Analysis of the Contact Stresses of Spur Gears Manufactured by 3D Printing from Composite Materials

Milan Zmindak, Michal Kaco, and Alzbeta Sapietova

Department of Applied Mechanics, Faculty of Mechanical Engineering, University of Zilina, Univerzitna 1, 01026 Zilina, Slovakia

Abstract. Additive manufacturing is a technology rapidly expanding on a number of industrial sectors. However, it is still hampered by low productivity, poor quality and uncertainty of final part mechanical properties. Materials for production of given components are composite materials, especially on the basis of so-called CFRP, CRP (carbon fibre-so-called polymers reinforced by carbon fibres). Composite fibers have properties that can add high strength and stiffness to the printed part. Like all manufacturing processes, composite printing is only suitable for certain design applications. The fibers have a predetermined width during printing and, due to geometrical limitations, can only be used to reinforce structures meeting a certain minimum thickness. The objective of this paper is to predict the contact stresses of carbon/onyx spur gear using Finite Element Analysis (FEA) results through the torque load. Finite element software ANSYS Workbench 20.0 is used to study the stress intensity factor of spur gear. A 3D solid model has been established for simulation of the spur gear which enables to understand the mechanical strength and strain at failure of the composite materials.

1 Introduction

The gear drive transmits power with comparatively smaller dimensions, light weight, less stress, runs reasonably free of noise and vibration with least manufacturing and maintenance cost [1-3]. Spur gear is a cylindrical shaped gear in which the teeth are parallel to the axis. Spur gears are easy to manufacture and it is mostly used to transmit power from one shaft to another shaft up to certain distance & it is also used to vary the speed & Torque, e.g. watches, gearbox etc. They form vital elements of main and ancillary mechanisms in many machines such as automobiles, tractors, metal cutting machine tools etc. Toothed gears are used to change the speed and power ratio as well as direction between input and output.

Composite materials are widely used in various industries. It is well known that their main advantages are very good mechanical properties, chemical resistance, lower material consumption and low product weight [4, 5]. From an economic point of view, the main disadvantage of composite materials is their relatively high price. The properties of composite materials often depend on the position and direction of their individual
components in a fixed coordinate system. In the mechanics of flexible bodies, these materials are referred to as heterogeneous and anisotropic. These materials consist of two or more components, which usually have different material properties. The supporting part of the composite are reinforcing fibers (long or short, eventually particles), the connecting element being a matrix. Fiber composites and laminates are represented at the macro level of homogenized substances with an equivalent or effective modulus of elasticity.

The trend of recent years is the production of composite materials using 3D printing technology weight [6]. The principle of 3D printing is to gradually add material layer by layer. There are several technologies for 3D printing, but fusion deposit modeling (FDM) and continuous fiber production (CFF) technologies are most commonly used for composite printing [7]. For example, the MarkTwo 3D printer, co. Markforged (which will also be used to make test samples) has two print nozzles. One nozzle is used to print material from a nylon or onyx matrix, the other nozzle is used to print a reinforced fiber. The reinforced fiber is coated with nylon or onyx. Fibers of various materials are available (kevlar, glass fiber, carbon fiber) [8].

In this paper, MarkTwo3D printer, manufactured by Markforged[9] has been the ALM/FDM system used to produce a spur gears. The contact stress are analyzed through the FEM for steel and composite spur gear. Finally FE model with a segment of three teeth is considered for crack propagation using SMART (Separating-Morphing-Adaptive and Remeshing Technology) Crack-Growth method.

2 3D printing technique of gears

3D printing is an additive part manufacturing process that is different from traditional manufacturing techniques as is machining, casting and forming [10]. Additive production is controlled by automated systems. Physical 3D models are made from computer models using metallic plastic, ceramic materials. Composite gears have better mechanical characteristics like wear resistance, corrosion resistance, lubricant free, noiseless high strength to weight ratio, etc..

