AUTOMATED DESIGN ANALYSIS OF ANTI-ROLL BARS

A THESIS SUBMITTED TO
THE GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES
OF
THE MIDDLE EAST TECHNICAL UNIVERSITY

BY

KEMAL ÇALIŞKAN

IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF
MASTER OF SCIENCE
IN
THE DEPARTMENT OF MECHANICAL ENGINEERING

SEPTEMBER 2003

Approval of the Graduate School of Natural and Applied Sciences

Prof. Dr. Canan Özgen
Director

I certify that this thesis satisfies all the requirements as a thesis for the degree of Master of Science.

Prof. Dr. Kemal İder
Head of the Department

This is to certify that we have read this thesis and that in our opinion it is fully adequate, in scope and quality, as a thesis for the degree of Master of Science.

Asst. Prof. Dr. Serkan Dağ
Co-Supervisor

Prof. Dr. Y. Samim Ünlüsoy
Supervisor

Examining Committee Members

Prof. Dr. Demir Bayka

Prof. Dr. Y. Samim Ünlüsoy

Asst. Prof. Dr. Serkan Dağ

Prof. Dr. Metin Akkök

Prof. Dr. Mehmet Akgün (AEE)

ABSTRACT

AUTOMATED DESIGN ANALYSIS OF ANTI-ROLL BARS

Çalışkan, Kemal
M.S., Department of Mechanical Engineering
Supervisor: Prof. Dr. Y. Samim Ünlüsoy
Co-Supervisor: Asst. Prof. Dr. Serkan Dağ

September 2003, 123 Pages

Vehicle anti-roll bars are suspension components used for limiting body roll angle. They have a direct effect on the handling characteristics of the vehicle. Design changes of anti-roll bars are quite common at various steps of vehicle production, and a design analysis must be performed for each change. Finite Element Analysis (FEA) can be effectively used in design analysis of anti-roll bars. However, due to high number of repeated design analyses, the analysis time and cost problems associated with the use of general FEA package programs may create considerable disadvantages in using these package programs for performing anti-roll bar design analysis.

In this study, an automated design program is developed for performing design analysis of vehicle anti-roll bars. The program is composed of two parts, the user interface and the FEA macro. The FEA macro includes the codes for performing deformation, stress, fatigue, and modal analysis of anti-roll bars in ANSYS 7.0. The user interface, which is composed in Visual Basic 6.0, includes the forms for data input and result output procedures. By the developed software, the FEA of the anti-roll bars is simplified to simple data entry via user interface. The flow of the analysis is controlled by the program and the finite element analysis is performed by ANSYS at the background.

The developed software can perform design analysis for a wide range of anti-roll bars: The bar centerline can have any 3D shape, the cross section can be solid or hollow circular, the end connections can be of pin or spherical joint type, the bushings can be mounted at any position on the bar with a user defined bushing length.

The effects of anti-roll bar design parameters on final anti-roll bar properties are also evaluated by performing sample analyses with the automated design program developed in this study.

Keywords: Anti-Roll Bar, Design Automation, FEA, Fatigue Analysis

ÖZ

OTOMOBİL DENGE ÇUBUĞUNUN OTOMATİK TASARIM ANALİZİ

Çalışkan, Kemal
Yüksek Lisans, Makina Mühendisliği Bölümü
Danışman: Prof. Dr. Y. Samim Ünlüsoy
Eş Danışman: Y. Doç. Dr. Serkan Dağ

Eylül 2003, 123 Sayfa

Otomobil denge çubukları araçlarda yalpa açısını sınırlandırmak amacıyla kullanılan süspansiyon elemanlarıdır ve otomobilin doğrultu kontrol ve stabilizasyon özellikleri üzerinde etkilidirler. Denge çubuğu tasarımı otomobil üretiminin çeşitli safhalarında değişmekte ve her değişim için yeni bir tasarım analizi yapılması gerekmektedir. Denge çubuğu tasarım analizinde Sonlu Elemanlar Metodu etkili bir biçimde kulanılabilmektedir. Fakat, genel amaçlı Sonlu Elemanlar Analizi paket programlarının kullanımınında karşılaşılan analiz zamanı ve maliyetiyle ilgili problemler, tasarım analizlerinin sürekli tekrarlanması nedeniyle, bu programların denge çubuğu tasarım analizinde kullanılmasında dikkate değer dezavantajlar yaratabilmektedir.

Bu çalışmada, otomobil denge çubuklarının tasarım analizlerini gerçekleştirmek amacıyla otomatik bir tasarım programı geliştirilmiştir. Bu program, kullanıcı arayüzü ve analiz dosyası olmak üzere iki kısımdan oluşmaktadır. Analiz dosyası, denge çubukları için deformasyon, gerilme, yorulma ve titreşim analizlerini ANSYS 7.0 programı aracığıyla gerçekleştirmeyi sağlayan kodları içermektedir. Visual Basic

6.0 ortamında geliştirlmiş olan kullanıcı arayüzü ise, veri girme ve sonuç görüntüleme amaçlı formları içermektedir. Hazırlanan program sayesinde denge çubuklarının Sonlu Elemanlar Analizi, kullanıcı arayüzü vasıtasıyla veri girme işlemine dönüştürülmüştür. Analiz akışı program tarafından kontrol edilmekte ve Sonlu Elemenlar Analizi ANSYS tarafından arka planda gerçekleştirilmektedir.

Geliştirilen program, denge çubuğu özellikleri açısından geniş bir uygulanabilirliğe sahiptir. Çubuğun merkez çizgisi üç boyutlu herhangi bir geometriye sahip olabilir, çubuk kesit alanı içi dolu veya boş silindirik olabilir, uç nokta birleşimleri pim veya küresel bağlantılarla sağlanabilir ve yataklar çubuk üzerinde herhangi bir noktaya, istenilen genişliğikle monte edilebilir.

Bu çalışmada ayrıca, denge çubuğu tasarım parametrelerinin denge çubuğu özellikleri üzerindeki etkileri, hazırlanan yazılım vasıtasıyla gerçekleştirelen analizler yardımıyla değerlendirilmiştir.

Anahtar Kelimeler: Otomobil Denge Çubuğu, Tasarım Otomasyonu, Sonlu Elemanlar Analizi, Yorulma Analizi

To Neva

ACKNOWLEDGEMENTS

I would to like to express sincere appreciation to Prof. Dr. Y. Samim Ünlüsoy and Asst. Prof. Dr. Serkan Dağ for their guidance and patience throughout the study.

I owe special thanks to my precious Neva for being in my life and making everything easier for me.

I want to extend my thanks to my family for their moral support and having faith in me, and especially to my brother Mustafa Çalışkan for his helps during my study.

Evren Özatay is also gratefully acknowledged for being the best officemate even possible.

I also send my special thanks to Ali Özzeybek for his support in computer programming.

TABLE OF CONTENTS

LIST OF TABLES

TABLE

LIST OF FIGURES

FIGURE

CHAPTER 1

INTRODUCTION

1.1 Statement of the Problem

Engineering design is the process of devising a system, component, or process to meet desired needs. It is a decision making process (often iterative), in which the basic sciences, mathematics, and engineering sciences are applied to convert resources optimally to meet a stated objective. Among the fundamental elements of the design process are the establishment objectives and criteria, synthesis, analysis, construction, testing and evaluation.

The primary way that engineers utilize the forces and materials of nature for the benefit of mankind is through new and innovative designs. The first step into a design study is to recognize the need. The need, which is the purpose of the design study, is established via a general statement of the client's dissatisfaction with the current situation. Problem definition is the second step of design which should include all the specifications for the thing that is to be designed, the constraints on the design, design considerations (strength, fatigue life, cost etc.) and the criteria to be used for evaluating the design according to the design considerations. The third step is information gathering which requires collection of related information, including theoretical information and previous studies on the subject, from the available sources. Concept generation comes as the fourth step of design, which is the most creative part of the design process. Concept generation is followed with concept selection, in which the generated concepts are compared with respect to basic design criteria. This step may require some simple analysis. After the selection of the concept to be applied for the solution, the detailed design and analysis are performed as the sixth step of design. The seventh and the last step of the design is to present the final design to the clients. Some imperative objectives that must be met in presenting any design solution to the clients are credibility, explanation, thoroughness and clear answers.

The effect of available products on design is considerably important. In today's world, computers are the most important and valuable of these products for design engineers. The engineering environment has been revolutionized by the advent of computer technology. Computers not only enabled the engineers to perform the previously applied design methods easier and faster, with higher precision, but also changed the methods of design process. As an example, recent advantages in computer hardware technology coupled with increased availability of sophisticated, user friendly Computer Aided Engineering software, has lead to a significantly increase in the role of analysis in the product development process. Today, various Computer Aided Design and Analysis methods are being used, which improve the speed and quality of design [1].

It's clear that the computers and the package programs had lead to great advances in engineering design. But a new phenomenon, competitive market, is forcing the limits of these advances. The recent market conditions makes every single moment and every small amount of money spend on the production of a part very important. As implied by Shih et al. [2]; "Only products with high quality, low cost and short concept-to-customer time will continue to have a high market share." Therefore, synthesizing new design concepts with advanced computing technology is the key to competitive product design in order to respond to challenges.

The above discussion is also valid for the automotive industry, which is one of the most dynamic industries in all over the world. Automotive designers are required continually to reduce lead times to the market place by exploiting computational tools. In addition to the role of design, the engineers will also have to manage and integrate computer-aided design, computer-aided engineering, computer-aided manufacturing and product data management tools into corporate strategies and provide more efficient ways for companies to operate.

As mentioned previously, with the help of computers, there had been a huge improvement in applications of design steps. The Finite Element Method (FEM) is a good example for this improvement. The FEM is a numerical technique to obtain approximate solutions to a wide variety of engineering problems where the variables are related by means of algebraic, differential and integral equations. Although originally developed to study stresses in complex airframe structures, it has since been extended and applied to the broad field of continuum mechanics. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in industry. The number of equations is usually rather large for most real-world applications of the FEM, and requires the computational power of the digital computer. Thus, the FEM has little practical value if the digital computer were not available. Advances in and ready availability of computers and software has brought the FEM within reach of engineers. Today, FEM is widely used for detailed analysis step of the design process and it's well known that, use of Finite Element Analysis (FEA) in product development will significantly reduce cycle time and improve product quality. The merit of using FEA in product design is evidenced by the mandate of its practice in the auto industry's QS 9000 quality standard [2]. However, the competitive market conditions give rise to a new problem: Most of the companies use Finite Element Method, thus take the advantage of using it. Therefore, a company has to use it more efficiently than others do in order to gain a competitive advantage.

As made clear above, the usefulness of FEA in engineering product design is no longer an issue. Rather, the availability and cost for its extensive usage for product development is of concern. This is due to some characteristics of FEA. First of all conducting FEA requires highly trained FEA specialists. There has been a significant increase in the capabilities of FEA software in the past 10 years. The main focus has been making software easier to use for less specialized users [3]. Also hardware improvements in the recent years have also contributed significantly to the reduction of analysis time which also reduced the need for model simplifications. Even with these improvements in software and hardware, it is still quite "difficult" for the non-specialists to be able to conduct FEA. Second, if FEA is used extensively in a company, the FEA package program must be available on a high number of high performance computers, which means increased software license and computer equipment costs. And the third problem carried with use of FEA for product development is the increase of analysis time. This problem has two parts: First, FEA is itself a rather time consuming process that requires attention on details at each step. Second, since the FEA analysis can only be conducted by FEA specialists, other design engineers have to wait until FEA specialist make the analysis for the part being designed, which sometimes creates queues for the analysis. These problems associated with FEA should be defeated in order to gain the competitive advantage that's necessary for a company to survive in the current market conditions [4].

The problem about effective use of FEA takes another view for the parts that are repeatedly analyzed in the production of system, since the cost and time of design is multiplied with number of repeated analyses. It has been estimated that in some auto parts manufacturing companies, more than 80% of the FEA analyses are of basic and repeat type [5]. Thus, the merits of having standardized FEA procedures for each product are numerous. Anti-roll bars are good examples for this case. It will be better to start with presenting some brief information about anti-roll bars.

Anti-roll bar, also referred to as stabilizer or sway bar, is a rod or tube, usually made of steel, that connects the right and left suspension members together to resist roll or swaying of the vehicle which occurs during cornering or due to road irregularities. The bar's torsional stiffness (resistance to twist) determines its ability to reduce body roll, and is named as "*Roll Stiffness"*. An anti-roll bar improves the handling of a vehicle by increasing stability during cornering or evasive maneuvers. Most vehicles have front anti-roll bars. Anti-roll bars at both the front and the rear wheels can reduce roll further. Properly chosen (and installed), anti-roll bars will reduce body roll, which in turns leads to better handling and increased driver confidence. A spring rate increase in the front anti-roll bar will produce understeer effect while a spring rate increase in the rear bar will produce oversteer effect. Thus, anti-roll bars are also used to improve directional control and stability. One more benefit of anti-roll bar is that, it improves traction by limiting the camber angle change caused by body roll. Anti-roll bars may have irregular shapes to get around chassis components, or may be much simpler depending on the car.

There are two important facts to be considered about the anti-roll bars within the presented information. First, the anti-roll stiffness of the bar has direct effect on the handling characteristics of a vehicle. And second, the geometry of the bar is dependent on the shape and location of other chassis components. In addition to these two facts, considering that anti-roll bar design is simpler than design of other chassis components, it is clear that in case of a problem about the handling of the vehicle or in case of a geometry change in one of the chassis components that leads to an interference with the anti-roll bar geometry, the first thing to be done is to change the design of the anti-roll bar. Therefore, design changes of the anti-roll bars at various steps of the vehicle production are quite common. The phrase "various steps" includes design, testing and manufacturing phases of the vehicle production and furthermore, in some cases, it can occur after marketing according to the customer responses.

The discussion on the availability and cost of FEA clarifies the fact that, it should be used more effectively. This necessity increases further as the number of analyses performed for a part increase. Methods that automate the design of such parts should be developed. Anti-roll bar design can be a suitable objective for a study of automating the FEA.

1.2 Objectives of the Study

This study aims to develop a software that automates the design analysis of anti-roll bars. The software will take the required parameters for the design of the anti-roll bar via user interface, perform FEA of the bar at the background and present the results of the analysis to the user. FEA part of the program will be run in the background and the user need not know or see the FEA program.

The major goal in the development of this software is to present a method for effective use of general purpose FEA package programs in the design analysis, which is a requirement in the current competitive market conditions.

The anti-roll bar is selected as the design objective since its design analysis must be performed many times during vehicle production due to design changes. Also, design parameters of anti-roll bars, which will be discussed in Section 3.2.2, are suitable for automated design.

The study itself includes two problems, first of which is the development of the automated design software while the second is to perform the detailed design analysis of the anti-roll bar. For the development of the user interface Visual Basic 6.0 [6] will be used. The FEA of the bar will be performed in ANSYS 7.0 [7]. The program code for FEA is going to be written in ANSYS Parametric Design Language (APDL). The main reasons for the selection of the these software are their availability in METU, existence of the studies using these software for similar purposes and familiarity of the author with these software.

1.3 Organization of the Thesis

The study is documented in the thesis within six chapters:

Chapter 1 presents the basic concepts and definitions for the subject, and states the problem. The objectives of the thesis and the organization of the dissertation are also included.

Chapter 2 is devoted to the review of literature for the regarding fields of the study.

In Chapter 3, finite element analysis of an anti-roll bar in ANSYS is studied in a detailed manner. Also, the characteristics of the employed ANSYS finite elements are mentioned.

Chapter 4 deals with the software prepared for the automated design analysis. First, the main structure and the flow chart of the program are given. Then, the ANSYS macro file, ANSYS parameters file and VISUAL BASIC Interface program are introduced.

The program capabilities and reliability of the program outputs are verified in Chapter 5. Also, the program is compiled with different input combinations in order to discuss the effects of design parameters on anti-roll bar properties.

In Chapter 6, the research is summarized and some conclusions are derived. Finally, recommendations for future studies are presented.

CHAPTER 2

LITERATURE REVIEW

2.1 Anti-roll Bar

2.1.1 Anti-Roll Bars and Vehicle Performance

Ride comfort, handling and road holding are the three aspects that a vehicle suspension system has to provide compromise solutions. Ride comfort requires insulating the vehicle and its occupants from vibrations and shocks caused by the road surface. Handling requires providing safety in maneuvers and in ease in steering. For good road holding, the tires must be kept in contact with the road surface in order to ensure directional control and stability with adequate traction and braking capabilities [8]. The anti-roll bar, as being a suspension component, is used to improve the vehicle performance with respect to these three aspects.

The anti-roll bar is a rod or tube that connects the right and left suspension members. It can be used in front suspension, rear suspension or in both suspensions, no matter the suspensions are rigid axle type or independent type. A typical anti-roll bar is shown in Figure 2.1.

The ends of the anti-roll bar are connected to the suspension links while the center of the bar is connected to the frame of the car such that it is free to rotate. The ends of the arms are attached to the suspension as close to the wheels as possible. If the both ends of the bar move equally, the bar rotates in its bushing and provides no torsional resistance. But it resists relative movement between the bar ends, such as shown in Figure 2.2. The bar's torsional stiffness-or resistance to twist-determines its ability to reduce such relative movement and it's called as "*roll stiffness*".

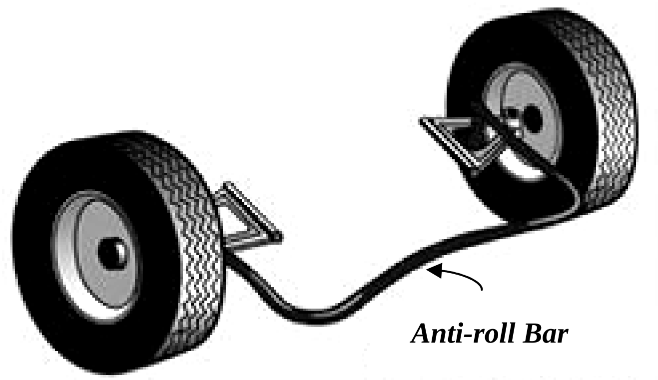


Figure 2.1 - A typical anti-roll bar

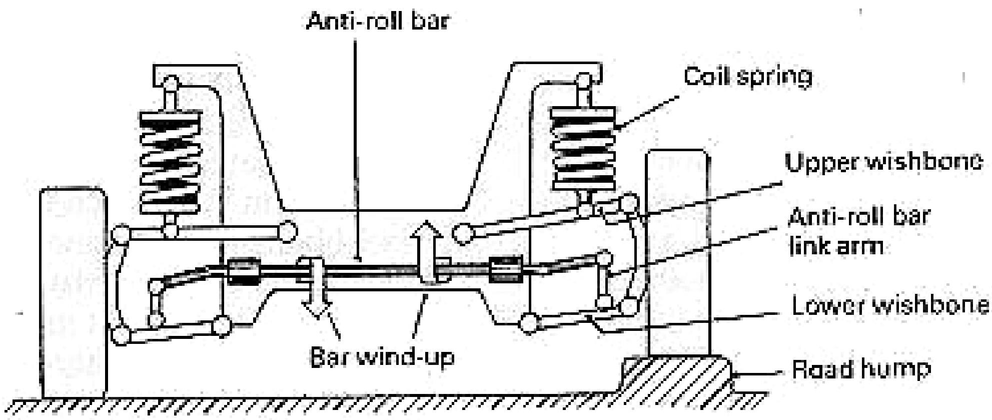


Figure 2.2 – An anti-roll bar attached to double wishbone type suspension. (The vehicle is crossing over a road bump on one side)

The main goal of using anti-roll bar is to reduce the body roll. Body roll occurs when a vehicle deviates from straight-line motion. The line connecting the roll centers of front and rear suspensions forms the roll axis roll axis of a vehicle. Center of gravity of a vehicle is normally above this roll axis. Thus, while cornering the centrifugal force creates a roll moment about the roll axis, which is equal to the product of centrifugal force with the distance between the roll axis and the center of gravity. This moment causes the inner suspension to extend and the outer suspension to compress, thus the body roll occurs (Figure 2.3). Body roll also occurs when a wheel crosses a bump at one side only, which was the case in Figure 2.2.

Figure 2.3 – A vehicle experiencing body roll during cornering.

Actually, body roll is an unwanted motion. First reason for this is the fact that, too much roll disturbs the driver and gives a feeling of roll-over risk, even in safe cornering. Thus, the driver cannot drive the vehicle with confidence. Second reason is its effect on the camber angle of the tires, which is the angle between the central plane of symmetry of the wheel and the vertical plane at the center of the contact patch. The purpose of camber angle is to align the wheel load with the point of contact of the tire on the road surface. When camber angle is changed due to body roll, this alignment is lost and also the tire contact patch gets smaller. The smaller the contact patch of the tire, the less traction exists against the road surface [9]. Therefore, body roll should be prevented.

The first way to prevent body roll is to eliminate its source, roll moment. This moment can be reduced by increasing the roll center heights of the front and rear suspensions. But, this will cause considerable lateral wheel displacements during bump and rebound with track variations during operation. Another negative effect is the higher camber angle change. Another method for preventing excessive body roll is to use stiffer suspension springs, thus making it harder for the suspensions to move in opposite directions at the same time. This however, reduces the ride comfort. A compromise solution is to use softer suspension springs to provide ride comfort, lower roll centers to avoid lateral wheel displacement and anti-roll bar(s) to reduce body roll.

Anti-roll bars serve two key functions. First they reduce body roll, as explained above, and second provide a way to redistribute cornering loads between the front and rear wheels, which in turns, gives the capability of modifying handling characteristics of the vehicle. This can be done by arranging the roll stiffnesses of the anti-roll bars at the front and rear suspensions. If a firmer anti-roll bar is installed at the front, then the distribution of lateral load transfer increases toward the front tires, since a firmer anti-roll bar allows less deflection, thus transfers lateral loads at a faster rate. And the overall result is additional understeer effect. Adversely, increasing roll stiffness at the rear by using firmer anti-roll bar will create an oversteer effect. Thus, anti-roll bars are also used to improve directional control and stability.

One negative effect of anti-roll bars is that, too stiff bars can reduce the adhesion on slick surfaces. This is especially true on snow and ice. They can also be a disadvantage for serious off-road driving [10].

After clarifying the need for use of anti-roll bars in vehicle suspension systems, it will be better to present some basic properties of anti-roll bars:

i. <u>Geometry:</u>

Packaging constraints imposed by chassis components define the path that the anti-roll bar follows across the suspension. Anti-roll bars may have irregular shapes to get around chassis components, or may be much simpler depending on the car. But, whatever the shape of the bar, it can be defined by a single curved bar centerline with a cross section swept along this centerline. Two sample anti-roll bar geometries are shown in Figure 2.4.

Figure 2.4 - Sample anti-roll bar geometries

ii. Cross-Section:

Anti-roll bars basically have three types of cross sections: solid circular, hollow circular and solid tapered. Among these three cross-section types, solid tapered bars are the most expensive ones and their use is not common. The solid circular bars are the oldest type of anti-roll bars. Their use is still the most common. However, in recent years use of hollow anti-roll bars became more widespread due to the fact that, mass of the hollow bar is lower than the solid bar that has the same anti-roll stiffness and the same bar centerline geometry.

iii. Material and Processing:

Anti-roll bars are usually manufactured from SAE Class 550 and Class 700 Steels. The steels included in this class have SAE codes from G5160 to G6150 and G1065 to G1090, respectively. Operating stresses should exceed 700 MPa for the bars produced from these materials. The bars are heated, formed (die forged or upset), quenched and tempered. The high stress regions should be shot peened and then coated in order to improve the fatigue life of the bar [11]. Use of materials with high strength to density ratio, such as titanium alloys, is an increasing trend in recent years.

iv. Connections:

Anti-roll bars are connected to the other chassis components via four attachments. Two of these are the rubber bushings through which the anti-roll bar is attached to the main frame. And the other two attachments are the fixtures between the suspension members and the anti-roll bar ends, either through the use of short links or directly.

Bushings:

There are two major types of anti-roll bar bushings classified according to the axial movement of the anti-roll bar in the bushing. In both types, the bar is free to rotate within the bushing. In the first bushing type, the bar is also free to move along bushing axis while the axial movement is prevented in the second type.

Figure 2.5 – Type-1 Bushing (rubber bushings and metal mounting blocks)

The bushing material is also another important parameter. The materials of bushings are commonly rubber, nylon or polyurethane, but even metal bushings are used in some race cars. The increase in the spring stiffness of bushing material also increases the roll stiffness of the bar.

Connections to Suspension Members:

One type of connections used between suspension member and the anti-roll bar is the pin joint shown in Figure 2.6. Spherical joints are also used to provide this connection.

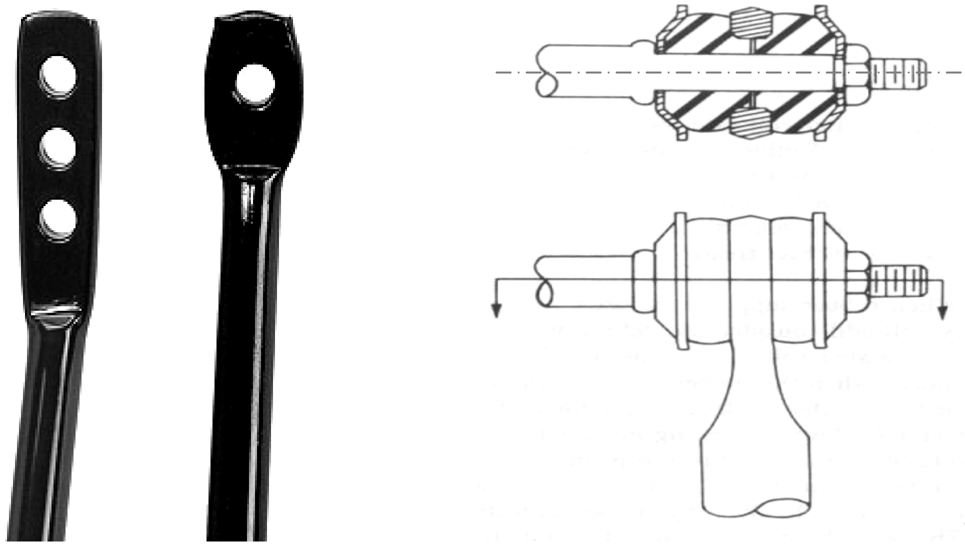


Figure 2.6 – Pinned connection between suspension member and the anti-roll bar (first one is adjustable)

The good thing about anti-roll bars is that they are very tunable by changing bar diameters, mixing and matching bushing materials or adjusting the moment arm length.

2.1.2 Studies on Design of Anti-roll Bars

The design of an anti-roll bar actually means to obtain the required anti-roll stiffness that improves the vehicles' stability and handling performance without exceeding the mechanic limitations of the bar material. Since, it's a straightforward process to analyze the anti-roll bar, it's not possible find published studies in the literature. The standard design analyses are performed by manufacturer companies, and the results are not published. Rather, the studies focused on the bushing characteristics and fatigue life analysis of the anti-roll bars are available. Also, some design automation studies about anti-roll bars are present.

Society of Automotive Engineers (SAE), presents general information about torsion bars and their manufacturing processing in “Spring Design Manual” [11]. Anti-roll bars are dealt as a sub-group of torsion bars. Some useful formulas for calculating the roll stiffness of anti-roll bars and deflection at the end point of the bar under a given loading are provided in the manual. However, the formulations can only be applied to the bars with standard shapes (simple, torsion bar shaped anti-roll bars).The applicable geometry is shown in Figure 2.7.

