# MODELING AND ANALYSIS OF OFFSHORE JACKET PLATFORM

Poonam Mohan[1], K. R. Aswin Sidhaarth[2], V. Sanil Kumar[3]
[1]IIndME.Structural Engineering, Anna University, Regional Centre, India
[2]Assistant professor, Anna University, Regional Centre, Coimbatore, India
[3]Principal Scientist, CSIR-National Institute of Oceanography, Dona Paula, Goa, India

***ABSTRACT***

*This paper details the results obtained from static analysis, considering both operational and ultimate limit state performance characteristics of a jacket platform, designed at Mumbai high basin. The existing platforms at Mumbai high region are affected by continuous sea state and operational state loading. So, it becomes a necessity that the structures design for the purposes should be a reliable one. The Static load analysis done in this study, using the STRUCAD *3D software helps to analyze and predict the performance of a typical jacket structure at Mumbai High, when subjected to various load cases. The pile analysis performed using PILE 19.4 OASYS software gives the ultimate load carrying capacity, which the structure can withstand. The static analysis is also able to replicate the details of observed behavior of the designed structure. Based on the axial tension/compression and the bending stresses obtained through simulation, the tubular members are designed and the code unity check is done manually for two offshore standards i.e. ISO 19902[8], API RP 2A [1], on which the code provision comparison is performed to find more reliable code standards suiting the state of loading. The basic design of deck portion of the structure is also performed so as to obtain the preliminary design factors and member dimensions with respect to sea-state loading, thus the selection of dimensions based on existing jacket platforms in the considered location is also being discussed[3]. The design load level is utilized to design structural elements, while the operational load level is applied to evaluate the capacity of the structure.*

***KEYWORDS:*** *API RP2A, ISO 19902, Jacket Structure, Static Analysis, Wave Loading*

## I. INTRODUCTION

With the recent innovative ideas of analysis using software, it is now easier for the offshore engineers to do simplified and realistic evaluation of the static operational and ultimate limit state characteristics of template or jacket platforms, which are subjected to various environmental conditions. The Mumbai High Basin is in the Arabian Sea (northern region of the Indian Ocean), about 160 km off the Mumbai coast. It is India's biggest hydrocarbon reserves. The field has more than 551 oil wells.

Recently, many researchers have worked on studying the reliability of existing jacket platform by considering various locations [Bea *et al*, 1995; Thomas Gjerde, 2011; Bill Madock, 1992].This is because, these platforms are highly susceptible to wave effects and considering it, being huge structures, stability is highly influencing factor. In the case of pile of these offshore structures, exposed to waves, several researchers have developed and implemented various effective methodologies to control such a huge system and to maintain the stability of the structure one among which is Rupam Mahanta, 2011[11].The solution adopted in such situations is to increase the number of elements and their cross sections, thus improving the stiffness of the structure, which contribute to the structural stability [4].

A typical jacket platform at 60 meter water depth in this basin is analysed, to hold the operational, self-weight, deformation and environmental load acting simultaneously on the structure. The wave force on slender tubular members is calculated by the addition of the effective drag and inertia forces in Morison's equation [5]. For this work, unity check and the basic joint design is also done for critical member of the jacket, thus studying the variation in code standards, such as API RP 2A and ISO 19902. Also, the pile load capacity is checked in software analysis, based on the geotechnical survey report obtained [12]. The seismic effects are not considered as the region considered is seismically inactive zone. This work is emphasized on the effect of load acting on the structure and the design which falls within the standard code recommended practices [2] leading to the generation of final stable offshore jacket platform. The comparison of the results between the manual and the simulation indicates the consistency of design and construction of the similar model and reliability of the manual design standards. Thus, the present study deals with static effects of structure, and balancing the forces acting on it, through simulation using software STRU CAD 3D.

