Jurnal Teknologi, 49(F) Dis. 2008: 311–322

FINITE ELEMENT STRESS ANALYSIS ON NATURAL GAS MOTORCYCLE INTAKE MANIFOLD

ZULKEFLI YAACOB[1], RAHMAT MOHSIN[2*], ZULKIFLI ABD MAJID[3], IRENE TEH HUI LYNN[4] & ROSDI BAHARIM[5]

Abstract. Stress analysis is an essential part of the design process and is also an indispensable tool for optimizing structural design to meet multidisciplinary performance requirements. The purpose of this study is to determine natural vibration frequency of intake manifold on the natural gas motorcycle by using finite element modal analysis. Besides that, the stress and displacements of the manifold structure are also been determined. When the motorcycle engine is running, the engine will cause some vibration due to unbalance condition of the engine. Vibration created by the engine will be transmitted to all neighboring component including intake manifold as well as the air fuel mixer. Intake manifold is next to the engine which is the source of the vibration. Thus, it takes greater impact of the transmitted vibration as compare to other components. Computer simulation technique using Finite Element method has been used to simulate this problem. Finite Element structure analysis was employed to study the stress distribution and natural frequency of the intake manifold. Finite Element model of the intake manifold is developed and analyzed. The effect of localize stiffening with ribs on intake manifold is considered to reduce the level of the vibration. It was found that by adding a stiffened rib to the manifold structure, the natural frequencies are increased. This may due to the stiffening of the manifold structure which dominates its vibration characteristics. Moreover, the strength of the manifold structure is also increased by adding a rib.

Keywords: Stress analysis; finite element method; natural frequency; vibration; stiffening rib

Abstrak. Analisis ketegasan merupakan satu keperluan bagi proses reka bentuk dan ia juga amat dikehendaki bagi mengoptimumkan reka bentuk struktur bagi memenuhi pelbagai displin prestasi yang dikehendaki. Tujuan kajian ini adalah untuk menentukan frekuensi semulajadi untuk manifold bagi motorsikal gas asli. Selain itu, peralihan dan tegasan struktur manifold juga ditentukan. Apabila enjin motorsikal dijalankan, gerakan enjin akan menghasilkan getaran yang disebabkan oleh keadaan gerakan enjin yang tidak stabil. Getaran yang dihasilkan daripada enjin akan beralih ke komponen-komponen motorsikal yang bersebelahan termasuk manifold masukan yang bertindak sebagai percampur udara dan bahan api. Maka, komponen ini akan mengalami kesan yang besar daripada getaran yang beralih daripada enjin berbanding dengan komponen lain. Simulasi komputer menggunakan kaedah unsur tak terhingga telah digunakan untuk menyelesaikan masalah ini. Kaedah analisis struktur unsur tak terhingga telah digunakan untuk menganalisis pertaburan ketegasan dan frekuensi semulajadi struktur manifold. Model manifold telah dicipta dan dianalisiskan. Kesan penegangan dengan menggunakan penegang tetulang dipertimbangkan untuk

[1,2,3,4] Gas Technology Center, Faculty of Chemical and Natural Resources Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor Bahru, Malaysia
[5] Malaysian International Shipping Company (MISC), Menara Dayabumi, Jalan Hishamuddin, 50050 Kuala Lumpur, Malaysia
[*] Corresponding author: Email: rahmat@fkkksa.utm.my

merendahkan tahap getaran. Apabila penegang tetulang ditambahkan kepada struktur ini, didapati bahawa frekuensi semulajadi telah dipertinggikan. Ini mungkin disebabkan oleh penegangan struktur manifold mendominasi sifat-sifat ketegasan. Selain itu, analisis juga menunjukkan bahawa ketegasan maksimum tidak melebihi ketegasan reka bentuk. Maka, boleh disimpulkan bahawa struktur manifold ini boleh menahan tekanan operasi yang maksimum. Kekuatan struktur juga bertambah dengan penambahan penegang tetulang.

