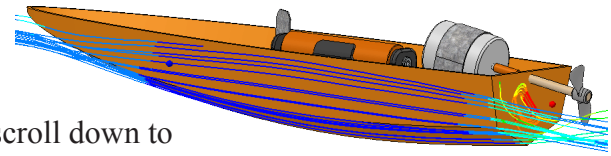
Boat
Flow Sim

A. Set Up.

Step 1. If necessary, open your **ASSEMBLY** file.

Step 2. Click Tools Menu > 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.

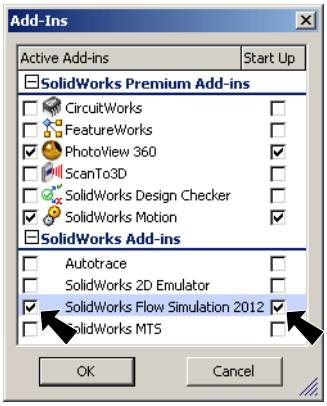
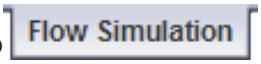


Fig. 1

B. Create Flow Simulation Project.

Step 1. Click **Flow Simulation** tab 

on the Command Manager toolbar.

Step 2. Click **Flow Simulation Main**

on the Flow Simulation toolbar and **Wizard**

from the menu.

Step 3. Use **Default** for Configuration name, **Fig. 2**.

Step 4. Click Next.

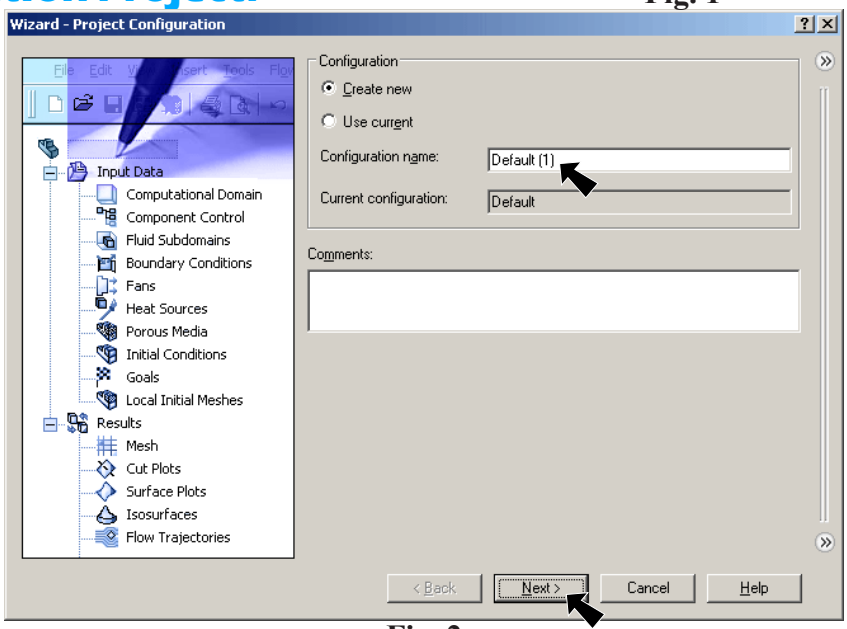


Fig. 2

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**.

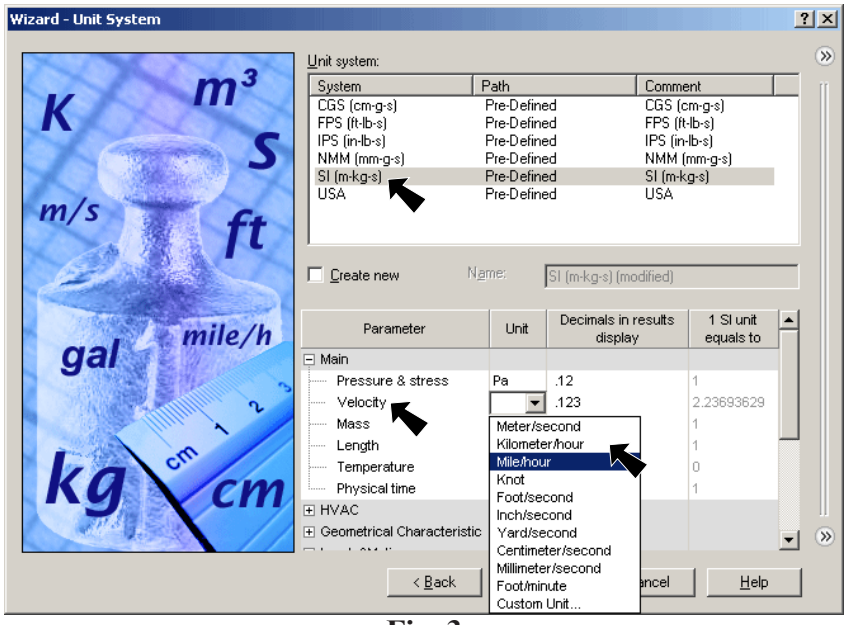


Fig. 3

© Cudacountry.net Tech Ed
http://www.cudacountry.net email:cudacountry@hotmail.com

Step 7. Expand **Loads & Motion,** 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.

