ТЕХНИКАЛЫҚ ҒЫЛЫМДАР

IRSTI 50.09

Y. Amirgaliyev[1], D. Khassanov[1], M. Meraliyev[1]
[1]Suleyman Demirel University, Almaty, Kazakhstan

3D MODELING OF THE OFFSHORE WIND TURBINES INTEGRATED STRUCTURE FOR KAZAKHSTAN REGION

Abstract. In this scientific paper we examine meteorological characteristics of the field "Kashagan" located in the north Caspian Sea, the types of offshore oil and gas constructers, offshore wind turbine structure integrated for the middle depths of the Caspian shelf, and the load acting on them during operation on a shelf.

Dimensional solid 3D model and 3D animation presentation of offshore wind turbine is built in the program Autodesk Maya 2012. Created the design scheme of supporting jacket for offshore wind turbine. Jacket designed for the field in the middle depths of the Caspian Sea. Carried out joint account of supporting columns and turbine structure on the static load—own weight of construction and the wind turbine weight of the structure, and the wind load by finite-elements. Also the calculations studied the stress-strain state of the structure.

The finite element method are mastered and calculated by using a software package Autodesk Inventor Professional 2014.

The calculation results can be used in the design of offshore wind turbine structures for oil and gas platforms. Also, the thesis presents the basic calculation of the estimated cost of construction and installation works supporting truss design and offshore wind turbine, its payback period. Sections health and the environment are considered potential risks to personnel and possible threats to the environment, and provide measures for their prevention and reduction.

Key words: Autodesk Maya, Autodesk Inventor, 3D Modelling, AutoCAD, Wind Turbines.

***

Аңдатпа. Бұл ғылыми статья оффшорлық жел турбинасы құрылымын жобалауды талқылайды. Бұл жұмыстың мақсаты оффшорлық жел турбинасы конструкциясын негіздеуде дамыту және дизайн болып табылады. Бұл мақсат жұмыстың ғылыми жаңалығы болып табылады. Деформация, қысым, бұралу жүктерінің барлық түрлері бойынша есептеулер жүзеге асырылыды.

Бұл ғылыми жобада Каспий теңізінде орналасқан “Қашаған” кен орнының гидрометеорологиялық сипаттамасы, теңіз мұнай-газ құрылымдарының түрлері және теңіз мұнай-газ платформаларын жел турбиналары арқылы қамтамасыз ету қарастырылған.

Көрнектік үшін Autodesk Maya программасында жел турбинасының кеңістіктегі қатты денелі 3D моделі құрастырылды және анимациялық видео жасалынды. Берілген тапсырмаға сай Autodesk Inventor Professional 2014 программасында Солтүстік Каспий теңізінде орналасқан “Қашаған” кен орнына арнайы жоба бойынша мұнай платформасын энергиямен қамтамасыз ету үшін жел турбинасының тірек фермалық бағанының есептік схемасы салынып, оған тұйықталған элементтер әдісін қолдануы арқылы өз салмағына, фермалық бағананың үстіңгі бөлігінен түсетін салмағына, циклондық ағын және мұз күштеріне байланысты есептеулер жүргізілген. Inventor 2014 жүйесінде жел турбинасының тіректі фермалық бағаны, іргетасы мен топырақ негізі кешенді түрде есептелген. Есептеулер нәтижелері бойынша құрылымның кернеулік-деформацияланған күйі зерттелген.

Кілт сөздер: Autodesk Maya, Autodesk Inventor, 3D Modelling, AutoCAD, Жел Турбиналары.

***

Аннотация. В данной научной статье рассмотрены гидрометеорологические характеристики месторождения “Кашаган”, расположенное на севере Каспийского моря, типы морских нефтегазовых сооружений, морская ветровая турбина для Каспийского шельфа в средних глубинах и нагрузки, действующие на них во время эксплуатации на шельфе.