Kata kunci: Analisis ketegasan; kaedah unsur tak terhingga; frekuensi semulajadi; getaran; penegang tetulang

1.0 INTRODUCTION

Research by the Gas Research Institute (GRI) has shown that NGVs have the potential to provide substantial reductions in greenhouse gas emissions. Thus, Gas Technology Centre UTM (GASTEG) has developed a dedicated natural gas (NG) powered motorcycle fuel system to reduce the emission and to meet rising demand for alternative fuel. Natural gas is the cleanest burning alternative fuel. Exhaust gas emissions from natural gas vehicles are much lower than those from gasoline-powered vehicles.

In order to convert fuel to energy, the engine needs a proper fuel delivery system. It should include fuel storage, fuel transmission line, air-fuel mixer, flow/volume controller, and the manifold to drive fuel-air mixture to the engine (combustion chamber). Intake manifold is the passageway from the air-fuel mixture to combustion chamber. The mixture will be collected in the combustion process.

When the motorcycle engine is running, the engine will cause some vibration due to unbalance condition of the engine. This vibration will be transmitted to all

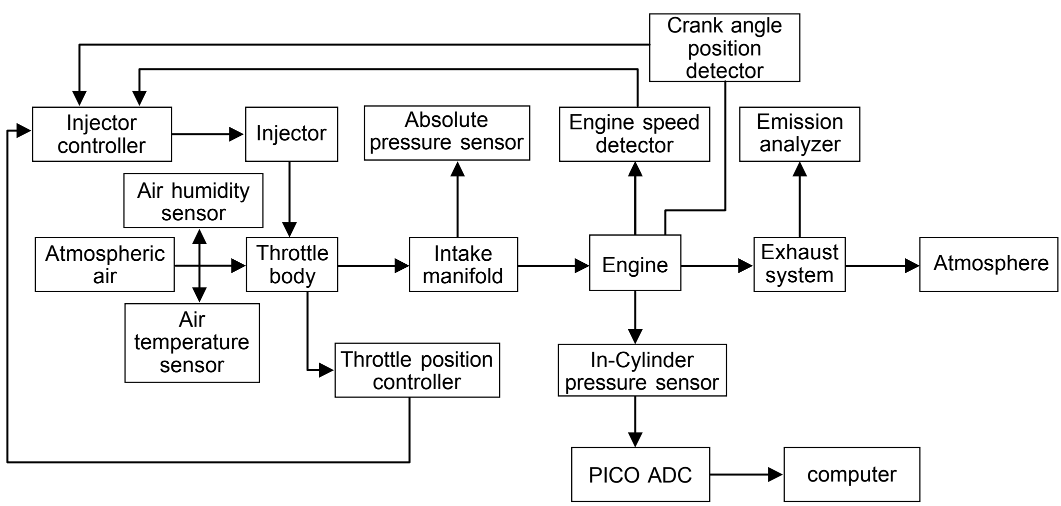


Figure 1 Full engine system for Modenas Kriss 111

neighboring component including intake manifold as well as the air fuel mixer. Intake manifold is closer to the engine which is the source of the vibration. Thus, it takes greater impact of the transmitted vibration as compare to other components. Resonance vibration is occurring when the excitation frequency approached the natural frequency. The natural frequencies are the frequencies at which a structure will tend to vibrate if subjected to a disturbance. A natural frequency is a source of resonance vibration that may lead to structure failure [5]. This resonance vibration is a critical vibration which propagates in the system or component causing component failure or fatigue [3]. Besides that, the system needs to configure correctly so that can provide safe, reliable and predictable service for life of motorcycle. If the component interaction is not completely understood by the system designer, it may cause the component to respond in undesirable manner and leading to unsatisfactory performance and life.

The finite element analysis is very useful tool to find out the vibration sources correctly and reduce the overall vibration level [6]. Finite element analysis is used to find out the modal properties of the structure. By using FEM technique, we can analyze how the structure exactly vibrates under given excitation, condition and with what amplitude [4].

The finite element method is comprised of three major phases:

(i) **Pre-processsing,** in which the analyst develops a finite element mesh to divide the subjected geometry into sub domains for mathematical analysis, applies material properties and boundary conditions.
(ii) **Solution**, during which the program derives the governing matrix equations from the model and solves for the primary quantities.
(iii) **Post-processing,** in which the analyst checks the validity of the solution, examines the value of primary quantities (such as displacement and stresses), and derives and examines additional (such as specialized stresses and error indicators) [8].

