
INNOVATION. UNLEASHED.
Ansys. Engineering What's Ahead.

Minimizing the Preprocessing Time Sink: A Guided, Fluent Meshing Workflow for the Aerospace and Defense Industry

Varun Chitta, PhD

Application Engineer, Fluids

Advanced Modeling and Simulation (AMS) Seminar Series
NASA Ames Research Center, August 17, 2021

Ansys

Your Presenter Today



Varun Chitta

Application Engineer, Fluids at Ansys

About Me:

- **Work closely with Aerospace and Automotive industry customers for deployment and adoption of fluids and multiphysics solutions**
- **Background: Ph.D. in aerodynamics and turbulence modeling**

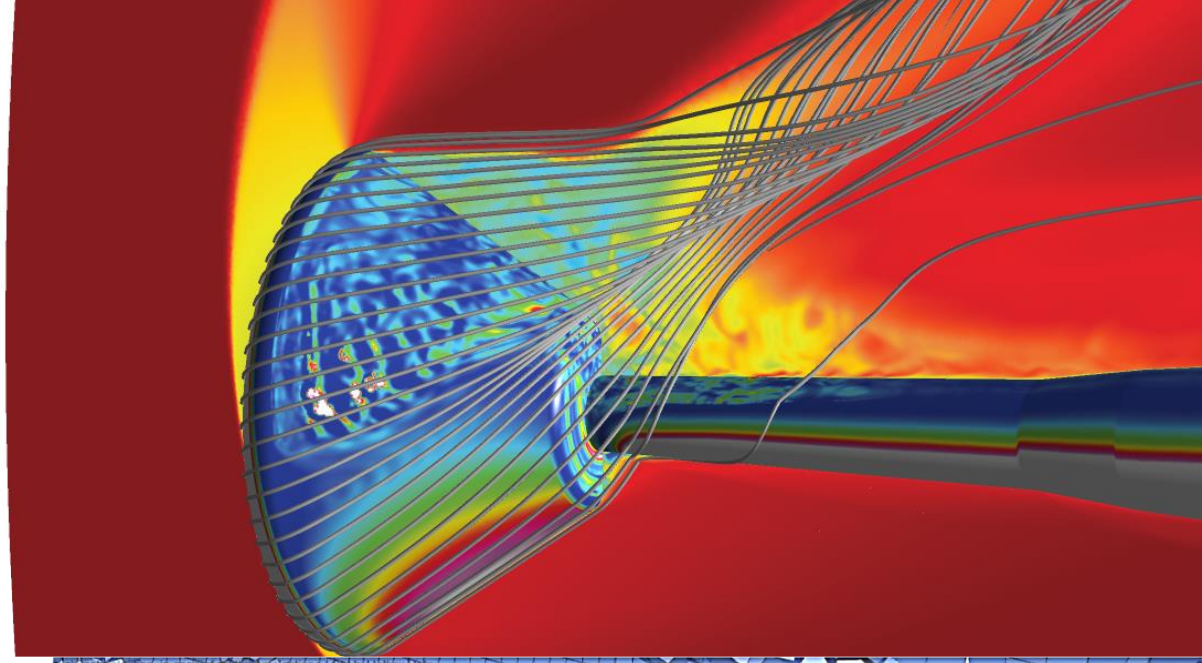
Let's stay connected!

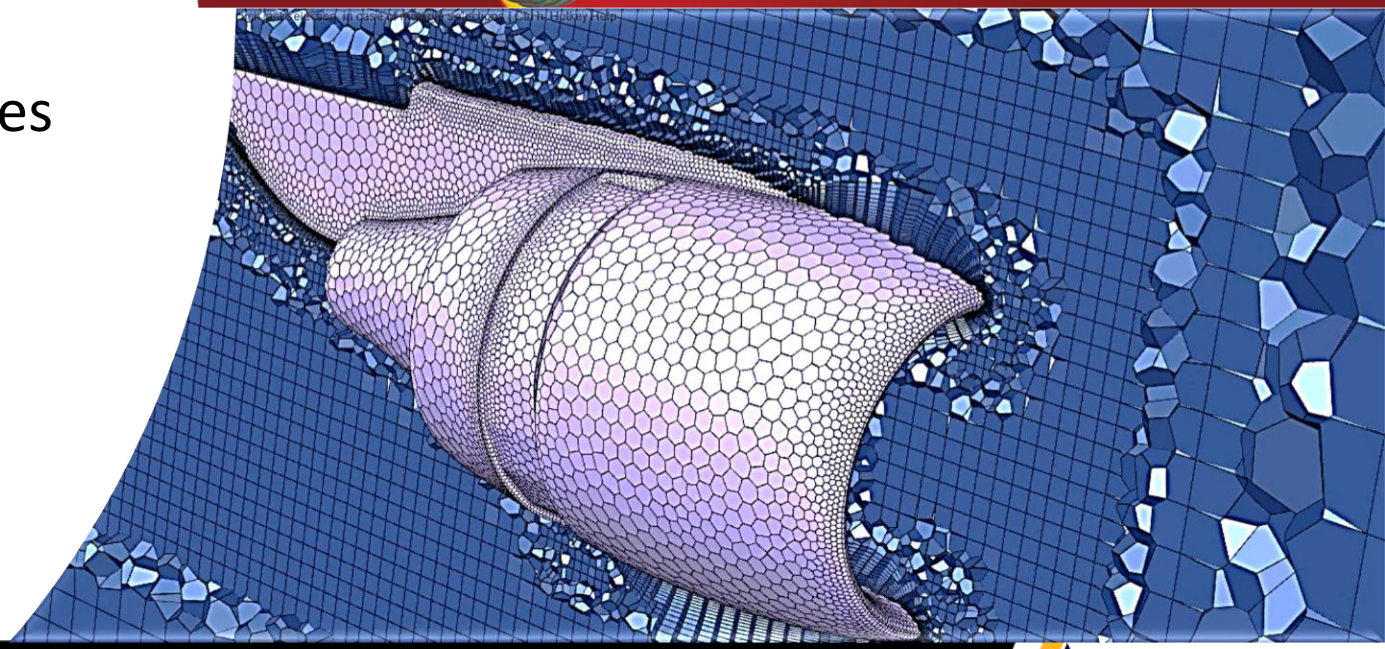
varun.chitta@ansys.com

©2020 ANSYS, Inc. / Confidential

Agenda

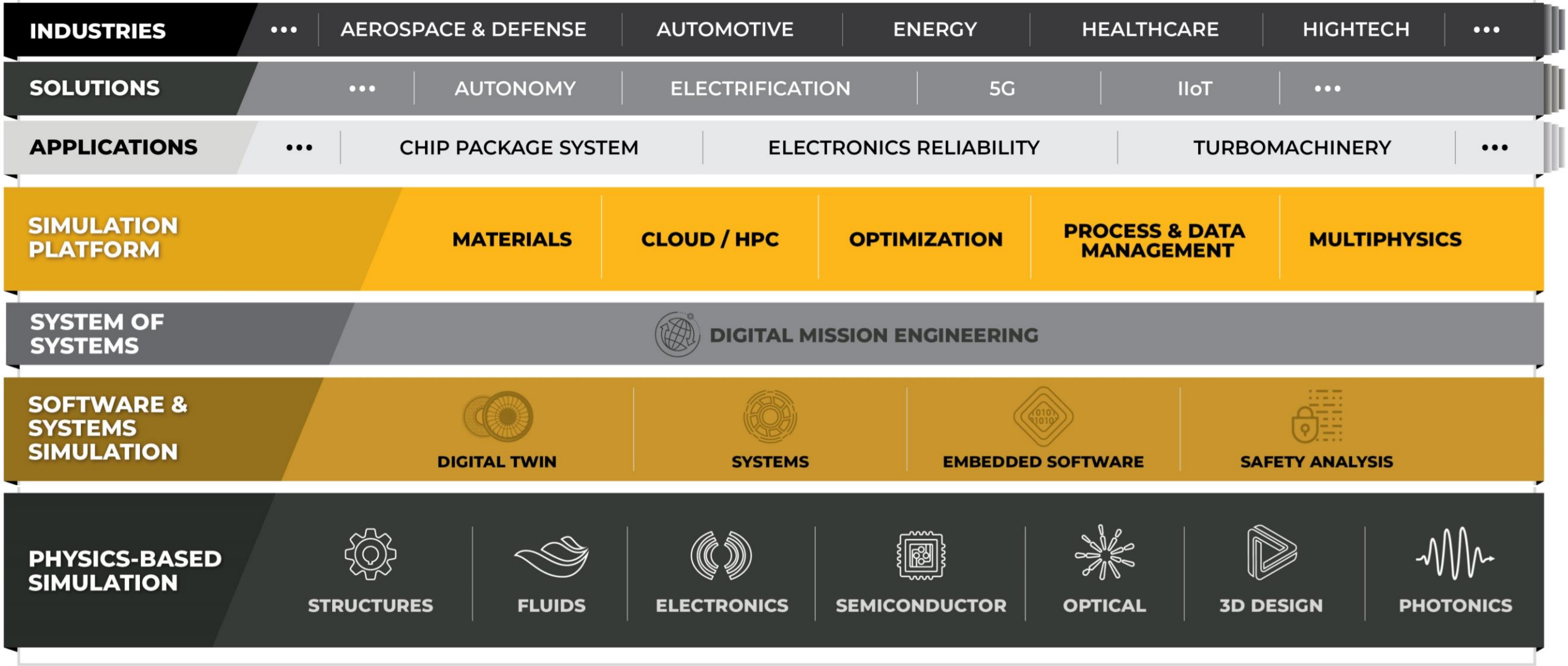
- Fluids meshing tools: Overview
- Fluent meshing workflows
  - WTM
  - FTM
- Mosaic technology
- Fluent meshing workflows: Case studies
  - Live demonstration: WTM workflow
- Automated workflows





©2020 ANSYS, Inc. / Confidential

Best-Of-Breed Simulation Across All Major Physics

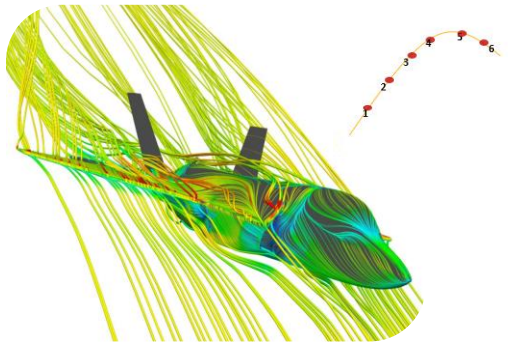


©2020 ANSYS, Inc. / Confidential

Ansys Driving Simulation Themes For Aerospace & Defense Industry

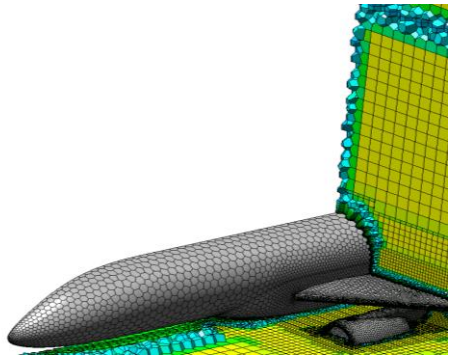
Improve engineering productivity while improving trust in simulation technology

Workflow



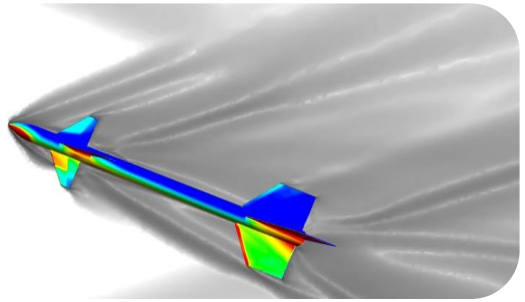
- Dedicated external aero workflow: parametric, scriptable
- End-to-end workflows

Preprocessing



- Mosaic technology
- Parallel meshing
- Clean/dirty CAD
- Integrated in workflow

