Advanced Modeling & Simulation (AMS) Seminar Series
NASA Ames Research Center, August 11th, 2022

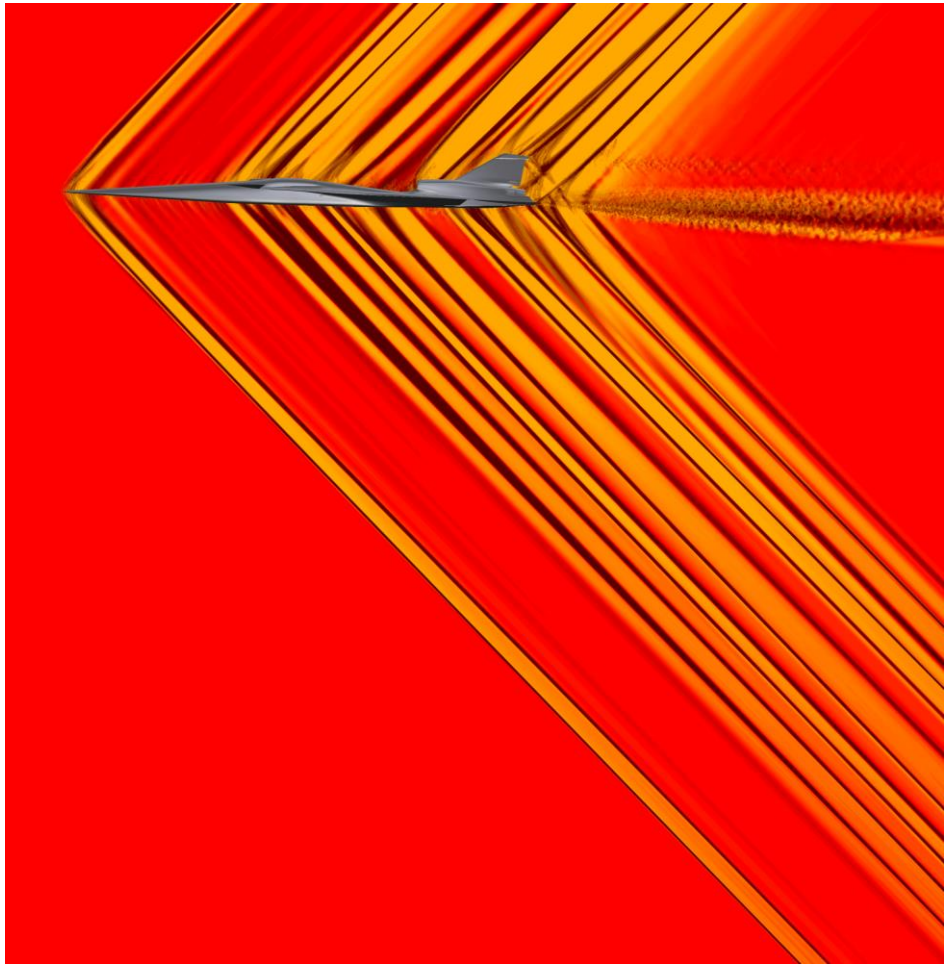
Improved CFD Methodology and Workflow Using Solution-Based Mesh Adaption in Ansys Fluent



Adam Norman
Lead Application Engineer
Ansys Inc.

Agenda

- Introduction to Mesh Adaption
- Error-Based Adaption Criteria
  - Aerospike
  - 25 – 55 Sharp Double Cone
  - C608
- Standard Deviation Adaption Criteria
  - Jet in Cross Flow
  - Premixed Hydrogen-Air Flame Propagation
  - Generic Supersonic Impingement
- Anisotropic Mesh Adaption
- Summary



©2021 ANSYS, Inc.

Introduction to Mesh Adaption and the Hessian

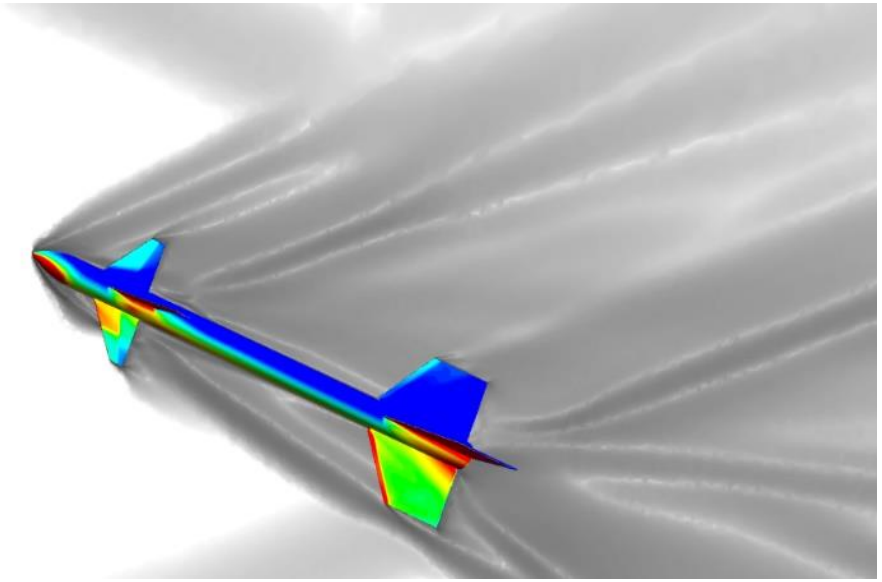
Ansys

Why use adaption?

- Resurgence in hypersonic technologies driving design investigation

- In numerical analyses, high-quality mesh generation can be time-consuming
  - Knowing where various phenomena will occur, is very difficult if not impossible

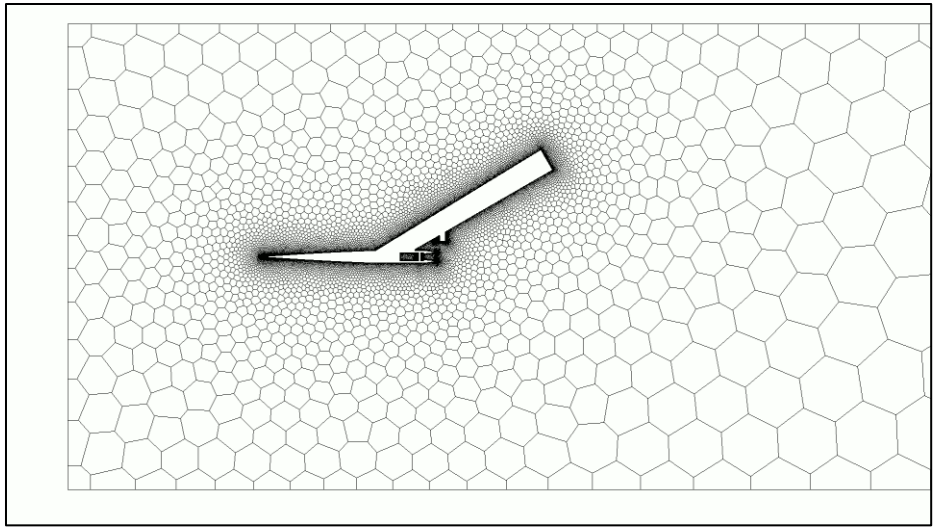
- This generally leads to an over-refinement of the computational mesh in non-key areas



©2021 ANSYS, Inc.

Solution-based Mesh Adaption

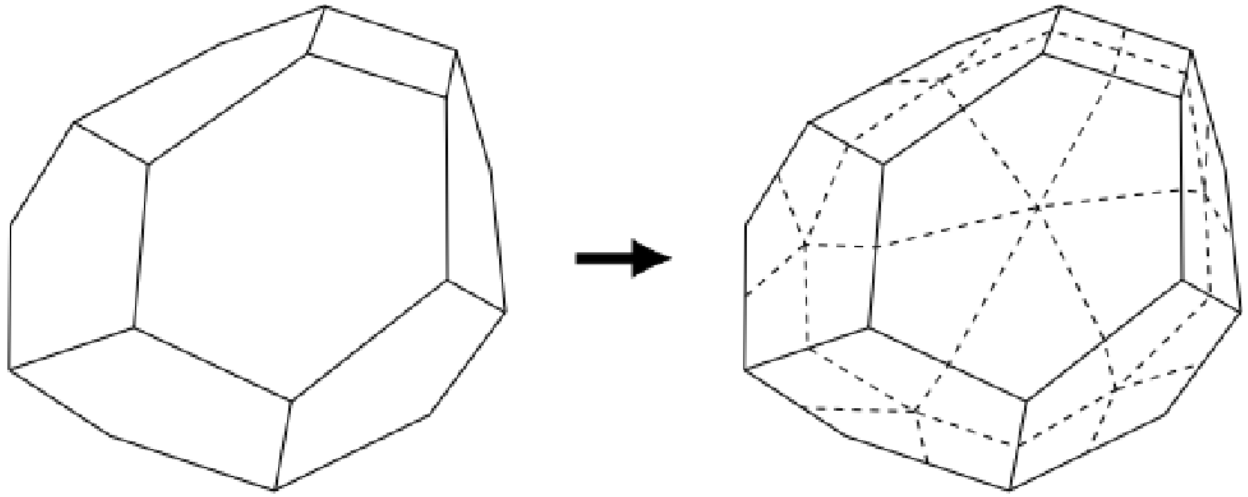
- The solution-based mesh adaption is a feature of Ansys Fluent that allows you to refine and/or coarsen your mesh based on numerical solution data
  - For example, refine the mesh along a bow shock to more accurately predict drag
- This allows users to start with a coarse mesh, and let the solution dictate where additional cells are needed to resolve the flow
- When used properly, this eliminates the need for excessively fine meshes or operating condition specific meshes
  - Saves time in both pre-processing and solving



©2021 ANSYS, Inc.

Polyhedral Unstructured Mesh Adaption (PUMA)

- Generalizes the adaption technique to meshes containing arbitrary polyhedra
- Can refine any 3D type (polyhedra, tetrahedra, hexahedra, etc…) since it does not rely on templates for refinement
- Consumes less memory during mesh refinement when compared to the hanging node method.



©2021 ANSYS, Inc.

Error-Based Adaption Criteria

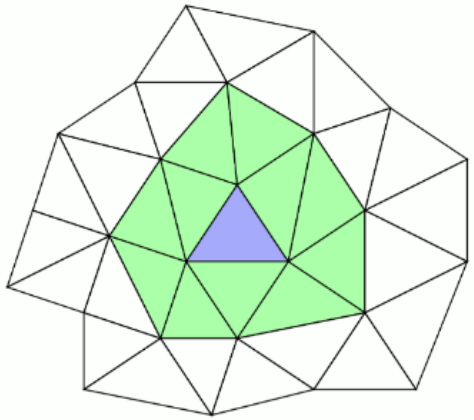
Ansys

Hessian-Based Indicator

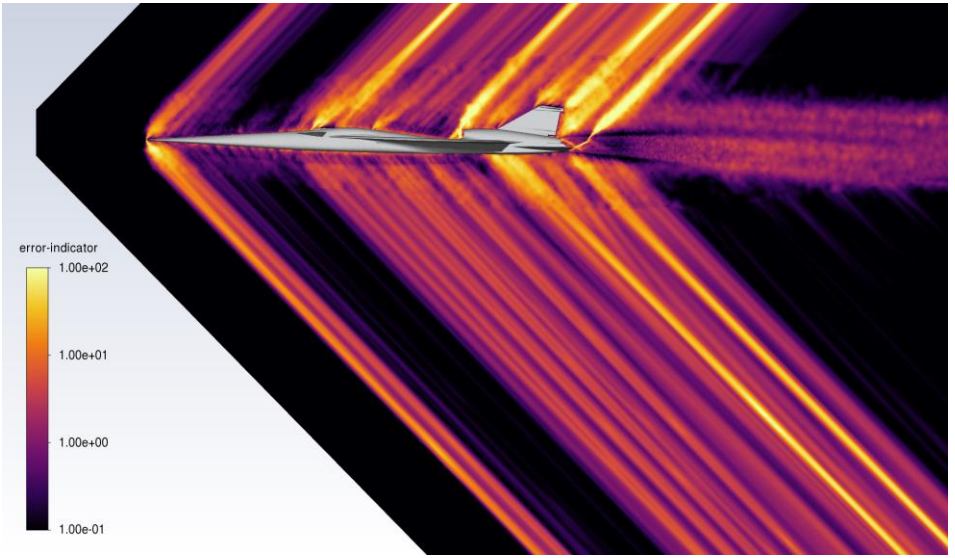
- Error-based adaption marks cells guided by the local errors in the solution relative to mesh size
- Error is estimated from the Hessian matrix, with components obtained through a least squares fit

$$e(x)=|u(x)-u_h(x)|\approx\left|x^T H x\right|,\quad H=\begin{bmatrix}\partial_{xx}u & \partial_{xy}u\\ \partial_{xy}u & \partial_{yy}u\end{bmatrix},$$

- Overall, using a multiscale approach, this provides a weighted estimate for solution error across a variety of element types, scales and flow regimes



