Experimental Study and Numerical Simulation of Sheet Metal Bending Process

A Project Report

Submitted by

S. Venkat Prasat - 71204402007

in partial fulfillment for the award of the degree
of

Master of Engineering
in
CAD/CAM

DEPARTMENT OF MECHANICAL ENGINEERING
KUMARAGURU COLLEGE OF TECHNOLOGY
COIMBATORE – 641 006

ANNA UNIVERSITY :: CHENNAI 600 025

APRIL – 2006

ANNA UNIVERSITY :: CHENNAI 600 025

BONAFIDE CERTIFICATE

Certified that this project report entitled **"Experimental Study and Numerical Simulation of Sheet Metal Bending Process"** is the bonafide work of

Mr. S. Venkat Prasat - Register No. 71204402007

Who carried out the project work under my supervision.

(Signature of the Head of the Department)
Dr. T. P. MANI
HEAD OF THE DEPARTMENT

(Signature of the supervisor)
Mr. S. KALIDASS
LECTURER

29/06/06
Internal Examiner

Dr. T.P. Mani
B.E., M.E., Ph.D., DML., MIE., MNIQR., MISTE.,
Dean & HoD / Dept. of Mech. Engg.
Kumaraguru College of Technology
Coimbatore - 641 006

Dr. Rajendran
29/6/06
External Examiner

DEPARTMENT OF MECHANICAL ENGINEERING

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE 641 006

ANNARI AMMAN
STITUTE OF TECHNOLOGY
Stay Ahead

TÜV CERT
EN ISO 9001
Certificate : 0410021309
RWTÜV

National Conference on

Optimization Techniques in Engineering Sciences and Technologies

OPTEST - 2006

held at

BANNARI AMMAN INSTITUTE OF TECHNOLOGY
Sathyamangalam-638 401 during
April 11-12, 2006

Certificate

This is to certify that Mr./~~Ms.~~/~~Mrs~~ ....K. S. VENKATA PRASAD....

has ~~participated~~ / presented a paper entitled ....NUMERICAL SIMULATION AND ANALYSIS OF MECHANICS OF SHEET METAL BENDING PROCESS....

in the **National Conference on "Optimization Techniques in Engineering Sciences and Technologies (OPTEST-2006)"** during **11-12 April 2006**, organized by the **Department of Mechanical Engineering, Bannari Amman Institute of Technology**, Sathyamangalam.

C SASIKUMAR/ G SASIKUMAR
Organizing Secretaries

Dr K THIRUNAVUKKARASU
Convener

Dr A SHANMUGAM
Chairman

ABSTRACT

Sheet metal bending is a manufacturing process in which initially flat sheets are permanently deformed into intricate shapes by the use of plate rolls or dies. The use of simulation software in sheet bending process has increased significantly in recent years as the benefits of troubleshooting and optimizing processes on the computer rather than through extensive shop trials have been realized. The simulation of the springback in sheet metal bending process requires the application of advanced tools. Springback or elastic recovery relates to change in shape between the fully loaded and unloaded configuration the material encounters during operation. Springback causes changes in shape and dimensions that creates major problems in the assembly. Thus accurate springback prediction is imperative for robust design of tooling; thereby saving costs and die try-out times and increasing productivity.

In addition to experimental methods, numerical simulation is needed to visualize the springback in sheet metal bending process. Therefore the objective of this project is to present the Experimental measurement and Numerical simulation of the Springback in sheet metal bending and their comparison. A test matrix has been developed based on the orthogonal array of Taguchi design of experiments (DOE) approach. Experiments have been conducted for the V - bending process using 6022-T4 Aluminium alloy to study the variation of springback due to both process and material parameters such as bend radius, sheet thickness, grain size, punching speed and time. The design of experiments has been used to evaluate the predominate parameters for a specific lot of sheet metal. It is observed that bend radius had greatest effect on springback. Next the finite element simulation of springback in V - bending process using ANSYS LS-DYNA is studied. A 2-D finite element modeling is considered in the springback simulations. A nonlinear isotropic material model is used. The experimental results compare well and are in good agreement with the simulated results for the effect of bend radius on springback.

ஆய்வுச் சுருக்கம்

உலோகத்தகட்டை வளைத்தல் என்ற உற்பத்தி செய்முறையால் வார்ப்பை பயன்படுத்தி ஒரு சமதளமான உலோகத்தகட்டை பல நிரந்தரமான வடிவங்களில் மாற்றி அமைக்கலாம். உலோகத் தகட்டை வளைத்து மடிப்புகள் ஏற்படுத்தும் போது ஏற்படும் எதிர் விளைவுகளை கணிப்பதற்கு நவீன கருவிகள் தேவைப்படுகிறது. எதிர் விளைவு என்பது தகட்டை சுமை அழுத்தும் போது உள்ள வடிவத்திற்கும் சுமை அகற்றிய பின் உள்ள வடிவத்திற்கும் உள்ள வேறு பாட்டை குறிக்கிறது. உலோகத் தகட்டின் எதிர்விளைவினால் வடிவங்கள், அளவுகள் மாறுவதால் பாகங்களை பொருத்துகையில் பல விதமான இன்னல்கள் ஏற்படுகிறது. துள்ளிய வார்ப்பு வடிவமைப்பிற்கும், மிகக் குறைந்தபட்ச செலவிற்கும், மற்றும் உற்பத்தி பெருக்கத்திற்கும் அதிநுட்ப எதிர்விளைவின் கண்ககீடு அவசியமாக தேவைப்படுகிறது.

எதிர் விளைவுகளை அளப்பதற்கு பரிசோதனை முறைகளுடன் எண் வழி சார்ந்த முறைகளும் தேவைப்படுகிறது. உலோகத் தகட்டை வளைத்தலில் ஏற்படும் எதிர் விளைவுகளின் கணக்கீட்டை எண் சார்ந்த முறை மற்றும், பரிசோதனை முறையில் சமர்ப்பிப்பதே இந்த ஆய்வின் நோக்கமாகும். Taguchi முறையின் ஆர்த்த கோணல் நிறைநிரல் அடிப்படையில் சோதனை தேற்றம் உருவாக்கப்பட்டது. 6022-T4 அலுமினிய உலோகக் கலவையைக் கொண்டு V-வடிவ வளைத்தல் செய்முறையில் அப்பொருளின் வழி அலகுகள் மற்றும் செய்முறை அலகுகளாகிய வளை ஆரம், தகடு தடிமன், தூள் அளவு, அழுத்த வேகம், நேரம் ஆகியவற்றை மாற்றி அமைத்து எதிர் விளைவுகளை கணக்கிடப்பட்டது. ANSYS LS-DYNA முறையில் மாறுபட்ட வளையாரத்திற்கு எதிர்விளைவின் கோணங்கள் கணக்கிடப்பட்டது. 2D தனிம உருப்படிவம் எதிர் விளைவுகளை கண்டறிய பயன் படுத்தப்பட்டது. பரிசோதனையின் முடிவும் ANSYS LS-DYNA வில் கண்டறிந்த முடிவும் நல்ல ஒப்பிகையானது.

ACKNOWLEDGEMENT

The author expresses his gratitude to his guide **Mr. S. Kalidass,** Lecturer, Department of Mechanical Engineering, Kumaraguru College of Technology for his untiring support, timely suggestion, constant encouragement, enthusiasm and inspiration throughout the study and for completing this project successfully.

The author wishes to express his deep sense of reverential gratitude and sincere thanks to **Dr. T. P. Mani,** Head of the Department, Department of Mechanical Engineering, Kumaraguru College of Technology, for supporting him to do the project work smoothly.

The author wishes to express his deep sense of humble gratitude and sincere thanks to **Dr. K. K. Padmanabhan,** Principal, Kumaraguru College of Technology for providing facilities to carry out the project work.

The author wishes to express his deep sense of gratitude and sincere thanks to **Dr. N. Gunasekaran,** Professor, Department of Mechanical Engineering, Kumaraguru College of Technology for his excellent motivation and valuable advice.

The author wishes to thank **Mr. Sriharan** and **Mr. N. Sivakumar,** Technicians, CAD Laboratory for helping him in completing the project successfully.

The author indebts his thanks for the opportunity provided by the management of Dynamax Pvt. Ltd., Bangalore to carry out a part of his project work in their unit.

The author owes his sincere thanks to his parents, teachers and Almighty God who have bestowed upon him their generous blessings in all his endeavors.

CONTENTS

Title **Page No.**

LIST OF TABLES

LIST OF FIGURES

NOMENCLATURE

e = Strain

s = Flow Stress

E = Young's Modulus

k = Material Constant

n = Strain Hardening Exponent

$F(s_{ij})$ = Yield Function

s_1, s_2, s_3 = Principal Stresses

n_1, n_2, n_3 = Principal Directions

DOF = Degrees of Freedom

S = Sum of Squares

V = Variance between the individual Control Factor effects

V_e = Variance in Experimental data due to random experimental error

S' = Pure Sum of Squares

S_T = Sum of Squares for Total

CHAPTER 1

1.1 PROJECT OBJECTIVE

The objective of this project is to study and analyze the sheet metal forming or the stamping aspects of automotive manufacture, specifically the relative contribution of material and process parameters on the formability of lightweight auto-body materials. This project was motivated because the production of lightweight car exteriors with enhanced repeatable dimensional tolerances is important to the automotive industry. Many of these parts are formed using sheet metal forming. The use of aluminum alloy sheets in the manufacture of auto-body panels has increased fourfold in the automotive industry because of its high strength, low density and corrosion resistance. But one of the major concerns of stamping lightweight aluminum alloys is springback. Hence, to be cost effective, accurate predictions must be made of its formability. The automotive industry places rigid constraints on final shape and dimensional tolerances. Compensating for springback becomes critical in this highly automated environment.

1.2 IMPORTANCE OF THE PROJECT WORK

Springback or elastic recovery relates to the change in shape between the fully loaded and unloaded configurations the material encounters during a sheet metal forming operation. This results in the formed component being out of tolerance and can create major problems in the assembly or installation.

Accurate springback prediction is imperative for robust design of tooling; thereby saving costs and die try-out times. The effect of the various process and material parameters on spring-back and spring-in are examined in this research. A 90 degree V-Bend process was selected for this study. The sheet material used is 6022-T4 Aluminum Alloy.

1.3 PROJECT OVERVIEW

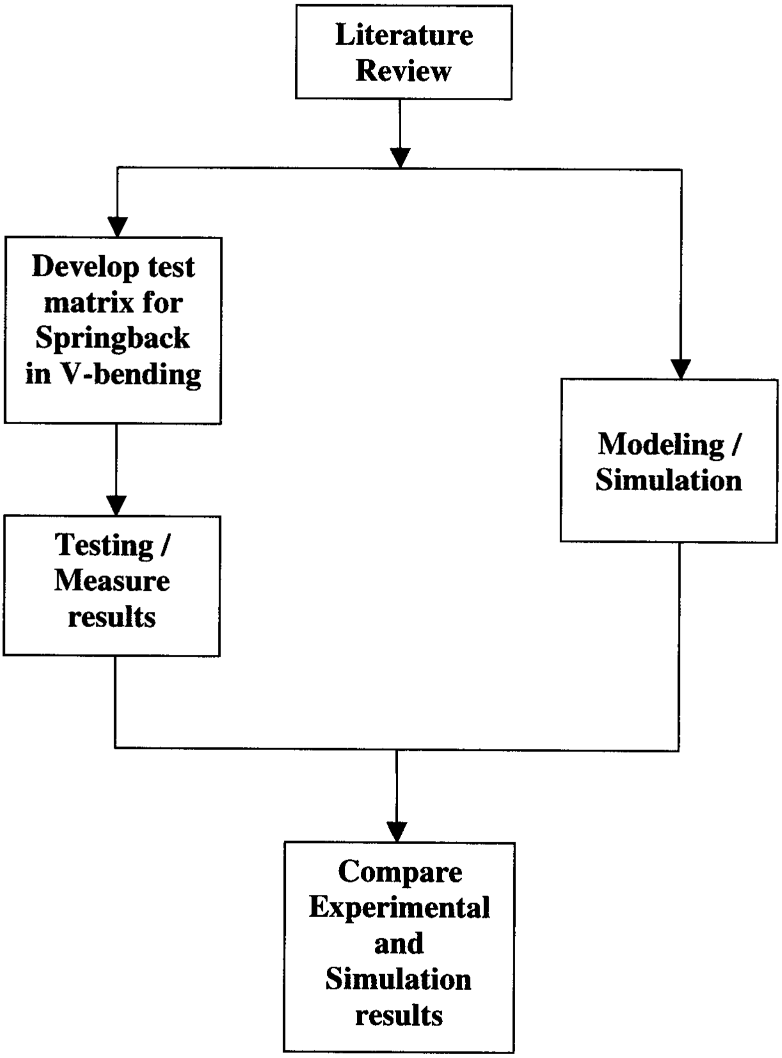


FIGURE 1.1 PROJECT OVERVIEW

This project work is comprised of two parts as shown in Figure 1.1. The first part involves the experimental measurement of springback, and the second part deals with springback simulation. Previous investigations have studied the effect of certain process or material parameters on springback. This study will evaluate processing parameters in addition to material parameters in relation to their overall contribution to springback.

In the first part, experiments were designed using the Taguchi design of experiments (DOE) methodology to include the various process and material parameters that effect springback as illustrated in Figure 1.2. V-bend fixtures were designed and fabricated and the test matrix was set up. Results were recorded and analyzed, and the relative significance of the various factors that have been reported to effect springback was determined. Based on the experimental results, appropriate material models were used to simulate the process.

The literature concerning factors affecting springback is split between processing parameters and material parameters. Once a relative ranking of process parameters and material parameters for a given lot of material is obtained, the process can be modeled using FE.

