

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

AUTODESK SIMULATION

SIMULATION TV RESOURCES EVENTS DISCUSSIONS LEARNING BLOG MORE SIGN IN

Simulation Community

Find answers, share expertise, and connect with your peers

Simulation TV

More Videos →

Autodesk Simulation Mechanical 2016 What's New Overview

Autodesk Simulation Mechanical 2016 What's New

Sualp Ozel, PE
Product Manager

Events

View and register for simulation events like user groups, webinars and tradeshows in your local area.

SEE ALL EVENTS →

Discussions

All Discussions →

- NEW Bracket error? 1
- NEW API NewFromTemplate 1
- NEW Envelope 2
- NEW RC SLAB REINFORCEMENT COLUMN HEADS 1
- NEW User Coefficient Seismic Load Pattern 2
- NEW Why is the 3D Filter tool for visualizing re... 1
- Student version of Robot 2014 2
- Save Intervals Not Working 2
- API: CFT Concrete fillet column in Robot 2
- Interaction Diagram 2

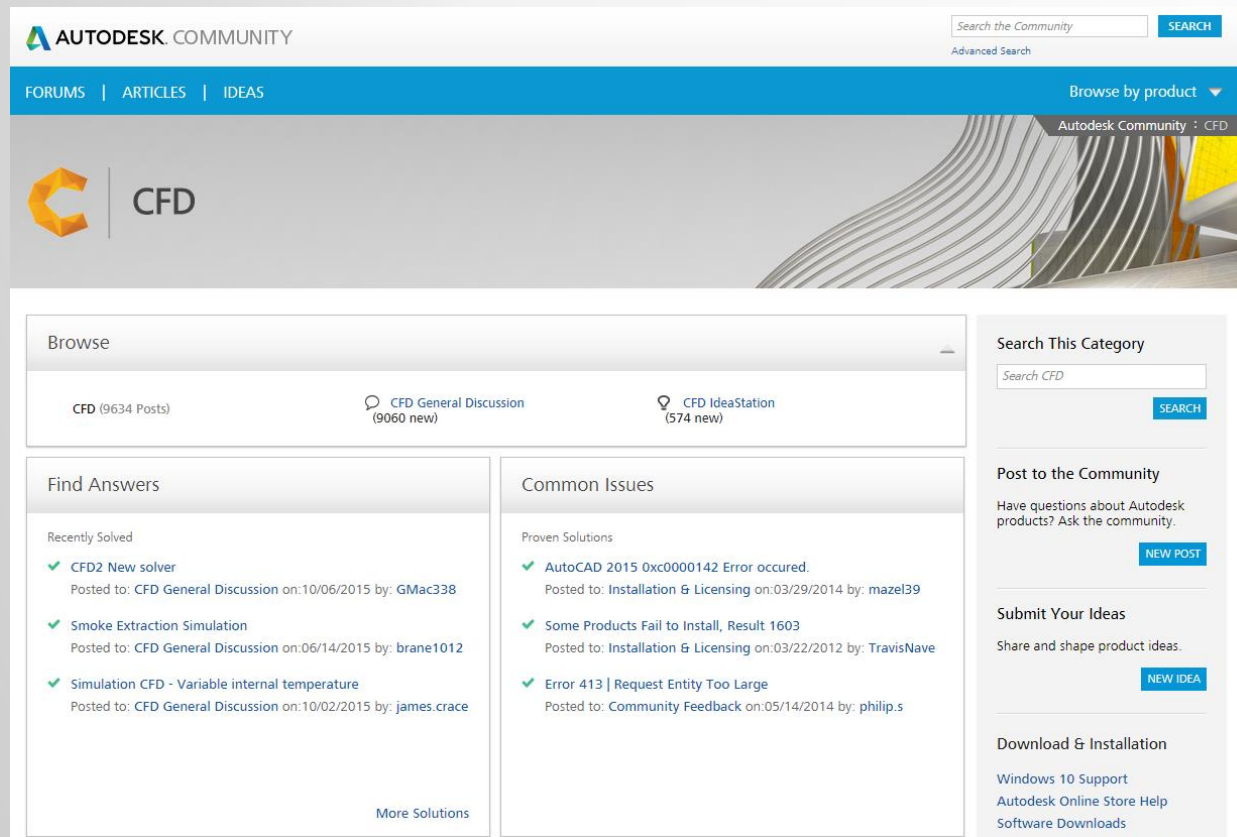
Resources

More Resources →

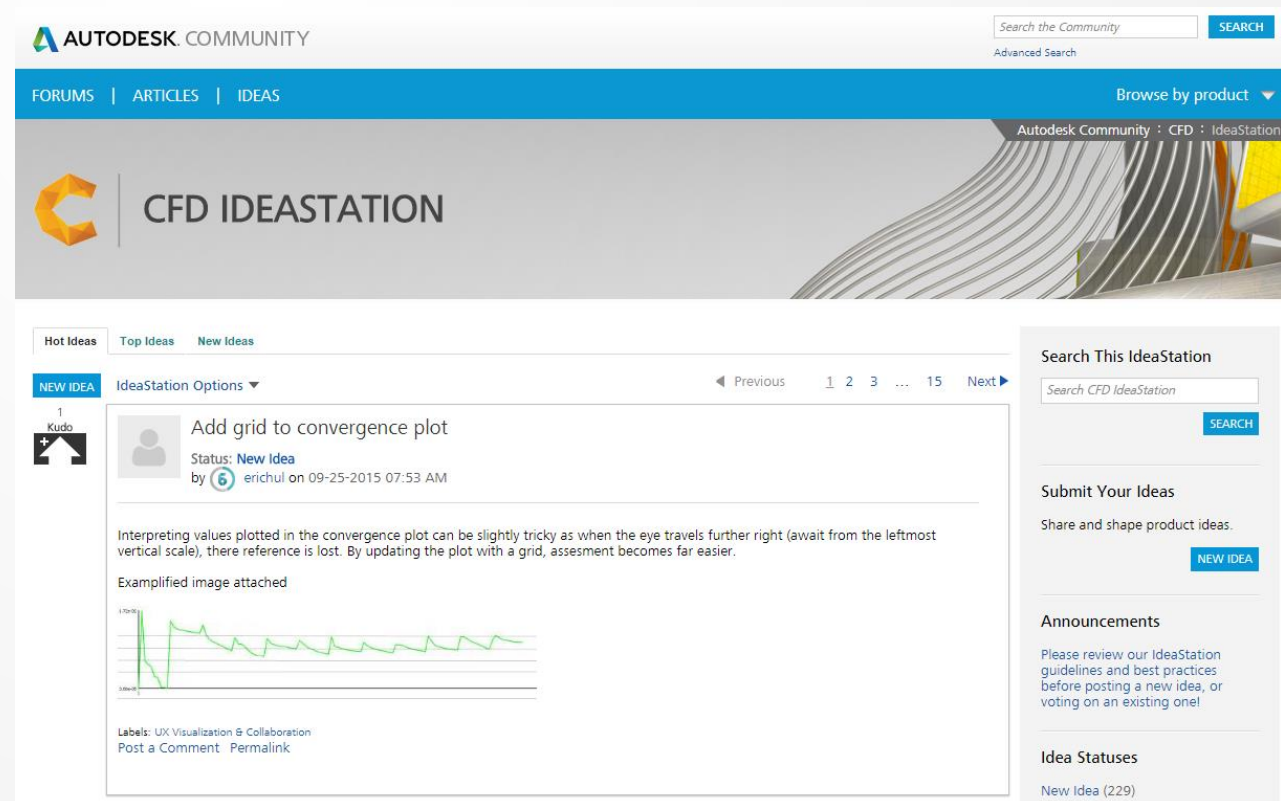
Join The Discussion!

- 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.



SUPPORT & LEARNING

CUSTOMER SERVICE

COMMUNITY

What is an Autodesk® Help Webinar?

Like (8) Share

Apr 09 2015

AUTODESK® HELP WEBINARS

The Autodesk Help Webinar series is designed to showcase time-saving tips and tricks that can be used to immediately increase your productivity. This live set of interactive training sessions hosted by our very own Autodesk Technical Support Specialists offers a unique look into some of the most impactful in-product features and functionality that you may not already be familiar with.

The webinar format is a 20 - 40 min period of in-depth topic discussion. The remainder of the hour is an open question & answer session covering any topic of interest by the audience. The webinar is considered a series and after signing up you will be notified of future events.

Translate this Page

English

Translate

☐ Use this language while I browse

This is machine translation.

Need Assistance?

BUILD YOUR NASTRAN IN-CAD IQ

Nastran In-CAD leverages the advanced Autodesk Nastran solvers within both Autodesk Inventor and Dassault Solidworks to allow designers and engineers to embrace analysis-driven design. Sessions will be hosted by Autodesk finite-element analysis experts and cover everything from the theory behind FEA to advice on troubleshooting common issues and errors.

SIGN UP HERE

BUILD YOUR ROBOT SIMULATION IQ

Join the Autodesk Support Team as we help you explore how to leverage Robot to its fullest potential, covering: best modeling techniques as well as tips and tricks. We will help you to improve your workflow and reduce the time you need to complete your projects.

SIGN UP HERE

BUILD YOUR SIMULATION MOLDFLOW IQ

Are you interested in enhancing your Autodesk Moldflow skills? Join the Autodesk Support team for monthly webinars to learn different techniques for various road blocks you may encounter with your simulations. We will cover best practices, tips and tricks as well have Q&A throughout the session!

SIGN UP HERE

BUILD YOUR SIMULATION CFD IQ

The CFD industry is a complex and growing field for engineers around the world. Join the Autodesk Technical Support team as we help you explore how to leverage Simulation CFD to its fullest potential, covering: best practices, validation models, new product enhancements, and much more. Come and add your own personal insight and experiences during our open Q&A!

SIGN UP HERE

BUILD YOUR SIMULATION MECHANICAL IQ

Simulation Mechanical is Autodesk's finite-element analysis package for engineers and analysts, now with added advanced solvers from Autodesk Nastran. Learn about new features and tips and tricks to help boost your Mechanical IQ in this reoccurring webinar series hosted by Autodesk Technical Support Specialists and guest speakers who work with the software every day.

SIGN UP HERE

Upload

Autodesk Simulation

Subscribed

Home

Videos

Playlists

Channels

About

Featured

Build your Simulation IQ

by Autodesk Simulation • 49 videos • 5,033 views • Last updated on Sep 23, 2015

A webinar series brought to you by Autodesk Subscription Services. Most of these technical webinars are presented by the Autodesk Technical Support team, but also includes invited guests from product development, product design, and technical sales.

Play all

Share

Save

1

Working with Single Elements to Understand Simulation Mechanical

by Autodesk Simulation

48:50

2

Pre-Simulation Geometry Prep using Autodesk SimStudio Tools with Moldflow Insight

by Autodesk Simulation

45:04

3

Helius PFA-Advanced Material Exchange Workflow

by Autodesk Simulation

36:14

4

Recurdyn Interoperability with Simulation Mechanical 2016

by Autodesk Simulation

48:04

5

Utilising the Decision Center in Autodesk CFD 2016

by Autodesk Simulation

39:07



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

Google Royce

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 ?

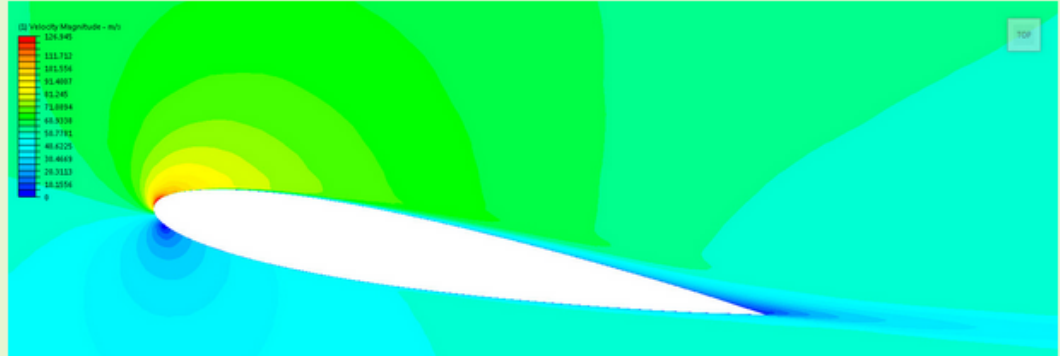
I have sent this also to Autodesk Support, with relevant files included.

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%



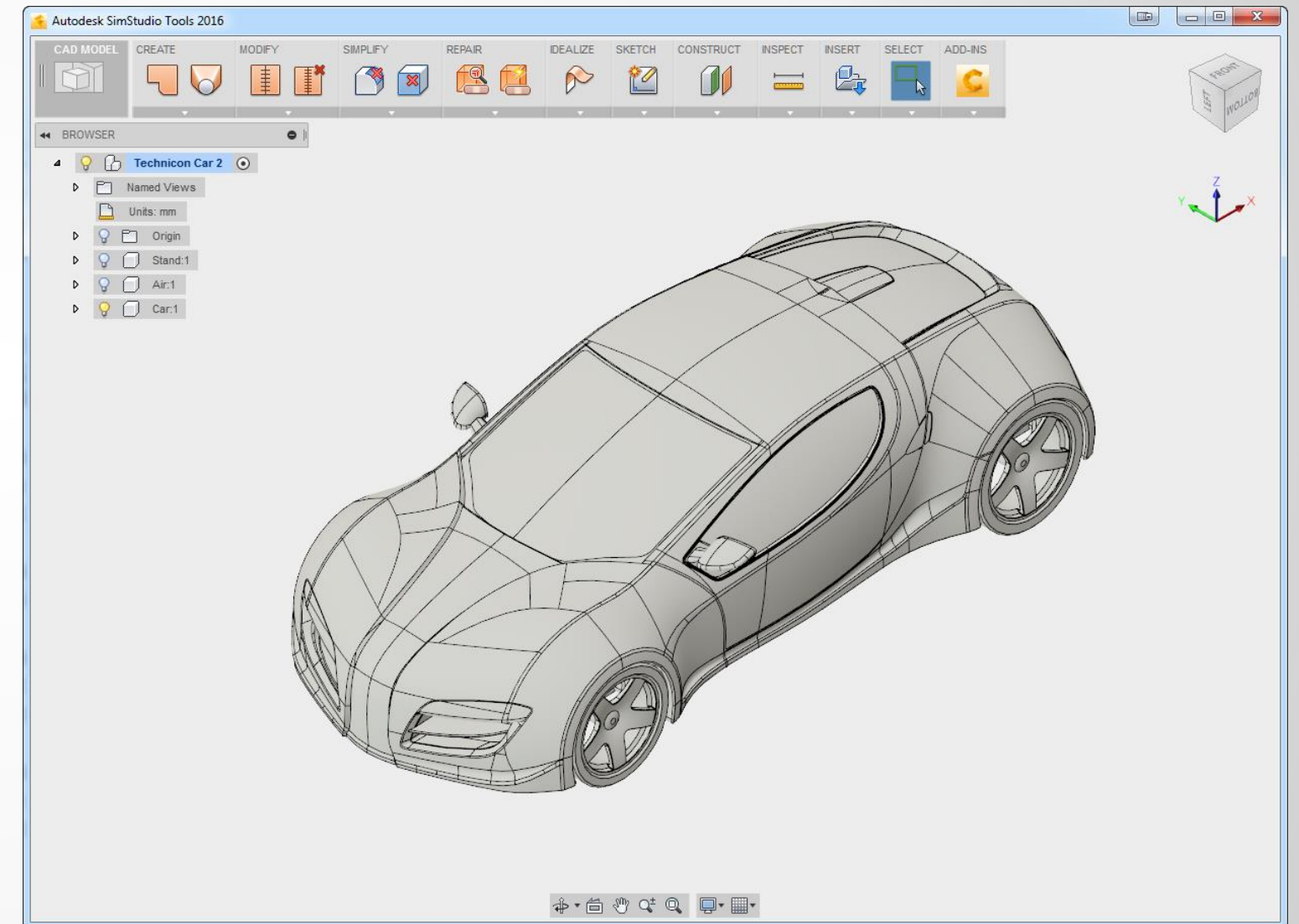
Capitalize on SimStudio Tools or Surface Wrapping

An aerial perspective of a city skyline featuring a large bridge spanning a river. The bridge has a rainbow-colored line along its edge. In the background, a dense cluster of skyscrapers is visible under a clear blue sky. The foreground shows green grass, trees, and a small landscaped area with purple flowers.

