FINITE ELEMENT ANALYSIS AND WEIGHT REDUCTION OF FLANGE COUPLING USING CAE TOOLS

Somvir Singh[1] Bharat Bhushan[2]

Research Scholar [1], Assistant Professor [2]

Department of Mechanical Engineering

Manav Institute of Technology & Management

Hisar (Haryana), India

ABSTRACT

Coupling is a mechanical component used for transmitting power from one shaft to another shaft rotating at particular RPM at a long distance. In flange coupling two shafts are connected together at their ends with the help of flanges for transmitting power. The rigid flange coupling is specially designed and developed for horizontal shaft mounted gear unit applications. Today every manufacturing industry has a challenge to increase the strength to weight ratio of the component. In the project our objective is to decrease the weight of flange coupling. For this, composite material aluminium silicon carbide is used for the flange which is light in weight and good thermal and mechanical properties. In the project we used Finite Element Analysis method to optimize the flange coupling design. In the work design of flange coupling is made by using CATIA V5 design software and static structural analysis is carried out in ANSYS workbench software and the stress and deformation was calculated. The Finite Element Analysis results are significant to improve the component design at the early developing stage. The main objective of the work is to reduce the weight of the flange coupling without affecting the performance so as to minimize the cost of the material.

Key Words: *FEA, CATIA, ANSYS, Flange Coupling.*

---

1. INTRODUCTION

Coupling is a mechanical component used to connect two shafts together at their ends for the purpose of transmitting power. The rigid flange coupling is specially designed and developed for horizontal shaft mounted gear unit applications. These applications require a rigid link between the low speed shaft of the gear unit and the machine, such as for conveyor drive, bucket elevator, travel drive applications etc. In rigid flange coupling, flanges are brought together and are bolted in the annular space between the hub and the protecting flange [3]. The Figure 1 shows the various parts of flange coupling [6].

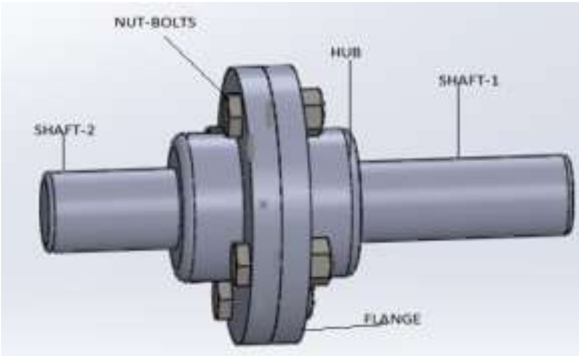

Figure 1 Flange coupling

1.1 Types of flange coupling

A flange coupling typically used to join the two shafts of same diameter aligned in same axis. Flange couplings do not allows axis deviation between two shafts. It consists of two flanges generally made up of cast iron. Each flange is mounted on the shaft end and keyed to it. Two flanges are holds together in position with help of bolts. The number of bolts may depend on the perimeter of the pitch circle in turn the diameter of the shat used to connect. Flange couplings generally classified into three types

1. Protected Flange Coupling
2. Unprotected Flange Coupling
3. Marine Flange Coupling

The difference between these coupling is in protected flange coupling an extra hallo shaft type layer is casted to protect the bolts, such layer is not provided in unprotected type and in marine type couplings tapered bolts are used instead of regular headed bolts [2].

2. LITERATURE REVIEW

D. H. Vardhan et al. [2] performed the java computation for mechanical design on dimensions of the various flange couplings. In the work they considered standard materials for the elements and also gave provision to change the material and stresses in the machine members. With the help of java they computed the couplings in shorter duration with precise values. With this program any one could compute the dimensions of the coupling with in fraction of seconds.

D. B. Shah et al. [3] performed parametric modelling and drawing automation for flange coupling using excel spreadsheet. In the work he integrated commercially available packages Autodesk Inventor with Microsoft excel spreadsheet for creation of modelling and manufacturing drawing. Various product variants of the flange coupling have been executed by parametric designing concept in Inventor. Then he used one feature crated in Inventor software the 3D modelling and manufacturing drawings would be generated automatically and efficiently.