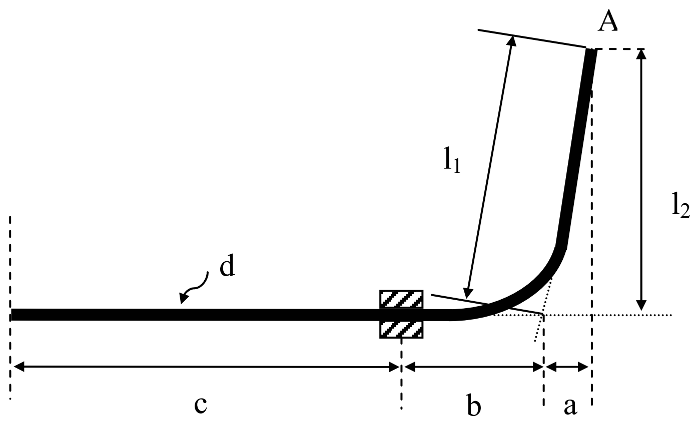


Figure 2.7 Anti-roll bar geometry used in SAE Spring Design Manual

The loading is applied at point A, inward to or outward from plane of the page. The roll stiffness of such a bar can be calculated as:

$$L = a + b + c \qquad \text{(Eqn. 2.1)}$$

(L :half track length)

$$f_A = \frac{P}{3EI}\left[l_1^{\,3} - a^3 + \frac{L}{2}(a+b)^2 + 4l_2^{\,2}(b+c) \right] \qquad \text{(Eqn. 2.2)}$$

(f_A : Deflection of point A)

$$k_R = \frac{PL^2}{2f_A} \quad \text{Nmm/ rad} \qquad \text{(Eqn. 2.3)}$$

(k_R : Roll Stiffness of the bar)

In the study of Shih, Kuan and Somnay [5] the aim was to develop standard FEA procedures to guide FEA jobs. Six benchmarking examples are reviewed in the paper including anti-roll bar analysis. First of all, general finite element analysis procedures to be standardized are determined. Then all steps are defined in detail for each product type. According to the authors, for the anti-roll bar, understanding the problem means understanding the function of this suspension member on vehicle's roll performance. The anti-roll bar bushings considered in the paper are made of rubber and have both some radial and axial stiffness, but allow rotation about their axis. Loading, mounting and attachment to other components are presented as main points of consideration. For model clean up and de-featuring, the pin hole and the bushing locations are required. Auto meshing is employed for meshing the model with finite elements. Displacement inputs are specified at the surface of pinholes that connect the bar to the steering knuckles. Thus, representative vertical displacements at the ends of the stabilizer bar are imposed. The boundary conditions are applied at the bushing locations. The bushing model consists of a set of 6 degrees-of-freedom springs. Zero rotational stiffness along the axis of the bushing was assigned to the bushing model so that free rotation of the bushing is allowed along this axis. The end of each spring is grounded by connecting it to the vehicle body. Direct comparison of stresses/strains and deflections from two parallel FEA jobs, one for an existing design whose performance has been validated in the field, and one for the new but similar design, presented as method to reach conclusions on a structural analysis. It's also claimed that, both the Von Misses stress/strain and/or the maximum principal stress/strain reversals can be used for design approvals in the fatigue analysis.

ArvinMeritor Inc. engineers added one more step to the above study as reported by J. Saxon and Chip Beaulieu [12]. The software, with the name "*Stabar*", was developed to fully automate the design analysis of the anti-roll bars by applying the standardized analysis steps defined by the previous reference. In the study, the bar geometry is regarded as a constraint while the cross-section, material and the process are seen as the design issues. The geometry details at bar ends are neglected due to moment free connection with the suspension member. Analyses of bars with three types of cross sections are available in the tool: solid circular, tubular and solid round tapered. Each cross-section type is modeled in ANSYS using BEAM189, SHELL93 and SOLID45 elements respectively. Bushing elements are modeled with ANSYS COMBIN14 elements. Constant amplitude displacements are imposed at the bar ends in reversed directions as the loads. A linear static analysis is performed in ANSYS to obtain the solution. Endurance test is typically performed using the stress results obtained from the solution assuming fully reversed loading cycles. The software developed also creates a report including input parameters and results of the analysis.

Thi [13], developed a software for finding the optimum torsion bar design using the strain energy capacity as the optimization criteria. The anti-roll bars in Mac-Pherson type suspensions are analyzed in the study. The length and diameter of the torsion bar, maximum suspension deflection and allowable shear stress on the bar are used as the constraints on design. Two case studies are presented with two different material types, AISI 2340 Steel and a titanium alloy.

In the paper by Visteon Corporation engineers Gummadi, Cai, Lin , Fan and Cao [14] , five different types of anti-roll bar bushings are investigated for their effects on the anti-roll bar performance. The five bushing types analyzed in the study are: Conventional Bushing, Grippy Flat Bushing, Bushing with Upset Ring, Chemically Bonded Bushing and Compressively Bonded Bushing. Some of these bushing types were developed by the authors and compared with the conventional bushing. The authors claim that, the axial movement of the anti-roll bar within the bushing reduces its effectiveness, thus it should be prevented. In the four types of the analyzed bushings, other than conventional bushing, the axial movement is prevented by different methods. The comparison of the bushing types was based on three criteria: Roll Stiffness, Maximum Stress and Manufacturing Cost. Finite element analysis is performed for the solid bar bushing with different bushing types, using ABACUS program. The loading was given by constraining one end of the bar while applying a vertical load on the other end. The results show that, roll stiffness increases by preventing the axial movement of the bar in the bushing. The highest roll stiffness is obtained for compressively bonded bushing while a close value is obtained for the chemically bonded bushing. The maximum stress differs on a narrow range except the bushing with grippy flats, on which stress concentration occurs near flattened portions. The manufacturing cost of the conventional bushings is the lowest, which makes their use the most widespread, while the chemically bonded bushing has the highest cost.

Palma and Santos [15] presented a detailed study on the fatigue life analysis of the anti-roll bars. In the study, fatigue damage correlation of a stabilizer bar in front suspension (McPherson) of a passenger car between laboratory and road experiments is presented. Cumulative fatigue damage theories together with experimental and analytical techniques of stress analysis are used to determine the fatigue damage imposed on the stabilizer bar, under both conditions (laboratory and actual conditions). FEM models of the stabilizer bars were used to determine the local stresses at critical regions. These stresses were then measured in laboratory, by using strain gages bonded on the material. The assessments of fatigue damage of the stabilizer bar under actual conditions were performed with a component mounted on a vehicle, which was driven over different road surfaces and velocities. The results of both experiment types were correlated and discussed. The material of the bar used in the study is SAE 5160 steel submitted to shot peening and painted. Finite element analysis of the bar is performed in I-DEAS program using static displacement loads of ± 41 mm at bar ends. The maximum stress is observed at bushing locations. The stress on the bar is also calculated by an analytical methodology. Three methods – strain gages, FEA and analytical method – gave consistent results and strain-gage results are corrected according to FEM results. Then maximum Von-Misses stresses are calculated using strain gage measurements and the mean and the alternating stresses are converted to fully reversed stress cycles using Goodman relationship. Fatigue life of the bar is calculated using S-N curves. Same methodlogy is performed for the results of the road experiments after employing rainflow method for cycles number counting. Severe road tests resulted in a maximum Goodman stress lower than obtained for 41 mm displacement in the laboratory. The fatigue life of the bar under fully reversed 41 mm displacement cycles is calculated as 78,000 cycles, the service life under sever road conditions will be higher than this value. This means practically infinite life according to the authors.

2.2 Finite Element Method as a Design Tool

Many problems in engineering and applied science are governed by differential or integral equations. The solutions to these equations would provide an exact, closed-form solution to the particular problem being studied. However, complexities in the geometry, properties and in the boundary conditions that are seen in most real-world problems usually means that an exact solution cannot be obtained or obtained in a reasonable amount of time. Current product design cycle times imply that engineers must obtain design solutions in a relatively short amount of time. They are content to obtain approximate solutions that can be readily obtained in a reasonable time frame, and with reasonable effort. The FEM is one such approximate solution technique. The FEM is a numerical procedure for obtaining approximate solutions to many of the problems encountered in engineering analysis.

As a computational method, the finite element method originated in the engineering literature, where in the mid 1950s structural engineers had connected the well established framework analysis with variational methods in continuum mechanics into a discretization method in which a structure is thought of as divided into elements with locally defined strains or stresses. Basic concepts have evolved over a period of 150 or more years. Some of the pioneering work was done by Turner, Clough, Martin and Topp (1956) [32] and the name of the *finite element method* appeared first in Clough (1960) [33]. In the early 1960s, engineers used the method for approximate solution of problems in stress analysis, fluid flow, heat transfer, and other areas. In the late 1960s and early 1970s, the FEM was applied to a wide variety of engineering problems. The 1970s marked advances in mathematical treatments, including the development of new elements, and convergence studies. Most commercial FEM software packages originated in 1970s and 1980s. [16]

In the FEM, a complex region defining a continuum is discretized into simple geometric shapes called *finite elements*. The properties and the governing relationships are assumed over these elements and expressed mathematically in terms of unknown values at specific points in the elements called *nodes*. An assembly process is used to link the individual elements to the given system. When the effects of loads and boundary conditions are considered, a set of linear or nonlinear algebraic equations is usually obtained. Solution of these equations gives the approximate behavior of the continuum or the system. In solid mechanics, the so-called Rayleigh-Ritz technique uses the Theorem of Minimum Potential Energy (with the potential energy being the functional, π) to develop the element equations. The trial solution that gives the minimum value of π is the approximate solution. [17]

A typical FEA job consists of five steps: information gathering, pre-processing, analysis, post-processing, and analyzing the FEA results to arrive at conclusions. Information gathering includes collecting background data, defining the analysis goals, and developing an action plan. Quite a few steps are involved in the "preprocessing" or preparing the CAD model for analysis. The meshing job starts with the "clean up", "de-featuring" and "feature-adding". This includes getting rid of unnecessary entities and features, and introducing "test fixtures" in the CAD model in preparation for load and boundary condition implementations. This is followed by the specifications for material properties, loading, and boundary conditions. Stress/strain, deformation, and fatigue analyses are major tasks to be carried out in the post-processing. Conclusions as to whether the design is sound are made based on relative comparisons of stress and deformation of similar designs, material yield, or durability requirements [2]. The flow diagram of a typical finite element analysis job is presented in Figure 2.8.

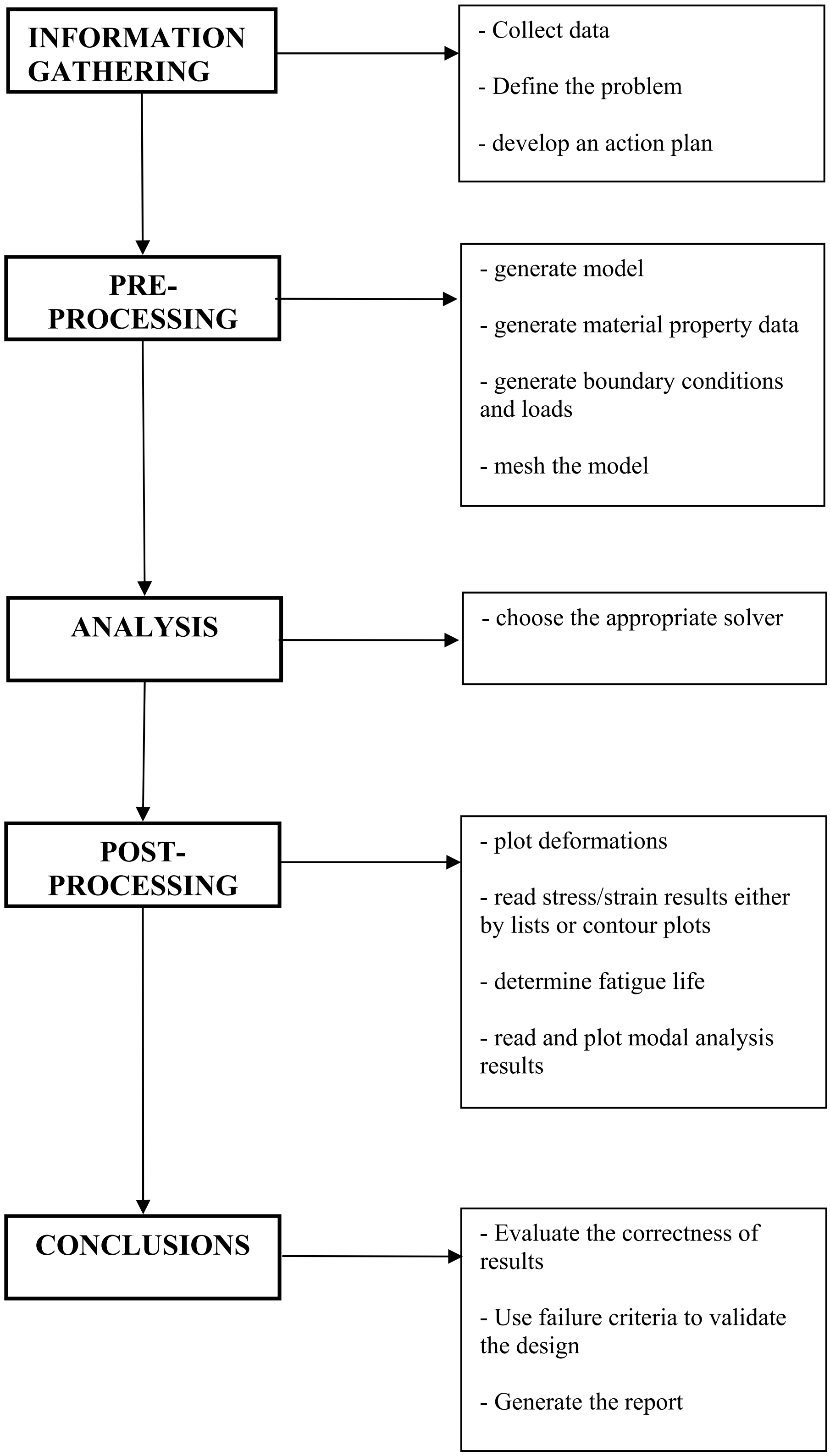


Figure 2.8 – Steps involved in a typical FEA

2.3 Design Automation

i) Automation of FEA for Design Analysis:

The need for effective use of finite element design analysis is well stated in reference [2]. There are two primary reasons for such a study. First, high quality and low cost products increase customer satisfaction and market share, and second, the efficient use of a company's FEA resources has the immediate benefit of reducing product development cycle times. It's noted by the authors that, FEA is carried out by a companies central analysis group and this approach creates problems due to job queues at various steps such as job request, problem definition, data transfer, modeling, analysis, redesign which increase the design cycle time. A methodology must be established by the FEA specialists in providing the design engineers with product oriented specific FEA guidelines, thus allowing them to conduct FEA for specific products. FEA by design engineers will lead to significant cycle time reductions. Support must be provided to the design engineers for selecting the appropriate tool (software and hardware) for analysis and determining the scope and type of FEA. Also, guidelines for each specific FEA job must be developed and the design engineers must be made familiar to the approaches for solving specific problems based on the experiences of the FEA specialists. The general FEA procedure is also discussed in the study.

The first step into the automated finite element design analysis is to standardize the analysis procedures as stated by reference [6]. The focus of the study is on, how to develop standard FEA procedures to guide FEA jobs. In the paper, first the FEA procedures in six benchmarking examples are reviewed. And considerations that must be taken into account when developing the standard FEA procedure for a heavy-duty axle's brake hub are presented in detail. The six benchmarking examples include parking brake bracket, axle housing, suspension beam, axle carrier housing, marine driveline double yoke and stabilizer bar. The job in developing the standard FEA procedure is not to solve any particular problem but to have a general understanding on what considerations should be taken and how to define the process.

General guidelines are developed for the following steps; understanding the problem, cad model clean-up, mock-up, and de-feature, auto mesh and model de-bugging, material properties, loading, boundary conditions, analysis, post-processing and conclusions. The authors also mention the importance of the design and analysis databases formed as a result this study. It's claimed that the standard FEA procedures have been proven to be beneficial and the time-saving in job execution can be as high as 5-10 times.

The result of the above two studies was the software "Stabar" which performs automated anti-roll bar analysis [12]. This program performs previously defined anti-roll bar analysis procedures automatically through a user interface. The analysis procedures performed by the automated anti-roll bar design software "Stabar" were presented in the previous section. The authors regard the analysis as a straightforward but time consuming procedure. However the company, ArvinMeritor Inc., was suffering from the delay typically occurred between the request for analysis, made by the design engineer, and the actual analysis performed by an FEA specialist. The analysis and documentation required several hours of the specialist's time. Productivity could obviously be improved if a program or macro could be created that drove the FEA software to produce the analysis and documentation, thereby freeing up the FEA specialist for more difficult projects. The complete solution required a combined Engineering/IT effort and resulted in a software tool that combines Active Server Page (ASP) technology with an ANSYS APDL macro. The authors complain about several difficulties that had to be overcome to bring these technologies together, but they are satisfied with the result since the solutions opened the door for development of similar tools that efficiently distribute engineering solutions through web technologies familiar to the end user. The software is available to all offices of the company through intranet.

Reference [13] presents a computer program developed with Visual C++ to find the best solution for a torsion bar installed in the front wheel suspension system. The computer program, which can run in Windows environment on any PC, gets the necessary inputs from the user and gives the optimum results for dimension and maximum strain energy capacity of the bar. Optimization techniques are employed to find the length and diameter of the straight portion of the multi-piece anti-roll bars.

Automated design studies for finite element analysis of other components are also available in the literature. The work by Abd El-Ghany and Farag [18] describes an expert system that provides an intelligent interface between the non-destructive testing engineer and the finite element analysis (SDRC/I-DEAS) software. The system helps in efficiently evaluating the stress concentration resulting from the presence of volumetric discontinuities inside the body of the material. This expert system has a wide knowledge base and decision-making skills that are taken from the published documentation and the experience of human experts. In addition, it contains a large number of rules that determine the appropriate type of elements, meshing and solving techniques that should be used for different welded joints. The expert system asks the user questions about the shape and dimensions of the basic part that contains the discontinuities and the shape and dimensions of the embedded discontinuities. Then, it operates I-DEAS, creates the geometry of the basic part and the discontinuities, prepares surfaces for meshing, meshes according the appropriate set of rules, applies loads and boundary conditions and solves the model using the appropriate solving techniques. Finally, it produces a report describing the stress concentration around each discontinuity and checks whether it is harmful to the structure or not. The expert system has a modular structure that can be easily updated and applied for more sophisticated jobs.

The paper by Padhi and McCarthy [19] discusses the development of a software tool for design of composite bolted joints, using three-dimensional finite element analysis. The tool, with the name BOLJAT, allows the user to create the joint geometry through a menu-driven interface and then generate a customized mesh according to the user's needs. Contact parameters are defined automatically, which shields the user from the most difficult part of the process. Boundary conditions, bolt pre-loads, and material properties can also be set. Only a few manual steps are necessary to complete the finite element code generation process. By automating the time-consuming model creation process, the tool facilitates the increased use of three-dimensional finite element analysis in the design of composite bolted joints. The GUI of the software was developed by adding a BOLJAT menu item to MSC.Patran main menu. The model generated by BOLJAT is ready to solve with MSC.MARC. The program does not have post-processing capabilities.

İlhan [20] developed a user interface for investigating the response of filament wound composite tubes and pressure vessels under various loading conditions by finite element method. The interface was developed with Visual Basic 6.0 and it calls the finite element analysis program after preparing the necessary input parameters. The finite element analysis is performed in ANSYS 5.6, using batch mode, which means the analysis program works in the background. The user interface is used for data input about geometry parameters, element type, mesh density, failure criteria etc. The program is capable of calculating failure loads, stress, strain and displacement values as well as the optimum winding angle for a given material, geometry and loading combination. The results are presented numerically on the user interface.

In the study of Alagöz [21], the input file for the finite element analysis of long fiber reinforced composite spur gears is created via user interface. The numerical results of the analysis with a contour plot of safety factors can also be reviewed using the program interface. The user interface is created in Borland Delfi Pascal 3.0 and runs in WINDOWS environment. The interface produces the input files for finite element analysis software ABAQUS running on HP Series Workstations in Unix environment. The input files created by the interface must be transferred to the HP Workstation. The user runs the ABAQUS program manually and specifies the transferred files as input files of for the analysis. The result files of the analysis are then transferred to PC for post-processing via interface.

ii) <u>Other Design Automation Examples:</u>

The design automation studies are not limited to finite element analysis. Sapa Aluminum's engineers found many opportunities for automation in the process of creating extrusion dies, because even though each extruded product is different, every die begins as a slice of round metal and the basic steps in creating it are the same each time [22]. They developed a Visual Basic program that automated many of the stages normally required for toolpath generation - such as automatically importing the CAD geometry; verifying the model for surface continuity; determining the size of the end mills; and so on. The program has significantly reduced toolpath programming time for aluminum extrusion dies from four hours, to 30 minutes.

Tang, Ogarevic and Tsai [23] introduce a flexible, general purpose, integrated Computer-Aided Engineering (CAE) system, called Durability and Reliability Analysis Workspace (DRAW). It carries out the simulation-based spectral fatigue damage and failure probability analysis of mechanical components. The engineering capability of DRAW is to predict durability (fatigue life) and reliability of mechanical components based on duty cycle information. The corresponding CAE tools for the purpose of aiding engineers in the durability and reliability analysis have been developed and implemented in the system. The system also provides a graphical, menu-driven user interface for quick and easy interaction with these tools. Advanced Computer Integrated Technology is utilized to integrate CAE tools bound into a system with automatic control, coordinate, and communicate. The main objective of the research is to provide a layer of network computational services for reliable remote computations and data transfers between an engineering workstation and a computation server on a high-speed computer. It incorporates methods of both engineering and computer science to allow the engineer to solve engineering problems through automation and reliability, utilizing high-speed procedures.

2.4 User Interface Design

i) <u>The Basics of User Interface Design</u>

The user interface of an application has the greatest impact on the user's opinion. No matter how technically brilliant or well optimized the code may be, if the user finds the application difficult to use, it won't be well received. In designing the user interface for an application, the user must be kept in mind. A well-designed user interface insulates the user from the underlying technology, making it easy to perform the intended task.

The stages of interaction between the interface and the user are described by Marinilli [24] as follows: First, the user forms a conceptual intention from her/his goal (i). Second, user tries to adapt this intention to the commands provided by the system (ii) and from these commands carries out the action (iii). Then, the user attempts to understand the outcomes of her/his actions (iv). This is particularly important for computer systems, where the inner workings are hidden and users have to figure out the internal state only from few hints. The last three stages help the user to develop her/his idea of the system. The whole process is performed in cycles of action and evaluation. The user refines the model of the system she/he has in mind by interpreting the outcome of her/his actions. The interaction styles found in the literature are listed in the article as: *Menu Selection* (if there are a number of items), *Form Filling* (used for data input), *Direct Manipulation* (users can manipulate the characters on the screen as if they were real, ex: word processors), *Command Language* (express commands by using the command line prompt facility), *Natural Language* (voice recognition and speech synthesizers). According to the author, it is essential to provide feedback for the system's internal state. This important feature can be achieved by using different techniques. The most commonly used techniques are the followings: *Changing the pointer shape* (waiting pointer), *Animations* (progress bar, ad-hoc), *Messages* (message dialogs, status bars).

According to Kennedy [25], users evaluate the softwares with three aspects: *Usefulness (*does the job, makes them more efficient, gives them additional power), *Usability (*ease of use*,* easy to learn, user friendly) and *Subjective appeal (*aesthetic appeal*,* similarity to other products used, good experiences). The author refers to Jakob Nielsen's Nine Heuristics about interface design which are:

i. Simple and natural dialogue
ii. Speak the user's language
iii. Minimize the user's memory load
iv. Be consistent
v. Provide feedback
vi Provide clearly marked exits
vii Provide shortcuts
viii. Provide good error messages
ix. Prevent errors

Kennedy also reminds the Great Law of Interface Usability "A system should be usable - without assistance or instruction – by someone inexperienced with the system but knowledgeable and experienced in the domain of the application" which requires focusing on the first time user in program development.

ii) <u>Composition of the Interface</u>

The composition or layout of the form not only influences its aesthetic appeal, it also has a tremendous impact on the usability of the application. Composition includes such factors as positioning of controls, consistency of elements, affordances, use of white space, and simplicity of design [26].

In most interface designs, not all elements are of equal importance. Careful design is necessary to ensure that the more important elements are readily apparent to the user. Important or frequently accessed elements should be given a position of prominence; less important elements should be relegated to less prominent locations. Grouping of elements and controls is also important. Information should be grouped logically according to function or relationship. In many cases, frame controls can be used to help reinforce the relationships between controls. Position of the controls must be related to the normal work flow of the user.

Consistency is a virtue in user interface design. A consistent look and feel creates harmony in an application while lack of consistency can be confusing, and can make an application seem chaotic and disorganized even causing the user to doubt the reliability of an application. For visual consistency, a design strategy and style conventions must be established before beginning development. A subset of controls should be chosen that best fit a particular application among wide variety of controls available for use in Visual Basic and used appropriately according to their properties. Consistency between different forms in the application is also important.

A user interface also makes use of affordances. For instances, the three-dimensional effects used on command buttons make them look like they are meant to be pushed. Text boxes also provide a sort of affordance; users expect that a box with a border and a white background will contain editable text.

Too many controls on a form can lead to a cluttered interface, making it difficult to find an individual field or control. White space must be incorporated in the design in order to emphasize the design elements. Consistent spacing between controls and alignment of vertical and horizontal elements can make the design more usable as well.

Perhaps the most important principle of interface design is the simplicity. From an aesthetic standpoint, a clean, simple design is always preferable. By creating logical groupings of fields and using a tabbed interface or several linked forms, all of the information can be presented without requiring the user to scroll. Additional controls, such as a list box preloaded with choices can also be used, which reduce the amount of typing required of the user. Providing defaults can sometimes simplify an application. Wizards can also help to simplify complex or infrequent tasks. The best test of simplicity is to observe the application in use. If a typical user can't immediately accomplish a desired task without assistance, a redesign may be in order.

The use of color in the interface can add visual appeal. Small amounts of bright color can be used effectively to emphasize or draw attention to an important area. As a rule of thumb, the number of colors should be limited in an application, and the color scheme should remain consistent. The use of icons can also add visual interest to the application. In designing icons, standards that are already established by other applications must be considered. Again, design consistency is important in choosing fonts. Too many fonts can leave the application looking like a ransom note.

CHAPTER 3

FINITE ELEMENT ANALYSIS OF ANTI-ROLL BARS

In this chapter, the procedures applied in finite element analysis of an anti-roll bar are explained in detail. Although, the procedures in FEA are standard, data input methods, analysis options and result viewing methods show differences among different FEA package programs and also among different versions of the same package program. In this study, the analysis of the anti-roll bar is performed with ANSYS Release 7.0. Therefore, the procedures explained in the following sections are valid for this version of the ANSYS program.

3.1 ANSYS Finite Elements Used in Anti-roll Bar Analysis

i) <u>BEAM189 Element</u>

BEAM189 is a quadratic (3-node) beam element in 3-D, as shown in Figure 3.1. BEAM189 is defined by nodes I, J, and K in the global coordinate system and accounts for the initial curvature of the beams. Node L is always required to define the orientation of the element. This element is suitable for analyzing slender to moderately stubby/thick beam structures. Shear deformation effects are included. This element is based on Timoshenko beam theory, which is a first order shear deformation theory: transverse shear strain is constant through the cross section; that is, cross sections remain plane and undistorted after deformation. BEAM189 has six or seven degrees of freedom at each node. By default six degrees of freedom occur at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. A seventh degree of freedom (warping) can also be added by setting element options. This element is well-suited for linear, large rotation, and large strain nonlinear applications. BEAM189 can be used with any beam cross section defined in the program. Elasticity, creep, and plasticity models are supported. BEAM189 ignores any real constant data. Using keyoption settings, torsion-related shear stresses and flexure-related transverse shear stresses can be output together or separately. Element output is available at element integration stations and at section integration points [27].

