International Journal of Research in Applied Sciences & Engineering (IJRASE)
Online ISSN: 2456-2920; Volume 1, Issue 1; January 2016 pp. 43-50

Finite Elementmodal Analysis and Mesh Optimization of a Typical Turbo Fan Engine – Fan Hubframe

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

Department of Aeronautical Engg, DSCE, Shavige Malleshwara hills, Bangalore-560078, India
charlesgmartin.aero@gmail.com, swamy.aroks@gmail.com

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 ModalAnalysis. Finally First order and Second order tetrahedral mesh is compared with the test results.**

Keywords: **ModalAnalysis, 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.

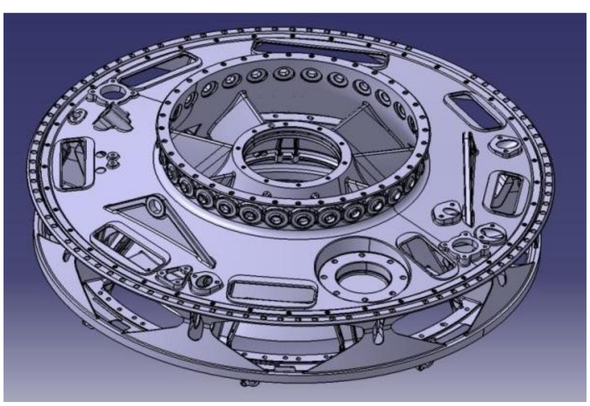


Fig. 1. Fan HubFrame

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 amesh.[11]

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)[1-10].

2. FLOWCHART

The whole project procedure is briefly explained in the flow chart as shown in the Fig2

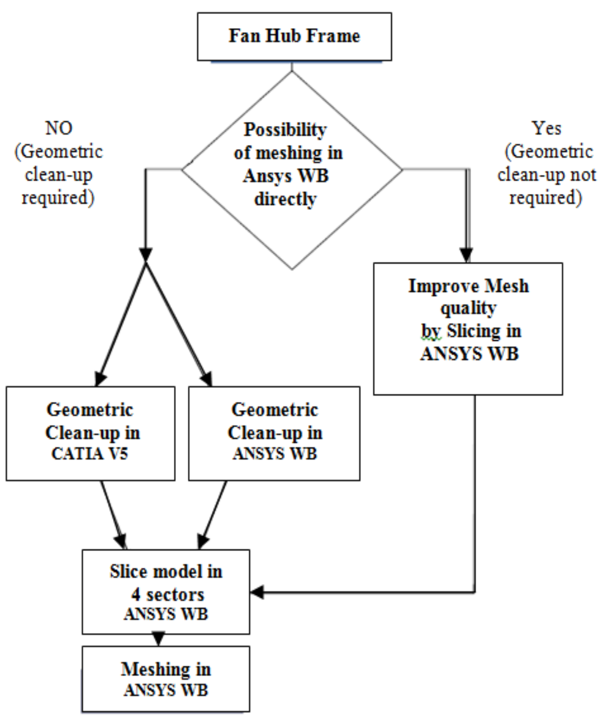


Fig. 2. Fan Hub Frame -FlowChart

3. POSSIBILITY OF MESHING DIRECTLY IN ANSYS WB BY CHECKING THE MODEL VISUALLY.

Check 1 : Check for presence of multiple splits on surfaces and fillets (as mentioned in Fig 4,5,6).If these split surfaces are present, most likely Ansys Work bench will fail to generate amesh.

