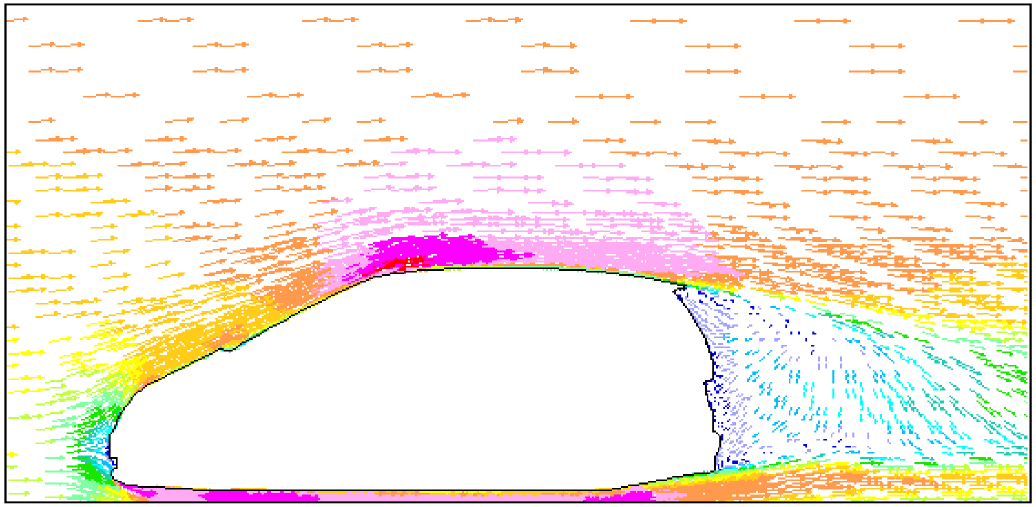
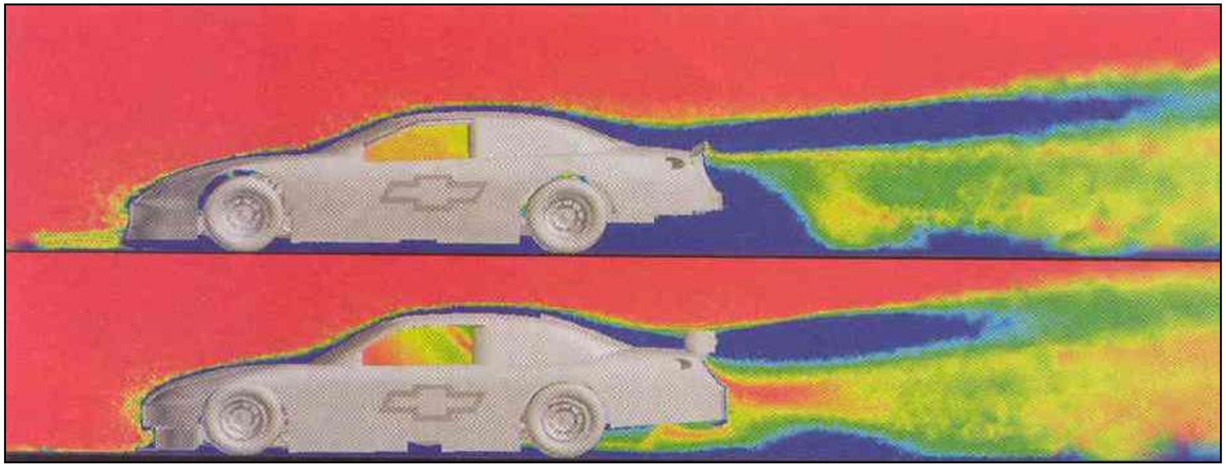
EXTERNAL AERODYNAMICS: FLOW OVER A CAR





