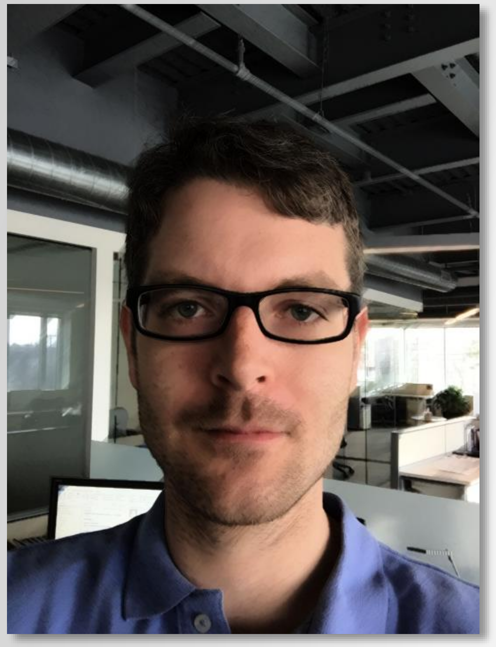
Aerodynamic CFD simulations just got a whole lot better!

Royce Abel
Technical Support Manager
@royceabel

Your Instructor's Schedule



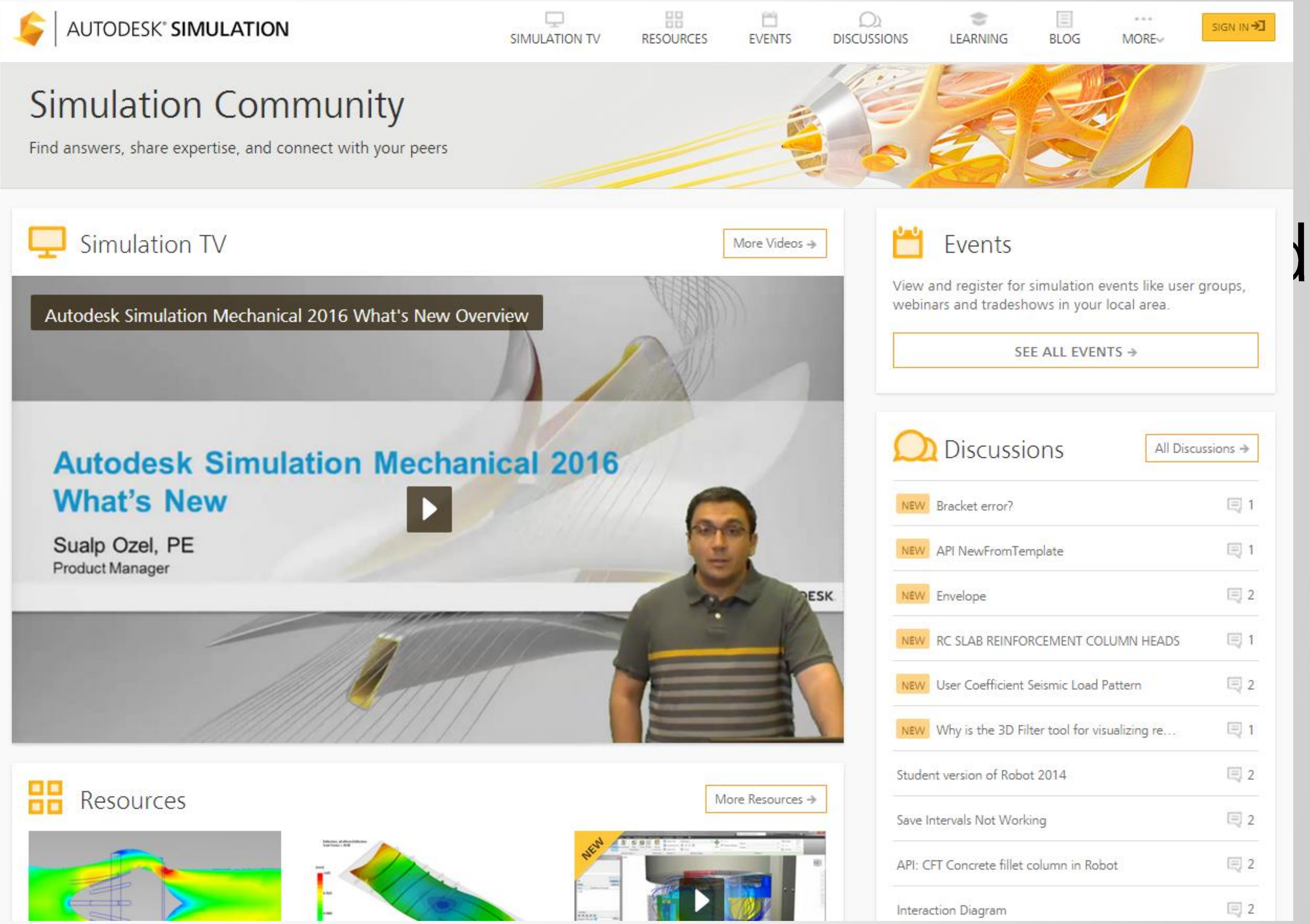
Royce Abel

Time	Event
Wednesday 10:00 am	Aerodynamic CFD simulation just got a whole lot better!
Wednesday 2:30 pm	Answer Bar – Bring your unanswered questions!

Resources – SimHub

simhub.autodesk.com

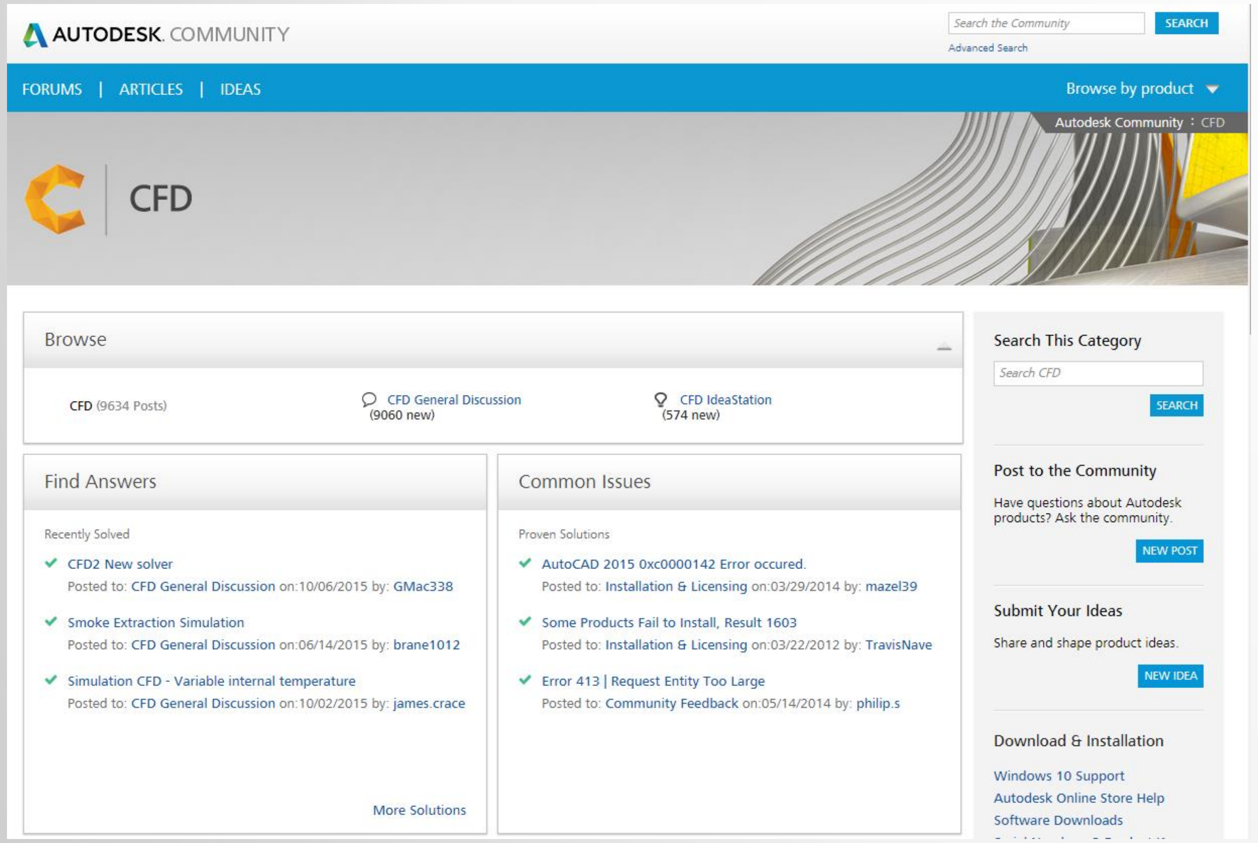
Simulation TV	Functionality Features and What's New videos
Resources	White papers and validation documents
Discussions / Idea Station	Ask questions, share your knowledge and ideas
Blog	Read feature stories, learn tips and tricks or get the latest news
Learning	Archive of AU-online presentations



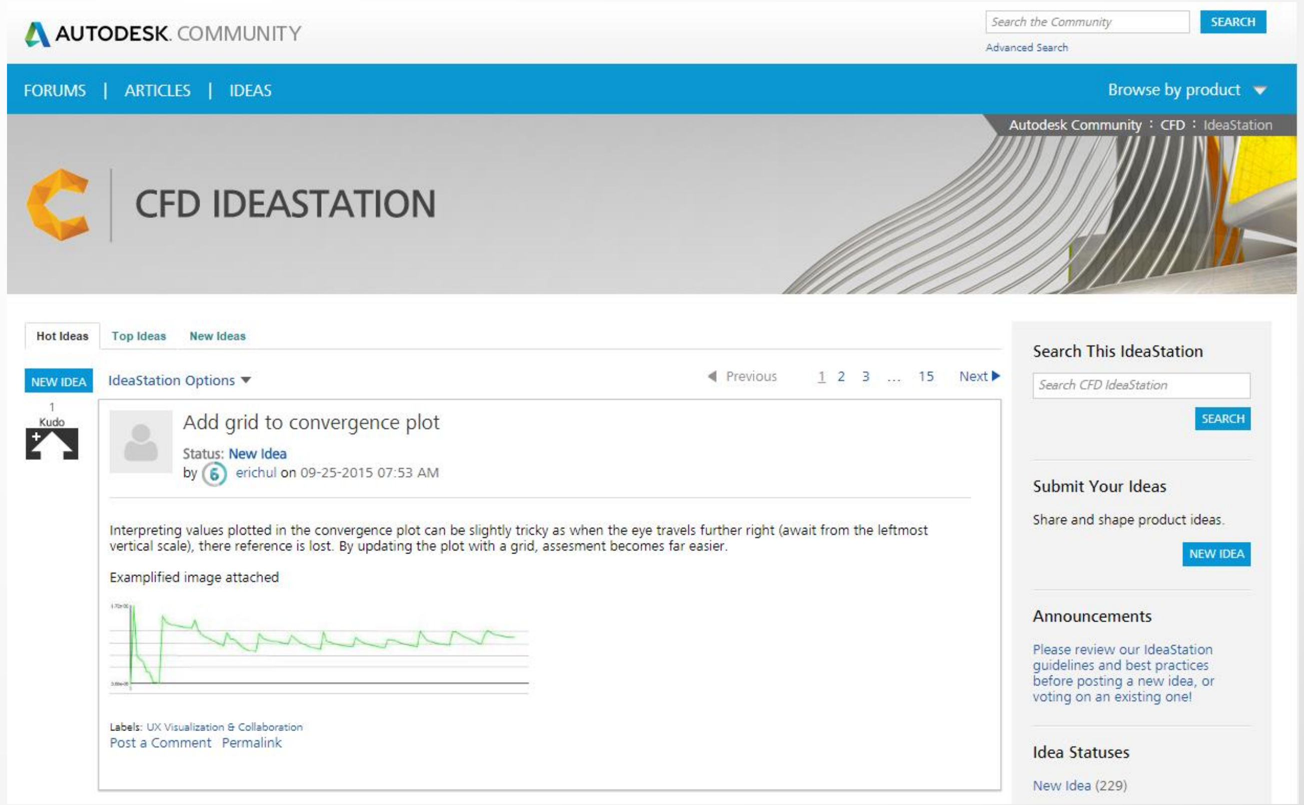
Join The Discussion!

- Autodesk customers and industry partners ask questions and share information about Autodesk products.
- Regularly monitored by Autodesk employees

CFD Forum

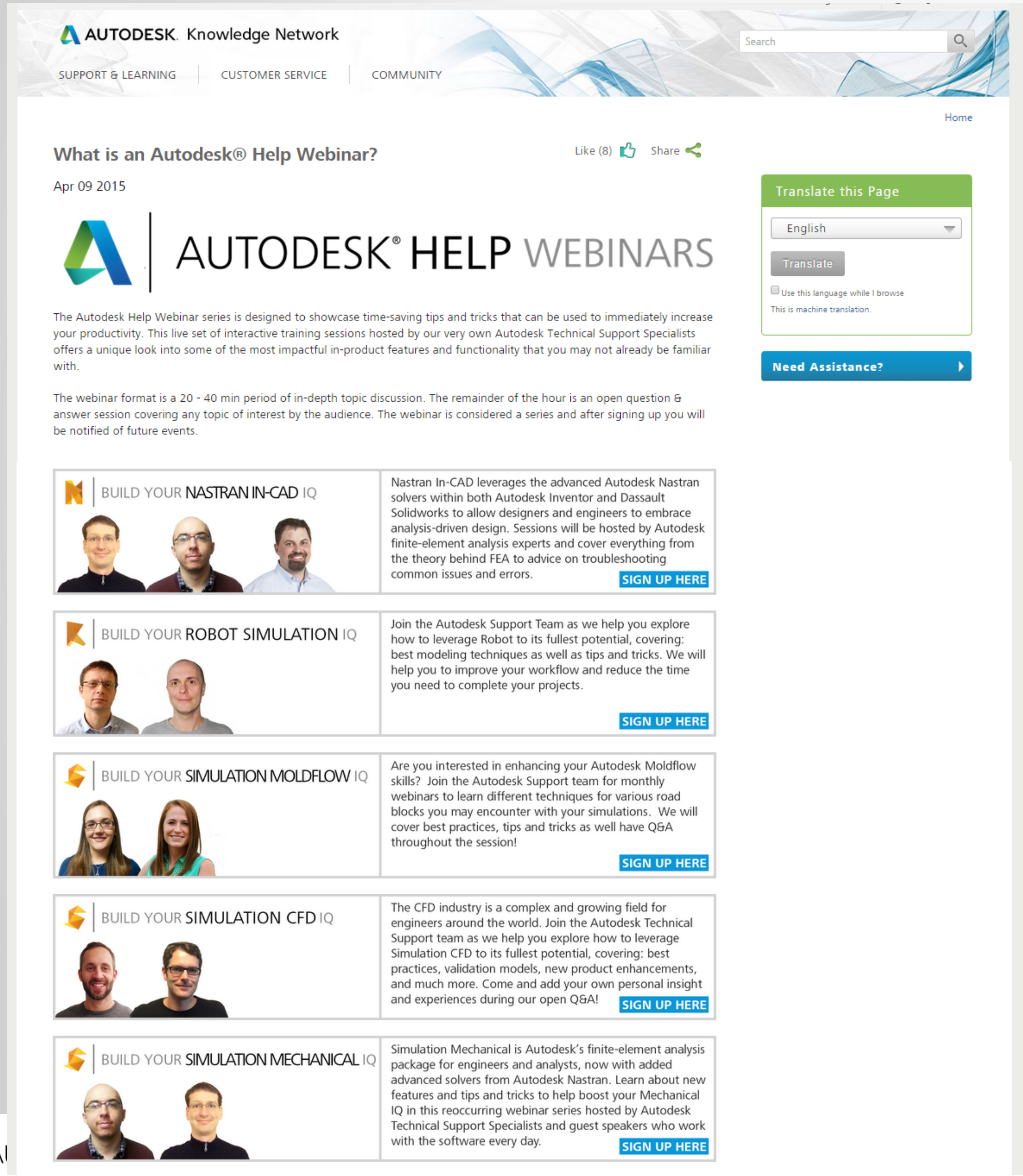


CFD Idea Station

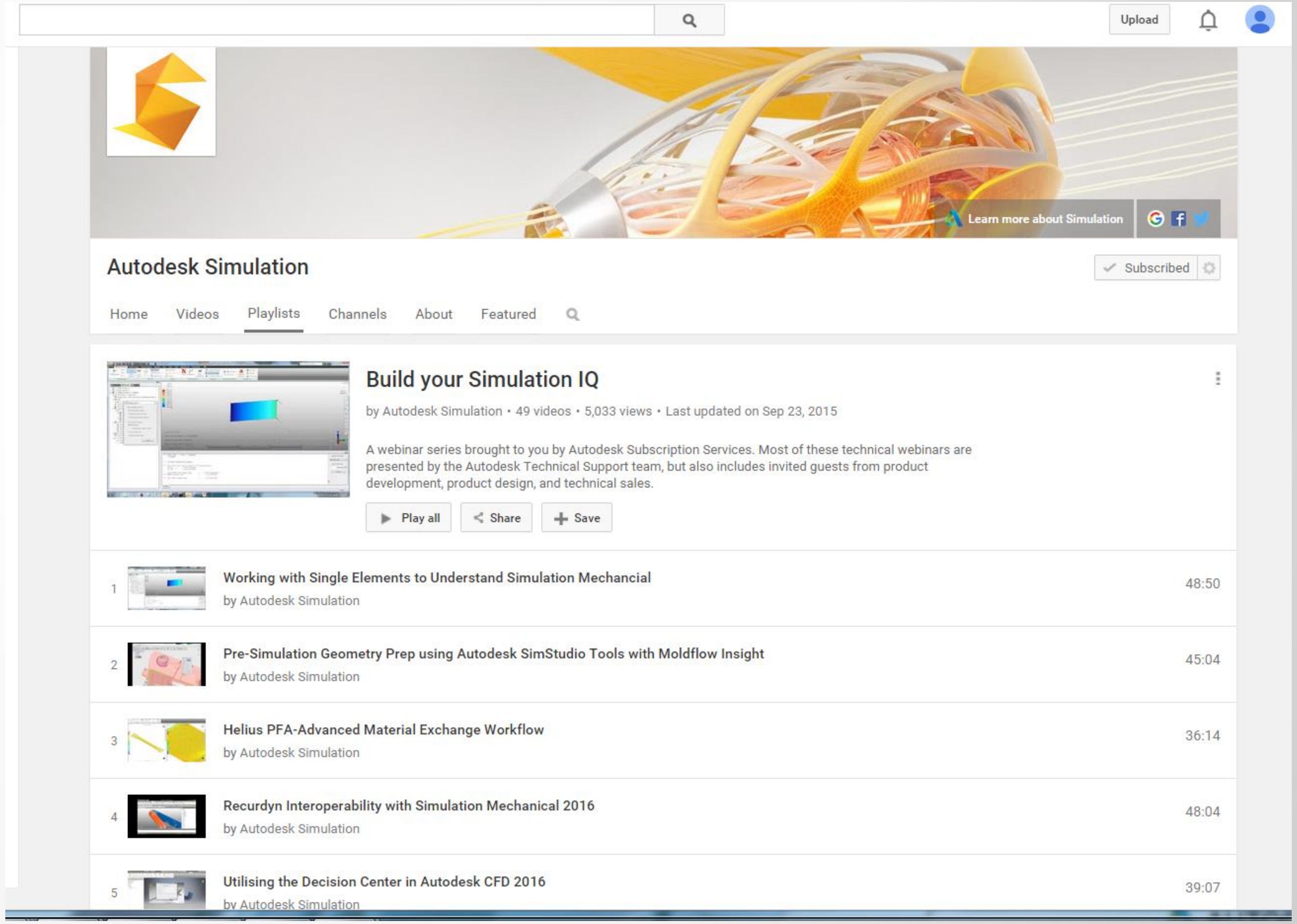


→ Can be found via the **Knowledge Network** or the **SimHub**

Resources – Build Your Simulation IQ Webinars



Register for live webinars, or watch them on-demand on Youtube.



Class summary

We will examine new workflows available to capitalize on your CAD data using SimStudio Tools for CAD cleanup or surface wrapping to nearly eliminate any CAD preparation. We will then do a deep dive into the new turbulence models available and the best practices and which ones to use for aerodynamics simulations. We will touch on mesh adaption best practices for aerodynamics to reduce the typical meshing guesswork and increase solving efficiency for complex aerodynamics models. The discussion will conclude with methods to accurately capture the wall force results to measure lift and drag on bodies using efficient techniques.

Key learning objectives

At the end of this class, you will be able to:

- Capitalize on **SimStudio Tools** or **Surface Wrapping for Simulation CFD** to reach the desired simulation aerodynamics goals
- Define which **advanced turbulence models** should be used for aerodynamics analysis
- Be more comfortable with implementing **mesh adaption** to eliminate the guess work from meshing complex aerodynamics models
- Extract accurate **steady state or transient wall force** results efficiently from different areas of a complex model.

What are some general goals with aerodynamics simulation?

- Lift
- Drag
- Side Forces
- Moments
- Pressure Distribution
- Pressure/Force mapping for FEA
- Force distribution around body
- Wake Structure/Frequency
- Visualization

- Series/Array of objects and their effects on each other
- Building Layouts within a city or campus
- Flow control: Breaks, radiator, sensors

My motivation

naca 0012 benchmark

Web | Images | Maps | Shopping | Videos | More | Search tools

About 19,100 results (0.26 seconds)

2D NACA 0012 Airfoil Validation
turbmodels.larc.nasa.gov/naca0012_val.html
Jul 27, 2015 - 2DN00: 2D NACA 0012 Airfoil Validation Case. The purpose here is to provide a validation case for turbulence models. Unlike verification ...
You've visited this page 2 times. Last visit: 9/30/14

[PDF] experimental flutter boundaries with unsteady pressure ...
ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19910018823.pdf NASA
by MG Farmer - 1991 - Related articles
achieved in testing the first model in the Benchmark. Models Program. Experimental flutter boundaries are presented for a rigid semi-span model (NACA 0012.

NACA 0012 benchmark model experimental flutter results ...
www.researchgate.net/.../23881779_NACA_0012_benchm... ResearchGate
by A Jose - 1992 - Cited by 68 - Related articles
NACA 0012 benchmark model experimental flutter results with unsteady pressure distributions on ResearchGate, the professional network for scientists.

Test Cases for Flutter of the Benchmark Models Rectangular ...
www.dtic.mil/cgi-bin/GetTRDoc?AD=ADP010713
by RM Bennett - 2000 - Related articles
As a portion of the Benchmark Models Program at NASA Langley (Ref 1), ... The NACA 0012 airfoil has a forward loading and for transonic flows, a shock forms.

Re: Autodesk Simulation CFD NACA0012 Benchmark Test - ...
forums.autodesk.com › CFD › General Discussion
Aug 22, 2014 - We have been conducting our own attempt to validate Simulation CFD 2015 by modelling the well researched and documented NACA0012 ...
More results from forums.autodesk.com

Autodesk Simulation CFD NACA0012 Benchmark Test Edited
Options
1240 Views, 23 Replies
08-22-2014 03:49 AM

We have been conducting our own attempt to validate Simulation CFD 2015 by modelling the well researched and documented NACA0012 aerofoil section in 2D. We have been very disappointed with our results and would be interested if anybody else has attempted a similar test. Following the advice in the Autodesk help files, we have the following settings:

- Section: naca0012 (Have tried both sharp point and blunted TE profiles)
- AOA : 10 deg (Have also looked at 8deg but has similar problems)
- Reynolds Number : 1e6 (Match to data)
- Mach Number : 0.15 (Match to data)
- Farfield : 30chords (have tried ranges 10c to 100c, only about 1% different)
- Mesh : Four concentric zones, mesh size =1/300th chord near section, increasing upto about 1 chord in farfield. Mesh growth rate reduced to 1.01 to smooth transitions. Typically produces mesh with more than 100000 fluid nodes.
- BL Mesh Enhancement : ON (BL Layers : 10).
- Physics : Compressible & incompressible (compressible is often unstable)
- Turbulence Model : SST K-omega
- Free stream turbulence intensity : 0.01 (reduced from default of 0.05 as this is external flow, all other turbulence settings default).
- Advection Scheme : ADV2 (ADV5 is not significantly better and more often unstable).
- Convergence Tolerances : Tight

Over a range of settings we have never achieved an error in lift force lower than 15%, and computed drag is typically 6x higher than published data for these conditions. I have seen considerably better results published on-line from Fluent and Cosmol simulations.
Is this really the best we can expect for this software, or am I doing something wrong ?

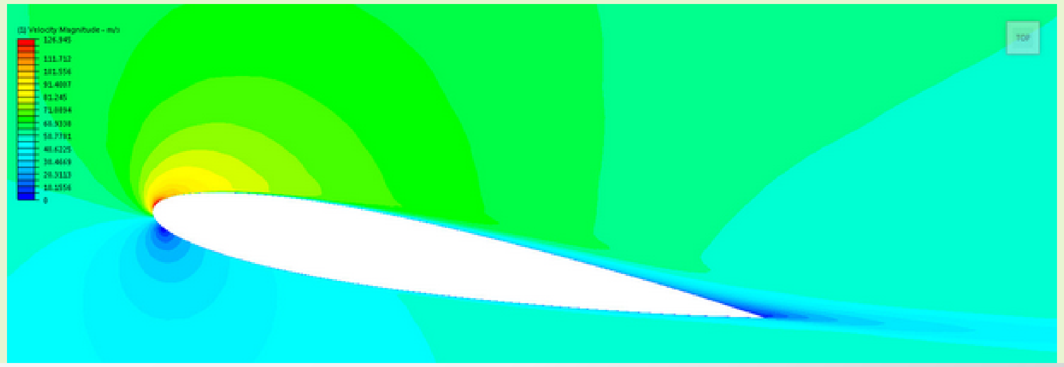
I have sent this also to Autodesk Support, with relevant files included.

