Boat
Flow Sim

A. Enable Flow Simulation.
Step 1. If necessary, open your ASSEMBLY file.

Step 2. If necessary, enable Flow Simulation, click the flyout of Options on the Standard toolbar and click Add-Ins.

Step 3. In the dialog box, scroll down to Flow Simulation and place a check in the check box under Active Add-Ins and Start-Up, Fig. 1. Click OK.

B. Create Project.
Step 1. Click Flow Simulation tab on the Command Manager toolbar.

Step 2. Click Wizard on the Flow Simulation toolbar.

Step 3. Use Project(1) for Project name, Fig. 2.

Step 4. Click Next.

Step 5. Unit System: Select SI (m-kg-s) for Unit system, Fig. 3.

Step 6. Under Parameter, set Velocity units to Mile/hour, Fig. 3.
Step 7. Scroll down Parameters, expand **Loads & Motion** and set **Force** units to **Gram force**, Fig. 4. Gram force unit is p.

Step 8. Click Next.

Step 9. **Analysis Type:**
Under Analysis type, select **External**, Fig. 5.

Step 10. Check **Exclude cavities without flow conditions** and **Exclude internal space**, Fig. 5.

Step 11. Set **Reference axis** to Z, Fig. 5.

Step 12. Click Next.
Step 13. **Default Fluid:**
Expand Liquids, scroll down and select **Water**, then click **Add**, **Fig. 6**.

Step 14. Click Next.

Step 15. **Wall Conditions:**
Use the default values for wall condition, **Fig. 7**.

Step 16. Click Next.

Step 17. **Initial Conditions:**
Under Velocity parameters set **Velocity in Z direction** to **-15 Mile/h**, **Fig. 8**. (click and key-in **-15**).

Step 18. Click Finish.
C. Computational Domain.

Step 1. Click Tools Menu > Flow Simulation > Computational Domain.

Step 2. Set values as shown here and Fig. 9.

\[
\begin{align*}
X_{\text{max}} & = 0.09 \\
X_{\text{min}} & = 0 \\
Y_{\text{max}} & = 0.033 \\
Y_{\text{min}} & = -0.06 \\
Z_{\text{max}} & = 0.13 \\
Z_{\text{min}} & = -0.4
\end{align*}
\]

Step 3. Set At X min to Symmetry, Fig. 10.

Step 4. Click OK in Property Manager.

Step 5. Click Front on the Views toolbar (Ctrl-1). Use Z key to zoom out to view domain, Fig. 11.

Step 6. Only half the boat should be inside the domain, Fig. 12. This will reduce the solver time. Solver can take several minutes.
D. Insert Global Goals.
Step 1. Click Isometric on the Standard Views toolbar. (Ctrl-7)
Step 2. Click the Flow Simulation tab in the Feature Manager, Fig. 13.
Step 3. Click Flow Simulation Features on the Flow Simulation toolbar and click Global Goals from the menu.
Step 4. Drag the edge of the Feature Manager to the right to expand, Fig. 13.
Step 5. In the Global Goals Property Manager: scroll down the Parameters to Force (Y), Fig. 13 and check: Force (Y) Force (Z) click OK.
Step 6. Rename the goals to Lift and Drag. To rename, click Force (Y), press F2 key and key-in Lift, Fig. 14. Rename Force (Z) to Drag.
Step 7. Save. Use Ctrl-S.

E. Run Analysis.
Step 1. Click Run on the Flow Simulation toolbar.
Step 2. Click Run in the Run dialog box, Fig. 15.
**F. Solver.**

Step 1. In the Solver dialog box you can view Calculation time left, Fig. 16. The CPU time runs around 2-3 minutes.

Step 2. When the calculation is done view the drag in the Solver, click Insert Menu > Goal Table, Fig. 17. Our drag was -914.7.

Step 3. Close the Solver dialog box, click File Menu > Close.

Step 4. Save. Use Ctrl-S.

**G. Surface Plots.**

Step 1. Right click Computational Domain in Flow Simulation tree and click Hide, Fig. 18.

Step 2. Click Surface Plot on the Flow Simulation toolbar.

Step 3. In the Property Manager, set: under Selection, Fig. 19 check Use all faces under Display select Contours under Contours click Adjust Minimum and Maximum 120900 for Maximum pressure 76100 for Minimum pressure click OK.

Step 4. After viewing the Surface Plot, expand Results in Flow Simulation tree and expand Surface Plots. Right click Surface Plots 1 and click Hide, Fig. 21.

Step 5. Save. Use Ctrl-S.
**H. Flow Trajectories.**

Step 1. Click **Flow Trajectories** on the Flow Simulation toolbar.

Step 2. In the Flow Trajectories Property Manager:
- under Starting Points, **Fig. 22**
- Select References by selecting all the faces of the hull on left side and rear face, **Fig 23**. To view rear, hold down middle mouse button (wheel) and drag to rotate view

  Number of Points ✂️ 20

  under Appearance
  - Draw Trajectories As 🌊 Lines
  - Line Width ✗ 2
  - click OK ✔️.

Step 3. After viewing the flow trajectories, right click **Flow Trajectories 1** in Flow Simulation tree and click **Hide**, **Fig. 25**.

Step 4. Save. Use Ctrl-S.
1. Goal Plot.

Step 1. Click **Goal Plot** on the Flow Simulation toolbar.

Step 2. In the Goal Plot Property Manager:
- under Goals, **Fig. 26**
  - check **All**
- under Options
  - click **Export to Excel**

An Excel file is opened. Note the Drag, **Fig. 27**.
Click the **Drag tab** at the bottom of the Excel file to view the chart, **Fig. 27**.
Close the Excel file.
Click OK.

**Fig. 26**

**Fig. 27**

**Fig. 28**
**J. Animate Flow Trajectories.**

Step 1. **Click Right** on the Standard Views toolbar. (Ctrl-4)

Step 2. **Right click Flow Trajectories 1** in Flow Simulation tree and click Animation, Fig. 29.

Step 3. At the bottom right of the display in the animation control panel, click **Expand** to expand the panel, Fig. 30.

Step 4. **Click Play** in animation control panel, Fig. 30.

Step 5. **Right click Flow Trajectories 1** in the Animation tree and click **Edit Definition**, Fig. 30.

Step 6. In the Flow Trajectories Property Manager:

- **under Appearance**, Fig. 31
  - **Draw Trajectories As Spheres**
  - **Width** .002
  - **click OK**.

Step 7. **Click Loop** and **Play** in animation control panel, Fig. 30.

  **Click Stop**.
K. Create Animation Movie.

Step 1. In the Animation control panel turn on **Capture Region**  Fig. 33.

Step 2. Drag the red frame capture region in the graphics area to resize/move capture region,  Fig. 34.

Step 3. Click **Record**  Fig. 34.

Step 4. Click **Open Folder** to view Animation AVI file.

Step 5. To exit, click **OK** in Animation control panel.