IJASRM
International Journal of Advanced Scientific Research and Management, Vol. 1 Issue 3, March 2016.
www.ijasrm.com
ISSN 2455-6378

Finite Element Modal Analysis and Mesh Optimization of a Typical Turbo Fan Engine – Fan Hub Frame

Charles.G.Martin[1] and Dr. A. Arokkiaswamy[2]

[1,2] Department of Aeronautical Engg, DSCE, Shavige Malleshwara Hills, Bangalore- 560078, India

Abstract:
This technical paper investigates a the procedure to mesh a complex component such as Fan Hub Frame (FHF) as shown in the Fig 1 in specified time frame of 8hrs .This component has more than 250 holes ,curved extrusions and problematic geometry. Various iterations were carried out and the geometry simplifications were done in cad tools namely CATIA-V5-by removing the holes, fillets and replacing some of the problematic geometry, by considering mass, stiffness and volume deviation to generate mesh in Ansys Workbench within the specified time limit for Modal Analysis. Finally First order and Second order tetrahedral mesh is compared with the test results.

.***Keywords:*** *Modal Analysis, Fan hub Frame, Ansys Workbench, Free vibrational Analysis.*

1. Introduction

. The component used in this project is FHF- which is the outer skeletal structure of the turbofan engine. FHF is the main structure of the turbofan engine which is connected directly to the aeroplane wing and it consists of stator aerofoil and a hollow conical structure. The main objective of the present work is to find a proper methodology to mesh the whole component in specified time limit for modal analysis.

The main aim is to capture the geometry of the component effectively while meshing and to achieve all the specified boundary conditions. Since the component is very complicated (with variable fillets and many holes) which makes the meshing difficult and takes lot of computational time to generate a mesh.[1]

The component was meshed in HYPERMESH it took almost 80hrs and the whole purpose of the project is to mesh the component within 24hrs and the maximum node count to be maintained is 250000 without compromising on result accuracy of modal analysis (natural frequencies and mode shapes)[2-10].

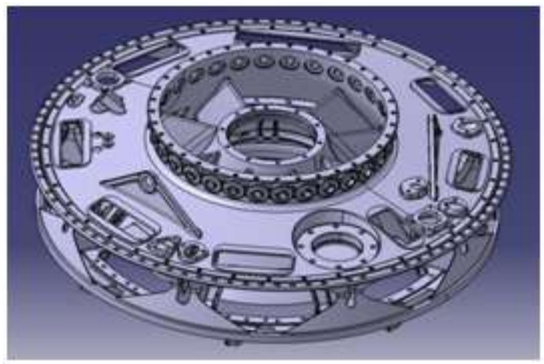

Fig 1 Fan Hub Frame

2. Flow Chart

The whole project procedure is briefly explained in the flow chart as shown in the Fig 2

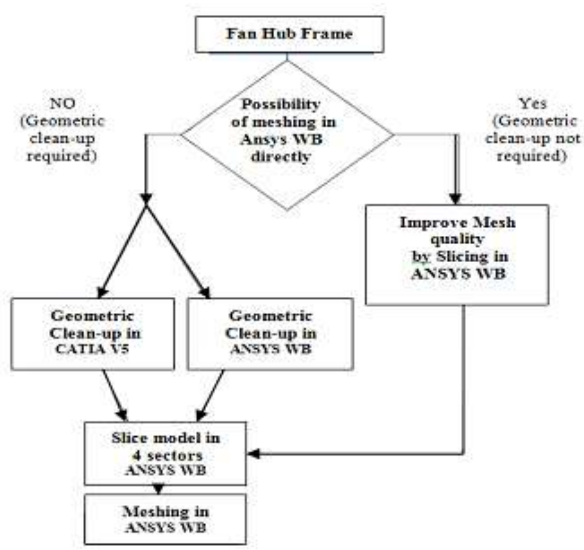

FIG 2 FAN HUB FRAME -FLOW CHART

3. Problematic Geometry Check Methodology

3.1 Possibility of meshing directly in ANSYS WB by checking the model visually.

ISSN 2455-6378

3.1.1 Check 1:

Check for presence of multiple splits on surfaces and fillets (as mentioned in Fig 3,4,5,6).If these split surfaces are present, most likely Ansys Work bench will fail to generate a mesh.

