International Journal of Engineering and Manufacturing Science.
ISSN 2249-3115 Volume 7, Number 2 (2017), pp. 259-272

http://www.ripublication.com

Finite Element Analysis of Hydroforming Components (Bellows) Using ANSYS

G. Jagan Naik[1], Dr. M. Devaiah[2], Pogu Raamcharaaun[3]

[1, 3]Department of Mechanical Engineering, Avanthi's Scientific Technological Research and Academy, Hayathnagar, R.R. Dist, Telangana, India.

[2]Department of Mechanical Engineering, Geethanjali College of Engineering & Technology, Cheeryal (V), Keesara (M), Medchal Dist. 501301, Telangana, India.

Abstract

The application of finite element Analysis (FEA) in the area of metal forming and material processing has been increasing rapidly during the recent years. Hydro forming is a manufacturing process where fluid pressure is applied to ductile metallic blanks to form desired component shapes. The blanks are either sheet metal or tabular sections. If sheet metal blanks are used, the process is called sheet metal hydro forming, and if tabular-section blanks are used, it is called tube hydro forming. In either of these processes, a hydro forming tool (or die), a hydraulic press and a fluid-pressure intensification system are required.

The challenges that are present during sheet hydro forming processes are divided into two categories viz Material and Fluid Pressure. The material challenge refers to the choice and behavior of the sheet metal. One of the major obstacles concerns the balance between the fluid pressure and the ductility of the material chosen for the hydro forming process. The fluid pressure needs to be high enough to stretch and bend the sheet through its radius of curvature to conform to the shape of the punch yet the material needs to be ductile enough to form without rupturing. Thus the Project mainly focussses on study of effect of the two parameters i.e. material and fluid pressure on the hydro formed components. Finite element modeling and analysis of sheet hydroforming has been performed using ANSYS.

1. INTRODUCTION

Hydro forming, sometimes referred to as forming or rubber diaphragm forming was developed during the late 1940's and early 1950's in response to a need for a lower cost method of producing relatively small quantities of deep drawn parts. The hydro forming technology is well-known since several years and has undergone an extremely dynamic development in automotive applications, especially in Germany and the United States. The increasing interests on hydro forming operations in automotive field have pushed a strong scientific interest in various investigations of this process.

YU Ying, Zhu Qing-nan, YU Xiao-Chun, LI Yong-Sheng [1] has carried out their study on application of particle swarm optimization algorithm in bellow optimum design.

In this paper, Particle Swarm Optimization Algorithm (PSOA) is used in problem of the bellow optimum design with constraints, in which the design variables are discrete. To implement bellow optimum design by PSOA, an augmented objective function is constructed based on penalty function and a new updating scheme of penalty parameter '**s**' is proposed. A new Discrete PSOA (DPSOA) is proposed. The mathematic model of bellow optimum design is established. Through numerical examples of bellow design, comparing the results of examples by proposed DPSOA with the theory solutions by Net method, it shows that the particle swarm optimization algorithm can be applied to the bellow optimum design successfully and satisfactory results by DPSOA are obtained, which is discrete optimal solution in the feasible domain.

Zhang Yong, Luen Chow Chan, Wang Chunguang, and Wu Pei [2] have carried out their study on 'optimization for loading paths of Tube Hydro forming using a Hybrid method'.

In this article, a hybrid method is proposed to optimize loading paths of THF. Firstly, a three-layer back-propagation artificial neural network (BP-ANN) is built, and 200 samples from finite element (FE) simulations are applied to train and test the artificial neural network (ANN). Then genetic algorithm (GA) is adopted to search the optimal loading paths in the specified bounds of the design variables by using the trained ANN as the solver of the objective function and constraint functions. After 59 iterations, the optimal loading paths are obtained. Finally, the verified experiments are performed on the special hydro forming press. The results show that the

Proposed method can effectively search the optimal loading paths of THF and remarkably improve the quality of the final formed parts.

F. Capece Minutolo, M. Durante, A. Formisano, and A. Langella [3] has carried out their study on optimization of a hydro forming process to realize asymmetrical aeronautical components by FE analysis.

In this work a numerical simulation has been carried out using the explicit finite element code LS-DIANA with the purpose to optimize a hydro forming process related to the achievement of asymmetrical aeronautical component. In particular, the number of the steps of of the manufacturing cycle actually used to produce the component has been reduced considering the FEM results.

F. Djavanroodi and M. Gheisary [4] has carried out experimental tests and finite element analysis to establish a basic understanding of Double Bulge Tube Hydro-Form processing of stainless steel deep drawn cups.

By measuring bulge height in both formed curves by Coordinate measuring machine (CMM) and thickness variation specimen by Ultrasonic thickness measurement device (UTM), it has been shown that maximum thinness occurred where the bending is maximized. A finite element model is constructed to simulate the Double Bulge Tube Hydro Forming process and asses the influence of friction coefficient, tube Material properties and spring back. It has been shown that material hardening coefficient had the most significant influence on formability characteristics during double bulge tube hydro forming. Also it is shown that spring back has significant effect on tolerances of formed tube. Finally fracture strain was estimated by analytical method and compared with simulation results; also fracture location was predicted on Double Bulge Tube Hydro-Forming (DBTHF) by simulating the process.

