ADDIS ABABA UNIVERSITY

ADDIS ABABA INSTITUTE OF TECHNOLOGY
SCHOOL OF MECHANICAL AND INDUSTRIAL ENGINEERING

Failure Analysis of Hydraulic Hammer Tool by Finite Element Method.

By
Agegnehu Tamirat

A thesis is submitted to the school of Graduate School of Addis Ababa University in partial fulfillment of the requirement for Master Of Science in Mechanical Engineering (Mechanical Design Stream)

Advisor: Dr. Daniel Tilahun

2018 G.C
Addis Ababa

Addis Ababa institute of Technology

School of Mechanical and Industrial Engineering

Failure Analysis of Hydraulic Hammer Tool by Finite Element Method.

By

Agegnehu Tamirat

A THESIS SUBMITTED TO ADDIS ABABA UNIVERSITY, SCHOOL OF MECHANICAL AND INDUSTRIAL ENGINEERING GRADUATE STUDIES IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF MASTER'S OF SCIENCE IN MECHANICAL ENGINEERING (Mechanical Design Stream)

JUNE, 2018

ADDIS ABABA, ETHIOPIA

Addis Ababa University

Addis Ababa institute of Technology

School of Mechanical and Industrial Engineering

Failure Analysis of Hydraulic Hammer Tool by Finite Element Method.

By

Agegnehu Tamirat

APPROVED BY BOARD OF EXAMINERS

______________________	______________________	______________________
Chairman of the School	Signature	Date
Dr. Daniel Tilahun	______________________	______________________
Adviser	Signature	Date
______________________	______________________	______________________
Internal Examiner	Signature	Date
______________________	______________________	______________________
External Examine	Signature	Date

Acknowledgement

It is my pleasure to thank all those who have in one way or another contribute to this thesis .First ,to my parents I express my heartfelt appreciation for their constant inspiration ,encouragement and patience.

My special thanks go to my advisor Dr.Daniel Tilahun who helped me in shaping of this thesis by his thoughtful reviews and criticisms for the thesis and his many valuable suggestions. I express special appreciating to the staff member of AAIT school of Mechanical and Industrial Engineering, for their continuous suggestion and providing me the necessary material. I would like to thank especially to Mr. Muluken Masresha who helped and provide me a target full and precise taught to follow the right path for solving and analyzing this thesis.

I am grateful to all staff of Ries Engineering Share Company. Specifically, I want to express to all Machinery Sales team and product support department for providing me the necessary material and their valuable advice.

I also have special thanks to all my friends for their continuous advice to be motivated in my work.

Contents

List of Figure

List of Table

Nomenclature

$\boldsymbol{\tau}$ Impact time iv

ii

$\mathbf{F_k}$ Linear impulse force

$\mathbf{S_{imp}}$ Impulse of force of impact

$\mathbf{F_{imp}}$ Force of impact or impulsive force

$\mathbf{E}$ Modulus of elasticity

ρ Density

$\boldsymbol{\nu}$ Poission’s ratio

$\boldsymbol{\varepsilon_i}$ Strain for the tool

$\mathbf{A_{ct}}$ Cross section area of the tool.

$\Delta\mathbf{l}$ Deformation of the tool

$\mathbf{l}$ length of the tool

$\boldsymbol{\sigma_x}$ Normal stress in x-direction

$\boldsymbol{\sigma_y}$ Normal stress in y-direction

$\boldsymbol{\tau_{xy}}$ Shear stress in y-direction

$\boldsymbol{\sigma_1}$, $\boldsymbol{\sigma_2}$, $\boldsymbol{\sigma_3}$ Principal stresses

$\boldsymbol{\sigma_{ut}}$ Ultimate tensile strength

$\boldsymbol{\sigma_{uc}}$ Ultimate compression strength

$\boldsymbol{\sigma_u}$ Ultimate strength

$\mathbf{D_n}$ insert tip diameter(n=1,2,3,4,5)

$\boldsymbol{\sigma_t}$ Tensile strength

$\boldsymbol{\sigma_{0.2}}$ offset yield strength

$\boldsymbol{\sigma_{max}}$ average maximum stress at a point

$\boldsymbol{\sigma_{min}}$ average minimum stress at a point

$\Delta\boldsymbol{\sigma}$ Range of stress

$\mathbf{K_I}$ Mode-1 stress intensity factor

$\boldsymbol{a}$ crack length

c & m Empirical material constant

$\boldsymbol{a}$ crack length

ai initial crack length

af Final crack length

N_f Number of Cycle/Blow to failure

K_{IC} Fracture toughness of material

d diameter of the tool

Abstract

The hydraulic hammers exposed to heavy-duty working condition occasionally generate premature failure of the tool. In actual case tool failure is caused by varies type of load. Under this thesis tensile stress developed on tool is considered for the investigation of tool failure. During rock breaking; both the axial and combined loading conditions mainly depend on thrust force acting on the tool. One particular existing design model H130ES hydraulic hammer tool is chosen for investigation of failure analysis. The dimensions, mechanical property and maximum hydraulic pressure are the input for both analytical and finite element method analysis. Using Finite Element Method with some analytical calculation this thesis tries to address the root causes responsible for tool failure. ANSYS 15 software is used as a strategic tool to perform each step in an orderly manner. The finite element simulation based on ANSYS identifies the maximum stress area and crack propagation simulation on tool. Analytically the stress concentration factor is solved as function of stress and assumed crack length. The analytical result is compared with FEM analysis. From the result obtained; the root cause of tool failure is an increase in thrust force on tool during rock breaking. To minimize such failure some recommendation is putted mainly with investigating tool manufacturing and design processes. Most important issue this thesis also addresses is the local problem of early tool failure caused by improper operation as well as lack of proper maintenance of hydraulic hammer.

Chapter One: Introduction

1.1. Background

Hydraulic hammer tool is the hardened steel used in rock breaking hammer for construction and mining industry. Excavators often use them to fracture the ground or bed rock to produce materials (aggregates) and to demolish structures. Hydraulic hammer typically are equipped with piston impacting the tool or steel that does the actual work. Design challenges of this type equipment include the high stress wave travelling through the tool including piston and associated parts [7]. The high stress waves can cause fatigue especially at stress/strain concentration areas. In construction machinery impact load is generated during normal operation[7], it is necessary to study its effect in performance improvement of the machine and failure of parts .Drilling performance for penetrating a visco-elasto-plastic rock material is evaluated for several percussive force profiles and rock material, in percussive drilling an impact tool continuously rises and drops to generate short duration compressive loads to crush the rock material [11].However there is limited study on the effect of impact load on tool failure of hydraulic hammer. It is reported that H130ES hydraulic hammer tool breakage happen suddenly and prematurely during rock breaking. Failure on tool caused by fracture (fig-1.1). The tool under investigation break/fracture at time interval of 240-720hr, but it is expected to work a minimum of 6,000hr working 4hr/day. For this thesis analysis; Blow/Impact energy, operating parameter and dimensions are all based on hydraulic hammer model H130ES.

As condition at tool bit and rock interface change due to geological variation at rock surface energy transfer efficiency will also vary [9]. The impact energy is transferred to the steel in the form of a stress wave that travels to the bit rock interface. Part of the energy in the wave goes to the rock, causing failure, and part of the energy is reflected .The reflected wave from bit rock interface is , as tensile stress wave. During reflection at the rock face, the total stress in the steel drill shank will be double, which is the critical zone for fatigue of the steel [25]. The tensile stress wave on the steel may cause any flaws /defect to progress to the stage of crack propagation and finally tool breakage happen (See figure1.1). Mode-I failure is expected when the tool break completely. According to[13] a tool metal "fatigue" failure generally occurs within 100mm above and below the face of front head, less common failure area is about 200mm from the face of front head, depending on working condition(See figure1.1).

A wide variety of defect can be found on the tool. These defects/flaws may result from such sources as material imperfections (See Fig. 1.2), defect generated during service, and defect introduce as a result of faulty design practice. The list of typical manufacturing defects includes machining, grinding, improper case hardening. Defect can be introduced into the component during service condition as a result of seizure (See Fig. 1.2).

Defects can also be introduced into the tool through faulty design .These human errors include the presence of severe stress concentrations, improper selection of tool material properties and surface treatments.

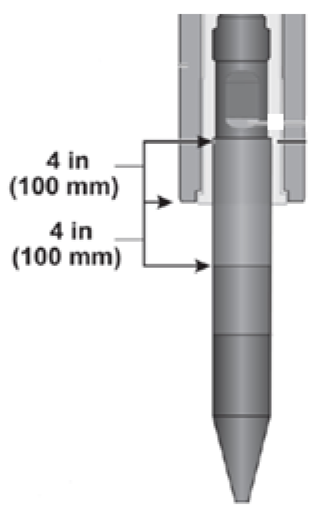


Fig.1.1. Fatigue fracture location & fracture surface on tool.

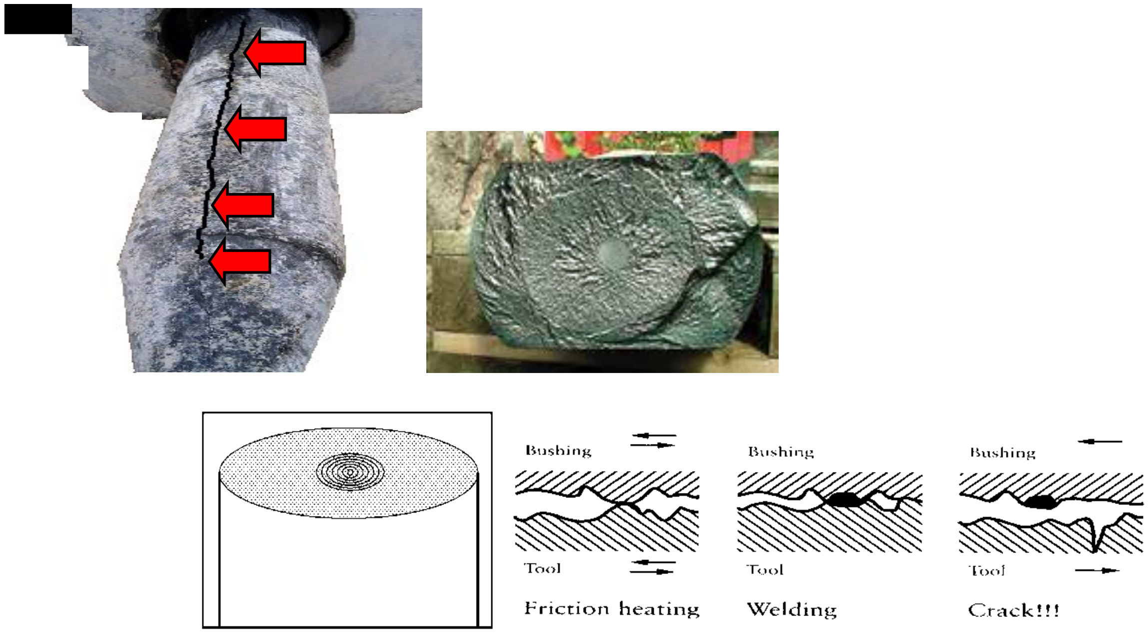


Fig.1. 2.Tool failure due to material defect and seizure.

1.2. Statement of the Problem

Most researches related to hydraulic hammer focused on the effect of impact load in relation to performance improvement of hydraulic hammer. The performance improvement mainly studies on how to increase breaking efficiency of the hydraulic hammer such as ROP (Rate of penetration). But no study has applied on the effect of impact load on tool failure. The actual situation of tool bit and rock interface is not simulated to study how stress is developed on the tool. In most researches only a specific rock behavior and tool mechanical property are taken, but in actual situation of impact the effect of coefficient of friction and contact type should also be considered.

The tool failure investigated under this thesis is imported from abroad. During design stage the designer may anticipate the design parameter such as rock behavior according to in house working condition; as a result design and manufacturing will depend on this specific condition. Even though this thesis indicate the improvement factors in tool failure analysis with considering design and manufacturing aspect but it ultimately try to suggest how to solve the overall problem by import substitution with improvement in design with investigating the cause of early tool failure.

Experimental analysis to determine tool failure is expensive since it requires a prototype of tool with specific test environment. The modeling technique for impact simulation of rock bit interface is very important in determining the actual stress distribution on tool. ANSYS has a power to simulate the actual contact behavior of bit rock interface and determine the stress distribution on the tool as function of time.

1.3. Objective of the study

1.3.1. Main Objective

To analyze the possible cause(s) of hydraulic hammer tool failure and propose necessary corrective action(s) for tool breakage/fracture in-service, based on predicted failure phenomena.

1.3.2. Specific Objectives

- Identify loading pattern on the tool.
- Define modeling Geometry of the tool and rock.
- Discuss the material property of the tool and rock.
- Define the most important parameter to model the impact.
- Determine critical stress area location under static load condition.
- Locate the maximum dynamic tensile stress area during impact load condition.
- Explain the predominant factor for fatigue failure development.
- Determine the fatigue life of the tool.
- Discuss on most important parameter from (material, dimension, design stress level, flaw size…)in order to minimize the fatigue failure.

1.4. Organization of the Thesis

This thesis is organized into five Chapters. In the First chapter, the background information and statement of research problem discussed; unresolved issue of the tool failure and its main objective is elaborated in this chapter. In chapter two, a review of literature with up-to-date relevant information to the thesis will be cited. In Chapter three, a static and dynamic 3D model analysis using

ANSYS software will develop. The boundary condition will be identified here according to practical application of the tool. In chapter four, result of the analysis part and discussions are made based on output of the analysis. In chapter Five, the conclusion of the thesis work and future work will be proposed.

Chapter Two : Literature review

Material

Paolo BALOSSI [1] in his research paper on experimental and Numerical simulation of penetration on sand stone explains that the hydraulic hammer tool has two major parts. The two major parts are; Penetrator (shank) and Insert. Penetrator (shank) is tempered steel with alloying elements. Due to high point stress must be developed at bit-rock interface so that the rock can be fragmented; usually tungsten-carbide inserts are mounted on tool bit face .Tungsten-carbide is most resistance to abrasion as applicable to rock drilling bit face insert.

Lucio Masriera, Marcos Fernandez, Juan Marani, Gianfranco Antonel,Claudio Acosta[2] according to their research project on Scientific Method Applied To Failure Analysis. Shank adapters or start bars are slim parts of hardened steel used in rock drilling hammers for the mining industry. The material the drill shank made of SAE-3310 steel bar with a 1.2mm layer generated by a case hardening heat treating. The chemical composition of the material 0.08/0.13(%C),0.45/0.60(%Mn),0.20/0.40(%Si),1.40/1.75(%Cr),3.25/3.75 (%Ni).Metallographic test shows that the drill shank has a martensite microstructure at the surface and a bainite at the core. According to the chemical composition of the drill shank the actual material of the shank for this thesis is 0.22(%C), 0.3(%Si), 0.7(%Mn), 1.3(%Cr), 2.9(%Ni), 0.2(%Mo), 0.020(%P Max), 0.025(%S Max),Proof yield strength=900MPa,Tensile strength=1250MPa.

Norman A.waterman and Michael F. Ashby [3] as per the material selector second edition book the corresponding material used for modeling the tool insert part is made of Tungsten-Carbide which is a hard alloy prepared by powder metallurgy, made of 84% (WC) and 16%(Co) and has high elastic modulus 530GPa , Poisson ratio(ν)= 0.22, Bending strength =1830MPa , Density = 14Mg/m^3.

Rasoul Moradi, Michael L. McCoy and Hamid M. Lankarani [4] in the academy published paper of Impact Analysis of Mechanical Systems Using Stress Wave Propagation Methodology describe that the solution to problems in impact mechanics requires the application of the basic laws of mechanics and physics, as well as a description of the behavior of the material being considered as a homogeneous, isotropic, linearly elastic body consists of the stress equations of motion, the Hook's law, and the strain-displacement relationships is to be applied.

Tool manufacturing processes

Lucio Masriera,Marcos Fernandez, Juan Marani, Gianfranco Antonel,Claudio Acosta; according to their research project on Scientific Method Applied To Failure Analysis ,the tool manufacturing processes with the following sequence ; The cold worked drawn steel bar going through a stress relief processes to avoid the possible deformation during machining operation(Turning, Milling) ,then the Carburized processes follow to increase the bar strength and harden ability at surface .After Carburizing quenching processes will follow for the purpose of refining the core by austenizing and at a reheat temperature of 650^0c stress relief processes is performed. Then second quenching for hardening again proceeds. Tempering is done for stress relief as well as to reduce brittleness without case softening at a temperature of 200^0C.Finally shoot penning follow to increase the compressive strength at the surface .Occasionally a straightening process is necessary to avoid any possible deformation (Bending) during previous heat treatment processes.

Method

Rasoul Moradi, Michael L. McCoy and Hamid M. Lankarani ; in the academy published paper of Impact Analysis of Mechanical Systems Using Stress Wave Propagation Methodology suggested that in practical aspect in modeling and understanding of impact problem with FEA it is necessary to develop a static model of the impact problem using dynamic load factors calculated by the energy method in order to understand the critical areas of stress. Refine the mesh where high stress levels are exhibited in the static model. Likewise, increase the mesh size in low and constant stress level regions. These lower ordered elements were found be more resistant to distortion and reduce computational effort. For 3D modeling of impact bodies they suggested to use 8-node, 24-DOF brick elements.

Dr. Wenyi Yan & Pr. Thanh Tran- Cong [5]. Hydraulic impact problem of a non-explosive rock breaking technology has been studied by them. Dimensional analysis and the finite element method have been applied to systematically investigate the influence of all the parameters involved in the impact process, which includes the geometrical parameters and the properties of rock.

Mr. V. Sivananth, Dr.S.Vijayarangan , & Mr.R.Aswathama [6] studied fatigue & Impact analysis of Automotive steering knuckle under operating load . Finite Element Analysis was performed on developed model of knuckle using hyper mesh. Failure region were first identified from the static analysis.

Loading , Boundary and Contact conditions

Changheon Song[1,2], Dae Jikim[2], Jintai Chung[1] ,Kang Won Lee[2],Sang Seuk Kweon[2],and Young Ky Kang[3] [7] in their research article of estimation of Impact loads in hydraulic breaker by transfer path Analysis ; the deformation of the chisel was measured by strain gage and they found that the maximum deformation ε_i of the chisel tool as 8.42×10^{-4} from the graph of shock wave profile of impact energy strain and they calculate the Impact/Impulse force by the formula $F_{imp} = E_C \times \varepsilon_i \times A_c$,where E_C is the young modulus of the chisel tool and A_c is it's cross sectional area.

Paolo BALOSSI ; in his research paper on experimental and Numerical simulation of penetration on sand stone suggest that a load scaling technique is used in finite element analysis to reduce computational effort. The load rate scaling consists in reducing the total analysis time until the kinetic energy is less 1% of the internal energy. The very small magnitude of kinetic energy suggests that the inertial effects of the model are not significant and the problem is solved *quasi-statically*. In three dimensional finite elements model is used to simulate the impact between tool bit and rock .The model allows to simulate the energy transmission from bit to rock interaction and processes of fragmentation, in this case the rock is assumed to be linear elastic. But if we want to simulate the stress distribution on hydraulic hammer tool the reverse should be assumed i.e the *rock to be considered as rigid* and the *tool as elastic body.*

Gang Han, Mike Bruno, Maurice B.Dusseault.[8] under the research paper of Dynamic modeling Rock Failure in Percussive drilling present a *blocky rock 3D model configuration* with dimension of square cross-section area with side length of 1.5m and a total height of 3m.The applied boundary condition is a *constant lateral confinement of 12.5MPa* and a *fixed displacement boundary condition at the bottom surface.*

Dr. Wenyi Yan & Pr. Thanh Tran- Cong. In their research paper on Investigation of hydraulic impact technology in rock breaking they discuss about the contact simulation as; there are three pairs of contacts involved in the impact process, i.e., the contact between the piston and the water, the contact between the piston and the rock, and the contact between the water and the rock. They use the hard contact algorithm from ABAQUS without damping to simulate these contacts. The friction is neglected in their simulation. They suggest the need for further study to be carried out to obtain reliable friction value involved in the contact between piston and rock.

Ming Jian [9] the research concludes that as condition at the bit-rock interface(contact condition) change, due to feed force or geological variations, the percussive energy transfer efficiency will also vary.

M.Ahmed[1],K.A.Ismail[2] and F.Mat[1][10] according to their journal of Impact model and coefficient of restitution ; Evaluation of impact between two rigid bodies can be treated by using discrete or continuous model. In discrete model, the velocity is instantaneously changed during the impact and is always measured by the impulse-momentum method. Mean while, *the continuous model represent collision in a finite duration* .The *force indentation* relationship during impact could be measured by using *continuous model.*

Simulation

Sazidy, M. S., Rideout , D. G., Butt, S. D. and Arvani,F.[11] in their paper of Modeling Percussive Drilling Performance using Simulated Visco- Elasto-Plastic Rock Medium they describe that , the range of parameters for simulation were selected in such a way that it closely matched with force(P) profiles generated by *real field hammer*. Accordingly they define the percussive interval as the total summation of Impact rise time (Tr) , Fall time(Tf) and time of gap(Tg) between two blows. Where *impact duration* is the sum of impact rise time (Tr) and impact fall time(Tf).

Failure Mode

Atlascopco, Working Tools for Hydraulic Breakers[12],according to product line Catalog ;the macroscopic crack path may also provide useful information about the toughness exhibited by a tool prior to failure. During fatigue failure as fracture propagated the crack often leaves a series of fracture marking, the curved lines called the chevron marking curves in toward tool fracture surface and point back to the crack origin. The cause of failure is due to the stress developed on the tool. Mode-I type of failure is clearly indicated on fractured/break tool.

RBI HYDRAULIC BREAKER MANUAL[13],according to the rock breaker manual to guide a warranty claims; when a tool fail to give satisfactory service life a brief visual inspection usually reveals the cause. A tool metal fail “fatigue” failure generally occurs within *100mm above and below the face of front head* .A less common failure areas is about 200mm from the face of the front head, depending on the work the breaker is doing.

Chapter Three: Finite Element Method and Conditions

3.1. Loading Conditions on the tool

3.1.1 Theory of Impact

If we represent linear impulse of the piston force (F_K) in time interval $\boldsymbol{\tau}$ in the form $F_K^{av}\,\boldsymbol{\tau}$, F_K^{av} is average value of this force in time interval $\boldsymbol{\tau}$ [17].

Theorem of change in momentum becomes;

$$m(v_1 - v_0) = \sum F_K^{av}\,\boldsymbol{\tau}$$

As $\boldsymbol{\tau}$ is infinitesimal (tends to zero), for conventional forces the velocity increment $\Delta v = v_1 - v_0$, tends to zero. If however, the acting force include very large force (of the order $1/\ \boldsymbol{\tau}$),the velocity increment in the small time interval $\boldsymbol{\tau}$ become finite quantity.

The phenomenon in which the velocities of the points of a body suffer a finite change in a very small time interval $\boldsymbol{\tau}$ is called Impact.

The force develop during impact is called impulsive forces or force of impact and very small time interval $\boldsymbol{\tau}$ during which collision takes place is called the impact time.

Theory of impact considers not the forces themselves as a measure of the interaction of colliding bodies but their impulses.

$$S_{imp} = \int_0^{\tau} F_{imp}dt = F_{imp}^{av}\boldsymbol{\tau}$$

The impulse of a force of impact (S_{imp}) is the finite quantity.

The action of non- impulsive forces (the force of gravity, for example) in time of impact can be neglected. But in this research the weight effect such as the thrust force is considered in the analysis.

3.1.1.1.Direct and Central impact

The impulsive force of the piston on the tool is considered to be direct and central impact. Excessive piston to tool angle will cause damage .Problem start with worn out tool bushings[14].

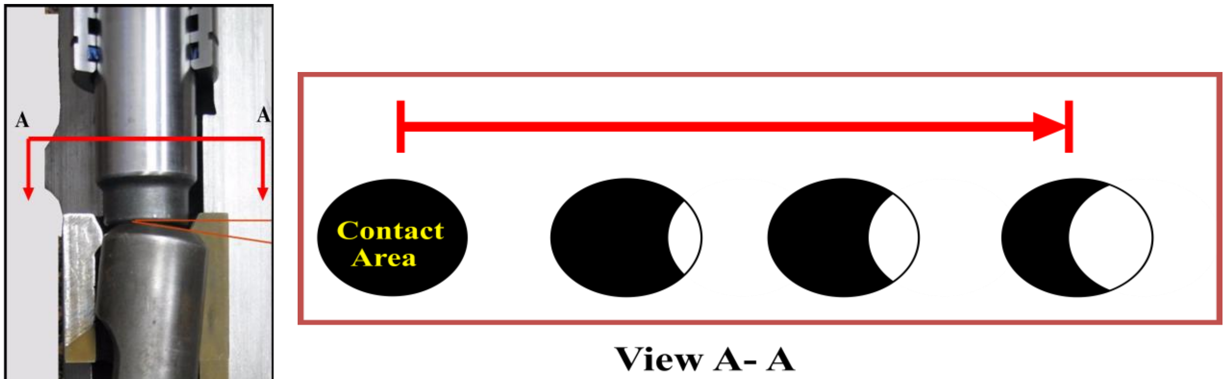


Figure 3.1. Piston tool impact

3.1.2. The Effect of Impact Force on Hydraulic Hammer tool

All hydraulic hammers worked by transmission of stress waves from piston through the tool. As the piston impacting the tool a transient incident stress $\sigma_i(t)$ generate in the tool. When the incident stress is transmitted into the interface of tool tip and rock, it will be partially reflected back into the tool in the form of the reflected stress $\sigma_r(t)$.Both the incident stress and the reflected stress are dynamic in nature i.e time dependent stress.

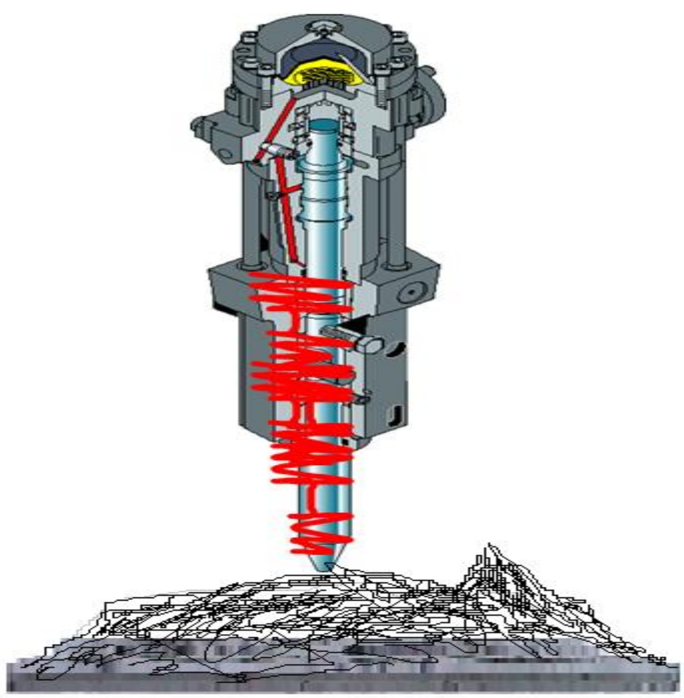


Fig.3.2 . Stress wave generated on the tool

3.1.3. Evaluation of Impact Model

Impact between two rigid bodies can be treated by using;

1. Discrete model
2. Continuous Model

3.1.3.1. Discrete model

Impulse momentum principle is a discrete model that was conventionally used to solve the problem in impact. This principle assumed that the impact is happen at very brief of period and there is no change in configuration of impact bodies during impact. The analysis is divided into two intervals, which are pre and post interval.COR (coefficient of restitution) from experiment is employed to determine post impact velocity.

Given by the formula;

$m_1u_1 + m_2u_2 = m_1v_1 + m_2v_2$

This method is not recommended for this thesis analysis since the result obtain by this principle will be inconsistent due to friction will not considered at tool bit and rock interface. Also it is difficult to determine the COR between tool tip and rock surface.

3.1.3.2. Continuous model

Continuous model, also known as compliance based method is proposed to overcome the problem in discrete model, the compliance of a small contact region around the initial contact point is represented by lumped parameter, which are stiffness and damping elements. As the contact force between the bodies is continuously act during impact, so the *impact amplitude against time* is the output of continuous model.

The continuous model analysis is recommended for this thesis analysis because;

1. Contact element concept is used during the analysis.
2. Both target and contact surface considered during analysis.
3. At contact region lumped parameters will be defined by ;
   - Stiffness parameter (E, ρ, ν)
   - Dumping element such as Coefficient of friction (μ).

In order to understand tool bit and rock interface contact parameters it will be important to consider the simulated Visco-Elasto-Plastic rock medium model [11]. From this modeling technique one can visualize how to relate the contact model parameters during the analysis.

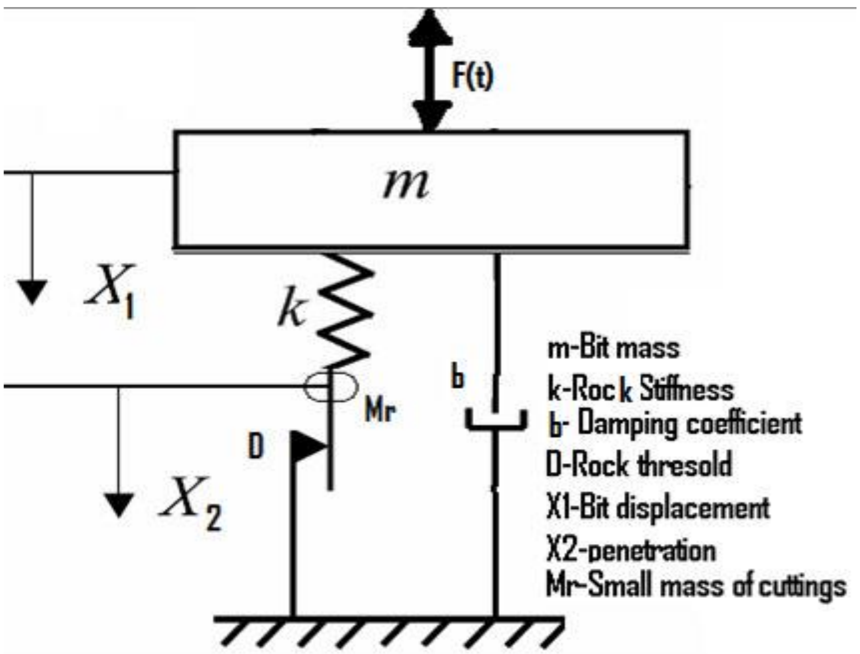


Fig.3.3. Diagram of visco-elasto-plastic rock model

3.1.4. Determination of impact and thrust force

It is already discussed about the theoretical background of impact as well as how impact load is applied in actual operation of hydraulic hammer. It is also described the type of impact model to be applied and its importance in visualizing the parameters needed in simulating the actual rock-bit contact region during analysis of the impact.

