# COMPARISON BETWEEN NUMERICAL AND THEORETICAL STUDIES FOR: FLANGE COUPLING DESIGN

**Asya A. Gabbasa[1*], Othman M. Daas[2]**

[1] Faculty of Oil and Gas Engineering, University of Zawia
[2] Faculty of Engineering, University of Zawia
[*] a.gabbasa@zu.edu.ly

## Abstract

The purpose of this paper is to compare the numerical and theoretical studies for designing and analyzing the stresses of a gray cast iron-protected flange coupling that connects a motor to a pump.

The flange was designed according to the analysis of the theoretical structure, and was modeled into a three-dimensional model by using the Solid Works program. The simulation was performed by using ANSYS Static Structural where an appropriate mesh, boundary conditions, and solution parameters were selected. Boundary conditions were the torque on the surface of the driving shaft and the fixed driven shaft.

The results have shown that the shaft and key are mostly affected by the applied torque. The applied torque had relatively little effect on the resulting parameters of the flange and the only significant effect on the bolt was at the points of contact with the flange.

The simulation results were compared with the theoretical and permissible pressures of the selected material; this comparison showed that the flange coupling material can withstand the applied torque without failure or deformation and within the acceptable margin of protection.

**Keywords:** Machine Design; Stress Analysis; Flange coupling.

## Introduction

To connect shafts together for the purpose of power transmission and torque, a device known as the coupling is used. Couplings are used to join separate manufacturing units, and shaft units, such as electric motors, motors, and generators, and centrifugal pumps.

For the duration of transporting shafts of greater length, it becomes essential to join or greater shafts by coupling. The shafts, which might be linked by coupling, need to be easily sufficient to assemble and dismantle for repair purposes, and altering extreme failures due to shear bolts-heads head, key head, nuts, and other projecting parts that may cause accidents.

Therefore, it must be covered by giving a proper shape to the flange or by providing guards. Flange couplings are further classified into rigid and flexible coupling. Rigid flange coupling contains two separate grey cast iron flanges; one is attached to the drive shaft and the other to the driven shaft by nuts and bolts which are arranged on a concentric circle with the axes of the shafts.

Rigid flange connections are of two types, namely shielded and unshielded rigid flange couplings. In a protected rigid flange coupling, a protective circumferential rim covers the nut and bolt head. So, in case of failure of bolts during operation, a broken piece of the bolt will dash against this rim and eventually fall down, protecting the operator from any possible injuries. In unprotected rigid flange coupling, a protective circumferential rim is absent. So, the failure of bolts may hit and harm the operator.

Rigid flanges are usually synthetic by casting as it includes projection and recess. The typically used material for flange coupling is gray cast iron that is characterized by graphitic microstructure inflicting fracture of the material to have a grey appearance. It is one of the mostly used variety for its casting properties. Kondru et al (2017) showed that maximum alloys of iron contain 2.5-4% carbon, and 1-3% silicon and the rest of weight proportion is iron. It has much less tensile strength and shock resistance as compared to its compressive strength. Its mechanical properties are manipulated by using the size and morphology of the graphite flakes, which deflect a passing crack and initiate counter less cracks as the material breaks; it has the right wear resistance and damping ability. Moreover, it has much less solidification shrinkage than the other types of cast iron that does not form a graphitic microstructure at some stage in the casting technique. Silicon promotes good corrosion resistance and increases fluidity whilst casting. It additionally gives a proper weld capability (Bhandari, 2010).

Protective flanges are supplied to guard the heads and nuts of the fallen bolts. The bolts are positioned at the same distances on the diameter of the bolt circle and the range of bolts relies upon the shaft diameter (Bhandari, 2010). For easy assembly, a spigot is furnished on one flange and a recess on the other face. Design procedure is totally based on determining the diameter of the shaft to transmit a sure torque after which empirical relationships are obtained for different dimensions of the coupling. (B. Kondru et al, 2017)

**Methodology**

The model used to represent the flange coupling that connects the motor to the pump is shown in Figure (1). d, L, D, D1, and D2 constitute the shaft diameter, flange duration, hub diameter, bolt pitch circle diameter, and flange outer diameter respectively. tp and tf represent the thickness of the protective circumferential flange and flange thickness respectively. The model is based on the following assumptions: no time-varying masses are implemented, the load is static, material properties are continuing, homogeneous, and isentropic, no internal stresses are there before loading, and finally accelerations equal zero. Tables (2) and (3) show gray cast iron allowable stresses and material properties respectively.

