RESEARCH ARTICLE OPEN ACCESS

Design and Analysis of Flange Coupling

Chandrakant M Patil And Anand Mattikalli
Dept. of Mechanical Engineering, Maratha Mandal Engineering College, Belagavi, Karnataka, India.

Abstract
The approach utilizes standard design equations of these couplings and links them together in computer software to determine the design parameters of the couplings. In general, most flange coupling is available in transformation system and automobile industries. A flange coupling usually applies to a coupling having two cast iron flanges. To achieve a require goal, a design of bolted unprotected flange coupling is modeled in to a cad package named Solid works. Furthered the finite element analysis module is created in ANSYS Workbench by using ANSYS Static Structural module which has a predefined process to obtain optimum results.
Keywords— Flange Coupling, Boundary condition, Loading condition

I. INTRODUCTION

The cylindrical flange coupling or a bolted joint or is used to union between different assemblies or modules into the engine. The component integrity is one of the important factors. The flange is a sensitive to the moment, torque and load path, a stress concentration area in the joint. Normally, the theory of beam is used for the analysis of maximum coupling problems. Coupling is a mechanical component and it is used to join the two shafts for transmitting power. The rigid flange coupling is specially designed and developed for horizontal shaft mounted gear unit applications. Shivaji G. Chavan [1] this paper deals with finite element analysis of bolted joint in different load conditions. This paper presents a theoretical model and a simulation analysis of flange and bolted joints deformation, stresses. The flange and Nut-Bolts force and contact stiffness factor are considered as parameters which are influencing the joint deformation. The reduced model bolted joint simulations were carried out in order to study its behaviour under the action of external loads taking into account as parameters the normal stiffness, the pretension force and friction coefficient. The flanged joint is modelled and simulation using ANSYS 14 Software. The finite element analysis procedure required in ANSYS simulation is presented as a predefined process to obtain accurate results. It is clear that force is directly proportional to stress in flanged and nut-bolted jointed. Swati N. Datey [2**],** in this project, analysis of rigid flange coupling is carried out which is similar to the universal joint. Traditional design has been done by simple calculation. But with increase in product performance and reliability it is difficult to follow the traditional iterative design procedures.

To satisfy the market needs it is necessary to provide a computational capacity along with the creativity of the human being. A numerical method is widely used to solve static structural analysis problems in both as in industries as well as in academic level. In this Finite Element Method analysis of rigid flange coupling with the help of ANSYS workbench Software for different torque and load condition. This analysis results are validated by theoretical calculation.

S.B. Jaiswal [3], this paper constitutes the failure analysis of a flange that had been welded to a high-water transmission\ pipeline. The flange had failed during the operation and that was the major point to take care. The analysis was conducted by using ANSYS Workbench 11.0 the integrity of the flange as well as that of the weld joint. The failure occurred along the weld on the flange side. The flange, which was a having material as structural steel, was expected to exhibit good weld ability. However, the analysis revealed that the structural steel had a less strength to allow the pressures. Hence, the new material alloy steel is suggested for the same and around 3 iterations are taken on Flange model in ANSYS. Jerome Montgomery [8], Modelling bolts for three-dimensional finite element applications have, and still continue to raise questions. The limitations on model size sometimes make modelling of solid bolts impractical. Therefore, many analysts choose other methods to model bolts. Line elements with coupled nodes and line elements with spider beams are a couple of alternative approaches. This paper looks at a few methods for modelling pretension bolted joints using the finite element method in ANSYS. Pretension is modelled using ANSYS pretension elements which can be used on solid or line element types. Surface-to-surface contact elements are used to account for varying contact distribution along flanges. Bolt head and nut behaviour is modelled by, coupled nodes, beam elements, rigid body elements, or solids. Bolt stud is modelled by solid elements, beam elements, pipe elements, or link elements. Doug Oatis [9**]**, Analysing a bolted flange used to be a serious undertaking, in part because of difficulties in including pretension loads produced by the installation torque of tightening the bolt. Analysts resorted to a variety of methods to account for pretension, including running a dummy thermal analysis to induce a thermal expansion loads or creating beams and constraint equations on the flange to add equivalent compressive flange loads. Pretension elements available in the ANSYS Workbench platform allow the analyst to more readily specify known axial loads or adjustments to groups of elements in accounting for these bolt installation loads. Indeed, bolt pretension is a great example of the user-friendly nature of simulation within the ANSYS Workbench environment. Through a combination of automatic contact detection, multiple meshing controls and an easy-to-use bolt-loading interface, simulation using the ANSYS Workbench platform has made including bolt pretension intuitive and speedy.

