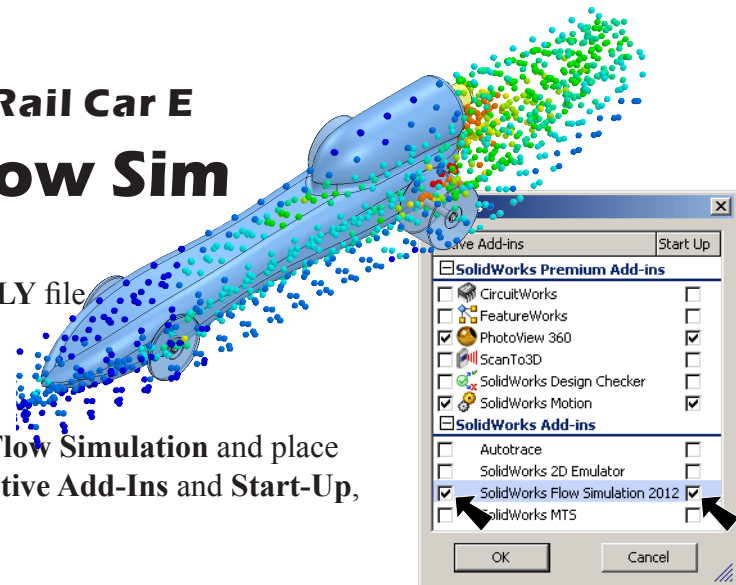


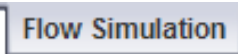


## Rail Car E Flow Sim



### 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.

### 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**.

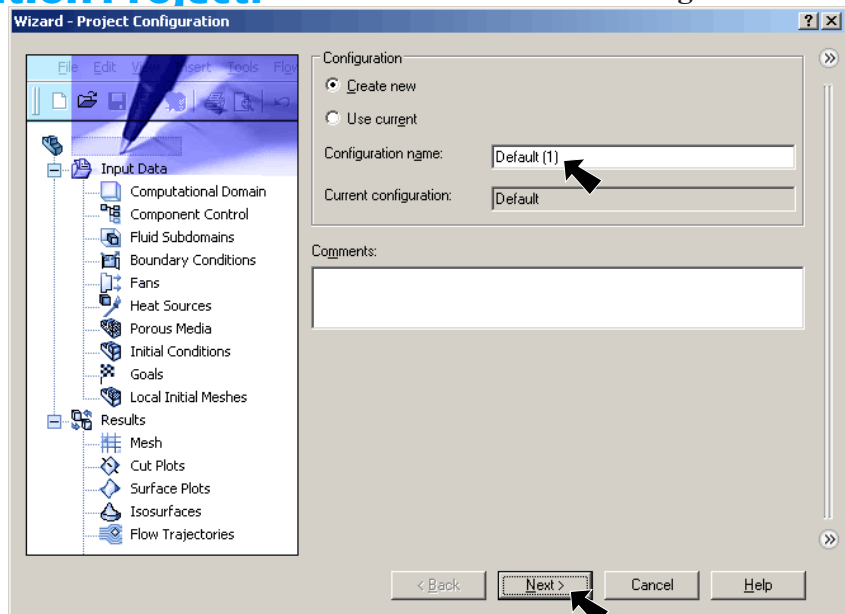


Fig. 2

- Step 4. Click Next.
- Step 5. **Unit System:** Select **SI (m-k-g-s)** for Unit system, **Fig. 3**.
- Step 6. Under Parameter, set **Velocity** units to **Mile/hour**, **Fig. 3**.

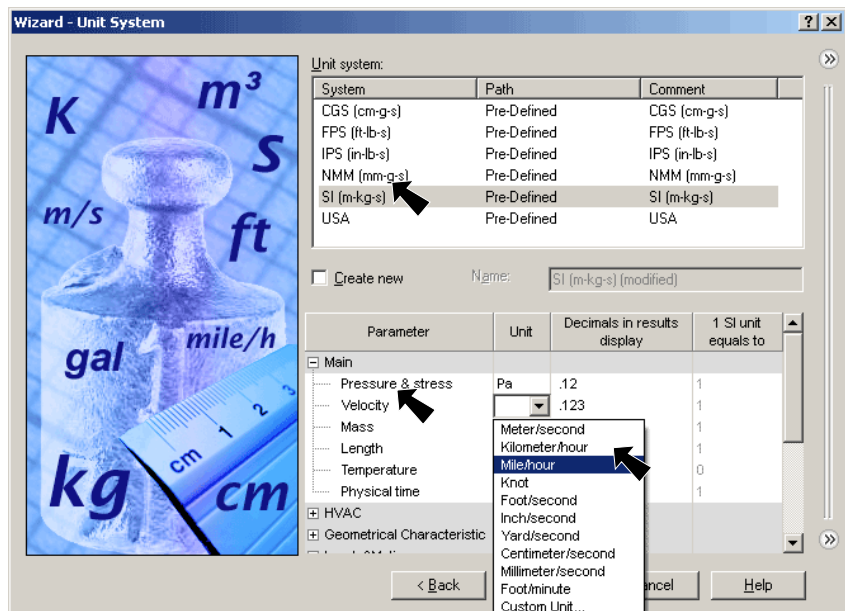


Fig. 3

Step 7. 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 Gases, select **Air** and click **Add**, Fig. 6.

Step 14. Click Next.

