UNIVERSITI PUTRA MALAYSIA

FINITE ELEMENT ANALYSIS OF HYDRAULIC BULGING OF METAL TUBES

SALEEM NA'MI

FINITE ELEMENT ANALYSIS OF HYDRAULIC BULGING OF METAL TUBES

By

SALEEM NA'MI

Thesis Submitted to the School of Graduate Studies, Universiti Putra Malaysia, in Fulfilment of the Requirement for the Degree of Master of Science

November 2006

TO THE MEMORY OF MY FIRST TEACHERS,

MY MOTHER AND FATHER

Abstract of thesis presented to the Senate of Universiti Putra Malaysia in fulfilment of the requirement for the degree of Master of Science

FINITE ELEMENT ANALYSIS OF HYDRAULIC BULGING OF METAL TUBES

By

SALEEM NA'MI

November 2006

Chairman: **Professor Abdel Magid S. Hamouda, PhD**

Faculty: **Engineering**

The use of finite element simulation in the development of bulge forming procedures is becoming more important as it provides a cheap and efficient way to determine important process parameters. Further improvements to the bulge forming method will be realized with the use of simulation and the design of advanced tooling. One major area where tube hydroforming is applied is in automotive structures.

In this current research, numerical analysis was conducted using the explicit finite element code ANSYS 2D. One-fourth model of the whole geometry consisting of the tube and the dies was adopted in consideration for symmetric property of the tube deformation. The model of metal tube is assumed as a bilinear isotropic model approximating the characteristics of annealed mild steel was adopted as the material for the tube. The final model contains 630 nodes and 560 elements included 19 contact elements between die and tube. The interface between the die and the tube was modeled using an automatic node to surface contact.

The effects of friction, geometric parameters; tube thickness , tube length , die corner radius and diameter of bulge width and varying internal pressures only and with axial load were evaluated in bulge hydroforming. Furthermore, two types of metal were tested, namely, mild steel tube and copper.

Numerical results were verified with available experimental values obtained from the literature were carried out and the percentage error is about 4.5%.

Finite element analysis showed that for a particular amount of wall thinning there is an increase of around 5.23% in bulge height for combined internal pressure with axial force. Results of this study indicate that tube hydroforming with combined internal pressure with axial force can increase expansion with less von Mises stress (8.32%) i.e. more difficult parts can be designed and manufactured. Furthermore, the minimum (optimum) friction coefficient displayed the highest value of bulging with the lowest value of decreasing of wall thickness was recorded for friction coefficient (μ) 0.15. Also the tube bulging increases with increasing die corner radius and bulge width. It also decreases with increasing initial tube length and initial thickness of the tube material. All these parameters are crucial to the success of the hydroforming operation.

Abstrak tesis yang dikemukakan kepada Senat Universiti Putra Malaysia sebagai memenuhi keperluan untuk ijazah Master Sains

ANALISIS UNSUR TERHINGGA UNTUK PEMBONJOLAN HIDRAULIK BAGI TIUB LOGAM

Oleh

SALEEM NA'MI

November 2006

Pengerusi: Profesor Abdel Magid S. Hamouda, PhD

Fakulti: Kejuruteraan

Penggunaan simulasi unsur terhingga dalam pembangunan prosedur-prosedur pembentukan bonjol telah menjadi semakin penting kerana simulasi ini memberikan satu cara yang murah lagi cekap untuk menentukan parameter-parameter proses yang penting. Peningkatan selanjutnya dalam kaedah pembentukan bonjol akan direalisasikan dengan penggunaan simulasi yang cekap dan reka bentuk alat yang lebih maju. Satu bidang utama di mana pembentukan tiub secara hidro digunakan adalah dalam pembuatan struktur-struktur automotif.

Dalam kajian terkini ini, analisis kaedah berangka telah dijalankan dengan menggunakan kod tak tersirat elemen unsur terhingga iaitu ANSYS 2D. Satu per empat model daripada keseluruhan geometri mengandungi tiub dan acuannya telah diubah dengan mengambil kira sifat simetri bagi pembentukan tiub. Model tiub keluli ini dianggap sebagai satu model isotropi bilinear yang menyerupai sifat keluli lembut sepuh lindap yang telah dipilih sebagai bahan bagi tiub ini. Model terakhir mengandungi 630 nod and 560 unsur termasuk 19 unsur yang bersentuhan di antara

acuan dan tiub. Permukaan di antara acuan dan tiub telah dimodelkan dengan menggunakan nod automatik kepada sentuhan permukaan.

