

# Design And Analysis Of Kick-out Control Kit Stud

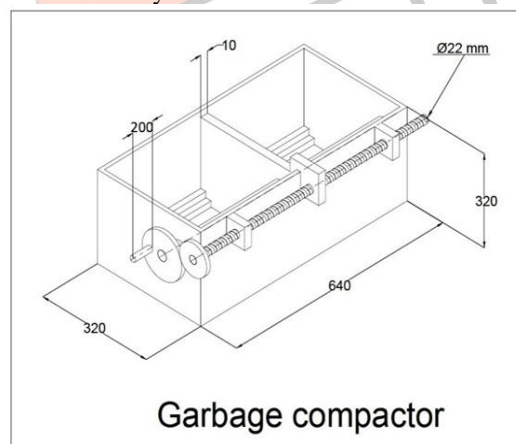
1Prem Dhang, 2Mrs. Shanthala N.l  
 1Student, 2Assistant Professor  
 1Bangalore Institute Of Technology, Bangalore,  
 2Bangalore Institute Of Technology, Bangalore

**Abstract** - directional control valve is a device that receives an external signal (mechanical, fluid pilot signal, electrical or electronics) to release, stop or redirect the fluid that flows through it. The function of a directional control valve is to control the direction of fluid flow in any hydraulic system. a directional control valve does this by changing the position of internal movable parts. D16 control valve having a capacity of 1 to 12 sections and oil rated flow of 250 l/min to 291 l/min. the application of valve in wheel loaders, Truck cranes, drilling machines and sea platform cranes, and garbage compactor. in this project work ,garbage compactor is considered and has a simple mechanism involving use of manual force to control the working of lead screw and gear arrangement system. a compactor consists of cylindrical block in which the garbage is collected. the compaction is performed through the action of gear arrangement and lead screw that is mounted upon the cylindrical shell which is operated by the simple hand lever. in this also consists of the moving plate which is attached to the lead screw by the nut. in garbage compactor the kick out control kit is used in the D16 valve. this control kit works under the pressure of 7bar.when the lever actuates, it may lock up below set of pressure or pressure may exceeds above it, then the lever will come to the neutral position. as this process continues the stud in the control kit failed due to necking around 50,000 cycles. to overcome this problem selection of different high strength material is required. the analysis is carried to check yield & ultimate strength of new material using ansys software. so that the material should run up to the 10,00,000 cycles.

**keywords** - kick-out control kit stud, finite element analysis, material selection, design optimization.

## I. INTRODUCTION

It is a machine or system utilized to diminish the size of material, for instance, waste material or bio mass through compaction. A trash compactor is regularly utilized by a home or business to lessen the volume of refuse it produces. A baler-wrapper compactor is frequently utilized for producing smaller and enveloped bundles by request to advance coordination. It is a straightforward component including utilization of manual power to control the working of manual worked lead screw and apparatus plan framework. It comprises of tube shaped square in which the trash is gathered from the individuals in the general public. The compaction is performed through the activity of rigging course of action and lead screw that is mounted upon the round and hollow shell which is worked by the straightforward hand switch. In this likewise comprises of the moving plate which is connected to the lead screw by the nut.



**Fig1:** Garbage Compactor

### D16 Directional Control Valve

The fig 2 shows D16 valve is accessible with manual, pressure driven remote, pneumatic, electro water powered, and electro pneumatic controls. Various setups and arrangements are conceivable. Working segments have assistant valves and a wide scope of tradable spools. Unique forms for LS variable siphons can be acknowledged on solicitation.

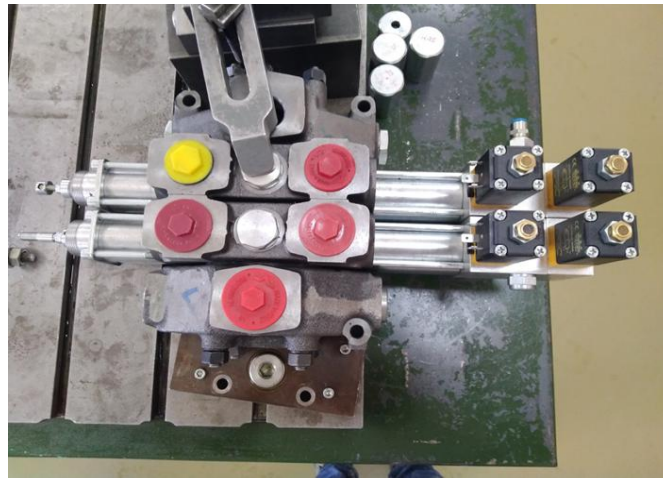


Fig2: D16 Directional Control Valve

**Technical specification of D16 valve**

- Working section number - 1-12
- Rated Flow - 150 L/min
- Spool stroke – 7+7 mm
- Circuit type –Parallel, Series, Tandem
- Rated Pressure – 350Bar