2.1 Markforged and Eiger

Markforged 3D printer was used for printing spur gears (Fig.1). The material used in the Markforged 3D printer contains an Onyx matrix. The material Onyx is produced from hard nylon, which provides a stiffness equal to or greater than any pure thermoplastic used to create products on a 3D printer. The Onyx filament contains chopped micro-carbon fibers, combing the toughness of nylon and thermal properties of carbon[11]. This composite filament provides a dimensionally stable, stiff, and heat tolerant engineering material with high quality surface finish. The Onyx is suitable for customers who require high demands on the visual side of the product. Surface quality exceeds the properties of plastic 3D prints made, for example, from PLA (Polylactic acid) material.
components in a fixed coordinate system. In the mechanics of flexible bodies, these materials are referred to as heterogeneous and anisotropic. These materials consist of two or more components, which usually have different material properties. The supporting part of the composite are reinforcing fibers (long or short, eventually particles), the connecting element being a matrix. Fiber composites and laminates are represented at the macro level of homogenized substances with an equivalent or effective modulus of elasticity.

The trend of recent years is the production of composite materials using 3D printing technology [6]. The principle of 3D printing is to gradually add material layer by layer. There are several technologies for 3D printing, but fusion deposit modeling (FDM) and continuous fiber production (CFF) technologies are most commonly used for composite printing [7]. For example, the MarkTwo 3D printer, co. Markforged (which will also be used to make test samples) has two print nozzles. One nozzle is used to print material from a nylon or onyx matrix, the other nozzle is used to print a reinforced fiber. The reinforced fiber is coated with nylon or onyx. Fibers of various materials are available (kevlar, glass fiber, carbon fiber) [8].

In this paper, MarkTwo3D printer, manufactured by Markforged [9] has been the ALM/FDM system used to produce a spur gears. The contact stress are analyzed through the FEM for steel and composite spur gear. Finally FE model with a segment of three teeth is considered for crack propagation using SMART (Separating-Morphing-Adaptive and Remeshing Technology) Crack Growth method.

2 3D printing technique of gears

3D printing is an additive part manufacturing process that is different from traditional manufacturing techniques as is machining, casting and forming [10]. Additive production is controlled by automated systems. Physical 3D models are made from computer models using metallic plastic, ceramic materials. Composite gears have better mechanical characteristics like wear resistance, corrosion resistance, lubricant free, noiseless high strength to weight ratio, etc..

2.1 Markforged and Eiger

Markforged 3D printer was used for printing spur gears (Fig.1). The material used in the Markforged 3D printer contains an Onyx matrix. The material Onyx is produced from hard nylon, which provides a stiffness equal to or greater than any pure thermoplastic used to create products on a 3D printer. The Onyx filament contains chopped microcarbon fibers, combing the toughness of nylon and thermal properties of carbon [11]. This composite filament provides a dimensionally stable, stiff, and heat tolerant engineering material with high quality surface finish. The Onyx is suitable for customers who require high demands on the visual side of the product. Surface quality exceeds the properties of plastic 3D prints made, for example, from PLA (Polylactic acid) material.

Fig. 1. Markforged 3D printer.

2.2 Fiber reinforcement

The most important rule for the fiber path at gears is the rule that applies to teeth. A gear printed on a 3D printer needs to have the reinforced fibers located in its teeth, as this is the most critical location for pitting damage (Fig.2). In order for the tooth of a gear to be properly reinforced with fiber all the way from the root to the tip, it must be wider than the minimum 3.8 mm threshold at the tip of the gear tooth. The minimum part width for a thin region connected to a larger part at both ends is 2.9 mm [12].

Fig. 2. Reinforcement of gearing from the root to the tip of the tooth.