Solved! by Royce_adsk. See the answer in context. Kudo | 4 Kudos

Hi Wolfgang,

Here is a summary of the results using below recommended settings for this analysis in Simulation CFD 2015:

- Lift Error: < 3%
- Drag Error: < 4%
- Your "F%" (resultant force) error: < 2%

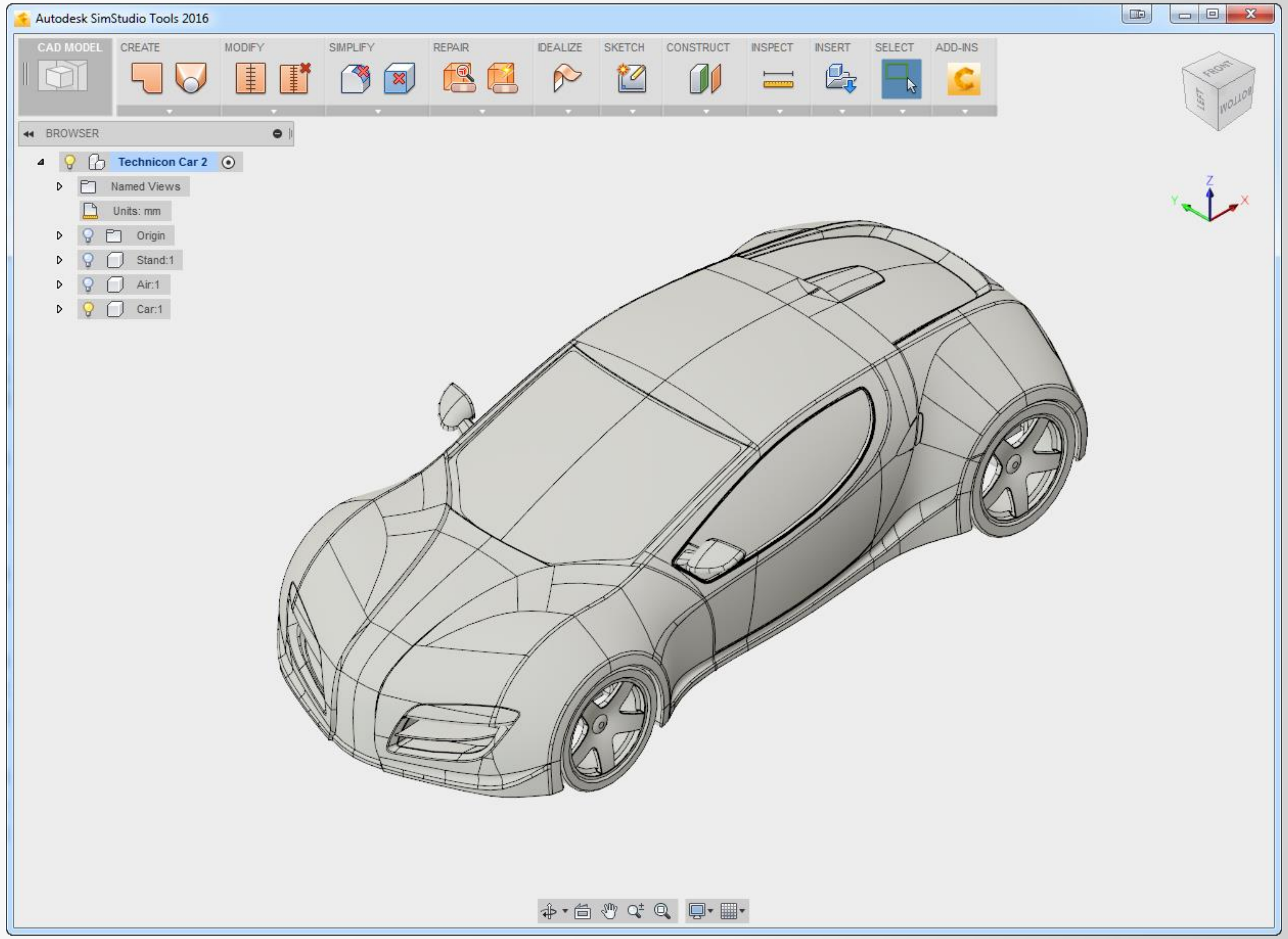


Capitalize on SimStudio Tools or Surface Wrapping

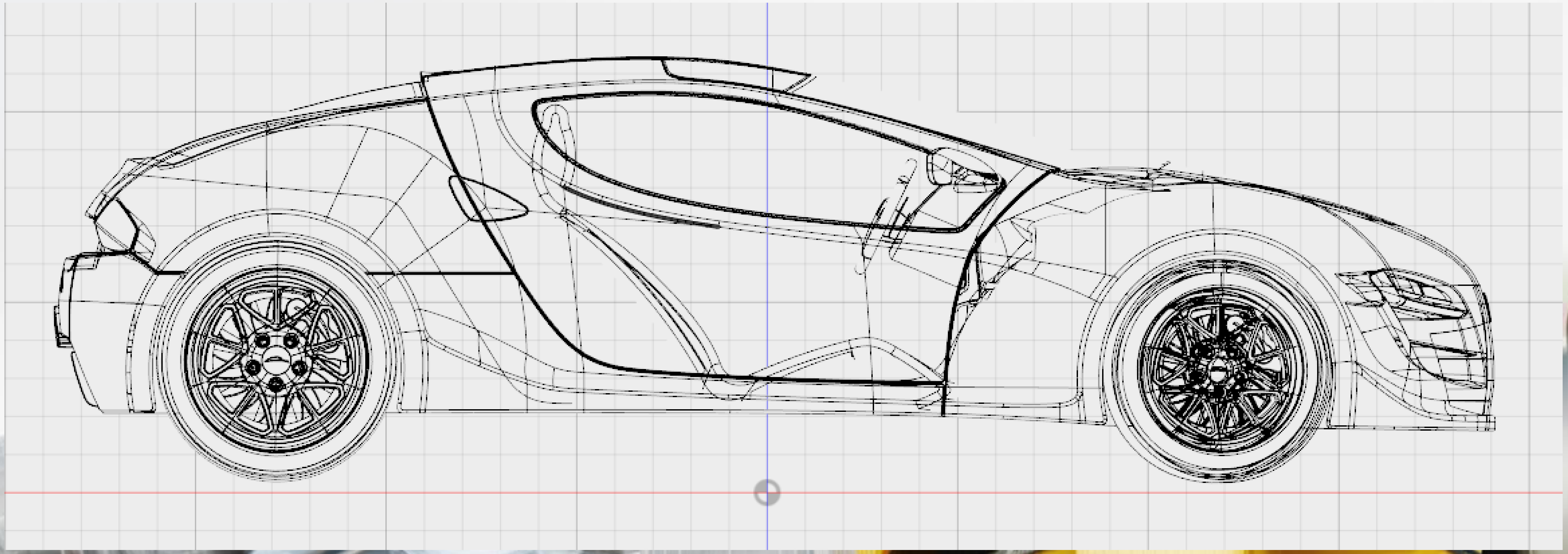
SimStudio Tools – CAD Prep

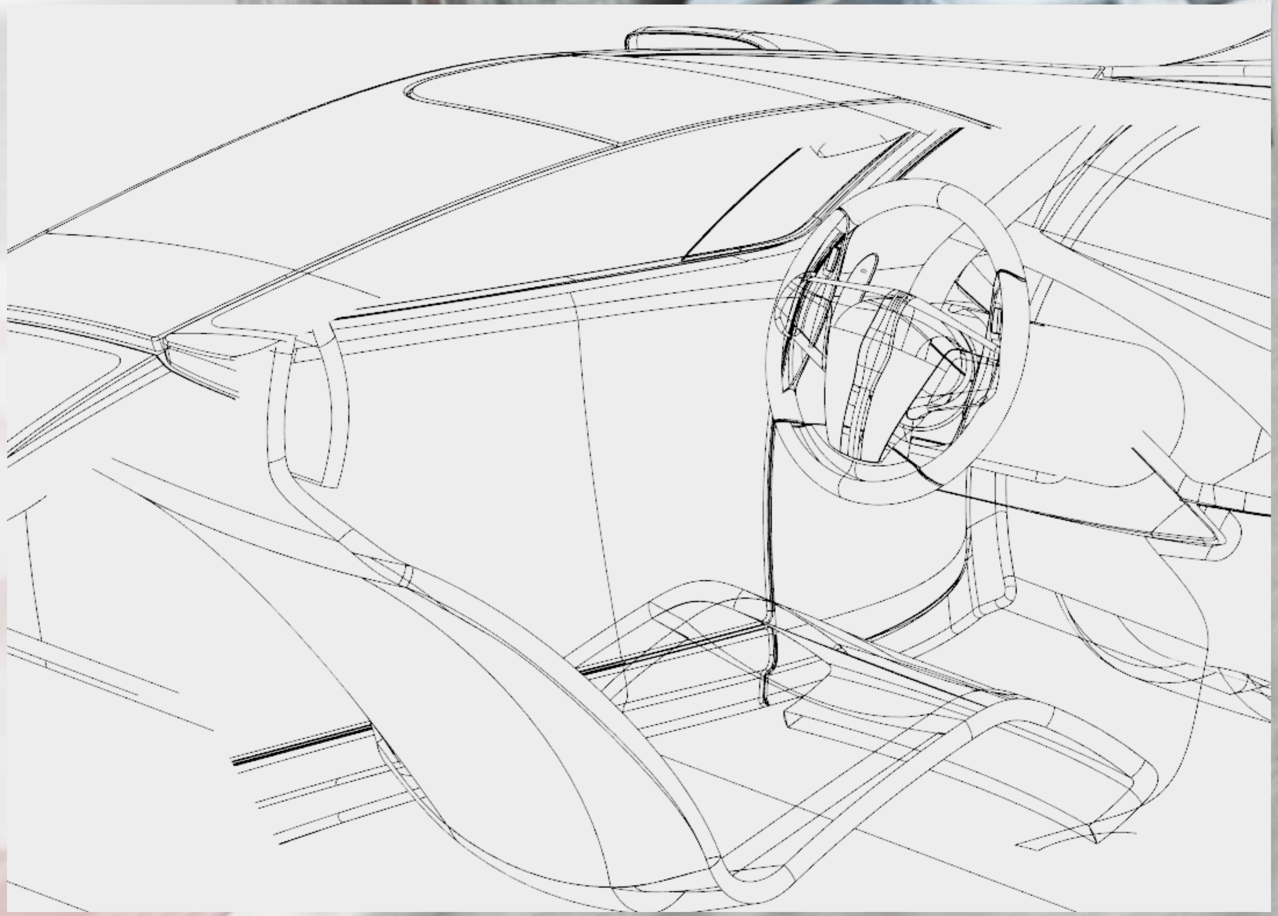
What is SimStudio Tools?

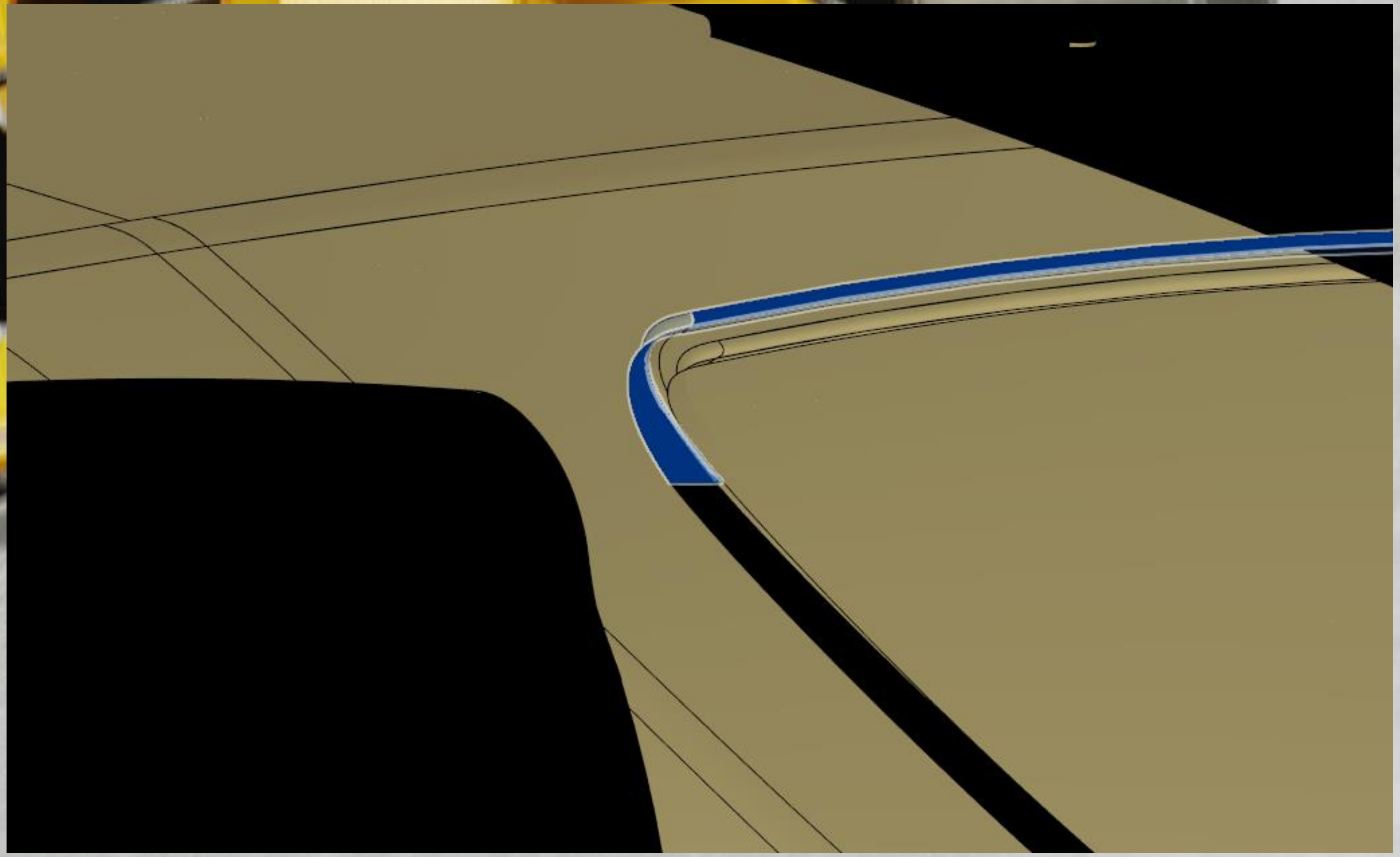
- Built off Fusion360
- CAD tool geared for Simulation Community
- Fastest way to prepare your geometry to be simulation ready.



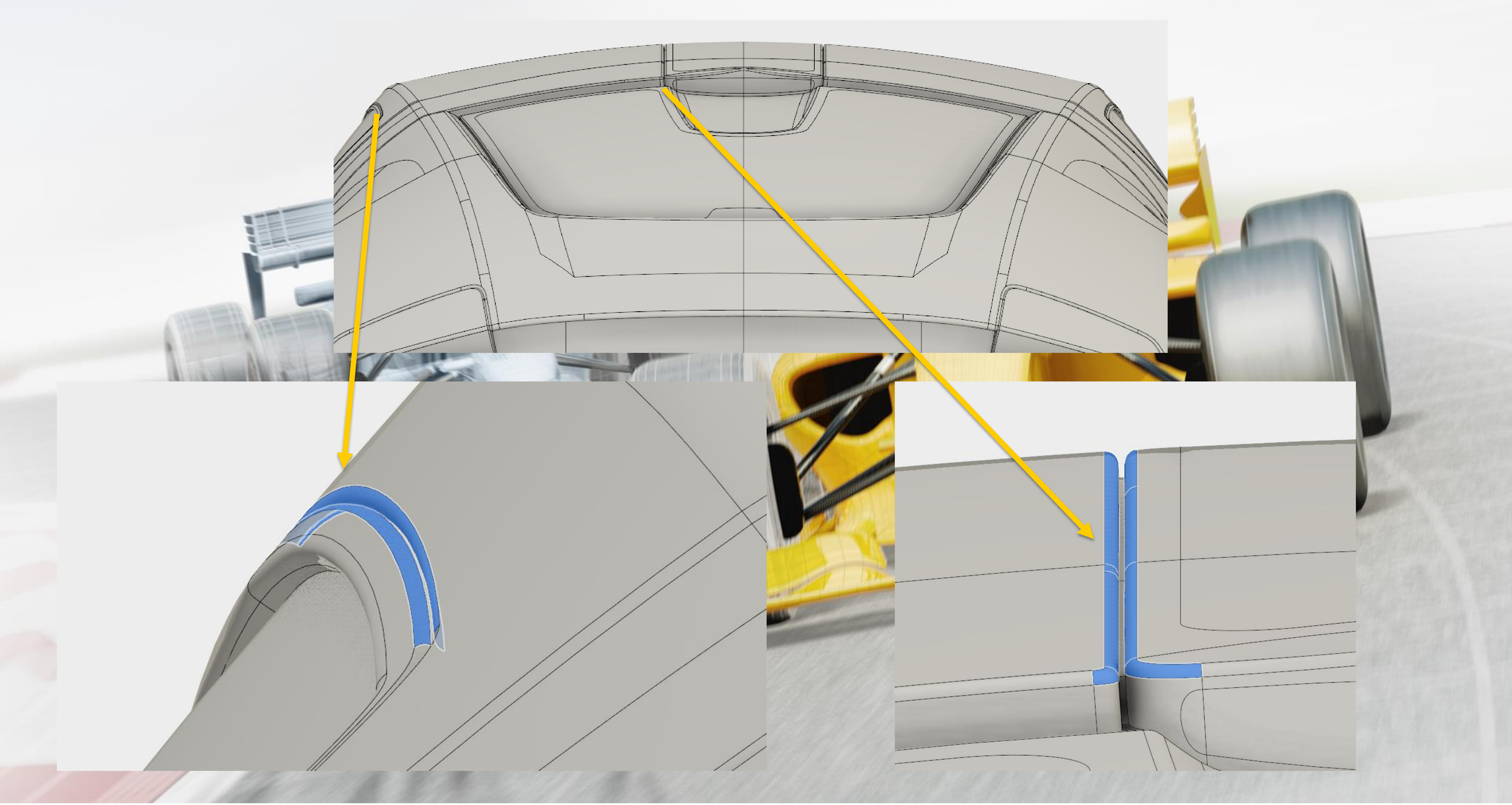
Alias Dirty CAD





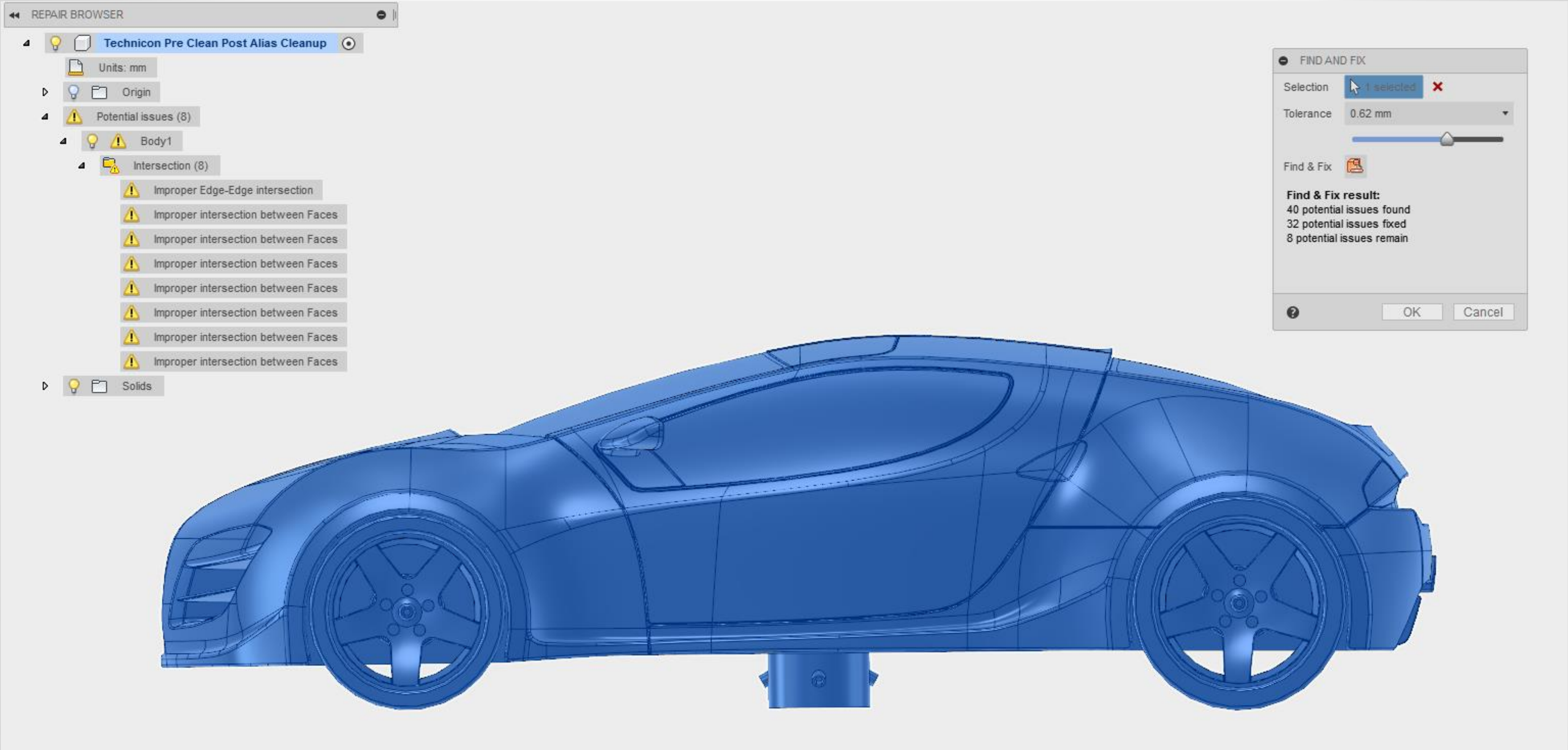


Alias ‘Clean’ CAD

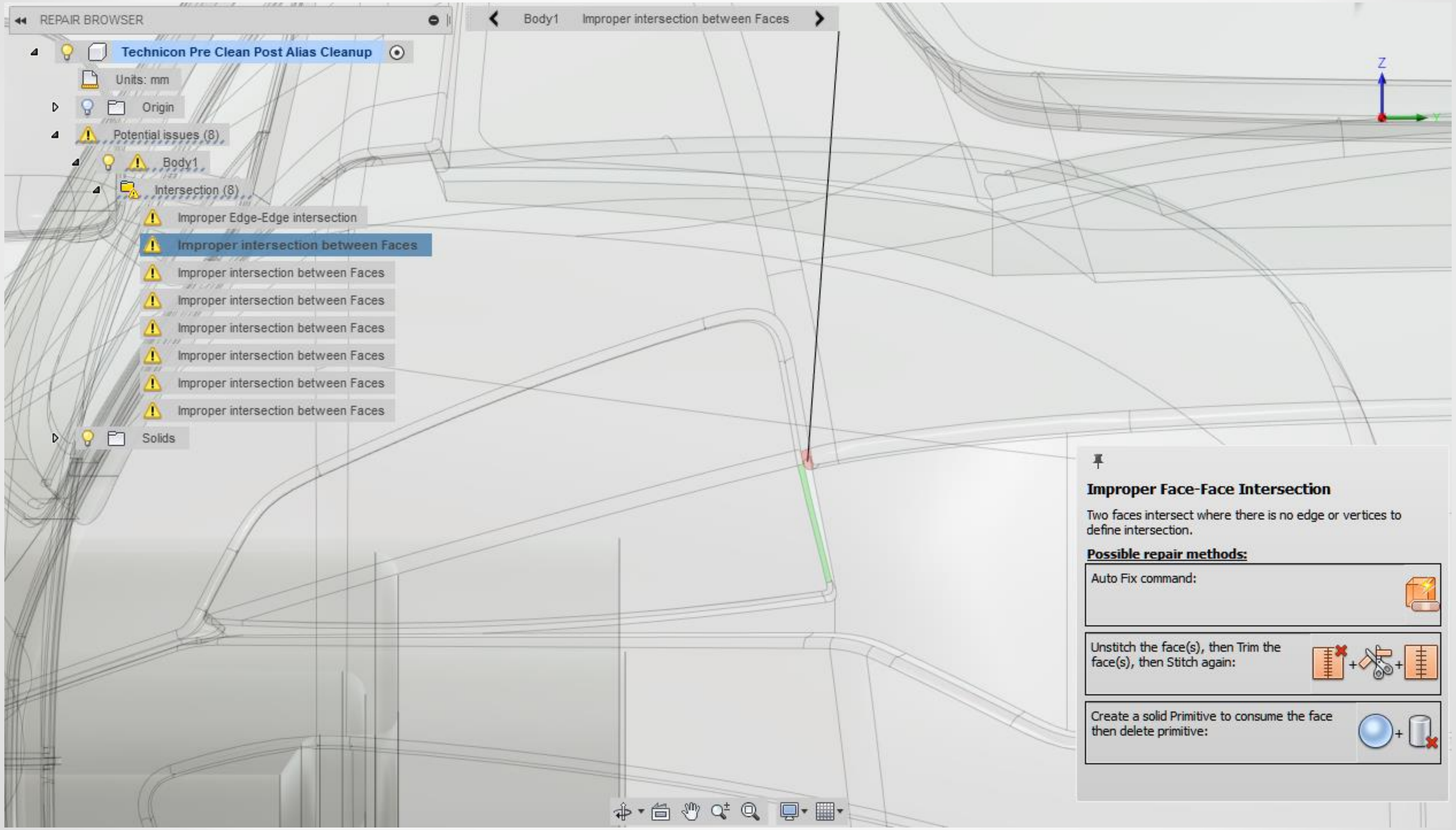


SimStudio Tools – Find and Fix

8 Potential Issues Remain

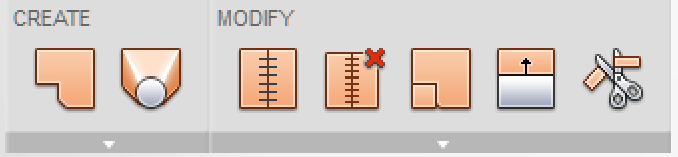


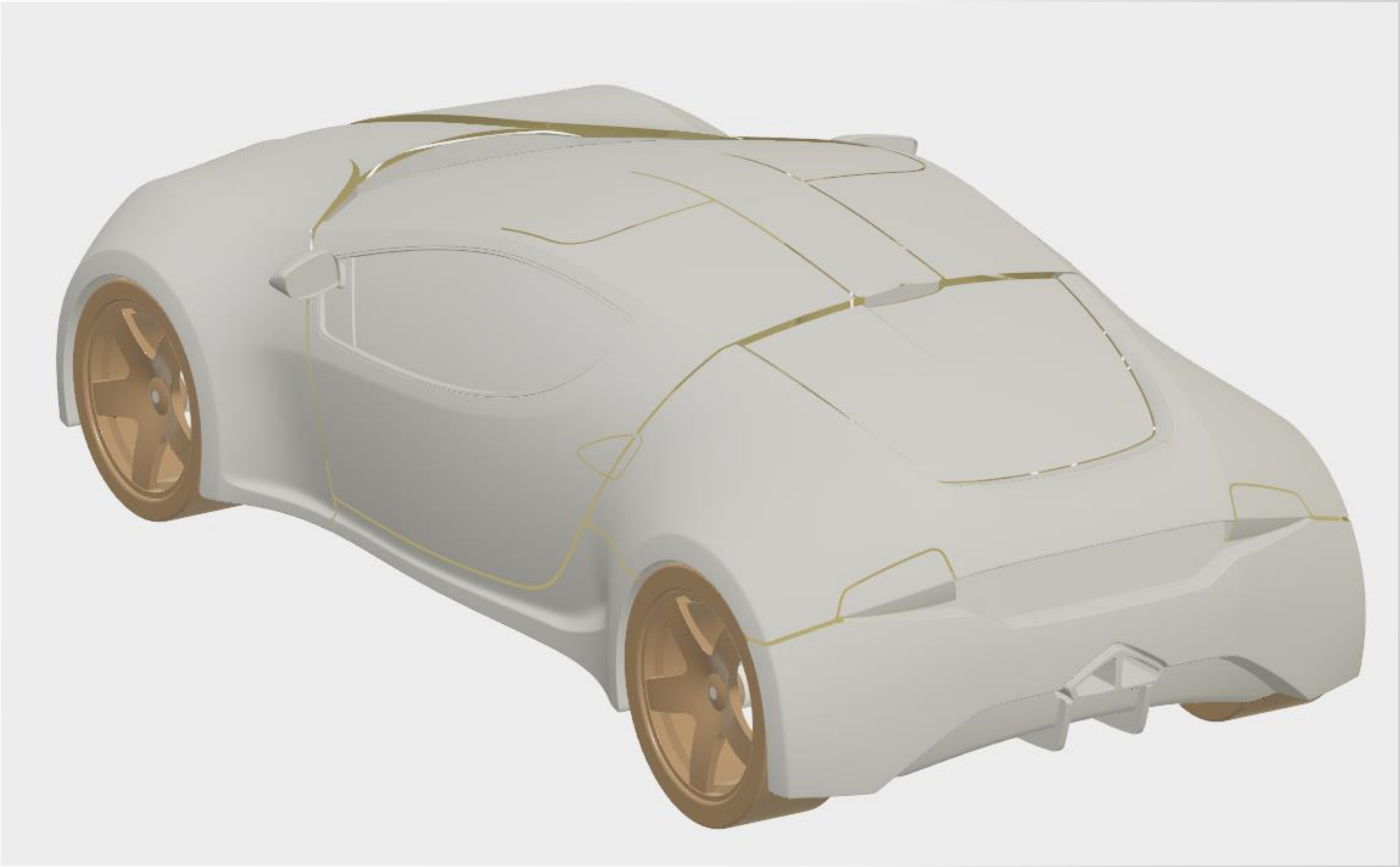
SimStudio Tools – Find and Fix
Locates issues with suggested solutions!



