Finite Element Analysis of Offshore Drilling Structure

Ragesh Sathiyan M[1*], Shreeprakash B[2]
[1]*Post-Doctoral Fellow, Department of Mechanical Engineering, Srinivas University, Mangalore, India.*
[2]*Professor, Department of Mechanical Engineering, Srinivas University, Mangalore, India*

Abstract- The offshore module drilling rig has developed into an essential component of equipment for offshore oil and gas extraction, particularly for its large potential use in deep water, analysis of its load-carrying capacity assessment is of great significance. The strength analysis of an offshore drilling structural assembly with a lifting load of 15 tonnes and a partial safety factor of 2 is explored in this work. Finite Element (FE) software ANSYS is used to calculate the loads based on the loading application. ANSYS used to define the materials, geometrical dimensions, loads, and boundary conditions for the drilling structures. As a result, deformations are obtained at 13.2 mm, which is less than the permitted maximum of 35 mm. The highest allowable limit of 235 MPa is determined to have stresses that are substantially below the offshore drilling structure. Therefore, the offshore drilling structure fulfils the strength criteria.

Keywords: Offshore drilling structure, FEA, ANSYS, cutting unit, winch, floating platform, pontoon, suction pipe, slurry, boundary conditions and loads

1. INTRODUCTION

Energy resources are a key factor in a nation's continual development and advancement, and the development of many nations depends on the management of such energy sources. The most extensively used energy sources are oil and gas, and the development and use of these resources are crucial measures of the state of an economy. Consequently, there are rigorous standards for both the operation's safety and the equipment's dependability. As the supply of onshore and inshore petroleum resources declines, the petroleum industry has come to agree that deep water oil and gas resources should be exploited. However, the harsh ocean environment makes Deepwater drilling less safe. As a result, severe specifications for offshore drilling equipment are required. As opposed to onshore drilling, offshore drilling uses a drilling riser system to transport drilling rigs and drilling mud to the wellhead. The length of the riser system is dependent on the depth of the sea. While a flexible joint and telescopic joint on its topside connect it to a drilling platform or a drilling vessel. The riser will deflect and vibrate as it swings with the ocean environmental loads during offshore drilling. Once the drilling riser breaks or falls into the ocean, there will be significant environmental damage and significant financial loss. Under these conditions, it is crucial to accurately predict and manage the dynamics of the drilling riser in order to guarantee the safety of the Deepwater drilling operation. Currently, the theory of strength, stiffness, dependability, and stability as well as the limited test data on derricks are used to construct the methodology for derrick assessment. The evaluation of Derrick made above using linear extrapolation of the scant test data is not exhaustive or scientific. To assess the stress and stresses occurring on its structure, simulation techniques are applied.

Studies on the use of drilling risers are widely available today. The marine riser exposed to a combination of random waves and constant current is investigated using the spectrum method to determine the parameters that affected its dynamic response. Then, the static/dynamic assessments of an offshore drilling riser under various environmental circumstances are performed using the ANSYS software. In order to analyse a drilling riser with its two ends hinged in the floating platform, the dynamic analysis model is used. Some researchers prefer to incorporate the dynamic positioning system onto the drilling platform to facilitate platform accessibility while avoiding the excessive riser end angle.

In this work, the strength analysis of an offshore drilling structure with a lifting load of 15 tonnes and a partial safety factor of 2 is explored. Finite Element (FE) software ANSYS is used to calculate the loads based on the loading application. ANSYS used to define the materials, geometrical dimensions, loads, and boundary conditions for the drilling structures.

2. PROPOSED METHODOLOGY

Development of a FE model in ANSYS Workbench for an offshore drilling structure, which analyse the boundary conditions and loads. The cutting unit evenly distributes loads to the structure. One end of the structural is regarded as a fixed support. Cutting load is referred as the cutter's cutting force. Winch load lifting is considered as occurring upward. The force applied by side winches is measured horizontally. Along with 1.3t/m3-density slurry, the mass of the suction pipe is taken into consideration. Every welded connection is regarded as a bonded contact. ANSYS is used to do deflection and static stress calculations.

