International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 18 (2018) pp. 13466-13474

Modal and Stress Analysis of X71A Sport Motorcycle Framebody Virtual Testing Model based on Finite Element Analysis

Andi Wibowo[1], Djoko Setyanto[2]

Departement of Mechanical Engineering, Atma Jaya Catholic University of Indonesia.

Jl. Jenderal Sudirman 51 Jakarta 12930

Abstract

The X71A framebody is new model development of sport chasis motorcycle, designed for two pasangger vehicle. 3D CAD geometry is modeled using CATIA V5R22. Two 3D CAD model geometry using weld bead and without weld bead was built to get updating FEA model. SimDesigner V2015 is used to generate 3D solid mesh. These two both model were meshed with various mesh size. Finite Elemen Analysis was used to solve the modeling and simulation. Modal analysis of X71A framebody model was done using Msc Nastran 2014 SOL 103 normal mode solution to obtain eigenvalue and eigenvector. The frequency response analysis is performed using Msc Nastran 2014 SOL 111at 2G with range frequency 1-100Hz. Boundary constraint was used represent the actual condition. The load was given at the center of front and rear axle position using RBE2 element. Using frequency response analysis, 1st mode at 3Hz dominant frequency of framebody shown stress distribution at the resonance condition and used to determine strenght acceptable of the framebody. The use of FEA test model is very helpful to avoid trial and error at the development stage.

Keywords: Finite Element Analysis, Motorcycle framebody, Modal Analysis, Eigenvalues, Eigenvectors, Normal Mode, Frequency Response

INTRODUCTION

For many organizations the production of a physical prototype is an essential step in the process of developing a new product. However, a physical prototype often requires manual tooling, skilled hand assembly, delicate testing instrumentation and time spent interpreting prototype data. As such it represents a necessary but ultimately time consuming step in the development cycle. Engineers typically understand and incorporate what was learned from constructing and testing a prototype by revising the design, making a new prototype, and repeating the entire process[1].

Virtual product development includes all IT-supported, virtual product-model-based processes for the generation of a new product. Virtual product models are used to perform optimization and testing procedures in a virtual environment with the goal of saving development time and costs, while simultaneously increasing the product quality[2].

Framebody is very important part in Motorcycle. Maturity design framebody is the main key to avoid trial and error at durability testing.The high cost trial and error process should be avoided at this development process to reduce time to market in order to fill market demand quickly. One of the efficient proces is using Numerical Metode with Finite Element Analysis in the early development stage. The valid FEA model of the framebody will help the designer to reduce trial and error using real prototype that need high cost and takes along time to modify until comply with the design target. And the prediction of strenght criteria should be achieved before the real prototype is manufactured. By using FEA simulation testing the strenght cirteria can be achieved at the development stage. From the critical area at resonance condition, stress analysis can be obtained in the early step of development.

Numerical Metode Finite Element Analysis

The Finite Element Analysis or Finite Elemen Method (FEM) is a numerical approximation method. It is a method of investigating the behavior of complex structures by breaking them down into smaller, simpler pieces called elements. The elements are connected to each other at the nodes and the assembly of elements and nodes is called a finite element model[3]. A modern computational approach based on finite element analysis (FEM) can be use to help obtained the stress analysis and modal characteristic using numerical methode. Type of element of modeling becomes the important consideration in FEA modeling. In 3D solid meshing modeling, the 10-node tetrahedron element with reasonable mesh size has been used. And today 3D CAD solid models are typically meshed with quadratic tetrahedral elements[4].

METHODOLOGY

Modeling 3D using Catia V5 R25

The X71A framebody geometry is modeled with CATIA V5R25. The geometry of 3D CAD solid modeling can be seen in Figure 1.

Figure 1. 3D CAD geometry model of X71A framebody

All of the framebody component of 3D CAD model was unite together in one 3D CAD solid body in CATIA V5R25.

Model updating by mesh size generated and weld bead geometry refinement.

Meshing of 3D CAD solid framebody data is processed using SimDesigner V2015. By using SimDesigner V2015, mesh generation from the 3D solid Catia V5 will be easier and the software can be used as FEA pre processing software, since this software works in Catia working environment. In this paper, the framebody meshing is generated with meshing size 20,15,10,7,4 to evaluate the frequency of normal mode result distribution. Another model updating used for framebody FEA meshing, mesh was built from modified 3D data Catia as close as actual shape to improve the FEM results by adding 3D weld bead shape. The shape of the weld bead geometry can be seen in Figure 2.

