



THREE DIMENSIONAL DYNAMIC STRESS ANALYSES FOR A GEAR TEETH USING FINITE ELEMENT METHOD

Haval Kamal Asker

Department of Mechanical Engineering, Faculty of Agriculture and Forestry, Duhok University, Duhok, Iraq

E-Mail: havaldosky78@yahoo.com

ABSTRACT

One of the main goals during load calculations and vibration analysis for gears is the determination of deformation and stress state in teeth contact zones. This paper describes development of the finite element model for simultaneously monitoring the deformation and stress state of teeth flanks and teeth fillets for spur gears during the contact load period. The paper also describes the Finite Element Method simulation of contact conditions for spur gears teeth with complete 3 dimension meshing model. A suitable analysis is performed in order to select a meshed gears model which is sufficiently economic and in same time sufficiently geometrically accurate. The special algorithm for the tooth profile drawing is developed and built in currently available software (Ansys 10) for Finite Element Analysis to assure drawing of real flanks contact geometry. The optimal mesh size level is chosen, too. The described finite element models are made for the spur gear teeth during the dynamic loading state. The obtained numerical results are suitable for tracking deformation and stress hesitation with time and investigating the effect of damping coefficient on the obtained results.

Keywords: spur gear, load distribution, finite element analysis.

INTRODUCTION

The contact between parts is a common phenomenon which in some cases can be treated with rigorous mathematical theory. Formulae for special cases can be found in machine design books and papers, e.g. two sphere, two parallel cylinders, cylinders on a flat plane, gear teeth, roller bearings, etc. however, these theories only describe the stress in contact region and take into consideration many assumptions and simplifications.

Engineering problems often include requirements for determination of stresses far from the contact zone, so the contact behavior must be simulated properly to find the important stresses in the system.

The calculation of a gear load capacity is one of these cases. A proper computation of forces and deformation of stress throughout the model, i.e., in teeth roots.

Experimental determination of stress state in spur gear teeth with real contact conditions is associated with many limitations. The F.E.M is the most appropriate numerical method that can solve this nonlinear problem, i.e., the F.E. analysis gives the possibilities for developing appropriate models in accordance with needs in modern research of nonlinear tasks.

Ulagu S. *et al.*, analyze contact problems of gears [1]. But their FEM models calculate only contact zone stresses. Dimitrijevic, D. *et al.*, use FEA for the 2 D dynamic analysis of the stress and strain state of the spur gear [2], and Pasta, A. described a 2D FEM model developed for the analysis of a spur gear with a corrected profile [3], also, Marunic, G. [4] Studied spur gears with low values of rim thickness and investigated the maximum tooth root stress.

In this paper a particular real gear is analyzed. The main characteristics of the gear is $z = 17$ standard tooth profile, face width = 20 mm, pressure angle α , $n = 20$, rotation wheel speed = 4.15 m / min, wheel torque $T =$

1264 KN. m, material is steel with modulus of elasticity = 206000 N/mm², Poisson's ratio $\nu = 0.3$, for defined geometry characteristics and torque the normal nominal load that this gear transmits [5] is $F_{bn} = 850$ KN.

The study will investigate the dynamic stresses and deformations induced in the gear tooth in different locations, also the study will describe the hesitation behavior of the gear tooth with time.

THEORY OF SPUR GEARS

There are many types of gear failures when someone analyzes actual gear in services however; these failures are categorized in two general groups. One is the failure of the root of the teeth when the bending strength is inadequate and the other is created on the surface of the gear. There are two theoretical formulas, which deal with the above two fatigue failure problems. One is the Lewis formula that is used to calculate the bending stress and Hertzian equation that is used to compute the contact stress. Various researchers are used variety of method to determine these stresses. The finite element method is very often used to analyze the state of stress in elastic bodies, which have complicated geometry like gears.

The bending stress is one of the crucial parameters during the analysis of spur gears. When the total repetitive load acting on the gear tooth is greater than its strength then the gear tooth will fail in bending. Bending failure in gears is predicted by a formula developed by Wiltred Lewis [6]. The formula uses the bending of cantilever beam to simulate the bending stress acting on the gear as shown in Figure-1; the tangential load (F_t) induces bending stress which tends to break the tooth. The maximum bending stress induced by this force is given by:

$$\sigma_b = \frac{Mc}{I} \quad (1)$$



Where

M = Maximum bending moment = $F_t \cdot l$

F_t = Tangential load acting at the tooth

l = Length of the tooth

C = Half thickness of the tooth (t) = $t/2$

I = Moment inertia = $\frac{bt^3}{12}$

b = Width of gear face.

Substituting the values for M , C and I in equation (1) the stress will be:

$$\sigma_b = \frac{6F_t l}{bt^2} \quad (2)$$

From similarity of the two triangles which is shown in Figure-1,

$$\frac{r/2}{x} = \frac{l}{t/2} \quad \text{This implies } l = \frac{t^2}{4x} \quad (3)$$

Substituting equation (3) into (2) and multiplying by a term $\frac{pd}{pd}$ we get,

$$\sigma_b = \frac{3F_t}{2bx} = \frac{3F_t pd}{2bpd x} = \frac{F_t pd}{bY} \quad (4)$$

Where pd = diametral pitch

$$Y = \frac{2}{3} x pd$$

Equation (4) is known as Lewis equation, which considers only static loading and doesn't take the dynamics of meshing teeth into account. The above stress formula must be modified to account different situations like stress concentration and geometry of the tooth. Therefore, Equation (5) that is shown below is the modified Lewis equation to account for variety of conditions that can be encountered in service.

$$\sigma_b = \frac{F_t}{mn b j} \frac{K_a K_s K_v}{k_v} \quad (5)$$

Where,

K_a = Application factor

K_s = Service factor

K_v = Velocity factor

F_t = Tangential load

m_n = Normal module

J = Geometry factor

k_m = Load distribution factor

Each of these factors can be obtained from machine design books [7]. This analysis considered only the component of the tangential force acting on the tooth and doesn't considered the effects of the radial force, which will cause in compressive stress over the cross

section on the root of the tooth. Suppose that the greatest stress occurs where the force is exerted at the top of the tooth, usually there are at least two pairs in contact. In fact, the minimum stress at the root of the tooth occurs when the contact point moves near the pitch circle because there is only one tooth pair in contact and these teeth pairs carries the entire torque or load. When the load is moving at the top of the tooth, two teeth pairs share the whole load if the ratio is longer than one and less than two.

If one tooth pair is considered to carry the whole load and it acts on the top of the tooth, this loading condition is enough to analyze the gear bending stress.

