Rail Car Form
CFD Flow Sim

A. New Wind Tunnel Component.

Step 1. Confirm **Autodesk CFD 2019 Ultimate** is installed. https://www.autodesk.com/education/free-software/cfd-ultimate

Step 2. Open your **BODY RAIL FORM** file.

Step 3. **Right click** the top node the **BODY RAIL FORM** Assembly in the Browser and select **New Component**, **Fig. 1**.

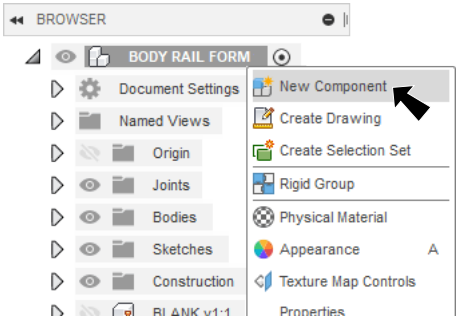


Fig. 1

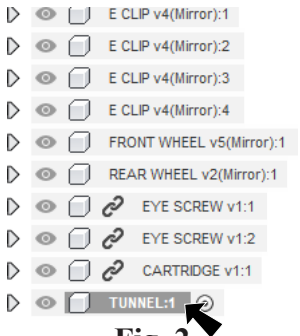


Fig. 2

Step 4. **Rename** New Component **Tunnel**, **Fig. 2**.

Step 5. Expand **Sketches** folder and **Show** **Sketch7** (sketch created in Render Chapter), **Fig. 3**.

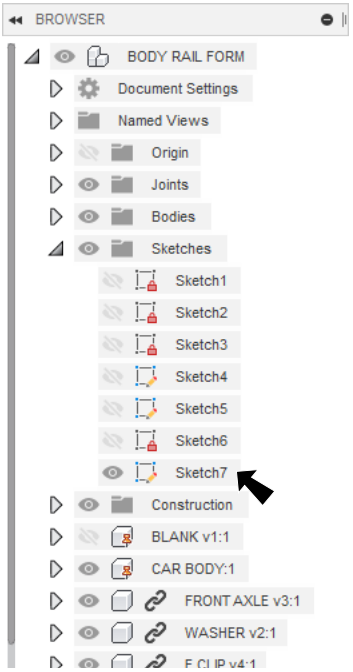


Fig. 3

Step 6. Expand Construction folder, **right click Plane2** (Plane created in Render Chapter) and click **Create Sketch** from menu, **Fig. 4**.

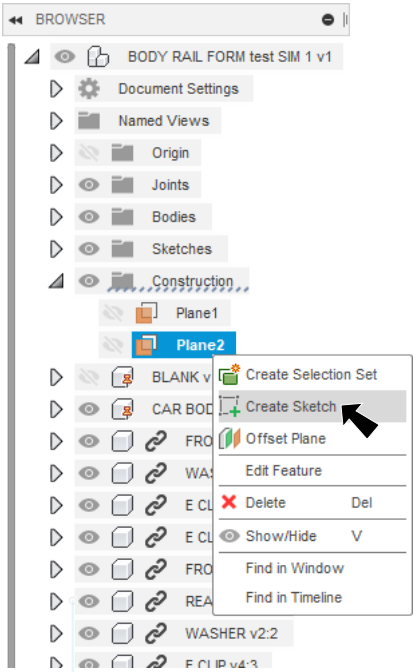


Fig. 4

Step 7. On the Sketch tab SKETCH click Create Menu > Rectangle > **Center Rectangle**.

Step 8. Click **Front** on View Cube FRONT.

Step 9. Sketch a center rectangle, **Fig. 5**.

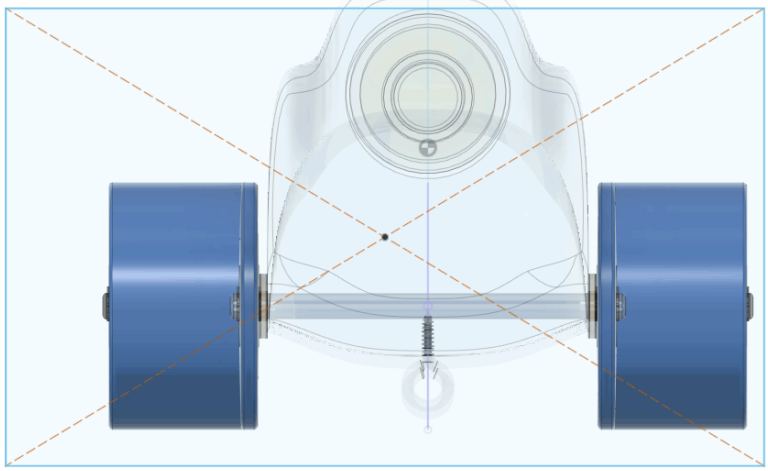


Fig. 5

Step 10. Click **Midpoint** in the Constraints panel of toolbar and click **bottom endpoint of vertical line in Sketch7** and **bottom horizontal line**, **Fig. 6**.

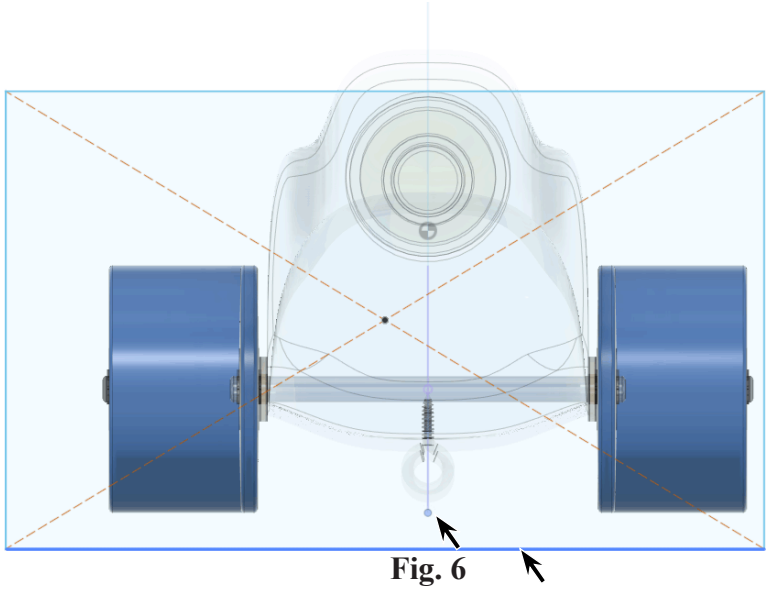


Fig. 6

Step 11. Click **Dimension** **(D)** on the toolbar.

Step 12. Add dimensions, **Fig. 7**.

Step 13. Click **Right View** on View Cube .

Step 14. Switch Visible Style to **Wireframe** (**Ctrl-7**). To switch to Wireframe, click the Display Settings pull-down in the Navigation Bar at the bottom of the canvas and select **Visual Style > Wireframe**, **Fig. 8**.

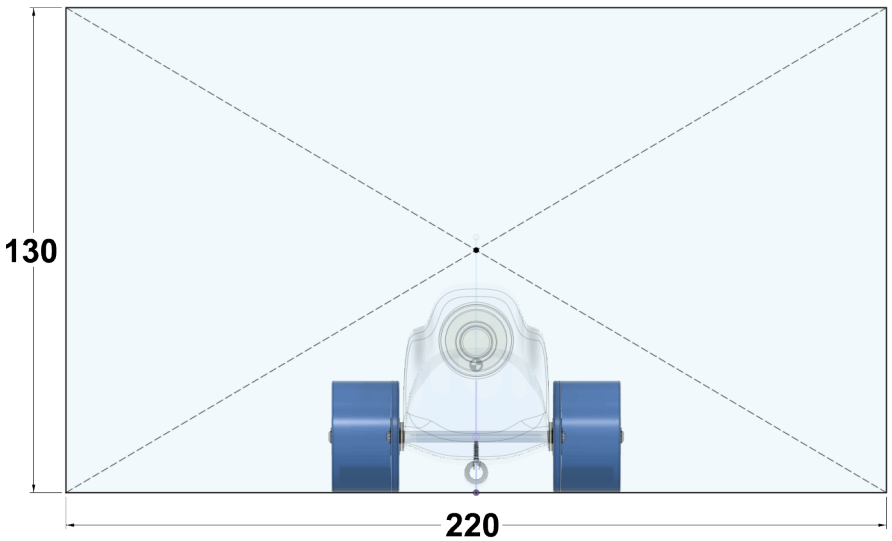


Fig. 7

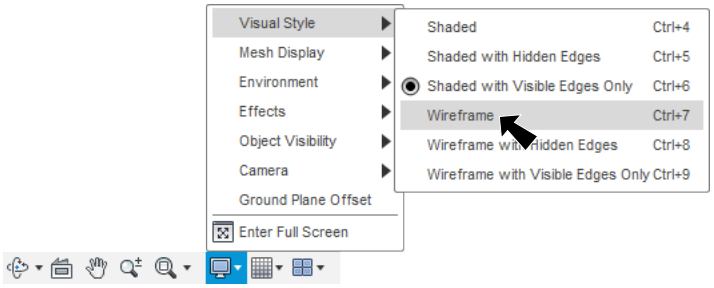


Fig. 8

Step 15. On the Solid tab SOLID click **Extrude** (**E**).