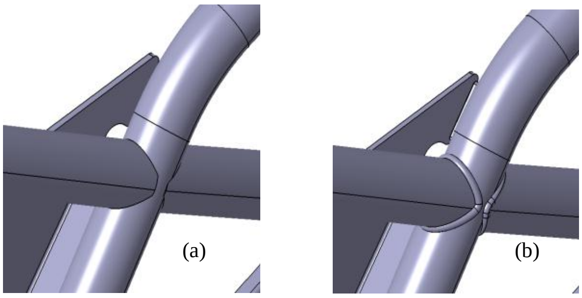

Figure 2. Comparation of X71A framebody prototipe geometry (a) without weld bead and (b) with weld bead

The modelling of framebody meshing without weld bead and framebody meshing with weld bead, were processed in variations of meshing size 20,15,10,7,4. In this paper both mesh size refinement and updating 3D weld bead are used in the FEA modeling to observe the comparation of both modeling and will be used for next FEA modeling.

Numerical Analysis of X71A Framebody using Finite Element Analysis

In this paper Finite Element Analysis was used to solve eigenvalue and eigenvector, and also used to perform frequency response analysis to solve the modal response in the range of frequency. The normal mode analysis is used to perform modal analysis for unconstrained framebody and constrained framebody which represent the actual condition. The eigen value and eigenvector is observed to evaluate all mode of constrained framebody. Base on this normal mode result, frequency response analysis is performed to obtain stress analysis of constrained framebody by frequency response analysis using excitation load.

Material of X71A framebody prototype is STAM 390 which has mechanical properties as table 1.

Table 1. Mechanical properties of X71A framebody

Property Material	Specification
Modulus Elastisitas	95000 Mpa
Poisson ratio	0.30000001
Shear Modulus	75000
Density	7.8599998E-009 Kg/mm^3

Modal Analysis of X71 Framebody

To obtain the modal characteristic of existing X71A framebody, FEA modal analysis was done using Msc NASTRAN 2014 SOL 103 normal mode solution. The natural frequencies and mode shapes are found. The eigenvalue results obtained from FEA simulation is observed to perform next evaluation. This Msc NASTRAN 2014 SOL 103 normal mode solution is also used for unconstrain framebody and constraint model X71A framebody analysis. In normal mode constrain model, boundary condition of the real unit motorcycle is used to ensure the system is represent real condition.

Frequency Response Analysis

Next evaluation used frequency response analysis which used acceleration load at range frequency. Frequency response analysis is performed using Msc Nastran 2014 SOL 111. The acceleration load was used 2G and the range frequency is used within 1 Hz - 100 Hz. Base on motorcycle road load excitation usualy with undulation road within 1-80 Hz[5] and road load excitation for framebody simulation is under 100 Hz[6], this frequency range excitations will produce resonance condition at the framebody, and only related mode shape observed for the stress analysis result from frequency response will be analyzed.

Table 2. Nodal & Element of Tet10 mesh size 20.15.10.7.4 using SimDesigner V2015.

Deviation	Type	Mesh size				
		m20	**m15**	**m10**	**m7**	**m4**
Without bead	Nodal	57,036	76,842	137,537	254,143	746,700
	Element	28,291	38,241	68,910	127,984	388,876
With bead	Nodal	66,090	86,851	156,734	273,267	780,336
	Element	33,122	43,550	79,031	137,807	404.714

RESULTS AND DISCUSSION

Finite Element Model Updating Result

The meshing result of X71A framebody using simDesigner V2015 can be seen at Figure 3.

