Meso-scale-Cantilevers-Abaqus-Automation Documentation

By Robert J Scales & Rory Rose

# Introduction

This code is primarily designed to allow one to serially run multiple ABAQUS simulations for the resonant fatigue of cantilevers, where the parameters between jobs can be altered by hand or handled via MATLAB, thus allowing for basic parametric studies to be quickly performed.

The base code that is stored within the code on the GitHub uses cantilevers of the following design:

INSERT PICTURE OF MODEL HERE

With this design of cantilever, the current available methods are the following:

1. Standard static – This is used for modelling how a concentrated force acting on the centre of the cantilever paddle will deform the cantilever; thus, giving one the stiffness value of said point for that cantilever.
2. Frequency eigenmodes - …
3. Direct dynamic - …
4. Calibration of standard static – This is like the *standard static* method, however now the load point is varied along the cantilever axis along the paddle. Thus, outputting a predicted stiffness vs loading point location plot, and allowing one to predict where their experimental loading location could be.

# Structure of the code

The following flowchart describes the MATLAB flow in which the code operates, with the **other** flowchart describing how the MethodFile.py should be constructed, and how it is run within ABAQUS.

Diagram, text

Description automatically generated