International Journal of Scientific & Engineering Research Volume 10, Issue 2, February-2019
ISSN 2229-5518

Finite Element Model Updating of Motorcycle Visor Stay Bracket base on Experimental Modal Analysis

Andi Wibowo, Isdaryanto Iskandar

Abstract— Motorcycle Visor is a new trend fashion of accessories in motorcycle. The Motorcycle visor used as shield and adjust the air flow in desire direction. The other function, black motorcycle visor is also can protect excessive sun light from front area, that may reflected to the speedometer display. But lately motor visor becomes the new trend of motor cycle accecories and make motor cycle apparence become more better and stylish. The wide surface shape of visor has specific characteristic respon due to the vibration excitation from road load and engine vibration. The dynamic response characteristic of visor depend on the visor stay bracket as mounting to the framebody that usualy has cantilever shape design which sensitive to the vibration excitation. In order to reach quick maturity design of motorcycle visor stay bracket and also to avoid trial and error, It was very important to create the model in CAD modeling and perform the analysis in the development stage.The analysis of visor stay bracket was done using Finite Element Methode. The Finite element modeling should update and verified to the actual test to ensure the validity of the model. Most comon methode to ensure the validity of Finite Element Methode was used EMA. The Visor stay bracket finite element modeling was verified using Experimental Modal Analysis result. The Experimental Modal Analysis is performed using impact hammer testing with Digital Signal Analyzer to obtain its natural frequency of the Visor stay bracket. The Motorcycle Visor stay bracket prototype geometry was modeled using CATIA V5R2017 , the meshing process used SimDesigner 2017, and the analysis is done using Nastran 2014 SOL 103 normal mode. The natural frequency obtained from normal mode FEM simulation compared with Experiment Modal Analysis result to evaluate and find the optimum FEM modeling. By updating mesh size refinement of FEM modeling, comparison values between both of the FEM normal mode analysis and Experimental Modal Analysis using impact hammer test shows a quite satisfactory result.

Index Terms— Finite Element Methode, Motorcycle visor stay bracket , Experimental Modal Analysis, impact hammer testing Eigenvalues, Modal Testing.

---

1 Introduction

Commonly at the motorcycle development process, the trial and error will affected high cost development especialy at testing stage. Beside this, trial and error process also followed by long time development and become time consuming step in the development cycle [1]. This should be avoided in order to fill market demand quickly. The study of virtual prototype using numerical metode Finite Element Analysis is used to built Finite Element Model testing in the development stage. And the validity of this Finite Element Methode result should be first verified using Experimental Modal Analysis result. From the experimental modal testing we can obtain the dynamic characteristic such as natural frequency, damping, model shapes. This modal analysis involves process of determining the modal parameters of a structure in order to construct a modal model of the response. In this study, experimental modal analysis will be done by impact hammer testing and numerical methode using Finite Element Methode .

The validity of the results of this FEM normal mode analysis will greatly affect the results of further analysis processes such as visor stay bracket resonance due to the load path excitation and engine vibration which will be forwarded to the entire structure and affect the strength of fatigue of the visor stay bracket and also parts or components that attached to the framebody in testing

2. Experimental Modal Analysis (EMA)

The goal of modal analysis is to identify and find out natural frequencies, damping ratio and mode shapes of specimen or structure. Experimental modal analysis is based on determining the modal parameters by testing. In Experimental modal analysis, frequency response functions (or impulse response functions) are calculated from measured input forces and output responses of a structure. The FRF model can be seen at Fig. 1. The analysis in modal testing is performed in the frequency domain inside the analyzer.

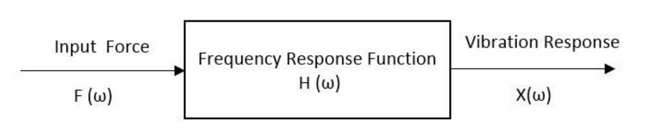

Fig. 1. FRF model used to obtain the dynamic characteristic of prototype using EMA, used to validate FEM result

The structure response usualy can be define in displacement, velocity, and acceleration [2].

