FACTA UNIVERSITATIS
Series: **Mechanics, Automatic Control and Robotics** Vol.3, N° 15, 2003, pp. 1055 - 1066

THE ANALYSIS OF CONTACT STRESS ON MESHED TEETH'S FLANKS ALONG THE PATH OF CONTACT FOR A TOOTH PAIR

UDC 532.12+ 620.10+521.8

Vera Nikolić-Stanojević, Ivana (Atanasovska) Cvejić

Faculty of Mechanical Engineering Kragujevac
Sestre Janjić 6, 34 000 Kragujevac, Serbia

Abstract. *The basic task of this paper is analysing and determining the shape of the function which defines the change of contact stresses on tooth flanks along the path of contact for a tooth pair. The determination of contact stress maximum point and the calculation of its value is possible if the course of change of contact stress is provided. This enables the precise calculation of a gear pair pitting resistance.*
The numerical method – Finite Element Method (FEM) is used for modeling the contact of tooth flanks. This paper gives the detailed description of model development procedure. The results provided for the stress state of tooth flanks are also presented and discussed.
The comparison of analyticly and numericly obtained curves of change in stress state on meshed tooth flanks, confirmed the accuracy of the developed model. This model opens new possibilities for succesful application in calculations and research involved in improving the geometry of gears pairs.

Key words: *Gears, Contact stress, The Finite Element Method*

INTRODUCTION

Starting point in studying the contact strains on tooth flanks is to define the contact surfaces as the parts of equivalent imagined cylinders. The radii of these cylinders are identical to the radii of flanks' curvatures in contact point (ρ_x), [4]. During the contact period in unloaded state, the active surfaces of meshed teeth's flanks are in contact with each other through the instantaneous contact lines, which coincide with the common generatrix of involute surfaces. The forces of mutual action exist in teeth's mesh. They act in the direction of the common normal for contact surfaces, so they are known as normal load (F_{bn}).

Received April 20, 2003

The action of normal load F_{bn} provokes elastic deformations on the contact surfaces of meshed teeth. Because of that, there is not flank contact through the contact line, but through the elastically deformed flank surfaces. In the case of involute spur gear, the curvatures' radii of the meshed teeth are continuously changing during the meshing period, so the determination of contact stresses on meshed flanks is a task analogical with Hertz's task for two cylinders in contact.

THE THEORETICAL ANALYSIS

The maximum local strain that appears on tooth flanks during the mesh period is relevant for the calculation of gear's load capacity, according to tooth flanks' strength criterion. In order to determine the maximum strain value, one must, first of all, determine the nominal value of tooth flanks' loading and the corresponding load factors. Load factors take into consideration real working conditions and translate the nominal load value into the maximum load value. The maximum load is defined as maximum normal load F_{bn} on tooth flanks, which is result of maximum tangential load, in accordance with:

$$F_{bn1} = F_{bn2} = F_{bn} = \frac{2T_1}{d_{b1}} = \frac{2T_2}{d_{b2}} = \frac{F_{tmax}}{\cos\alpha_w} = \frac{K_A K_V K_\alpha K_\beta F_t}{\cos\alpha_w} \tag{1}$$

where: K_A – is the application factor;

K_V – is the internal dynamic factor;

K_α - is the factor of load distribution among simultaneously meshing tooth pairs;

K_β - is the factor of load distribution over the facewidth.

In the case of involutes spur gear, the curvatures' radii of the meshed teeth are continuously changing during the meshing period of a tooth pair, in accordance with following expression:

$$\rho_x = r_b \cdot \mathrm{tg}\alpha_x = r_x \cdot \sin\alpha_x \tag{2}$$

When the expressions that correspond for tooth flanks' contact:

$$q = \frac{F_{bn}}{b}\ ;\ \rho = \frac{1}{2\lambda} = \frac{\rho_1 \cdot \rho_2}{\rho_1 + \rho_2} \text{ and } k_i = \frac{1-\nu_i^2}{E_i}, i = 1,2 \tag{3}$$

are inserted in Hertz's solution for two cylinders in contact, [5]:

$$p_{max} = \frac{2q}{\pi \cdot a} = \sqrt{\frac{2}{\pi} \cdot q \cdot \frac{\lambda}{k_1 + k_2}} \tag{4}$$

then it takes a form applicable for the calculation of the maximum Hertzian contact stresses in the points of mesh without slipping for single tooth pair meshing period:

$$\sigma_H = \sqrt{\frac{F_{bn}}{b \cdot \rho} \cdot \frac{1}{\pi \cdot \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)}} \tag{5}$$

where: ρ is the equivalent curvature's radius in a contact point.

