The Design of an aluminium alloy wheel using three dimensional Finite Element Analysis and Fatigue Life Prediction.

By

Mary M. Doyle BSc Eng, Dip Eng.

This Thesis is submitted to fulfil the requirement for the award of Master's Degree in Engineering by research to:

Dublin City University.

Supervisor : **Professor M. S. J. Hashmi.**
Head of the School of Mechanical and Manufacturing Engineering

January 1996

Some of the technical content of this work is based on the European Community Research and Development BRITE/EURAM project No 5349

The project Co-ordinators were based in Germany and the place of work for the Irish Partners was The Department of Experimental Physics, St Patrick's College, Maynooth, Co Kildare

The Author acknowledges, with thanks, permission which has been given to use the technical content.

The terms and conditions of this agreement is to keep this Thesis confidential until December 2000 The Author and Supervisor have agreed to this condition

Signed

Mary M Doyle

Professor M S, J Hashmi

Janurary 1996

DECLARATION

I hereby certify that this material, which I now submit for assessment on the programme of study leading to the award of Master's Degree by research, is entirely my own work and has not been taken from the work of others save and to the extent that such work has been cited and acknowledged within the text of my work.

Signed :

Mary M. Doyle　　　　I.D. No. : 94970637

1.1.1996

ABSTRACT

The Design of an aluminium alloy wheel using three dimensional Finite Element Analysis and Fatigue Life Prediction.

By Mary M. Doyle BSc Eng

Styling has always played a very important role in automobile design. This factor as well as the demands of new safety legislation in Europe and through out the world makes it a very competitive industry. This often leads to complex car designs which need to be produced and proof tested with a minimum lead time and expenditure. But these new designs and manufacturing technologies must be reliable, thus the automobile manufacturer is increasingly investigating and developing new design tools to help improve the quality of their products. Computer aided engineering helps reduce the time necessary to produce a new design. It also improves the quality of design. In this study computer aided design, finite element analysis and fatigue life prediction are the tools which have been used.

The design of a cast aluminium alloy wheel has been optimised using the Finite Element technique. It simulates the behaviour of the wheel under it's working load conditions. IDEAS Master Series has been used to develop a three dimensional linear elastic structural model. The wheel has been loaded with static load cases which represent the working load conditions. Maximum and minimum principal stresses were calculated and a comparison of these with measured test results was made to establish a correlation with acceptable accuracy. Stress-time histories from the tested wheel are used for this purpose.

Once the predicted results were validated, the technique was used to simulate stress patterns under a variety of possible load cases. The

mechanism of load transfer from the tyre to the wheel rim was studied in detail and suggestions made as to how to optimise the FEA's model load cases. It was found that the wheel's stress level in the critical areas was below the material's allowable fatigue stress level. Thus, the geometry of the wheel has been modified to optimise the volume of the aluminium alloy used in the manufacture of the wheel, yet still keep the stress amplitudes to an acceptable level.

Finally, a Procedure for the fatigue life prediction of the wheel was developed to verify that the actual lifetime of the wheel was greater than, or at least equal to the required lifetime using the Local Stress approach. Turbo Pascal is the programming language used here.

ACKNOWLEDGEMENTS

I would like to express my gratitude to Professor Hashmi, my supervisor, who continuously encouraged me throughout the past two years. In difficult times he never lost faith in either myself or my ability to complete this work. I express my heartfelt thanks to him for his patience and understanding, he always made himself available to deal with my questions no matter how basic.

To all the staff and postgraduates in D.C.U., I wish to acknowledge their assistance over the past two years. It has been greatly appreciated over the last couple of months when the intense pressure was on. Thanks especially to Lesley, Lisa, Mahiuddin and Liam.

Particular thanks is extended to Professor Slevin of Saint Patrick's College, Maynooth. His help has ensured that this thesis has come to it's completion. He has put the facilities of the Experimental Physics Department at my disposal and I am very grateful to him for this. Derek Gleeson has also been most helpful and supportive throughout the past two years. The expertise and assistance of Derek Coburn in the Turbo Pascal programming has been greatly appreciated. In fact I wish to express my thanks to all the staff and postgraduates in Experimental Physics, Bill, Clare, Marie, Naill, Paul ... Thanks to them all for the last two memorable years!!

I would like acknowledge the support of Professor Grubisic and Dr. Fischer from LBF Darmstadt, Germany. Their agreement in allowing me to use data from the BRITE/EURAM project No. 5349 has made this work possible. There expertise and experience has inspired me throughout.

Finally I wish to sincerely thank Aidan, who has patiently encouraged me over the last number of years. Thanks also to my Mother and Father - for their unlimited support and interest.

CONTENTS | PAGE NO.

PAGE NO.

PAGE NO.

APPENDIX

1.0 INTRODUCTION

1.1 Wheel design

In a car every component is important but some are more critical than others Components which may fail but will not cause fatal accidents are not catastrophic in failure The car electrics are an example of this Wheel, breaks, steering or tyre failure, on the other hand, can cause catastrophic accidents in failure

Road wheels are one of the most important safety components from a structural point of view They are required to be lighter and more attractive to the customer all the time This means that it has become necessary to perform more rigorous strength evaluations on new wheel designs

The designer must keep the following in mind when designing a new wheel.

* The wheel is required to be an aesthetically pleasing feature of a car

* It is classified as a safety component of a vehicle

* It is a very highly stressed safety component

* Modern car manufacturers have to meet very strict reliability specifications

* Fuel consumption must be reduced to a minimum, this means that cars must be as light as possible, because these two factors are directly related

* They are required to recycle as much of the material used in the product as possible and keep all manufacturing costs to a minimum

* Car companies are producing cars with a reduced design cycle time.

All these factors lead to a very competitive design process. To ensure the future of the manufacturer it is now vital for them to be able to produce the component quickly, inexpensively with a proven design approach which satisfies the required reliability.

The use of a light alloy wheel minimises weight and fuel consumption. They have high strength and rigidity characteristics and good fatigue resistance. They can be easily manufactured. They allow a very high level of recycling and have a high resistance to corrosion. Aluminium alloys have the best combination of all these requirements.

1.2 Literature survey

1.2.1 Finite element analysis (FEA)

1.2.1.1 General

The finite element analysis technique is a mathematical solution applied to engineering systems and is implemented on a computer system. It helps designers understand the system better and produce a quality product for the market faster, while reducing production costs. This is because there is often an optimisation of the design and fewer prototypes are required. A more comprehensive description is given by Cook *et al.* [1]:

The finite element method is a numerical procedure for analysing structure and continua. Usually the problem addressed is too complicated to be solved satisfactorily by classical analytical methods. The problem may concern stress analysis, heat condition, or any of several other areas. The finite element procedure produces many simultaneous algebraic equations, which are generated and solved on a digital computer. Finite element calculations are performed on personal computers, mainframes and all sizes in between. Results are rarely exact however, but errors are decreased by processing more equations. Results are accurate enough for engineering purposes and are obtainable at reasonable cost.

It is easy to analyse simple structures, such as a beam in a framework, using differential equations In the 'real' world the vast majority of structures are a lot more complicated than this and it is very difficult to solve their stresses, strains and displacements under operational loading conditions This is true for any section of engineering To solve these complex structures they first are divided (hypothetically) into finite elements which are so small that the shape of the displacement and stress field can be approximated without too much danger, leaving only the magnitude to be found Secondly all the individual elements have to be assembled together in such a way that the displacements and stresses are continuous in some fashion across the element interfaces, the internal stresses are in equilibrium with each other and the applied loads and prescribed boundary conditions are satisfied [2].

The finite element method combines several mathematical concepts to produce a system of linear and non-linear equations These can range in number from 20 to 20,000 or more and requires the computational power of a computer The method has little practical value if a computer is not used [3]

Although the Finite Element method is new to us, with it's development and success expandıng with the rapid growth of the digital computer, the idea of piecewise approximation is far from new The early geometers used 'finite elements' to determine an approximate value for π They did this by bounding the quadrant of a circle with inscribed and circumscribed polygons, the straight line sections being the approximation of an arc of the circle In this way they were able to obtain extremely accurate estimates Upper and lower bounds were obtained, and by taking an increasing number of elements, monotonic convergence to the exact solution would be expected Archımedes used these ideas to determine areas of plane figures and volumes of solids, although he did not have a precise concept of a limiting procedure It was only this fact that prevented him from discovering the integral calculus some two thousand years before Newton and Leibnitz ! [4]

The finite element method dates back to the early 1940s when the mathematician Courant published an article in the Bulletin of the American

Mathematical Society It is not possible to say who exactly invented the finite element technique because a number of mathematicians, engineers and physicists proposed discretization methods in the 1940s and 1950s [5] The papers by Turner et al [6] and Argyris and Kelsey [7] are regarded as important contributions Clough is reported to have been the first to use "finite elements" [8].

With the advent of digital computers in the 1950s, the finite element method was implemented by researches in aerospace and in civil engineering [9, 10]. The fact that their designs had to be correct first time and could not be tested before construction made this option an appealing one In the 1960s the fact the FEM could be derived from energy principles using variational calculus opened the method to all areas of continuum mechanics and physics [11] Soon after, the method was brought out from the research facilities and governmental bodies because general purpose programs were put onto the market. FEA became widespread for lots of applications in the 1970s and '80s Not only was it used for design verification but it was also used to optimise design at the concept stage There are many reasons for this Two of the most important are· The highly competitive nature of manufacturing and engineering, and, more access to hardware and software

Today FEA is used in most industries as a vital element of computer aided engineering Some fields where it is currently in use are· structural engineering; the computer, automotive, electronics, medical, transportation industries as well as in metalworking and metallurgy, fluid analysis, aircraft design and the space industry The list is endless and is continually growing. Numerous successful applications have be reported from the civil, mechanical and electrical engineering areas as well as the fluid mechanics and medical fields FEA plays an important role in pioneering new or improving traditional technologies [12]

In the automotive industry the finite element method is used in developing most components, from the steering column [13] to the car body joints [14]. Different studies have been made of the wheel also [15], and it has proved to produce very interesting results

Finite element software designers are constantly updating and improving their softwares. This is an important factor in keeping the method in daily use in industry.

1.2.1.2 Future trends

The workstation is an important design tool in engineering today. The finite element method is a key element in product design. Tools are now becoming available which allow FEA to be used routinely by the non specialist in product design, thus increasing the user productivity. These tools have significant potential to increase the competitiveness of the engineering industry [16]. Universities now integrate these tools into their curriculum [17]. They see that more and more companies are investing in this technology because it allows a faster design turn around, reduction in materials and labour costs and these saving justify the investment.

The engineer has historically the following tools available in the design of new products [18] :

Experience and intuition
Handbooks
The elusive exact solution
Numerical methods

A high percentage of product design today uses experience and intuition. This method will remain a very valuable asset in any design office. Handbooks are also an important design tool but are limited when more complex designs are required. The elusive exact solution refers to design using partial differential equations. These prove to be nearly impossible to solve. If the problem is greatly simplified then it no longer represents the problem in question, even though a solution can be found. Numerical methods are popular with the engineering community but not with mathematicians. This is because results are in numerical form and give an approximation of the exact solution, which is acceptable to the engineer.

Technology developments which have an impact on finite element analysis being used every day in industry are [19]

* *User interface.*
  Modern systems have the ability to guide new users through the analysis. The current trend is to use interactive graphics prompts. Help commands are on line making them a lot more accessible to the user.

* *Graphics*
  Interactive graphics are an important tool in FE packages both in pre and post processing. Current trends include 3-D volume visualisation, animation and quality X-Y data display. The future of graphics leads us into a more interactive environment, in which users can access movable 3-D models to examine numerous aspects of behaviour [20].

* *Solid Modelling*
  Solid models can be used to automatically generate mesh in FE packages. This is currently available but improvements are constantly being made and more and more CAD solid models are being used in this way.

* *Design Optimisation*
  Current design optimisation routines allow virtually any aspect of a design to be optimised, including shape, stress, weight, natural frequencies temperatures. In the future design optimisation will become common practice rather than the exception [21].

* *User Access*
  In the future FEA Programs will have to support users with advanced analysis needs and provide interfaces for this purpose.

* *CAD Interfaces*
  customers have, for a number of years demanded open standards, thus allowing users access to multiple systems. UNIX and MS-DOS are accepted as standards at present. For complex geometry

representation NURBS (non-uniform rational B-splines) is emerging as the industry standard

* *Interfaces with other Engineering Packages*
  Currently the ability of FEA packages to interface with other packages allows the user to choose the 'best of class' software Interfaces have been established with kinematics, plastic moulding, crash analysis etc In the future bi-directional data exchange will allow data exchange with many other packages such as stereolithography and 3-D laser sintering

Finite Element Analysis is not used for the design of the less expensive products This is because the software and hardware can still seem expensive to the smaller company and the benefits of FEA are not always immediate, thus making it harder to justify than production equipment

Management commitment, engineering expertise and the appropriate software and hardware are all equally necessary to implement production FEA Management commitment is needed because FEA requires a long term investment therefore technical managers need to consider FEA an up-front cost in design in the same way that a machine tool is considered an up-front cost in production [22]

1.2.1.3 Linear elastic models

Historically, the Finite Element Method has predominantly been used for linear applications In the early 80's , 90-95 % of Finite Element calculations were linear This is rapidly changing [2]

In general the world does not operate with linear rules It is very important, however, to know when the linear behaviour approximation can be used for a calculation, and, when it cannot Good engineering practice often uses approximation and usually, the assumptions of a linear behaviour are sufficient for designs

The fundamentals of the Finite Element linear elastic analysis are not discussed here The reason for this is that they are standard rules and can be found in many text books

1.2.3.4 3-Dimensional structural models

Although there are many ways to construct an FEA model, only a few element types are used for most models [23]

> 1-Dimensional stick models use beam elements, as do structural components such as trusses, bars and shafts
>
> 2-Dimensional shell and plate elements are used for both flat plate and shell components as well as for flanges and ribs when such detail is required
>
> 3-Dimensional "brick" elements are used when the geometry or loading requires full structural representation

Finite Element Analysis computational overheads increases dramatically with the increase in model detail but time and resources are of course limited It is important to analyse the structure, it's loading conditions and the required results before a first attempt is made to model it Modelling is possible at several levels of abstraction Choosing the appropriate level of formalisation is therefore a matter for the engineer's experience and education Analysis can be used effectively if the appropriate investments in people, hardware and software are made [22]

Some of the more updated Finite Element Packages can automatically produce a solid mesh from a C A D (Computer Aided Design) solid model The analyst must examine the part for unnecessary details, such as small chamfers and radii and eliminate or suppress as many as possible (without affecting the overall stress analysis results) This ensures that when the automatic mesh is generated, the mesh density will be optimised [24]

1.2.2 Fatigue

1.2.2.1 General

From as early as 1830 it has been noticed that a metal which is subjected to a repetitive or fluctuating load will fail at a stress level lower than that required to cause fracture on a single application of the load

The fatigue life of structures is an extremely important design criterion and the safety and reliability of the structure depends on it [25] The failure of a metal due to repeated loads was first documented in 1829 by Albert Since then considerable effort has been paid to the deformation behaviour of metal under reversed loading conditions In the early 1800s during the Industrial Revolution with the invention of rotating machinery, failure due to repeated loads became a recognised problem Fatigue still plays an important part in service failures in ground, air and sea vehicles It has been estimated that between 50 and 90% of all mechanical failures are due to fatigue (Fuchs and Stephens, 1980) Failures due to fatigue result in cracks or fracture after a sufficient number of load fluctuations

The problem of fatigue has been investigated from many different viewpoints and some of the pioneering work that has been done is described by R M Mitchell [26]

1829 Albert in Germany failure because of repeated loads is first documented

1839 Poncelet in France introduces the term *FATIGUE*

1849 Institute of Mechanical Engineers in England "crystallisation" theory of metal fatigue is debated

1864 Fairbairn first experiments on effects of repeated loads

1871 Woehler: first systematic investigation of fatigue behaviour of railroad axels. Rotating-bending test; S-N Curve; concept of "endurance limit"

1886 Bauschinger: notes the change in "elastic limit" caused by cycling; stress-strain hystersis loop

1903 Ewing and Humfrey: microscopic study disproves old "crystallisation" theory; fatigue deformation takes place by slip similar to monotonic deformation

1910 Bairstow: investigates changes in stress-strain response during cycling; hystersis loop measured; multiple-step tests; concepts of cyclic hardening and softening

1955 Coffin and Manson (working independently): thermal cycling; low cycle fatigue, plastic strain considerations

It has been noted over the years that fatigue failure falls into two categories [27, 28]:

* *Low cycle fatigue* (10 to 100,000 cycles)
  Here there is significant plastic strain occurs during some of the loading cycles, at least, and has a relatively short life.

* *High cycle fatigue* (over 100,000 cycles)
  Here long life and low loading is characteristic.

The type of loading is also critical to the fatigue analysis. Constant amplitude and variable amplitude are the two different loading conditions. Figure 1 shows the different scenarios possible [29].

E. Haibach describes the different scenarios in the following manner [30] :

With reference to Figure 2 :
The monotonic stress-strain curve of the material (a) indicates the ultimate strength Rm and the yield strength Re from which one derives the upper

limit value of stress that, being exceed just once, would mean failure of the component On the other extreme end the endurance limit SE specifies a limit value of stress up to which a fluctuating stress (b) can be endured any number of times without failure In between a constant amplitude stress sequence (c) exceeding the endurance limit but not the ultimate strength leads to a failure after a finite number of cycles, the higher the stress the sooner failure occurs This dependency is presented by the S-N curve which has to observe the endurance limit and the ultimate strength as the lower and upper bound values respectively. A convenient analytical description of the S-N curve in the finite life regime is

$$N = N_E * (S_a / S_E)^{-k}$$

$$\begin{aligned} \textit{for} \quad & S_a \geq S_E, \\ & R = S_{max} / S_{min} = const., \\ & S_{max} = (2 / (1-R)) * S_a \\ \text{and} \quad & S_{max} < R_e \end{aligned}$$

where the endurance limit = SE,
the endurance cut off point = NE,
the slope k and the stress ratio R are the determining parameters [31].

Under a stress history of variable amplitude (d), as is characteristic for the service stress histories of most components, the endurance will exceed the S-N curve If the stress is assessed in terms of the peak to peak cycle and the number of cycles being defined by half the number of reversal points, then according to Gaβner [32] and in equivalence to the S-N curve, a fatigue curve depicts the interdependence between the range or amplitude of the peak to peak cycle and the endurance value Gaβner also says that in the majority of cases the magnitude and frequency of the stresses follow a particular distribution law which can be clearly defined by statistical analysis of a sufficiently large number of measured values

The position of the fatigue life curve may be determined experimentally from variable amplitude fatigue tests [32 & 33] , or it may be found by calculation from the S-N curve by using some cumulative damage hypothesis [34]

How the fatigue life curve will be located in excess of the S-N curve will depend on the amplitude spectrum of the history in question : the greater the number of cycles that are comparable in size to the peak to peak cycle, the more the position of the fatigue life curve will be towards the S-N curve. Hence on the basis of the above definition the constant amplitude S-N curve represents a lower limit case of the variable amplitude situation.

Three design situations result from the above :

* Design and dimensioning for infinite life based on the endurance limit in cases where the peak to peak cycle of the stress spectrum will occur several million times.

* Design and dimensioning for finite life based on the useful life of the structure and by making particular allowance of the variable amplitude loading condition.

* Design for finite or infinite life but dimensioning made according to some limit value of maximum stress derived from the yield strength.

Fatigue - resistant engineering structures have evolved primarily through experience based on proven performance. The control of fatigue resistance through micro structural manipulation remains a difficult goal [35]. Progress has been made in improving our micro structural effects on fatigue behaviour, at least for simple alloy systems [36, 37]. Recent developments in fatigue analysis and material characterisation procedures have greatly improved our quantitative treatment of the fatigue process. Modern approaches to fatigue design emphasise finite life behaviour and view fatigue as a problem in cyclic deformation.[26, 38]. Properties which characterise the material, such as material strength, strain hardening and ductility are primary elements in the assessment of the influences of various micro structural features on the material fatigue resistance.

A major goal in engineering today is to predict the service life of a component as early as possible [27]. In engineering design programs, life prediction analysis can be integrated with the design to optimise the fatigue life before undertaking durability testing This reduces the overall length of the design cycle and can prove to be a very important point where the marketing of the product is concerned

1.2.2.2 Strain and stress approaches

As mentioned above the fatigue life of a component is affected by many different factors (ASM, 1975), for example [39] .

1 Type of load (umaxial, bending, torsion)

2. The nature of the load displacement curve (linear, non-linear)

3 The frequency of load repetitions or cycling

4. The load history (cyclic load with constant or variable amplitude, random load etc [Gautier and Petrequin, 1989, Bauxbaum et al, 1991]

5. The size of the member

6. The presence of material flaws

7 The manufacturing method (surface roughness, notches)

8 The operating temperatures (high temperatures that results in creep, low temperatures that result in brittleness)

9 The envıronmental operating conditions (corrosion, see Clark and Gordon, 1973)

In reality it is often difficult to make an accurate estimate of the fatigue life because, for many materials, slight changes in any of the above factors may greatly affect the fatigue life of the component

Two very important factors in fatigue are the number of cycles that the component is required to undergo without failure and the nature of the load displacement curve Figure 1 shows the classification of the different fatigue strength possibilities The engineer must know, very early on, which category the design falls into [29].

When the number of cycles is low (N = 10 to 100,000) and the cyclic loads are relatively large, having significant amounts of plastic deformation, the local strain or critical location approach is used This type of behaviour has been commonly referred to as *'low -cycle fatigue'* or *'strain controlled-fatigue'*.

The foundation for the strain based approach to fatigue is called the *'strain life relationship'*

Strain has two components it's elastic component and plastic component

$$\varepsilon = \varepsilon e + \varepsilon p$$

Where ε is the total strain
εe is the elastic component
εp is the plastic component

Or expressed as strain amplitudes from a constant -amplitude, zero-mean-strain controlled test

$$\Delta\varepsilon/2 = (\sigma'f/E)*\{(2Nf)^\wedge b\} + (\varepsilon'f)*\{(2Nf)^\wedge c\}$$

Where $\Delta\varepsilon/2$ is the total strain amplitude,
$\sigma'f$ is the fatigue strength co-efficient,
E is the modules of Elasticity,
2Nf is reversals to failure (1 cycle = 2 reversals),
b is the fatigue-strength exponent (Basquin's exponent),
$\varepsilon'f$ is the fatigue ductility co-efficient,
c is the fatigue ductility exponent

Figure 3 shows the log strain versus log reversals to failure.

This relationship applies to wrought metals only [26]. When internal defects govern life (as is the case with cast metals, higher-hardness wrought steels, weldments, etc) these principles are not directly applicable, and modifications for "internal micro-notches" must be applied [40]

In the second case, high cycle fatigue, the nominal stress approach was the first approach developed to try and understand the failure process It is still widely used in applications where the applied stress is within the elastic range of the material and the number of cycles to failure is large (N > 1,000,000)

Figure 4 shows some typical fatigue stress cycles, (a) fully reversed, (b) offset, and (c) random

(a) This is typical of the loading condition found in rotating shafts operating at constant speed without overloads

(b) A general loading condition where the maximum and minimum stresses are not equal In this case they are both tensile and so define an offset for the cyclic loading

(c) This represents a more complex random loading pattern which is more representative of the cyclic stresses found in real structures

From Figure 4 it can be seen that a fluctuating stress can be considered to be made up of two components

* A static or steady state stress - **Sa**

* An alternating or variable stress amplitude - **Sr**

The stress range $Sr = Smax - Smin = (Smax - Smin) / 2$
The stress amplitude $Sa = Sr/2$

The mean stress	$Sm = (Smax + Smin) / 2$
The stress ratio	$R = Smin / Smax$
The amplitude ratio	$A = Sa / Sm$

Between 1852 and 1870 the German railway engineer August Wohler set up and conducted the first systematic fatigue investigation He conducted cyclic tests and plotted the results in terms of nominal stress versus cycles to failure, on what has now become known as the S-N diagram The S-N relationship is determined for a specific value of **Sm**, **R** or **A.**

When plotted on log-log scales, the relationship between alternating stress, S, and the number of cycles to failure, N can be described by a straight line, Figure 5 The following relationship exists

$$N = No *(S/So)^k$$

Where k is equal to **1/b**
and **b** i s equal to the slope of the line

As mentioned previously the S-N approach is applicable to situations where the cyclic loading is essentially elastic Great care should be taken in using the above S-N equations in situations where lives are less than 10,000 cycles Figure 6 shows the S-N curve for two metals, one ferrous, one non-ferrous Note that the mild steel has a fatigue limit while the aluminium alloy does not

Mean stress plays a very significant role in the fatigue life of a component Failures tend to be more sensitive to tensile mean stress than compressive The Gerber and Goodman curves relate the important factors here and allow the number of experimental measurements be reduced to a minimum

1.2.2.3 Local stress

Figure 7 illustrates how difficult it is sometimes to use the fatigue data available for the design to produce the calculations required for the fatigue life of the component under operational loading conditions

Designers are very often faced with complex geometry's which are subjected to irregular loading conditions The critical regions within the structure commands the most attention and a lot of time and effort are expended in analysing these critical areas.

Local material response which is observed in the critical areas, is analysed by cutting smooth specimens from the structure and subjecting them to local stress or strain histories. Plastic strain will often occur locally and this will result in crack growth and eventual failure when no strain energy can be accommodated by the local region, due to lack of ductility.

Computer based material modelling and damage accumulation techniques use these concepts and they have effectively applied the problem of predicting the initiation and early growth of fatigue cracks in such situations [41, 42]

The Ford Motor Company has requirements for durability which all there vehicles must meet before they can go into production Due to the fact that they find the process of finding and fixing these fatigue criteria costly and time demanding, sometimes even causing a delay in production schedules, they are now looking at ways of using analytical methods of predicting durability failures in large body systems early on in the design process. This will result in a reduction in design, engineering, manufacturing, tooling and prototype costs [43, 44]. FLAP (Fatigue Life Analysis Procedure) is the methodology used in the above papers.

Automotive industries world-wide have developed many durability requirements for their vehicles which have to be met before going into production [45, 46] Vehicles are usually tested by driving them over a pre-defined durability track. This "find and fix" testing process is very expensive and time consuming and may extend the vehicle development cycle time

The best known and most widely used cumulative fatigue damage procedure is the Palmgren-Minor procedure which uses either (a) a material S-N curve, based on nominal stresses, or (b) a component S-N curve, based on local stresses in critical areas

In the case of (a) it has been found that results from such specimen can be transferred to components with considerable limitations [47] In the case of (b) where the basic data was determined from the components, the manufacturing conditions as well as the mulitaxality loading conditions, if existent, will be represented (Note This is particularly important in wheel design where S-N curves required for each specific component must reflect exactly the stress conditions in that critical area of the component)

The Modified Minor hypothesis will be explained in more detail in Chapter 6 where it is used in a Fatigue life Prediction Procedure.

1.3 Scope of work

The objective of this work is to apply a finite element modelling technique and a fatigue life prediction method to optimise the design of an aluminium alloy wheel In general the use of these design tools will help the designer reduce the time necessary to completely design and validate a new wheel concept In addition to this the quality of design can be greatly improved upon, several variations can be investigated and the influences of modifications easily analysed Thus, the use of this computer based design technique can reduce design lead time and costs whilst producing a reliable product and increasing customer confidence and overall satisfaction

Computer aided design techniques have been used to optimise the design of the wheel This is achieved by firstly analysing the stresses induced on the wheel model under operational loading Then secondly, by using this stress amplitude information in the fatigue life prediction software it can be seen whether the design has met or superseded its design requirements The design can then be easily modified is several ways to optimise volume, material, manufacturing technique or any other design variable which may be required.

The work was divided into two areas:

(1) The finite element modelling and optimisation of geometry
(2) The fatigue life prediction.

A three dimensional linear elastic finite element analysis technique has been used. The software I-DEAS Master Series (versions 1.3 and later on 2.1) was the software used to produce both the initial solid model and the finite element model. The hardware platform was a Silicon Graphics Indy workstation.

The model was checked by initially using the checking facilities provided by the software and then by analysing the model behaviour under a straight driving load case.

The loading conditions used in the study represent the European standard road loading conditions. The principal stress results of the calculations have been compared to the measured results and a correlation has been deduced. This has be done by using stress-time histories.

An investigation into mechanism of load transfer from the tyre to the wheel rim has been carried out. Here different distributions have been looked into and one case chosen for the remainder of the study. The criterion for choosing this case was it's principle stress correlation with measured results at critical areas of the wheel (i.e. where the spoke and rim join).

The calculated principal stress results were also used to predict the fatigue life of the wheel. It was found that wheel stress amplitudes at critical areas of the wheel were well below the allowable material stress levels.

It was then decided that rather that changing the material or manufacturing processes, the geometry of the wheel was to be optimised. Two variants were made by modifying the mesh elemental co-ordinates. The resulting total wheel masses were reduced by 1% and 1.9% respectively. These modifications increased the principal stresses at the critical location by 8.8% and 16 9%

This did not prove to have a radical affect on the fatigue life of the wheel as so the design variation no 2 was accepted

To estimate the fatigue life of the wheel no standard software existed To automate complicated 'hand' calculations a prediction software was developed The language used was Turbo Pascal The local stress approach is employed

Maximum and minimum principal stresses at critical locations, information on material and manufacturing techniques, required design life as well as wheel and tyre definitions are the inputs into the procedure It calculates (for the defined area). the design spectrum and the S-N material curve The Palmgren Miner Damage calculation is then performed and both the expected damage and expected life of the wheel defined for that critical location are outputted

One particularly good feature of the software is it's feedback loop. The presence of this loop means that if the expected life does not give the required value then modifications can be introduced into the calculation For example a different manufacturing process can be selected to increase the life of the wheel This reduces the time required to obtain a suitably optimised design

This thesis has been divided into eight chapters Chapter one presents the literature survey Fatigue life prediction and the finite element method are discussed here The past and future of these techniques are looked into and their relevance in today's industry is analysed This chapter also gives a brief summary of the objectives of this research Chapter two discusses the software and hardware used to produce the finite element model It explains how the model was constructed and checked Chapter three is a description of the testing method The testing was not performed by the author. The analysis and results of the finite element calculations are addressed in chapter four The comparison of the calculated and measured results is made in chapter five Here, a description is made of the attempted optimisation of loading conditions Chapter six introduces us to the fatigue life prediction software The results for the test case are calculated and an optimisation of geometry is discussed This is the subject of chapter seven Conclusions are reached in chapter eight, this being the final chapter The thesis is concluded by appendices which contain software and hardware specification, sample list file output, stress-time histories, plots of the finite element mesh, parabolic distribution information, strain gauge location information and maximum and minimum stress results.

Finally, to make a finite element study which accurately represents 'real life' conditions a lot of information is required If this information is not available the reliability of the results can be questioned If this information is available, care must be taken to validate both the model and results as inaccuracies can easily be introduced Here, a considerable amount of time and effort was expended in validating the accuracy of the model and results. Care was also taken in the fatigue life prediction area The material data used in the S-N curves was found by testing actual components in a special set up which will be described later on in this thesis All testing and material data was kindly supplied by an external source which are very reliable

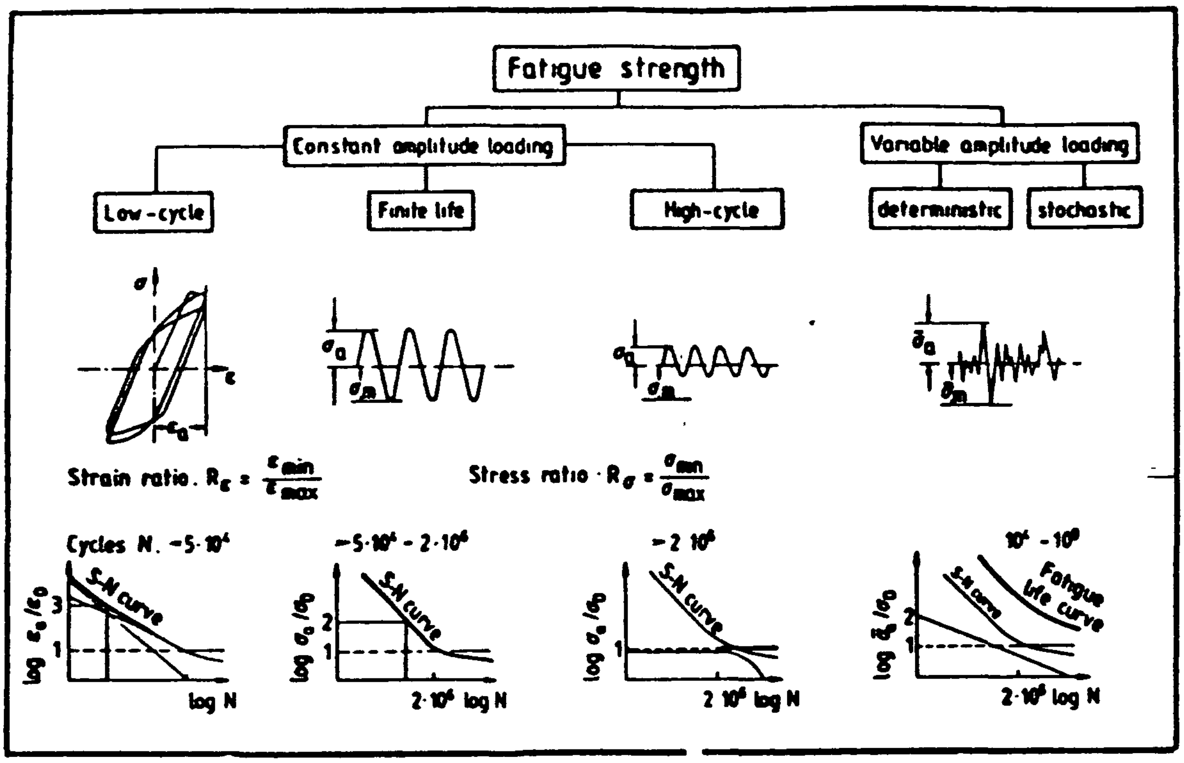


Figure 1. The classification of fatigue strength.

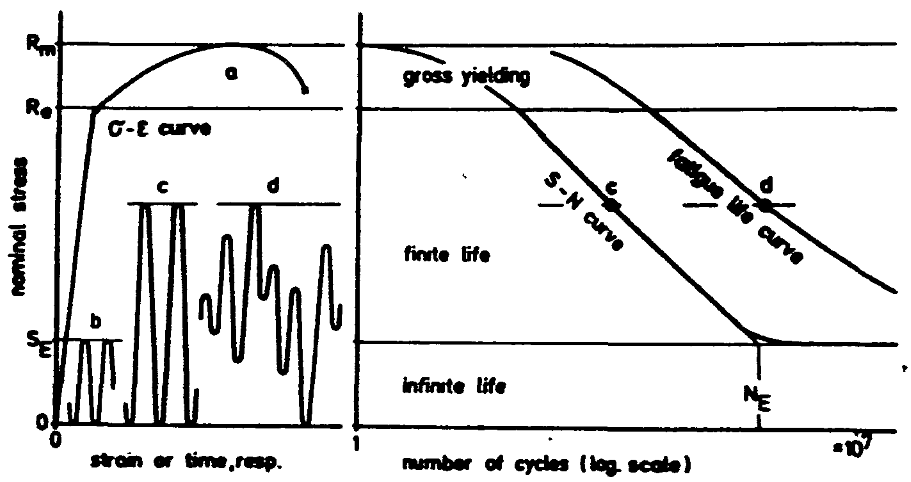


Figure 2. The relationship between stress and life.

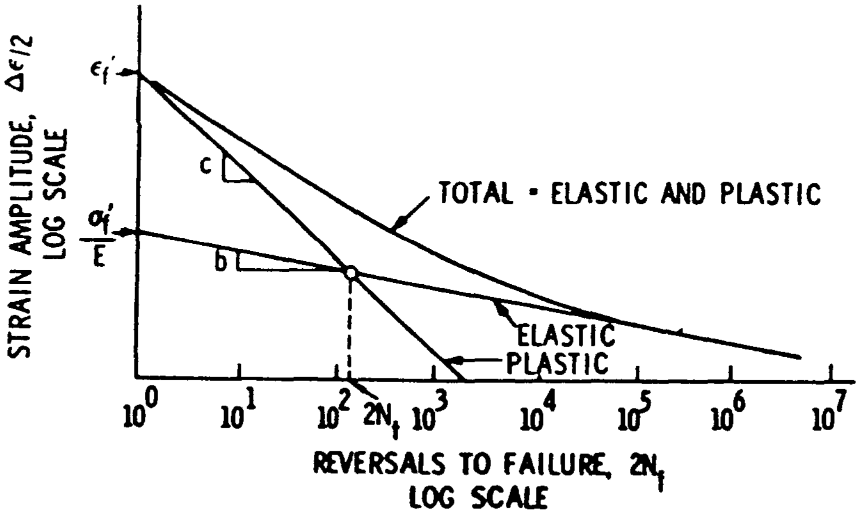


Figure 3. Log strain versus Log reversals to failure.

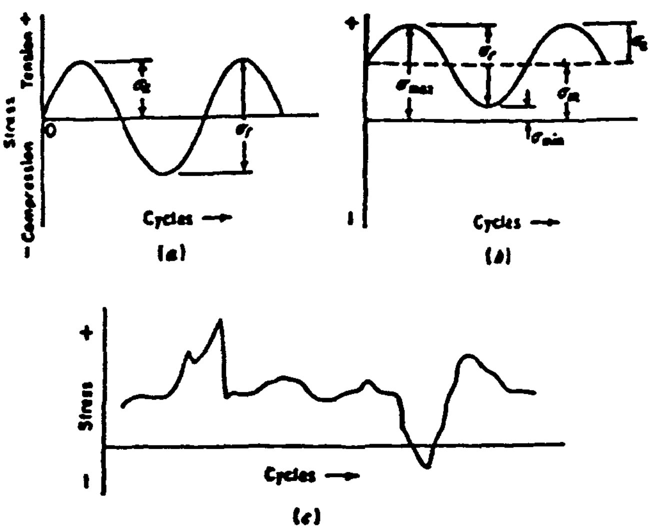


Figure 4. Typical fatigue stress cycles :
(a) fully reversed, (b) offset (c) random.

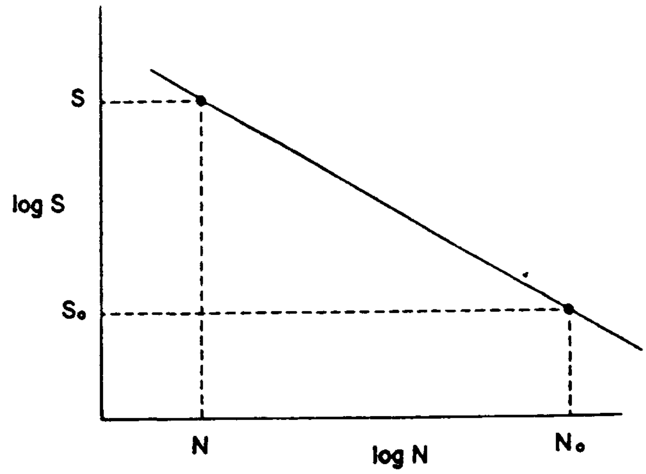


Figure 5. Idealised form of the SN curve.

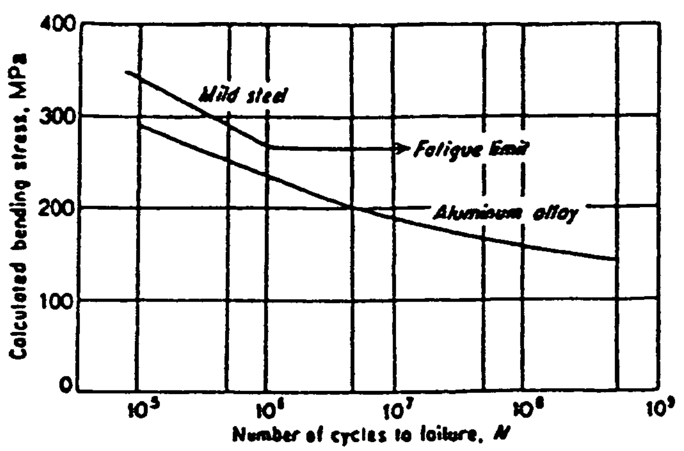


Figure 6. Typical SN curves for ferrous and non ferrous metals.

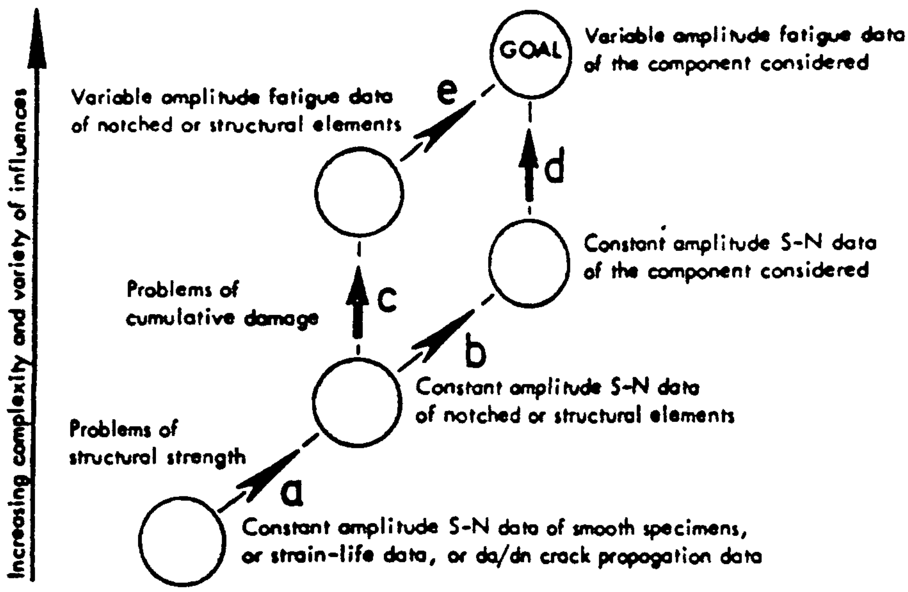


Figure 7. Conversion of available fatigue data to service conditions.

2.0 MODELLING SOFTWARE AND HARDWARE

The software package used for this work is the I-DEAS finite element pre and post processing and linear model solution modules

SDRC's I-DEAS Master Series is a complete mechanical CAE/CAD/CAM system with more than 70 tightly integrated modules that automate the entire mechanical product development process from design through drafting, simulation, testing and manufacturing Figure 8 illustrate these modules

The following description is taken from the I-DEAS Master Series product catalogue [48]

The Master Modeler software is a high performance 3-dimensional design system. The solid based approach simplifies the construction of complex geometry and facilitates design changes. The geometry is created in the Master Modeler module and is used directly in other I-DEAS applications, for example finite element modelling, drafting and manufacturing.

An integrated data management system provides the foundation for concurrent engineering by maintaining associatively between Master Modeler, drawings, finite element models and NC data. I-DEAS provides 'concurrent associatively'. Here, the project designer can give early 'snap shots' of the design to other team members involved in the design allowing them to start their solutions, drawings etc. At the same time the designer can continue to work and when the design is 'checked in' to the master model, all the related applications will be updated to reflect the revised design.

NURBS (Non Rational B Splines) geometry is used to create the model. The flexibility of this modelling approach is due to the integration of wireframe, surface and solid modelling. The surfacing software is the sculptured modelling compliment to the Master Modeler. Tools provided include lofting, sweeping and blending surfaces giving excellent local and overall control of the surface shape. The result is a completely integrated unified wireframe, surfaces, trimmed surfaces and solid data structure.

I-DEAS Model Solution Linear software is a general finite element analysis program for linear structural, thermal and flow analysis. In addition to the standard beam, shell and solid elements it supports a p-version tetrahedron element. It offers a very large list of modelling elements and analysis types.

The I-DEAS Master Series runs on an Indy Silicon Graphics platform [see appendix A for specification] It has 64 mega bytes in RAM and 199 mega bytes in SWAP The system disk is 630 mega bytes and there is an extra external disk of 2 giga bytes which is necessary for the finite element calculations

2.1 Description of model construction

Mesh definition is an important feature of the finite element model The type of mesh depends on geometry, loading and boundary conditions Several different elements are available in most modern systems for mesh generation beam, rod, solid, shell, plane strain, plain stress, gap, spring, p-elements etc Most of these have variations such as linear and parabolic It is not always obvious which type of element should be used for a particular study Sometimes a few models using different elements should be tried out in the study and acomparison of the results made, thus leading to an optimised choice

Mesh generation can be manual or automatic depending on the software used A combination of these techniques is usually used

After the mesh has been drawn (i e given co-ordinates and the connectivity decided) the following must be defined material properties of the elements, physical properties (to complete the modelling of beam and shell elements), loading conditions and constraints

It is vital to have accurate information for the above if an accurate analysis is to be carried out Loading information must represent actual conditions Some studies require dynamic models Here inertial force affects are taken into account. The stiffness and inertia of the system must be defined The dynamic analysis solves for natural frequencies and modal shapes of the restrained or unrestrained structure If a study of heat transfer is required thermal loads can be used. Applied displacements, mechanical loads (point loads, inertia forces, surface pressures etc) and initial strain or stress loads (thermal loads) can be represented by the software Boundary constraint conditions are usually one of the following . restraints applied to nodes and kinematic degrees of freedom (unrestrained nodes) in all six directions of motion. They are applied to nodes only

After the mesh is completed several checks must be performed before any analysis can be run These include checking for co-incident nodes and elements, free edge and element distortion

Unless a dynamic or heat transfer analysis is required a static analysis is performed

A solid model of the wheel was constructed. As the wheel was symmetric about each spoke centre, initially only a one tenth section of the wheel was modelled. The solid model was developed using I-DEAS Master Series 1 3 with the extrusion, revolution and master surfacing tools

This spoke was reflected about its centreline to produce a single spoke, which was revolved four times to produce the full wheel

The finite element model was produced from the one tenth solid section. The high level of curvature and the varying wall thickness dictated the type of mesh produced Manual meshing techniques were used to obtain an acceptable mesh in solid linear brick elements Solid linear wedge elements were used only where necessary, for example, to join different regions of the mesh in the hub area.

Some pictorial representations of the mesh are shown in Appendix B.

