

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.

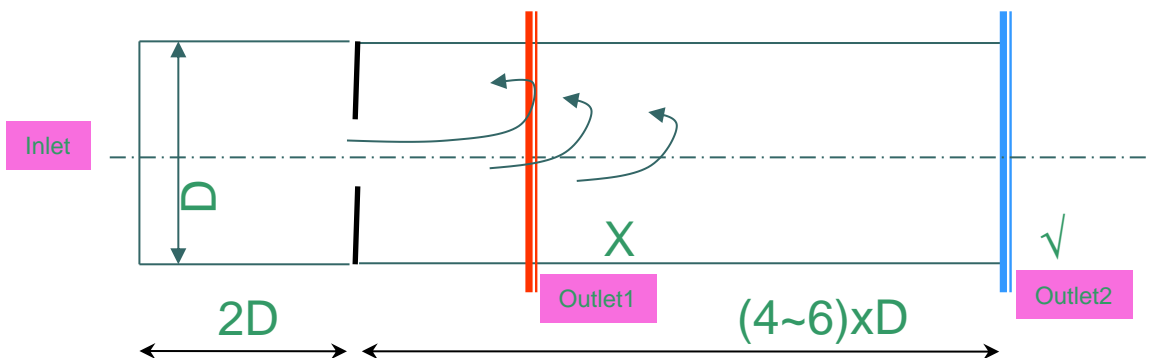
1. Why selection of computational domain is important?
2. Basic principles of selection (size and envelop) of the computational domain
3. Geometry simplification recommendation for computational domain
4. Examples

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.

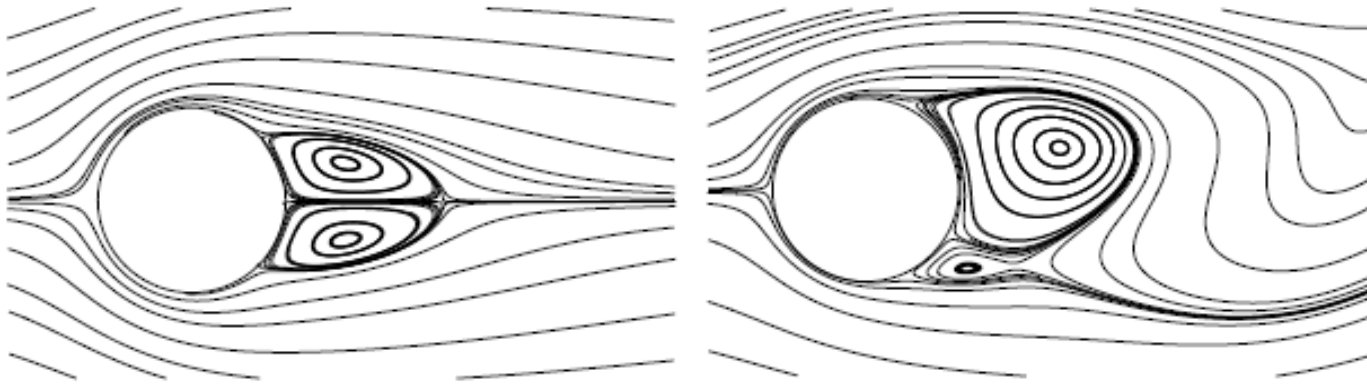
Basic: Any CFD simulation involves 3D region or its simplified 2D counterpart) where fluid is supposed to occupy the space and flow. Hence, if one says that he is simulating flow through a U-bend, it is always assumed that the complete space inside the tube is filled with fluid under consideration.

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. Lets us start with some examples before going into other details. In example below, for Outlet-1, clearly CFD will never produce



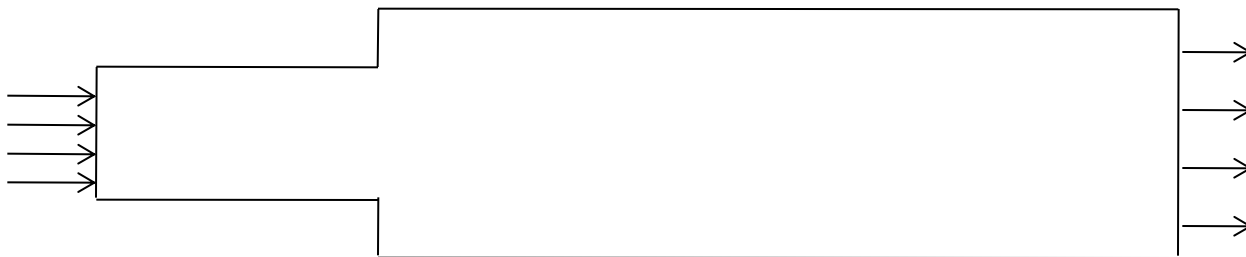
correct result even though the pipe length downstream the orifice-meter or control valve is very short in actual application. Hence, to get a fairly accurate performance characteristics of an orifice-meter or a Control Valve, it is vital to have a prior understanding of the highly turbulent zone & length 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.