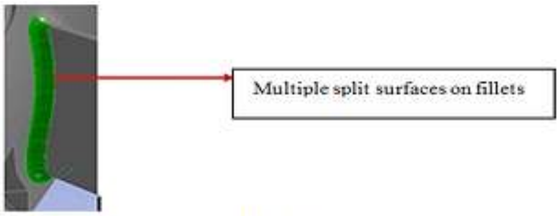

Fig 3 Multiple spilt in fillets

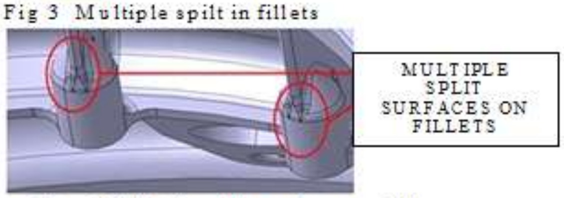

Fig 4 Multiple split surfaces on fillets

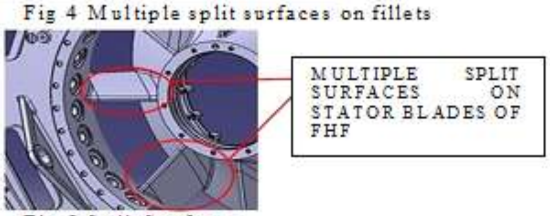

Fig 5 Split Surfaces

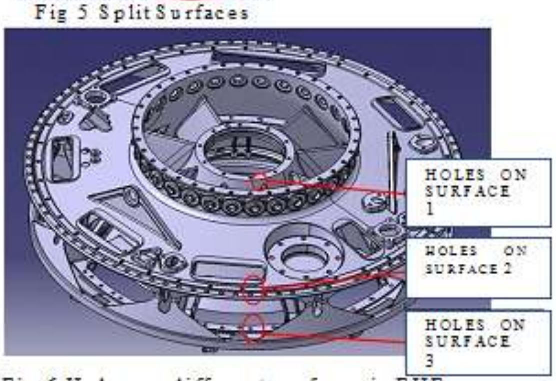

Fig 6 Holes on different surfaces in FHF

3.1.2 Check 2:

Check for the presence of many holes as shown in Fig 3,4,5 &6 and other features like extrusion, brackets ..Etc. on each surface of FHF, which creates sharp edges and uneven cuts on the holes when a slicing is done on the model.

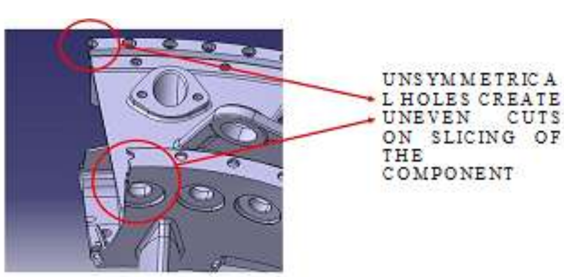

Fig 7: Unsymmetrical sliced holes

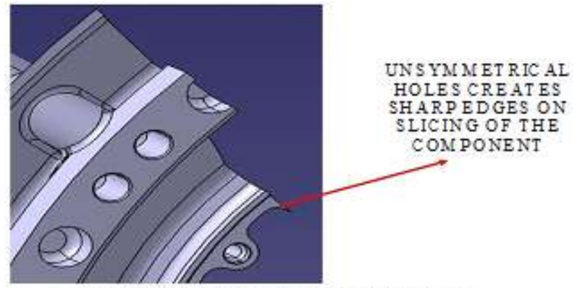

Fig 8 Sharp Edge formed during slicing

If any of above mentioned feature check are not satisfied then, most likely Ansys Work bench will fail to generate a mesh.

