Experimentální Analýza Napětí 2005

THE STRESS INTENSITY FACTOR DETERMINATION USING FEM AND IT'S EXPERIMENTAL VERIFICATION

URČENÍ LOMOVÝCH PARAMETRŮ TRHLINY METODOU KONEČNÝCH PRVKŮ A JEJICH OVĚŘENÍ EXPERIMENTEM

Jiří Žák [1]

This article is interested in the stress intensity factor (SIF) determination; the SIF is one of the most important and also most used fracture parameters in these days.

The cracked body is a nonstandard oblique gear wheel that is used in an automobile gear-box. The crack is placed in the bottom of the cog wheel's cog, next to the "active tooth flank". "Active tooth flak" means the tooth is loaded on this tooth flank and that is why the crack can be opened by the load and can growth too.

The SIF solution was performed using FEM software Ansys. There were created and analyzed many FE models of the cracked body – the differences among these models were the crack size and the number of teeth modelled there. First, the experiment was simulated so, only one pair of teeth was modelled. Second, the FE model which consisted of five teeth in every gear was created to realize existing gears in the automobile gear-box. From every solution the SIFs chart along the crack tip was done. Then the SIF charts depending on the crack size and on the loading magnitude was done too. From these charts the critical crack size or the critical gear loading can be estimated when the fracture toughness of the material is known.

The last part of this article is about experimental verification of the FE solutions. In the school laboratory, there were realized several experiments – the experimental verification of stresses on the gear tooth surfaces and the experimental verification of the critical gear loading. The crack in the bottom of cog whell's cog was performed by cyclic loading.

Keywords

Stress intensity factor, SIF, FEM, Ansys, gear.

Introduction

The stress intensity factor (SIF) is one of the most important and in these days also most used fracture parameters of fracture mechanics. This parametr describes the state of stress in the crack tip vicinity. Several methods can be used for its determination, e.g. analytical solution of the stress tensor components in the crack tip vicinity, experimental methods (e.g. using photoelasticity) and also numerical method – the finite element method (FEM). The last method is very powerful for its universality and is used here, in this work.

[1] Ing. Jiří Žák: Katedra částí a mechanismů strojů, Technická univerzita v Liberci; Hálkova 6, 461 17 Liberec 1, tel.: +420 483 533308, e-mail: jiri_zak@seznam.cz

Solutions for comparison with an experiment

Cracked body

The cracked body is a nonstandard oblique gear wheel. The crack is placed in the bottom of the cog wheel's cog and next to the „active tooth flank" as can be seen in fig.1. „The active tooth flank" means the tooth is loaded on this flank so, the crack can be opened and growths too.

