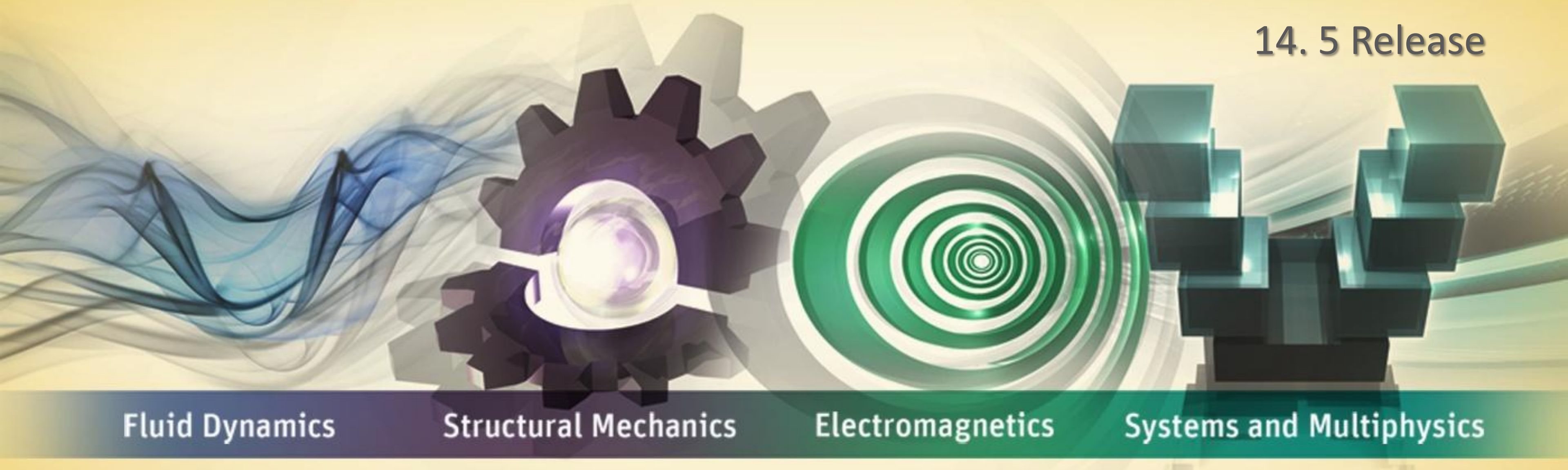
ANSYS

Realize Your Product Promise™

Lecture 8
Mesh Quality

14. 5 Release



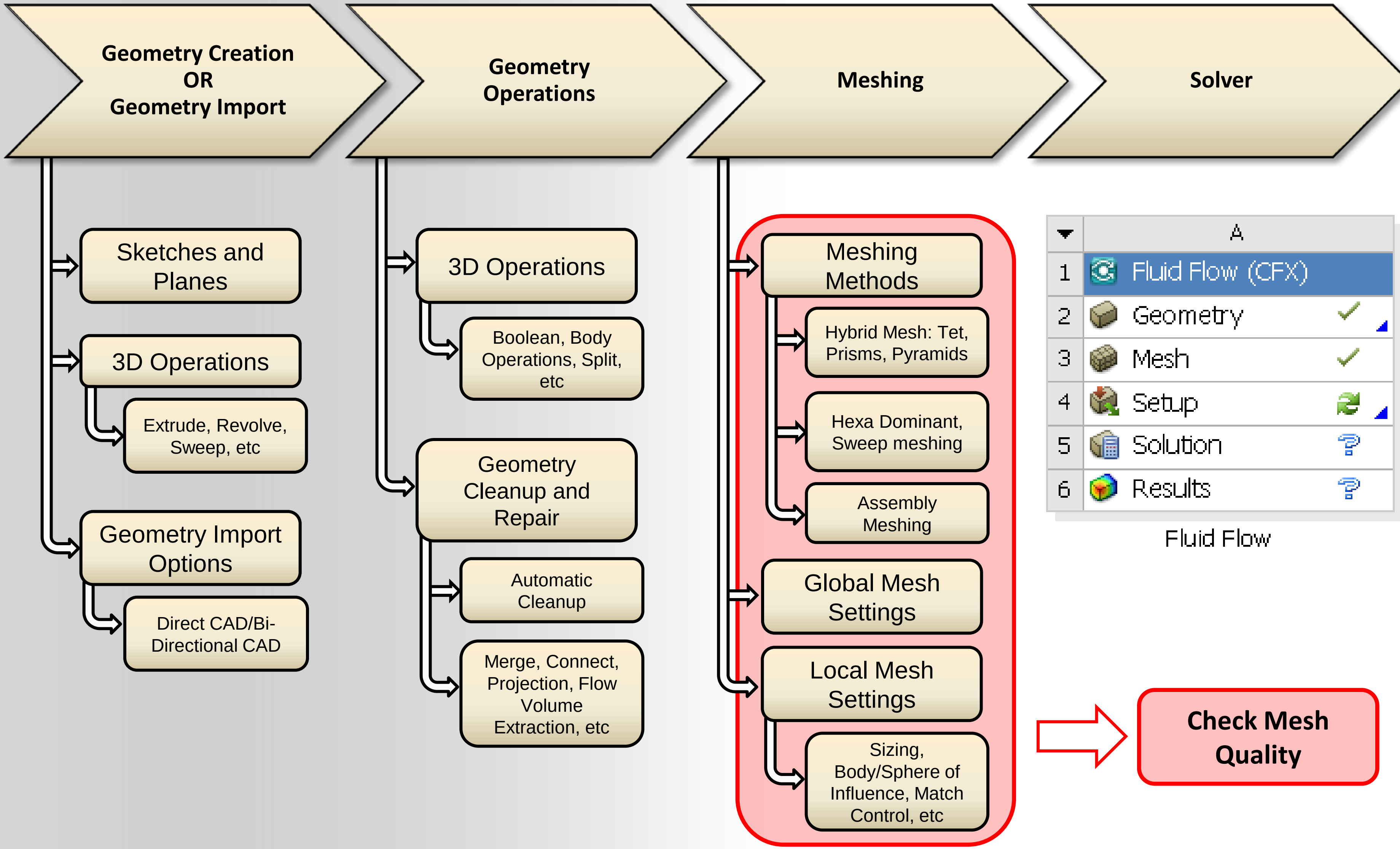
Introduction to ANSYS Meshing

Mesh Quality

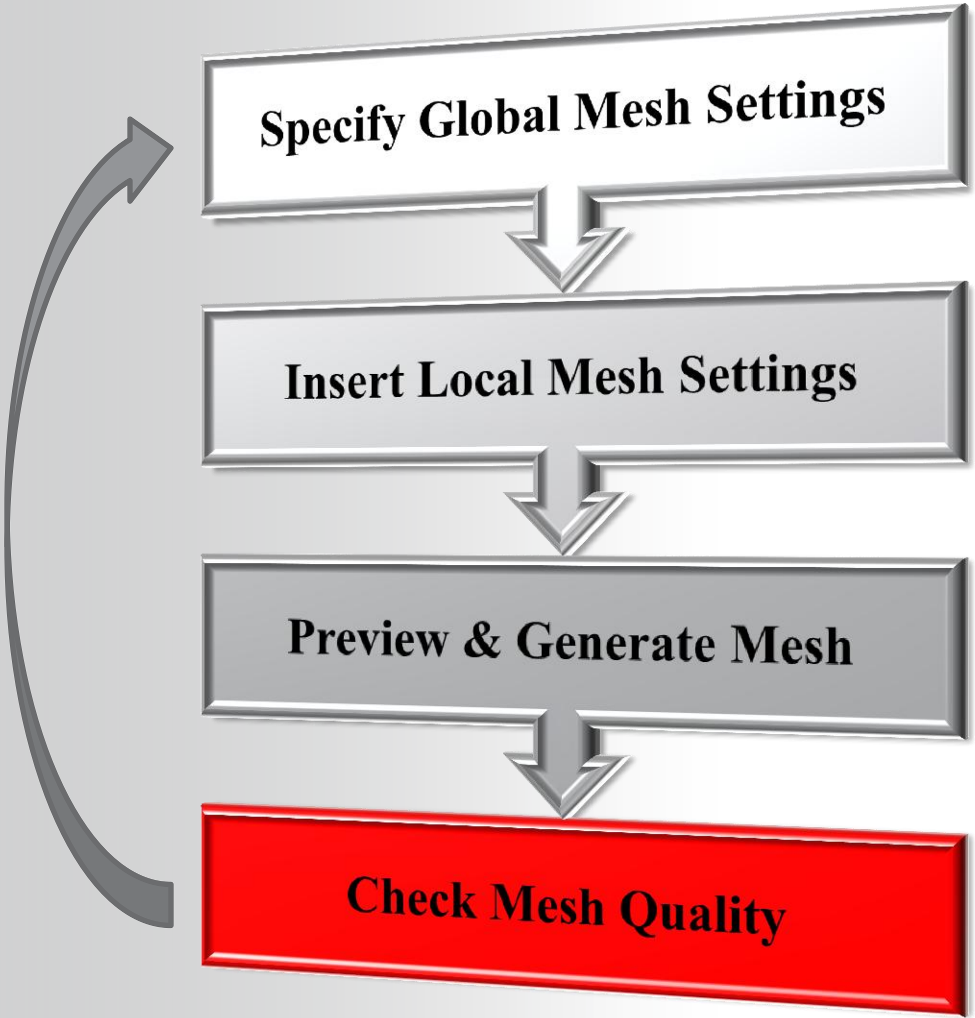
What you will learn from this presentation

- Impact of the Mesh Quality on the Solution
- Quality criteria
- Methods for checking the mesh quality
- Tools to improve quality in Meshing
- Pinch
- Virtual topology

Preprocessing Workflow



Meshing Process in ANSYS Meshing



Impact of the Mesh Quality

Good quality mesh means that…

- Mesh quality criteria are within correct range
  - Orthogonal quality …
- Mesh is valid for studied physics
  - Boundary layer …
- Solution is grid independent
- Important geometric details are well captured