Source: Automotive Engineering

OBJECTIVE OF AERODYNAMIC CALCULATIONS

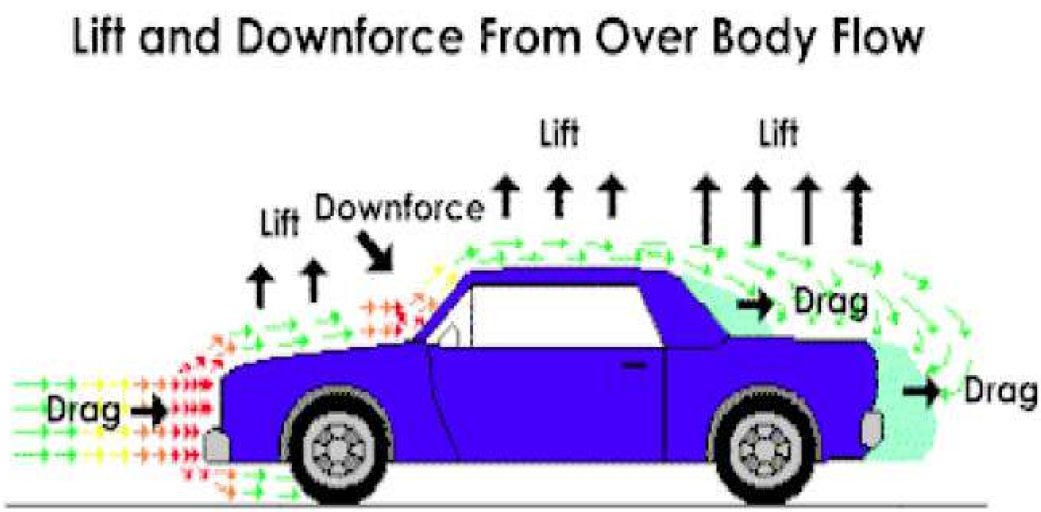
- To compute & visualize velocity and pressure contour over car body surfaces
- To Predict Aerodynamic forces to compute lift and drag coefficients
- To minimize aerodynamic forces (drag, lift, and side force) by modifying the body shape without changing the original structural frame of the vehicle.

EXTERNAL AERODYNAMICS TERMINOLOGY

- When a road vehicle moves through air, its body experiences aerodynamic/viscous forces and moments
- Force experienced OPPOSITE the direction of the moving body: *drag. Around 60% of power required for cruising at high speed in spent in overcoming the air drag. In America, a speed limit of 55 mph was imposed during 1973 oil crisis keeping in mind this drawback!*
- Vertical force experienced perpendicular to the drag: *lift*
- The body also experiences a *side force* and a *rolling moment* additional to the drag and lift when there is a crosswind.

EXTERNAL AERODYNAMICS TERMINOLOGY

Summary of Forces



F_d = Drag Force

C_d = Drag Coefficient

$$\mathbf{F}_d = \frac{1}{2}\rho \mathbf{v}^2 C_d A$$

V = Free Stream Velocity

ρ = Free Stream Density

A = Frontal or Reference Area

A . C_d = Drag Area

This parameter is used to consider various designs of Automotive bodies.

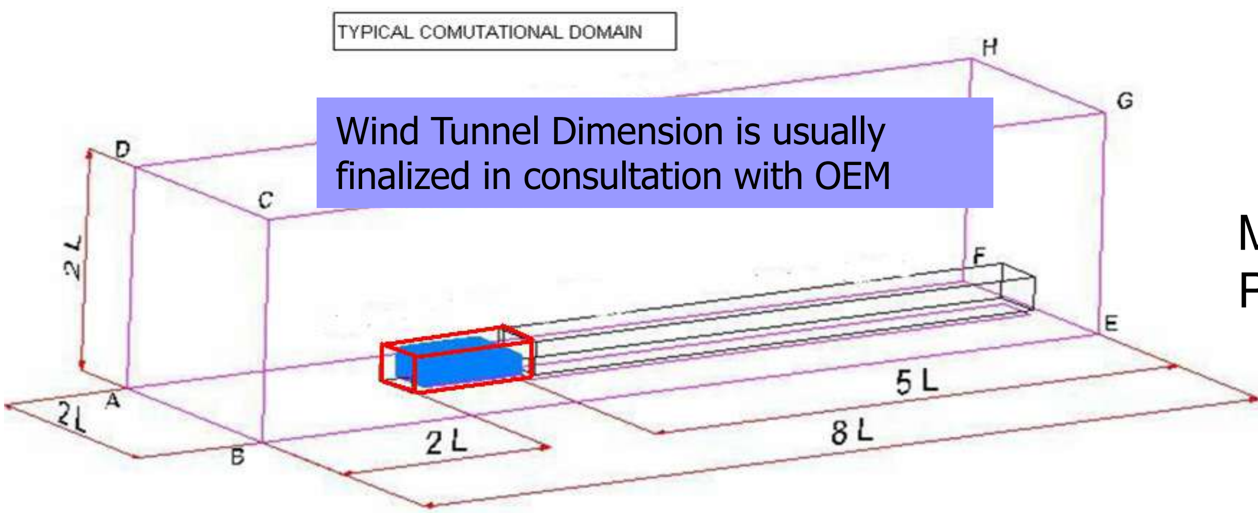
1. Typical range of 'Drag Area' is 0.60 ~ 0.80 m^2 (VW Polo (Class A): 0.636)

