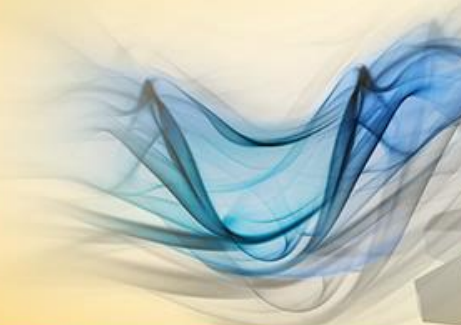


Lecture 9: Best Practice Guidelines

16.0 Release

A visualization of fluid flow, showing blue, wavy, translucent streamlines that represent the movement of a fluid around an object.

Fluid Dynamics

A 3D model of a purple gear with a glowing white center, surrounded by other smaller gears, representing structural analysis.

Structural Mechanics

A series of concentric green circles with a glowing center, representing electromagnetic field lines or wave propagation.

Electromagnetics

A 3D structure composed of several blue and black rectangular blocks stacked together, representing a complex system or multiphysics simulation.

Systems and Multiphysics

Introduction to ANSYS Fluent

Introduction

Lecture Theme:

The accuracy of CFD results can be affected by different types of errors. By understanding the cause of each different error type, best practices can be developed to minimize them. Meshing plays a significant role in the effort to minimize errors.

Learning Aims:

You will learn:

- Four different types of errors
- Strategies for minimizing error
- Issues to consider during mesh creation such as quality and cell type
- Best practices for mesh creation

Learning Objectives:

You will understand the causes of error in the solution and how to build the mesh and perform the simulation in a manner that will minimize errors

Motivation for Quality

CFD-Results are used for many different stages of the design process:

- Design & optimization of components and machines
- Safety analyses
- Virtual prototypes

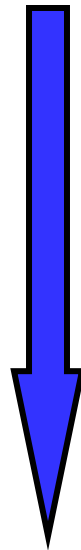
When undertaking a CFD model, consideration should be given to the purpose of the work:

- What will the results be used for?
- What level of accuracy will be needed?

Different Sources of Error

There are several different factors that combine to affect the overall solution accuracy. In order of magnitude:

- **Round-off errors**
 - Computer is working to a certain numerical precision
- **Iteration errors**
 - Difference between ‘converged’ solution and solution at iteration ‘n’
- **Solution errors**
 - Difference between converged solution on current grid and ‘exact’ solution of model equations
 - ‘Exact’ solution → Solution on infinitely fine grid
- **Model errors**
 - Difference between ‘exact’ solution of model equations and reality (data or analytic solution)



Round-Off Error

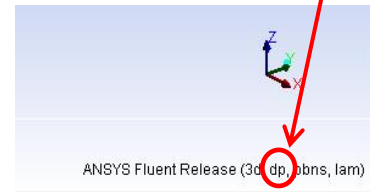
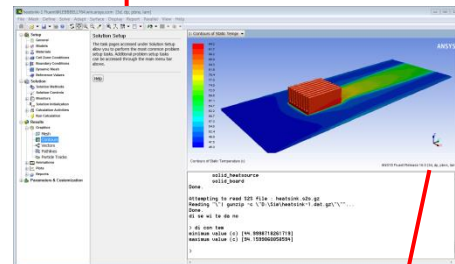
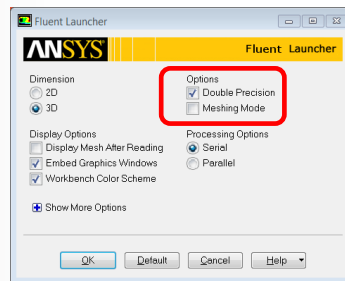
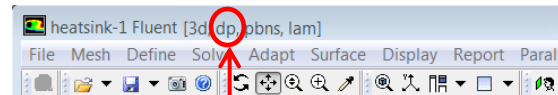
Inaccuracies caused by machine round-off:

- High grid aspect ratios
- Large differences in length scales
- Large variable range

Tip: Look for "dp" in the title bar of the Fluent window or the lower right corner of the graphics window to check if your current session is using double precision

How to identify if round-off error is a problem:

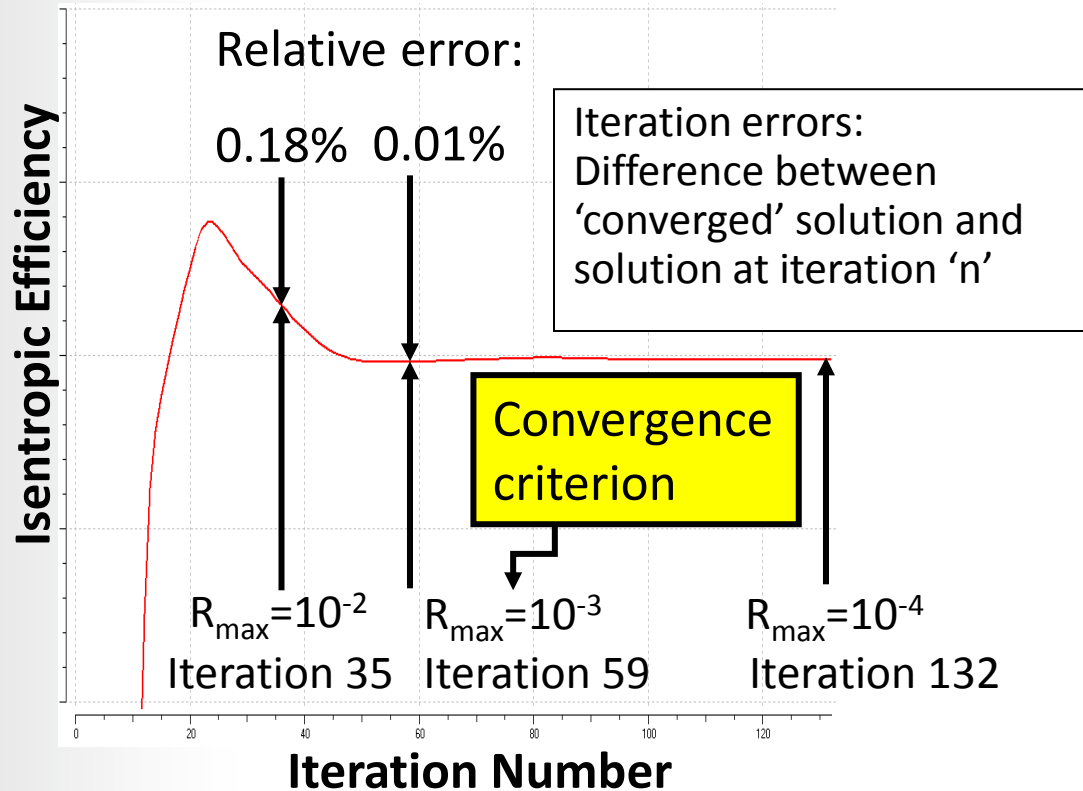
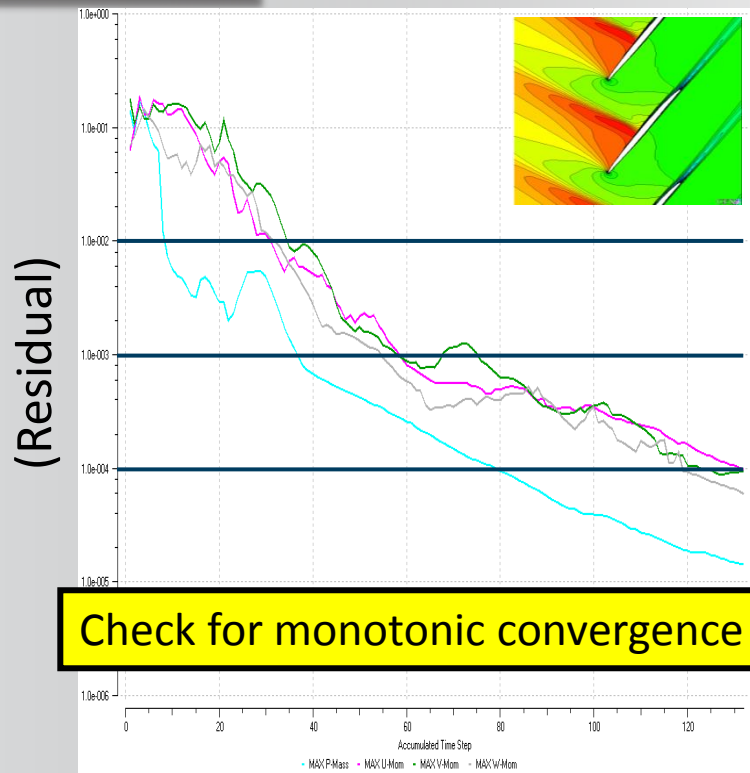
- Calculate with double precision if your case meets the above criteria
- Compare results with a solution that has been calculated with single precision
- If important quantities (target variables) are different, double precision should be used for subsequent calculations



Iteration Error - Best Practice

- **Define quantity or quantities of interest for your simulation (target variables):**
 - Head rise
 - Efficiency
 - Mass flow rate
 - ...
- **Select convergence criterion for the residuals**
- **Plot the value of the quantities of interest as the solution iterates**
- **Select a tighter convergence criterion and continue iterating and plotting**
- **Repeat until the values of the quantities of interest no longer change**
 - This will identify what residual level it is necessary to achieve in order to ensure the solution is free from iteration error
- **Monotonic convergence of residuals (next slide) is desirable, although not always possible**
- **Report mass and energy fluxes to ensure these are being conserved**

Iteration Error Example: 2D Compressor Cascade



Discretization Error

All discrete methods have solution errors:

- Finite volume methods
- Finite element methods
- Finite difference methods
- ...

