CRASH ANALYSIS ON A TRUCK CABIN BASED ON THE ECE R29 STANDARD

A PROJECT REPORT

Submitted by

R. SARAVANAN - Reg. No. 71206402008

in partial fulfillment for the award of the degree of

MASTER OF ENGINEERING

in

CAD/CAM

DEPARTMENT OF MECHANICAL ENGINEERING

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE – 641 006

ANNA UNIVERSITY :: CHENNAI 600 025

JUNE - 2008

Certificate

ANNA UNIVERSITY :: CHENNAI 600 025

BONAFIDE CERTIFICATE

Certified that this project report entitled **"Crash Analysis on a Truck Cabin Based on ECE R29 Standard"** is the bonafide work of

Mr. R. Saravanan - **Register No.71206402008**

Who carried out the project work under my supervision.

C Sivanandan
Signature of the HOD

Signature of the Supervisor
6/6/08

Internal Examiner

External Examiner

DEPARTMENT OF MECHANICAL ENGINEERING

KUMARAGURU COLLEGE OF TECHNOLOGY

COIMBATORE – 641 006

Dr. M.G.R. EDUCATIONAL AND RESEARCH INSTITUTE

(University u/s 3 of UGC act 1956)

DEPARTMENT OF MECHANICAL ENGINEERING

National Conference on

"Recent Advanced Trends in Mechanical, Automobile and Production Engineering"

RATMAPE 2008

24-25 April, 2008

CERTIFICATE

This is to certify that Mr.R.SARAVANAN

Of Final Year, M.E (CAD/CAM),Kumaraguru College of Technology,Coimbatore-6.

Has presented a technical paper in the conference titled CAE Simulation of Crash Worthiness of a Truck

Cabin using Ls Dyna-A Case Study.

Co-ordinator | Head of the Department | Registrar

Abstract

ABSTRACT

According to the recent survey conducted by the department of transport, U.S., it is calculated that nearly 16 million people are involved either directly or indirectly in road accidents. And this rate is increasing steadily to alarmingly height. Hence a safe means of transportation is required in the near future. Regarding which a number of experiments are being conducted to achieve the necessary objective.

And one such area that has been recently developed in order to improve the safety is that of "crash analysis". Here, the work is being carried out in the area of crash analysis of a truck cabin. The comparison of the result is done with that of the ECE R29 (economic commission for Europe) standard accordingly.

The work is being carried under two phases, the initial phase is with that of the meshing. The given structure is first meshed by means of the platform "Hypermesh V8". The meshing is done on the basis of the given criteria's, and once the optimum level of meshing has been completed, it is proceeded with that of the second phase. In this phase the work begins with that of analysis. The suitable platform selected for this phase is of "Ls Dyna". The application of this software is of nearly endless. And later on the results are obtained and compared to that of the standard.

ஆய்வு சுருக்கம்

அமேரிக்காவில் உள்ள போக்குவரத்து கழகத்தால் புதிதாக எடுக்கப்பட்ட ஆய்வறிக்கையின் படி வருடத்திற்கு சுமார் 1.6 கோடி மக்கள் நேர்முகமாக மற்றும் மறைமுகமாக விபத்தினால் பாதிக்கப்படுகின்றனர் எனக் கூறுகிறது. இதன் காரணமாக வாழ்க்கை ஒரு பாதிப்புக்குள்ளானதாக கருதப்படுகிறது. இதனை தடுக்க பாதுக்காப்பிற்காக வாகனங்களில் பல்வேறு சோதனைகளை மேற்கொள்ளப்படுகிறது.

இச்சோதனையின் மூலம் உருவான புதிய முறையே "நொருங்குதல் பகுப்பாய்வு". இந்த ஆய்வில் மேற்கண்ட முறையை பயன்படுத்தி பழு ஊந்துகளில் உள்ள ஓட்டுநரின் அறையை சோதனை செய்யப்படுகிறது. இச்சோதனையை **இசிஇஆர்** 29 (எக்கனாமி கமிஷன் ஆப் யூரோ நிபந்தனை 29) என்ற தரக்கட்டுபாட்டை அடிப்படையாக கொண்டு மேற்கொள்ளப்படுகிறது. இத்தரக்கட்டுபாடு பாதுகாப்பிற்கு முக்கிய பங்காற்றுகிறது மேலும் இது கனரக ஊர்தி; பேருந்து மற்றும் பளு ஊந்து போன்றவற்றிக்கு மட்டும் பொருந்தக் கூடியதாகும்.

இந்த ஆய்வில் இரு கட்டங்களாக சோதனை மேற்கொள்ளப்படுகிறது. முதற்கட்டத்தின்படி வடிவமைக்கப்பட்ட பாகத்தை சிறுசிறு பாகங்களாக பிரித்து அவை ஒன்றோடொன்று பொருந்தும் படிவையாக மாற்றப்படுகிறது. இவ்வாறு ஒன்றோடொன்று பொருந்தும் படிவையாக மாற்றி, கொடுக்கப்பட்ட கட்டுபாடுகளின்படி செய்யபட்டுள்ளது. அதற்குபின் இரண்டாவது கட்டமாக படிவையாக மாற்றப்பட்ட பாகங்களை பகுப்பாய்வுக்கு உட்படுத்தபடுகிறது.

மேற்கண்ட இவ்வாய்வு, அசோக் லேலேண்டு நிறுவனத்தால்

Acknowledgment

ACKNOWLEDGEMENT

I acknowledge with sincere thanks to the **Management of Kumaraguru College of Technology** for their kind patronage, which helped me to work on the project successfully.

I thank our principal **Dr. Joseph V. Thanikal, M.E., Ph.D., PDF, CEPIT** for providing us a chance to do this project.

I am very grateful debted to **Dr. C. Sivanandan, Ph.D.**, HOD – Department of Mechanical Engineering for approving this project and extending his support both technically and morally throughout then project.

I wish to express my deepest gratitude to my guide **Dr. V. VeL Murugan, Ph.D.**, Associate Professor - Department of Aeronautical Engineering, for guiding me through my project, providing valuable suggestions and for supporting me in all possible ways for computing this project.

I express my heartfelt gratitude and sincere thanks to **Dr. P. Palanisamy, Ph.D.**, Project Coordinator – Department of Mechanical Engineering, for his help, valuable advice and motivation during the entire project.

I thank M/s. Ashok Leyland Ltd., for giving me an opportunity to work in this project and also thank my guide **Mr. Nagendhra Appa Rao, Assistant Manager CAE**, for his assistance throughout this work.

I also wish to express my gratitude and gratefulness to my Parents, Family Members and Friends for encouraging and motivating me through out the project. Successful work will have in its backdrop blessings of God, hence I place my humble salutations and prayers to God Almighty for His all pervading grace that sustained throughout the study.

Contents

CONTENTS

List of Figures

LIST OF FIGURES

Chapter 1

Introduction

CHAPTER 1

INTRODUCTION

Roads and streets form a part of workplace of many employees. Besides professional drivers, it includes home-help and house nursing personals, security staffs and sales personals. And considering a report by **RAIS (Road Accident Information system)** of 2004, it is such that;

Number of accidents reported	41220
Fatal	3059
Grevious injuries	25631
Minor injuries	25594

Total number of accident cases during the year 2002 was 38761 and increased to 39496 in 2003 and 41220 by 2004.And the percentage increase was about 1.89% (735) from 2002-2003 and about 4.36% (1724) in 2004.An estimation death of 3000 people every year. Hence it is necessary to avoid the ill health and accidents on the road. And from the overall about 20% of the fatal accidents is with that of the heavy vehicles.

Apart from the other protocols regarding the safety, improved design also plays a vital role in saving one's life. And here the project deals with one such an issue. Here the concentration is on the Vehicle cabin (Truck).The cab design is such a way that, sufficient survival space is to be provided in the event of an accident. And this project has been carried out under one of the well known manufacturer of the trucks, "ASHOK LEYLAND". Chennai.

Chapter 2

Company Profile

CHAPTER 2

COMPANY PROFILE

2.1 INTRODUCTION

Ashok Leyland, over five decades in the transport solution industry. offering a world class range of trucks, buses, special application vehicles and engines.

For over five decades, Ashok Leyland has been the technology leader in India's commercial vehicle industry, moulding the country's commercial vehicle profile by introducing technologies and product ideas that have gone on to become industry norms.

From 18 seater to 82 seater double-decker buses, from 7.5 tonne to 49 tonne in haulage vehicles, from numerous special application vehicles to diesel engines for industrial, marine and genset applications, Ashok Leyland offers a wide range of products.

Eight out of ten metro state transport buses in India are from Ashok Leyland. With over 60 million passengers a day, Ashok Leyland buses carry more people than the entire Indian rail network!

The origin of Ashok Leyland can be traced to the urge for self-reliance, felt by independent India. Pandit Jawaharlal Nehru, India's first Prime Minister persuaded Mr. Raghunandan Saran, an industrialist, to enter automotive manufacture. In 1948, Ashok Motors was set up in what was then Madras, for the assembly of Austin Cars. The Company's destiny and name changed soon with equity

Since then Ashok Leyland has been a major presence in India's commercial vehicle industry with a tradition of technological leadership, achieved through tie-ups with international technology leaders and through vigorous in-house R&D.

Access to international technology enabled the Company to set a tradition to be first with technology. Be it full air brakes, power steering or rear engine busses, Ashok Leyland pioneered all these concepts. Responding to the operating conditions and practices in the country, the Company made its vehicles strong, over-engineering them with extra metallic muscles. "Designing durable products that make economic sense to the consumer, using appropriate technology", became the design philosophy of the Company, which in turn has moulded consumer attitudes and the brand personality.

Ashok Leyland vehicles have built a reputation for reliability and ruggedness. The 5,00,000 vehicles we have put on the roads have considerably eased the additional pressure placed on road transportation in independent India.

In the populous Indian metros, four out of the five State Transport Undertaking (STU) buses come from Ashok Leyland. Some of them like the double-decker and vestibule buses are unique models from Ashok Leyland, tailor-made for high-density routes.

In 1987, the overseas holding by Land Rover Leyland International Holdings Limited (LRLIH) was taken over by a joint venture between the Hinduja Group, the Non-Resident Indian transnational group and IVECO. (Since July 2006, the Hinduja Group is 100% holder of LRLIH).

The blueprint prepared for the future reflected the global ambitions of the company, captured in four words: Global Standards, Global Markets. This was at a time when liberalization and globalization were not yet in the air. Ashok Leyland embarked on a major product and process up gradation to match world-class standards of technology.

... for quality, Ashok Leyland reached a win the ISO 9002 certification. The more comprehensive ISO 9001 certification came in 1994, QS 9000 in 1998 and ISO 14001 certification for all vehicle manufacturing units in 2002. It has also become the first Indian auto company to receive the latest ISO/TS 16949 Corporate Certification (in July 2006) which is specific to the auto industry.

2.2 RESEARCH AND DEVELOPMENT

2.2.1 World-Class Technology

To offer world-class technology that is relevant and affordable to the Indian customer is the philosophy that drives R&D at Ashok Leyland. Over the years, this philosophy has been translated time and again into products that seamlessly integrate international technology with local needs. "The role of R&D is central in fulfilling the company-wide commitment to total customer satisfaction" states Mr. R. Seshasayee, Managing Director, and adds that the increased infrastructural and financial support expresses the company's determination to become self-reliant in R&D.

2.2.2 Value to the Customer

The immediate R&D priorities are to pro-actively address safety and environmental issues, harness and adopt technologies that provide value to the customer in an atmosphere enabling creativity and innovation. Powering those who "engineer tomorrows" with an enabling infrastructure has been top priority for the company.

2.2.3 Test Tracks

But our R&D is not confined within walls. It extends to the test tracks as well. Rigorous tests are carried out under stringent simulated conditions that replicate the most treacherous landscapes.

Vehicle ruggedness and longevity are a prime customer concern, as they directly impact earnings. Ever conscious of this, Ashok Leyland makes extensive use of a modern CAD set-up, a comprehensive test track facility (where cobble-stones are durability testing facilities. Together they ensure that there is a constant improvement in the life and on-road performance of every make of Ashok Leyland vehicle to hit the roads. Safety, durability, through our R&D efforts.

2.2.4 Innovations

Ashok Leyland product development successes have come from a keen sense of anticipation and attentiveness. The company initiated research into alternative fuels well before legislative debate had even begun in the country. The result was the implementation of CNG technology ahead of the rest promising a breath of fresh air for polluted cities.

2.3 RESEARCH AND DEVELOPMENT MILESTONES

1966	Introduced full air brakes
1967	Launched double-decker bus
1968	Offered power steering in commercial vehicles
1979	Introduced multi-axle trucks
1980	Introduced the international concept of integral bus with air suspension
1982	Introduced vestibule bus
1992	Won self-certification status for defence supplies
1993	Received ISO 9002
1997	India's first CNG powered bus joined the BEST fleet
2001	Received ISO 14001 certification for all manufacturing units
2002	Launched hybrid electric vehicle

2.4 ASSOCIATE COMPANIES

2.4.1 AUTOMOTIVE COACHES AND COMPONENTS LTD (ACCL)

ACCL was promoted by Ashok Leyland and the Tamil Nadu Industrial Development Corporation (TIDCO) in the 1980's

2.4.2 LANKA ASHOK LEYLAND

Established in 1982, this is a joint venture between Ashok Leyland and the Government of Sri Lanka

2.4.3 ENNORE FOUNDARIES

Established in 1959, Ennore Foundries is India's largest automotive jobbing foundry with production capacity of 45,000 MT in Grey Iron and 3000 MT in aluminum gravity die castings per annum. Certified to ISO - 9001 and QS 9000 Quality systems.

2.4.4 IRIZAR-TVS

Started in 2001, IRIZAR-TVS is a joint venture between Ashok Leyland, TVS & Sons Ltd and IRIZAR, the internationally reputed bus body builder from Spain. This joint venture addresses the growing demand for luxury coaches in the country.