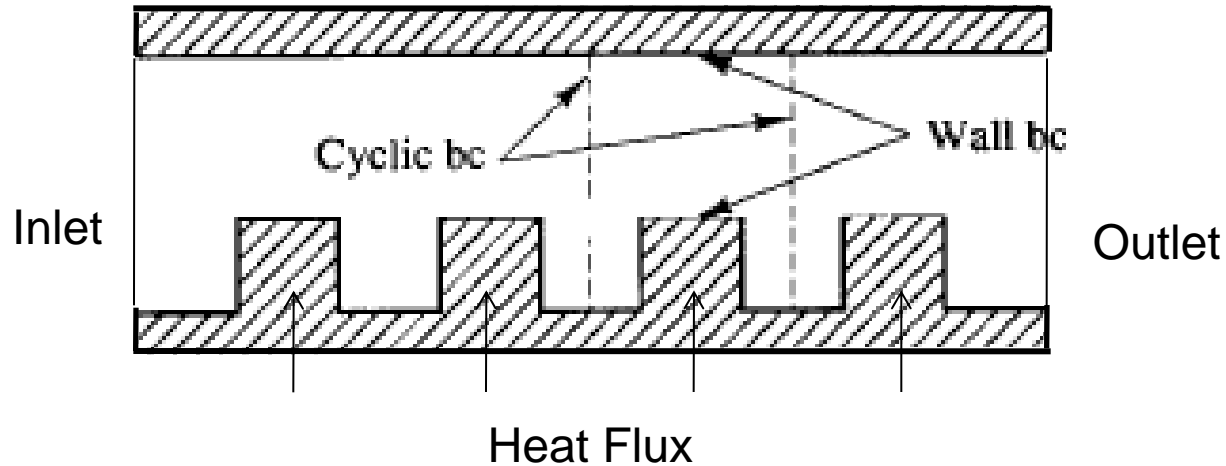


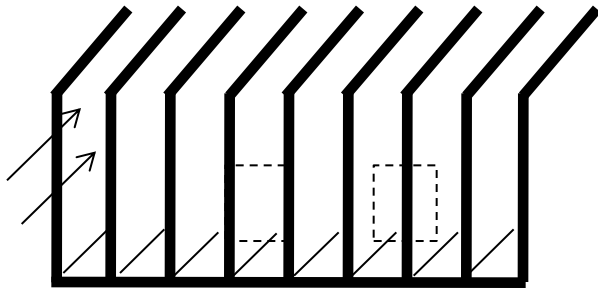
Example-1: The much read and common flow configuration, flow over a cylinder ("Bluff Body") is not symmetrical at Reynolds Number typically encountered in industrial applications.

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.





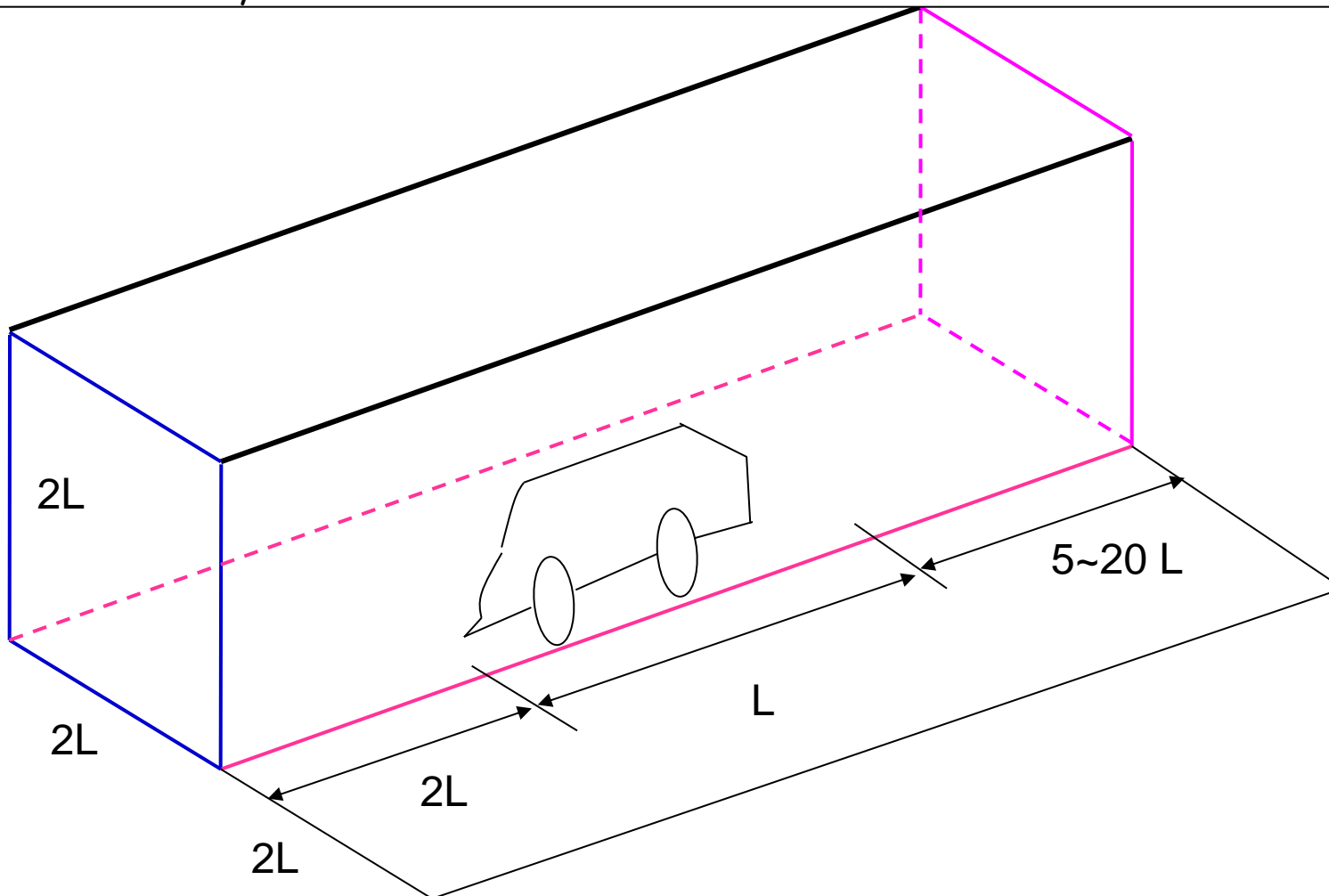
C1 C2

Tips & Tricks-1: The above picture demonstrates flow over an 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 \Leftrightarrow RH Fin), there are two possible scenarios to select a domain incorporating "SYMMETRY". Given the fact that the fin thickness cannot be neglected as compared to the gap between two consecutive fins, which configuration would you choose and why?

