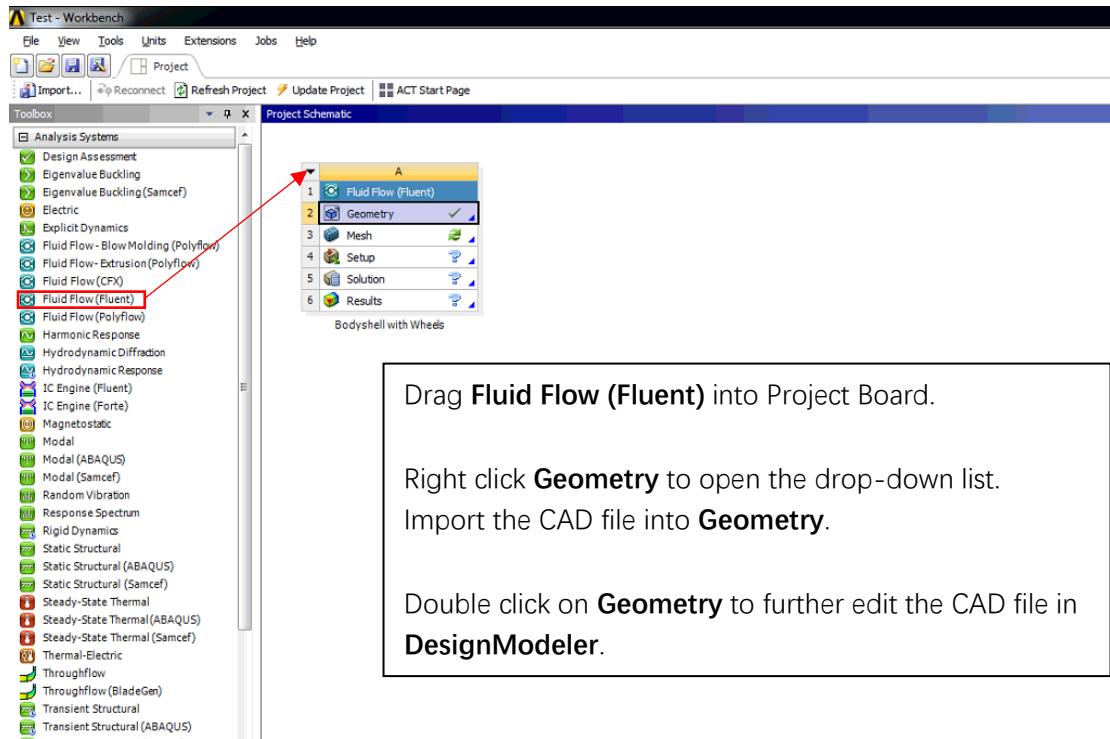


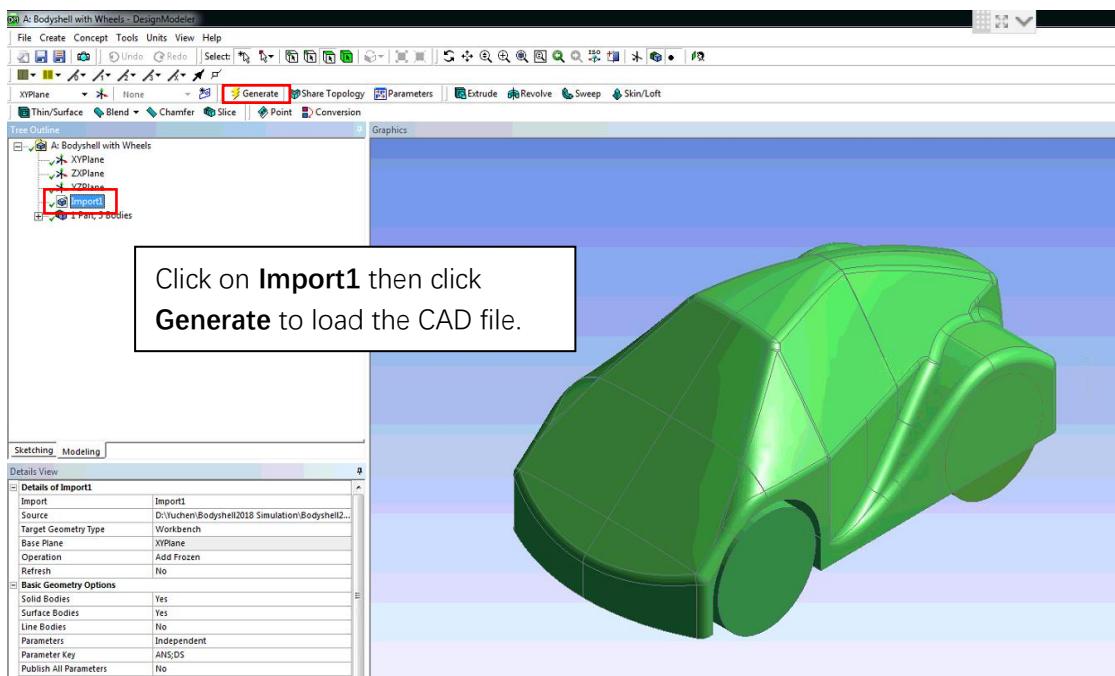
ANSYS Fluent Tutorial

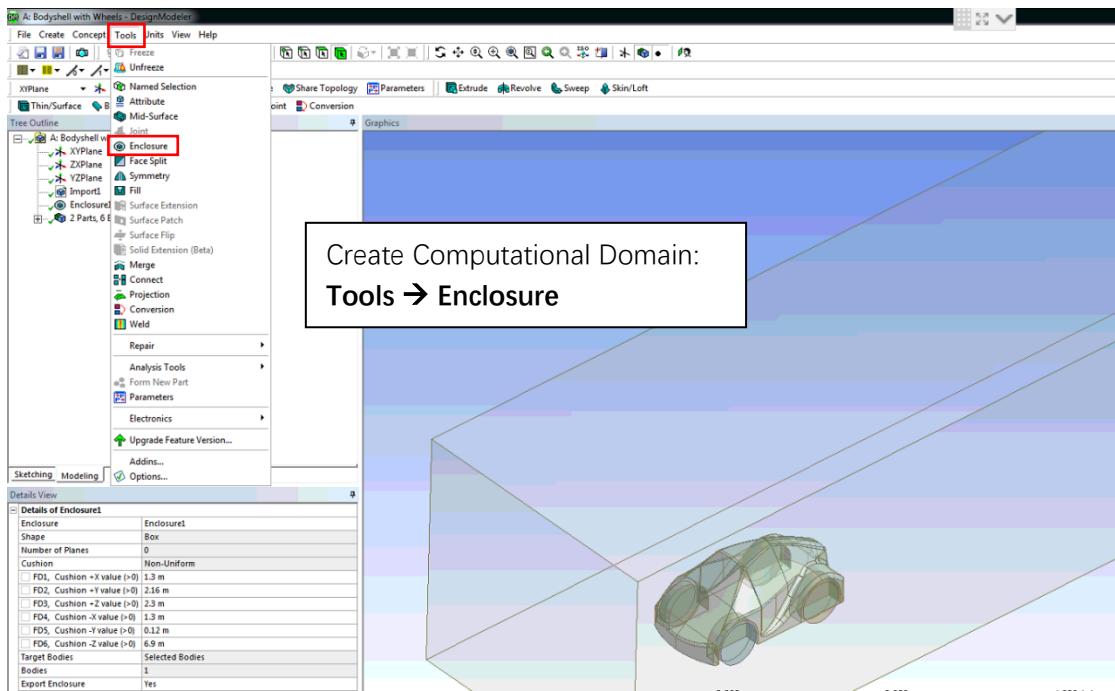
1. Set Up Geometry for Fluent simulation

1.1 Open Workbench



1.2 Edit CAD file in DesignModeler



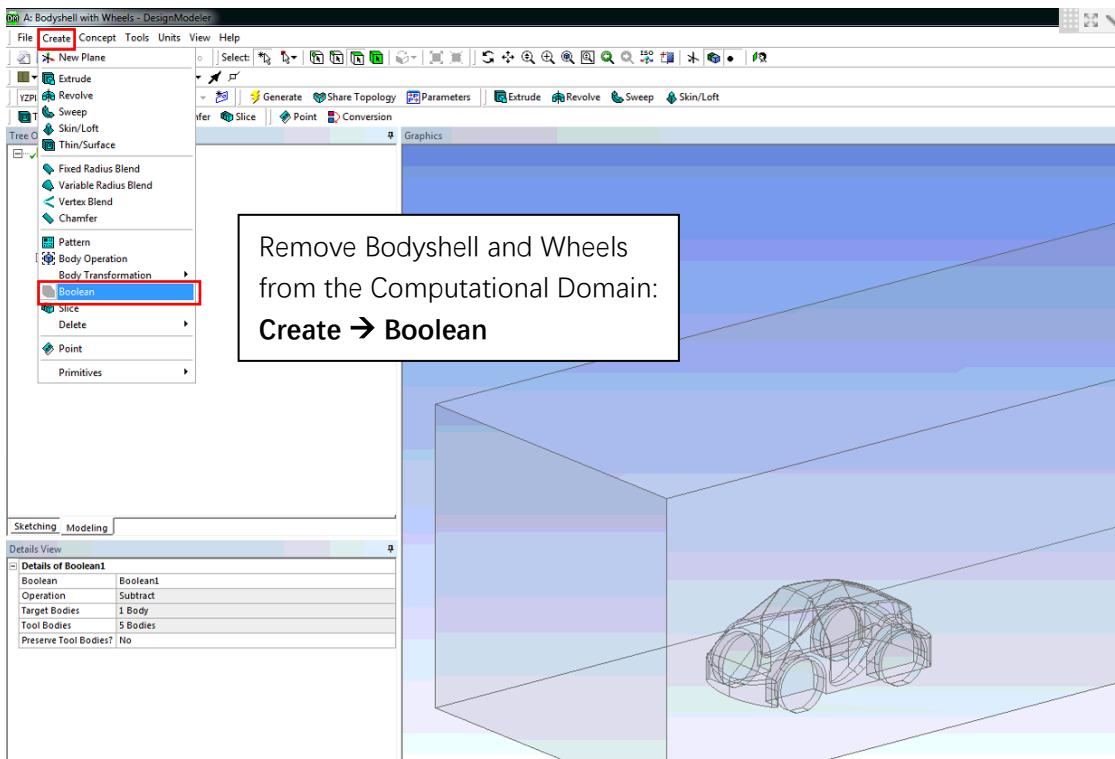


Details View	
Details of Enclosure1	
Enclosure	Enclosure1
Shape	Box
Number of Planes	0
Cushion	Non-Uniform
FD1, Cushion +X value (>0)	1.3 m
FD2, Cushion +Y value (>0)	2.16 m
FD3, Cushion +Z value (>0)	2.3 m
FD4, Cushion -X value (>0)	1.3 m
FD5, Cushion -Y value (>0)	0.12 m
FD6, Cushion -Z value (>0)	6.9 m
Target Bodies	Selected Bodies
Bodies	1
Merge Parts?	No
Export Enclosure	Yes

Give Dimensions to the Domain:
 Inlet (Upstream): 1 Car Length
 Outlet (Downstream): 3 Car Length
 Total Width: 3 Car Width
 Total Height: 3 Car Height
 Total Length: 5 Car Length

Select ONLY the Car Body without
Wheels as the Target Body.

Click Generate.



Remove Bodyshell and Wheels
from the Computational Domain:
Create → Boolean

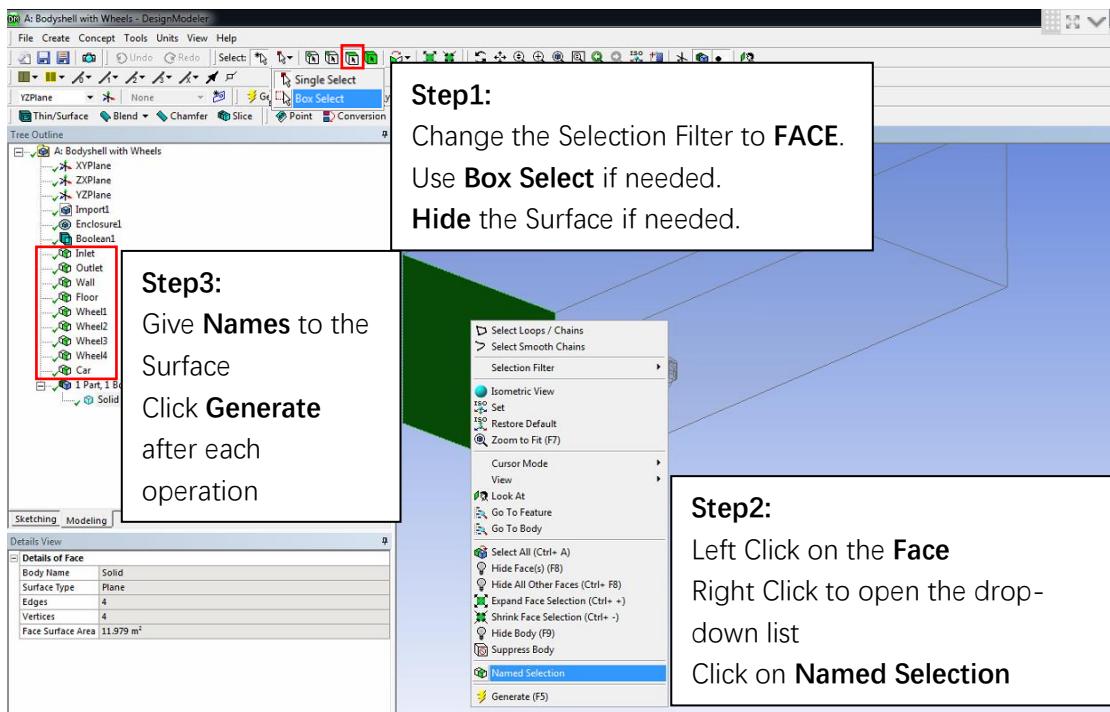
Details View	
Details of Boolean1	
Boolean	Boolean1
Operation	Subtract
Target Bodies	1 Body
Tool Bodies	5 Bodies
Preserve Tool Bodies?	No

Select **Subtract** from **Operation** drop-down list.

Select the **Enclosure Body** as the **Target Body**.

Select the **Bodyshell and 4 Wheels** as the **Tool Bodies**.

Click **Generate**.

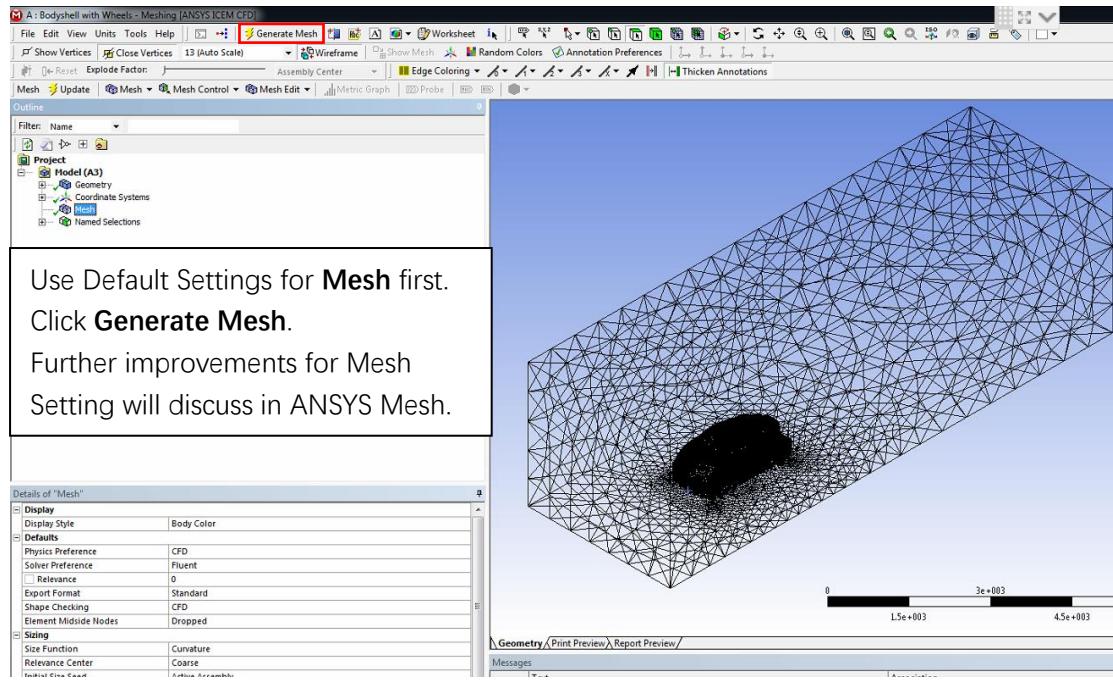


Close **DesignModeler** window once finished all the steps above.

Go back to **Workbench**.

2. Generate Mesh for Computational Domain

2.1 Double Click on **Mesh** to open **Meshing** window.

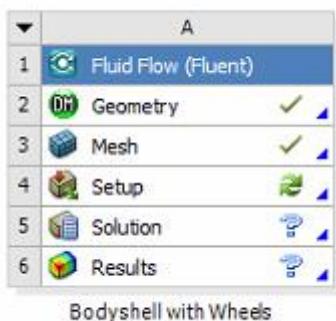


Close **Meshing** window once finished.

Go back to **Workbench**.

Right Click on **Mesh** to open the drop-down list, Left Click **Update**.