SimStudio Workflow model simplification

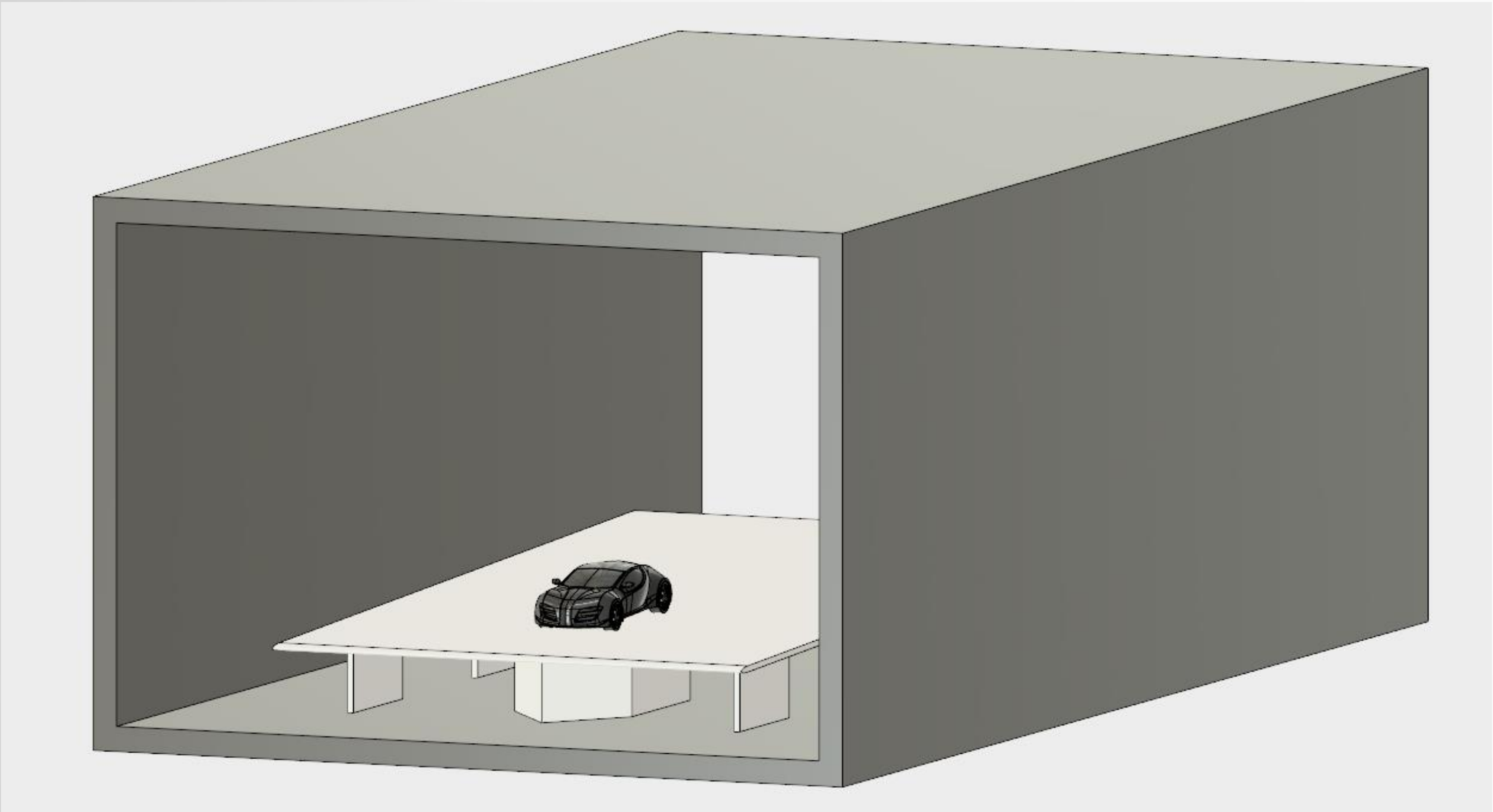
1. Remove Seams
2. Fill Seams
   - Patch
   - Loft
   - Stitch/Unstitch
   - Merge
   - Extend Face/Trim

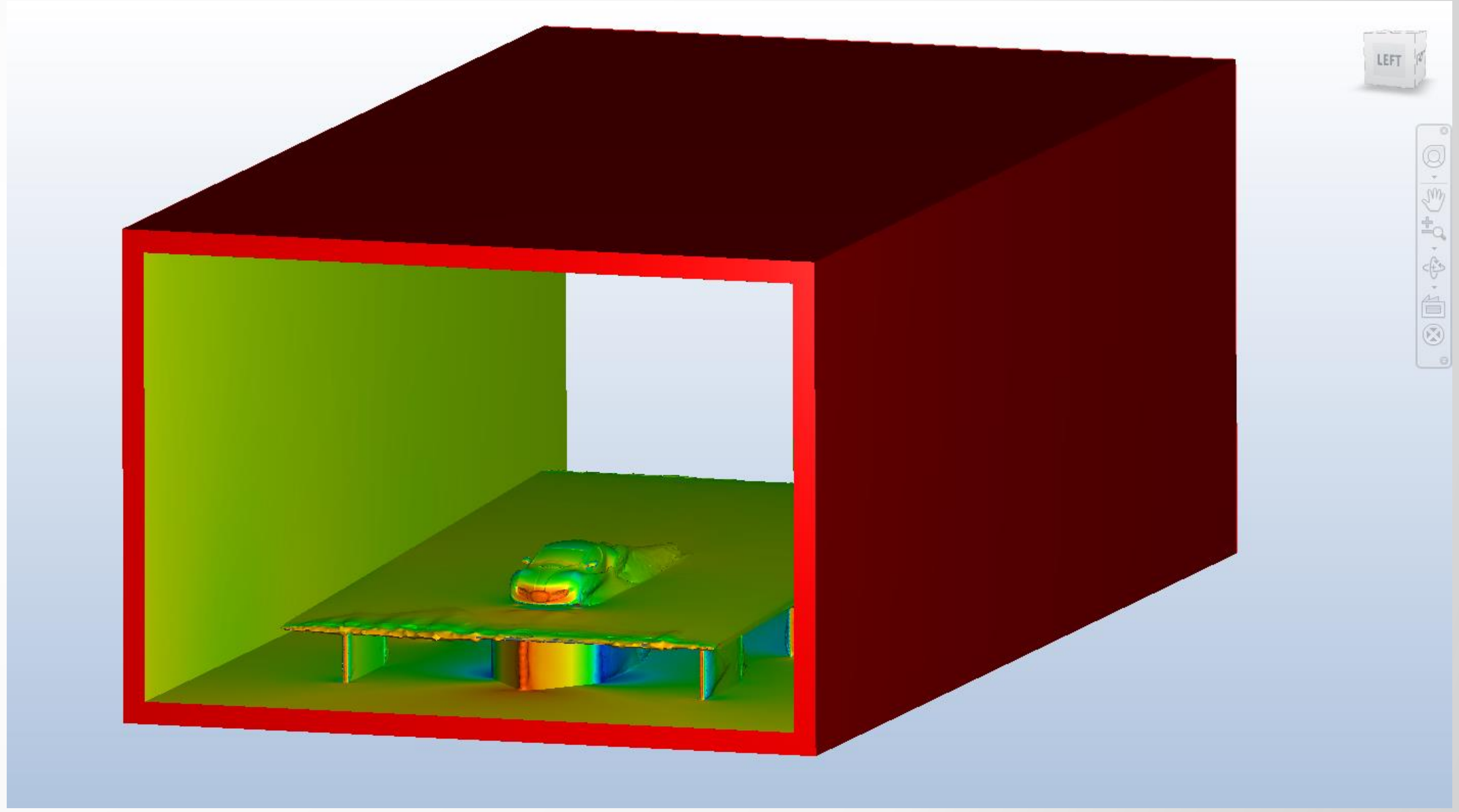




Examine Work

Car Prepared for Wind Tunnel and Simulation Ready!

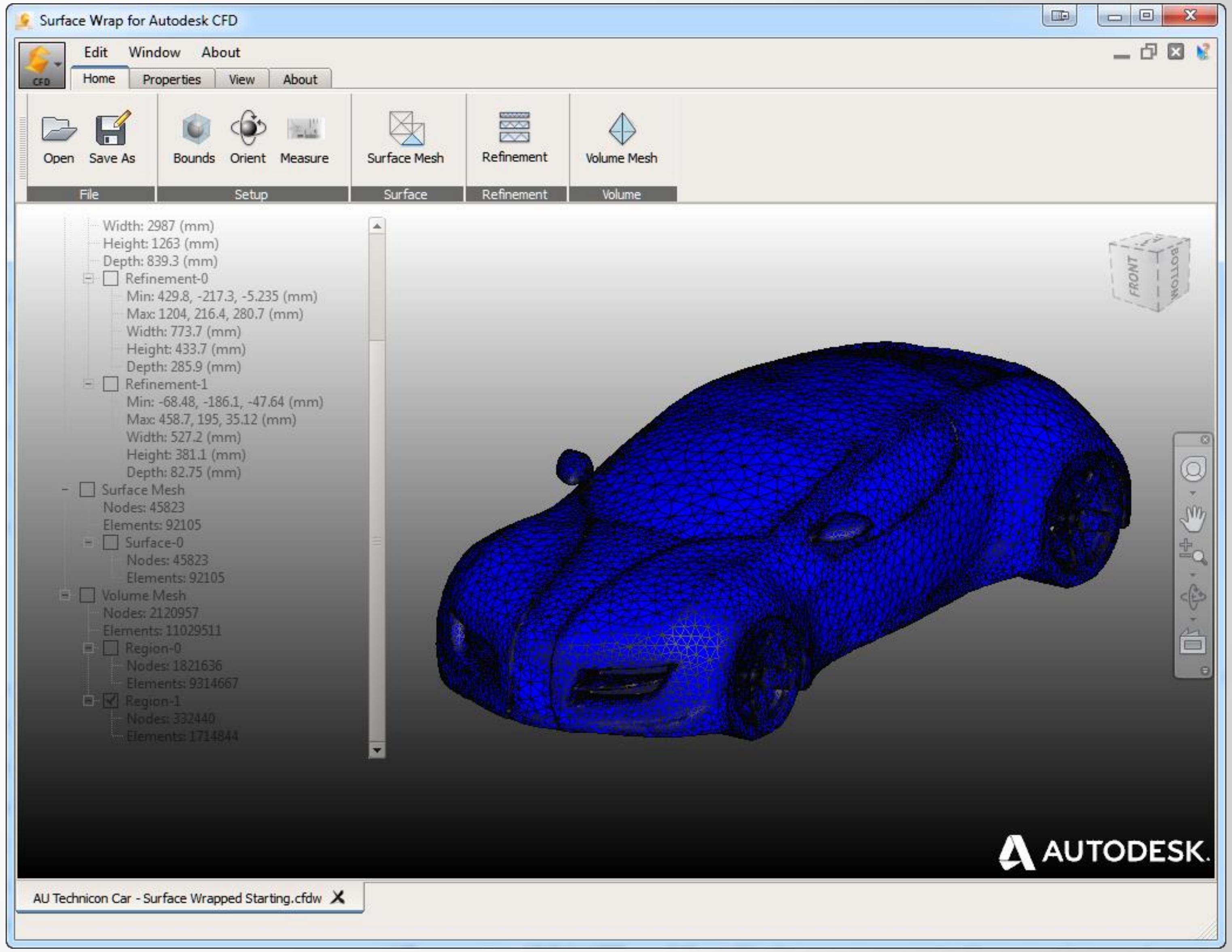




Surface Wrapping – Avoid CAD

What is Surface Wrapping for Autodesk CFD?

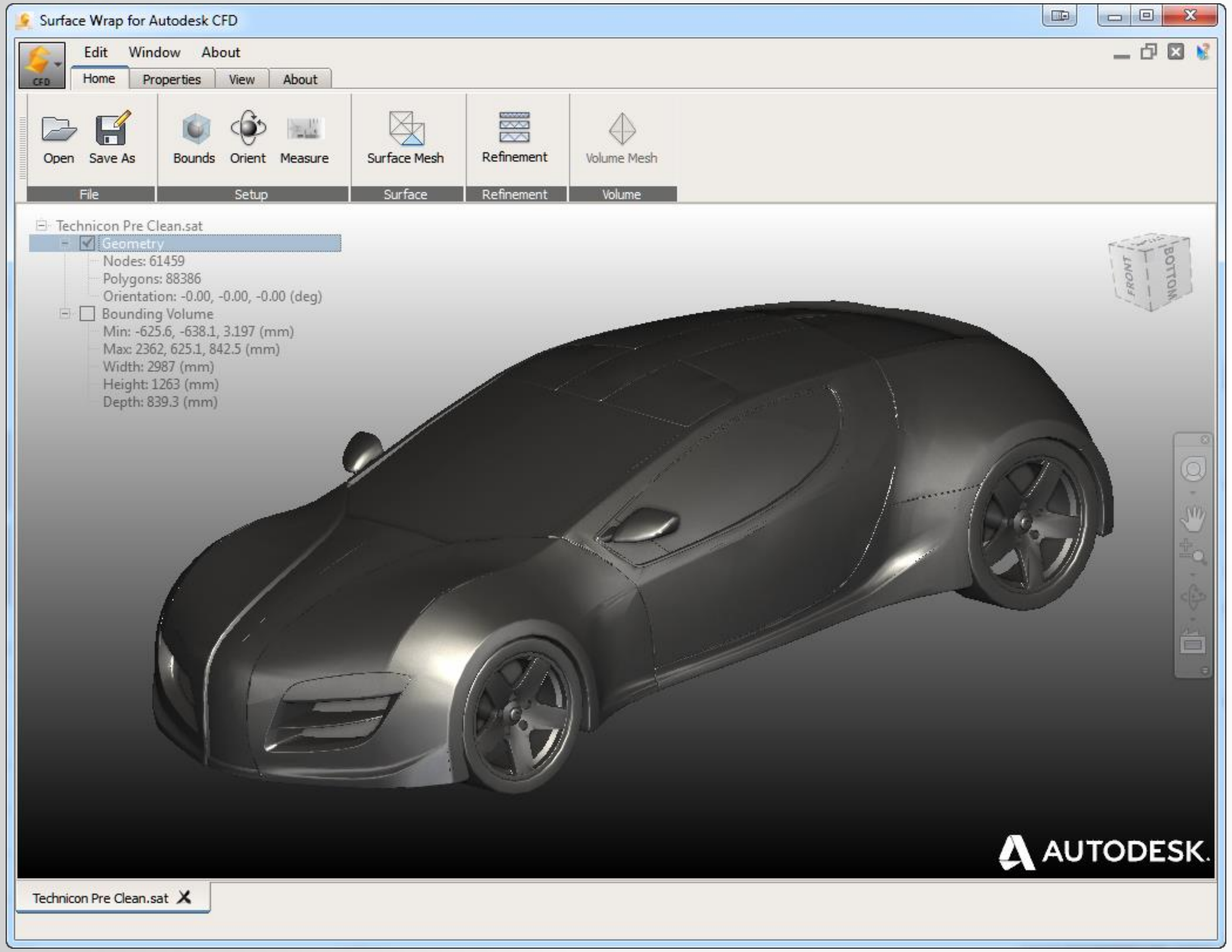
- Geometry Surface Wrapping technology
- External Flow/Wind Tunnel
- Can simulate difficult to nearly impossible geometry without extensive CAD cleanup

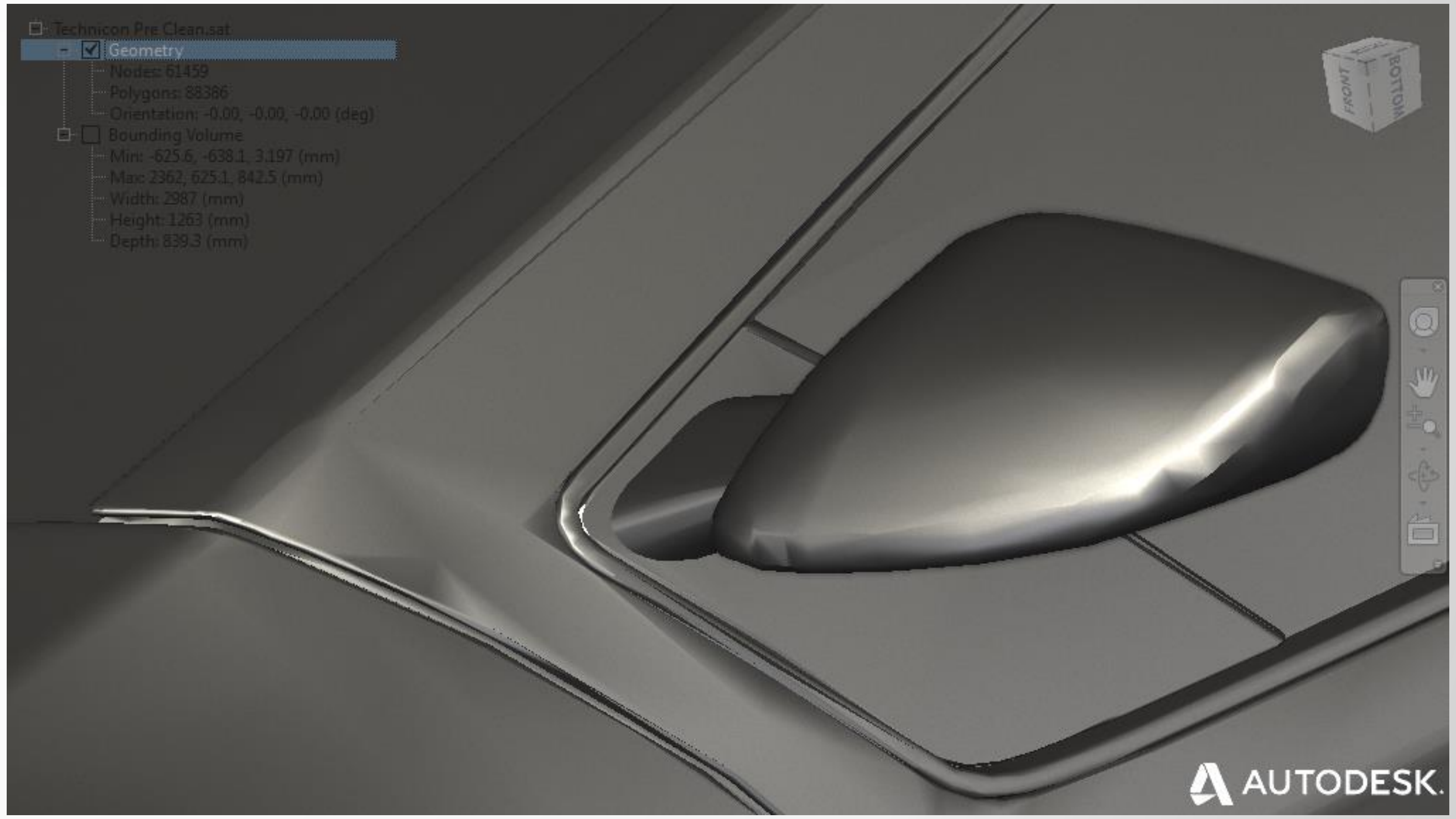


Basic Steps for Surface Wrapping

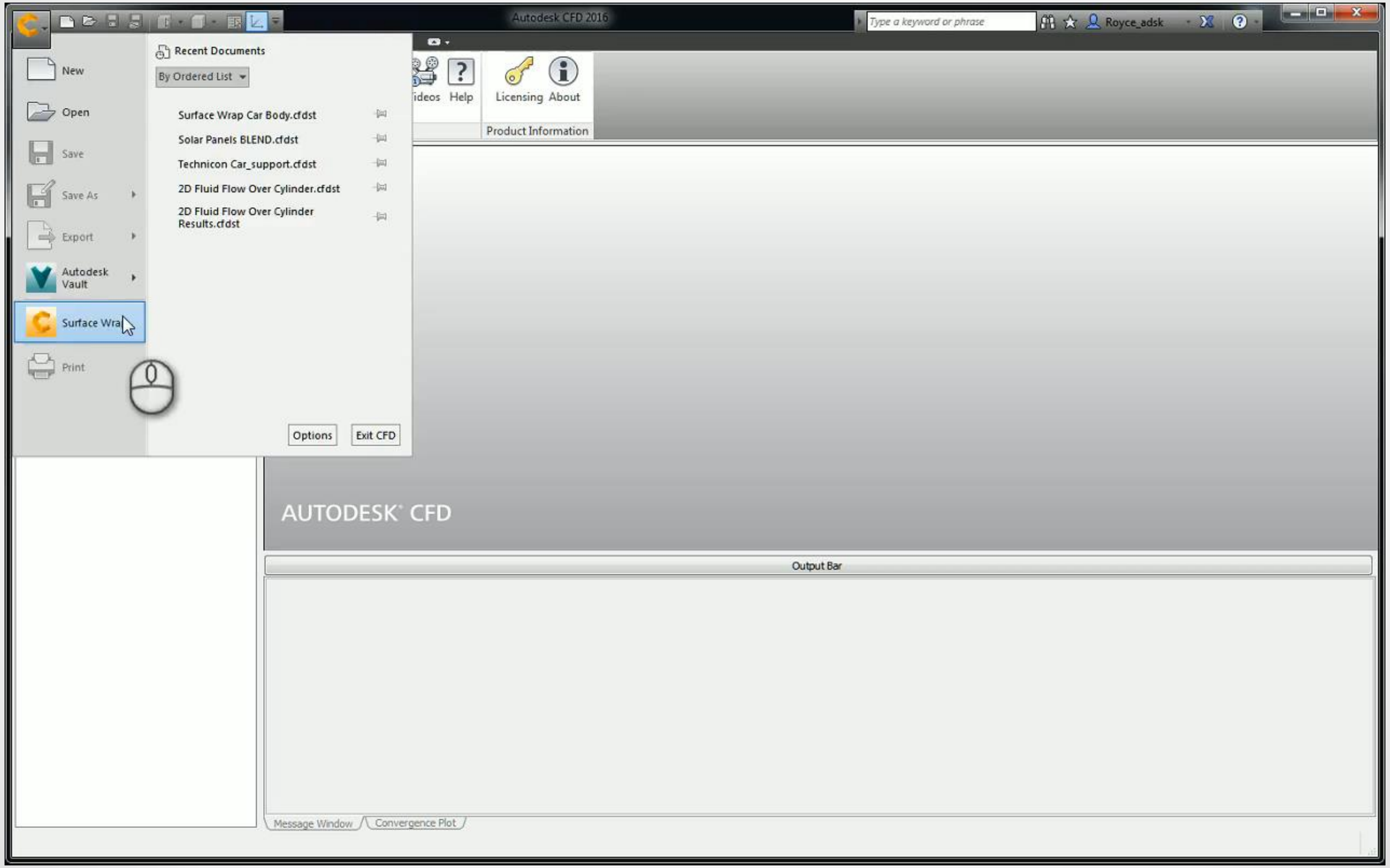
1. Open geometry
2. Define Wind Tunnel Geometry
3. Build Surface Wrap
4. *Optional*: Define Refinement Regions
5. Generate Volume Mesh
6. Export NAS file
7. CFD: Create new design study and CAD Mesh File

Dirty CAD Ready for Surface Wrapping

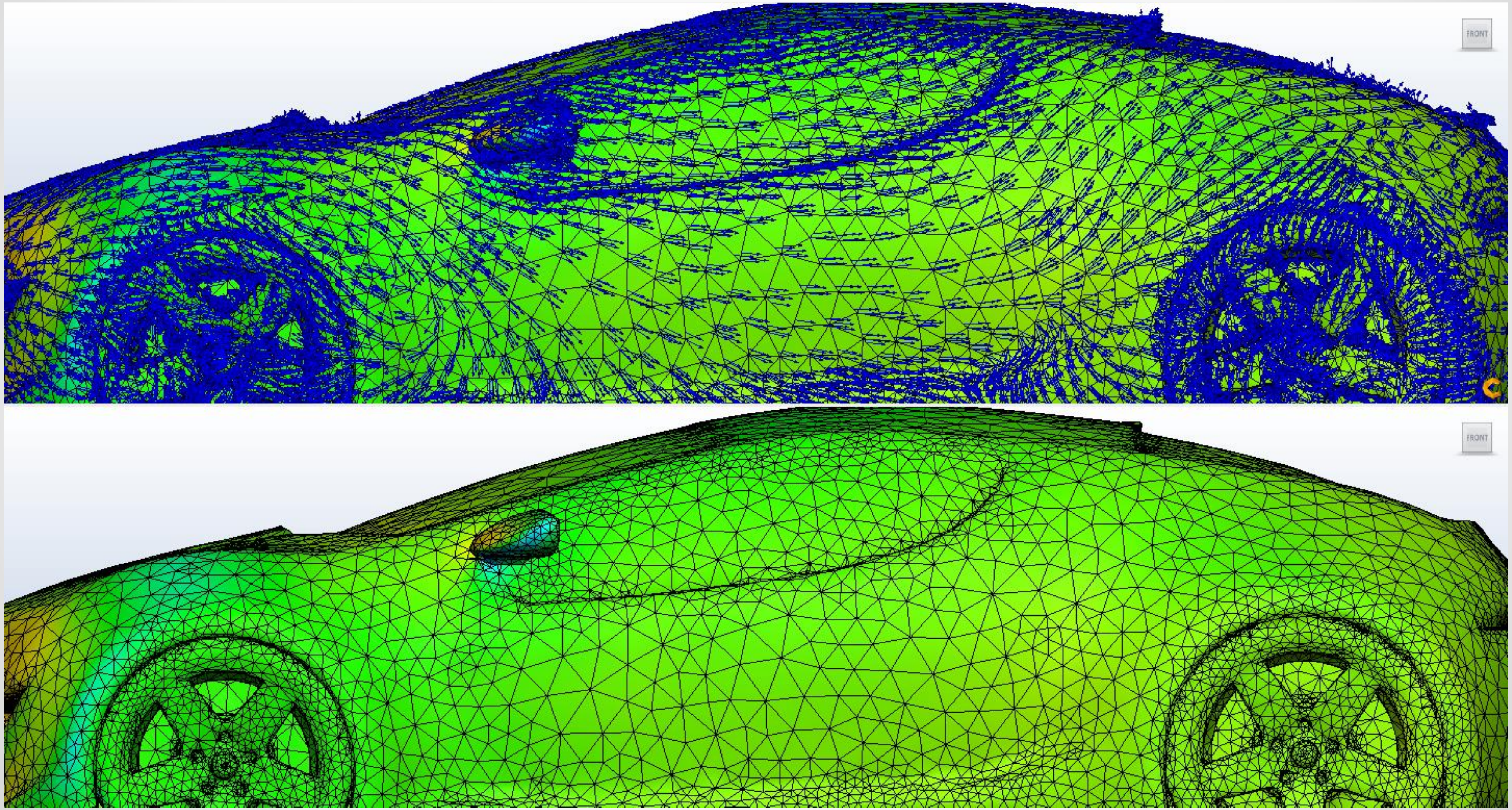




Surface Wrapping Demo



Surface Wrapped Car – Side View Results



Which method should be used?