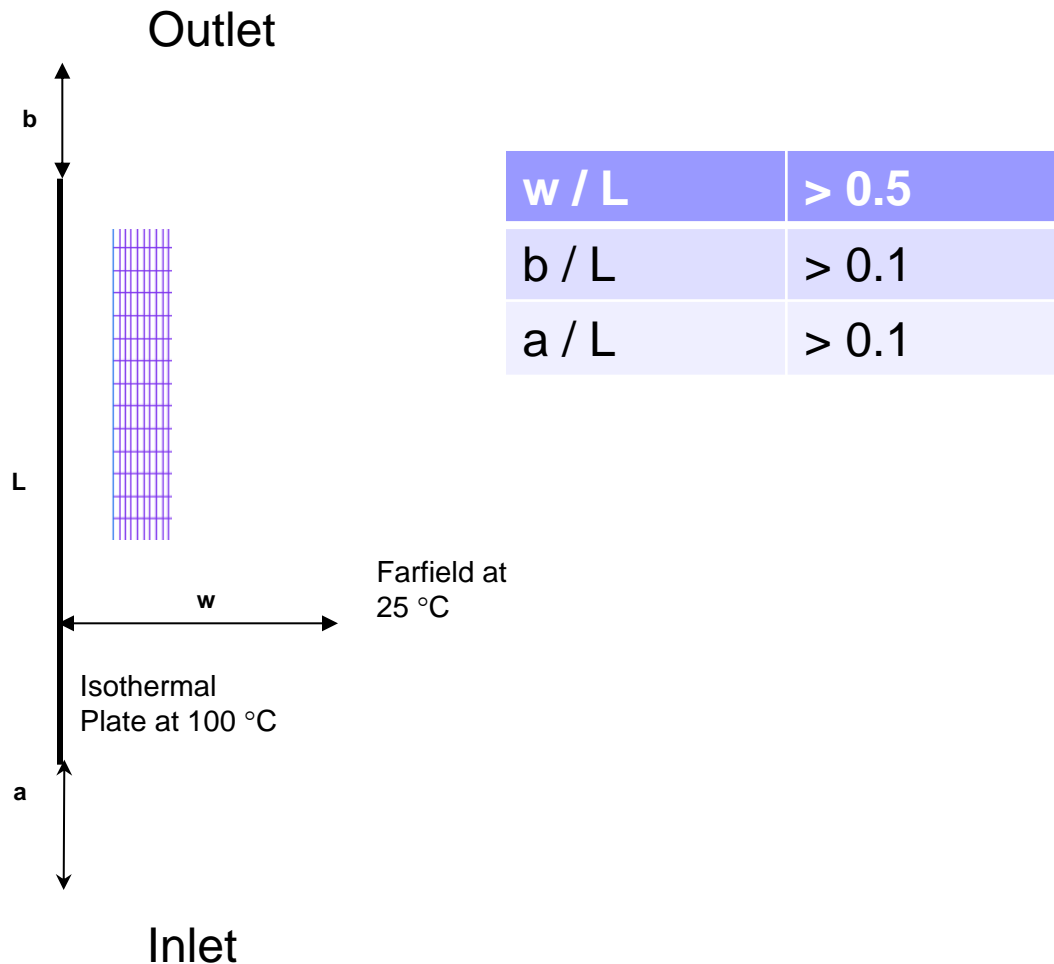
Answer: C1, Why?

Domain Selection: Automotive External Aerodynamics

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.

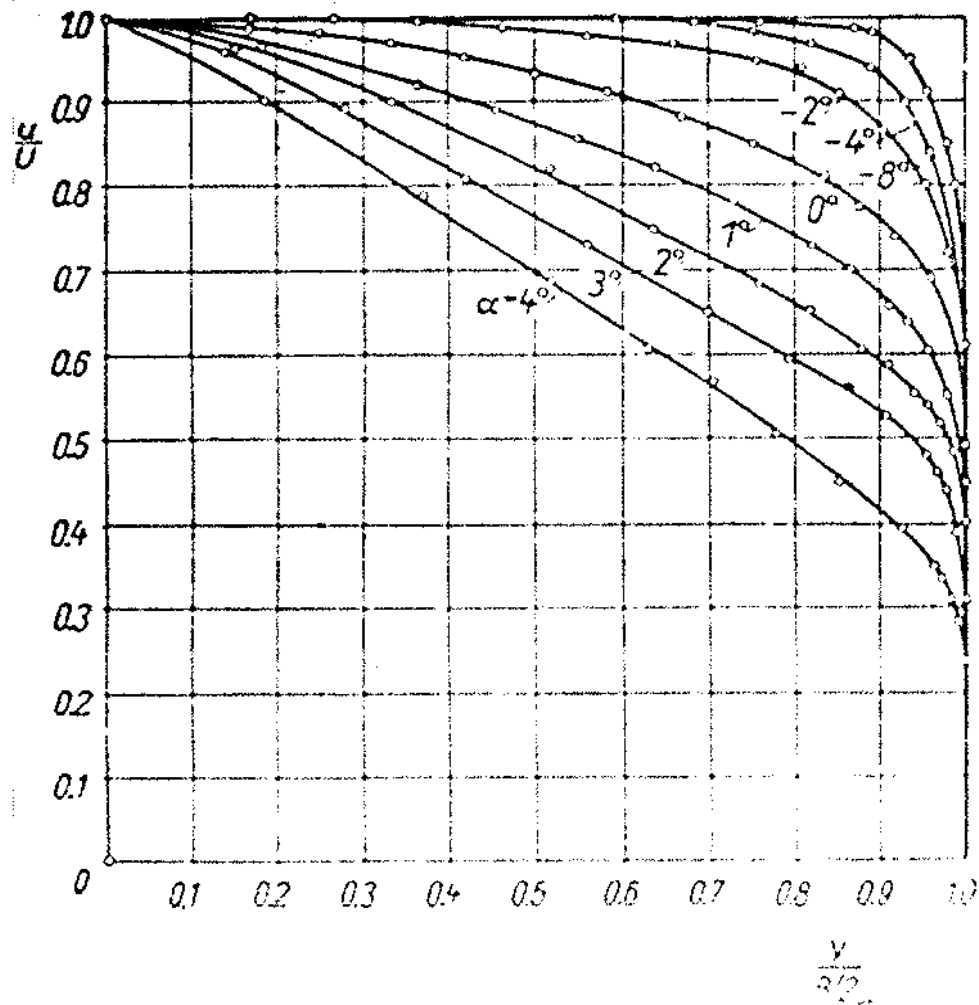


Tips & Tricks-4: Systematic experiments on 2D flows with pressure drop and pressure rise in convergence and divergent channel with flat walls have been carried out by F. Doench, J. Nikuradse, H. Hochschild and J. Polzin. The included angle of the channels ranged over -16° , -8° , -4° , 0° , 2° , 4° , 6° , 8° .

