SKILL ADVADVANCED COURSE-II REPORT
(PRACTICAL FINITE ELEMENT ANALYSIS USING HYPERMESH AND LS-DYNA)

A Skill Advanced Course-II
(STRUCTURAL ANALYSIS ON COMPONENTS OF RADIAL ENGINE USING HYPERMESH & LS-DYNA)
Report submitted to
JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY, KAKINADA
In partial fulfilment for the award of the Degree of

BACHELOR OF TECHNOLOGY
IN
MECHANICAL ENGINEERING
By
DOKKU SIRI VENKATA NAGA GOPI
(20761A0358)

Under the esteemed guidance of
DR. CH. SIVA SANKARA BABU

DEPARTMENT OF MECHANICAL ENGINEERING

LAKIREDDY BALI REDDY COLLEGE OF ENGINEERING (AUTONOMOUS)
(Approved by AICTE, Affiliated to JNTUK, KAKINADA, Accredited by NBA (Tier-1), NAAC (A Grade) and an ISO 9001-2015 certified Institution)
L. B. REDDY NAGAR, MYLAVARAM, KRISHNA DIST –521230
ANDHRA PRADESH

LAKIREDDY BALI REDDY COLLEGE OF ENGINEERING
DEPARTMENT OF MECHANICAL ENGINEERING

CERTIFICATE

This is to certify that the Skill advanced course (PRACTICAL FINITE ELEMENT ANALYSIS USING HYPERMESH AND LS-DYNA) report entitled **"STRUCTURAL ANALYSIS ON COMPONENTS OF RADIAL ENGINE USING HYPERMESH & LS-DYNA"** at **"MayinKrish Ventures Pvt, Ltd."** that is being submitted by **DOKKU SIRI VENKATA NAGA GOPI** bearing **20761A0358** in partial fulfilment for the award of the degree of Bachelor of Technology in Mechanical Engineering is a record of the summer internship work carried out by him under our guidance and supervision.

Staff In-charge
Dr. Ch. Siva Sankara Babu

Head of the Department
Dr. M.B.S. Sreekara Reddy

Internal Examiner

External Examiner

ACKNOWLEDGEMENT

It was a great pleasure to undertake the two-week skill advanced course - **PRACTICAL FINITE ELEMENT ANALYSIS USING HYPERMESH AND LS-DYNA.** It was a great learning Experience through-out the program. I would like to express my special thanks of gratitude to my project mentor **Mr. BRMV Krishna** from Hyderabad, Telangana whose valuable Suggestions, worthy experience and encouragement helped me to draft this Report.

I humbly express my thanks to our management and Principal **Dr. K. Appa Rao** for extending his support for providing us with an environment to complete our Skill advanced course successfully.

I am indebted to our Head of the Department **Dr. M.B.S. Sreekara Reddy** who modeled us both technically and morally for achieving greater success in life.

Also, I would like to thank the **Dr. Ch. Siva Sankara Babu**, Sr. Assistant Professor of Lakireddy Bali Reddy college of Engineering, who gave me this wonderful opportunity at this program.

Without them and their consistent guidance and motivation, this project would Not have been completed.

I am thankful to my friends who helped me sharing knowledge and by providing material to complete the Skill Advanced Course in time.

DOKKU SIRI VENKATA NAGA GOPI.
(20761A0358).

DECLARATION

I am here by declaring that the project entitled "**STRUCTURAL ANALYSIS ON COMPONENTS OF RADIAL ENGINE USING HYPERMESH & LS-DYNA**" work done by me. I certify that the work contained in the report is original and has been done by me under the guidance of my supervisor. The work has not been submitted to any other institute in preparing for any degree or diploma. I have followed the guidelines provided by the institute in preparing the report. I have conformed to the norms and guidelines given in the Ethical Code of Conduct of the institute. Whenever I have used materials (data, theoretical analysis, figures and text) from other sources, we have given due credit to them by citing them in the text of the report and giving their details in the references. Further, we have taken permission from the copyright's owner of the sources, whenever necessary.

Signature of the student

ABSTRACT

Abstract:

The project titled "Structural Analysis on Components of Radial Engine Using HyperMesh & LS-DYNA" focuses on conducting a comprehensive structural analysis of key components within a radial engine, including the piston, slave rod, master rod, and crankshaft assembly. This endeavor employs advanced engineering tools, HyperMesh and LS-DYNA, to simulate and evaluate the performance and integrity of these crucial engine parts. By subjecting them to rigorous computational testing and stress analysis, this project aims to ensure the durability, safety, and optimal functioning of radial engine components, contributing to the advancement of aviation and mechanical engineering industries.

Keywords:

- Structural Analysis
- HyperMesh
- LS-DYNA
- Radial Engine Components
- Piston
- Slave Rod
- Master Rod
- Crankshaft

SKILL ADVANCED COURSE CERTIFICATE:

MAYINKRISH VENTURES PVT. LTD.

A registered MSME firm

www.mayinkrish.com Nafeez residency, Begumpet, Hyderabad - 500016

Certificate of completion

This is to certify that

Mr./Ms. DOKKU SIRI VENKATA NAGA GOPI (20761A0358) He/she has successfully completed a skill-oriented program on Practical finite element analysis using HyperMesh & LS-DYNA during 07th Sep - 2023 to 16th Sep - 2023

Organised by Department of Mechanical Engineering, Lakireddy Bali Reddy College of Engineering, Mylavaram, A.P.

in association with Mayinkrish Ventures Pvt. Ltd, Hyderabad.

During this period, their dedication and performance were commendable. We found them to be hardworking, sincere, and devoted. We wish them all success in their future Endeavour.

Managing Coordinator

Project InCharge

Dr. S. Pichi Reddy
Head of the Department
Mechanical Engineering

CONTENTS

Project Title *i*

Certificate *ii*

Acknowledgement *iii*

Declaration *iv*

Abstract *v*

Skill Advanced Course Certificate *vi*

List of Figures *viii*

1. INTRODUCTION **1**

1.1. Radial Engine 2

1.2. Principle 3

2. DESIGN OF RADIAL ENGINE COMPONENTS USING CATIA **6**

2.1. Components of Radial Engine 6

2.2. Design of Components using CATIA 6

3. INTRODUCTION TO NUMERICAL METHODS **10**

3.1. Types of methods to solve an engineering problem 10

3.2. Phases of numerical methods 11

3.3. HyperMesh 11

3.4. LS-DYNA 12

4. ANALYSIS ON RADIAL ENGINE COMPONENTS **17**

4.1. Types of Analysis 17

4.2. Structural Analysis 17

4.3. Pre-processing 18

4.4. Solving 25

4.5. Post-Processing 25

4.6. Results 29

5. CONCLUSION & REFEREMCES **32**

LIST OF FIGURES

S.NO.	FIGURES	PAGE NO.
1.	Fig-1: Radial Engine	3
2.	Fig-2: Design of Piston using Commands like Shaft	7
3.	Fig-3: Design of Master Rod using commands like Pad	7
4.	Fig-4: Design of Slave Rod using commands like Pad	8
5.	Fig-5: Design of Crankshaft using commands like Shaft	8
6.	Fig-6: Interface of HyperMesh	18
7.	Fig-7: Deletion of Solids	18
8.	Fig-8: 2D Mesh of Piston Component	19
9.	Fig-9: 3D Mesh of Piston	19
10.	Fig-10: Properties	20
11.	Fig-11: Materials	21
12.	Fig-12: Assigning property and material	21
13.	Fig-13: Creating Nodes	22
14.	Fig-14: Creating Nodes	22
15.	Fig-15: Creating Nodes	23
16.	Fig-16: Overall Node Points	23
17.	Fig-17: Fixing the rigid nodes	24
18.	Fig-18: Applying Load on piston	24
19.	Fig-19: LS-DYNA	25
20.	Fig-20: HyperView Interface	25
21.	Fig-21: Von-mosses stress of Piston	26
22.	Fig-22: 2D Mesh of Slave Rod	26
23.	Fig-23: 3D Mesh of Slave Rod	27
24.	Fig-24: Von Mosses stress of Slave Rod	27
25.	Fig-25: Von Mosses stresses of Master Rod	28
26.	Fig-26: 26: Von Mosses Stress of Crank Shaft	28

CHAPTER-I

1. INTRODUCTION

An engine is a machine designed to convert one form of energy into mechanical energy. Heat engines, like the internal combustion engine, burn a fuel to create heat which is then used to do work.

The word engine derives from Old French engine, from the Latin ingenium–the root of the word ingenious. Preindustrial weapons of war, such as catapults, trebuchets and battering rams, were called siege engines, and knowledge of how to construct them was often treated as a military secret. The word gin, as in cotton gin, is short for engine. Most mechanical devices invented during the industrial revolution were described as engines—the steam engine being a notable example. However, the original steam engines, such as those by Thomas Savory, were not mechanical engines but pumps. In this manner, a fire engine in its original form was merely a water pump, with the engine being transported to the fire by horses. In modern usage, the term engine typically describes devices, like steam engines and internal combustion engines, that burn or otherwise consume fuel to perform mechanical work by exerting a torque or linear force (usually in the form of thrust). Devices converting heat energy into motion are commonly referred to simply as engines. Examples of engines which exert a torque include the familiar automobile gasoline and diesel engines, as well as turbo shafts. Examples of engines which produce thrust include turbofans and rockets.

When the internal combustion engine was invented, the term motor was initially used to distinguish it from the steam engine—which was in wide use at the time, powering locomotives and other vehicles such as steam rollers. The term motor derives from the Latin verb moto which means to set in motion, or maintain motion. Thus, a motor is a device that imparts motion. A heat engine may also serve as a prime mover—a component that transforms the flow or changes in pressure of a fluid into mechanical energy. An automobile powered by an internal combustion engine may make use of various motors and pumps, but ultimately all such devices derive their power from the engine. Another way of looking at it is that a motor receives power from an external source, and then converts it into mechanical energy, while an engine creates power from pressure.

1.1.RADIAL ENGINE:

The Radial Engine as shown in fig 1 is a reciprocating type internal combustion engine configuration in which the cylinders point outward from a central crankshaft like the spokes on a wheel. This type of engine was commonly used in most of the aircrafts before they started using turbine engines. In a Radial Engine, the pistons are connected to the crankshaft with a master-andarticulating-rod assembly. One of the pistons has a master rod with a direct attachment to the crankshaft. The remaining pistons pin their connecting rods` attachments to rings around the edge of the master rod. Four-stroke radials always have an odd number of cylinders per row, so that a consistent every-other-piston firing order can be maintained, providing smooth operation. This is achieved by the engine taking two revolutions of the crankshaft to complete the four strokes. Which means the firing order for a 9-cylinder radial engine is 1,3,5,7,9,2,4,6,8 and then again back to cylinder number 1.

This means that there is always a two-piston gap between the piston on its power stroke and the next piston on fire (the piston on compression). If an even number of cylinders was used the firing order would be something similar to 1,3,5,7,9,2,4,6,8,10 which leaves a three-piston gap between firing pistons on the first crank shaft revolution, and only one piston gap on the second crankshaft revolution. This leads to an uneven firing order within the engine, and is not ideal.

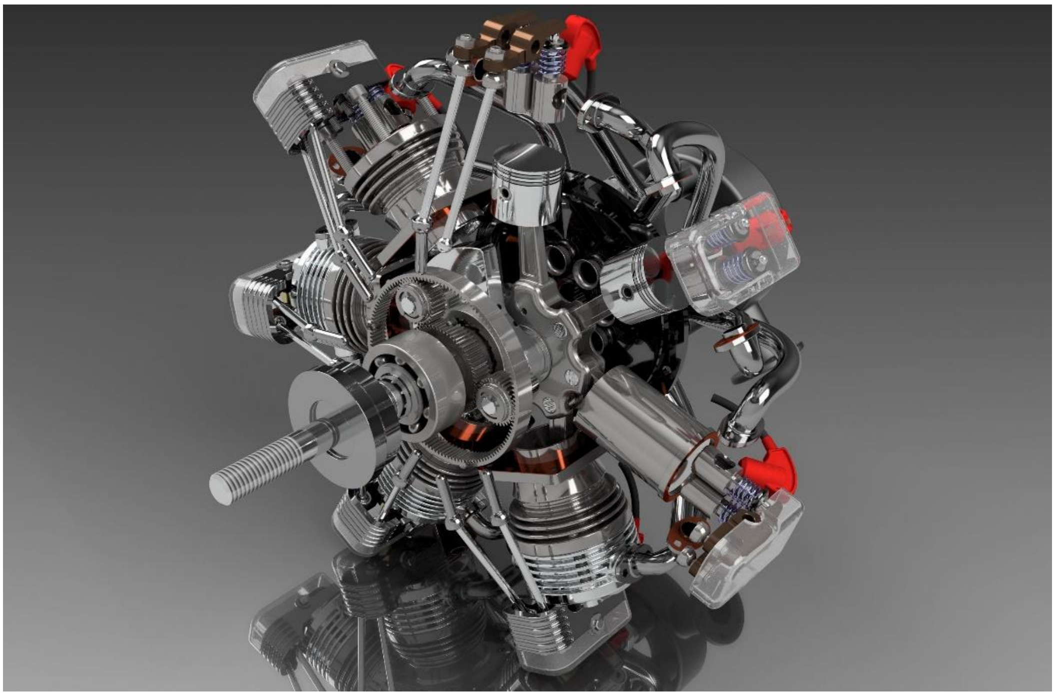


Fig- 1: Radial Engine

The Four-stroke consequence of every engine is:

a) Intake

b) Compression

c) Power

d) Exhaust

Most radial engines use overhead poppet valves driven by pushrods and lifters on a cam plate which is concentric with the crankshaft, with a few smaller radials. A few engines utilize sleeve valves instead.