The mesh was refined in the rim and spoke regions where necessary. It was left coarser in the hub regions As a guide, two elements were used for the wall thickness of the rim except where it was intersected by the spoke Here four elements were used In the spoke region, generally three elements model the two edge ribs whilst four are used for the central rib In the hub region a course mesh was created around the bolt holes. This proved sufficient to allow the application of the bolt hole restraints

A suitable material property table for the G-AlSi 7 Mg grade of aluminium alloy was defined

Elastic modulus	E = 70 MPa
Poissons ratio	$\mu = 0\ 33$
Mass density	$\rho = 2.7$ g/cm³
Shear modulus	$G = E/2\ (1+\mu)$

The total number of nodes =19,361
The total number of elements =13,978

2.2 Description of model checking

The initial one tenth section structural mesh was checked using the following tests to ensure good element formulation and continuity.

- *Free edge* to ensure accurate element continuity i e no cracks.
- *Co-incident element* to ensure no duplication or overlaying of elements
- *Co-incident node* to ensure no duplication or overlaying of nodes
- *Element interior angles* not less than 45 degrees for bricks and 35 degrees for wedges
- *Element distortion* not less than 0.6 for bricks and 0 4 for wedges

2.3 Description of Boundary conditions

The wheel has been restrained at the five bolt holes in the following manner:

The nodes around the edge of each bolt hole under the head of the bolts were restrained in three degrees of translational freedom. This was felt to be representative of the test conditions.

2.4 Description of loading

Unit load cases were selected to minimise the computational solve time. For each loading condition, four unit load cases were configured comprising of:

* Unit vertical load (1N)on the outer rim - Fvo
* Unit vertical load (1N)on the inner rim - Fvi
* Unit lateral load (1N) on the outer rim - Fho
* Unit lateral load (1N) on the inner rim - Fhi.

Each of these loads was distributed in a parabolic fashion and applied as point forces at the nodes on the rim in a position closest to that specified. Figure 9 shows the parabolic load distribution

Tables 1 to 18 in Appendix C show the magnitude of the loads at each nodal position on the rim. Here the total unit load was distributed over a range of angles and at two different locations on the rim. The first location is where the centre of the parabolic distribution is centred on spoke one (as in Figure 9), and the second location is where the centre is located between spoke one and spoke two.

The loading in these Tables was calculated by using the following equation

$$F_i = F_{total} * \{1-(\theta_i / \theta_{total})\}$$

Where F_i = force at node i

F_{total} = total force,

θ_{total} = total angle of the parabolic distribution

θ_i = angle i of the parabolic distribution

and i varies from 0 to total angle of the parabolic distribution

The loads were applied to nodes at distances on the rim close to the specified offsets

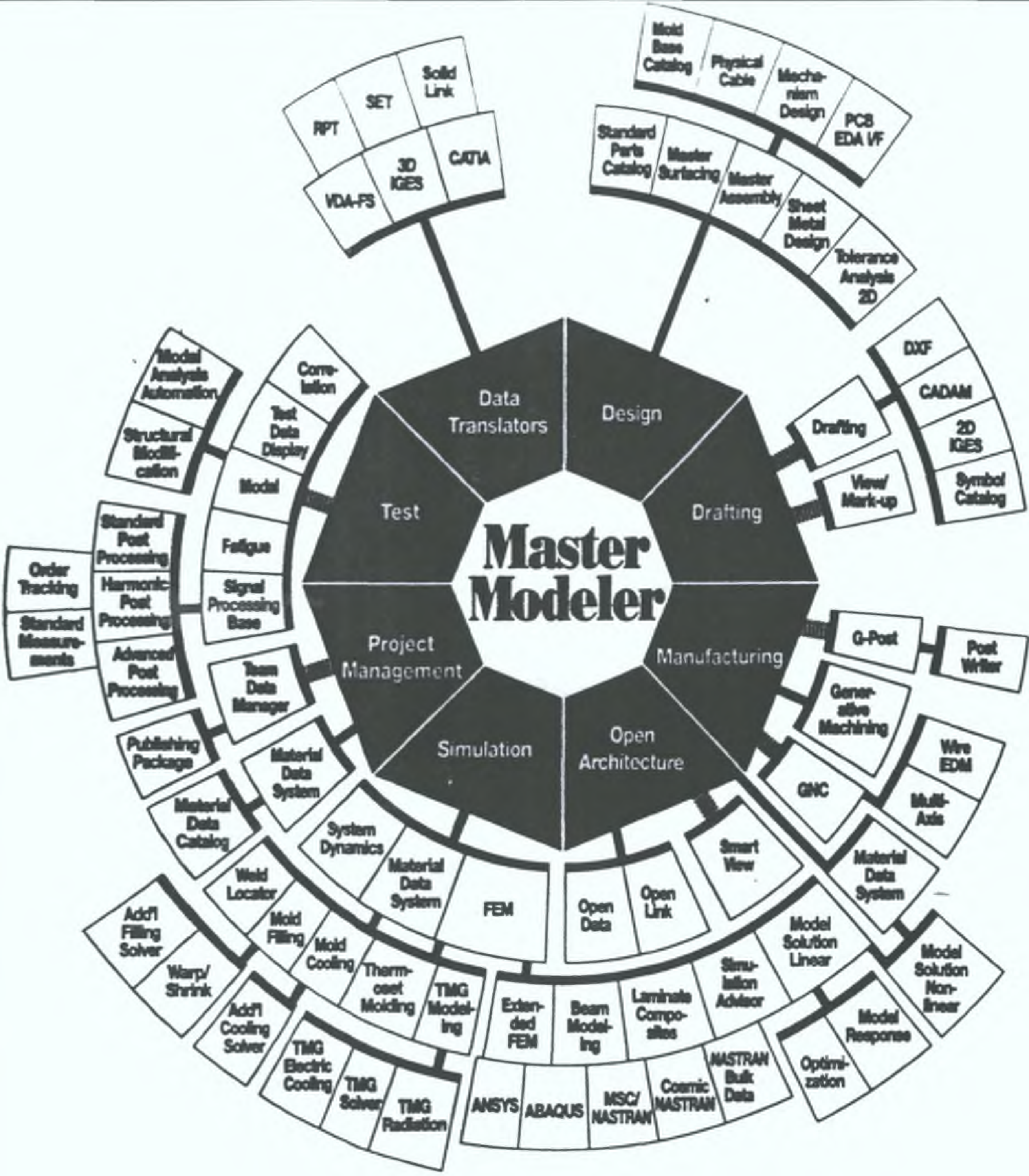
Lateral forces offsets = 5-10 mm

Vertical forces offsets = 190 mm

Figure 10 shows these details

The final straight driving and cornering load cases were computed by the scale and combine function of the I-DEAS Post-Processor

Note Loading resulting from the bolt tightening torque and the tyre inflation pressure were not applied as these were not included in the measured stress time histories



Master Modeler—Single User
- **I-DEAS Product—Single User**
- **I-DEAS Feature—Functionality only**
- **Unlimited Users**

Figure 8. I-DEAS product catalogue.

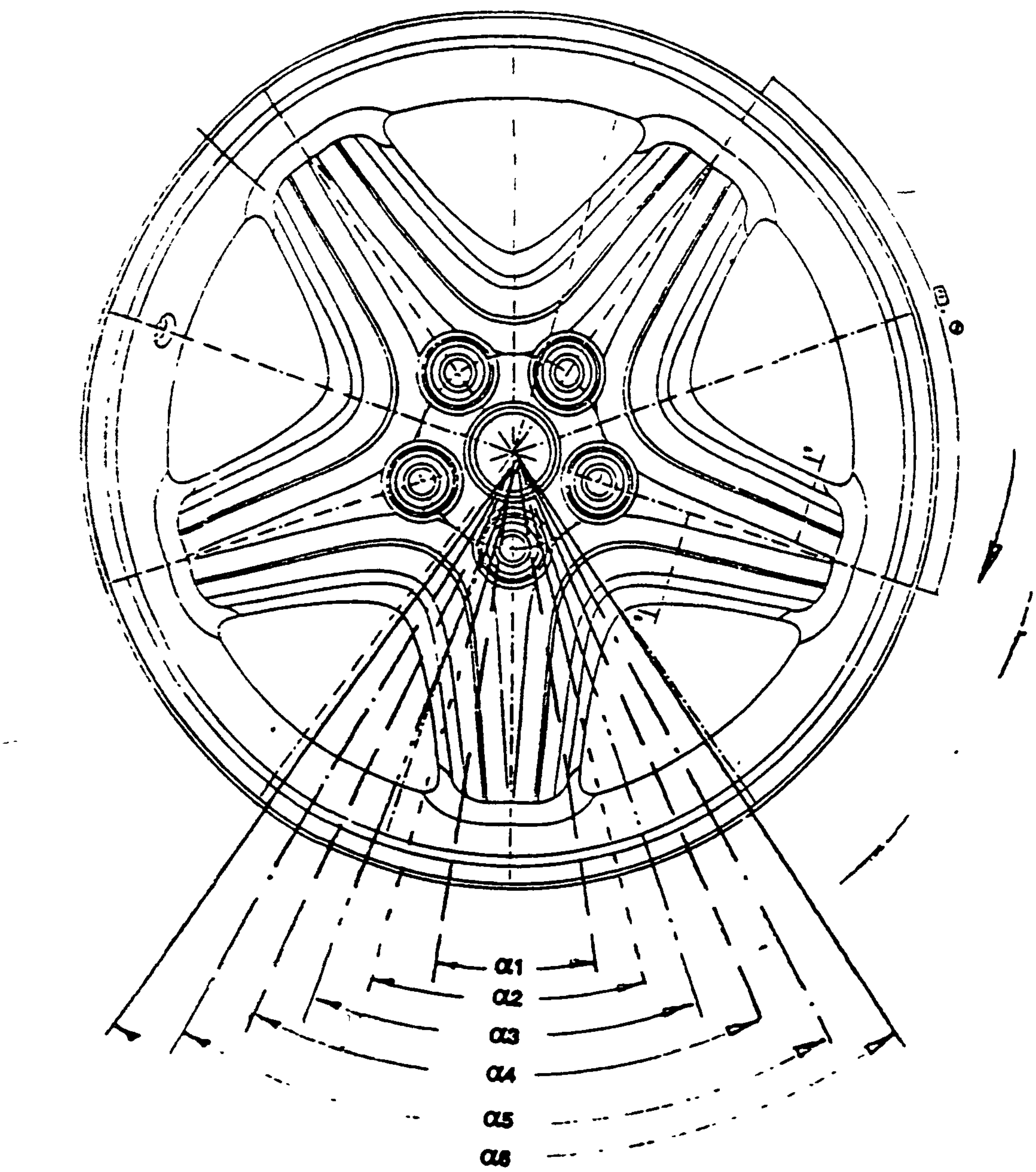
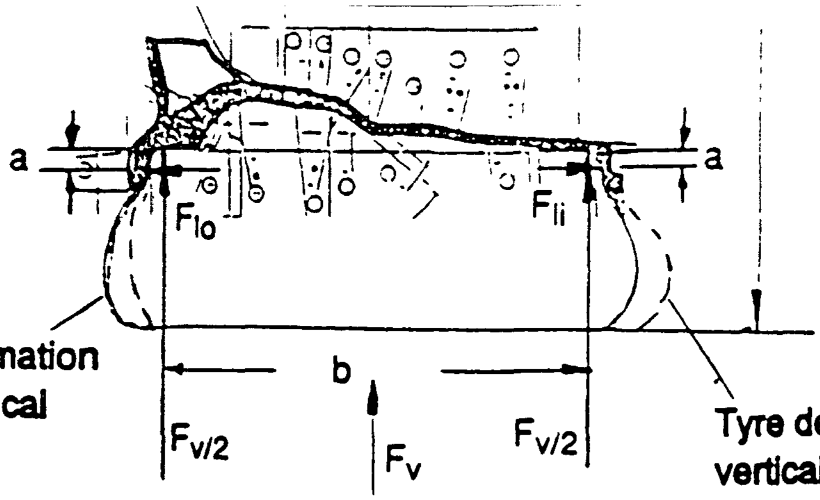
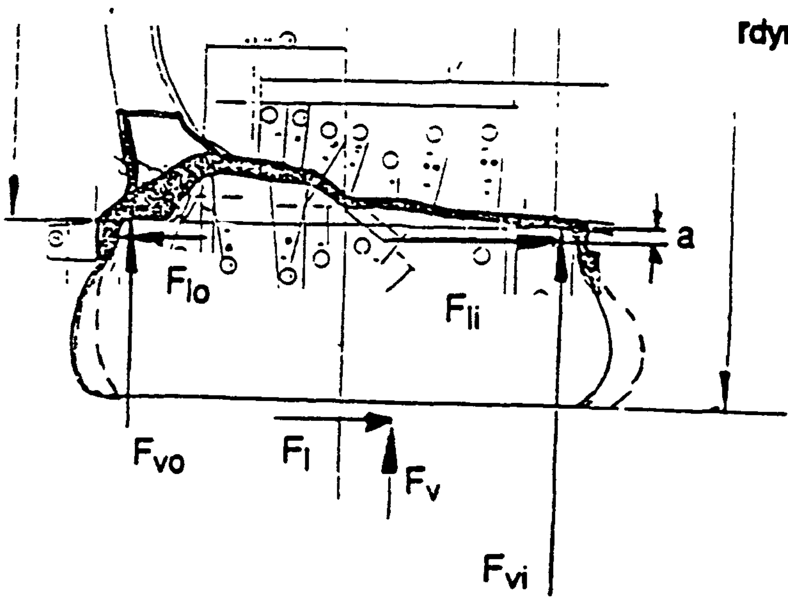


Figure 9. **Parabolic load distribution on wheel rim.**



Distance of lateral force position : a = 5 mm / 10 mm
Lateral force position : Fli = Flo = 0,1 ; 0,2 ... 0,5 · Fv / 2
Distribution : Parabolic
Distance of vertical force b = 190 mm

rdyn = 305 mm



F_{vi} : $0{,}5 \cdot F_v$; $0{,}6 \cdot F_v$; $0{,}7 \cdot F_v$; $0{,}8 \cdot F_v$

a = 10 mm = const.

F_{vo} : $0{,}5 \cdot F_v$; $0{,}4 \cdot F_v$; $0{,}3 \cdot F_v$; $0{,}2 \cdot F_v$

F_{li} : F_l ; $1{,}3 \cdot F_l$; $1{,}5 \cdot F_l$

F_{lo} :

Figure 10. Distribution of the vertical and lateral loads over the inner and outer wheel rim.

3.0 EXPERIMENTAL TEST DATA

3.1 General

There are several different methods in operation today for testing wheels They can be divided into three categories

(1) Test method to find stresses imposed on the wheel during operational loading
For example The Roll Test Facility for stress analysis

(2) Test method to find the allowable stresses on the wheel For this the tests can be carried out on the whole wheel without the tyres or on a uniaxial test specimens
For example Drop centre Rim Test and the Rotating Bending Test

These tests are also being used for rapid Quality Control tests in production

(3) Durability test method for wheels
Historically the Dynamic Radial Fatigue Test and the Dynamic Cornering Fatigue Test have been used to determine the Fatigue Life of the wheel They have both shown to be limited in predicting the Fatigue Life of the complete wheel. The Biaxial Wheel test facility, on the other hand, which is quite a recent development in wheel testing has been proven to be a better option in Durability testing

Figures 11, 12, 13 and 14 show the set ups for the above testing methods [49]

3.2 Test method and set up

The stresses which occur in individual areas of the wheel are best determined by experimental stress analysis with strain gauges. The Roll Test facility is often used for this. Here a wheel with the tyre mounted is rotated on a surface of rollers. In this study this is the test method used.

To measure the stresses on the wheels, firstly the location of the strain gauges must be determined. It is absolutely necessary to make sure that they are measuring in the critical areas of the wheels. Firstly a qualitative stress analysis with a brittle or photostress can be used [50, 51].

The stresses on the wheel caused by the inflation of the tyre and mounting (bolting) the wheel to the hub are determined first. Then the wheel is rolled under different loads in the test facility and the stresses determined [52]. Photographs can be seen in Figures 15 and 16 of the wheel being tested under straight driving and cornering loading conditions in LBF.

The Flat Base Roll test machine can replicate actual road service conditions for passenger car wheels. These service conditions were previously analysed by LBF by testing the car on typical European roads. Figure 17 and 18 show the car and wheel being tested.

This data is then converted into a program for the Flat Base Roll test machine. The strains measured by the strain gauges at the different locations of the wheels are converted into stresses according to the fatigue criteria for the given material behaviour [51].

3.3 Test results

The test results for straight and cornering load cases for the aluminium alloy wheel were very generously made available to myself by LBF. They are presented in the form of stress- time histories for each strain gauge location. This, together with the maximum and minimum principal stress values, is the total available data.

In the next chapter a comparison will be made between this data and the Finite Element calculations

Maximum and minimum principal stresses have been used for the basis of comparison with the measured stress For any position and orientation of the gauge, the stresses derived from the measured strains will change from maximum principal to minimum principal during the cycle, particularly where bending is the dominant mechanism, as is found in the spokes.

Provided that the gauge is oriented to a principal direction at the peak of a cycle, then the choice of principle stress and comparison with measured data is a reasonable at the extremities of the wheel cycle.

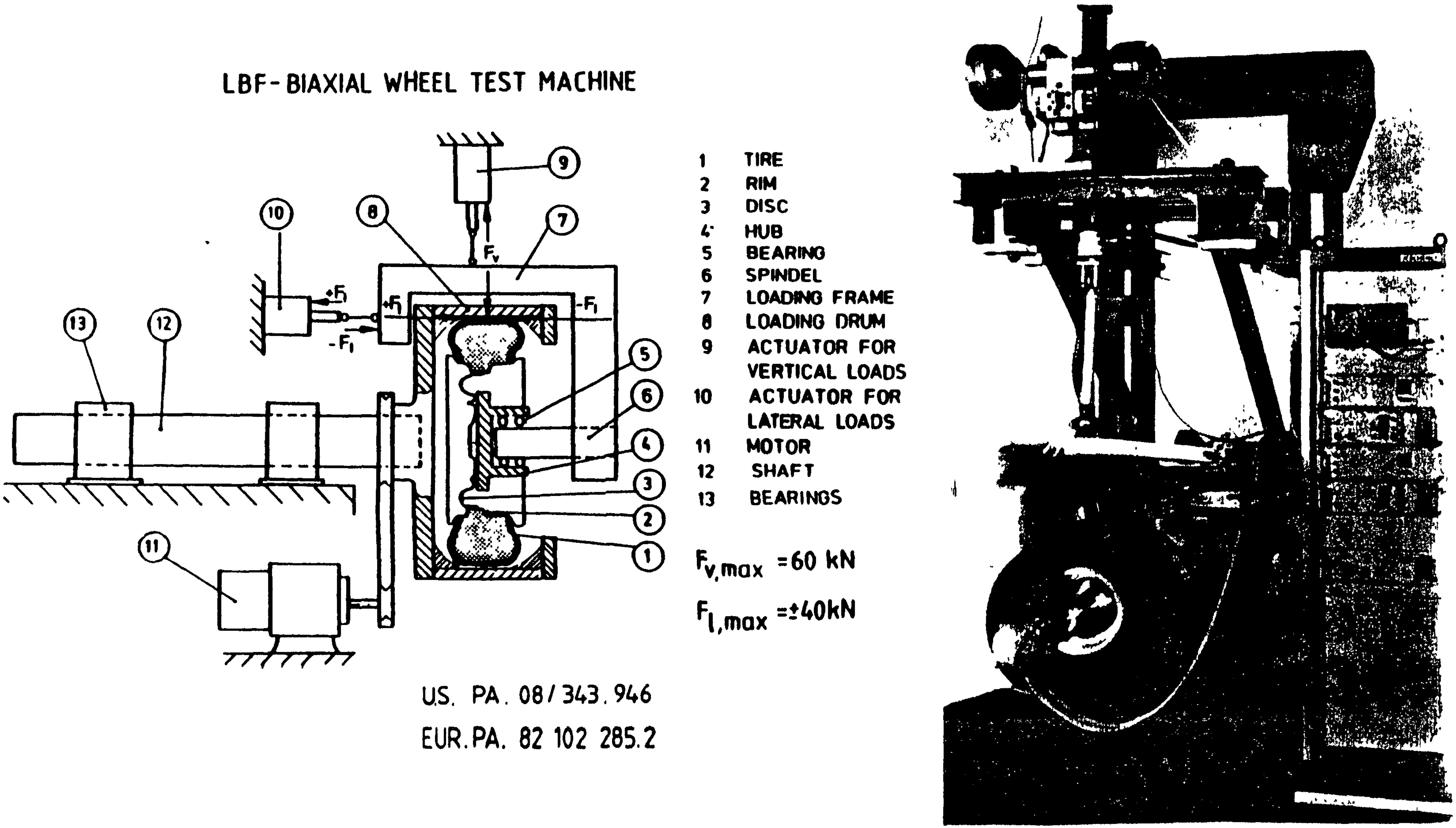


Figure 11. Roll test facility for stress analysis.

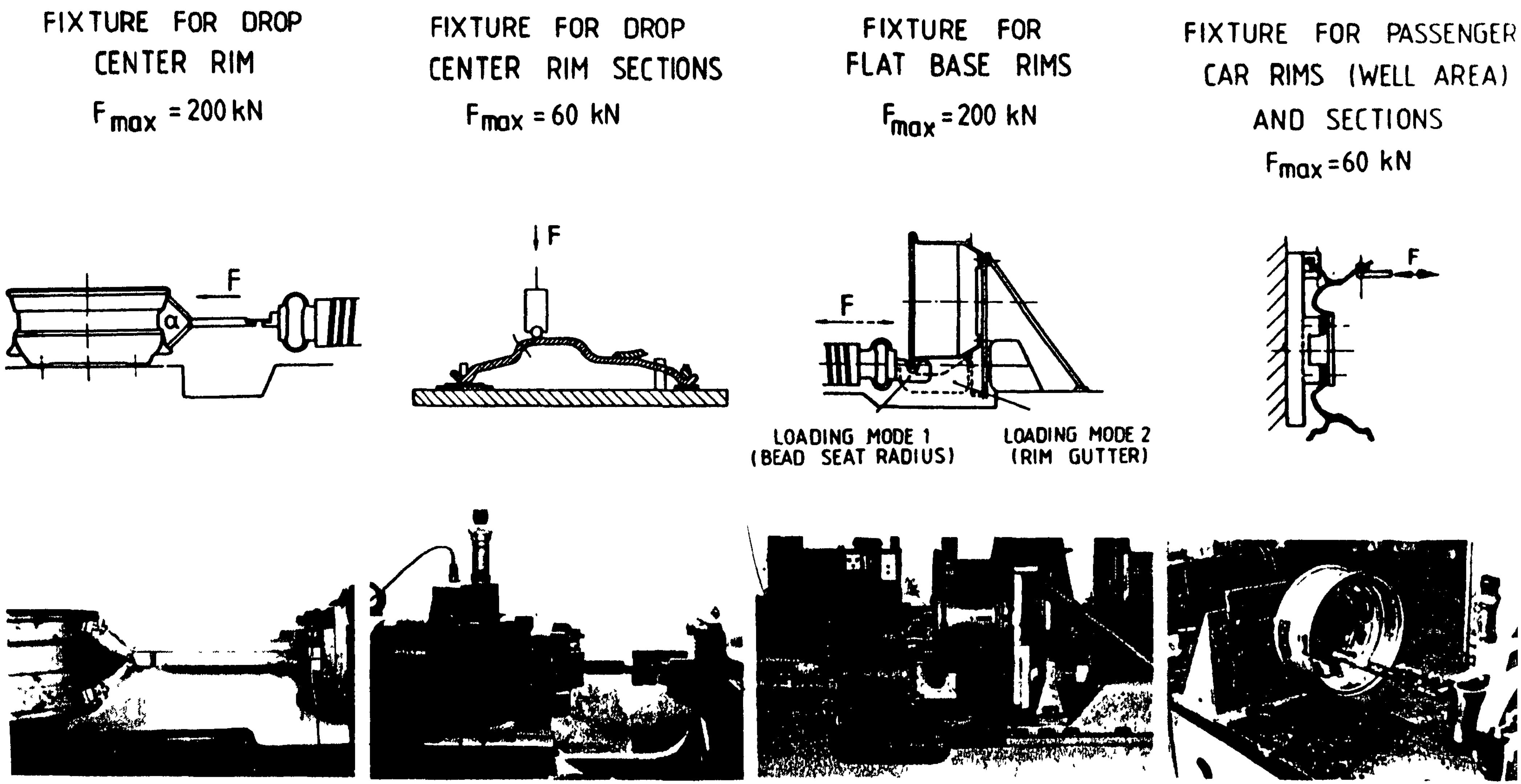
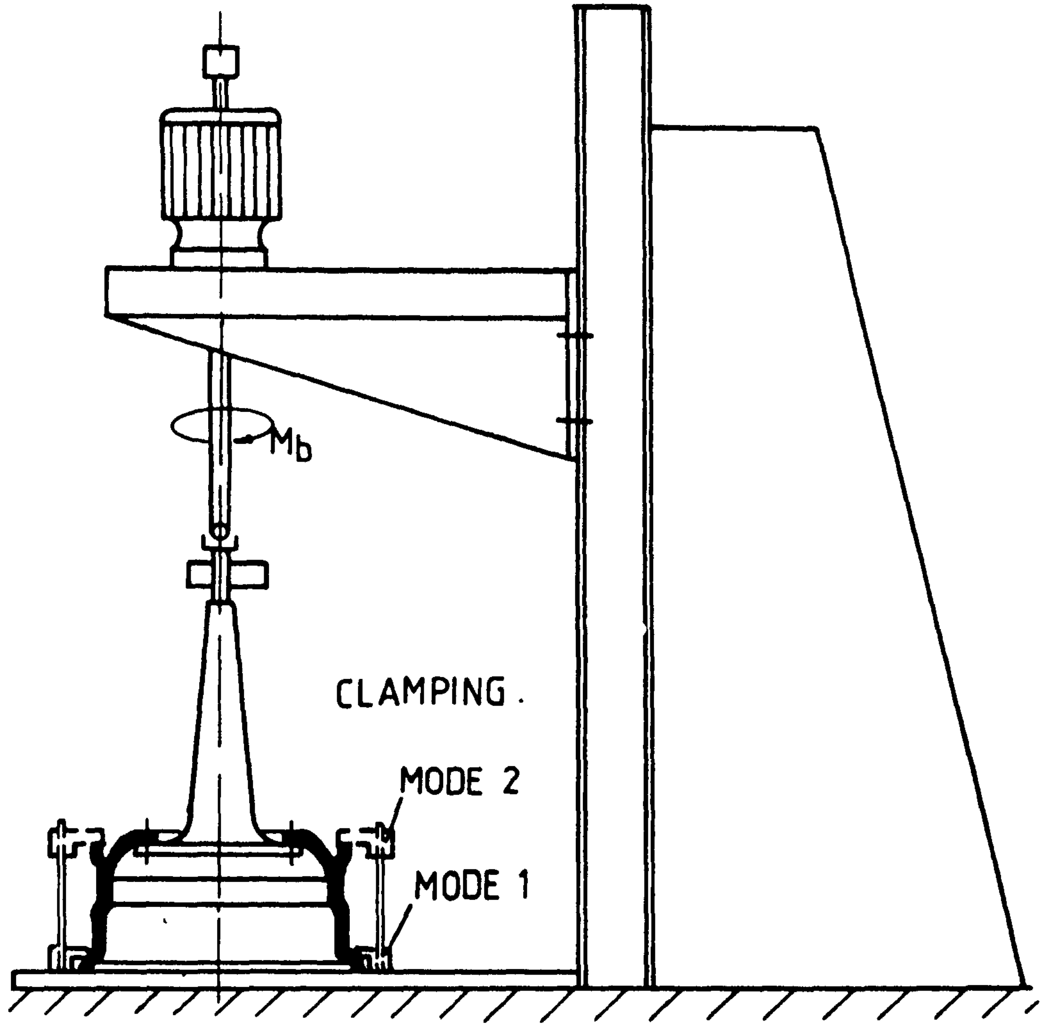


Figure 12. Test fixture for determining the fatigue strength on wheel rims.



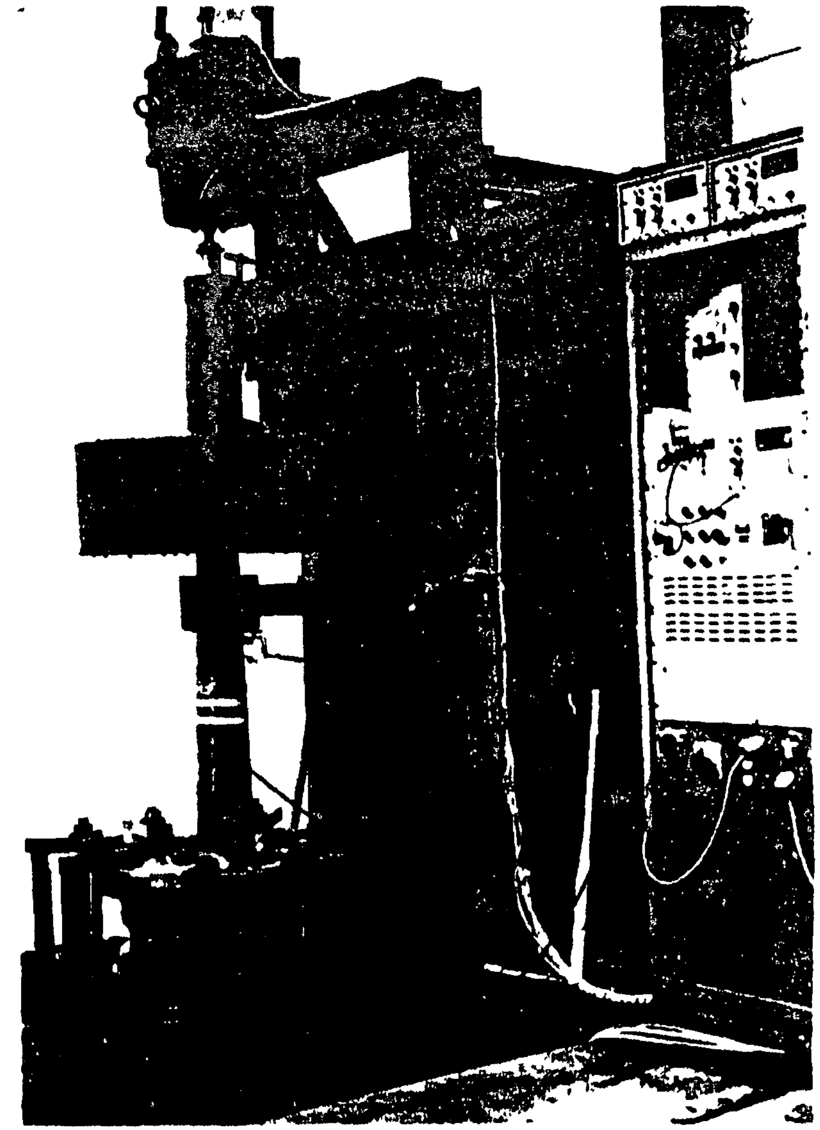


Figure 13. Rotating bending test machine with ex centre excitation.

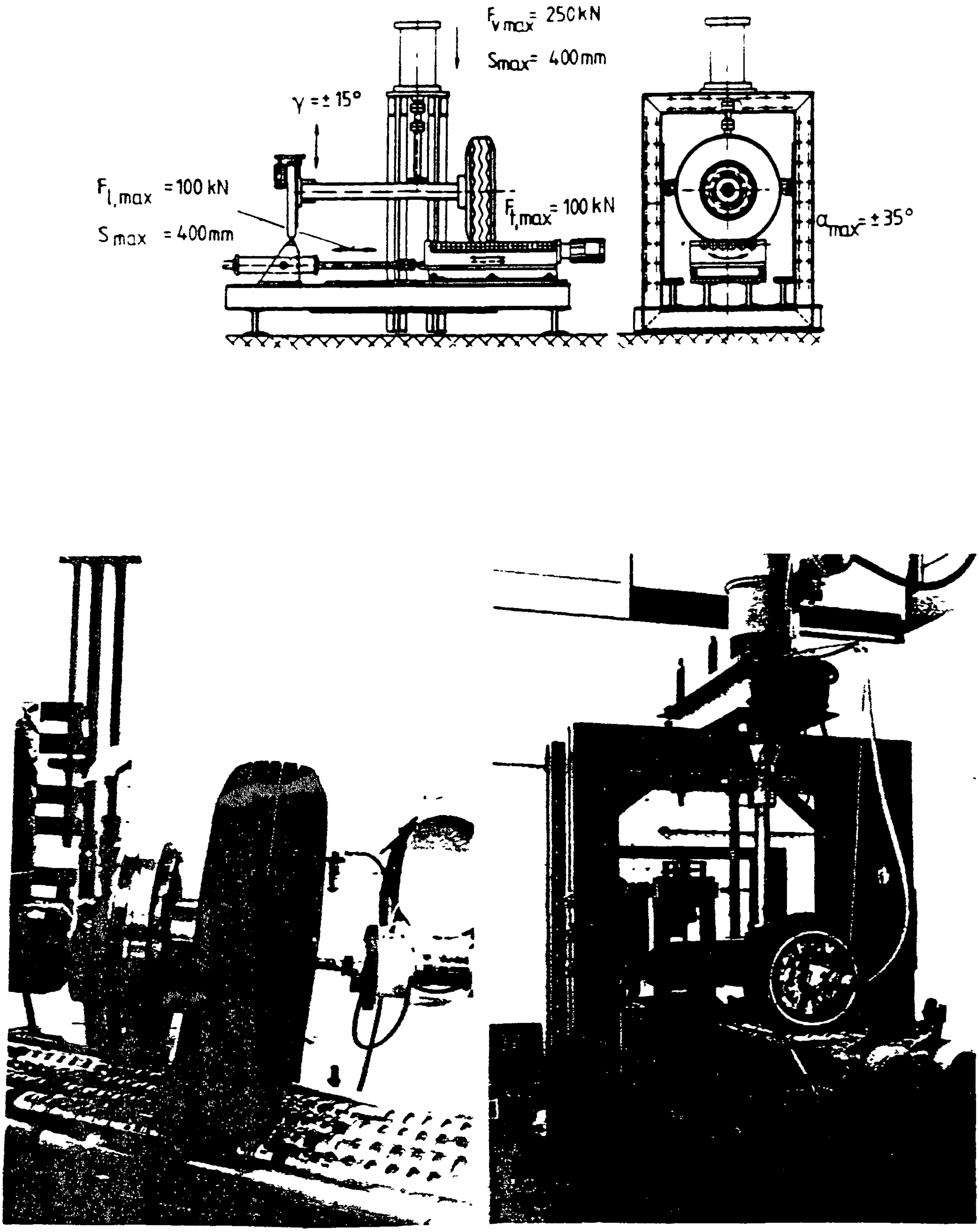


Figure 14. Biaxial wheel test facility for durability testing of passenger car wheels.

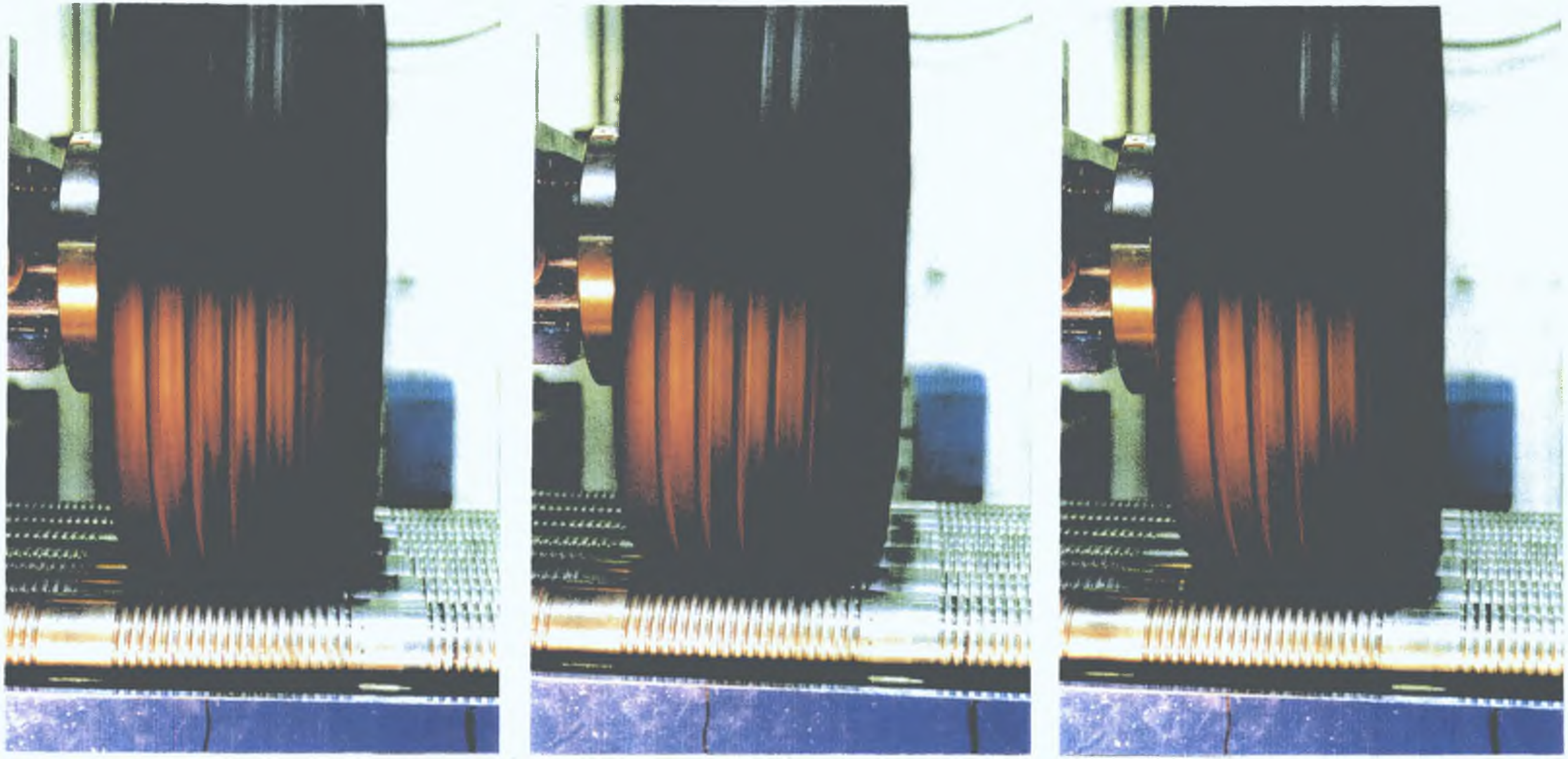


Figure 15. Photo of the aluminium alloy wheel being tested under straight driving loading conditions.

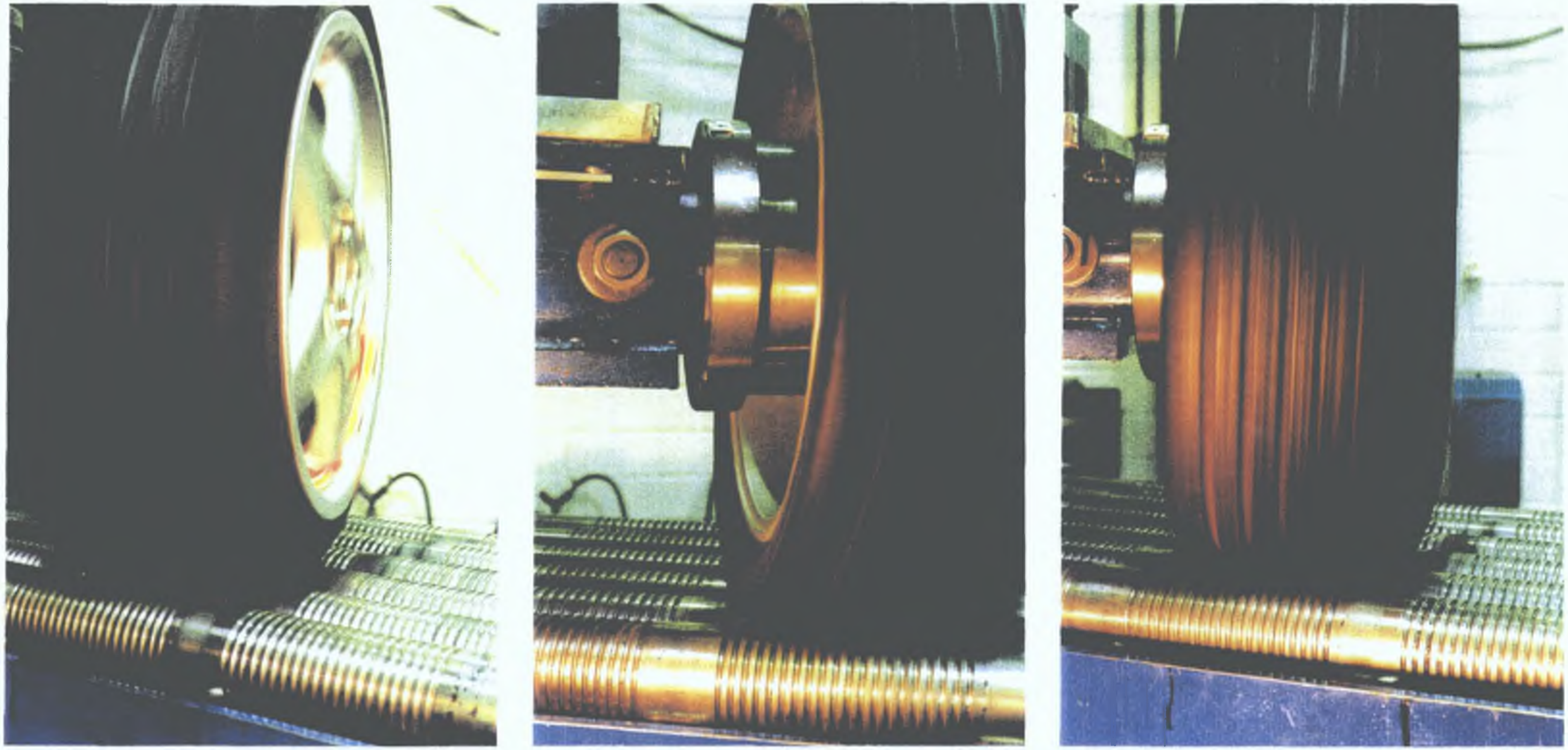
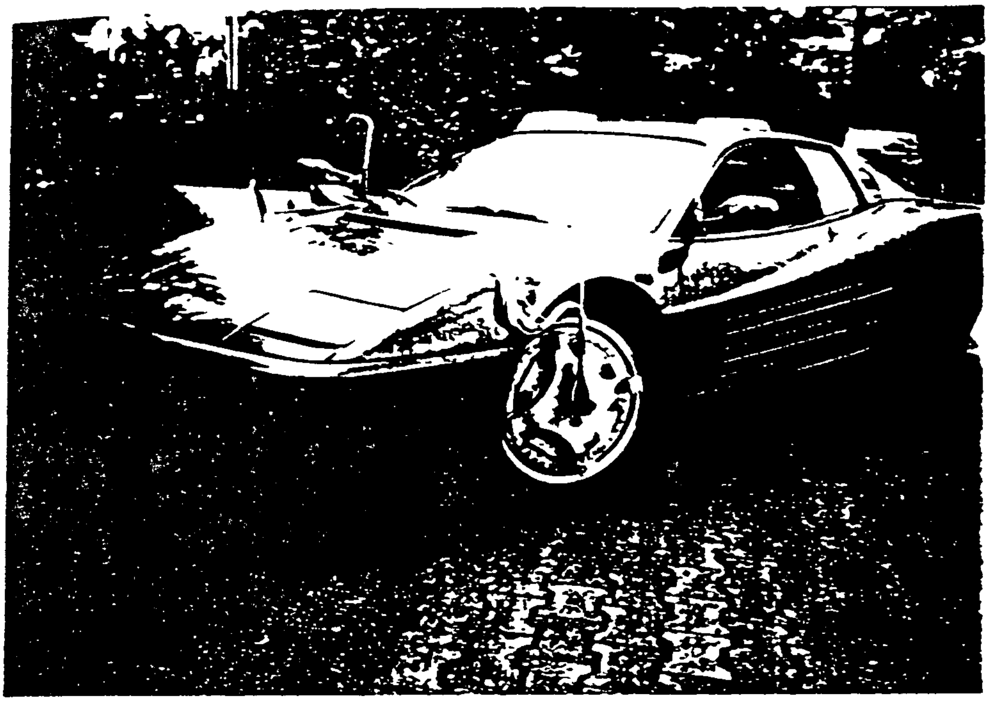


Figure 16. Photo of the aluminium alloy wheel being tested under cornering loading conditions.