**3D Model of Kick-Out Control Kit**

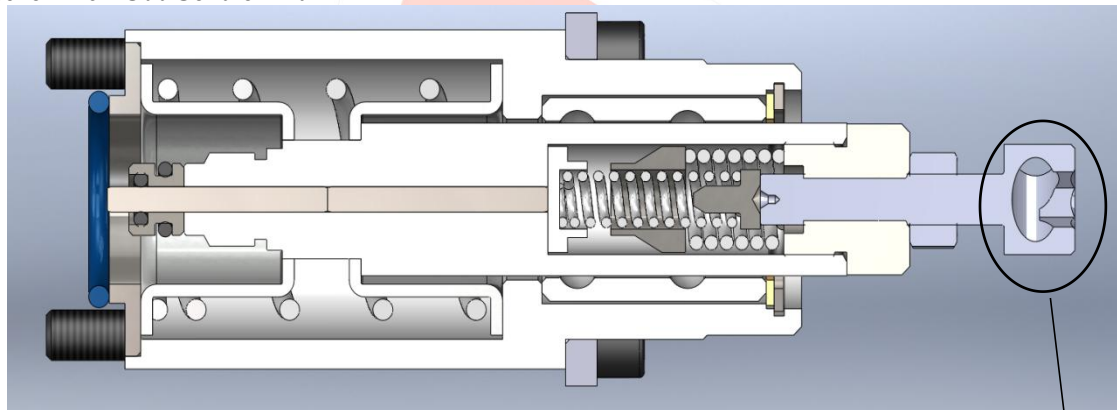


Fig 3: Assembly of Kick-Out Control Kit

ENIA Material Stud

In the fig 3 shows the assembly of kick-out control kit, It is connected to the D16 directional control valve to actuate lever for the flow of fluid.

**II. LITERATURE REVIEW**

**Dr.Santosh B (Prof. MechDept) G.H.Raisoni College of Eng. Nagpur: Failure analysis of Exhaust manifold stud of Mahindra tractor using finite element. [1]:** Tractor is most helpful hardware has significant offer in planting, holding, reaping tasks and after that in motorization part. Consequently so as to arrive at manageable horticulture machine and furthermore amount must be come to ideal level. Just couple of instances of break disappointment investigation of agrarian vehicles have been accounted for till date. In the disappointment of stud should do in FEA examination. Accordingly reproduction for cyclic pressure assessment and redistributions are basic for foreseeing crack, improving the precision of break life forecast of mechanical segments.

**Zhang Boshan Professor Tongji University, Shanghai, China Behaviour of stud Under Monotonic and cyclic loading based on FEM simulation [2]:** The steel solid complex organisation can provide full play to the substantialbit of leeway of steel and cement. So the steel cement is broadly utilized in structure development of extensions, modern structures and high rises. Steel solid composite structures are commonly jointed by stud connectors to move shear. Be that as it may, the examination about stud's mechanical presentation system still deficient. The investigation about the mechanical properties of broke stud is particularly deficient. Accordingly, steel solid composite structure can be absconded and venture the structure might be squandered.

**DOUG OATIS senior mechanical engineer, (Phonix Analysis and Design Technologies) [3]:** He issued about breaking down of Bolt demand in Ansys workbench. Dissecting a shot spine used to be a genuine under taking, to a limited extent on account of challenges in including demand burdens created by the establishment torque of fixing the jolt. Experts depended on an assortment of techniques to represent claim, including running a "sham" warm investigation to instigate warm development loads or making shafts and imperative conditions on the spine to include proportionate compressive rib loads.

Claim components accessible in the ANSYS Workbench stage enable the examiner to all the more promptly determine known hub burdens or changes in accordance with gatherings of components in representing these jolt establishment loads.

**MICHEL OLIVER, oliver testing and Consulting, USA [4]:** He Published about Modeling Threaded Bolted joints in Ansys Workbench. Dated joints are incredibly basic latches in development and machine structure. Anyway making a limited component model of a strung shot joint is a convoluted undertaking however certainly justified regardless of the exertion. The means are: create asolid model, make contacts, work the model and afterward set up limit conditions. Every one of these assignments has its own arrangement of difficulties and issues; these all effect the capacity of the product not exclusively to combine, however to give exact outcomes.

**BIST. BIHER (Asst.Professor,Department of Mechanical Engineering) [5]:** Published about analysis of stress square threaded power screws using Ansys. An example of torture test is used in analysis of stresses in power screw. Torture test is used in industries manufacturing screw jacks so that there is no misalignment or problem while using screw jack. One end of the shaft of the screw jack is fixed and flange is given a torque. The model is drawn in pro E. Then the model is transferred to ANSYS. The stresses between the flange and the shaft are analyzed using ANSYS. The analytical value of stresses are found out and compared to the stresses found out using ANSYS. The result is shown in a tabular format.

