THE UNIVERSITY OF ZAMBIA

SCHOOL OF ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

METAL FORMING ANALYSIS

BY

SAMUEL KUMWENDA

AUGUST 2003

THE UNIVERSITY OF ZAMBIA

SCHOOL OF ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

METAL FORMING ANALYSIS

BY

SAMUEL KUMWENDA

"Report submitted in partial fulfillment of the requirements for the award of the degree of Bachelor of Engineering, University of Zambia"

2003

ACKNOWLEDGEMENTS

The author wishes to put on record his grateful acknowledgements of the kind assistance and helpful advice received from a number of people during the compilation of this report.

To Dr H.M. Mwenda and Mr. C.G.Chizyuka, my supervisors, for their professional advice and guidance.

Thanks are also gratefully tendered to the workshop staff in the mechanical engineering workshop, for their assistance during experimentations.

To the good Lord God, for his inspiration and strength when the project seemed challenging.

DEDICATIONS

This report is dedicated to my mother, brothers and sisters.

TABLE OF CONTENTS

Page

CHAPTER 1: INTRODUCTION

CHAPTER 2: METAL SHEARING AND FORMING THEORY

CHAPTER 3: INSTRUMENTATION

CHAPTER 4: PRINCIPLES OF FINITE ELEMENT ANALYSIS

CHAPTER 5: JL ANALYZER

CHAPTER 6: EXPERIMENTAL DESIGN

CHAPTER 7: DISCUSSION, CONCLUSION, AND RECOMMENDATIONS

CHAPTER 6: EXPERIMENTAL DESIGN

CHAPTER 7: DISCUSSION, CONCLUSION, AND RECOMMENDATIONS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

P, f	Force
Π	3.14
D_1	Diameter of drawn cup, mm
T, S_o, T	Thickness of sheet metal
σ_{ult}	Ultimate tensile strength
F_d,	Ideal drawing load
d_m	Mean cup diameter
$\sigma_{f,m}$	Mean flow stress
d	Flange diameter
$F_{d,max}$	Maximum drawing force
d_o	Blank diameter
W_d	Drawing work, joules
K	Constant
h, h_c	Cup height
r_{ir}	Minimum bend radius ,mm
r_m	Mean radius, mm
β	Drawing ratio
R	Planner anisotropy
α	Angle of workpiece when bending force is acting, degrees
α_r	Final angle of workpiece, degrees
r_i	Internal radius, mm
l_w	Workpiece length, mm
a_1, a_2	Straight lengths of the bent part, mm
M	Bending moment, Nm
b	Width of stock, mm
σ_y	Yield strength, N/m^2
W	Die width, mm
K_s	Springback ratio

A_p	Punch area, m^2
σ_p	Stress on the punch, N/m^2
P_c	Pressure on the main cylinder of the press, N/m^2
S F	Factor of safety
τ_{ult}	Ultimate shear stress

SUMMARY

This is a project carried out to develop analysis procedures for particular metal deformation processes in the department of mechanical engineering.

It involves the analysis of force displacement as well as stress distribution and effects in the bending and deep drawing die sets in the mechanical workshop by comparing the Analytical method to the Finite Element Analysis method and the Angela Data Logging System method.

Analysis method on sheet metal drawing, bending and shearing utilizes the state of the art computer based instrumentation ANGELA and the finite element software JL ANALYSER 8.0 AND 5.0.

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Production by pressing in its many forms together with sheet metal working constitutes one of the most absorbing branches of production engineering, and one which plays an important role in our domestic life, being largely responsible for many of the comforts which have been brought within then reach of all classes.

Metal shearing is a chipless cutting operation in which the metal is stressed in shear between two cutting edges beyond its ultimate strength.

Metal forming is a manufacturing process which changes size or shape of a sheet metal by application of stresses beyond the yield strength but below the fracture strength.

1.2 OBJECTIVES

To develop analytical method and procedures to illustrate metal deformation processes and then comparing them to the data logging system Angela and the finite element software JL Analyser8.0 and 5.0.

1.3 PREVIOUS WORK DONE

Other students did the following work previously:

a) The design and manufacture of tooling for deep drawing and shearing.

b) The design and fabrication of transducer mountings.

c) Installation of Data Acquisition and Control (DAC) card into the personal computer.

d) To develop experimental tooling and procedures to illustrate metal deformation processes, with the help of the data logging system Angela.

1.4 PROJECT SCOPE

Work on this project involved:

a) Conducting experiments on shearing, deep drawing and sheet bending to be carried out on the hydraulic press.

b) Developing experimental procedures on the finite element software.

c) Analyzing Visio plasticity (or grain movement on the finite element simulation).

d) Conducting experimental procedures on the Angela data logging system.

1.5 METHODOLOGY

a) Learn metal deformation theory

b) Learn data logging procedures and the Angela system.

c) Learn the finite element method as well as the corresponding software

d) Learn the technique of Visio plasticity

e) Test metal forming processes and the instrumentation part of the hydraulic press not forgetting the operation of the potentiometer and the various tools to be used.

f) Analyzing metal deformation using the Angela system, finite element method as well as Visio plasticity.

g) Report writing.

CHAPTER 2

METAL SHEARING AND FORMING THEORY

2.1. DEEP DRAWING THEORY

Deep drawing is used to shape flat sheets into shell. The sheet of appropriate size is placed over a die and is pressed into the die cavity. The configuration of the process is shown in the figure 2.1 below

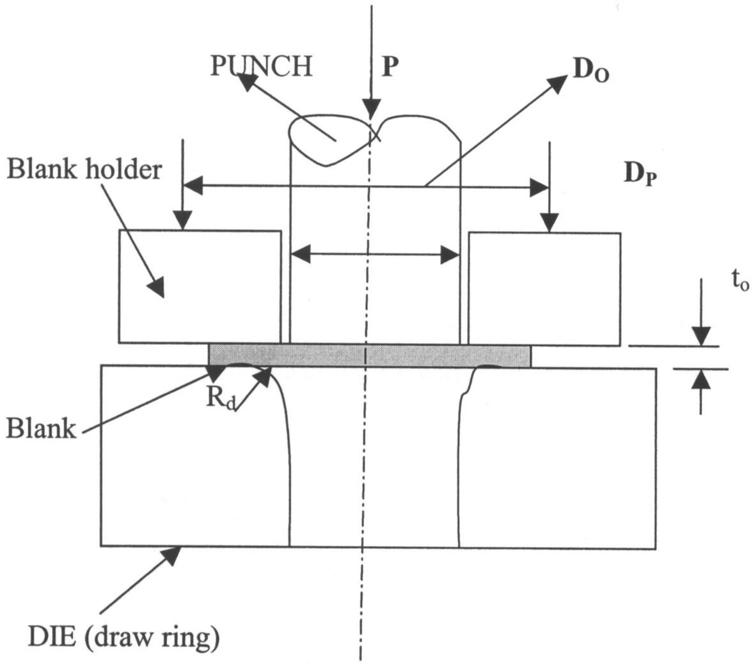


Figure 2.1: Deep drawing

2.1.1 PURE DRAWING

In pure drawing, the blank can flow freely into the die cavity. The blank holder does not exert significant restraint on the flow of the work material. Pure drawing can make parts that do not require high accuracy features or uniform wall thickness.

2.1.2 IRONING

To form uniform wall thickness sheets the ironing process is used. In this process, the punch draws a preformed sheet through a draw ring. The space between the punch and the draw ring or die is less than the initial cup wall thickness, so the wall becomes thinner, and the cup height increases. Close tolerance cups require high precision tooling and little allowable tool ware.

2.1.3 REDRAWING

This operation is done to reduce the diameter and increase the height of a previously drawn cup. This can be full length or a specific distance from the bottom.

2.2 STRESS STATE ACTING

As a punch advances, the deforming blank assumes the shape of the punch and eventually the shape of the space between the punch and the die. Depending on the part work material may be forced to deform in a complex way. Considering figure 2.2 below

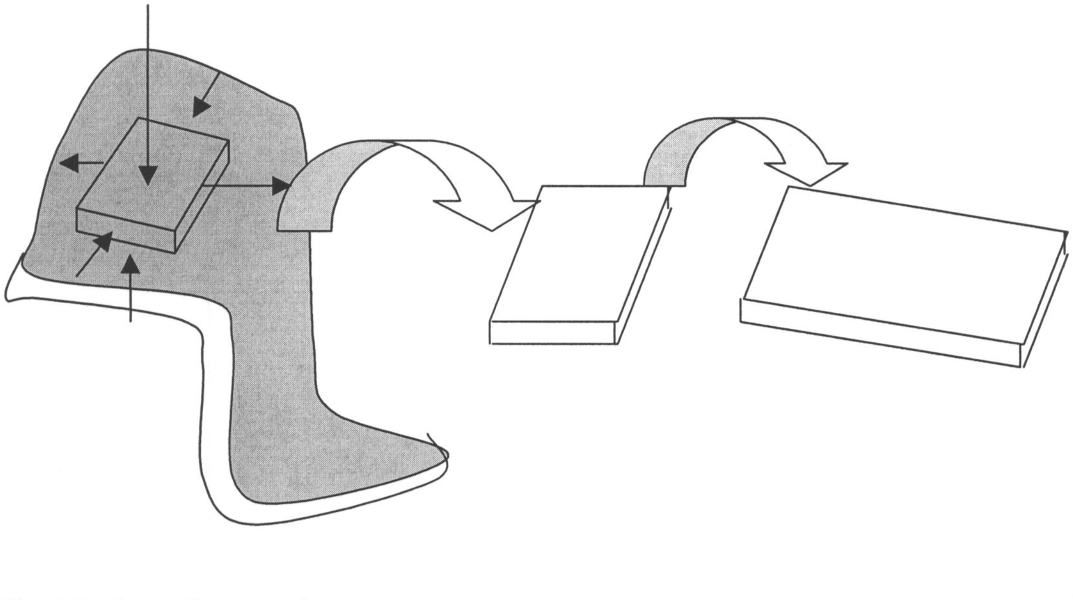


Fig. 2.2: Stress State Acting

As the cup is deformed, the flange is subjected to a tensile stress due to the punch pulling the blank into the die cavity. As the blank moves into the die, the flange diameter or the peripheral length is decreased. A compressive stress acts in the tangential or hoop direction. This compressive tangential stress can lead to wrinkling of the flange and so a blank holder is used. The blank holder force can prevent wrinkling but also gives rise to a frictional force, which restrains blank flow into the die.

2.2.1 FORCE REQUIREMENTS FOR DRAWING

In most drawing operations the metal is worked close to the ultimate strength. For this reason the ultimate strength of the material is used as the basis for estimating force requirements.

The drawing force is given as:

$$F = \pi d_1 S_O \sigma_{ULT} \text{ -- } (2.1)$$

2.2.2 LOAD-STROKE DIAGRAMS

A typical load-stroke diagram for deep drawing is shown in figure

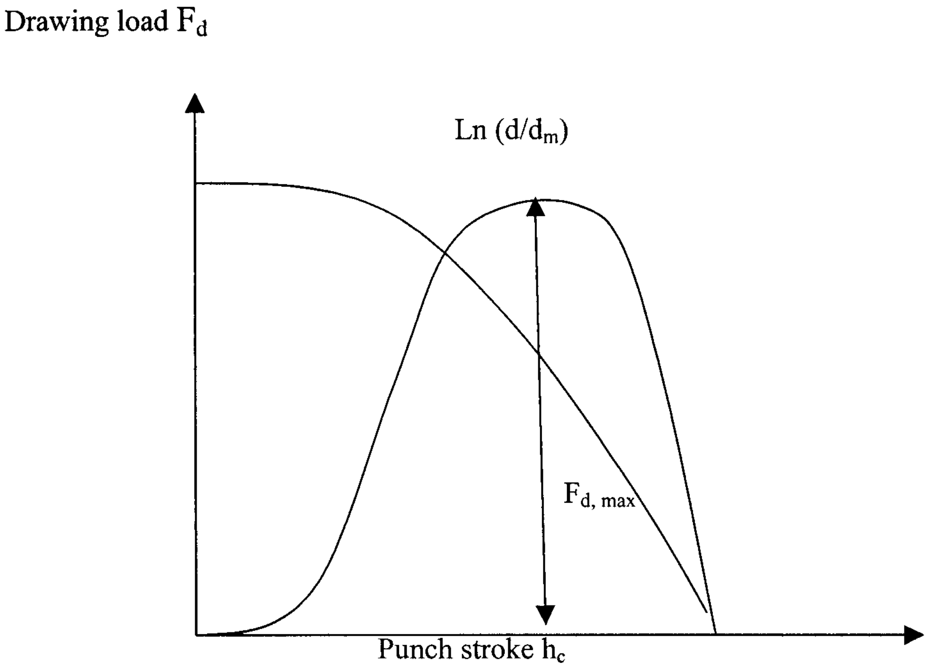


Fig. 2.3: A load – stroke diagram for First Draw

The ideal drawing load can be considered as:

$F_{d,id} = \pi d_m S_o \sigma_{f,m} \ln (d/d_m)$ --(2.2)

As shown in the figure, the flow stress increases continuously with progressing deformation because of strain hardening, while the flange diameter becomes smaller and smaller such that In (d/d_m) decreases continuously until it reaches at zero at the end of the draw. The product of these two quantities shows a distinct maximum which is reached when the ratio of the flange diameter to the blank diameters:

$d_{f,\max}/d_o \approx 0.77$ ---(2.3)

the work required W_d for deep drawing a cup is given by the area under load-stroke diagram. One can estimate the work from the maximum drawing load using a correction factor K_d as:

$W_d = KF_{d,\max} h_c$ --(2.4)

2.2.3 BLANK HOLDER FORCE

The amount of blank holder force required to prevent wrinkles and puckers is largely determined by trial and error. The force required to hold a flat blank for a cylindrical draw varies from zero to a maximum of about one-third of the drawing force.

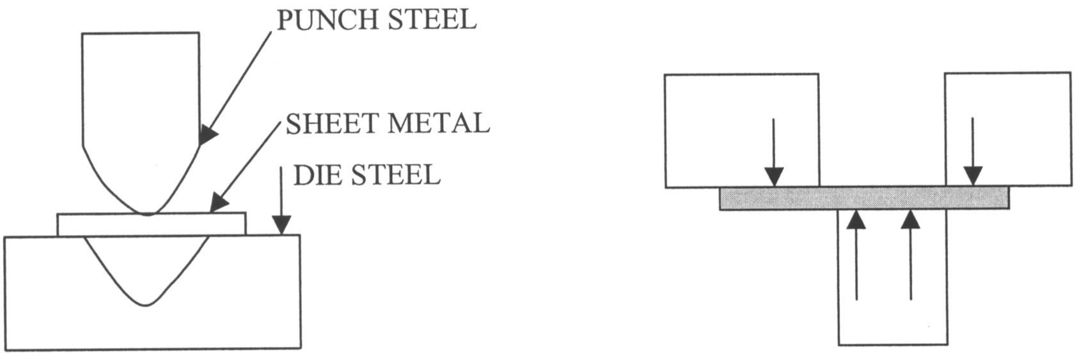
2.3 BENDING

Bending is on of the most common metal working operations. Bending sheet stock makes parts and bending also is a component of more complex sheet metal forming operations. Bending is the plastic deformation of metals about a linear axis called the bending axis with little or no change in the surface area.

