UDK: 621.83:519.673
COSATI: 13-09, 12-01, 20-12

Finite Element Model for Stress Analysis and Nonlinear Contact Analysis of Helical Gears

Ivana Atanasovska, PhD (Eng)[1)]
Vera Nikolić-Stanojlović, PhD (Eng)[2)]
Dejan Dimitrijević, PhD (Eng)[3)]
Dejan Momčilović, MSc (Eng)[4)]

One of the main goals during load calculations and load capacity check for gears is determination of deformation and stress state in teeth contact zones and teeth fillets. This paper describes development of the finite element model for simultaneously monitoring the deformation and stress state of teeth flanks, teeth fillets and parts of helical gears during the tooth pair meshing period. The paper also describes the Finite Element Method simulation of contact conditions for helical gears teeth with an involute profile.

A suitable analysis is performed in order to select a meshed gears model which is sufficiently economic and in same tame sufficiently geometrically accurate. The special algorithm for the tooth involute profile drawing is developed and built in a currently available software 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 particular helical gear pair. The obtained numerical results are suitable for tracking deformation and stress variables during the tooth pair meshing period.

Key words: **gear, helical gear, strain, contact strain, load distribution, stress calculation, 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 spheres, two parallel cylinders, cylinders on a flat plane, [1], gear teeth, roller bearings, [2] 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 behaviour 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 deformations in the contact zone is crucial for determination of stress throughout the model, i.e. in teeth roots and bodies.

Experimental determination of stress and strain state in helical gear teeth with real contact conditions is associated with many limitations. The Finite Element Method (FEM) is the most appropriate numerical method that can solve this nonlinear problem, i.e. the Finite Element Analysis (FEA) gives the possibilities for developing appropriate models in accordance with needs in modern research of nonlinear tasks. Ulaga, S. et al analyse contact problems of gears using Overhauser splines [3], but their FEM models calculate only contact zone stresses. Dimitrijevic, D. et al use FEA for the dynamic analysis of the stress and strain state of the spur gear [4], and Pasta, A. and Virzı´Mariotti, G. described a 2D FEM model developed for the analysis of a spur gear with a corrected profile [5]. Also, some authors as Jovanovic, M. et.al, [6] analysed approximate contact models of the rolling supports using FEA software.

In this paper a particular real gear pair with a high value of transmission ratio is analyzed. The main characteristics of the gear pair are: number of teeth z_1=20, z_2=96; standard tooth involute profile [7], addendum modification coefficients x_1=0.3, x_2=0.2; face width b=175 mm; module m_n=24; pressure angle α_n=20°; helix angle β=15°; rotational wheel speed n_2=4.1596 min^{-1}; wheel torque T_2=1264.4 KN·m, material: steel with E = 206000 N/mm^2; ν =0.3, pinion teeth inclination – right, wheel teeth inclination – left. For defined geometry characteristics and torque, the normal nominal load that this gear pair transmits, [2], is F_{bn}= 1168.0354 KN.

The theoretical, numerical, [8, 9] and experimental results, [10, 14] are used for the verification of the developed gear FEM model. The analyzed gear pair has a high value of the transmission ratio (u=4.8) i.e., high ratio of meshed gears dimensions. During FEM modelling it gives more possibilities for detection of problems and bugs than gear pairs with a transmission ratio value near 1.

Characteristics of helical gears geometry

An involute of a circle is a curve that is traced by a point on a taut cord unwinding from a circle, which is called a base circle, [11, 12]. The involute is a form of spiral, the

1) Institute Kirilo Savić, Vojvode Stepe 51, 11000 Belgrade, SERBIA
2) Faculty of Mechanical Engineering, University of Kragujevac, Sestre Janjić 6, 34000 Kragujevac, SERBIA
3) SIEMENS AG – Industrial Solutions and Services, Werner von Siemens, Soprien, 91052 Erlangen, GERMANY
4) Institute for testing of Materials (IMS), Bulevar Vovode Mišića 43, 11000 Belgrade, SERBIA

curvature of which becomes straighter as it is drawn from a base circle and becomes a straight line at infinity. In an involute gear, the side surfaces of the teeth are involutes of a circle. Fig.1 shows involute definitions in polar coordinates:

$$inv\,\alpha_x = tg\,\alpha_x - \alpha_x\,;\ \alpha_x \text{ in rad} \tag{1}$$

The involute radii in any point ρ_x are defined as:

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

The angle of action α_x is the angle with the vertex at the gear centre, one leg on the point where mating teeth first make contact, the other leg on the point where they disengage:

$$\cos\alpha_x = \frac{r_b}{r_x} \tag{3}$$

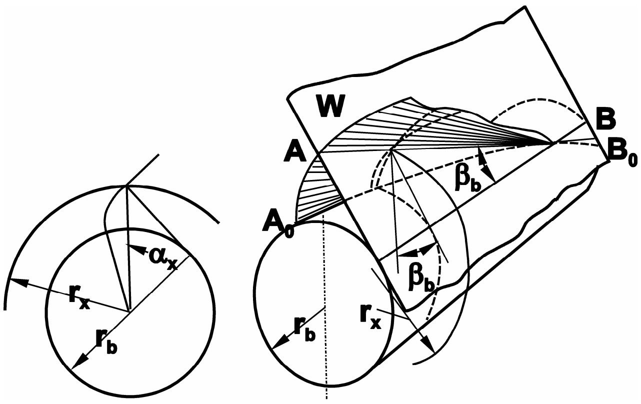


Figure 1. Helical involute surface

For helical gears, a helical involute surface of a cylinder is a surface that is traced by a line AB on a taut cord unwinding from a cylinder, which is called a base cylinder, Fig.1. The line AB is a strength line that lies at the tangent plane W and makes an angle with the generating line of the base cylinder – helix angle β_b, Fig.1. During the line AB tracing a helical involute surface, the intersection the of line AB and the base circle cylinder is the helix A_0B_0 with helix angle β_b. The cross sections of the helical involute tooth surface with the planes normal to the base cylinder axis are involutes with the same base circle. Their start points lie on the helix A_0B_0. Helical gears come in pairs where the helix angle of one is the negative of the helix angle of the other; such a pair might also be referred to as having a right handed helix and a left handed helix of equal angles.