Drag coefficient for Automobiles varies in the range 0.3 ~ 0.5
(VW Polo: 0.37)

CFD SIMULATION DOMAIN SET-UP

Simulation Assumptions:

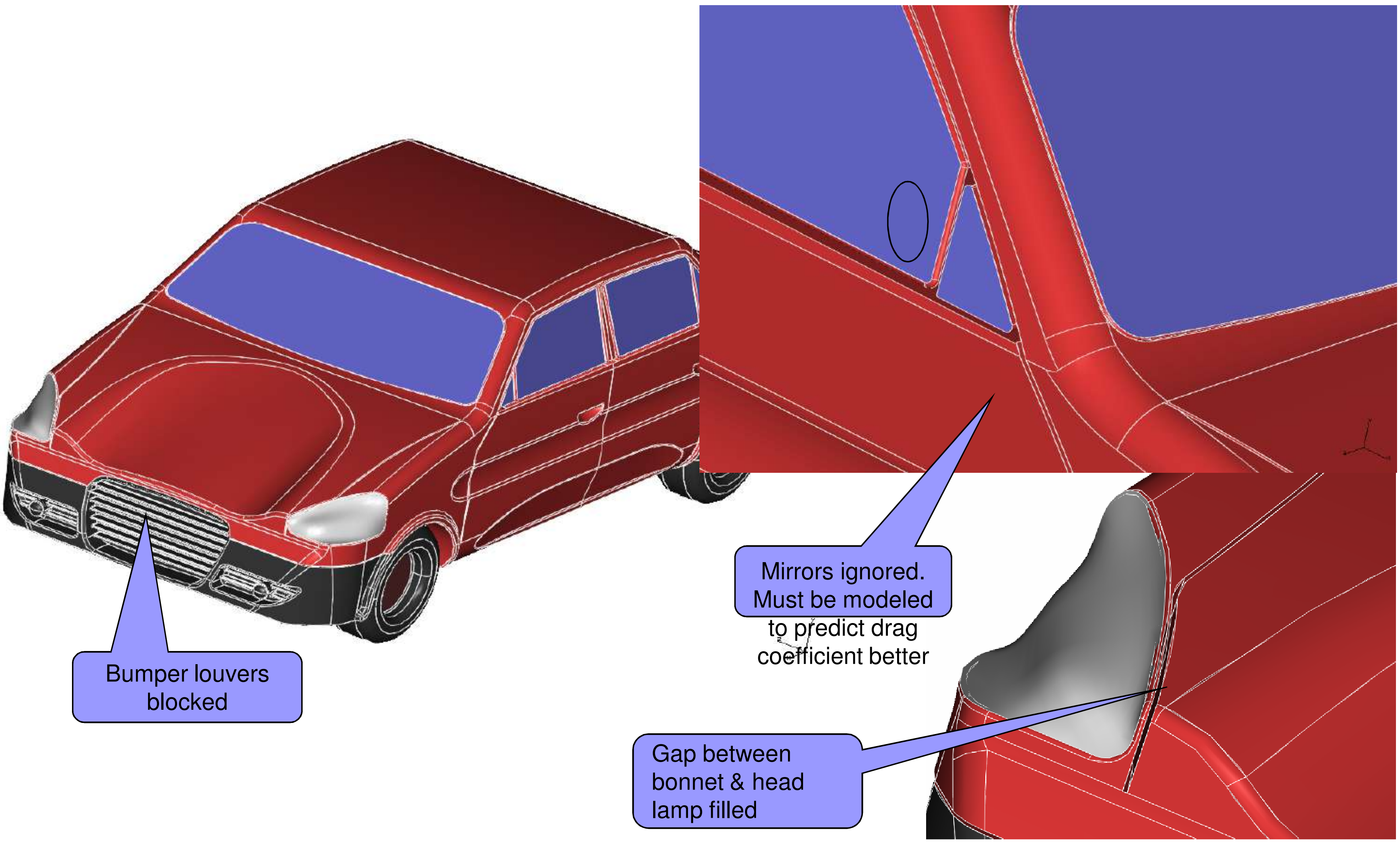
1. Car body surfaces and ground have been assumed (hydraulically) smooth.
2. RVSM (Rear View Side Mirror) not considered. Local step in door glasses and body panels, weatherstrips, door handles, roof trims, etc have not considered.
3. Complex flow geometries like bumper grille & tire have initially been not considered, geometry have been assumed symmetrical and only half the domain have been used.
4. Computational domain have been selected based on approximate WIND TUNNEL DIMENSION.
5. Components in Engine Compartment have been ignored.