For further analysis purpose the impact amplitude force is calculated here. According to [7] the impact/Impulse amplitude force on the tool can be calculated by.

$$F_{imp} = F_{imp} = E_t \times \varepsilon_i \times A_c ,$$

Where E_t =180GPa

$\varepsilon_i = 8.42 \times 10^{-4}$ according to [7]

$Ac_t = 22/7 \ * (0.065)^2 m^2$

$F_{imp} = 210 \text{ X } 10^9 \text{ X } 8.42 \text{ X } 10^{-4} \text{ X } 0.0133$ N.

$F_{imp} = 2351706N = 2352KN.$

3.1.4.1. Maximum Deformation on tool

If σ_y is the nominal normal stress due to impact on tool cross section and relative longitudinal strain (ε_i) can be defined as[18];

$$\varepsilon_i = \frac{\partial u}{\partial x} = \frac{\Delta l}{l}$$

l = length of the tool, before deformation.

Δl= change in length, maximum deformation under impact.

$\Delta l = (\varepsilon_i)(l)$ but l=110cm , $\varepsilon_i = 8.4 \text{ x}10^{-4}$[7]

$\Delta l = 8.42 \text{ x } 10^{-4} \text{ x } 110cm = 0.09262cm.$

The expected deformation of the tool during impact is 0.09262cm.

3.1.4.2. Thrust Force on tool

Proper thrust force consideration is one of the parameter in the design of hydraulic hammer tool. This force acts at the start of impact and at time gap between consecutive impact forces. Here the minimum thrust force calculated as;

Thrust Force (F_t) = Hammer maximum weight + stick weight

F_t= (1199Kg) (9.8N/Kg) + (1890Kg) (9.8N/Kg)

F_t= 11750.2N + 18522N

F_t= 30272.2N

F_t=30500N

3.1.5. Failure Criteria

Since the tool under investigation is suddenly break/fracture, a yield criterion of failure is not reliable to evaluate this thesis analysis. In applying fracture criteria, the ultimate strength in tension and compression, σ_{ut} and σ_{uc} ,are needed .As the material fail in brittle manner, yield will not well defined event, ultimate strength and fracture event occur at the same point. Therefore σ_{ut} for brittle material is the same as engineering fracture strength, $\sigma_{f.}$

3.1.5.1.Maximum Normal Stress fracture Criteria

With considering tool failure mode, this thesis will apply failure criteria that consider the largest principle normal stress with respect to the uniaxial strength of the tool material. Material which fractures if the ultimate strength σ_u exceeded either in tension or compression.

Assume $\sigma_{ut} = |\sigma_{uc}| = \sigma_u$

Maximum Normal stress fracture criteria would be specified by function of ;

$\sigma_u = \text{Max}(|\sigma_1|, |\sigma_2|, |\sigma_3|)$ at fracture.

The direction of the main loads applied by the hydraulic breaker follows the y-axis.

Since; $\sigma_x = 0$, $\tau_{xy} = 0$

$\sigma_1 = \sigma_y$ (For tensile case).

$\sigma_3 = \sigma_y$ (For compression case).

3.2. Finite Element Method Using ANSYS

3.2.1. Geometry of the Tool.

3.2.1.1. Simplified two dimensional model of the tool

The tool investigated in this thesis is moil type and applicable in breaking sedimentary and metamorphic rock into which the tool can penetrate. It also applicable for breaking concrete. Most of the time this tool is used for soft and non-abrasive rock type.

Specific application in construction site are; for road maintenance for breaking road surface. In quarry area for breaking the oversize rock to prepare for crusher feed.

A tool dimension with simplified 2D model is shown using AutoCAD drawing below. (See Fig.3.4).

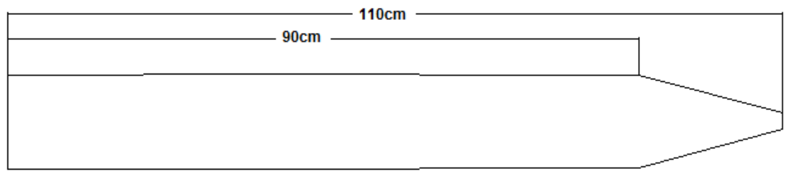


Fig.3.4. Simplified tool Model

3.2.1.2. Condition to model Hydraulic hammer tool by using ANSYS

The ultimate goal of solid modeling of tool is to create a finite element mesh of the physical system. Solid modeling method involves creation of geometrical volumes that represent actual problem.

Rock breaking insert is mounted on the tool shank either by brazing or mechanical fastening to the shank part.(See figure .3.5. below)

Therefore the tool has two parts which is made of different material.

1. Insert section
2. Shank/Penetrator section.

The insert part of the tool most of the time made of tungsten carbide to resistant abrasion, while the shank/penetrator is made of tempered steel bar with alloying elements [1].

Bit Section

In the case of direct bearing bits on yielding rock, experiment have shown that a wedge –shaped bit will penetrate until the average bearing pressure on the oblique faces & tool edge is equal to the uniaxial compressive strength of the rock[25].

3.2.1.3.Steps to model the hydraulic hammer tool.

1. Modeling of the insert

As already mentioned above the tool insert (D) is made of tungsten carbide the remaining part of the bit(E) is also part of the shank/penetrator.

In order to model the tool ANSYS use a special command known as Glue.

Glue takes two entities that are touching and make them contiguous or congruent so that when meshed they will share common nodes.

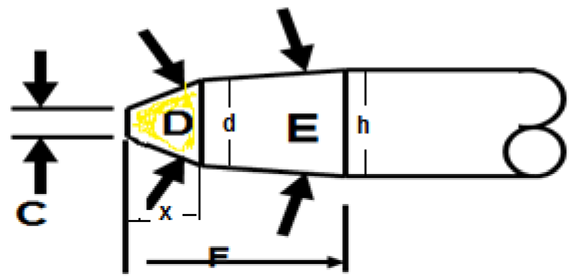


Fig.3.5. Tool bit part

Dimensions of bit for H130ES

C = 20mm D=60⁰ E= 24⁰ F=200mm

Detail dimensions and Modeling of the insert

Let the length of the insert is x and it's diameter at the tip of the shank is d

Therefore the length of the bit section excluding the insert

=200-x

For the insert;

$Tan30 = OP_1/x$

$OP_1 = x(tan30)$

$d = 2(OP_1) + 20$

For bit section excluding the insert

$Tan12 = OP_2/200\text{-}x$

$OP_2 = 200\text{-}x\ (tan12)$

$h = d + 2(OP_2) = 130$

- $2(tan12(200-x)) + 2x(tan30) + 20 = 130$
- $0.426(200-x) + 1.154x = 110$
- $85.2-0.426x + 1.154x=110$
- $0.728x=24.8$
- $X = 34mm$

Therefore the length of insert is 34mm

$d = 2(34)(0.577) + 20$

$d = 59mm$

Length of the bit section excluding the insert=200-34 =166mm

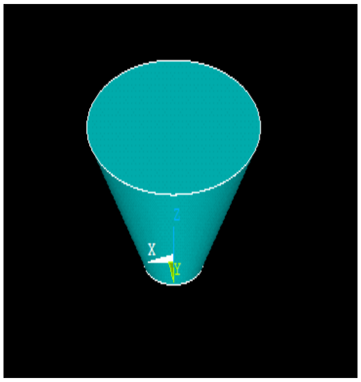


Fig.3.6 Insert Model

2. Model the shank

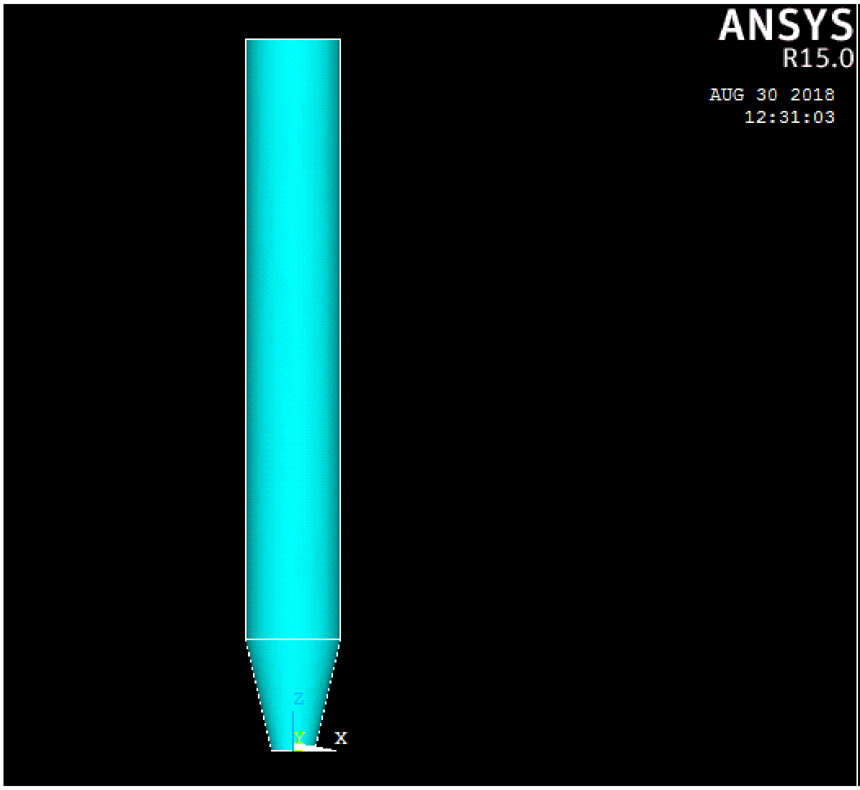


Fig.3.7. Shank/penetrator Model

3. Glue the insert and shank(penetrator) part using Boolean operation.

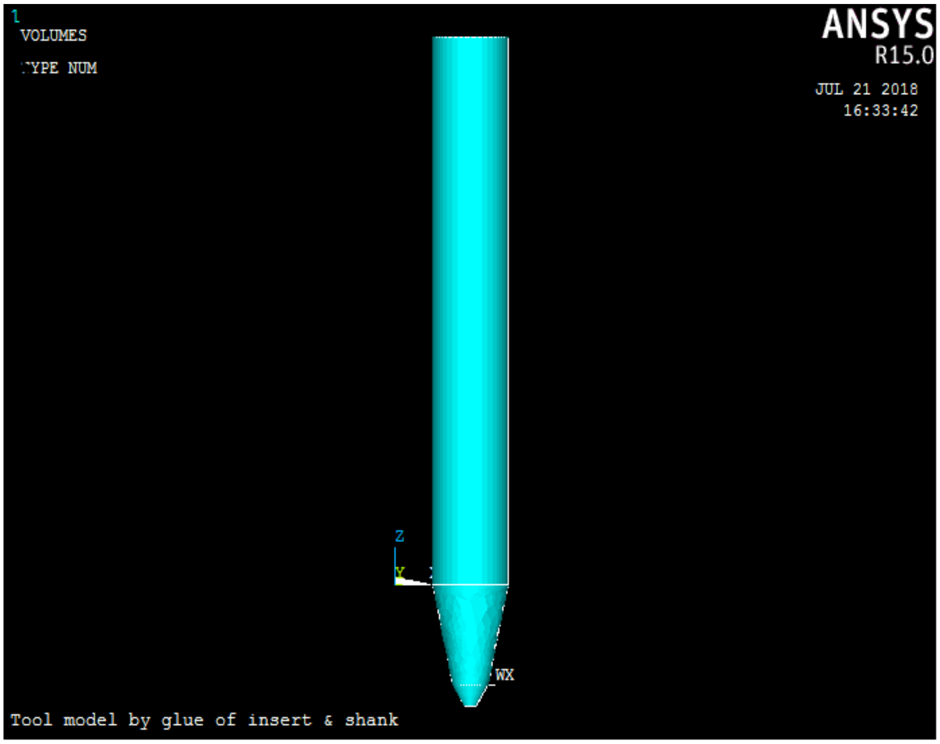


Fig .3.8. Tool Model

3.2.2. Geometry of the Rock Model.

3.2.2.1.Rock Model Size

Rock formation is in layered and blocky.

Massive: - When the distance between layers and joints is more than six feet. (> 1.83meter).

Blocky : - When joint spacing is greater than one foot and less than six feet.(0.305 < x < 1.83meter).

Broken:-Individual blocky are smaller than one foot.(< 0.305meter).

Very Broken: - fragments are smaller than three inches.(< 2.54cm).

In this thesis a blocky confined type rock model is taken for the analysis purpose [8].Furthermore, many of the practical problem that are concerned with the strength involve is a configuration where the rock is in confined state; consequently, the actual bed rock condition is simulated. The rock can also be classified as intact i.e rock containing no joints.

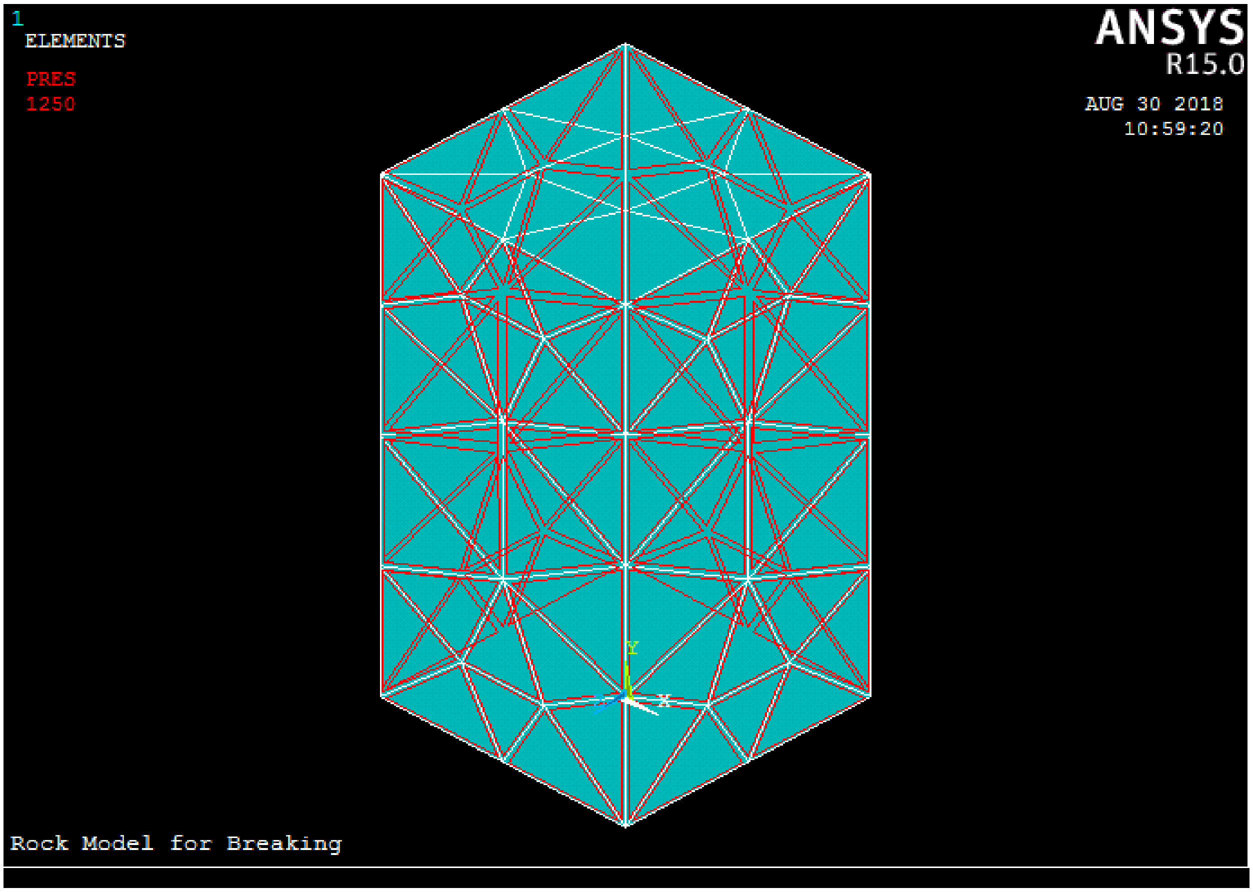


Fig 3.9. Confined Rock model

3.2.3. Material property of the Tool

The material of the penetrator/shank is Carburized and tempered steel with alloying elements as reported in [2].The insert is made of Tungsten carbide which is hard alloy prepared by powder Metallurgy and it's alloying element is as per [3].In defining tool material each body of the assembly should correspond with its material type.

N.B. The modulus of elasticity is the measure of the stiffness of the tool material.

3.2.3.1.Mechanical properties of Penetrator/Shank Table 3-1

M. property	$\sigma_{ts}/\sigma_{0.2}$ (Tensile/Offset yield strength),MPa	E (Modulus of Elasticity),GPa\ N/cm^2	ν (Poisson's ratio)	Density Kg/cm^3
Actual	1250/900	210/2.10x10^7	0.31	0.0085

3.2.3.2.Mechanical properties of insert

Table 3-2

M. property	σ_b(Bending strength),MPa	E (Modulus of Elasticity),GPa\N/cm^2	ν (Poisson's ratio)	Density Kg/cm^3
Actual	1830	530/5.30x10^7	0.22	0.014

3.2.4. Rock property

Rock Strength

The magnitude of stresses under which a rock disintegrates characterizes its strength. It is to compression that a rock offers the greatest resistance, where as its tensile strength usually does not exceeds 10% of the compressive strength.

Elasticity of rock: -most of rock –forming minerals are elastico-brittle bodies, e.g. they obey Hooke's law and disintegrate when stresses reach the elastic limit. For this reason it is but approximately that one may refer rocks to the category of elastic bodies.

E(Modulus of Elasticity) of rock and Poisson's ratio(ν) characterize the elastic property of rock. Moreover the modulus of Elasticity is a measure of the stiffness of the rock material.

Density of rock:- is the mass contained in unit volume of it's solid phase. Generally rock strength increases with increasing of rock density.

Coefficient of friction(μ):- is taken from the experimental value of coefficient of friction between the rock and steel [1].

3.2.4.1. Rock Mechanical properties

Table 3-3

M. property	E (Modulus of Elasticity),GPa\N/cm^2.	ν (Poisson's ratio).	Density, Kg/cm^3	Coefficient of friction(μ) between rock and insert
Actual	55/0.55x10^7	0.26	0.00261	0.14

3.2.5. Assembling of tool and rock models

In the assembling to model the tool and rock. First the rock model (Volume-1) is done in ANSYS workbench .Next the tool insert (tungsten carbide part) modeled as Volume-2 in contact with the rock surface. Then the rest of the tool part i.e penetrator/shank is modeled with Booleans operation of glue to the tungsten carbide insert.

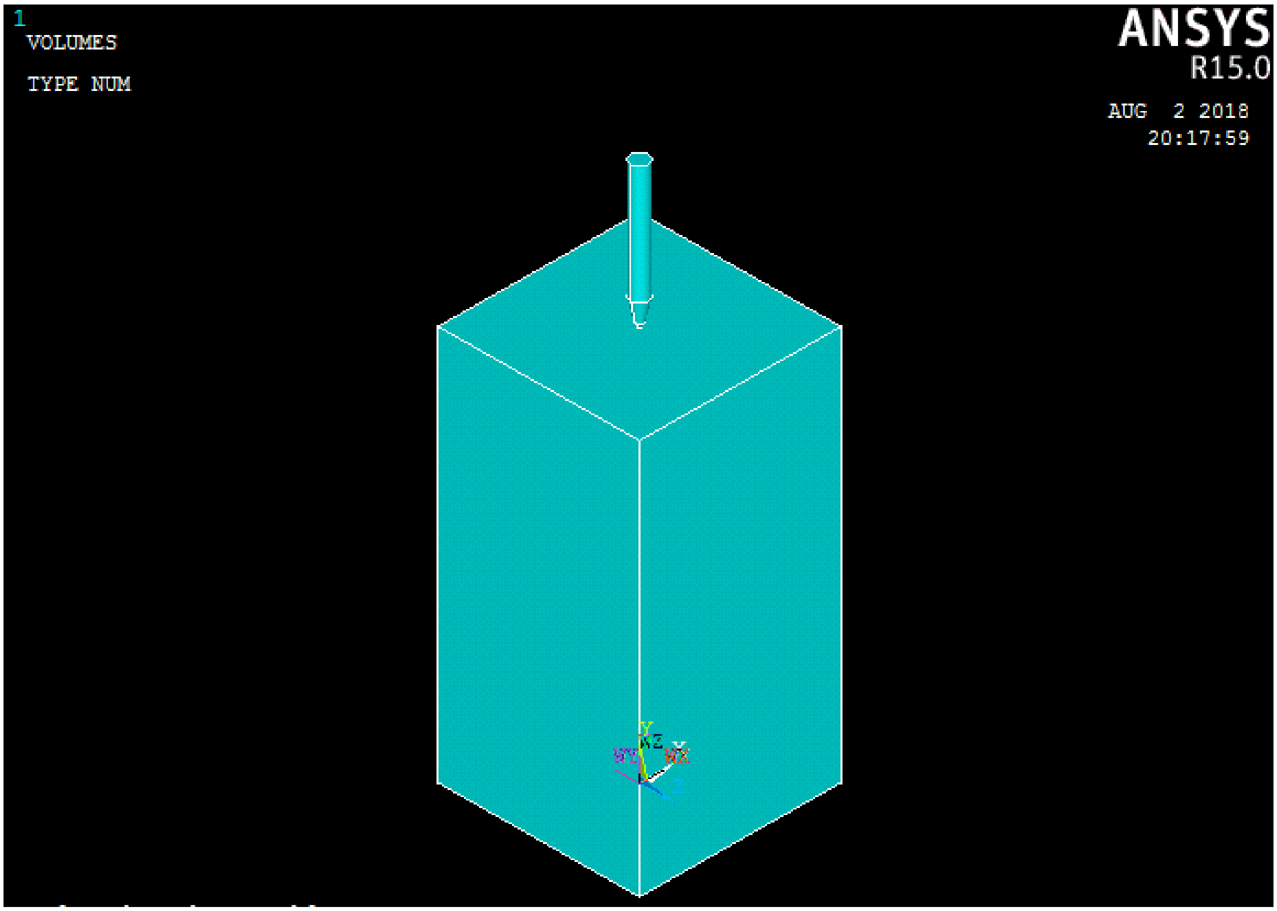


Fig.3.10. Tool and Rock Assembly Model.

3.2.6. Meshing of Assembled tool / rock model

Before starting meshing, element type must be defined. ANSYS software contains more than hundred elements in its library. Elements are with unique number classified according to dimensionality, analysis discipline and material behavior. ANSYS classifies the element in 21 different groups[23].

One of these groups is structural. For structural analysis the primary variable is the displacement and is called Degree Of Freedom (DOF). A general three dimensional structural model has six degree of freedom at each node, translation in X, Y and Z directions, and rotation about those three axes. All of the basic calculations in the finite element are performed at the node. What happen inside an element between the nodes is controlled by the element shape function .This an assumed distribution of the primary variable and can be linear, quadratic, cubic etc [22].

Element type selection

Consider the elements Quad 4 node 182, Quad 8 node 183, Brick 8 node from structural solid sub group. The first two element type Quad 4 node 182 & Quad 8 node 183 are used for two dimensional structural problems (plane stress, plane strain, or ax symmetric).Whereas the Brick 8 node used for three dimensional structural problems. The difference between Quad 4 node 182 & Quad 8 node 183 elements is that they have a different number of nodes per element. In this part case ,the variation of displacements along element edge is assumed be linear for Quad 182 & quadratic for Quad 183.The interpolation of brick 8 node element are linear [23].

** 20 node 186 is higher order version of brick 8 node element. It has quadratic displacement behavior.

**10 node187 has quadratic displacement behavior particularly useful for modeling solid with irregular geometry.

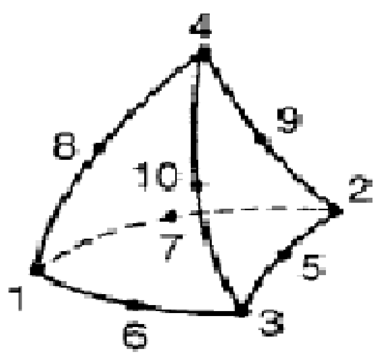
Summary of Elements characteristic

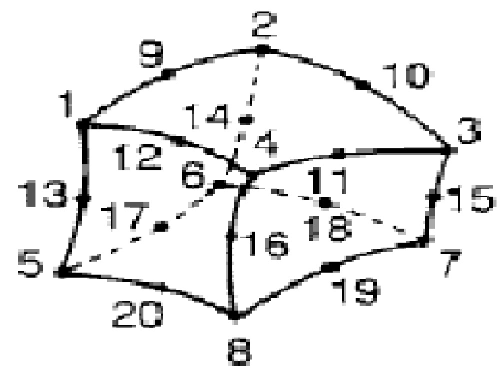
Table 3-4

Element Type	Application	Element Output Data
Brick 8 node	For 3D modeling of solid structure.	Nodal displacement, principal stresses, stress intensity, elastic and principal strain.

20 node 186	For 3D modeling of solid structure with irregular geometry.	Nodal displacement, principal stresses, stress intensity, strain.
10 node 187	For 3D modeling of solid structure with irregular geometry.	Nodal displacement, principal stresses, stress intensity, strain.

Since both 20 node 185 and 10 node 187 are used for meshing of irregular geometry, both element type will be applied and compare the error of meshing to select the best element type.





a.10 node 3-D space
DOF: UX, UY, UZ

b.20 node 3-D space
DOF: UX, UY, UZ

To refine meshing with minimum error the assembly model will be meshed with bricks 10 node 186 element type. More ever Impact bodies should be meshed with low order elements to be more resistance to distortion and reduce computational effort [4]

3.2.6.1.Refining the mesh

It is important to refine the mesh size in the tool contact area .

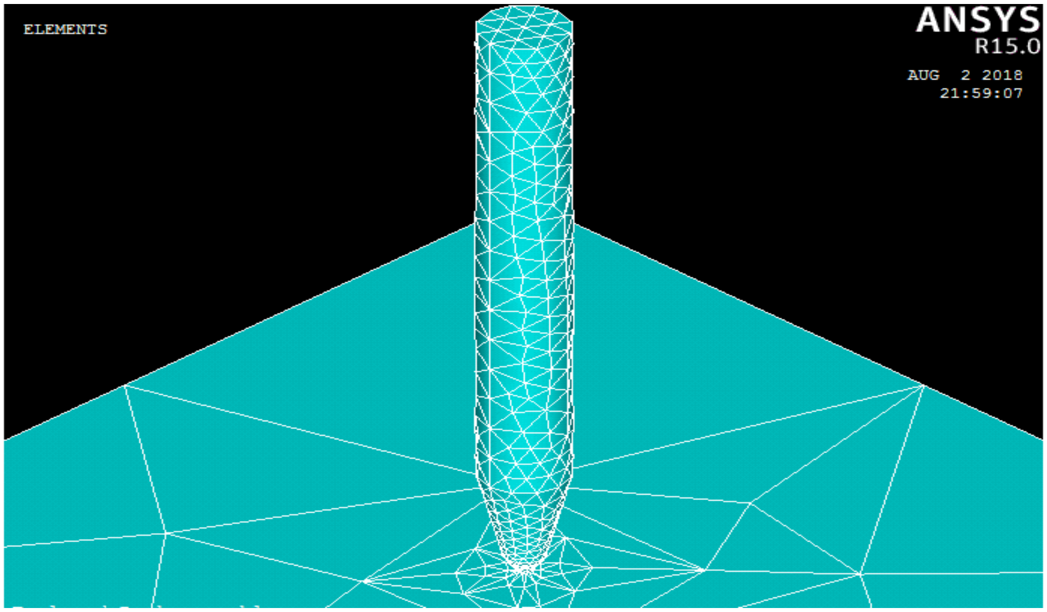


Fig.3.11. Meshed Rock & tool model

3.2.6.2.Mesh attribute for material specification

The step to follow to model tool rock assembly with the correct material specification is very important in the analysis stage especially to analyze the stress distribution on tool and tool sank part separately. As per the ANSYS model(Figure 3.12) shown below number-1 represent the rock model, 2 represent the tip part and 3 represent the shank/penetrator.

Step to follow to specify the material is;

1.Select the Element type.

2.Assign material property corresponding to material model.

3.Model the rock as block.

4.Model the tool tip part(Tungsten carbide tip) as truncated cone.

5.Model the shank/penetrator part of the tool.

6.Glue the shank/penetrator with tool tip.

7.Perform mesh attribute corresponding to each material model number.

8.Mesh overall assembled model.