Построена пространственная твердотельная модель и создана анимационная презентация ветровой турбины в программе Autodesk Maya 2012. Создана расчетная схема опорной фермовой колонны ветровой турбины, проектируемой для месторождения “Кашаган” в средних глубинах Каспийского моря. Проведен совместный расчет опорной колонн и надводной части строения на статические нагрузки – собственный вес сооружения и вес надводной части турбины, и ветровой нагрузки методом конечных элементов. По результатам расчетов изучено напряженно-деформированное состояние конструкции. Освоена методика расчета методом конечных элементов с использованием программного комплекса Autodesk Inventor Professional 2014. Результаты расчета могут быть использованы при проектировании морских ветровых турбин.

Также в работе приводится расчет базовой сметной стоимости строительно-монтажных работ опорной фермовой колонны и морской ветровой турбины и срока его окупаемости. В разделах охраны труда и окружающей среды рассматриваются потенциальные риски для

персонала и возможные угрозы для окружающей среды, а также приводятся меры по их предотвращению и сокращению.

Ключевые слова: проектирование, месторождение, 3D - моделирование, ветровые турбины.

1. Justification of the choice of the method of calculation tasks

This problem is solved in a linear formulation. In linear problems there is a direct proportionality between the loads and displacements due to the small displacements, as well as between strains (effort) and deformations in accordance with Hooke's law. Therefore for linear problems the best choices are the principle of superpositionand the independenceof the forces.

In physics a non-linear problem is when there is no direct proportionality between stress and strain. Material of construction is the subject to the law of the non-linear deformation. Deformation law can be symmetric or asymmetric—with a variety of out side resistance to tension and compression. In geometrical nonlinear problems there is no direct proportionality between strains and displacements. In practice, the most widely used is the case of large displacementsat small strains.

This problem cannot be solved by any analytical method. In this regard, it is advisable to use one of the most effective methods for the approximate solution of engineering problems—the finite element method (FEM).

For engineering practice the great importance have approximate methods of calculation, the importance of which is constantly increasing due to the inability to obtain accurate solutions to complex engineering problems.

Suppose that the state of the system is described by a function. Let this function is the only solution of a mathematical problem, formulated on the basis of physical laws. The solution is to find an infinite set of functions, one that satisfies the task. For complex tasks, rather than to look for the desired function of an infinite set of features, the problem is simplified. We consider a family of functions defined by a finite number of parameters. As a rule, among the functions there is no exact solution of the problem. However, the appropriate choice of parameters can try to approximately satisfy the equations of the problem and thereby build an approximate solution. This general approach is not unique to the FEM, but also for other approximate methods. Specific to the finite element method is to construct a family of functions defined by a finite number of parameters[1].

We note several important advantages of FEM

a) The finite element method allows us to construct a convenient scheme of formation of the system of algebraic equations for the nodal values of the unknown function. Approximate approximation solutions with simple polynomial functions and all the necessary operations are performed on the individual data element. Next, the association of elements which leads to the desired system of algebraic equations. This algorithm is a separate element from the transition to their full range is particularly suitable for physically and geometrically complex systems.

b) Every single algebraic equation obtained on the basis of FEM contains a small part of the nodal unknowns of the total number. In other words, many of the coefficients in the equations of the algebraic system is zero, which greatly facilitates the solution.

c) The tasks that describe the functions that satisfy the functional equations, is called the continuum. In contrast, the decision of the so-called discrete tasks accurately defined by a finite number of parameters which satisfy the corresponding system of algebraic equations. FEM, as well as other numerical methods, essentially replacing the continuum approximation to the discrete problem. In FEM the whole procedure of such replacement has a simple physical meaning. This allows you to more fully imagine the whole process of solving the problem, avoid the many possible errors and to properly assess the results.

d) In addition to the continual problems finite element method is used to connect the elements and the formation of algebraic equations for solving discrete problems directly. It extends the scope of application of the method.

Active introduction to the theory and practice of design of computer technology and the latest information technology also contributes to the wide application of approximate methods of solutions, including the FEM.

There are software packages that solve engineering problems using finite elements, such as:

- Autodesk Maya 2012;
- MSC NASTRAN for Windows2006;
- Autodesk Inventor Pro2014;
- SolidWorks2008;
- LIRA 9.6;
- FEM Models etc.