R. Di Lorenzo, G. Ingarao, F. Gagliardi and L. Filice [5] has carried the experimental validation of optimization strategies in hydro forming of T-shaped tubes.

C.T. Kwan and F.C. Lin [6] have carried out the investigation of T-shape tube Hydro forming with Finite element method.

In this paper the finite element method is used to investigate the cold hydro forming process of a T-shape tube. A series of simulation on hydraulic expansion, axial feeding and the counter force of the tubes was carried out using the program DEFORM-3D. The influences of the process parameters such as the internal pressure, the fillet radius and counterforce on the minimum wall thickness of formed tube are examined. A suitable range of the process parameters for producing an acceptable T-shape tube that fulfills the industrial demand was also found.

The material assumptions and friction are the same. The feed is only from the top in the experiments and in the simulations. The piston initially pushes the blank axially but the pin is fixed so as to incur little pressure increase. The result is a large axial load in the blank which tends to buckle the blank as soon as the blank begins to yield very early in the process. Double wrinkle is also clearly observed in the experiments. Axial load on the tube ends continues to increase until at some point the tube begins to soften axially as well. They observed that original both bulges occurred at about the same time and grew at the same rate but because the valley of the lower bulge is being forced into the lower die radius, this produce an apparent stabilizing effect and inhibits the lower bulging. Also observed in the pressure signal is the sudden increase when the peaks of the bulges reach the die. These simulations suggest a strategy for minimizing the potential for bursting while preventing excessive wrinkling.

2. FINITE ELEMENT DESIGN AND ANALYSIS OF HYDRO FORMING COMPONENTS

ANSYS is the general-purpose Finite element-modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both Linear and Non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

The name Finite element summarizes the basic concept of the method of transformation of an engineering system with an infinite number of unknowns (the response at every location in a system) to one that has a finite number of unknowns related to each other by elements of finite size. The unknowns called degrees of freedom represent the responses to applied action. The degrees of freedom and the actions are related by a set of basic equations. The purpose of the finite element method is to determine the basic solution to these equations across the entire engineering system being analyzed.

In general, a finite element solution may be broken into fallowing three stages. This is a general guideline that can be used for setting up any finite element analysis.

1. Preprocessing
2. Solution
3. Post processing

In the pre-processing phase, you describe the problem to be solved by providing the model geometry and finite element data, material data, boundary conditions etc., to the program. In the solution phase, the program puts all the model data together, assembles and solves simultaneous equation and evaluates the stresses. In the post-processing phase, you evaluate the results calculated in the solution phase by producing contour displays, tabular listings etc.

3. DESIGN OF BELLOWS AND ITS STRESS ANALYSIS

The cylindrical bellows are design by using ANSYS; Metal bellows are cylindrical vessels that can be compressed when pressure is used on the top or the bottom of the vessel (or both). When the pressure is removed, the bellow will return back to its original shape (provided the material has not been 'pushed' past its yield strength.

3.1. Tube Hydroforming Analysis of Bellows

To design the bellows in FEM the following material and geometry properties are used

Material Name	Young's Modulus	Poisson's Ratio	Yield Stress	Tangent Modulus	Internal Diameter	Thickness	Height
316L Stainless Steel	1.93e5 N/mm^2	0.3	290 N/mm^2	2000 N/mm^2	400 mm	0.6 mm	400 mm

The bellows are designed and analysis was done with internal pressure values varies from 0.25 N/mm^2 to 2 N/mm^2 in the interval of 0.25 N/mm^2

4. RESULTS

(a) Deformation in Bellow at Various Pressures

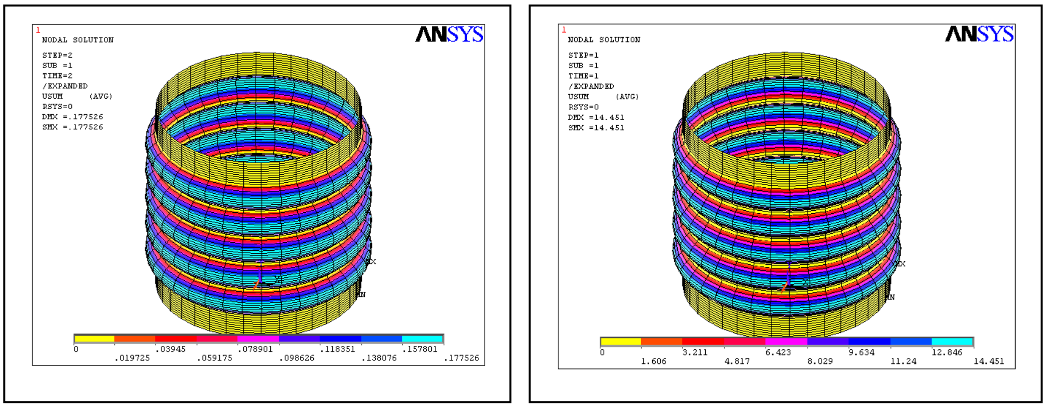


Figure 1: Represents the values of deformation in bellow at (A) a pressure of 0.5 N/mm^2 (B) a pressure of 1.5 N/mm^2

