IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X
PP. 15-25
www.iosrjournals.org

"Nonlinear Contact Pressure Analysis of Oil Sump Using FEA & Experimental Method"

Amol Nandgave[1], A.S.Patil[2]
[1](Assistant Manager, Bharat Forge Ltd, Baramati, Pune India)
[2](Asst. Professor, Mechanical Department, JSPM NTC College/Pune University, India)
Corresponding Author: Amol Nandgave

Abstract: *Incidents of fluid leaking from a joint containing an oil seal is one which many engineers have encountered. The use of an oil seal in a joint to prevent leakage is only effective as long as there is sufficient clamp force generated by the bolts to allow effective sealing. If insufficient pressure is applied to the oil seal then in such regions, leakage can result. In this oil sump paper gaskets have used for sealing when clamping pressure is applied. Without knowing the standard values of torque, installation is done then installation can be found problematical. And due to vibrations caused in vehicles also reduces the torque and may cause to loosing of bolt which further cause to oil leakage. So in this paper, by applying different torque range to check leakage & find out optimum torque, where oil leakage do not occurs by FEA and experimental method.*
Keywords : *Non linear contact pressure, sliding deformation, Torque, FEA.*

I. Introduction

Now a day's oil leakage from oil sump is observed in high range, from engine oil sump. This may cause to engine jamming problem, further its gets highly maintenance problem, also can cause the problem of less efficiency by an engine. The sump is attached to the bottom of the cylinder block underneath the crankcase. The functions of the sump are to store the engine's lubrication oil for circulation within the lubrication system. To collect the oil draining from the sides of the crankcase walls and if ejected directly from the journal bearings. To give a short period of time to recover its temperature (to cool the oil) after oil once has circulated in engine. The sump may be made from a single sheet-steel pressing or it may be an aluminum-alloy casting with cooling fins and strengthening ribs. Both the constructions have a flanged joint face, which matches with a corresponding joint face on the underside of the crankcase. A soft flexible gasket is provided in between to seal the joint and is tightened down by set-screws. The sump generally has a shallow downward slope at one end, which changes into a relatively deep but narrow walled reservoir at the other end. The incoming oil flows towards the deep end, where it submerges the pick-up pipe and strainer of the lubricating system. A drain plug is located at the lowest level in the sump for easy drainage of used oil.

One only really notices the oil-sump when it's not function properly or when driving in very uneven road it came into contact with the ground, which is unseen now a days because the entire underside of the car is paneled or covered. So because of the special underside protection, it almost never occurs leakages caused due to uneven roads. There may also be something wrong with the sealing to the engine block when the mechanic may have done something wrong work.

It is pretty unsuitable point to jack up the engine by applying jack to the oil sump. There should be a suitable fixture which reaches around the oil-sump and is mounted to a fastening flange. If the sump is made of pressed sheet metal, it can easily be dented, possibly also causing it to get damage and making a good seal unlikely. More expensive oil-sumps made of Al-Si alloy can crack, or even break when wrongly handled.

II. Literature Survey

"Design Optimization of Oil Pan using Finite Element Analysis, Kandru Kamalakar, Talada Suresh Prakash, Mechanical Engg.Department, Swami Vivekananda Engineering College, Bobbili, A.P., India."

Over the previous years an increasing attention has been paid to vibration and noise control in automotive engineering. The control of noise and vibration is essential in the design process of an automobile, since it contributes to the comfort, efficiency and safety. Considering trucks, the power train represents one of the main noise sources. The major contributor to the power train noise emission is the engine oil pan. The purpose of this literature survey paper is to design a truck oil pan for vibration reduction using numerical simulations.

In the analyses Finite Element Method (FEM) is applied to model the structural behavior of the oil pan. Simultaneously there are static load and dynamic load acting on a vehicle structure. The current work explores the effects of pre-stress forces on modal parameters of oil pan. The harmonic response analysis of pre-stressed oil pan is one of the key objectives in this literature paper. The harmonic response analysis of pre-stressed oil pan using ANSYS has been explored. FE simulations of the oil pan are presented, which are aimed to indentify the most dominant mode shapes within a frequency range of 0-1200 Hz. First pre-stress modal analysis is performed using block lanczo's method on the oil pan and then harmonic analysis of the pre-stressed oil pan is completed using full harmonic method. Based on the results obtained, efforts are made to optimize the design of oil pan to reduce the resonance effect caused due to vibrations. NX-CAD software is used to design the oil pan.

A truck oil pan was designed and analyzed for vibration reduction. Finite element analysis was done to model the structural behavior of oil pan. Both static and dynamic loads were considered for the analysis. In this literature paper the effects of pre-stress forces on modal parameters of oil pan and harmonic response analysis of pre-stressed oil pan using ANSYS has been performed. First pre-stress modal analysis was performed using block lanczo's method on the oil pan and then harmonic analysis of the pre-stressed oil pan was done using full harmonic method. Based on the results obtained, it was observed that the original oil pan was not safe for the operating loads. Later design changes were implemented to increase the stiffness of the oil pan. From the FE simulation results of the modified oil pan, it is concluded that the modified oil pan is safe for the mentioned operating loads [2]

"Design of Oil Sump using Coupled Eulerian – Lagragian (CEL) Technique, By A. Vishnu Prasanna Moorthy"

The performance of an IC engine has an ionic bond and with the performance of lubricating oil system. Oil sump and Oil Suction Pipe is one such assembly. The oil sump is acting as partially filled reservoir of lubricating oil. The oil pump sucks the oil through the oil strainer which is immersed inside the oil sump. Design of the oil sump involve designing for capacity, engine tilt angles, layout, and NVH. This paper deals with design of oil sump for engine tilt angles. When the vehicle is cornering / moving on roadways with inclinations, a set of complicated inertial and gravitational forces act on the lubricating oil. Due to these forces, the oil is pushed towards oil sump sides depending on the resultant of the above forces. This resulting oil movement may have serious implications on the engine performance, if the oil strainer is not immersed in oil and the pump sucks air into the lubricating oil system Since the problem is highly non linear and considering the importance of this design aspect, the verification has been attempted using Fluid-structure interaction capability in Abaqus Explicit. Coupled Eulerian- Lagragian technique has been used and the oil was modeled using Equation of State (EOS) – Mie Gruneisen form. The oil movement in dynamic tilting condition has been visualized and verified that sump provides oil to engine at all required tilting conditions.

