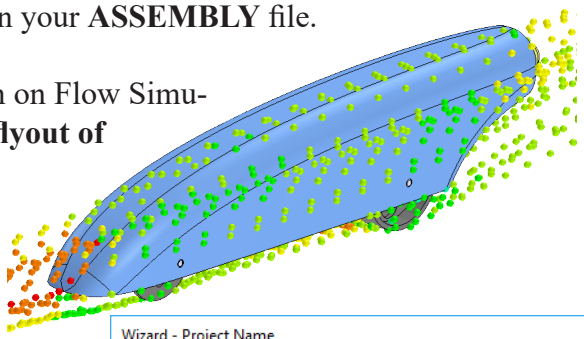
CO2 Shell Car
Flow Sim

A. Enable Flow Simulation.

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

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



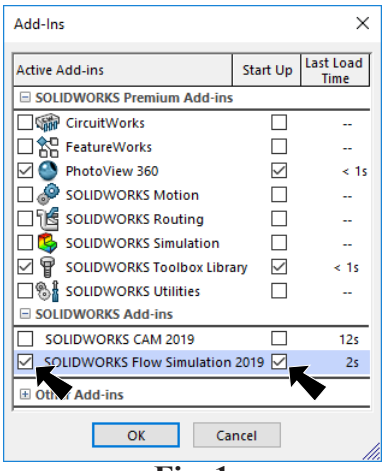


Fig. 1

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**.

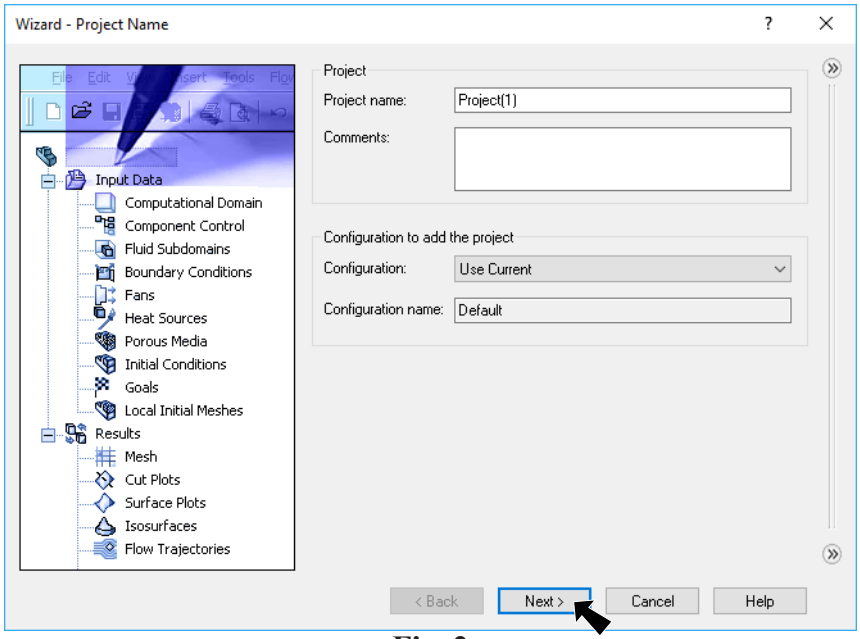


Fig. 2

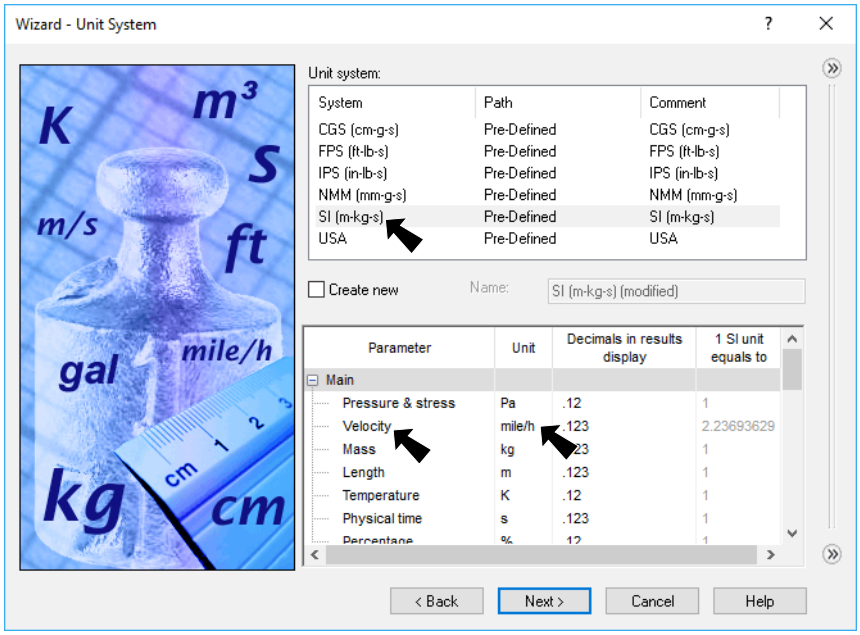


Fig. 3

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. Click Next.

