ANNALS of the ORADEA UNIVERSITY.
Fascicle of Management and Technological Engineering, Volume XI (XXI), 2012, NR1

CYLINDRICAL PARTS STRESS STUDY BASED ON THE BOTTOM GEOMETRY SHAPE

Pop Alin[1], Grebenişan Gavril[2], Radu Ioan[3]
[1]University of Oradea, afpop@uoradea.ro
[2] grebe@uoradea.ro, [3] iradu@uoradea.ro

Keywords: finite element analysis, cylindrical parts, stress study

Abstract: The level of stress for different types of loads, affects the future processing that are made to the parts. The bottom part geometry is one of the most important elements that affect the place where the stress is at high level. This paper aims to determine the relationship between the bottom geometry of the part and the position where maximum stress occurs.

1. INTRODUCTION

As it was demonstrated in [1] maximum stress occurring in the cylindrical parts are located at the bottom of the parts. Geometry bottom part is an important factor in the stress study. The two main features are the geometry part and the bottom part radius that connect the bottom with sidewalls. In this paper we determine how two parameters influence the value and the place where the stress occurs, with finite element analysis and response surface method.

The surfaces responses are functions in witch the output parameters are described based on input parameters. They are determined in Design of Experiment tools from Ansys, in order to provide in the fastest way, values that are close to the output parameters, without generating a new finite element analysis.[4] Accuracy of response surface method depends on several factors: the complexity of solution diversity, number of attempts set in "Design of Experiment" and the type of response surface. DesignXplorer ANSYS gives us tools to estimate and improve the quality of results. Once obtained response surfaces can create and manage points of response and graphics. Post-processing tools allow us to understand how results are influenced by the input parameters.

2. FINITE ELEMENT MODEL.

In this study the deformation process simulation is performed using ANSYS program. Type of analysis used is the Static Structural type (ANSYS) - Mechanical. The part geometry was created in environment modeling program DesignModeler from Ansys. At the blank form is distinguished several geometric features that allow obtaining the desired results, the cone-shape of the blank and the bottom radius that is connecting with the sidewalls. At the blank manufacturing, we leave from a flat sheet and the first operation is performed to obtain a conical bottom piece. To achieve the desired geometry of the part-formed, we need more deep drawing operations.

We chose a blank that is made from Steinless Steel (material that is used throughout the study), thickness of the at 0.3 mm.

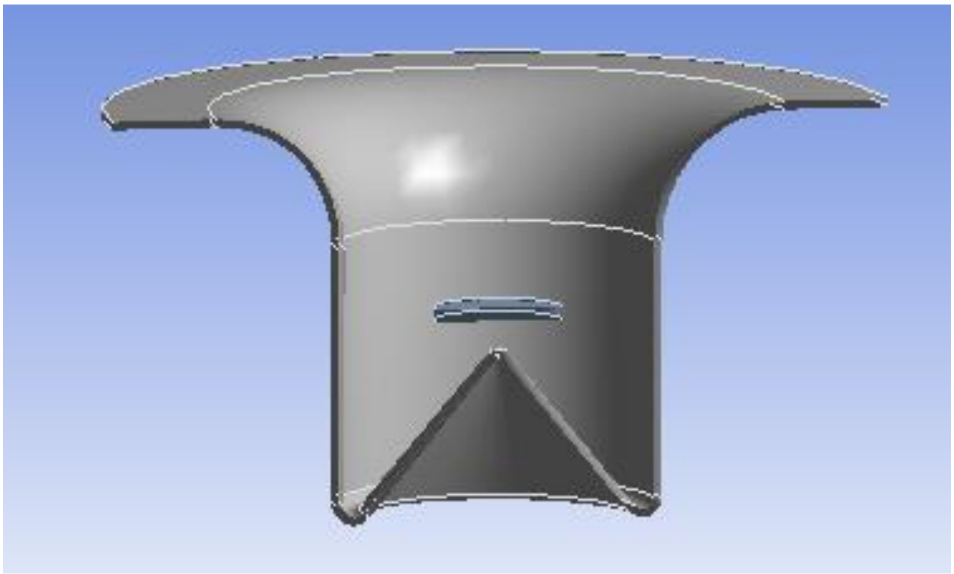

Figure 1. Blank geometry

3. GEOMETRY PART INFLUENCE

Simulation processes using finite elements is a very powerful tool for investigation and observation of interactions between parameters and material behavior. It provides very useful information for the production process. In this case the first stage we analyze the influence of cone-shape angle on the stress value and the place where the stress is maximal.

In the analysis as input parameter we defined the cone-shape angle and as output parameter we determined the equivalent stress value occurred.

