Finite Element Analysis of A Muff Coupling Using CAE Tool

Rajeev Kumar[1,2], Manpreet Singh[1], Jujhar Singh[3],JaiinderPreet Singh[1], Piyush Gulati[1], Nitin Chauhan[1], Sumit Shoor[1]

[1]Department of Mechanical Engineering, Lovely Professional University
Phagwara, India
[1]rajeev.14584@lpu.co.in
[2]Research Scholar, Inder Kumar Gujral Punjab Technical University
Kapurthala, India
[3]Department of Mechanical Engineering Inder Kumar Gujral Punjab Technical University
Kapurthala, India

Abstract: Performance of muff coupling relies heavily on successful power transmission from one shaft to another shaft, which strongly affects the various parameters and the reliability of the muff coupling. Advances taking place in servo application have increased the need for a predictive tool for simulating the stress analysis and displacement analysis under operating conditions. A stress analysis and displacement analysis of a muff coupling has been done. This paper depicts the validation of the design of a muff coupling using Finite Element Analysis. The analysis has been carried out with actual design considerations and loading conditions. A coupled stress and displacement linear static analysis has been carried out for a given torque to determine the maximum deflection, stress distribution and its location in the muff coupling. It has been observed that stresses, displacement, in key, shaft1, shaft2, have been found within safe limits and structure could withstand the given torque.

Keywords: Coupling, Shaft, Key

Introduction

Couplings, for basic shaft connections are historically imprecise inexpensive and mostly homemade components in the pasts in any servo applications most people did not consider using a rigid coupling. A rigid coupling having smaller size especially made of aluminum, cast iron, steel etc. are mostly used in motion control applications because of its high torque capacity and zero backlash. Rigid couplings are torsionally stiff couplings with virtually zero windup under torque loads. If any deviation is present in the system the unbalanced forces will cause the shafts, bearings or coupling to fail prematurely. Rigid couplings are mostly not suitable to run at extremely high rpms because of it cannot compensate any thermal changes in the shafts due to high rpm, however in situations where deviation or misalignment can be fully controlled rigid couplings often excellent performance characteristics. A sometimes over looked advantage of rigid couplings is it can be used to determine shaft alignment in misaligned systems.

Problem Definition

Finite element analysis of a muff coupling finds widespread application in various industries, it has been carried out using CAE tools. The design of the muff coupling has been done by taking the data from various reference and books. Muff coupling is designed as a part structure for easy assembly, disassembly and removal from the shaft.The CAD model has been prepared in SOLID EDGE & the meshing tool used is Hyper mesh & the solver used is Redioss linear. The results are to be validated by comparing with the experimental results. The 2-D drawing of the muff coupling is shown below in Fig-1.1

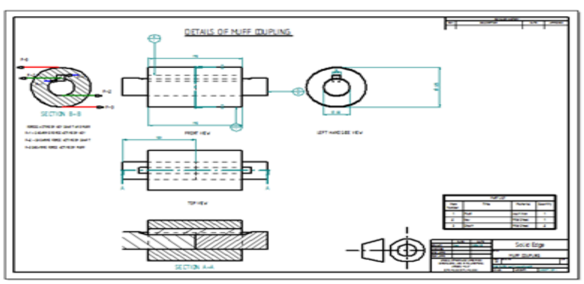


Fig 1.1-2D drawing of muff coupling

Geometry Preparation

After the CAD data has been imported, an Edit/Surface/Edge Match is performed on the geometry in order to prepare the surfaces for meshing. This involves the task of removal of features by changing the shape of a part in order to simplify the geometry. details of the shape, such as blends, small holes may simply not be necessary for the analysis purpose. When these details are removed, the analysis can run more efficiently. Meshing is done to reduce the degree of freedom from infinite to finite. The geometric surfaces of all the components of muff coupling are meshed using 2-D mixed elements. Mixed mode is commonly preferred due to better mesh transition pattern. Based upon the analysis and hardware configuration, an element edge length of 2 mm is used. For better representation of hole geometry and smooth mesh flow lines, holes are modeled with even no. of elements. After all surfaces have been meshed, next step involves equivalency all nodes.