2.4.5 ASHOK LEYLAND PROJECT SERVICES LTD

Ashok Leyland Project Services Limited (ALPS), spearheads the project development activities of the Hinduja Group in India. Apart from assisting the investment entities of the Group identify and implement successfully projects in India, ALPS also provides professional services to help international companies interested in projects in India.

2.4.6 ASHLEY DESIGN AND ENGINEERING SERVICES (ADES)

ADES is a venture by Ashok Leyland, the trusted name in the automobile industry and the flagship of the Hinduja Group, the multi-billion dollar transnational conglomerate.

ADES caters to a global need in transportation and other select industry verticals by offering end-to-end solutions in the spectrum of Design, Engineering, Prototyping, Testing and Validation.

CHAPTER 3

PROBLEM IDENTIFIED

As the vehicles on road increases day by day, so is the rate of accidents. Road safety has already emerged as a major social and economic concern in India. With over 60,000 deaths on the road every year, the situation is indeed alarming. The prospects for the future does not look brighter either. With the vehicle manufacture and purchase poised for big leap and no dramatic increase in the width and length of the road network, urgent steps are needed even to stay where one is.

A recent research by the transport research laboratories in the U.K. puts china well ahead of India on road deaths. In Thailand, Korea and Malaysia where per capita vehicle population is more than ten times that of India, the situation is much worse. Whether India will go the East Asian way is a moot point.

Spread of safety awareness is perhaps most important. There are several areas in which concerted action is needed. One such area is of going for the optimum design. And here the work is done in the area of studying the given design for its optimum level. And here the concentration is on that of a truck cabin. The testing is done to that of "ECE R-29" standard, kept as reference.

Technology can only solve the problem up to a point but it is the human factor which needs training and education.

Chapter 3

Problem Identified

Chapter 4

Literature Survey

CHAPTER 4

LITERATURE SURVEY

The following are some of the work initially been carried out related to the topic mentioned. It gives the description of literature reviewed from various research papers published in international and national journals, proceedings of various conferences and books.

1. Steven W. Kirkpatrick, Robert Mac Niell, and Robert T. Bocchiri, Etr (November 2002)

This paper presents the development and validation of an LS-DYNA finite element occupant model suitable for use in crash analyses of roadside safety features. The addition of an occupant model in crash simulations can provide a link between vehicle accelerations and occupant injury. Injury measures for the vehicle occupants can provide additional evaluation criteria for the effectiveness of roadside safety features. The use of an occupant model in roadside hardware crash analyses will also improve the fidelity of the crash analysis by correctly modeling the internal effects of the occupants and their interaction with the vehicle motions.

2. Heavy vehicle research center, Etr (September 2005)

The main objectives of this research task were to conduct an in-depth evaluation of the single-unit truck (SUT) finite element model with respect to its ability to accurately simulate its interaction with roadside safety hardware and to identify areas of possible improvements. The model's primary purpose is to be used as a "bullet" object for computational evaluation of roadside safety hardware.

3. Annemarie R.Vakonyi-Koczy,Andras Rovid,Varlaki, Etr (January 2007)

The energy absorbed by the deformed car body is one of the most important factors affecting the accidents thus it plays a very important role in car crash tests and accident analysis. There is an ever-increasing need for more correct techniques, which need less computational time and can more widely be used. Thus, new modeling and calculating methods are highly welcome in deformation analysis.

4. Tai Suke Wantanabe,Shigera Hirayama,KIazuhiro Obayashi,Tomosaburo Okabe, Etr (March 2005)

To achieve good frontal impact compatibility, it is necessary to help match stiffness between vehicles in addition to the enhancement of structural interaction. In this paper, the issues of helping stiffness matching in frontal SUV-to-car impacts were studied using MADYMO vehicle simulation and MADYMO occupant dummy simulation.

5. T.Kim Parnell, M.ASCE Christopher, V.White and Shari E.Day, Etr (January 2002)

The computer program LS-DYNA 3D was used to simulate the behavior of a specific, though representative, heavy truck cab-over tractor-trailer vehicle during a full 180° rollover event. These simulations provide a key component in the development of a physical testing procedure for evaluating integrity and occupant crash protection system designs in heavy trucks.

Chapter 5

ECE R29

CHAPTER 5

ECE R-29

This automotive industry standard specifies the requirements of survival space for the protection of occupants of the cab of a commercial vehicle of category N as defined in IS: 14272 (Part 1): 1995 which are intended for the carriage of goods. It does not apply to agricultural tractors and machinery and construction equipment vehicles.

- Specifies the general specification and requirements, test methods and survival space for approval of the cabin.
- This code in the form of AIS 29 is to be implemented in India from the year 2007/08.

5.1 SURVIVAL SPACE REQUIREMENT AS PER ECE R-29

Para 5.3 and 5.4 of the standard specifying the survival space and other requirements, are reproduced below:

"5.3.1 After undergoing each of tests referred to in para 5.2, the cab of the vehicle shall exhibit a survival space allowing the accommodation for the manikin defined in Annexure III of ECE R-29, when the latter is in its median position, without contact between the manikin and non-resilient parts. To facilitate installation, the manikin may be inserted in dismantled form and assembled in the cab.

5.3.2 The space so defined shall be verified for every seat provided by the manufacture.

5.4.1 During the tests, the components by which the cab is secured to the chassis

5.4.2 None of the doors shall open during the tests, but the doors shall not be required to open after testing."

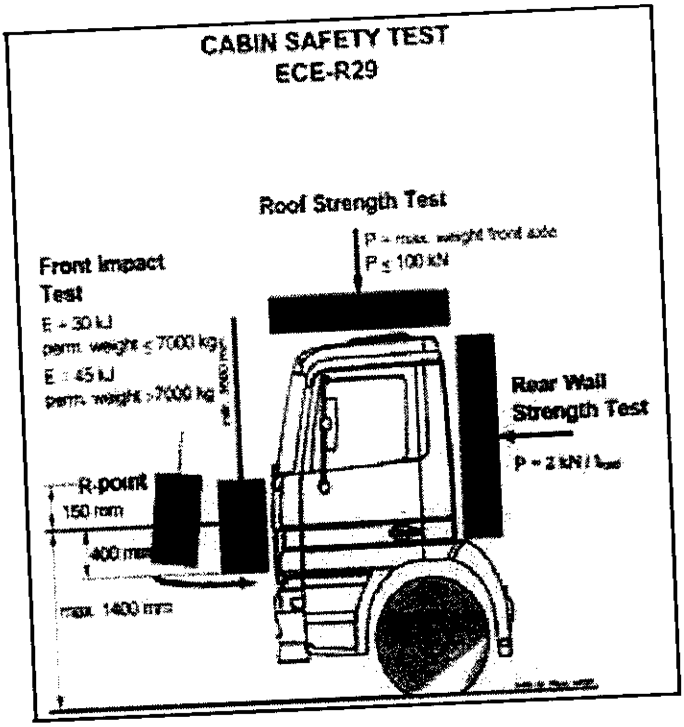


Fig. 5.1 Basic Test Setup

5.2 FRONTAL IMPACT TEST (TEST A)

The frontal impact test simulates the head on collision condition. The frontal test calls for a 3.5 m long pendulum with rectangular flat striking face 2.5 m * 0.8 m of steel of mass 1500 kg. C.G. of the pendulum is ----------- mm below the R-Point of driver's seat and in longitudinal plane of the vehicle. The impact energy is 80 kJ for vehicle having payload capacity up to 7000 kg and 45 kJ for more than 7000kg.

5.2.1 MOUNTING OF THE CAB

The cab shall be mounted on the vehicle. But at the request of the manufacturer, the test can be carried out with the cab mounted on a special frame, which is equivalent to mounting of vehicle.

5.2.2 PHYSICAL TEST

For frontal impact test, a pendulum impact test rig was designed and developed at ARAI according to ECE R-29 guidelines. The cab was mounted on the long members of the cab as on the vehicle. The test set up is shown as follows.

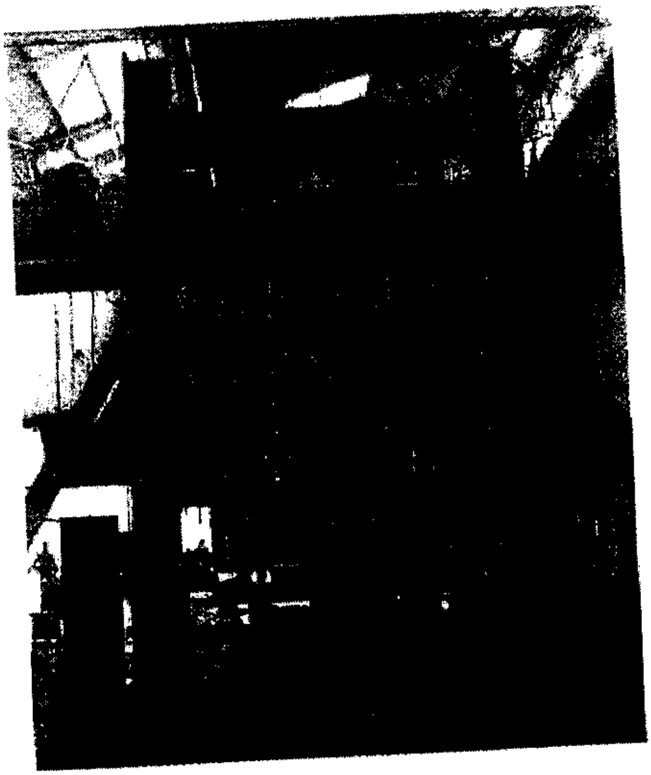


Fig 5.2 Test Setup – Frontal Impact

5.2.3 FE SIMULATION

The standard allows FE simulation for roof and rear wall strength tests. Even though the standard does not consider frontal impact test, this was simulated for development purpose using Non-Linear FE software LS-DYNA and the results are compared with actual test. Fig. 3 shows the deformed shape obtained from actual testing. It is observed that simulation and actual test results are comparable.

5.2.4 OBSERVATIONS

Para 5.4.1 of the standard specifies that "during the tests, the components by which the cab is secured to the chassis frame may be distorted or broken, provided that the cab remains attached to the chassis frame." This is not very clear. During mountings get sheared, it is difficult to say whether the cab remain attached to the chassis or not. If mounting cannot sustain the impact load, one should consider that the cab is not meeting the requirements.

The standard specifies the survival space after undergoing the test (Para 5.3.1) but does not talk about survival space during the test. During an actual test, high-speed photography was used, which showed that the steering wheel intruded dynamically in the survival space as the steering wheel assembly is very flexible and elastic. This is also confirmed by simulation. Other standards like ECE R-12 [3], and FMVSS 204 [4] specifies that the dynamic movement of the steering should be measured during and after the test. ECE R=66 [5] also talks about the survival space to be maintained during and after the test. The above standards also suggest use of high-speed photography of understand the behavior of the vehicle during the impact test. Hence it is suggested that the survival space should be checked by suitable means like high-speed photography during and after the impact test and the standards should specify the survival space during the test, besides after.

5.3 ROOF STRENGTH TEST (TEST B)

The roof strength test simulates the roll over and toppled accident condition and front axle weight is applied on the roof of the cab.

5.3.1 TEST PROCEDURE

Para 5 of Annex 3 of ECE R-29 [1] giving the test procedure reads:

The roof of the cab shall withstand a static load corresponding to the maximum mass authorized for the front axle of the vehicle, subjected to maximum of 10 tones. This load shall be distributed uniformly all ever bearing members of the roof structure of the drivers cab or compartment by means of any suitably shaped rigid former."

5.3.2 OBSERVATIONS

The standard specifies that load shall be applied by a suitably shaped rigid former out with rigid formers, which showed that as the load increases, the roof gets deformed and looses contact with the former in some areas and further uniform loading is not possible, which is also proved by simulation as shown in fig. 4. Hence in the physical test, the load was applied with two different methods viz hydraulic actuator and dead weight as shown in fig. 5 and 6 respectively and results were compared.

With the hydraulic actuator, it was found that the plate moves in the vertical direction only and does not change its orientation with deformation of the cab and looses contact in some areas resulting in non-uniform distribution of load. While in other case, the dead weight was kept on the roof without any restraint so that the orientation could change with deformation of cab and uniform loading was possible. Hence, for applying uniform load, the dead weight option without any constraints is a better practical method and is suggested to be specified.