FEM can be solved with a high degree of accuracy virtually any engineering problem. The solution to this problem is also possible boundary element method, however, it is limited to in sufficient provision of commercial programs.

2. Fundamentals of the finite element method

The method of finite elements, substantially lies in its name: the system being studied (rod or continuum) is divided into a large number of individual parts of the final sizes (finite element) having the same physic-mechanical characteristics, as given year. After that exactly or approximately investigate the stress-strain state of each finite element known methods of structural mechanics and the theory of elasticity: strength, movement, or a combination in order to determine, depending on the adopted method of analysis effort or movement, or both of them at the points of connection between the finite elements themselves (nodes). These factors make the unknowns of FEM. To find the unknown and algebraic equations are solved, usually of a very high order.

In practical calculations of structures and facilities engineering is currently the most common version of the finite element method, a method based on the idea of movement, so will restrict the study only this form of the method. The Finite Element Method movements was very much geared to the use of computers, as in the analysis of individual finite elements have to deal with simple geometric objects such as standard fixed on a path. The matrix of the system of algebraic equations is symmetric and positive definite band. Such a system is relatively easy to solve. Using the same example of an embodiment in the form of a mixed finite element method or force does not always lead to a system of equations with symmetric positive definite matrices. For their solutions usually require special more complex algorithms.

When implementing the finite element method in displacements as unknowns take necessary translational movement, and in some finite element models and optional angles nodes.

Approach to strength calculations, based on the finite element method in displacements, is a single rod for both systems, and for continuum: plates, shells, massive bodies. The difference lies only in the applicable basic types of finite elements: core, flat triangular and rectangular, similar to the shell, curved shell and volume.

The core elements can be hinged at the ends, working only in tension and compression, flexural flat and spatial, and general appearance, experiencing all kinds of deformation: tension, compression, bending and shear in two planes, and torsion.

Plane elements can be deformed in the plane (plane problem of elasticity) or plane (the problem of bending the plate).

Flat shell elements combine both types of deformation: in its own plane and out of plane, but do not include the mutual influence of these kinds of deformations. The curved shell elements allow for interaction between the two types of strains, rather describe the geometry of a given system being studied, but in the realization they are more cumbersome.Three-dimensional finite element models have the form of pyramids, prisms, parallelepipeds, or appropriate curved objects. They are commonly used in the calculation of massive bodies: dams, piers, arrays, soils, etc., that is where the bulk of the problem requires a decision by the theory of elasticity.

3. Software systems, Integrated Motion Simulation and Assembly Stress Analysis

Autodesk Inventor, developed by software company Autodesk, is 3D mechanical CAD design software for creating 3D digital prototypes used in the design, visualization and simulation of products.

Autodesk Inventor uses Shape Manager, their proprietary geometric modeling kernel. Autodesk Inventor competes directly with SolidWorks and less directly with Creo Elements/Pro, CATIA, and NX (Unigraphics).

Inventor includes an integrated motion simulation and assembly stress analysis environment. Users can input driving loads, friction characteristics, and dynamic components, then run dynamic simulation tests to see how a product will work under real-world conditions. The simulation tools can help users optimize strength and weight, identify high-stress areas, identify and reduce unwanted vibrations, and size motors and actuators to reduce energy consumption. Finite element analysis (FEA) lets users validate component design by testing how parts perform under loads (using actual load information instead of estimates)[2].

Inventor's Parametric Studies and Optimization technology lets users modify design parameters from within the assembly stress environment and compare various design options, then update the 3D model with the optimized parameters.

4. Functionality, Data Interoperability and Exchange

Layout circuits combine individual parts and assemblies. Users can check the possibility of assembling an object; add new parts and position, as well as to correct the interference between the parts of the project.

Molds and tooling. The program automates key aspects of the design of injection molds under pressure. Users can quickly create and validate the design of forms and then export them to the Autodesk Moldflow.

Information sheet. Special design environment flat products automate many aspects of the work. Users can create parts sweep formed shapes, to form the flanges by 3D-modeling and paste the details of specialized fasteners.