Once set the input parameters, we can update the "design of experiments" tool witch can send directly the parameter of the system for finding the right solution for each value. Once the solutions have been found based on response surface method, it will generate dependency diagram of the equivalent stress according to the cone-shape angle. In the figure below are shown how the angle affects the equivalent stress values. On figure 2 we have the half values on the angle radius

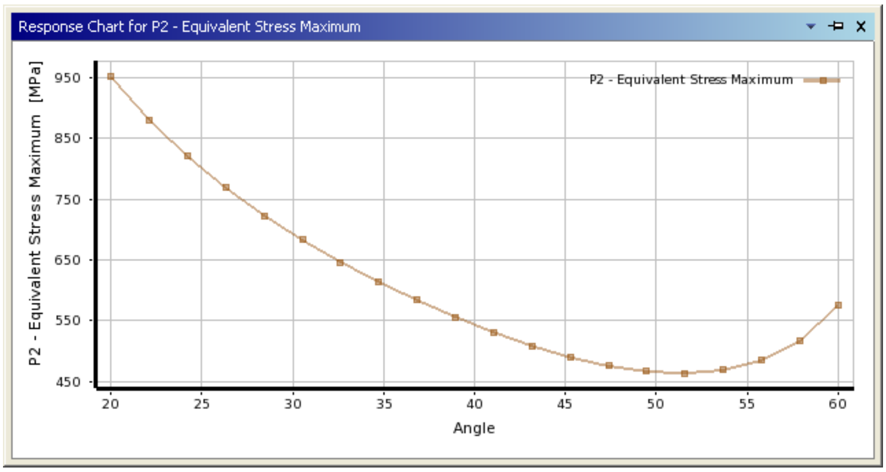

Figure 2. Dependency diagram of the equivalent stress

It can be see how the maximum stress occurs at small angles of cone-shape. But the place where it has maximum value is not the same. In the figures below we observed that for angle of 120 degrees, maximum stress occurs in the lower area at the base of the part and the values of the angle of 90 degrees it appears in the upper area (figure 3)

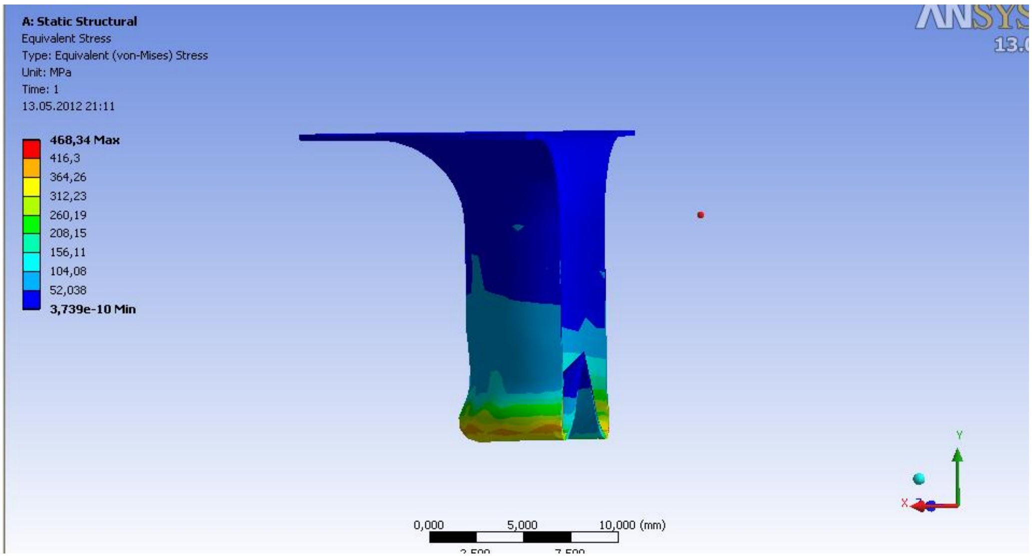

Figure 3. Finite Element Analysis of 90 degrees cone-shape angle

A second parameter that influences the level of the stress on cylindrical parts is the radius of the bottom part and side walls. we studied as a range of radius with values between 0.3 and 1.2 mm. Were generated in the tool Design of Experiment