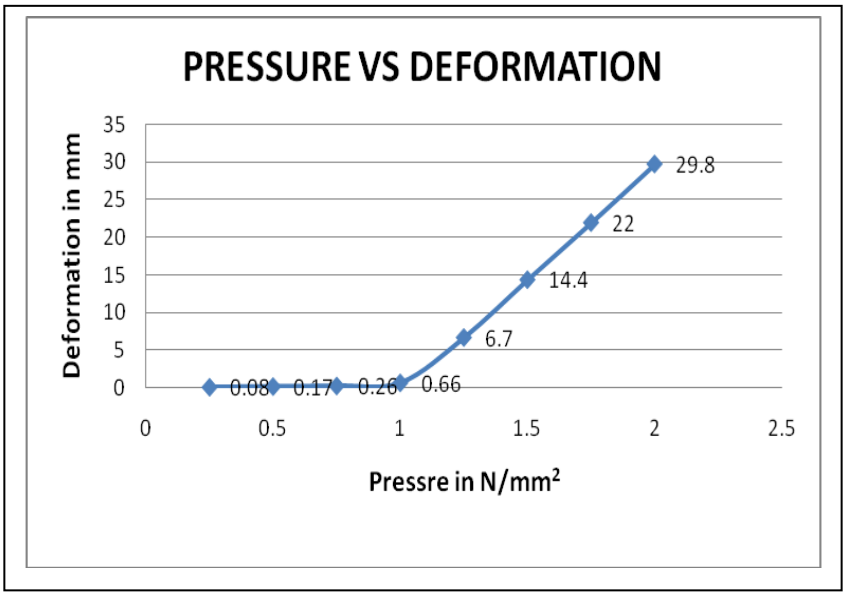


Figure 2: Shows the effect of pressure on deformation in bellow.

Figure 1 shows the deformation in cylindrical bellows at different pressures such as 0.5 N/mm^2 and 1.5 N/mm^2 and the deformation values at these pressures are 0.177 mm and 14.45 mm respectively.

Figure 2 show the variation of deformation with respect to different pressure values.

(b) **Elastic Stress in Bellows at Various Pressures**

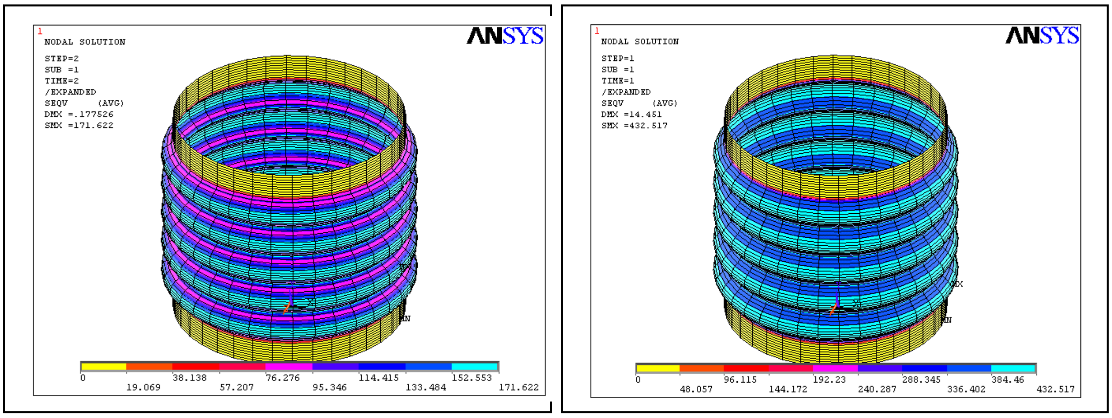


Figure 3: Represents the values of elastic stress in bellow at (A) a pressure of 0.5 N/mm^2 (B) a pressure of 1.5 N/mm^2

Figure 3 shows the elastic stress in cylindrical bellows at different pressures such as 0.5 N/mm^2 and 1.5 N/mm^2 and the elastic stress values at these pressures are 171.422 N/mm^2 and 432.517 N/mm^{2}respectively.

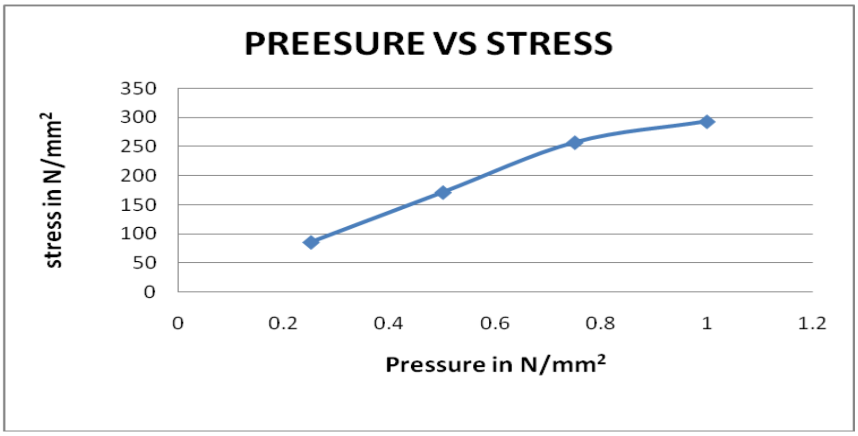


Figure 4: shows the effect of pressure on elastic stress in bellow

Figure 4 show the variation of elastic stress with respect to different pressure values.

