

APPLICATION DEVELOPMENT FOR ANALYSIS OF BEVEL GEARS ENGAGEMENT USING FEM

Paweł FUDALI, Jacek PACANA

Katedra Konstrukcji Maszyn, Politechnika Rzeszowska
Al. Powstańców Warszawy 8, 35 - 959 Rzeszów, fax (0-17) 854 31 16, pacanaj@prz.edu.pl

Summary

Knowledge of diagnosed object includes issues of design, manufacture and operation of the facility. One of the objectives of production and structural diagnosis is to identify sources of potential threats and thus in this paper will be presented computer tools for trace detection of gears engagement. In this way it is possible to assess the correctness of their cooperation, determine the causes of high stress, and also reduce the number of tests on a real object.

The study contains selected aspects of creating an application used to facilitate the analysis of gears using the finite element method. There was developed software application, plug-in extending the capabilities of ABAQUS, which was written in Python programming language. Libraries available in the ABAQUS package were used to design a graphical interface for a created application as well as to implement the respective functions of performing specific tasks. Thanks to this application it is possible to easily prepare the model for simulation, as well as to display desirable results, including determined traces of gears cooperation.

Keywords: Design diagnostics, FEM, bevel gears

OPRACOWANIE APLIKACJI DO ANALIZY WSPÓŁPRACY KÓŁ ZĘBATYCH STOŻKOWYCH W ŚRODOWISKU MES

Streszczenie

Wiedza o obiekcie diagnozowania obejmuje zagadnienia z dziedziny projektowania, wytwarzania i eksploatacji obiektu. Jednym z celów diagnostyki wytwarzania oraz konstrukcyjnej jest identyfikacja źródeł możliwych zagrożeń i dlatego w niniejszym artykule zostaną przedstawione narzędzia pozwalające na komputerowe rozpoznawanie śladu współpracy kół zębatych. W ten sposób możliwa jest ocena poprawności ich współpracy i ustalenie przyczyn zwiększonych naprężeń, a także minimalizacja liczby testów na obiekcie rzeczywistym.

Przedstawione opracowanie zawiera wybrane aspekty tworzenia aplikacji służącej do ułatwienia przeprowadzania analiz przekładni zębatych z wykorzystaniem metody elementów skończonych. Została opracowana aplikacja w języku Python rozszerzająca możliwości programu ABAQUS. Dzięki aplikacji możliwe jest w prosty sposób przygotowanie modelu do symulacji, jak również wyświetlenie pożądaných wyników m. in. w postaci wyznaczonych śladów współpracy kół zębatych.

Słowa kluczowe: Diagnostyka konstrukcyjna, MES, przekładnia stożkowa

1. INTRODUCTION

Design of mechanical equipment is often realized using computer computational programs, among which the great interest in the finite element method (FEM) [2, 4]. Numerical calculations using the FEM structures are often very complicated and time-consuming. You must carry out a number of preparatory activities for the calculation, so-called 'preprocessing', as well as perform processing and analysis of the results, so-called "postprocessing". In the case of the analysis and diagnosis of gear for each model every time you need to define the boundary conditions, load and define materials. Similarly, in the case of the analysis of simulation

results. Creating charts, generating aggregated pressure or trace cooperation is a long process. Many of these steps can be reduced by using process automation of numerical analysis. In order to achieve this task, the application has been developed, extending the capabilities of the ABAQUS program.

1.1. Model of analyzed gear

Sample calculations were performed for the bevel gear of aggregates box drive of aircraft engine. Basic data of transmission is: transverse module $m=1,954\text{mm}$, teeth number of pinion $z_1=6$, teeth number of wheel $z_2=59$, shaft angle $\Sigma=70^\circ$, Hypoid

offset $a=0\text{mm}$, face width $b=13,665\text{mm}$, transverse pressure angle $\alpha_n=20^\circ$, mean spiral angle of pinion $\beta_{m1}=35^\circ$. In the present case full gear models were made in CATIA software, and then were cut, limiting each wheel bevel gear only to seven teeth. Prepared in such way models were written to step format. Geometric restriction of wheels enabled to reduce the number of finite elements used in the process of digitization, but did not affect the correctness of the results. Transmission model and its section is shown in Fig. 1.