3.2 Alternate Check (Quick Check)

However, if somebody would like to check still the possibility of mesh in Ansys WB, then the following quick check can be done (Fig 9)

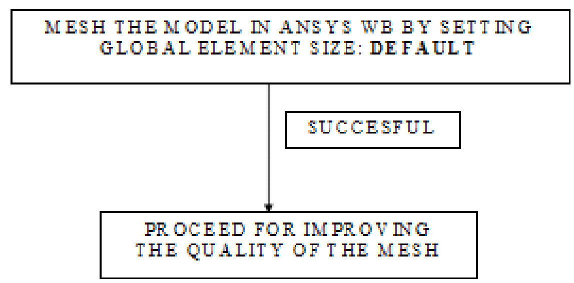

Fig 9 Flow chart of alternative check

4. Geometry Clean Up in CATIA V5

4.1 Hole filling operations

- Pad operation
- Surface extract operation

4.1.1 Pad operation (Extrusion)

- Pad operation is used to fill all holes on a surface (surface should be free from projections on the bottom of the surface)
- For further understanding refer Fig 10& Fig 11
-

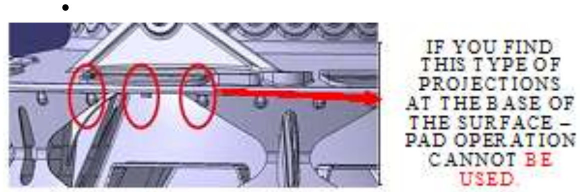

Fig 10 Projections below the surface of the model

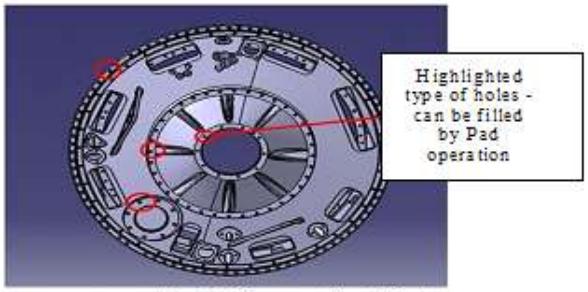

Fig 11 Symmetrical Hole

4.1.2 Surface extract operation

- When there are complicated holes or pocket with variable thickness as shown in Fig 12 then the pad operation for filling the holes cannot be used, therefore one as to use surface extract operation to eliminate the complicated hole.
- The extracted hole using is shown in the Fig 13
- After the surface extraction if the holes are symmetric as shown in Fig 12 &13, then one hole can be manually filled using surface extract and then circular pattern can be used to fill all the holes in one operation.
- If the holes are not symmetric then each hole should filled manually.

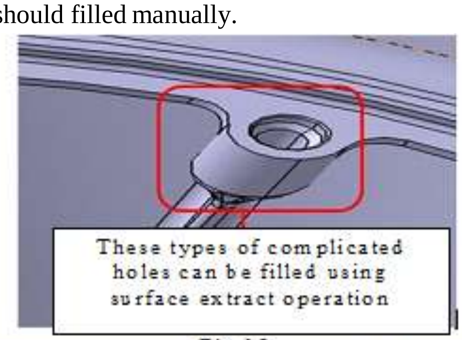

Fig 12

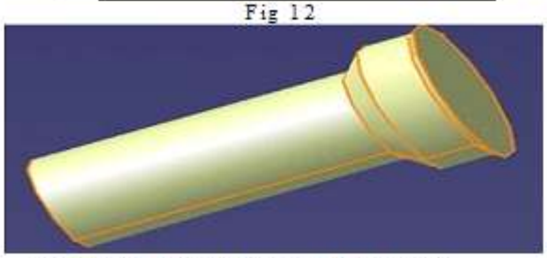

Fig 13 Surfaces Extracted from a hole

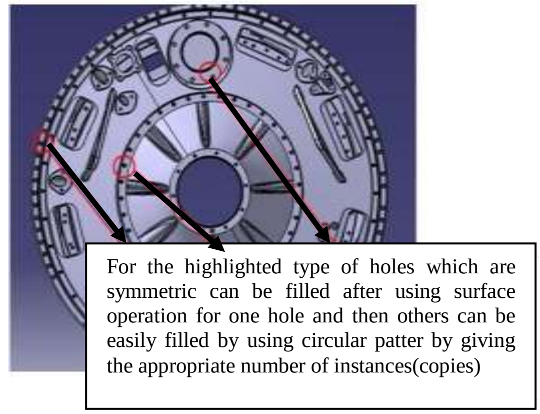

Fig 14 Symmetric Hole.