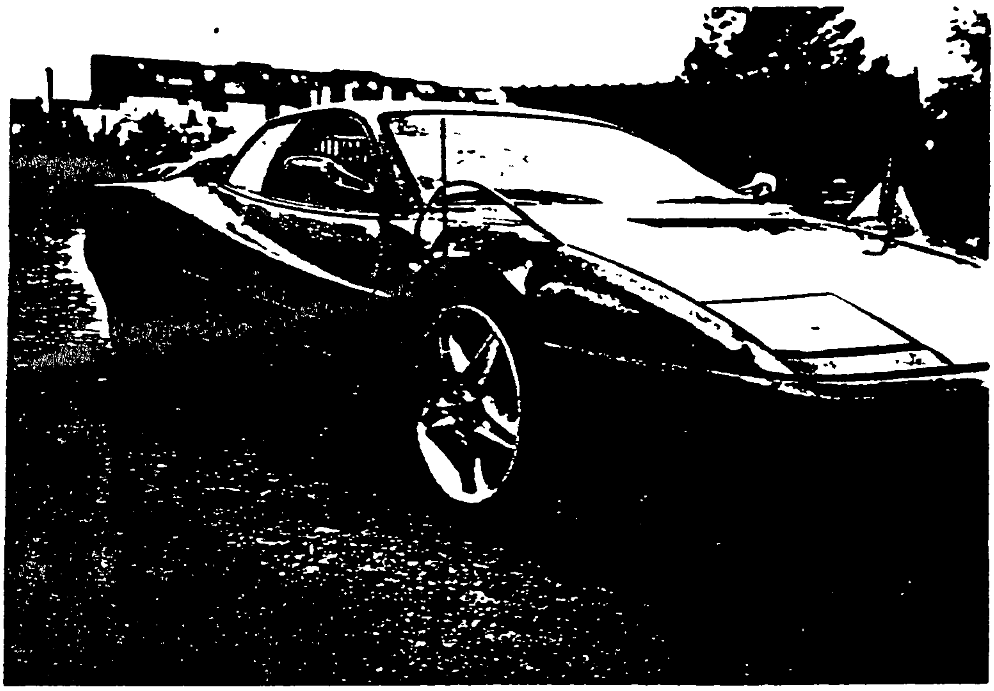
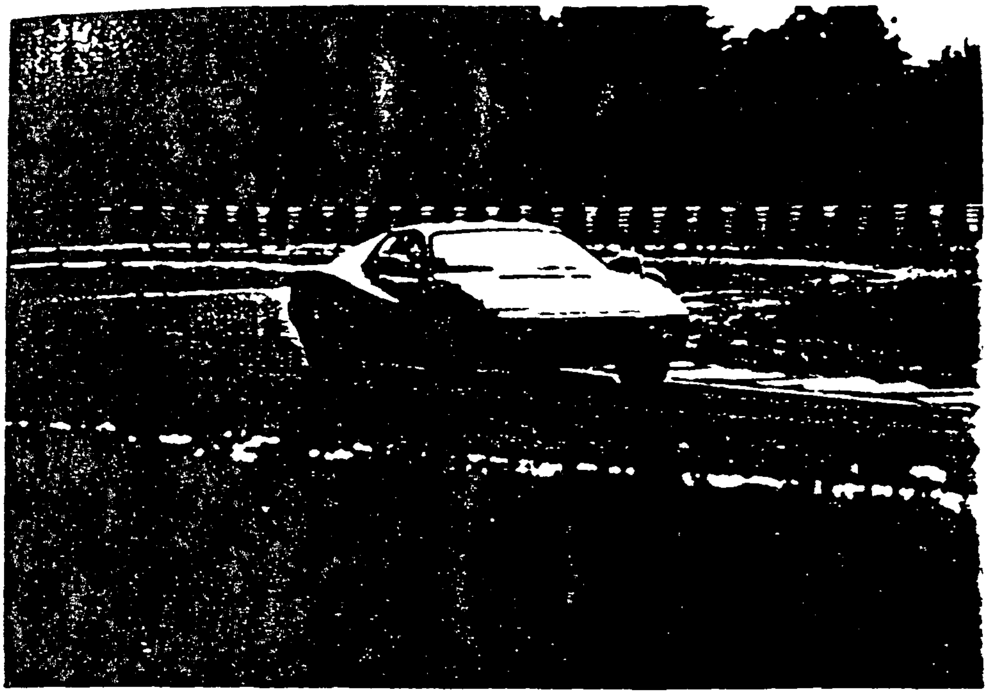


Figure 17. Test set up of the car under road service conditions.



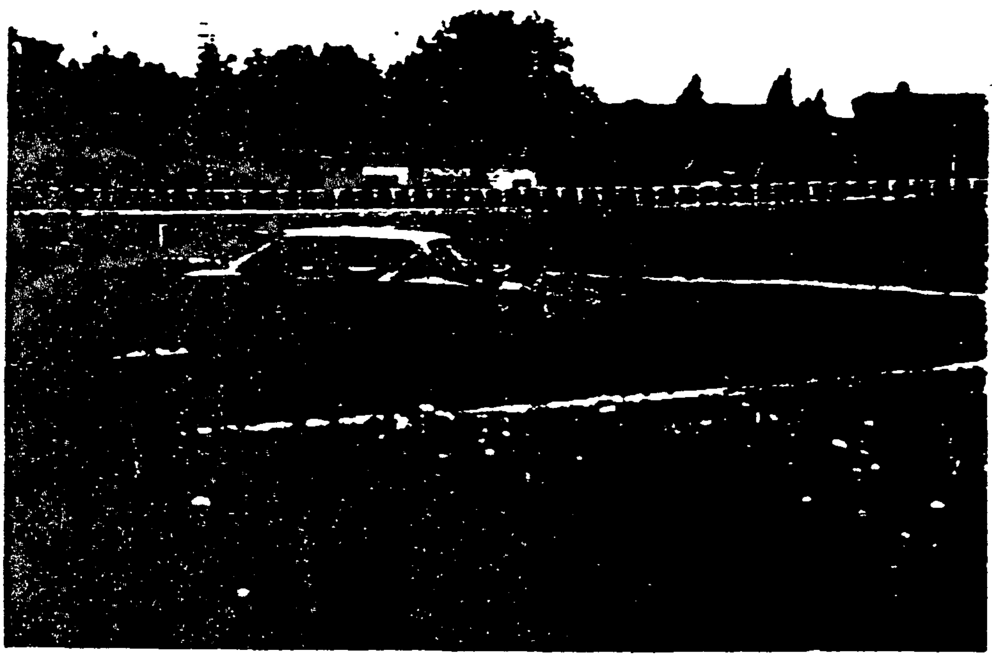


Figure 18. Car measuring service conditions.

4.0 ANALYSIS AND RESULTS

During the study most of the effort invested in comparing the calculated stress with strain gauge stress results has been concentrated in the spoke area of the wheel. This is because these gauges are the least affected by external factors related to the tyre etc. and they are the critical design locations.

The first step in the in analysis of the wheel behaviour is to prove that the model response closely represents the actual wheel response under loading. As mentioned previously the type of finite element analysis being used here is linear elastic, thus if the geometric data, loading and boundary conditions as well as the material properties are accurately represented by the model, the results should represent the actual case.

At this stage it worth describing the convention used for spoke numbering. As viewed from the outside of the wheel, spoke 1 is vertically upwards, i.e. at an angle of 0 degrees and the spokes are numbered in a clockwise sense so that spoke 2 is at 72 degrees etc. Figure 19

Initially to prove that the model was symmetric for a straight driving load case a load case was applied over a full revolution of the wheel and the maximum and minimum principle stress resultants were studied at different locations on the wheel.

(a) On a straight line through the centre of the wheel rim and spoke (see Figure 20).

(b) Offset locations on the spoke only (see Figure 21).

The results displayed in Figure 22 were found. The graph shows that the model reacts symmetrically to loading. For example when the load is applied to spoke 1, the maximum and minimum principal stresses induced on spoke 2 and 5 are equal and almost likewise on spokes 3 and 4. This initial test was carried out on different areas of the wheel.

It is also important to look at the stress patterns of points which do not lie along the centre of symmetry of the spoke These points are located at the offsets along the spoke and rim In Figure 23 the results of the straight driving load case are applied The resultant maximum and minimum principal stresses for the point 1 when rotating clockwise and point 4 when rotating anti-clockwise are practically identical This, together with the previous tests proves that the wheel model reacts in a symmetrical manner under the straight driving load case

The next step in the analysis is to make a comparison of the calculated principal results to the results measured from the wheel being tested under conditions identical to the finite element model

To compare the calculated stress results with the measured strain gauge ones, initially, a graph of maximum and minimum principal stress versus distance along the spoke was produced for the different strain gauge locations See Figures 24 and 25 for an example showing stress along spoke 1

This was then compared to the measured stress results at the corresponding angle (in this case the angle = 0 degrees)

The measured strain gauge results have been captured over the complete rotation of the wheel (360 degrees). This has been modelled by applying a load to the static wheel and recording the results of the five spokes Thus, for each 360 degree rotation, 5 calculated points for each strain gauge are located.

By rotating the load on the static wheel by 36 degrees a further 5 calculated points for each strain gauge are found giving a total of 10 points.

It was found from testing several different load distributions that the optimum solution was found by applying 60% of both the vertical and horizontal load to the outer edge of the wheel

The model was solved in I-DEAS Model Solution linear statics to obtain displacements and stresses for each of the unit load cases

An output list file for a sample run can be seen in Appendix D Here the unit load cases were checked for applied load and reaction load to ensure that the loading had been correctly applied and that no erroneous moments had been introduced

The load cases were scaled and combined in the Post Processing module of I-DEAS New load cases containing the following combinations were created for comparison to stresses obtained from strain gauge measurements

Fv 10.6kN Straight Drivıng case

* Fvo . 0.5 Fv, 0 6 Fv, 0 7 Fv, 0 8 Fv

* Fvi · 0.5 Fv, 0 4 Fv, 0 3 Fv, 0 2 Fv

* Fho = Fhi = 0 5 Fv/2

Fv 6.72 kN + Fh 5.4 kN Cornerıng Case

* Fvo 0 6 Fv
* Fvi 0 4 Fv
* Fho · 1 0 Fh
* Fhi : 0.0 Fh

For a second analysis the parabolic loading distrıbution was applied over an angle of 60 degrees centred directly between spoke 1 and 2, i e at an angle of 36 degrees

The calculated stress results for each load case are presented in the form of

(a) Table showing the maximum and minimum principal stresses at gauge locations along each spoke and

(b) Stress-time history at each 'gauge location for a single rotation of the wheel The FEA calculated results are effectively a snapshot in time

Measured results at the same locations have been obtained from the dynamic test.

Appendix E shows the tables and stress time histories for both straight driving and cornering cases Results for several different loading parabolic distributions are shown, as well as results for geometry variations, and these will be discussed later on The location of the strain gauges on the wheel is also shown here

For the initial analysis conducted with the load distribution centred on spoke 1, the stress distribution of immediate interest is in spoke 1. However for this case the stress distributions that are predicted in spokes 2, 3, 4 and are also valid points for comparison with the measurement stress-time histories They can be considered as 4 other points on the stress time history that occur as the wheel rotates The stress distributions in spoke 2 are valid for an anticlockwise wheel rotation of 72 degrees, the stress distributions in spoke 3 are valid for an anticlockwise wheel rotation of 144 degrees and so on. In this way 5 points on the stress-time history can be obtained from a single position of load application.

From solutions with the loading distribution applied at different angles between two adjacent spokes e g spoke 1 and 2, an additional set of 5 points on the stress-time history have been obtained

The same argument is true for the rim region and this has been used to derive stress-time histories for comparison to measured results

Initial comparisons were focused on the spokes as the measurements indicated that these resulted in the highest values of stress Stress distributions from the model were obtained radially along the spokes for comparison with the following groups of gauges:

* gauges 1, 2, 27, 3 and 4 running up the rear outer edge of the spoke ribs.

* gauges 31, 32 and 33 running up the rear inner edges of the spoke ribs

* gauges 6, 28 and 29 running up the rear edges of the central rib

* gauges 27, 31, 28 and 30 running across the rear edges of the spoke at approximately the same radial station.

* gauges 7, 36 and 35 running up the outside of the central rib.

For each gauge location on the spokes, stress values were taken from the nearest node in the model to build up points on the stress-time history.

A similar exercise was conducted for selected gauges on the rim.

As mentioned previously, maximum and minimum principal stresses have been used as the basis for comparison with the measured stresses. For any position and orientation of the gauge, the stresses derived from measured strains will change from maximum principal to minimum principal during the cycle, particularly where bending is the dominant mechanism, as in the spokes. Provided that the gauge is oriented to a principal direction at a peak of a cycle, then the choice of principal stress and comparison with measured data is reasonable at the extremities of the cycle. However, there is a considerable uncertainty between the extremes of the cycle. Direct comparison of strains in the gauge orientation are possible in the model and would be expected to give a higher level of correlation.

Fv 10.6kN Straight Driving case

Upon examination of the results from the initial case of Fvo = Fvi = 0.5 Fv it was found that there was a net moment on the wheel causing too much tension on the outside of the spoke and too much compression on the inside of the spoke compared to the strain gauge results. On the outside of the spoke, the level of compressive stress in the cycle was comparable to the strain gauge results. Similarly, on the inside of the spoke the level of tensile stress in the cycle was comparable to the strain results To reduce this net moment lead to the notion of reducing the proportion of vertical load applied to the inner rim. The combination giving the best agreement to the test results was Fvo = 0.6 Fv and Fvi = 0 4 Fv This has some practical reality as the load path from the force applied at the outer rim to the hub is significantly stiffer than the load path from the force applied at the inner rim to the hub and should therefore attract more load

The lateral load components were distributed in the same manner so that Fho = 0 6 x 0.5 Fv/2 and Fhi = 0 4 x 0 5 Fv/2

The predicted stress-time histories for this case are shown in Appendix 5 for gauges 2, 3, 6, 28, 29, 30 and 35 The minimum and maximum values in the cycle are listed in the Table 1

Fv 6.72 kN + Fh 5.4 kN Cornerıng Case

The cornering load case of Fv = 6 72kN and Fh=5.4kN takes into consideration the assumption of the best combination of loading to reduce net moment of Fvo=0 6Fv and Fvi=0.4Fv, as well as the net lateral component Fh=5.4kN

Upon examination of results we see that the stress calculation points correlate quite well with the stress-time histories This can be readily see in Appendix 5 for gauges 2, 3, 6, 28, 29, 30 and 35 The minimum and maximum values in the cycle are listed in the Table 2.

Comparison of the calculated stresses to the measured stress shows that there are some differences in the results even though there is a good correlation between them

These differences could be due to errors made during test measurements, computational inaccuracies, or a combination of both of these It is difficult to improve this correlation but efforts are made to increase the accuracy of the finite element model

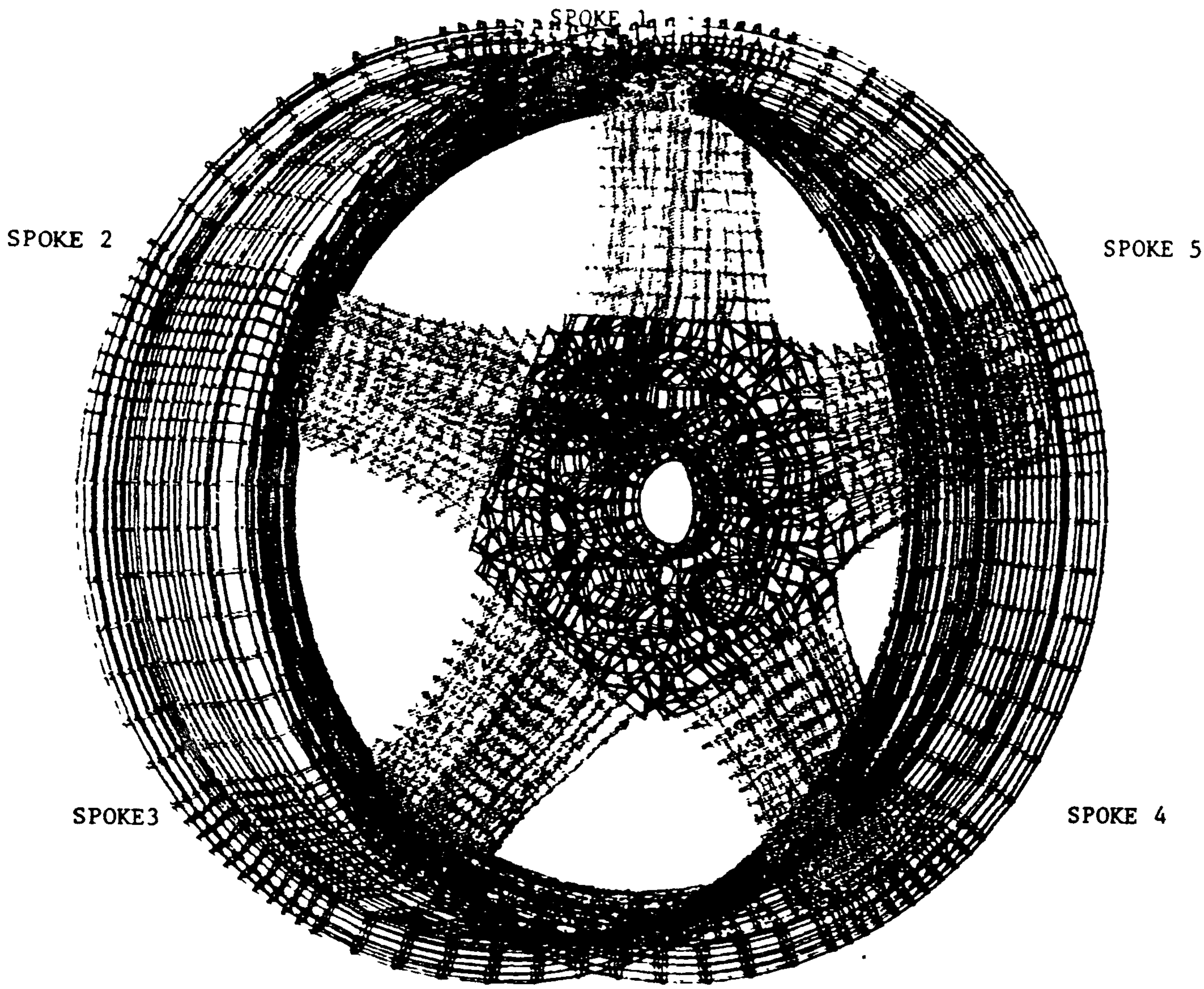


Figure 19. Convention used for spoke numbering.

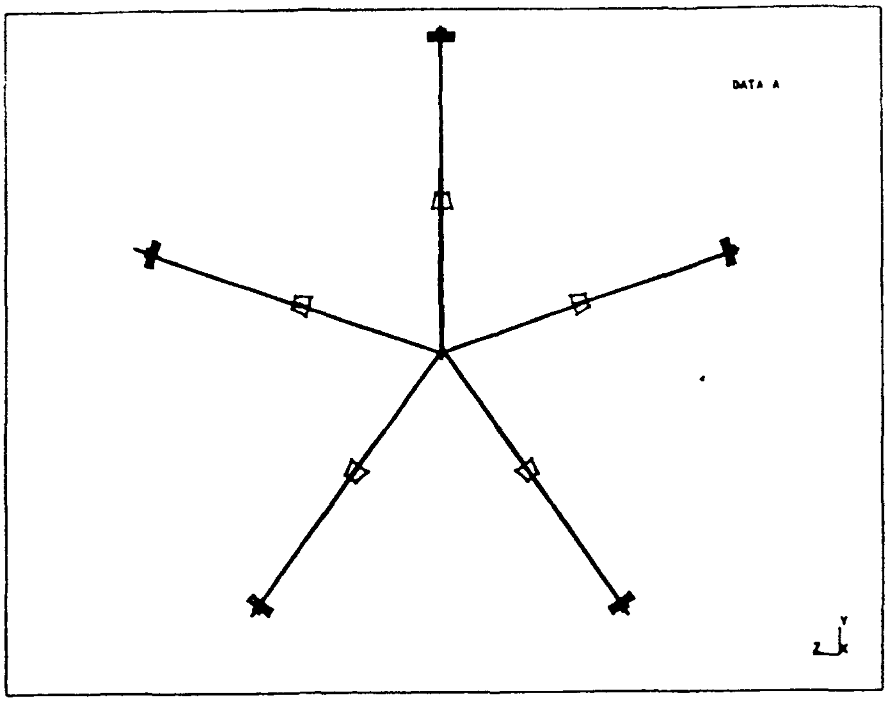


Figure 20. Elements on the centre line of the wheel.

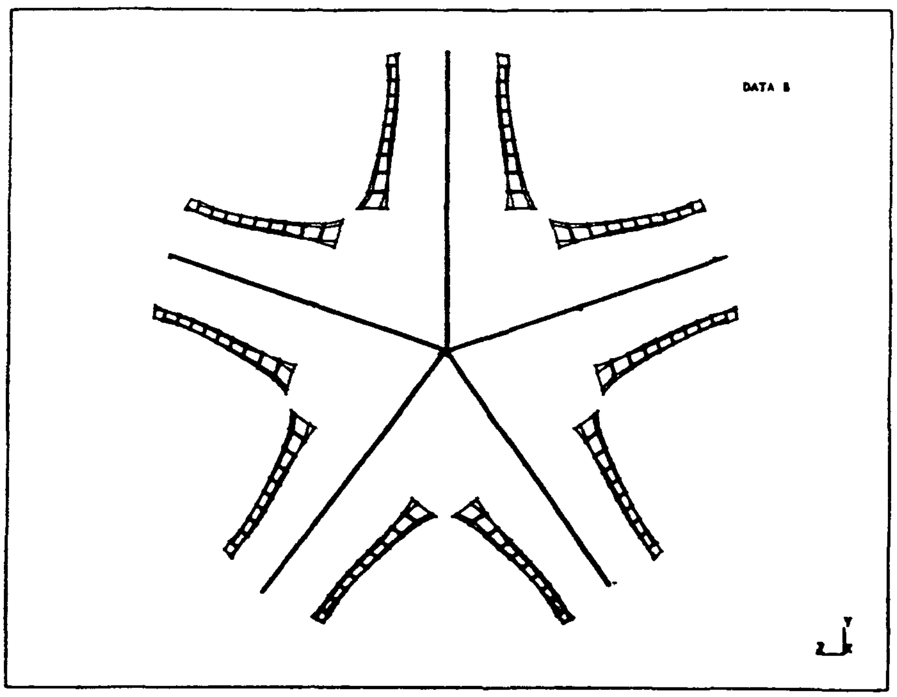


Figure 21. Elements at off-set locations on the spoke only.

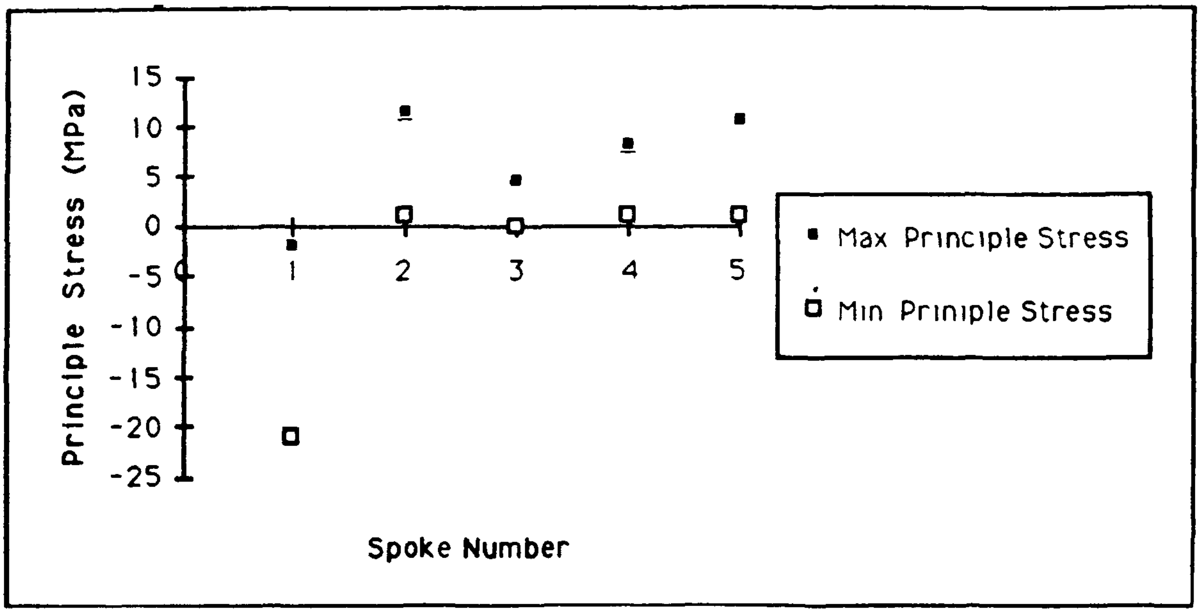


Figure 22. Principle stress versus position on the wheel for spoke elements located at the centreline positions on the spoke near the hub.

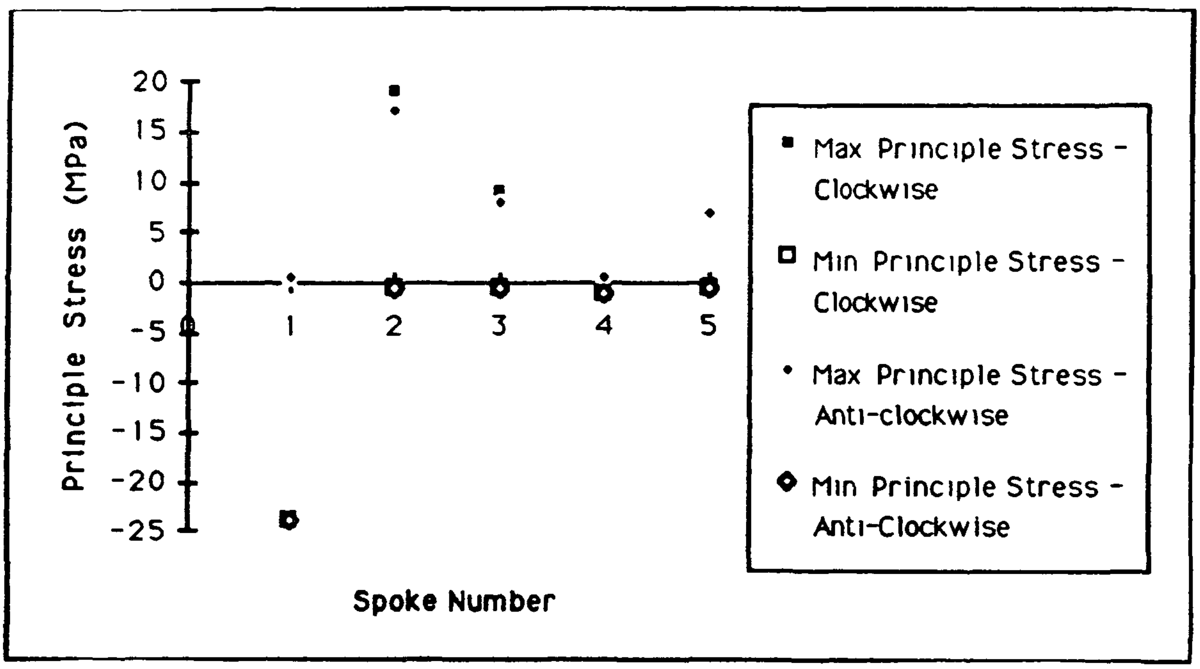


Figure 23. Principle stress versus position on the wheel for spoke elements located at the offset positions on the spoke near the hub.

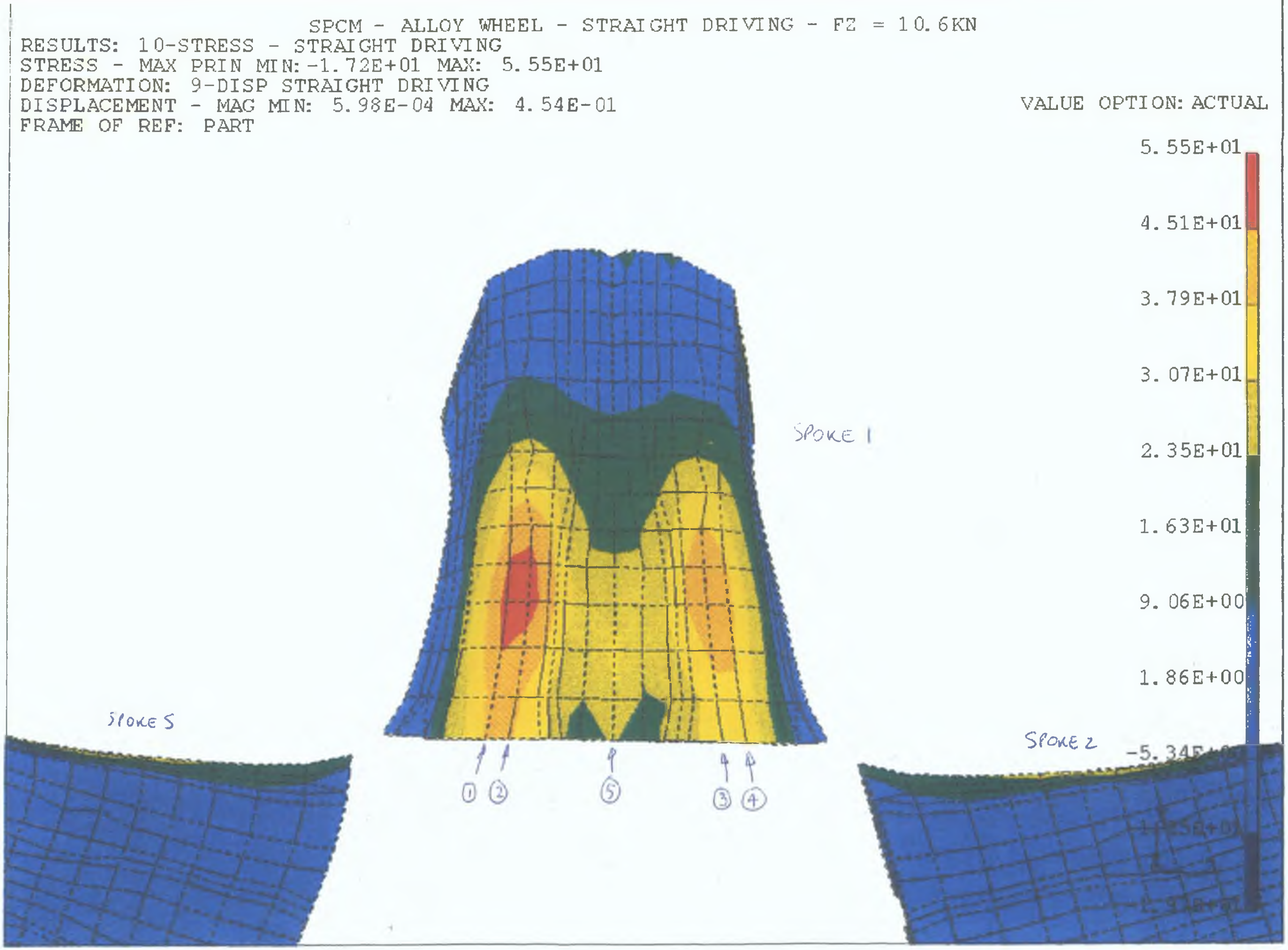
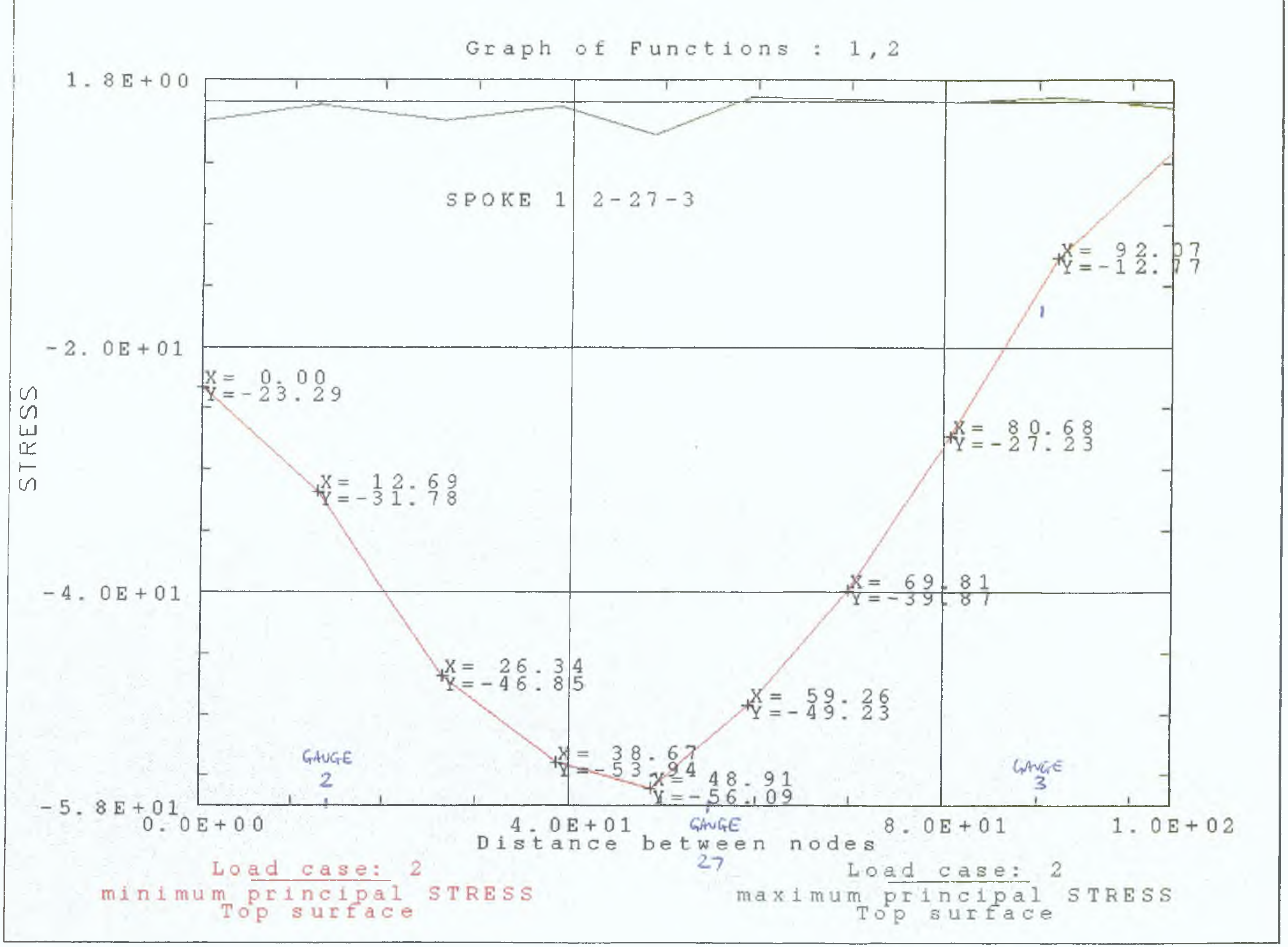


Figure 24. Stress vs distance along spoke 1 for straight driving load case



Graph

Figure 25.

Gauge Number	Measured max	Measured min	Calculated max	Calculated min	Measured alternating	Calculated alternating	% difference in alternating results
	Spoke gauges						
1	13	16			14.5		
2	18	24	18	21	21	19.5	7 %
3	15	20	14	11	17.5	12.5	29 %
4	21	17			19		
5	5	12			8.5		
6	15	29	14	30	22	22	0 %
7	15	7	17	10	11	13.5	23 %
27	24	30	29	35	27	32	19 %
28	12	23	13	26	17.5	19.5	11 %
29	9	13	8	9	11	8.5	23 %
30	25	35	29	29	30	29	3 %
31	24	36	21	25	30	23	23 %
32	21	15	30	14	18	22	22 %
33	13	16	32	21	14.5	26.5	86 %
34	4	3			3.5		
35	7	5	8	8	6	8	33 %
36	13	7	10	10	10	10	0 %
	Rim gauges						
8	9	13	10	12	11	11	0 %
9	4	2	30	7	3	18.5	516 %
10	9	6			7.5		
11	6	7			6.5		
12	6	4			5		
13	6	5	10	6	5.5	8	45 %
14	9	8			8.5		
15	4	3			3.5		
16	12	10	43	20	11	31.5	186 %
17	9	6	19	7	7.5	13	73 %
18	2	2			2		
19	7	5			6		
20	4	6			5		
21	4	4			4		
22	5	1			3		
23	15	36	19	89	25.5	54	112 %
24	1	4			2.5		
25	5	5			5		
26	8	10			9		

Note : When no value is present in table cell, this means that no result has been calculated for that strain gauge location.

TABLE 1. **Straight driving (10.5kN = Fv) Maximim and Minimum stresses. (All stresses are measured in MPa).**

Gauge Number	Measured max	Measured min	Calculated max	Calculated min	Measured alternating	Calculated alternating	% difference in alternating results
	Spoke gauges						
1	17	29			23		
2	24	41	22	41	32.5	31.5	3 %
3	29	28	36	14	28.5	25	12 %
4	36	23			29.5		
5	9	21			15		
6	24	51	16	62	37.5	39	4 %
7	28	12	46	11	20	28.5	43 %
27	31	39	15	17	35	16	54 %
28	14	28	10	15	21	22.5	7 %
29	14	14	14	11	14	12.5	11 %
30	33	46	37	59	39.5	48	22 %
31	30	50	30	38	40	34	25 %
32	30	21	42	22	25.5	32	25 %
33	24	24	70	28	24	49	104 %
34	6	2			4		
35	8	7	9	13	7.5	11	47 %
36	19	8	26	12	13	19	46 %
	Rim gauges						
8	12	17	15	17	14.5	16	10 %
9	1	3	10	15	2	12.5	525 %
10	2	7			4.5		
11	7	8			7.5		
12	0	6			3		
13	8	9	12	17	8.5	14.5	71 %
14	12	14			13		
15	2	4			3		
16	16	18	20	32	17	26	53 %
17	11	1	13	10	6	11.5	92 %
18	3	4			3.5		
19	15	4			9.5		
20	10	5			7.5		
21	4	8			6		
22	8	1			4.5		
23	10	34			22		
24	2	5			3.5		
25	3	3			3		
26	16	12			14		

Note : When no value is present in table cell, this means that no result has been calculated for that strain gauge location.

TABLE 2. Cornering (6.72kN = Fv & 5.4kN = Fh) Maximim and Minimum stresses. (All stresses are measured in MPa).

5.0 COMPARISON AND DISCUSSION OF FE AND MEASURED RESULTS

5.1 Results

The predicted results have been compared to the stresses obtained from test plotting of stress-time histories These have been produced for many of the gauge positions on the spokes and rim except those in the close vicinity to application of load in the FE model The predicted values are likely to be artificially high in this area and should therefore not be used in the comparison Gauges 10, 11, 12, 19, 20, 21, 22 and 24 fall into this category

Fv 10.6kN Straight Drιvιng case

For the straight driving case of Fv = 10 6 kN the predicted stress-time histories are shown in Appendιx E for gauges 2, 3, 6, 28, 29, 30 and 35 The maximum and minimum values in the cycle are listed in Table 1

For gauges located on the spokes (1 through 7 and 27 through 36), the comparison is generally quite good It ιs evident that the maximum (tensile) stress in the cycle shows a better level of agreement with the test values than the minimum (compressive) stress in the cycle. If anything, the model is slightly over predicting the maximum stress and under predicting the minimum stress in the cycle

In terms of stress range, or the alternating component which is half the stress range, the comparison is good The model over predicts as much as it under predicts

For the gauges located in the rim (8 through 26) the comparison is not as good compared to the spoke For all gauge locations assessed the model over predicts both the maximum and minimum stress in the cycle This means that the alternating stress ιs also over predicted (See table 1)

Fv 6.72 kN and 5.4 kN Cornering Case.

For the cornering case of Fv = 6.72 kN and Fh = 5.4 kN the predicted stress-time histories are shown in Appendix 5 for gauges 2, 3, 6, 28, 29, 30 and 35 The maximum and minimum values of the cycle are also listed in Table 2.

For the gauges located on the spokes (1 through 7 and 27 through 36) the comparison is generally quite good but less so than the straight driving case It is evident that generally the maximum (tensile) stress in the cycle still shows a better level of agreement with the test values than the minimum (compressive) stress in the cycle For this case the model is generally over predicting both the maximum stress and the minimum stress of the cycle.

The alternating stress component is therefore generally over predicted compared with the measurements

For the gauges located in the rim, the comparison is again not as good when compared with the spoke For all gauge locations assessed, the model over predicts both the maximum and minimum stress in the cycle. This means that the alternating stress is also over predicted

From the finite element analysis it was found that the critical areas of the wheel are the strain gauge locations M2 and M3 I e where the spoke interfaces with the hub and rim on the inner wheel the stress values are particularly high in these areas under the straight and cornering driving load cases. These areas of the wheel are the design 'weak spots' and are the most likely to fail first under operational loading

Figure 26 shows the stress-strain diagram for the aluminium alloy used in the wheel, i.e Gk - Al Si 7 Mg This is a cast, age hardened aluminium alloy Figure 27 shows the S-N curves to crack initiation obtained under strain control for this material From these two Figures it is possible to see the affect that the stresses induced on the wheel have on the crack initiation.

The major problem with this method is, that it is difficult to relate the S-N strain controlled life to the actual wheel life, but, it can be used as a basis for comparison

Because the stresses at strain gauge location M2 are the most critical, this is the area that we will look at in most detail The straight driving load case is the most influential load case and likewise, it is the one chosen here to study in the most detail

The stress amplitude at M2 under straight driving loading conditions is 19 5 MPa From Figure 26 this gives a percentage strain value of 0 23% (This is obtained by recognising that 0 2% of the proof stress is used as the yield stress of the material) Figure 27 then shows that with strain value of 0 23 %, the number of cycles to crack initiation is 100,000

Generally, comparisons of strain gauge results and stresses at nodes of finite element models should be viewed with caution. The FE results may not be at the maximum stress or strain locations but they will be able to be interrogated to understand the stress gradients and distribution. If stress gradients are too severe, the worst condition may not have been captured and greater fe mesh refinement would be needed to overcome this The strain gauge results may be inaccurate for any of the following reasons,

* If the gauges are not fully bonded then full coupling of the surface and gauge will not occur

* The direction of single gauges may not align with the principal stress and the principal direction may vary during a complete wheel cycle

* In areas of high strain gradient, the length of the gauge will 'smooth' the values.

* The conversion of strain to stress for single gauges may not fully account for Poisson's effects (this is overcome for rosettes) and

* Noise and signal conditioning may give stray inaccuracies.

Only biaxial and uniaxial gauges have been used in the testing Experience and previous measurements with rosettes has been used to select locations and orientations of the gauges In the spokes most gauges have been oriented in the radial dırection

In our case we have stresses at ten points from the model to contribute to the generation of a predicted stress-time history compared to the continuos description from the test This means that the available data points from the model may not correspond exactly to the maximum or minimum values in the cycle However, examination of the stress-time histories will give a good indication as to whether this data/value mismatch condition has occurred.

The calculated values used in the comparison tables are the maximum and minimum of the values obtained from the ten points in the model. A more detailed examination of the stress-time history plots reveals a concern regarding some of the points used to build up the calculated plot

5.2 Study of loading profile

It was mentioned earlier that although the magnitude of the total load applied to the rim is known accurately, it's distribution over the rim area is assumed. Therefore it was decided to investigate the effect of this load distribution and isolate the one which correlates most accurately with the measured results

It has been noted already that the shape of the stress time histories are very similar for the measured and calculated results In the following work only the *amplıtudes of the maxımum and minımum prıncıpal stresses* are investigated

The transfer mechanism of the load from the road surface to the tyre, and from the tyre to the wheel rim is a complicated one Even now the tyre manufacturers are not sure of this load transfer function and in assumptions hitherto made, it was assumed that a single parabolic distribution transfers both the vertical and lateral loads. The angle of this distribution is unknown.

In this work a single value of the angle (α) was used for the application of both the inner and outer loads, i e both angles were equal and ranged from 20 degrees to 120 degrees

When a comparison was made of the calculated resulting stresses at the gauge locations with the measured time histories the following was noted

* some results correlated better, both in terms of the mean and amplitude values, when α =30 degrees and when α = 120 degrees

* Therefore it seems that the load distributions varies in reality from the outside of the wheel to the inside.

Thus different 'hybrid' load cases were investigated. Figure 28 shows the 'hybrid' parabolic distribution on the wheel rim

Table 3 shows a selection of results for the 'non-hybrid' single angle load cases, the 'hybrid' load cases and finally the measured test data from the Fraunhofer-Institute für Betriebsfestigkeit, Darmstat (LBF)

Different strain gauge locations are shown, but because the spoke area is the most highly stressed area of the wheel, this study concentrates in this area The most critical area is where the spoke and hub join together (location of strain gauge M2), and where the rim an spoke join (location of strain gauge M3).

From the table it is apparent that for these critical areas the 'Hybrid' load case α_I = 60 degrees and α_O =120 degrees correlates best with these measured results

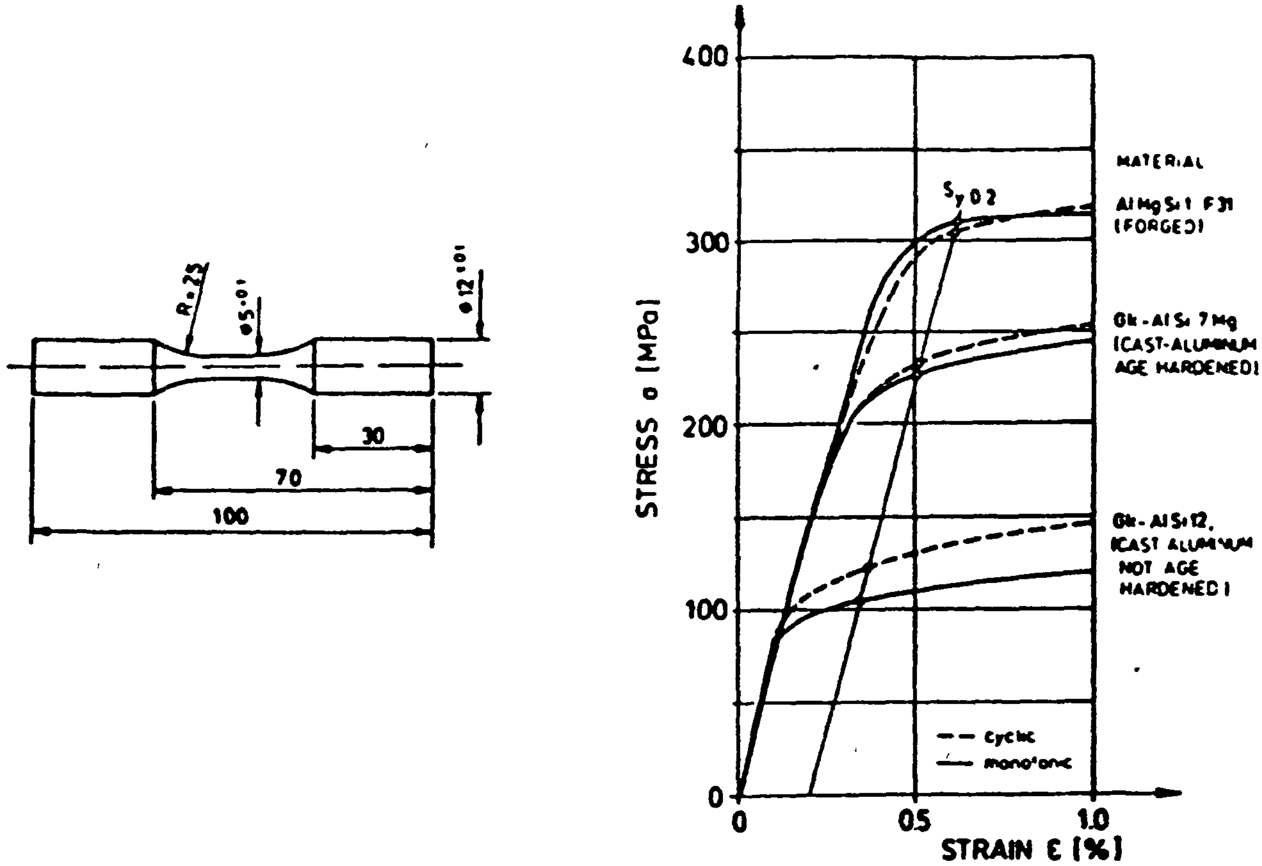


Figure 26. Stress - Strain diagram.