Solver



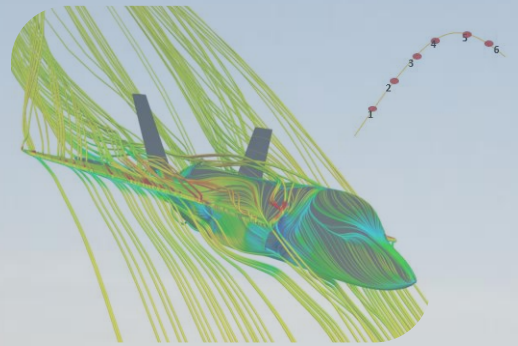
- Improved robustness: HSN, PMN
- Built-in NASA 9-coeff curve fits for material properties
- Built-in partial slip-wall BC
- Built-in surface ablation model

©2020 ANSYS, Inc. / Confidential

Ansys Driving Simulation Themes For Aerospace & Defense Industry

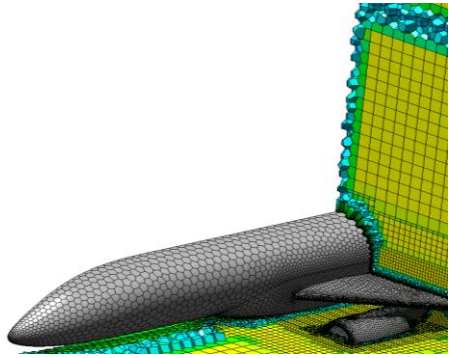
Improve engineering productivity while improving trust in simulation technology

Workflow



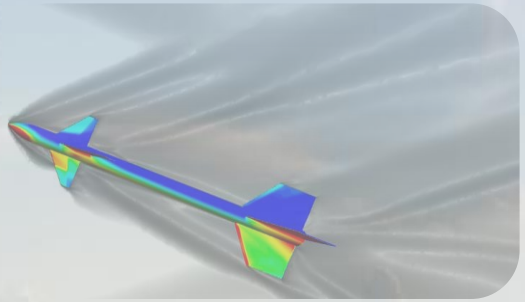
- Dedicated external aero workflow: parametric, scriptable
- End-to-end workflows

Preprocessing



- Mosaic technology
- Parallel meshing
- Clean/dirty CAD
- Integrated in workflow

Solver

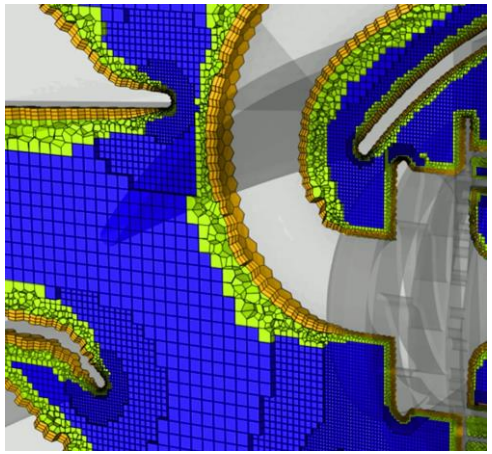


- Improved robustness: HSN, PMN
- Built-in NASA 9-coeff curve fits for material properties
- Built-in partial slip-wall BC
- Built-in surface ablation model

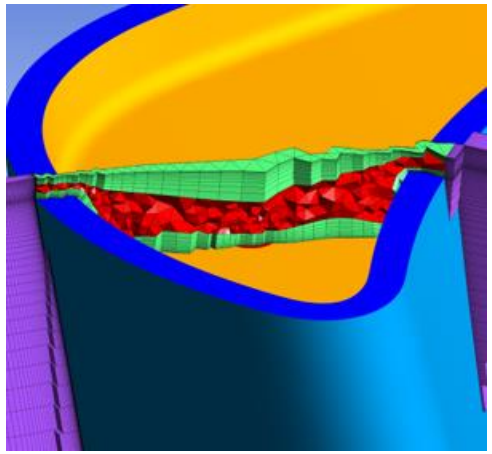
©2020 ANSYS, Inc. / Confidential

Fluids Meshing Tools

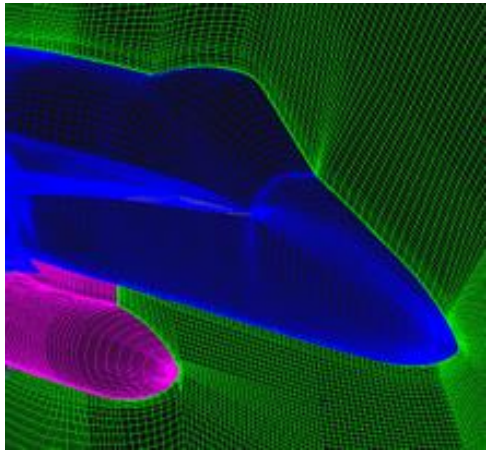
Fluids Meshing Tools: Overview



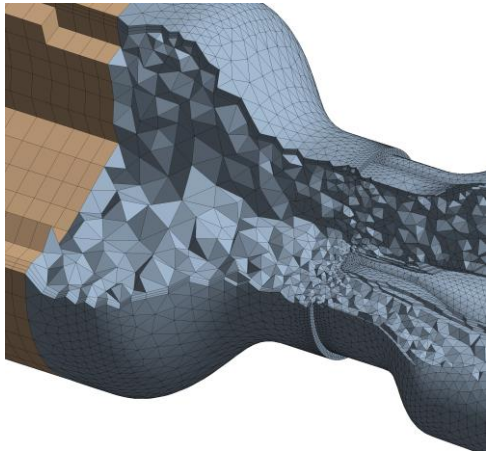
Unstructured Meshing
Fluent Meshing & Workflows



Turbomachinery
TurboGrid



Block-Structured Hex
ICEM CFD & SpaceClaim Meshing

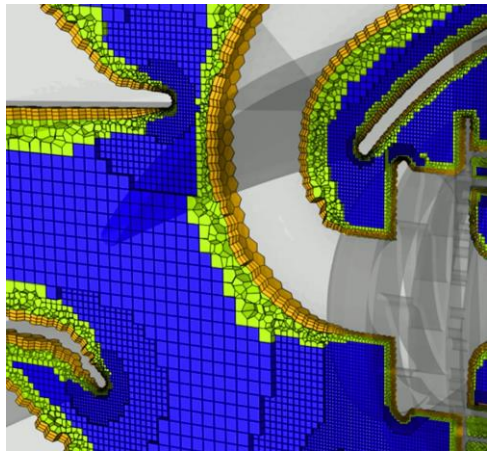


Multiphysics Users
Ansys Meshing

©2020 ANSYS, Inc. / Confidential

Fluent Meshing

Robust mesh generation for complex assembly



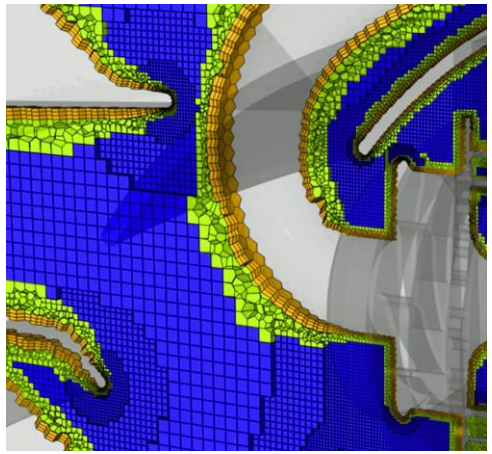
Unstructured Meshing
Fluent Meshing

- Handles large geometry
- Generates huge meshes
- Parallel meshing
- Advanced methods
- Poly-hexcore / Poly / Tet-hexcore / Tet
- CAD import to solution in one interface
- Linux and Windows support
- Automation
  - Workbench
  - Scripting

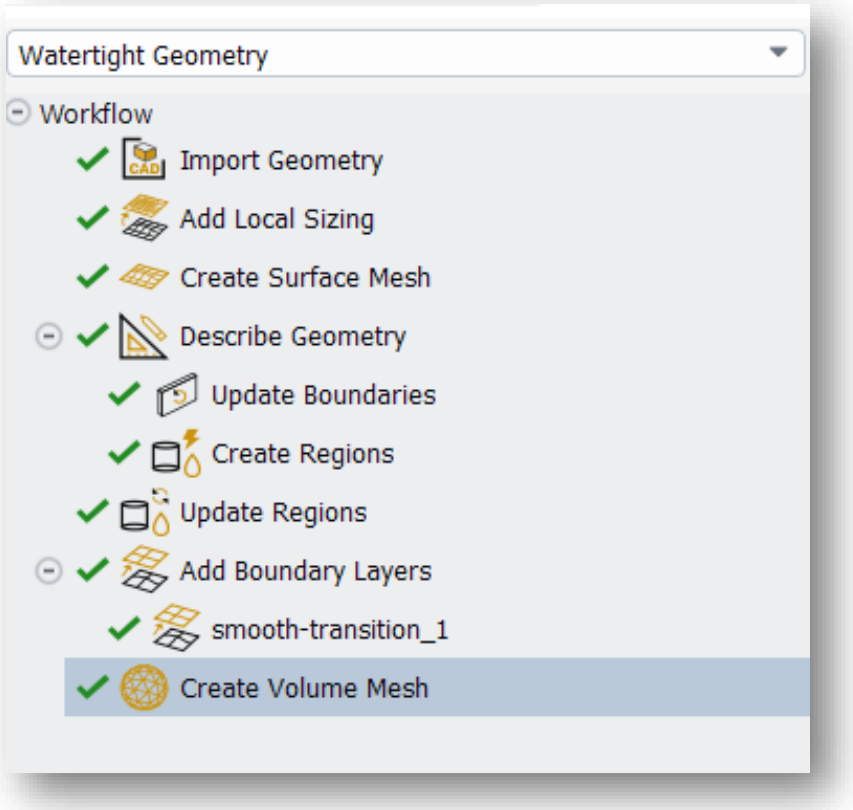
©2020 ANSYS, Inc. / Confidential

Fluent Meshing Workflows

Robust mesh generation for complex assembly



Unstructured Meshing
Fluent Meshing Workflows



Step by step guided workflows

- ✓ Fast learning
- ✓ Automated
- ✓ Persistent
- ✓ Efficient parallel meshing
- ✓ Customizable workflow
- ✓ Can be automated with python-based journal file

©2020 ANSYS, Inc. / Confidential

Fluent Meshing Workflows

Watertight Meshing Workflow (WTM)

- Prerequisite: clean watertight geometry
- Share topology at CAD/mesh level
- Conformal CHT mesh

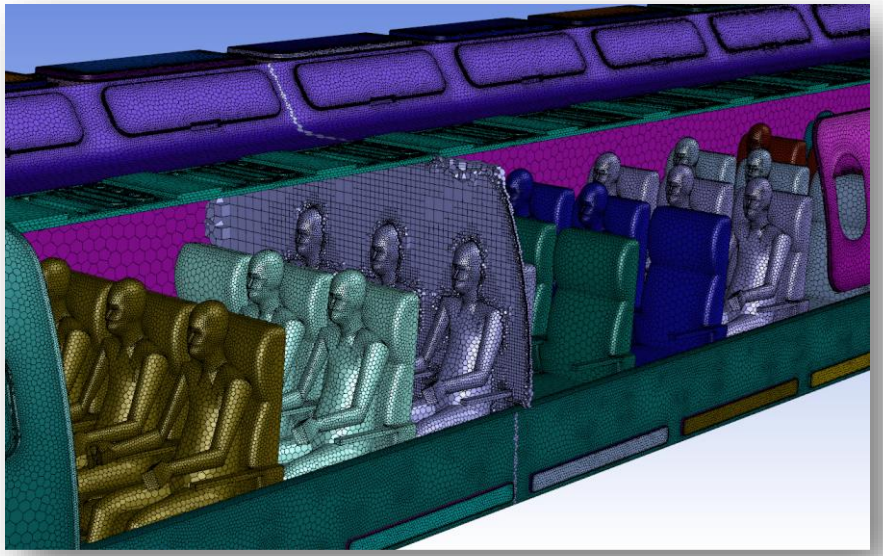
Fault-Tolerant Meshing Workflow (FTM)

- Tolerant to bad quality CAD/STL
- For dirty, non-watertight geometries
- Wrapping technology
- Automatic handling of gaps and overlaps
- Significantly reduces the pre-processing and CAD-cleanup effort required to prepare the models for simulation
- Non-conformal CHT mesh

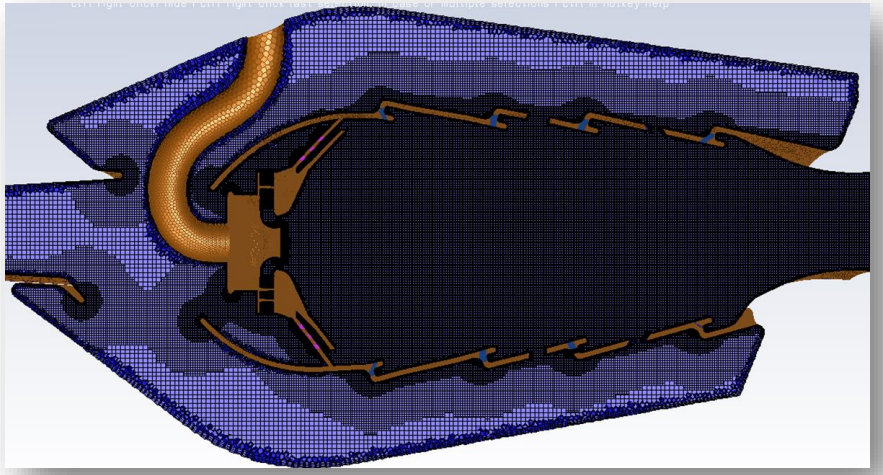
©2020 ANSYS, Inc. / Confidential

When to Use the Watertight Geometry (WTM) Workflow?