In this study, the Experimental Modal Analysis is performed using impact hammer test. And the response was measured using accelerometer. Using this EMA, modal parameter Can be

obtained easily since the method is simple and has good result [2]. The Visor stay bracket prototype using rod 5 mm and plate 1.6 mm was small object category, so for EMA impact hammer was still sufficient to excite the system [2]. The experimental should fulfill the modal testing condition in order to obtain the good result. To perform the impact hammer test, the preparation includes rigid jig preparation, free free condition setting for the visor stay bracket , and equipment setting.
The prototype and equipment was setting as specified figure 2.

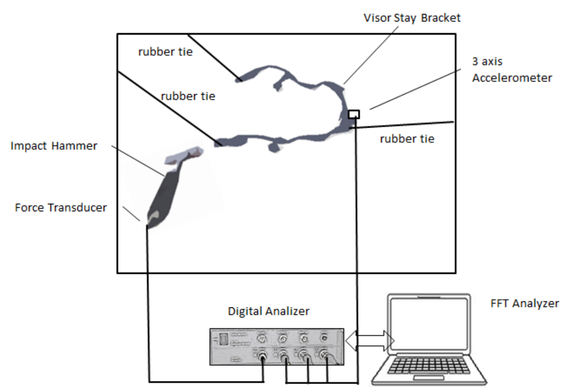

Fig. 2. Experimental Setup using impact hammer test.

The force input and corresponding responses are then used to compute the FRFs.

3. Modeling 3D using catia V5 R2017

The Motorcycle visor stay bracket geometry was modeled using CATIA V5R2017. The visor stay bracket geometry image of Catia V5R2017 3D data can be seen in figure 3. Main component of visor stay bracket consist of 5 mm rod and sheet metal plate 1.6 mm and unite together using bead shape at each assembling wich represent the actual shape of visor stay bracket. The 3D weld bead shape modeling will increase the stiffness of the modeling and will improve the FEM results and can be used reduce error spread of the eigenvalue modeling result [3].

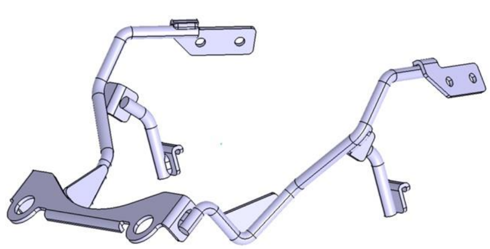

Fig. 3. 3D geometry model of Motorcycle visor stay bracket, modeled using Catia V5R2017

This 3D solid visor stay bracket from Catia V5R2017 data was processed with SimDesigner V2017. By using SimDesigner will allow us to generate meshing from the 3D solid Catia V5 R2017 easily and used as pre processing FEM software, since the simDesigner works in Catia V5 environment.

4. Materials & Speciment

Visor stay bracket was designed for two wheller vehicle in order to make a better style and appearance . The visor stay bracket prototype assembling was developed from 3D data and ploted into 2D Drawing using Catia V5R2017. The material is prepared using STAM 390 with 5 mm diameter rod and 1.6 mm thickness plate . Both diameter rod size and thickness plate, same with the dimension of Catia V5R2017 3D data, each pipe material of the Visor stay bracket is joined by MIG welding. And measurement of visor stay bracket prototype has good dimension condition and good welding penetration same as 3D data design.
The finished prototype of visor stay bracket can be seen in figure 4.

Fig. 4. Geometry of visor stay bracket prototype after welding assembling for Experimental Modal Analysis

The finished prototype of this visor stay bracket was used for next analysis using Experimental Modal Analysis to obtain characteristic dynamic of the visor stay bracket prototype.

5. Numerical Analysis of visor stay bracket using Finite Element Methode

FEM is a numerical method for solving problems of engineering and mathematical physics, that predict the response of physical systems. A modern computational approach based on finite element methode can be use to help obtained the modal characteristic using numerical methode. FE modeling has been shown to be a useful tool in optimizing performance characteristics [4], it is also a useful technique that can be used to reduce the time and financial cost.