Difference between solution on a given grid and “exact” solution on an infinitely fine grid

Exact solution not available → Discretization error estimation

Discretization Error Estimation

Impinging jet flow with heat transfer

2-D, axisymmetric

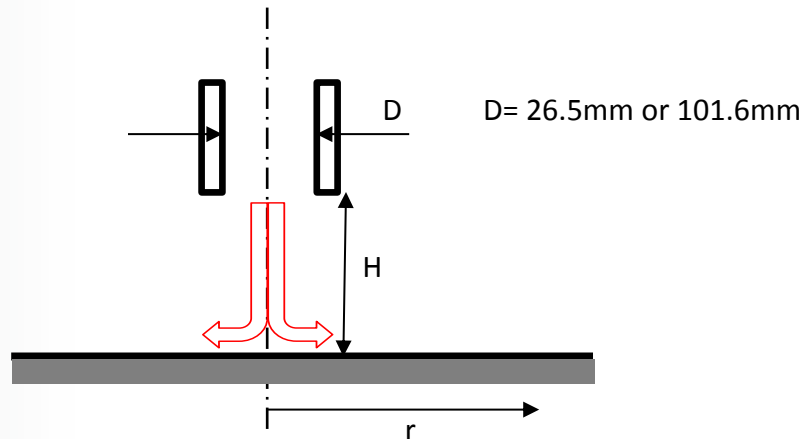
Compared Grids:

- $50 \times 50 \rightarrow 800 \times 800$

SST turbulence model

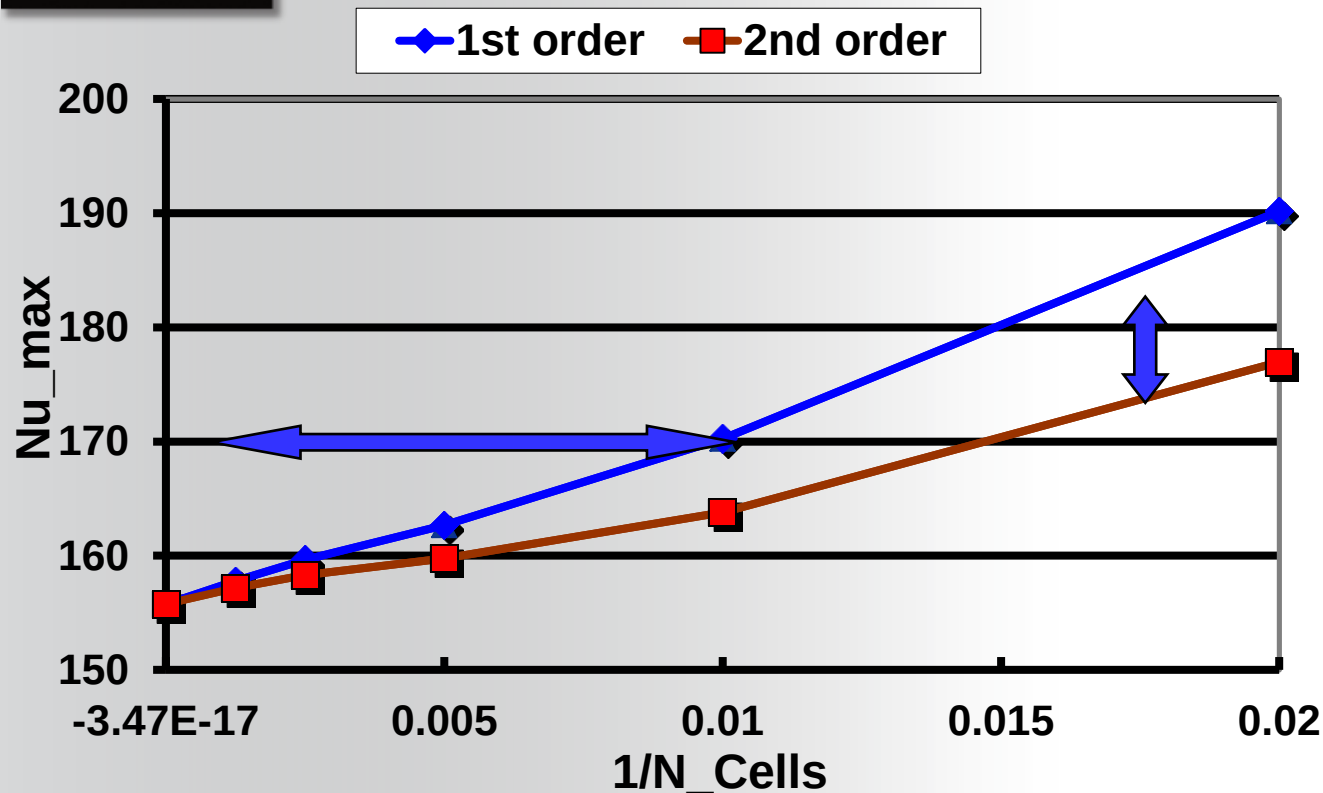
Discretization schemes:

- 1st order Upwind
- 2nd order Upwind



- Target quantities:
 - Heat transfer
 - Maximum Nusselt number

Discretization Error Estimation



The plot shows

- If the grid is fine enough, 1st and 2nd order solutions are the same
- On coarser meshes, the 2nd order solution is closer to the final solution

Practical alternatives for industrial cases are:

- Compare solutions from different order schemes
- Compare solutions on locally or regionally refined meshes

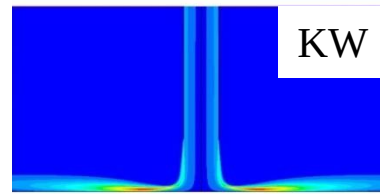
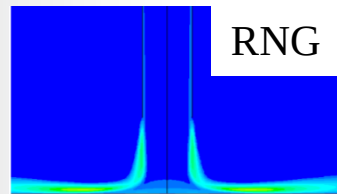
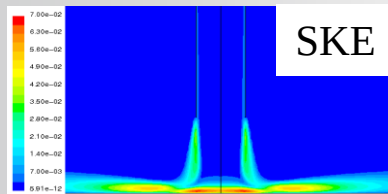
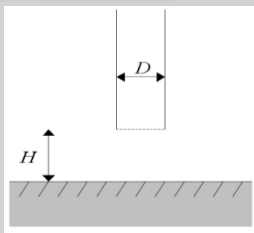
Model Errors

Inadequacies of (empirical) mathematical models:

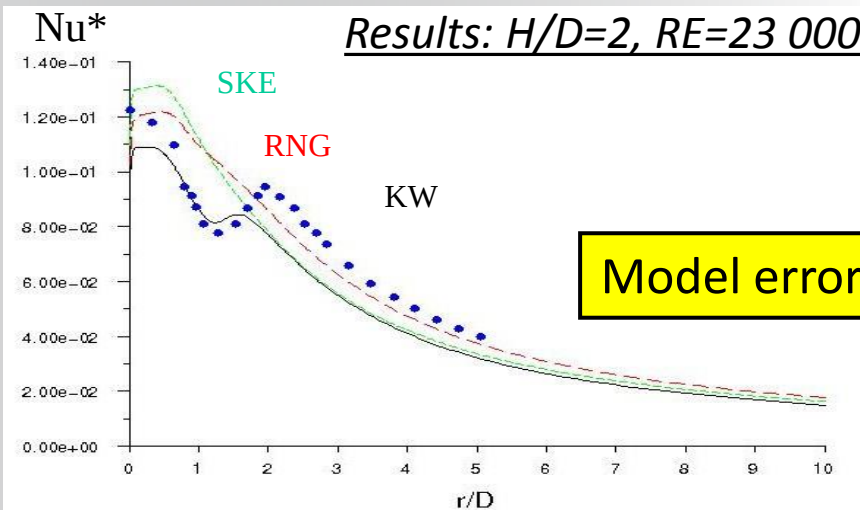
- Base equations (Euler vs. RANS, steady-state vs. unsteady-state, ...)
- Turbulence models
- Combustion models
- Multiphase flow models
- ...

Discrepancies between data and calculations remain, even after all numerical errors have become insignificant!