- What do we mean by watertight geometry?
  - Relatively clean
  - Watertight solid/fluid regions
- Single body: any imported CAD model
- Multi bodies: share topology at CAD level
- Supports surface mesh file as input
- Disconnected bodies are allowed if one body is entirely inside another, e.g., aircraft inside farfield domain
- Can be meshed by surface meshing and then volume filling, i.e., no wrapping required



Aircraft cabin



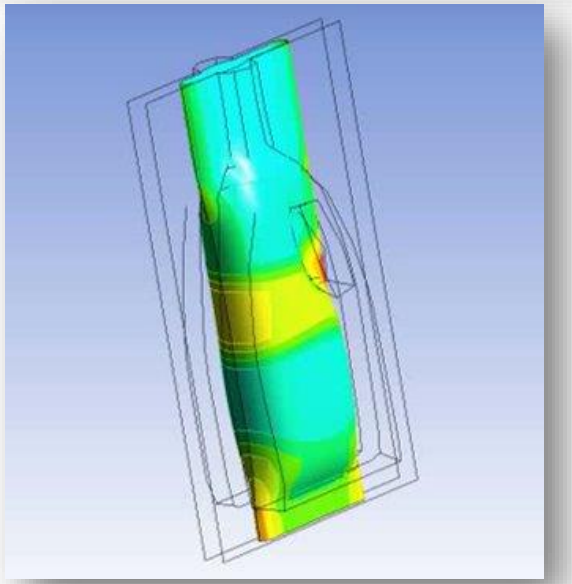
Generic combustor

©2020 ANSYS, Inc. / Confidential

What is Wrapping?

- The wrapping approach can be likened to
  - Shrink-wrapping luggage to wrap from outside of the geometry, or
  - Blow molding to wrap from inside of the geometry
- Wrapping has several strengths:
  - Automated handling of gaps and overlaps
    - Removes need for manual geometry-based closure of the geometry
    - No need to have geometry in form of a watertight region
  - Feature suppression
    - Mesh sizes > local features can be used for local defeaturing
- Inside Fluent Meshing FTM workflow
  - Part management for assembly preparation
  - Wrap-extract of external and internal flow domain
  - Creation of porous blocks, e.g., heat exchangers
  - Automatic offset and wake refinements
  - CHT applications via non-conformal auto-pairing workflow

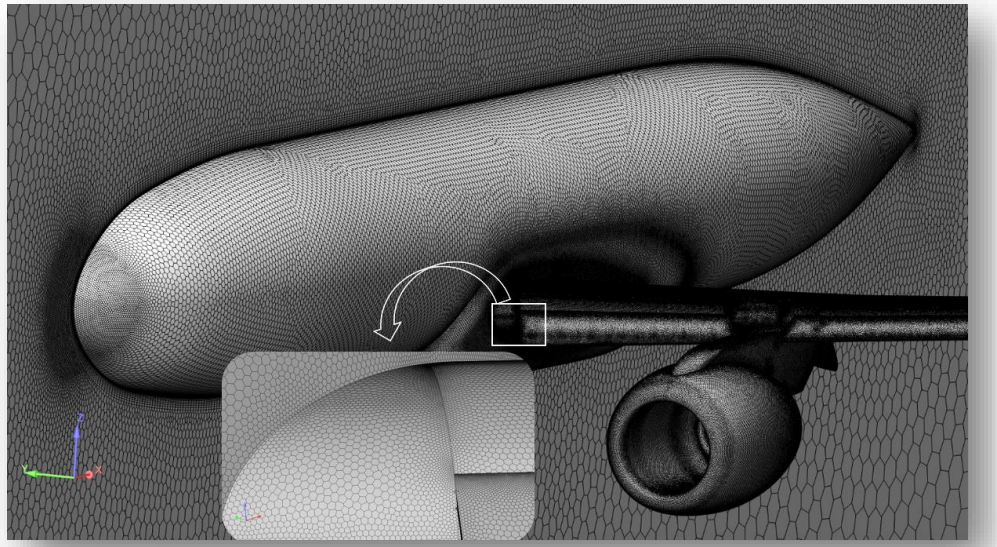


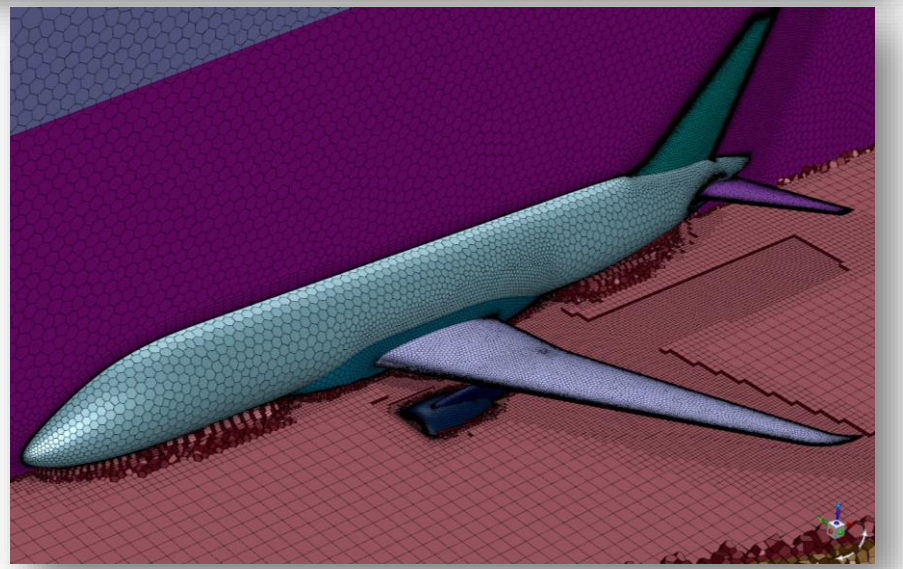


©2020 ANSYS, Inc. / Confidential

When to Use the Fault-Tolerant Meshing (FTM) Workflow?

- Complex faulty CAD where it would be impractical to clean into a good quality watertight geometry
- Faceted geometry
  - Scanned geometry
  - STL geometry
- Watertight CAD with complex surface patches where surface meshing results in poor quality mesh
- Cases where mesh-based defeaturing is required
- Applicable for all industries





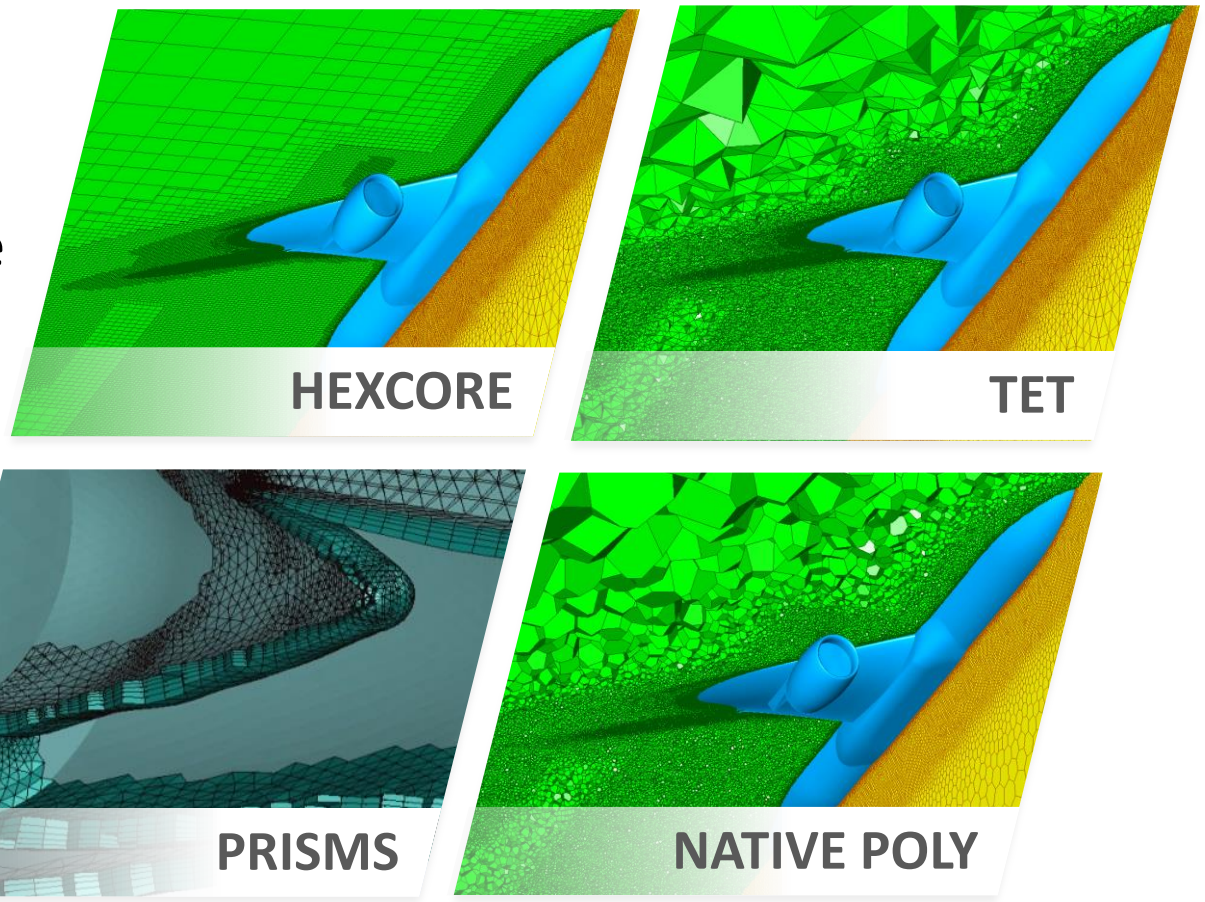
External aerodynamics application examples

©2020 ANSYS, Inc. / Confidential

Mosaic Technology

Accuracy and Solution Time Are Highly Dependent on the Mesh

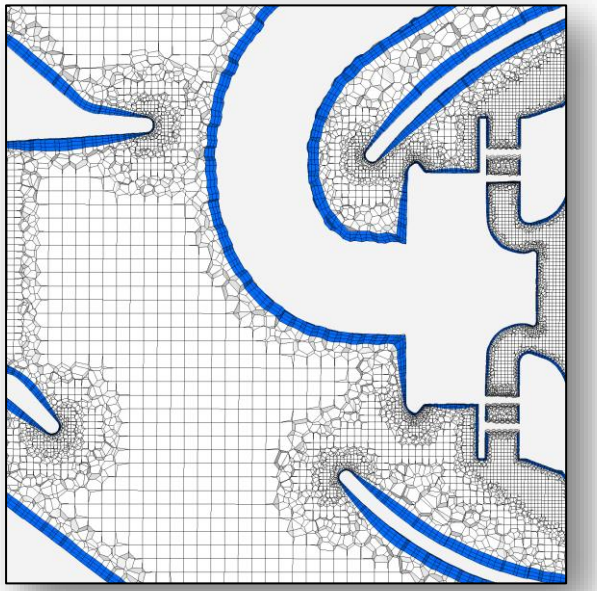
- Various geometries and flow regimes require different meshes
- Hybrid meshes containing hex in the core are good alternatives
- But there's no good solution for mesh transitions
  - Reduced mesh quality
  - Excessive cell counts
  - Forced to compromise on a common element type to minimize mesh transitions