(c) Elastic Strain in Bellows at Various Pressures

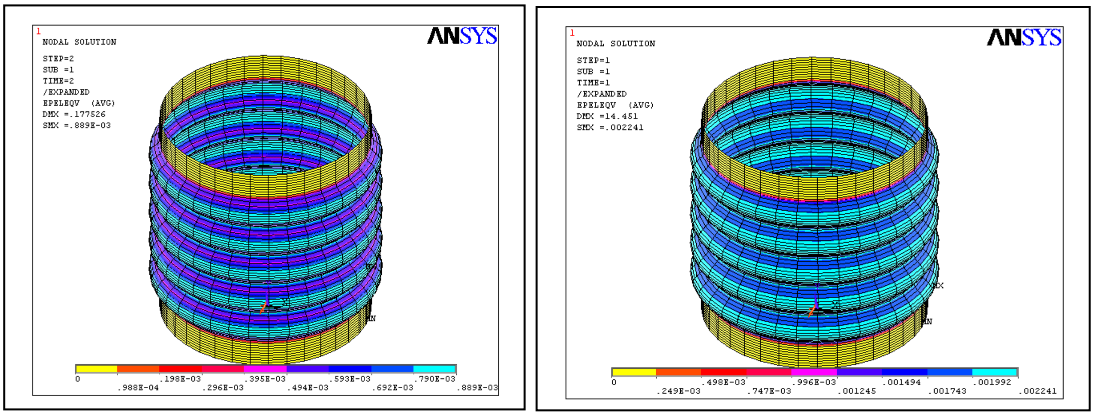


Figure 5: Represents the values of elastic strain in bellow at (A) a pressure of 0.5 N/mm^2 (B) a pressure of 1.5 N/mm^2

Figure 5 shows the elastic strain in cylindrical bellows at different pressures such as 0.5 N/mm^2 and 1.5 N/mm^2 and the elastic stress values at these pressures are 889E-03 and .002241 respectively.

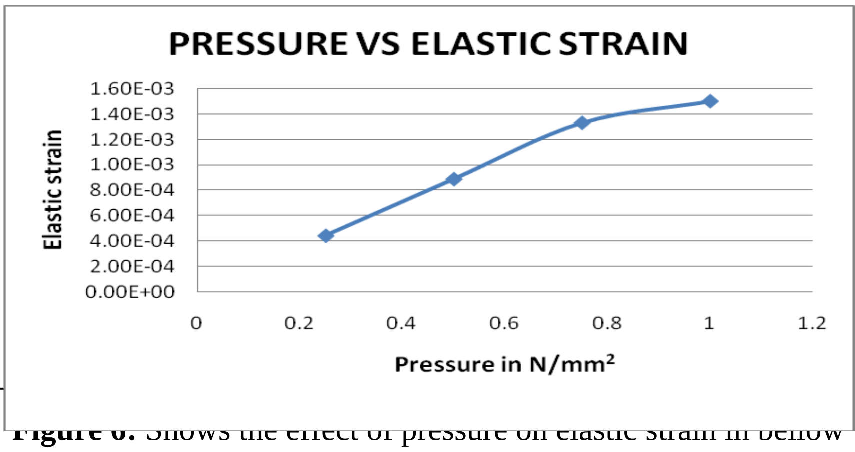


Figure 6: Shows the effect of pressure on elastic strain in bellow

Figure 6 show the variation of elastic strain with respect to different pressure values. The elastic strain increases as pressure increases.

(d) Plastic Strain in Bellow at Various Pressures

Figure 7 represents the values of plastic strain in bellow at (A) a pressure of 0.25 N/mm^2 (B) a pressure of 1.5 N/mm^2

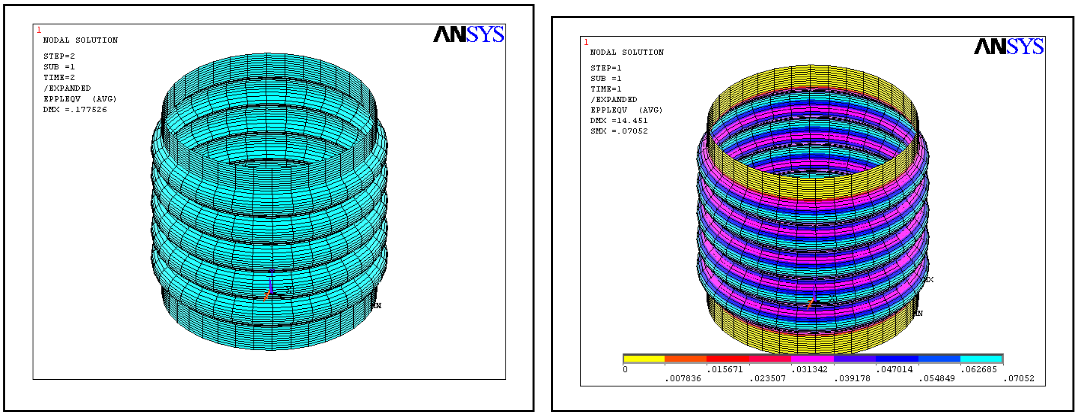


Figure 7: Shows the plain strain in cylindrical bellows at different pressures such as 0.25 N/mm^2 and 1.5 N/mm^2 and the elastic stress values at these pressures are 0 and 0.07052 respectively.