Frame Generator is used to design frames (frames) based on the standard profiles. The frames are created by placing standard steel sections to the frame. Formation of the final conditions is simplified thanks to the standard options for the corner joints and butt joints. Users can create their own profiles and add them to the library.

Cable and pipeline systems. A framework for creating piping helps to design them in such a way as to fit into a complex assembly or confined. It includes a library of standard fittings, pipes and hoses, and provides the creation of assembly drawings, which are updated as changes to the original 3D-model.

Inventor uses specific file formats for parts (IPT), assemblies (IAM), and drawing views (IDW or DWG). Files can be imported or exported in DWG format. Design Web Format (DWF) is Autodesk's preferred 2D/3D data interchange and review format[3].

Inventor includes a Building Information Modeling (BIM) Exchange tool, used to create and publish simplified 3D representations, intelligent connection points, and additional information in native file formats for AutoCAD MEP software. Users can export 3D geometry to AutoCAD Architecture, Revit-based software, and AutoCAD software, and exchange data with industrial design software such as Autodesk Alias Design. Inventor is also used to design mechatronic systems since it is interoperable with electrical software applications such as AutoCAD Electrical and Eplan.

Inventor can exchange data with applications such as CATIA V5, UGS, SolidWorks, and Pro/ENGINEER. Inventor supports direct import and export of CATIA V5, JT 6, JT 7, Parasolid, Granite, UG-NX, SolidWorks, Pro/E, and SAT files. The Inventor Construction Environment provides fault-tolerant import of large STEP and IGES data sets.

5. Creating a three-dimensional model of a jacket of wind turbine

On the PC, Autodesk Inventor Professional 2014 was created a solid model of offshore wind turbine components and constructed the jacket, whichis shown in Figure1.