5. Geometry Correction by Replacement of Alternate Geometry

4.1 TYPE I

When the Problematic geometries of regular shapes are found as shown in the Fig 15 a then it must be replaced with alternative geometry as shown in Fig16 to make meshing process smooth and effective on these type of geometries.

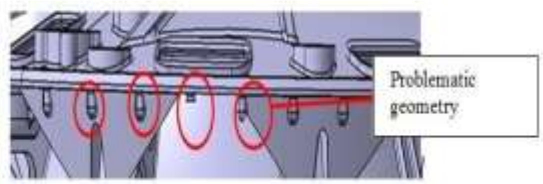

Fig 15

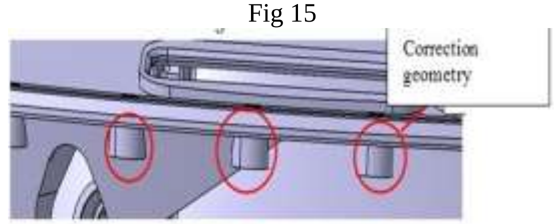

Fig 16 Correction for Type

4.2 TYPE II

- When there are cut edges on the circular edges of a solid extrusion or whole geometry, it becomes a problematic geometry, which in-turn makes ANSYS WB - impossible to mesh such surfaces.
- Therefore, alternative geometry should be replaced as shown in Fig 17

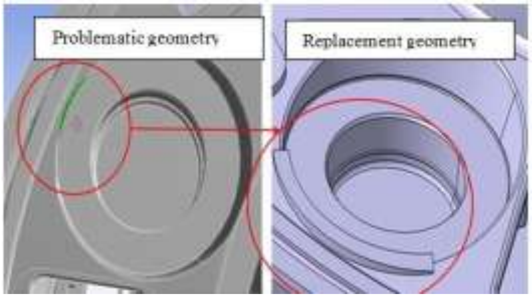

Fig 17 Problematic and Alternative replacement geometry

6. Meshing Procedure in ANSYS WB

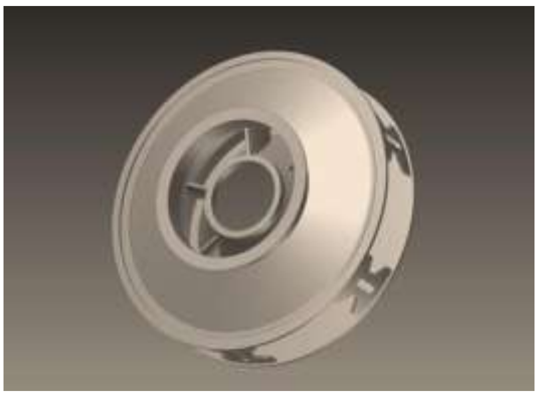

Fig 18 FHF after geometry Clean-Up

- The fully cleaned up geometric model as shown in the Fig 18.
- Before starting the meshing we should slice the component into 4 sectors as shown in the Fig 19.

6.1 *Why slicing is done?*

- Slicing is use to simplify the model and to divide it into many sub parts which in-turn reduces the mesh computational time.
- Each part can be meshed efficiently and effectively when the model is divided.
- Even computer takes less time comparatively to generate a mesh when divided than considering the whole has single volume.