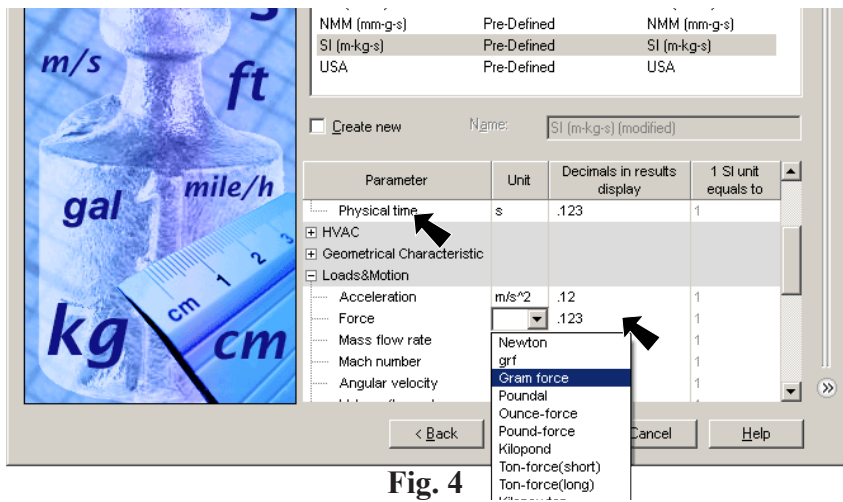


Fig. 4

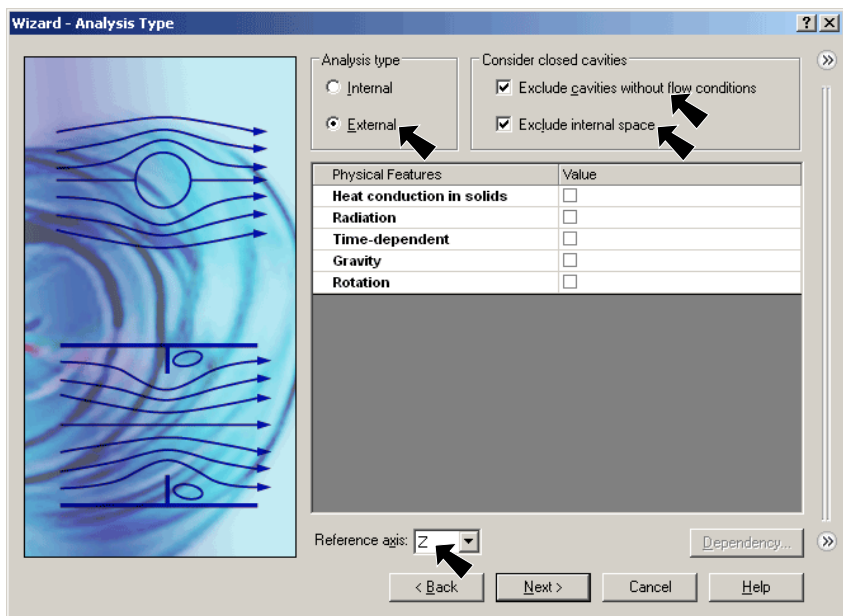


Fig. 5

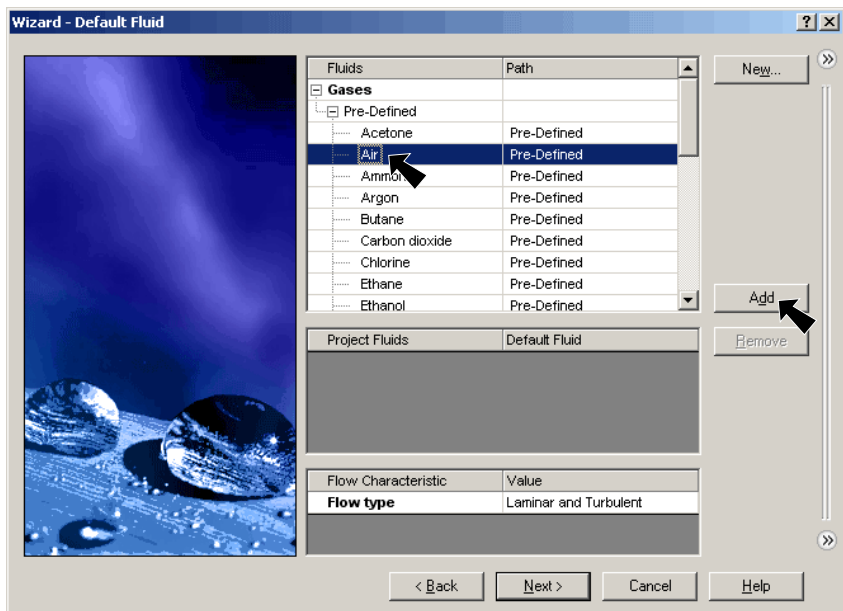


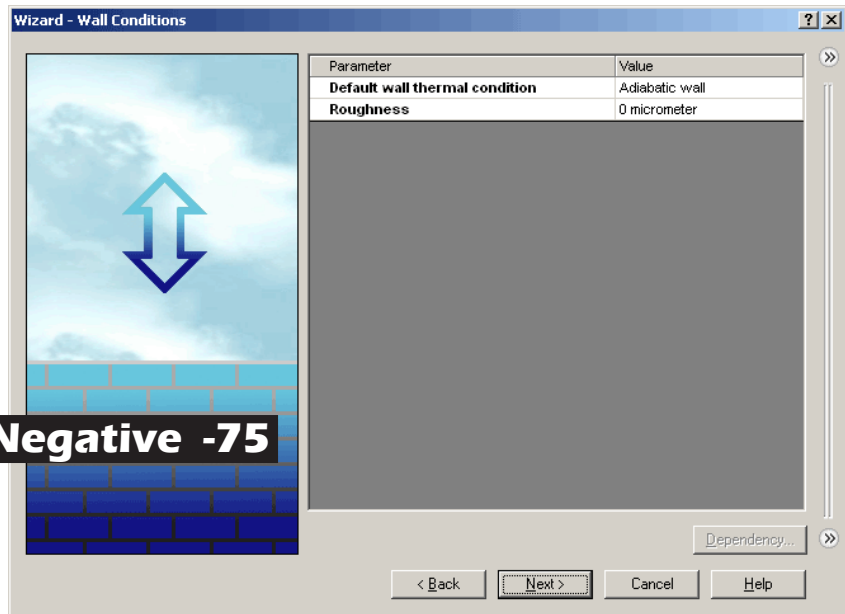
Fig. 6

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

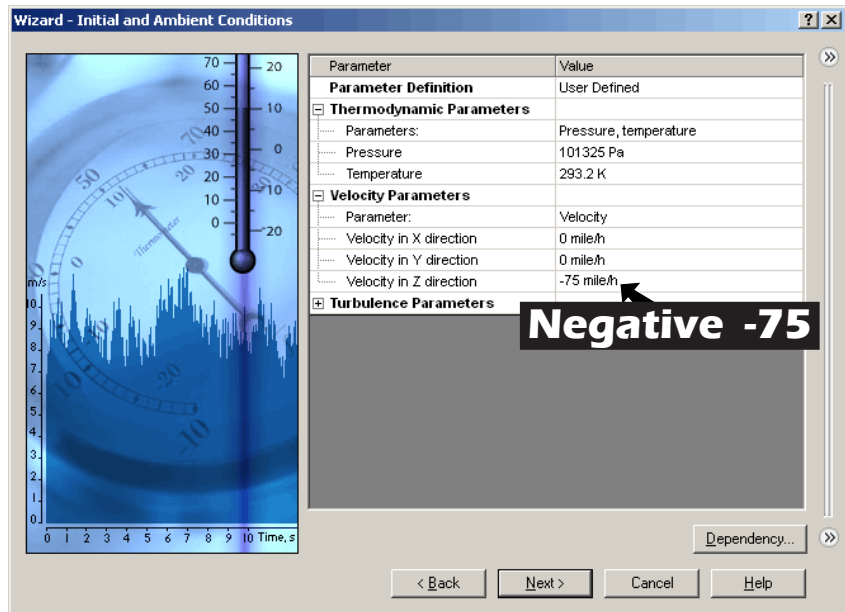
Step 16. Click Next.

Step 17. **Initial Conditions:**  
Under Velocity parameters set **Velocity in Z direction to -75 Mile/h, Fig. 8.** (click and key-in -75).

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.

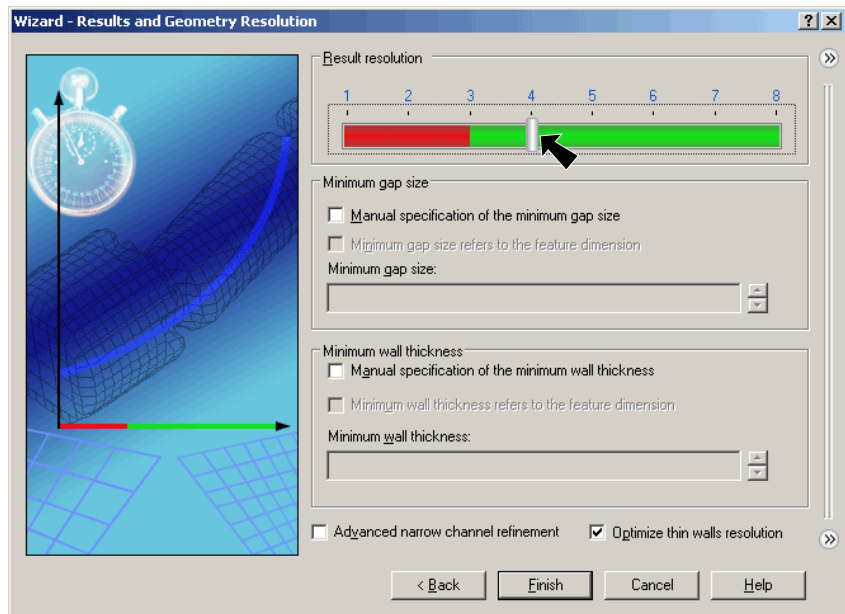


Fig. 9

### C. Computational Domain.

Step 1. Click Flow Simulation Menu > Computational Domain.