I. J. Jadeja et al. [4] developed GUI based design software in VB Environment to Integrate with CREO for design and modelling using case study of coupling. They reviewed the procedural steps involved in the design of couplings and the development of the software package using visual basic as a tool for the design. This system is carried out on the case study of flange coupling and standard design equation being carried out together with the use of programming software and used CREO as modelling software after getting output from the designing software.

R. T. Salunkhe et al. [5] designed and carried out analysis of coupling using ANSYS. The project presented the concept of reducing maximum shear stresses by adding a new material between shaft and hub. The modelling of rigid flange coupling was done in CREO 2.0 and analysis of rigid flange coupling was carried out with the help of ANSYS 15 Software for calculated torque. They concluded that the shear stresses developed in the flange coupling with Ti alloy ring were less than the shear stresses developed in coupling without Ti alloy ring.

3. METHODOLOGY

The main objective of this work is to perform the Finite Element Analysis of flange coupling using ANSYS workbench. Firstly the model of flange coupling is designed in CATIA V5 software and saved in iges format. The iges file is then imported in Solidworks software and saved in Parasolid (x_t) format file for no data loss. This Parasolid file of flange coupling is then imported in ANSYS workbench. The deformation and stress contours have been plotted. Then random designs are made with modification in dimension and by optimising the material and analysis is done. The design optimization of flange coupling is carried out with a view to reduce the weight. Figure 2 shows the model of flange coupling made in CATIA V5 software.

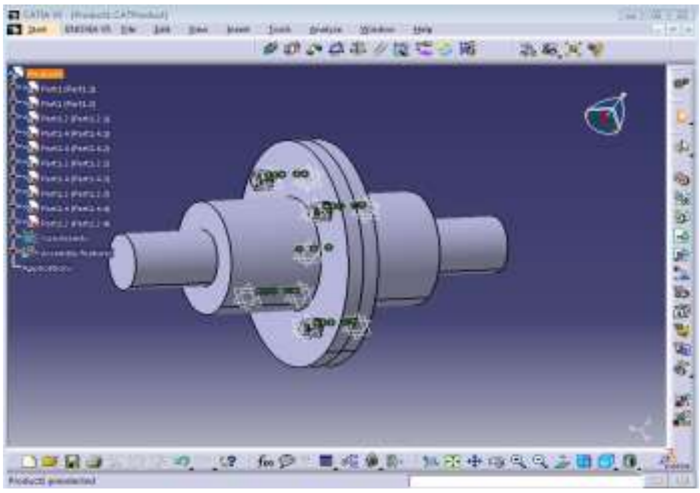

Figure 2 CAD Model of Flange coupling in CATIA

3.1 Material Properties

The material selected for the analysis is aluminium silicon carbide which is a composite material and having low mass density as compared to grey cast iron. Table 1 shows the mechanical properties of aluminium silicon carbide material [1].

Table1 Properties of aluminium silicon carbide

Material	Aluminum silicon carbide Composite
Density	2.8 g/cc
Young's Modulus	115 GPa
Poisson's Ratio	0.27
Ultimate Tensile strength	680 MPa
Ultimate Yield strength	340 MPa

3.2 Meshing and Boundary Condition

Finite Element mesh is generated using parabolic tetrahedral elements. The Von-Mises stress is checked for convergence. An automatic method is used to generate the mesh in the present work. The total number of nodes is 27435 and elements are 10893. Figure 3 shows the meshed model of flange coupling in AYSYS workbench.

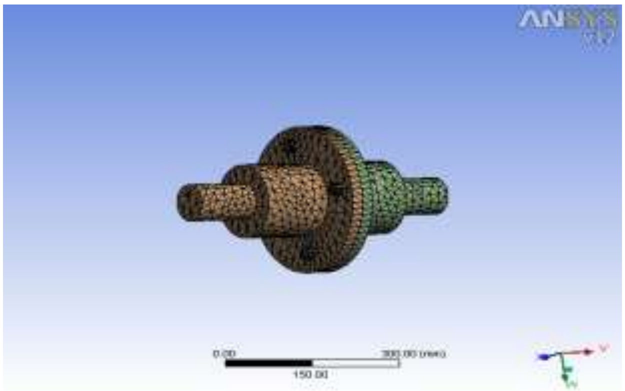

Figure 3 Meshing of Flange coupling in ANSYS

3.3 Boundary Condition

After completion of the meshing, boundary condition and loads are applied. User can define constraints and loads in various ways. This helps the user to keep track of load cases. The boundary condition is the collection of different forces, supports, constraints and any other condition required for complete analysis. Loading conditions involves applying a moment of 3.44 X 106 N-mm at the one end of shaft and other end is fixed as shown in Figure 4 [7].