In the case of smaller gears, fiber can often help reinforce the tooth, but may not be able to fit all the way to the tip. This will still result in a gear that is stronger than a pure plastic one, but be weaker locally long the tooth than desired. In this case fiber reinforcement can only reach slightly beyond the pitch circle of the gear.
3 Finite element modeling and simulation

Finite element method (FEM) is particularly interesting for modelling 3D printed part due to its flexibility in analyzing complex geometries in both macro and micro scale [13,14]. A very important parameter when designing a gear pair is the maximum contact stress that exists between two gear teeth in mesh, as it affects surface fatigue (namely, pitting and wear) along with gear mesh losses. A lot of attention has been targeted to the determination of the maximum contact stress between gear teeth in mesh, resulting in many "different" formulas. Moreover, each of those formulas is applicable to a particular class of gears (e.g., hypoid, worm, spiroid, spiral bevel or cylindrical-spur and helical) [15-18]. More recently, FEM has been introduced to evaluate the contact stress between gear teeth. Presented below is a single methodology for evaluating them a maximum contact stress that exist between gear teeth in mesh. FEM software. ANSYS allows specify the required boundary conditions to the composite model, input values for composite dimensions and material properties of the composite, etc. [19-21].

3.1 Spur gear geometry

Spur gears are used to transmit power between parallel shafts. The Fig. 3 shows a pair of identical spur gears. In Fig. 4 is shown the basic spur gear geometry. Spur gears have their teeth cut parallel to the axis of the shaft on which the gears are mounted. To maintain a constant angular velocity ratio, two gears must satisfy a fundamental law of gearing: the shape of the teeth must be such that the common normal (8) at the point of contact between two teeth must always pass through a fixed point on the line of centers (5). This fixed point is called the pitch point (6). The angle between the line of action (8) and the common tangent of the pitch circles (7) is known as the pressure angle. The parameters defining a spur gear are its pitch radius $r_p$, pressure angle ($\alpha = 20^\circ$) and number of teeth (N). The teeth are cut with a radius of addendum $r_a$ (9) and a radius of addendum $r_d$ (10). The shaft has a radius $r_s$ (11) and the fillet radius is $r_f$ (12). The thickness of the gear is 20.0 mm. The spur gears parameters are used from [22] and are given in Table 1.

![Fig. 3. A pair of gears.](image-url)
Finite element modeling and simulation

Finite element method (FEM) is particularly interesting for modeling 3D printed parts due to its flexibility in analyzing complex geometries in both macro and micro scale [13,14].

A very important parameter when designing a gear pair is the maximum contact stress that exists between two gear teeth in mesh, as it affects surface fatigue (namely pitting and wear) along with gear mesh losses. A lot of attention has been targeted to the determination of the maximum contact stress between gear teeth in mesh, resulting in many different formulas. Moreover, each of those formulas is applicable to a particular class of gears (e.g., hypoid, worm, spiroid, spiral bevel or cylindrical - spur and helical) [15-18].

More recently, FEM has been introduced to evaluate the contact stress between gear teeth. Presented below is a single methodology for evaluating the maximum contact stress that exist between gear teeth in mesh. FEM software, ANSYS allows specify the required boundary conditions to the composite model, input values for composite dimensions and material properties of the composite, etc. [19-21].

3.1 Spur gear geometry

Spur gears are used to transmit power between parallel shafts. The Fig. 3 shows a pair of identical spur gears. In Fig. 4 is shown the basic spur gear geometry. Spur gears have their teeth cut parallel to the axis of the shaft on which the gears are mounted. To maintain a constant angular velocity ratio, two gears must satisfy a fundamental law of gearing: the shape of the teeth must be such that the common normal (8) at the point of contact between two teeth must always pass through a fixed point on the line of centers (5). This fixed point is called the pitch point (6). The angle between the line of action (8) and the common tangent of the pitch circles (7) is known as the pressure angle. The parameters defining a spur gear are its pitch radius \( r_p \), pressure angle \( \alpha = 20° \) and number of teeth \( N \). The teeth are cut with a radius of addendum \( r_a \) and a radius of dedendum \( r_d \). The shaft has a radius \( r_s \) and the fillet radius is \( r_f \). The thickness of the gear is 20.0 mm.

The spur gears parameters are used from [22] and are given in Table 1.