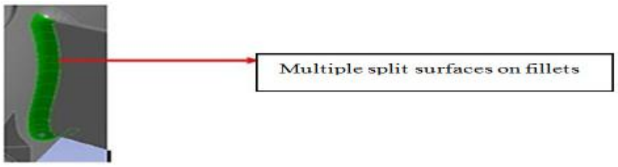


Fig. 4. Multiple spilt infillets

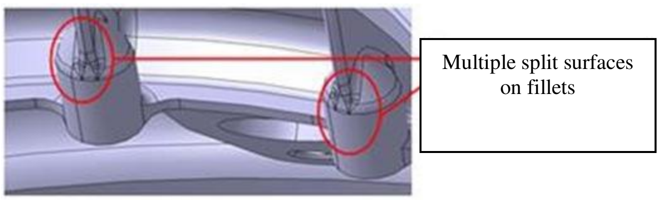


Fig. 5. Multiple split surfaces onfillets

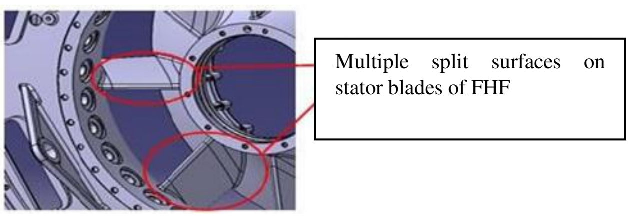


Fig. 6. SplitSurfaces

3.2 Check 2: Check for the presence of many holes as shown in Fig 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 themodel.

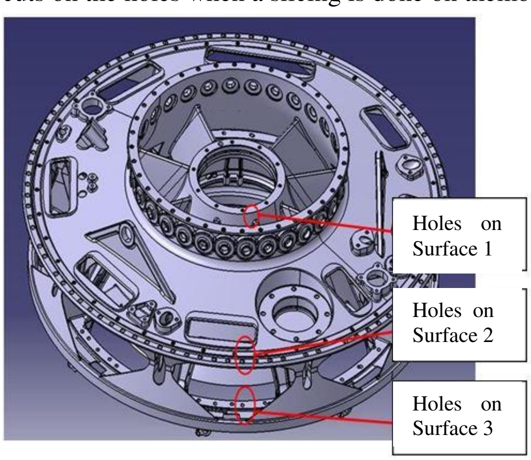


Fig. 7. Holes on different surfaces inFHF

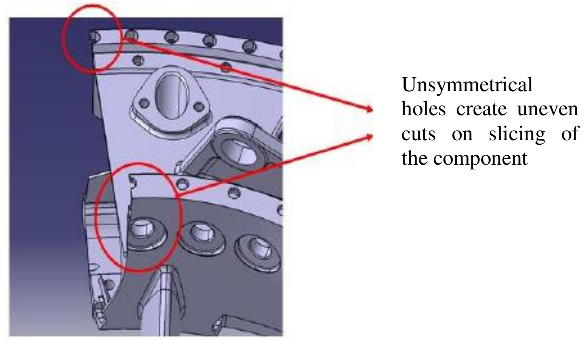


Fig. 8. Unsymmetrical sliced holes

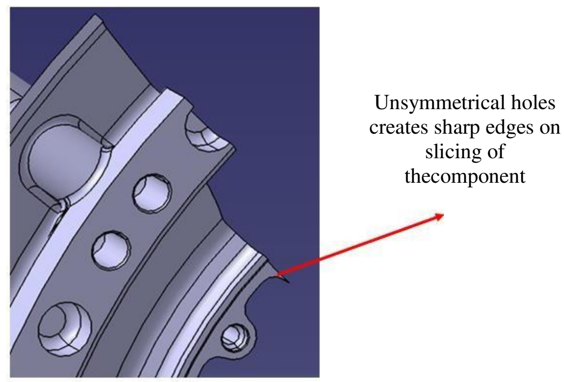


Fig. 9. Sharp Edge formed during slicing

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

3.2 Alternate Check (QuickCheck)

However, if somebody would like to check still the possibility of mesh in Ansys WB, then the following quick check can bedone (Fig 10)