Hessian Reconstruction Stencil, Ω_k

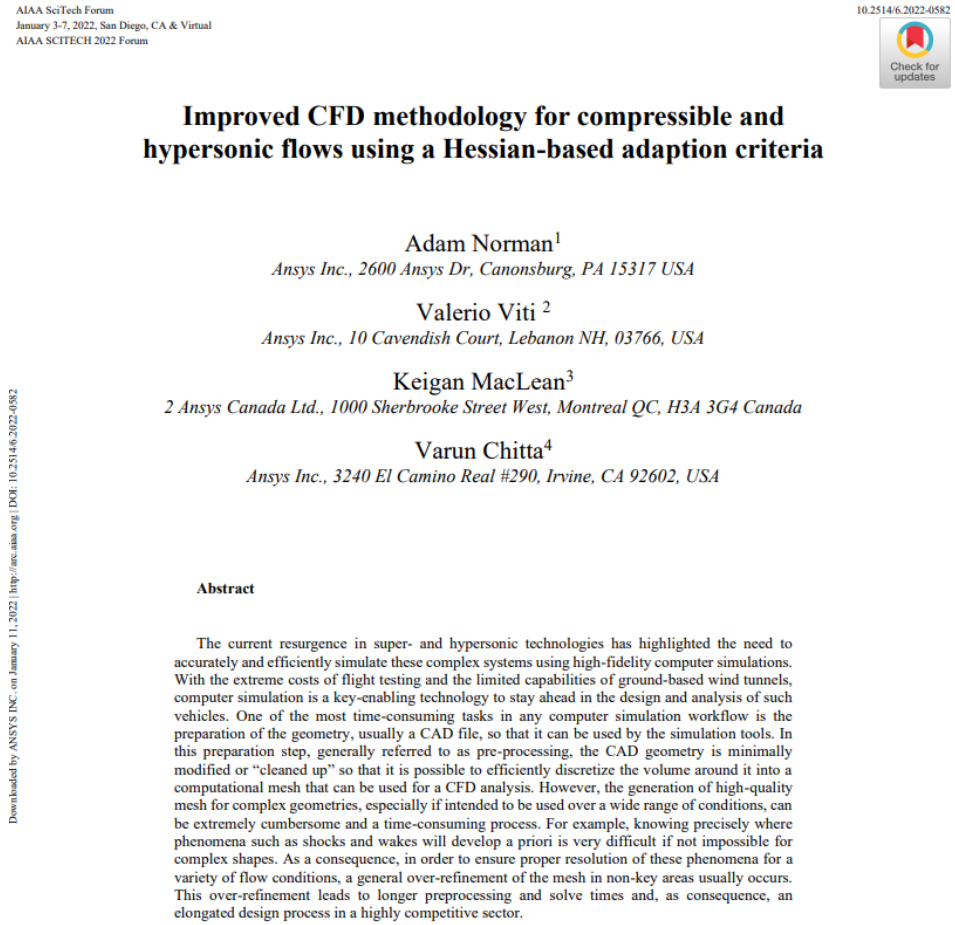


C608 Error-Indicator, $\eta(H_p)$

©2021 ANSYS, Inc.

Hessian-Based Indicator

- *Improved CFD methodology for compressible and hypersonic flows using a Hessian-based adaption criteria*
  - **AIAA 2022-0582**
  - **Presented at SciTech 2022**



AIAA SciTech Forum
January 3-7, 2022, San Diego, CA & Virtual
AIAA SCITECH 2022 Forum

10.2514/6.2022-0582

Check for updates

Improved CFD methodology for compressible and hypersonic flows using a Hessian-based adaption criteria

Adam Norman[1]
Ansys Inc., 2600 Ansys Dr, Canonsburg, PA 15317 USA

Valerio Viti [2]
Ansys Inc., 10 Cavendish Court, Lebanon NH, 03766, USA

Keigan MacLean[3]
2 Ansys Canada Ltd., 1000 Sherbrooke Street West, Montreal QC, H3A 3G4 Canada

Varun Chitta[4]
Ansys Inc., 3240 El Camino Real #290, Irvine, CA 92602, USA

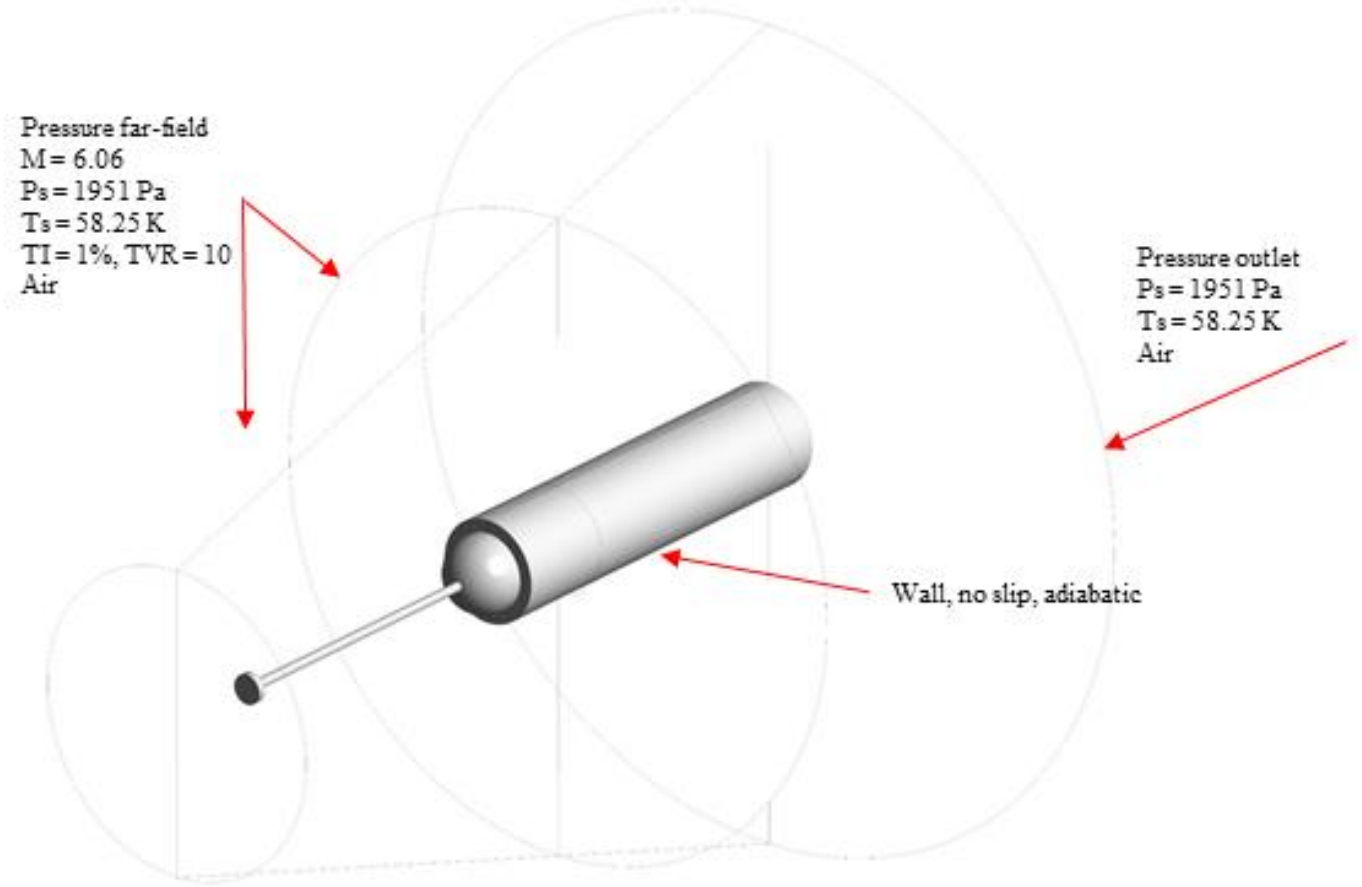
Downloaded by ANSYS INC. on January 11, 2022 | http://arc.aiaa.org | DOI: 10.2514/6.2022-0582

Abstract

The current resurgence in super- and hypersonic technologies has highlighted the need to accurately and efficiently simulate these complex systems using high-fidelity computer simulations. With the extreme costs of flight testing and the limited capabilities of ground-based wind tunnels, computer simulation is a key-enabling technology to stay ahead in the design and analysis of such vehicles. One of the most time-consuming tasks in any computer simulation workflow is the preparation of the geometry, usually a CAD file, so that it can be used by the simulation tools. In this preparation step, generally referred to as pre-processing, the CAD geometry is minimally modified or "cleaned up" so that it is possible to efficiently discretize the volume around it into a computational mesh that can be used for a CFD analysis. However, the generation of high-quality mesh for complex geometries, especially if intended to be used over a wide range of conditions, can be extremely cumbersome and a time-consuming process. For example, knowing precisely where phenomena such as shocks and wakes will develop a priori is very difficult if not impossible for complex shapes. As a consequence, in order to ensure proper resolution of these phenomena for a variety of flow conditions, a general over-refinement of the mesh in non-key areas usually occurs. This over-refinement leads to longer preprocessing and solve times and, as consequence, an elongated design process in a highly competitive sector.

©2021 ANSYS, Inc.

Aerospike Computational Setup

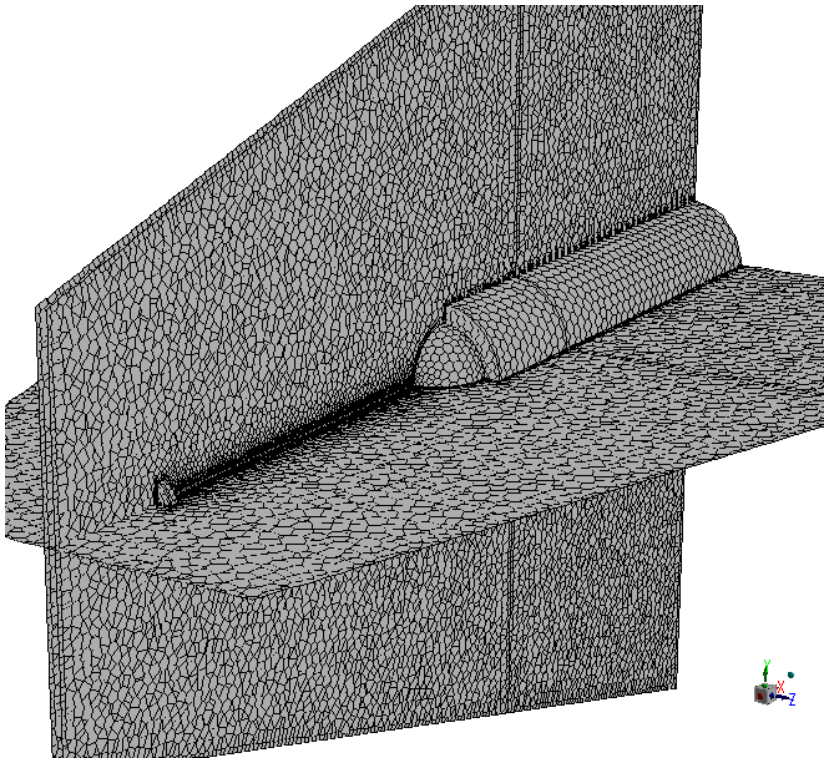


©2021 ANSYS, Inc.

Aerospike Results

- Three meshes were analyzed for the aerospike
  - Coarse (123,000 cells)
  - Adapted (2,440,000 cells)
    - Initial cell count 123,000 cells
  - Fine (15,650,000 cells)

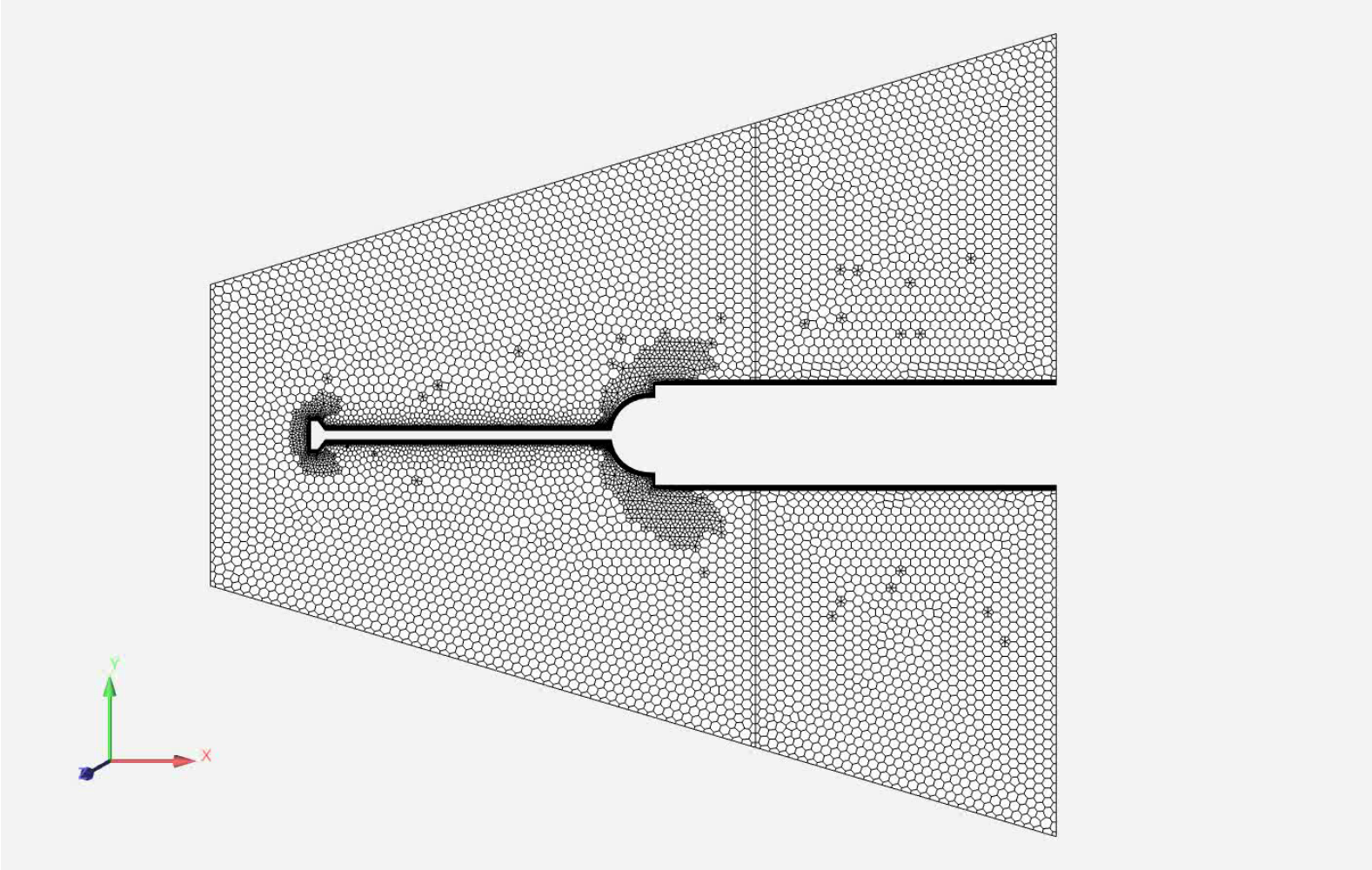
- The mesh was dynamically adapted every 100 iterations