L = Length of Bluff Body i.e. Car

Model: L x W x H:
Prototype: L x W x H:

GEOMETRY SIMPLIFICATION



DOMAIN SIZE REDUCTION THRU SIMILARITY

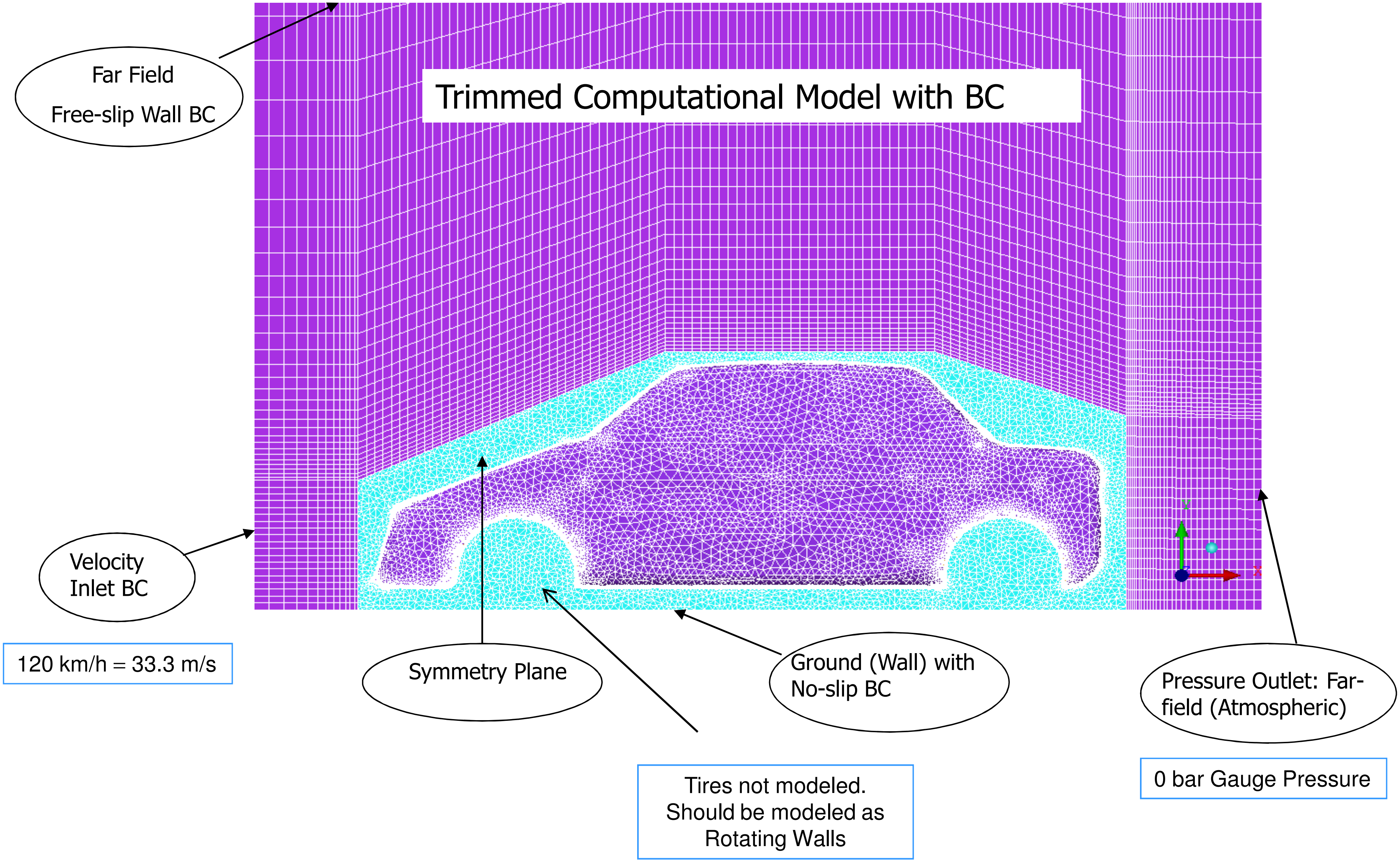
Motivation: "Scale Down" the computational domain to meet the system memory constraints without affecting the computational accuracy