## II. MODEL DATA

The platform considered in the study is a four legged production platform. Water depth at the location is 60 m. The platform is designed based on the API recommended criteria for 50 years return period for a wave height of 10 meters. The cellar and the main deck elevation are +77.5 m and +85 m respectively. The major deck framing is 50 by 50 m in plan and the jacket legs are battered at one to eight in both broad side and end on framing. The deck legs, the jacket column and the pile are of 1.372 m, 1.42 m, 1.524 m OD obtained from the preliminary design. The average wall thickness is 20 mm uniform to the whole structural members. The jacket is sub divided into 5 bays of 10 meter each. The bottom three bays are provided with x-bracings of 61 cm outer diameter both on end on and broad sides. The bracing provided on the fourth bay from the bottom is 50.8 cm and the top most bay is provided with 45.7 cm outer diameter braces. The model is generated using 204 beam elements and 69 plate elements. The jacket bracing and the horizontal framing is of 695 Mpa steel materials with average yield strength of 106 ksi. The members are considered to be cold rolled pipes and small members of rolled pipe sections. The soil data is obtained for driven pile of 1.524 m diameter at Mumbai High region.

## III. LOAD CALCULATIONS

A design approach that minimizes static load effects through judicious selection of the platform geometry is one of the keys to increased capability of platform. Static loads could be reduced through decreasing member sizes and through adjustment of the jacket geometry at the top load action zone, so that force cancelation could be used to advantage. The type of loads considered in the analysis (API RP 2A) [1], includes the following:

- Dead load (G)
- Live load (Q)
- Environmental loads (E) (wind /wave/buoyancy)
- Deformation load (D)

The load combination used for the analysis purpose includes:

- $1\,G + 1\,D = U$
- $1\,G + 1\,E = V$
- $1.3\,G + 1.3\,Q + 1\,E + 1D = X$
- $1G + 1\,Q + 1\,E + 1\,D = Z$

The load calculations are done based on the standard DNV, ISO19902, API RP 2A. The wave load is calculated based on the Morrison’s equation which is shown in equation (1) below:

$$F = 0.5\,\rho\,|u|\,u\,C_d\,D + \rho\,C_m\left(\frac{\pi d^2}{4}\right)\mathring{u} \qquad \ldots\ldots\ldots.. (1)$$

Where,
$C_m$, $C_d$ = hydrodynamic inertia & drag coefficient, $\rho$ = water density, D = pile diameter, u = water particle velocity, $\mathring{u}$ = water particle acceleration.
The wind load is calculated for the critical wind direction for the whole topside module using the following API recommended formula:
Mumbai basic wind speed (zone III) $V_b$ = 50.6 m/s (API)
Observed critical angle at Mumbai High = 34° & 56°

$$F = q\,(h/10)^{0.22}\,c.A.\mathrm{Sin}\,\alpha \qquad (2)$$

$$Q = 0.613\, V_b^2\, (h/10)^{0.22} \qquad (3)$$

*C* = shape coefficient, h =datum height, A= exposed area, *q* = basic wind pressure or suction, α = angle between the direction of the wind and the axis of the exposed member, $V_b$ = basic wind speed (m/s)

Considering the above equations, the calculation of the wave forces in each member is generated both manually and theoretically. The lateral forces are resolved and the total axial force in the structure is found out, which is later compared with the result obtained through analysis. The obtained value is checked and compared to determine the stability of the structure.