1.2.PRINCIPLE:

Since the axes of the cylinders are coplanar, the connecting rods cannot all be directly attached to the crankshaft unless mechanically complex forked connecting rods are used, none of which have been successful. Instead, the pistons are connected to the crankshaft with a master-and-articulating-rod assembly. One piston, the uppermost one in the animation, has a master rod with a direct attachment to the crankshaft. The remaining pistons pin their connecting rods' attachments to rings around the edge of the master rod. Extra "rows" of radial cylinders can be added in order to increase the capacity of the engine without adding to its diameter.

Four-stroke radials have an odd number of cylinders per row, so that a consistent every-other-piston firing order can be maintained, providing smooth operation. For example, on a five-cylinder engine the firing order is 1, 3, 5, 2, 4, and back to cylinder 1. Moreover, this always leaves a one-piston gap between the piston on its combustion stroke and the piston on compression. The active stroke directly helps compress the next cylinder to fire, making the motion more uniform. If an even number of cylinders were used, an equally timed firing cycle would not be feasible. The prototype radial Zoche aero diesels (below) have an even number of cylinders, either four or eight; but this is not problematic, because they are two-stroke engines, with twice the number of power strokes as a four-stroke engine per crankshaft rotation.

1.3.APPLICATION:

Radial engines are generally used on aeroplanes because the shape allows it for the efficient packaging and air cooling behind a propeller.

1.4.ADVANTAGES:

- Easy maintenance
- Less vulnerable to critical damage
- Reliability – could still run with cylinder damage
- Versatility – becoming more desirable, increasingly more popular in experimental types
- Large frontal area provides for evenly distributed air cooling

1.5.DISADVANTAGES:

- Requires hydraulic lock avoidance
- Larger displacement equates to bigger frontal area and higher drag component
- Engine has a higher number of parts, increases the possibility of oil leaks
- Less well controlled operating temperature
- Less capable of high-altitude performance

CHAPTER-II

2. DESIGN OF RADIAL ENGINE COMPONENTS USING CATIA

2.1. COMPONENTS OF RADIAL ENGINE:

A 7-cylinder radial engine, like other radial engine configurations, consists of several key components that work together to generate power and drive an aircraft's propeller. Here are the main components of a 7-cylinder radial engine:

2.1.1. PISTON:

In a 7-cylinder radial engine, each cylinder is equipped with a piston. Pistons are essential components that play a crucial role in the engine's operation by converting the energy generated from the combustion of air and fuel into mechanical motion.

2.1.2. MASTER ROD AND SLAVE ROD:

Master rod and slave rod in a 7-cylinder radial engine are the connecting rods that connects piston and crankshaft. The connecting rod is a crucial component that connects the piston to the crankshaft and converts the linear motion of the piston into rotational motion of the crankshaft. Each piston in an internal combustion engine, including a 7-cylinder radial engine, is attached to a slave rod. And all the 7 slave rods are connected to the master rod which is attached to the crank shaft.

2.1.3. CRANK SHAFT:

The crankshaft is a central component in an internal combustion engine, including a 7-cylinder radial engine. It is responsible for converting the reciprocating motion of the pistons, which move up and down in the cylinders, into rotational motion that drives the engine and powers external systems such as propellers. In case of 7-cylinder radial engine, the piston movements are radial, as they are attached to the master rod whose movement is radial due to the connection of crankshaft with master rod at its centre.

2.1.4. PISTON RING:

Piston rings are essential components in internal combustion engines, including 7-cylinder radial engines. These rings are positioned around the outer diameter of the piston and play a critical role in ensuring efficient engine operation. Piston rings provide a tight cylinder space. They work at high temperature and variable loads. The main requirements about them are to have high elasticity, durability and low coefficient of friction with the cylinder walls.

2.2. DESIGN OF COMPONENTS USING CATIA:

CATIA is among the very few software which has its application in about every industrial sector. It is mostly used by the designer team. The designer team of any organization needs to create a digital copy of any object which has to be manufactured. It is mostly found in companies who are associated with design and manufacturing of products.

For designing of Radial engine Catia software is used. CATIA is an acronym for Computer Aided Three-Dimensional Interactive Application. It is one of the leading 3D software used for Part Design, Generative Shape Design and also for the Assembly. These are only three of the many workbenches that CATIA offers. So, the parts required for our model are designed as follows:

2.2.1. PISTON:

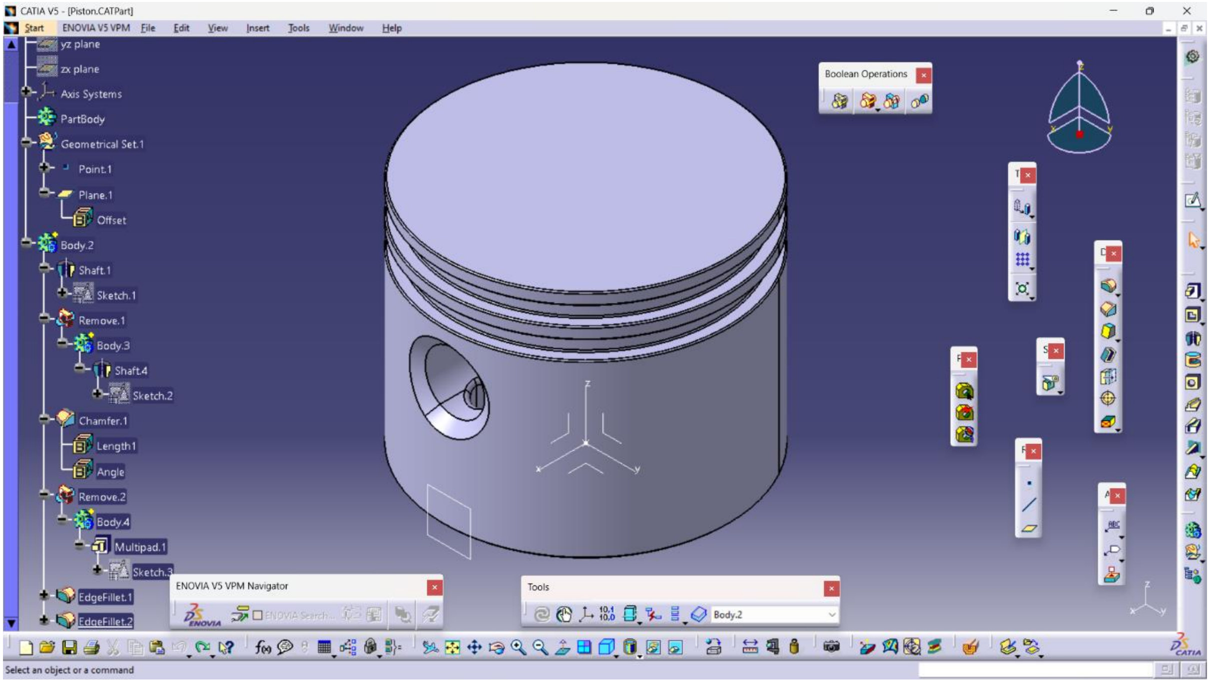


Fig- 2: Design of Piston using Commands like Shaft

2.2.2. MASTER ROD:

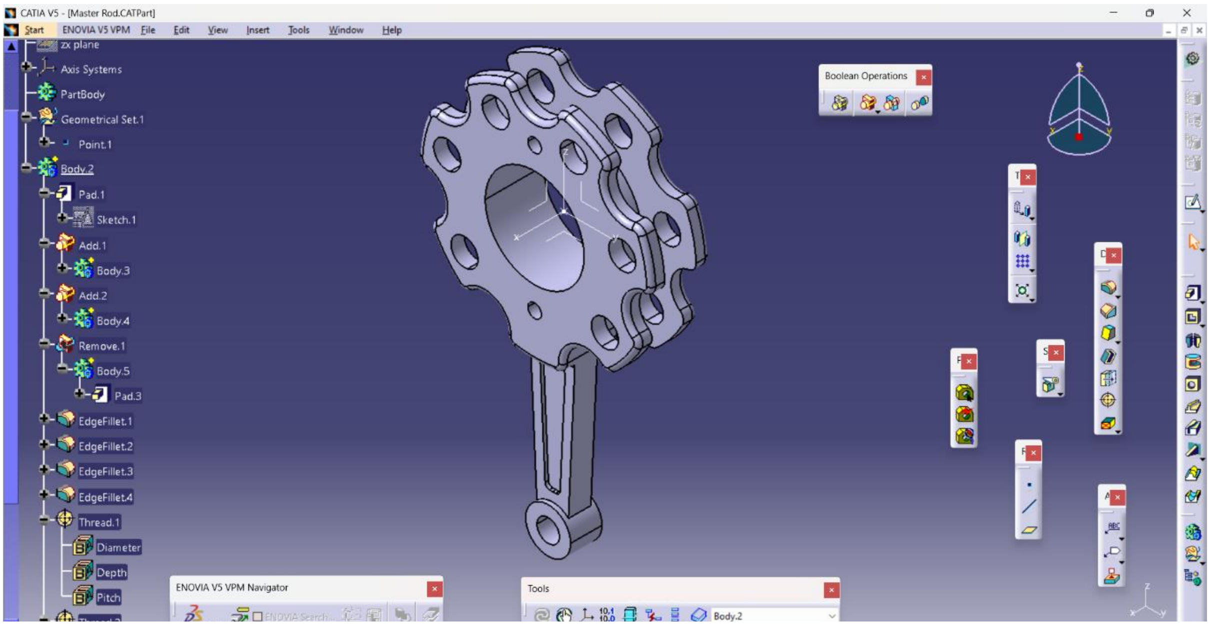


Fig- 3: Design of Master Rod using commands like Pad

2.2.3. SLAVE ROD:

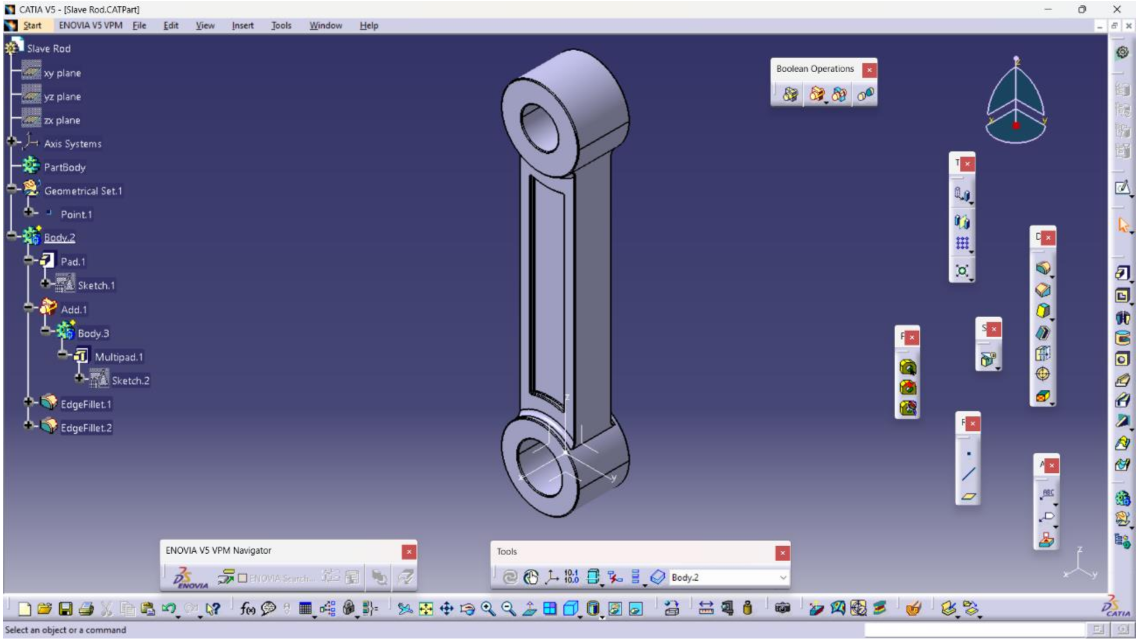


Fig- 4: Design of Slave Rod using commands like Pad

2.2.4. CRANK SHAFT:

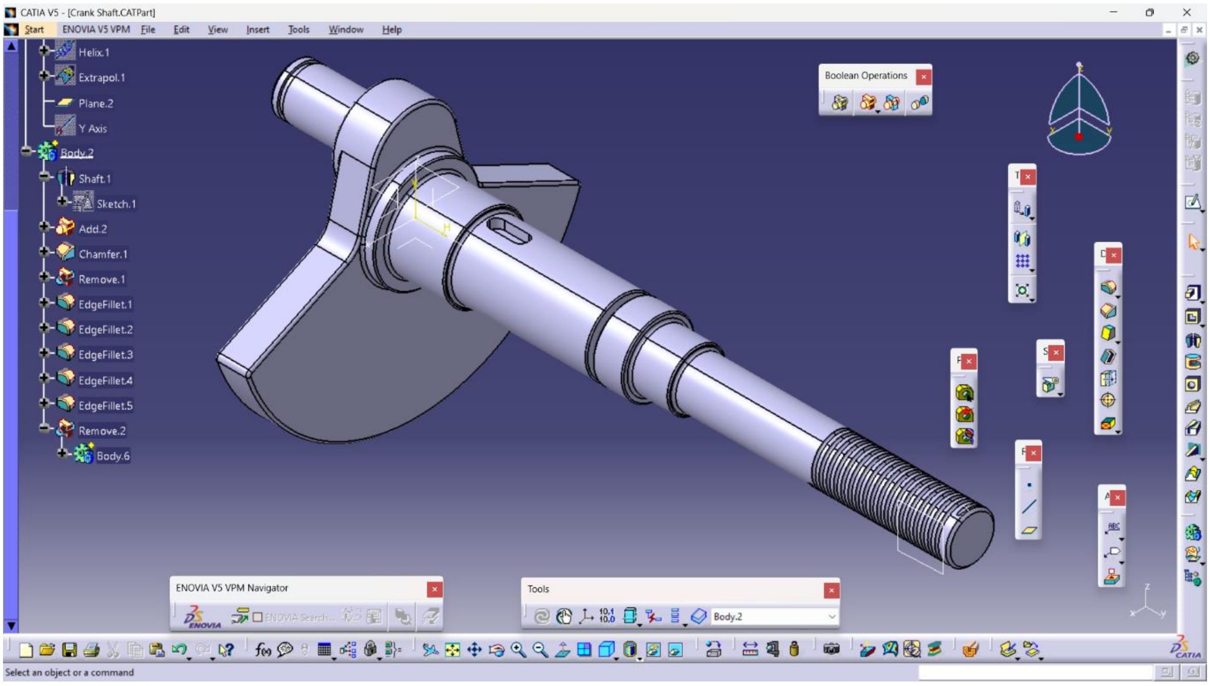


Fig- 5: Design of Crankshaft using commands like Shaft

CHAPTER-III

3. INTRODUCTION TO NUMERICAL METHODS

3.1. TYPES OF METHODS TO SOLVE AN ENGINEERING PROBLEM:

There exist three types of methods where an engineering problem can be solved in all sort of ways.

