XVI Konferencja Naukowo-Techniczna
TKI2022
TECHNIKI KOMPUTEROWE W INŻYNIERII
18–21 października 2022

Automation of modelling, analysis and post-processing of Rod-End parts

Damian Andrzejczyk[1], Artur Faflik-Brooks[1]
[1]Airbus Helicopters Polska Sp. z o.o.
email: damian.andrzejczyk@airbus.com, artur.faflik-brooks@airbus.com

ABSTRACT: It was found that the Rod Ends parts, in certain operating conditions, were experiencing excessive wear on some helicopter models. The cause was soon identified: some combinations of relative dimensions of the part produced high contract pressures and stress, which caused crack formation. A series of tools were developed to automatically model a Rod End assembly, depending on the input dimensions, generate the FEM model, launch the analysis, post-process and compare the results in order to optimize the geometry of the part and ensure a satisfying life-time of the part.

KEYWORDS: rod end, optimization, automation, FEM, life-time

1. Introduction

The Rod Ends are used to transfer loads from a shaft to a rod. They are localized throughout a helicopter's main and tail rotor mechanism assemblies, for some models adding up to over 200 such parts affected per helicopter.

An overview of the complete Rod End, with the Joint and the Bearing is presented in Fig. 1.

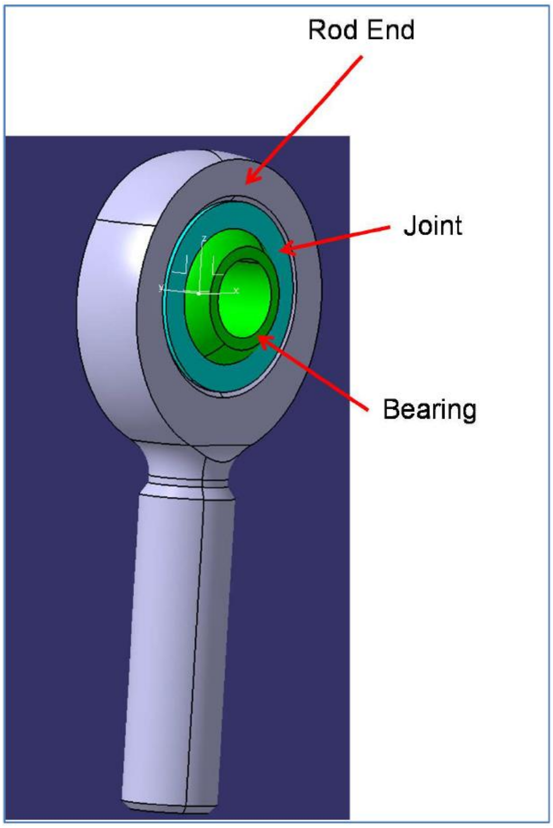

Fig. 1. Rod End assembly

The helicopter's operation environment has a major influence over the crack occurrence frequency, with dusty regions being the most affected. Frequent inspections are required and, in most extreme cases, cracks can occur in the part after a very short flight time, counted in the order of magnitude of tens of hours. The likelihood of such cracks occurring is assumed to be related to the amount of stress and contact pressure in the parts, hence it is desired to identify the geometrical variables which would enable the stress and contact pressure, as well as the displacement, to be decreased to the maximum extent. A first investigation, based on the company's engineering experience, backed by theoretical calculations from [1], soon found out that the critical zones, located on the Rod End's internal surface edges, were highly sensible to the width dimension. However it was hard to quantify the correlation of the contact pressure and displacements with multiple dimension variables, so a detailed FEM analysis was necessary. A project was launched with the purpose of investigating geometrical variables, or combinations of variables, which would enable cracks to occur less frequently on the Rod End parts. It is likely that such an investigation may ultimately involve a large number of analyses, hence the first step was to create a tool, or set of tools, which would enable the automation of the Rod End geometry creation, mesh generation, structural analysis and results post-processing.

2. Overall workflow

As a first step, a Rod End baseline geometry is created in CATIA. The geometry model needs to be parametric. The Excel-based Rod End tool updates the dimensions of the baseline model of the geometry in CATIA, and then generates an FE model in HYPERMESH using the TCL language, with a higher-quality hexagonal mesh on the Rod End inner contact surfaces of interest. This process can be automatically performed for multiple sets of input parameters. The code must be robust, and it must guarantee a proper mesh generation for all the possible combinations of parameters, so it was necessary to extensively consult [2] and [3]. Therefore, this step was the most complicated and time-consuming of the entire project. A separate tool, the SAMCEF autorun tool was then developed, to run the FEM static analysis for each parameter set in series using the SAMCEF FE solver, while a further tool, the HYPERVIEW post-processing tool, was also developed to output plots of the Critical Areas, as well as importing the maximum stress values in such areas into an Excel table. It was assumed that these conclusions will be independent of the materials used, however the created set of tools contain the possibility for material properties (Young's modulus) to be set individually for the Rod End, joint and bearing parts.