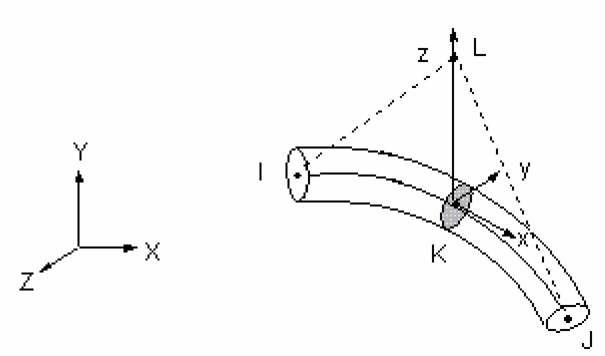


Figure 3.1 – ANSYS BEAM189 element

By default, ANSYS divides a cross-section into sub-sections in order to provide accurate results. The elements are provided with section relevant sub-sections, automatically by the program, at a number of section points. The number of sub-sections can be increased by the user. Each sub-section is assumed to be an assembly of predetermined number of 9 node cells. Figure 3.2 illustrates a sub-section model of a rectangular cross-section. Each cell has 4 integration points

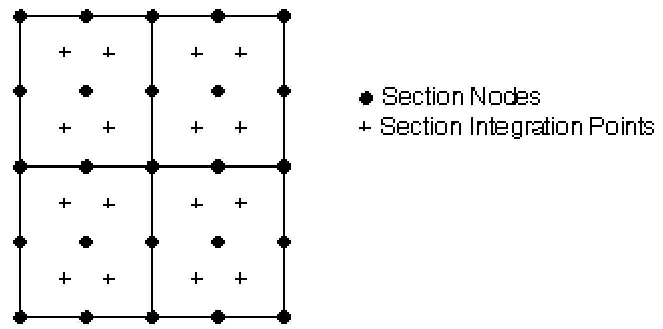


Figure 3.2 - Sub-sections of a rectangular cross-section

ii) COMBIN14 Element

COMBIN14 is a spring-damper combination element that has longitudinal or torsional capability in one, two, or three dimensional applications. The longitudinal spring-damper option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered. The torsional spring-damper option is a purely rotational element with up to three degrees of freedom at each node: rotations about the nodal x, y, and z axes. No bending or axial loads are considered. Keyoption settings are used for defining the element as 1D or 3D, longitudinal or torsional spring. The element has no mass. The geometry, node locations, and the coordinate system for this element are shown in Figure 3.3. The element is defined by two nodes, a spring constant (k) and damping coefficients (c_{v1}) and (c_{v2}). The spring or the damping capability may be removed from the element by setting k or c_v equal to zero, respectively. If (c_{v2}) is not zero, the element is nonlinear and requires an iterative solution [27].

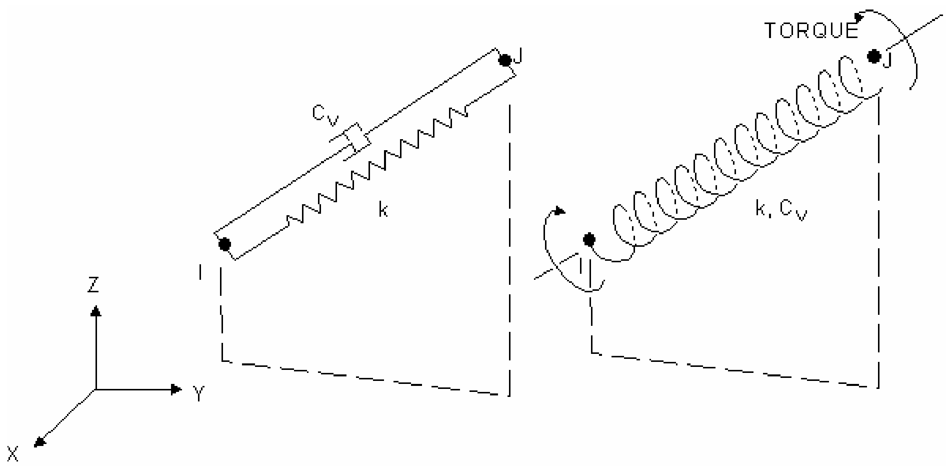


Figure 3.3 – ANSYS COMBIN14 element

3.2 Design Analysis of Anti-roll bars in ANSYS

A typical ANSYS analysis has three distinct steps:

1. Build the model.
2. Apply loads and obtain the solution.
3. Review the results.

These 3 steps are performed using pre-processing, solution and post-processing processors of the ANSYS program. Actually, the first step in an analysis is to determine which outputs are required as the result of the analysis, since the number of the necessary inputs, analysis type and result viewing methods vary according to the required outputs. After determining the objectives of the analysis, the model is created in pre-processor. The next step, which is to apply loads, can be both performed in pre-processor or the solution processor. However, if multiple loading conditions are necessary for the required outputs and if it is also necessary to review the results of these different loading conditions together, solution processor must be selected for applying loads. The last step is to review the results of the analysis using post-processor, with numerical queries, graphs or contour plots according to the required outputs.

3.2.1 Determination of Design Outputs

As mentioned in the pervious chapters, the basic goals of using anti-roll bars are to reduce body roll during cornering and to improve handling characteristics of the vehicle. The roll-stiffness property of the anti-roll bar is used to provide extra roll-stiffness to the front or rear suspensions. Therefore, to perform an anti-roll bar analysis basically means to determine its roll stiffness. In order to determine the roll stifffness, the deflection of the bar ends under a defined loading, in the direction of suspension motion, must be obtained. This deformation value, with some trigonometric relationships, can be then used for calculating the roll-stiffness of the bar.

In all machine component analysis, a component must be designed such that the stresses and strains occurring during operation will not exceed material limits. The material limits are determined by material properties and some known deformation theories (Maximum Normal Stress Theory etc.). In order to check the safety of using the part in operation, the designer must know the maximum stress and strain values on the part with material properties and the formulation of the deformation theory to be applied. Therefore, the maximum stress and strain values, in the case of maximum suspension deflection (maximum loading), must be obtained as the result of the anti-roll bar analysis.

The anti-roll bar, as being a vehicle suspension component, is subjected to alternating loads during its life time. Therefore, fatigue life evaluation becomes another necessity for the anti-roll bar analysis. A long operational life is required for an anti-roll bar (above 70000 cycles under fully reversed cycles of maximum loading), thus the final design have to satisfy this condition.

Modal analyses of automobile components have great importance in ride comfort studies. The vibrations and noise, to which the passenger is exposed, should be kept within certain limits. This fact brings the requirement of determining the natural frequencies and mode shapes of the vehicle components, and this applies clearly for the anti-roll bar.

The mass and finished length of the bar are the last of the required outputs from anti-roll bar analysis. It's obvious that the mass of the bar should be minimized, which is a general consideration for all automobile components. Also, the length is an important parameter since it affects the mass and production cost of the component.

3.2.2 Determination of Design Parameters

The parameters of anti-roll bar design are:

- Bar geometry

- Bar cross-section
- Bar material
- Bushing type
- Bushing location
- Bushing length
- Stiffness of the bushing material
- End connection type

Bar geometry is defined by a single curved bar centerline. The cross-section types that will be considered in this study are solid circular and hollow circular, since use of tapered bars are not common. Two types of bushings explained in Section 2.1.1 will be considered in the analysis, one of which constraints the bar movement along bushing axis while the other not. Also spherical and pin joints will be used for providing bar ends' connection to the suspension members.

3.2.3 Determination of Constraints and Loads

The anti-roll bar is connected to the other chassis components via four attachments explained in Section 2.1.1. Two of these are the bushings through which the bar is connected to the main chassis of the vehicle, while the other two are the connections between the bar and the suspension links at bar ends. At the bushing connections, the bar is free to rotate within the bushing and its vertical and lateral movements are constrained by the bushing material in both bushing types. However, movement along bushing axis is dependent on the bushing type. This movement may be constrained or not. At the bar ends, since the bar is to travel vertically along with the suspension member, bar ends' lateral displacements are constrained. These constraints may create some erroneous results if the suspension member does not travel absolutely vertical, but this is not a common case. If end connections are provided with spherical joints, there are no rotational constraints while only the rotational degree of freedom about x-axis exists for the pin joint and the other two rotational freedoms are constrained.

When the vehicle experiences body roll, one wheel will pull one end of the stabilizer bar down while the other wheel will pull the opposite end of the stabilizer bar up. The loading of the bar is the relative displacement of the bar ends which are connected to the suspension members. Hence, the stabilizer bar will be under combined bending and torsional loading.

The deflection of the bar ends is related to maximum permissible body roll angles. For passenger cars this angle is limited around 3.5°. Assuming a track length of 1300 mm with a beam axle suspension, the maximum deflection at the bar ends will be around:

$$f_A = (1300/2) \cdot \sin(3.5) \cong 40mm$$

This displacement will be smaller for independent type suspensions.

3.2.4 Analysis

Now steps involved in a typical anti-roll bar analysis will be explained in detail. An ANSYS session can be conducted both by using menus of ANSYS Graphical User Interface (GUI) or the Command Line. Most of the menu operations performed during the analysis have command replicates. Here, the analysis procedures will be explained but neither GUI operations nor the command replicates will be presented.

ANSYS has 10 processors each of which is specialized for special purposes. During the anti-roll bar analysis 4 of these processors will be used. These are: AUX15 (IGES file transfer processor), PREP7 (model creation preprocessor), SOLUTION processor, POST1 (Database results postprocessor). Use of AUX15 processor is dependent on the model creation method which will be explained in Part-(iii) of this section.

The nodal degrees of freedom are denoted by UX, UY, UZ, ROTX, ROTY, ROTZ meaning translations in the x, y, and z directions and rotations about the x, y, and z axis respectively. The coordinate axis convention that will be used in the analysis is shown in the figure below:

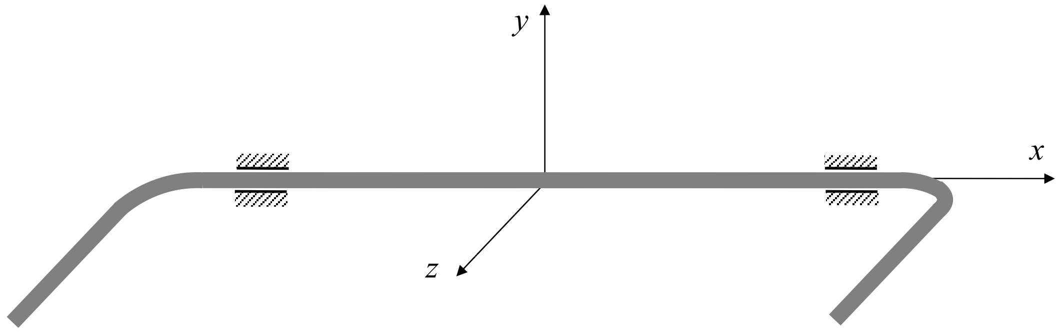


Figure 3.4 – Orientation of the anti-roll bar in Cartesian Coordinates

i. Basic Procedures

After starting the ANSYS session, a jobname and an analysis title should be defined before entering any processors. This task is not required for an analysis, but is recommended. The jobname is a name that identifies the ANSYS job and by using a different jobname for each analysis you ensure that the jobname becomes the first part of the name of all files the analysis creates, thus no files are overwritten. If an analysis title is defined, ANSYS includes the title on all graphics displays and on the solution output.

The ANSYS program does not assume a system of units for the analysis, thus no initial setup for the units is required. Instead, the user must decide on the unit system and make sure to use that system for all the data inputs.

ii. Define Element Types, Element Real Constants and Material Properties

After performing the basic analysis procedures, the user must enter PREP7 preprocessor in order to continue with the analysis.

The first thing to be done in the pre-processor is to define the element types. Two different element types are required for modeling the anti-roll bar with its bushings. The bar will be meshed with BEAM189 elements while the bushings will be modeled by COMBIN14 elements. Actually, the anti-roll bar can be analyzed with solid, beam or shell elements (in case of hollow cross-section). However in this study beam elements are preferred. Beam elements are used to create a mathematical one-dimensional idealization of a 3-D structure. They offer computationally efficient solutions when compared to solid and shell elements. Since the aim of the study is to create an automated design, computational efficiency is very important. Also control of the meshing operation is easier when using beam elements. The basic features of the BEAM189 and COMBIN14 elements are given in Section 3.1. The main reasons of selecting these elements can be listed as follows:

<u>Reasons of using BEAM189 elements</u>:

- Computationally effective
- Easy to use, both in pre-processing and post-processing phases
- Nonlinear analysis capabilities
- Does not require real constants
- User defined section geometry can be assigned
- Accounts for initial curvature of the beams
- Torsional and transverse shear stresses can be reviewed separately or together

<u>Reasons of using COMBIN14 elements</u>:

- Damping property can be removed
- Simpler data input than other combination group elements
- Nonlinear analysis capabilities

Choice of keyoptions for the element types is an important part of the analysis. An element can behave completely different with different keyoption arrangements. For COMBIN14 keyoption-3 is selected as “0” in order to obtain a 3D Longitudinal

Spring while keyoption-4 of BEAM 189 is selected as “2” to obtaining result output state of both torsion-related and transverse shear stresses together.

After defining the element types, the real constant sets must be defined for these element types. BEAM189 element does not require any real constants, while stiffness and damping ratio are the required real constants for COMBIN14 element. However, since the analysis type will be static, we only need to define spring stiffness of the COMBIN14 element. For BEAM189 elements a cross-section must be defined instead of real constants. A cross section defines the geometry of the beam in a plane perpendicular to the beam axial direction. As denoted before, anti-roll bars with solid circular and hollow circular cross-sections will be analyzed, thus these cross-sections must be created with appropriate dimensions. ANSYS itself divides a defined cross-section into sub-sections in order to ensure accurate solution and the number sub-sections can be supplied by the user. If this information is not supplied by the user, the program uses its default number of divisions for the cross-section. In this study 10 circumferential and 3 radial subsections are used for solid circular cross-section, while hollow cross-section is divided into 20 circumferential sub-sections. These decisions are based on some analyses performed with different sub-section numbers. The results of this study are given in Section 3.2.5. The subsections for the two cross-sections are represented below:

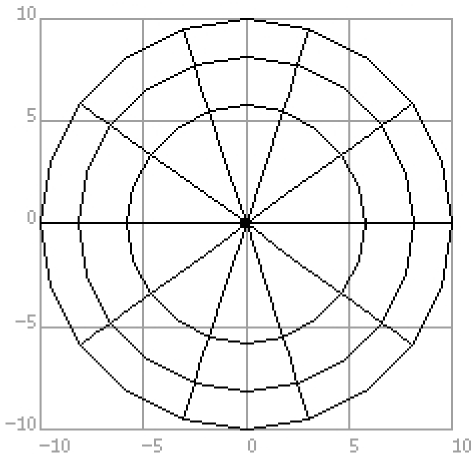


Figure 3.5 – Sub-sections for the solid cross-section (10C-3R)

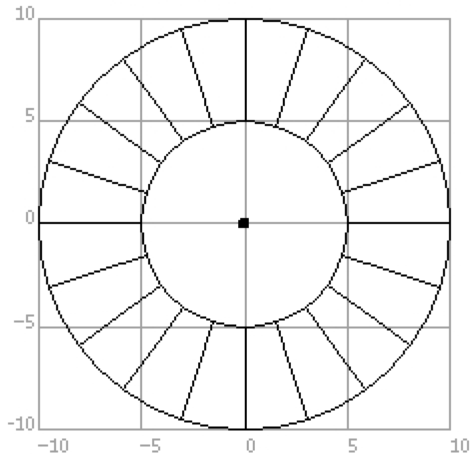


Figure3.6 - Sub-sections for the hollow cross-section (20C)

The next step is to define material properties for the element types. Opposite to the real constants, this time COMBIN14 element does not accept any material properties. Therefore, material properties will only be defined for BEAM189 elements. Since there is no material non-linearity, a structural linear isotropic model is created with supplying Modulus of Elasticity (E) and Poison's Ratio (υ) as the inputs. Density of the material is also added to this model.

In an ANSYS analysis, all defined element types, real constants, cross-sections and material properties have a reference number. At the current stage of anti-roll bar analysis, there exist 2 element types with reference numbers 1 for BEAM189 and 2 for COMBIN14 and 1 set of Real constants, 1 cross-section type (either solid or hollow) and 1 material model each with reference numbers 1.

iii. Modeling the Anti-Roll Bar

The next step in the analysis is generating a finite element model -nodes and elements- that adequately describes the model geometry. To represent the 3-D structure of a beam, one-dimensional beam elements are used. In order to model a beam with beam elements, first a line that follows the beam axis must be created and then this line must be meshed with beam finite elements. The beam finite elements should have the same cross-sectional properties with the beam to be modeled.

The geometry of any anti-roll bar, even with irregular shapes, can be defined by a single curved bar centerline and a cross-sectional area swept along this centerline. The cross-section of the bar was created in the previous sections. This cross-section will be assigned to the beam elements during meshing of the beam. However, before any other operations the bar centerline must be created. Two alternatives exist at this stage, creation of the model in ANSYS or importing the model from an IGES (Initial Graphics Exchange Specification) file. The import of the model from IGES file can only be performed in AUX15 processor of ANSYS. However, this operation must be completed before entering any other processors which means all the procedures explained from the beginning of Part-ii up to this point should be skipped first and performed after IGES file import. The IGES file to be imported should satisfy some requirements such as:

- Dimensions must be same with system of units that will be used during the analysis.
- The geometry can be imported as a single line or combination of lines connected to each other which define the anti-roll bar centerline. Anti-roll bar cannot be imported as a volume.
- Line connections with zero fillets are not allowed and sharp corners should be avoided for a good finite element solution.
- Any labels, symbols and extra lines must be cleaned from the drawing.
- The orientation of the bar centerline must be conform to the coordinate system convention to be used in the analysis.

ANSYS asks the user to select one of the two options, smooth and faceted, during file import process. However, these options deal with solid models and choosing any of the options does not create difference for line model import. After importing the model, the geometry must be checked first to see whether it's correctly imported or not. Then all lines in the drawing must be combined to obtain single curved bar centerline.

In case of creating the model in ANSYS, "from the bottom up" method is used. First of all, keypoints, which are the "lowest-order" solid model entities, must be created. Then these keypoints are connected with straight lines which are then connected to each other with fillets. The final step in creating the bar centerline is to combine all these lines and fillets, as we have done in the case of file importing. A sample bar centerline is shown in Appendix B.1.

Now the bar centerline can be meshed with BEAM189 elements. However, there is still a problem, which is locating the bushings on the bar. In order to model the bar correctly nodes are needed at the midpoints of the bushings. Actually, this problem can be solved by creating hardpoints at the bushing locations on the bar centerline and then meshing the line. The program will automatically create nodes at hardpoint locations. But, this method can only be used with free meshing. In an ANSYS analysis, mapped meshing, if it is possible, is always preferred to free meshing. A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it while a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. Here, the concern is on the pattern of the mesh and it's preferred to have elements with equal size. Thus, another method is employed by dividing the centerline into 3 lines, using the bushing positions as division points. During meshing of these lines the ANSYS program will define a node at the ends of all three lines and merge the coincident nodes. Thus, nodes at the midpoints of both bushings are created.

Before meshing the line, some attributes to be associated with the generated beam elements must be defined. These attributes include:

- The beam element type
- The material set number
- The cross section ID
- The real constant set number
- The orientation keypoints

The first three of these attributes are defined as; Element Type 1, Material Set Number 1 and Cross-section ID 1, for all three lines. No real constants are required for the BEAM189 so the fourth attribute is not defined. The orientation keypoints are used to determine the orientation of the cross section with respect to the beam element axis. This property is actually not used for circular cross-sections since a circle's orientation is meaningless. But an orientation keypoint is asked by the program whatever the cross-section is. Defining a keypoint at a far distance in y axis will solve this problem.

Now all three lines can be meshed and the final mesh with real element shapes is shown in Appendix B.2. There exists a single node at the intersection points of lines, which means the model will behave like single beam. Number of elements used for meshing is an important issue. The results of some sample analyses performed with different number of elements are discussed in Section 3.2.5. As a result of the discussion, use of an element number between 100 and 200 is suggested. This case may change according to the geometry but 200 elements will guarantee a good solution and using more elements will only increase the analysis time.

There is an obvious difference between the model shown in Appendix B.2 and the real shape of the anti-roll bar; the bar ends. This is a simplification which is based on the fact that, anti-roll bars have spherical joints or bar and bayonet type (pin) connections at the end points both of which are moment free. Thus, stresses at the ends are not an issue [12]. This assumption is validated with some sample analyses in Section 3.2.5. It should be also noted that, the end connections -either pin or spherical joints- can be designed using the reaction force data without requiring finite element solution.

iv. Modeling the Bushings

Modeling the bushings of the anti-roll bar requires careful attention on the structures of the bushings. Two types of bushings were discussed in Section 2.2.1. In both types, bar is free to rotate within the bushing. This property is automatically accomplished by setting COMBIN14 elements as longitudinal springs. Since longitudinal spring elements have only translational degrees of freedom at their nodes, they cannot resist rotation. The basic difference between the two bushing types is the movement of the bar along bushing axis. Consider the first type, where the bar is free to move axially within the bushing. Here, the only restriction on the bar exists for its radial movement, which can be modeled with circumferentially distributed springs as shown in Figure 3.7. Here the springs are connected to the nodes at the division points of bar centerline (note that, the bar centerline was divided into 3 lines using 2 bushing positions). This arrangement can be simplified to two springs, one of which acts along y axis while the other acts along z axis, demonstrated in Figure 3.8. Here the most important issue is the determination of the equivalent spring stiffness for the simplified model.

The numerical analysis that follows the figures also shows that simplified model has the same stiffness along all radial directions and can be used safely for modeling the bushings provided that correct stiffness values are assigned to the springs.

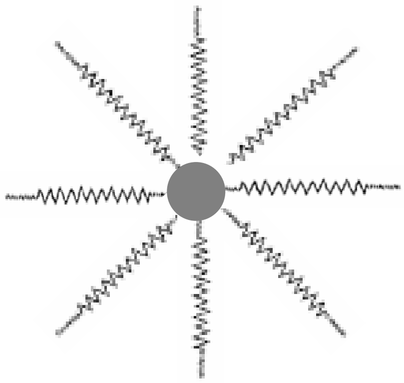


Figure 3.7 – Springs distributed around the circumference

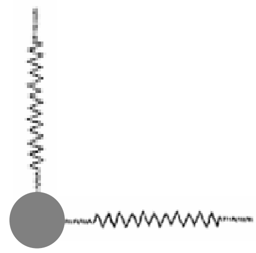


Figure 3.8 - Two spring arrangement used for modeling the bushings

In the simplified model, for a displacement of magnitude "*a*" with an angle of "*θ*" with the horizontal, the opposing forces of the springs with stiffnesses *k* are F_1 and F_2.

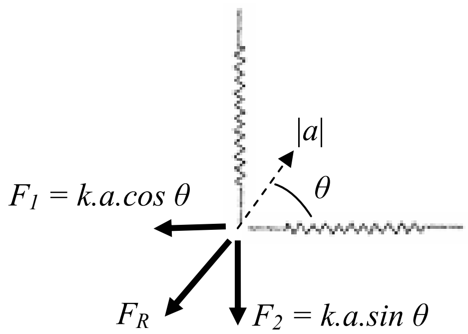


Figure 3.9 – Force analysis of the two spring bushing model

The resultant force due to the springs is:

$$F_R = (F_1{}^2 + F_2{}^2)^{0.5} = k.a$$

Thus two spring model has the same stiffness in all directions.

Another important parameter is the length the bushings. The two springs used for modeling the bushings must be distributed along the bushing length in order to obtain accurate solution. The need for this operation is clarified in Section 3.2.5 using some analyses results. The analyses demonstrate two cases: i) springs are attached to the node at the midpoint of the bushing, ii) spring are distributed to the nodes on the both sides of this node. For the longitudinal distribution of the springs, the spring stiffness of springs should be determined carefully. Since the springs work in parallel, the overall spring stiffness must be divided by the total number of nodes to which springs are attached.

Now the decided bushing model -bushing springs distributed to the nodes with two springs attached to each node- will be created. First of all, element type number is defined as 2 and real constant set number is defined as 1. No other attributes are necessary for COMBIN14 elements. In order to create a spring element two nodes are required. Luckily, one of these nodes exists on the bar model for each spring. The second nodes must be created at a distance from the bar centerline, in y direction for the springs that will work along y axis and in z direction for the springs that will work along z axis. The distance between the nodes is not important for the solution but can be around 50mm to represent the real figure. The spring elements are then created simply by using two nodes, one on the bar centerline and the other created in one of y or z directions. A sample bushing model is given in Figure 3.10.

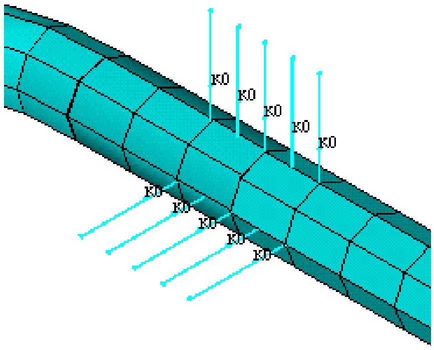


Figure 3.10 - Detailed View of Anti-roll Bar Bushing Model

v. Applying Boundary Conditions and Loads

This step can be performed in PREP7 preprocessor or SOLUTION processor. Since there are two loading conditions, one for obtaining roll stiffness and one for determining maximum stresses under maximum loading, and since it's preferred to review the results of these two loadings together, SOLUTION processor must be selected for applying loads. The *load step* method is used for accomplishing this task, in which user applies each loading condition separately and solves multiple load steps. A load step can be defined as one set of loading conditions for which the solution is obtained. By using multiple load steps the structure's response to each loading condition is isolated. There exists another loading for determination of the natural frequencies and the mode shapes of the bar. However, this is a different analysis type -*modal analysis*- and can not be solved as a third load step of the *static analysis*.

The displacement constraints exist at two locations: at the bar ends and at bushing locations. The *UX, UZ* degrees of freedom are constrained at the bar ends for spherical joints. *ROTY* and *ROTZ* degrees of freedom are also constrained if pin joints are used. At the bushing locations, free ends of the springs are constrained in all *UX, UY* and *UZ* degrees of freedom. These elements have no rotational dof's. The other ends of the spring, attached to the beam, are constrained according to the type of the bushing. *UX* dof is constrained for the second bushing type which does not allow bar movement along bushing axis.

The loading for the first load step -determination of roll stiffness- is a known force, *F*, applied to the bar ends, in +y direction at one end and in –y direction at the other end as shown in Figure 3.11.

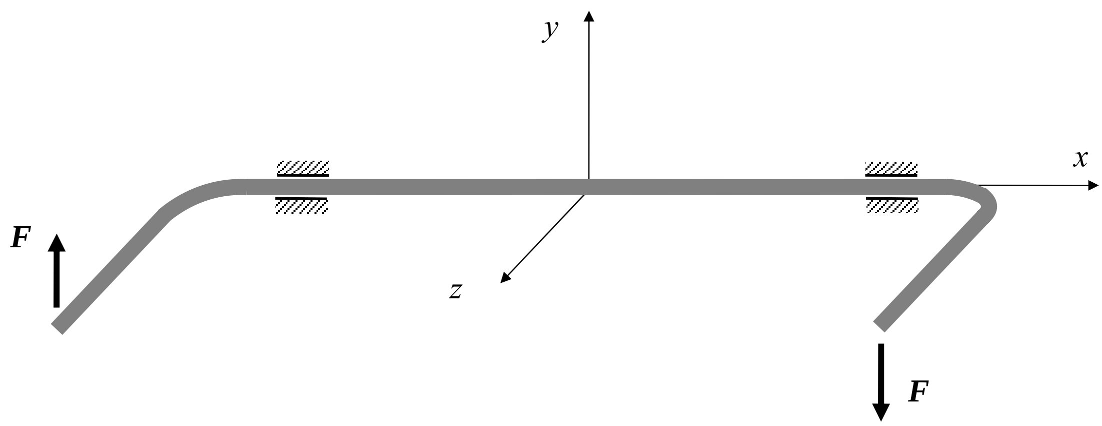


Figure 3.11 - Load Step1

For the second load step the force loads of the first load step are removed and displacement loads, representing maximum suspension deflection, are applied to the bar ends again in opposite directions.

There are no loads for the modal analysis and both force and displacement loads at the bar ends must be deleted while the constraints remain.

vi. Solution

The first step of solution is to choose the analysis type based on the loading conditions and the required outputs. For the first two loading cases given in the previous section, analysis type is *static*, since the loading is steady. The analysis type is selected as *modal* for the third case since the natural frequencies are to be determined.

Second decision is to determine whether the analysis will be linear or non-linear. A static analysis can be either linear or nonlinear. Some types of nonlinearities in a model are: large deformations, plasticity, creep, stress stiffening, contact (gap) elements, hyperelastic elements etc. In the anti-roll bar problem, only large deformations can create non-linearity. It should be better to check the difference between the results obtained form linear and non-linear analysis. The analysis of an anti-roll bar is performed using both linear and non-linear options and the results are compared in Section 3.2.5. The comparison shows that linear solution may not be adequate. A modal analysis cannot be non-linear, thus linear analysis option must be chosen.

The third step is the selection of the solver that solves the simultaneous set of equations that the finite element method generates. For the static analysis, this selection is left to the ANSYS program which selects a solver based on the physics of the problem. For the modal analysis a mode extraction method must be selected by the user. The Block Lanczos method is well suited for this analysis. This method uses the sparse matrix solver. The number of modes to be extracted will be defined as 5 (this is not a restriction), and the starting and ending frequencies are given as 0 and 200 Hz respectively. The extracted modes must also be expanded for obtaining the mode shapes.