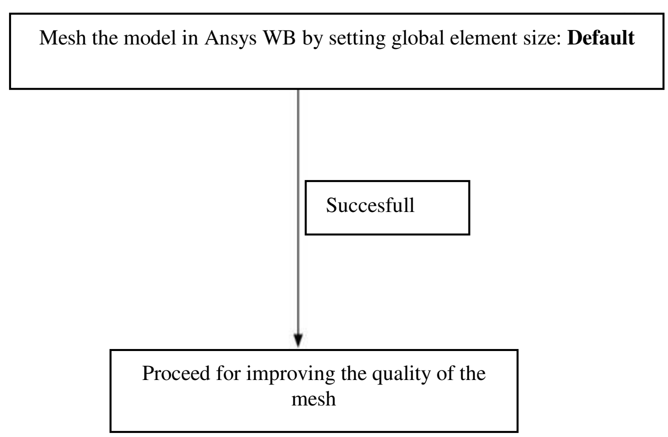


Fig 10 Flow chart of alternativecheck

4. GEOMETRY CLEAN UP IN CATIAV5

4.1 Hole fillingoperations

- Padoperation
- Surface extractoperation

4.1.1 Pad operation(Extrusion)

- Pad operation is used to fill all holes ona surface (surface should be free from projections on the bottom of thesurface)
- For further understanding refer Fig 11&Fig 12

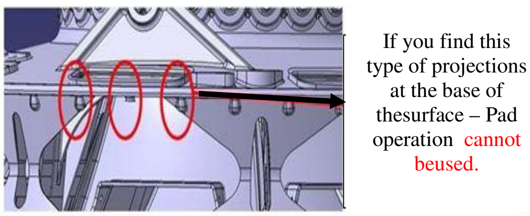


Fig 11. Projections belowthe surface of the model

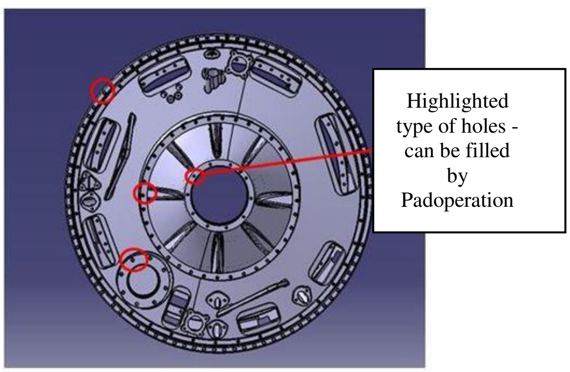


Fig12 SymmetricalHole

4.1.2 Surface extractoperation

- When there are complicated holes or pocket with variable thickness as shown in Fig 13 then the pad operation for filling the holes cannot be used, therefore one as to use surface extract operation to eliminate the complicatedhole.
- The extracted hole using is shown inthe Fig14
- After the surface extraction if the holes are symmetric as shown in Fig 13 &14, then one hole can be manually filled using surface extract and then circular pattern can be used to fill all theholes in oneoperation.
- If the holes are not symmetric then each hole should filledmanually.

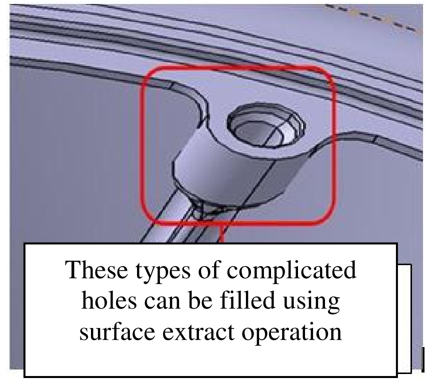


Fig13

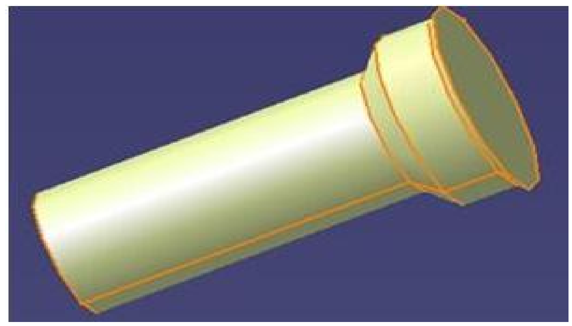
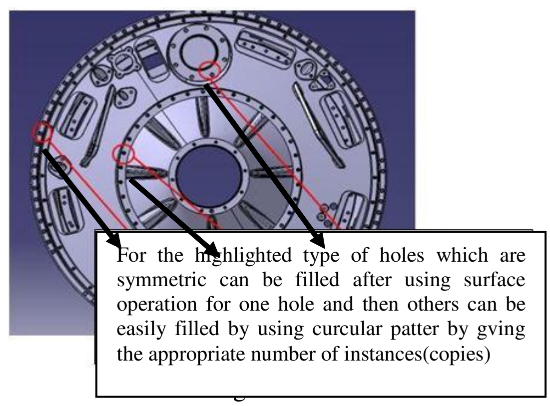