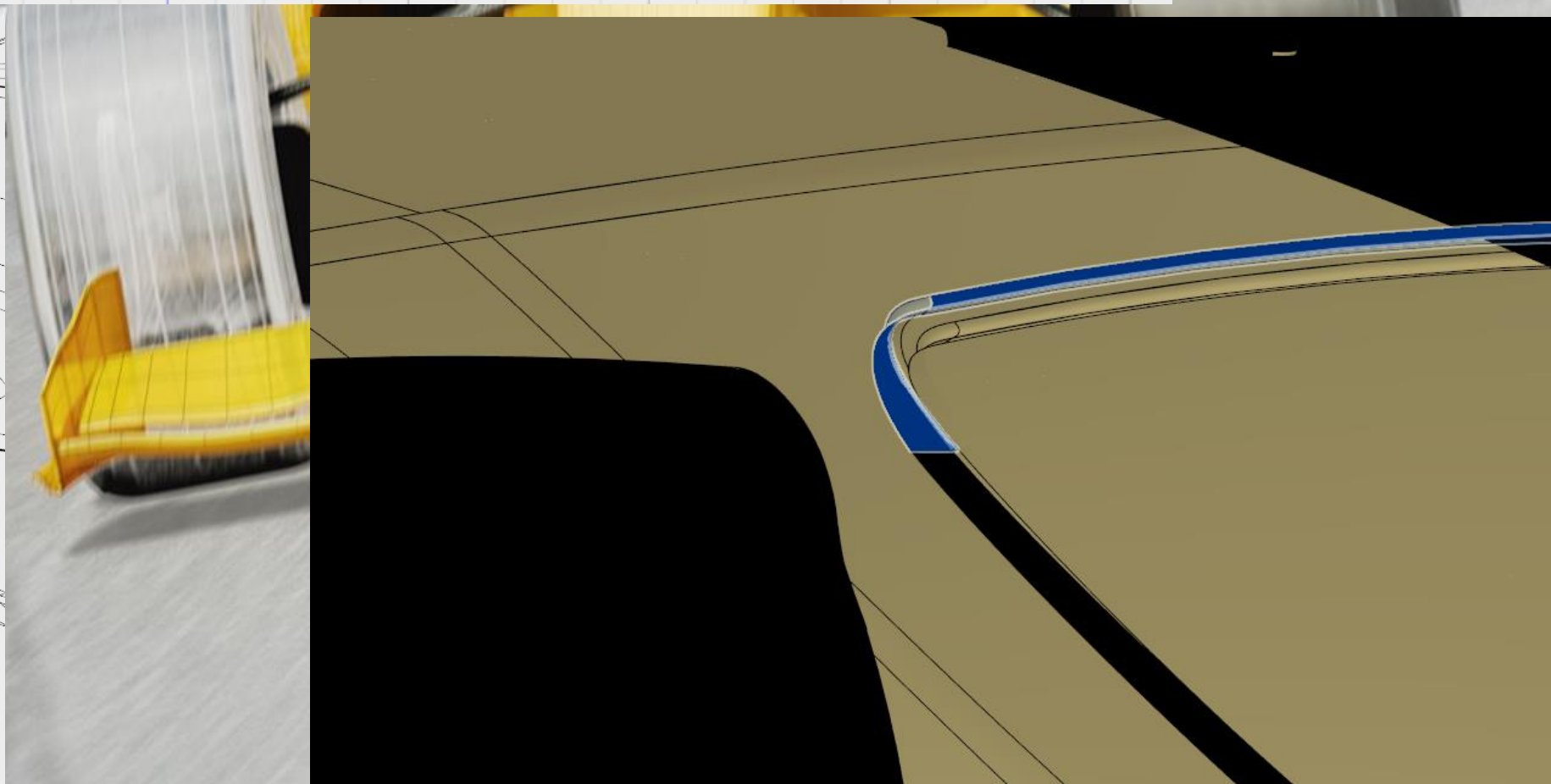
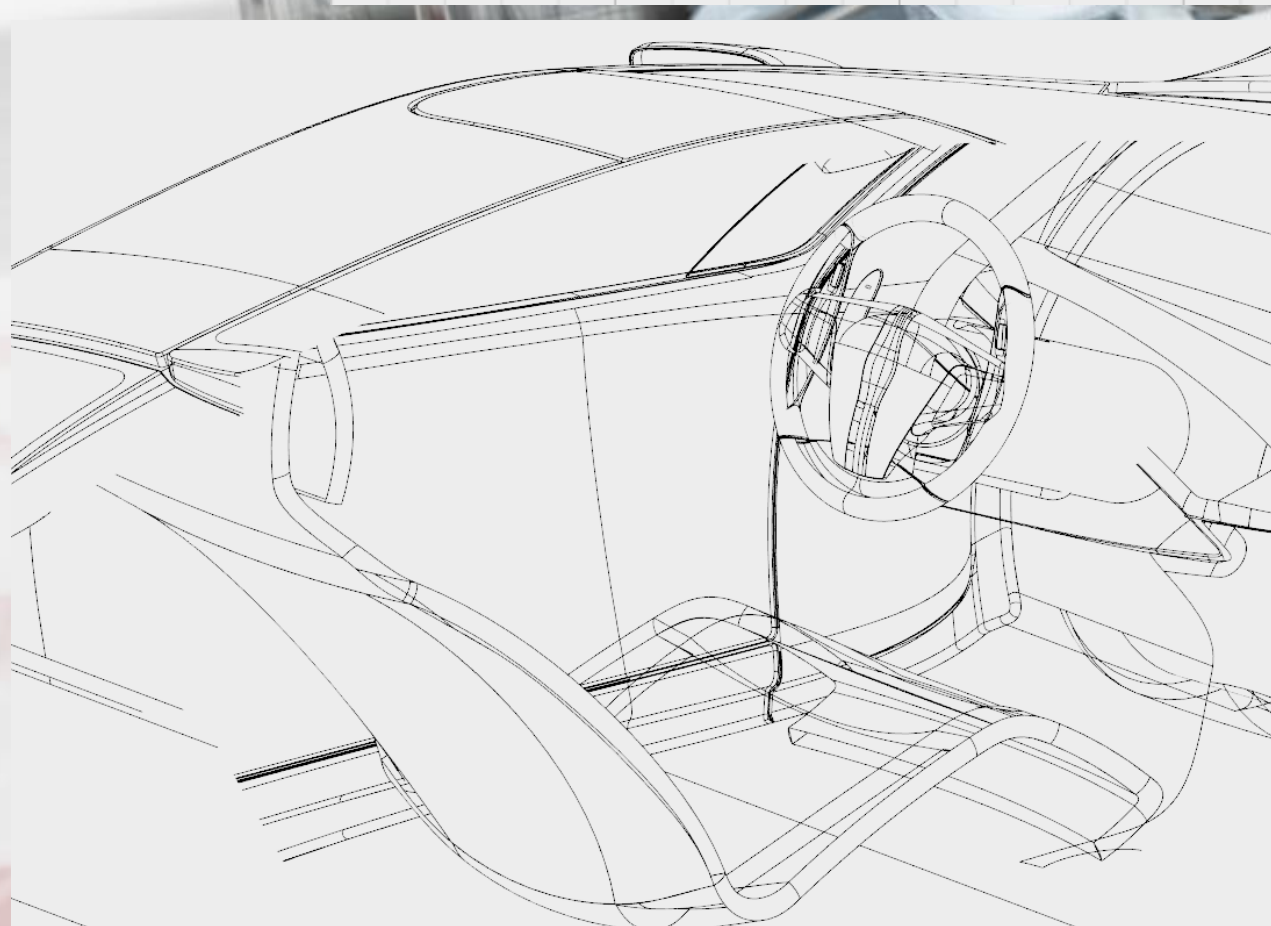
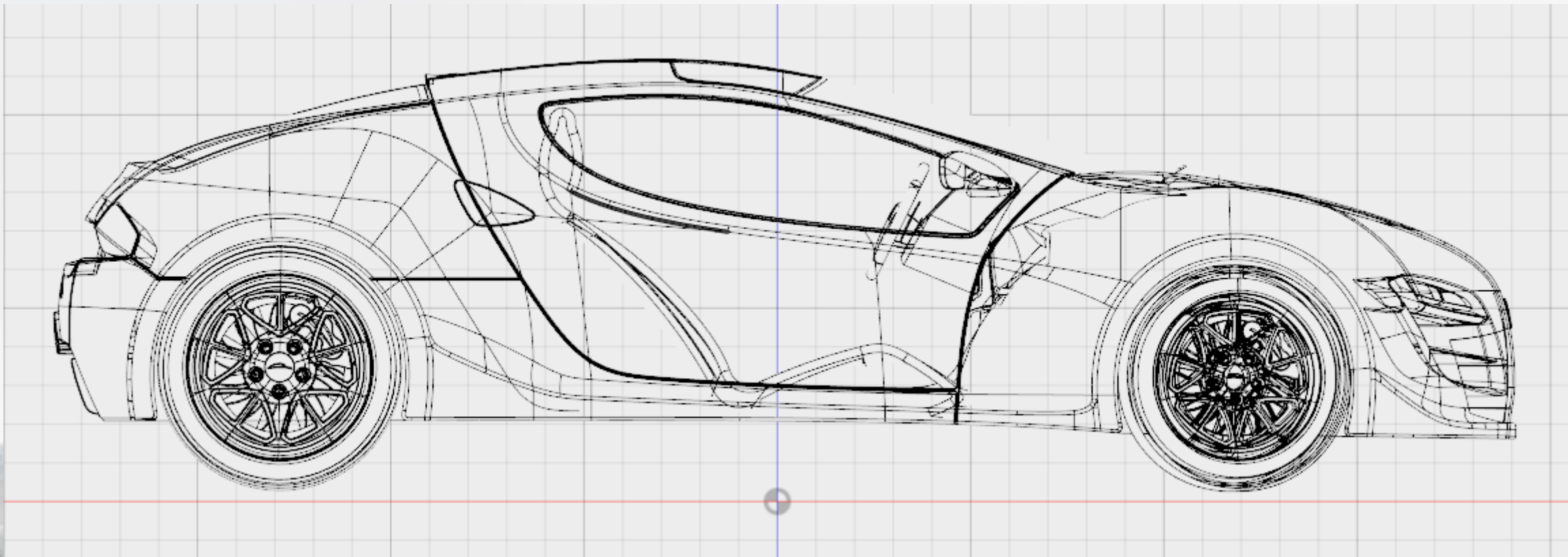
SimStudio Tools – CAD Prep

What is SimStudio Tools?

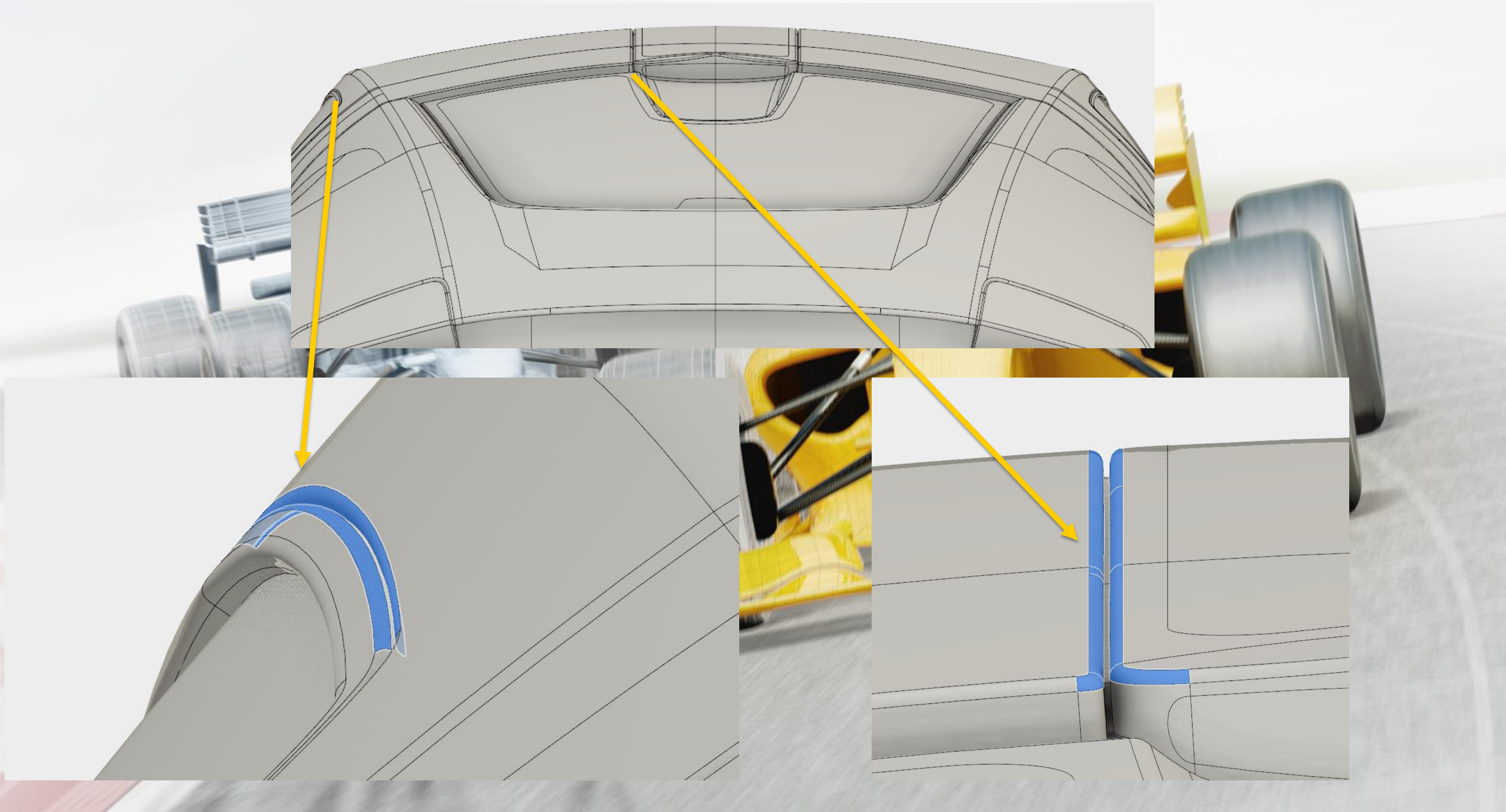
- Built off Fusion360
- CAD tool geared for Simulation Community
- Fastest way to prepare your geometry to be simulation ready.