A. Modelling Of Finite Element

The finite element analysis is explained briefly in below,

a) Model characteristics

- Linear elastic material properties considered for all components.
- Quasi-static loading of structure.
- FE models of offshore drilling structure assembly are meshed with linear hexahedral elements.
- Neglect of small holes and edges.
- Model units: N, mm, t, s.

b) Coordinate system

The CAD models determine the coordinate system. The axes are arranged so that the y-axis goes upward along the tower axis and that the x- and z-axes produce a right-handed coordinate system. Figure 1 specifies the geometry of the drilling unit with the suction pipe.

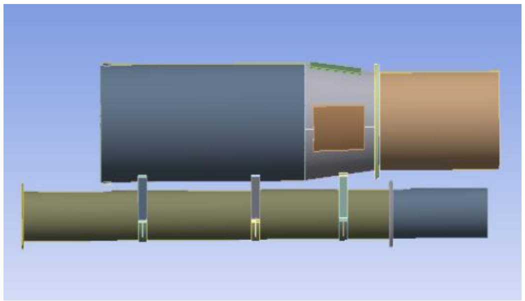


Fig. 1. Geometry of cutter unit, elevation

c) Mesh Modelling

The Hex Dominant method in ANSYS Workbench is adopted to generate the mesh. The mesh element is sized from 3 to 50 mm in order to get fine mesh.

The offshore drilling structure assemblies' whole mesh, utilised to determine stresses at every component, is represented in Figure 2. For the cutter unit assembly and its parts, mesh sizes are typically used between 3mm and 80mm.

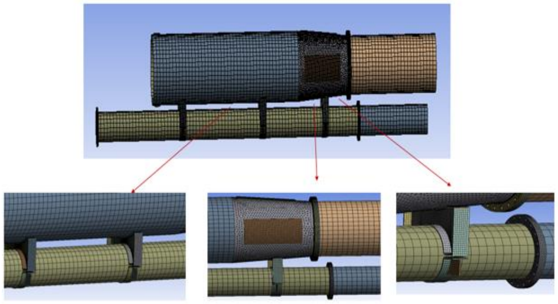


Fig. 2. Meshing of cutter unit – detailed view.

B) Material Properties

Material properties of all used materials are given in Table 1. The cutter unit assembly is fabricated by utilizing IS 2062 grade B steel.

Table 1. Material properties

Part	Material	Young's modulus (MPa)	Poisson's ratio	Yield strength $R_k=R_p0.2$ (MPa)	UTS R_m (MPa)	Density (kg/m^3)
Cutter Unit	IS 2062 Gr B	2.1e005	0.3	235	360	7850

C. Contact and Boundary Conditions

Bonded contact is used to simulate every welded and bolted connection. This means that if a node is discovered to be in contact, it is confined in all directions normal and tangential to the contacted body. As a result, no separation, penetration, or sliding may take place in the event of discovered contact.

a) Fixed support

The cutter unit's centre of gravity is 1.5 metres from the edge of the pipe, and it is attached to the pipe surface by a rigid connection at a remote position, as specified in Figure 3 (a). In order to determine the precise weight of the cutter ladder assembly, which is seen in figure 3 (b), the centre of gravity of the cutter ladder unit must be related to its remote location by a rigid equation.

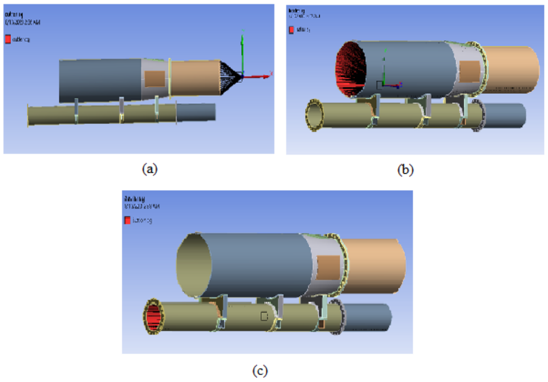

(a) (b) (c)

Fig. 3. (a) Remote point for cutter CoG (b) Ladder structure CoG (c) Suction pipe CoG

To accurately capture the weight of the suction pipe and slurry, as shown in figure 3 (c), the centre of gravity of the pipe must be coupled to its remote location through a rigid equation.

The combination of remote point location for the centre of gravity of following components are illustrated in figure 4.

