# FLOW 3D SIMULATIONS

Start by setting up a new folder titled something like CFD on the desktop of the host computer to store the simulations and their results. Next activate FLOW3D. Then under NAVIGATOR click on FILE and create a new WORKSPACE tilted something like CASES and put it the CFD folder. Next under NAVIGATOR click on CASES and then FILE to create a new simulation titled something like MINE. Then click on MINE and then on the MODEL SETUP menu to develop the simulation. Under GENERAL set things like FINISH TIME, TYPE OF INTERFACE, SIMULATION UNITS and PRECISION. Under PHYSICS set things like GRAVITY, GMO, VISCOSITY and TURBULENCE. Under FLUIDS click on MATERIALS and load the simulation fluid. Under MESHING & GEOMETRY right click on MESH and add a cell block. Expand the x and y and z menus on the left. Set the mesh planes and the number of cells between the planes. Next click on BOUNDARIES to set the BOUNDARY TYPES. Use the WAVE boundary for bodies in waves. Set the water depth and define the wave. Use the SPECIFIED PRESSURE boundary for air over water problems. Make sure to set FLUID FRACTION to 0 and void STAGNATION PRESSURE. Use the OUTFLOW boundary when fluid or waves move from inside to outside. For wave problems allow

fluid to enter the boundary. Use the WALL boundary for WALLS. Next click on INITIAL and set the pressure in the mesh to be hydrostatic in the z direction and set the water depth in the mesh. Next click on STL and add the General Moving Objects or GMOs. Double click on COMPONENT to show the GMO menus. Then click on SUBCOMPONENT and move the GMO to the proper location using the TRANSFORMATIONS sub menu. Next click on COMPONENT PROPERTIES and set the DENSITY and the DOFs. Next click on OUTPUT and set the output time step. Then click on NUMERICS and set LIMITED COMPRESSIBILITY. Next click on SIMULATION and SIMULATE and RUN to run the simulation. Next click on ANALYZE to see the simulation results. Load the NAME.flsfrg file. Check results using the 3D and PROBE menus. Under 3D use default settings and click on STL and RENDER to get to a 3D display. Click on the CAMERA icon to see a 3D movie of the GMO motion. Under PROBE click on GENERAL HISTORY and GRAPHICAL OUTPUT. Scroll down the list of outputs and pick an item of interest such as a particular GMO motion. Click on RENDER to see a graph of the motion. Click on TEXT to see the numbers in the graph.