International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 08, August-2015

Behaviour of Jacket Platform Subjected to Ship Impact

Vaibhav Dahiwalkar*
Structural Engineering Department
V. J. T. I.
Mumbai , India

J. G. Solanki
Structural Engineering Department,
V. J. T. I.
Mumbai , India

***Abstract*- The offshore structure is subjected to the possible collision of supply ships. Therefore, it is necessary to study the behaviour of jacket structure after the collision because it affects the structural integrity. The theoretical method and code provision are presently available but the solution is quite conservative. Therefore, It is necessary to evaluate the results from ANSYS are compared with theoretical formulation based on a rigid plastic mechanism approach. Tubular members are subjected to impact load for the investigation of the collapse behaviour of the jacket platform. Together with the theoretical calculation and FEM program results, study the effect of the localised damage on tubular member has been examined.**

Index Terms*- *Explicit Dynamics, Impact force, Mesh, Deformation

I. INTRODUCTION

In an offshore industries, ship have to serve the offshore operation and during the transportation, due to harsh weather conditions or engine failure or manual error or combination of the all, it may sometime hit the offshore structure i.e. column or braces.

In splash zone, the broadside impact and bow or stern impact are the two critical impact scenarios. In present research work, only the bow impact scenario is considered. The jacket legs and braces in the splash zone shall be designed of such loads to avoid premature failure and collapse of the platform. Supply boat collision is one of the most critical accidents that can affect structural safety and integrity of offshore structure.

II. FEM MODELLING APPROACH

FEM model of the jacket leg segment is modeled from the dimensions provided by Kjetil Qvale (2012) in their master thesis. The column and ship bow is modeled by using ANSYS Geometry modeler and export in Mechanical i.e. ANSYS WM analysis module.

Fig.1 is represent the sketch of the Jacket leg i.e. column and Table 1 show the column cross sectional properties, Length and slope.

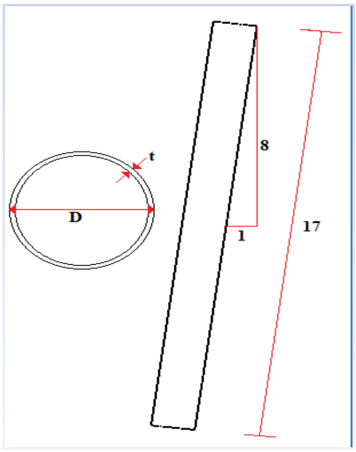

Figure 2:-Column Model

Length	17	m
Diameter	1.5	m
Slope	1/8	
Thickness	60	mm

Table 1:-Column Model Dimensions

The ship model was developed based on available data of bulbous bow. The finished model (Fig. 2) and its dimensions are presented in Table 2.

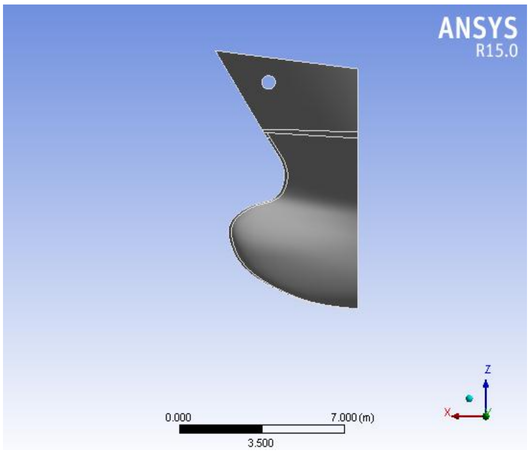

Figure 1:-Bow Model

Length	6.01	m
Width	9.73	m
Height	10.24	m
Thickness	15	mm

Table 1:-Model dimensions (Bow model)

a. Mesh and Elements

The offshore jacket leg i.e. column geometry is simple (hollow cylindrical pipe) and meshed with 4-noded quadrilateral elements with element size is 100 mm. The effective length to thickness ratio of column element is 1.67 which is quite small compared to the previous convergence studies. The advantage of convergence is to achieve a good representation of the circular geometry using 4-noded quadrilateral elements.

It is also recommended from previous research work for Ship model to use 4-noded quadratic shell elements at an element size of 5 to 10 times the plate thickness would give a good representation of the results. Therefore; the 4-noded quadratic shell element mesh size of 150 mm is used for the ship model.

b. Boundary Conditions

The column ends are considered as completely fixed in all translational and rotational degrees of freedoms. In this case the surrounding structure is assumed strong enough to resist the loads that arise when the column deforms. The Displacement of ship is restricted in global Y direction with a travelling speed of 2 m/s.

c. Material Properties

The Jacket leg segment is modeled using two different material models. A power law material with isotropic hardening is used in analyses where the jacket leg is allowed to deform and dissipate compact energy. This plasticity material model is often used in large strain analyses

A bilinear stress-strain curve requires that you input the Yield Strength and Tangent Modulus. The slope of the first segment in the curve is equivalent to the Young's modulus of the material while the slope of the second segment is the tangent modulus.
The material properties of the jacket leg i.e. column member are presented in table 3.