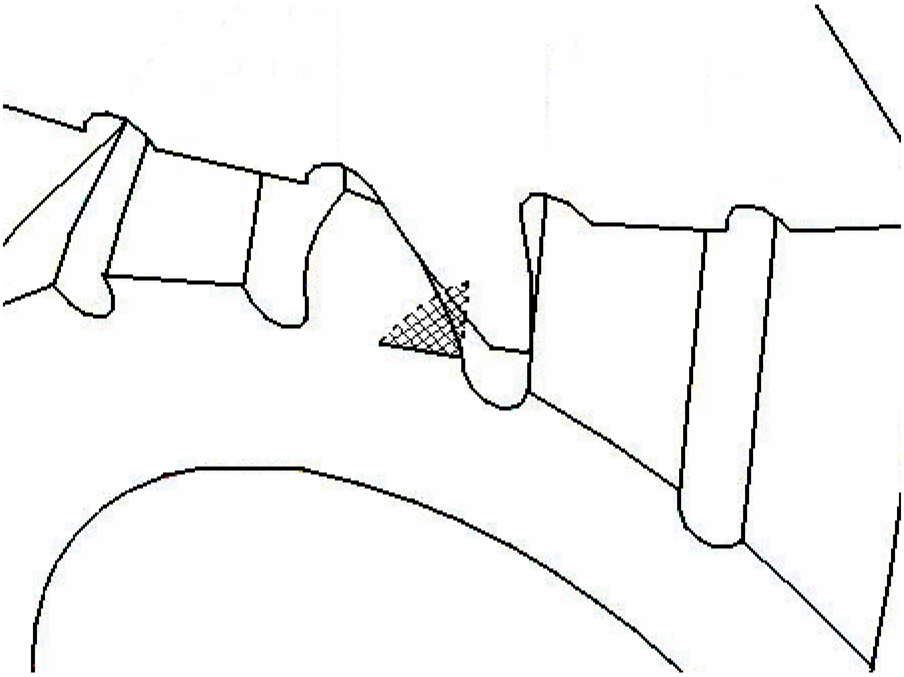

Fig.1 – The crack placing in the bottom of the pinion tooth and next to the „active tooth flank".

FE model description

The CAD model of the gear was modelled in Pro/Enginer software. Then this model was imported to Ansys software and the crack surface was created. Linear material properties of steel was defined. The element types which were used for generating mesh were Solid45 for the gear volumes, Link4 for gear centers substitution and contact elements Targe170 and Conta174 for the pinion and gear interaction.

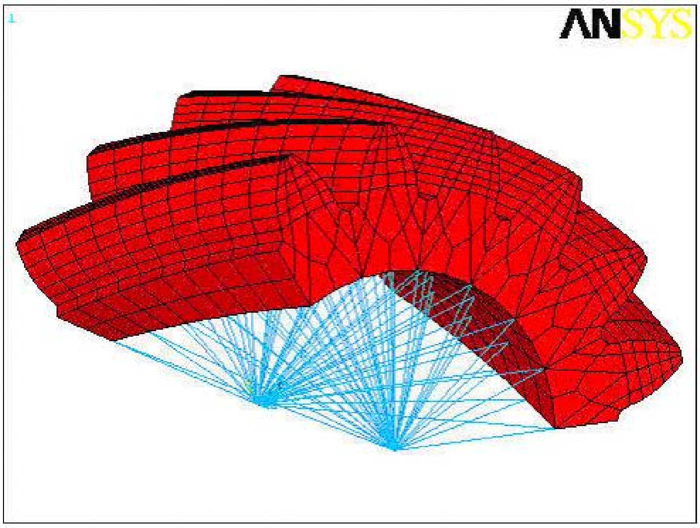

Fig.2 – An example of gear FE model.

Boundary conditions and model solution

FE model was fixed in space by applying displacements to four nodes which were placed in the pinion and gear axes ($u_x=u_y=u_z=0$). Next pinion boundary condition was displacement applied in the tangent direction to some nodes – this boundary condition set pinion rotation. The gear was loaded applying nodal forces also in the tangent direction.

After applying all of boundary conditions the FE model solution performed.

Submodel

For the SIF solution possibility in the Ansys software the FE model of a cracked body must be created according to some necessary conditions. These conditions can be seen for example in [1, 2]. One of the necessary conditions is that the element type in the crack tip vicinity must be Solid95 – a prism with midside nodes. The second necessary condition is the crack tip elements must be so called „singular" – it simulates the „infinite stress" in the crack tip. The submodel respected all of these necessary requirements. The submodel is an FE model of one tooth of pinion and one tooth of gear - the teeth are in contact.

As can be seen in [2] the singular elements shape isn't necessary to be triangular. This claim was verified by SIF study of a cracked body which was the finite planar plate with semicircular surface crack.

After boundary conditions applying (strictly according to Ansys submodelling technique) the stress analysis was performed. In the end the SIF values and its equivalent values were determined.

Real cracked gear FE solution

The procedure used here was almost the same as the procedure mentioned above in this article. The only one step which wasn't the same was that it was necessary to find the worst loading state for the cracked tooth. It was found using various tangent displacements of the pinion as a boundary condition. The worst loading state of the cracked tooth was characterized by the maximum crack opening.