To perform the modal characteristic of existing visor stay bracket prototype by FEM, The Finite Elemen Methode of modal analysis was done using Msc NASTRAN 2014 SOL 103 normal mode solution. The natural frequencies and mode shapes were found. Base on this normal mode, the eigenvalue results obtained from FEA simulation is observed to perform next evaluation. In normal mode constrain model, free free condition of the prototype is used to ensure the system is represent

represent EMA same as normal mode condition.

Material of Visor stay bracket prototype is STAM 390 which has mechanical property as follow [5]:

Mechanical properties of Visor stay bracket

Material Property	Specification
Modulus Elastisitas	95000 Mpa
Poisson ratio	0.30000001
Shear Modulus	75000
Density	7.8599998E-009 Kg/mm3

Tabel 1. Mechanical properties of Visor stay bracket prototype

6. Modal Analysis of Visor stay bracket using Impact hammer test

To obtain the modal characteristic of modal analysis of visor stay bracket was done by performing impact hammer test. The impact hammer test as spicified at figure 5.

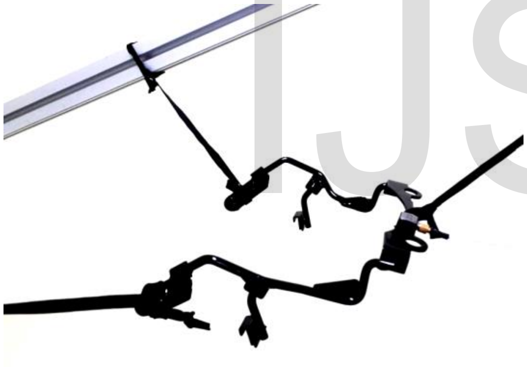

Fig. 5. The Visor stay bracket Test Structure Impact Hammer FRFs Measurement free free condition

The visor stay bracket was mounted on the rigid frame with the rubber tie at three places. The rubber mounted is used to perform free free condition for modal testing using impact hammer. The 3 axis accelerometer was mounted at center front plate bracket to acquire the response of x, y, z, direction. An Impact Hammer with the force transducer at the lower side handle used to measure the impact wich work in the testing.

To compute the measurement and response of the impact, Digital Signal Analizer was used to perform the FRF of the structure at x, y, z.

The quality of the reponse that sometimes disturbed with noise outside the system, can be seen at coherence result. The result of EMA Impact Hammer FRFs Coherence measured from the visor stay bracket prototype can be seen at figure 6.

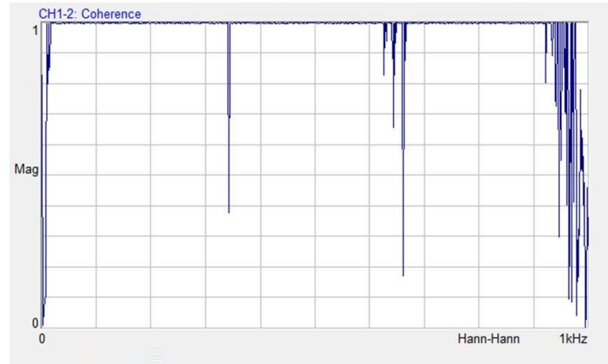

Fig. 6. EMA Impact Hammer FRFs Coherence measured from the Visor Stay Bracket prototype.

The natural frequencies of x direction of the visor stay bracket prototype which were identified with the peaks in x, display FRFs plot and the values can be seen at figure 7.