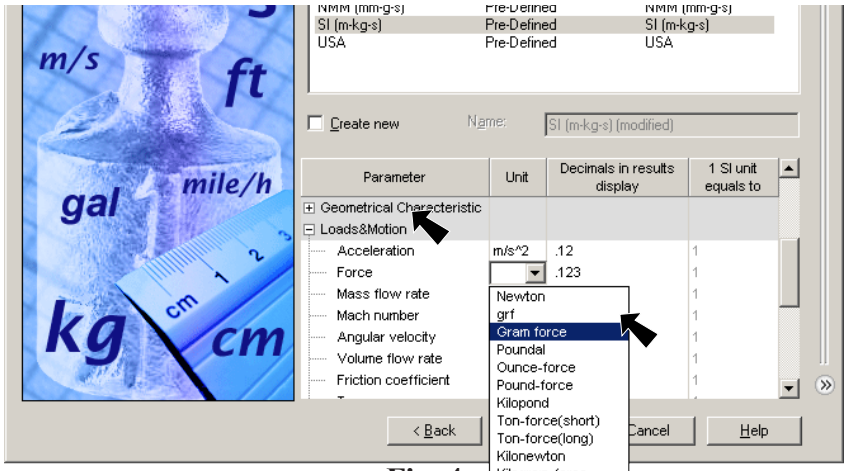


Fig. 4

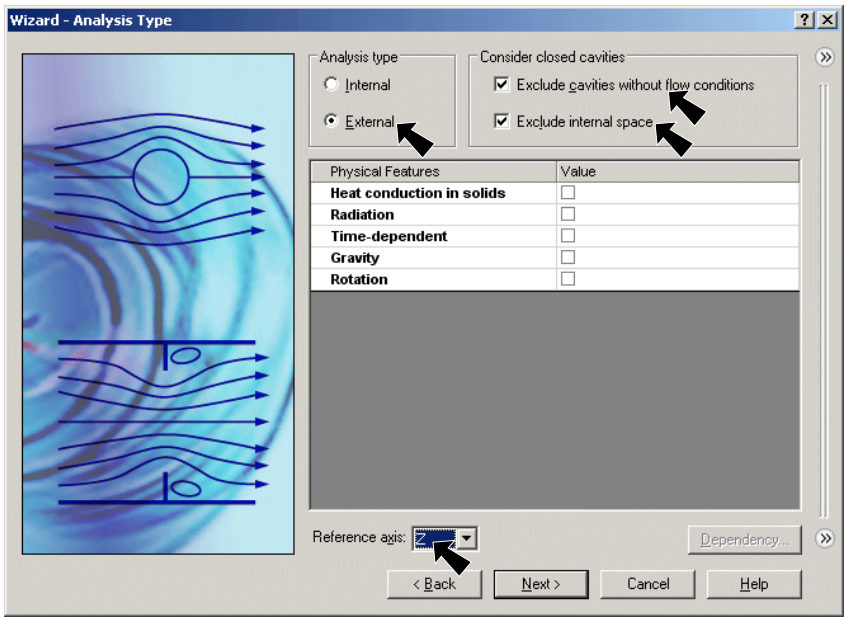


Fig. 5

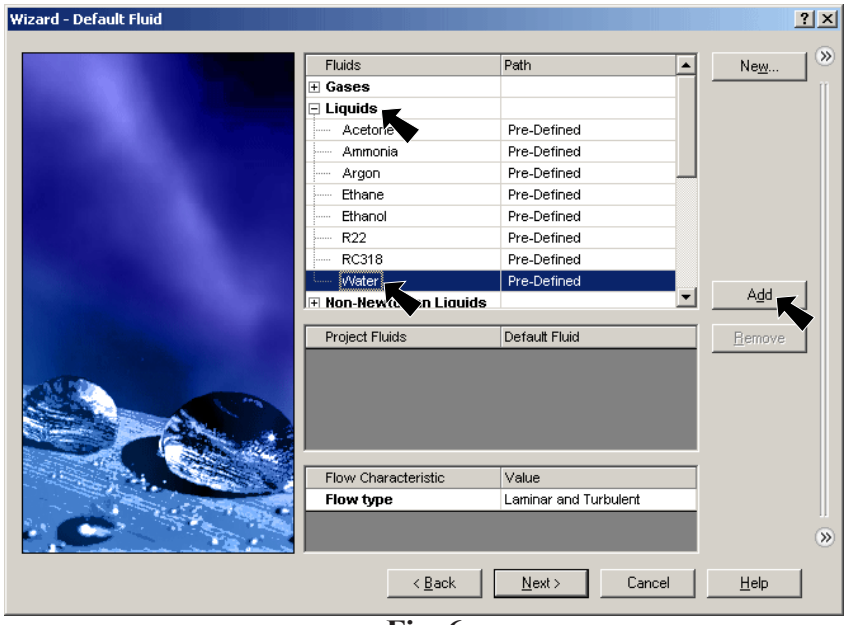


Fig. 6

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** (click and key-in **-15**), **Fig. 8**.