**Hemant K. Wagh Assistant Professor, Mechanical Engineering Department,RCPIT, Shirpur, MS, India [6]:** Claim burden assumes significant job in modern gatherings so as to stay away from mishap by relaxing of congregations. The point is to decide most extreme breaking point of claim load without string disappointment utilizing all inclusive testing machines. The vibration test apparatus is created for estimating the slackening execution of nut fastener gatherings. The examples of ISO Metric electrical discharges size, for example, M8, M10, M12, M16 and M20 jolt having 1.25, 1.5, 1.75, 2 and 2.5 mm pitches individually are utilized for testing.

**Jianzhong Zhou, School of Hydropower and Information Engineering, Huazhong University of Science and Technology, Wuhan 430074, China [7]:** This paper exhibits an examination on the unfortunate vibration of an angled stud stator outline in an enormous hydropower generator by methods for trial and limited component (FE) investigation. In the first place, field trials were accomplished, and the outcomes demonstrate that the primary vibration segment originates from electromagnetic elements. At that point, a 2D-attractive and 3D-mechanical FE model was created to research the vibration of the stator outline under the activity of electromagnetic powers. A lot of complexity models was set up to examination the impacts of various types of capriciousness and various structures. In view of the relative examination between the aftereffects of recreations and exploratory tests, it tends to be surmised that the anomalous vibration is produced as a result of the absence of firmness in the upper piece of structure and the presence of dynamic unconventionality in the rotor-stator framework.

**Prof. N.P.Doshi,(Department of Mechanical Engineering, BapuraoDeshmukh College of Engg, Sewagram, India) [8]:** The associating pole is a significant connection within a burning motor. It interfaces the cylinder to the crankshaft and is answerable for moving force from the cylinder to the crankshaft and sending it to the transmission. There are various kinds of materials and generation techniques utilized in the production of interfacing poles. The most widely recognized kinds of materials utilized for associating poles are steel and aluminum. Interfacing poles are broadly utilized in assortment of motors, for example, in-line motors, V-motor, contradicted chamber motors, spiral engines and limit chamber engines. For the endeavor work we have picked partner bar used in light business vehicle of farewell motors had starting late been impelled in the market. We utilized PRO-E out of control fire 4.0 programming for demonstrating of interfacing pole and ANSYS 11 programming for examination. ANSYS Workbench component had been utilized for examination of associating pole.

**Selvaraji Muthu (Mahle Engine Components India PVT.LTD) [9]:** There is a developing interest for a wide range of screw blowers in the business because of client necessities. Structure and development of screw blowers are requesting undertakings that require propelled estimations and hypothetical learning. The clearances in screw blowers assume a significant job in execution and unwavering quality. Screw blowers work with tip accelerates to 100 m/sec and the release temperature up to 250 °C. A hypothetical methodology was required so as to limit the clearances while keeping away from contact between the rotors and the packaging. It was built up to ascertain the clearances precisely by thinking about the basic and warm distortions. It is fundamental to fuse the Computer Aided Engineering in the Screw Compressor structure and advancement procedure to approve the hypothetical model. Investigation of the pressure and diversion brought about by outer power and weight, examination of the warm pressure and distortion brought about by warmth move has been performed in the screw blower rotors, rotor and bearing lodgings by utilizing Ansys.

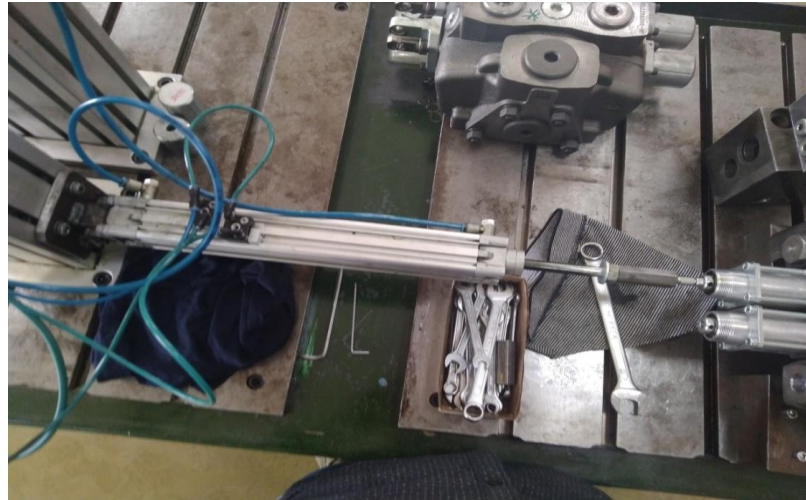
### III. PROBLEM DEFINITION

Kick out control kit having set of Pressure about 160 bar. In the garbage compactor the compressor compresses the drainage beyond this pressure, then the lever will be kick back to the neutral position. While testing of Kick-Out control kit in the Pneumatic cylinder the stud was failed about 50,000 cycles.