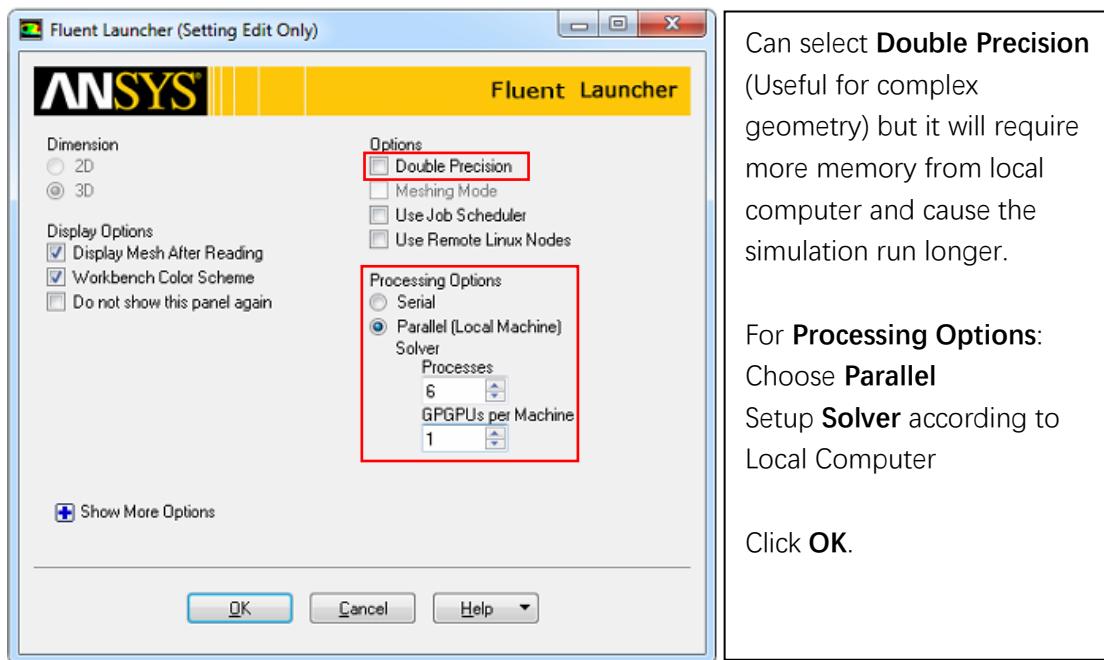
Green tick will show once finish updating mesh.



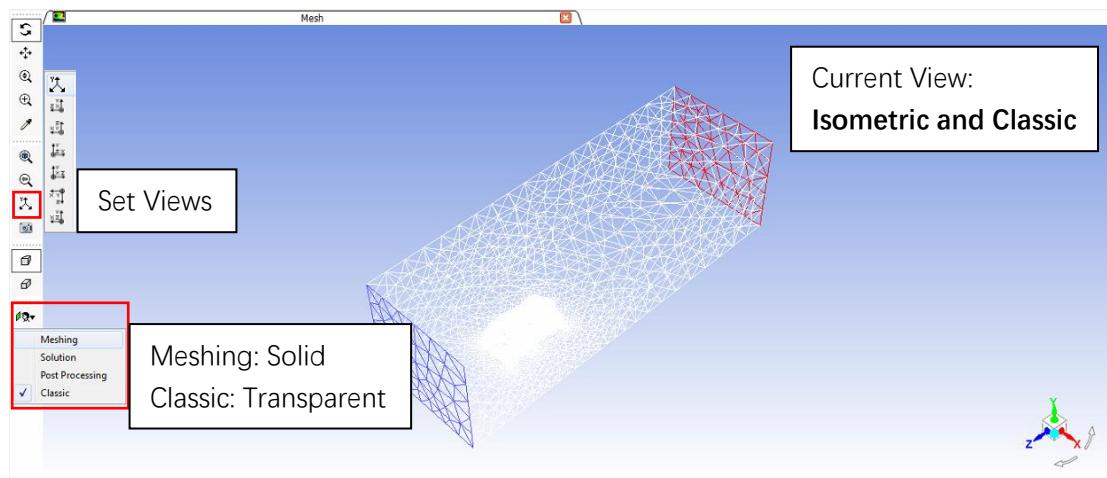
3. Set Up Fluent simulation (Steady Flow)

3.1 Fluent Launcher Settings

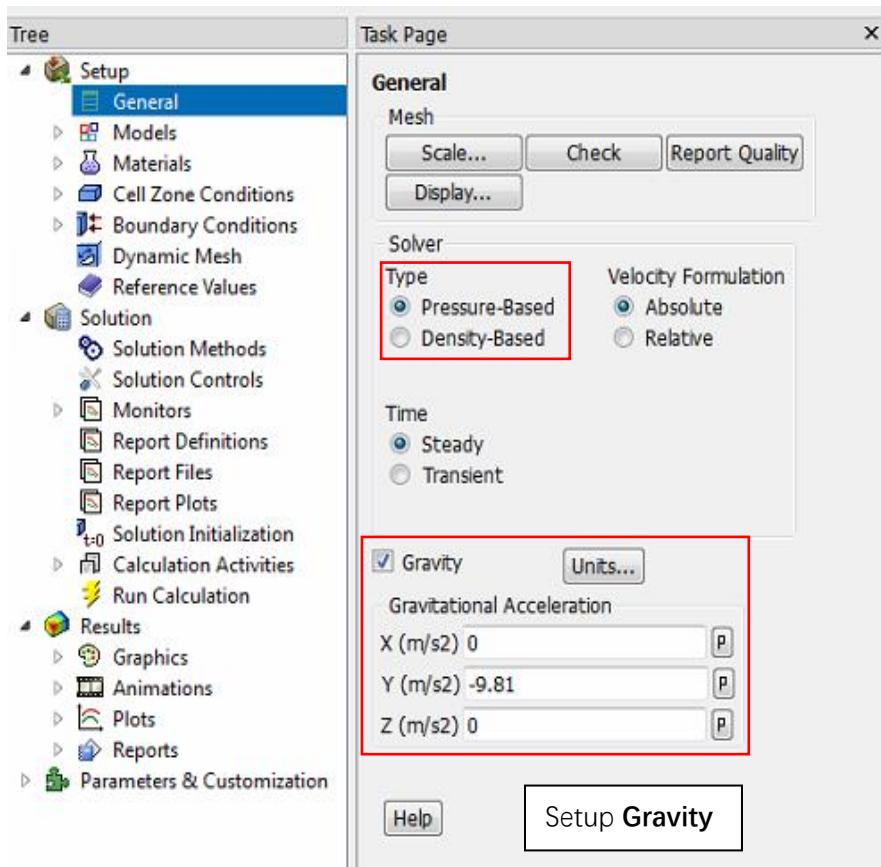
Double Click **Setup** to launch **Fluent**.



3.2 View Window



3.3 Setup → General



Follow the settings as shown above.

Use **Steady State Flow** for current Simulation.

Suggest using **Pressure-Based Solver** for this case, require less memory and more flexible in solution procedure.

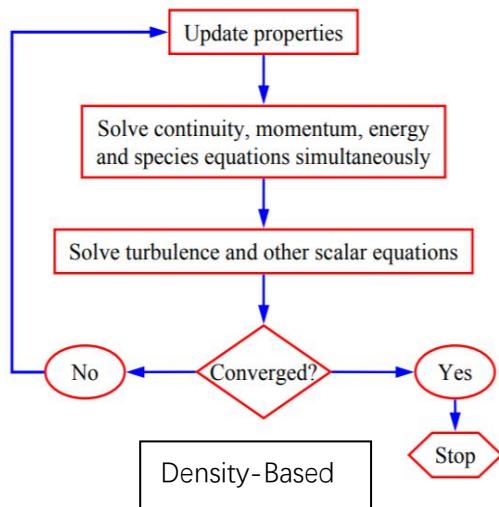
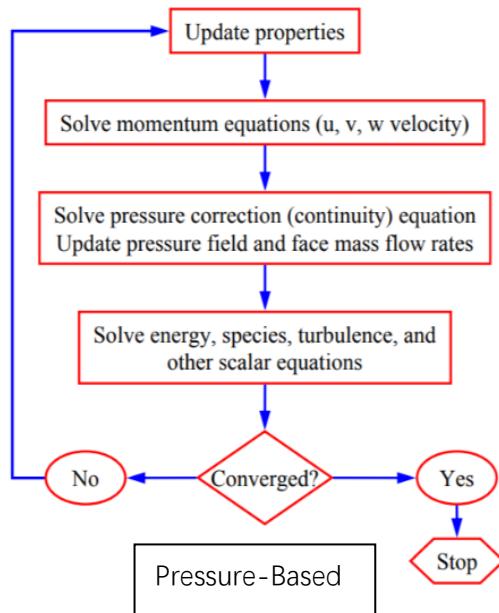
Pressure-Based Solver is applicable for most **single-phase flows**.

Not available for multiphase (Eulerian), periodic mass-flow and NITA cases.

Density-Based Solver is applicable when there is a strong coupling, or interdependence,

between density, energy, momentum, and/or species.

Such as high speed compressible flow with combustion, hypersonic flows, shock interactions



3.4 Setup → Models

The screenshot shows the ANSYS Fluent interface with the 'Task Page' window open. In the 'Models' section, 'Energy - Off' is selected. A sub-dialog titled 'Energy' is shown, containing a single checked option 'Energy Equation'. Below the Task Page is the 'Viscous Model' dialog box. It lists various turbulence models: Inviscid, Laminar, Spalart-Almaras (1 eqn), k-epsilon (2 eqn) (selected), k-omega (2 eqn), Transition k-kl-omega (3 eqn), Transition SST (4 eqn), Reynolds Stress (7 eqn), Scale-Adaptive Simulation (SAS), Detached Eddy Simulation (DES), and Large Eddy Simulation (LES). Under 'k-epsilon Model', 'Standard' is selected. In the 'Near-Wall Treatment' section, 'Enhanced Wall Treatment' is selected. Under 'Enhanced Wall Treatment Options', 'Pressure Gradient Effects' is checked. Other options like 'Viscous Heating' and 'Full Buoyancy Effects' are also listed.

Step1: For New Study:
Use default settings (**Laminar**) for Models. **Skip this step**

Step2: For Improvement of simulation result accuracy:
Click Energy → Edit → Energy Equation → OK
Click Viscous – Laminar → Edit → k-epsilon (2 eqn) & Enhanced Wall Treatment → OK

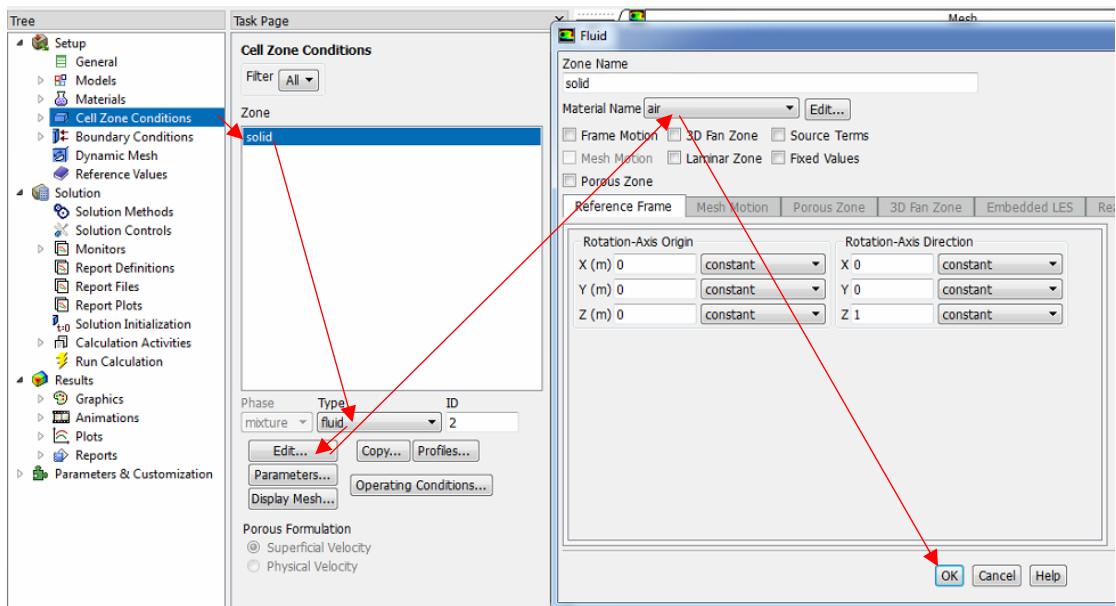
* Can Select **Pressure Gradient Effects** but may cause simulation not easy to converge

$$\text{For this case, } \text{Re} = \frac{\rho u L}{\mu} = \frac{1.225 \times 6.944 \times 11.5}{1.81 \times 10^{-5}} = 5.4 \times 10^6 > 4000, \text{ hence use turbulent flow}$$

model. Below gives the detailed explanations for the common models used in CFD.

Model	Explanation	Application
Laminar	Model solved by using Navier-Stokes equations, which gives the velocity and the pressure fields.	Flow past a Cylinder: Unsteady but laminar flow
k-epsilon (2 eqn) Kinetic energy and epsilon equations	Low Reynolds number turbulent flow model. Low Reynolds number refer to the region close to the wall where viscous effects dominate. Give accurate description of the boundary layer but require high mesh resolutions. Good convergence rate and relatively low memory requirements. Recirculating flow, and model with planar shear layers.	External flow problems around complex geometries. Airflow around a bluff body
k-omega (2 eqn) Kinetic energy and the specific rate of dissipation of kinetic energy equations	Low Reynolds number model, similar to the k- ε model but more nonlinear, more difficult to converge than the k- ε model, sensitive to the initial guess of the solution. Commonly used when k- ε model is not accurate enough.	Internal flows, flows that exhibit strong curvature, separated flows, and jets. Flow through a pipe bend
Transit SST(4 eqn) k-ε model (free stream) and k-ω model near the walls	Low Reynolds number model. Similar mesh resolution requirements to the k- ω model. More difficult to converge than the k- ε model unless a good initial guess is provided, Perform better for flows with adverse pressure gradients than k- ε model.	Flow over airfoil
Reynolds stress (7 eqn)	High degrees of anisotropy, significant streamline curvature, flow separation, flows with zones of re-circulation or flows influenced by mean rotational effects.	Swirl component Buoyant flow Secondary flow More complex flow

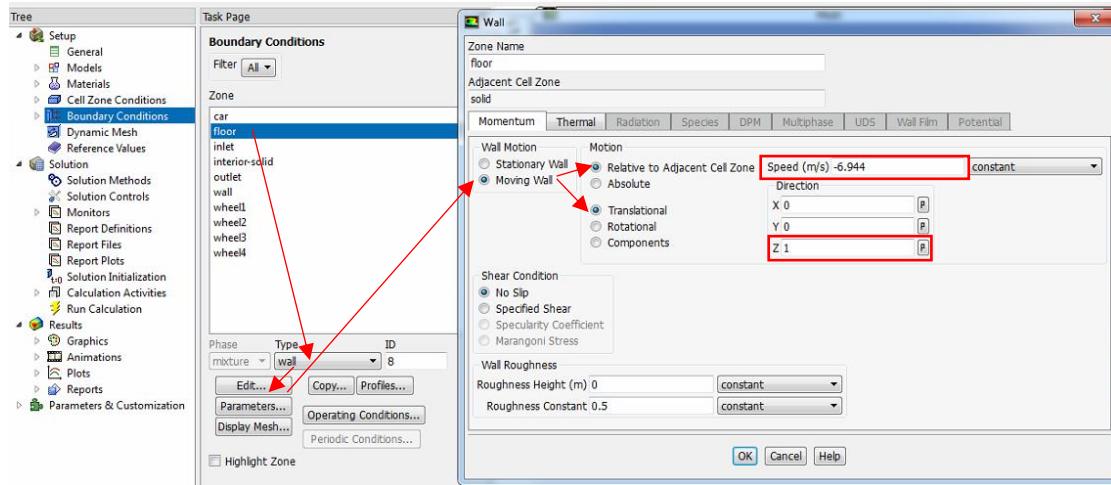
3.5 Setup → Cell Zone Conditions



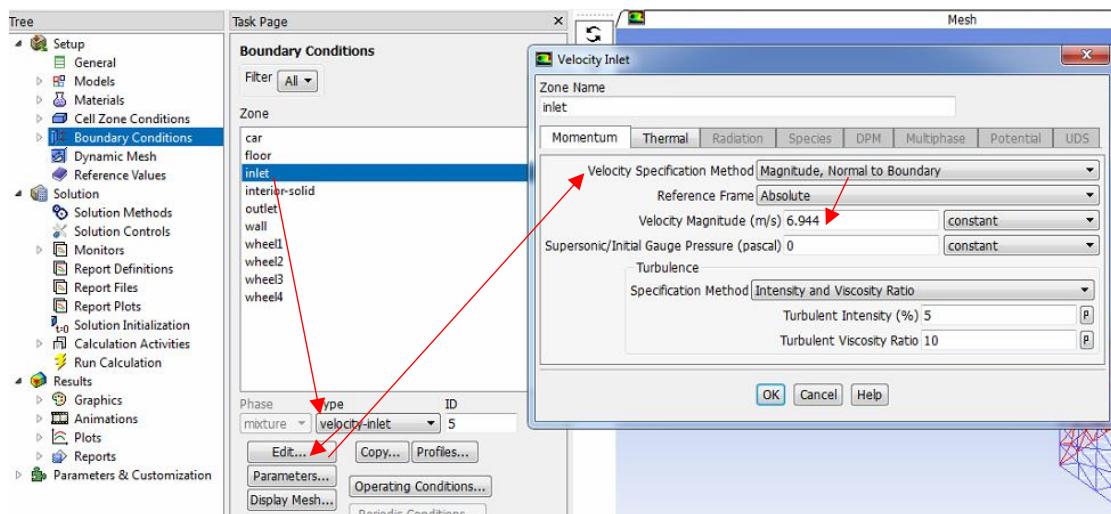
Click solid → Type: fluid → Edit → Choose air → OK