Step 18. Click Next.

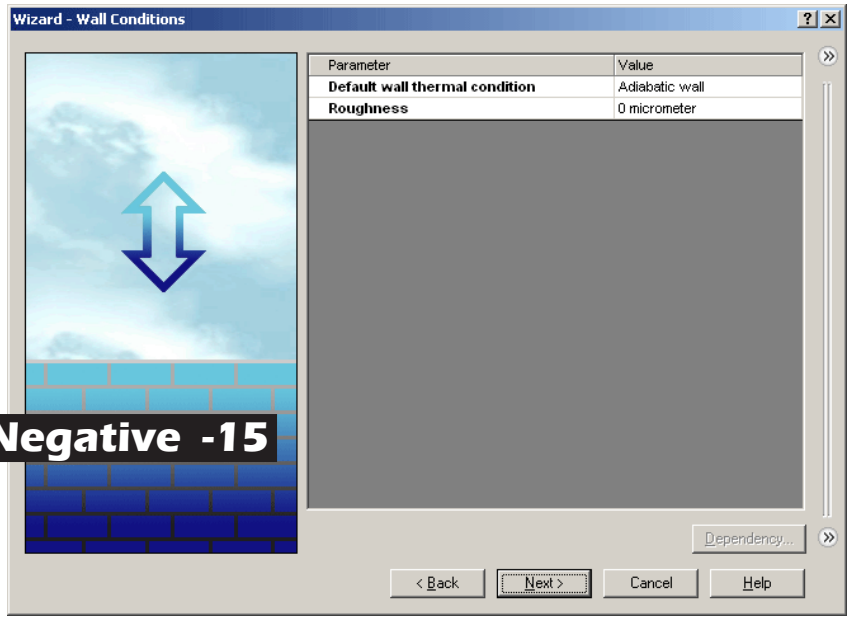


Fig. 7

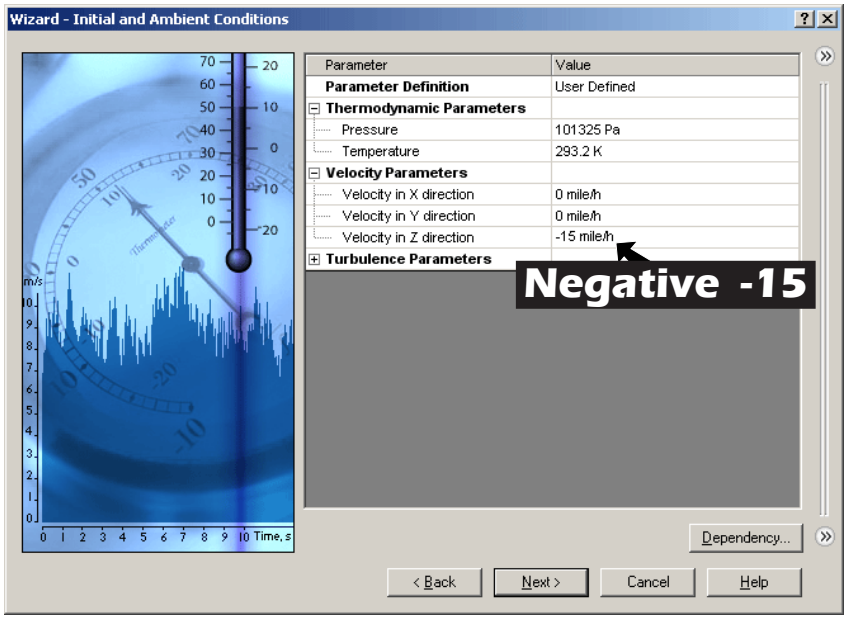


Fig. 8

Step 19. **Results and Geometry Resolution:**
Set **Result resolution to 4**, **Fig. 9**.

Step 20. Click Finish.