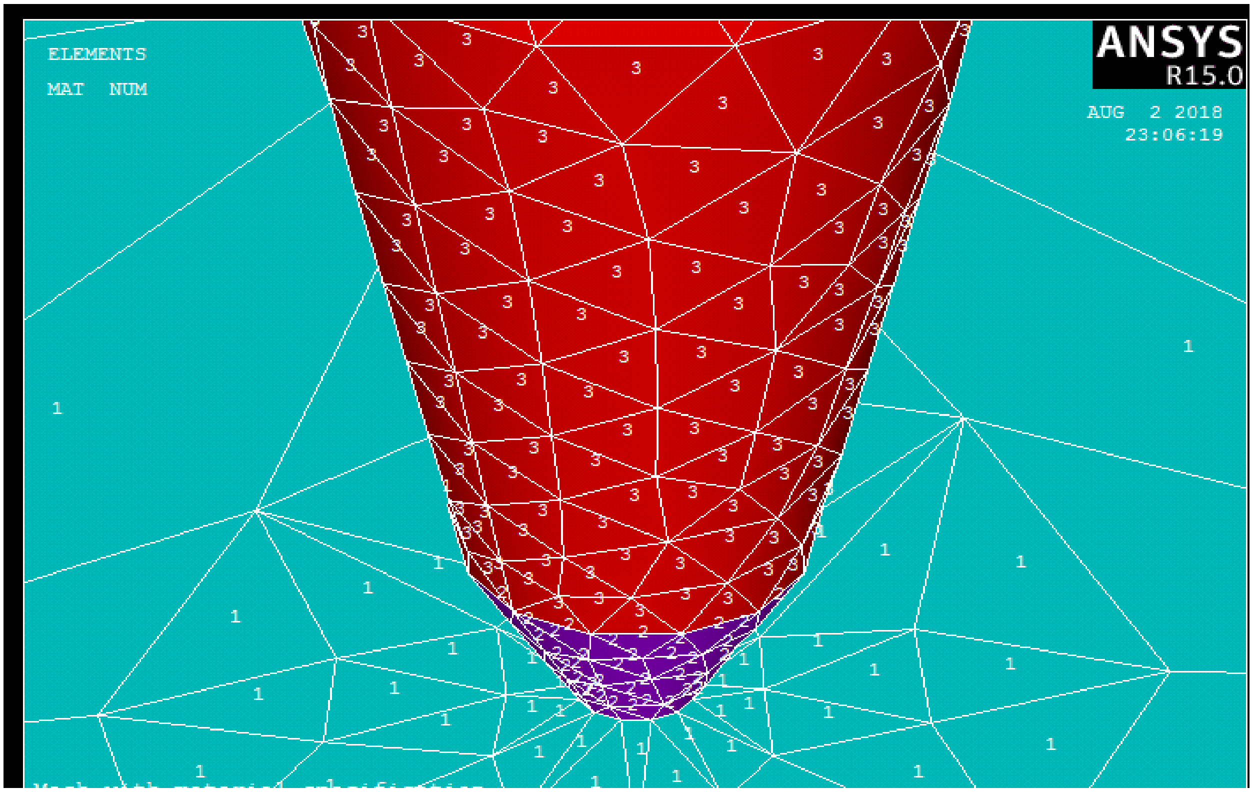


Figure 3.12.Material attribute

3.2.7. Contact Elements and Boundary Conditions.

3.2.7.1.Contact between surfaces

When two engineering surface are loaded together there will always be some distortion of each of them. These distortions may be purely elastic or may involve some additional plastic, and so permanent, change in shape. Due to above reason nonlinearity in contact condition arises on the boundary and it depends on the deformation of the structure. Furthermore, no-interpenetration conditions are enforced while the extent of the contact area is unknown[21].

Contact between two bodies with no bonding (such as glue, solder, or weld) is a challenging problem, mainly stemming from the lack of prior knowledge of the contact regions. Another complication is that, in most of the cases, there is friction between contacting bodies. Both of these two factors make contact analysis highly nonlinear. In addition of theses, if the material involved exhibit nonlinear material behavior or transient effect, achieving convergence become more difficult .Classical contact mechanics assumes the deforming material to be isotropic and homogenous; in principle, its result can be applied both to global contacts and those between interacting asperities[23].

3.2.7.2. Finite Element Method in Contact Problem

When the contact problem involves nonlinear material behavior such as plasticity or when the half-space assumption is no longer valid (i.e when the dimensions of the body are not large compared to the region of contact), then it may be necessary to employ a finite element method.

A number of commercial finite element codes are available which are suitable for the solution of frictional contact eg. ANSYS.

A contact bodies are divided into an array of either two-dimensional axi symmetric or planer element or three dimensional brick elements. A set of gap or contact elements is defined between surface nodes which are likely to come into contact during loading.

A friction coefficient may be defined for the contact. Appropriate material properties, boundary conditions, and nodal loading are also specified.

The nonlinearities associated with contact, such as friction, and plasticity mean that the solution is ***loading-path dependent*** .This requires that the applied load is divided into a number of small steps and a solution found at each iteration. These two features can often leads to long computational times [4].

3.2.7.3.Selection of contact model type

Before starting the analysis, the two main considerations are;

i.The difference in stiffness between the tool and the rock .

ii. Location of possible contact between tool tip and rock surface.

Then based on stiffness of contact bodies, rigid-to-flexible contact option can be used in ANSYS.

There are three contact models in ANSYS[23]. They are;

1.Node-to-node :- if contact region is accurately known a prior.

2.**Node-to-surface**:- a specific point on one of the surface make contact.

3.**Surface-to-surface**; - when contact regions are not known accurately and significant amount of sliding is expected. In this model, one of the surfaces is called the contact surface and the other, the target surface.

Since the contact region in the tool impact model is not known a prior and sliding is expected between the tool tip and rock surface, the surface-to-surface contact model is applied to this thesis analysis.

3.2.7.4.Boundary Conditions

Constrain the appropriate degree of freedom based on nature of support provided.

The steps in applying boundary conditions are [22];

- Select the kind of constraint to be applied.
- Select the geometry entity on which it is to be applied.
- Enter the value and direction for it.

Essential Boundary condition

In actual operation during impact the tool should move in vertical direction so that the impact to be direct and central.

The rock model is assumed to be as a rigid body.

Table 3-5

Boundary condition	**Type of Boundary condition.**	**Geometry entity.**	**Value**	**Constraint**
Displacement	Essential BC	Lower surface of the rock model.	All DOF = 0	On Bottom of confined rock model.

Natural Boundary condition

The tool is acted upon a force or forces that are applied on top of the tool surface.

There are two type of force to be considered during the analysis of this thesis;

1. Force of impact or Impulse force

This force is the main cause for the generation of dynamic stress inside the tool. It's value is calculated in section (3.1.4).

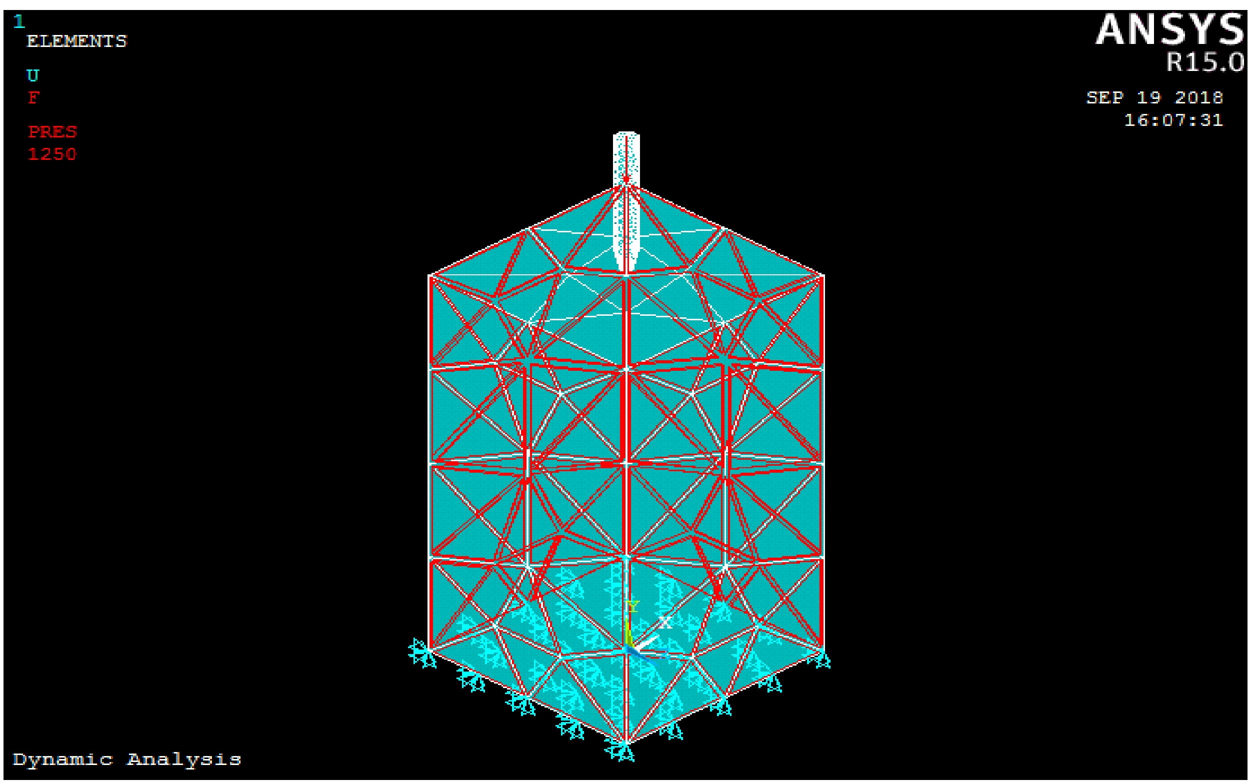
2. Weight on Bit/Thrust force.

This force is applied to keep the tool in contact with the rock surface during rock breaking. It is a static type force and its value is calculated in section (3.1.4).

The confinement of the rock model is used to represent the rock to be broken is in natural rock bed condition. Therefore a lateral pressure has to be applied on the rock model to show the rock is in confined state(See figure 3.9 above).

Table 3-6

Load Type	Type of BC	Geometric entity	Value
Impact Force	Natural BC	On top edge surface of tool	2351706N / 2352KN
Weight on Tool/Thrust force.	Natural BC	On top edge surface of tool	30500N/30.5KN
Confinement pressure	Natural BC	On lateral surface of rock model	1250N/cm^2 [8]



3.2.8. Identification of the critical stress area using a static load

3.2.8.1.Condition

Before doing the actual problem by dynamic analysis the possible failure area should be located by critical stress zone on tool under static loading condition. According to research paper [4] in impact analysis of mechanical system, in order to understand the problem developing a static model for actual impact situation using dynamic load factors calculated by the energy method will result in locating of the critical areas of stress on the tool.

Dynamic load Factor

It is generally very difficult to determine the exact magnitude of stresses produced by impact or shock. These are usually estimated by multiplying the static stresses with a dynamic or shock factor[19].

$$K_i = \frac{Impact/Dynamic\ \ Load}{Gradual/Static\ \ Load} = 1 + \sqrt{1+2h/y}$$

h is impact travel.

y is deformation by static load.

K_i = 1, for gradually applied or steady load.

= 1.0 to 1.5 for minor shock.

= 1.5 to 2.0 for heavy shock.

Let's calculate the static load by using the calculation done in section (3.1.4)

Take $K_i = 2$

$F_{imp} = 2351706N$

$F_{stat} = 2351706N/2 = 1175853N$

$F = F_{stat} + F_t = 1206353N$

Analysis step for static Analysis

The main purpose of the static analysis is to identify the critical stress area on the tool so that the possible fatigue failure area is easily identified during dynamic analysis.

1. Select the element type.
2. Specify materials.

3. Model tool rock assembly.
4. Mesh tool rock assembly.
5. Specify the contact pair on tool rock assembly
6. Applying the coefficient of friction at contact area .
7. Apply the essential boundary condition.
8. Apply the natural boundary condition.
9. Define the analysis type by solution control.
10. Solve the problem.
11. Separate the tool model from the assembly to identify the stress develops on the tool.

Tabulation of Parameters for Modeling

I. Tabulate the dimensions for the volume of Rock

Volume No	X_1(cm)	X_2(cm)	Y_1(cm)	Y_2(cm)	Z_1(cm)	Z_2(cm)
1	75	-75	0	300	75	-75

II. Tabulate the dimensions for the Volume of Tool.

Volume No	RBOT(cm)	RTOP(cm)	Z_1(cm)	Z_2(cm)	THETA 1	THETA 2
2	1	2.95	300	303.4	0	360
3(Shank)	2.95	6.5	303.4	320	0	360
4(Shank)	6.5(R1)	0(R2)	320	410	0	360

III. Tabulate the volume attribute

Volume No	Poisson's Ratio	Young's Modulus(N/cm^2)	Density(Kg/cm^3)
1(Rock)	0.26	$0.55X10^7$	0.00261
2(Insert)	0.22	$5.30X10^7$	0.014
3(Shank)	0.29	$2.10X10^7$	0.0078

IV. Tabulate the Essential Boundary Condition (See table 3.5 above)

V. Tabulate the Natural Boundary condition

Volume No.	Area(cm^2)	Y-Coordinates(cm)	Force(N)

3	132.79	410	1206353

NB 1:- To generate the solution the Analysis type defined by a solution control following those steps;

1. Selecting small displacement under static condition.
2. Specifying time at end of load step.
3. There is no need for time stepping program since the analysis is under static condition.
4. Specifying the number of sub step, it should be one for static loading condition.

NB 2:-The result of the analysis is focused on the principle stress theory for failure criteria. Since failure will occurs under compression stress with least principle stress (SMIN) is the criteria for failure.

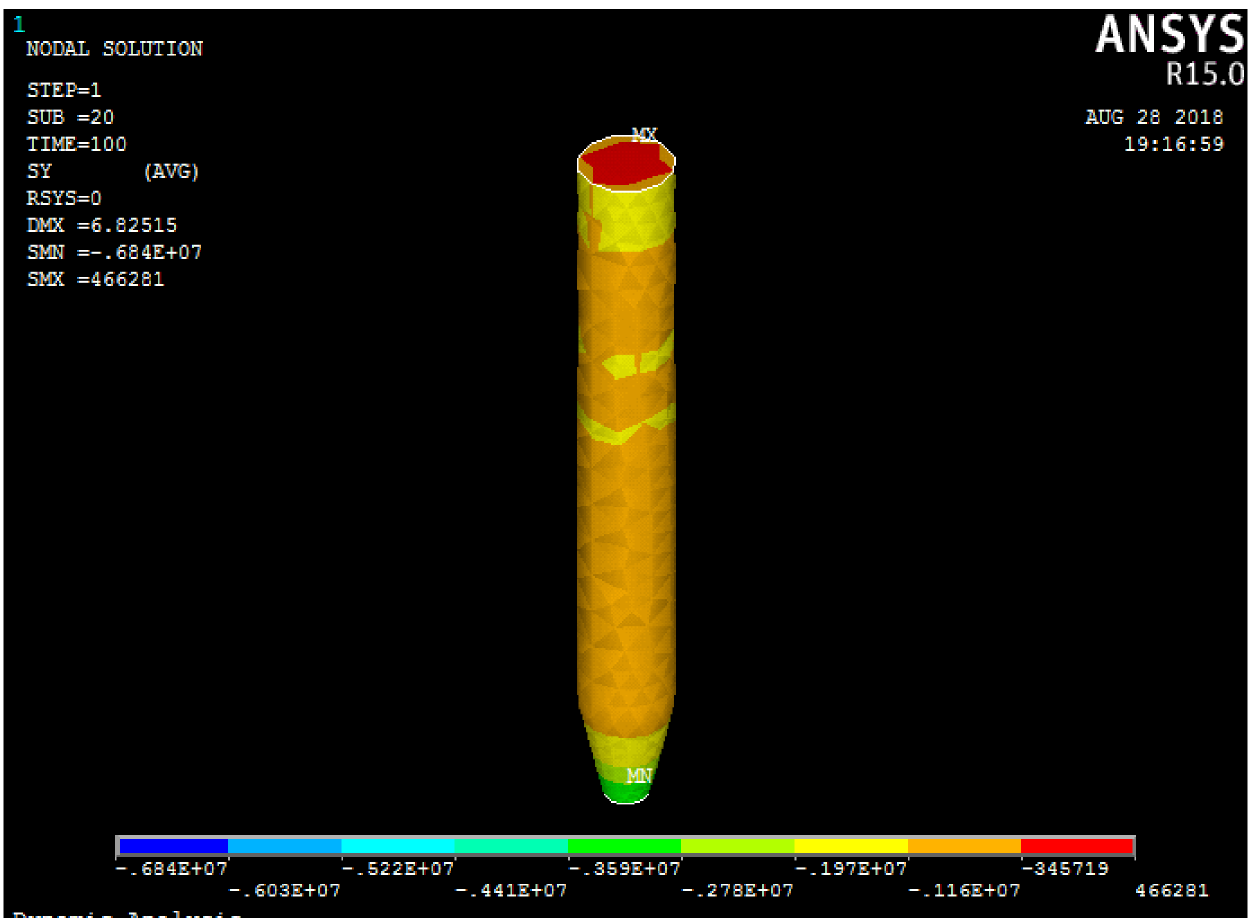


Figure 3.13 Static analysis result

From static analysis of critical stress area it can conclude that possible least compression stress is middle section of the tool

There fore in order to identify the most critical stress area that cause the tool to fail, the actual phenomena of dynamic analysis should be done with the impact loading condition.

3.2.9.Dynamic Analysis Condition

A static analysis might ensure that the design will withstand steady state loading conditions, but it might not be sufficient, especially if the load varies with time. Dynamic analyses take an account of an impact as per discussion in the introduction part of hammer blow. Impact is handled by a specific type of dynamic analysis.

The hydraulic hammer tool should be designed to resist the impact of piston when it breaks the rock, such a problem will be solved by transient dynamic analysis to calculate the tool response to time during impact load of the piston.

3.2.9.1.General Equation of motion

General equation of motion for transient analysis is;

$[M]\{\ddot{U}\} + [C]\{\dot{U}\} + [K]\{U\} = \{F(t)\}$

Under this thesis the general equation of motion above is solved by direct integration. For transient analysis, the equation remains a function of time can be solved using either an explicit or implicit method[26].

Generally explicit method can handle nonlinearities easily with small integration time step Δt and ANSYS uses this method.

Depending on the method of solution, ANSYS allows all type of nonlinearities to be included in a transient dynamic analysis like **contact problem and friction** consideration, those problems requires an iterative solution at each time point.

3.2.9.2.Direct integration method of solving.

Equation of motion is directly integrated step by step over time .At each time point (time = 0 , Δt , $2\Delta t$,$3\Delta t$, ….) a set of simultaneous static equilibrium equation (F = ma) is solved.

Δt = time increment from one time point to the next.

3.2.9.3.Nonlinear Response

Nonlinearities such as friction in this thesis problem is non- conservative in nature and **require the load history** to be followed accurately a small integration size helps in following the load history accuracy.

3.2.10. The gap and simulation time

According to paper [9] the range of parameters for simulation were selected in such a way that it closely matched with force profiles generated by field hammer. Accordingly, the percussive interval is total summation of impact rise time(T_r), Fall time(T_f) and time of gap (T_g) between two blows. Impact duration is the sum of Impact rise time (T_r) and Impact fall time (T_f).In this particular analysis, applied thrust (WOB) was kept constant at a level equal to the dry friction threshold force. The impact force periodically brings the total force above this threshold force[11].

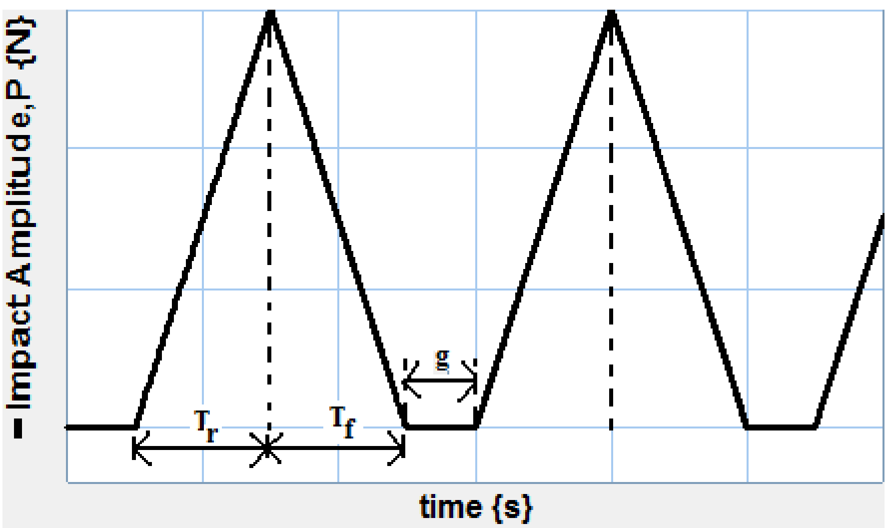


Figure 3.14. Time History Load Diagram [9]

3.2.10.1.Determination of Parameter for Optimization

As per [11] the impact duration or impact time(τ) is the sum of rise time(T_r) and fall time(T_f)

$\tau = T_r + T_f$

From theorem of change of momentum ;

Change in momentum of a system during impact is equal to the total impulse of all external forces of impact acting on the system.

$m(v_1 - v_2) = \sum F_K^{av}\, \boldsymbol{\tau}$

$m(v_1 - v_2) = F_{imp}\, \boldsymbol{\tau}$

When the piston impact the tool;

$m_p(v_f - v_i) = F_{imp}\, \boldsymbol{\tau}$

But initial velocity of the piston(v_i)= 0

Blow/Impact Energy = $1/2 m_p v_f^2$

$V_f = \sqrt{4745/40}$ m/sec

V_f =10.89m/sec

V_f =10.9m/sec

It is already calculated that the impact force;

F_{imp} = 2351706N

$\boldsymbol{\tau} = m_p v_f / F_{imp}$

$\boldsymbol{\tau}$ = 80 x 10.9 / 2351706 sec

$\boldsymbol{\tau}$ =0.000371

$\boldsymbol{\tau} = T_r + T_f$ but $T_r = T_f$ [11]

$T_r = T_f$ = 0.000371/2sec

$T_r = T_f$ = 0.000186sec

According to research paper [11] for constant threshold force(Thrust force) the gap between blows (T_g) is 0.001 – 0.1sec.

3.2.11.Simulation of dynamic stress on the tool

In order to simulate especially the dynamic stress and deformed shape at critical time with transient effect on this thesis will apply the time – history loads and solve the problem by ANSYS with applying multiple load step method.

N.b. In actual application of hydraulic hammer the tool allowable breaking time is 15sec until the rock fracture.

The main objective of the simulation is to determine the response of the tool for 0.015sec.

3.2.11.1.Tabulation of Time-dependent loading on tool.

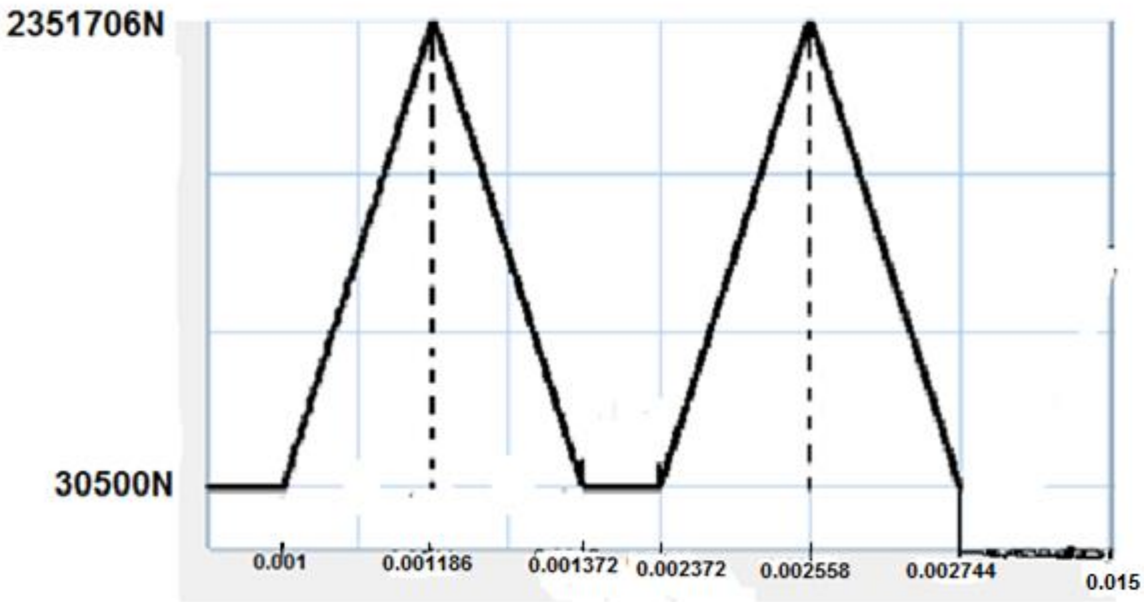


Figure 3.15.Tabulated time dependent loading on tool

3.2.11.2.Multiple Load Step method

In ANSYS this method allows each segment of the load-vs-time to be written in separate load step.

For the above time history load graph, seven load steps are necessary;

1.For constant load of initial thrust force.

2.For up-ramp load of first impact.

3.For down- ramp load of first impact.

4.For gap between impact (with constant thrust load)

5.For up-ramp load of second impact.

6.For down-ramp load of second impact.

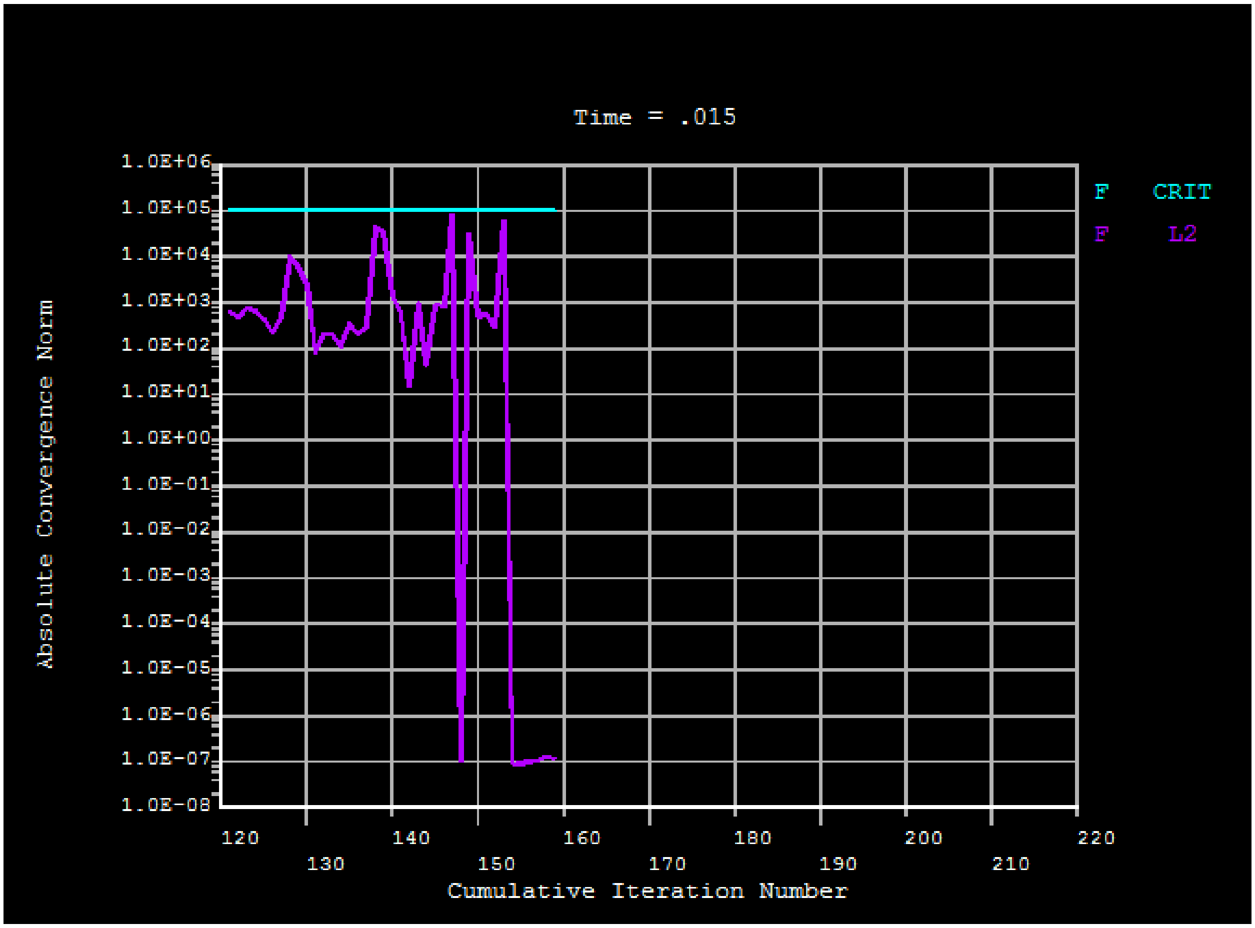
7.Load removal step.

In order to solve the problem it is necessary to write each of the above load steps using ANSYS software program commands.

3.2.11.3.Contact Frequency

Twenty points per cycle are recommended .This is sufficient to capture the momentum transfer between the two objects. A larger Δt might result in energy loss and the impact may not be perfectly elastic[26].

3.2.11.4.Convergence of the Nonlinear solution under Dynamic Load



Firuge 3.16 Convergence of nonlinear solution

3.2.11.5.Dynamic stress on the Rock and Tool assembly

According to ANSYS result with total simulating time of 0.015sec with 7(seven) step loads ,it is clearly shown that the dynamic stress developed on the tool depends on time .It is observed that when the tool impact the rock to break ,the dynamic stress travel from the insert to it's upper most part of the tool. Here it is important to note that the dynamic stress develop in most upper part of the tool is tensile in nature.

The above facts are shown taking the dynamic stress distribution during the impact at time 0.001711sec, 0.003372sec, 0.005033sec, 0.006694sec, 0.008356sec 0.010017sec, 0.011678sec, 0.013339sec, 0.015sec . There are three critical time that maximum tensile stress developed.