Structural vibration analysis is one of the important areas that should be considered in order to study the behaviour of any component operated in a rotating machine [4]. The ride comfort is affected by the vibration and noise which experienced by the user during the travel. These vibration are caused by road roughness, mass unbalance etc [6].

In order to predict the structural behaviour, a high quality system model such as vehicle body, engine is a prerequisite. The first step is to apply classical analysis tools like normal modes analysis, frequency response and frequency deformation to understand the behaviour of the system. The next step is to define the design variables relating to the region. It should be taking into account fatigue, crash and production requirement.

2.0 RESEARCH METHODOLOGY

Prior to this research, researchers have done serious study on various field of NGV such as Natural Gas Vehicle fuel system, natural gas fuel characteristic, development of dynamometer, study on engine oil for NGV engine. So this study will focus on the intake manifold system. Structural analysis on the intake manifold system will be carried out to support the theory in order to generate new idea for the design [6].

Firstly, the factors that contribute to stress on the structure of the intake manifold is considered. There are including the transmitted vibration from the engine, pressure and temperature loadings and prescribed displacement.

After verifying the source of stresses, the methods related to this research is identified and used as guideline. Nowadays, finite element and other numerical models are very commonly for predicting deflections and stresses of structures [7]. Finite element analysis has been widely used in noise, vibration and harshness simulation especially in the fast growing automotive industries. MSC.PATRAN is the platform to develop and establish all the requirement of the model need to perform finite element analysis. That was very critical part to done before it passes through to run in its solvent well known as MSC.NASTRAN. Both of the two programming is coming in package. The result analysis then will present back in MSC.PATRAN.

Model updating is a step in model validation process that modifies the values of parameters in a finite element model. This is an important stage where the dynamic behavior of the structure is modified. The modification that been done is adding a

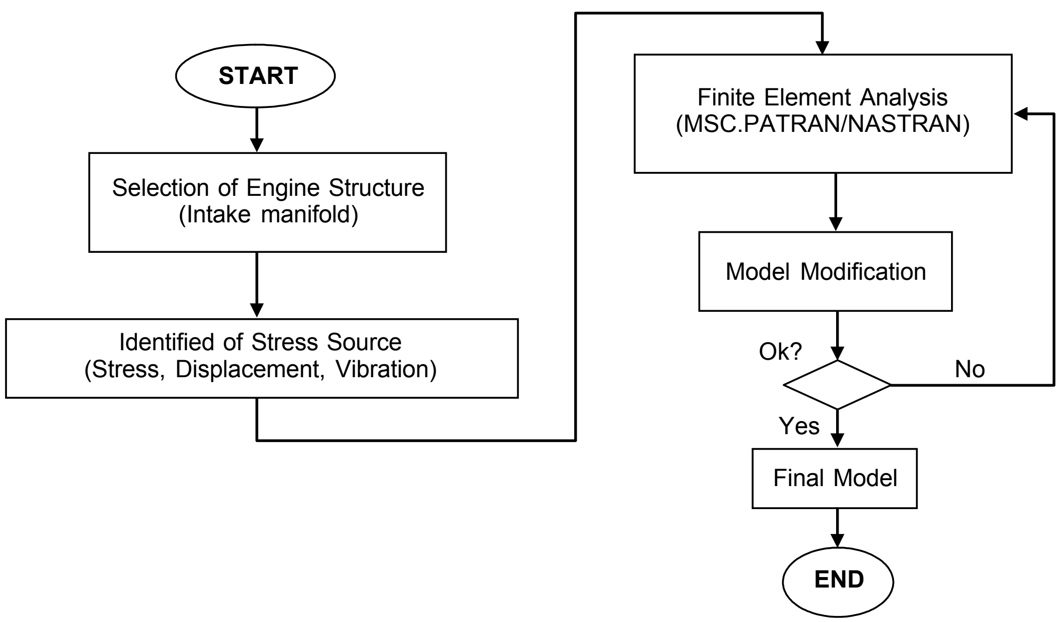


Figure 2 Methodology flowchart

stiffening rib to manifold, changing the thickness of the manifold and replace cast iron with other materials.