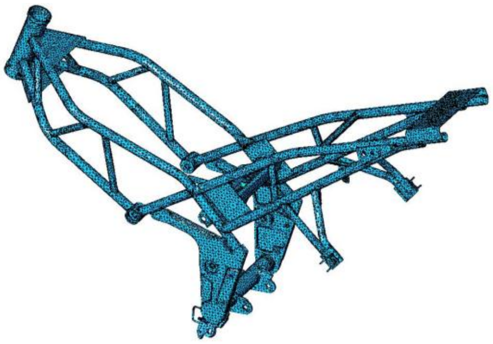

Figure 3. Mesh Modeling Using Simdesigner V2015

The eigenvalue of the 3D solid framebody are calculated using 10-node tetrahedron element using Msc Nastran 2014. The meshing is generated using Msc SimDesigner V2015, total number of nodes and the total number of elements create with meshing size 20,15,10,7,4 is found to be shown in the Table 2.

Table 2 provides comparison the number of nodes and elements of some mesh size evaluation alternatives. The smaller the mesh size will give a better geometric shape approach to the area that needs detailed mesh geometry. On the other hand the smaller the mesh size it will take longer time in computation process of the analysis. The meshing also updated with the bead geometry and without bead gemetry. The geometry shape comparation of weld bead geometry can be seen at figure 4.

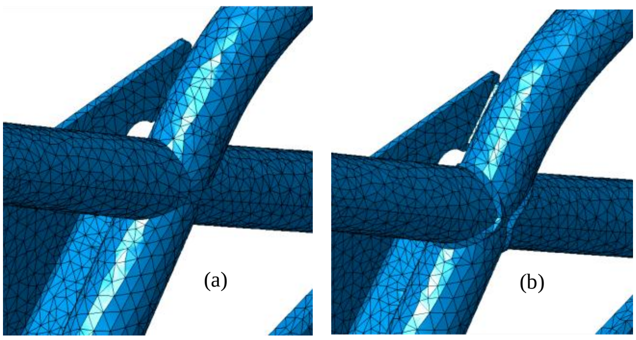

Figure 4. Comparatio of Tet10 Mesh using sim Designer : (a)without weld bead and (b) using weld bead

In the figure 4, clearly seen the shape of meshing result from 3D framebody with weld bead is more represent the actual shape of real framebody.

Normal Mode Analysis result using SOL 103

The Normal mode analysis is performed using Msc Nastran 2014 SOL 103, the analysis was done with the variation of meshing size, without and with weld bead. In the first FEA Modeling analysis, Finite Element Analysis is done by running normal mode solution to the model without weld bead. The meshing preparation was completed with simDesigner V2015 in 3D solid modeling meshing using tet 10 element with mess size 20, 15, 10, 7, 4, and the Msc Nastran 2014 normal mode analysis result are tabulated in Table 3. By comparing the results of normal mode eigenvalue results of FEA. Obviously seen distribution eigenvalue after 4th mode of variation mesh size result has bigger differences at 5^{th} 6^{th}, 7^{th}, 8^{th}, 9^{th} mode than modes under 5^{th} mode. The mesh with bigger size indicate has bigger eigenvalue result especially after 4^{th} mode.

Table 3. Nomal Mode Analysis result Tet10 Mesh modeling without bead

Mode No.	Normal Mode				
	m20	m15	m10	m7	m4
	f (Hz)	f (Hz)	f (Hz)	f (Hz)	f (Hz)
Mode 1	89.60	86.24	87.43	85.44	82.97
Mode 2	121.22	117.03	118.84	116.88	113.80
Mode 3	121.58	120.17	121.54	120.95	118.69
Mode 4	124.30	123.72	123.64	122.52	120.22
Mode 5	154.08	151.84	148.00	144.34	140.48
Mode 6	167.35	161.20	156.57	151.64	148.87
Mode 7	185.47	180.71	177.34	173.35	168.55
Mode 8	208.67	197.72	197.17	189.65	187.92
Mode 9	219.95	216.11	213.82	207.15	201.68
Nodal	57,036	76,842	137,537	254,143	746,700
Element	28,291	38,241	68,910	127,984	388,876

Comparation of normal mode in nine natural frequency analysis result of FEA mesh variation framebody without weld bead at each mode can be seen at figure 5.