Rock and Tool simulation at the end of load step(0.015sec)

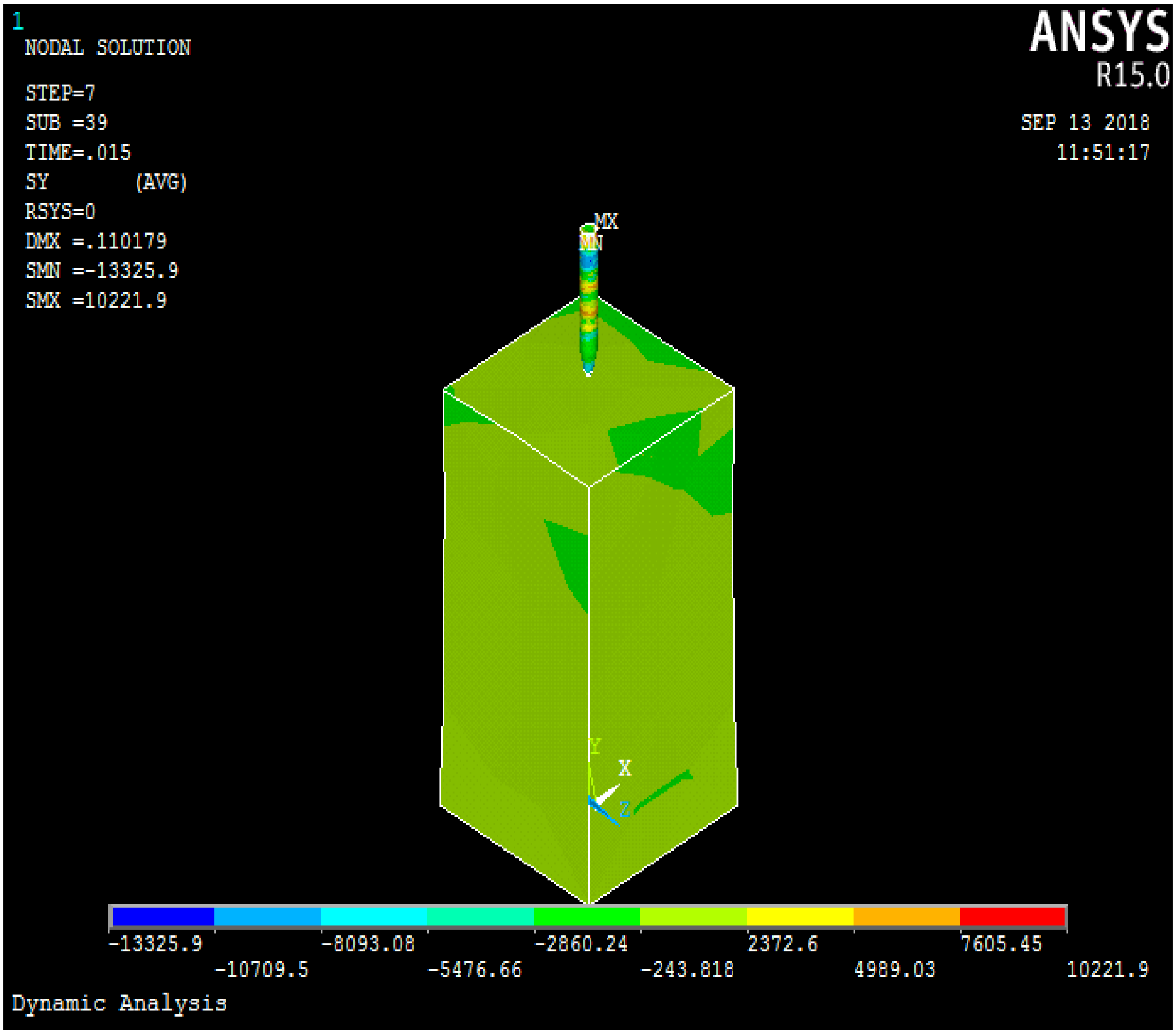


Figure 3.17 Simulated rock tool model

3.2.12. Dynamic Behavior of the shank(penetrator)

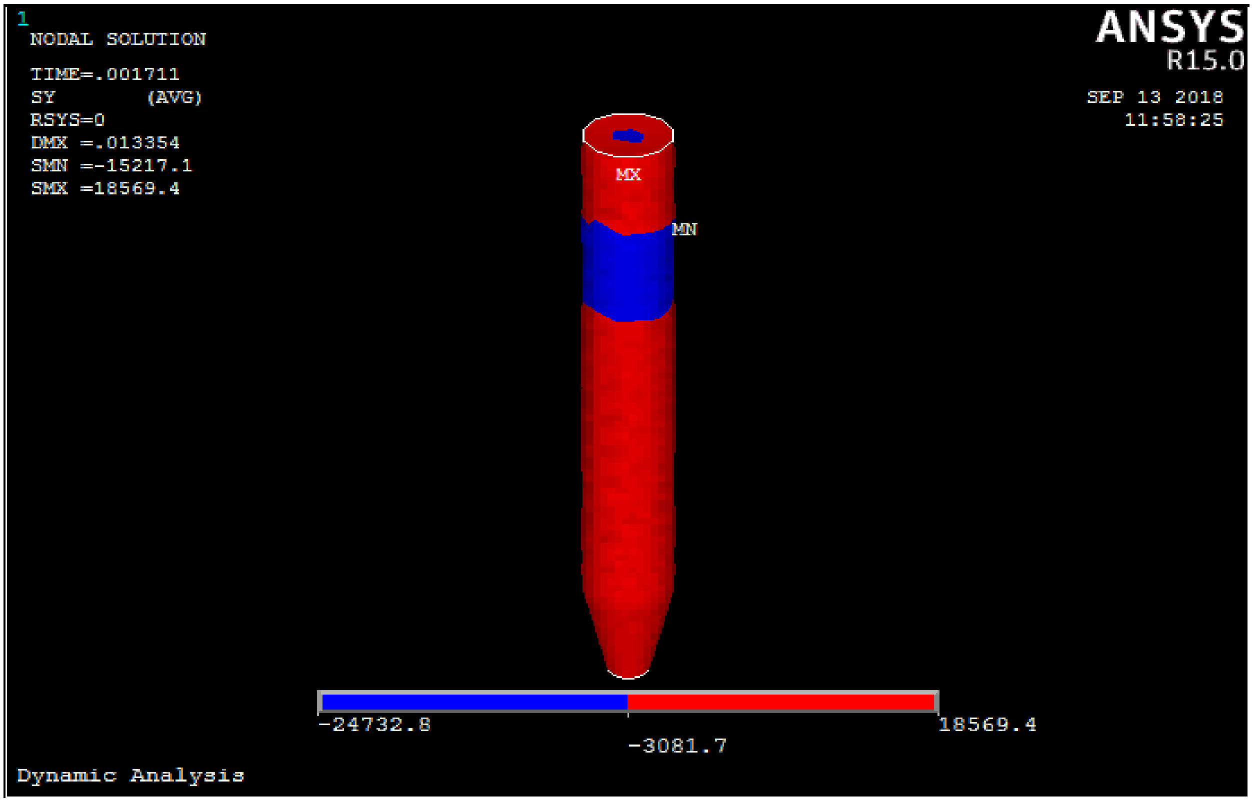


Figure 3.18.Dynamic stress at time, t=0.001711sec

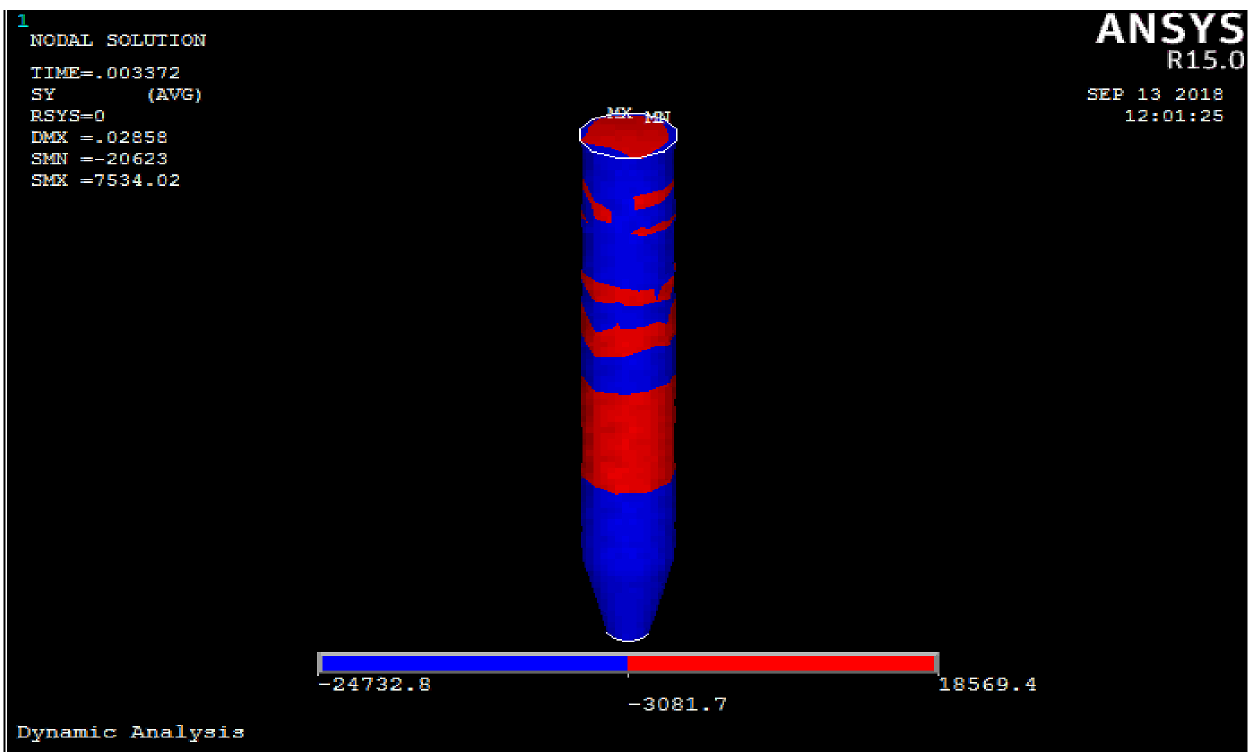


Figure 3.19.Dynamic stress at time, t=0.003372sec

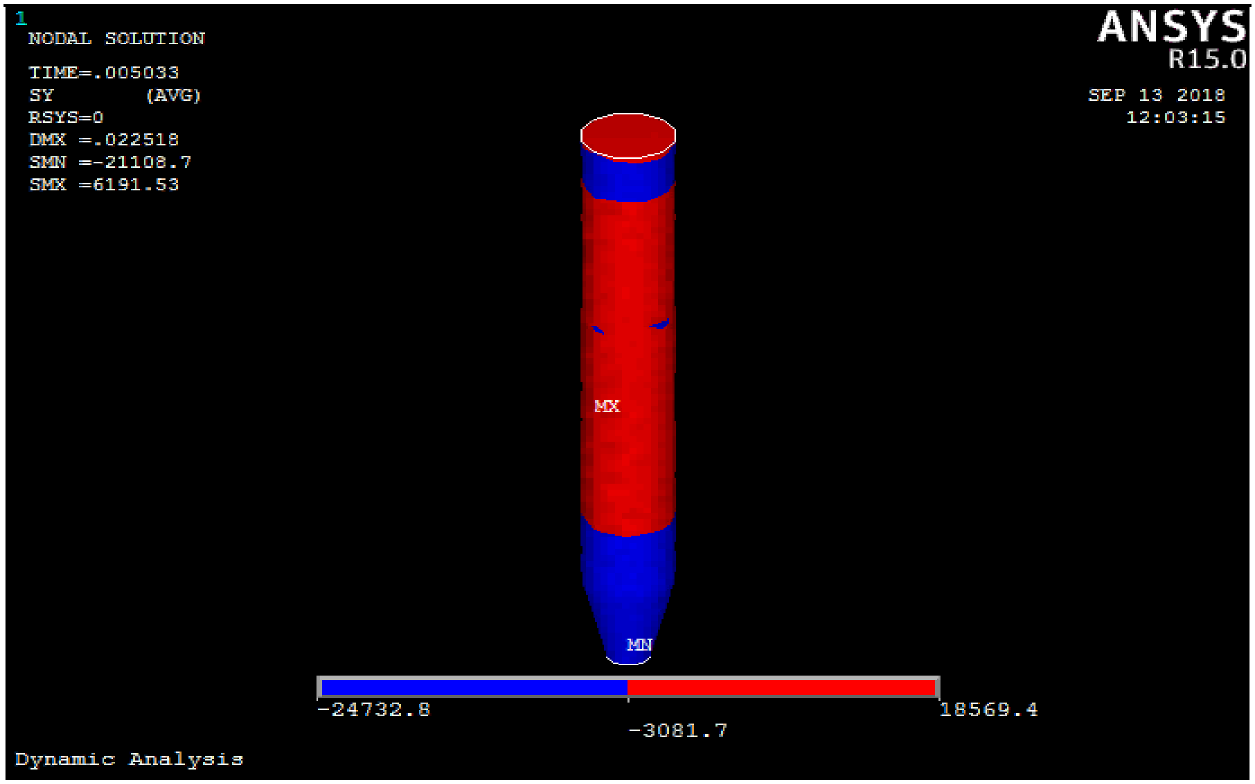


Figure 3.20.Dynamic stress at time, t=0.005033sec

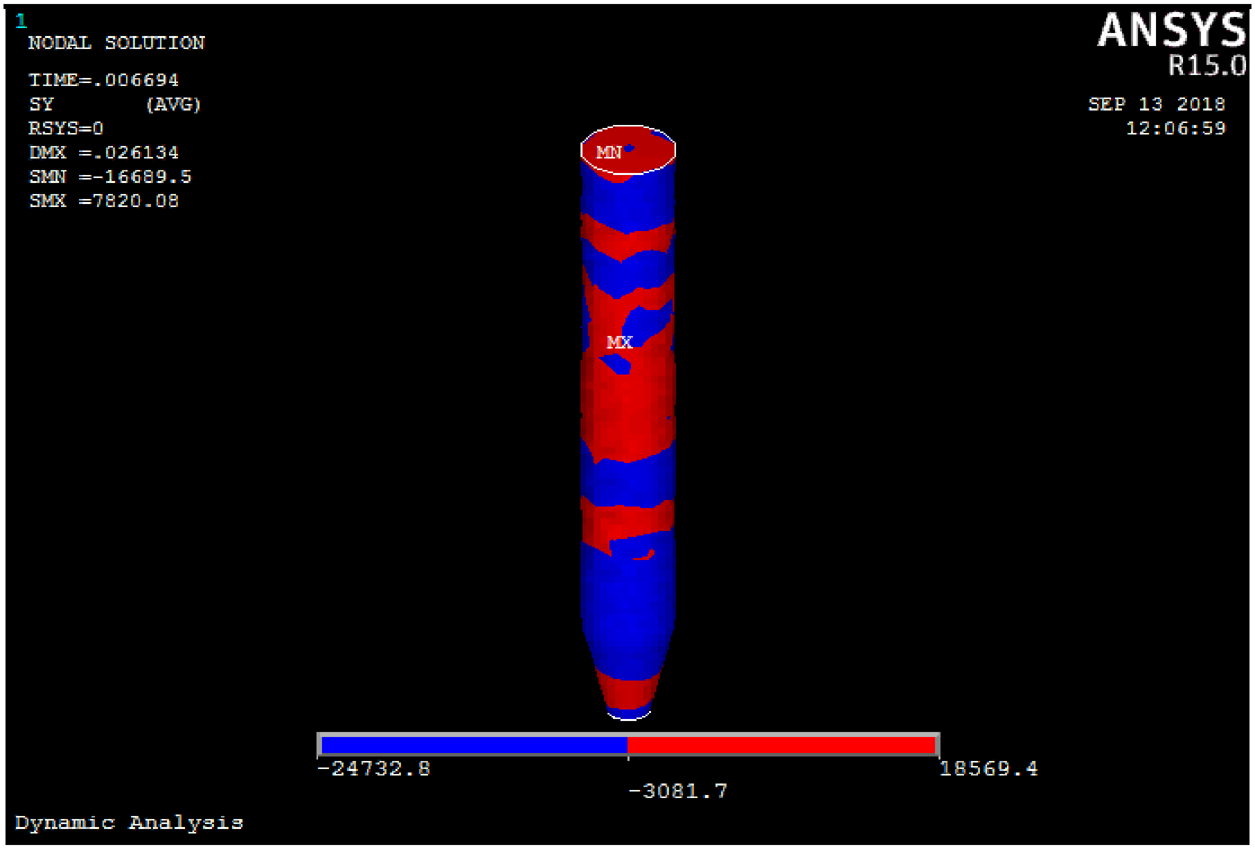


Figure 3.21.Dynamic stress at time, t=0.006694sec

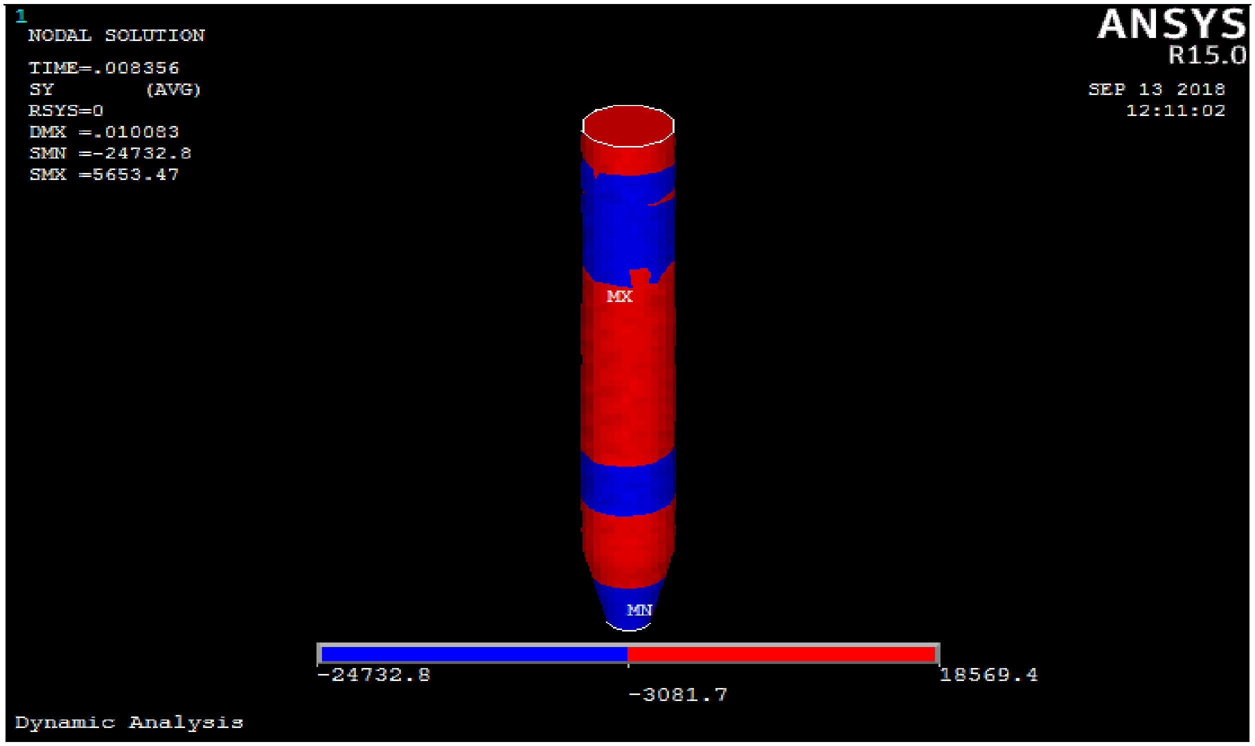


Figure 3.22.Dynamic stress at time, t=0.008356sec

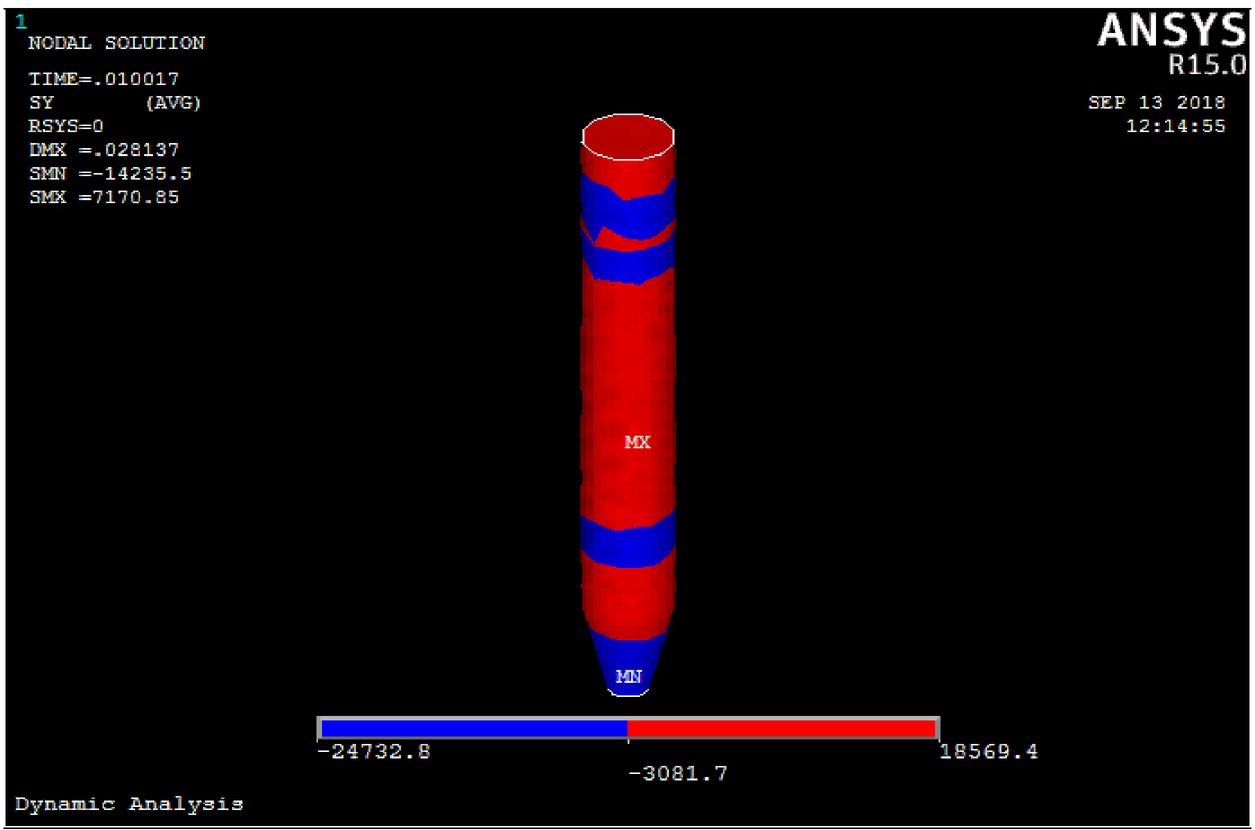


Figure 3.23.Dynamic stress at time, t=0.010017sec

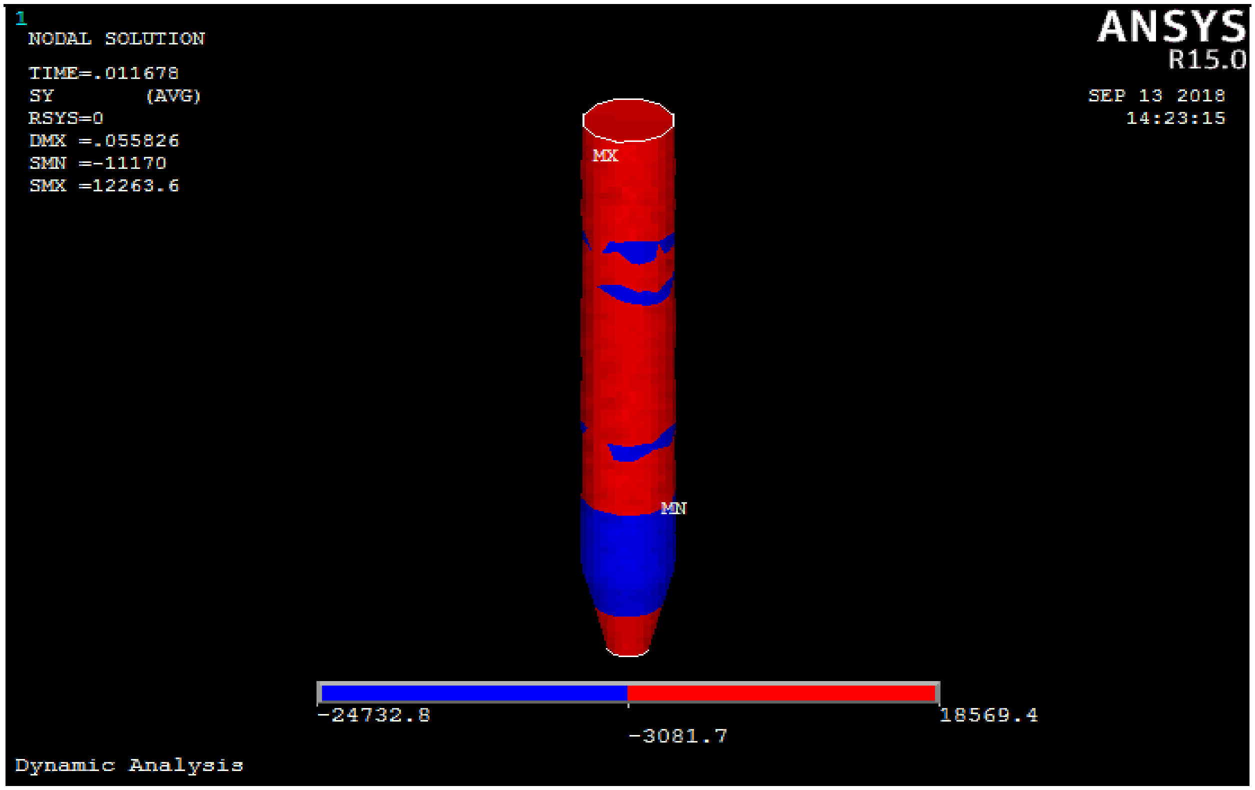


Figure.3.24. Dynamic stress at time, t=0.011678sec

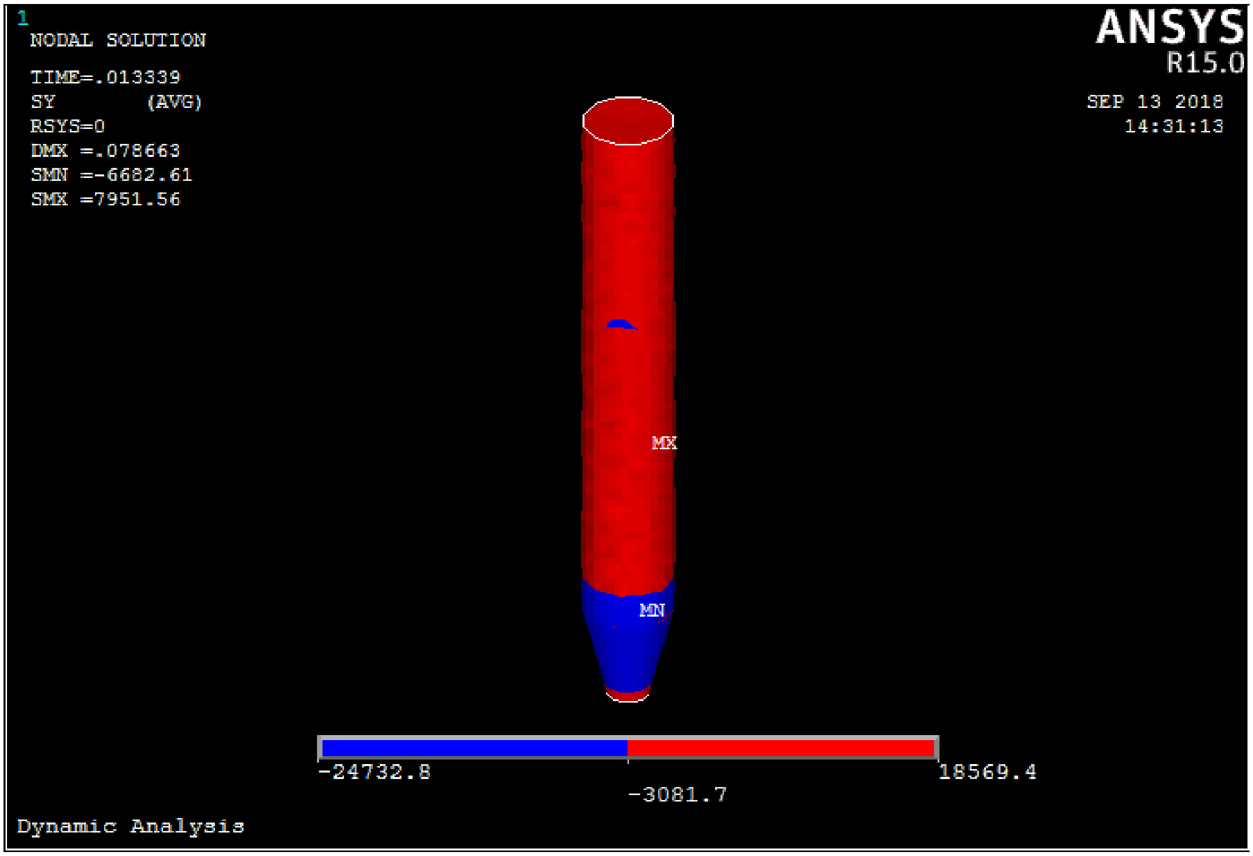


Figure.3.25 Dynamic stress at time, t=0.013339sec

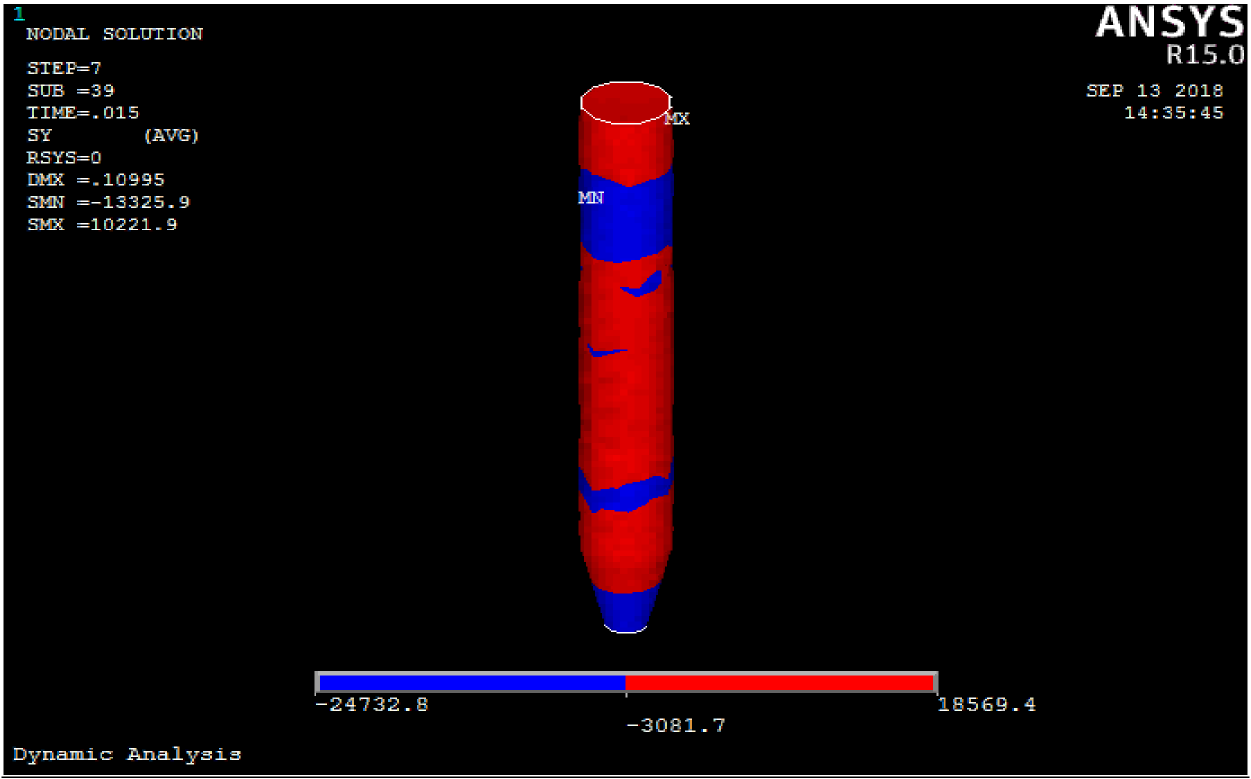


Figure 3.26.Dynamic stress at time, t=0. 015sec

3.2.7. Identification of the Maximum tensile stress Area on tool

If the compressive stress is instantaneously built up and rapidly diminish, on reflection from the bottom of the tool a tensile stress wave is generated and superimposed on compressive stress wave ,at some distance(δ) from the bottom of the tool give rise to the tensile stress on the tool. Therefore at this location with maximum tensile stress will cause fatigue failure of the tool.