Fig 14 Surfaces Extracted from ahole



5. GOEMETRY CORRECTION BY REPLACEMENT OF ALTERNATE GEOMETRY

5.1 TYPEI

When the Problematic geometries of regular shapes are found as shown in the Fig 16a then it must be replaced with alternative geometry as shown in Fig17 to make meshing process smooth and effective on these type ofgeometries.

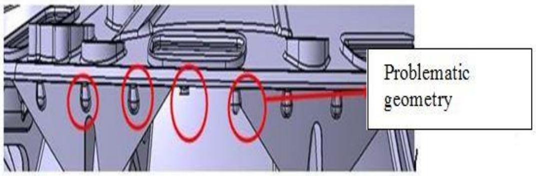


Fig 16

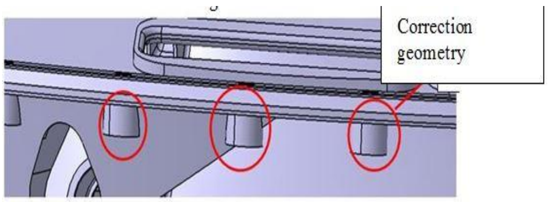


Fig17

5.2 TYPEII

- 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 suchsurfaces.
- Therefore, alternative geometry should be replaced as shown in Fig18

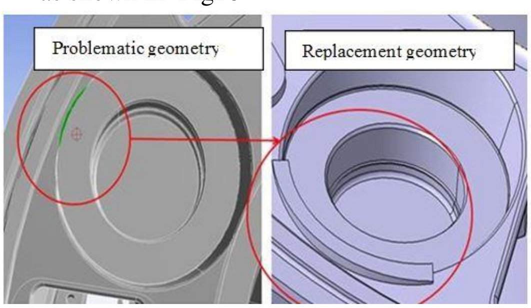


Fig 18 Problematic and Alternative replacementgeometry

6. MESHING PROCEDURE IN ANSYSWB

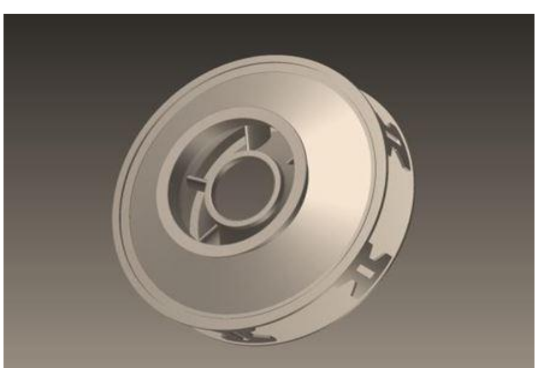


Fig 19 FHF after geometry Clean-Up

- The fully cleaned up geometric model as shown in the Fig6.1
- Firstly before starting the meshing we should slice the component into 4sectors as shown in the Fig6.2

6.1 Why slicing isdone?

- Slicing is use to simplify the model and to divide it into many sub parts which in-turn reduces the mesh computationaltime.
- Each part can be meshed efficiently and effectively when the model isdivided.
- 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 cleanup.(Fig 20)
- 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 aplane.

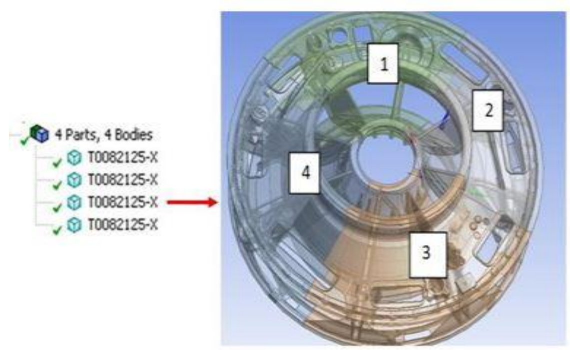
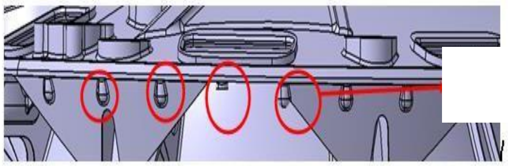