SimStudio Tools

- Time Available
- Leverage all CFD Meshing Tools/Adaptation
- Highly Accurate Model Required
- Forces measured at different zones in model
- Complex Wind Tunnel Setups
- More precise geometry position control

Surface Wrapping

- Need Results ASAP!
- Rough Measurements/Trending
- Geometry too complex
- Only Visualization required
- System Level Forces

Advanced Turbulence model for Aerodynamics

Agenda

- What is Turbulence?
- How is Turbulence modeled in CFD Software?
- General Timeline of Turbulence Models Academic Development

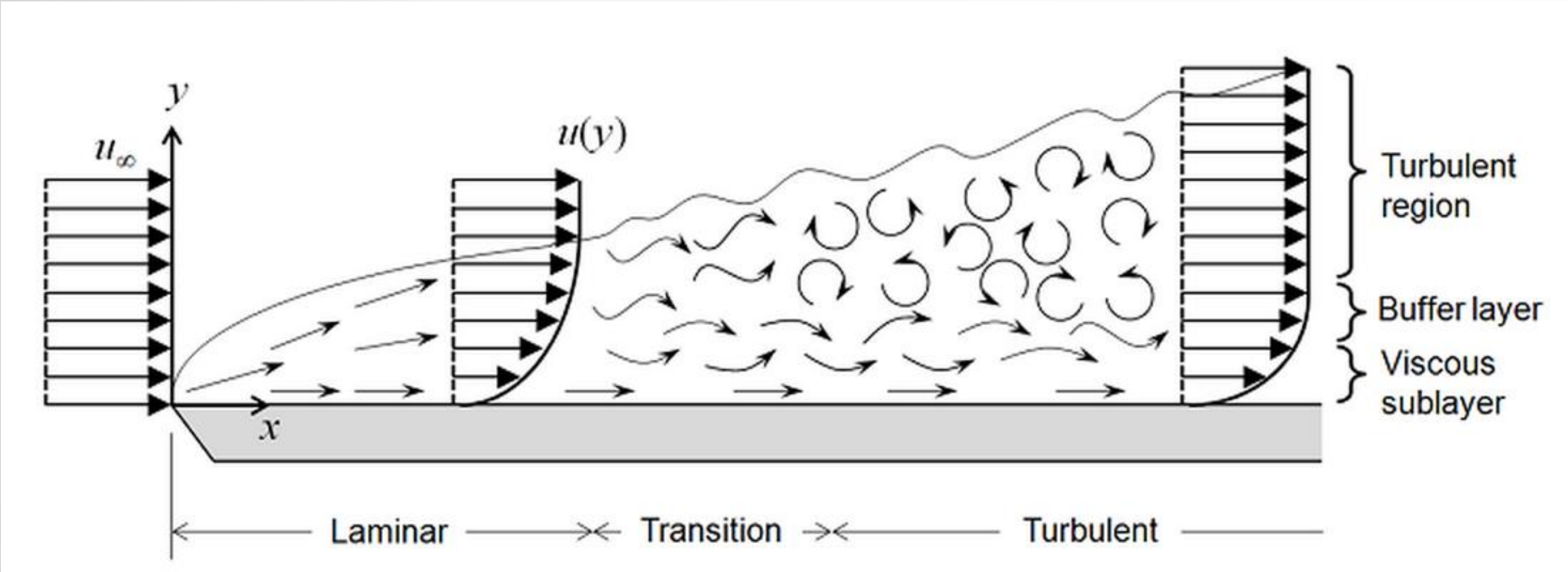
- Overview of Select Turbulence Models
  - K-omega SST
  - K-omega SST RC (Hellsten)
  - K-omega SST RC (Smirnov-Menter)

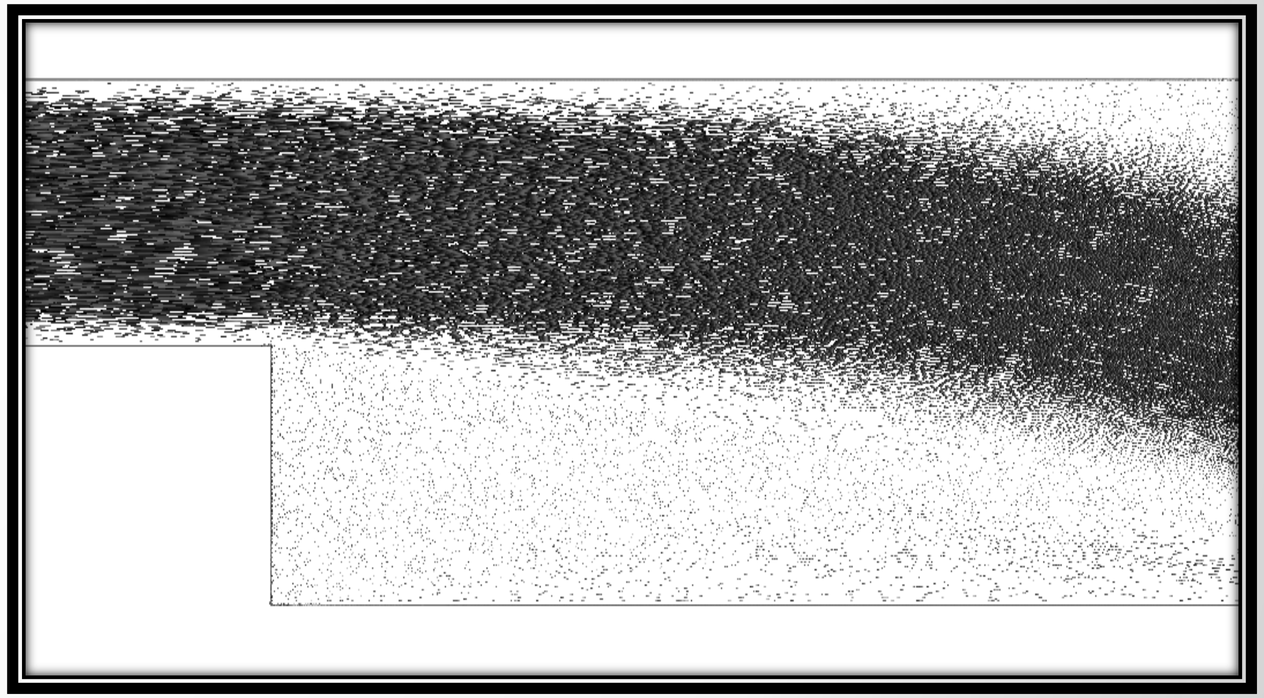
  - K-omega SST SAS
  - K-omega SST DES

- Best Practices
- Examples

What is Turbulence?

Noun: Violent or unsteady movement of a fluid.





How is Turbulence modeled in CFD Software?

- **Flow Averaging** (Reynolds Averaged Navier-Stokes *RANS*)
- Subgrid-Scale Models (Large Eddy Simulation *LES*)
- Hybrid of Flow Averaging and Subgrid-Sale Models (Detached Eddy Simulation *DES*)
- Direct Numerical Simulations (DNS) – No turbulence model used

- Wall Functions

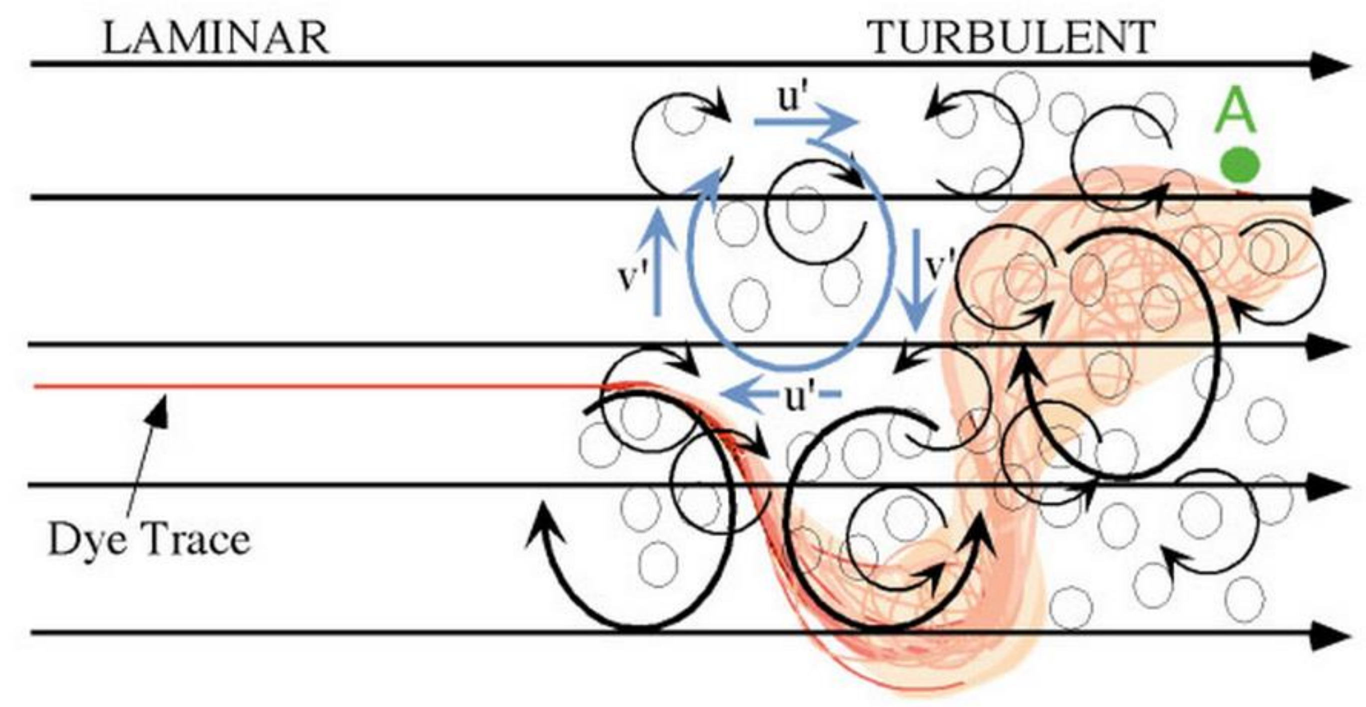
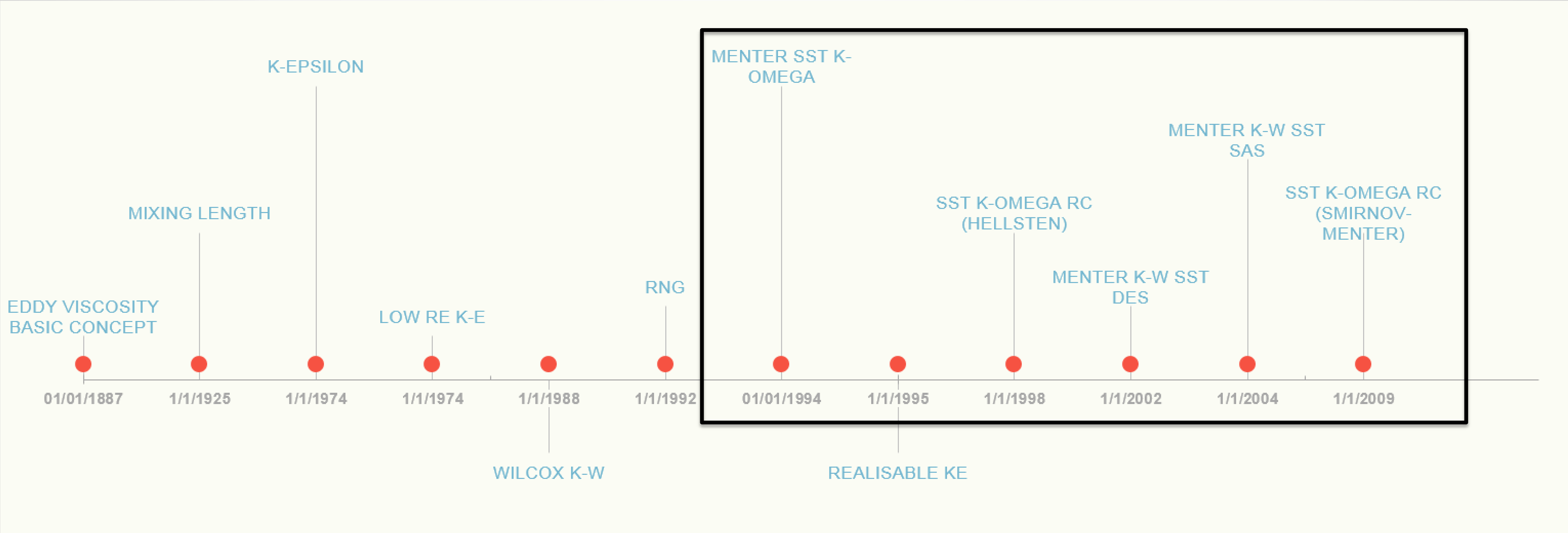


Image Courtesy dspace.mit.edu

Selected Turbulence Model Research Timeline



SST k-omega

Two-equation, linear eddy viscosity RANS

- SST (shear stress transport) k-omega is a hybrid turbulence model combining the best of k-epsilon and k-omega.
- Near-wall regions are resolved with k-omega, allowing for simulation down to the viscous sub-layer.
- The model transitions into k-epsilon as the distance from a wall increases.
- Wall Considerations
  - y+ can be up to 100
  - Ideal y+ value of less than 2 with a goal of 0.3
  - Very Robust 10-100
  - Blending of direct calculation when refined mesh is present and wall functions for higher y+ value

Pros:

- Robust across a wide range of flows
  - Accurate for predicting flow separation
  - Ideal for adverse pressure gradients (occurs with flow separation)
- Accurate for high Prandtl number (liquids) wall heat flux prediction

Cons:

- Can require a very fine boundary layer mesh 10-15 mesh enhancement layers
- Lacks ability to capture high curvature

SST k-omega RC (Hellsten)

Two-equation, linear eddy viscosity RANS

- SST (shear stress transport) k-omega is a hybrid turbulence model combining the best of k-epsilon and k-omega.
- Near-wall regions are resolved with k-omega, allowing for simulation down to the viscous sub-layer.
- The model transitions into k-epsilon as the distance from a wall increases.
- Wall Considerations
  - y+ can be up to 100
  - Ideal y+ value of less than 2 with a goal of 0.3
  - Very Robust 10-100
  - Blending of direct calculation when refined mesh is present and wall functions for higher y+ value

Pros:

- Robust across a wide range of flows
  - Accurate for predicting flow separation
  - Ideal for adverse pressure gradients (occurs with flow separation)
- Accurate for high Prandtl number (liquids) wall heat flux prediction
- Slight mathematically change to Standard SST k-omega
- Theoretically, correction shouldn't impact when not needed
- Good pressure accuracy for cyclones

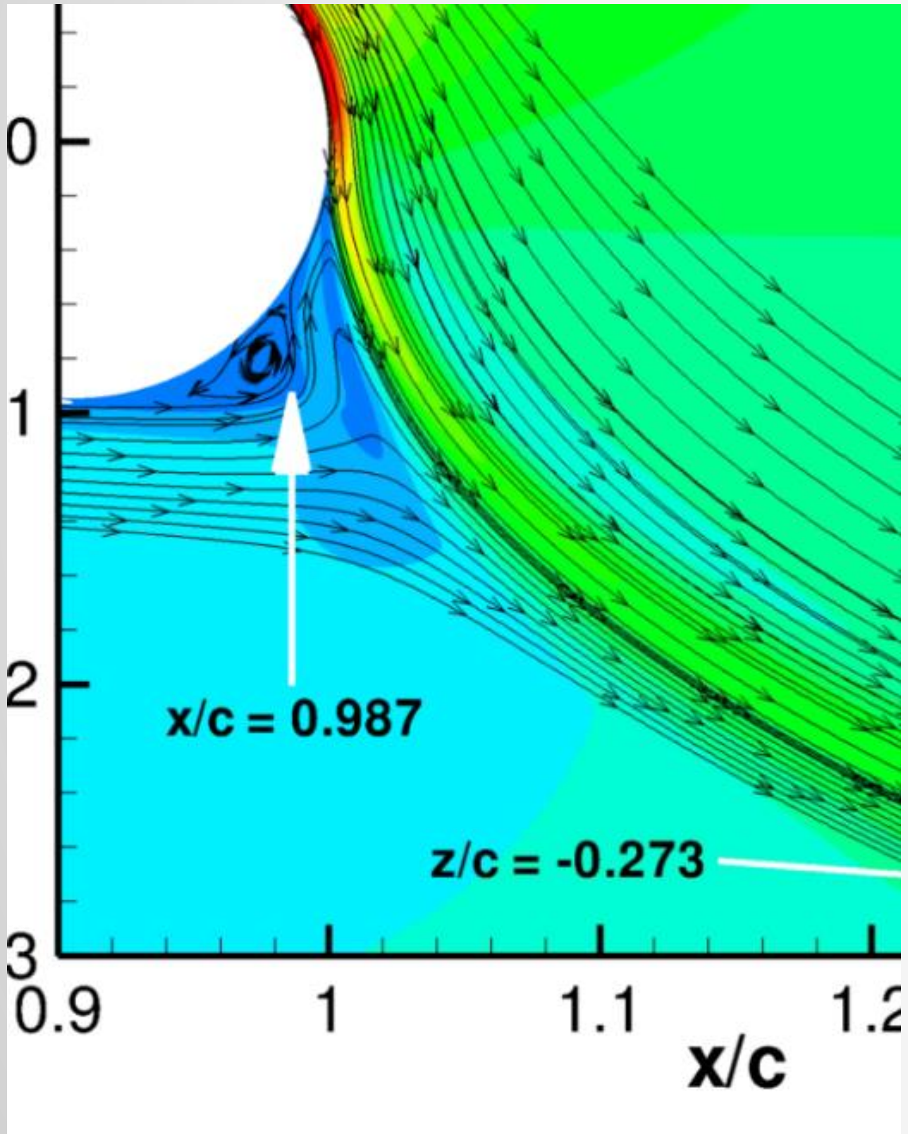
Cons:

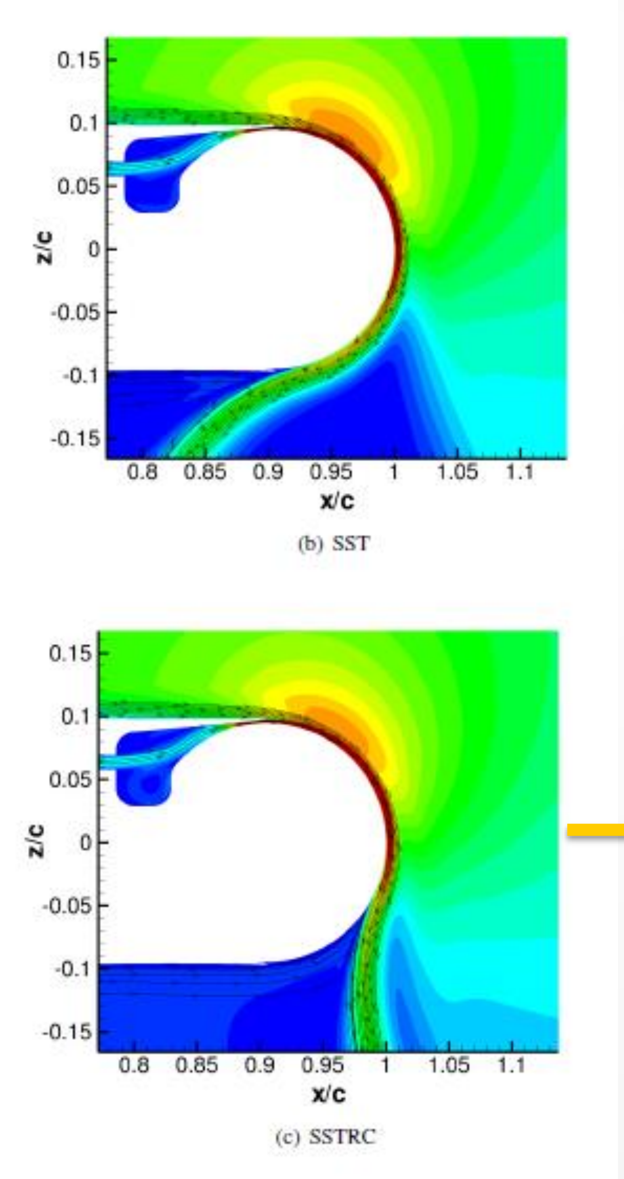
- Can require a very fine boundary layer mesh 10-15 mesh enhancement layers
- Moderate flow profile accuracy for cyclones

Why use the Hellsten model?

Coandă Effect: Tendency for fluid jet to be attracted to a nearby surface

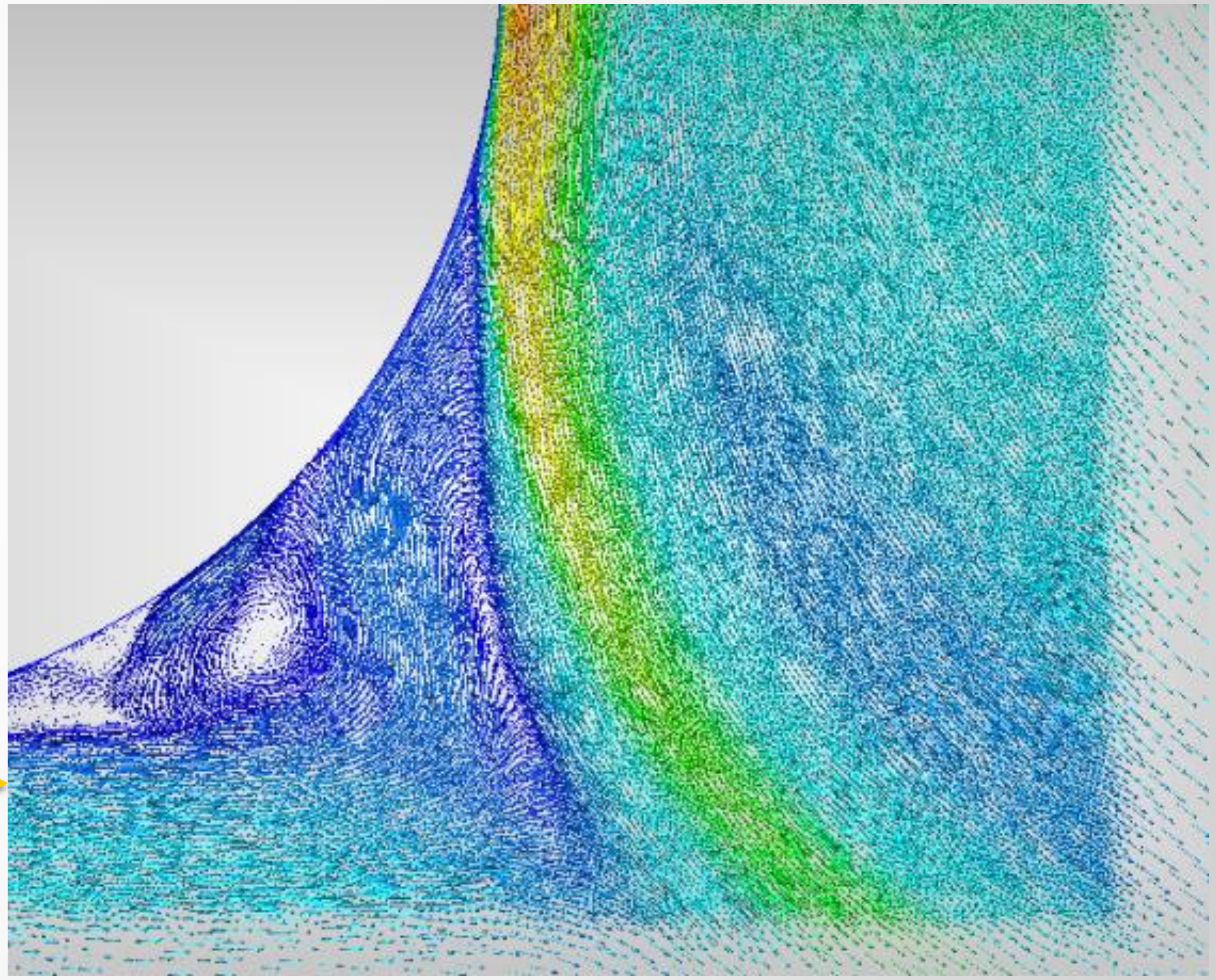
NASA Langley Study of SST Models





(b) SST
(c) SSTRC

Autodesk CFD Results w/Hellsten



Numerical Study Comparing RANS and LES Approaches on a Circulation Control Airfoil Rumsey (NASA Langley)

SST k-omega RC (Smirnov-Menter)

Two-equation, linear eddy viscosity RANS

- SST (shear stress transport) k-omega is a hybrid turbulence model combining the best of k-epsilon and k-omega.
- Near-wall regions are resolved with k-omega, allowing for simulation down to the viscous sub-layer.
- The model transitions into k-epsilon as the distance from a wall increases.
- Wall Considerations
  - y+ can be up to 100
  - Ideal y+ value of less than 2 with a goal of 0.3
  - Very Robust 10-100
  - Blending of direct calculation when refined mesh is present and wall functions for higher y+ value

Pros:

- Robust across a wide range of flows
  - Accurate for predicting flow separation
  - Ideal for adverse pressure gradients (occurs with flow separation)
- Accurate for high Prandtl number (liquids) wall heat flux prediction
- Mathematically more complex than Hellsten
- Theoretically, correction shouldn't impact when not needed
- Similar pressure drop accuracy for cyclones as Hellsten
- More accurate flow profiles for cyclones

Cons:

- Can require a very fine boundary layer mesh 10-15 mesh enhancement layers

SST k-omega Scale Adaptive Simulation (SAS)

Two-equation, linear eddy viscosity URANS

- Simulates the formation of turbulent structures in unsteady regions
- Can be used for Steady State and Transient simulations. *Using in a transient analysis is more appropriate.*
- Dynamically adjusts to resolved structures which results in a LES-like behavior in unsteady flow regions

Pros:

- State of the art turbulence modeling technique for simulations
- More reasonable for use with industrial models when it is needed
- Less mesh dependent compared to DES while delivering similar accuracy

Cons:

- Can be more computationally intense than previous mentioned models
- Requires fine boundary layer mesh for maximum accuracy
- Transient is best use case which requires longer runtimes

SST k-omega Detached Eddy Simulations (DES)

Two-equation, linear eddy viscosity URANS

- Simulates the formation of turbulent structures in unsteady regions
- Hybrid between URANS (transient) SST k-omega and LES (Large Eddy Simulation)
- Unsteady RANS employed in near-wall
- LES employed in the turbulent core region where large turbulent scales play a dominant role

Pros:

- Max possible accuracy for separated, high Re external aerodynamics
- More efficient than full LES

Cons:

- More computationally intensive than SST and SST w/ SAS
- Works best with smooth (more uniform) mesh distributions
- Highly sensitive to mesh sizing
- Very academic in use

3 Levels of External Aero

Attached Flow

SST k-omega RC (Hellsten)

- Mesh
  - 10-15-30 Layers
  - Factor: 1.0
  - Enhancement Blending
  - *Optional Flag*
    mesh_enhance_thick: 100-600
    mesh_boundarylayer_blend
- Solver
  - **Convergence** – Custom w/2 Orders added

Complex Turbulence Structures

SST k-omega SAS

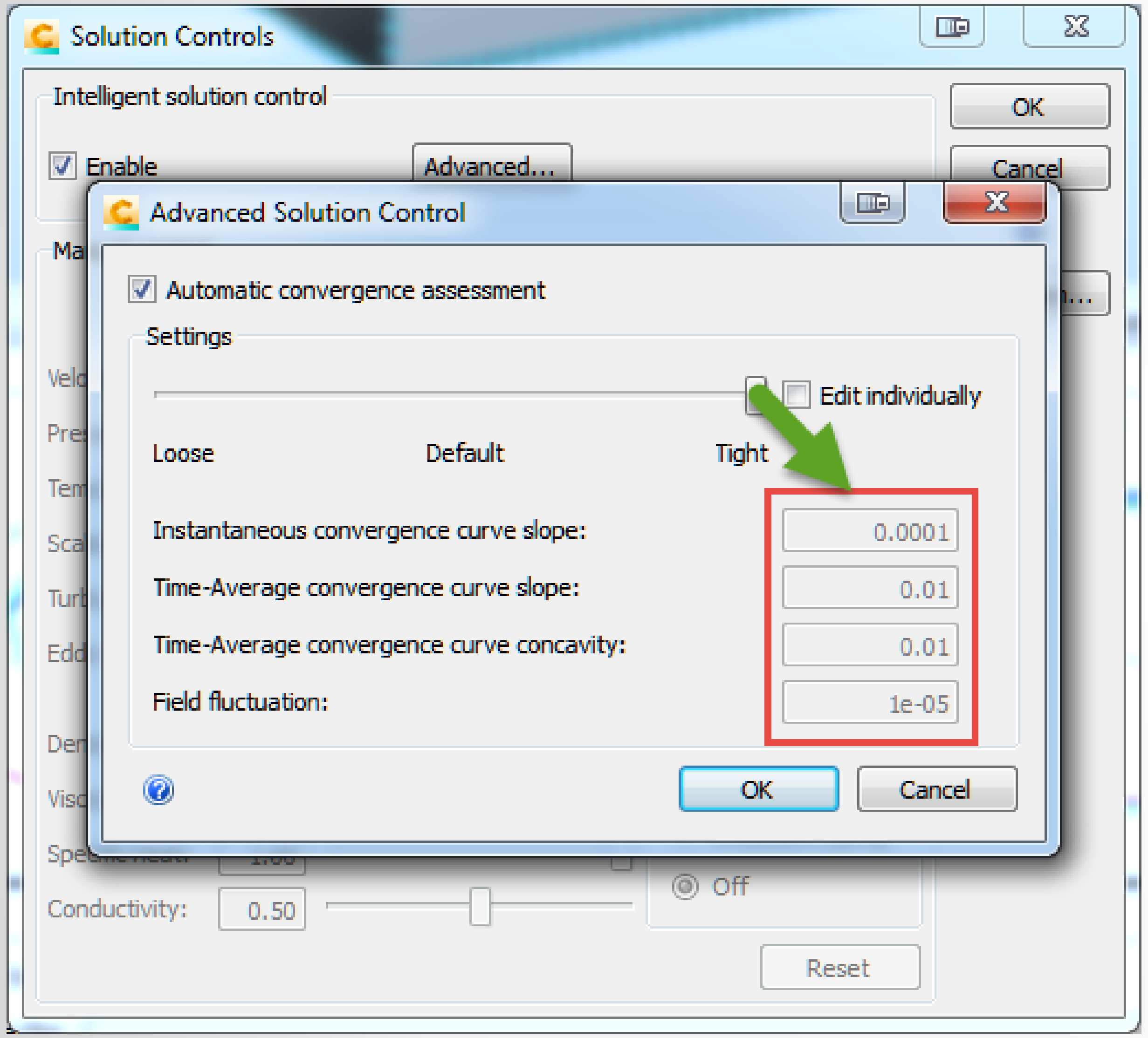
- Mesh
  - Same as **Attached Flow**
  - *Suggested Flag*:
    mesh_boundarylayer_blend
- Solver
  - **Required Flag:**
    sst_new_iwf 1
  - **Steady State Convergence** – Custom w/2 Orders added
  - **Transient** with testing smaller time steps

