FINITE ELEMENT ANALYSIS OF RACK-PINION SYSTEM OF A JACK-UP RIG

[1]KHONDAKER SAKIL AHMED, [2]MD WAHIDUL ISLAM, [3]IBRIJU IBRAHIM

[1,2,3]Department of Civil Engineering, Military Institute of Science and Technology, Bangladesh

Abstract - In offshore mobile jack-up rig, platforms are generally supported by movable jacking system where strength of rack and pinion are very crucial in determining the design life of the superstructure. In this study, a number of finite element models have been developed for noble 7 teeth-pinions to investigate the strength of this jacking system. Since rack-pinion contact patterns change with the rotation of the shaft, models are developed for every 10 degree rotation starting from 0 to 50 degree where elasto-plastic properties of both rack and pinion is considered. Therefore, it is expected that each pinion tooth experiences different stress and displacement as it contacts with rack at different orientation. Elasto-plastic finite element analyses are conducted for worst possible load cases of the gear including self-weight of the platform, wind load, hydrodynamics force as per American Bureau of Shipping (ABS) guideline. The stress distribution of the pinion shows that maximum stress occurs at the root of the tooth, mainly due to bending effect. The analysis reveals that maximum stress in rack and pinion are found to be 1103 MPa and 682 MPa, respectively which is very close to the corresponding material capacity. The outcome of this study will certainly contribute in designing future rack-pinion system for offshore jack-up rigs and similar other structures.

Keywords - Rack-pinion, Finite element, Geometric Analysis, Stress Analysis.

I. INTRODUCTION

Jack-up drilling rigs were first introduced in the late 1950s. Rack-and-pinion-type jack-up units were introduced soon after that and have dominated the industry ever since. Offshore jack-up platforms are designed to operate at various locations with different sea-bed conditions, great water depths and various sea conditions. Recently, offshore mobile jack-up rig has become a very popular subsea infrastructure for oil and gas exploration primarily because of its easy installation, reapplication and excellent adaptability to water depth [1]. The offshore industry has ventured into deeper waters and more severe sea conditions. As a result, offshore platforms are experiencing severe environment condition that leads to increased investigations about the safety of these offshore structures. The jacking system of an offshore rig is very important since the system is always subjected to heavy load [2]. The rack and pinion of the system used to raise and lower the rig which is a crucial responsibility for a mobile jack up rig. The entire stress transfer from pinion to rack involves the other critical parameters such as gear pitch, pressure angle, tooth distance and rack valley depth standards [3, 4]. Based on the importance and their role in jack up rig, the rack pinion system should be investigated comprehensively. Many researcher have focused on the stress analysis of the gear system [1, 3-5]. This study investigates a rack pinion system of real life offshore platform using elasto plastic Finite Element Analysis. After a recent failure of such system of an offshore platform, this study includes a detail stress analysis of a proposed rack-pinion system before their practical application.

II. GEOMETRIC ANALYSIS

Initially, 7 teeth pinion as well as rack geometry is developed by using Excel spread sheet. The geometry is then modelled in 3D Finite Element (FE) software. Based on prescribed material properties and loading, FE analysis is conducted for every 10 degree interval (up to 50 degree) to obtain the maximum stress in rack and pinion. After successful completion of stress analysis. The key dimensions for rack -pinion geometry are

Parameters	Values
Pressure angle	25^0
Pitch radius, r_p	177.8 mm
Tooth distance	159.59 mm
Outside radius	218.995 mm
Rack valley depth	91.44 mm
Rack valley width	20.11 mm
Base radius, r_b	161.142 mm