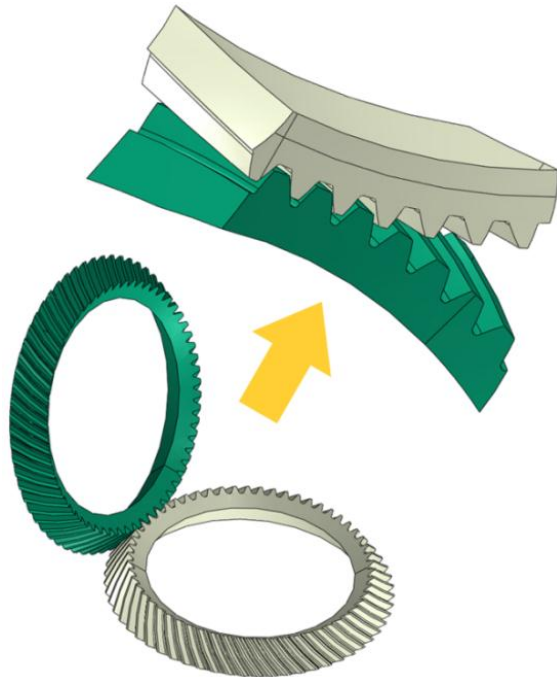


Fig.1. Model of analyzed gear and its section used for the simulation

Models should be made and saved in millimeters. Accordingly, the readout values of the stresses and pressures are given in MPa.

2. PLUG-IN APPLICATION

The plug was divided into two sub-programs, one called *przygotowanie_obliczen*, is used to assist in creating an computation, the other: *wyniki_obliczen* is helpful in processing the results and facilitates fast and clear presentation of selected results of the simulation models including: the state of stress, surface pressure, the contact trace, graph of irregularity work. The plugin is a collection of several text files with the .py extension, in which instructions are written in Python language. In ABAQUS environment there is located compiler that when you start the main program it compiles .py files located in the work folder to .pyc format. The compiler recognizes all commands of Python programming language syntax. Furthermore, the

"pure" Python language is supplemented by a number of libraries, with commands, functions, and other elements of the programming language syntax, which relate directly to the command supported by the ABAQUS program. The plug-in is visible in the ABAQUS program if two conditions are met:

- 1) files with scripts should be placed in one of the specified directory:
 - <abaqus_dir> / abaqus_plugins
 - <user folder> / abaqus_plugins
- 2) At least one of the files should contain the command registering plug-in Abaqus environment.

The following is a fragment of the script responsible for registering a plug-in in ABAQUS program, responsible for presentation the pressure on the surfaces of the teeth [1].

```
thisPath = os.path.abspath(__file__)
thisDir = os.path.dirname(thisPath)
```

```
toolset=getAFXApp().getAFXMainWindow().getPluginToolset()
toolset.registerGuiMenuButton(
buttonText='przekladnia|sumanaciski',
object=sumarycznenaciski_plugin(toolset),
messageId=AFXMode.ID_ACTIVATE,
kernelInitString='import obliczeniaim',
applicableModules=ALL,
author='Pawel Fudali, PRz',
description='modul sluzacy wyswietlaniu
naciskow',
)
```

2.1. Preparation of the calculations

Numerical calculations of gears by using ABAQUS program, are carried out with the use of models imported into it, and prepared at any other CAD program. Preparation of the calculations consists of several basic, routine but necessary steps as defining the boundary conditions, but also a number auxiliary operations that must be completed for each imported model like: setting the models, division of surfaces, etc. Additional operations significantly prolong the preparatory calculations, and their numbers may cause mistakes. Automating this process will keep the same settings for multiple variants carried out calculations and significantly shorten their time. In ABAQUS program it typically includes the following basic steps:

- import the model
- define the material properties and assigning them to specific models,
- define the loads and restraint,
- generate the finite element mesh,
- specify the parameters of the simulation,
- define the contact conditions between elements.