3.1.1. Experimental Methods:

In the Experimental type of methods, an engineering problem can be solved by conducting some practical sessions.

- Laboratory Experiments: Engineers conduct experiments to test hypotheses, validate models, and gather data on physical properties and behaviours of materials and systems.
- Field Testing: Field experiments involve collecting data and conducting tests in real-world conditions, often used in civil engineering, environmental engineering, and geotechnical engineering.

Drawbacks: Initial cost, specimen, space occupied by the experimental setup, an experimental setup is used only for a required experiment.

3.1.2. Analytical Methods:

In the Analytical methods, an engineering problem can be solved using mathematical equations and conducting some statistical surveys.

- Mathematical Analysis: Engineers often use mathematical equations and models to describe and analyze physical phenomena. Techniques such as calculus, differential equations, and linear algebra are frequently employed.
- Statistical Analysis: Statistical methods can be used to analyze data, assess uncertainty, and make predictions or decisions based on experimental or observational data.

Drawbacks: This method is only suitable for small problems. It takes a lot of time to solve a complex body and requires a huge knowledge on solving problems using formula itself.

3.1.3. Numerical Methods:

In Numerical methods, to solve an engineering problem, it involves the use of numerical algorithms and computational tools to find approximate solutions, often in the context of complex mathematical models or systems. An engineering problem can be solved through various types of Numerical methods.

- Finite Element Analysis (FEA): FEA is a numerical technique used to analyze complex structures and systems by dividing them into smaller, manageable elements.
- Computational Fluid Dynamics (CFD): CFD simulations are used to analyze fluid flow and heat transfer in systems, such as pipes, pumps, and aircraft.
- Finite Difference Methods: These are numerical techniques used to approximate solutions to partial differential equations, often applied in heat transfer and fluid flow problems.

Drawbacks: Numerical methods may not give exact analytical solutions. It gives approximate solutions.

So as the work to be done in small interval of time and obtain approximate results, the numerical methods are used in analysis software tools like ANSYS, HyperMesh etc. The problem is solved by conducting some analysis and simulations, with respect to real time operations, in a software tool without a physical prototype of the product.

3.2. PHASES OF NUMERICAL METHODS:

There exist three phases of numerical methods to solve a problem.

3.2.1. Pre-processing:

Pre-processing is the first phase of numerical method to solve a problem, where it is known to be the process of preparing a model to solve. The preparing of material is done as follows:

1.) Meshing
2.) Properties(*Section)
3.) Material(*MAT)
4.) Assembly
5.) Intersections and Penetrations
6.) Connections
7.) Loads
8.) Boundary Conditions
9.) Contacts
10.) Control Cards
11.) Database Cards

Software tools used for pre-processing: HyperMesh, ANSA, ANSYS, LS-prepost.

3.2.2. Solving:

The second phase of numerical method is Solving the pre-processed model, using a solver software tool. The solver tool solves the problem by using formulas and algorithms associated within it and arises error if there are any.
Software tools used for Solving: LS-DYNA, Abaqus, Nastrom, Pam-crash, Actron, ANSYS, Radioss, Opstra.

3.2.3. Post-processing:

Post-processing is the final phase of numerical method, where the outputs like stresses, strains, deformations and energies of a solved problem are shown. Based up on the outputs obtained, the CAD model may go over some small changes if any.

Software tools used for Post-processing: HyperView, LS-Prepost, ANSYS.

3.3.HYPERMESH:

Altair Hyper Mesh is a powerful simulation solution that redefines CAE design processes across industries with unmatched pre- and post-processing capabilities for finite element modeling. Powered by AI-augmented 3D modeling and visualization tools, next-gen design and optimization workflows, and an open, programmable, user-friendly interface, Hyper Mesh empowers users to effortlessly manage large and complex models, unveil critical insights, optimize designs, and foster innovation. It revolutionizes engineering workflows with unparalleled precision to enhance productivity and scale operations.

Hyper Mesh’s pre- and post-processing capabilities for finite element modeling streamlines workflows and manages complex models to boost productivity. Hyper Mesh offers specialized workflows, direct integration with PDM systems, seamless solver integration and I/O support, and extensive applications in various industries such as aerospace, automotive, heavy equipment, marine, architecture and control, and consumer goods to empower swift decision-making and cost-effective design processes throughout all stages of product development.

Democratizing the design process, Hyper Mesh offers an intuitive user interface and support of domain-specific AI-powered workflows. In-app support, workflow guidance, and convenient mouse controls and keyboard shortcuts further elevate the user experience. This powerful FEM software optimizes efficiency by offering workflows tailored to specific industries or disciplines. By incorporating AI powered workflows, it automates complex processes and delivers intelligent recommendations.

3.3.1. Key features of HyperMesh:

1) Direct FE and Geometry Modeling
2) Build and Manage Large Models and Assemblies
3) PDM Integration and Revision Management
4) High-Fidelity Meshing
5) Interactive Morphing
6) Precise, Efficient Sketching
7) Optimization-Enabled Design Space
8) Optimize Designs with Skeleton Modeling
9) Powerful Post-processing and Data Analysis

3.3.2. Functions of HyperMesh:

3.3.2.1. Geometry Cleanup and Preparation: HyperMesh allows engineers and analysts to import 3D CAD models and prepare them for finite element analysis (FEA). This includes tasks like geometry simplification, removal of unnecessary details, and ensuring that the model is suitable for meshing.

3.3.2.2. Meshing: It provides powerful meshing capabilities, enabling users to create finite element meshes from the prepared geometries. HyperMesh offers a wide range of meshing methods, including tetrahedral, hexahedral, and shell element meshing, along with advanced controls for mesh quality and refinement.

3.3.2.3. Boundary Conditions and Loads: Engineers can define boundary conditions, such as constraints and supports, as well as apply various loads and forces to simulate real-world scenarios. This is essential for accurately representing the physical behaviour of the system under analysis.

3.3.2.4. Material Properties: HyperMesh allows users to assign material properties to different parts of the model. This is crucial for performing structural and thermal analyses, as material properties impact the system's response to various loading conditions.

3.3.2.5. Assembly and Connections: Engineers can assemble complex systems with multiple components, ensuring proper connections and interactions between parts. This is particularly important in simulations of assemblies and mechanisms.

3.3.2.6. Model Visualization: The software provides powerful visualization tools, making it easier to review and validate the model before conducting simulations.

3.3.2.7. Customization and Automation: HyperMesh offers a scripting language that allows users to automate repetitive tasks and customize the software to meet specific needs.

3.3.2.8. Integration: It seamlessly integrates with various CAE solvers, allowing users to export the pre-processed models and simulation data to perform analyses in tools like Altair OptiStruct, Abaqus, ANSYS, and more.

3.4. LS-DYNA:

HyperMesh provides a complete pre-processing environment for preparing LSDYNA data decks for analysis. Hyper Mesh can read existing LS-DYNA decks, create a model, display and edit LS-DYNA cards as they will look in the deck, and write a deck for analysis. To create LS-DYNA decks in Hyper Mesh, you must load the LS-DYNA user profile with the appropriate template to access the full pre-processing capability.

Key features and applications of LS-DYNA include:

Explicit Dynamics: LS-DYNA excels in modeling highly dynamic events, such as crash simulations, impact analysis, and explosions. It can accurately capture the transient behavior of structures and materials under extreme loading conditions.

Nonlinear Analysis: The software is capable of handling a wide range of material models, including plasticity, viscoelasticity, and hyper elasticity. This allows for the simulation of materials that behave nonlinearly under various loading conditions.

Coupled Multi-Physics Simulations: LS-DYNA can perform coupled simulations involving multiple physics, such as fluid-structure interactions (FSI), thermal analysis, and electromagnetic simulations, providing a holistic view of complex systems.

Material Models: LS-DYNA offers a broad library of material models, enabling users to accurately represent the behaviour of various materials, including metals, composites, foams, and elastomers.

Geometric Modeling and Meshing: It supports 2D and 3D modeling, as well as sophisticated meshing techniques to create finite element models that accurately represent the physical geometry of the system.

Crash and Safety Analysis: LS-DYNA is commonly used in the automotive industry to simulate vehicle crash tests, airbag deployments, and occupant safety evaluations.

Structural Analysis: It is employed in aerospace and civil engineering for structural analysis of components and systems, helping engineers assess the integrity of designs under various loading conditions.

Virtual Prototyping: LS-DYNA allows for the development of virtual prototypes, reducing the need for physical testing and speeding up product development cycles.

Research and Development: It is used in research institutions and laboratories to investigate a wide range of phenomena, including impact dynamics, fluid-structure interactions, and material behaviour.

3.5.IMPORT AND EXPORT:

- Hyper Mesh supports LS-DYNA solver versions through solver version 971_R12.0.
- For an LS-DYNA R11.1 and later profile, keyword attribute comments are written in the exported deck.
- Hyper Mesh supports LS-DYNA Long and i10 formats.
- Solver specific import options are available during import in the Solver Options tab.
- Hyper Mesh supports LS-DYNA Dummy models with the Primer and LSTC dummy information format. Hyper Mesh writes out the dummy information on Primer format.
- Most IDs in the solver deck are preserved in Hyper Mesh. If a keyword is not supported in a dedicated Hyper Mesh entity to ensure its unique ID-Pool, then Hyper Mesh renumbers those keywords when ID conflicts are detected. The new IDs are posted during the import process.
- The LS-DYNA interface supports a smart, reliable FE input reader that warns you when your input deck contains unsupported fields and unsupported data lines.
- Hyper Mesh supports parameterized IDs for Components, Materials, Properties, and Curves.
- Hyper Mesh supports undefined entities. These are entity IDs which are referenced in keywords (for example a Material ID in a *PART) but not defined in the deck. In this case, Hyper Mesh creates a default card (for example a material of type *MAT_ELASTIC is then created) in order to preserve the ID. This keyword has the Defined checkbox toggled off and is automatically not exported.

3.6.LS-DYNA MASS CALCULATION:

The mass reported is not simply calculated by Density x Volume for each part. It follows the many LS-DYNA requirements to handle rigid body mass, non-structural mass, and lumped mass.

Contributing Total Mass factors: total mass = structural mass + lumped mass + nonstructural mass + rigid body mass + transferred mass + distributed mass

Structural Mass: Volume x density; except in case of *PART_INERTIA in which it is also the total mass. Lumped Mass: Accounts for contributions from *ELEMENT_MASS, *ELEMENT_MASS_NODE_SET, and *ELEMENT_INERTIA. This does not take into account the transfer of lumped mass to rigids.

Non-structural Mass (NSM): Accounts for contributions from ELEMENT_MASS_PART, ELEMENT_MASS_PART_SET, and NSM in *SECTION. This does not take into account the transfer of lumped mass to rigids.

Rigid Body Mass: Mass of *CONSTRAINED_NODAL_RIGID_BODIES. Transferred mass: Mass transferred from deformable nodes to rigid materials. This includes lumped mass transferred from rigid or deformable nodes to the rigid materials.

- For rigid material, this is the mass gained from deformable (+).
- For deformable parts, this is the mass lost to rigid material (-).

Distributed Mass: Mass distributed from nodal rigid bodies to free nodes.

Engineering Mass: Mass of the part that most closely matches its real engineering meaning. The engineering mass is the most useful for possible mass adjustments. Engineering mass is the sum of structural, non-structural, and lumped mass.

CHAPTER-IV

4. ANALYSIS ON RADIAL ENGINE COMPONENTS

4.1. TYPES OF ANALYSIS:

There are several types of analysis in the field of engineering and science, each serving a specific purpose and providing insights into different aspects of a system or structure. Some of the common types of analysis include:

1.) Structural Analysis: This type of analysis assesses how a structure or component responds to loads and stresses, ensuring that it can support its intended purpose without failure. It includes linear and nonlinear static analysis, as well as dynamic analysis to study the response of structures to dynamic forces like vibrations and impacts.
2.) Thermal Analysis: Thermal analysis evaluates how a system or component responds to temperature changes. It is essential in understanding how temperature gradients affect materials and structures, ensuring they can withstand temperature variations without deformation, fatigue, or damage.
3.) Modal Analysis: Modal analysis identifies the natural frequencies and mode shapes of a structure, which is important for understanding how a structure will behave in response to external forces or vibrations.
4.) Buckling Analysis: Buckling analysis assesses the stability of structures under compressive loads to determine if they are prone to buckling or collapsing. This is crucial for designing columns, beams, and other slender structures.
5.) Drop Test Analysis: A drop test, as the name suggests, simulates the impact or shock experienced by an object when it is dropped from a certain height or subjected to a sudden change in velocity or position. This type of analysis is used to evaluate the structural integrity of the object and to assess how it withstands impact forces.
6.) Crash Test Analysis: Crash analysis is primarily associated with the automotive and aerospace industries. It simulates the behaviour of vehicles or aircraft during impact events, such as collisions, crashes, or emergency landings. Crash analysis is used to assess occupant safety, structural integrity, and overall vehicle performance in the event of an accident.