![Fig. 4. Basic spur gear geometry.](image)

<table>
<thead>
<tr>
<th>Dimensions</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shaft centre distance [mm]</td>
<td>( a )</td>
<td>91.5</td>
</tr>
<tr>
<td>Effective face width [mm]</td>
<td>( b )</td>
<td>20</td>
</tr>
<tr>
<td>Working pitch diameter [mm]</td>
<td>( d_w )</td>
<td>91.5</td>
</tr>
<tr>
<td>Tip diameter [mm]</td>
<td>( d_t )</td>
<td>99.89</td>
</tr>
<tr>
<td>Module [mm]</td>
<td>( m )</td>
<td>4</td>
</tr>
<tr>
<td>Number of teeth [-]</td>
<td>( N )</td>
<td>23</td>
</tr>
<tr>
<td>Profile shift coefficient [-]</td>
<td>( x )</td>
<td>-0.061</td>
</tr>
<tr>
<td>Pressure angle [°]</td>
<td>( \alpha )</td>
<td>20</td>
</tr>
<tr>
<td>Working pressure angle [°]</td>
<td>( \alpha_w )</td>
<td>19.12</td>
</tr>
<tr>
<td>Overlap ratio [-]</td>
<td>( \varepsilon_{\alpha} )</td>
<td>1.69</td>
</tr>
</tbody>
</table>

3.2 Boundary conditions

First of all, we have prepared assembly in Creo Parametric for spur gear and save as this part as IGES for Exporting into ANSYS Workbench Environment [23]. Based on the
assumptions of Lewis equation, the boundary conditions are set in ANSYS Workbench. The fixed support is used at the root end of the tooth and the Z component of the moment is applied, red colour (Fig.5).

![Fig. 5. Boundary conditions.](image)

### 3.3 Meshing

Meshing is basically the division of the entire model into finite elements so that at each and global the stiff equations are solved. The number of elements and the type elements significantly affects the accurate of solution and also improves the quality of solution. In our case quality, the quality of the mesh is controlled by the choice adaptive sizing and mesh defeaturing. The target quality is 0.05 and transition ratio is 0.272. The maximum layers are 5 and growth rate is 1.2. Here the element size of 1 mm with medium smoothing is considered for mesh generation (Fig.6). The geometry is created by 49 faces, number of nodes is 106868 and number of elements is 59727. The contact is frictional, number contacting is 23, penetration is 6.3968e-04, geometric penetration is 6.3968e-04, geometric gap is 4.0283e-05 and resulting pinball is 0.9126.
assumptions of Lewis equation, the boundary conditions are set in ANSYS Workbench. The fixed support is used at the root end of the tooth and the Z component of the moment is applied, red colour (Fig. 5).

Fig. 5. Boundary conditions.

3.3 Meshing

Meshing is basically the division of the entire model into finite elements so that at each and global the stiff equations are solved. The number of elements and the type elements significantly affects the accurate of solution and also improves the quality of solution. In our case quality, the quality of the mesh is controlled by the choice adaptive sizing and mesh defeaturing. The target quality is 0.05 and transition ratio is 0.272. The maximum layers are 5 and growth rate is 1.2. Here the element size of 1 mm with medium smoothing is considered for mesh generation (Fig. 6). The geometry is created by 49 faces, number of nodes is 106868 and number of elements is 59727. The contact is frictional, number contacting is 23, penetration is 6.3968e-04, geometric penetration is 6.3968e-04, geometric gap is 4.0283e-05 and resulting pinbal is 0.9126.

Fig. 6. FEM mesh.

3.4 Frictional model

In the basic Coulomb friction model, two contacting surfaces can carry shear stresses. When the equivalent shear stress is less than a limit frictional stress ($\tau_{\text{lim}}$), no motion occurs between the two surfaces [24]. This state is known as sticking. The Coulomb friction model is defined as:

$$\tau_{\text{lim}} = \mu p + b$$  \hspace{1cm} (1)

$$\|\tau\| \leq \tau_{\text{lim}}$$  \hspace{1cm} (2)

where

- $\tau_{\text{lim}}$ - limit frictional stress
- $\|\tau\|$ - \{ $|\tau|$ equivalent stress fro 2D contact $\sqrt{\tau_1^2 + \tau_2^2}$ equivalent stress for 3D contact \}
- $\mu$ -coefficient of friction for isotropic friction
- $p$ -contact normal pressure
- $b$ -contact cohesion