Young's module (E)	200	GPa
Yield strength	250	MPa
Tangent modulus	1450	MPa
Density	7850	kg/m3
Poisson's ratio	0.3	
Bulk Modulus	166.7	MPa
Shear Modulus	0.769	MPa

Table 3 –Material Properties

d. Location of Impact

The location of initial impact may have an effect on the capacity of the tubular member. The impact location is set to middle span (in the middle of the column) for all the analyses in this thesis. The central impact on jacket leg is most critical position in the analysis of beam. Figure 4- illustrates the impact point for jacket platform leg and ship bulbous bow impacts.

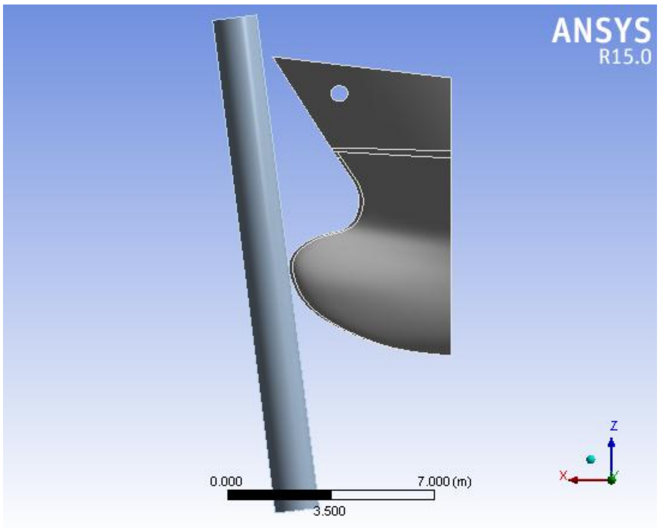

Figure 4:- Middle Span Impact location

III. RESULT AND VALIDATION

a. Finite element analysis results

In present research work, the jacket leg segment is subjected to impact load with combination of lateral load and axial force. To investigate the effect of impact load on jacket leg in which both ends are restrained against rotation and translation. The jacket leg segment with a diameter of 1.5 m and thickness of 60 mm is considered. An explicit dynamics analysis is used to determine the dynamic response of a structure due to impact.

The required force to produce a 0.91 m deformation of the column is 31.16 MN for the case with fixed end boundary condition and axial force. The effect of impact on the ship is quite negligible. The Total energy stored in body due to deformation of tubular member is 14.81MJ.

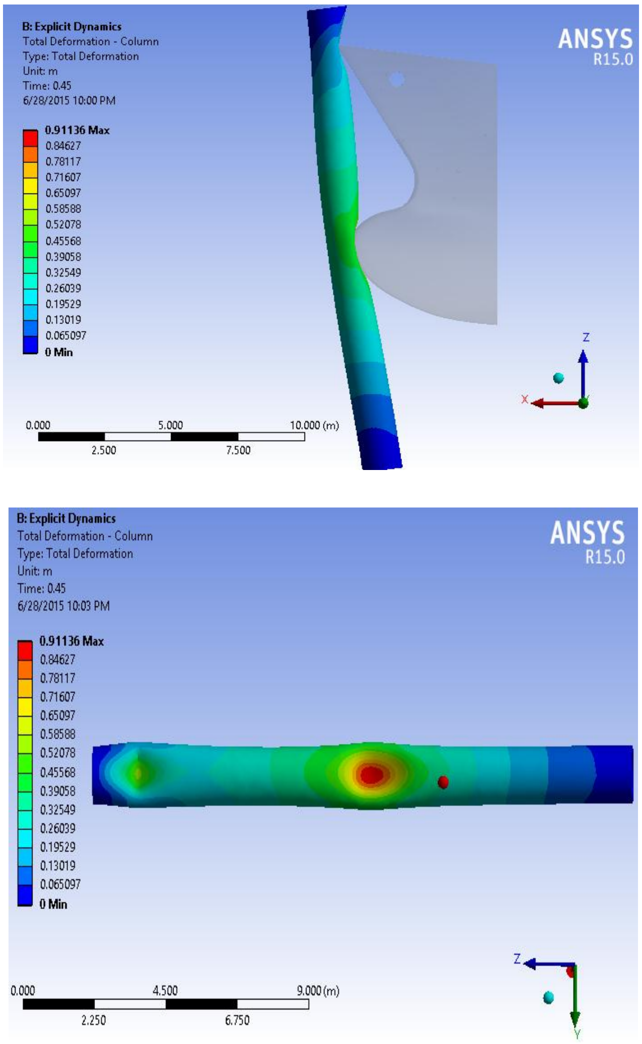

Figure 4:- Total deformation of Column in ANSYS

b. *Validation of Finite element analysis results from theoretical calculations*

The finite element analysis results are validated by using available theoretical method. The Theoretical calculation is simple and based upon the plastic analysis under assumption of point load model.