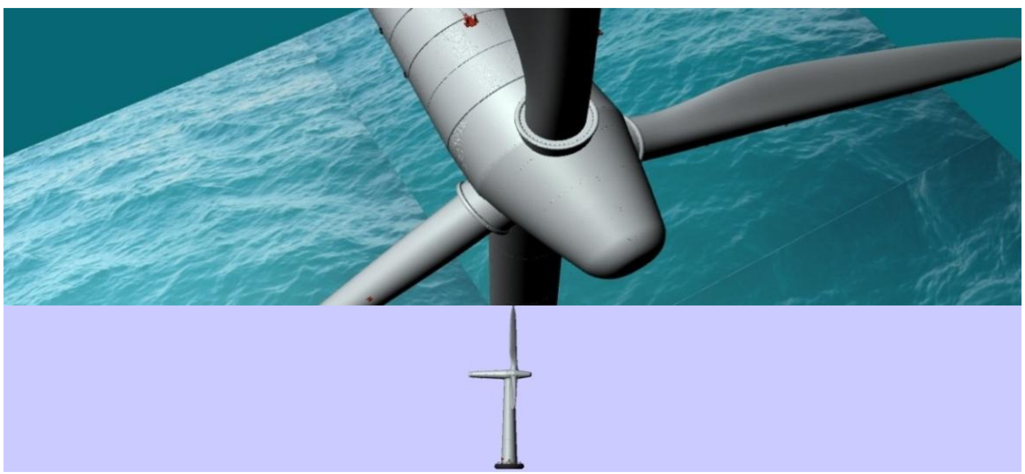

Figure 1- Solid model of wind turbine body

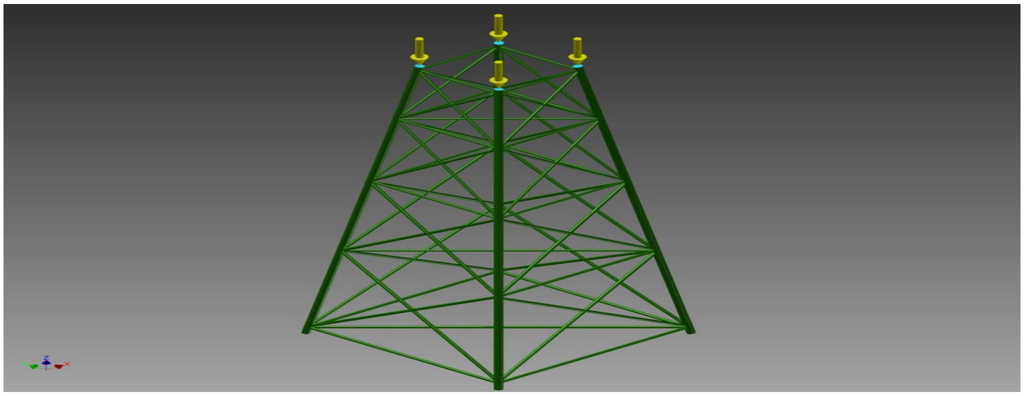

Figure 2 - Solid model of jacket

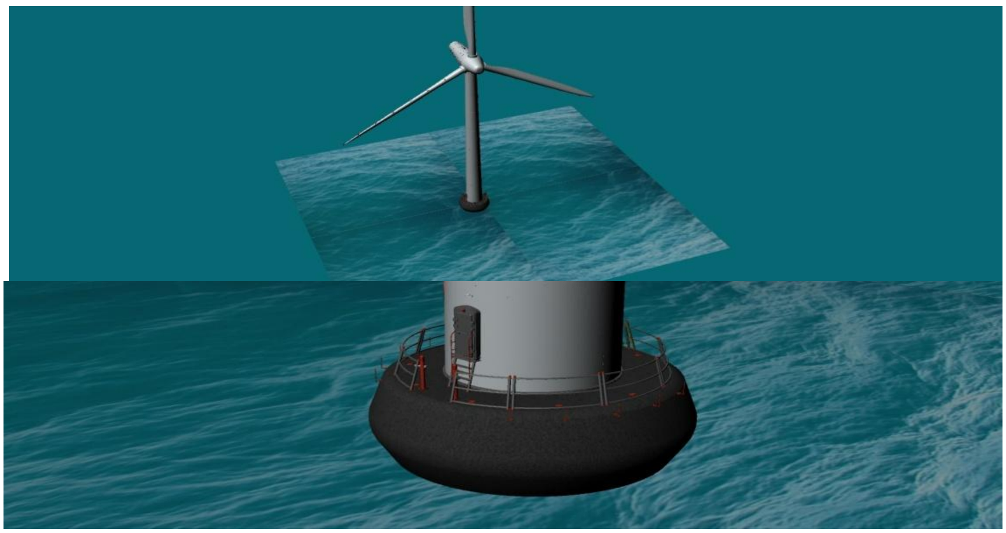

Figure3- Solid model of constructed wind turbine

Table 1- Main data for 3.6 MW wind turbine (without tower)

Power	3.6 MW
Rotororientation	Upwind
Numberofblades	3
Control	Variablespeed, collectivepitch
Turbinediameter	126 m
Ratedrotorspeed	0,20 Hz
Bladepassagefrequency	0,59 Hz
Ratedtipspeed	78 m/s
Bladeweight (each)	18,8 ton
Hubweight	53,6 ton

Rotorweight	110 ton
Nacelleweight	240 ton
Towertopweight	350 ton

Trends in geometric size and rated power capacity of offshore wind turbines shows the main components of an OWT system, including a typical monopile foundation, the substructure, transition piece, tower, rotor blades and nacelle (hub). Modern OWTs are installed with either pitch-regulated blades or variable rotational speed systems in order to allow optimization of power production over a wide range of prevailing wind speeds. The rotational speed of the main rotor shaft is typically between 10 and 20 rpm[4]. The nacelle contains key electromechanical components of the wind turbine, including the gearbox and generator. The gearbox may cause efficiency losses for the wind turbine and is a particular source of noise.