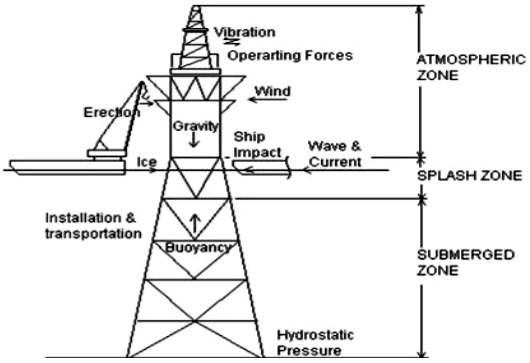


**Fig 1:** The loads on the structure (Chakrabarti, 2005)

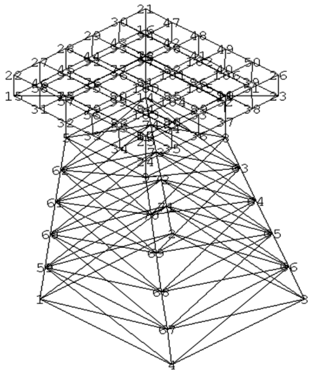


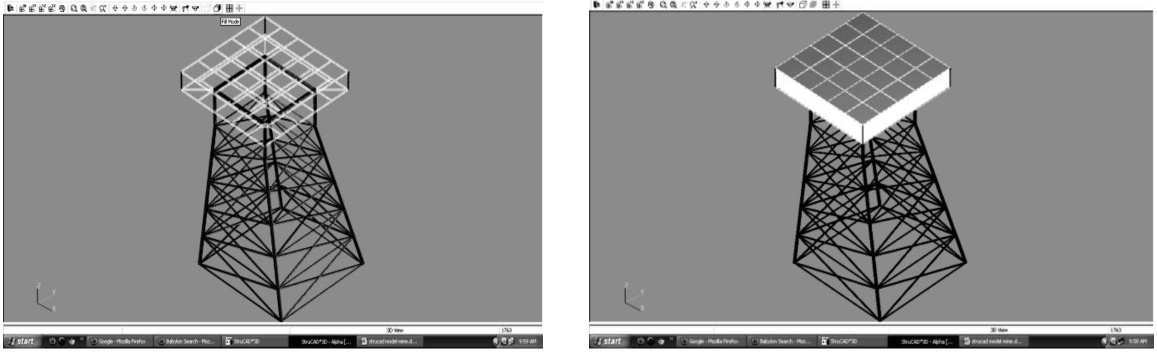
**Fig 2:** Joint Number Provided in STRUCAD 3D model

Fig.1 illustrates all kinds of loads acting on the offshore template structure. In the present work the wave, wind, operational, buoyancy and dead load are of main interest and considered in the analysis.

## IV. STRU CAD *3D MODELING & ANALYSIS

StruCAD*3D is a general purpose oriented interactive, graphics-assigned system for constructing, analyzing and designing three-dimensional structural engineering models where elements definition is done by standard user libraries supplied for the section types supported by the AISC[1]. The program is composed of several modules (Alpha, Beta, Omega, Noah, Stru-Plots) which translate the state-of-the-art of research into the state-of-practice of design, thus reducing the complexities, uncertainties in solving problems.

The Alpha module helps develop all the geometric information and the load data (Joint, Member, Plates and Load statements). The Beta module is the counterpart of the Alpha module in generating the input data for structural analysis and it acts as a spreadsheet-style interface, the best way to describe the non-geometric tabular data. The Noah module is a structural analysis program for basic analysis and it is comprised of the **Scprep , Scload , Scsolve ,** and **Scstress** programs which is used for the purpose of model Preparation, Load generation, Solve and calculating the Stresses respectively.



**(a)** **(b)**

**Fig 3(a, b):** Jacket platform model generated in STRUCAD

The analysis focuses on response of jacket structure to environmental load acting throughout the structure. The jacket is modeled as a frame in three dimensional spaces. The estimates of the environmental parametric values are based on data obtained from related literature reviews [12, 9]. The response of the structure under various loading combinations is obtained by performing the analysis. The hydrodynamic loading is generated using the Hydrostatic collapse option card. The STRU CAD 3D models are subjected to repeated full cycle of wave forces at consecutive time periods [5]. The distributed wave forces are calculated as equivalent point loads acting in the joints, by using the theory of equilibrium. For each time period the load is multiplied with deflection to obtain the generalized moment acting on the structure. Static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads. When loads are applied to a body, the body deforms and the effect of loads is transmitted throughout the body. The external loads induce internal forces and reactions to render the body into a state of equilibrium. All loads are applied at full magnitudes. On applying the full magnitudes, loads remain constant (time-invariant). Time-variant loads induce considerable inertial or damping forces, leading to the static analysis. Here we used static analysis to calculate the structural response of bodies affected with constant velocities or travelling with constant accelerations since the generated loads do not change with time. The relationship between loads and induced responses is linear that is stress is directly proportional to strain and the induced displacements are small enough to ignore the change in stiffness caused by loading. The boundary conditions do not vary during the application of loads which are constant in magnitude, direction, and distribution while the model is deforming. Stiffness of the jacket structure below the deck is accounted for by use of linear beam elements. The deck is modeled by increasing material density, mainly in the deck plating.