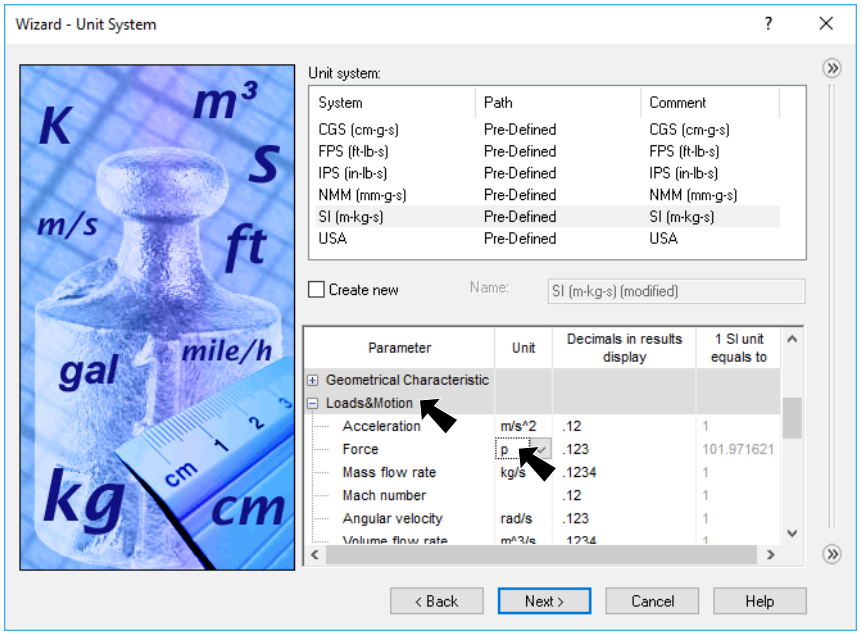


Fig. 4

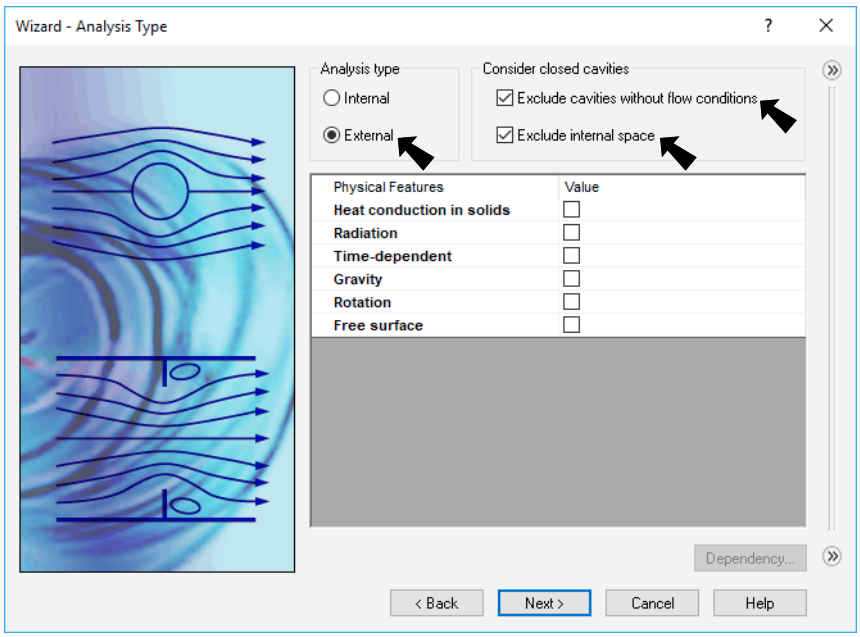


Fig. 5

Step 12. **Default Fluid:** Expand Gases, select **Air** and click **Add**, **Fig. 6**.

Step 13. Click Next.

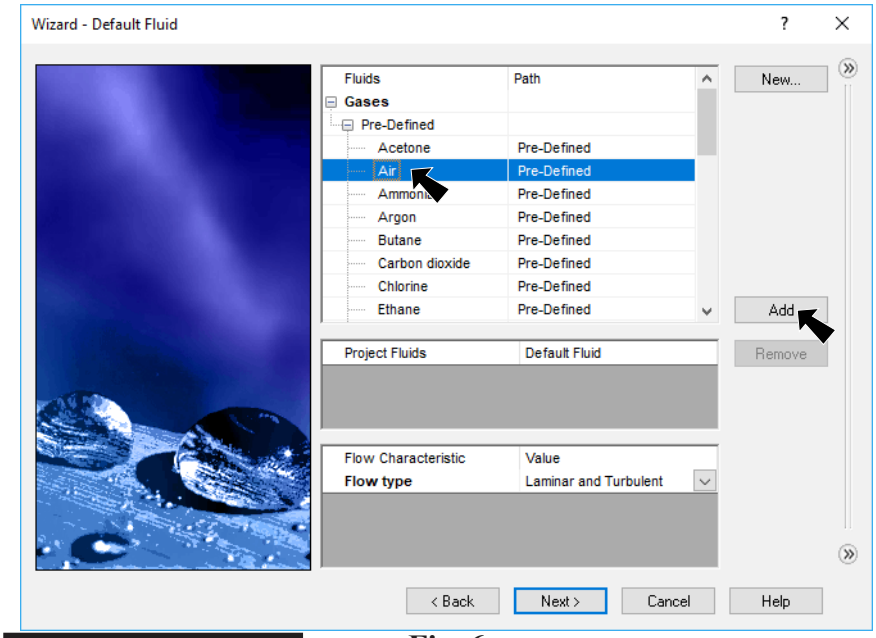
Step 14. **Wall Conditions:** Use the default values for wall condition, **Fig. 7**.

Step 15. Click Next.

Step 16. **Initial Conditions:** Under Velocity parameters set **Velocity in Z direction** to **-75 Mile/h**, **Fig. 8**. (click and key-in **-75**).

Step 17. Click Finish.

Tip: Good idea to turn off decals. View Menu > Hide\Show > Decals. Or...