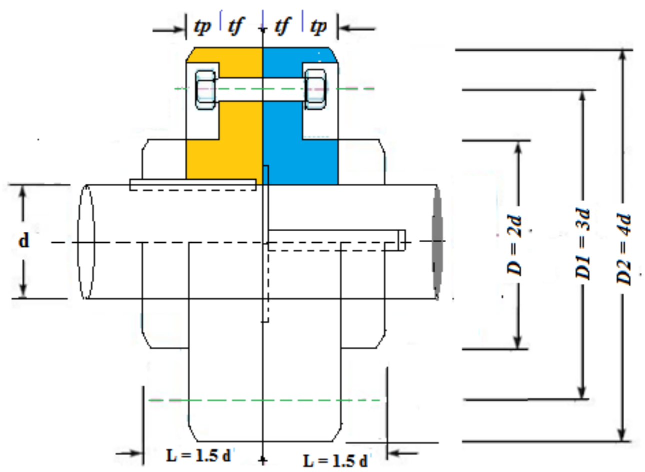


**Figure (1): Flange Coupling Model. (R.S Khurmi; J.K Gupta, 2005).**

## Numerical Analysis Method

### Finite Element Analysis

ANSYS Static Structural is a general-purpose finite element modeling package that digitally solve a variety of mechanical problems. These problems include static/dynamic structural analysis (linear and nonlinear), heat and fluid transfer problems, as well as acoustic and electromagnetic problems.

In general, a finite element solution may be broken into the following three stages, which constitute a general guideline that can be used for setting up any finite element analysis: -

1. Preprocessing: defining the problem
   The most important steps in the quantity of detail required will rely on the dimensionality of the analysis (i.e., 1D, 2d, axisymmetric, 3-D).
2. Solution: assigning loads, constraints, and solving
   Here we specify the loads (point or pressure), and constraints (translational and rotational) and eventually solve the resulting set of equations.
3. Post-Processing: in addition, to processing and viewing the outcomes;

### Boundary Conditions

Boundary conditions include a fixed support on the front surface of the left shaft and a clockwise moment of 215 KNm. caused by transmitting 15Kw at 900rpm on the front surface of the right shaft. The method used to calculate the momentum will be explained in the next section. Figure (2) shows the boundary conditions used.

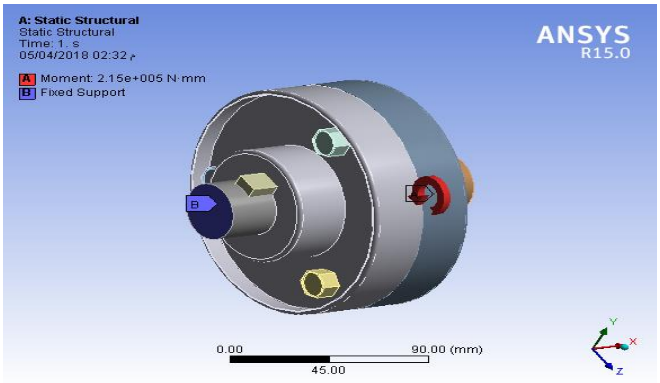


**Figure (2): Boundary Conditions Used.**

**Simulation Process**

This program was used for its accuracy and flexibility. The procedures taken when running the simulation are explained next.

- selecting an appropriate mesh since static structural is a finite element-based solver. The mesh is chosen so that the time taken to reach conversion is kept to the minimum without affecting results accuracy. A reference value of 35 was found to be appropriate; the referee was used for the mesh. Select update to apply the mesh, Figure (3) shows the mesh used while Table (1) shows the mesh properties used.