Once the equivalent frictional stress exceeds $\tau_{\text{lim}}$ the contact and target surfaces will slide relative to each other (Fig. 7). This state is known as sliding. The sticking/sliding calculations determine when a point transitions from sticking to sliding or vice versa. The contact cohesion provides sliding resistance even with zero normal pressure. The coefficient of friction 0.2 was specified for our case.
3.5 Material

Selecting different materials for gears plays an important role in gear technology. Material selected for making a gear must satisfy two conditions: (1) manufacturability and processing requirement; (2) achieving required service life. Manufacturability requirement includes its forgeability and its response to heat treatment. Whereas, to achieve required service life, gears should transmit power to a satisfactory level when working in loading conditions as well as fulfilling mechanical property requirement such as fatigue, strength and response to heat treatment.

The material used for this analysis is Onyx matrix reinforced by carbon fibres. Material properties of both materials are given in Table 2 and Table 3. Homogenized material properties for volume fraction of gear teeth \( v_f = 0.4 \) are given in Table 4.

### Table 2. Material properties of Onyx.

<table>
<thead>
<tr>
<th>Material</th>
<th>Modulus of elasticity (E) [GPa]</th>
<th>Poisson’s ratio (( \mu ))</th>
<th>Density (( \rho )) [g/cm³]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Onyx</td>
<td>1.4</td>
<td>0.3</td>
<td>1.2</td>
</tr>
</tbody>
</table>

### Table 3. Material properties of carbon fiber.

<table>
<thead>
<tr>
<th>Material property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Modulus of elasticity of lamina in the direction of fibers ( E_1 ) [GPa]</td>
<td>64.7</td>
</tr>
<tr>
<td>Transverse modulus of elasticity ( E_2 ) [GPa]</td>
<td>22.4</td>
</tr>
<tr>
<td>Shear modulus of elasticity of the lamina ( G_{12} ) [GPa]</td>
<td>22.1</td>
</tr>
<tr>
<td>Shear modulus of elasticity of the lamina ( G_{23} ) [GPa]</td>
<td>8.3</td>
</tr>
<tr>
<td>Poisson’s ratio ( \mu_{12} )</td>
<td>0.3</td>
</tr>
<tr>
<td>Poisson’s ratio ( \mu_{23} )</td>
<td>0.35</td>
</tr>
<tr>
<td>Density ( \rho ) [g/cm³]</td>
<td>1.4</td>
</tr>
</tbody>
</table>
Table. 4. Homogenized material properties of unidirectional composite.

<table>
<thead>
<tr>
<th>$E_{11}$ [GPa]</th>
<th>$E_{22}$ [GPa]</th>
<th>$E_{33}$ [GPa]</th>
<th>$\nu_{12}$ -</th>
<th>$\nu_{13}$ -</th>
<th>$\nu_{23}$ -</th>
<th>$G_{12}$ [GPa]</th>
<th>$G_{13}$ [GPa]</th>
<th>$G_{23}$ [GPa]</th>
<th>$\rho$ [kgm$^{-3}$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.960</td>
<td>1.501</td>
<td>1.501</td>
<td>0.30</td>
<td>0.30</td>
<td>0.4</td>
<td>4.369</td>
<td>3.902</td>
<td>3.902</td>
<td>1490</td>
</tr>
</tbody>
</table>

4 Numerical results

Fig. 8 shows the von Mises stress contour for steel spur gears and Fig.9 for material onyx + carbon. The maximum values of von Mises stress are 59.28 MPa for material steel and 99.35 for material onyx + carbon. The next FEA results are compared in Table 5.
### Table 5. Simulation results.