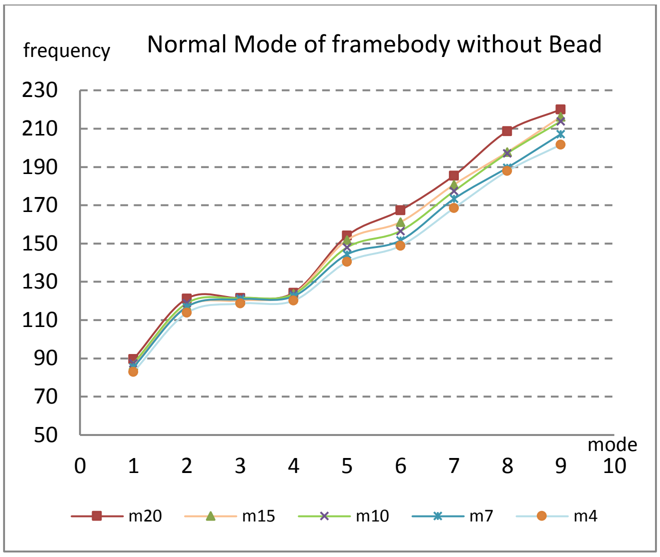

Figure 5. Comparation of normal mode result of nine natural frequency of FEA mesh variation framebody without weld bead

Figure 5 shown the spread frequency value of each mode. Although the spread after 4^{th} mode is bigger than the spread at the lower mode, but both was relatively still small and will not affected since the analysis will observed under 100Hz. The FEA model of framebody without weld bead is quite good enough.

For the other model, to get the result more represent the actual conditions of the prototype framebody. 3D modeling is made just same with weld bead shape in actual conditions. 3D Meshing model with weld bead adjustment was built and performed with normal mode analysis, and obtained eigenvalue analysis results shown in table 4. It shows for various meshing size that after 4^{th} mode. The distribution result trend of framebody without weld bead almost same with the framebody modeling using weld bead normal mode result .

Table 4. Nomal Mode analysis result Tet10 Mesh modeling with bead

Mode No.	Normal mode				
	m20	m15	m10	m7	m4
	f (Hz)	f (Hz)	f (Hz)	f (Hz)	f (Hz)
Mode 1	88.25	86.57	87.62	85.92	83.87
Mode 2	108.82	115.79	117.80	116.36	113.79
Mode 3	114.54	118.73	119.75	119.27	117.02
Mode 4	126.75	125.62	125.81	124.64	122.42
Mode 5	154.36	151.25	147.08	143.51	139.70
Mode 6	166.28	163.70	158.12	153.75	149.70
Mode 7	181.60	178.95	174.99	171.32	166.46
Mode 8	209.44	193.61	193.31	186.19	185.53
Mode 9	218.04	212.66	211.06	208.19	203.34
Nodal	66,090	86,851	156,734	273,267	780,336
Element	33,122	43,550	79,031	137,807	404,714

Comparation of normal mode analysis result in nine natural frequency of FEA mesh variation framebody with bead at each mode can be seen at figure 6.

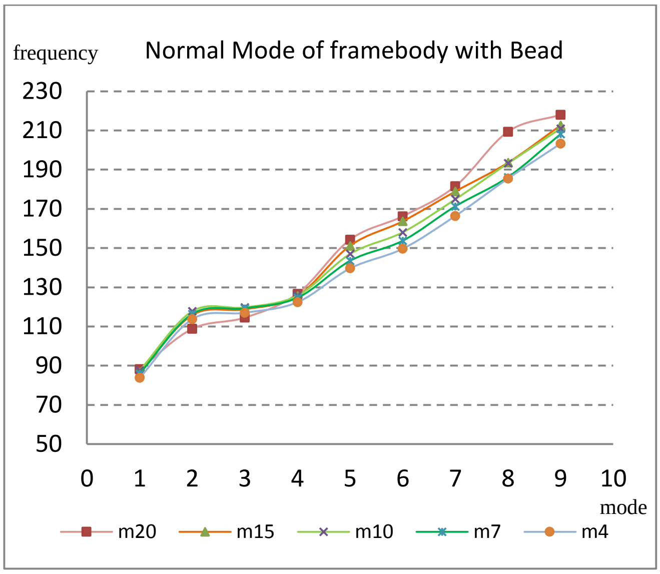

Figure 6. Comparation of normal mode in nine natural frequency of FEA mesh variation framebody with bead

In the other hand, In the lower than 5th mode, the spreads at each mode of all mesh size of the modelling, shows that the modeling with bead or without bead give small diferences.