Step 16. In the Extrude panel set, **Fig. 9**
Profile click **rectangle**, **Fig. 10**
Start **Offset Plane**
Offset **-150**
Direction **One Side**
Distance **520**
Operation **New Body**
click OK.

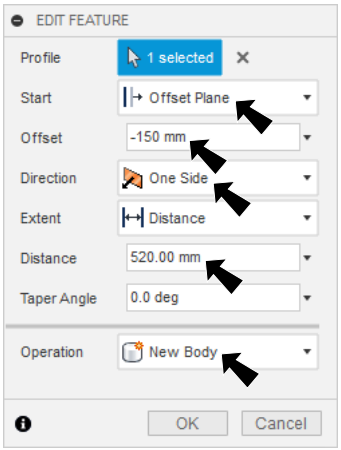


Fig. 9

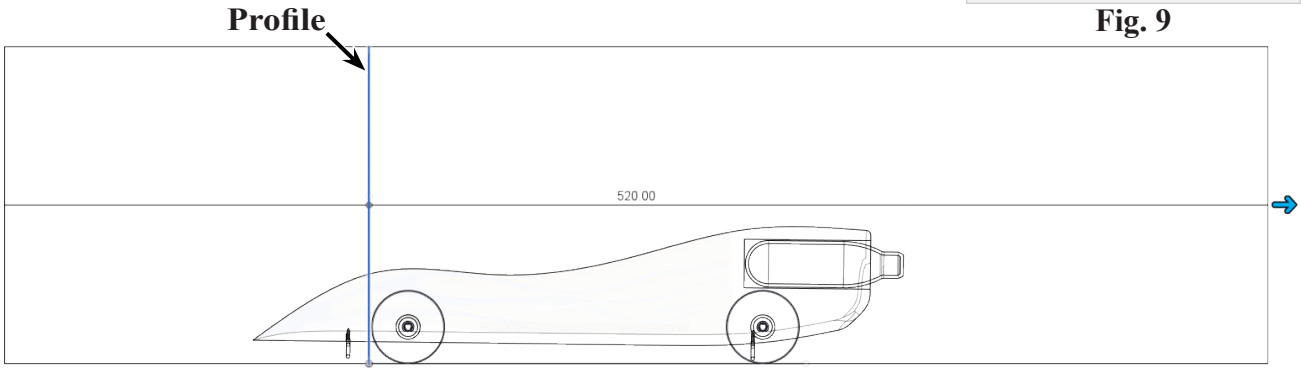


Fig. 10

Step 17. Click **Home** (Isometric) on View Cube .

Step 18. Switch Visible Style to **Shaded with visible Edges Only** (**Ctrl-6**). To switch to Wireframe, click the Display Settings pull-down in the Navigation Bar at the bottom of the canvas and select **Visual Style > Shaded with visible Edges Only**, **Fig. 11**.

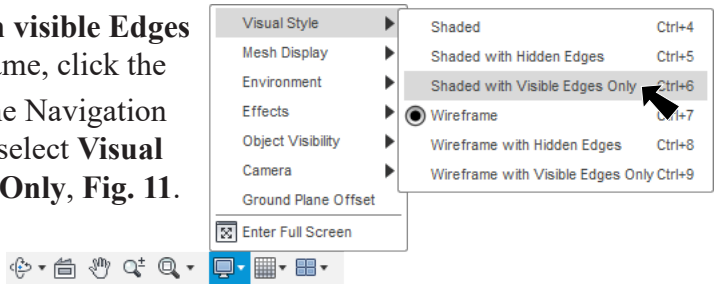


Fig. 11

B. Appearance.

Step 1. Display the Appearance panel, use **A** key.

Step 2. In the Appearance Panel:
under Library, **Fig. 12**.
click **Glass**
expand **Smooth**
drag **Glass(Clear) onto Tunnel**, **Fig. 13**.
click Close.

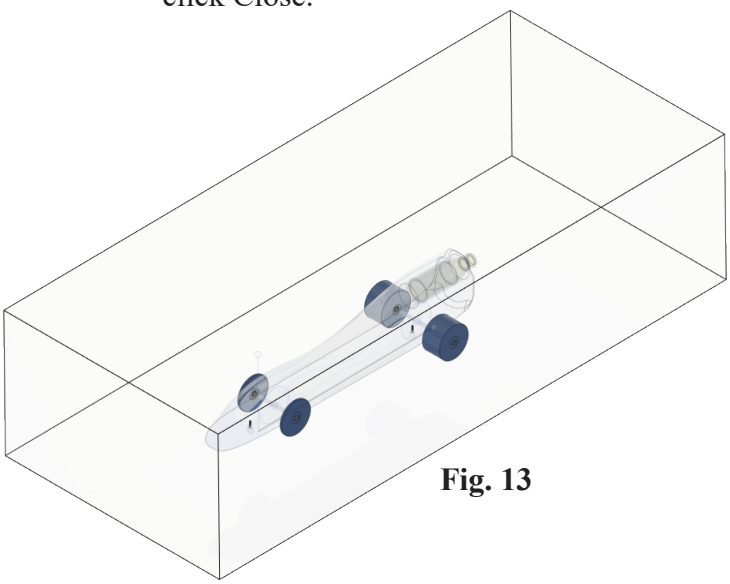


Fig. 13

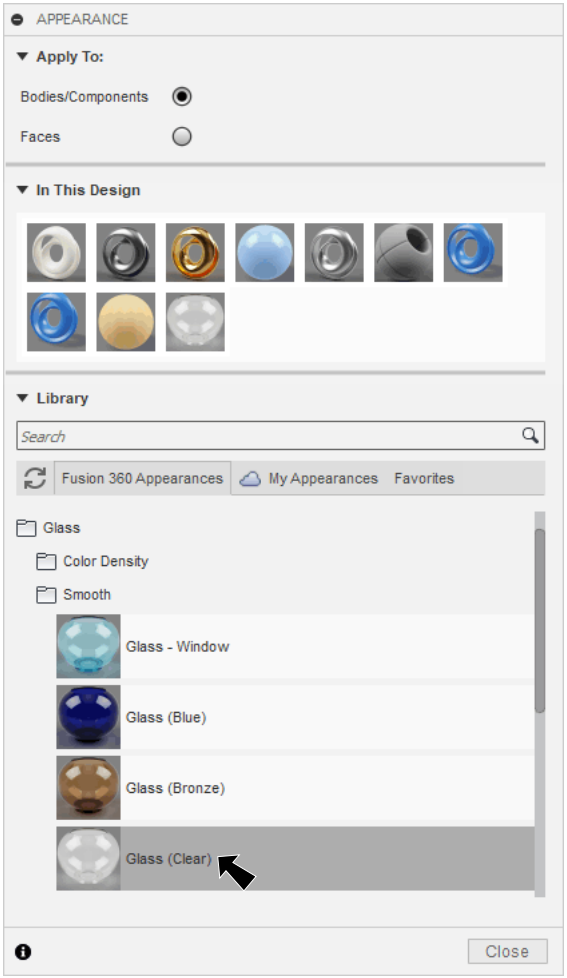


Fig. 12

C. Ground Tunnel and Hide Sketch.

Step 1. **Right click Tunnel** in Browser and click **Ground**, **Fig. 14**.

Step 2. Expand **Sketches** folder and **Hide** **Sketch7**, **Fig. 15**.

Step 3. **Activate** **BODY RAIL FORM** Assembly, **Fig. 15**.

Step 4. Save. Use **Ctrl-S** and press ENTER.

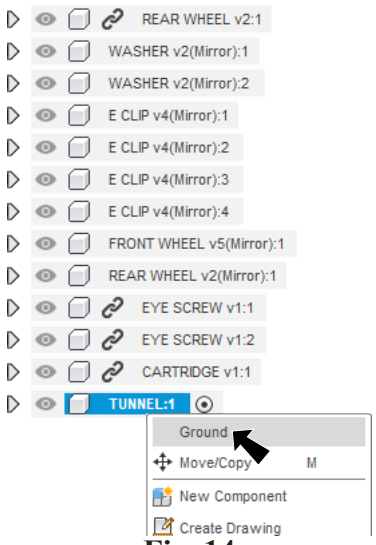


Fig. 14

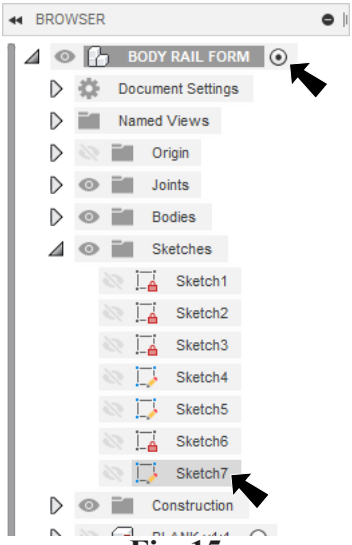


Fig. 15

D. Simulation Workspace.

Step 1. Confirm **Autodesk CFD 2019 Ultimate** is installed.
https://www.autodesk.com/education/free-software/cfd-ultimate

Step 2. Switch to the Simulation workspace. To switch, click Design DESIGN in the Change Workspace toolbar and click **Simulation** SIMULATION from the menu.

Step 3. In the New Study dialog box **double click Simplify** , **Fig. 16**.