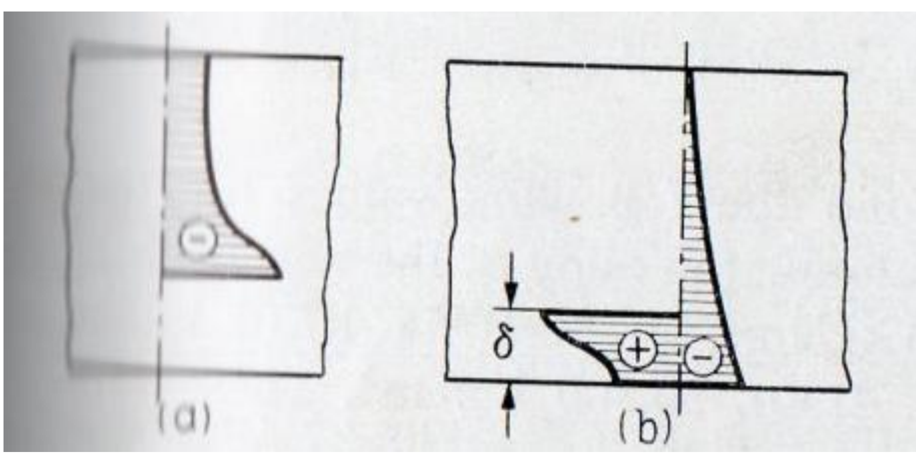


Figure 3.27.Location of maximum tensile stress

The average maximum and minimum stress value at a point at maximum tensile stress area as function of time is shown below.

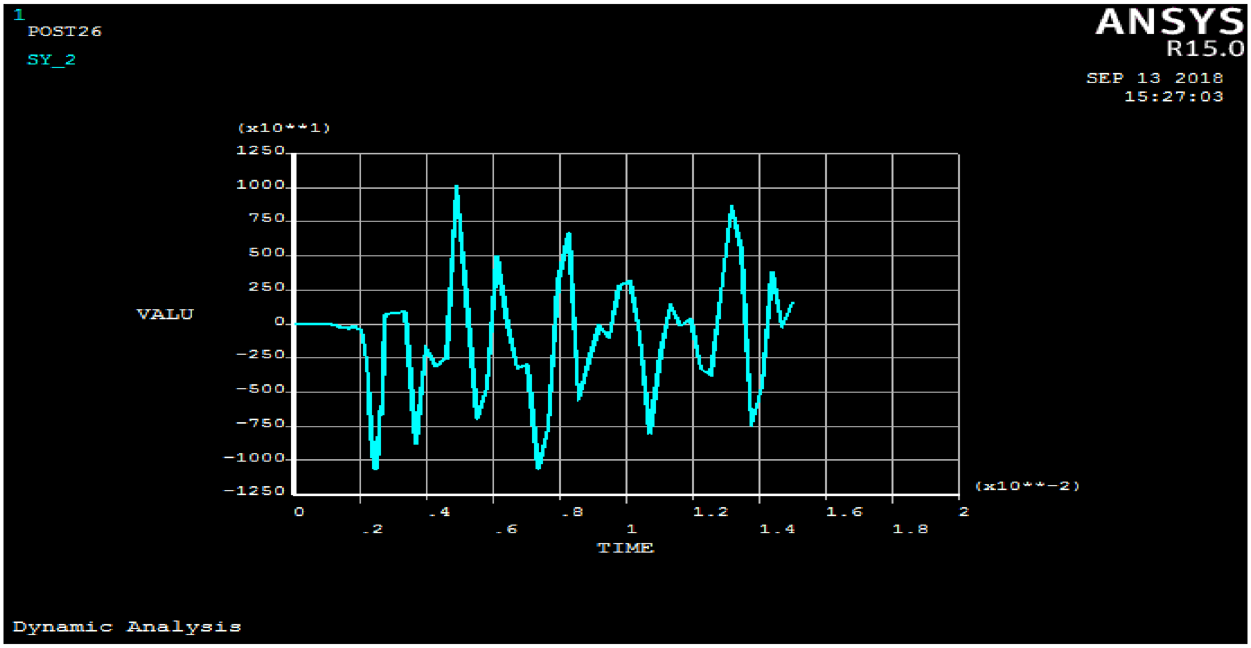


Figure 3.28.Stress against time,t= 0.005033sec

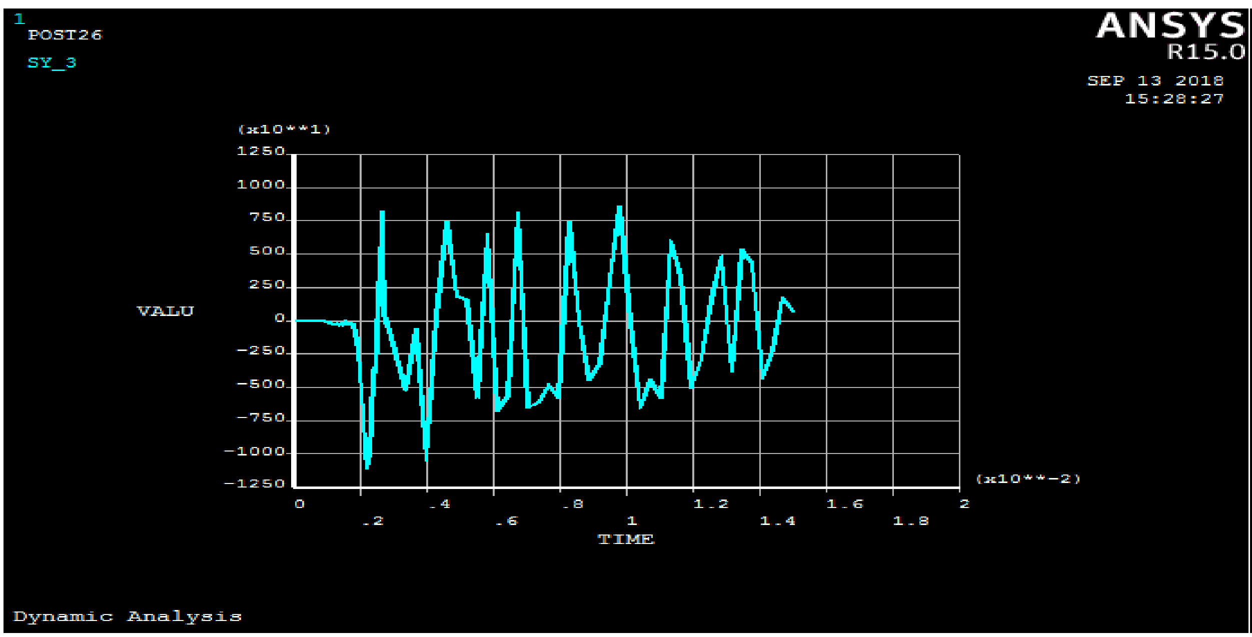


Figure 3.29.Stress against time,t= 0.006694sec

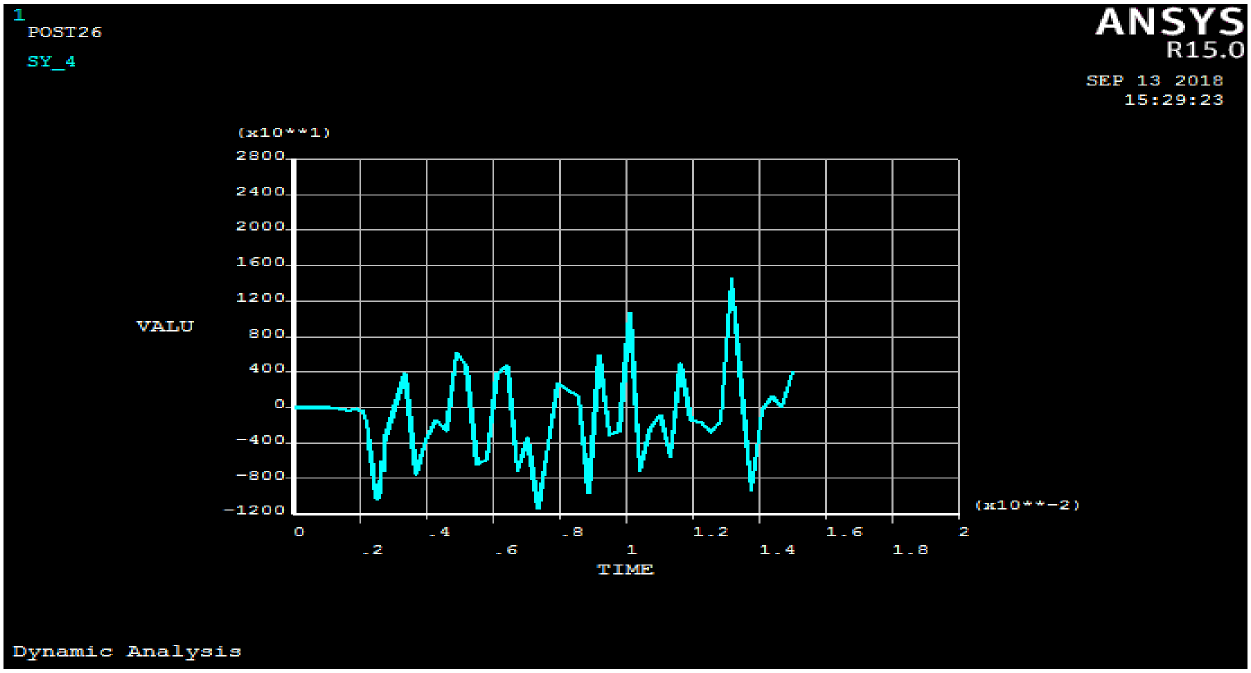


Figure 3.30.Stress against time,t= 0.010017sec

Average maximum and minimum stress Table 3-7

Critical time(sec)	$\sigma_{max}(N/cm^2)$	$\sigma_{min}(N/cm^2)$	Node location
0.005033	10116.8	-10641.2	964
0.006694	8601.95	-11113.7	1439
0.010017	14585.5	-11335.2	1454

3.2.14. Fatigue Failure development on tool

A fatigue failure in a specimen with circular section shown in the figure (3.28) below is compared with the actual fracture of the tool fig(3.29).In both figure below the outer region of the fracture is a smooth shiny surface where as the inner region has a mat granular surface .This type of fracture suggests the following explanation of the way in which the fatigue failure developed. Initially a number of fatigue cracks formed on the surface of the tool where the normal stress is maximum .

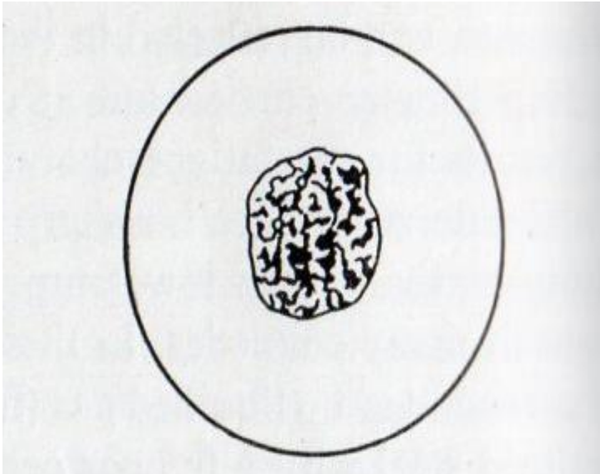


Fig.3.31 ,Fatigue failure in circular section

Fig.3.32 ,Fatigue failure in tool section.

These cracks then merged and spread towards the center of the tool. At the same time the surface of the crack in the compression zone was worn smooth. At some instant when the crack had spread to a sufficient depth the stress concentration created conditions for brittle failure of the remaining part of the section, in this way completing the fatigue failure.

NB:- From figures above it clearly indicate that the tool failure under fatigue load with crack initiation at the surface.

3.2.15.Fatigue life of the tool

According to [15] there are three major approach used in design and analysis to predict when, if ever , a cyclically loaded machine component will fail in fatigue over a period of time.

They are ;

1.The stress-life method.

2.The strain-life method.

3.Linear- Elastic fracture mechanics Method.

For this thesis analysis according to fatigue failure of the tool surface shown in above figure 3.32, it will be advisable the third method of approach to determine the number of cycle or blow to produce a failure after the initial detectable crack is formed.

3.2.15.1.Linear-Elastic fracture mechanics method

The first phase of fatigue cracking on tool failure can be designated as stage I fatigue. This crack is invisible during the service life of the tool. The second phase is that of crack extension is called stage II fatigue with progress of micro crack to macro crack. Final fracture/breakage of the tool occurs during stage III. At this stage ,although fatigue is not involve the type of failure is brittle in nature. Quite often the beach marks, if they exist and possible pattern in stage III fracture called chevron lines ,see figure (3.32).When the crack is sufficiently long that $K_I = K_{IC}$ for stress amplitude involved. K_{IC} is the critical stress intensity for undamaged metal.

3.2.15.2.Estimation of Number of cycle(Blow) to failure

Fatigue crack nucleated and grow on the tool when the stress vary and there is some tension in each stress cycle.

The stress fluctuating on tools between limit σ_{min} and σ_{max} at tensile stress area at a point discussed above.

Stress range($\Delta\sigma$) = $\sigma_{max} - \sigma_{min}$

From fracture mechanics the stress intensity is given by;

$K_I = \beta\sigma\sqrt{\pi a}$ ....(a)

If $\Delta\sigma$ is a stress intensity range per cycle.

➯ $\Delta K_I = \beta(\sigma_{max} - \sigma_{min})\sqrt{\pi a}$

$= \beta\Delta\sigma\sqrt{\pi a}$ ....(b)

Crack nucleated at or very near a free surface of the tool.

Assume the initial crack length is a_i ,and crack growth as function of N will depend on $\Delta\sigma$ and crack is discovered early in stage II.

Crack growth in stage II approximated by Paris equation,

$$\frac{da}{dN} = c(\Delta KI)^m$$

C and m are empirical material constant

Substitute equation (b) and integrate gives ;

$$\int_0^{Nf} dN = N_f = \frac{1}{c}\int_{ai}^{af} \frac{da}{(\beta\Delta\sigma\sqrt{\pi a})m}$$

ai = initial crack length

af = final crack length corresponding to failure.

N_f = Estimated number of cycle to produce a failure after the initial crack is formed.

β = Integration variable.

For brittle fracture, designated the crack length af ,with $K_I = K_{IC}$, approximate af as;

$$af = \frac{1}{\pi} (K_{IC}/\beta\sigma_{max})^2$$

Minimum detectable Crack length (ai)

In reality, the minimum detectable crack length $\boldsymbol{ai}$ is not absolute limit, as the probability of finding a crack in inspection increases with crack size ,but it never 100% .Note that $\boldsymbol{ai}$ is usually defined as the depth of surface crack. Cracks as small as $\boldsymbol{ai}$ = 0.1mm can be found, but $\boldsymbol{ai}$ values this small can be justified in special cases[16]

Determination of the Critical (Final Crack) Length(af)

The fracture toughness of the shank is 93MPa$\sqrt{m}$. The maximum average stress at critical area of fatigue failure area is σ_{max} = 16284.2N/cm^2(163MPa).

$$af = \frac{1}{\pi} (K_{IC}/\beta\sigma_{max})^2$$

$$\beta = 0.728 \text{ (for } a/d < 0.2)$$

a/d=0.1/130=0.000769

$$af = \frac{1}{\pi} (93/(0.728*163)^2$$

$$af = 0.1954$$

Estimation of Fatigue life

$$N_f = \frac{1}{c}\int_{ai}^{af} \frac{da}{(\beta\Delta\sigma\sqrt{\pi a})m}$$

Since after the outer hardened layer of 1.2mm is most of the tool material is austenitic steels.

$C=5.61(10^{-12})$ and $m=3.25$ [15]

3.2.15.3.Factor affecting the Fatigue life of the tool

From the above calculation the fatigue failure at the critical area on the tool mainly depend on the range or maximum stress developed on the tool surface. It is already discussed that at the critical area the reflected tensile stress from lower tip of the tool superimposed on the compressive stress causing the increase in tensile stress .At a point on the critical area the maximum/range of stress is depend on the tensile stress developed at this area.

Factors affecting the tensile stress developed at the critical area will depend on the hardness of the insert material and contact parameters. In this thesis the contact parameter such as surface contact area between the tool and the rock surface will be investigated to evaluate the fatigue failure of the tool.

3.2.16.The Effect tip insert diameter on Maximum tensile stress

The designed insert tip diameter was 20cm.Let's investigate the tensile stress developed at critical area with recommended standard tip insert diameter with predefined material of the tool.

Selected diameters for the insert tip of the tool were;