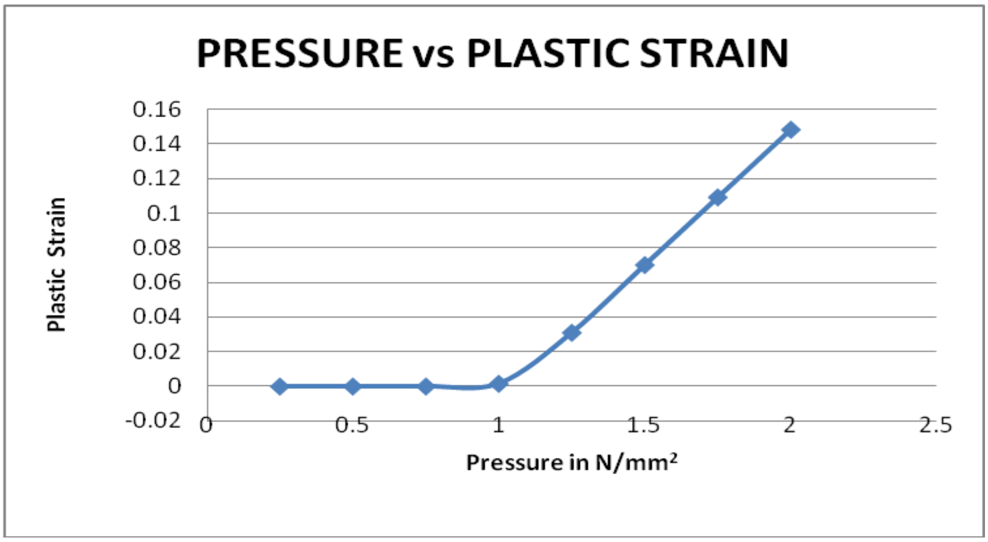


Figure 8: Shows the effect of pressure on plastic strain in bellow.

Figure 8 show the variation of plain strain with respect to different pressure values. The plain strain increases as pressure increases.

(e) Stress and Total Strain in Bellows at Various Pressures

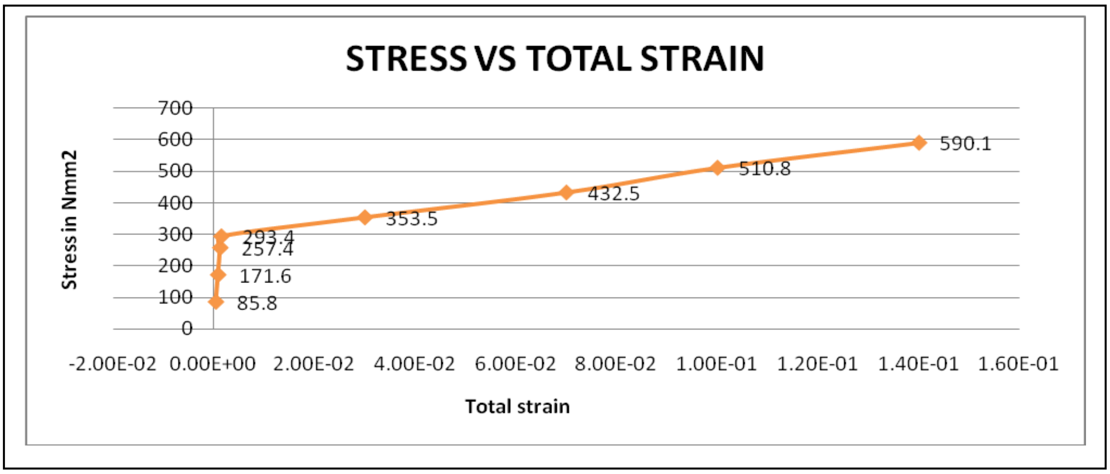


Figure 9: Shows the stress vs total strain in bellows at various pressures

Below the yield point plastic strain is almost nil, only elastic strain is occurred but that is very small (0.0015). Aftre the yield point plastic strain is started.