After determining the solution controls the two static load steps are solved without exiting the solution processor. The results of these loads steps can be reviewed as described in the following section. After reviewing and storing the results of static analysis "New Analysis" option is selected from solution menu. After selecting analysis options according to modal analysis, the loadings of the static analysis are deleted and the modal analysis is performed.

vii. Post-processing the Results

POST1 post-processor of ANSYS is used for reviewing the analysis results. POST1 has many capabilities, ranging from simple graphics displays and tabular listings to more complex data manipulations such as load case combinations.

The first step in POST1 is to read data from the results file into the database. When each load step is solved in the SOLUTION processor, the results of that load step are written to a results file. This results file must be read into database for post-processing.

So, post-processing is started by reading the results of the first load step. Here, the required output is the deflection of bar ends. This deflection value can be obtained by first plotting the "DOF Solution - UY displacement" contour plot and then using the query picker to read displacement value of the node at the bar end. Another method is to list results "DOF Solution - UY displacement". The result can be directly read from the printed list if node numbers at the bar ends are known. The obtained displacement value is stored for use in roll stiffness calculation that will be presented in the following section.

Now, the results of the second load step of the static analysis can be read to database. Here, the stress and strain distributions on the bar under maximum loading are the considerations. The 1.Principal and Von Misses stresses and strains are important for failure analysis. Therefore, contour plots of these stresses and strains must be plotted. Maximum value occurring in each contour plot is printed as a label on the plot. Thus, this maximum value can be easily stored simply after reading it from the label. The stress or strain values at any point on the bar can be read by query picker. However, storing these results is the most difficult part of post-processing the beam elements. ANSYS does not store the maximum or minimum stresses or strains at a beam cross-section. The user can only list all the stress or strain values at integration points (a cross-section is divided into subsections by the program) of faces and middle section of each beam element. Thus, the maximum values at each section of the beam can be determined outside the ANSYS program after saving these listed results. Deformed shape of the bar should also be plotted and stored in order to see the shape of the bar under loading.

After reading and storing the results of the static analysis, user must exit POST1 and enter SOLUTION processor to perform modal analysis. After completing the modal analysis and returning again to POST1, results summary is viewed. This summary shows the first five natural frequencies of the anti-roll bar. Different from the static analysis, here determination of each natural frequency is regarded as a different load step. Thus, in order to see the mode shape of the 3. natural frequency, 3. load step must read into database. The mode shape, can be observed by plotting the deformed shape or animating the deformed shape.

viii. Determination of the Roll Stiffness and the Fatigue Life of the Bar

The roll stiffness and fatigue life calculations are performed outside the ANSYS program using the results of the FEA analysis.

<u>Roll Stiffness Calculation:</u>

Supposing that the load "F" that was shown in Figure 3.11 caused a deflection "f_A" at the bar ends, the roll stiffness of the bar can be calculated using the geometry presented below. Figure 3.12 shows the new orientation of the line that connects the bar ends. In case of rigid axle suspensions the movement of the bar ends is equal to the wheel movement, thus the vehicle body rolls with an angle "ψ". If the suspension is independent type, suspension members which are connected to the anti-roll bar ends move the same amount with the bar ends. Thus the ratio of the wheel travel to the suspension member travel is required for calculating the body roll angle for independent suspensions. [11]

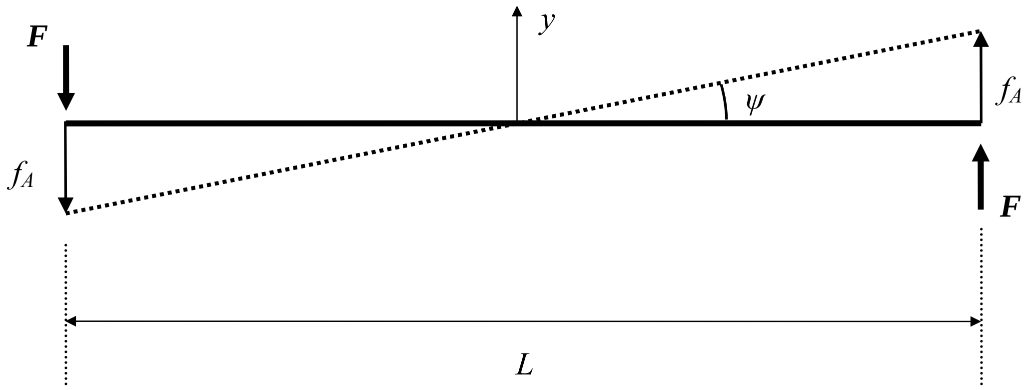


Figure 3.12 – The orientation of the line connecting bar ends, before and after deformation.

Assuming rigid axle suspension, the anti-roll stiffness (k_R) can be calculated with three different units as follows:

$$k_R = \frac{FL}{f_A} \quad \left(\frac{\mathrm{Nm}}{\mathrm{mm}}\right) \qquad \text{(Eqn. 3.1)}$$

$$k_R = \frac{F}{f_A} \quad \left(\frac{\mathrm{N}}{\mathrm{mm}}\right) \qquad \text{(Eqn. 3.2)}$$

$$k_R = \frac{FL}{\tan^{-1}(\frac{f_A}{L/2})} \quad \left(\frac{\mathrm{Nm}}{\mathrm{deg}}\right) \qquad \text{(Eqn. 3.3)}$$

All these three units are used in the literature for expressing the anti-roll bar stiffness.

Fatigue Life Calculation:

Fatigue life calculation is based on the maximum equivalent stress calculated on the bar as a result of the maximum suspension deflection. Then, fully reversed cycles of this maximum stress are assumed and the fatigue life is calculated using S-N curves. Actually, in real road conditions the loading is not always maximum and mean stresses exists as well as the alternating the stresses. However, the study presented in the reference [15] shows that the mean stresses in real road conditions are less than 1 percent of the alternating stresses. Assuming fully reversed load cycles of maximum suspension deflection is a simplification on the safe side.

For steels there exists an endurance limit ($S_e^{'}$) below which fatigue failure does not occur. The endurance limit can be estimated by using the ultimate tensile strength (S_{UT}) of the material. Actually, the relationship between S_{ut} and $S_e^{'}$ is dependent on the microstructure of the steel. Mischke [31], suggests the following formulation for relating the endurance limit of the steels to their ultimate tensile strength as:

$$S_e^{'} = \begin{cases} 0.504 \cdot S_{UT} & \text{if } S_{UT} < 1400 \text{ MPa} \\ 700 \text{ MPa} & \text{if } S_{UT} > 1400 \text{ MPa} \end{cases} \quad \text{(Eqn. 3.4)}$$

Here, $S_e^{'}$ is the endurance limit of the test specimen. Some modification factors must be used to obtain the endurance limit of a particular machine element (S_e). In this study, three endurance limit modifying factors are considered for endurance limit determination. The first modification factor is the size factor. Since the anti-roll bars are non-rotating components, the effective diameter must be found. For both solid and hollow cross- sections, the effective diameter is:

$$d_e = 0.370\,D \quad \text{(Eqn. 3.5)}$$

(D is the outer diameter for the hollow cross-section)

The size factor is then calculated as [28]:

$$k_1 = \left(\frac{d_e}{7.62}\right)^{-0.1133} = (0.0486D)^{-0.1133} \quad \text{for } 7.5 \text{ mm} < D < 137.8 \text{ mm} \quad \text{(Eqn. 3.6)}$$

The second modification factor is surface factor, which is related with the manufacturing method of the anti-roll bar. SAE recommends anti-rolls to be manufactured by die forging or upsetting. Some portions of the bar may require machining (especially the ends of the bar). Surface factor is calculated as follows [28]:

$$k_2 = a \cdot (UTS)^b \qquad \text{(Eqn. 3.7)}$$

UTS = Ultimate Tensile Strength of the material

a = 2.70 b = 4.51 for machining

a = 39.9 b = 272 for forging and upsetting

The effect of shot peening on the fatigue strength should be considered also. By shot peening process the surface of the anti-roll bar is pre-stressed in order improve fatigue performance. This improvement is dependent on the quality of shot peening process and the hardness of the material. An optimum shot peening applied to a hard material increases fatigue life by 30 percent or even more [29].

There third modification factor (k_3) accounts for miscellaneous effects. The effect of corrosion and coating must be included in this factor. Also temperature effect and reliability may be considered. It should be noted that, stress concentration factor is not used since finite element solution gives the exact stress values at each point of the part.

After determining the Modification factors, the Endurance Limit of the bar "S_e" is calculated as:

$$S_e = k_1 \cdot k_2 \cdot k_3 \cdot S_e' \qquad \text{(Eqn. 3.8)}$$

Note that, S_e' is the endurance limit of the bar material after quenching and tempering.

Now the S-N curve can be constructed as shown in Figure 3.13.

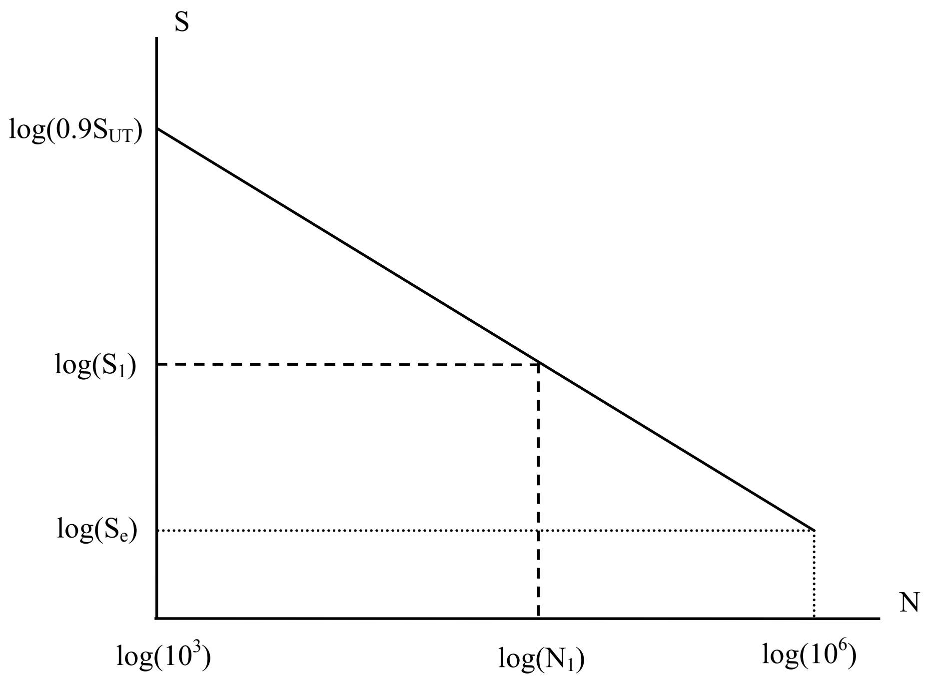


Figure 3.13 – S-N curve

Fatigue life of a bar "N_1" under fully reversed cycles of "S_1" can be calculated using:

$$\frac{6-\log(N_1)}{6-3} = \frac{\log(S_1)-\log(S_e)}{\log(0.9\cdot S_{UT})-\log(S_e)} \qquad \text{(Eqn. 3.9)}$$

For anti roll bars a fatigue life over 70000 cycles under fully reversed cycles of maximum loading is accepted as practically infinite operation life [15].

3.2.5 The Comparative Analyses about Finite Element Model

In this section, the comparative analyses that are performed to decide on a correct finite element anti-roll bar model are presented. First of all, a bar geometry is selected for use in all comparative analysis. This geometry is shown in Appendix B.1. The bushings are free x movement type and the end connections are provided by spherical joints. The units used in all graphs, plots and tables are in terms of N, mm and MPa.

i) <u>Number of sub-sections for solid and hollow cross-sections:</u>

It is obvious that, increasing the sub-section number will increase the accuracy of the solution. However, the analysis time is an important factor which is adversely affected from increased number of sub-sections. Therefore, an optimum sub-section number should be found where the results of finite element solution converge.

The solid cross-section is divided into 8 circumferential and 2 radial subsections by the program default. First the analysis is performed with these default values. Then, the analysis is repeated by increasing the sub section number. The results of the analysis are tabulated in Table 3.1.

<u>Table 3.1</u> – Subsection Number vs. Analysis Results for Solid Bar

Sub-section Number	8 Cir. – 2 Rad.	10 Cir – 3 Rad.	12 cir – 4 Rad.
Roll Stiffness (N/mm)	52.4	52.5	52.5
Max. Principal Stress (MPa)	454.1	446.2	447.6
Max. Equivalent Stress (MPa)	550.0	540.3	539.4

As seen from the above table, increasing the number of sub-sections has an obvious effect on the analysis results. 10 circumferential, 3 radial sub-sectioning will provide a compromise solution both accuracy and duration of the analysis.

The hollow cross-section can only be divided into circumferential sub-sections. Program's default value for the hollow cross-section is 8 sub-sections. The analysis for the hollow bar is first performed with this default values, and then repeated by increasing the number of sub sections. The results of the analysis are given Table 3.2.

Table 3.2 shows that, increasing the number of sub-sections considerably affects the analysis results and 20 subs-subsections seems to be the convergence point.

Therefore, 20 circumferential sub-sections should be used for the analysis of hollow bars.

Table 3.2 – Subsection Number vs. Analysis Results for Hollow Bar

Sub-section Number	8 Cir.	14 Cir.	20 Cir.	26 Cir.
Roll Stiffness (N/mm)	68.4	68.5	68.5	68.5
Max. Principal Stress (MPa)	487.7	482.4	477.1	477.0
Max. Equivalent Stress (MPa)	597.9	584.3	578.5	578.0

ii) Number of elements used for meshing:

As implied for sub-sectioning, increasing the number of finite elements used for meshing the anti-roll bar will also improve the accuracy of solution. Two limits must be set; a lower limit to ensure adequate accuracy and a higher limit to avoid waste of time. For this purpose, the analysis of the anti-roll bar shown in Appendix B.2 is performed with varying element numbers starting from 50 up to 150.

Table 3.3 – Number of Finite Elements vs. Analysis Results

Element Number	50	80	100	150
Roll Stiffness (N/mm)	52.1	52.3	52.4	52.4
Max. Principal Stress (MPa)	485.0	459.5	454.1	454.0
Max. Equivalent Stress (MPa)	559.7	553.9	550.0	549.9

It's clear from the above table that, satisfactory results are obtained with element numbers around 100. However, this number will vary with the geometry complexity and the length of the bar. Element number also affects the number of springs used for meshing the bushings. A finer mesh means a better constraining at the bushings. Therefore, a range of element numbers between 100 and 200 is suggested.

iii) Stresses at end points:

Typical anti-roll bars have spherical joints at the ends. When the anti-roll bar analysis is conducted with a vertical displacement of 65 mm on both ends of the anti-roll bar, in opposite directions, the equivalent stress distribution on the bar given in Figure 3.14 is obtained. Here, it's observed that the stress near bar ends is much lower than the maximum stress on the bar.

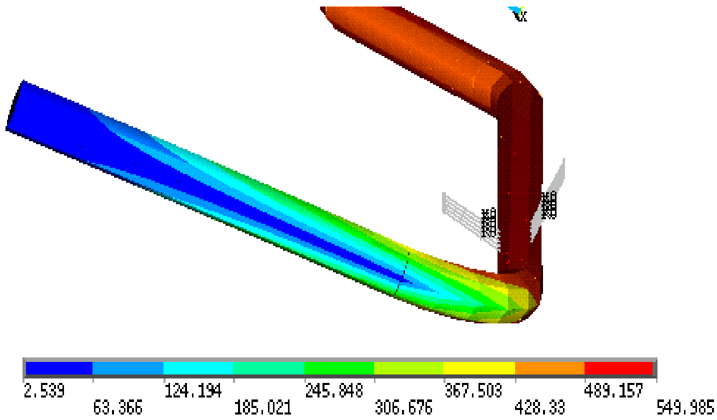


Figure 3.14 - Equivalent Stress Distribution at Bar Ends (Spherical Joint)

For the pin connection between the bar end and the suspension member, the same analysis results in a stress distribution shown in Figure 3.15. In this case, the stress near the bar ends has increased up to 170 MPa, but it is still much lower than the maximum stress which is around 530 MPa.

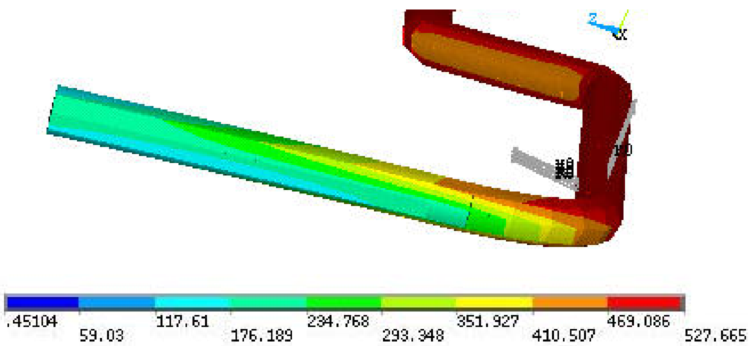


Figure 3.15 - Equivalent Stress Distribution at Bar Ends (Pin Joint)

iv) Distributing the bushing springs along bushing length:

Stress concentration, which occurs due to the application of loads and constraints on a single node, is an important fact that leads to erroneous results in finite element solution. To avoid this problem, loads and constraints should be distributed to nodes.

The two spring model, shown in Figure 3.7, is first applied to a single node at the center of the bushing, then distributed among 3 and then among 5 nodes (See Figure 3.10 for the distributed model). The results of the analysis with these three arrangements are presented in Table 3.4.

The results verify the stress concentration effect caused by using a single node for attaching the springs. In this case, connecting the springs at a single node would create an error of 10 MPa compared to using 5 nodes. The number of nodes that the springs are distributed is dependent on two factors which are the length of finite elements and the length of the bushings.

Table 3.4 – Number of Nodes used for Bushing Model vs. Analysis Results

Number of Nodes	1	3	5
Roll Stiffness (N/mm)	52.2	52.3	52.4
Max. Principal Stress (MPa)	465.1	460.7	454.1
Max. Equivalent Stress (MPa)	559.7	554.8	550.0

v) Linear or Nonlinear Analysis:

The stress analysis of the anti-roll bars can be conducted both with linear and non-linear solutions. The non-linearity in the anti-roll bar analysis can only be caused due to large deformations. The non-linear solution requires longer analysis times. Thus, if a compromise result is obtained using linear solution, linear solution should be preferred. Another important point about non-linear analysis is the change in direction of 3D longitudinal suspension springs used for modeling the bushings. In the linear solution, the springs in y direction resist movements only along y axis while the springs in z direction resist movements along z axis. However, in nonlinear solution the orientation of the spring change at all steps of the solutions and the springs start to resist movements along their new orientations. Thus, the effective bushing stiffness changes at all steps of the solution. To prevent this change, 1 dimensional longitudinal springs are used instead of 3D springs. This requires defining new element types using COMBIN14 finite element with different keyoption settings.

Table 3.5 summarizes the results obtained from the linear and non-linear analysis of the same anti-roll bar with a loading of 60 mm.

Table 3.5 – Analysis Type vs. Analysis Results

Analysis Type	Linear	Non-Linear
Max. Principal Stress (MPa)	454.0	629.1
Max. Equivalent Stress (MPa)	549.9	633.9

The contour plots showing the stress distribution on the bar explains the difference between the maximum stress results. Figures 3.16 and 3.17 show the equivalent stress distribution plots for the linear and non-linear cases respectively. It's obvious from the figures that, linear solution ignores the axial stresses caused due to the vertical displacement of the bar ends. Since the anti-roll bar is connected to the suspension member, bar ends move vertically up and down with the movement of suspension. This vertical movement will try to extend the bar. Thus the reaction forces will exist in all x, y and z directions at the bar ends. However, for the linear case following reaction forces are obtained at one end of the bar.

$R_x = -0.32181\ \text{E-06}\ \text{N}$

$R_y = 3407.7\ \text{N}$

$R_z = 2027.6\ \text{N}$

The reaction force along x direction is found as zero due to characteristics of linear solution. If the moment arm of the bar lied in x-z plane, The reaction force along z direction would also be 0. The real reaction forces are found from the non-linear solution as follows:

R_x = - 3153.1 N

R_y = 4017.8 N

R_z = 5050.5 N

In non-linear solution, roll stiffness is dependent on the deformation of bar. If roll stiffness is calculated near the undeformed position, equal roll stiffness is obtained form both linear and non-linear solutions, which is around as 52.4 (N/mm).

From all above, it can be concluded that non-linear solution should be preferred.

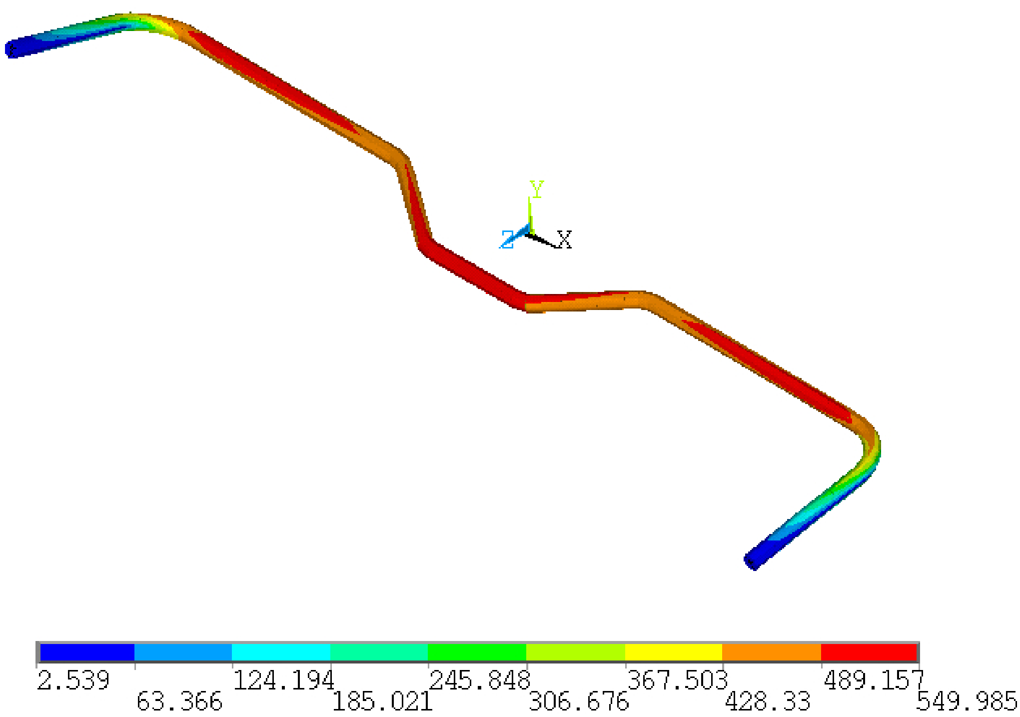


Figure 3.16 – Equivalent stress distribution on the bar - Linear solution

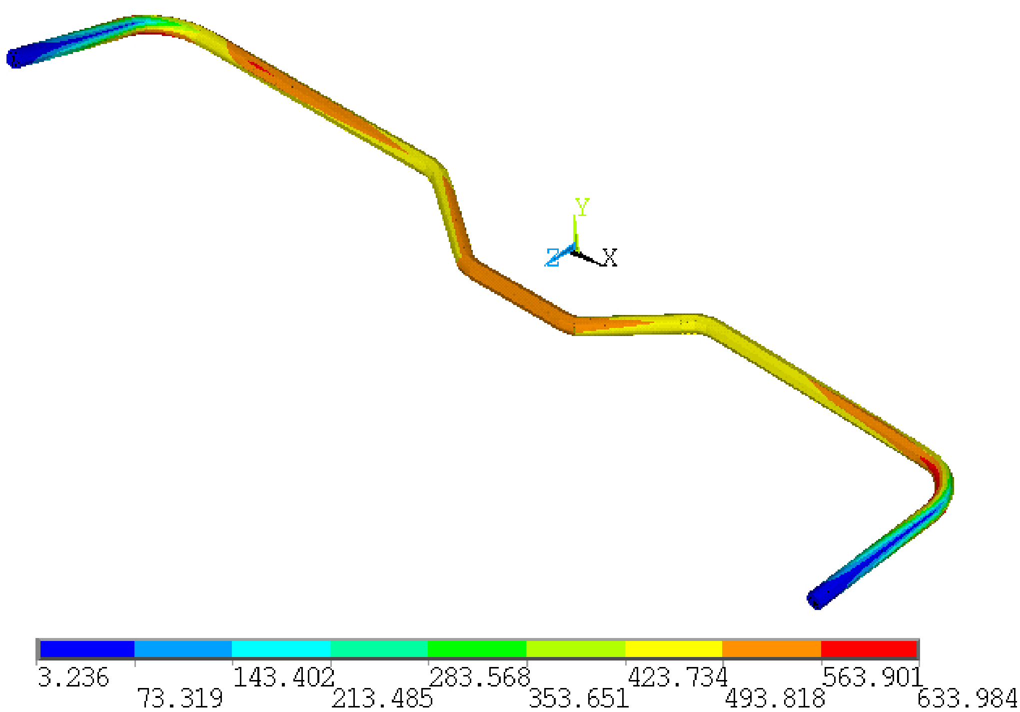


Figure 3.17 – Equivalent stress distribution on the bar – Non-linear solution

CHAPTER 4

THE AUTOMATED DESIGN SOFTWARE

4.1 Structure of the Software

The automated anti-roll bar design software is composed of two main parts; the user interface and the finite element analysis macro. The user interface is developed in Microsoft Visual Basic 6.0, which is a powerful tool to create graphical user interface applications for Microsoft Windows. The user interface of the automated design software acts as the pre-processor and the post-processor of the finite element analysis of anti-roll bars. Solution step of the analysis is performed using ANSYS 7.0, the well known finite element analysis program. The macro file capability of the ANSYS program is used for automating the design analysis. The macro file prepared for analysis purpose is composed in ANSYS Parametric Design Language (APDL).

The program can be executed using "autobar.exe" from any PC on which ANSYS 7.0 is installed. Some commands must be added to the "start.ans" file in the ANSYS setup directory, and these commands are given in the help file of the program. The user interacts only with the user interface of the program during the analysis. Only the start-up picture of ANSYS is seen at the beginning of the solution step for a few seconds.

The main structure of the program is presented in Figure 4.1. The interface provides data input forms to the user for modeling the anti-roll bar. The geometry parameters file is automatically created by a user command after finishing the data input. After creating the anti-roll bar model, solution is performed by simply clicking the "Start

Analysis" button on the interface. The user is prompted for waiting the ANSYS to complete its job. ANSYS saves the necessary results files in a folder for use in post-processing. Finally, the results of the analysis can be reviewed using the interface.

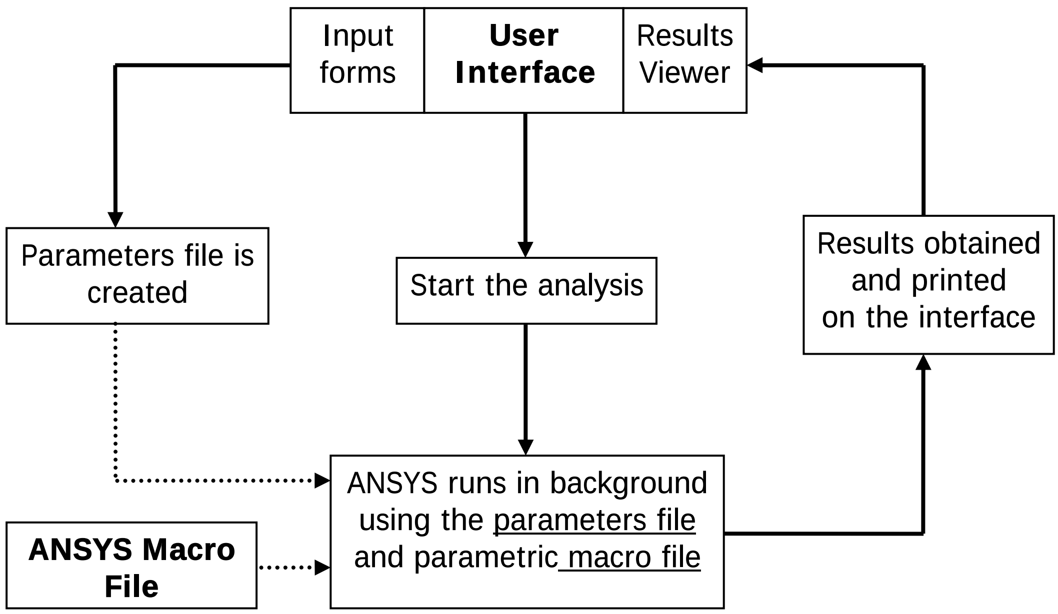


Figure 4.1 - Main structure of the program

The detailed flow chart of the program is presented in Figure 4.2.