Coarse Mesh

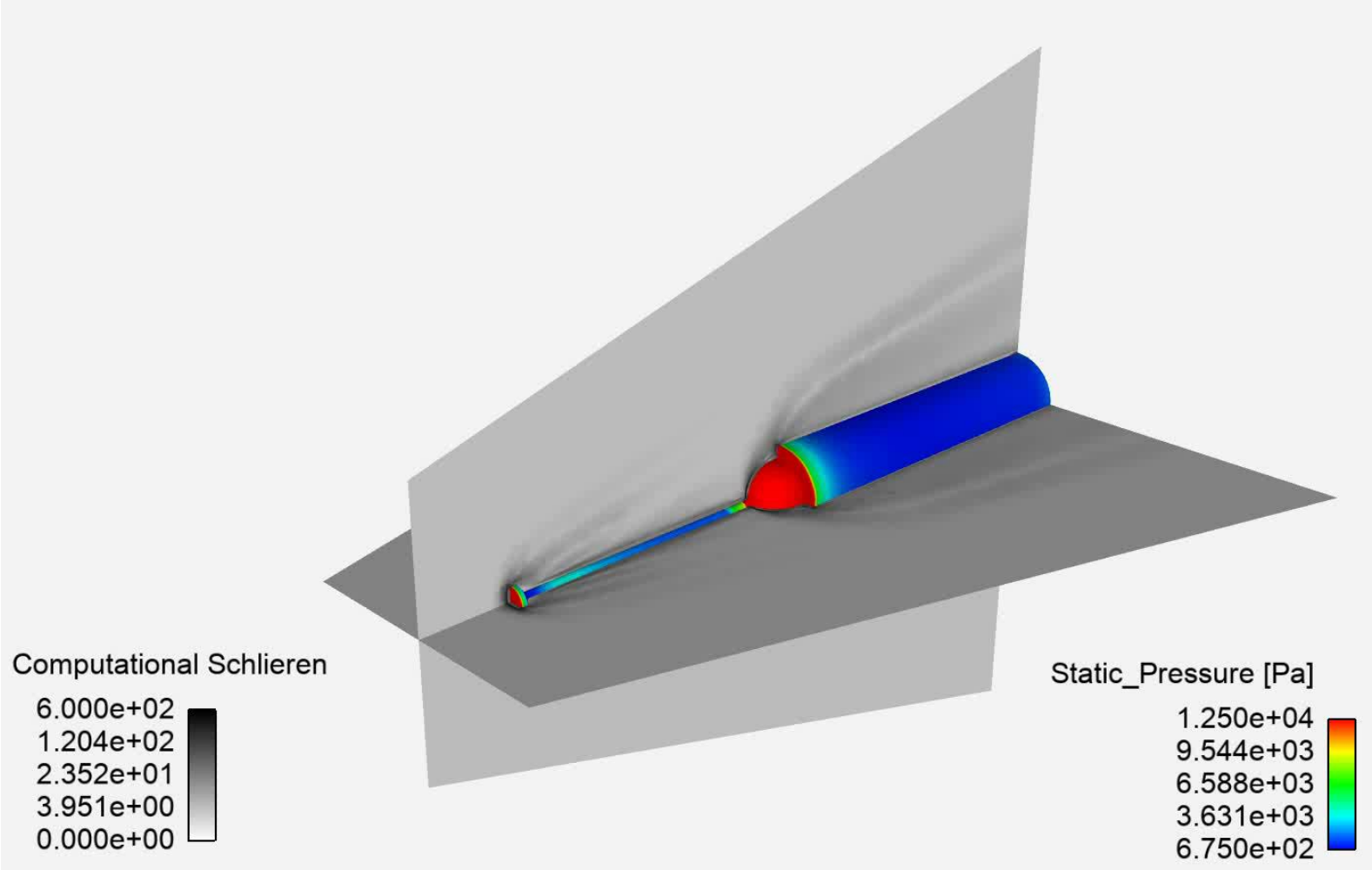
©2021 ANSYS, Inc.

Aerospike Adaption Animations



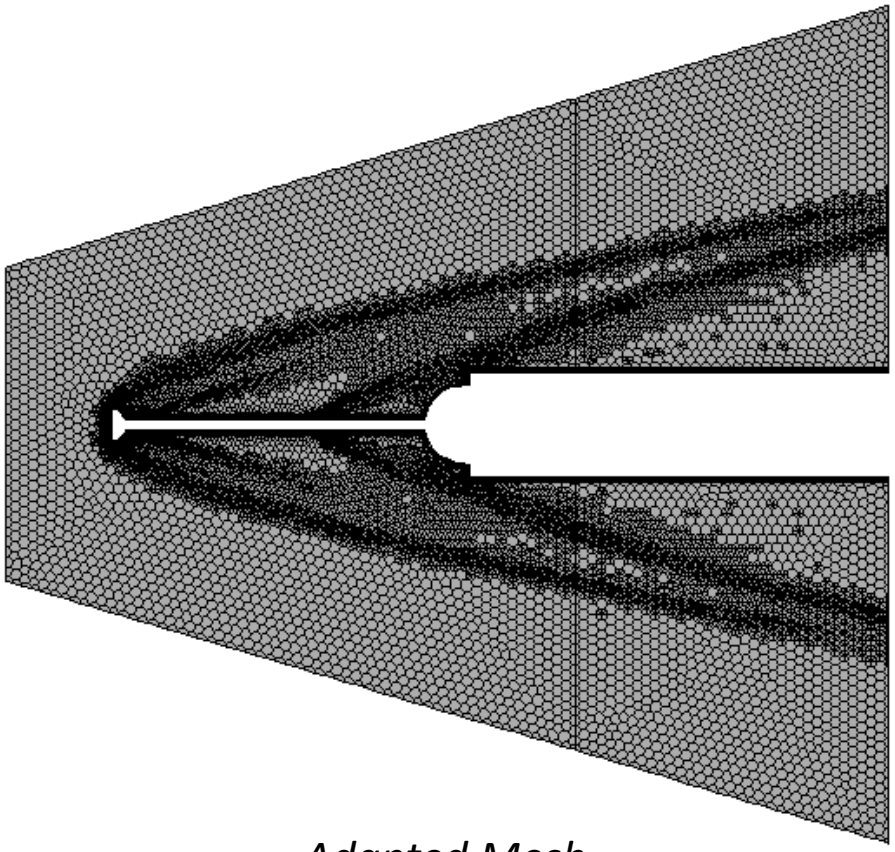
©2021 ANSYS, Inc.

Aerospike Adaption Animations



©2021 ANSYS, Inc.

Aerospike Results



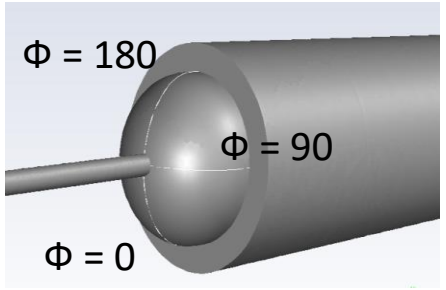
Adapted Mesh

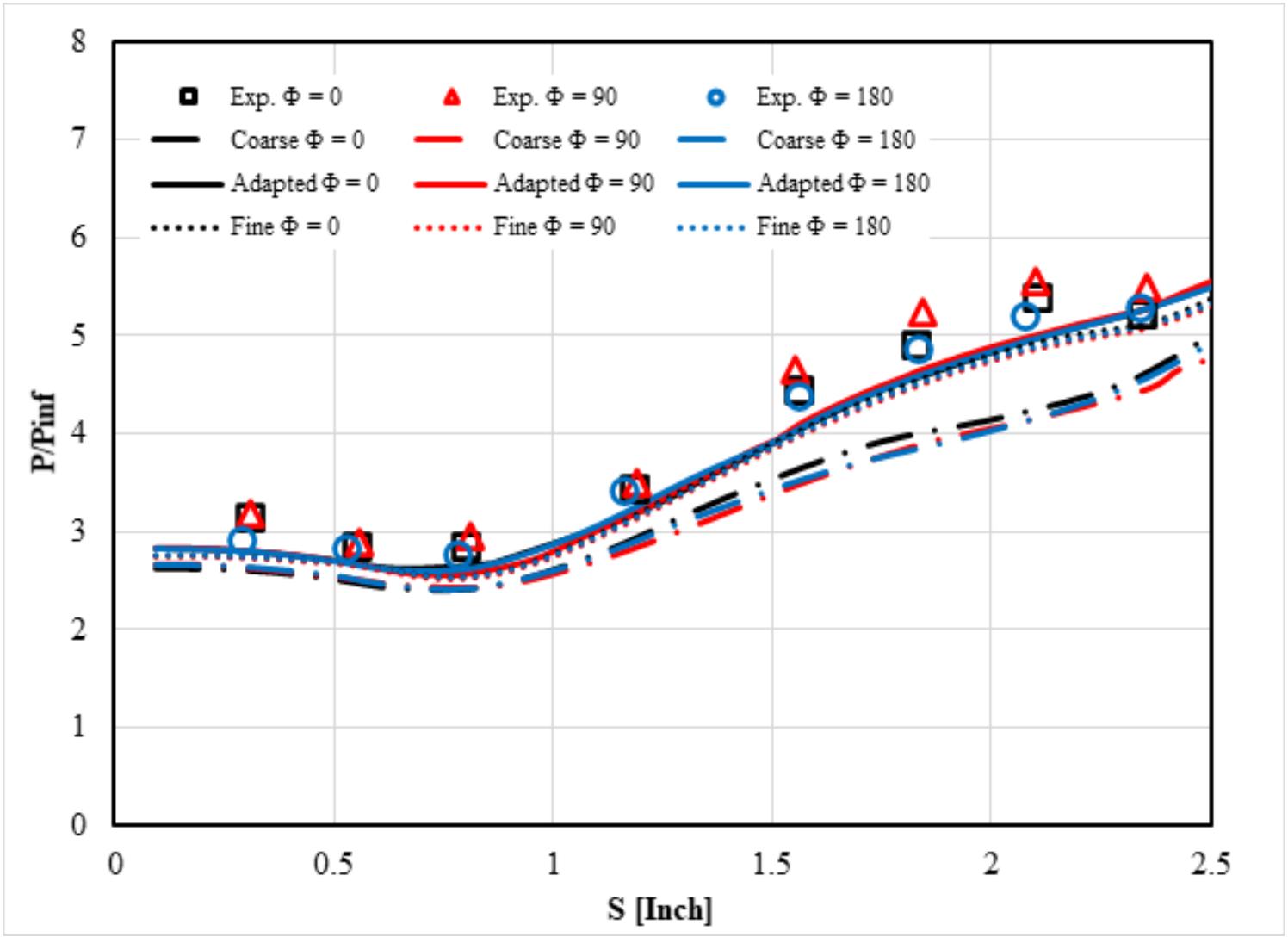
Mesh	Final Cell Count (millions)	Number of Cores	Run Time (s)
Coarse	0.123	64	88
Adapted	2.44	64	3094
Fine	15.65	64	53920

Mesh	Drag Coefficient	% Difference
Coarse	0.238	6.86
Adapted	0.256	-0.18
Fine	0.255	0.00

©2021 ANSYS, Inc.