The analysis of Radial engine components is done in HyperMesh. This is Mechanical finite element analysis software used to simulate computer models of structures or machine components for analysing strength, toughness, elasticity, temperature distribution and many other attributes. So many types of analysis system are there in HyperMesh but I worked with static structural analysis system.

4.2. STRUCTURAL ANALYSIS:

Structural analysis is a branch of civil and mechanical engineering that involves the examination and assessment of the behaviour and stability of various structures, such as buildings, bridges, and mechanical components, to ensure they can withstand loads and environmental conditions. It employs principles of physics and engineering to predict how structures will respond to forces, stresses, and deformations, ultimately helping engineers and designers make informed decisions about the materials, dimensions, and configurations needed to create safe, efficient, and durable structures.

4.3. PRE-PROCESSING:

Open the HyperMesh software tool and select LS-DYNA as solver tool. And the interface appears as given below.

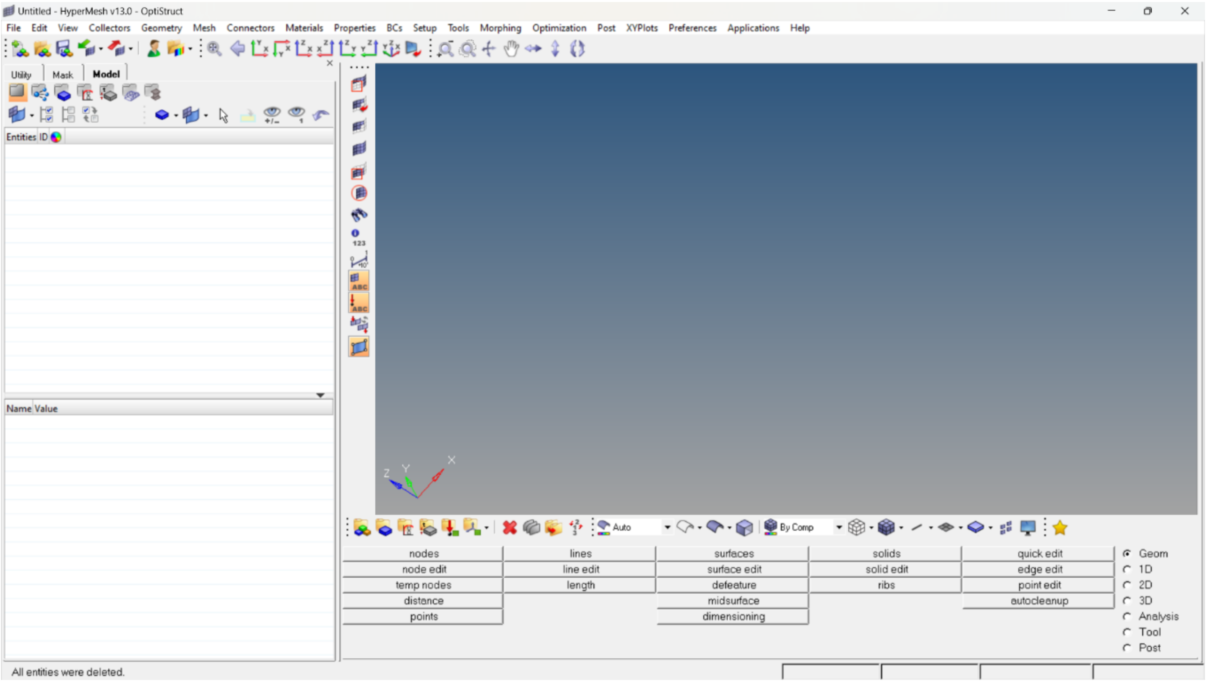


Fig- 6: Interface of HyperMesh

4.3.1. Meshing:

- Import the geometry from the "File" tool on top left side of docking area and make the imported geometry as "Current Component".
- Delete the solids from the imported geometry if any, using "Delete Panel", which can be accessed by clicking "F2" > Select "Solids" > Select "All" > Click "Delete".

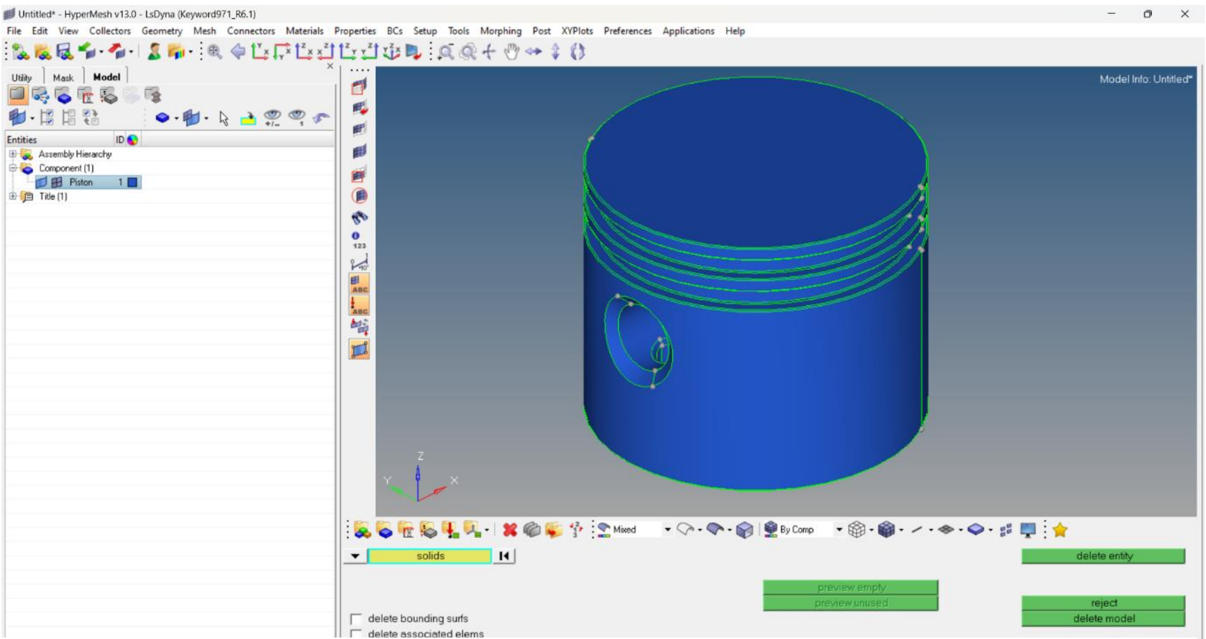


Fig- 7: Deletion of Solids

- Delete the duplicate surfaces if any, using “Defeature” option on “Geometry” panel. Select “duplicate” > Select “Surfaces” > Click “find” and delete duplicate surfaces if found.
- Set the average element length of 1mm in options panel by clicking “O”.
- Create a new component and name it as “2D Mesh”, and mesh the piston component using “Auto-mesh” option by clicking “F12” > Select “R trias”> Select all surfaces > Create mesh.

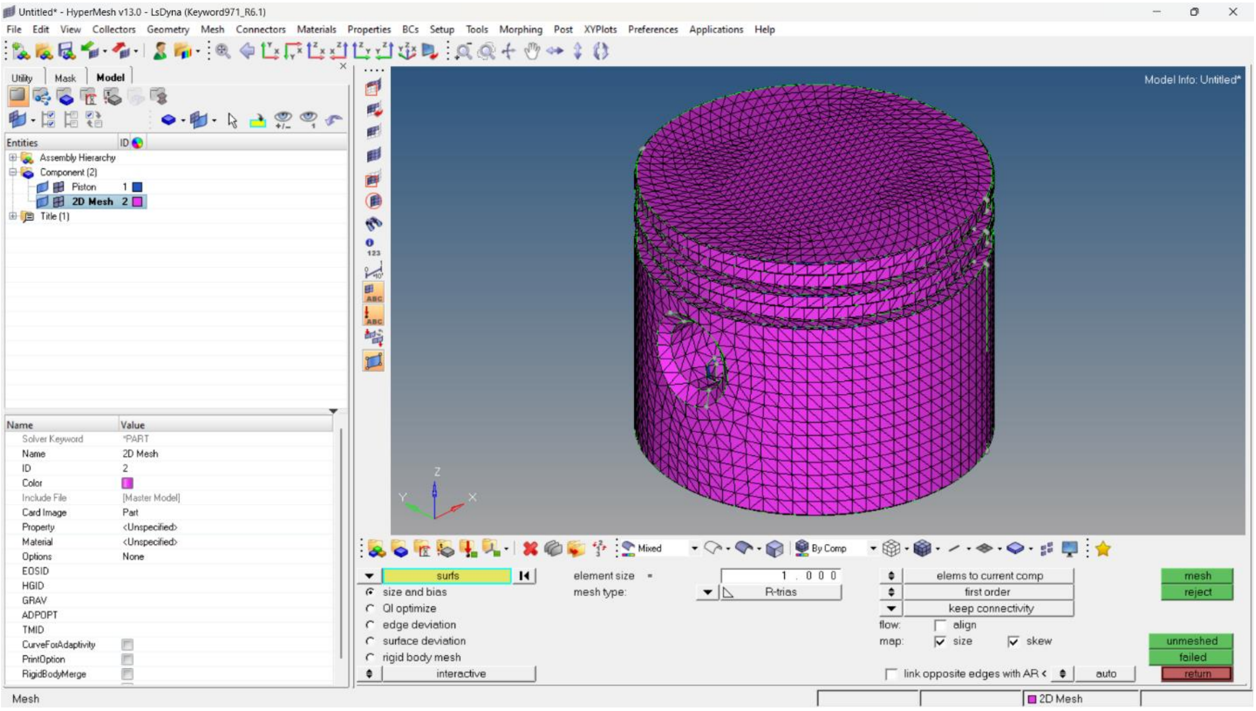


Fig- 8: 2D Mesh of Piston Component

- Check whether the model encloses a volume using shortcut “Shift + F3”.

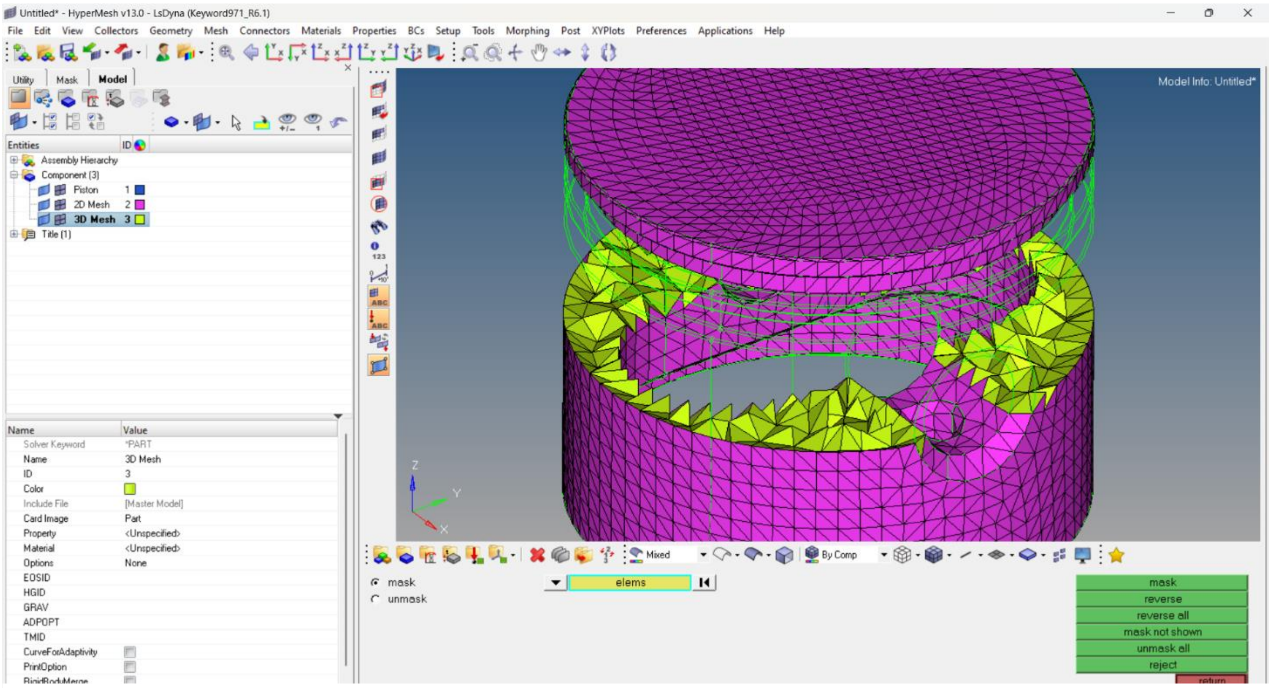


Fig- 9: 3D Mesh of Piston

- Create a new component as “3D Mesh”, and 3d meshing of piston is as follows. Select “3D” from interface > Select “Tetra Mesh” > Select “Components” to be meshed > Select “Simple Pyramid” > Create Mesh.

4.3.2. Properties:

- Creating property cards for piston is as follows: Select “Tools” from docking area > Select “Create Cards” > Select “*Section_Solid” > ELFORM=1 and name it as “Aluminium_prop”.