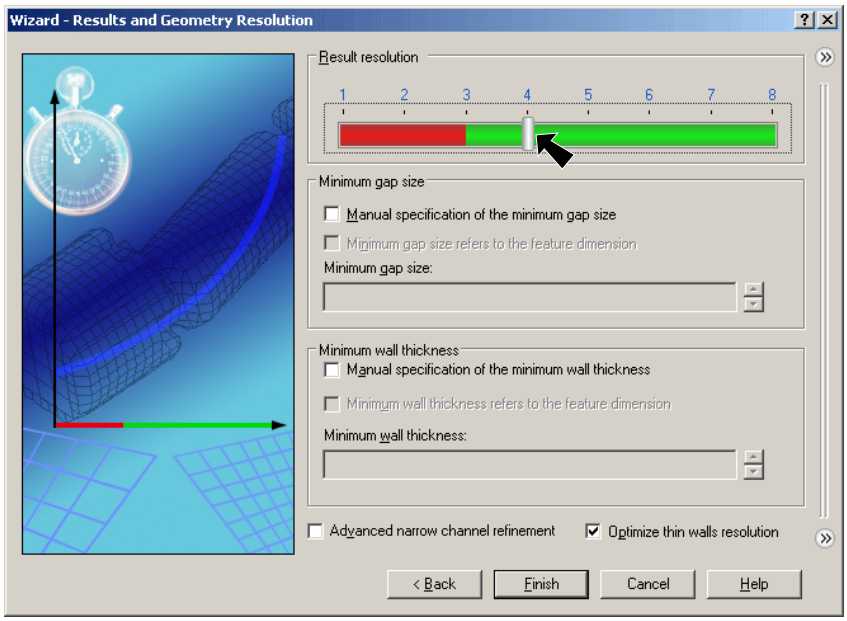


Fig. 9

C. Computational Domain.

Step 1. Click Flow Simulation Menu > Computational Domain.

Step 2. Set values as shown here and **Fig. 10**.

X max .09
X min 0

Y max .033
Y min -.06 **Negative**

Z max .13
Z min -.4 **Negative**

set **At X min** to **Symmetry**, **Fig. 11**

click OK.

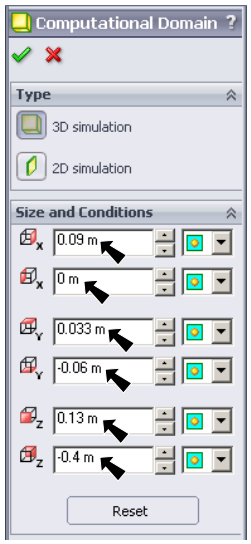


Fig. 10

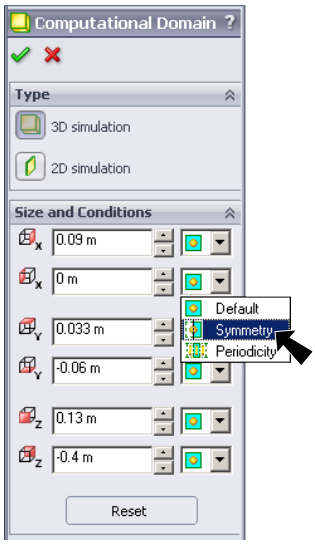


Fig. 11

Step 3. Click **Front** on the Views toolbar (**Ctrl-1**). Use **Z** key to zoom out to see domain, **Fig. 13**.

Step 4. **Only half the boat** should be inside the domain as shown in **Fig. 13**. This will reduce the solver time. Solver can take several minutes.

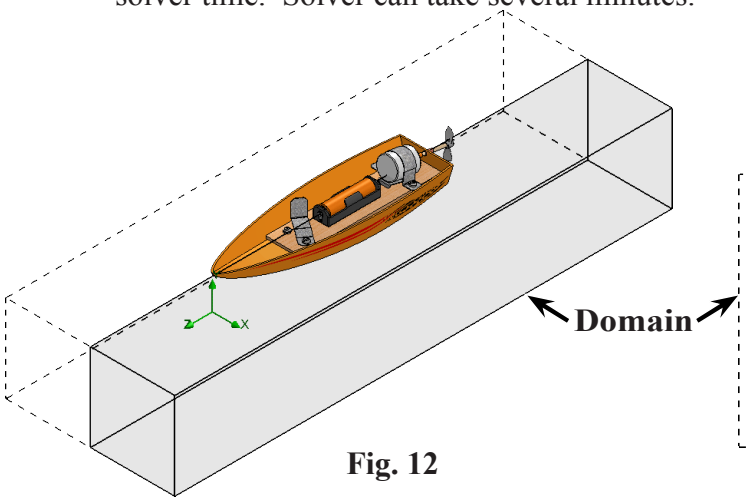


Fig. 12

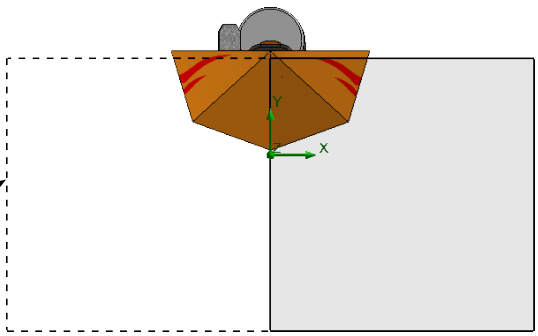


Fig. 13

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. 14**.