Table 1: Proposed geometric parameter

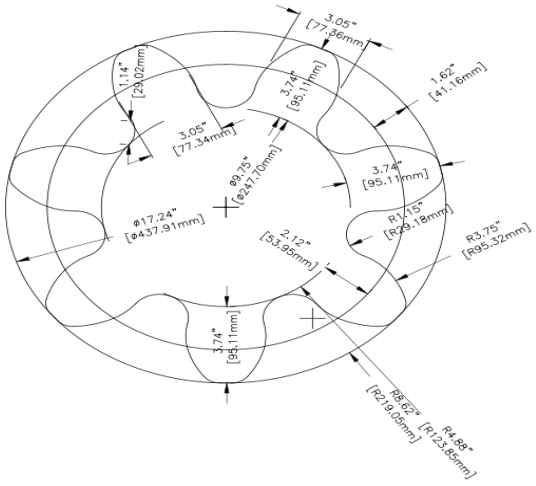


Fig.1 Proposed Pinion geometry

Based on the dimensions stated on Table 1, the geometric shape of the pinion is presented on Fig.1. The drawing is corrected in fraction of millimeter scale. The curvature, root of tooth and pressure angle are corrected as prescribed to the geometric requirement. During operation, the pinion will move over the rack where tooth will be subjected to different contact orientation as presented on Fig.2. In this geometry analysis, it is observed that the similar contact phenomenon will repeat after every 51.43^0 rotation. In view of this occurrence, an interval of every 10 degree starting from 0 to 50 degree rotation that results a total six numbers of rack pinion orientation is presented on Fig.2. The geometry analysis also confirms that both single and double contact can be occurred in the process of rack pinion rotation. The figure shows that at 0, 20, 30 and 50 degree rotation, there will be single contact surface between rack and pinion teeth. However, the contact can be at two surface when the pinion rotation angle is at 10 and 40 degree.

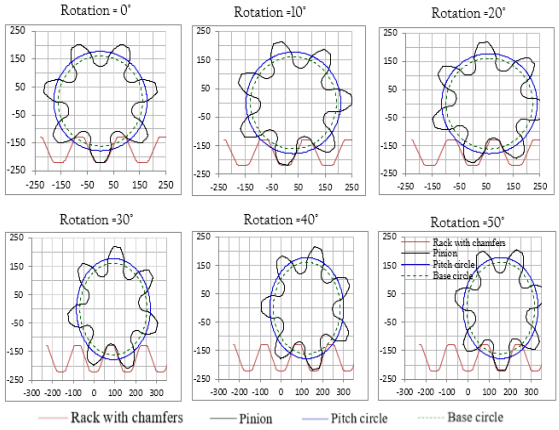


Fig.2 Orientation of Pinion in different angle with the rack

III. FINITE ELEMENT (FE) ANALYSIS

Previously, a number of research works have been conducted in developing methods to investigate the stress analysis of gear drive, among them finite element method is more prominent and is widely used for its high level of accuracy [5-7]. There are many well established FEM software packages such as Abaqus, ANSYS, and ADINA that are commonly used in industries and research centers. In this investigation, three dimensional Finite Element models have been developed to simulate full scale practical rack pinion system using Abaqus 6.14.1.

The prescribed mechanical properties of pinion material that can be achieved through carbon integration and post heat treatment are presented in table.

Properties	Values
Ultimate tensile strength	180 ksi
Yield Strength	155 ksi
Minimum elongation	16%
Hardness (Brinell)	363-401 BHN

Table-2: Mechanical Properties of pinion material

IV. FE MODEL

As presented on table 2, the parameters have been exactly used as proposed for the system with an accuracy of 0.01mm (10 μm). Total number of teeth Z=7, thickness of the pinion = 165.1mm, rack thickness = 127mm and the mounting hole diameter = 165.1mm. The rack profile is straight and the profile of the pinion is involute as presented on the geometry analysis. In the process of 3D finite element model, one tooth is formed first using the coordinates from the spread sheet that is developed for geometry analysis. Then the tooth is replicated for other 6 teeth in 360 degree with solid module having a thickness of 165.1mm and that creates the pinion.

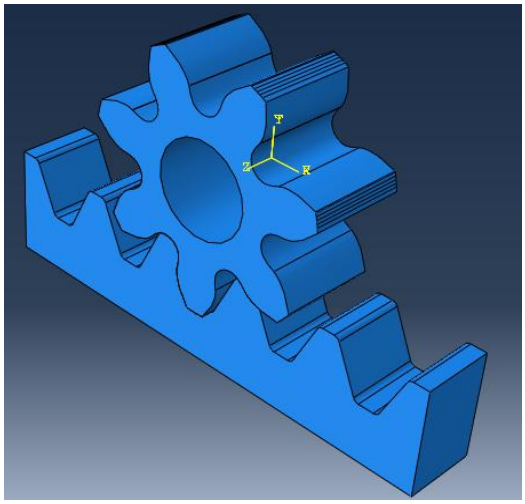


Fig.3 Finite Element Model of the rack-pinion system

In this study, we have modelled the entire rack/pinion system as it practically works in an offshore rig as shown on Fig. 3 rather conducting investigation on a

single tooth. 20 node solid elements are used for meshing of the model. Mesh biasness is considered at the effected tooth and at the contact regions that results a total 3950 numbers of element as shown in Fig.4.