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

- X max .15
- X min 0
- Y max .2
- Y min -.02
- Z max .3
- Z min -.4

**Negative**

**Negative**

Step 3. Set At X min to Symmetry, Fig. 11.

Step 4. Click OK in the Property Manager.

Step 5. Click **Front** on the Views toolbar (Ctrl-1). Use Z key to zoom out to see domain, Fig. 13.

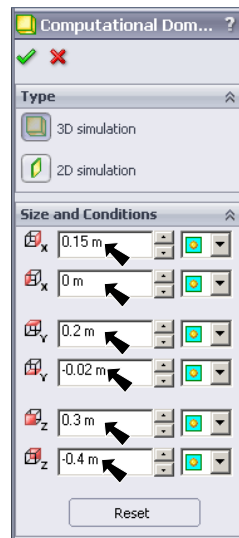


Fig. 10

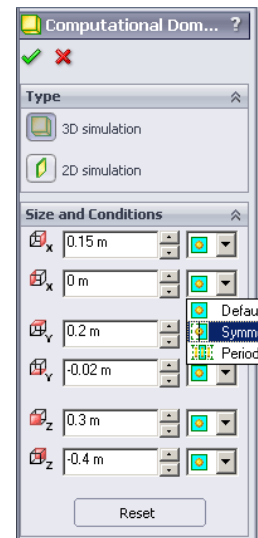


Fig. 11

Step 6. **Only half the car** should be inside the domain as shown in Fig. 13. This will reduce the solver time. Solver can take several minutes.