3.6 Setup → Boundary Conditions

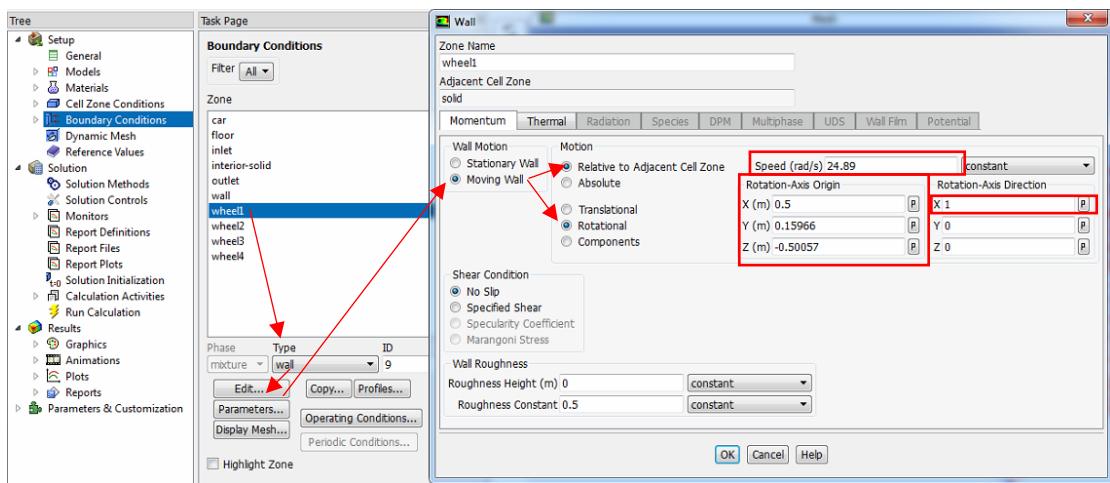
- Click **car** → Type: wall (Stationary Wall)
- Click **floor** (Setup Moving Condition) → Type: wall → Edit → Moving Wall → Relative to Adjacent Cell Zone & Translational → Speed: -6.944 → Direction: Z=1 → OK



- Click **inlet** (Setup flow speed) → Type: velocity-inlet → Edit → Magnitude, Normal to Boundary → Velocity Magnitude: 6.944 → OK



- Click **interior-solid** → Type: **interior**
- Click **outlet** → Type: **Pressure-outlet**
- Click **wall** (3 Side Walls) → Type: **Pressure-outlet**
- Click **wheel1** (Setup Rotating Condition) → Type: **wall** → Edit → Moving Wall → Motion: Relative to Adjacent Cell Zone & Rotational → Speed: 24.89 → Rotation-Axis Origin: Coordinate of the wheel center → Rotation-Axis Direction: X=1 (Right Hand Rule) → OK



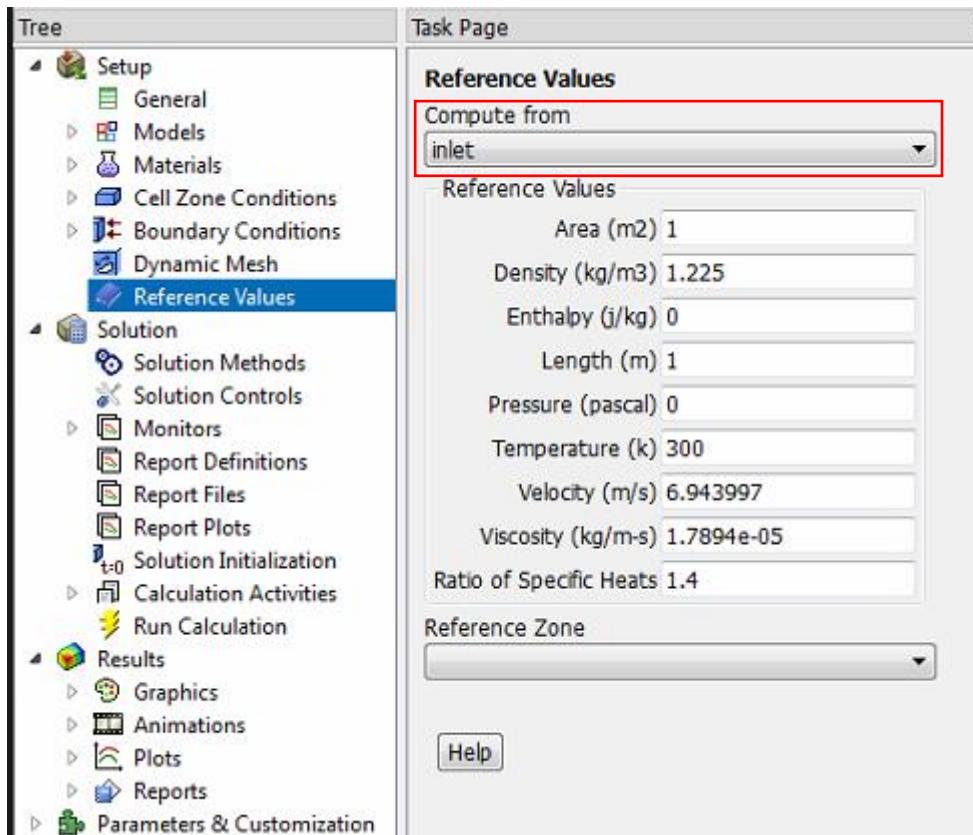
For rest of the **wheels**, keep same settings with **wheel1** except the **Rotation-Axis Origin** needs to change accordingly.

Relative to Adjacent Cell Zone	Absolute
Specify velocities relative to the frame motion	Specify velocities in absolute frame
E.g. A velocity of zero means that the wall is stationary in the relative frame, and therefore moving at the speed of the adjacent cell zone in the absolute frame.	E.g. A velocity of zero means that the wall is stationary in the absolute frame, and therefore moving at the speed of the adjacent cell zone--but in the opposite direction--in the relative reference frame

*If the adjacent cell zone is not moving, the absolute and relative options are equivalent.

<https://www.sharcnet.ca/Software/Fluent6/html/ug/node253.htm#sec-wall-inputs> for more information

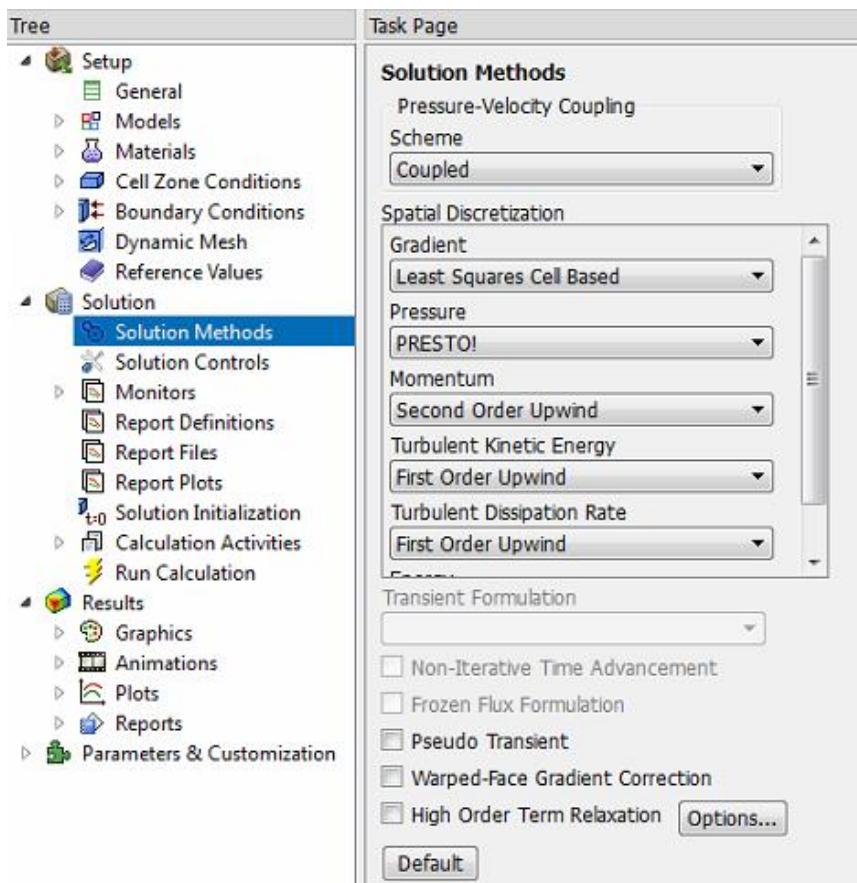
3.7 Setup → Reference Values



Select **inlet** from **Compute from** drop-down list

Keep **Reference Zone** empty

3.8 Solution → Solution Methods

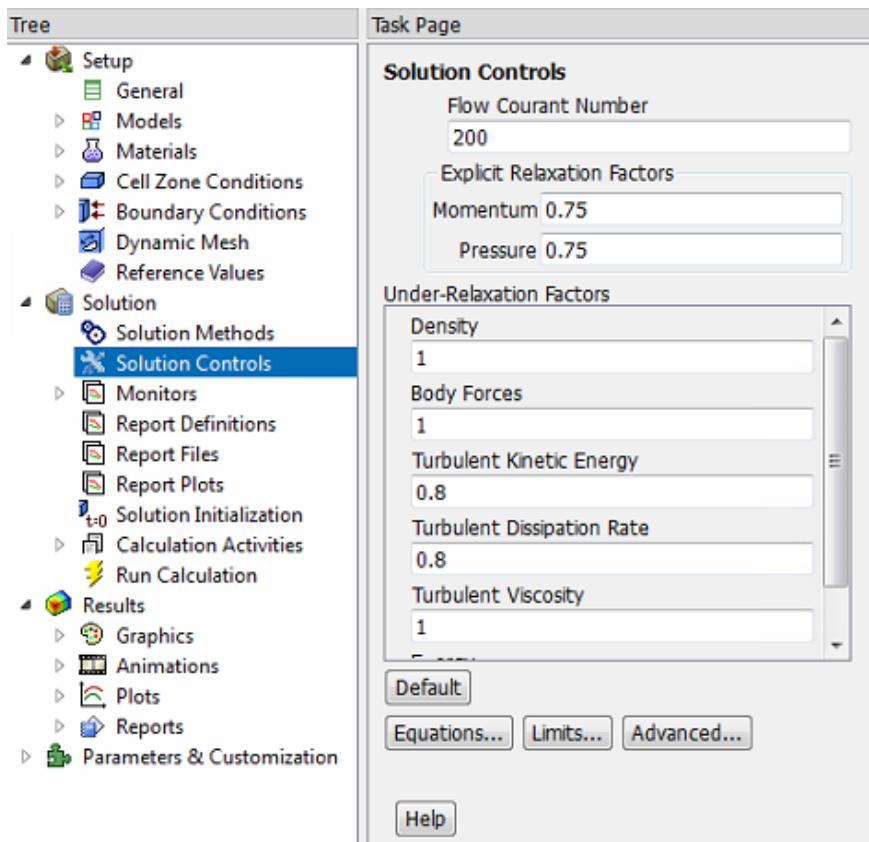


- Select **Coupled** from the **Scheme** drop-down list. A combination of continuity and momentum equations to derive an equation for pressure (or pressure correction).
- Select **Least Squares Cell Based** from the **Gradient** drop-down list.
- Select **PRESTO!** from the **Pressure** drop-down list. Suitable for highly swirling flows, flows involving steep pressure gradients (porous media, fan model, etc.), or in strongly curved domains
- Select **Second Order Upwind** from the **Momentum** drop-down list. Second Order Accuracy.
- Select **First Order Upwind** from the **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** drop-down lists. First Order Accuracy.
- Select **Second Order Upwind** from the **Energy** drop-down list.

<http://www.engr.uconn.edu/~barbertj/CFD%20Training/Fluent/4%20Solver%20Settings.pdf>

for more information

3.9 Solution → Solution Controls



Flow Courant Number = 200 (A fluid particle moves through 200 cells at each time step; higher → difficult to converge)

Explicit Relaxation Factors:

Momentum = 0.75

Pressure = 0.75

Under-Relaxation Factors:

Density = 1

Body Forces = 1

Turbulent Kinetic Energy = 0.8

Turbulent Dissipation Rate = 0.8

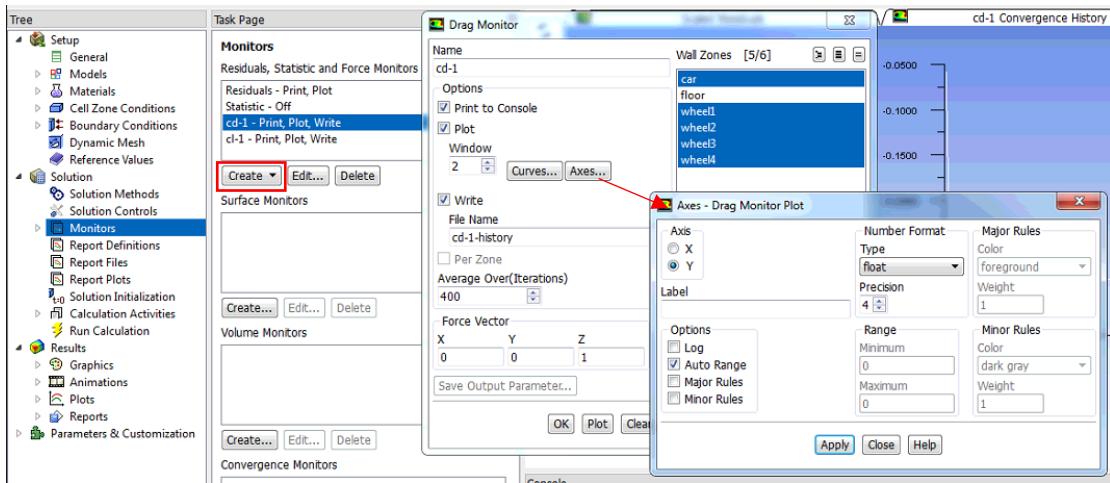
Turbulent Viscosity = 1

Energy = 1

***Relaxation Factors:** Under and over relaxation factors control the stability and convergence rate of the iterative process. The under-relaxation factor increases the stability while over relaxation increases the rate of convergence.

3.10 Solution → Monitors

Drag coefficient monitor

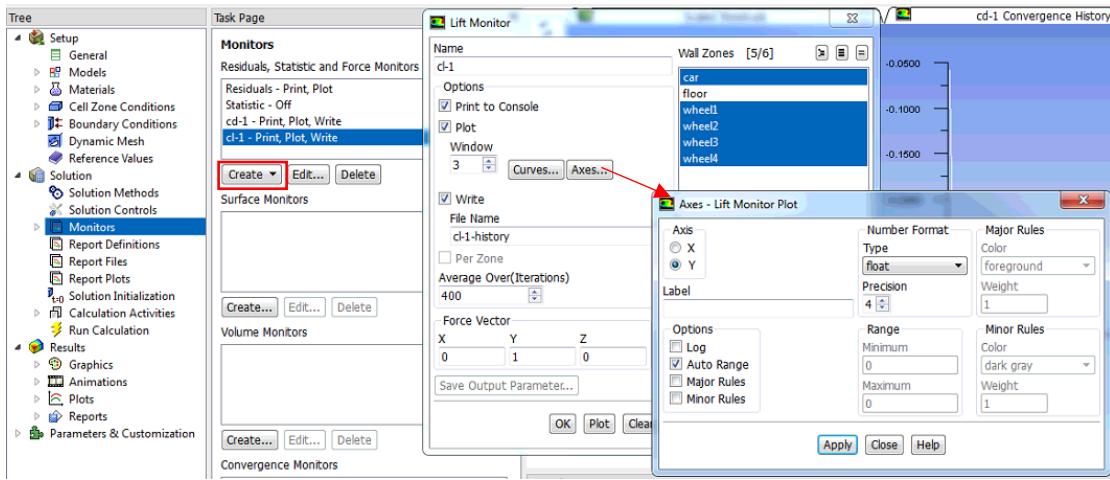


Select **Drag** from the **Create** drop-down list.

Click **Axes** to setup Drag Monitor Plot. Click **Apply** once finished.

Setup the **Drag Monitor** as shown above. Click **OK** once finished.