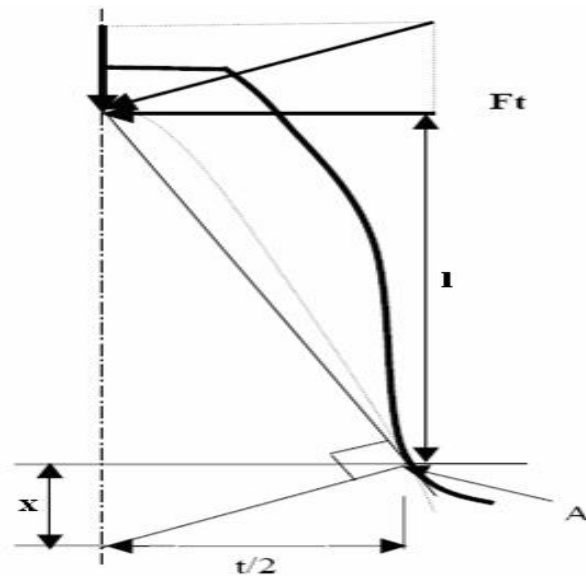


Figure-1. The cross section and the dimensions of gear tooth profile to determine the lewis formula.

BASIC CONSIDERATION OF THREE DIMENSIONAL FEM PINION MODEL

The base of pinion was constrained radially and axially, and in tangential direction was left free. The described boundary conditions of pinion related to the load and constraint conditions.

The state of stress of spur gear was determined by means of FEM [8]. The pinion-wheel gear models with the boundary conditions as described before were meshed with (brick 8 node 45) solid elements. In relation with the time required for the calculation, the attention was devoted to the density of grid in areas of interest, which are the surfaces the contact is accomplished through the area of the loaded tooth fillet.

Also, the dynamic analysis was made by this paper with a damping coefficient = 0.05, to investigate the effect of time increments on the resulting stresses and deformations.



NUMERICAL EXPERIMENT PROCEDURE

Identifying the stress state is a more complex task for 3 D gear teeth model than for 2 D models.

The load distribution in the spur gear mesh should be solved only by resolving the load distribution along the contact line and along the gear teeth, thus calculation of spur gear teeth stress and deformation could be made only by a three-dimensional finite element model. For developing such model the study defined the procedure of numerical experimental Figure-2.

The first step in the procedure is developing a plane finite element model for the spur gear contact area. The geometry of a plane finite element model is teeth profile geometry in the face plane. In the second step, the developed plane contact model has to be extruded. Then, the next step is developing three dimensional solid finite element models for the selected contact line positions by extruding plane model. The dynamic finite element analysis follows then and gives numerical results with respect to time that could be presented by Figure-2, which is a diagram showing the flow chart for the numerical experimental process.

THREE DIMENSIONAL SOLID FINITE ELEMENT MODELS FOR SELECTED LINE OF CONTACT POSITIONS

This sup - step procedure implies meshing of the whole developed plane finite element models for the teeth profile at the face surface as well as the width surface which gives appropriate 3 D solid finite element models for numerical calculations.

The finite element types chosen for the 3 D FEM spur gear teeth model developing are:

The 3 D isoperimetric structural solid element defined by eight points - for the 3D gear modeling; and the 3D point to surface contact element for the tooth contact modeling.

The boundary conditions on the developed 3D FEM model is defined by the displacement constraint at the direction normal to the surfaces which separate the modeled gear segment from the rest of the gear body. Also, displacement constraints at the direction normal to the traverse plane (Z - axis of the global Cartesian coordinate system) set at the nodes placed on gear rim border. The external load is defined on the teeth flanks opposite the contact flanks at the direction of the line of action (the same as the path of the contact direction for spur gear and Y - axis of the global Cartesian coordinate system) by specific concentrated forces set at the nodes placed on the middle part of the gear face width. The sum of these forces is a calculated normal force that this gear transmits [5].

Figure-3 is a three dimensional gear tooth and it shows the gear tooth before meshing and applying loads. While Figure-4 is showing the fine meshed gear tooth with

the created nodes and elements, so the study can calculate the nodal solutions and elements stresses by using F.E.M. and Figure-5 is the meshed gear tooth applied to loading condition in the line of contact along the face width and is also the meshed gear tooth applied to constraints in the (x, y, z) directions at the bottom of the gear tooth base.

NUMERICAL RESULTS ANALYSIS AND DISCUSSIONS

The 3 D diagrams Figures (7, 8, 9) are used for tracking the variation of gear teeth deformations, gear teeth stiffness, mesh stiffness, load distribution in the mesh.

Figure-6 shows the case study applied to the gear teeth which conducts three regions to study in addition to the dynamic nodal and element results.

In the first region (line of contact), von mises stresses Figure-10 indicates the position of the max. Stress which occur in node (2741) and equal to (2240 MPa), also the deformations in X direction Figure-11 indicates to an increase in the position of max. Stress node (2741) and equal to (0.034 mm). While the deformations in Y direction Figure-12 indicates to a decrease in the position of maximum stress node (2741) and equal to (- 0.022 mm).

In the second region of study (line of profile) which through by the middle face width of the gear teeth. Von Mises stresses Figure-13 reflect the region of maximum stress (2000 Mpa) near the line of contact region which is logically clear. Also, the deformations in X direction Figure-14 show an increase in the position of the line of contact (0.032 mm). But there is decrease in the deformations in Y direction Figure-15 in the same position of the line of contact (-0.019 mm).

The third region of study (gear tooth root line), Von mises stresses Figure-16 show approximately a steady values but the maximum stress occur in node (2407) and equal to (73.5 Mpa), and the deformation in X direction Figure-17 indicate to also a steady deformation and the maximum deformation in node (2407) and equal to (0.0119 mm), while for the deformation in Y direction Figure-18 there is a smooth curve represent the Y deformation and the maximum deformation was (-0.001 mm).

The dynamic behavior of stress and deformations were declared in Figures (19, 20, 21), with the effect of time increments (10 increments with 0.5 sec for each step) and the damping coefficient (0.05) and the results of dynamic analysis are taken for the node of maximum Von Mises stress which is in node (2741).

The 3 D (FEM) stress calculations enabled the insight into the tooth root stress distribution, face width stress distribution, and the profile stress distribution, which is not available by analytical 2 D approach.