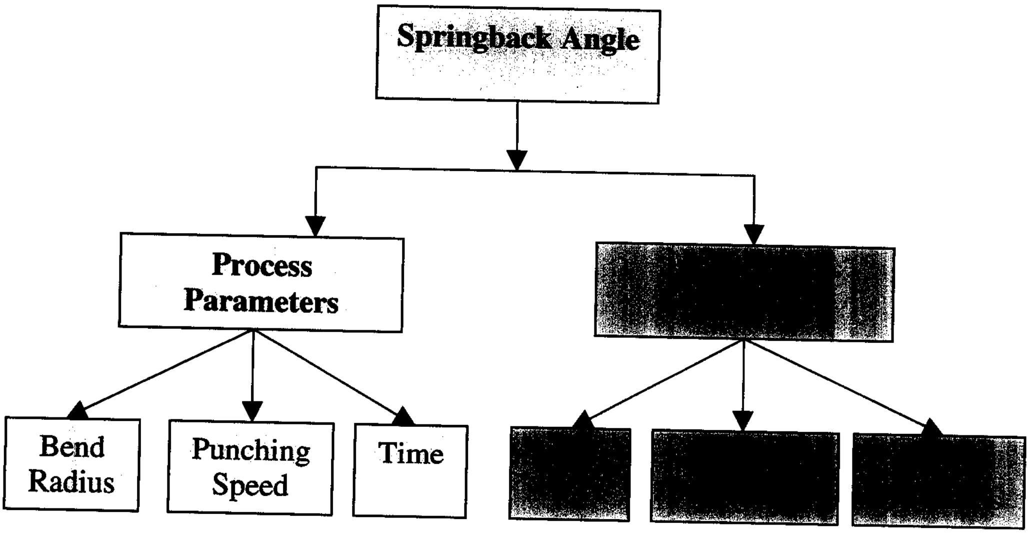


FIGURE 1.2 DESIGN PARAMETERS THAT EFFECT SPRINGBACK

CHAPTER 2

LITERATURE REVIEW

Cleveland et al. (2002), describes in his research paper about the Inelastic effects on springback in metals. The details of nonlinear recovery were studied for two different test materials which were 6022 T4 Aluminium and a high strength steel. A model based on physical mechanisms for strain recovery and compliance was proposed to describe both the tension-unloading and compression segments of deformation. Improved predictions of springback and resumption of reverse flow are possible using this model.

Yoon et al. (2002), in his research paper describes about the springback prediction for sheet metal forming process using a 3D hybrid membrane/shell method. To reduce the computational time of finite element analyses for sheet forming, a 3D hybrid membrane/shell method has been developed and applied to study the springback of anisotropic sheet metals. In the hybrid method, the bending strains and stresses were calculated as post-processing, considering the incremental change of the sheet geometry obtained from the membrane finite element analysis beforehand. The hybrid method was applied for a 2036-T4 aluminium alloy square blank formed into a cylindrical cup, in which stretching is dominant.

Kawka et al. (1998), this paper describes the simulation of multi-step sheet metal forming processes by a static explicit FEM code. Here two topics related to the FEM simulation of multi-step stamping processes are discussed. They are: (1) the difficulty in simulating the springback and trimming operations, and (2) the way to select a most appropriate element. Finally the entire simulation of multi-step forming processes of a car wheel disk is presented and compared with the experiment data.

Lee et al. (1998), has published a research paper on the assessment of numerical parameters influencing springback in explicit finite element analysis of sheet metal forming process. In this work, numerical factors influencing springback have been evaluated quantitatively using the Taguchi method. To clarify the effect of each factor, the U-draw bending process was chosen as an evaluation problem

Li et al. (2002), describes in his paper the simulation of springback. Here springback has been simulated with 2-D and 3-D finite element modeling. Simulations using solid and shell elements have been compared with draw-bend measurements presented in a companion paper. Plane-stress and plane-strain simulations revealed the dramatic role of numerical tolerances and procedures on the results.

Math et al. (2002), has described the finite element approach in the plate bending process in his technical paper. Here inorder to determine the inevitable mechanical springback of bent plates, designed for assembling spherical tanks, an elastic-plastic incremental finite element calculation has been carried out to analyze axisymmetric strain in sheet metal bending process.

Narasimhan et al. (1999), in his research paper published has described about predicting springback in sheet metal forming by an explicit to implicit sequential solution procedure. Using this simultaneous solution technique on an actual automotive component, numerically predicted springback deformations were found to be within 1% of production values.

Rojek et al. (1998), has stated about the application of explicit FE codes to simulation of sheet and bulk metal forming processes. An original formulation of a triangular shell element without rotational degrees of freedom is reviewed.

Zhang et al. (1997), in his paper named V-shaped sheet forming by deformable punches describes about the experimental investigations into the deformation mechanisms of V-shaped sheet forming by deformable punches and rigid dies.

CHAPTER 3

SHEET METAL

3.1 INTRODUCTION TO SHEET METAL FORMING

Sheet Metal forming (SMF) is one of the most common metal manufacturing process used. Its applications are wide in aircraft components, automobile components etc. The characteristics of sheet metal forming processes are:

- ✓ The work piece is a sheet or a part fabricated from a sheet
- ✓ The surfaces of the deforming material and of the tools are in contact
- ✓ The deformation usually causes significant changes in shape, but not in cross-section (sheet thickness and surface characteristics) of the sheet
- ✓ In some cases, the magnitude of permanent plastic and recoverable elastic deformation is comparable, therefore elastic recovery or springback may be significant.

The technical-economic advantages of Sheet Metal Forming are that it is a highly efficient process that can be used to produce complex parts. It can produce parts with high degree of dimensional accuracy and increased mechanical properties along with a good surface finish. But the limitation is that the deformation imposed in Sheet Metal Forming process is complicated.

Stamping is one type of Sheet Metal Forming process, which is widely used in automotive industries. The popularity of stamping is mainly due to its high productivity, relatively low assembly costs and the ability to offer high strength and lightweight products.

In general, the deformation of sheet materials in the stamping process is classified by four types of deformation modes; i.e., bending, deep drawing, stretching and stretch flanging. Since this project deals with the bending process, this study will be focused on the bending operation.

3.1.1 Advantages and Disadvantages of Sheet Metal Forming process

Advantages of Sheet Metal Forming

- Highly efficient process that can be used to produce complex parts.
- High degree of dimensional accuracy and increased mechanical properties.
- Good surface finish.

Limitation of Sheet Metal Forming

- Deformation imposed is complicated.

3.1.2 Stamping process

The Stamping process is a Sheet Metal forming process and is widely used in automotive industries. The advantages of Stamping are:

- High productivity
- Relatively low assembly costs
- Ability to offer high strength and lightweight products

Classification of Stamping

Deformation of the Sheet Metal in the Stamping process is classified according to type of deformation mode.

- Bending
- Deep drawing
- Stretching
- Stretch Flanging

3.2 SHEET METAL BENDING

Sheet Metal Bending process is the plastic deformation of metals about a linear axis called the bending axis with little or no change in the surface area. Bending types of forming operations have been used widely in sheet metal forming industries to produce structural stamping parts.

Sheet metal bending is a fabricating process and in which initially flat sheets are permanently deformed into intricate shapes by the use of plate rolls or dies. Thus it is the operation of deforming a flat sheet around a straight axis where the neutral axis lies.

The work piece is a sheet or a part fabricated from a sheet. The surfaces of deforming material and of the tools are in contact. The deformation usually causes significant changes in shape, but not in cross-section (sheet thickness and surface characteristics) of the sheet. The magnitude of permanent plastic and recoverable elastic deformation is comparable, therefore elastic recovery or springback may be significant.

One of the important characteristics noticed during the bending operation is that the tensile stress decreases toward the center of the sheet thickness and becomes zero at the neutral axis whereas the compressive stress increases from the neutral axis toward the inside of the bend as shown in Figure 3.1. Even with large plastic deformation in bending, the center region (elastic metal band or zone) of the sheet remains elastic and so on unloading elastic recovery occurs.

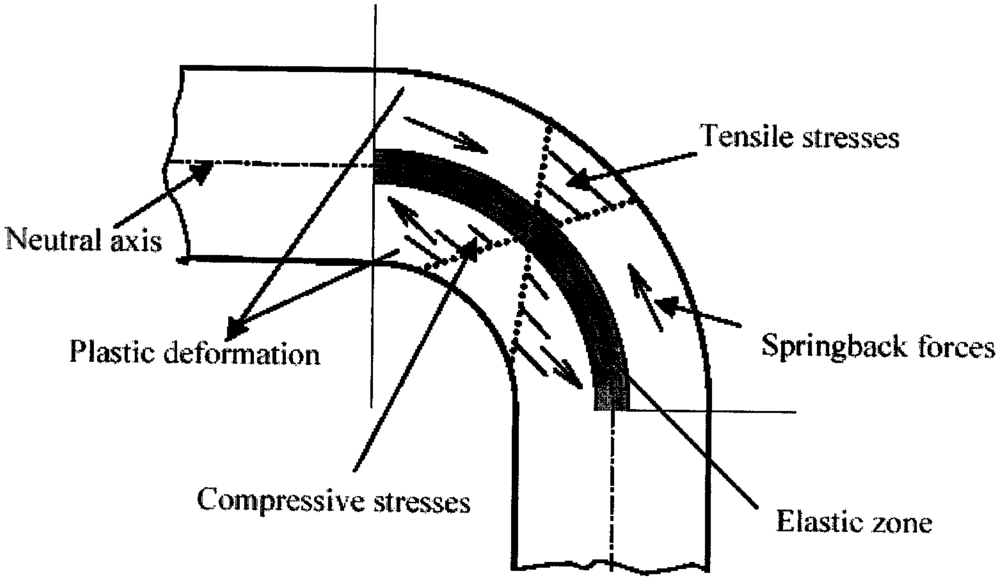


FIGURE 3.1 BENDING OF SHEET METAL

The list of parts produced by Sheet Metal Bending process are :

- Braces
- Bracket
- Supports
- Hinges
- Angles
- Frames
- Channels
- Other nonsymmetrical sheet metal parts

3.2.1 Types of Bending process

The Bending process can be broadly classified into :

1. Air Bending
2. Bottoming
3. Coining

The other important types of bending operations are :

- ✓ V-bending
- ✓ U-die bending
- ✓ Wiping die bending
- ✓ Double die bending
- ✓ Rotary bending

3.3 V - BENDING PROCESS

Figure 3.2 illustrates the V-bending process. Sheet metal is placed over the die and bent as the punch descends into the die. The V-die bending process falls into two categories, namely Air bending and Bottom bending. This study is limited to Bottom bending, in which the punch fully sets in the die. The first diagram shows the loading process and the second one shows the forming of the V-bend. Upon unloading (third diagram) springback or spring-in (negative springback) is observed depending on the process and material parameters used, this context is explained in the later chapters.

The clearance between punch and die is constant and is equal to the thickness of sheet blank. The thickness of sheet ranges from approximately 0.5 to 25 mm. It is widely used.

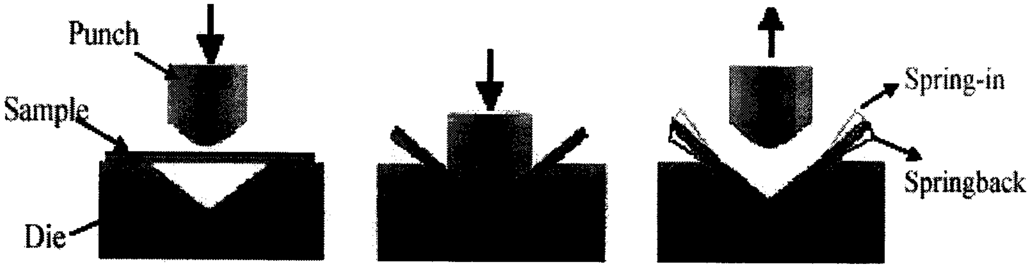


FIGURE 3.2 V - BENDING PROCESS

3.4 SPRINGBACK

Springback or elastic recovery refers to the shape discrepancy between the fully loaded and unloaded configurations. The stress strain plot shown in Figure 3.3 illustrates the springback phenomenon. Upon unloading in a Bending process there is elastic recovery, which is the release of the elastic strains and the redistribution of the residual stresses through the thickness direction, thus producing springback.

Springback causes changes in shape and dimensions that can create major problems in the assembly; hence springback prediction is an important issue in sheet metal forming industry.

Many factors could affect springback in the process, such as material variations in mechanical properties, sheet thickness, tooling geometry (including die radius and the gap between the die and the punch), processing parameters and lubricant condition.

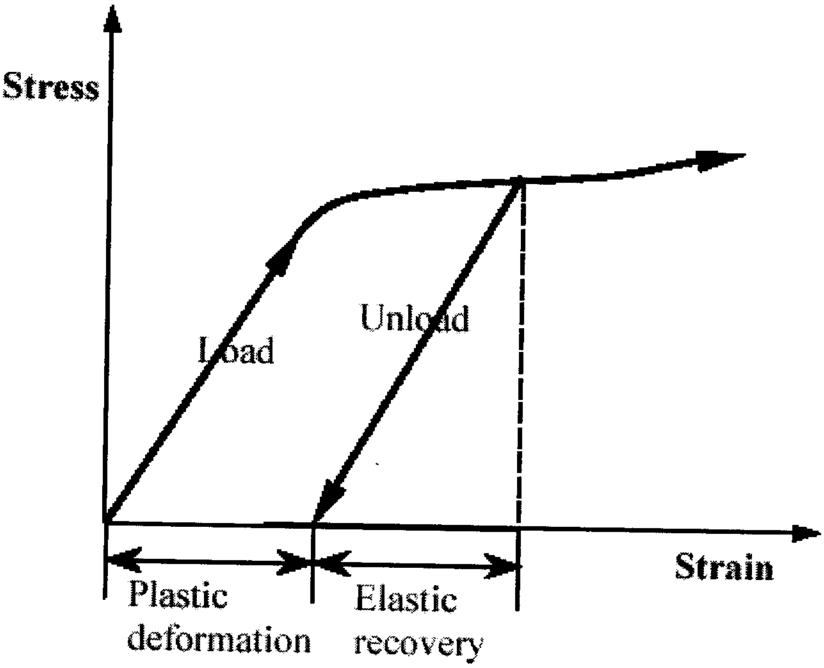


FIGURE 3.3 SPRINGBACK PHENOMENON

3.4.1 Factors affecting Springback

Process parameters

- Bend radius
- Die gap
- Punching speeds

Material properties

- Sheet Thickness
- Flow stress
- Texture and grain size

Various investigations of springback prediction show that process parameters such as bend radius, die gap and punching speeds, and material properties such as sheet thickness, flow stress, texture and grain size have considerable influence on springback. Many of the factors affecting springback are also manifested in the minimum bending radius or bendability limit in addition to the surface or edge condition of the sheet. Research showed the effect of increasing the die profile radius as the material strength increases. Larger springback has been correlated with an increase in normal anisotropy and decrease in strain hardening exponent.