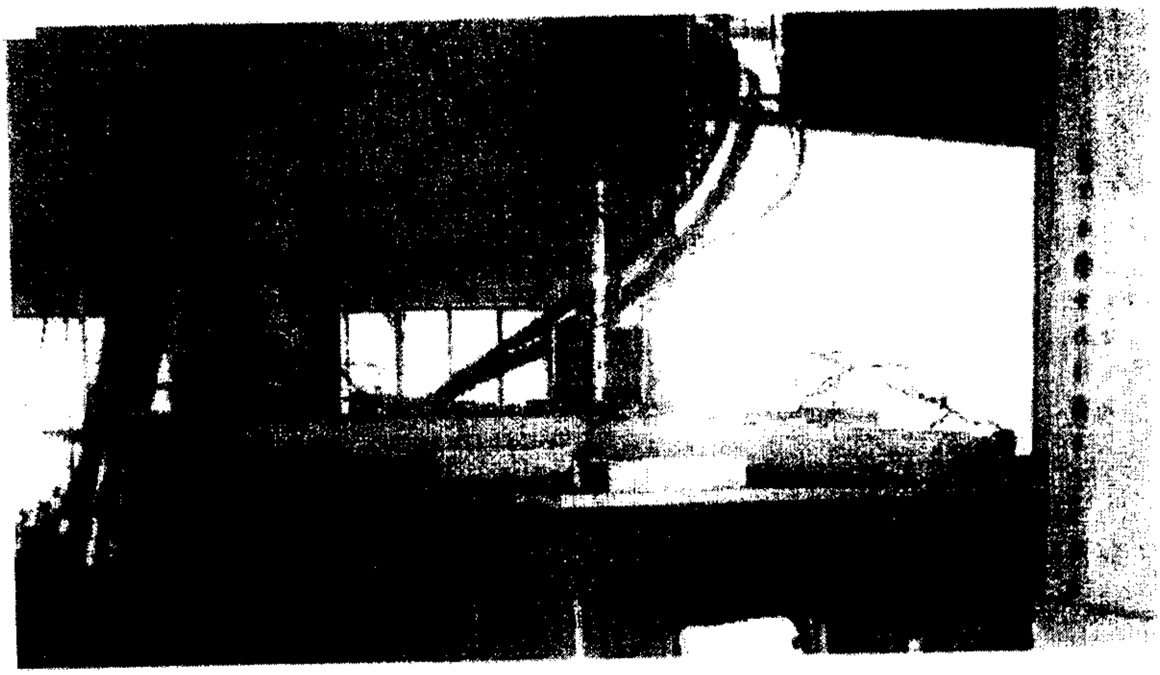


Fig 5.3 Test with Hydraulic Actuator

During one of the test with dead weight, it was observed that the cob could take the load initially for some time (approximate 10 s), and then started yielding under the load subsequently. Though standard does not talk about the duration of

Fig 5.4 Test with Dead Weight

It is suggested that the duration of the application of full load should be at least 5 min.

In the roof strength test, the load is applied in vertical direction but in most accidental conditions, the cab gets rolled over and the loading is in inclined direction. Hence the test does not represent the accidental condition practically. Other standards like ECE R-66 [6] and FMVSS 216 [7] mention about application of load at an angle of 25 (+ 0, -5) to the central longitudinal vertical of the roof. Hence it is suggested that the load should be applied with some inclination(25), which will simulate the accidental condition in a better way.

5.4 REAR WALL STRENGTH TEST (TEST C)

The rear wall strength test simulates the braking and head on collision test where the payload can hit the rear wall. For the test, cab should be capable of withstanding static load of 200 kg per ton of permissible payload. This load shall be applied by means of a rigid barrier perpendicular to longitudinal median axis of the vehicle, covering at least the whole of the cab rear wall situated above the chassis frame and moving parallel to that axis. The rear wall strength test was roof strength test this test specifies clearly that the rigid barrier should move perpendicular to longitudinal median axis. Hence use of hydraulic actuator for application of the load satisfies this condition.With relevant to the literature survey carried out, the following methodology were adopted in this paper.

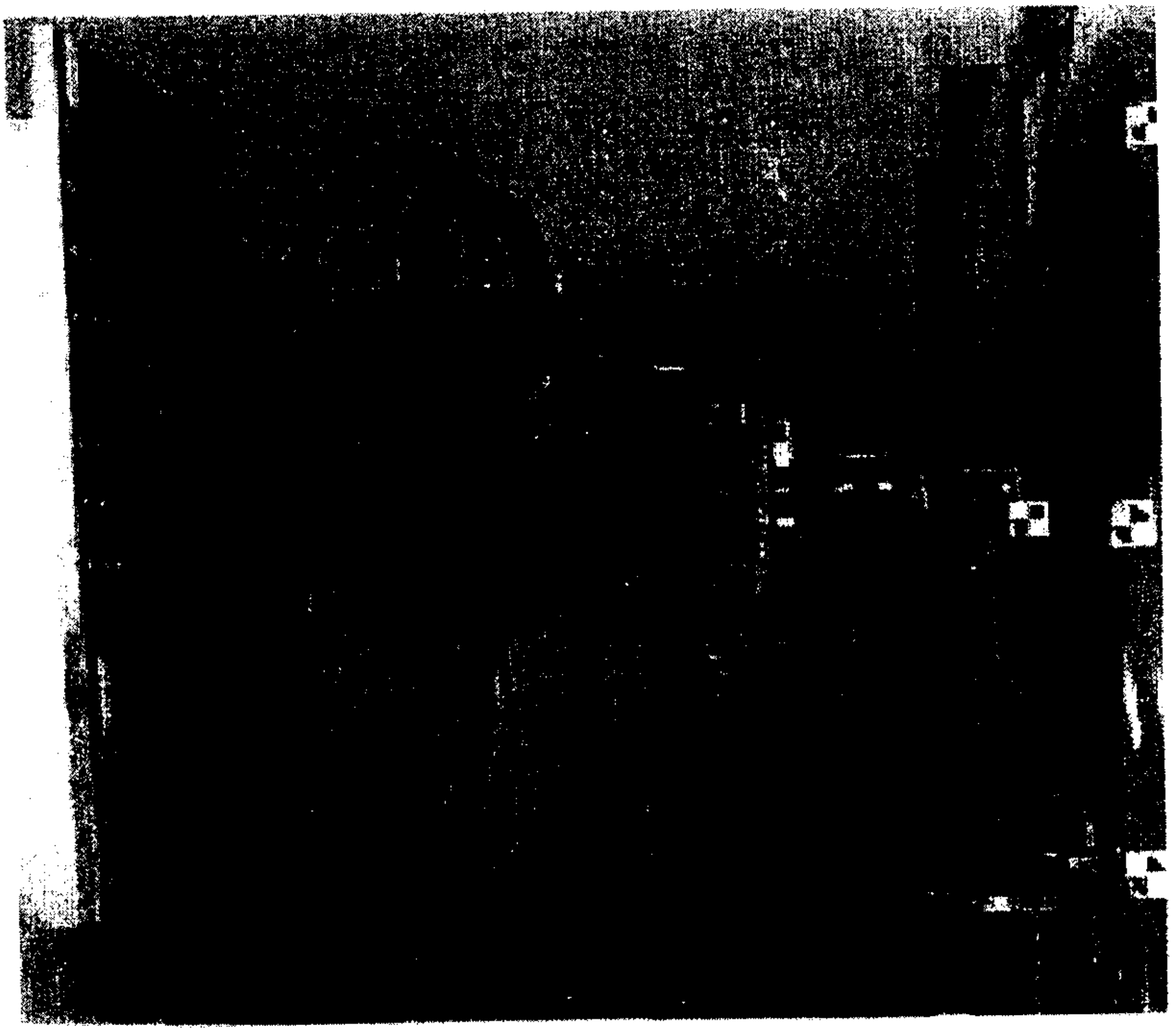


Fig 5.5 Rear Wall Test

Chapter 6

Methodology

CHAPTER 6

METHODOLOGY

The following are the methodology that has been and to be followed in order to achieve the objective accordingly.

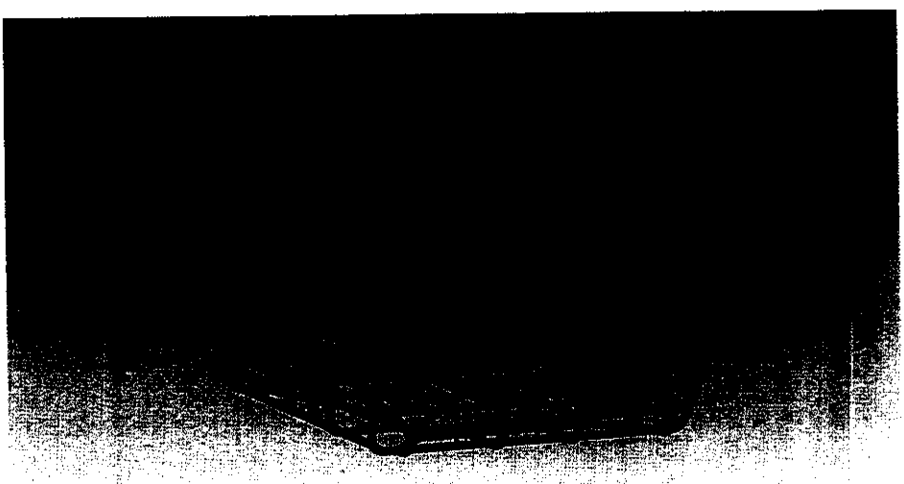
And these are as follows;

- Modeling
- Meshing
- Analysis

All though the work being carried out is from the area of meshing, let us see a short description regarding the modeling respectively.

6.1 MODELING

Here the term modeling denotes the conversion of a blue print design which is of 2D to a model of 3D using the computer with the help of the suitable software. And the software practiced in this industry is of the "CATIA V5".



6.1.1 SOFTWARE DESCRIPTION – CATIA

CATIA V5 is the leading product development solution for manufacturing organizations of all sizes.

Apply its capabilities to a variety of industries such as aerospace, automotive, industrial machinery, electrical, electronics, shipbuilding, plant design, and consumer goods.

Provides an integrated suite of Computer Aided Design (CAD), Computer Aided Engineering (CAE), and Computer Aided Manufacturing (CAM) applications for digital product definition and simulation addresses the complete product development process, from product concept specifications through product-in-service, in a fully integrated and associative manner facilitates true collaborative engineering across the multidisciplinary extended enterprise, including style and form design, mechanical design, equipment and systems engineering, digital mock-up, machining, analysis, and simulation enables enterprises to reuse product design knowledge and accelerate development cycles helps companies speed their responses to market needs and frees users to focus on creativity and innovation based on the open, scalable V5 architecture.

6.1.2 BENEFITS

Features and capabilities commonly referred to as 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAX). The stages range from conceptualization, through design (CAD) and manufacturing (CAM), until analysis (CAE).

CATIA provides open development architecture through the use of interfaces, which can be used to customize or develop applications. The supporting application programming interfaces are as follows:

- The FORTRAN and C programming languages for version 4 (V4).
- The Visual Basic and C++ programming languages for version 5 (V5).

These APIs are referred to as CAA for V4 and CAA2 (or CAA V5) for V5. The CAA2 are component object model (COM) like interfaces. They provide integration for products developed on the CATIA suite of software.

Although later versions of CATIA V4 implemented NURBS, version 4 principally used piecewise polynomial surfaces. CATIA V4 uses a non-manifold solid engine.

CATIA V5 features a parametric solid/surface-based package which uses NURBS as the core surface representation and has several workbenches that provide KBE support. As of 2007, the latest release is V5 release 18 (V5R18).

One of the main reasons customers choose CATIA V5 is its ability to seamlessly interact and work in tandem with a host of other applications like Enovia, Smarteam, various CAE Analysis applications etc.

6.1.3 APPLICATIONS

CATIA is found in a variety of industries throughout the world. Some of these industries include; Aerospace, Appliances, Architecture, Automotive, Construction, Consumer Goods, Electronics, Medical, Furniture, Machinery, Mold and Die, and Shipbuilding.

6.2 MESHING

6.2.1 INTRODUCTION TO FEA

Finite element analysis was first developed for use in the aerospace and nuclear industries where the safety of structures is critical. Today, the growth in usage of the method is directly attributable to the rapid advances in computer technology in recent years. As a result, commercial finite element packages exist that are capable of solving the most sophisticated problems, not just in structural analysis, but for a wide range of phenomena such as steady state and dynamic temperature distributions, fluid flow and manufacturing processes such as injection molding and metal forming.

FEA consists of a computer model of a material or design that is loaded and perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

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

Within each of these modeling schemes, the system behaves linearly or non-linearly. Linear systems are far less complex and generally ignore many subtleties of model loading & behavior. Non-linear systems can account for more realistic behavior such as plastic deformation, changing loads etc. and is capable of testing a component all the way to failure.

6.2.2 FINITE ELEMENT ANALYSIS

Finite element analysis is a computer-based numerical technique for calculating the strength and behavior of engineering structures. It can be used to calculate deflection, stress, vibration, buckling behavior and many other phenomena. It can be used to analyze either small or large-scale deflection under loading or applied displacement. It can analyze elastic deformation, or "permanently bent out of shape" plastic deformation. The computer is required because of the astronomical number of calculations needed to analyze a large structure. The power and low cost of modern computers has made finite element analysis available to many disciplines and companies.

In the finite element method, a structure is broken down into many small simple together to build the whole structure, the equations describing the behaviors of the individual elements are joined into an extremely large set of equations that describe the behavior of the whole structure. The computer can solve this large set of simultaneous equations. Form the solution, the computer extracts the behavior of he individual elements. From this, it can get the stress and deflection of all the parts of the structure. The stresses will be compared to allowed values of stress for the materials to be used, to see if the structure is strong enough.

The term "finite element" distinguishes the technique form the use of infinitesimal "differential elements" used in calculus, differential equations, and partial differential equations. The method is also distinguished from finite difference equations, for which although the steps into which space is divided are finite in size, there is little freedom in the shapes that the discreet steps can take. Finite element analysis is a way to deal with structures that are more complex than can be dealt with analytically using partial differential equations. FEA deals with complex boundaries better than finite difference equations will, and gives answers to "real world" structural problems. It has been substantially extended in scope during the roughly 40 years of its use.

6.2.3 HOW IS FINITE ELEMENT ANALYSIS USEFUL?

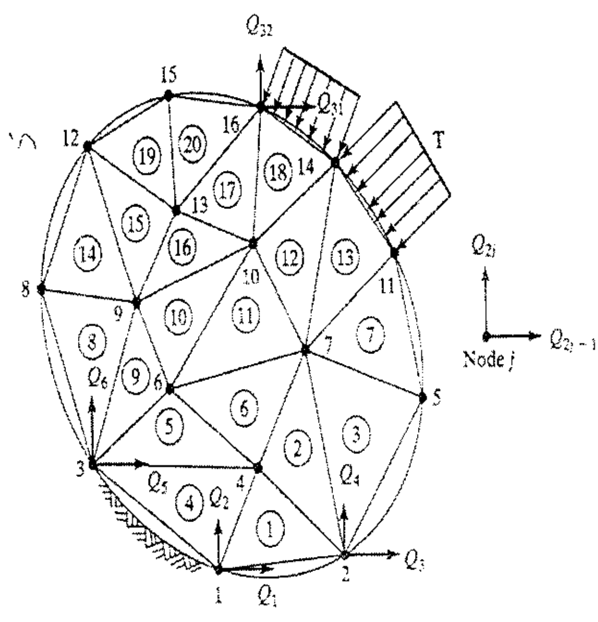
Finite Element Analysis makes it possible to evaluate a detailed and complex structure, in a computer, during the planning of the structure. The demonstration in the computer of the adequate strength of the structure and the possibility of improving the design planning can justify the cost of this analysis work. FEA has also been known to increase the rating of structures that were significantly over designed and built many decades ago.

