International Journal of Analytical, Experimental and Finite Element Analysis (IJAEFEA), Issue. 3, Vol. 2, Sept 2015.
e-ISSN: 2394-5141, p-ISSN: 2394-5133, pp 101-103

Computation of Hydraulic Force Required to Reduce the Bounce Back Effect of Material in Process of Pipe Bending Using FEA

Mahesh M. Dadwe
M.Tech Scholar
maheshdadwe99@gmail.com

Vivek Padole
Assistant Professor

Department of Mechanical Engineering,
Abha Gaikwad Patil College of Engineering, Nagpur, India.

***Abstract* - In this paper is presented a finite element model for simulation of pipe Bending Process. The finite element model, used commercial code ANSYS. Also this paper is present some results for simulation of different bending distance in one direction of the bending. Bending of metal pipes is very important production method considering that metal pipe are widely used in a great variety of engineering products, such as automobile, aircraft, air conditioner, air compressor, exhaust systems, fluid lines.**

The results show that the amount of hydraulic force required bending a pipe and how it varies in respect of amount of bend of pipe in one direction in distance mm. It is also noticeable that maximum stress developed in pipe after bending. Springback effect in the material is due to elastic and plastic properties of the material. In this project the hydraulic force is calculated to reduce the bounce back or springback effect of material.

Index terms - Finite element simulation, tubular parts, bending.

I. INTRODUCTION

One of the most troublesome problems that are facing the tube production industry is the defect like springbok and others. In the outside region, the tube is subjected to tensile stress and the wall becomes thinner, while in the inside region, the compressive stress causes a thickening of the wall. In the previous works, the authors have developed a finite element model for the simulation of the bending process. This finite element approach can be used to optimize the force applied. There are a few bibliographic sources that report the use of the finite element simulation for the study of the tube bending. The finite element model, developed in Inventor & ANSYS, has been used to calculate the actually force required to bend a pipe having some thickness which will not create or developed any defect in the pipe.

A. Pipe Bending Process

Pipe hydro forming is one of the most important manufacturing process in metal forming industries such as automobile, architectural structure building, etc. which is almost cumbersome. These operations are mainly carried out by means of hydraulic cylinder and press dies. These operations include deformation of metal pieces to the desired size and size by applying uniform hydraulic pressure or force, in which metal work pieces always working under variable load & contact condition with die, due to this, the uniform bending of the metal piece is hardly achieved result in failure of the metal piece. Therefore the project objective is to compute the hydraulic force desired for bending standard schedule pipes of different thickness. It is also helpful to optimize the structural mass of frame so that material saving and cost benefit will be considerable.

II. PRINCIPLE OF FINITE ELEMENT ANALYSIS

FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions which will receive large amounts of stress usually have a higher node density than those which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements.

A. 3D Modeling

The simplified model of the pipe bending machine along with its two die & Punch is generated with the help of Autodesk Inventor 2012 software. The pipe of 60.3mm dia, 1000mm in length & 3.41mm in thickness is also generated for finding out the force required to bend it as a specimen object.

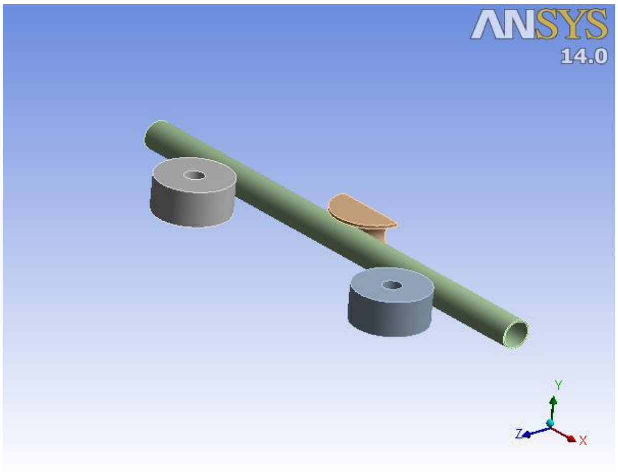

Figure 1. 3D Modeling of pipe & die arrangement

B. 3D Meshing

As Ansys 14.0 version is supported importing of different software model, the simplified model of the pipe bending machine is imported in Ansys 14.0. After importing the model the material conditions have been defined in the Ansys material library, Structural Steel with high stiffness valves is assigned to pipe die & Punch pie of the simplified model & Structural Steel NL i.e. Non Linear material properties assigned to the pipe.