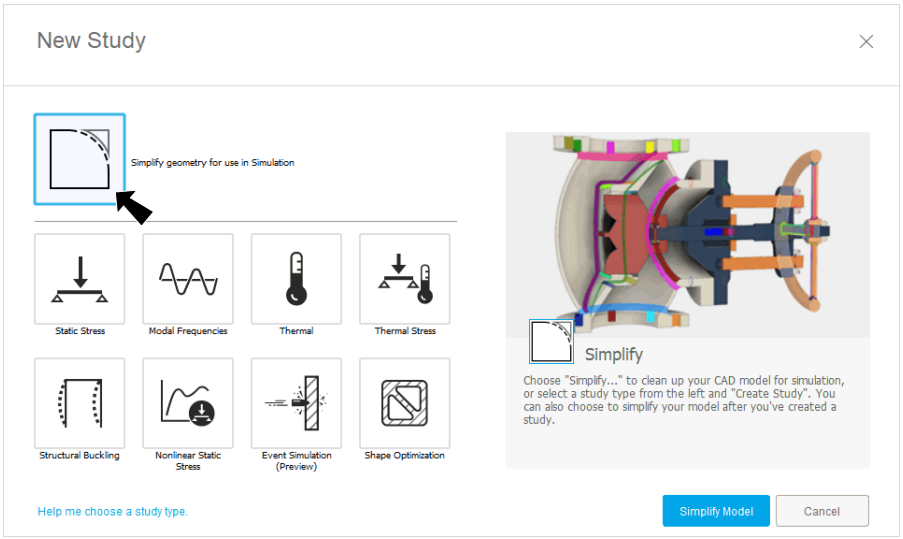


Fig. 16

E. Simplify.

Step 1. In the Browser expand **Model Components folder** and **Hide**
Bodies folder
Blank component
both **Eye Screw** components
Tunnel component, **Fig. 17**. (show Tunnel after Combine)

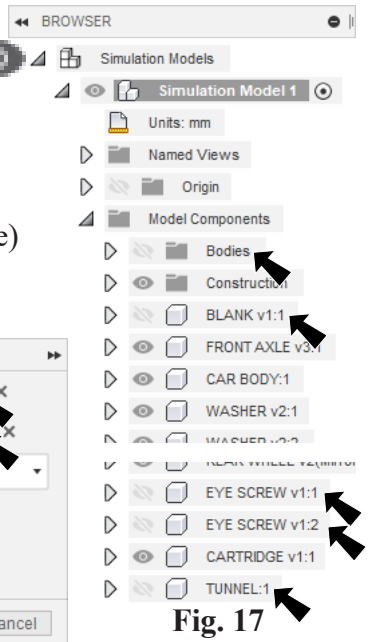


Fig. 17

Step 2. On the Simplify Solid tab
SIMPLIFY SOLID click Modify Menu > Combine .

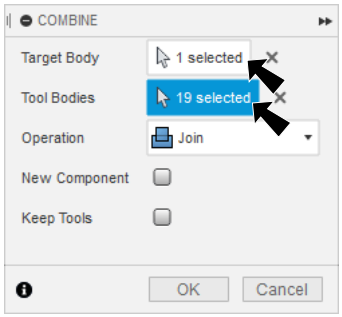


Fig. 18

Step 3. In the Combine panel set, **Fig. 18**
Target Body click **car body**, **Fig. 19**
Tool Bodies **drag selection around all other components**, **Fig. 20**
click OK.

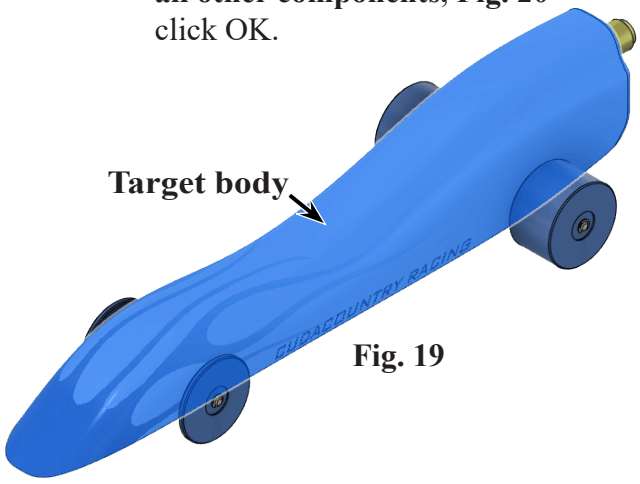


Fig. 19

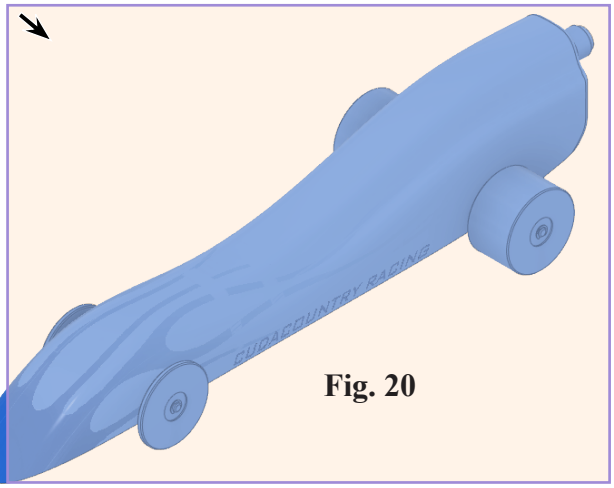


Fig. 20

Step 4. In the Browser, **Show** **Tunnel** component, **Fig. 21**.

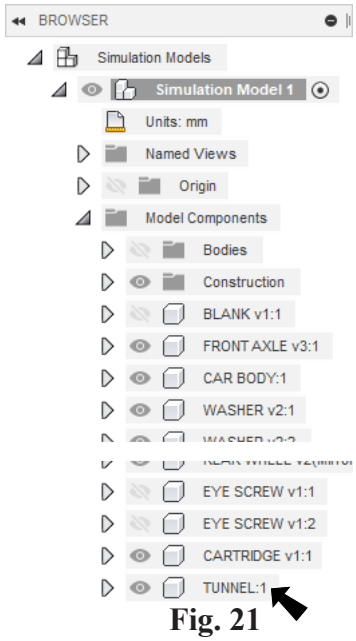


Fig. 21

F. Launch CDF.

Step 1. On the Tools tab TOOLS click **Simulate in CFD** on the toolbar.

Step 2. In the Design Study Manager dialog box click the **Path** icon, **Fig. 22**. Select a folder to save CFD files. Back in the Design Study Manager dialog box, click Launch.

Step 3. In the Geometry Tools dialog box on the **Edge Merge tab** Edge Merge click **Merge**, **Fig. 23**.

Step 4. Click the **Small Object** tab Small Object, **Fig. 24**
drag the **Tolerance slider to right** until the Remove button is highlighted, around **.00200488**
click **Remove** and **Close**.

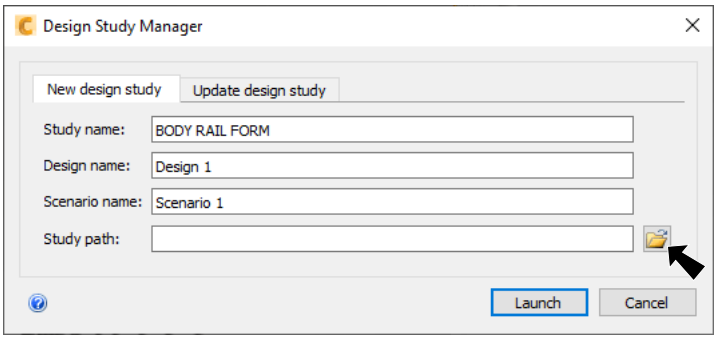


Fig. 22

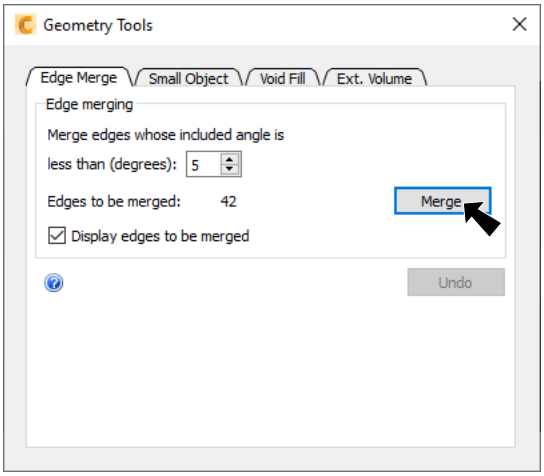


Fig. 23

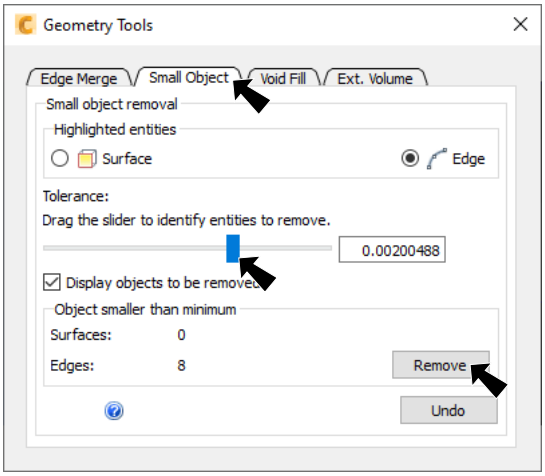


Fig. 24

G. Setup.

Step 1. **Right click** the Tunnel and click **Outline** from the menu, **Fig. 25**.