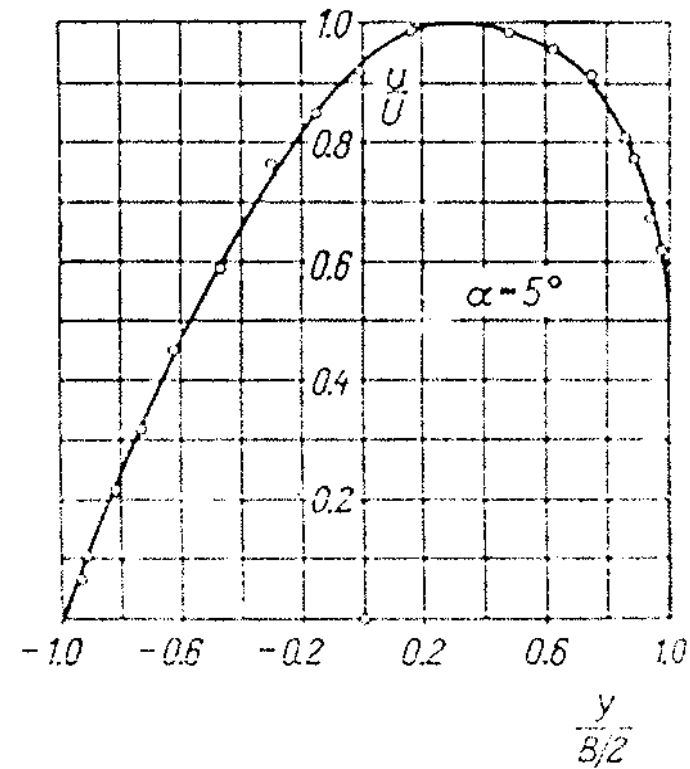
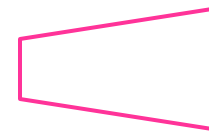
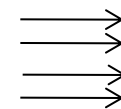
For included angles up to 8° in a divergence channel the velocity profile is fully symmetric over the width of the channel and shown no feature associated with separation. On increasing the included angle beyond 8° , there is a remarkable shift in velocity profile which cease to be symmetrical for channels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

Computational Domain Selection: A Prior Insight in Flow Field

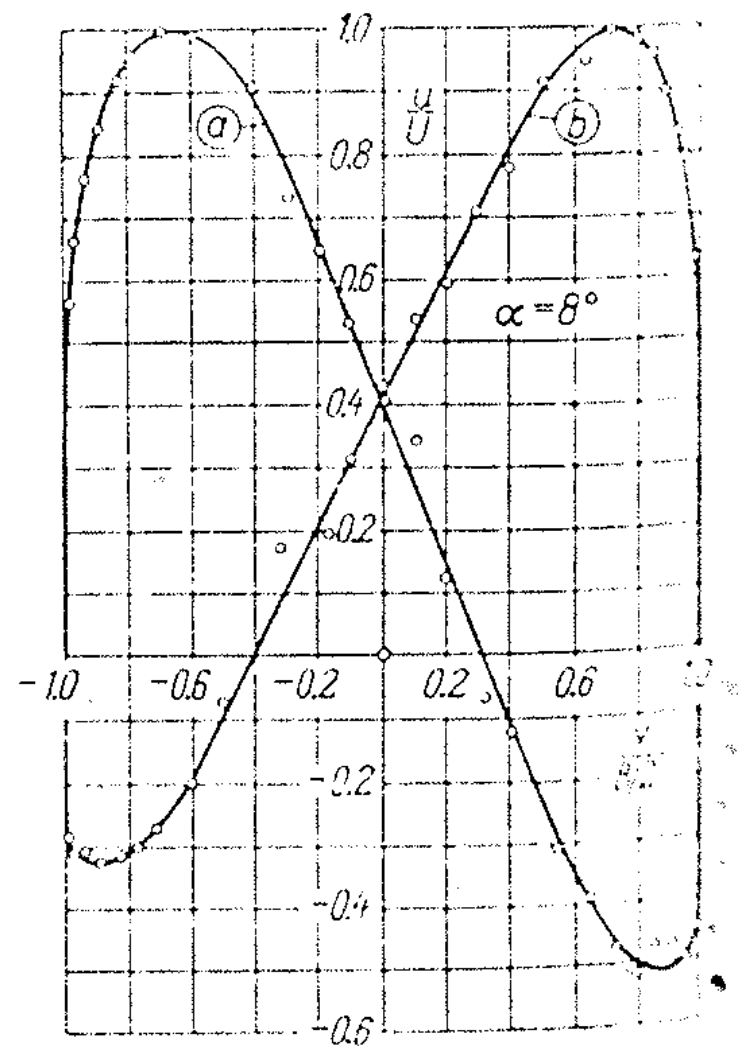
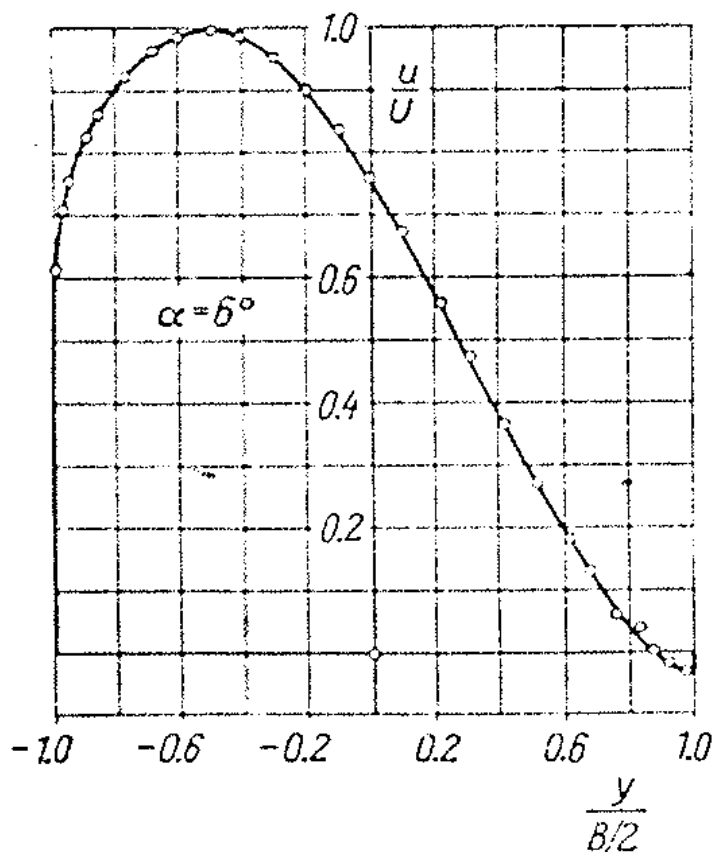
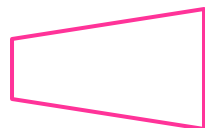
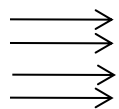


Velocity distribution in *convergent and divergent* channels with flat walls as measured by Nikuradse
 α — half included angle; B — width of channel



Velocity distribution in a *divergent* channel of half included angle $\alpha = 5^\circ$. The lack of symmetry in the velocity distribution signifies incipient separation

Computational Domain Selection: A Prior Insight in Flow Field



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)

Computational Domain Selection: Summary

- 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.