Negative -75

Fig. 6

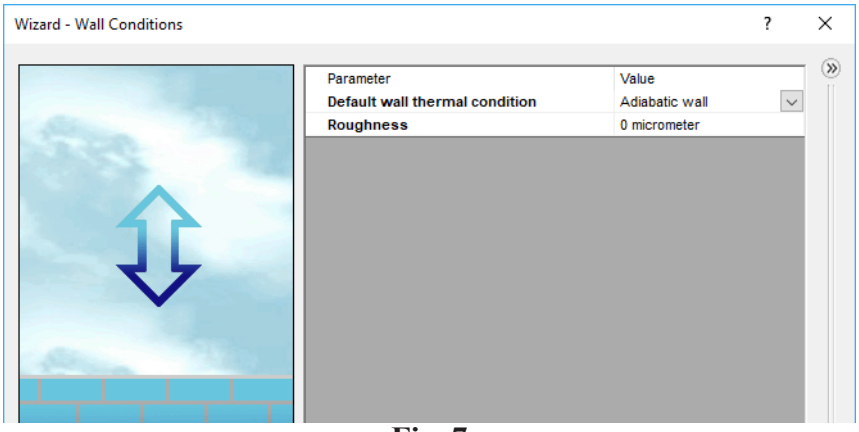
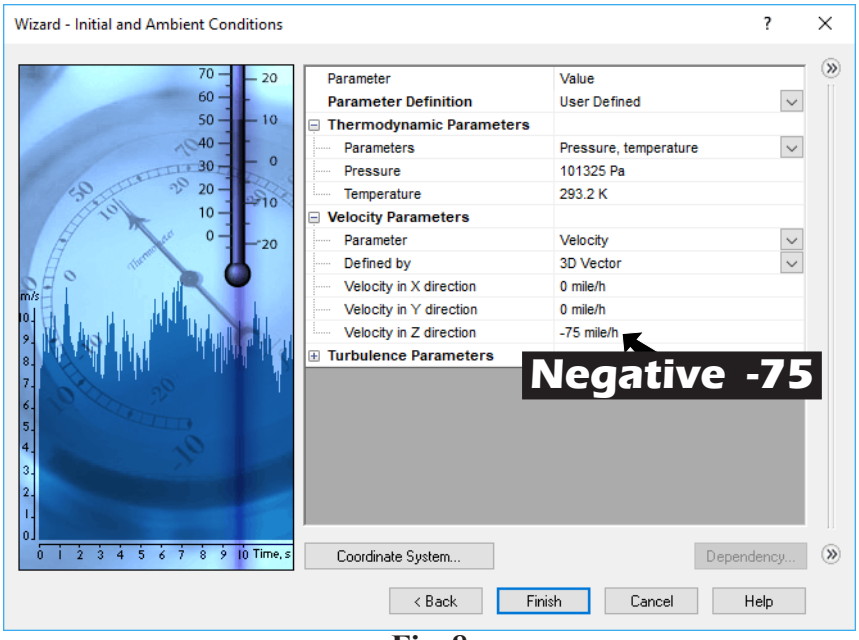


Fig. 7



Negative -75

Fig. 8

C. Computational Domain.

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

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

X max .15
X min 0

Y max .2
Y min -.02 ← **Negative**

Z max .3
Z min -.4 ← **Negative**

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

Step 4. Click OK in Property Manager.

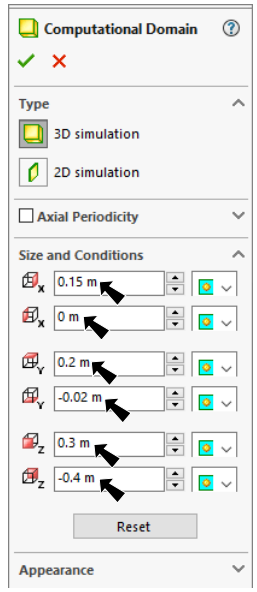


Fig. 9

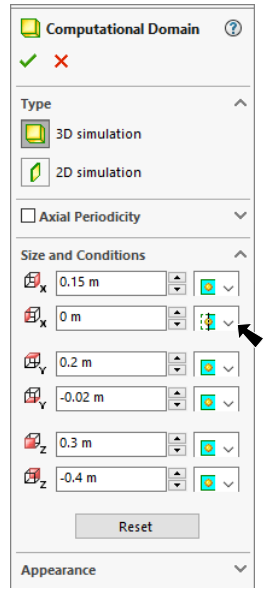


Fig. 10

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 car** should be inside the domain, **Fig. 12**. This will reduce the solver time. Solver can take several minutes.

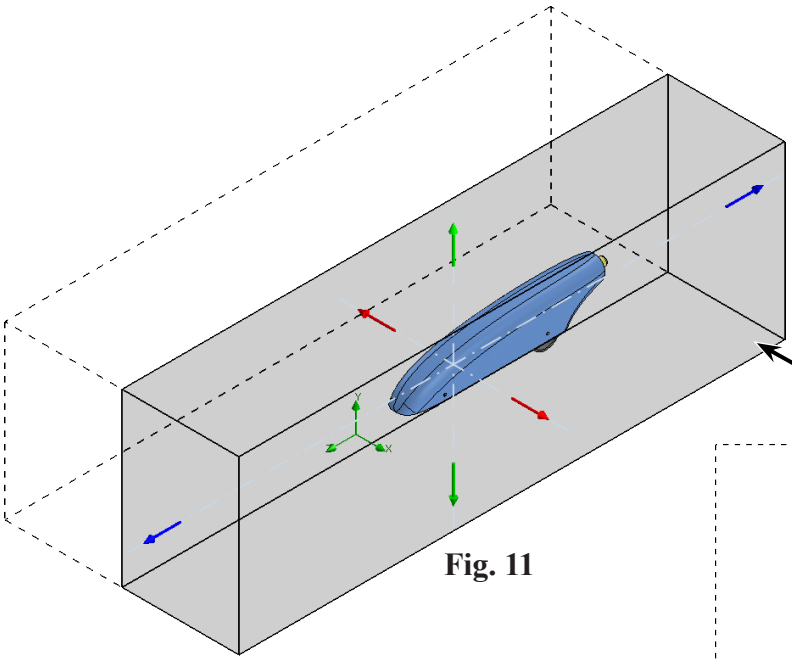


Fig. 11

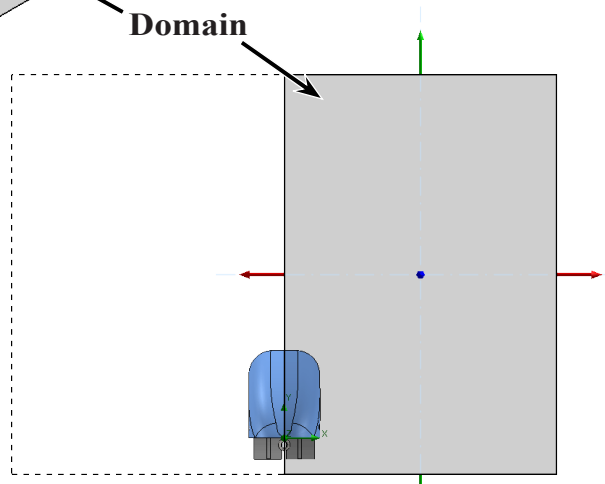


Fig. 12

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** (Flow Simulati...) 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**.