With application of the expression for the radii of flanks' curvatures and the expression for the equivalent curvature's radius in the contact point, the expression (5) takes a form applicable for the active stress calculation while the contact is in the pitch point C (σ_{H0}):

$$\sigma_{H0} = \sqrt{\frac{1}{\pi \cdot \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)}} \cdot \sqrt{\frac{F_{t\max}}{b \cdot d_1} \cdot \frac{u+1}{u} \cdot \frac{2}{\cos^2 \alpha \cdot \mathrm{tg}\, \alpha_w}} \tag{6}$$

It is possible to monitor the change of active stress on tooth flanks through the change of ratio of active stress in any contact point σ_{Hx} and active stress for contact in the pitch point σ_{H0}. If the equations for the calculation of the active stresses on meshing tooth flanks, which are developed from the Hertz's solution for contact stresses while two cylinders are in contact, are applied, then this ratio takes the following form:

$$\frac{\sigma_{Hx}}{\sigma_{H0}} = \frac{\sqrt{(F_{\mathrm{tmax}})_{\mathrm{x}} / (b \cdot \rho_x \cdot \cos\alpha_w)}}{\sqrt{(F_{\mathrm{tmax}})_{\mathrm{C}} / (b \cdot \rho_C \cdot \cos\alpha_w)}} \tag{7}$$

where: ρ_x and ρ_C – are the equivalent curvature's radii in any contact point and the pitch point, in accordance with: $\rho = \frac{1}{2\lambda} = \frac{\rho_1 \cdot \rho_2}{\rho_1 + \rho_2}$;

$(F_{\mathrm{tmax}})_{\mathrm{x}} = K_{\mathrm{A}} \cdot K_{\mathrm{V}} \cdot K_{\beta\mathrm{x}} \cdot K_{\alpha x} \cdot F_{\mathrm{t}}$ – is the maximum tangential load for contact in any point on the path of contact;

$(F_{\mathrm{tmax}})_{\mathrm{C}} = K_{\mathrm{A}} \cdot K_{\mathrm{V}} \cdot K_{\beta C} \cdot K_{\alpha C} \cdot F_{\mathrm{t}}$ – is the maximum tangential load for contact in the pitch point.

If n tooth pairs are simultaneously meshed and F_{i} is the load transferred by i^{th} tooth pair, then the factor of load distribution among simultaneously meshing tooth pairs for i^{th} tooth pair ($K_{\alpha\mathrm{i}}$) describes the level of interest in mesh for i^{th} tooth pair. It is defined with the following quotient:

$$K_{\alpha\mathrm{i}} = \frac{F_{\mathrm{i}}}{F}\ , \ F = \sum_{i=1}^{n} F_{\mathrm{i}} \tag{8}$$

In accordance with the assumption that tangential force per unit tooth width is constant during the meshing period (K_β=const.), the ratio (7) takes a form suitable for monitoring the changes of contact stress on tooth flanks during the contact period for a tooth pair:

$$\frac{\sigma_{\mathrm{Hx}}}{\sigma_{\mathrm{H0}}} = Z_{\rho\mathrm{x}} \cdot \sqrt{\frac{K_{\alpha\mathrm{x}}}{K_{\alpha\mathrm{C}}}} \tag{9}$$

where: $K_{\alpha\mathrm{x}}$ and $K_{\alpha\mathrm{C}}$ – are factors of load distribution among simultaneously meshing tooth pairs for contact in any point and in the pitch point; $Z_{\rho x} = \sqrt{\rho_C / \rho_x}$ – is radius factor for curvatures of meshed teeth's profiles for contact in any point on the line of contact.