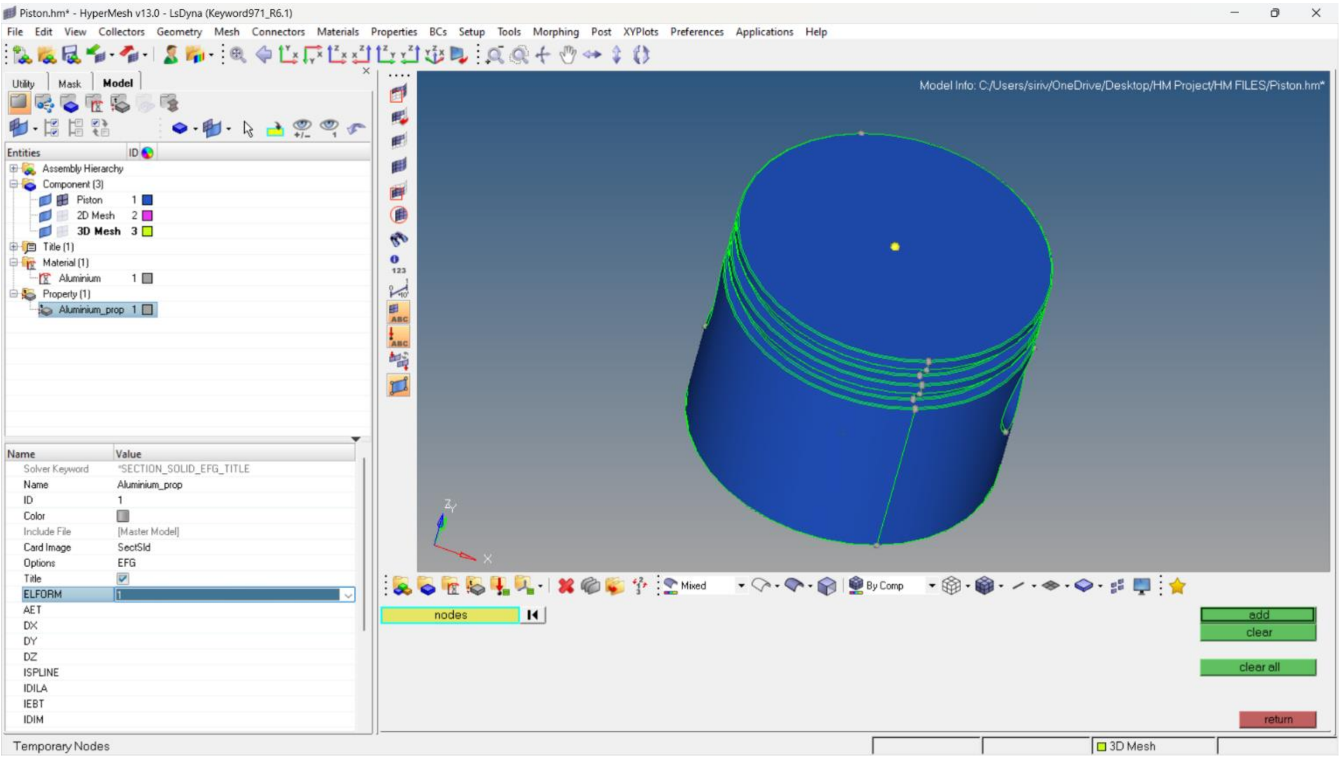


Fig- 10: Properties

4.3.3. Materials:

- Creating material cards is as follows: Select “Tools” from docking area > Select “Create Cards” > Select “*MAT01” > Density = 2700kg/m3 > Youngs Modulus = 70 GPa > Poisson’s Ratio = 0.33 and name it as “Aluminium”.

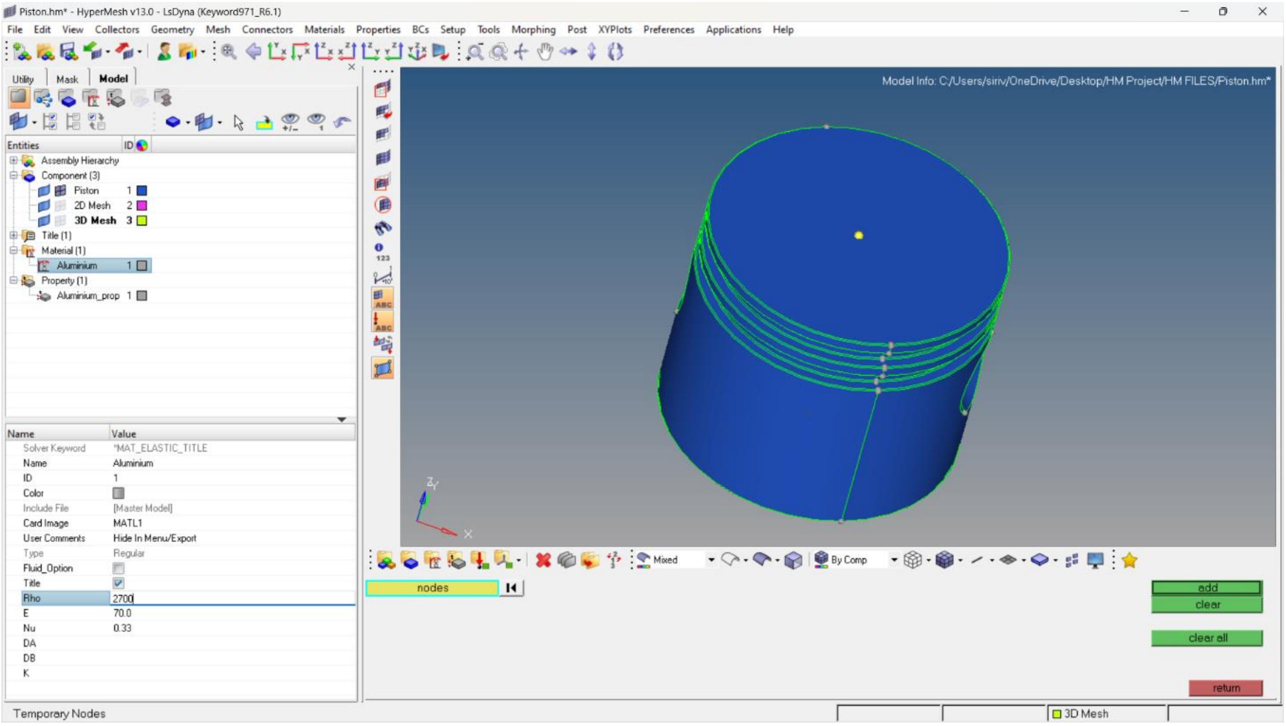


Fig- 11: Materials

- Assign the property and material to the Piston component.

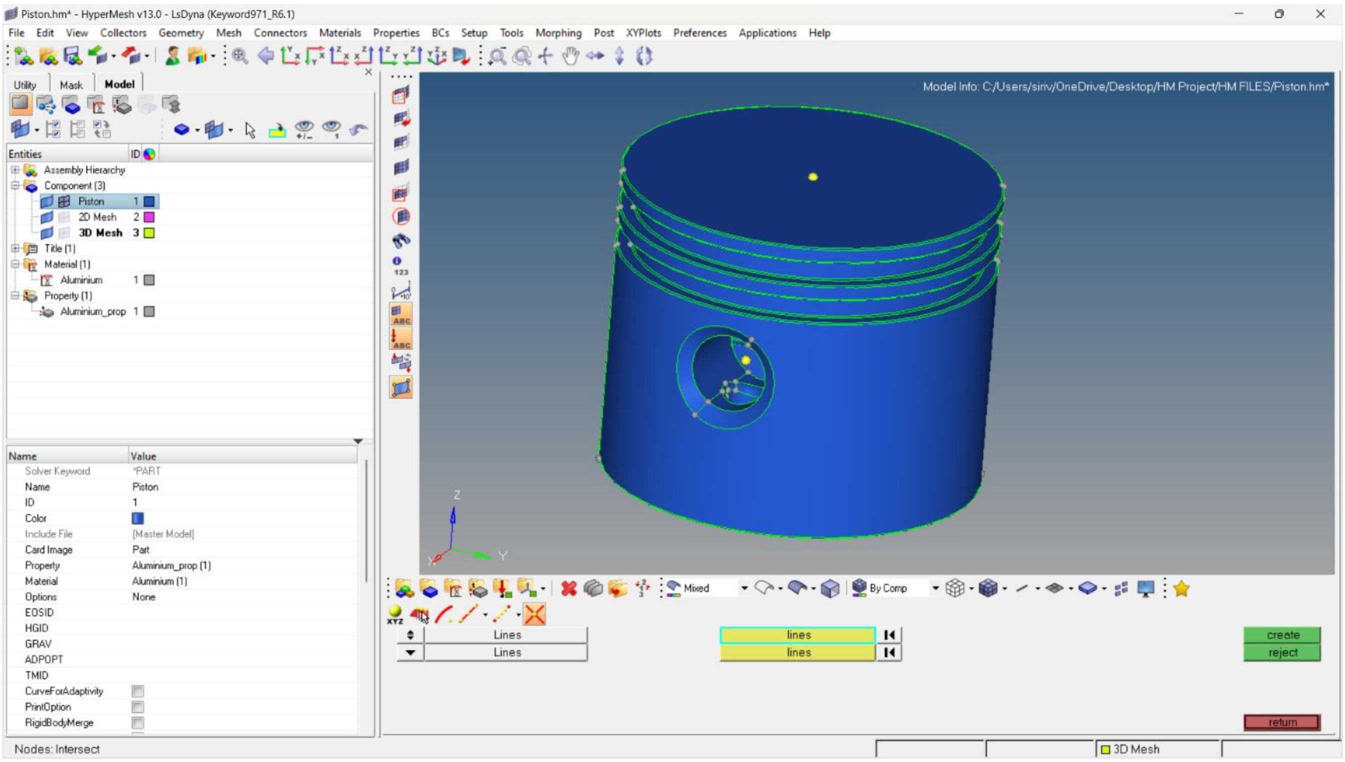


Fig- 12: Assigning property and material

4.3.4. Loads:

- Assign the nodes to the piston component, where action takes place on the component using “F8” shortcut.

