CORRELATION OF TEST DATA WITH ANALYSIS OF TURBOCHARGER HOT BOLTED JOINT

Smita S Abhang[1], Assistant Prof. Y.C. Jadhav [2], Prof. S. M. Jadhav [3]

[1]*P G Student, Department of Mechanical Engineering, N.B.N Sinhgad School of Engineering Ambegaon (Bk), Pune-41, India*

[2] *Professor, Department of Mechanical Engineering, NBN Sinhgad School of Engineering Ambegaon (Bk), Pune-41, India*

[3] *Professor, Department of Mechanical Engineering, NBN Sinhgad School of Engineering Ambegaon (Bk), Pune-41, India*

***

Abstract - *This project study variation of standard parameters which used in analysis of joint for checking the variation of it along with its effect on the stress and contact pressure to correlate the analysis with test results accurately. The model contains turbine housing, bearing housing, clam plate, heat shield, compressor cover and bolt. The modeling and simulation software used are CREO and ANSYS. In existing analysis procedure there are some standard values but in this project it will change to lower and higher side and observe its effects on the result and find best combination. This will help industry to match the results with test data and reduce time as well. The parameters are mesh, coefficient of friction, preload and coefficient of thermal expansion.*

***Key Words*: Finite element analysis, thermal analysis, structural analysis, bolted joint, mesh, coefficient of friction, Bending test, combine loading.**

1.INTRODUCTION

Turbocharger is the mechanical device which increases density of air entering into the combustion chamber of IC engine with compressor which is driven by a turbine driven by exhaust gas of same IC engine. Turbocharging increases quantity of air entering into the combustion chamber which promotes lean combustion, this further result into better performance and lower exhaust emissions. From last few years many researchers made effort to improve the power output of an engine and to reduce exhaust gases by making some changes in conventional turbocharger and installing some additional accessories like turbocharger and intercooler. Due to increase in the demand of fuel efficient engines with more power and minimum emissions more research will take place in this field. Basically two types of joints are using in turbocharger industry, V-band joint and bolted joint. The joint used between turbine housing and bearing housing is bolted joint.

There were types of loading applied on this joint –thermal load, bending load and torsional load. Preload is applied to clamp the joint components which generate clamping force between joint. The joint members and bolts both behave like stiff springs, one being compressed and the other stretched as suggested in Fig. 1

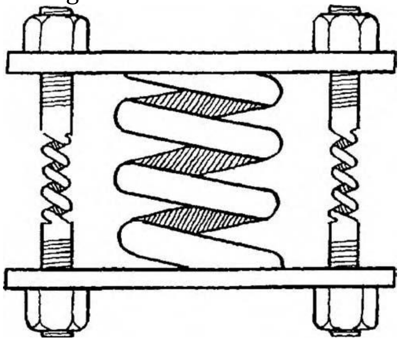

Fig.1 Bolt and joint members

Preload = Torque / (Nut factor* dia. of shank)

Turbine housing contains exhaust gas so the temperature of the components nearby is high. So thermal stresses will induced in the joint components. Bending and torsional load is generated due to boost loading, the loading due to attached accessory to the joint components.

2. METHODOLOGY

Finite element model of turbine housing to bearing housing bolted joint is prepared in ANSYS work bench. All solid metal parts are modelled using tetrahedron and wedge elements SOLID187, contact and target elements by CONTA174 and TARGE170 and pretension elements by PREST179. The contact established within joint has been carried out by standard procedure. Bonded as well as frictional contacts are used for the joint analysis. Coupled thermal structural analysis of turbine housing to bearing housing bolted joint assembly has been done in ANSYS workbench to determine the stresses induced in critical area that is fillet in this analysis. Bending test was conducted on the "A" frame and customer end test was conducted on "B" frame of turbo and aim of this analysis is to correlate that test along with examine the effect of various parameters on the stresses induced. In case of "A" frame bending load is the reason behind failure. The test is bending limit validation test. The behaviour of any component or assembly can be predicted in advance through simulation. Experimentation is one of way to predict the behaviour in advance before field use, but since it involves time, cost, complex set up and prototype or actual model, therefore experimental test is not recommended every time. Simulation is the one of the best method to predict behaviour of structure and to understand stresses induced stresses in advance to have more confidence in model before going for test. Hence to check strength of joint also at high temperature, thermal structural coupled analysis of joint assembly was performed.

3. THEORETICAL BACKGROUND

A computerized process helpful for analysis of structure can be defined as finite element [FE] analysis. To evaluate stresses due to structural along with thermal FE analysis can be used. Since the joint components are subjected to higher temperatures, thermal stresses will developed in the components of the joint. Simultaneously since the joint assembly is subjected to various external loads –thermal load and bending load structural stresses will get developed within the joint components.