Model Error: Impinging Jet



←
TKE Contours



- Note how the predictions differ depending on which turbulence model is used
- The $k-\omega$ model (KW) performs better than the standard (SKE) or RNG $k-\epsilon$ models in this case
 - The $k-\epsilon$ based models overestimate the production of turbulence at the stagnation point, causing the predicted Nusselt number to be too high

Systematic Errors

Discrepancies remain

- Even if numerical and model errors are insignificant

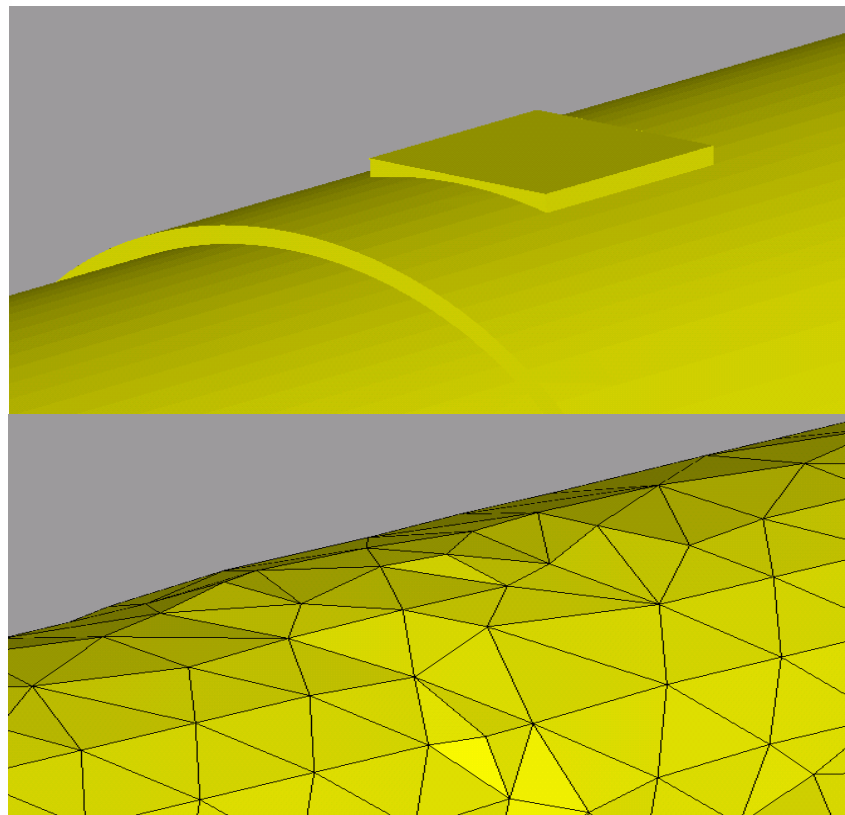
‘Systematic errors’:

- Approximations of:
 - Geometry
 - Component vs. machine
 - Boundary conditions
 - Fluid and material properties, ...

Try to ‘understand’ application and physics

Document and defend assumptions !

Perform uncertainty analysis



Meshing Best Practice Guidelines

Choosing your mesh strategy depends on

1. ACCURACY

Desired mesh quality

What is the maximum skewness and aspect ratio you can tolerate?

2. EFFICIENCY

Desired cell count

- Low cell count for resolving overall flow features vs High cell count for greater details

3. EASINESS TO GENERATE

Time available

- Faster Tet-dominant mesh vs crafted Hex/hybrid mesh with lower cell count

Goal: Find the best compromise between accuracy, efficiency and easiness to generate

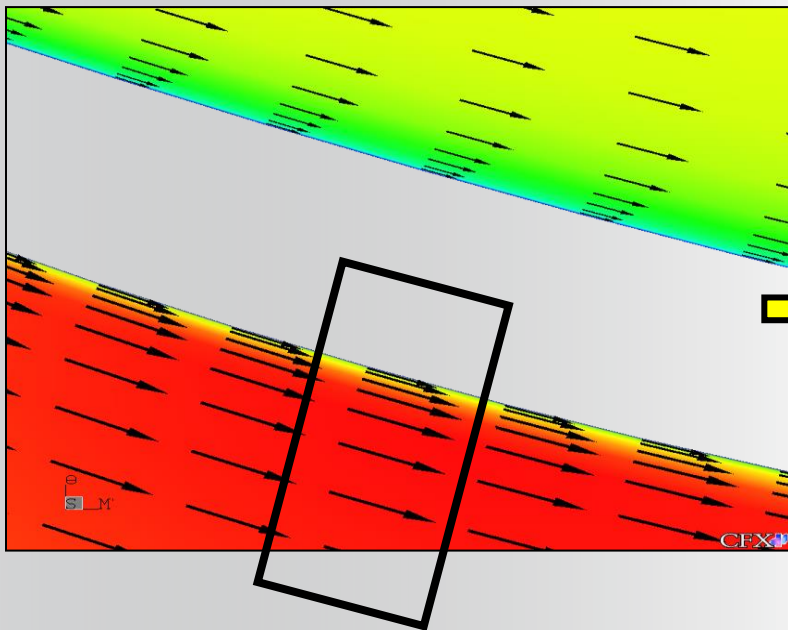
Meshing: Capture Flow Physics

- **Grid must be able to capture important physics:**
 - Boundary layers
 - Heat transfer
 - Wakes, shock
 - Flow gradients
- **Recommended meshing guidelines for boundary layers**
 - Both the velocity and thermal boundary layers must be resolved
 - There should be a minimum of 10-15 elements across the boundary layer thickness
 - The mesh expansion ratio in the wall normal direction should be moderate:
 - $\leq 1.2 \dots 1.3$
 - $y^+ \approx 1$ for heat transfer and transition modeling

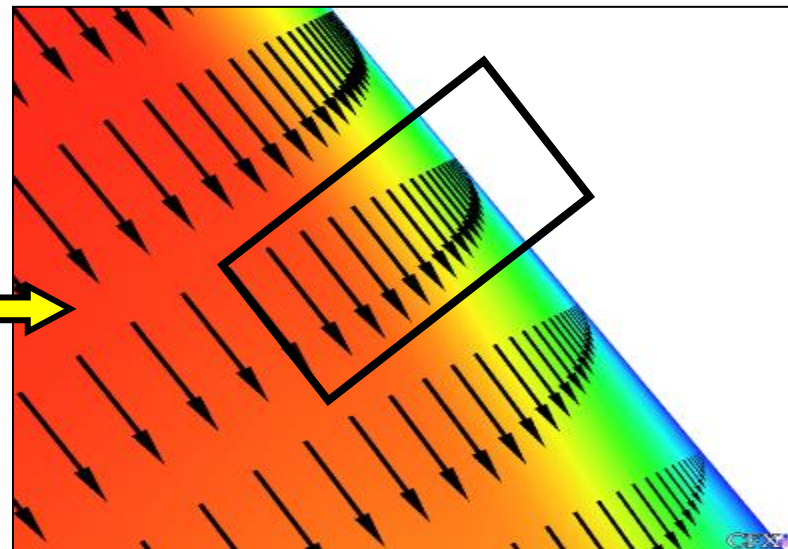
Meshing: Capture Flow Physics

- Example: Velocity profiles at airfoil

“Bad”



“Good”



A good mesh depends on :

- Cell not too distorted
- Cell not too stretched
- Smooth Cells transition

Good



Not Good

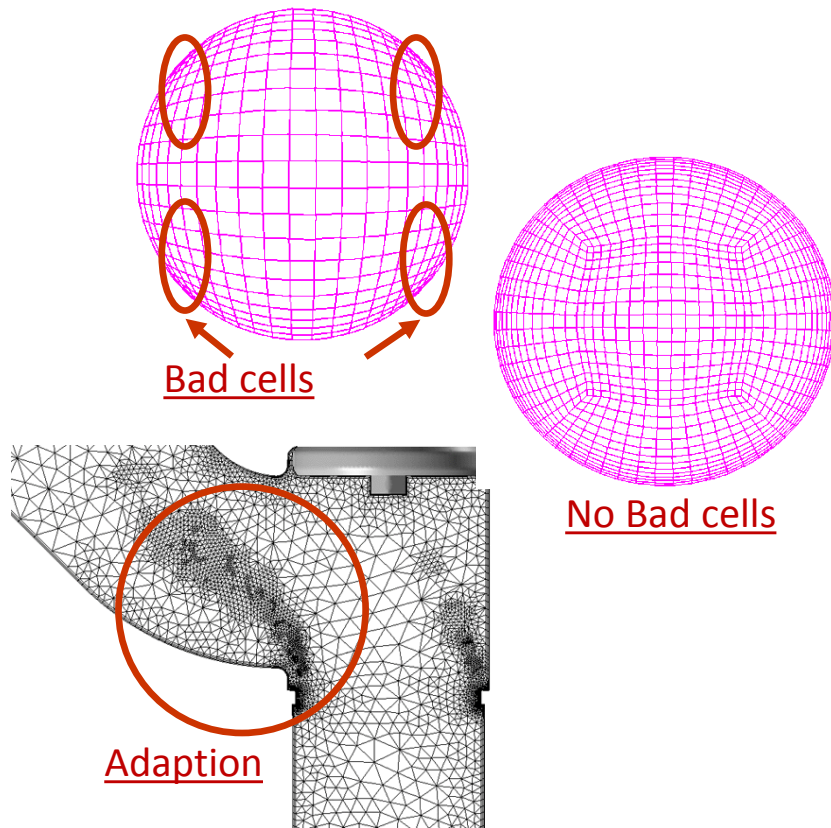


Grid generation:

- Scalable grids
- **Skewness < 0.95 (accuracy, convergence)**
 - also worst Orthogonal Quality $> .01$ and average value much higher
- Aspect ratios < 100
- Expansion ratios $< 1.5 \dots 2$
- Capture physics based on experience (shear layers, shocks)
- Angle between grid face & flow vector
- Concrete, quantitative recommendations for these factors presented in the Introduction to ANSYS Meshing course are included in the appendix of this presentation

