ISSN: 2456 - 4664
International Journal of Advanced Trends in Engineering and Technology
Impact Factor 5.965, Special Issue, January - 2018
1st International Conference on Innovations in Mechanical Engineering (ICIME-2018)
On 5th & 6th January 2018 Organized By
Guru Nanak Institute of Technology & Guru Nanak Institutions Technical Campus, Hyderabad

DEFORMATION ANALYSIS ON EXHAUST MANIFOLD

Vinod Angadi*, Praveen Siddanna* & Dr. S. Sreenatha Reddy**
* Department of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, Telangana
** Professor, Guru Nanak Institute of Technology, Hyderabad, Telangana

Cite This Article: Vinod Angadi, Praveen Siddanna & Dr. S. Sreenatha Reddy, "Deformation Analysis on Exhaust Manifold", International Journal of Advanced Trends in Engineering and Technology, Special Issue, January, Page Number 41-44, 2018

Abstract:

The exhaust manifold is placed on the cylinder head of the engine. It is attached to the catalyst converter at the other end. The exhaust manifold is related to high temperatures and pressures which will gives to thermo- mechanical failure. As exhaust gases are actual hot, the aqueduct charge be heat-resistant In a two stoke engine, the exhaust pipe is provided with a bulge known as expansion chamber More air and ammunition is fabricated to access the agent butt with the burden of the gases in the chamber. The architecture is advised safe afterwards the acclimation that the deflections and accent ethics do not beat the accustomed limits. The Execution Furthermore efficiencies of the motor depending upon those exhaust system. Also, correct plan of the ventilation system will bring about bring down fuel utilization.

Key Words: Manifold, Exhaust, Workbench, FEA, Geometry & Meshing

1. Introduction:

Exhaust manifolds are for the most part basic cast iron alternately stainless steel units which gather motor debilitates gas starting with various cylinders also convey it of the fumes channel. The word MANIFOLD derived from the Old English word manifold (refers to the folding together of multiple inputs and outputs. In contrast, an inlet or intake manifold supplies air to the cylinders) the manifold assorted is subjected to large temperatures and pressures which will advance to thermo-mechanical failure. A back burden is created due to not absolutely abandonment the manifold arrangement before the gases from the added butt is released. These Weight waves from gas limit the engine's valid execution possibilities. Concerning illustration the debilitate gasses would very hot, that channel must a chance to be heat-resistant. Those channels ought to have the ability to send the dangerous gasses far starting with those away from the operator. Those debilitate pipes would different for diverse sorts of engines. A stationary motor might need a fireplace serving as a fumes channel. To engine cycles those debilitate pipes rely on the kind of the motor it needs. In The situation from claiming trucks, the exhaust framework will be level Furthermore at times might have Vertical pipes. Some trucks are Gave with adaptable ducting the middle of those motor Furthermore silencers. This plan will abstain from the vibrations being exchanged of the exhaust framework (1). Thermal examination need will make done right away for discover those temperature distribution, high engineering transfer, warm gradients additionally heat flux. This might a chance to be took following to tension analysis, should remember those thermal anxieties. The design is considered safe after the conformance that the deflections and stress values do not exceed the allowed limits. The performance and efficiencies of the engine depend upon the exhaust manifold. Also, correct design of the manifold give us result in low fuel consumption.

2. Methodology:

The finite element analysis (FEA) is mathematical method that is used to get approximate. Solution to boundary valve problems. It uses a mathematical method called finite element method(fem).in FEA is the computer generated model of a structure that is loaded and analyses for unique results, much as stress, deformation, deflection, natural frequencies, mode shapes, temperatures distribution. The modal analysis is used to calculate the vibration characteristics such as natural frequency and mode shape (deformed shapes) of component. The results from modal analysis can be further used as input for the harmonic and transient analyses. For example, a cantilever beam, attached to a system vibrating at a certain frequency. It is important for the designer to find out whether the beam will sustain the vibrations induced by the machine to which it is connected. When the cantilever vibrates, various shapes are attained at certain frequencies. The shape of the component corresponding to a frequency is known as mode shape. The mode shape is a representations of the deformation attained due to vibration. The main aim of the modal analysis is to find whether the natural frequency of the component is closer to the vibrations induced in the component. In this example, with this cantilever, the more number of modes found is six. Through display the various mode shapes of first, second, 3rd, and 4th modes, respectively. If the natural frequency of a system is nearer excitation frequency, the component can get into resonance and fail. Therefore, to avoid the resonance, you need to strengthen the component on die basis of the mode shape. However, sometimes strengthening the component possible due to the design limitations. Also, in actual practice, the displacement produced at resonance may not be get due to the presence of damping. Therefore, you need to calculate the results of a system under the time/frequency based loads. If the stress/strain/displacement response is less than the permissible limit, the component will not be required to strengthen or redesign (2).