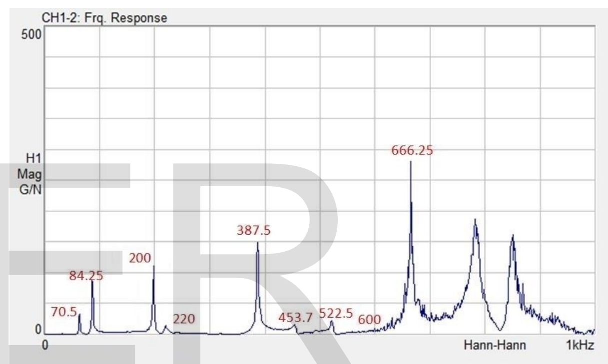

Fig. 7. EMA Impact hammer FRFs measured from the visor stay backet test structure prototype at x direction

The natural frequencies of y direction of the visor stay bracket prototype which were identified with the peaks in y, display FRFs plot and the values can be seen at figure 8.

.

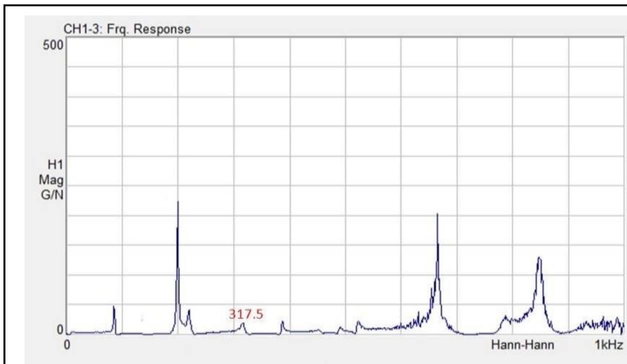

Fig. 8. EMA Impact hammer FRFs measured from the visor stay backet test structure prototype at y direction

The natural frequencies of z direction of the visor stay bracket prototype which were identified with the peaks in z, display

FRFs plot and the values can be seen at figure 9.

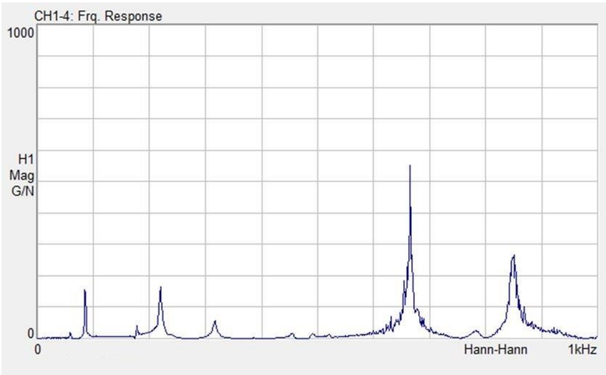

Fig. 9. EMA Impact hammer FRFs measured from the visor stay backet test structure prototype at z direction

The eight natural frequencies of the visor stay bracket prototype which were identified up to eight value are tabulated in Table 2.

EMA Impact hammer FRFs result

Mode Number	Natural Frequency (Hz)
Mode1	70.50 Hz
Mode2	84.25 Hz
Mode3	200.00 Hz
Mode4	220.00 Hz
Mode5	300.00 Hz
Mode6	387.50 Hz
Mode7	453.70 Hz
Mode8	522.50 Hz
Mode9	600.00 Hz

Table 2. The eight natural frequencie of EMA Impact hammer, FRFs measured from the visor stay backet test structure prototype.

Although motorcycle road load excitation usualy with undulation road within 1-80 Hz [6], [7], and in the other hand the excitation frequency from engine is almost 400 Hz. But in this validation, natural frequency observed until eight frequency to ensure the FEM validation compared with actual prototype is good.

7. Finite Element Model Updating Result

The Normal Mode of the 3D solid visor stay bracket are calculated using 10-node tetrahedron element using Msc Patran Nastran 2017. In 3D solid meshing modeling, the 10-node tetrahedron element with reasonable mesh size provide accuracy. 3D CAD solid models are typically meshed with quadratic tetrahedral elements [7], [8]. The Visor stay bracket meshing is generated using Msc SimDesigner V2017, total number of nodes and the total number of elements create with meshing size 10, 7, 4, 2 wais found to be shown in the Table 3.