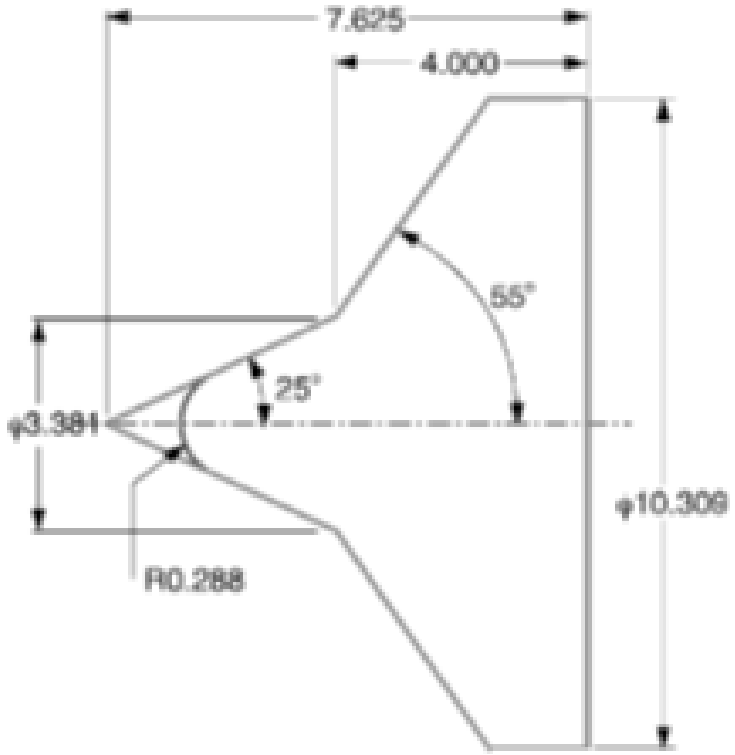
Aerospike Results: Sensor Pressure Coefficient

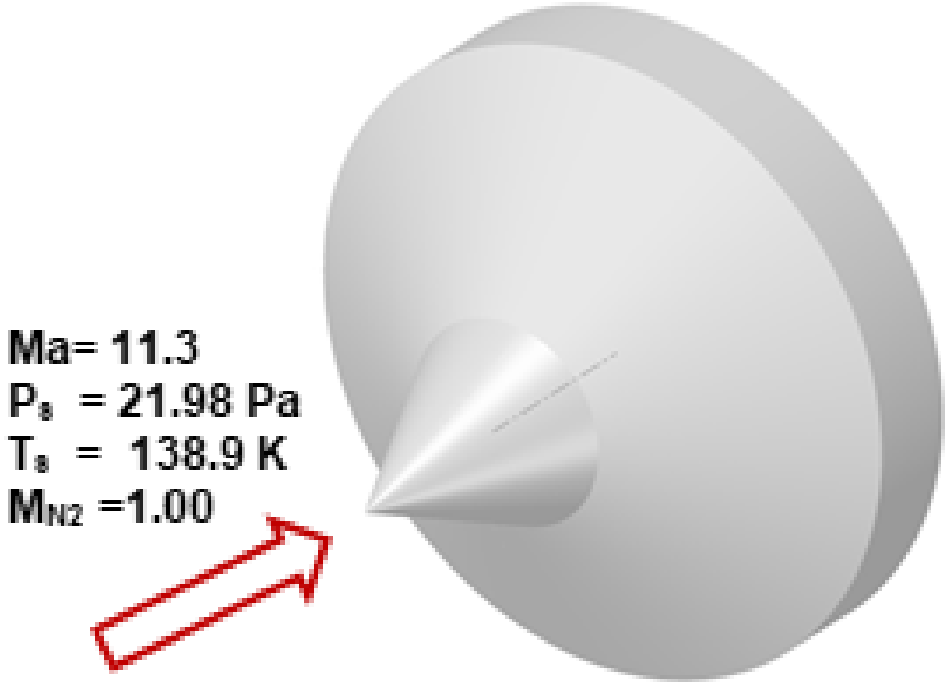




©2021 ANSYS, Inc.

25 – 55 Sharp Double Cone Computational Setup



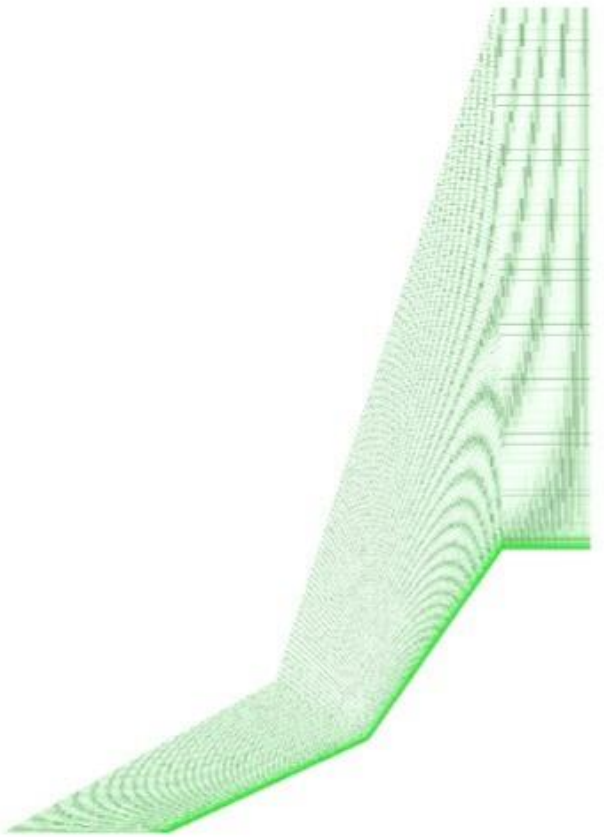


©2021 ANSYS, Inc.

25 – 55 Sharp Double Cone Results

- For the 25 – 55 sharp double cone, four different meshes were used
  - Coarse (24,000 cells)
  - Adapt-2 Levels (65,000 cells)
    - Initial cell count 24,000
  - Adapt-4 Levels (188,000 cells)
    - Initial cell count 24,000
  - Fine (6,380,000 cells)

- The mesh was dynamically adapted every 200 iterations

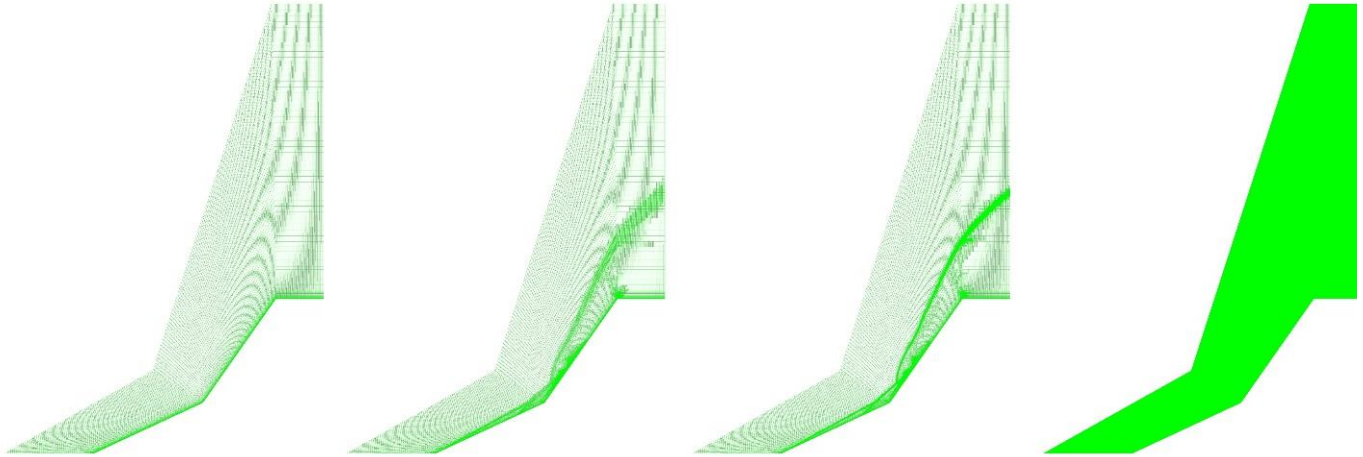


Coarse Mesh

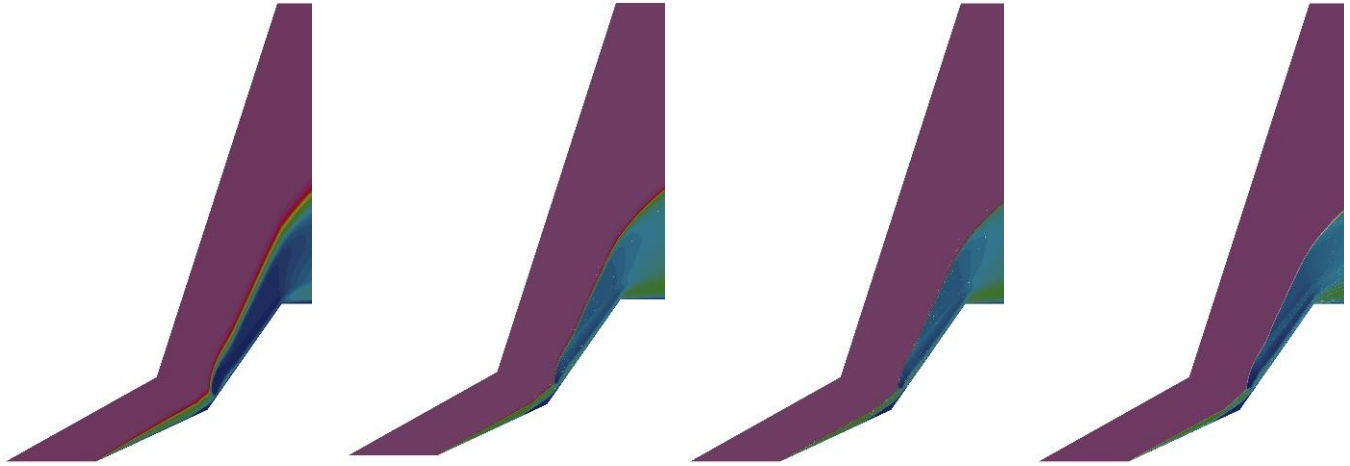
©2021 ANSYS, Inc.