Geometry Clean-up in Numerical Simulations

The guidelines are very generic in nature and has been explained with examples. However, the users may need to check their software user manual to understand all the features available.

1. Why geometry cleaning is required?
2. Basic operations of geometry clean-up tools
3. Examples

What happens when geometry information is translated from CAD format to a neutral file format? The answer to this question will require a knowledge of not only the way CAD data are stored but also the universal way (read mathematics) to store the geometrical information such as lines and curves. The most natural way to make a design is to use "regular curves" such as straight lines, circle and arcs, ellipses. However when the same information is translated into a neutral format say IGES, they are recorded into a more universal way of representing the curves 'Spline' and 'NURBS'!

Geometry clean-up or defeaturing is an inevitable activity of most of the numerical simulations. This extra activity may be required to simplify the CAD geometry by deleting the spurious details from Numerical Simulation point of view or to compensate for the data loss that might arise during the data conversion from CAD-kernel to FE-Kernel. There are two international neutral formats widely recognized and used for data transfer from native CAD format namely IGES (International Graphics Exchange Standard - American standard) and STEP (STandard for Exchange of Product model data - a European Standard).

The understanding of mathematical description and storage of geometrical entities by CAD kernels and FE-kernels will help the two set of engineers to smoothen the process. But, it may not be completely eliminated. Lot of resources are being allocated by the software development vendors to minimize or eliminate the data loss that may arise during geometry data translation. This article is aimed at:

1. Compiling all nomenclature and geometric transformation available in contemporary CAD and FE GUIs

2. Developing a comparative summary of defeaturing methods available in various meshing softwares
3. Making a summary of mathematical operations being used in data creation, translation and their limitations
4. Preparing Best Practice Guidelines for Design Engineers, CAD Operators and FE Analysts to minimize the "Defeaturing" efforts.

The information collected below is in nascent stage and lot of efforts is being put to make it a comprehensive and reliable resource on the subject. Any comments and feedback are welcome!

Geometry Clean-up: Summary of Features

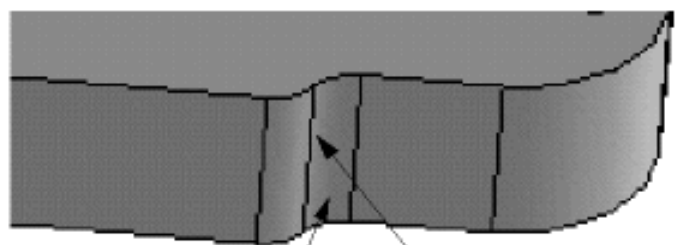
OPERATION/DESCRIPTION	ANSYS	HM	ICEM	GAMBIT
Shared Edge, attached to 2 surf.	GRN	GRN	RED	BLU
Non-manifold Edge	RED	YLO	BLU	RED
Suppressed Edge/Curve	x	BLU	Not Visible	x
Topology Correction Nomenclature	x	Geom Clean-up	Build Geometry	x
Line Collapse	√	x	X	√
Create Lines	√	√	√	√
Connect Vertices		√	√	√
Trim Curves	√	√	√	√
Split/Break Curves	√	√	√	√
Extend Curves	x	x	√	√
Bridge Curves	x	x	√	√
Join Curves	x	x	√	√
Equivalence	x	√	√	x
Edge Re-trim (ICEM)/Edge Toggle (HM)	√	x	√	x
Line Merge	√	√	√	√
Area Merge	√	√	√	√

Geometry Clean-up: Summary of Features

OPERATION/DESCRIPTION	ANSYS	HM	ICEM	GAMBIT
Area Collapse	√		x	√
Edge Fillets	x	√	x	x
Area Fill	√	√	√	√
Split/Trim Areas by Curve/Plane/3-Points	√	√	√	√
Non-manifold surfaces	x	x	√	x
Surface Fillets	x	x	x	x
Surface from Curves	√	x	√	√
Sweep Surface Along Curve	√	x	√	x
Sweep Surface Along Vector	√	x	√	x
Surface by Revolution	√	√	√	x
Loft Over Several Curve		√	√	x
Offset Surface	√	√	x	x
Extend Surface	√	√	√	√
Standard Shapes	√	x	√	√
Line Merge	√	√	√	√
Area Merge	√	√	√	√

Virtual Topology: A virtual entity doesn't have a geometric definition of its own, but is based on other entities. Virtual topological entities are (a) Surface Trim or split (b) Surface merge, etc. GAMBIT, FEMAP, HM supports this concept. ICEM does not follow this concept even though they support the operations of surface trim, merge, suppress, etc.

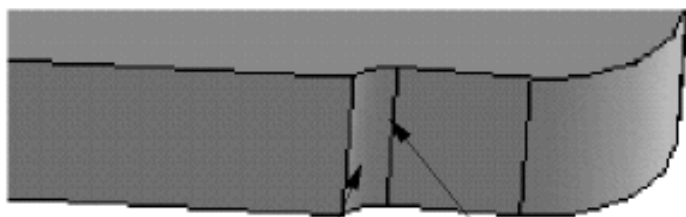
(a)



This area...

...collapses to this line

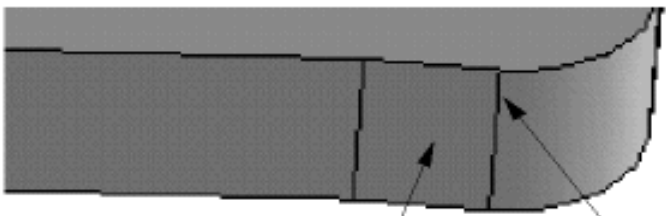
(b)



This area...