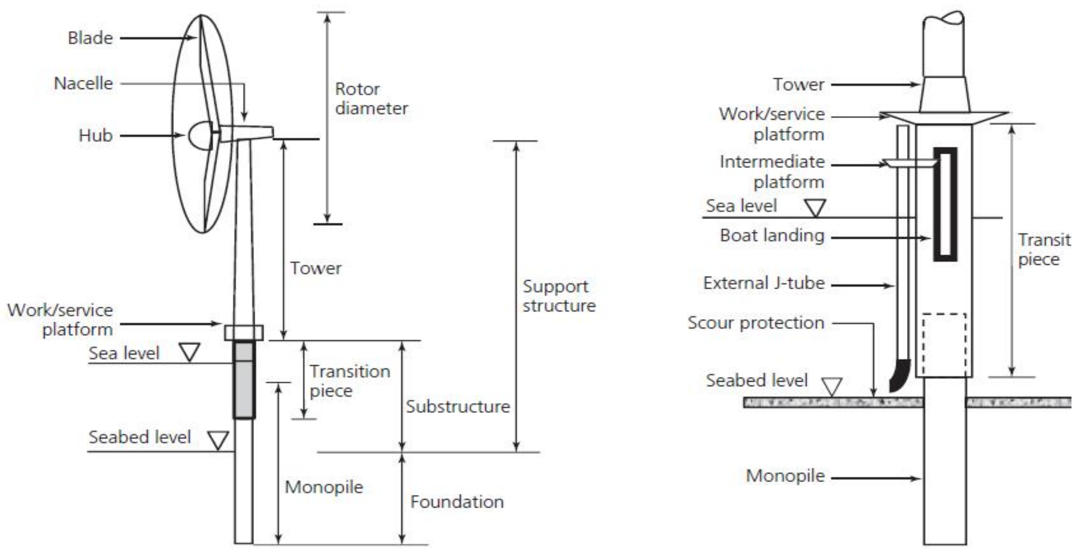

Figure 4 - Major components of OWT system.

Recent developments in the design of permanent magnet generators have made it possible to construct some types of wind turbines without the requirement for a gearbox. In this case, the rotor is connected directly to a low speed multi-pole generator that rotates at the same speed, termed a direct-drive unit. Removing the gearbox removes one of the key components requiring more maintenance and that is prone to failure. The substructure connects the transition piece or tower to the foundation at seabed level. Together the tower, substructure/support structure and foundation maintain the turbine in its correct operational position. The transition piece provides a means of correcting for any vertical misalignment of the foundation that may have occurred during its installation. In some cases, the foundation can extend to above the water surface, thereby also serving as a substructure by connecting directly to the transition piece or tower.

6. The calculation model of supporting of a wind turbine

This manual provides basic conceptual information to help get you started. It provides examples that introduce you to the capabilities of Stress and Modal Analysis in Autodesk Inventor Simulation.

Built on the Autodesk Inventor application, Autodesk Inventor Simulation includes several different modules. The first module included in this manual is Stress Analysis. It provides functionality for Structural Static and Modal analysis of mechanical product designs.

This chapter provides basic information about the stress analysis environment and the workflow processes necessary to analyze loads and constraints placed on a part or assembly.

Calculation of the wind turbine support on the stress analysis is carried out in several stages:

– To create three-dimensional model of an wind turbine;
– To create the calculated scheme of the jacket support;
– To put necessary dependences of the jacket's support. Dependence of the fixings is enclosed on the bottom of the jacket support;
– To set the own weight and distributed load on the deck of an wind turbine on an axis Z;
– In the conclusion we count the jacket bases owing to what we receive results on the SSS.

The calculation model ofsupportingrodcolumns andthe foundation of an wind turbinewas createdin Autodesk Inventor Professional 2012.