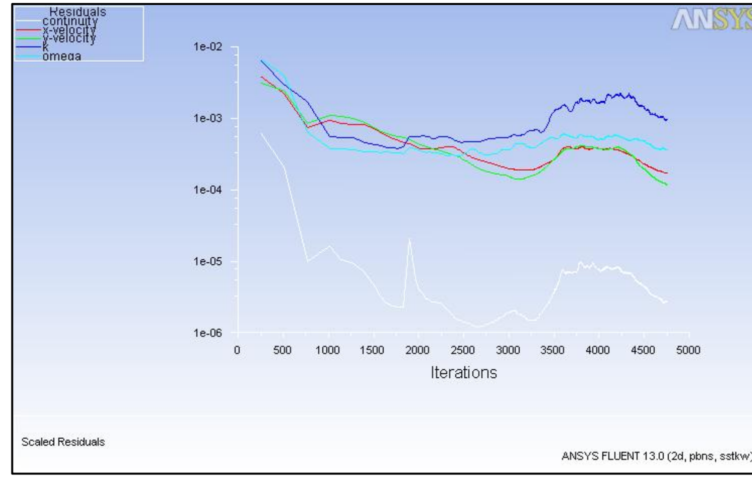


Table of Design Points

	A	B	C	D	E	F
1	Name	P1 - Sweep Method 3 Sweep Element Size	P2 - Sweep Method 2 Sweep Element Size	P3 - Sweep Method Sweep Element Size	P4 - Face Sizing Element Size	P6 - Dp
2		m	m	m	m	Pa
3	Current	0.04	0.04	0.04	0.02	747.88
4	DP 1	0.02	0.02	0.02	0.01	500.44
5	DP 2	0.01	0.01	0.01	0.005	361.4
6	DP 3	0.005	0.005	0.005	0.0025	307.6
7	DP 4	0.0025	0.0025	0.0025	0.00125	299.86
*						

Bad quality mesh can cause;

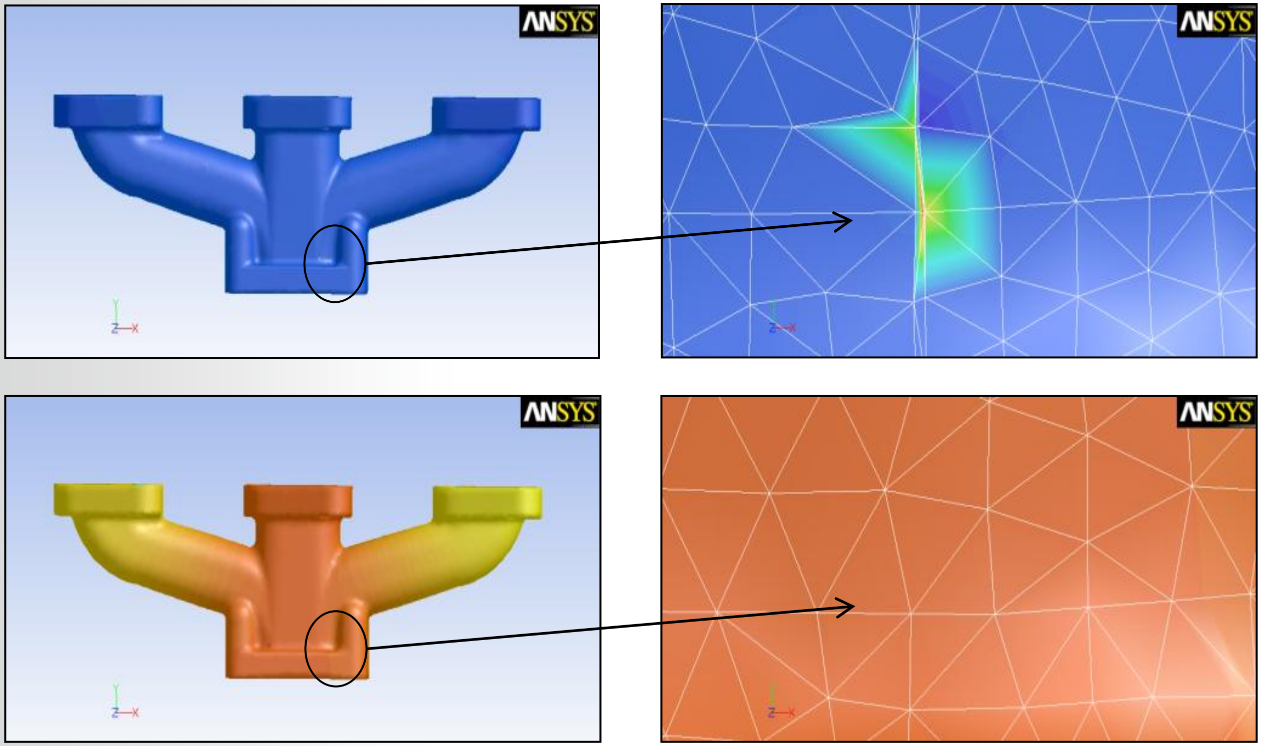
- Convergence difficulties
- Bad physic description
- Diffuse solution

User must…

- Check quality criteria and improve grid if needed
- Think about model and solver settings before generating the grid
- Perform mesh parametric study, mesh adaption …

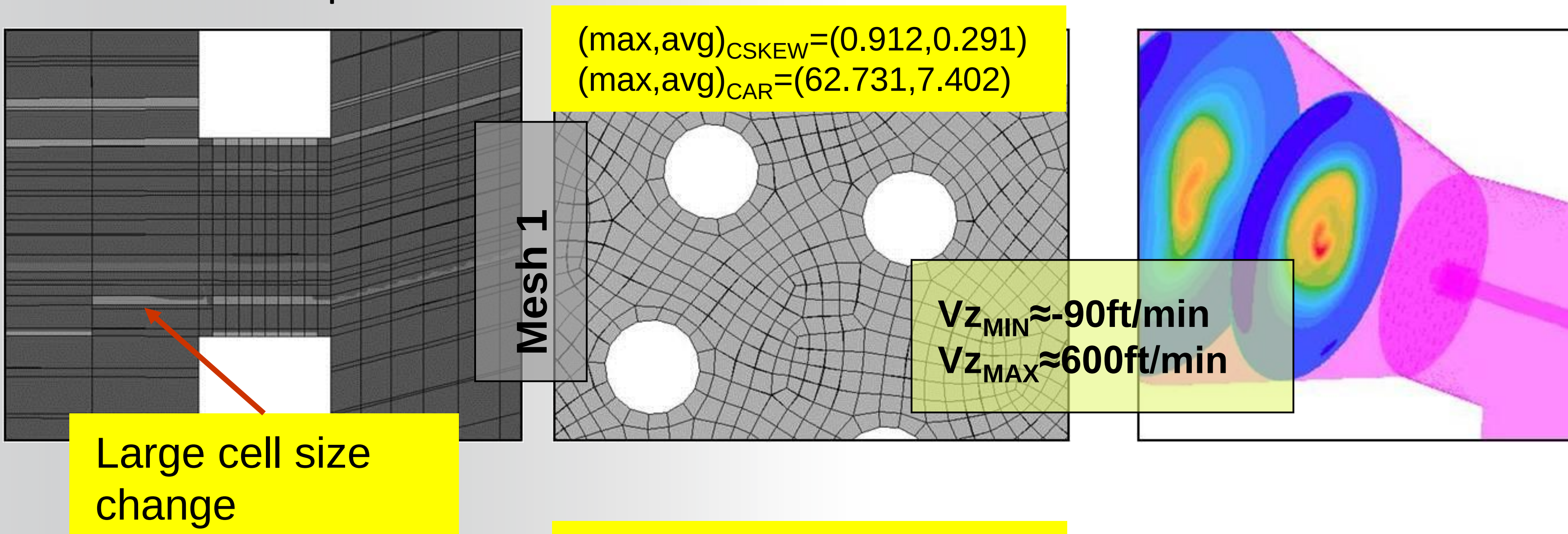
Impact of the Mesh Quality on the Solution

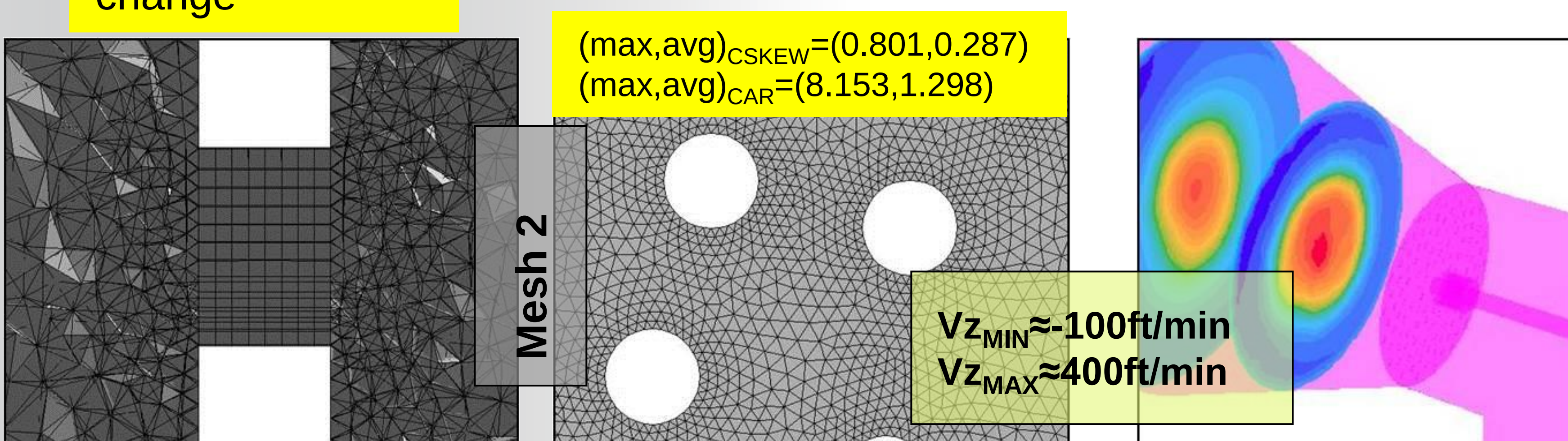
- Example showing difference between a mesh with cells failing the quality criteria and a good mesh
- Unphysical values in vicinity of poor quality cells