Criteria:

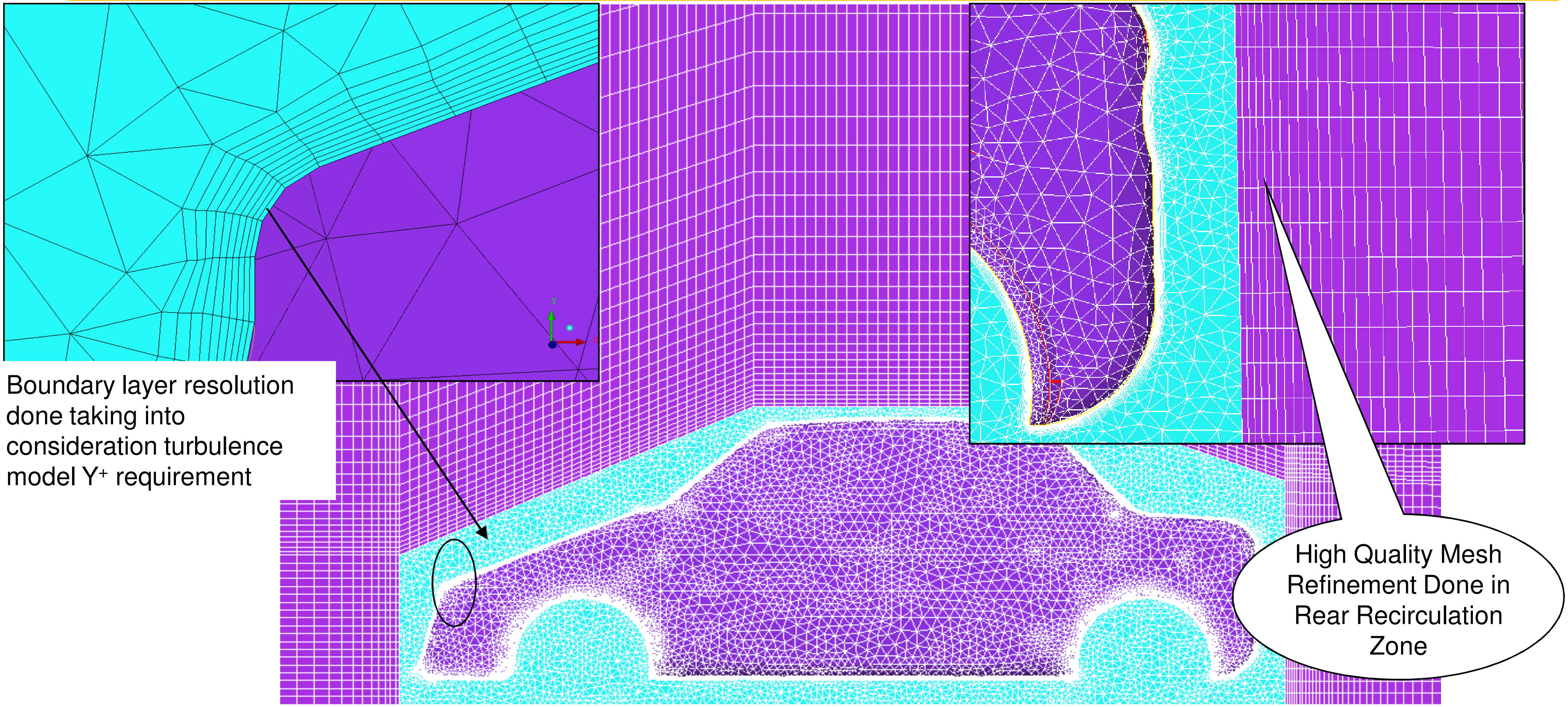
1. **Geometrical Similarity:** This requirement involves similarity of the form
2. **Kinematic Similarity:** This is necessity of similarity of motion. The scaling down or the car body retains the patterns & path of motion occurrences
3. **Dynamic Similarity:** This is the similarity of forces. Since mass in incompressible flow does not change, dynamic similarity is also maintained. $\{ Re_{Model} = Re_{Prototype} \}$