The bending process is used not only to form parts such as angle sections, flanges, Seams and corrugations, but also to impart stiffness to the part by increasing its moment of inertia. Often changes in cross-sectional shape can lead to increasing section stiffness without the addition of material.

2.3.1 METHODS

There are various bending methods. Some of the most common ones are shown in figure 2.4 below:



U -DIE

V-DIE

Fig. 2.4: Bending Methods

2.3.2 STRAIN IN BENDING

Predicting the bending strain is important in bending process design and operation. Successful bending processes require:

- Producing plastic strain so that the work piece is permanently deformed.
- The strain to be less than the relevant failure strain (usually fracture strain but inside radius wrinkling can be of concern).
- Estimating the amount of springback which depends on the strain imposed

Starting with the general definition of strain that is to change in length referred to the length over which the dimensional change takes place:

$$e=(l_f - l_o)/l_o \text{ --- } (2.5)$$

Since the length of the neutral axis stays constant, the length of the neutral axis in the deformed state can be taken as l_o.

Thus

And the length of the deformed stroke, taking the outer fibers length, is:

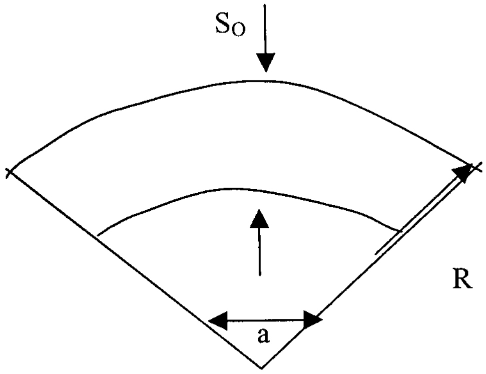


Fig. 2.5: Strain in a Bent Beam

Therefore, equation 2.5 becomes:

$$e_o = (R + S_o)\alpha - (R+S_o)/(R+S_o/2)/\alpha \text{ --- } (2.6)$$

Or

$$e_o = 1/\{(2R/S_o) + 1\} \text{ --- } (2.7)$$

The ratio R/S_o is very useful in describing sheet deformation in bending and appears in mos results and discussions in bending.

If the neutral axis shifts, toward the inner surface of the bend, then outer surface and inne surface strains will be different from this value.

2.3.3 WORKPIECE LENGTH

To calculate the length of workpiece needed to form a part empirical methods are used in industry:

Equation 2.8 is one such relationship. In this equation for blank length, the difference between the unlengthened and mean fiber is taken into account by means of a correction factor ξ given in appendix A, for different bend radii r_i and bend angles α.

$$L_W = a_1 + \pi/180\alpha(r_i + (S_o/2)\,\xi) + a_2 \text{ --- } (2.8)$$

2.3.4 MINIMUM BEND RADIUS

The minimum bend radius r_i is the smallest radius to which a material can be bent without cracking or weakening.

Referring to equation 2.11, we see that, as the ratio R/S_O decreases, tensile strain at the outer sheet surface increases. When the strain reaches the material fracture strain, further bending will cause material separation. The minimum radius is usually determined by the outer surface fracture.

The values of the bend radii for different steels is given in Table 2.1

Table 2.1 Table of minimum bend radii for materials of different hardness

MATERIAL	RADIUS
Dead-soft drawing quality steel	1/2T
1/4 hard temper	11/2T
1/2 hard temper	3T
3/4 hard temper	4 to 5T
Full hard temper	6 to 8T

When bending parallel grain at least twice the above values should be used.

2.3.5 BENDING FORCE

When a rectangular bar beyond its yield point, the bending moment is:

$M=b(S_o)^2\sigma_Y/4$ ---(2.9)

In the case of a V-shaped die centrally loaded, the moment is:

$M=P/2*W/2=PW/4$ ---(2.10)

Equating equations 2.9 and 2.10, we have:

$PW/4=b(S_o)^2\sigma_Y/4$ ---(2.11)

Also using the ultimate strength σ_u, since the values of ultimate strength in tension are more common. Moreover, to allow for dull cutting edges and for friction by the factor k we the have the bending load as:

$P=Kb(S_o)\sigma_u/W$ ---(2.12)

2.3.6 BENDING DIRECTION WITH RESPECT TO ANISOTROPY

Cold rolling of sheets produces anisotropy because of alignment of impurities, inclusions, and voids and this is called mechanical fibering. As shown in figure 2.6 below:

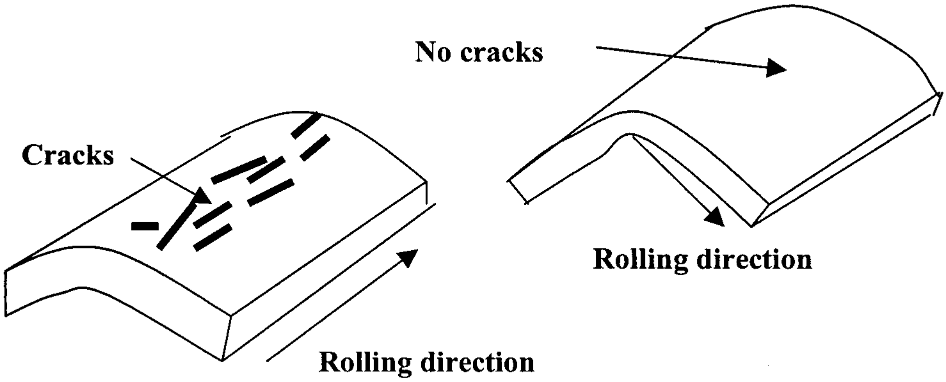


Fig. 2.6: The effects of Rolling Direction on Bending

When metal is bent parallel to the rolling direction, cracking sometimes occurs. To avoid this, bending lines should be perpendicular to the rolling direction.

2.3.7 SPRINGBACK AND COMPENSATION

Due to elastic forces, material fibers " spring back" from the profile to which it has been formed when the force is withdrawn.

Springback can be characterized by the springback ratio

$$K_S = \alpha_r/\alpha \tag{2.13}$$

$K_S = 1$ indicates no springback while $K_S = 0$ indicates complete elastic recovery.

Some variables and their effects on springback are as follows:

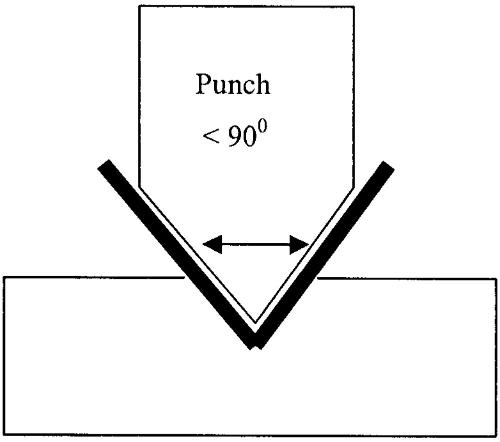
- Harder sheet metals have greater degrees of springback due to there high yield strength and the resulting larger elastic bands in the bends.
- A sharper or smaller bend radius reduces springback by creating a larger plastic zone.
- Thicker sheets have less spring back because more plastic deformation occurs for the same die radius.

Springback must be compensated for in bending operations; otherwise, the operation would not yield parts of desired dimensions and shape.

Several methods are used to overcome or counteract the effects of springback:

- **Over - bending**

Over bending of the formed area is one means of compensating. The method is illustrated in figure 2.7 below:



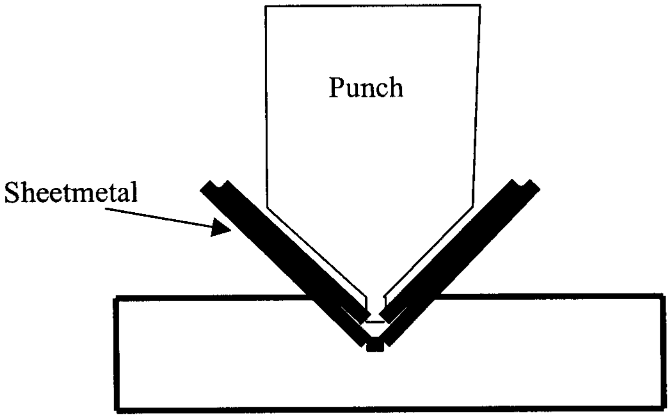
Over - bending

Fig. 2.7: Over - bending

This method was used in the development of the die in the previous project.

- **Bottoming or Setting**

Large short-time loading or impact type loading applied at the bottom of the bend produces high compressive stresses that set or hold the bend. Bottoming is illustrated in the figure 2.9:

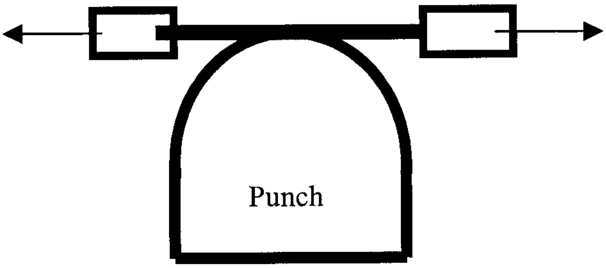


Bottoming

Fig. 2.8: Bottoming

- **Stretch Bending**

In stretch bending, the sheet is first stretched so that the blank is stressed past the yield strength. The sheet is then forced over the punch to obtain the desired contour. This addition of strain may be necessary if a small strain is to be produced in the formed sheet. Figure 2.9 illustrates stretch bending



Stretch bending

Figure 2.9: stretch bending

Only relatively large radii are bent by this method because the stresses added in bending to sharp radii would take the pre - stressed metal beyond its ultimate tensile strength.

2.3.8 MECHANISM OF SHEARING

Shearing includes three successive phases:
The elastic phase during which sheet metal is compressed across and slightly deformed between the punch and the die, the stress and deformation in the material do not exceed the elastic limit.

The plastic phase in which the punch penetrates into the material to a certain depth, pressing it into the hollow of the die with bending and stretching of the material in the clearance region. The deformation of the material becomes permanent and the stress exceeds the yield strength of the work material and increases steadily with further punch motion. At the end of this phase, the stress in the material close to the cutting edges reaches a value corresponding to the material shear strength.

The fracture phase during which the strain in the material reaches the fracture limit, microcracks, and the separation of the parts of the workpeice occurs. The cracks in the material start from the cutting edges and propagate along the slip planes until complete separation of part from sheet occurs.

2.3.9 SHEARING OPERATIONS

The various operations can be classified as follows:

a) **Punching (piercing):** in which the focus is on the hole left in the stock, and the slug is waste.
b) **Blanking:** in which the slug is the blank to be further processed.
c) **Die cutting:** which consists of the following operation used to produce more complex blanks.
    I. **Perforation;** or punching a number of holes in a sheet.

II. **Parting:** This is the separation of blanks by cutting away a strip of material between them. It can be done after notching or lancing has developed most of the part outline.

III. **Notching:** involves removing pieces of various shapes from the edges. This can be done to remove excess metal before forming.

IV. **Lancing:** in which a single line cut or slit is made part way across the strip stoke without removing any metal.

2.3.10 DIE CLEARANCE

Clearance refers to the space between the punch and die. Clearance is the most important factor determining the shape and quality of the sheared edge. When clearance increase, the zone of deformation becomes larger and the edges become rougher. The material is pulled into the clearance area, and the edges of the sheared zone more and more rounded.

Die clearance ranges from 4.5 to 7.5% of the stock thickness

CHAPTER 3

INSTRUMENTATION

3.1 INTRODUCTION

Computer based data acquisition and control systems are widely used in industry and laboratory application for monitoring, controlling, data acquisition and automated testing.

The system is usually made up of the following:

- Transducers and actuators signal conditioners
- Data acquisition and control hardware.
- The flow of data and the interconnection of these elements are shown in the figure 3.1 below.

The flow of data and the interconnection of these elements are shown in figure 3.1 below:

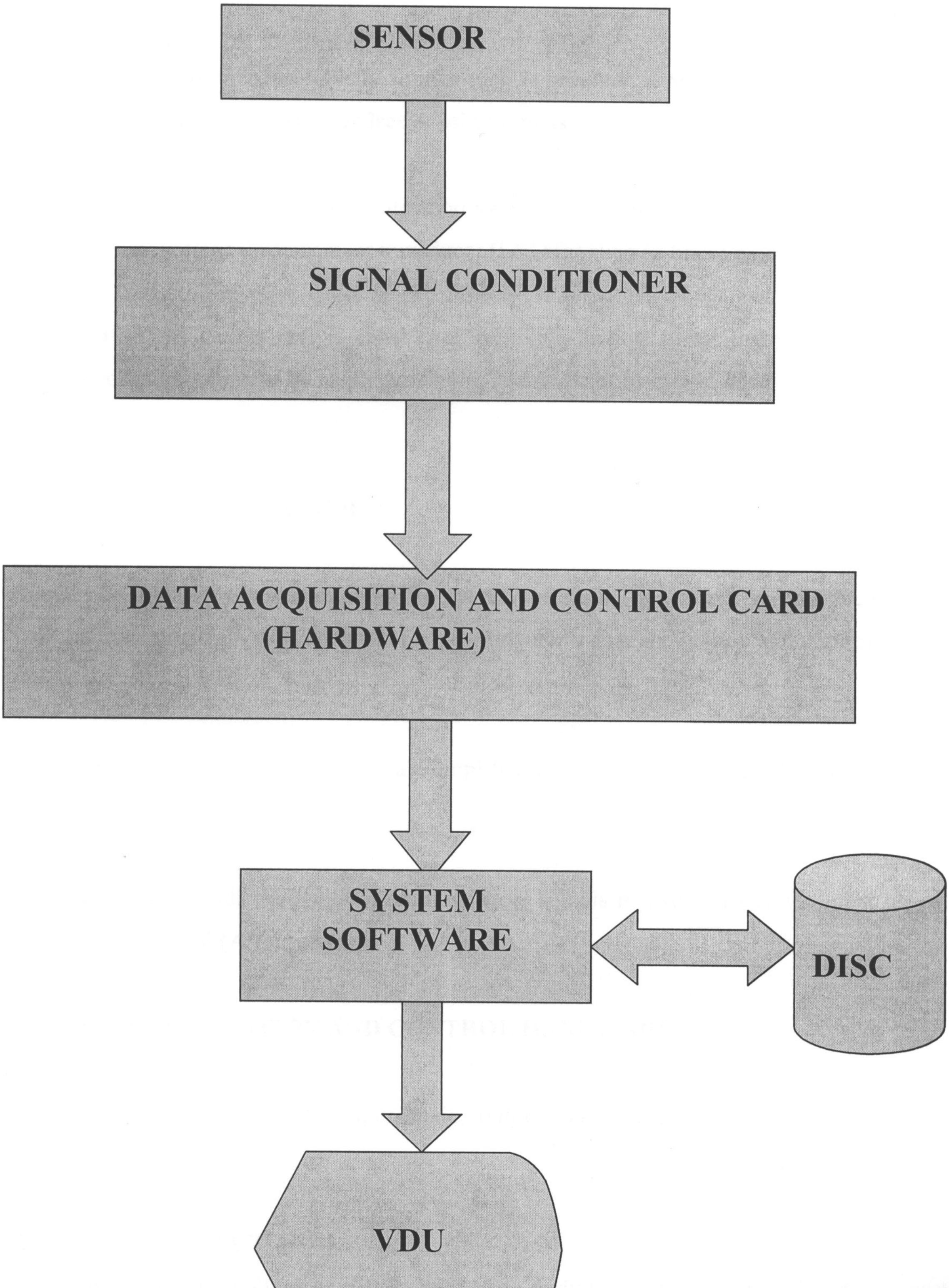


Figure 3.1: Flow of information in a data logging system

3.1.2 TRANSDUCER AND ACTUATORS

A transducer converts measurable quantities such as pressure, level, length, position, etc. Into voltage, current, frequency, pulses or other signals.