Impact of the Mesh Quality on the Solution

- Diffusion example



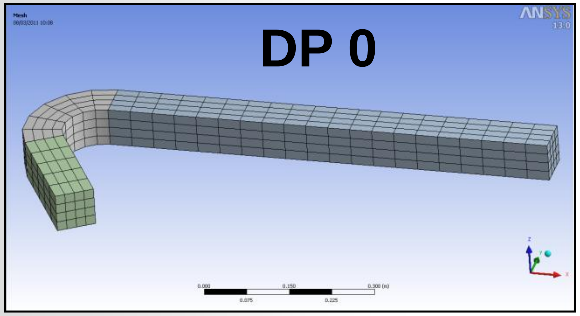


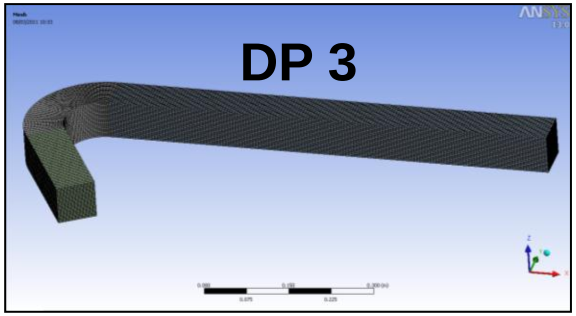
Grid Dependency

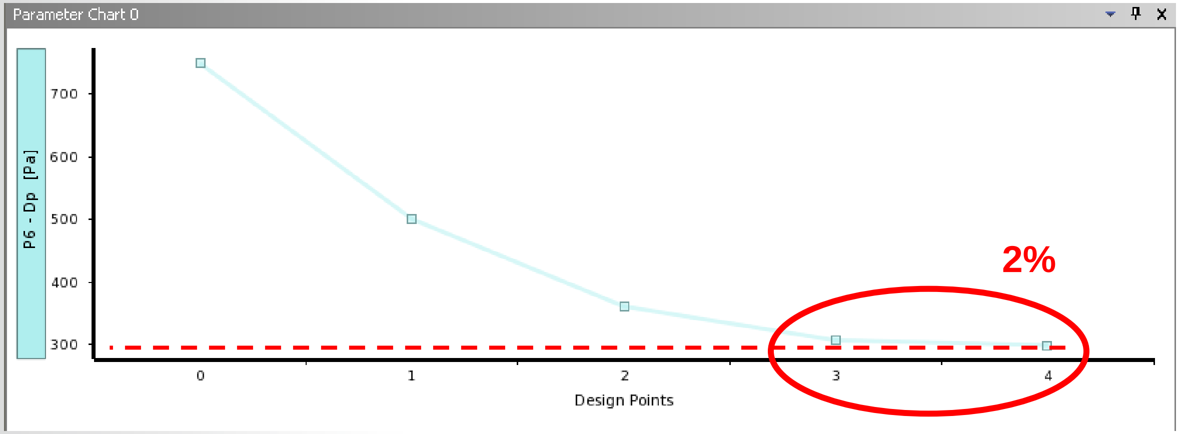
	DP 0	DP 1	DP 2	DP 3	DP 4
Nb Cells	500	3 000	24 000	190 000	1.5 M

x8

- **Solution run with multiple meshes**
- **Note : For all runs the computed Y+ is valid for wall function (first cell not in laminar zone)**

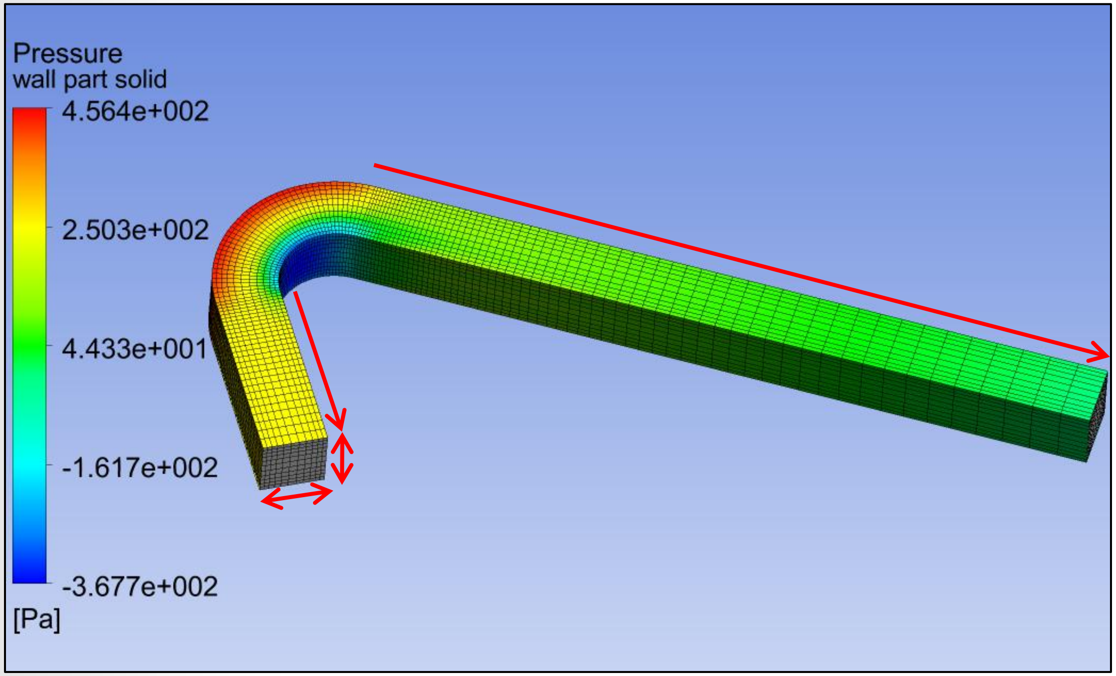






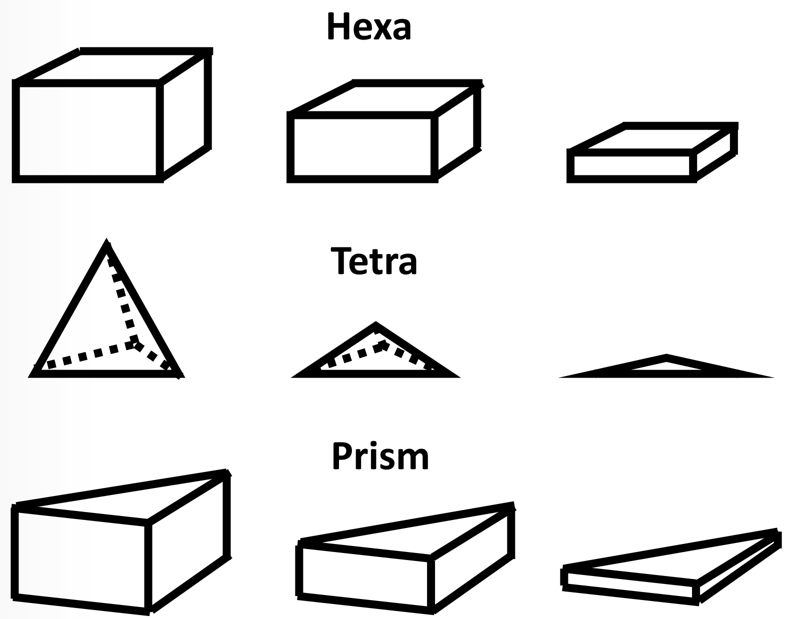
Grid Dependency

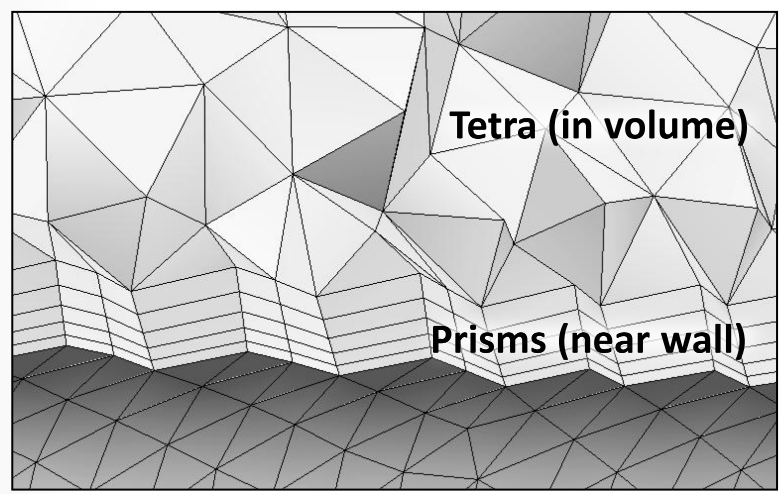
- **Hexa cells can be stretched in stream direction to reduce number of cells**
- **Bias defined on inlet and outlet walls**
- **Bias defined on inlet edges**
  - **16 000 cells (~DP2)**
  - **Delta P = 310 Pa (~DP3)**



Hexa vs. Tetra

- **Hexa: Concentration in one direction**
  - Angles unchanged
- **Tetra: Concentration in one direction**
  - Angles change
- **Prism: Concentration in one direction**
  - Angles unchanged
- **Solution for boundary layer resolution**
  - Hybrid prism/tetra meshes
  - Prism in near-wall region, tetra in volume
  - Automated
  - Reduced CPU-time for good boundary layer resolution





Mesh Statistics and Mesh Metrics

Displays mesh information for Nodes and Elements

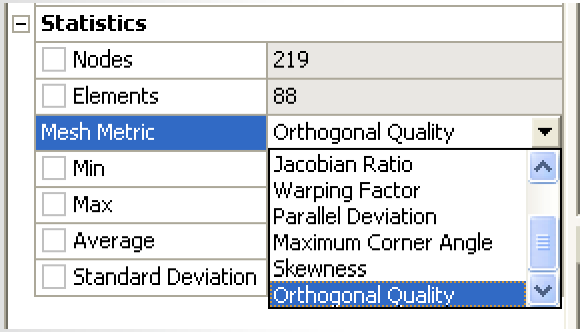
List of quality criteria for the Mesh Metric

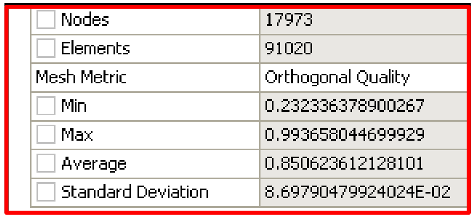
- **Select the required criteria to get details for quality**
- **It shows minimum, maximum, average and standard deviation**

Different physics and different solvers have different requirements for mesh quality

Mesh metrics available in ANSYS Meshing include:

- **Element Quality**
- **Aspect Ratio**
- **Jacobean Ration**
- **Warping Factor**
- **Parallel Deviation**
- **Maximum Corner Angle**
- **Skewness**
- **Orthogonal Quality**





For Multi-Body Parts, go to corresponding body in Tree Outline to get its separate mesh statistics per part/body

Mesh Quality Metrics

Orthogonal Quality (OQ)

Derived directly from

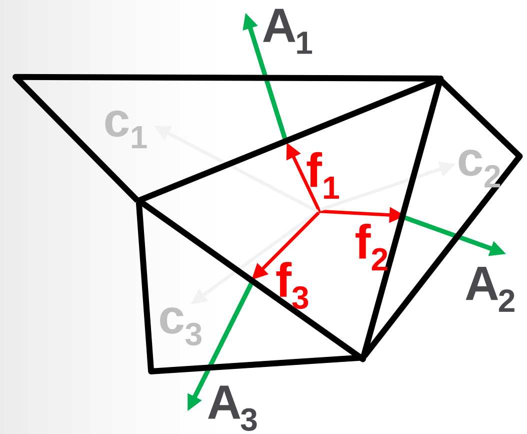
Fluent solver discretization

- ***For a cell*** **it is the minimum of:**

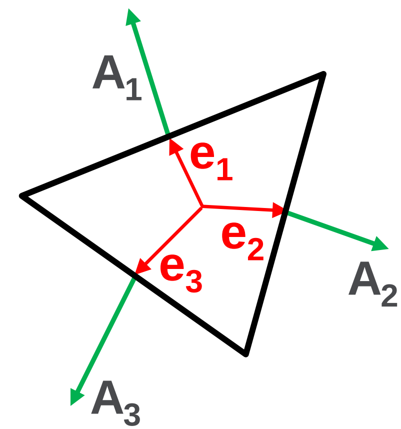
$$\frac{\vec{A_i}\cdot\vec{f_i}}{|\vec{A_i}||\vec{f_i}|} \qquad \frac{\vec{A_i}\cdot\vec{c_i}}{|\vec{A_i}||\vec{c_i}|}$$

computed for each face *i*

On cell



On face



For the face **it is computed as the minimum of** $\frac{\vec{A_i}\cdot\vec{e_i}}{|\vec{A_i}||\vec{e_i}|}$ **computed for each edge *I***