Lift coefficient monitor



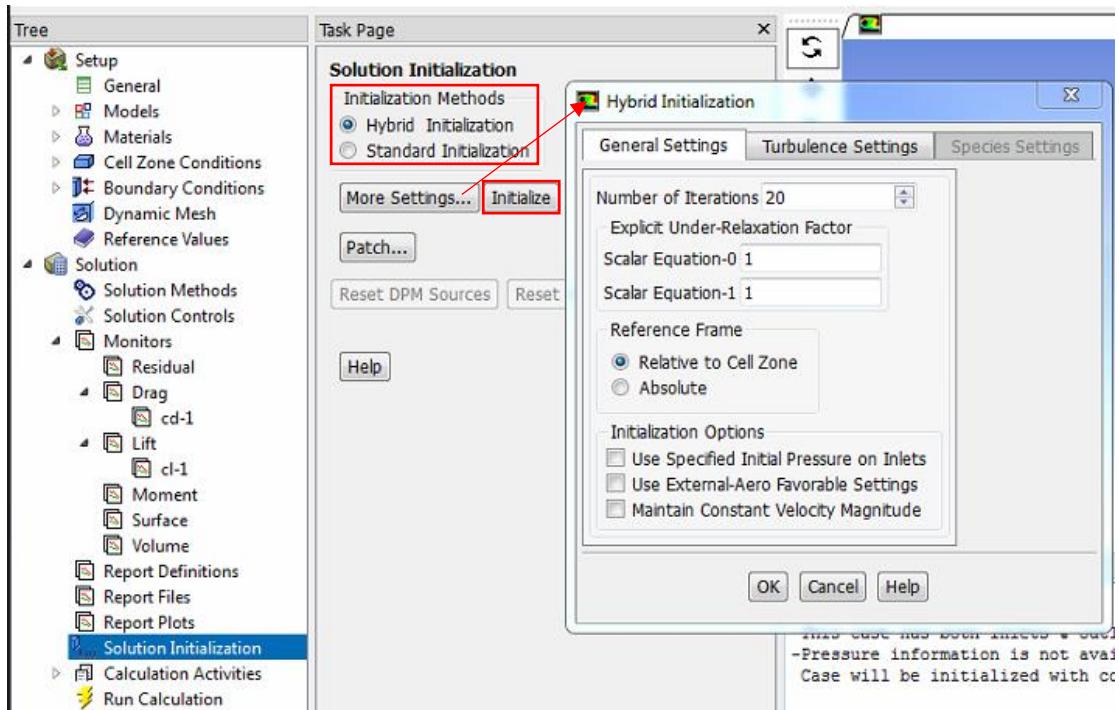
Select **Lift** from the **Create** drop-down list.

Click **Axes** to setup Lift Monitor Plot. Click **Apply** once finished.

Setup the **Lift Monitor** as shown above. Click **OK** once finished.

***Average Over (Iterations):** Set this number according to the Iteration number set in **Run Calculation** (See 3.12) for steady flow simulation.

3.11 Solution → Solution Initialization



Select **Hybrid Initialization** for **Solution Initialization**

Click **More Settings** → Set the **Number of Iterations: 20** → **OK**

Click **Initialize**

```
Console
Checking case topology...
-This case has both inlets & outlets
-Pressure information is not available at the boundaries.
Case will be initialized with constant pressure

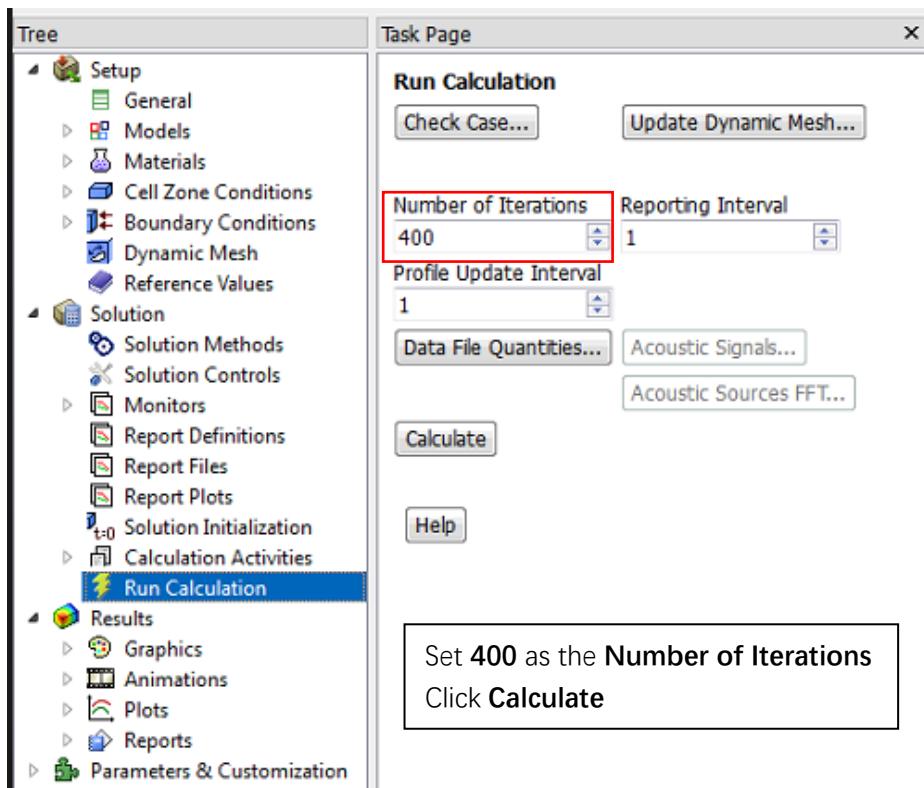
iter      scalar-0
1        1.000000e+00
2        5.872517e-03
3        1.153929e-03
4        4.227006e-04
5        1.253546e-04
6        4.722024e-05
7        1.690884e-05
8        6.348780e-06
9        2.501951e-06
10       1.064571e-06
11       5.655855e-07
12       3.725448e-07
13       3.466597e-07
14       3.124156e-07
15       3.497483e-07
16       3.589446e-07
17       4.147058e-07
18       4.318450e-07
19       4.891939e-07
20       5.279248e-07
hybrid initialization is done.
```

Once the initialization finished without warning means the simulation able to converge.

Warning: convergence tolerance of 1.000000e-06 not reached during Hybrid Initialization.

If there is a warning about convergence tolerance, need to check the simulation settings, mesh quality and try to simplify the geometry before **Run Calculation**.

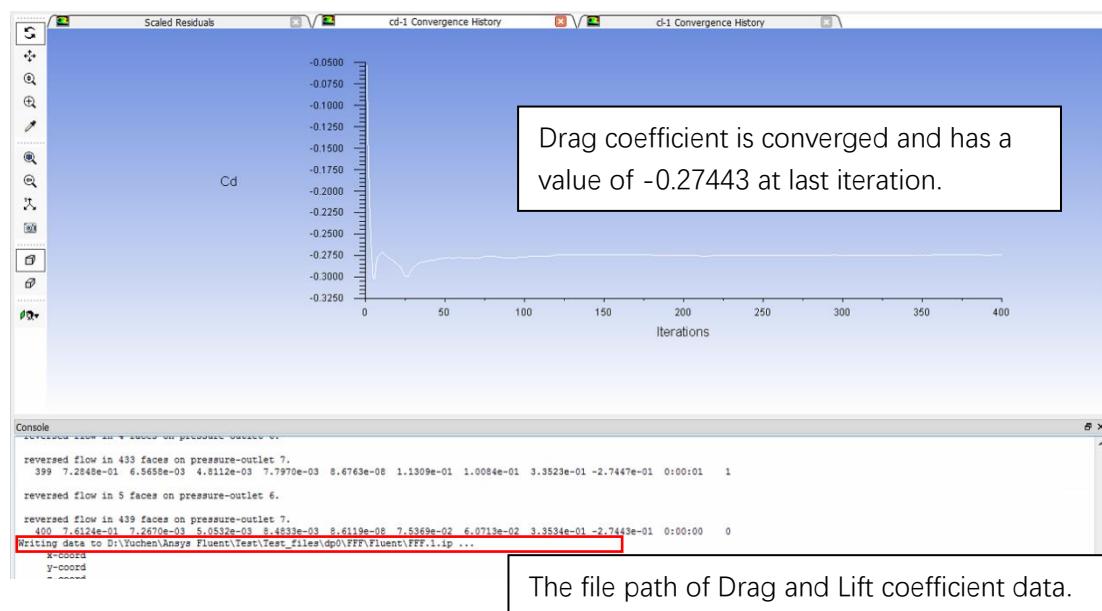
3.12 Solution → Run Calculation



*For Steady State Flow, it should be converged within 50 to 100 loops.

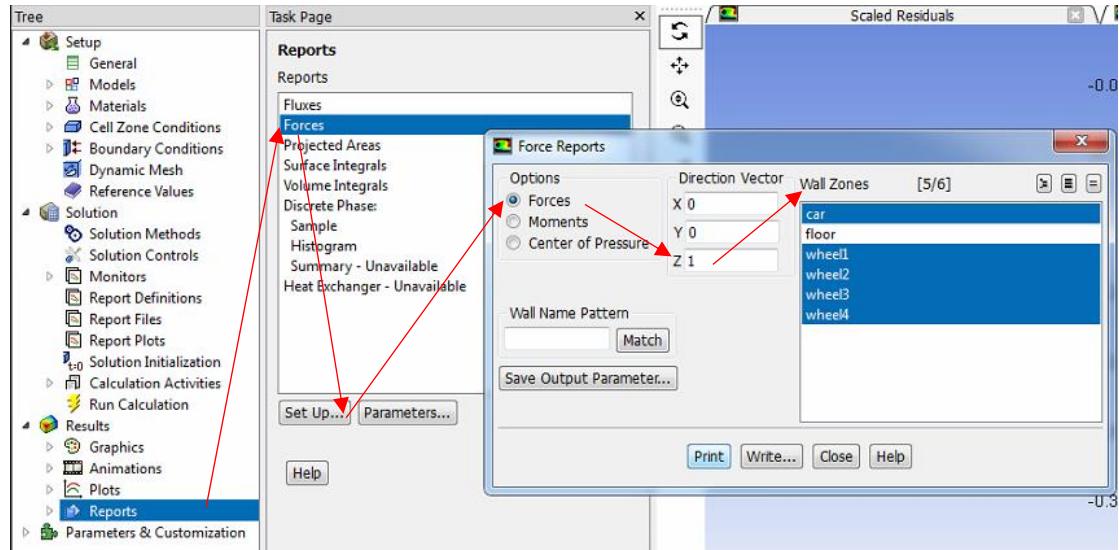
For this case, the drag coefficient and drag force tend to be stable after 200 iterations.

Hence, it is better to set an iteration number larger than 200.



3.13 Results → Reports

Calculate Drag Force



Results → Reports → Forces → Set up → Options: Forces → Direction Vector: Z=1 →

Wall Zones: Bodyshell with 4 Wheels → Print

Zone	Forces (n)			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
wheel1	-0.02508229	-0.031274609	-0.056356899	-0.00084926291	-0.0010589291	-0.001908192
wheel2	-0.21402603	-0.024736816	-0.23876285	-0.0072467216	-0.00083756548	-0.008084287
wheel3	-0.3009074	-0.002007443	-0.30291485	-0.010188444	-6.7970144e-05	-0.010256414
wheel4	-0.22899058	0.00020188733	-0.2287887	-0.0077534073	6.8357165e-06	-0.0077465715
car	-6.12988	-0.65619308	-6.786073	-0.20755201	-0.022218085	-0.22977009
Net	-6.8988863	-0.71401006	-7.6128963	-0.23358984	-0.024175714	-0.25776556

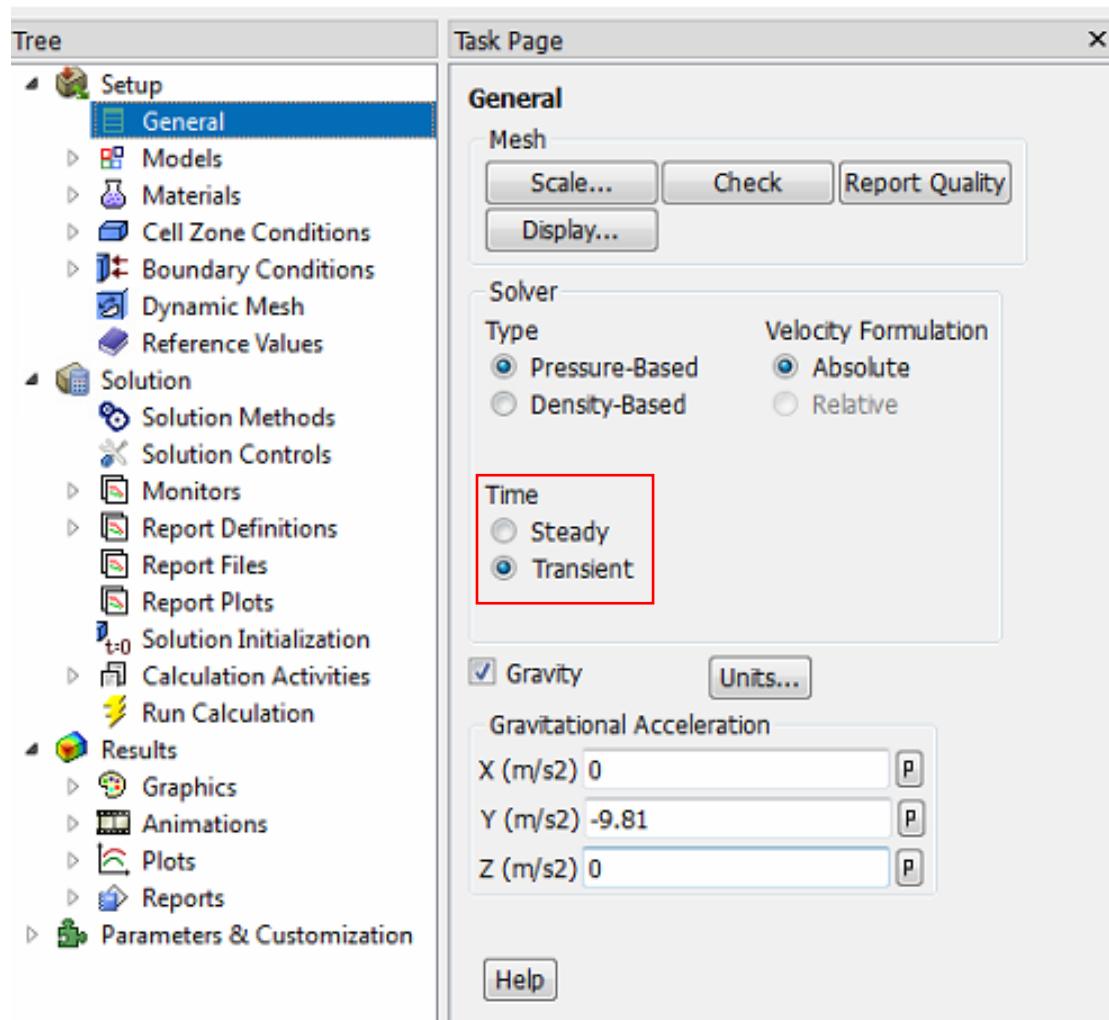
General Results shown as above.

Close **Fluent** window once the simulation finished.

Go back to **Workbench**.

4. Set Up Fluent simulation (Transient Flow)

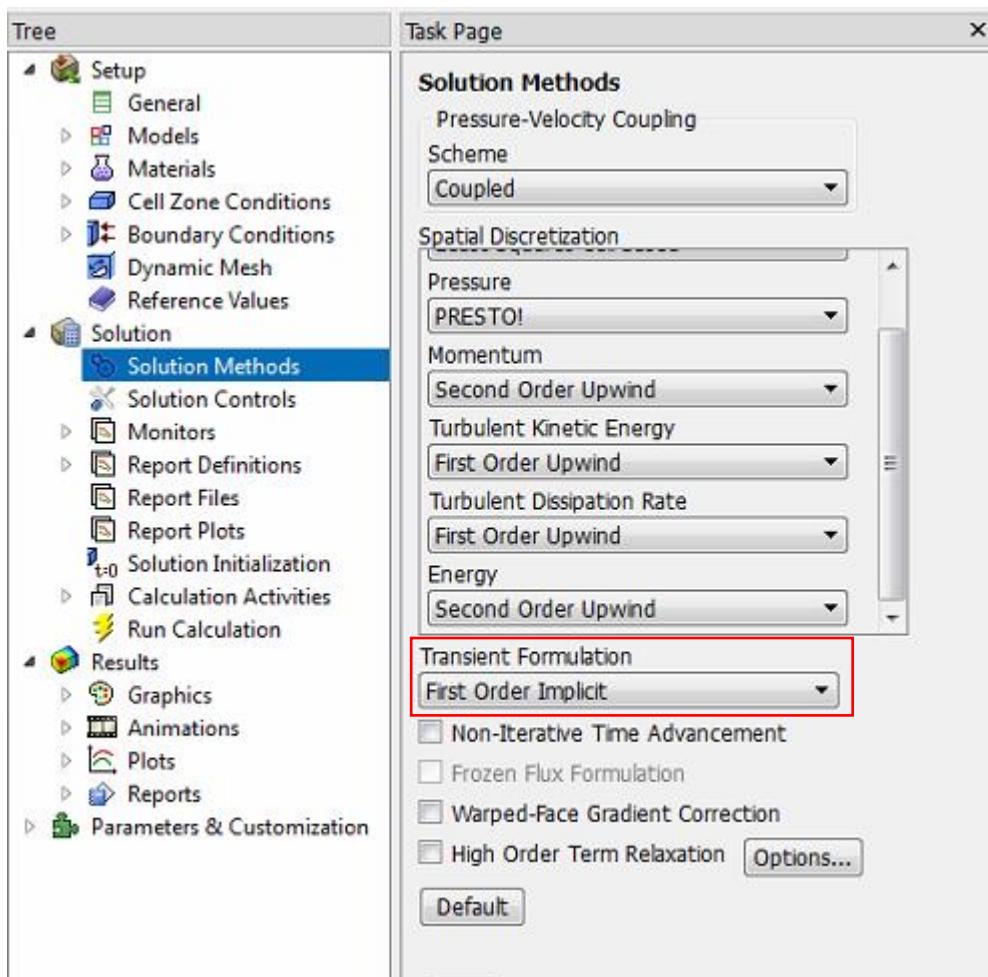
4.1 Setup → General



Follow the settings as shown above.