The step of helical involute surface could be calculated with expression:

$$T = 2\pi \cdot r_b \cdot ctg\,\beta_b \tag{4}$$

and the angle β of helix that is made by the intersection of the helix involute surface and the pitch point could be calculated through the following expressions:

$$2\pi \cdot r \cdot ctg\,\beta = T\ , \tag{5}$$

$$tg\,\beta = \frac{r}{r_b} \cdot tg\,\beta_b \tag{6}$$

Fig.2 shows intermeshing of two helix involute surfaces corresponding to cylinders with parallel centrelines and the base circles r_{b1} and r_{b2}. The surface of action for involute, parallel axis gears with helical teeth is plane of action W that is a tangent to the base cylinders.

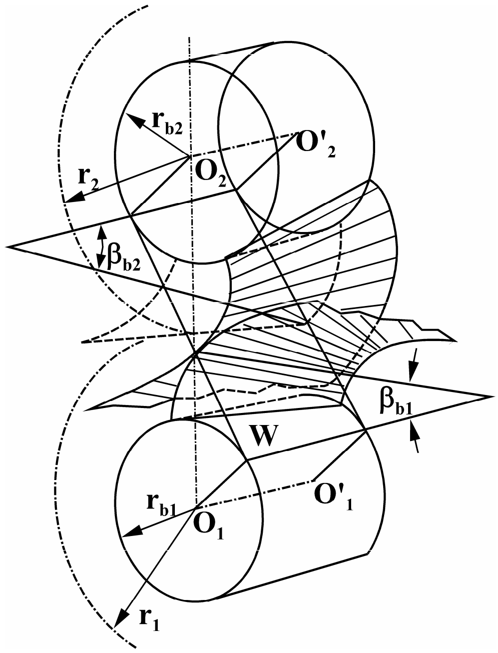


Figure 2. Meshing of helical involute surfaces

Lines of contact are not parallel to gears axis. Therefore, tooth pairs come in to the mesh gradually. The contact starts at the pinion tooth root on one face of gears and propagates through the face width towards the pinion tooth top on another face of gears. This leads to continuously changing the contact line length and makes a load distribution calculation very complex.

The zone of action (contact zone) for involute parallel-axis gears with helical teeth is the rectangular area in the plane of action bounded by the length of action and the effective face width, Fig.3. The expressions for the determination of length of action exist in machine elements books and standards, [2, 11].

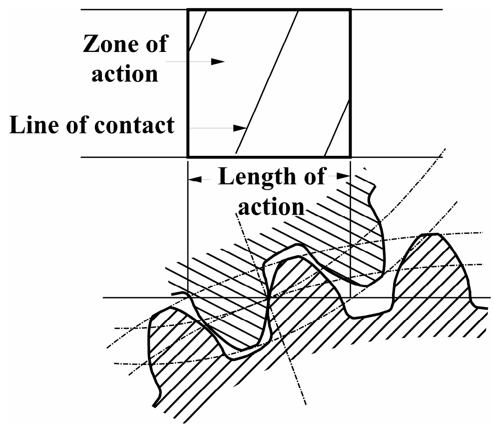


Figure 3. Zone of action for involute parallel-axis gears with helical teeth

Numerical experiment procedure

Research for identifying the stress state is a more complex task for the involute cylindrical gears with helical teeth than the same task for the involute straight spur gears. The load distribution in the helical gears mesh could be solved only by resolving the load distribution between simultaneously meshed teeth pairs and the load distribution along each of teeth pair contact line, at the same time. Thus, calculation of helical gear teeth stress and deformation could be made only by a three-dimensional finite element model. For developing such model, authors defined the procedure of numerical experiment, Fig.4.

The first step in the procedure is developing a plane finite element model for the helical gear pair contact. The geometry of a plane finite element model is a teeth profile geometry in the face plane. In the second step, the developed plane contact model has to rotate in characteristic contact positions during a teeth pair contact period. Then, the next step is developing three-dimensional solid finite element models for selected contact line positions by extruding rotated plane models through the helix defined in Chapter 2. The finite element analysis follows then and gives numerical results that could be presented by diagrams.

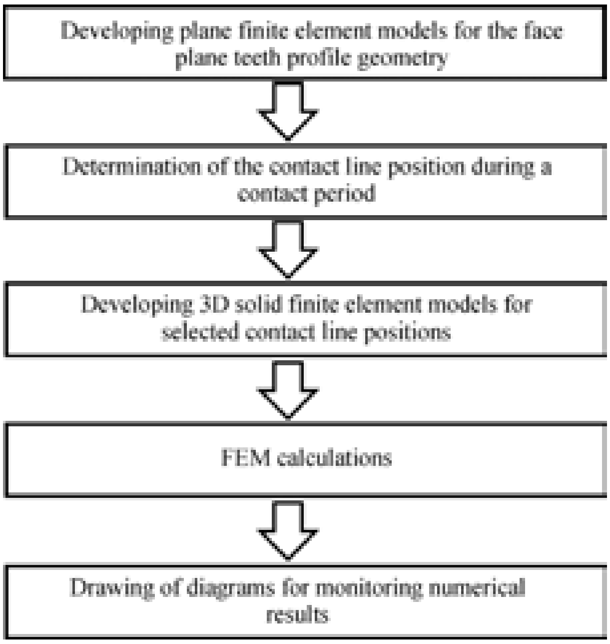


Figure 4. The procedure of the numerical experiment

The finite element contact model for a gear teeth contact simulation

The references [13, 16] discuss the capabilities and procedures for using finite elements to model contact problems, but do not give details needed for creating and solving such a model. In the process of problem solving, which must include some form of contact behaviour, numerous decisions have to be made. In many cases, it may be possible to use constraints to capture some of the contact problem behaviours and treat a contact problem as a linear condition. However, it is not often sufficient and a contact must be modelled with contact finite elements and treated as a nonlinear problem. Hence, during the development of the gear finite element analysis the authors solved a lot of problems in order to obtain a sufficiently economic and precise model in accordance with all defined targets.