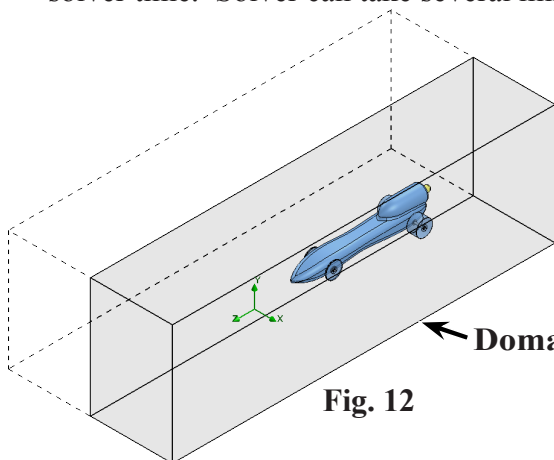


Fig. 12

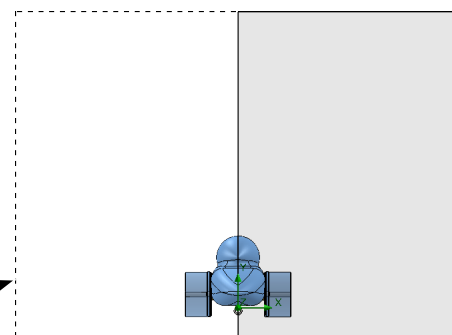



Fig. 13

## 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**.

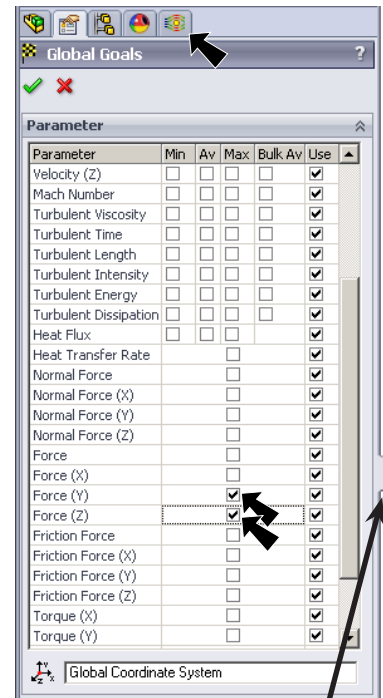
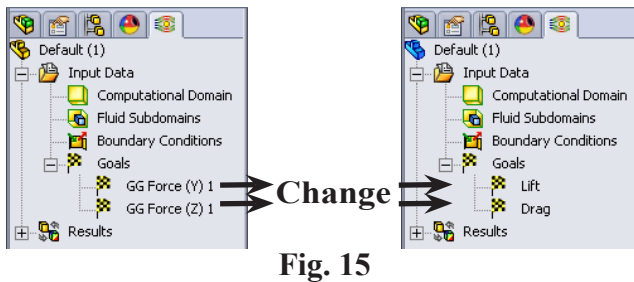
Step 4. Drag the edge of the Feature Manager to the right to expand, **Fig. 14**.

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**.

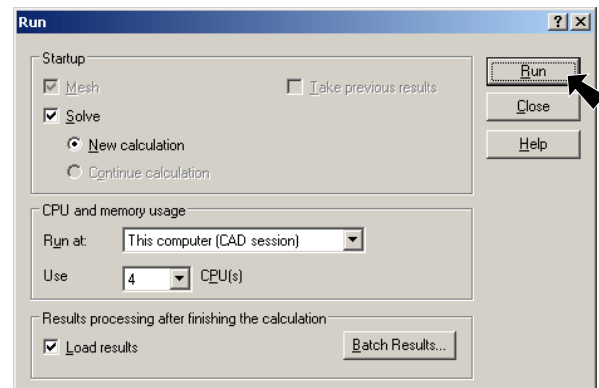


**Fig. 14**  
**Drag right**

## E. Run Analysis.

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

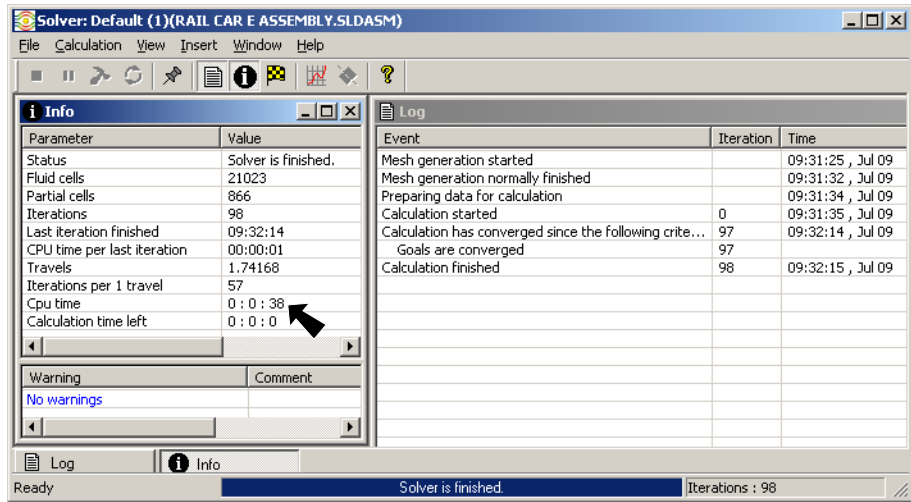
Step 2. Click **Run** in the Run dialog box, **Fig. 16**.



**Fig. 16**

## F. Solver.

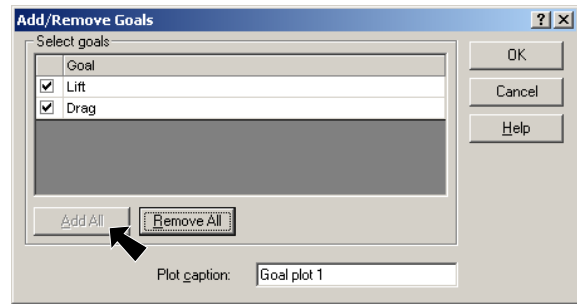
Step 1. In the Solver dialog box you can view Calculation time left, **Fig. 17**. The CPU time runs around 2-3 minutes depending on the car design and the computer.



**Fig. 17**

Step 2. Click Insert Menu > Goal Plot in the Solver, **Fig. 18**.

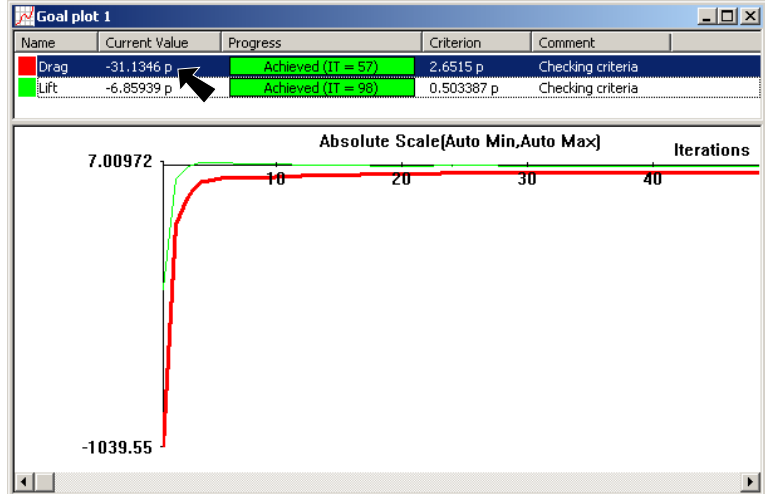
Step 3. Click **Add All** and OK in Add/Remove Goals, **Fig. 18**. Our **Drag** was **-31.1**, **Fig. 19**.