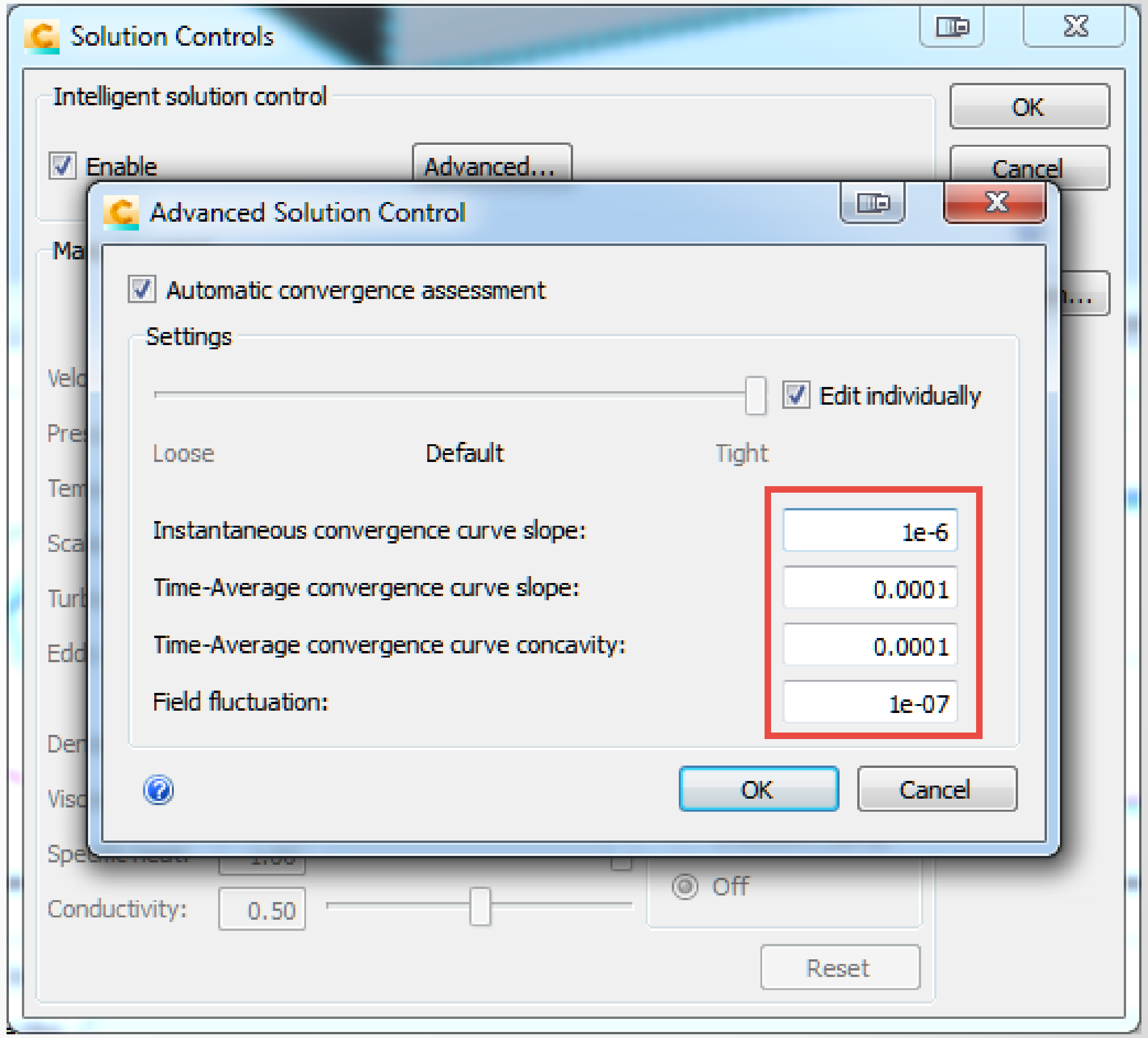
Strouhal Number

SST k-omega SAS/DES

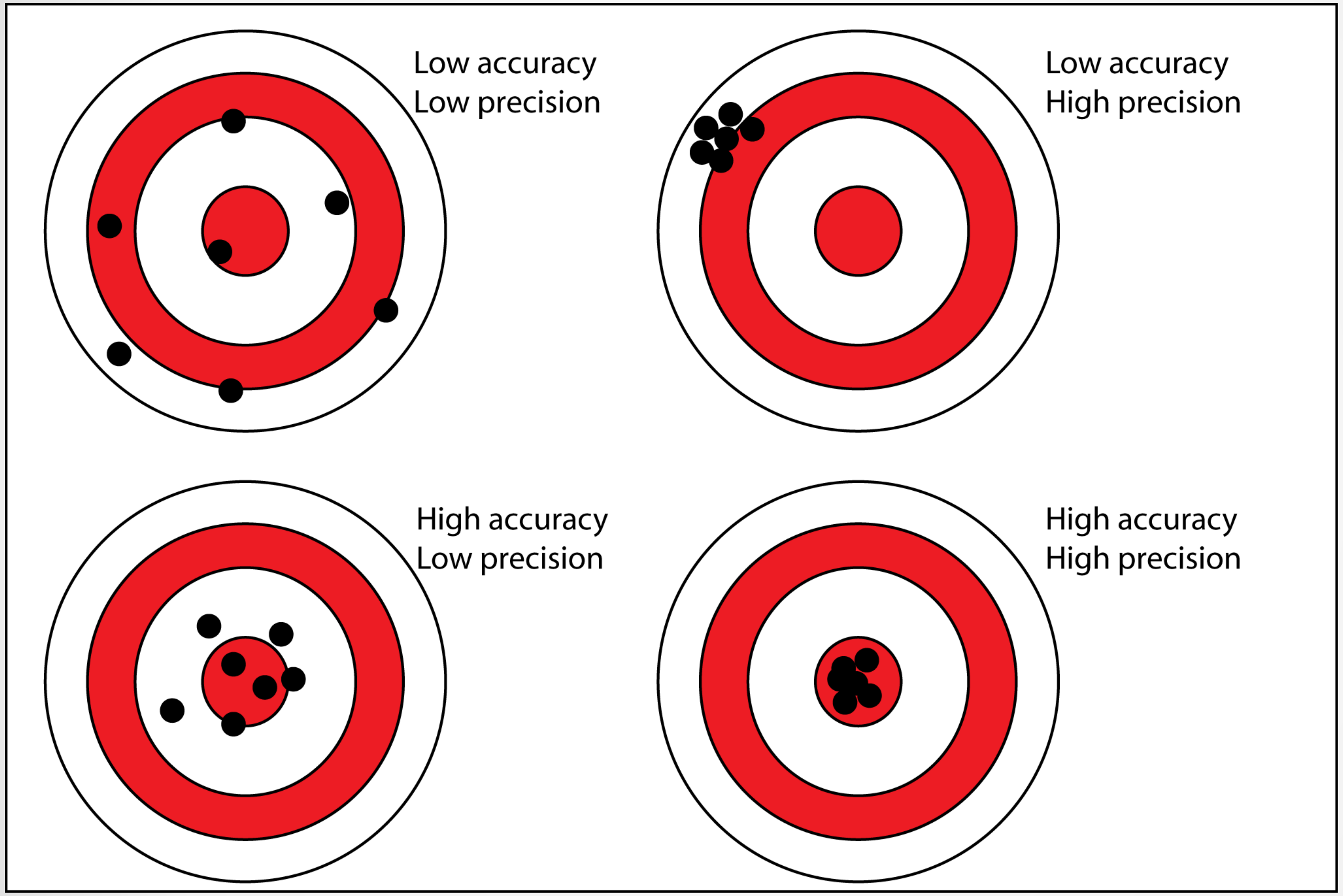
- Mesh
  - Same as Complex Turbulence Structures
  - Uniform mesh in wake zone to capture eddies
- Solver
  - **Required Flag:**
    sst_new_iwf 1
  - **Transient - Start with 100 timesteps per cycle**

What is 2 Orders of Magnitude for Convergence?



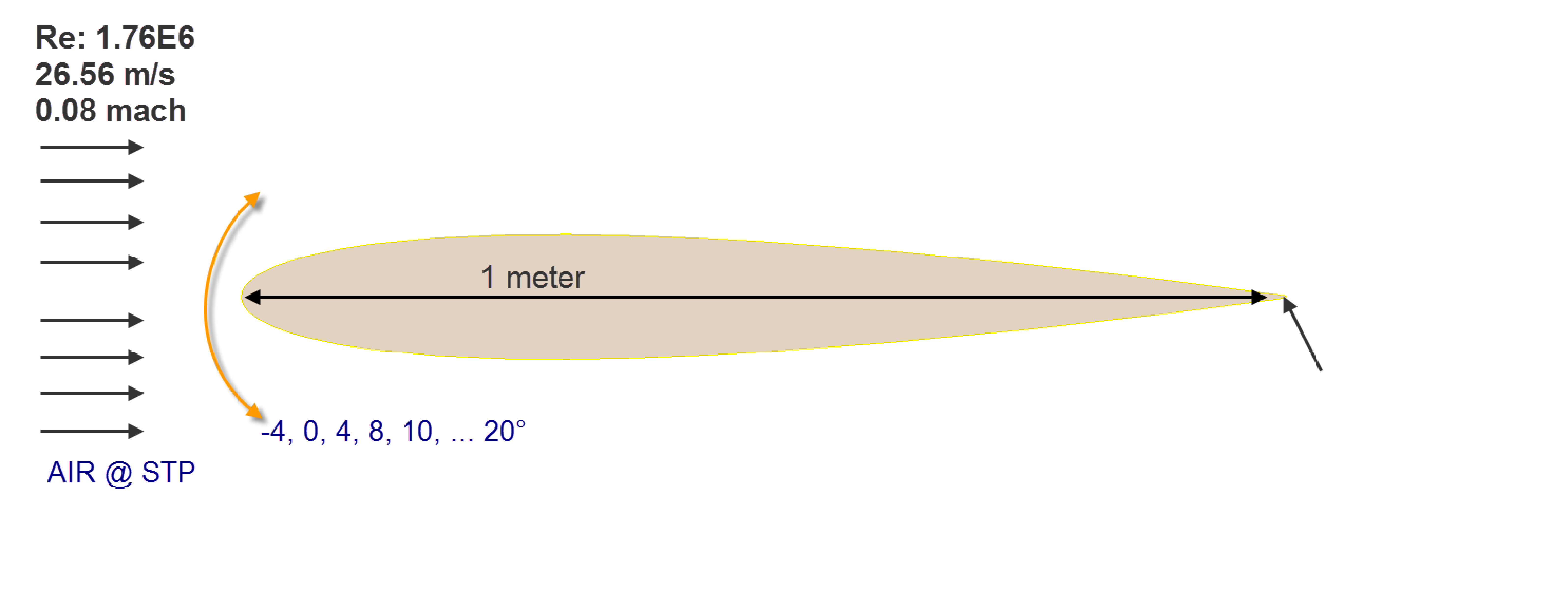


Accuracy vs. Precision for Product Design



Attached Flow Example

NACA 0012 CFD Setup Strategy

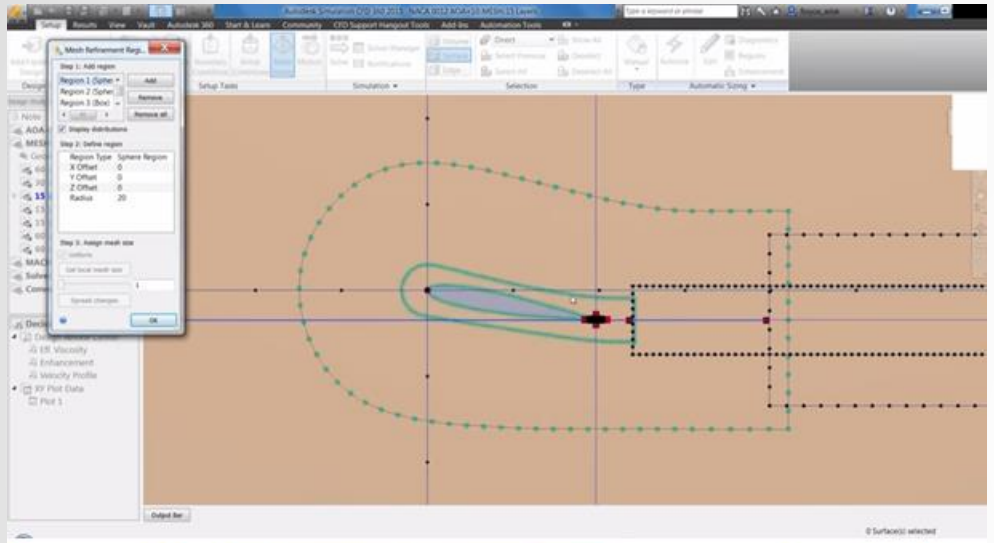


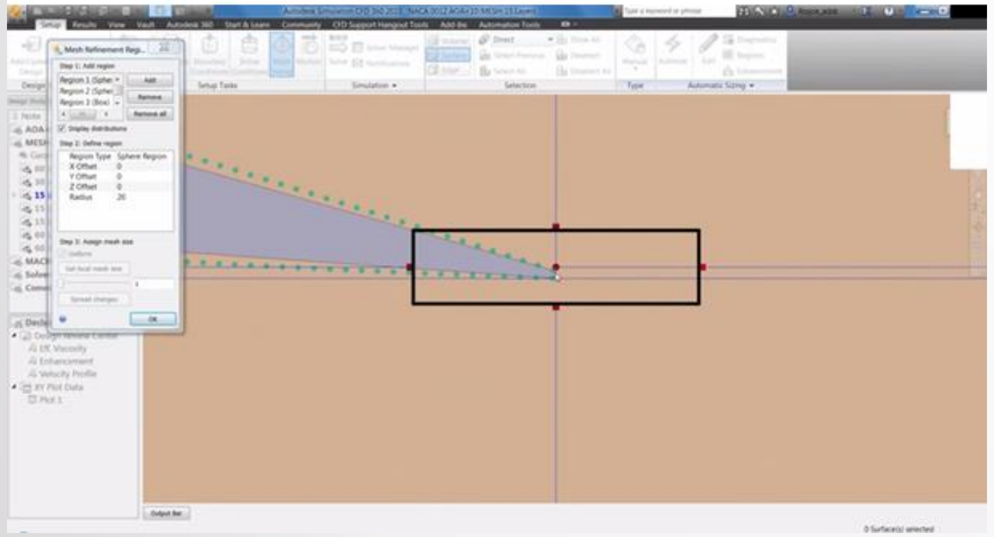
What is the NACA 0012 CFD setup strategy?

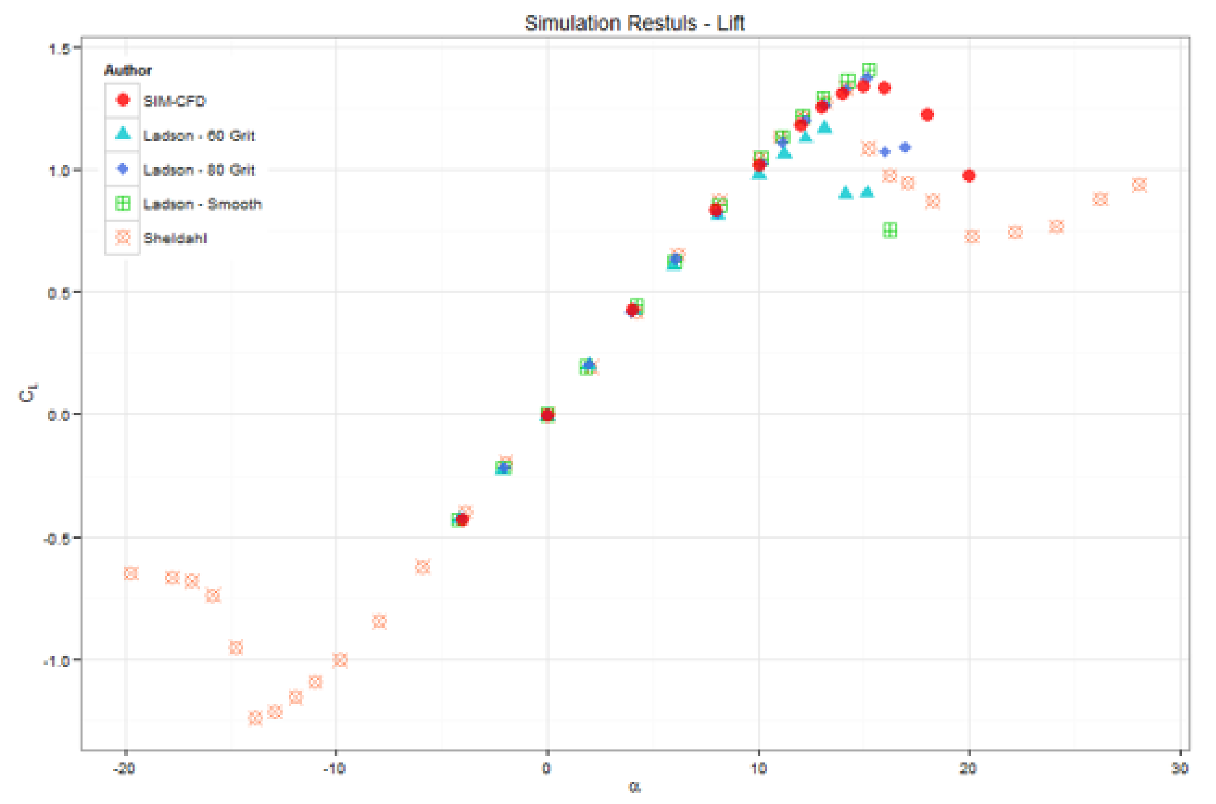
- **Advanced Turbulence**
  - Attached Flow Configuration
    SST k-omega RC (Hellsten)
- **Updated Wall Calculator**
  - resid_bdry_force_calc 1

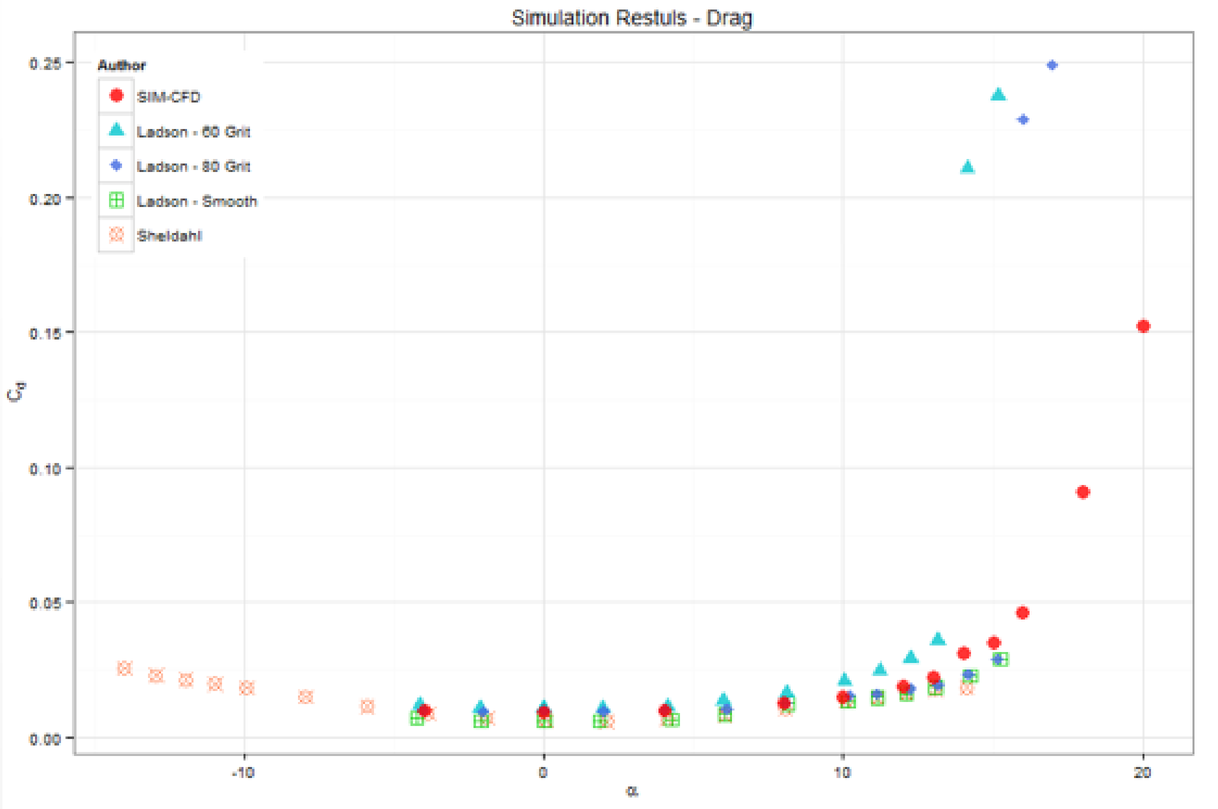
- **Meshing**
  - 30 to 60 Enhancement Layers
  - mesh_enhance_thick (%): 400
  - Layer Factor: 0.8
  - Airfoil Surface mesh: 2.5 mm
  - Airfoil Tail mesh: 0.5 mm
  - First Region: 5 mm

NACA 0012 Results

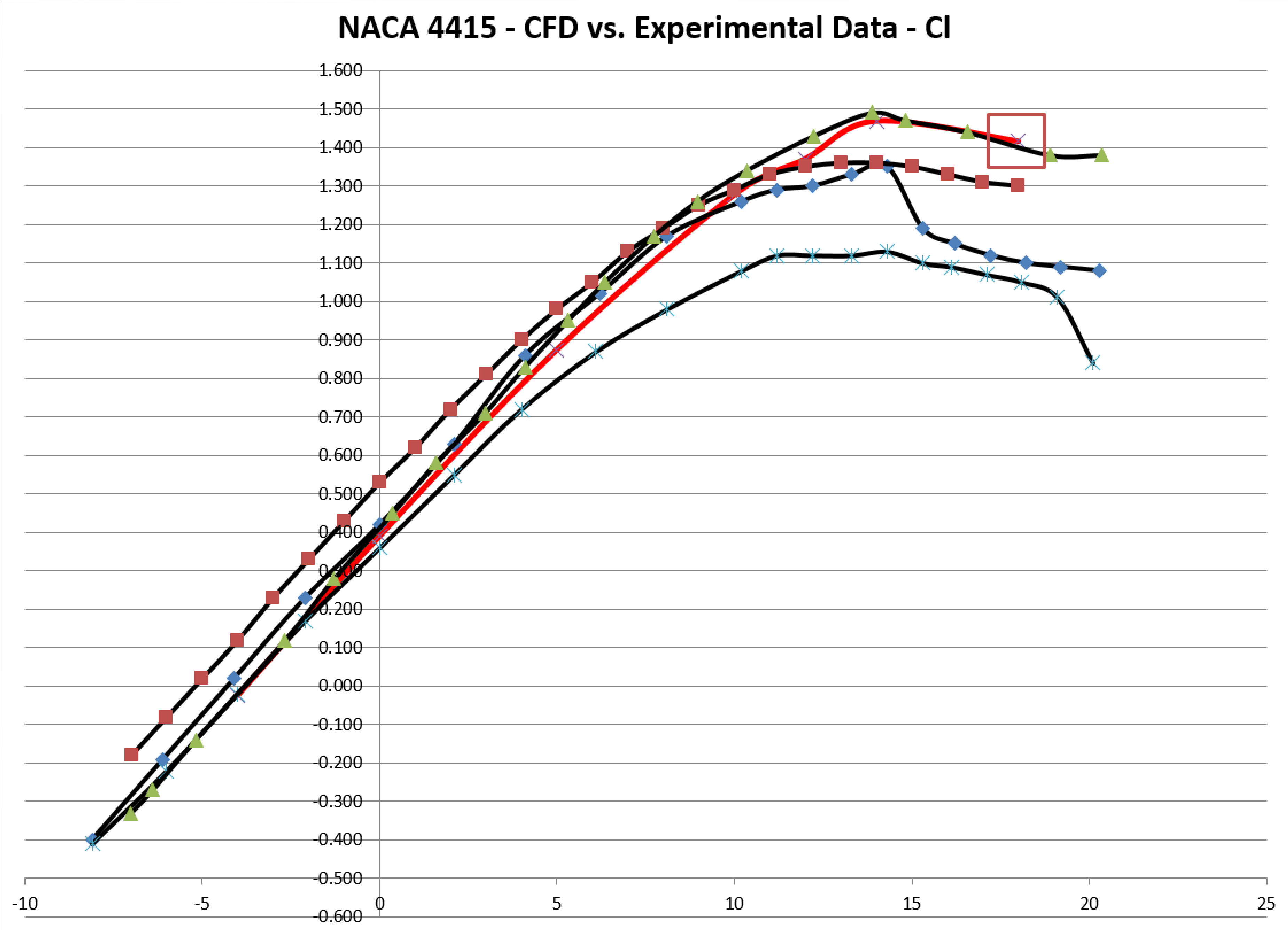


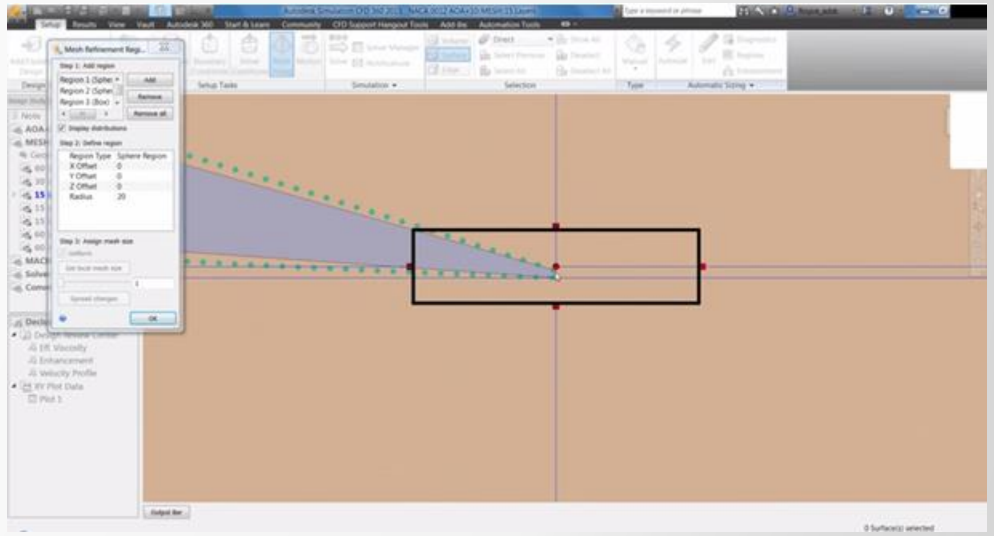






Customer Case: NACA 4415





Mesh Adaptation
Best Practice for Aerodynamics

Agenda

- Adaptation Workflow
- Aerodynamic Specific Adaptation Configuration
- Pitfalls

- Solar Panel Example

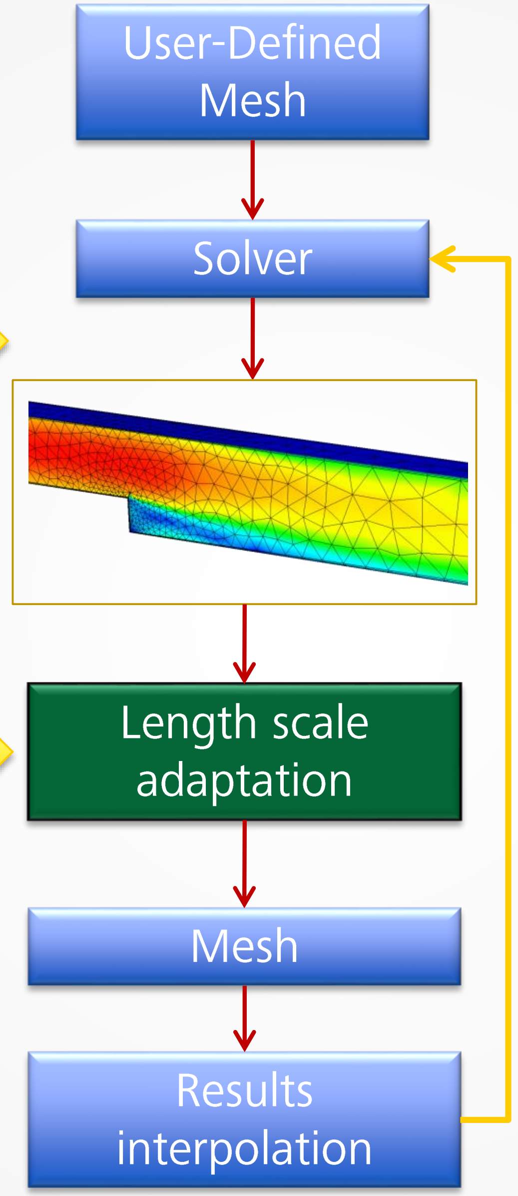
Mesh Adaptation Workflow

Generates *mesh independence indicators*:

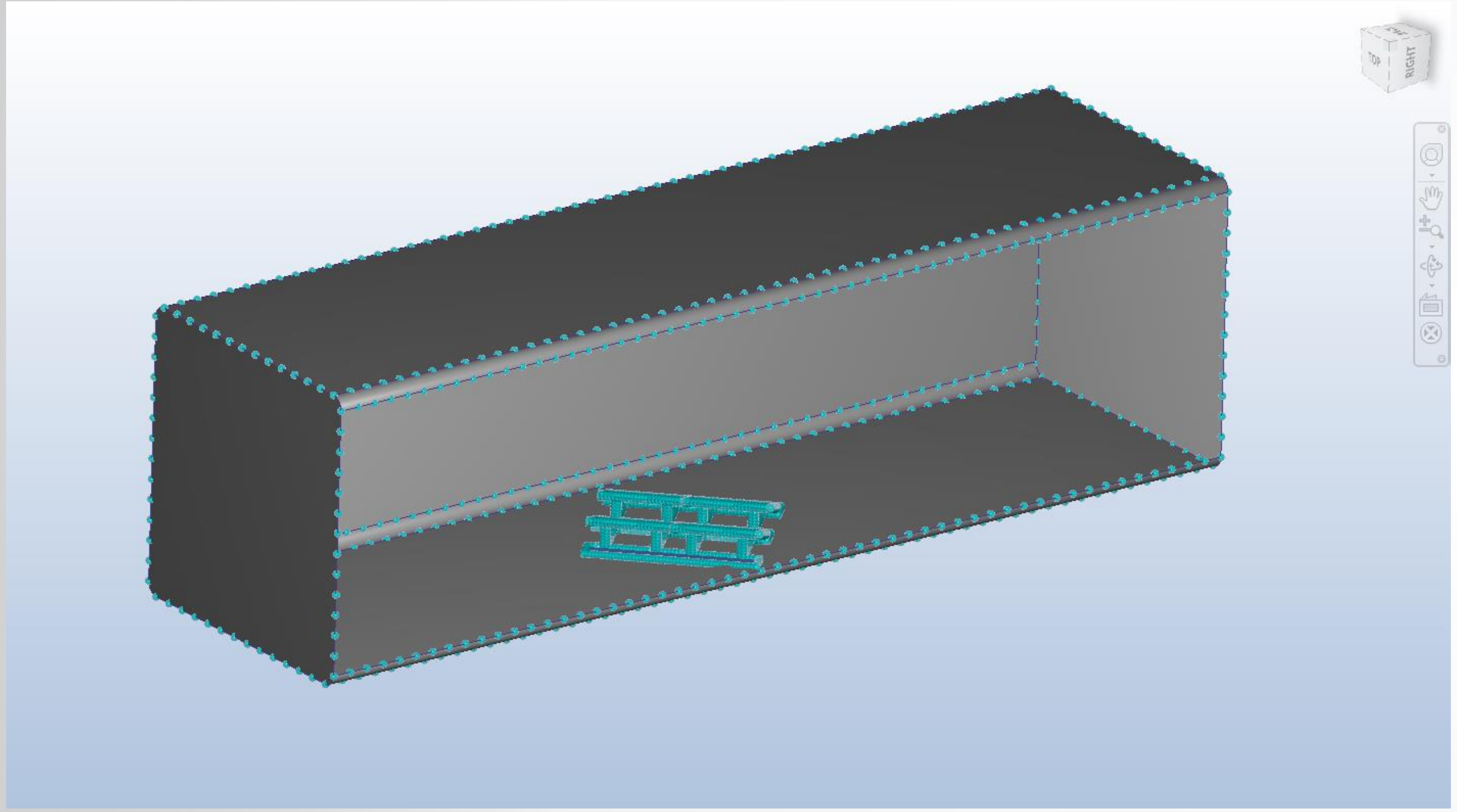
- ✓ These are *relative* rather than *absolute* indicators.
- ✓ Require that solutions be fully converged

Adaptation via regeneration:

- ✓ Less efficient
- ✓ More robust
- ✓ *If a mesh can be generated, it can be adapted*

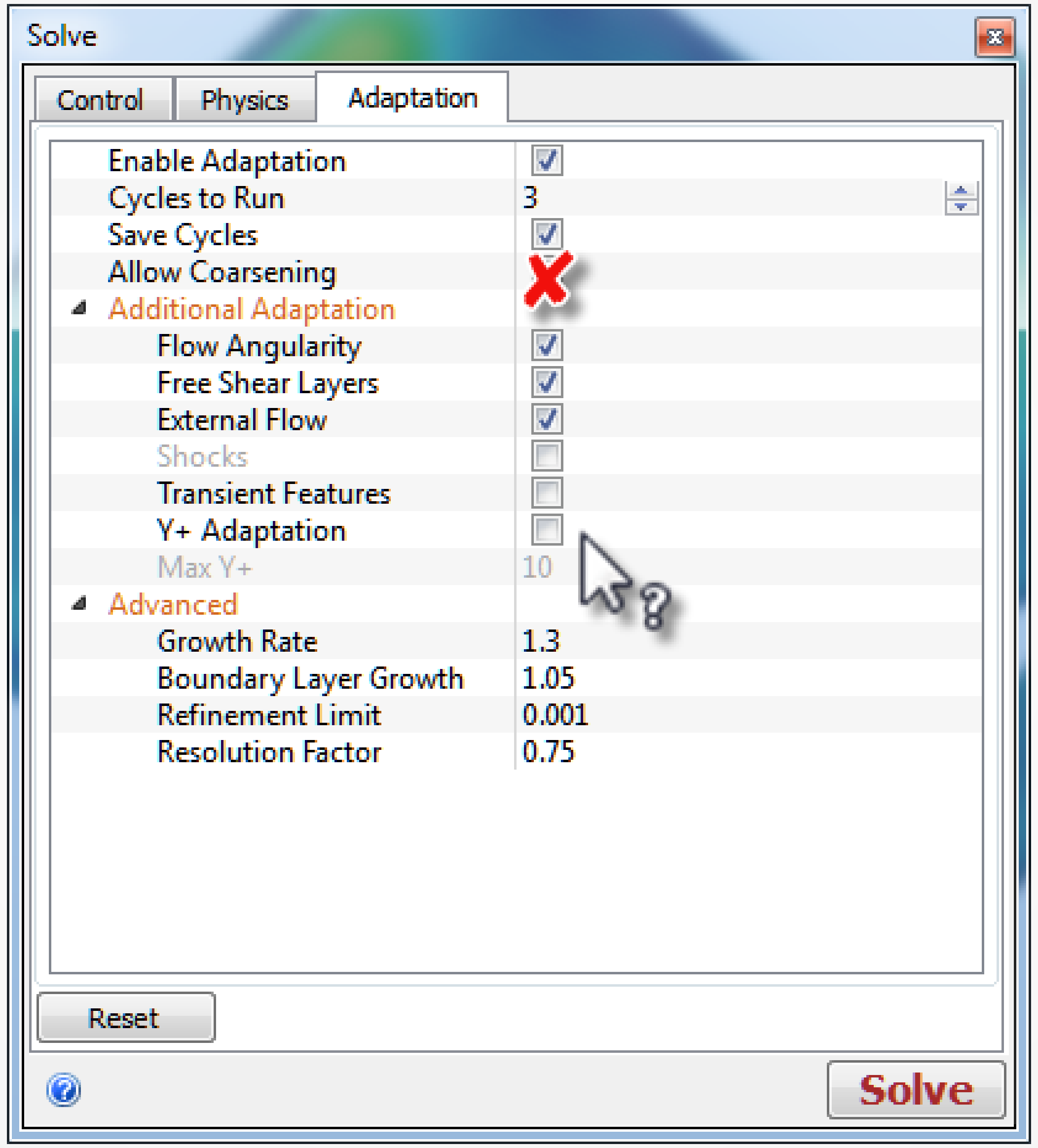


User-Defined Mesh



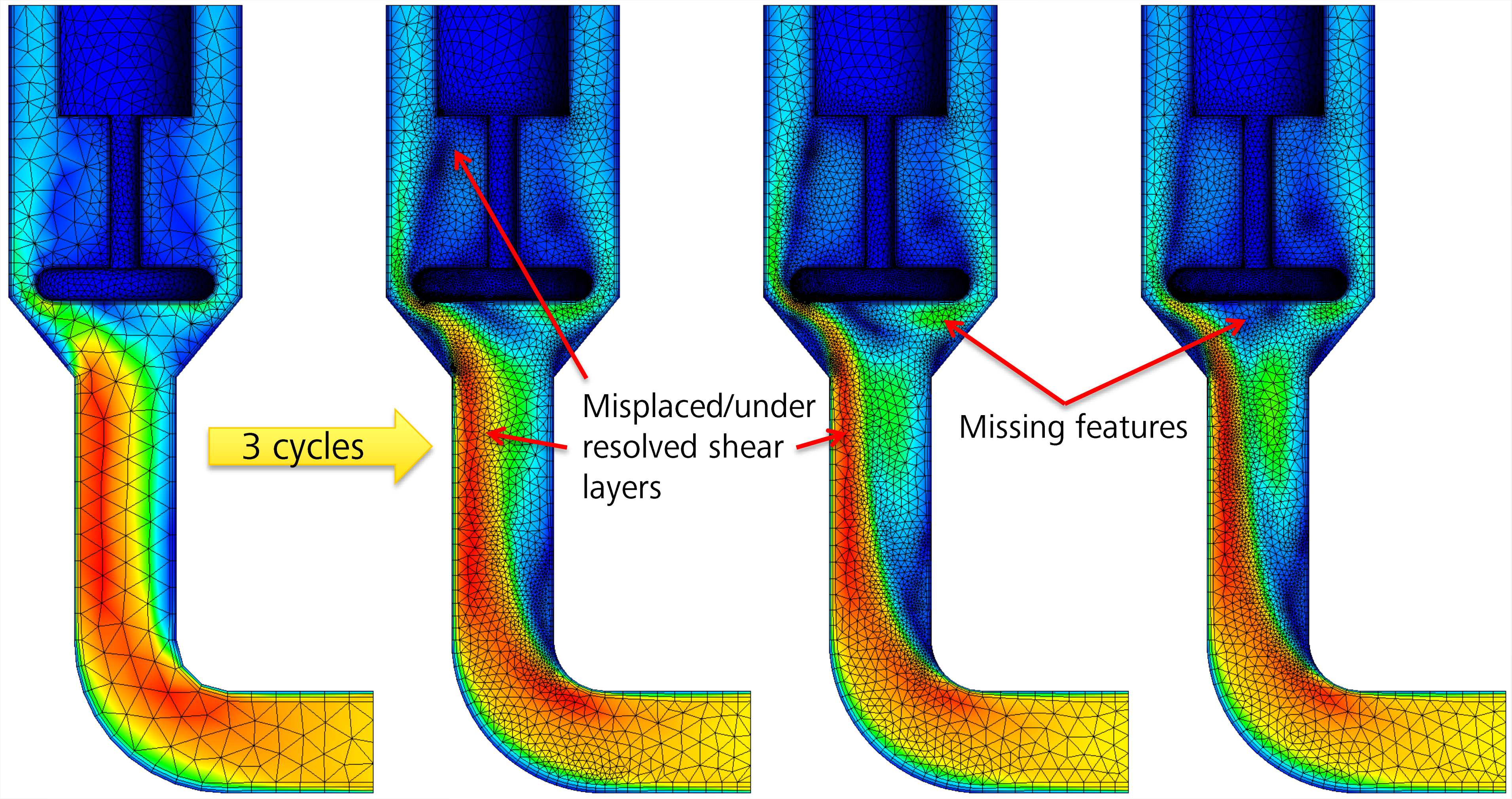
- **Manual Mesh**
- **Suppress Solids**
- Large size on external domain for a fast first cycle
- Tighter surface mesh compensate for air domain size
- Manual Mesh 4:1 rule?

Mesh Adaptation Setting Panel



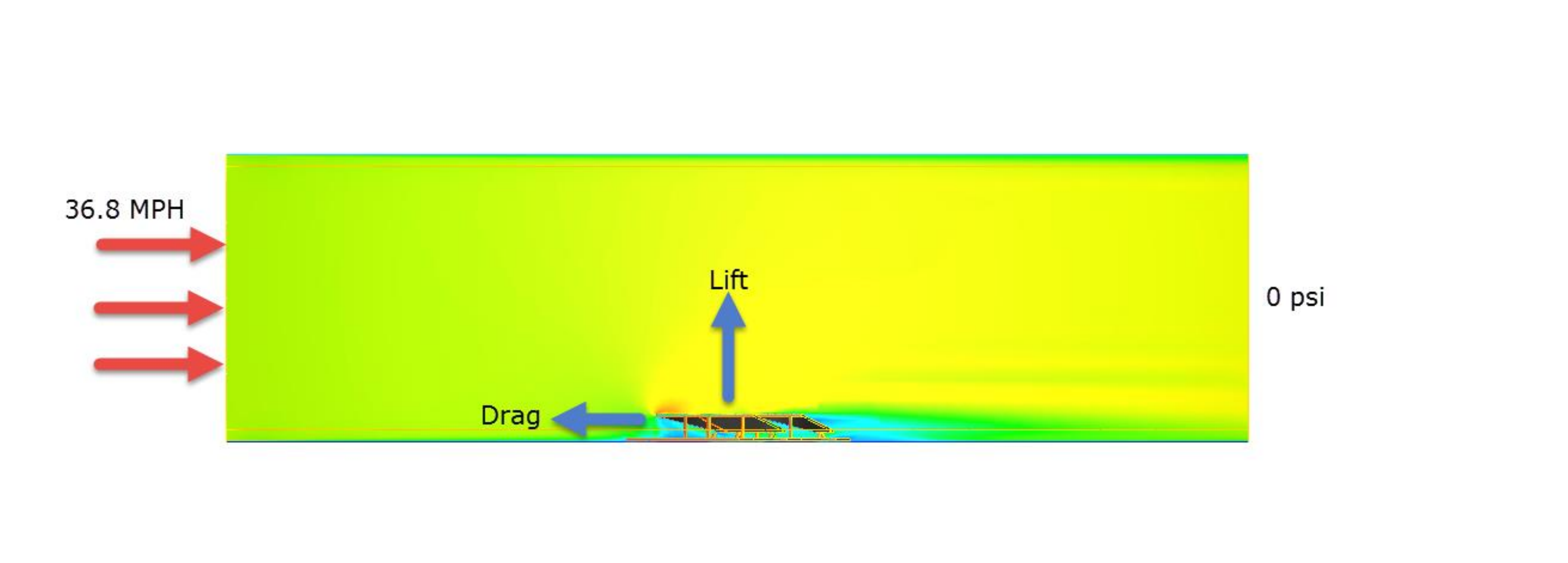
The Importance of Solution Convergence

Leverage 2nd Order Manual Automatic Convergence Assessment



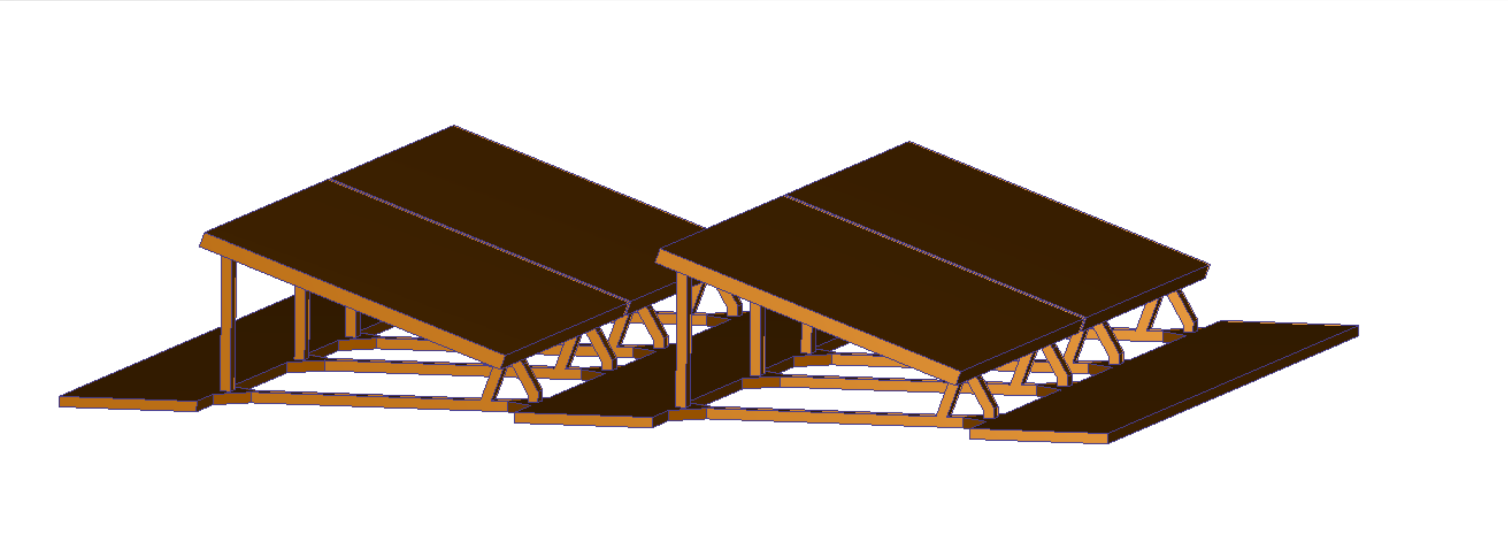
Initial solution | **25** solver iterations | **50** solver iterations | **250** solver iterations

Example: Flow around 2x2 Solar Panel Array

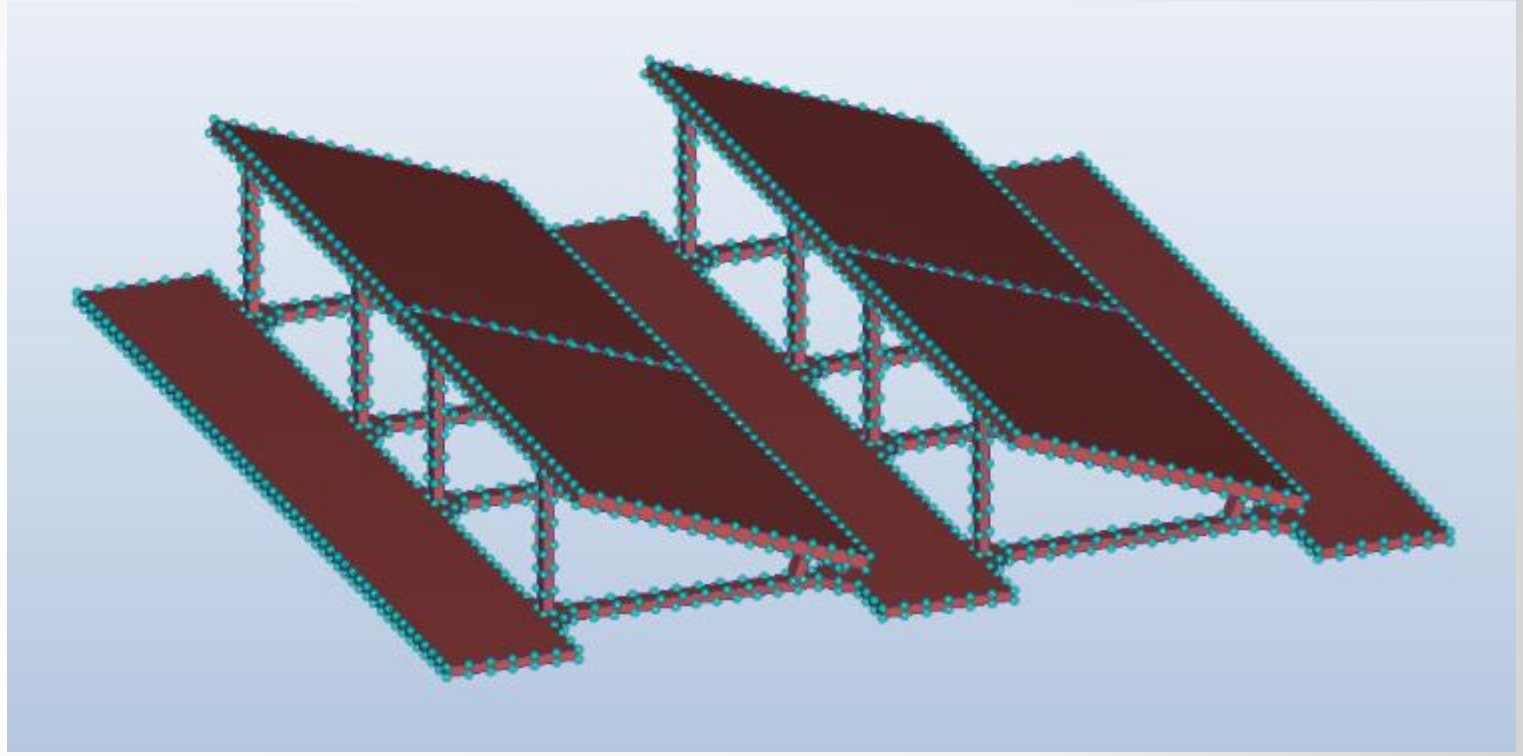


Lift: 0.9 lbs　　**Drag**: 0.3 lbs

Solar Panel Array

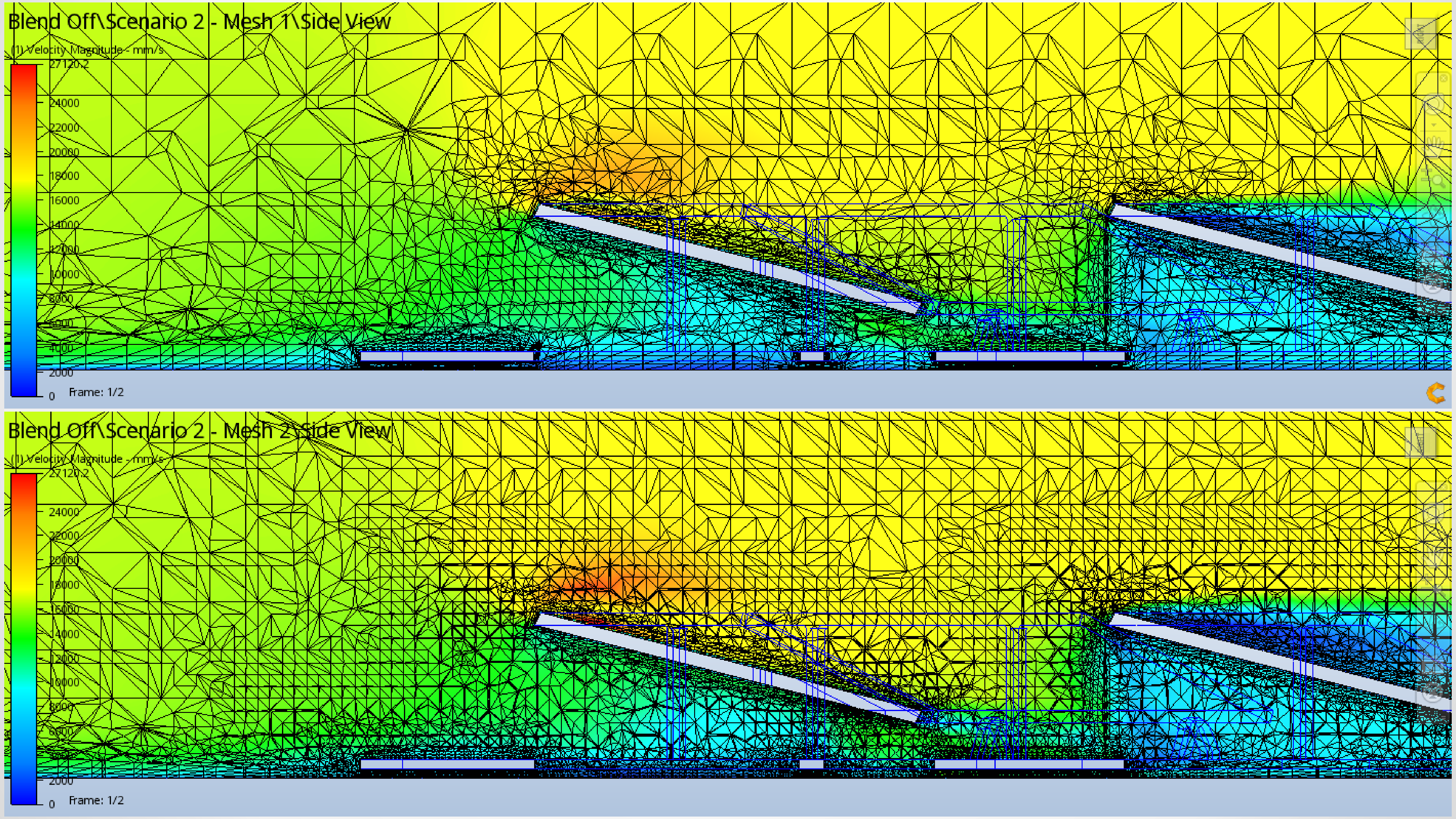


Size: 10" x 10"
Panels: 20°

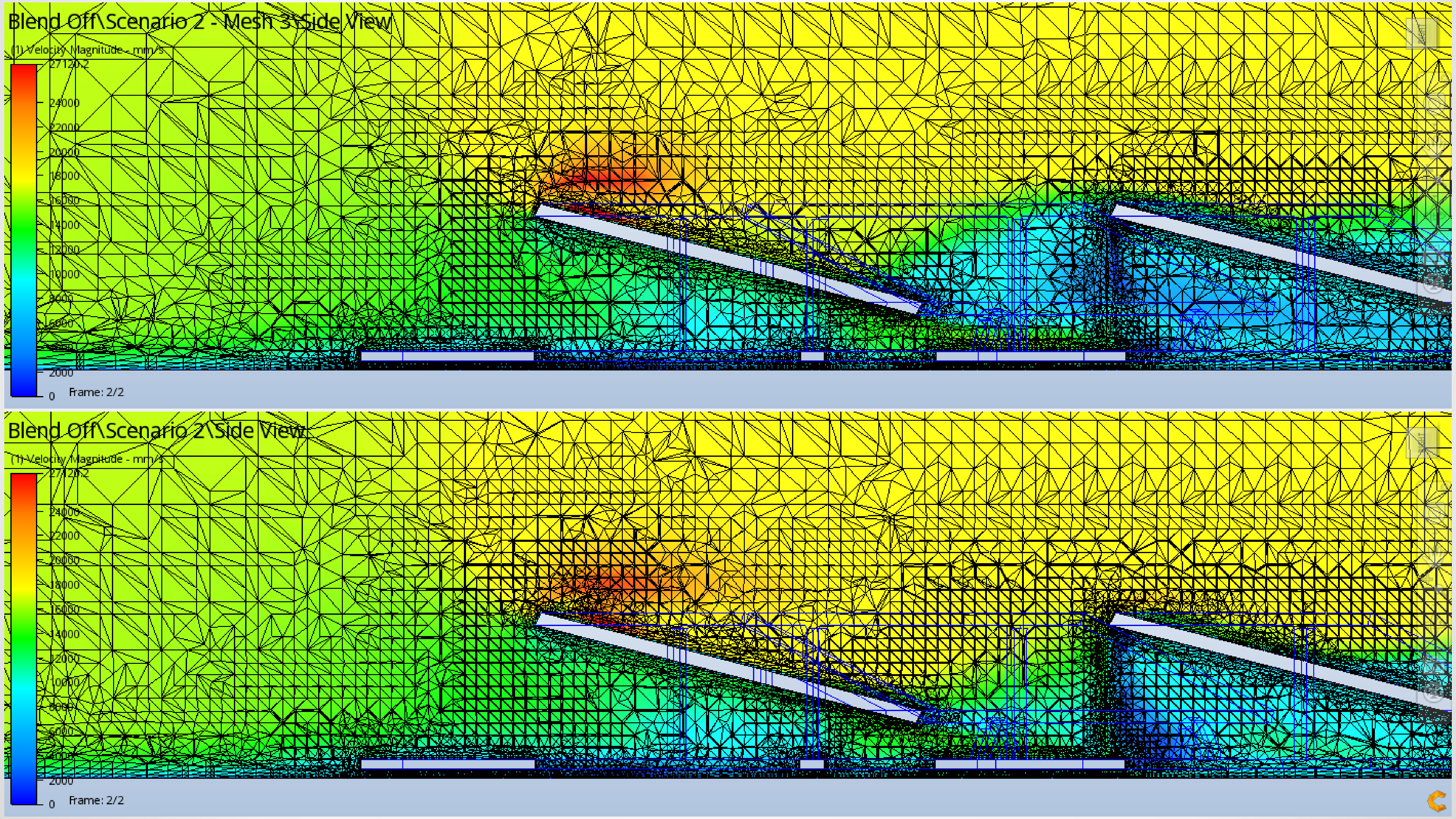


Surface: 5 mm
Air Volume: 30 mm
Attached Flow Strategy w/3 Layers
No Y+ Adaptation