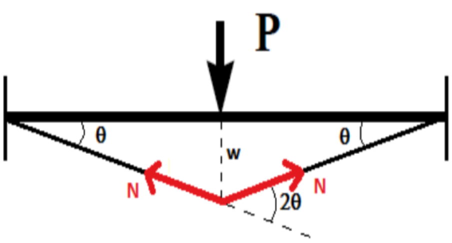

Figure 3:-Three Hinge Collapse Mechanism – Point Load

The comparison in between the theoretical calculation and the FEA results is presented in Table 5. In this section, the FEA results are compared against the ideal rigid plastic behaviour that takes into account the increase in strength due to the development of membrane forces during large column deformations.

The point-load calculation model is quite conservative compared to the FEA results. The assumption of a point load at the middle of the column might be a reasonable assumption in the initial phases of the impact.

Results	Deformation	Impact Force	Column Energy absorption
	m	MN	MJ
FEM results	0.91	31.16	14.81
Theoretical results	0.98	31.80	13.15

Table 2:- Comparison b/w Theo. Calculation and the FEA results

IV. CONCLUSION

The capacity of a platform-leg may be significantly weakened by local indentation. Accurate estimate of the resistance against local indentation are therefore very important. The FEM result are discussed and compared with theoretical calculations.

The results show that the columns with large local indentations have a significant reduction of the plastic collapse capacity compared with the ideal load. The capacity of the strongest columns almost reaches the ideal collapse load. The ideal rigid model is generally conservative for large deformations in the post-collapse range.

REFERENCES

[1] API2A -WSD- Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms—Working Stress Design
[2] NORSOK N- 004 (2004), Design of Steel Structures
[3] Kjetil Hatlestad Qvale (2012), “Analysis and Design of column in Offshore structure subjected to supply vessel Beam Collisions” The Norwegian Institute of Technology, Norway
[4] O. furnes and J. amdahl (1980), “Ship collision with offshore platforms”, The Norwegian Institute of Technology, Norway
[5] T.H. Soreide, T. Moan, J. Amdahl and J. Taby (1982), “Analysis of ship / platform impacts”, The Norwegian Institute of Technology, Norway
[6] C. G. Soares and T. soreide (1984), “Plastic analysis of laterally loaded circular tubes”, ASCE
[7] C. P. Ellinas (1984), “Ultimate strength of damaged tubular bracing members”, ASCE
[8] Jorgen Amdahl (1991), “Consequences of ship collisions”, The Norwegian Institute of Technology, Norway
[9] Jorgen Amdahl (1993), “Ship collision with offshore structures”, The Norwegian Institute of Technology, Norway
[10] M. Zeinoddinia, G.A.R. parkeb and J.E. Hardingb (2001), “Axially pre-loaded steel tubes subjected to lateral impacts: an experimental study”, International Journal of Impact Engineering
[11] W. Jina, J. Songa, S. Gonga, Y. Lub (2004), “Evaluation of damage to offshore platform structures due to collision of large barge”, Engineering Structures, ELSEVIER
[12] M. R. Khedmati and M. Nazari (2010), “A numerical investigation into strength and deformation characteristics of preloaded tubular members under lateral impact loads”, Marine Structures, ELSEVIER
[13] H. Qu, J. Huob, C. Xua and F. Fuc (2013), “Numerical studies on dynamic behaviour of tubular T-joint subjected to impact loading”, International Journal of Impact Engineering, ELSEVIER
[14] J. Travanca, H. Hao (2013), “Numerical analysis of steel tubular member response to ship bow impacts”, International Journal of Impact Engineering, ELSEVIER
[15] L. Li, Z. Hu and Z. Jiang (2013), “Plastic and Elastic Responses of a Jacket Platform Subjected to Ship Impacts”, Hindawi Publishing Corporation ,Mathematical Problems in Engineering
[16] M. Storheim and J. Amdahl (2013), “Design of offshore structures against accidental ship collisions”, Marine Structures, ELSEVIER