6.2 *Important steps for creating a plane for slicing .*

- Selecting the plane is the main part in the FHF meshing methodology after geometry clean-up.(Fig 19)
- Plane should be created in such a way that it should not cut any of the features creating sharp edges and complicated surfaces
- So for this operation plane from face method is used to create a plane.

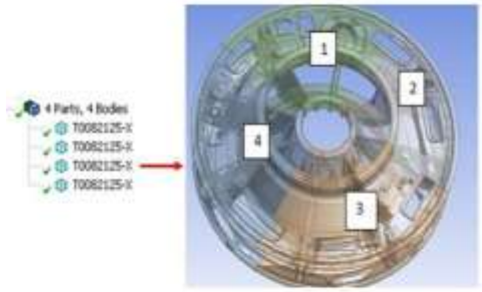

Fig 19 Final sliced 4 sector FHF Model

- First by setting global element size: Default try to mesh and if the mesh is successfully generated, then go for further iterations by changing the global element size : 10 and 20 to get further mesh refinement.
- After trying all the types of element size select the best global element size which can produce a good mesh with least node count.
- 1st order mesh-Using Element size:10 (Fig 20)
- 2nd order mesh–Using Element size:20 Medium Coarse (Fig 21)

Table 1: 1ST Order Mesh Count

TYPE	1ST ORDER
NODES	187497
ELEMENTS	621699

Table 2: 2ND Order Mesh Count

TYPE	2ST ORDER
NODES	513409
ELEMENTS	288801

Fig 20 1st order mesh-Using Element size:10

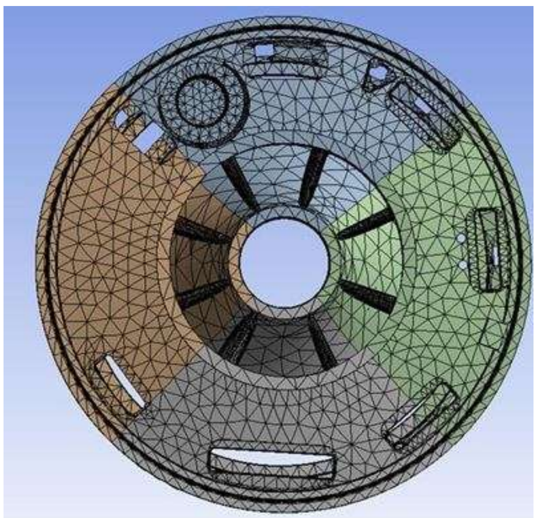

Fig 21 2nd order mesh–Using Element size:20 Medium Coarse

6.3 *Mass Deviation*

$$\frac{Final\ Mass - Original\ Mass}{Original\ Mass} * 100$$

$$\frac{94.49 - 84.118}{84.118} * 100 = 8.7\%$$

Mass Deviation=8.7%

6.4 *Volume Deviation*

$$\frac{Final\ Volume - Original\ Volume}{Original\ Volume} * 100$$

$$\frac{1.89x10^7 - 2.055x10^7}{2.055x10^7} * 100 = 8.7\%$$

Volume deviation: 8.7%

7. Modal Analysis Results Report

7.1 Material Used:

Name	Titanium	
General	Mass Density	4.51 g/cm^3
	Yield Strength	275.6 MPa
	Ultimate Tensile Strength	344.5 MPa
Stress	Young's Modulus	102.81 GPa
	Poisson's Ratio	0.361

7.2 Boundary condition.

a) The Exhaust Face Fixed (as mentioned in the figure 22 below) for conducting Modal analysis.

b) No of Modes of extraction :10

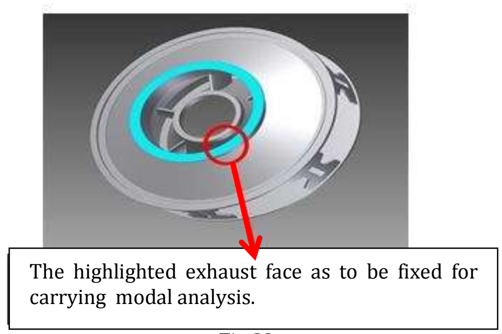

Fig:22

7.3 Time evaluation.

Table 3 : Time Evaluation of Analysis

Time evaluation	Hrs (approx)
Time taken to geometry clean-up	5
Time taken for meshing	3
Time taken for Analysis	2
Total time taken	10