When the load-application device is released, the sheet tends to return back to its initial shape. Springback deformation must be taken into consideration by deforming the sheet initially to an extent beyond that which is finally required, by an amount usually determined from experience or experimental data.

3.5 IMPORTANCE OF SPRINGBACK CALCULATION

- Accurate springback prediction is imperative for robust design of tooling; thereby saving costs and die try-out times.
- Springback causes changes in shape and dimensions that creates major problems in the assembly. The automotive industry places rigid constraints on final shape and dimensional tolerances. Hence springback prediction and simulation is critical in this highly automated environment.
- With increased demand from the industries to shorten the lead times and with increased usage of lightweight, higher strength materials in manufacturing, springback simulation is essential for proper design of the forming tools.

3.6 SHEET METAL FORMING SIMULATION

Numerous research have been done on the use of the finite element code for sheet metal forming simulations. Others have developed an elasto-plastic finite element code based on static-explicit FE, which is suitable for plastic instability and also springback problems. This code was used by another researcher to analyze the automotive sheet metal forming process. Paulsen et al. (1996) analyzed the bending process with an implicit code and reported good agreement between experimental and simulation results. Journal publications have compared implicit and explicit simulations, finding that the response differences were almost negligible.

Investigations have shown that explicit method is well suited for solving large sheet forming models, and implicit codes handle the springback calculation very efficiently. Hence, recently researchers have adopted a new method of coupling the implicit and explicit methods to solve complex sheet metal forming processes would save design effort and production time.

Narasimhan et al. (1999) have used ANSYS LS-DYNA explicit coupled with ANSYS implicit for springback simulation. Others combined the commercial codes LS-DYNA3D and NIKE3D for prediction of springback in automotive body panels. Researchers have discussed the numerical solution of sheet-metal forming applications using the ABAQUS general purpose implicit and explicit finite-element modules.

CHAPTER 4

EXPERIMENTAL DESCRIPTION

4.1 MATERIAL CHARACTERIZATION

6022-T4 Aluminum Alloy developed in early 1990's is becoming popular in automotive industry and it is the material that was used in this project. This aluminum alloy is a precipitation-strengthened alloy with major alloying elements Mg and Si. It is intended for automotive body sheet applications. The T-4 processing includes a high temperature solution heat treatment, a quench, and then natural aging to a microstructurally stable condition. The chemistry and material properties of the material are summarized in Table 4.1.

TABLE 4.1 COMPOSITION AND MATERIAL PROPERTIES

PROPERTY	VALUE
Composition	Si 0.8-1.5, Mg 0.45-0.70, Fe 0.05-0.20, Mn 0.02-0.10, Cu 0.01-0.11, Ti 0.15, Cr 0.10, Zn 0.25
Young's Modulus	69 GPa
Ultimate Tensile Strength	236-237 MPa
Yield Strength	125-126 MPa
Elongation	27.5%

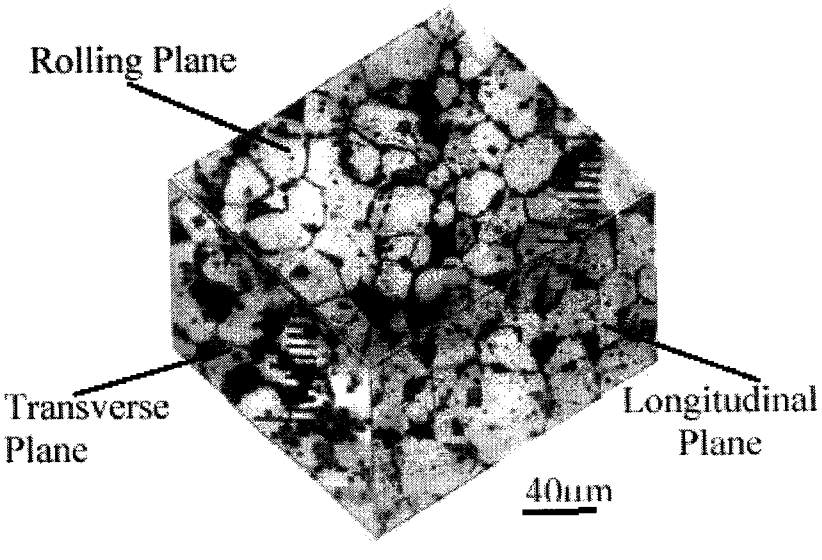
4.1.1 Microstructure

A common lot of 6022-T4 Aluminium Alloy was used for all tests. The 6022-T4 aluminum alloy sheet which was received from Dynamax Pvt. Ltd., Bangalore was sectioned in three orientations and was mounted, polished (as described in Table 4.2) and etched using a modified Keller's etchant (25ml methanol, 25 ml hydrochloric acid, 25ml nitric acid and one drop of hydrofluoric acid).

TABLE 4.2 POLISHING PROCEDURE

Abrasive / Surface	Lubricant	Time
240 grit SiC paper 320 grit SiC paper 400 grit SiC paper 600 grit SiC paper	Water Water Water Water	1 minute 1 minute 1 minute 1 minute
6μm pad	Water + Alpha alumina	3 minutes
1μm pad	Water + Alpha alumina	2 minutes
0.05μm pad	Colloidal Silica polishing suspension	1 minute

The microstructure of the 6022-T4 Aluminium Alloy sheet material is shown in Figure 4.1. The microstructure shows equiaxed grains with an average size of 40 μm.



4.1.2 Heat Treatment / Hardness

A comprehensive heat treatment study was conducted to increase the grain size of the material without changing its hardness. This was done to vary the grain size for the test matrix. Negligible grain growth was observed in Aluminium 6022-T4 below 560° C. The software Scion Image was used to measure the diameter of the grains. The dark areas constitute the second phase Mg2Si particles. The grain sizes of 125 μm and 185 μm were obtained by heat-treating the material at 560° C for 4hrs and 24 hrs respectively.

Rockwell B hardness tests indicated negligible change in the hardness of the material due to heat treatment as illustrated in Table 4.3.

TABLE 4.3 ROCKWELL B HARDNESS VALUES AT VARIOUS TEMPERATURES

Test Points	1	2	3	4	Average
Room Temperature	64.5	68.0	71.0	68.5	68.0 +/- 4.0
560° C, 4 hrs.	67.0	71.5	68.0	72.5	69.8 +/- 4.0
560° C, 24 hrs.	63.0	67.0	70.5	77.5	69.5 +/- 7.0

4.1.3 Tensile Testing

Uniaxial tension tests were conducted on 6022-T4 Aluminium Alloy specimens (0.85mm thick approximately) to plot the stress strain plots in 0, 45 and 90 degrees to the rolling direction. An Instron model 5800 electromechanical load frame with mechanical grips was used to test the specimens in uniaxial tension as

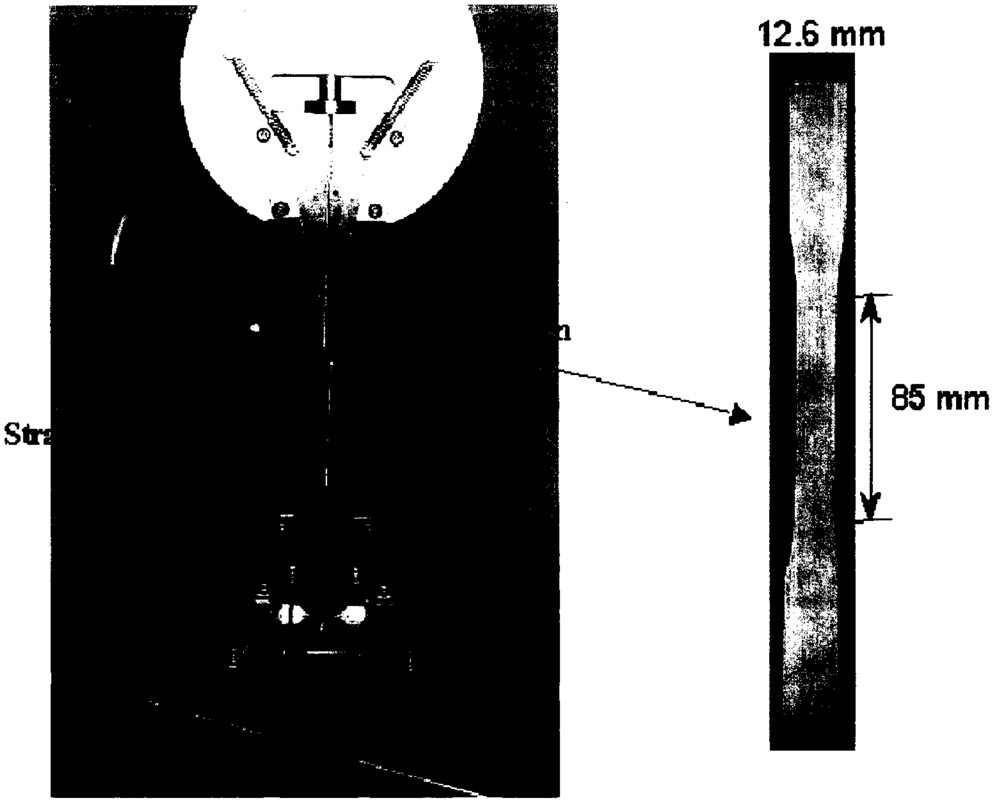


FIGURE 4.2 UNIAXIAL TENSILE TESTING

Three repeats at each orientation were conducted. Also the tests were run at different speeds i.e. at minimum and maximum speed of the Instron machine (5.1mm/min and 510mm/min), but it was found that the material properties were rate independent as seen in the graphs, Figure 4.3 to 4.5

The graphs from Figure 4.3 to Figure 4.5 illustrates the engineering stress strain plots recorded at different speeds of the load frame (5.1mm/min and 510mm/min) with three repetitions each. The uniaxial tests are conducted at 0, 45 and 90 degrees to the rolling direction. As seen from the plots, there is not much variation in the properties of the material at the varying speeds; this signifies that the material properties are rate independent.