The overview of the workflow is presented in Fig. 2.

3. Conclusions

Due to this set of tools, the initial study evaluated the impact of changing certain geometrical variables of a baseline Rod End part, on the stress, contact pressure and displacement. There is also a high potential for further development of the tool and usage in different projects. Specifically, the following conclusions were reached:

1. it was proven that this approach is correct and financially justified,
2. this set of tools enabled significant reduction in the time to perform a study of different parameters of the Rod End geometry,
3. it was found that varying the Rod End width, outer diameter or inner diameter may hold the largest potential for optimizing the results of the part,
4. the tools may be used as the basis for further, more detailed investigation to optimally configure the Rod End parts to prolong their LCF life,
5. the SAMCEF autorun tool and HYPERVIEW post-processing tool were later used on multiple other projects to enable further reduction in manual processing time, contributing to on-time delivery of the projects.

4. References

[1] Young W. C., Budynas R.G., *Roark's Formulas for Stress and Strain*, 7th Edition, McGraw-Hill, 2001, Chapter 14.

[2] Ashok P. Nadkarni, The TCL programming language – A comprehensive guide, 2017.

[3] Altair Support online, Hypermesh 2020 Script reference guide.

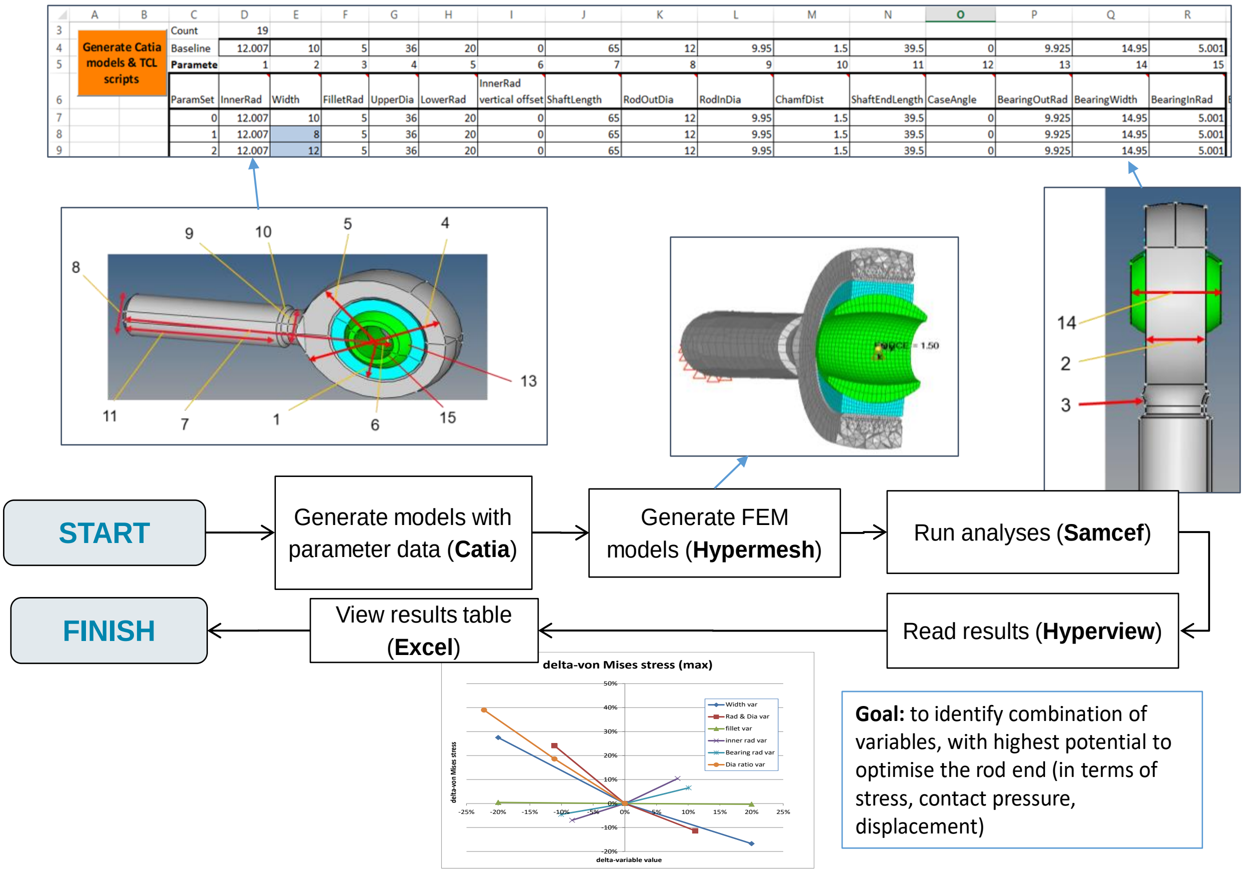

Fig. 2. Overall workflow