#### Testing of Stud

Below Fig 3 is shows the setup of testing stud in Double acting Pneumatic cylinder. The Pneumatic cylinder has following specifications

- i. Pressure applied- 7bar
- ii. Piston diameter- 32mm
- iii. Stroke length- 225mm



**Fig4: Pneumatic Cylinder**

**Finding compressive load and tensile load of stud**

**i. Compressive load of the stud.**

a)  $F = P * 3.142 * (d1^2 - d2^2) / 4 \dots\dots N$  (a)

Where,

P=Pressure (Bar) =7bar

d1=Full bore piston diameter (meters) =0.032m

d2=Piston diameter (meters) =0.0121m

$F = (7 * 10^5) * 3.142 * (0.032^2 - 0.0121^2) / 4$

$F = 483N.$

**ii. Tensile load of the stud.**

$F = P * 3.142 * (d1^2) \dots\dots N$  (b)

P=Pressure (Bar) =7bar

d1=Full bore piston diameter (meters) =0.032m

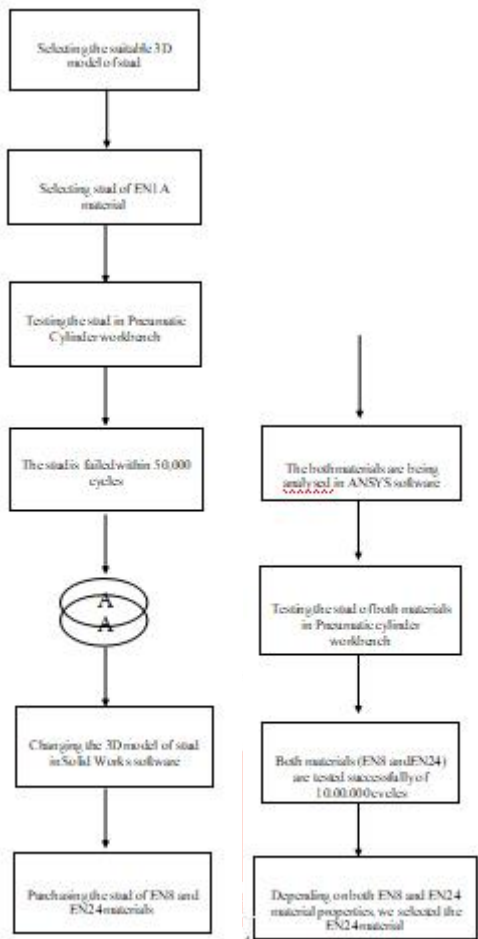
$F = 7 * 3.142 * (0.032^2)$

$F = 563N$

The above equation a and b are compressive load 483N and tensile load 563N respectively. These loads are applied in boundary conditions while making analysis in ansys software.

**IV. METHODOLOGY**

**Block Diagram**



**Material Selection**

Here for stud EN1A, EN8 and EN24 materials are selected because less in cost, more durability, high yield and tensile strength.

**Properties of EN1A, EN8 and EN24 Materials**

**Table 1:** Properties of EN1A, EN8, and EN24 Materials

Materials	Yield strength	Ultimate strength	Poisson's ratio	Young's modulus
EN1A	240MPa	370MPa	0.3	200e3MPa
EN8	465MPa	700MPa	0.3	200e3MPa
EN24	650MPa	(850-1000)MPa	0.3	207e3MPa

**V. MODELING AND FINITE ELEMENT ANALYSIS**  
**3D Model of EN1A Material Stud**

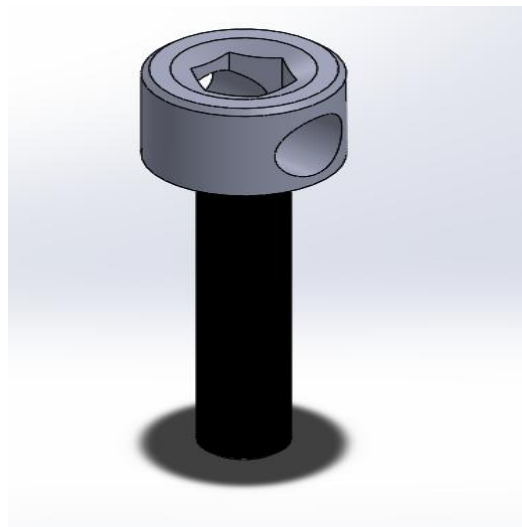


Fig 5: 3Dmodel of EN1A Stud

**Meshing**

CAD model of part converted into STEP file. This model is imported into Ansys Workbench simulation. Geometry cleanup was performed prior to meshing of model. For better quality of mesh fine element size is selected.

