simulation Modeling and Analysis (2nd ed.). New York, NY: McGraw-Hill.

[6.] Repenning, N. 2002. A Simulation-Based Approach to Understanding the Dynamics of Innovation Implementation. Organization Science, 13(2): 109-127

[7.] S. Hartmann, The World as a Process: Simulations in the Natural and Social Sciences, in: R. Hegselmann et al. (eds.), Modelling and Simulation in the Social Sciences from the Philosophy of Science Point of View, Theory and Decision Library. Dordrecht: Kluwer 1996, 77-100

AUTHORS & AFFILIATION

[1]Nazimuddin QURESHI,
[2]Muhammad Arsalan FAROOQ,
[3]MD. Mushtafizur REHMAN,
[4]S.M. Sohaib TARIQ

[1.] MECHANICAL & MANUFACTURING ENGINEERING, NED UNIVERSITY, KARACHI, PAKISTAN
[2.] INDUSTRIAL ENGINEERING, LUCIAN BLAGA UNIVERSITY OF SIBIU, ROMANIA
[3] COMPUTER ENGINEERING, LUCIAN BLAGA UNIVERSITY OF SIBIU, ROMANIA
[4] MECHANICAL ENGINEERING, UNIVERSITY OF FREIBURG, GERMANY

ACTA TECHNICA CORVINIENSIS
– BULLETIN of ENGINEERING
ISSN: 2067-3809 [CD-Rom, online]