Alias Dirty CAD

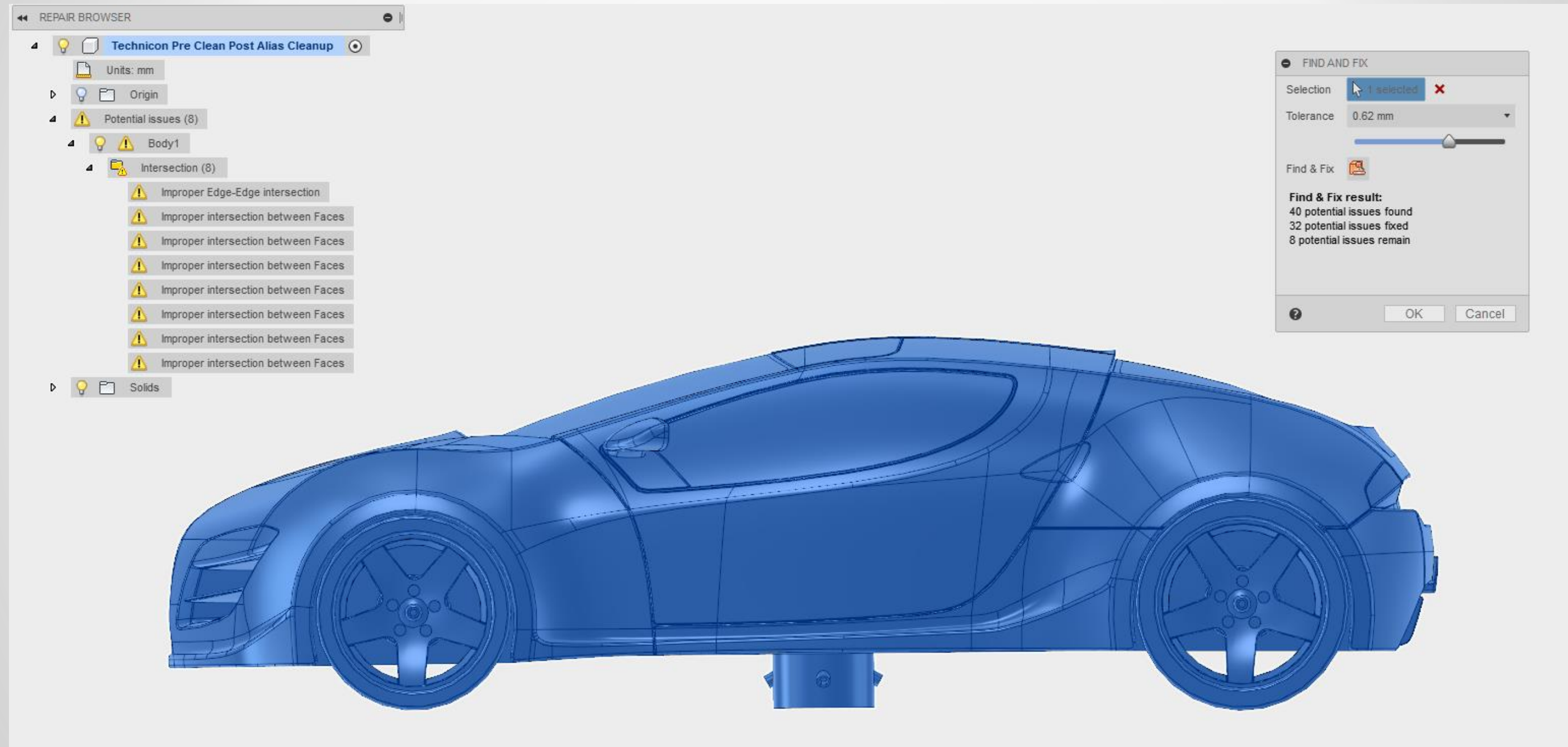


Alias 'Clean' CAD



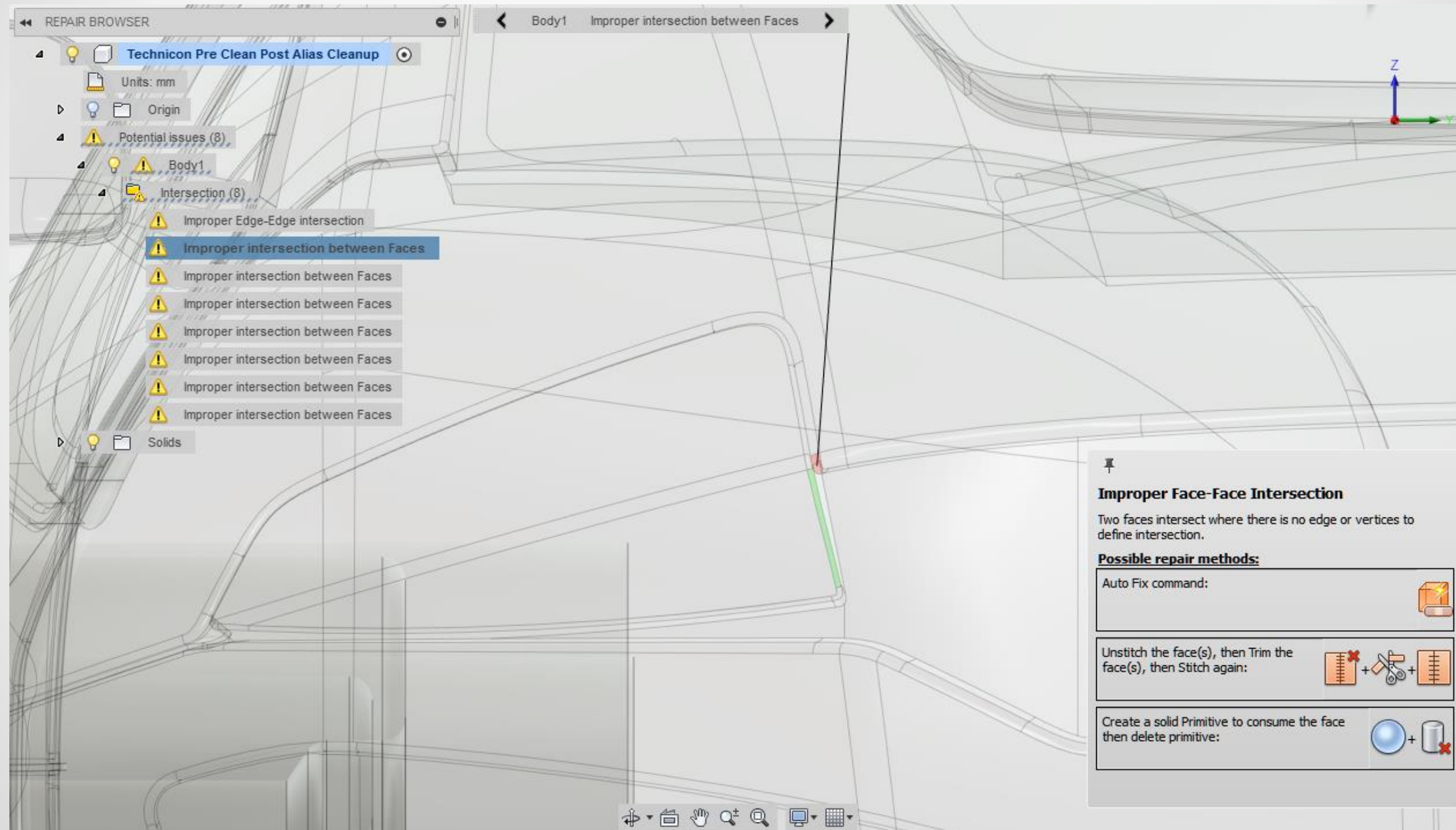
SimStudio Tools – Find and Fix

8 Potential Issues Remain



SimStudio Tools – Find and Fix

Locates issues with suggested solutions!

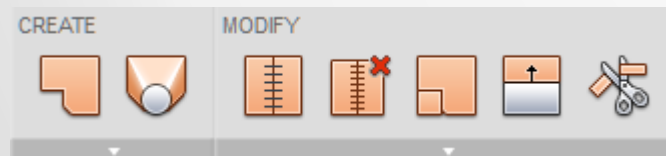


SimStudio Workflow model simplification

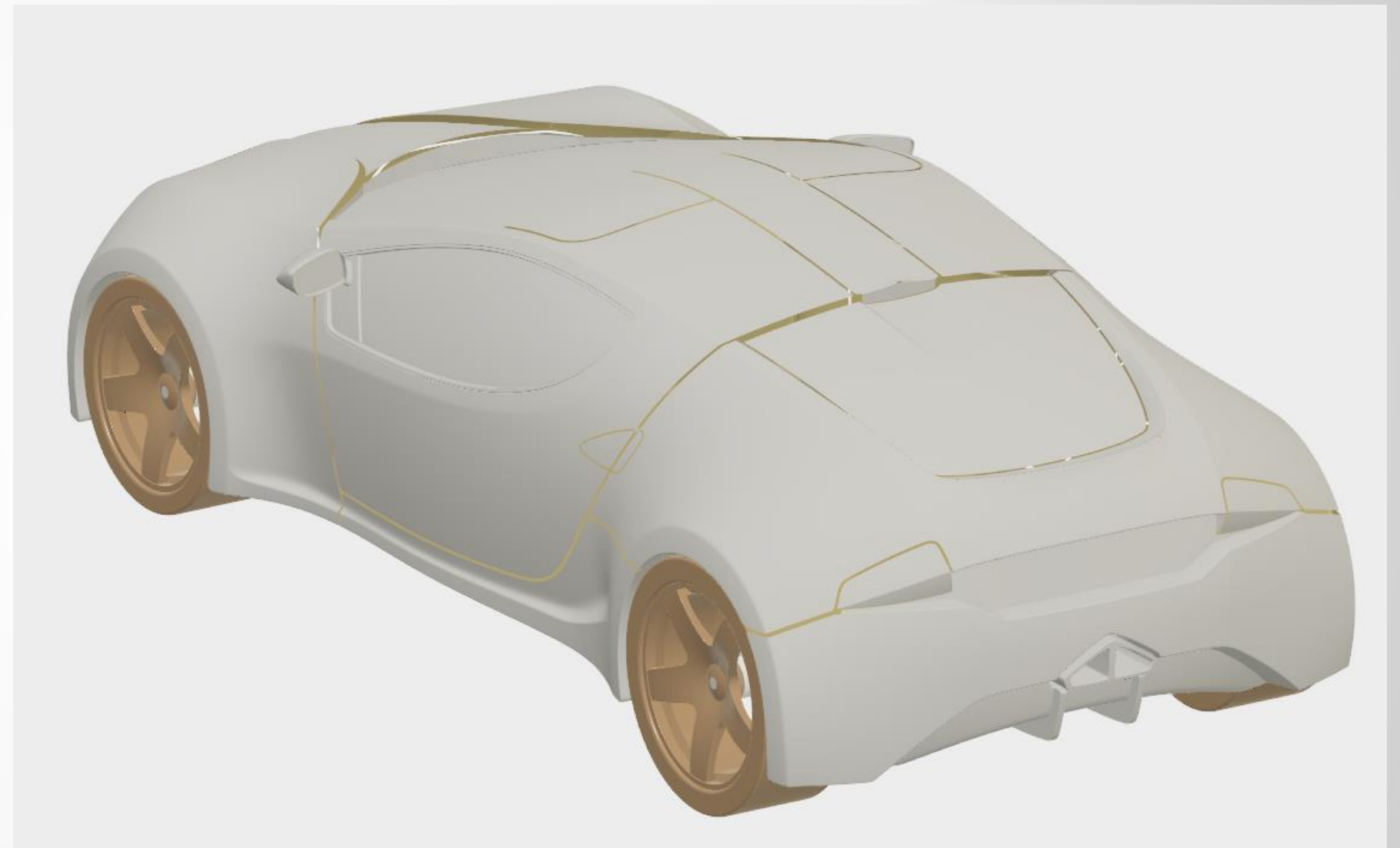
1. Remove Seams

2. Fill Seams

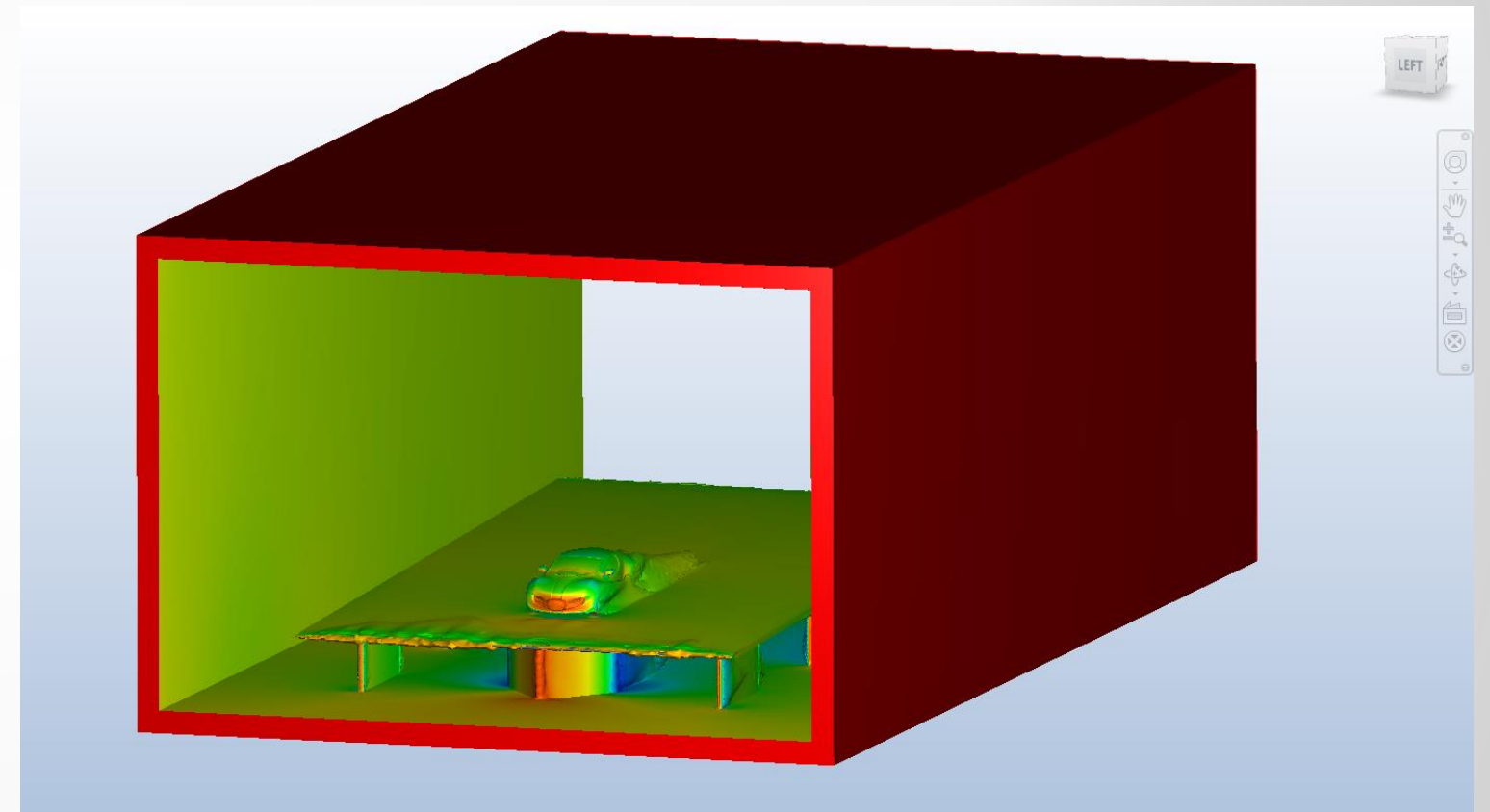
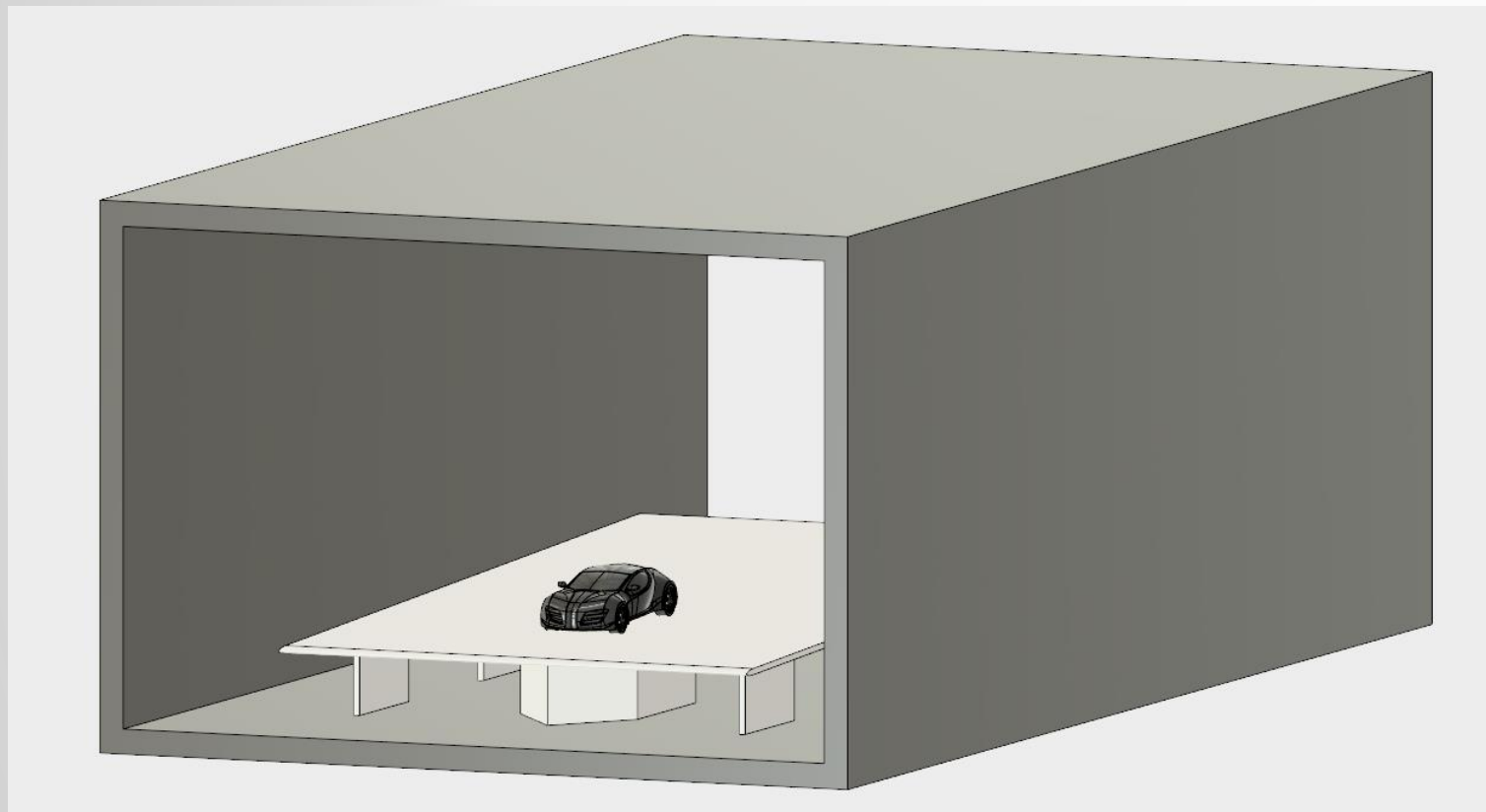
- Patch
- Loft
- Stitch/Unstitch
- Merge
- Extend Face/Trim