...collapses to this line

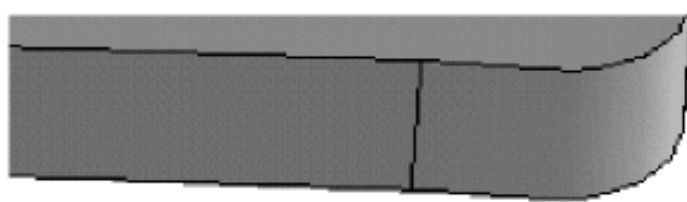
(c)

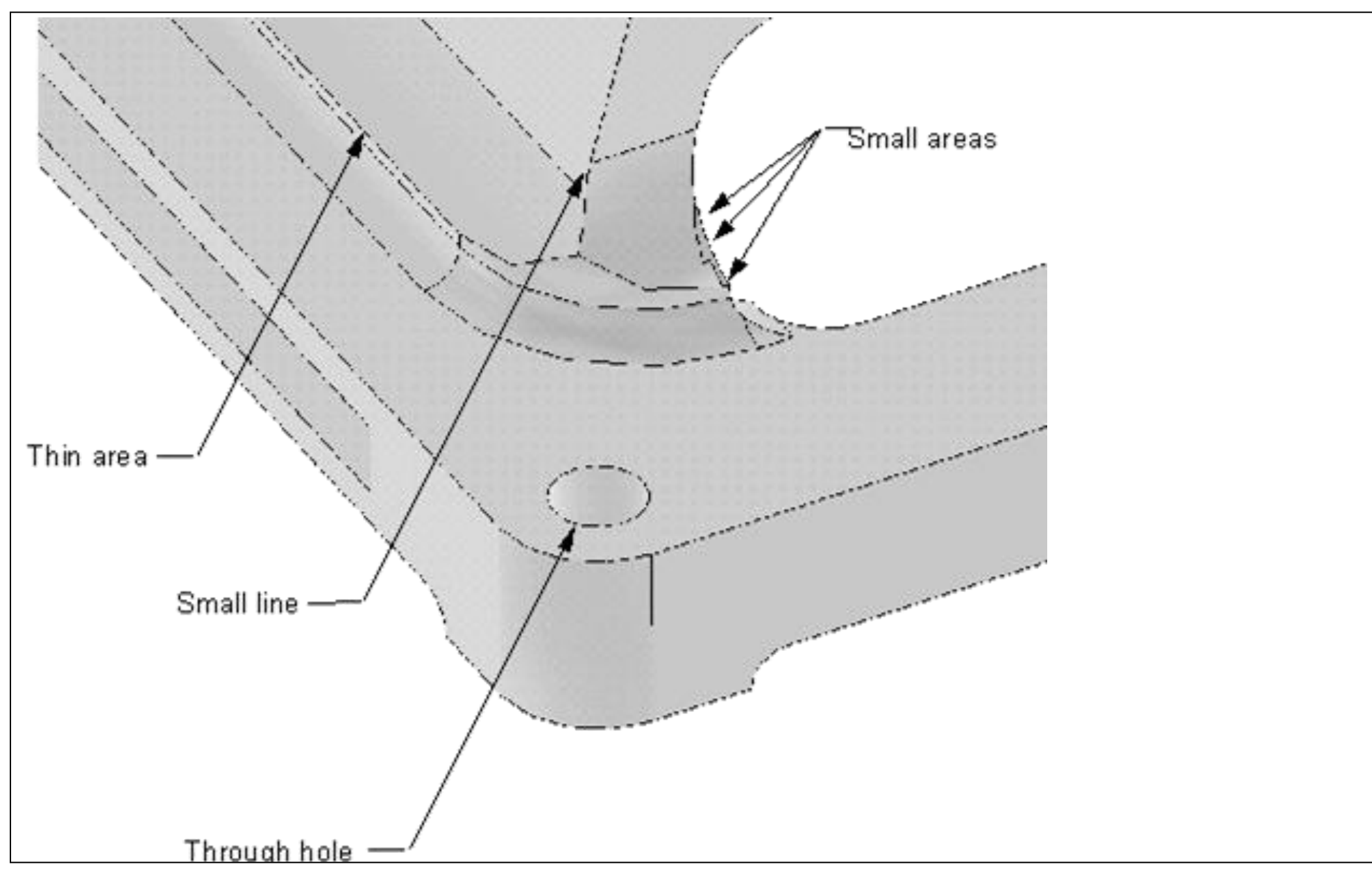


This area...

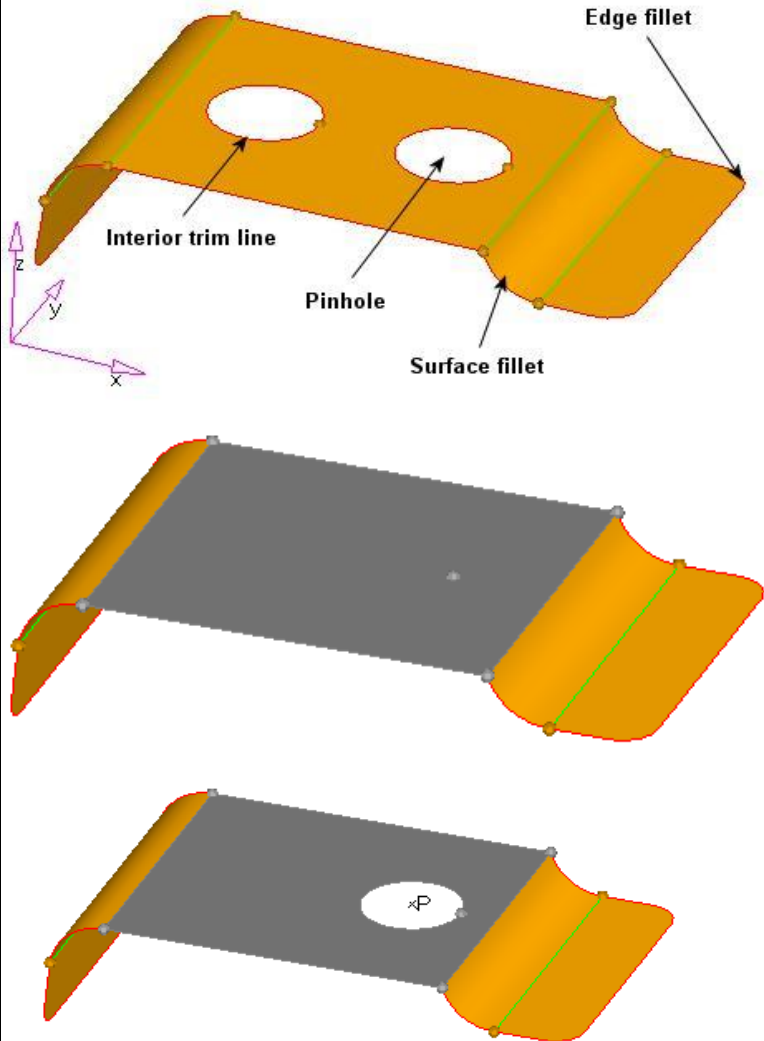
...collapses to this line

(d)