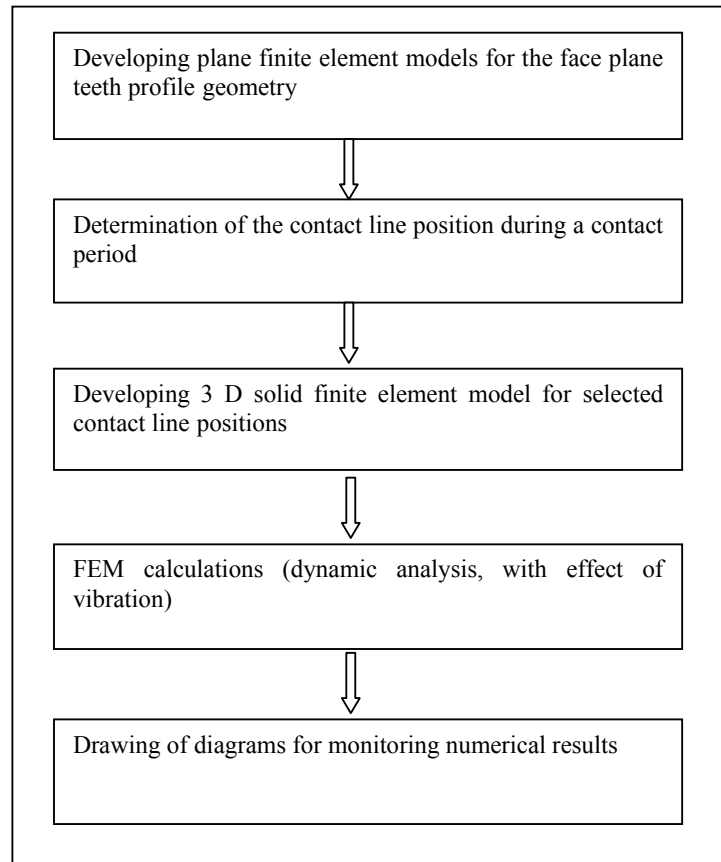


Figure-2. The procedure of the numerical experiments.

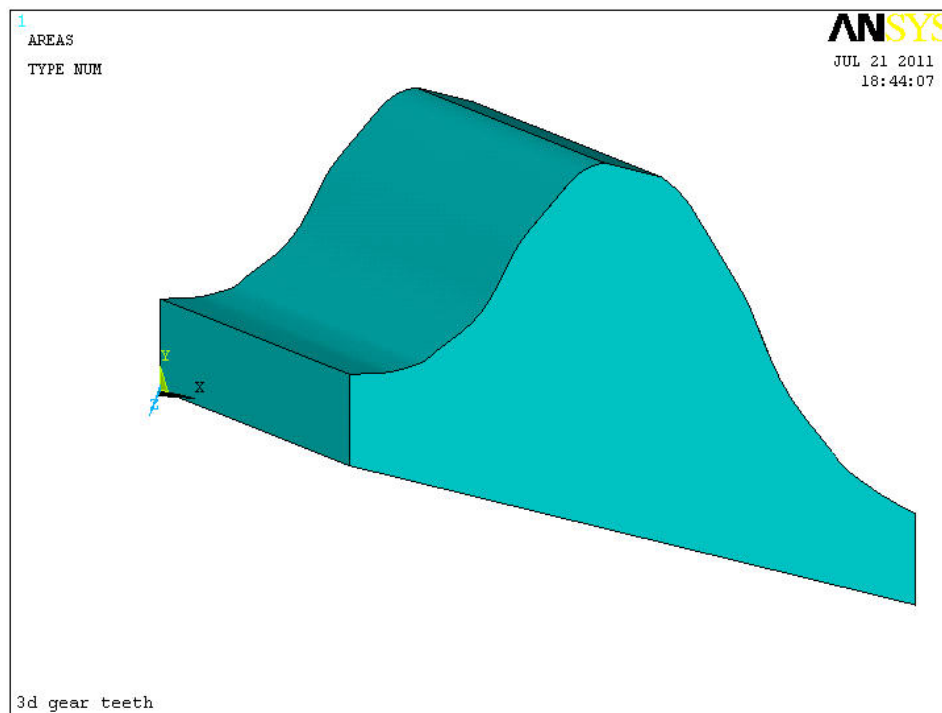
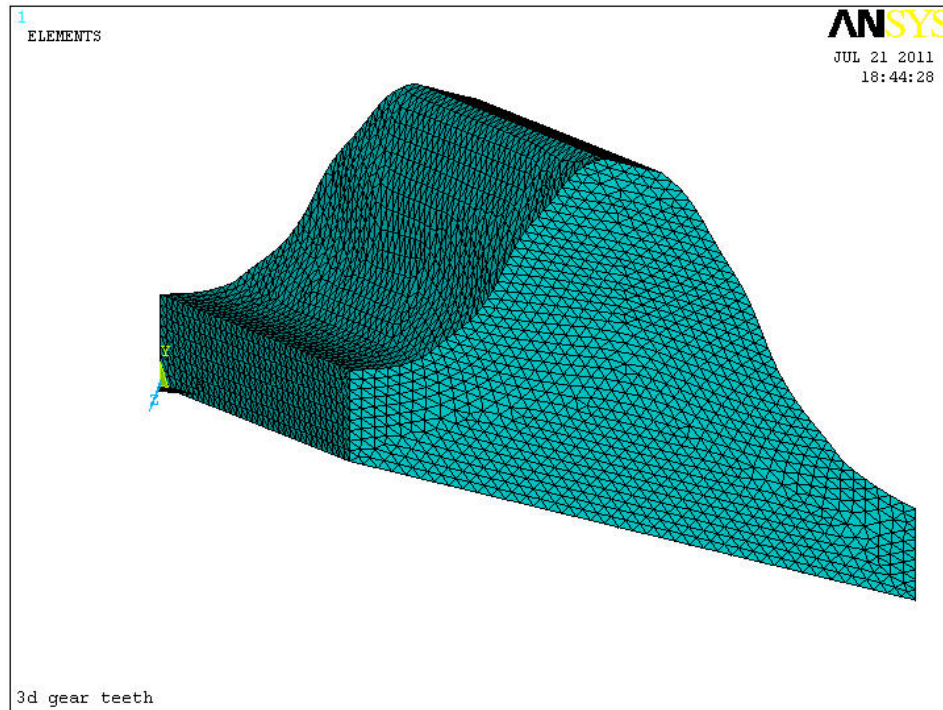
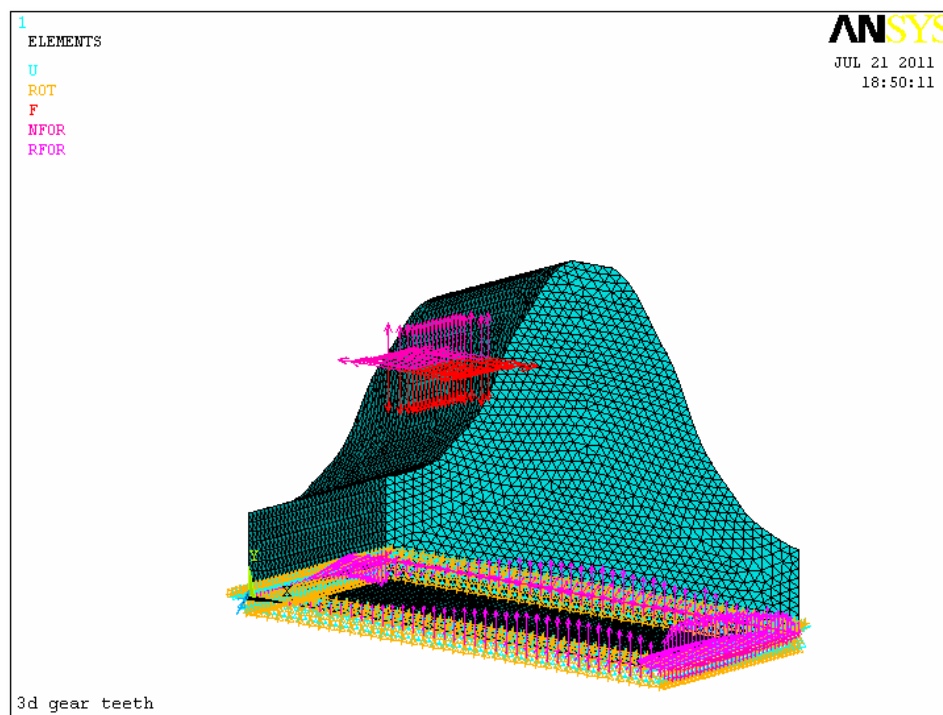