Hypermesh has tools for determining the causes of mesh failure such as self intersections, free edges, problems with element normal's, or duplicate elements. The 3-D tetra meshing, available with Hypermesh, is used to create the solid tetra mesh. Quad elements are splitted to trias and converted to four-noded tetras. The overall analysis approach is to create a refined 3D tetra mesh from the imported CAD geometry for the muff coupling. After completing meshing part of the model, than quality check is performed to check the quality of elements because result quality is directly proportional to element quality. Different quality parameters have been performed to measures the deviations. RBE2 elements are used to distribute the forces and moment equally among all the connected nodes irrespective of forces and moment application. RBE3 elements are used to transmit torque to a body.

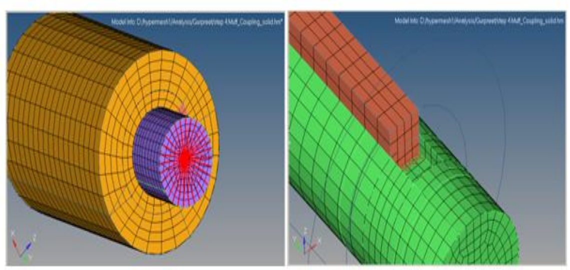


Fig 1.2 RBE3 and RBE2 Element

In material collector we assign the material to the component we created in component collector. There are four component of muff coupling. While creating the material collector we specify MAT 1 Card image to it for isotropic material. During meshing use different surfaces in different component collector. for example, the muff is in muff component collector, shaft is in shaft component collector etc.

After creating material collector, assign properties by creating property collectors. Card image used is PSOLID. For cross checking that whether the properties are assigned or not, it can be check the property tables, in utility tab.

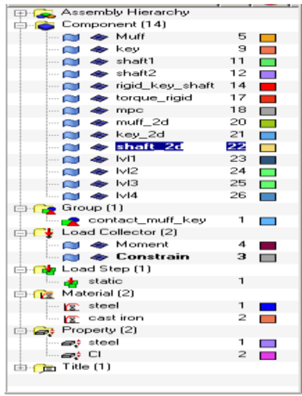


Fig 1.3 Component, property, material etc. collector

To collect or specify the forces, loads, moment, torque, pressure, velocity etc applied i.e boundary conditions. Applying load is very crucial consideration in Analysis. In muff coupling, shaft is mostly used for torque application, by creating rbe 3 element, then creating independent and dependent node and then applying the moment on the rbe3 dependent node as shown in figure given below.

.

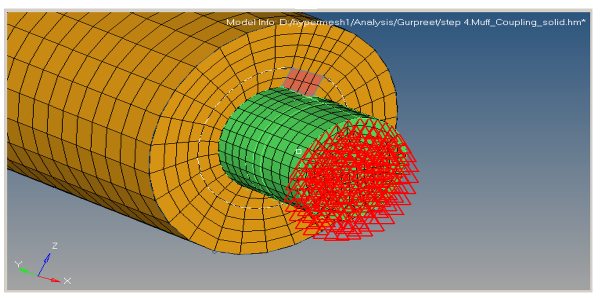


Figure 1.4 Constraint applied on shaft

The model created in the earlier steps is now taken up for solution - the computer program reads the data, generate FE model, calculates matrix entries, solves the matrix equation and writes the data out for interpretation. The finite element model was developed to predict the stress distribution and displacement distribution at key, Shaft1, and shaft2 instead of those costly and time-consuming experimental trials and errors. This task is CPU intensive, and is often called processing. Most of the time very little interaction from the user is required. The analysis of muff coupling produces different type of data files having different type of information

After the program has evaluated the results, the next step is to examine and interpret the results in HyperView. HyperView gives the information and results such as: displacement contours in x, y and z direction, resultant displacement contours, von misses stress contours.

Results and Discussions

The Figure 1.5 shows displacement distribution in the shaft1 of muff coupling. It is observed from the displacement contours that displacement is decreasing towards end of the shaft. The maximum displacement observed is 0.0081 mm at the upper surface of the shaft1. The Figure 1.6 shows displacement distribution in the shaft2 of muff coupling. The maximum displacement observed is 0.0804 mm. at the lower surface of the shaft. The Figure 1.7 shows. displacement distribution in the key of muff coupling the maximum displacement observed is 0.03622 mm.