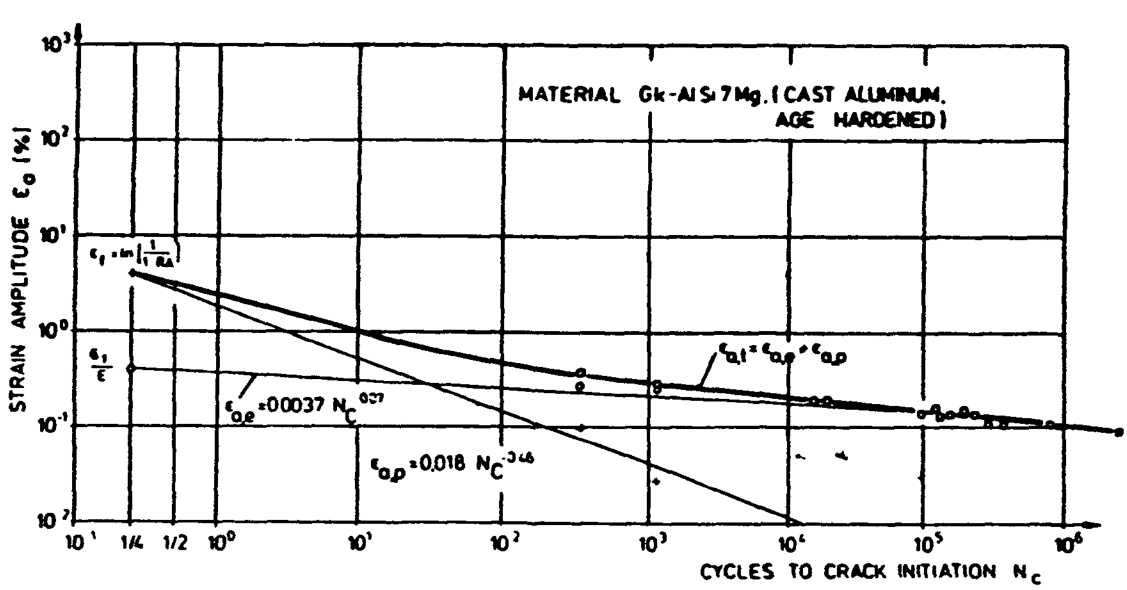


Figure 27. SN curves obtained under strain control.

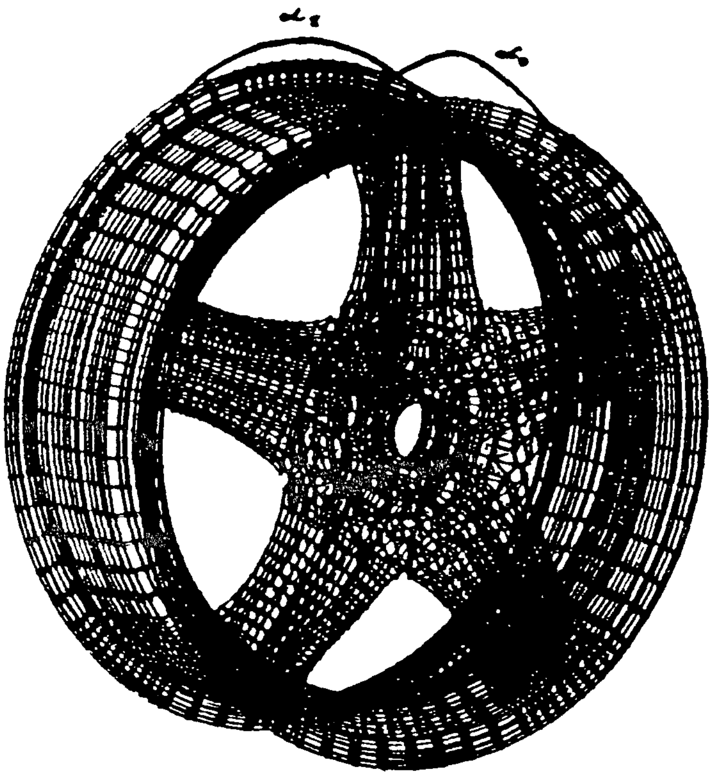


Figure 28. Hybrid parabolic distribution on wheel rim.

Parabolic distribution	M2 max stress	M2 min stress	M27 max stress	M27 min stress	M3 max stress	M3 min stress	M6 max stress	M6 min stress	M7 max stress	M7 min stress	M17 max stress	M17 min stress
outer 30°	+19 90	-21.67	+32 12	-35 31	+13 96	-15 31	+15 19	-30 58	+17 29	-10 28	+23 86	-11.37
inner 30°	mean = -0.88 ampl = ±20 79		mean = -1 59 ampl = ±33.72		mean = 0 67 ampl = ±14 64		mean = -7 69 ampl = ±22 89		mean = +3 51 ampl = ±13 79		mean = +6.25 ampl = ±17 62	
outer 60°	+18 13	-20 71	+28 85	-34 38	+13 21	-11 44	+14 48	-29 11	+16 67	-10.15	+19 02	-6 97
inner 60°	mean = -1 29 ampl = ±19.42		mean = -2.76 ampl = ±31.62		mean = +0 89 ampl = ±12 33		mean = -7.31 ampl = ±21 80		mean = +3 26 ampl = ±13.41		mean = +6 06 ampl = ±13 00	
outer 120°	+13 21	-16 42	+20 36	-28 17	+9 57	-6 97	+11 16	-22.70	+13 24	-8 24	+9 83	-4 66
inner 120°	mean = -1 6 ampl = ±14.82		mean = -3.90 ampl = ±24.27		mean = +1 3 ampl = ±8 27		mean = -5 77 ampl = ±16.93		mean = +2.5 ampl = ±10.74		mean = +2.59 ampl = ±7.25	
outer 50°	+15.23	-24.57	+24 94	-38.74	+21 72	-10 67	+12 31	-22 68	+27 66	-8 68	+7.91	-5 87
inner 120°	mean = -4 67 ampl = ±19.90		mean = -6 90 ampl = ±31.84		mean = +5 53 ampl = ±16 20		mean = -5 18 ampl = ±17 5		mean = +9 49 ampl = ±18 17		mean = +1 02 ampl = ±6.89	
outer 60°	+15.22	-24 00	+24.90	-38.70	+21 72	-10.73	+12.34	-36.72	+27.69	-8 72	+7.85	-5.87
inner 120°	mean = -4 39 ampl = ±19 61		mean = -6 90 ampl = ±31.80		mean = +5 5 ampl = ±16 23		mean = -12.19 ampl = ±24.53		mean = +9 49 ampl = ±18 21		mean = +0 99 ampl = ±6 86	
outer 70°	+15 22	-24 58	+24.87	-38.84	+21 72	-10 75	+12 35	-36.73	+27.73	-8 74	+7 78	-5 87
inner 120°	mean = -4 68 ampl = ±19.90		mean = -6 98 ampl = ±31.86		mean = +5 49 ampl = ±16 24		mean = -12.19 ampl = ±24 54		mean = +9 5 ampl = ±18 24		mean = +0 96 ampl = ±6 83	
measured	+18	-24	+24	-30	+15	-20	+15	-29	+15	-7	+9	-6
results	mean = -3 ampl = ±21		mean = -3 ampl = ±27		mean = -2 5 ampl = ±17 5		mean = -7 ampl = ±22		mean = +4 ampl = ±11		mean = +1.5 ampl = ±7 5	

Table 3 The calculated Principle stresses in the strain gauge locations resulting from different load cases and the actual measured stresses.

6.0 FATIGUE LIFE PREDICTION SOFTWARE

6.1 Description of the Procedure

Here a Procedure was developed which calculates the fatigue life of the wheel under operational loading conditions Turbo Pascal is the programming language used to produce this Procedure.

This complex task was broken down into the following sub-tasks

1. *Load assumptions which simulate operational loading.*
2. *Calculation of local stresses by using numerical calculations such as FEA.*
3. *Derivation of Design Spectrum.*
4. *Calculation of S-N curves for material and manufacturing processes and the estimation of damage using the modified damage calculation of Palmgren Minor*
5. *Estimation of the Fatigue Life of the component*

A feedback loop was incorporated into the Procedure to allow the user to

> Optimise either the wheel design, material or manufacturing processes
> or
> Optimise the allowable stresses on the wheel components (S-N curve)

Figure 29 shows the flow chart which illustrates this Fatigue Life Prediction Procedure.

This Procedure is composed of two Turbo Pascal programs. They are :

For sub-task 1	***SPCM_TP1***
For sub-tasks 2,3,4,5 & 6	***SPCM_TP2.***

They can be seen in appendix F.

6.1.1 General Procedure description

This Procedure is used to predict the Fatigue Life of those areas of the component which are regarded as critical to the overall life of the automotive wheel. Because the wheel represents a complicated component it is difficult, if not impossible, to determine the **Nominal** stresses for the individual areas. In this case the manufacturing influences as well as the stress distributions (stress gradient, multi-axiality) and the residual stresses, if existent, are represented.

In this Procedure the Nominal approach is not used. The **Local** stress approach with stress analysis (FEA) gives us the necessary stress calculations.

The Modified Damage Accumulation Hypothesis is used to calculate the Damage in the critical areas of the wheel with the S-N curve of the component-like specimens and this Damage calculation allows the Service Lifetime to be estimated. Figure 30 illustrates life prediction based on the Local stress approach [47].

6.1.2 Results strategy

For a given wheel under given loading conditions by using FEA the critical areas of the wheel can be isolated and studied in detail. A Design Spectrum for these critical areas is developed. From the material and manufacturing data, the S-N Wohler curve is produced and a modified Damage

Calculation can be performed This Cumulative Damage calculation allows the calculation of the Expected Lifetime for that particular area of the wheel

If the calculated Expected Lifetime is too low then the wheel is not optimised and the following may need to be investigated to increase the lifetime :

* New material and/or manufacturing processes

* Improved wheel design (geometry)

If the Expected Lifetime is too high then the wheel is over designed The above modifications may be considered to reduce it to the required Lifetime

NOTE : This Procedure has yet to be validated against experimental data from industry

<u>STEP 1</u>: ***Load Assumptions***

The objective of this step is to deduce what the forces on the wheels are from the following information

* ***What type of road is the wheel being driven on?***
  For example off road, pot holed, secondary, primary
  Usually the wheel is designed for the worst case.

* ***What type of driving is the wheel being subjected to?***
  For example. aggressively driven, average driving, slow driving.

* ***What type of tyre is being used on the wheel?***

I E how stiff is the tyre?

The tyre stiffness depends on the tyre design and pressure It can be derived from the data in the tyre catalogues.

Typical values are:	car tyres	C1=1 5 kN/cm
	Truck tyres	C1= 15 kN/cm

The vertical static force on the wheel must be known

This information is then analysed by the Procedure and the following data is outputted onto the screen ·

Fv,s	Vertical wheel force for the straight driving load case
Fl,s:	Lateral wheel force for the straight driving load case
Fv,c	Vertical wheel force for the cornering load case
Fl,c	Lateral wheel force for the cornering load case

This data, together with the Fv,static value, are used in the Finite Element Analysis model.

Figure 31 shows the characteristic curves for straight driving and cornering

<u>STEP 2</u>: ***Local Stress Calculation using finite element analysis***

A software package such as I-DEAS can be used to produce a solid geometric model of the wheel From this a finite element model can be constructed and the loading conditions for this model are produced by STEP 1 of the Procedure

The maximum and minimum Principal stresses at critical locations on the wheel for a complete rotation can be extracted from the post processor results This information can then can

1 Be directly compared with the test data stress-time history at critical locations (i e the strain gauge that is glued to the wheel at that point This corresponds to a node on the fea model)

2 For each critical location the results of maximum and minimum principal stress for straight driving , cornering and static load cases can be recorded and inputted into STEP 3. These stresses are then the basis for the stress amplitude calculation which eventually leads to Life Estimation

<u>STEP 3</u>: ***Design Spectrum***

From the previous STEP the following data is inputted into the Procedure for each critical location :

*	Straight driving	maximum and minimum principal stresses
*	Cornering	maximum and minimum principal stresses
*	Static	maximum and minimum principal stresses

These are called Service Stresses and they are a result of wheel loads of which the vertical forces Fv and the lateral forces Fl are the most important [52] The Service stresses can be seen as consisting of two parts

1 The basic stress Sa,stat due to the wheel just rolling on a smooth surface under it's static vertical load Fv,static (the rated wheel load)

and

2. The superimposed stresses due to service loads acting on the wheel. These loads result from the roughness of the road and the manoeuvres of the vehicle, and they vary in frequency of occurrence as well as in phasing [49]

The stress amplitude, mean stress and the stress ratio are then calculated in the Procedure. The following equations are used to calculate these values

$$\text{Stress Amplitude} = (\sigma max - \sigma min) / 2$$

$$\text{Mean Stress} = (\sigma max + \sigma min) / 2$$

$$\text{Stress Ratio} = \sigma min / \sigma max$$

The stress amplitude values for each critical area of the wheel are used to create the Design Spectrum for that area of the wheel. Each critical area has it's own specific Design Spectrum

The Procedure then uses the stress amplitude values as the Y-axis values of the Design Spectrum. The X-axis values (cycles) are computed in the following way ·

The User is asked to input · 1 The required design life value for the wheel (km)
2 The Dynamic tyre radius (m)

From this Ntot, Ns, Nc and Nes (=Nec) are calculated using the equations given in Figure 32 [53] This also shows that the Design Spectrum is the "worst case" of the static, straight driving and cornering load cases

STEP 4: ***SN Curve***

In order to determine the fatigue behaviour and to achieve optimum design of a wheel, two conditions must be known

1. The loads that the wheel will be subjected to (Design Spectrum). These depend on the vehicle parameters and operational loading conditions.

2. The allowable stresses These depend on material properties, prestress and manufacturing conditions.

As mentioned previously the **Local** Stress Approach is used in this Procedure, i e. the SN curves are determined from simplified tests under sinusoidal loading Figure 12 shows how this test was set up [54]. These SN curves reflect the exact loading conditions in the critical areas of the wheel They depend on :

* Material
* Manufacturing Processes
* Area of the wheel

In the Procedure the following options are available

MATERIAL	MANUFACTURING PROCESS	AREA *	Sendurance	k	k'
Steel		A	130	5	2k-1
		B	90	4	2k-2
		C	160	6	2k-1
		D	90	4.5	2k-2
Aluminium	Cast	All	40	4 5	2k-2
	Cast - Heat treated	areas	60	4 5	2k-2
	Forged		80	4 5	2k-2

*The critical areas of the wheel are given in figure 33

This data was supplied by LBF

From this data the SN curve is drawn

STEP 5: ***Modified Damage Calculation***

Having both the Design Spectrum and the SN curve for a particular area of the wheel, the Cumulative Damage to the wheel can be estimated. Because the Local stress approach is used, the Hypothesis used in the Procedure is the Modified Minor Hypothesis Figure 34 illustrates the method used in the Damage calculation [54]

The allowable Damage for car wheels is **D ≤ 0.5**

STEP 6: ***Service Life Calculation***

The required service life (Lr) of a car is 300,000 km. The required damage ≤ 0.5 (Da).

From the damage calculation the Da the damage sum is found

The expected lifetime L, is calculated by the simple calculation

$$L = \{Da / D\} * Lr \qquad \text{where L is in km}$$

If L is found to be very large in comparison to Lr, the wheel is said to be *'over-designed'* and similarly if L is very small it is said to be *'under-designed'* and will fail under service load conditions. In both cases it must be redesigned. The following options can be investigated

1 Modifications of the material and / or manufacturing processes.

> Here the SN curve is effectively changed and the allowable stresses on the components are increased. The damage calculation is updated and the expected lifetime is recalculated

2. Modifications of the design.

Here the design is analysed and optimised taking into account the information from the FE analysis I e the critical areas are redesigned to reduce the stress amplitude resulting from operational loading conditions.

After each design modification the new stress values must be loaded into the Procedure. This will result in a new improved design spectrum in the critical areas and ultimately for the same SN curve an increase in the expected lifetime of the design.

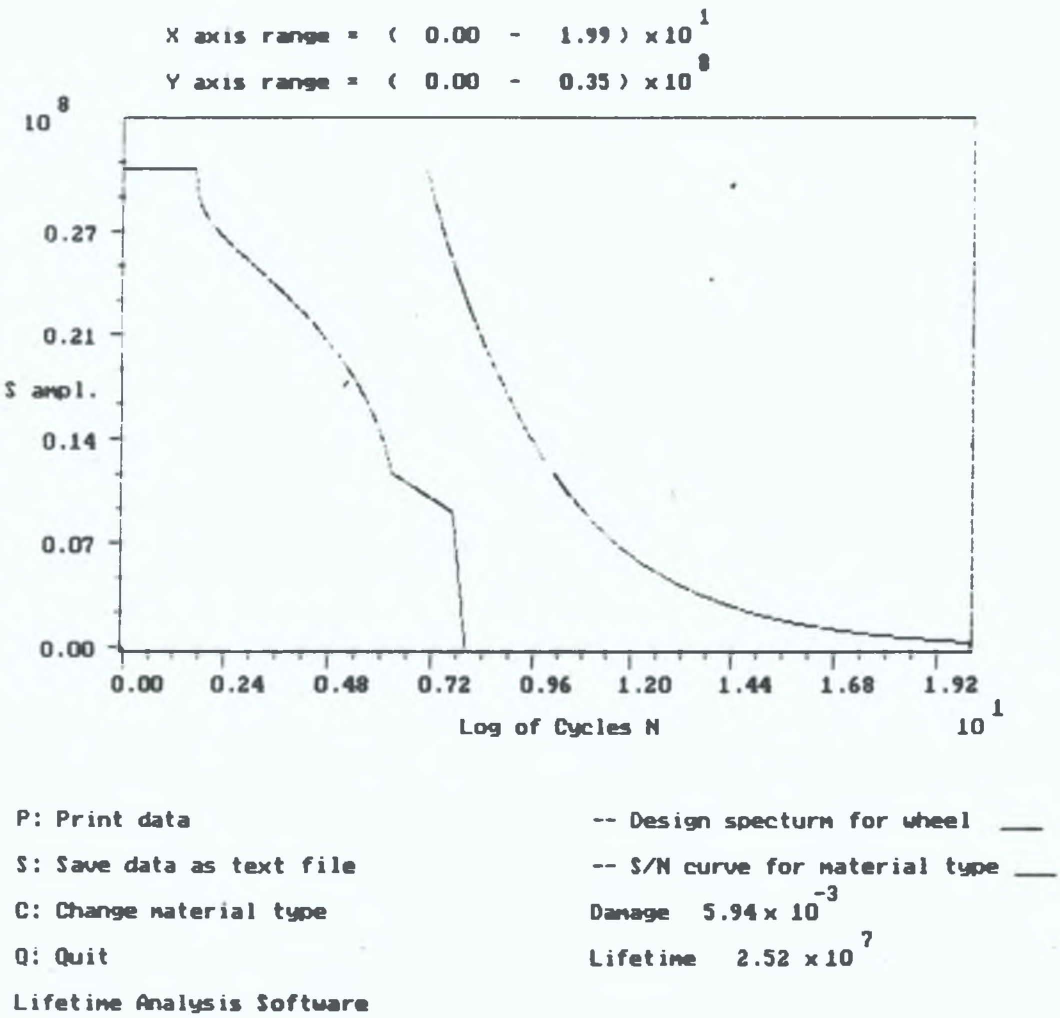
6.2 Fatigue results

For a given wheel design and specification the Procedure was used to predict the Damage in the aluminium alloy wheel at the critical location M2 (at the spoke and hub interface), under its operational loading conditions

Stress amplitudes :	***static***	***Sa,stat = ± 9 Mpa***
	straight driving	***Sa,s = ± 19.5 MPa***
	cornering	***Sa,c = ± 31.5 MPa***
The required wheel Design Life :	***Lr = 300,000 km***	
The Dynamic Tyre Radius :	***Rdyn = 0.45 m***	
Material characteristics* :	***Material = Aluminium***	
	Manufacturing process = cast, non-heat treated	
	k = 4.5	
	k' = 7	
	Sendur = 40 Mpa	
	Nendur = 2 x 10^6 cycles	

*Note These need to be verified for the Aluminium alloy.

The Damage calculation for the Aluminium alloy wheel is shown in the following



Therefore, according to the Procedure the damage to the wheel at a critical location, spoke / hub intersection D = 0.0006. This gives an expected lifetime of 25,200,000 km which far exceeds the required lifetime of 300,000 km.

This indicates that the wheel is *very* over designed and there are different possibilities for an optimisation which could reduce the product cost.

For example .

* A change in the material or manufacturing methods. This could allow a less expensive material or manufacturing process to be used in the wheel manufacture.

* An optimisation of geometry reducing the volume of aluminium used in the manufacture of the product

Due to the fact that the wheel, like most automotive components, is a high volume product, a saving due to either of the above could result in an important saving for the company. The car manufacturer is also very interested in weight saving and offer monetary incentives to component suppliers to find new ways of reducing the weights of there products

A reduction in the quantity of material used to cast the wheel is the saving investigated here. In the following chapter two variations of the original geometry are studied.

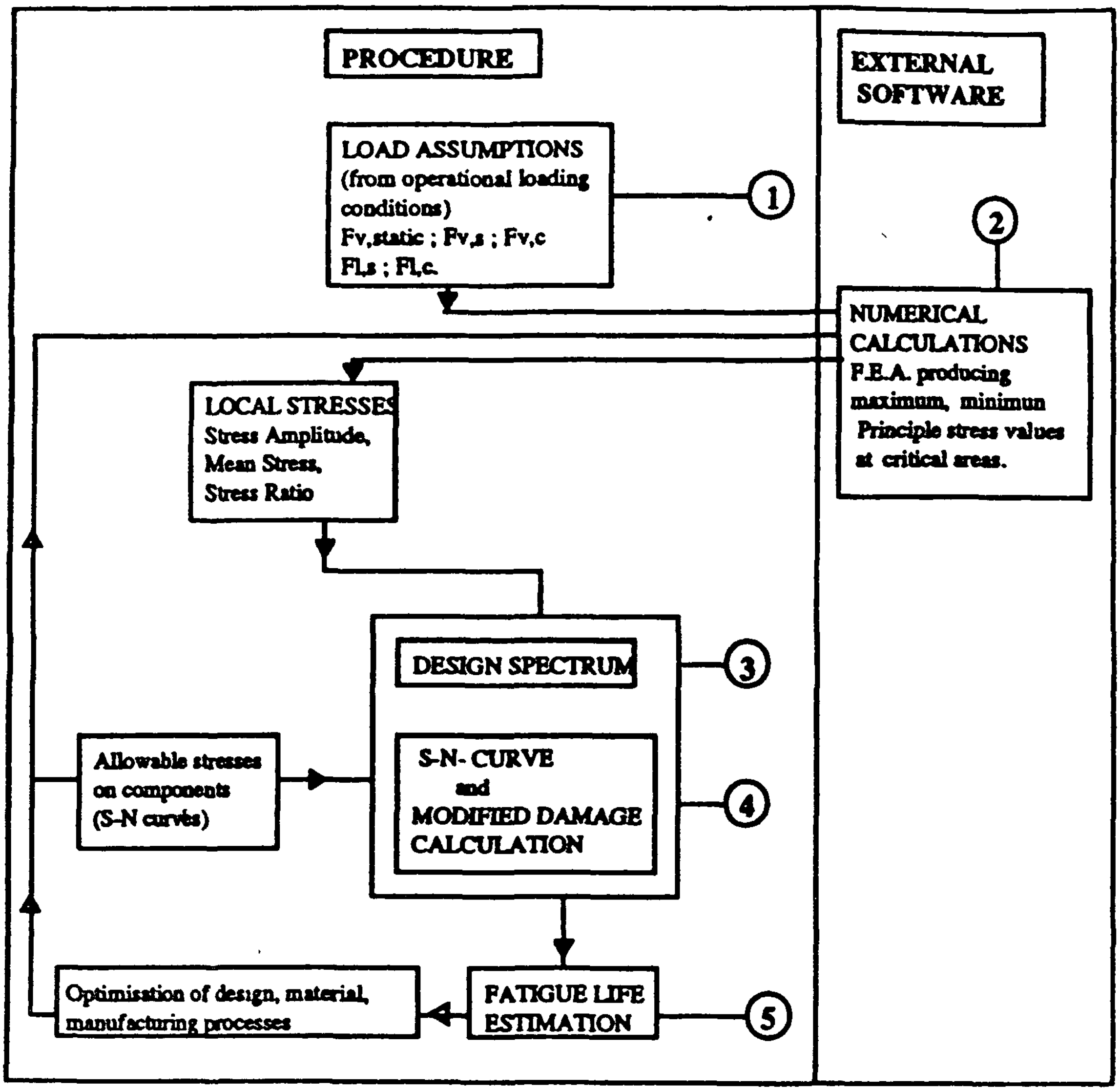


Figure 29. Flow chart for the Fatigue Life Estimation and optimisation.

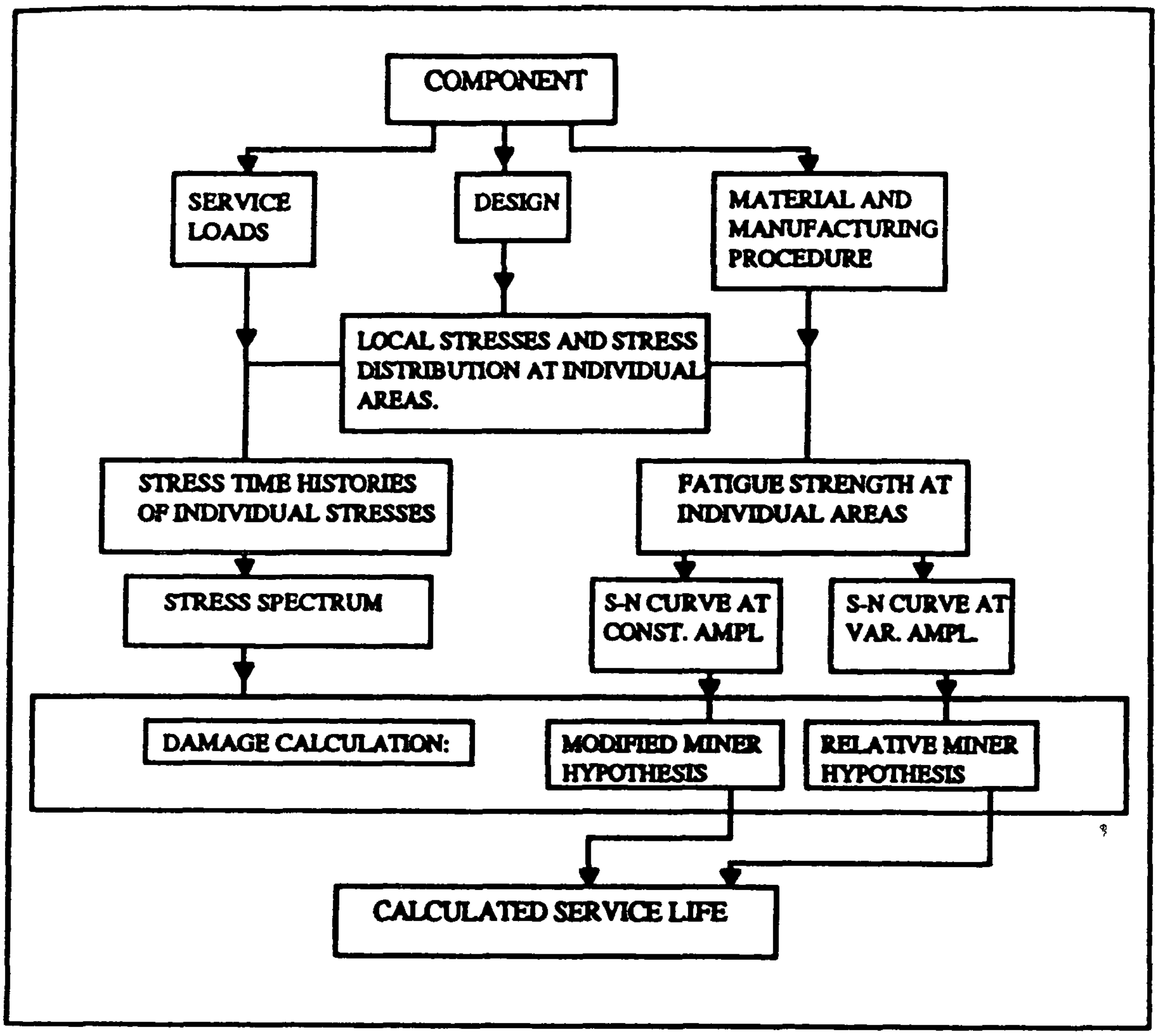
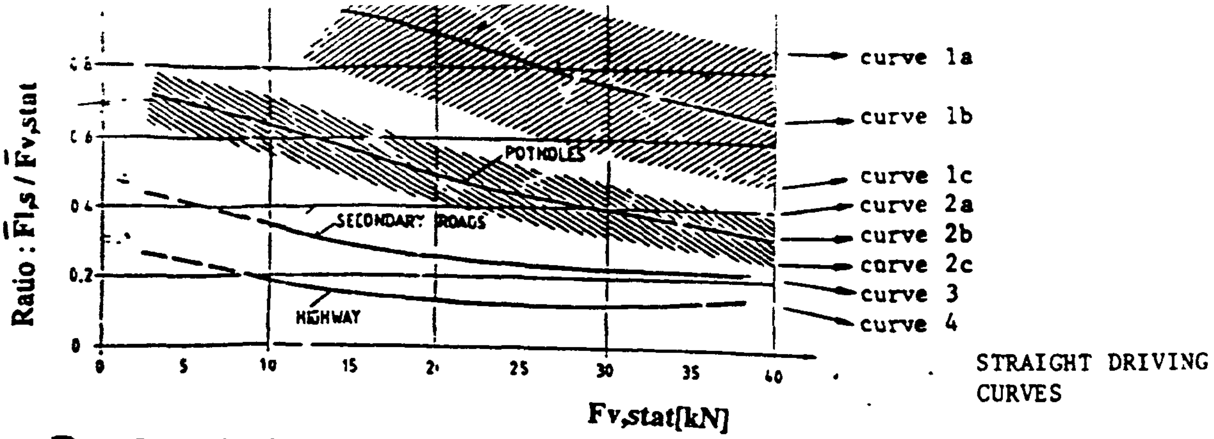
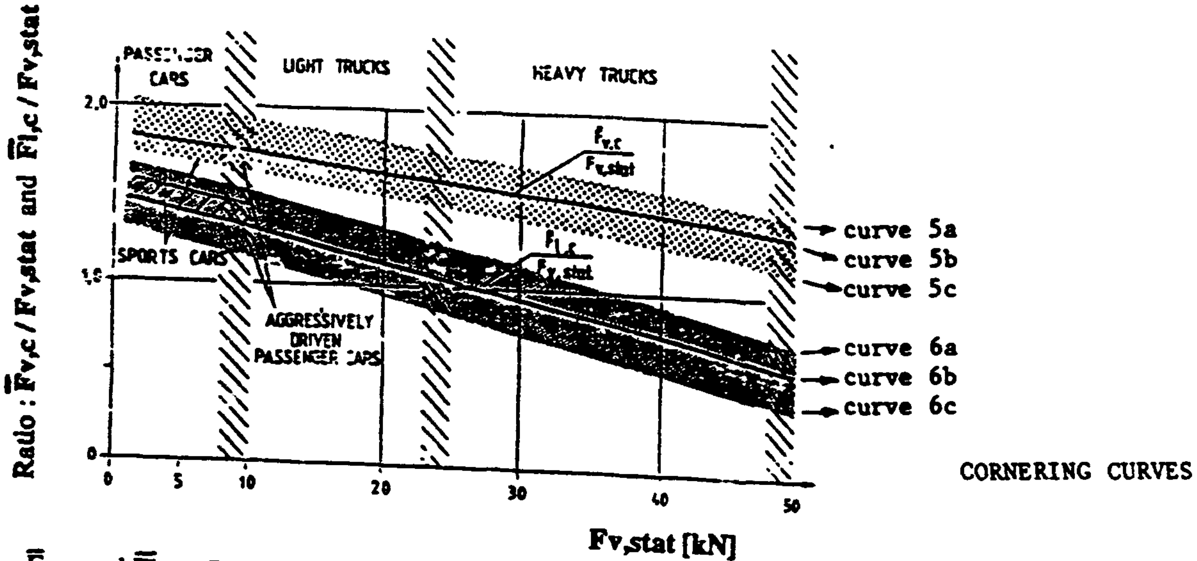


Figure 30. Life Prediction based on Local stress approach.



$\overline{Fl}_{,s}$ = Dynamic wheel load (front lateral)

Fv,stat = Static wheel load (front vertical)



$\overline{Fv}_{,c}$ and $\overline{Fl}_{,c}$ = Dynamic wheel load (front vertical and lateral)

Fv,stat = Static wheel load (front vertical)

1a: $Fl_{,s}=(1.4759-0.02362 * Fv_{,stat} + 0.0002018 * Fv_{,stat}^2) * Fv_{,stat}$
1b: $Fl_{,s}=(1.2465-0.02206 * Fv_{,stat} + 0.0001809 * Fv_{,stat}^2) * Fv_{,stat}$
1c: $Fl_{,s}=(1.0603-0.02333 * Fv_{,stat} + 0.0002036 * Fv_{,stat}^2) * Fv_{,stat}$
2a: $Fl_{,s}=(0.8503-0.01714 * Fv_{,stat} + 0.0001482 * Fv_{,stat}^2) * Fv_{,stat}$
2b: $Fl_{,s}=(0.7796-0.01815 * Fv_{,stat} + 0.0001691 * Fv_{,stat}^2) * Fv_{,stat}$
2c: $Fl_{,s}=(0.6765-0.01715 * Fv_{,stat} + 0.0001663 * Fv_{,stat}^2) * Fv_{,stat}$
3: $Fl_{,s}=(0.467-0.0146 * Fv_{,stat} + 0.0002131 * Fv_{,stat}^2) * Fv_{,stat}$
4: $Fl_{,s}=(0.296-0.01304 * Fv_{,stat} + 0.0002367 * Fv_{,stat}^2) * Fv_{,stat}$

5a: $Fv_{,c}=(2.0601-0.01227 * Fv_{,stat}) * Fv_{,stat}$
5b: $Fv_{,c}=(1.8416-0.01016 * Fv_{,stat}) * Fv_{,stat}$
5c: $Fv_{,c}=(1.7389-0.01178 * Fv_{,stat}) * Fv_{,stat}$
6a: $Fl_{,c}=(1.8936-0.01848 * Fv_{,stat}) * Fv_{,stat}$
6b: $Fl_{,c}=(1.4936-0.01848 * Fv_{,stat}) * Fv_{,stat}$
6c: $Fl_{,c}=(1.0936-0.01848 * Fv_{,stat}) * Fv_{,stat}$

Figure 31. Characterisation curve for straight driving and cornering.

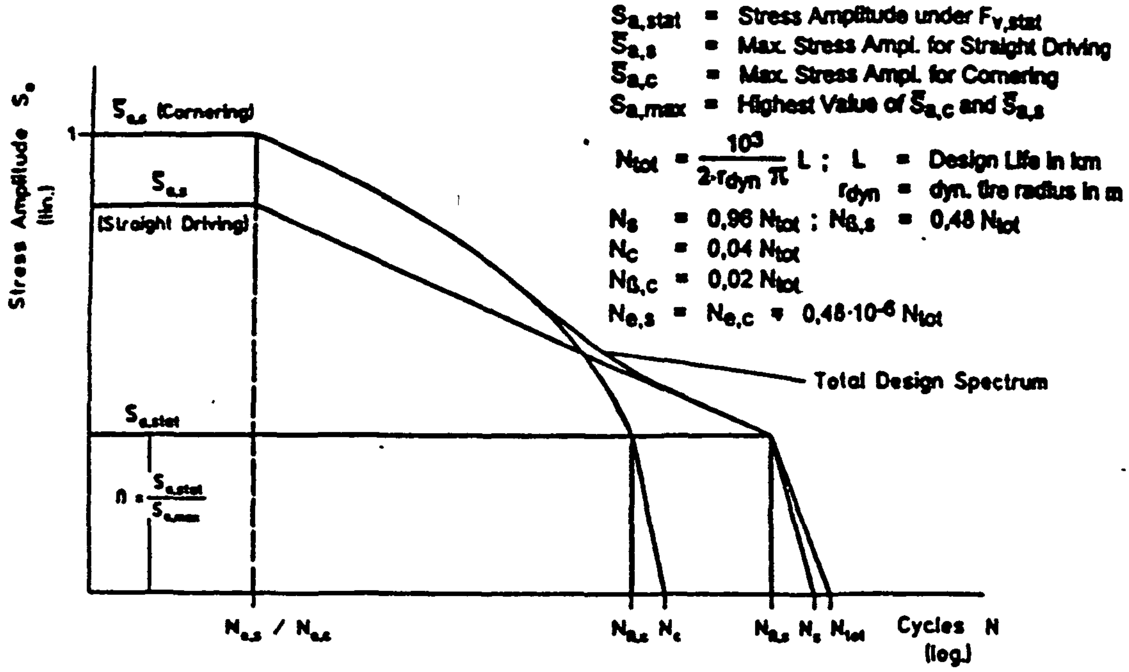


Figure 32. Design spectrum for passenger car wheel.

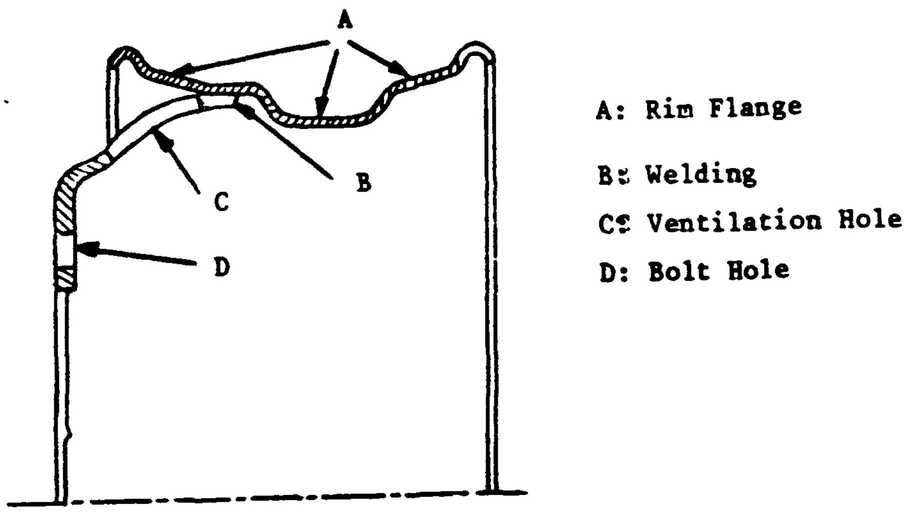


Figure 33. The critical areas of the wheel.

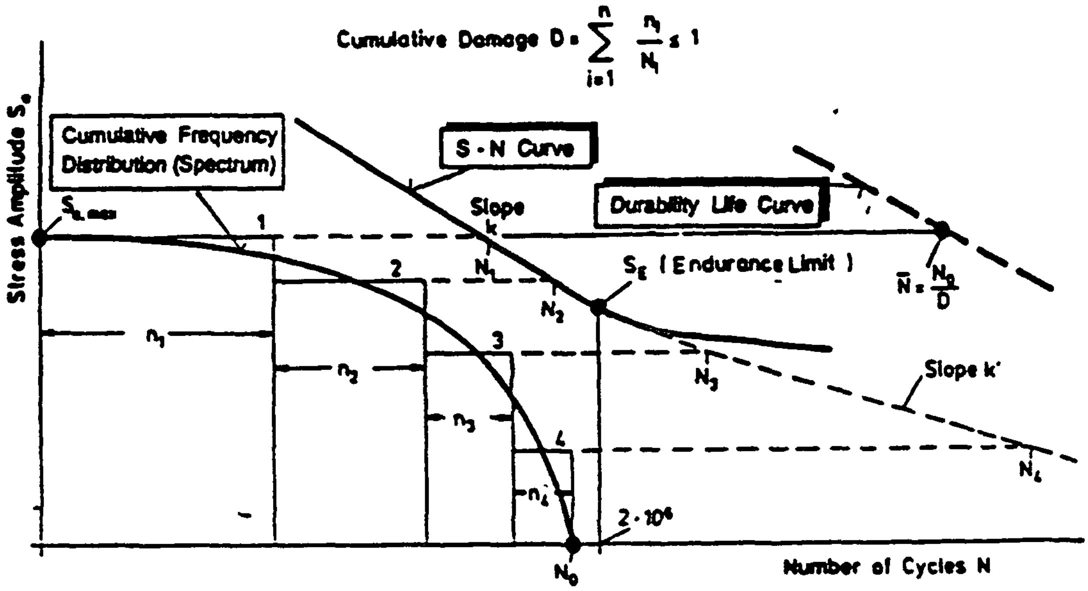


Figure 34. The modified damage calculation.

7.0 OPTIMISATION OF WHEEL GEOMETRY

It was found in the previous chapter that the life estimation for the aluminium alloy wheel gave results showing a greatly over designed product (i e the required wheel lifetime was 300,000 km and the calculated lifetime was 25,200,000 km). The action taken here is the redesign of the critical areas of the wheel (spoke - cross sectional area) This will allow a reduction in the volume of material used to cast the wheel, thus give a cost saving, without compromising the reliability of the product during the required lifetime

Care was taken with these modifications The original design criteria must be taken into account and the aesthetic features of the wheel considered all the time

Two design variations are considered here Both of these are a reduction in the cross sectional area of the spoke. The stress results from the finite element analysis, as well as intuition and experience, guided the author in the choice of these variations They are simple modifications of geometry rather than a methodical systematic approach to design optimisation

Figure 35 shows the original spoke section as well as variation one and two. There are two half sections shown here, section AA which is the section through the spoke near the hub region Section BB is the section through the spoke near the rim region A half section is shown because they are symmetrical around the spoke centreline

The modifications to the geometry were made by modifying the co-ordinates of the nodes in the relevant areas of the finite element mesh. The unit load case used in the finite element analysis was the optimised parabolic distribution for the spokes i e. the distribution on the outer rim was over 60 degrees and on the inner rim was 120 degrees

After comparing the results for the straight driving load case for all three geometry's it was found that, as expected, with a reduction in cross sectional area the stress amplitudes increased in the critical wheel areas The increase was relative to the reduction in volume.

Table 4 shows the increase in maximum and minimum principal stresses and amplitude due to a reduction in cross sectional area. The results at strain gauge locations M2, M27, M3, M6, M7 and M17 are presented

For Variation 1 the increase in amplitude varies from 2 3% in gauge M7 to 12% in gauge M27. Gauges M2 and M3 are the critical ones and they experience an increase in amplitude of 7 2% and 8 8% respectively.

For Variation 2 the increase in amplitude with respect to the original geometry varies from 5 6% to 18.9% Strain gauge location M6 shows the increase of 4.6% and M27 the increase of 18 9% M2 shows an increase of 11 4% and M3 of 16 9%

These increases in stress amplitudes at the different gauge locations are due to the reduction in volume of the complete wheel. These decreases are of 1 0% in Variation 1 and 1 8% in Variation 2 These can be seen in Table 5

These increases in stress amplitudes do not greatly affect the wheel life estimate From Figures 26 and 27 the following is found ·

Geometry	*Stress from FEA at M2 (MPa)*	*Strain for Figure 26 (%)*	*No. cycles to crack initiation Figure 27*
Original	19 5	0 23	100,000
Variation 1	21 5	0 25	50,000
Variation 2	21 8	0 25	50,000

From this it can be seen that although there is a small change in the number of cycles to crack initiation, it is difficult to interpret the results in terms of wheel cycles

Similarly the difference in fatigue life when calculated by the Procedure was minimal A more radical decrease in volume (perhaps 10 to 20%) is necessary to substantially reduce the life To check the influence of such a modification, the finite element model would have to be completely reconstructed.