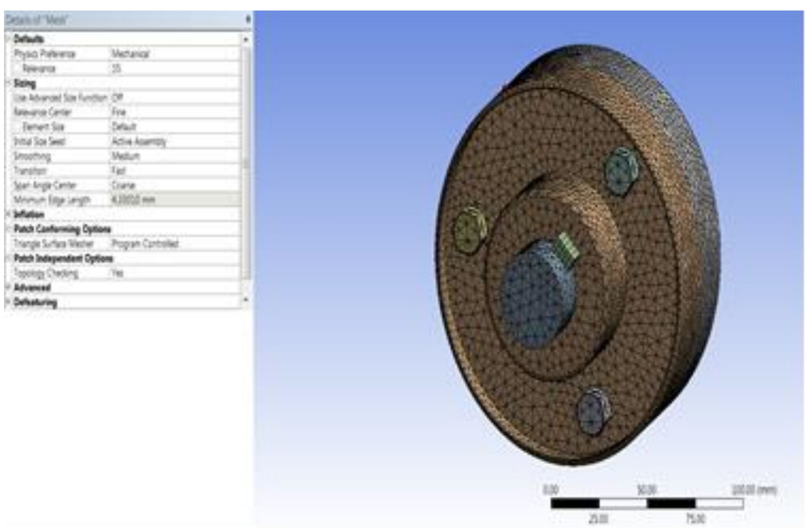

**Figure (3): Setting the Mesh.**

**Table 1 Mesh Properties.**

| | |
|---|---|
| Element size | 0.001mm |
| Reference value | 35 |
| No. elements | 1055061 |
| No. nodes | 1675177 |

The boundary conditions; select the left side of the flange coupling driven shaft then select “support” then “fixed support” then select "apply"

To apply the moment on the right side, select “load” then “moment” After that select “Geometry” then select the right side of the flange coupling driving shaft then enter the desired magnitude, in this case, 215000 N.m.

select the solution parameters needed to study which are maximum principle stress, minimum principle stress, normal stress, maximum shear stress, shear elastic stress, equivalent (von Mises), normal elastic stress and total deformation then select “Solve”, the result parameters where chosen based on a previous study by (Shivaji and Chavan,2014).

**Results and Discussion**

In this section, simulation results will be presented, analyzed, and discussed. Results include an analysis of each flange coupling part, the entire flange coupling as one part, and a comparison between theoretical and simulation results.

**Flange Coupling Analysis**

1- Maximum principal stress

The shaft and key had the highest maximum principal stress while the rest of the flange coupling was hardly affected. This is due to the fact that flange coupling absorbs the forces, which act on it because of the applied torque.

The highest maximum principal stress occurred at the contact points between the shaft and the key as well as between the key and the hub; this is due to the forces that act on the key from the shaft and due to the forces of the key that act on the hub. Figure (4) shows the flange coupling’s maximum principal stress.

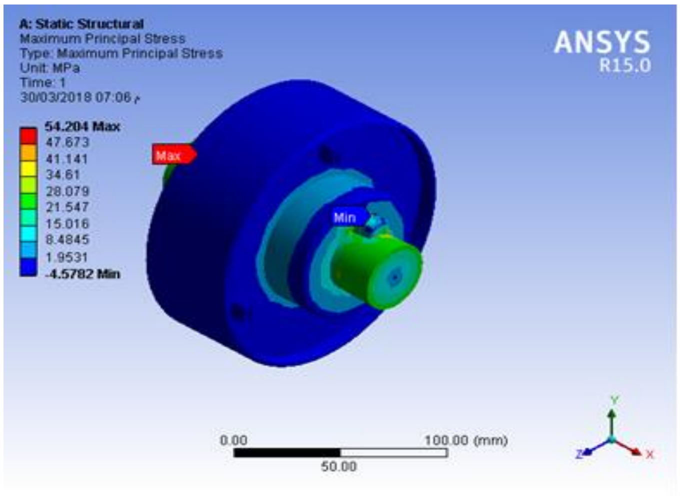


**Figure (4): Flange Coupling Maximum Principal Stress.**

2- Total deformation

Since the flange coupling is made from the same material, the total deformation (Z-axis) increases linearly with the vertical distance from the applied torque. The lowest total deformation occurred at the center of the shaft and the highest total deformation occurred at the outer layer of the flange. Figure (5) shows the total deformation of the flange coupling.