The Airy's Wave theory [5] is used and the member loads is calculated based on API RP 2A guidelines. The wind profile description card is used to generate the wind profile and the wind load is calculated for the critical angle at the Mumbai high region [9] i.e. 56 and 34 degree from global axis. The basic sectional properties if provided, the structure will calculate the self-weight. Jacket is modeled using beam element. The deck portion of the structure is provided as plate element, and the related load acting on the structure, is provided as plate pressure, which is uniformly distributed to the whole structure. On providing the member properties the structure is rendered to, as the model shown in Fig. 3(a, b). The element is modeled in accordance with the structural drawings made. Each beam element is having 2 nodes and six degrees of freedom. To maximize the load on pile, the conductors are considered to be horizontal components of environmental loading.

## V. METHODOLOGY

In the work done by Bill Madock (1992) [3], he considered the similar procedure adopted in this work, to check the stability of the structure. In the present work, a change in the type of bracing and the structural dimensions obtained are varied, together with varied environmental conditions. Only the basic static analysis is performed and seismic loading is not taken into account. The method by which the project is progressed begins with the collection of the environmental and field data's of Mumbai high. The environmental load is calculated i.e. the wind and the wave, in the above explained manner. The next step involves the calculation of the preliminary dimensions based on the soil data obtained from the geotechnical investigation for 1.525 m outer diameter piles. This is followed by calculating the approximate UDL acting on the structure and the determination of the assumed self-weight. Based on the references made from the earlier performed research work of Bill Madock, 1992[3] the approximate load acting on the platform is finalized.

The adopted column diameter should be less than or equal to the pile outer diameter. After fixing the preliminary pile dimensions and checking for safety, the structure is modeled in the StruCAD 3D software. The deck height from mean sea level is calculated based on wave height, tide and storm surge and is 16.54 m. The concerned input is provided i.e. member dimensions, group identities, material properties etc. The software automatically calculates the total dead weight of the structure. Then the environmental wave load and the wind load are generated using the hydrodynamic card and the wind profile and wave function cards. After forming the structural model, the static push over analysis/linear analysis is done. The obtained result is checked to find out the critically loaded member, for which the member is replaced or redesigned. Now, on having all the parameters to design the joint and connections of the jacket portion, the design is performed for the critically loaded member i.e. member 61.The connection design for tube to tube and tube to I beam connection is done. Then, for a considered member and joint, the code unity check for API RP 2A [2] and ISO19902 [8] is studied. The usual analysis practice involves the consideration of the anode weight and the conductor weights. In this particular work the anode weight and conductor weight is taken to be as the horizontal component load [10], so that more stability is attained for the structural components as a whole. This is the basic methodology adopted for the analysis, design and modeling and the results obtained is briefly explained in results and discussion.

## VI. RESULTS AND DISCUSSIONS

### 6.1 Preliminary design (deck beam, girder, pile)

Based on the approximate load assessed, the dead load and live load of the structure is determined. The UDL acting on each deck is obtained as 3900 kN/m on both main deck and the cellar deck. The yield stress of steel, used for the deck beam and girder is 385 kN/m$^2$.To find the deck beam and girder sizes, the deck is considered to be continuous overhanging beam. The maximum moment acting on the deck at two particular conditions is checked for (1) no equipment (2) fully loaded. The maximum deck shear is also calculated for these maximum loading conditions, from which the section modulus is calculated. Thus, W24x176 section is selected as deck beam member [1].

The deck girder size is obtained by performing the three moment equation analysis for positioning of load on the deck at various points. The support reactions at various supports are found out and the designing is done for the support at which maximum combined shear, combined reactions and maximum moment is obtained. Based on AISC ASD -PART 5, the girder size is selected as W24x76 A375 grade steel followed by the selection of the brace dimension to be 10'' 40ST [1]. The jacket is modeled as beam element in accordance with the structural drawings made. Each beam element is having two end nodes and six degrees of freedom. To maximize the load on pile [2], the conductors are considered as horizontal component of environmental load for the pipes with weld stiffener plates.

### 6.2 Stru cad 3D analysis results

The structure is analysed using static analysis. The member failure up to first member, this is due to the loading conditions and thereafter failure occurred due to the excessive displacement due to overturning moments. This is found to be more for load case X. Those failed members where redesigned for the concerned moment in z direction, as it is maximum. The results reported here do not incorporate the dynamic conditions related to the structure but only moments at three directions.