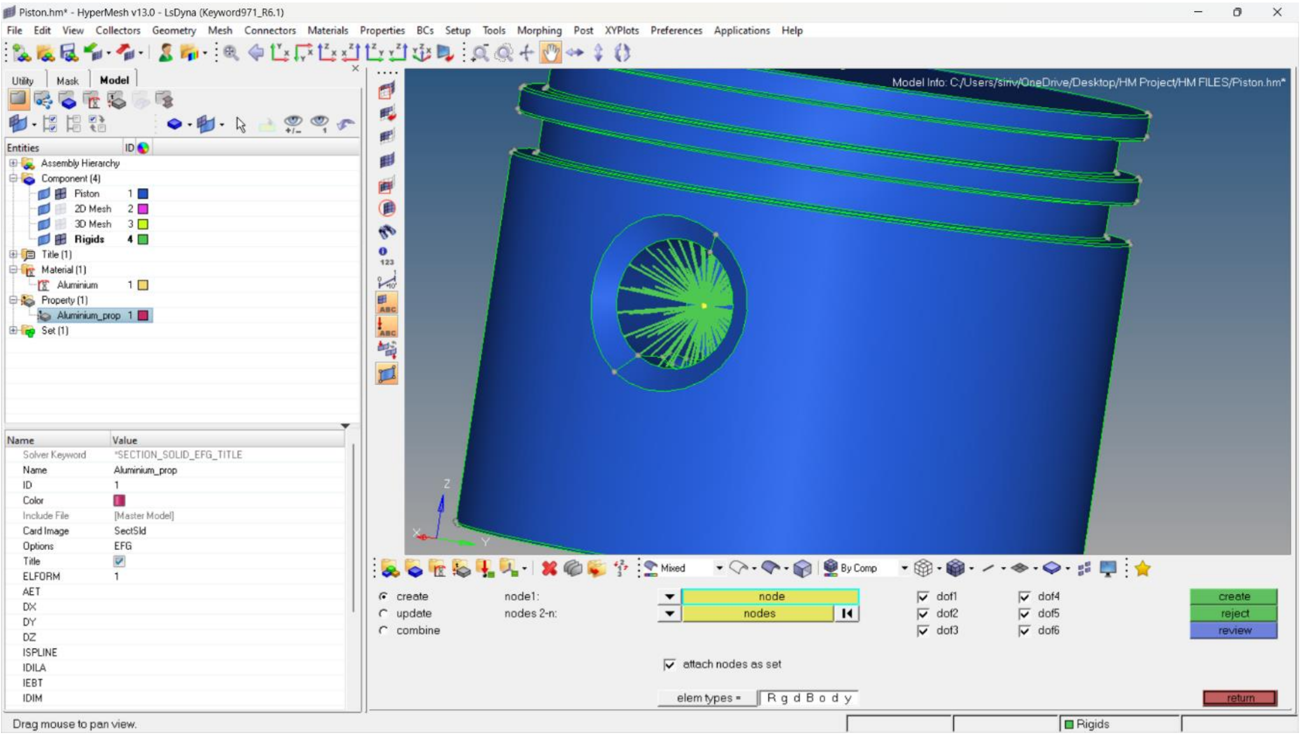


Fig- 14: Creating Nodes

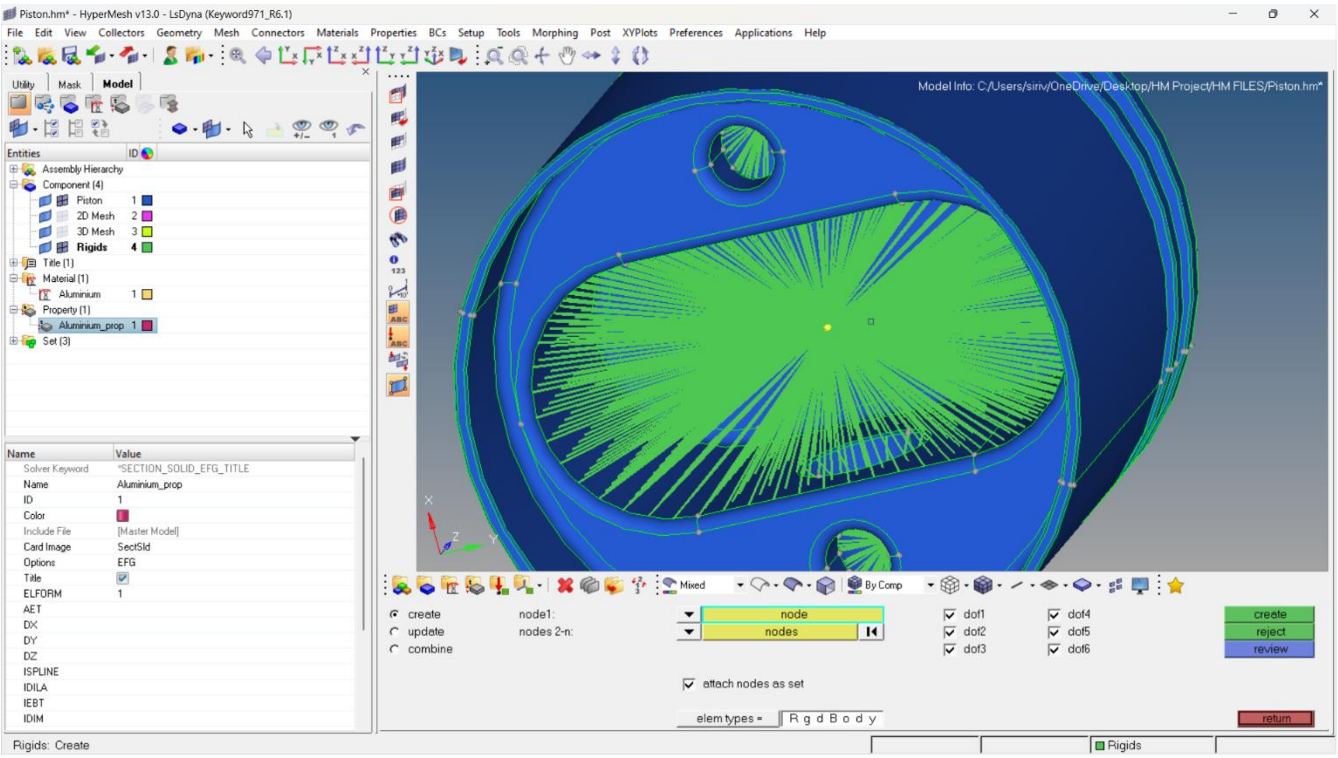


Fig- 13: Creating Nodes

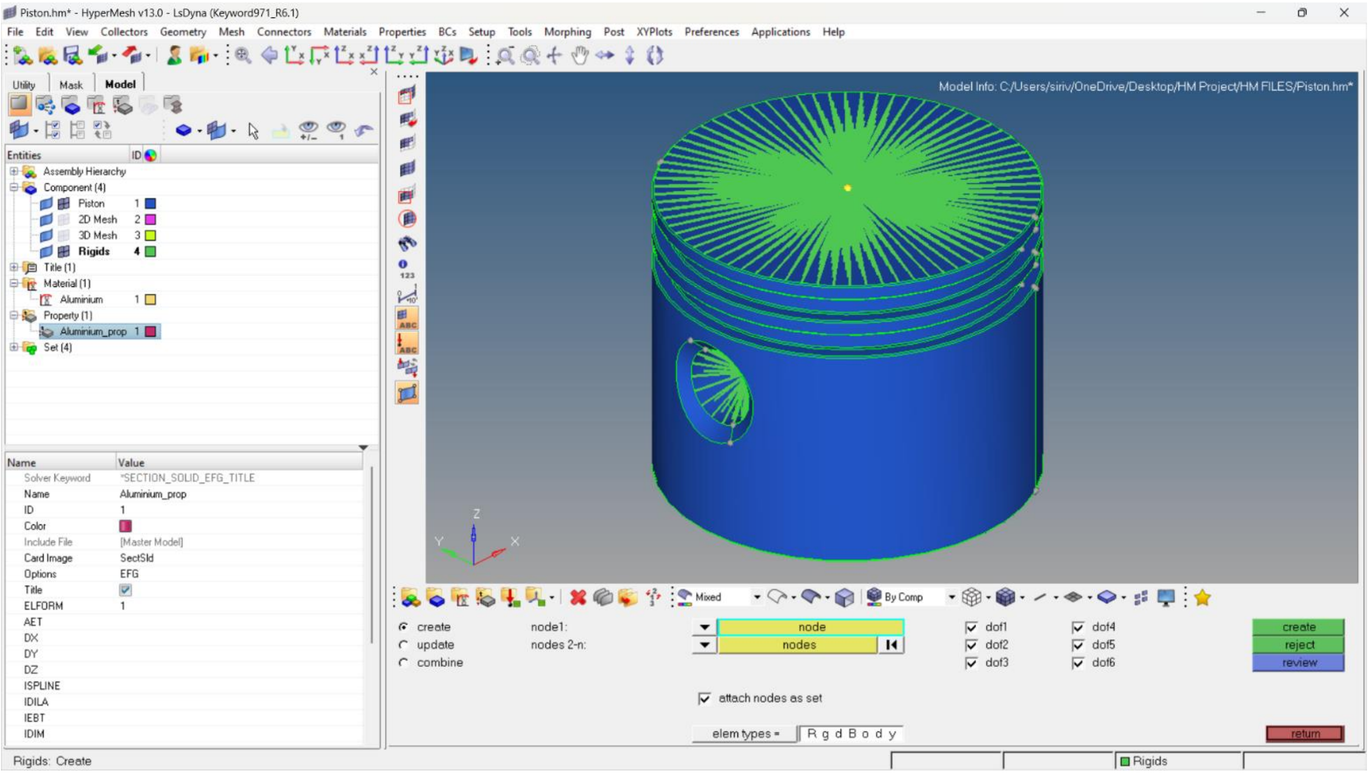


Fig- 16: Creating Nodes

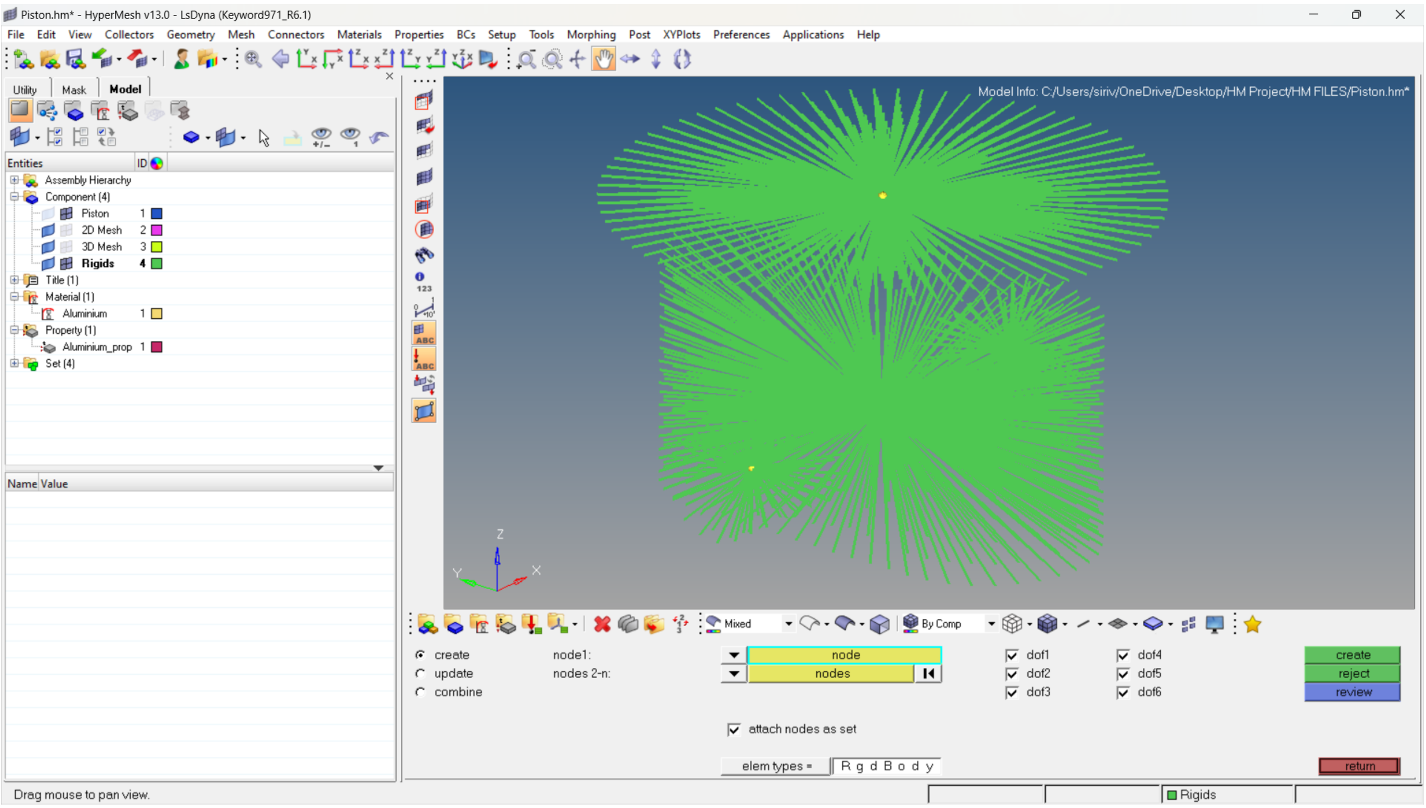


Fig- 15: Overall Node Points

- Fix the rigid nodes as follow: Analysis > Constraints > Nodes > Size = 100 > Create.

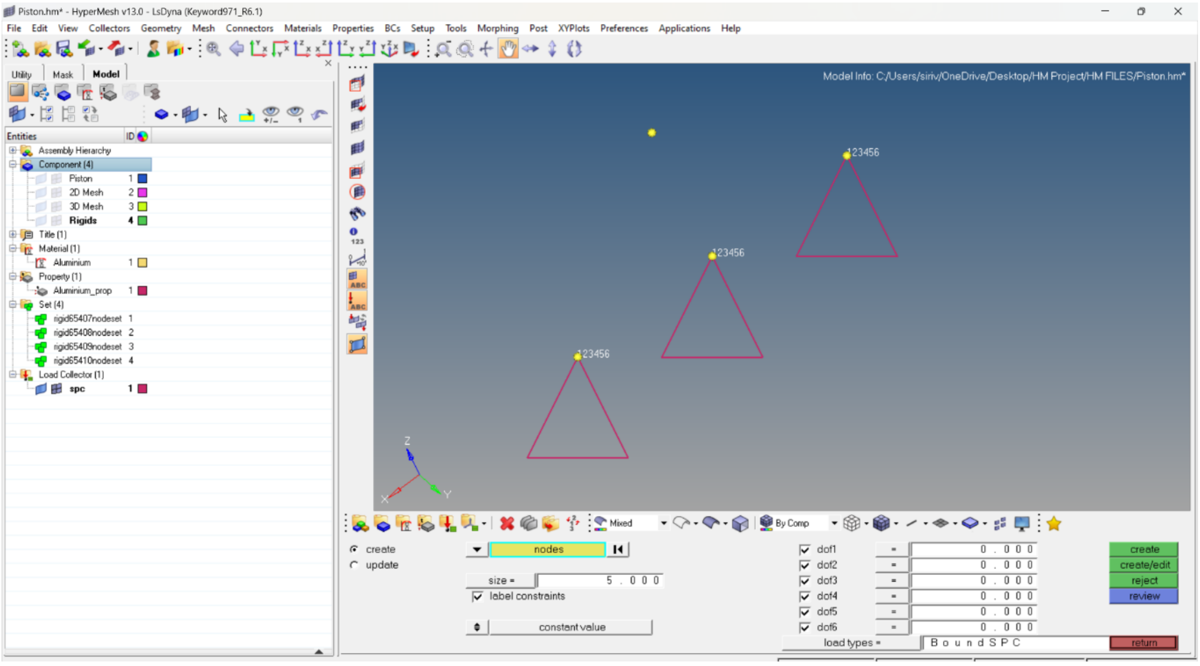


Fig- 17: Fixing the rigid nodes

- Apply the load on top node of the piston as follows: Analysis > Forces > Nodes > Select Node > Load = -1000 N > Z-axis > Create.