Figure-3. Gear teeth area.

**Figure-4.** Gear teeth meshing.**Figure-5.** Gear teeth loading.

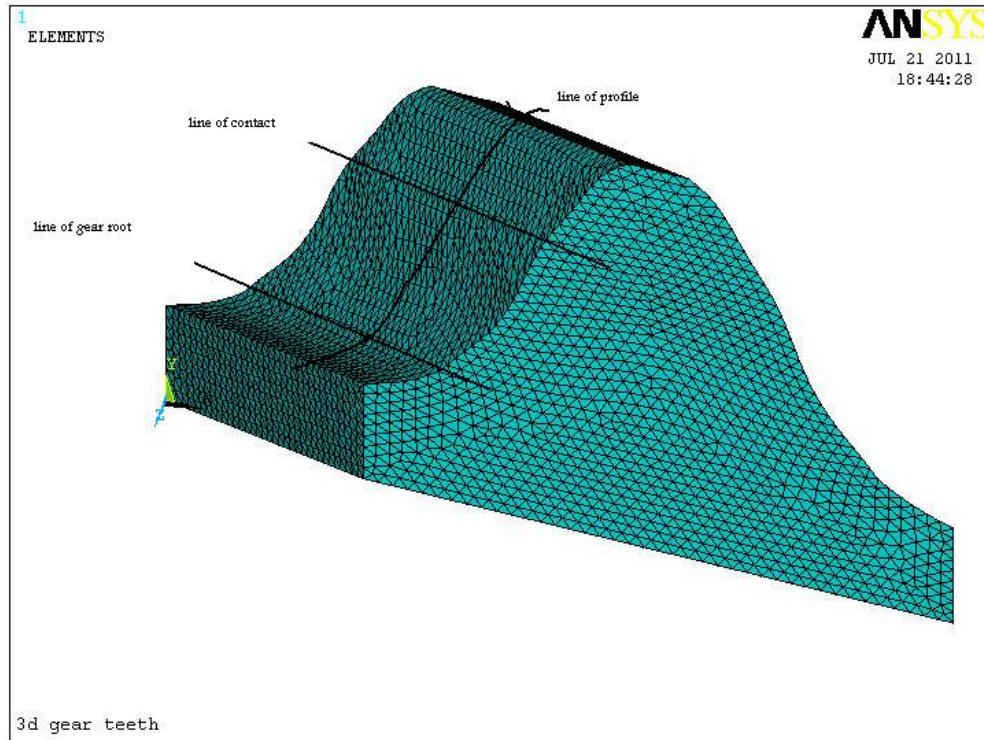


Figure-6. Studied lines among the body.

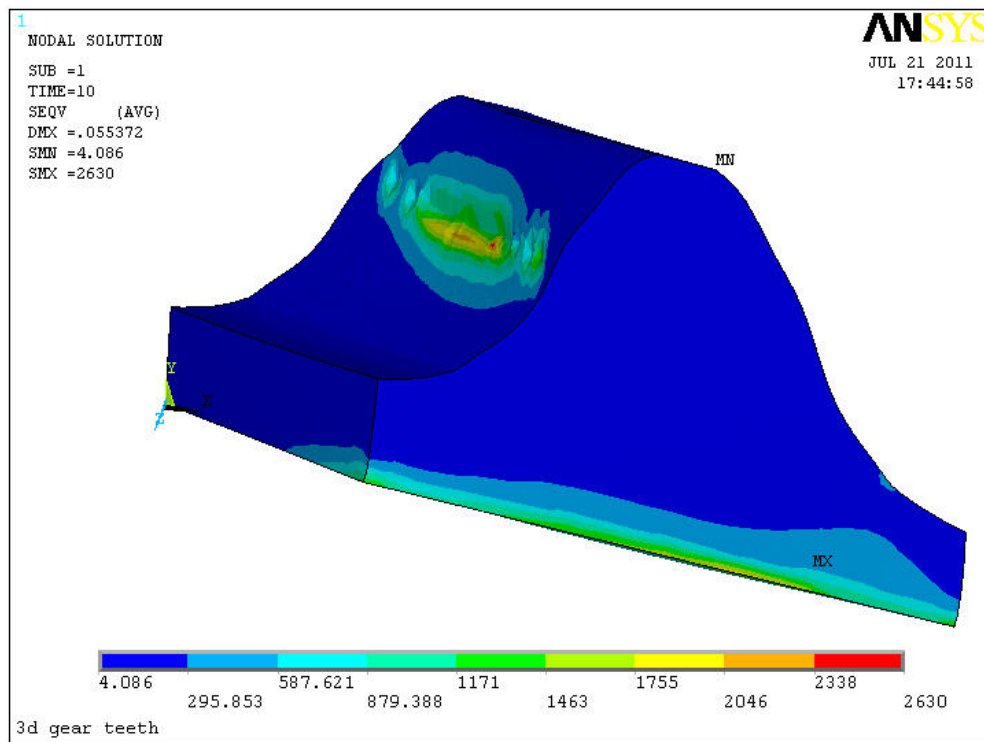


Figure-7. Von mises stresses in the line of contact.