Step 2. Click the **Tunnel** outline and click **Edit** in the pop-up, **Fig. 26**.

Step 3. In the Materials dialog box:
confirm Name **Air**, **Fig. 27**
click **Apply**.

Step 4. In the Design Study Bar:
under Material, **Fig. 28**
right click Unassigned and
click **Edit** from menu.

Step 5. In the Materials dialog box:
set Type **Solid**, **Fig. 29**
ABS is OK.
click **Apply**.

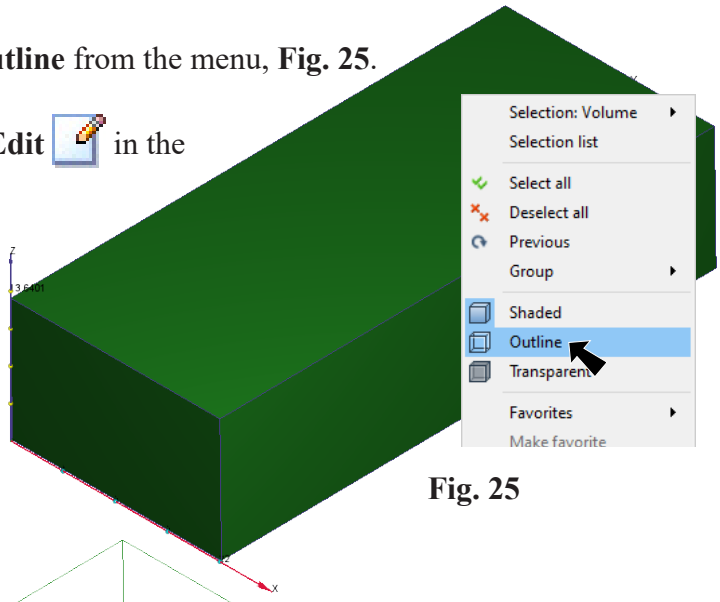


Fig. 25

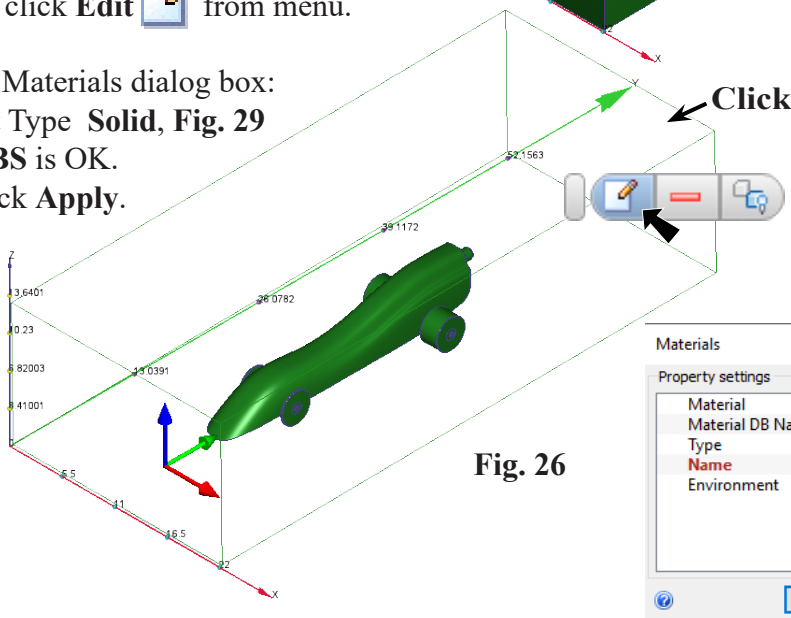


Fig. 26

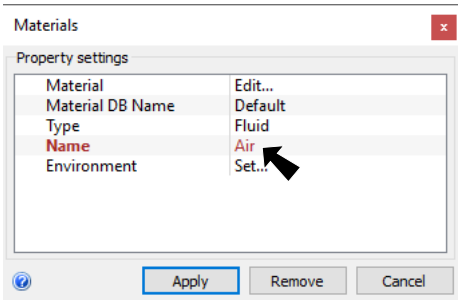


Fig. 27

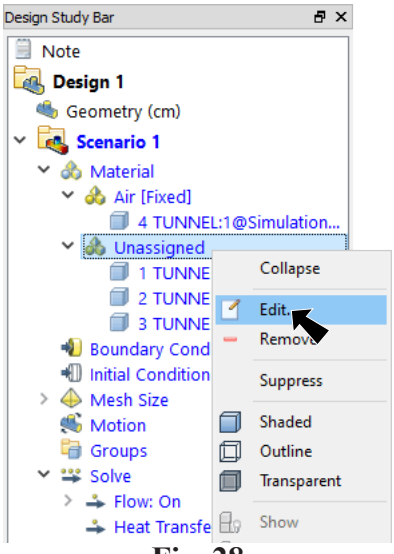


Fig. 28

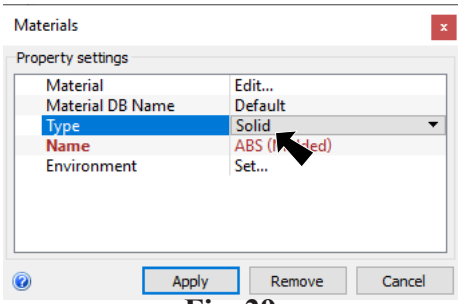
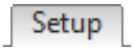
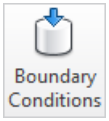


Fig. 29

H. Boundary Conditions.

Step 1. On the Setup tab 

click **Boundary Conditions** 

in the Setup Task panel of toolbar.

Step 2. Click the **front surface of Tunnel** outline and click **Edit**  in the pop-up, **Fig. 30**.

Step 3. In the Boundary Conditions dialog box:
Velocity Magnitude **75** (m/s), **Fig. 31**
click **Apply**.

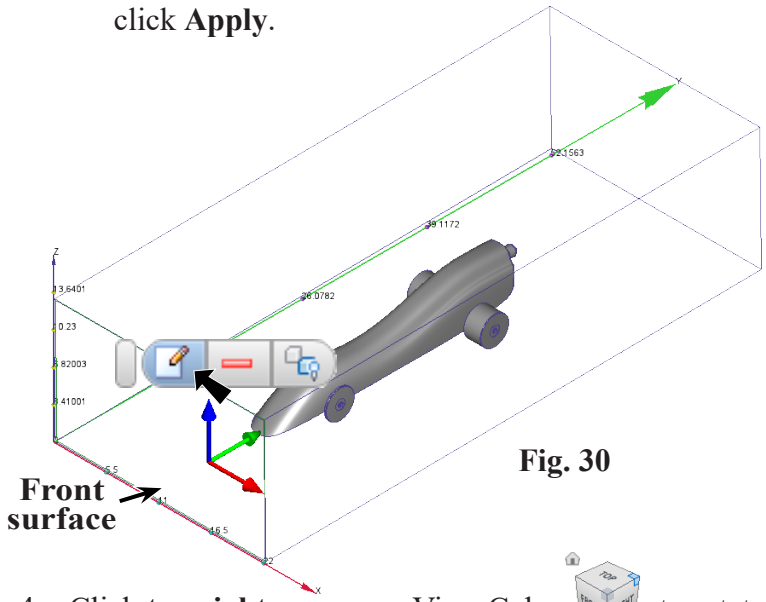


Fig. 30

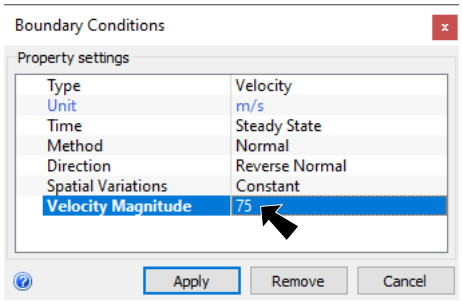


Fig. 31

Step 4. Click **top right corner** on View Cube  to rotate view to view rear face of Tunnel.

Step 5. Click the **rear surface of Tunnel** outline and click **Edit**  in the pop-up, **Fig. 32**.

Step 6. In the Boundary Conditions dialog box:
Type **Pressure**, **Fig. 33**
confirm Pressure **0**
click **Apply**.