These modifications are outside the scope of the present work but could be the subject for a further study

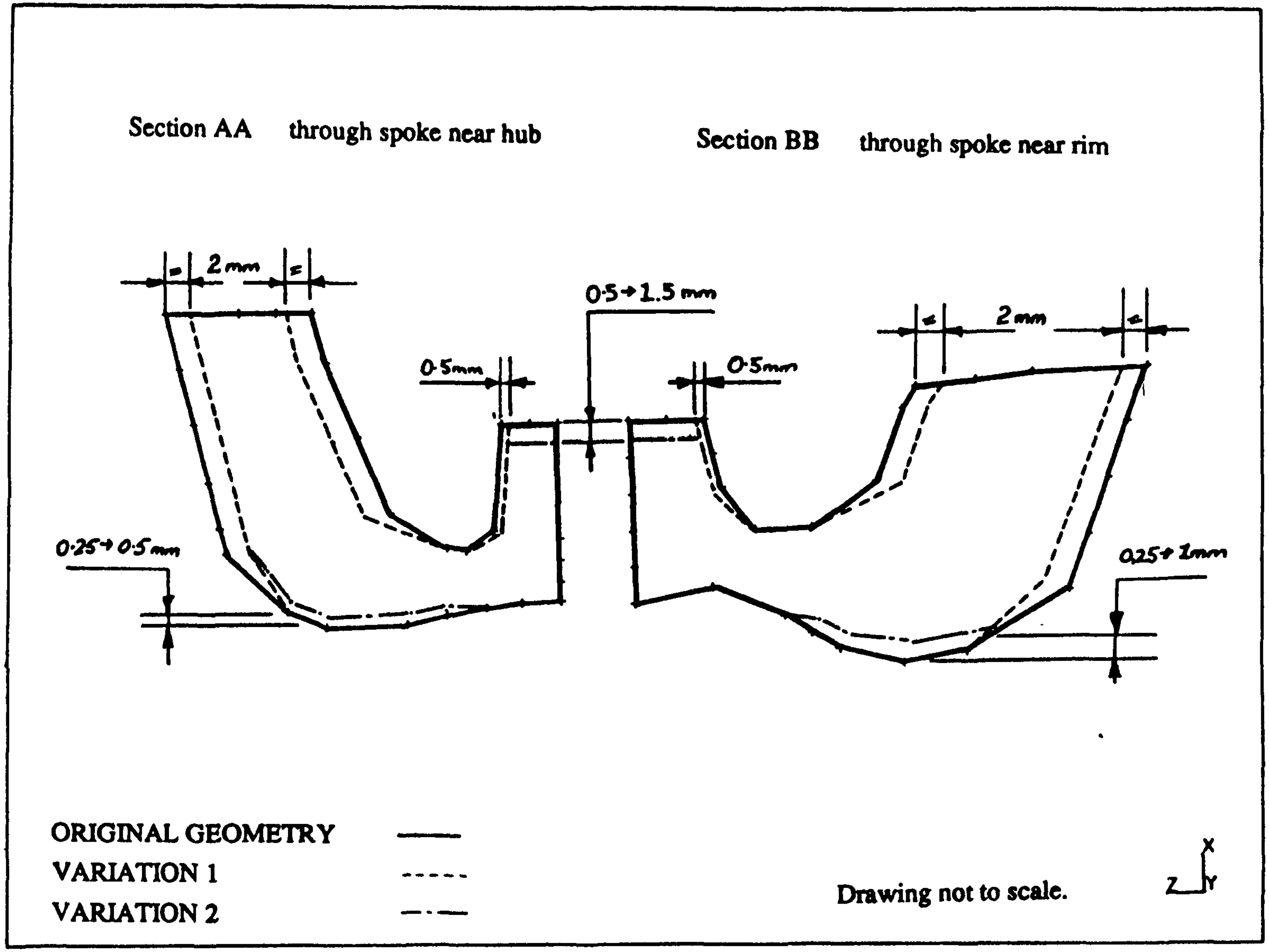


Figure 35. Modification to the spoke geometry.
(a) Original geometry, (b) Variation 1, (c) Variation 2

	M2		M27		M3		M6		M7		M17	
parabolic distribution *O=60°; I=120°*	max stress	min stress	max stress	min stress	max stress	min stress	max stress	min stress	max stress	min stress	max stress	min stress
Original geometry	+15.22	-24.00	+24.90	-38.70	+21.72	-10.73	+12.34	-36.72	+27.69	-8.72	+7.85	-5.87
	mean = -4.39 ampl = ±19.61		mean = -6.90 ampl = ±31.80		mean = +5.5 ampl = ±16.23		mean = -12.19 ampl = ±24.53		mean = +9.49 ampl = ±18.21		mean = +0.99 ampl = ±6.86	
Variation 1	+16.67	-25.37	+28.19	-43.04	+22 98	-12.34	+12.73	-38.57	+28.39	-8.86	+9.15	-5.61
Change in Ampl. from Original	mean = -4.35 ampl = ±21.02 ↑ *7.2%*		mean = -7.42 ampl = ±35.62 ↑ *12%*		mean = +5.32 ampl = ±17.66 ↑ *8.8%*		mean = -12.92 ampl = ±25.65 ↑ *4.6%*		mean = -9.76 ampl = ±18.63 ↑ *2.3%*		mean = +1.77 ampl = ±7.38 ↑ *7.6%*	
Variation 2	+17.24	-26.44	+29.69	-45.91	+24 43	-13.52	+13.05	-37.80	+30.21	-9.54	+6.69	-5.92
Change in Ampl. from Original	mean = -4.6 ampl = ±21.84 ↑ *11.4%*		mean = -8.11 ampl = ±37.8 ↑ *18.9%*		mean = +5.56 ampl = ±18.98 ↑ *16.9%*		mean = -12.38 ampl = ±25.43 ↑ *4.6%*		mean = -10.34 ampl = ±19.88 ↑ *9.2%*		mean = +0.38 ampl = ±6.31 ↑ *8.0%*	

Table 4 Affect of varying the geometry on the Principle stresses calculated at the strain gauge locations.

Geometry	Total wheel volume (cm^3)	Total wheel mass (kg)	Volume of 5 spokes (cm^3)	Mass of the 5 spokes (kg)
Original	4,990	13.84	995	2.69
Variation 1	4,941 ↓ *1.0%*	13.34 ↓ *1%*	943.5 ↓ *5.2%*	2.55 ↓ *5.2%*
Variation 2	4,901 ↓ *1.8%*	13.23 ↓ *1.9%*	905.3 ↓ *9%*	2.44 ↓ *9%*

Table 5 Changes in mass and volume resulting from changes in geometry.

8.0 CONCLUSIONS AND SUGGESTIONS FOR FURTHER WORK.

8.1 Conclusions

The calculation of stresses in the selected five spoke wheel component under its operational loading has been satisfactorily achieved.

A combination of SDRC's I-DEAS Finite Element Pre and Post Processing and Linear Model Solution software on a Silicon Graphics Indy platform has been used to construct and analyse a model of the selected wheel component

The predicted results for a number of different combinations of vertical and lateral loads have been compared with the experimental data This indicated that the best comparisons were obtained, at critical gauge locations, under the following simplified load inputs

Straight Driving : Fv 10.6 kN

* Fvo = 0 6 Fv and Fvi = 0 4 Fv, Fho = Fhi = 0 5 Fv/2
* Parabolic load distributed over 60 degrees on the outer rim

and 120 degrees on the outer rim

Cornering : Fv 6.72 kN + Fh 5.4 kN

* Fvo = 0 6 Fv and Fvi = 0 4 Fv,
* Fho = 5 4 Fh
* Fhi = 0 0 Fh
* Parabolic load distributed over 60 degrees on the outer rim and 120 degrees on the outer rim

The predicted results generally show a good level of agreement when compared with the measurements for the spoke region of the wheel for the straight driving case. In the rim region there is a lower level of agreement but it is still acceptable The level of alternating stress predicted is better than +/- 25% for all the higher stressed areas of the spokes

The level of agreement is not as good when comparing the predicted results with the measurements for the spoke region of the wheel for the cornering case For this case the level of alternating stress is never under predicted so the model results are likely to be conservative for component life assessment.

The model is regarded as representative of the wheel structure and can be used as the basis for further work The time taken for a single solve containing several unit load cases is approximately 8 hours on an appropriately configured Silicon Graphics Indy R4000 processor workstation This was reduced to 4 hours when the software version was updated from 1.3 to 2.1

The alternating stress predicted using the model are considered to be acceptable for the determination of fatigue life under operating loads The predicted stresses from the current model are likely to lead to conservative life predictions in most areas of the wheel

The stresses predicted using the wheel model show the same general behaviour as the dynamic test measurements However, the model can be used to generate far more detailed information regarding stress distribution which may be used for subsequent life assessments under operating conditions. For example, the model can be used to locate important regions for fatigue and failure analysis, guide more detailed inspections for fatigue cracking and assess the effects of design changes of materials, construction methods or local geometric features

The parabolic distribution of the loading profile was investigated Different 'mono' and 'hybrid' distributions were analysed and a compromise 'hybrid' load case was chosen to represent actual loading conditions This is comprised of a distribution of load on the inner rim of 120 degrees and outer rim of 60 degrees

A Procedure has been developed to predict the life of the wheel under operational loading conditions. The programming language used is Turbo Pascal and it runs in a PC in a DOS environment.

The Procedure allows the user to try out different materials and manufacturing processes This allows the optimisation of the lifetime calculations if the first calculation does not give the required lifetime

The lifetime of the aluminium alloy wheel was calculated by using the Procedure Stress data calculated by the finite element analysis was used in this calculation The wheel was found to be greatly over designed, i e the expected lifetime was much greater that the required lifetime by a factor of 100

The geometry in the spoke region was modified. Two reduced volume variations were analysed and an increase in stress amplitude resulted This increase was not enough to affect the lifetime estimation result by a significant amount

8.2 Further work

Although the results of the predictive work performed are generally representative, it is recommended that further work could be undertaken in the following theoretical and experimental areas :

A more detailed study of simplified load assumptions and distributions is required to more completely understand and characterise the behaviour of the tyre and its interface with the wheel It is recommended that some future effort is directed towards modelling and understanding the behaviour of the tyre / structure interface.

Future work in the tyre and its interface with the wheel should also develop a test method for measuring the actual stresses at the tyre / rim interface

The current approach has been to use a linear elastic solution which will not accurately represent the larger displacements and membrane stresses at the extremity of the inner rim in particular This can be seen by the high stresses measured at the strain gauge location 23 It is recommended that non-linear solution methods be used to predict this local effect.

It is recommended that comparisons be made against raw strain measurements, taking into account any adjustments needed for dc offsets, uncertainties of positions, size and orientation of gauges This will remove some of the uncertainties of believed to be present in the processed stresses An assessment of cycle to cycle variability of strain measurements is also desirable

The fatigue life prediction Procedure has not been tested with industrial test cases This will be a necessary step to ascertain the Procedure's reliability

To optimise the geometry of the wheel and reduce material production costs to a minimum it will be necessary to develop a revised solid model with large reductions in cross sectional area A methodical systematic approach could be developed to do this rather than optimisation by simple design modification

REFERENCES

[1] Cook, R D., Malkus, D. S and Plesha M. E *'Concepts and Applıcatıons of Finite Element Analysıs'*, (1989), Published by John Wiley & Sons, Inc., New York.

[2] *NAFEMS -A Fınıte Element Primer,* (1992)

[3] Segerling, L J., *Applıed Fınıte Element analysis,* (1984), Published by John Wiley and Sons, Inc , New York

[4] Davies, A J., *The Fınıte Element, a Fırst Approach,* (1980), Published by Clarendon Press, Oxford.

[5] Strang, G., Fix, G. J.; *An Analysıs of the Fınıte Element Method,* (1973), Published by Prentice-Hall, Inc , Englewood cliffs, N. J.

[6] Turner, M. J , Clough, R W, Martin, H C and Topp, L J , *'Stıffness and Deflection Analysis of Complex Structures'*, (1956), Journal of Aeronautical Science, Vol 23, pp 805-832

[7] Argyris, J. H., and Kelsey, S , *'Energy Theorems and structural analysıs Aırcraft Engıneerıng'*, (1955), Vols 26 & 27.

[8] Clough, R W. *'The Fınıte Element ın Plane Stress Analysis'*, (1960), Proc. 2nd A S.C.E Conf. on Electronic Computation, Pittsburgh

[9] Robinson, J, *'The Lıves and Work of Early FEM Pioneers'*, (1985), Robinson and Associates, Horton Road Woodlands, Wimborne, Dorset, England

[10] Fenner, D N, *Engıneering Stress Analysis, a Finite Element Approach with Fortran 77 software*, (1987), Published by John Wiley and Sons, New York.

[11] Dietrich, D. E, *'Twenty years of Fınıte Elements in Desıgn'*, (1991), Swanson Analysis Systems Inc, Houston, Pennsylvania

[12] Helmut F Schweiger, *'Fınıte Elements ın Engıneerıng - a Short Introduction'*, (1990), Radex Rundschau, pp 317-326.

[13] Gotoh, E., Ohsawa, S, Satoh, Y, Yasuda, S, *"The Investıgatıon of Steerıng Column Collapse Behavıour using Finite Element Analysıs"*, (1992), SAE, International Congress and Exposition, Detroit, Michigan, February 24-28.

[14] Sharman, P W, Al-Hammoud, A, *"The Effect of Local Detaıls on the Stiffness of Car Body Joints"*, (1987), Int J of Vehicle Design, Vol 8, nos. 4/5/6, pp 526-537

[15] Morita, Y., Kawashima, H, Ishıhara, K, *"Fınite Element Stress Analysis of Automotıve Steel Road Wheel"*, (1987), Sumitomo Metals, V 39, p 245-263, 3 rd August

[16] Wood, Jim, ' *The Evolvıng shape of Finıte Elements*', (1993), Professional Engineering, Vol. 6(5), pp. 15-17.

[17] M.Khosrowjerdi, *'Utilisation of a Commercial Finite Element Package for Correlation of Analytical and Experimental Analysis'*, (1989), ASME, International Computer in Engineering Conference and Expo., Vol. 2 (of 2).

[18] W. Charles Paulsen, *'Product Design on a PC'*, (1992), Vol. 26 (1)pp. 58-64 , V Sound and Vibration.

[19] Swanson, J. A., 'Current and Future Trends in Finite Element Analysis', (1994), International Journal of Computer Applications in Technology, Vol. 7, Nos. 3-6, pp. 108-117.

[20] Gallagher, R. S and Selker, P. J., *'Three Dimensional volume visualisation in FE Analysis'*, (1992), mechanical Engineering, pp 54-57.

[21] Rakin, J. J and Ott, D A., *'The Open Approach to FEA Integration in the Design Process'*, (1992), Mechanical Engineering, pp 70-75.

[22] Berry, D. T., Leewood, A. R., *'FEA for Production Design'*, (1989), Machine Design, September 21.

[23] West, M. P., *'Finite Element Analysis : a 3-D View'*, (1993), Aerospace America, June issue.

[24] Dubensky, R. G., *"Automated Finite Element Mesh Generation from solids"*, 1992, SAE, International Congress and Exposition, Detroit, Michigan, February 24-28.

[25] El-Sayed, M. E. M. and Lund, E. H., *'Structural Optimisation with Fatigue Life Constraints'*, (1990), Engineering Fracture Mechanics Vol. 37, No. 6, pp. 1149-1156.

[26] Mitchell, M R, *'Fundamentals of Modern Fatigue Analysis for Design'*, Rockwell International Science Centre, Thousand Oaks, California

[27] Tivey, C. J., *'The utilisation of Fatigue Life Prediction Techniques in Support of a Major Vehicle Project'*, (1986), proceedings of the Institute of Mechanical Engineers, Suffolk, Part D.

[28] Fatigue Analysis, *'A technical background'*, nCode International ltd

[29] Sonsino, C. M., *'Fatigue Life Prediction for Large Size Components'*, (1991), Methods for Fatigue Life Prediction of Structures, Split

[30] Haibach, E, *'Fatigue Data for Design Applications'*, (1981), Materials, Experimentation and Design in Fatigue, Proceedings.

[31] Spindel, J. E. and Haibach, E, *'Statistical analysis of Fatigue data'*, (1981), ASTM.

[32] Gaβner , E, *'Effect of variable load and cumulative damage of fatigue in vehicle and aeroplane structures'*, (1956), Proceedings of the International Conference on Fatigue of metals London - New York, Ed. by Institution of Mechanical Engineers, London, pp 304-309.

[33] Swanson, R., *'Random Load Fatigue Testing - a state of the art survey'*, (1968), Materials Research and Standards 8, pp 10-44

[34] Landgraf, R W, *'Control of Fatigue Resistance Through Micro structure - Ferrous Alloys'*, Ford Motor Co , Dearbourn, Michigan

[35] Feltner, C. E., and Beardmore, P , (1970), ASTM STP 467, p77

[36] Grosskreutz, J C , ASTM STP 495, (1971), p5.

[37] Laird, C., *'Alloy and Micro structural Design'*, (1976), New York, Academic press, p175.

[38] 'Fatigue under complex Loading', SAE Warrendale, PA , (1977)

[39] Boresi, A P , Schmidt, R J , Sidebottom, O M , *Advanced Mechanics of Materials,* fifth edition, (1993), Published by John Wiley and Sons, Inc

[40] Mitchell, M R , *"A Unified Predictive Technique for the Fatigue Resistance of Cast Ferrous-Based Metals and High Hardness Wrought Steels,"* .Sept (1976), College of Engineering, Univ of Illinois, Fracture Control Report No. 23, SAE, 1978.

[41] Landgraf, R W , and LaPointe N R., SAE Trans., Vol. 83, (1974), P 1198

[42] Landgraf, R. W., Richards F D and LaPointe N R , SAE Trans , Vol. 84, (1975), p249

[43] Agrawal, H., Gopalakrishnan, R , Rivard, C , *'Durability assessment of large automotive body structures using Fatigue Life Analysis Procedure (FLAP)'*, (1994), Int J of Computer Applications in Technology, Vol 7, Nos. 3-6, pp. 250-270

[44] Gopalakrishnan, R, Agrawal, H, *'Durability Analysis of Full Automotive Body Structures'*, (1993), Simulation and Development in Automotive simultaneous Engineering, SAE special publication n973, pp 1-17

[45] Tivey, C. J., *'The Utilisation of Fatigue Life Prediction techniques in support of a major vehicle project'*, (1986), Proceedings of Institution of Mechanical Engineers, Part D

[46] Jung, W. W., Min, B H, *'Fatigue Life Prediction using S-N Curve modification'*, (1991), Proceedings of the sixth International Pacific Conference on Automotive Engineering

[47] Grubisic, V., Lowak, H, *'Life Prediction and Test Results of Aluminium Alloy Components'*, (1986), Fatigue Prevention and Design', Ed by J T Barnby, Engineering Materials Advisory Services, (EMAS), Warley, S 171-187

[48] *IDEAS Master Series Product Catalogue*, Version 2, July, 1993

[49] *Technical note TM Nr. 92/83*, Fraunhofer-Institut fur Betriebsfestigkeit,
Darmstadt.

[50] Grubisic, V, Mahing, F, Pavlossky, R., *'A Method for the Design of Wheel Spiders and Rims'*, (1973), ATZ 75, pp 324-330

[51] Buxbaum, O., Grubisic, V, *'A Method for the Optimum Design of Aircraft Wheels'*, (1979), Proceeding of the tenth ICAF Symposium

[52] Grubisic, V., Fischer, G , *'Automotive wheel, Method and Procedure for*
Optimal Design and Testing', (1983), SAE Technical Paper Series, International Congress and Exposition, February 28 -March 4th.

[53] V.Grubisic and G.Fischer, *'Biaxial Wheel / Hub Test Facility'*, Proceedings of the first International User's Meeting, (1993), September 30 th, Darmstadt.

[54] V.Grubisic, *'Fatigue of Materials and Structures under Operational Stresses'*, (1990), Split, Seminar Pogonsica Cvrstoca

APPENDIX

Appendix A

Software and hardware specification.

HARDWARE

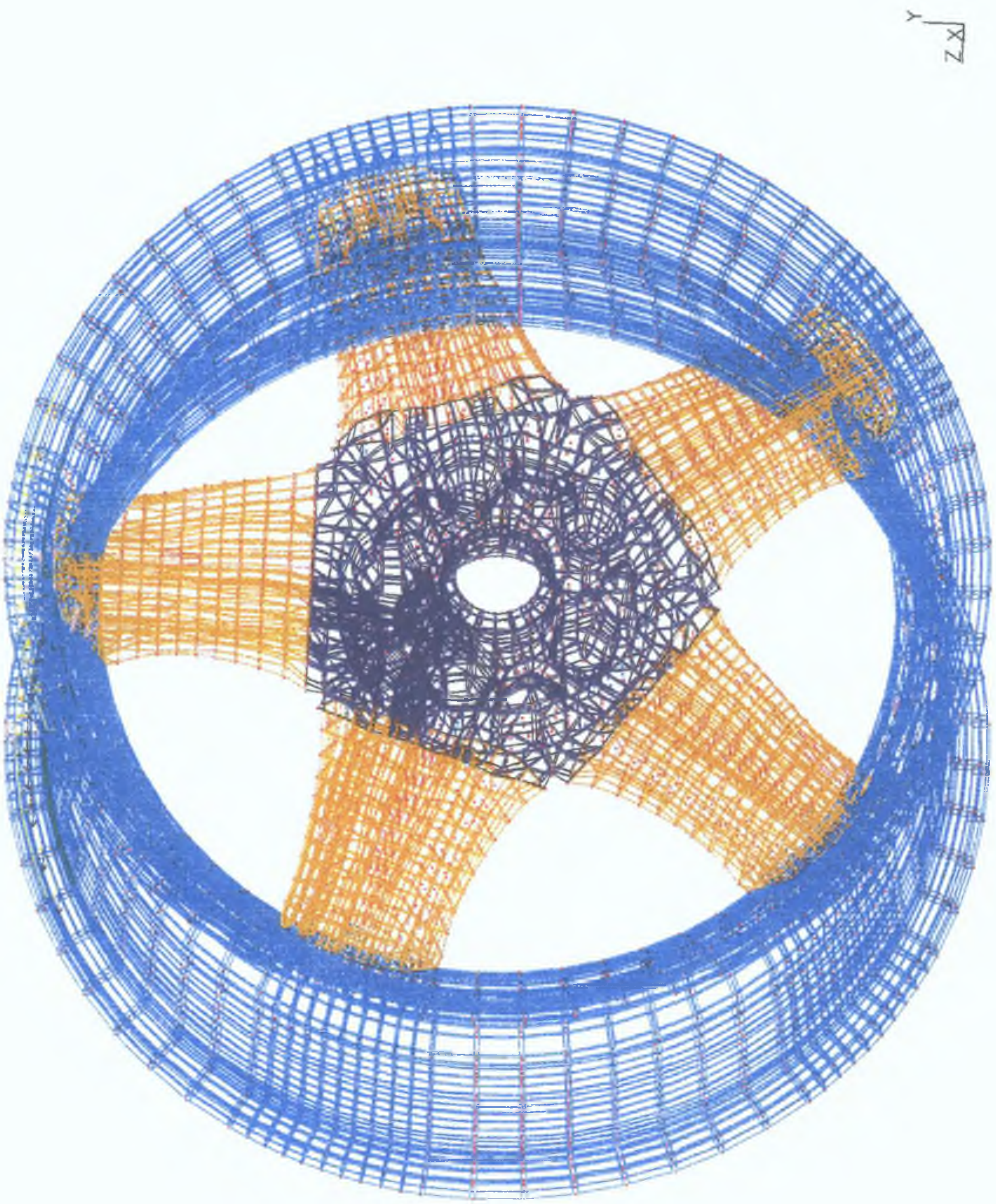
INDY SC R4000	(64 MB RAM / 199 MB SWAP)	W8B-5064-E
630 MB SYSTEM DISK		
2 GB EXTERNAL DISK		
EXTERNAL 2GB 4mm SCSI TAPE DRIVE		
EXTERNAL CD-ROM SCSI DRIVE		
8 BIT COLOUR 19 inch MONITOR		
INDYCAM CAMERA		

SOFTWARE

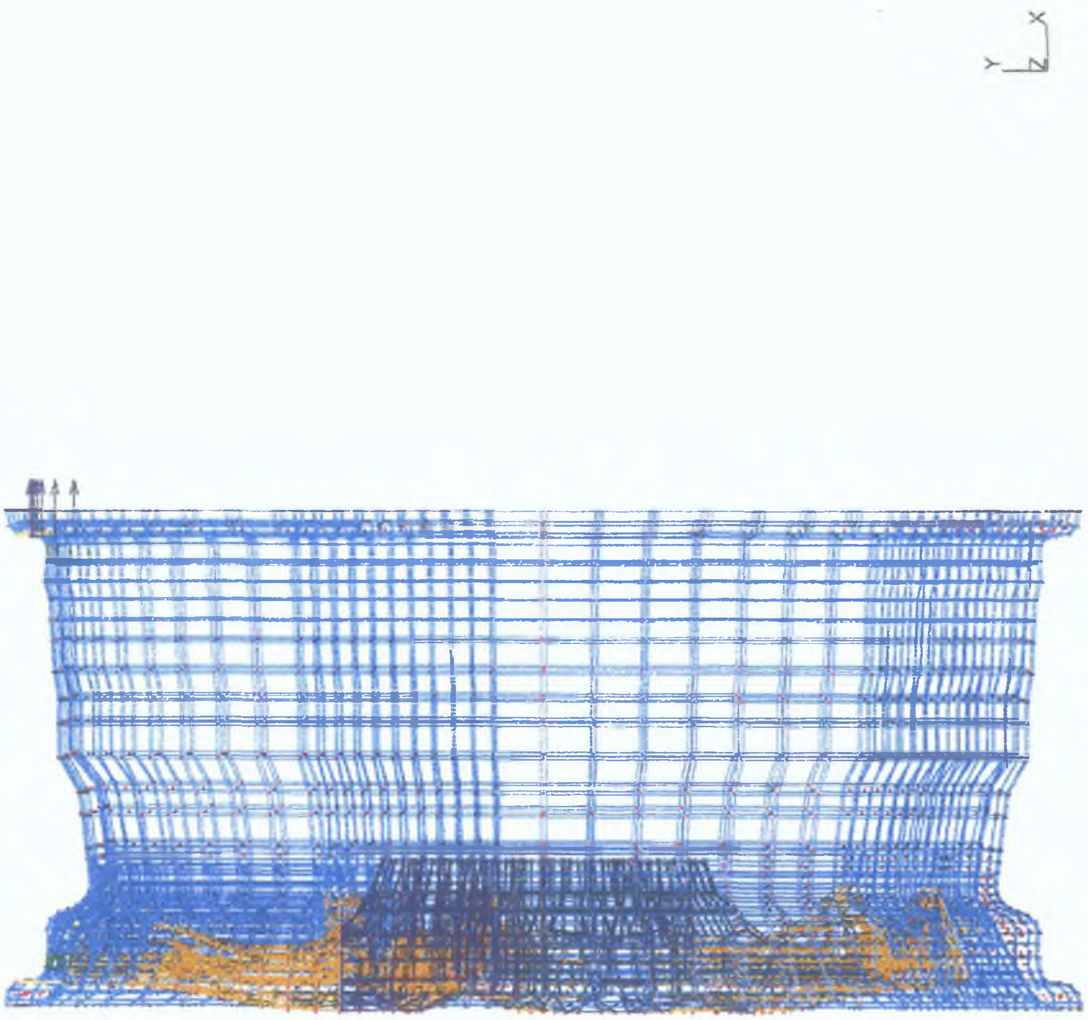
IRIX MASTERCOPY	SGI
NFS LICENCE	SGI
I-DEAS SMARTVIEW ENGLISH	SDRC
ADVANCED FEM PACKAGE	"
SIMULATION ADVISOR	"
MODEL SOLUTION	"
MASTER SURFACING	"

Appendix B

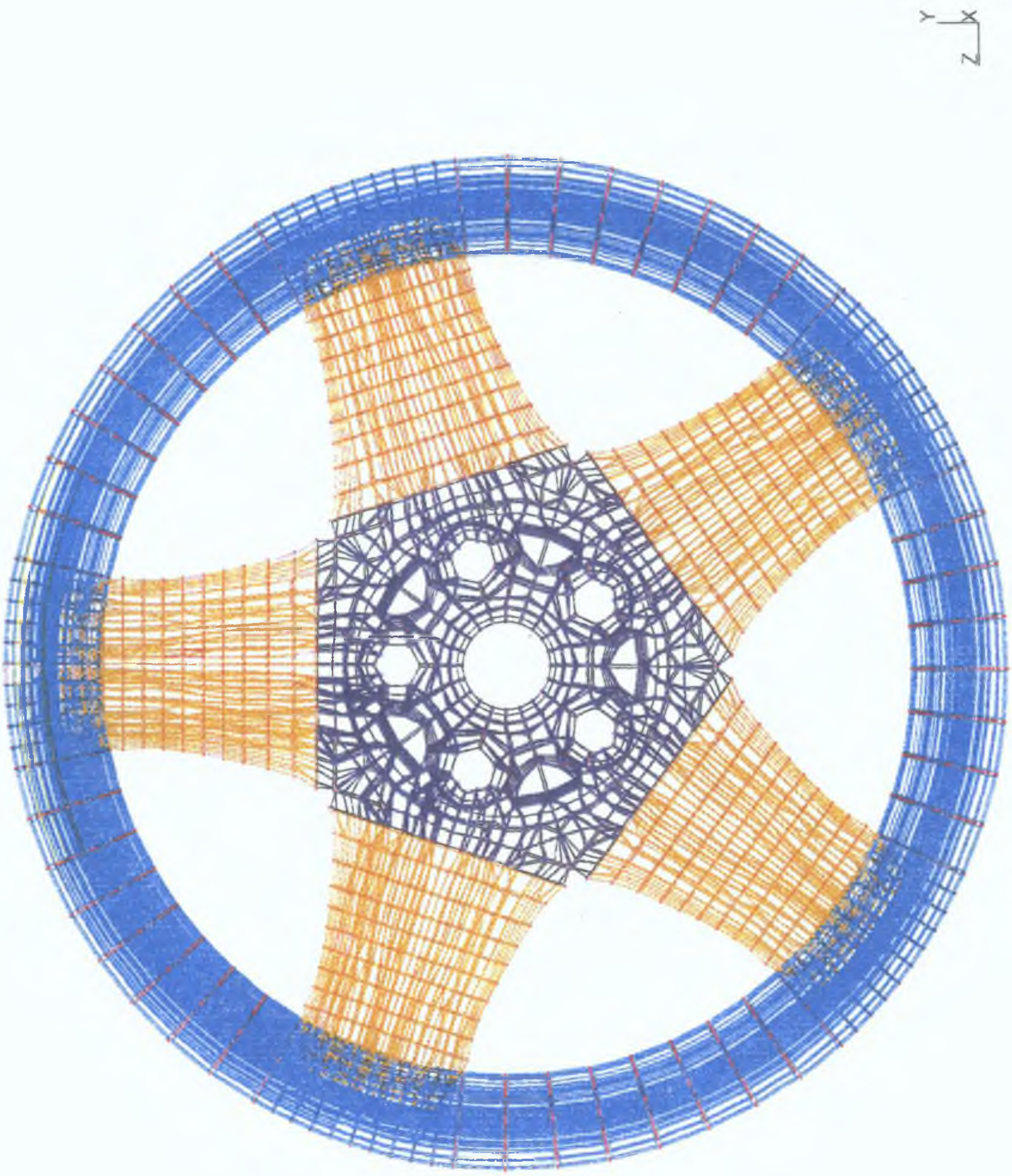
Some pictorial representations of the mesh.



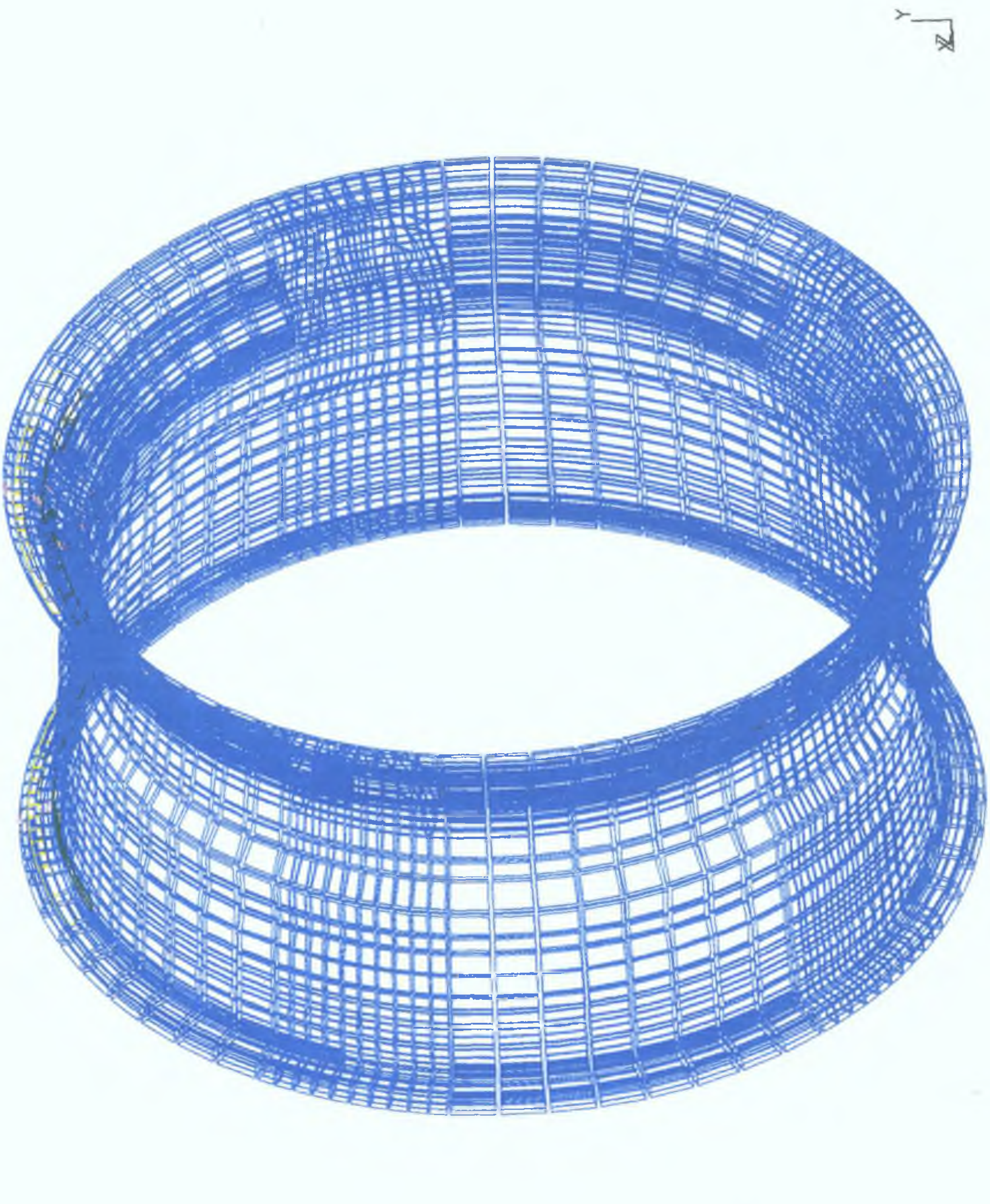
Structural mesh-wheel overview



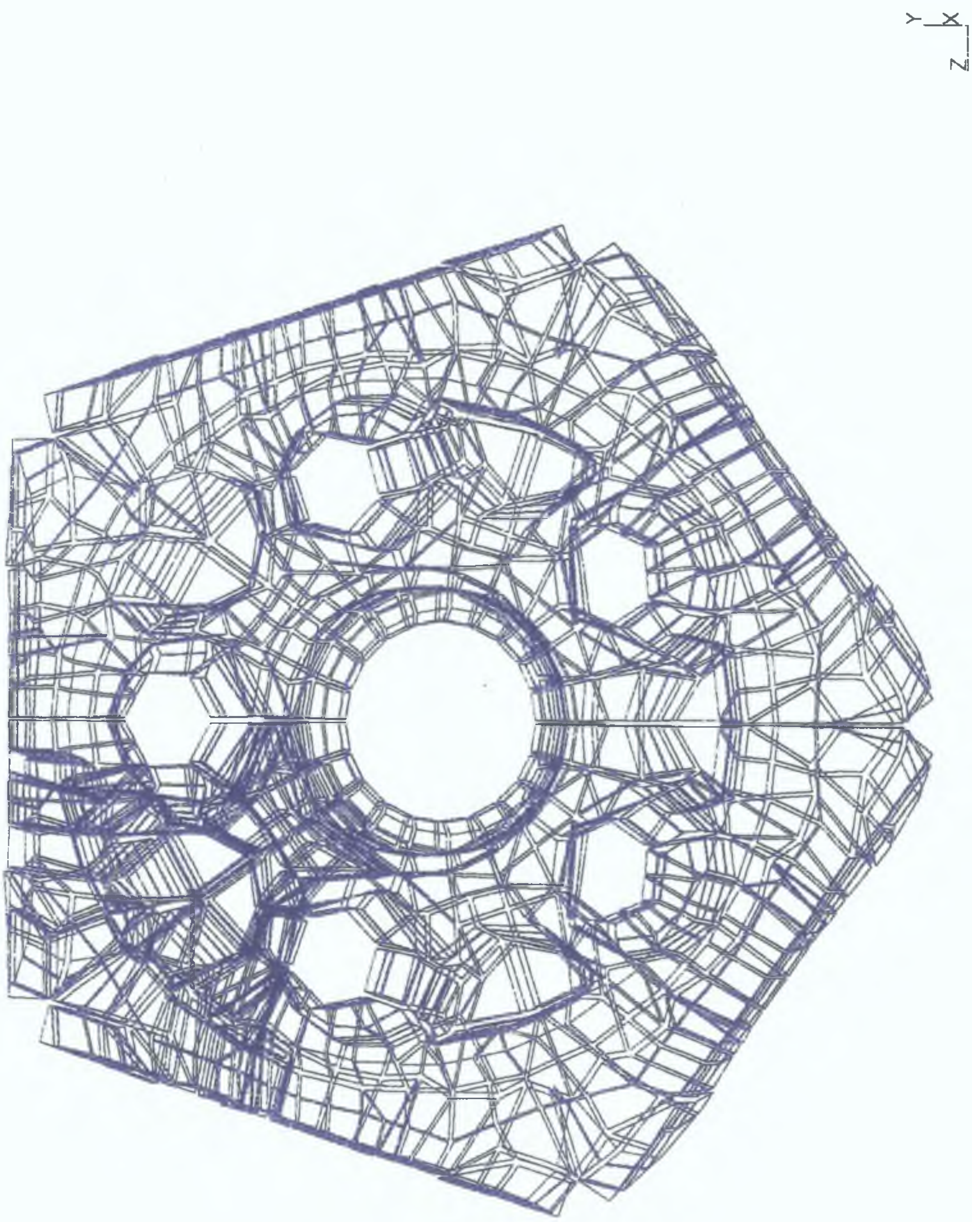
Sturctural mesh - side view



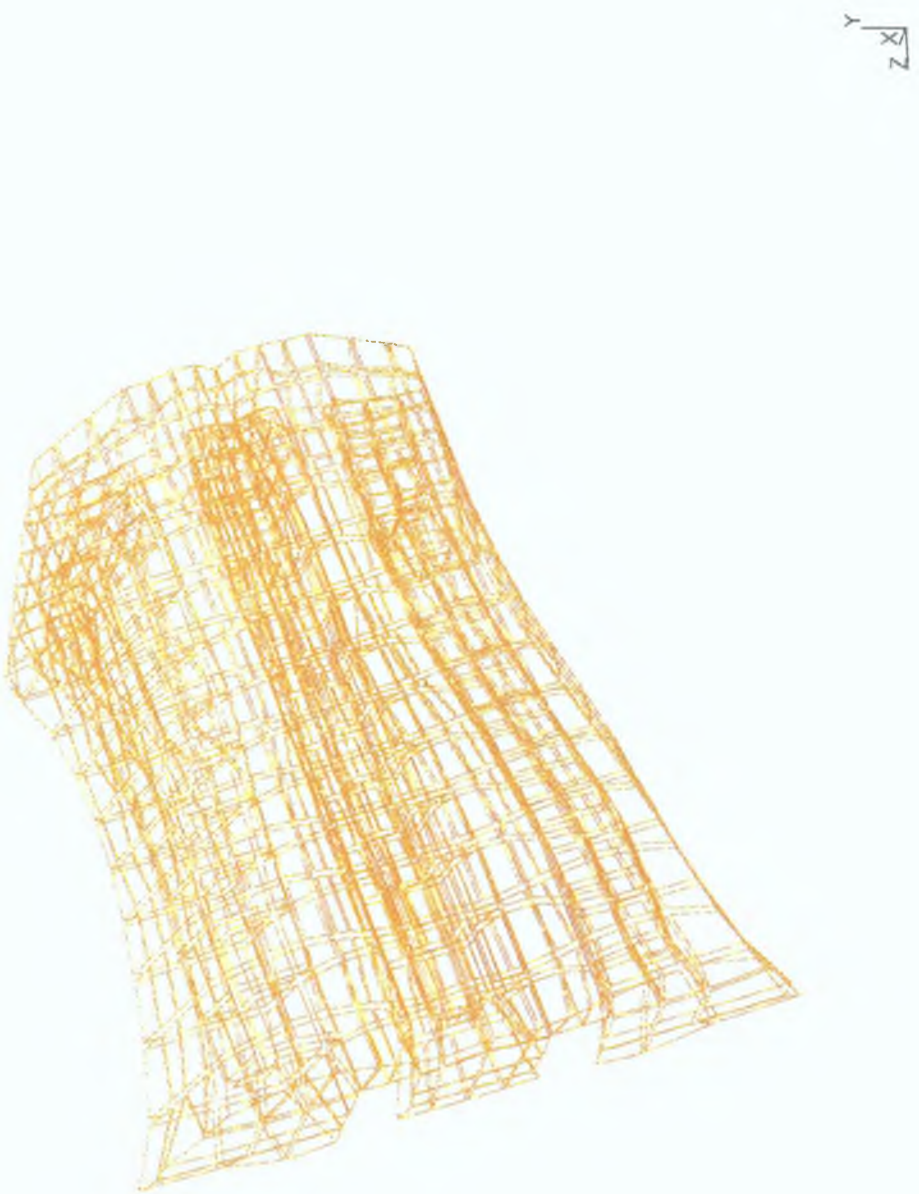
Structural mesh - wheel



Structural mesh - rim detail



Structural mesh - hub detail



Structural mesh spoke detail

Appendix C

The magnitude of the loads at each nodal position on the wheel rim

Load case	R30VI / R30HO / R30HI		Location	Through spoke one
			Total angle	30 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	12.00	0.00	0.00
	9.60	36.00	5.45
	7.20	64.00	9.70
	4.80	84.00	12.73
	2.40	96.00	14.55
	0.00	100.00	15.14
	2.40	96.00	14.55
	4.80	84.00	12.73
	7.20	64.00	9.70
	9.60	36.00	5.45
	12.00	0.00	0.00
Total	24 degrees	6.60 N	1.00 N

TABLE 1

Load case	R30VO		Location	Through spoke one
			Total angle	30 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	12.00	0.00	0.00
	9.60	36.00	5.33
	6.40	71.56	10.60
	4.80	84.00	12.44
	2.40	96.00	14.22
	0.00	100.00	14.82
	2.40	96.00	14.22
	4.80	84.00	12.44
	6.40	71.56	10.60
	9.60	36.00	5.33
	12.00	0.00	0.00
Total	24 degrees	6.7512N	1.00N

TABLE 2

Load case	**F30 VI / F30HO / F30HI / F30VO**	Location	Between spokes
		Total angle	30 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	12.00	0.00	0.00
	6.00	75.00	30.00
	0.00	100.00	40.00
	6.00	75.00	30.00
	12.00	0.00	0.00
Total	24 degrees	2.5 N	1 N

TABLE 3

Load case	**R50 VI / R50HO / R50HI**	Location	Through spoke one
		Total angle	50 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	24.00	0.00	0.00
	18.00	43.75	4.06
	12.00	75.00	6.96
	9.60	84.00	7.80
	7.20	91.00	8.45
	4.80	96.00	8.91
	2.40	99.00	9.19
	0.00	100.00	9.26
	2.40	99.00	9.19
	4.80	96.00	8.91
	7.20	91.00	8.45
	9.60	84.00	7.80
	12.00	75.00	6.96
	18.00	43.75	4.06
	24.00	0.00	0.00
Total	48 degrees	10.775 N	1.00 N

TABLE 4

Load case	**R50VO**		Location Total angle	Through spoke one 30 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	24.00	0.00	0.00
	18.00	43.75	4.01
	12.00	75.00	6.87
	8.00	88.89	8.15
	6.40	92.89	8.51
	4.80	96.00	8.80
	2.40	99.00	9.07
	0.00	100.00	9.18
	2.40	99.00	9.07
	4.80	96.00	8.80
	6.40	92.89	8.51
	8.00	88.89	8.15
	12.00	75.00	6.87
	18.00	43.75	4.01
	24.00	0.00	0.00
Total	48 degrees	10.9106 N	1.00 N

TABLE 5

Load case	**F50VI / F50HO / F50HI / F50V0**		Location Total angle	Between spokes 50 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	24.00	0.00	0.00
	18.00	43.75	8.33
	12.00	75.00	14.29
	6.00	93.75	17.86
	0.00	100.00	19.04
	6.00	93.75	17.86
	12.00	75.00	14.29
	18.00	43.75	8.33
	24.00	0.00	0.00
Total	48 degrees	5.25 N	1.00 N

TABLE 6

Load case **R60VI / R60HO / R60HI**

Location: Through spoke one
Total angle: 60 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	30.00	0.00	0.00
	24.00	36.00	3.43
	18.00	64.00	6.09
	12.00	84.00	8.00
	7.20	94.24	8.97
	4.80	97.44	9.28
	2.40	99.36	9.47
	0.00	100.00	9.52
	2.40	99.36	9.47
	4.80	97.44	9.28
	7.20	94.24	8.97
	12.00	84.00	8.00
	18.00	64.00	6.09
	24.00	36.00	3.43
	30.00	0.00	0.00
Total	60 degrees	10.5008 N	1.00 N

TABLE 7

Load case	**R60VO**		**Location**	**Through spoke one**
			Total angle	**60 degrees**

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	30.00	0.00	0.00
	24.00	36.00	2.91
	18.00	64.00	5.17
	12.00	84.00	6.78
	8.00	92.89	7.51
	6.40	95.45	7.71
	4.80	97.44	7.87
	2.40	99.36	8.01
	0.00	100.00	8.08
	2.40	99.36	8.01
	4.80	97.44	7.87
	6.40	95.45	7.71
	8.00	92.89	7.51
	12.00	84.00	6.78
	18.00	64.00	5.17
	24.00	36.00	2.91
	30.00	0.00	0.00
Total	60 degrees	12.3828 N	1.00 N

TABLE 8

Load case	**F60VI / F60HO / F60HI**		Location Total angle	Between spokes 60 degrees
	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)	
	28.80	0.00	0.00	
	26.40	15.97	2.38	
	24.00	30.56	4.55	
	18.00	60.94	9.07	
	12.00	82.64	12.31	
	6.00	95.66	14.24	
	0.00	100.00	14.90	
	6.00	95.66	14.24	
	12.00	82.64	12.31	
	18.00	60.94	9.07	
	24.00	30.56	4.55	
	26.40	15.97	2.38	
	28.80	0.00	0.00	
Total	57.6 degrees	6.7154 N	1.00 N	

TABLE 9

Load case	F60VO	Location	Between spokes
		Total angle	60 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	29.60	0.00	0.00
	28.00	10.52	1.46
	26.00	22.85	3.17
	24.00	34.26	4.76
	18.00	63.02	8.75
	12.00	83.56	11.61
	6.00	95.89	13.31
	0.00	100.00	13.88
	6.00	95.89	13.31
	12.00	83.56	11.61
	18.00	63.02	8.75
	24.00	34.26	4.76
	26.00	22.85	3.17
	28.00	10.52	1.46
	29.6	0.00	0.00
Total	59.2 degrees	7.20 N	1.00N

TABLE 10

Load case	**R70VI / R70HO / R70HI**	Location	Through spoke one
		Total angle	70 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	36.00	0.00	0.00
	30.00	30.56	2.57
	24.00	55.56	4.68
	18.00	75.00	6.32
	12.00	88.89	7.48
	7.20	96.00	8.08
	4.80	98.22	8.27
	2.40	99.56	8.38
	0.00	100.00	8.44
	2.40	99.56	8.38
	4.80	98.22	8.27
	7.20	96.00	8.08
	12.00	88.89	7.48
	18.00	75.00	6.32
	24.00	55.56	4.68
	30.00	30.56	2.57
	36.00	0.00	0.00
Total	72 degrees	11.8758 N	1.00 N