To automate these steps, you can use a specially prepared subroutine as a plug-in in preprocessor of

ABAQUS program. This routine includes above steps and step by step guides you through the process of preparing calculations. Launching plug is possible by selecting *przekladnia-przygotowanie obliczen* from the main menu of the program in the Plug-ins tab. After launching the application the main window appears (Figure 2), in which the user can perform various stages of preparation of the model calculations. These steps should be executed in the order in which they were defined. All operations in the presented plugin you can according to your needs change, they are all available on the feature tree.

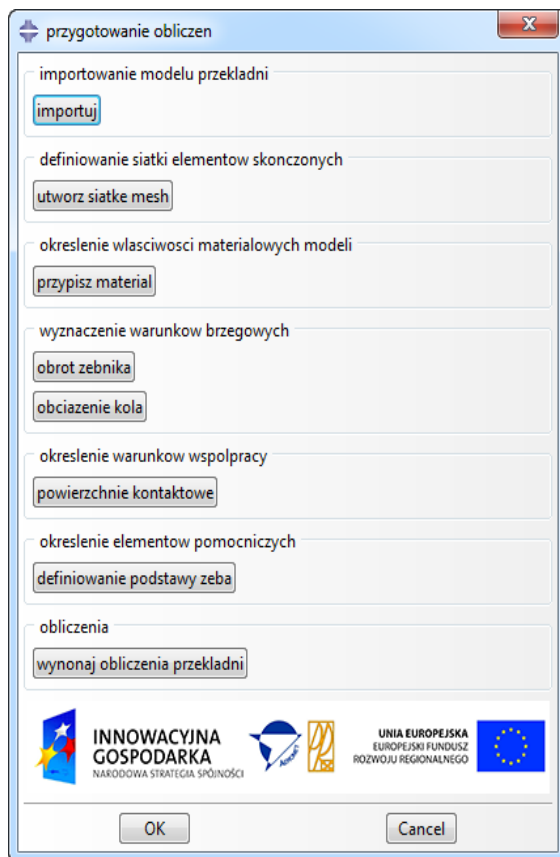


Fig. 2. The main window of the program for the preparation of calculations

Each stage of the model preparation listed in the main window during start-up displays a window for entering parameters or indicating objects. Among the steps required to prepare the analysis include division of the continuous model into finite elements. If you select 'utworz siatke mesh' window appears (Fig. 3), which allows to define the basic parameters of the mesh.

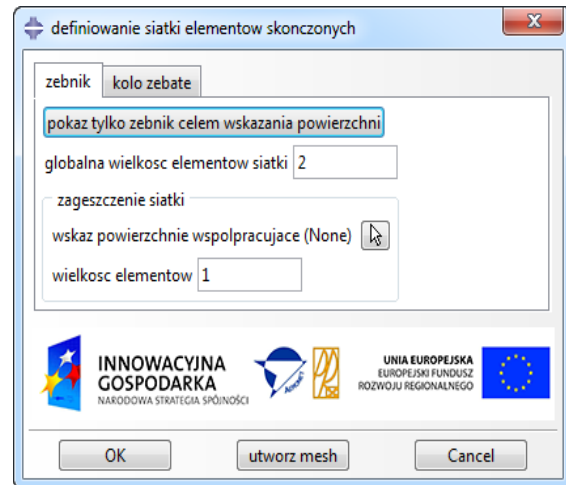


Fig.3. A window for entering basic parameters of mesh

In a similar way it is necessary to input the information for each of the stages.

