Shaft Parametric FEA with deal.II

INPUT

The following diagram presents the basic idea behind the step_18_cyl_force_modeal_v3 program.

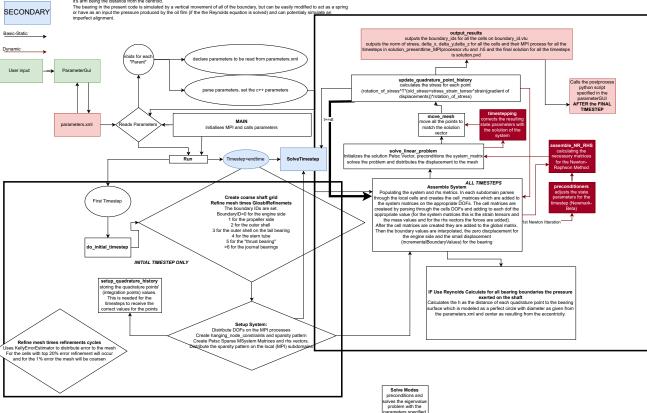
The code in question receives input from an parameters.xml file, which can be maniplulated by deal.II's ParameterGui. The parameters include the length of the shaft, the radius, material properties, engine specifics, ship speed, solver parameters, and bearing parameters. For the time being only one bearing is beign considered, but the code can be easily modified to receive multiple bearings

The FEA analysis can be dynamic, which includes inertia forces and damping, or static. The goal of the goal is to be explicitly accessible enclosurements and purpose of the goal PetsC matrices and solvers. In the code the shaft is divided into partitions (MPI cores) which can be examined in Paraview.

The code is written and used in ubuntu OS, it uses cmake to create the Makefile and the libraries with which deal. It is compilled with are: HDF5 (used to output the results for postprocessing), MPI, Petsc, Arpack, MPI, ScalaPack, petsc4py, with complex numbers

The executable also calls the python postprocessing script which in turn outputs the maximum norm of stress per dx of the shaft and a OUTPUT 3D surface plot that examines the outer shell points, ploting their norm of stress in the z-axis, angle=0 is at z=radius of the shaft In the code the Engine Power applied on the propeller side of the shaft is calculated based on the propeller law and divided by the

ship's velocity to calculate the froce. The torque is also calculated based on the propeller law and applied on each point as a force with it's arm being the distance from the centroid.



in parameterGUL but consumes ALL the RAM, not viable for fine meshes

if eigenmodes==True