TABLE 11

Load case	R70VI / R70HO / R70HI	Location Total angle	Through spoke one 70 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	36.00	0.00	0.00
	30.00	30.56	2.22
	24.00	55.56	4.03
	18.00	75.00	5.44
	12.00	88.89	6.44
	8.00	95.06	6.89
	6.40	96.84	7.02
	4.80	98.22	7.12
	2.40	99.56	7.22
	0.00	100.00	7.24
	2.40	99.56	7.22
	4.80	98.22	7.12
	6.40	96.84	7.02
	8.00	95.06	6.89
	12.00	88.89	6.44
	18.00	75.00	5.44
	24.00	55.56	4.03
	30.00	30.56	2.22
	36.00	0.00	0.00
Total	72 degrees	13.7938 N	1.00 N

TABLE 12

Load case **F70VI / F70HO / F70HI** Location Between spokes
Total angle 70 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	31.20	0.00	0.00
	28.80	14.79	1.93
	26.40	28.40	3.71
	24.00	40.83	5.34
	18.00	66.72	8.73
	12.00	85.21	11.15
	6.00	96.30	12.60
	0.00	100.00	13.08
	6.00	96.30	12.60
	12.00	85.21	11.15
	18.00	66.72	8.73
	24.00	40.83	5.34
	26.40	28.40	3.71
	28.80	14.79	1.93
	31.20	0.00	0.00
Total	62.4 degrees	7.645 N	1.00 N

TABLE 13

Load case	F70VO	Location	Between spokes Total angle 70 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	31.20	0.00	0.00
	29.60	9.99	1.25
	28.00	19.46	2.44
	26.00	30.56	3.83
	24.00	40.83	5.11
	18.00	66.72	8.36
	12.00	85.21	10.68
	6.00	96.30	12.07
	0.00	100.00	12.57
	6.00	96.30	12.07
	12.00	85.21	10.68
	18.00	66.72	8.36
	24.00	40.83	5.11
	26.00	30.56	3.83
	28.00	19.46	2.44
	29.60	9.99	1.25
	31.20	0.00	0.00
Total	62.4 degree	7.9814	1.00 N

TABLE 14

Load case	**R120VI / R120HO / R120HI**	Location total angle	Through spoke one 120degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	60.00	0.00	0.00
	54.00	19.00	11.00
	48.00	36.00	20.84
	42.00	51.00	29.52
	36.00	64.00	37.05
	30.00	75.00	43.41
	24.00	84.00	48.62
	18.00	91.00	52.68
	12.00	96.00	55.57
	7.20	98.56	57.05
	4.80	99.36	57.52
	2.40	99.84	57.79
	0.00	100.00	57.90
	2.40	99.84	57.79
	4.80	99.36	57.52
	7.20	98.56	57.05
	12.00	96.00	55.57
	18.00	91.00	52.68
	24.00	84.00	48.62
	30.00	75.00	43.41
	36.00	64.00	37.05
	42.00	51.00	29.52
	48.00	36.00	20.84
	54.00	19.00	11.00
	60.00	0.00	0.00
Total	120 degrees	17.2752 N	1.00 N

TABLE 15

Load case	**R120VO**		Location Total angle	Through spoke one 120 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	60.00	0.00	0.00
	54.00	19.00	9.87
	48.00	36.00	18.71
	42.00	51.00	26.50
	36.00	64.00	33.25
	30.00	75.00	38.97
	24.00	84.00	43.65
	18.00	91.00	47.28
	12.00	96.00	49.88
	8.00	98.22	51.04
	6.40	98.86	51.37
	4.80	99.36	51.63
	2.40	99.84	51.87
	0.00	100.00	51.96
	2.40	99.84	51.87
	4.80	99.36	51.63
	6.40	98.86	51.37
	8.00	98.22	51.04
	12.00	96.00	49.88
	18.00	91.00	47.28
	24.00	84.00	43.65
	30.00	75.00	38.97
	36.00	64.00	33.25
	42.00	51.00	26.50
	48.00	36.00	18.71
	54.00	19.00	9.87
	60.00	0.00	0.00
Total	120 degrees	19.2456 N	1.00 N

TABLE 16

Load case	**F120VI / F120HO / F120HI**	Location	Between spokes
		Total angle	120 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	60.00	0.00	0.00
	54.00	19.00	9.92
	48.00	36.00	18.79
	45.60	42.24	22.05
	43.20	48.16	25.14
	40.80	53.76	28.06
	38.40	59.04	30.81
	36.00	64.00	33.40
	33.60	68.64	35.83
	31.20	72.96	38.08
	28.80	76.96	40.17
	26.00	81.22	42.39
	18.00	91.00	47.50
	12.00	96.00	50.11
	6.00	99.00	51.66
	0.00	100.00	52.18
	6.00	99.00	51.66
	12.00	96.00	50.11
	18.00	91.00	47.50
	26.00	81.22	42.39
	28.80	76.96	40.17
	31.20	72.96	38.08
	33.60	68.64	35.83
	36.00	64.00	33.40
	38.40	59.04	30.81
	40.80	53.76	28.06
	43.20	48.16	25.14
	45.60	42.24	22.05
	48.00	36.00	18.79
	54.00	19.00	9.92
	60.00	0.00	0.00
Total	120 degrees	19.1596 N	1.00 N

TABLE 17

Load case **F120VO** Location Between spokes
Total angle 120 degrees

	Node Angle	Force on Node (parabolic distribution)	Force on node (for unit load case)
	60.00	0.00	0.00
	54.00	19.00	9.51
	48.00	36.00	18.02
	44.00	46.22	23.13
	40.80	53.76	26.91
	38.40	59.04	29.55
	36.00	64.00	32.03
	33.60	68.64	34.35
	31.20	72.96	36.51
	28.00	78.22	39.15
	26.00	81.22	40.65
	24.00	84.00	42.04
	18.00	91.00	45.54
	12.00	96.00	48.05
	6.00	99.00	49.54
	0.00	100.00	50.04
	6.00	99.00	49.54
	12.00	96.00	48.05
	18.00	91.00	45.54
	24.00	84.00	42.04
	26.00	81.22	40.65
	28.00	78.22	39.15
	31.20	72.96	36.51
	33.60	68.64	34.35
	36.00	64.00	32.03
	38.40	59.04	29.55
	40.80	53.76	26.91
	44.00	46.22	23.13
	48.00	36.00	18.02
	54.00	19.00	9.51
	60.00	0.00	0.00
Total	120 degrees	19.9812 N	1.00 N

TABLE 18

Appendix D

An output list file from the I-DEAS software for a sample linear elastic analysis

```
MODEL FILE : /usrl/people/mary/mod/Var1-modified_wheel.mf1
MODEL FILE DESCRIPTION : /usrl/people/mary/mod/Var1-modified_wheel.mf1
ACTIVE UNITS SYSTEM : User defined
TEMPERATURE MODE    : Relative Temperatures

 Executing: Check matrix statistics
 Total number of elements processed      : 13978
 Total number of nodes processed         : 19361
 Maximum node degree                     : 32

 Matrix statistics
                            Existing      Resequence
                            Sequence         Map
  Max bandwidth              18815          3681
  Avg bandwidth                781           396
  RMS bandwidth               2267           613
  Profile                 15140157       7671328

11:48:25 (CP     12.56      12.56)     LINEAR STATIC ANALYSIS
11:48:26 (CP      0.34      12.90)      Solution No Restart
11:48:26 (CP      0.01      12.91)
11:48:26 (CP      0.25      13.16) Hypermatrix File Opened
11:49:41 (CP     66.81      79.97) Physical Properties Formed
11:49:50 (CP      8.01      87.98) Offset Tables Formed
 W  21639 FOR MATERIAL NO. 8
         SHEAR MODULUS INCONSISTENT WITH ELASTIC MODULUS
         AND POISSONS RATIO. FOR AN ISOTROPIC MATERIAL
         G=E/2(1+NU) WILL BE USED. IF YOU WISH TO VIOLATE
         THIS CONSTRAINT USE ORTHOTROPIC MATERIALS.
11:50:04 (CP     12.43     100.41) Material Tables Formed
11:50:04 (CP      0.05     100.46) Freedom Table and Constraint Matrices Formed
11:50:42 (CP     36.13     136.59) Boolean Formation Complete
11:50:42 (CP      0.02     136.61) Begin Constraint Partitioning
11:50:43 (CP      1.02     137.63) Constraint Partitioning Complete
11:50:43 (CP      0.30     137.93) Constraint Elimination Complete
11:50:43 (CP      0.02     137.95) Begin Stiffness Matrix Formation
11:53:38 (CP    165.05     303.00) Stiffness Matrix Formation Complete
11:55:35 (CP     99.34     402.34) Stiffness Partitions Formed
11:56:17 (CP     33.45     435.79) Sparse solve will be done with minimum memory♦
11:56:20 (CP      0.16     435.95) Increase application memory 15 mb for normal ♦
11:56:47 (CP     24.01     459.96) Est. decomp time = 1429 cpu seconds

  NET APPLIED LOAD FOR LOAD SET 14
     FX =  0.00000D+00, FY = -9.83829D+01, FZ = -1.08014D-06
     MX = -2.77556D-14, MY =  6.49112D-05, MZ = -5.91233D+03
     MOMENTS TAKEN ABOUT THE ORIGIN
```

```
MODEL_SOLUTION_SOLVE

 NET APPLIED LOAD FOR LOAD SET 15
    FX = -1.00000D+02, FY =  0.00000D+00, FZ =  0.00000D+00
    MX =  0.00000D+00, MY = -4.40050D-04, MZ =  2.39744D+04
    MOMENTS TAKEN ABOUT THE ORIGIN

 NET APPLIED LOAD FOR LOAD SET 30
    FX =  0.00000D+00, FY = -7.82030D+01, FZ = -5.68178D+01
    MX =  5.55112D-13, MY =  3.41447D+03, MZ = -4.69962D+03
    MOMENTS TAKEN ABOUT THE ORIGIN

 NET APPLIED LOAD FOR LOAD SET 31
    FX = -1.00000D+02, FY =  0.00000D+00, FZ =  0.00000D+00
    MX =  0.00000D+00, MY = -1.39336D+04, MZ =  1.91779D+04
    MOMENTS TAKEN ABOUT THE ORIGIN

 NET APPLIED LOAD FOR LOAD SET 40
    FX =  0.00000D+00, FY = -8.56595D+02, FZ = -7.46982D-06
    MX =  2.22045D-13, MY =  1.79775D-03, MZ = -2.06156D+05
    MOMENTS TAKEN ABOUT THE ORIGIN

 NET APPLIED LOAD FOR LOAD SET 41
    FX =  1.00000D+03, FY =  0.00000D+00, FZ =  0.00000D+00
    MX =  0.00000D+00, MY = -1.06171D-06, MZ = -2.18212D+05
    MOMENTS TAKEN ABOUT THE ORIGIN

 NET APPLIED LOAD FOR LOAD SET 49
    FX =  0.00000D+00, FY = -7.11944D+02, FZ = -5.17257D+02
    MX = -5.55112D-13, MY =  1.24488D+05, MZ = -1.71342D+05
    MOMENTS TAKEN ABOUT THE ORIGIN

 NET APPLIED LOAD FOR LOAD SET 50
    FX =  1.00000D+03, FY =  0.00000D+00, FZ =  0.00000D+00
    MX =  0.00000D+00, MY =  1.22848D+05, MZ = -1.69085D+05
    MOMENTS TAKEN ABOUT THE ORIGIN
11:58:33 (CP    88.92    548.88) Loads Constructed
11:58:34 (CP     0.26    549.14) Begin Decomposition

  CHOLESKY DECOMPOSITION STATISTICS:
  SINGULARITY CRITERIA =  1.0E-14
  NUMBER OF EQUATIONS  =  57723

  MINIMUM                                  MAXIMUM
  CHOLESKY   EQUATION   NODE AND           CHOLESKY   EQUATION   NODE AND
  PIVOTS      NUMBER    DIRECTION          PIVOTS      NUMBER    DIRECTION
```

6.2825D+06	50488	17901-X	4.7324D+09	57010	20339-X
9.9126D+06	29608	10419-X	4.7324D+09	40228	14226-X
1.1261D+07	6418	2155-X	4.7324D+09	34765	12153-X
1.2401D+07	50401	17872-X	4.7324D+09	17038	5962-X
1.3221D+07	18013	6287-X	4.7324D+09	23170	8021-X
1.3494D+07	29500	10383-X	4.7324D+09	5443	1830-X
1.4873D+07	52230	18493-Z	4.7324D+09	51715	18322-X
1.4947D+07	6310	2119-X	4.7324D+09	28633	10094-X
1.5710D+07	17905	6251-X	4.7324D+09	11575	3889-X
1.5987D+07	21895	7596-X	4.7324D+09	46360	16285-X
1.6498D+07	55912	19961-X	4.7324D+09	57004	20337-X
1.6694D+07	50489	17901-Y	4.7324D+09	34759	12151-X
1.7044D+07	16500	5782-Z	4.7324D+09	17032	5960-X
1.7803D+07	29609	10419-Y	4.7324D+09	5437	1828-X
1.8410D+07	51111	18120-Z	4.7324D+09	40222	14224-X
1.8657D+07	29610	10419-Z	4.7324D+09	28627	10092-X
1.8737D+07	10300	3464-X	4.7324D+09	46354	16283-X
1.8825D+07	55825	19932-X	4.7324D+09	11569	3887-X
2.1757D+07	38872	13762-X	4.7324D+09	51709	18320-X
2.2184D+07	56593	20200-X	4.7324D+09	23164	8019-X

```
12:22:53 (CP   1260.08   1809.22) End Of Decomposition
12:22:55 (CP      0.98   1810.20) Utility Matrices Constructed
12:26:33 (CP    146.95   1957.15) Displacement Calculation Complete
12:36:45 (CP    516.71   2473.86) Analysis Data Preparation Complete
12:36:45 (CP      0.31   2474.17) END OF SOLUTION
12:36:46 (CP      0.14   2474.31) END OF ANALYSIS
```

Appendix E

Parabolic loading distributions, strain gauge locations, stress-time histories for straight and cornering load cases and stress results for the FEA model

	Parabolic loading Outer 30 degrees		Inner 30 degrees		Parabolic loading Outer 50 degrees		Inner 50 degrees	
	Load case A		*Load case F*		*Load case A*		*Load case F*	
gauge and node no.	σ max.	σ min	σ max.	σ min	σ max.	σ min	σ max.	σ min
Spoke 1								
M3 398	-0.03	-5.31	7.70	-6.61	-0.05	-5.86	7.28	-6.20
M2 452	0.94	***min -21.67***	0.87	-15.18	-0.12	***min -21.15***	0.84	-14.22
M6 624	0.40	***min -30.58***	0.61	-13.95	0.39	***min -29.80***	0.56	-13.71
M7 623	***max 17.29***	0.48	10.54	0.30	***max 16.95***	0.47	9.86	0.29
M17 36	***max 23.86***	-0.15	1.85	-11.37	***max 20.92***	-0.14	2.70	***min -9.00***
M27 367	-1.50	***min -35.31***	0.87	-24.73	-1.49	***min -34.84***	0.91	-22.80
Spoke 2								
M3 16926	***max 13.96***	-0.55	5.71	-15.31	***max 13.60***	-0.60	5.98	***min -13.25***
M2 16980	12.35	-0.43	1.69	-4.27	12.19	-0.41	1.62	-4.74
M6 15127	11.70	-0.73	0.99	-13.65	11.19	-0.70	0.91	-13.41
M7 15126	-0.15	-8.66	10.29	0.54	-0.13	-8.16	9.63	0.51
M17 14587	-0.02	-5.98	1.85	-11.37	-0.02	-5.90	2.70	-8.97
M27 16895	23.39	0.10	4.37	-14.30	22.90	0.08	4.11	-14.17
Spoke 3								
M3 12794	2.83	-7.41	12.59	0.09	2.81	-7.01	11.43	0.03
M2 12848	7.10	-0.04	***max 19.90***	0.49	7.32	-0.02	***max 19.02***	0.44
M6 10995	5.53	-0.31	***max 15.19***	-0.38	5.74	-0.32	***max 14.84***	-0.40
M7 10994	-0.08	-4.71	-0.12	-10.20	-0.08	-4.75	-0.18	-10.15
M17 10455	4.52	0.03	5.08	-1.41	4.53	0.02	4.72	-1.55
M27 12763	7.99	-1.66	***max 32.12***	1.40	8.33	-1.42	***max 30.48***	1.34
Spoke 4								
M3 8662	0.71	-9.66	0.16	-9.70	0.66	-9.37	0.14	-9.77
M2 8716	0.43	-0.89	1.36	-0.10	0.46	-0.75	1.19	-0.10
M6 6863	5.69	-0.40	3.82	0.29	5.91	-0.39	3.53	-0.29
M7 6862	-0.07	-5.08	-0.07	-3.83	-0.08	-5.12	-0.06	-3.72
M17 6323	4.45	0.02	4.03	0.05	4.46	0.02	3.82	0.05
M27 8631	1.64	-3.74	0.22	-2.42	1.66	-3.36	0.14	-2.73
Spoke 5								
M3 4530	10.11	-1.12	3.21	-0.81	9.90	-1.15	3.00	-0.84
M2 4584	5.23	-0.25	2.50	-0.24	4.73	-0.27	2.68	-0.23
M6 2731	11.49	-0.47	15.16	-0.46	10.95	-0.47	14.82	-0.46
M7 2730	-0.23	-9.25	-0.29	***min -10.28***	-0.23	-8.75	-0.28	***min -10.22***
M17 2191	-0.02	-5.90	5.08	-1.41	-0.02	-5.80	4.72	-1.55
M27 4499	14.99	0.09	11.09	0.55	14.04	-0.04	10.98	0.55

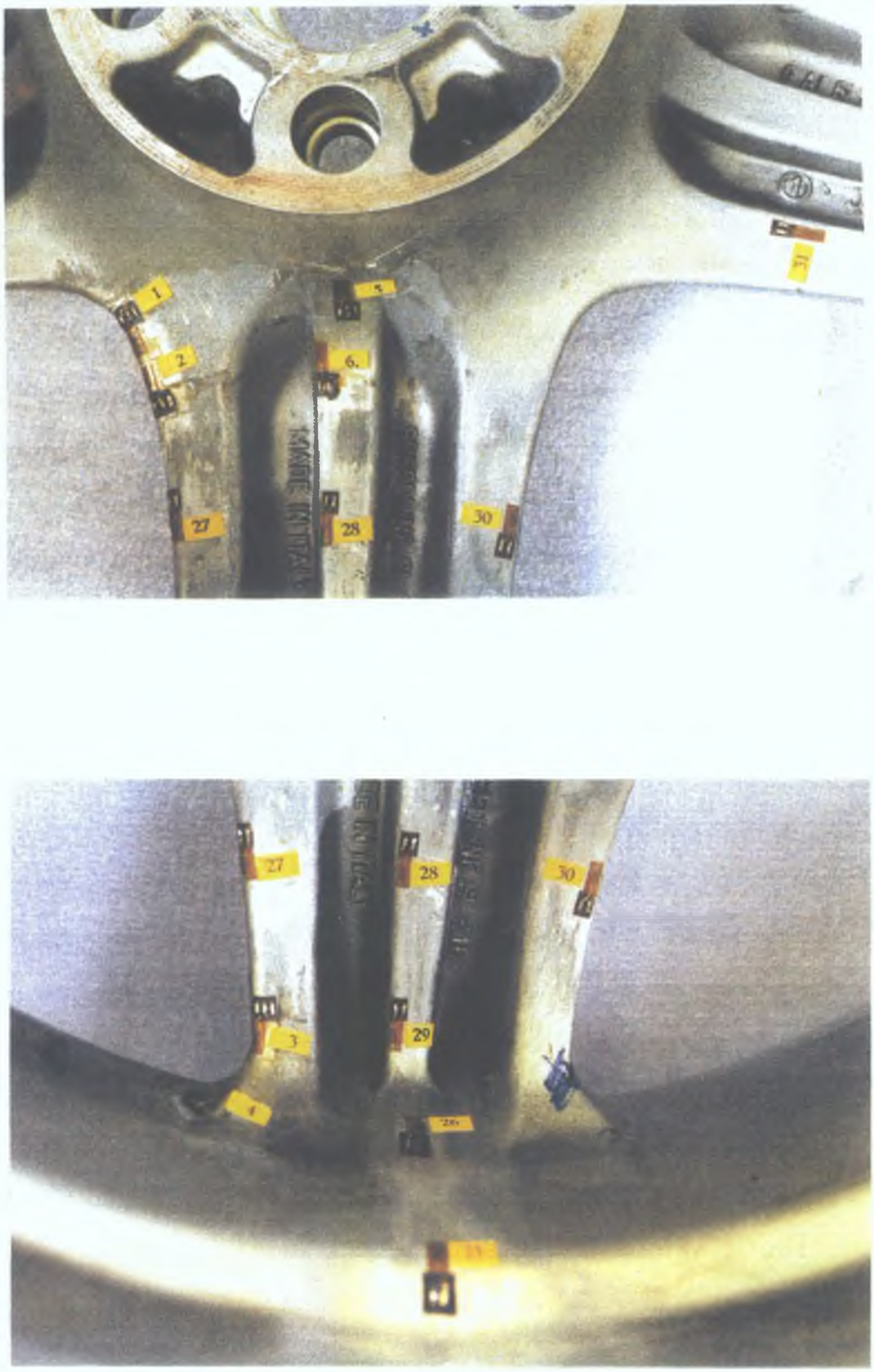
	Parabolic loading Outer 60 degrees Inner 60 degrees				Parabolic loading Outer 70 degrees Inner 70 degrees			
	Load case A		*Load case F*		*Load case A*		*Load case F*	
gauge and node no.	σ max.	σ min	σ max.	σ min	σ max.	σ min	σ max.	σ min
Spoke 1								
M3 398	-0.08	-6.23	6.88	-5.86	-0.09	-6.43	6.61	-5.57
M2 452	-0.13	***min -20.71***	0.81	-13.17	-0.14	***min -20.16***	0.77	-12.77
M6 624	0.38	***min -29.11***	0.50	-13.53	0.38	***min -28.23***	0.47	-13.21
M7 623	***max 16.67***	0.46	9.23	0.28	***max 16.28***	0.45	8.81	0.27
M17 36	***max 19.02***	-0.13	3.34	***min -6.97***	***max 16.74***	-0.11	3.71	***min -5.86***
M27 367	-1.48	***min -34.38***	0.97	-20.98	-1.45	-33.72	0.93	***min -20.09***
Spoke 2								
M3 16926	***max 13.21***	-0.62	6.21	***min -11.44***	***max 12.70***	-0.65	6.06	***min -10.73***
M2 16980	12.09	-0.39	1.56	-5.47	11.91	-0.37	1.53	-5.40
M6 15127	10.79	-0.68	0.82	-13.22	10.27	-0.65	0.78	-12.88
M7 15126	-0.11	-7.77	9.01	0.49	-0.09	-7.28	8.57	0.47
M17 14587	-0.02	-5.84	3.34	-6.97	-0.02	-5.75	3.76	-5.83
M27 16895	22.50	-0.07	3.85	-14.18	21.91	0.06	3.74	-13.67
Spoke 3								
M3 12794	2.78	-6.68	10.31	-0.03	2.75	-6.28	9.66	-0.04
M2 12848	7.48	0.00	***max 18.13***	0.39	7.67	0.01	***max 17.59***	0.37
M6 10995	5.90	-0.32	***max 14.48***	-0.41	6.08	-0.31	***max 14.16***	-0.41
M7 10994	-0.08	-4.78	-0.18	-10.10	-0.08	-4.81	-0.17	-9.94
M17 10455	4.54	0.02	4.37	-1.70	4.53	0.02	4.16	-1.73
M27 12763	8.60	-1.24	***max 28.85***	1.27	8.92	-1.04	***max 27.87***	1.22
Spoke 4								
M3 8662	0.62	-9.12	0.13	-9.83	0.58	-8.80	0.12	-9.70
M2 8716	0.50	-0.64	1.00	-0.11	0.55	-0.52	0.99	-0.10
M6 6863	6.09	-0.39	3.22	-0.29	6.31	-0.39	3.18	-0.29
M7 6862	-0.08	-5.16	-0.06	-3.59	-0.09	-5.21	-0.06	-3.55
M17 6323	4.46	0.02	3.61	0.05	4.46	0.02	3.50	0.05
M27 8631	1.69	-3.04	0.07	-3.08	1.74	-2.65	0.07	-3.04
Spoke 5								
M3 4530	9.72	-1.18	2.78	-0.88	9.45	-1.22	2.65	-0.85
M2 4584	4.32	-0.29	2.87	-0.21	3.85	-0.32	2.86	-0.20
M6 2731	10.53	-0.47	14.46	-0.46	10.00	-0.49	14.13	-0.46
M7 2730	-0.22	-8.36	-0.27	***min -10.15***	-0.21	-7.87	-0.26	***min -9.98***
M17 2191	-0.02	-5.71	4.37	-1.70	-0.02	-5.60	4.15	-1.73
M27 4499	13.27	-0.15	10.89	0.55	12.32	-0.29	10.65	0.54

	Parabolic loading Outer 120 degrees Inner 120 degrees				**Parabolic loading Outer 60 degrees Inner 120 degrees**			
	Load case A		*Load case F*		*Load case A*		*Load case F*	
gauge and node no.	σ max.	σ min	σ max.	σ min	σ max.	σ min	σ max.	σ min
Spoke 1								
M3 398	-0.12	-6.17	4.68	-3.33	0.92	-2.25	7.80	-3.28
M2 452	-0.14	***min -16.42***	0.35	-10.46	0.10	***min -24.57***	0.22	-14.48
M6 624	0.30	***min -22.70***	0.27	-11.67	1.20	***min -36.72***	1.02	-22.72
M7 623	***max 13.24***	0.36	6.41	0.21	***max 27.69***	0.63	17.80	0.38
M17 36	***max 9.83***	-0.07	4.88	-0.10	***max 7.85***	-0.10	2.84	-1.98
M27 367	-1.22	***min -28.17***	0.25	-15.23	-1.89	***min -38.78***	-0.16	-21.87
Spoke 2								
M3 16926	***max 9.57***	-0.61	4.46	-5.91	***max 21.72***	-0.55	13.54	-4.64
M2 16980	10.90	-0.29	1.13	-5.29	6.40	-1.06	2.37	-14.65
M6 15127	8.00	-0.53	0.46	-11.03	4.06	-1.69	1.47	-22.14
M7 15126	-0.03	-5.38	6.24	0.36	2.24	-1.92	17.41	0.62
M17 14587	-0.02	***min -4.66***	4.88	-0.10	-0.07	-5.85	2.84	-1.98
M27 16895	18.13	0.06	2.22	-10.74	17.87	-1.20	2.96	-20.41
Spoke 3								
M3 12794	2.33	-4.59	5.46	0.08	2.97	-5.07	10.89	0.32
M2 12848	7.48	0.06	***max 13.21***	0.24	10.81	0.06	***max 15.22***	0.27
M6 10995	6.04	-0.28	***max 11.16***	-0.39	9.28	-0.49	***max 12.34***	-0.37
M7 10994	-0.08	-4.48	-0.14	-8.22	-0.12	-6.89	-0.12	-8.72
M17 10455	4.07	0.02	2.29	-2.16	5.52	-0.04	2.73	-3.82
M27 12763	9.06	-0.43	***max 20.36***	0.94	12.99	-0.15	***max 24.90***	1.18
Spoke 4								
M3 8662	0.38	-6.97	0.12	***min -8.28***	0.24	-9.76	0.18	***min -10.73***
M2 8716	0.73	-0.18	1.95	-0.08	1.85	-0.17	4.48	-0.09
M6 6863	6.37	-0.34	4.39	-0.32	9.85	-0.53	8.00	-0.55
M7 6862	-0.01	-4.90	-0.07	-3.63	-0.16	-7.40	-0.12	-5.81
M17 6323	4.00	0.01	2.96	0.04	5.50	-0.10	3.94	0.06
M27 8631	1.80	-1.34	0.55	-1.18	2.76	-0.82	2.54	-0.48
Spoke 5								
M3 4530	7.58	-1.20	1.25	-0.43	11.11	-1.63	1.65	-0.36
M2 4584	2.19	-0.48	2.41	-0.17	2.13	-1.36	2.63	-0.19
M6 2731	7.64	-0.53	11.15	-0.41	4.84	-2.28	12.32	-0.43
M7 2730	-0.17	-5.91	-0.21	***min -8.24***	2.75	-2.92	-0.23	***min -8.72***
M17 2191	-0.01	-4.57	2.29	-2.16	-0.05	***min -5.87***	2.73	-3.82
M27 4499	8.41	-0.75	8.02	0.42	9.04	-2.04	9.46	0.48

	Parabolic loading Outer 50 degrees Inner 120 degrees				Parabolic loading Outer 50 degrees Inner 70 degrees			
	Load case A		*Load case F*		*Load case A*		*Load case F*	
gauge and node no.	σ max.	σ min	σ max.	σ min	σ max.	σ min	σ max.	σ min
Spoke 1								
M3 398	0.95	-2.19	7.79	-3.27	0.00	-0.48	0.64	-0.55
M2 452	0.10	***min -24.57***	0.21	-14.60	-0.01	***min -2.00***	0.07	-1.45
M6 624	1.20	-36.72	1.02	*min -22.68*	0.03	***min -2.81***	0.05	-1.27
M7 623	***max 27.66***	0.63	17.83	0.38	***max 1.55***	0.00	0.92	0.00
M17 36	***max 7.91***	-0.01	2.75	-2.03	***max 1.80***	0.00	0.27	-0.69
M27 367	-1.89	***min -38.74***	-0.19	-22.02	-0.14	***min -3.27***	0.05	-2.22
Spoke 2								
M3 16926	***max 21.72***	-0.55	13.31	-4.66	***max 1.27***	-0.07	0.46	***min -1.27***
M2 16980	6.37	-1.06	2.36	-14.45	1.11	0.00	0.14	-0.33
M6 15127	4.06	-1.69	1.47	-22.09	1.02	-0.06	0.09	-1.25
M7 15126	2.23	-1.93	17.43	0.63	-0.01	-0.75	0.90	0.05
M17 14587	-0.07	-5.84	2.75	-2.02	0.00	-0.55	0.27	*min -0.69*
M27 16895	17.83	-1.20	2.95	-20.29	2.12	0.00	0.36	-1.23
Spoke 3								
M3 12794	2.97	-5.09	10.96	0.32	0.26	-0.65	1.06	0.00
M2 12848	10.79	0.06	***max 15.23***	0.27	0.73	0.00	***max 1.77***	0.00
M6 10995	9.27	-0.49	***max 12.31***	-0.37	0.58	-0.03	***max 1.37***	0.00
M7 10994	-0.12	-6.89	-0.12	***min -8.68***	0.00	-0.47	0.00	-0.93
M17 10455	5.52	-0.04	2.73	-3.79	0.44	0.00	0.42	-0.15
M27 12763	12.95	-0.15	*max 24.94*	1.18	0.83	-0.12	***max 2.85***	0.13
Spoke 4								
M3 8662	0.24	-9.72	0.18	***min -10.67***	0.06	-0.89	0.00	-0.89
M2 8716	1.84	-0.17	4.54	-0.09	0.05	-0.06	0.18	0.00
M6 6863	9.82	-0.53	8.09	-0.56	0.59	-0.04	0.44	0.00
M7 6862	-0.16	-7.39	-0.12	-5.86	0.00	-0.50	0.00	-0.39
M17 6323	5.49	-0.10	3.96	0.06	0.43	0.00	0.37	0.00
M27 8631	2.75	-0.82	2.66	-0.45	0.16	-0.29	0.05	-0.15
Spoke 5								
M3 4530	11.09	-1.63	1.66	-0.35	0.89	-0.12	0.27	-0.06
M2 4584	2.13	-1.34	2.58	-0.19	0.43	0.00	0.22	0.00
M6 2731	4.83	-2.27	12.29	-0.43	1.00	-0.04	1.37	0.00
M7 2730	2.74	-2.93	-0.23	-8.68	0.00	-0.81	0.00	***min -0.94***
M17 2191	-0.05	***min -5.87***	2.73	-3.79	0.00	-0.55	0.42	-0.15
M27 4499	9.05	-2.03	9.41	0.47	1.27	0.00	0.98	0.05

	Parabolic loading Outer 120 degrees Inner 60 degrees				Parabolic loading Outer 70 degrees Inner 120 degrees			
	Load case A		*Load case F*		*Load case A*		*Load case F*	
gauge and node no.	σ max.	σ min	σ max.	σ min	σ max.	σ min	σ max.	σ min
Spoke 1								
M3 398	0.01	-5.48	0.04	-3.13	0.90	-2.31	7.81	-3.28
M2 452	6.01	-0.28	2.76	-0.23	0.10	***min -24.58***	0.22	-14.43
M6 624	***max 11.14***	-0.89	10.29	-0.70	1.20	***min -36.73***	1.02	-22.73
M7 623	-0.22	***min -12.79***	-0.15	-10.48	***max 27.73***	0.63	17.79	0.38
M17 36	***max 3.93***	0.00	2.68	0.03	***max 7.78***	-0.10	2.88	-2.00
M27 367	7.21	0.51	4.45	0.34	-1.89	***min -38.84***	-0.15	-21.80
Spoke 2								
M3 16926	0.00	***min -11.09***	0.07	-10.96	***max 21.72***	-0.55	13.62	-4.63
M2 16980	5.87	-0.83	***max 9.00***	-1.11	6.44	-1.05	2.37	-14.72
M6 15127	6.39	-0.58	10.00	-1.07	4.07	-1.70	1.47	-22.14
M7 15126	-0.17	-5.83	-0.22	-10.27	2.25	-1.91	17.39	0.62
M17 14587	1.11	0.02	2.68	0.03	-0.07	***min -5.87***	2.89	-1.95
M27 16895	4.90	-1.15	***max 8.44***	-0.51	17.91	-1.20	2.96	-20.43
Spoke 3								
M3 12794	0.03	-0.58	0.07	-4.72	2.98	-5.06	10.85	0.32
M2 12848	0.00	***min -2.61***	0.42	-0.40	10.83	0.06	***max 15.22***	0.27
M6 10995	0.18	-2.70	0.94	-0.46	9.30	-0.49	***max 12.35***	-0.37
M7 10994	2.23	0.03	0.05	-1.40	-0.12	-6.90	-0.13	-8.74
M17 10455	0.57	-1.02	1.51	-0.03	5.53	-0.04	2.72	-3.82
M27 12763	-0.16	***min -3.29***	0 33	-2.10	13.02	-0.14	***max 24.87***	1.18
Spoke 4								
M3 8662	***max 2.04***	0.02	1.48	-0.07	0.24	-9.76	0.18	***min -10.75***
M2 8716	0.04	-1.25	0.00	-2.48	1.85	-0.17	4.45	-0.09
M6 6863	0.16	-2.94	0.20	***min -3.28***	9.87	-0.53	7.96	-0.55
M7 6862	2.33	0.05	***max 2.63***	0.05	-0.16	-7.41	-0.12	-5.78
M17 6323	0 45	***min -1.07***	0.00	-0.63	5.51	-0.10	3.94	0.06
M27 8631	-0.05	-1.55	-0 12	-2.86	2.78	-0 81	2.50	-0.49
Spoke 5								
M3 4530	0.32	-2.58	0.38	-0.68	11.14	-1.63	1.64	-0.36
M2 4584	1.66	-0.12	0.31	-0.26	2.12	-1.38	2.64	-0.19
M6 2731	5.43	-0.45	1.01	-0.45	4.84	-2.29	12.33	-0.43
M7 2730	-0.06	-5.81	0.00	-1.60	2.76	-2.91	-0.23	***min -8.74***
M17 2191	1.33	0.00	1.51	0.00	-0.05	-5.88	2.72	-3.82
M27 4499	2.40	-0 41	0.54	-0.89	9.02	-2.05	9.48	0 48

	VARIATION 1 Parabolic loading Outer 60 degrees Inner 120 degrees				VARIATION 2 Parabolic loading Outer 60 degrees Inner 120 degrees			
	Load case A		*Load case F*		*Load case A*		*Load case F*	
gauge and node no.	σ max.	σ min	σ max.	σ min	σ max.	σ min	σ max.	σ min
Spoke 1								
M3 398	1.23	-2.31	10.35	-2.05	2.03	-1.93	11.35	-1.86
M2 452	-0.02	***min -25.37***	0.04	-14.90	-0.10	***min -26.44***	-0.02	-15.67
M6 624	1.22	***min -38.57***	1.01	-24.04	1.80	***min -37.80***	1.34	-23.81
M7 623	***max 28.39***	0.70	18.21	0.41	***max 30.21***	0.70	19.21	0.41
M17 36	***max 9.15***	0.74	3.70	-1.10	***max 6.69***	-0.01	2.32	-2.23
M27 367	-2.12	***min -43.04***	-0.51	-23.78	-2.30	***min -45.91***	-0.63	-25.17
Spoke 2								
M3 16926	***max 22.98***	-0.63	14.13	-4.86	***max 24.43***	-0.62	14.99	-4 89
M2 16980	6.70	-1.10	2.48	-15.29	6.99	-1.18	2.65	-15.48
M6 15127	4.14	-1.76	1.50	-22.96	3.96	-2.19	1.64	-23.12
M7 15126	2.40	-1.86	17.92	0.67	2.66	-1.74	18.27	0.68
M17 14587	-0.09	-4.77	3.89	-1.10	-0.08	***min -5.92***	2.80	-2.31
M27 16895	18.87	-1.26	3.11	-21.23	19.80	-1.39	3.33	-21.49
Spoke 3								
M3 12794	3.35	-4.24	12.02	0.42	3.36	-4.54	12.02	0.41
M2 12848	12.00	0.00	***max 16.67***	0.21	12.65	0.03	***max 17.24***	0.23
M6 10995	9.80	-0.50	***max 12.73***	-0.35	10.39	-0.52	***max 13.05***	-0.36
M7 10994	-0.13	-7.11	-0.13	-2.84	-0.13	-7.70	-0.13	***min -9.54***
M17 10455	5.09	-0.50	2.89	-3.51	5.33	-0.54	3.16	-3.38
M27 12763	13.95	-0.25	***max 28.19***	1.25	15.09	-0.20	***max 29.69***	1.33
Spoke 4								
M3 8662	-0.21	-12.26	0.37	***min -12.34***	-0.21	-13.33	0.42	***min -13.52***
M2 8716	2.24	-0.15	4.63	-0.12	2.75	-0.15	5.07	-0.13
M6 6863	10.66	-0.54	8.62	-0.57	11.16	-0.59	9.21	-0.60
M7 6862	-0.18	-7.69	-0.13	-6.12	-0.19	-8.37	-0.14	-6.62
M17 6323	5.05	-0.65	2.95	-0.05	6.01	-0.21	4.08	0.66
M27 8631	3.06	-0.55	2.47	-0.62	3.40	-0.40	2 78	-0.73
Spoke 5								
M3 4530	15.07	-0.66	1.62	-0.41	15.81	-0.71	1.77	-0 40
M2 4584	1.90	-1.12	3.19	-0.15	1.90	-1.37	3.30	-0.15
M6 2731	4.79	-2.44	12.69	-0.44	4.40	-3.00	12.10	-0.49
M7 2730	2.81	-2.92	-0.26	***min -8.86***	3.05	-2.90	-0.27	-8.99
M17 2191	-0.05	***min -5.61***	2.88	-3.73	-0.06	-5.61	3.00	-3.80
M27 4499	9.36	-2.00	10.61	0.55	9.69	-2.19	11.04	0 57