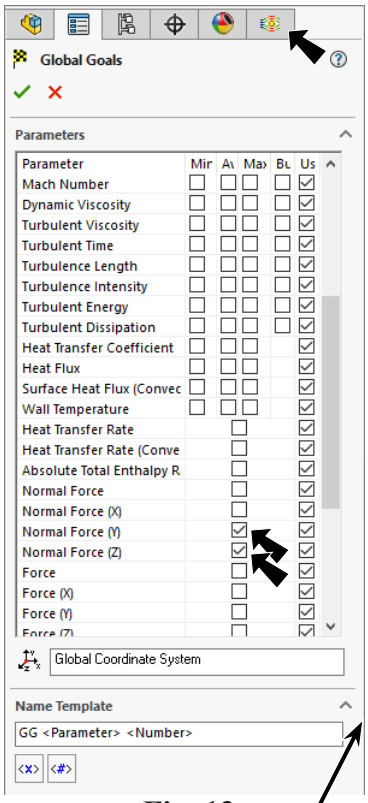


Fig. 13

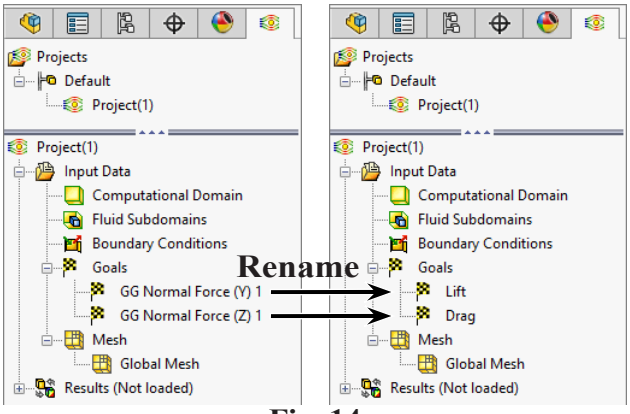


Fig. 14

E. Run Analysis.

Step 1. Click **Run** (Run) on the Flow Simulation toolbar.

Step 2. Click **Run** in the Run dialog box, **Fig. 15**.

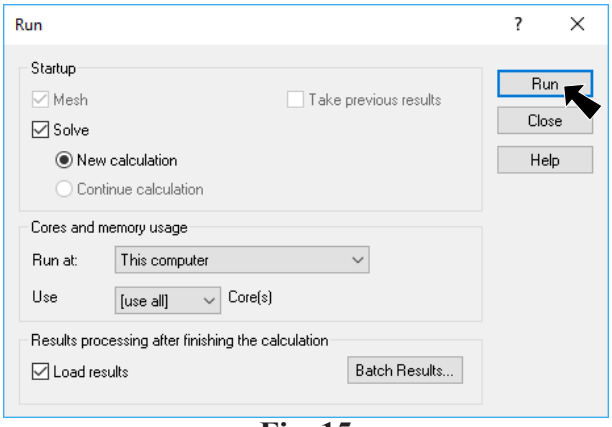


Fig. 15

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 depending on the car design and the computer.

Step 2. When the calculation is done view the drag in the Solver, click Insert Menu > Goal Table, **Fig. 17**. Our drag was **-14.09**.

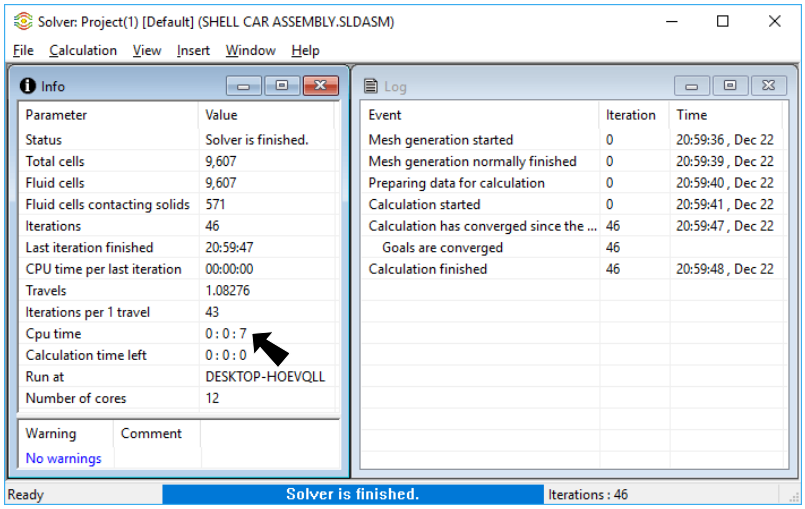


Fig. 16

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

Step 4. Save. Use **Ctrl-S**.

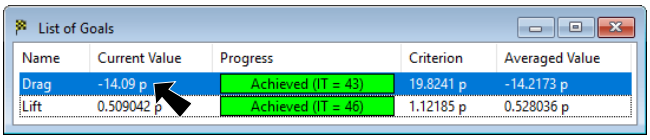


Fig. 17

G. Surface Plots.

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

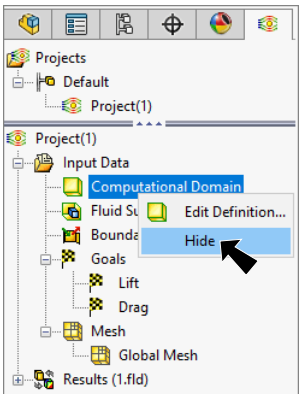


Fig. 18

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

Step 3. In the Surface Plot Property Manager, set:
under Selection, **Fig. 19**
check **Use all faces**
under Display
select **Contours**
under Contours
click **Adjust Minimum and Maximum**
101575 for Maximum pressure
100930 for Minimum pressure
click OK .