1. The Cutter unit remote location.
2. The cutter ladders structural remote location.
3. The suction pipe remote location.

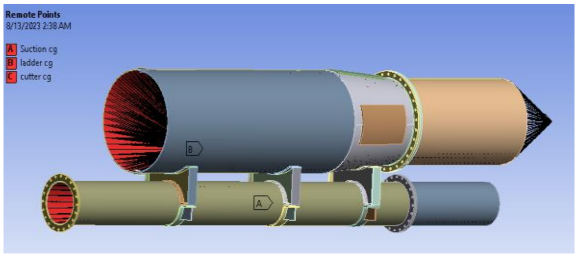


Fig. 4. Combined remote point in cutter ladder structure

b) Gravity load

All calculations include gravity load:

$F_{Grav} = g_M \cdot m \cdot g$

Where, m denotes the mass and g specifies the gravitational constant

$g_M = 1.35$ safety factor for gravity.

Gravitational constant is applied as acceleration by defining

$a = g_M \cdot g$

$a = 1.35 \cdot 9.81\ m/s^2 = 13243.5\ mm/s^2$.

This is important because changing the value of g to include the factor of 1.35 would violate the preset earth acceleration. Figure 5 displays the acceleration.

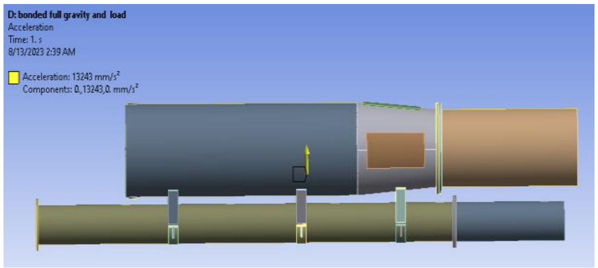


Fig. 5. Acceleration applied in cutter ladder structure.

c) Fixed support

Figure 6 illustrates how the cutter ladder pipe face is restrained from all degrees of freedom by a fixed support.

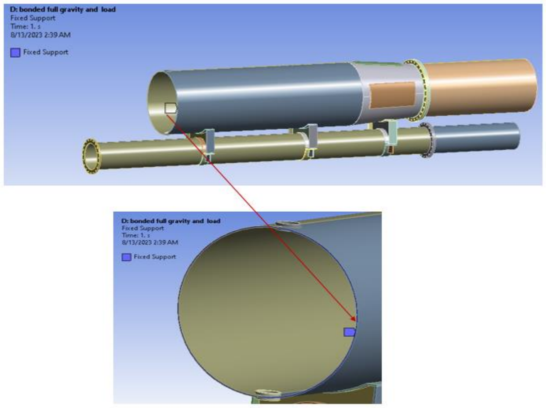


Fig. 6. Fixed support on pipe face.

d) Lifting load

The lifting load is applied 7.5 tons with a partial safety factor of 2.

$F_{cift} = F_{lift} * \gamma_F$

$F_{lift} = 7.5$ ton $* 2 = 15$ ton.

Safety factor γ_F is set to 2 according to (2).

The 100kN mass is applied on the Cutter unit remote location, which is shown in Figure 6 (a). 5kN mass is applied on the suction pipe remote location, which is shown in Figure 6 (b). 150kN mass is applied on the cutter ladders structural remote location, which is shown in Figure 6 (c).

Table 2. Load combination applied on cutter ladder structure

Load Case No.	Load Case Name	load (N)	Gravity (mm/s^2)
1	Cutter load	100000	13243.5
2	Cutter ladder mass	150000	13243.5
3	Suction pipe mass	25000	13243.5

4	Center winch	130000	13243.5
5	Left side winch	130000	13243.5
6	Right side winch	130000	13243.5

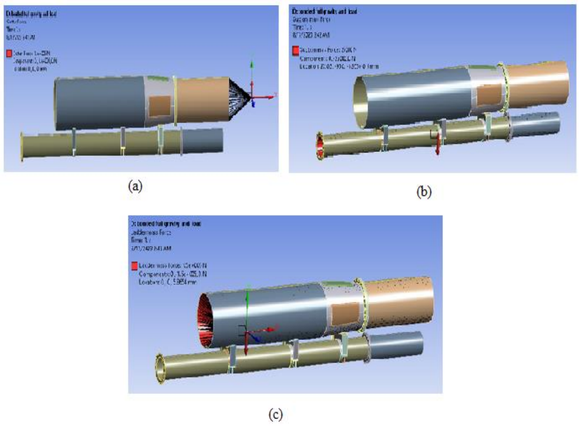


Fig. 6. Remote load applied on the (a) Cutter unit (b) suction pipe and (c) Cutter ladder structure

e) Load pattern

Calculations are performed with ANSYS workbench in a same way all the load cases were summarized in one calculation run. Gravity load is applied in all load cases as specifies in Table 3.