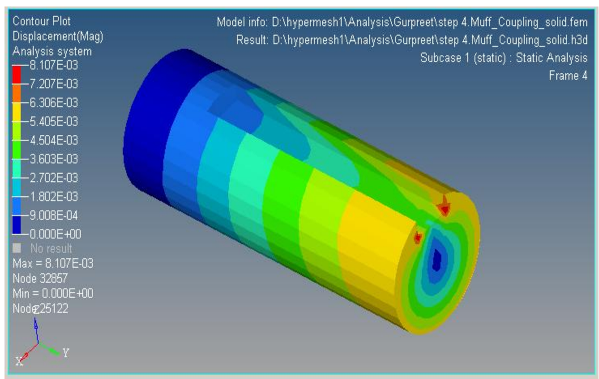


Figure 1.5 Displacement Contours in Shaft 1(torque applied)

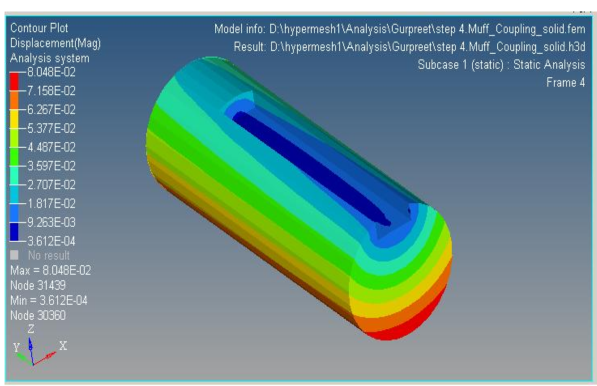


Figure 1.6 Displacement Contours in Shaft 2(constraint applied)

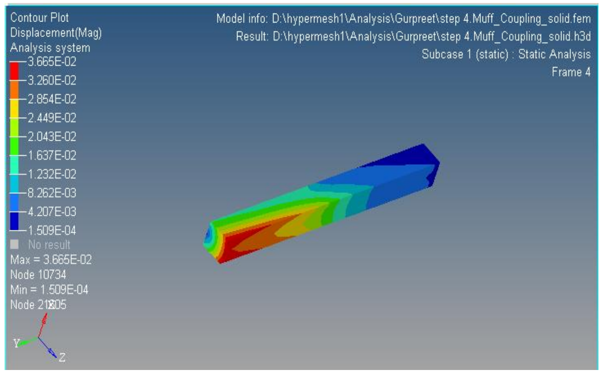


Figure 1.7 Displacement Contours in key

With the same Finite element model, the stresses in key, shaft1 and shaft2 is also predicted. The Figure 1.8 shows stress contours of the shaft1. The maximum stress observed is 29.44 Mpa and is decreasing towards end of the shaft the Figure 1.9 shows stress contour of the shaft2. The maximum stress observed is 30.31 Mpa and is also decreasing towards end of the shaft. The Figure 1.10 shows stress contour of the key. The maximum stress observed is 28.67 Mpa. The maximum stress observed in the shaft2 is 30.31 Mpa.

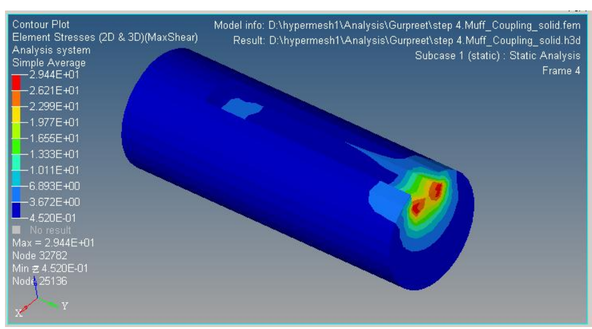
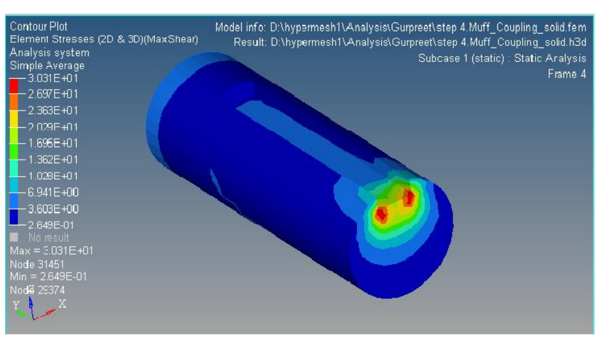