Contact problems are highly nonlinear and require significant computer resources to solve. It is important to understand the physics of the problem and take the time to set up a model to run as efficiently as possible. The Finite Element Analysis (FEA) 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 can 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 rigid-to-flexible body contact. But in the case of a flexible-to-flexible body contact, the FEA procedure is complex and requires excellent knowledge of the character of the contact to be analyzed. When the bodies are in contact, contact forces appear. These forces prevent mutual penetration of bodies and provoke deformations in the contact areas.

For the purpose of the meshed gear modeling the authors choose the ANSYS FEA software, which supports three types of contact models: point-to-point, point-to-surface, and surface-to-surface. Each type of models uses a different set of contact elements types and is appropriate for specific types of problems. The point-to-surface contact model has been chosen for the contact modeling of gear teeth in the mesh.

The point-to-surface contact model is represented by following the positions of points on one surface (the contact surface) relative to the lines or the areas of another surface (the target surface). The program uses contact elements to track the relative positions of two surfaces. In 2D models, point-to-surface contact elements are triangles and 3D model point-to-surface contact elements are pyramids. Both of these elements have a base made of the nodes from the target surface and a remaining vertex made of the node from the contact surface. These elements are not compatible with higher-order solid elements because the mid-side nodes of the quadratic elements are not used for the contact faces. Therefore, 2D four-nodes structural solid elements and 3D eight-nodes structural solid elements have been used for developing gear finite element models. The detailed description of the used finite element types could be found in references [15, 16]. First, the areas where a contact might occur during the deformation of the model have been identified and then contact elements are defined. For the most efficient solution smaller localized contact zones have been defined, thereby the chosen zones are adequate to capture all necessary contacts.

For point-to-surface contact elements, there are two different ways to approach the contact – the penalty method alone or a LaGrange multiplier added to the penalty method. The penalty method uses a contact “spring” to establish a relationship between two contact surfaces. The spring stiffness is called the contact stiffness. The penalty method modifies the present stiffness matrix by adding large terms to prevent too much penetration. The LaGrange method is an iterative series of penalty methods. The contact tractions (pressure and frictional stresses) are augmented during equilibrium iterations so that the final penetration is smaller than the allowable tolerance. Compared to the penalty method, the LaGrange method usually leads to better conditioning and is less sensitive to the magnitude of the contact stiffness. However, in some analyses, the augmented LaGrange method may require additional iterations, especially if the deformed mesh becomes too distorted.

Some experienced analysts feel that „better“ contact performance and results may be found when the penalty formulation alone is used [15]. During the helical gear finite element model developing, the authors have chosen the penalty method alone for teeth contact, and have obtained an enough precise and, at the same time, economic model. For the penalty method (mentioned above), the proper choice of the contact stiffness was critical. It should be large enough that it reasonably restrains the model from over-penetration, yet it should not be so large that it causes ill-conditioning.

One more decision is to select an asymmetric or symmetric contact. The asymmetric contact is defined as having all contact elements on one surface and all target elements on the other surface. This is usually the most efficient way to model the surface-to-surface contact. However, under some circumstances the asymmetric contact does not perform satisfactorily, i.e. each surface to be both a target and a contact surface. That means generation of two sets of contact pairs between the contacting surfaces. This is known as the symmetric contact. Obviously, the symmetric contact is less efficient than the asymmetric contact. However, many analyses will require its use (typically to reduce penetration). Specific situations that require the symmetric contact include models where the distinction between the contact and target surfaces is not clear and when both surfaces have very coarse meshes. The symmetric contact algorithm enforces the contact constraint conditions at more surface locations than the asymmetric contact algorithm. In accordance with this, the symmetric contact is chosen for the simulation of contact conditions on the gear teeth.

Modelling of face plane teeth profile geometry for helical gears in the mesh

The required accuracy of the FEA calculation of stress and deformation states of meshed gears is very high. It is greatly conditioned by the accuracy of the model geometry. Because of that, the special algorithm TOOTH for tooth involute profile drawing is programmed at the base of equations (1) to (6) and built in the present FEM software to assure very precise drawing (up to 1 μm) of the contact geometry of real flanks. The capability for entering a desired accuracy is also built in the algorithm. For gear ring dimensions the recommendations from literature [2] was used. Fig.5 shows a drown helical involute teeth at the face plane section for a particular helical gear pair described above modelled by the programmed algorithm TOOTH.

