Computational Domain Selection for CFD Simulation
The guidelines are very generic in nature and has been explained with examples. However, the users may need to check their problem set-up with the intended operating conditions, testing environment to establish a realistic simulation set-up.

1. Why selection of computational domain is important?
2. Basic principles of selection (size and envelop) of the computational domain
3. Geometry simplification recommendation for computational domain
4. Examples
The underlying assumptions and mathematical constraints involved with Computational Fluid Dynamics has made it an ART rather than a science or industrial engineering application. A scientifically correct simulation not only demand the correct understanding of underlying physics of the problem at hand (which is true in most of the engineering applications), a keen sense of testing, measurement and instrumentation is equally important. For example, if a CFD engineer (for purist “CFD analyst”) is planning to generate the performance characteristics of a Venturi-meter or an Orifice-meter, he must give attention to the test set-up historically used for this purpose. It will involve asking questions like “What is the length of pipe upstream the actual measurement section?”, “Where is the pressure measured downstream the Venturi?”, Is the pressure measured at only one location or at multiple angular position at a given cross-section? If yes, why?”, so on and so forth.

In the following section, we will see how answer to these questions help (rather lead) us to decide the computational domain scientifically correct.
**Basic:** Any CFD simulation involves 3D region or its simplified 2D counterpart) where fluid is supposed to occupy the space and flow. Hence, if one says that he is simulating flow through a U-bend, it is always assumed that the complete space inside the tube is filled with fluid under consideration.

Selection of computational geometry is not only dependent on the actual shape and size of the product. At the same time it must address the mathematical constraints under which a CFD software solves the governing Differential Equations. The applicability of “Fully Developed” and “Developing” flow should be carefully thought of while deciding the computational domain. Let us start with some examples before going into other details. In example below, for Outlet-1, clearly CFD will never produce
correct result even though the pipe length downstream the orifice-meter or control valve is very short in actual application. Hence, to get a fairly accurate performance characteristics of an orifice-meter or a Control Valve, it is vital to have a prior understanding of the highly turbulent zone & length formed downstream such narrow passages. It must be emphasized that this requirement of knowledge should come from your basic understandings of Fluid Mechanics and not from theory behind CFD. Questions such as the following was encountered in a forum, “I have written a CFD program to calculate laminar flows. Can someone provide me a benchmark data to validate my code?” I wonder who is the actual author of the code being refereed by the member! It is ironic to develop a CFD code development expertise without having the knowledge of Hagen-Poiseuille flow equations.

**Symmetry:** Domain simplification is a very common approach and if used judiciously, results in accurate results faster with lesser computational resources. However, the inherent nature of lack-of-symmetry found in most industrial flow problems warns us to use this approach with a pinch of salt.
Example-1: The much read and common flow configuration, flow over a cylinder ("Bluff Body") is not symmetrical at Reynolds Number typically encountered in industrial applications.

Example-2: Flow with sudden expansion tend to be asymmetric even though geometry is symmetrical.
Example-3: In the figure below, the flow domain proposed by dashed lines is incorrect since the flow in the wake region (region behind any “bluff body”, here the cubical step) is far from being symmetrical or periodic.
Tips & Tricks-1: The above picture demonstrates flow over a extended surface also called Heat Sinks and an integral part of CPUs. Assuming that the end effects are negligible and there is no conduction losses in transverse direction (LH Fin ↔ RH Fin), there are two possible scenario to select a domain incorporating “SYMMETRY”. Given the fact that the fin thickness cannot be neglected as compared to gap between two consecutive fins, which configuration would you choose and why?

Answer: C1, Why?
**Tips & Tricks-2:** Domain length downstream the car (shown as “5~20 L”) can only be guessed by an expert aero-dynamist or by trial & error. These lengths have been selected based on typical wind-tunnel used in industry. If you are simulating aerodynamics for a racing car, this size of domain may not suffice.
Tips & Tricks-3: The dimension 'w' should be large enough to affect the Boundary Layer growth on the vertical wall.

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Requirement</th>
</tr>
</thead>
<tbody>
<tr>
<td>w / L</td>
<td>&gt; 0.5</td>
</tr>
<tr>
<td>b / L</td>
<td>&gt; 0.1</td>
</tr>
<tr>
<td>a / L</td>
<td>&gt; 0.1</td>
</tr>
</tbody>
</table>
**Tips & Tricks-4:** Systematic experiments on 2D flows with pressure drop and pressure rise in convergence and divergent channel with flat walls have been carried out by F. Doench, J. Nikuradse, H. Hochschild and J. Polzin. The included angle of the channels ranged over -16°, -8°, -4°, 0°, 2°, 4°, 6°, 8°.

For included angles up to 8° in a divergence channel the velocity profile is fully symmetric over the width of the channel and shown no feature associated with separation. On increasing the included angle beyond 8°, there is a remarkable shift in velocity profile which cease to be symmetrical for channels with 10°, 12° and 16° included angle.

With 10° deg angle of divergence, no back flow can yet be discerned, but separation is about to begin on one of the channel walls. In addition the flow becomes unstable so that, depending on fortuitous disturbances, the stream adheres alternatively to the one or the other wall of the channel. Such an instability is characteristic of incipient separation and 1st occurrence at an angle between 4.8° and 5.1° was observed by J. Nikuradse.
Velocity distribution in **convergent** and **divergent** channels with flat walls as measured by Nikuradse

\( \alpha \) — half included angle; \( B \) — width of channel

Velocity distribution in a **divergent** channel of half included angle \( \alpha = 5^\circ \). The lack of symmetry in the velocity distribution signifies incipient separation.
Velocity distribution in a divergent channel of half included angle $\alpha = 6^\circ$. Reverse flow and separation are seen to be setting in on the right-hand wall; $B$ — width of channel.

Velocity distribution in a divergent channel of half included angle $\alpha = 8^\circ$. Reverse flow is completely developed. The flow oscillates at longer intervals between patterns (a) and (b).
• Inlet and Outlet boundary conditions should be always free from recirculation zones.

• Overall dimension of the numerical domain should match the actual experimental set-up such as Wind tunnel for car External Aerodynamics.

• Choice of Porous domain or detail modelling of the geometry should be made judiciously on past experimental data and the complexity of the flow domain.

• Use of ‘Symmetry’ should be used keeping in mind actual flow phenomena. A “Geometric Symmetry” does not necessarily imply a “Flow Symmetry” such as “Flow with a Sudden Expansion” over a geometrically symmetric domain.

• Main parameter which gives a preview of symmetrical behaviour is the Reynolds number. If the Reynolds number is high the flow tends to be asymmetric.

• Geometry simplifications such as reducing a 3D problem to 2D should be done keeping in mind boundary effects. The length scale which is being simplified should be at least an order of magnitude larger than the other flow length scales.