25 – 55 Sharp Double Cone Results

Computational Mesh



Mach Contours



Coarse Mesh
24,000 cells

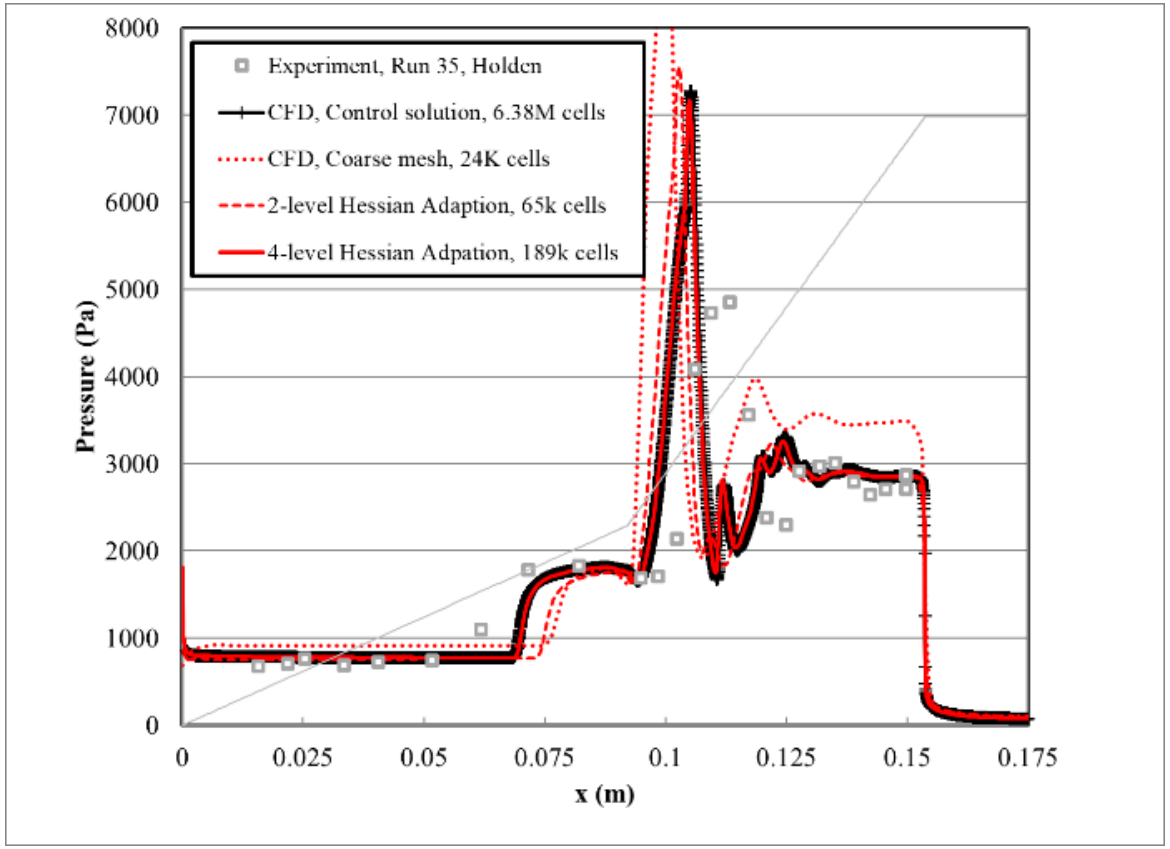
Adapt-2 Mesh
65,000 cells

Adapt-4 Mesh
188,000 cells

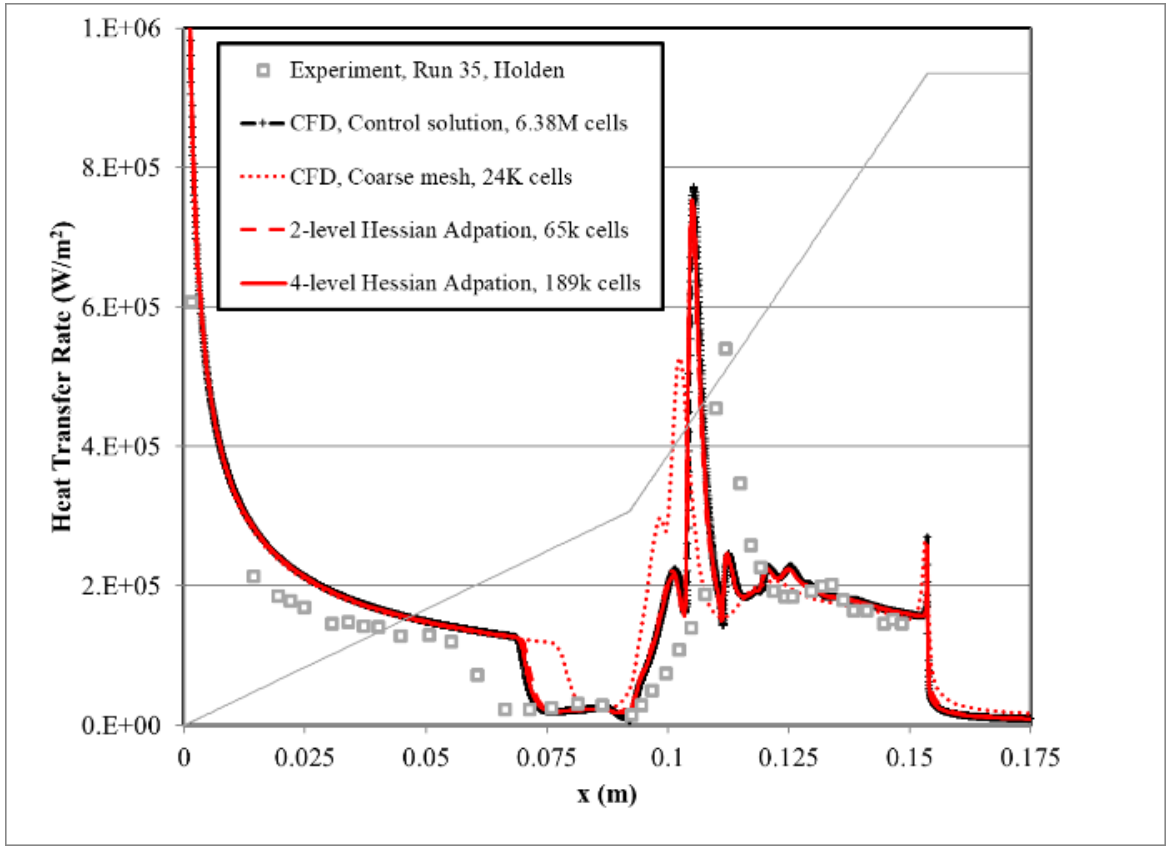
Fine Mesh
6,380,000 cells

©2021 ANSYS, Inc.

25 – 55 Sharp Double Cone Results: Pressure and Heat Flux



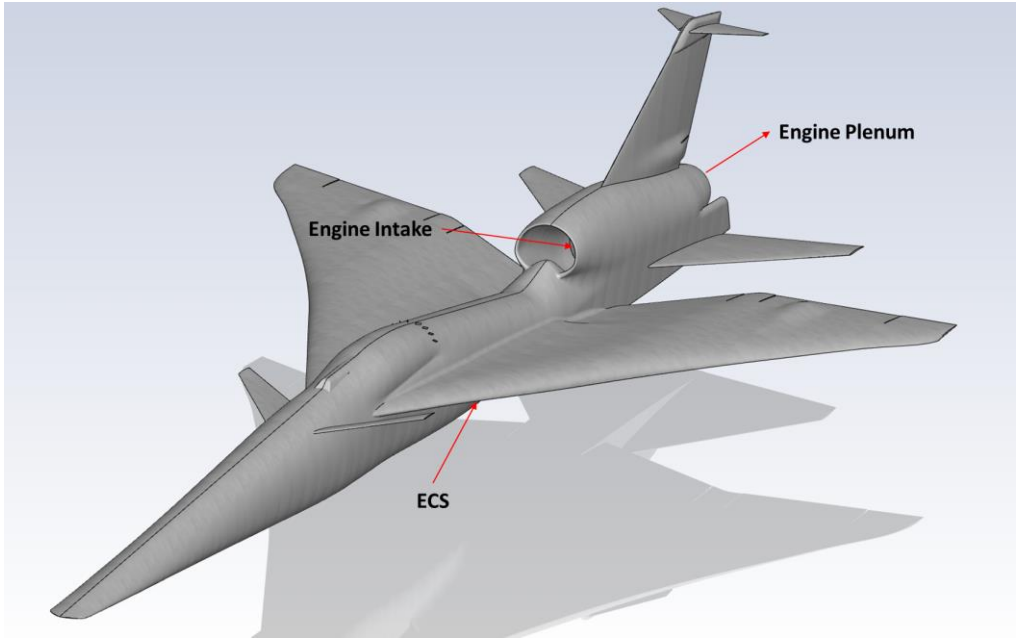
Pressure



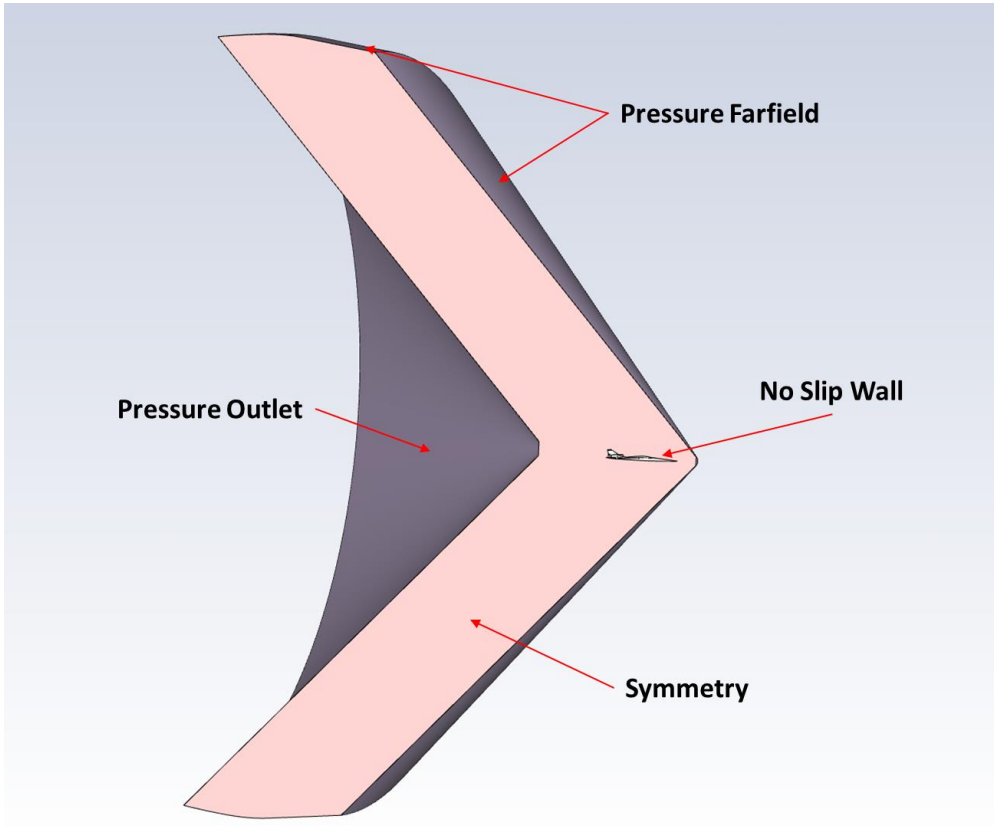
Heat Flux

©2021 ANSYS, Inc.

C608 Computational Setup



C608 configuration with ECS boundary conditions

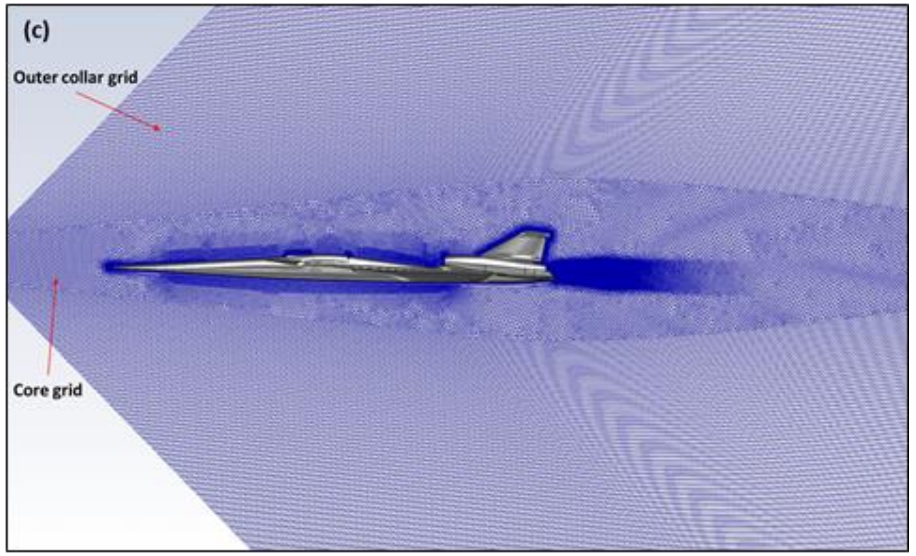


Computational domain for C608

©2021 ANSYS, Inc.

C608 Results

- On the C608 geometry, five different meshes were used
  - Baseline (30M cells)
  - Adapt-1 (47M cells)
  - Adapt-2 (99M cells)
  - Adapt-3 (127M cells)
  - Adapt-4 (148M cells)



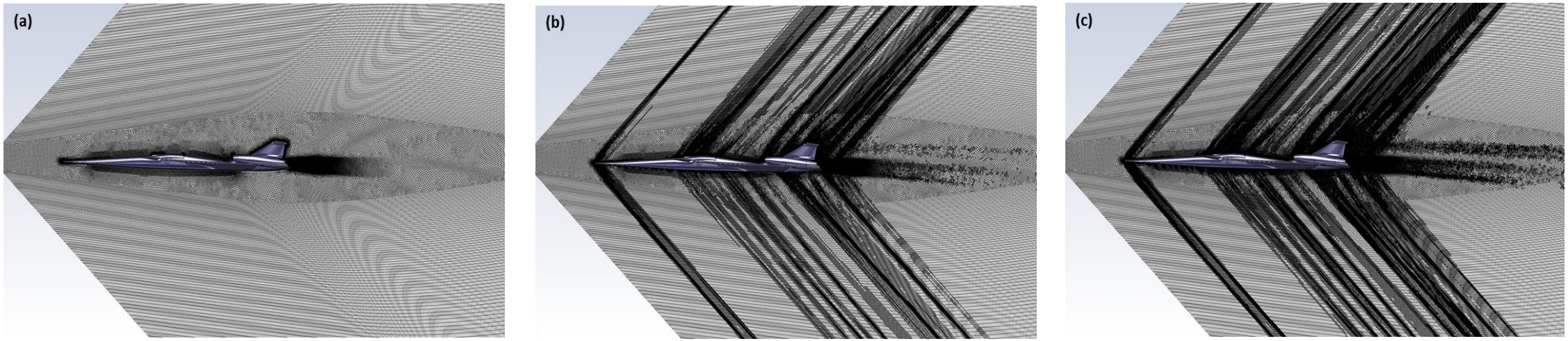
Baseline Mesh

- The mesh was dynamically adapted every 200 iterations

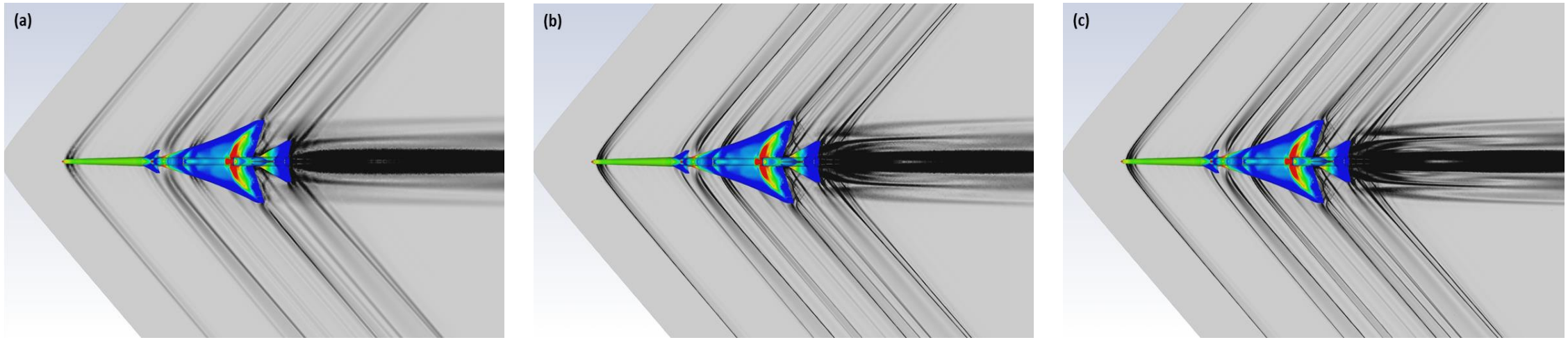
©2021 ANSYS, Inc.