Based on the changes in results, the design of intake manifold is continuous improved until the best design is found. Finally, when result has been generated, the analyst must first check to see the numerical solution is satisfactory before determine the required engineering data from the solution.

3.0 RESULTS AND DISCUSSION

3.1 Solidified Model

Solid model for intake manifold is successfully drawn as a prototype to carry out an analysis in term of the stress and deformation. The normal mode analysis which runs to determine natural frequencies is also carry out to the model. The prototype is developed by using SOLIDWORKS program. The solid prototype intake manifold without and with rib is shown in Figure 3.

(a) Without rib (b) With rib

Figure 3 Solid drawing for manifold via SOLIDWORKS

Solid model has to be drawn and saved by using IGES format file before being exported to MSC.PATRAN program. This step is successfully done and MSC.PATRAN software is capable to read the model.

3.2 Stress

Finite element analysis using MSC.PATRAN/NASTRAN are capable to perform analysis of deformation and stress of manifold structure when it is pressurize at 2.5×10^5 Pa from throttle body. Analysis will focus the maximum tensor stress caused by the applied upon pressure.

The materials that used for finite element analysis are stainless steel, cast iron, aluminium and cast aluminium alloys. For example, the result analysis shows that maximum stress tensor for cast iron is shown in Table 1 below.

Table 1 Stress and Displacement for manifold structure

(a) Material: Stainless Steel

Thickness (mm)	Without Rib Stress ($\times10^6$ Pa)	Without Rib Displacement ($\times10^{-4}$ mm)	With Rib Stress ($\times10^6$ Pa)	With Rib Displacement ($\times10^{-4}$ mm)
2.5	2.18	1.43	1.84	1.19
2.0	2.55	1.77	2.27	1.53
1.5	2.96	2.52	2.70	2.07

(b) Material: Cast Iron

Thickness (mm)	Without Rib Stress ($\times10^6$ Pa)	Without Rib Displacement ($\times10^{-4}$ mm)	With Rib Stress ($\times10^6$ Pa)	With Rib Displacement ($\times10^{-4}$ mm)
2.5	2.11	2.64	1.84	2.30
2.0	2.49	3.31	2.30	2.81
1.5	3.05	4.80	2.70	3.55

(c) Material: Aluminum

Thickness (mm)	Without Rib Stress ($\times10^6$ Pa)	Without Rib Displacement ($\times10^{-4}$ mm)	With Rib Stress ($\times10^6$ Pa)	With Rib Displacement ($\times10^{-4}$ mm)
2.5	2.15	4.38	1.85	3.75
2.0	2.54	5.35	2.27	4.61
1.5	2.92	7.39	2.89	5.81

(d) Material: Cast Aluminum Alloys

Thickness (mm)	Without Rib Stress ($\times10^6$ Pa)	Without Rib Displacement ($\times10^{-4}$ mm)	With Rib Stress ($\times10^6$ Pa)	With Rib Displacement ($\times10^{-4}$ mm)
2.5	2.16	3.99	1.86	3.38
2.0	2.41	4.87	2.21	4.28
1.5	2.86	6.62	2.73	5.62

For cast iron, when the manifold is pressurizes at a maximum pressure of 2.5×10^5 Pa, the maximum stress tensor is 2.11×10^6 Pa and it is occurred node 1647. Figure 4 shows the detail of node 1647 location. Even though the maximum stress tensor occurs at node 98 but maximum displacement does not occur at the same node. From the result, it is occurred at node 401. After stiffening rib is added to the manifold structure, the maximum stress is decreased. This show that stiffened rib is increasing the strength of the structure.