The moment for five basic load cases and four combined load combinations is obtained which is less than the calculated moments, shown in Fig 4.The Table 1 shows, comparison of the load values obtained through manual design and software load generation. The load generated is within the permissible limits [7] as specified in code provisions. The manual design proved to exceed the analysis results. The Table 2 shows the manual design values greater, and the percentage difference shows the variation is design. The axial load shows a comparatively higher difference. This is due to the addition of extra vertical component of wave loading in the software analysis.

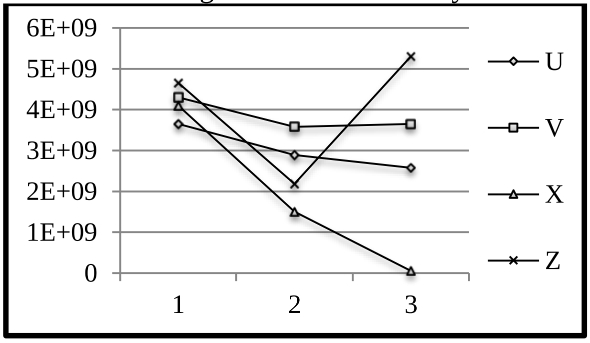


**Fig 4:** Moment (kNm) in X, Y, Z axis in jacket

**Table 5:** Comparison of Dead/Live Load Result

| **Item** | **Weight (tonnes)** | |
|---|---|---|
| | **Manual design** | **Project design** |
| Jacket | 22186.8 | 18000 |
| Buoyancy Effect | 3759.88 | 1573.088 |
| Topside | 35227 | 35227 |

The Top, Middle and Bottom diagonals members where considered to find the critically one, and the axial shear and Bending properties was studied for Operational and extreme environmental state loading of the members.

**Table 2: Comparison of manual and software analysis**

| Item | Force (kN, kNm) | | |
|---|---|---|---|
| | Project value | Manual design value | Percentage difference |
| Max overturning moment | 1145583.8 | 1146732.2 | 0.1 |
| Max base shear | 15575 | 16253.42 | 4 |
| Max axial load | 15842.25 | 13593.4 | 14.19 |
| Axial force(base) | 130015.9 | 129817.28 | 0.153 |

### 6.3 Pile 19.4 Oasys analysis result

Pile 19.4 is a program (http://www.oasys-software.com/download?pid=11), which is used to calculate the vertical load carrying capacity in this present work. The individual pile performance characteristic, in a layered soil deposit, is studied. The theory is based on conventional and new methods for undrained (frictional) test. The pile is of circular cross-section. The soil layers and P-Y curve is provided as input apart from the pile dimensions [12]. The obtained ultimate capacities and cumulative skin friction of pile in software are greater than the calculated values (Fig 5, 6) so the pile seemed to be stable and reliable and maximum load acts at corner pile. Skin friction is the main component of ultimate axial capacity generally in the range of 60%-90% of the total capacity. The Fig 8 shows the variation in bending moment at various depths. It shows that maximum bending moment occurs at the pile head and the piles have quite high amount of rotational restraint at the pile heads.