Ladder winch lifting load is applied 130kN as illustrated in figure 7 (a). Left side winch load is applied 130kN as shown in figure 7 (b). Right side winch load is applied 130kN as shown in figure 7 (c). All load combinations are shown in figure 7 (d).

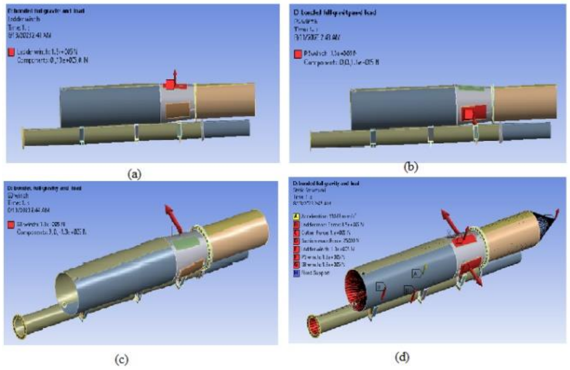


Fig. 7. Load applied on the cutter ladder structure (a) Ladder winch lifting (b) Left side winch (c) Right side winch and (d) All load combination.

3. RESULTS AND DISCUSSION

The strength analysis of an offshore drilling structure with a lifting load of 15 tonnes and a partial safety factor of 2 is explored. Finite Element (FE) software ANSYS is used to calculate the loads based on the loading application.

Deformation plots are examined to confirm that the direction of cutter ladder drilling unit deformation is compatible with applied loading in order to demonstrate the plausibility of results. As an illustration, the following image illustrates directional deformation in the y-direction for applied load, which is consistent with applied loading. Thus, the FE model's plausibility is established.

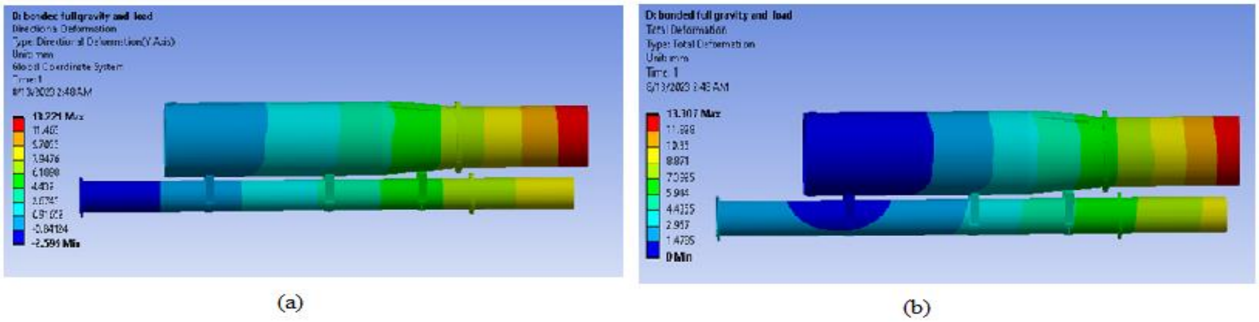
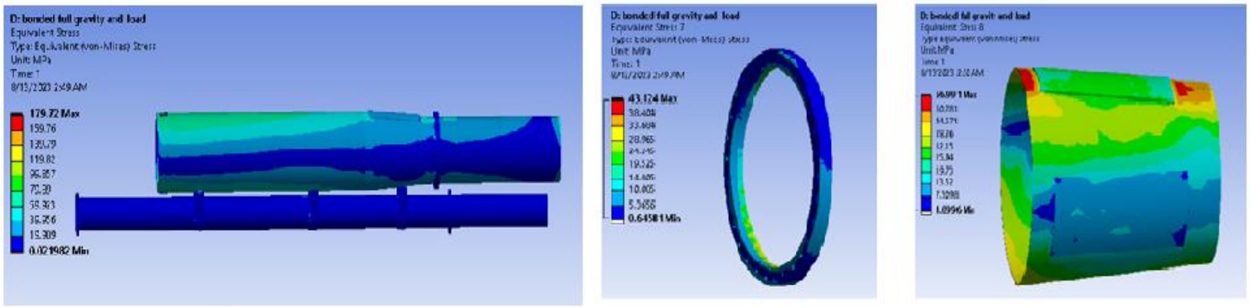