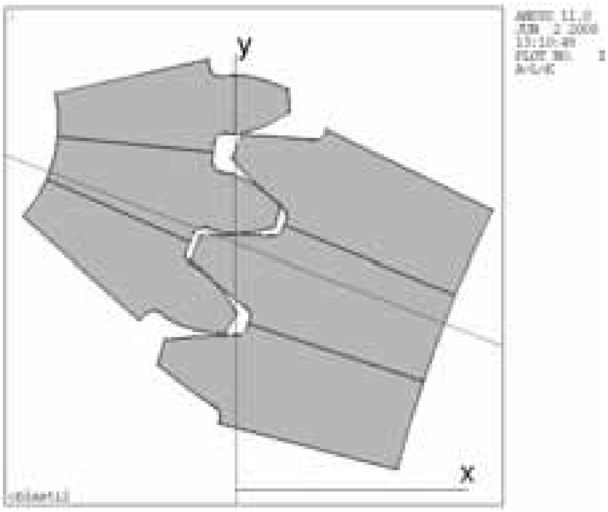


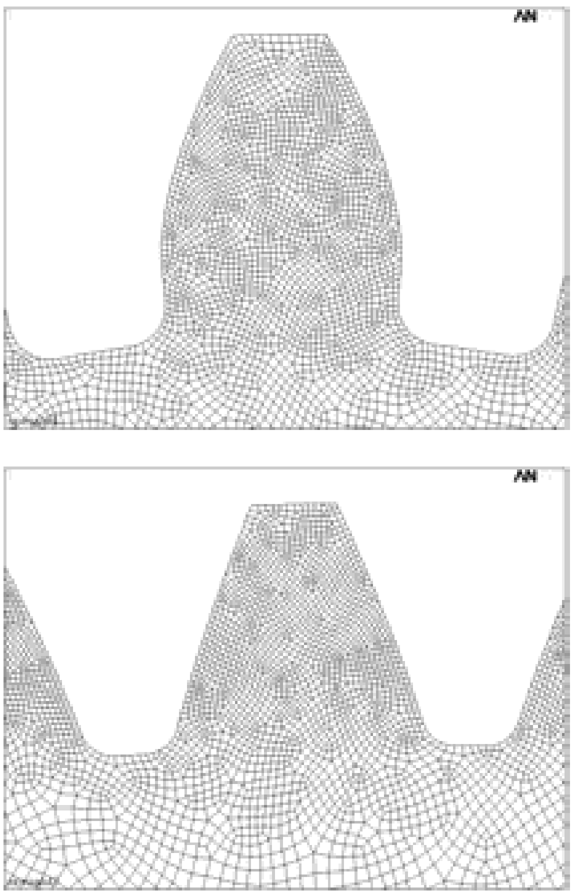
Figure 5. Physical model for a gear pair

One of the main steps during the helical gear pair modelling is selection of a simplified physical model for the simulation of deformation and stress state at the contact zones of the meshed teeth, as well as at other zones of the meshed teeth and the gear bodies. With the analysis of different models, and with the emphasis on the balance between the accuracy and FEA calculation time, the physical models of gear segments with three teeth were adopted, Fig.5. In order to simulate load more easily and examine the results, the models are turn in such a way that the path of action (line of action) of the meshed teeth matches the *y* axis of the global Cartesian coordinate system (*x,y,z*).

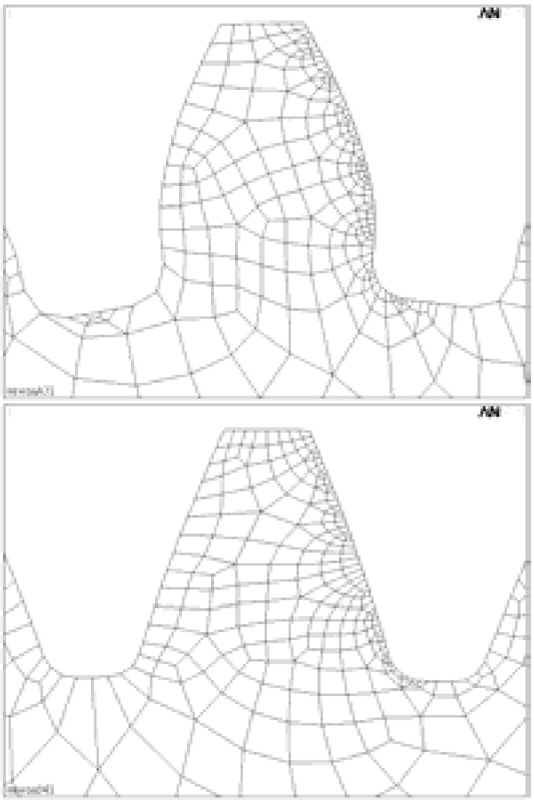
Development and examination of the plane meshed gear FEM models

A spur gear pair with the strength teeth and the teeth profiles the same as the helical teeth profiles at the face plane section was used for the development and examination of plane meshed gear finite element models. The 2D FEM gear pair models use finite element types described in Part 2 of this paper: The 2D is a parametric structural solid element is defined by four points– for the 2D gear modeling and the 2D point-to-surface contact element – for the “symmetrical” contact modelling. The used FEM software gives an option for friction contact modelling, but asked for an exact friction coefficient, which is unknown in all real tooth contacts So, an assumption in the first step of the investigation described in this paper is neglecting friction.

Two different finite element meshes of involute gear teeth are taken into consideration. Fig.6a represents 2D finite element meshes for pinion teeth and wheel teeth with very fine finite element meshes. And Fig.6b represents corresponding finite element meshes with coarse meshes and same finite element division at contact surfaces.



a)



b)

Figure 6. 2D finite element meshes for the pinion teeth and the wheel teeth
a) fine mesh; b) coarse mesh

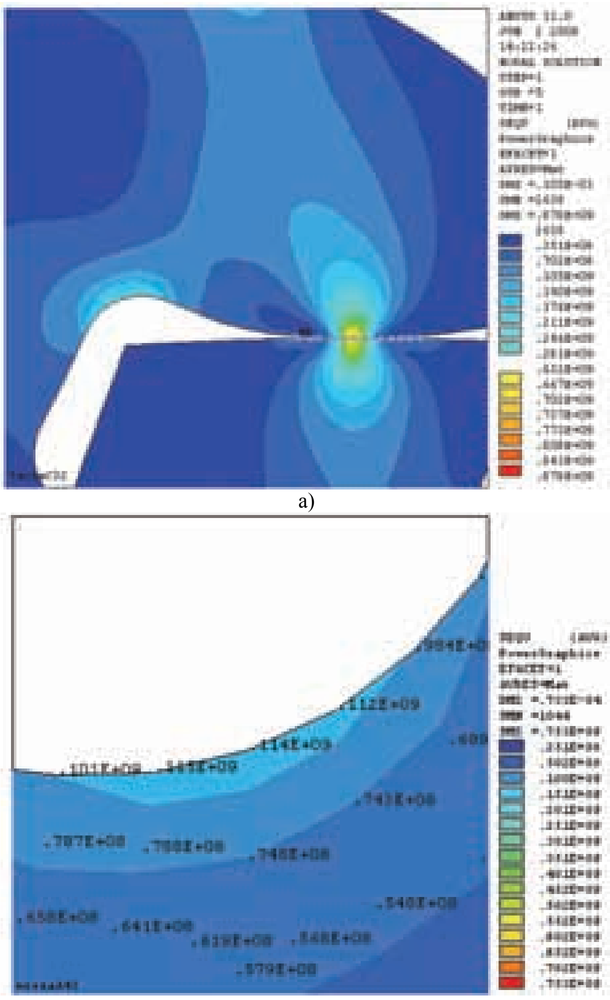
Having positioned the developed finite element model onto the characteristic positions of line of contact, the boundary conditions and the external loads are defined. The models enable the calculation of all important variables (deformation, stiffness, load distribution, flank stress state, fillet stress state), as well as the determination of variable changes along the path of the contact for a tooth pair. During the finite element calculation, the external load was simulated through one load step and ten sub steps, which gives a satisfactory fast solution convergence of a nonlinear contact problem. The boundary conditions on any 2D FEM model are defined by displacement constraints at a direction normal to the surfaces which separate the modelled gear segment from the rest of the gear body. The normal forces that this gear pair transmits are used as external loads in the gear FEM models and acts on the teeth flanks opposite the contact flanks at the direction of line of action (the same as the path of the contact direction for involute cylindrical gears).