Use **Transient Flow** for current Simulation.

4.2 Solution → Solution Methods



Select **Coupled** from the **Scheme** drop-down list.

Select **Least Squares Cell Based** from the **Gradient** drop-down list.

Select **PRESTO!** from the **Pressure** drop-down list.

Select **Second Order Upwind** from the **Momentum** drop-down list.

Select **First Order Upwind** from the **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate** drop-down lists.

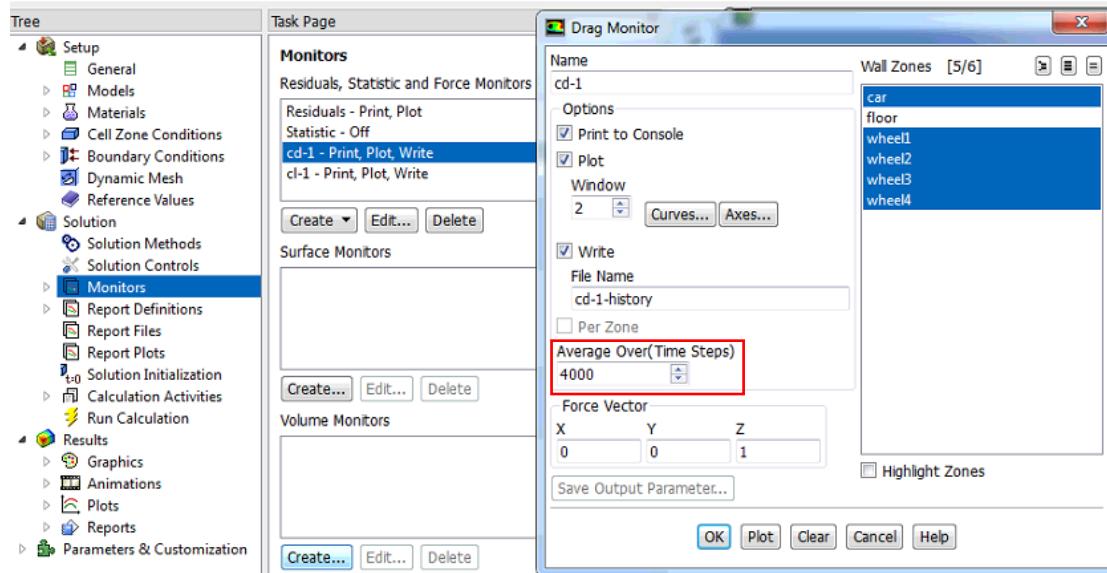
Select **Second Order Upwind** from the **Energy** drop-down list.

(Above settings can keep as same as **Steady State Flow**)

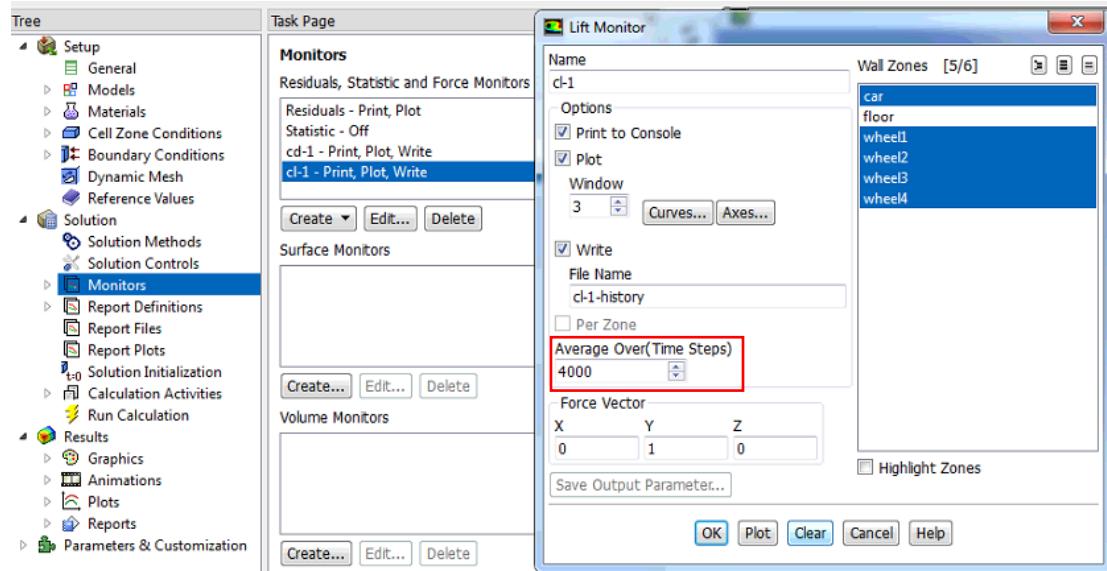
Select **First Order Implicit** from the **Transient Formulation** drop-down list (First Order Accuracy, Easy to converge)

4.3 Solution → Monitors

Drag Coefficient Monitor



Lift Coefficient Monitor



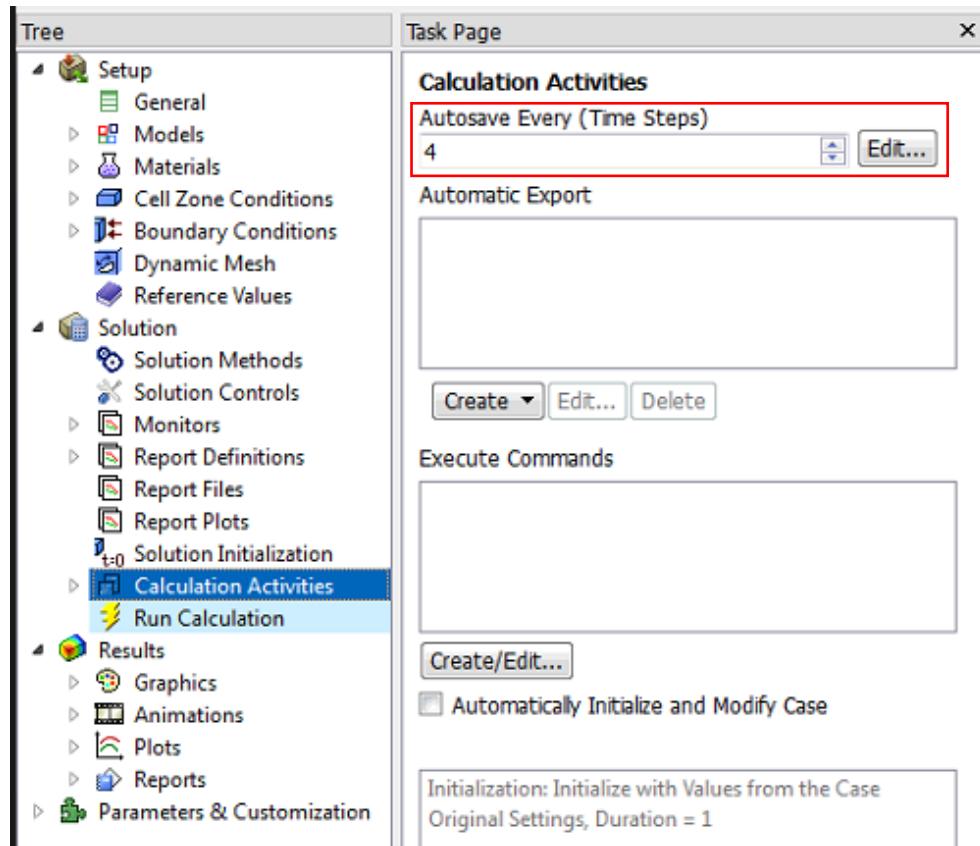
***Average Over (Time Steps):** Set this number according to the Time Steps number set in

Run Calculation (See 4.5) for transient flow simulation.

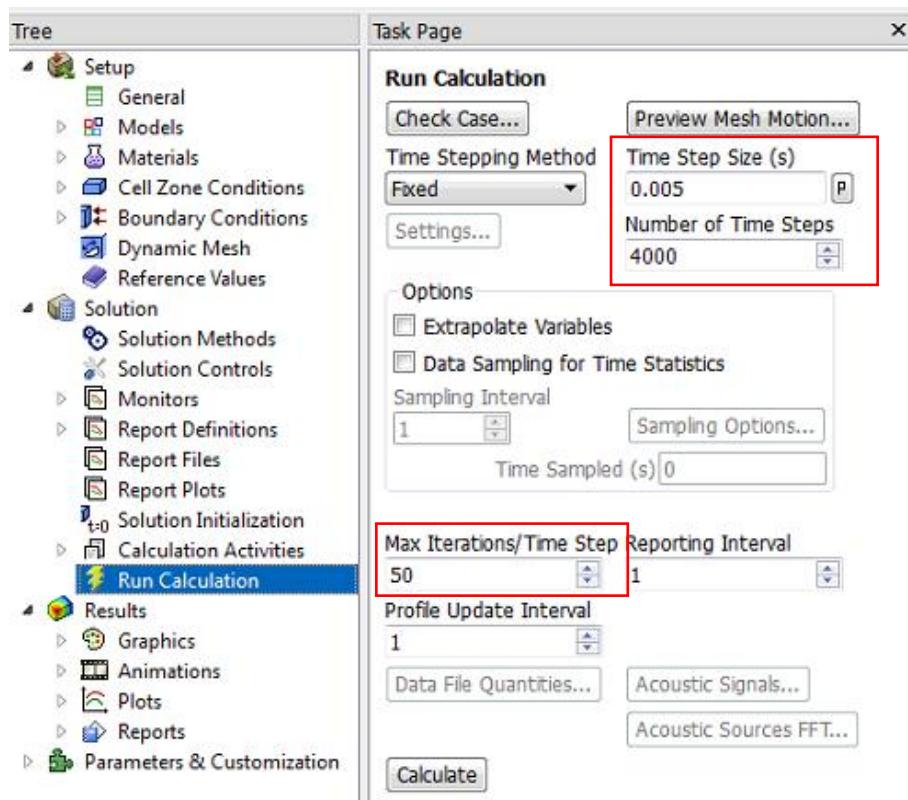
4.4 Solution → Calculation Activities

To generate **animation** of the flow, set **Autosave** for Time Steps in Calculation Activities.

For large Time Steps (like 4000), set larger number for **Autosave** (every 4 time steps).



4.5 Solution → Run Calculation



Follow the settings as shown above for this case.

$$\text{Time Step Size (s)}: \Delta t = \frac{\text{Typical Cell Size}}{\text{Characteristic Flow Velocity}} ; \text{ Typical Cell Size} = \frac{\text{Volume of Geometry}}{\# \text{ of Mesh Element}}$$

If the mesh is relatively uniform, better use the minimum cell size for typical cell size.

To check Mesh size: Click Info → Size and Check (Details show in Console window)

The screenshot shows the Mesh menu with the 'Info' and 'Check' buttons highlighted with red boxes. The 'Console' window below displays mesh statistics:

Level	Cells	Faces	Nodes	Partitions
0	542503	1105685	101099	6

1 cell zone, 10 face zones.
Domain Extents:
x-coordinate: min (m) = -1.85000e+00, max (m) = 1.85000e+00
y-coordinate: min (m) = -1.10000e-01, max (m) = 3.127518e+00
z-coordinate: min (m) = -9.20000e+00, max (m) = 2.30000e+00
Volume statistics:
minimum volume (m3): 1.108174e-08
maximum volume (m3): 7.686543e-02
total volume (m3): 1.364134e+02
Face area statistics:
minimum face area (m2): 6.083771e-06
maximum face area (m2): 3.923733e-01
Checking mesh.....
Done.

From Console window:

$$\text{Time Step Size} = \frac{\sqrt[3]{\text{Minimum Volume}}}{\text{Inlet Flow Velocity}}$$

$$= \frac{\sqrt[3]{1.108 \times 10^{-8}}}{6.944} = 3.21 \times 10^{-4} \text{s}$$

Or $\text{Time Step Size} = \frac{\sqrt[3]{\text{Average Volume}}}{\text{Inlet Flow Velocity}}$

$$= \frac{\sqrt[3]{1.36 \times 10^2 / 542503}}{6.944} = 9.09 \times 10^{-3} \text{s}$$

To save time, set the time step size for the transient flow simulation to be **0.005s** (between two Time Step Size values calculated from above steps).

Number of Time Steps: Specify the desired number of time steps.

For this case, the length of the computational domain is 11.5m and the inlet velocity of the air flow is 6.944m/s. Hence, the **free flow** needs 1.66s to pass the whole computational domain. For the transient flow simulation, set **20s** as the end time hence set the number of time steps to be **4000**.

Max Iterations/Time Step: This parameter sets a maximum for the number of iterations per time step. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step. To ensure the simulation converged within each time step, set Max Iterations/Time Step to be **50**.

Rest of the settings not mentioned in transient flow simulation keep as same as **steady state flow**.

Click **Calculate**.

4.6 Results Reports

Calculate Drag Force

Forces - Direction Vector (0 0 1)				Coefficients		
Zone	Forces (n)	Viscous	Total	Pressure	Viscous	Total
wheel1	0.0904333	-0.030001603	0.060431696	0.003061987	-0.0010158263	0.0020461608
wheel2	-0.2178342	-0.028048821	-0.24588303	-0.007375626	-0.00094970687	-0.0083253695
wheel3	-0.306759	-0.00076510711	-0.30752411	-0.010386573	-2.5905812e-05	-0.010412479
wheel4	-0.22763354	0.0010958122	-0.22653772	-0.0077074589	3.7103177e-05	-0.0076703557
car	-8.8615894	-0.77379525	-9.6353847	-0.30004514	-0.026199984	-0.32624512
Net	-9.5233829	-0.83151497	-10.354898	-0.32245285	-0.02815432	-0.35060717

Using **Transient Flow** model, the simulation has higher overall results (End time: 20s) for drag force and drag coefficients than steady state flow.

4.7 Steady State Flow VS. Transient Flow

Steady State Flow	Transient Flow
Ignore many of the cross terms and higher order terms dealing with time (such as transient non-linearities)	More accuracy from transient solution because it is time marching solution
"Local timescale factor" is used as it can accelerate convergence nicely but as different timescales are used across the domain can cause accuracy problems Local timescale factor: Too high → Diverge (Floating point exception) Too low → Converge slow	Require real time information to determine the time intervals Used for: Initially changing boundary condition Steady-state condition is never reached (such as Buoyancy)
*Fully converged simulation run to steady state by either steady state or transient approached should be the same.	

https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/cfx_mod/i1298451.html for more information

Close **Fluent** window once the simulation finished.

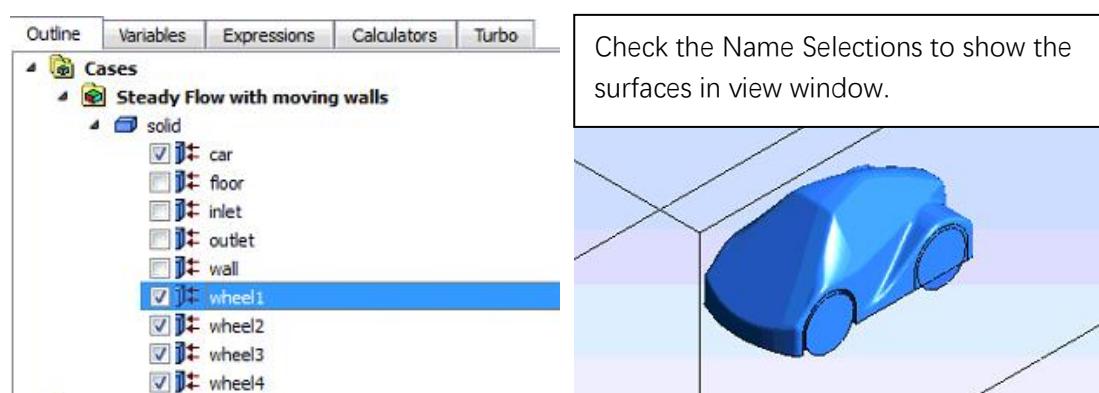
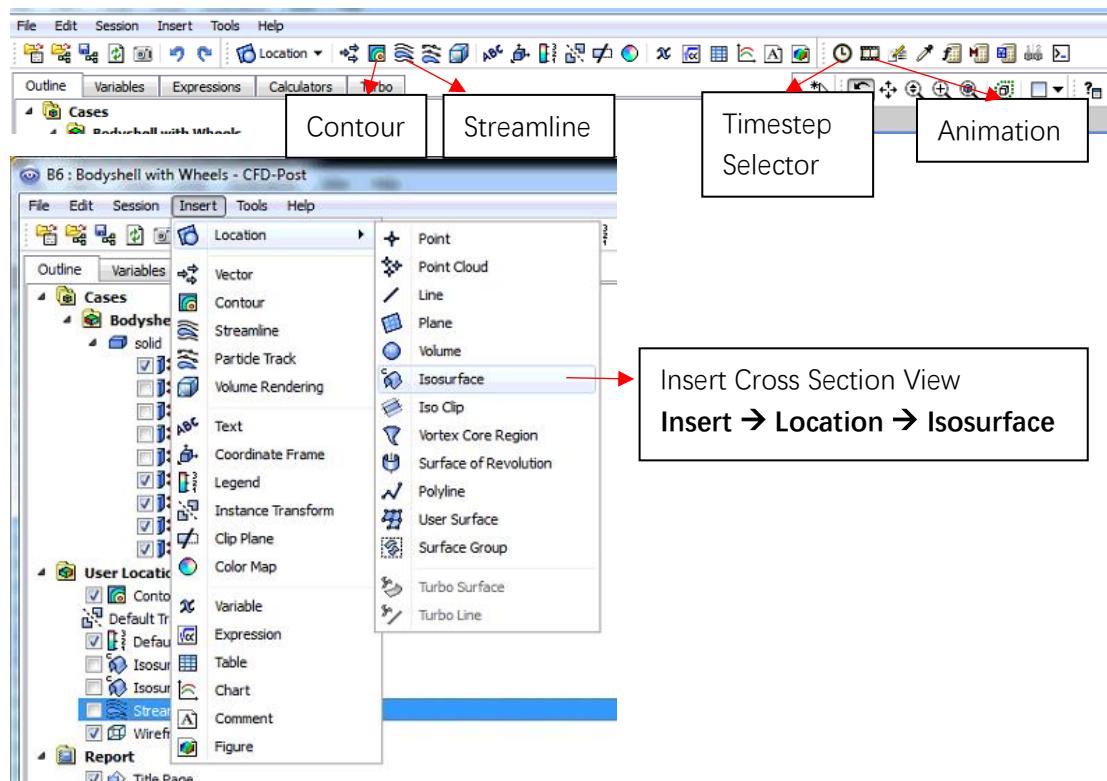
Go back to **Workbench**.