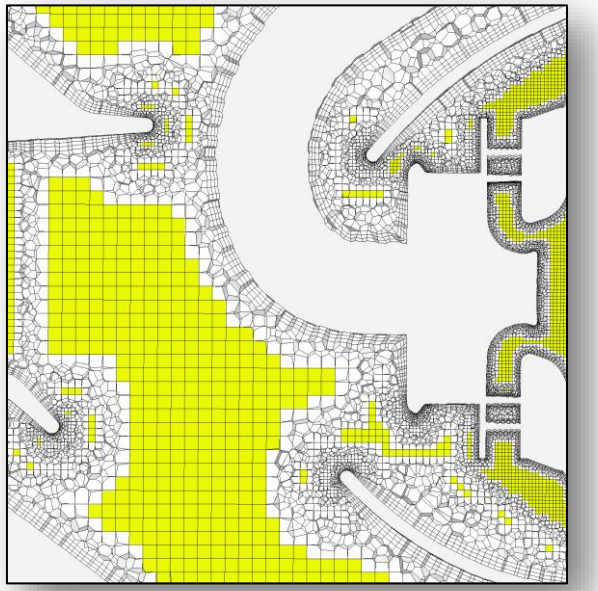
©2020 ANSYS, Inc. / Confidential

Mosaic Meshing Technology

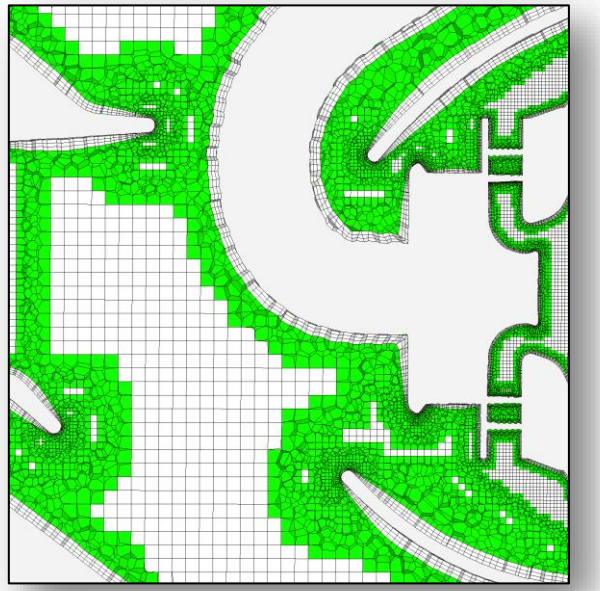
- Mosaic technology enables polyhedral connections between disparate mesh types
  - Conformally connects polygonal prisms on the boundary to bulk Cartesian aligned hexahedral cells
- Hex dominant allows for faster meshing and a better-quality volume mesh
  - Solve time reduction of 30-50%



Isotropic Poly Prisms



Hexahedral Cells

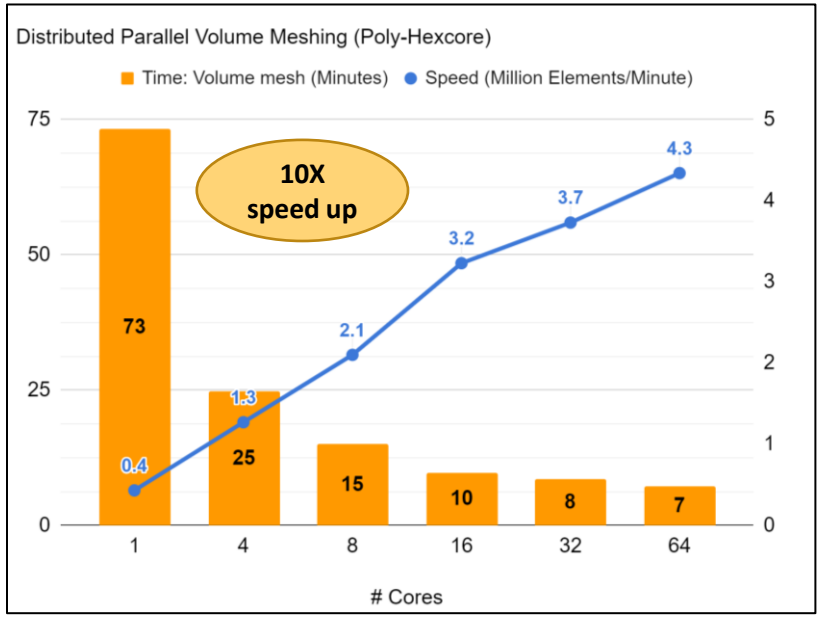


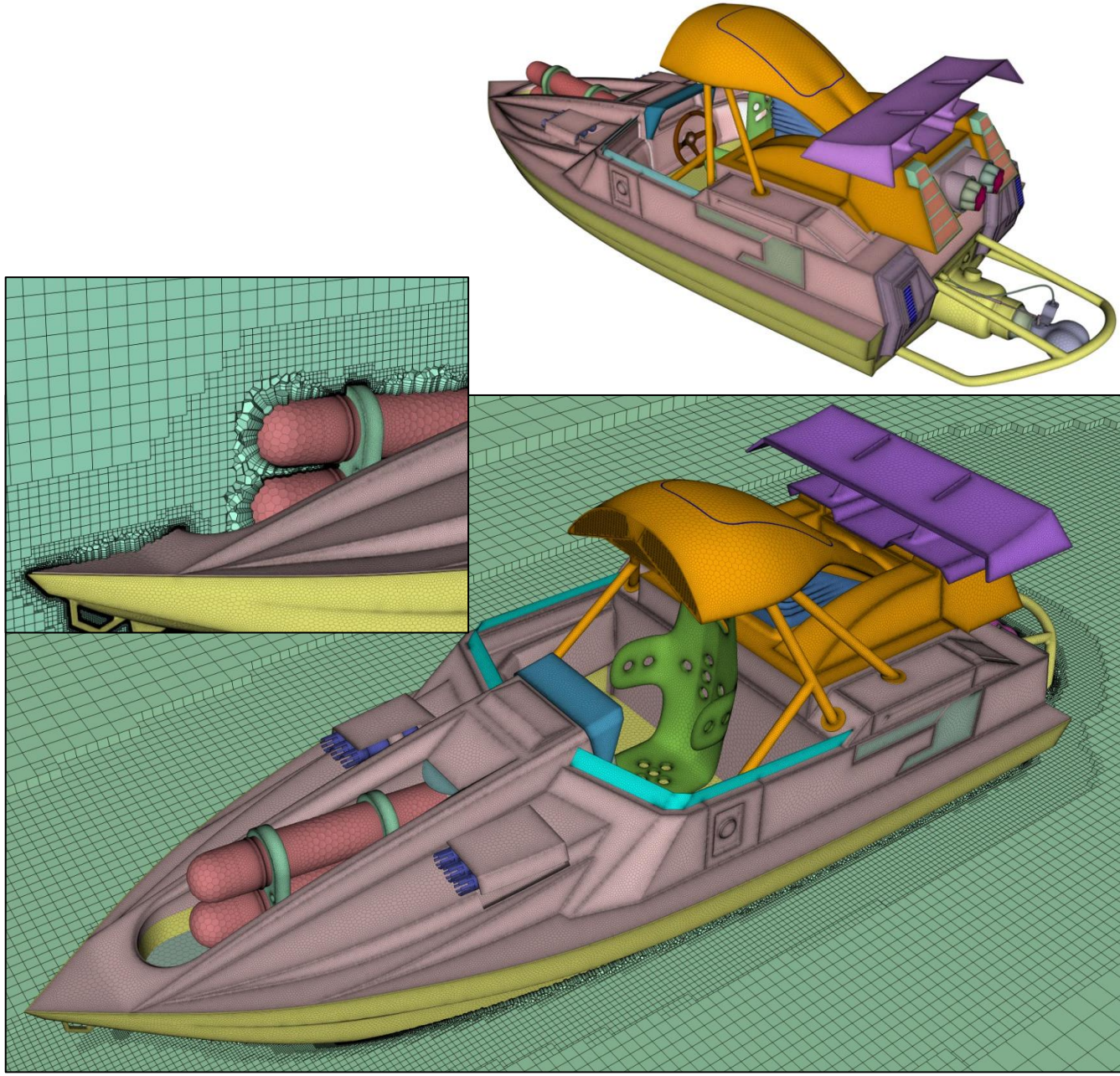
Mosaic Polyhedral Cells

©2020 ANSYS, Inc. / Confidential

Parallel Poly-Hexcore Metrics

- Parallel poly-hexcore quality, robustness, scalability improvements
  - 31.5M elements Poly-Hexcore mesh includes
  - 5.2M Cartesian aligned Hexahedral elements and
  - 15.4M polyhedral prismatic elements



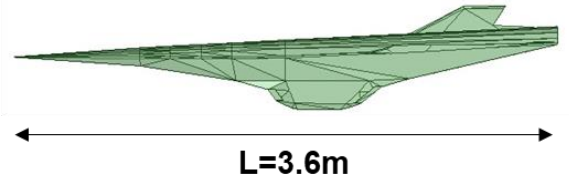


©2020 ANSYS, Inc. / Confidential

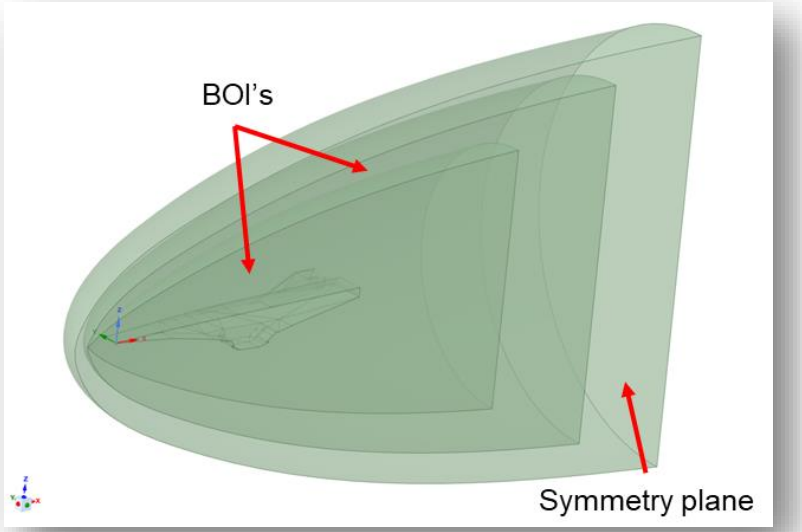
Fluent Meshing Workflows: Case Studies

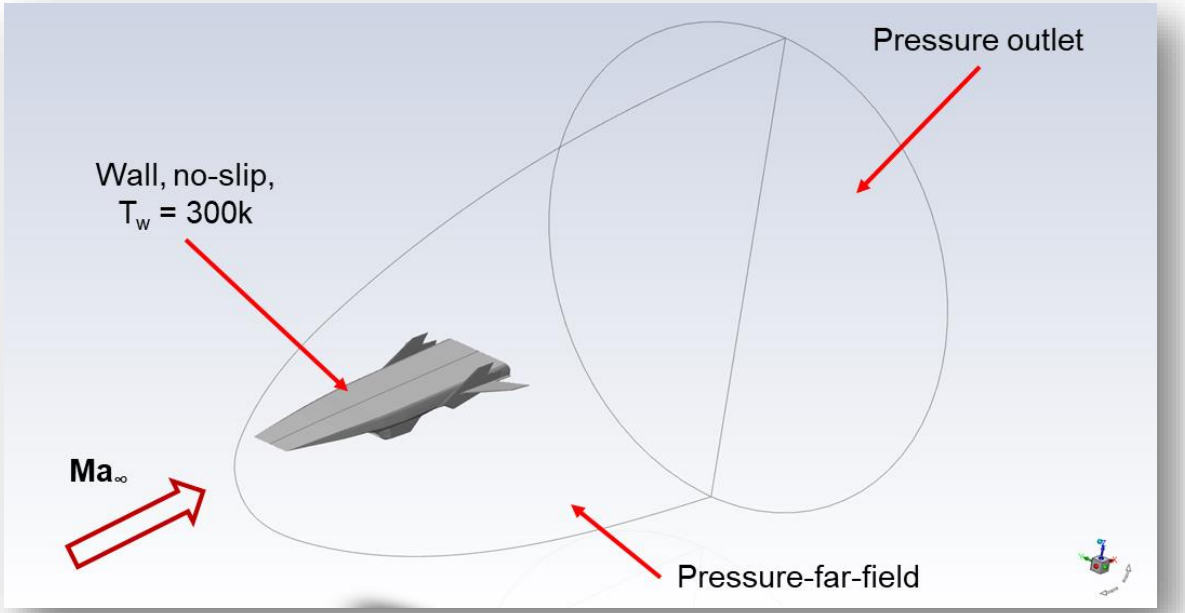
Ansys

Watertight Meshing of NASA X-43



- Hypersonic flow past a X-43 (mockup) geometry at Mach 9
- No yaw angles are considered -> used symmetry assumption