Where A_i is the face normal vector and f_i is a vector from the centroid of the cell to the centroid of that face, and c_i is a vector from the centroid of the cell to the centroid of the adjacent cell, where e_i is the vector from the centroid of the face to the centroid of the edge

At boundaries and internal walls

c_i is ignored in the computations of OQ

0 Worst

1 Perfect

Mesh Quality Metrics

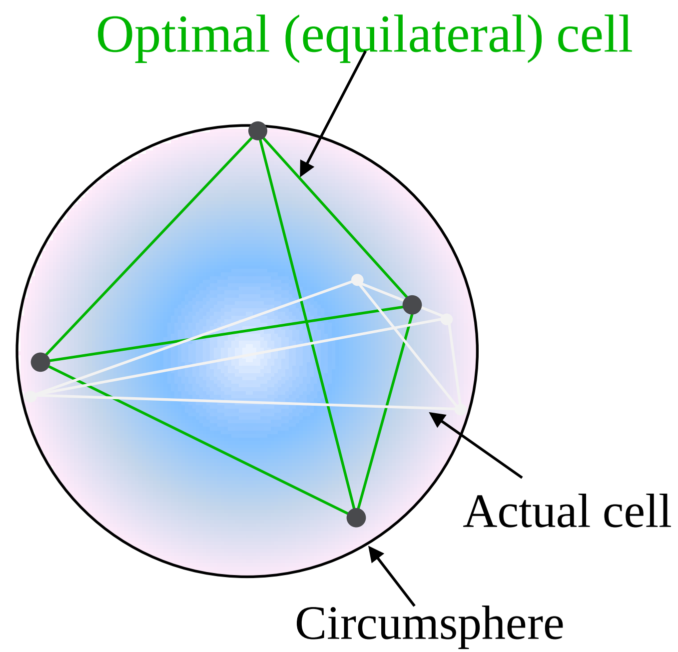
Skewness

Two methods for determining skewness:

1. Equilateral Volume deviation:

$$\text{Skewness} = \frac{\text{optimal cell size} - \text{cell size}}{\text{optimal cell size}}$$

Applies only for triangles and tetrahedrons



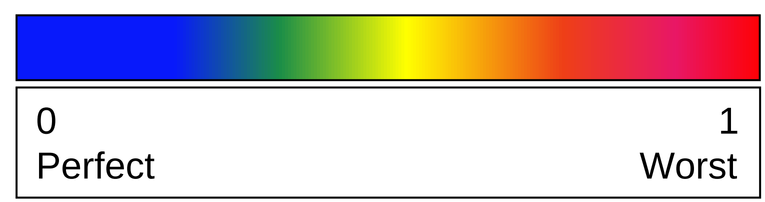
2. Normalized Angle deviation:

$$\text{Skewness} = \max\left[\frac{\theta_{max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{min}}{\theta_e}\right]$$

Where θ_e is the equiangular face/cell (60 for tets and tris, and 90 for quads and hexas)

– Applies to all cell and face shapes

– Used for hexa, prisms and pyramids



Mesh Quality

Mesh quality recommendations

Low Orthogonal Quality or high skewness values are not recommended

Generally try to keep minimum orthogonal quality > 0.1, or maximum skewness < 0.95. However these values may be different depending on the physics and the location of the cell

Fluent reports negative cell volumes **if the mesh contains degenerate cells**

Skewness mesh metrics spectrum

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

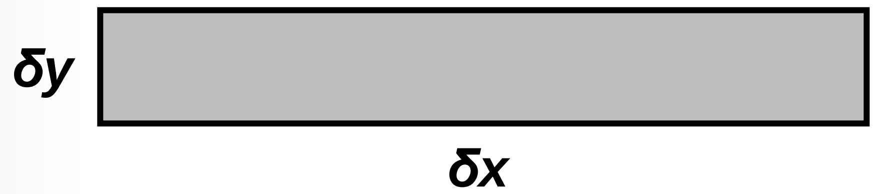
Orthogonal Quality mesh metrics spectrum

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

Aspect Ratio

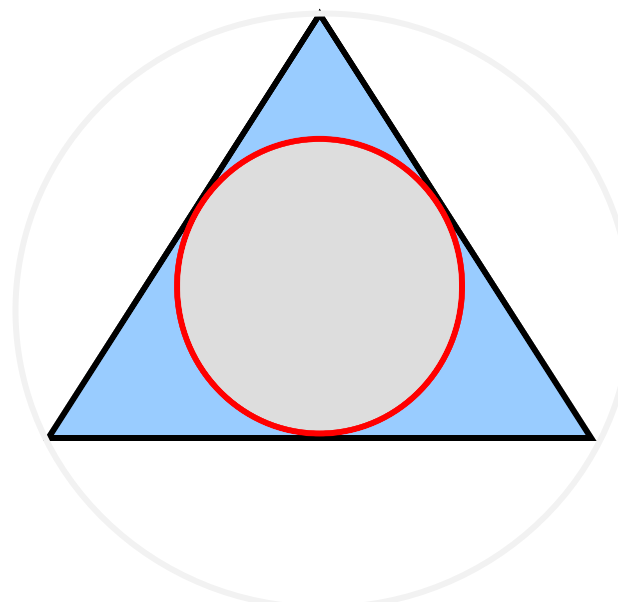
2-D:

- Length / height ratio: $\delta x/\delta y$



3-D

- Area ratio
- Radius ratio of circumscribed / inscribed circle



Limitation for some iterative solvers

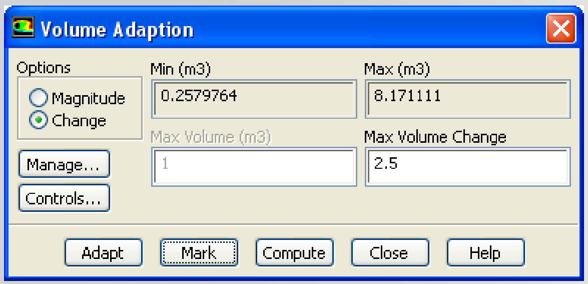
- A < 10 … 100
- (CFX: < 1000)

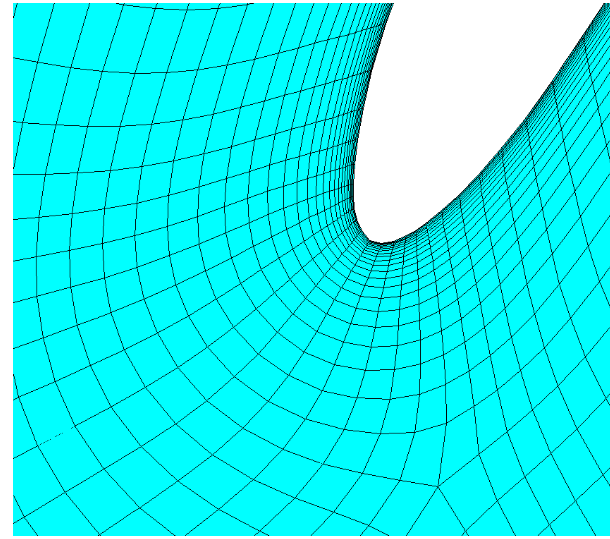
Large aspect ratio are accepted where there is no strong transverse gradient (boundary layer ...)

Smoothness

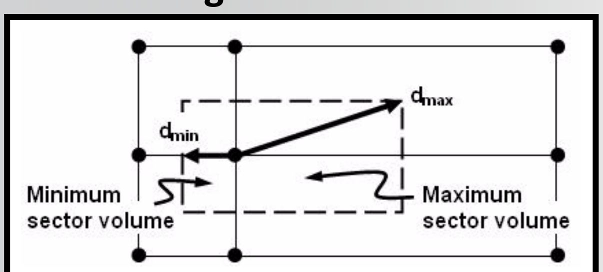
Checked in solver

- **Volume Change in Fluent**
  - Available in Adapt/Volume
  - *3D :* $\sigma_i = V_i / V_{nb}$





- **Expansion Factor in CFX**
  - Checked during mesh import
  - Ratio of largest to smallest element volumes surrounding a node



Recommendation:

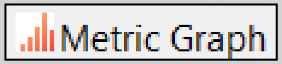
Good: 1.0 < σ < 1.5

Fair: 1.5 < σ < 2.5

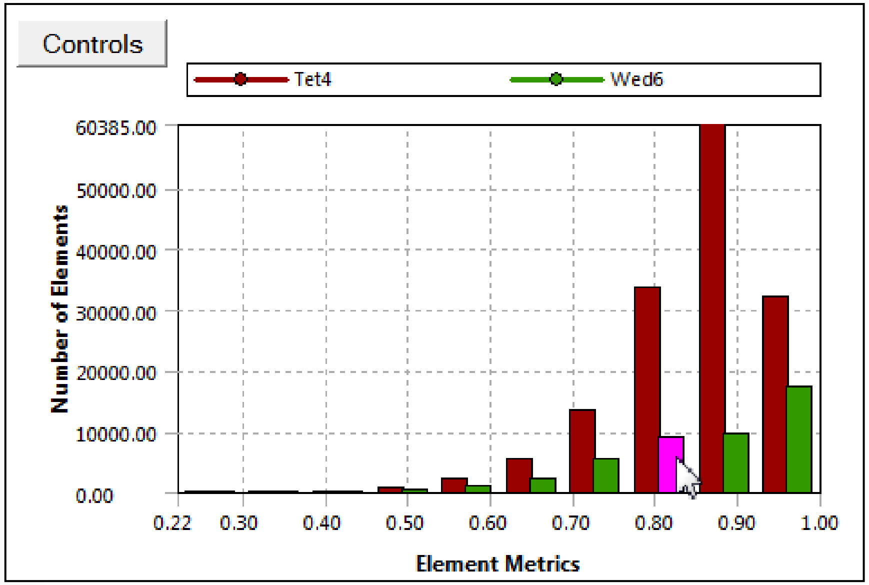
Poor: σ > 5 … 20

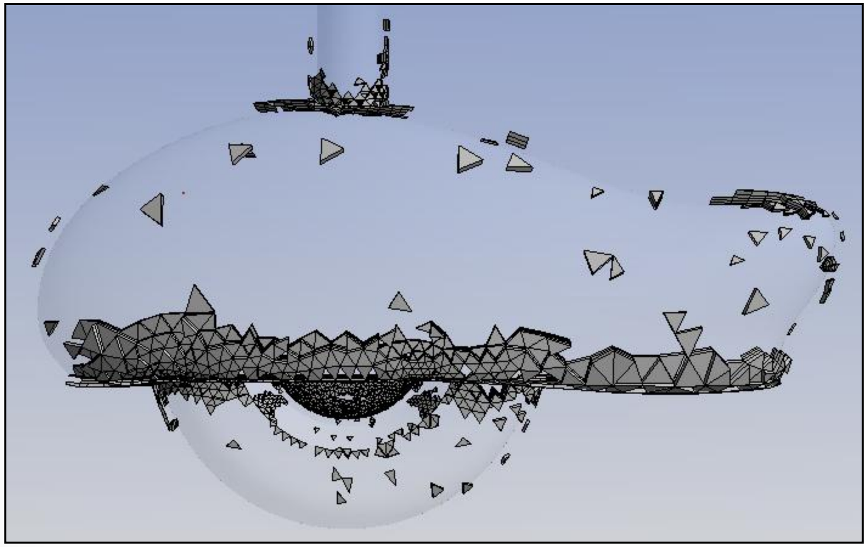
Mesh Metric Graph

- **Displays Mesh Metrics graph for the element quality distribution**
- **Different element types are plotted with different color bars**
- **Can be accessed through menu bar using Metric Graph button**

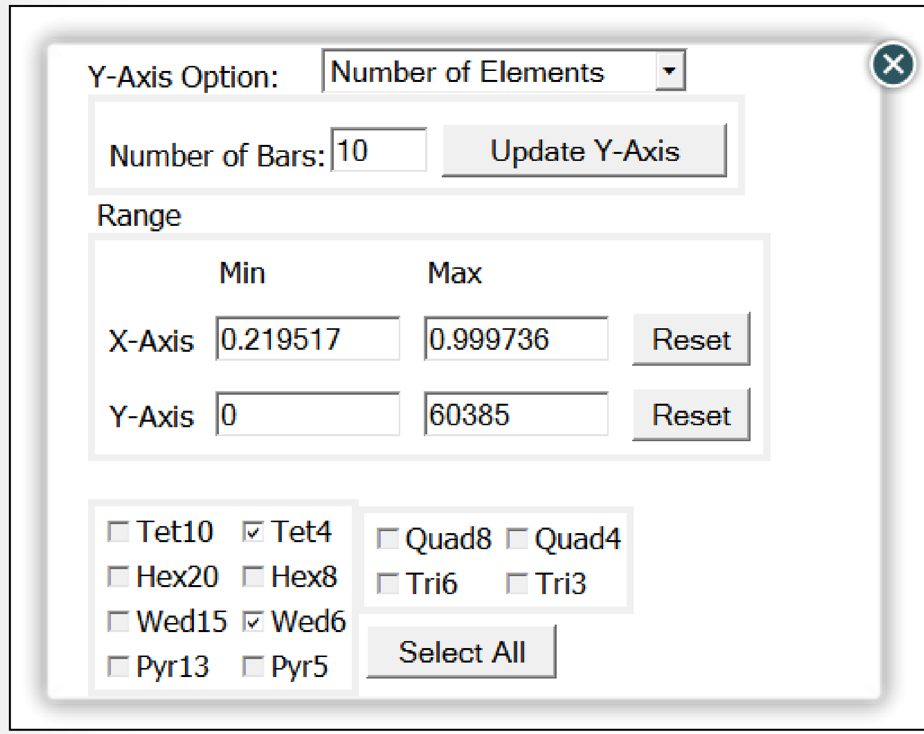
- **Axis range can be adjusted using controls button (details next slide)**
- **Click on bars to view corresponding elements in the graphics window**
  - Use to help locate poor quality elements





Mesh Metric Graph Controls

- **Elements on Y-Axis can be plotted with two methods;**
  - Number of Elements
  - Percentage of Volume/Area
- **Options to change the range on either axis**
- **Specify which element types to include in graph**
  - Tet4 = 4 Node Linear Tetrahedron
  - Hex8 = 8 Node Linear Hexahedron
  - Wed6 = 6 Node Linear Wedge (Prism)
  - Pyr5 = 5 Node Linear Pyramid
  - Quad4 = 4 Node Linear Quadrilateral
  - Tri3 = 3 Node Linear Triangle
    - Te10, Hex20, Wed15, Pyr13, Quad8 & Tri6 non-linear elements



Section Planes

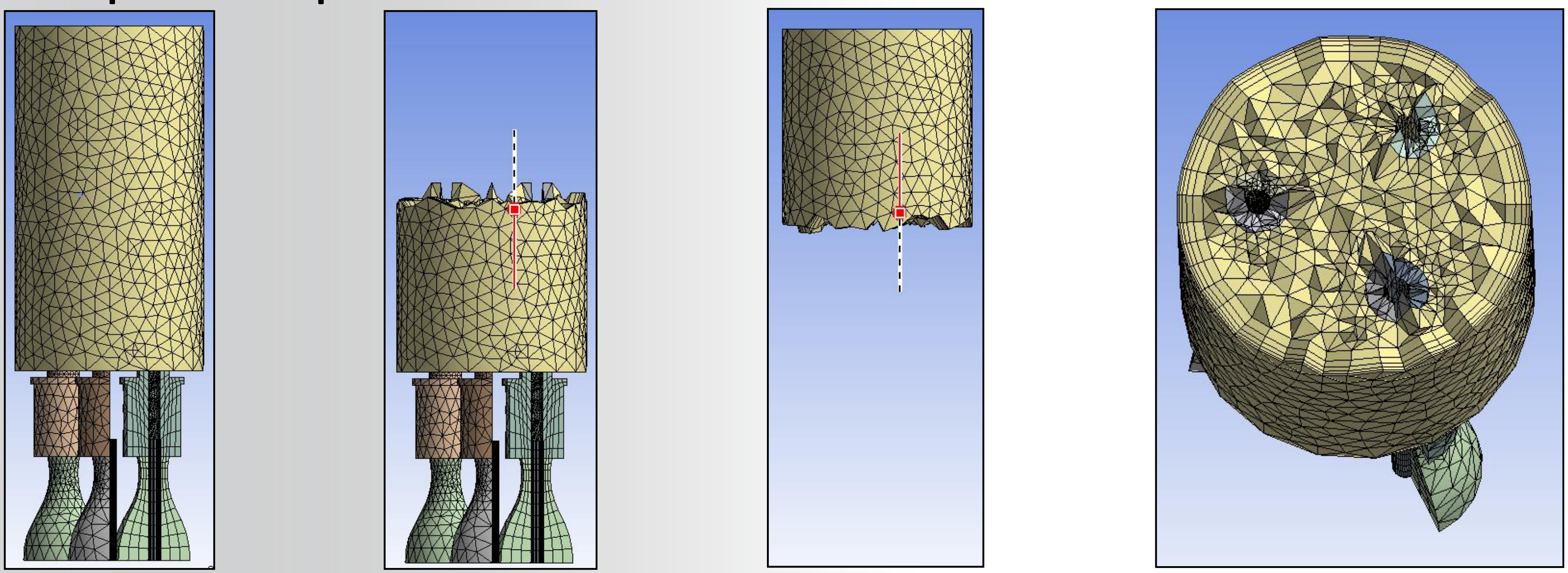
Displays internal elements of the mesh

- **Elements on either side of plane can be displayed**
- **Toggle between cut or whole elements display**
- **Elements on the plane**

Edit Section Plane button can be used to drag section plane to new location

- **Clicking on "Edit Section Plane" button will make section plane's anchor to appear**

Multiple section planes are allowed



For large meshes, it is advisable to switch to geometry mode (click on geometry in the Tree Outline), create the section plane and then go back to mesh model

Mesh Quality Check for CFX

- The CFX solver calculates 3 important measures of mesh quality at the start of a run and updates them each time the mesh is deformed
- Mesh Orthogonality
- Aspect Ratio
- Expansion Factor

```
+--------------------------------------------------------------------+
|                          Mesh Statistics                           |
+--------------------------------------------------------------------+
 Domain Name: Air Duct
     Minimum Orthogonality Angle [degrees]                =     20.4 ok
     Maximum Aspect Ratio                                 =     13.5 OK
     Maximum Mesh Expansion Factor                        =    700.4  !
 Domain Name: Water Pipe
     Minimum Orthogonality Angle [degrees]                =     32.8 ok
     Maximum Aspect Ratio                                 =      6.4 OK
     Maximum Mesh Expansion Factor                        =     73.5  !
Global Mesh Quality Statistics :
     Minimum Orthogonality Angle [degrees]                =     20.4 ok
     Maximum Aspect Ratio                                 =     13.5 OK
     Maximum Mesh Expansion Factor                        =    700.4  !
```

Good
(OK)

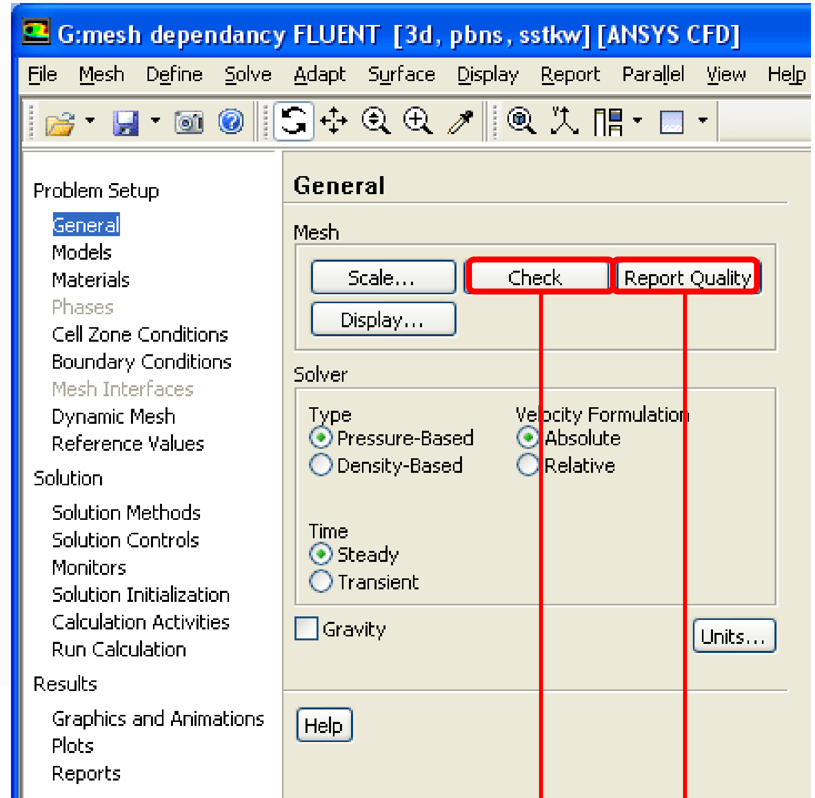
Acceptable
(ok)

Questionable
(!)