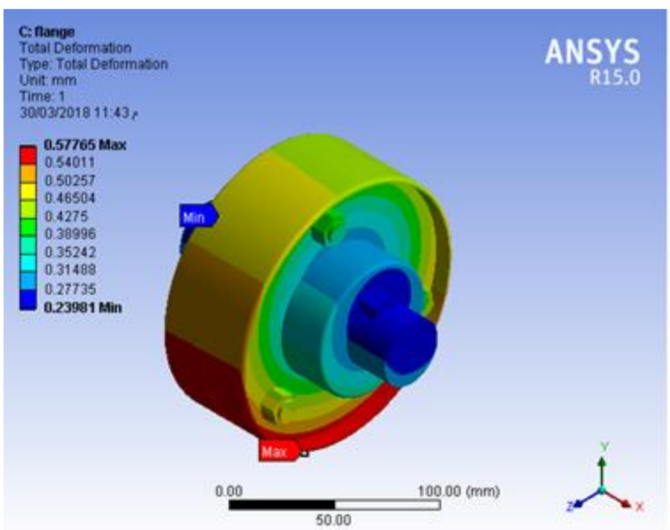


**Figure (5): Flange Coupling Total Deformation.**

1- Maximum shear stress

The highest maximum shear stress occurred at the key and shaft especially at their contact points however the maximum shear stress continued to decrease as it moved away from the contact point where most of the flange and hub were hardly affected. Figure (6) shows the flange coupling's maximum shear stress.

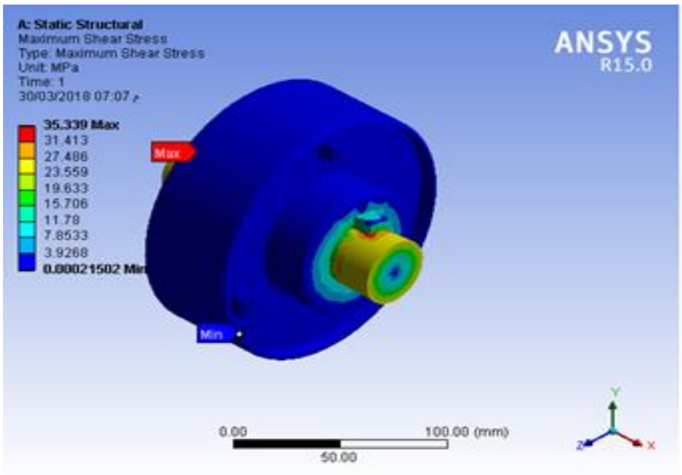


**Figure (6): Flange Coupling Maximum Shear Stress.**

2- Minimum principal stress

The highest minimum principal stress occurred at the key and shaft, especially at the contact points between the shaft and key and the contact points between the key and the flange. Most of the flange coupling had a significantly lower minimum principal stress as compared to the contact points. Figure (7) shows the flange coupling's minimum principal stress.

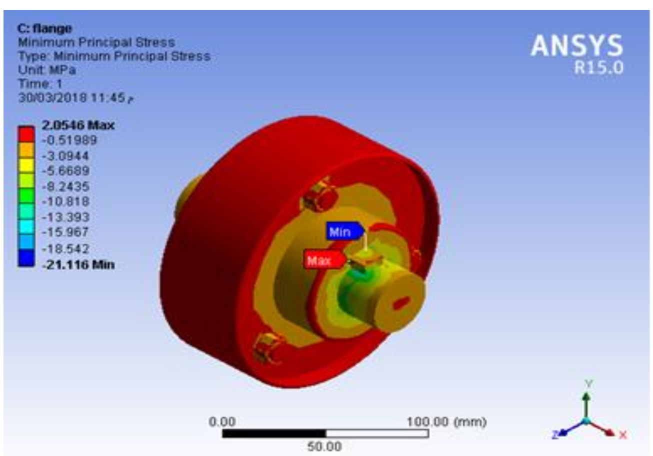


**Figure (7): Flange Coupling Minimum Principal Stress.**

a. Normal stress

The applied torque had a relatively insignificant effect on the normal shear stress where only the contact point between the shaft, key, and flange had any significant normal stress. Figure (8) shows the flange coupling's normal stress.

