HYDROFORMING OF A STEEL TUBE

Greg Wilmes
University of Massachusetts

ABSTRACT

The focus of this project is to create a finite element model to accurately show the deformation of a thin steel tube into a square corner by simulating a hydro-forming process. To achieve this goal, this paper will discuss the use of the creation of the model to simulate this deformation in Ansys 5.7. This project uses 2-dimensional elements and non-linear material properties to model a three dimensional tubular hydro-forming problem. It further investigate the use of load steps and solution controls to create a quasi-static model.

INTRODUCTION AND OBJECTIVE

Hydro-forming is a manufacturing process which deforms thin metal “blanks” in complex shapes using incompressible fluids. Hydro-forming’s ability to create complex shapes and high, repeatable tolerances have made this process attractive, however, until recently the cycle time of the process was slow. About ten years ago, the automotive world started to embrace this process due to the superior consistency of mechanical properties and were use in higher-end, lower production vehicles. This lead to shorter cycle times and therefore the process moved into higher production vehicles. Today, hydro-forming is widely used in the automotive world for applications like engine chassis, automotive frames, exhaust systems, hoods, roofs, instrument panel supports.

The concept of hydro-forming is relatively simple; it involves the use of an incompressible fluid to form complex shapes out of thin metal “blanks.” There are two version of hydro-forming, tubular and sheet, which describes the types of blanks used. Tubular hydro-forming starts with a thin-walled tube and uses pressure to expand the bank into a more complex shape. Figure 1 shows the six steps of the hydro-forming process:

A. Placement of the blank,
B. Close the shape die around the circumference
C. Fill interior with liquid
D. Close the axial portion of the die
E. Compress the part
F. Evacuate the liquid and remove the part.

The objective of this project is to use the finite element method to accurately model this process and to use this model to transform a round tube into a square tube.

Figure 1. Tubular Hydroforming Primer [1]: Show the 6 basic steps to tubular hydroforming.
MODEL CREATION

Although the concepts of hydro-forming are simple, actual applications are very complex. The Engineering Research Lab for Net Shape Manufacturing at Ohio State University has identified 4 main areas of design concerns, as seen in Figure 2, which need to be addressed in any model. This project focuses on the processing and material behavior and the main concerns in designing the model are the non-linear, anisotropic material behavior of steel, high deformation, friction, complicated 3-D geometry.

Pressure

Figure 3: Simplified model Geometry

Hydro-forming is used to create complex 3-D shapes, the scope of this term project allows for a 2-dimensional model of the hydro-forming process to be created representing the cross-section of a tube. This cross-section is also symmetric, the model will capitalize on this to reduce elements to increase processing speed, Figure 3 shows the basic model. A thickness of 1.6mm was selected because that is the thickness of a standard square steel tube.

Modeling material properties became the next focus of the model. Ansys allows both linear and non-linear material behavior. This model uses both the linear elastic material properties and then the kinematic hardening method of non-linear properties. Kinematic hardening requires the input of a table of 20 stress and strain points from the plastic portion of a stress-strain curve, Figure 4 shows the curve that was used. This curve was created from experimental tensile data. The model further assumes that the material will behave isotropically.

Element selection was relatively simple, due to the nature of the model and the assumptions, the Plane42 element was selected. This element is a 4 noded, 2-dimensional element which handles non-linear behavior. This element has options which were looked at:

- Plane-Stress Option - Due to the thin nature of the deforming area, normal stress cannot be supported. This is a plane stress condition; however the model was run both ways and this option did not have a noticeable effect
- Local Coordinate System - Due to the curved area this option was used. This sets a local coordinate system for each element.
- Extra Shape Functions - Due to the high deformation of this model, the extra shape function make sure that the elements match.

Figure 4: Experimental stress-strain curve used to model plastic deformation
MESHING AND CONTACT ELEMENTS

After the geometry was drawn as shown, the tube was meshed using mapped meshing. To use this mesh type with the quarter circle geometry, the mesh must be defined in angular divisions and transverse slices, (fig. 5) The area which acts as the die was not meshed due to the requirement it act as a rigid body during contact.

Using the Contact Wizard, the contact constraints were set such that the model had the following options: a rigid target, no separation, sliding allowed, Coulomb friction, and a deformable contact.

SOLUTION CONTROLS

After fully constraining the model, the stresses were added using a series of load steps, set to auto time-step. The first set of load steps was created to increase step the pressure from 0 - 380 MPa. This was achieved using a “do” loop to increase the pressure in 180 steps of 2MPa. Not only did this method slowly increase the applied pressure, it also provided for “bursting” of the tube. If the pressure got too large the model no longer worked.

The second set of load steps was used to emulate a constant pressure of 380MPa over time. Each of these load steps was solved as a static problem, converged and then the next load was applied to form a quasi-static solution of hydroforming.

Figure 5: View of the mapped mesh of the model

Figure 6: Deflection of tube partially through process, showing original shape

Figure 7: Maximum Displacement of the finite element model
FINDINGS, CONCERNS AND ANALYSIS

The model was run several with several meshes, concentrating on maximum combined deflection. Stress state was generally ignored, because of the functionally of the load steps to burst the model if stresses got too high.

Figure 7 shows the maximum displacement of the model. This displacement shows the distance into the corner that the model. As can be seen if the figure, results of between 90 and 1440 nodes produced a maximum displacement of about 12.5mm. The difference between solutions with 90 elements and 1440 elements was 0.032mm or about 0.3%. Since the 90 element model runs significantly faster, this difference seems acceptable. Furthermore this difference is well within common dimensioning tolerances for sheet metal parts.

The model produced two main concerns which need to be investigated further and the model needs to be refined to account for them. Both of these concerns are related, there seemed to be no plastic deformation of the part, although a pressure of 380MPa is much higher than the yield hoop stress. This paper will assumes that this is do to the load steps method applied. Each load step increased to the start of plastic deformation, then the next load step would be applied.

FUTURE WORK

• Expand the model to 3-Dimensions
• Work with different material models
• Deform the parts more complex shapes
• Experiment with different pressures, loads and thicknesses of parts
• Look deeper into the effect of the number of loops in the second set of load steps to see it effect.
• Analyze the stress state of each model.

ACKNOWLEDGMENTS

Mei Yu for patience in helping me to solve my contact element problems and showing me how to use solution controls to solve my project.

Dr. Grosse for giving me hints and suggestions about how to get around roadblocks and telling me to turn around several times.

REFERENCES