Both normal mode result shown that refinement mesh into smaller mesh size produce more detail modeling geometry and influenced the normal mode result become into more lower value. And the modelling framebody with bead or without bead give almost same result. To determine which modeling is better and represent the real natural frequency in the real prototype, both modeling should be validate with the real protototype by using Experimental Modal Analysis.

Since both the modeling framebody with bead or without bead give almost same result. Both modeling can be used for next evaluation. Spread comparation at each modeling, shown that with changes the meshing size to the mesh size 4 both modeling has bigger spread than mesh size 7. Then for the next frequency response analysis used mesh size 7.

The eigenvector FEA normal mode result mesh Size 7 of X71A framebody using weld bead can be seen also in the figure 7.

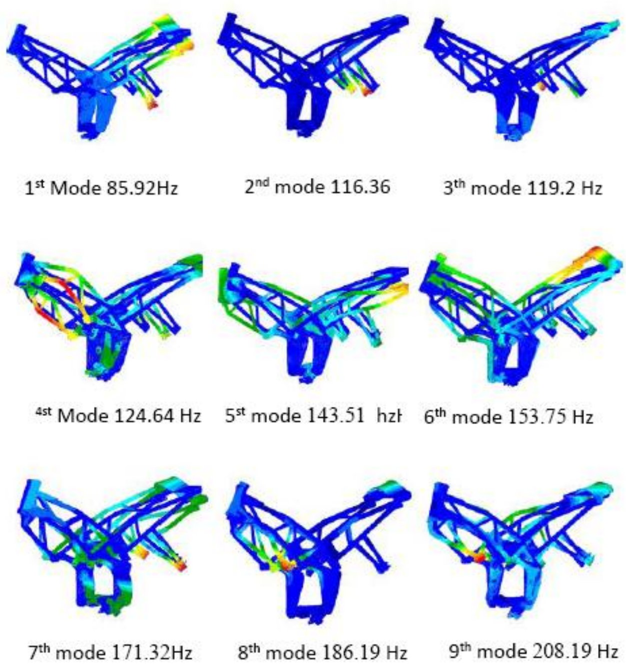

Figure 7. Eigenvector from normal mode result framebody with weld bead.

Normal Mode Analysis result of constrained framebody modeling using SOL 103

The Normal mode analysis for constrained framebody is performed, the analysis used mesh size 7 with weld bead base on the previous normal mode analysis and real condition after manufacture.

To represent the load of the actual condition, contrains system at the X71A framebody is applied at the framebody. The excitation of load is given at the axle at the front fork X71A framebody model. To simplified the constrained structure analysis but still represent the actual condition, excitation of the axle is received at the center of suspension, RBE2 is used to conect the axle to the framebody steering steem system. And at the front and rear pivot axle, the boundary is given revolute joint in order to allow y axis or lateral movement.

The contrains system of the frequency response can be seen at figure 8.

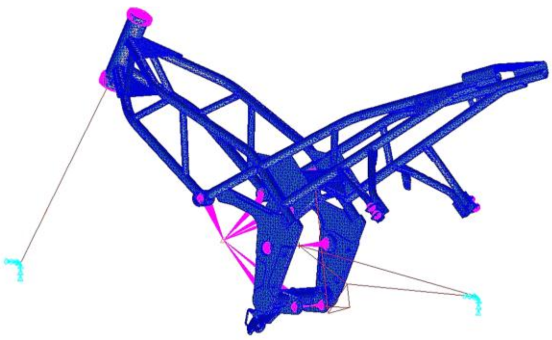

Figure 8. Boundary Condition of constrained framebody

The properties of boundary condition of contstrained framebody are shown in the table 5.

Table 5. System property used for frequency response

system	property	input
Front Fork	Spring Coeff.	9.42 N/mm
	Damping Coeff.	0.05 N.s/mm
Rr Cushion	Spring Coeff.	88.1 N/mm
	Damping Coeff.	4.1 N.s/mm

For 27 kg mass is used for represent the engine. And located in the engine mounting position. The Fr fork and Rear Cushion was used 1D Spring Element.To connect pivot center of framebody and center of load was used 1D Element beam – rigid element. The eigen value result of normal mode constrained framebody modeling using Msc Nastran 2014 SOL 103 can be seen at table 6.