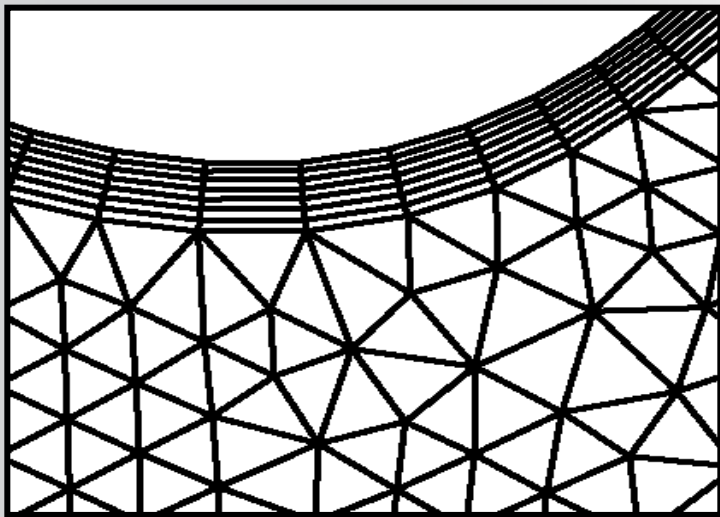
Grid refinement:

- Manual, based on error estimate
- Automatic adaptive based on 'error sensor'

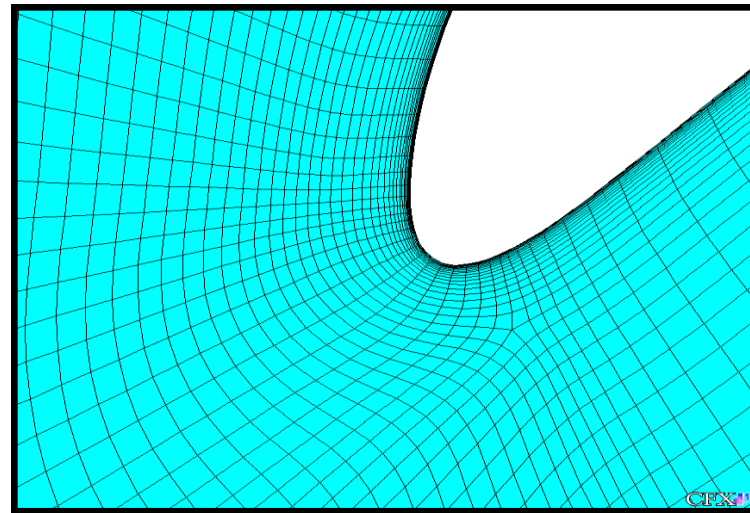


Mesh Quality

Avoid sudden changes in mesh density



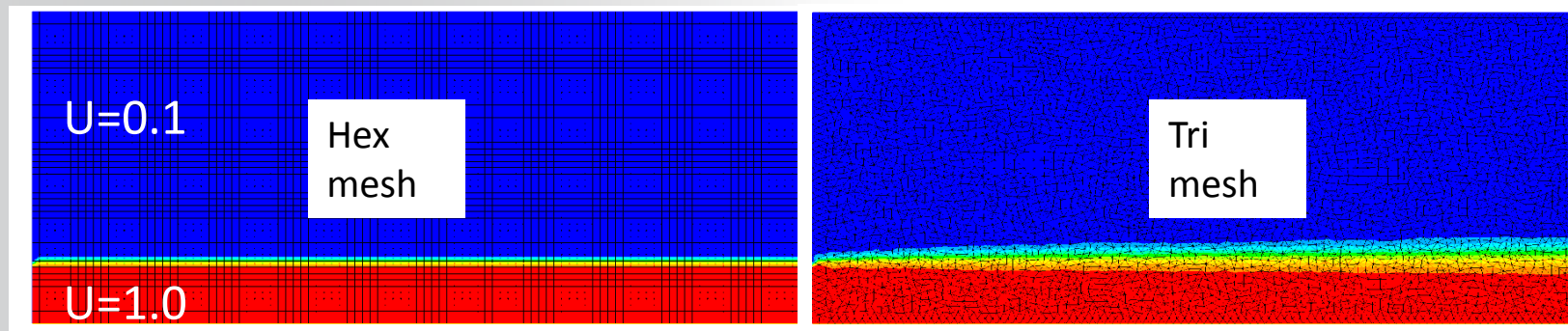
Not good



Good

Hex vs Tet Mesh : Accuracy Comparison

- Direction of the flow well known
 - ⇒ Quad/Hex aligned with the flow are more accurate than Tri with the same interval size

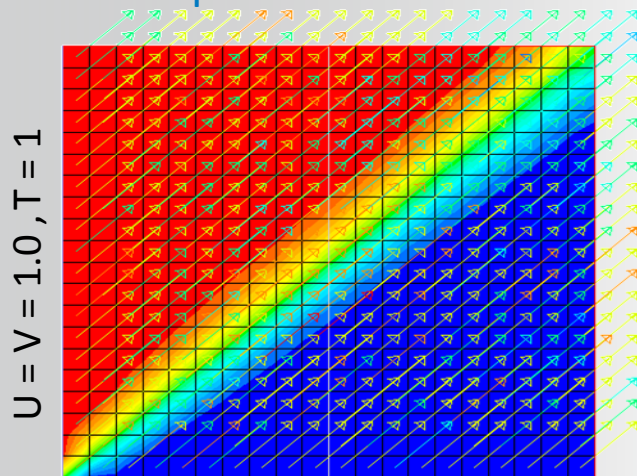


Contours of axial velocity magnitude for an inviscid co-flow jet

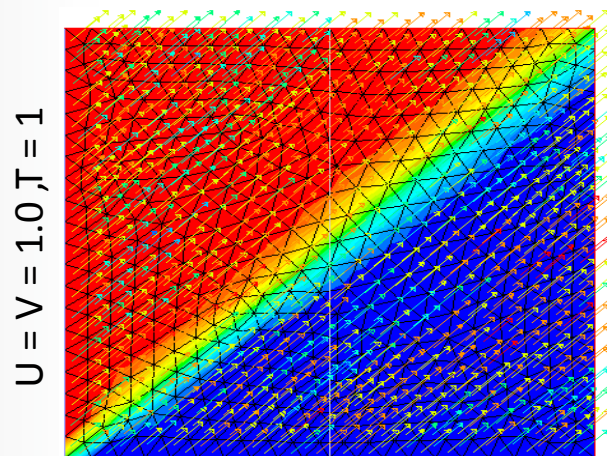
Hex vs Tet Mesh : Accuracy comparison

- For complex flows without dominant flow direction, Quad and Hex meshes lose their advantage

⇒ Quad & Tri equivalent



$U = V = 1.0, T = 0$



$U = V = 1.0, T = 0$

Contours of temperature for inviscid flow

Summary

- Try to ‘understand’ application and physics of the application
- Distinguish between numerical, model and other errors
- Document and defend assumptions
 - Geometry
 - Boundary conditions
 - Flow regime (laminar, turbulent, steady-state, unsteady-state, ...)
 - Model selection (turbulence, ...)
- Sources of systematic error
 - Approximations
 - Data
- Accuracy expectations vs. assumptions?

ERCOFTAC SIG: ,Quantification of Uncertainty in CFD‘

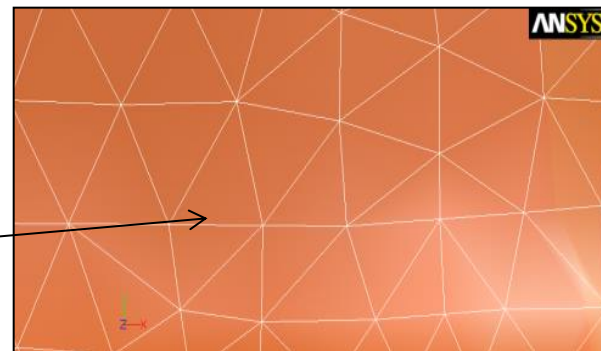
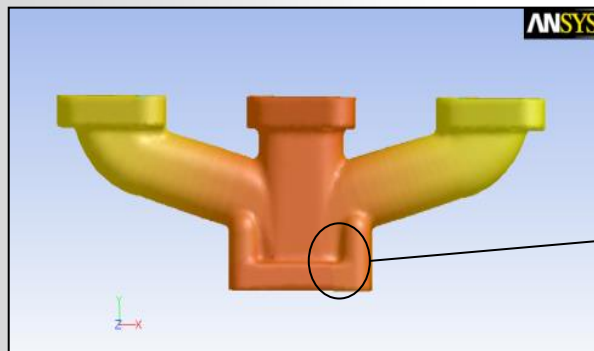
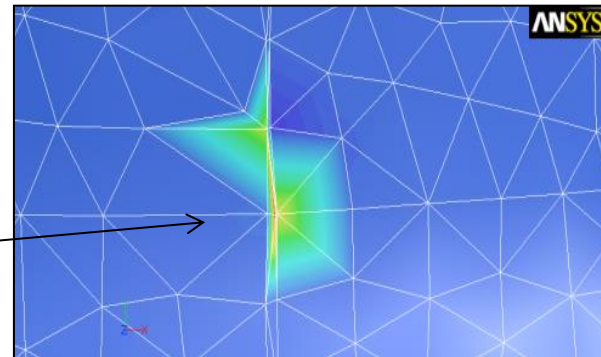
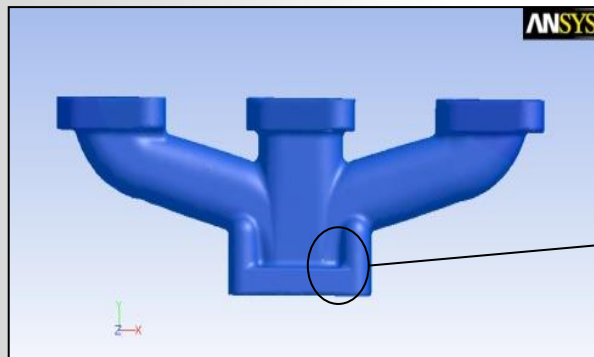
**Roache, P.J., *Verification and Validation in Computational Science and Engineering*,
Hermosa Publishers, 1998**