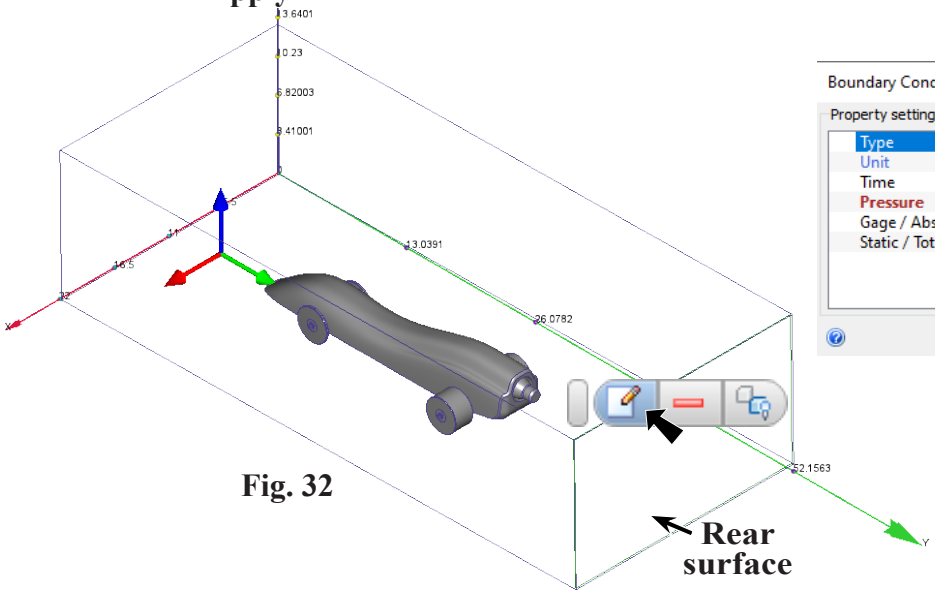


Fig. 32

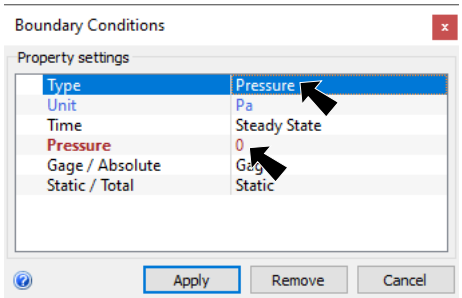


Fig. 33

Tip: If the Solver fails unselect use Cluster on Solver Computers and Configurations. To unselect Cluster, click Simulation menu > Solver Computers and uncheck **Use Cluster** of MyComputer, **Fig. 34**.

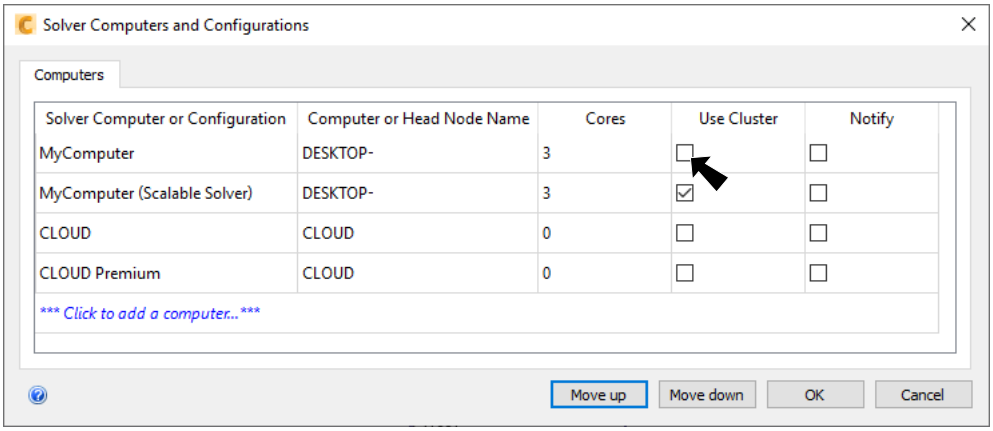


Fig. 34

I. Solve.

Step 1. On the Setup tab Setup click **Solve** Solve in the Simulation panel of toolbar.

Step 2. In the Solve dialog box on the **Control** tab Control, **Fig. 35**.
set **Iterations to Run 130**
click **Result quantities** button.

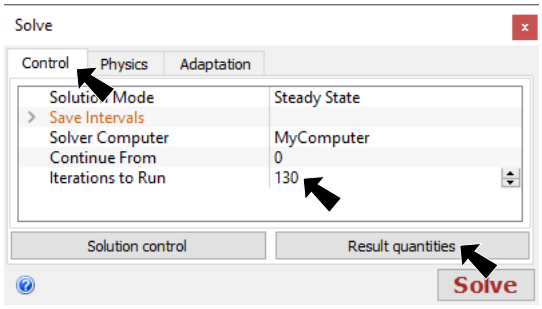


Fig. 35

Step 3. In Result Quantities dialog box set:
under Basic flow values, **Fig. 36**
uncheck **Temperature**
under Turbulence
check **All button**
under Miscellaneous
check **Pressure coefficient**
uncheck **Wall heat flux**
and **Wall film coefficient**
click OK

Step 4. Back in Solve dialog box
click **Solve**.

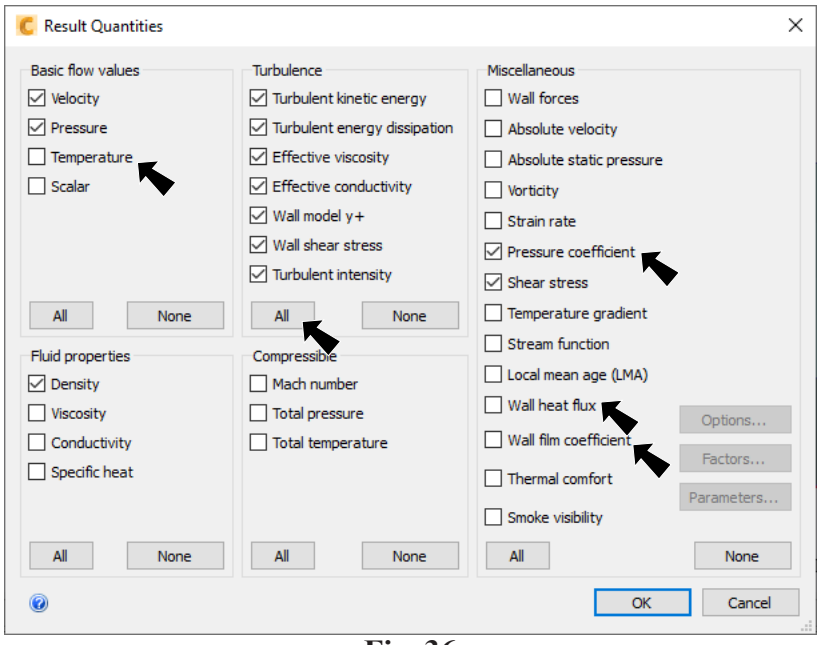


Fig. 36

Step 5. In the Output Bar the mesh starts calculating, **Fig. 37**.

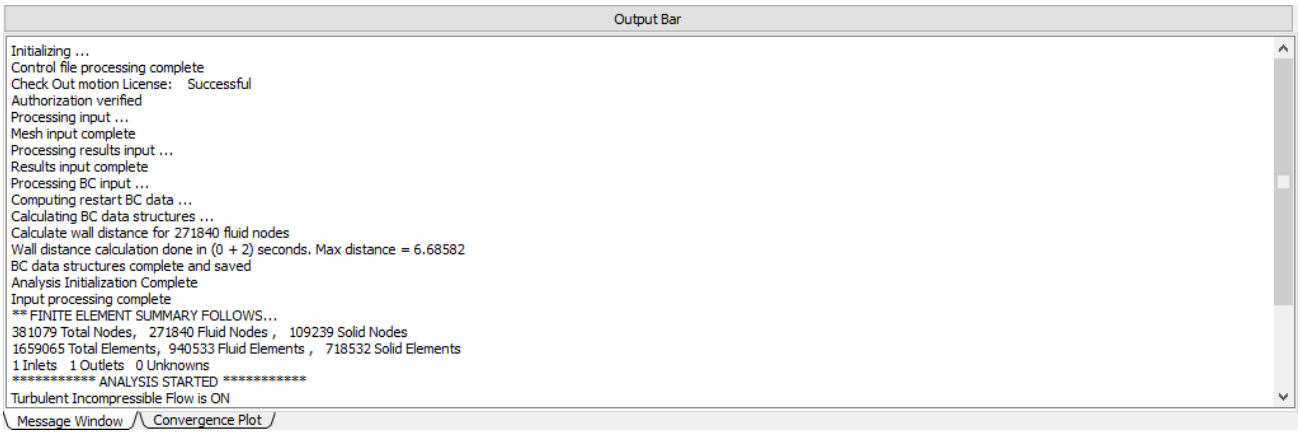


Fig. 37

Step 6. After the mesh is calculated CFD will switch to the **Convergence Plot** tab Convergence Plot and start generating curves. When the curves converge or flatten out the analysis is complete, **Fig. 38**.

The analysis could take several minutes depending on your computer. While the analysis is running we will create a plane.