Table 6. Nomal Mode Analysis Tet10 Mesh modeling with bead constrained framebody

No.	Mode	Frequency
1	1 st mode	3 Hz
2	2 nd mode	55.279 Hz
3	3 th mode	91.347 Hz
4	4 th mode	107.77 Hz
5	5 th mode	118.67 Hz

From the normal mode analysis of contsrained framebody is only two mode, first was 1st mode at 3 Hz and second at 2nd mode at 55.279 Hz, were in the road load frequency range and can be seen at table 5. Even so the stress analysis should obtained from the frequency response analysis. The eigenvector result of normal mode constrained framebody modeling using Msc Nastran 2014 SOL 103 can be seen at figure 9,10,11,12,13.

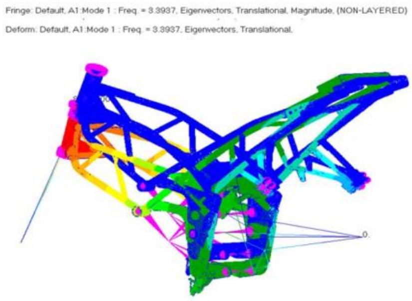

Figure 9. 1st Mode 3 Hz

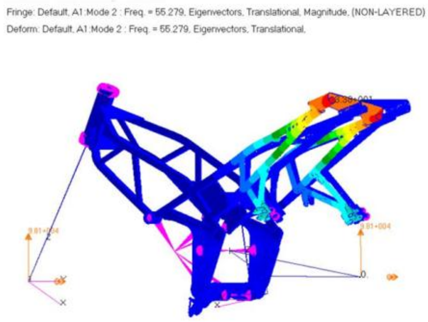

Figure 10. 2 nd mode 55.279 Hz

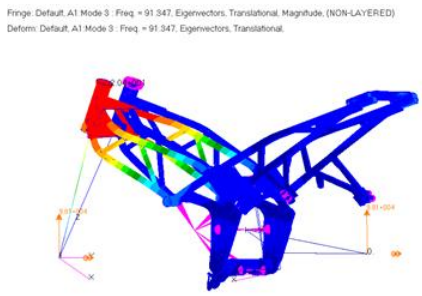

Figure 11. 3th mode 91.347 Hz

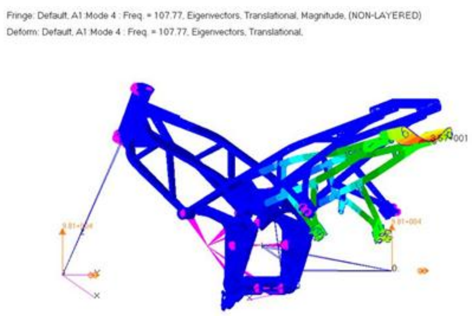

Figure 12. 4th mode 107.77 Hz

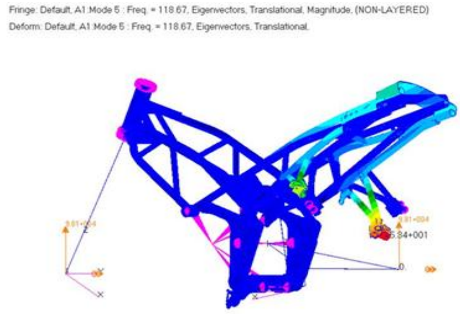

Figure 13. 5 th mode 118.67 Hz

Stress Analysis of Modal model using Frequency Response SOL 111

The frequency response of constrained framebody is performed, the analysis was used mesh size 7 with weld bead. The frequency response analysis X71A Framebody was done using Msc Nastran 2014 SOL111 using frequency response solution. The frequencies is given from 1 sd 100 Hz and the acceleration load is 2 G at two position at the center of front and rear axle at Z direction. The position load at the framebody can be seen at figure 14.