On the ANSYS customer portal, search for "best practice"

Appendix

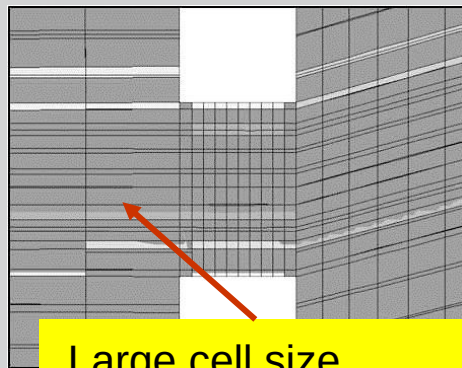
Impact of the Mesh Quality on the Solution

- 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

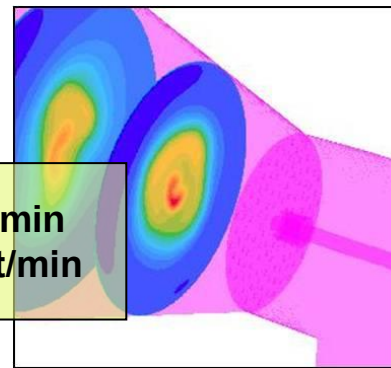


Large cell size change

Mesh 1

$(\max, \text{avg})_{\text{CSKEW}} = (0.912, 0.291)$
 $(\max, \text{avg})_{\text{CAR}} = (62.731, 7.402)$

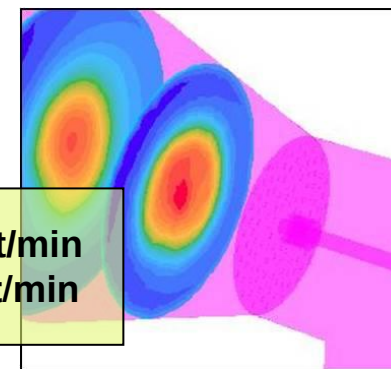
$Vz_{\text{MIN}} \approx -90 \text{ ft/min}$
 $Vz_{\text{MAX}} \approx 600 \text{ ft/min}$



Mesh 2

$(\max, \text{avg})_{\text{CSKEW}} = (0.801, 0.287)$
 $(\max, \text{avg})_{\text{CAR}} = (8.153, 1.298)$

$Vz_{\text{MIN}} \approx -100 \text{ ft/min}$
 $Vz_{\text{MAX}} \approx 400 \text{ ft/min}$



Mesh Statistics and Mesh Metrics

Displays mesh information for Nodes and Elements

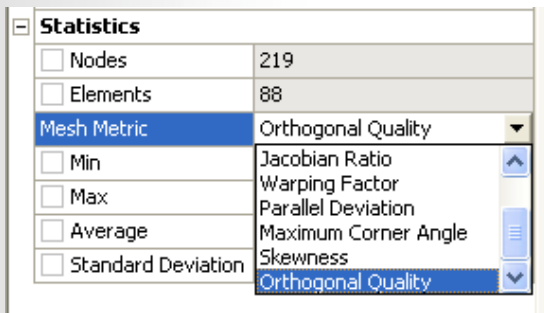
List of quality criteria for the Mesh Metric

- 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



☐ Nodes	17973
☐ Elements	91020
Mesh Metric	Orthogonal Quality
☐ Min	0.232336378900267
☐ Max	0.993658044699929
☐ Average	0.850623612128101
☐ Standard Deviation	8.69790479924024E-02



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:

$$\frac{A_i \cdot f_i}{|\vec{A}_i| |\vec{f}_i|} \quad \frac{A_i \cdot c_i}{|\vec{A}_i| |\vec{c}_i|}$$

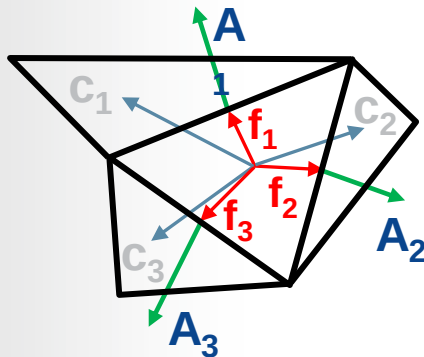
computed for each face i

For the face it is computed as the minimum of $\frac{A_i \cdot e_i}{|\vec{A}_i| |\vec{e}_i|}$ computed for each edge i

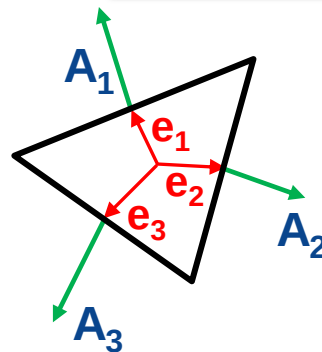
Where A_i is the face normal vector and f_i is a vector from the centroid of the cell to the centroid of that face, and c_i is a vector from the centroid of the cell to the centroid of the adjacent cell, where e_i is the vector from the centroid of the face to the centroid of the edge

At boundaries and internal walls c_i is ignored in the computations of OQ

On cell



On face



Skewness

Two methods for determining skewness:

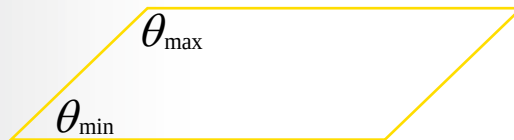
1. Equilateral Volume deviation:

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

Applies only for triangles and tetrahedrons

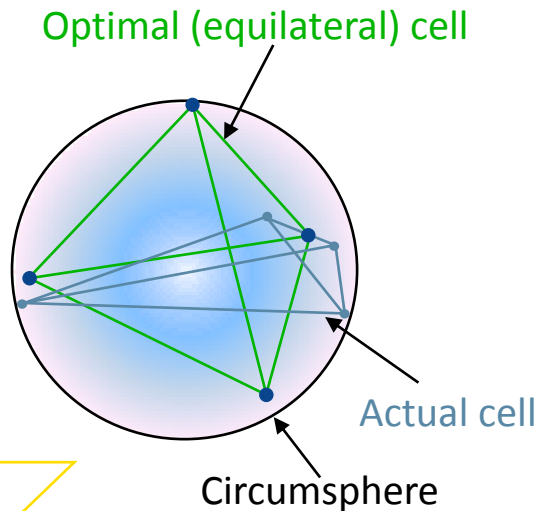
2. Normalized Angle deviation:

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



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

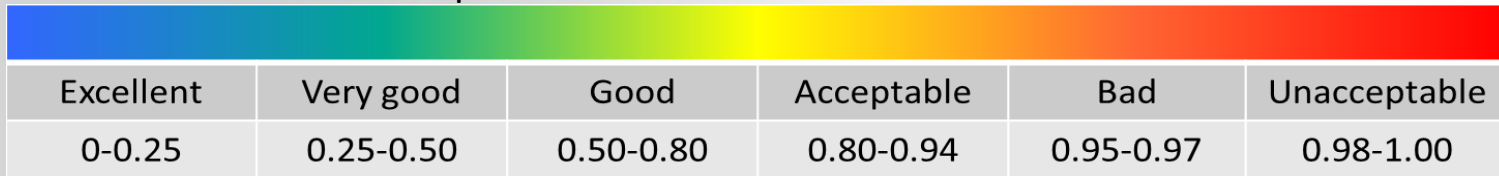
Mesh quality recommendations

Low Orthogonal Quality or high skewness values are not recommended

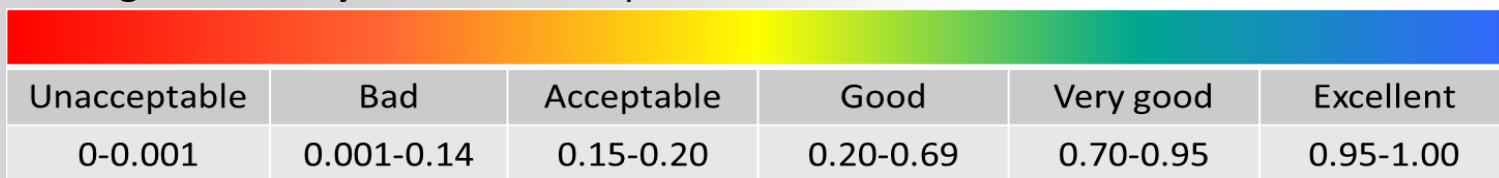
Generally try to keep minimum orthogonal quality > 0.1 , or maximum skewness < 0.95 . However these values may be different depending on the physics and the location of the cell

Fluent reports negative cell volumes if the mesh contains degenerate cells

Skewness mesh metrics spectrum



Orthogonal Quality mesh metrics spectrum



2-D:

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

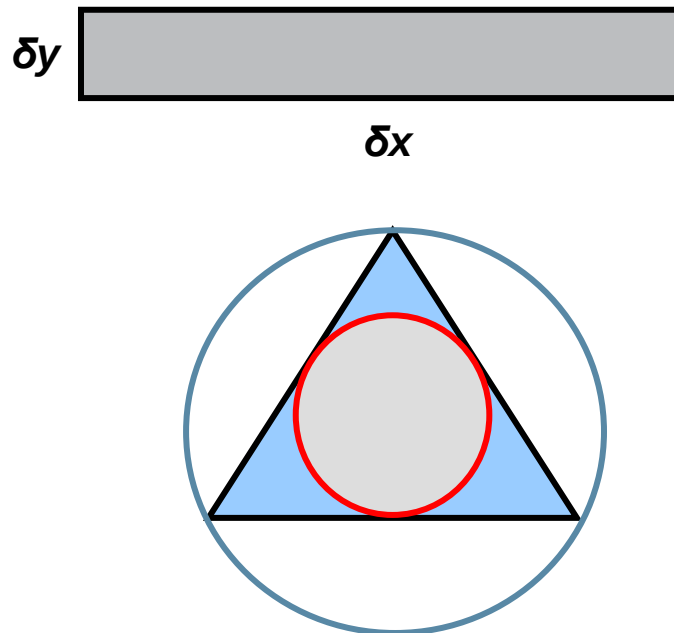
3-D

- Area ratio
- Radius ratio of circumscribed / inscribed circle

Limitation for some iterative solvers

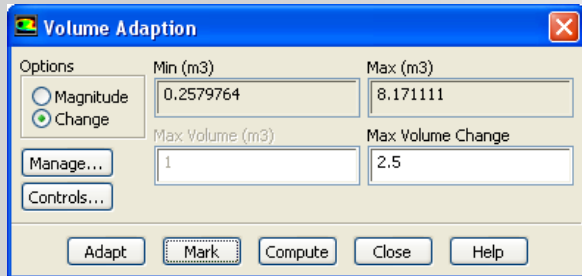
- $A < 10 \dots 100$
- (CFX: < 1000)

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

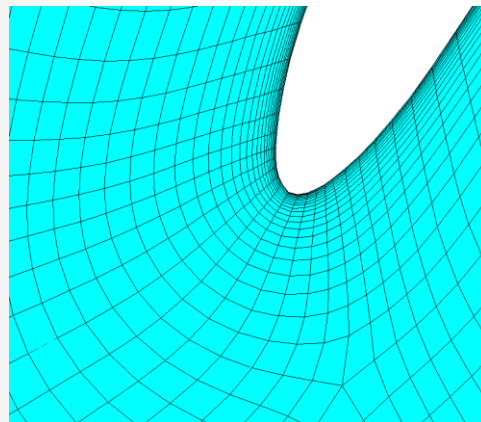


Checked in solver

- 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

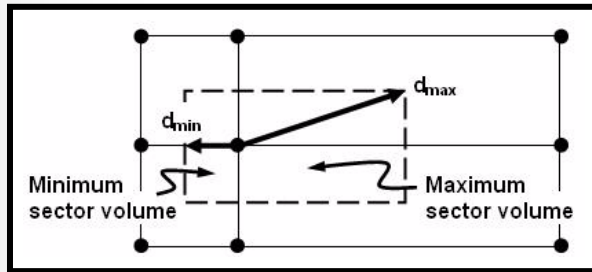


Recommendation:

Good: $1.0 < \sigma < 1.5$

Fair: $1.5 < \sigma < 2.5$

Poor: $\sigma > 5 \dots 20$



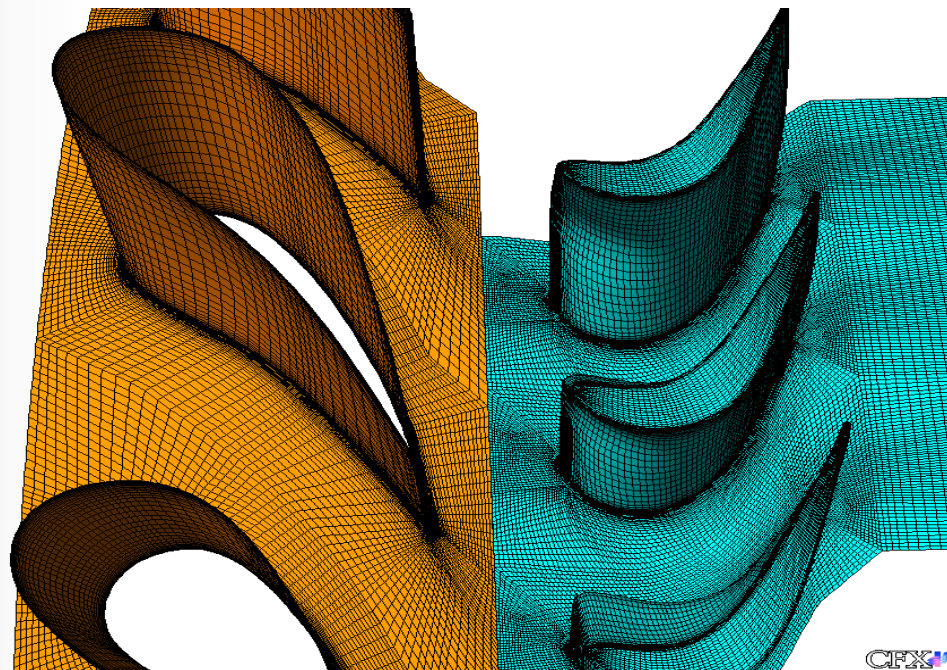
Elements: Hex

Pro:

- Good shear layer element
- Best element wrt. memory & calculation time per element

Con:

- Degree of automation for grid generation

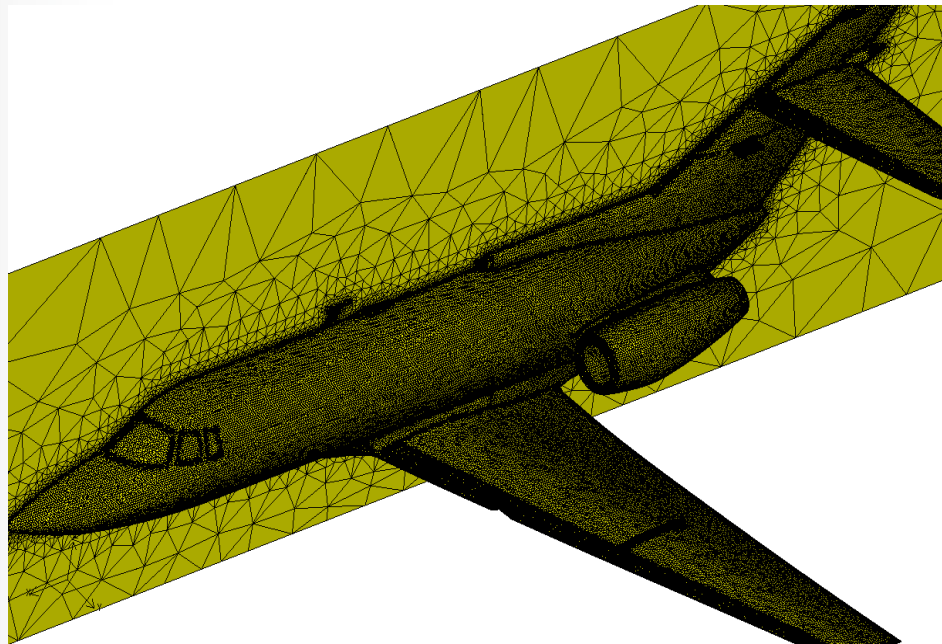


Pro:

- High degree of automation for grid generation

Con:

- Memory & calculation time per node $\approx 1.5 \times$ hex
- Poor shear layer element
- No streamline orientation
- Quantity must (and can) make up for quality



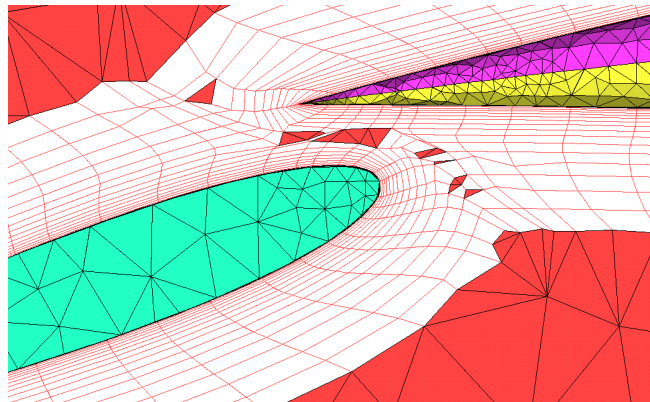
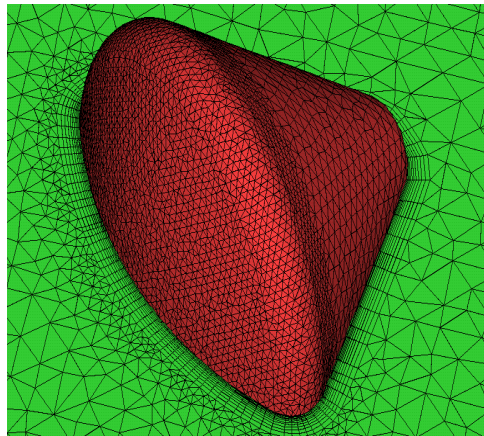
Elements: Prism

Pro:

- Better shear layer resolution than tet
- High degree of automation
- Tet/prism combination

Con:

- Less efficient than hex
- Topological difficulties (corners, ...) → poor grid quality (angles, ...)
- Manual repair