During the meshing period for a tooth pair of a gear pair, the factor of load distribution among simultaneously meshing tooth pairs K_α is defined as the function of the loading value, the tooth stiffness and the difference between meshing teeth's base pitches. In meshing periods for two simultaneously meshing tooth pairs, factor K_α is changing in accordance with the complex continuous functions. However, the knowledge of these functions is not important. For the calculation of tooth flanks strength, the important information is only maximum loads in the moments of load exchange among tooth pairs in mesh. Consequently, these functions can be assumed as linear functions.

In this paper, for the theoretically and numerically analyses, one particular gear pair for large transport machines (dradges) are taken. Its characteristics are given in tab.1. The results of theoretically analysis of the changes of active stress on meshed tooth flanks, obtained through the computer program developed in [1], are given in figure 1. The curve in figure 1 is obtained for ideally manufactured (Δp_b=0) gear pair and in accordance with the described assumptions.

Table. 1.

Quantity	Symbol	Value
Number of teeth	z_1	20
	z_2	96
Addendum modification coefficient	x_1	0
	x_2	0,328
Facewidth	b	350 *mm*
Module	$m=m_n$	24
Pressure angle	$\alpha=\alpha_n$	20°
Helix angle	β	0°
Rotational speed of wheel	n_2	4,1596 min^{-1}
Pinion torque and wheel torque	T_1	526,41667 $KN\cdot m$
	T_2	2526,8 $KN\cdot m$
Useful level	η	1

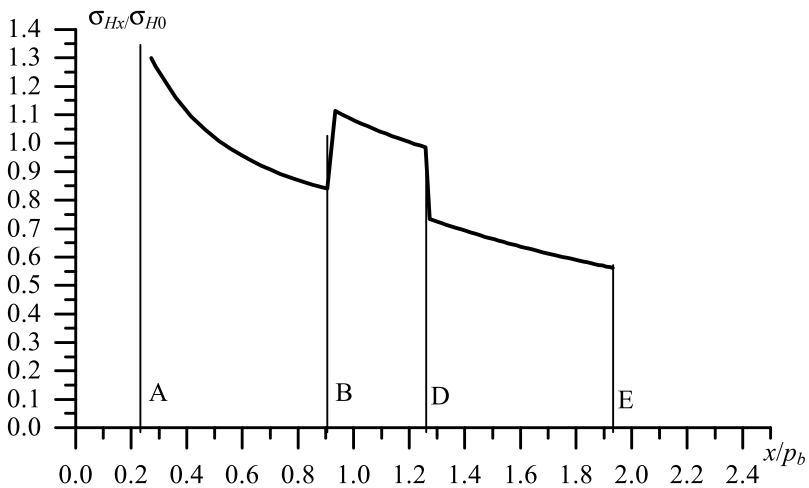

Fig. 1. Active stress on tooth flanks for the studied gear pair, while Δp_b=0

THE FINITE ELEMENT METHOD AND PLANE CONTACT PROBLEMS

In accordance with the described procedure for calculation of stresses on tooth flanks, it can be concluded that the solving of the defined task by the analytical methods is very complex and possible only with many assumptions. So, for this problem, as well as for almost all problems in Theory of elasticity, the numerical methods must be used. Instead of a system of differential (simple, partial and integral) equations, which are defined for solving some problem in Theory of elasticity, numerical methods form corresponding system of simple algebraic equations. By using computers, that system can be solved quickly and successfully. So, for studying the problem of contact stresses on tooth flanks, the Finite Element Method (FEM) is chosen. The Finite Element Method formulates the differential equations of balance of an elastic body. By taking into consideration the boundary conditions, the number of unknown quantities in these equations becomes smaller.

The Finite Element Method is based on the principles of the mechanics of continuum. The mechanical values, which are used in mechanics of continuum, are continuous functions of points' coordinates, so they are differentiable in accordance to coordinates. Material point of continuum presents an infinitely small particle of continuously disposed material, and its position is defined by geometrical point of space.

When variational principles, i.e. energy principles, are applied, variational equations take the form of integral equations. The variational principle of the maximum potential energy is the most frequently used principle in the Finite Element Method and it is described in [2], [3], [7].