**Fig. 18**

Step 4. Close the Solver dialog box, click File Menu > Close.

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



**Fig. 19**

## G. Surface Plots.

Step 1. **Right click Computational Domain** in Flow Simulation tree and click **Hide**, Fig. 20.

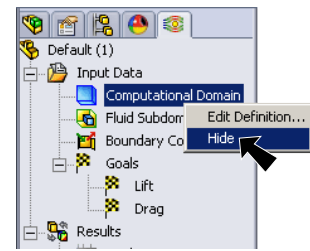




Fig. 20

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

Step 3. In the Property Manager set:  
check **Use all faces**, Fig. 21  
select **Contours**

click **Adjust Minimum and Maximum** 

**Maximum**  pressure **101575**

**Minimum**  pressure **100930**

click **OK** .

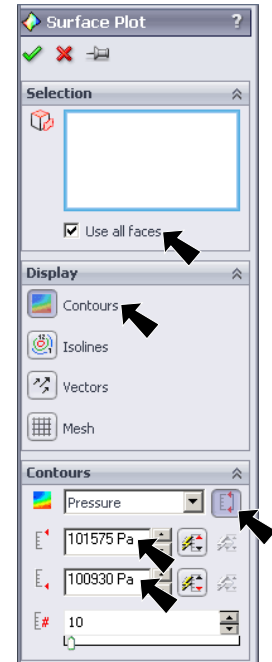


Fig. 21

Step 4. After viewing the surface plot, expand **Results** in Flow Simulation tree, **right click Surface Plots 1** and click **Hide**, Fig. 23.

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

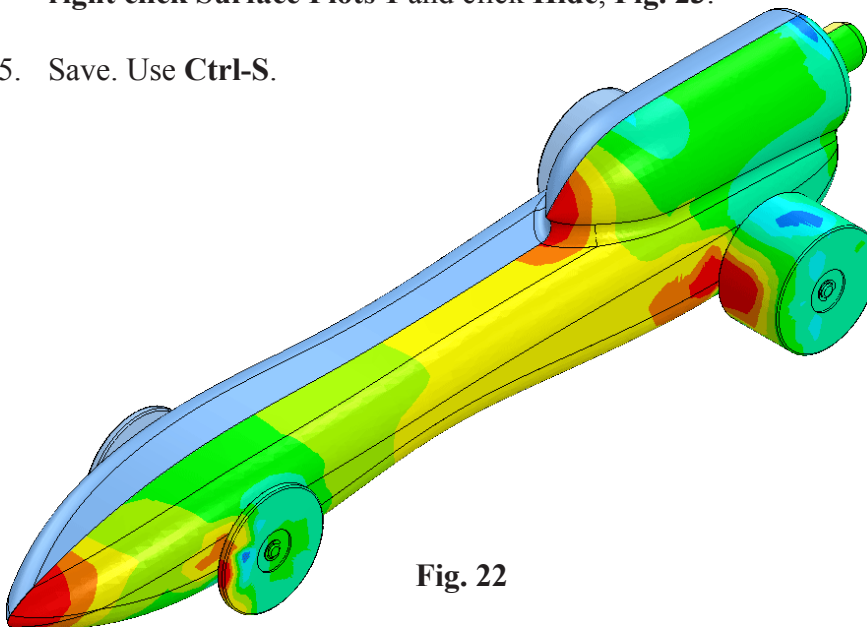


Fig. 22

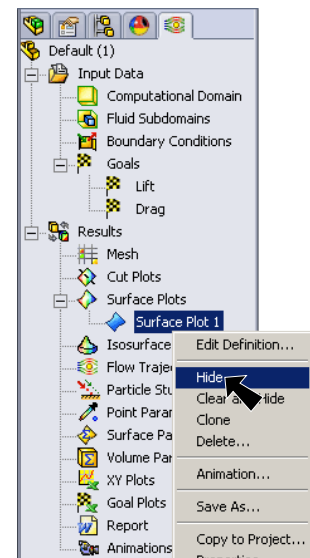


Fig. 23

## H. Flow Trajectories.



Step 1. Click **Filter Faces**  (X) on the **Selection Filter toolbar** at the bottom of the display, **Fig. 24**. If necessary use **F5** key to display the toolbar.



Fig. 24

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

Step 3. In the Flow Trajectories Property Manager:  
under **Starting Points**, **Fig. 25**

**Select References**  by selecting all faces of right side of car. To select References, **right click** a face of the body and click **Select Tangency** from menu, **Fig. 26**. View the underside of body to make sure all faces are selected. To view under underside, hold down middle mouse button (wheel) and drag to rotate view. **Right click** both wheels on left side of car and select **Select Tangency** to select faces of wheels. Also, Select Tangency faces of cartridge, **Fig. 27**

set **Number of Trajectories**  **50**, **Fig. 25**

under **Appearance**

set **Draw Trajectories As**  **Lines with Arrow**

set **Cross Size**  **.002**

click **OK** , **Fig. 28**.

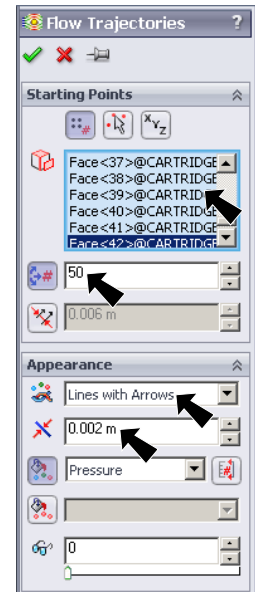



Fig. 25

Step 4. After viewing the flow trajectories, **right click** **Flow Trajectories 1** in Flow Simulation tree and click **Hide**, **Fig. 29**.