Analysis and verification of the developed models with a fine mesh

For the analysis and verification of the developed FEM models of a chosen gear pair, four times less load compared with real loads (defined by the torque on wheel T_2) is used. During the development of spur gear FEM models, an assumption was used about uniform load distribution along the teeth face width and nominal load was defined as unit load q=1.6667 KN/mm. Low load values enable easier result analysis.

The FEM calculations gave deformation and stress values for defined constraints and external loads. The analysed variables are equivalent Von Mises Stresses, the definition of which is given in reference [16], and normal stresses for the Cartesian coordinate system, oriented as Fig.5 shows. In the course of the verification of the developed 2D FEM model and the calculation procedure of deformation and stress state, various aspects were used:

- The FEM calculation on 2D models for the tooth contact in the pitch point C, gives a maximum value of equivalent Von Mises contact stress $\sigma_{H\max} = 778\cdot 10^6$ N/m^2 and a maximum value of normal stress (y-axis direction) $\sigma_{y\max} = -849\cdot 10^6$ N/m^2. On the other side, the analytical calculation, according to [17] and without taking into account the load factors, gives an expected value of the Hertz contact stress in the pitch point σ_0=806.6·10^6 N/m^2. This confirms excellent match numerical and analytical results for contact stresses. Fig.7a shows a detailed zoom the contact FEA stresses with the stress distribution zones concurring with the experiment results for the contact teeth stresses obtained by the photoelasticity method [14].
- The standard ISO calculation of maximum stresses at tooth fillets, [17] without taking into account load factors, gives the values of $\sigma_{F1\text{VonMises}} = 226\cdot 10^6$ N/m^2 for the pinion and $\sigma_{F2\text{VonMises}} = 198.7\cdot 10^6$ N/m^2 for the wheel. The numerical results deviation from these analytical results is in average 10%. However, in this comparison, it is necessary to emphasize that the ISO standard for stresses at teeth fillets accepts bending stresses only, as at the same time neglects compression stresses and deformation of gear bodies. On the other side, the developed FEM procedure calculates stresses at tooth fillets based on total teeth strains.



a)

b)

Figure 7. Numerical results verifications
a) contact FEM stresses; b) FEM tooth fillet stresses

- Another verification of the FEM calculation results for teeth fillet stresses is obtained by their comparing with the experimental results of Nikolić and Bogdanović, [10], obtained by the strain gauges method for the wheel. The set of experimental results corresponds to the nominal load F_{bn}=25 KN, on a model of the reduced length of gears (to b=35 mm). The maximum stress (total normal stress) experimentally obtained for this case is $\sigma_{n\max}=100.52\cdot10^6$ N/m^2. For a successful check of the developed FEM models, the calculation with an external load of F_{bn}=25 KN and plane stress state with element thickness of 30 mm is performed. The obtained value in the tooth fillet is $\sigma_{\mathrm{VonMises}}=115\cdot10^6$ N/m^2.

These side-by-side analyses confirm a successful choice of FEM models.

Analysis and verification of the plane FEM models with a coarse mesh

Although the developed plane FEM models of meshed teeth with q fine mesh display good results, they are inadequate from the point of consuming the calculation time. These models could be replaced with appropriate models with a coarse mesh of elements, for more economic time management.

In order to ensure precision in the determination of values needed for tracking the load distribution and deformation and the stress state of the teeth flanks in contact zone and the teeth fillets in the critical (tensile) zones, the mesh of elements is preserved with the same density in these zones, but is strongly reduced in other zones of models, Fig.6b.

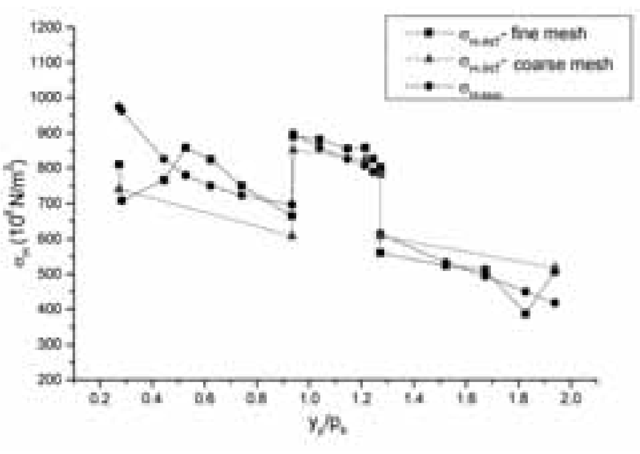


Figure 8. Comparative diagrams for optimal mesh density selection

The comparative analysis of FEM models with a fine mesh and a coarse mesh, as a starting point for a selection of an optimum mesh size, is best described by tracking the diagrams of changes of contact stresses (equivalent stress $\sigma_{\mathrm{VonMises}}$) during the contact period for a teeth pair (y_p/p_b – relative contact point distance from the start of the path of contact). Fig.8 shows the comparative diagrams for the equivalent Von Mises stress σ_H in the contact zone of meshed teeth obtained by: FEA of meshed gear teeth with fine meshes, FEA of meshed gear teeth with coarse meshes and theoretical analysis of gears teeth contact stresses with the Hertz theory described in [18, 19], and serves for a choice of FEM models with course meshes as optimal.