8. Results and Discussion

- Results for 10 mode shapes in Modal analysis were solved and the following results were tabulated .
- The time estimation for the analysis is 30 mins
- Frequency(Hz) Vs Modes graph is plotted as shown in the Fig 23

ISSN 2455-6378

Table 4

Properties	
Volume	1.4519e+007 mm³
Mass	67.078 kg
Centroid X	174.04 mm
Centroid Y	9.7724e-002 mm
Centroid Z	-0.14496 mm
Moment of Inertia	1.0677e+007
Moment of Inertia	6.1241e+006
Moment of Inertia	6.1153e+006
Statistics	
Nodes	129889
Elements	65794

Table 5 Modal Analysis tabulation

Mode	Frequency [Hz]
1.	157.14
2.	157.25
3.	163.51
4.	163.76
5.	178.58
6.	181.51
7.	275.86
8.	276.97
9.	305.34
10.	308.2

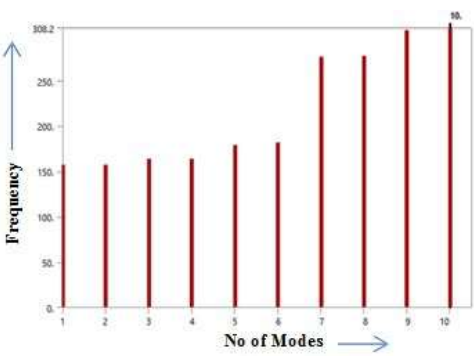

Fig 23 Frequency v/s No of Modes.

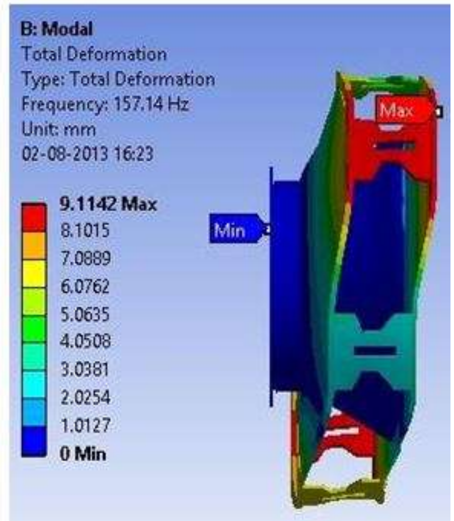

Fig 24 Total Deformation

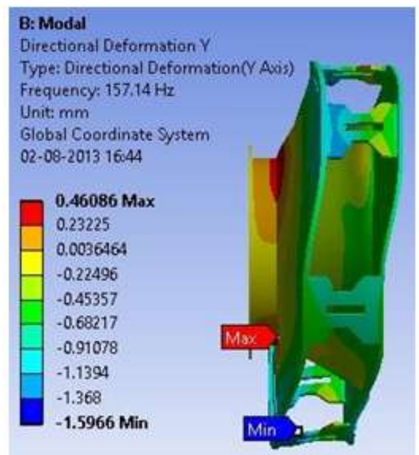

Fig 25 Directional deformation(Y axis)

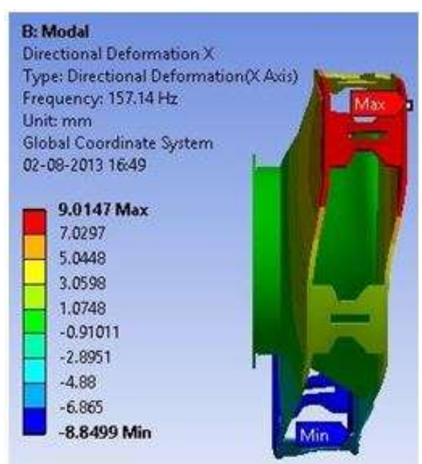

Fig 26 Directional deformation(X axis)