In the absence of Finite Element Analysis (or other numerical analysis), development of structures must be based on hand calculations only. For complex structures, the simplifying assumptions required to make any calculations possible can lead to a conservative and heavy design. A considerable factor of ignorance can remain as to whether the structure will be adequate for all design loads. Significant changes in designs involve risk. Designs will require prototypes to be built and field tested. The field tests may involve expensive strain gauging to evaluate strength and deformation.

With Finite Element Analysis, the weight of a design can be minimized, and there can be a reduction in the number of prototypes built. Field-testing will be used to establish loading on structures, which can be used to do future design improvements via Finite Element Analysis.

The term meshing is nothing but the conversion of a CAD model to an FEA model. i.e., the discretiation of a model into a number of individual fine elements. And by doing so the model can be studied to an optimum level.

And now let us see a brief description regarding the finite element modeling. The following figure shows a typical triangulation. The points where the corners of the triangles meet are called "nodes", and each triangle formed by the three nodes and three sides are called the "elements". The elements fill the entire region except a small region at the boundary. This unfilled region exists for the curved boundaries, and it can be reduced by choosing smaller elements or elements with the curved boundaries. The idea of the finite element method is to solve the continuous problem approximately, and this unfilled region contributes to some part of this approximation.



And once the meshing part has been completed the later phase is that of the analysis. And here the suitable software selected is that of the "Hypermesh".

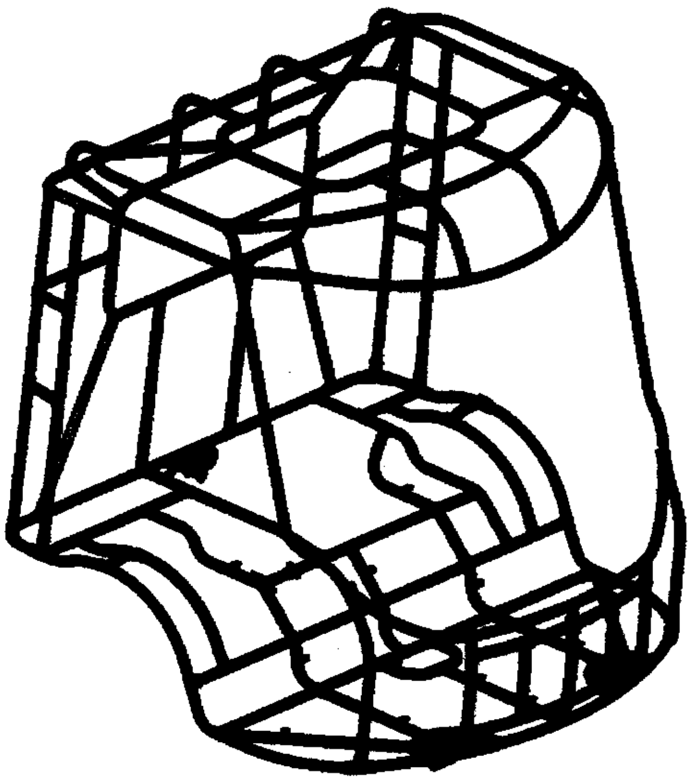


Fig 6.3 Meshed Component

6.2.4 HIGHLIGHTS OF MESHING

- The preferred type of mesh is of "QUAD".
- The connectivity between the elements to be maintained.
- Number of elements to be kept to a minimum.
- Wherever possible go for the larger element size.
- It is found that size "8" is of optimum level.

6.2.5 SOFTWARE DESCRIPTION – HYPERMESH

Altair Hypermesh is a high-performance finite element pre-processor that provides a highly interactive and visual environment to analyze product design performance.

With the broadest set of direct interfaces to commercial CAD and CAE systems, Hypermesh provides a proven, consistent analysis platform for the entire enterprise.

With a focus on engineering productivity, Hypermesh is the user-preferred environment for:

- Solid Geometry Modeling
- Shell Meshing
- Model Morphing
- Detailed Model Setup
- Surface Geometry Modeling
- Solid Mesh Generation
- Automatic mid-surface generation
- Batch Meshing

6.3 ANALYSIS

In this phase the meshed model is being used. Here the analysis is being carried out according to the standard ECE R-29. And the appropriate software selected is that of the "LS-DYNA".

6.3.1 SOFTWARE DESCRIPTION – LS DYNA

LS-DYNA is an advanced general purpose multiphysics simulation software package, actively developed by the "Livermore software technology corporation"

6.3.2 CAPABILITIES

LS-DYNA potential applications are numerous and can be tailored to many fields. LS-DYNA is not limited to any particular type of simulation. in a given simulation any of LS-DYNA's feature can be combined to model a wide range of physical events. An example of a simulation, which involves a unique combination of features, is the NASA JPL Mars pathfinder landing simulation which simulated the space probe's use of airbags to aid in its landing.

6.3.3 APPLICATION

LS-DYNA is widely used by the automobile industry to analyze vehicle design. LS-DYNA accurately predicts a car's behavior in a collision and the effects of the collision upon the car's occupants. With LS-DYNA, automotive companies and their suppliers can test car designs without having to tool or experimentally test a prototype, thus saving time and expense.

Chapter 7

Meshing

CHAPTER 7

MESHING

The following are the steps that have been followed during the meshing cycle:

7.1 GEOMETRIC CLEANUP

Here, the power of meshing begins with that of extraction of the mid surface of the given component. Since the given components are of hollow tubes, which is considered as thin sheet rolled into pipes accordingly. If the structure is of solid, then direct meshing is been carried out without the extraction of the mid surface.

Once the extraction of the mid surface has been done, the next step is that of the geometry cleanup. Once the model has been imported to the hypermesh, the original geometry suffers some data losses. These data losses causes the problems such as formation of duplicate geometry, loss of geometry, formation of unwanted surfaces etc,

The duplicate surfaces has been eliminated by means of deleting the surface, the loss of geometry has been cleared by means of using the surface command in the 2D page accordingly. Finally the formation of the unwanted surface has been deleted using the appropriate option.

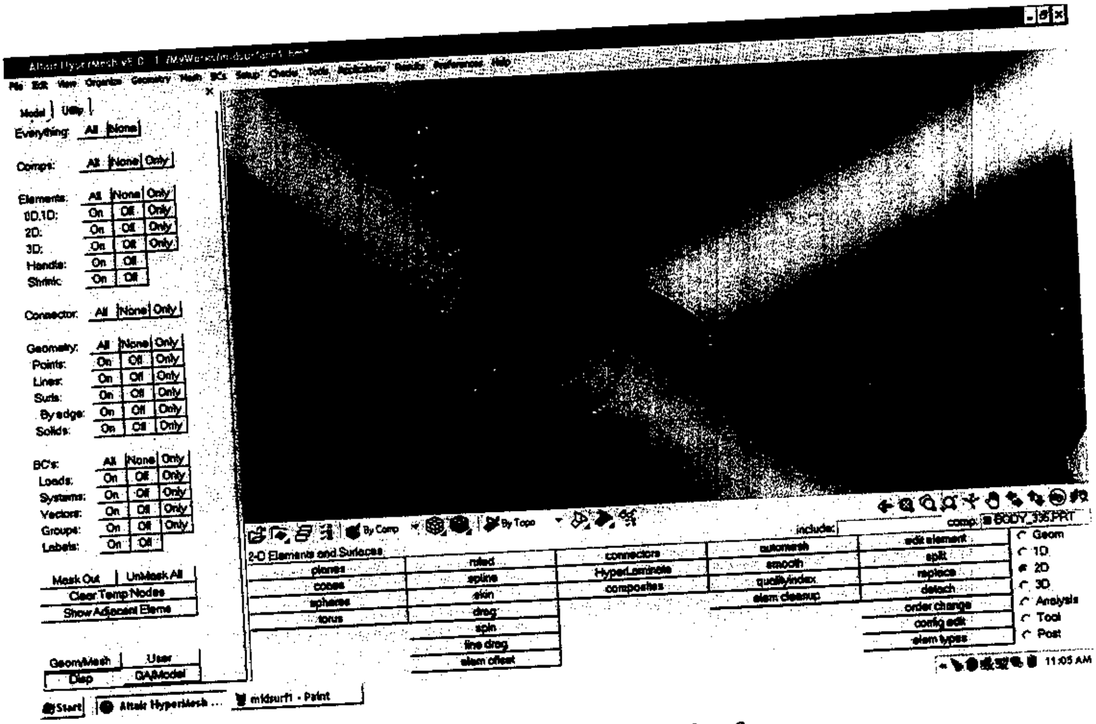


Fig 7.1 Loss of Geometric Surface

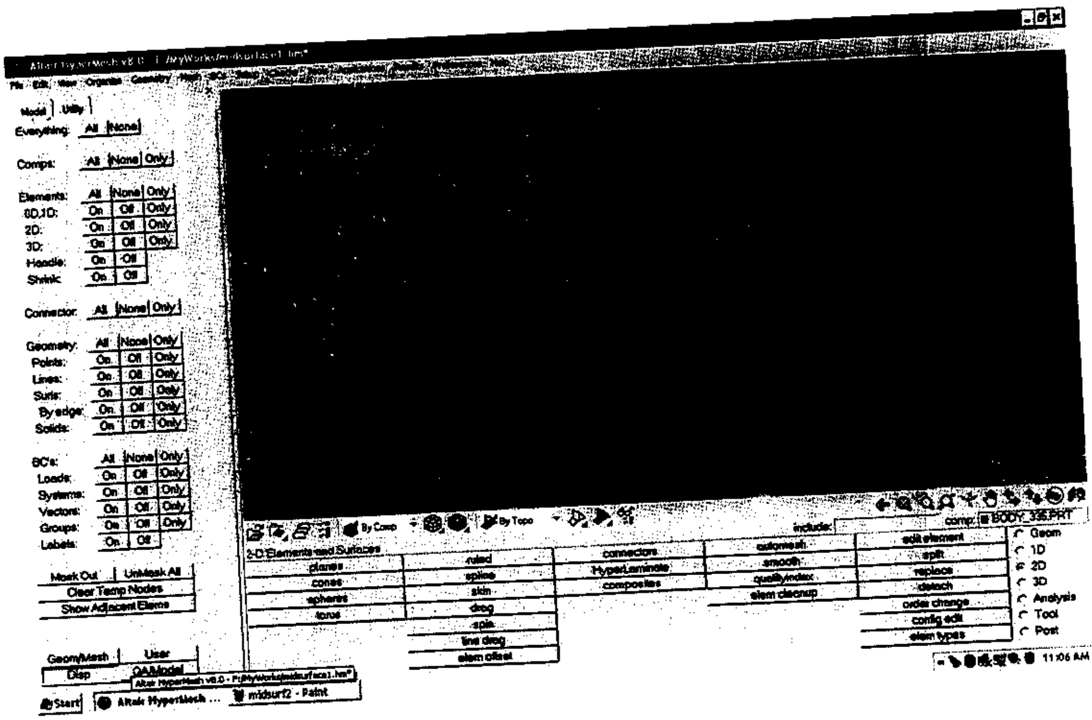


Fig 7.2 Representation of Mixed Surfaces

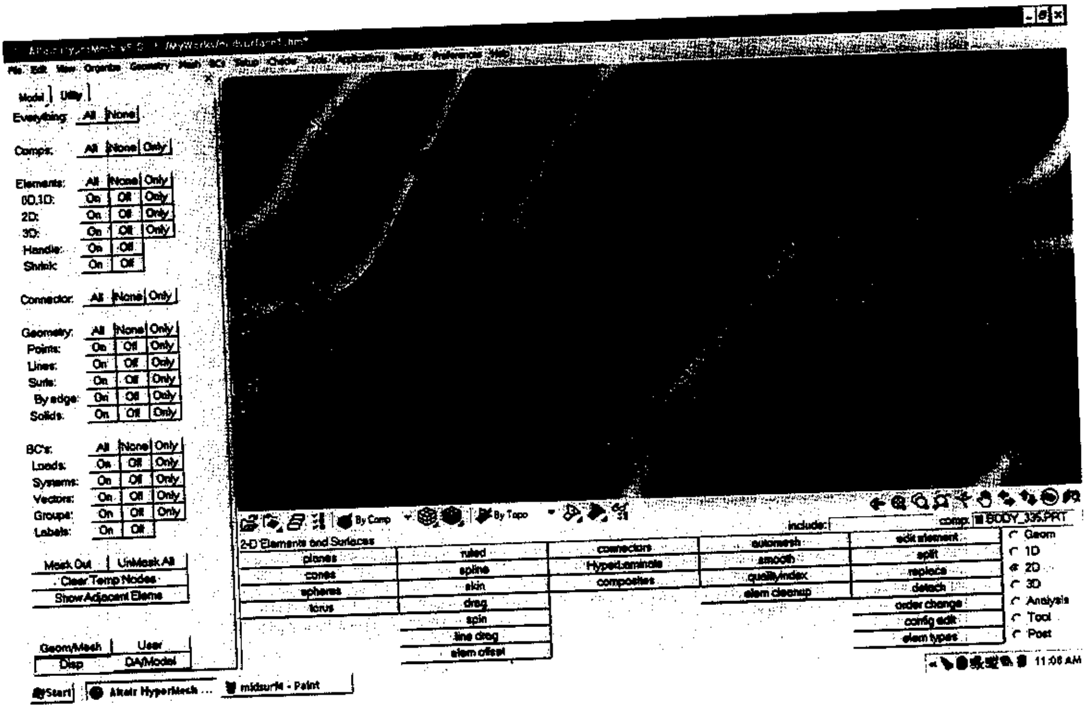


Fig 7.3 Formation of Duplicate Surfaces

7.2 MESHING

In two dimensions there is more than one simple geometric shape that can be used as a finite element. The interpolation functions depend not only on the number of nodes in the element and the number of unknowns per node, but also on the shape of the element. The shape of the element must be such that its geometry is uniquely defined by a set of points, which serve as the element nodes in the development of the interpolation functions. A triangle is the simplest geometric shape, followed by a rectangle.