The model setup described in this is from the final iteration. But previous iterations consist of various modifications in the setup. They are described as below:

Since the sump is a sheet metal part with 1.5mm thickness, initially it was analyzed as shell model. After analysis it was found that lot of Eulerian material penetrations were happening through the contact interface. This was primarily due to the reason that outward normal directions of all shell elements in a model are not unique. So the model setup was changed by modifying the CAD model with 10mm thickness and analyzed as solid model. The solid model as they have unique outward normal direction the penetrations across the interface were reduced to a good extent.

Initially iterations were carried out with seed size of 15 for Eulerian model as the computation time was very high. (9 hours 40 minutes with 8 processors) using this setup, the penetrations were more and loss of material from the boundary was high. Then a seed size of 10 was used. This increase in mesh density has increased the analysis time to 23 hours and 20 minutes with the same domain level parallelization. This has greatly reduced the material loss. This can be found out from the history output for Total energy (ETOTAL). This in general has to be constant for accurate analysis. But when there are oscillations in the system due to improper time/mass scaling, Total energy will not be constant and when there is material loss from the domain, it will further change to a negative quantity.

In actual condition it will take approximately 5 -7 s for 30° tilting. But performing this analysis for such time period will take extremely high amount of increments (>> 20,00,000) and computation time will run into days. So computation time can be reduced by using time scaling. The consequence of such time scaling will be extremely high accelerations and sloshing effects. The final iteration is performed with 2 seconds. It took 4,91,204 increments to complete the analysis. It is relatively low compared to the estimated one for actual tilting (as indicated above) and also the reason for choosing double precision Abaqus/Explicit executable. The acceleration of the one of the nodes in the Eulerian model (oil) at the end of the step.

The seed size of Eulerian model can still be reduced from 10 to 5 to reduce the material loss and to get constant Total energy. But the present model gives fair results and can be relied upon.

However special method called "Selective subcycling" must be used if seed size is reduced to 5. In this analysis model, as automatic time incrementation is used, the minimum increment is based upon the critical element. From the message file of various iterations, it has been observed that only few elements in Eulerian model those near the sump wall boundary /interface are found to be critical elements. Therefore using "Selective subcycling", solver uses small time increments for those critical elements only and large increments for the other elements. By using this process the total computation time is reduced drastically. To use this, node sets has to be created as per according to it. With the time scaling, compared to the boundary conditions the accelerations experienced are the worst case. Even with this condition, in static circle, the oil doesn't reach oil seals and the trigger wheel and in dynamic circle, the strainer is found to be completely immersed in oil. So the design is found to be safe [3]

III. Objective

To find the minimum contact pressure where the leakage from oil sump not occurs and to find the minimum torque so that leakages should not occurs. Also will find the contact pressure, contact status, deformation and von-misses stresses by using experimental and FEA analysis.

IV. Experimental Setup

Fig.1; Oil sump used for experiment

Fig.2; Torque wrench

Fig.3: compressed air filling in oil sump

Fig. 4: Oil leakage after filling compressed air

Oil sump used for project is of TATA vehicle Truck which is shown in figure 1. The oil sump is having two parts which are joined with bolted joints. As per the standard sheet of bolt M8 bolts is used, with having quantity in 8 numbers, by using gas-kit in between them. For experimental purpose added a valve (used to fill the air in tiers) at the top of the oil sump to fill the compressed air.

To apply the load on bolt, torque wrench is needed, here the wrench taken is screw type torque wrench is used to apply torque on bolts of oil sump (as shown in figure 2). The toque wrench used is in Ft-lb, so converted the torque from Nm to Ft-lb (as shown in table 1).

As going to find the oil leakage from the oil sump by using different torques, we need a compressed air to be filled in oil sump as shown in figure 3, so we have added the valve externally.

Experimental procedure:

First of all to fill the oil in oil sump, then cover the both parts by adding gas-kit in between them and apply the bolt on it, all the bolts should keep loose, and then have to apply the torque on all bolts equally. After applying torque to all bolts, fill the compressed air through the valve which is added for experimental purpose.

If the bubbles with oil comes out from the joint parts then it is said that at that torque oil leakage occurs, and if air with oil bubbles not come through the joint then it said that at particular torque leakage is not there. Then by reducing the torque do the same procedure till the oil gets leakages. At particular torque the oil will gets leakage, which shows that the minimum torque where the leakage is not found.

Cases of experimental readings, which shows what percent of torque is going to apply as compared to standard given torque:

CASE 1: In this case going to apply 100% load w.r.t. standards (ie; 14060 N).
CASE 2: In this case going to apply 90% load w.r.t. 100% load (ie; 12654 N).
CASE 3: In this case going to apply 80% load w.r.t. 100% load (ie; 11248 N).
CASE 4: In this case going to apply 75% load w.r.t 100% load (ie; 10545 N).
CASE 5: In this case going to apply 70% load w.r.t. 100% load (ie; 9842 N).
CASE 6: In this case going to apply 50% load w.r.t. 100% load (ie; 7030 N).

By applying different torque will find out at what percent load, oil will get leakage. Also with this different load cases will find out Contact pressure, Deformation, VON MISES stresses and contact status by using ANSYS.

V. Design And Analysis

A. Material Specification:

The material of the specimen is having the following composition:

Oil sump	: TATA truck
Bolt used	: M8
Material	: Steel
Modulus of Elasticity	: 200 GPa
Poisson's Ratio	: 0.30
Density	: $7.85e^{-6}$ kg/mm^3
Yield Strength	: 520 MPa

B. CAD MODEL:

To prepare the CAD model of specimen ANSYS 15.0 Design modeler is used.

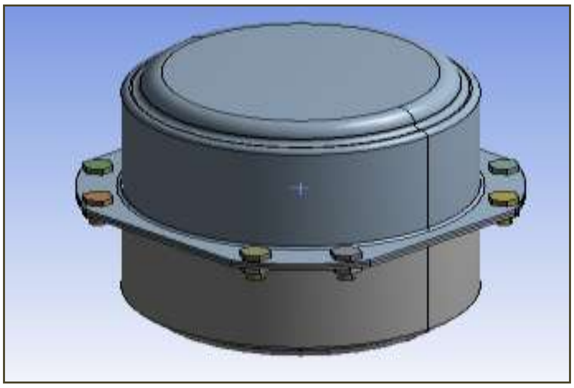


Fig 5: CAD Model of specimen

C. Discretization or Meshing:

For the discretization of model a hexahedron element and tetrahedron element with solid program controlled mesh is used. Where node population count is 118938 node and element population count is 59692 element.