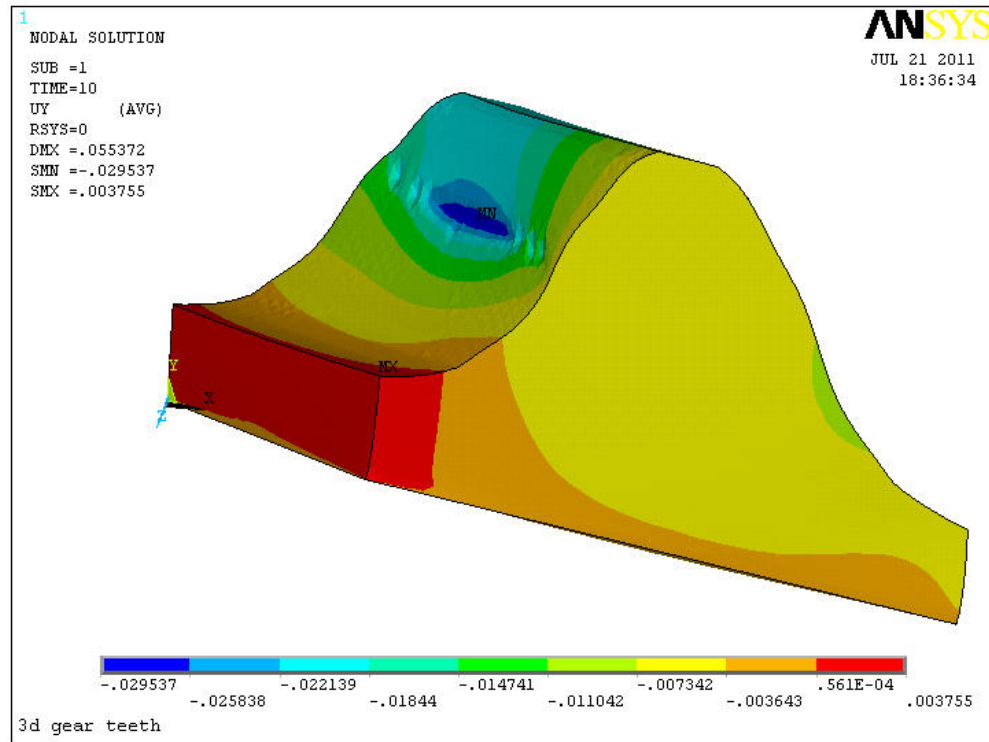


Figure-8. Deformation in Y direction in the line of contact.

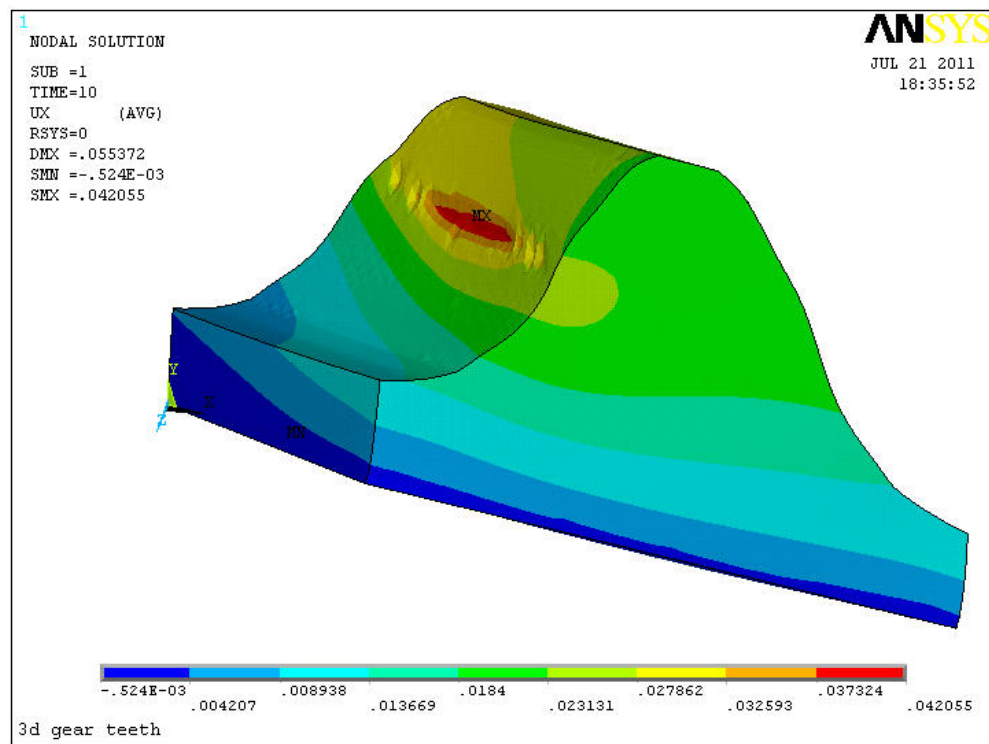


Figure-9. Deformation in X direction in the line of contact.

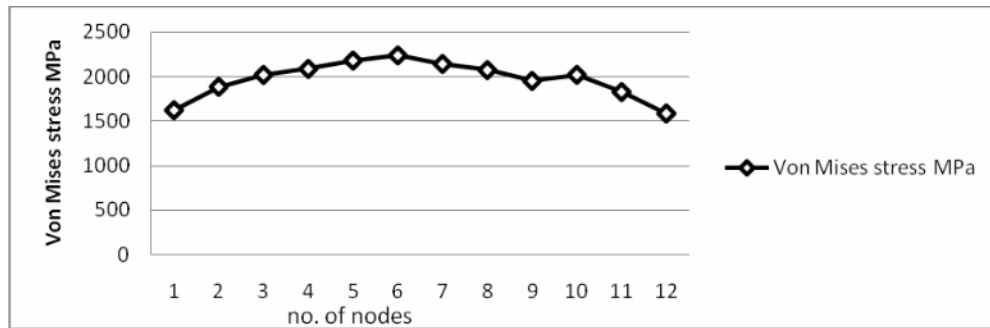


Figure-10. Von Mises stress along the face width of the gear teeth.