3D solid finite element models for a selected line of contact positions

Determination of the line of contact positions at the zone of action essential for load distribution in a mesh

This step of the developed procedure uses the expressions for the determination of the essential line of contact positions that will be modelled within a numerical experiment. The relevant positions are: the first point of the period with two teeth pairs in contact (when the previous teeth pair is out coming of the mesh), the first point of the period with three teeth pairs in contact (when the next teeth pair is incoming in to the mesh), the position during the period with two teeth pairs in contact when a sum of lengths of the meshed teeth pair lines of contact has the maximum value and the position during the period with three teeth pairs in contact when that sum has the minimum value. Some more relevant contact line positions could be defined. It is determined by requirements of a numerical calculation and available calculation time.

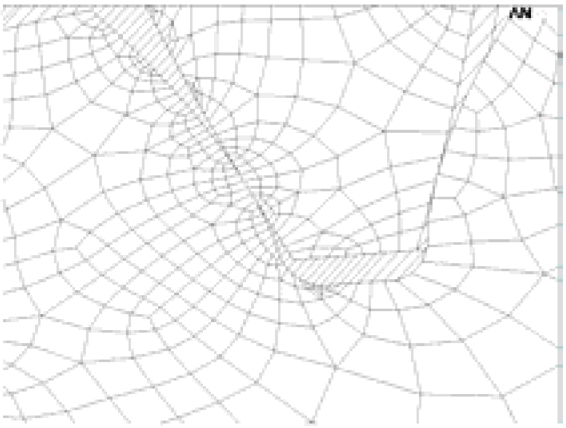
3D solid finite element models for selected line of contact positions

In this sub-step the procedure implies rotations of the developed plane finite element models for the teeth profile at the face surface section to the defined relevant line of contact positions.

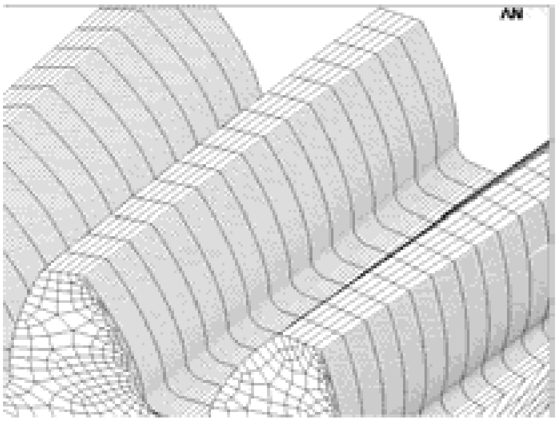
Then, the extractions of the rotated plane models through helix with the helix angle β_b, give appropriate 3D solid finite element models for numerical calculations. The modelling of helical teeth flanks is based on the following: the start points of all involutes that make the helical teeth flank surface lie on the helix with the helix angle β_b and all these involutes have the same base circle (with the radius r_b), as described in in first part of this paper in detail. The finite element types chosen for the 3D FEM helical gear pair model developing are: the 3D isoparametric structural solid element defined by eight points– for the 3D gear modelling; and the 3D point to surface contact element – for the tooth contact modelling.

Fifteen sections (segments) along the gear face width, which were used during extractions in the 3D FEM models, give a possibility for a very precise determination of stress state and load distribution along the lines of contact. The contact nonlinearity in the tooth contact zones was modelled by symmetric contact element groups, as at 2D finite element models. Fig.9a represents a detailed zoom of the 3D finite element model of the chosen helical gear pair for one of the chosen line of contact positions and Fig.9b represents a finite element mesh of the pinion.

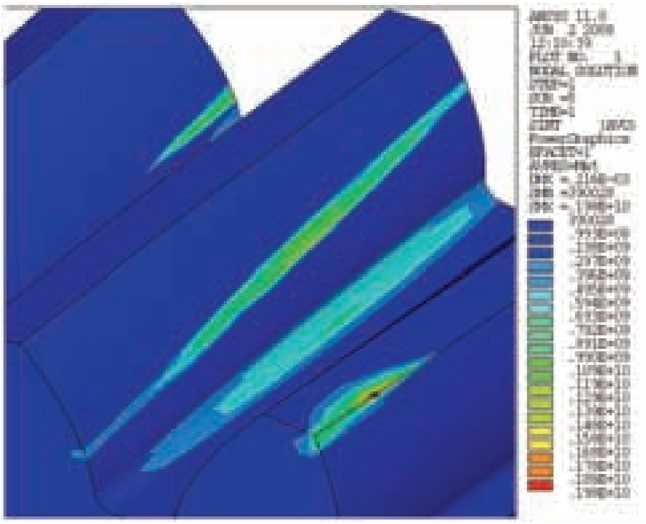
The boundary conditions on the developed 3D FEM models are defined by the displacement constraints at the direction normal to the surfaces which separate the modelled gear segment from the rest of the gear body. Also, displacement constraints at the direction normal to the transverse plane (z-axis of 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 involute cylindrical gears and the y-axis of the Global Cartesian Coordinate System) by few 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 pair transmits [17], F_{bn}= 1168.0354 KN.



a)



b)



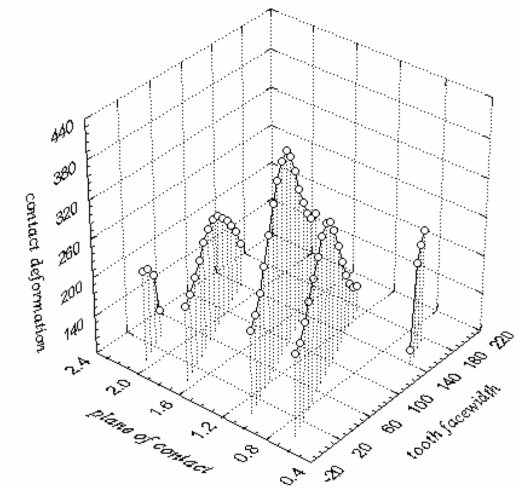
c)