5. Post processing for Fluent simulation

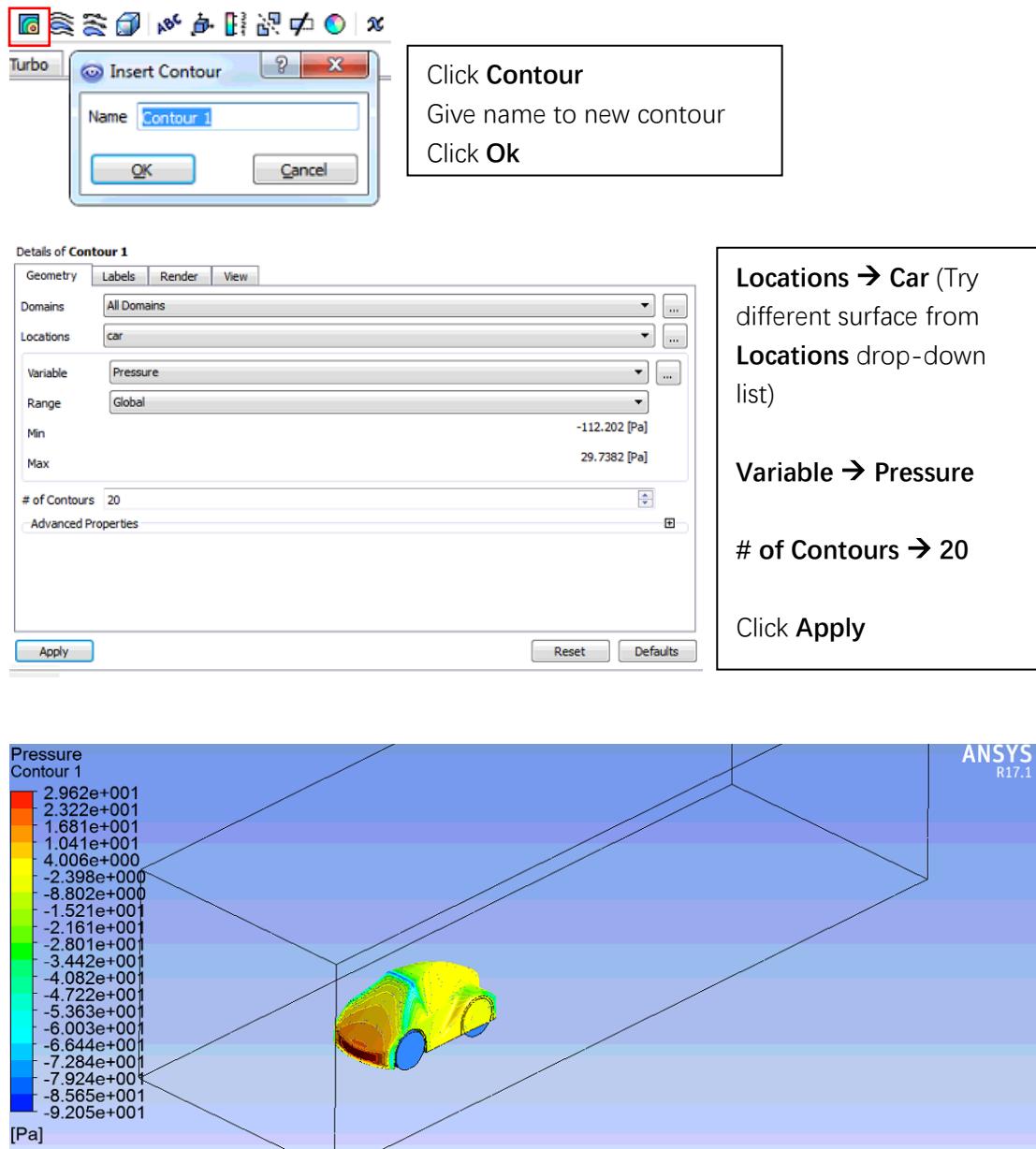
Right Click on **Results** to open the drop-down list, Left Click **Update**.

Green tick will show once finish updating results.

Double click on **Results** to launch **CFD Post** window.

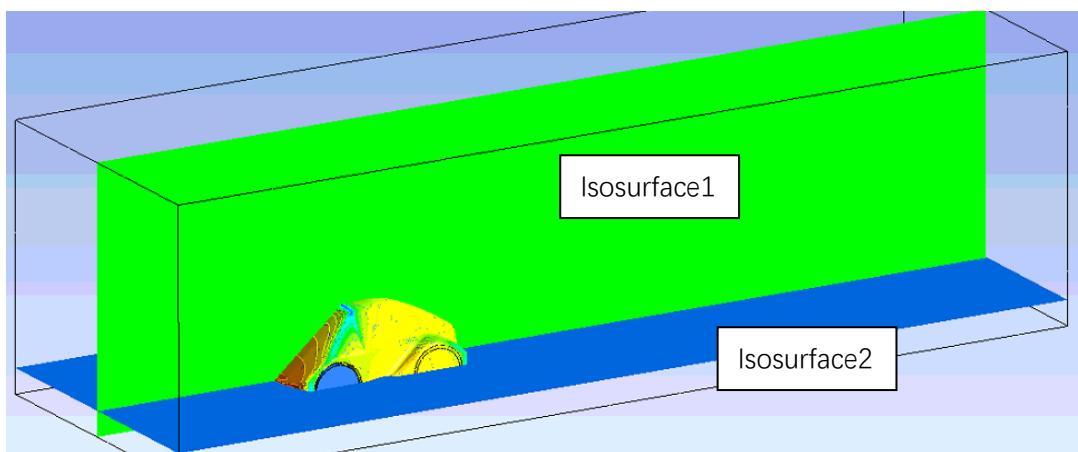
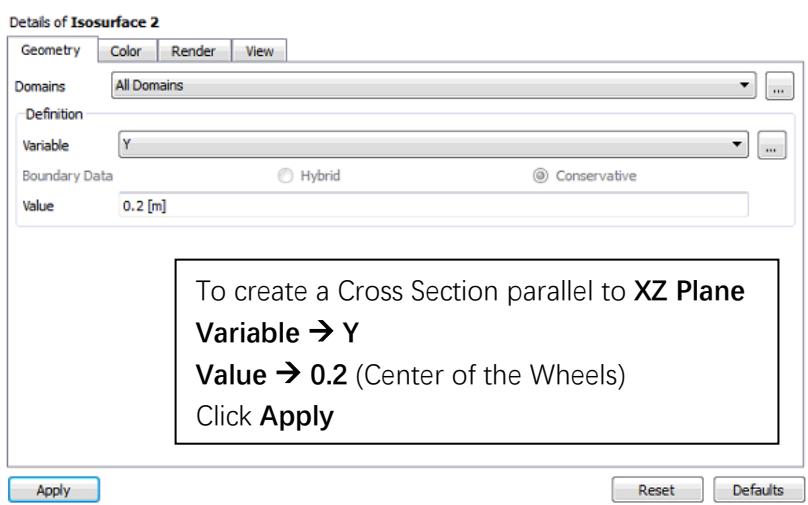
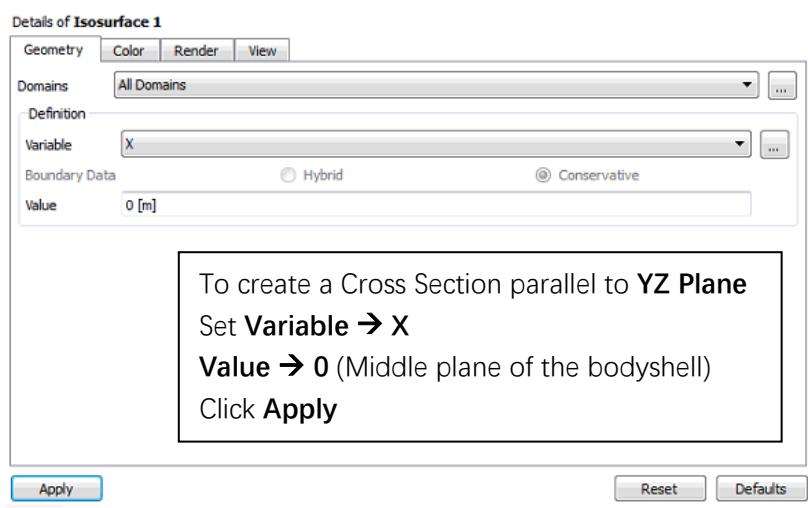


5.1 Insert **Contour** to show **Pressure** on the Bodyshell



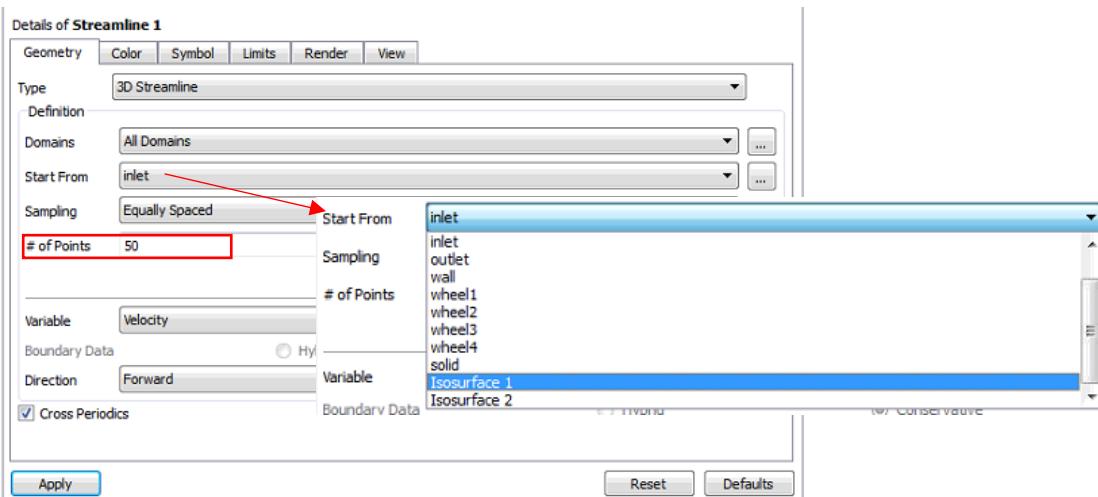
Above shown the pressure contour on the bodyshell from the steady flow simulation.

5.2 Insert Isosurface to show the Cross Section of the bodyshell



Above shown the two Isosurfaces.

5.3 Insert Streamline to check fluid movement around the bodyshell

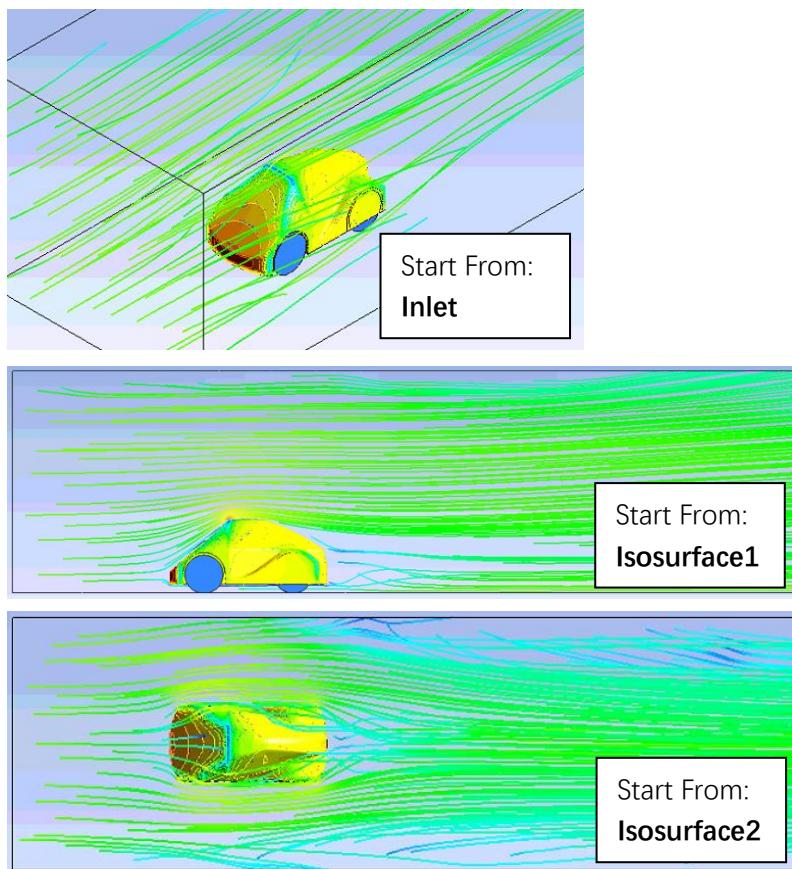


Select **inlet** (or **Isosurface**) from the **Start From** drop-down list.

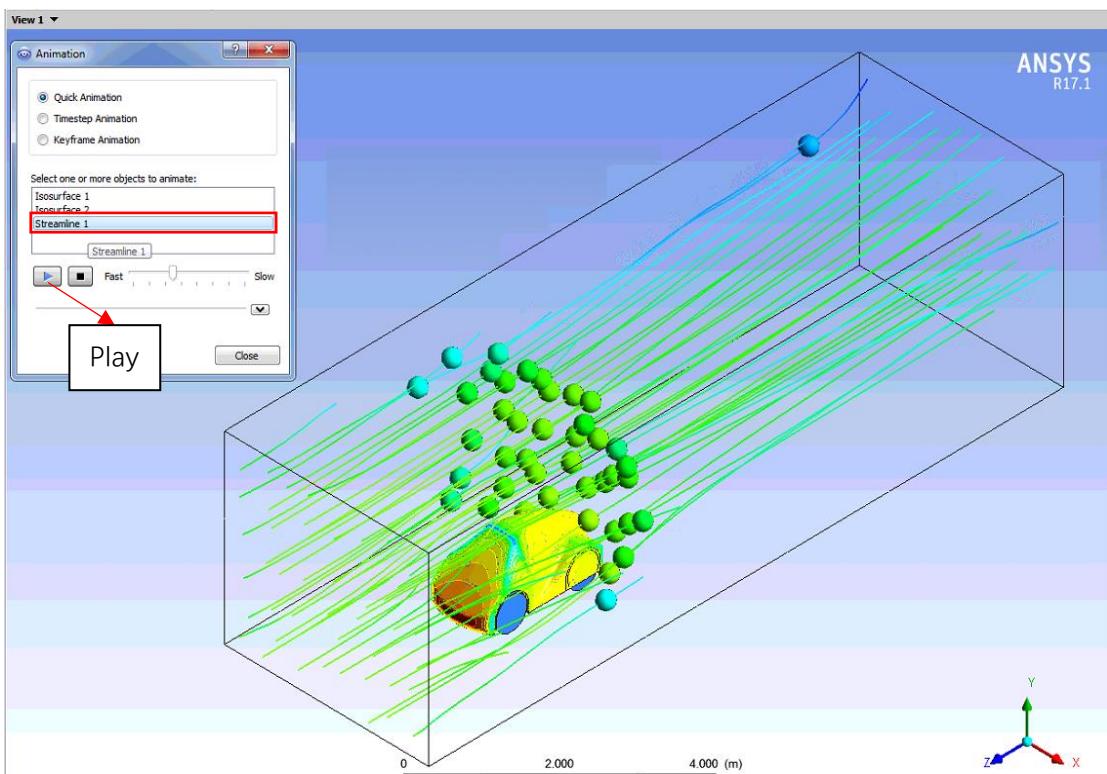
Set **# of Points** accordingly.

Set **Variable** to be **Velocity**.

Click **Apply**.