C608 Results

Computational Mesh



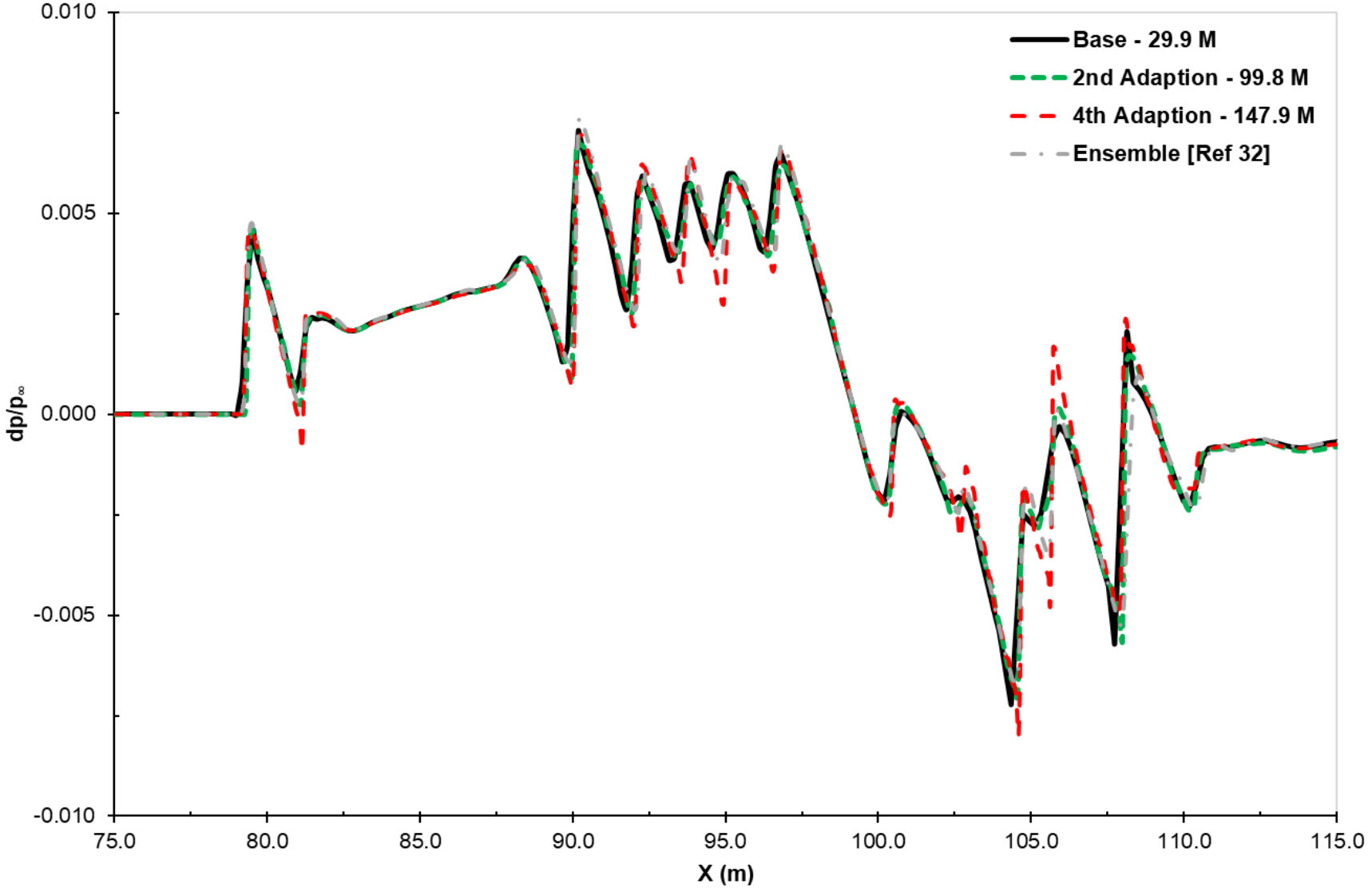
Computational Schlieren and Pressure Coefficient



Baseline Mesh (30M cells) | *Adapt-2 Mesh (99M cells)* | *Adapt-4 Mesh (148M cells)*

©2021 ANSYS, Inc.

C608 Results: Nearfield Normalized Pressure Signatures



Pressure measured from a horizontal line located three body lengths directly below the aircraft in the symmetry plane

©2021 ANSYS, Inc.

Standard Deviation Adaption Criteria

Ansys

Workflow

AMR: Cell Marking Method

- For every selected flow quantity ‘i’, corresponding value of $f\left(x_{cell}^{i}\right)$ is calculated as

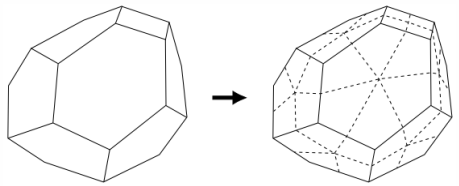
$$f\left(x_{cell}^{i}\right) = 1\,, \quad If\ x_{cell}^{i} \geq \bar{x^{i}} + \alpha\sigma_{x^{i}}$$
$$= 0\,, \quad If\ x_{cell}^{i} < \bar{x^{i}} + \alpha\sigma_{x^{i}}$$

$x_{cell}^{i} = Cell\ value\ of\ a\ given\ flow\ quantity$
$\bar{x^{i}} = Spatially\ avg.value\ of\ the\ flow\ quantity$
$\alpha = User\ controlled\ scaling\ parameter$
$\sigma_{x^{i}} = Standard\ deviation\ of\ the\ \ quantity$

- Combined expression is derived as
  - ‘True’ if $f\left(x_{cell}^{i}\right)$ is ‘1’ for at least one of the selected flow variable ‘1’ else ‘False’

Cell Adaption Method

- Adapted using inbuilt Polyhedral Unstructured Mesh Adaption (**PUMA**) algorithm : No hanging nodes



	Refinement (Properties of Child cell)	**Coarsening (Properties of merged cell)**
Intensive Properties	Same as parent cell	Volume-weighted average of merging cells
Extensive Properties	Distributed among child cells as a fn. of their volume	Summed up across merging cells
Flux across faces	Calculated using properties of child cells	Flux values are calculated using newly computed cell quantities

©2021 ANSYS, Inc.

Standard Deviation Based Indicator

- *Solution-based mesh adaption criteria development for accelerating flame tracking simulations*
  - **GT2022-82620**
  - **Presented at IGTI 2022**

Proceedings of ASME Turbo Expo 2022
Turbomachinery Technical Conference and Exposition
GT2022
June 13-17, 2022, Rotterdam, The Netherlands

GT2022-82620

SOLUTION-BASED MESH ADAPTION CRITERIA DEVELOPMENT FOR ACCELERATING FLAME TRACKING SIMULATIONS

Sourabh Shrivastava
Ansys Inc.
Pune, India

Ishan Verma
Ansys Inc.
Pune, India

Rakesh Yadav
Ansys Inc.
San Diego, US

Pravin Nakod
Ansys Inc.
Pune, India

ABSTRACT

Accurate flame tracking plays a vital role in predicting the combustion characteristics of a system. This is even more critical for systems that evolve over time. Predicting relight performance of an aero combustor, predicting flame propagation due to gas leakage from a storage tank or during the thermal runaway of batteries, are some examples of such dynamic systems. Predicting accurate flame position also plays an important role in deriving the correct pollutant formation rate from a combustion system.

The challenge with flame tracking through a 3D computational fluid dynamics (CFD) simulation comes from the requirement to have a good resolution of gradients along the flame front. This requirement can push the overall mesh count of any industrial cases to a very large value (several million-mesh count). Further, the global drive towards using hydrogen or hydrogen blended fuels for different combustion applications pushes the limits on having even finer cells since hydrogen is a fast-burning fuel flow configurations. The functions based on selected flow-quantities derived from these studies are then tested to predict the results for the more complex set of published flames like the Engine Combustion Network (ECN) spray flame and Knowledge for Ignition, Acoustics and Instabilities (KIAI) five-burner configuration (liquid and gas fuel). Derived adaption criteria are found to predict the correct flame tracking behavior in terms of transient evolution of flame front, flame propagation, and ignition timing of burners. The parameters used for the study are identified keeping genericity as the key point, and thus making sure that the derived adaption functions can be applied across different types of fuel blends, combustion systems (gaseous or liquid fuel-based systems) and combustion models, for example species transport or mixture fraction-based models.

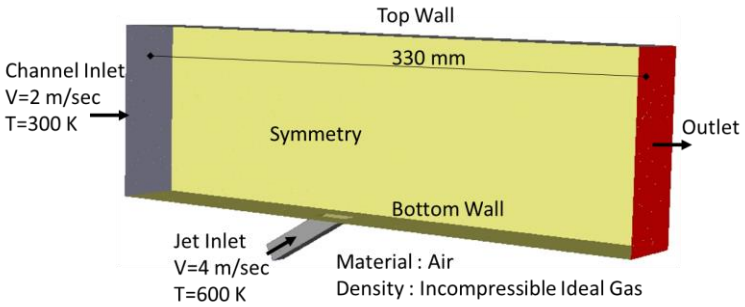
Keywords: Dynamic Mesh Adaption, Solution Based Mesh Adaption, Flame Tracking, Relight Modeling

©2021 ANSYS, Inc.

Jet in Cross Flow

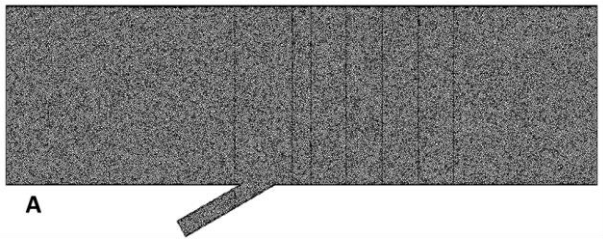
Aim: To evaluate accuracy of solution-based AMR against refined static mesh.

Domain



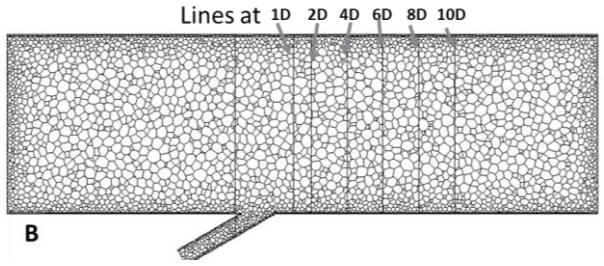
Fine Mesh

Mesh Count 2 M



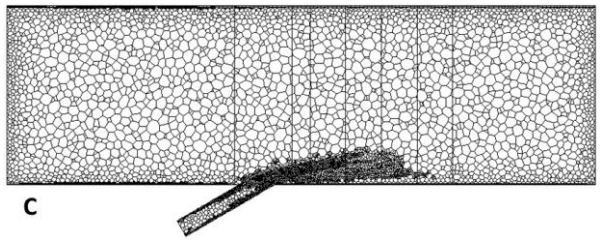
Coarse Mesh

Mesh Count 115K

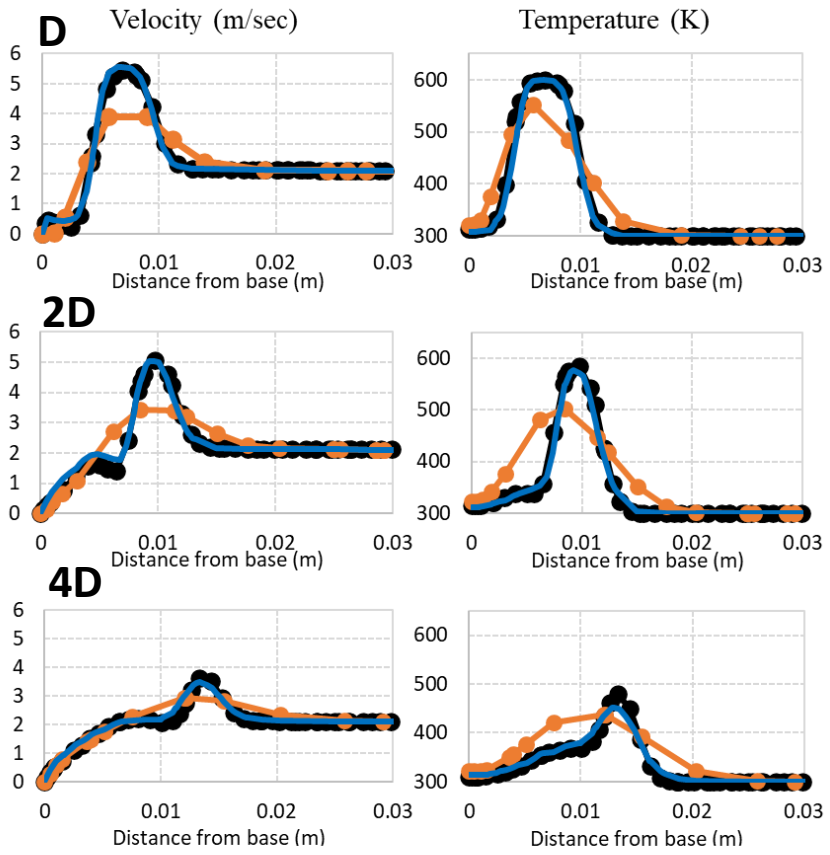


Adapted Mesh

Mesh Count 306K



Results



Velocity and temperature profiles for three meshes, along the lines normal to the walls, located at specified axial distance from jet pipe connection

©2021 ANSYS, Inc.