Primary task in studying a deformable body by the FEM, is to choose a discrete model, which gives the most proper approximations for stress state, strain state and boundary conditions. The discreted model of continuum is obtained by its division on finite elements. Finite elements are mutually joined with common nodes. For modeling the whole construction, one can simultaneously use either the finite elements of only one type or the finite elements of a few different types. The basic differences among the finite elements of different types are differences in their shapes and interpolation functions. The special finite element types, which are used in solving engineering problems, are: the isoparametric finite elements (which use the same interpolation functions for the approximation of element geometry and for the approximation of basic unknown quantities in the field of a finite element) and the subparametric finite elements (the number of nodes for the approximation of element geometry is bigger than the number of nodes for the approximation of other functions).

The choice of an optimum discreted model for exact and economical calculation is directly dependent on the engineering experience and the knowledge about the characteristics of stress and deformation states for the studied construction. Basically, the discreted model selection is based on two major tasks: the selection of finite element type and the selection of necessary mesh density.

The method of deformation is used for modeling the contact problem. It is the most frequently used method in the FEM, and its basic task is to determine the functions of displacement. The displacements of any point of a finite element can be determined as the functions of displacement values for mesh nodes. It adopts that displacements, deformations and stresses are the continuous functions of coordinates of a finite element's points.

The finite element's equation, if the influence of temperature strains is ignored, gives the connection between the node displacements and node forces:

$$\{F\} = [K^e]\{S\} \tag{11}$$

where: $\{F\}$ – is a vector of external node forces for a finite element;
$\{S\}$ – is a vector of node desplacement for a finite element and
$[K^e]$ – is a stiffness matrix for a finite element.

The elements of these vectors and matrix depend on physical material characteristics, characteristics of chosen finite element type and choosed interpolating functions.

The construction equation is obtained by combining the equations for all finite elements that make the mesh of a body or a construction:

$$\{F\} = [K]\{S\} \tag{12}$$

where:

$$\{F\} = \begin{Bmatrix} \{\overline{F}^{(1)}\} \\ \{\overline{F}^{(2)}\} \\ \vdots \\ \{\overline{F}^{(n)}\} \end{Bmatrix}; \quad \{S\} = \begin{Bmatrix} \{\overline{S}^{(1)}\} \\ \{\overline{S}^{(2)}\} \\ \vdots \\ \{\overline{S}^{(n)}\} \end{Bmatrix} \tag{13}$$

In these vectors, n is the total node number, which is equal to the sum of the total node number in finite element mesh and the number of construction support. Every term of the vectors (13) is a vector with the number of elements equal to the number of degrees of freedom for finite element's nodes. Therefore, both of the vectors$\{F\}$ and $\{S\}$ have $p\times n$ elements (p – number of the degrees of freedom, n – number of nodes), and the stiffness matrix $[K]$ has nxn submatrices $\overline{K}_{rs}$.

Some node displacements are known in advance. They are defined by construction support and present the boundary conditions. When they are included in the construction equation, some equations become eliminated. Thereby, the total number of the degrees of freedom becomes smaller. The Finite Element Method takes into consideration the external forces (which come into action in some mesh nodes and present the loading of a construction) by including the force components, in the directions of coordinate axis, in the construction equation. If a construction is loaded with concentrated forces, there is a mesh node in each point struck by concentrated force's action. If a construction is loaded with continuously distributed load, this load is replaced with equivalent node forces. In that case, the knowledge of unit load along the line or along the loading surface is required.

The FEM treats a plane contact problem as a part of the general problem of bodies' movement in space and their interaction. The problem of this type ca be described by two bodies that get in contact because of the action of external forces. One of these bodies is defined as a contactor body, and the other as a target body. The procedure of choosing a contact body is simple when the Finite Element Method examines a stiff body and a deformable body in contact. But in the case of two deformable bodies in contact, the

FEM procedure is complex and requires excellent knowledge of the character of contact which is to be analysed. When the bodies are in contact, contact forces appear. These forces prevent mutual penetration of bodies and provoke deformations in contact areas. For the calculation of contact problems (nonlinear numerical problems) the Finite Element Method most frequently uses the Method of Lagrange multipliers.

The simplest FEM procedure for modeling contact conditions, is the one that uses finite elements in the form of nonlinear sticks with the assigned characteristic of pressure capacity and without stiffness in the case of tension ("gap" elements). Only the finite elements, for which exists the possibility that they will be in contact, are defined as the contact elements. These elements consist of nodes and lines that are located on the borders of bodies. In order to reduce the calculating time, only the part of the border that will possibly be in contact with the contactor body is defined as the contact surface of the target body. A contact element is defined as a connection between a node of a contactor body (contactor node) and a target segment of a target body.