Thermal analysis:

Matematically thermal analysis in Ansys can be done as follow. In this analysis only heat flow due to conduction is cosidered.

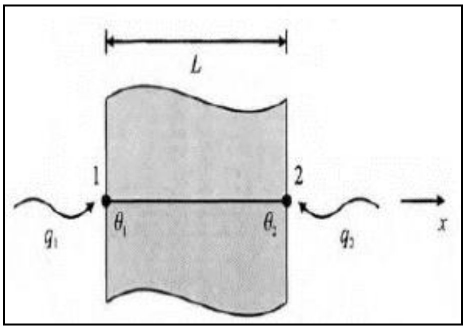

Fig. 2 : One Dimensional Heat flow.

This heat flow by conduction and covection

Where K= thermal conductivity,

By combining both conduction and convection the final equation can be written in matrix form as;

[K] [T] = [F]

Where [K] =global stiffness matrix. This can be obtained by assembling stiffness matrix of each element.

[T]= Temperature of that node.

[F]= Boundary condition.

This matrix form of equation can be used in FEM to solve problem. In summery if [K] and [F] can be formed, and then temperature distribution can be determined by any matrix solver procedure. These are basic question of FEA.

A. *Structural Analysis:*

For structural analysis we are using output of thermal analysis i.e. node wise temperature. This temperature can be applied as body force on component.But in case of structural analysis temperature data for each node is available. So we applied each node temperature as body force and allow it to expand. Mathematical equation for calculating thermal stress is

…where α=thermal expansion and ΔT= temperature difference, v=poisons ratio.

In FEM this can be written as in matrix form such as

[σ]= [K] [T]

[K] =matrix called as global stiffness matrix. Equation shows matrix for one element. Global matrix can be calculated by assembling all elements stiffness matrix. These are basic question of FEA.

Material properties:

Table 1: Material properties of joint component.

Component	Material	Density (Kg/m3)	Poisson's ratio
Bearing housing	Grey cast iron	7050	0.26
Turbine housing	S.G. cast iron	7100	0.3
Bolt	B16 alloy steel	7850	0.3
Clamp plate	Stainless steel	9000	0.28
Heat shield	Stainless steel	9000	0.28
Compressor cover	Aluminum	2890	0.34

4. SIMULATION WORK

Analysis objective is to perform coupled thermal structural analysis to evaluate stresses on critical area that is fillet of the joint component.

A. *A frame*

In the analysis for "A" frame assembly is not available so I assembly has been carried out in CREO and then import it to ANSYS workbench. Geometry clean-up has been done in design modular and pre-processing of the proposed structure has been done in mechanical window of ANSYS workbench. Number of nodes of joint assembly is around 15 lac. For "A" frame. Meshed model of the bolted joint assembly has been shown in following fig. 3.

In finite element modelling of the bolted joint assembly the elements are used as per table 1.

Table 2: Elements used in analysis

Sr. No.	**Component's name**	**Element's Type**
1.	All solid parts	SOLID 187
2.	Contact Element	CONTA174
3.	Target Element	TARGE170
4.	Pretension	PREST179

CAD model of bolted joint assembly is shown in following fig. 3.

Analysis summary: For this finite element model of bolted joint assembly first steady state thermal analysis is carried out then structural analysis.

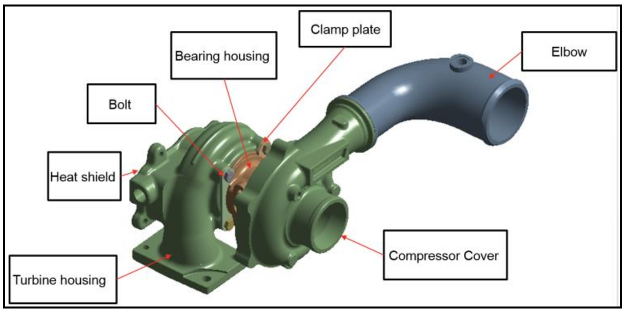

Fig. 3: CAD model of bolted joint assembly of "A" frame

For design of experiment four parameters are selected, which are preload, coefficient of friction, coefficient of thermal expansion and mesh. There are some standard values for the parameter, these values varied as per given table no. 2to lower and higher side for 3 parameters.