Fig 20 Final sliced 4 sector FHFModel

- First by setting global element size : Defaulttry to mesh and if the meshis 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 21)



	1st order
Nodes	**187497**
Elemen	**621699**

- 2nd order mesh–Using Element size:20 MediumCoarse (Fig 22)

Type	**2st order**
Nodes	**513409**
Elements	**288801**

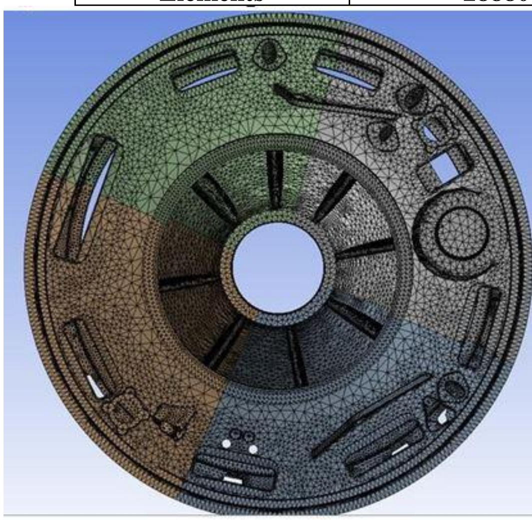


Fig. 21. 1st order mesh-Using Element size:10

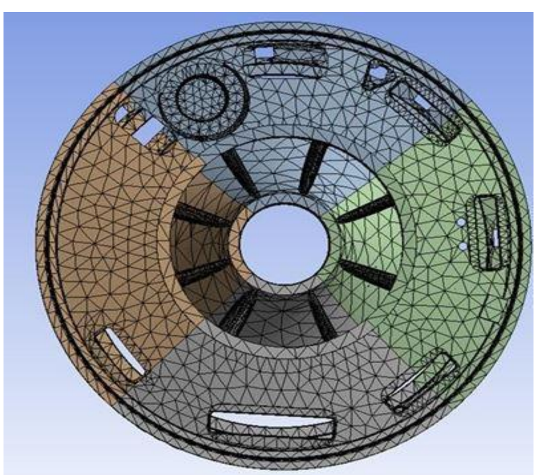


Fig. 22. 2nd order mesh–Using Element size:20 MediumCoarse

7. MASS DEVIATION

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

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

Mass Deviation=8.7%

8. 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%

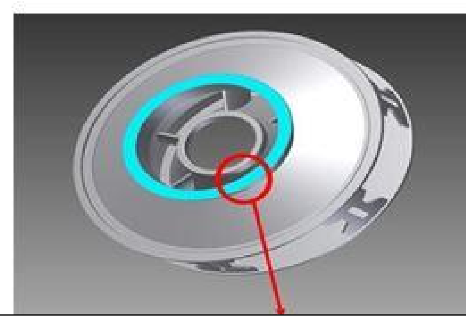
9. MODAL ANALYSIS RESULTSREPORT

9.1 MaterialUsed:

Name	Titanium	
General	MassDensity	4.51g/cm^3
	Yield Strength	275.6MPa
	Ultimate Tensile Strength	344.5MPa
Stress	Young'sModulus	102.81GPa
	Poisson'sRatio	0.361

9.2 Boundaryconditions

- The Exhaust Face Fixed (as mentioned in the figure below) for conducting Modal analysis.()
- No of Modes of extraction :10



The highlighted exhaust face as to be fixed for carrying modal analysis.