Kesan daripada geseran, parameter-parameter geometri, ketebalan tiub, panjang tiub, jejari bucu acuan dan diameter bagi lebar bonjol serta tekanan dalaman yang berlainan dan dengan paksi beban telah diuji dalam pembonjolan hidraulik. Selain itu, kajian perbandingan antara sifat-sifat bahan bagi tiub keluli lembut dan kuprum juga telah dilaksanakan.

Keputusan berangka yang diperolehi telah dibandingkan dengan hasil keputusan eksperimen dari kajian ilmiah yang lain. Keputusan menunjukkan peratusan ralat adalah sebanyak 4.5%.

Analisis unsur terhingga menunjukkan peningkatan ketinggian bonjolan sebanyak 8% bagi amaun penipisan dinding tiub yang tertentu untuk kombinasi di antara tekanan dalam tiub bersama daya paksi. Keputusan kajian ini menunjukkan pembentukan tiub secara hidro dengan kombinasi di antara tekanan dalam tiub bersama daya paksi akan meningkatkan amaun pengembangan dengan tekanan yang lebih rendah pada tiub tersebut. Oleh itu bahagian produk yang lebih kompleks dapat direka dan dihasilkan. Tambahan lagi, koefisien geseran minimum (optimum) yang menunjukkan ganjakan yang tertinggi dengan nilai terendah bagi pengurangan ketebalan dinding telah tercatat pada koefisien geseran (μ) 0.15. Ganjakan tiub juga bertambah dengan pertambahan jejari penjuru acuan dan lebar ganjak. Ia juga berkurangan dengan pertambahan panjang tiub awal dan ketebalan awal bahan tiub. Kesemua parameter berkenaan adalah amat genting dalam menjayakan operasi hidro digunakan

ACKNOWLEDGEMENTS

All praise to supreme almighty Allah s.w.t. the only creator, cherisher, sustainer and efficient assembler of the world and galaxies whose blessings and kindness have enabled me to accomplish this project successfully.

Gratefully acknowledge the guidance, advice, support and encouragement received from my supervisor, Professor Dr. Abdel Magid S. Hamouda, who keeps advising and commenting throughout this project until it turns to real success.

I am also grateful to my supervisory committee, Associate Professor Ir. Dr. Mohd Sapuan Salit, for their advice and helpful discussion during this period of study.

I am also grateful to Professor Dr. A. kostanjevec Senior Researcher, University of Ljubljana, Slovenia and Associate Professor Dr. I. Mupende, Technical University Clausthal, Germany, I wish to express my sincere thanks for their valuable remarks, assistance and advice me towards a successful completion of this research.

Appreciation also to all the staff in Department of Mechanical and Manufacturing Engineering, Universiti Putra Malaysia for providing the facilities and the components required for undertaking this project.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

D_o, d_o, D	Outer tube diameter
d_i	Inner tube diameter
D_p	Protrusion diameter
E	Young modulus
F, F_a, F_r	Axial force (Load)
F_t	Total axial force
F_f	Axial friction force
k	Shear strength
K_R	Ratio of the axial force to the internal pressure
K	Strength coefficient
l_f	The free tube length
L, x	Length of tube
μ	Coefficient of friction
m	Friction factor
n	Strain hardening exponent
P_i	Internal pressure
P_o	Outside pressure
$(P_i)_{max}$	Calibrating pressure
$(P_i)_b$	Bursting pressure
$(P_i)_y$	Yielding pressure
p_{oc}	Pressure under constrained area
R_b, r_b	Smallest die corner radius

t_i	Finial tube wall thickness
t_o, t	Initial tube wall thickness
T_p	Tangent modulus
W_b	Bulge width diameter
ν	Poisson's ratio
x_o	Punches enter a length
σ_k	Buckling stress
σ_f	Flow stress of the material
σ_u	Ultimate tensile strength of the material
σ_y , S	Yield strength of the material
σ_θ	Axial stress
σ	Von Mises Stress
ε_t	Plane strain
ρ	Density

CHAPTER I

INTRODUCTION

1.1 Introduction

Hydroforming is one kind of plasticity working or metal forming process, which can be used to increase the diameter, to change the geometry, or to expand the outer walls of a cylindrical shell or tube. This process is very important, especially for industrial products of light weight and high strength.