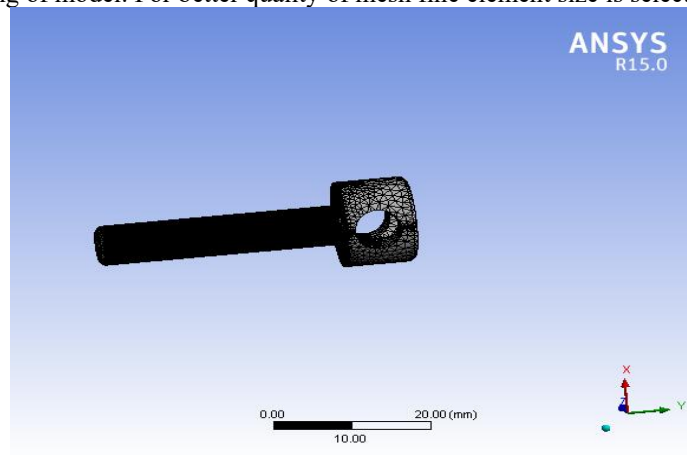


Fig 6: Meshing of EN1A Stud

**Boundary Conditions**

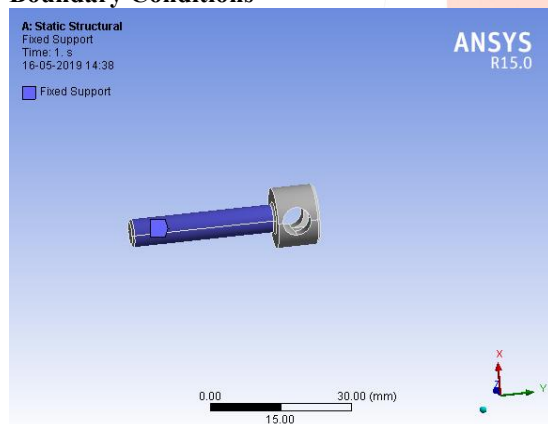


Fig 7: Fixed Support of stud

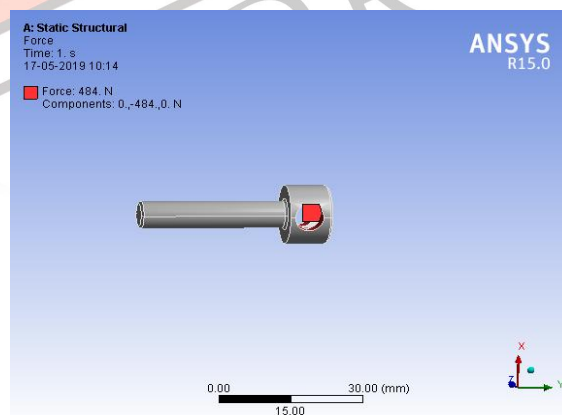


Fig8: Loading Condition of Stud

In static analysis we have to consider the force acting on the component and boundary it means where the component is stationary. By observing the displacement contour the maximum deformation at the small eye end or in small hole and here the blue colour contour indicates the minimum displacement and red colour indicates the maximum.