4.2. BELLOWS HYDROFORMING FOR MATERIAL WITH DIFFERENT PROPERTIES

4.2.1. STRESS ANALYSIS IN BELLOWS

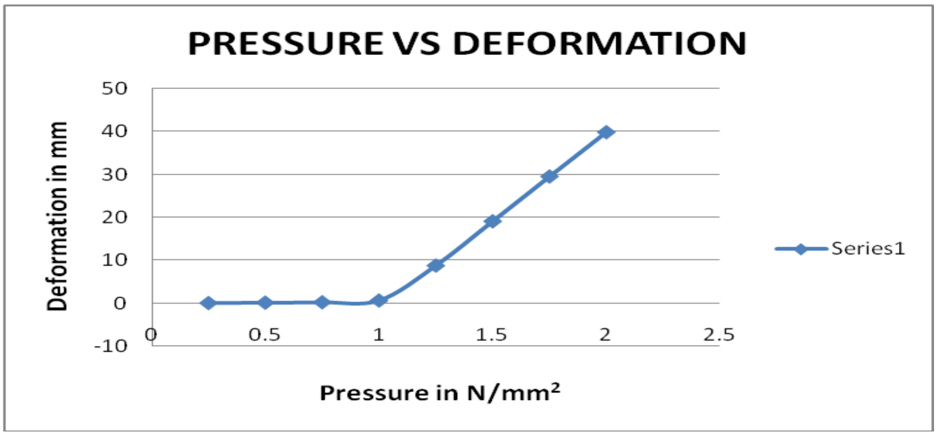
The following material and geometry properties are used.

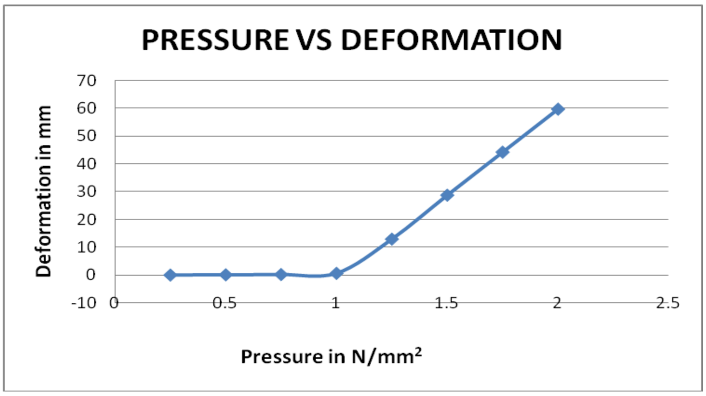
Material Name	Young's Modulus	Poisson's Ratio	Yield Stress	Tangent Modulus	Internal Diameter	Thickness	Height
316L Stainless Steel	1.93e5 N/mm^2	0.3	290 N/mm^2	2000, 1500, 1000, 500 N/mm^2	400 mm	0.6 mm	400 mm

The cylindrical bellows are designed using ANSYS and analysed the same at different tangential modulus in the range of 2000 N/mm^2, 1500 N/mm^2, 1000 N/mm^2 and 500 N/mm^2.

The bellows are designed and analysis was done with internal pressure values vary from 0.25 N/mm^2 to 2 N/mm^2 in the interval of 0.25 N/mm^2 and the results are plotted as follows.

(a) Deformation in Cylindrical Pipe at Various Pressures





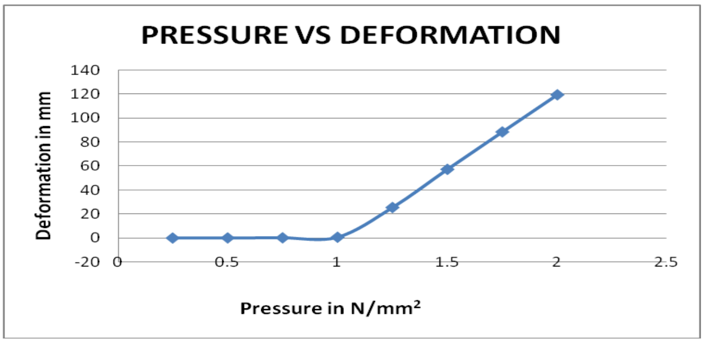


Figure 10: Shows the effect of pressure on deformation in bellow at (a) tangential modulus is 1500 N/mm^2 (b) tangential modulus is 1000 N/mm^2 (c) tangential modulus is 500 N/mm^2

Figure 10 shows the effect of pressure on deformation in cylindrical bellows at different tangential modulus such as 1500 N/mm^2, 1000 N/mm^2 and 500 N/mm^2 and the deformation is constant upto certain pressures and increases as increases the pressure and the deformation varies with tangential modulus as shown in above figure, varies from 40 mm to 120 mm at pressure 2 N/mm^2 for different tangential modulus values same can be observed in figure 11.

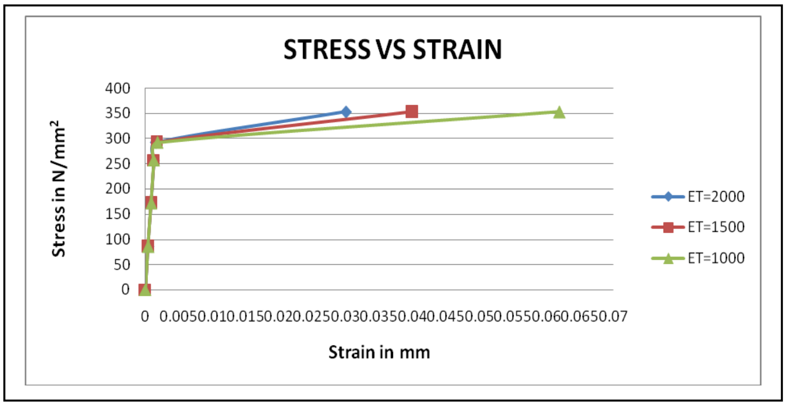


Figure 11: Shows the strain in the bellows at different stress of the tangential modulus.