Mesh Quality Check for Fluent

Grid check tools available

- ***Check*** : Perform various mesh consistency checks
- ***Report Quality*** : lists worse values of orthogonal quality and aspect ratio
- TUI command ***mesh/check-verbosity*** sets the level of details in the report




```
 Domain Extents:
   x-coordinate: min (m) = -1.349580e-01, max (m) = 8.000000e-01
   y-coordinate: min (m) = -2.407051e-01, max (m) = 1.350000e-01
   z-coordinate: min (m) = -3.500000e-02, max (m) = 3.500000e-02
 Volume statistics:
   minimum volume (m3): 2.067421e-08
   maximum volume (m3): 3.187442e-07
     total volume (m3): 5.925829e-03
 Face area statistics:
   minimum face area (m2): 6.187846e-06
   maximum face area (m2): 1.274684e-04
 Checking mesh.........................
Done.
```

```
Mesh Quality:
 Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
Minimum Orthogonal Quality =   9.99641e-01
Maximum Aspect Ratio =  2.03929e+01
```

Factors Affecting Quality

Geometry problems

- Small edge
- Gaps
- Sharp angle

⇨ **Geometry cleanup in Design Modeler**
or
Virtual topology & pinch in Meshing

Meshing parameters

- Sizing Function On / Off
- Min size too large
- Inflation parameters
  – Total height
  – Maximum angle
- Hard sizing

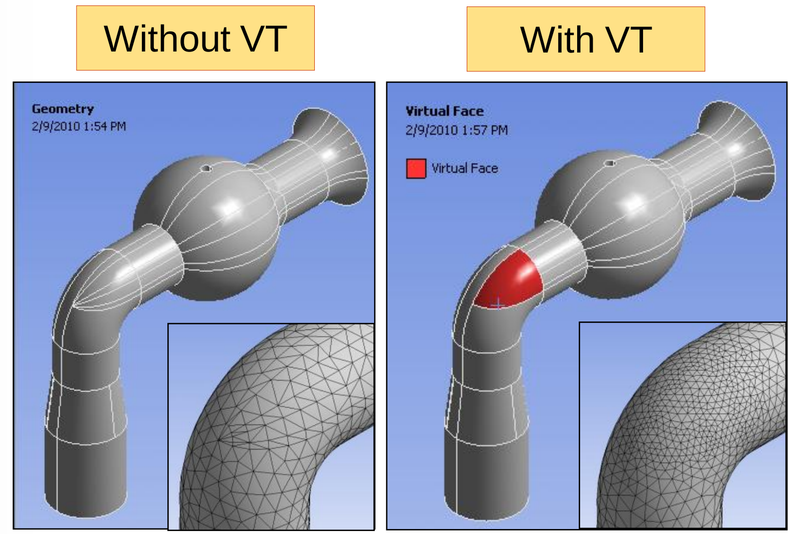
⇨ **Mesh setting change**

Meshing methods

- Patch conformal or patch independent tetra
- Sweep or Multizone
- Cutcell

⇨ **Mesh setting change**

Virtual Topology



When to use?

- To merge together a number of small (connected) faces/edges
- To simplify small features in the model
- To simplify load abstraction for mechanical analysis
- To create edge splits for better control of the surface mesh control

Virtual cells modify topology

- Original CAD model remains unchanged
- New faceted geometry is created with virtual topology

Restrictions

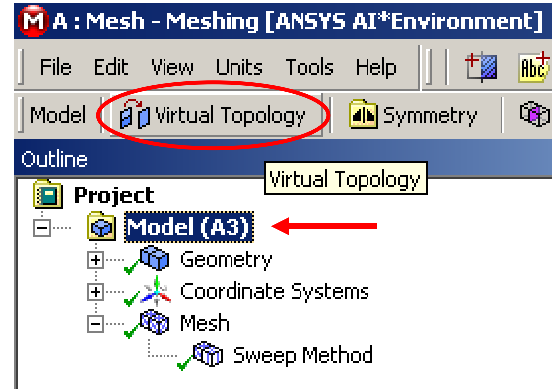
- Limited to “developable” surfaces
- Virtual Faces cannot form a closed region

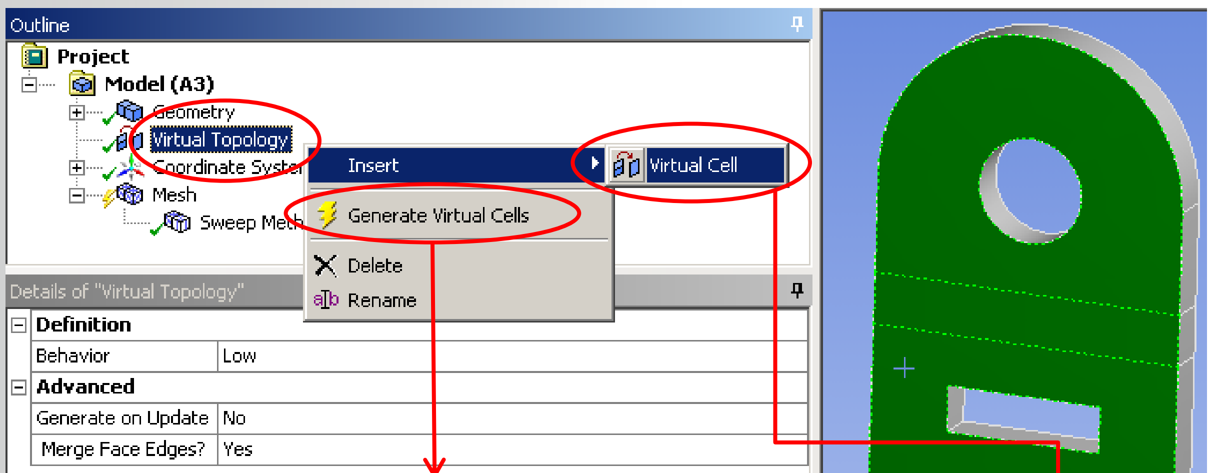
Details of "Virtual Topology"

Definition	
Method	Automatic
Behavior	Low
Advanced	
Generate on Update	No
Merge Face Edges	Yes
Lock position of dependent edge splits	Yes
Statistics	
Virtual Faces	1
Virtual Edges	0
Virtual Split Edges	0
Virtual Split Faces	0
Virtual Hard Vertices	0
Total Virtual Entities	1

Creating Virtual Topology

- To acces VT menu, click on Model and then on Virtual Topology
- Right click on VT menu to access automated and manual VT tools





Creates VT automatically

Creates VT manually

ANSYS

Automatic Virtual Topology

Automatically creating *Virtual Faces*

- Left Click *Virtual Topology* in *Model Tree*
- Set *Behaviour* in *Details*
  - Controls aggressiveness of automatic VT algorithm
  - Low: merges only the worst faces (and edges)
  - Medium & High: try to merge more faces
- Select if Face Edges shall be merged
- Right Click *Virtual Topology* and click *Generate Virtual Cells*

Manually creating a Virtual Face

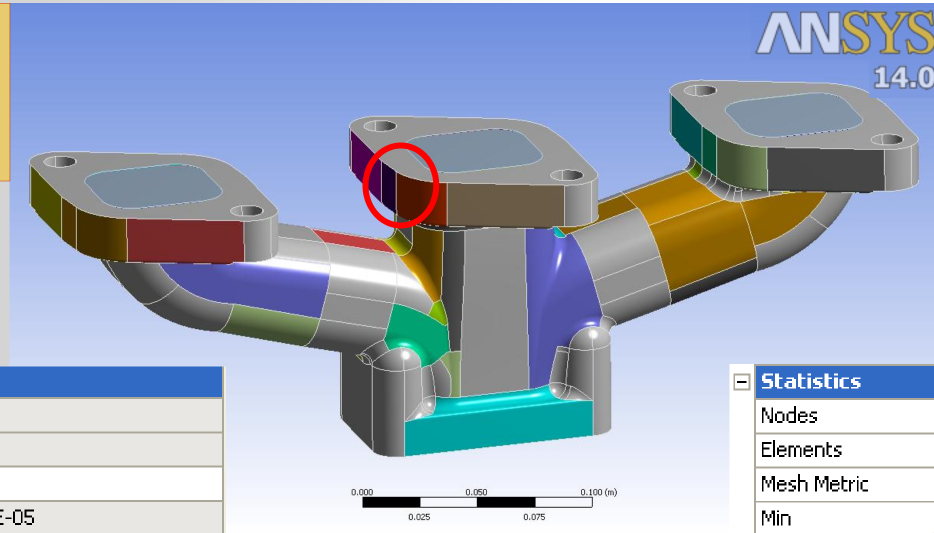
- RMB on Model tree and select Insert Virtual Topology
- Select Virtual Topology from the Tree Outline
- Pick faces or edges, RMB and Insert Virtual Cell

All VT entities created can be seen in different colors if Virtual Topology is selected in Tree Outline

Virtual Topology : Example

Without Virtual cells:

Edges are respected while creating surface mesh

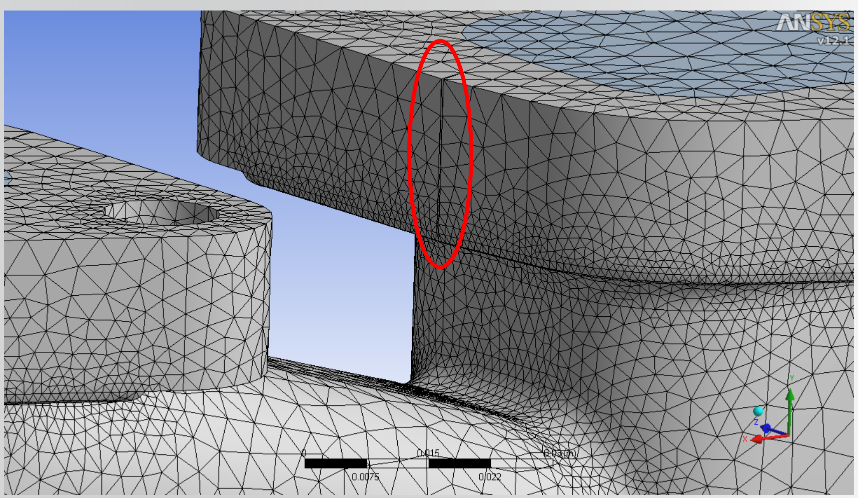


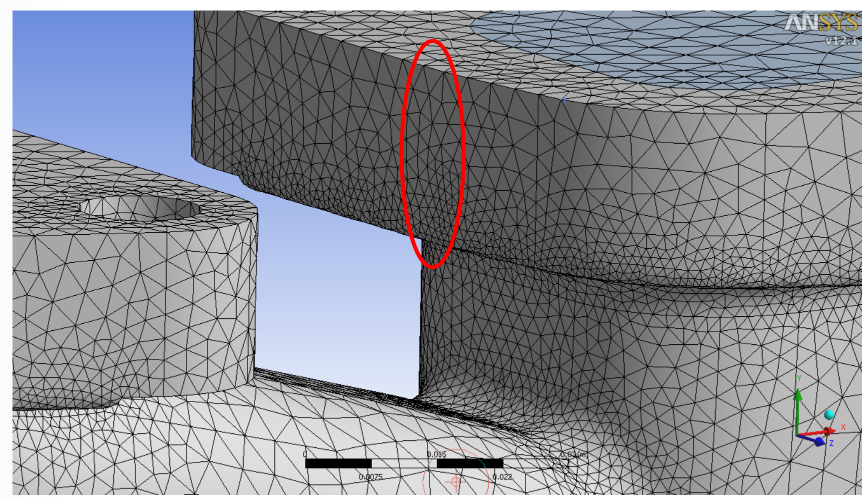
With Virtual cells:

Small faces are merged to form a single virtual face and edges of the original set of faces are no longer respected for meshing

Statistics	
Nodes	122036
Elements	640547
Mesh Metric	Skewness
Min	5.78935566496463E-05
Max	0.987372255333058
Average	0.245608757966614
Standard Deviation	0.131328150948273

Statistics	
Nodes	120644
Elements	635831
Mesh Metric	Skewness
Min	8.0149887573544E-06
Max	0.860292667417595
Average	0.24688413449344
Standard Deviation	0.12782486922467





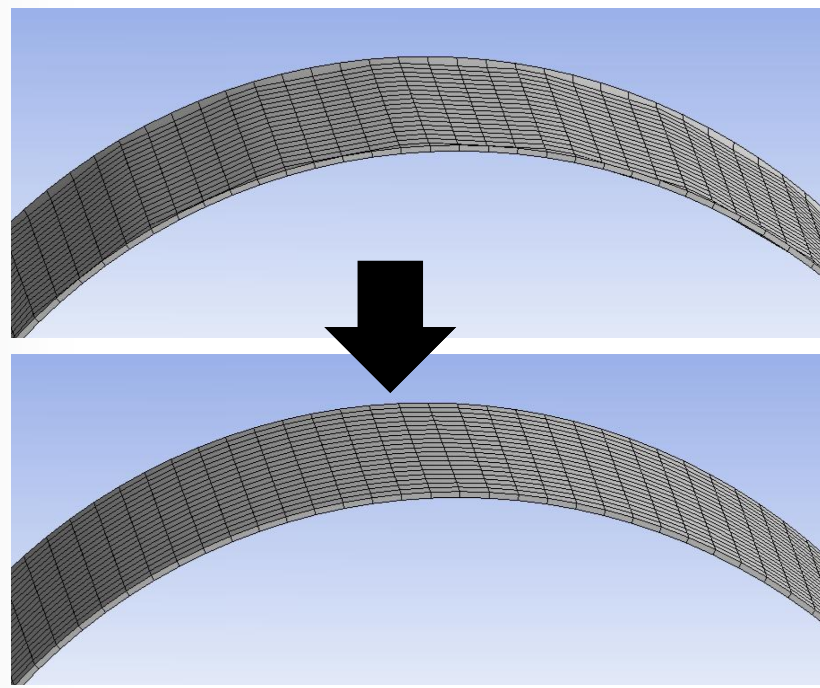
Project to underlying Geometry

– Virtual topologies are a faceted representation of the original geometry. By default mesh is projected to the facets

– Improved projection can be obtained by projecting back to the underlying geometry

Details of "Virtual Face 2"

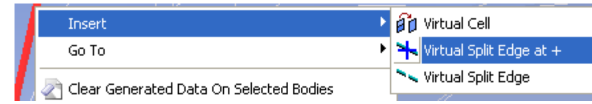
Definition	
Cell Class	Virtual Face
Suppressed	No
Project to Underlying Geometry	Yes
Scope	
Geometry	280 Faces

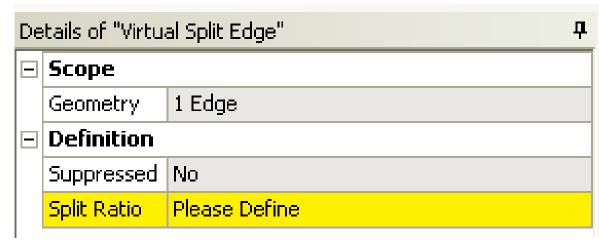


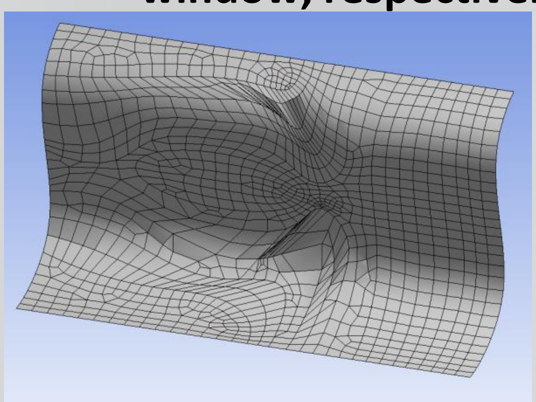
Virtual Topology : Example

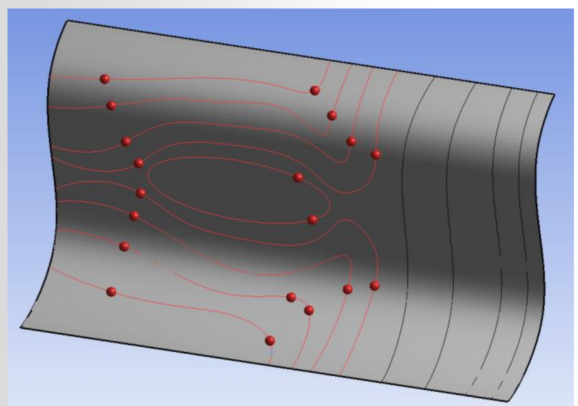
Creating edge split

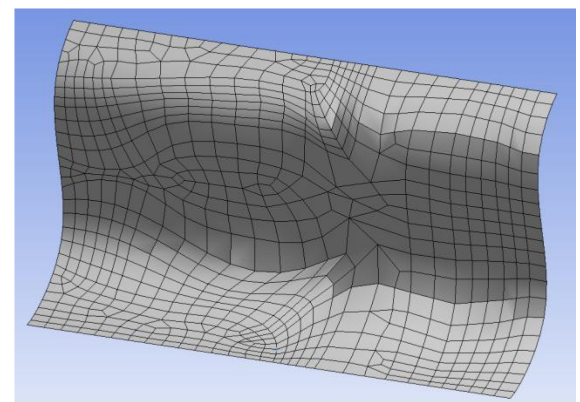
- Select Virtual Topology from the Tree Outline
- Pick the edge(s)
- RMB and select ‘Virtual Split Edge at +’ or ‘Virtual Split Edge’ to split the edge at the location specified by the selection, or to enter the split ratio in the Details window, respectively









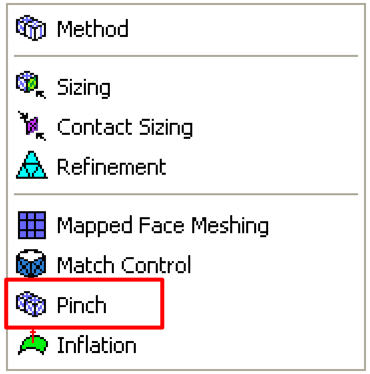


With edge splits:

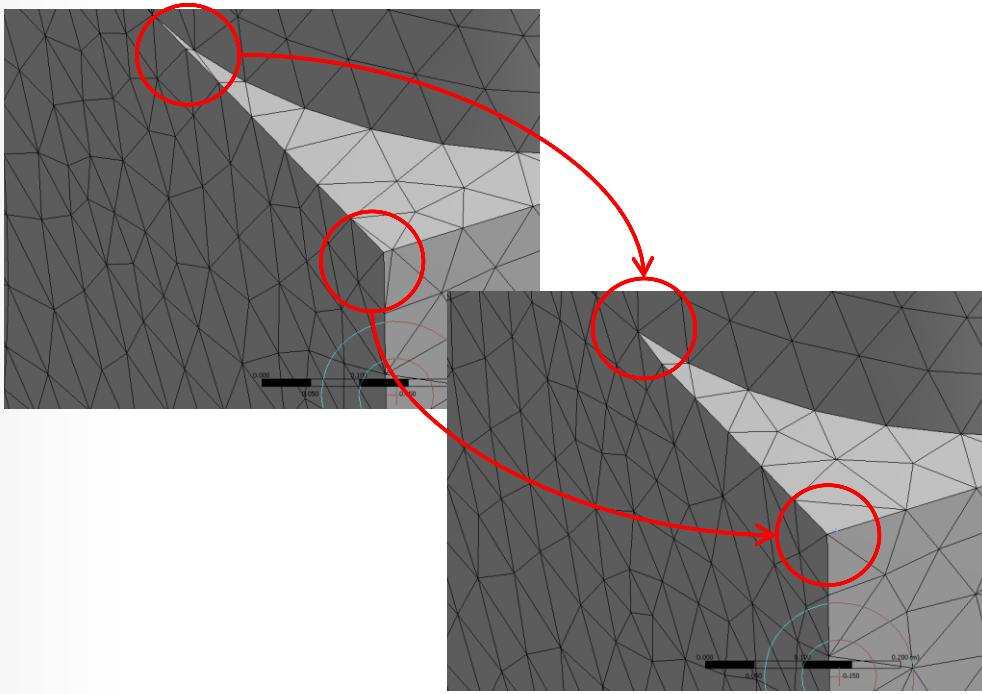
We can add edge constrains to improve the mesh

Edge splits can be moved interactively. Pick the virtual edge, hold the F4 key and move the red node along the edge with the mouse

Pinch



- **Pinch control removes small features at the mesh level**
  - Slivers
  - Short Edges
  - Sharp Angles
- **The Pinch feature works on vertices and edges only**
- **The Pinch feature is supported for the following mesh methods:**
  - Patch Conforming Tetrahedrons
  - Thin Solid Sweeps
  - Hex Dominant meshing
  - Quad Dominant Surface Meshing
  - Triangles Surface meshing
- **Not supported for**
  - CutCell
  - Patch Independent
  - Multizone
  - General Sweep