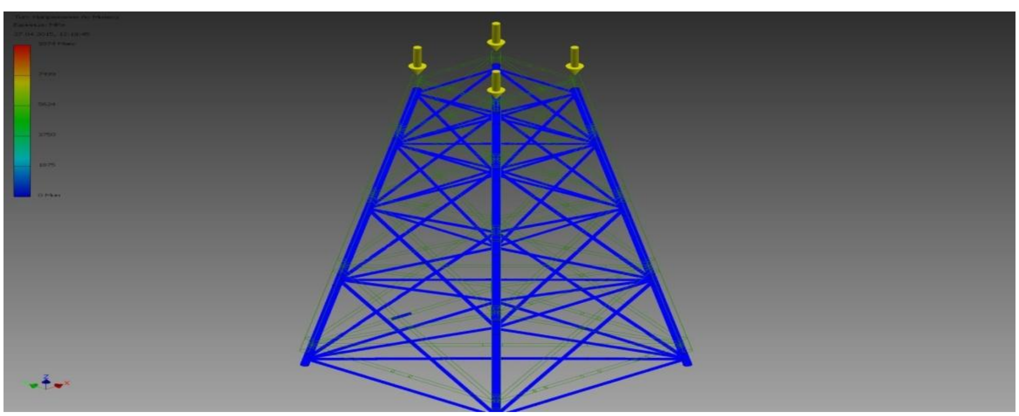

Figure 5 - Calculation Model

The results interpretation phase is where the most critical thinking must take place. You compare the results (such as the numbers versus color contours, movements) with what is expected. You determine if the results make sense, and explain the results based on engineering principles. If the results are other than expected, evaluate the analysis conditions and determine what is causing the discrepancy.

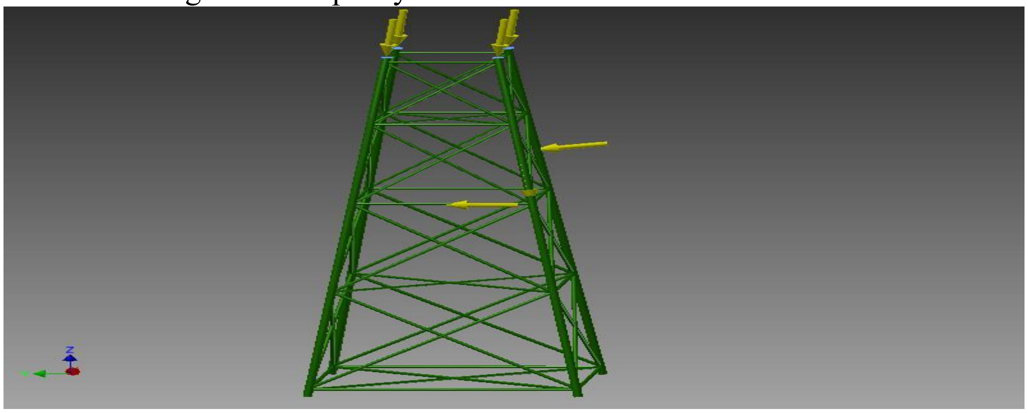

Figure 6 - Dependence of fixings

Conclusion

In this project designed offshore wind turbine with jacket structure by Autodesk Inventor 2014 and Autodesk Maya 2012.

In carrying out degree project:

– Built dimensional solid model and calculation method selected and successfully implemented by means of calculation software package Autodesk Inventor Professional 2014.

– Creation of the methodological basis of the main version of choice of wind turbines, with enhanced functional responsibility

– Choose the method of calculation and successfully implemented calculations using the software package Inventor 2014.

– Applied Materials studied courses: "The Finite Element Method in Structural Mechanics", "Structural Mechanics", "Marine engineering structures and platforms", "Development of offshore structures" and "Computer Design".

Based on the results of calculation of wind turbine for the oil and gas platform and jacket can be concluded that the conditions of strength and stiffness are fully implemented, therefore, you can begin to build the structure.

References:

1 Martijn V.W. Concept design of steel bottom founded support structures for offshore wind turbines / V.W. Martijn. – Norway: Delft University of Technology. Faculty of Civil Engineering and Geosciences, 2007. – 127 p.

2 Steelwork Design Guide to BS 5950 Section Properties: The Steel Construction Institute and The British Constructional Steelwork Association Ltd. – 2001. – 71 p.

3 API RP 2N-95. Recommended Practice for Planning, Designing and Constructing Fixed Offshore Structures and Pipelines for Arctic Conditions. Second Edition. – Dallas: American Petroleum Institute, 1995. – 127 p.

4 Nallayarasu S. Offshore Structures Analysis and Design / S. Nallayarasu. – Madras: Department of Ocean Engineering, Indian Institute of Technology Madras, 2009. – P. 31-56.