D_1=1.6cm, D_2=1.8cm ,D_3=2.0cm ,D_4=2.2cm, D_5=2.4cm, D_6=2.4cm

1. Tensile stress developed on tool (at t=0.005033sec)

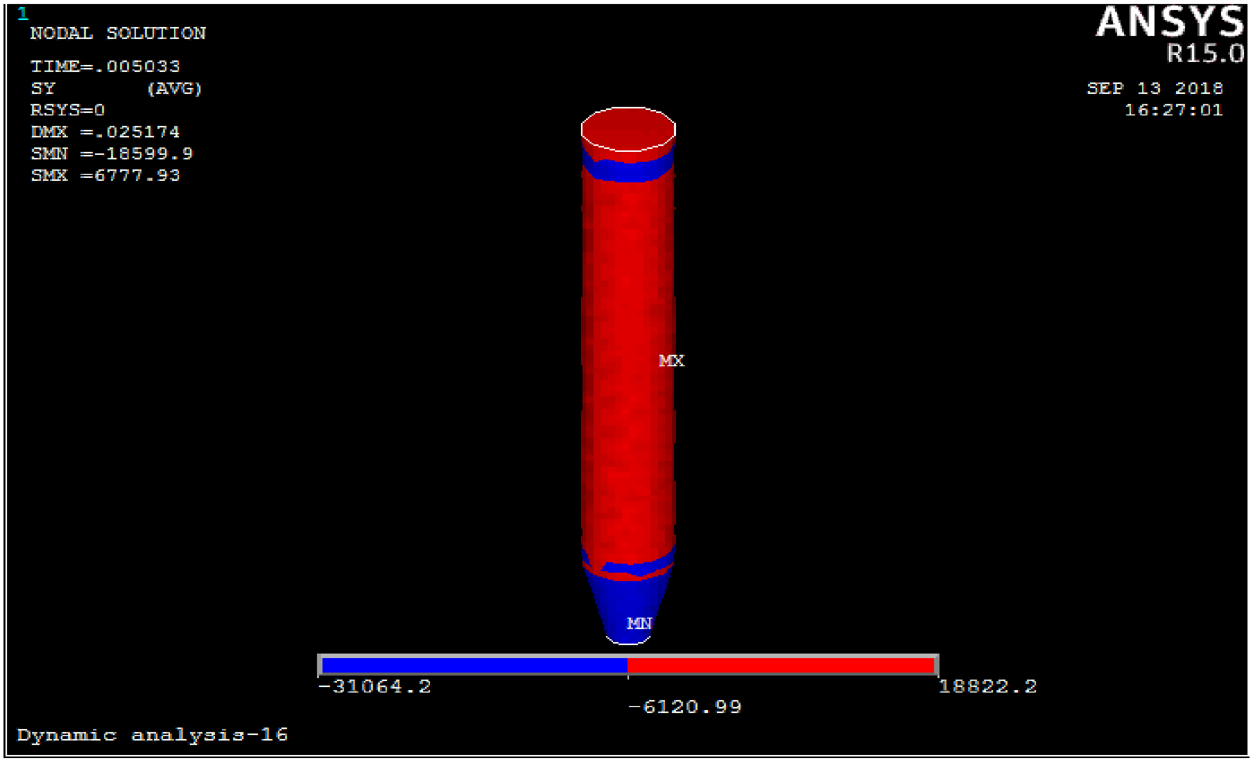


Figure 3.30.Maximum tensile stress , D_1=1.6cm

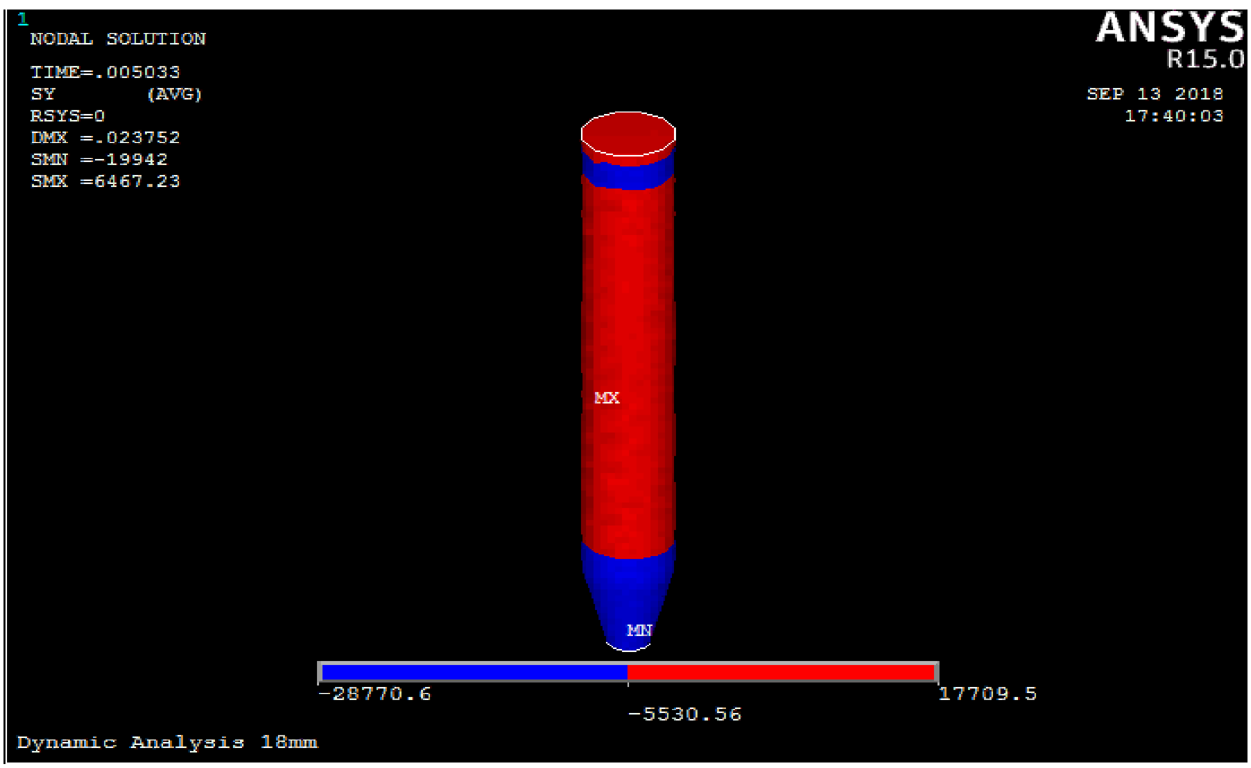


Figure 3.31.Maximum tensile stress , D_2=1.8cm

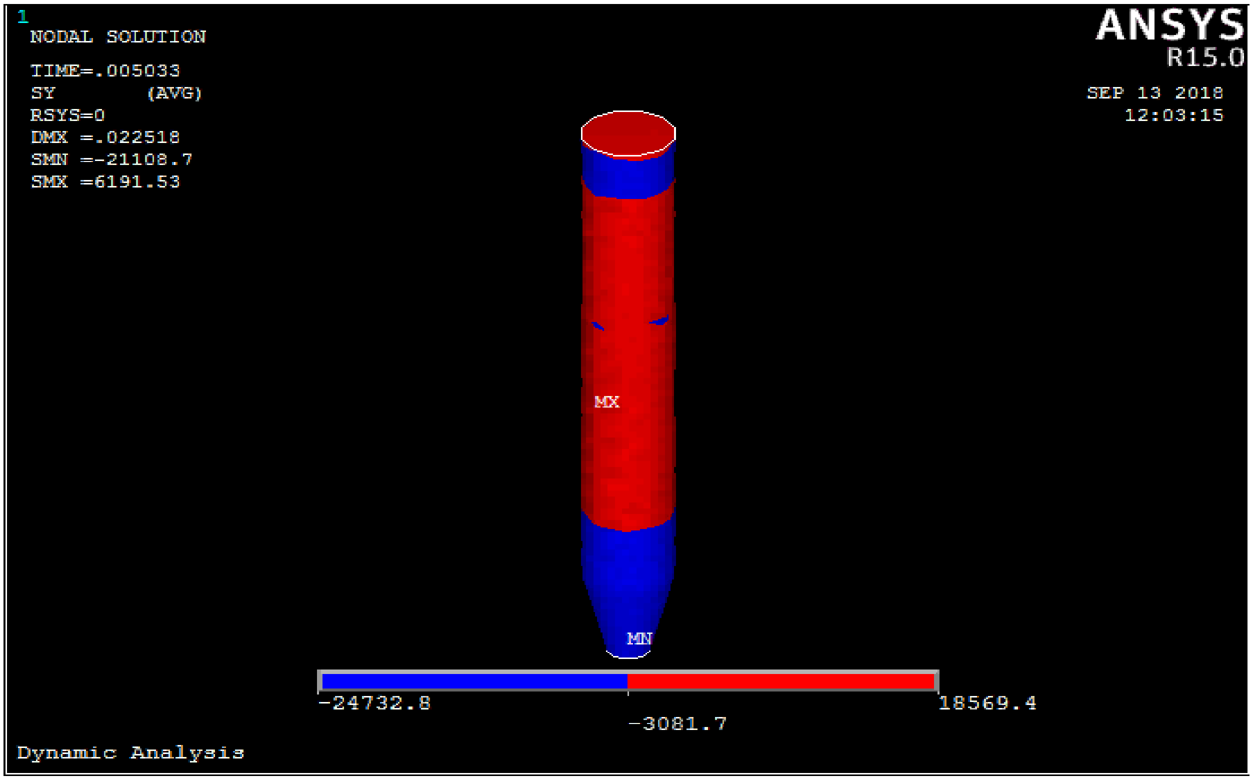


Figure 3.32.Maximum tensile stress , D_3=2.0cm

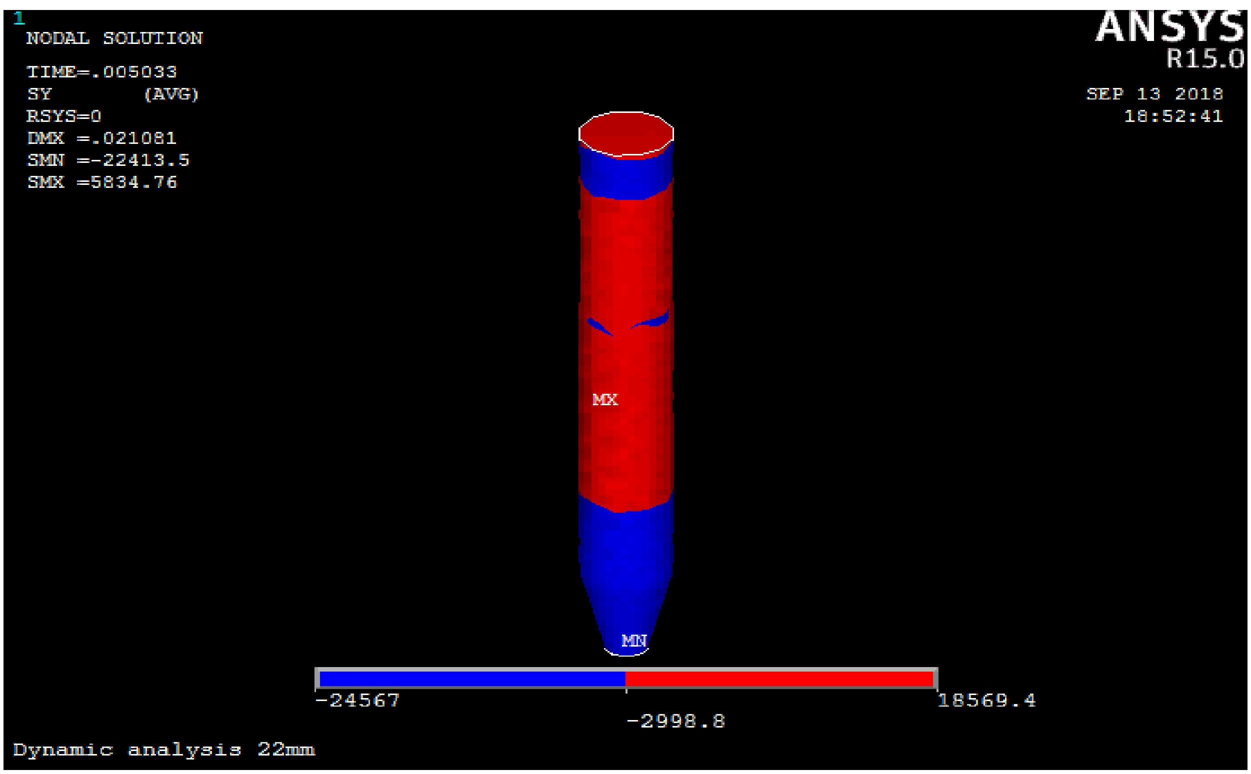


Figure 3.33.Maximum tensile stress , D_4=2.2cm

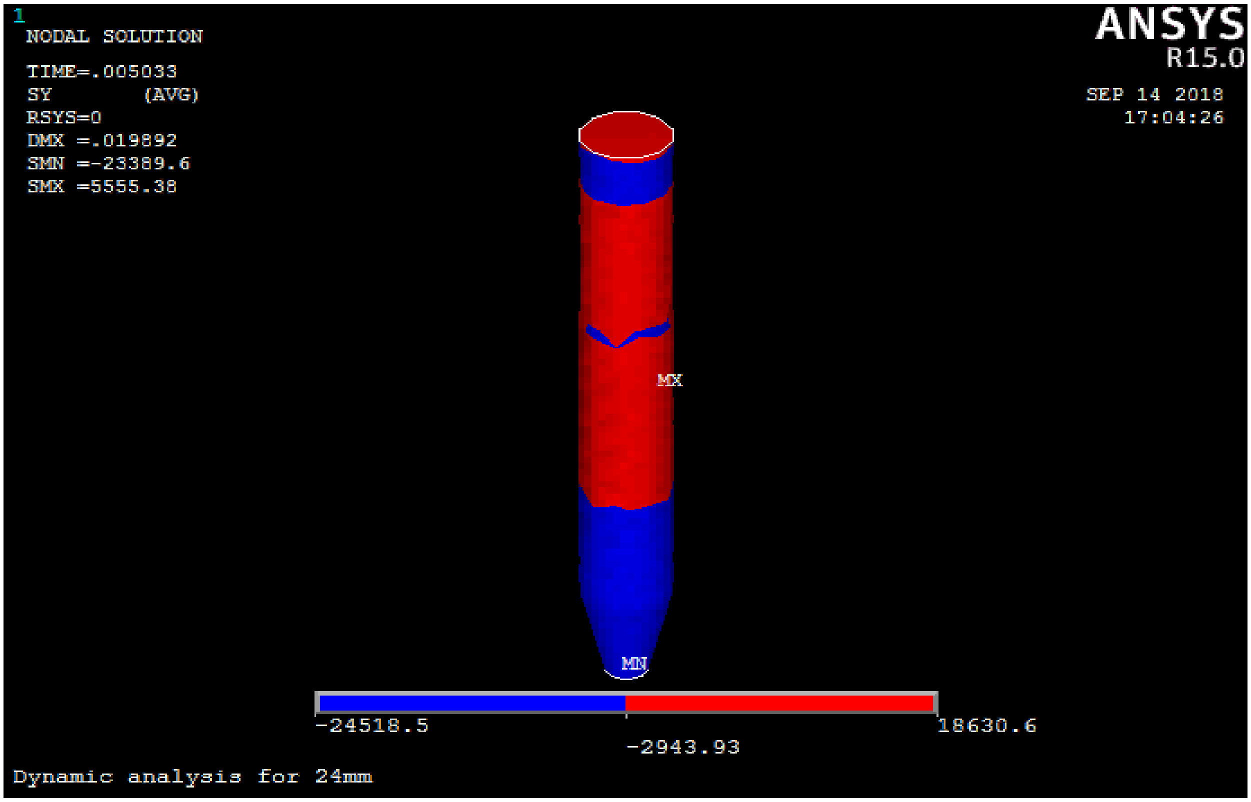


Figure 3.34.Maximum tensile stress , D_5=2.4cm

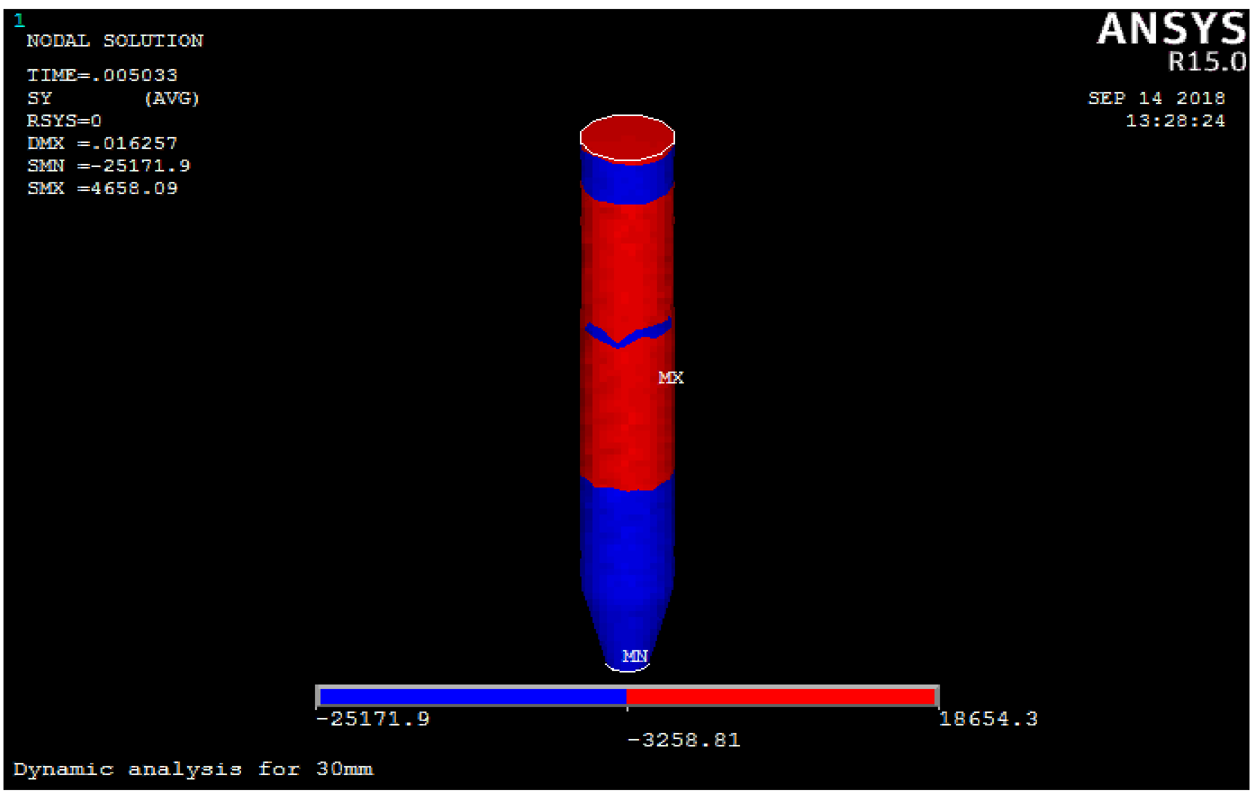


Figure 3.35.Maximum tensile stress , D_6=3.0cm

3.2.17.Tensile stresses developed on tools at critical times

Here the summarized table of tensile stresses developed is putted on tables for the actual design and the recommended tool insert diameter at critical times.

Tensile stress at critical time Table 3.8

No	Time(sec)	Tensile Stress (N/cm²)					
		D=1.6cm	D=1.8cm	D=2.0cm	D=2.2cm	D=2.4cm	D=3.0cm
1	0.001711	18222.2	17709.5	18569.4	18569.4	18630.6	18654.3
2	0.003327	7165.41	5918.23	7534.02	7534.24	6924.58	6876.96
3	0.005033	6777.93	6467.23	6191.53	5834.76	5555.38	4658.09
4	0.006694	8007.05	7894.64	7820.08	7859.18	7716.47	8156.64
5	0.008356	6379.56	6285.42	5653.47	5693.9	5461.62	5721.8
6	0.010017	4677.28	4914.11	7170.85	7247.82	7488.18	7947.12
7	0.011678	14668.4	12888.5	12263.6	12387	13540.7	13571.1
8	0.013339	7053.89	7387.24	7951.56	8204.58	8177.95	7362.89
9	0.015	9761.42	10780.8	10221.9	10164.2	10727.6	11100.5

3.2.18 Maximum tensile stress area

In my thesis proposal I have already suggested that tool failure due to fatigue generally occur within certain distance above and below the face of front head of hydraulic hammer. Especially a tool metal fatigue failure generally occurs within 4in (10cm) above and below the face of font head, and below top edge of the tool.

The working length at normal condition for this tool when measured from bottom edge of the insert is 67.5cm.So that the fatigue failure area for the tool under investigation is expected to be happen from 57.5cm to 77.5cm.

Table 3.9

No	Critical time(sec)	Tool insert tip diameter(cm)	Maximum tensile stress location (cm)	Tool section position	Remark
1	0.005033	1.6	64.7	Below the face of front head of hammer housing	Length measured from bottom tool edge.
		1.8	58.24		
		2.0	57.6		
		2.2	57.6		
		2.4	64.7		
		3.0	64.7		
2	0.006694	1.6	76.4	Above the face of front head of hammer housing	Length measured from bottom of tool edge.
		1.8	76.4		
		2.0	76.4		
		2.2	76.4		
		2.4	76.4		
		3.0	76.4		
3	0.010017	1.6	63.4	Below face of front head of hammer housing	Length measured from bottom of tool edge.
		1.8	63.4		
		2.0	53.1		
		2.2	53.1		
		2.4	63.4		
		3.0	-		

3.2.19.Estimated number of cycle(Blow) to failure

Estimated number of cycle/blow to failure Table3.10

No	Critical Time(sec)	Tool insert tip diameter(cm)	Stress at a point in tensile region(MPa)			
			Average maximum stress, σ_{max} (MPa)	Average minimum stress, σ_{min} (MPa)	Range of stress($\Delta\sigma$) (MPa)	Estimated cycle/Blow to failure
1	0.005033	1.6	106.87	-110.09	216.96	$9.99 x 10^5$
		1.8	105.72	-109.17	214.89	$10.31 x 10^5$
		2.0	101.17	-106.41	207.58	$11.54 x 10^5$
		2.2	97.83	-106.52	204.35	$12.14 x 10^5$
		2.4	92.92	-108.58	201.5	$12.71 x 10^5$
		3.0	85.66	-108.96	194.62	$14.24 x 10^5$
1	0.006694	1.6	79.28	-111.66	190.94	$15.16 x 10^5$
		1.8	81.69	-111.40	193.09	$14.61 x 10^5$
		2.0	86.02	-111.14	197.16	$13.65 x 10^5$
		2.2	84.29	-111.16	195.35	$14.07 x 10^5$
		2.4	94.98	-112.34	207.34	$11.59 x 10^5$
		3.0	99.37	-112.19	211.56	$10.85 x 10^5$
1	0.010017	1.6	139.35	-119.02	158.37	$5.65 x 10^5$
		1.8	144.29	-117.82	262.11	$5.39 x 10^5$
		2.0	145.86	-113.35	259.21	$5.59 x 10^5$
		2.2	133.66	-110.84	244.5	$6.76 x 10^5$
		2.4	75.56	-144.50	220.06	$9.56 x 10^5$
		3.0	-	-	-	-

3.2.20.The Effect of thrust force on tool Failure

According to the theory of impact the action of non-impulsive forces such as thrust force can be neglected in time of impact .But let's investigate the tool rock impact simulation without thrust force.

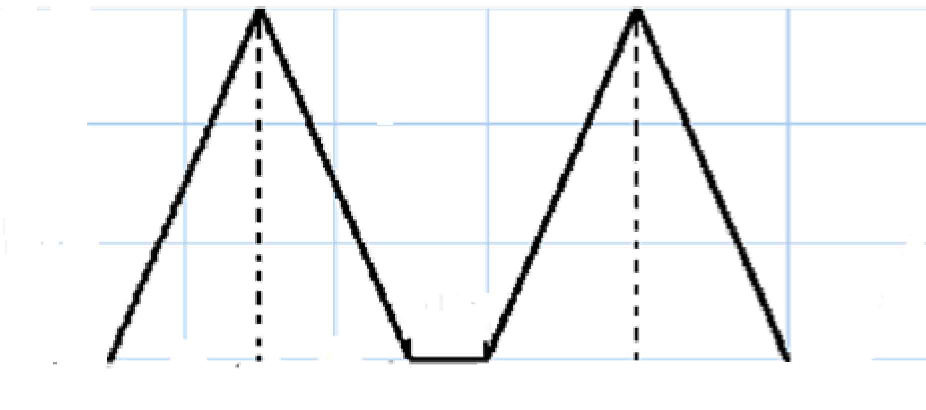


Figure 3.36 Tabulated time dependent loading without thrust force

3.2.21.Location of Maximum tensile stress developed on tool without thrust force

According to ANSYS dynamic stress analysis result simulating for 0.015sec with 5 step loads the effect of neglecting the thrust force is shown below for the actual design case of the tool.