Fig 6: Discretization of specimen

D. Boundary Condition & Loading:

Boundary conditions are fully constrained at upper surfaces. And the three load cases are taken which are as shown below. Preloading specifications:

- Metric thread= M8
- Initial tensile stress=0.6
- Thread coefficient Of friction =0.15
- Head face coefficient of friction=0.15
- Total tightening torque =22.62 Nm
- Initial preload (F_i)=14.06 KN

E. Standard specification for M8 bolt:

Metric thread :	M8 - 8.8	
Initial tensile stress : σ_t =	0.6	$\cdot R_{p0.2}$
Thread coeff. of friction : μ_{thread} =	0.15	
Head - face coeff. of friction : μ_{head} =	0.15	
Solve Reset Print		
pitch P	1.25	mm
pitch diameter d_2	7.188	mm
root diameter d_3	6.466	mm
tensile stress area $A_t = \pi/4\ d_0^2$; $d_0=(d_2+d_3)/2$	36.61	mm^2
ultimate tensile strength R_m	800	MPa
yield strength $R_{p0.2}$	640	MPa
tensile stress $\sigma_t = \sigma'_t \times R_{p0.2}$	384	MPa
torsional stress $\tau = M_G / (\pi/16\ d_3^3)$	219.64	MPa
equivalent stress $\sigma_e = (\sigma_t^2 + 3\tau^2)^{1/2}$	540.53	MPa
thread friction M_G	11.66	Nm
thread friction M_{G-}	-5.9	Nm
head face friction $M_{WD} = F_i\ \mu_{head}\ 1.3\ d/2$	10.96	Nm
Total tightening torque $M_A = M_G + M_{WD}$	22.62	**Nm**
Initial preload $F_i = \sigma_t \cdot A_t$	14.06	**kN**
Load at Yield $F_{02} = R_{p02} \cdot A_t$	23.43	**kN**
Load reserve $P_b = F_{0.2} - F_i$	9.37	**kN**

1) stress by torsion is relaxed after tightening.
2) equivalent stress should remain below the yield strength (linear elastic).

www.tribology-abc.com

F. Torque wrench values in foot pound and settings:

TORQUE	Nm	Ft-lb
100	22.62	16.7
90	20.358	15.04
80	18.098	13.35
75	16.965	12.52
70	15.834	11.67
50	11.31	8.64