Figure 1.8 Stress Contours in Shaft 1(torque applied)



ISSN: 0971-1260

Vol-22-Issue-17-September-2019

Figure 1.9 Stress Contours in Shaft 2(constraint applied)

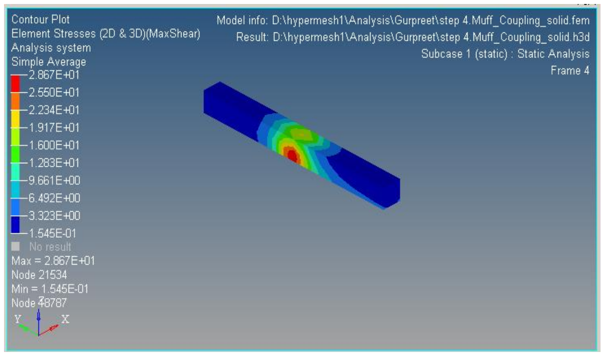


Figure 1.10 Stress Contours in key

To validate the analysis, the results have been compared with the available experimental and standard results. As the experimental results were available as shown in table 1.1 for the various component of muff coupling under a torque of $11x10^5$ Nmm the same FE analysis has been carried out for the torque of $11x10^5$ Nmm. The maximum deflection observed in shaft2 is 0.08048, which is well below the experimental result for the torque of $11x10^5$ Nmm. The stresses and displacement comparison for key also validate the FEA of the muff coupling. As from the results obtained many discussions have been made and it will be concluded that with the applied torque of 1100000 Nmm on shaft 1, there is maximum displacement on shaft 2 is found to be 0.0804 which is in the safe limit. The shear stress under same loading condition in the shaft 1, shaft 2 and key are 15.8%, 19.22% and 25.8 % error as compared with FEA. The results of the comparison have been depicted in tabular form in table.

Parameters	**Exp. Results**	**FEA Results**	**% error**
Deflection Shaft 2 (mm)	0.023	0.08048	2.49%
Deflection Key (mm)	0.01	0.03622	2.6%

ISSN: 0971-1260

Vol-22-Issue-17-September-2019

Max Shear Stress Shaft 1(Mpa)	25.422	29.44	15.8%
Max Shear Stress Shaft 2 (Mpa)	25.422	30.31	19.22%
Max Shear Stress Key (Mpa)	22.79	28.67	25.8%