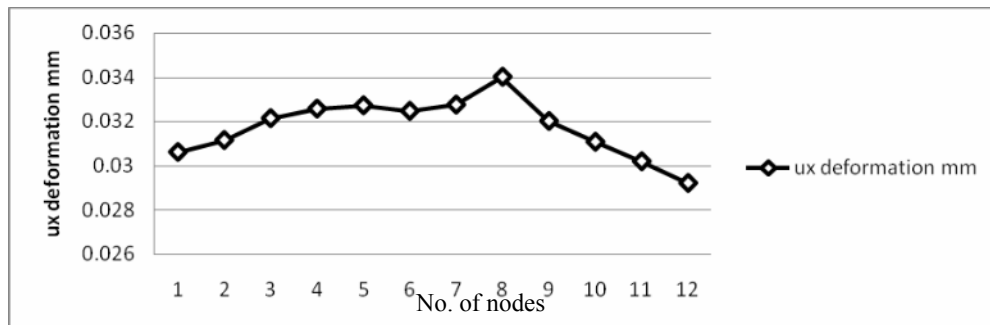


Figure-11. Deformation in the X direction along the face width of the gear teeth.

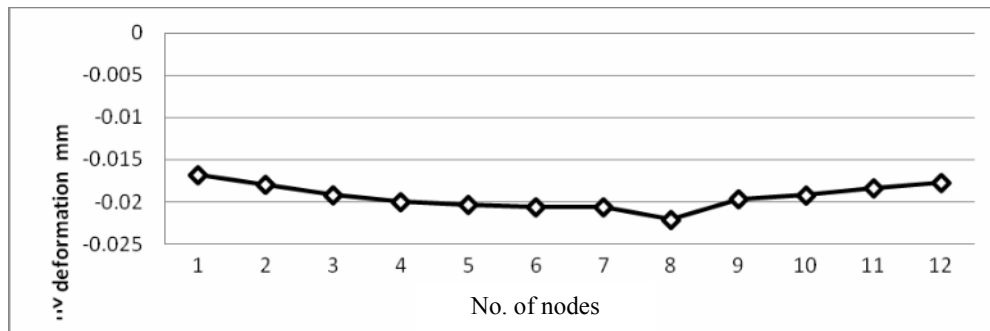


Figure-12. Deformation in the Y direction along the face width of the gear teeth.

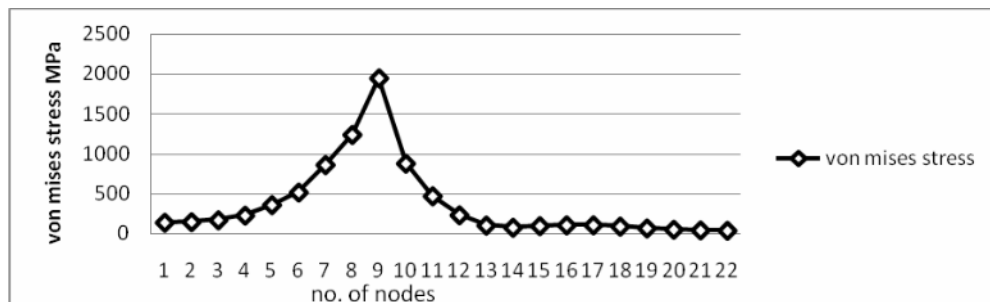


Figure-13. Von Mises stress along the profile of the gear teeth.

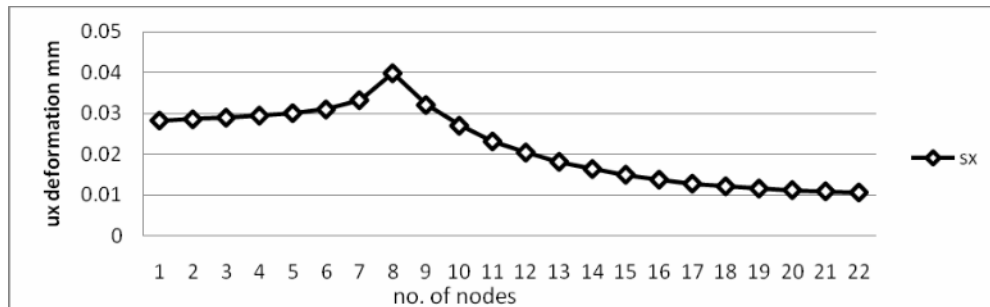


Figure-14. Deformations in the X direction along the profile of the gear teeth.

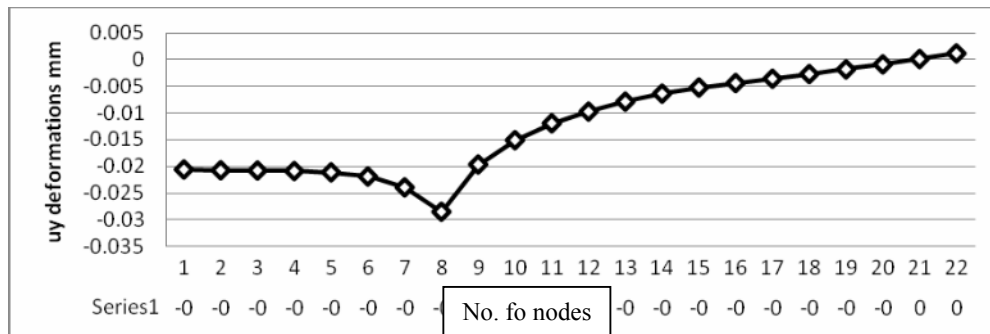


Figure-15. Deformations in the Y direction along the profile of the gear teeth.

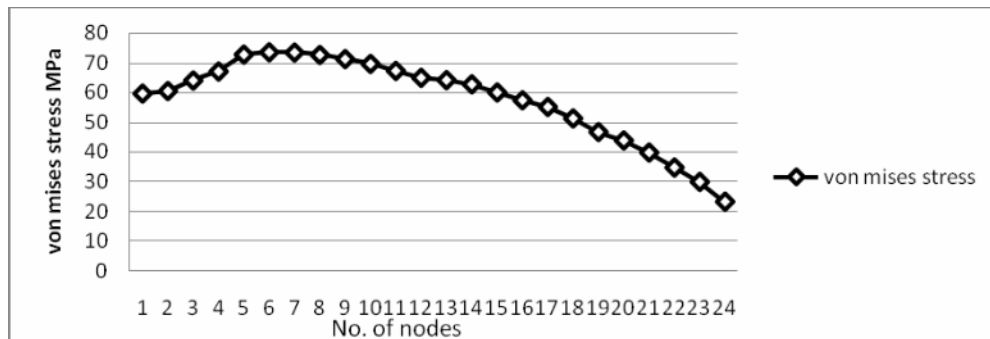


Figure-16. Von Mises stresses in the root line of the gear teeth.