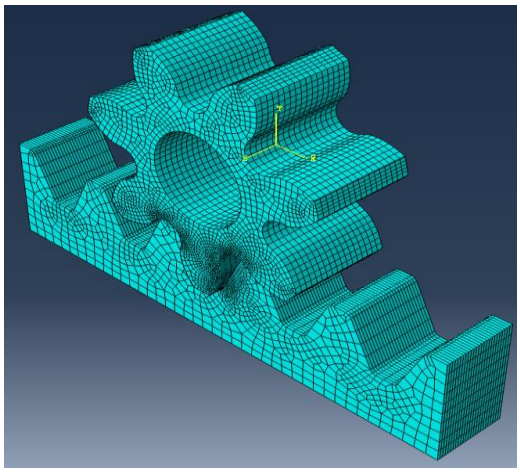


Fig.4 Meshing with the biasness of FE model

Both material and geometric nonlinearity is accounted in the analysis procedure including the reduced integration to save time and lighten the calculation complexity. The material properties of the pinion commensurate with the SAE 4340 with the elasticity of 206 GPa with a Poisson's ratio of 0.29. The surface to surface contact is considered as hard in which the contact surface of the pinion is the master surface and the rack surface is the slave. The boundary condition applied for the rack is fixed support where the pinion is allowed to rotate subjected to the torsional load. The loading of the rack pinion system comes from different combinations and the critical combination is presented on Table 3.

Working Condition	Kn-M	Remarks
Storm Holding	394.2	SF of 2.0 × Jacking
Normal Holding	262.8	SF of 1.8× Jacking
Abs Proto Test	394.2	SF of 1.5× Normal Holding
Max Preloading	262.8	SF of 1.8× Jacking
Maximum Jacking	146	SF of 2.7× Jacking
Brake Holding	315.36	SF of 1.2× Normal Holding

Table 3: Loading the rack/pinion under different condition

V. ANALYSIS RESULTS & DISCUSSIONS

Finite Element analyses have been run for all six rotational cases of rack pinion system. The main target of this analyses is to determine the maximum stress developed in the pinion and hence figure out the suitability of the system based on the material strength. In view of this the analysis for 0 degree orientation that keeps the pinion tooth just in contact with the rack. The result in terms of von meshes stress distribution for 0 degree orientation is presented on Fig.5.

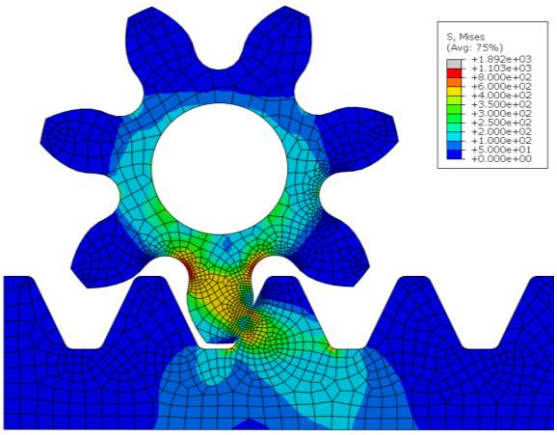


Fig.5 Stress distribution of rack-pinion at 0^0

The non-linear FE analysis shows that the maximum stress is developed at two positions; one is at the root of the tooth that generally occurs due to the bending effect and the other position is at the point of contact between rack and pinion. The maximum von mises stress is found to be 1103 MPa that results at the root of the corresponding tooth which is just below the ultimate strength of the proposed material (1242 MPa).