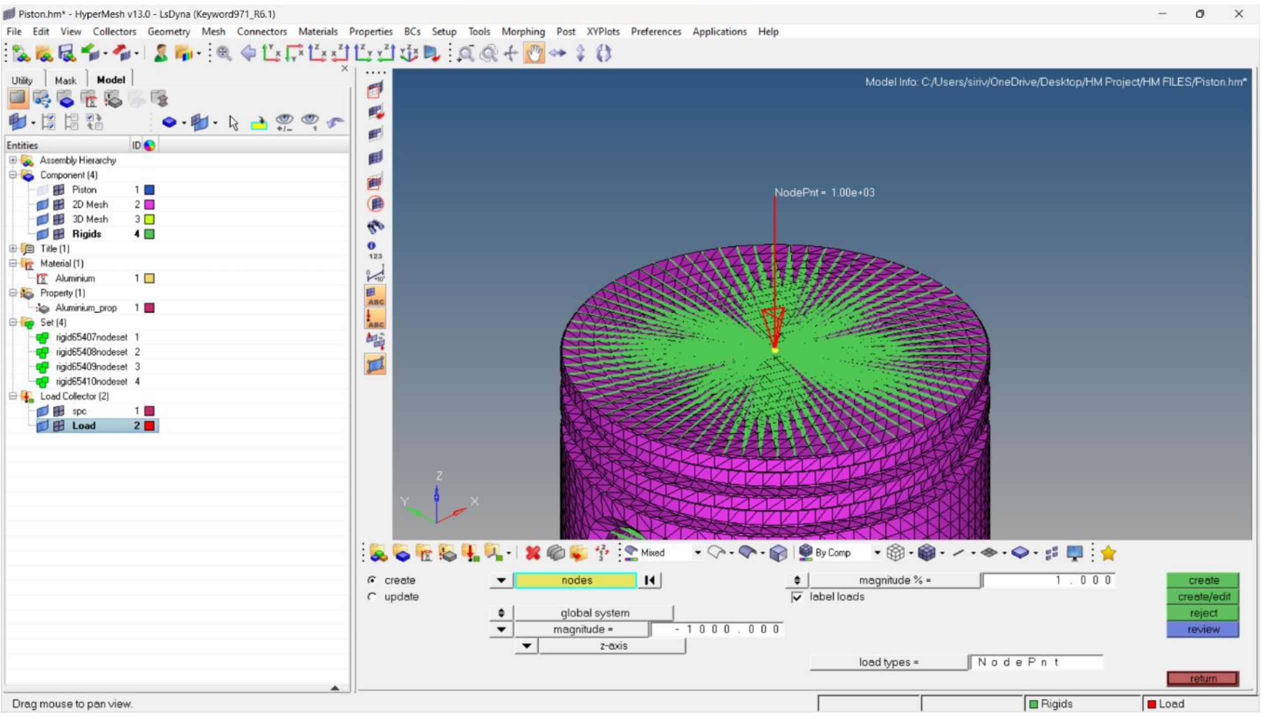
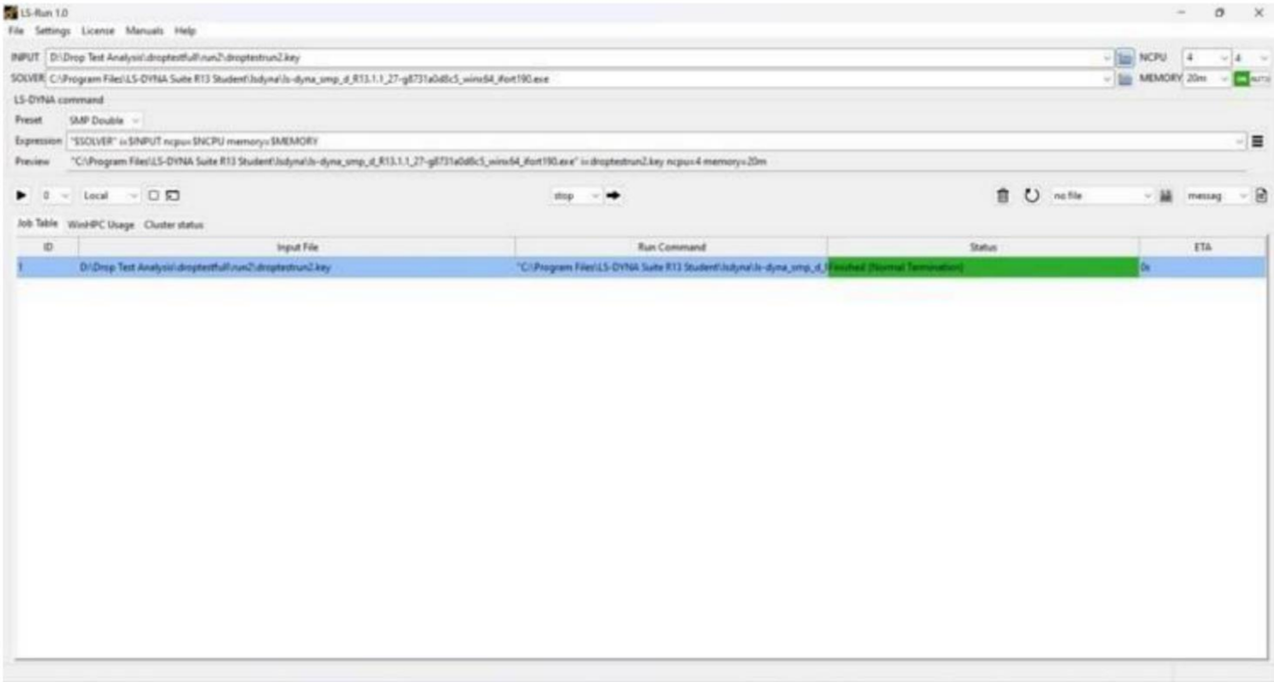


Fig- 18: Applying Load on piston

- Save the model file types as “.hm” and also as “.key”.

4.4. SOLVING:

- Now open the LS Dyna Manager (Solver) to run the file.
- After the Successful termination it shows Normal termination as shown in picture.



4.5. POST-PROCESSING:

- Now for the Post-Processing Open HyperView and import d3plot file which we get after solving process.
- The interface of the HyperView looks like this.

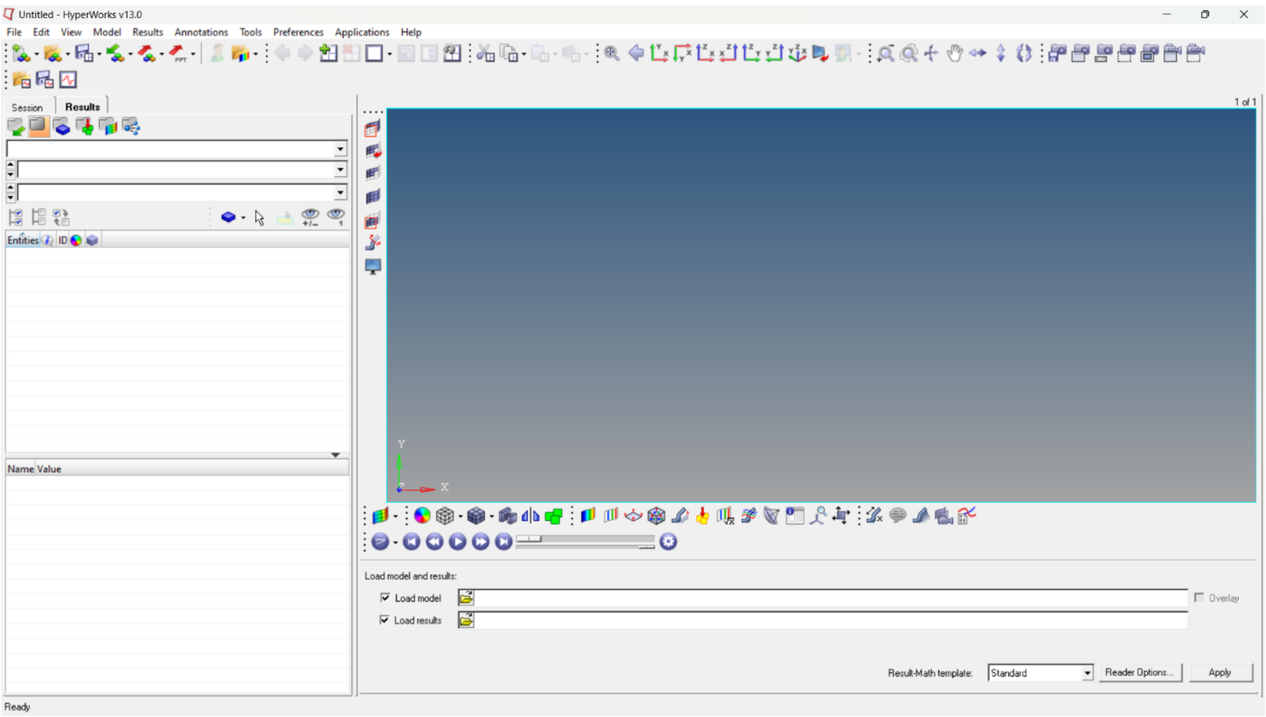


Fig- 20: HyperView Interface

- We can select any type of results in this interface, let us select Von-moisses stress from Piston Component.
- And the results of those stresses will look like this.

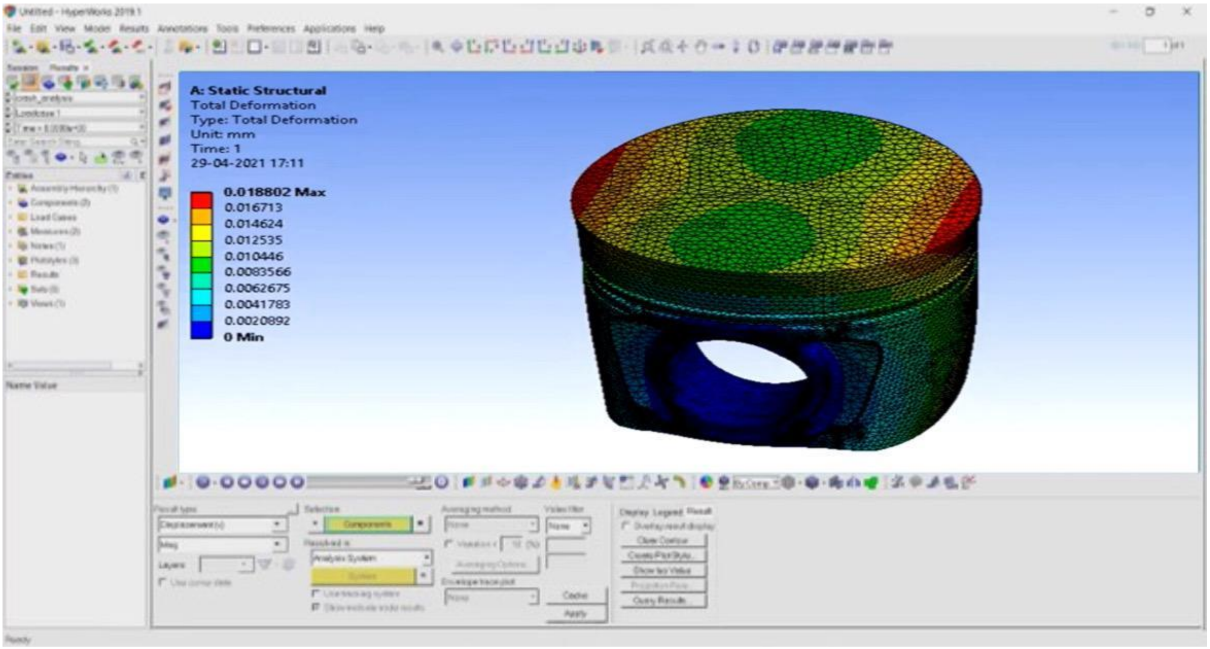


Fig- 21: Von-moisses stress of Piston

- Likewise, the remaining components of the radial engine are simulated and analyzed.

Slave Rod:

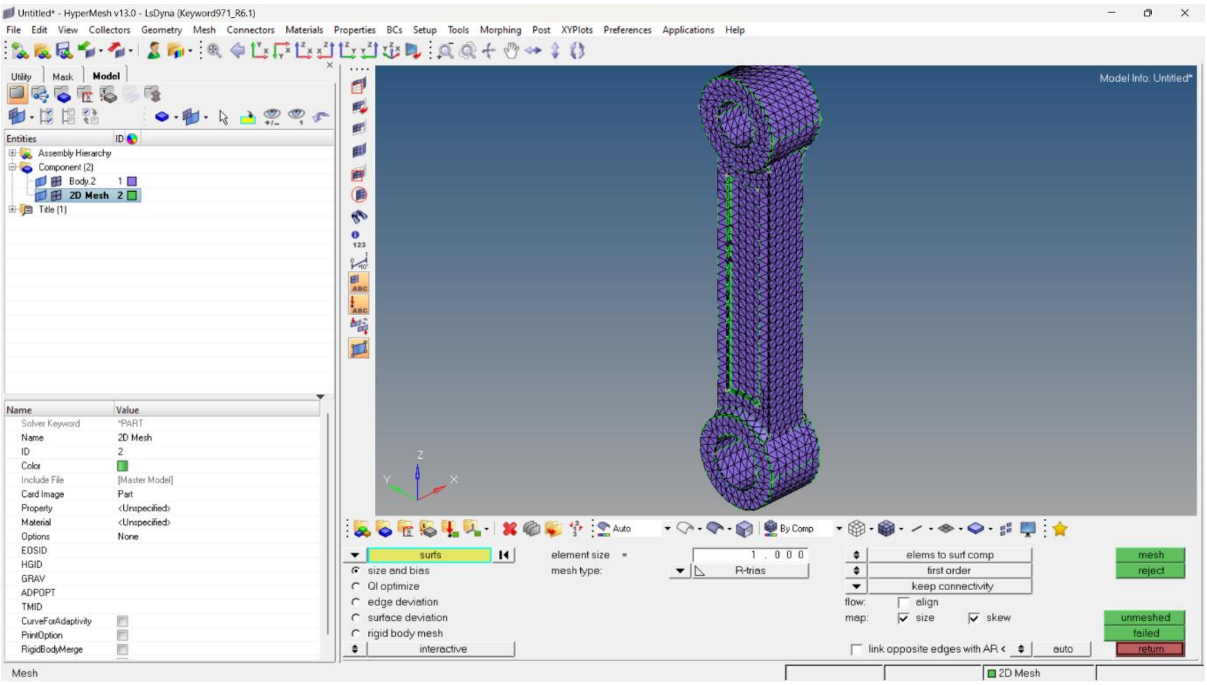


Fig- 22: 2D Mesh of Slave Rod

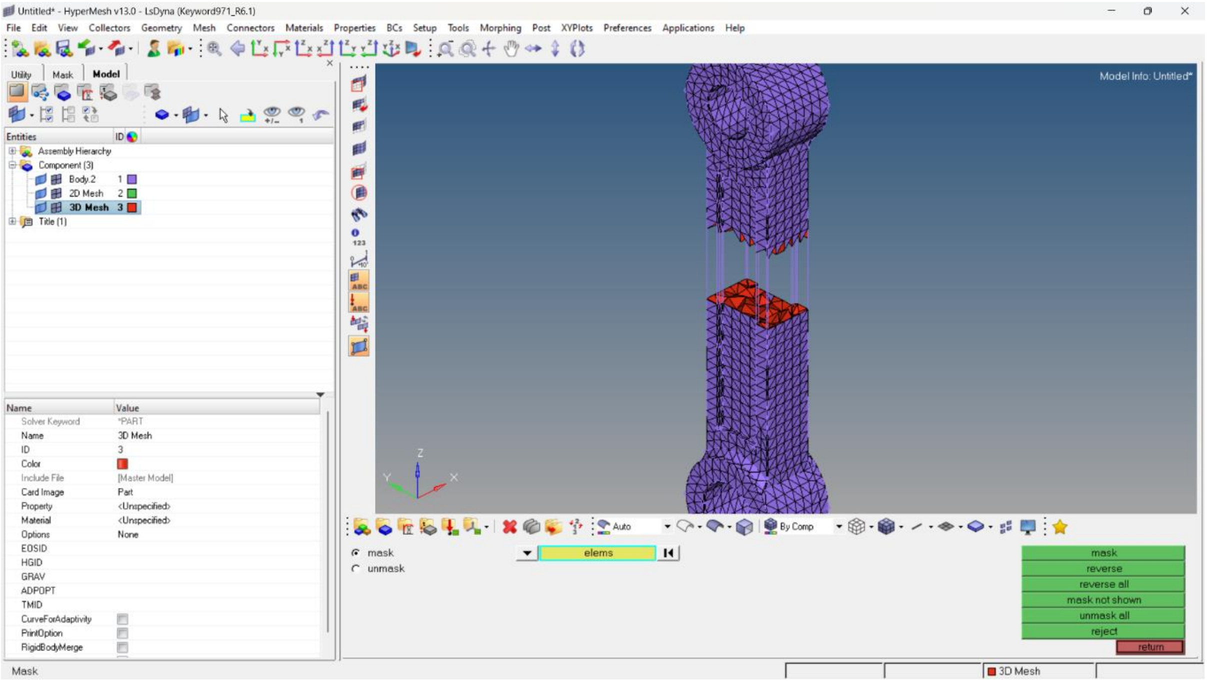


Fig- 23: 3D Mesh of Slave Rod

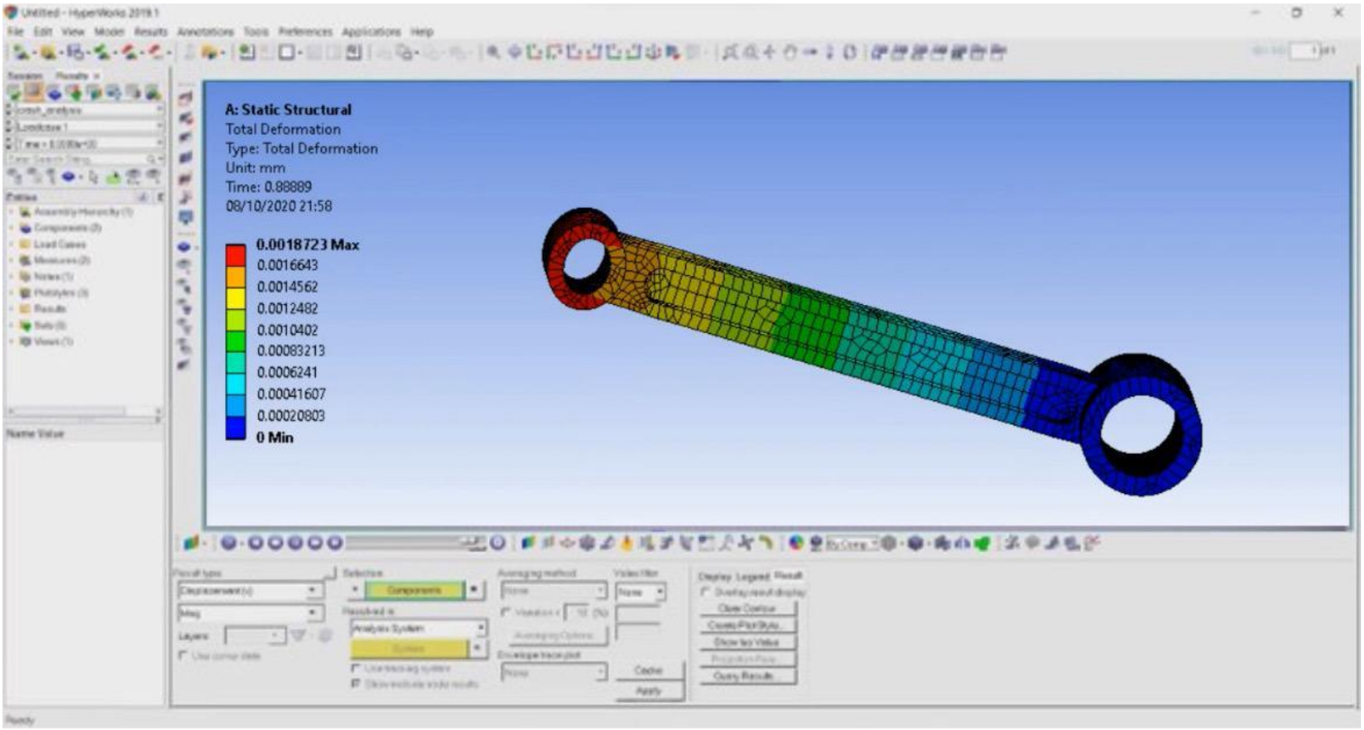


Fig- 24: Von Mosses stress of Slave Rod

Mater Rod:

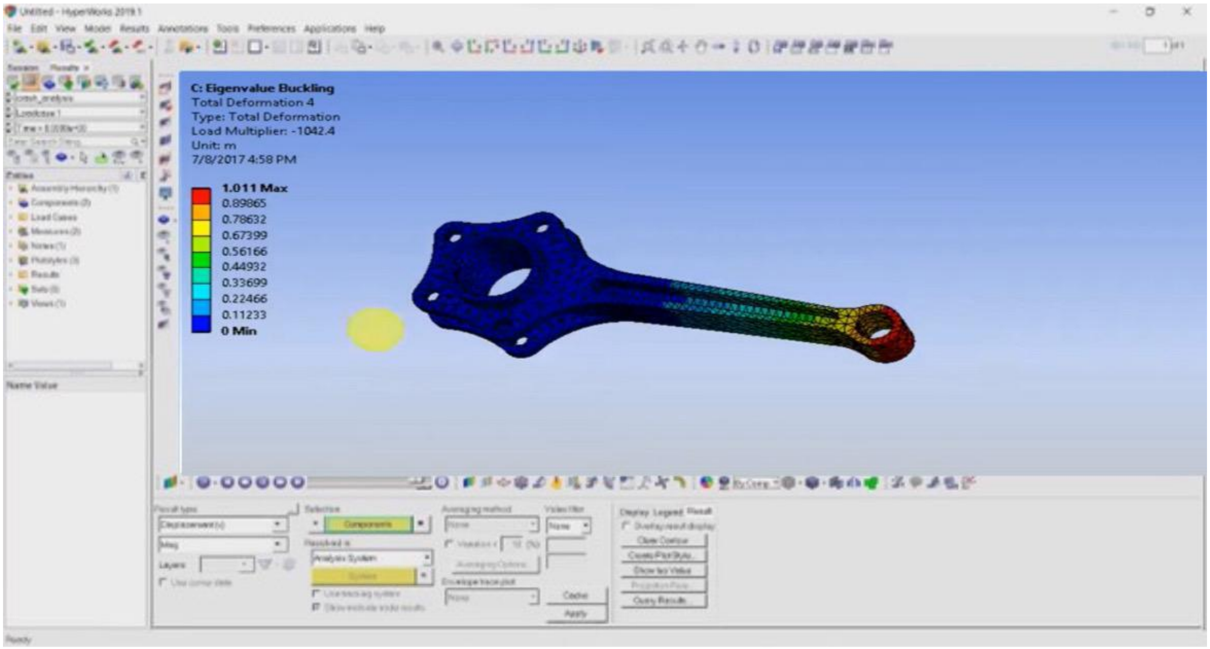


Fig- 25: Von Mosses stresses of Master Rod

Crank Shaft:

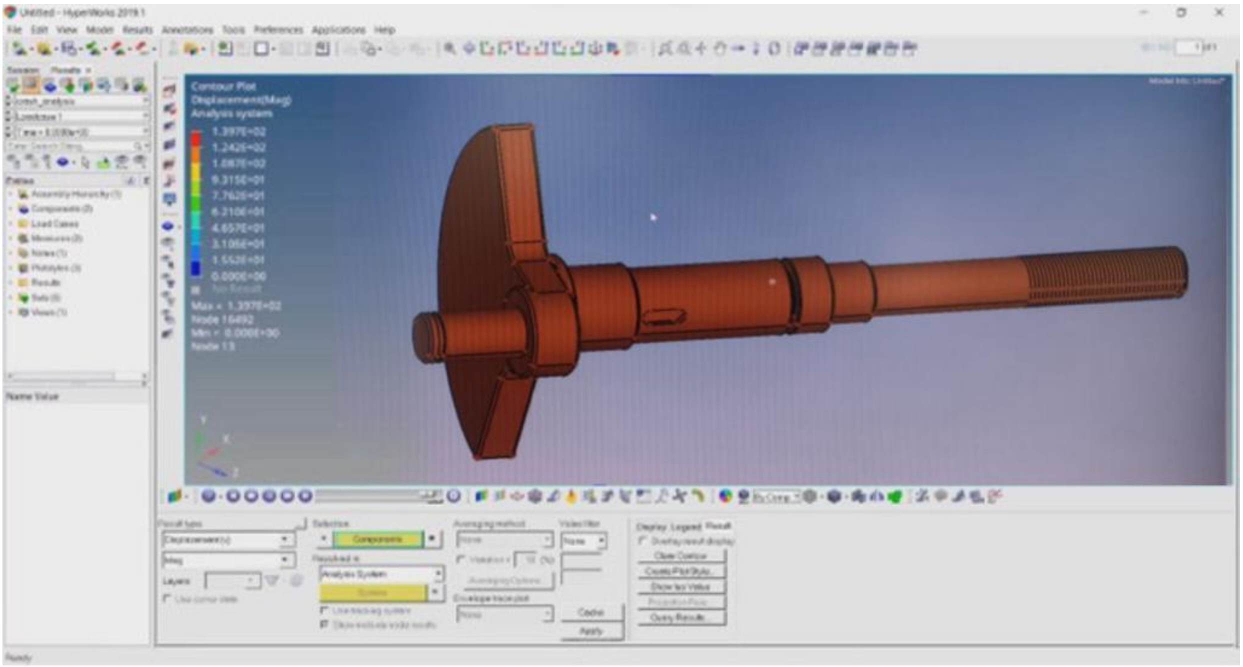


Fig- 26: Von Mosses Stress of Crank Shaft

4.6. RESULTS:

As the components of Radial Engine are analysed with two different types of materials, the results obtained in two different situations are compared.

4.6.1. Inputs:

Material	Density(kg/m^3)	Young's Modulus (GPa)	Poissons's Ratio
Aluminium Alloy	2700	70	0.33
Cast Iron	7800	200	0.2998

4.6.2. Outputs:

1.) Piston:

Material	Total Deformation (mm)	Total equivalent Strain	Equivalent Stress (MPa)	Safety Factor
Aluminium Alloy	7.0535	7.6402e-002	4879	1.6958e-002
Cast Iron	297.19	1.086e-002	3846.6	4.6742e-002

2.) Slave Rod:

Material	Total Deformation (mm)	Total equivalent Strain	Equivalent Stress (MPa)	Safety Factor
Aluminium Alloy	1.3789	1.4011e-003	99.474	0.83177
Cast Iron	0.48166	4.9076e-004	99.815	1.8013

3.) Master Rod:

Material	Total Deformation (mm)	Total equivalent Strain	Equivalent Stress (MPa)	Safety Factor
Aluminium Alloy	0.29737	1.9755e-004	14.003	5.9086
Cast Iron	0.10383	6.8969e-005	14.005	12.838

\

CHAPTER-V

5. CONCLUSION & REFERENCES

5.1.CONCLUSION:

In summary, two effective software tools that are frequently utilised in the fields of explicit dynamics and finite element analysis (FEA) are HyperMesh and LS-DYNA. When combined, they offer a complete solution for dynamic events and complex structures in a variety of industries, including modelling, simulating, and analysing.

Numerous applications, such as automotive crash simulations, aerospace impact analysis, structural dynamics, manufacturing process optimization, military and defence applications, sports equipment design, medical device safety, and consumer product safety, can benefit greatly from the integration of these two tools. By combining HyperMesh and LS-DYNA, engineers and analysts may construct rich finite element models and then run extremely accurate and efficient simulations, streamlining the simulation workflow.

In industries where safety, performance, and structural integrity are important factors, this integrated approach is very beneficial. In a broad range of applicable domains, it helps engineers to make well-informed design decisions, optimise products and processes, and ultimately improve safety, reliability, and performance.

5.2. REFERENCES:

1.) Design and Analysis of Radial Engine, Vaishnavi Barangule et. al.,
2.) Prof. Dr. I. Satyanarayan "Design and analysis of Radial engine".
3.) Prof. N.P. Doshi, "Analysis of Connecting Rod Using Analytical and Finite Element.
4.) Kuldeep B, "Analysis and optimization of connecting rod using Alfasic composites".
5.) Debbati Venkatesh, "Design and Thermal Analysis of Radial Engine"