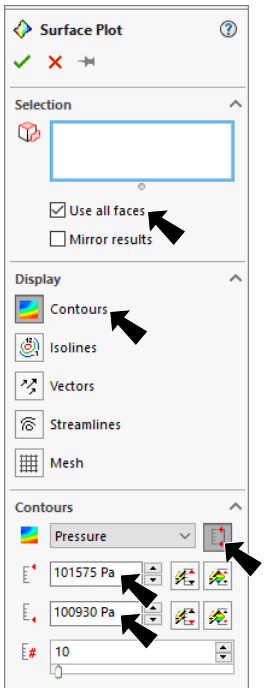


Fig. 19

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**.

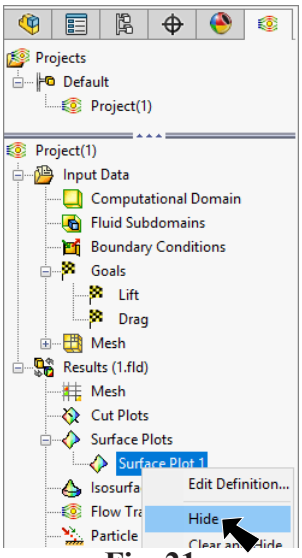


Fig. 21

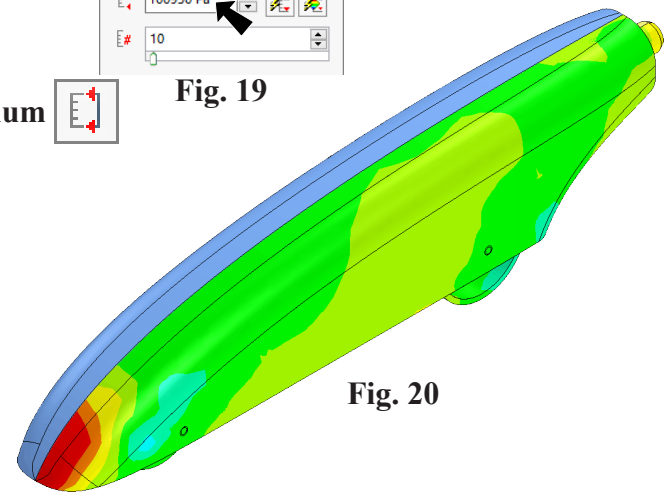


Fig. 20

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 faces on right side of car. To select References, click each face of the body, wheels and cartridge, **Fig 23**. Rotate view to select faces on bottom and rear, **Fig. 24**.

Tip: Turn on **Filter Faces** (**X**) on the **Selection Filter toolbar** at the bottom of the display to select only faces and not edges.

Number of Points **50**

under Appearance

Draw Trajectories As **Arrows**

Arrow Size **.002**

click OK .

Step 3. After viewing the flow trajectories, **right click Flow Trajectories 1** in Flow Simulation tree and click **Hide**, **Fig. 26**.

Step 4. Save. Use **Ctrl-S**.