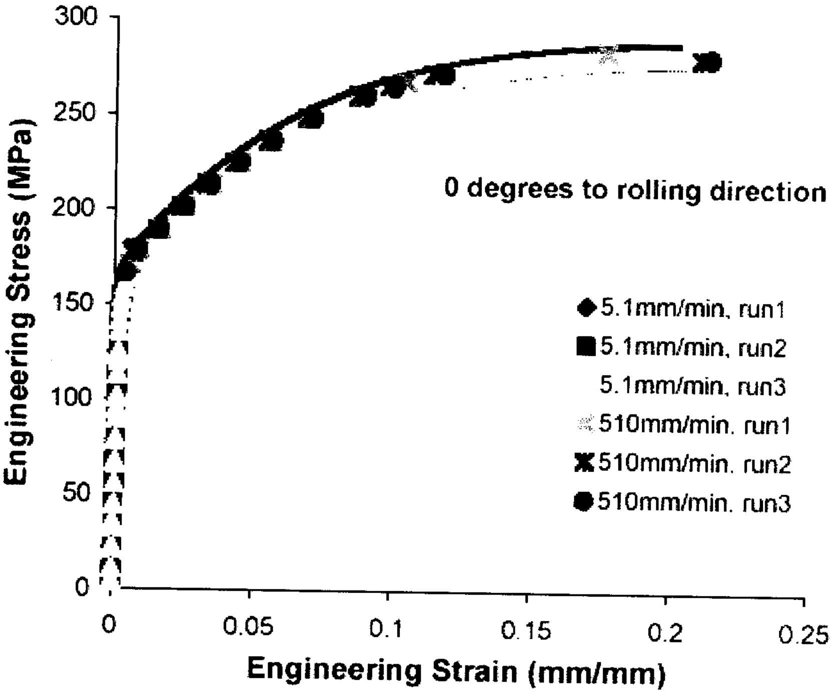


FIGURE 4.3 STRESS STRAIN PLOT AT DIFFERENT SPEEDS WHEN TESTED 0° TO ROLLING DIRECTION

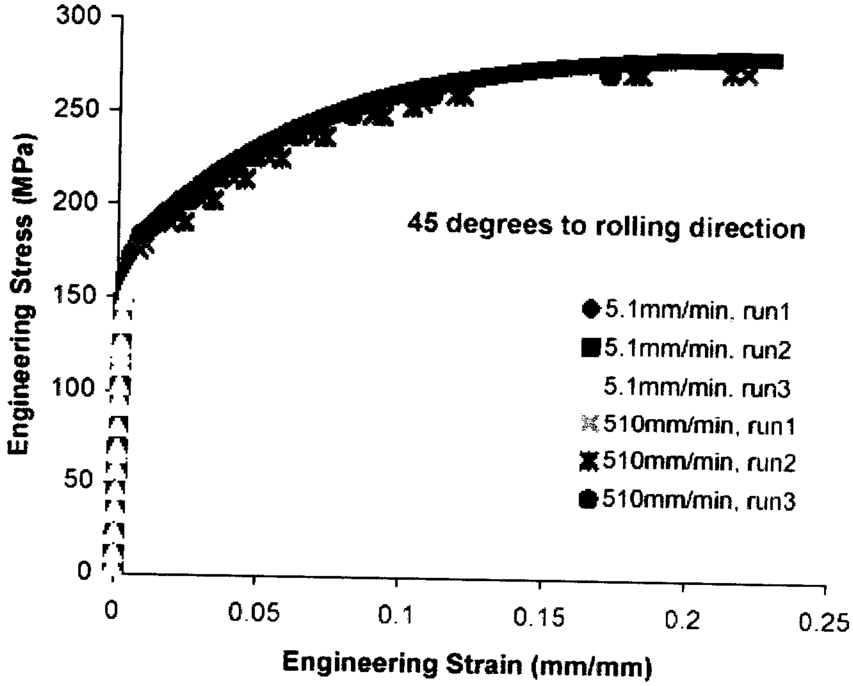


FIGURE 4.4 STRESS STRAIN PLOT AT DIFFERENT

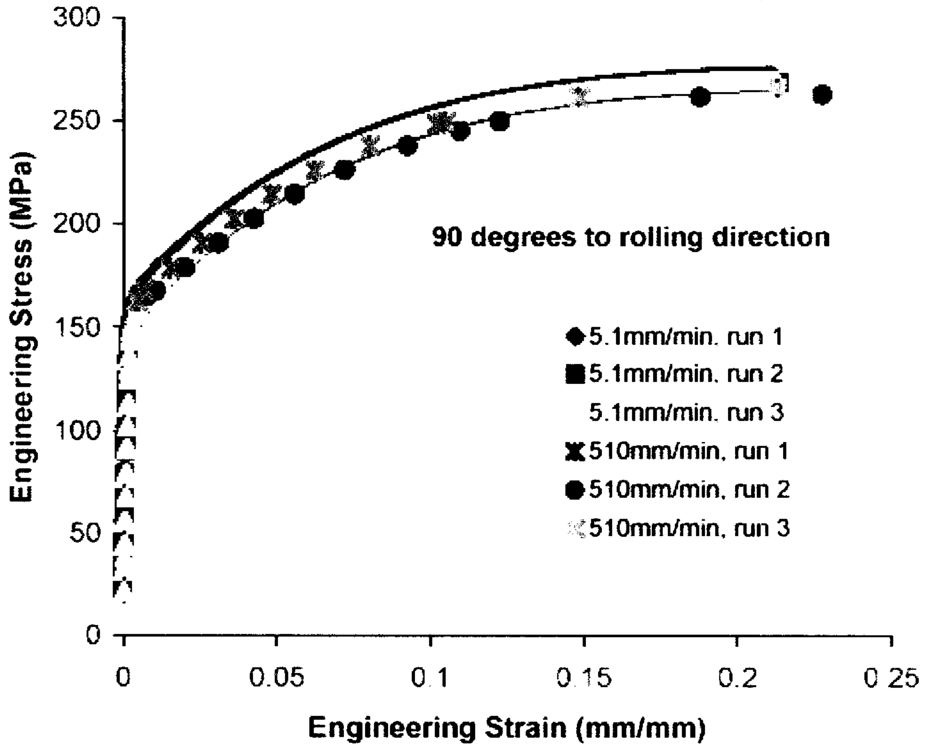


FIGURE 4.5 STRESS STRAIN PLOT AT DIFFERENT SPEEDS WHEN TESTED 90° TO ROLLING DIRECTION

The following Figure 4.6 depicts the true stress-strain plots of 6022-T4 Aluminium Alloy in all three different directions.

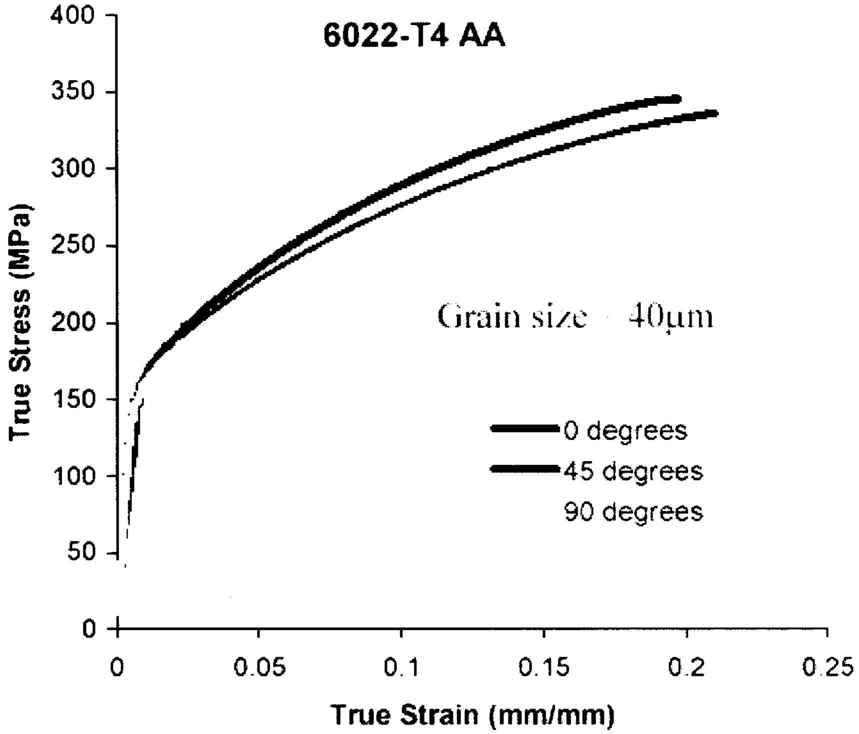


FIGURE 4.6 TRUE STRESS-STRAIN PLOTS FOR 6022-T4 AA IN THREE DIRECTIONS TO THE ROLLING PLANE

This data will be used in the simulations to observe the effect of material behavior (texture) on springback. The other data in three directions to rolling direction (RD) are listed in Table 4.4 and the respective plots are illustrated in Figure 4.7

TABLE 4.4 MECHANICAL PROPERTIES OF 6022-T4 AA TESTED IN UNIAXIAL TENSION AT DIFFERENT DIRECTIONS

Property	**0° to Rolling Direction**	**45° to Rolling Direction**	**90° to Rolling Direction**
Young's Modulus	69 GPa	69 GPa	69 GPa
Yield Strength	170 MPa	160 MPa	163 MPa
Ultimate Tensile Strength	285 MPa	272 MPa	273 MPa
% Elongation to Failure	20.27	22.63	21.52
Toughness	46.21 MPa	45.48 MPa	46.16 MPa
Resilience (Work)	0.51 MPa	0.4 MPa	0.489 MPa
r value	0.7	0.48	0.59

The graphs in Figure 4.7 describe the mechanical properties of 6022-T4 Aluminium Alloy tested uniaxially at three directions to the rolling plane, i.e., 0, 45 and 90 degrees. It shows that 0 degrees has the highest strength followed by 90 degrees. 45 degrees to the rolling plane has the least strength.

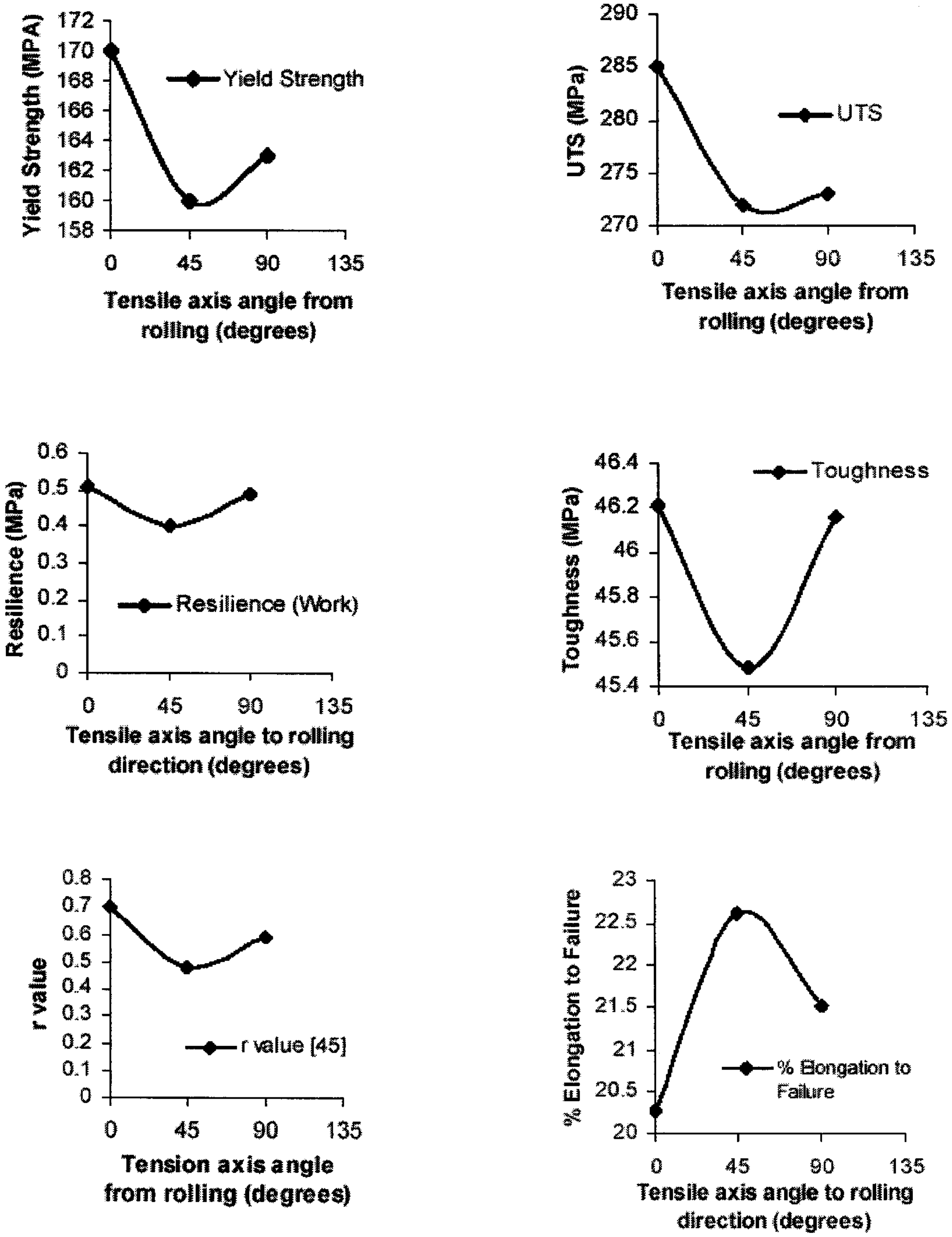


FIGURE 4.7 MECHANICAL PROPERTIES OF 6022-T4 ALUMINIUM ALLOY

4.2 DESIGN OF EXPERIMENTS

4.2.1 Taguchi Methodology

The Taguchi design of experiments method was used in this project to evaluate the relative contribution of process and material parameters on springback in V-die bending. According to Taguchi, quality characteristic is a parameter whose variation has a critical effect on product quality, e.g., weight, cost, target thickness, strength, material properties, etc. The Taguchi quality strategy is to improve quality in the product design stage by:

(1) Making the design less sensitive towards influence of uncontrollable factors
(2) Optimizing the product design.

Designing an experiment:

Taguchi method uses a special set of arrays called orthogonal arrays. These standard arrays stipulate the way of conducting the minimal number of experiments, which could give the full information of all the factors that affect the performance parameters.

4.2.2 Taguchi Matrix

Experiments were designed to study the parameters that effect springback. Three levels of sheet thickness were chosen to represent variations in the received sheet. Bending the sheet metal at three different directions, parallel, perpendicular and forty five degrees to the rolling direction, was the boundary condition used to evaluate the contribution of planar anisotropy on springback. Previous research has shown that forming speed has a great effect on springback behavior of the formed part. The maximum tool velocity of the Instron model 5800 load frame used for V-Bending is 8.5 mm/s; hence, the punching speeds are varied between 0.085mm/s and 8.5mm/s. Carden et al. (2002) have speculated that for 6022-T4 aluminum, the springback angles continued to increase for periods up to several months after sheet metal forming. Hence, shelf life and dwell time were included

Experimental investigations and analytical calculations showed that when the ratio of the die gap to sheet thickness is slightly greater than one, the effect of die gap on springback is greatest. For this study the die gap was set to be equal to the sheet thickness, to eliminate the effect of die gap. Carden et al. (2002) showed that friction in normal industrial ranges (i.e., well lubricated to dry conditions) has no measurable effect on springback, although very low friction conditions increase springback for 6022-T4.

In this study, lubrication was not taken into consideration. A common tool radius of 9.5mm is referenced for the automotive industry. For this study, three bending radii were selected, 3, 5, and 9.5 mm.

The following Tables 4.5 and 4.6 show the Taguchi test matrix for the tests to be performed on prediction of springback during V-bending process. To design experimental matrix for seven factors with three levels, the L_{18} orthogonal array was most applicable. The L_{18} array requires the minimum number of tests (18) to investigate the factor effect on springback. In this study only the individual effect of each factor on springback was investigated. The L_{18} is not structured to study interactions.

TABLE 4.5 FACTOR AND LEVEL DESCRIPTIONS

Factor	**Factor Description**	**Level 1**	**Level 2**	**Level 3**
A	Bend Radius	3 mm	5 mm	9.5 mm
B	Sheet Thickness	0.84 mm	0.86 mm	0.89 mm
C	Grain size	40 μm	125 μm	185 μm
D	Rolling Direction	Parallel	Perpendicular	45 degrees
E	Punching speeds	0.085 mm/sec	0.85 mm/sec	8.5 mm/s
F	Shelf life	None	15 days	2 months
G	Dwell Time	None	30 min	1 hour
e	Error	N/A	N/A	N/A

TABLE 4.6 L_{18} TEST MATRIX

Run #	Factors							
	e	A	B	C	D	E	F	G
1	1	1	1	1	1	1	1	1
2	1	1	2	2	2	2	2	2
3	1	1	3	3	3	3	3	3
4	1	2	1	1	2	2	3	3
5	1	2	2	2	3	3	1	1
6	1	2	3	3	1	1	2	2
7	1	3	1	2	1	3	2	3
8	1	3	2	3	2	1	3	1
9	1	3	3	1	3	2	1	2
10	2	1	1	3	3	2	2	1
11	2	1	2	1	1	3	3	2
12	2	1	3	2	2	1	1	3
13	2	2	1	2	3	1	3	2
14	2	2	2	3	1	2	1	3
15	2	2	3	1	2	3	2	1
16	2	3	1	3	2	3	1	2
17	2	3	2	1	3	1	2	3
18	2	3	3	2	1	2	3	1

4.3 V-BEND TEST

A V-bend fixture was designed and built to install on a Model 5800 Instron EM load frame as shown in Figures 4.8 and 4.9. The experiments were performed using the bend fixture at Dynamax Pvt. Ltd., Bangalore. The dimensions of the sheet metal specimens used in the V-bend test are 56mm length and 30.5mm width. The sample is not restrained during the bending process. The two linear ball bearing / bushing assemblies as shown in the figure guarantee accurate punch guidance.