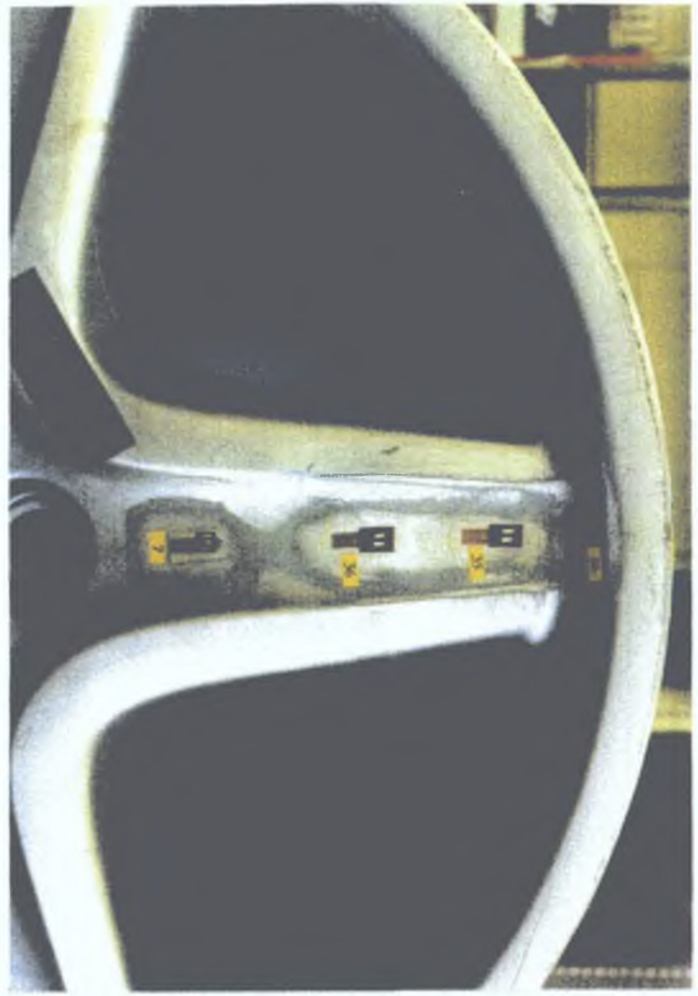


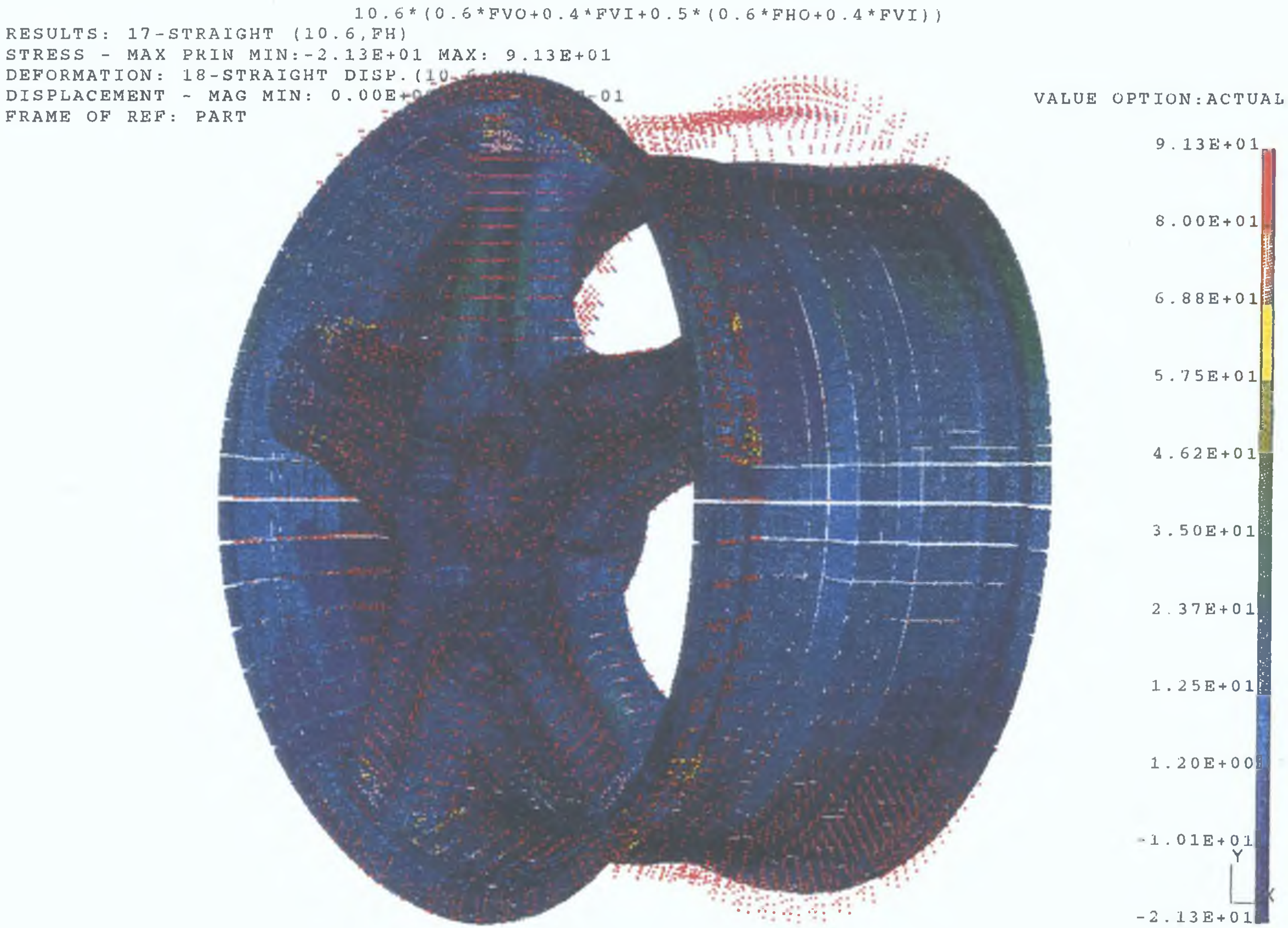
LBF 7396

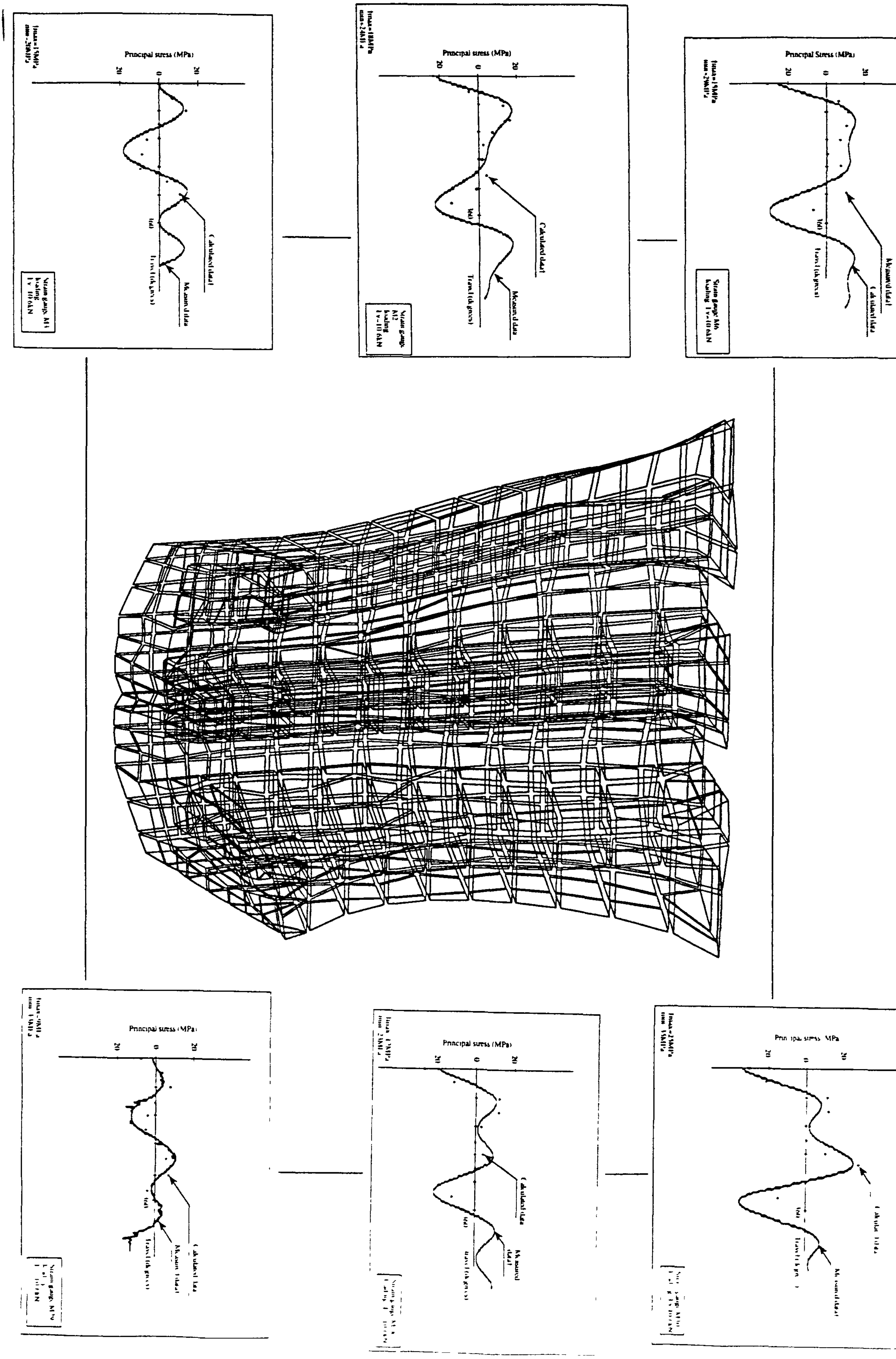
LOCATION OF STRAIN GAUGES

LOCATION OF STRAIN GAUGES

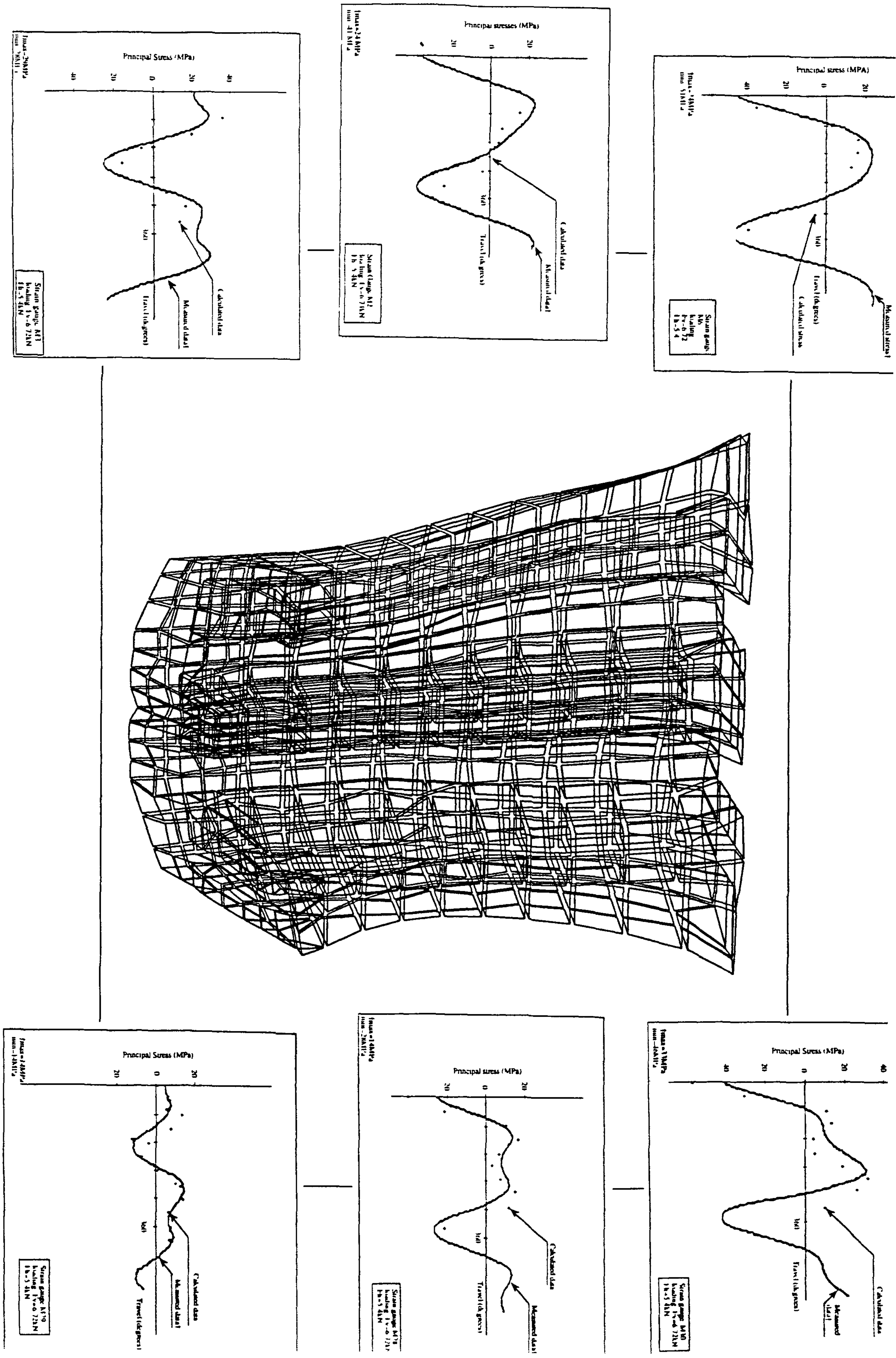
LOCATION OF STRAIN GAUGES



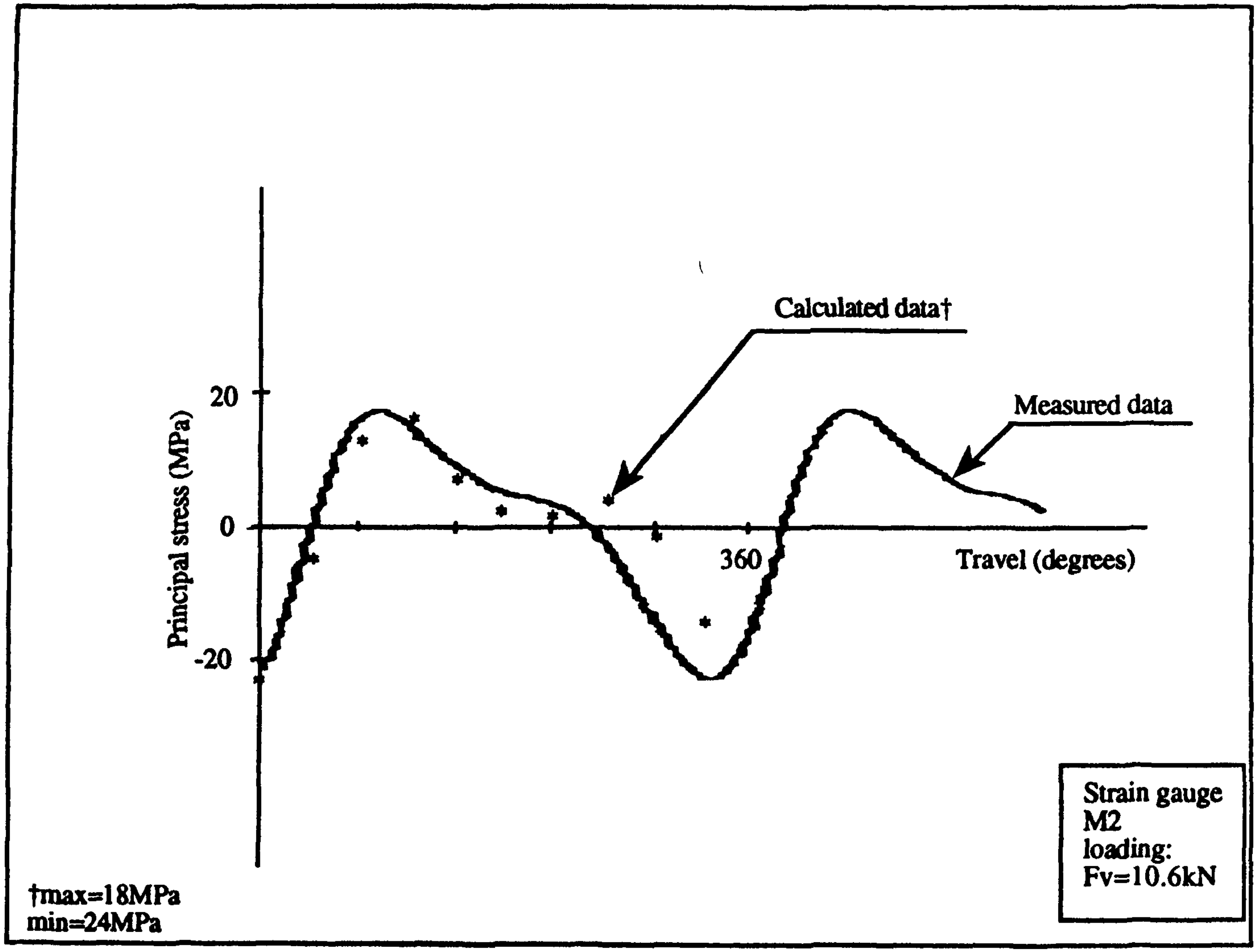

10.6*(0.6*FVO+0.4*FVI+0.5*(0.6*FHO+0.4*FVI))
RESULTS: 17-STRAIGHT (10.6,FH)
STRESS - MAX PRIN MIN:-2.13E+01 MAX: 9.13E+01
DEFORMATION: 18-STRAIGHT DISP.(10
DISPLACEMENT - MAG MIN: 0.00E+0
FRAME OF REF: PART
VALUE OPTION:ACTUAL
9.13E+01
8.00E+01
6.88E+01
5.75E+01
4.62E+01
3.50E+01
2.37E+01
1.25E+01
1.20E+00
-1.01E+01
-2.13E+01
Y
X

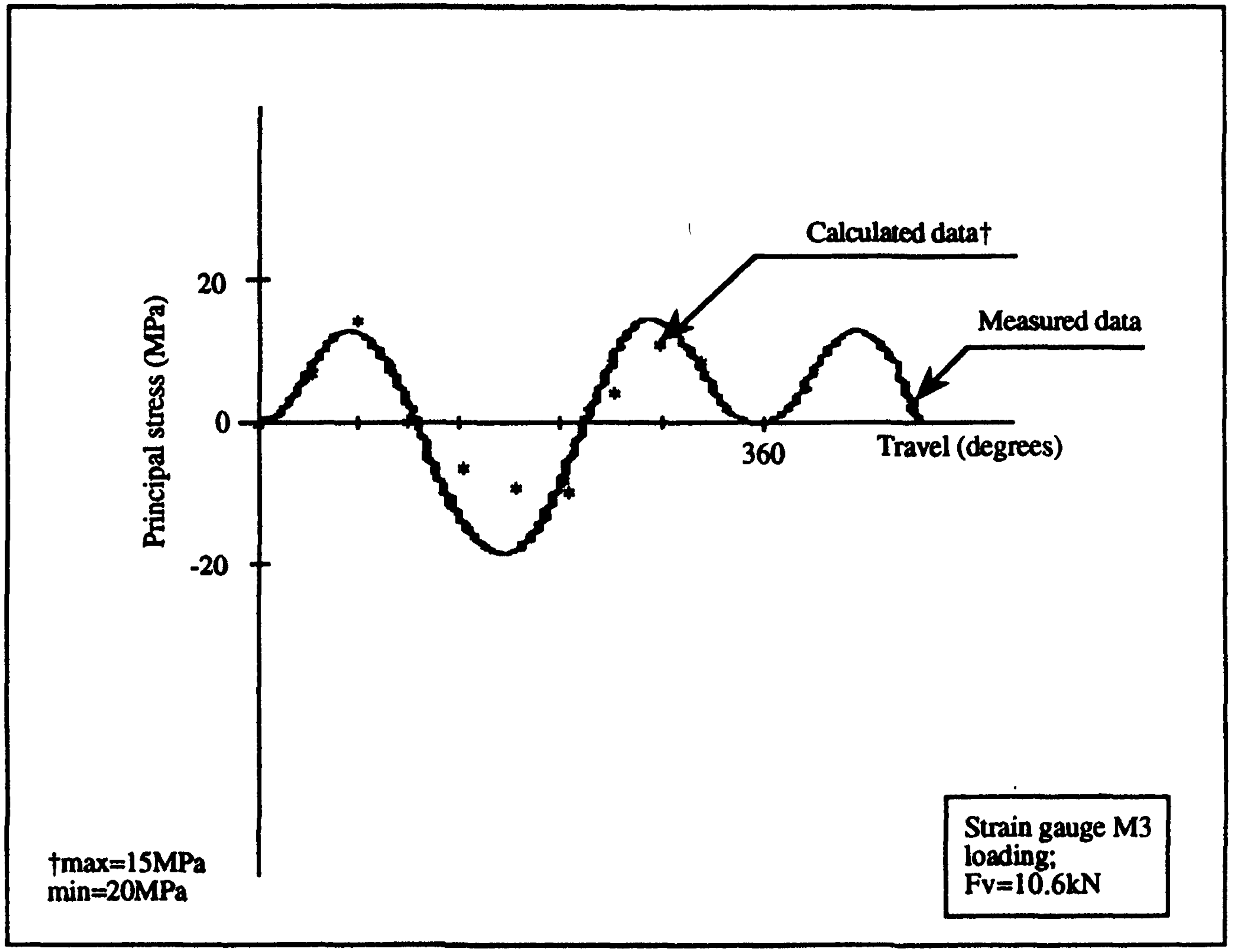


Principal stress (MPa)
Principal Stress (MPa)
Calculated data
Measured data
Travel (degrees)

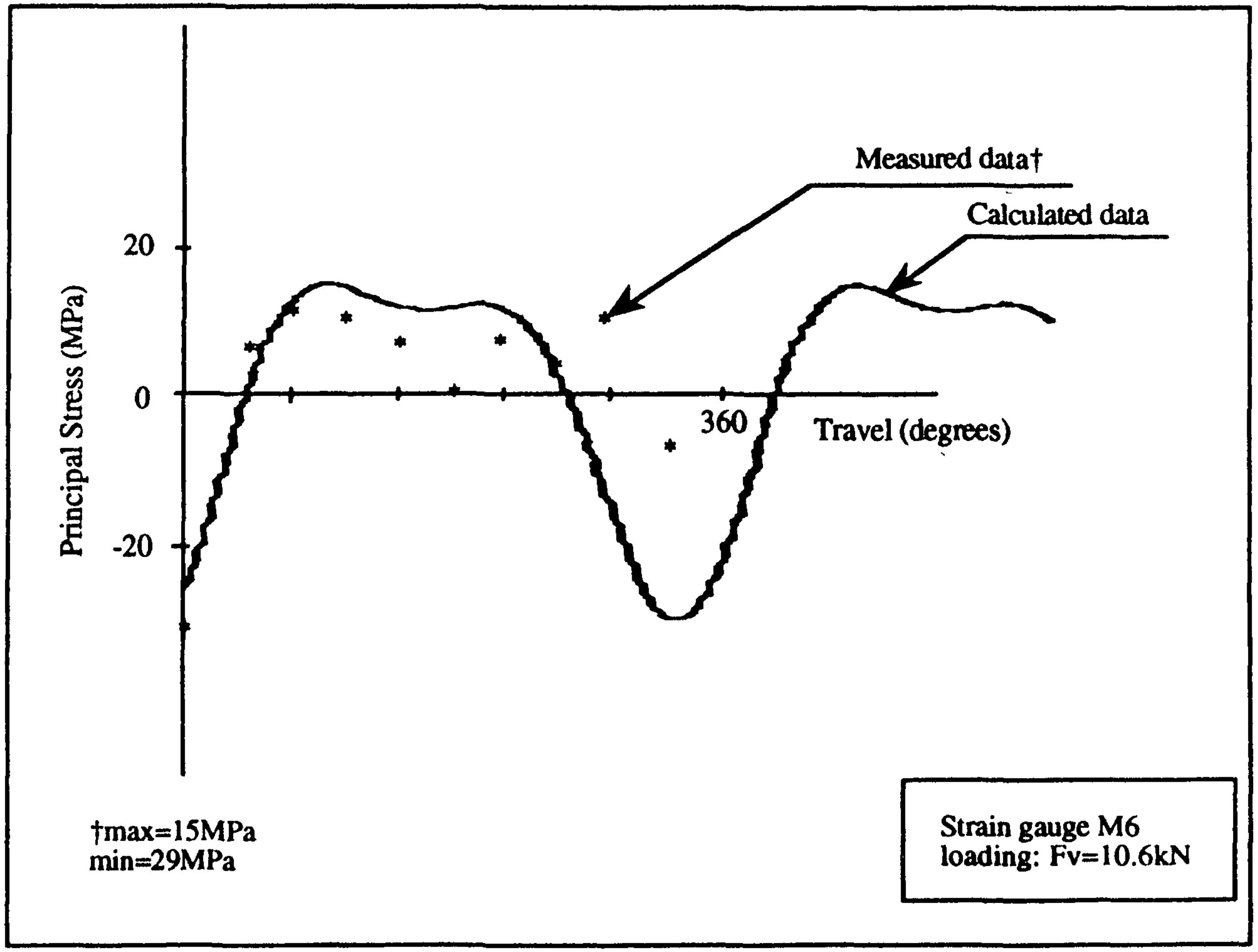


Principal Stress (MPa)
Principal stresses (MPa)
Principal stress (MPA)
Principal Stress (MPa)
Principal stress (MPa)
Principal Stress (MPa)
Calculated data
Measured data
Travel (degrees)

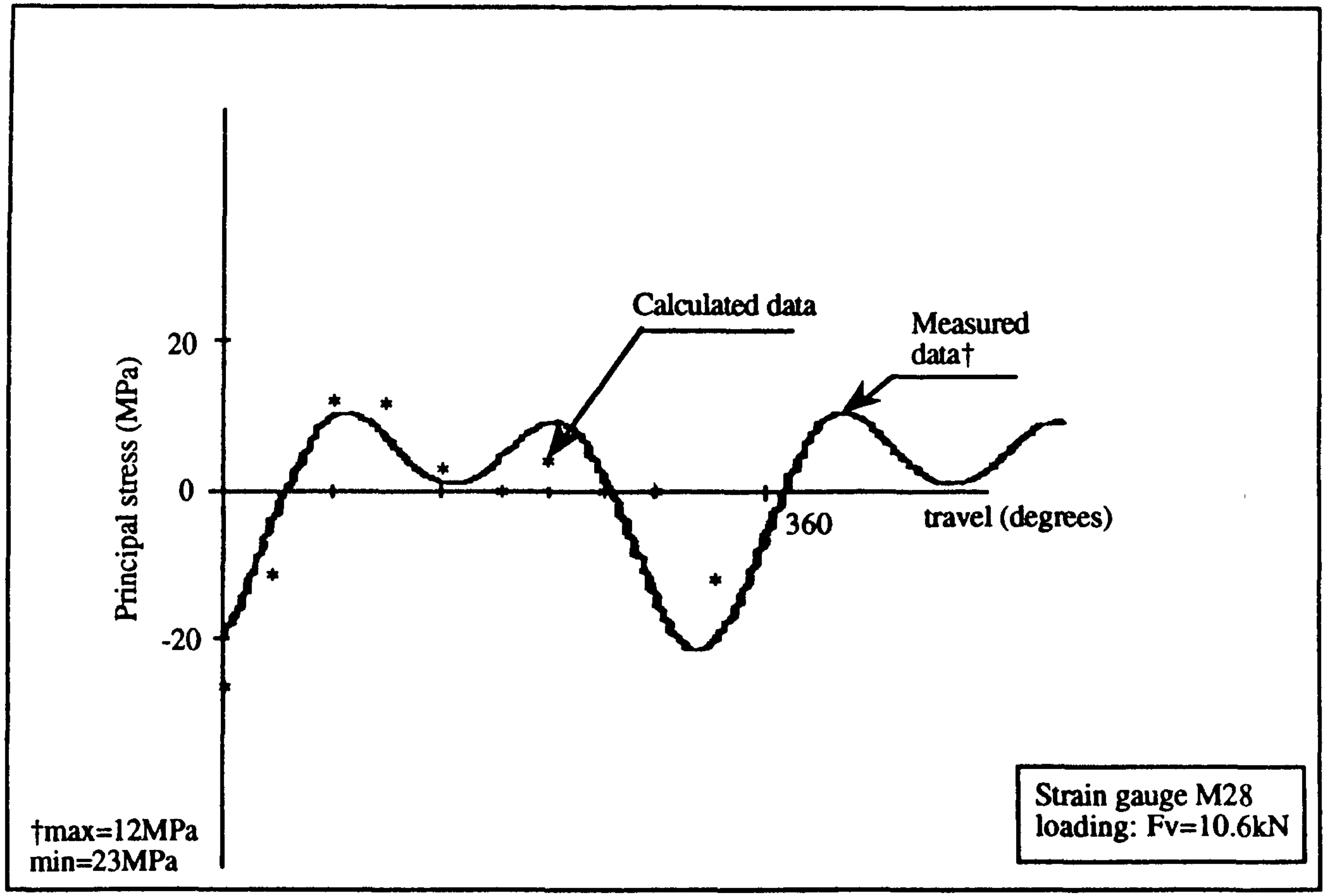

Calculated data†
Measured data
20
0
-20
Principal stress (MPa)
360
Travel (degrees)
Strain gauge
M2
loading:
Fv=10.6kN
†max=18MPa
min=24MPa

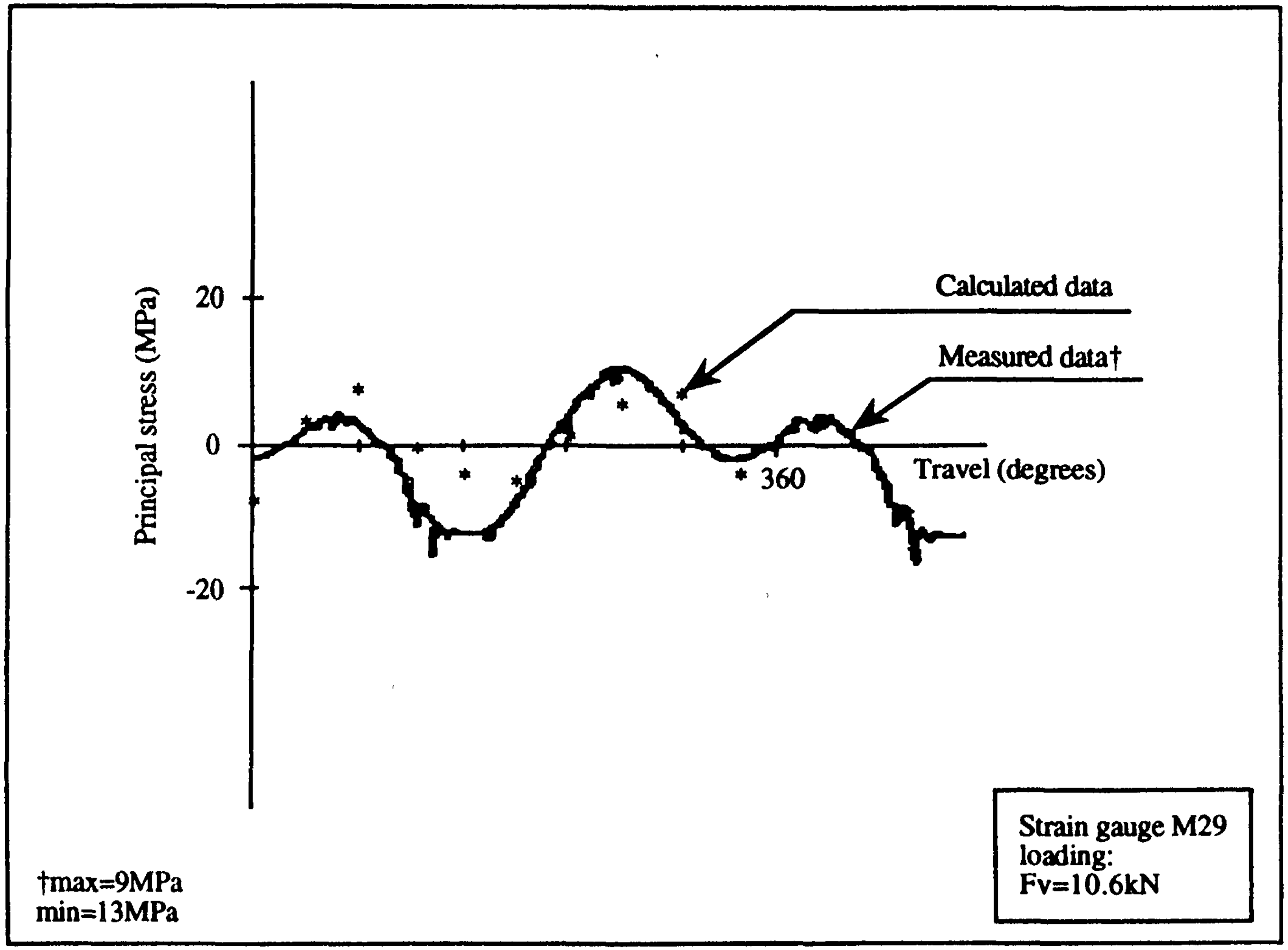


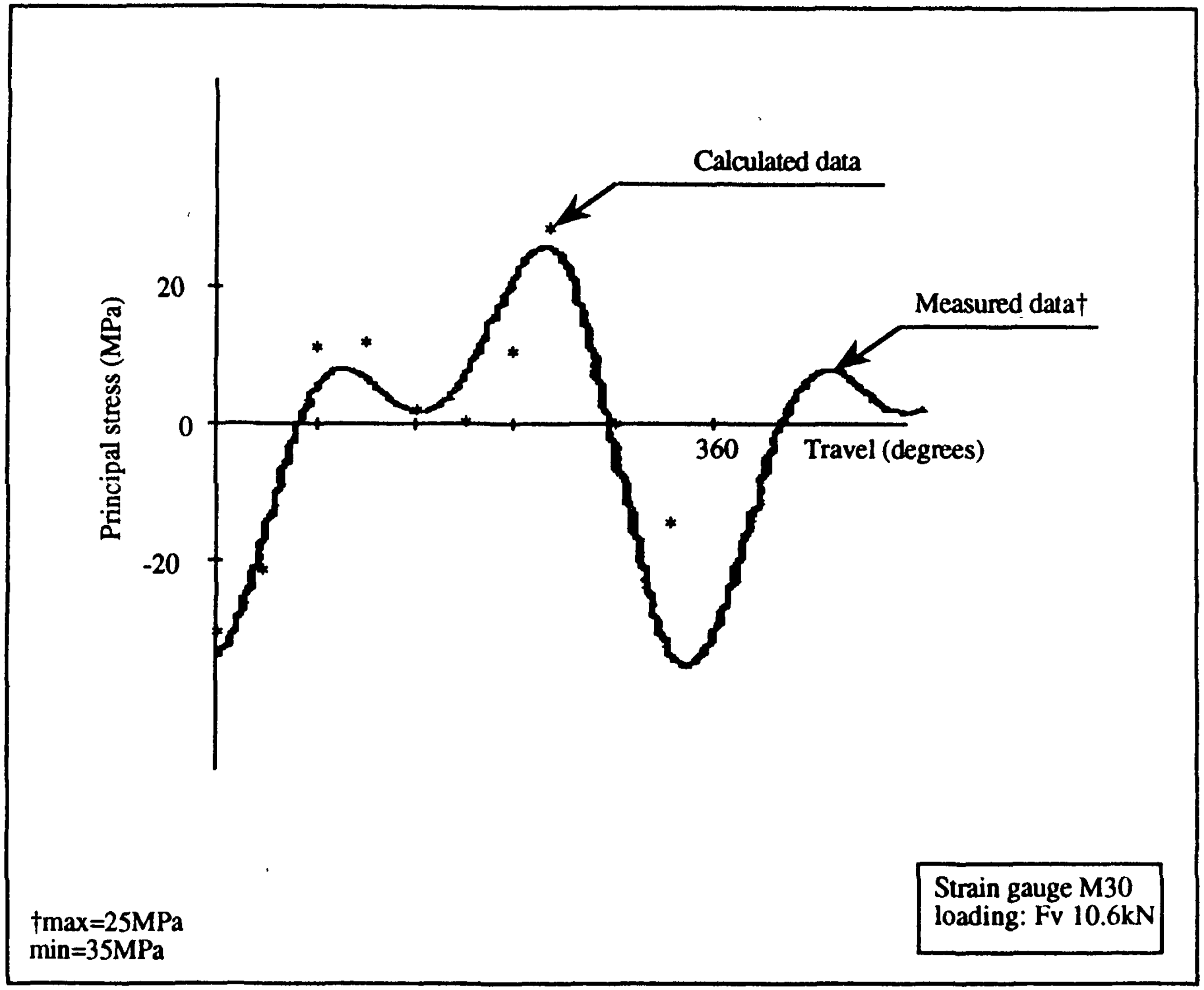
Calculated data†
Measured data
20
0
-20
360
Travel (degrees)
Principal stress (MPa)
Strain gauge M3 loading; Fv=10.6kN
†max=15MPa
min=20MPa

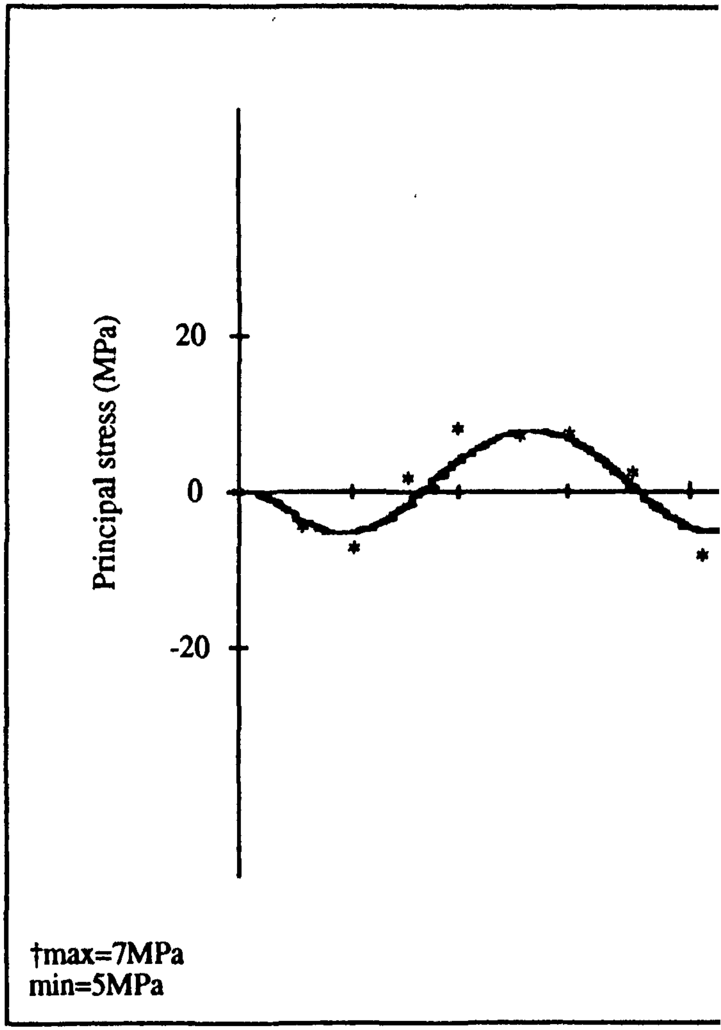


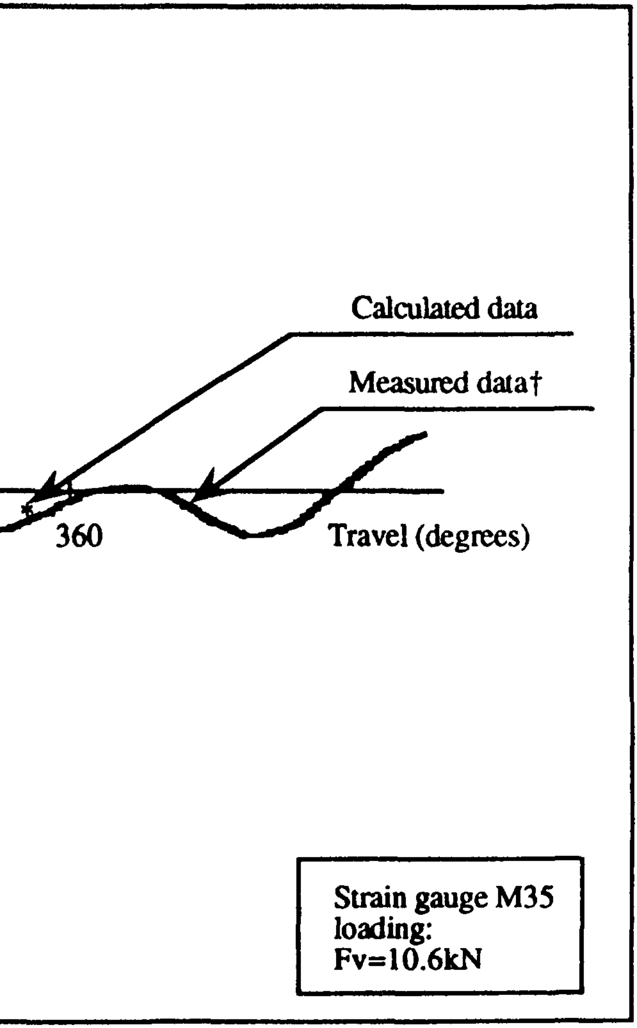
Measured data†
Calculated data
20
0
-20
360
Travel (degrees)
Principal Stress (MPa)
†max=15MPa
min=29MPa
Strain gauge M6
loading: Fv=10.6kN

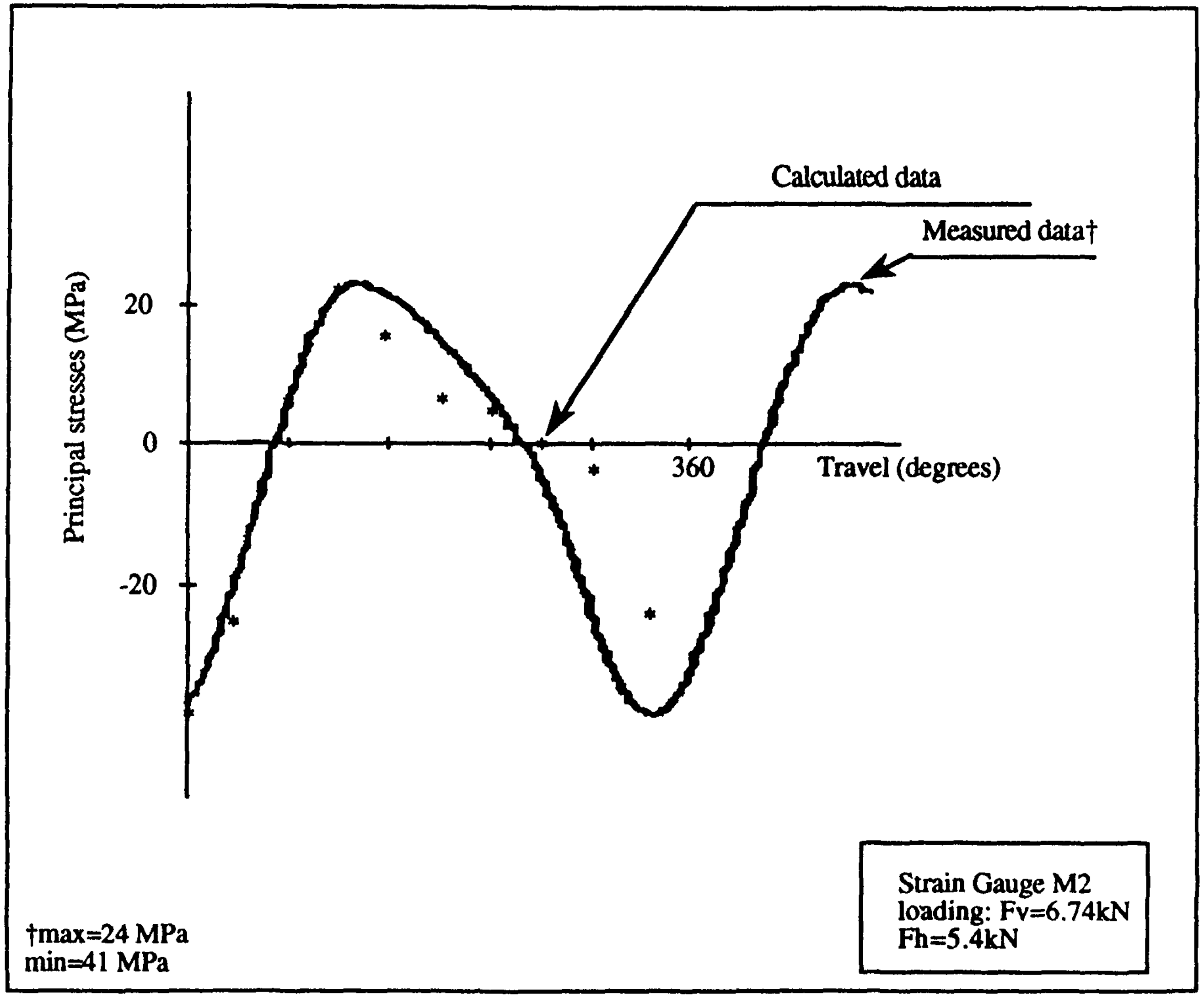

Calculated data
Measured data†
20
0
-20
Principal stress (MPa)
360
travel (degrees)
Strain gauge M28
loading: Fv=10.6kN
†max=12MPa
min=23MPa


Principal stress (MPa)
20
0
-20
360
Travel (degrees)
Calculated data
Measured data†
Strain gauge M29
loading:
Fv=10.6kN
†max=9MPa
min=13MPa

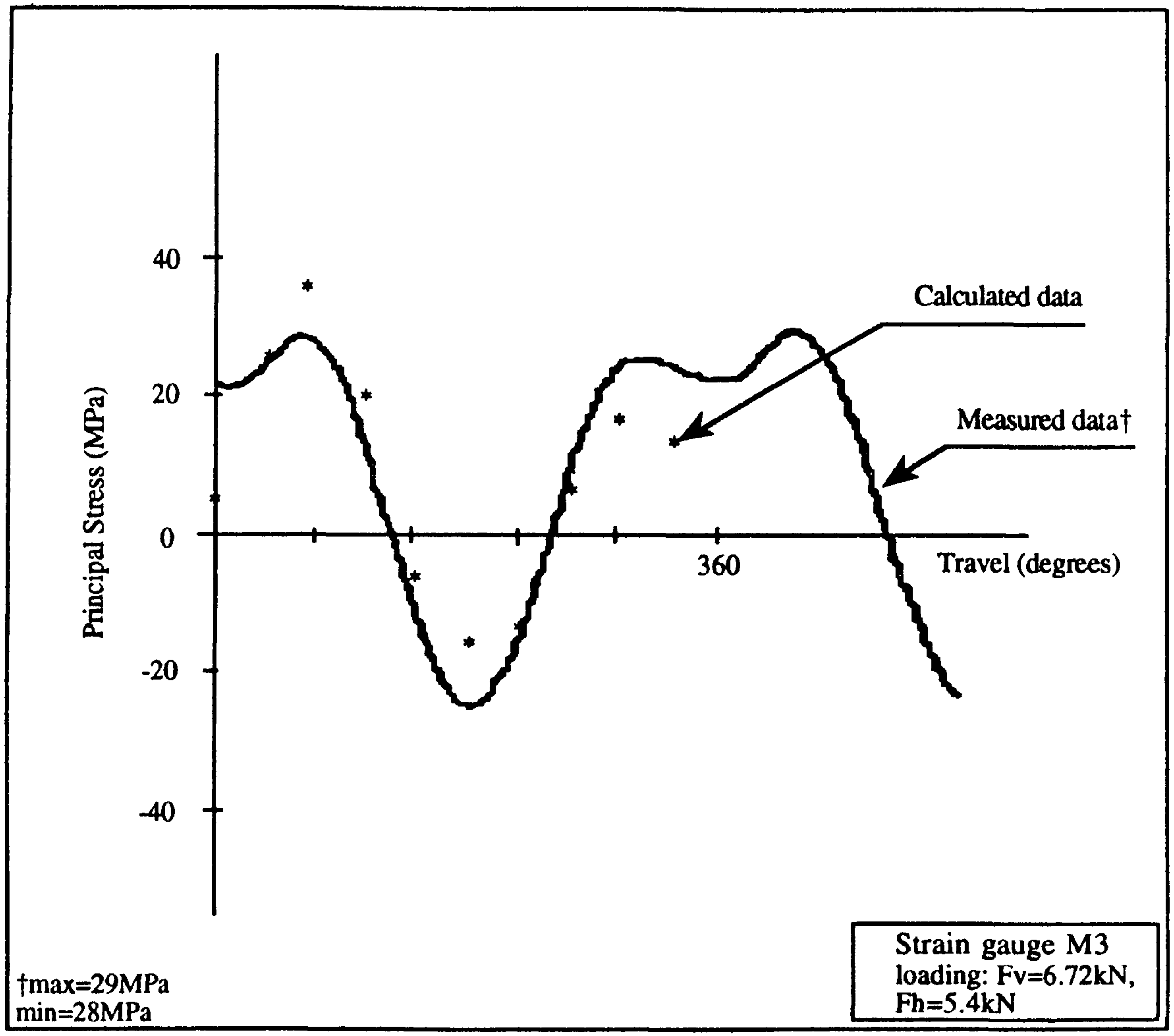

Calculated data
Measured data†
Principal stress (MPa)
20
0
-20
360
Travel (degrees)
†max=25MPa
min=35MPa
Strain gauge M30
loading: Fv 10.6kN