Nodal & Element

	M10	M7	M4	M2
Nodal	2946	4582	9906	36855
Element	1095	1796	4405	18749

Tabel 3. Nodal & Element of Visor stay bracket using Tet10 mesh size 10, 7, 4 , 2 using SimDesigner 2017

Table 3 provides a comparison of the number of nodes and elements of some mesh size evaluation alternatives. The bigger mesh size 10, has less nodal and element but the shape is too coarse. The smaller the mesh size gave a better geometric shape approach to the shape that needs detailed mesh geometry, on the other hand the smaller mesh size, will effect to the computation process of its analysis become longer. But for this visor stay bracket prototype mesh modeling the computation using Tet10 mesh size 2, still fast enough the next normal mode computation because the geometry was not complicated.

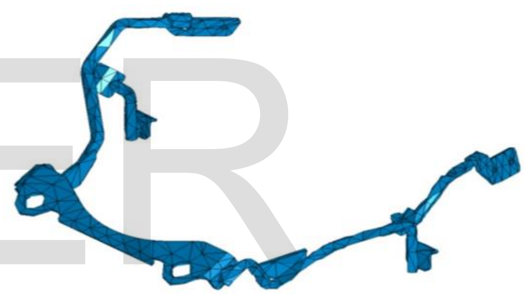

Fig. 10. Tet10 Mesh Size 10 using simDesigner 2017

In figure 10. using Mesh size 10 clearly seen that the shape of visor stay bracket was not generated good enough, the rod bar, hole and the small detail was not good generated, because the mesh size 10 was too big for this model. So for the next modeling should refine to the smaller size. The next smaller mesh size used using mesh size 7 is shown in figure 11.

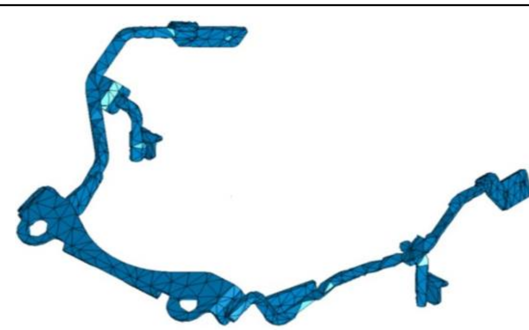

Fig. 11. Tet10 Mesh Size 7 using simDesigner 2017

Using Mesh size 7, the shape of visor stay bracket generated better, but for same area was not good. The small detail at hole was better than using mesh10. The mesh result of mesh size 7 is shown in figure11. The mesh size of smaller size using mesh size 4 is shown in figure 12.

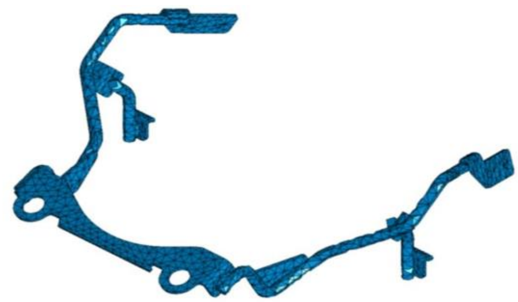

Fig. 12. Tet10 Mesh Size 4 using simDesigner using weld bead

Using Mesh size 4, the shape of visor stay bracket was generated good, The hole and small shape was built good enough, and at the rod the shape is good also.

The mesh size of smaller size using mesh size 2 is shown in figure 13.

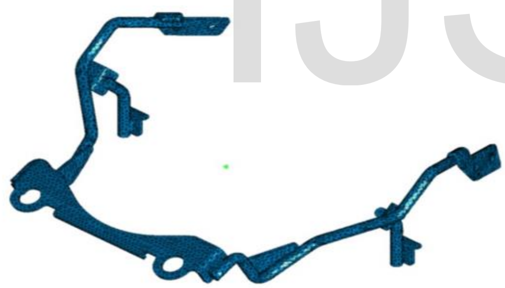

Fig. 13. Tet10 Mesh Size 2 using simDesigner using weld bead

Using Mesh size 2, the shape of visor stay bracket was generated very good. The hole and small shape is built very good and also in the rood visor stay bracket.