**Equivalent stress, Factor of Safety, and Fatigue life of EN1A material Stud**

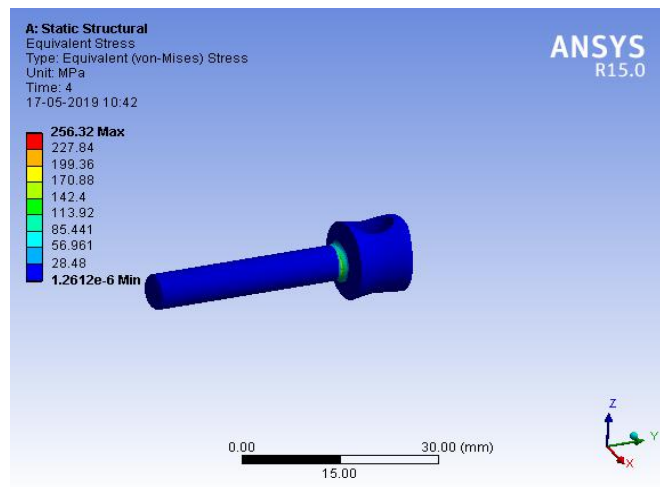


Fig 9: Equivalent Stress of stud

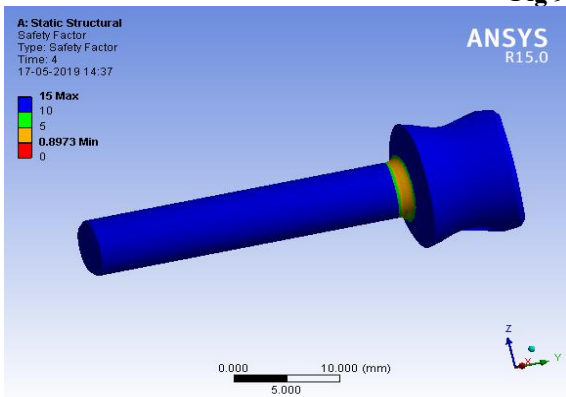


Fig 10: Safety Factor of Stud

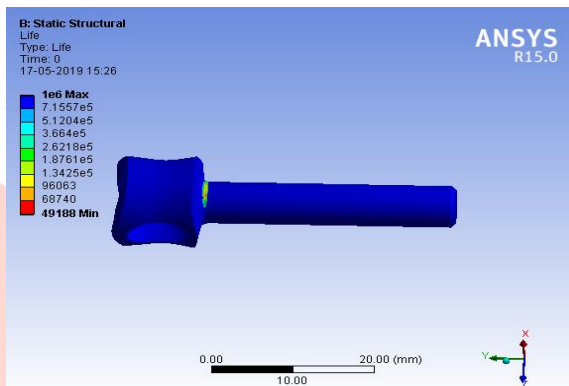


Fig 11: Fatigue life of Stud

- The EN1A material stud has Equivalent stress about 256.32MPa, where it is above the yield limit(Refer table 1) then the material will get fail as shown in fig 9.
- In the fig 10 shows the Stud has 0.8973 Safety Factor, which is not safe. The Safety Factor value should be equal or above 1
- In the fig 11 shows Fatigue Life of EN1A stud has 49188 cycles, which is not safe.

**Results and Discussion**

Material	Equivalent Stress	Safety Factor	Fatigue Life
EN1A	256.32MPa	0.8973	49188cycles

Table 2: Results of EN1A Material

In the above table 2 shows that the Equivalent stress is 256.32MPa which is more than the yield strength of the EN1A Material, Safety Factor is 0.8973 and Fatigue life is 49188 cycles. Which is not safe in theoretically, So we will go for change of 3D model.

**New 3D Model of EN8 and EN24 materials Stud**



Fig 12: New 3D Model of Stud

**Meshing**

CAD model of part converted into STEP file. This model is imported into Ansys Workbench simulation. Geometry cleanup was performed prior to meshing of model. For better quality of mesh fine element size is selected.

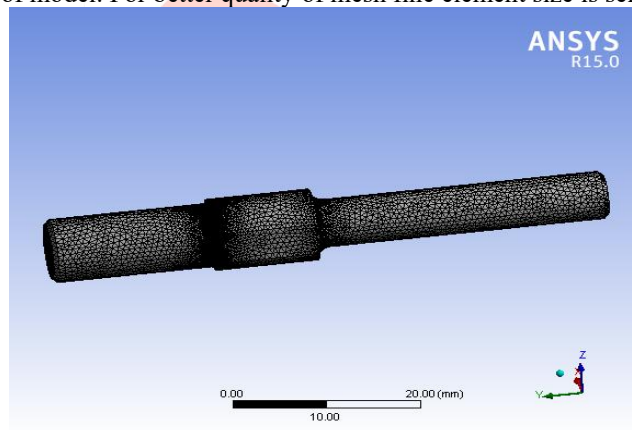


Fig 13: Meshing of New Stud

**Boundary Conditions**

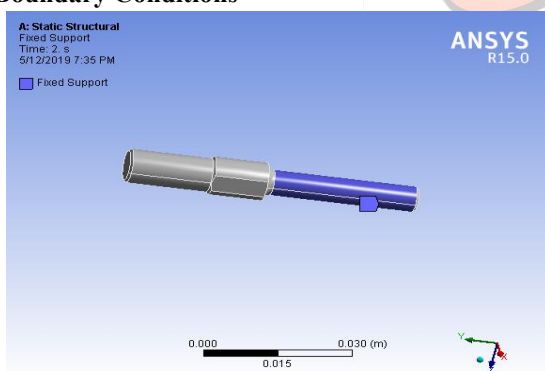


Fig 14: Fixed Support of Stud

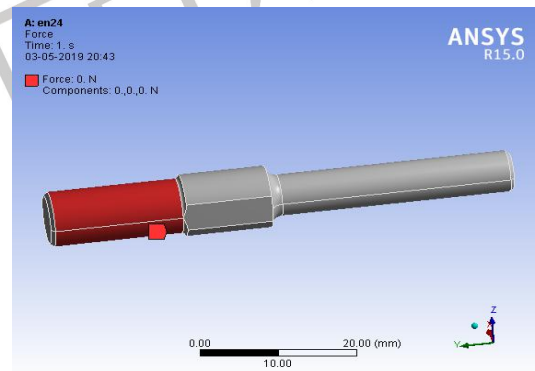


Fig 15: Loading Condition in Stud

In static analysis we have to consider the force acting on the component and boundary it means where the component is stationary. By observing the displacement contour the maximum deformation at the one end and here the blue colour contour indicates the minimum displacement and red colour indicates the maximum.