International Journal of Advanced Scientific Research and Management, Vol. 1 Issue 3, March 2016.
www.ijasrm.com
ISSN 2455-6378

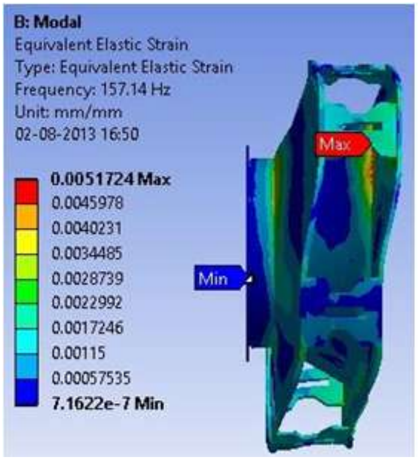

Fig 27 Equivalent Elastic Strain

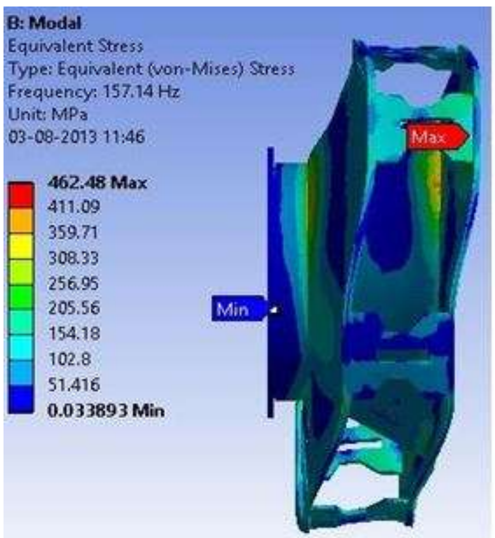

Fig 28 Equivalent Elastic Stress

Table 6 ANSYS WORKBENCH RESULTS

DIRECTIONAL DEFORMATION		
Orientation	X AXIS	Y AXIS
	Minimum	Maximum
Results	0mm	9.1142 mm
	-1.5966 mm	0.46086 mm
	-8.8499 mm	9.0147 mm
TOTAL DEFORMATION		
Minimum	Maximum	
0mm	9.1142 mm	
EQUIVALENT ELASTIC STRAIN		
Minimum	Maximum	
7.1622e-007 mm/mm	5.1724e-003 mm/mm	
EQUIVALENT STRESS		
Minimum	Maximum	
3.3893e-002 MPa	462.48 MPa	

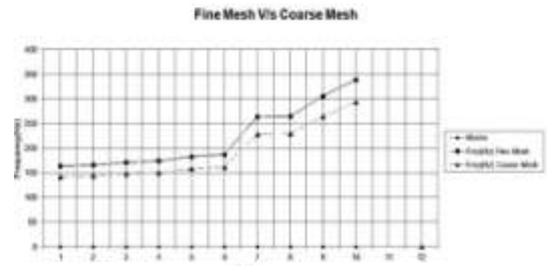

Fig 29 Fine Mesh V/s Coarse Mesh Comparison

8.1 Graphical Comparison

Comparison Of Fine Mesh V/S Coarse Mesh.

From the plot Fig 29 , It can justify that fine mesh (Element Size: 10-20) when compared to Coarse mesh (Element Size:20-30) . The results are discussed and compared with those obtained using a very fine mesh of solid finite elements, which results in better result accuracy when compared to the corase mesh.

Comparison Modes v/s Frequency for Variable stator airfoil thickness Fig 30

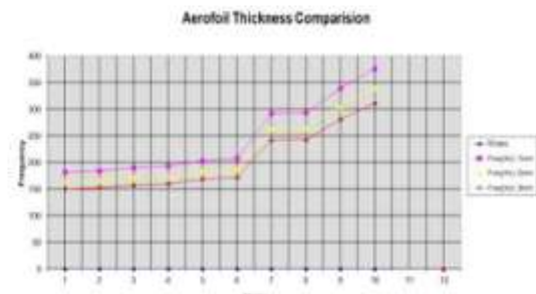

Fig 30 Modes v/s Frequency for Variable Stator airfoil thickness

Comparison was done with the ANSYS Results V/S Test Bench Results As shown in the Fig 31.