Common examples of transducers are thermocouples, strain gauges, pressure sensors, load cells, and low voltage displacement transducer (LVDTs) such as linear potentiometers.

An actuator is a device that is capable of receiving analog signal instructions to activate process control equipment by using pneumatic, hydraulic or electrical power.

3.1.3 SIGNAL CONDITIONER

Signal conditioning circuits improve the quality of signal generated by transducers before they are converted into digital signals by the computer's data acquisition and control (DA&C) hardware.

Common among signal conditioning are amplification, signal scaling, excitation, attenuation, etc.

Amplification expands the range of the transducer signals that they match the input range of the analog to digital (A/D) converter.

3.1.4 DATA ACQUISITION AND CONTROL HARDWARE

This hardware performs: analog input, analog output and counter/timer functions.

3.1.5 ANALOG INPUT A/D

A/D converts analog voltage or current levels into digital information, to enable the computer store or process the signals.

3.1.6 ANALOG OUTPUT D/A

The operation converts digital information into analog voltage or current. This allows the computer to control real world events.

3.1.7 DIGITAL INPUT AND OUTPUT I/O

These accept digital input and are capable of outputting digital information. They are used for switching and in communication.

3.1.8 COUNTER/ TIMER

This does the frequency counting, time period measuring etc.

3.1.9 SOFTWARE

These are the programs that run the DA&C system. This can be done at package level, driver level or hardware level. The system drivers support a wide range of popular programming languages including:

C: Microsoft C/C_{++}, Boland C/C_{++}, and Turbo C/C_{++}.

Basic: Microsoft Quick Basic V.4.0 and 4.5, GWBASIC versions 2.02 and 3.20.

Pascal; Turbo Pascal.

3.1.10 THE ANGELA MEASURING SYSTEM

This Angela measuring system is a universal system for measuring with different kinds of sensors. It mainly consists of:

- Sensors
- The MULTILAB Bus
- ADVANTEC DA&C Card installed in the personal computer Angela software.

CHAPTER 4

PRINCIPLES OF FINITE ELEMENT ANALYSIS

4.1 INTRODUCTION

The finite element method is a powerful numerical technique available today for the analysis of complex structure and mechanical systems. It is to obtain numerical solutions to a wide range of problems. Finite Element Analysis (FEA) consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in product design, and existing product refinement. There are generally two types of analysis that are used in industry; 2-D modeling and 3-D MODELING. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions that make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and are capable of testing a material all the way to fracture.

Mathematically, the structure is subdivided into a mesh of finite sized elements of simple shape. Within each element, the variation of the displacement was assumed to be determined by simple polynomial shape function and nodal displacements. Equations for the strains and stresses are developed in terms of the unknown nodal displacement. From this, the equations of equilibrium are assembled in a matrix form, which can be easily programmed and solved on a computer. After applying the appropriate boundary conditions, the nodal displacements are found by solving the matrix stiffness equations. Once the nodal displacements are known element stresses and strains can be calculated.

A complete study of the finite element methods was beyond the scope of this project. However, despite the proliferation and power of commercial software it was essential to have an understanding of the technique (F.E.A), so that an appropriate and accurate analysis model could be selected, correctly defined and interpreted.

4.2 MESHING

4.2.1 MESH SPECIFICATION

Meshing is the procedure of applying a finite number of elements to a model. In order to conduct a finite element analysis the structure must be first idealized into some form of mesh. The art of successful application of the technique, as far as the user is concerned, lies in the combined choice of element type and associated mesh. As the method is approximate, it is necessary that the user has a good idea of the form of the solution, together with an understanding of the consequences of the assumptions made within the element types to be used.

4.2.2 MESH GENERATION

The element type that is to be used and the expected stress distribution throughout the structure dictate the choice of the mesh density. Any finite element is only approximate and the inherent approximations must be considered when the mesh is being generated. Some elements, especially plates and shells, do not satisfy compatibility internally thus leading to structural displacement divergence and meaningless results are obtained. Complete compatibility will only be satisfied if elements with the same interpolation function, at least along the common edge are used.

4.2.3 MESH DENSITY

The art of using Finite Element Method (FEM) lies in choosing the correct mesh density required to solve a problem. If the mesh is too coarse, then the inherent element will not allow a correct solution to be obtained. Alternatively, if the mesh is too fine, the cost of analysis in computing time can be out of proportion to the results obtained. In order to define a relevant mesh, some idea of the tress distribution within the component is required. If the answer is known, then a good mesh can be defined. A fine mesh is required where there are high rates of change of stress and strain and coarse mesh is sufficient in area of reasonably constant stress. This still begs the question as to what constitutes a fine mesh. A linear displacement element requires a finer mesh than a parabolic (quadratic), which in turn requires a finer mesh than a cubic element. It follows that any user of a finite element package must have some knowledge of structural analysis and the manner in which a given structure behaves. He/she must be able to identify regions of stress concentration and able to estimate die away lengths associated with a given form of discontinuity (a geometrical feature of the component being analyzed, which produces high rates of changes of stress, e.g. sharp changes in cross section etc.).

Every element is defined in terms of the basic shape of a parent element. Due to the geometry of a component, this element may become slightly distorted to fit within the boundary of the drawn component model. When elements are distorted from their parent shape they are found to be less accurate. As the distortion is increased, the greater the errors in the element behavior after the loads are applied. Therefore, in setting up a mesh, the user should attempt to keep the element as near to the basic parent element shape as possible.

4.3 ASSEMBLY AND SOLUTION

Once the mesh for a finite element analysis has been constructed, then the analysis can be performed. This requires much less input from the user than the mesh preparation stage as the computer does this work.

4.4 EXCESSIVE ELEMENT DISTORTION

On viewing the element displacement on the graphic user interface, the formulation of all finite elements will ultimately be based on the assumption of a basic shape for the element (e.g. For a triangular element, the basic shape is an isosceles triangle). As the element is distorted from the basic shape within the component, then possible errors occur in the associated mapping. The more distorted the element is from the parent element shape, after the displacement due to the load is experienced, the less it will model the behavior of the actual element, since higher order ones (parabolic, cubic etc.) are generally more tolerant to distortions. There are various methods in use for measuring these distortions, some of which are general and some are particular to certain element.

4.5 A TYPICAL SESSION

Finite element analysis provides a reliable numerical technique for analyzing engineering designs. The process starts with the creation of geometric model in one of the CAD packages available commercially. Then the model is exported to a pre-processor (usually via SAT or IGES file format) and is sub-divided (meshed) into small pieces (elements) of simple shapes connected at specific node points. The process assumes that the behavior of each element varies in a particular, known fashion for various conditions. The FEM predicts the behavior of the model by manipulating the information obtained from the element making up the model. Meshing is a crucial step in FEA. As mentioned above, in meshing, the program breaks up the geometric model into a number of interconnected elements. Usually an automatic mesher performs this task. Thus, a network of discrete elements connected at points called nodes represent the continuous model.

The size of the element generated by the auto-mesher depends on the geometric dimensions of the model and the degree of analysis accuracy desired. In the early stages of the process, a ballpark result may suffice, but if a designer wants real information affecting life span of a product or the safety of its operation, high accuracy results will always be needed.

The success of any analysis depends on the how closely the geometry, material behavior, and boundary conditions of the design in its actual operating conditions approximate reality. Figure 4.1 shows the process of finite element analysis.

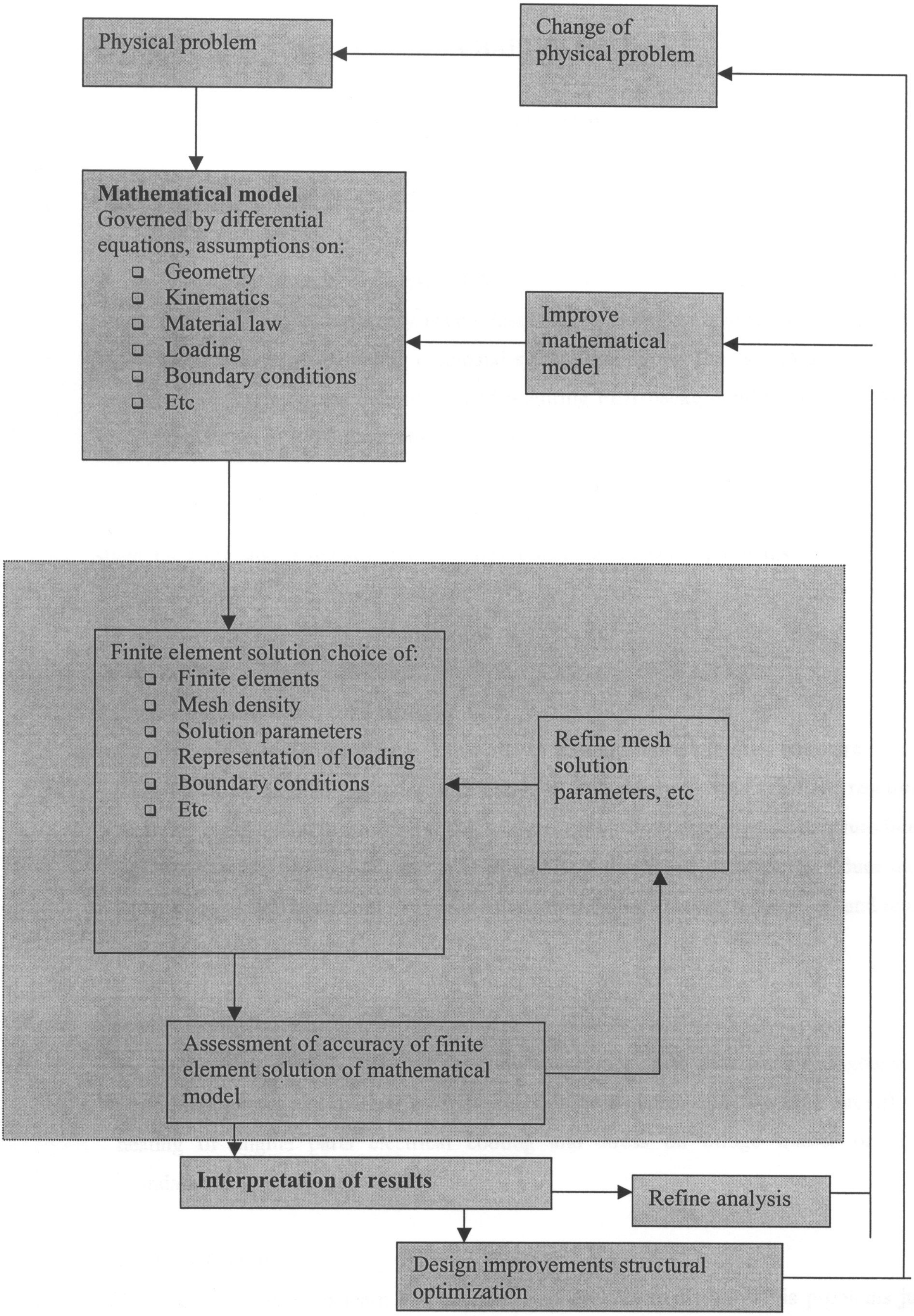


Fig. 4.1: The Process of Finite Element

CHAPTER 5

JL ANALYZER

5.1 INTRODUCTION

JL Analyzer is an engineer's analysis FEA software that offers a complete aid through three simple processes: Modeling, Analysis and Post Process. Based on engineering common sense, it developed a series of operating command system that allows the user to use the program without remembering many commands and requiring FEA background because JL Analyzer has a complete set of FEM functions that any designer may come to rely on.

JL Analyzer combines a full solid modeling with an advanced finite element analyzer to solve static, buckling, and frequency, dynamic, heat transfer and electric problems.

5.2 ADVANTAGES OF JL ANALYZER

(a) Cost effective and time efficient

JL Analyzer is an engineering tool that allows an engineer to subject a structure to various loads (such as thermal, vibration and pressure) and determine the resultant reaction. JL Analyzer enables one to quickly model, analyze, check for feasibility and structure integrity and redesign. The greatest advantage of using that is that it can significantly reduce the cost and save tremendous amount of time involved in building, testing, redesigning and retesting prototypes until optimized.

(b) Wide application

JL Analyzer is used extensively in civil, mechanical and aeronautical engineering to perform structural and thermal analysis, such as aerodynamic effects on the aircraft wings, heating of engine parts electrical cooling and stress on bridge trusses or building foundations.

(c) User friendly

JL Analyzer is easy to learn and use. An engineer can easily solve his problems just by following the menu on the screen step by step. It does not require the user to remember commands. The online help for each command enables the user to understand the command.

Through these three steps, an engineering solution toward any problem can easily and quickly be obtained.

5.3 JL ANALYZER PROGRAM FUNCTIONS

JL Analyzer FEA session consists of three main stages, namely the Modeling (pre - processing) stage, the analysis (solution) stage and the Post processing stage.

5.4 PROCESSES

5.4.1 STEP A: MODELING

JL Analyzer has a well- defined and thoroughly tested menu-command system in both structural and logical. It is designed based on the common sense if engineers, and with its open-menu system, the user can easily obtain an intuitive understanding of the whole system. One line helps the user understand the highlighted command and also give more explanation for selected command in the communication window. Unlimited asking analysis function allows the user to click on analysis that shows you what stage of current process. If modeling is not complete, it shows the next step.