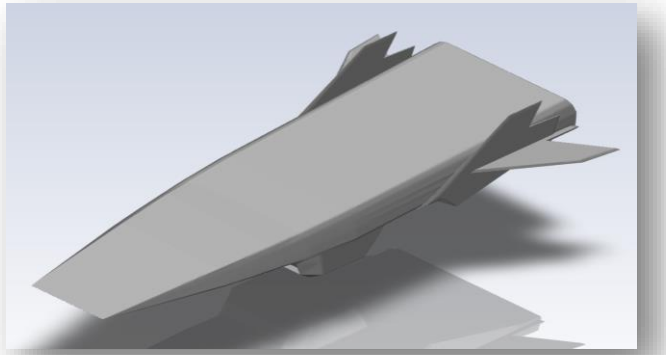
SpaceClaim geometry

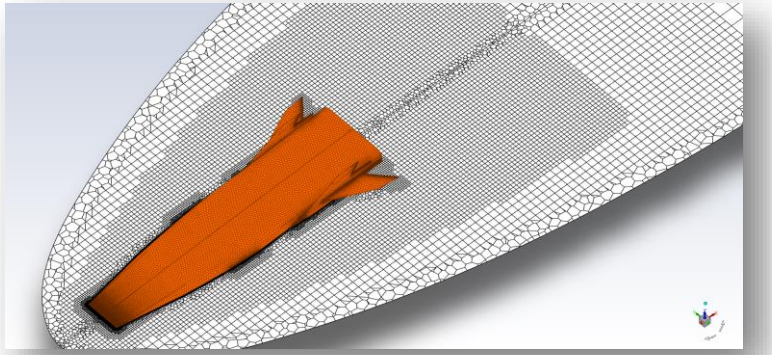
Geometry cleanup performed in Ansys SCDM

Domain with boundary conditions

©2020 ANSYS, Inc. / Confidential

Watertight Meshing of NASA X-43







Mesh stats:

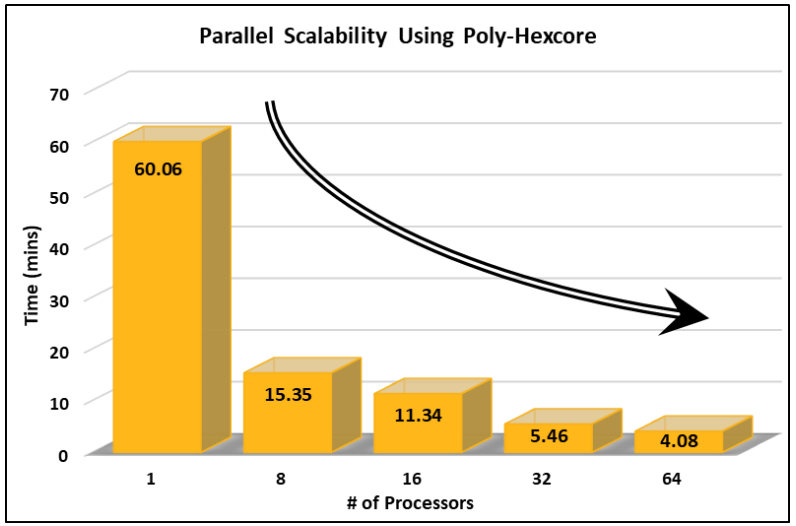
Total mesh count = 24.6M

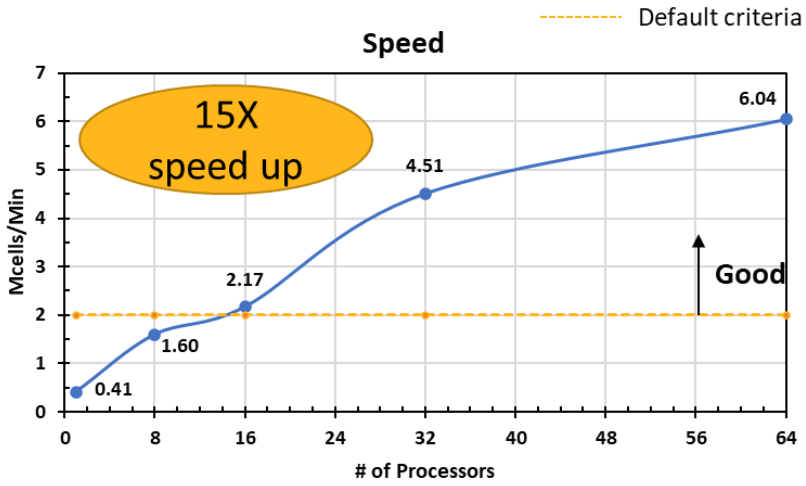
- **3.4M hexahedral cells**
- **21.2M polyhedral cells**

Total mesh time ≈ 4.08 mins with parallel poly-hexcore elements

Max cells/min seen = 6.04M (64 processors)

Max skewness = 0.88

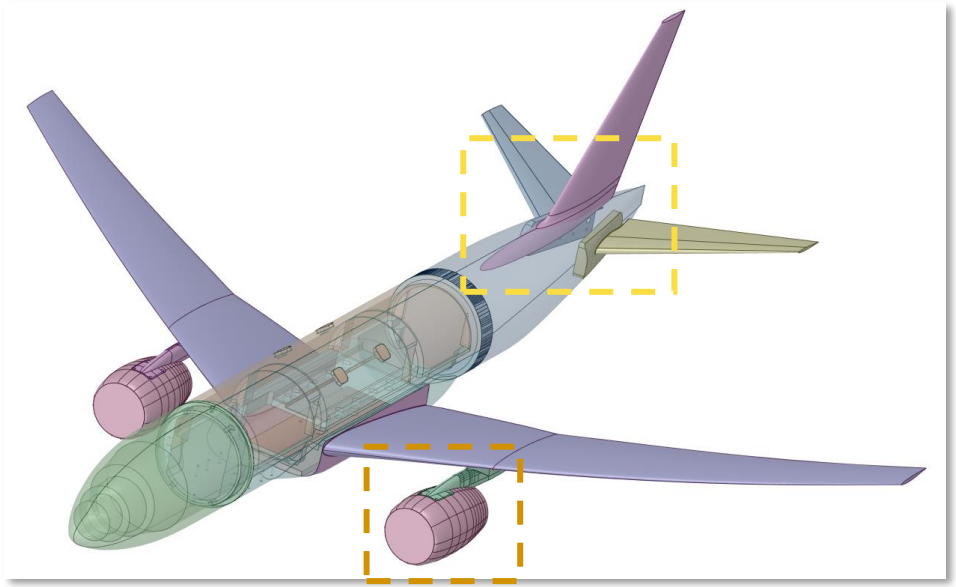




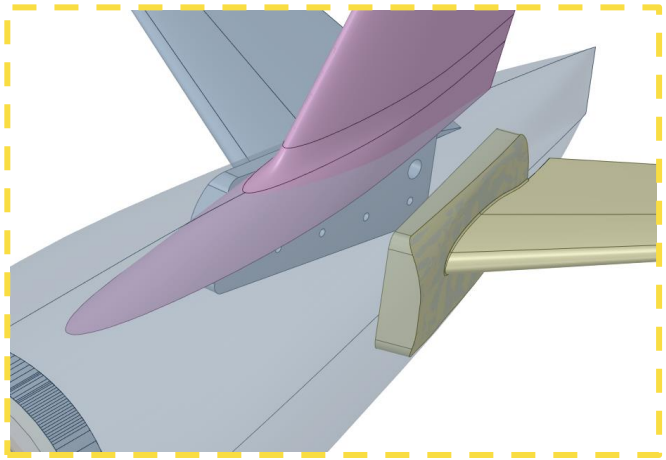
©2020 ANSYS, Inc. / Confidential

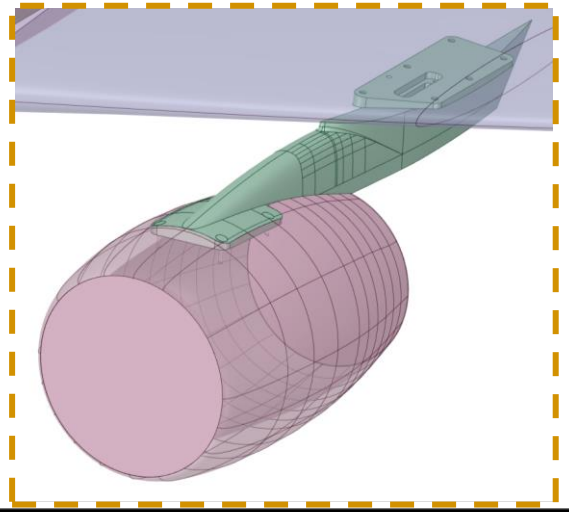
Fault-Tolerant Meshing of NASA High-Speed CRM

- Transonic flow past a NASA common research model (CRM) representative of a wide-body commercial transport aircraft



Original geometry

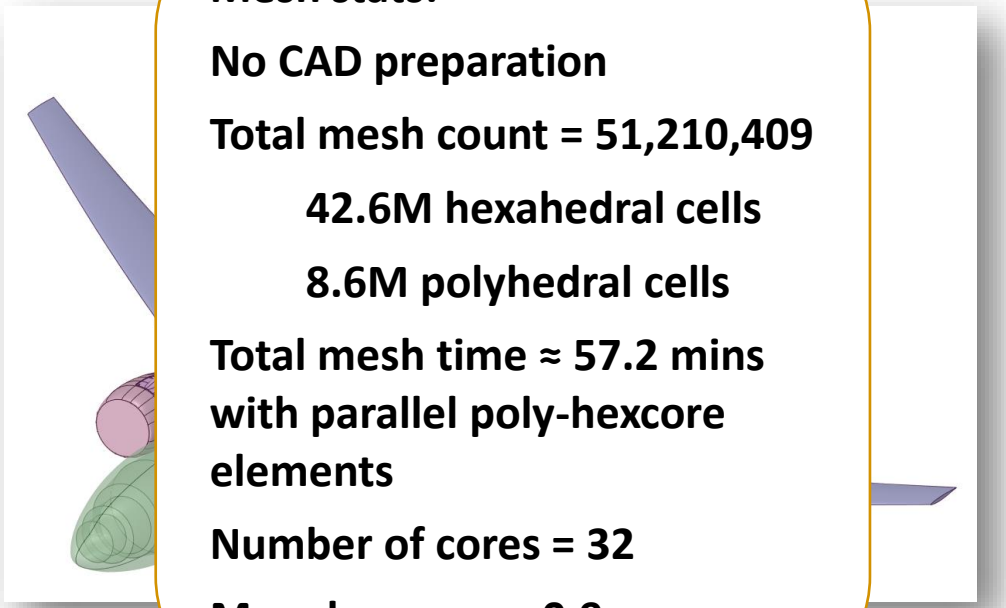




Unstitched surfaces
Intersecting bodies
Overlapping bodies
Internal bodies
Gaps/Holes

©2020 ANSYS, Inc. / Confidential

Fault-Tolerant Meshing of NASA High-Speed CRM



Mesh stats:

No CAD preparation

Total mesh count = 51,210,409

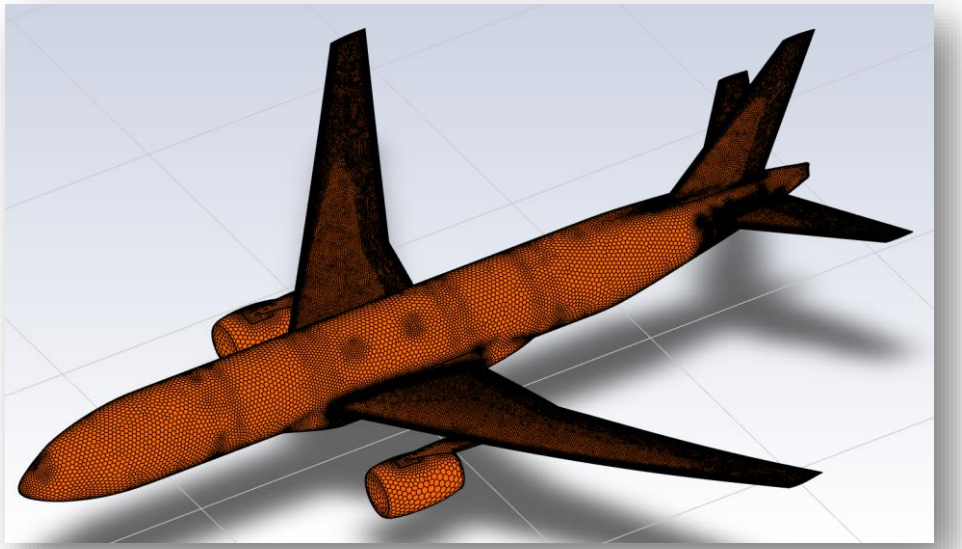
- **42.6M hexahedral cells**
- **8.6M polyhedral cells**

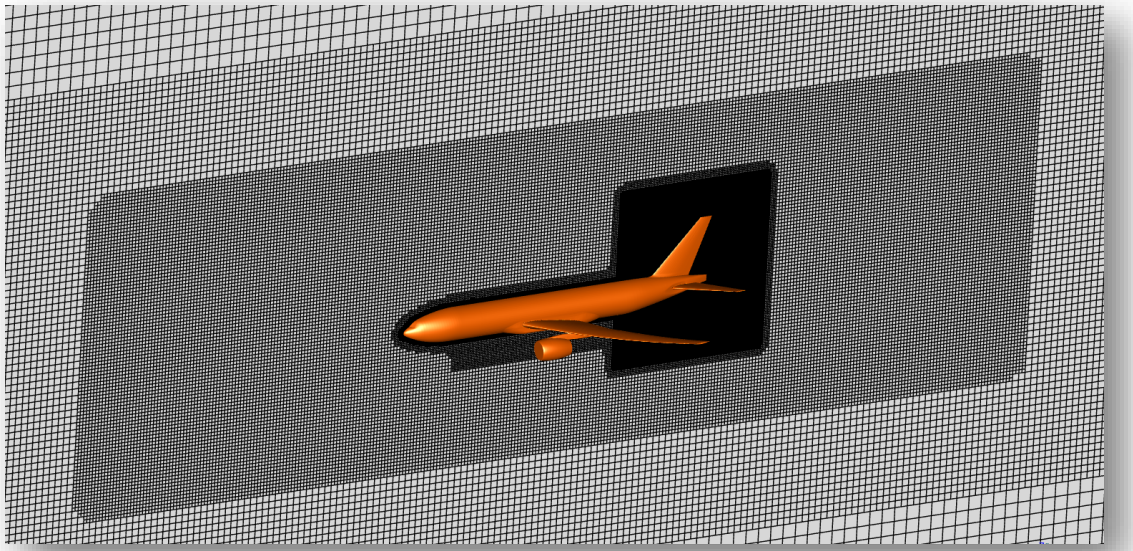
Total mesh time ≈ 57.2 mins with parallel poly-hexcore elements

Number of cores = 32

Max skewness = 0.9

Original geometry



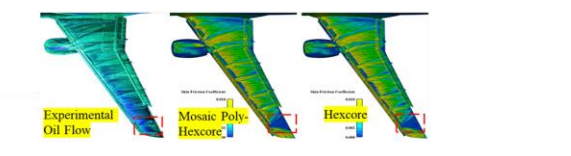


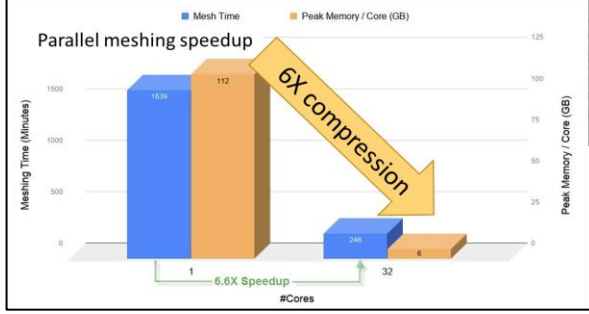
Mesh

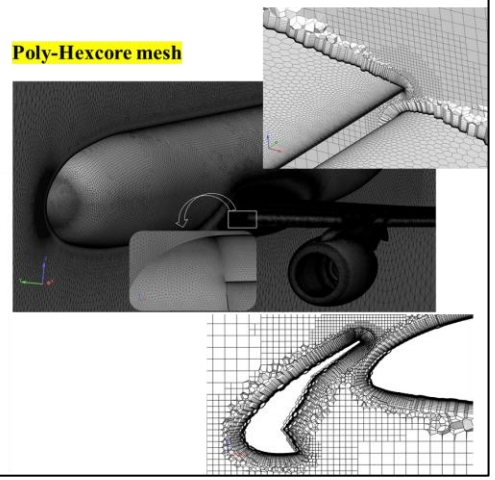
©2020 ANSYS, Inc. / Confidential

Additional Case Studies: Pre-processing

Case study: Mosaic Meshing for High-Lift Aircraft Configuration



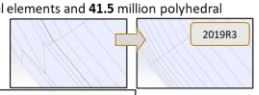


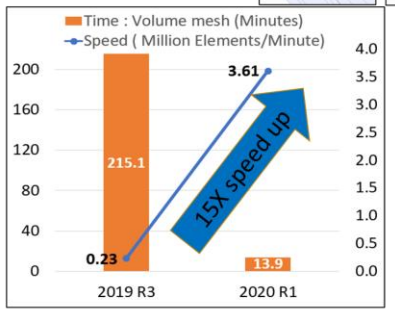


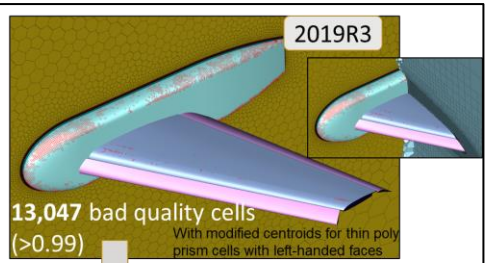
Case study: Parallel Poly-Hexcore

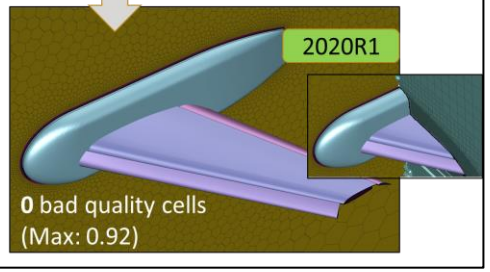
High Aspect Ratio Poly Prisms quality improvements

HiLift Workshop case: (Parallel Poly-Hexcore mesh on **32 cores**)
50.2 million element Poly-Hexcore mesh includes
4.5 million Cartesian aligned Hexahedral elements and **41.5** million polyhedral prismatic elements
(Aspect Ratio: ~10,000 AR)



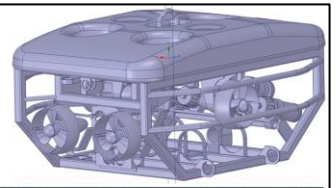


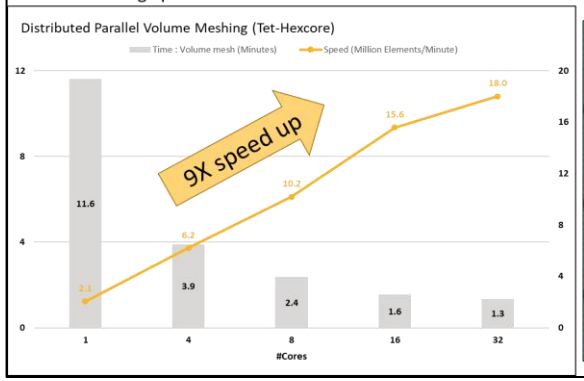


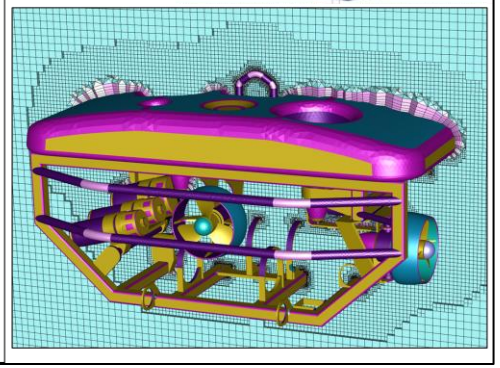


Case study: UROV, Parallel Tet-Hexcore

UROV case:
24 million element Tet-Hexcore mesh includes
4.7 million Cartesian aligned Hexahedral elements and
6 million wedge prismatic elements.