Examine Work



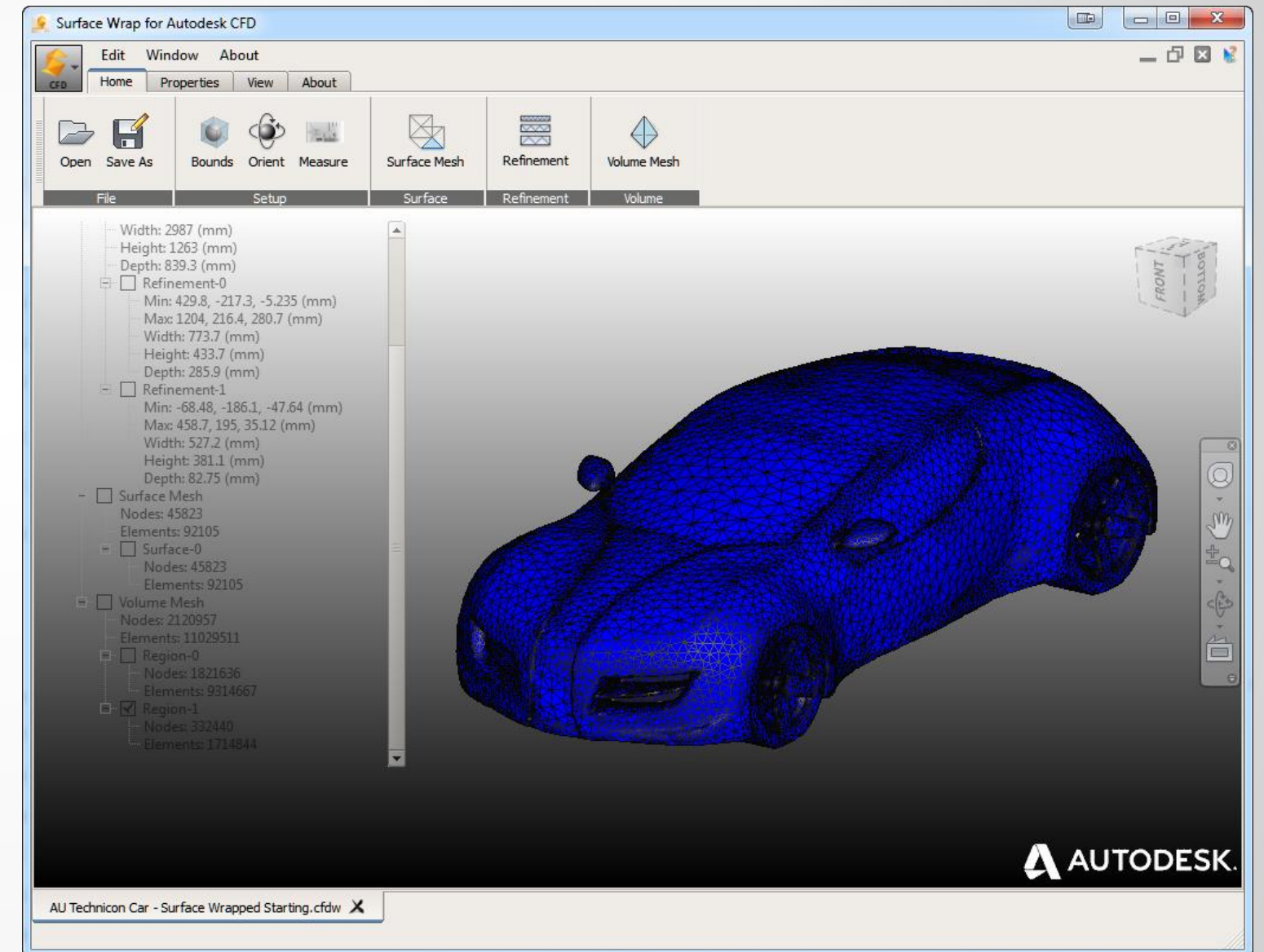
Car Prepared for Wind Tunnel and Simulation Ready!



Surface Wrapping – Avoid CAD

What is Surface Wrapping for Autodesk CFD?

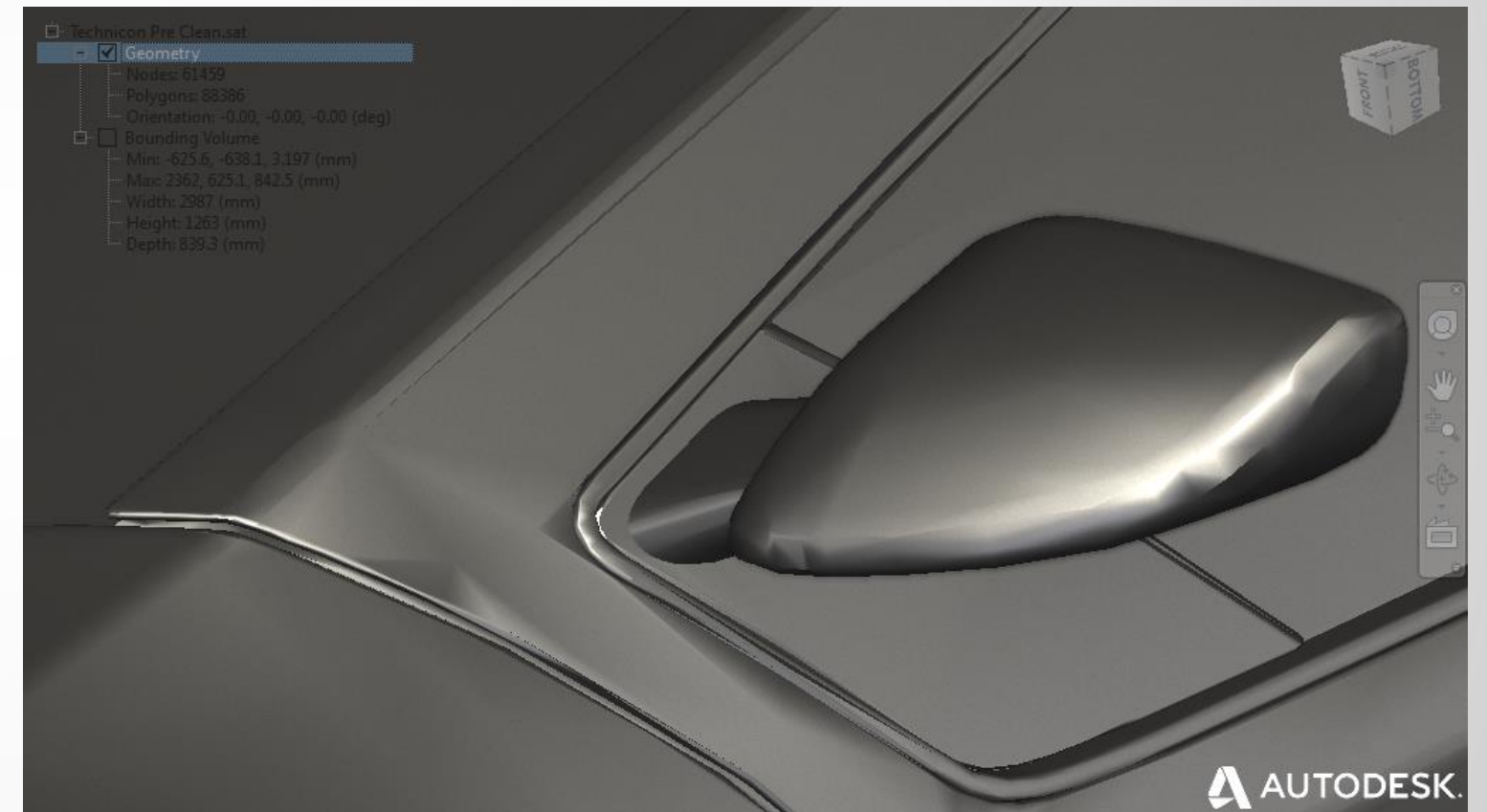
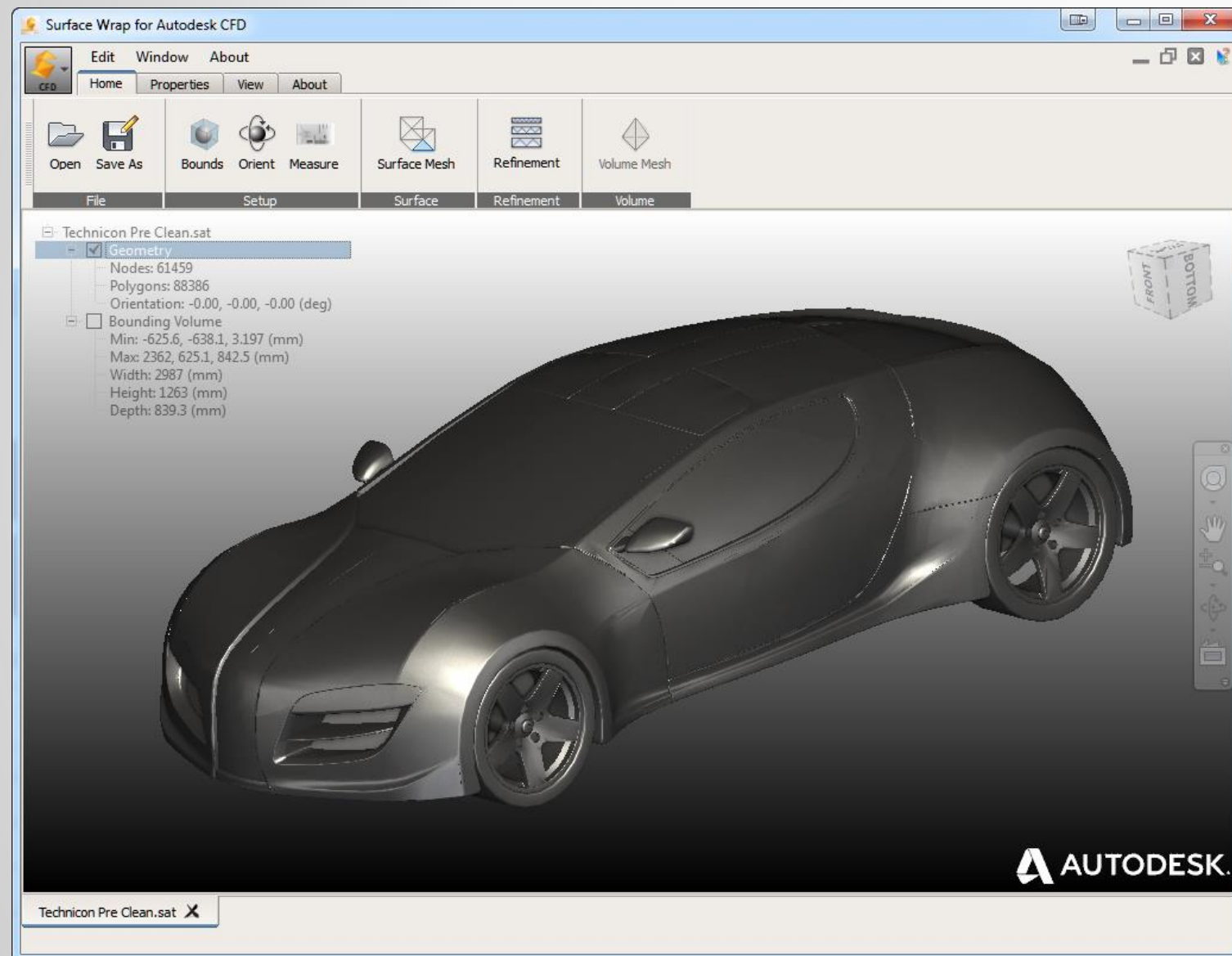
- Geometry Surface Wrapping technology
- External Flow/Wind Tunnel
- Can simulate difficult to nearly impossible geometry without extensive CAD cleanup