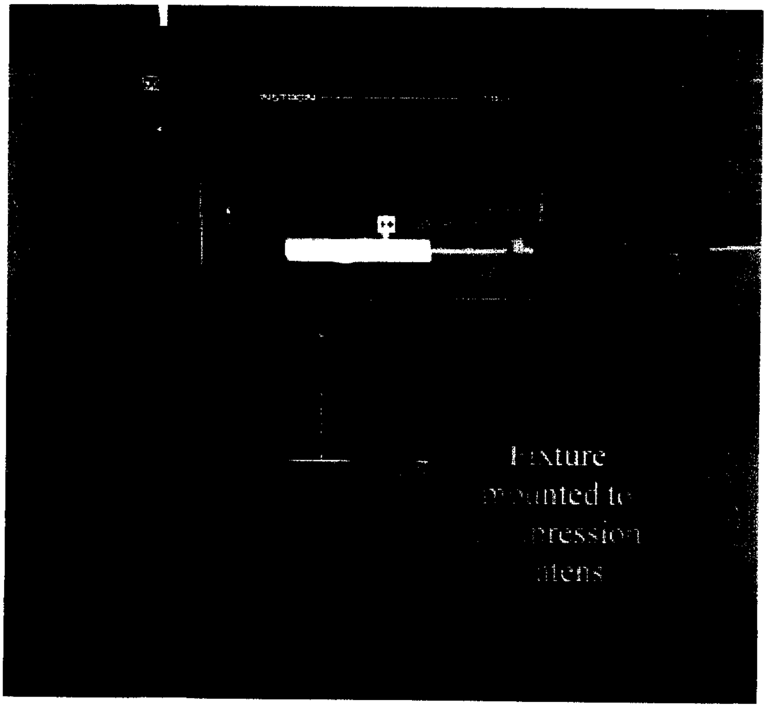


FIGURE 4.8 V-BEND FIXTURE INSTALLED ON MODEL 5800 INSTRON MACHINE

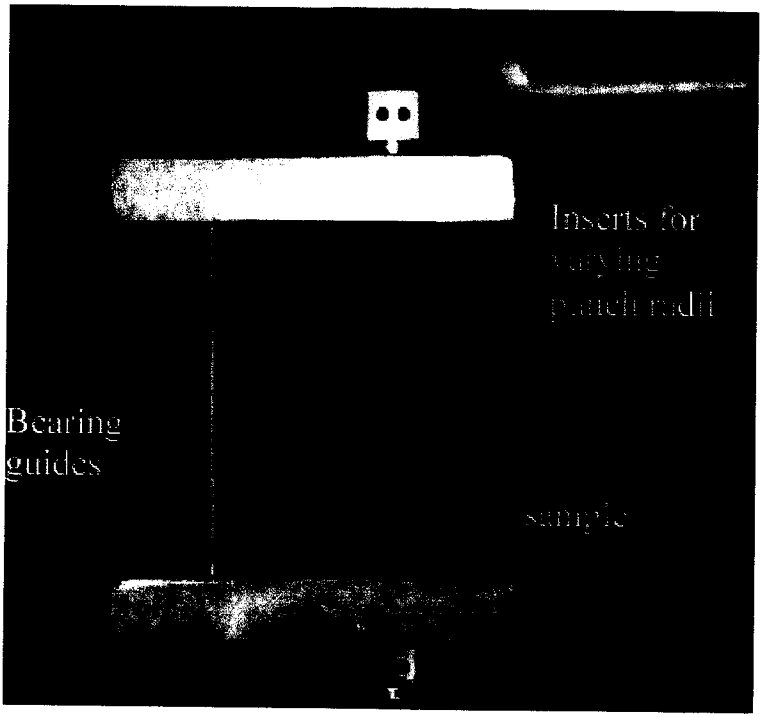


FIGURE 4.9 V-BEND PROCESS

CHAPTER 5

NUMERICAL SIMULATION

5.1 FINITE ELEMENT ANALYSIS

The finite element method, a powerful numerical technique, has been applied in the past years to a wide range of engineering problems. Although much FE analysis is used to verify the structural integrity of designs, more recently FE has been used to model fabrication processes. When modeling fabrication processes that involve deformation, such as SMF, the deformation process must be evaluated in terms of stresses and strain states in the body under deformation including contact issues. The major advantage of this method is its applicability to a wide class of boundary value problems with little restriction on work piece geometry.

The three basic requirements for the successful commercial application of numerical simulation are :

(1) Simplicity of application

(2) Accuracy

(3) Computing efficiency.

<u>The characteristic features of the finite element method are :</u>

The domain of the problem is represented by a collection of simple sub domains, called *finite elements.* The collection of finite elements is called *the finite element mesh.* Over each finite element, the physical process is approximated by functions of desired type (polynomials or otherwise), and algebraic equations relating physical quantities at selective points, called *nodes*, of the element are developed. The use of finite element analysis is beneficial in the design of tooling in sheet metal forming operations because it is more cost effective than trial and error. The prime objective of an analysis is to assist the design of the product by:

(1) predicting the material deformation and

(2) predicting the forces and stresses necessary to execute the forming operation.

5.1.1 Comparison of Implicit and Explicit codes

Implicit code solves for equilibrium at the every time step *(t + ?t)*. Depending upon the procedure chosen, each iteration requires the formation and solution of the linear system of equations. Explicit method solves for equilibrium at time *t* by direct time integration. This explicit procedure is conditionally stable since iterative procedure is not implemented to reach equilibrium and also *?t* is limited by natural time. The Table 5.1 summarizes the difference in implicit and explicit codes.

Numerous research have been done on the use of the finite element code for sheet metal forming simulations. Paulsen et al. (1996) analyzed the bending process with an implicit code and reported good agreement between experimental and simulation results. Clausen et al compared implicit and explicit simulations, finding that the response differences were almost negligible.

Investigations have shown that explicit method is well suited for solving large sheet forming models, and implicit codes handle the springback calculation very efficiently. Hence, recently researchers have adopted a new method of coupling the implicit and explicit methods to solve complex sheet metal forming processes would save design effort and production time.

Narasimhan et al. (1999) have used ANSYS/LS-DYNA explicit coupled with ANSYS implicit for springback simulation. Others have combined the commercial codes LS-DYNA3D and NIKE3D for prediction of springback in automotive body panels.

TABLE 5.1 IMPLICIT AND EXPLICIT CODES COMPARISON

S.No	Implicit	Explicit
1.	Large time increment can be adopted and the equilibrium is rigorously satisfied at the end of the time step.	It restricts the time increment to very small size in order to maintain the out of balance force within admissible tolerance.
2.	In some cases implicit finite element analysis may develop convergence problems associated with sudden changes on the contact conditions between work piece and tools.	The solution procedure is stable even if the deformation dependent contact problem is included in the process.
3.	Several equilibrium iterations must be performed for each time step and for each iteration it is necessary to solve a set of linear equations.	It requires fewer computations per time step. Complex geometries may be simulated with many elements that undergo large deformations.
4.	They are not well suited to solving the interaction of a large number of nodes with rigid tooling, but they do handle the springback calculation very efficiently.	Although explicit codes are well suited to solving large sheet-forming models with large number of deformable elements, calculation of geometry after springback may be difficult.
5.	Generally favored for relatively slow problems with static or slowly varying loads.	Generally favored for fast problems such as impact and explosion.

5.1.2 Summary of Elements used in SMF Simulation

FEA of the sheet metal forming problem usually adopts one of three analysis methods based on the membrane, shell and continuum element. The Table 5.2 summarizes the elements used in FE method for sheet metal forming simulation.

TABLE 5.2 ELEMENTS USED IN SMF SIMULATION

Element	Advantage	Limitation
Membrane	Computational efficiency and better convergence in contact analysis than the shell or continuum element	It does not consider bending effect and has to tolerate inaccuracy in the bending dominant problems.
Shell	Can capture the combination of stretching and bending as opposed to membrane elements. Use of shell element gives more number of degrees of freedom to capture accurate stress distribution including in- plane and out-plane deformation.	It takes a substantial amount of computational time and computer space for its 3-D calculation with integration in the thickness direction.
Continuum	Are used where fully 3-D theory is needed to describe the deformation process. They can handle through - thickness compressive straining, whereas shell elements cannot.	More elements are needed to describe the shell-type structures, so that a large system of equations must be solved.

5.2 MATERIAL BEHAVIOR

The history of plasticity theory dates back to 1864 when Tresca published his yield criterion based on his experimental results on punching and extrusion. When a material body is subjected to external forces, it deforms. The type of deformation is dependent on the load applied and the material. A reversible and time independent deformation is called elastic. A reversible but time-dependent deformation is known as viscoelastic, where the deformation increases with time after application of load, and it decreases slowly after the load is removed. The deformation is called plastic if it is irreversible or permanent.

Plasticity theory deals with the establishment of stress-strain and load-deflection relationships for a plastically deforming ductile material or structure. This involves the experimental observation, and the mathematical representation.

The theories of plasticity can be divided into two groups:

(1) The mathematical theory and

(2) The physical theory.

Mathematical theories are formulated to represent the experimental observations as general mathematical formulations. They are based on hypotheses and assumptions from experimental results. Whereas, the physical theories require a deep knowledge of the physics of plastic deformation at the microscopic level and it should explain why and how the plastic deformation occurs.

Mechanics of plastic deformation can be understood and quantified if one looks into the microstructure of the material and explores the mechanisms of the plastic deformation or flow at the microscopic level.

The four fundamental elements of plastic deformation are :

(1) Initial yield surface

(2) Constitutive equations for hardening parameters

(3) Constitutive equations for plastic strain and

(4) Loading and unloading criteria

5.2.1 Yield Criterion

The yield surface is an important concept in plasticity since it defines the critical stress levels beyond which plastic deformation occurs, and it serves as a potential for the strains. It divides the stress space into the elastic and plastic domains. It is used together with a constitutive equation as the material input for numerical simulations of forming processes.

A yield criterion is a basic assumption about a material for the purpose of determining the onset of the plastic deformation. The yield function can be written mathematically in the general form.

$F(s_{ij}) < 0$ (5.1)

with $F(s_{ij}) < 0$ for elastic deformation domain (5.2)

$F(s_{ij}) = 0$ for plastic deformation domain (5.3)

If the material is isotropic, the yielding depends only on the magnitudes of the principal stresses. For such materials the yield criterion is given by:

$$F(s_1, s_2, s_3) = 0 \tag{5.4}$$

For fully anisotropic materials the initial yield criterion should be expressed in terms of six independent components of the stress tensor s :

$$F(s_{ij}) < 0 \text{ or } F(s_{xx}, s_{yy}, s_{zz}, s_{xy}, s_{yz}, s_{zx}) = 0 \tag{5.5}$$

In terms of principal stresses and principal directions n_i ($i = 1, 2, 3$)

$$F(s_1, s_2, s_3, n_1, n_2, n_3) = 0 \tag{5.6}$$

Isotropic material is one whose properties do not vary with distance or direction whereas for an anisotropic material it is vice versa. The above equation indicates that the yielding of an anisotropic material depends both on the "intensity" of the stress tensor (the principal stresses) and also on its "orientation" (the principal directions).

The purpose of applying plasticity theory in metal forming is to investigate the mechanism of plastic deformation in metal forming processes. Such investigations allows the analysis and prediction of :

- ✓ Metal flow behavior (velocities, strain rates and strains)
- ✓ Temperatures and heat transfer
- ✓ Local variation in material strength or flow stress
- ✓ Stresses, forming load, pressure and energy.
- ✓ Limit strains above which failure occurs.

The nonlinear isotropic / kinematic-hardening model with the von Mises yield criterion predicts springback very accurately for bending dominant problems.

5.2.2 Constitutive models

Constitutive equations describe the non-linear stress-strain relationship of the material used in the structural components being analyzed. They relate the stress to strain and / or strain rate that characterize the behavior of a material under an application of forces or loads. These equations vary for different materials. They can even differ for the same material in different regimes of deformation. These constitutive models are in relation to the material parameters, which have to be determined. The equations below show the basic constitutive relations.

$$s = E\,e \quad \text{in the elastic region} \tag{5.7}$$

$$s = k\,e^{n} \quad \text{in the plastic region} \tag{5.8}$$

where s is the flow stress, e is the strain, *k* is the material constant and *n* is the strain hardening exponent.

Describing the flow stress of a material has to incorporate factors such as degree and rate of deformation and temperature during processing. The combined effect of these factors on flow stress is rather complicated. Hence there is a need for a constitutive equation that quantifies the effects of these factors on the flow stress of the work material.

A constitutive model must be computationally efficient so that it can be implemented in large computer codes. Many constitutive models have been proposed and used in the past but they vary in complexity and adaptability to numerical computational schemes. Some of these models describe only the variation of yield stress with strain rate changes, while others describe strain and strain rate hardening effects without softening effects caused by temperature.

Materials subjected to large deformation and very high rates of strain require information on the mechanical behavior in the form of a constitutive equation, which relates the stress system in the material to the instantaneous values of strain, strain rate and temperature. While strain is not a true state function, constitutive equations of this form usually allow an adequate description of the structural response to be predicted for most engineering purposes.