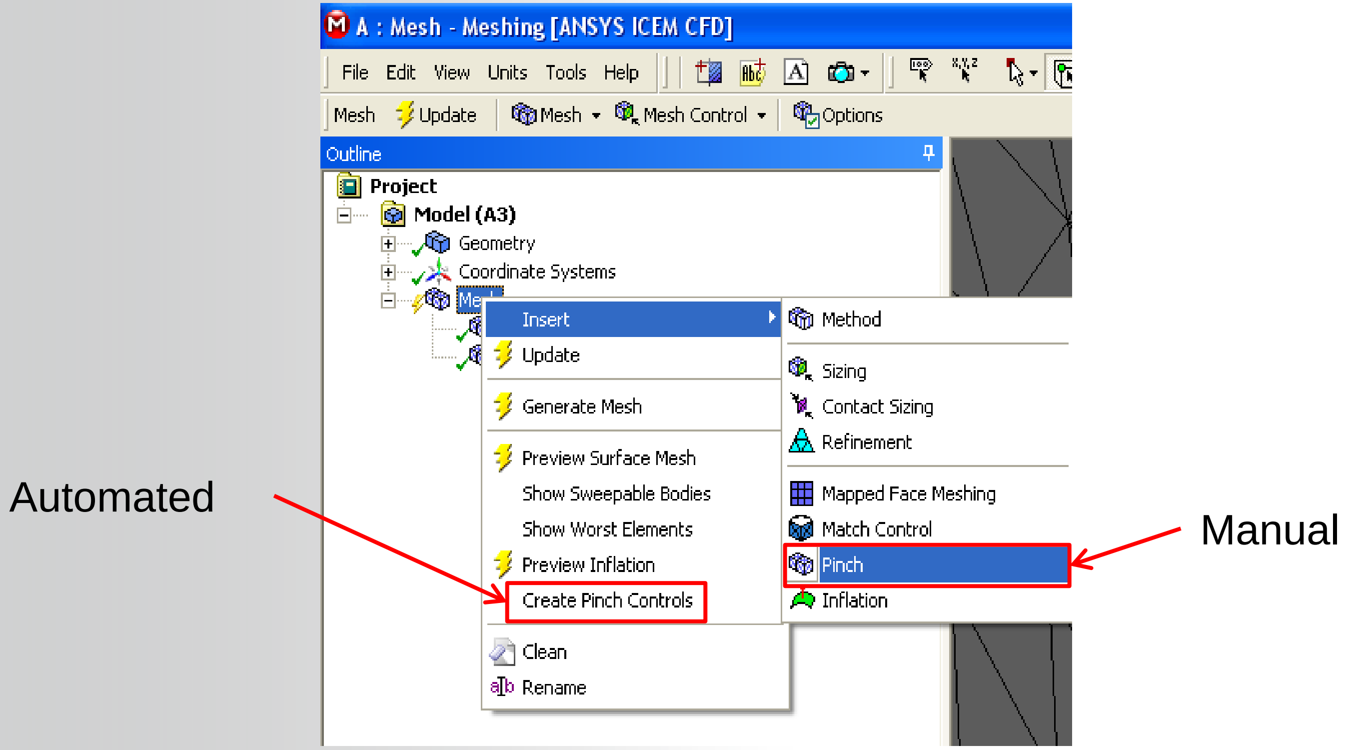


Pinch Control

Pinch features can be defined 2 ways

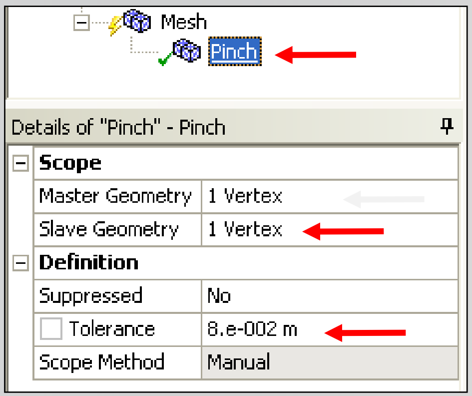
- **Automatically : pinches created based on global pinch tolerance in Mesh Detail**
- **Manually : pinch created one by one by user with local tolerance**

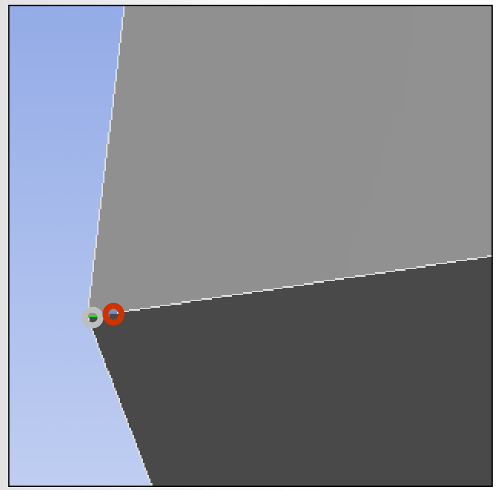
All pinches are listed in Model Tree under Mesh menu with methods and local controls



Pinch: Vertex-Vertex

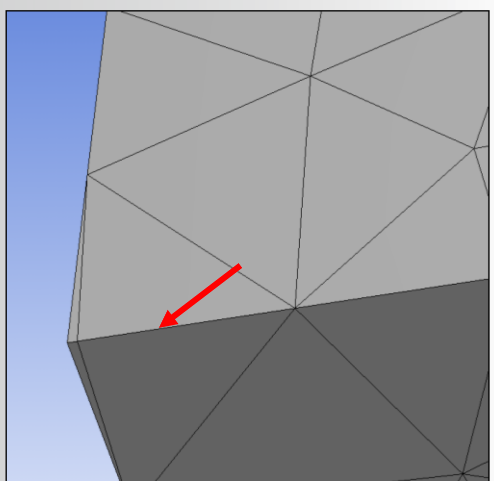
Vertex-vertex 'Pinch controls' will be created on an edge with length less than the specified tolerance

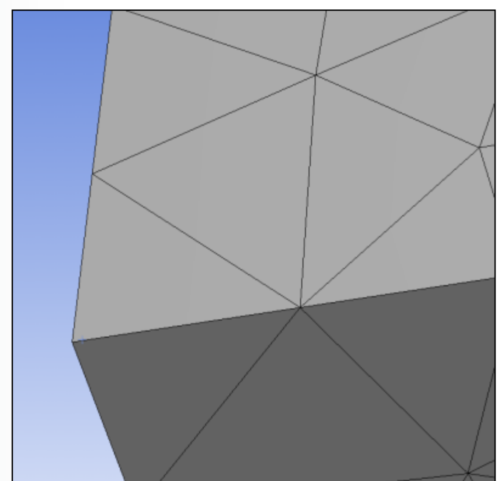




Will pinch out the slave geometry into the master geometry

Mesh without Pinch Control

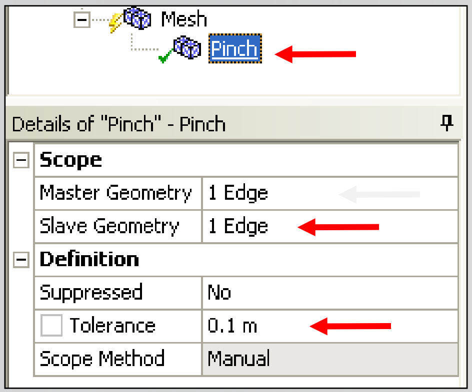


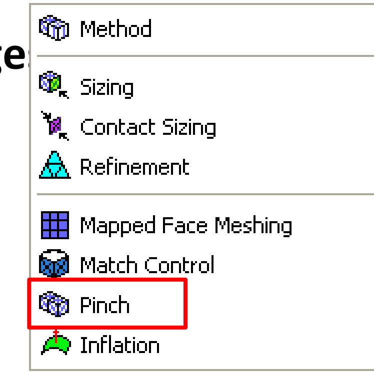


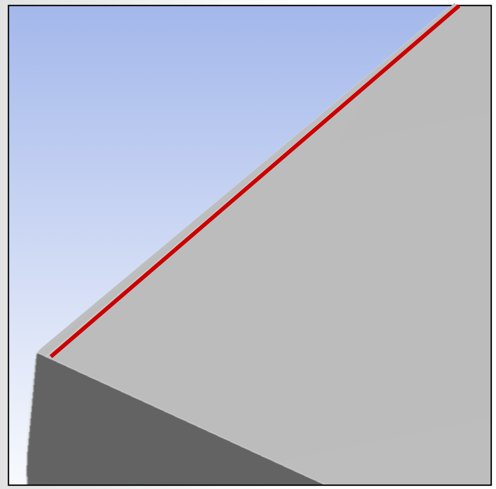
Mesh with Pinch Control

Pinch: Edge-Edge

Edge-Edge ‘Pinch controls’ will be created on any face for which two edges are within the proximity of specified tolerance

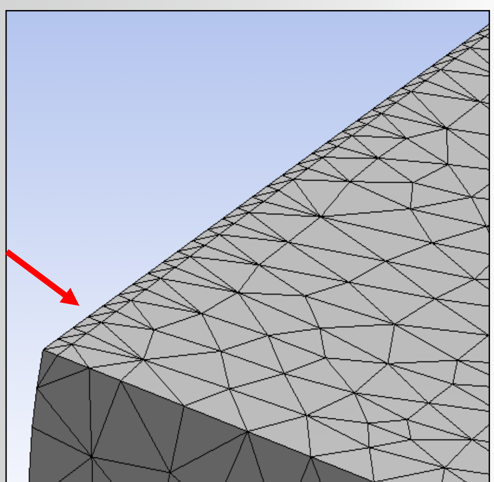


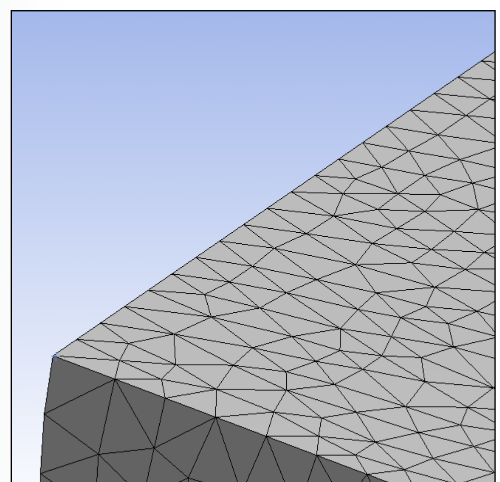




Will pinch out the entire or a portion of the slave geometry into the master

Mesh without Pinch Control





Mesh with Pinch Control

Workshops 5 (Applications Choice)