Premixed Hydrogen-air Flame Propagation In Closed Duct

- Flame ignited at t=0 sec, by patching progress variable = 0 in spark region
- Flame propagation capture using iso-surface of progress variable = 0.5
- Minimum mesh edge length for adaption kept as 0.1 mm

Fine Mesh

Mesh count 373K



Temperature Field

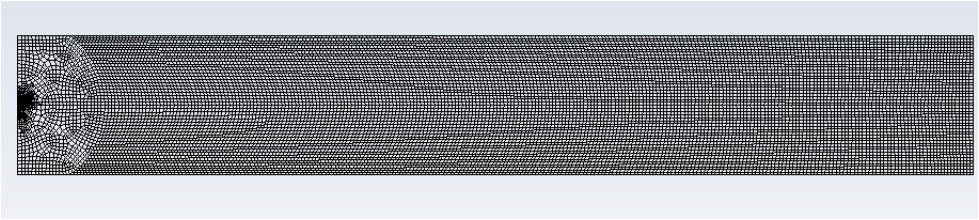
Coarse Mesh

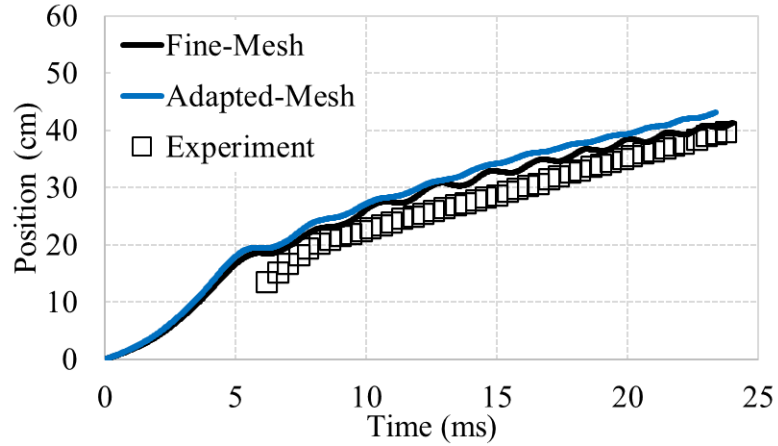
Initial Mesh count 13K



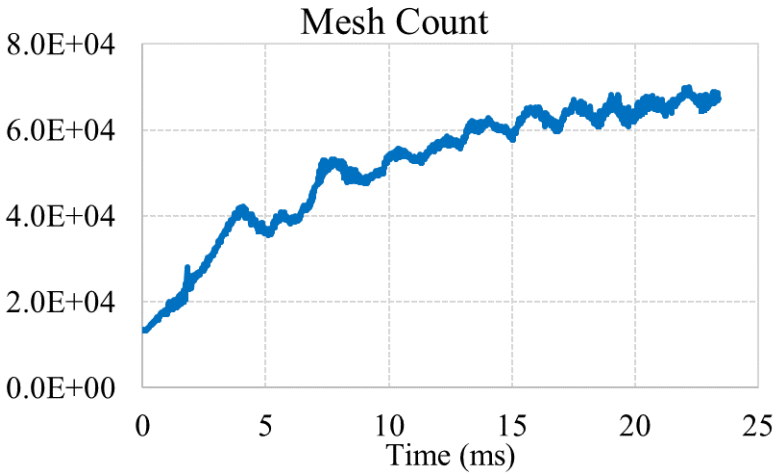
Mesh

Mesh behavior with adaption





Comparison of flame front position with time, with respect to experimental data



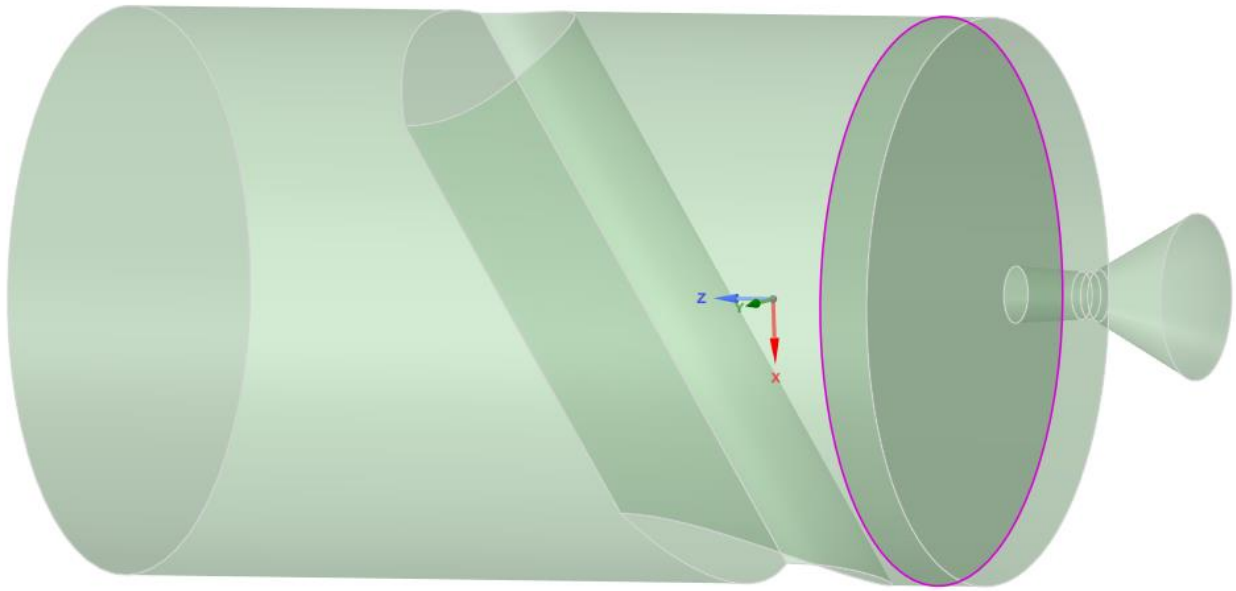
Increase in cell count with time

©2021 ANSYS, Inc.

Supersonic Rotor Impingement Generic Use Case

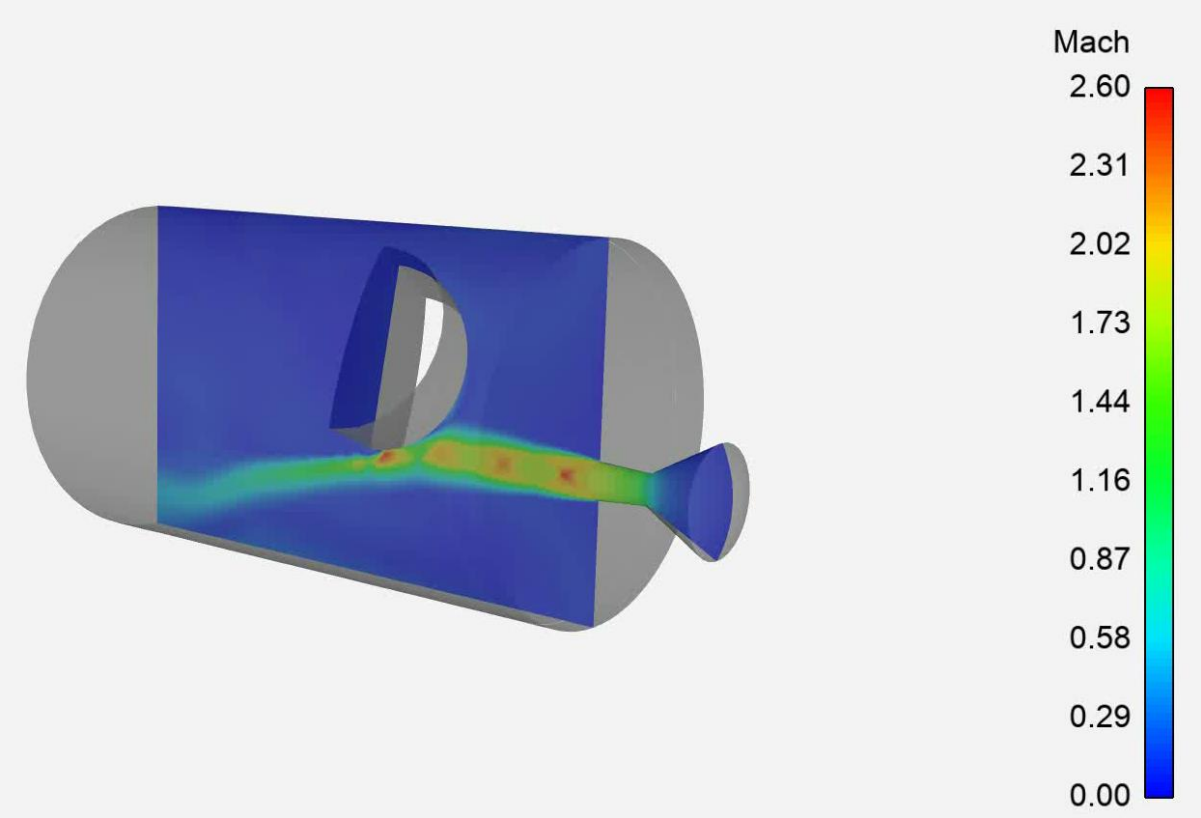
Case Parameters

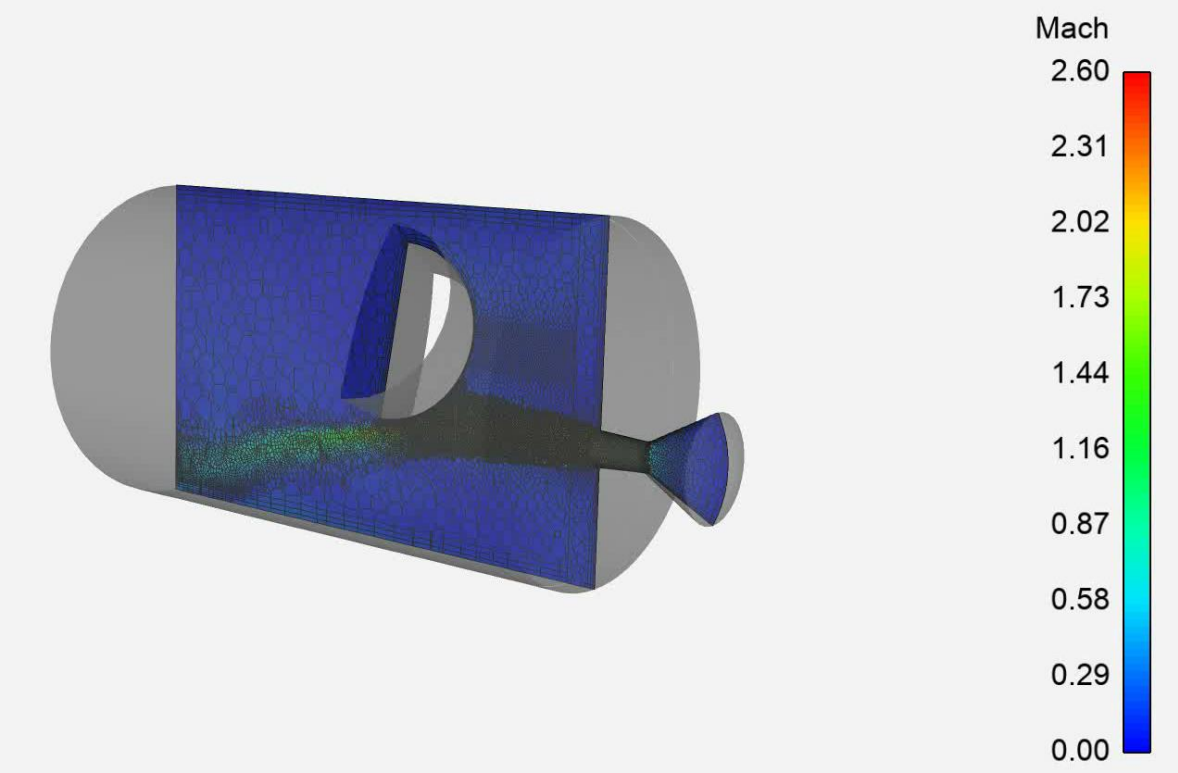
- 3,400 RPM
- Inlet Pressure: 700,000 Pa
- Outlet Pressure: 1 atm



©2021 ANSYS, Inc.