Experimental verification of FEM solutions

In the practical use of gears there are several teeth pairs in contact but it wasn't possible to obtain the fatique fracture in an acceptable testing time. So, it was decided to solve and test gears with only one teeth pair in contact. A tester can be sen in fig.3. Both pinions are fixed in a bottom part of the tester and the gear loads pinions by cyclic loading.

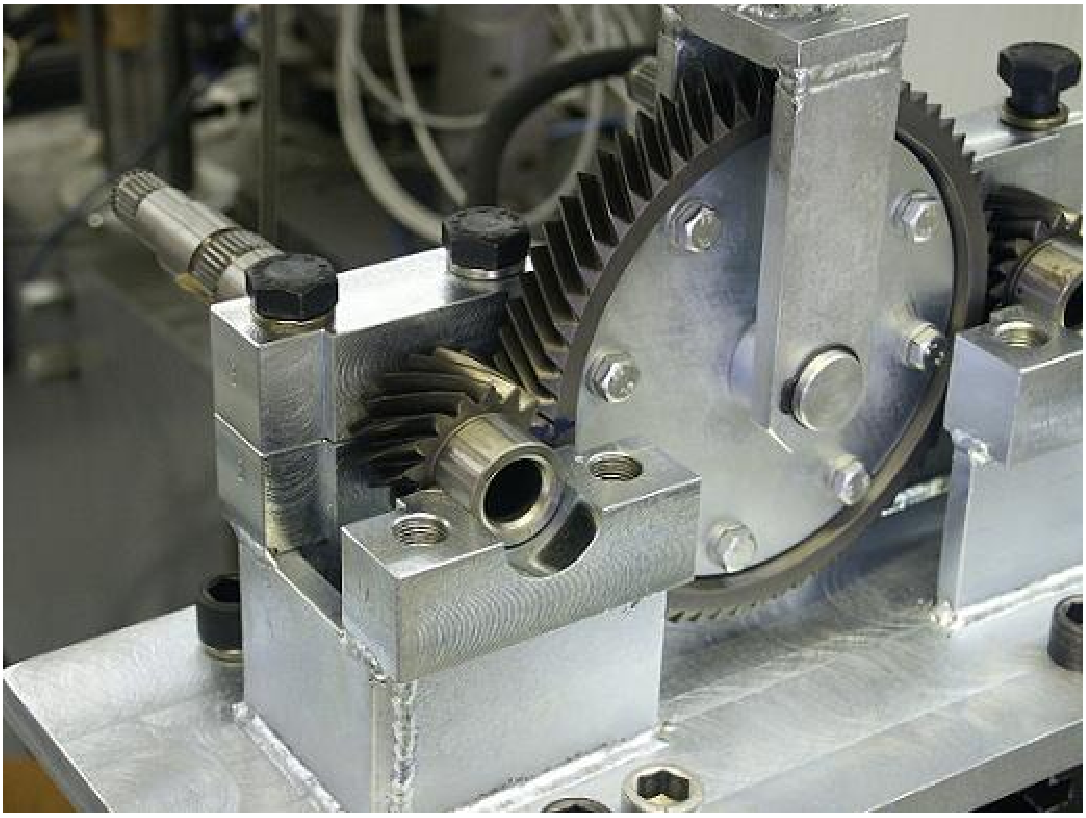

Fig.3 – A tester (a pinions have all of it‘s teeth).

A tensiometer was placed on an „inactive tooth flank“ for strain verification calculated in this place.

The next step was a verification of the critical crack loading. So, first the fatique crack was performed by cyclic loading. Then the load magnitude was increasing very slowly from 0N till the tooth was broken – till the critical cracked body loading.

Conclusion

The SIF determination was performed using a very modern and a very powerfull method – the FEM. One of its big advantages is that it can be used for the cracked body of various schapes. The results obtained using FEM was verificated by experiment – experimental strain determination and experimental critical loading identification. Both experimental results are in deuce with FEM solution.

References

[1] Ansys help

[2] Sfakiotakis, Anifantis: Finite element modeling of spur gearing fracture, in: Finite Elements in Analysis and Design 39 (2002), p.79-92.