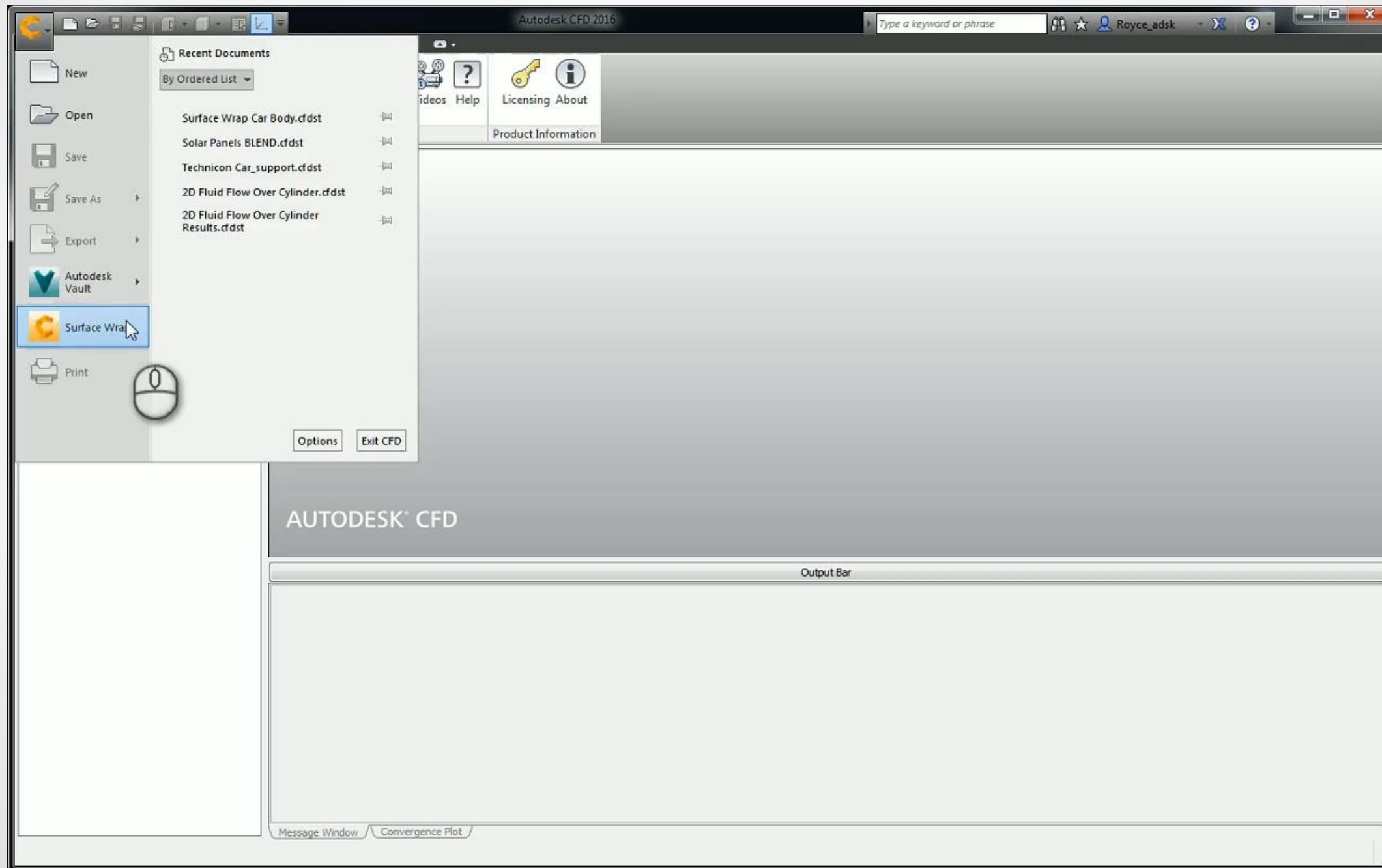
Basic Steps for Surface Wrapping

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

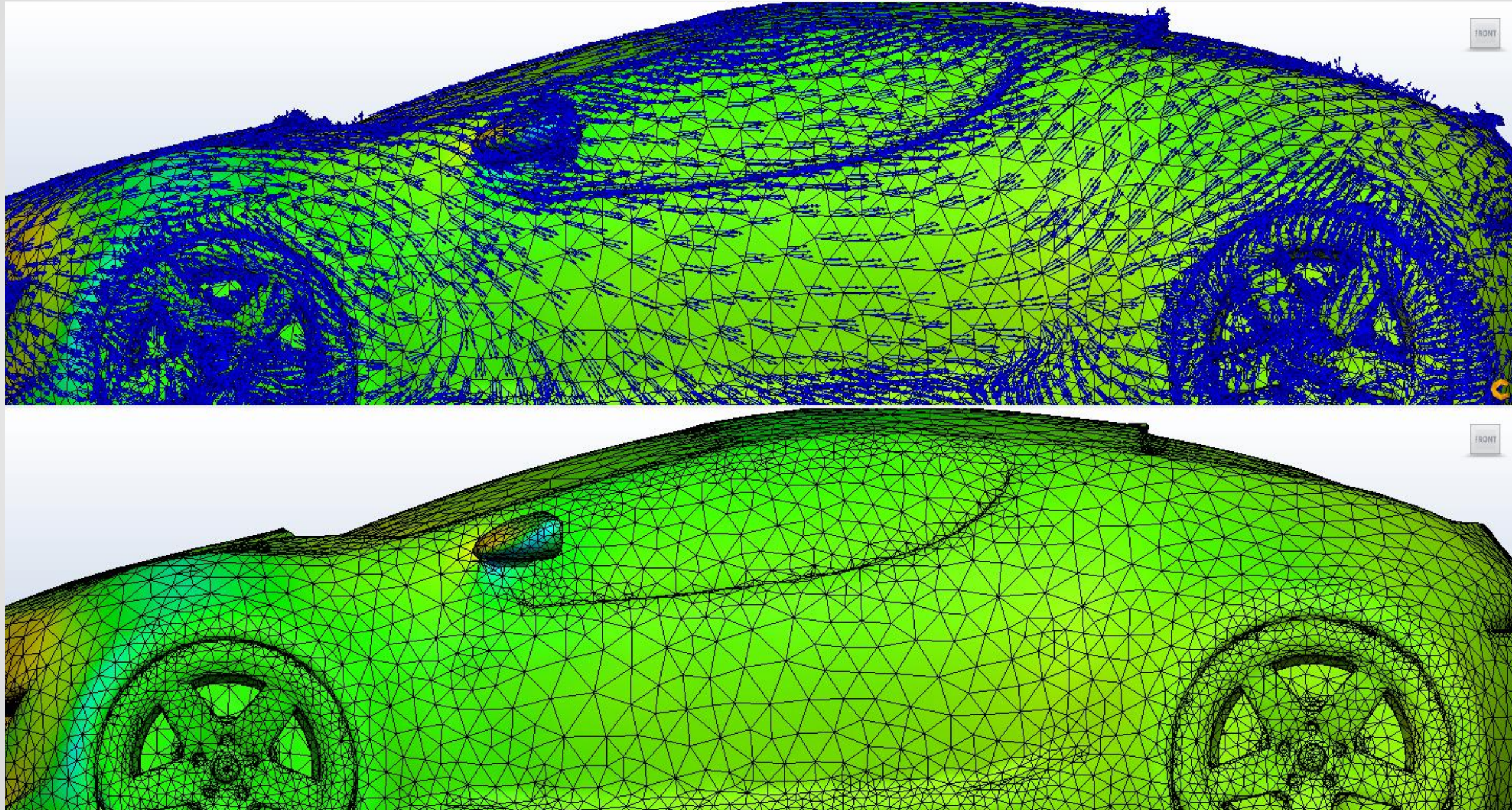
Dirty CAD Ready for Surface Wrapping



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

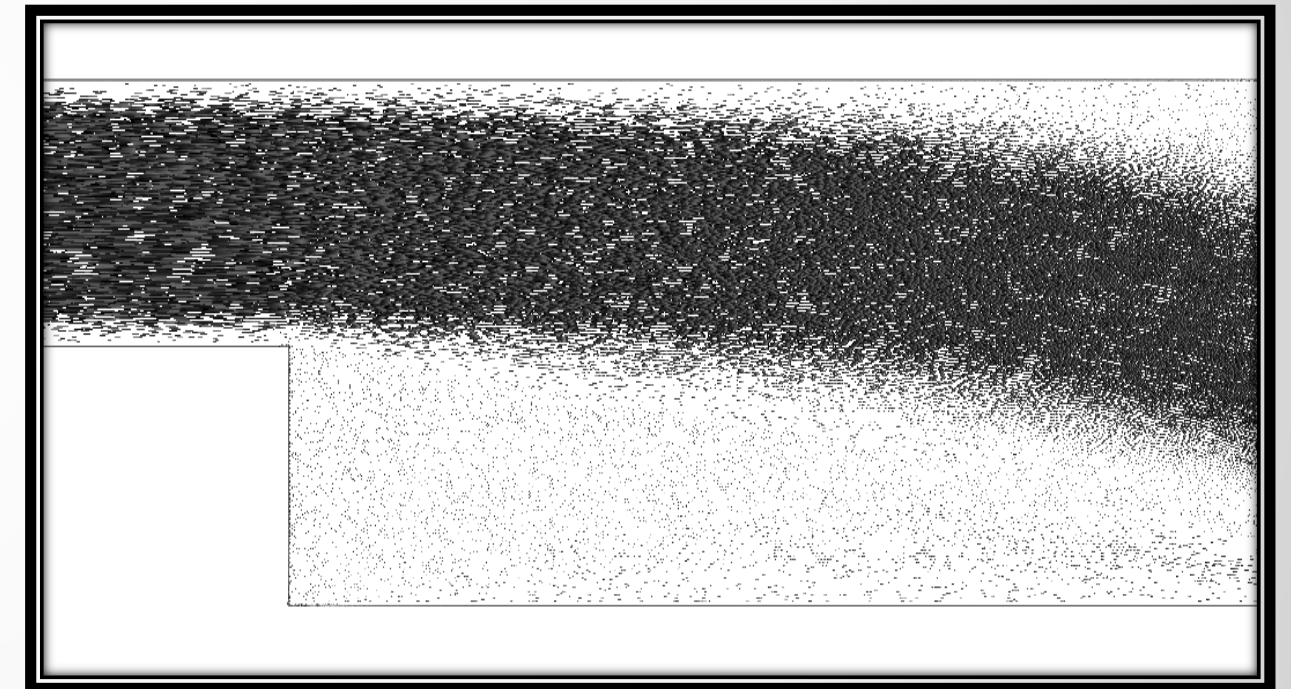
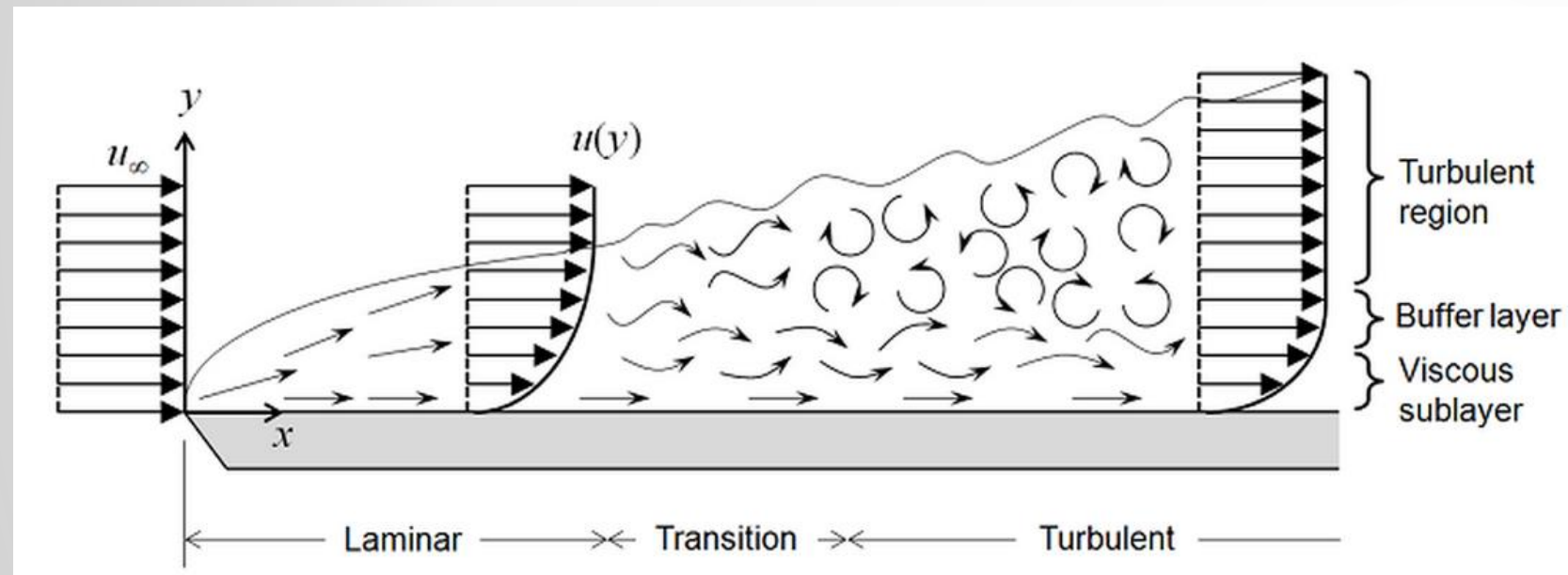
Advanced Turbulence model for Aerodynamics

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

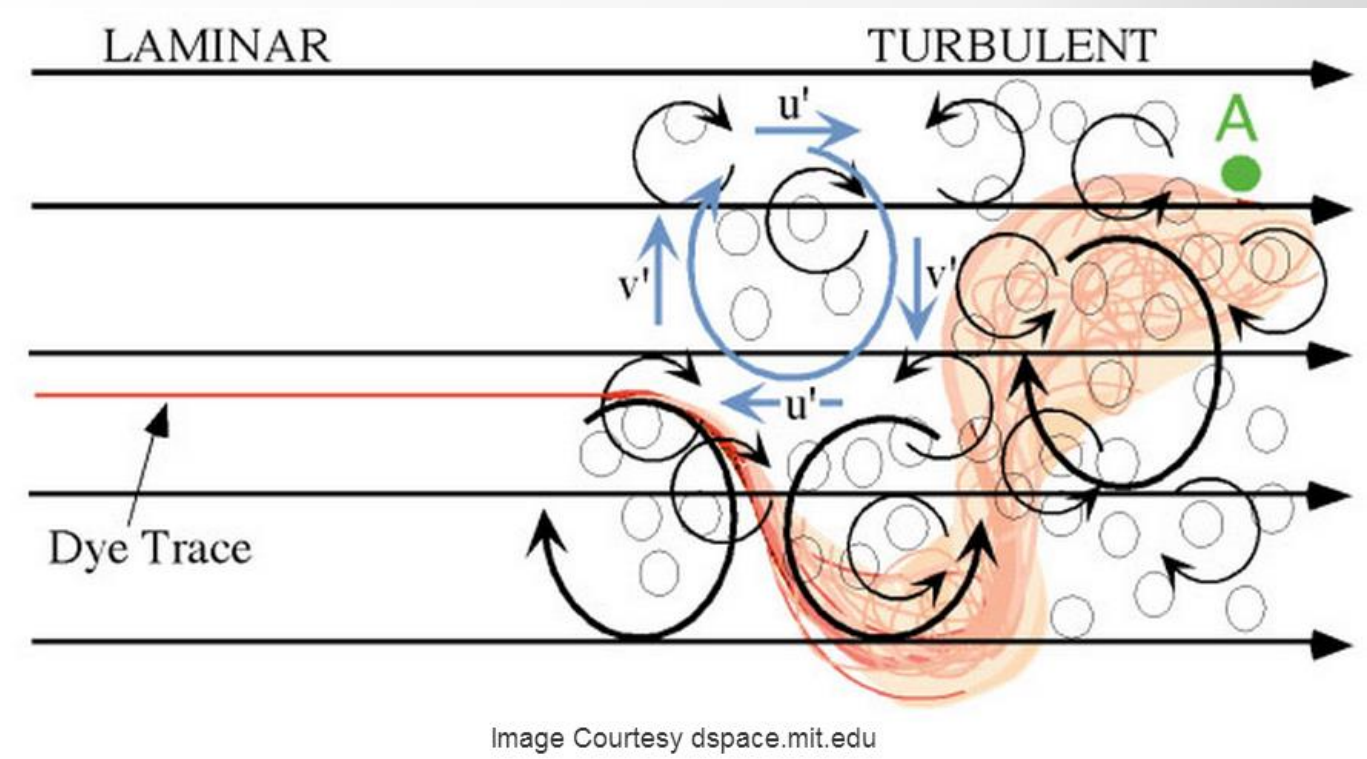
What is Turbulence?

Noun: Violent or unsteady movement of a fluid.

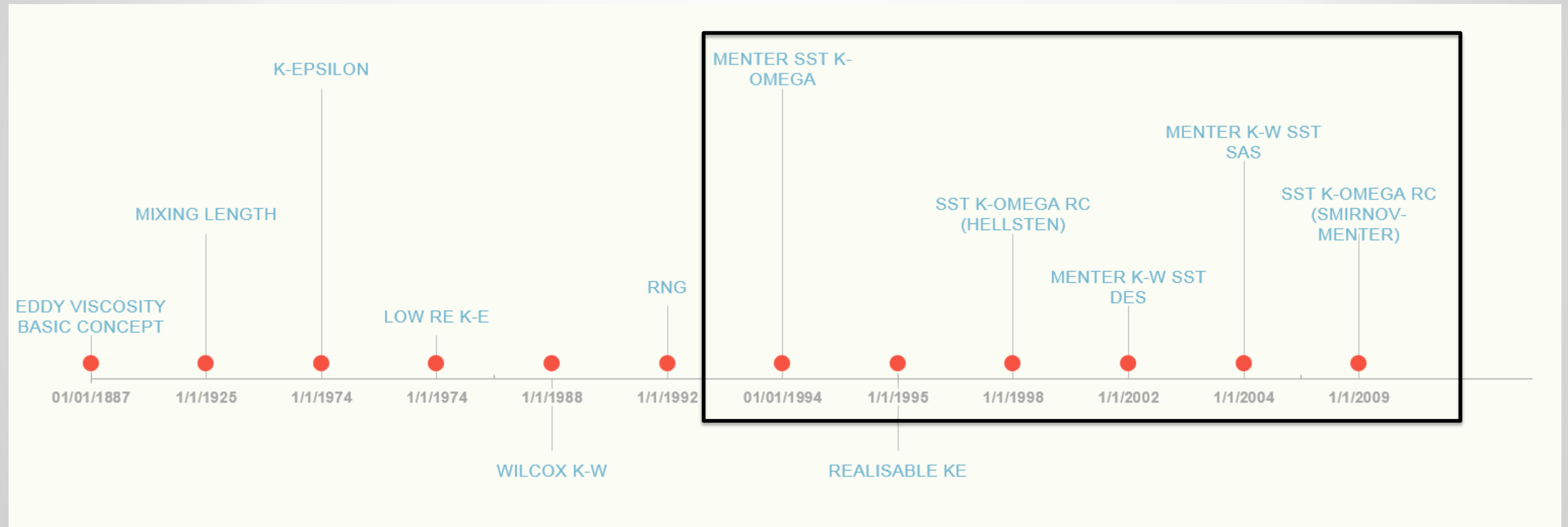


How is Turbulence modeled in CFD Software?