In the past several years tube hydroforming technology has proved itself as a vital metal forming process for manufacturing variety tubular parts, e.g. household piping components, fittings, complex automotive parts such as exhaust pipes and structural components.

The rapid growth of this technology has been due to the advantages Tube hydroforming (THF) offers compared to conventional manufacturing via stamping and welding, namely (a) part consolidation; (b) weight reduction through more efficient section design and tailoring of the wall thickness in structural components;(c) improved structural strength and stiffness via optimized section geometry; (d) lower tooling costs due to fewer parts; (e)fewer secondary operations (less welding and punching of holes during hydroforming); (f) tighter tolerances and reduced spring-back that facilitates assembly and (g) reduced scrap since trimming of excess material is far less in THF than in stamping . The development of this process for automotive industries is relative new and many process variables have been studied, like: friction, material properties, pressures and displacement path during the process. The simulation is a very important method to help to develop this process. Using finite element method many researches have been studying the influence of these variables in the process and they are applying forming limit expressions to define whether the material will resist to the deformation or not. Successful tube hydroforming process requires proper combination of part design, material selection, and application of internal pressure and axial feeding. By using Finite Element Analysis methods, process parameters can be determined before manufacturing the dies and starting die try-out and process development.

The development of hydroformed parts for series production necessitate efficient methods to meet the requirements of short development times, high part quality with an optimized process chain. An important factor in achieving these short development lead times is process simulation exploiting the potential of finite element analysis (FEA).

1.2 Methods

The bulge forming of tubular components is accomplished by the application of hydrostatic pressure to tube blanks either in free expansion mode or using a die bearing the shape of the component to be formed. The pressure is transmitted via a medium such as a liquid (e.g. hydraulic fluid or water), an elastomer (e.g. rubber or polyurethane), or a soft metal (e.g. lead or a lead alloy). Bulge forming using pure internal pressure has a major limitation for producing excessive thinning of the tube wall which lead to the rupture of the tube for only moderate expansions. However, if a compressive axial load is applied to the ends of the tube simultaneously with the internal pressure, metal can be fed into the deformation zone during forming enabling more expansion and less thinning.

In tube hydroforming THF, compressive stresses occur in regions where the tube material is axially fed, and tensile stresses occur in expansion regions. The main failure modes are buckling, wrinkling (excessively high compressive stress) and bursting (excessively high tensile stress). It is clear that only an appropriate relationship between internal pressure and axial load, guarantees a successful THF process without any of the failures. Hence, it is imperative to establish a systematic way for determining loading paths and using finite element analysis (FEA) is one possible way. Internal pressure and axial load are then applied simultaneously to form the tube to fill the die cavity. To obtain a successful part, coordination of the pressurization and axial feeding is required. High internal pressure without end feeding will result in bursting of tube blank. On the other hand, large end feeding with insufficient internal pressure will lead to the development of wrinkles on the tube wall.

Effective classifications of hydroformed tubular parts are necessary for development of THF part design and process systematically. Finite element analysis (FEA) simulations can be used as a tool to extensively analyze THF. Design of the process parameters are normally selected through time-consuming, trial-and-error iterative FEA simulations.

FEA simulation enhanced with optimization schemes can greatly reduce the lead-time spent in the process development.

1.3 Benefit and Limitations

Tube hydroforming has been identified as a new technology to manufacture parts. Tube hydroforming has many advantages in comparison with conventional manufacturing via stamping and welding. It can reduce the weight of the component, retain and even improve the strength and stiffness, reduce tooling cost due to fewer parts and tube hydroforming requires fewer secondary operations.

With the aid of FEA simulation, the part quality control, and the design of the tube hydroforming process can be easily implemented and monitored. FEA simulations provide insights on the necessary process parameters internal pressure and axial load, part geometry, and part formability by analyzing the thinning, thickening, and strain distribution in the deformed tube.

In all metal forming processes, part and process design is an essential step in successful manufacturing of any products. Tube hydroforming (THF) process demands a lot of engineering knowledge starting from the part design which is constrained by part functionality and geometry, to the process design where appropriate combination of internal pressure and axial feed need to be determined. It has always been of a primary concern in the industry to reduce the lead time in part and process design developments