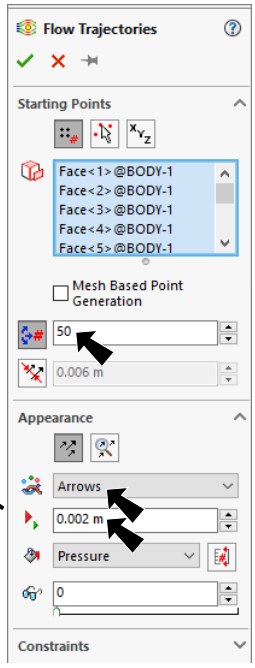


Fig. 22

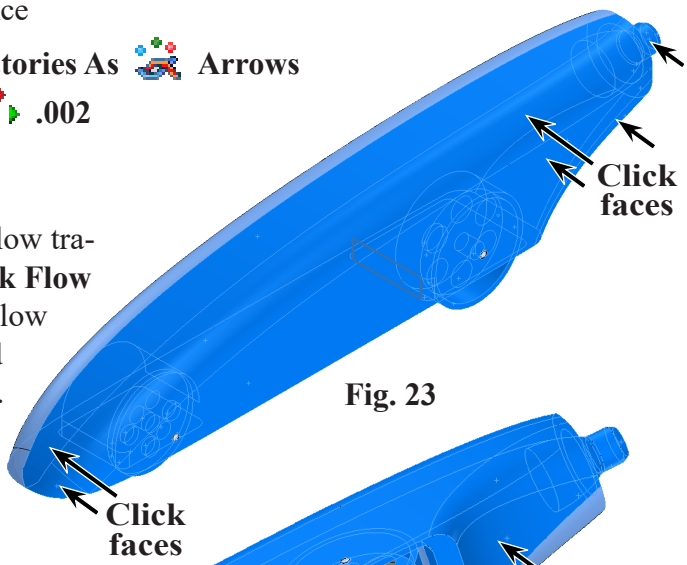


Fig. 23

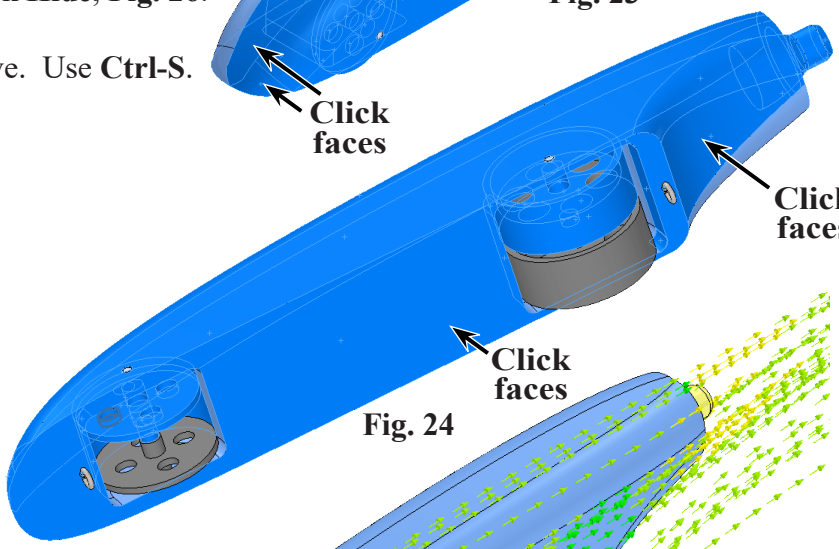


Fig. 24

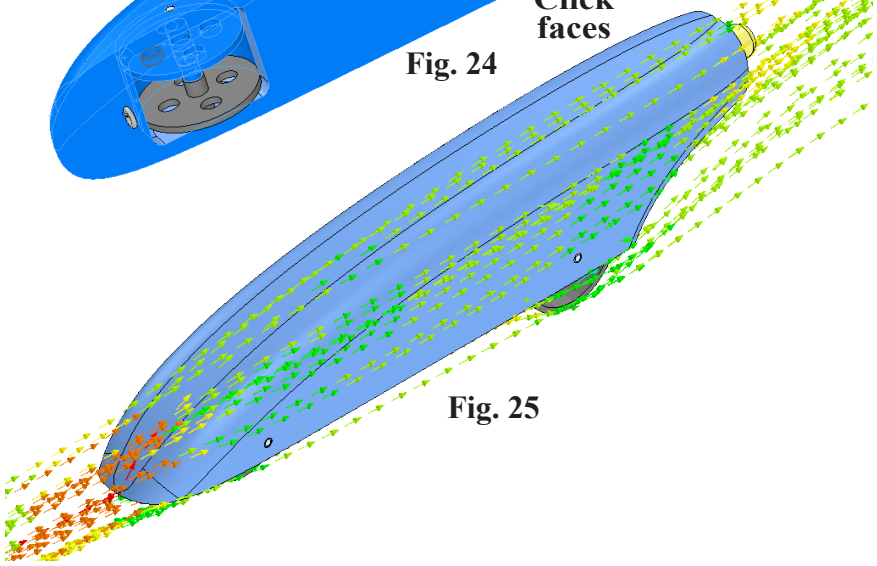


Fig. 25

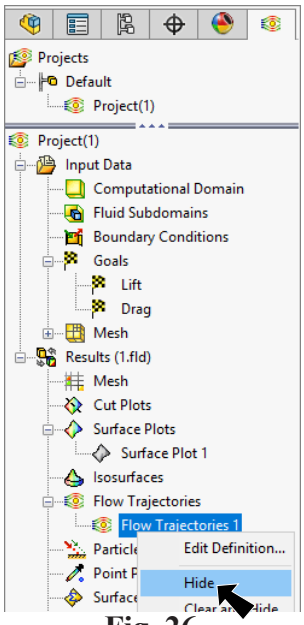


Fig. 26

I. Goal Plot.

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

Step 2. In the Goal Plot Property Manager:
under Goals, **Fig. 27**
check **All**
under Options
click **Export to Excel**
An Excel file is opened. Note the Drag, **Fig. 28**.
Click the **Drag tab** at the bottom of the Excel file to view the chart, **Fig. 29**.
Close the Excel file.
Click OK .

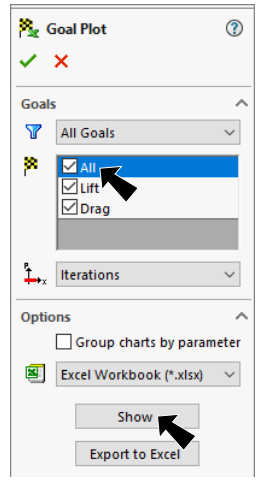
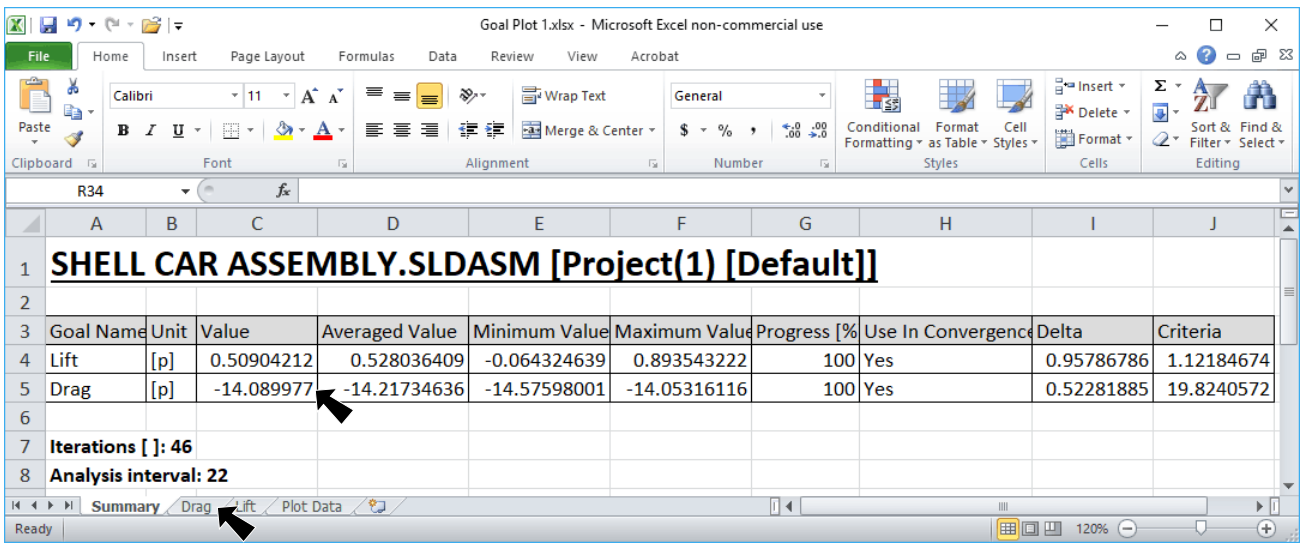