Step 3. Click Flow Simulation Menu > Insert > **Global Goals**.

Step 4. Drag the edge of the Feature Manager to the right to expand, **Fig. 14**.

Step 5. Scroll down the Parameters to **Force** and **check:**
Force (Y)
Force (Z) Fig. 14

click OK .

Step 6. Rename the goals to **Lift** and **Drag**. To rename, click and hold over **Force (Y)** and key-in **Lift**, **Fig. 15**. Rename **Force (Z)** to **Drag**.

Step 7. Save. Use **Ctrl-S**.

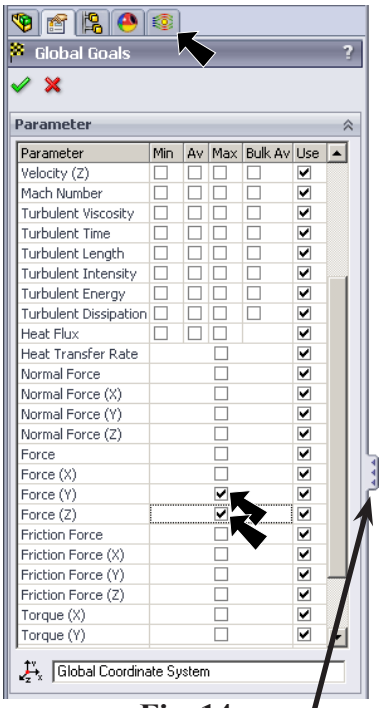


Fig. 14

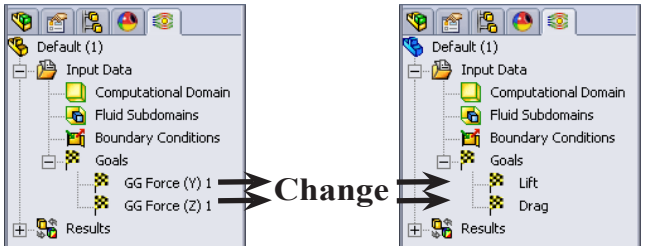


Fig. 15

E. Run Analysis.

Step 1. Click **Flow Simulation Main** on the Flow Simulation toolbar and **Run** from the menu.

Step 2. Click **Run** in the Run dialog box, **Fig. 16**.

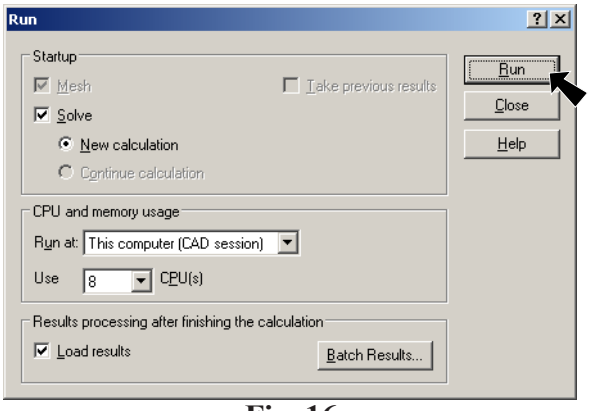


Fig. 16

F. Solver.

Step 1. In the Solver dialog box you can view Calculation time left, **Fig. 17**. The CPU time runs around 1-3 minutes depending on computer.

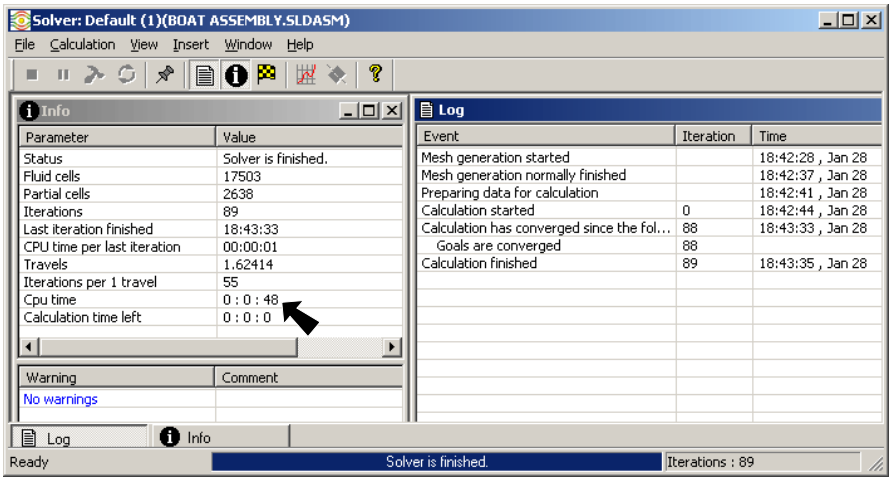


Fig. 17

Step 2. When the calculation is done check the lift. To check lift in the Solver, click **Insert Menu > Goal Table**, **Fig. 18**. Our Drag was **-879.2**.