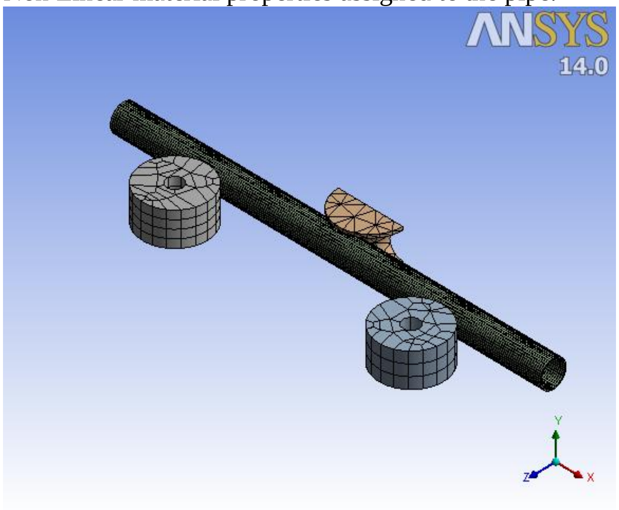

Figure 2. Finite Element Meshed Model Pipe Bending process arrangement

C. Preparing the 3-D Basic Model Simulation

For computing the simulation of the pipe bending we need to specify the contact condition between the each of the 3d object. The assigned contact condition drives the behavior of the simulation, how the modeled parts will behavior to assigned conditions.

TABLE I
ASSIGNED CONTACT CONDITION

Contact Conditions	Frictional - 2inch-Pipe:1 To Pipe Dia:1	Frictional - 2inch-Pipe:1 To Pipe Dia:2	Frictional - 2inch-Pipe:1 To Pipe Die:1

From the above table it is clearly seen that for all the bodies, frictional contact condition has been defined.

Now, the meshing has been done of the simplified model with medium sized mesh, the number of nodes & elements generated are 73493 & 11614 respectively. The mapped meshing function is used to create the mesh for the pipe.

Transient Structural analysis environment is selected for the final analysis from the Ansys work space. As our aim & objective of the project is to calculate the force required to bend the pipe & reduce the spring back effect, this type of simulation can been done in transient structural analysis because it can give the quick results with respect to bending. To find out the unknown force valve to bend the pipe we set the displacement of the punch dia to 50mm in z direction, that means the punch die will travel a distance of 50mm & the material properties of the pipe will try to resist the displacement because of the yield strength of the material. Ansys will run the simulation & give the force valve in force convergence graph shown in below graph. After running the simulation for more than 5 hours we get the analysis results described in the result analysis chapter.

I. RESULT AND DISCUSSION

A. Hydraulic Force

Due to hydraulic force created by the machine, the bending is developed in the pipe. In bending of pipe the tensile stress is developed in lower region and compressive are developed in the upper region. The maximum stress developed in the pipe is calculated from simulation done in ANSYS software. The graph shown in figure gives the idea of stress induced.

As per title, the hydraulic force required to bend the pipe for a particular distance is required to be calculated. The below graph showing the maximum force given by the bending machine to bend the pipe 40 mm displacement in one direction.

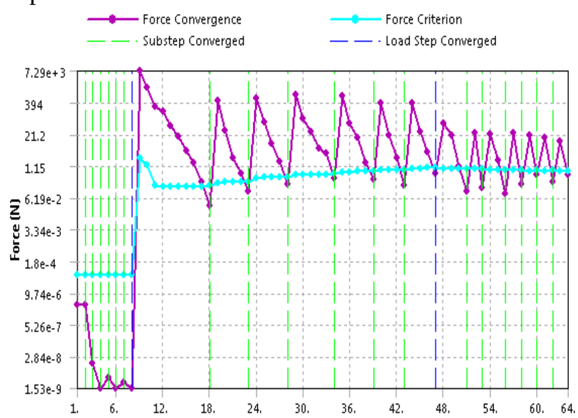

Figure3: Variation of hydraulic force for bending of pipe to 40 mm displacement

B. Maximum Stress

The figure is showing how the stress developed in the pipe in bending condition. The maximum stress induced in the material of pipe is 12.399 MPa for 40 mm displacement of pipe in one direction.