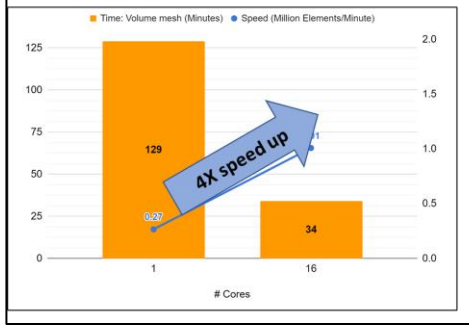


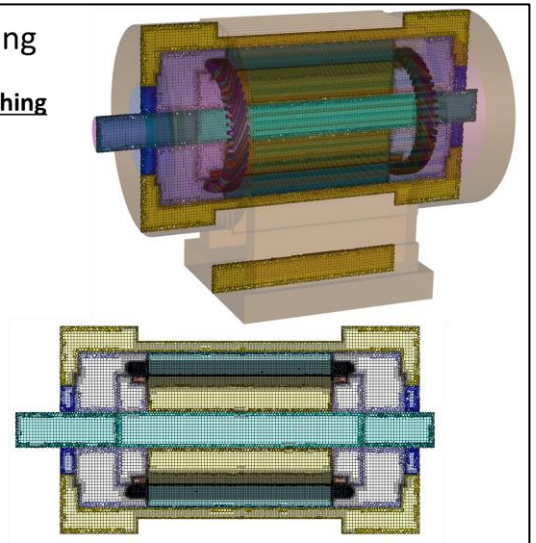


Case study: Electric Motor Meshing

Multi-object <u>distributed parallel volume meshing</u>

Useful for:
- Sliding interface cases
- Non-Conformal mesh
- Overset mesh

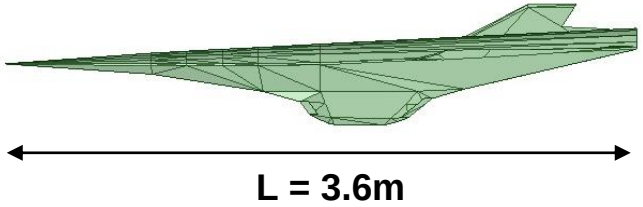




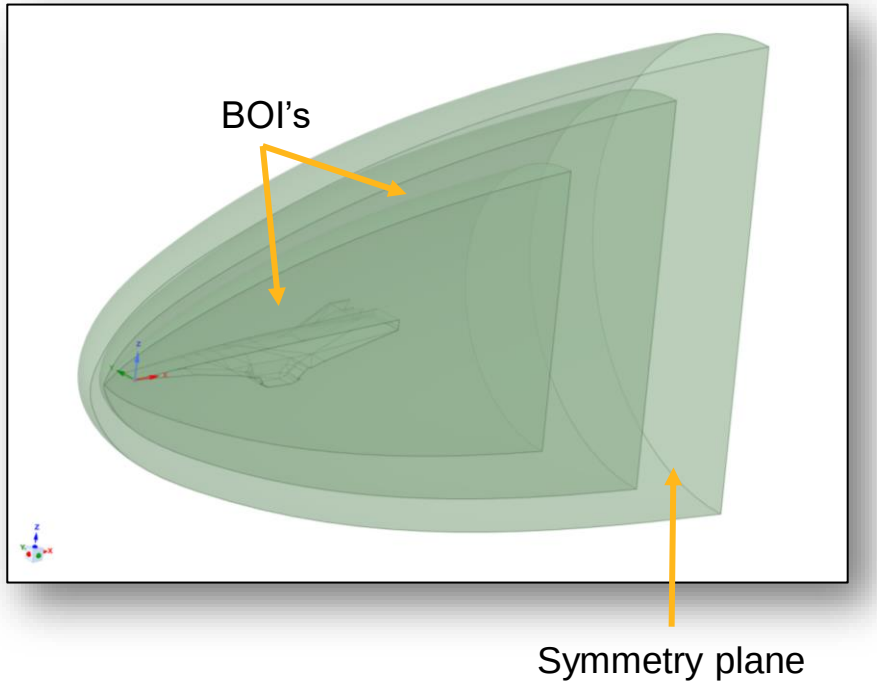
©2020 ANSYS, Inc. / Confidential

Live Demonstration: WTM Workflow

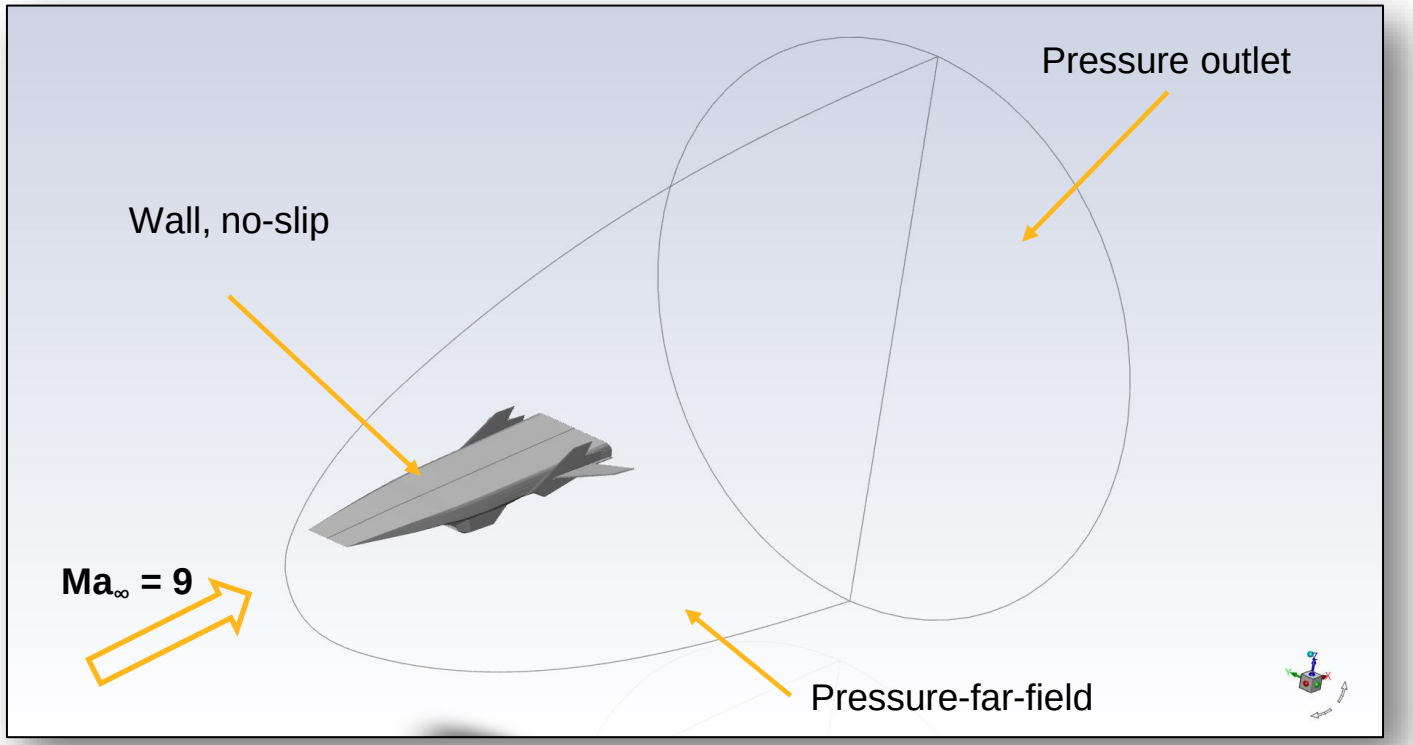
Problem Definition



- Hypersonic flow past a NASA X-43 (mockup) geometry at Mach 9



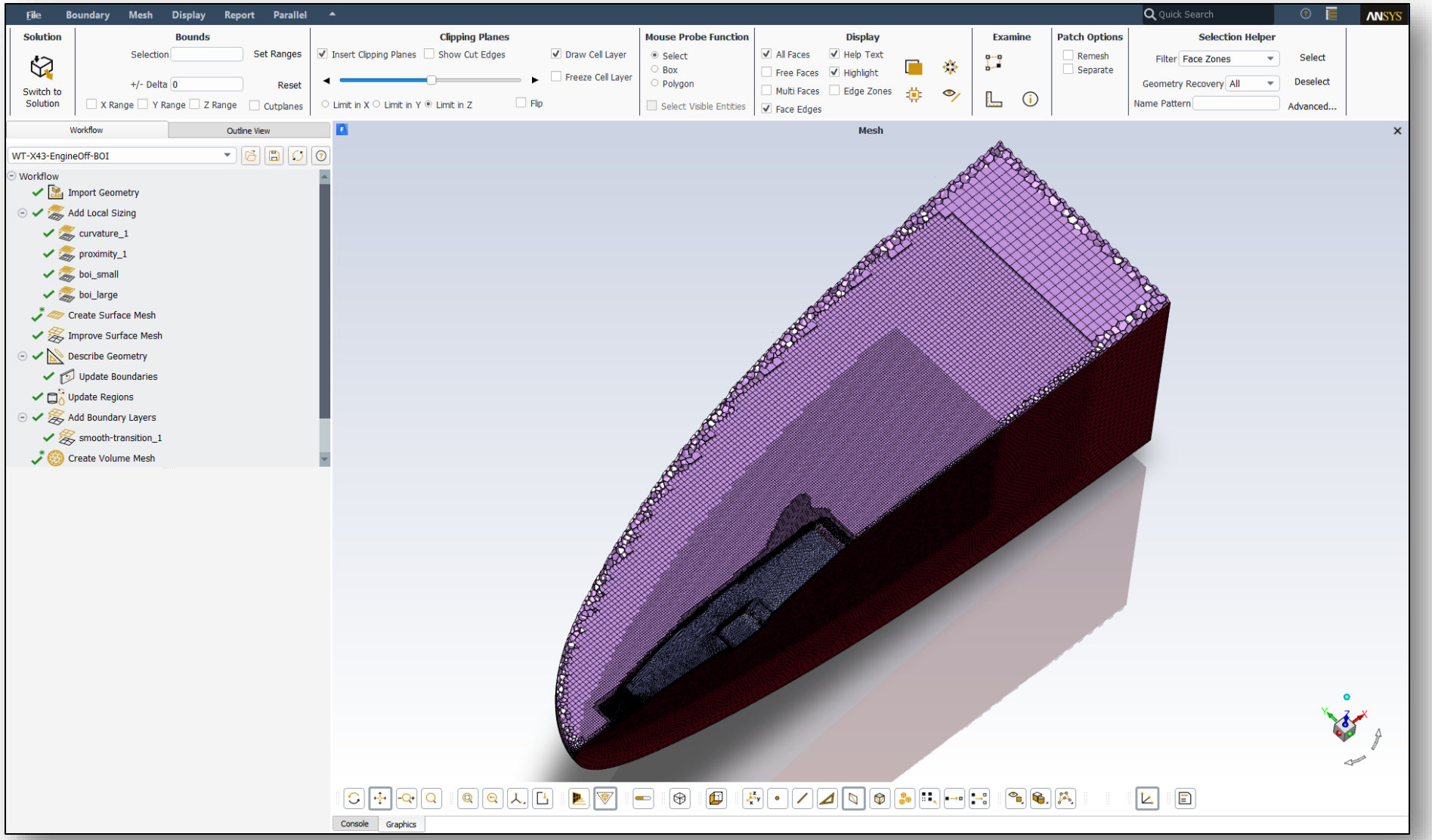
SpaceClaim Geometry



Domain with Boundary Conditions

©2020 ANSYS, Inc. / Confidential

Watertight Meshing of NASA X-43: Live Demo



©2020 ANSYS, Inc. / Confidential

Watertight Mesh to Simulation Workflow for NASA X-43

Geometry
- Clean geometry in SCDM
- Setup named selections for boundaries

Meshing
- Transfer clean CAD into Watertight workflow
- Setup mesh and boundary layers controls
- Generate surface and volume mesh
- Save workflow for future mesh edits / mesh automation

Setup/Solve
- Load mesh into Fluent and solve using Density-based solver
- Solution transcript written by default
- Save the journal file for future simulations (automation)

Post-Process
- Post-process in Fluent
- Can be exported into Ansys Ensight / CFD-Post

©2020 ANSYS, Inc. / Confidential

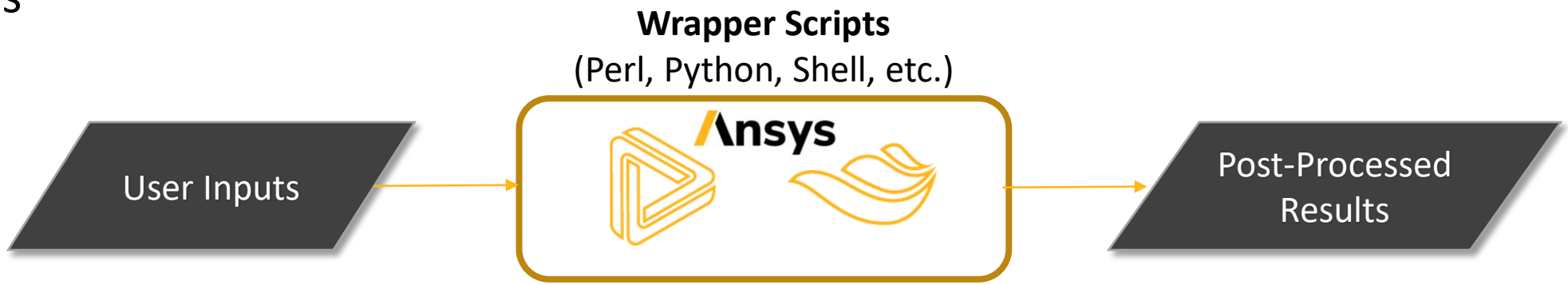
End-to-End Automated Workflow for Pre-Processing

Automation Framework

- Ansys simulation software can be readily automated and incorporated as part of a pre-existing solution with python scripting

- Use wrapper scripts and journal files to automate workflow execution
  - CAD manipulation, meshing, simulation execution and more…
  - Highly customizable

- Allows for batch processing and can further extend integration with 3rd party/in-house tools



©2020 ANSYS, Inc. / Confidential

Case Study: Automated Workflow Flowchart

- A fully automated workflow was developed. Using Python scripting to drive the CAD-preparation, meshing, setup, execution and post-processing of the simulation

Named Selection creation in SpaceClaim

Ansys SPACECLAIM

FMD File

Journal File

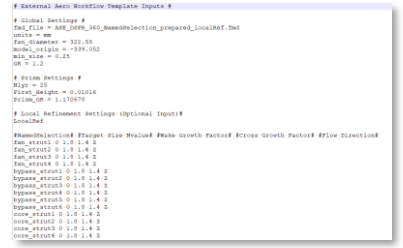
Mesh is generated automatically using the Fault tolerant workflow