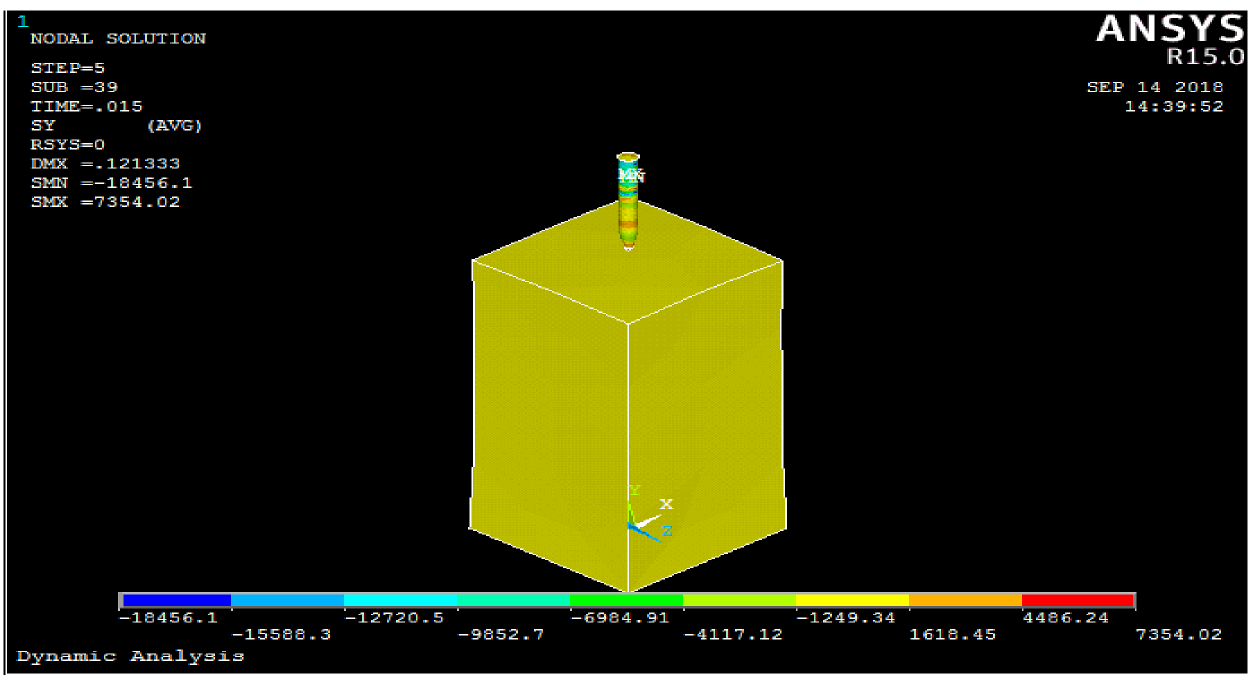


Figure 3.37. Simulated rock tool model

Dynamic Behavior of the shank without thrust force

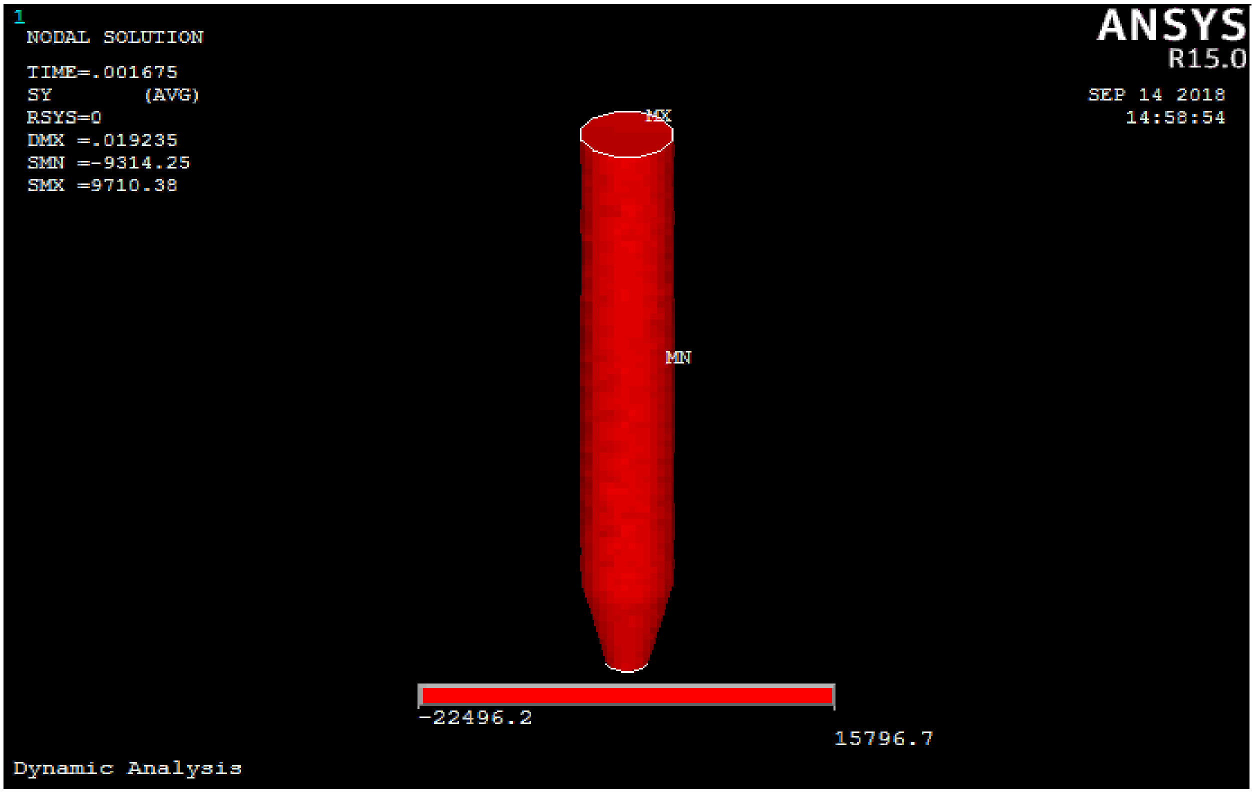


Figure 3.38.Dynamic stress at time, t=0.001675sec

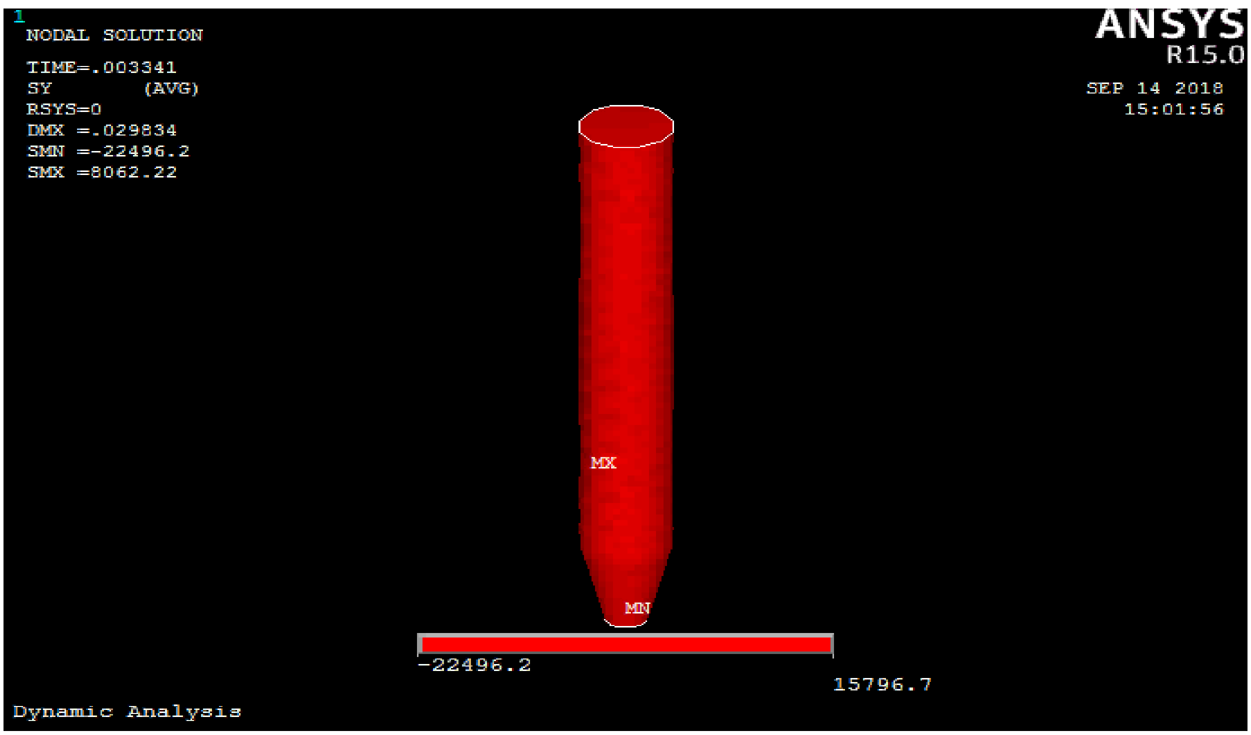


Figure 3.39.Dynamic stress at time, t=0.003341sec

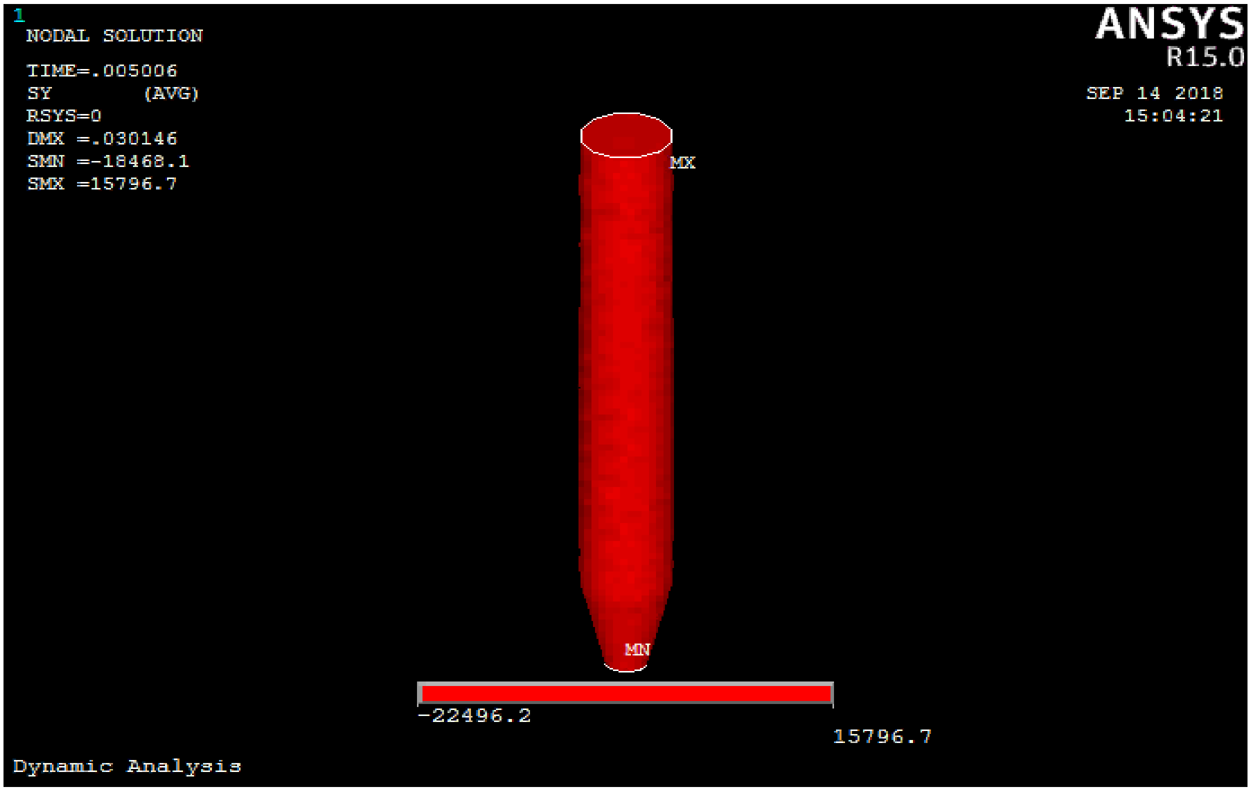


Figure 3.40.Dynamic stress at time, t=0.005006sec

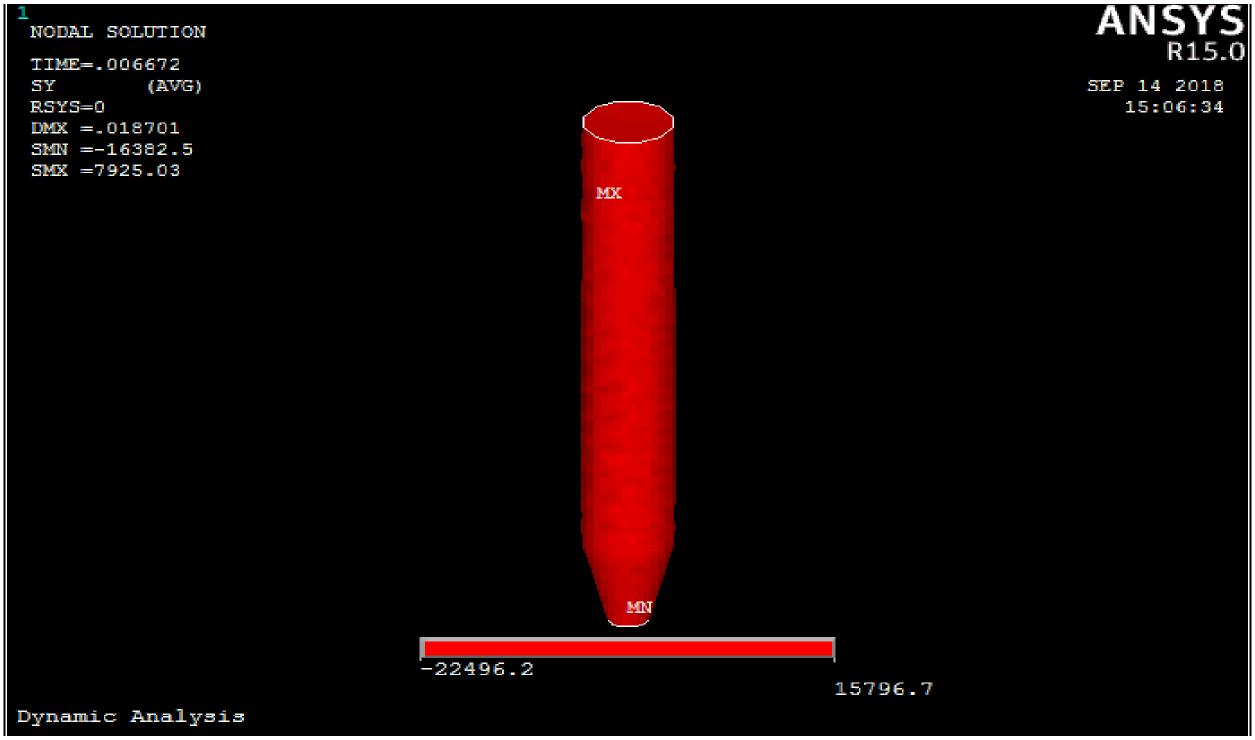


Figure.3.41, Dynamic stress at time, t=0.006672sec

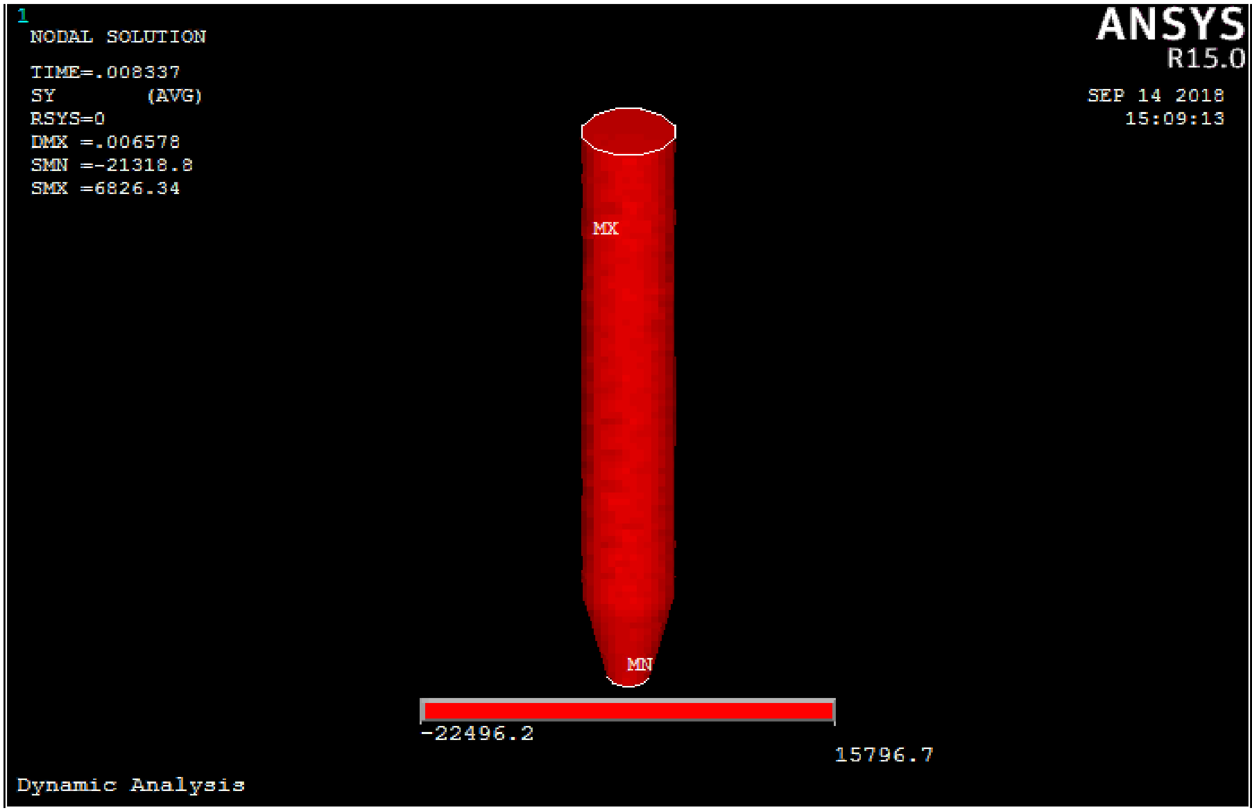


Figure.3.42. Dynamic stress at time, t=0.008337sec

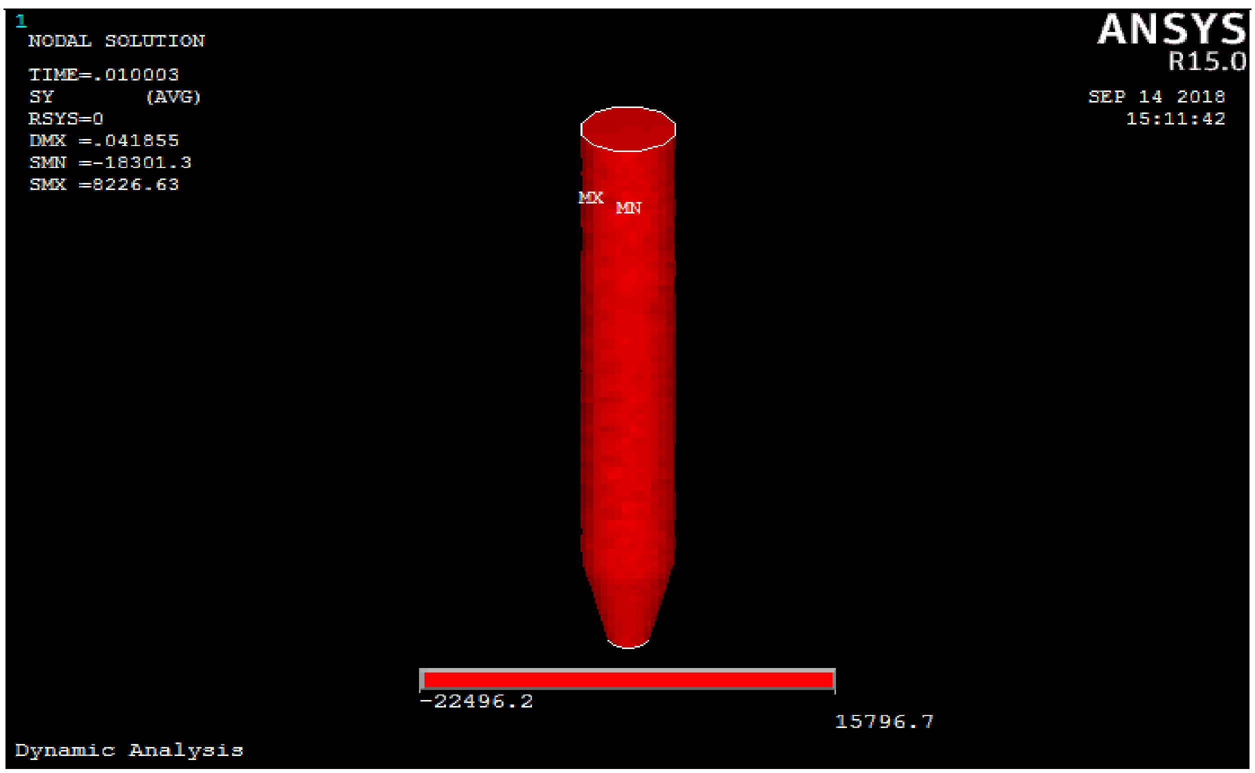


Figure.3.43. Dynamic stress at time, t=0.010003sec

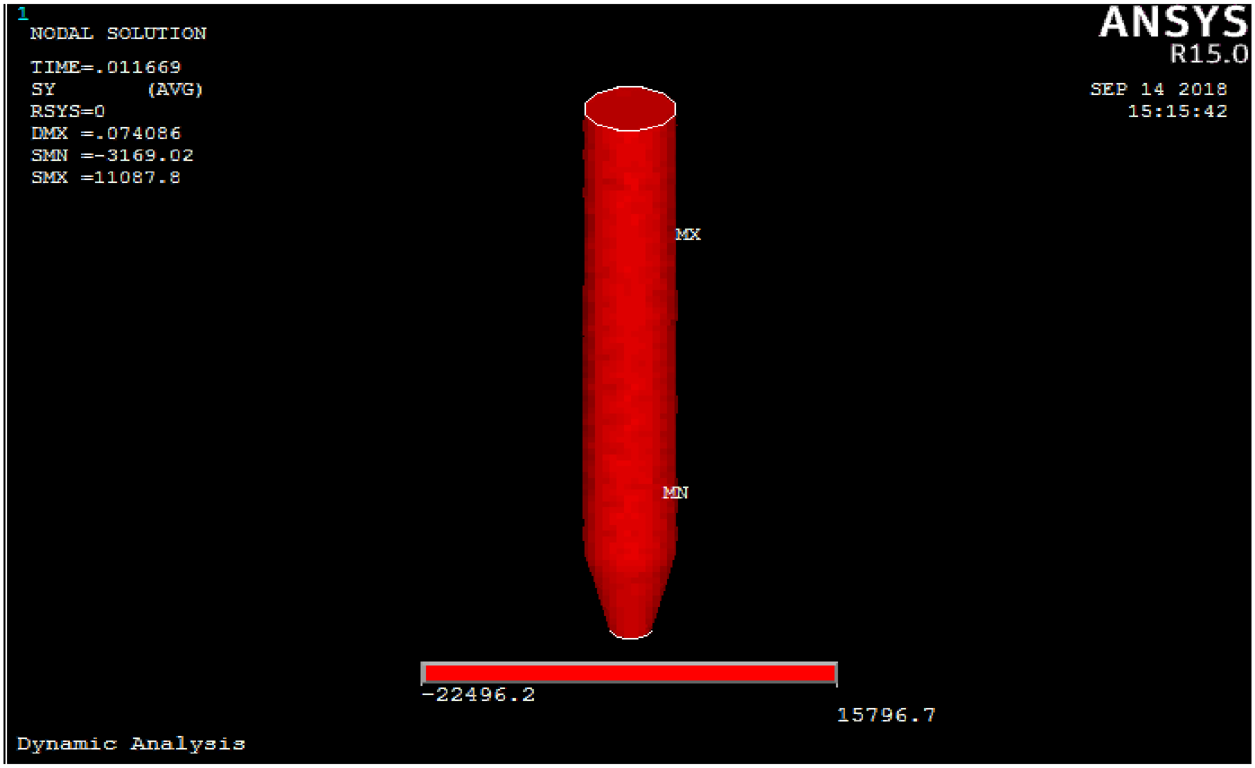


Figure 3.44. Dynamic stress at time, t=0.011669sec

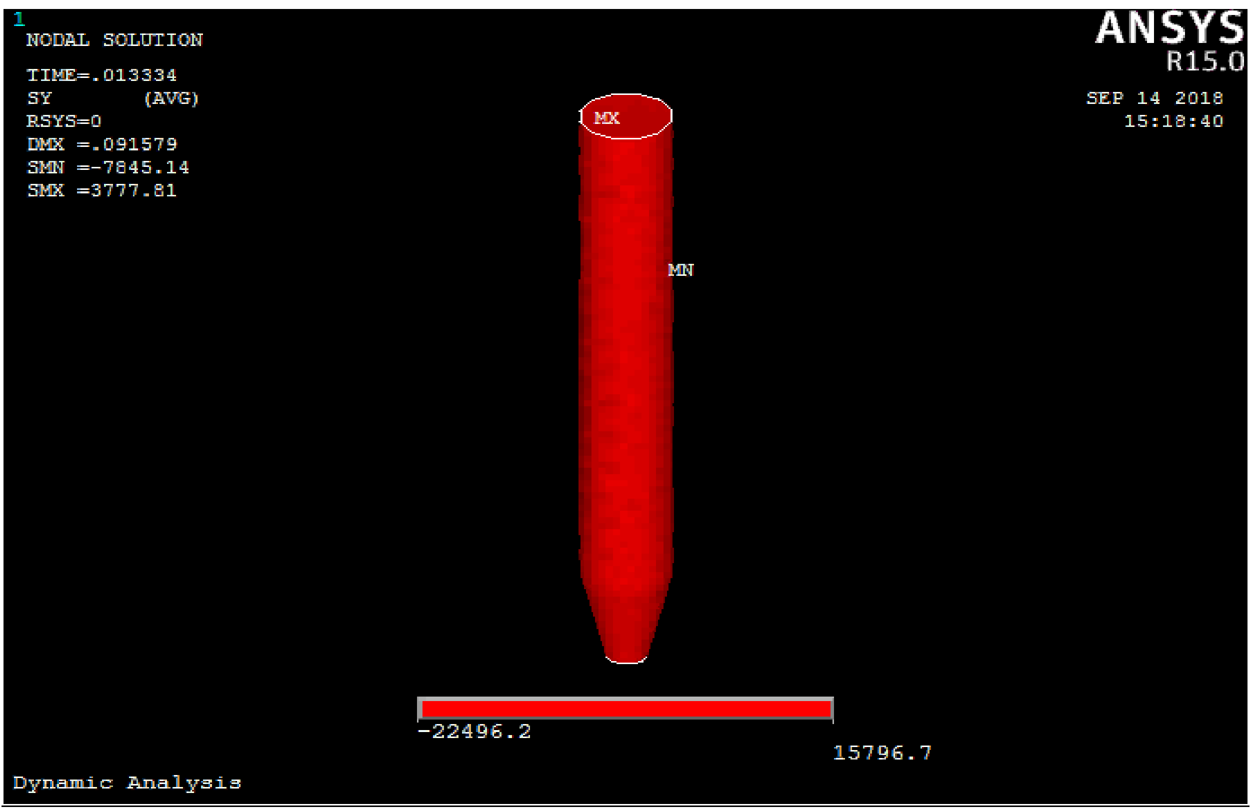


Figure 3.45. Dynamic stress at time, t=0.013334sec

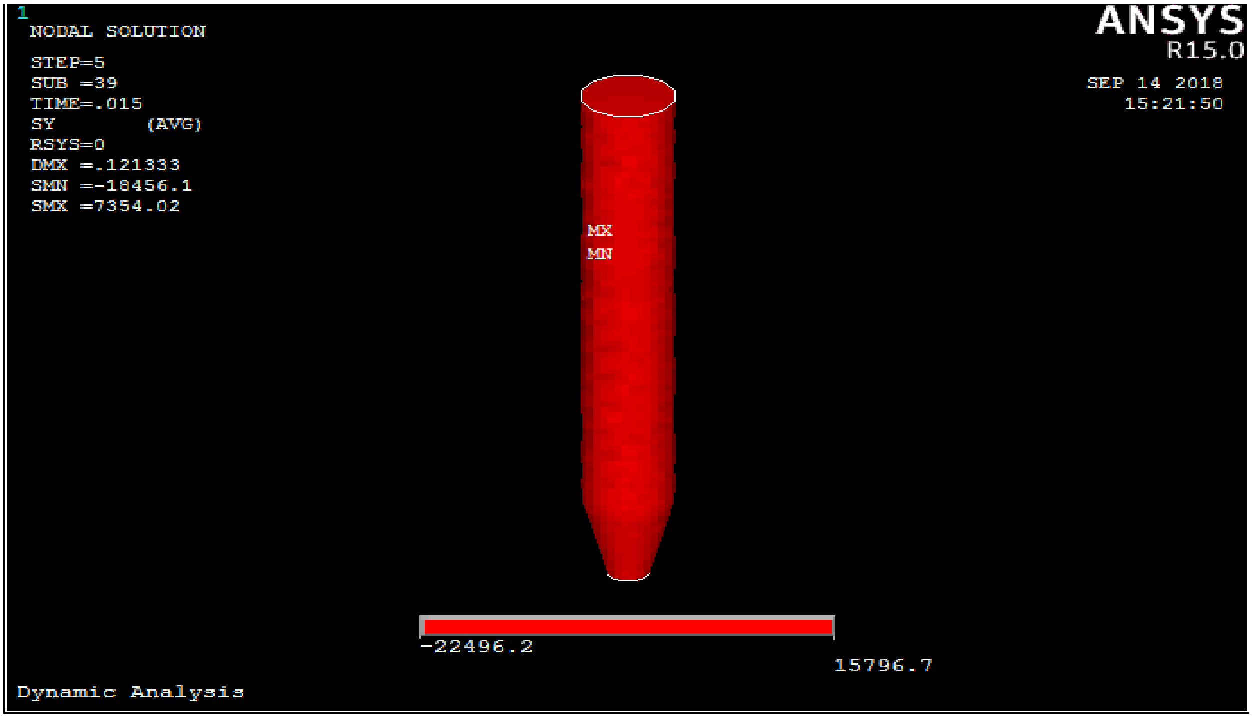


Figure.3.46. Dynamic stress at time, t=0.015sec

According to the above ANSYS dynamic stress simulation there are four critical times where the stress is maximum in the tool part. Those times are t=0.003341sec, t=0.005006sec , t=0.006672sec and 0.08337sec.

Maximum tensile stress without thrust force Table 3-11

No	**Tool insert tip diameter (cm)**	**Maximum tensile stress location**	**Remark**
1	2.0	In most case above middle section	Length measured from bottom of insert/Bottom tool edge.

According to the ANSYS result, the theory of impact to neglect the thrust force causes the solution of the analysis to deviate from the solution in section(3.2.18) . This is shown by the maximum tensile stress observed on tool area above the mid length. Therefore the thrust force must be considered for the analysis .So that thrust force is one of the main parameter considerations in tool design.

Chapter Four : Result & Discussion

4.1A.Dynamic behavior Analysis

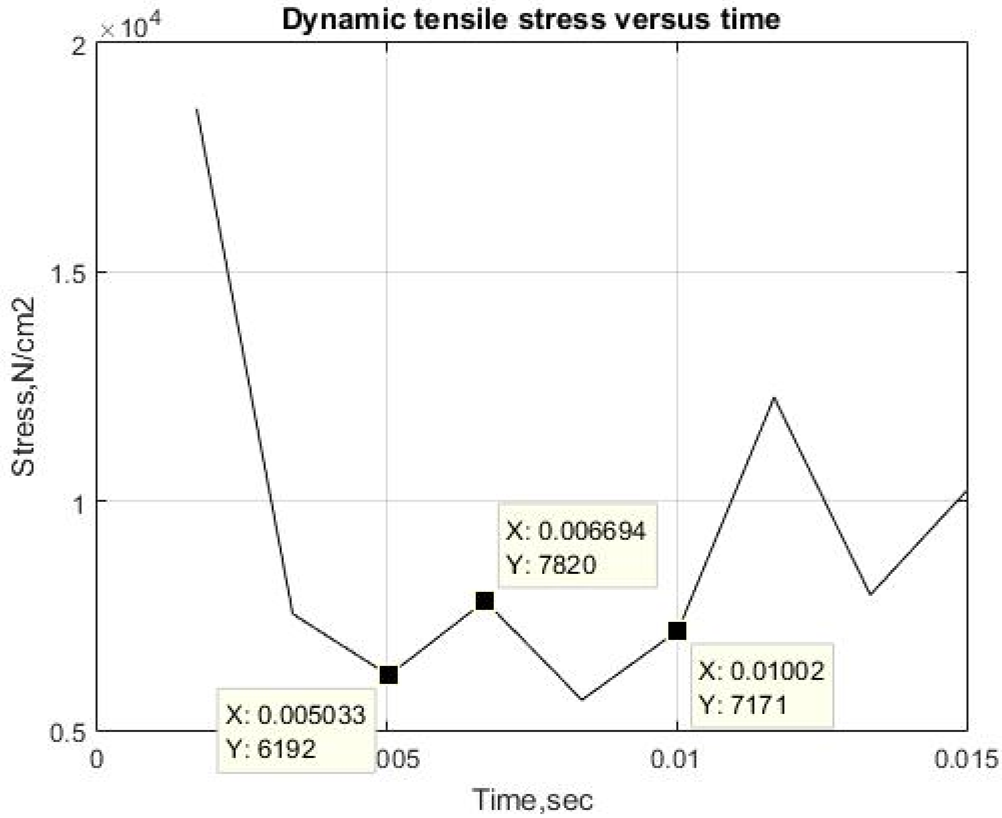
In this section ANSYS result for dynamic tensile stress analysis will be put for actual tool design.

Time(sec);

0.001711,0.003372,0.005033,0.006694,0.008356,0.010017,0.011678,0.0133339,0.015

Stress(N/cm^2)

18569.4, 7534.02, 6191.53, 7820.08, 5653.47, 7170.85, 12263.6, 7951.56, 10221.9



4.2A.The Effect of Dynamic behavior at different tool section

Case I – For the actual tool investigated on this thesis

To study the dynamic behavior the actual tool design simulated using ANSYS.As per the above graph there are three critical times at which the tensile stress has maximum value according to impact theory.

At critical time of 0.005033sec the maximum tensile stress is generated at tool section below the face of front head of hammer housing (see table 3.9).This maximum tensile stress developed at distance measured 57.6cm from bottom tip of the tool. The other critical time was 0.006694sec,here the maximum tensile stress developed at a distance of 76.4cm from bottom tip of the tool(see table 3.9).But at critical time of 0.010017sec the maximum tensile stress observed at a distance of 53.1cm from bottom tip of the tool. When the result compared with the manufacturer recommendation of fatigue area location between 57.5cm and 77.5cm, the critical times, 0.005033sec and 0.006694sec satisfy the manufacturer recommendation .But at critical time 0.010017sec the distance of 53.1cm does not much the manufacturer recommendation. Therefore at critical time of 0.010017sec, fatigue failure may not be the problem for the actual tool design in this thesis.

Case II – For the recommended tool insert tip diameters

In order to see the effect of insert tip diameters on dynamic behavior of the tool ANSYS simulation was done by tool insert tip diameters other than the actual design of tool tip diameter of 2.0cm.

At critical time of 0.005033sec the maximum tensile stress developed at distance measured 64.7cm from bottom tool edge(tip) for recommended tool tip insert diameters of 1.6cm,2.4cm,and 3.0cm.For tool tip insert diameter of 2.2cm the maximum tensile stress developed at 57.6cm from bottom of tool tip, which correspond the actual design case of the tool investigated in this thesis(see table 3.9).All the proposed tip diameters satisfy the manufacturer fatigue area location which specify a minimum distance of 57.5cm from bottom tip of the tool.

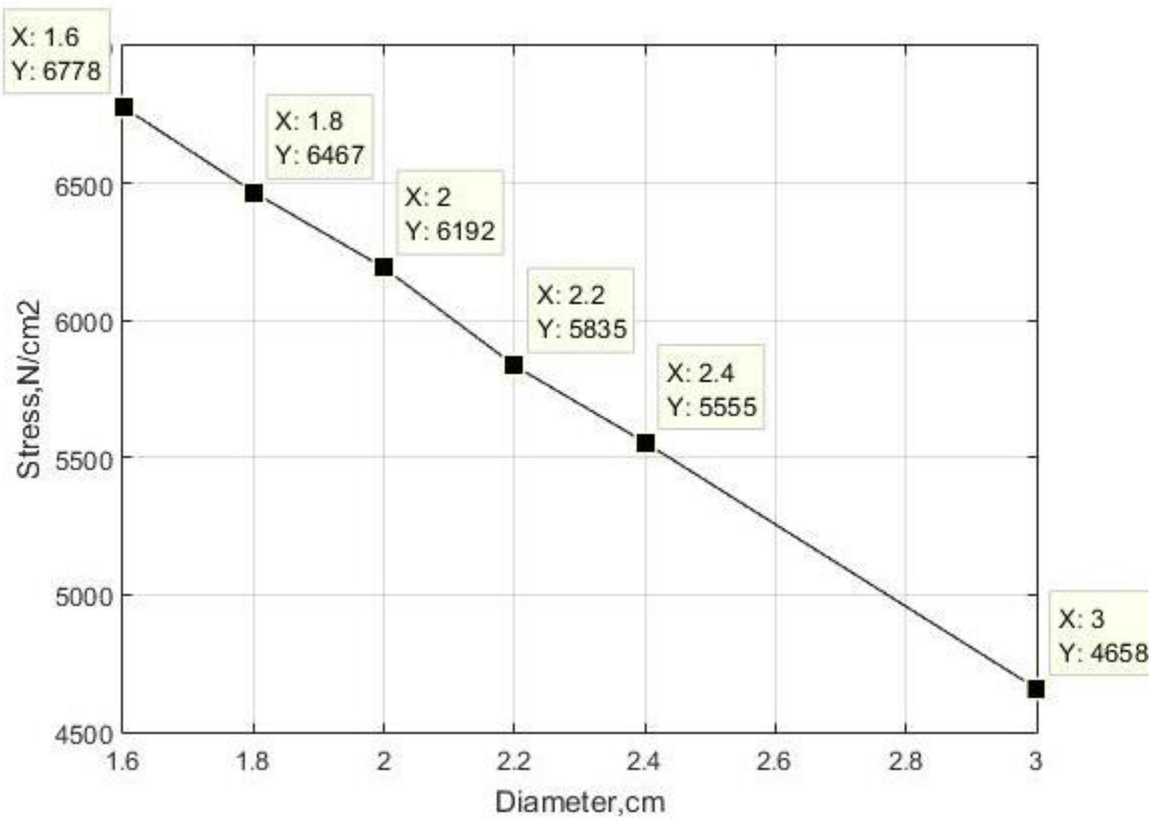
At critical time 0.006694sec all the recommended insert tip diameters have the same maximum tensile stress area location to the actual tool design which is 76.4cm.It also satisfy the manufacturer recommended fatigue area location specification with maximum distance 77.5cm from bottom of tool tip.

At critical time 0.010017sec except the proposed tool tip diameter of 2.2cm the others (1.6cm,1.8cm & 2.4cm) the maximum tensile stress lies at distance of 63.4cm which also satisfy manufacturer recommendation. A critical time 0.010017sec may not have a fatigue problem in case of 2.2cm recommended tip diameter like the actual design case. But in case of 3.0cm no specific maximum tensile stress area is observed below the face of front head of hydraulic hammer face.

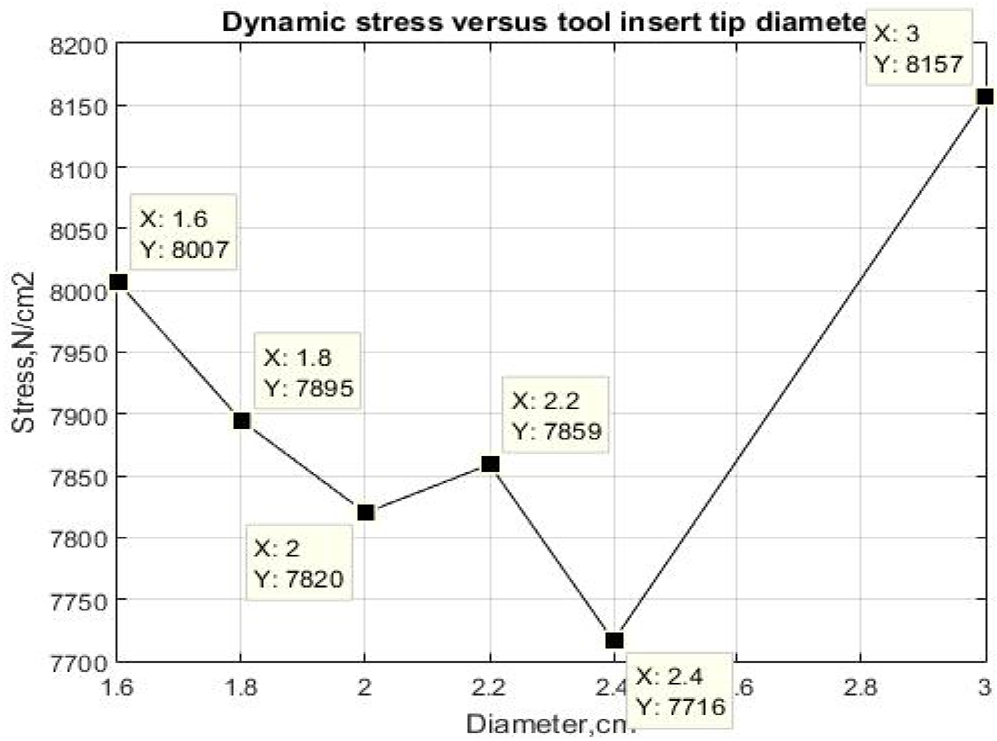
4.1B.Maximum tensile stress against tool tip diameters.

In this section the value of maximum tensile stress compared for designed and proposed tool insert tip diameter.

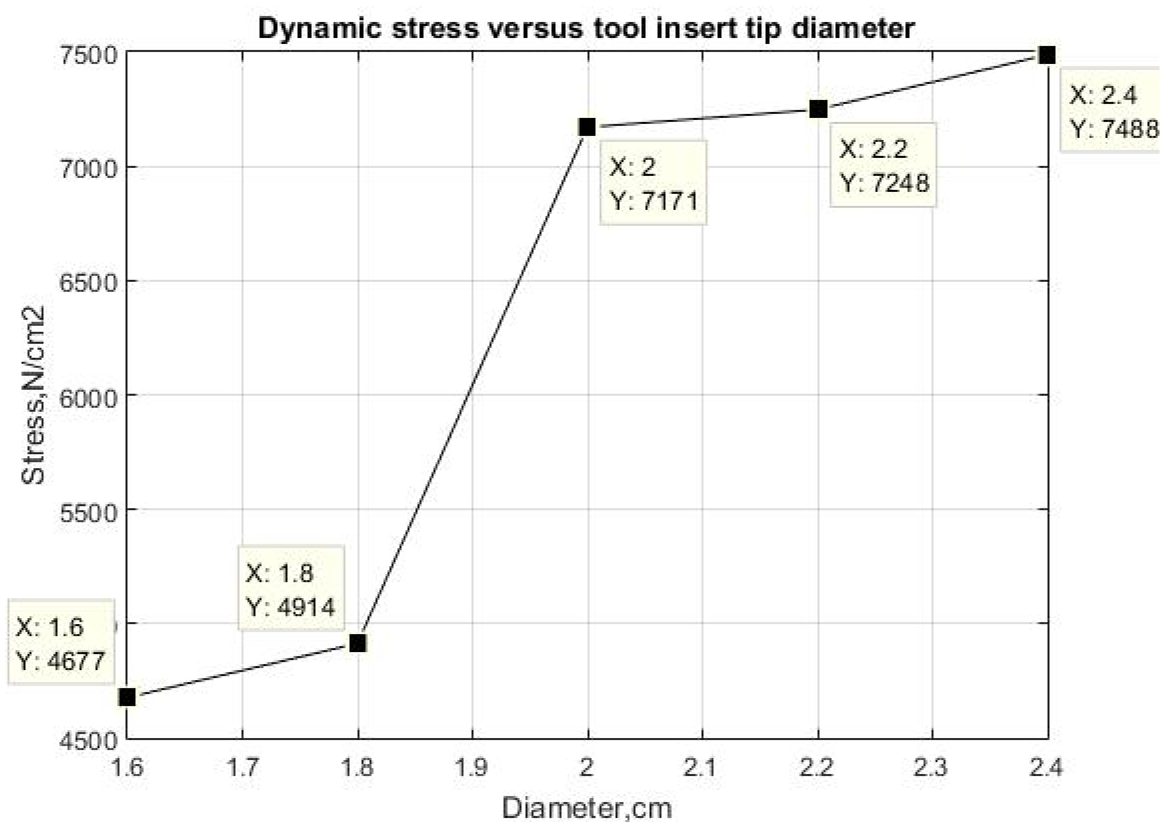
I. For Critical time 0.005033sec



II. For Critical time 0.006694sec



III. For Critical time 0.010017sec



4.2B.The Effect of tip insert diameters on tensile stresses

As discussed above ,for the actual and recommended tool insert design, the critical time 0.005033sec shows the maximum stress area to be developed below the face of front head of hammer housing(or on working length of the tool).

At this critical time i.e 0.005033sec in case of actual tool design a maximum tensile stress of 6192N/cm^2 is developed at specified location.

From the above graph it can conclude that at critical time of 0.005033sec the maximum tensile stress developed on tool decreases as the tool insert tip diameter increases from 1.6cm to 3cm.Compared to the critical time of 0.010017sec the maximum tensile area location of the critical time 0.005033sec is far below the front head of hammer housing(see table 3.9).There for we can conclude that, in order to reduce the maximum tensile stress as we move far below the front head of hammer housing, designing the tool with large insert tip diameter(3.0cm) has an advantage even compared to the actual design case.

At critical time 0.006694sec, the maximum tensile stress area to be developed above the face of front head of hammer housing (which is in the area of lower bushing).

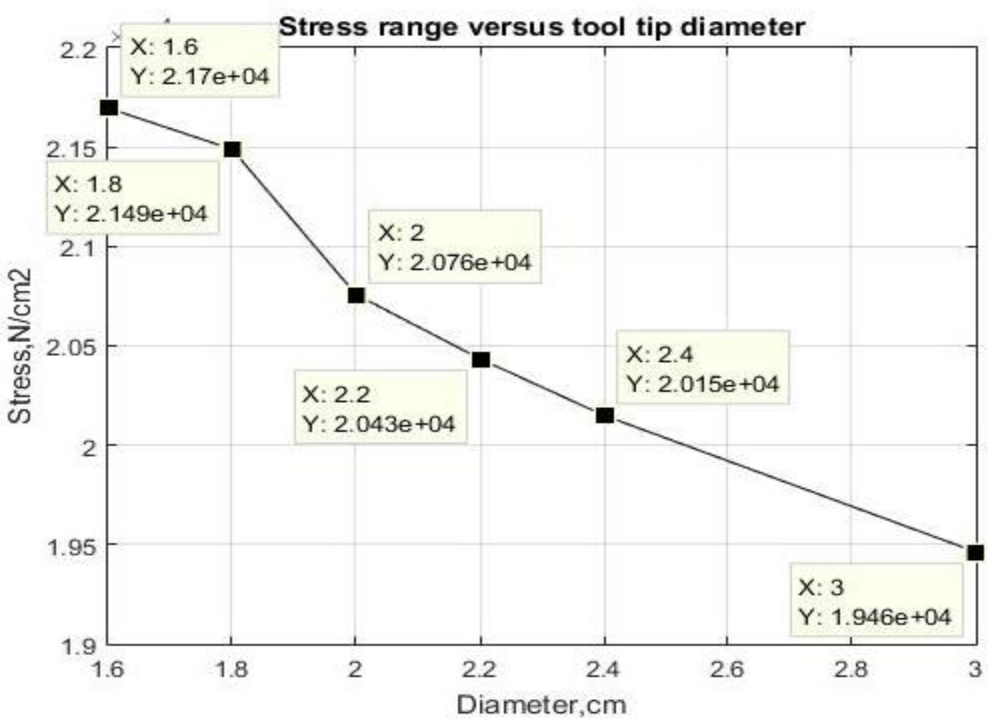
For the actual design case at critical time of 0.006694sec a maximum tensile stress of 7820N/cm^2 developed above front face of hammer housing(at lower bushing area).Generally as tool insert tip diameter increases from 1.6cm to 2.0cm(design case) the maximum tensile stress developed decrease in value. The maximum tensile stress becomes 7859 N/cm^2 for proposed 2.2cm tip diameter. The minimum tensile stress at this critical time is 7716 N/cm^2 which correspond to insert tip diameter of 2.4cm.Tha maximum tensile stress at this critical time is 8157 N/cm^2 correspond to 3.0cm tip diameter. Therefore in order to achieve a minimum tensile stress on tool bushing area it will be better to design the tool with insert tip diameter of 2.4cm.