Fig. 27



SHELL CAR ASSEMBLY.SLDASM [Project(1) [Default]]

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
Lift	[p]	0.50904212	0.528036409	-0.064324639	0.893543222	100	Yes	0.95786786	1.12184674
Drag	[p]	-14.089977	-14.21734636	-14.57598001	-14.05316116	100	Yes	0.52281885	19.8240572

Iterations [ ]: 46
Analysis interval: 22

Fig. 28

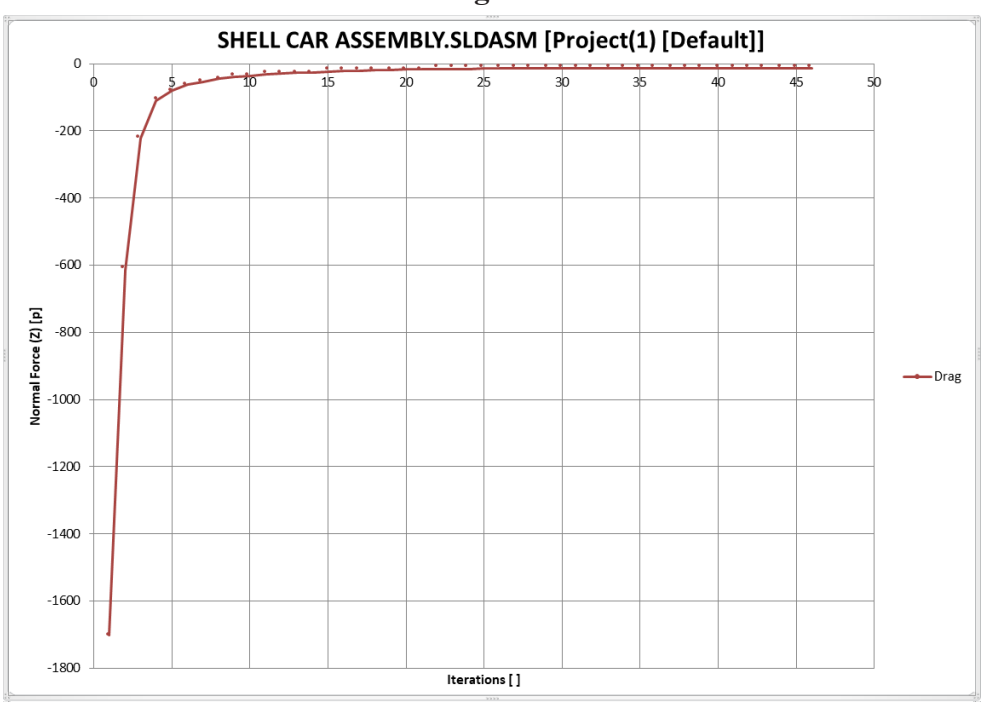


Fig. 29

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. 30**.

Step 3. At the bottom right of the display in the animation control panel, click **Expand** to expand the panel, **Fig. 31**.

Step 4. Click **Play** in animation control panel, **Fig. 31**.

Step 5. **Right click Flow Trajectories 1** in the Animation tree and click **Edit Definition**, **Fig. 31**.

Step 6. In the Flow Trajectories Property Manager:
under Appearance, **Fig. 32**

Draw Trajectories As **Spheres**
Width **.003**
click OK .

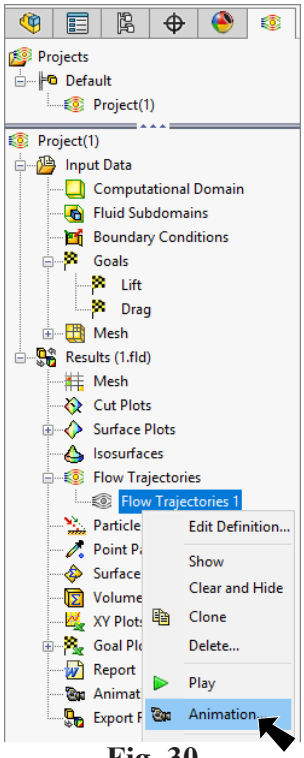


Fig. 30

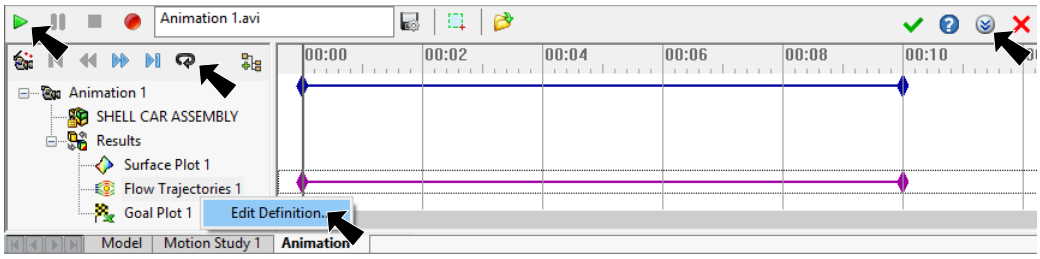


Fig. 31

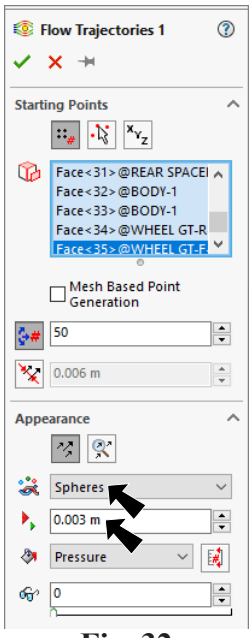


Fig. 32

Step 7. Click **Loop** and **Play** in animation control panel, **Fig. 31**.
Click Stop .