The representation of a given region by a set of elements (i.e., discretization or mesh generation) is an important step in finite element analysis. The choice of element type, number of elements, and density of elements depends on the geometry of the domain, the problem to be analyzed, and the degree of accuracy desired. Of course, there are no specific formulae to obtain this information. In general, the analyst is guided by his or her technical background, insight into the physics of the problem being modeled (e.g., a qualitative understanding of the solution), and experience with finite element modeling. The general rules of mesh generation for finite element formulations include:

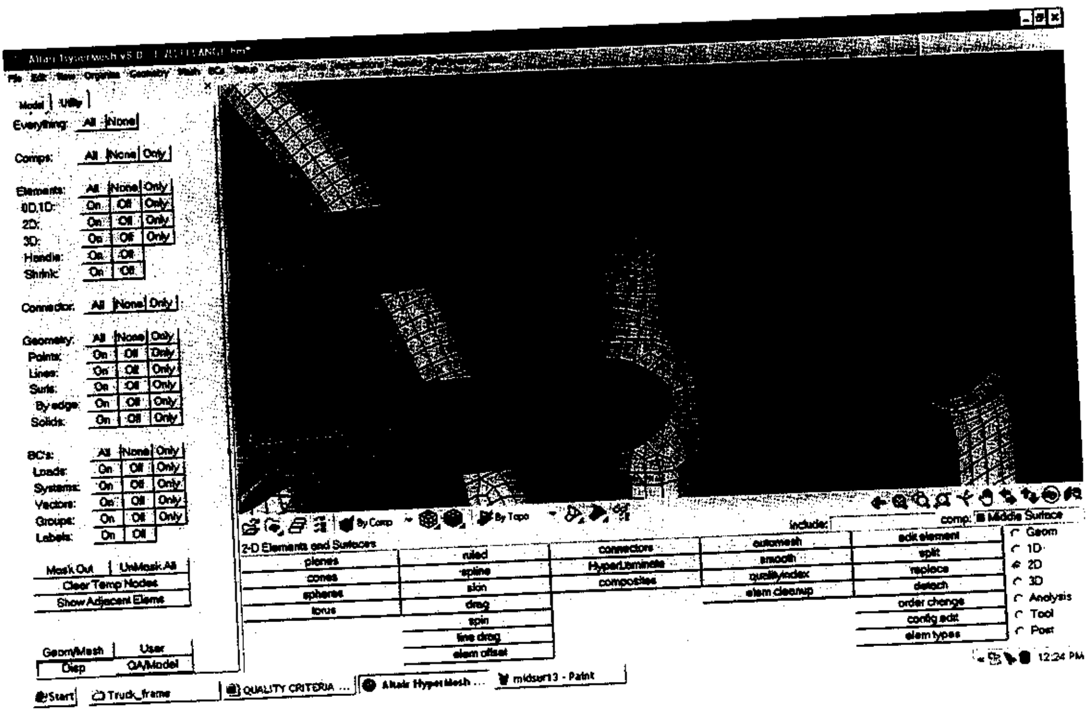
1. The elements that are selected should characterize the governing equations of the problem.

2. The number, shape, and type (i.e., linear or quadratic) of elements should be such that the geometry of the domain is represented as accurately as desired.

3. The density of elements should be such that regions of large gradients of the solution are adequately modeled (i.e., use more elements or higher-order elements in regions of large gradients).

4. Mesh refinements should vary gradually from high-density regions to low-density regions. If transition elements are used, they should be used away from critical regions (i.e., regions of large gradients). Transition elements are those that connect lower-order elements to higher-order elements (e.g., linear to quadratic).

And hence we concentrate on that of the quadratic (rectangular) element rather than that of the triangular element respectively. The reason for selecting the quadratic element is such that the number of nodes is more than that of the triangular element, which is of effective in control.



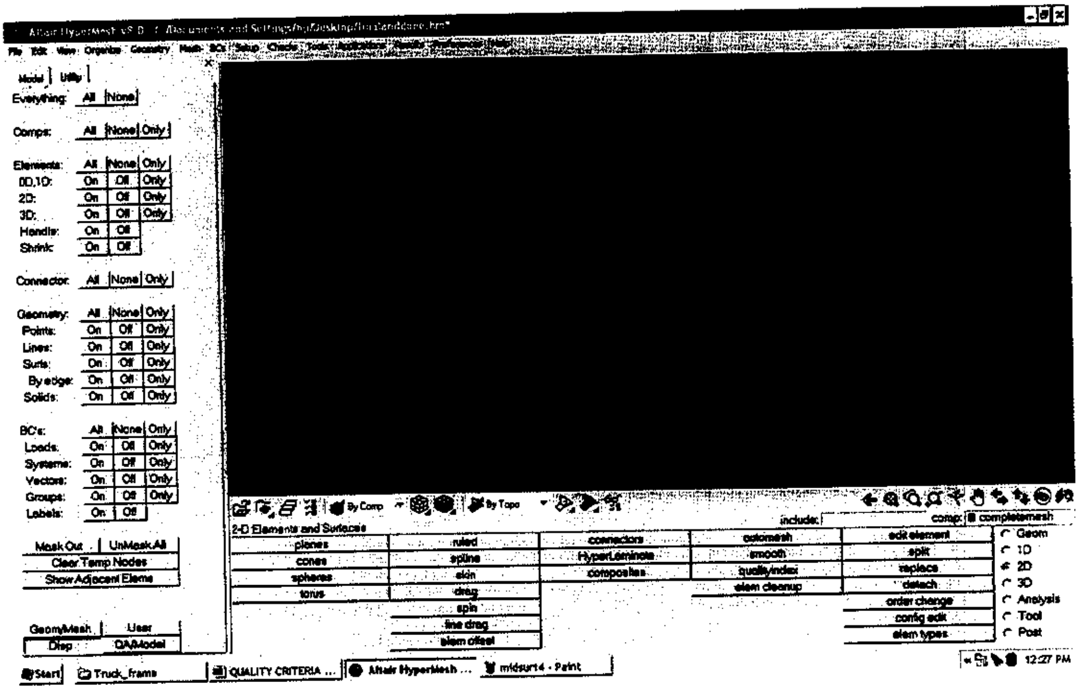


Fig 7.5 Mesh using 1D Elements

7.3 ELEMENT OPTIMIZATION

Once the machine has been done the created elements has to be checked for that of the given criteria, else in other words the optimization of the elements is required in order to obtain the expected result.

The optimization here reference to that of maintaining the required element size, and the factor also to be considered is that of reducing the number of elements, because it leads to the delayed process time. And later is that of the connectivity. The reason for maintaining the connectivity is for the transmission of the force as expected. And other important factors that are to be concentrated are such as skewness and warpage.

The term skewness and warpage has been explained as follows,

1. Skewness – It is defined as the ratio between the sides of the rectangle i.e., the ratio between the large side to that of the smaller side accordingly.

2. Warpage – It is defined as the deviation of an element from the normal flow of the elements respectively.

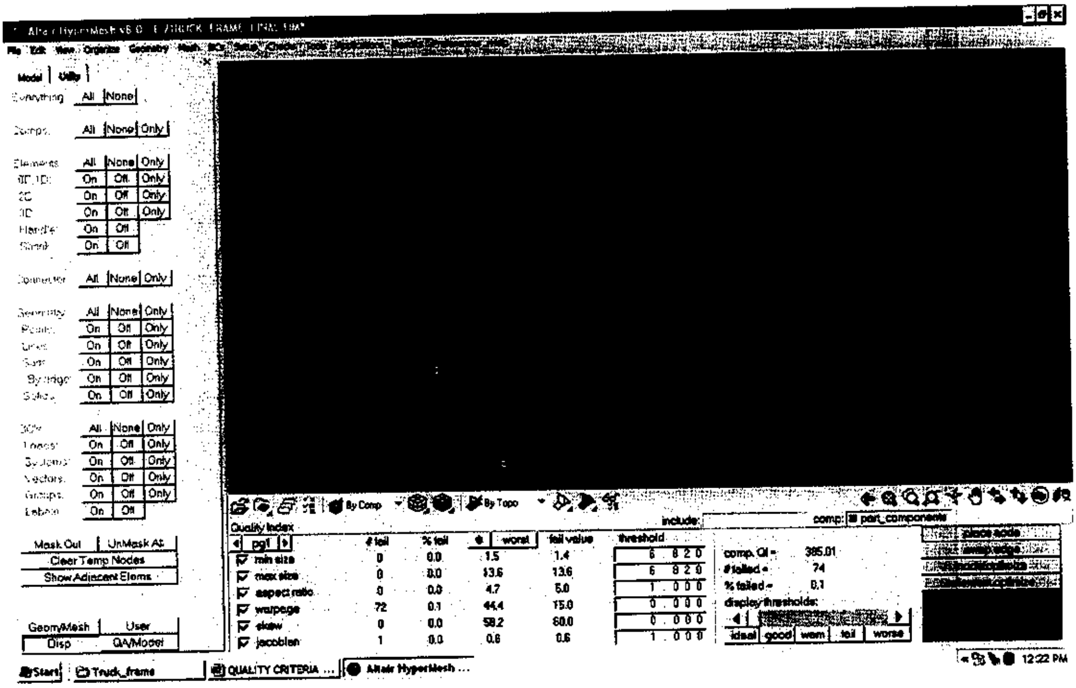


Fig 7.6 Quality Check – Initial

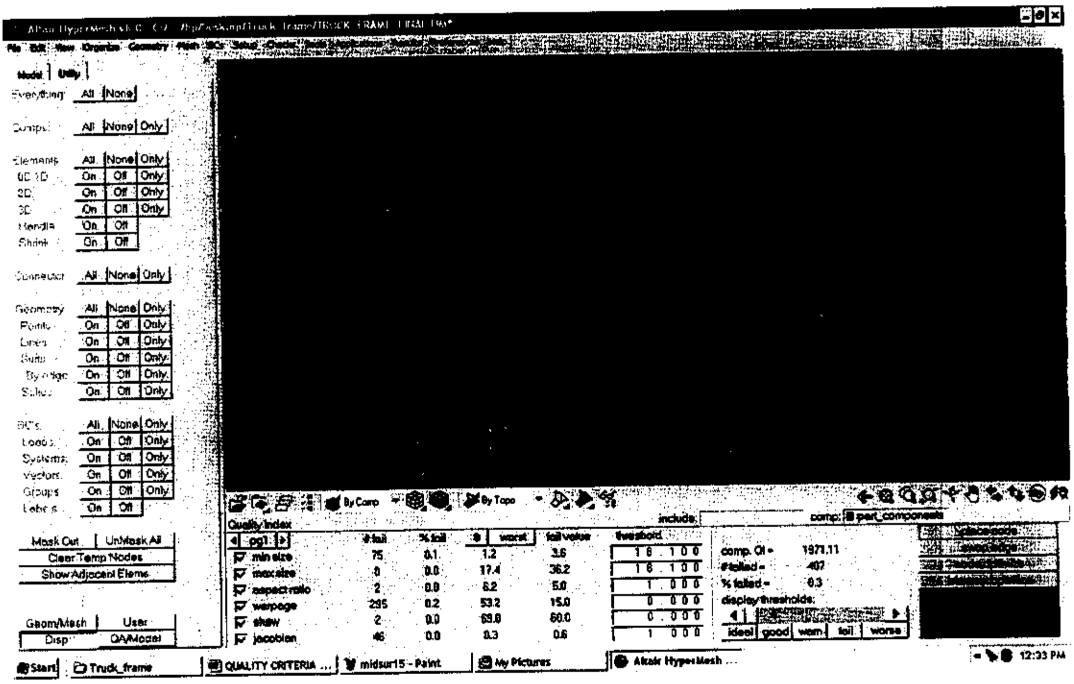


Fig 7.7 Representation of Warning Elements

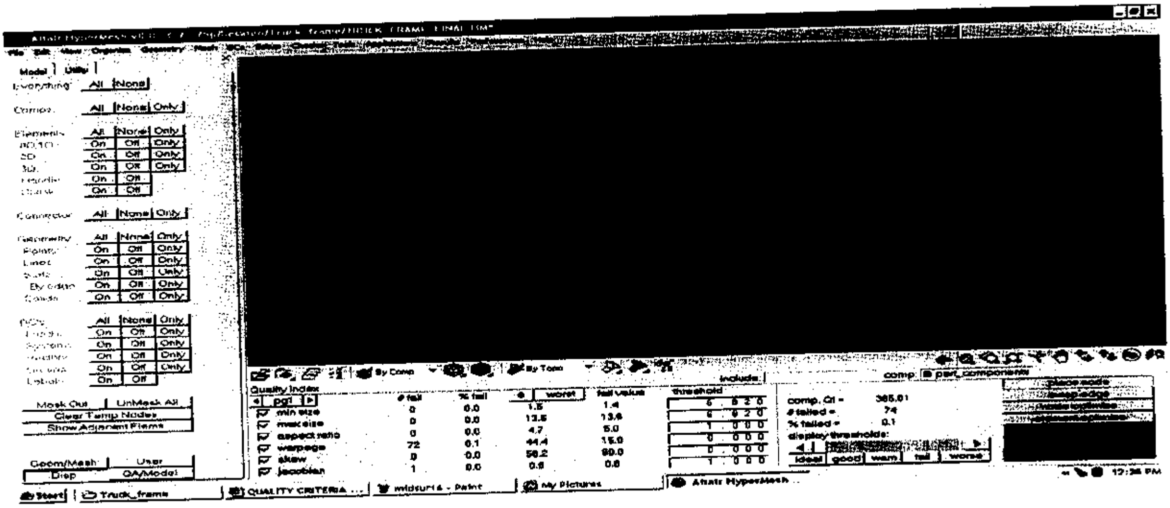


Fig 7.8 Representation of Fine Elements

7.3.1 MESH GUIDELINES

General Mesh Guideline

1. Mesh size of 10-15 mm for main structural areas, 15-20 mm for non structural areas.

2. Fine mesh required at Frontal, roof and rear wall regions where large deformations are expected for tests specified in AIS-029.