Before starting the analysis, the created model can be previewed by the user. This option is provided to avoid waste of time for the analysis of an incorrect model. Also if the model is completely wrong, this may cause the ANSYS program to collapse. For the model preview, ANSYS is called by the interface, and the model created by ANSYS is printed on the interface. The user can change the geometry parameters if the created model is not satisfactory.

In both preview and the solution runs of the ANSYS, two files are used; one of which is the parameters file created by the interface and the other is the ANSYS macro. There are two macro files composed for preview and solution steps. These macros are written in parametric format and they work with the parameters file.

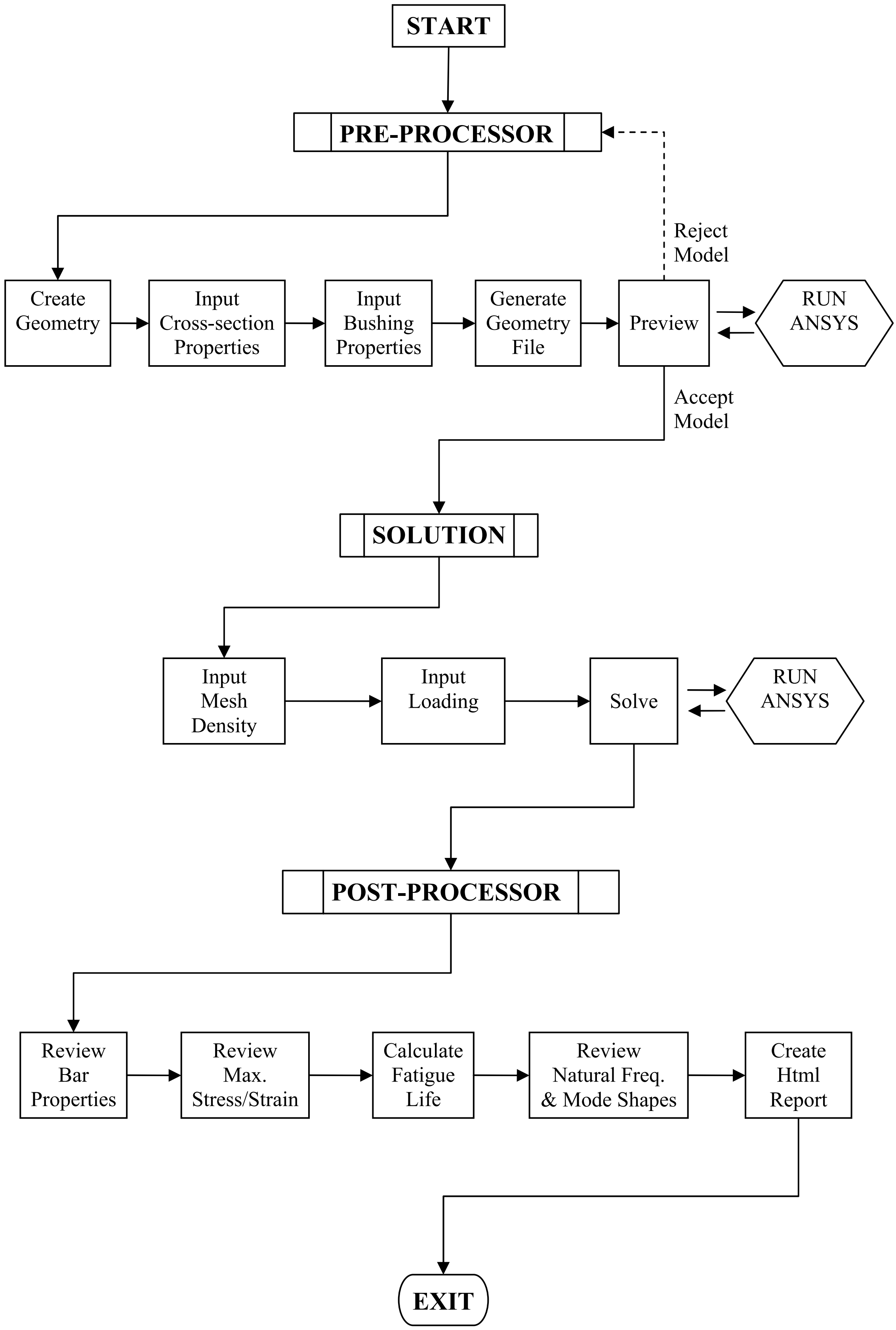


Figure 4.2 - Program Flow Chart

4.2 The Analysis Files

Analysis files are actually the text files composed of the commands required for an ANSYS analysis. Graphical user interface (GUI) of the ANSYS, provides menus and dialog boxes to the user for performing analysis steps. Almost all the operations performed in an analysis using the GUI have a command replicate. The user can write these commands directly to the command field provided in the GUI instead of using the menus. Another and more efficient way is using the macro files that include a sequence of commands. The macro files can be created outside the ANSYS program using a text editor. When a macro file is read by the ANSYS as input file, the commands in the file are performed. The macro files can also contain standard programming features such as logical statements, loop operations etc. The code for these operations is written in APDL (ANSYS Parametric Design Language). A complete ANSYS analysis can be performed with a macro file that includes all the necessary pre-processing, solution and post-processing commands.

In anti-roll bar analysis, there are two macro files and one parameters file. One of the macro files includes the commands up to the end of model creation. This file is used for creating a preview of the model. The user can change the model parameters according to the created preview. The other macro file includes all the steps from modeling to post-processing and this macro file will be explained here in detail.

4.2.1 Parameters File

The parameters file is created by the user interface and includes necessary analysis parameters of the anti-roll bar. This file contains both numerical parameters input by the user and selections among different options. The information that is written to the parameters file is:

- Model Creation Option (import or create in Ansys)
- Location of the CAD file to be imported (if to be imported)
- Number of Keypoints (if to be created)

- Keypoint Coordinates
- Fillet Radii
- Cross-section Type (solid or hollow)
- Section Dimensions (radius for solid case, inner and outer radii for hollow case)
- Section mesh (number of radial and circumferential divisions)
- Bushing Type
- Bushing Parameters (locations, length, stiffness)
- End Connection Type
- Material Properties (modulus of elasticity, poisons ratio, yield strength, ultimate tensile strength, endurance limit, density)
- Number of elements for meshing (defaults to 100 for preview)
- Bar ends' displacement loading
- Analysis Type (linear or non-linear)

The parameters file also includes commands for creating arrays for storing the keypoint coordinates (x, y, z) and the fillet radii.

4.2.2 ANSYS Macro File

The ANSYS macro file includes all the steps from beginning to the end of the solution. The following operations are accomplished by the commands in the macro file:

- The parameters file is read into the database
- According to the geometry creation method (declared in the parameters file), either AUX15 processor is started first for model import and the defined IGES file is imported to ANSYS or /PREP7 started directly at the beginning.
- After starting the /PREP7 pre-processor, BEAM189 and COMBIN14 elements are defined and keyoptions of these elements are arranged. For COMBIN14 keyoption-3 is selected as "0" which makes it a 3D longitudinal spring, while keyoption-4 of BEAM 189 is selected as "2" to obtain result output state of both torsion-related and transverse shear stresses together.

- A linear elastic material model is created with the user defined elasticity modulus and poison’s ratio.
- The cross-section is created. According to the selected option, either solid or hollow section is created with the defined dimensions.
- The keypoints are created with the coordinates written in the parameters file.
- Keypoints are connected by straight lines.
- The lines are connected with fillets.
- All lines and fillets are connected to obtain the single curved bar centerline.
- Start and end keypoints of the bar centerline is stored.
- Two keypoints are created on x axis at user defined bushing locations.
- Two large diameter disks are created in *yz* plane at these two keypoints.
- The intersection points of the bar centerline with these disks -the exact coordinates of the bushings on the bar- are determined. The disks are then deleted.
- The bar centerline is divided into three lines using the bushing points.
- BEAM189 is set as the active element type for meshing the beam.
- A keypoint is created at a long distance from the bar centerline in y axis and this keypoint is defined as the orientation keypoint for all three lines.
- The lines are meshed with user selected number of elements. The total number of elements is distributed among three lines according to their lengths.
- Number of spring elements to be used for modeling bushings is determined by using the user defined bushing length and size of the beam elements.
- User defined spring stiffness is divided to the number of springs to be used for modeling each bushing and this value is then defined as a real constant set for COMBIN14.
- COMBIN14 is set as the active element type
- Nodes are created 50 mm away from the nodes on the bar corresponding to bushing points, in y and z directions,
- Spring elements are created between these nodes and the nodes on the bar.
- Node numbers on the bar ends and bushing locations are stored.
- SOLUTION processor is started.
- Free spring ends are fixed.

- If bushing type is 2, x dof of the nodes at spring ends on the bar are constrained.
- x and z dof of one of the bar ends are fixed.
- For roll stiffness calculation, 1000 N force is applied in +y direction at one end and in -y direction at the other end.
- Analysis type is selected as static and the solution is performed
- Analysis type is selected as resume and the forces applied at the previous load step are deleted. User defined maximum suspension deflection is applied as displacement load on both ends of the bar in opposite directions along y axis.
- Second load step is solved and POST1 post-processor is started.
- Results of the first load step is read into the database and relative displacement of the bar ends along y axis is stored.
- Result of the first load step is read into the database.
- Labels, text fonts, image quality and graphics output size are arranged in order to obtain best graphics.
- Deformed bar shape and contour plots of principal stress, equivalent stress, principal strain and equivalent strain are captured from 7 different views and saved in graphics files.
- Maximum values of principal stress, equivalent stress, principal strain and equivalent strain on the bar stored in variables and output to text files.
- Relative movement of bar ends along y axis and the initial distance between bar ends along x axis stored and output to a text file. The length of the beam elements is also output to a text file.
- The principal stress and strains at each subgrid of each beam element is captured as listing and output to files with .js extension using the report generator of the ANSYS program.
- The POST1 post-processor is finished and the PREP7 pre-processor is started again for modal analysis.
- Modulus of Elasticity of the material model and the bushing spring stiffness is converted to relevant units for modal analysis and density is added to the material model.
- Displacement loads from the second load step are deleted.

- Translational x and z and rotational y and z degrees of freedoms of the bar ends are constrained.
- SOLUTION processor is started again and analysis type is selected as modal.
- Block Lanczos Eigensolver is selected as the solver with requesting 5 natural frequencies and corresponding mode shapes to be extracted.
- Solution is performed and first five natural frequencies are stored.
- POST1 is then started for saving the images of the animation sequences of mode shapes for the extracted frequencies. For this purpose Report Generator is used and 20 image sequences are saved after reading each load step (each natural frequency).
- A test file is created for verifying the ANSYS job finished.
- ANSYS is exited.

4.3 The User Interface

The user interface is composed of three forms; main form, wait form and about form. During the analysis, user interacts with the main form which contains all necessary controls for a complete anti-roll bar analysis. The form has a menu bar on the top which contains menu items for both classical interface actions such as file operations (new, open, save, save as, exit), for access to help, program information and analysis stages (pre-processor, solution, post-processor). The contents of the user interface are shown in Appendix-A with screen shots taken at different stages of an analysis.

The Pre-processor is composed of 6 tabs for defining geometry, cross-section, bushing and material options, creating the geometry file and previewing the model. The input geometry tab contains two options for importing the model from an .igs file or creating the model in ANSYS. At both stages quick help is available that reminds basic points about geometry. If the user selects the import option, the File Selection frame appears through which the user selects the .igs file to be imported. Important points about file import are also presented in this frame. If the create option is selected instead, Ansys Modeling frame is shown to user. Through this menu, user can input the keypoint coordinates and fillet radii for the bar centerline geometry. Symmetric and asymmetric geometry options are made available and by selecting the symmetric model, full geometry can be defined by inputting the keypoint coordinates on one side. Symmetry is about yz plane and there are two different symmetry types. In the first type, a keypoint exists at the plane of the symmetry while the second is completely symmetric about yz plane. Maximum number of keypoints for defining a bar centerline geometry is limited to 20. The second tab of the Pre-processor is the Cross-section Tab. Here, user can select the solid or hollow section options, and define the radius for the solid section, and inner and outer radii for the hollow section. The next tab is the Connections tab through which the user supplies data for the bushing and end connections of the anti-roll bar. For the bushings; type (x-movement free or x-movement constrained), location, length and stiffness are required parameters. Two options are provided for the connections between the bar and the suspension members; spherical joint or pin joint. Material is the last of data input tabs of the Pre-processor. Here, the mechanical properties of the bar material are input. User can either input the material properties manually or select a material from the material database of the program.

After inputting the necessary parameters using the first four tabs of Pre-processor, geometry file must be created before continuing the analysis. This step ensures that there are no missing inputs and user is aware of any change in the parameters. When user clicks the “Create Geometry File” button, which is found in the fifth tab, the parameters file required for the analysis is created. The contents of this file are explained in Section 4.2.1. Preview is the last tab in the Pre-processor, where a preview of the created model is made available. In order to create the preview ANSYS program is called and run using the preview macro as input file. The Preview tab includes bar centerline preview with keypoint numbers from different views, a list of keypoint coordinates, fillet radii and bushing locations, meshed beam model preview with bushing springs distributed to nodes and a cross section preview. Many problems can exist in the model created by ANSYS which may occur due to errors in file importing, inputting the keypoints in an incorrect sequence, defining bushing locations on curved parts of the bar etc. Obviously, an incorrect model will produce incorrect results and time will be wasted for waiting a wrong solution. Also, ANSYS program can be locked in case of fatal errors, thus program can end with an abnormal exit. In such a case, it will be required to restart the ANSYS out of the automated design software after making some changes in the configuration files. Proving the correctness of the model will be a good prevention for these problems.

The second stage of the analysis is the Solution, which contains a single tab. The analysis type -linear or non-linear-, number of elements to be used for the analysis and the displacement loads at the bar ends must be input using the corresponding fields in this tab. A command button for starting the analysis also exists in this tab. After pressing this button ANSYS is called by the interface and analysis is performed in the background using the analysis macro file.

The last analysis stage is the Post-processor. There are five tabs under this stage, and first four of these tabs are for reviewing resulting bar properties, stress/strain on the bar, fatigue life and modal properties, while the fifth tab is for generating the analysis report. In the first tab -Bar Properties tab- roll stiffness, total length and mass of the bar is presented to user. The second tab is the Stress/Strain Results tab. Maximum principal and Von-Misses stress and strain values, deformed shape of the bar, contour plots of principal and Von-Misses stress and strains, graphs showing the variation of principal and Von-Misses stress and strains along bar length are presented in this tab. The deformed shape and the contour plots can be viewed from 7 different views (isometric, oblique, front, top, bottom, left and right) in order to provide the user to observe all portions of the bar. Fatigue Life is the third tab in the Post-processor. Before calculating the fatigue life, the maximum equivalent stress on the bar and the user defined fatigue strength is reminded. User can input 3 fatigue strength modification factors for size, surface finish and corrosion and coating effects. The fatigue life calculation is performed by pressing the “Calculate” button. The Modal Analysis tab is the last of result reviewing tabs. Here, the first five natural frequencies and the corresponding mode shapes of the bar are presented. The first natural frequency usually represents the rigid body motion, and the user is warned for this fact. The modes shapes are presented as animations. ANSYS saves the animations as sequences of .png files and the animations are created by viewing these files one after one with a small time delay. The Visual Basic program does not have the capability of viewing the .png files, thus a code is embedded into the software for creating the animations.

The last step in the automated anti-roll bar analysis is the creation of the report document. Some items are restrictly included in the report while some items are optional. Input data, bar properties, maximum stress and strain values, fatigue life and natural frequencies are always included in a report. The user can also add the full geometry preview, stress/strain variation graphs, selected contour plots and mode shapes to the report. User defines a title for the analysis then presses "Generate Report" button. Script codes of the word file that contains the report is generated and the file is saved as a word document in the report folder. The report file has a cover page that contains the report title and date of the analysis. A sample analysis report is given in Appendix C.

There are two other menu items, about and help, which are not associated with analysis. Clicking the about menu item opens the form that contains information about the program. The help menu item is a link to html help file of the program. The help file is the user's manual of the program, which explains the program in detail and contains a sample analysis.

4.4 Program Features

The basic advantage of the Automated Anti-roll Bar Analysis software is the simplicity of repeating the analysis after changing any of the parameters. This gives the advantage of finding an optimum solution by trying different parameter combinations. The only thing to be done after changing a parameter is to recreate the geometry file and restart the analysis by simply pressing two buttons.

Another important feature of the program is the database of analyzed anti-roll bars, which is formed by generating a report after each analysis. This leads to an improvement in the quality of design by observing the previous analysis results available in the database.

The program also provides new, open, save and saveas operations for the Pre-processor part. By these operations data input times can be reduced since different input configurations can be saved in text files and for later use. When a previously saved file is opened all input boxes are filled and selections are made according to the saved file. File type filters are used in the file system browsers for open and saveas operations. Filter for .igs file type is also used in import file selection in geometry creation since only .igs files can be used for defining the bar centerline geometry.

Warnings for errors, guidelines about analysis and information about the program progress are presented with three ways; tooltips, message boxes, labels on the form. Some of the important points are already reminded using colored labels on the form. Information about all objects on the forms is given with tooltips which appears when the mouse is paused over an object. The user is warned against missing input parameters and incomplete tasks using message boxes. A message box also appears when ANSYS is called by the interface and runs in the background. No operations can be done at this stage until ANSYS completes its job and the message box disappears.

CHAPTER 5

SAMPLE ANALYSES AND DISCUSSION OF THE RESULTS

5.1 Verification of the Program Results

The results of the automated anti-roll bar design software will be verified in this chapter. Hand calculations will be compared with program outputs. A simple bar will be used to simplify the analysis. The model of the bar to be analyzed is given in Appendix B.3 with its dimensions. The properties of the bar and the bushings are listed in part (i) of Section 5.2.

First, the bar will be analyzed with the software to obtain the roll stiffness. Then, the roll stiffness will be calculated using the formulations suggested by SAE, for comparison.

The automated design software gives the end deflection of the anti-roll bar under a load of 1000 N as:

$f_A = 24.44\ mm$

And the roll stiffness of the anti-roll bar as:

$$k_R = \frac{1000N \cdot 1100mm}{\tan^{-1}(\frac{f_A}{550})} = 432.0 \quad Nm/deg$$

According to the SAE formulations, roll stiffness can be calculated from Eqn 2.2 as:

$$f_A = \frac{1000}{3 \cdot 206000 \cdot \dfrac{\pi \cdot 20^4}{64}} \left[250^3 - 90^3 + 550 \cdot (160)^2 + 4 \cdot 230^2 \cdot (460) \right] = 26.0 \quad mm$$

$$k_R = \frac{1000 \cdot 1100^2}{2 \cdot 26.0} = 23.26 \quad Nmm/rad \;=\; 405\ Nm/deg$$

There is % 6 difference between the two results, and this is an acceptable error. It should be noted that, SAE formulations include some approximations which can lead to errors. It's stated in the *SAE Spring Design Manual* that, the result can be affected up to %30 from the stiffness of the bushings.

Now, the bar will be analyzed with the software to obtain stress results at defined points on the bar and then classical solid mechanics calculations will be performed to calculate the stresses at these points. The first point that will be used for comparison is located between the bushing and the center of the bar, 100 mm away from the bushing and is at the top of the cross-section. The second point is the 15 mm away from the bushing in the opposite direction and again at the top of the section.

Finite element analysis is performed with a loading of 1000 N applied at bar ends in opposite directions. In order to simplify the hand calculations, translational degrees of freedom of the nodes are constrained within the bushing. The query picker of ANSYS program is used for determining the stresses at the defined points.

For the first point, the program outputs the shear stress as 153.5 MPa. This stress is caused due to torsion on the bar. With hand calculations [30]:

$\tau = T \cdot c / J = 1000 \cdot 230 \cdot 10 / (\pi \cdot 20^4 / 32) = 146.4$ MPa

For the second point, the axial stress (σ_x) is 193.7 MPa. This stress is caused due to bending at this section. With hand calculations [30]:

$\sigma = M \cdot c / I = 1000 \cdot 145 \cdot 10 / (\pi \cdot 20^4 / 64) = 184.6$ MPa

The results show that the stresses obtained from the FEA and hand calculations are close to each other with a difference about %5, and this difference is acceptable.

5.2 Sample Analyses

i) Sample Analysis with a Typical Anti-roll Bar

The first sample analysis is performed with a solid round anti-roll bar. Moment-free bushings and spherical joints at bar ends are used for connections. The results obtained in this analysis will be used for demonstrating the program outputs. Also, the same anti-roll bar geometry will be used with different connection types and parameters in other sample analysis in order to discuss the effects of the design parameters on anti-roll bar performance. The units used in all graphs, plots and tables are in terms of N, mm and MPa.

Inputs:

The preview of the geometry to be analyzed is given in Appendix B.3. This geometry is created in ANSYS, and the keypoint coordinates and fillet radii are given below the figure. (Note: When same model is imported from an igs file, ANSYS assigns different keypoint numbers and possibly at different locations)

After generating the model, other design parameters are assigned as follows:

Cross-section type	= Solid round cross-section		
Section radius	= 10 mm		
Bushing type	= 1 (x movement free)		
Bushing locations	= ± 390 mm		
Bushing length	= 40 mm		
Bushing Stiffness	= 1500 N/mm		
End connection type	= 1 (spherical joint)		
Bar material	= SAE 5160	E	= 206 GPa

υ = 0.27

S_y = 1180 MPa

S_{uts} = 1400 MPa

S_E = 500 MPa

ρ = 7800 kg/m^3

Number of elements = 100

Loading = ± 50 mm on both sides

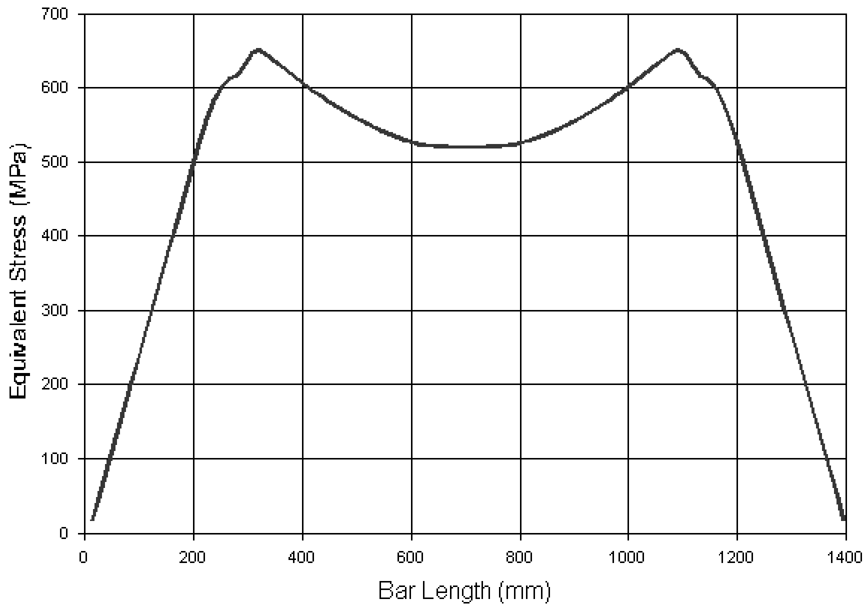
<u>Results</u>

The results obtained from the analysis of this bar are:

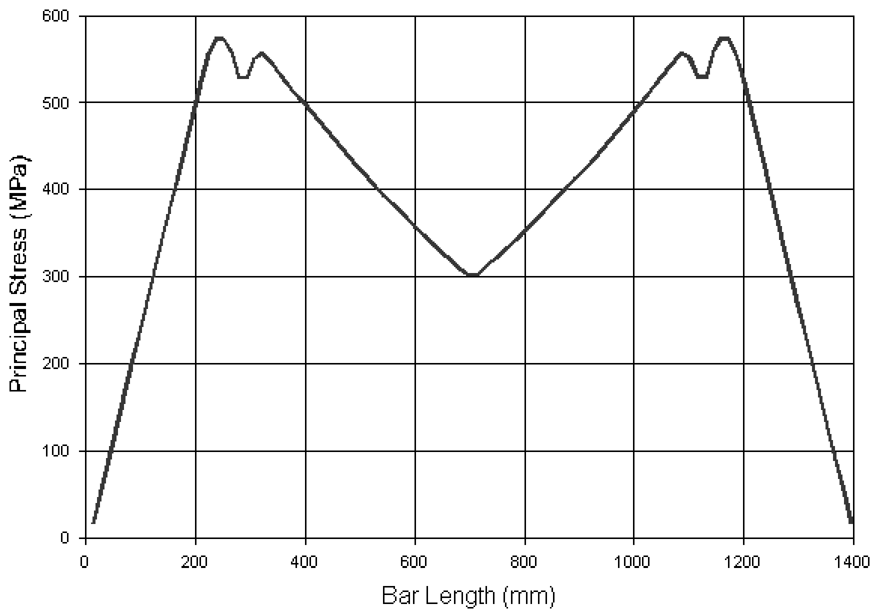
Roll Stiffness	= 410.2 Nm/deg
Total Length	= 1394 mm
Mass	= 3.416 kg
Max. Prin. Stress	= 578.9 MPa
Max. Eqv. Stress	= 652.1 MPa
Max. Prin. Strain	= 0.304 %
Max. Eqv. Strain	= 0.372 %
Fatigue Life	= 139600 cycles (modification factors are taken as 1 except size factor, which is calculated automatically by the program)
Lowest Natural Freq.	= 71.1 Hz

The variation of equivalent and principal stresses along bar length are plotted in Figures 5.1 and 5.2.

The contour plot of equivalent stress on the bar is given in Figure 5.3, and the mode shape of the lowest natural frequency is presented in Figure 5.4.



<u>Figure 5.1</u> - Variation of Equivalent stress along bar length



<u>Figure 5.2</u> - Variation of Principal Stress along bar length

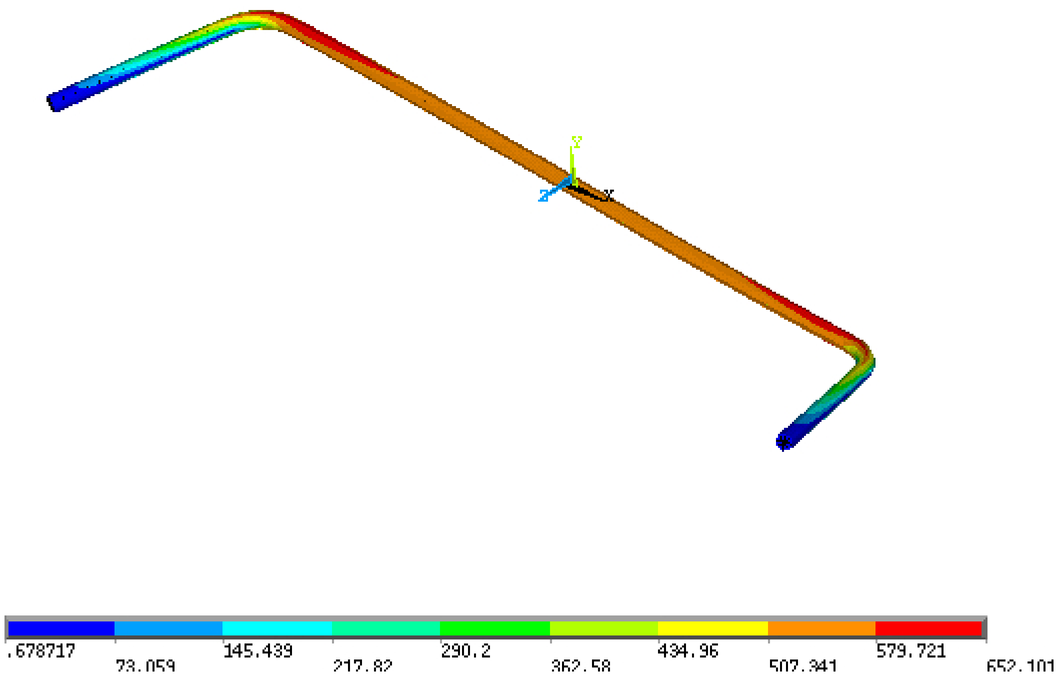


Figure 5.3 - Equivalent Stress Distribution on the Bar

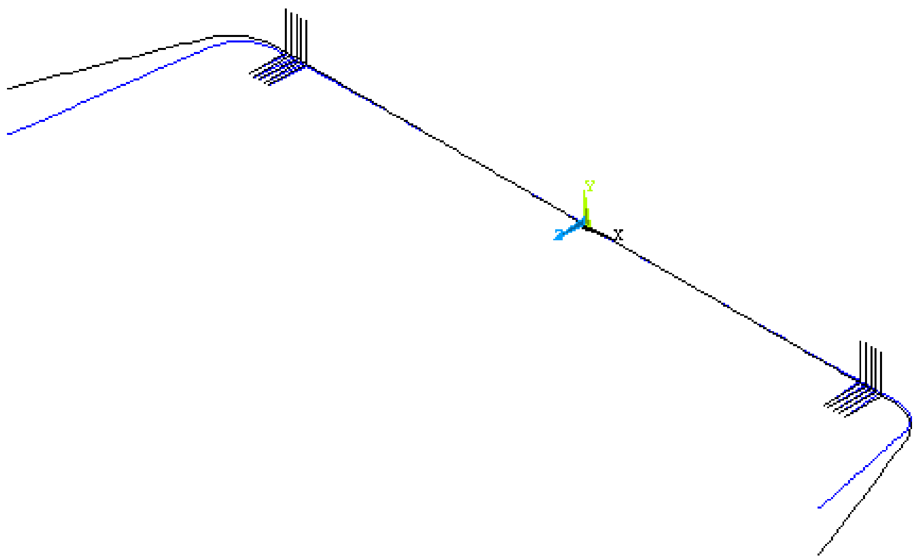


Figure 5.4 - Mode Shape of the First Natural Frequency

ii) Effects of Bar Cross-Section Type and Dimensions:

In this analysis the effect of the bar cross-section on the bar properties will be presented. The primary parameter to be considered for an anti-roll bar is the roll stiffness. Therefore, the anti-roll bar stiffness obtained in part (i), will be obtained with a hollow bar and the other analysis results will be compared.