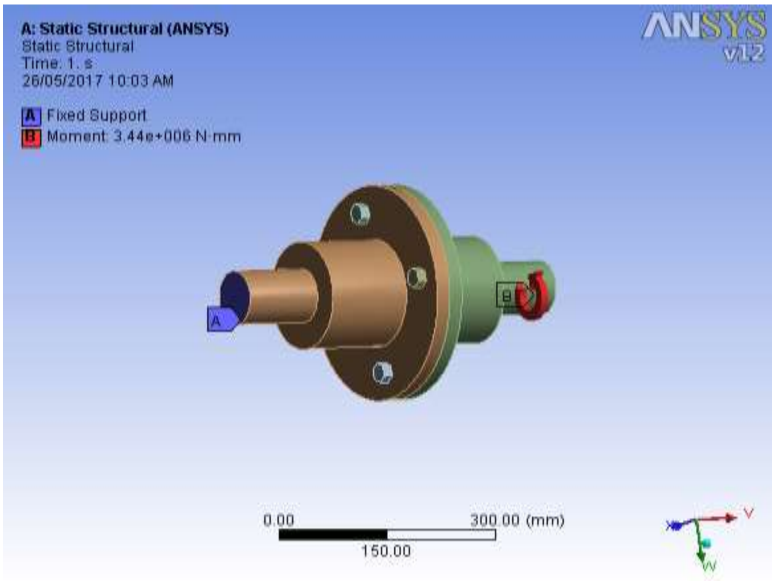

Figure 4 Boundary Condition

4. STATIC STRUCTURAL ANALSYS

Static structural analysis is done in ANSYS workbench to find out the equivalent von-mises stress, total deformation and maximum shear stress. The maximum von-mises stress, deformation and maximum shear stress is shown in Figures 5, Figure 6 and Figure 7 respectively.