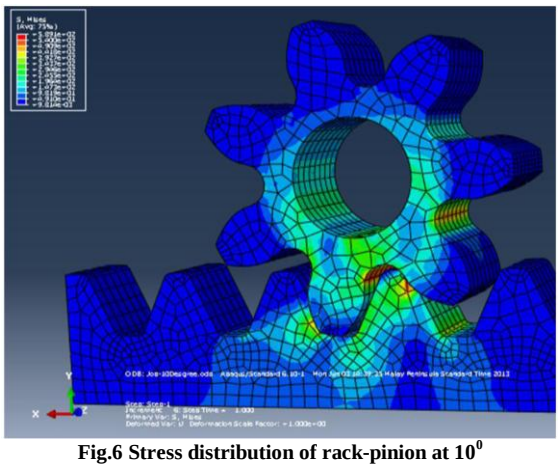
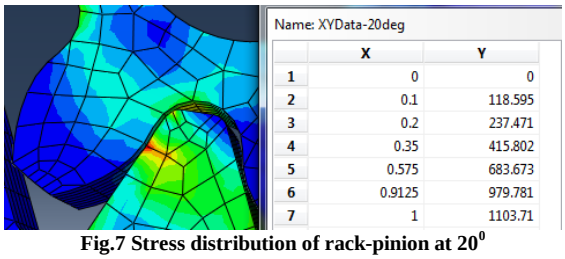


Fig.6 Stress distribution of rack-pinion at 10^0

Finite element model for 10 degree rotation is simulated to obtain the von mises stress distribution for both rack and pinion. It is important to highlight that rack and pinion contact each other at two surface positions at this rotational angle. The maximum stress obtained at this contact angle is found to be 682 MPa which is far below than that obtained at 0 degree rotation. This can be explained by the fact that with the change of rotational angle the positioning of

pinion change with the rack and hence the contact surface also changes. In addition, at 10 degree two surface shares the total amount of torsional load that ultimately results smaller amount of stress at the pinion tooth.

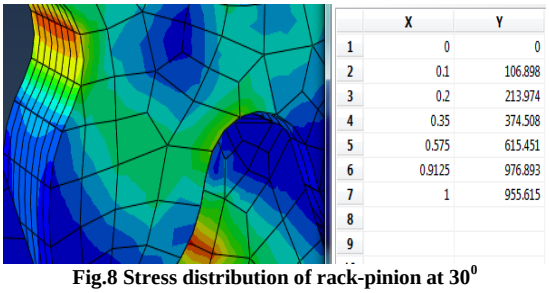


Name: XYData-20deg

	X	Y
1	0	0
2	0.1	118.595
3	0.2	237.471
4	0.35	415.802
5	0.575	683.673
6	0.9125	979.781
7	1	1103.71

Fig.7 Stress distribution of rack-pinion at 20^0

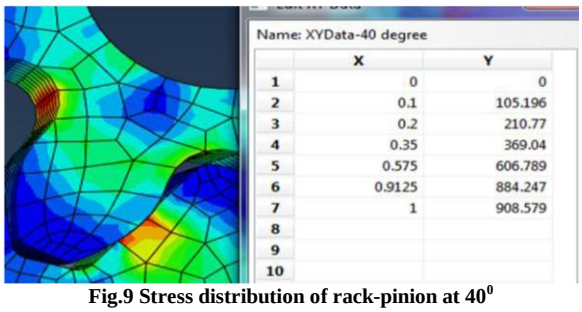
Results extracted from the finite element analysis for 20, 30, 40 and 50 degree are presented on Fig 7 to Fig. 10 respectively. It can be seen from the Fig.7 that maximum stress for 20 degree orientation is obtained same as that resulted for 0 degree. At this rotational angle, rack and pinion contacts at a single surface and maximum stress is obtained at the root of the affected tooth.



	X	Y
1	0	0
2	0.1	106.898
3	0.2	213.974
4	0.35	374.508
5	0.575	615.451
6	0.9125	976.893
7	1	955.615
8		
9		

Fig.8 Stress distribution of rack-pinion at 30^0

Fig.8 shows the von mishes stress distribution of the rack pinion for the rotational angle of 30 degree. The maximum von mises stress is observed to be 955.6 MPa which is larger than that obtained for 10 degree but smaller than that obtained for 20 degree. In addition, it is interesting to highlight that the contact stress is significantly high for this particular angle.