Table 3: Parameters for the Design of experiment

Sr. No.	Parameter	Lower limit	As per standard	Higher limit
1	Mesh	2X	X elements at critical area	3X
2	Preload	30% less	As per Std.	30% more
3	Coefficient of friction	50% less	As per Std.	50% more
4	Coefficient of thermal expansion of bolt	10% less	As per Std.	10% more

Thermal boundary condition

After meshing the next step is to apply the boundary conditions. The boundary conditions are applied as per standard procedure. In this, the heat transfer coefficient and temperatures are given as thermal boundary condition for the below mentioned areas.

1) Structural boundary condition:

After meshing the next step is to apply the boundary conditions. Selecting the proper boundary condition has an important role in structural analysis. For a static analysis, the turbine inlet flange is fixed same as test condition. For the loading, there are two external forces acting on the joint components. Hydraulic force is applied at the elbow face to achieve bending moment of the "Y" N-m and the thermal load which generate due to high temperature.

2) Analysis approach –

Load step 1 – bolt preload

Load step 2 – bolt preload + thermal load

Load step 3 – bolt preload +thermal load + bending load.

The assembly is fixed at the turbine housing flange and the structural boundary conditions have been applied at the compressor end elbow.

B. *"B" Frame*

In the analysis for "B" frame assembly is available so it imported to ANSYS workbench. Geometry clean-up has been done in design modular and pre-processing of the proposed structure has been done in mechanical window of ANSYS workbench shown as fig.4. No. of nodes of joint assembly is around 16 lac. for "B" frame. Fine mesh is applied at the critical area. Parameter study has been carried out as per the "A" frame.

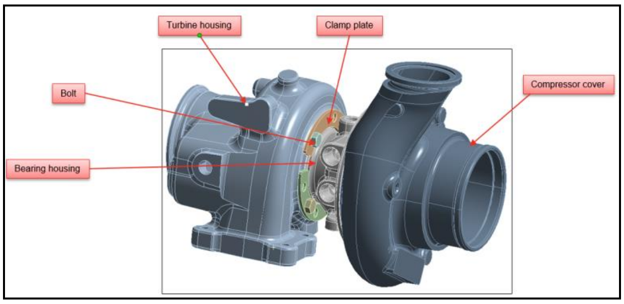

Fig. 4: CAD model of bolted joint assembly of "B" frame

Thermal boundary condition

The boundary conditions are applied as per standard procedure. In this, the heat transfer coefficient and temperatures are given as thermal boundary condition for the below mentioned areas.

3) Structural boundary condition:

4) Analysis approach –

Load step 1 – Bolt preload

Load step 2 – Bolt preload + Thermal load

Load step 3 – Bolt preload + Thermal load + bending load+ Rotational torque.

The assembly is fixed at the turbine housing flange and the structural boundary conditions have been applied at the compressor outlet.

5. RESULTS AND DISCUSSION

Thermal results:

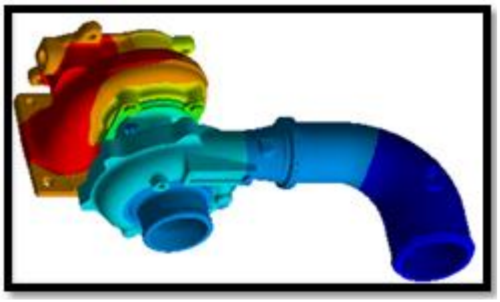

Fig.7: Results of thermal analysis for "A" frame

Exhaust gas has been entering into the turbine inlet and atmospheric air enters into the compressor inlet so the temperature flows from turbine housing to the compressor cover and elbow. The temperature at the components in actual test is correlated with simulation results.

Table 4: Result of steady state thermal analysis

Sr. No.	Component	Maximum temperature (°C)	Minimum temperature (°C)
1	Bearing housing	57	20
2	Turbine housing	100	57
3	Bolt	79	56
4	Heat shield	65	34
5	Compressor cover	35	25
6	Clamp plate	74	46
7	Elbow	22	14

Fig.8: Results of structural analysis for load step 1

The bearing housing made by brittle cast iron so maximum principle stresses theory is used. Out of all loading the bending load is dominant. Stresses are low in bolt preload case 11.15, in second load step two – 26.3, and third load step – 100. All results are scaled as per company policy.

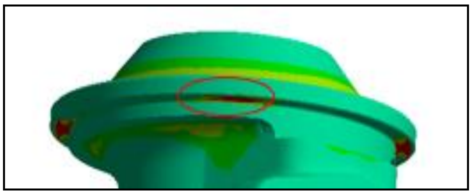

Fig.9: Results of structural analysis for load step 2

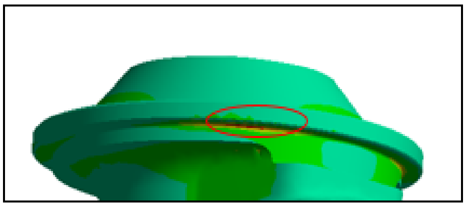

Fig.10: Results of structural analysis for load step 3

Table 5: Result of static structural analysis

Sr. No.	Load Step	Maximum Principal Stresses (MPa)	Maximum Total deformation (mm)
1	LS1	5	0.22848
2	LS2	26	1.4894
3	LS3	75	3.6393

DOE is carried out on this joint assembly.

