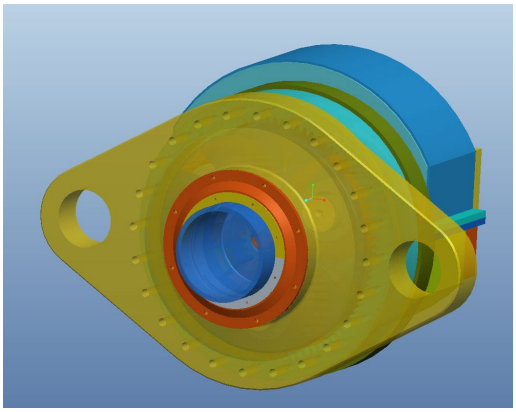
Proposal for Master's thesis in Applied Mechanics

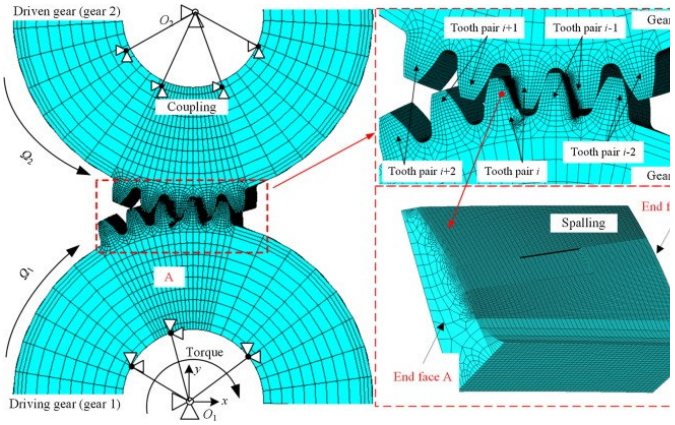
Finite element modelling of a generic 2MW wind turbine gearbox

Aim

The aim of the project is to develop a finite element (FE) model of a generic 2MW wind turbine driveline. The model is useful to consider various stiffness components and is regarded as the most suitable and effective tool as computer performance increases. The focus of this project is to investigate the effect of gear-mesh misalignment, gear-body flexibility, shafts, bearing stiffness, gearbox stiffness and gear-mesh flexibility.



Source: Nagaiah, A. (2019). A 2 MW Wind Turbine Gearbox Layout. Chalmers University of Technology, Sweden. Technical report.



Source: Chen, K., Ma, H., Che, L., Li, Z., & Wen, B. (2019). Comparison of meshing characteristics of helical gears with spalling fault using analytical and finite-element methods. Mechanical Systems and Signal Processing, 121, 279-298.

Methodology

The general methodology involves developing a full 3D FE model of the planetary and the spur gear systems in ABAQUS. A Pro/Engineer model of a generic 2MW gearbox is available at the division. This model will provide the geometry of the gearbox that can be imported into ABAQUS. Development of the FE model includes the following key steps:

- Sensitivity analysis of the different *Finite Elements* available in ABAQUS and identification of the one most suited to the requirements.
- Mesh convergence study to determine how many elements are required in a model to ensure that the results of an analysis are not affected by changing the size of the mesh. This may include both *h* and/or *p* convergence study.
- Modelling contact interaction to solve gear-meshing problem.
- Accurate modelling of the roller bearings as it is essential for static and dynamic analysis of the gear system.

Every key step must be numerically verified and well documented. A few optional specialized topics that may be considered under the project includes:

- Study of component flexibility (including bearings) and its effect on gear-mesh misalignment.
- Creating ABAQUS *super-elements* representing flexible bodies that can be imported in multibody simulation software such as SIMPACK for rapid dynamic simulations.
- Modelling the lubrication process in gear trains to simulate oil presence in gear trains.
- Root bending fatigue analysis of the gears.

Expected outcome

The FE model is to serve as a high-level simulation model capable of providing insights into the internal stresses of the various components. This will help in identifying critical components and estimating their fatigue life. The cyclic variation of tooth bending stress, which creates a varying gear-mesh stiffness, is a source of fatigue crack initiation. The project it expected to be useful for predicting the fatigue life of the critical drivetrain components for nominal/design operational loads. The FE model is also expected to provide information about the torsional stiffness of the gearbox with and without misalignment.

Student background: This project is suitable for (preferably two) students with an interest in computational mechanics, contact mechanics, machine dynamics, wind turbines dynamics.

Supervisors and contacts:

Saptarshi Sarkar, e-mail: ssarkar@chalmers.se
Håkan Johansson, e-mail: hakan.johansson@chalmers.se
Examiner: Viktor Berbyuk, e-mail: viktor.berbyuk@chalmers.se