II. METHODOLOGY

A. CAD - Models

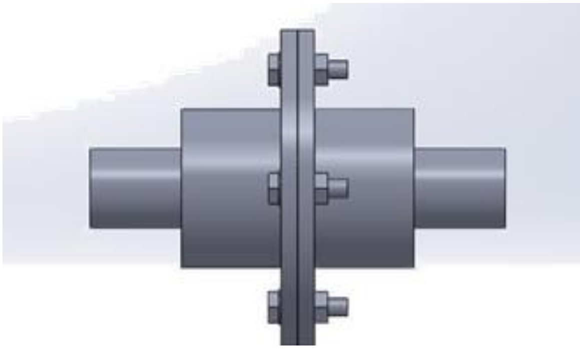

Figure 1. Front view of CAD model of flange coupling

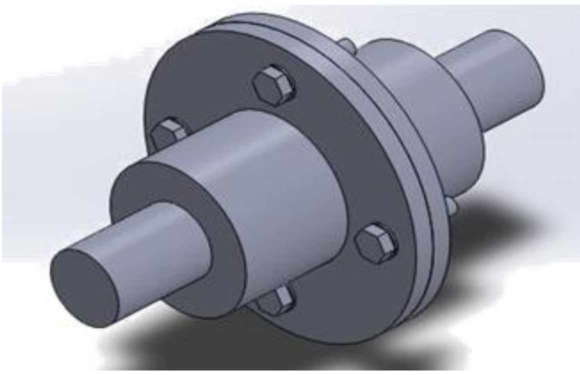

Figure 2. Isometric view of CAD model of flange coupling

The solid model of bearing component is created in SOLID WORKS V 2012 software.

B. Analytical Design of Flange Coupling

Design & draw a cast iron flange coupling for a mild steel shaft transmitting 90 Kw at 250 rpm. The allowable shear stress in the shaft is 40 MPa and the diameter of shaft is 80 mm. calculate different stresses in it. Also create a Microsoft excel sheet for this calculation.

Given data,
P = 90 kW
N = 250 rpm
d = 80 mm
The torque transmitted by shaft;
$T = (p*60)/ (2\pi N)$
$= (90*10^3*60)/ (2\pi*250)$
$= 3440*10^3$ N-mm
Considering strength of the shaft, we know that,
$(T/J) = (T_s / (d/2))$
$((3440*10^3)/ ((\pi/32)*80^4) = (T_s / (80/2))$
$T_s = 34.21$ MPa (shear stress induced in shaft)
Since the induced shear stress in the shaft is less than 40 MPa therefore the design is safe.

Design for hub,
Outer dia. Of hub = 2d = 160 mm
Length of hub =L= 120 mm
Shear stress induced in hub by considering it as hollow shaft.
$T = \pi/16* T_s ((D^4 - d^4)/D)$
$3440*10^3 = \pi/16* T_s *((160^4 - 120^4)/160)$
$T_{sh} = 4.56$ MPa
Since the induced shear stress in the hub is less than 40 MPa therefore the design is safe.

Design for key
Width of key (w) = 25mm
Thickness of key (t) = 14mm
Length of key (l) = L = 120
$T = l*w* T_{sk}*(d/2)$
$3440*10^3 = 120*25* T_{sk}*40$
$T_{sk} = 28.7$ MPa
Since the induced shear stress in the key is less than 40 MPa therefore the design is safe.