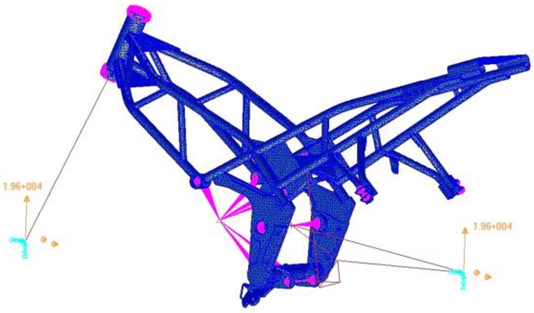

Figure.14. Load Boundary Condition of constrained framebody

The measurement point is picked up at two position in order to know the response when the load applied. First measurementat the steering steem and the socond at the rear cross plate. The result of frequency response at two point measurement can be seen at figure 15.

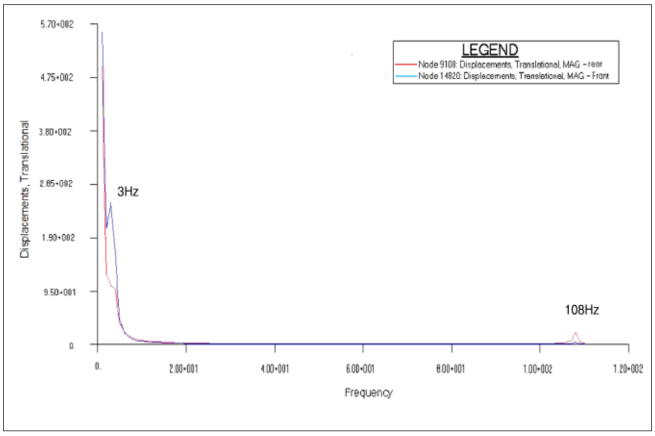

Figure.15. Displacement graph of Frequency response result using 0 Hz -100Hz with input acceleration 2G

In the figure 15, It shows dominant response 1st Mode at 3 Hz. This resonant mode is used for strenght evaluation for single durability testing. The mode shape of frequency response 1st Mode at 3 Hz can be seen at figure 16.

From the frequency response of 1st Mode 3 Hz the critical area stress contour and the max and min principle stress can be obtained, and compared with the framebody single durability testing standard criteria.

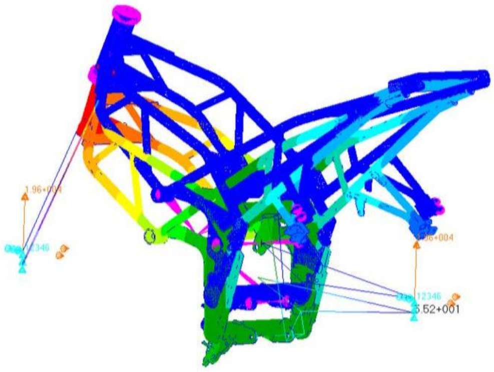

Figure 16. X71A framebody 1 st resonance mode at 3Hz

The position of the critical area can be seen at figure 17. This result was simetrically on right side and left side.

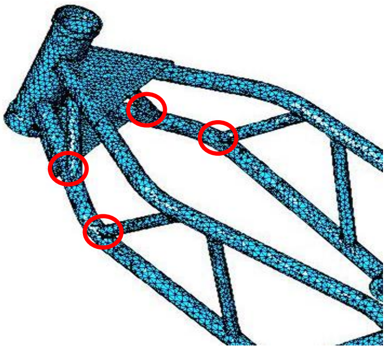

Figure 17. Max principle stress and min principle stress area of 1st Mode at 3 Hz left side

The position of the max principal stress of critical area of frequency response 1st Mode at 3 Hz left side can be seen at figure 18. And also the position of the max principal stress of critical area of frequency response 1st Mode at 3 Hz right side can be seen at figure 19.

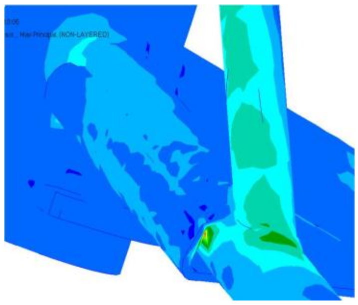

Figure 18. X71A framebody Max principle stress of 1st Mode at 3 Hz left side

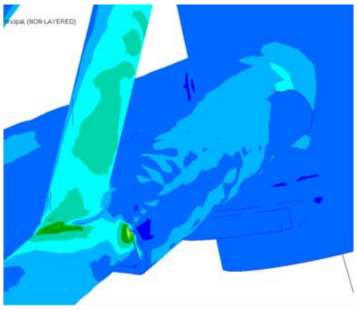

Figure.19. X71A framebody Max principle stress of 1st Mode at 3 Hz at right side

The position of the min principal stress of critical area of frequency response 1st Mode at 3 Hz left side can be seen at figure 20. And also the position of the min principal stress of critical area of frequency response 1st Mode at 3 Hz right side can be seen at figure 21.