**Equivalent Stress, Factor of Safety and Fatigue life EN8 and EN24 materials stud**



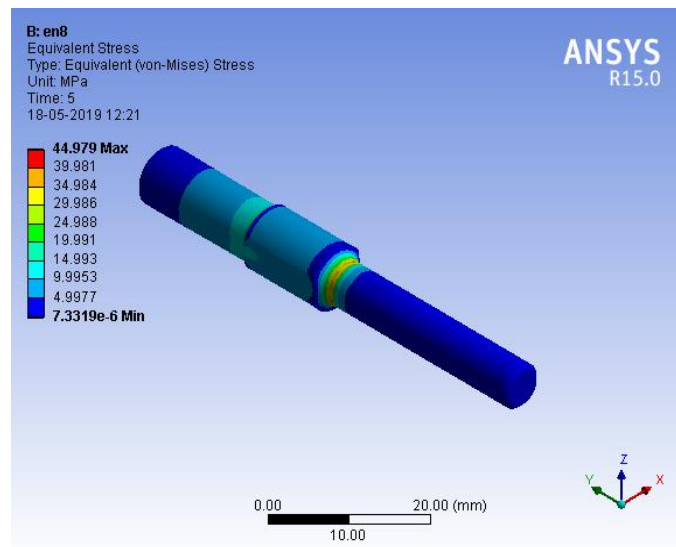


Fig 16: Equivalent stress of EN8 Stud

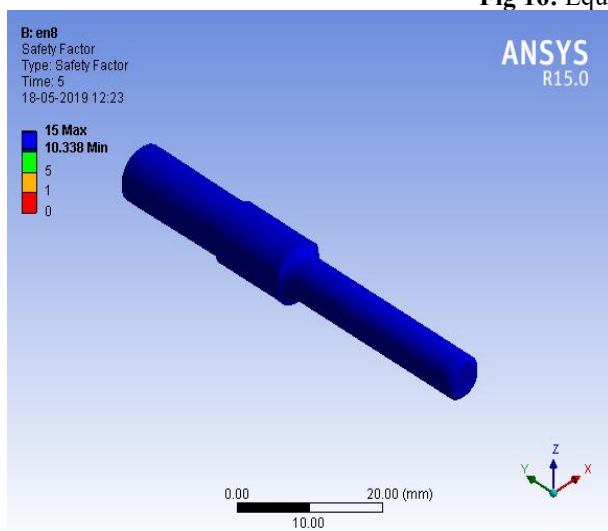


Fig 17: Safety Factor of EN8 Stud

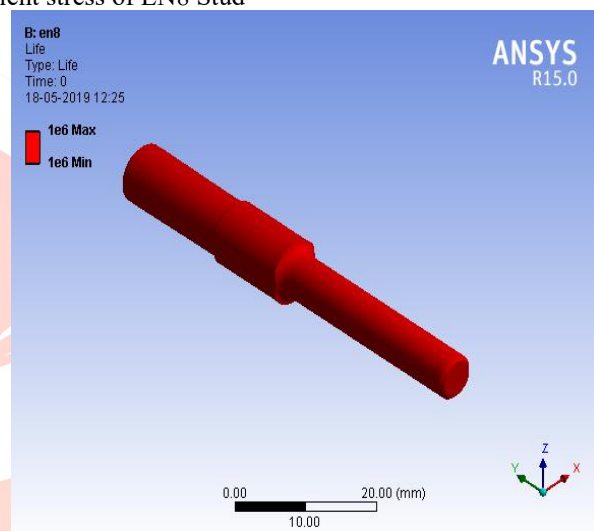


Fig 18: Fatigue Life of EN8 Stud

- Equivalent stress of EN8 Material stud is 44.979MPa which is below the yield limit(Refer Table 1) as shown in fig 16.
- Safety factor of EN8 material Stud is 10.338, So which is safe in design as shown in fig 17.
- Fatigue life of EN8 material stud is minimum of 10,00,000 cycles, So which is safe in design as shown in fig 18.

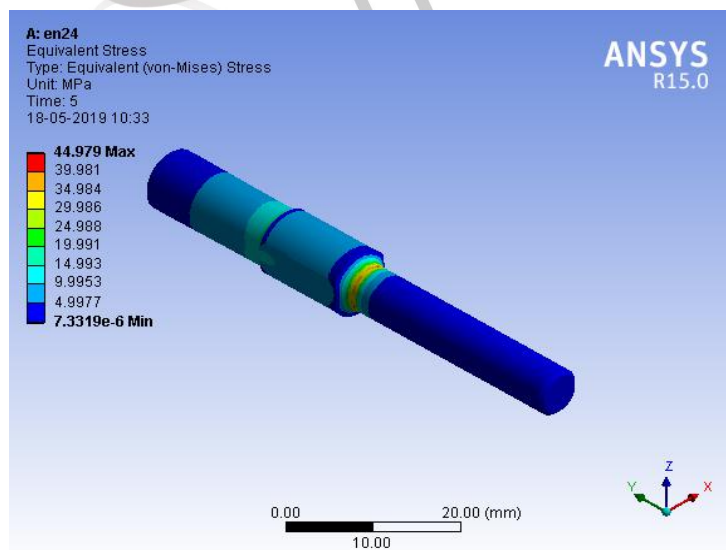
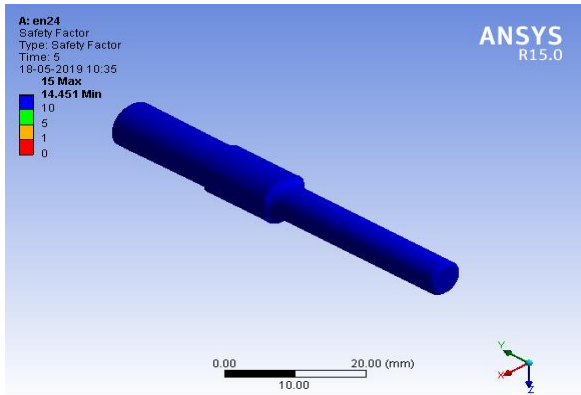
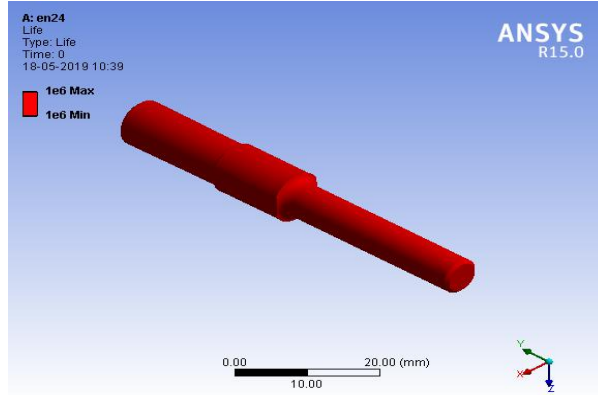