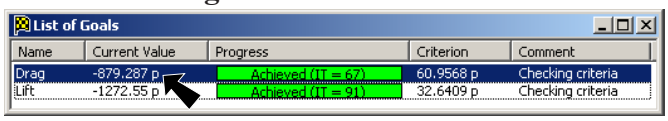


Fig. 18

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. 19**.

Step 2. Click **Flow Simulation Results Features** on the Flow Simulation toolbar and **Surface Plot** from the menu.

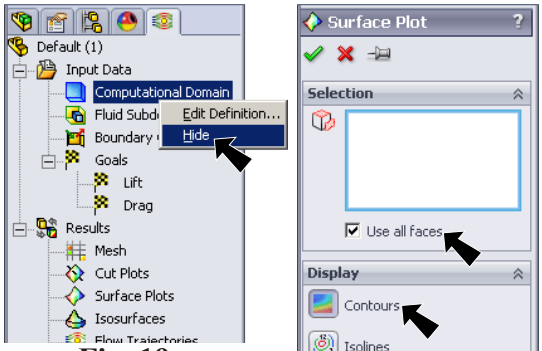


Fig. 19

Fig. 20

Step 3. In the Property Manager set:
check **Use all faces**, **Fig. 20**
under **Contours**
click **Adjust Minimum and Maximum**
Maximum pressure **120900**
Minimum pressure **76100**
click OK.

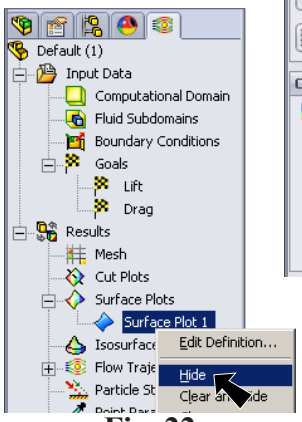


Fig. 22

Step 4. After viewing the Surface Plot hide the Surface Plot. To hide, in Flow Simulation tree expand Results and Surface Plots, then **right click Surface Plots 1** and click **Hide**, **Fig. 22**.

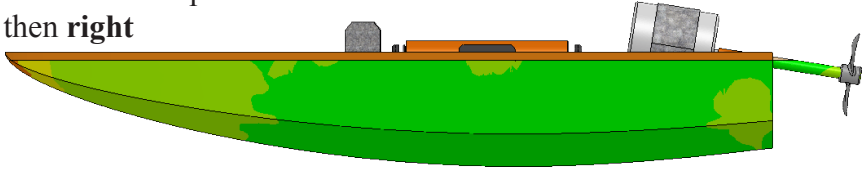


Fig. 21

Step 5. Save. Use **Ctrl-S**.

H. Flow Trajectories.

Step 1. Click **Flow Simulation Results Features** on the Flow Simulation toolbar and **Flow Trajectories** from the menu.

Step 2. In the Flow Trajectories Property Manager set:

under **Starting Points**, **Fig. 23**

Select References . To select References**,** click all the faces of the hull on left side and rear face, **Fig 24**. To view rear, hold down middle mouse button (wheel) and drag to rotate view

Number of Points to **20**, **Fig. 23**

under **Options**

Draw Trajectories As to **Lines**

Line Width **2**

click OK .

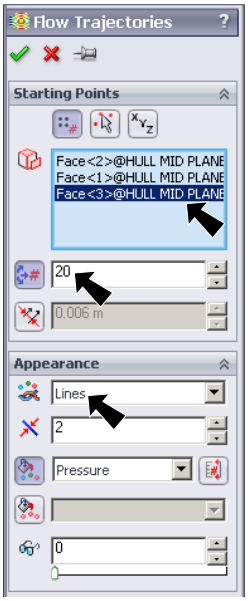


Fig. 23

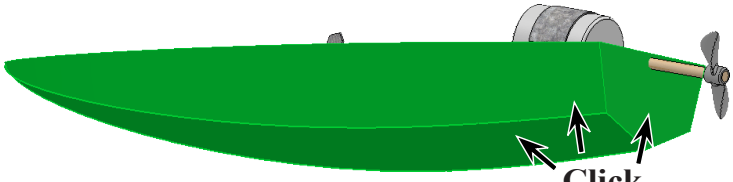


Fig. 24

Step 3. After viewing the flow trajectories, **right** click **Flow Trajectories 1** in Flow Simulation tree and click **Hide**, **Fig. 27**.