JL Analyzer models an engineering problem into four simple steps, which are called Modeling 1-2-3-4.

1. Geometry Modeling
2. Mesh Generation
3. Assigning Properties
4. Setting Boundary Condition and Loads

TABLE 5.1: PROCESSES

PROCESS
Geometry Mesh Property BC/Load Initial Condition Exam Results

THE FIRST STEP: Geometry Modeling allows the user to create and modify geometry model. JL Analyzer has a comprehensive set of geometry modeling tools to help users to construct a drawing of geometry. JL Analyzer geometry modeler is feature parametric technique.

THE SECOND STEP: Mesh Generation allows the user to generate mesh on the geometry to perform finite element analysis. JL Analyzer provides its user with automatic mesh generation.

THE THIRD STEP: Assigning Properties allows the user to assign material and geometric properties. JL Analyzer has included a predefined material library containing common materials. The users may add their own materials into this library.

THE FOURTH STEP: Setting Boundary Conditions and Loads allows the user to define constraints and loads, such as Joints, predefined displacements, multiple constraints, force, pressure, varying pressure, temperature gravity, centrifugal, thermal heat flow, heat generation, convection, radiation, seepage and electrical boundary conditions. The boundary conditions are second most critical stage of modeling (element type is first).

5.4 .2 STEP B: ANALYSIS

Analysis Overview

JL ANALYZER SUPPORTS THE FOLLOWING ANALYSIS MODULES

Ø Analysis Status
Ø Static Analysis
Ø Frequency Analysis
Ø Buckling Analysis
Ø Dynamic Analysis
Ø Dynamic Stress Calculation
Ø Response Spectrum Analysis
Ø Thermal (steady and transient) Analysis
Ø Seepage Analysis
Ø Electric Conductive Analysis
Ø Nonlinear Static Analysis

Ø Analysis Window

In this project, much emphasis was on stress analysis. Stress analysis deals with the Von Mises stress as well as the X, Y and Z stress.

5.5 DESCRIPTION OF THE PROBLEM

To perform the stress analysis on the sheet metal by the bending experiment, the following A-B-C steps were done. The mild steel sheet metal of dimension 12cm x 7cm and of thickness 1.6mm was utilized. The punch is constrained on all the surfaces. The stress distribution and its mode shapes were calculated as follows.

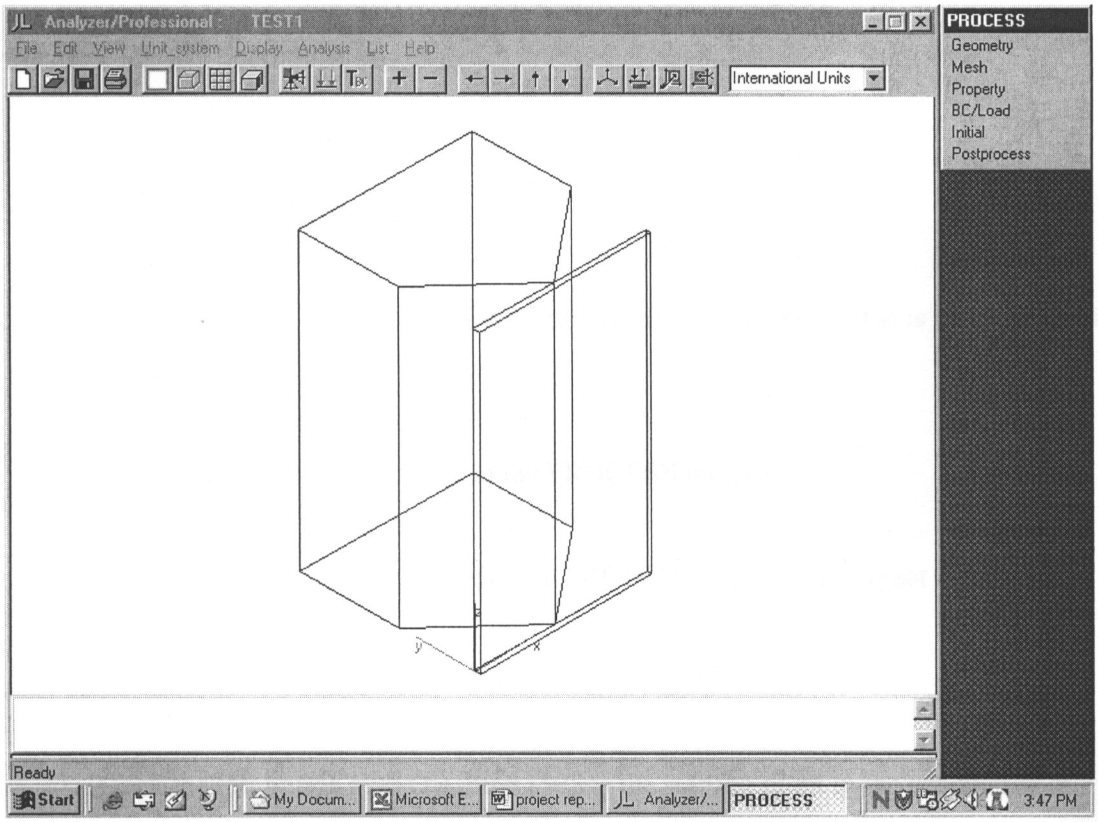


Fig. 5.1: Wire frame presentation of the Punch and Blank

Modeling 1-2-3-4 will now be used to model this problem

5.6 STEP A: MODELING

Modeling 1-2-3-4 are the 4 steps that are good for modeling any problem:

Step 1: geometry modeling

Step 2: mesh modeling

Step 3: assign properties

Step 4: boundary conditions and loads

STEP 1: GEOMETRY MODELING

This section shows how to create a 3D bending diagram in the sketch that is basic in geometry modeling.

1. Select the geometry command in the PROCESS menu.
2. Select the sketch command in the GEOMETRY menu (input project name).
3. Select the polyline command in the SKETCH menu.
4. Click the left button at the position (50) as the starting point in the production of a 2D drawing.
5. Select the DONE command in the sketch menu (to finish the sketch and then save the data to the data base).

6. Select the EXTRUDE command under the sketch command and then in the small input box enter the distance through which you want to extrude the 2D diagram and in this case it is 120mm.

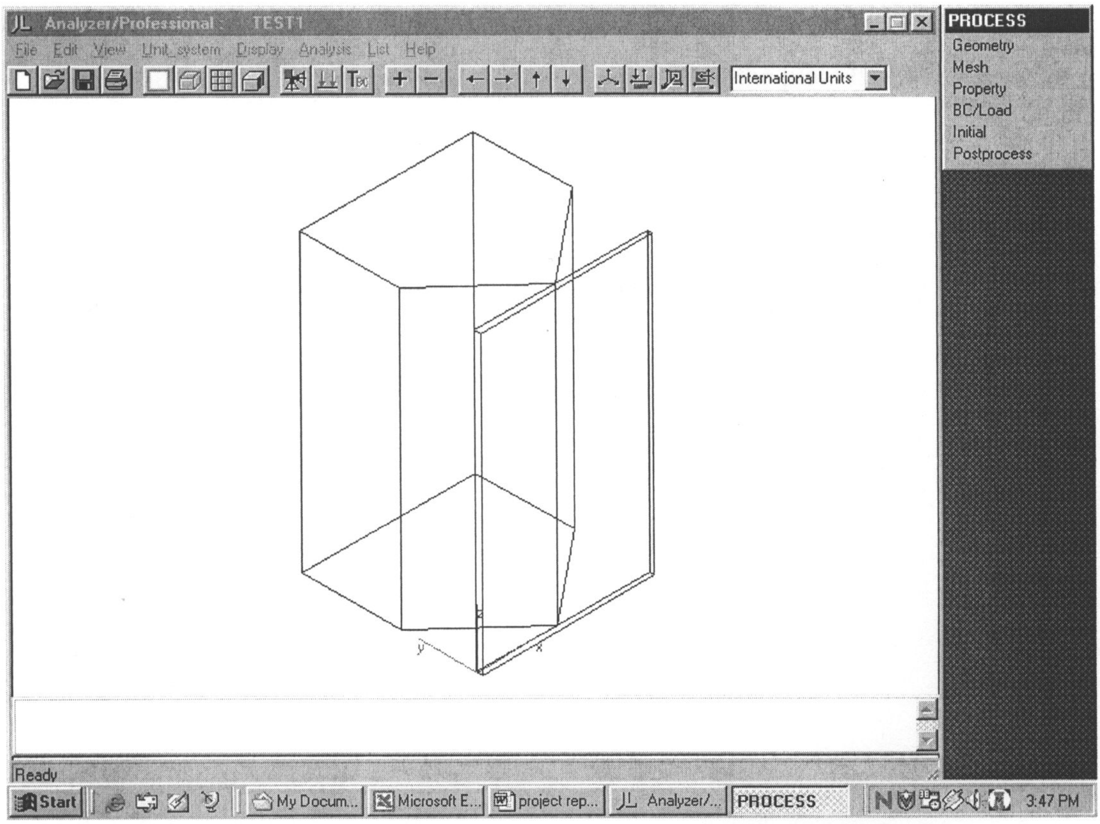
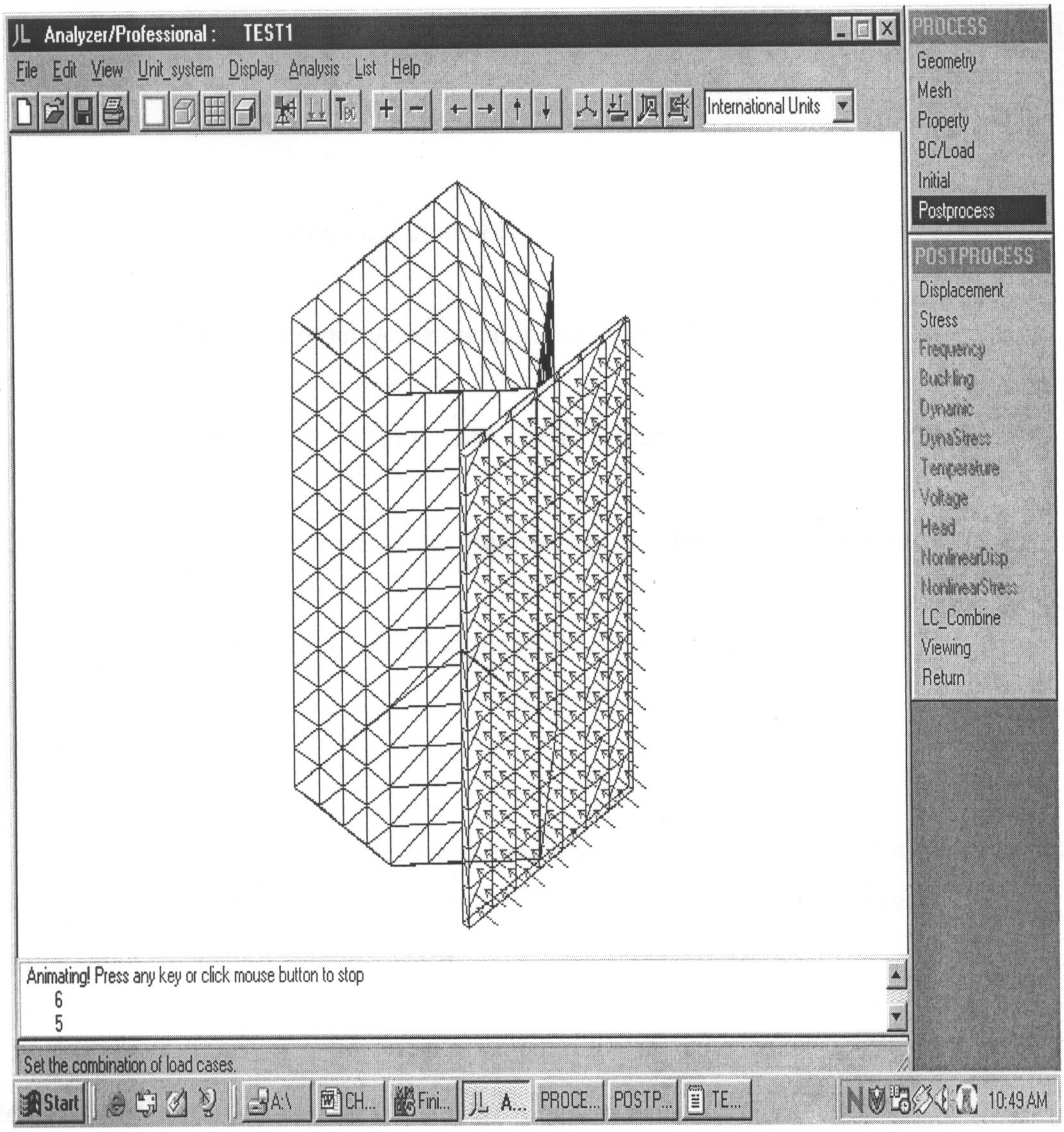


Fig. 5.2: Punch and the sheet metal in contact as shown on the screen.

STEP 2: MESH GENERATION.

Step 2 is to generate the mesh. JL analyzer supports automatic mesh generation. It requires the user to select the element type and input mesh size.

1. Select the MESH command in the PROCESS menu.
2. Select the generate command in the MESH menu.
3. Select the plane stress command in the MESH plate menu.
4. Select the Quad 4, Whole part and Accept commands in the plate ELEMENT menu
5. Input mesh length 1.0 in the dialog box.
6. Click OK then the mesh is shown on the screen (Figure 5.3)



```
Animating! Press any key or click mouse button to stop
    6
    5
```

```
Set the combination of load cases.
```

Fig. 5.3: Punch and the sheet metal as shown after meshing.

STEP 3: ASSIGN PROPERTIES

MATERIAL PROPERTY

JL Analyzer has an internal material library that allows the user to select a material by the select command. The user also can define his/her own material property by the Define command to input the data of the material and then can save it to the internal material library.

1. Select the Property command in the PROCESS menu.
2. Select the Material Property command in the PROPERTY menu
3. Select the Select command in the MATERIAL menu
4. Click on the MILD STEEL in the Material Library list box, and then click OK.
5. Select the Whole part command in the M ASSIGN menu.

Note:

The material property parameters are shown in another list box. The user also may select the unit system that allows you to check the parameters in different unit system. This would not change defined unit system.

GEOMETRY PROPERTY

Assign geometry property is required for plane stress problem that is to input the plate thickness by the thickness command:

1. Select the geometry property command in the PROPERTY menu
2. Select the thickness command in the GPROP menu
3. Select the Whole Part command in the G SHELL menu.
4. Input the plate thickness of 1 in the dialog box, and then click OK.