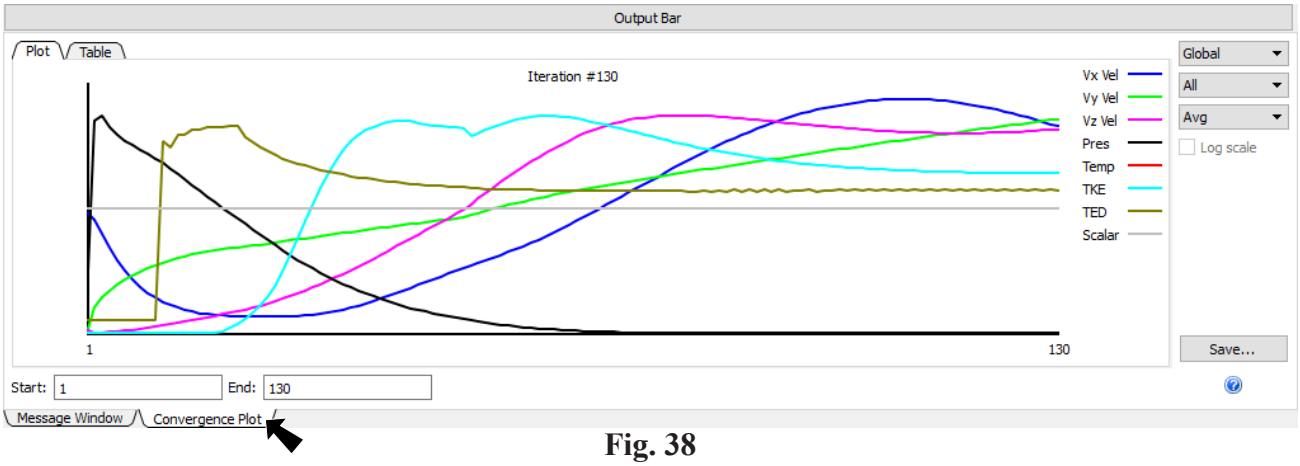


Fig. 38

J. Create Plane.

Step 1. **Right click** the Tunnel and click **Outline** from the menu, **Fig. 39**.

Step 2. On the Results tab Results click **Planes** Planes in the Results Tasks panel of toolbar.

Step 3. Click **Add** in the Planes panel of toolbar.

Step 4. Click **Edit** Edit in Planes panel of toolbar.

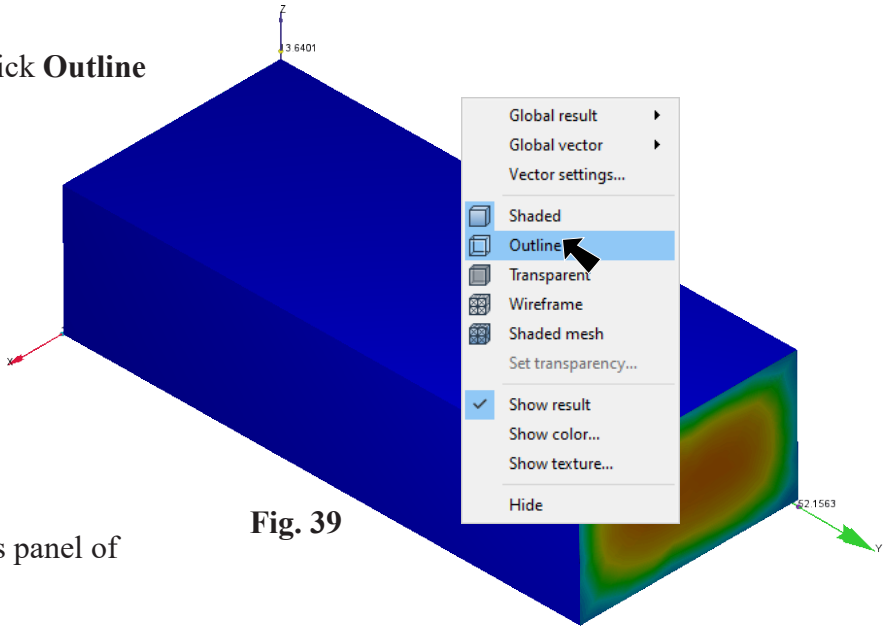


Fig. 39

Step 5. In the Plane Control dialog box:

On the Controls tab Controls, **Fig. 40**

The Plane should be on the Y axis and centered on the Tunnel, **Fig. 41**

To position Plane on Y axis, click **Normal** button

In the Edit plane normal dialog box, **Fig. 42**

set **X 1** and click OK.

Back in the Plane Control, set Appearance **Transparent**

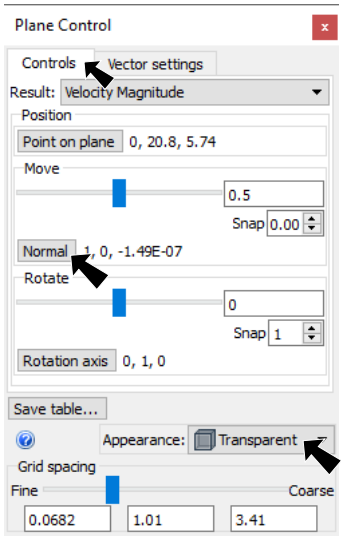


Fig. 40

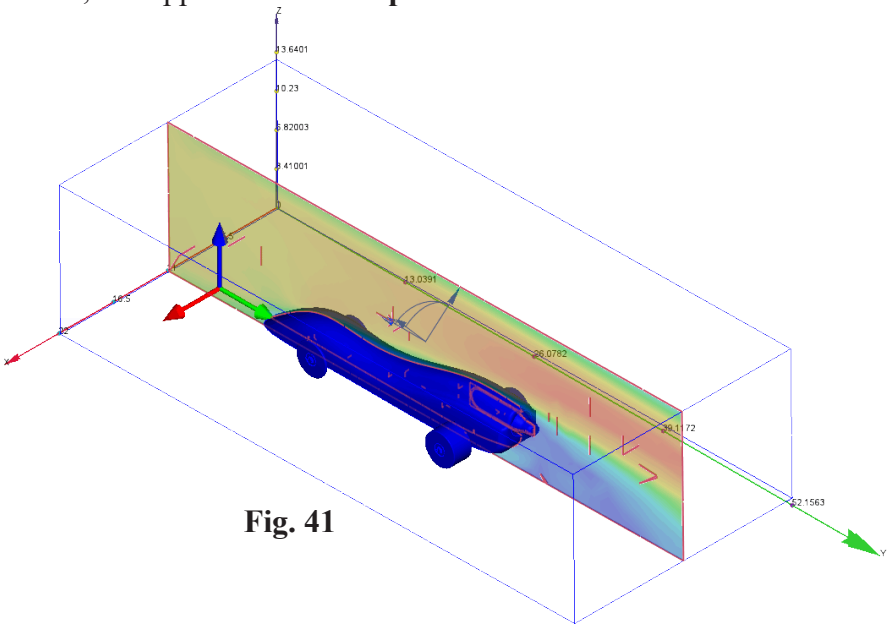


Fig. 41

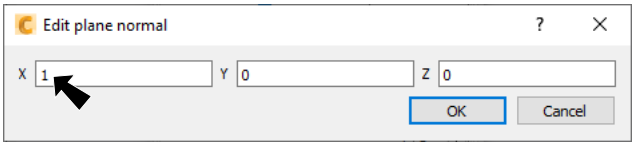
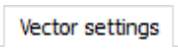


Fig. 42

Step 6. Click **Right** on View Cube 

.

Step 7. In the Plane Control dialog box set:

Click the **Vector setting** tab 

, **Fig. 43**

Results **Velocity Vector**

drag the **Length slider** to around **.4**

Click **More** button

Arrowhead size **.15**

Scale factor **.1**

Grid spacing drag slider to **around 1**

click **Close** .

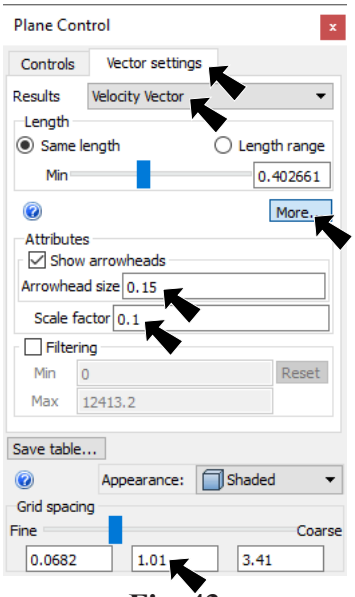


Fig. 43

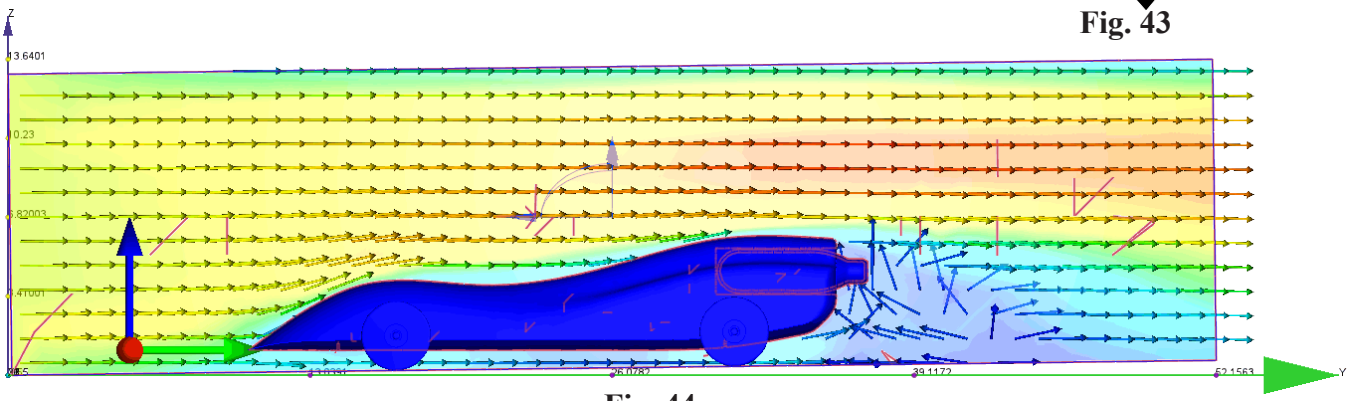
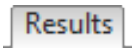


Fig. 44

Step 8. On the Results tab 

click **Static Image** 

in the Image panel of toolbar and same image. Also, save a Dynamic Image 

.