Numerical models require equations, which account for variations in behavior. The constitutive equation, when incorporated in a suitable finite element code, allows the changing deformation and stress state within the component to be determined during the course of the impact for the given boundary and loading conditions. To obtain the appropriate constitutive equation for a given material, data are usually obtained from standard specimen tests, which are performed at constant temperature and strain rate.

5.3 SHEET METAL FORMING SIMULATION

Finite element method is generally composed of three basic steps, namely: preprocessing of input data, computational analysis, and post-processing of results. The description of these terms when simulating sheet metal forming process are :

Preprocessing:

It is the creation of a geometric model for the part to be formed, the imposition of the appropriate boundary conditions for the forming process, the selection of the constitutive equation for plastic deformation, and the selection of material and process variables.

Computational analysis:

It involves solving appropriate equations to obtain the deformed shape of the part.

Post-processing:

Results from numerical simulation runs provide predicted shapes of the panels as well as stress and strain distribution data for the entire surface area of the formed parts. Surface stress and strain data for the deformed shapes are given in the form of color-coded or contour plots to facilitate the interpretation of results.

5.3.1 Convergence Criteria

The modeling of sheet metal forming processes is one example of highly nonlinear problems where the iterative solution procedure can become very slow or diverge. The non-linearities observed can be categorized as geometrical non-linearities, material non-linearities due to plastic deformation, friction force reversals and contact mode changes. In a static finite element code, high mesh elements. But a sufficiently fine mesh must be generated so that the fine features are captured.

In order to obtain optimal and reliable convergence, it is essential that the rigid tool surface representation is smooth. Numerous authors have reported such convergence problems in analyzing sheet metal forming processes. Studies were conducted to overcome these problems.

Researchers have suggested implementing rigid-body motion constraints for unloading elements. Some others have presented a contact element with damping in the direction normal to the sheet which smoothens the transition from opened to closed contact elements and vice versa. Investigators paid special attention to the continuity in the tool surface description, a careful treatment of the contact condition, a smoothened friction and applied an anisotropic hardening law.

5.3.2 INTRODUCTION TO ANSYS LS-DYNA

The premier software package for explicit nonlinear structural simulation LS-DYNA was combined with one of the industry's most recognized and respected finite element pre and post processors ANSYS to get ANSYS LS-DYNA, the result of a collaborative effort between ANSYS, Inc. and Livermore Software Technology Corporation (LSTC). First introduced in 1996, ANSYS LS-DYNA's capabilities and robustness have helped thousands of customers in numerous industries resolve highly intricate design issues.

ANSYS LS-DYNA is built upon successful ANSYS interface. ANSYS LS-DYNA is an integrated pre and post processor for the world's most respected explicit dynamic solver LS-DYNA. This combination makes it possible to solve combined explicit/implicit simulation.

Limited-duration events (such as severe collisions and blade containment) and large, permanent deformations (within the stamping and forming industries) these challenges by fusing LSTC's LS-DYNA explicit dynamic solver technology with the pre-/post-processing power of ANSYS software. This powerful pairing helps engineers understand the elaborate combinations of nonlinear phenomena found in crash tests, metal forging, stamping and catastrophic failures.

ANSYS LS-DYNA supports both 2-D and 3-D explicit elements, and features an extensive set of single-surface, surface-to-surface and node-to-surface contact as well as a contact analysis option that automatically creates the contact surfaces. ANSYS LS-DYNA also gives optional methods for fast solution processing. Because ANSYS LS-DYNA is created from the same powerful technology as ANSYS Multiphysics, it is easy to combine with other ANSYS products and tailor to your specific needs. For instance, you can incorporate the ANSYS Drop Test Module for an easy-to-use GUI for simulating dynamic impacts. Add ANSYS Structural or ANSYS Mechanical into your ANSYS LS-DYNA package for comprehensive implicit structural simulation capabilities.

LS-DYNA is the world's premier analysis tool for simulating time-dependent mechanical events. Built from the ground up as an explicit solver to deal with things like impact, crashes, ballistic penetration, and very large deformation, the program is used by anyone who is serious about these types of events. ANSYS users have an added advantage through the use of the ANSYS/LS-DYNA package to pre- and post-process runs without having to learn an entirely new set of tools.

Material Models of ANSYS LS-DYNA :

Elastic

- Isotropic
- Orthotropic
- Anisotropic
- Fluid

Nonlinear Elastic

- Blatz-Ko rubber
- Mooney-Rivlin rubber
- Viscoelastic

Elastoplastic

- Elastic-plastic hydrodynamic
- Bamman rate-dependent
- Zerilli-Armstrong rate-dependent
- Bilinear isotropic
- Bilinear kinematic
- Plastic kinematic
- Powerlaw plasticity
- Strain rate-dependent plasticity
- Rate-sensitive powerlaw plasticity
- Three-parameter Barlat
- Barlat anisotropic plasticity
- Piece-wise linear plasticity
- Transversely anisotropic elastic plastic

Foam

- Closed-cell
- Low-density
- Viscous
- Crushable
- Honeycomb

Damage

- Composite
- Concrete

Equations of State

- Johnson-Cook

Overview of ANSYS LS-DYNA Analysis capabilities :

- ✓ Nonlinear Dynamics
- ✓ Rigid Multi-Body Dynamics
- ✓ Quasi-Static Simulations
- ✓ Element-based Failure Analysis
- ✓ Design Optimization
- ✓ Implicit Capabilities
- ✓ Static Analysis and Transient Analysis
- ✓ Linear / Nonlinear Analysis

Applications of ANSYS LS-DYNA

- ✓ Metal forming
- ✓ Drop testing
- ✓ Glass forming
- ✓ Plastics, mold, and blow forming
- ✓ Machining
- ✓ Automotive industry - Vehicle crashes, Air bag inflation and impact
- ✓ Seismic loading of structures
- ✓ Military and Defense Applications - Weapons loading during firing, Ballistic penetration
- ✓ Aerospace Industry Applications - Bird impact on engines, wings and windscreens
- ✓ Ice impact
- ✓ Behavior of fabrics/shields
- ✓ Explosion containment/resistance

Sheet Metal Forming With ANSYS LS-DYNA

One of ANSYS LS-DYNA's most widely used applications is sheet metal forming. ANSYS LS-DYNA accurately predicts the stresses and deformation experienced by the metal and also determines whether the metal will fail.

Metal forming applications for ANSYS LS-DYNA include:

- Sheet metal bending
- Metal stamping
- Hydroforming
- Forging
- Deep drawing
- Multi-stage processes

5.4 SPRINGBACK SIMULATION

The automotive industry places rigid constraints on final shape and dimensional tolerances. Hence springback prediction and compensation is critical in this highly automated environment. In the past two decades, finite element method has proven to be a powerful tool in simulating sheet metal forming processes. With the increasing demand from the industries to shorten the lead times and with increased usage of lightweight, higher strength materials in manufacturing auto-body panels, the simulation of springback has become essential for proper design of the forming tools.

Springback simulation is difficult and complicated; to obtain useful results requires accurate material and geometric description of the process in the formulation.

The interaction of the work piece with the tooling needs to be described precisely. Various approaches were adopted and formulated by researchers in the past years to accurately simulate springback in sheet metal forming processes. Advances in springback simulation to reduce computational time and increase overall efficiency of the process have been tremendous. Studies involving the use of different types of materials models, finite element codes, elements, hardening rules, frictional constraints, etc.

Researchers have simulated springback in V-die bending for several materials neglecting friction. The true stress-strain curve from a tensile test was used as the material description. They noted a good correlation between simulation and experimental results. Huang and Leu have used elasto-plastic incremental finite-element computer code based on an updated Lagrangian formulation to simulate the V-die bending process of sheet metal under the plane-strain condition. Isotropic and normal anisotropic material behavior was considered including nonlinear work hardening.

Engineers have studied springback and springforward phenomena in V-bending process using an elasto-plastic incremental finite element calculation. Ogawa et al. did a somewhat similar study but used different element mesh sizes and compared their results with experimental predictions. Lee and Yang evaluated the numerical parameters that influence the springback prediction by using FE analysis of a stamping process. Others have showed that a material property described by the kinematic hardening law provides a better prediction of springback than the isotropic hardening law. Analytical model and FEA results were compared with the experimental results.

Li et al. (2002) used a linear hardening model and an elasto-plastic power-exponent hardening model to study the springback in V-free bending. According to their results, the material-hardening mode directly affects the springback simulation accuracy. Others have also showed that the simulated springback angle depends intimately on both hardening law after the strain reversal and on the plastic anisotropy. They anisotropic hardening model that extends existing mixed kinematic / isotropic and non-linear kinematic formulations. Li et al. (2002) compared the use of solid and shell elements in their springback simulation with 2D and 3D finite element modeling.

5.4.1 Finite Element Codes

Commercially available Finite Element Codes for Sheet Metal Forming Simulation are listed below with their specialty and disadvantages:

LS-DYNA explicit can handle complex problems with large deformation and has no convergence problems. But the limitation is that it does not have preprocessor and complex post-processor.

ABAQUS implicit-explicit can solve problems with large deformation, no convergence problems. But the limitation is that it does not have preprocessor.

ANSYS implicit has the advantage of the availability to use user material model But there is convergence problem during nonlinear analysis and contact conditions.

Thus in this project springback simulation of V-die bend process using ANSYS LS-DYNA is studied. The nonlinear isotropic / kinematic-hardening model with the von Mises yield criterion predicts springback very accurately for bending dominant problems. Thus a 2-D model is considered with von Mises yield criterion.

5.4.2 V-Bend Simulation Approach

There are various factors that need to be considered while simulating sheet metal forming processes, which is a large deformation problem, such as the complications of geometrical and material nonlinear behaviors, frictional contact boundary conditions, solution procedure for convergence, etc. The following describes the approach adopted in this research to simulate springback in V-die bending process.

- Finite element code: 2-D finite element modeling is considered in the springback simulations. Research results in the literature have shown that for the forming phase, an explicit code is suitable and for the springback phase of the simulation, a static implicit time integration approach is preferable. However, since modeling of V-Bending is not very complex, this study used ANSYS implicit for both loading and unloading process.
- Material model: Due to rolling processes, metal sheets before stamping operations usually exhibit significant plastic anisotropy that can be attributed principally to the presence of crystallographic texture. Hence, anisotropy is an important parameter that has to be considered for more realistic modeling of sheet metal bending. In this study two classes of material behavior are compared. Initially the multilinear isotropic material model is used, where the true stress-strain material description is input in discrete form. These results will be compared to Barlat's 2000 anisotropic material model. This plane stress anisotropic yield function describes the planar anisotropic behavior of aluminum alloy sheets.
- Modeling assumptions: The tooling was treated as rigid surfaces (tool steel) and lubrication was not taken into account. The 6022-T4 Aluminium Alloy sheet metal was taken as deformable. The coefficient of friction between the work piece and the tool was assumed to remain constant during the process. The value of friction coefficient used was 0.16. The boundary and load conditions were set to the same condition as in the experiment. The deformation is achieved by prescribing the displacement of the punch, which corresponds with how the deformation is achieved in reality. Guided by the experimental results, the approach toward modeling is to vary dominant parameters. Due to symmetry only one half of the geometry was modeled.

- Element description: Bending dominated problems are generally simulated with solid or shell elements. Use of shell element gives more number of degrees of freedom to capture accurate behaviour including in-plane and out-plane deformation. In this analysis the sheet is modeled with deformable contact shell elements whereas the tooling is modeled with rigid shell elements. Four noded quadrilateral elements have been used in the simulations since investigations have shown that triangular finite elements can cause numerical problems or deteriorate the solution accuracy.

CHAPTER 6

RESULTS

6.1 TAGUCHI DESIGN OF EXPERIMENTS TEST RESULTS

The L_{18} matrix has been used to conduct the experiments and the springback angles are recorded with three test points for each experiment as illustrated in the same Table 6.1. A vernier protractor is used to measure the V-angle. The least count of the protractor is 5'. The factor descriptions can be referred back to the Table 4.5. The Taguchi analysis approach consists of :

ANOVA (Analysis of Variance)

This is done to find the relative contribution of each control factor to the overall measured response.

TABLE 6.1 L_{18} TEST MATRIX AND RECORDED SPRINGBACK ANGLES

Run #	Factors								Springback angle - 3 test points			Total (radians)
	e	A	B	C	D	E	F	G	(a)	(b)	(c)	
1	1	1	1	1	1	1	1	1	$-4^{0}35'$	$-4^{0}45'$	$-3^{0}45'$	-13.083
2	1	1	2	2	2	2	2	2	$-3^{0}20'$	$-3^{0}25'$	$-3^{0}40'$	-10.417
3	1	1	3	3	3	3	3	3	$-3^{0}20'$	$-3^{0}0'$	$-3^{0}30'$	-9.833
4	1	2	1	1	2	2	3	3	0^{0}	$-0^{0}35'$	$0^{0}35'$	0
5	1	2	2	2	3	3	1	1	$-1^{0}30'$	$-1^{0}15'$	$-0^{0}55'$	-3.667
6	1	2	3	3	1	1	2	2	$-0^{0}55'$	$-1^{0}0'$	$-0^{0}35'$	-2.5
7	1	3	1	2	1	3	2	3	$2^{0}25'$	$0^{0}40'$	$1^{0}0'$	4.084
8	1	3	2	3	2	1	3	1	$2^{0}10'$	$1^{0}30'$	$1^{0}25'$	5.084
9	1	3	3	1	3	2	1	2	$4^{0}15'$	$4^{0}30'$	$3^{0}50'$	12.58
10	2	1	1	3	3	2	2	1	$-3^{0}45'$	$-4^{0}55'$	$-4^{0}45'$	-13.417
11	2	1	2	1	1	3	3	2	$-3^{0}45'$	$-3^{0}30'$	$-3^{0}0'$	-10.25
12	2	1	3	2	2	1	1	3	$-4^{0}10'$	$-4^{0}0'$	$-3^{0}20'$	-11.5
13	2	2	1	2	3	1	3	2	$-2^{0}10'$	$-2^{0}50'$	$-1^{0}40'$	-6.667
14	2	2	2	3	1	2	1	3	$-1^{0}25'$	$-2^{0}15'$	$-3^{0}10'$	-6.834
15	2	2	3	1	2	3	2	1	$-0^{0}50'$	$-0^{0}50'$	$-1^{0}25'$	-3.083
16	2	3	1	3	2	3	1	2	$1^{0}20'$	$1^{0}20'$	$1^{0}15'$	3.916
17	2	3	2	1	3	1	2	3	$3^{0}35'$	$3^{0}50'$	$3^{0}20'$	10.746
18	2	3	3	2	1	2	3	1	$1^{0}30'$	$0^{0}30'$	$1^{0}15'$	3.25

6.1.1 TAGUCHI ANALYSIS DESCRIPTION

The relative contribution of each control factor to the overall measured response is obtained by using an analysis of variance (ANOVA). A mathematical technique known as the sum of squares is used to quantitatively evaluate the deviation of the control factor effect response averages from the overall experimental mean response. An F-ratio is used to test for the significance of factor effects. This is done by comparing the variance between the individual control factor effects (V) against the variance in the experimental data due to random experimental error (Ve). Table 6.2 summarizes the initial ANOVA with the F-Statistics for the factors.