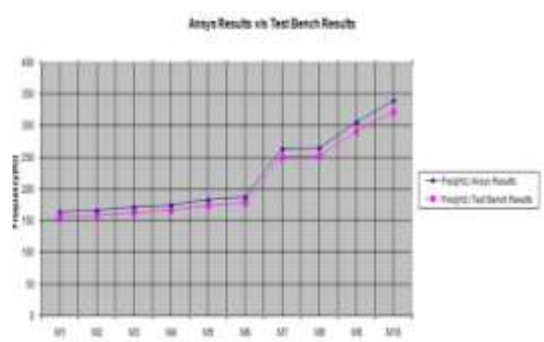

Fig 31 ANSYS Results v/s Test bench results

9. Conclusion

In this paper geometric clean-up iteration was carried out on a complex FHF component, in order to achieve a proper meshing for structural analysis within the specified computational time limit.

The modal analysis results obtained for coarse mesh and the fine mesh were compared with the standard test bench results and the significant observations are as follows:-

- The mesh operation was executed using fine mesh (Element Size range of 10- 20mm)
- Second order tetrahedron, 3-D elements were used.
- Meshing operation was carried out within specified computational time limit.
- It is observed that the frequency was between 157.14-308.2Hz by considering 10 modes.
- Maximum Deformation was observed to be 9.1142 mm
- The difference between the ANSYS WB results and the standard test bench results is 5% which is within the acceptable design limit.
- The total computational time was significantly reduced from 80hrs to 10hrs.

References

[1] ANSYS Workbench Tutorials. http://www.ansys.com/Industries/Academic/Tools/Curriculum+Resources/Tutorials,+Example s+&+Curriculum

[2] Aminbaghai M, Murín J and Kutiš V, Modal analysis of the FGM-beams with continuous transversal symmetric and longitudinal variation of material properties with effect of large axial force. Engineering Structures, Volume 34, Pages 314-329, January (2012)

[3] Banwell G, Mohr S, Rothberg S and Roberts J, Using experimental modal analysis to validate a finite element model of a tennis racket. Procedia Engineering, Volume 34, Pages 688-693, (2012).

[4] Chen S, An equivalent direct modelling of a rotary shaft with hot-fit components using contact element modal analysis results. Computers & Mathematics with Applications, Volume 64 (5),Pages 1093-1099, September (2012).

[5] Li H, Li P and James Hu S, Modal parameter estimation for jacket-type platforms using noisy free-vibration data: Sea test study. Applied Ocean Research, Volume 37, Pages 45-53, August (2012).

[6] Li Y, Wang C and Wang R, The thermal stress analysis and structure optimum of neck tube with vertical cryogenic insulated cylinders based on ANSYS. Nuclear Engineering and Design, Volume 252, Pages 144-152, November (2012).

[7] Murin J, Aminbaghai M, Hrabovský J, Kutiš V and Kugler S, Modal analysis of the FGM beams with effect of the shear correction function. Composites Part B: Engineering, Volume 45 (1),Pages 1575-1582, February (2013).

[8] Murin J, Aminbaghai M, Kutis V and Hrabovsky J, Modal analysis of the FGM beams with effect of axial force under longitudinal variable elastic Winkler foundation. Engineering Structures, Volume 49, Pages 234-247, April (2013).

[9] Ribeiro D, Calçada R,Delgado R, Brehm M, Zabel V, Finite element model updating of a bowstring-arch railway bridge based on experimental modal parameters. Engineering Structures, Volume 40,Pages 413- 435, July (2012).

[10] Wei L, Cheng H and Li J, Modal Analysis of a Cable-stayed Bridge. Procedia Engineering, Volume 31, Pages 481-486, (2012).