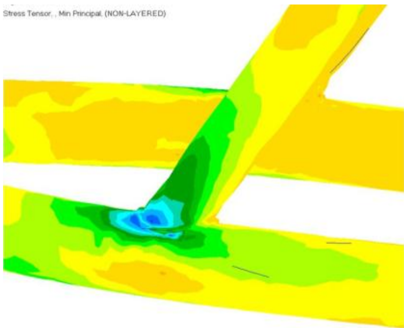

Figure 20. X71A framebody Min principle stress of 1st Mode at 3 Hz left side

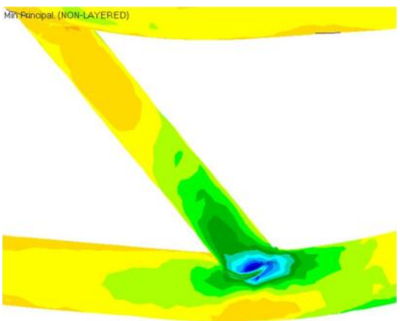

Figure.21. X71A framebody Min principle stress of 1st Mode at 3 Hz right side

Max Principle stress of frequency response 1st Mode at 3 Hz right side and left side is shown 193.37 Mpa and min principle stress of frequency response 1st Mode at 3 Hz right side and left side is shown -151.66 Mpa.

The stress evaluation of single durability testing should not not exceed 200 Mpa[5]. This value already meet with the strenght criteria of single durability testing framebody sport motorcycle. This stress still also under yields strenght of STAM 390 is 235 Mpa with standard designation JIS G3472:1988[7].

CONCLUSION AND RECOMMENDATIONS

This paper presents Finite Element Analysis with model updating of X71A motorcycle framebody to obtain modal analysis and stress assesment of the model. Finite Element Analysis modeling of the X71A framebody has been done, and it has a good and quite satisfactory result. Meshing size refinement contribute to improving FEA normal mode result and accuracy of critical strees position. The addition of weld bead in 3D modeling will increase the stiffness of the modeling and can be used to improve the modeling also, Since both the modeling framebody with bead or without bead give almost same result. Both modeling can be used for next evaluation. Evaluation using weld bead modeling is selected since the model has same shape with real condition.

The critical area 1st mode at 3Hz obtained from the frequency response has quite good result. Max Principle stress right side and left side is shown 193.37 Mpa, and min principle stress right side and left side is shown -151.66 Mpa and still meet with the design required. By using frequency respone analysis, early prediction testing assesment lead to guide critical area position and stress allowed will reduce trial and error using prototype at development stage. And this will greatly help saving development time and costs.

As future work to be done on this X71A Sport Motorcycle Framebody is verification of Frequency Response Function of actual framebody. To determine which modeling is better and represent the real natural frequency at each mode real prototype after manufactured, both modeling should be validated. The prototype should be built in order to perform experimental modal analysis (EMA) eg. impact hammer to find natural frequency of the real framebody. The result of Experimental Modal Analysis impact hammer can be use to validate the eigen value result of finite element analysis. Validation to EMA result will indicate the value error of finite element analysis allready build. And better model updating can be choose base on this experimental result.

ACKNOWLEDGEMENTS

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

REFERENCES

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

[2] Hirz. M. ; Dietrich. W.; Gfrerrer. A.; Lang, 2013, "Integrated Computer Design in Automotive

Development. Development Processes. Geometric Fundamentals. Methodes of CAD. Knowledge-Base engineering Data management". pp. 29

[3] Donald Cristensen, 2011," Introduction to Finite Element Analysis for University Courses and Research". pp. 4

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

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

[6] ETC, 2016, "Motorcycle Design & Simulations", CAD/CAE AETC Indonesia. pp. 20-25

[7] John E. Bringas. Handbook of comparative World Steel Standards.ASTM DS67A, 2nd Edition