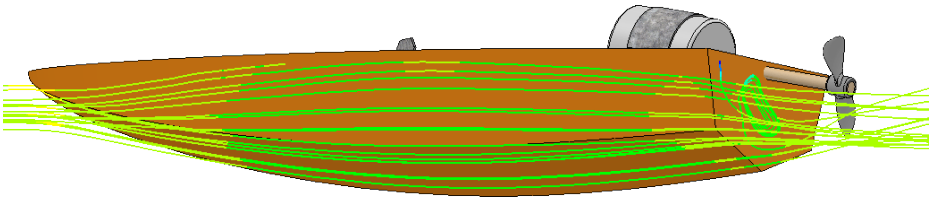


Fig. 25

Fig. 26

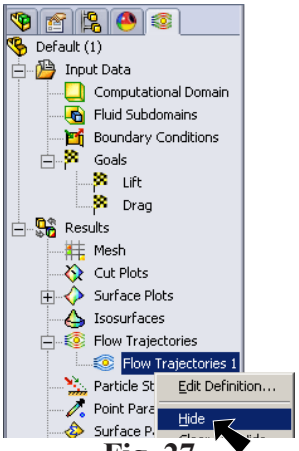


Fig. 27

I. Goals Plot.

Step 1. Click Flow Simulation Menu > Results > **Goal Plot**.

Step 2. In the Goal Plot Property Manager set:
check **All**
click OK , **Fig. 28**.

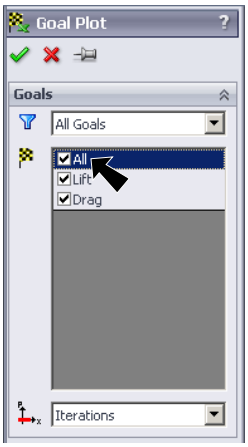


Fig. 28

Step 3. An Excel file is opened. What is your **Drag?** Is it less than **-879.2**, **Fig. 29**.

Step 4. Close the Excel file.

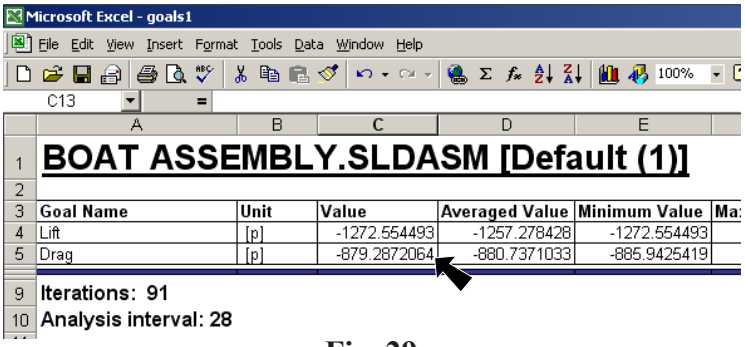


Fig. 29

J. Animate Flow Trajectories.

Step 1. **Right click Flow Trajectories 1** in Flow Simulation tree and click **Animation**, **Fig. 30**.

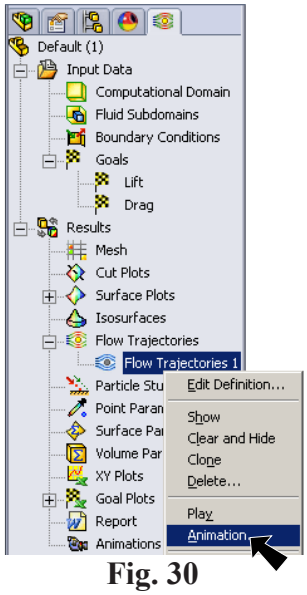


Fig. 30

Step 2. At the bottom right of the display in the animation control panel, click **Expand** to expand the panel, **Fig. 31**.

Step 3. Click **Play** in animation control panel, **Fig. 31**.

Step 4. **Right** click **Flow Trajectories 1** in the Animation tree and click **Edit Definition**, **Fig. 31**.

Step 5. In the Flow Trajectories Property Manager set:
under **Appearance**, **Fig. 32**

Draw Trajectories As to **Spheres**

Width to **.002**

Color Parameter
Velocity(Z)

click OK .

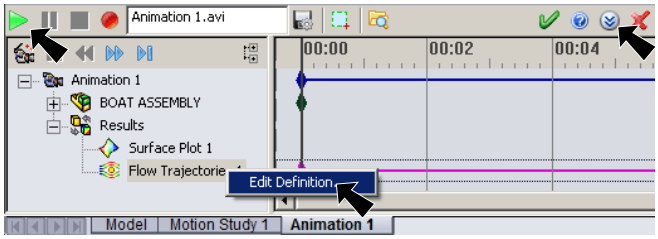


Fig. 31

Step 6. Click **Play** in animation control panel, **Fig. 31**. Click Stop .

Step 7. Experiment with other Flow Trajectories setting. To change setting, **right** click **Flow Trajectories 1** and click **Edit Definition**, **Fig. 31**. Try changing **Pressure** to **Z Velocity**, **Fig. 32**.