During the calculation, the FEM procedure finds the nodes of the target surfaces and the corresponding contactor nodes that together make "the instantaneous contact elements". For each instantaneous contact element, the FEM procedure forms the stiffness matrix, the vector of displacement and the vector of internal forces in accordance with the analysis of the variation of contact forces potential. These matrices and vectors present the influence of an instantaneous contact finite element on the system, which consists of matrices for all finite elements of contact bodies (structural finite elements and contact finite elements).

THE FEM MODEL DEVELOPED FOR TOOTH FLANKS IN CONTACT

Because the uniformly distributed loading along the tooth face width is assumed in this paper, the contact of meshed tooth flanks presents a plane contact problem. Therefore, for modeling a tooth contact, which will be used for static analysis, the FEM can use the double-dimension (2D) isoparametric elements (figure 2). These elements are the special case of the 3D isoparametric finite elements.

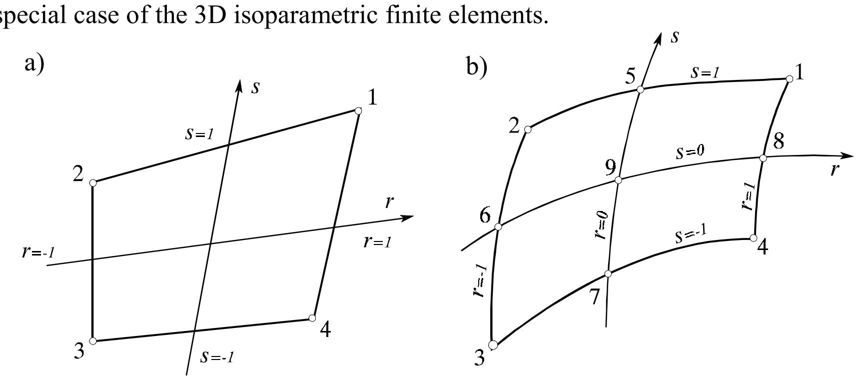

Fig. 2. 2D finite elements a) linear element; b) parabolic element

The FEM tooth models use the parabolic 2D degenerate isoparametric finite elements with 6 nodes and the case of plane state of deformation, which means that material is identically deforming in all planes parallel with *x,y* plane. The degenerate element, shown in figure 3, is obtained by overlapping of two corner nodes (1 and 4), so that the correction of the interpolation functions b_1 appears. Then, the function is equal to the sum of the interpolation functions for nodes 1 and 4:

$$b_1^* = b_1 + b_4 = \frac{1}{2}(1+r) \tag{14}$$

The same procedure is used for the degeneration of the parabolic 2D elements. Triangular 2D isoparametric element with 6 nodes is obtained by the degeneration of 2D element with 8 nodes.

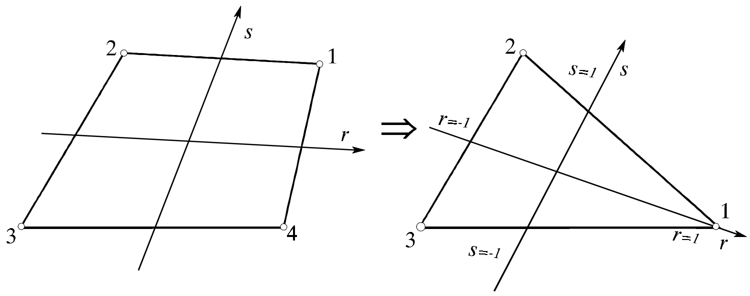

Fig. 3. The degeneration of 2D element

The tooth contact models are developed for the gear pair with characteristics given in table 1. Each of these models consists of three groups of elements: the group of elements that define the tooth of the pinion, the group of elements that define the tooth of the wheel and the group of contact elements. This paper's goal is to investigate the change of contact stress on meshed teeth, along the path of contact, so the most important thing is the determination of contact stress states for the contacts in characteristics points on the tooth profile (*A*, *B*, *C*, *D* and *E*). A computer program developed in [1] provides the radii that correspond to these points. For contact in each characteristic point, the special model in the FEM was made, so that the global coordinate system remained unchanged. That is especially suitable for monitoring the results.