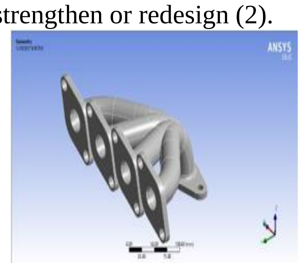


Figure 2.1: Model Geometry

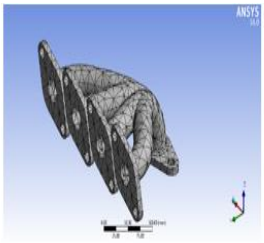


Figure 2.2: Meshing of Model

ISSN: 2456 - 4664
International Journal of Advanced Trends in Engineering and Technology
Impact Factor 5.965, Special Issue, January - 2018
1st International Conference on Innovations in Mechanical Engineering (ICIME-2018)
On 5th & 6th January 2018 Organized By
Guru Nanak Institute of Technology & Guru Nanak Institutions Technical Campus, Hyderabad

2.1 Simulation and Boundary Conditions: After the meshing done to model next step is to set the boundary conditions under which the analysis is to be performed. For that right click on modal node in the tree outlines and then click insert fixed support from the menu displayed; fixed support with a question symbol is added under the modal nodes in the tree outline. After specifying the boundary conditions in the mechanical window, you need to set the variables to find the results and solve the analysis. Select analysis settings under the modal node in the tree outline. And make sure that no is chosen in the limit search to range drop-down list. Expand the solver controls node in the details of “analysis settings” window, if it is not already expanded. In the damped drop-down list, select the no option, if not already selected. By right clicking on the solution node, then choose the solve option from the shortcut menu displayed; the analysis is solved. Select the solution node in the tree outline; the graph and tabular data windows are displayed, Right-click on the solution node again and then choose to evaluate all results from menu displayed; all the six results are ready to be viewed. Select total deformation under the solution node; the find mode is shown in the graph screen, as shown in figure. Select total deformation 2 under the solution node; the second mode shape is displayed in the graph screen, as shown in figure Similarly, select other results from the solution node to view the respective mode shape in the graphics view.Go to solution and right click on solution- summary of results from workbench can be obtained (3) Tabular data window shows selecting temperature from the steady-state thermal node in the tree outline

2.2: Material Properties of Cast Iron:

- ✓ Thermal conductivity, K (w/m k)………………50
- ✓ Density, (kg/m3)…………………………………7200
- ✓ Specific heat, c (J/Kg k)………………………..1.88
- ✓ Poisson’s ratio, v………………………………..0.3
- ✓ Thermal expansion, á (10ı6 / k)……………….0.3
- ✓ Elastic modulus, E (GPa)………………………105
- ✓ Coefficient of friction, µ………………………...0.2
- ✓ Yield strength (MPa)……………………………130

Table 2.1: Temperature applied in firing order

Step	Order	Temp 1	Temp 2	Temp 3	Tempe 4
1	1	600	22	22	22
2	4	22	22	22	600
3	2	22	600	22	22
4	3	22	22	600	22