2.2. Displaying results

In order to analyze and present the results of the calculations of the gear it has been created second subprogram. It is available by selecting '*przekladnia-wyniki obliczen*' from the Plug-ins top menu. When you select it there appears the main window shown of Fig.4. This plugin allows you to: load file with calculations, generating stress and pressure for each step, generating summary pressure state (Fig. 5), determining momentary contact trace (Figure 6), determining total trace of cooperation (Fig. 7), generating a graph of irregularity movement of the gear (Fig. 8) and the stress distribution at the base of the tooth (Fig. 9).

Displaying momentary stresses, pressures or contact trace is an activity that can be done relatively easily using a graphical interface. As for the cumulative pressure and total trace of cooperation situation becomes complicated. In the presented application there was used the following algorithm: all values were analyzed for each time point and for case of summary pressure the maximum value of pressure is stored (in the case of total trace the smallest COPEN parameter value) for each of the nodes of the sides of the teeth, and then the stored values have been applied to model of wheel gear.

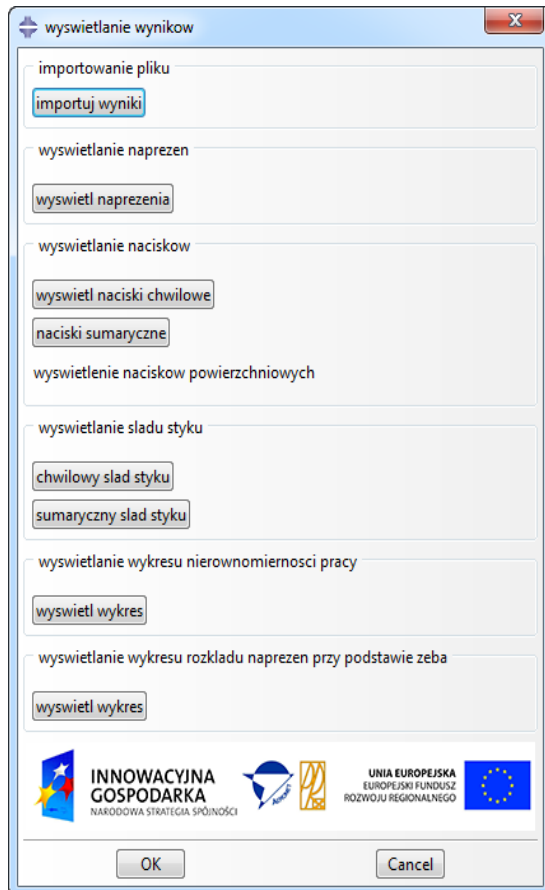


Fig. 4. The main window to display results

In the case of a showing trace of cooperation ink thickness value means that there are marked in red places where the distance from the pinion gear wheel is smaller than a predetermined value.

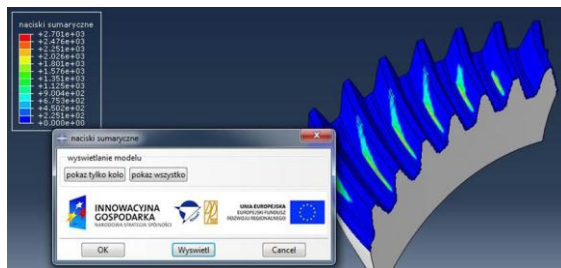


Fig. 5. The summary pressure

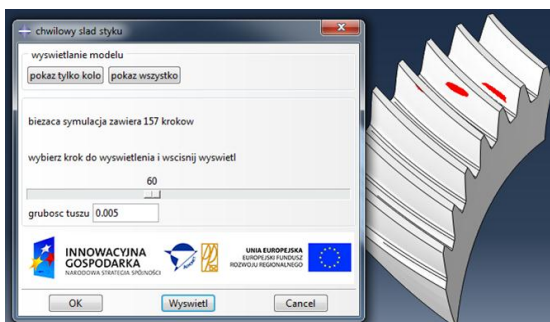


Fig. 6. Momentary trace of cooperation



Fig.7. The total trace of cooperation

Graph of irregularity of the gear unit is based on the equation (1) [3, 5] and for the investigated example using the plug-in is shown in Fig. 8.