Modeling improvement of updated geometry is done, and the result of normal mode analysis to obtain the eigenvalue as in Table 4.

The first eight natural frequencies of visor stay bracket normal mode with the variation of meshing size are calculated compare to the EMA result. The values up to eight natural frequency and the error are tabulated in Table 4.

Mode No.	EMA	Normal mode							
		m10		m7		m4		m2	
	f (Hz)	f (Hz)	%Error	f (Hz)	%Error	f (Hz)	%Error	f (Hz)	%Error
Mode 1	70,500	63,461	9,984	67,63	4,068	65,07	7,705	64,21	8,93
Mode 2	84,250	89,382	6,091	91,67	8,804	87,03	3,303	85,51	1,499
Mode 3	200,000	209,670	4,835	218,86	9,430	223,31	11,655	222,35	11,175
Mode 4	220,000	219,160	0,382	227,47	3,395	227,34	3,336	229,90	4,500
Mode 5	317,500	295,920	6,797	299,64	5,625	299,71	5,603	300,42	5,380
Mode 6	387,500	426,580	10,085	402,49	3,868	474,29	22,397	498,16	28,557
Mode 7	453,700	461,860	1,799	463,02	2,054	516,12	13,758	534,78	17,871
Mode 8	522,500	533,500	2,105	501,79	3,964	571,62	10,123	598,27	14,501
NODAL		82.935		98.968		163.606		284.600	
ELEMENT		41.844		49.753		82.206		143.616	

Table 4. Comparation of EMA and normal mode with mesh variation m10, m7, m4, m2

The spread of comparation result of EMA and normal mode eigenvalue also shown in figure 14.

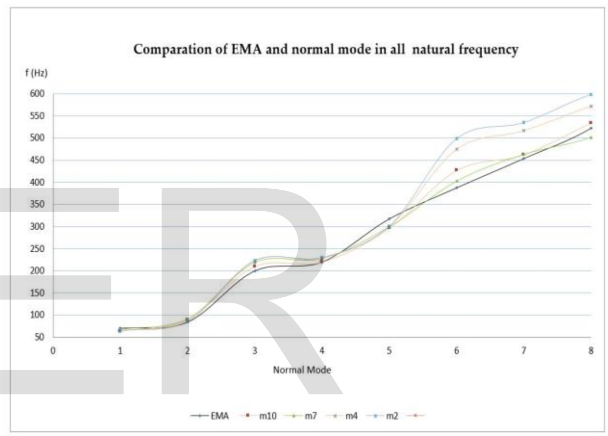

Fig. 14. Comparation of EMA and normal mode in eight natural frequency

In table 4, eigenvalue of FEM modeling with mesh size 10, mesh size 7, mesh size 4, mesh size 2 modeling already gives variation error. The result of mesh size 10 modeling in 6[rd] mode at 426,580 Hz with 10.085 % error, was still not enough to give optimum result to get eigenvalue with error below 10% error compare to EMA result[5]. The result of refinement mesh size 7 modeling already gives better result. The biggest error found at 218,86 Hz with 9.4% error in all mode.

The result of smaller mesh size 4 modeling, which have better shape mesh modeling, has normal mode eigenvalue error up 11,655 % at low frequency and 22,397 % error in high frequency.

The result of smaller mesh size 2 modeling, which have better shape mesh modeling, has normal mode eigenvalue error up 11,175% and 28,557% error in high frequency.

Thus the result of mesh size 4, and mesh size 2 modeling was still not enough to give optimum result to get eigenvalue with error below 10% compare to EMA result, even the visual modeling is very good.