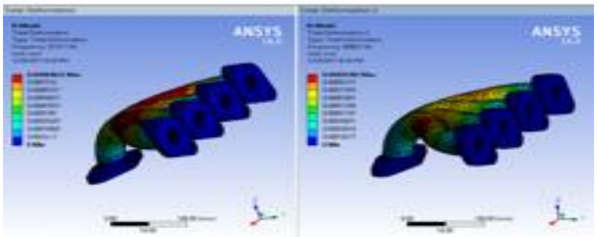


Figure 2.1.1: Deformation 1 and 2

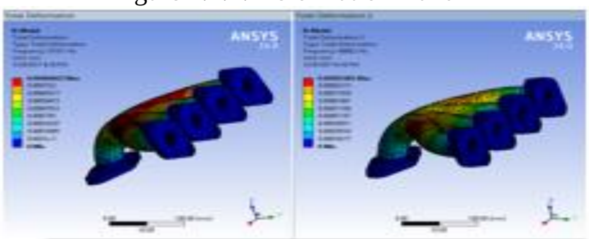


Figure 2.1.2: Deformation 3 and 4

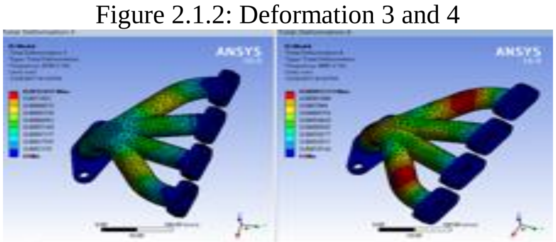


Figure 2.1.3: Deformation 5 and 6

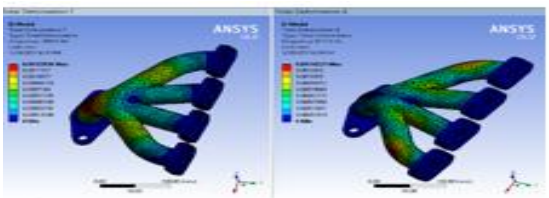


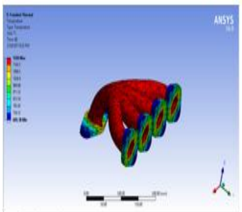
Figure 2.1.4: Deformation 7 and 8

Table 2.2: Deformation Values

Results	Minim...	Maximum	Units	Reported Frequency (Hz)
Total Deformation	0.	8.4622e-004	mm	3519.1
Total Deformation 2	0.	9.2492e-004	mm	4000.2
Total Deformation 3	0.	1.3397e-003	mm	4347.5
Total Deformation 4	0.	1.3814e-003	mm	6218.1
Total Deformation 5	0.	1.2411e-003	mm	6742.5
Total Deformation 6	0.	9.1131e-004	mm	6861.2
Total Deformation 7	0.	1.2956e-003	mm	7019.2
Total Deformation 8	0.	1.4237e-003	mm	8517.1

3. Results and Analysis:

Sectioned view of the model shows the colour contours that indicate the distribution of temperature along various regions of the model. Red colour contour displays the region with the maximum temperature, whereas blue colour contour shows the region with minimum temperature. The other colours represent various temperatures that lie within the maximum (red contour) and minimum (blue contour) temperatures. The colour contours and their respective values can be seen in the legend, refer to figure. The temperature on the dome of the piston is maximum because the temperature boundary condition is applied and the component has attained maximum temperature in this region. The effect of the temperature on the farther regions is minimum, refer to figure. In this model the maximum temperature is 2000 °c and the minimum temperature is 22 °c in figure 3.1 and 3.2 respectively.



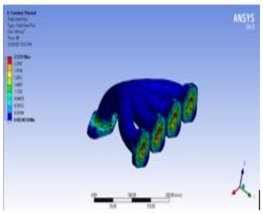
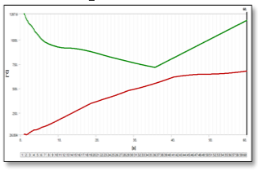


Figure 3.1: Temp distribution Figure 3.2: Heat flux on model



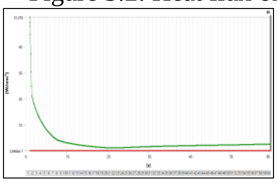


Figure 3.3: Temp v/s time Figure 3.4: heat flux v/s time

A body is called to be deformed if its shape is changed temporarily or permanently. The temporary change of shape is known as elastic deformation and a permanent change of shape is known as plastic deformation. In workbench, you can determine deformation in terms of total and directional deformations. Total deformation is the total change of shape in a given working condition. You can view the total deformation induced in any component by using the total tool from the deformation drop-down in the solution contextual toolbar. Directional deformation is the total change of shape in a particular axis, due to given working conditions. You can view directional deformation by using the directional tool from the deformation drop-down in the solution contextual toolbar (4). Total deformation is the summation of all directional deformations produced in a certain region of the model. The following equation describes the total deformation:

If deformation in the x-axis u_x

Deformation in the y-axis u_y

Deformation in the z-axis u_z

Then total deformation. U will be given as follows: $U = (u_x^2 + u_y^2 + u_z^2)^{1/2}$

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	0.23825	mm	1.
Equivalent Stress	0.23372	497.84	MPa	1.
Life	4.4858e+005	1.e+008	Units Unavailable	0.
Damage	10.	2229.3	Units Unavailable	0.