Ansys FLUENT

Solver execution
Post-processing

Python script produces meshing journal file

Python Input File

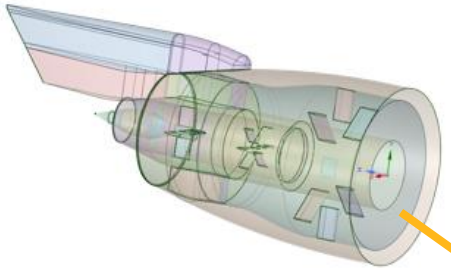


Text-based input file

©2020 ANSYS, Inc. / Confidential

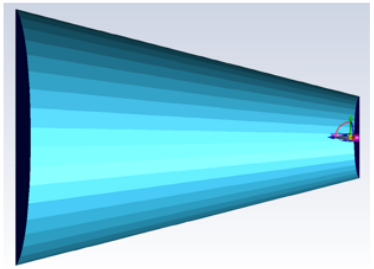
Case Study: Automated Meshing Operation

Model Preparation



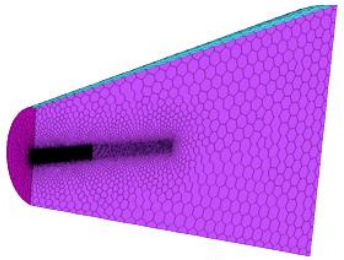
Named Selection Creation
(No Surface Cleanup Needed)

Enclosure Creation



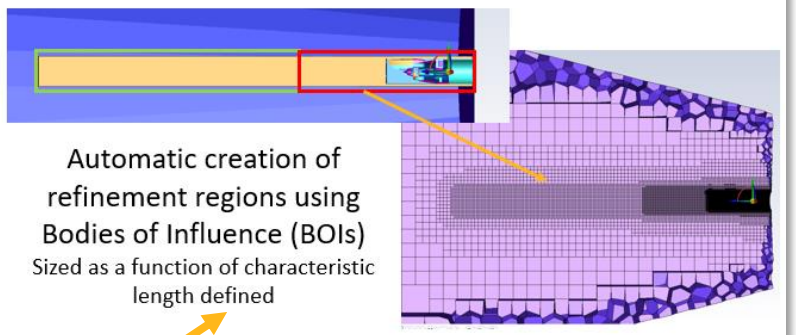
Automatic creation Enclosure Domain
(parametrically sized as a function of the model, best practices engrained)

180º Sector Calculation



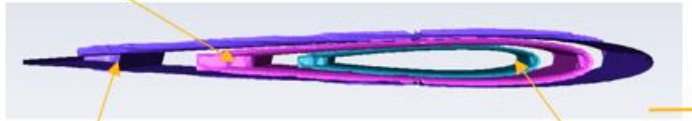
Automatic creation of Symmetry Plane
(optional)

Automatic BOIs Creation

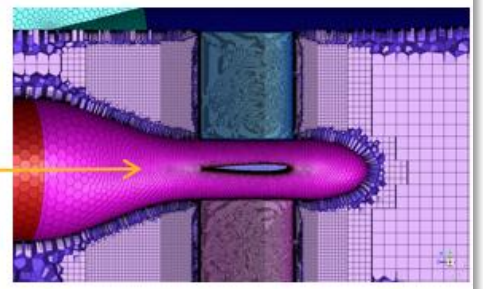


Automatic creation of refinement regions using Bodies of Influence (BOIs)
Sized as a function of characteristic length defined

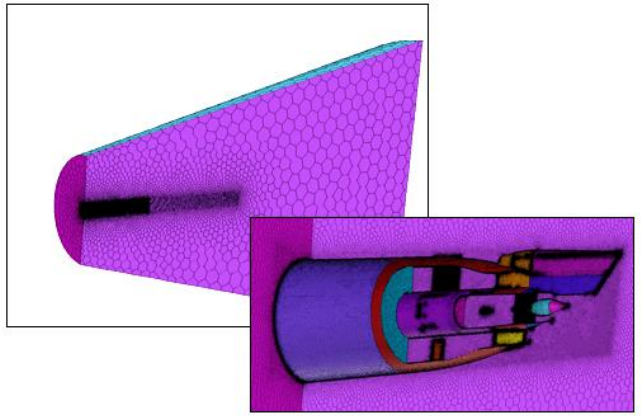
Local Refinement Regions



Cross and wake growth factors control offset dimensions



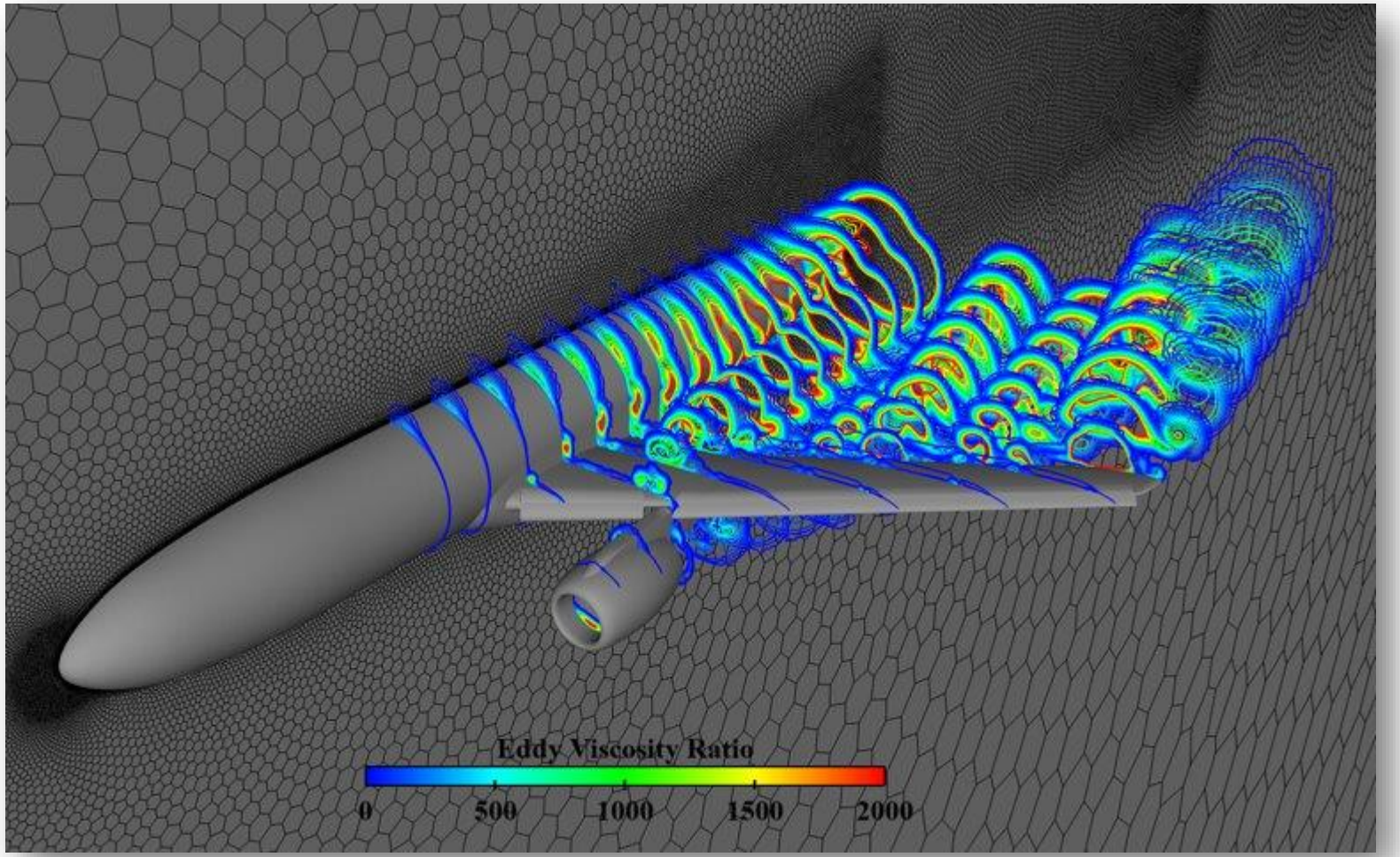
Finalized Mesh



©2020 ANSYS, Inc. / Confidential

Top Reasons to Use Ansys Fluent Meshing for A&D Applications

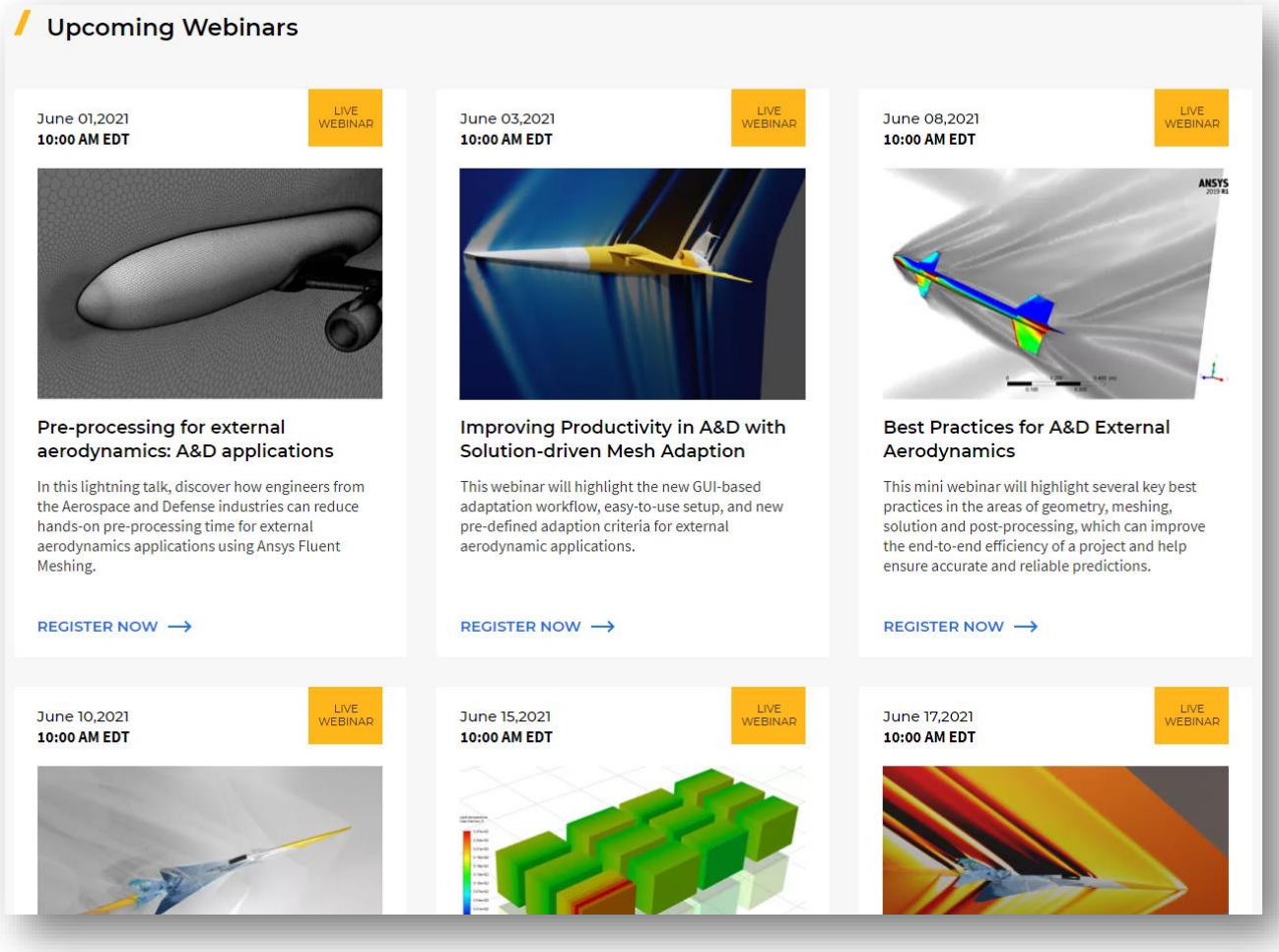
- *User friendly*
- *Parallel scalability*
- *Automated meshing*
- *Robust mesh offerings*
- *Customizable workflows*
- *Ecosystem*



©2020 ANSYS, Inc. / Confidential

Learning Resources

- **Ansys Learning Hub (ALH)**
- **Fluent Webinar Series**
- **Ansys Customer Excellence (ACE) Support**



©2020 ANSYS, Inc. / Confidential

Thank You!




Ansys