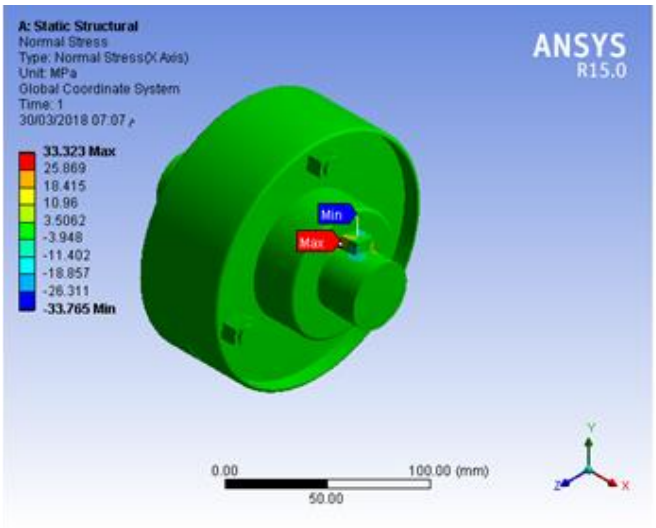


**Figure (8): Flange Coupling Normal Stress.**

b. Shear stress

The highest shear stress occurred in the shaft and the lowest shear stress occurred in the bolt, hub, and flange. Figure (9) shows the flange coupling's shear stress.

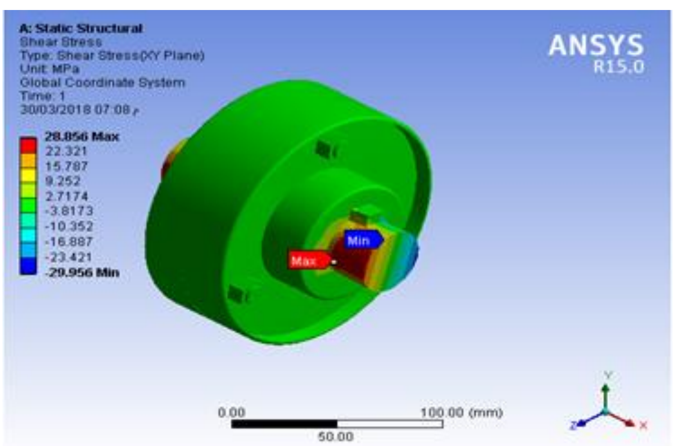


**Figure (9): Flange Coupling Shear Stress.**

Figure (9) shows that the only significant shear stress was at the shaft. On the other hand, the insignificant shear stress was at the rest of the flange coupling's components.

c. Equivalent stress (von Mises)

The applied torque had a relatively insignificant effect on the equivalent stress where only the contact point between the shaft, key, and flange as well as the blots and flange had any significant equivalent stress. Figure (10) shows the flange coupling's equivalent stress.

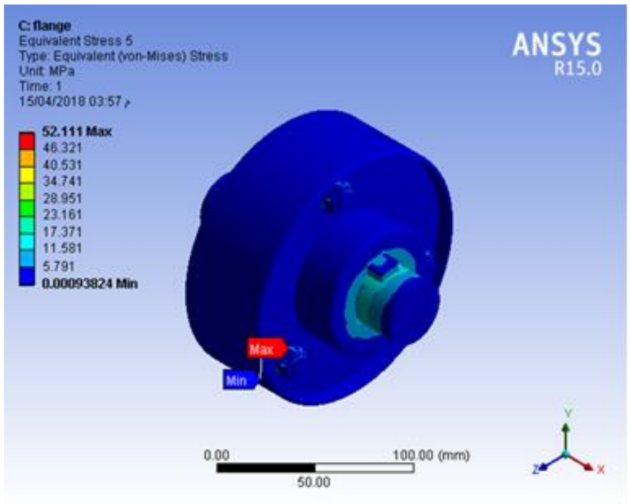


**Figure (10): Flange Coupling Equivalent Stress.**

d. Allowable stresses

To ensure the safety of the flange coupling, it is necessary to compare the simulation stresses with the allowable stress of the chosen material and component. Table (2) shows the comparison between the theoretical stresses and the allowable stresses, and Table (3) shows the simulation stresses. (Gabbasa et al, 2022).