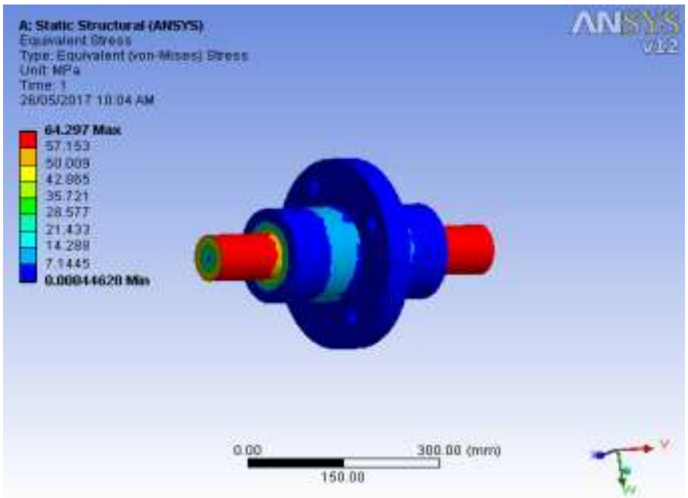

Figure 5 von-mises stress

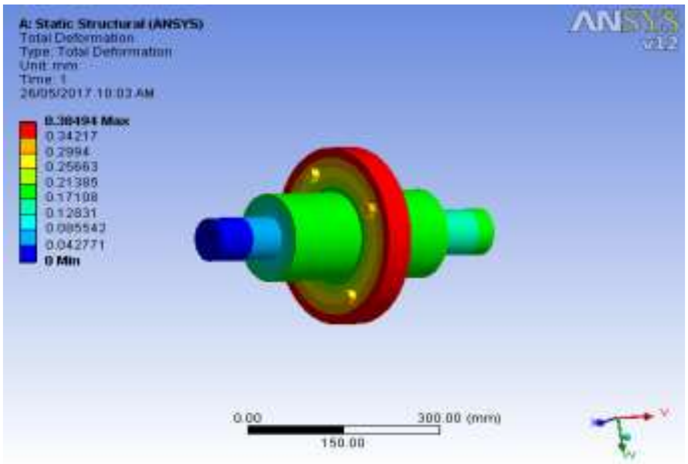

Figure 6 Total Deformation

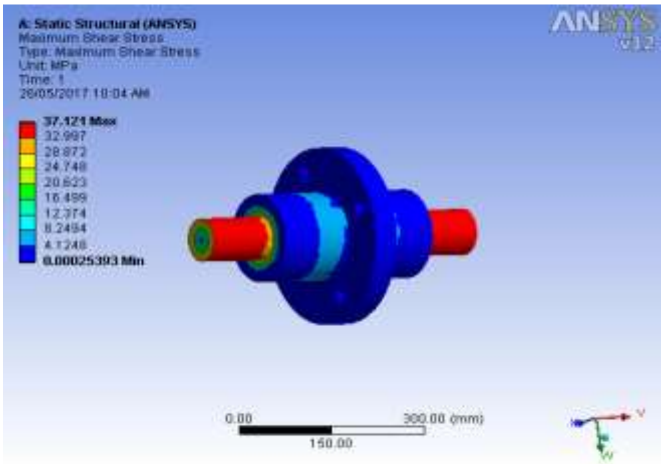

Figure 7 Maximum shear stress

The results obtained from the static structural analysis of flange coupling are shown in Table 2

Table 2 Result Comparisons of Static Structure Analysis

Sr. No.	Parameters	Existing Results	Modified Results
1.	Total Deformation in mm	-	0.384
2.	Equivalent (von-mises) stress in MPa	-	64.297
3.	Maximum shear stress in MPa	36.75	37.121
4.	Mass in kg	67.1	27.1

The FEA results are within the limit and are expectable. The values of total deformation, equivalent (von-mises) stress and shear stress are much below the yield limit and present design is safe. The mass of the flange coupling is reduced from 67.1 kg to 27.1 kg which reduces the cost of the material and thereby cost of the flange coupling.

5. CONCLUSION

The following conclusions are drawn from the present work of static structural analysis of flange coupling.

1. The maximum von mises stress is 64.121 MPa which is less than the ultimate stress.
2. The maximum total deformation for the flanged coupling is 0.384 mm.
3. The maximum shear stress found to be 37.121 MPa.
4. The mass of the flange coupling is reduced from 67.1 kg to 27.1 kg.

On the basis of the current work, it is concluded that the design parameters of the flange coupling with modification give sufficient improvement in the existing results. So main advantage of using aluminium silicon carbide composite is it reduces the weight of the component and withstands maximum applied loads.

REFERENCES

[1] Chandra Sekhar Katta, Kamana Srinivasa Rao "Design and Analysis of Flange Coupling" IJPRES, Volume VI, Issue 4, Aug 2016.

[2] D. Harsha Vardhan, M. HariPrasad, D. Mahammad Rafi, Manu Ravuri "Java for Mechanical Design Computation: Dimensions of the Various Flange Couplings", ISSN: 2278-6856, IJETTCS, Volume 2, Issue 6, November – December 2013.

[3] Dhaval B. Shah "Parametric Modeling And Drawing Automation For Flange Coupling Using Excel Spreadsheet" IJRET, Volume 1, Issue 2, July 2013.

[4] Indrajitsinh J. Jadeja, Kiran kumar M. Bhuptani "Developing a GUI based Design Software in VB Environment to Integrate with CREO for Design and Modeling using Case Study of Coupling" ISSN: 2277-9655, IJESRT, Volume 3, Issue 4, April 2014.

[5] Prof. Salunkhe R.T., Mr. Patil N.T., Mr. Lokhande V.S., Mr. Bandelkar D.M., Mr. Patil R.D. "Design, Modelling, Analysis of Propeller Shaft and Couplings" ISSN: 2249-9954, IJST, Issue 3 volume 5, Sep.-Oct. 2013.

[6] Shivaji G. Chavan "Stress Analysis of Flanged Joint Using Finite Element Method" ISSN: 2319-7064, IJSR, Volume 3 Issue 8, August 2014.

[7] Swati N. Datey , S.D. Khamankar, Harshal C. Kuttarmare "Finite Element Analysis of Universal Joint" ISSN: 2320-334X, IOSR-JMCE, Volume 11, Issue 3 Version V May- Jun. 2014.