Name: XYData-40 degree

	X	Y
1	0	0
2	0.1	105.196
3	0.2	210.77
4	0.35	369.04
5	0.575	606.789
6	0.9125	884.247
7	1	908.579
8		
9		
10		

Fig.9 Stress distribution of rack-pinion at 40^0

Fig.9 shows the von mishes stress distribution of the critical region of the rack pinion for the rotational angle of 40 degree. The maximum vms stress is observed to be 908 MPa which is close to that obtained for 30 degree orientation. Local contact stress of the rack is very high at this angle.

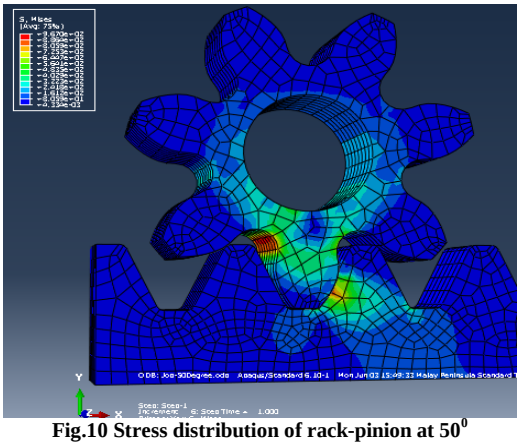


Fig.10 Stress distribution of rack-pinion at 50^0

Fig.10 shows the von mishes stress distribution of the rack pinion for the rotational angle of 50 degree. At this orientation, the pinion comes close to the initial position that genarated at 0 degree. The maximum von mises stress is observed to be 1050 MPa. In any of the abovementioned analyses, no excessive distortion or deformation is observed either in pinion or in the rack surface.

VI. CONCLUSION

A series of finite element models have been run to investigate the proposed rack-pinion system and their effectiveness against all possible loading condition that can occur in regular operation and offshore environment. The maximum stress resulting from the critical load cases and its location for all six analyses is briefly presented on Table 4.

Angle of Rotation	Maximum Stress (Mpa)	Remarks
0^0	1103	Pinion
10^0	682	Both Rack & Pinion (Double Contact)
20^0	1103	Pinion
30^0	955.6	Rack & Pinion (Local Stress)
40^0	908.6	Pinion
50^0	1049	Pinion

Table 4: Maximum stress with different angle of rotation

The result shows that the maximum stress that can be developed in rack pinion system of this particular jack up rig is 1103 which is less than proposed material strength. Therefore, the noble 7 tooth rack pinion system with proposed shape and material can be effective solution for such type offshore jack up rig.

ACKNOWLEDGEMENT

This paper would not have been possible without the cooperation and active assistance of Singapore Amerine, Singapore.

REFERENCES

[1] Cao, Y.G. and S.H. Zhang, Failure analysis of a pinion of the jacking system of a jack-up platform. Engineering Failure Analysis, 2013. 33: p. 212-221.

[2] Montestruc, A.N., Allowable Contact Stresses in Jacking Gear Units Used in the Offshore Industry. Geartechnology, 2010(01): p. 53-57.

[3] Hwang, S.-C., et al., Contact stress analysis for a pair of mating gears. Mathematical and Computer Modelling, 2013. 57(1): p. 40-49.

[4] Refaat, M.H. and S.A. Meguid, On the contact stress analysis of spur gears using variational inequalities. Computers & Structures, 1995. 57(5): p. 871-882.

[5] Mohammed, J., Contact Stress Analysis of Spur Gear Under the Different Rotational Speed by Theoretical and Finite Element Method. 2019.

[6] MackAldener, M. and M. Olsson, Tooth Interior Fatigue Fracture — computational and material aspects. International Journal of Fatigue, 2001. 23(4): p. 329-340.

[7] Gupta, S., N. Shabakhty, and P. van Gelder, Fatigue damage in randomly vibrating jack-up platforms under non-Gaussian loads. Applied Ocean Research, 2006. 28(6): p. 407-419.

★★★