Figure 9. The Helical gear FEM model and equivalent stresses for the pinion gear

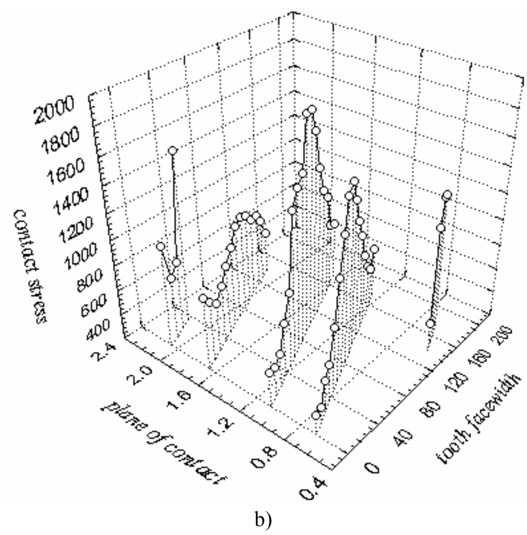
Numerical results analysis and conclusions

The 3D diagrams are used for tracking the variation of gear teeth deformations, gear teeth stiffness, mesh stiffness, load distribution in the mesh and gear stress state during a meshing period of helical involute cylindrical gears. These diagrams show the variation of the variables at different relevant contact line positions during the meshing period and along every single contact line, simultaneously, enabling identification of maximum stresses competent for gears load capacity calculations. Fig.10 shows the diagrams for: total teeth pair deformation at contact (Fig.10,a), equivalent Von Misses contact stresses at the pinion teeth flanks (Fig.10,b), equivalent Von Misses stresses at the pinion teeth fillets (roots) (Fig.10,c) and equivalent Von Misses stresses at the wheel teeth fillets (roots) (Fig.10,d).

The Finite Element Method calculations gives also the numerical values of the maximum contact stress and the maximum teeth fillet stresses referent for gear load capacity calculations, [17]. The maximum contact stress exists at the contact line position that lies at the period with three teeth pairs in contact and is characterized with a minimum sum of the lines of contact lengths: $\sigma_{H\max} = 2080 \cdot 10^6$ N/m^2. This value can be compared with an appropriate value of the maximum contact stress for the spur gear pair with the same characteristics, [9], $\sigma_{H\max}$ =2320·10^6 N/m^2. The maximum pinion teeth fillet stress is $\sigma_{F1\max}$ =704·10^6 N/m^2, and the maximum wheel teeth fillet stress is $\sigma_{F2\max}$ =672·10^6 N/m^2. These values correspond to the same line of contact position as for the maximum contact stress and can be compared with appropriate values for the spur gear pair with the same characteristics, [9]: $\sigma_{F1\max}$ =867·10^6 N/m^2 and σ_{F_2} =823·10^6 N/m^2. The comparative analysis shows significant increases of load capacity of the gear pair with the helical involute teeth with respect to the gear pair with the strength involute teeth.



a)



b)

Figure 10. Diagrams for result tracking
a) total teeth pair deformation at contact (μm); b) equivalent Von Misses contact stresses at the pinion teeth flanks (10^6 N/m^2); c) equivalent Von Misses stresses at the pinion teeth fillets (roots) (10^6 N/m^2); d) equivalent Von Misses stresses at the wheel teeth fillets (roots) (10^6 N/m^2)

The detailed analyses of the numerical experiments on the developed FEM models for the real helical gear pair lead to the verification of models and procedures. The developed models enable a study of a real contact between the involute surfaces of the meshed teeth flanks and a simultaneous calculation of the total teeth deformation and both the flank and the fillet stress state.

These calculations do not include any theoretical formulae for the determination of deformation, stiffness and stress states of meshing teeth contact zones. The described numerical experiment procedure and the analysed results are a new viewpoint in gear capacity calculation. Integrating new methods in to standard calculations of helical gears could be very useful for engineers and scientists who investigate involute helical gears.

Acknowledgment

Some parts of this research were supported by the Ministry of Sciences of the Republic Serbia through the University of Belgrade, Faculty of Mechanical Engineering, Grant TR14033: “Research of methods and approaches for increasing service life and reliability of mechanical systems”.

References

[1] FISCHER-CRIPPS,A.: *Introduction to Contact Mechanics*, Springer, 2007.

[2] NIEMANN,G., WINTER,H.: *Mashinenelemente*, Band 2, Springer, 2002.

[3] ULAGA,S. et al.: *Contact problems of gears using Overhauser splines, International Journal of Mechanical Science*, 41, Elsevier Publishing, 1999. pp.385-395.

[4] DIMITRIJEVIĆ,D. et al.: *Dynamic analysis of the stress and strain state of the spur gear*, Scientific Technical Review, 2007, Vol. LVII, No. 3-4.

[5] PASTA,A., VIRZI´MARIOTTI,G.: *Finite element method analysis of a spur gear with a corrected profile*, Journal of Strain Analysis for Engineering Design, 2007, Vol.42, pp.281-292.

[6] JOVANOVIĆ,M. et.al.: A*pproximate contact models of the rolling supports*, FACTA UNIVERSITATIS, Series: Mechanical Engineering, 2004, Vol.2, No.1, pp.69-82.

[7] ISO 53, *Cylindrical gears for general and heavy engineering — Standard basic rack tooth profile*, 1998.

[8] NIKOLIĆ,V., CVEJIĆ-ATANASOVSKA,I.: *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, 2003, Vol.3, No.15, pp.1055-1066.

[9] ATANASOVSKA,I., NIKOLIĆ,V.: *3D spur gear FEM model for the numerical calculation of face load factor*, The Scientific Journal: Facta Universitatis, Series: Mechanics, Automatic Control and Robotics, 2007, Vol.6, No.1, pp.131...143.

[10] NIKOLIĆ,V., BOGDANOVIĆ,G.: *Determination of stress state of gear drive at static and dynamic loadings,* MVM-International Journal for Vehicle Mechanics, Engines and Transportation Systems, September 1994, Vol.20, No.3.

[11] LITVIN,F., FUENTES,A.: *Gear Geometry and Applied theory*, Cambridge University press, 2004.