**Table (2): Allowable Stresses and Theoretical Stress Comparison.**

| | Allowable (MPa) | Theoretical result (MPa) |
|---|---|---|
| Shear stress in the shaft material | 40 | 25.53 |
| Shear stress in key | 40 | 19.5 |
| Shear stress in the bolt | 40 | 3.5 |
| Crushing stress in key | 80 | 39 |
| Crushing stress in the bolt | 80 | 7 |

**Table (3): Theoretical Stresses and Simulation Stress Comparison.**

| Parameters | Theoretical result (MPa) | Simulation (MPa) |
|---|---|---|
| Crushing stress of bolt | 7 | 17.56 |
| Shear stress of bolt | 3.5 | 17.52 |
| Crushing stress of key | 39 | 38.652 |
| Shear stress of key | 19.5 | 19.968 |
| Shear stress of shaft | 25.53 | 25.57 |

**Table (4): Allowable Stresses and Simulation Stress Comparison.**

| Parameters | Allowable (MPa) | Simulation (MPa) |
|---|---|---|
| Crushing stress of bolt | 80 | 17.56 |
| Shear stress of bolt | 40 | 17.52 |
| Crushing stress of key | 80 | 38.652 |
| Shear stress of key | 40 | 19.968 |
| Shear stress of hub | 40 | 24.32 |
| Shear stress of shaft | 40 | 25.57 |

It can be seen in Table (4) that all the stresses gained from the simulation were less than the allowable stresses of the chosen material; this proves that the coupling can endure the forces acting on it without failing or deforming.

## Conclusion

For the design and analysis of a shielded flange coupling, the dimensional model of a flange coupling using Solid Works was designed on bases of the results of the theoretical analysis of a flange coupling.

Flange coupling components include a shaft, key, hub, and flange and bolts. Gray cast iron was chosen as the flange coupling material. The simulation model was designed and developed by using ANSYS Static Structural, and grid, boundary conditions, and result coefficients were set there. Boundary conditions included a fixed support on the left shaft and a torque of 215,000 Nm on the right shaft.

The results showed that the shaft and key are the most affected components by the applied torque, especially at the points of contact between the shaft, key, and hub. This effect decreased steadily as it moves away from the touchpoints.

The applied torque had relatively little effect on the resulting parameters of the flange and was the only effect it had on the screw at the points of contact with the flange. It should be noted that bolt stresses were higher in simulation than in the theoretical analysis.

The results gained from the simulation were compared to the permissible stresses of the selected material to ensure the integrity of the flange coupling and that the flange connection would not fail or permanently deform under the applied torque. The results of this comparison showed that the flange can hold out the applied torque coupling material without malfunction or deformation and were within an acceptable margin of safety.

**References**


- Asya A. Gabbasa and Othman M. Daas (2022) "Theoretical design and stress analysis for flange coupling", Journal of Applied Science Issue (9) September (2022).
- B. Kondru, C. Syed, D. Khatroth, J. Kachve (2017) 'Design and Failure Analysis of Flange Coupling with Uniform Strengthen Bolts', International Journal of Innovative Research in Science, Engineering, and Technology, Vol. 6, Issue 1.
- D. Shah, (2013),' Parametric Modeling and Drawing Automation for Flange Coupling Using Excel Spreadsheet', International Journal of Innovative Research in Science, Engineering, and Technology, Vol. 1, Issue 2,187-192.
- G. Andrzej; B. Michał (2010) 'Fundamentals of Machine Design', University of Technology, polish.
- R, Saurav, D. Debayan, J. Pawan, (2015) 'Design and Stress-Analysis of a Rigid Flange Coupling using FEM' International Journal of Innovative Research in Science, Engineering, and Technology, Vol. 4, Issue 10.
- R.S Khurmi; J.K Gupta, (2005) 'Machine Design', Eurasia Publishing House (PVT.) LTD.
- Shivaji G. Chavan "Stress analysis of flanged joint using finite element method". (IJSR), Volume 3 Issue 8, August 2014, ISSN (Online).