At critical time of 0.010017sec the maximum tensile stress developed is 7171 N/cm^2 for the case of actual tool design. This critical times also shows the maximum tensile stress area below the front head of hammer housing (on working length of the tool)

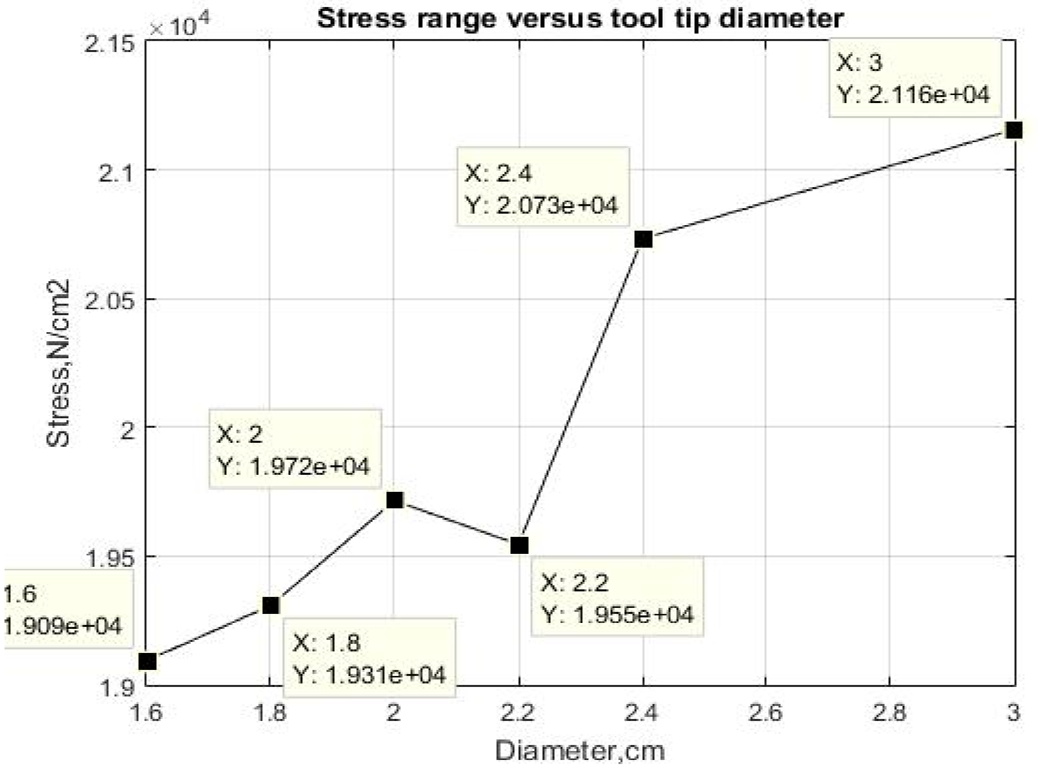
Here also compared to critical time of 0.005033sec,the critical time 0.010017sec shows relative maximum tensile stress area near to front head of hammer housing. From the above graph as the diameter of tool insert tip increases from 1.6cm to 2.4cm the maximum tensile stress also increases in magnitude. There for it can conclude that to reduce the maximum tensile stress as we move near to front head of housing it is better to design the tool with small diameter (1.6cm) tool insert tip.

4.1C.Range of stress($\Delta\sigma$) against tool tip diameter.

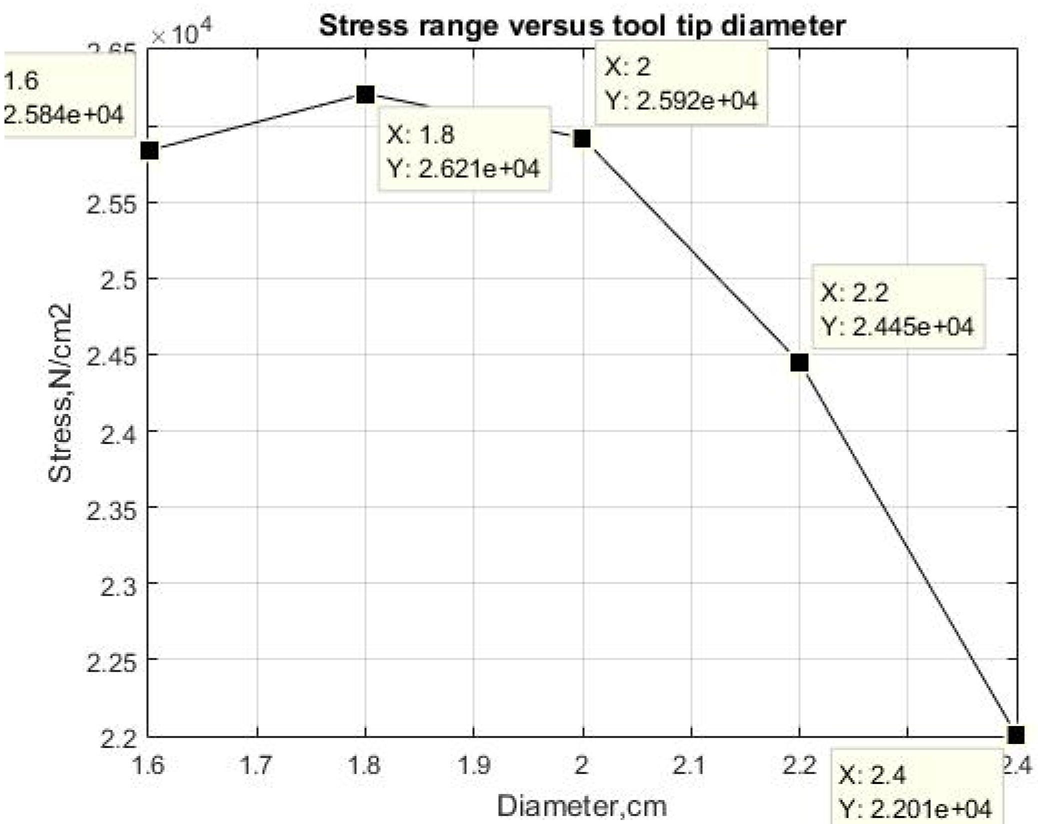
I. For critical time of 0.005033sec



II. For critical time of 0.006694sec



III. For critical time of 0.010017sec



4.2C.The Effect of tool tip diameter on Range of stress

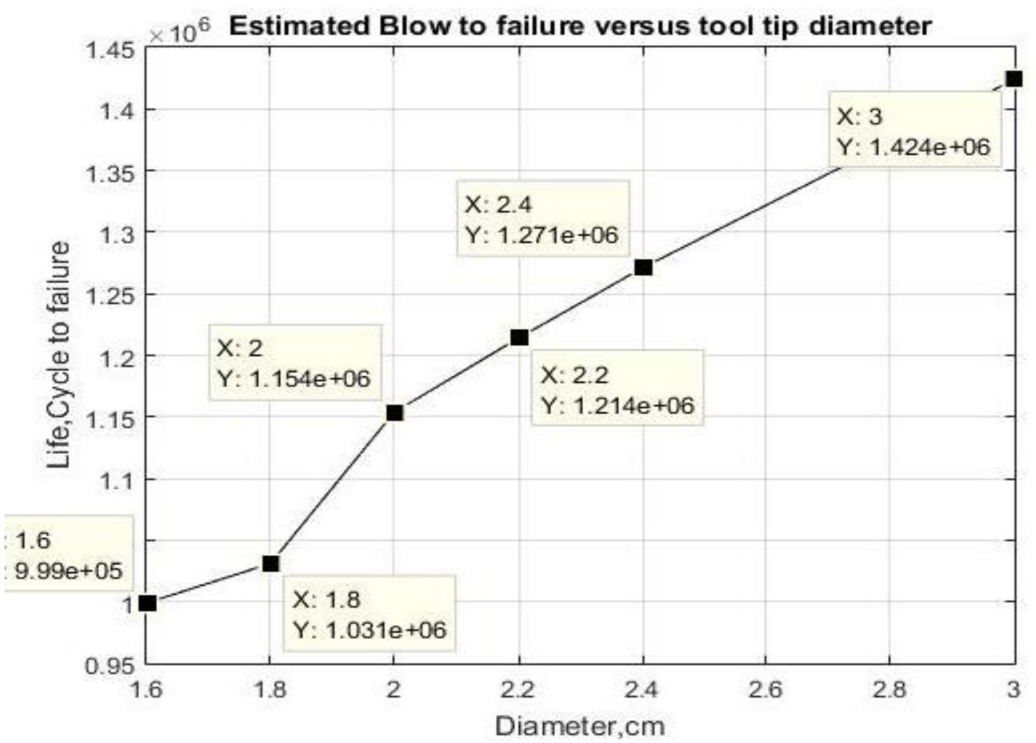
The range of stress for critical time of 0.005033sec i.e when the maximum tensile stress is below the face of front head of hammer housing .Generally at this critical time it is maximum for tool tip diameter of 1.6cm and it's value decreases with increasing the tip diameter. The range of stress shows the same train with the graph of maximum tensile stress graph. The actual tool design has a stress range value of 20760N/cm^2.

At critical time of 0.006694sec i.e when the maximum tensile stress is inside lower bushing area. The range of stress increase when the tool tip insert diameter increases from 1.6cm to 2cm(design case) and then decrease until 2.2cm tool insert tip diameter. The range of stress has maximum value for tool insert tip diameter of 3cm.

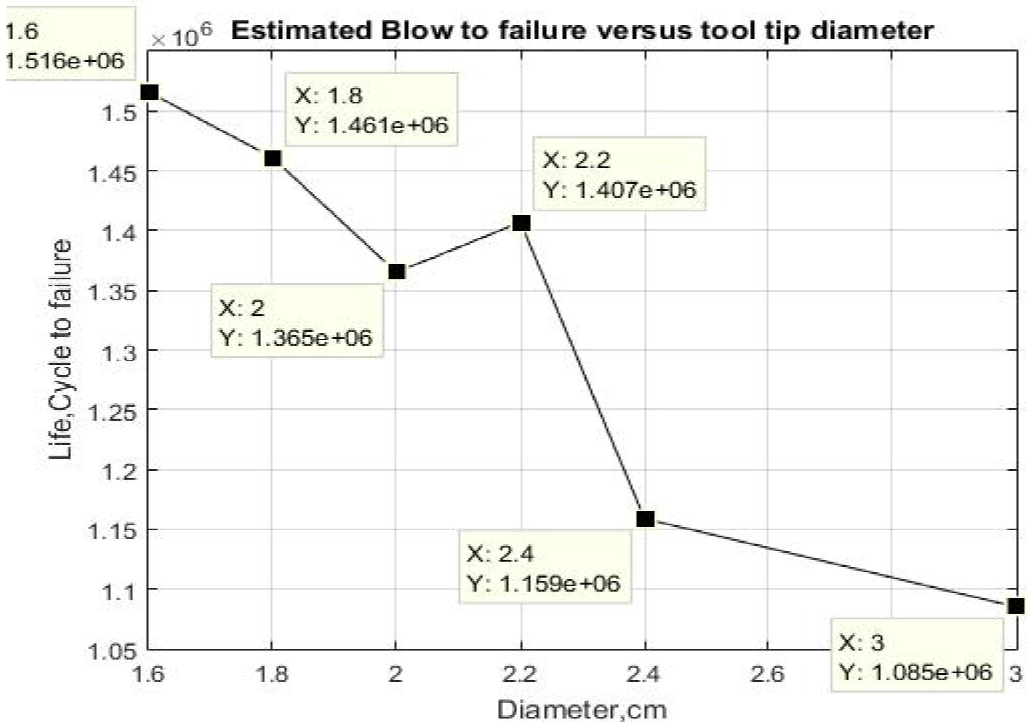
At critical time of 0.010017sec when the maximum stress is observed below the face of front head of hammer housing. The range of stress has maximum value at tool tip diameter of 1.8cm and decrease in value with increasing tool tip diameter.

4.1D.Estimated Cycle/Blow to failure against tool tip diameter.

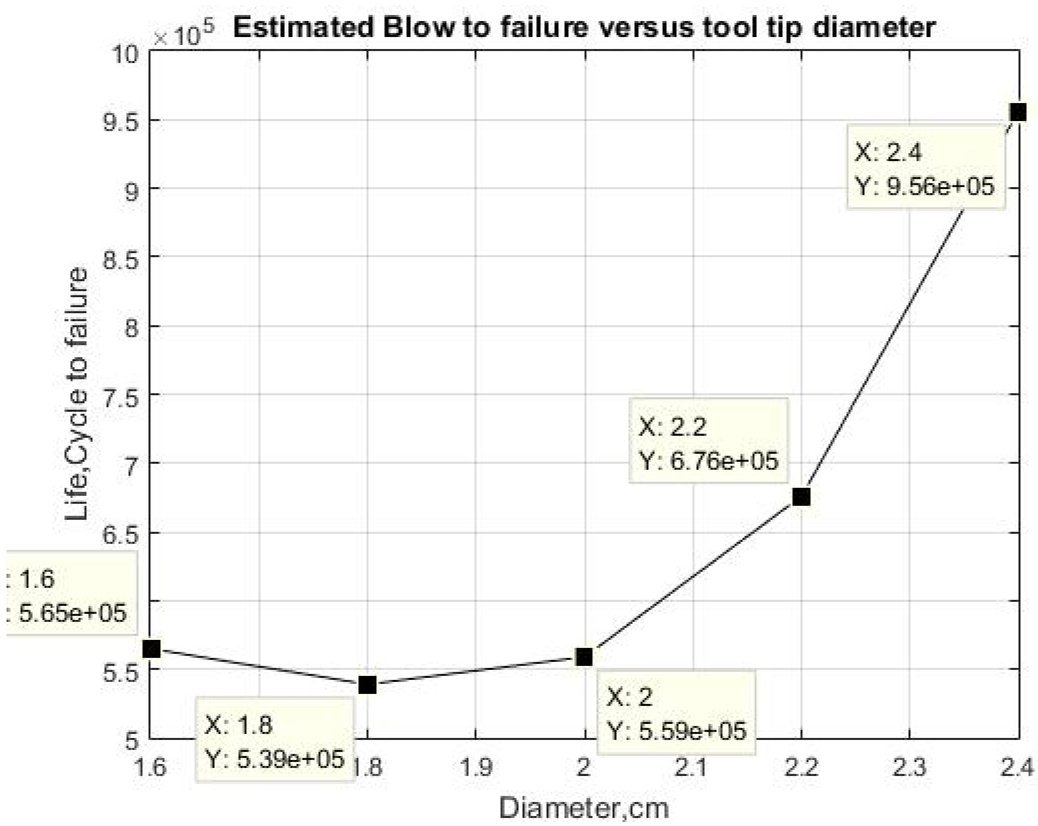
I. For critical time of 0.005033sec



II. For critical time of 0.006694sec



III. For critical time of 0.010017sec



4.2D.The Effect of tool tip diameter on tool failure

For critical time of 0.005033sec i.e when the expected tool failure area will happen below the front head of hammer housing .In case of actual tool design the estimated number of cycle/blow to failure below the front head is 11.54 $x10^5$ **.**In case of the minimum recommended diameter of 1.6cm the estimated number of cycle to failure is $9.99x10^5$, so that the tool will fail early compared to other tool tip diameter. *Generally the estimated number of cycle/Blow to failure increases with increasing tool tip diameter* .For example when tool tip insert diameter designed with 3cm ,the estimated number of cycle/Blow to failure becomes $14.24x10^5$ .Therefore the tool will not fail early in a position far below the face of front head if the tool is designed with the recommended diameter of 3cm.

For critical time of 0.006694sec i.e when the expected failure area is above front head of hammer housing or within lower bushing area. For the actual tool investigated in this thesis the estimated number of cycle/blow to failure is 13.65 x 10^5 .The result of this thesis shows that as tool tip diameter increases from 1.6cm to 3.0cm the number of cycle/Blow to failure decreases .If the tool were designed with the recommended tip diameter of 1.6cm the maximum estimated number of cycle/Blow to failure is obtained, therefore after 15.16 $x10^5$ number of cycle/blow the tool will fail after the initial detectable crack size is measured during inspection. Generally to avoid failure in lower bushing area the tool shall be designed with smallest recommended tip diameter(1.6cm).

At critical time of 0.010017sec when the tool failure is expected relatively near to front face of hammer housing compared to the critical time 0.005033sec the number of cycle/blow to failure has minimum value compared to the other case. Generally at this critical time the number of cycle/blow to failure increase with increasing tool tip diameter. Compared to other cases the tool will fail early in this region.

Chapter Five

Conclusion, Recommendation, Future Work

5.1. Conclusion

In this thesis work, ANSYS software programming were develop to study the dynamic behavior of hydraulic hammer tool to identify the possible area of failure .ANSYS were the main tool to locate the maximum dynamic tensile stress area based on the actual time history loads graph for hydraulic hammer tested on the field. After simulation with calculated impact and thrust force the manufacturer recommendation of fatigue failure area were compared to the analysis result.

Generally from this thesis work the following conclusion can be figured out;

- ✓ Actual material property of both tool and rock are the main factor for this thesis end result.
- ✓ Parameters of contact such as stiffness and coefficient of friction are the main factor for the dynamic analysis.
- ✓ The maximum dynamic tensile stresses on the tool is due to a relative stiffness difference between tool shank and insert part.
- ✓ The average maximum stress and the range of stress at a point is the cause of fatigue failure propagation on the tool.
- ✓ Thrust force is one of the most important parameter during hydraulic hammer tool design.
- ✓ During hydraulic tool design the rock property should be considered.
- ✓ Maximum tensile stresses were located at distance between 57.6 to 76.4cm when measured from bottom tip of the tool.
- ✓ The critical simulation time to locate maximum tensile stress should be taken only when minimum compression stress is obtained at lower tip of the tool.
- ✓ Tool deformations were below 0.09262cm during impact.
- ✓ Tool insert diameter is one of the factors for tool failure.
- ✓ Crack initiation in the region between 57.6cm and 76.4cm is the main cause for fatigue failure.

5.2. Recommendation

1.During hydraulic hammer tool design modeling of the tool part only will not give reliable result. So that the designer should anticipate all design parameters such as the corresponding rock mechanical property especially the stiffness of the rock. The working condition also affect the tool design.. This thesis clearly shows that the sevier working condition of breaking bed rock were modeled by applying the confinement pressure to be applied in all direction of rock model, which resulted in reliable dynamic stress simulation on tool part. Considering the damping effect like friction also leads to ultimate good result of tool design.

2.Maximum tensile stress on tool shank were taken when there is minimum dynamic compression stress developed at tool tip. The thesis result clearly shows that all dynamic tensile stress is transferred to the tool steel shank part. The tool animation shows that always maximum stress first generated at the contact area between tool and rock. So that the tip should be hard material compared to the tool shank part. Tool shank is relatively low in stiffness therefore this maximum stress developed at the contact area immediately goes to the relatively lesser stiffness part .

3.During analysis the dynamic behavior of the tool were simulated with out the thrust force. The simulation of this condition shows that the maximum tensile stress were observed above the middle and below the top edge of the tool part. Therefore this result will misled the designer. So always thrust force should be considered during hydraulic tool design. The impact theory which proposes to neglect the weight effect during impact is not valid here.

4. Generally from actual tool design investigated and the recommended tool insert diameters. It can conclude that in order to avoid early tool failure in lower bushing area the tool should be designed with relatively smaller tool tip insert diameter (ex. 1.6cm).This is mainly related to a relative minimum magnitude of the range of stress and the average maximum stress at a point at maximum tensile stress area. Designing the tool with 1.6cm insert

diameter the tool life will increase with 10% that the actual tool design(2.0cm).

At the same time if the expected failure area is on the working length (below the front face of hammer housing)to achieve a better life of the tool should be designed with relatively large tip diameter(ex. 3cm).Here also with a relative large tip radius there is relative minimum average maximum stress and minimum range of stress at maximum tensile stress area. Designing the tool with 3.0cm insert diameter the tool life will increase with 19% that the actual tool design(2.0cm).

5.According to the strain value found from the literature the expected deformation of the tool calculated as 0.09262cm.But the ANSYS result of the tool deformation during impact were found to be 0.022518cm, 0.026134cm,0.028137cm at critical times of 0.005033sec, 0.006694sec, 0.010017sec respectively. This shows the total simulation time of 0.015sec taken for tool rock impact analysis is the optimum value for all ANSYS result found in this thesis analysis.

6.Avoiding crack initiation and propagation in the region between 57.6cm to 76.4cm will minimize the fatigue failure. This is done by;

6.1. Avoid tool straightening processes.

- During tool manufacturing there is occasional straightening. During heat treatment processes the tool may not support with appropriate fixture or support. This conditions prone the tool to bend. During straightening micro crack may be generated at the surface of the tool. Therefore the use of proper support or fixture is mandatory to avoid bending of the tool.

6.2. Quality control on original material of tool steel shank

- The source of material imperfection may be found within original material supply. A quality control of original material supplied is mandatory. The best surface testing is the use of either liquid penetration or Die penetrates methods. Both methods indicate the flaw found on surface regardless of size, configuration, internal structure, chemical composition and flaw orientation.

6.3. Increasing of the fracture toughness of the tool Material

- Increasing the fracture toughness value of the tool material has an advantage for the tool not to fail due to further progress in crack length. Example if fracture toughness value of the tool material increased to $110MPa\sqrt{m}$ at the failure at the working length(below the front face of hammer)the cycle to failure will increase from 14.24×10^5 to 14.30×10^5.

6.4.Maintenance Aspect

- As far as possible the hydraulic hammer bushings should be greased as per the specification on manufacturer operation manual based on the recommended interval. With improper type of lubricant seizing commonly occur between tool surface and bushing. This is the main cause for crack initiation on surface of the tool.

5.3. Future Work

The theses try to investigate dynamic behavior of the tool and the possible area it's failure. Therefore further studied should be conducted on;

1. Experimental failure analysis of the hydraulic hammer tool.
2. The effect of varying the thrust force on hydraulic hammer tool failure under dynamic load condition
3. The effect of heat treatment on hydraulic hammer tool failure

Reference

1. **Paolo Balossi**.:"Experimental and Numerical Simulation of penetration on sand stone ". Research project for Master of Science Degree in Mechanical Engineering,2016/2017,27-46.
2. **Lucio Marsriera, Marcos Fernadez, Juan Marani, Gianfranco Antonel, Claudio Acosta, Jorge Halabi**.:"Scientific Method Applied to Failure Analysis on Engineering Components". Research project School of Engineering, Procedia Material Science 8(2015) ,117-127.
3. **Norman A.Waterman and Michael F.Ashby.:**"THE MATERIALS SELECTOR SECOND EDITION"CHAPMAN & HALL, Volume 2, 1997,pp 660-666.
4. **Rasoul Moradi`, Michael L.Mc Coy and Humid M.Lankarani** "Impact Analysis of Mechanical Systems Using Wave Propagation Methodology" Academy Publish. Org., 211-249.
5. **Dr.Wenyi Yan and Pr. Thanh Tran - Cong** "Investigation of a hydraulic impact a technology in rock breaking". A research paper, Department of Mechanical Engineering, VIC800, p.20.
6. **Mr.V.Sivananth, Dr.S.Vijayarangan , Mr.R.Aswathama.:** "Fatigue and Impact Analysis of Automotive steering Knuckle under operating Load Cases". Altair Technology Conference, Engineering and Technology, Pollachi-642003,2015,P 1.
7. **Changheon Song[1,2], Dae Jikim[2], Jintai Chung[1] ,Kang Won Lee[2],Sang Seuk Kweon[2],and Young Ky Kang[3].:** "Estimation of Impact load in a hydraulic Breaker by Transfer path analysis" Research article, Volume2017,Article ID 8564381,15pages.

8. **Gang Han, Mike Bruno, Maurice B.Dusseault.:**"Dynamic Modelling Rock failure in percussion Drilling "Research paper,ARMA/USRMS 05-819, pp 1-7.
9. **Ming Jian.:** "Theoretical and experimental studies of ITH percussive drill vibration". A thesis submitted to the Faculty of Graduate Studies and Research in Partial fulfillment of the requirements for Degree of Master of Engineering, Department of Mining and Metallurgical Engineering, March 1994, p.13.
10. **M.Ahmed[1],K.A.Ismail[2] and F.Mat[1].:**"Impact models and coefficient of restitution; a review" Journal of Engineering and Applied science,ISSN 1819-6608,Vol.11,No.10,May 2016,pp 6549-6555.
11. **Sazidy, M. S., Rideout , D. G., Butt, S. D. and Arvani,F.:**"Modeling percussive drilling performance using simulated visco-elasto-plastic rock medium" Research paper,ARMA 10-434,pp1-9.
12. **Atlascopco, Working Tools for Hydraulic Breakers.:**"Black line". Working Tools for Hydraulic Breakers , Atlascopco, p 14-15
13. **RBI HYDRAULIC BREAKER MANUAL.:** "RBI Hydraulic Breaker owner's Manual" Rock Breakers, Inc. www.Rbibreakers.com. P. 65.
14. **Caterpillar Hammer Service Presentation.** .CATERPILLAR ,Global Work Tools and service, for H110E-130E,2011.
15. **Joseph Edward Shigley.:**"Mechanical Engineering Design" .Tenth Edition in SI unit, 2016, p 274-290.
16. **Norman E.Dowling .:**"Mechanical Behavior of Materials". International Edition, Fourth Edition, 2013 ,p 348-563.
17. **S.Targ.:**"Theoretical Mechanics". A short course, MIR PUBLISHER, Moscow,1976, p 454-468.
18. **A.A.ILYUSHIN and V.S.LENSKY.:** "Strength of Materials",Moscow,1967.p327-364.
19. **P.C.SHARMA,D.K.AGGARWAL.:**"Machine Design"New Delhi,p124-125
20. **Paleti Srnivas, Krishna Chaitanya Sambana, Rajesh Kumar Datti.:**"Finite Element Analysis using ANSYS 11.0".New Delhi-110001,2012,p 247-249.
21. **J.A.WILLIAMS.:**"Engineering Tribology" OXFORD UNIVERSTY PRESS,1994, P73-164.
22. **R.B.Choudary.:**"Introduction to ANSYS 10.0".New Delhi, 2011,p 3-23.
23. **Erdogan Madenci ,Ibrahim Guven.:**"The Finite Element Method and Applications in Engineering using ANSYS". Printed in United State of America,2006, p 459-471.

24.N.G.Sereda, E.M.Solovyov. :"Drilling of Oil and Gas Wells". MIR PUBLISHER, Moscow,1977.p 24- 203.
25.D.F. Coates.:"Rock Mechanics Principle". Mining Research Center,1972, p 46-50.
26.ANSYS training manual.:"Transient Dynamic analysis" Module 4,Inventory #001809,March 14,2003,p 1-58.

Annex-I

SUGGESTED CHECK LIST OF DATA DESIRABLE FOR FAILURE ANALYSIS

1. Description of component size , shape ,and use;
   Size: - ***Length=110cm, Diameter=130mm***
   Shape: - ***Most part cylindrical***
   Use:-***To Break the rock***
2. Specify areas of design stress concentrations:- ***At mid section of the tool***
3. Magnitude of stress concentration at failure site:-***It Vary depending on location***
4. Type of stress (e.g. Mode I, II , III or combinations):- ***Mode I (See Figure 1.1)***
5. State of stress; Plane strain vs. Plane stress:- ***Plain strain***
   Fracture surface appearance (percent shear lip):-***No shear lip.***
6. Type of Loading Pattern
   Axial impact
   Thrust force at initial and between impact.
7. Details of critical flaw
   Surface or Imbedded Flaw :- ***Surface flaw (See Figure 3.5)***
   Direction of crack propagation as determined by ;
   Chevron marks :-***Curves in toward tool fracture surface and point back to crack origin(See Figure 3.5).***
   Manufacturer flaws related to crack initiation
   Scratches:- ***Expected due to turning and milling machining operation. (See Fig. 3.4)***
   Metallurgical flaws related to crack initiation.
   Expected due to poor heat treatment processes.
8. Component material specifications
   Mechanical Properties

	E	ν	σ_{ys}	K_{IC}

Specified	***180GPa***	***0.31***	***1250MPa***	***93*** $MPa\sqrt{m}$

Melting Practice;

Not specified

Ingot breakdown;

Cold rolled

Component Manufacture;

Machined

Surface Treatment;

Shot peened

Cold Rolled

Carburized

Annex II

Tool Failure Data

1. Distributer: - ***CATERPILLAR***

2. Hammer Model :- ***H130ES***

3. Type of Tool: - ***Moil***

4. Tool Part No :- ***3Q -7638***

5. Tool Batch No :- ***355-5876***

6. Date of Installation: - ***28/02/2013***

7. Date of Failure: - ***01/06/2013***

8. Working hour :- ***720hr (Even, < 720hr depend on application)***

9. Application: - ***To break bed rock.***

10. Type of Material: - ***Limestone***

11. Carrier Make & Model: - ***Make; CATERPILLAR***

    Model; 324DL Hydraulic Excavator

12. Failure: - ***Suddenly Broken***

13. Overall length:- ***110cm***

14. Length before failure (L1) :- ***110cm***

15. Length of Upper part (L2) :- ***Not fixed in length but happen in lower bushing/and at working length.***

16. Fracture Type: - ***Fatigue***

17.Max. Diameter :- ***130mm***

18. Min. Diameter: - ***130mm***

19. Type of Lubricant: - ***Hammer base grease***

20. Grease dispenser type: - ***Manual***

Annex-III

Specification and system operation

Design specification of model H130ES hydraulic Jack Hammer

1	Blow/Impact Energy(E)	4745Joul
2	Working weight	1700 – 1890Kg
3	Mass of piston(m_p)	80Kg
4	Striker(Piston) speed	10.89m/s
5	Hammer operating Pressure(P_H)	15000kpa / 2175psi
6	Area of piston section(A_{ps})	$2486 \times 10^{-6} m^2$
7	Acceptable oil flow	120 – 220 l/min

Firing Cycle

- **Two forces acting on piston**
- **Piston moves down with F2**

F2 = 8 x F1

F = Pressure x Area

Area: A2 > 8 x A1