Design for flange
Thickness of flange (tf) = 0.5d = 40 Shear stress in flange
$T = ((\pi*D^2)/2)* tf* T_{sf}$
$3440 \times 10^3 = ((\pi \times 160^2)/2)* 40* T_{sf}$
$T_{sf} = 2.14$ MPa
Since the induced shear stress inthe flange is less than 40MPa therefore the design is safe.

Design for bolt
d_1 = nominal dia. of bolt = 18 mm
d = dia, of the shaft = 80 mm
Number of bolts (n) = 4
Pitch circle dia. Of bolt (D1) =3*d = 240 mm
The bolt are subjected to shear stress due to torque

transmitted

$T = (\pi/4)*d1^2 *n* T_{sb}*(D1/2)$

$3440*10^3 = (\pi/4)*18^2 *4* T_{sb}*(240/2)$

$T_{sb} = 28.16$ MPa

D2=4d =4*80 = 320 mm

Thickness of protective circumferential flange

Tp = 0.25d = 0.25*80 = 20 mm

Meshing of flange coupling

In this work, ANSYS WORKBENCH is used for a meshing of flange coupling. It creates sufficient smooth meshing as shown in figures below. All components meshed with 10-node tetrahedral structural solid element that is SOLID 185. Patch confirming method is used for meshing. It has quadratic displacement behavior and is well suited to modeling irregular meshes. The element is defined with 4 nodes and have three degrees of freedom (DOF) at each node .

Statistics	
Nodes	15827
Elements	69855
Mesh Metric	Element Quality
Min	0.127532447084992
Max	0.999637394487151
Average	0.792067944802517
Standard Deviation	0.113713449180591

Figure 3 Meshing statistics of flange coupling

Statistics	
Nodes	14926
Elements	60785
Mesh Metric	Element Quality
Min	0.151397349742798
Max	0.999945012159449
Average	0.791917571433636
Standard Deviation	0.115321350962879

Figure 4 Meshing statistics of flange coupling.

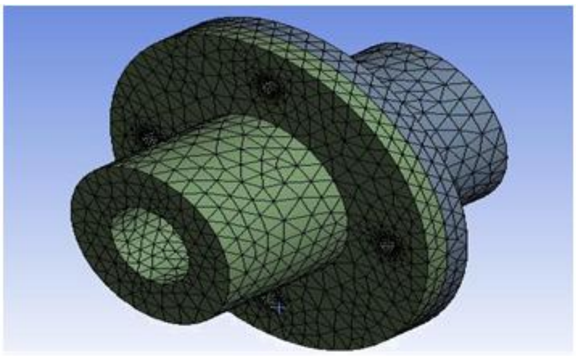

Figure 5 Meshing of hub in workbench.

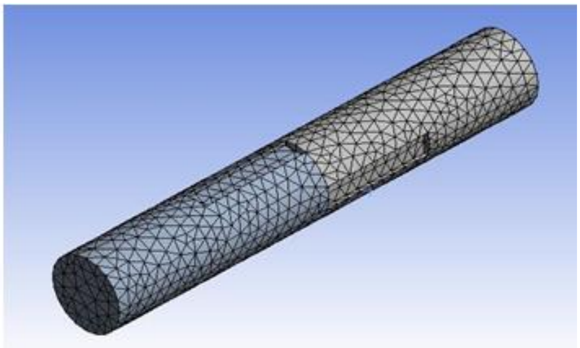

Figure 6 Meshing shaft in workbench.

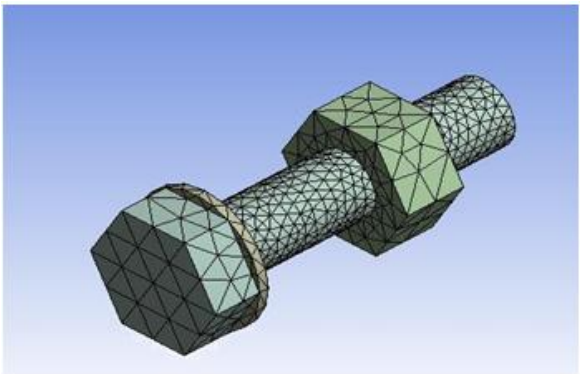

Figure 7 Meshing of nut-bolt-washer assembly in workbench.