The analyzes shows that with meshing refinement into smaller mesh size and create more detail modeling geometry to the optimum mesh size, will improve the results of the eigenvalue normal mode close to the EMA result. But if the mesh size was too small, the the results of the eigenvalue normal error become bigger again become divergent and resulting big spread error to EMA result. Using the 10-node tetrahedron element with mesh size 7 is the optimum modeling for visor stay bracket prototype and shows quite good result error in low and high frequency. The detail mesh generated was not too coarse and not time consuming when modeling used for this normal mode analysis to obtain eigenvalue and eigenvectore at each mode.

The Eight Eigenvector FEM normal mode result at each mode can be seen also in the figure 15 untill figure 22.

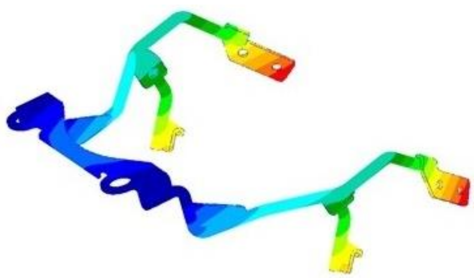

Fig. 15. Eigenvector 1st mode shape at 67.63 Hz obtained from normal mode result analysis of visor stay bracket

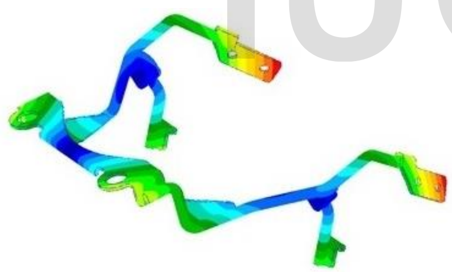

Fig. 16. Eigenvector 2nd mode shape at 91.67 Hz obtained from normal mode result analysis of visor stay bracket.

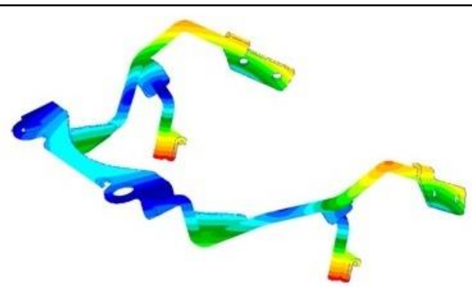

Fig. 17. Eigenvector 3st mode shape at 218.86 Hz obtained from normal mode result analysis of visor stay bracket

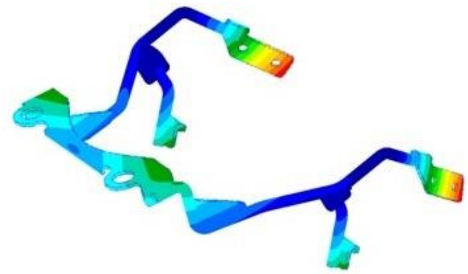

Fig. 18. Eigenvector 4rd mode shape at 227.47 Hz obtained from normal mode result analysis of visor stay bracket

.

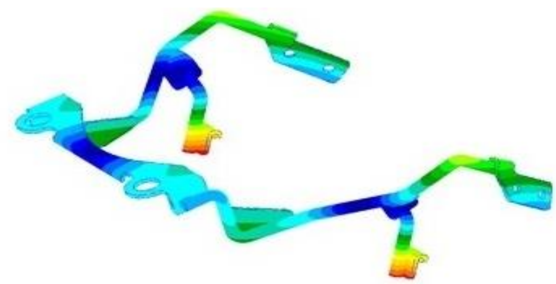

Fig. 19. Eigenvector5rd mode shape at 299.64 Hz obtained from normal mode result analysis of visor stay bracket

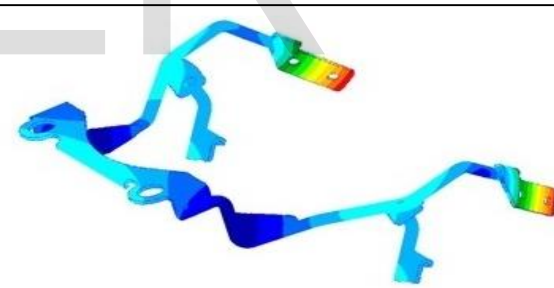

Fig. 20. Eigenvector 6rd mode shape at 402.49 Hz obtained from normal mode result analysis of visor stay bracket