			Model	Prototype	Remark
Flow Velocity	V	m/s	33.3	66.7	Velocity Doubled
Flow Length Scale	L	m	1.395	0.6975	Length Scales Halved
Free Stream Temperature	T	K	298	298	No Change
Free Stream Pressure	P	Pa	101325	101325	No Change
Free Sream Density	ρ	kg/m3	1.18	1.18	No Change
Free Stream Viscosity	μ	kg/m-s	1.84E-05	1.84E-05	No Change
Reynolds No. of Flow	Re	---	2.99E+06	2.99E+06	No Change
Free Stream Flow Mach No.	Ma	---	0.10	0.20	< 0.3. Flow can still be assumed incompressible

CFD SIMULATION B.C. SET-UP



COMPUTATIONAL MESH DESCIPTION

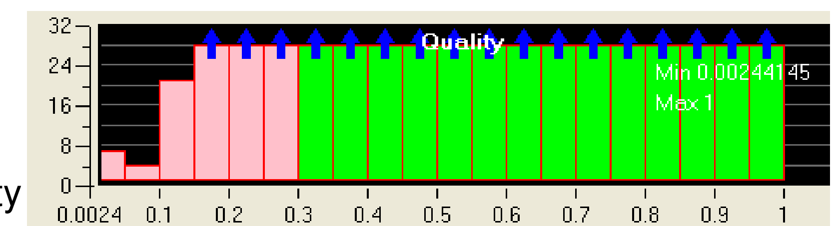


Mesh Statistics:

- Tet Elements: 163588
- Hex Elements: 568370
- Wedge Elements: 211552
- Pyramid Elements: 20990

Mesh Quality:

- High quality mesh, only 0.029% elements have quality < 0.3



The quality deterioration has been observed primarily in the “prism (wedge) – tet” transition region

CFD SIMULATION SOLVER APPROACH

•Use UPWIND difference scheme with standard k-ε turbulence modeling to compute initial velocity field.