ANOVA (Analysis of Variance)

Analysis of variance is a computational technique that quantitatively estimates the relative contribution of each control factor to the overall measured response and expresses it as a percentage. It uses a mathematical technique known as the sum of squares to quantitatively examine the deviation of the control factor effect response averages from the overall experimental mean response. The significance of the individual control factors is quantified by comparing the variance between the control factor effects against the variance in the experimental data due to random experimental error and the effects of unrepresented interactions. This is given by the F-ratio. It is used to test for the significance of factor effects.

$$F-ratio=\frac{a}{b}$$

Where a = mean square due to a control factor

b = mean square due to experimental error

In the Taguchi approach, ANOVA can be applied to two forms of data. First, it can be applied to the data as measured in engineering units to find the

Second, it can be applied to the data after it has been transformed into S/N ratios. This is done to find the effect of noise due to repetition of runs.

The first step in this analysis is to calculate the totals for each of the factors and the error column. The first level of factor A would be represented by all the data points where the first level of Factor A occurred in the experiment. The first level of factor A occurred in experiments 1, 2, 3, 10, 11, and 12. This is done in the same manner for each level of each of the factors and the error column. Table 6.2 represents the totals table for the analysis of variance.

TABLE 6.2 TOTALS TABLE FOR ANOVA

	Totals	Mean
A1	-68.500	-3.80556
A2	-22.751	-1.26394
A3	39.660	2.20333
B1	-25.167	-1.39817
B2	-15.338	-0.85211
B3	-11.086	-0.61598
C1	-3.090	-0.17167
C2	-24.917	-1.38428
C3	-23.584	-1.31022
D1	-25.333	-1.40739
D2	-16.000	-0.88889
D3	-10.258	-0.56989
E1	-17.920	-0.99556
E2	-14.838	-0.82433
E3	-18.833	-1.04628
F1	-18.588	-1.03267
F2	-14.587	-0.81039
F3	-18.416	-1.02311
G1	-24.916	-1.38422
G2	-13.338	-0.74100
G3	-13.337	-0.74094
e1	-17.752	-0.65748
e2	-33.839	-1.2533

Sum of squares :

Sum of Squares for the total S_T is calculated by taking each of the 54 data points and square them. From the sum of those squared observations, subtract away the total of all the data points, squared, divided by the total number of data points that is 54.

$S_T = (-4.583)^2 + (-4.75)^2 + (-3.75)^2 + (-3.333)^2 + \ldots\ldots\ldots\ldots\ldots + (1.25)^2 - (-51.591)^2$
$= 378.047$

Sum of Squares for factors:

$S_* = (*1)^2 / n_{*1} + (*2)^2 / n_{*2} + (*3)^2 / n_{*3} - (*1 + *2 + *3)^2 / n$
$S_A = A_1^2 / 18 + A_2^2 / 18 + A_3^2 / 18 - (A_1 + A_2 + A_3)^2 / 54$
$= 376.82 - 49.29 = 327.53$

Similarly for other factors:

$S_B = 55.085 - 49.29 = 5.795$
$S_C = 65.923 - 49.29 = 16.633$
$S_D = 55.72 - 49.29 = 6.43$
$S_E = 49.776 - 49.29 = 0.486$
$S_F = 49.8579 - 49.29 = 0.5679$
$S_G = 54.2547 - 49.29 = 4.9647$
$S_{e1} = (-17.752 + 33.839)^2 / 54 = 4.79$
$S_{e2} = S_T - (S_A + \ldots + S_{e1}) = 10.8504$

Degrees of Freedom:

The Degrees of Freedom must be calculated for each of the factors, the error term and the total. The Degrees of Freedom for the total is obtained by taking the total number of data points and subtracting 1. The Degrees of Freedom for any factor is computed by taking the number of levels for that factor and subtracting 1. The Degrees of Freedom for Error is the left over Degrees of Freedom not accounted for by the factors.

Total: $df_T = n-1 = 54-1 = 53$

Factors: $df_A \ldots\ldots df_G = 3-1 = 2$

Error (primary) $df_{e1} = 2-1 = 1$

Error (secondary) $df_{e2} = df_T - (df_A + df_B + df_C + df_D + df_E + df_F + df_G + df_{e1})$

$= 53-15 = 38$

Variance:

Variance is the sum of squares divided by the Degrees of Freedom.

F-Statistic:

A test statistic is now calculated to help test for the significance of the difference that is demonstrated by the experiment for the outcome of interest. To calculate the F-Statistic for a factor divide the Variance for that factor by the Variance for error, forming a ratio of variances. The bigger this ratio is the more difference there is between the levels for a factor. Thus an initial ANOVA Table is evaluated and is illustrated in Table 5.3.

Since $V_{e1} >= V_{e2}$ the secondary error is discarded and all the factors are tested versus the primary error.

TABLE 6.3 INITIAL ANOVA FOR SPRINGBACK MEASUREMENT

Factor	**DOF (df)**	**Sum of squares (S)**	**Variance = S/df**	**F - Statistics = V/V$_e$**
A	2	327.53	163.765	34.189
B	2	5.795	2.898	0.605
C	2	16.633	8.317	1.736
D	2	6.43	3.215	0.672
E	2	0.486	0.243	0.051
F	2	0.568	0.284	0.059
G	2	4.965	2.482	0.518
el (primary)	1	4.79	4.79	
e2 (secondary)	38	10.850	0.286	
Total	53	378.047		

The primary error term is because one column of L_{18} is not filled and secondary error term is because there are repetitions of runs. Next pooling of the factors whose F-ratio is less than one is done with the error term. Concentration of the factors is made to better analyze the experiment. Table 6.3 illustrates the pooled ANOVA table for our experiments. A Percentage Contribution (P%) is also computed for the remaining terms.

This is done by taking the Pure Sum of Squares (S') for each of the terms and dividing by the sum of squares for total (S_T).

Pooling rules:

1. Factors that have an F-Statistic less than or equal to 1 should be pooled into an error term and thought of as a random variation component.
2. When after pooling all the factors where the F ratio is less than or equal to one, the number of factors remaining is not equal to or less than ½ the number of columns in the array, then additional pooling must be done.

Calculations for pooled error term: Pooling factors whose F<1 with the error term

$S_e\,(\text{pool}) = S_B + S_D + S_E + S_F + S_G + S_e = 23.034$

$df_e\,(\text{pool}) = 2*5 + 1 = 11$

$V_e\,(\text{pool}) = S_e\,(\text{pool}) / df_e\,(\text{pool}) = 23.034/11 = 2.094$

Next the pure sum of squares must be calculated. This is given by the formula as below:

$S_A = S_A - (df_A)V_e = 323.342$

$S_C = S_C - (df_C)V_e = 12.445$

$S_e = S_e - (df_T)V_e = 23.034 + (15\text{-}11)\,2.094 = 31.41$

A Percentage Contribution (P%) is also computed for the remaining terms. This is done by taking the Pure Sum of Squares for each of the terms and dividing by the sum of squares for total. The pooled ANOVA Table is described in Table 6.4.

TABLE 6.4 POOLED ANOVA TABLE

Factor	**DOF**	**S**	**V**	**F (pool)**	**S'**	**P% = S'/S_T**
A	2	327.53	163.77	78.206*	323.34	88.06
C	2	16.633	8.3165	4.077**	12.445	3.39
e (pool)	11	23.034	2.094		31.41	8.55
Total	15	367.20			367.20	100

* Significant at 99 % confidence $F_{.99}$ (2, 11) = 7.20

* Significant at 95 % confidence $F_{.95}$ (2, 11) = 3.98

Table 6.4 shows that factor A that is bend radius is the major factor that contributes to springback in V-Bending process. Factor C that is grain size ended up being a factor to a lesser extent. The rest of the factors that are pooled in the error term are thought of as a random variation component.

6.2 SIMULATION TEST CASES

Experimental results have shown that bend radius is the only predominant factor that effected springback and it was significant at 99% confidence level. Grain size and texture had some contribution to springback but it was significant at 95% confidence level only. The contribution of shelf life was valid below 95% confidence level. Hence to validate the significance of process parameters versus the material parameters on springback in V-bending, finite element analysis was conducted using ANSYS LS-DYNA.

In this project, ANSYS LS-DYNA which is a generic code was used due to lack of availability of other special sheet metal forming codes.

Table 6.5 which is depicted below shows the Simulation test matrix that was conducted.

TABLE 6.5 FINITE ELEMENT SIMULATION TEST MATRIX

Run No.	Bend Radius (mm)
1	3
2	5
3	9.5

Model:

The Figure 6.1 illustrates the meshing and boundary condition description of the model.

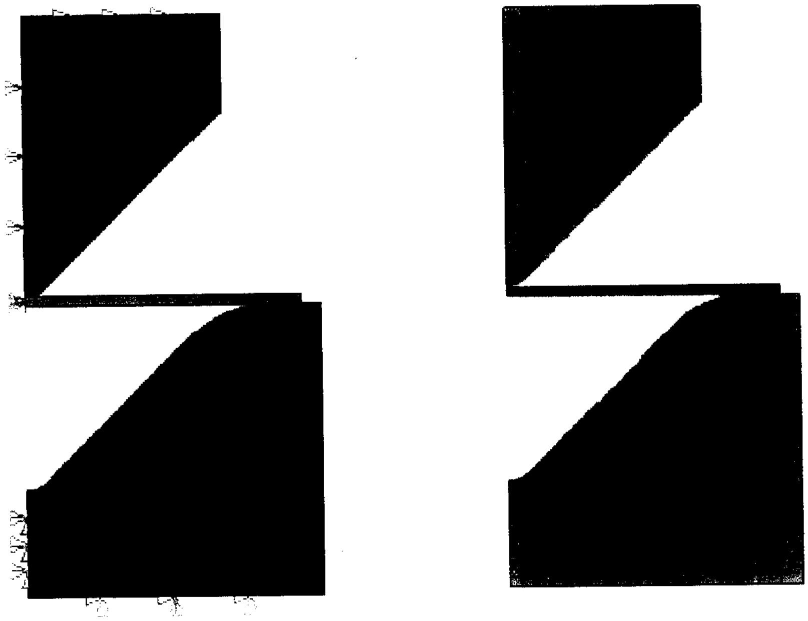


FIGURE 6.1 V-BEND MODEL WITH BOUNDARY CONDITIONS AND MESH

6.2.1 V-Bend Simulation Results

Effect of Bend Radius:

The test matrix for simulation was conducted and the results are shown in the Table 6.6.

TABLE 6.6 V-BEND SIMULATION RESULTS

Run No.	Bend Radius (mm)	Springback angle (degrees)
1	3	- 3.5
2	5	- 0.5
3	9.5	4.5

Figures below show the simulated results for 3mm, 5mm and 9.5mm radius. Figure 6.2 specifically shows the exploded view of the sheet under deformation.

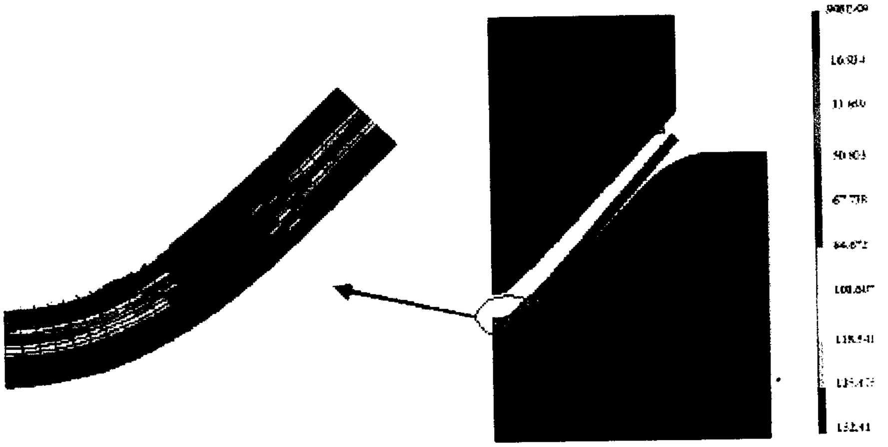


FIGURE 6.2 SIMULATED RESULT FOR BEND RADIUS 3MM

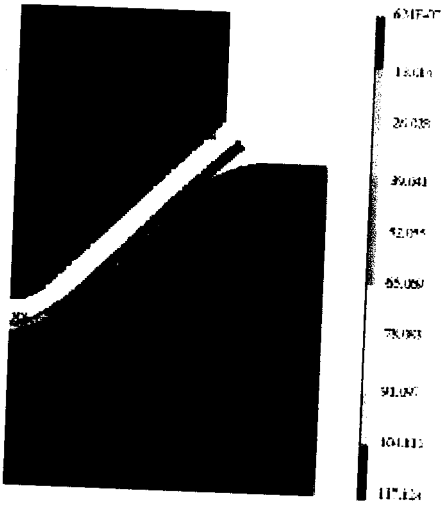


FIGURE 6.3 SIMULATED RESULT FOR BEND RADIUS 5MM

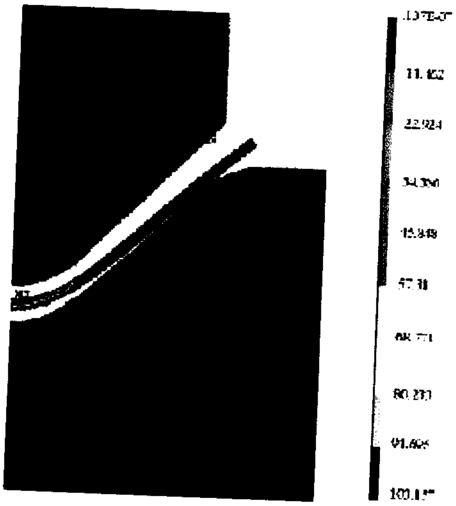


FIGURE 6.4 SIMULATED RESULT FOR BEND RADIUS 9.5MM

CHAPTER 7

DISCUSSIONS

7.1 TAGUCHI EXPERIMENTAL RESULTS

This study revealed some interesting results. Negative springback or spring-in was observed in experiments with bend radius less than 5mm. Previous investigations have shown that this behavior is noted for thin sheets ($R/t < 5$, where *R* is the bend radius and *t* is the thickness of the sheet). Some post-forming deformations may have occurred during removal of the part from the forming press, accounting for the flexibility of the thin sheets.