<table>
<thead>
<tr>
<th></th>
<th>Structural steel</th>
<th>Onyx + Carbon</th>
</tr>
</thead>
<tbody>
<tr>
<td>von Mises stress [MPa]</td>
<td>59.28</td>
<td>99.35</td>
</tr>
<tr>
<td>Contact pressure [MPa]</td>
<td>67.55</td>
<td>83.77</td>
</tr>
<tr>
<td>Friction stress [MPa]</td>
<td>0.43</td>
<td>11.08</td>
</tr>
<tr>
<td>Sliding [mm]</td>
<td>12.26*10^{-5}</td>
<td>720.14*10^{-5}</td>
</tr>
<tr>
<td>Penetration [mm]</td>
<td>24.7*10^{-5}</td>
<td>1825*10^{-5}</td>
</tr>
</tbody>
</table>

### 5 Fracture mechanics

Fracture mechanics mainly used for predicting strength and life of cracked structures. Using the computational methods of applied mechanics, the stress and deformation states at the crack are determined. Linear elastic fracture mechanics can be used to describe the behaviour of cracks. The basic assumption of this theory is determining the crack growth behaviour in elastic domain of material by using stress intensity factors (SIF). Therefore, calculating the SIFs is very important to achieve accurate results [25].

Fracture analysis is typically accomplished using either the energy criterion or the SIF criterion. For the energy criterion, the energy required for a unit extension of the crack (the energy-release rate) characterizes the fracture toughness. For the SIF criterion, the critical value of the amplitude of the stress and deformation fields characterizes the fracture toughness. Under some circumstances, the two criteria are equivalent [26].

We note that the fracture toughness of composites, made by combining engineering polymers, with greatly exceeds the individual fracture toughness of the constituent materials.

Computational methods for evaluating the fracture parameters can be broadly categorized as[27]:

- Methods wherein the singular fields near the crack tip or crack front are modeled explicitly by spatial discretization schemes such as finite element methods (FEM) or boundary element methods (BEM).
- Superposition methods wherein the singular fields are treated entirely analytically (through infinite body solutions), and spatial discretization techniques are employed to solve only the uncracked structure.

The FEM has been successfully used to compute the SIFs in stress analysis of crack problems with finite domains and general, complex boundary conditions, for which analytical solutions are not available. When conventional, conforming displacement elements are used, the stress singularities developing at the tip of a crack or at the tree of a tinted crack are modeled either through suitable geometric transformations of by adopting appropriate interpolation functions. In general, the computation of SIFs is performed indirectly and requires the use of highly refined finite element meshes in the neighborhood of the crack to shape with accuracy the characteristics high stress gradients developing locally. Static and dynamics linear and nonlinear fracture mechanics analysis can be performed with many commercials software for example ADINA, ANSYS, MARC etc.

The SIF is one of the most important and currently also the most used mechanical quantities describing the state of stress in a cracked body. This parameter includes both the size and method of external loading, as well as the basic qualitative and quantitative characteristics of the geometry of the body and the crack.
The SIFs describe the magnitude of the elastic stress field at a crack front. The SIF in general is mainly dependent on load, crack length
\[ K = f(\text{load, crack length, geometry}) \] (3)
The stress fields in any linear elastic cracked body are expressed as\[ \sigma_{ij} = \left( \frac{K}{\sqrt{2\pi r}} \right) f_\theta(\theta) + \sum_{m=0}^{\infty} A_m r^m g_m^{(m)}(\theta) \] (4)
where \( \sigma_{ij} \) are the Cauchy stresses, \( r \) is distance from the crack and \( \theta \) is the angle with respect to the plane of the crack (Fig. 10), \( f_\theta \) is a dimensionless functions of \( \theta \) which is dependent on the geometry of crack and loading conditions. For the higher-order terms, \( A_m \) is amplitude and \( g_m^{(m)} \) is a dimensionless function of \( \theta \) for the \( m \)th term. These equations apply to any of the three fracture modes (opening mode, shearing mode, tearing mode). Since the quantity \( f_\theta \) is dimensionless, the SIF can be expressed in units of \( \text{MPa} \sqrt{\text{m}} \).