- **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



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)

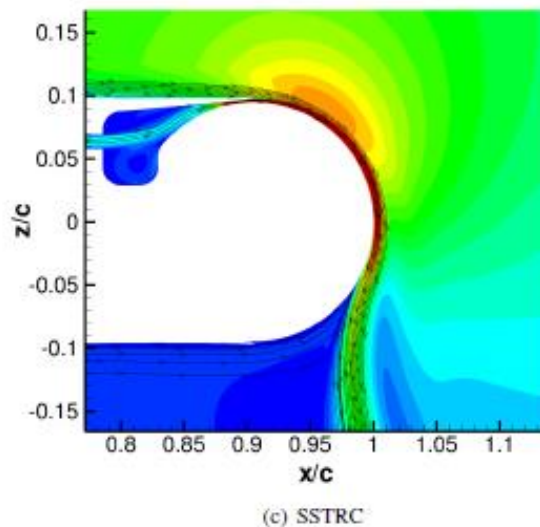
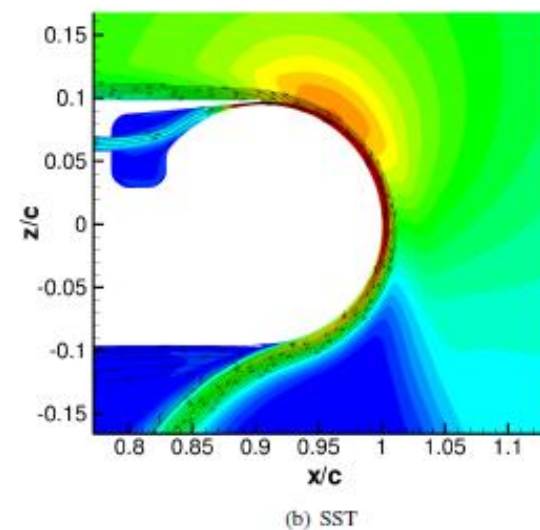
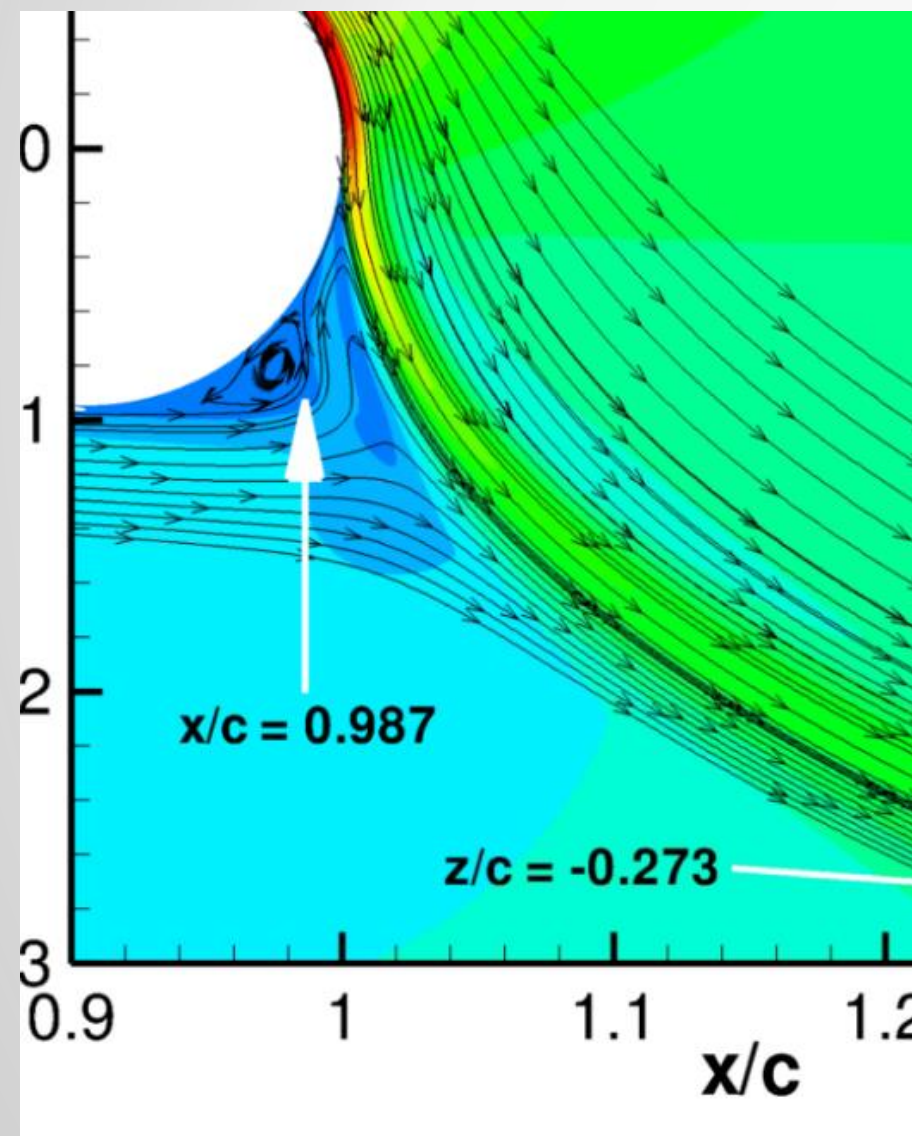
Two-equation, linear eddy viscosity RANS

- 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

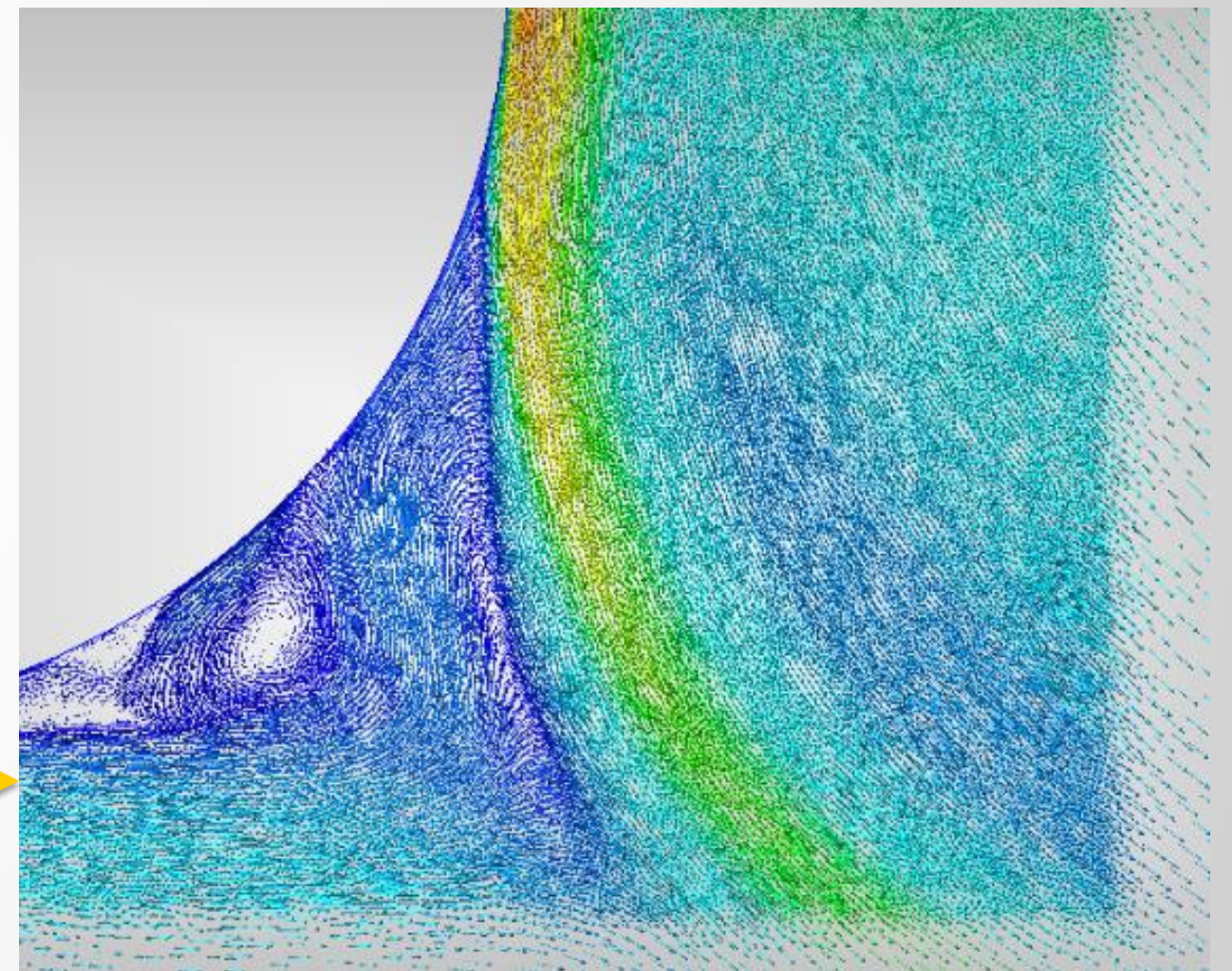
Why use the Hellsten model?

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

NASA Langley Study of SST Models



Autodesk CFD Results w/Hellsten



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

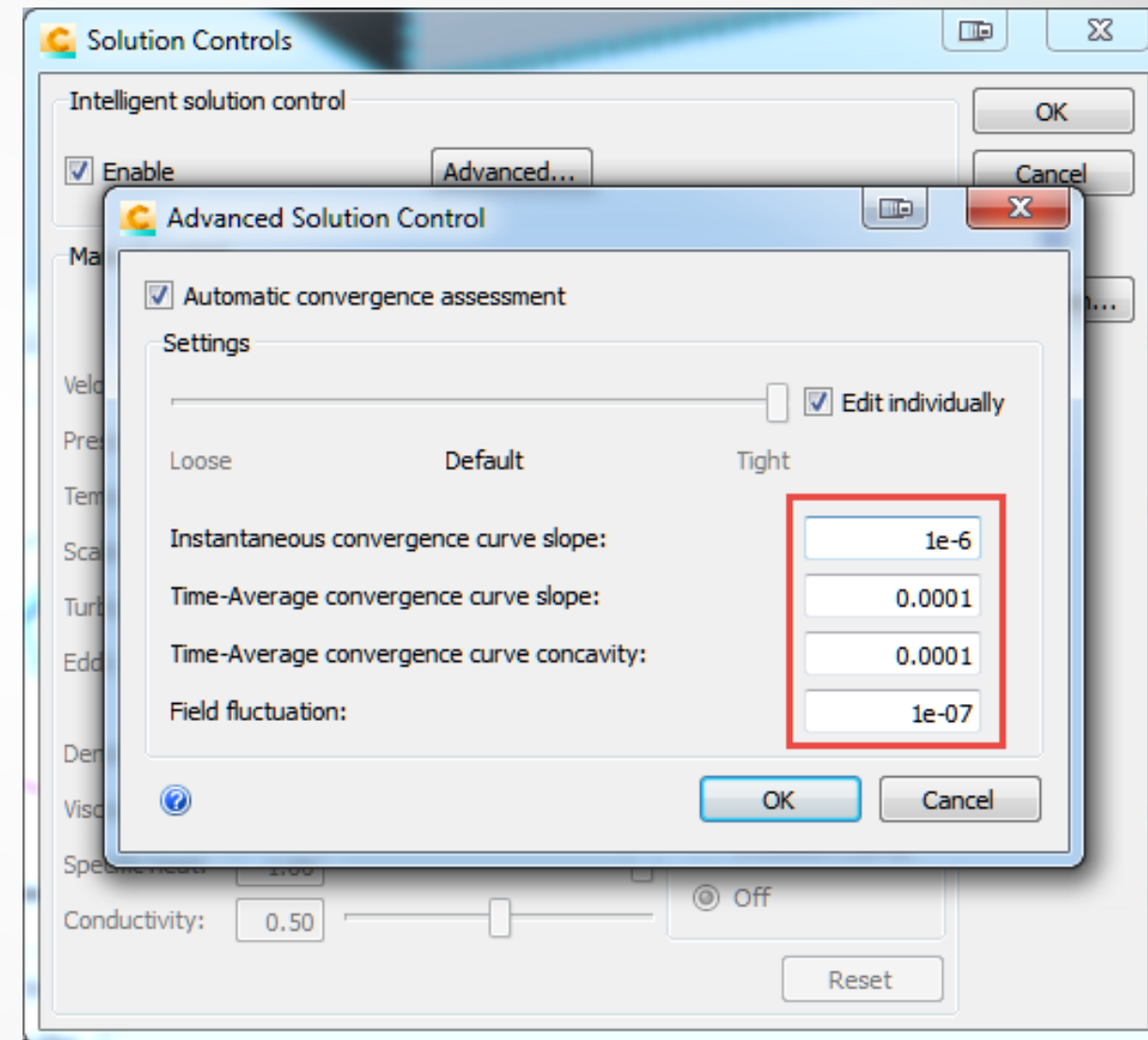
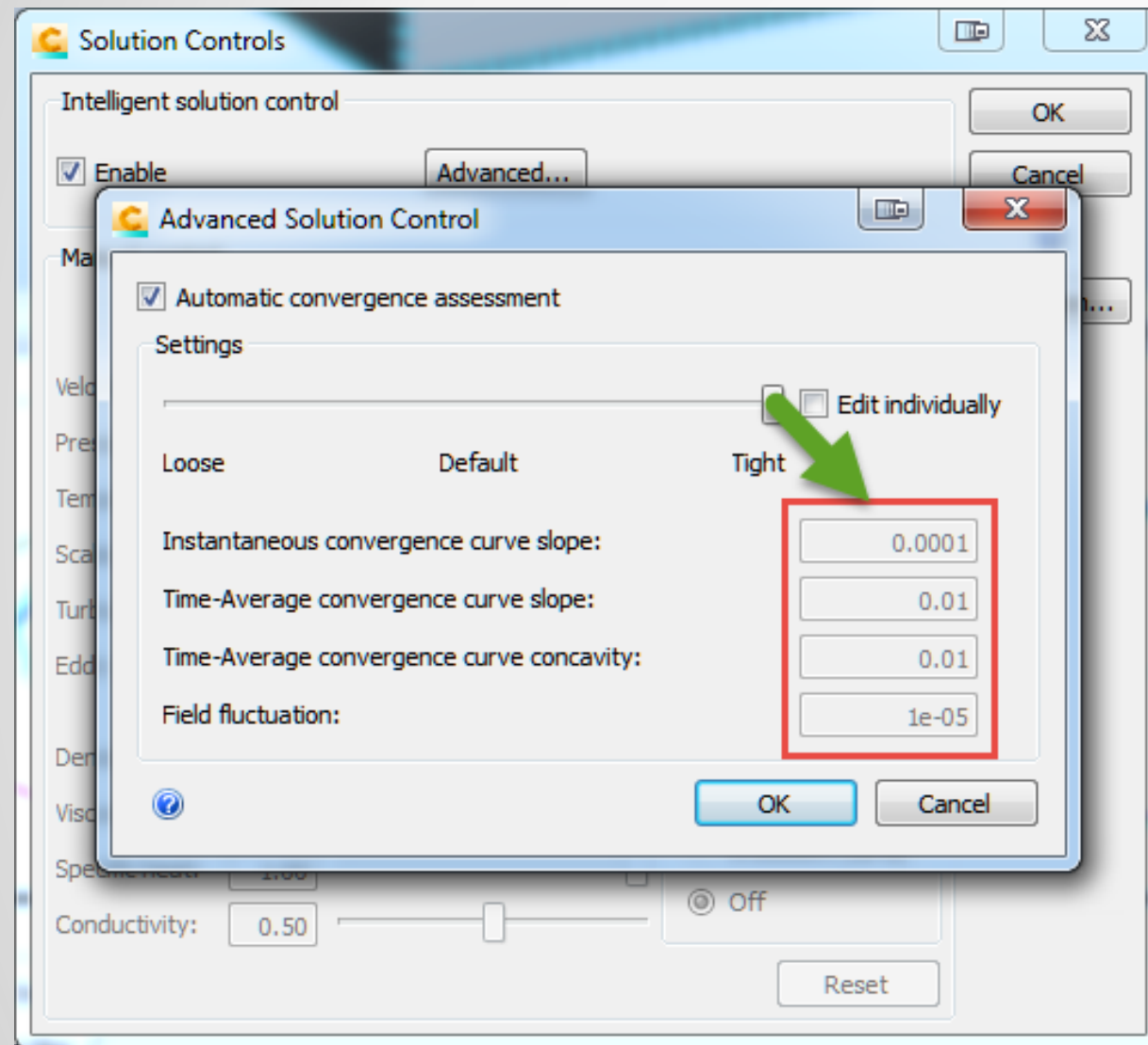
- Mesh

- Same as Complex
Turbulence Structures
- Uniform mesh in wake
zone to capture eddies

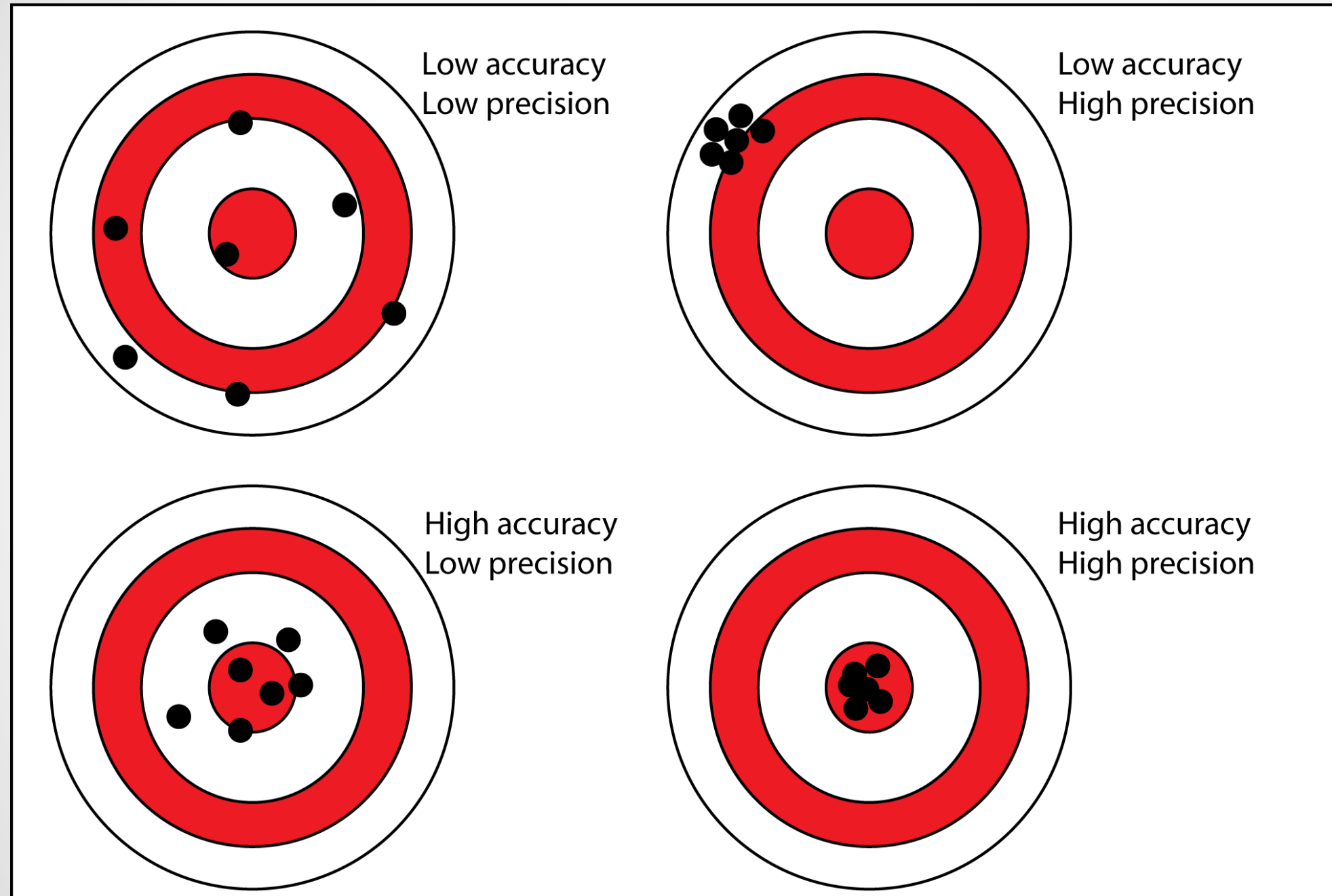
- Solver

- **Required Flag:**
sst_new_iwf 1
- **Transient - Start with
100 timesteps per cycle**

What is 2 Orders of Magnitude for Convergence?