Table 6: Stress results by varying preload values

	Load step	% change	Test Model	% change
1	LS1	18	5	27
2	LS2	16	26	16
3	LS3	6	75	7

Due to preload variation % change in stress is maximum 27% and minimum 2 %.

Table 7: Stress results by varying coefficient of thermal expansion values

	Load step	% change	Test Model	% change
1	LS1	0	5	0
2	LS2	0	26	0
3	LS3	0	75	2

There is no effect of variation of coefficient of thermal expansion.

Table 8: Stress results by varying coefficient of friction.

	Load step	% change	Test Model	% change
1	LS1	18	5	9
2	LS2	0	26	2
3	LS3	1	75	2

There is effect of coefficient of friction but not in the case of combine loading.

Table 9: Stress results by varying mesh.

	Load step	Test Model	% change	% change
1	LS1	5	4	4
2	LS2	26	0	0
3	LS3	75	0.4	0.4

Mesh with X elements at the fillet gives the accurate results within less time

Thermal results of "B" frame:

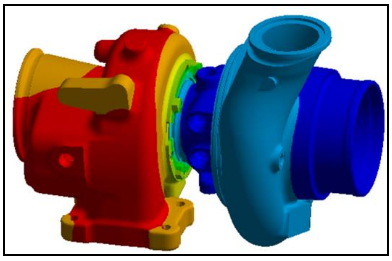

Fig.11: Results of thermal analysis for "B" frame

The temperature at the components in actual test is correlated with simulation results.

Table 10: Result of steady state thermal analysis

Sr. No.	Component	Maximum temperature (scaled values)	Minimum temperature (scaled values)
1	Bearing housing	53	15
2	Turbine housing	100	55
3	Bolt	75	50
4	Heat shield	83	35
5	Compressor cover	30	13
6	Clamp plate	74	42

As turbine exposed to hot gas and compressor cover to atmospheric condition, the temperature gradient generated. The temperature at the components in actual test is correlated with simulation results. The maximum temperature archived at the turbine end.

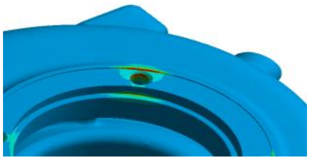

Fig.12: Results of structural analysis of load step 1 for "B" frame

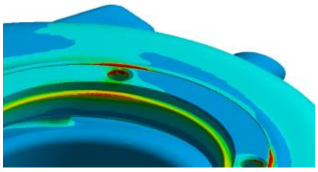

Fig.13: Results of structural analysis of load step 2 for "B" frame

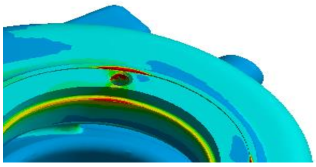

Fig.14: Results of structural analysis of load step 3 for "B" frame

The material of the turbine housing is ductile cast iron so equivalent von-Mises stress is used. From the results in table 11, it is clearly seen that the preload is more dominant in the analysis. So the focus of this DOE is the effect parameters on LS1.

Table 11: Result of static structural analysis for "B" frame

Sr. No.	Load Step	Von Mises Stresses (scaled values)	Maximum Total deformation (scaled values)
1	LS1	125	0.26
2	LS2	93	1.63
3	LS3	80	1.67

DOE has been carried out.

Table 12: Stress results by varying preload values

	Load step	% change	Test Model	% change
1	LS1	30	125	34
2	LS2	20	93	27
3	LS3	25	80	29

Due to preload variation % change in stress is maximum 34% and minimum 20 %.

Table 13: Stress results by varying coefficient of thermal expansion values

	Load step	% change	Test Model	% change
1	LS1	0	125	0
2	LS2	14	93	12
3	LS3	15	80	15

Due to coefficient of thermal expansion variation % change in stress is maximum 15% and minimum 12 %.

Table 14: Stress results by varying coefficient of friction.

	Load step	% change	Test Model	% change
1	LS1	1	125	0
2	LS2	2	93	0.2
3	LS3	3	80	2

There is no effect of variation of coefficient friction.

Table 15: Stress results by varying mesh.