![Fig. 10. Schematic of a crack tip.](image)

For linear elastic problems, the displacements near the crack tip (or crack front) vary as 
\[ 1/\sqrt{r} \]. The stresses and strains are singular at the crack tip, varying as
\[ \sigma_\theta = -\frac{K}{\sqrt{r}} f_\theta(\theta) \] (5)
\[ \varepsilon_\theta = -\frac{K}{\sqrt{r}} g_\theta(\theta) \] (6)
To produce this singularity in stresses and strains, the crack tip mesh should have certain characteristics: A mesh of circular rings of elements near the crack tip is needed, the quarter-point element is widely used in linear elastic fracture computations with the FEM. In this case the shape functions contain singular crack-specific functions the free parameters of which are related to the \( K \) factors. Special elements of this type are called crack tip elements (CTE) (Fig. 11). They are utilized to discretize the direct surroundings of the crack tip, while regular elements are then used to model the rest of the structure.
5.1 Orthotropic mixed mode fracture and SIF calculation

An orthotropic composite material behaviour can be further idealized in order to simplify the process of crack modelling; a fibre composite structure is assumed as a homogeneous orthotropic continuum, where the crack growth takes place in an idealized material with anisotropic constituents. In this approach, the details of local failures of the composite, such as broken fibres or cracked matrix, are not considered and an equivalent orthotropic continuum is adopted [29].

**SIF Calculation by Interaction Integral**

The SIF is one of the important parameters representing fracture properties of a crack tip. The domain integral method is adopted to evaluate the mixed modes stress intensity factors in homogenous orthotropic media

\[
J = \int_{\Gamma} \left( w_i \delta_{ij} - \sigma_{ij} \frac{\partial u_i}{\partial x_j} \right) n_j \, d\Gamma
\]  

where \(w_i\) is the strain energy density for linear elastic material, \(\delta_{ij}\) is the Kronecker delta, \(\Gamma\) is an arbitrary contour around the crack tip which encloses no other cracks or discontinuities and over which the integration is carried out, \(n_j\) is the \(j\)th component of the outward unit normal to \(\Gamma\), and \(u_i\) is the component of the displacement vector. Eq. (5) is not well suited for the finite element solutions, and an equivalent form of the \(J\) integral can be obtained by exploiting the divergence theorem in the form of the domain integral approach

\[
J = \int_{A} \left( \sigma_{ij} - \frac{\partial u_i}{\partial x_j} w_j \delta_{ij} \right) \frac{\partial q}{\partial x_j} \, dA
\]

where \(A\) is an area surrounding the crack tip (the interior region of \(\Gamma\)) and \(q\) is a smoothly varying function. \(\Gamma\) is usually assumed as a circular or rectangular area whose center locates on the crack tip. The extended finite element method (XFEM) to overcome the singularity, \(1/r\) problems at the crack tip by incorporating the singularities in local approximation. The XFEM is based on enriching the degrees of freedom in the model with additional displacement functions that account for the jump in displacements across the crack surface. The method is used to propagate cracks in linear elastic materials based on user-specified fracture criteria. With SMART (separating morphing and adaptive remeshing technology) fracture modeling functionality in ANSYS Mechanical, crack growth analysis is not dependent on the mesh. Fracture modeling is faster and more accurate, and requires much less special expertise. For more information, see XFEM-Based Crack Analysis and Crack-Growth Simulation [26].
The approach for evaluating the energy-release rate is based on the virtual crack-closure technique (VCCT). VCCT is based on the assumption that the energy needed to separate a surface is the same as the energy needed to close the same surface. The implementation described here uses the modified crack-closure method (a VCCT-based method) and assumes further that stress states around the crack tip do not change significantly when the crack grows by a small value of crack length (\(\Delta a\)). The VCCT method for energy-release rate calculation supports the following material behaviors: linear isotropic elasticity, orthotropic elasticity, anisotropic elasticity.

\[ f = f\left(G_I^c, G_{II}^c, G_{III}^c, G_I, G_{II}, G_{III}\right) \]  

Fracture occurs when the fracture criterion index is met, expressed as:

\[ f \geq f_c \]  

where \(G_r^c\) is the fracture criterion ratio. The recommended ratio is 0.95 to 1.05. The default is 1.0.