Fig 19: Equivalent Stress of EN24 Stud



**Fig 20:** Safety Factor EN24 Stud



**Fig 21:** Fatigue Life of EN24 Stud

- Equivalent stress of EN8 Material stud is 44.979MPa which is below the yield limit(Refer Table 1) as shown in fig 19.
- Safety factor of EN8 material Stud is 14.451, So which is safe in design as shown in fig 20.
- Fatigue life of EN8 material stud is minimum of 10,00,000 cycles, So which is safe in design as shown in fig 21.

**Results and Discussion of EN24 and EN8 Materials**

**Table 3:** Results of EN8 and EN24 Materials

Materials	Equivalent Stress	Safety Factor	Fatigue Life
EN8	44.979MPa	10.338	1e6 Cycles
EN24	44.979Mpa	14.451	1e6 Cycles

In above table 3 Equivalent stress, Safety factor and fatigue life of both EN24 and EN8 materials are safe in design. But as compare to EN8, EN24 has better safety factor. So we will select EN24 material.

**VI. CONCLUSION**

In this Analysis work, the effect of tensile load and compressive load in the stud causes the failure, by overcoming these failures we have changed the 3D model of the stud.

The first 3D model of stud is made up of with EN1A material and Second New 3D model of Stud is made up with EN8 and EN24 Materials. As compare to EN1A which have the better material properties of EN24 and EN8.

In this analyses work we found out Equivalent Stress, Factor of Safety, and Fatigue Life.

Among these two materials(EN1A and EN24) to select the EN24 is good because which has high safety factor as compare to other material and also having high yield and ultimate stress.

After the analyse the EN24 stud is tested in the Pneumatic cylinder, where it ran 10, 00,000 cycles successfully.

**Scope for Future Work:**

- In this project fatigue life analysis is carried out for constant amplitude, but variable amplitude loading conditions can be applied for further investigations.
- Optimization for the present work can be taken as separate project in future.

**VII. REFERENCES**

1. James R.Dale “Connecting Rod Evaluation” ASME Journal of Mechanical Engineering January 2005, Vol-123.
2. Mansour Rasekh, Md. Reza Asaid, Ali Jafari, Kamran Kheiralipour “Obtaining maximum stresses in different parts of tractor (Mf-285) connecting rods using Finite element method” Australian Journal of Basic and Applied Sciences,3(2): 1438-1449,2013.,ISSN 1991-8178.
3. Santosh B. Jaju “Failure analysis of Exhaust manifold stud of Mahindra tractor using Finite Element Analysis” IEEE Journal of Mechanical Engineering, 2011.
4. The Focus(www.padtinc.com/epubs/focus) from engineering consulting firm PADT(Phoenix Analysis & Design Technologies).
5. Moliverconsulting.com
- 6(i).Arun Kumar N., Srinivasan V., Krishna Kumar P., Analysing the strength of unidirectional fibre orientations under transverse static load, International Journal of Applied Engineering Research, v-9, i-22, pp-7749-7754.
- (ii)Srinivasan V., Optimizing air traffic conflict and congestion using genetic algorithm, Middle - East Journal of Scientific Research, v-20, i-4, pp-456-46.
- 7.Shinji Kasei “Study of self-loosening performance of bolted joints due to Repetition of small amount of slippage bearing surface” Journals of advance mechanical Design system and manufacturing. Vol 1 p 358-367.
- 8.Lashkari, N.; Poshtan, J.; Azgomi, H.F. Simulative and experimental investigation on stator winding turn andunbalanced supply voltage fault diagnosis in induction motors using Artificial Neural Networks. ISA Trans., 59, 334–342.
- 9.FEM analysis of connecting rod by R.Vozenilek,C.Scholz (The Technical University of Liberec,Halkova ).