Fig. 23

9.3 Timeevaluation

Timeevaluation	Hrs(approx)
Time taken to geometryclean-up	5
Time taken formeshing	3
Time taken forAnalysis	2
Total timetaken	10

10. RESULTS ANDDISCUSSIONS

- Results for 10 mode shapes in Modal analysis were solved and thefollowing results weretabulated.
- The time estimation for the analysisis 30 mins
- Mode shapes v/s Frequency(Hz) graphis plotted as shown in the Fig11.1

Properties	
Volume	1.4519e+007mm³
Mass	67.078kg
CentroidX	174.04mm
CentroidY	9.7724e-002mm
CentroidZ	-0.14496mm
Moment of InertiaIp1	1.0677e+007kg·mm²
Moment of InertiaIp2	6.1241e+006kg·mm²
Moment of InertiaIp3	6.1153e+006kg·mm²
Statistics	
Nodes	129889
Elements	65794

Modal Analysistabulation

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

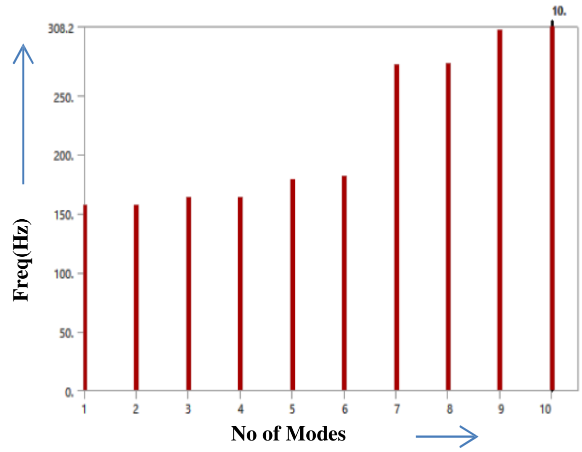


Fig. 24

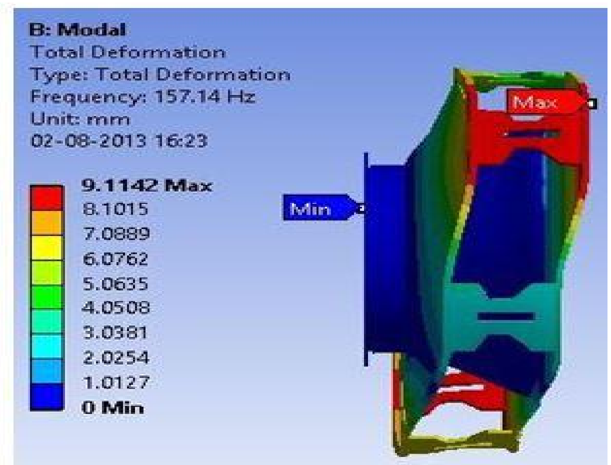


Fig. 25 TotalDeformation

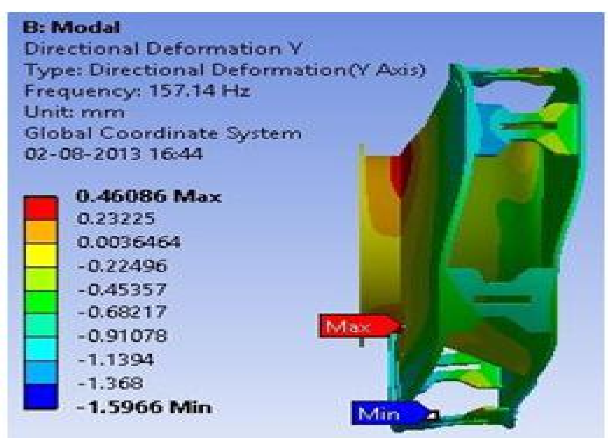


Fig. 26 Directional deformation(Yaxis)

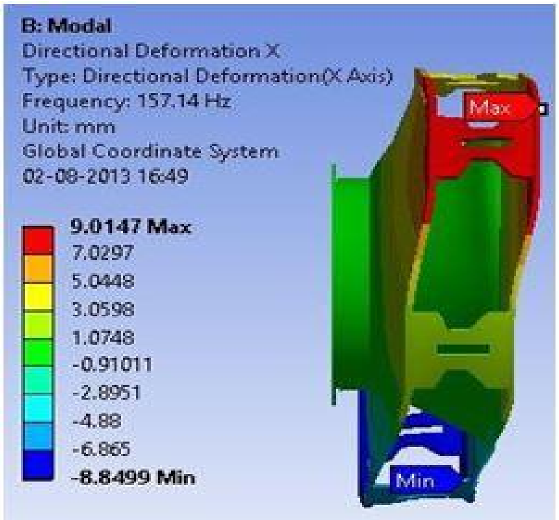


Fig. 27 Directional deformation(Xaxis)