- **Case1 -** Hollow Cross-Section vs. Solid Cross-Section

Inputs:

All inputs are same as part (i) except cross-section properties. After some iterations the following section dimensions are determined in order to obtain specified roll stiffness:

Cross-section type = Hollow cross-section
Section inner radius = 8 mm
Section outer radius = 10.9 mm

Results:

Following results are obtained from the analysis of the hollow bar:

Mass = 1.872 kg
Max. Prin. Stress = 606 MPa
Max. Eqv. Stress = 683 MPa
Max. Prin. Strain = 0.319 %
Max. Eqv. Strain = 0.390 %
Fatigue Life = 93595 cycles
Natural Freq. = 95.94 Hz.

As seen from Figure 5.5 variation of equivalent stress along bar length is same as part (i), except the values of the peaks. Same situation is valid for principal stress variation along bar length. The contour plot of equivalent stress shown in Figure 5.6 is also similar to part (i).

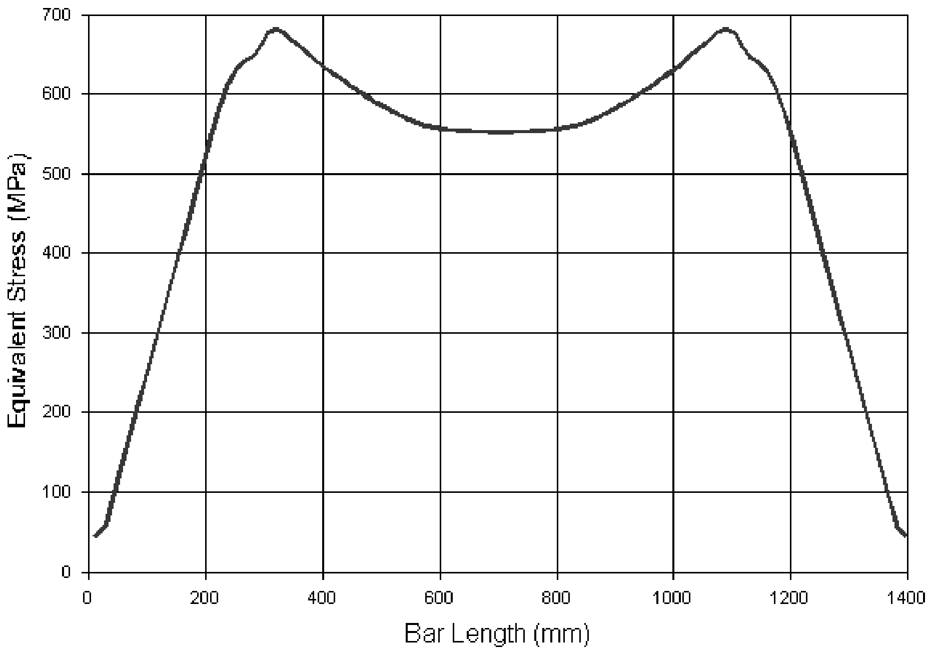


Figure 5.5 - Variation of Equivalent stress along bar length

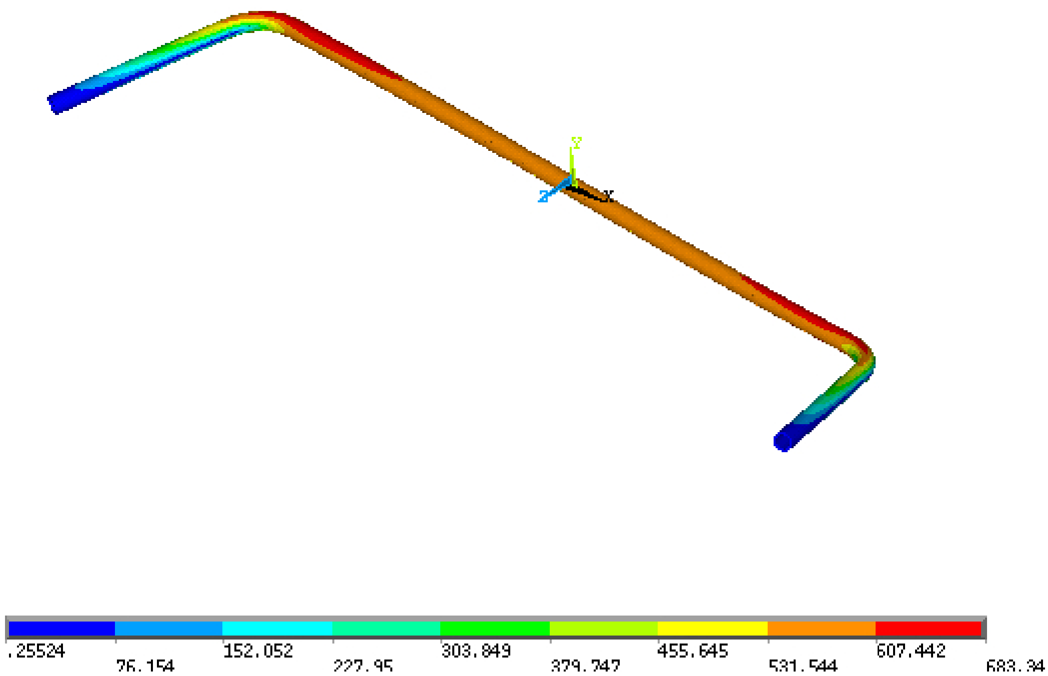


Figure 5.6 - Equivalent Stress Distribution on the Bar

• **Case2** – Increasing the Diameter of Solid Cross-Section

<u>Inputs:</u>

All inputs are same as part (i) except cross-section dimension.

Section radius = 12 mm

<u>Results:</u>

Following results are obtained from the analysis of the hollow bar:

Roll Stiffness = 805 Nm/deg
Mass = 4.919 kg
Max. Prin. Stress = 657 MPa
Max. Eqv. Stress = 741 MPa
Max. Prin. Strain = 0.346 %
Max. Eqv. Strain = 0.424 %
Fatigue Life = 48736 cycles
Natural Freq. = 80.8 Hz.

The variations of the stresses on the bar are same as part (i) except the peak values.

iii) Effects of Bushing Type and Parameters:

In this part, anti-roll bar performance will be investigated for different bushing type, stiffness and lengths.

The first parameter to be analyzed is the stiffness of bushing material. To demonstrate the effect of bushing stiffness, the bar will be analyzed with a higher bushing stiffness compared to part (i).

• **Case1-** Increasing Bushing Stiffness

Inputs:

All inputs are same as part(i) except bushing stiffness.

Bushing stiffness = 5000 N/mm

The maximum relative displacement of the bar ends used in part (i) is assumed to occur at the maximum permissible body roll angle, which is about 3.5°. Thus, the loading will be again ±50 mm although the roll stiffness will be changed.

Results:

Roll Stiffness	= 426.4	Nm/deg
Max. Prin. Stress	= 601.8	MPa
Max. Eqv. Stress	= 674.6	MPa
Max. Prin. Strain	= 0.314	%
Max. Eqv. Strain	= 0.386	%
Fatigue Life	= 93595	cycles
Lowest Natural Freq.	= 91.27	Hz

The stress distribution is again similar to part (i).

• **Case 2 -** Changing Bushing Type

Changing bushing type will create difference if the bar moves within the bushing along bushing axis. The bar that has been analyzed in the previous parts does not move along bar axis too much, due to its geometry. For this reason, the bar shown in Appendix B.1 will be analyzed for evaluating the effect of the bushing type.

Inputs:

The geometry details of the bar are given in Appendix B.1. All other properties are used as they are in part (i).

The analysis is first performed with bushing type 1:

Results:

Roll Stiffness	= 945.2 Nm/deg
Max. Prin. Stress	= 457.1 MPa
Max. Eqv. Stress	= 537.9 MPa
Lowest Nat. Freq.	= 40.38 Hz

The analysis is then performed with bushing type 2:

Results:

It should be noted that, the method used of preventing the movement of the anti-roll bar along bushing axis may cause stress concentrations at the bushing positions.

Roll Stiffness	= 997.6 Nm/deg
Max. Prin. Stress	= 474.1 MPa
Max. Eqv. Stress	= 538.8 MPa
Lowest Nat. Freq.	= 48.45 Hz

The stress variations along bar length and contour plots are given in Figures 5.7 to 5.10.

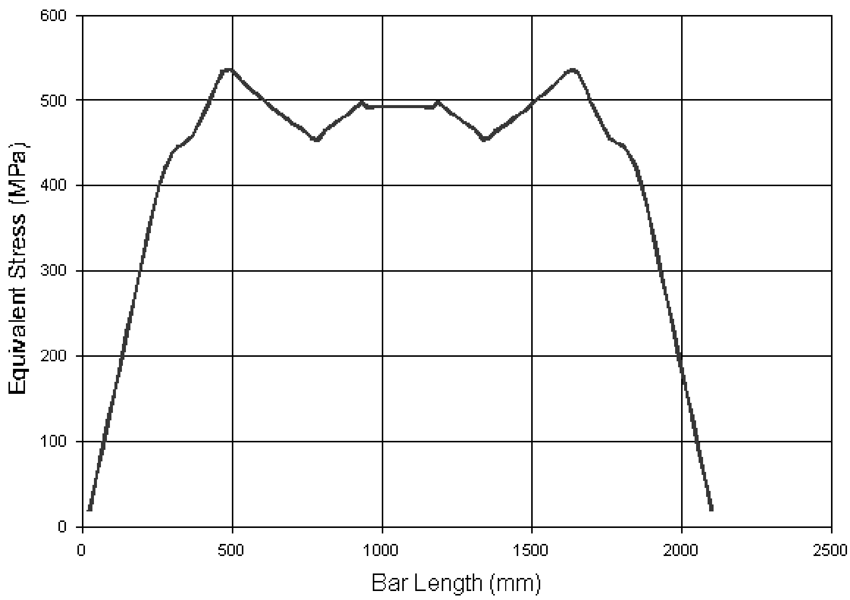


Figure 5.7 - Variation of Equivalent stress along bar length – Bushing type 1

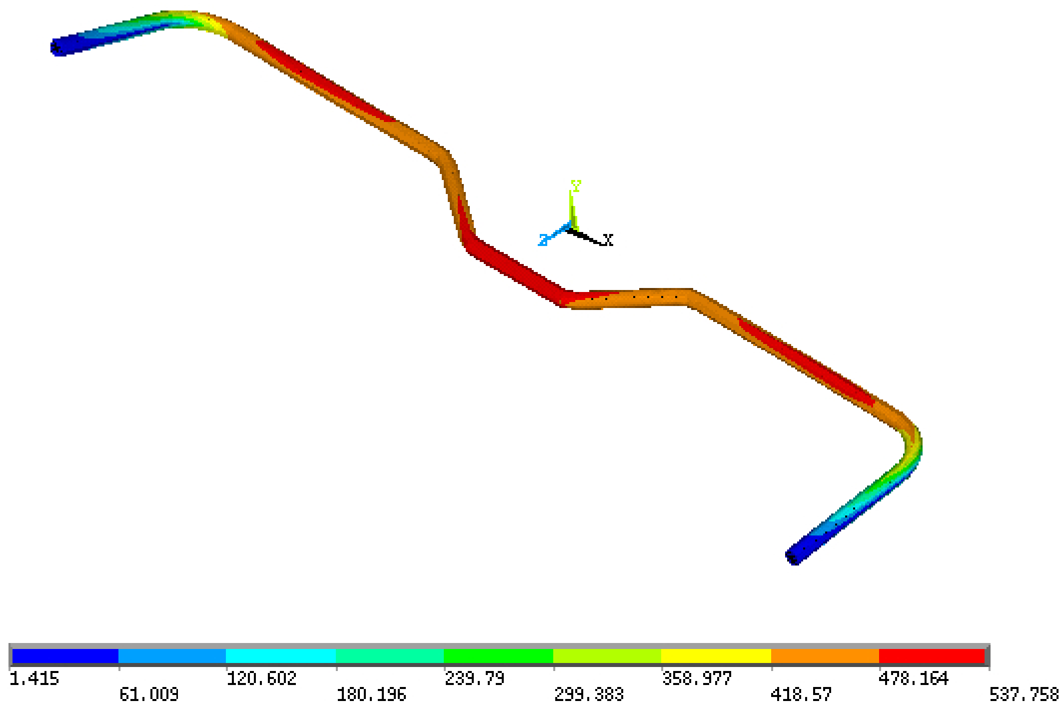


Figure 5.8 – Equivalent Stress Distribution on the Bar- Bushing type 1

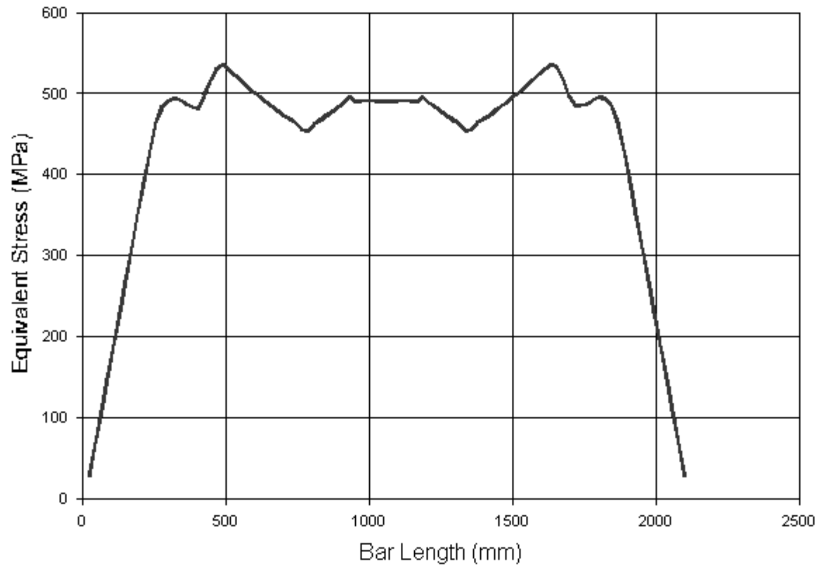


Figure 5.9 - Variation of Equivalent stress along bar length - Bushing type 2

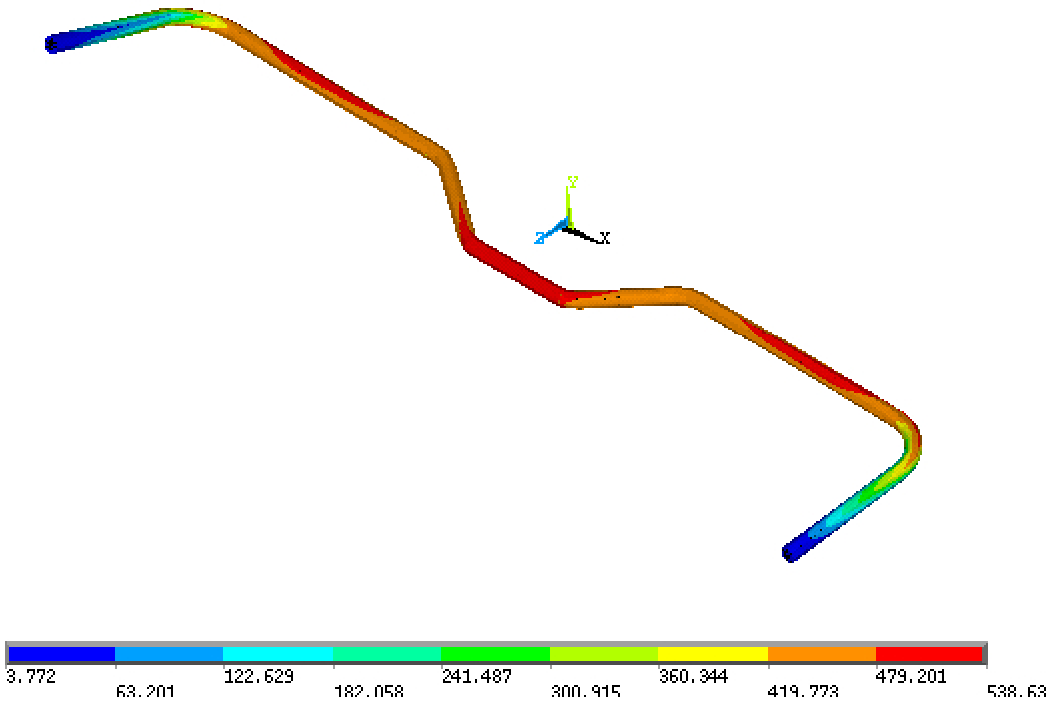


Figure 5.10 – Equivalent Stress Distribution on the Bar - Bushing type 2

- **Case 3 -** Changing Bushing Locations

Anti-roll bars may have various shapes since their geometry depends on the availability of space in the chassis. Also, bushings can be located at different positions. Normally bushings are fitted near bend portions as close as possible. The effect of bushing locations will be now analyzed by using bushings closer to center compared to part(i).

Inputs:

All inputs are same as part(i) except bushing locations.

Bushing locations = ± 300 mm on both sides

Results:

Roll Stiffness = 342.8 Nm/deg

Max. Prin. Stress = 622.6 MPa

Max. Eqv. Stress = 678.0 MPa

Natural Freq. = 64.0 Hz

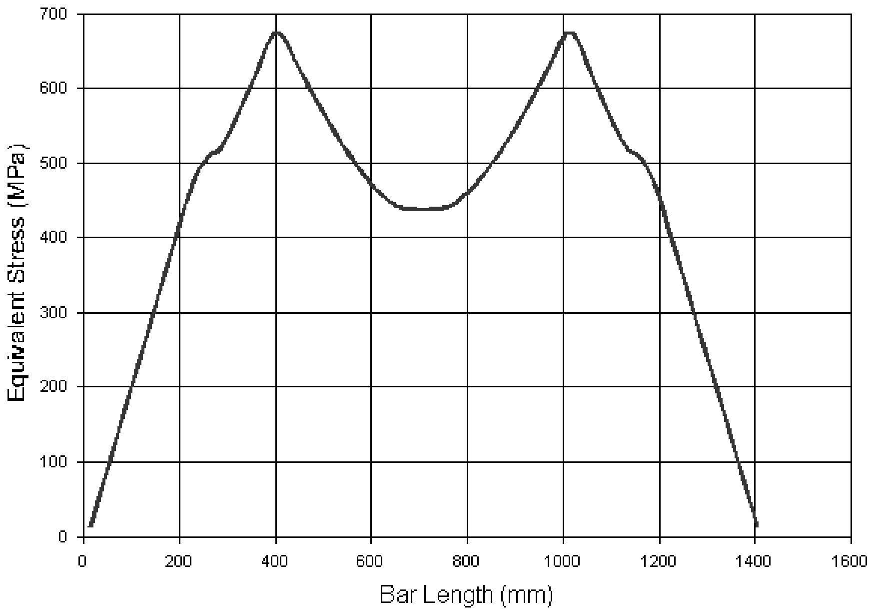


Figure 5.11 - Variation of Equivalent stress along bar length

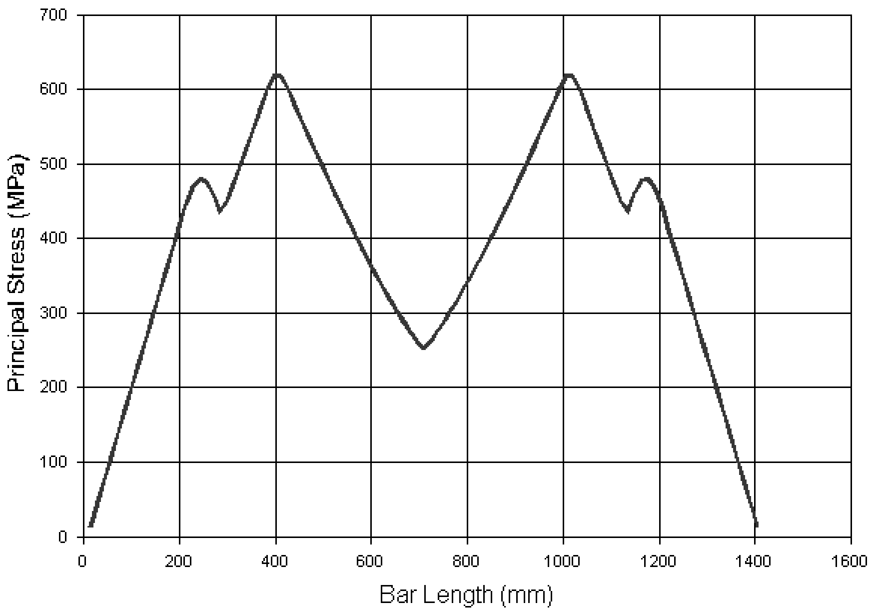


Figure 5.12 - Variation of Principal Stress along bar length

The equivalent stress distribution on the bar is given in Figure 5.13.

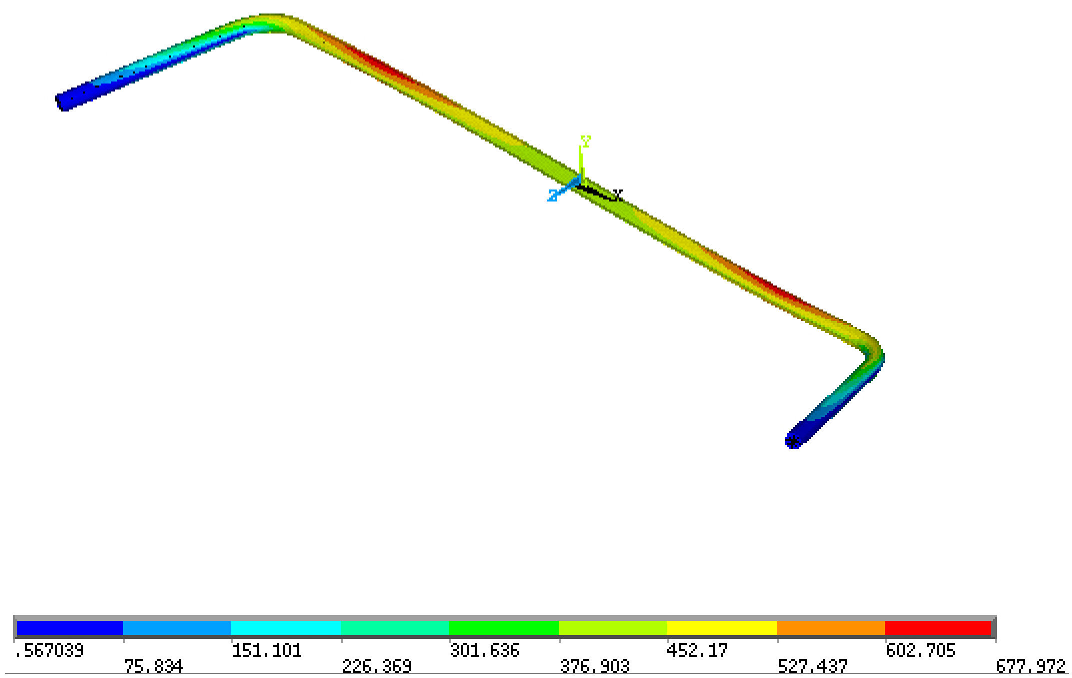


Figure 5.13 - Equivalent Stress Distribution on the Bar

iv) Effect of end Connection Type:

There are two joint types used for connecting anti-roll bar to the suspension members; spherical joints and pin joints. In part(i) spherical joints were used for the providing end connections. Now, pin joints will be used and analysis results will be compared with part(i).

Inputs:

End Connection Type = 2 (pin joint)

Results:

Roll Stiffness = 449.4 Nm/deg

Max. Prin. Stress = 528.0 MPa

Max. Eqv. Stress = 600.8 MPa

Max. Prin. Strain = 0.273 %

Max. Eqv. Strain = 0.362 %

Natural Freq. = 71.05 Hz

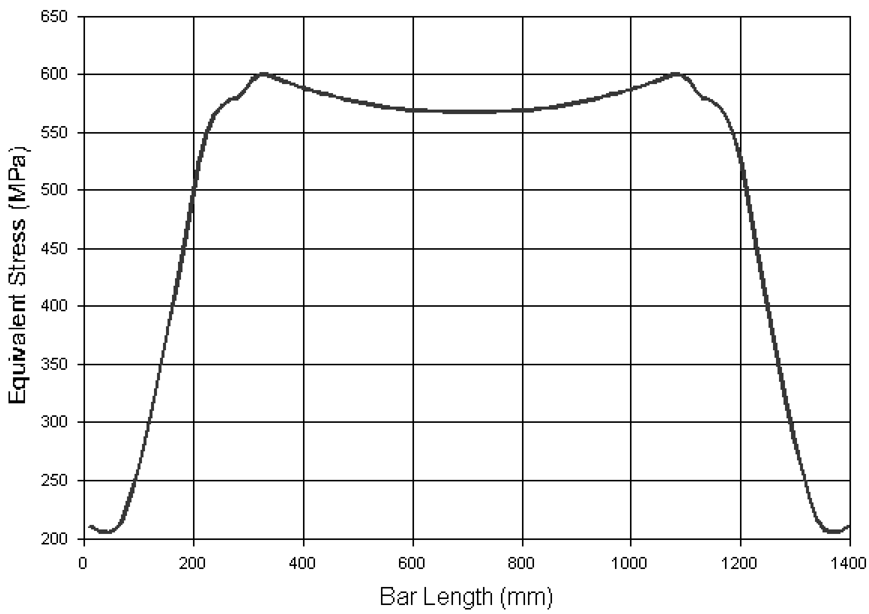


Figure 5.14 - Variation of Equivalent stress along bar length

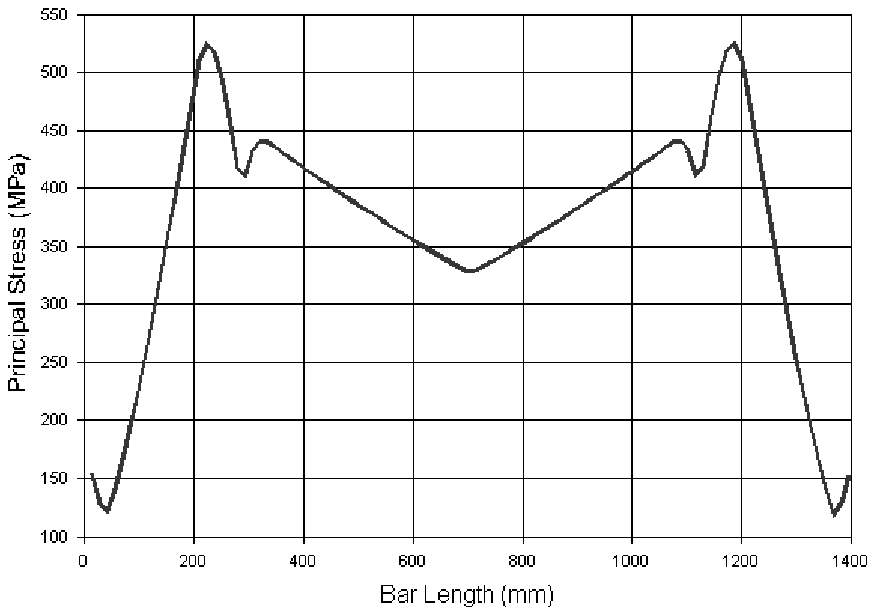


Figure 5.15 - Variation of Principal Stress along bar length:

The equivalent stress distribution on the bar is given in Figure 5.16.