STEP 4: DEFINE BOUNDARY CONDITIONS AND LOADS

CONSTRAINTS: Fixed on all the surfaces of the punch.

Defining constraint is critical. Different constraints will give different solutions. One rule of thumb for defining constraints is to be sure that the problem is stable, and no free motion.

1. Select the BC/Load command in the PROCESS menu
2. Select the Constraint command in the BC/Load menu
3. Select the Define command in the CONSTRAINT menu.
4. Select the Surface, Rigid joint and Accept commands in the CNST DEFINE menu.
5. Pick a particular surface by picking the two lines defining that surface.
6. Right click to finish the command.

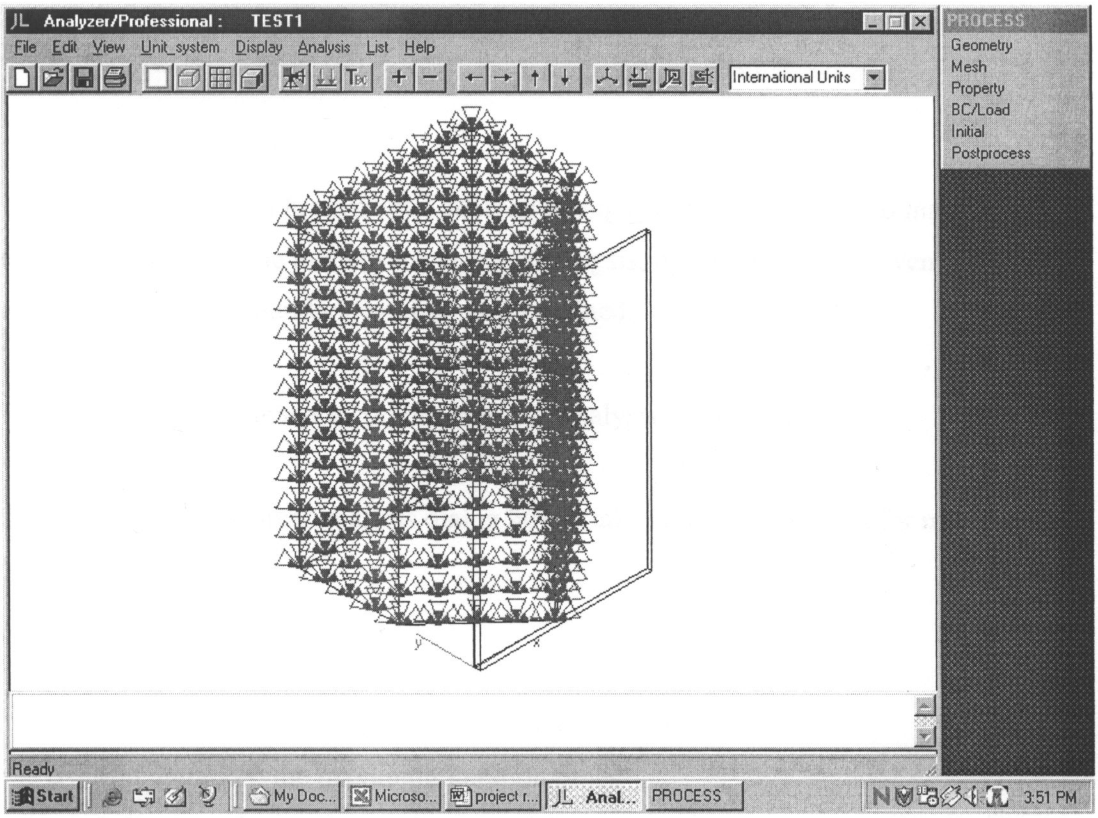


Fig. 5.4: Defining the constraints

Note:

1. Before running analysis, the program automatically checks the necessary conditions. If some basic requirements are missing, the program will ask the user to define it that is called Asking Analysis in the JL Analyzer. The user can use Ask Analysis feature any time to check if you defined all necessary conditions or what is the next step.

5.7 STEP B ANALYSIS

The stress analysis is a fundamental analysis in engineering. The user also may run other analysis from this same model such as static analysis, dynamic analysis, even thermal analysis, if thermal boundary conditions are defined.

1. Select the stress analysis command in the Analysis of main menu

2. Click OK in the stress analysis dialog box to take all default options for most users.

- Eigen values NO; the number of engine values to calculate
- Max iteration: the maximum iteration times
- Tolerance: the converge tolerance of the iteration
- Mode shift: the flag of the mode shape output in the output file.
- Print details: the flag of printing details in output file
- Stiffening: the flag of stiffening effects.
- P-delta: the flag of p- delta effects

Note:

1. The subspace iteration method is used for the stress analysis.

2. The maximum number of the Eigen values to calculate is 50.

3. The default value of the maximum iteration time is 15.

4. The default value of converge tolerance is 0.0001.

5. The flag of the mode shape is for writing the mode shape in the output file. The output file is a text file, which allows user to read and edit.

6. Stiffening is the flag of stiffening effect of pre-loading problem. The results of per-loading analysis are saved in the load case 1. this stiffening will automatically be added to other load case if this flag is on.

7. P-delta is the flag of modifying the nodal coordinates by the pre-loading displacement results. The pre-loading analysis results are always saved in the load case (1).

8. The program will automatically enter into Analysis window, then back to the graphics window once the calculation is completed.

5.8 STEP C: EXAMINE RESULTS

Examining results is unique for every analysis. The commands Plot, Animate, At node, On edge, On surface are the commands in the post process that allow the user to view the solution in contour plot, animation and at any particular node, edge or surface.

Examine stress

Contour plot

To check the stress in contour plot

1. Select the stress command in the RESULTS menu.

2. Select the Plot command in the STRESS menu.

3. Select the Deformed, Y displacement and Accept commands in the STRESS PLOT menu.

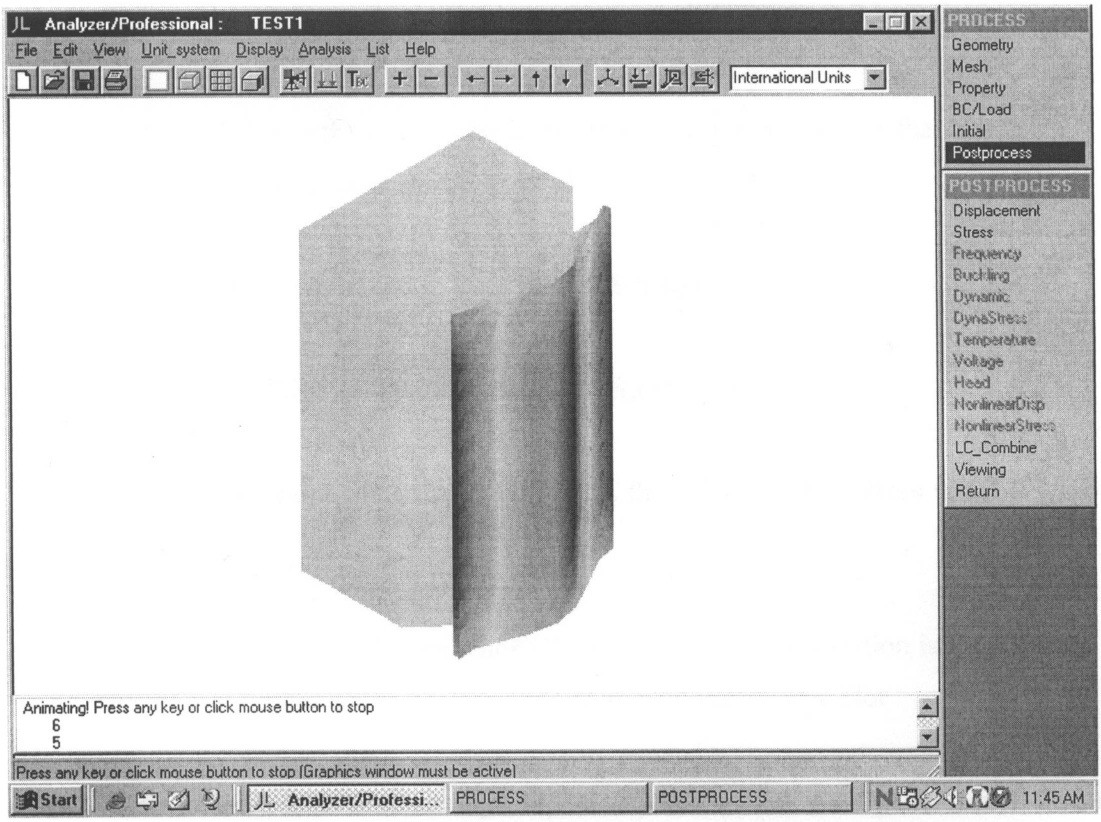
Note:

1. Rainbow color is shown grades of stress (figure 5.5)
2. Select the List Results menu to check or sort the solutions

Animate

To show stress animation:

1. Select the stress command in the RESULTS menu.
2. Select the Animate command in the STRESS menu.
3. Select the Deformed Y displacement and Accept command in the STRESS PLOT menu.
4. Click the left button to STOP the animation.



Animating! Press any key or click mouse button to stop
6
5

Press any key or click mouse button to stop (Graphics window must be active)

Fig. 5.5: Stress contour plot for mode shape <1>

At node

This command allows the user to check the stress ant any node.

1. Select the Stress command in the RESULTS menu.

2. Select the At Node command in the stress menu.

3. Now the element nodes are shown on screen, pick a node then its stresses and reaction forces are shown in the communication window.

4. Repeat step 3 for the next node until clicking the right button to finish.

On Edge

This command allows the user to check the stress solution on an edge that stress distribution along an edge is shown in xy plot.

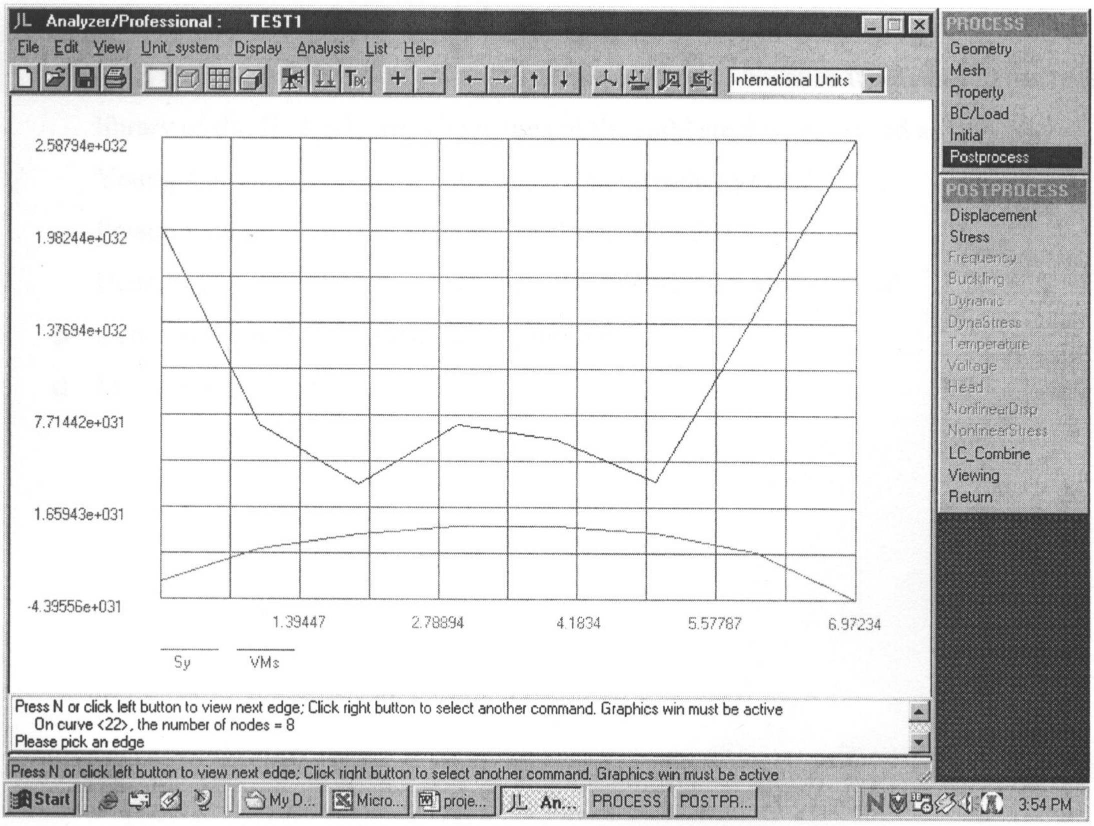
1. Select the stress command in the RESULTS menu.

2. Select the On Edge command in the STRESS menu.

3. Now the wire frame plots shown on screen, then pick an edge, stress variable window is shown up.

4. Select the stress variables and click OK, then the stress distribution is shown and (figure 5.6) each color represents one variable on the xy type plot

5. Click the left button for next edge, repeat step and 4, click the right button to finish.



```
Press N or click left button to view next edge; Click right button to select another command. Graphics win must be active
   On curve <22>, the number of nodes = 8
Please pick an edge
```

```
Press N or click left button to view next edge; Click right button to select another command. Graphics win must be active
```

Fig. 5.6: Stress distribution for an edge.

5.9 USE OF THE JL ANALYZER SOFTWARE TO INVESTIGATE THE EFFECTS OF STRESS ON THE BLANK UNDER BENDING AND DEEP DRAWING.

This section covers the study that was carried out to investigate the effects of stress on bending and deep drawing of the blank. Stress analysis was used to look at the various mode shapes and the stress levels. The procedure that was followed is similar to that given in the last section.

