Student assistant/ Studentische Hilfskraft

Data generation for training Machine learning algorithm using Abaqus models of composite bolted joints

Description

 Large Composite parts are often joined together using bolted joints and accurate Finite element simulation of such joints is essential for designing composite structures. The time required to simulate a single composite joint is extremely high because of which it becomes impossible to simulate large structures with field of bolts. The project aims to address this problem by using shell-beam models, supplemented with Machine learning algorithms to reduce the computational time substantially without compromising on accuracy. The aim of the current task is to develop training data for the machine learning algorithm from Abaqus models. The tasks can be tailored to some extent according to the knowledge and skills of the student

Tasks

- Python Scripting in Abaqus
- MATLAB Programming
- Stress Analysis in Abaqus

Prerequisites

- Basic programming skills in Python and MATLAB
- Some experience with Abaqus or FEM solver
- Basic Knowledge of FEM and ML is a plus
- Motivated and Independent way of working

Contact Person

Aditya Bansod, M. Sc.
Institut für Statik und Dynamik
Appelstr. 9A
30167 Hannover
Tel.: 0511-762-17426
E-Mail an: a.bansod@isd.uni-hannover.de