Numerical modelling of contact-driven bearing fixation in staking-type joints for electric motor housings

Sz.Szczygieł[1] and M. Szczepanik, PhD, DSc[2]

[1,2] Faculty of Mechanical Engineering, Silesian University of Technology, Gliwice, Poland
e-mail: Szymon.Szczygiel@polsl.pl; Miroslaw.Szczepanik@polsl.pl

KEYWORDS: *Contact mechanics, finite element method, staking joints, bearing fixation*

This work presents the first stage of a study focused on the numerical modelling of bearing fixation mechanisms in electric motor housings applied in steer-by-wire systems using staking-type joints. This stage is dedicated to the development of a computational approach enabling the analysis of contact interactions and load transfer mechanisms governing the mechanical performance of such connections. The final objective of the study is to develop a comprehensive methodology for the design and optimisation of these joint systems.

A three-dimensional finite element model was developed in ANSYS Mechanical using a cyclic symmetry approach to represent a segment of the staking-type joint [1]. The model includes an elastic bearing and a deformable housing region, where fixation is achieved through geometric interference, with material behavior of the housing defined in the elastic–plastic regime. Contact interactions were defined using a frictionless formulation to ensure numerical stability. The loading scenario consists of an axial displacement applied to the bearing (see Fig. 1), simulating the extraction process and enabling determination of the force–displacement response. The analysis focuses on contact pressure, equivalent stress distribution, and reaction force evolution, while a parametric study is prepared to assess the influence of selected geometric parameters.

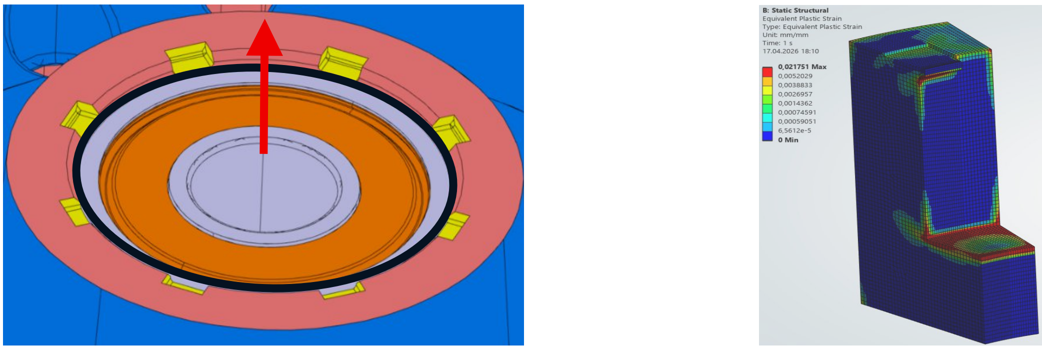

Figure 1. Left: geometry and loading scheme of the staking-type bearing fixation system (yellow - staking features; salmon red - motor housing; gray – bearing outer race; navy - bearing to housing contact zone; red – bearing push off force applied to bearing during the simulation); right: plastic strain simulation result.

The obtained results show the evolution of the reaction force with imposed displacement, allowing identification of the initial slip threshold and the maximum resistance force of the joint. The results indicate a strong dependence of the load–displacement response on staking geometry, where changes in design significantly affect stress concentration and joint stiffness. The numerical analysis reveals the occurrence of plastic deformation in the staking region, with a maximum equivalent plastic strain of 0.022, indicating localized material yielding governing the mechanical response of the joint. The maximum displacement obtained in the simulation reaches approximately 0.4mm, reflecting the progressive deformation of the joint under axial loading. Further work will include correlation of numerical results with experimental tests conducted on physical samples of the joint, aimed at refining the contact-based formulation.

ACKNOWLEDGEMENT:

[1] O.C. Zienkiewicz and R.L. Taylor, J.Z. Zhu, The Finite Element Method: Its Basis and Fundamentals, Butterworth-Heinemann, 7th ed., 2013. https://doi.org/10.1016/C2009-0-24909-9