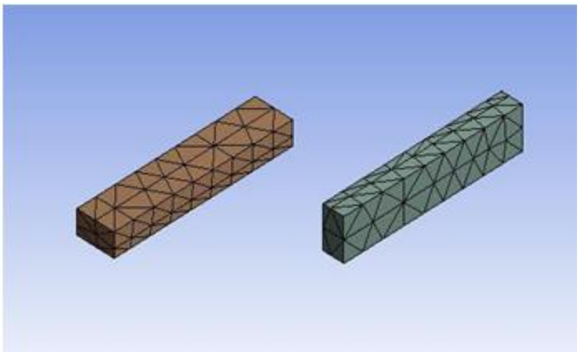

Figure 8 Meshing of keys in workbench

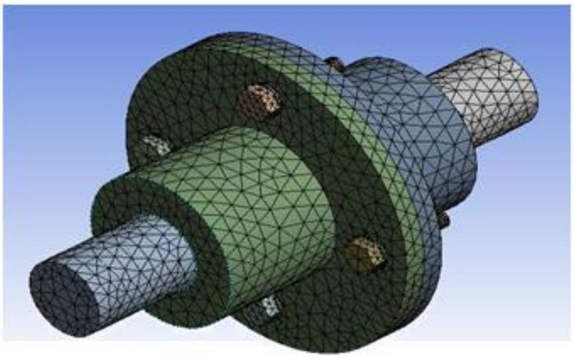

Figure 9 Meshing of flange coupling in workbench

C. Boundary condition

A fix support is used to fix a flange from one end.

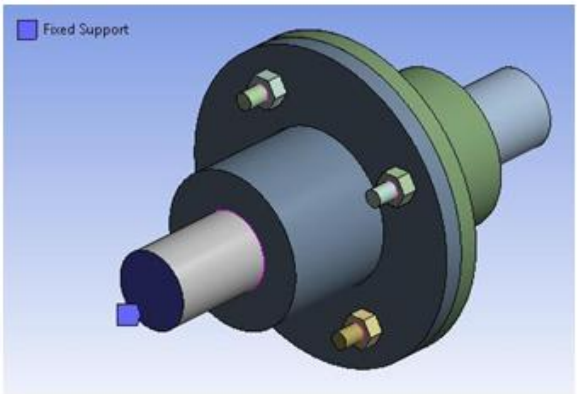

Figure 10 Fixed support.

Loading condition

A load is provided in two stages. In 1st stage bolt pretension is provided on flange bolts. A 1000N pretension load is applied on each bolt which will acts as a tightening load. In 2nd stage a constant torque is applied with a value of 3440000MPa along X-axis.

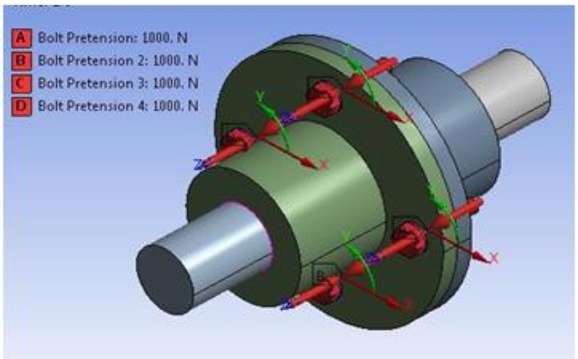

Figure 11 Bolt pretension.