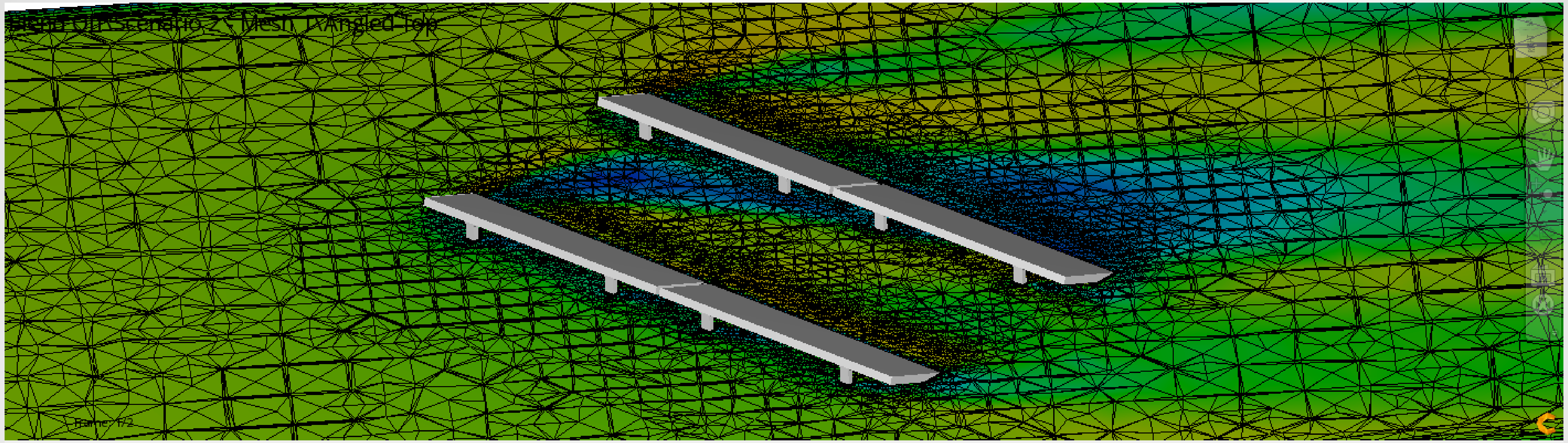
Side View of Mesh Adaptation (1 & 2)

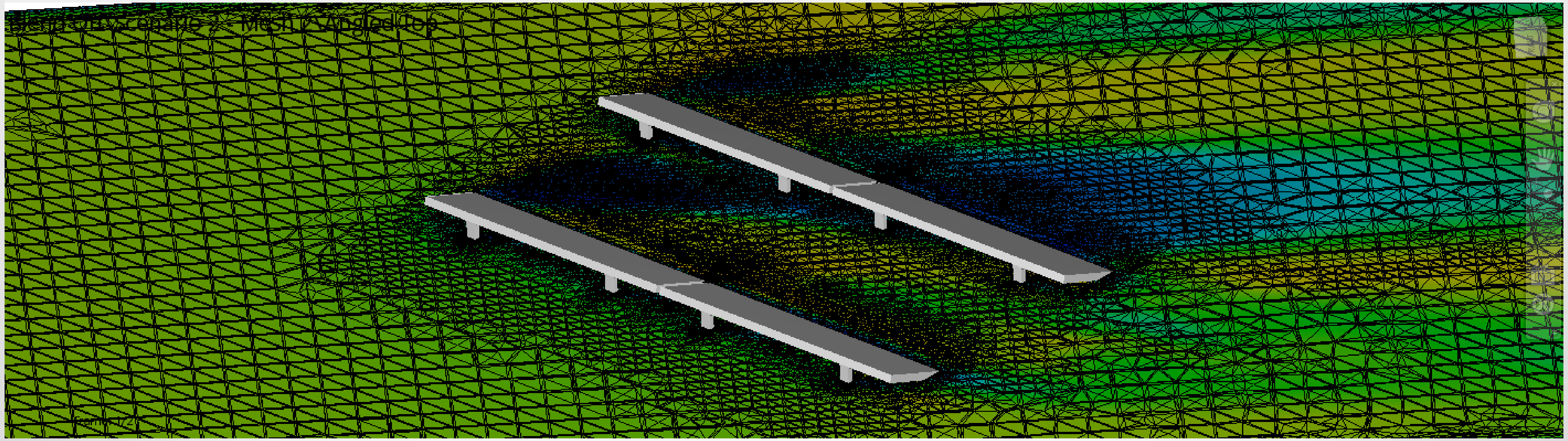


Side View of Mesh Adaptation (3 & 4)

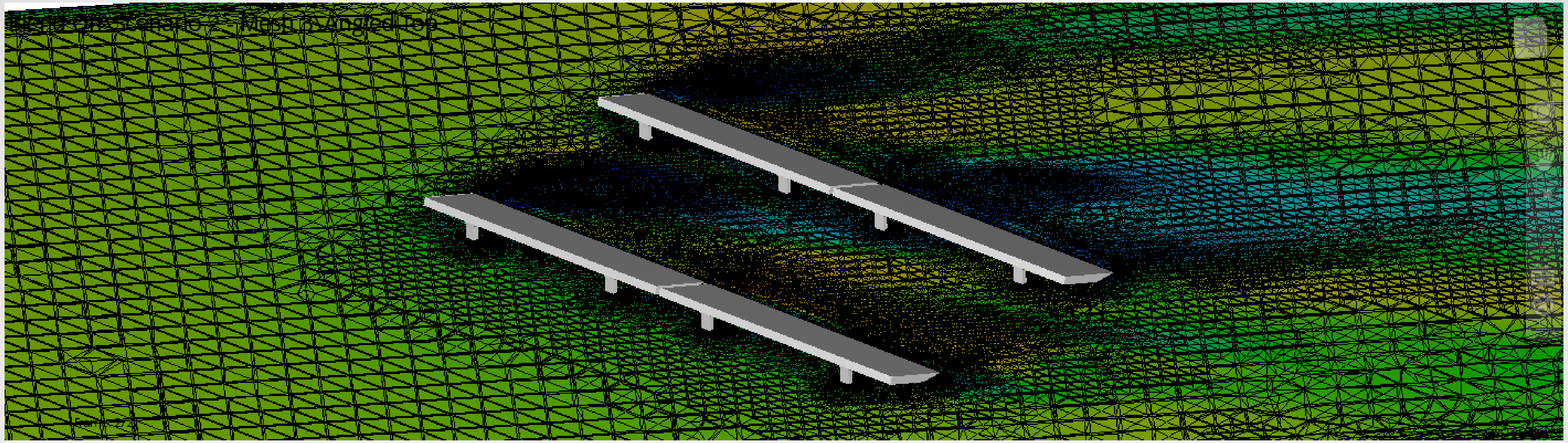


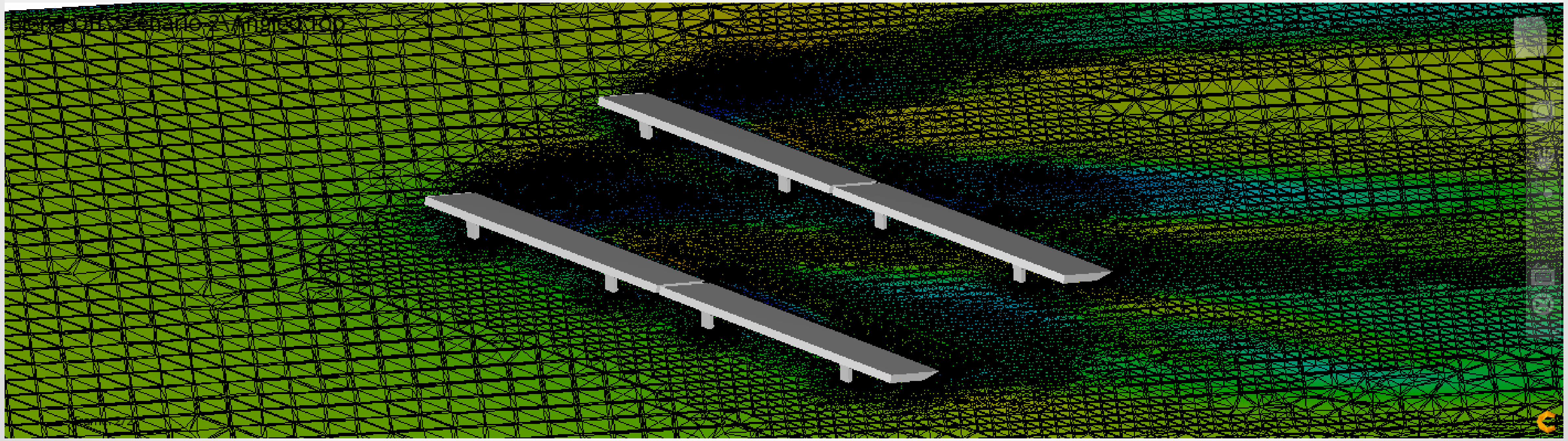
Top View of Mesh Adaptation (1 & 2)



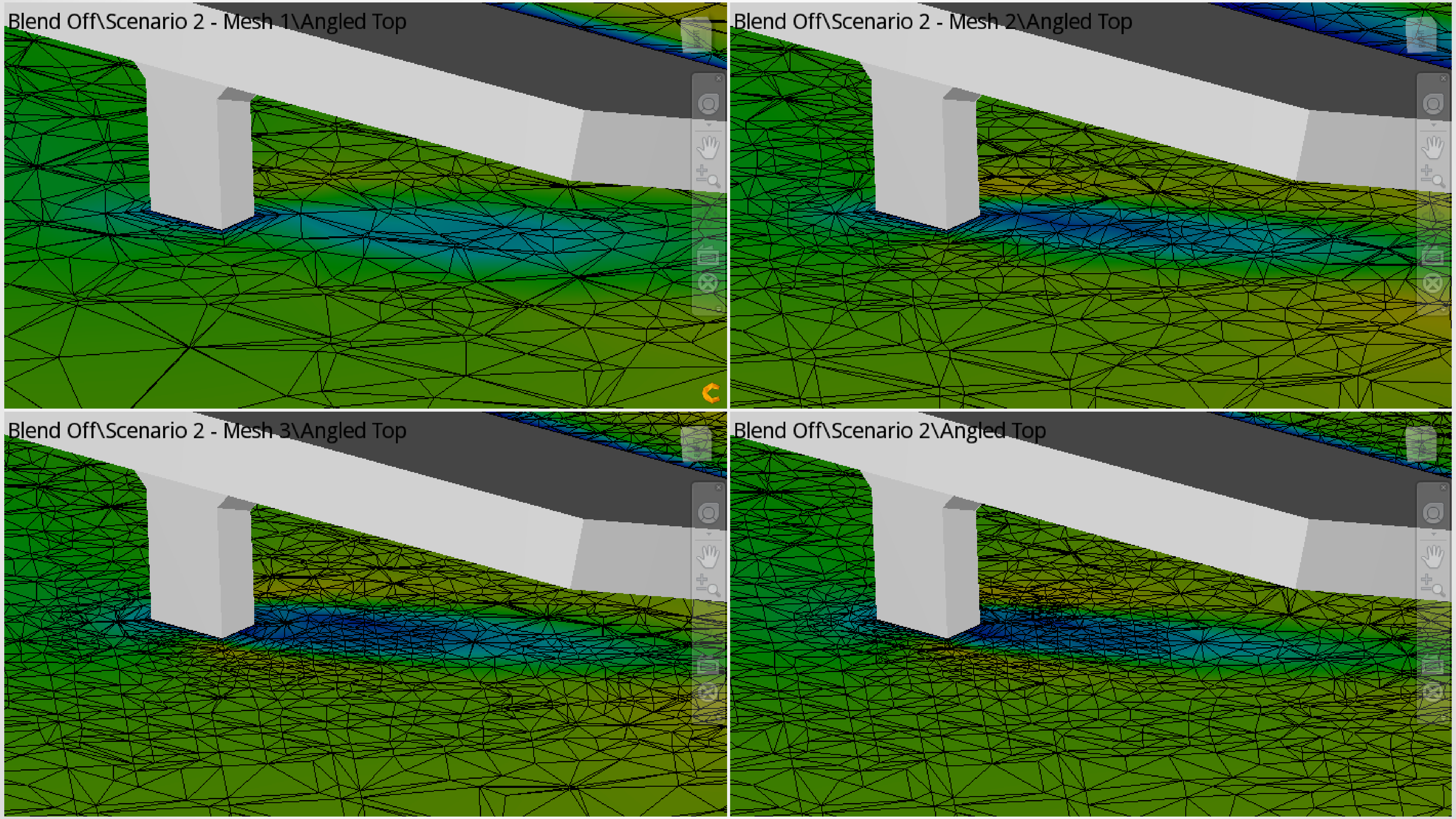


Top View of Mesh Adaptation (3 & 4)

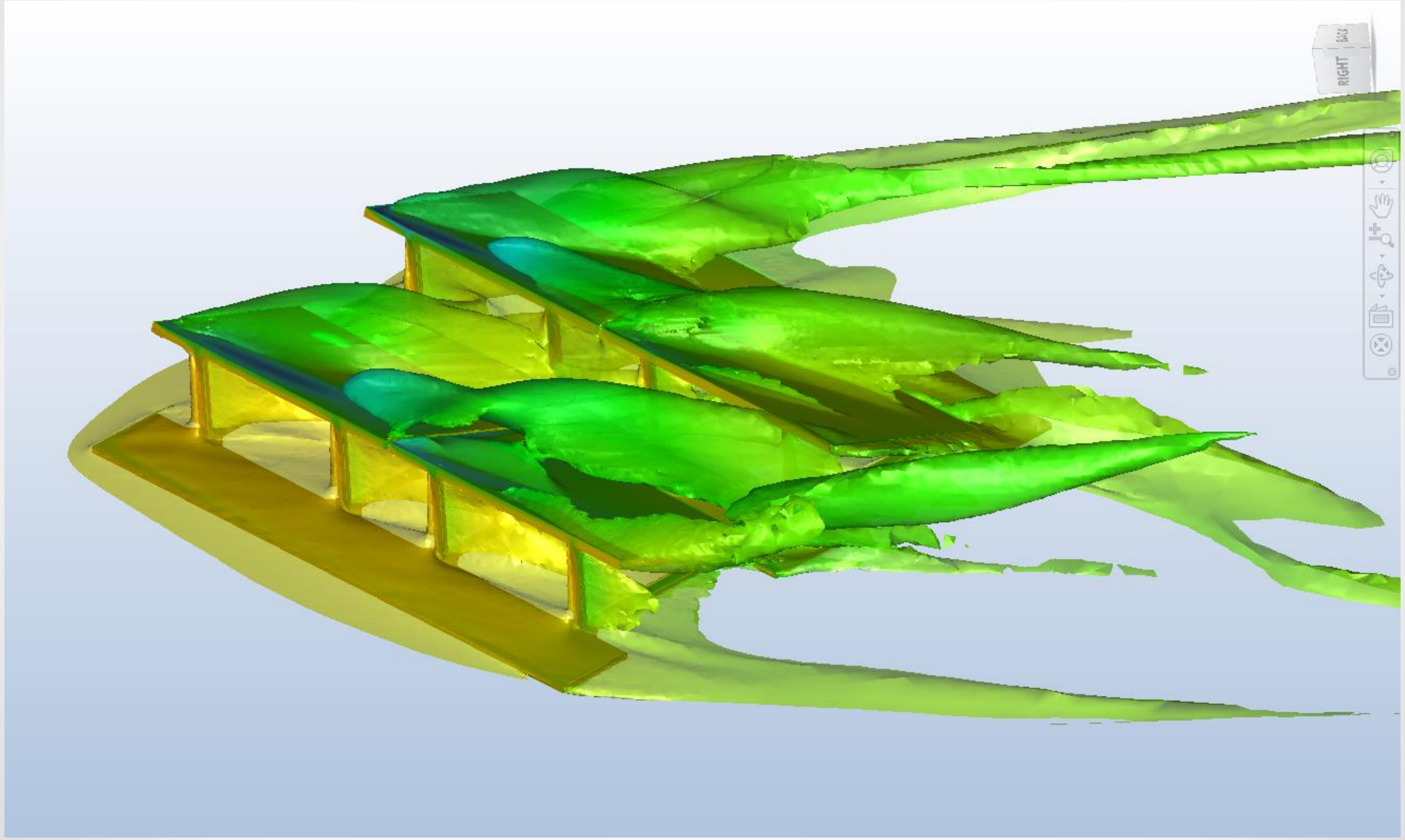




Flow Around Leading Leg



Attached Flow: 20 mph Iso-Surface



Complex Turbulence Structures

- Continued Final Mesh Adaptation for Transient
- Transient: 1e-5 s
- Intelligent Solution Control Enabled

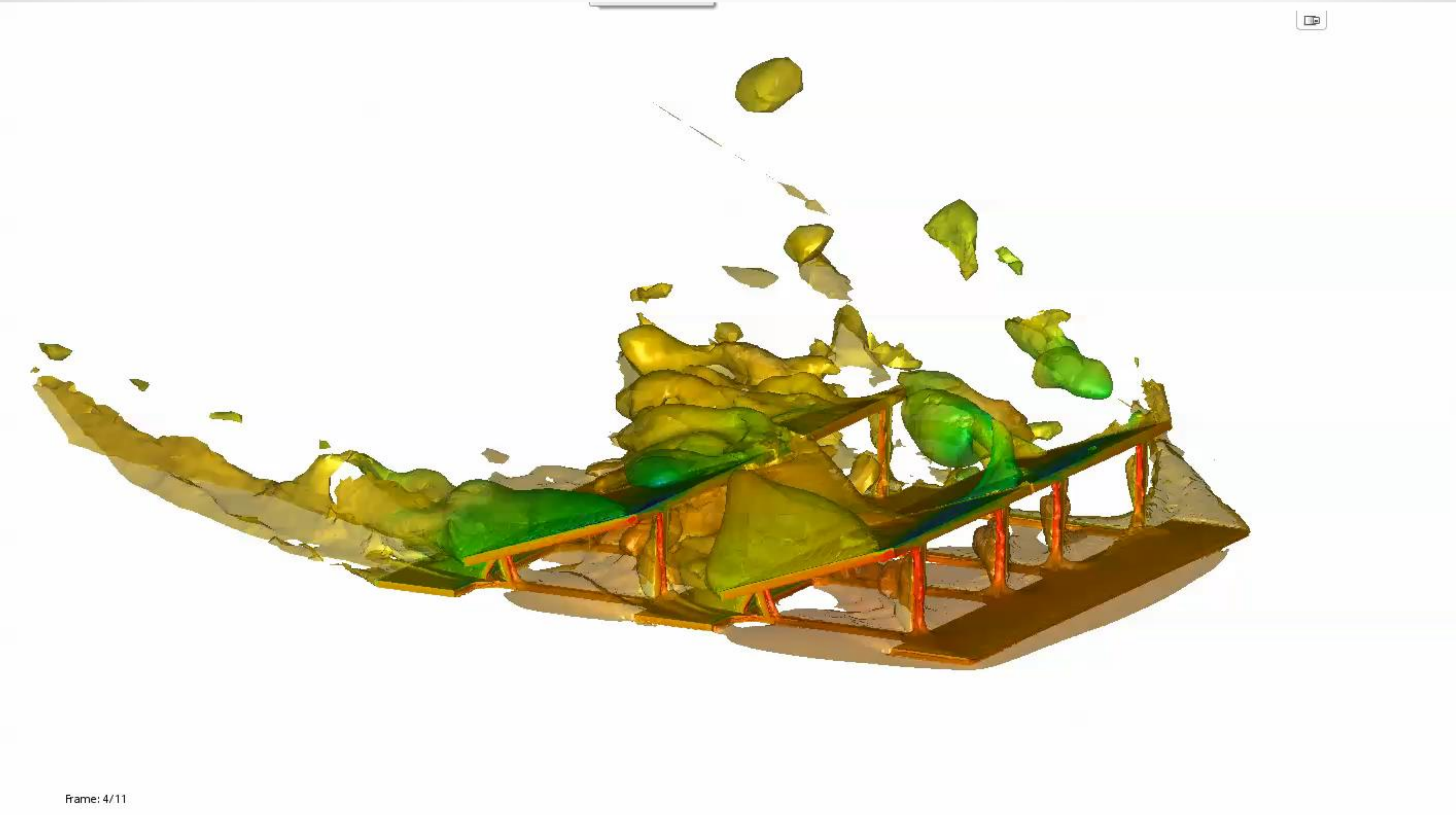
SST k-omega RC (Hellsten)

- Mesh
  - 10-15-30 Layers
  - Factor: 1.0
  - Enhancement Blending
  - *Optional Flag*
    mesh_enhance_thick: 100-600
    mesh_boundarylayer_blend
- Solver
  - **Convergence** – Custom w/2 Orders added

SST k-omega SAS

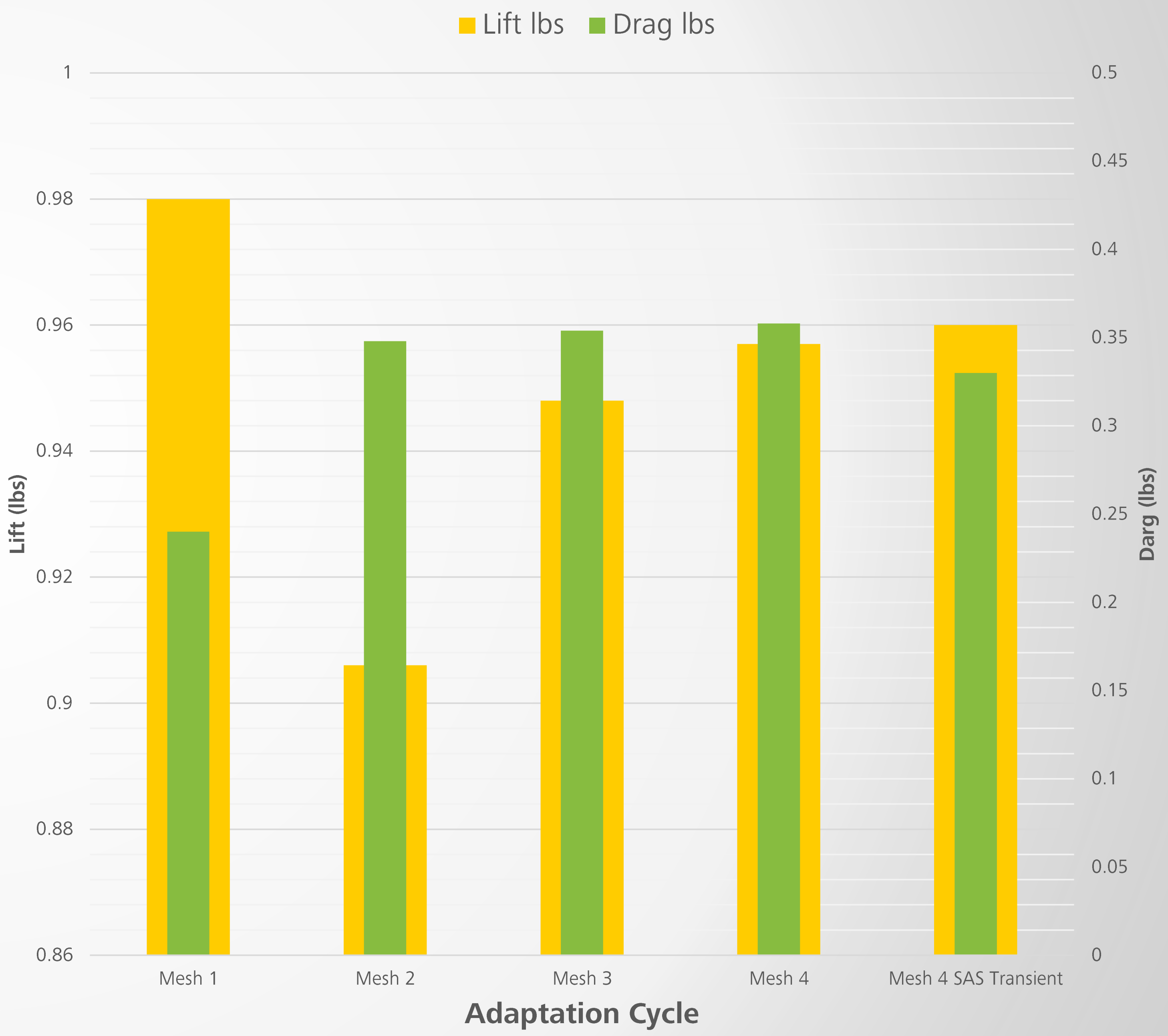
- Mesh
  - Same as **Attached Flow**
  - *Suggested Flag*:
    mesh_boundarylayer_blend
- Solver
  - **Required Flag:**
    sst_new_iwf 1
  - **Steady State Convergence** – Custom w/2 Orders added
  - **Transient** with testing smaller time steps

Complex Turbulence Structures
20 mph Iso-Surface



Final Outcome

	Lift lbs	Drag lbs	Iterations	Error Lift	Error Drag
Mesh 1	0.98	0.24	2431	9%	-20%
Mesh 2	0.906	0.348	5000	1%	16%
Mesh 3	0.948	0.354	4105	5%	18%
Mesh 4	0.957	0.358	2525	6%	19%
Mesh 4 SAS Transient	0.96	0.33	12295	7%	10%



Extract accurate steady state or transient wall force results

Agenda

- Review Best Practices
- Newer Wall Calculator algorithm

- Wall History Application
  - Flow over a Cylinder to capture Strouhal Number
  - Get the Installer & Code

Best Practices for Accuracy

1. Leverage SimStudio to clean the geometry

2. Select the correct meshing/turbulence scheme
   - *Attached Flow*
   - *Complex Turbulence Structures*
   - *Strouhal Number* / Model Wake Structures

3. Leverage Mesh Adaptation if Needed

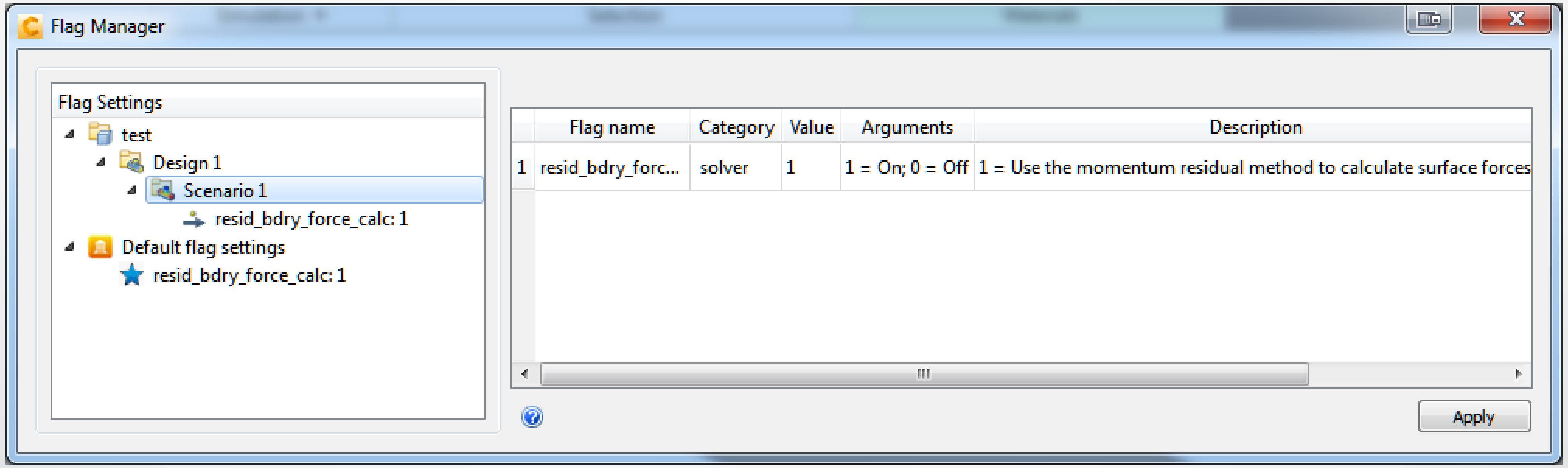
Newer Wall Calculator algorithm

resid_bdry_force_calc A

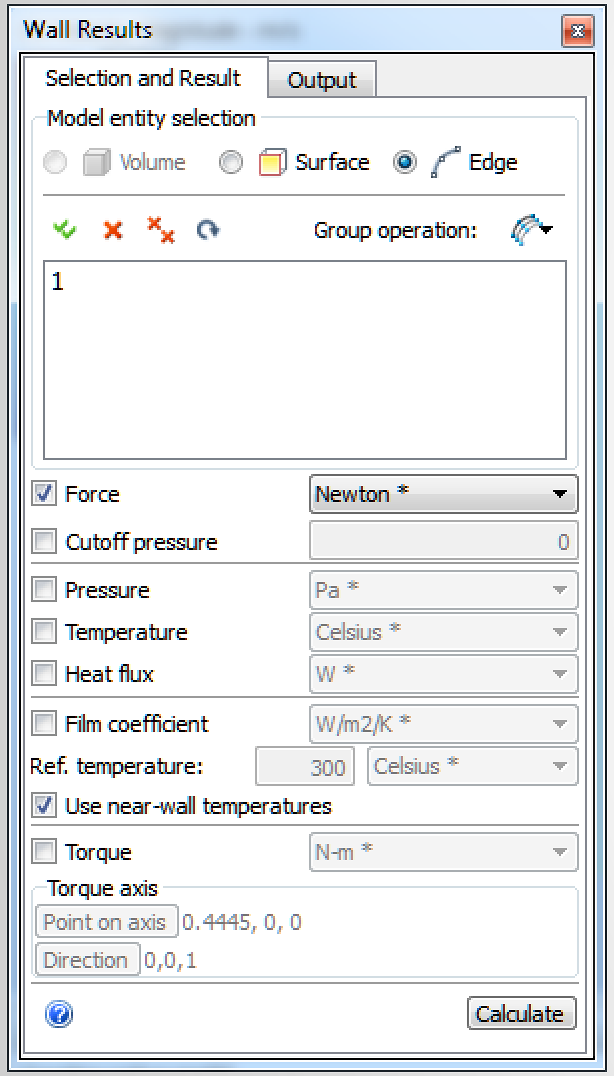
- A = 1 (On): Causes the fluid forces at the walls to be computed in a consistent manner and includes the shear forces during run-time and results visualization.
- A = 0 (Off): Disables the flag. (Default)

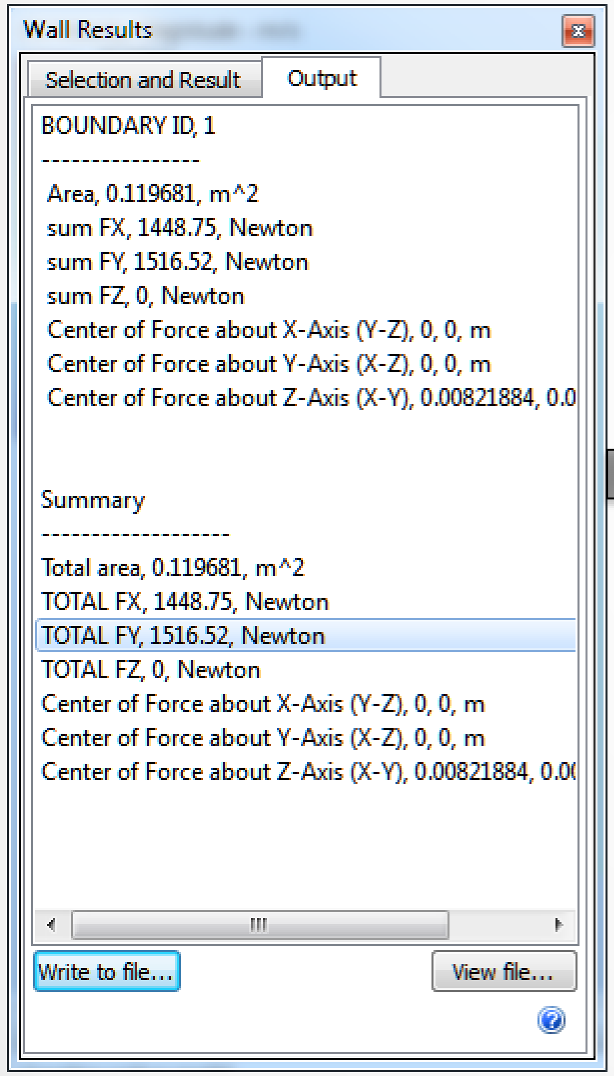
Description

- The resid_bdry_force_calc flag computes the wall forces and properly distributes it from the nodes to the logical CAD surfaces to report in the wall calculator. Another benefit is that the same calculation results are available during run-time. Without it, the run time wall calculator force results only involve the pressure and not shear.