Step 5. Turn off **Filter Faces**  (X) on the **Selection Filter toolbar** at the bottom of the display, **Fig. 29**. Or use **F6** key to turn off filters.

Step 6. Save. Use **Ctrl-S**.

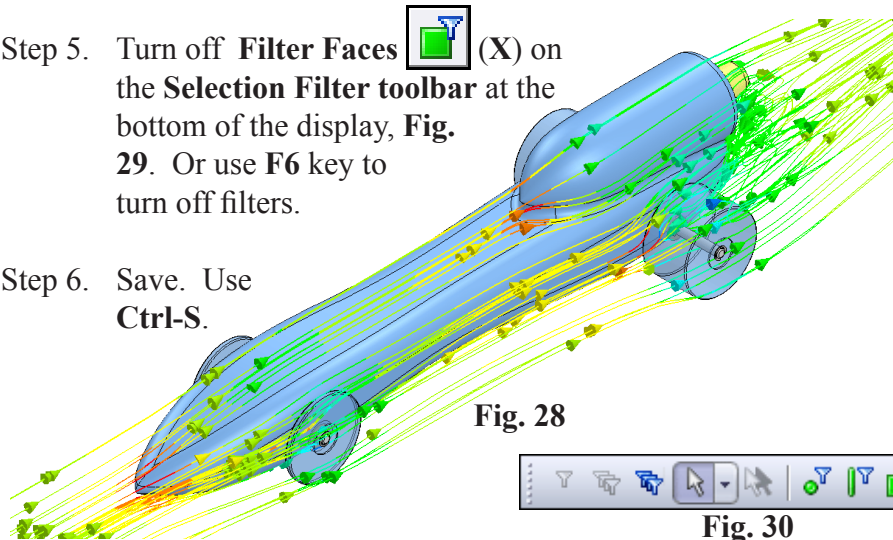


Fig. 28



Fig. 30

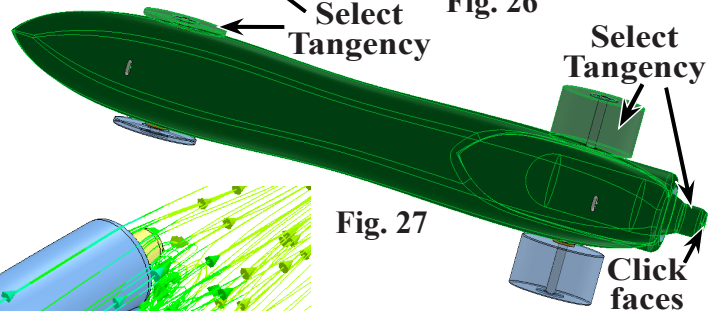
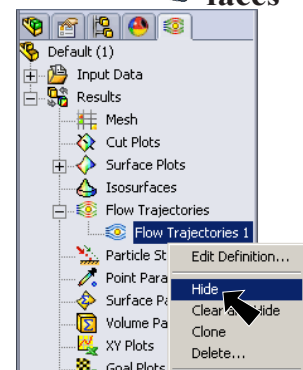


Fig. 26


Fig. 27

Fig. 29



# I. Goal Plot.

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

Step 2. In the Goal Plot Property Manager:  
 check All, Fig. 31  
 click OK .

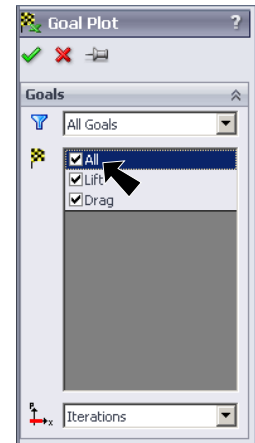


Fig. 31

An Excel file is opened. What is your Drag, Fig. 32.  
 Click the Force (Z) tab at the bottom of the Excel file to view the chart, Fig. 33.  
 Close the Excel file.

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress %	Use In Convergence	Delta	Criteria
Lift	[p]	-6.869389787	-7.2561646	-7.988241946	-6.586699918	100	Yes	0.483056321	0.503387182
Drag	[p]	-31.13461192	-31.23350413	-32.12527211	-30.37824747	100	Yes	0.549861858	2.651503507