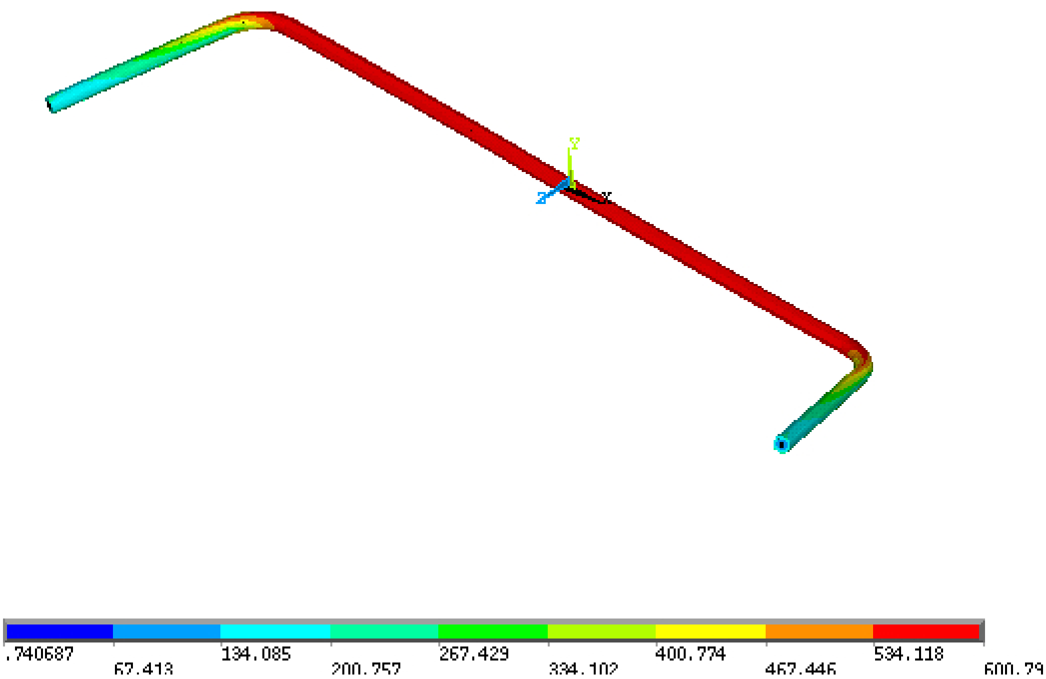


Figure 5.16 - Equivalent Stress Distribution on the Bar

v) Material Properties:

It's clear that using a material with high stiffness increases the stiffness of anti-roll bar. Also, high stress values can be handled with high strength materials. To demonstrate the effects of the bar material, a titanium bar will be analyzed compared with part(i). For this purpose, the diameter of a titanium bar having a roll stiffness equivalent to the bar of part(i) will be determined.

Inputs:

All inputs are same as part(i) except material properties and cross section.

Material = Ti-6Al-4V Bar STA

$E = 110$ Gpa

$\upsilon = 0.3$

$E_{yield} = 1100$ MPa

$E_{uts} = 1185$ MPa

S_E = 895 MPa (smooth surface)

ρ = 4.43 kg/m^3

Section Radius = 11.8 mm

Results:

Mass = 2.7 kg

Max. Prin. Stress = 357.9 MPa

Max. Eqv. Stress = 403.2 MPa

Max. Prin. Strain = 0.357 %

Max. Eqv. Strain = 0.439 %

Natural Freq. = 80.34 Hz

Stress-Strain variations are same as part (i) except the peak values.

5.3 Discussion of the Results

With the help of the analysis results obtained in the previous section, some conclusions can be derived for the effects of design parameters on the resulting bar properties.

In sample analysis (i), the basic characteristics of an anti-roll bar are obtained. The bending stress on the bar, starts from 0 and increases up to its maximum value at the bend portion or bushing positions, and then reduces again between the bushings. The torsional shear stress is maximum and constant between the bushings. With the combination of these stresses, the graphs in Figures 5.1 and 5.2 are obtained. These graphs are symmetric with respect to the center of the bar and there are two peaks at each side of the graphs (more obvious in figure 5.2). The peaks correspond to bend part of the bar and the bushing locations. Thus, the maximum stresses on the bar will occur at one of these locations.

The natural frequencies of the anti-roll bars are above the critical limit of forming resonance with wheel hop frequency of the vehicles which is around 10 Hz.

The most obvious effect of using hollow section is the reduction in mass of the bar. However, maximum stresses on the bar have increased, while the variation along bar centerline remained same as part (i). Almost 50% mass reduction is provided in excess of 5% increase in the stresses.

Increasing the radius of the bar from 10 mm to 12 mm doubles the anti-roll stiffness. However, the stresses on the bar are increased by 10% and this bar is 40% heavier than the previous one. Another negative effect of increasing the bar diameter is the reduction in the fatigue life. The Endurance limit modifying factor was about 1.00 for 20 mm diameter but it decreased to 0.98 for 24 mm diameter.

Increasing the bushing stiffness obviously improves the anti-roll stiffness by resisting deformations within the bushings. This method seems to be the easiest way of obtaining higher roll stiffness without changing the anti-roll bar. The increased stresses should be again considered.

The two different bushing types lead to different anti-roll bar properties. If the movement of the anti-roll bar along the bushing axis is considerably high than the using a bushing that constrains this movement will improve roll stiffness. However, an increase in the stresses will accompany this improvement. The variation of the stress along bar length is also affected from the change in the bushing type. From Figures 5.7 to 5.10, it’s observed that the bend portion of the bar becomes more stressed relative to the bushing locations by using second bushing type. One important fact about the stresses at bushing locations is the stress concentration effect which may occur due to method that is utilized for constraining x movement. Since the program does not account for the details of the constraining the x movement the user must be aware of the problems that will be carried with x movement constraining method.

Bushing location is another parameter of anti-roll bar design. By placing the bushings closer to the center of the bar, the stress at the bushing becomes higher than the stresses at the bend portion. This situation can be observed from Figures 5.11 and 5.12 where the peaks at bushing position are much higher than the peaks at the bends. Roll stiffness of the bar decreased while the max stresses increased. Thus, the overall effect of using bushing far away from the bends is completely negative and should be avoided.

Changing the end fixture type especially has an effect on the stress distribution near bar ends. The stresses near the ends are increased for the pin joint connection due to additional rotational constraints associated with this joint type. The roll stiffness of the bar is increased with reduced maximum stresses. Stress is more equally distributed along the length of the bar. This is absolutely the desired combination of results. However, the pin connection details become important as well the direction of motion of the suspension members, which is not always absolutely vertical.

The use of materials other than steels for anti-roll bar production can be discussed from many aspects. Various materials can be used for manufacturing anti-roll bars. The primary goals of using a different material may be lower weight, lower cost and longer fatigue life. In last part of the sample analysis a titanium alloy is used as the bar material. The required roll stiffness is obtained with lower bar mass and lower maximum stresses. If a compromise solution can be found for the raw material and production cost of the titanium, its use for anti-roll bar production would be a good choice. No comments are made about the fatigue life of the titanium bar since the fatigue life prediction procedures employed in the program are valid for steels.

CHAPTER 6

CONCLUSIONS AND RECOMMENDATIONS FOR FUTURE WORK

6.1 Summary and Conclusions

In this study, a software is developed for automating the design analysis of vehicle anti-roll bars. The aim of the development of this software was to present a method for effective use of general purpose FEA package programs in the design analysis. The software developed in the study is then used for evaluating the effects of design parameters on anti-roll bar characteristics.

The presented work provides a general method for automating FEA. The procedures applied for automating the anti-roll bar design formed general guidelines for design automation of other machine components.

The design automation study started with defining the analysis steps, which requires a throughout understanding of the problem. The design inputs and outputs are determined according to the requirements. Then, the finite element analysis steps are standardized considering different input combinations. The next procedure is to prepare the codes of the finite element analysis macro that performs the standardized analysis steps. Finally the user interface is developed for data input and result output operations.

By completing this study, the basic program algorithms and codes for performing the design automation tasks are obtained. These codes can be used in other studies with small changes according to the properties of the designed object. As an example, providing the interaction between finite element analysis software and the user interface was one of the greatest problems faced in this study. However, once it is managed, the only thing to be done in the further studies is to copy the program code that establishes the interaction. Some other programming capabilities acquired during the study are; capturing images from a non-gui ANSYS session, viewing the file types output by ANSYS on the user interface, reading data from the script type of lists created by ANSYS, etc..

The software developed in this study is successful considering the aim of the study. First of all, the software can handle a wide group of anti-roll bars with different parameters. The analysis time is short and can be repeated simply after changing any of the input parameters which provides an easy way to find an optimum solution for anti-roll bar design. The provided analysis results cover a complete set of design aspects including resulting bar properties, stress/strain distributions on the bar, fatigue life, natural frequencies and mode shapes. Also, the user is not interacted with the FEA software during the analysis and help is provided if decisions are to be made by the user. A ready to print report can be generated at the end of the analysis and the report content can be determined by the user.

The important result of the report files created at the end of each study is the database of different anti-roll bar designs. This database can be searched by the user before starting a design job, thus makes it easier to reach the solution in shorter times. The more important effect of the database is the improvement of the design quality.

Anti-roll bars are tunable vehicle components which have direct effect on the vehicle's performance. Bar properties can be changed by changing the parameters that define the bar. Thus, the study is completed by using the developed software for performing sample anti-roll bar analysis and the effects of design parameters on anti-roll bar characteristics are discussed using the results of these analysis. Following conclusions are derived about anti-roll bar design parameters:

- Increasing the cross-sectional diameter of an anti-roll bar will increase its roll stiffness. But larger stresses occur on the bar for the same bar end deflection. The size factor used for endurance limit modification is also affected from the diameter of the bar.

- The weight of the hollow anti-roll bar is less than the solid bar having the same roll stiffness. However, the stresses on the hollow bar are higher. The size factor is also adversely affected from the outer diameter of the anti-roll bar.

- Increasing the bushing stiffness increases the anti-roll stiffness. The stresses are again increased.

- Constraining the x movement of the within the bushing increases the roll stiffness if the amount of this displacement is high in the unconstrained case.

- Locating the bushings closer to the center of the bar increases the stresses at the bushing locations while roll stiffness of the bar decreases.

- If the pin joints are used at the bar ends, the stresses near the ends are increased. The roll stiffness of the bar is increased while the maximum stresses are decreased due to distribution of the stresses along the length of the bar.

- Required roll stiffness can be obtained with a lower weighting bar by changing the bar material.

6.2 Recommendations for Future Work

The following recommendations can be made for future studies:

- In order to obtain desired anti-roll bar properties, the automated design software must be run with trial parameters. Optimization can be utilized for finding the value of an input parameter (bar diameter) to obtain a specified property (roll stiffness) while keeping the other input parameters same.

- In some situations zooming may be very useful in post-processing the results. Thus, zooming capability can be made available in the post-processor.

- Although their use is not common, the analysis of bars with variable cross-section can be added to program capabilities.

- A database of bushing materials with material properties can be made available to user.

- Fatigue analysis can be performed by using real road data.

- Automated design software can be developed for other machine components.

- The use of the software via internet can be provided.

REFERENCES

[1] Somnay, R., Shih, S., **“Product Development Support with Integrated Simulation Modeling”**, SAE Technical Paper Series, Paper No:1999-01-2812, 1999

[2] Shih, S., Bennett, J., Baldwin, S., Basas, J., Somnay, R., **“Product Development Cycle Time Reduction with FEA – A New Consideration”**, SAE Technical Paper Series, Paper No: 982805, 1998

[3] Orr, J., **“Friendlier Finite Element Modeling”,** Computer-Aided Engineeering, , v.16, p.54, October 1997

[4] The Brueau of European Designers’ Association, **“BEDA Communication Series – Report 9”**, http://www.beda.org/pdf/BEDA_Report_09.11.00.pdf , 2000

[5] Shih, S., Kuan S., Somnay, R., **“Effective FEA for Product Development Support”**, SAE Technical Paper Series, Paper No: 2002-01-3124, 2002

[6] Microsoft VISUAL BASIC 6.0, Retail 6.0.8169, Microsoft Corp.

[7] ANSYS, Release 7.0, October 12, 2002, SAS IP

[8] Ünlüsoy, Y. S., “**ME 436 Automotive Engineering Lecture Notes**”, METU, 2000.

[9] Comesky, J., **“http://www.grmotorsports.com/swaybars.html”**, 2002

[10] Domingues, R., **“http://www.tokicogasshocks.com/suspension/components.html”**, 2002

[11] SAE Spring Committee, **“Spring Design Manual”**, 2nd Ed., SAE, pp. 215-267, 1996

[12] Saxon, J., Beaulieu, C., **“Web Site Streamlines Analysis”**, ANSYS Solutions, Vol. 4, Number 1, p. 7-9, 2001

[13] Thi, T., V., **“Automated Torsion Bar Design for Vehicle Suspension System”**, MS Thesis, Department of Mechanical Engineering, University of Louisville, 1999

[14] Gummadi, L. N. B., Cai, H., Lin, S., Fan, X., Cao, K. **“Bushing Characteristics of Stabilizer Bars”**, SAE Technical Paper Series, Paper No: 2003-01-0239, 2003

[15] Palma, E. S., Santos, E. S., **“Cumulative Fatigue Damage in an Automobile Stabilizer Bar: Correlation between Laboratory and Road Experiments”**, SAE Technical Paper Series, Paper No: 2001-01-3175, 2001

[16] Thomée, V., **“From finite differences to finite elements; A short history of numerical analysis of partial differential equations ”,** Journal of Computational and Applied Mechanics, Vol. 128, pp. 1-54, 2001

[17] M. Barton , S. D. Rajan, **“Finite Element Primer for Engineers”**, http://ceaspub.eas.asu.edu/structures/FiniteElementAnalysis.htm , 2000

[18] Abd El-Ghany, K. M., Farag, M. M., **”Expert System To Automate The Finite Element Analysis For Non-Destructive Testing “,** NDT & E international, pp. 259-267, 2001

[19] Padhi, G.S., McCarthy, M.A., McCarthy, C. T. **“BOLJAT: A Tool For Designing Composite Bolted Joints Using Three-Dimensional Finite Element Analysis”**, Composites, Part A 33, pp. 1573–1584, 2002

[20] İlhan, K. A., **“Computational Design of Composite Tubes and Pressure Vessels”**, METU M.S. Thesis, September 2001

[21] Ç. Alagöz, **“Finite Element Analysis of Long Fiber Reinforced Composite Spur Gears”**, METU M.S. Thesis, September 2001

[22] DP Technology Corp, **“CAM Reduces Extrusion Die Programming Cycle”**, Aluminium International Today, Vol. 14, Issue 3, May 2002

[23] Tang, J., Ogarevic, V., Tsai, C. S., **“An Integrated CAE Environment For Simulation-Based Durability And Reliability Design”**, Advances in Engineering Software, Vol. 32, pp. 1-14, 2001

[24] Mauro Marinilli, **“The Theory Behind User Interface Design”**, http://www.developer.com/design/article.php/1545991, 2003

[25] Kennedy, G., “**Organisational Database Systems, User Interface Design**”, University of Sydney, 2003

[26] VISUAL BASIC Help for Version 6.0

[27] ANSYS Help for Version 7.0

[28] J. E. Shigley, C.R. Mischke, **“Mechanical Engineering Design”**, 5th Ed. , McGraw-Hill, pp. 282-289, 1989

[29] SAE Fatigue Design and Evaluation Committee, **"SAE Fatigue Design Handbook"**, 3rd Ed., SAE, p.82-91, 1997

[30] F. P. Beer, E. R. Johnston, **"Mechanics of Materials"**, 2nd Ed., McGraw Hill, p.121, p.191, 1992

[31] Mischke, C. R., **"Prediction of Stochastic Endurance Strength"**, Trans. of ASME, J. Vibration Acoustics, Stress, and Reliability in Design, vol. 109, pp. 113-122, January 1987

[32] Turner, M. J., Clough, R. W., Martin, H. C. and Topp, L., **"Stiffness and deflection analysis of complex structures"**, J. Aero. Sci. 23, pp. 805–823, 1956

[33] Clough, R. W., **"The finite element method in plane stress analysis"** Proceedings of Second ASCE Conference on Electronic Computation, Vol. 8, Pittsburg, Pennsylvania pp. 345–378, 1960