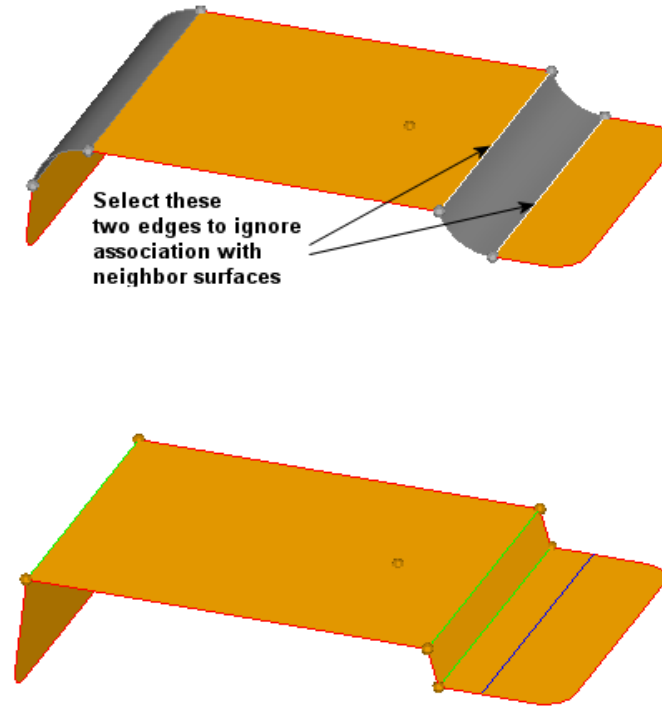




Remove Small Fillets



Remove Interior Holes



A powerful technique for finding problem areas is to use the auto-mesh panel in interactive mode to preview a mesh on the surfaces. Use your target element size and review areas with a node density of 1. These are an indication that the feature may be too small for the element size you plan on using. This helps identify short surface edges to be suppressed or any fixed points that should be combined.

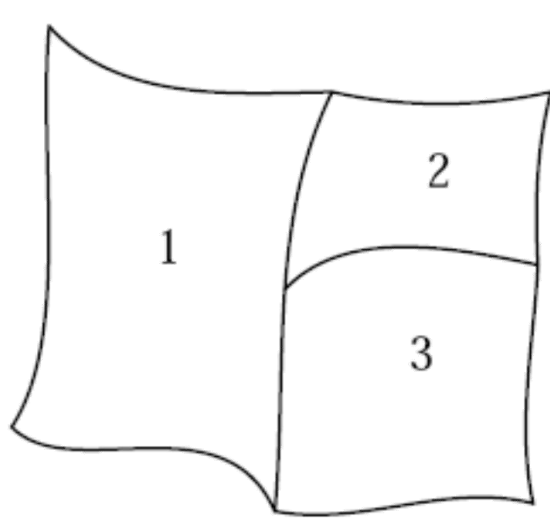
An incongruency problem means that two (or more) surfaces, appearing to have an edge in common, do not share same edge vertices.

1. Two points are coincident if they are separated by a distance equal to or less than the global model tolerance. If, upon creation, two points are coincident, they are considered the same and the second point is not created.
2. Two surfaces sharing the same edge are topologically congruent. Similarly, any two surfaces not separated by a distance greater than the global model tolerance are considered to be the same and only one is created.
3. Two solids sharing the same face are topologically congruent.
4. When you perform meshing, if the resulting mesh does not match at boundaries between two surfaces and they appear to be topologically congruent, it is probably because the global model tolerance was too small at the time the surfaces were created. To prevent this from occurring, we recommend a tolerance of 0.05% of the expected maximum model size. The correct tolerance may vary, however, based on the size of the smallest mesh element or geometric entity you plan to model. If gaps greater than the

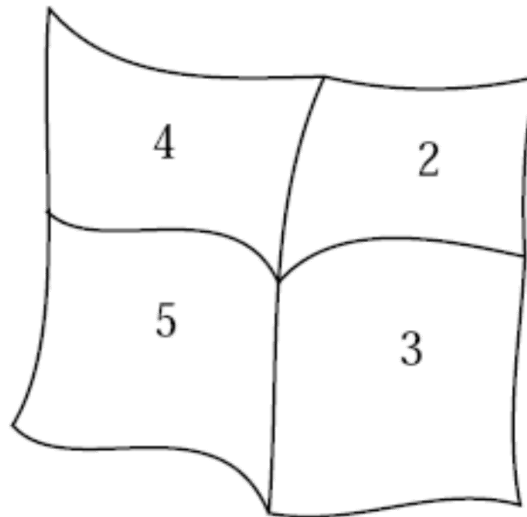
Geometry Clean-up: Topological Definition

tolerance exist where geometry should connect, you may need to increase the global model tolerance. Too large a tolerance value can result in problems with other parts of the model, so consider carefully before you increase the tolerance.

5. CFX-Build determines connectivity (topology) of the model during the creation phase, or via CAD access or import of geometry. Once connectivity is determined you cannot modify it unless you first delete the geometry, change the tolerance and then re-create the geometry.