**Critical Energy-Release Rate Criterion**

The critical energy-release rate criterion uses total energy-release rate (GT) as fracture criterion. The total energy-release rate is summation of the Mode I (GI), Mode II (GII), and Mode III (GIII) energy-release rates, expressed as:

\[ f = \frac{G_r}{G_r^c} = \frac{G_I + G_{II} + G_{III}}{G_r^c} \]  

where is the critical energy-release rate.

Therefore, the SIF at the crack tip is studied to indicate the behavior of crack propagation towards the remote stress applied, especially on the strength of the cracked gear tooth [30].

**5.2 Numerical results**

In this section we will present some results using SMATR modelling in ANSYS Workbench software. Only a finite element model with a segment of three teeth is considered for analysis. Zero displacements and rotations (fixed support) are prescribed on the lower surface of the gear. Here the element size of 2 mm with medium smoothing is considered for mesh generation.

The course of the von Mises stress is shown in the Fig. 12 and its maximum value is 855.98 MPa. The SIF was determined from the finite element nodal displacements and forces using J-integral method. A local coordinate system is defined for the crack tip. The components indicate crack propagation direction (x) and crack opening direction (y). Mode I loading refer to loads applied normal to the crack plane, which tends to open the crack. Mode II
refers to in plane shear loading. We select three named geometric regions to define the crack. These are the crack edge, the top surface of the crack and the bottom surface of the crack.

Each of these regions is then associated with a node set for use in the analysis. The top surface of the crack and the bottom surface of the crack have prescribed displacements 8 mm for X and Y components. Options for crack growth are: Initial crack is defined as pre-meshed crack, failure criteria option is SIF and critical rate is $6.3246 \times 10^{-5}$ MPa.mm$^{1/2}$ and element size is 2 mm.

SIF for crack propagation under static load is shown in the Fig. 13 and its maximum value is $-86.71$ MPa.mm$^{1/2}$. We see that its value is negative. In physical sense, the negative mode I SIF is incorrect. When the crack is within a compressive stress field, the negative mode I SIF often occurs. The negative mode II stress intensity factor is attributed to the direction of the tangential component (shear) of the applied load in the mixed mode conditions.
The graph plots the distance of the crack front node from the origin and the energy release rate as it moves along the crack front. The number of solution contours is set to 6. These are the “loops” through the mesh around the crack tip, which are used to evaluate the SIF by integrating the crack tip region strain energy. The fracture mechanics approach avoids the stress singularities at the crack tip in the analysis.

The curves in Fig. 14 present the six contours (cont 1 to cont 6) requested in the Pre-Mesh Crack object. They are used to check for convergence of the KI value. The first red curve is from a contour close to the crack tip; the other curves represent contours at increasing distances. The curves converge quickly, showing the mesh is adequate.

![Graph showing SIF contours](image)

**Fig. 14. SIF- Contour 6.**

The meaning of colours: cont 1-red, cont 2-light green, cont 3 - blue, cont 4 – purple, cont 5- orange, cont 6, dark green.

### 6 Conclusions

In this paper, a 3D deformable-body (model) of spur gears is developed. This study provides a foundation for future studies on calculation of contact stresses and SIF for crack propagation under static load using FEM. The model is applied onto commercial FEA software ANSYS Workbench. A general finite element model was developed for evaluating the contact stress and SIF in spur gears of equal geometry in both gears. From the result it could be concluded that the von Mises stress and contact pressure of composite material is slightly larger compared to steel. Friction stress is very larger for composite materials with friction coefficient 0.2.

Finite element analysis has been carried out for gear crack propagation. SIF for crack propagation under static load was calculated. We see that its values are negative. This is probably due to the crack is within a compressive stress field.
Acknowledgments

This work was supported by the Scientific Grant Agency of the Ministry of Education of Slovak Republic and the Slovak Academy of Sciences under Projects VEGA 1/0141/20 and VEGA 1/0510/20

References

15. S. Bodzas Tooth contact analysis of helical gears having modified straight teeth by changing of the number of teeth on the pinion, StrojnickyCasopis - Journal of Mechanical Engineering, 70 (1), 1-16 (2020).