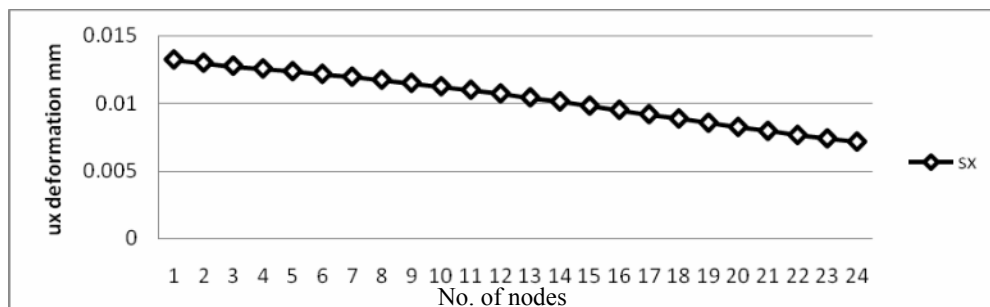


Figure-17. Deformation in the X direction in the root line of the gear teeth.

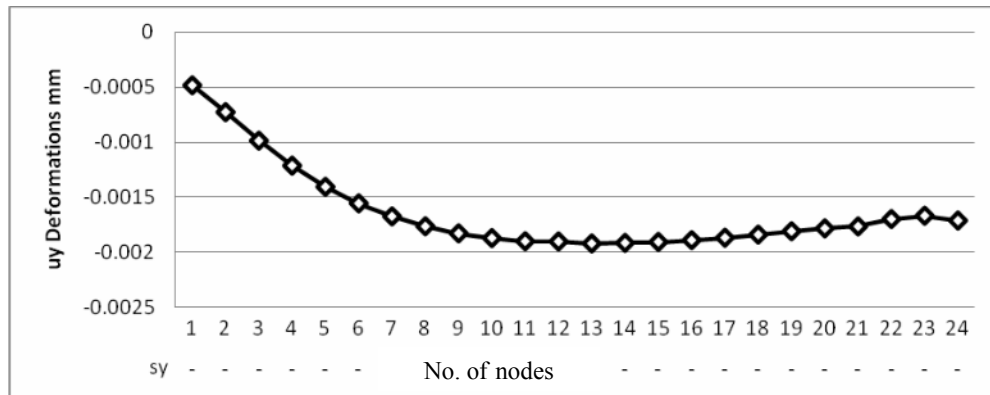


Figure-18. Deformation in the Y direction in the root line of the gear teeth.

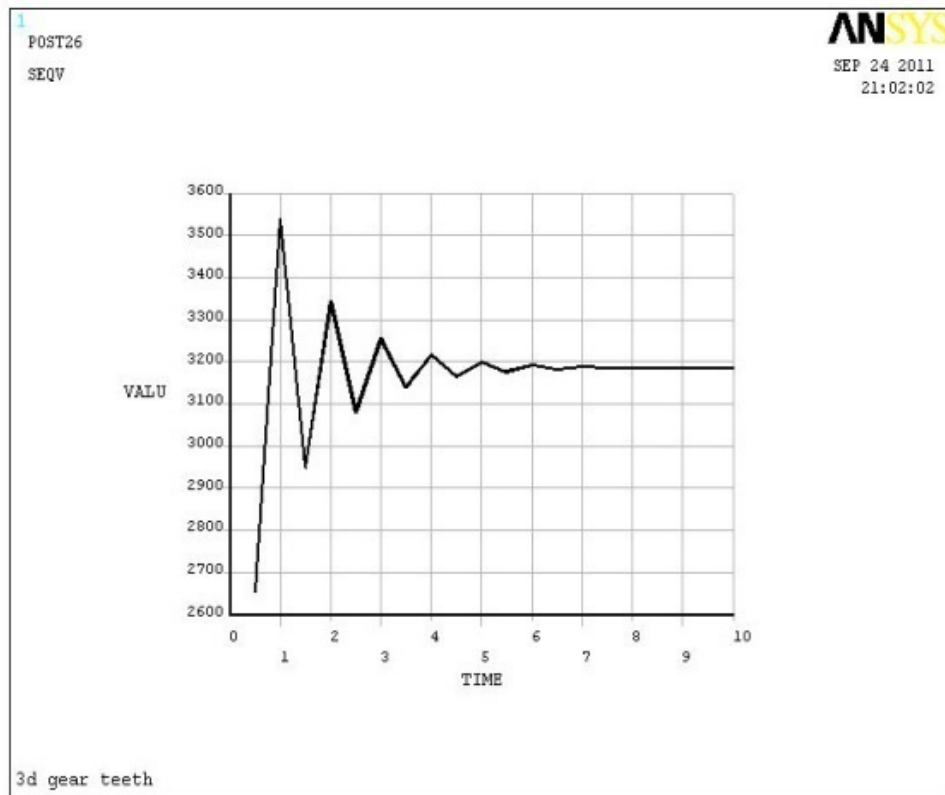


Figure-19. Variation of Von Mises stress with time in node (2741) in which the maximum Von Mises stress occurs.