D. Analysis

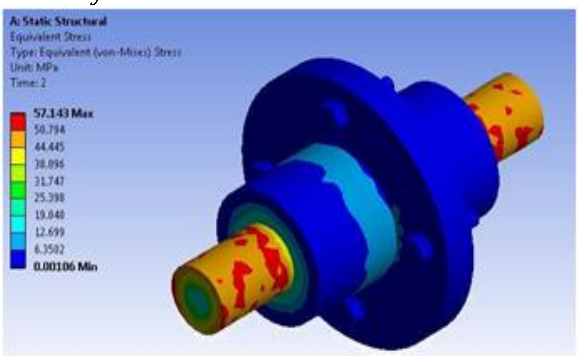

Figure 12 Equivalent stress of flange coupling in workbench

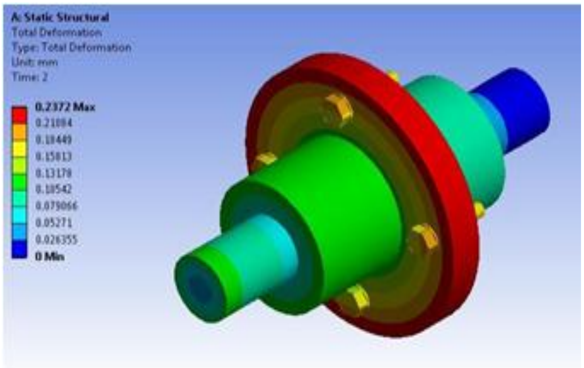

Figure 13 Total deformation of flange coupling in workbench.

E. Results and Discussion

The reduced model of flanged-bolted joint simulations was carried out in order to study its behavior under the action of external loads taking into account as parameters the pretension force. The influence of these parameters on the deformation of the bolted flange jointed is analyzed.

Typically, flanged–bolted jointed are designed for carrying tensile loads and shear loads. In a tensile bolted joint the most critical parameter is the applied pretension load. In such joints, a fastener that can provide the maximum tensile load is preferable. Similarly, if the bolt is subject to shear loads the effective shear area at the shearing planes needs to be maximized. In general, it is assumed that the failure will occur in the bolt (which is preferable and the design criterion for failure). In a first order analysis (assuming total load is evenly distributed over the entire engagement length) on the load carrying capacity of a bolt, it is found to be directly related to the shank diameter.

The figures show different contacts between the bolted flange assemblies as well as in between bolt and nut. The penetration of contacts and targets has been checked and occurred a very small which is considerable for this analysis system. Also, The gap between contacts and targets has been checked and occurred a very small value, which is also considerable for this analysis system of flange coupling.

Defining joints is one of the most difficult aspects when simulating the behavior of machine-tools, because there are many variables that can affect the joint's properties.

By using finite element analysis software we can optimize the design process of machine tool components by identifying the parameters that has a influence on the static behavior of machine tools.

By analyzing the results obtained in the post-processing phase, the user can evaluate the properties of machine tools still in the design stage, without the need to make prototypes. Based on these preliminary results of flange coupling deformation as well as shear analysis further research will be carried on the model optimization. By knowing the influence of the parameters on the contact deformation we can optimize the shape of the structural components of coupling.

III. ACKNOWLEDGMENT

The authors would like to thank MM Engineering College, Belagavi for support given to this project work.

REFERENCES

[1] Shivaji G. Chavan, Stress Analysis of Flanged Joint Using Finite Element Method, International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064 Volume 3 Issue 8, August 2014.

[2] Swati N. Datey, S.D. Khamankar, Hershel C. Kuttarmare, Finite Element Analysis of Universal Joint of IOSR.

[3] S.B. Jaiswal1, M.D. Pasarkar, Failure Analysis of Flange Coupling In Industry, International Journal of Emerging Technology and Advanced Engineering, ISSN 2250- 2459, Volume 2, Issue 5, May 2012.

[4] Jerome Montgomery, Methods for Modeling Bolts in the Bolted Joint, Siemens Westinghouse Power Corporation, Orlando, FL.

[5] Doug. J.Oatis, Analyzing Pretension in the ANSYS Workbench Platform, ANSYS Advantage • Volume I, Issue 4, 2007

AUTHORS' BIOGRAPHIES

Mr. Chandrakant M Patil student at Maratha Mandal Engineering College, Belagavi, completing his M.Tech in Mechanical Machine Design of Visvesvaraya Technological University, Belagavi, Karnataka, India.

Mr. Anand Mattikalli is a Professor, Dept. of Mechanical Engineering, Maratha Mandal Engineering College, Belagavi, and he is my project guide.