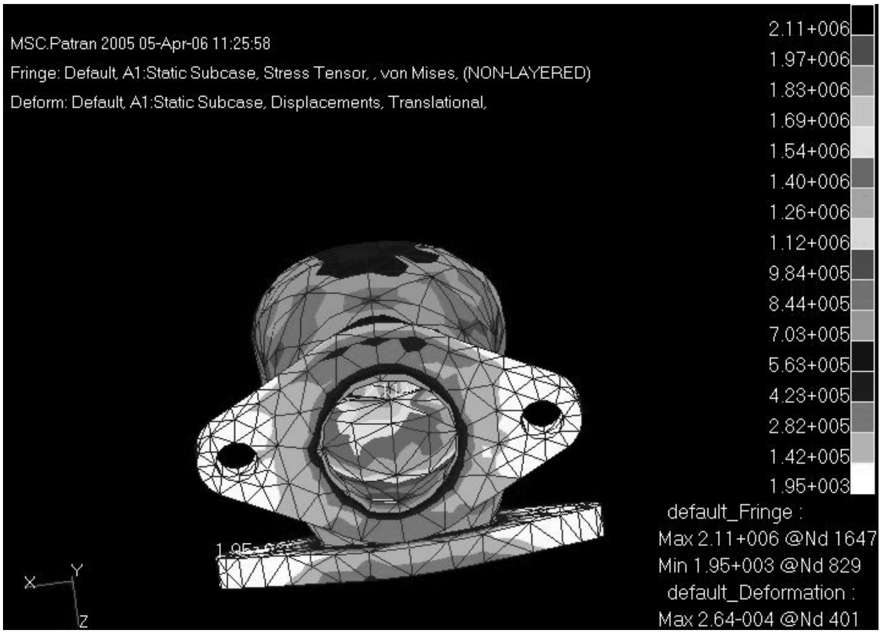


Figure 4 Maximum stress tensor (2.11×10^6 Pa) on Node 1647

For structural applications, the yield strength is usually a more important property than the tensile strength, since once it is passed, the structure has deformed beyond acceptable limits. Yield strength is the pressure which a substance is capable of supporting without fracturing. In industry, yield stresses are usually not even approached because the applied stresses are kept well below the yield strength by a safety factor on the order of 1.5 to 2.0.

$$\text{Design Stress} = \frac{\text{Yield Strenght}}{\text{Safety Factor}}$$

On this analysis, the safety factor that will be considered is 2.0. From the simulation result, the maximum stresses of each manifold structure did not excess the design stresses. Thus, the manifold structure is safe to use. The gas flow pressure is decreased after the pressure regulator. Thus, the pressure of the gas flow through the manifold does not affect so much on the manifold structure.

3.3 Displacement

The pressure is applied at the inner surface of manifold. It must be tied to vehicle body to bind the prototype and prevent movement from its origin position. The applied pressure will force the elements to move from its origin and analysis will show the displacement of elements node. Figure 5 below show the displacement

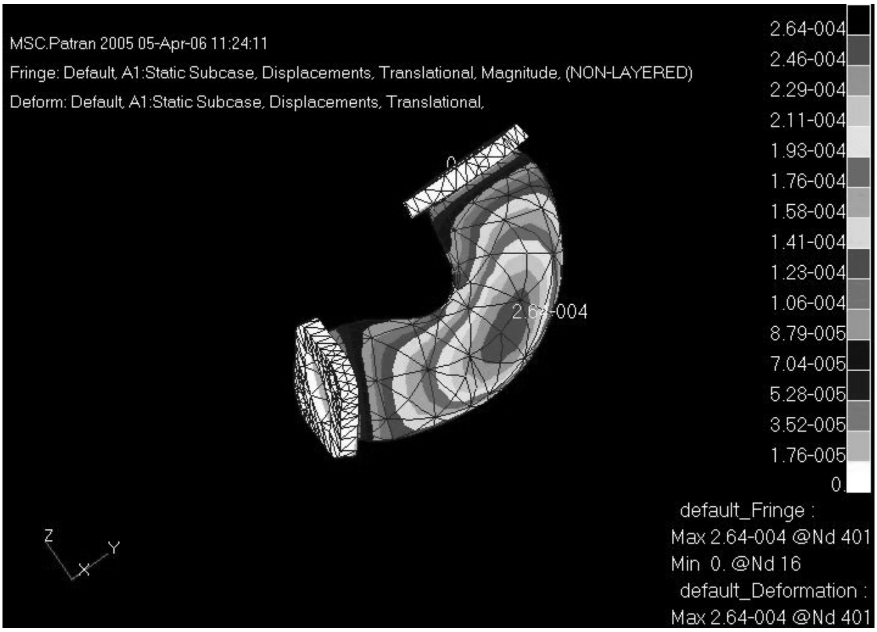


Figure 5 Maximum displacement region (2.64×10^{-4} mm) on Node 401

value for manifold model at pressure of 2.5×10^5 Pa in the inner surface. The result shows the node displacement for original model using cast iron as its material of construction. From the figure, the maximum value of displacement is 2.64×10^{-4} mm and occurs at node 401 in the red contour.