Table: 1 conversion of torque from Nm to Ft-lb

G. BOUNDRY CONDITIONS:

LOAD CASE 1: *14060N (100 PERCENT PRELOAD)* LOAD CASE 2: *12654N (90 PERCENT PRELOAD)*

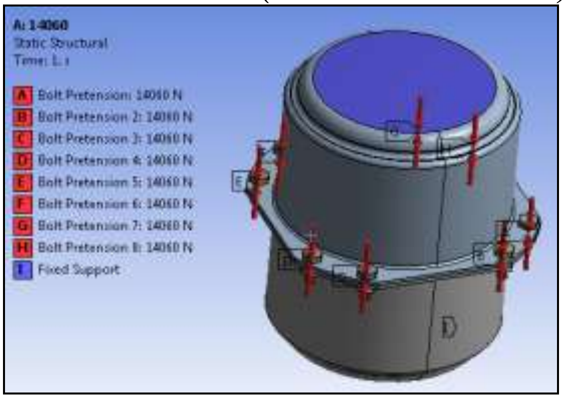


Fig.7; Boundary condition for load 100 %

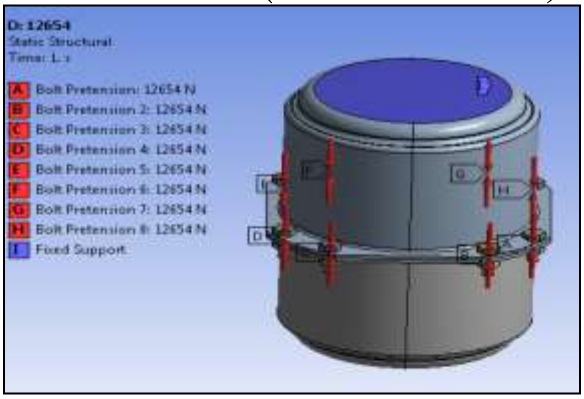


Fig.8; Boundary condition for load 90 %

LOAD CASE 3: *11248N (80 PERCENT PRELOAD)*

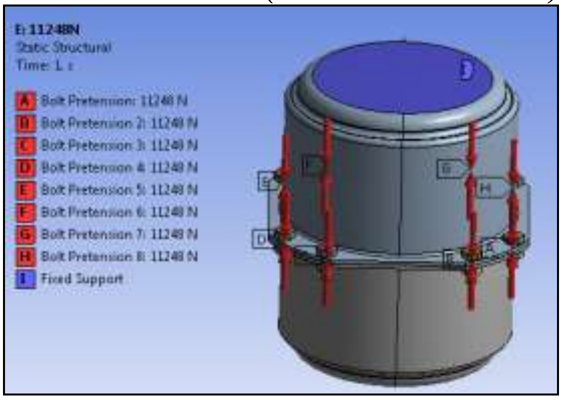


Fig.9; Boundary condition for load 80 %

LOAD CASE 4: *10545N (75 PERCENT PRELOAD)*

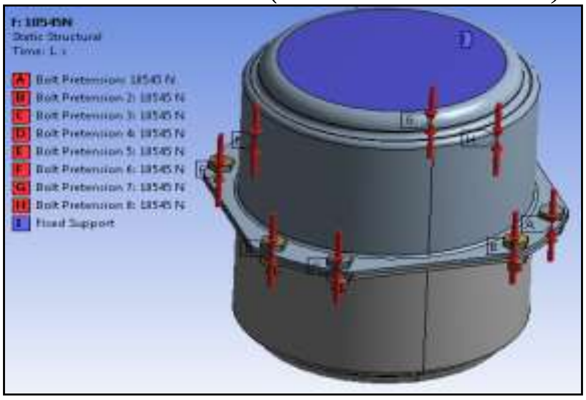


Fig.10; Boundary condition for load 75 %

LOAD CASE 5: *9842N (70 PERCENT PRELOAD*

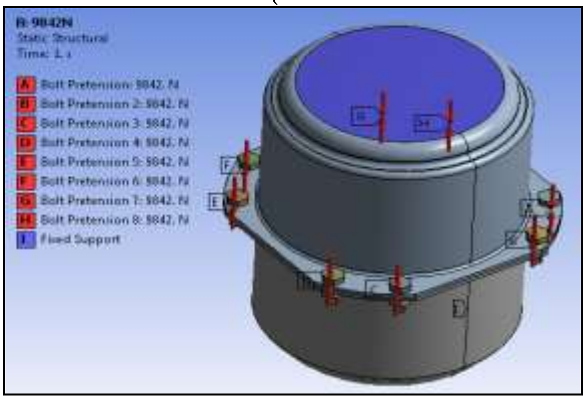


Fig.11; Boundary condition for load 70 %

LOAD CASE 6: *7030N (50 PERCENT PRELOAD)*

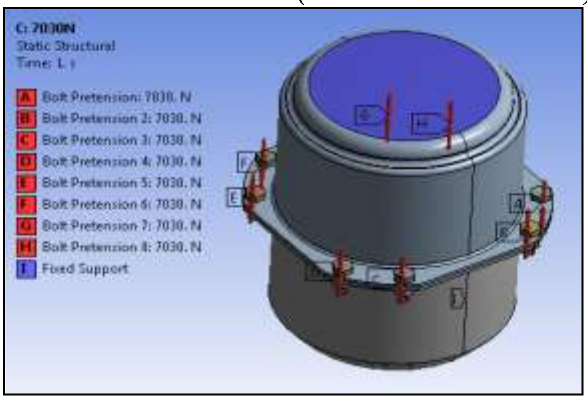


Fig.12; Boundary condition for load 50 %

Boundary conditions are taken for all loading conditions are same, and they are as fully constrained at the upper surfaces. (As shown in fig. 7,8,9,10,11 & 12)