Iterations: 98  
 Analysis interval: 29

Fig. 32

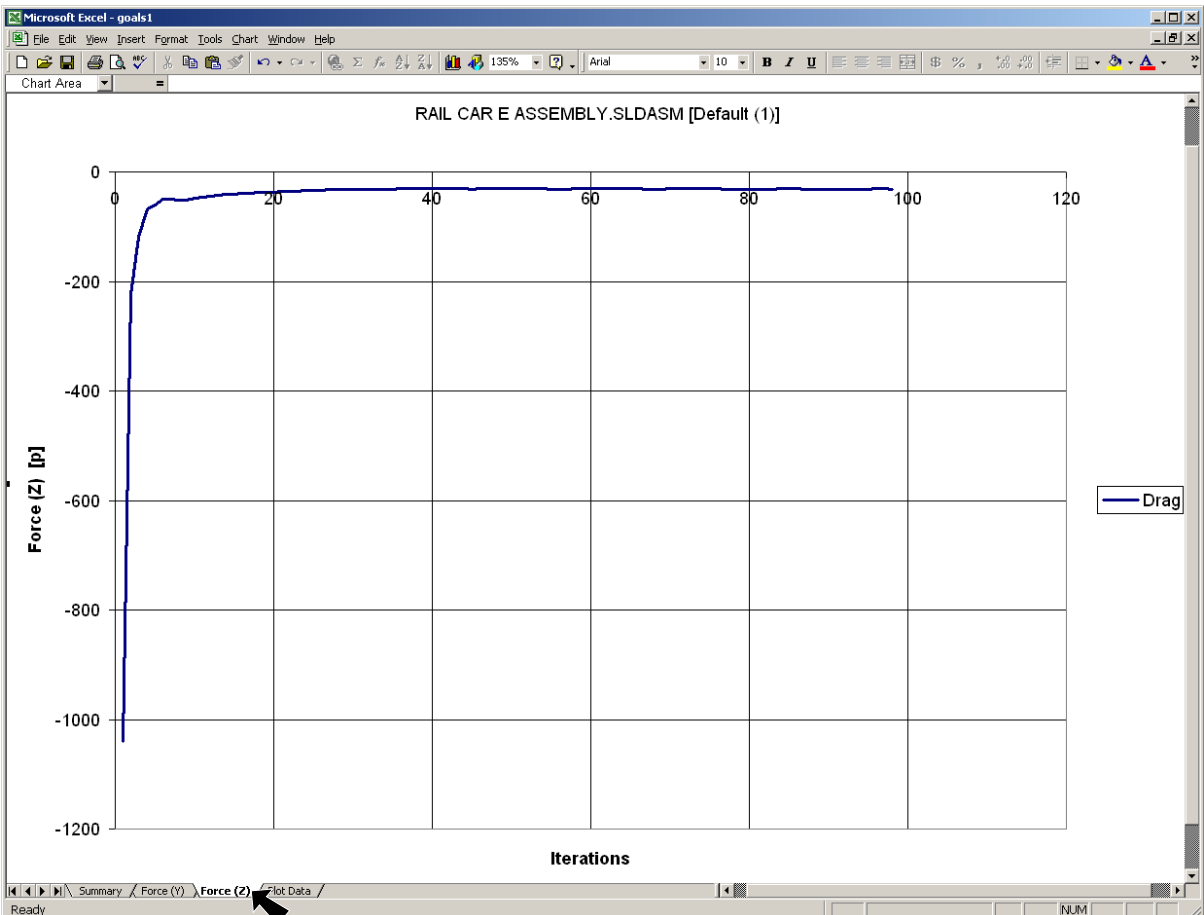



Fig. 33

## J. Animate Flow Trajectories.

Step 1. **Right click Flow Trajectories 1** in Flow Simulation tree and click **Animation**, Fig. 34.

Step 2. At the bottom right of the display in the animation control panel, click **Expand**  to expand the panel, Fig. 35.

Step 3. Click **Play**  in animation control panel, Fig. 35.


Step 4. **Right click Flow Trajectories 1** in the Animation tree and click **Edit Definition**, Fig. 35.

Step 5. In the Flow Trajectories Property Manager:  
under **Appearance**, Fig. 36

set **Draw Trajectories As**  **Spheres**

set **Cross Size**  .003

click **OK** .

Step 6. Click **Play**  in animation control panel, Fig. 35.

Step 7. Experiment with other Flow Trajectories setting. To change setting, **right click Flow Trajectories 1** and click **Edit Definition**, Fig. 35.

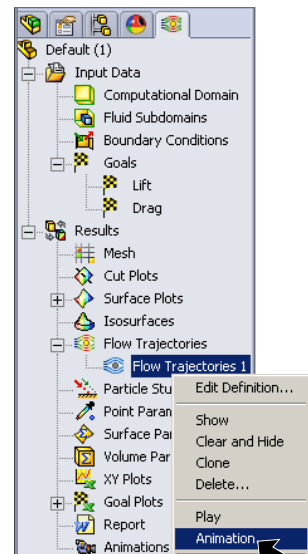


Fig. 34

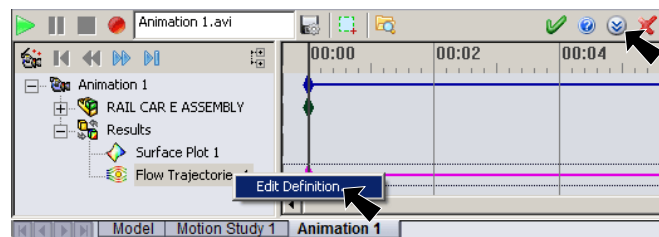


Fig. 35

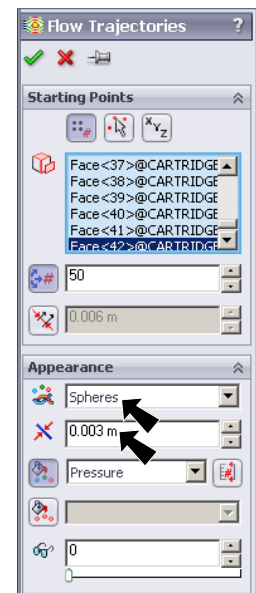


Fig. 36

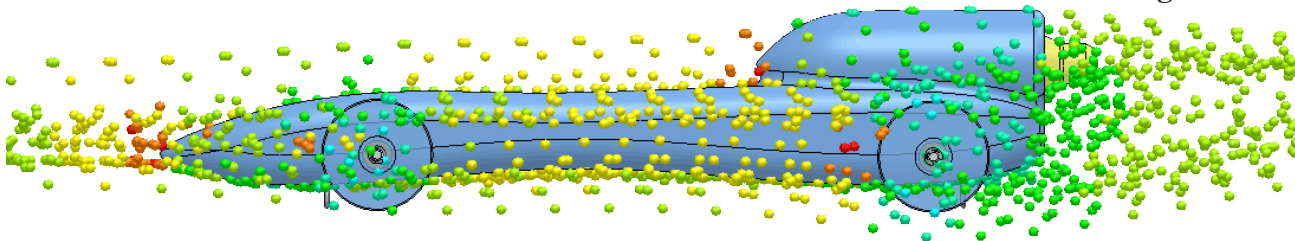


Fig. 37

## K. Velocity Z.

Step 1. Right click **Flow Trajectories 1** in the Animation tree and click **Edit Definition**, Fig. 38.