The displacement for manifold structure for each material is not critical due to small distance of node displacement which is lower than global model tolerance 0.005 mm as stated by MSC.PATRAN. For example, the maximum displacement for Stainless Steel 304 manifold without rib 1.43×10^{-4} mm for thickness 2.5 mm which is lower than global model tolerance 0.005 mm. Thus, the manifold structure is safe to operate at that pressure.

Comparing to Stainless Steel 304, Cast Iron, Aluminium and Cast Aluminium Alloys are give 2.64×10^{-4} mm, 4.38×10^{-4} mm and 3.99×10^{-4} mm node displacement. The displacement occurs at same node for every types of material. Though the results show the variation values of node displacement, but the displacement occur at same node for each of material. This situation explains that the node will receive the maximum force when pressure us supplied.

3.4 Natural Frequency

The models of the manifold system are used to illustrate some of the "classical" vibration mode shapes typically encountered with manifold systems. While every system may not exhibit every mode shown, these are the typical modes of concern.

Table 2 Natural frequencies for manifold without and with rib

(a) Material: Stainless Steel

Mode Shape	Natural Frequencies (Hertz)	
	Without Rib	With Rib
7	118.32	170.39
8	134.24	128.95
9	155.60	154.78
10	212.86	209.27

(b) Material: Cast Iron

Mode Shape	Natural Frequencies (Hertz)	
	Without Rib	With Rib
7	90.84	131.76
8	104.02	100.01
9	122.47	122.15
10	164.14	162.19

(c) Material: Aluminum

Mode Shape	Natural Frequencies (Hertz)	
	Without Rib	With Rib
7	111.27	161.11
8	126.06	121.20
9	144.72	144.39
10	199.68	197.81

(d) Material: Cast Aluminum Alloy

Mode Shape	Natural Frequencies (Hertz)	
	Without Rib	With Rib
7	116.54	168.95
8	132.29	127.09
9	151.41	151.12
10	208.49	200.57

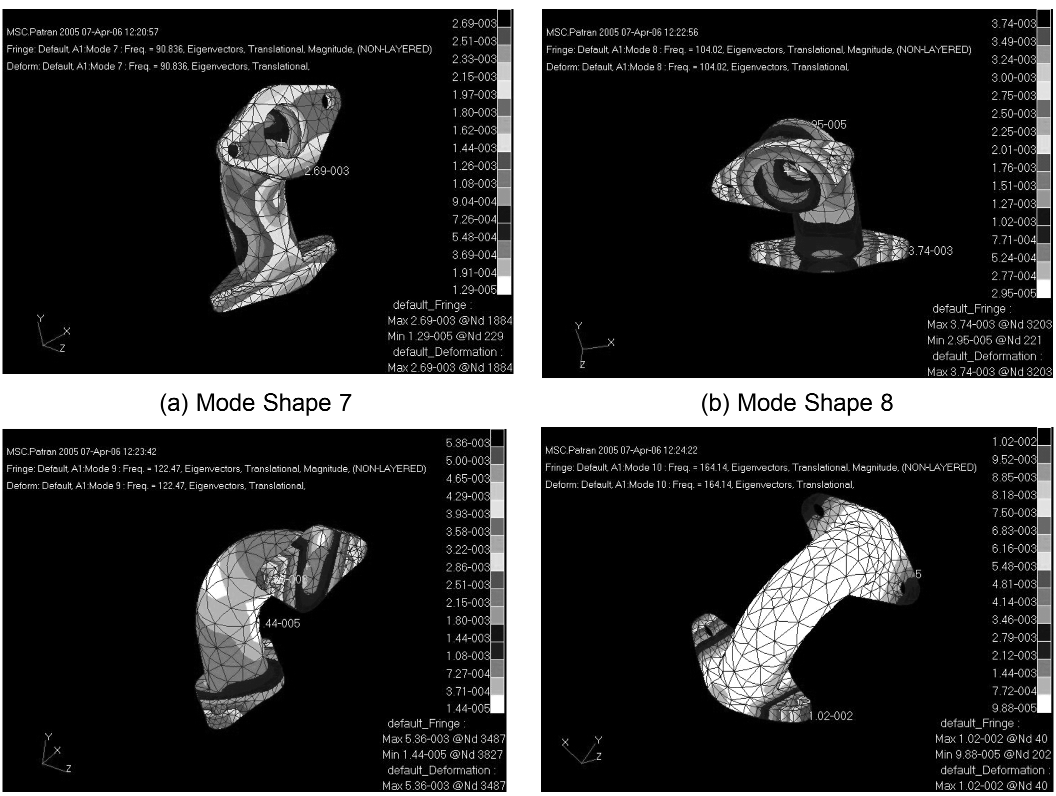
Therefore, it is useful to understand these modes and effective means of controlling them. The natural frequencies for each material were calculated by MSC.NASTAN and tabulated in Table 2.