VI. Finite Element Analysis

DEFORMATION:-

LOAD CASE 1- *14060N (100 PERCENT PRELOAD)*

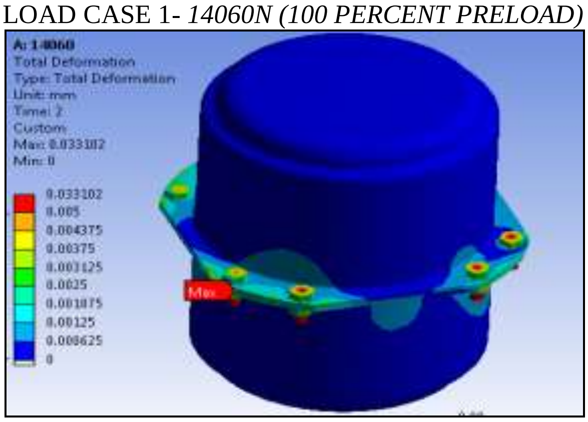


Fig.13; Deformation in 100 % load

LOAD CASE 2: *12654N (90 PERCENT PRELOAD)*

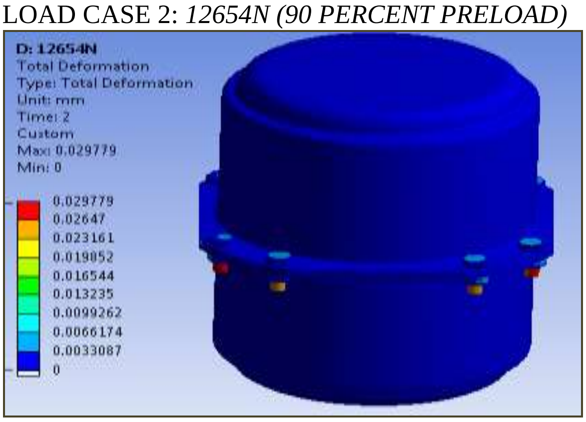


Fig.14; Deformation in 90 % load

LOAD CASE 3: *11248N (80 PERCENT PRELOAD)*

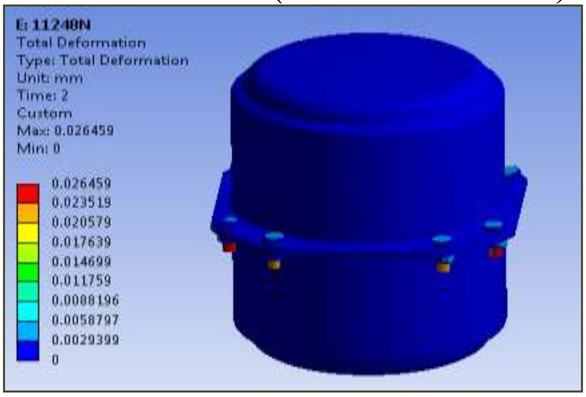


Fig.15; Deformation in 80 % load

LOAD CASE 4: *10545N (75 PERCENT PRELOAD)*

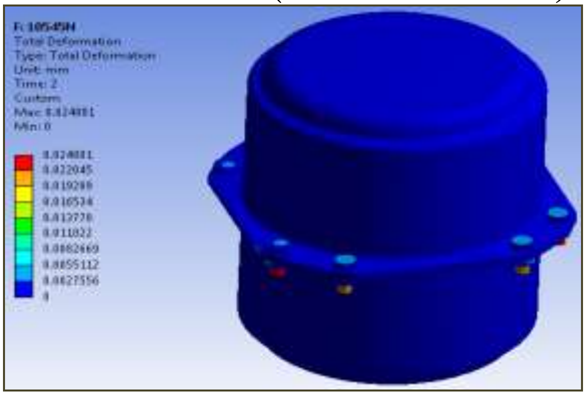


Fig.16; Deformation in 75 % load

LOAD CASE 5: *9842N (70 PERCENT PRELOAD)*

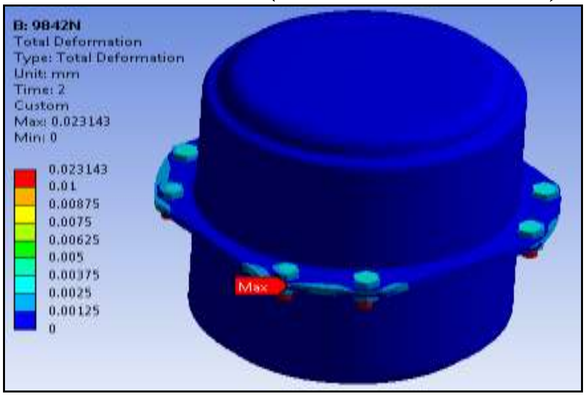


Fig.17; Deformation in 70 % load

LOAD CASE 3: *7030N (50 PERCENT PRELOAD)*

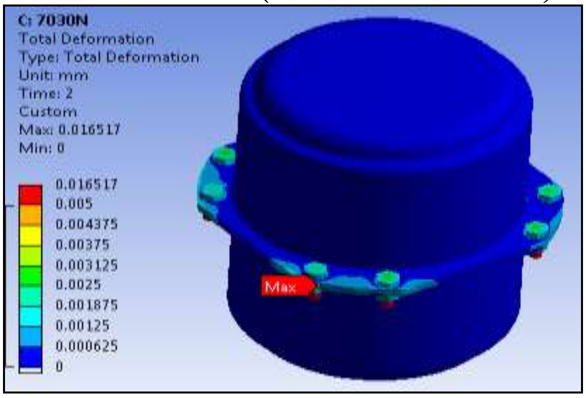


Fig.18; Deformation in 50 % load

Above three figure shows the deformation in all different loading conditions, where figure 13, shows the higher deformation of 0.0331mm which is having 100 % load on it. Where 90% & 80% load having deformation of 0.0297mm & 0.0264mm respectively as shown in figure 14 & 15. Also 75% load is having deformation of 0.0248mm as shown in figure 16. Whereas at 70% and 50% are reducing the deformation to 0.0231mm and 0.0165mm respectively as shown in figure 17 & figure 18.

VON MISSES STRESS:-

LOAD CASE 1: *14060N (100 PERCENT PRELOAD*)