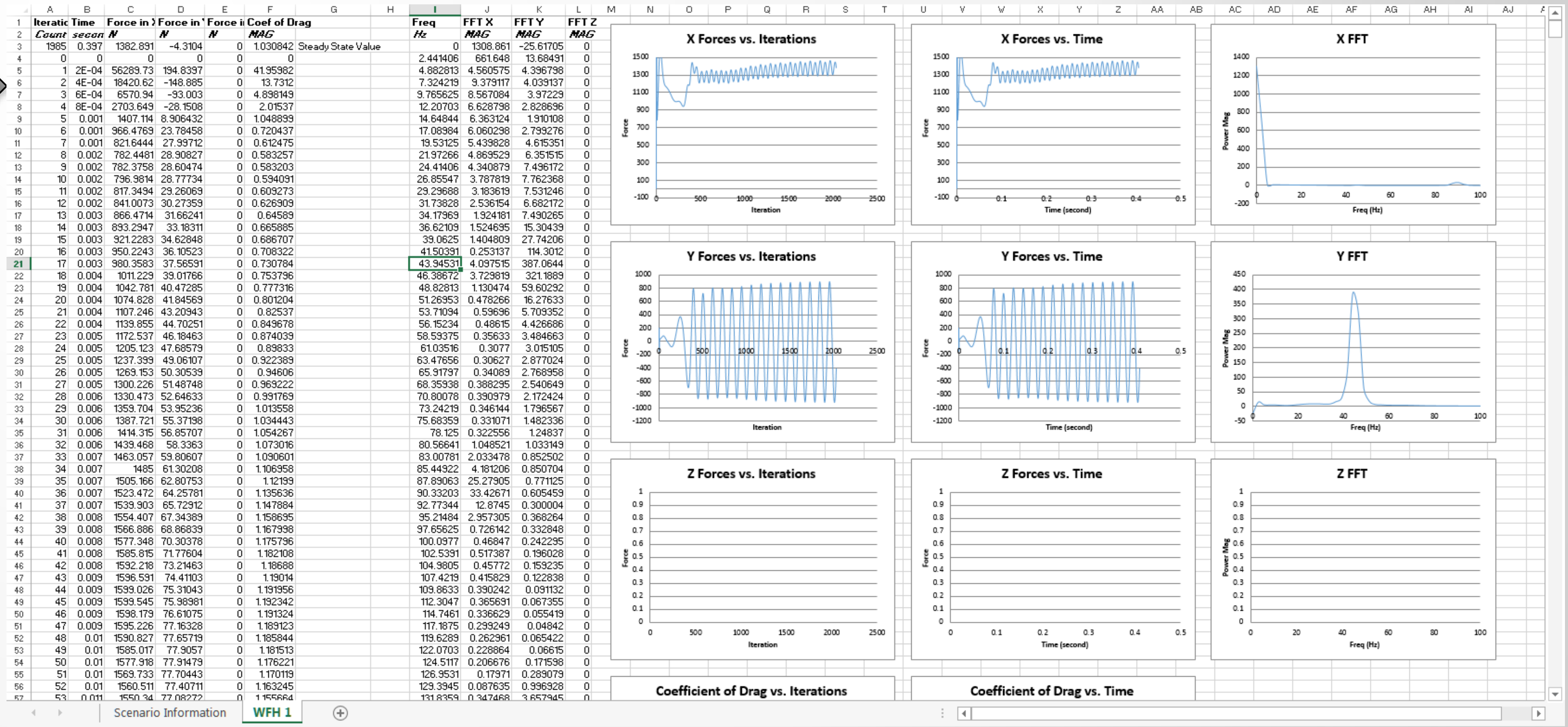


Wall Force History Tool



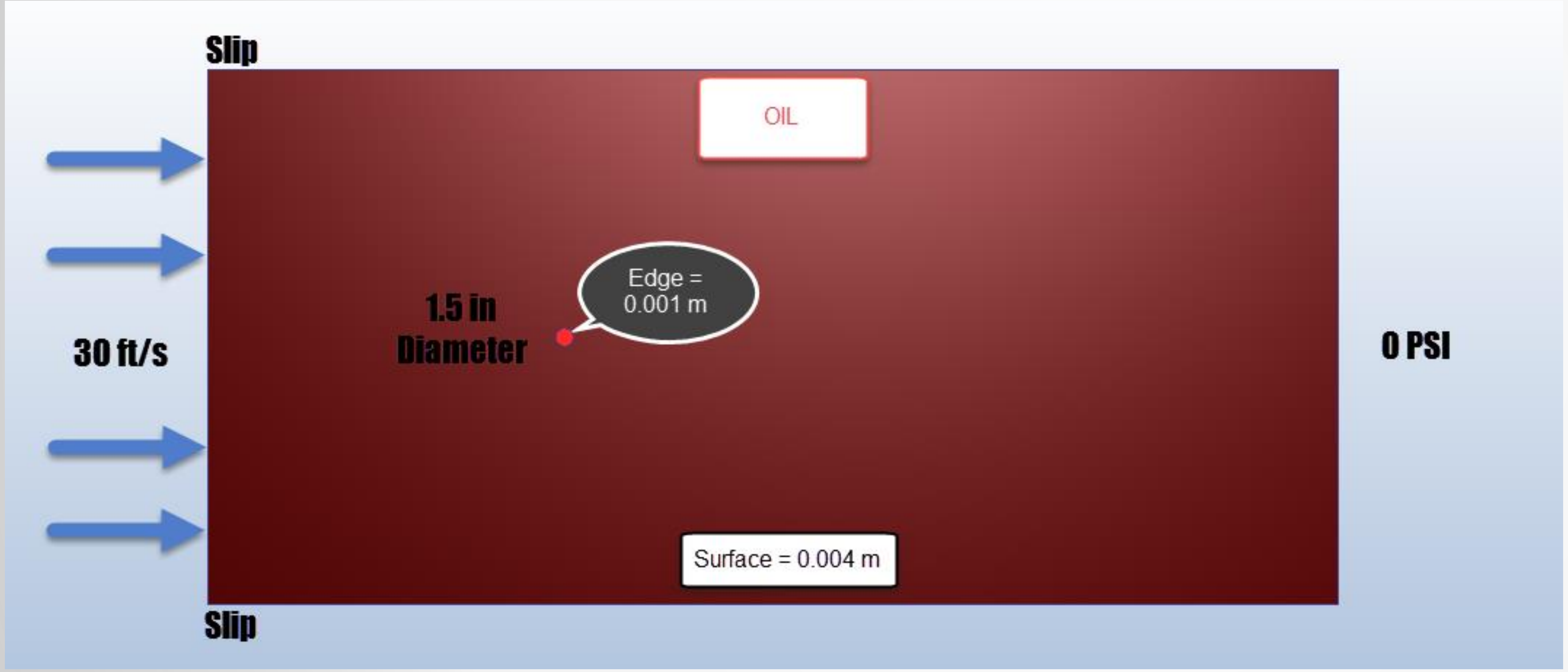


- Forces extracted to excel for every iteration
- Coefficient of Drag/Lift
- Frequency Response (FFT)
- Driven off Surface Groups
- Automatically generated plots of all data
- Steady State and Transient
- Extracts necessary material/simulation values



Strouhal Number Example

Flow over a Cylinder



SST k-omega SAS

- Mesh
  - 30 Layers
  - Factor: 1.0
  - *Y+ max = 0.2*
- Solver
  - sst_new_iwf 1
  - **Transient - Start with 100 timesteps per cycle**
    - 0.0002 s
  - ISC = Off

Flow over a Cylinder – Expected Results

St = 0.19
Frequency = 45.94 Hz

Drag = 1619.55 N

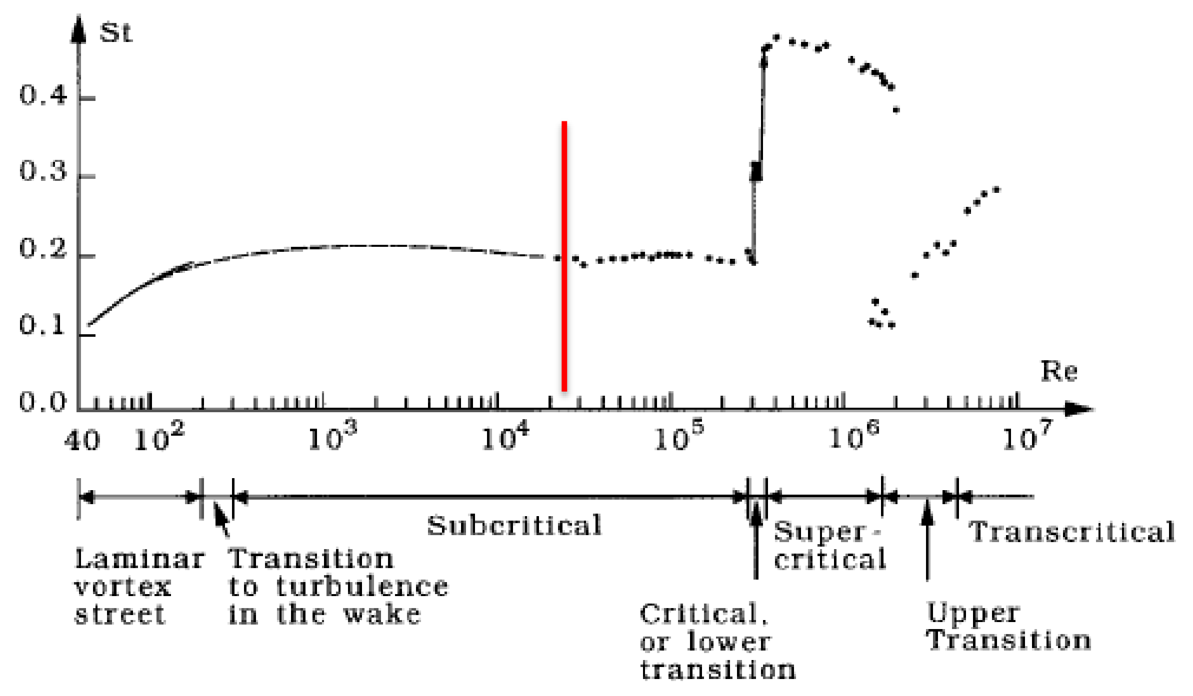


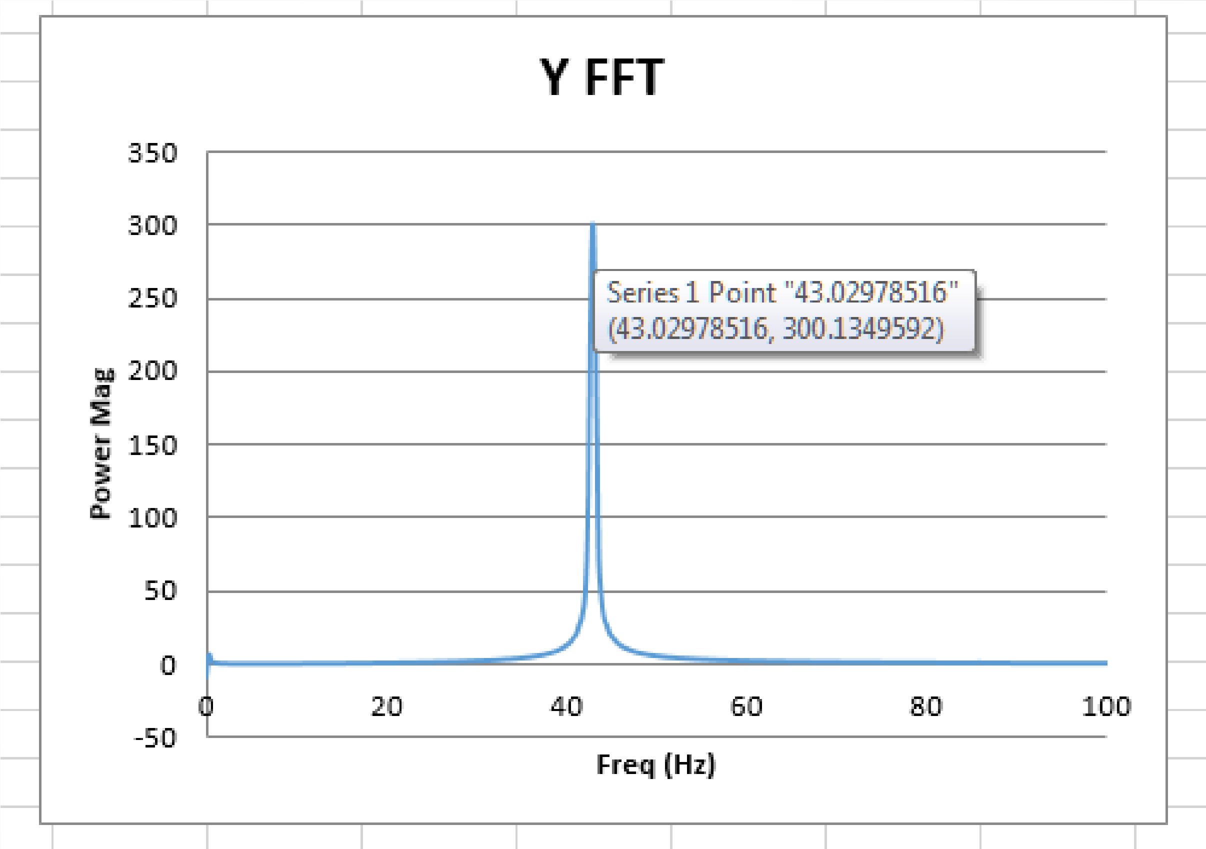
Figure 1.9 Strouhal number for a smooth circular cylinder. Experimental data from: Solid curve: Williamson (1989). Dashed curve: Roshko (1961). Dots: Schewe (1983).

Demo of Extracting Results

Flow over a Cylinder – Results Summary Strouhal Number Setup

St = 0.18
Frequency = 43.03 Hz (6%)

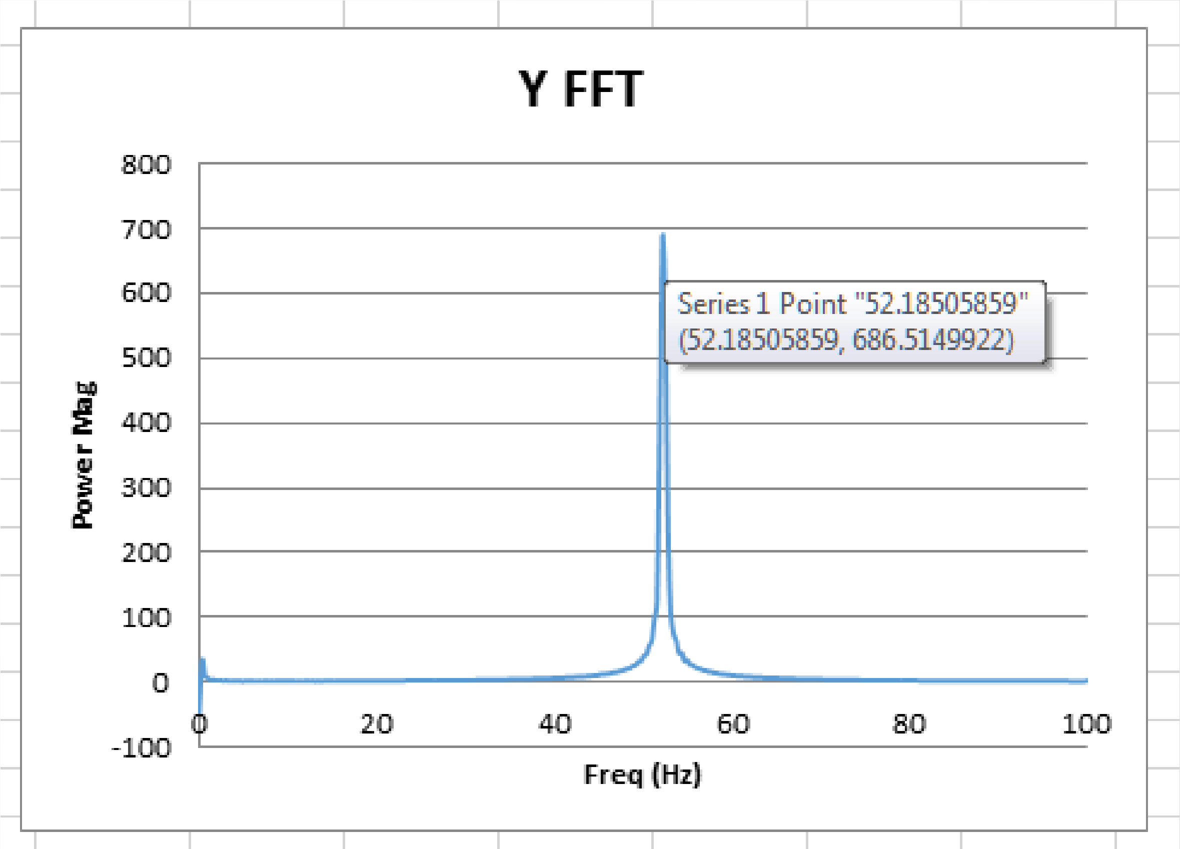
Drag = 1433 N (12%)



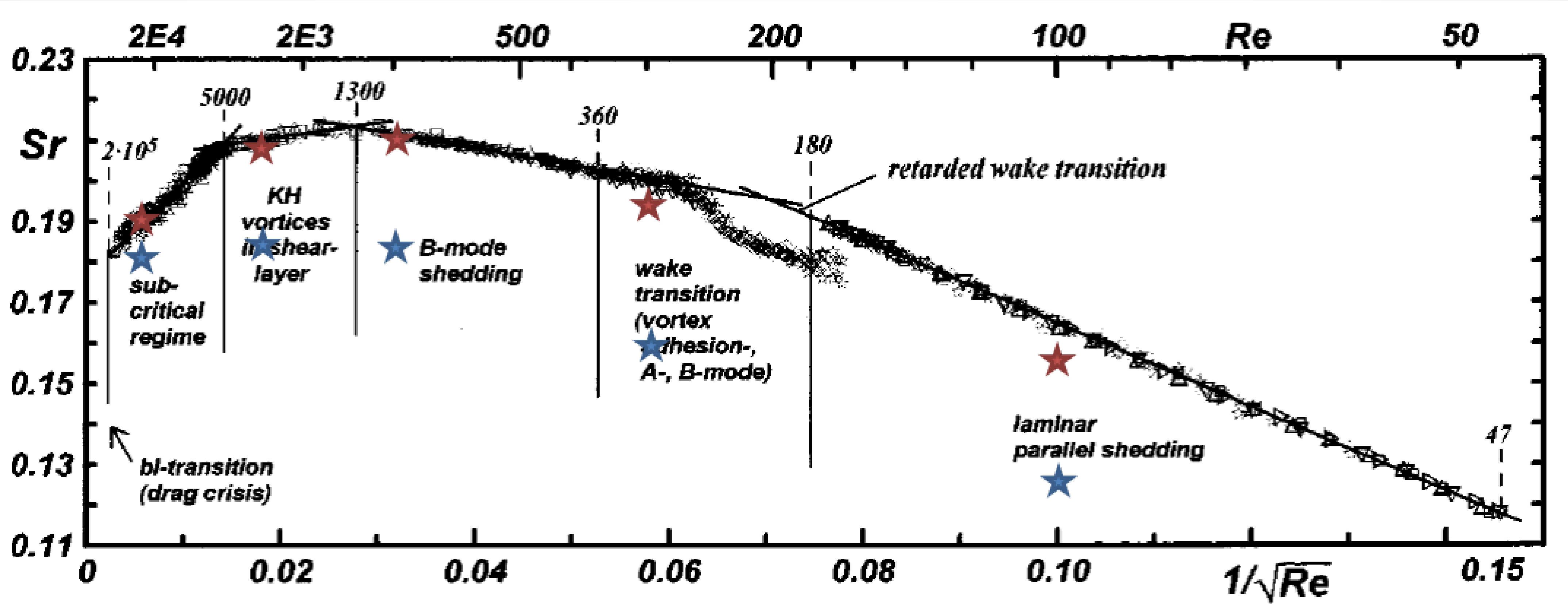
Flow over a Cylinder – Results Summary
Attached Flow Setup

St = 0.22
Frequency = 52.2 Hz (14%)

Drag = 1502 N (7%)



Flow over a Cylinder – Re Sweep



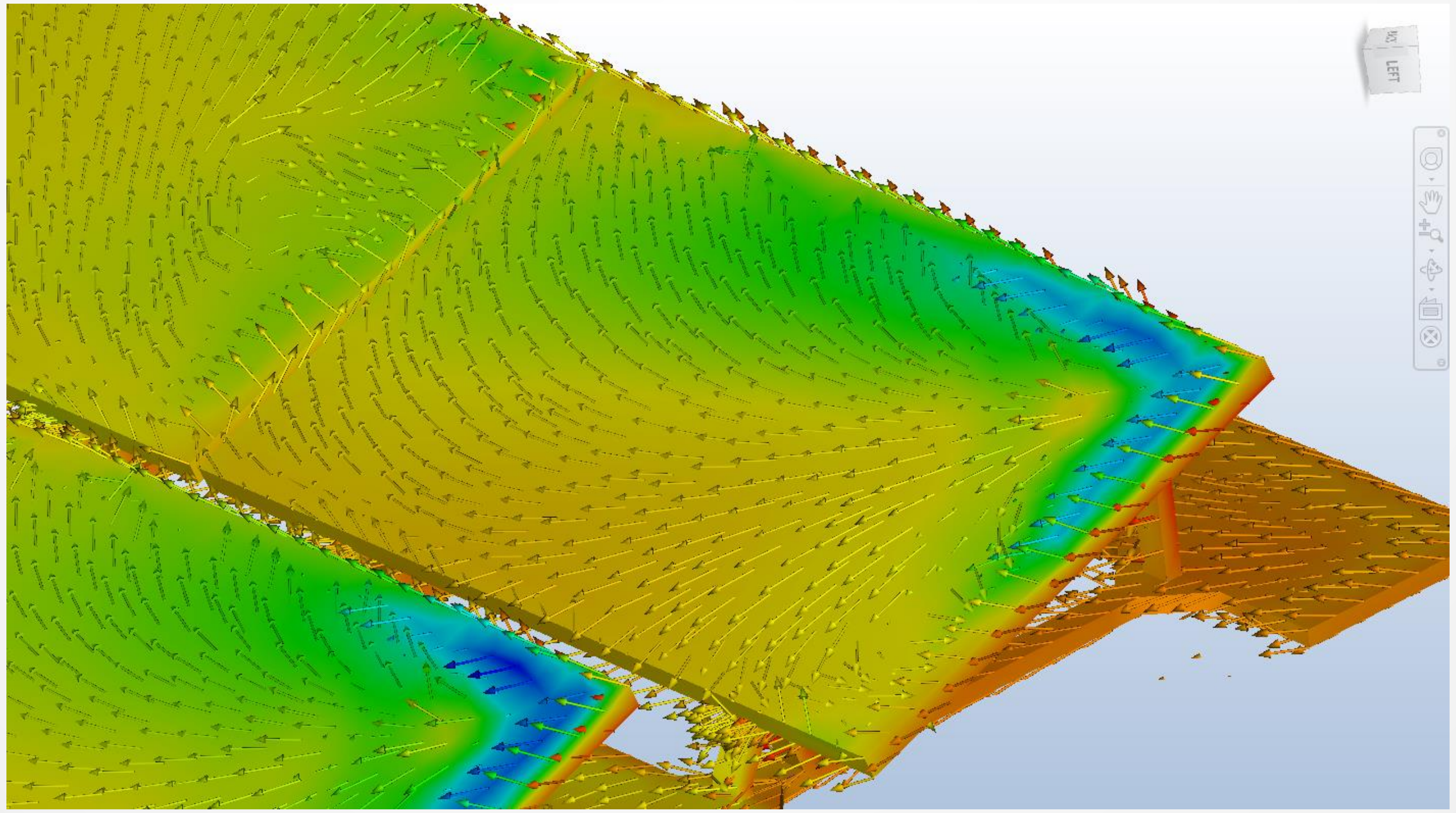
Where to get the Wall Force History Tool?

- Download
  - Today: https://autodesk.box.com/DE10379
  - Eventually: https://apps.autodesk.com/SCFD/en/Home/Index

Bonus! Results Visualization

Flow profile along wall?

- **Flag**: write_wall_dist 1
- Overwrites the Y+ result with a distance from the wall
- **Iso-Surface**: ‘Y+’ with a small value + Vectors



Summary

You should now:

- Understand when to use **SimStudio Tools** or **Surface Wrapping for Simulation CFD**
- Choose one of three approaches to correctly leverage **advanced turbulence models** for aerodynamics analysis
- Have a simple workflow to follow for **mesh adaption** that can then be used for transient analysis
- Use the wall force history tool to capture **steady state or transient wall force** results efficiently from different areas of a complex model.