Table 3.1: Deformation Values

Table 3.1: Values from Model Analysis

Modes	Total Deformation Mm	Frequency Hz
1	8.46e-04	3519.1
2	9.25e-04	4000.2
3	1.34e-03	4347.5
4	1.38e-03	6218.1
5	1.24e-03	6742.5

ISSN: 2456 - 4664
International Journal of Advanced Trends in Engineering and Technology
Impact Factor 5.965, Special Issue, January - 2018
1st International Conference on Innovations in Mechanical Engineering (ICIME-2018)
On 5th & 6th January 2018 Organized By
Guru Nanak Institute of Technology & Guru Nanak Institutions Technical Campus, Hyderabad

6	9.11e-04	6861.2
7	1.30e-03	7019.2
8	1.42e-03	8517.1

Table 3.2: Values from Static Analysis

S.No	Results	Minimum	Maximum
1	Total Deformation Mm	0	0.23825
2	Equivalent Stress Mpa	0	497.84
3	Life Hours	4.49e+05	1.00e+08
4	Damage	10	2229.3

Table 3.3: Steady State Temperature Applied

	Temperature ^{O}C		
Time	Applied	Minimum	Maximum
1	300	270.77	300.01
2	600	476.45	600.07
3	900	617.35	900.24
4	1200	713.55	1200.6
5	1500	781.61	1501.2

4. Conclusion:

Thermal analysis provides the thermal loads for Structural analysis. Coupled Field analysis results show maximum deflection of 0.1 mm and Von Mises Stress as115MPa.And As per Maximum Yield Stress theory, the design is accepted and further analyses can be carried out. And From Modal analysis, 6 natural frequencies for the structure is identified. Maximum Mass participation factor in the different directions are obtained. Also Harmonic Response analysis is done and the amplitudes at the forcing frequency for each pipe is obtained. Deflections and stresses at these critical frequencies are obtained. Hence it is observed that the Von Mises stress at frequencies of 1250, 1300 and 1450 Hz are equal to the yield strength of Cast Iron. The given design of the exhaust manifold is safe.

5. References:

1. Bin Zou, Hu Yaqian, Zhien Liu, Fuwu Yan and Chao Wang, "The Impact of Temperature Effect on Exhaust Manifold Thermal Modal Analysis", Research Journal of Applied Sciences Engineering and Technology, PP 2824-2829.(2013)
2. J.David Rathnaraj, "Thermo mechanical fatigue analysis of stainless steel exhaust manifolds", IRACST – Engineering Science and Technology, Vol 2, No. 2, PP 265 – 267.(2012)
3. K.H. Park, B.L. Choi, K.W. Lee, K.-S. Kim and Y.Y. Earmme, "Modelling and Design of Exhaust Manifold under Thermo mechanical Loading".
4. Sweta Jain, AlkaBani Agrawal, "Coupled Thermal – Structural Finite Element Analysis for Exhaust Manifold of an Off-road Vehicle Diesel Engine", International Journal of Soft Computing and Engineering, Vol 3, Issue 4, PP 226-230.(2013)
5. A.K.M. Mohiuddin, Ataur Rahamn and Mohd. Dzaidin, "Optimal design of automobile exhaust system using gt- power", International Journal of Mechanical and Materials Engineering, Vol 2 No. 1, PP 40 – 47.(2007)
6. H. Julia, Buckland, Mrdjan Jankovic, J. W. Grizzle and J. S. Freudenberg, "Estimation of exhaust manifold pressure in turbocharged gasoline engines with variable valve timing".
7. Swathi Satish, Mani Prithiviraj and Sridhar Hari, "Comparison of predictions obtained on an exhaust manifold analysis using conformal and indirect mapped interface", International Congress on Computational Mechanics and Simulation, IIT Hyderabad,(2012)
8. Masahiro Kanazaki, Masashi Morikawa, "Multiobjective Design Optimization of Merging Configuration for an Exhaust Manifold of a Car Engine", Parallel Computational Fluid Dynamics, International Conference Parallel, Kyoto, Japan,(2002)
9. M.Rajasekhar Reddy, Dr K.Madhava Reddy, "Design and Optimization of Exhaust Muffler in Automobiles", International Journal of Engineering Research and Applications, Vol 2, Issue 5, PP 395 – 398.