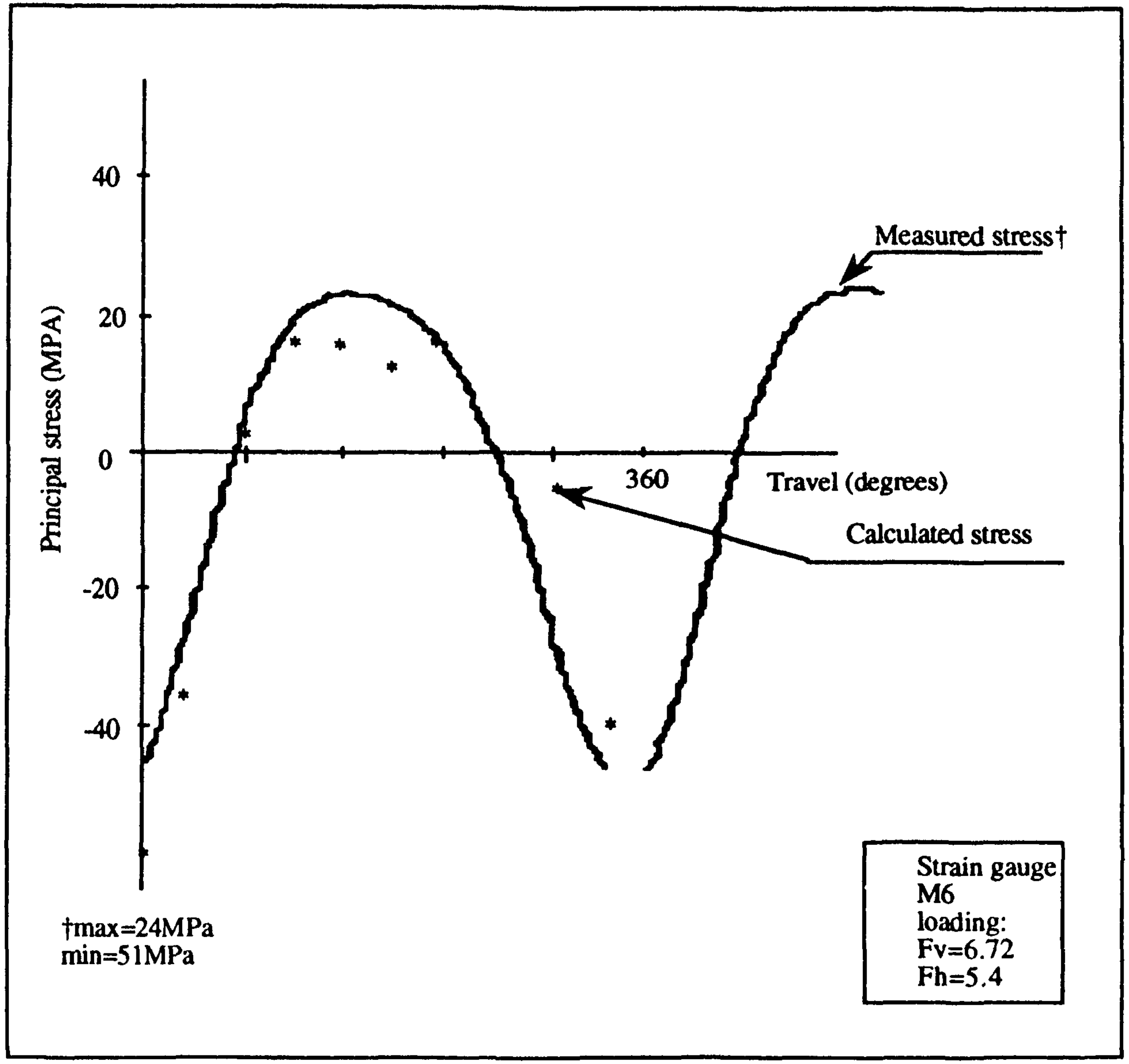

Principal stress (MPa)
20
0
-20
†max=7MPa
min=5MPa

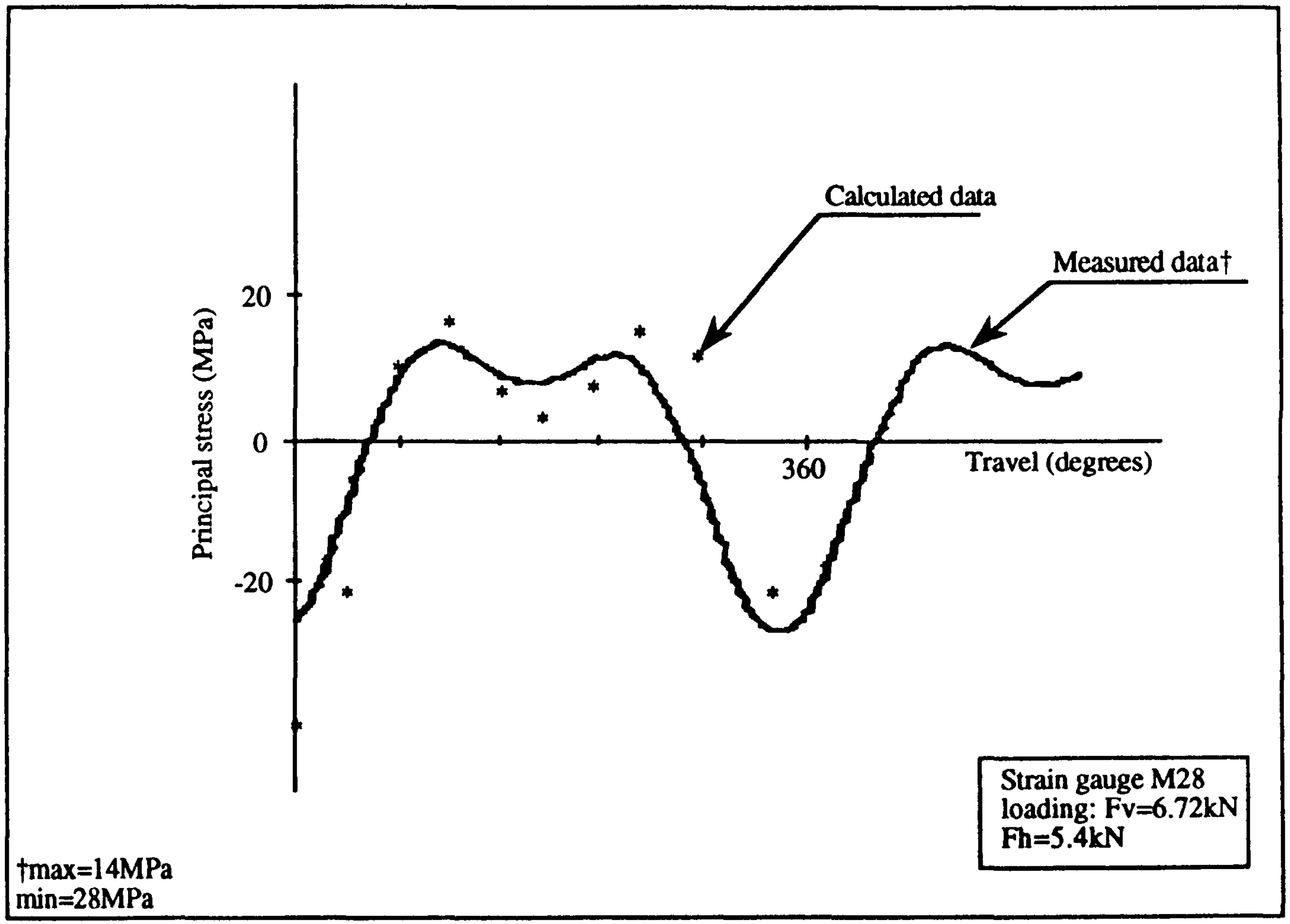

Calculated data
Measured data†
360
Travel (degrees)
Strain gauge M35
loading:
Fv=10.6kN

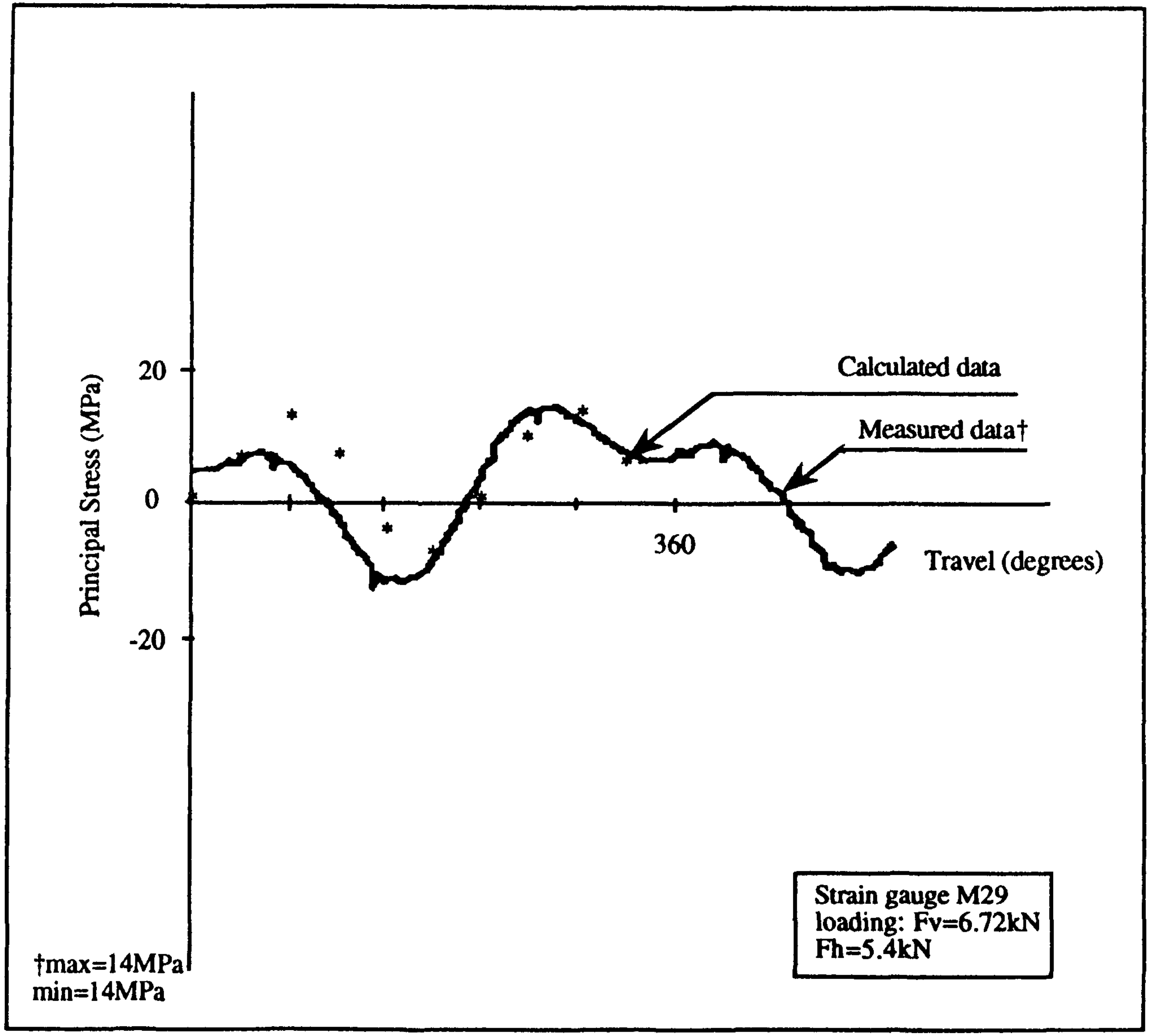


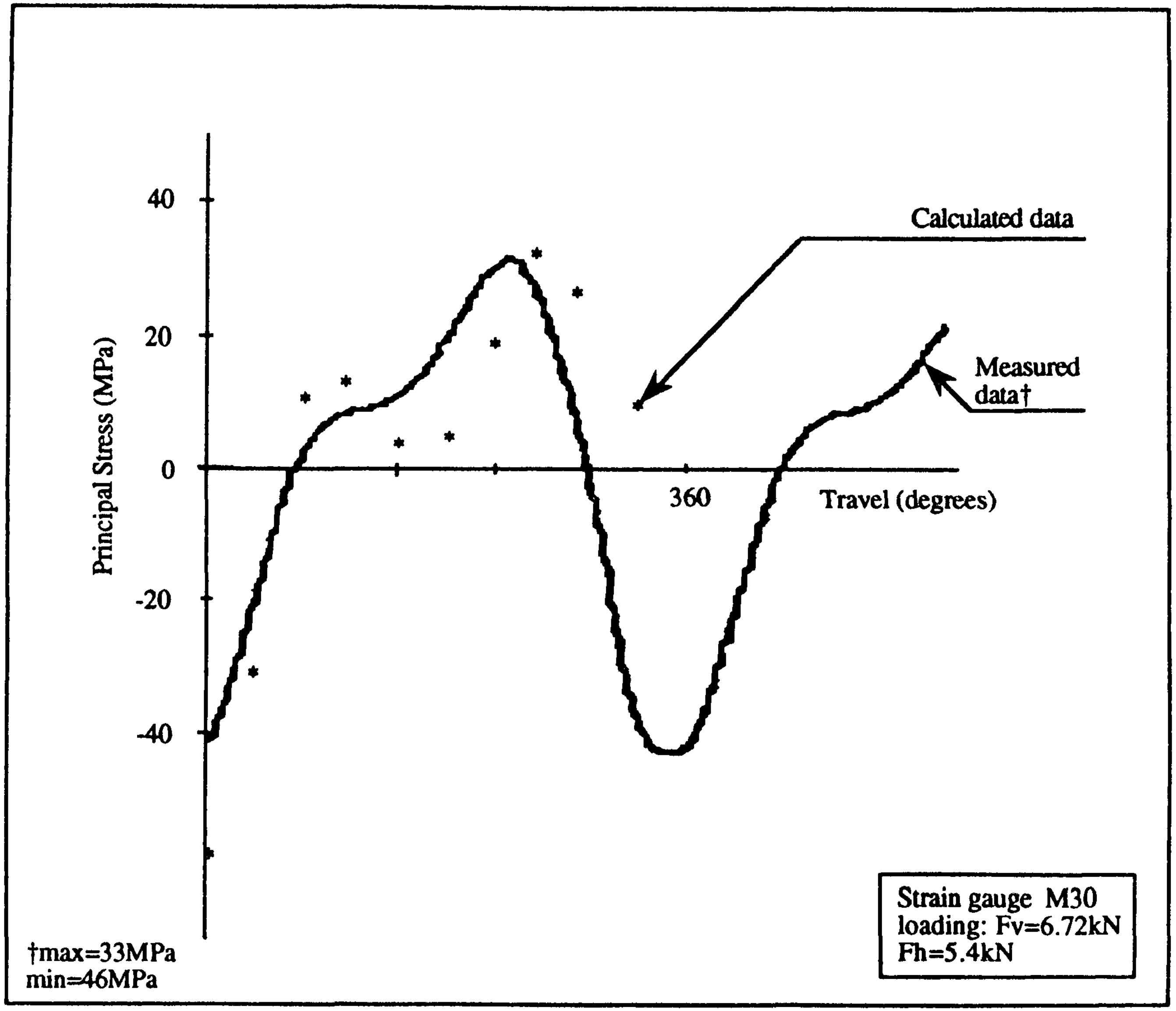
Calculated data
Measured data†
Principal stresses (MPa)
20
0
-20
360
Travel (degrees)
Strain Gauge M2
loading: Fv=6.74kN
Fh=5.4kN
†max=24 MPa
min=41 MPa


Principal Stress (MPa)
40
20
0
-20
-40
360
Travel (degrees)
Calculated data
Measured data†
†max=29MPa
min=28MPa
Strain gauge M3
loading: Fv=6.72kN,
Fh=5.4kN

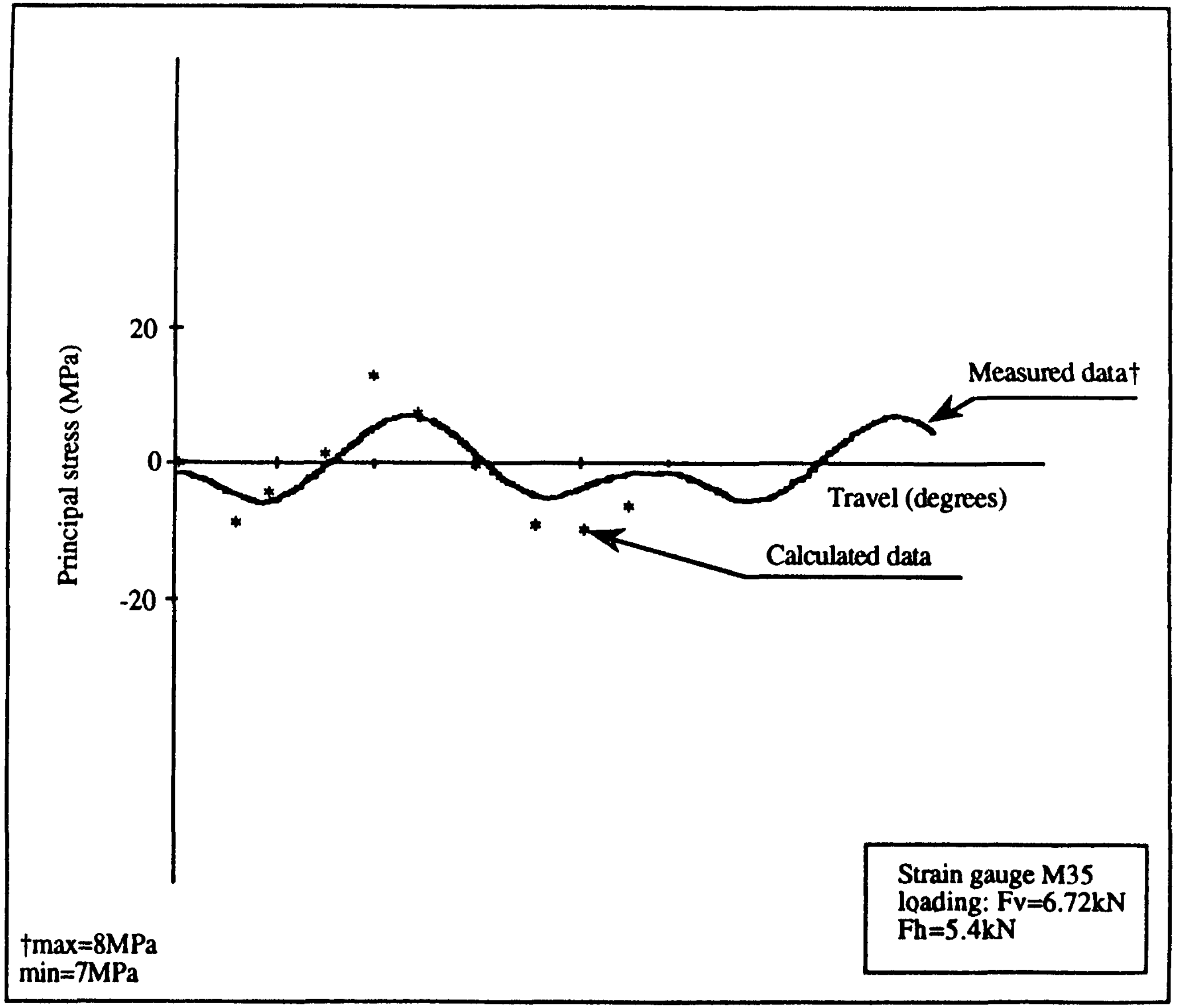

Principal stress (MPA)
40
20
0
-20
-40
360
Travel (degrees)
Measured stress†
Calculated stress
†max=24MPa
min=51MPa
Strain gauge
M6
loading:
Fv=6.72
Fh=5.4


Calculated data
Measured data†
Principal stress (MPa)
20
0
-20
360
Travel (degrees)
Strain gauge M28
loading: Fv=6.72kN
Fh=5.4kN
†max=14MPa
min=28MPa


Principal Stress (MPa)
20
0
-20
360
Travel (degrees)
Calculated data
Measured data†
Strain gauge M29
loading: Fv=6.72kN
Fh=5.4kN
†max=14MPa
min=14MPa



40
20
0
-20
-40
Principal Stress (MPa)
360
Travel (degrees)
Calculated data
Measured
data†
Strain gauge M30
loading: Fv=6.72kN
Fh=5.4kN
†max=33MPa
min=46MPa



20
0
-20
Principal stress (MPa)
Measured data†
Travel (degrees)
Calculated data
Strain gauge M35
loading: Fv=6.72kN
Fh=5.4kN
†max=8MPa
min=7MPa

Appendix F

Turbo Pascal programs

```
Program   mary1;
(              {Comments   30/Aug/95}
Uses crt;
Var
  x,    y1,y2,y3,y4,y5,y6,y7,y8,y9,y10,y11,y12,y13,y14   : Real;
  I, RoadType, DrivingType                                                    ♦
  C1, Fstatic, FVstraight                                                : Rea♦
  FVCornering, FLStraight, FLCornering        : Real;
  out1        : Char;

    Function   F_For_Curve_5b(Fstatic  : Real) : Real;
    {Outputs the Fverticalmid for cornering load case}

    Begin
    F_For_Curve_5b  := (1.8416 - 0.01016 * Fstatic) * Fstatic;
    End;


    Function   F_For_Curve_5c(Fstatic  : Real) : Real;
    {Outputs the Fverticalmax for cornering load case}

    Begin
    F_For_Curve_5c  := (1.7389 - 0.01178 * Fstatic) * Fstatic;
    End;


    Function   F_For_Curve_5a(Fstatic  : Real) : Real;
    {Outputs the Fverticalmin for cornering load case}

    Begin
    F_For_Curve_5a  := (2.0601- 0.01227 * Fstatic) * Fstatic;
    End;


    Function   F_For_Curve_6b(Fstatic  : Real) : Real;
    {Outputs the Flateralmid for cornering load case}

    Begin
    F_For_Curve_6b  := (1.4936 - 0.01848 * Fstatic) * Fstatic;
    End;


    Function   F_For_Curve_6c(Fstatic  : Real) : Real;
    {Outputs the Flateralmax for cornering load case}

    Begin
    F_For_Curve_6c  := (1.0936 - 0.01848 * Fstatic) * Fstatic;
    End;


    Function   F_For_Curve_6a(Fstatic  : Real) : Real;
    {Outputs the Flateralmin for cornering load case}

    Begin
    F_For_Curve_6a  := (1.8936 - 0.01848 * Fstatic) * Fstatic;
    End;


    Function   F_For_Curve_1b(Fstatic  : Real) : Real;
    {Outputs the Flateralmid for straight load case}
```

```
Begin
F_For_Curve_1b  := (1.2465 - 0.02206 * Fstatic + 0.0001809
             * (Fstatic * Fstatic)) * Fstatic ;
End;


Function    F_For_Curve_1a(Fstatic  : Real) : Real;
{Outputs the Flateralmax for straight load case}

Begin
F_For_Curve_1a  := (1.4759 - 0.02362 * Fstatic + 0.0002018 *
             (Fstatic * Fstatic)) * Fstatic ;
End;


Function    F_For_Curve_1c(Fstatic  : Real) : Real;
{Outputs the Flateralmin for straight load case}

Begin
F_For_Curve_1c  := (1.0603 - 0.02333 * Fstatic + 0.0002036 *
             (Fstatic * Fstatic )) * Fstatic ;
End;


Function    F_For_Curve_2b(Fstatic  : Real) : Real;
{Outputs the Flateralmid for straight load case}

Begin
F_For_Curve_2b  := (0.7796 - 0.01815 * Fstatic + 0.0001691 *
             (Fstatic * Fstatic )) * Fstatic ;
End;


Function    F_For_Curve_2a(Fstatic  : Real) : Real;
{Outputs the Flateralmax for straight load case}

Begin
F_For_Curve_2a  := (0.8503 - 0.01714 * Fstatic + 0.0001482 *
             (Fstatic  * Fstatic)) * Fstatic ;
End;

Function    F_For_Curve_2c(Fstatic  : Real) : Real;
{Outputs the Flateralmin for straight load case}

Begin
F_For_Curve_2c  := (0.6765 - 0.01715 * Fstatic + 0.0001663 *
             (Fstatic  * Fstatic)) * Fstatic ;
End;


Function    F_For_Curve_3(Fstatic  : Real) : Real;
{Outputs the Flateralmid for straight load case}

Begin
F_For_Curve_3  := (0.467 - 0.0146 * Fstatic + 0.0002131 *
             (Fstatic  * Fstatic)) * Fstatic ;
End;


Function    F_For_Curve_4(Fstatic  : Real) : Real;
{Outputs the Flateralmid for straight load case}

Begin
```

```
F_For_Curve_4  := (0.296 - 0.01304 * Fstatic + 0.0002367 *
             (Fstatic  * Fstatic)) * Fstatic ;
End;


Function    F_Vertical_Straight(Fstatic,C1 : Real; RoadType  : Integer)
                                                              : Real;
{Outputs the Fvertical for straight load case from equation
      where Nvs=load factor, K= roughness factor and C1=tyre stiffness}
{RoadType 1 -> Highway / 2-> Secondary/ 3->PotHole / 4-> Off road}
Var
    K, Nvs   : real;
Begin
Case RoadType of
 1 : K := 1.3;    {HighWay}
 2 : K := 2.0;    {Secondary}
 3 : K := 2.6;    {PotHole}
 4 : K := 3.5;    {Off Road}
End;
Nvs := 1 + K*(C1/Fstatic);
F_Vertical_straight := Nvs * Fstatic;
End;

  Function     F_Lateral_Straight
                      (Fstatic : Real;RoadType, Drivingtype :Integer)
                                                : Real;
{Make decision between curve 1a, 1b, 1c,

Var
                             Dummy : Real;
Begin
    Case Roadtype of
    1 : Dummy := F_For_Curve_4(Fstatic);
    2 : Dummy := F_For_Curve_3(Fstatic);
    3 : Begin
                Case DrivingType of
            1 : Dummy := F_for_Curve_2a(Fstatic);
          2 : Dummy := F_for_Curve_2b(Fstatic);
                                                  3 : Dummy := F_for_Curve
         End;
        End;
  4 : Begin
                Case DrivingType of
            1 : Dummy := F_for_Curve_1a(Fstatic);
          2 : Dummy := F_for_Curve_1b(Fstatic);
                                                  3 : Dummy := F_for_Curve
         End;
        End;
  End;
F_Lateral_Straight  := Dummy;
End;




Function     F_Vertical_Cornering  (Fstatic : Real; Drivingtype :Integer)
                                                       : Real;
{Make decision between curve c1.1a, c1.1b, c1.1c}
Var Dummy : Real;
Begin
    Case Drivingtype of
    1 : Dummy := F_For_Curve_5a(Fstatic);
    2 : Dummy := F_For_Curve_5b(Fstatic);
    3 : Dummy := F_For_Curve_5c(Fstatic);
  End;
```

```
    F_Vertical_Cornering  := Dummy;
    End;


    Function     F_Lateral_Cornering  (Fstatic : Real; Drivingtype :Integer)
                                                          : Real;
    {Make decision between curve 6a, 6b, 6c}
    Var Dummy : Real;
    Begin
        Case Drivingtype of
        1 : Dummy := F_For_Curve_6a(Fstatic);
        2 : Dummy := F_For_Curve_6b(Fstatic);
        3 : Dummy := F_For_Curve_6c(Fstatic);
      End;
    F_Lateral_Cornering  := Dummy;
    End;


Begin
    {Test the validity of the FVstraight equation as given in the notes,
                as well as FLstraigth, FVcornering and Flcornering as given in
                 various curve equations}
ClrScr;
WriteLn ('This is a test program which will output the values of Fvs, Fvc, Fls')♦
WriteLn  ('and Flc for a given Fstatic and other driving parameters');
WriteLn;
WriteLn ('The program repeats to enable you to test function validity until the'♦
WriteLn ('user deciedes to quit by typing Q at the request');
WriteLn;
WriteLn ('Hit Return to continue');
ReadLn;
Repeat
Begin
                ClrScr;
                Repeat
    Begin
                        WriteLn ('Input the static Force');
        WriteLn ('Note vehicle weigths    0 ->  10 kN Passanger Car');
        WriteLn ('                        10 -> 24 kN Light Truck');
        WriteLn ('                        24 -> 50 kN Heavy Truck');
        ReadLn (Fstatic);
        if (Fstatic < 0) or (Fstatic > 50) Then
        WriteLn ('Non valid option. Try again');
    End;
    Until Not ( (Fstatic < 0) or (Fstatic > 50) );
    WriteLn;

    WriteLn ('Now give me the Tyre stiffness');
    ReadLn (C1);
    WriteLn;

    Repeat
    WriteLn ('Input the road type Note 1-> Highway');
    WriteLn ('                         2-> Secondary');
    WriteLn ('                         3-> PotHole');
    WriteLn ('                         4-> Off Road');
    ReadLn (RoadType);
    if (RoadType < 1) or (RoadType > 4) Then
        WriteLn ('Non valid option. Try again');
    Until Not ( (RoadType < 1) or (RoadType > 4) );
    WriteLn;

    Repeat
    WriteLn ('Input the driving type Note 1-> Aggressivly');
    WriteLn ('                            2-> Average');
    WriteLn ('                            3-> Slowly');
```

```
    WriteLn;
    ReadLn (DrivingType);
    if (DrivingType < 1) or (DrivingType > 3) Then
        WriteLn ('Non valid option. Try again');
    Until Not ( (DrivingType < 1) or (DrivingType > 3) );
    WriteLn;

    FVStraight :=       F_Vertical_Straight(Fstatic, C1, RoadType);
    WriteLn ('Vertical Force for straight driving = ',FVStraight:5:3,' kN');

    FLStraight :=       F_Lateral_Straight(Fstatic, RoadType, Drivingtype);
    WriteLn ('Lateral Force for straight driving = ',FLStraight:5:3,' kN');

    FVCornering := F_Vertical_Cornering  (Fstatic, Drivingtype);
    WriteLn ('Vertical Force for cornering = ',FVCornering:5:3,' kN');

    FLCornering := F_Lateral_Cornering  (Fstatic, Drivingtype);
    WriteLn ('Lateral Force for cornering = ',FLCornering:5:3,' kN');

    WriteLn;
    WriteLn ('Type Q to quit or any other key to continue test on');
            WriteLn ('new set of parameters');
    Repeat Until Keypressed;
    Out1 := ReadKey;
End;
Until (Out1 = 'q') or (Out1 = 'Q');

End.
```

```
Program Spectrum_Test_4;

Uses   Crt, Dos, Graph, PlotNwl;

Var

{Used for graphics procedures}
   PlotNumber                              : Integer;
   XLabel, YLabel                          : Labels;
   Out1                                    : Char;
   Xdata                                   : PlotData;
   YdataPlot1                              : PlotData;
   YdataPlot2                              : PlotData;
   YdataPlot3                              : PlotData;
   LogXdata1, LogXdata2                    : PlotData;
   DesignSpectrum                          : PlotData;
   Xdata2                                  : PlotData;
   SN_Curve                                : PlotData;
   HValue, Ho                              : Real;

{General Use variables}
   Index1, Index2                          : Integer;

{Design Spectrum Variables}

   PlotSlope, PlotConstant, dN             : Real;

   Sstatmax, Sstatmin, Sstrmax, Sstrmin                         : Real;
   Scornmax, Scornmin, SA_static, SA_str, SA_corn               : Real;
   SM_static, SM_str, SM_corn, SR_static, SR_str, SR_corn       : Real;
   rdyn, Lr, Ntot, Nb_s, Nb_c, Ne_c                             : Real;
   Damage, LifeTime                                             : Real;
   Inputname                                                    : String;

{Material SN variables}

   k, kdash, Sendur, Nendur, S1, N1                             : Real;
   Material_Type, WheelArea, Manuf_process                       :Integer;

 Function  Log (Input  : real) : real;
 {Gets the log to the base 10 of the input}
 Begin
   Log := Ln(input)/Ln(10.0);
 End;

 Function  ALog (Input  : Real) : real;
 {Gets the Alog to the base 10 of the input}
 Begin

    ALog := Exp(Ln(10)*Input);
 End;

 Procedure      Stress_Amplitude (Sstatmax, Sstatmin, Sstrmax, Sstrmin,
        Scornmax, Scornmin :real; Var  SA_static, SA_str, SA_corn :Real);
   Begin
     SA_static := (Sstatmax-Sstatmin)/2.0;
     SA_str := (Sstrmax-Sstrmin)/2.0;
     SA_corn := (Scornmax-Scornmin)/2.0;
   End;

 Procedure      Mean_Stress (Sstatmax, Sstatmin, Sstrmax, Sstrmin,
        Scornmax, Scornmin :real; Var  SM_static, SM_str, SM_corn :Real);
   Begin
     SM_static := (Sstatmax+Sstatmin)/2.0;
     SM_str := (Sstrmax+Sstrmin)/2.0;
```

```
    SM_corn := (Scornmax+Scornmin)/2.0;
  End;

Procedure       Stress_Ratio (Sstatmax, Sstatmin, Sstrmax, Sstrmin,
        Scornmax, Scornmin :real; Var  SR_static, SR_str, SR_corn :Real);
  Begin
    SR_static := Sstatmin/Sstatmax ;
    SR_str := Sstrmin/Sstrmax;
    SR_corn := Scornmin/ Scornmax;
  End;

Procedure       Design_Spectrum_Parameters (rdyn, Lr : Real; Var Ntot, Nb_s,
                             Nb_c, Ne_c :Real);

  Begin
    Ntot := Lr * ((1000)/(2*rdyn*PI)) ;
    Nb_s := 0.48 * Ntot;
    Nb_c := 0.02 * Ntot;
    Ne_c := 0.00000048 * Ntot;
 End;

Procedure        SN_Parameters (Var k, kdash, Sendur :Real; Material_Type,
                  WheelArea, Manuf_process :Integer);

{This procedure will calculate the Material SN curve for a given material}
 {and manufacturing process. The user must input the relevant area of the}
 {wheel, the type of material (ie steel or aluminium), and the type of}
 { manufacturing process used.}


 Begin
 WriteLn; WriteLn;
 WriteLn ('You have selected the following input parameters');
  Case  Material_Type Of
  1 : Begin
      WriteLn ('Material type Steel');
      {Material is Steel}
      Case  WheelArea  of
            1 : Begin
                {Region A Rim Flange}
                  WriteLn ('Wheel Area - Rim Flange Region');
                  k    := 5;
                  kdash := (2.0 * k) - 1.0;
                  Sendur := 130.0e6;
                End;
            2 : Begin
                {Region B Welding}
                  WriteLn ('Wheel Area - Welding Region');
                  k := 4 ;
                  kdash := (2.0 * k) - 2.0;
                  Sendur := 90.0e6;
                End;
            3 : Begin
                {Region C Ventilation Hole}
                  WriteLn ('Wheel Area - Ventilation hole Region');
                  k := 6 ;
                  kdash := (2.0 * k) - 1.0;
                  Sendur:= 160.0e6;
                End;
            4 : Begin
                {Region D Bolt Hole}
                  WriteLn ('Wheel Area - Bolt Hole Region');
                  k := 4.5;
                  kdash := (2.0 * k) - 2.0;
                  Sendur := 90.0e6;
                End;
```

```
     End;  {End Wheel Area Case}
     End;  {End steel case 1}
 2 : Begin
     {Material is Aluminium}
     WriteLn ('Material type Aluminium');
     Case Manuf_process of
          1 : Begin
              {Cast, non-heat treated aluminium}
                WriteLn ('manufacturing process - cast, non-heat treated');
                k := 4.5;
                kdash:= (2.0 * k) - 2.0;
                Sendur:= 40.0e6;
              End;

          2:  Begin
              {Cast, heat treated aluminium}
                WriteLn ('manufacturing process - cast, heat treated ');
                k := 4.5;
                kdash:= (2.0 * k) - 2.0;
                Sendur:= 60.0e6;
              End;

         3:  Begin
             {Forged Aluminium}
               WriteLn ('manufacturing process - forging');
               k := 4.5;
               kdash:= (2.0 * k) - 2.0;
               Sendur:= 80.0e6;
             End;
     End;  {End Manufac Process Case}
     End;  {End Aluminium Case}
  End; {End material case}
End;  {End Proc}


Procedure    InitialSN_Curve;
{This procedure is used to initate the SN curve data}
Begin
   Repeat
   ClrScr;
   WriteLn ('Is the wheel manufactured from : (enter 1 or 2)      ' );
   WriteLn ('                          1-> STEEL');
   WriteLn ('                          2-> ALUMINIUM');
   ReadLn  ( Material_Type);
   if (Material_Type < 1) or (Material_Type > 2) Then
   WriteLn ('Non valid option. Try again');
   Until Not ( (Material_Type < 1) or (Material_Type > 2) );
   WriteLn;

   {Wheel Area}
   Repeat
   WriteLn ('What area of the wheel are we studying :  (enter 1,2,3 or 4)
   WriteLn ('                          1-> Rim Flange');
   WriteLn ('                          2-> Welding');
   WriteLn ('                          3-> Ventilation Hole');
   WriteLn ('                          4-> Bolt Hole');
   ReadLn  (WheelArea);
   if (Wheelarea < 1) or (Wheelarea > 4) Then
   WriteLn ('Non valid option. Try again');
   Until Not ( (Wheelarea < 1) or (Wheelarea > 4) );
   WriteLn;

   {Manufacturing Process}
   If (Material_Type = 2) Then
   Begin
   Repeat
```

```
   WriteLn ('For Aluminium only: ');
   WriteLn ('What manufacturing processes are used : (enter 1,2 or 3)        ' ♦
   WriteLn ('                           1-> Cast, non-heat treated aluminium');
   WriteLn ('                           2-> Cast, heat treated aluminium');
   WriteLn ('                           3-> Forged Aluminium');
   ReadLn  (Manuf_process);
   if (Manuf_process < 1) or (Manuf_process > 3) Then
   WriteLn ('Non valid option. Try again');
   Until Not ( (Manuf_process < 1) or (Manuf_process > 3) );
   WriteLn;
   End;
   WriteLn;
   {Calculate SN curve Parameters Based on Above}
   SN_Parameters( k, kdash, Sendur, Material_Type,
           WheelArea, Manuf_process);
   {Write Results to the screen to be view if required}
   WriteLn ('k = ',  k:5:3);
   WriteLn ('kdash = ', kdash:5:3);
   WriteLn ('Sendur = ', Sendur:5:3, ' Pa');
   Nendur  :=2e+6;
   WriteLn ('Nendur = ', Nendur:5:3,' cycles');
   WriteLn ('Press return to continue and Wait');
   ReadLn;
End;  {End of the procedure}

Function     N_ForInput_S  (SInput, k, kdash, Sendur, Nendur  : real):real;
{This Function gives you the N value for an input S value on the SN curve}
Var
   Slope, xvalue :  real;
Begin
     If SInput > Sendur Then
     {Sort out what region of the curve your in}
     Slope := k
     Else
     Slope := kDash;
     If SInput = 0 then
     {Send out -6 to tell you that the N value = infinity}
       xvalue := -6
     Else
       xvalue := Nendur*ALog( Slope*(log(Sendur/SInput)) );
    N_ForInput_S := xvalue;
End;

Procedure   DamageCalc;
{Procedure to perform the damage calculation}
Var
   Index1   : Integer;
   x1, x2   : real;
   Dummy    : Real;
   Little_N, Big_N   : Real;
Begin
   Damage   := 0.0;
   Lifetime := Ntot;
   For Index1 := 0 to 500 Do
   Begin
        SN_Curve[Index1] := DesignSpectrum[Index1];
        Dummy    :=  N_ForInput_S  (SN_Curve[Index1], k, kdash,
                                    Sendur, Nendur);
        If Dummy = -6 Then
          {Singularity of the SN Curve i.e. N -> infinity}
          Xdata2[index1] := Xdata2[Index1-1]
        Else
          Xdata2[Index1] := Dummy;
        X1   := Xdata[Index1];
        X2   := Xdata2[Index1];
        Big_N := (x2-x1);
```

```
          If (Index1 > 0) and (Index1 < 500) Then
          Begin
               Little_N := (Xdata[Index1+1] - Xdata[Index1-1]);
               Damage := Damage + Little_N/Big_N;
          End;
          If (Index1 > 0) and (Index1 < 500) Then
          Begin
          Lifetime := Lr*(0.5/Damage);
          End;
     End;
  End;


Begin
{Initation of Input Data for use in calculating the design spectrum}
{*****}

        ClrScr;
        WriteLn ('Enter the Maximum Static Stress value (Pa)');
        ReadLn  (Sstatmax);
        WriteLn ('Enter the Minimum Static Stress value (Pa)');
        ReadLn  (Sstatmin);
        WriteLn ('Enter the Maximum Straight Driving Stress value (Pa)');
        ReadLn  (Sstrmax);
        WriteLn ('Enter the Minimum Straight Driving Stress value (Pa)');
        ReadLn  (Sstrmin);
        WriteLn ('Enter the Maximum Cornering Stress value (Pa)');
        ReadLn  (Scornmax);
        WriteLn ('Enter the Minimum Cornering Stress value (Pa)');
        ReadLn  (Scornmin);

{test data to save us input new data each time valid procedure is above}


{Calculates the design spectum parameters on the y axis}
        Stress_Amplitude (Sstatmax, Sstatmin, Sstrmax, Sstrmin,
                          Scornmax, Scornmin, SA_static, SA_str, SA_corn);
        WriteLn ('The Stress Amplitude for the Static Case = ', SA_Static:5:4,'P♦
        WriteLn ('The Stress Amplitude for the Straight Driving Case = ', SA_Str♦
        WriteLn ('The Stress Amplitude for the Cornering Case = ', SA_Corn:5:4,'♦
        Mean_Stress (Sstatmax, Sstatmin, Sstrmax, Sstrmin,
                     Scornmax, Scornmin, SM_static, SM_str, SM_corn);
        WriteLn ('The Mean Stress for the Static Case = ', SM_Static:5:4,'Pa');
        WriteLn ('The Mean Stress for the Straight Driving Case = ', SM_Str:5:4,♦
        WriteLn ('The Mean Stress for the Cornering Case = ', SM_Corn:5:4,'Pa');
        Stress_Ratio (Sstatmax, Sstatmin, Sstrmax, Sstrmin,
                 Scornmax, Scornmin, SR_static, SR_str, SR_corn);
        WriteLn ('The Stress Ratio for the Static Case   = ', SR_Static:5:4);
        WriteLn ('The Stress Ratio for the Straight Driving Case = ', SR_Str:5:4♦
        WriteLn ('The Stress Ratio for the Cornering Case  = ', SR_Corn:5:4);
        WriteLn ('Press return to continue');
        ReadLn;



{Calculates the design spectum parameters on the x axis}
        WriteLn ('Enter the required Design Life value for wheel (km)');
        WriteLn ('                         eg: Lr for car       = 3*10^5 km');
        WriteLn ('                         eg: Lr for truck     = 5*10^5 km');
        ReadLn  (Lr);
        WriteLn ('Enter the Dynamic Tyre Radius (m)');
        ReadLn  (rdyn );

        Design_Spectrum_Parameters (rdyn, Lr, Ntot, Nb_s, Nb_c, Ne_c);

        WriteLn ('Ntot = ', Ntot:5:4);
```

```
        WriteLn ('Nb_s = ', Nb_s:5:4);
        WriteLn ('Nb_c = ', Nb_c:5:4);
        WriteLn ('Ne_c = ', Ne_c:5:4);
        WriteLn ('Press return to continue and Wait');
        ReadLn;

{Interval spacing for xdata of design spectrum}
         Dn := Log(NTot)/500.0;


{Sort out Static design curve}
     For Index1 := 0 to 500 Do
     Begin
      {Calculate the xdata}
      XData [Index1]  := ALog(Index1*dN);
      {Y for region 1 for Static Curve}
      If Xdata [Index1] <= Nb_s Then
        YdataPlot1 [Index1] := SA_static;
      {Y for region 2 for Static Curve}
      If Xdata [index1] > Nb_s Then
       Begin
           PlotSlope :=(-SA_static) /(Log(Ntot)-Log(Nb_s)) ;
           PlotConstant :=  (SA_static*Log(Ntot)) / (Log(Ntot)-Log(Nb_s));
           YdataPlot1 [Index1] := PlotSlope * Log(Xdata [Index1])
                                + PlotConstant;
      End;
     End;

{Sort out straight driving design curve}
     For Index1 := 0 to 500 Do
     Begin
      {Y for region 1 of Straight Driving Curve}
      If Xdata [Index1] <= Ne_c Then
        YdataPlot2 [Index1] := SA_str;
      {Y for region 2 of Straight Driving Curve}
      If (Xdata [index1] > Ne_c) and (Xdata [index1] <= Nb_s) Then
      Begin
           PlotSlope := (Sa_static-Sa_str)/(Log(Nb_s)-Log(Ne_c));
           PlotConstant := Sa_static-(  ( Log(Nb_s)*(Sa_static-Sa_str) ) /
                               (Log(Nb_s)-Log(Ne_c))    );
           YdataPlot2 [Index1] := PlotSlope * Log(Xdata [Index1])
                               + PlotConstant;
      End;
      {Y for region 3 of Straight Driving Curve}
      If (Xdata [index1] > Nb_s)  Then
      Begin
           YdataPlot2 [Index1]:= 0.0;
      End;
     End;

{Sort out cornering design curve}
     For Index1 := 0 to 500 Do
     Begin
      {Y for region 1 of Cornering Curve}
      If Xdata [Index1] <= Ne_c Then
         YdataPlot3 [Index1] := Sa_corn;
      {Y for region 2 of Cornering Curve}
      If (Xdata [index1] > Ne_c) and (Xdata [index1] <= Nb_c) Then
      Begin
         Hvalue := Xdata[index1]-Ne_c;
         Ho     := Nb_c-Ne_c;
         YdataPlot3 [Index1] := (Sa_corn-Sa_Static) *
                    sqrt((1.0/ln(Ho))*ln(Ho/HValue))
                    + Sa_Static;
      End;
```

```
        {Y for region 3}
        If (Xdata [index1] > Nb_c)  Then
        Begin
           YdataPlot3 [Index1]:= 0.0;
        End;
      End;

{Here the largest value of curves is selected to produce the Design Spectrum}
      For Index1 := 0 to 500 Do
      Begin
          If (YdataPlot1[Index1] > YdataPlot2[Index1]) and
              (YdataPlot1[Index1] > YdataPlot3[Index1]) Then
              DesignSpectrum [Index1] := YdataPlot1[Index1];
          If (YdataPlot2[Index1] > YdataPlot1[Index1]) and
              (YdataPlot2[Index1] > YdataPlot3[Index1]) Then
              DesignSpectrum [Index1] := YdataPlot2[Index1];
          If (YdataPlot3[Index1] > YdataPlot1[Index1]) and
              (YdataPlot3[Index1] > YdataPlot2[Index1]) Then
              DesignSpectrum [Index1] := YdataPlot3[Index1];
         If ( (YdataPlot3[Index1] = 0) and (YdataPlot2[Index1] = 0) )
              and (YdataPlot1[Index1] = 0) Then
              DesignSpectrum [Index1] := 0.0;
      End;


{Initation of Input Data for use in calculating the S/N Curve}
{*****}
{Here the user is asked to input the relevant area of the wheel as well}
      { as the material and manufacturing process used.}
      InitialSN_Curve;


{room for SN curve and damage calculation calcaulation}
{Perform damage calculation and generate sn plot based on this data}
{***}
      DamageCalc;



{Convertion of xdata for plot to log scale}
      For Index1 := 0 to 500 Do
      Begin
      {Note Log (a) = ln (a)/ ln (10)}
           LogXdata1 [Index1] := Log(Xdata[Index1]);
           LogXdata2 [Index1] := Log(Xdata2[Index1]);
      End;

{now plot SN and design spectrum}
      Init_Graphic;
      PlotNumber := 500;
      {plot this just to start with}
      Xlabel := 'Log of Cycles N';
      Ylabel := 'S ampl.';
      {This is the valid plot procedure}
      PlotDataSets(6, 7, PlotNumber, Xlabel, Ylabel,
                   LogXdata1, DesignSpectrum, LogXdata2, SN_Curve);


{** Menu options **}
      DrawMenu1(Damage, Lifetime);
      Repeat
      Repeat Until KeyPressed;
      Out1 := ReadKey;

{menu option1 save data}
      If (Out1 = 's') Or (Out1 = 'S') Then
```

```
     {Write design spectrum, S/N curves to a series of text files}
     Begin
       ReStoreCrtMode;
       Passout (Xdata, DesignSpectrum, PlotNumber, 'design spectrum');
       Passout (Xdata, DesignSpectrum, PlotNumber, 'S/N data');
       Passout (Xdata, DesignSpectrum, PlotNumber, 'straight driving');
       Passout (Xdata, DesignSpectrum, PlotNumber, 'cornering');
       SetGraphMode(CurrentGraphicsMode);
       {This is the valid plot procedure}
       PlotDataSets(6, 7, PlotNumber, Xlabel, Ylabel,
                 LogXdatal, DesignSpectrum, LogXdata2, SN_Curve);
       DrawMenul(Damage, Lifetime);
     End;

{menu option2 print data}
     If (Outl = 'p') Or (Outl = 'P') Then
       Hardcopy(False, 4);

{menu option3 change material type and go back to plot new data}
     If (Outl = 'c') Or (Outl = 'C') Then
     {Change S/N curves and perform new damage calc}
     Begin
       ReStoreCrtMode;
       {Initation ofnew input data for use in calculating the S/N Curve}
       InitialSN_Curve;
       {Perform damage calculation and generate sn plot based on this data}
       DamageCalc;
       SetGraphMode(CurrentGraphicsMode);
       {This is the valid plot procedure}
       PlotDataSets(6, 7, PlotNumber, Xlabel, Ylabel,
                 LogXdatal, DesignSpectrum, LogXdata2, SN_Curve);
       DrawMenul(Damage, Lifetime);
     End;


{menu option4 quit}
     Until (Outl = 'q') Or (Outl = 'Q');
     CloseGraph; {quit bgi graphics}
End.
```