Mode shapes 1 to 6 are called the low mode because it is often the lowest frequency mode. The mode shapes are basically rigid body motion of the structure in phase. The mode shape 7 is represented bending failure of the model. On this mode, the manifold structure is bended up and down (toward z-direction).

Mode shape 8 is also represented bending failure of the manifold structure. The different between of this mode shape with mode shape 7 is the direction of bending. For this mode shape, the manifold structure is bended to left and right (x-direction). Mode Shape 9 is the torsion mode. The manifold structure is twisted along y-axis. Mode shape 10 is bending mode for the fringe part on y-direction for the bottom and z-direction for the top. Mode shapes for original structure are show in Figure 6.

The manifold structure with additional stiffened rib is created to analyze the effect of the rib toward the dynamic behavior of the manifold structure. By adding a stiffened rib to the manifold structure, the natural frequencies are increased. This may due to the stiffening of the manifold structure which dominates its vibration characteristics. The mode shape 7 affected by the location of the stiffener. The deformations associated with these modes are prevented by the stiffener.

In contrast, the natural frequencies for mode shape 8, 9 and 10 are decreased. The mode shapes does not affected so much by the stiffened rib. This is because the location of the stiffened rib did not help in preventing the deformation but only



(a) Mode Shape 7 (b) Mode Shape 8

(c) Mode Shape 9 (d) Mode Shape 10

Figure 6 Mode shape of original manifold structure (Cast Iron) increases total mass of the manifold. From this study, it can be concluded that to reduce the vibration and noise from a vibrating structure, the dominant natural frequency and mode shape have to be identified for effective stiffening. Mode shapes for modified structure are show in Figure 7.

In general, the natural frequency for the manifold structure is given by the following equation.

$$F_n = \frac{1}{2\pi}\sqrt{\frac{k}{m}}$$

where F_n is the natural frequency
k is the stiffness
m is the mass

From this, it is seen that if the stiffness increases, the natural frequency also increases. If the mass increases, the natural frequency decreases.