Topologically Incongruent



Topologically Congruent

Basic Operation: Keyboard Short-cuts and Mouse Button Action

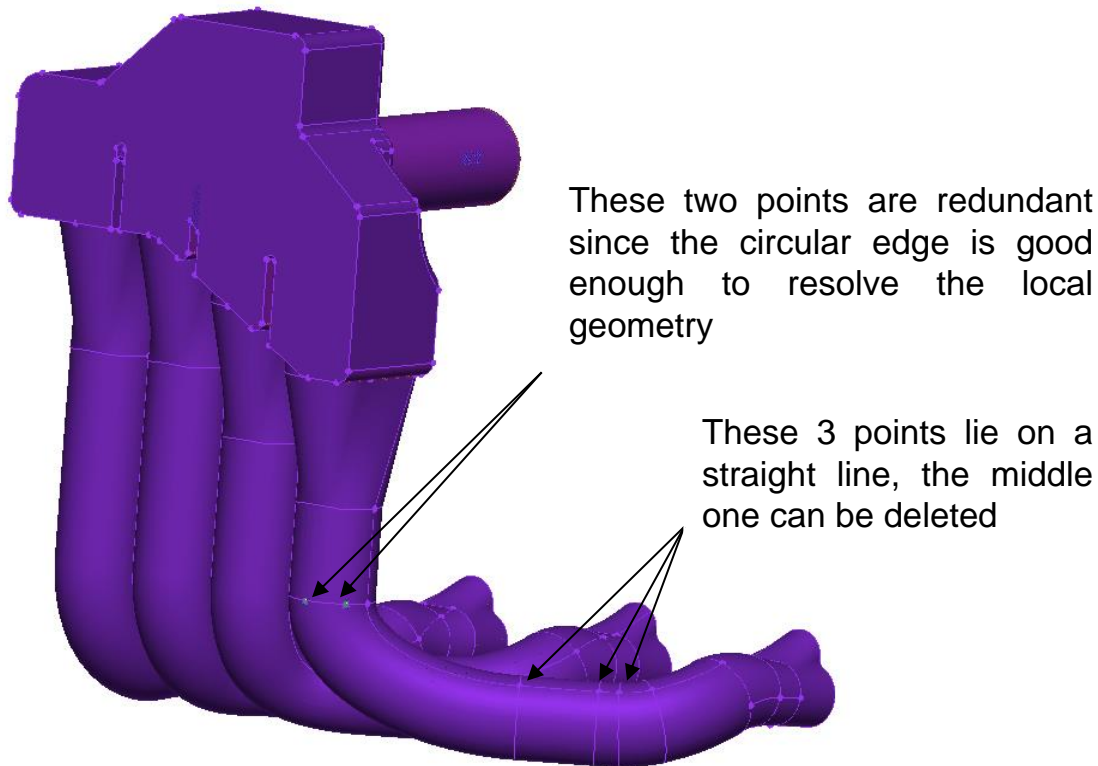
Operation Description	UG	HM	GAMBIT	ANSYS	ICEM	CFX-Build
Rotate	MB2	Arrow Keys, r, (a, MB1)	MB1	^+MB3	MB1	^Z, ^X, ^Y F2+MB2 F3+MB2
Pan	Shift+MB2	^+MB3	MB2	^+MB1	MB2	F4+MB2, Arrows
Zoom	^+MB2	(S, MB1), ^+MB2	MB3	^+MB2	MB3	z, Z, F5+MB2
Fit	^+F	f	GUI	GUI	GUI	^F
Window Zoom	MB1	^+MB2	^+MB2, ^+MB1	GUI	GUI	GUI
Refresh, Replot	F5	p	GUI	GUI	GUI	GUI
Select, Pick	MB1	MB1	Shift+MB1	MB1	MB1	MB1
Deselect, Unpick	Shift+MB1	Shift+MB1	Shift+MB2	GUI	MB3	MB3
Execute, Apply	MB2	MB2	Shift+MB3	MB2	MB2	MB2
Toggle Selection/Operation	--	b	MB2*MB2	MB3	GUI	---
Align to View	F8	--	--	--	Click on Axis	--
UNDO	^+Z	Not Available	GUI	Not Available	GUI	GUI
REDO	Not Available	Not Available	GUI	Not Available	GUI	GUI

Basic Operation: Keyboard Short-cuts and Mouse Button Action

Operation Description	UG	HM	GAMBIT	ANSYS	ICEM	CFX-Build
SAVE	^+S	GUI	GUI	GUI	GUI	GUI
FILE EXTENSION	*.prt	*.hm	*.dbs *.jou	*.db	*.tin, *.prj, *.uns,*.blk	*.db *.gtm
New Sesion	^+N	GUI	GUI	GUI	GUI	^+N
Open	^+O	GUI	GUI	GUI	GUI	^+O
Close	^+X	GUI	GUI	GUI	GUI	^+W
Revert/Resume	Not Available	GUI	Not Available	GUI	Not Available	Not Available
Rebuild	GUI	--	--	--	GUI	--
Quit/Exit	^+E	GUI	GUI	^+E	GUI	^+Q

Geometry Cleanup: Optimize Topology – Hard Points, Surface Edges

Delete hard points which are not required to guarantee sharp curvature of the computational geometry. For example, if there are three points on an straight edge, the middle one should always be deleted. This provides flexibility to the in-built smoothing algorithm.



Action	Geometry, Edit Mesh, Blocking
Undo	^+Z (Ctrl + Shift + z)
Redo	^+Y (Ctrl + Shift + y)
Scale to Fit	x
View-X	X =shift+x
View-Y	Y
View-Z	Z
View Reverse	R
View Isometric	I
Zoom	z
Build Topology	b
Delete (Make Dormant)	d
Delete Permanently	^+d (Ctrl + d)

Action	Edit Mesh /Blocking
Project Node/Vertice to Curve	p
Project Node/Vertice to Surface	^+p (Ctrl + p)
Project Node/Vertice to Point	P
Quality / Determinant	q
Smooth Mesh / Quality	^+q (Ctrl + q)
----- / Angle	Q

Geometry Cleanup: Entities defined by IGES Standards

ID	Form	Entity
100		Circular Arc
102		Composite Curve
104	0	General Arc
104	1	Ellipse
106		2D & 3D point set data
106	11	2D polygonal or line set data
106	22	3D polygonal or line set data
106	63	Closed 2D Curve
108	1	Bounded Plane
110	0	Independent Line Sets or Lines
112	0	Parametric B-Spline Curve
114	0	Parametric Spline Surface
116	0	Points
118	1	Rules Surface
120	0	Surface of Revolution

Geometry Cleanup: Entities defined by IGES Standards

ID	Form	Entity
122	0	Tabulated Surface
123	0	Direction
124	0	Transformational Matrix
125		Flash
126	0	Rational B-Spline Curve
128	0	Rational B-Spline Surface
130		Offset Curve
140	0	Offset Surface
141	0	Boundary of a Bounded Plane
142	0	Curve on a Parametric Surface
143	0	Bounded Surface
144	0	Trimmed Surface
150~		CSG Entities (150 ~ 169)
158		Sphere Entity
180~		Manifold Solid B-rep Entities (180 ~ 186)

Geometry Cleanup: Entities defined by IGES Standards

ID	Form	Entity
190		Plane Surface Entity
192		Right Circular Cylindrical Surface Entity
194		Right Circular Conical Surface Entity
196		Spherical Surface Entity
198		Toroidal Surface Entity
304		Line Font Definition
406		Name Associated with an Entity or Group of Entity
408		Singular Subfigure Instance
504		Edge Entity
508		Loop Entity
510		Face Entity
514		Shell Entity