	Load step	Test Model	% change	% change
1	LS1	125	0.5	2
2	LS2	93	0.5	1
3	LS3	80	0.3	0.05

Mesh with X elements at the fillet gives the accurate results within less time.

6.EXPERIMENTAL VALIDATION

Fig.15: Test set up

Test Procedure

The temperature of the component has been raised to reference temperature.

Then cyclic loading is applied to the assembly through force, till failure is achieved.

Instruments

1. Fixture
2. Data logger
3. Cyclic force generator
4. thermocouples

Experimental Results:

In this assembly of joint the bearing housing prior failed and reason behind it is bending load. The crack is generated at the bearing housing fillet which is below flange. After design modification it passes the test.

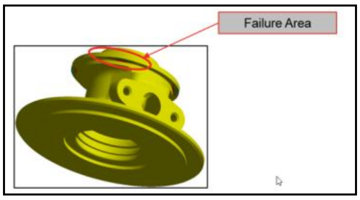

Fig. 16: failure of bearing housing.

7.CONCLUSION

The correlation between "A" frame model for bending test simulation and experimentation shows a good agreement. From the correlation it is observed that finite element model and experimentation test are matched, thus it can be concluded that failure of bearing housing is adequately captured in analysis.

Stresses on bearing housing fillet should be taken on tetrahedron element below inflation layer, which are realistic and show correlation with the test failure using Haigh diagram.

In case of "A" frame the preload have max impact but this preload load case is not the reason for failure. The bending load with combine loading is dominant in this analysis. This parameter has significant impact because the critical location is very close to the bolt.

Coefficient of thermal expansion doesn't have any impact on this analysis it is concluded as if there is no temperature dependent data of the material where stress will be plotted then do not run for variation of this parameter.

Coefficient of friction has no impact for both of cases.

From Mesh study it is concluded that the "X" elements at the fillet gives better result when mesh is either bearing housing fillet or turbine housing fillet.

Similarly for "B" frame the failure is on same location as shown in highly stress area that is fillet of turbine housing at bolt no. 3.

In this "B" frame analysis again bolt preload have more impact reason same as "A" frame the location of stress is near to the bolt along with this preload is dominant in this analysis.

The second important parameter is coefficient of friction. It has significant impact on the analysis result for "B: frame size.

REFERENCES

[1]Z.S. Lad, Nikhil S. Mane, H.M. Dange "Review on Turbo charging Of IC Engines" International Journal of Engineering Sciences & Research Technology, 2015, 327-331.

[2] li K. A. Khudhir, Mahesh C. Aggarwal, Michael Sirak and Michael J. Panza," A Theoretical and Computational Analysis Of The Bolted Joint – A Sensitivity Study Of Temperature Effect On The Turbocharger Inlet Bolted Joint In Locomotive Engines" ASME 2014 Gas Turbine India Conference GTINDIA2014 December, New Delhi, India, 2014, 1-8.

[3] Muhammad Abidand Saad Hussain, "Relaxation behaviour of gasketed joints during assembly using finite element analysis" Ghulam Ishaq Khan Institute of Engineering Sciences and Technology, 31-433.

[4] Sanjib Goswami," A Finite Element Investigation on Progressive Failure Analysis of Composite Bolted Joints under Thermal Environment" German Aerospace Center, Institute of Structures and Design, Pfaffenwaldring, D-70569 Stuttgart, Germany, 2005, 161-171.

[5] Nomesh Kumar, P.V.G. Brahamanandam and B.V. Papa Rao, "3-D Finite Element Analysis of Bolted Flange Joint of Pressure Vessel" MIT International Journal of Mechanical Engineering Vol. 1, No. 1, ISSN No. 2230 – 7699, Jan 2011, 35-40.

[6] W. Brown, "Recent Advancement in PVP Bolted Joint Technology" 14th International Conference on Pressure vessel technology, Australia, 2015, 158-167.

[7] Lreman, Tomas; Nyman, Tonny; Hellbom, Kurt"On design methods for bolted joints in composite aircraft structure" Composite Structures (ISSN 0263-8223), vol. 25, no. 1-4, 1993, 567-578.

[8] Hehui Wang , Yan Zhang, "Creep Analysis of Combined Bolted Flange Joints under High Temperature" international journal of Applied Mechanics & Materials, Issue 624, 2014, 187-192.

[9] W.BrownT.-Y.Lim, "Factors Affecting Nut Factors for PVP Bolted Joint Assembly"14th International Conference on Pressure vessel technology, Australia, 2015, 168-175.

[10] Dattatraya Parle, Amol Patil and Anirudha Ambulgekar, "Evaluation of Stress Concentration Factor using ANSYS" Pune, India,2011, 35-38.