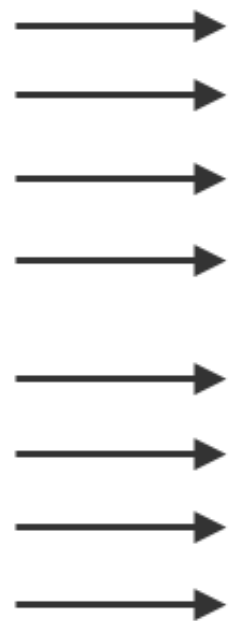
Accuracy vs. Precision for Product Design



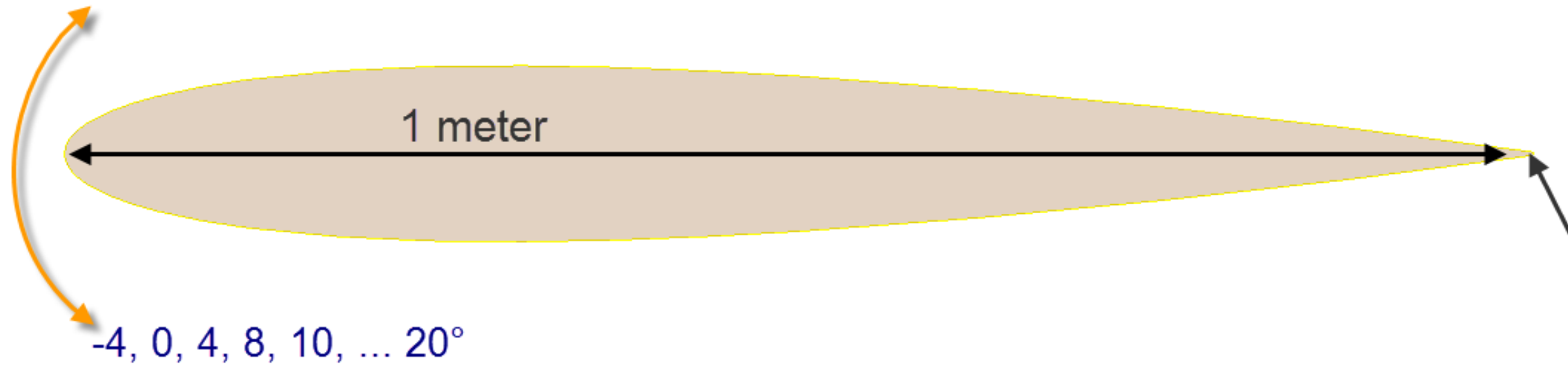
Attached Flow Example

NACA 0012 CFD Setup Strategy

Re: 1.76E6
26.56 m/s
0.08 mach



AIR @ STP



What is the NACA 0012 CFD setup strategy?

- **Advanced Turbulence**

- Attached Flow Configuration
SST k-omega RC (Hellsten)

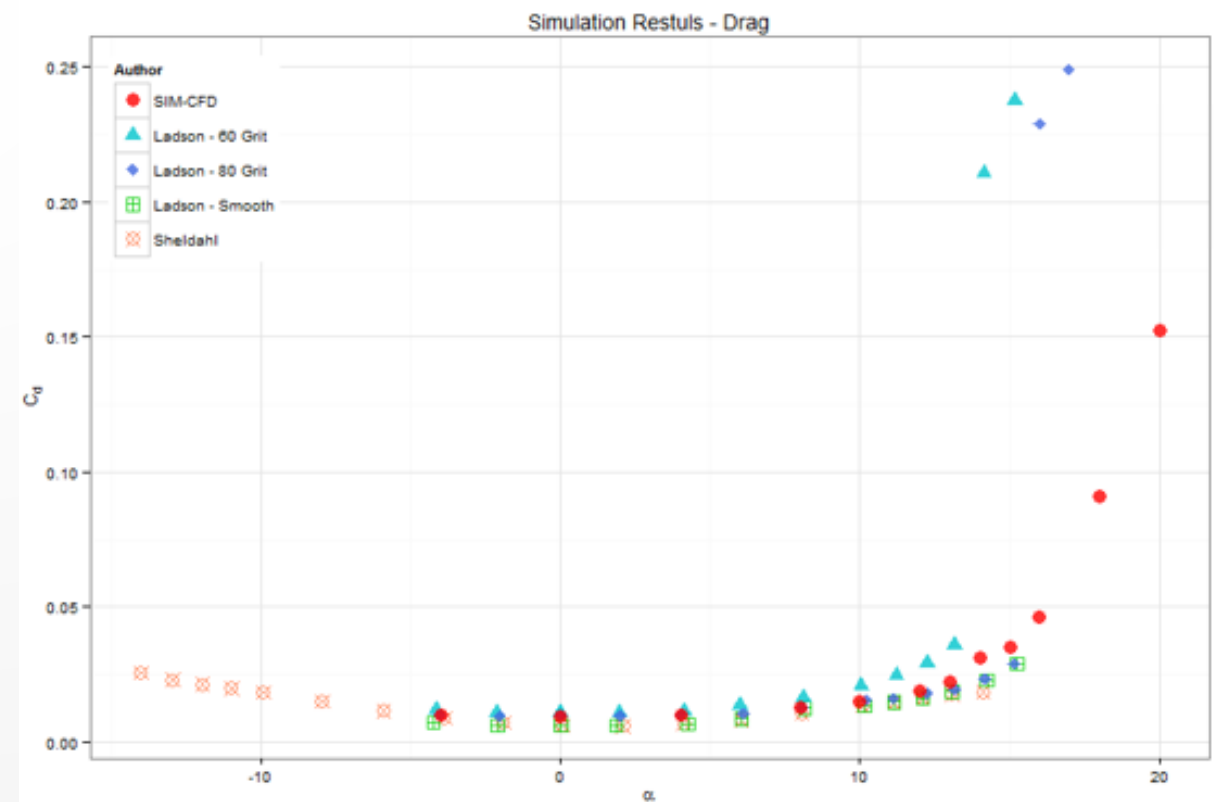
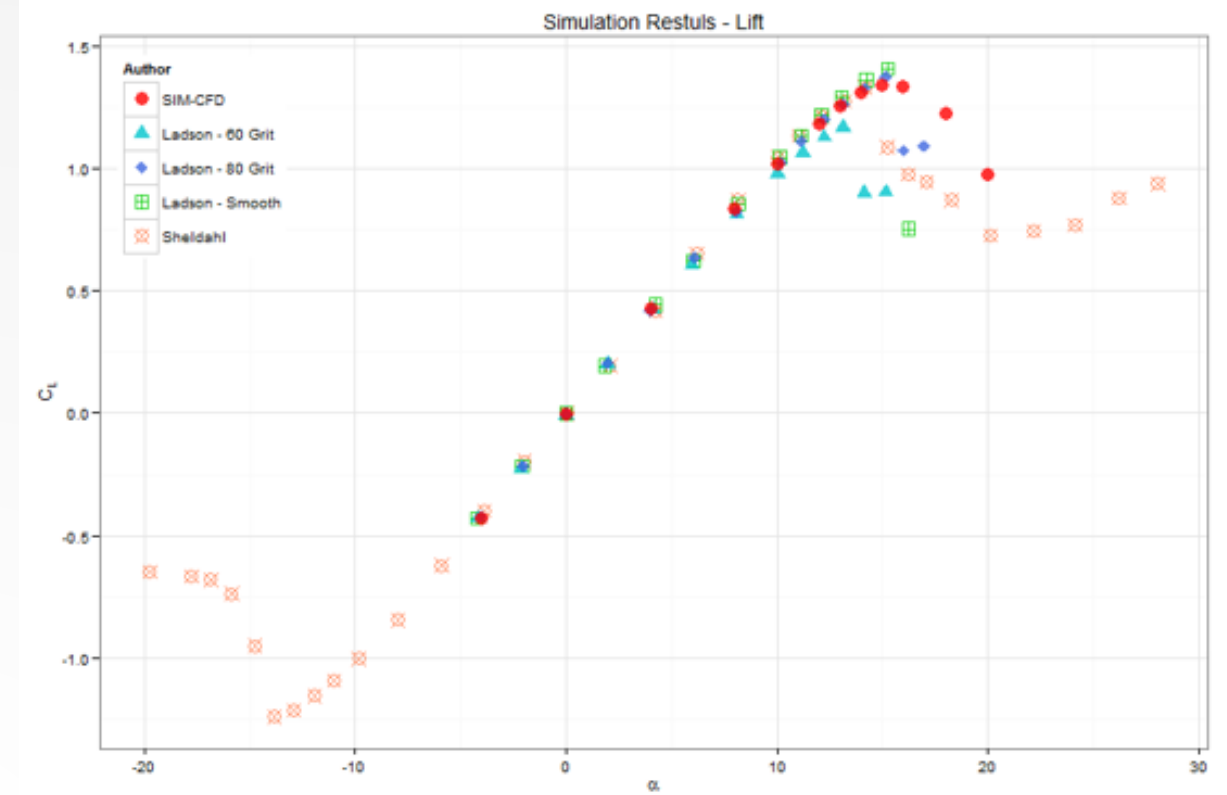
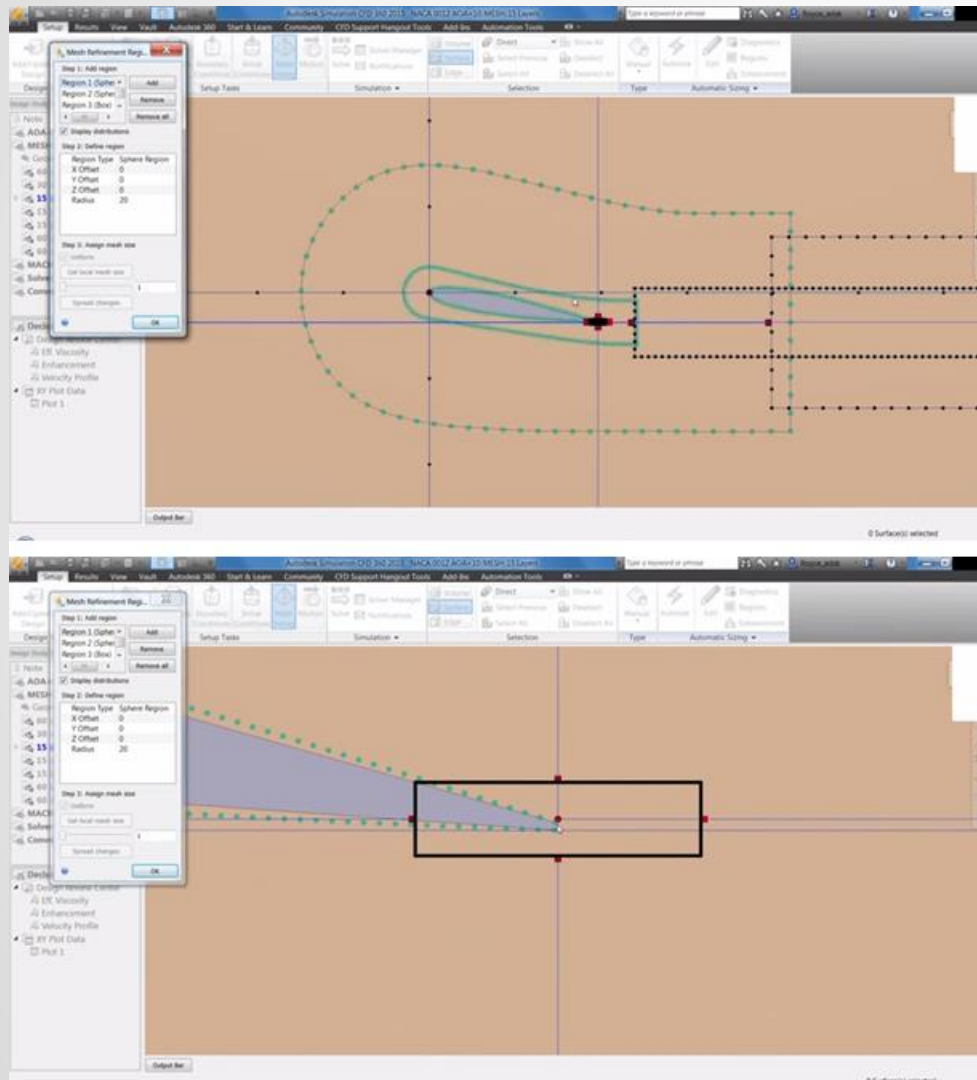
- **Updated Wall Calculator**

- resid_bdry_force_calc 1

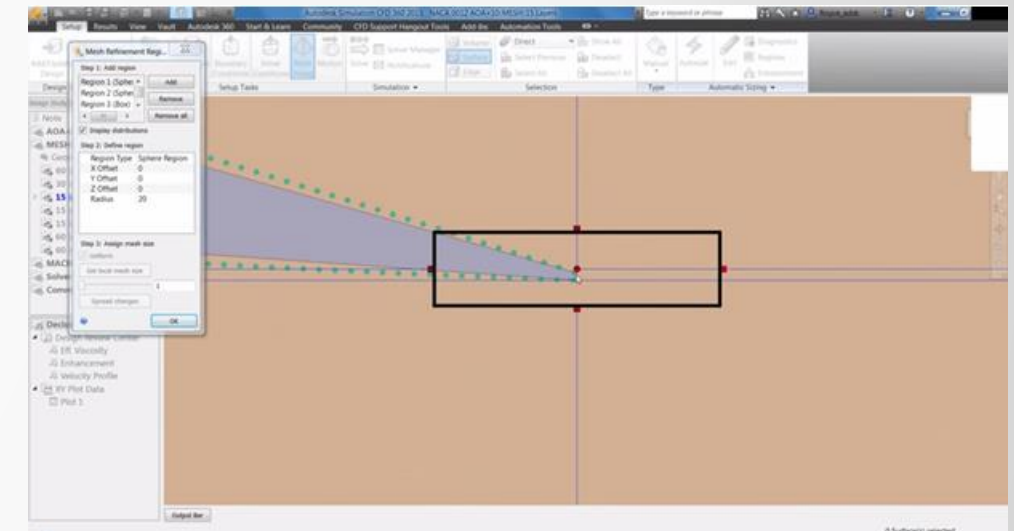
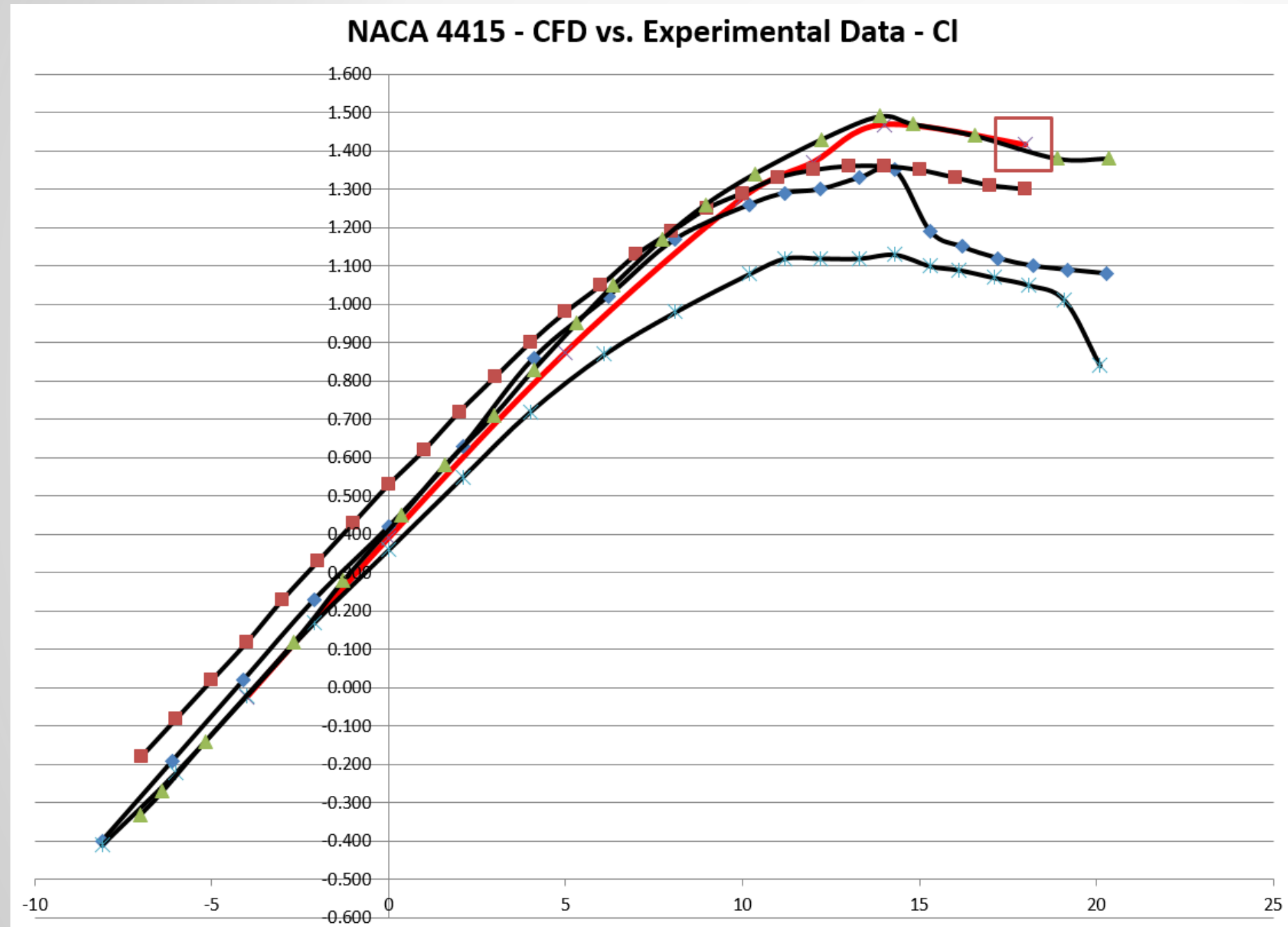
- **Meshing**