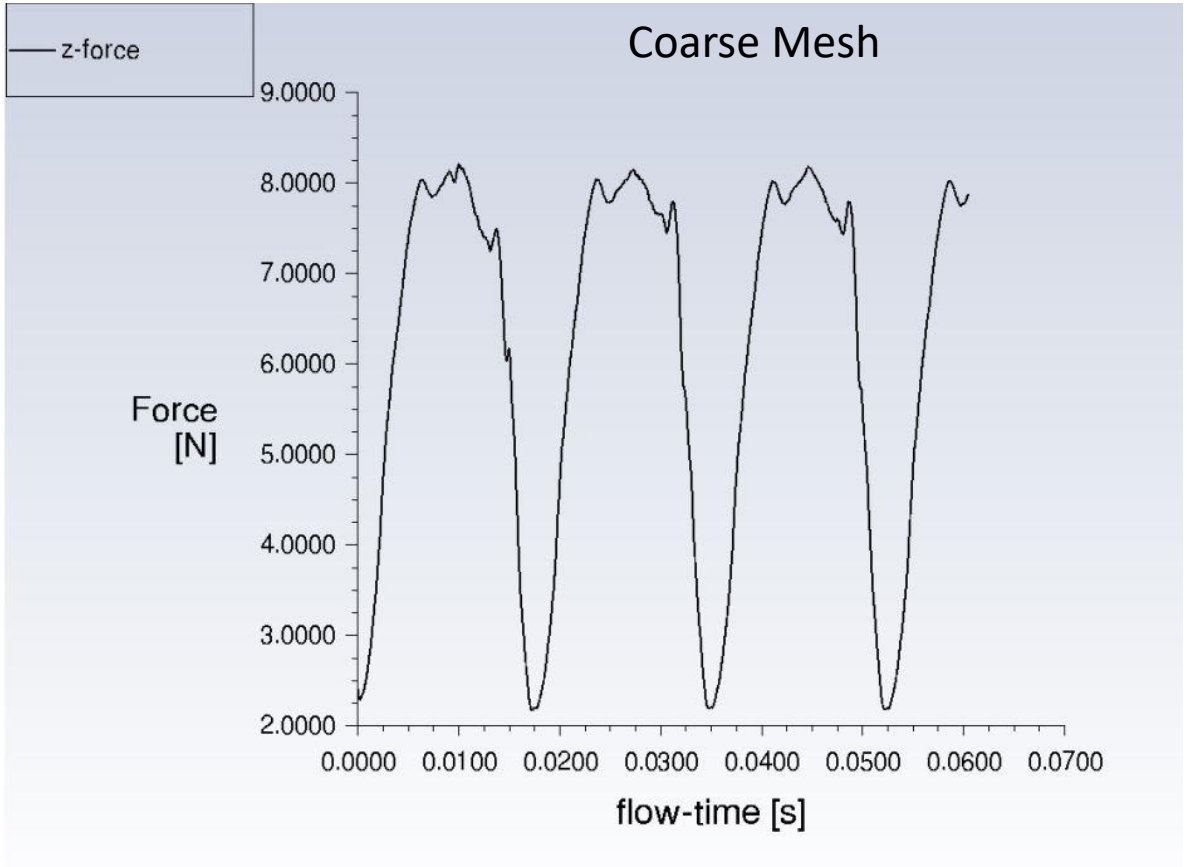
Supersonic Rotor Impingement Generic Use Case

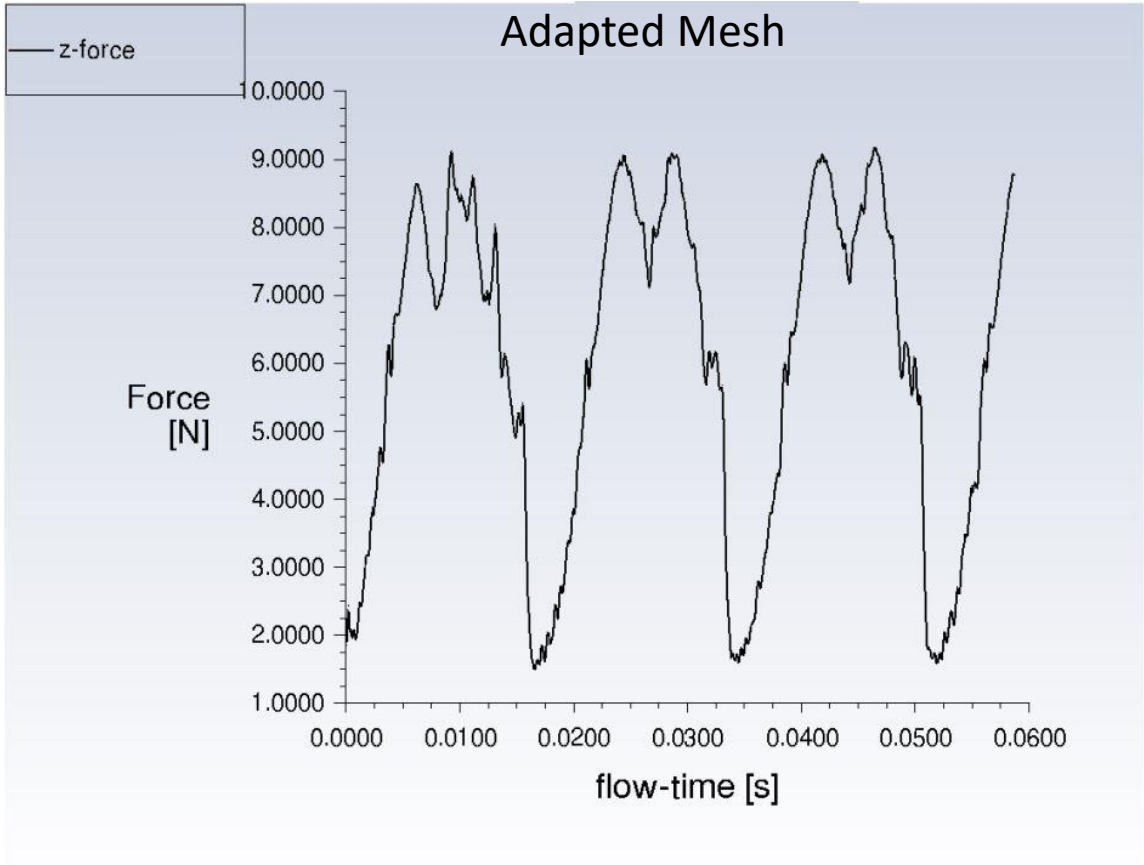




©2021 ANSYS, Inc.

Supersonic Rotor Impingement Generic Use Case

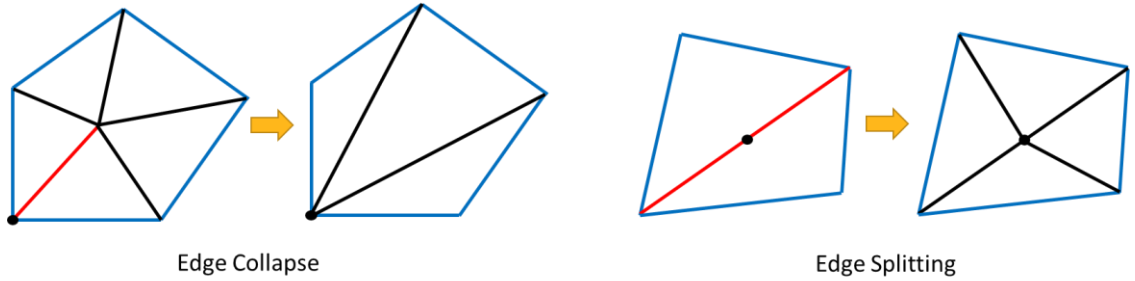




©2021 ANSYS, Inc.

Anisotropic Mesh Adaption

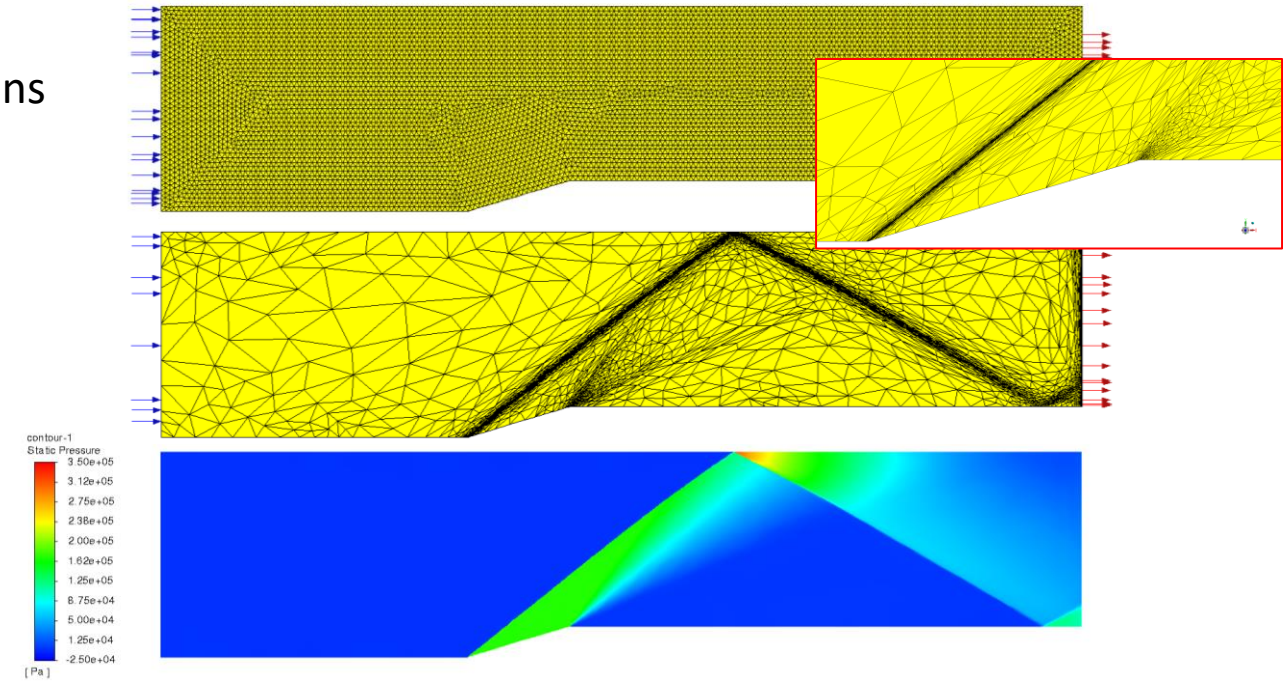
Anisotropic Adaption



- New adaption method currently in development
  - Integrating capabilities previously offered from OptiGrid

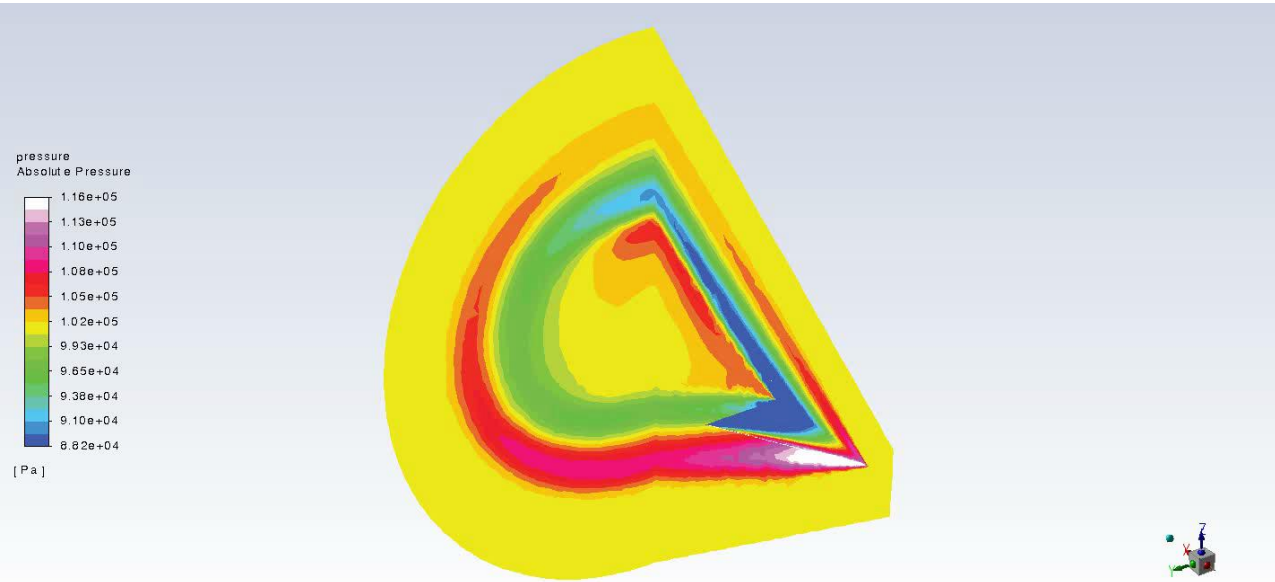
- Anisotropic, Unstructured, Metric-Based mesh adaption
  - Hessian based error estimates for capturing flow features
  - Targets ideal scaling and stretching of the mesh cells
  - Applies series of refine/coarsen/swap/move operations

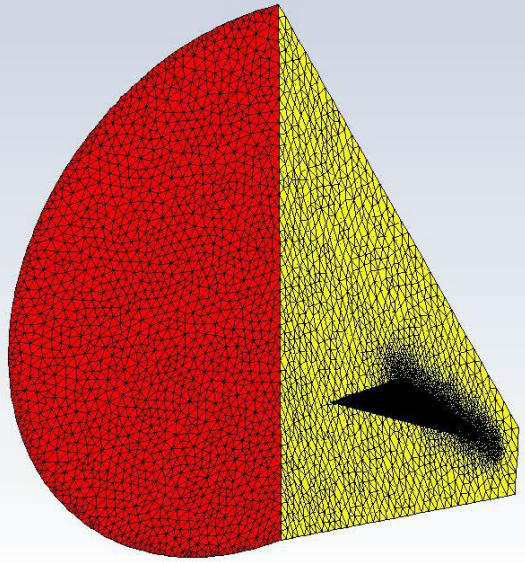
- Additional features:
  - L^p norm for multiscale phenomena
  - Metric controls for gradation, smoothing, limiting
  - Tetrahedral and hybrid mesh adaption



©2021 ANSYS, Inc.

Anisotropic Adaption – Delta Wing





©2021 ANSYS, Inc.


Ansys