$$\Delta\varphi^{(2)}(\varphi^{(1)}) = \varphi^{(2)}(\varphi^{(1)}) - (Z_1/Z_2) \cdot \varphi^{(1)} \quad (1)$$

where:

- $\varphi^{(1)}$ – rotation angle of the pinion,
- $\varphi^{(2)}$ – rotation angle of the wheel,
- Z_1 – teeth number of pinion,
- Z_2 – teeth number of wheel

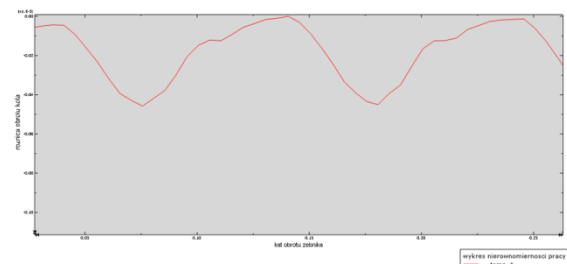


Fig. 8. Graph of irregularity work in gear

The application recognizes when there was not yet a full engagement, so when it creates a chart it skips first few steps of analysis.

2. CONCLUSIONS

The use of scripting in ABAQUS greatly simplifies and speeds up the work for repetitive tasks. On the other hand, it allows to obtain results that cannot be displayed by using the tools available in the program interface. The amount of work put into preparing the script is compensated by the simplicity and usability created plug. Determined gear characteristics allow us to assess the correctness of their work and reduce the number of tests on a real model during its design. The calculation results obtained by the application presented in this article will make it easier to carry out and analyze the results of diagnostic tests of cooperation traces of aircraft gears.

Acknowledgement

Financial support of Structural Funds in the Operational Programme - Innovative Economy (IE OP) financed from the European Regional Development Fund - Project "Modern material technologies in aerospace industry", No POIG.0101.02-00-015/08 is gratefully acknowledged.

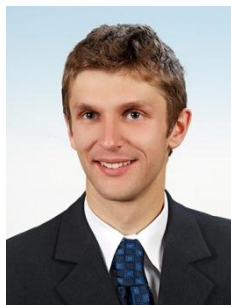
REFERENCES

- [1] Barbero E. J.: *Finite Element Analysis of Composite Materials: using Abaqus*, CRC Press, 2013.
- [2] Budzik G., Kozik B. Pacana J.: *Defining influence of load conditions on distribution and value of stresses in dual-power-path gear wheels applying FEM*, Aircraft Engineering and Aerospace Technology: An International Journal 85/6 (2013) 453–459, pages 453-459.
- [3] Marciniak A., Sobolewski B.: *Method of Spiral Bevel Gear Tooth Contact Analysis Performed in CAD Environment*; 7th International Conference AIRTEC, Aircraft Engineering and Aerospace Technology, Vol. 85 Iss: 6, p.467 – 474.
- [4] Pacana J.: *Development of Bevel Gear Motion Transmission Graphs with FEM*, Key Engineering Materials Vol. 490 (2012), Trans Tech Publications, Switzerland 2012.
- [5] Pisula J, Płocica M.: *Numerical Model of Bevel Gears Cutting by Duplex Helical Method*, Key Engineering Materials: Fundamentals of Machine Design, Vol. 490(2012), pp. 237-246.



design methods and computations of mechanical structures, in particular gears.

Jacek PACANA is an employee of the Department of Mechanical Engineering - Rzeszow University of Technology. In the research work deals with the design and numerical analysis of harmonic drives and their experimental research. He specializes in computer-aided



Pawel FUDALI is an employee of the Department of Mechanical Engineering - Rzeszow University of Technology. In the research work deals with the design and analysis of equipment in the field of biomechanics and rehabilitation. He specializes in numerical computations using modern CAE tools.