3. Smooth transition from coarse to fine mesh required. Also mesh line need to be parallel & perpendicular to material flow (shape expression mesh).

4. Element size should be uniform wherever possible.

5. Triangular elements should be avoided at all locations and especially for weld runs and at other stress raisers.

6. All shells should be at surface mid-plane.

7. Flanges to be modeled with

   a. 3 elements across flange if width is > 15 mm.

   b. 2 elements across flange if width is < 15 mm.

c. Bend Radii greater than 8mm are to have a minimum of 2 elements, 1 node, following the geometry.

d. Bend radii less than 8mm are to be modeled as a sharp corner angle.

8. Holes with

   a. Diameter < 10 mm to be ignored.

   b. Diameter > 10mm to be included and a minimum of 6 elements along circumference.

9. Initial penetration to be checked to material gauge.

7.3.2 MESH QUALITY GUIDELINES

1. Warpage < 10
2. Aspect Ratio < 1:5
3. Skew < 60
4. Jacobian > 0.7
5. Minimum angle
   a. quads 45
   b. trias 30
6. Maximum angle
   a. quads 135
   b. trias 120
7. Maximum trias per component - 5%

Chapter 8

Theory Concepts

CHAPTER 8

THEORY CONCEPTS

8.1 CONTACT MODELING IN LS-DYNA

Contact treatment forms an integral part of many large-deformation problems. Accurate modeling of contact interfaces between bodies is crucial to the prediction capability of the finite element simulations. LS-DYNA offers a large number of contact types. Some types are for specific applications, and others are suitable for more general use. Many of the older contact types are rarely used but are still retained to enable older models to run as they did in the past. Users are faced with numerous choices in modeling contact. This document is designed to provide an overview of contact treatment in LS-DYNA and to serve as a guide for choosing appropriate contact types and parameters.

8.2 HOW CONTACT WORKS

In LS-DYNA, a contact is defined by identifying (via parts, part set, segment sets, and/or node sets) what locations are to be checked for potential penetration of a slave node through a master segment. A search for penetrations, using any of a number of different algorithms, is made every time step. In the case of a penalty-based contact, when a penetration is found a force proportional to the penetration depth is applied to resist, and ultimately eliminate, the penetration. Unless otherwise stated, the contacts discussed here are penalty-based contacts as opposed to constraint-based contacts. Rigid bodies may be included in any penalty-based contact but in order that contact force is realistically distributed, it is recommended that the mesh defining any rigid body be as fine as that of a deformable body.

... effective to define a single contact that any number of contacts in a single model. It is generally recommended that redundant contact, i.e., two or more contacts producing forces due to the same penetration, be avoided by the user as this can lead to numerical instabilities.

To enable flexibility for the user in modeling contact, LS-DYNA presents a number of contact types and a number of parameters that control various aspects of the contact treatment.

8.3 CONTACT TYPES

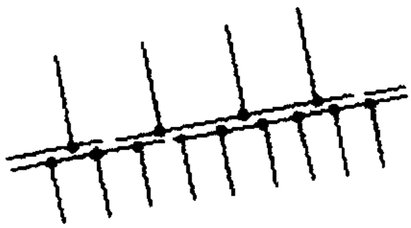
8.3.1 SINGLE SURFACE

These contact types are the most widely used contact options in LS-DYNA, especially for crashworthiness applications. With these types, the slave surface is typically defined as a list of part ID s. No master surface is defined. Contact is considered between all the parts in the slave list, including self-contact of each part. If the model is accurately defined, these contact types are very reliable and accurate. However, if there are a lot of interpenetrations in the initial configuration, energy balances may show either a growth or decay of energy as the calculation proceeds.

Type 1: *CONTACT_SLIDING_ONLY

Only sliding
No separation

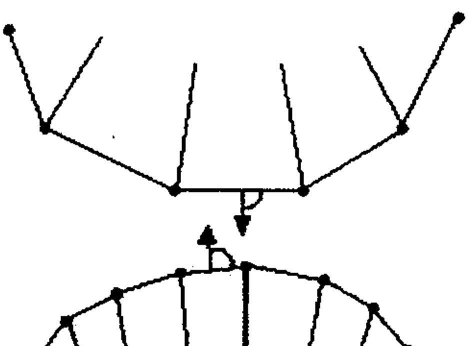
Type 2: *CONTACT_TIED_SURFACE_TO_SURFACE



Tying surfaces with translational degrees od freedom.

Distance between nodes an surface is been preserved with the OFFSET option.

Type 3: *CONTACT_SURFACE_TO_SURFACE



Each slave node is checked for penetration through the master surface

For shell surfaces segment normal orientation is important

Type 4: *CONTACT_SINGLE_SURFACE
Type 13: *CONTACT_AUTOMATIC_SINGLE_SURFACE
Type a13: *CONTACT_AIRBAG_SINGLE_SURFACE
Type 26: *CONTACT_AUTOMATIC_GENERAL
Type i26: *CONTACT_AUTOMATIC_GENERAL_INTERIOR

Fig 8.1 (a) Contact types

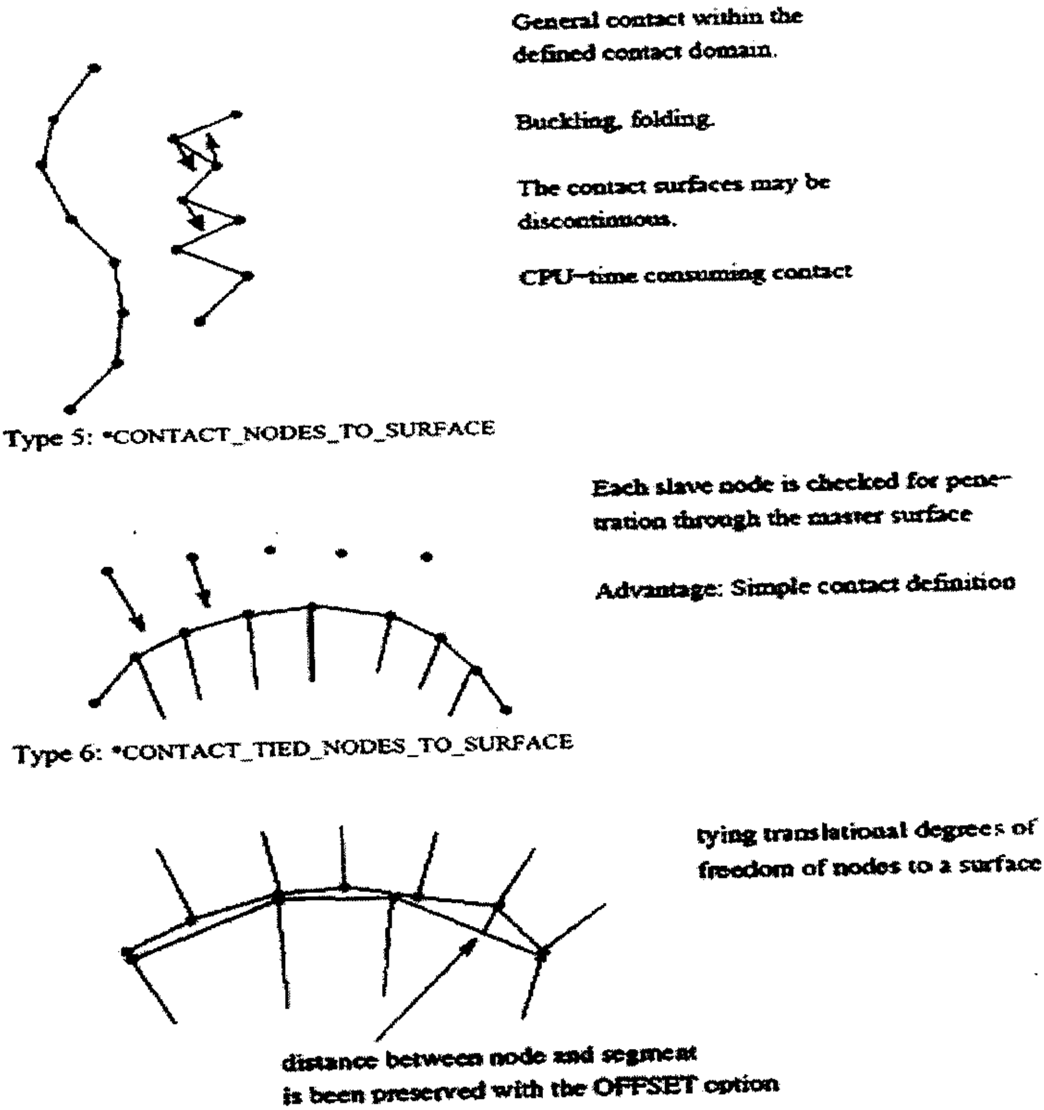
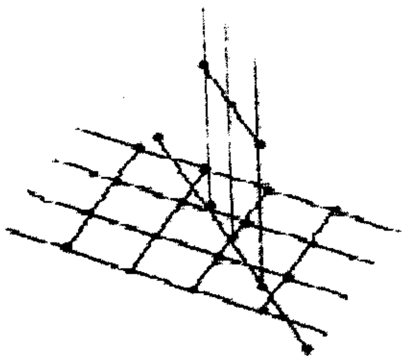


Fig 8.1 (b) Contact types

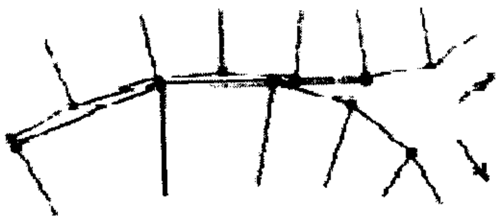
moments are not transmitted
(hinge)

Type 7: *CONTACT_TIED_SHELL_EDGE_TO_SURFACE



tying both translational and rotational
degrees of freedom of nodes to a surface

distance between node and surface
is been preserved with the options:
CONSTRAINED_OFFSET
BEAM_OFFSET



like type 6 but with failure criteria

failure criterion: (fn/Sn)^n+(fs/Ss)^m >= 1
fn ... normal interface force (nonzero for tensile values only)
fs ... shear interface force

Fig 8.1 (c) Contact types

For crash analysis, the contact type *CONTACT_AUTOMATIC_SINGLE_SURFACE is recommended. This contact has improved from version to version of LS-DYNA and is the most popular contact option.

The older single surface contact type *CONTACT_SINGLE_SURFACE should be avoided since it has not undergone improvement. It eventually will be removed or recoded. The difference between *CONTACT_SINGLE_SURFACE and *CONTACT_AUTOMATIC_SINGLE_SURFACE are twofold.

First, the older method uses nodal based bucket sorting where closest nodes are down if the segments vary appreciably in size and shape, especially, if aspect ratios are large. Secondly, the older method uses segment projection to determine the contact surface. This requires the calculation of nodal normal vectors that are area weighted by the segments that share the node, which in turns creates further difficulties for T-intersections and other geometric complications. The calculation of the vectors can require 25% of the total CPU required.

For modeling the deployment of airbags the following contact option is recommended: *CONTACT_AIRBAG_SINGLE_SURFACE.

With *AIRBAG_SINGLE_SURFACE, contact between nodes and multiple segments is considered. Much more searching is done than in the normal contact option and, consequently, this contact option is much more expensive. During the past several years, the soft constraint, on optional card A, in the contact definition, set to 2 has proved to deploy airbags very accurately. We current recommend this option for airbag deployment. The latter option is currently being implemented for MPP usage.

The final contact is: *CONTACT_AUTOMATIC_GENERAL.

The contact treatment with this option was similar to type 13 through the 950c release of LS-DYNA. The main difference was that three possible contact segments, rather than just two, were stored for each slave node. With 950d and later versions, type 13 was substantially improved and now type 13 is frequently more accurate. The main feature of the GENERAL option is that shell edge-to-edge and beam-to-beam contact is treated automatically. All free edges of the shells and all beam elements are checked for contact with other free edges and beams. Unlike type 13 contact, type 26 contact checks for contact along the entire length of beams and exterior shell edges, not just at the nodes. There is a new option in 960 to also check internal shell edges (INTERIOR option). This is quite expensive, however, and is not usually needed. We plan to update this contact type in version 970 of LS-DYNA to include all the recent improvement in the *AUTOMATIC_SINGLE_SURFACE contact.