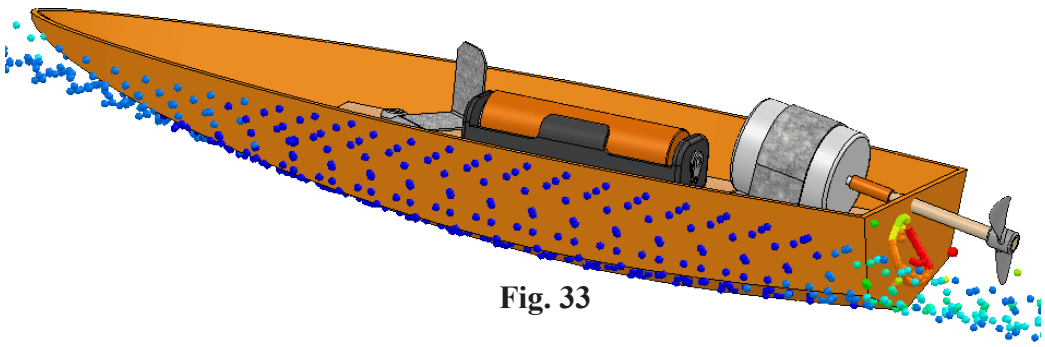


Fig. 33

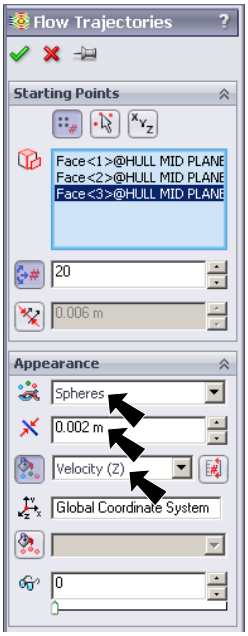


Fig. 32

K. Create Animation Movie.

Step 1. In the Animation control panel turn on **Capture Region** , **Fig. 34**.

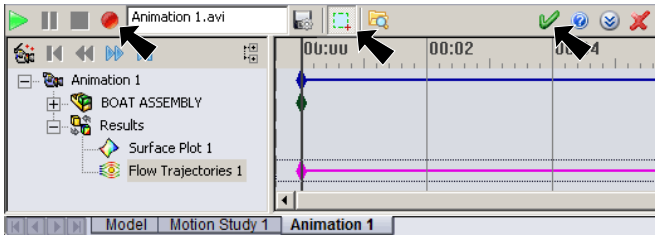


Fig. 34

Step 2. Rotate view to view **rear of boad** as shown in **Fig. 35**. To rotate, hold down middle mouse button (wheel) and drag.

Step 3. Drag the red frame capture region in the graphics area to resize and move capture region, **Fig. 35**.

Step 4. Click **Record** , **Fig. 34**.

Step 5. Click **Open Folder** to view Animation AVI file.

Step 6. To exit, click **OK** in Animation control panel.

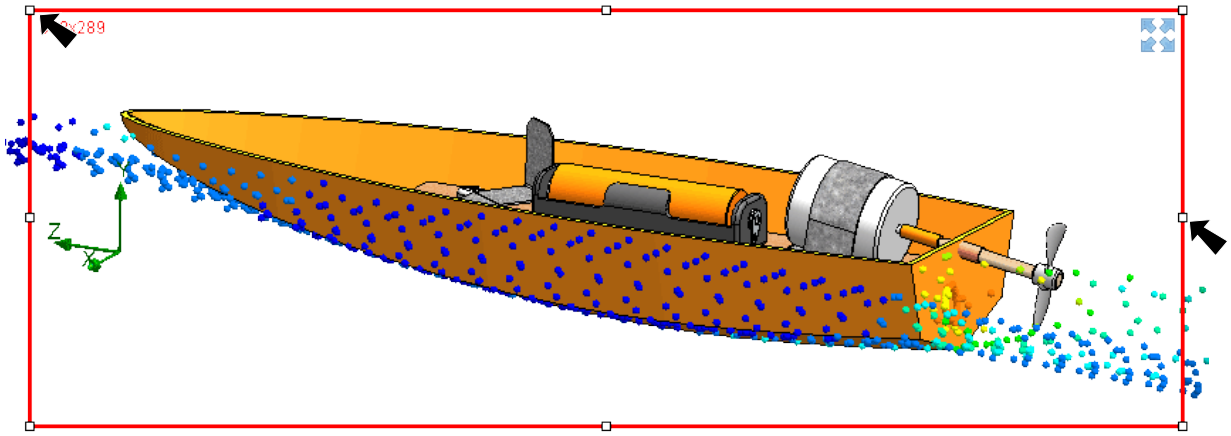


Fig. 42