K. Hide Plane.

Step 1. Click **Home**  (Isometric) on View Cube 

.

Step 2. In the Design Study Bar:

under Planes, **Fig. 45**

Hide ☐ **Plane 1**.

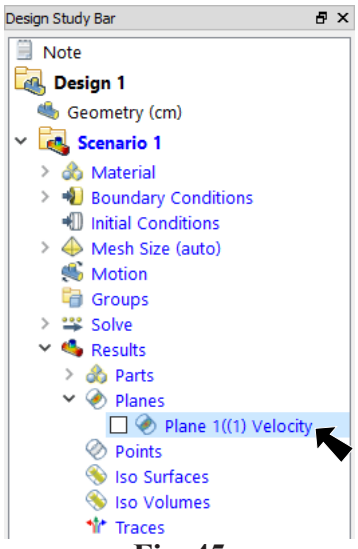
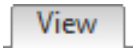


Fig. 45

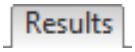
L. Hide Coordinates Axes.

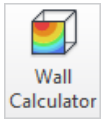
Step 1. On the View tab 

click **Axes** 

in the Appearances panel of toolbar, then uncheck **Show Bounding Coordinate Axes**.

M. Drag Coefficient .

Step 1. On the Results tab 

click **Wall Calculator** 

in the Results Tasks panel of toolbar.

Step 2. In the Wall Results dialog box:

Click Select All  to select all (all edges red), **Fig. 46**.

Unselect the Tunnel by selecting the six surfaces.

To unselect:

1) click the 3 surfaces visible in Iso view - 3 edge will be displayed unselect **blue**, **Fig. 47**.

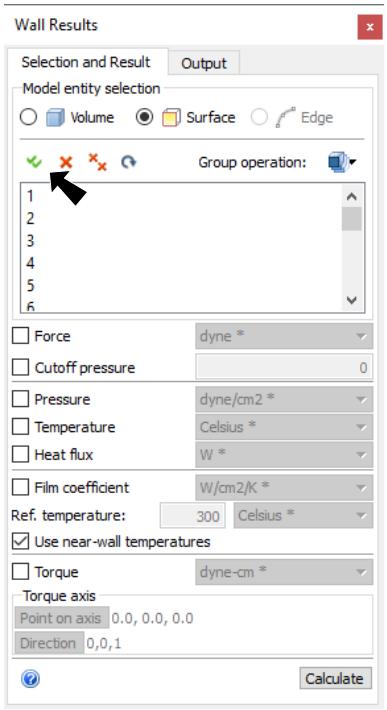


Fig. 46

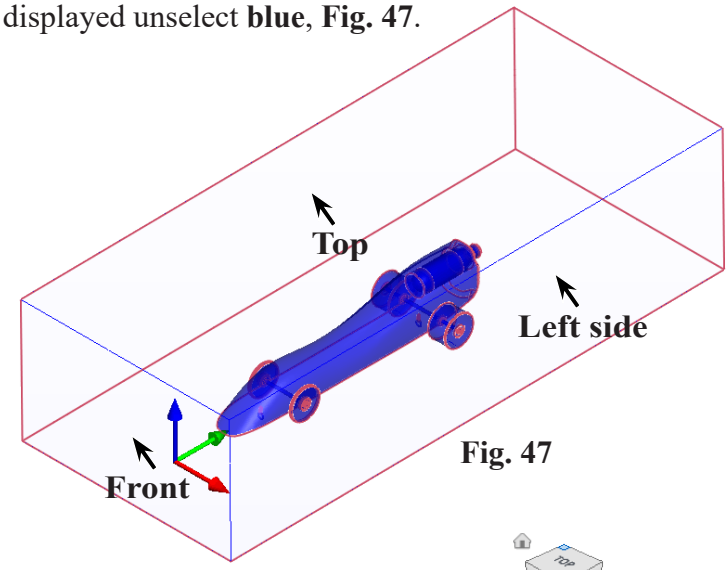


Fig. 47

2) Click **top rear corner of View Cube** 

to rotate.

3) Click **bottom front corner of View Cube** 

to rotate.

4) Click the 3 remaining surfaces, **Fig. 48**.
(Tunnel edges blue)

Tip: **Red edges selected**
Blue edges unselected

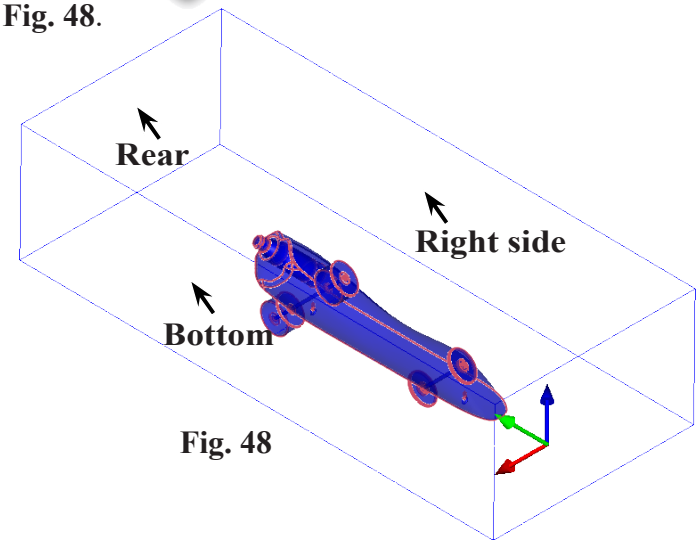


Fig. 48

Step 3. Still in Wall Results dialog box:

check **Force**, **Fig. 49**
set units **Newton**
click **Calculate**

click **Output tab** Output ,
Fig. 50
scroll down to bottom
under Summary
FY (drag) 5.48
FZ (lift) .87
close .

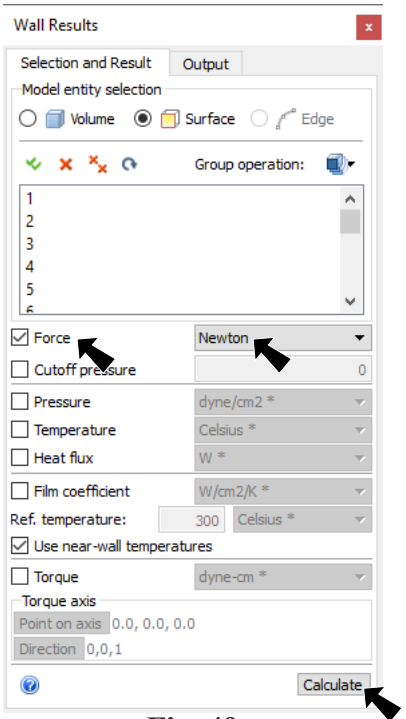


Fig. 49

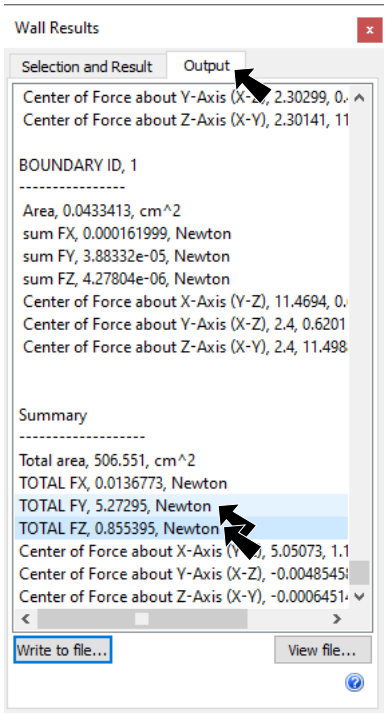


Fig. 50

Step 4. Save. Click **Save** on the Quick Access Toolbar.

N. Traces.

Step 1. Click **Home** (Isometric) on View Cube .

Step 2. Click **Look At** in the Navigation Bar on left edge of the canvas, **Fig. 51**.

Step 3. Click **front surface of Tunnel**, **Fig. 52**.

Step 4. On the View tab View click **Axes** Axes: in the Appearances panel of toolbar, then uncheck **Show Local Coordinate Axes**.

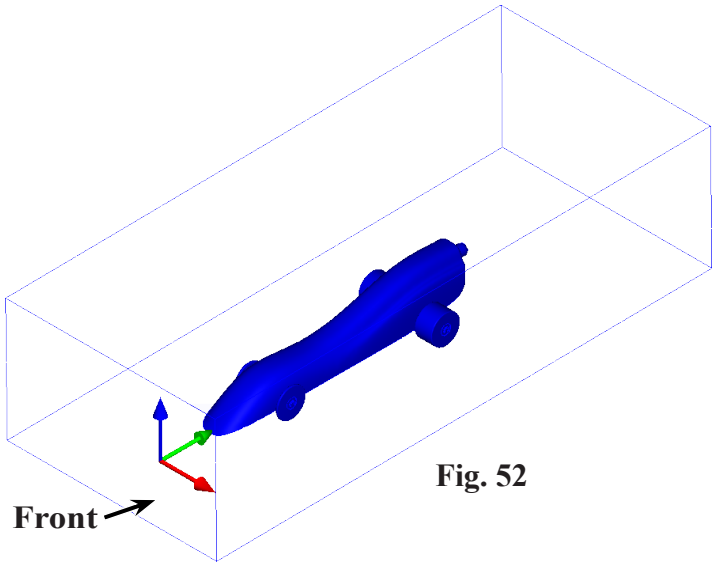


Fig. 52

Fig. 51

Step 5. On the Results tab Results click **Traces** Traces in the Create Set panel of toolbar.