Boundary conditions are defined as the limits of all translational and rotational displacements for those contour nodes in tooth models that present the connections with the gear body. A few concentrated forces along the external involute surfaces of meshed teeth simulate the loading, so that the total momentum that is loading the tooth mesh is equal to the momentum on the gear shaft. This choice for load simulation is the result of the investigation of different load simulations on the model of a tooth pair. The selected load simulation yields the distribution of stress and deformation state in the tooth contact, most similar to the distribution obtained from the experimental data acquired by the experimental photoelastic method, [3].

Tab.2 gives the view of node numbers and element numbers for the developed models and figure 4 presents discreted FEM models for all characteristic contact points (first contact point – *A*, point of passing from period with two tooth pairs in contact to single meshed tooth pair period – *B*, pitch point – *C*, point of passing from single meshed tooth pair period to period with two pairs in contact – *D*, last contact point – *E*). It is easy to notice previously mentioned groups of finite elements and the displacement limits for some contour nodes in mesh, that define the boundary conditions, as well as the concentrated forces that simulate the external loading.

Tab.2

Contact point	Node number for finite element mesh	Element number for finite element model
A	2006	941
B	1982	929
C	1344	621
D	1978	931
E	2030	953

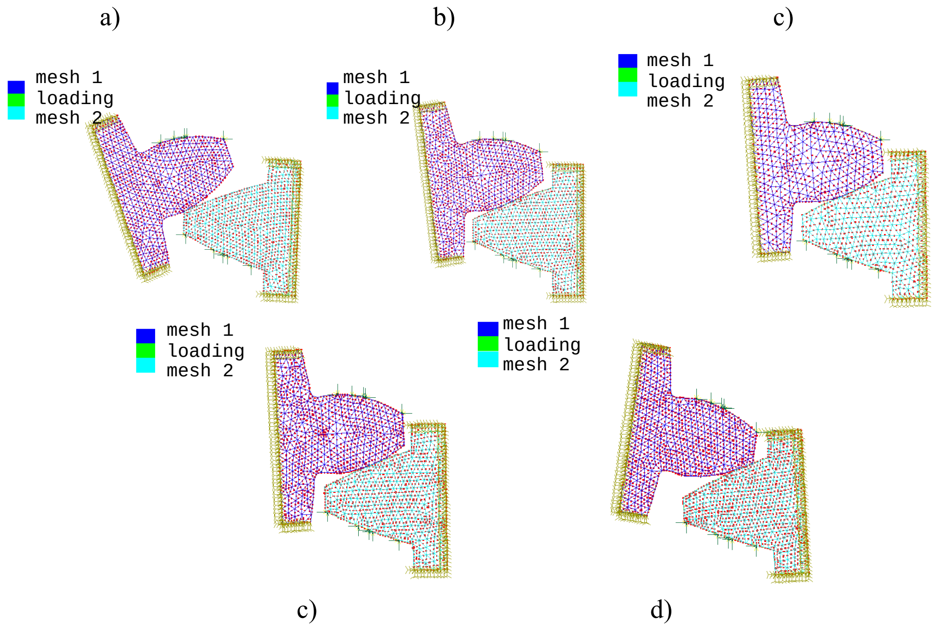

Fig. 4. Discreted models – finite element meshs for tooth pair contact:
a) in pointA; b) in point B; c) in point C; d) in point D; e) in point E

THE RESULTS OF THE DEVELOPED PROCEDURE

When the FEM calculation [6] is used on the tooth pairs contact models presented in figure 4, it gave the results given in figure 5. The results are presented through the Von Mises' equivalent stress fields. Reading of the maximum sresses in tooth contact, for different moments of the tooth pair meshing period, is very simple, so we can use these results for different type of analyses and comparisons.