[12] ISO 21771:2007. *Gears – Cylindrical involute gears and gear pairs – Concepts and geometry*, 2007.

[13] WRIGGERS,P.: *Computational Contact Mechanics*, Springer, 2006.

[14] http://www.mech.uwa.edu.au/DANotes/gears/failure/failure.html.

[15] STOLARSKI,T., NAKASONE,S., YOSHIMOTO,S.: *Engineering Analysis with Ansys Software*, Elsevier, 2006.

[16] ZIENKIEWICZ,O.C., TAYLOR,R.L.: *The Finite Element Method – Volume 1: The Basis*, Butterworth – Heinemann, 2000.

[17] ISO 6336: 2006. *Calculation of load capacity of spur and helical gears*, 2007.

[18] ATANASOVSKA,I.: *Influence of stiffness and base pitch deviation on load distribution between tooth pairs and involute gear load capacity*, Monograph MACHINE DESIGN, Publiching by University of Novi Sad, Faculty of Technical Sciences, 2007., pp.259-264.

[19] ATANASOVSKA,I., NIKOLIĆ,V.: *Influence of addendum modification coefficient on the gear load capacity*, 16th European Conference of Fracture (ECF16)”, Alexandroupolis, Greece, 2006, pp.787.

Received: 05.03.2009.

MKE model zupčanika sa kosim zupcima za analizu naponskog stanja i nelinearnu analizu kontakta

Određivanje naponskog i deformacionog stanja na bokovima i u podnožju zubaca jedan je od osnovnih zadataka u toku proračuna opterećenja i provere nosivosti zupčanika. U ovom radu opisan je postupak razvijanja modela konačnih elemenata za istovremeno praćenje naponskog i deformacionog stanja bokova zubaca, podnožja zubaca i segmanata zupčanika sa kosim zupcima u toku sprezanja jednog para zubaca. Takođe, opisano je i simuliranje kontaktnih uslova kod evolventnih zupčanika sa kosim zupcima metodom konačnih elemenata (MKE).

Izvršena je odgovarajuća analiza za izbor modela spregnutih zupčanika koji je istovremeno dovoljno ekonomičan i dovoljno geometrijski precizan. Razvijen je poseban algoritam za crtanje evolventnog profila zubaca. Ovaj algoritam ugrađen je u postojeći MKE softver i na taj način je obezbeđeno crtanje realne geometrije bokova zubaca u kontaktu. Takođe, izvršen je i izbor optimalne gustine mreže konačnih elemenata.

Opisani modeli konačnih elemenata razvijeni su za jedan konkretan par zupčanika sa kosim zupcima. Dobijeni numerički rezultati pogodni su za praćenje promene deformacionog i naponskog stanja u toku perioda sprezanja para zubaca.

Ključne reči: **zupčanik, zupčanik sa kosim zupcima, naponsko stanje, kontaktno naprezanje, raspodela napona, proračun napona, metoda konačnih elemenata.**

МКЭ модель косозубого зубчатого колеса для анализа напряжёного состояния и для нелинейного анализа контактов

Определение напряжёного и деформационного состояний на боковых поверхностях и на ножке зубчатого колеса представляет одну из основных задач в процессе расчёта нагрузки и проверки несущей способности зубчатого колеса. В настоящей работе описан способ развития метода конечных элементов для одновременного текущего контроля напряжёного и деформационного состояний боковых поверхностей, ножки зубчатого колеса и сегментов косозубого зубчатого колеса в процессе спаривания одной пары зубьев. Здесь тоже описано и имитационное моделирование контактных условий у эвольвентных косозубых зубчатых колёс методом конечных элементов (МКЭ).

Здесь тоже проведён соответствующий анализ для выбора моделей спаривания зубьев, который одновременно является довольно экономичным и довольно геометрически точным. Тоже развит особый алгоритм для рисования эвольвентного профиля зубьев. Этот алгоритм установлен в уже существующее МКЭ программное обеспечение и таким способом обеспечено и рисование реальной геометрии боковых поверхностей зубьев в контакте. Тоже проведён и выбор оптимальной плотности сети конечных элементов.

Описаные модели конечных элементов развиты для одной конкретной пары косозубого зубчатого колеса.

Полученые цифровые результаты пригодны для текущего контроля изменений напряжёного и деформационного состояний в процессе спаривания одной пары зубьев.

Ключевые слова: Шестерня, косозубое зубчатое колесо, напряжёное состояние, контактное напряжение, распределение напряжения, расчёт напряжения, метод конечных элементов.

Le modèle d’engrenage à dents hélicoïdales MKE pour l’analyse de l’état de tension et l’analyse non linéaire de contact

La détermination de l’état de tension et l’état de déformation sur les flancs et la racine des dents est l’une des taches principales pendant le calcul de la charge et la vérification de la portance de l’engrenage. Ce papier décrit le procédé du développement du modèle des éléments finis pour la poursuite simultanée de l’état de tension et la déformation sur les flancs des dents, la racine des dents et les segments de l’engrenage à dents hélicoïdales au cours de l’engrènement d’une paire de dents. On a décrit aussi la simulation des conditions de contact chez les engrenages à développante à dents hélicoïdales par la méthode des éléments finis (MKE). On a fait une analyse correspondante pour le choix du modèle des engrenages engrenés, qui est à la fois suffisamment économique et précis géométriquement. On a développé l’algorithme particulier pour dessiner le profil développant des dents. .Cet algorithme est incorporé dans le logiciel MKE existant et on a assuré ainsi le dessin de la géométrie réelle des flancs des dents en contact. On a fait aussi le choix de la densité optimale du réseau des éléments finis. Les modèles décrits des éléments finis ont été développés pour une paire concrète des engrenages à dents hélicoïdales. Les résultats numériques obtenus conviennent pour la poursuite du changement de l’état de déformation et de la tension au cours de la période d’engrènement d’une paire de dents.

Mots clés: **engrenage, engrenage à dents hélicoïdales, état de tension, contrainte de contact, distribution de tension, calcul de tension, méthode des éléments finis.**