Table 1.1 Comparison of experimental and FEA results

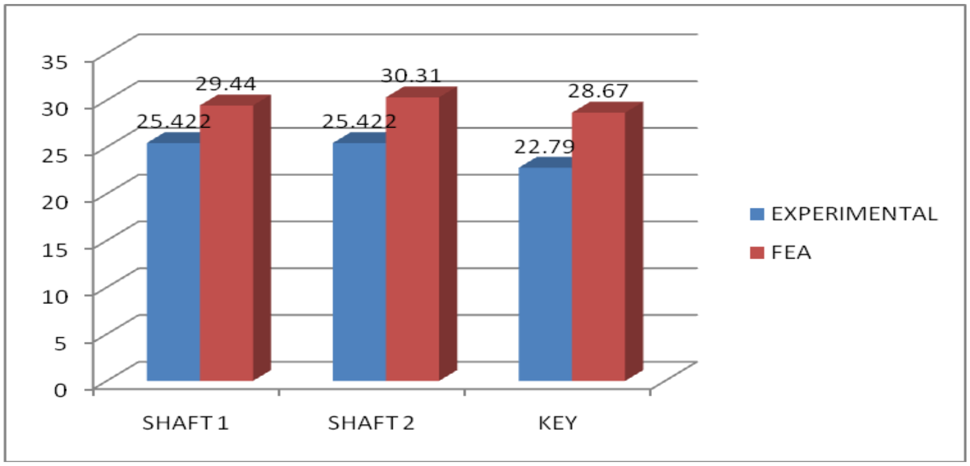


Fig- 1.11 Bar chart for Max Shear Stress (Mpa) in Experimental and FEA results

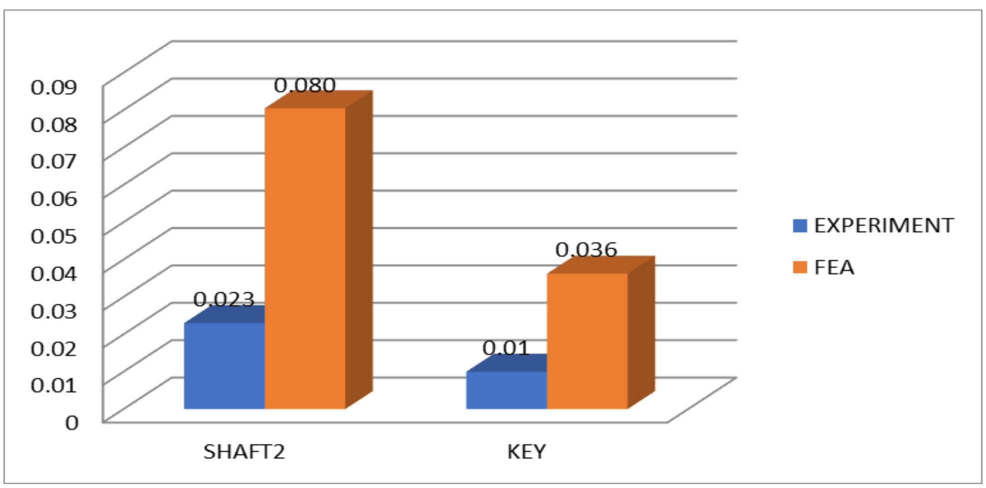


Fig- 1.12 Bar chart for Deflection in degree in Experimental and FEA results

References

[1] kumar, R., Singh, G., Singh, M., Singh, J.: Detection of crack Initiation in the ball bearing using FFT analysis, International Journal of Mechanical Engineering and Technology, 8(7), pp. 1376–1382, 2017

[2] Rabold, F. and Kuna, M.: Automated Finite Element Simulation of Fatigue Crack Growth in Three-dimensional Structures with the Software System ProCrack, Procedia Mater. Sci.: 3, pp. 1099–1104 (2014).

[3] Brezeanu, L. C.: Contact Stresses: Analysis by Finite Element Method (FEM), Procedia Technol., 12, pp. 401–410 (2014).

[4] CERLINCA Delia Aurora (2010) FEA of stress concentrator effect from a rotating disk with a keyway Fascicle of Management and Technological Engineering, Volume IX (XIX), 2010, NR2

[5] Jaiswal S.B (2012) Failure Analysis of Flange Coupling In Industry International Journal of Emerging Technology and Advanced Engineering ISSN 2250-2459, Volume 2, Issue 5, May 2012)

[6] Canann S A , Muthukrishnan S N (1998)Topological improvement procedures for Quadrilateral Finite element Meshes The MacNeal- Schwendler Corporation/PDA Engineering Technology Center, Costa Mesa CA , USA.

[7]Mahmood M Shokrieh " Analysis and Optimization of composite leaf springs, Composite Structure Volume 60, Issue 3, Pages 317-325

[8]Hyperview[Online]Available at : http://www.altairhyper works.com/ Product ,11,Hyper View.aspx

[9] Rigid Coupling[Online]. Available at :http://www.ruland.com/ps_couplings_rigid.asp (Acce ssed on 5th Dec 2012)

[10] Madan, M and Sujata, M and Raghavendra, K and Bhaumik, SK (2011) Failure analysis of an aeroengine components. Project Report. National Aeronautical Laboratory, Bangalore, India.

[11] Kenneth A. Williams (2006) Hub Design, Finite Element Analysis College of Engineering and Mineral Resources at West Virginia University