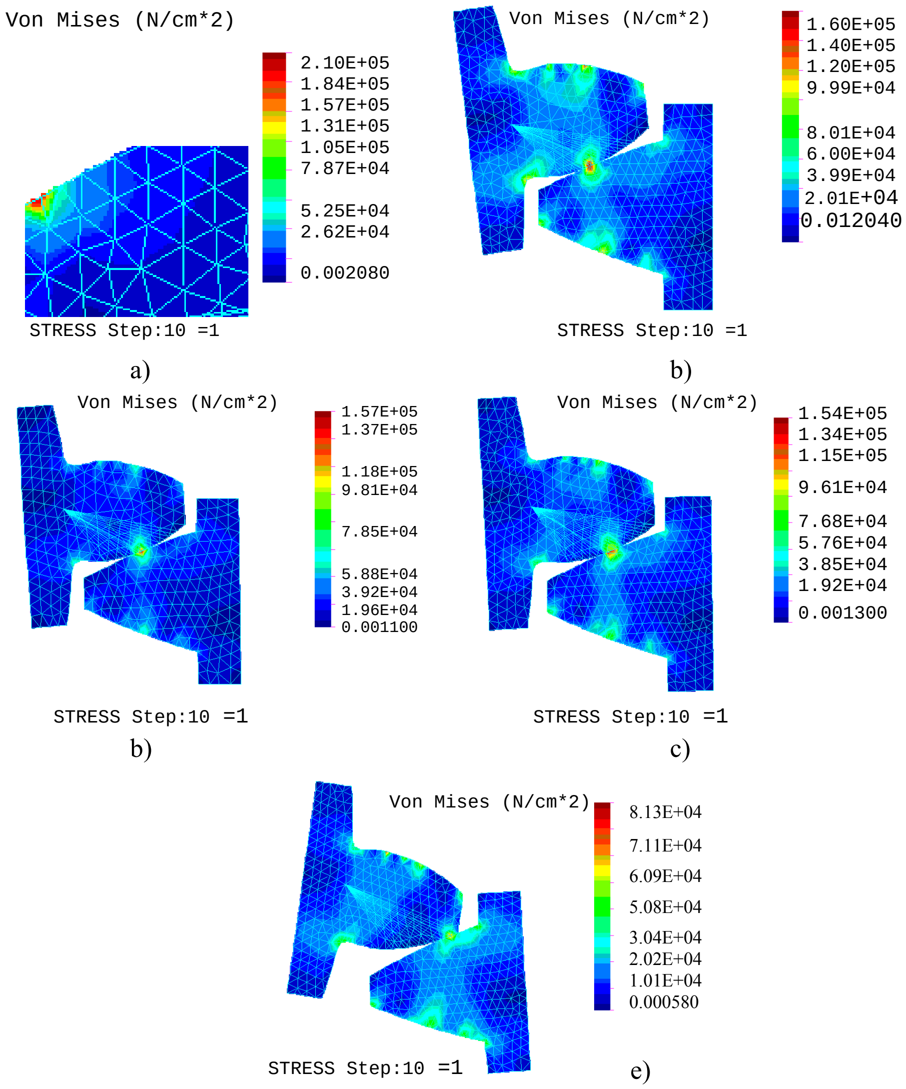

Fig. 6. Von Mises' equivalent stress fields for the tooth pair contact:
a) in pointA; b) in point B for the single meshed tooth pair period; c) in point C;
d) in point D for the single meshed tooth pair period; e) in point E

The model developed and described in this paper corresponds to an ideal gear, so these numerical results can be compared with the theoretical results shown in figure 1. Figure 6 presents a comparative diagram for the change of relative contact stress along the line of contact for a tooth pair in mesh. This diagram is a result of the theoretical analysis and the FEM calculation models given in figure 4. Because there are no results that correspond to every contact point on the line of contact, straight lines occur in the diagram obtained by the FEM calculation. Other results can be obtained by developing new models for the tooth pair in mesh, but this procedure would take much calculation time, and furthermore the results that would be obtained would not play important role in studying load distribution among simultaneously meshing tooth pairs from tooth flanks' strength aspect. The analysis of the comparative diagram leads to a conclusion that high level of coincidence exists between the results obtained by the application of the FEM and the ones obtained by the application of theoretical analysis. Thereby, these two methods certify one another.