Note:

- 2D modeling analysis was used in deep drawing and 3D in bending
- The co-ordinates of key points were mainly input by the keyboard
- D_y refers to the displacement in the y direction
- Measurements of the positions of the stress were done from he left edge of the blank
- Metric units were used in the analysis
- Mild steel was defined by using the define command because it was not in the material library of the JL Analyzer. The values of the mild steel were served as follows:
  Young's modulus, EX= 2.070000e+006 kgf/cm^2
  Poison's ratio, NUXY= 3.000000e-001
  Density, DENS = 7.830000e=006 kgf*s*s/cm^4
- Command type I the sketch menu: polyline
- Mesh length 1.5cm

CHAPTER 6:

EXPERIMENTAL DESIGN

6.1 BENDING EXPERIMENTS (Angela Data Logging System)

6.1.1 OBJECTIVES

- ☐ To demonstrate sheet metal forming
- ☐ To investigate the effects of grain direction on the bending sheet metal
- ☐ To investigate spring back effects
- ☐ To determine the bending forces
- ☐ To determine the stresses

6.1.2 THEORY

Refer to chapter 2

6.1.3 EQUIPMENT SETUP

- ☐ The VERMAC press
- ☐ 1.6 mm thick mild steel
- ☐ Bending die set
- ☐ Shifting spanner
- ☐ Gear box oil
- ☐ Protractor
- ☐ Rule

6.1.4 PROCEDURE

(a) Mount the die to the bottom die shoe.
(b) Mount the punch onto the top of the top die shoe.
(c) Mount the top die shoe with the punch onto the punch holder fix it firmly
(d) With the die directly below the punch, lower the ram until the punch mates with the die. You may need to place metal strips of the same thickness as the sheet stock inside the die to set clearance for the material.
(e) Fix the bottom die shoe securely to the press bed.
(f) Set this as lower limit
(g) Fix the linear potentiometer to the press.
(h) Move the punch 125mm clear of the die and insert the predetermined and lubricated stock over the die opening.
(i) Pull the linear potentiometer shaft to wedge into the rubber carrier.
(j) Set this as the press upper limit.
(k) Start Angela measurement.
(l) Move the ram downwards to bend until the lower limit.
(m) Repeat (c) and (I) but this time with the sheet placed in such a way that the grain direction is along the bending axis.
(n) Measure the bend angles of the two products
(o) Examine the two products

6.1.5 ANALYSIS/ POINTS OF DISCUSSION

(a) Subtract the displacement column from 150
(b) Multiply the pressure column by 262.09
(c) Plot the displacement against the forces
(d) Read off the maximum bending force.

(e) Estimate the bending force using equation – and compare with the value from the graph.
(f) Compare the quality of the bends on the two products
(g) Compare the bend angles on the products to the die angle.

6.2 DEEP DRAWING EXPERIMENT

6.2.1 OBJECTIVES

- ☐ To investigate the effects of blank holder force on deep drawn parts.
- ☐ To determine the maximum force to draw a cylindrical shell.

6.2.2 THEORY

Refer to chapter 2

6.2.3 EQUIPMENT/ SETUP

Before setting up the equipment, remove the linear potentiometer. The rest is as in the above diagram, except this time use the drawing die set.

6.2.4 PROCEDURE

As in the above experiment, except the blank holder replaces the stripper, and the drawing die set replaces the bending die set.

(a) Repeat step a. and k. above
(b) Tighten the bolts on the blank holder.
(c) Start the Angela measurement
(d) Start the press and, using the two-hand operation, draw.
(e) Save the measurements.
(f) Repeat this experiment, with less torque than is necessary, to demonstrate wrinkle formation.

6.2.5 ANALYSIS/ POINT OF DISCUSSION

(a) Subtract the column of the displacement values from 150.

(b) Multiply the column of pressures by 262.09

(c) Plot the new displacement column against the force

(d) Read off the maximums drawing force from the graph.

(e) Calculate the drawing force from the equation 2.1 in chapter 2.

(f) Compare the theoretical value of the maximum drawing force with the value read off the graph

6.3 RESULTS

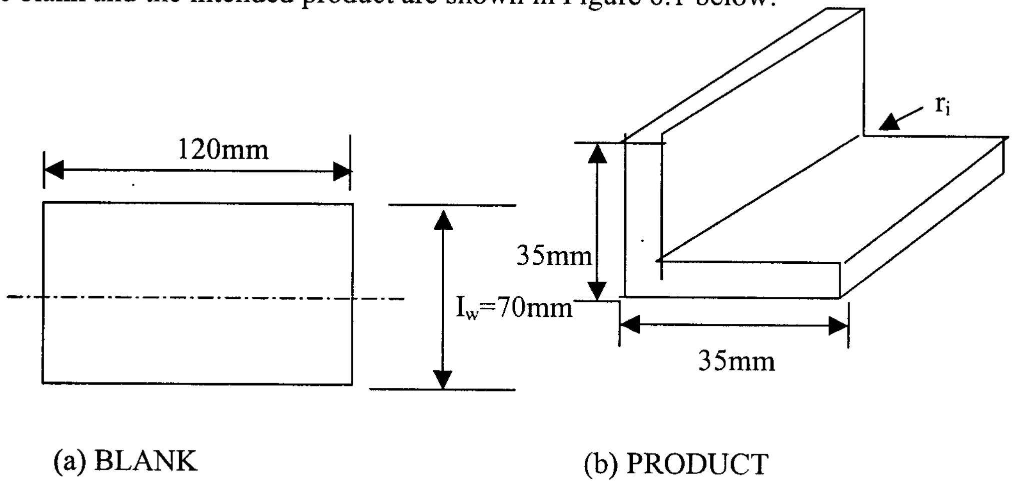
6.3.1 BENDING RESULTS (Analytical Method)

6.3.2 DEVELOPMENT OF THE 90° V – DIE BENDING

PART SPECIFICATION

The number of products that will be produced from the die set per annum is such that a low production tool could be designed for the job.

The blank and the intended product are shown in Figure 6.1 below:



(a) BLANK (b) PRODUCT

Fig. 6.1: Workpiece specifications

6.3.3 DETERMINATION OF THE MINIMUM BEND RADIUS

From the figure above taking our steel to be between dead soft and hard temper, we have:

$$r_i = 1\frac{1}{2}T \quad \text{--}(6.1)$$

Where T is the thickness of the blank, also to prevent weakening in case the bend axis falls parallel to the grain we multiply this value by 2. We then have:

$$r_i = 2 \times 1 \times 1\frac{1}{2}T \quad \text{--}(6.2)$$

Where:

- T is the blank thickness
- r_i is the minimum bend radius

Or r_i=3x1.5= 4.5mm

6.3.4 DETERMINATION OF WORKPIECE LENGTH

For this product, the mean radius r_m is:

$$r_m = r_i + \frac{S_O}{2} \quad \text{--}(6.3)$$

= 4.5+1.5/2 = 5.25mm

Where

- T = S_O which is the blank thickness
- r_m is the mean radius of the bend

And hence

$\frac{r_m}{S_o}$=5.25/1.5=3.5 --(6.4)

Thus from the appendix, for 90° bend, ξ is found to be 0.96.

Also a_l= 35 $-r_i$= 35-4.5= 31.5=a_2

From equation 2.9,

$$L_w = 31.5 + \frac{90\pi}{180}\left(4.5 + \frac{1.5}{2 \times 0.96}\right) + 3.5$$

.

$$\underline{L_w = 71.2}$$

6.3.5 SPRING BACK AND THE DETERMINATION OF DIE ANGLE

For the 1.5mm thick structural steel to be bend to have bend radius of 5mm, appendix gives k as 0.97.

Using the following:

Using that r_i/S_O= 5/1.5= 3.33

THEREFORE REARRANGING EQUATION 2.13, GIVES THE DIE ANGLE AS:

$$\alpha = \frac{\alpha_k}{k} = \frac{90}{0.97} = 92.8 \qquad \text{------------------------------------(6.5)}$$

Thus the included angle is 180-92.8=87.2°

6.3.6 DETERMINATION OF THE BENDING FORCES

For our workpiece and the die set

$W=2\{L_W/2Cos(\alpha/2)\}$

- W is the punch width
- b is he length of the blank
- S_U is the ultimate tensile strength
- t is the thickness of the blank

W=2(35Cos43.6)=50.7mm --(6.6a)

b=120mm

t=1.5mm

$S_U= 530X10^6$ N/m^2

Thus the bending load:

$$P=\frac{bt^2s_u}{w}$$ ---(6.6b)

$P= 0.120X.0015^2X530X10^6/0.0507 = 2.8225$ kN

Or

P= 2822.5/9810 tons = <u>0.288 tons</u>

<u>This is less than the Press tonnage of 100 tons.</u>

6.3.7 ANALYSIS OF THE DIRECT FORCES ON THE PUNCH

The punch has the smallest cross-sectional area and hence it was used as the design basis.

The projected area of the punch is:

$A_P = L \times B$ --(6.7)

=50.7X120

=6084mm^2

The load on the punch = bending force

P= 2.8225kN

Direct stresses:

$\sigma_P = \frac{P}{A_P}$ --(6.8)

= 2.8225X10^3/6.084X10^{-3}

= 0.463921761998685075608152531229454 MPa

Or

σ_P = 0.464 MPa

The yield stress for medium Carbon steel is 170MPa.

This gives a factor of safety of:

$$SF = \frac{\sigma_Y}{\sigma_P} \quad \text{--}(6.9)$$

$$\mathrm{SF} = \frac{170}{0.464} = 366.37931034482758620689655172413 8$$

SF=366

THIS IS VERY SAFE.

The die that was designed is shown in Figure 6.2 below.

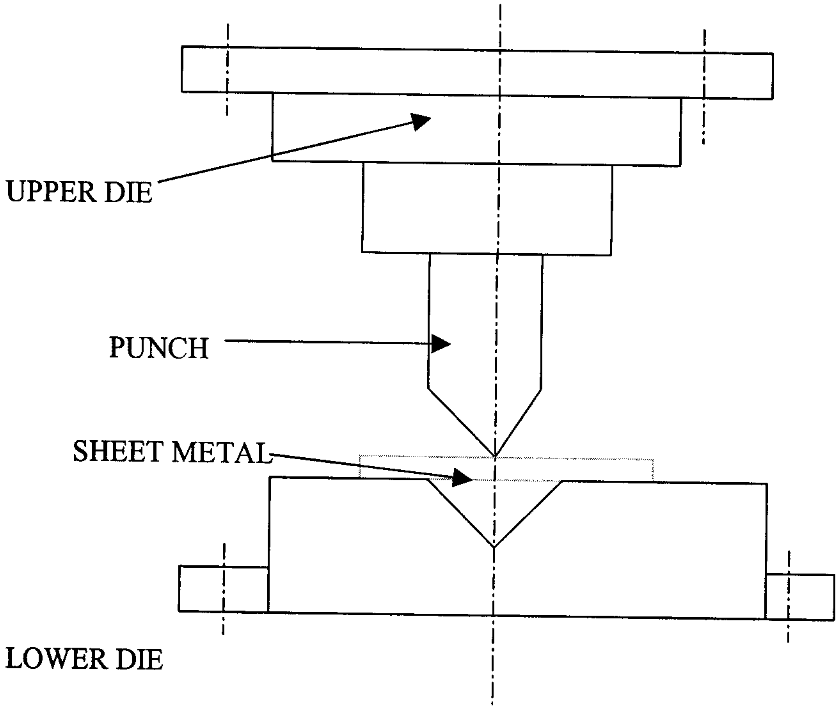
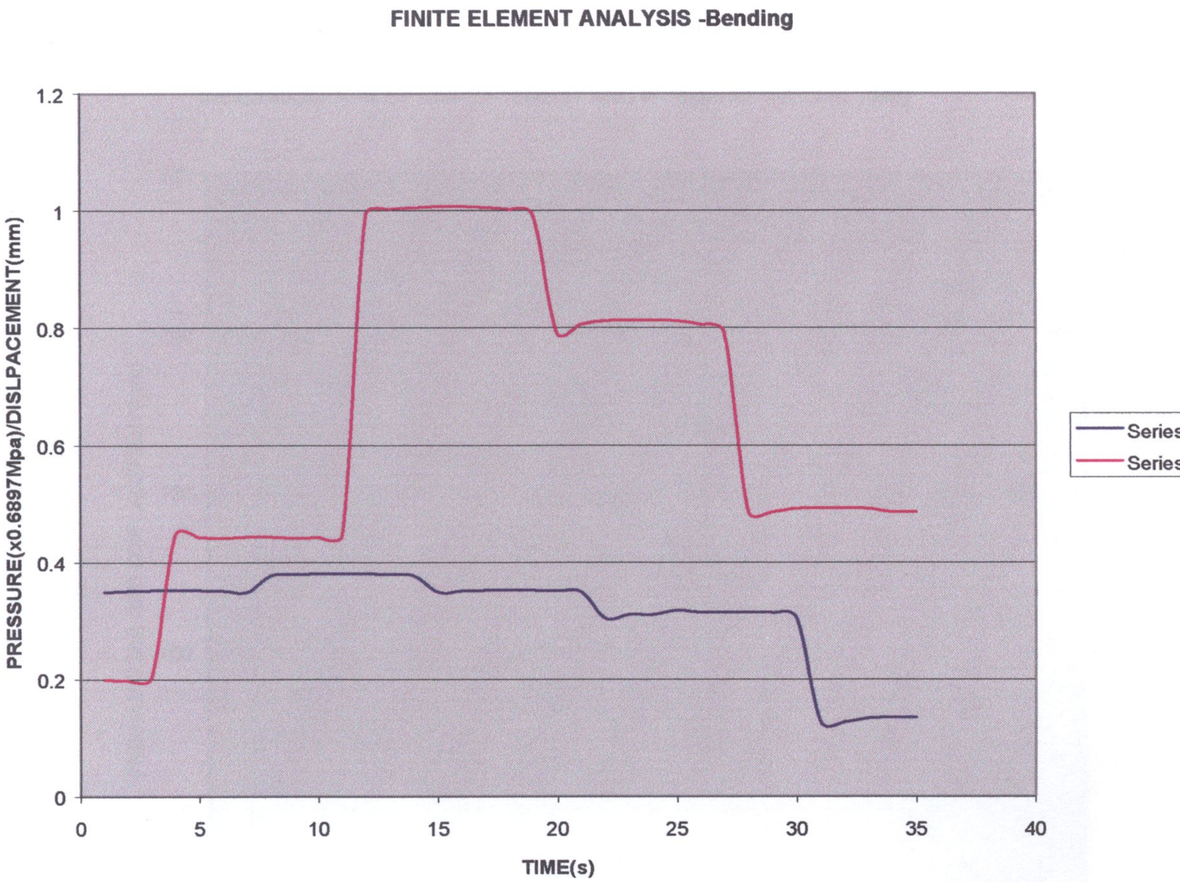


Fig. 6.2: The Bending Die set

6.3.8 ANALYSIS OF THE FINITE ELEMENT RESULTS



(Series 1 = Displacement, Series 2 = Pressure)