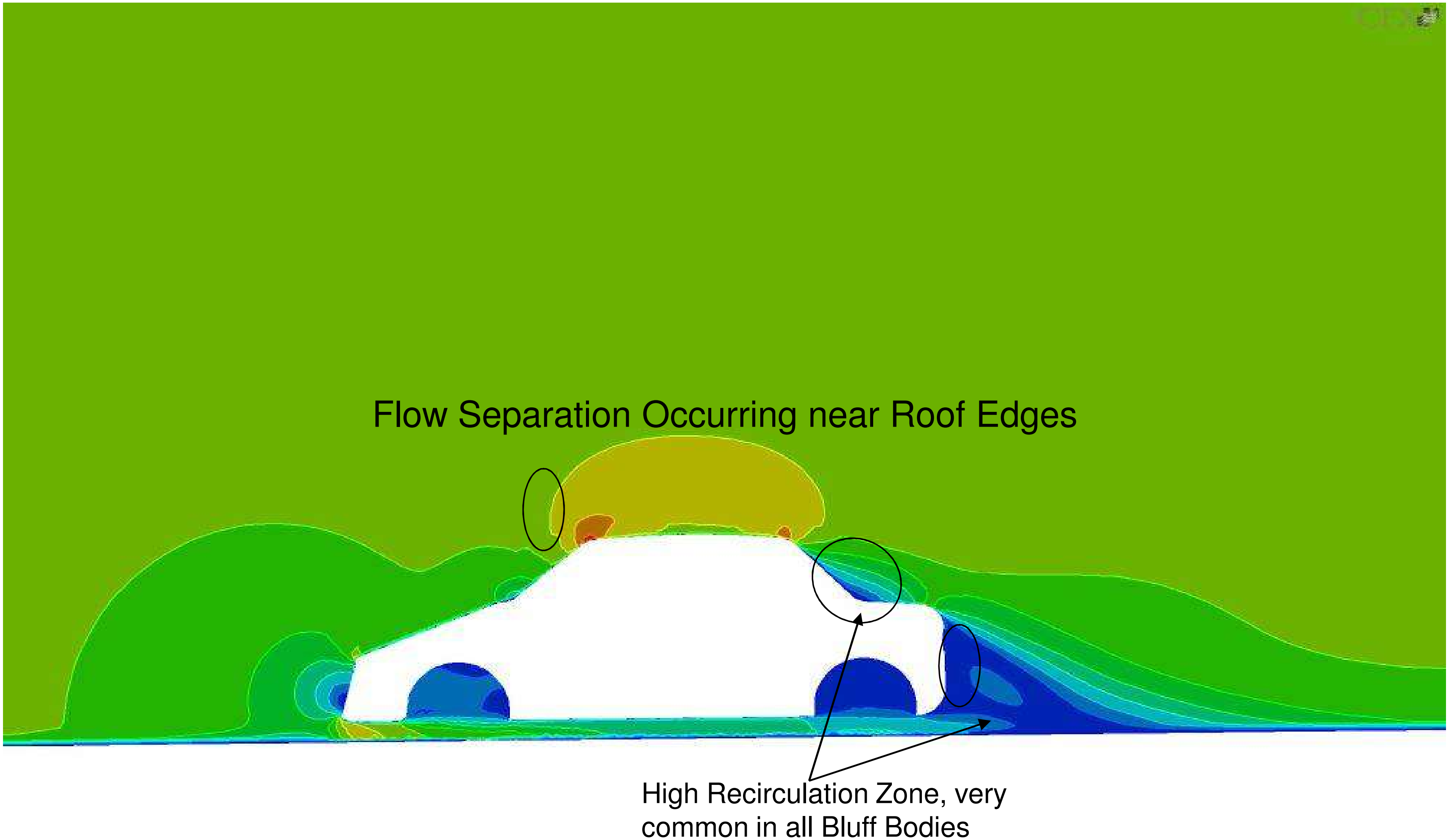
•Use more accurate SST (Shear Stress Transport) model available in CFX10 to get better results. The SST model is recommended when flow is characterized by flow separation and strong recirculation zones.