Moreover, aluminum alloys are known to adhere easily to the tools, hence on application of high loads, spring-in may have occurred on unloading, due to lower flow stresses. Bending at a sharp radius can cause the formation of invisible cracks, which might have caused a reduction in springback.

There was no stress cracking observed by light microscopy in the samples tested. Davies et al. (1994) found that for ratios of material thickness to bend radius greater than 0.2, there was a considerable amount of negative springback observed. This is because the higher the strength, the larger the deformed volume of the material in the bend.

According to Zhang et al. (1997) as the punch pushes down, the two ends of the specimens are forced towards the die surfaces and some plastic deformation may occur in the two contact regions of the specimen with the die resulting in negative springback. Other effects of various factors on springback are discussed below.

7.1.1 Effect of Bend Radius

Table 6.3 shows that bend radius has a significant effect on springback, and it overshadows all the other factors. Figure 7.1 shows the sectional profiles of specimens at various bend radii after forming.

smaller bending radii, the sheet is deformed more locally and severely, resulting in the increased plastic stiffness of the bent zone and hence creep would be reduced. Investigations show that the stress over the punch corner is the most significant factor that governs the magnitude of springback. Hence springback is greater for the larger die radius this is due to the comparatively small bending stress locked into the sheet at the punch corner.

When the sheet is bent with a smaller radius, the metal under the punch is stressed beyond the yield strength through almost the entire sheet thickness. Such enlargement of the plastic zone produces a reduction in the springback angle.

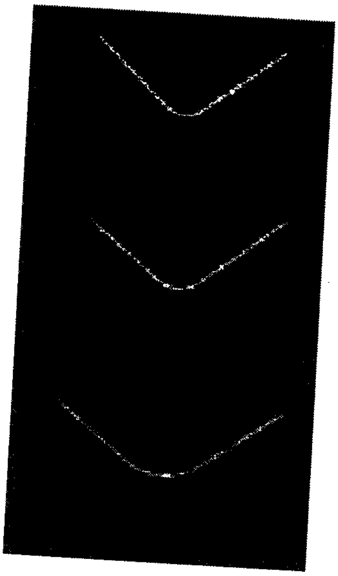


FIGURE 7.1 V-BENDS AT DIFFERENT BEND RADII

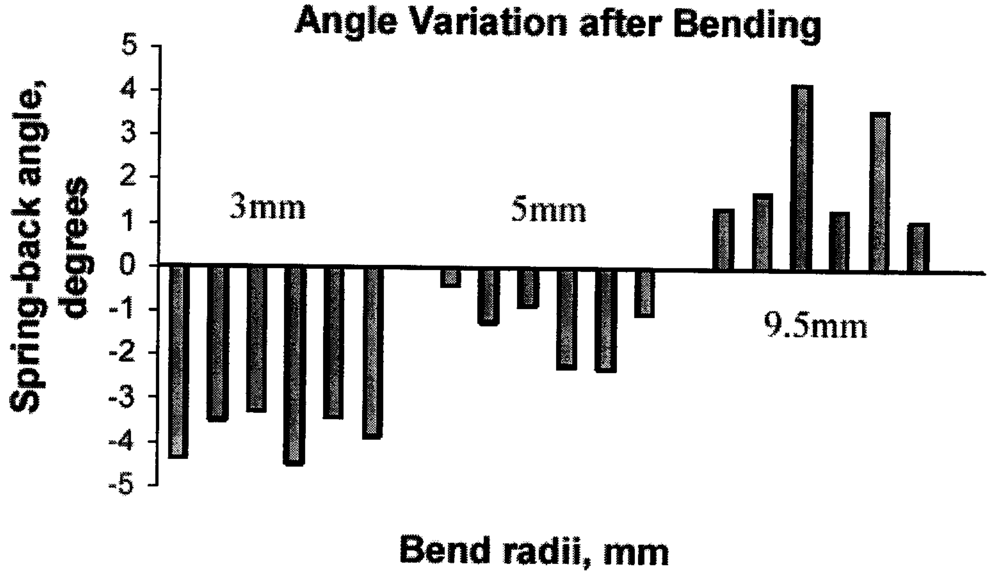


FIGURE 7.2 VARIATION OF SPRINGBACK ANGLE WITH RESPECT TO DIFFERENT BEND RADII

Gardiner conducted V-bend experiments to measure springback and studied the effect of bend radius. In this study tests with small bend radii were most difficult to conduct and resulted in a large data scatter. However, the experimental results shown in Figure 7.2 illustrate less scatter in the springback angle at small bend radius. Others have conducted experiments for springback of small radius-to-thickness bends in the range R/t<10 and showed larger springback than that predicted by any known theory.

7.1.2 Effect of Other Factors

Previous investigations have shown that using thicker material will reduce springback. For a given radius of the outer fiber under loading, the increase in the sheet thickness leads to an increase in the bending moment and the bending strain at the outer fiber, thus reducing the springback angle. In our study, the contribution of sheet thickness to springback is not significant when compared to the surface, develop less roughening and have better formability characteristics. That is springback is reduced when the material has a finer grain size. Present results show that grain size is significant only to a very small extent.

Sheet metals that exhibit different flow strengths in different directions in the plane of the sheet are defined as having planar anisotropy. Parallel, perpendicular and forty-five degrees to the rolling direction represent the three vectors of the planar anisotropy. Previous study showed that anisotropy has a great effect on the bending limit with the relative differences in yield strength. Springback is reported higher at higher strength, reflecting minimal spring observed for the bend perpendicular to the rolling direction. In this study the rolling direction was found to be somewhat significant.

Shelf life and dwell time are linearly proportional to the springback behavior; this may be related to the creep characteristics of aluminum alloy. Present results have shown that shelf life has very little effect on springback. Contribution of punch speed and dwell time to springback are almost negligible when compared to bend radius in this study.

7.2 FINITE ELEMENT SIMULATION RESULTS

A 2-D finite element model with multilinear isotropic material description gives a good prediction of springback. Simulated results are obtained for different bend radii 3mm, 5mm and 9.5mm. The Springback angles for the different bend radii are determined. The experimental results compare well and are in good agreement with the simulated results for the effect of bend radius on springback. Although an attempt that was made to study the effect of texture (planar anisotropy) on springback was not successful. This might well be a topic for further study. Hence this specific aspect recommended for follow-on studies by using different sheet metal forming codes, which support material models that describe the anomalous behavior of highly textured aluminum alloys.

7.3 COMPARISON OF EXPERIMENTAL AND SIMULATION RESULTS

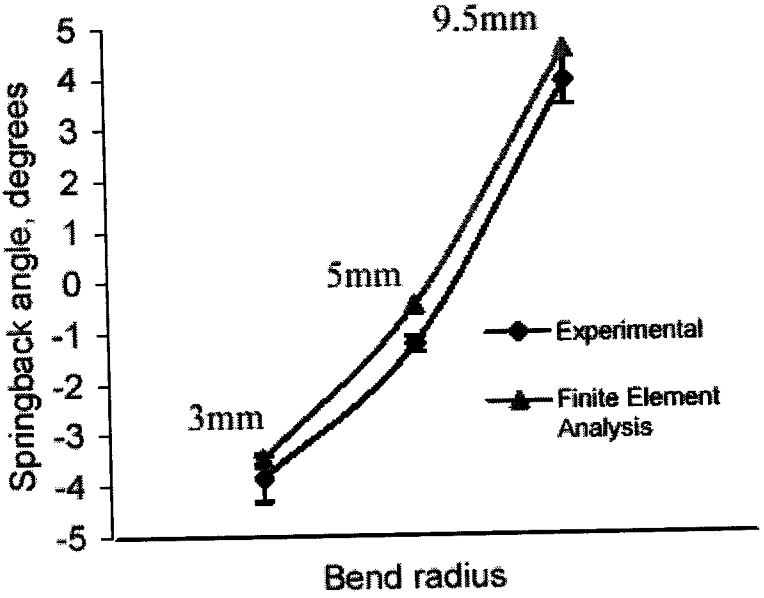


FIGURE 7.3 COMPARISON OF EXPERIMENTAL AND SIMULATION RESULTS

There appears to be only a small difference in the experimental and simulation results, the difference being less than one degree, this shows that the isotropic material model works well in the prediction of springback. Two important points should be noted in this context:

1. Figure 4.1 shows that the batch of 6022-T4 Aluminium Alloy material used for this study possesses equiaxed grains; this may well contribute to the material being isotropic.
2. The grain size of the material as seen in Figure 4.1 was observed to be 40μm, which is very small. It is presumed that the material was recrystallized and processed to render it nearly isotropic so that anomalous behavior is taken care of.

CHAPTER 8

CONCLUSIONS

Experimental measurement and finite element simulation of springback on V-bending 6022-T4 Aluminium alloy has been conducted. The relative contribution of process and material parameters on springback has been evaluated using a Taguchi Design of experiments approach. Experimental predictions showed that the bend radius has the greatest effect on springback, and it overshadows the other parameters. Spring-in or negative springback has been observed for very small bend radii. Grain size / microstructure showed very little effect on springback at 95% confidence level. It is inferred from the results that other factors including texture have negligible effect on springback when compared to bend radius.

Finite element simulation of springback using ANSYS LS-DYNA has been conducted to explore the limits regarding process control by boundary values versus material parameters. A 2-D model with nonlinear isotropic material description is considered. Simulated results are obtained for different bend radii 3mm, 5mm and 9.5mm. The Springback angles for the different bend radii are determined. The experimental results compare well and are in good agreement with the simulated results for the effect of bend radius on springback. Therefore this proves that a 2-D finite element model with multilinear isotropic material description gives a good prediction of springback.

REFERENCES

1. Cleveland, R. M. and Ghosh, A. K. (2001), Inelastic effects on springback in metals, *International Journal of Plasticity,* pp. 769-785.
2. Hsu, T.C. and Shien, I.R. (1997), Finite Element Modeling of Sheet Forming Process with Bending Effects, *Journal of Materials Processing Technology,* 63, pp. 733-737.
3. Jeong Whan Yoon, Farhang Pourboghrat, Kwansoo Chang (2002), Springback prediction for sheet metal forming process using 3D hybrid membrane/shell method, *International Journal of Mechanical Sciences,* pp. 2133-2153.
4. Kawka, M., Kakita, T. and Makinouchi, A. (1998), Simulation of multi-step sheet metal forming processes by a static explicit FEM code, *Journal of Materials Processing Technology,* 80-81, pp. 54-59.
5. Lee, S. W and Yang, D. Y. (1998), An Assessment of Numerical Parameters Influencing Springback in Explicit Finite Element Analysis of Sheet Metal Forming Process, *Journal of Materials Processing Technology,* 80-81, pp. 60-67.
6. Levy, B.S. (1984), Empirically Derived Equations for Predicting Springback in Bending, *Journal of Applied Metalworking*, 3, no. 2, pp. 135-141.
7. Li, K. P., Carden, W.P. and Wagoner, R. H. (2002), Simulation of Springback, *International Journal of Mechanical Sciences*, 44, pp.103-122.
8. Math, M. and Grizelj, B. (2002), Finite element approach in the plate bending process, *Journal of Material Processing Technology,* pp. 778-784.
9. Narkeeran Narasimhan and Michael Lovell (1999), Predicting Springback in Sheet Metal Forming: An Explicit to Implicit Sequential Solution Procedure, *Finite Elements in Analysis and Design,* 33, pp. 29-42.
10. Paulsen, F., Welo, T. (1996), Applications of Numerical Simulation in the Bending of Aluminum-Alloy Profiles, *Journal of Materials Processing Technology,* 58, pp. 274-85.
11. Rojek, J., Onate, E., Postek, E. (1998), Application of explicit FE codes to simulation of sheet and bulk metal forming processes, *Journal of Material Processing Technology,* pp. 620-627.
12. Sadhu Singh (1999), *Theory of Plasticity and Metal Forming Processes,* Khanna Publishers, New Delhi.
13. Zhang, L.C., Lu, G. and Leong, S.C. (1997), V - Shaped Sheet Forming by Deformable Punches, *Journal of Materials Processing Technology*, 63, pp. 134-139.
14. Zhang, Z.T. and Lee, D. (1995), Effect of Process Variables and Material Properties on the Springback Behavior of 2D-Draw Bending Parts, *Automotive stamping technology*, pp. 11-18.