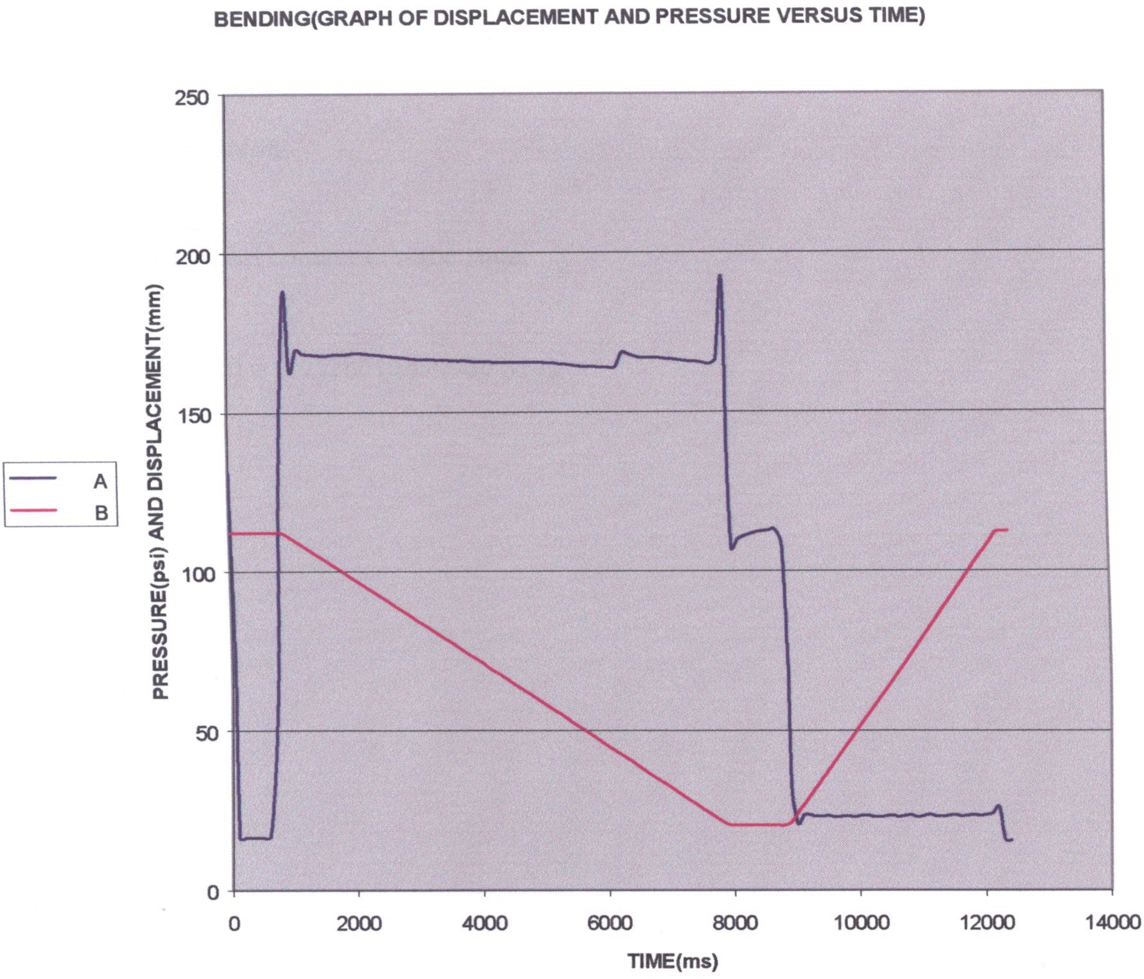
Fig. 6.3: Plot of Von Misses Stress, and Displacement against Time.

From the graph above it can be noted that the stress being analysed is the Von Misses stress. This stress is bigger than the analytical value because we are analysing the stresses in all the directions i.e. x, y, and z.

$$\sigma_V = \frac{1}{\sqrt{2}}\left[(\sigma_1-\sigma_2)^{\frac{1}{2}}-(\sigma_2-\sigma_3)^{\frac{1}{2}}-(\sigma_3-\sigma_1)^{\frac{1}{2}}\right]^{\frac{1}{2}} \quad \text{-----------------------------------(6.10)}$$

From the above graph the maximum stress has a value of **0.6897 MPa**, which is the average or von misses stress i.e. the stresses in all directions. In addition, because we are considering the comparison of this value with the analytical which is **0.464 MPa.** It is vital that we take the average value of the von misses stress, which is **0.643414 MPa,** which is derived from the Microsoft data file on finite element. It is vital to note that the data used to plot the above results is on the Output file of the finite element project simulation

6.3.9 ANALYSING THE STRESS LEVELS IN THE GRAPH OF ANGELA SYSTEM FOR BENDING



(A = Pressure, B = Displacement)

Figure 6.4: Load stroke diagram for bending 1.6mm steel plate

Plate thickness t=1.5mm

Punch actual angle i.e. die angle= 87.2°

Angle of plate after bending=84.5°

Pressure just when the punch touched the work piece was 169.79 psi

The highest pressure recorded was the pressure at which the punch hit the base of the bottom die and had a value of 190 psi.

Therefore the maximum bending pressure was

σ_p=169.79 psi

σ_p = 169.79 x 6894.7572 =1171339.984988 N/m^2

But 1psi = 6894.7572N/m^2

μ = 0.2 for steel-to-steel contact in static state.

σ_p = 1.17-(1.17 x 0.2) = **0.9370719879904 Mpa**

6.3.2 DEEP DRAWING

6.3.2.1 ANALYTICAL RESULTS

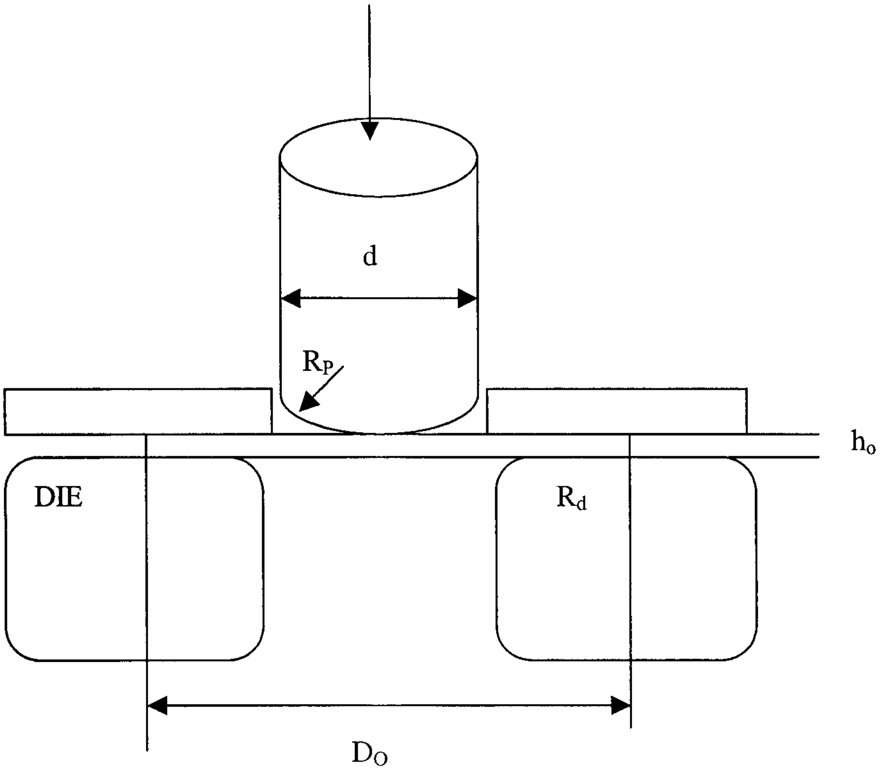


Fig. 6.5: Arrangement of the Punch and bottom Die in Deep Drawing

6.3.2.1a THE FORMULA FOR DETERMINING THE SHELL BLANK SIZE IS AS FOLLOWS:

$$D = \sqrt{(d^2 + 4dh)} \quad \text{---}(6.12)$$

D =√(60*60 +4*60*27)

D =100.4 mm

This is the theoretical blank size. However, to get a smooth edge, a trimming is necessary. For this, it is necessary to add extra metal. Rule of the thumb is to add about 3.2 mm to the blank diameter for each 2.5cm of the cup diameter. Since cup is 60mm, 64mm should be added for trimming.

D =100.4 + 6.4 = **106.4 mm**

6.3.2.1b RADIUS ON THE PUNCH AND THE DIE

For the blank to be drawn the following should be done:

$R_P = 4 X t$

$R_P = 4*1 =$ **4 mm**

DRAWING PRESSURE FORCE:

$$F = \pi d\sigma\left(\frac{D}{d} - c\right) \quad \text{-----(6.13)}$$

d= 60mm

t = 1mm

c = constant(0.6 to 0.7)

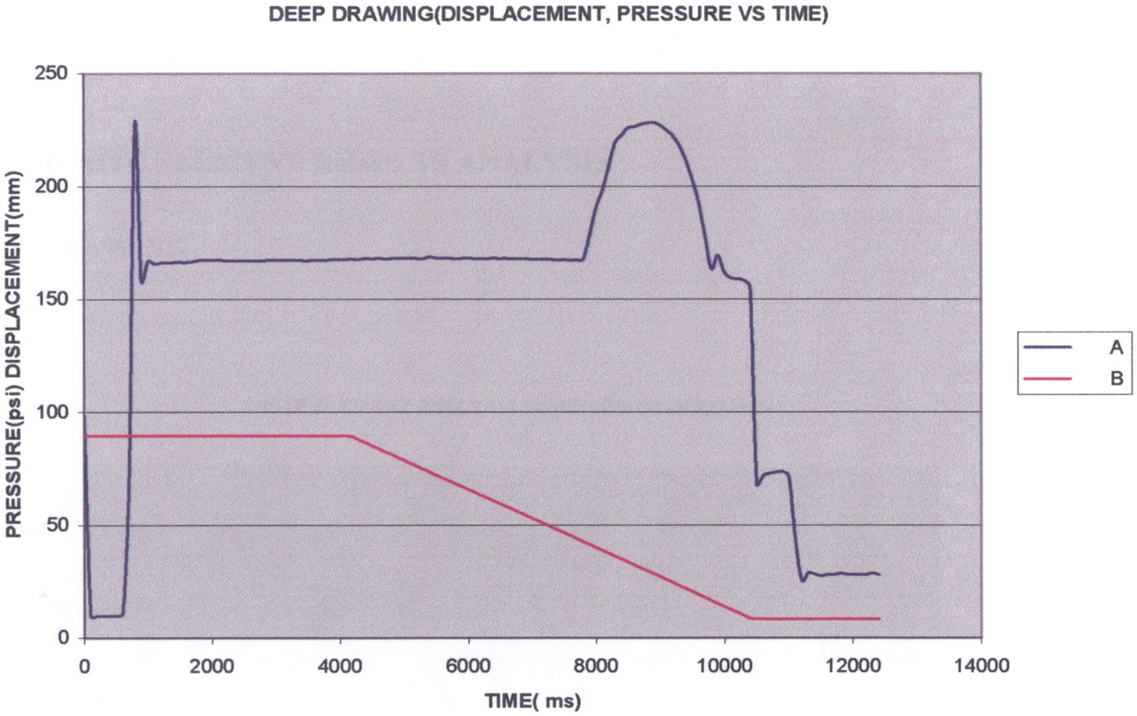
$\sigma_y = 530 N/mm^2$

F =3.14 X60 X1 X (2.37-0.65)

F = 171.832 kN

6.3.2 DEEP DRAWING RESULTS

6.3.2.2 ANGELA DATA LOGGING SYSTEM



(A = Pressure, B = Displacement)

Fig. 6.6: Load-stroke diagrams for Deep Drawing

Sample calculation of the force from the pressures

Piston area of the cylinder =0.038m^2

1 psi =6897.1 N/m^2

Therefore, to obtain forces from the pressures in pounds per square inch (psi)

We multiply the pressure column by 0.038 * 6897.1 =262.0898

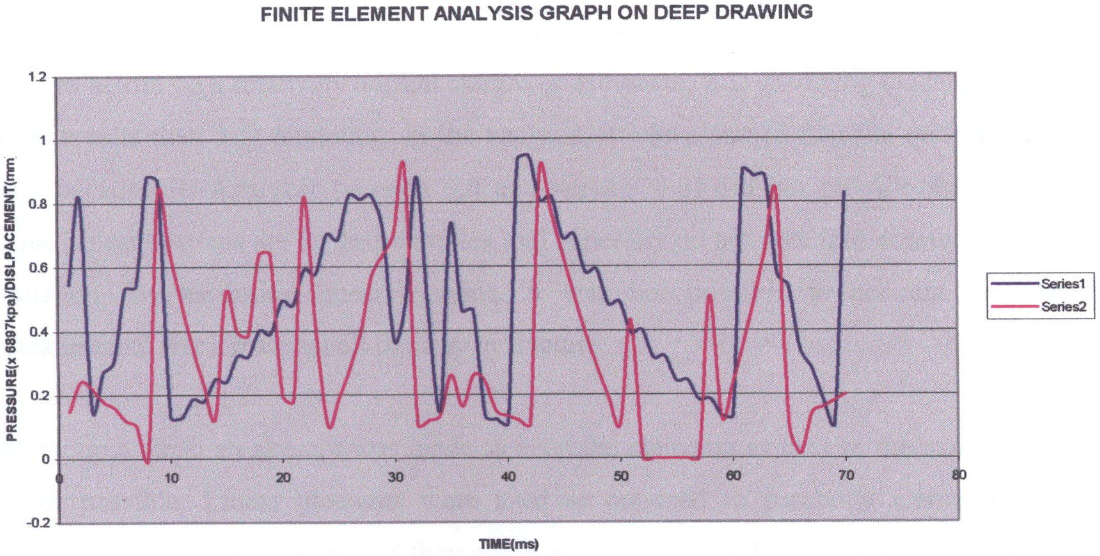
F =(225 –160) *262.09 = **170.358 kN**

NOTE:

It is vital to note that the results for the Finite Element Analysis on deep drawing could not be obtained accurately because of the limitation in the software itself regarding drawing of certain profiles, in this case the Deep drawing die set. However, the analysis can be done on one of the latest software called **GID**, were only the study of the software was done.

6.3.2.3 FINITE ELEMENT RESULTS ANALYSIS

DEEP DRAWING



(Series 1 = Pressure, Series 2 = Displacement)

Fig. 6.7: Graph of Pressure and Displacement against Time

From the above graph it can be noted that the maximum pressure is about (0.85x.6897000x0.038) = **222.776 kN** and the minimum force from the stress distribution of (0.5x6897000x0.038) = **131.043 kN** where:

- $0.038m^2$ is the punch area
- And 6897000 is the conversion multiplication factor of the stress into a force.

When the above values are compared to the analytical value of force, we observe that the analytical value of 171.832 kN is within the range of 131.043 kN and 222.776 kN.

CHAPTER 7

DISCUSSION, CONCLUSION AND RECOMMENDATIONS

7.1 DISCUSSION

By using JL Analyzer, it was possible to perform modeling, analysis, and post - processing in one environment. JL Analyzer is an easy to learn, easy to use engineering tool and allows the user to quickly master FEA.

There are generally two types of analysis, in 2-D modeling and 3-D modeling. In this project both the 2-D and the 3-D modeling was used to conserve on the simplicity and allowed the analysis to be run on a relatively normal computer. However, 2-D modeling tends to yield less accurate results than 3-D modeling. In the analysis, it was assumed that the system behaved linearly because JL Analyzer (version 5.0 and version 8.0) did not provide for non-linear systems. Linear systems are far less complex and generally do not take into account for plastic deformation. By assuming linear systems, it was not possible to account for plastic deformation and test a material all the way to fracture.

