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.

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. Click Next.
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...
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
- Z max .3
- Z min -.4

Step 3. Set 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 car 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 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.

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 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.

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.
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.
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

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**

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

Click **Stop**.
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.

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