•**Convergence Criteria:** 1E-04, MAX Residual

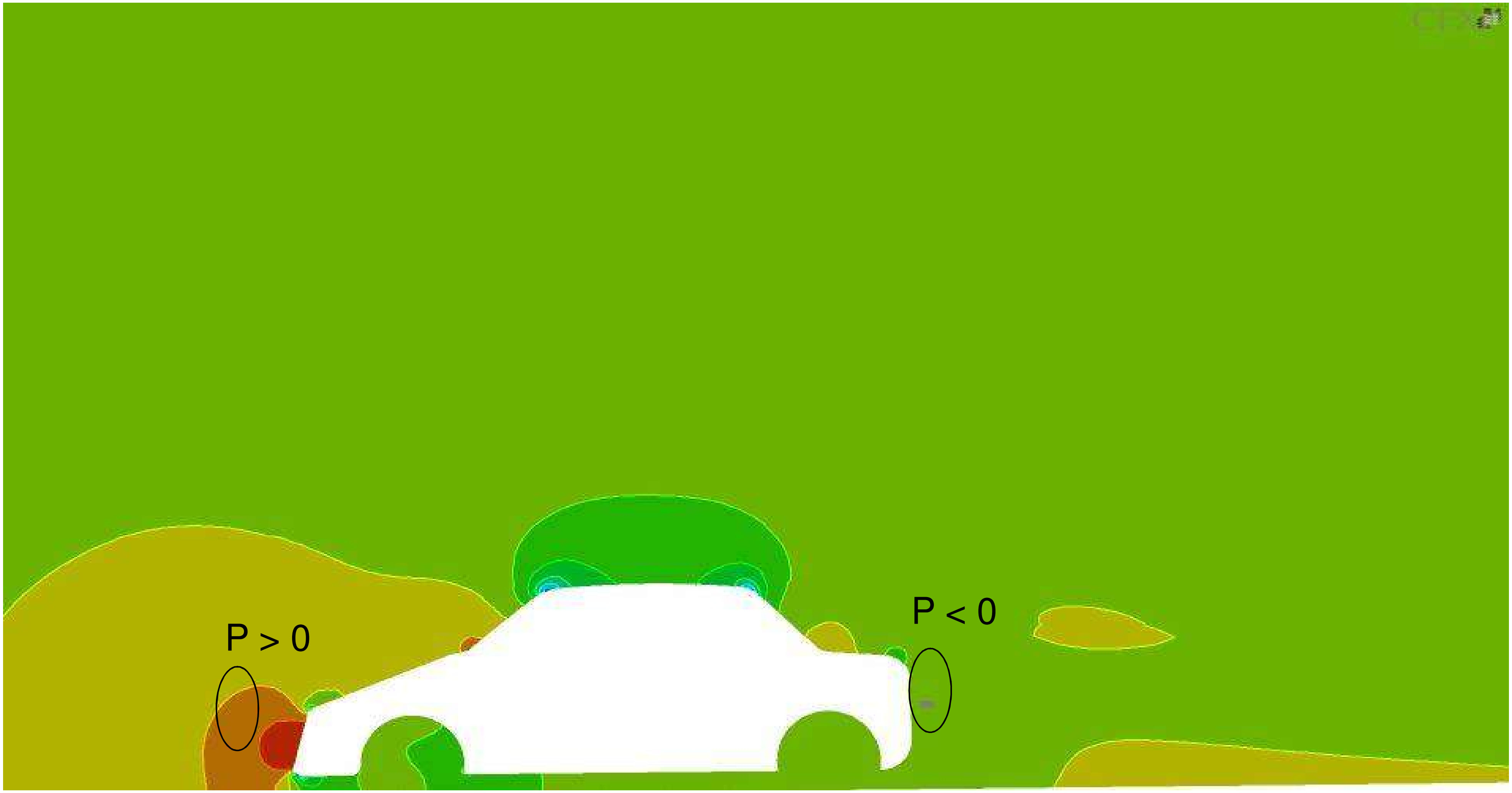
•**Time Step:** Physical Time Step, 0.02 s

•**Expert Parameters:** Reference pressure set to ambient in order to calculate pressure forces

SIMULATION OUTPUT: VELOCITY CONTOUR



SIMULATION OUTPUT: PRESSURE CONTOUR



High Dynamic Head Casing
Vehicle Push Force

Low Pressure (Vacuum) Head
Casing Vehicle Pull Force

CONCLUSION

•The front windshield inclination is well streamlined and no stagnant flow region observed near front cowl

•Rear windshield inclination should be reduced to prevent the small recirculation zone formed near its lower edge

•Similarity Theory can be successfully utilized for engineering problems to overcome the hardware constraints

•Symmetric Computational domain may be used to reduce the computation effort without significant loss of accuracy. Simulation with complete vehicle domain is recommended when a correlation with experimental data is being done.