Step 6. In the Create Set panel of toolbar set **Seed Density 1.4**, **Fig. 53**.

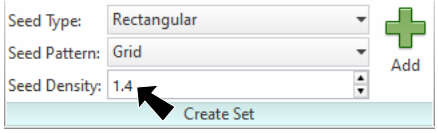


Fig. 53

Step 7. Click **Add** in the Create Set panel of toolbar.

Step 8. Click left of the wheel at ground level to start the pattern. Click to right of other wheel and click up above the top of car body, **Fig. 54**.

Tip: The seed of pattern can be moves by dragging an arrow head of the triad, **Fig. 55**.

Step 9. Click **Right View** on View Cube .

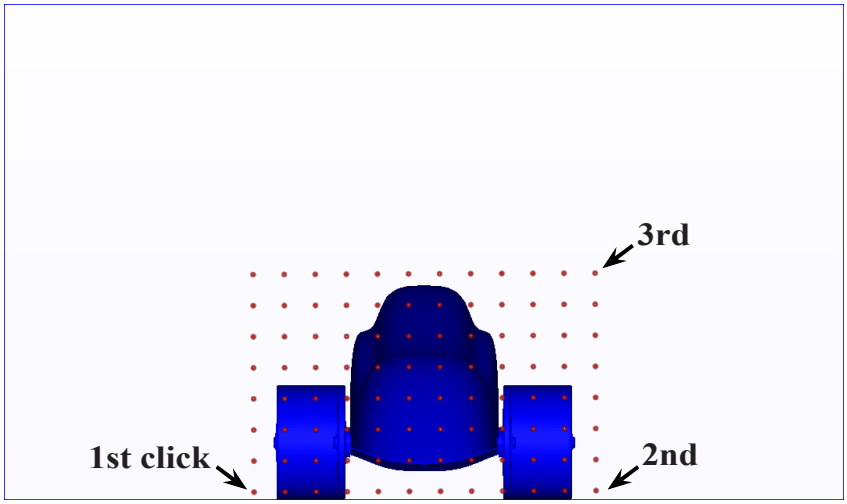


Fig. 54

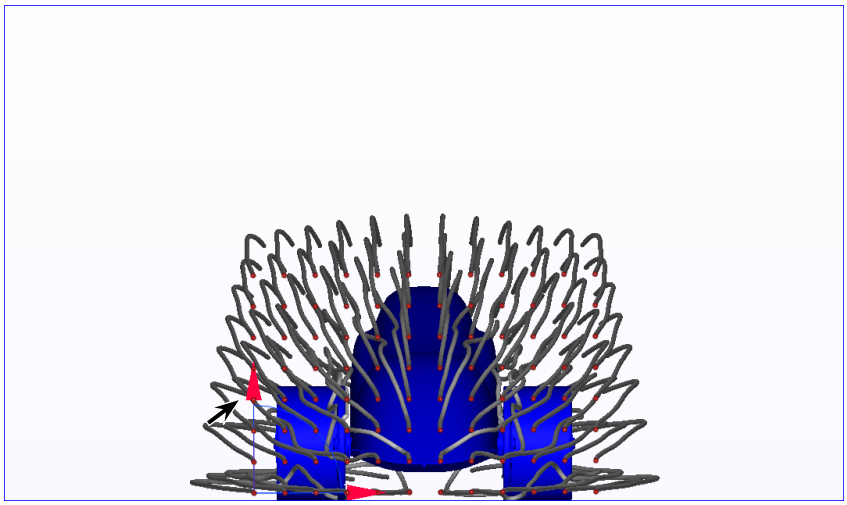


Fig. 55

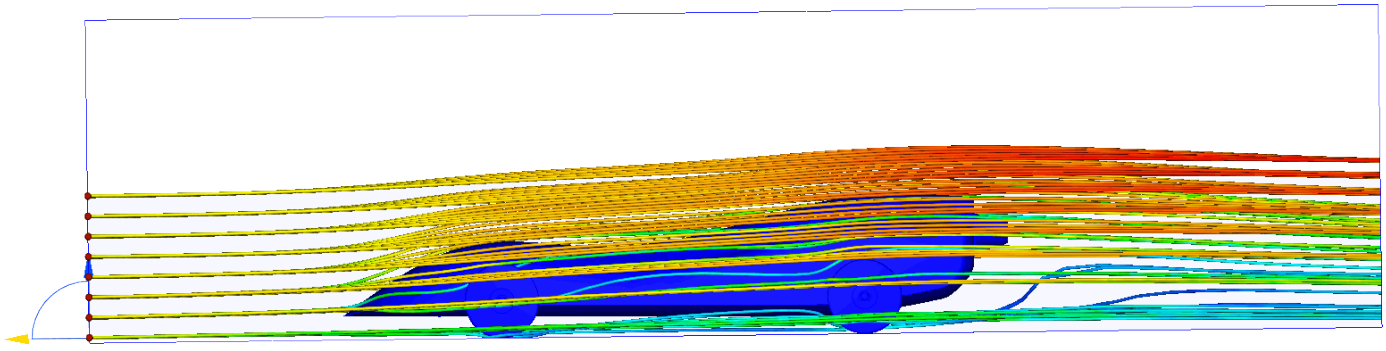
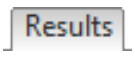


Fig. 56

Step 10. On the Results tab 

click **Edit** 

in the Modify Set panel of toolbar.

Step 11. In Edit Trace dialog box set:
under Properties, **Fig. 57**
Appearance **Spheres**
Width **75**
click Close.

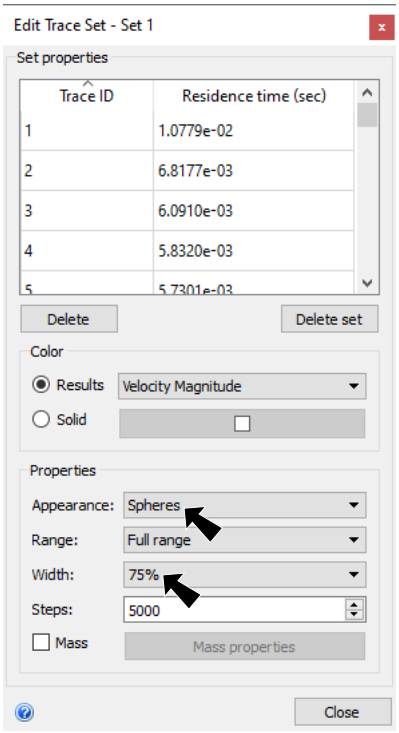
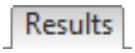
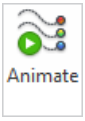


Fig. 57

Step 12. On the Results tab 

click **Edit** 

in the Process Traces panel of toolbar.

Step 13. In Animate Traces dialog box set, **Fig. 59**
Animation Time **8**
click **Play** 
click **Play**  to **Stop**
click **Save avi**
close ✕.

Step 14. Save. Click **Save**  on the Quick Access Toolbar.

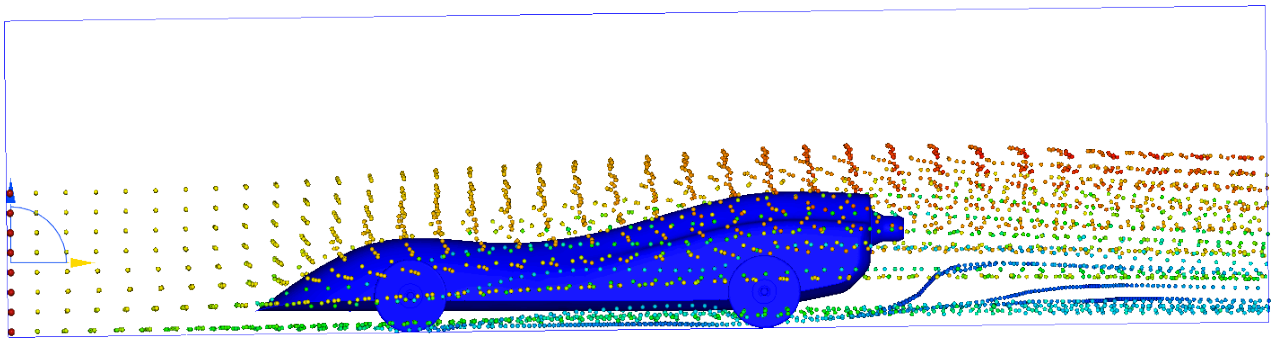


Fig. 58

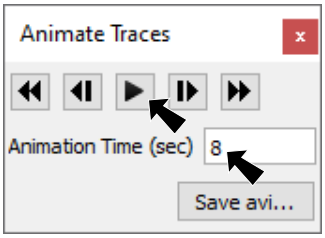


Fig. 59