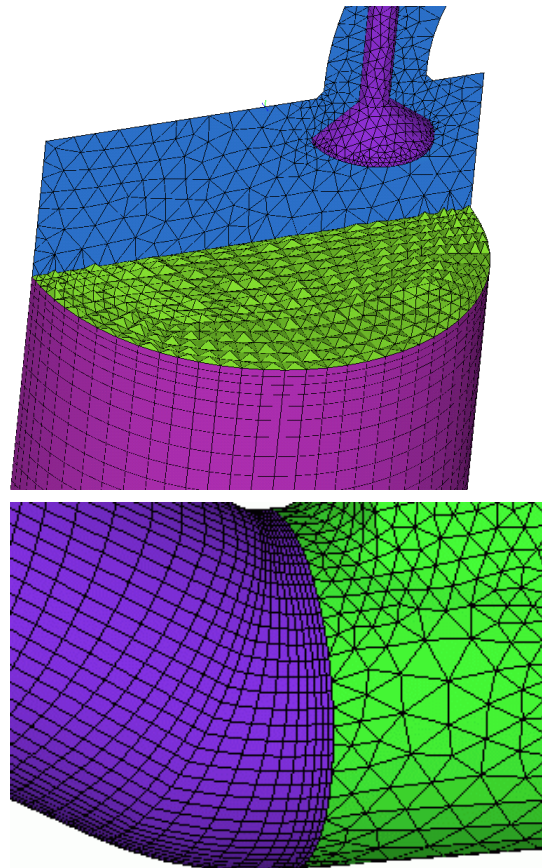


Use in hybrid grids

Transition element between hex and tet

Polyhedral grids

- ANSYS Fluent:
 - Generate base types
 - Convert
- ANSYS CFX builds polyhedrals around vertices



Recommendations

1st Option → Hex grid

- Best accuracy and numerical efficiency
- Time and effort manageable?

2nd Option → Tet/hex/pyramid grid

- Hex near walls & shear layers
- Developing technology ...

3rd Option → Tet/prism grid

- High degree of automation
- Quality (prism/tet transition, ...)

4th Option → Tet grid

- Shear layer resolution?

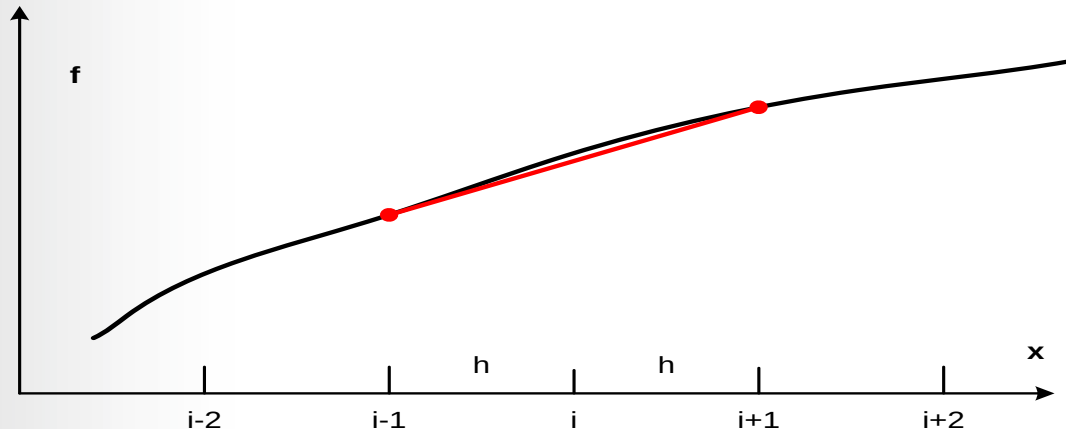
Truncation errors → source of discretisation errors

Minimize truncation errors → minimize discretisation errors

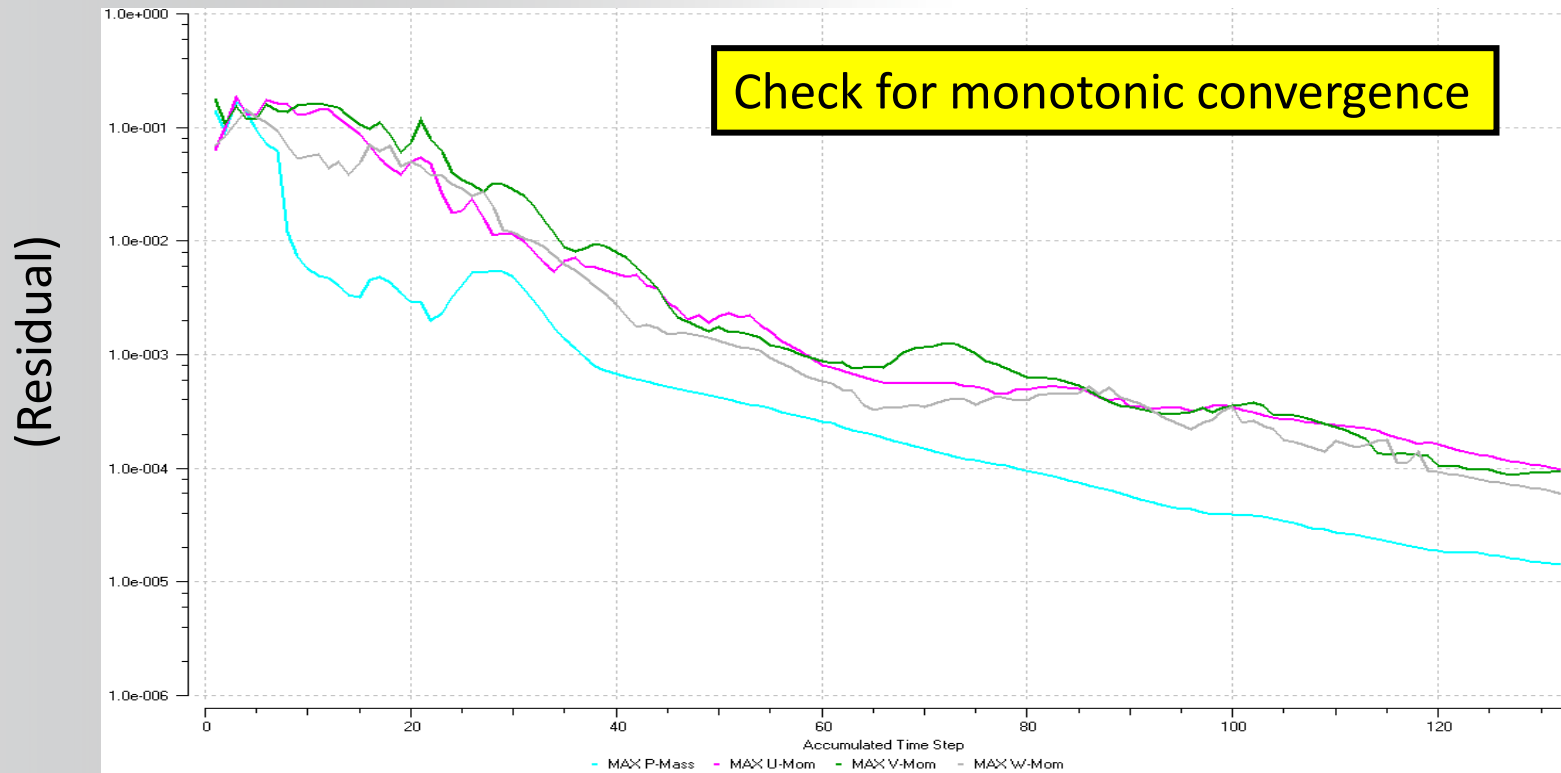
Truncation error → Difference between 'analog' and 'discrete' representation

$$\left(\frac{\partial f}{\partial x} \right)_i = \frac{f_{i+1} - f_{i-1}}{2h} + \tau_i$$

$$\tau_i = \frac{h^2}{6} \left(\frac{\partial^3 f}{\partial x^3} \right)_i + \dots$$



Iteration Error – Example

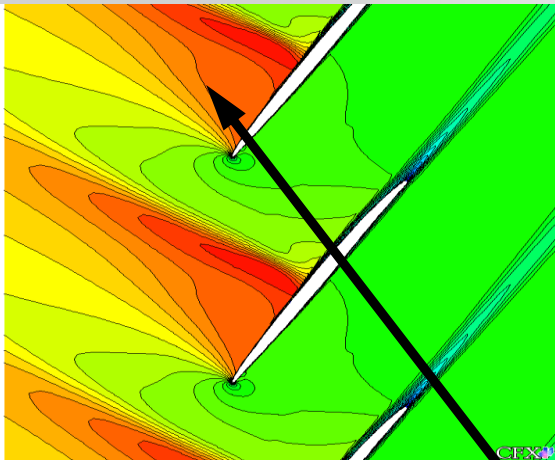


Iteration Error – Example

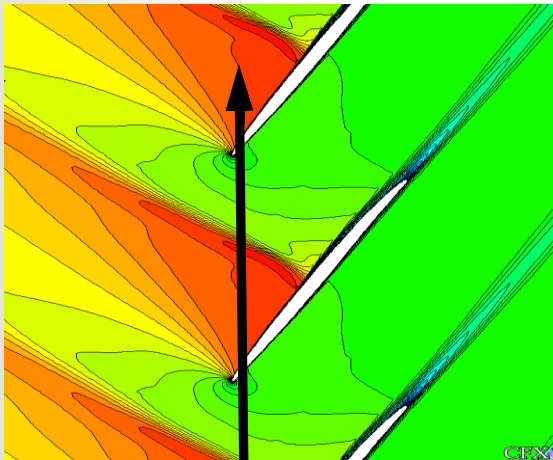
Effect of different residual limits during convergence:

- 2D Compressor cascade
- 2nd order

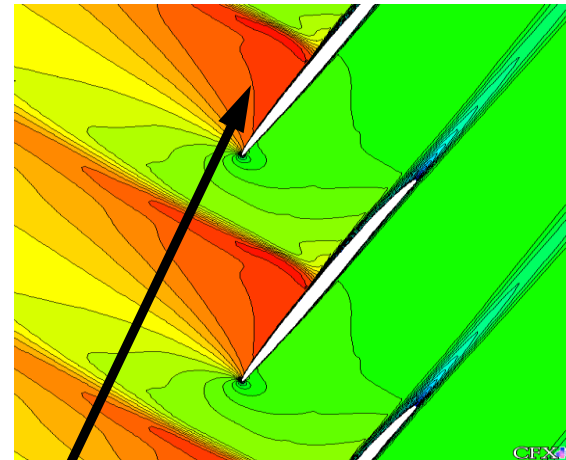
$$R_{\max} = 1 \times 10^{-3}$$



$$R_{\max} = 1 \times 10^{-4}$$



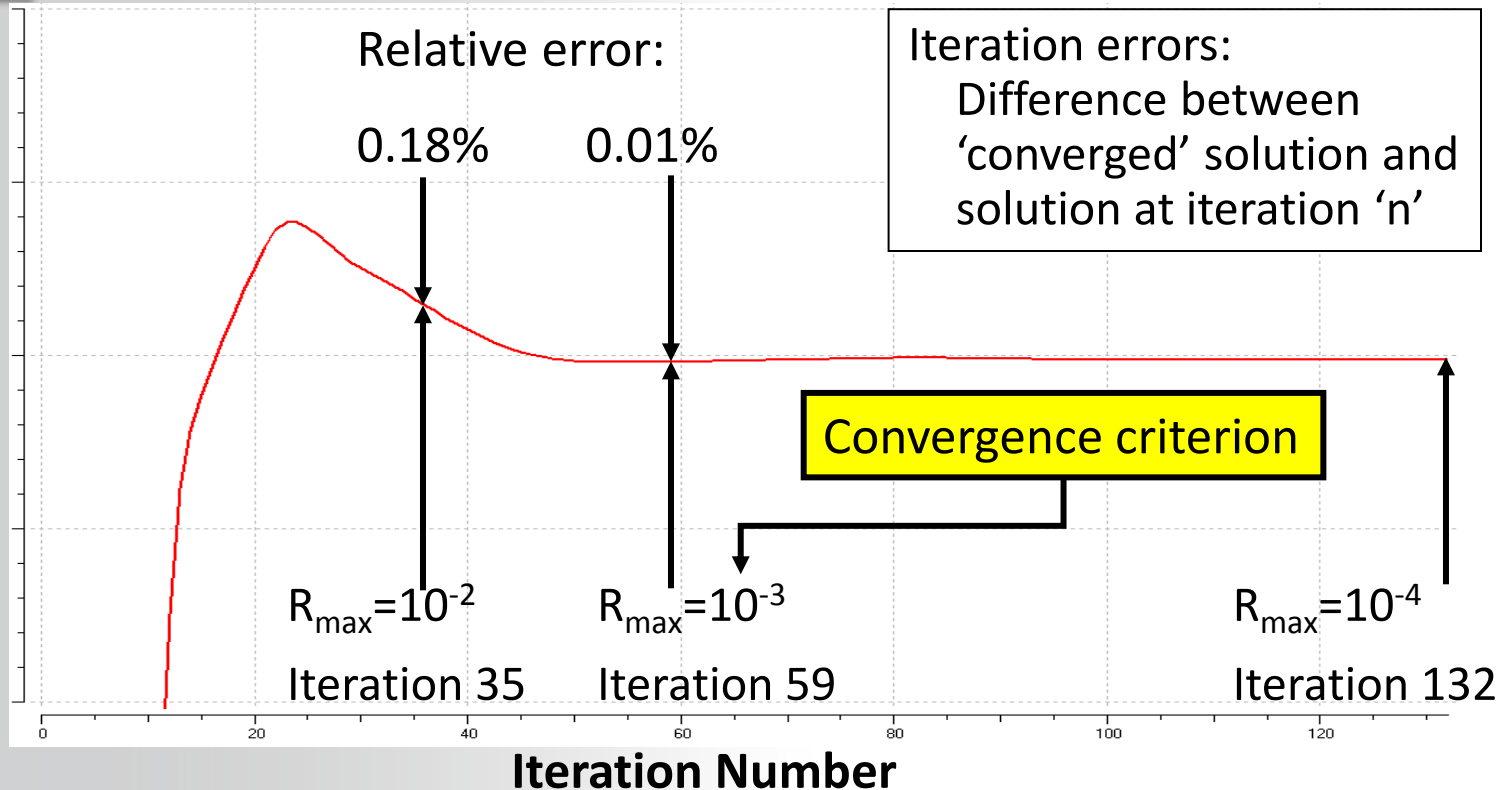
$$R_{\max} = 1 \times 10^{-5}$$



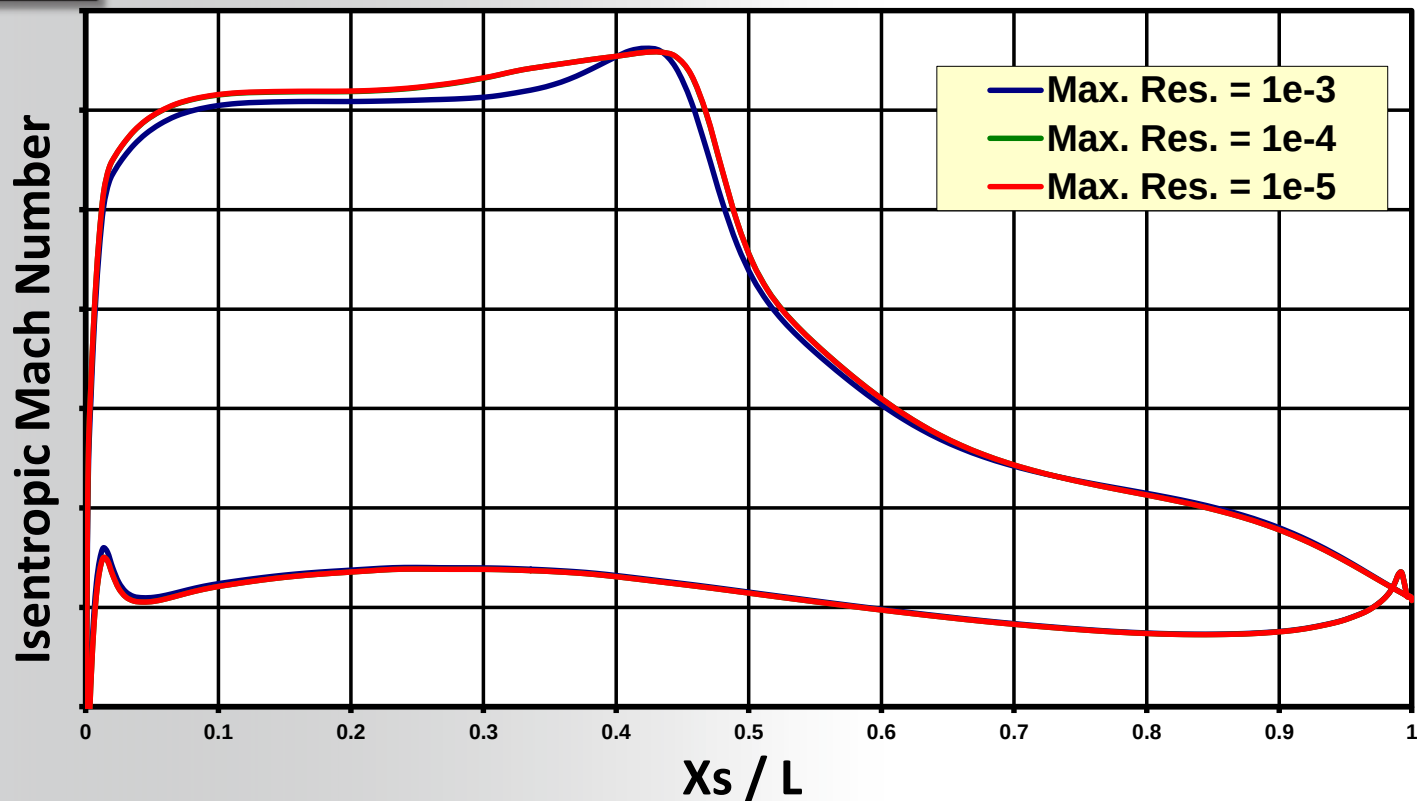
Change of Pressure Distribution

Iteration Error – Example

Isentropic Efficiency



Iteration Error – Example



Discretization Error Estimation

Grid	Nu		Error	
	1 st order	2 nd order	1 st order	2 nd order
50 × 50	190.175	176.981	22.1 %	13.6 %
100 × 100	170.230	163.793	9.3 %	5.1 %
200 × 200	162.664	159.761	4.4 %	2.6 %
400 × 400	159.646	158.296	2.3 %	1.4 %
800 × 800	157.808	157.168	1.1%	0.7 %
$\infty \times \infty$	155.751	155.777		