8.4 CONTACT ENTITY

This contact type is used for treating deformable nodes against rigid geometric surfaces. The analytical equations defining the geometry of the surface are used in the contact calculations. This is an improvement over the usual segmented surface as represented by a mesh. A penalty-based approach is used in calculating the forces that resist penetration. This contact type is widely used to couple LSDYNA with rigid body dummies, which have surfaces approximated by nice geometric shapes such as ellipsoids. An automatic mesh generator is used to mesh the rigid surfaces to aid visualizing the results. The mesh is not used in the contact calculations. The analytical rigid surfaces can be of the following types:

- Flat Planes (infinite and finite)
- Sphere
- Cylinder
- Hyper-ellipsoid
- Torus
- Load curve defining the line
- CAL3D/MADYMO plane
- CAL3D/MADYMO ellipsoid
- VDA surface (red from a file)
- IGES surface (red from a file)

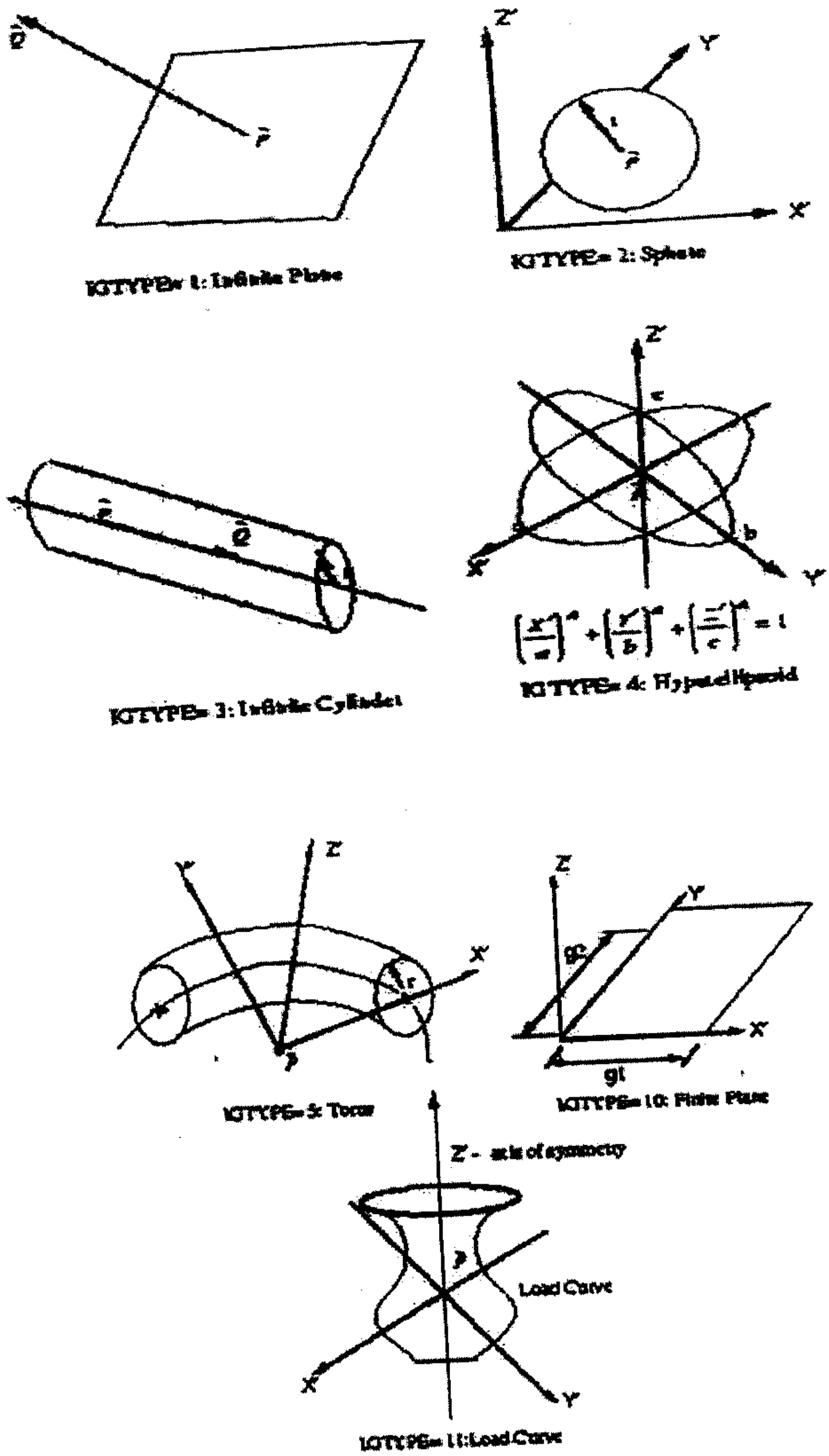


Fig 8.2 Contact entity

8.5 CONTACT STIFFNESS CALCULATION

Contact treatment is internally represented by linear springs between the salve nodes and the nearest master segments. The stiffness of these springs determines the force that will be applied to the slave nodes and the master nodes. There are currently two methods of calculating the contact spring stiffness and they are briefly discussed below.

Penalty-based approach (SOFT=0 **in Optional Card A in** '*CONTACT_')

The formula for the stiffness of a contact segment is as follows:

$$k = \frac{f_s x Area^2 x K}{Volume}$$ for segments on solid elements

$$k = \frac{f_s x Area x K}{Minimum Diagonal}$$ for segments on shell elements

Area	=	area of contact segment
k	=	bulk modulus of contacted element
f_s	=	SLSFAC x SFS (or SLSFAC X SFM)

SLSFAC.....penalty scale factor (0.1 by default)

SFS / SFM.....scale factor on default slave / master penalty stiffness

This method is the default method and uses the size of the contact segment and its material properties to determine the contact spring stiffness. As this method depends on the material constants and the size of the segments, it works effectively when the material stiffness parameters between the contacting surfaces are of the same order-of-magnitude. In cases where dissimilar materials come into contact, the contact might break down, as the stiffness, which is roughly the minimum of the slave and master stiffness, maybe too small. This frequently happens with soft dense foams contact metal materials. Consequently, for crash analysis we do not recommend the option, SOFT = 0, unless prior experience shows that no problems occur.

Soft Constraint-based approach (SOFT=1&2 **on Optional Card A in** *Contact_)

This non-default method calculates the stiffness of the linear contact springs based on the nodal masses that come into contact and the global time step size. The resulting contact stiffness is independent of the material constants and is well suited for treating contact between bodies of dissimilar materials. The stiffness is found by taking the nodal mass divided by the square of the time step size with a scale factor to ensure stability.

$$k = (SOFSCL)\frac{m}{\Delta t^2} \quad \{(0.1) \text{ by default}\}$$

Generally, for the case of metals contacting metals the resulting penalty stiffness for SOFT = 0 or SOFT = 1 is similar. For the case where soft dense foams contact metal, the option, SOFT = 1 often gives interface stiffness that are one or two orders-of-magnitude greater. The SOFT = 1 option is recommended for impact analysis where dissimilar materials come into contact.

The soft = 2 option uses mass and time step based penalty stiffness as in SOFT = 1. SOFT = 2 invokes a segment-based contact algorithm which has it origins in Pinball contact developed by Belytschko and his co-workers. With this contact algorithm, contact between segments is treated rather than using the usual node-to-segment treatment. When two 4-noded segments come into contact, forces are applied to eight nodes to resist segment penetration. This treatment has the effect of distributing forces more realistically and sometimes is quite effective for very stubborn contact problems. The SOFT = 2 option is currently being ported for MPP calculations. Beam contact is not handled by SOFT = 2 type contact. Further, SOFT = 2 is available only for surface-to-surface and single surface contacts and not for nodes-to-surface contacts. The optional parameter EDGE on Optional Card A should be used cautiously when segment-edge-to-segment-edge contact is anticipated and SOFT is set to 2.

8.6 CONTACT OUTPUT

... files pertaining to contact, which can be written by LScommon contact-related output file, RCFORC, is produced by including a *DATABASE_RCFORC command in the input deck. RCFORC is an ASCII file containing resultant contact forces for the slave and master sides of each contact interface. The forces are written in the global coordinate system. Note that RCFORC data is not written for single surface contacts as all the contact forces from such a contact come from the slave side (there is no master side) and thus the net contact forces are zero. To obtain RCFORC data when single surface contacts are used, one or more force transducers should be added via the *CONTACT_FORCE_TRANSDUCER_PENALTY command. A force transducer does not produce any contact forces and thus does not affect the results of the simulation. A force transducer simply measures contact forces produced by other contact interfaces defined in the model. One would typically assign a subset of the parts defined in a single surface contact to the slave side of a force transducer. No master side is defined. The RCFORC file would then report the resultant contact forces on that subset of parts.

The ASCII output file NCFORC reports contact forces at each node. The command *DATABASE_NCFORC is required in the input deck to produce such a file. Further, one or more contact print flags must be set. Only those surfaces whose print flag is set to a value of 1 will have their nodal contact force output to the NCFORC file. By including a *DATABASE_SLEOUT command, contact interface energies are written to the ASCII output file SLEOUT. In cases where there are two or more contact interfaces in a model and the global statistics file (GLSTAT) indicates a problem with contact energy, such as a large negative value, the SLEOUT file is useful for isolating which contact interfaces are responsible. In some cases, it can be very useful to visualize contact surfaces and produce fringe plots of contact stress both in directions normal and tangential to the contact surface. To do this, a binary interface file must be written by

1. Including a *DATABASE_BINARY_INTFOR command in the input deck.
2. Setting one or more contact print flags as detailed above.
3. And including the option s=filename on the LS-DYNA execution line where

Chapter 9

Mathematical Concepts

CHAPTER 9

MATHEMATICAL CONCEPTS

- In a typical crash simulation, the body structure is analyzed using spatial discretization.

- The discretization involves subdividing the surface of the constituent, thin, sheet metal parts into a large number (approaching one million in 2006) of quadrilateral or triangular regions, each of which spans the area between "nodes" to which its corners are fixed.

- Each element has mass, which is distributed as concentrated masses and as mass moments of inertia to its connecting nodes.

- A simple definition of the moment of inertia of any object, be it a point mass or a 3D-structure, is given by:

$$I = \int r^2 \, dm$$

  where

  m is the mass, and
  r is the (perpendicular) distance of the point mass to the axis of rotation.

- Each node has 6 kinematic degrees of freedom.

- The forces and moments of all nodes are collected into a column vector (or column matrix), and the time dependent equations of motion (in dynamic equilibrium) can be written as follows.

where vector $\mathbf{Ma}$ (mass times acceleration vector) collects the inertia forces at the nodes, $\mathbf{F}_{ext}$ collects the external nodal loads, and $\mathbf{F}_{int}$ collects the internal resisting forces from the deformation of the material.

- *M* is a diagonal matrix of the nodal masses. Each vector (*u*, *v*, *a*, *F*, etc.) has dimension 6 times the total number of nodes in the crash model (about 6 million "degrees of freedom" for every 1 million "nodes" in 3-D thin shell finite element models).

- In mathematics the **dimension** of a space is roughly defined as the minimum number of coordinates needed to specify every point within it.

- Classical physics theories describe three physical dimensions: from a particular point in space, the basic directions in which we can move are up/down, left/right, and forward/backward.

- In its simplest form: a line describes one dimension, a plane describes two dimensions, and a cube describes three dimensions.

- In linear algebra, a **column vector** is an $m \times 1$ matrix, i.e. a matrix consisting of a single column of m elements.

$$\mathbf{x} = \begin{bmatrix} x_1 \\ x_2 \\ \vdots \\ x_m \end{bmatrix}$$

  The transpose of a column vector is a row vector and vice versa.

- In physics, **equations of motion** are equations that describe the behavior of a system (e.g., the motion of a particle under an influence of a force) as a function of time. Sometimes the term refers to the differential equations that the system satisfies (e.g., Newton's second law or Euler–Lagrange equations), and sometimes to the solutions to those equations.

The equations that apply to bodies moving linearly (that is, one dimension) with uniform acceleration are presented below

Linear equations of motion

The body is considered at two instants in time: one "initial" point and one "current". Often, problems in kinematics deal with more than two instants, and several applications of the equations are required.

$$v_f = v_i + a\Delta t$$

$$d = \frac{1}{2}(v_i + v_f)\Delta t$$

$$d = d_i + v_i\Delta t + \frac{1}{2}a\Delta t^2$$

$$v_f^2 = v_i^2 + 2ad$$

Where, d_i is the body's initial position, v_i is the body's initial velocity and its current state is described by:

d, the distance travelled from initial state (displacement)

v_f, The final velocity

Δt, the time between the initial and current states

a, the constant acceleration, or in the case of bodies moving under the influence of gravity, g.

Note that each of the equations contains four of the five variables. When using the above formulae, it is sufficient to know three out of the five variables to calculate the remaining two.

Chapter 10

Analysis

CHAPTER 10

ANALYSIS

The following are the steps that have been followed for analysis using LS-DYNA:

Step-1 : From the user profile display select "LS-DYNA'

Step-2 : Retrieve the model using import option

Step-3 : Define the material using material collectors. Create/edit apply the required properties such as young's modulus, Poisson's ration, Density etc.

Step-4 : Define the property. Define the property using property collectors such as solid lement, shell element etc.

Step-5 : Updating material and properties to the component.

Note: The specific materials and properties are to the component assigned.

Step-6 : The entity set panel allows you to create update and review named sets of entities. The entity sets (Nodes, Elements, Material, Components, Properties) can be created and retrieve during the application of constraints, forces, loads to the component.

Step-7 : Forces and boundary conditions (or) load collectors:- This is used to specify the forces, velocities, loads and constraints to the components.

Note: This can be down by retrieving the entity sets created above.