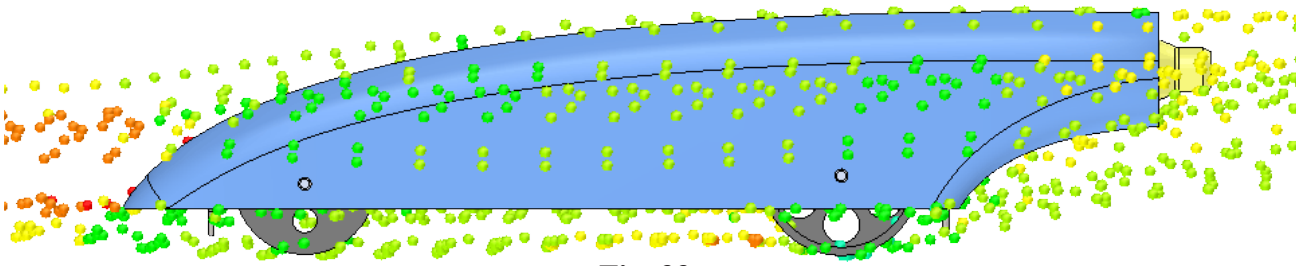


Fig. 33

K. Create Animation Movie.

Step 1. Click **Isometric** on the Standard Views toolbar. (**Ctrl-7**)

Step 2. In the Animation control panel turn on **Capture Region**, **Fig. 34**.

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

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

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

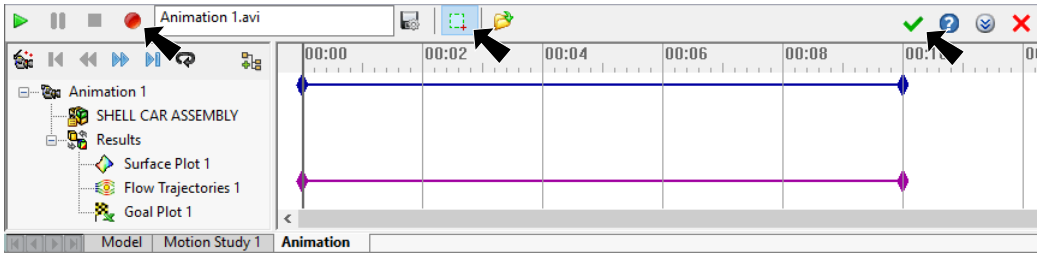


Fig. 34

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

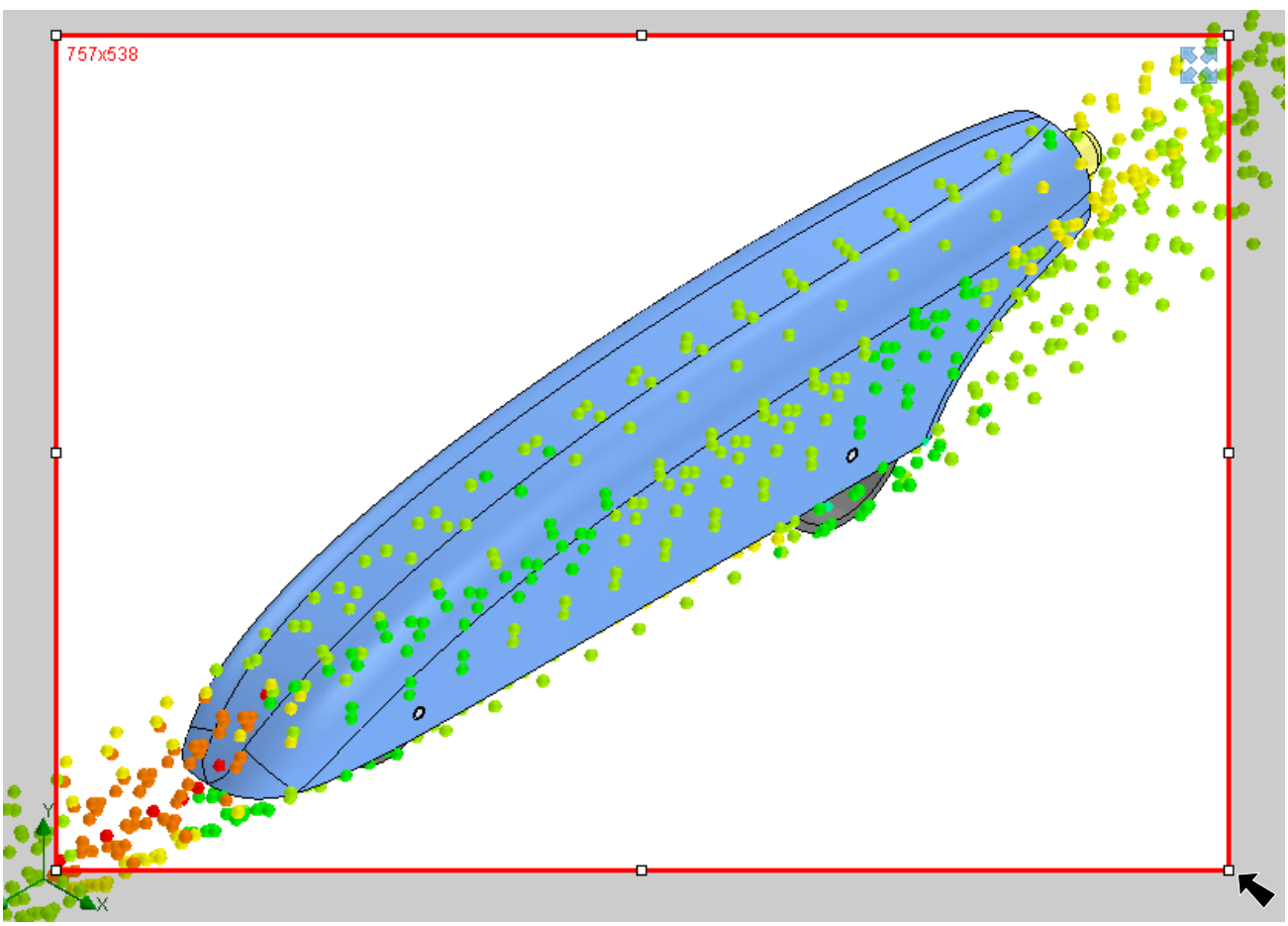


Fig. 35