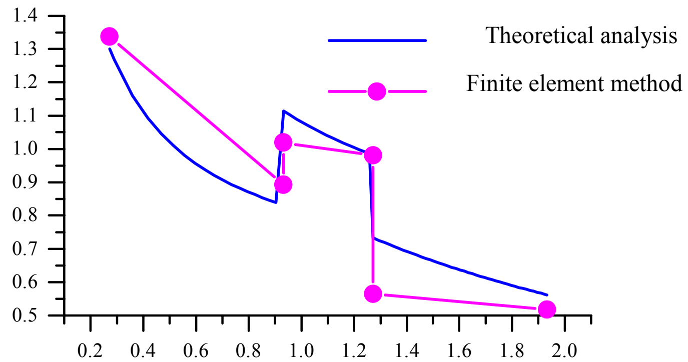

Fig. 6. The change of relative stress along the line of contact

CONCLUDING REMARKS

The results for equivalent stress fields in contact areas, which are presented in this paper and obtained by using the Finite Element Method, show that this numerical method is especially suitable for different analyses of contact stresses on meshed tooth flanks. Also, presented comparison between the theoretical results and the results for developed FEM models, confirms that all elements in the FEM models (finite element type, mesh density, boundary conditions and external loading simulation) are optimally determined for described contact problem.

The comparison between the shown equivalent contact stress fields for the tooth pair in contact and the existing experimental results, gives the conclusion that the Hertz's model of mutually pressed equivalent cylinders (that is used as the base for developing of the described FEM models) can be succesfully used as the basis in contemporary methods for the calculation of meshed tooth flanks' strength.

The procedure described in this paper calculates the maximum active stress and stress fields on tooth flanks during the meshing period, more precisely than any analitical method. So, these FEM models, especially with future improvements, can be used for calculation of all influence factors for the load distribution among meshing tooth pairs.

REFERENCES

1. Atanasovska I.: "Analiza raspodele opterećenja na spregnute parove zubaca sa osvrtom na čvrstoću boka zupca zupčanika", Magistarska teza, Kragujevac, 1999.
2. V. Nikolić: Mehanička analiza elemenata zupčastih prenosnika, monografija, Mašinski fakultet u Kragujevcu, 1999.
3. Kojić M., Slavković R., Živković M., Grujović N.: "Metod konačnih elemenata I, linearna analiza", monografija, Univerzitet u Kragujevcu, Mašinski fakultet, Kragujevac, 1998.
4. V. Nikolić: "Mašinski elementi, teorija i primeri", Kragujevac, 1995.
5. Lorenz H.: "Verbesserte Zahnfuβfestigkeit", Dortmund.
6. Rašković D.: "Teorija elastičnosti", Naučna knjiga, Beograd, 1985.
7. Hedrih (Stevanović) Katica, "Izabrana poglavlja teorije elastičnosti", Niš, 1988.
8. Software COSMOS Mexplorer, version 1-7 from SRAC.
9. V. Nikolić: "Istraživanje naponskog i deformacionog stanja glavnih strukturnih elemenata zupčastih prenosnika velikih snaga primenom metode konačnih elemenata", doktorska disertacija, 1989, Niš.
10. V. Nikolić: "Stress - state investigation of heavy loaded gear drives", MVM- nternational Journal for Vehicle Mechanics, Engines and Transportation Systems,Volume 19, Number 3, Septembar, 1993. str. 25...32.
11. V. Nikolić: "Analysis of the stress state of gear drive using numerical method", International Journal for Vehicle Mechanics, Engines and Transportation Systems, Volume 20, Number 4, December, 1994, str. 33... 39.
12. V. Nikolić, D. Dimitrijević: "On application of numerical methods in the studies of toothed gears", International Symposium "Machines and Mechanisms", Belgrade, September, 2-5. 1997.

ANALIZA KONTAKTNIH NAPONA NA MREŽI ZUBA DUŽ KONTAKTA ZA PAR SPREGNUTIH ZUBA

Vera Nikolić- Stanojević, Ivana (Atanasovska) Cvejić

Osnovni zadatak u ovom radu je analiza i odredjivanje oblika funkcije koja definiše promenu kontaktnih napona na ivici-boku zuba duž puta kontakta za par zuba. Odredjivanje tačke maksimalnog napona kontakta i njihov proračun je moguć ako je pravac kontakta zuba predvidljiv. Korišćena je numerička metoda – metoda konačnih elemenata za modeliranje kontakta zuba. Ovaj rad daje detaljan opis razvoja procedure. Uporedjeni su analitički i numerički rezultati.