5.4 Insert Animation



Select **Streamline 1**

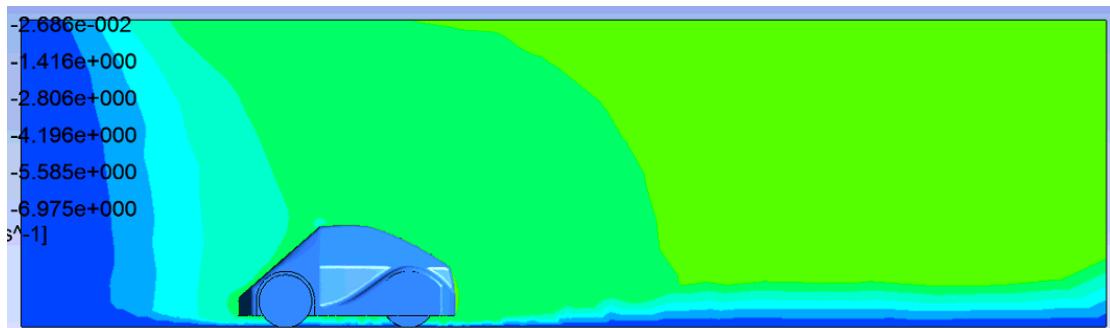
Adjust the Animation Speed accordingly

Click **Play**

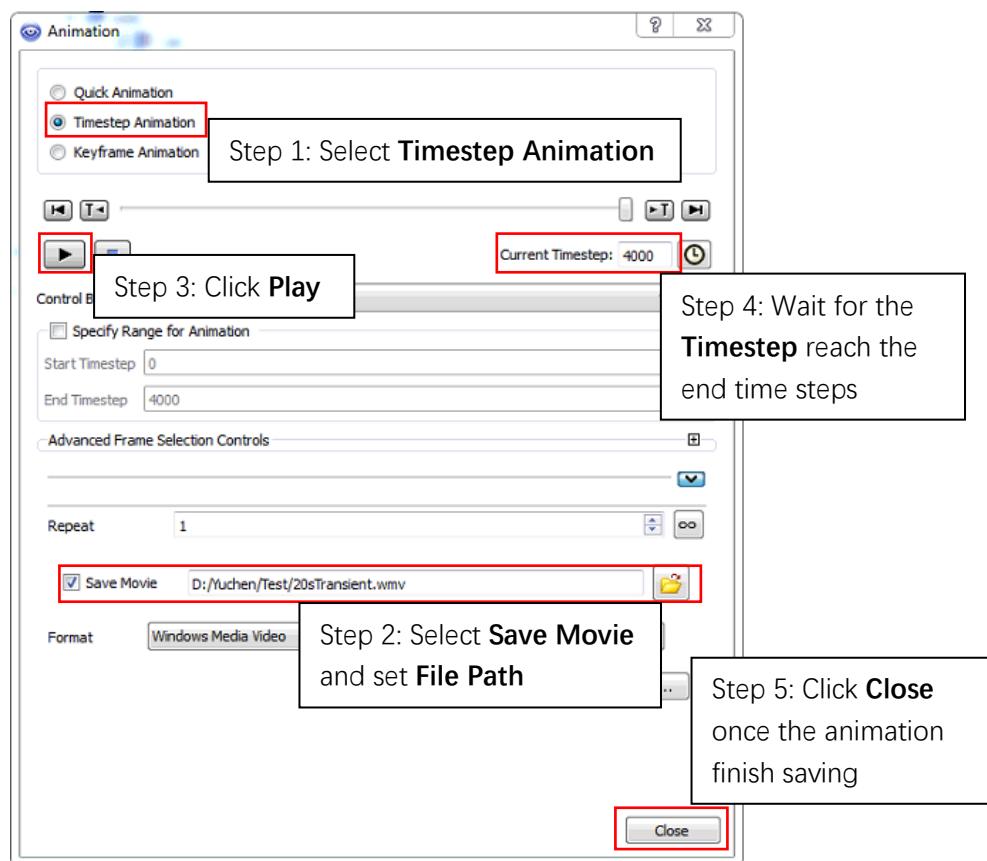
Press **Esc** to exit from **Animation**

5.5 Insert Animation for Transient Flow

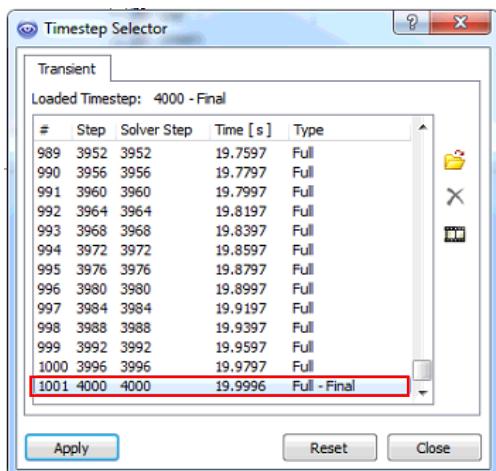
To generate the animation of the velocity changed through timestep on cross-section plane, need to set up the contour of the velocity (w: direction of the flow) on isosurface (right plane of the geometry) as shown below.



Click **Animation**.



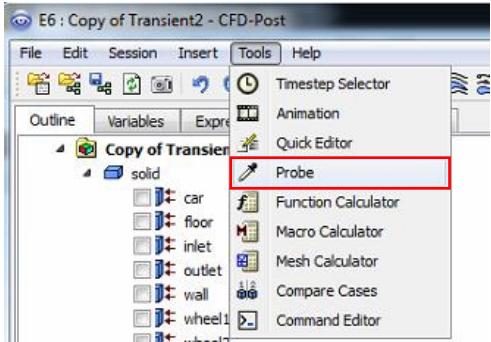
5.6 Insert Timestep Selector for Transient Flow



Select the specific **Timestep** to check the status of the flow

Click **Apply**

5.7 Display results at a Single Point

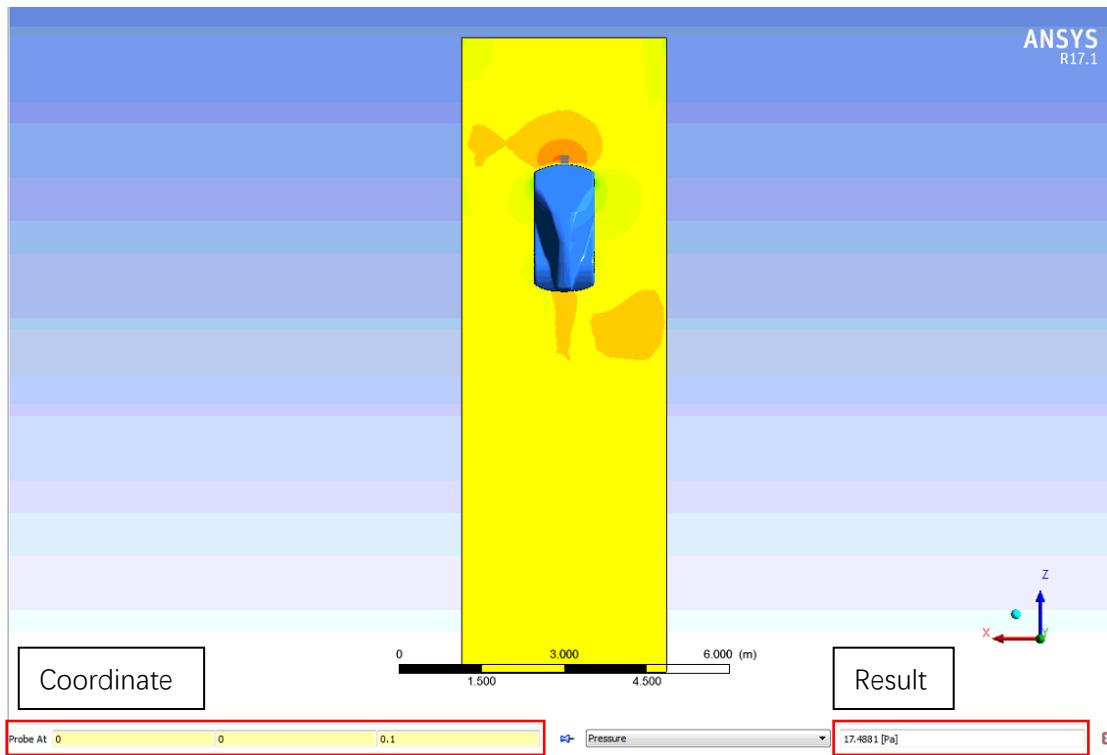


Tools → Probe

Give **Coordinate** to the Single Point

Select **Pressure** from the drop-down list

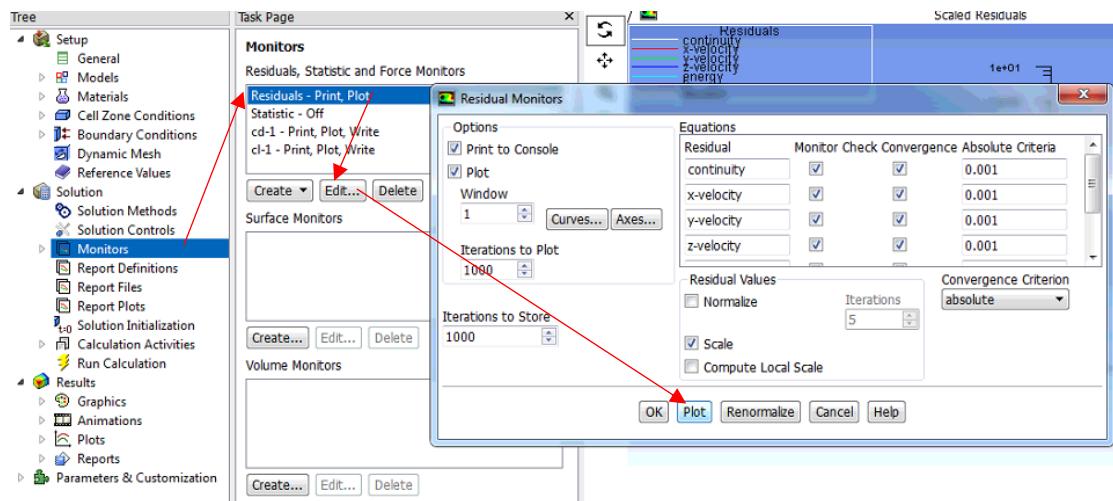
The result shows the pressure of the specific point.



Above shown the pressure of a single point from steady flow simulation.

6. Results Reports for different Solver Settings

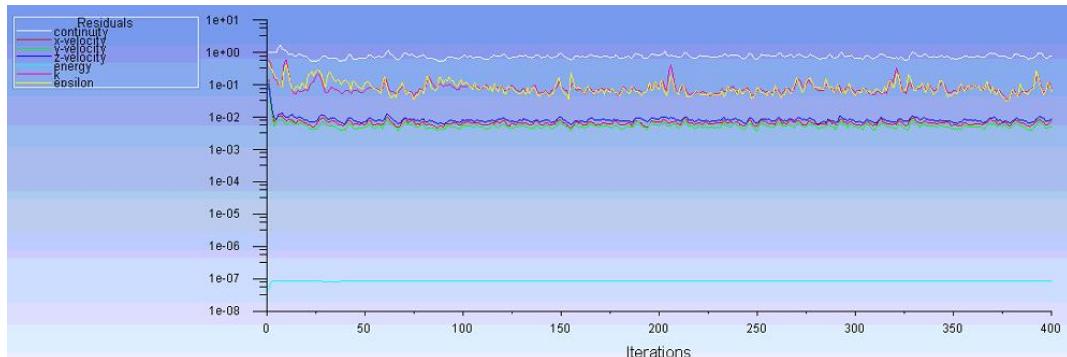
6.1 Check for Convergence



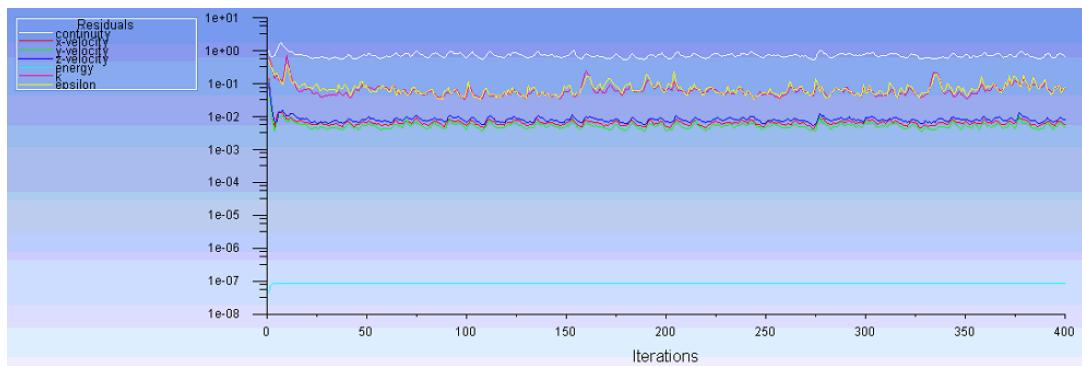
To print out the **Residuals** table in **Console** window:

Click **Monitors** → **Residuals-Print, Plot** → **Edit** → **Plot**

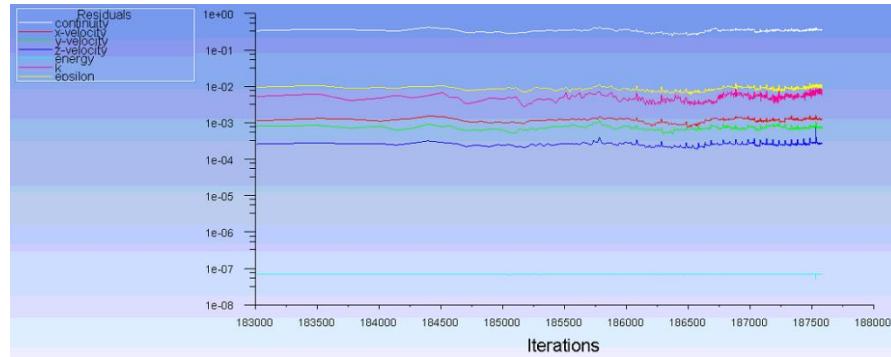
a. Steady Flow, Turbulent Models, Moving floor and Rotating wheels



b. Steady Flow, Turbulent Models, Stationary floor and Stationary wheels



c. Transient Flow, Turbulent Models, Moving floor and Rotating wheels



Summary

All three conditions are **converged**.

***Unconverged results** are misleading, should not be used.

Troubleshooting (Simulation cannot Converge)

- Decrease under-relaxation factors for equations having convergence problems (Pressure-based solver).

Solution → Solution Controls

- Decrease the Courant number (Density-based solver).

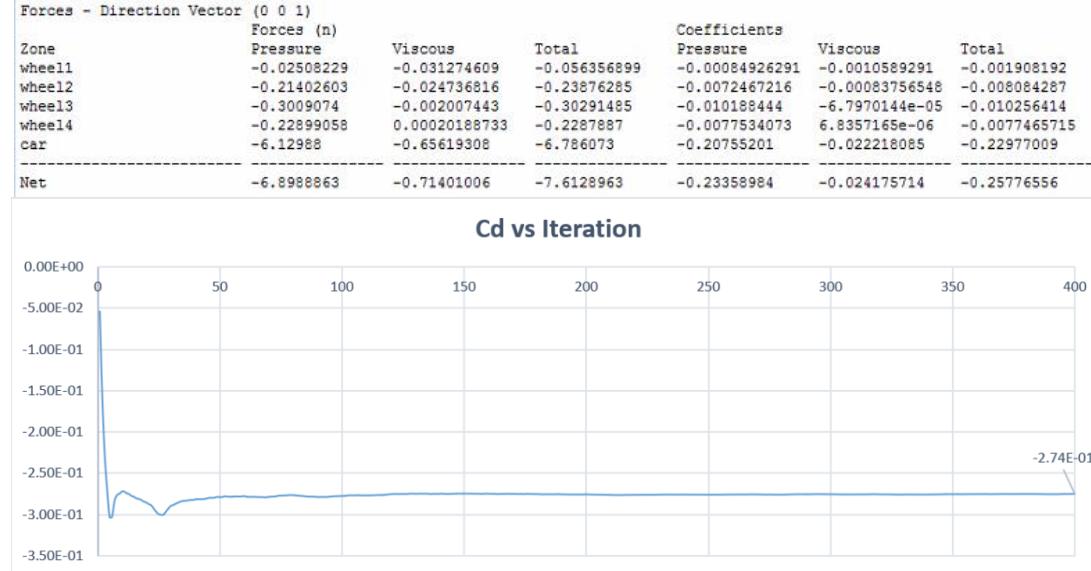
Solution → Solution Controls

- Remesh or refine cells which have large aspect ratio or large skewness.