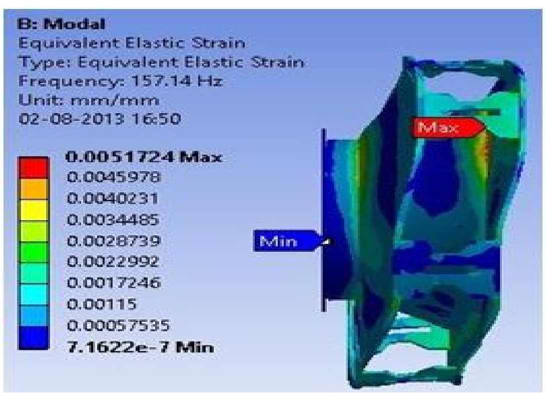


Fig. 28 Equivelent ElasticStrain

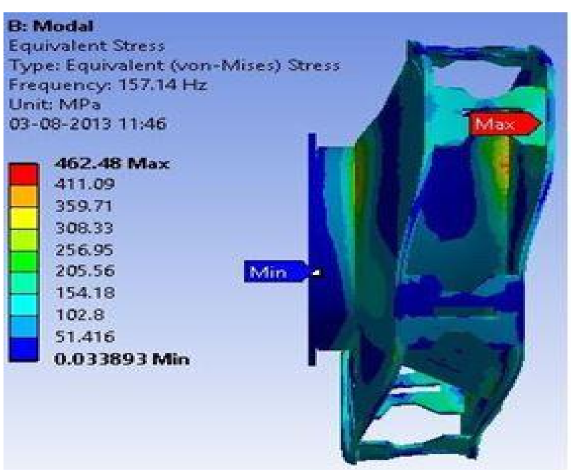


Fig. 29 Equivelent ElasticStress

DIRECTIONALDEFORMATION		
Orientation	XAXIS	YAXIS
	Minimum	Maximum
Results	0mm	9.1142mm
	-1.5966mm	0.46086mm
	-8.8499mm	9.0147mm
TOTALDEFORMATION		
Minimum	Maximum	
0mm	9.1142mm	
EQUIVALENT ELASTICSTRAIN		
Minimum	Maximum	
7.1622e-007mm/mm	5.1724e-003mm/mm	
EQUIVALENTSTRESS		
Minimum	Maximum	
3.3893e-002MPa	462.48MPa	

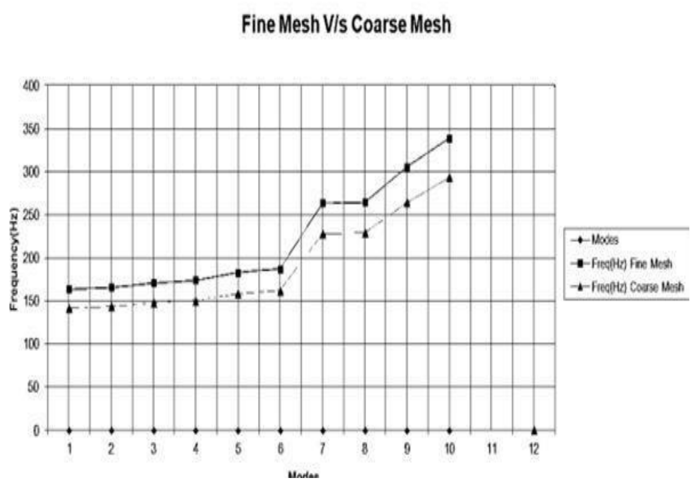


Fig 30 Fine Mesh V/s Coarse MeshComparison

Comparison of fine mesh v/s coarse mesh.

From the plot Fig 11.7, 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 corasemesh.