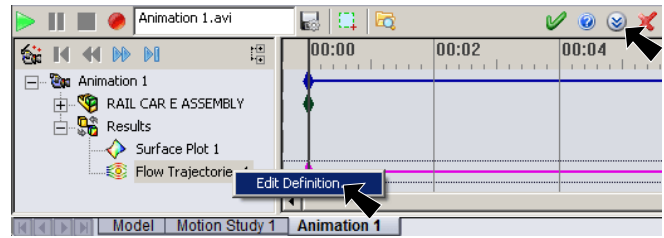


Fig. 38

Step 2. In the Flow Trajectories Property Manager:

under **Appearance**, Fig. 39

set **Draw Trajectories As**  **Spheres**

set **Cross Size**  .002

set **Color Parameter**  **Velocity (Z)**

click **OK** .

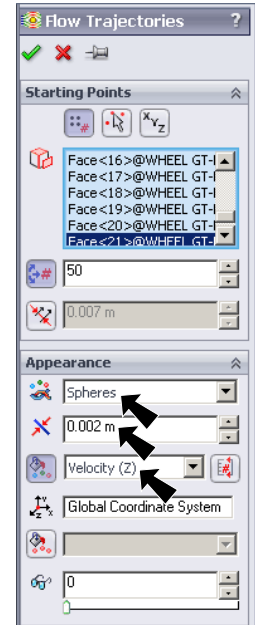


Fig. 39

Step 3. Click **Play**  in animation control panel, Fig. 38. Click **Stop** .

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

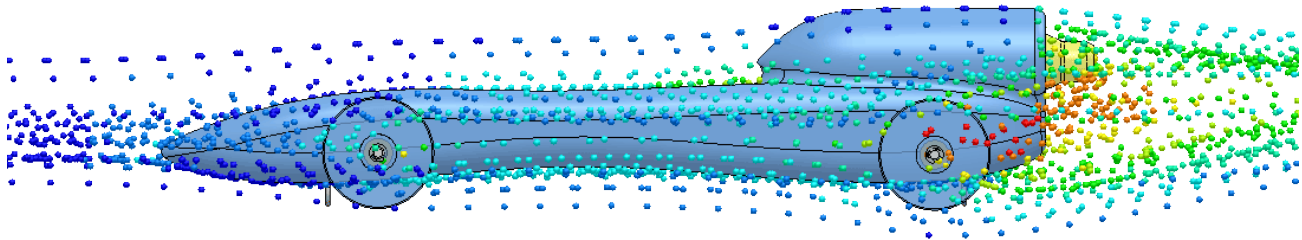



Fig. 40

## L. Create Animation Movie.

Step 1. In the Animation control panel turn on **Capture Region** , Fig. 41.

Step 2. Rotate view to view **rear of car** as shown in Fig. 42. 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. 42.

Step 4. Click **Record** , Fig. 41.

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

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

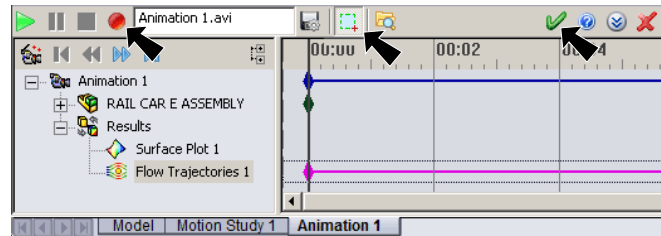


Fig. 41

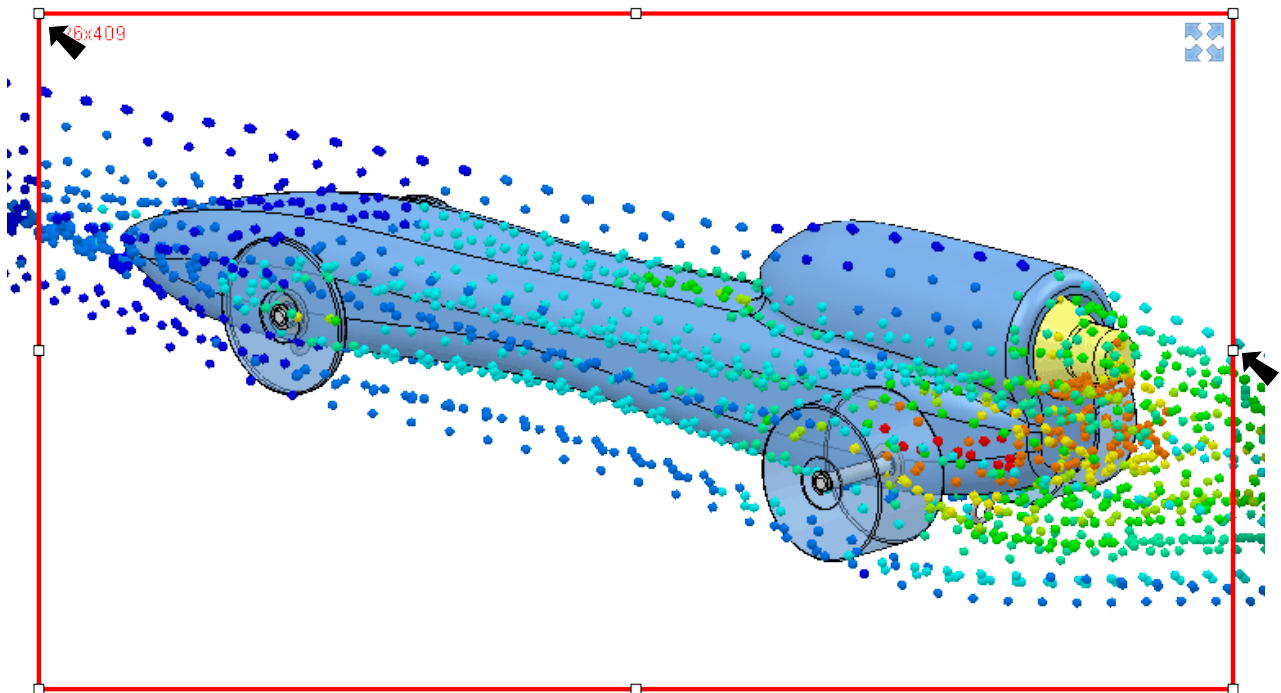


Fig. 42