APPENDIX

APPENDIX A- USER INTERFACE

APPENDIX B - ANTI-ROLL BAR MODELS

APPENDIX C - SAMPLE ANALYSIS REPORT

APPENDIX A

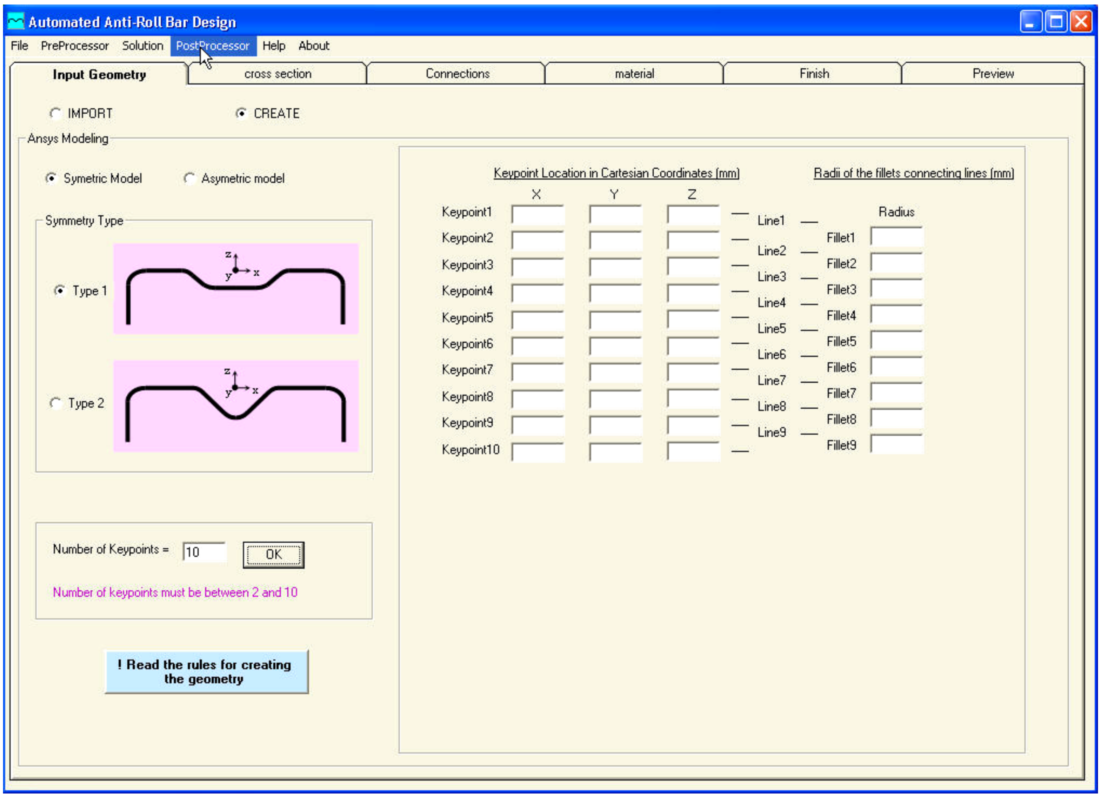


Figure A.1 – Pre-processor – Input Geometry Tab – Create Option

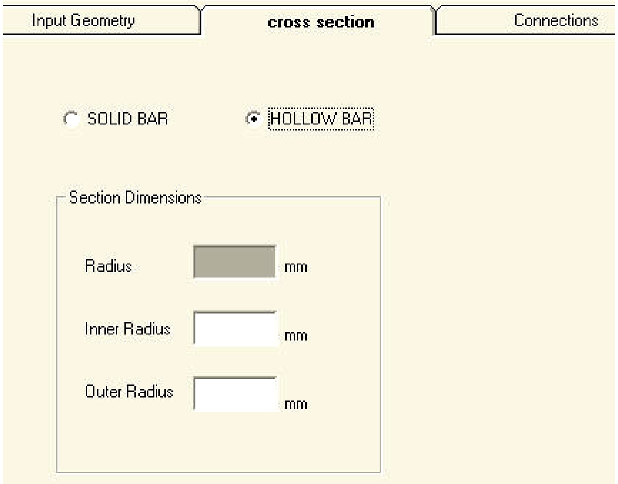


Figure A.2 – Pre-processor – Cross Section Tab

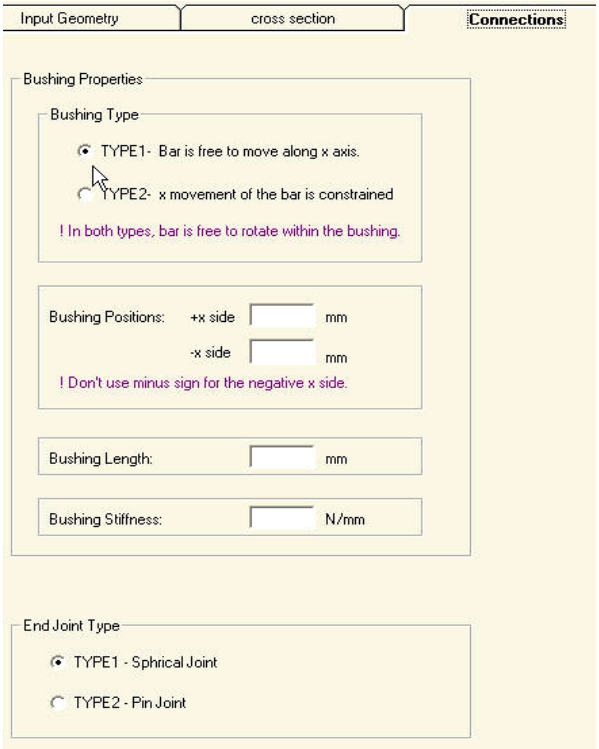


Figure A.3 – Pre-processor – Connections Tab

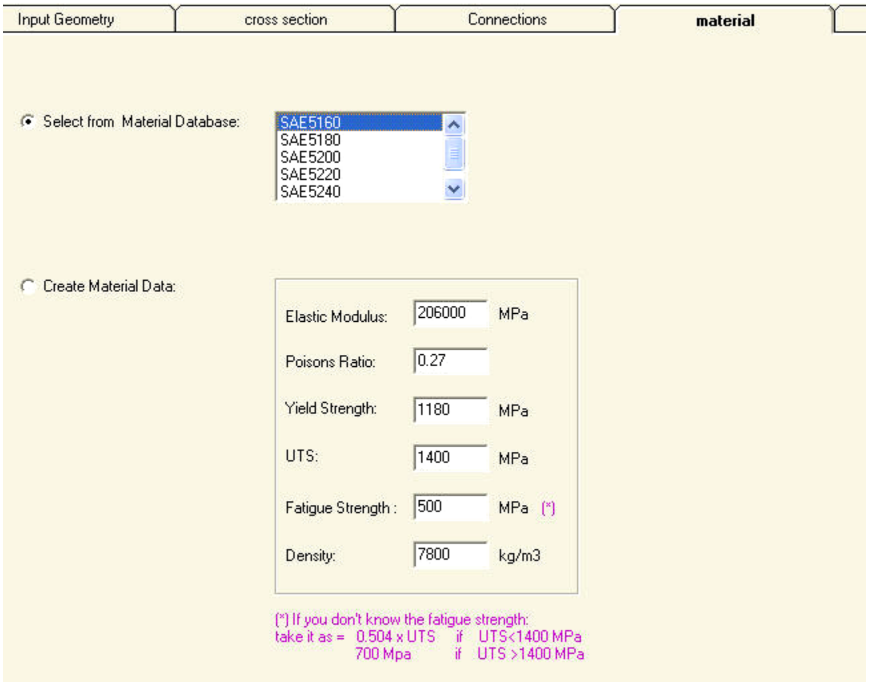


Figure A.4 – Pre-processor – Material Tab

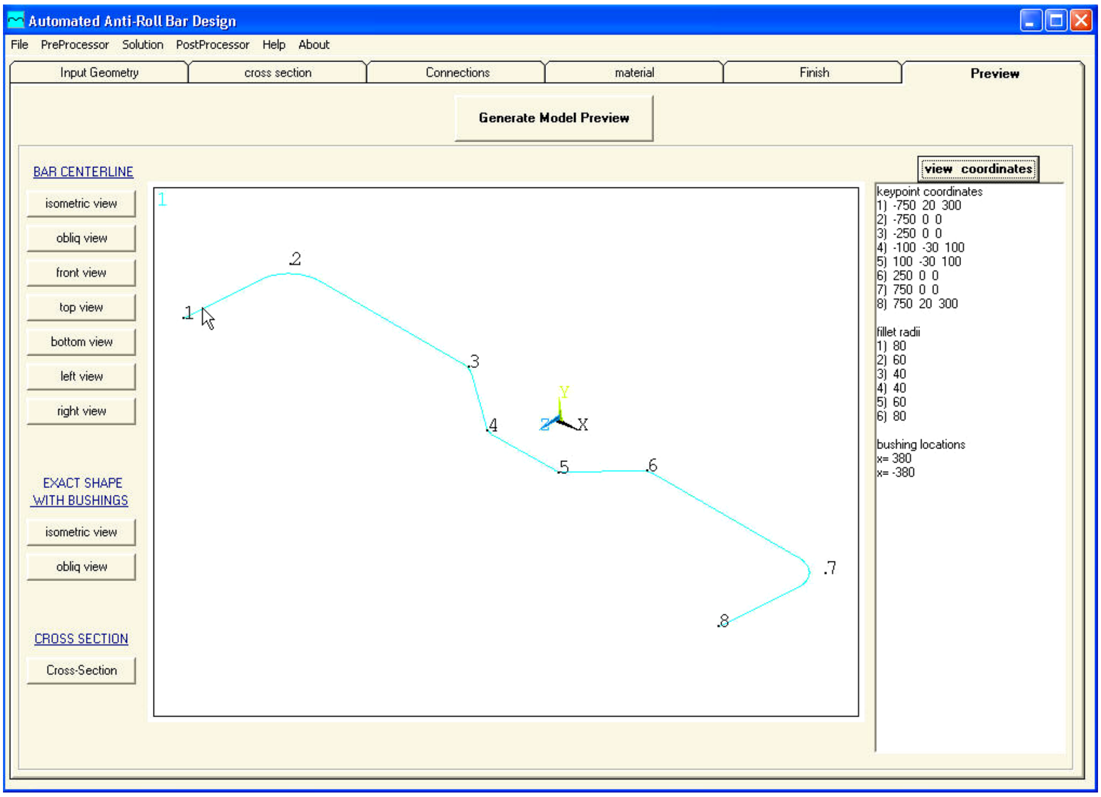


Figure A.5 – Pre-processor – Preview Tab

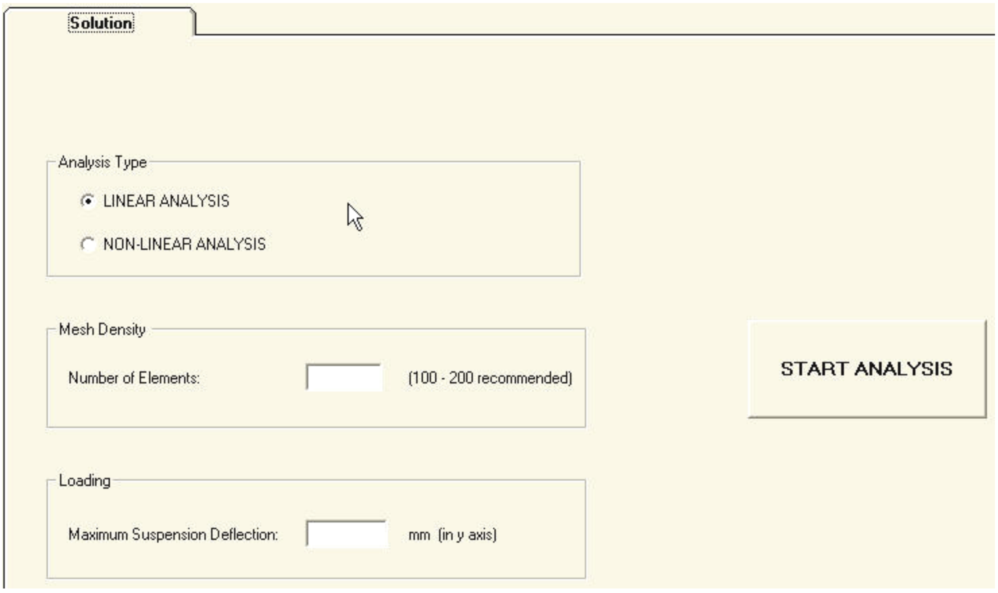


Figure A.6 – Solution

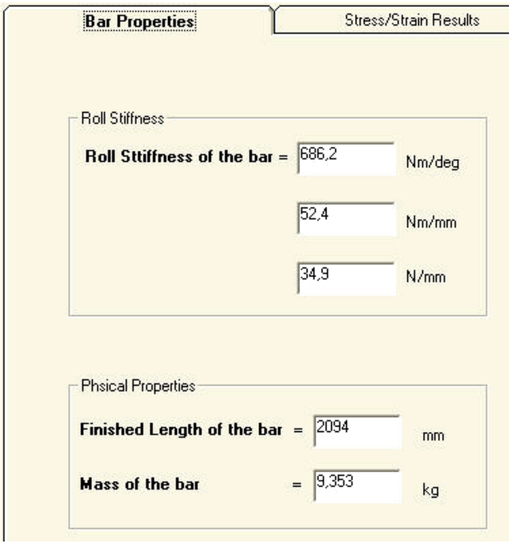


Figure A.7 – Post-processor – Bar Properties Tab

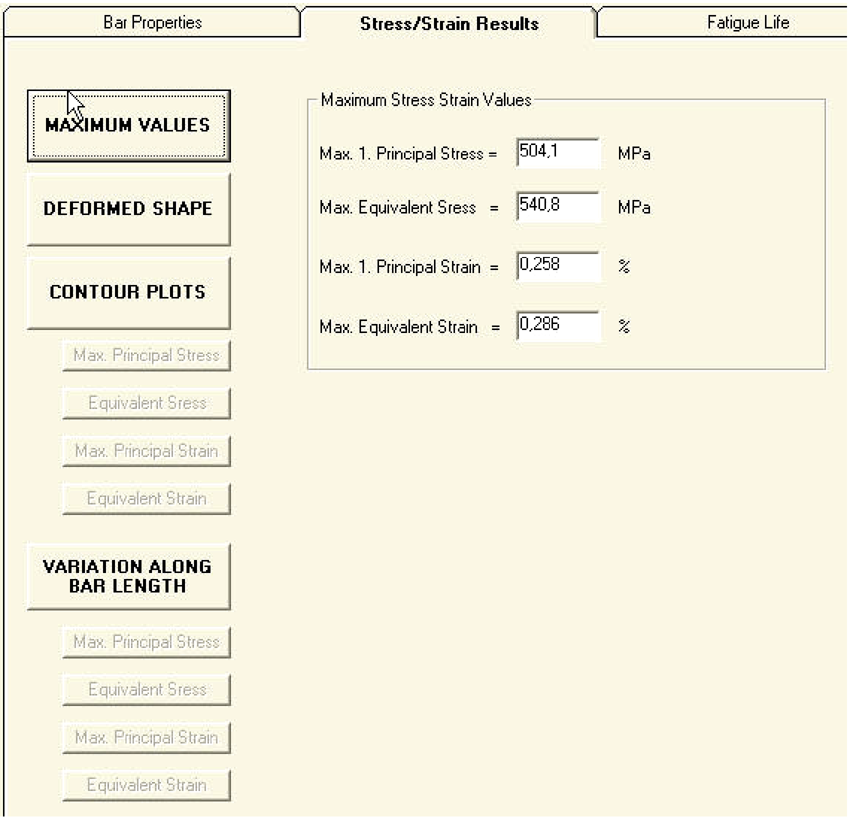


Figure A.8 – Post-processor – Stress/Strain Results Tab

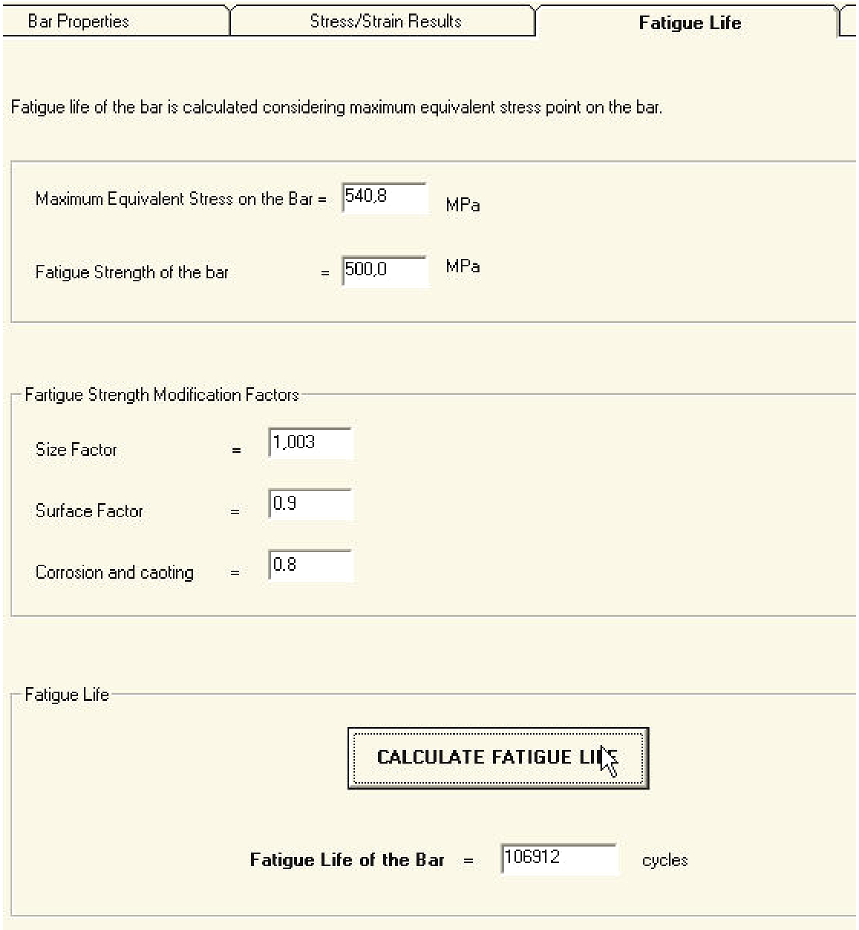


Figure A.9 – Post-processor – Fatigue Life Tab

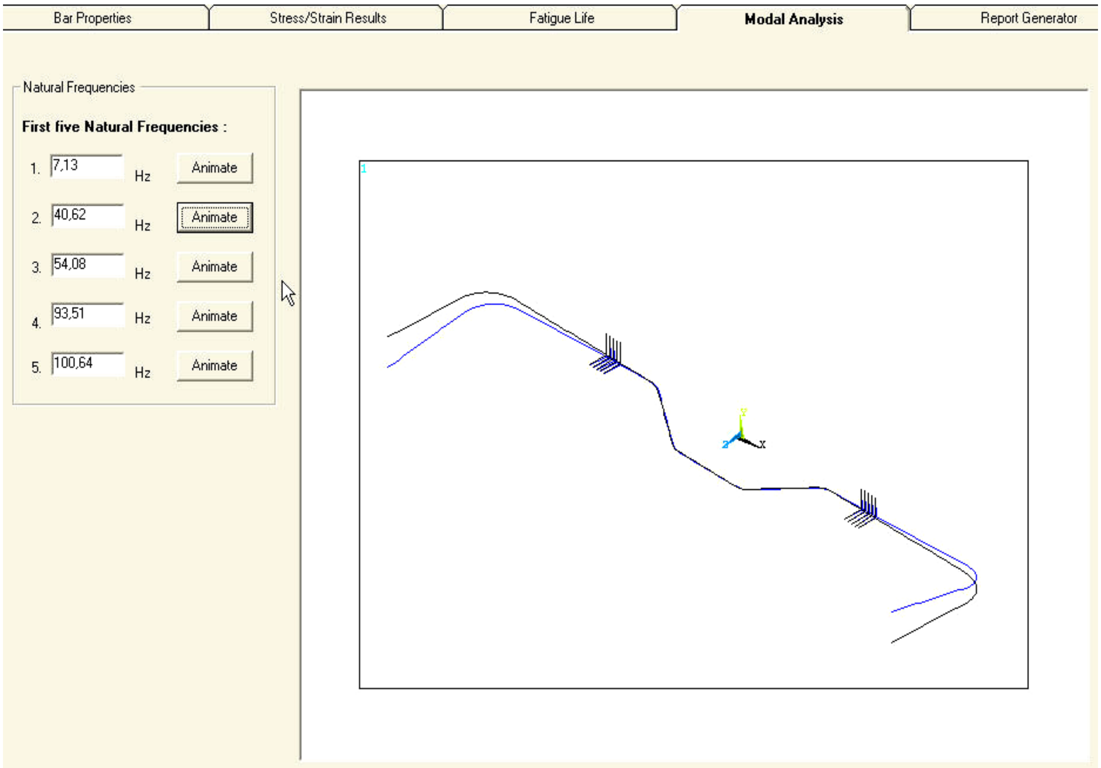


Figure A.10 – Post-processor – Modal Analysis Tab

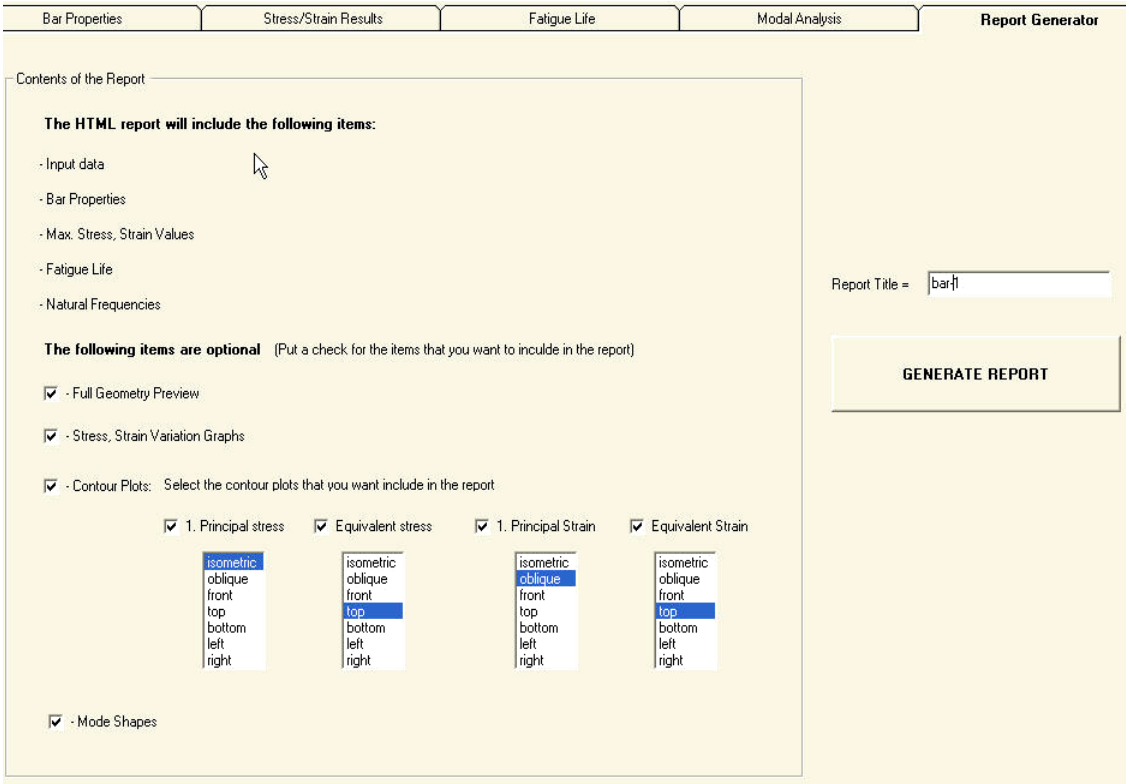


Figure A.11 – Post-processor – Report Generator Tab

APPENDIX B

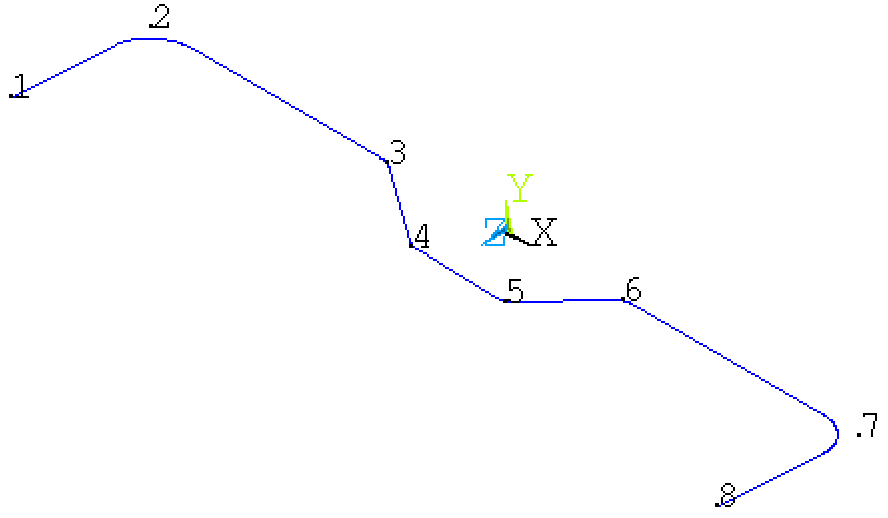


Figure B.1 – Sample Anti–Roll Bar Centerline - Isometric view

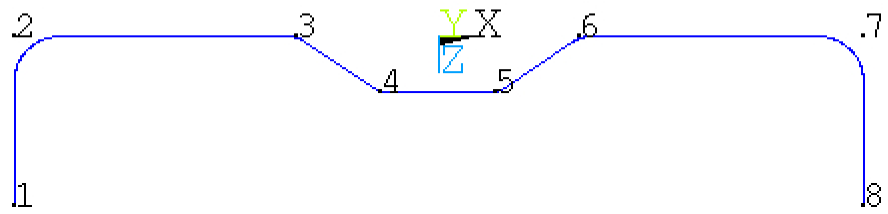


Figure B.2 – Sample Anti–Roll Bar Centerline - Top view

Keypoints	x coord. (mm)	y coord. (mm)	z coord. (mm)
1	-750	20	300
2	-750	0	0
3	-250	0	0
4	-100	-30	100
5	100	-30	100
6	250	0	0
7	750	0	0
8	750	20	300

Fillet Radii (mm) -Starting from KP1 side-	80	60	40	40	60	80

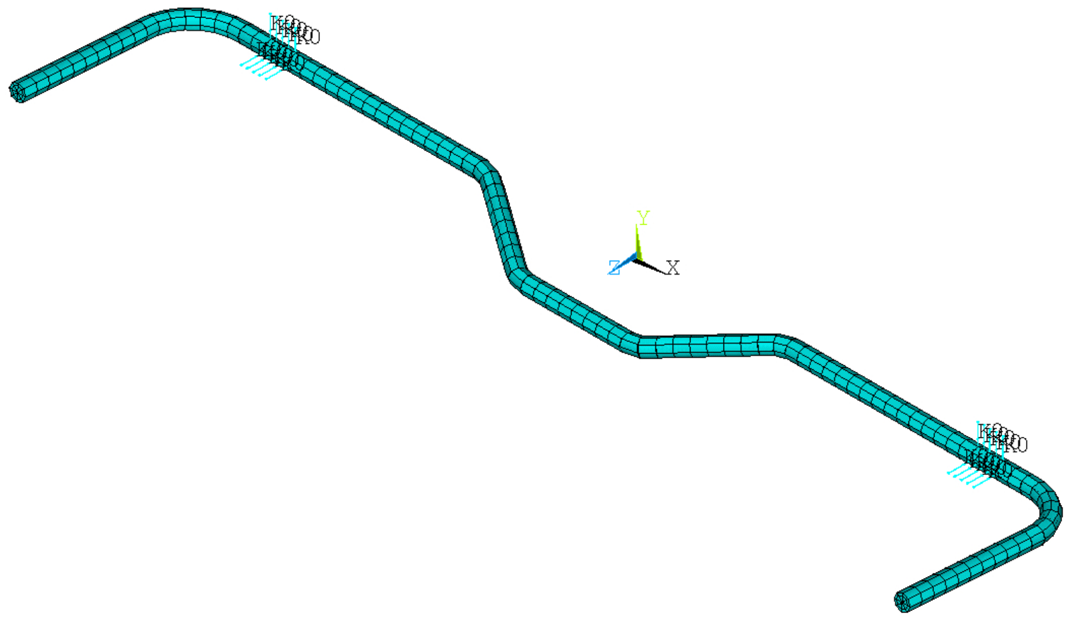


Figure B.3 – Meshed Model of the Anti-Roll Bar - with Bushing Springs

Diameter = 27 mm

Bushing poitions = ± 600 mm

Bushing Stiffness = 2300 / 5 = 475 N/mm

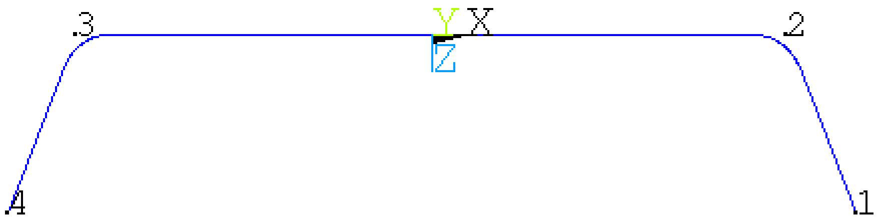


Figure B.4 – Anti-Roll Bar Model for Program Verification Calculations and Sample Analysis - Bar Centerline - Top view

Keypoints	x coord. (mm)	y coord. (mm)	z coord. (mm)
1	550	0	230
2	460	0	0
3	-460	0	0
4	550	0	230

Fillet Radii (mm)	50	50

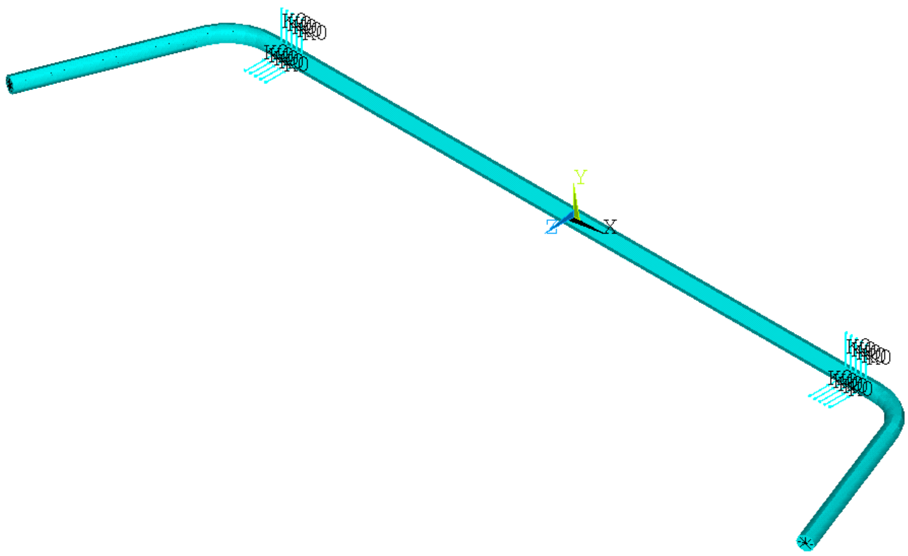


Figure B.5 – Anti-Roll Bar Model for Program Verification Calculations and Sample Analysis - Meshed Model - Isometric view

APPENDIX - C

SAMPLE ANALYSIS REPORT

ANTI-ROLL BAR ANALYSIS REPORT

Analysis Title = BAR-1

Date = 27.08.2003

INPUT DATA

Geometry (Dimensions are in mm)

keypoint coordinates
1) 550 0 230
2) 460 0 0
3) -460 0 0
4) -550 0 230

fillet radii
1) 50
2) 50

bushing locations
x1= 390
x2= -390

Cross-Section

Type = solid circular

Radius = 10 mm

Connections

Bushing Type = x-movement free

Bushing Location on +x side = 390 mm

Bushing Location on -x side = 390 mm

Bushing Length = 40 mm

Bushing Stiffness = 1500 N / mm

End Fixture Type = Spherical Joint

Material

Material Type = SAE 5160

Modulus of Elasticity = 206000 MPa

Poison's Ratio = 0.27

Yield Strength = 1200 MPa

Ultimate Tensile Strength = 1400 MPa

Endurance Limit = 706 MPa

Density = 7800 kg/m^3

Mesh Density

Number of Elements = 100

Loading

Max. Suspension Deflection = 50 mm

Analysis Type

Linear Analysis

Endurance Limit Modification Factors

k_{size} = 1.003

$k_{surface}$ = 0.661 (Cold Drawn)

k_{misc} = 1

BAR PROPERTIES

Roll Stiffness = 432,0 Nm / deg
45,08 Nm / mm
40,98 N / mm

Length = 1394 mm

Mass = 3,416 kg

MAXIMUM STRESS/STRAIN RESULTS

Max. Principal Stress = 578,9 MPa

Max. Equivalent Stress = 652,1MPa

Max. Principal Strain = 0,269 %

Max. Equivalent Strain = 0,395 %

FATIGUE LIFE

N = 98 957 cycles

NATURAL FREQUENCIES

1st Natural Freq. = 0 Hz (This may correspond to rigid body motion)

2nd Natural Freq. = 71,10 Hz

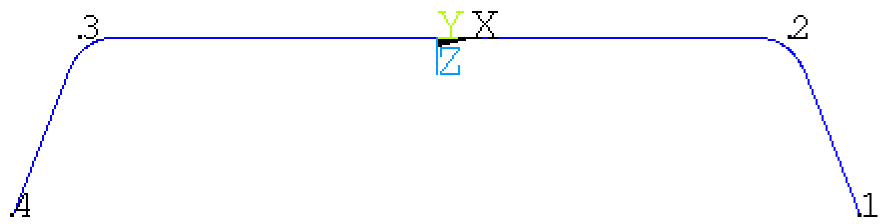
3rd Natural Freq. = 88,82 Hz

4th Natural Freq. = 109,78 Hz

5th Natural Freq. = 165,28 Hz

GEOMETRY PREVIEW

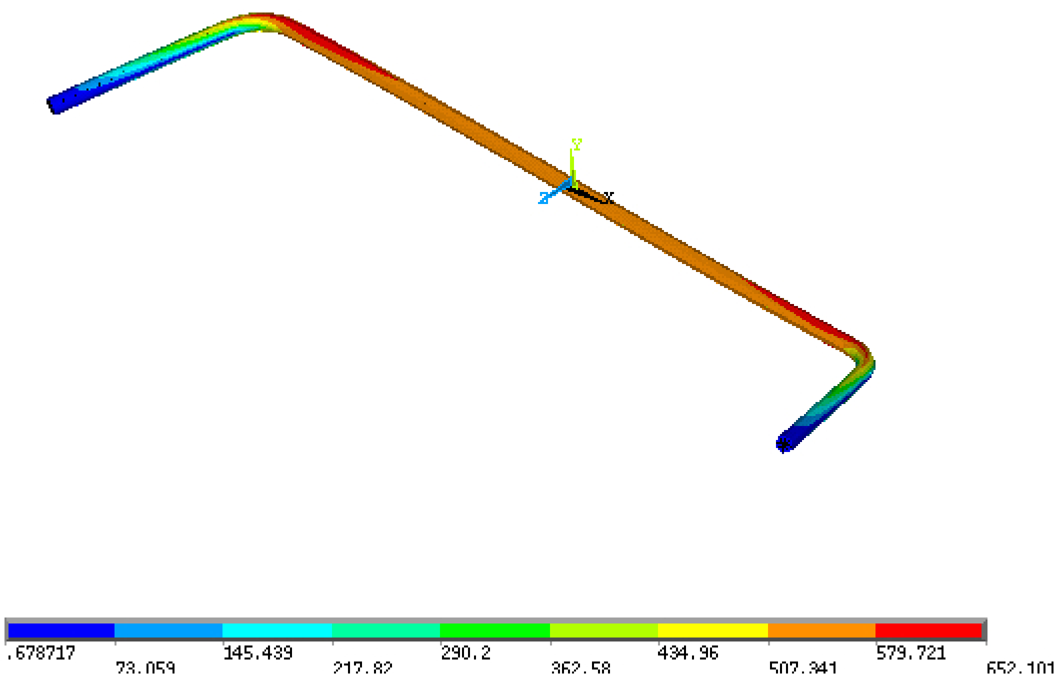
Bar Centerline Geometry - with Keypoint Numbers



STRESS / STRAIN VARIATION GRAPHS

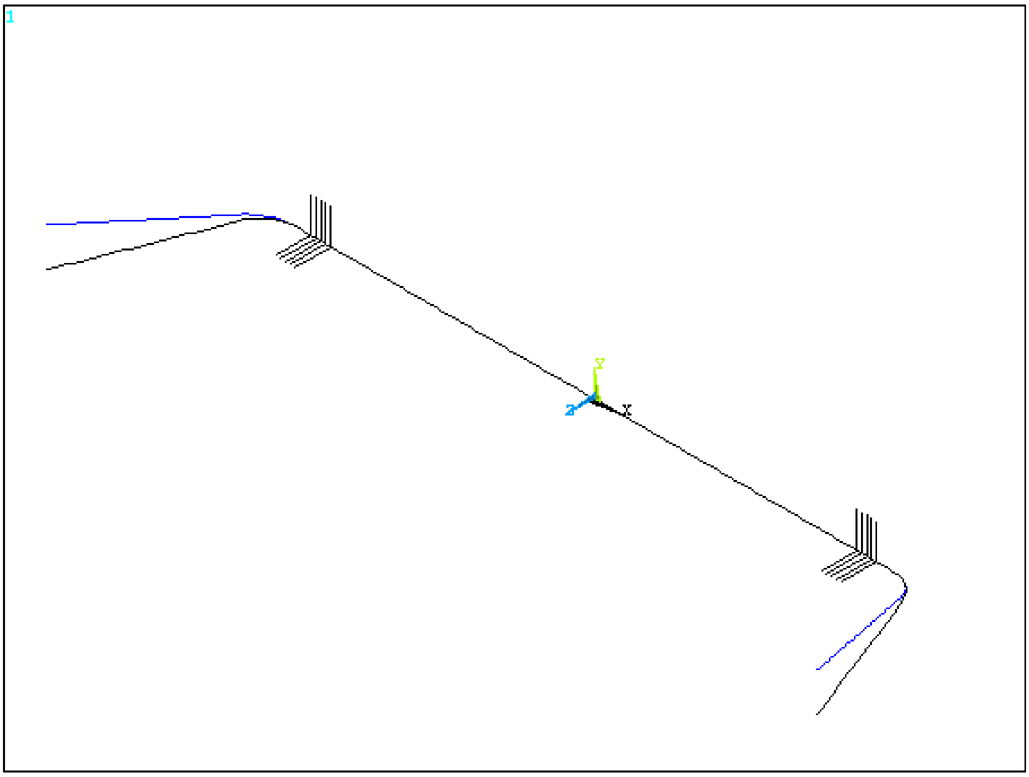
CONTOUR PLOTS

Equivalent stress – isometric view (MPa)



MODE SHAPES

Mode Shape of the First Natural Frequency (0 Hz)



Mode Shape of the Second Natural Frequency (71.1 Hz)