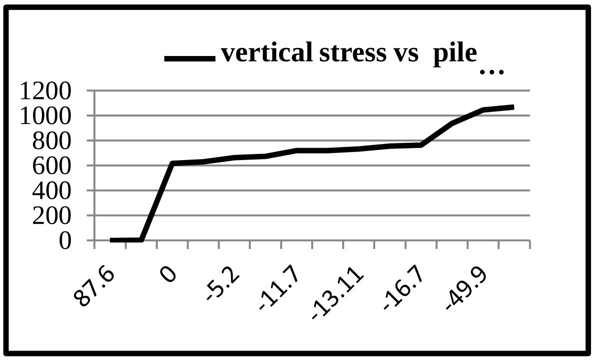


**Fig 5:** Vertical stress at corresponding pile depth

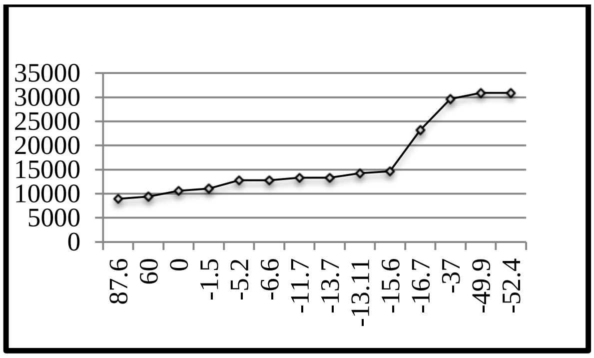


**Fig 6:** Cumulative Skin Friction (kN/m) Vs Depths

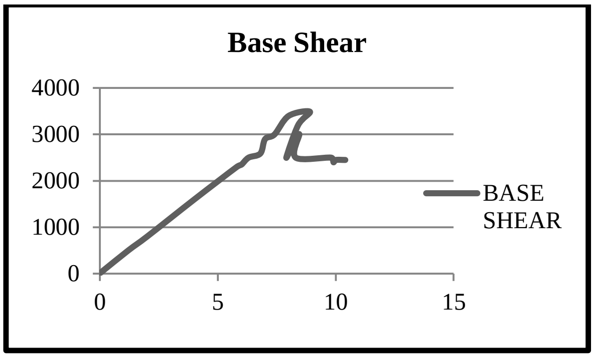


**Fig 7:** The base shear characteristic of static pile

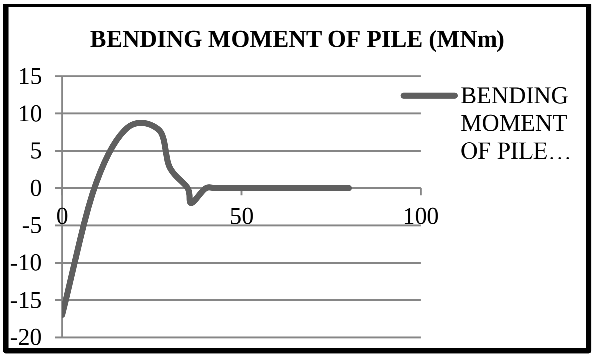


**Figure 8:** Bending Moment of Pile Vs Depth

The software showed that the pile is capable of resisting about15575 kN of loading and the calculated value was 16253.4 KN which is higher so analysis structure seemed to be stable. The probability of failure is mostly on the broadside bay due to buckling of compression bases. The Fig 7 shows base shear in kips (y- axis) with respect to displacement in inches (x axis).

### 6.4 Tubular joint design result

A systematic study of stresses in tubular x braces is done manually, which covers axial loading, in plane bending and out of plane bending etc. From analysis, one of the structural member on a complex multi-planar leg joint, appeared to have a high utilization of stress when assessed using API RP2A WSD (API, 1996). Therefore, a comparative study of the multi-planar joint has been carried out by considering API RP 2A and ISO19902 to determine both the absolute load capacity of the joint, the effect of the out of plane loads and braces.

The joint connecting the main column and the member 61 is chosen for joint design. The joints which are axially loaded are subjected to tensile forces. The tubular joint design is performed manually to see the joints reliability. The joint 5 is considered for which the point where the maximum wave load acts. The strength factor varies with various kinds of joints and load types [4]. The axial load varied from 5 to 15 kN. The obtained strength factor for considered joint is 0.94 and 0.5058 respectively. The chord load factor and yield axial capacity of chord is satisfactory. The brace axial load and bending capacity are within the allowable limits.

### 6.5 Code unity check result

The sensitivity of the jacket structure to variation in design parameters is investigated. The axial tension will lead to column buckling, local buckling, bending and shear effects. The column buckling obtained is i.e. kL/r value is 34.892, found within the allowable axial compressive stress. The parameters like axial tension, elastic and inelastic buckling, bending, beam shear, Torsional shear, hydrostatic pressure and hoop buckling is checked and the values are compared for the APIRP2A and ISO19902 .The beam shear, Torsional shear and hoop buckling values showed variation which is due to the utilization of factored values [11].