Comparison Modes v/s Frequency for Variable stator airfoilthickness Fig 31

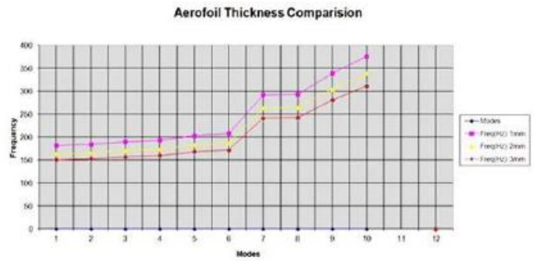


Fig. 31 Modes v/s Frequency for Variable Stator airfoil thickness

Comparison was done with the ANSYS results v/s Test bench results as shown in the Fig32.

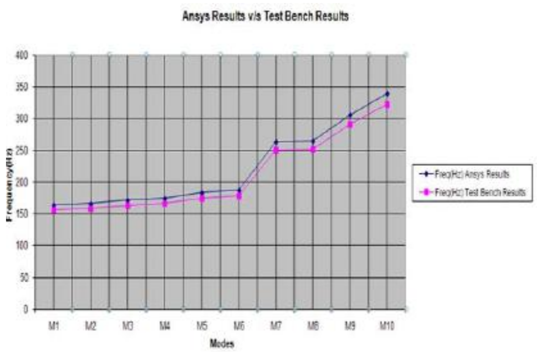


Fig. 32 ANSYS Results v/s Test benchresults

11. 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 timelimit.

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 asfollows:-

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

The total computational time was significantly reduced from 80hrs to10hrs.

REFERENCES

[1] Liuchuang Wei, Heming Cheng, Jianyun Li, Modal Analysis of a Cable-stayed Bridge, Procedia Engineering, Volume 31, 2012, Pages 481-486, ISSN1877-7058.

[2] J. Murin, M. Aminbaghai, V. Kutis, J. Hrabovsky, Modal analysis of the FGM beams with effect of axial force under longitudinal variable elastic Winkler foundation, Engineering Structures, Volume 49, April 2013, Pages 234-247, ISSN 0141-0296.

[3] J. Murin, M. Aminbaghai, J. Hrabovský, V. Kutiš, S. Kugler, Modal analysis of the FGM beams with effect of the shear correction function, Composites Part B: Engineering, Volume 45, Issue 1, February 2013, Pages1575-1582, ISSN1359-8368.

[4] M. Aminbaghai, J. Murín, V. Kutiš, 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, January 2012, Pages 314-329,ISSN0141-0296.

[5] Huajun Li, Ping Li, Sau-Lon James Hu, Modal parameter estimation for jacket-type platforms using noisy free-vibration data: Sea test study, Applied Ocean Research, Volume 37, August 2012, Pages 45-53, ISSN0141-1187.

[6] J. Murin, M. Aminbaghai, J. Hrabovský, V. Kutiš, S. Kugler, Modal analysis of the FGM beams with effect of the shear correction function, Composites Part B: Engineering, Volume 45, Issue 1, February 2013, Pages 1575-1582, ISSN1359-8368.

[7] D. Ribeiro, R. Calçada, R. Delgado, M. Brehm,V. Zabel, Finite element model updating of a bowstring-arch railway bridge based on experimental modal parameters, Engineering Structures, Volume 40, July 2012, Pages 413- 435, ISSN0141-0296.

[8] Yang Li, Caili Wang, Rongshun Wang, The thermal stress analysis and structure optimum of neck tube with vertical cryogenic insulated cylinders based on ANSYS, Nuclear Engineering and Design, Volume 252, November 2012, Pages 144-152, ISSN 0029-5493.

[9] Shin-Yong Chen, An equivalent direct modelling of a rotary shaft with hot-fit components using contact element modal analysis results, Computers & Mathematics with Applications, Volume 64, Issue 5, September 2012, Pages 1093-1099, ISSN 0898-1221.

[10] Guy Banwell, Stefan Mohr, Steve Rothberg, Jon Roberts, Using experimental modal analysis to validate a finite element model of a tennis racket, Procedia Engineering, Volume 34, 2012, Pages 688-693, ISSN1877-7058.

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