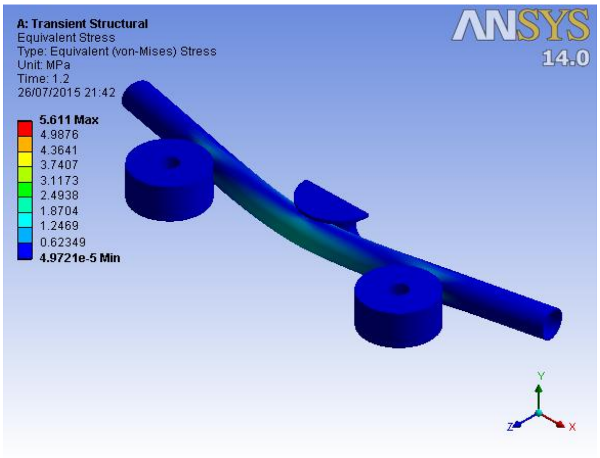

Figure 4. Maximum & Minimum Von Mises stress in the pipe

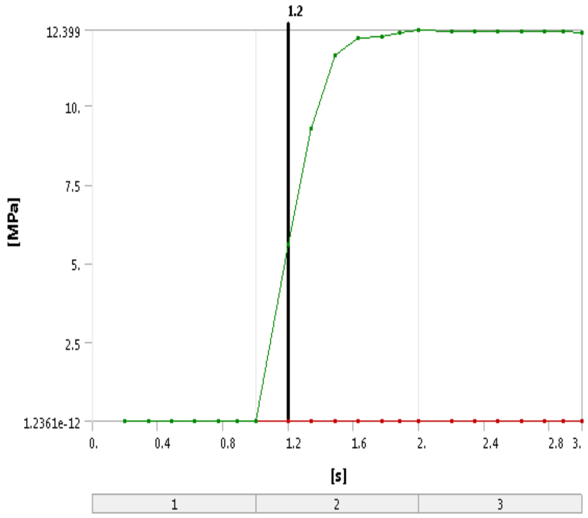

Figure 5 : Equivalent Stress

C. Directional Deformation

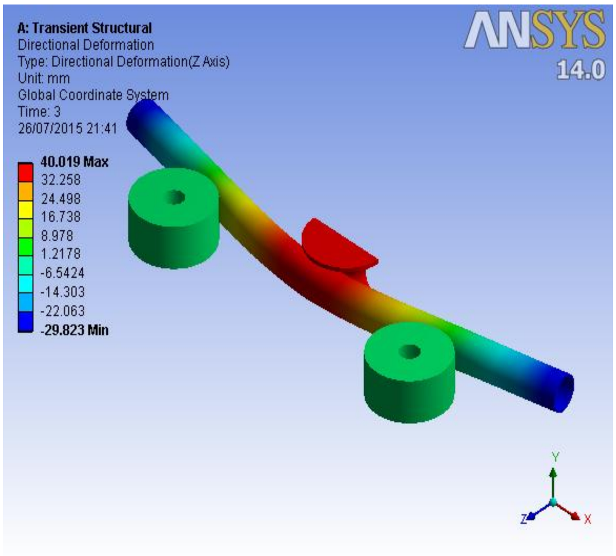

Figure 6. Maximum & Minimum Deformation of pipe

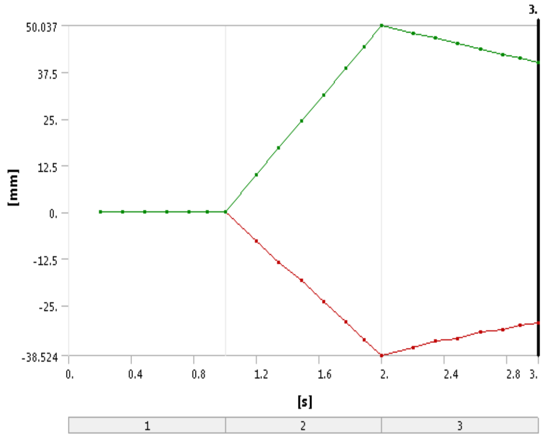

Figure 7. Directional Deformation

REFERENCES

[1] A.Bhattacharya and Daniel Long, "A FEA based investigation on Stress Intensification and flexibility factor for pie bends within and outside the limitations of ASME B31 piping code", CB&I UK Ltd, London.

[2] Peter Gantner, Herbert Bauer, David K. Harrison, Anjali K. M. De Silva, "FEA - Simulation of Bending Processes with LS-DYNA", 8th International LS-Dyna Users Conference, pp 2-33 to 2-40.

[3] Sutasn Thipprakmas (2010). Finite Element Analysis on V-Die Bending Process, Finite Element Analysis, David Moratal (Ed.), ISBN: 978-953-307-123-7, InTech Europe, 2010.

[4] Ivan Ivanovitsch Thesi Riagusoff, Paulo Pedro Kenedi, Luís Felipe Guimarães de Souza, Pedro Manuel Calas Lopes Pacheco, "Modelling Pipe Cold Bending A Finite Element Approach" – Ansy South American Conference, October 2010.

[5] Esther T. Akinlabi , Kagisho Matlou and Stephen A. Akinlabi , "Characterising the Effect of Springback on Mechanically Formed Steel Plates", World Congress on Engineering 2013, Vol. I, July 2013

[6] ZHAO Gang-yao, "Cross-sectional distortion behaviors of thin-walled rectangular tube in rotary-draw bending process", College of Materials Science and Engineering, State Key Laboratory of Solidification Processing, Northwestern Polytechnical University, Xi'an 710072, China.

[7] Nguyen Dang Van1, Namgung Ihn2 "An investigation of wall thinning and cross sectional geometry change bent tube of small diameters", Transactions of the Korean Nuclear Society Autumn Meeting Pyeongchang, Korea, October 30-31, 2014.

[8] Vasile Adrian Ceclan, Nicolae Bâlc, Alin Vasile Miron, Cristina Borzan1, Alexandru Popan, "Numerical simulation of the tube bending process and validation of the results", Academic Journal Of Manufacturing Engineering, Vol. 9, Issue 3/2011.

[9] Kunal Sharma, Dr. V.N Bartaria, "Design And Simulation Of Multi-Axis Pipe Bending System With Receiving Gauge", International Journal of Advanced Technology & Engineering Research (IJATER).

[10] Piotr Domanowski, Bartosz Nowak, "Numerical And Experimental Study On Aluminium Pipe Bending Process", University of Technology and Life Sciences in Bydgodszczal. Prof. S. Kaliskiego 7, 85-789 Bydgoszcz, Poland.