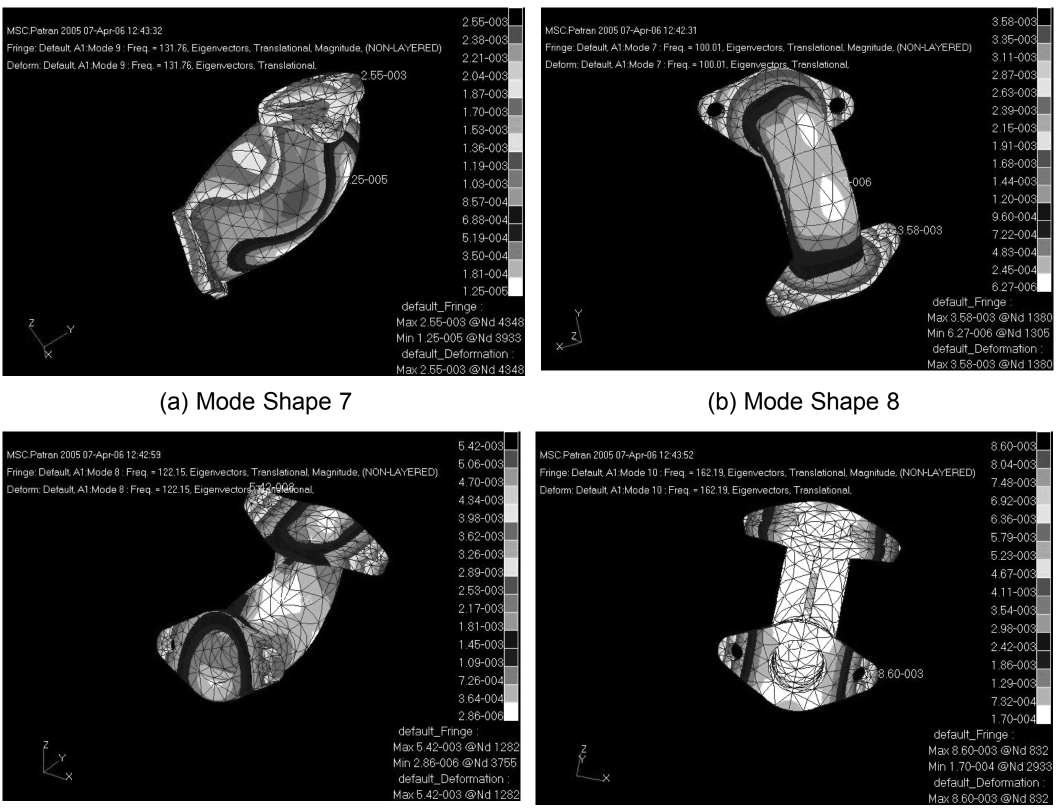


Figure 7 Mode shapes of modified manifold structure (Cast Iron)

4.0 CONCLUSION

In conclusion, the natural frequencies and the mode shape of intake manifold have been determined by using normal mode analysis. By adding a stiffened rib to the manifold structure, the natural frequencies are increased. This may due to the stiffening of the manifold structure which dominates its vibration characteristics. The mode shape 7 affected by the location of the stiffener. The deformations associated with these modes are prevented by the stiffener. From this study, it concluded that to reduce the vibration and noise from a vibrating structure, the dominant natural frequency and mode shape have to be identified for effective stiffening.

The stress tensor and deformation of the manifold is obtained by using linear static analysis. Adding a rib to the manifold structure is also increases the structure strength toward pressure. Thus, the manifold structure is sufficient to support maximum operating pressure.

REFERENCES

[1] Iremonger, M. J. 1982. *Basic Stress Analysis*. London: Butterworth.

[2] Leonard, M. 1975. *Element of Vibration Analysis*. New York: McGraw-Hill.

[3] Tao Li and Jimin He. 1999. Local Structural Modification Using Mass and Stiffness Changes. *Journal of Engineering Structures.* 21: 1028-1037.

[4] Vierck, R. K. 1967. *Vibration Analysis*. New York: International Textbook.

[5] Kalsule, D. J., R. R. Askhedkar and P. R. Sajanpawar. 1999. Engine Induced Vibration Control for a Motorcycle Chassis Frame by Right Combination of Finite Element Method and Experimental Techniques. The Automotive Research Association of India: *SAE paper.* 1999-01-1754.

[6] Heiner, S., S. Hartono and Ya Pu. 2001. Experimental Modal Analysis of Automotive Exhaust Structures. Purdue University: *SAE paper.* 2001-01-0662.

[7] Steven, J. R. 2004. "The Finite Element Method: A Four-Article Series." *Newsletter of the American Society of Mechanical Engineers* (ASME).

[8] Kingston, M. R. 1975. *Finite Element Stress Analysis as Aid to the Design of Automotive Component*. London: Joseph Lucas Ltd. H. G. Gibbs and T. H. Richards. 1975. Stress, Vibration and Noise Analysis in Vehicles. London: Applied Science Publishers Ltd.