Step-8 : Contacts:- (a) Set segment : It is used to create pyramid elements. To

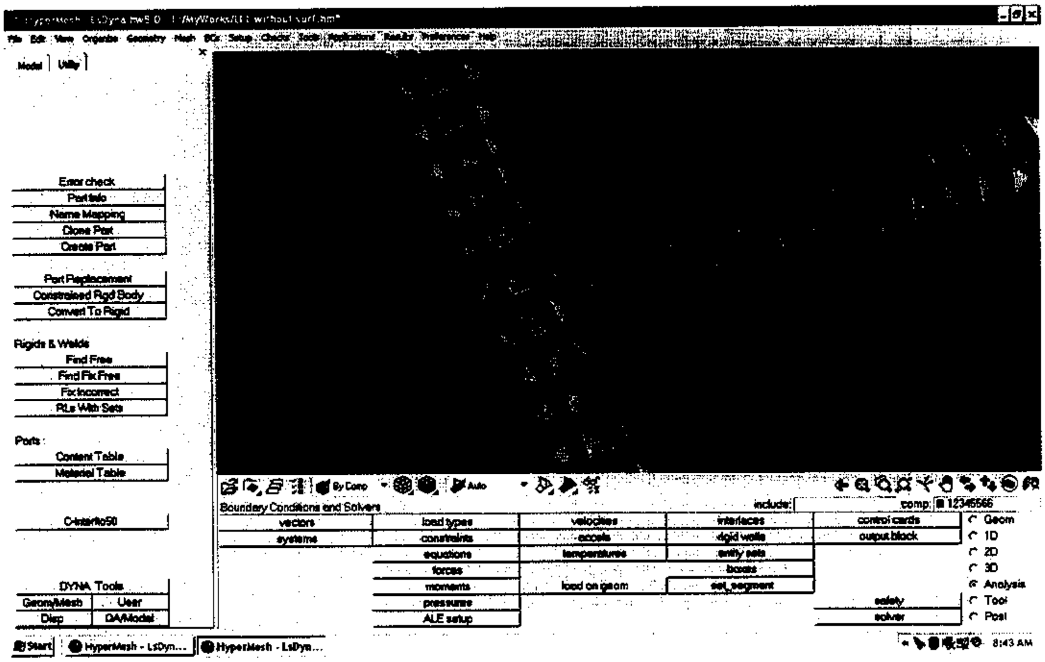


Fig 10.1 Representation of the Pyramid Structure

Step-9 : Control Cards:- This is used to control the output results.

E.g.: Control Termination ⟶ This specifies the end time of the Analysis.

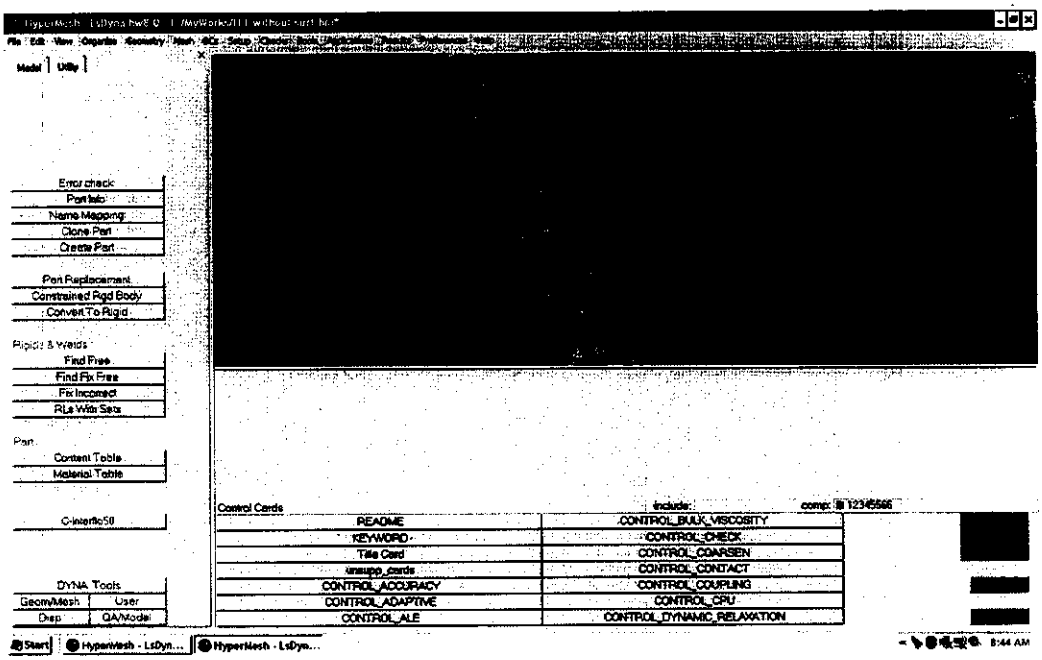


Fig 10.2 Control Cards

Step-10 : Solving:- The created deck is saved as key files and solved using Dyna solver.

Chapter 11

Results and Discussions

CHAPTER 11

RESULTS ANS DISCUSSION

The following are some of the results that have been obtained and are discussed as follows:

11.1 FRONTAL IMPACT TEST

Input Parameters

Area of the rigid body = 6 m^2

Weight of the rigid body = 2 ton's

If payload = 5000 kg then E = 15 kj/m

Height of the rigid body from ground = 1400 mm

Height of the rigid body from the reference point = 150 mm

Output – Discussion

For the given set input parameters the obtained result has been rejected. Hence it has been suggested to raise the frame of the cabin at the frontal region of about 0.5 m towards the top.

Reasons

- The cabin was not sufficient for free access of mankin.
- The joints have been sheared in case of the adhesive joints.

11.2 REAR IMPACT TEST

Input Parameters

Area of the rigid body = 12 m^2

Weight of the rigid body = 1.5 ton's

Output – Discussion

For the given set input parameters the obtained result has been accepted. As the load applied has been uniformly transmitted over the entire span of the surface, it was possible to with the test accordingly.

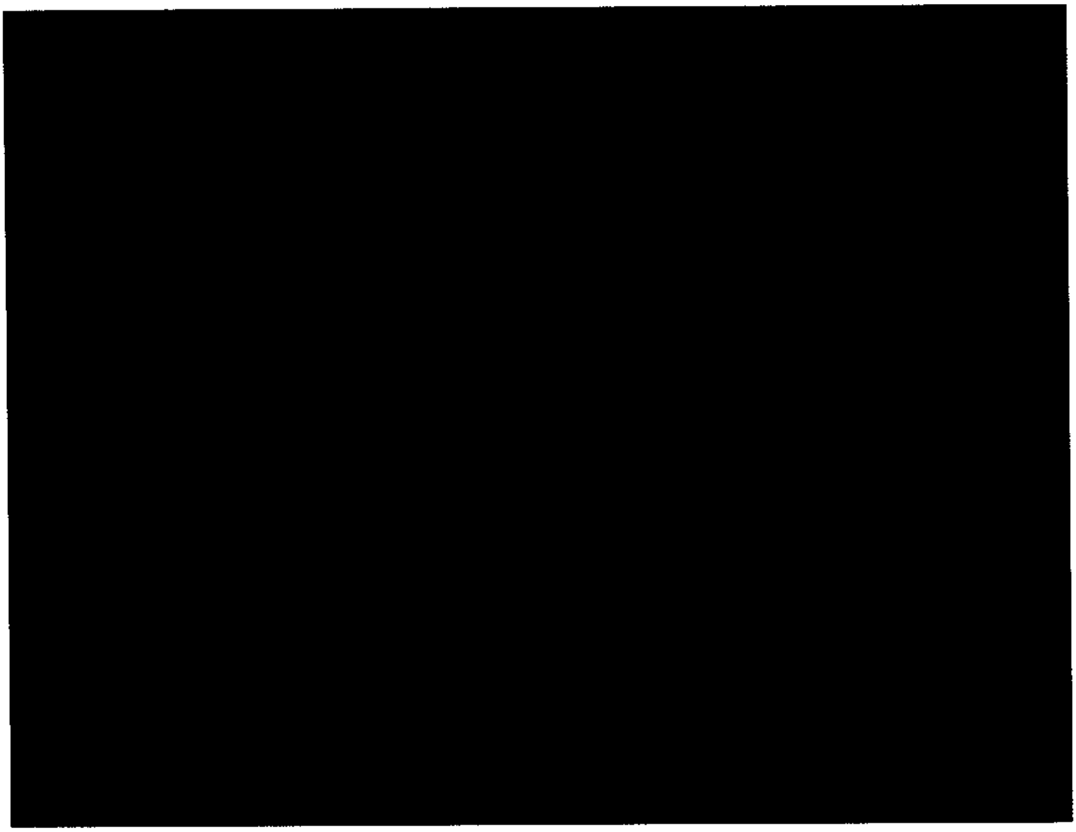


Fig 11.1 Rear Wall

11.3 ROOF STRENGTH TEST

Input Parameters

- Loading condition – Uniformly distributed load.
- Applied pressure = Max load of the front axel.

Hence, p = 50 kg

Output – Discussion

For the given set input parameters the obtained result has been accepted.

Note: for the UDL type

The reasons are such as the uniform distribution of the load over the entire span and the use of hollow cylindrical pipes.

Thus the obtained results and the related discussions are made accordingly.

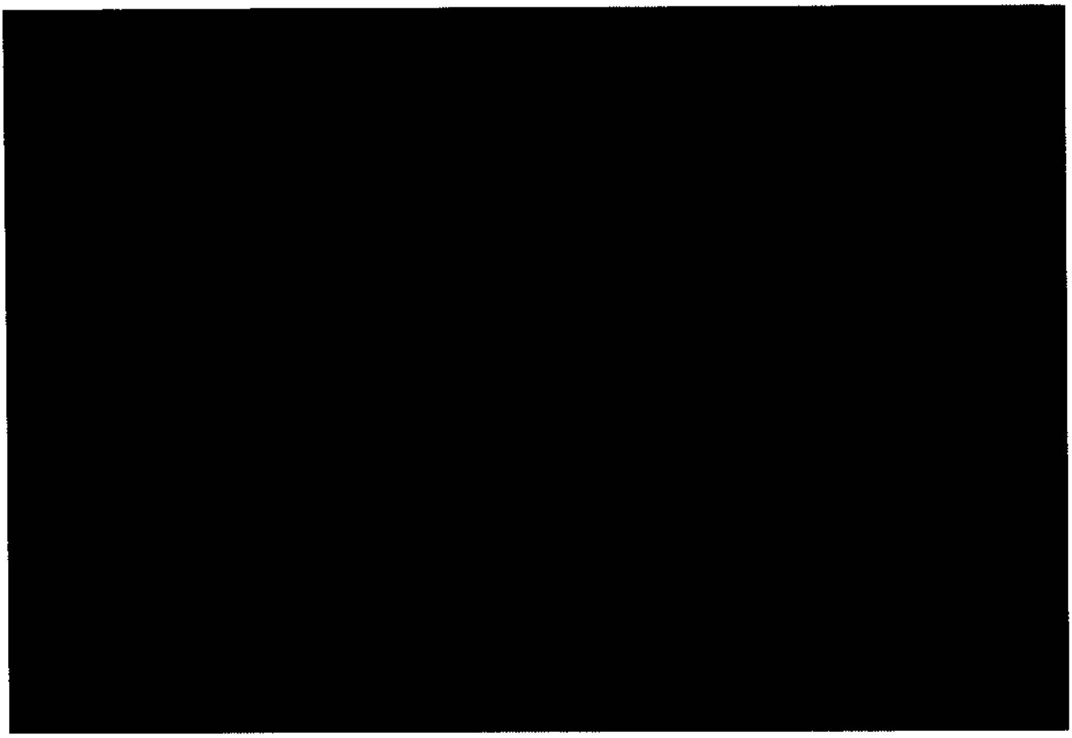


Fig 11.2 Roof

Chapter 12

Conclusions

CHAPTER 12

CONCLUSION

- In the present years the rate of accidents has increased to an alarming height. And hence the importance to the aspect of safety is very important, and one such way of doing so is by improving the design.

- The test preformed is by means of keeping truck as stationary but it is not so in case of reality.

- The transfer of model in the STEP format is found optimum than that of the IGES format.

- By doing so it is possible for us to reduce the loss of geometry, formation of duplicate surfaces etc.

- During meshing number of elements are to be kept to a minimum in order to reduce the processing time.

- Elements size that was found optimum was 2^n (e.g., 8). But to the size of 15 is being allowed.

- The use of quadratic elements is found more effective than that of the triangular element, as it is of every effective to control.

- The rigid welded joints are found more effective than that of the adhesive type of joints.

- The factors such as skewness and warpage are to be kept to an minimum, an error percentage of less than 5 percent is considered acceptable.

- The presences of warning elements represented by means of blue color are

REFERENCES

1. AIS-029 (2004), *Survival Space for the Protection of the Occupants of the Cab of a Commercial Vehicle,* Automotive Industry Standards Commitee

2. Annamaria R.Varkonyi, Koczy, Andras Brid (2007), *Intelligent Methods for Car Deformation Modeling and Crash Speed Estimation.*

3. Castro I.J., Michael Jokuszies (2001), *Simulation of Occupant Response in the ECE-R29 Safety Test.*

4. Central Institute of Road Transport (2004), *Road Safety Book of Readings.*

5. Erich Mayrhofer, Hermann Steffan (2005), *Enhanced Coach and Bus Occupant Safety.*

6. Horst Raich, DaimlerChrysler AG, Stuttgart (March 2001), *Safety Analysis of the New Actors MegaSpace Cabin According to ECE-R29/02.*

7. Kim Parnell. T, M.Asce Christopher V.White (January 2002), *Finite Element Simulation of 180° Rollover for Heavy Truck Vehiles.*

8. National Transportation Research Center (September 2005), Heavy *Vehicle Infrastructure Asset Interaction and Collision.*

9. Sadhusingh Dr. (2004), *Computer Aided Design and Manufacturing,* pp. 105 – 108

10. Seshu P., *Text Book Of Finite Element Analysis,* pp. 145 - 153

11. Steven M.Kirkpatrick, Robert Mac Neill (November 2002), *Development of an LS-DYNA Occupant Model for Use in Crash Analysis of Road Side Safe Feature.*

12. Taisuke Wantanabe, Shigeru Hirayamar (March 2005), *Research on Stiffness Matching Between Vehicles for Frontal Impact Compactibility.*

13. Thirupathy R.Chandrupatla, Ashok D.Belegundu (2004), *Introduction to Finite Elements in Engineering,* pp. 131 – 133.