(b) Plastic Strain in Bellows at Various Pressures

Figure 17 shows the effect of pressure on plastic strain in bellow at the tangential modulus is 1500 N/mm^2.

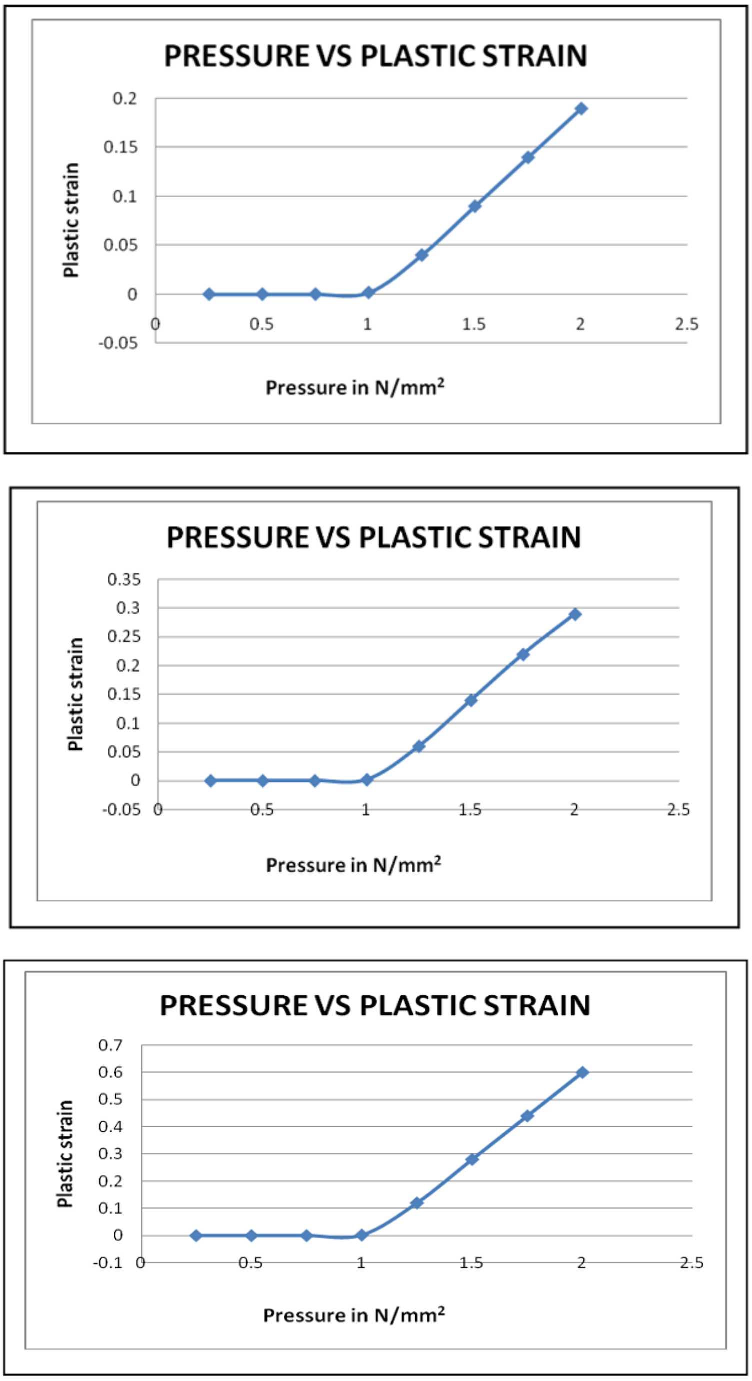


Figure 12: Shows the effect of pressure on plastic strain in bellow at (a) tangential modulus is 1500 N/mm^2 (b) tangential modulus is 1000 N/mm^2 (c) tangential modulus is 500 N/mm^2

Figure 12 shows the effect of pressure on plain strain in cylindrical bellows at different tangential modulus such as 1500 N/mm^2, 1000 N/mm^2 and 500 N/mm^2 and the plain strain is constant upto certain pressures and increases as increases the pressure and the deformation varies with tangential modulus as shown in above figure, varies from 0.175 to 0.6 at pressure 2 N/mm^2 for different tangential modulus values same can be observed in figure 13.