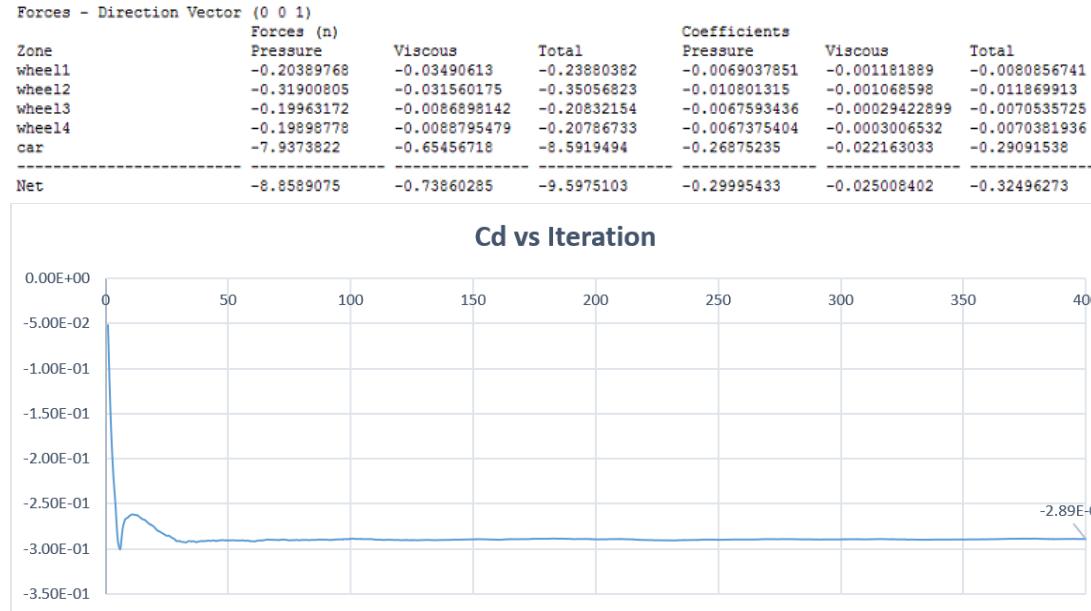
6.2 Compare Force and Coefficient

Drag Force and Coefficient

a. Steady Flow, Turbulent Models, Moving floor and Rotating wheels



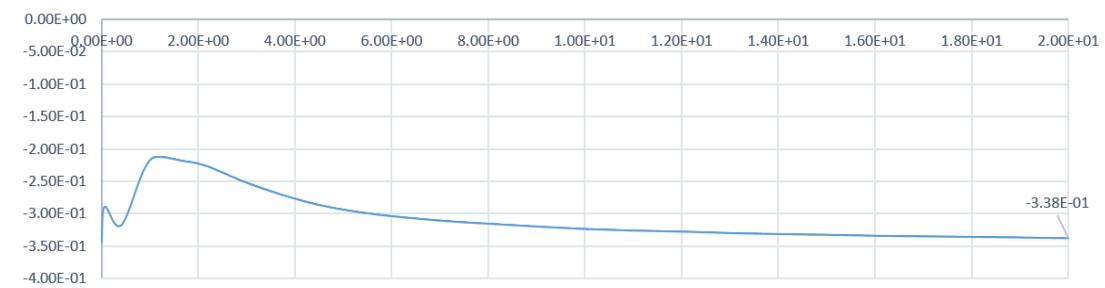
b. Steady Flow, Turbulent Models, Stationary floor and Stationary wheels



c. Transient Flow, Turbulent Models, Moving floor and Rotating wheels

Forces - Direction Vector (0 0 1)				Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
wheel1	0.0904333	-0.030001603	0.060431696	0.003061987	-0.0010158263	0.0020461608
wheel2	-0.2178342	-0.028048821	-0.24588303	-0.0073756626	-0.00094970687	-0.0083253695
wheel3	-0.306759	-0.00076510711	-0.30752411	-0.010386573	-2.5905812e-05	-0.010412479
wheel4	-0.22763354	0.0010958122	-0.22653372	-0.0077074589	3.7103177e-05	-0.0076703557
car	-0.0615894	-0.77379525	-9.6353847	-0.30004514	-0.026199984	-0.32624512
Net	-9.5233829	-0.83151497	-10.354898	-0.32245285	-0.02815432	-0.35060717

Cd vs Time

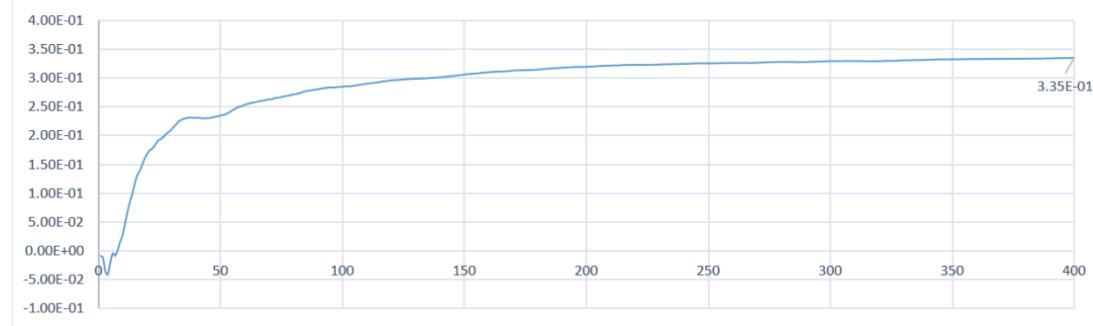


Lift Force and Coefficient

a. Steady Flow, Turbulent Models, Moving floor and Rotating wheels

Forces - Direction Vector (0 1 0)				Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
wheel1	0.80808306	-8.9483336e-05	0.80799357	0.027360937	-3.0298221e-06	0.027357908
wheel2	0.84924686	0.0030297767	0.85227664	0.028754705	0.00010258541	0.028857291
wheel3	0.062287919	-0.0017414708	0.060546448	0.0021090108	-5.8964573e-05	0.0020500462
wheel4	0.042895149	-0.0022584298	0.04063672	0.0014523897	-7.6468322e-05	0.0013759214
car	9.2436504	0.11386637	9.3575168	0.31298137	0.003855409	0.31683678
Net	11.006163	0.11280677	11.11897	0.37265841	0.0038195317	0.37647794

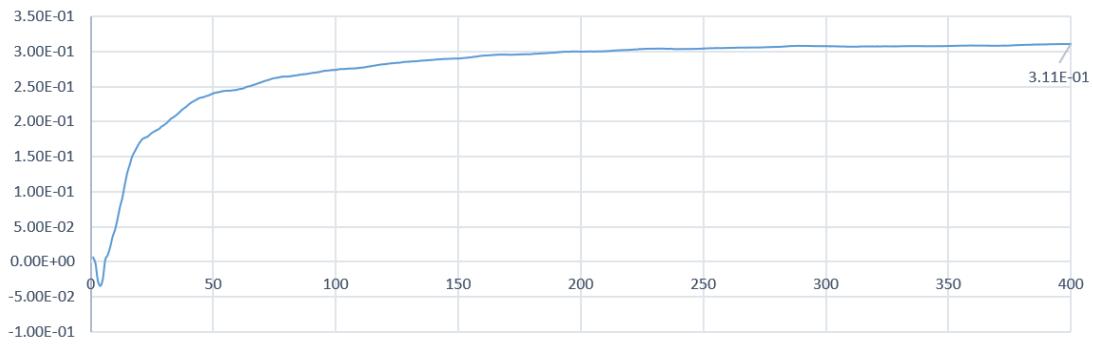
Cl vs Iteration



b. Steady Flow, Turbulent Models, Stationary floor and Stationary wheels

Forces - Direction Vector (0 1 0)				Coefficients			
	Forces (n)	Pressure	Viscous	Total	Pressure	Viscous	Total
Zone							
wheel1	0.73922914	0.0036874204	0.74291656	0.025029608	0.00012485261	0.025154461	
wheel2	0.73598117	0.0057437317	0.7417249	0.024919635	0.00019447739	0.025114112	
wheel3	0.043285355	-0.00047733681	0.042808018	0.0014656017	-1.6162179e-05	0.0014494395	
wheel4	-0.028064907	-0.00016667458	-0.028231581	-0.00095025152	-5.6434456e-06	-0.00095589496	
car	7.9170084	0.13099553	8.0480039	0.26806251	0.0044353862	0.27249789	
Net	9.4074392	0.13978267	9.5472218	0.3185271	0.0047329105	0.32326001	

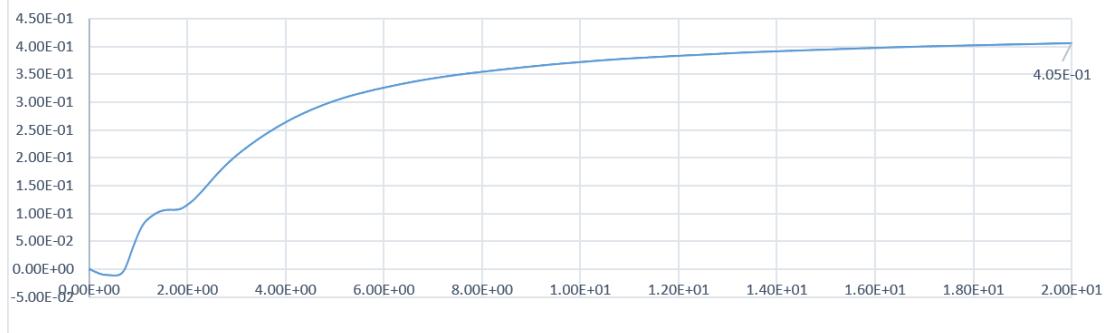
Cl vs Time



c. Transient Flow, Turbulent Models, Moving floor and Rotating wheels

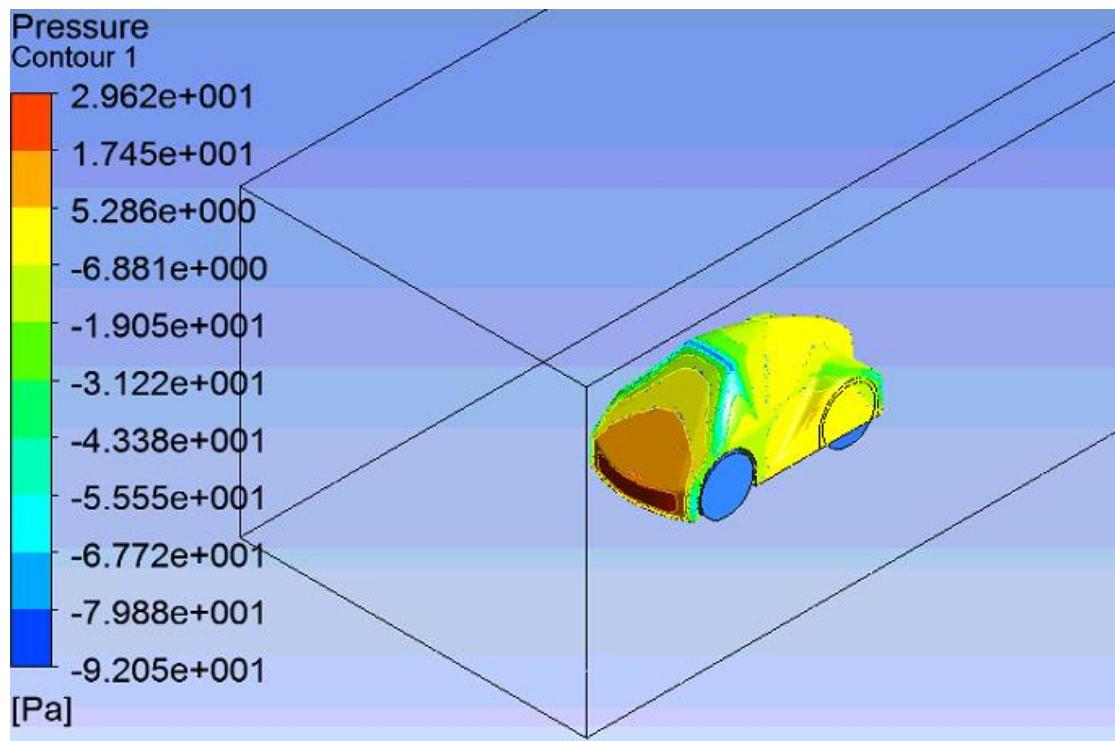
Forces - Direction Vector (0 1 0)				Coefficients			
	Forces (n)	Pressure	Viscous	Total	Pressure	Viscous	Total
Zone							
wheel1	1.0168421	0.00054132007	1.0173834	0.034429324	1.8328592e-05	0.034447653	
wheel2	0.9883852	0.0014946852	0.98987989	0.033465799	5.0608644e-05	0.033516408	
wheel3	0.077462435	-0.0026135817	0.074848853	0.0026228057	-8.8493435e-05	0.0025343122	
wheel4	0.17267396	-0.0032905035	0.16938345	0.0058465788	-0.00011141337	0.0057351654	
car	10.955781	0.1000594	11.05584	0.37095251	0.003387918	0.37434043	
Net	13.211145	0.096191325	13.307336	0.44731702	0.0032569484	0.45057397	

Cl vs Time

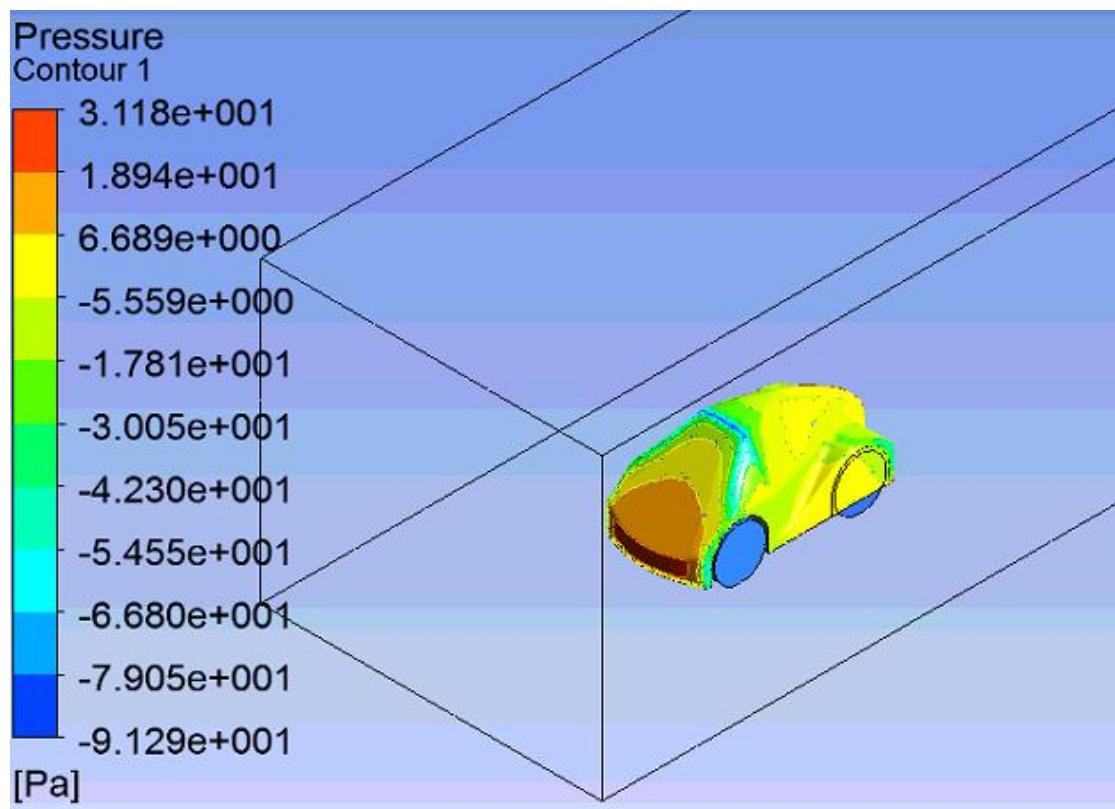


6.3 Compare Pressure on the Bodyshell

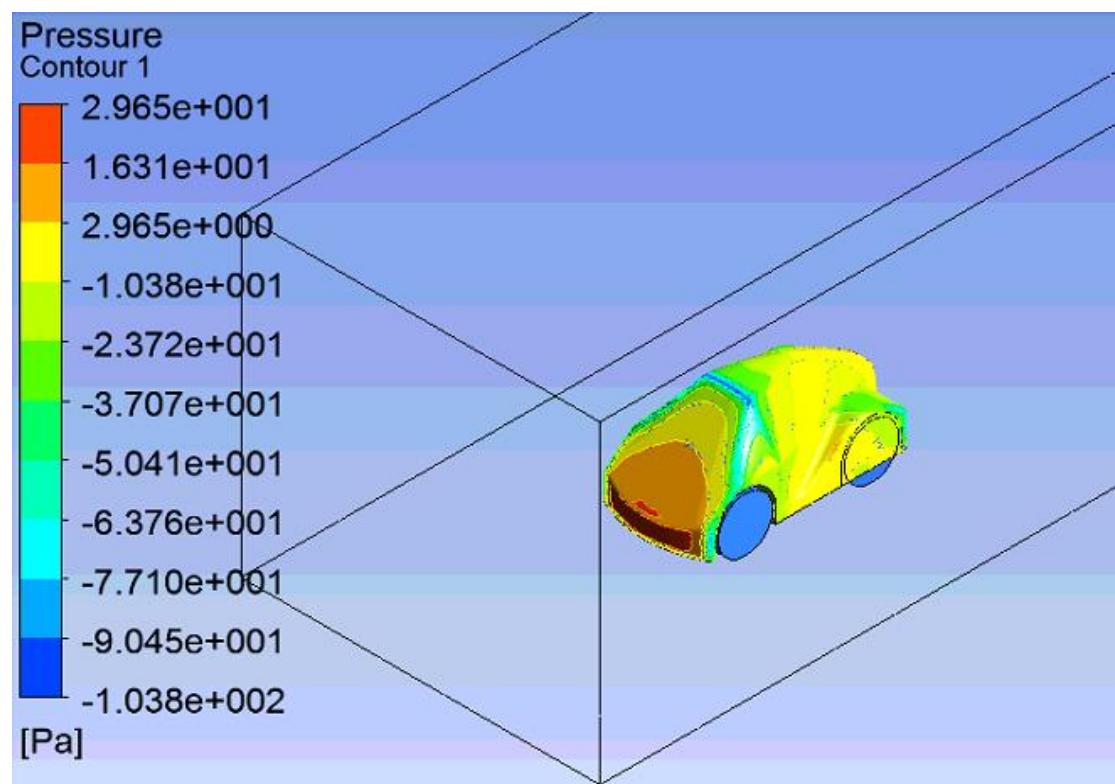
a. Steady Flow, Turbulent Models, Moving floor and Rotating wheels



b. Steady Flow, Turbulent Models, Stationary floor and Stationary wheels



c. Transient Flow, Turbulent Models, Moving floor and Rotating wheels



End