## VII. FURTHER WORK

Only the basic strength check is being performed through static analysis. Hence to ensure further stability the dynamic analysis should be performed which involves the construction, transportation, installation and fatigue studies are not done. For this separate analysis is to be performed further. A total stable structure can be obtained only by in depth analysis at each phase from the point at which the project commences. Topside facility of deck structure such as accommodation, process module etc. are not checked, instead the overall dead load is considered. The load such as those resulting from future changes in utility of the structure may be incorporated in further work.

## VIII. CONCLUSION

The static responses of a frame jacket type offshore platform in Mumbai High have been studied using a frame model in computer program STRU CAD 3D. This structure is capable of accounting for static load and buckling behavior of tubular struts. For linear study of braced frames (especially for the jacket type offshore structures) behavior, this methodology can be used. This structure predicts the overall behavior accurately. From the results of the presented study in this paper, the following conclusions can be drawn:

- The general tendency of the value of the vertical stress (Fig 5) is to be directly proportional to the depth for the considered wave period and the platform time period. The static effect of waves with the considered periods is significant since the probability of occurrence of such wave period waves is higher than longer period waves. Therefore, the study of the static behavior of the structure under such waves is of significant importance under static stresses which is less but harmful through constant action than the stresses generated from longer period waves.
- Comparing the Base Shear values obtained from the approximate equation and those obtained from the analysis, it was noted that the approximate equation overestimates the values by about 4% for the considered model (Table 2). This can be reduced by the inclusion on the dynamic effects on the structure, which is suggested in further study.
- Calculation of the moment acting on the structure for different load cases as found in Fig 3 showed that, by selecting the maximum moment producing load combination (z) gives maximum structural stability.
- The use of the analysis gave values of axial load, which is found to give a better representation for the structure model, since it was proven that the value lies at 14.19 % higher than the calculated results (Table 2), which can reduce the existence of stable structure. This provides more resistance to the structure against lateral forces, since laterally acting forces when resolved will lead to higher overturning moments on the structure.
- The existence of marine growth that can be accumulated with time has a significant effect on increasing stress check unity, especially for members at splash zone in case of jackets that has simple configurations

Thus the validity of this new approach is demonstrated by comparing the STRUCAD 3D analysis results with the manual calculations performed based on the parametric equations in standard offshore recommended practices

## ACKNOWLEDGMENT

The author would like to thank for the support provided by CSIR-National Institute of Oceanography, Goa, for successful completion of this work.

## REFERENCE

[1] American Institute of Steel Construction [AISC], (www.zentech-usa.com) Inc., Chicago, Illinois, USA.
[2] American Petroleum Institute [API], 1996. (RP-2A 20th edition, Recommended Practice for Planning, Designing and Constructing Fixed Offshore Platforms—Working Stress Design dated December 1996.
[3] Bill Madock,1992;"Verification of CSA code for fixed offshore steel structures"
[4] Bea et al ,1995;"Verification of simplified method to evaluate the capacities of Template Type Platforms",27th annual OTC, Houston, Texas, USA
[5] Chakrabarti.S.K, 2005, Hand book of offshore engineering, Elsevier.
[6] Dawson.T.H, 1983 ; "Offshore Structural Engineering", Elsevier.
[7] DNV (os-c201 &os-c101), Practice on the joint and connection design of jacket platforms, Environmental Conditions and Environmental load.
[8] ISO 19902, 2007; Petroleum and natural gas industries - Fixed steel offshore structures. - 1st edition.
[9] Rupam Mahanta, 2011; "Pile Design for Fixed Platform for Hydrocarbon Exploration in Indian Offshore", Proceeding of Indian geotechnical conference, Kochi.
[10] Sadeghi K, 2001; "Coasts, Ports and Offshore Structures Engineering"
[11] Thomas Gjerde, 2011;"Structural Analysis of Offshore Module"

[12] Soil Geotechnical Report, 2010; ONGC, Mumbai, www.lntenc.com.

## AUTHORS BIBLIOGRAPHY:

**Poonam Mohan** is structural engineering student at Anna University, Regional Centre, Coimbatore, TamilNadu. She completed her B-Tech from The University of Kerala. This work was performed under dissertation program, as part of academics for the fulfillment of M.E Structural engineering at CSIR-National institute of oceanography-Goa.