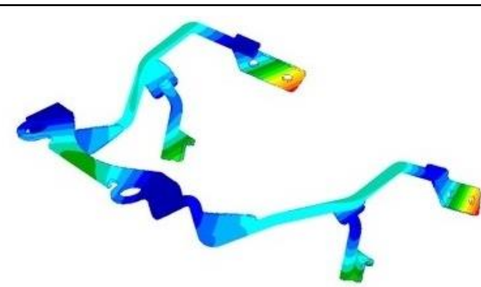

Fig. 21. Eigenvector 7rd mode shape at 463.02 Hz obtained from normal mode result analysis of visor stay bracket

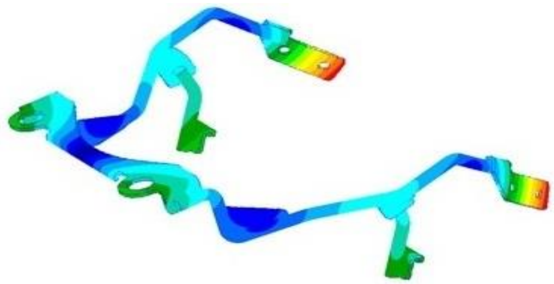

Fig. 22. Eigenvector 8rd mode shape at 501.79 Hz obtained from normal mode result analysis of visor stay bracket

8. Conclusion

This paper presents the Finite element validation, and its model updating of visor stay bracket based on impact hammer test. By performing experimental modal analysis impact hammer, the natural frequency of the visor stay bracket can be obtained. Finite Element model updating has been done and it has a good and quite satisfactory result. In this result, meshing size refinement contribute to improving FEA normal mode result. And using finite element model updating, the natural frequency obtained from numerical analysis is close to the experimental results. The maximum error obtained from verification between numerical and the modal testing is under 10%. So the FEA modeling is quite valid and can be used as the next evaluation visor stay bracket . The validated visor stay bracket FEM modeling can be used as next analysis such as FEM frequency respon analysis to reduce trial and error at development stage. And this will greatly help saving development time and costs.

Acknowledgment

The author would like to thank CAD/CAE ETC Indonesia facility in supporting this study & implementation.

References

[1] Peter McLeod, "The Availability and Capabilities of 'Low-End' Virtual Modeling (Prototyping) Products to Enable Designers and Engineers to ProveConcept Early in the Design Cycle", PRIME Faraday Technology Watch, Wolfson School of Mechanical and Manufacturing Engineering Loughborough University, Loughborough, 2001, 16-18

[2] C.Q.Liu and Ronald L. "Principles of Vibration Analysis, With Application in Automotive Engineering", Huston, SAE International, Warrendale, Pennsylvania, USA.2011, 449-450

[3] Andi Wibowo, Djoko Setyanto, "Modal and Stress Analysis of X71A Sport Motorcycle Framebody Virtual Testing Model based on Finite Element Analysis", International Journal of applied Engineering Research, ISSN 0973-4562 Volume 13, Number 18 (2018) pp. 13466-13474

[4] K.J. Bathe, E.J. Wilson, " Numerical Methods in Finite Element Analysis", Prentice-Hall, Englewood Cliffs, NJ, 1982.

[5] AETC, " Motorcycle Design & Simulations", CAD/CAE AETC Indonesia, 2016, 20-25

[6] Vittore Cossalter, "Motorcycle Dynamics", 2nd Edition 2006, 203-204.

[7] Erke Wang, Thomas Nelson, Rainer Rauch, " Back to Elements - Tetrahedra vs. Hexahedra", CAD-FEM GmbH, Munich, Germany,1-10

[8] Cornelia Thieme, " How to Create a Good Quality FE Model ", Msc Software. 2016, 3