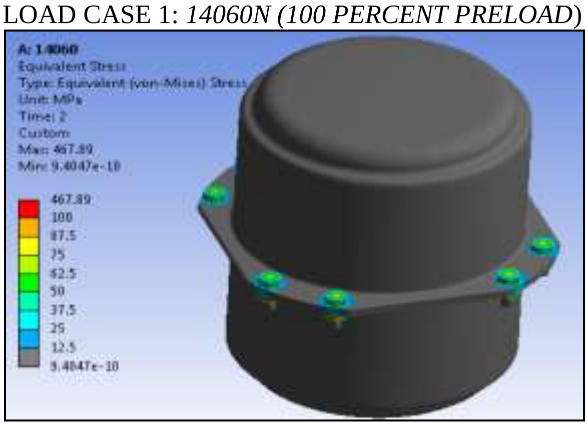


Fig.19; VON MISES Stress at 100 % load

LOAD CASE 2: *12654N (90 PERCENT PRELOAD)*

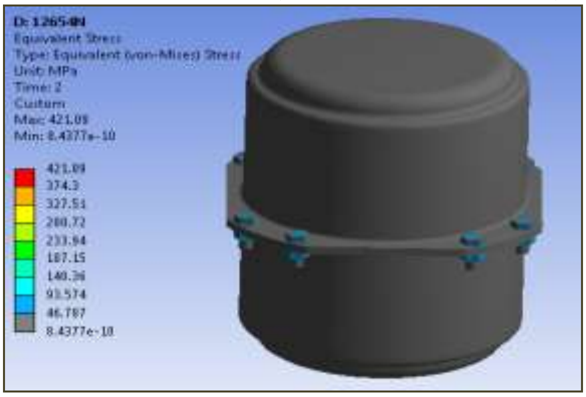


Fig.20; VON MISES Stress at 90 % load

LOAD CASE 3: *11248N (80 PERCENT PRELOAD)*

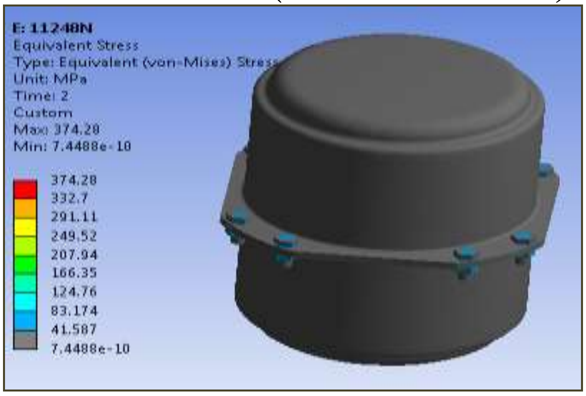


Fig.21; VON MISES Stress at 80 % load

LOAD CASE 4: *10545N (75 PERCENT PRELOAD)*

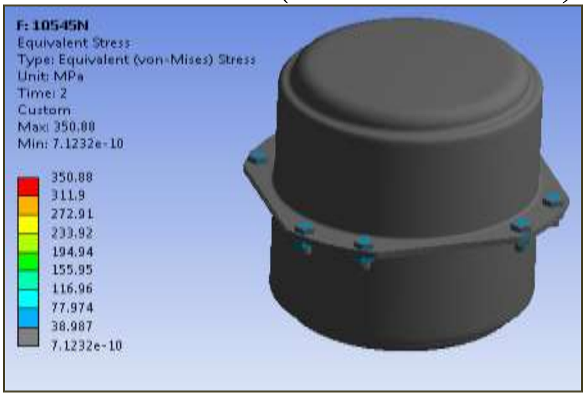


Fig.22; VON MISES Stress at 75 % load

LOAD CASE 5: *9842N (70 PERCENT PRELOAD)*

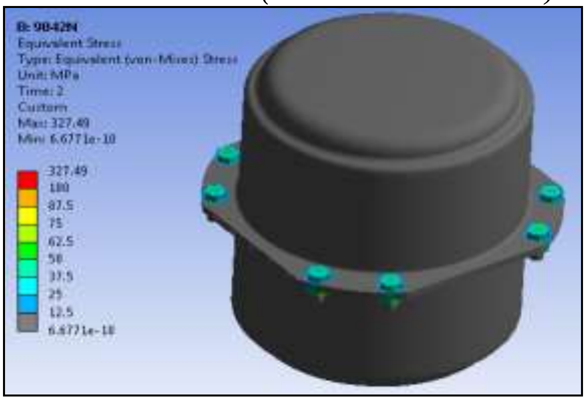


Fig.23; VON MISES Stress at 70 % load

LOAD CASE 6: *7030N (50 PERCENT PRELOAD)*

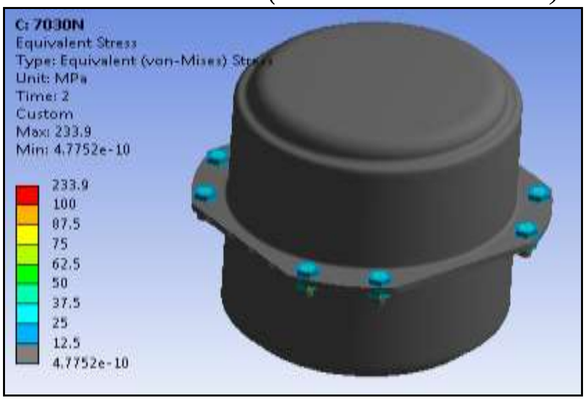


Fig.24; VON MISES Stress at 50 % load

VON MISES Stresses induced in all loading conditions are shown in above figures. From figure 19 it is seen that the stresses induced with applying load of 100% is 467.89 Mpa which is more as compare to 90% load which is 421.09 Mpa (shown in figure 20) and reducing to load 80% the stresses reduced to 374.28 Mpa (shown in figure 21) again load reducing to 75% load stresses induced are 350.88 Mpa (shown in figure 22), load reducing to 70% and 50% the stresses induced are 327.49 Mpa & 233.9 Mpa respectively as shown in figure 23 & 24 respectively.