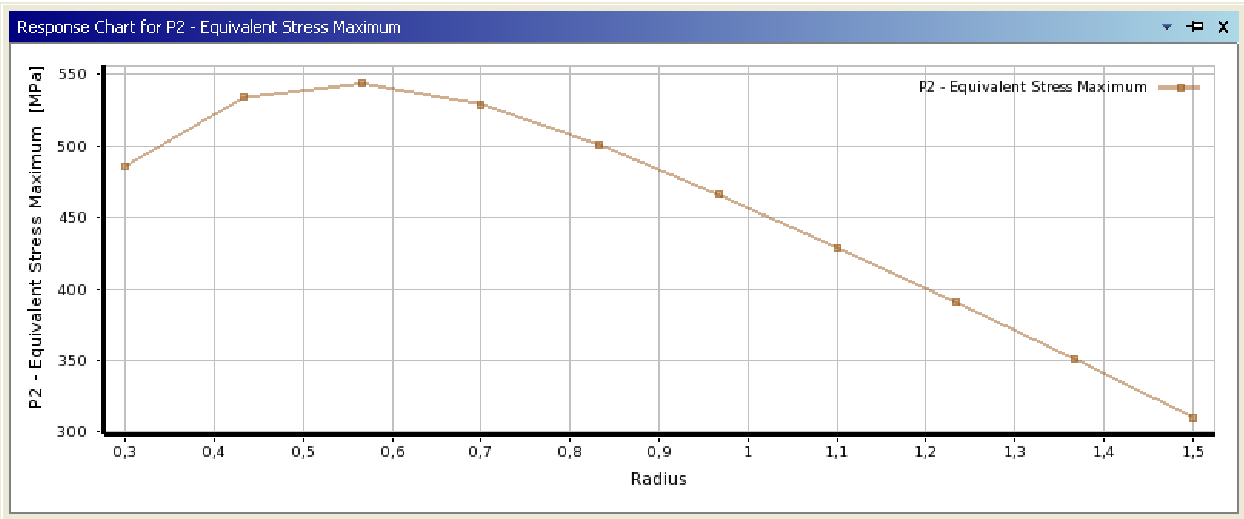

Figure 4. Response chart of radius influence on equivalent stress value

So far we have shown how each parameter influences the stress value in the part. We must take into consideration the fact that the combination of the two input parameters can influence the final outputs. It was made experiments with eight rays and three values of the cone-shape angle.

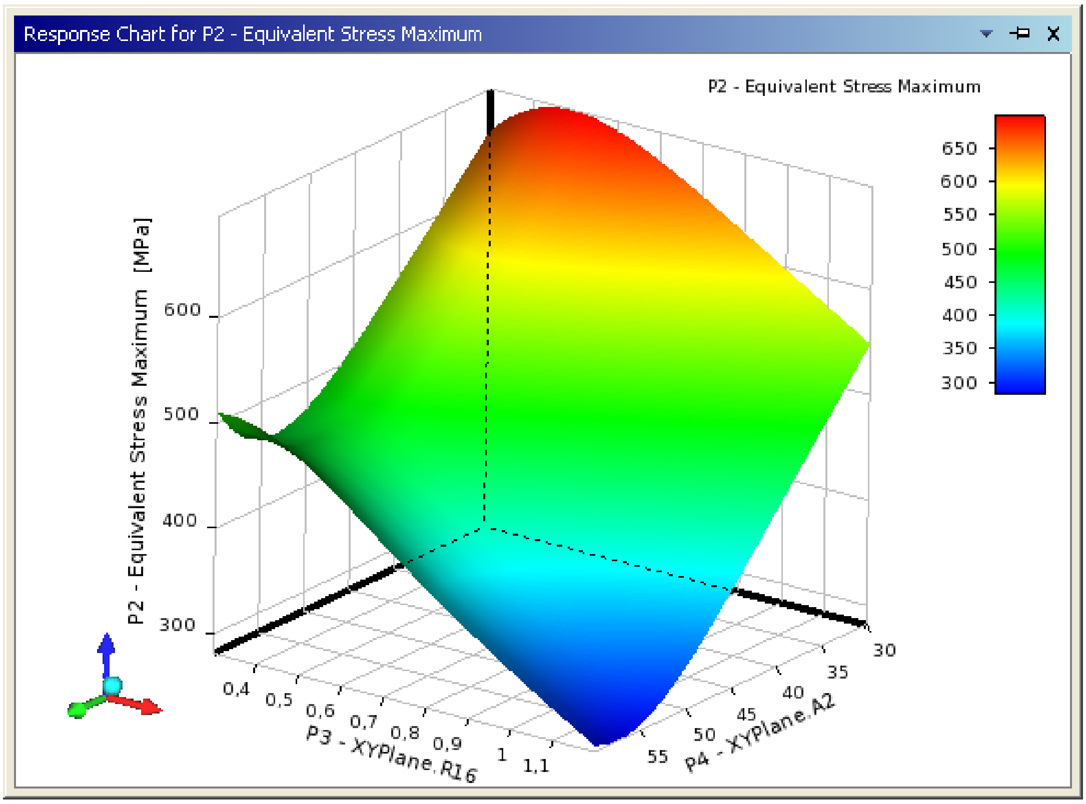

Figure 5 Response Char for Equivalent stress

4 CONCLUSION

Equivalent stresses occurring at the bottom of cylindrical components are important in terms of further processing that occurs on the part. The value and the place, in which these have maximum level, are influenced by several geometry parameter of the bottom part. In this paper we made an analysis of the influence of two parameters, the cone-shape angle and radius base part. We performed a finite element analysis of the model based on response surface methods were it has been set up the influence graphic of the two input factors for the output parameters.

Reference:

1. **Radu Ioan Eugen, Grebenişan Gavril,(2010) FINITE ELEMENT ANALYSIS OF A FORMING PROCESS, USING STATIC STRUCTURAL (ANSYS) AND ANSYS LS-DYNA, Annals of the Oradea University. Fascicle of Management and Technological Engineering, ISSN1583-0691, pp.1171-1187, articol în revistă acreditată CNCSIS „Clasa B+''**
2. **Radu Ioan –Tehnologia presarii la rece- Note de curs**
3. **Huei-Huang LeeFinite Element Simulations With Ansys Workbench 12 SDC Publication ISBN: 978 1-58503-604-2**
4. **http://www1.ansys.com/customer/content/documentation/121/wb_dx.pdf**