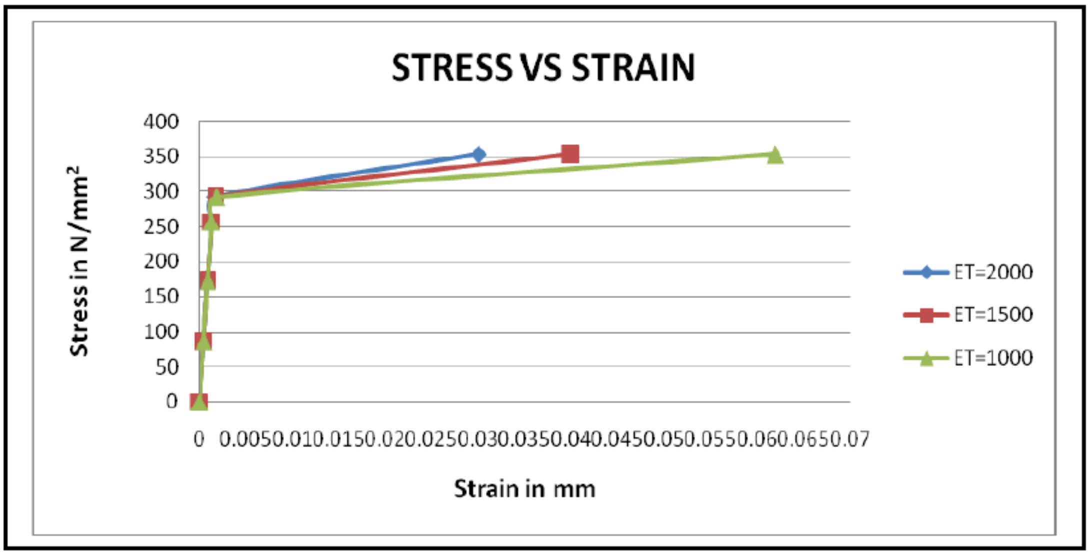


Figure 13: Shows the strain in the bellows at different stress of the tangential modulus.

Below the yield point plastic strain is almost nil, that is very small (0.0013). After the yield point plastic strain is started, similar to stress- strain curve for the material. To incresase the tangential modulous strsain is decreasess.

5. CONCLUSIONS

The effect of pressure on deformation, elastic stress and Plastic strains is observed. The Maximum values of deformation in bellow are 0.08 mm, 0.17 mm, 0.26 mm, 0.66 mm at pressures 0.25 N/mm^2, 0.5 N/mm^2, 0.75 N/mm^2 and 1N/mm2 respectively.

The Maximum values of elastic stress in bellow are 85.81 N/mm^2, 171.62 N/mm^2, 257.43 N/mm^2 and 293.4 N/mm^2 at pressures 0.25 N/mm^2, 0.5 N/mm^2, 0.75 N/mm^2 and 1 N/mm^2 respectively.

The Maximum values of elastic strain in bellow are 0.44e-3, 0.88e-3, 0.0013 and 0.0015 at pressures 0.25 N/mm^2, 0.5 N/mm^2, 0.75 N/mm^2 and 1 N/mm^2 respectively.

The Maximum values of plastic strain in bellow are 0, 0, 0, 0.001, 0.003, 0.07, 0.1 and 0.14 at pressures 0.25 N/mm^2, 0.5 N/mm^2, 0.75 N/mm^2, 1 N/mm^2, 1.25 N/mm^2, 1.5 N/mm^2, 1.75 N/mm^2 and 2 N/mm^2 respectively.

At the tangential modulus 1500 N/mm^2, 1000 N/mm^2 and 500 N/mm^2, the Maximum values of plastic strain in bellows are (0, 0, 0, 0.0017, 0.04, 0.09), (0, 0, 0, 0.0018, 0.06, 0.14) and (0, 0.08, 0.4, 0.7) at pressures 0.25 N/mm^2, 0.5 N/mm^2, 0.75 N/mm^2, 1 N/mm^2, 1.25 $N/mm^2$and 1.5 N/mm^2.

REFERENCES

[1] YU Ying, Zhu Qing-nan, YU Xiao-Chun, LI Yong-Sheng , *Novel Discrete Particle Swarm Optimization Based on Huge Value Penalty for Solving Engineering Problem,* Chinese Journal of Mechanical Engineering, 410 -418, Vol. 22, No. 3, 2009

[2] Zhang Yong, Luen Chow Chan, Wang Chunguang, and Wu Pei, Optimization for Loading Paths of Tube Hydroforming Using a Hybrid Method, journal of Materials and Manufacturing Processes, 700-708, Volume 24, 2009 - Issue 6

[3] F. Capece Minutolo, M. Durante, A. Formisano, and A. Langella, Optimization of a ydroforming process to realize asymmetrical aeronautical components by FE analysis,

[4] F. Djavanroodi and M. Gheisary, “Fracture and spring back on Double Bulge Tube Hydro - Forming ". American Journal of Applied Sciences 5 (8): 1041-1046. [2008].

[5] R. Di Lorenzo, G. Ingarao, F. Gagliardi and L. Filice, Experimental validation of optimisation strategies in hydroforming of T-shaped tubes, International Journal of Material Forming, April 2008, Volume 1, Supplement 1, pp 323–326

[6] C.T. Kwan and F.C. Lin, Influence of Process Parameters on T-Shape Tube HydroForming Characteristics for Magnesium Alloy, Key Engineering Materials, (Volumes 364-366) , 973-979, 2008.