CONTACT STATUS:-

LOAD CASE 1: *14060N (100 PERCENT PRELOAD)*

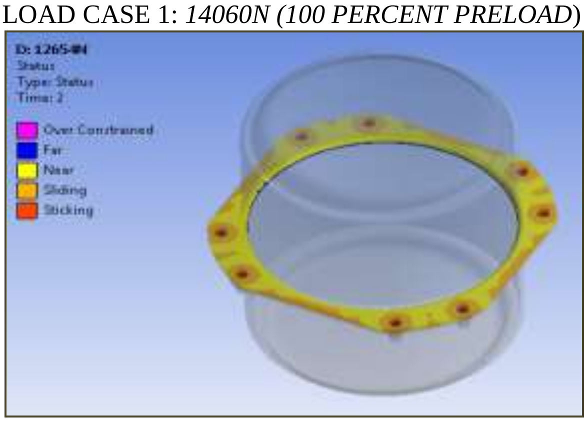


Fig.25; Contact status at 100 % load

LOAD CASE 2: *12654N (90 PERCENT PRELOAD)*

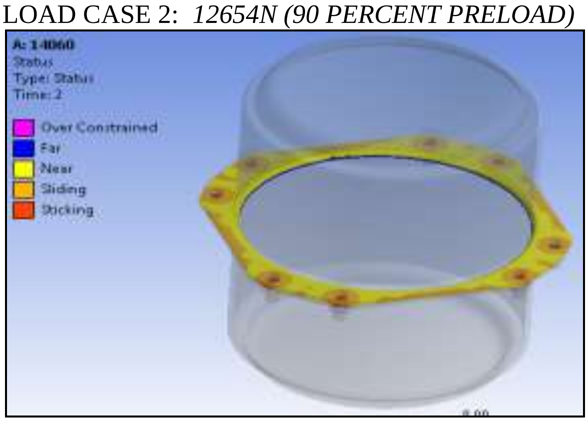


Fig.26; Contact status at 90 % load

LOAD CASE 3: *11248N (80 PERCENT PRELOAD)*

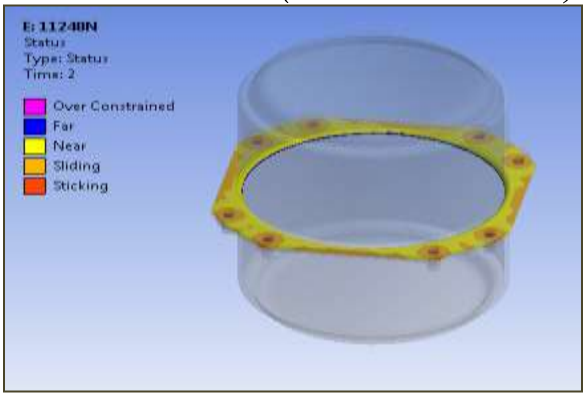


Fig.27; Contact status at 80 % load

LOAD CASE 4: *10545N (75 PERCENT PRELOAD)*

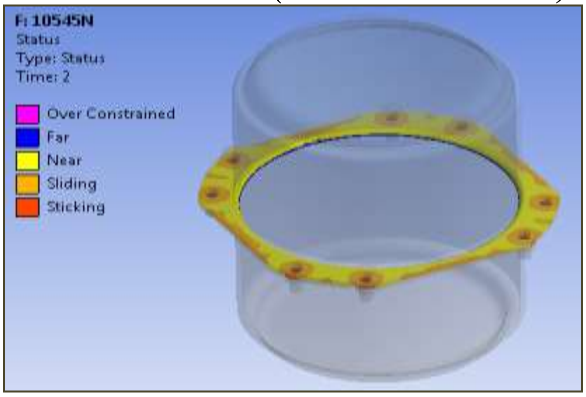


Fig.28; Contact status at 75 % load

LOAD CASE 5: *9842N (70 PERCENT PRELOAD)*

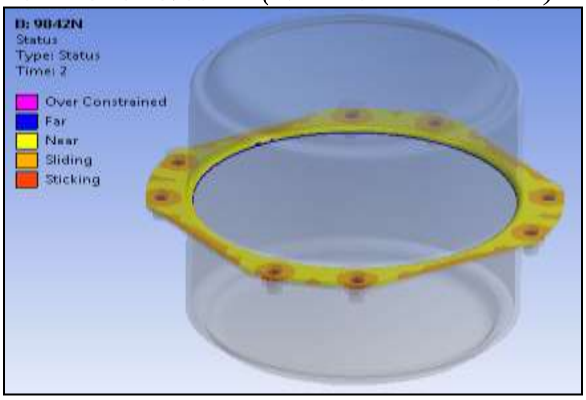


Fig.29; Contact status at 70 % load

LOAD CASE 6: *7030N (50 PERCENT PRELOAD)*

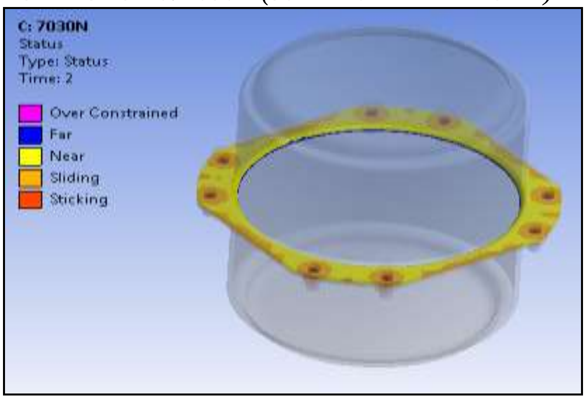


Fig.30; Contact status at 50 % load

Above all figure shows that as the torque on bolt is reduced contact status gets weak, from figure 27 where load is applied is 100% it is seen that the contact status is in sliding and some what sticking contact condition. Where as in 70% load applied it shows only sliding condition as shown in figure 29. And in 50% load applied condition the contact condition is to the somewhat sliding and somewhat near contact condition as shown in figure 30.

CONTACT PRESSURE:

LOAD CASE 1: *14060N (100 PERCENT PRELOAD)*

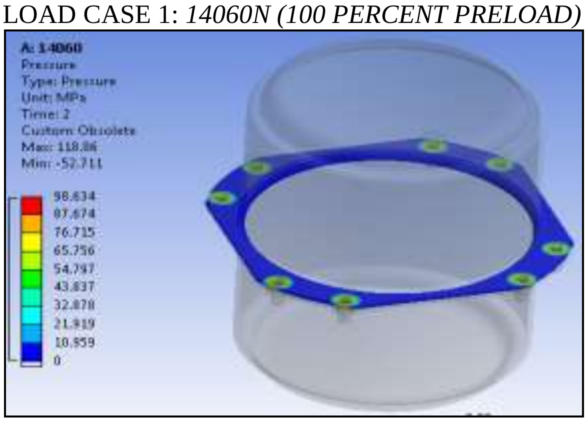


Fig.31; Contact pressure at 100 % load

LOAD CASE 2: *12654N (90 PERCENT PRELOAD)*

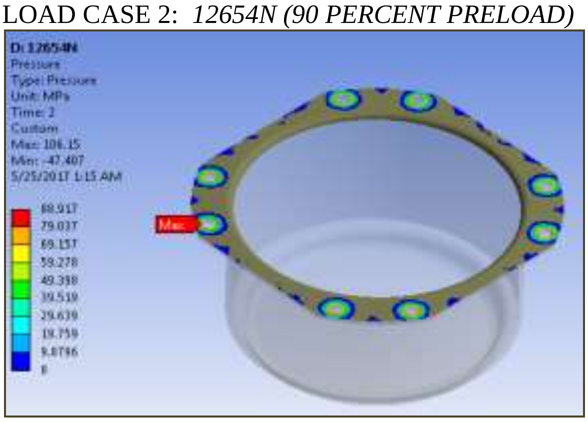


Fig.32; Contact pressure at 90 % load

LOAD CASE 3: *11248N (80 PERCENT PRELOAD)*

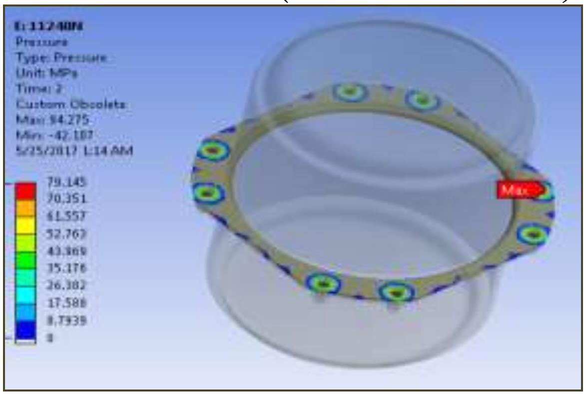


Fig.33; Contact pressure at 80 % load

LOAD CASE 4: *10545N (75 PERCENT PRELOAD)*

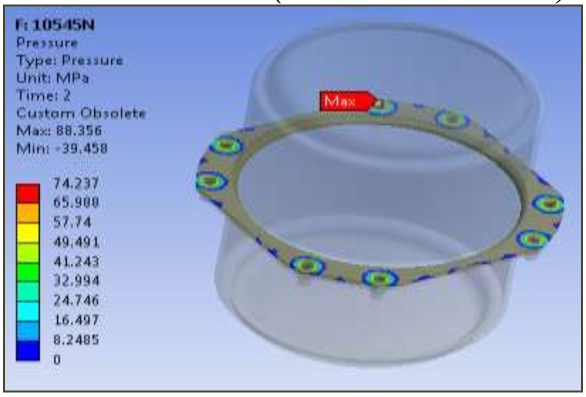


Fig.34; Contact pressure at 75 % load

LOAD CASE 5: *9842N (70 PERCENT PRELOAD)*

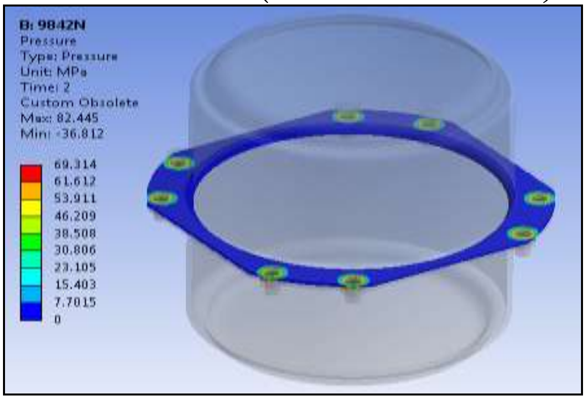


Fig.35; Contact pressure at 70 % load

LOAD CASE 6: *7030N (50 PERCENT PRELOAD)*

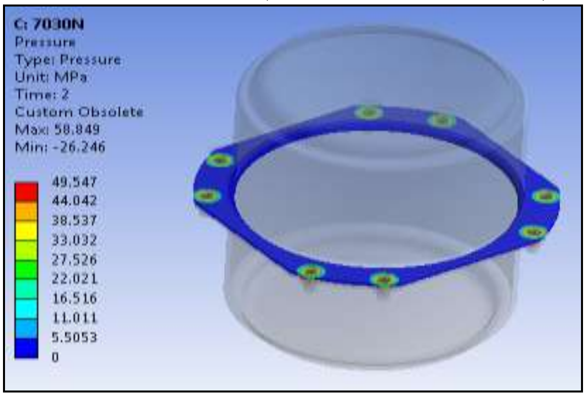


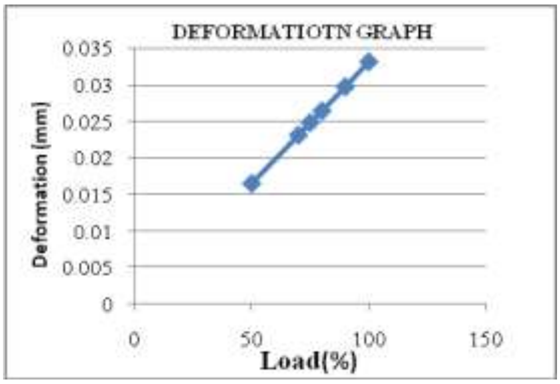
Fig.36; Contact pressure at 50 % load

As the load varies on bolt, contact pressure varies in direct proportional. If the load reduced than the contact pressure reduces. From figure 31 at 100% load the contact pressure is 118.06 Mpa, at 90% load it reduces to 106.15 Mpa (shown in figure 32), at 80% of load it reduces to 94.275 Mpa (shown in figure 33), at 75% load it reduces to 88.356 Mpa (shown in figure 34) 70 % load it reduces to the 69.314 Mpa (shown in figure 35) and at 50% load it reduces to 58.894 Mpa(shown in figure 36).

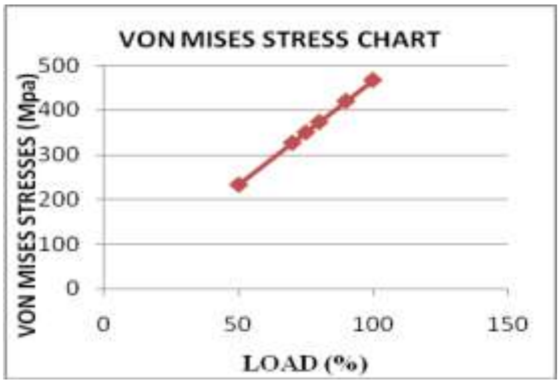
VII. Result And Discussion

LOAD %	DEFORMATION (mm)	VON MISES STRESSES (Mpa)	CONTACT PRESSURE (Mpa)
100	0.0331	467.89	118.06
90	0.0297	421.09	106.15
80	0.0264	374.28	94.275
75	0.0248	350.88	88.356
70	0.0231	327.49	82.445
50	0.0165	233.9	58.849

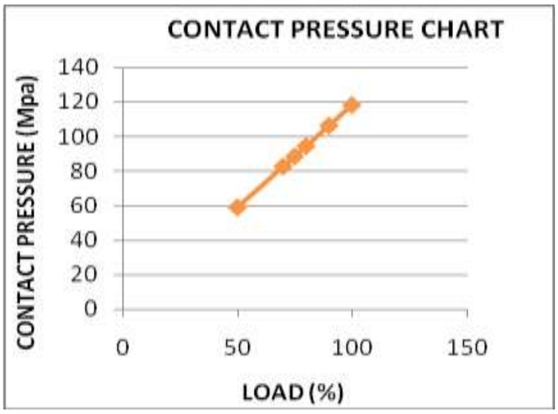
Table 2: FEA readings



Graph 1: Deformation in three loads in FEA



Graph 2: VON MISES Stresses at three loads in FEA



Graph 3: Contact pressure induced at three loads in FEA

Here are some graphs plotted with the values formed in analysis in ANSYS which shows that, as the load reduces on the bolt the deformation induced on the contact surface is reduced as shown in graph 1. Whereas VON MISES stresses increased as the load on bolt increased as shown in graph 2. In graph 3 it is seen that the load increases than the contact pressure also increases, it is required that in bolt joint the contact pressure should be maximum which can help to avoid oil leakages.

VIII. Conclusion

The minimum torque required to avoid the oil leakage from oil sump is 75% load (10545 N). So to avoid leakages minimum torque required is 75% torque as compare to standards. As load reduces contact pressure reduces, which hence cause to leakages. As bolt pretension reduces, sliding nature of contact reduces, which cause to leakages. As the load decreases the stresses induced reduces. Effect of pre load on assembly is also studied, where it is found that deformation, contact pressure and stresses reduces as load reduced, which can cause to leakages from assembly.

Refferences

[1]. Zhengui Zhang, Peixuan Wu, Haiyan H Zhang, “ Contact Stress Analysis of Interference Fit Between V-band Clamp and Piping Systems” International Journal of Engineering and Innovative Technology (IJEIT) Volume 4, Issue 3, September 2014
[2]. Design Optimization of Oil Pan using Finite Element Analysis Kandru Kamalakar, Talada Suresh Prakash, Mechanical Engg.Department, Swami Vivekananda Engineering College, Bobbili, A.P., India
[3]. Design of Oil Sump using Coupled Eulerian – Lagragian (CEL) Technique By A. Vishnu Prasanna Moorthy
[4]. A. Satish, Dr. K.Rambabu, M. Ramji, “Part Modeling with Reverse Engineering” International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 12, December – 2013 ISSN: 2278-0181
[5]. Dharun Lingam K, Arun Lingam K, “Design and Fatigue Analysis on Metal Matrix Composite Connecting Rod Using FEA” International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 12, December – 2013 ISSN: 2278-0181
[6]. Prasad Darapureddy, Ch. Srinivas, R. Lalitha Narayana, “Static and Modal Analysis of Base Frame for Steam Turbine” International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 12, December – 2013 ISSN: 2278-0181
[7]. Mechanical design handbook, Edition 16, page number 266.