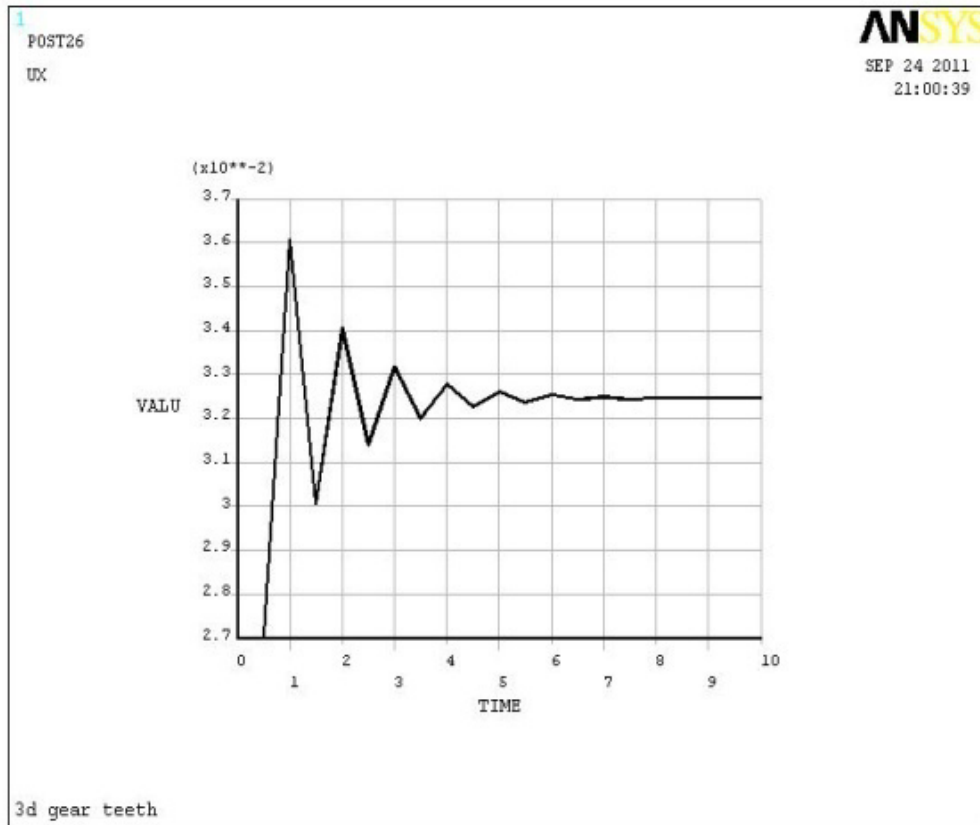


Figure-20. Variation of deformation in X direction with time in node (2741).

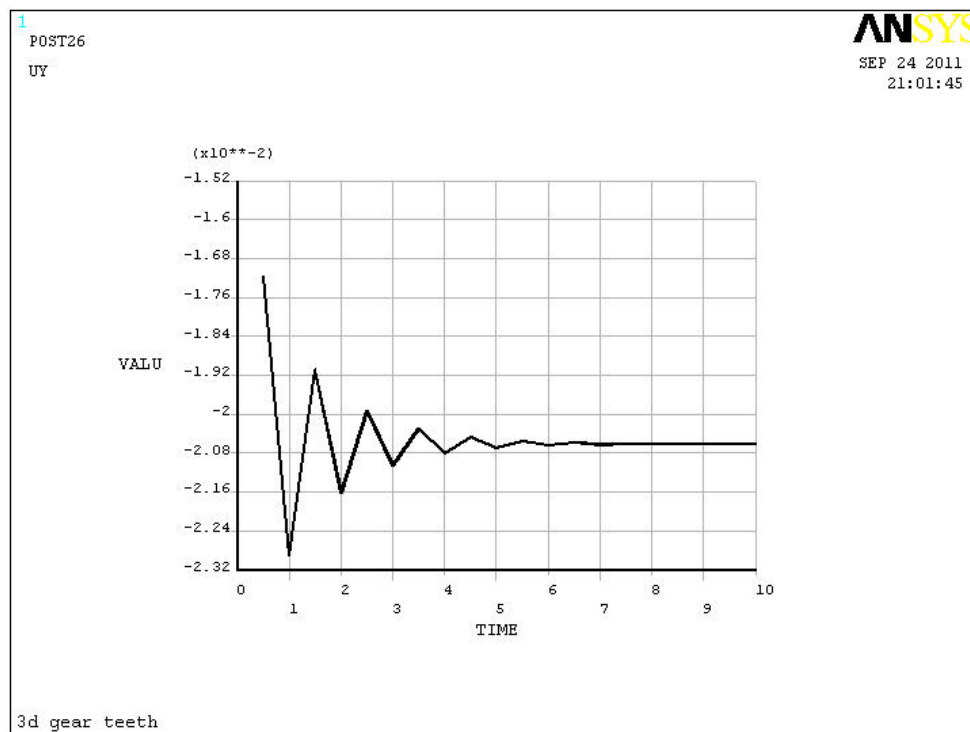


Figure-21. Variation of deformation in Y direction with time in node (2741).



CONCLUSIONS

Analytical method of gear analysis uses a number of assumptions and simplifications and it is intended to determine the maximum stresses and deformations values. In this paper, numerical approach has been used for predicting the static and dynamic contact stresses of spur gear.

The contribution of this work can be summarized as follows:

- a) It is shown that the effective method to estimate the contact stresses and deformations was by using three dimensional model of the gear tooth;
- b) The type and direction of loading has a big role in specifying the direction and increments of the resulted deformations;
- c) The 3D FEM stress calculations enabled the insight into the tooth root stress distribution along the face width, which is not available by analytical 2D approach;
- d) The stress increase significantly due to triangle load distribution. That means under non - uniform contact conditions in the meshing process; and
- e) The described numerical experiment procedure and the analyzed results are a new viewpoint in gear calculation. Integrating new methods into standard calculations of spur gears could be very useful for engineers and scientists whom investigating spur gear.

REFERENCES

- [1] Ulaga S. *et al.* 1999. Contact problems of gears using overhouser splines. International Journal of mechanical science. Elsevier publishing. pp. 385-395.
- [2] Dimitrijevic D. *et al.* 2007. Dynamic analysis of the stress and strain state of the spur gear. Scientific technical review. 57(3-4).
- [3] Pasta A. 2007. Finite element method analysis of a spur gear with a corrected profile. Journal of strain analysis for engineering design. 42: 281-292.
- [4] Marunic G. and Gregov G. 2010. Spur gear tooth root critical section, advanced engineering. ISSN 1846-5900.
- [5] Niemann G. and Winter H. 2002. Mashinenelemente. Band 2, springer.
- [6] Orthwein W.C. 2004. Machine Component Design. Jaun publishing house, Mumbai, India.
- [7] Shigley J.E. and Mischke C.R. 1996. Standard Handbook of Machine Design. McGraw-Hill, USA.
- [8] Nikolić V. and Bogdanović G. 1994. Determination of stress state of gear drive at static and dynamic loadings. MVM-International Journal for Vehicle Mechanics, Engines and Transportation Systems, September. 20(3).
- [9] Stolarski T., Nakasone S. and Yoshimoto S. 2006. Engineering Analysis with Ansys Software. Elsevier.
- [10] Zienkiewicz O.C. and Taylor r. L. 2000. The Finite Element Method. Vol. 1: The Basis. Butterworth - Heinemann.
- [11] Nikolić V. and Cvejić-atanasovska I. 2003. The analysis of contact stress on meshed teeth's flanks along the path of contact for a tooth pair. Facta Universitatis. Serie S: Mechanics, Automatic Control and Robotics. 3(15): 1055-1066.