Fig. 8 (a) Direction deformation (mm) of cutter ladder structure in y-direction and (b) Total deformation (mm) of cutter ladder assembly

The resulting stresses of cutter ladder unit are shown in figure 9. The stress is below the allowed limit of 235 MPa in cutter ladder structure, which shows the plots of von mises stresses of cutter ladder structure assembly for applied load (linear elastic material behaviour). Deformation scaling is set to 1.0 for all load cases.



(a) (b)

Fig. 9. Von mises stress plot of cutter (a) ladder structural assembly and (b) ladder flange and reducer

5. CONCLUSION

In this work, the strength analysis of a drilling structure used offshore with a lifting load of 15 tonnes and a partial safety factor of 2 is investigated. The loads are calculated using the Finite Element (FE) programme ANSYS based on the loading application. As a result, deformations at 13.2 mm, which is less than the allowed maximum of 35 mm, are obtained. It has been established that the offshore drilling structure is significantly more stressed than the maximum permissible limit of 235 MPa. Consequently, the offshore drilling structure satisfies the strength requirements.

REFERENCE

[1] Tang, Liping, Baolin Guo, Marcin Kapitaniak, Vahid Vaziri, and Marian Wiercigroch. "Finite element analysis of drill pipe-slip system." *Journal of Petroleum Science and Engineering* 220 (2023): 111163.

[2] Liao, Maolin, Gaowei Wang, Zhiying Gao, Yipeng Zhao, and Ruifeng Li. "Mathematical modelling and dynamic analysis of an offshore drilling riser." *Shock and Vibration* 2020 (2020): 1-13.

[3] Hajinezhadian, Mehdi, and Behrouz Behnam. "A probabilistic approach to lifetime design of offshore platforms." *Scientific Reports* 13, no. 1 (2023): 7101.

[4] Adamiec-Wójcik, Iwona, Lucyna Brzozowska, Łukasz Drąg, and Stanisław Wojciech. "Rigid finite element method in applications to dynamic optimization of motion of a riser in reentry." *Marine Structures* 78 (2021): 103006.

[5] Dandash, Alaa, WenSheng Xiao, and HuaLin Liao. "Unconstrained Dynamic Simulation on Offshore Dual Derrick." *International Journal for Engineering Modelling* 35, no. 2 Regular Issue (2022): 32-42.

[6] Dandash, Alaa, WenSheng Xiao, and HuaLin Liao. "Analysis of Offshore Dual Derrick Motion: The Impact of Accurate Modelling." In *Offshore Technology Conference Asia*. OnePetro, 2020.

[7] Dandash, Alaa, WenSheng Xiao, and HuaLin Liao. "Unconstrained Dynamic Simulation on Offshore Dual Derrick." *International Journal for Engineering Modelling* 35, no. 2 Regular Issue (2022): 32-42.

[8] Cheng, Po, Yong Liu, Yu Ping Li, and Jiang Tao Yi. "A large deformation finite element analysis of uplift behaviour for helical anchor in spatially variable clay." *Computers and Geotechnics* 141 (2022): 104542.

[9] Prabowo, Aditya Rio, Fajar Budi Laksono, and Jung Min Sohn. "Investigation of structural performance subjected to impact loading using finite element approach: case of ship-container collision." *Curved and Layered Structures* 7, no. 1 (2020): 17-28.

[10] Liao, Maolin, Gaowei Wang, Mu Li, Qing Zhao, and Zhiying Gao. "Dynamics of an offshore drilling tube system with pipe-in-pipe structure based on drift element model." *Applied Ocean Research* 118 (2022): 102978.