In setting up a mesh an attempt was made to keep the elements as near to the parent element shape as possible. Linear elements were used as opposed to parabolic elements. Linear elements have vertex nodes only and their edges are straight. Parabolic elements in addition to corner nodes, a mid- side node along each side creating quadratic type edges which can be a quadratic curve rather than a straight line. Stress and displacement analysis in most cases is done by considering the problem as plane stress (during meshing). Let us emphasize that, by our analysis, we can only obtain insight into the physical problem considered: we cannot predict the response of the problem exactly because it is impossible to reproduce even in the most refined mathematical model all the information that is present in nature and therefore contained in the physical problem.

From the results analysis on both the finite element and the Angela data logging system, it can be pointed out that the finite element produced results that were closer to the analytical values than the Angela data logging system. The following are the mean values of stress obtained:

MEAN STRESS VALUES:

ANALYTICAL METHOD

$\sigma_p = 0.464$MPa

FINITE ELEMENT

$\sigma_p = 0.643414$MPa

ANGELA DATA LOGGING SYSTEM

$\sigma_p = 0.937072$MPa

Some of the problems that were encountered in this project were:

(a) Limited access to computer facilities
(b) Limited memory space on most computers so the analysis was taking more time than expected in most cases.

7.2 CONCLUSION

Using Angela data logging system, JL Analyzer software and the analytical method it was possible to investigate the effects of stress on the work piece under deformation getting results with tolerance of 0.0001. Numerical values of the stress and displacements with their corresponding mode shapes were observed. The project also provided an engineering insight into possible failure modes for the work piece subjected to a pressure force under buckling.

7.3 RECOMMENDATIONS

FINITE ELEMENT SOFTWARE

The effects of stress on the work piece using JL Analyzer was done and reliable data obtained. The following are the recommendations for future work.

- ☐ a higher version of the software should be obtained which will be able to analyze non-linear systems.(JL Analyzer 5.0 was used in this project but the latest on the market is JL Analyzer 8.0).

- ☐ Computer with higher capacity should be used to save on the time i.e. the following should be the minimum system requirement.

    - Windows 98 or higher
    - At least 64 megabytes of extended memory (RAM)
    - Super VGA graphics card with sufficient video memory to support 256 colors.
    - CPU: 600NHz or more

- ☐ More models should be done in 3-D to give even more accurate results (most tests were done in 2-D due to unlimited memory space on most computers)

- ☐ Modeling of deep drawing die set could not give accurate approximations and results, thus GID software can be utilized because it can overcome some of the limitations that are encountered in the JL Analyzer.

ANGELA DATA LOGGING SYSTEM

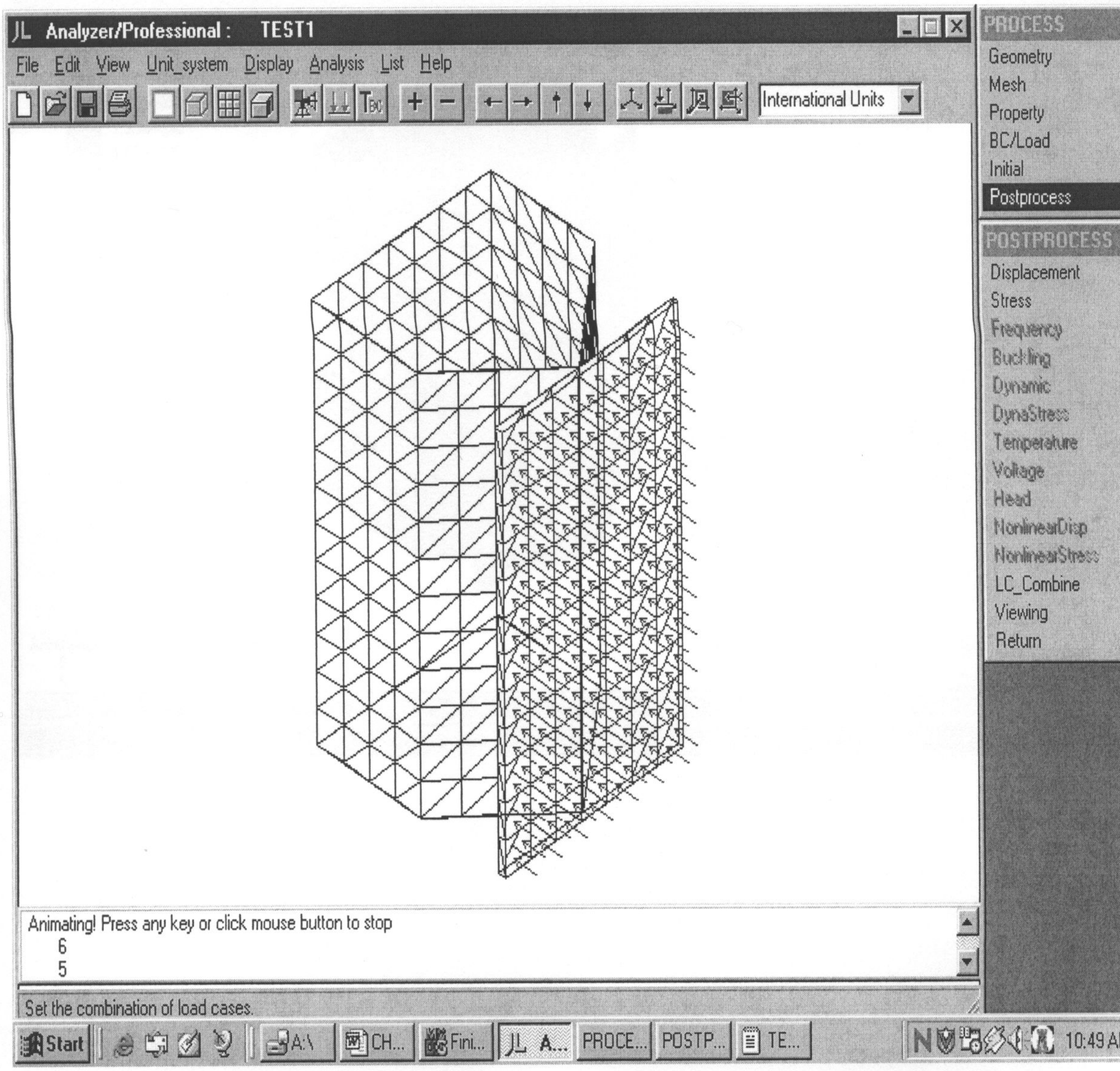
- For verification of the system, the pressure gauge on the press should be repaired
- The press faulty switch should be replaced
- The calibration of the DA/C card needs to be done in the presence of qualified personnel
- Sensors should never be left on the press after use.
- The cables for the sensors should be increased in length so that the MULTILAB BUS could be placed a little lower
- The tools should be hardened in order to increase their life. For this, pack carburisation could be most economical.

REFERENCES

1. Autofea engineering software technology inc. JL Analyzer user's guide (www.autofea.com).
2. Barson, J.M, Rolfe, S.T, fracture and fatigue- applications of fracture mechanics, prentice hall inc., 1977.
3. Beckwith, T.G, Buck, L.N., Mechanical Measurements, Addison-Wesley Publishing Company Inc., 1961
4. Monoghan, D, Finite Element Research group, the Queens University of Belfast' (www.sog1.qub.ac.uk)
5. Fundamentals of tool design, third edition, society of manufacturing engineers.
6. I. Lange handbook of metal forming, McGraw-Hill Book Company, 1985.
7. Moolela Malfred, Sheet Metal Forming Analysis, dissertation, UNZA, 1998.
8. Jere Donald, Metal Forming Tooling and Experimental Design, dissertation, UNZA, 2000.

APPENDICES

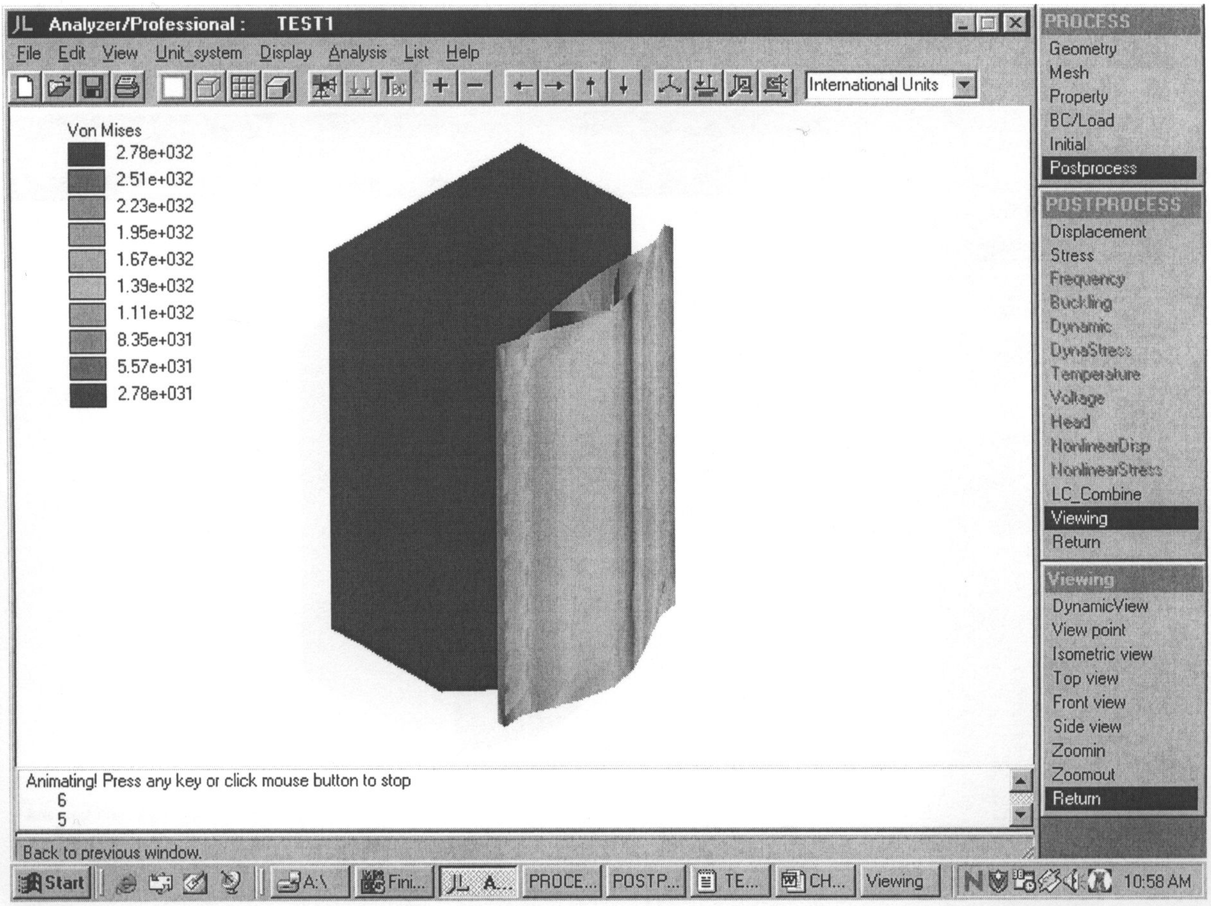
APPENNDIX A



```
Animating! Press any key or click mouse button to stop
    6
    5
```

```
Set the combination of load cases.
```

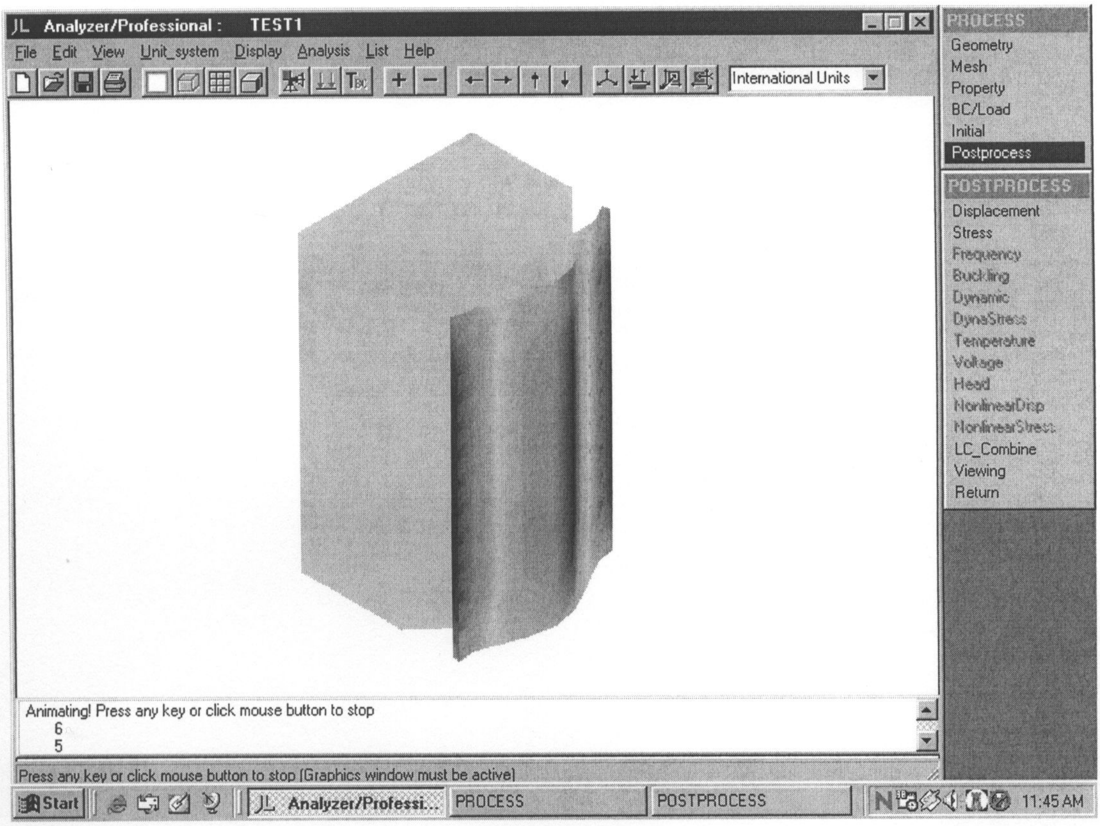
APPENDIX B



```
Animating! Press any key or click mouse button to stop
    6
    5
```

Back to previous window.

APPENDIX C



```
Animating! Press any key or click mouse button to stop
    6
    5
```

Press any key or click mouse button to stop [Graphics window must be active]