- 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

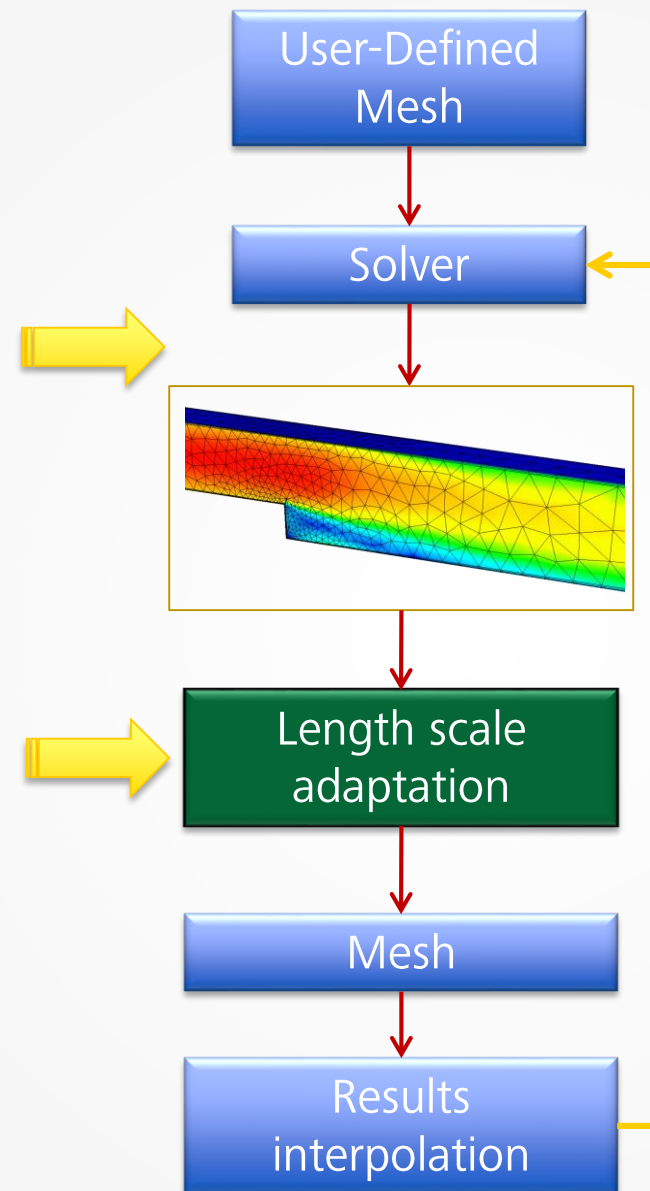
Mesh Adaptation Workflow

Generates *mesh independence indicators*:

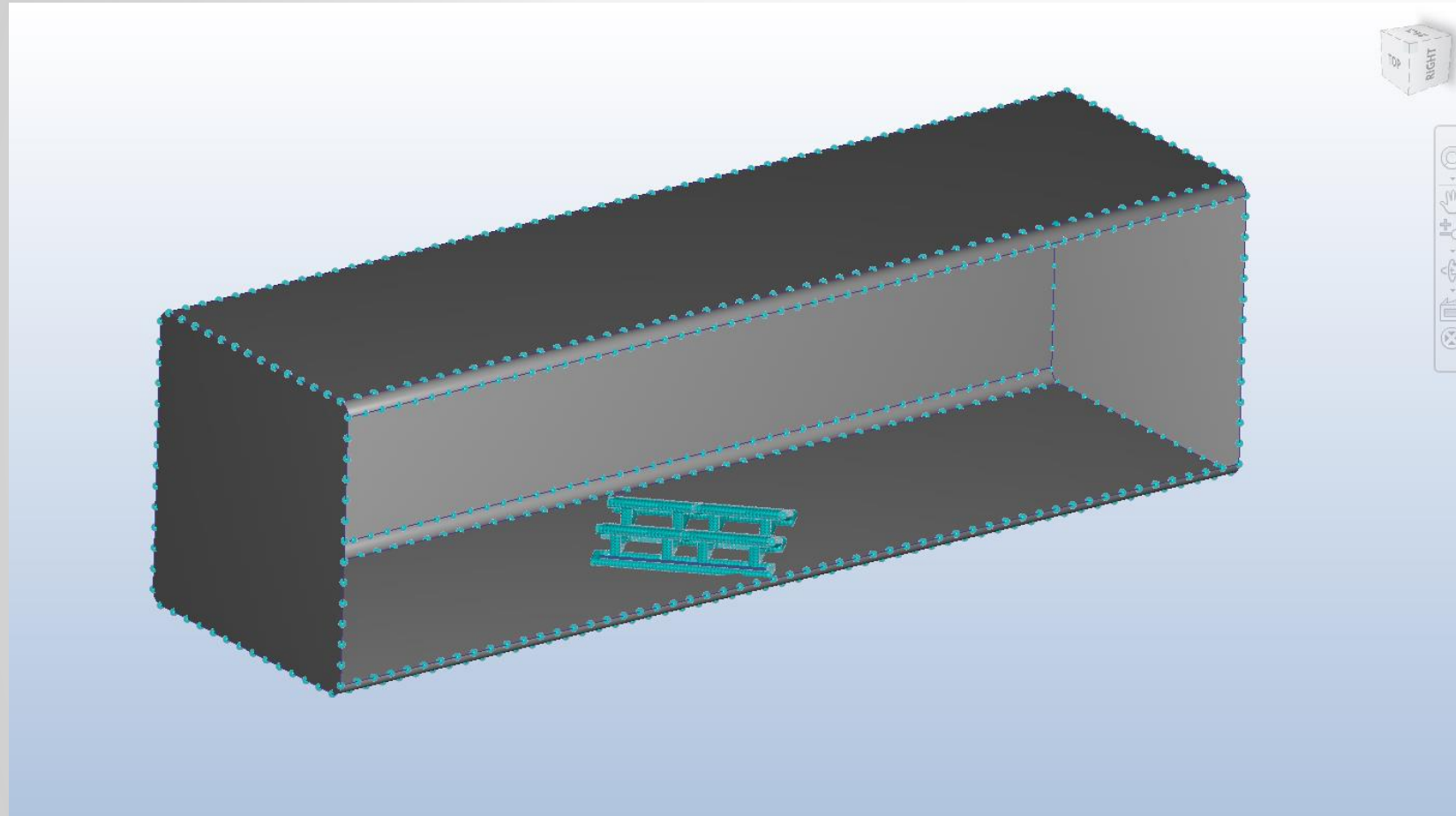
- ✓ These are *relative* rather than *absolute* indicators.
- ✓ Require that solutions be fully converged

Adaptation via regeneration:

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

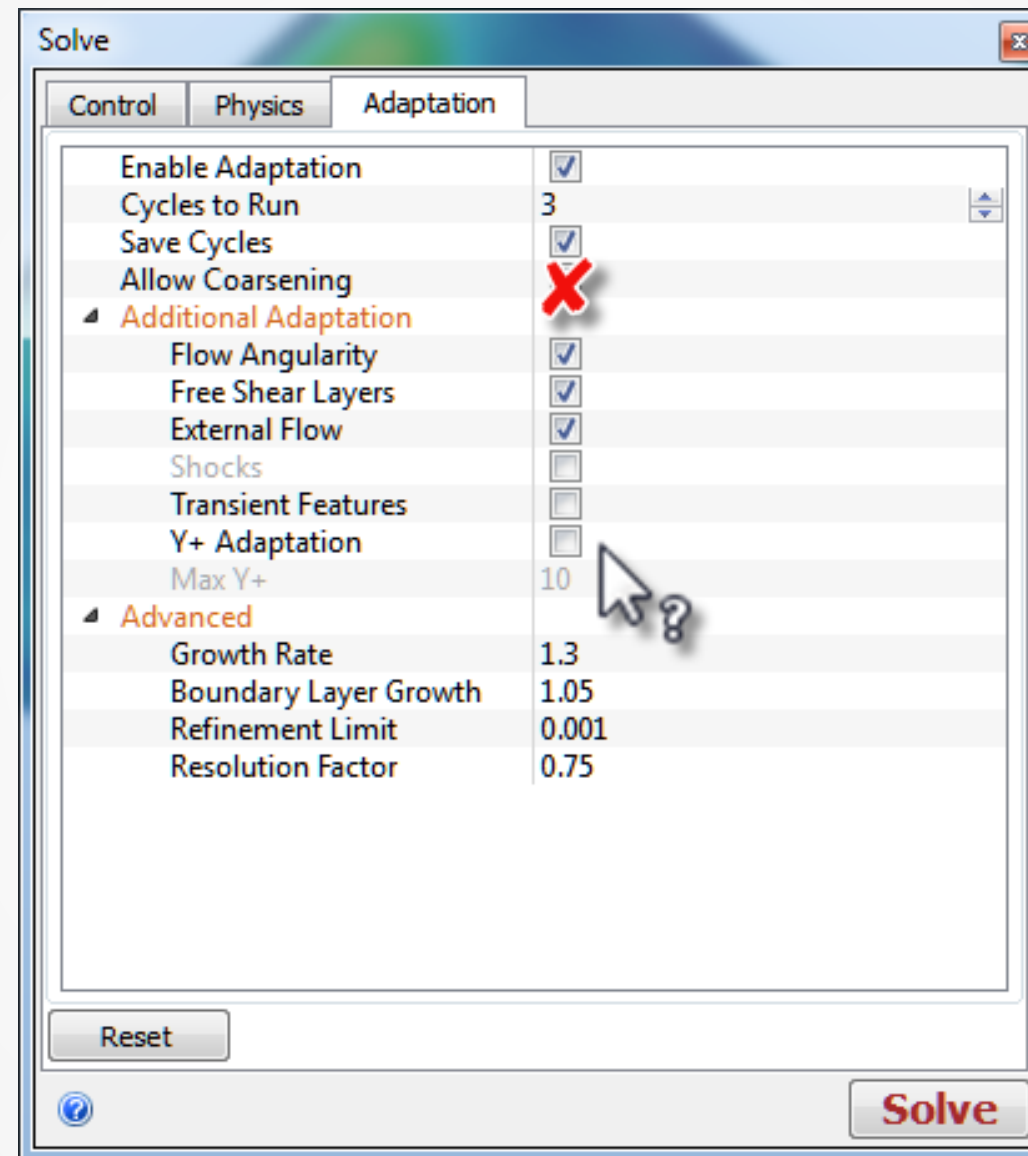


User-Defined Mesh



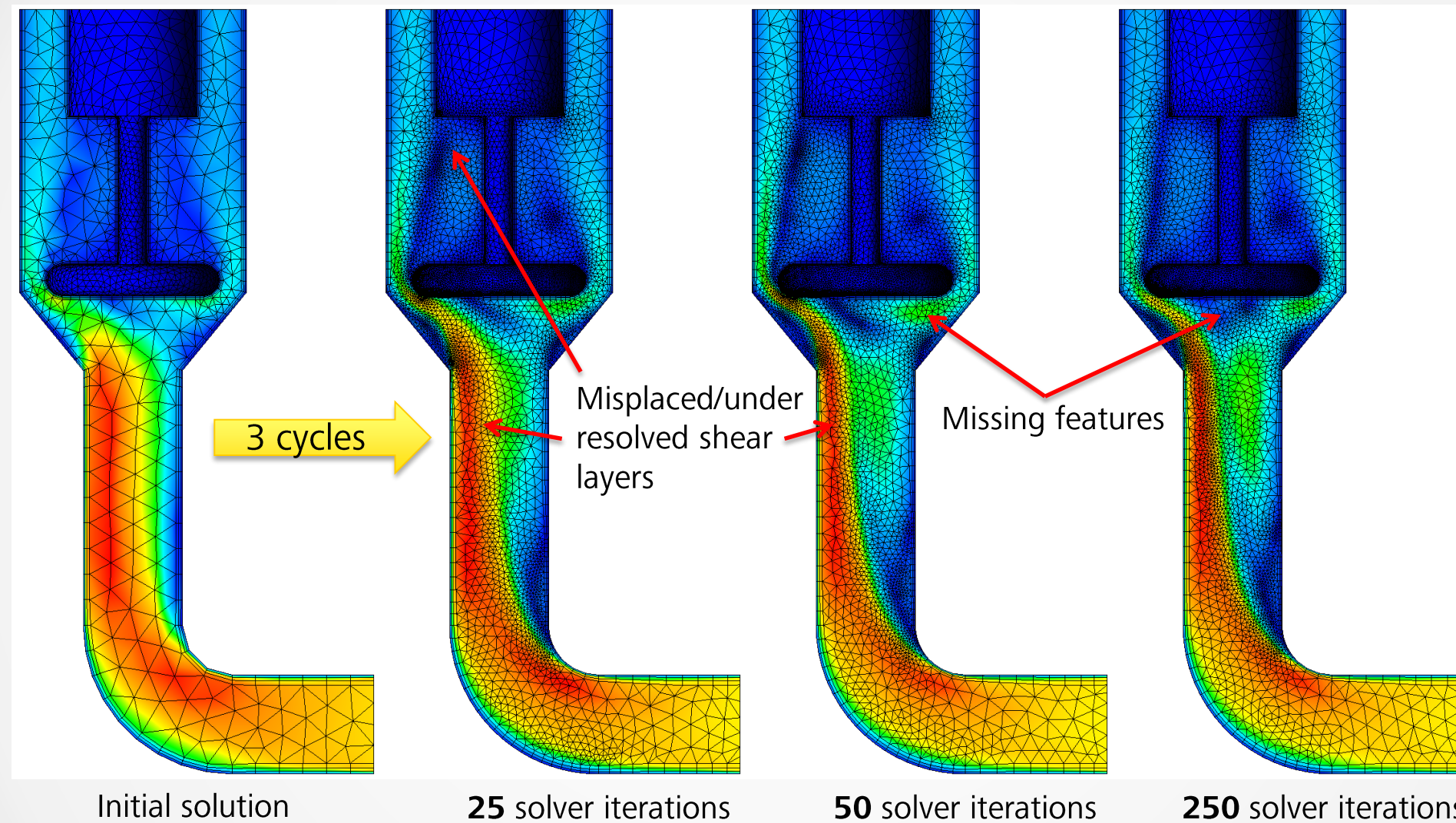
- **Manual Mesh**
- **Suppress Solids**
- Large size on external domain for a fast first cycle
- Tighter surface mesh compensate for air domain size
- Manual Mesh 4:1 rule?

Mesh Adaptation Setting Panel

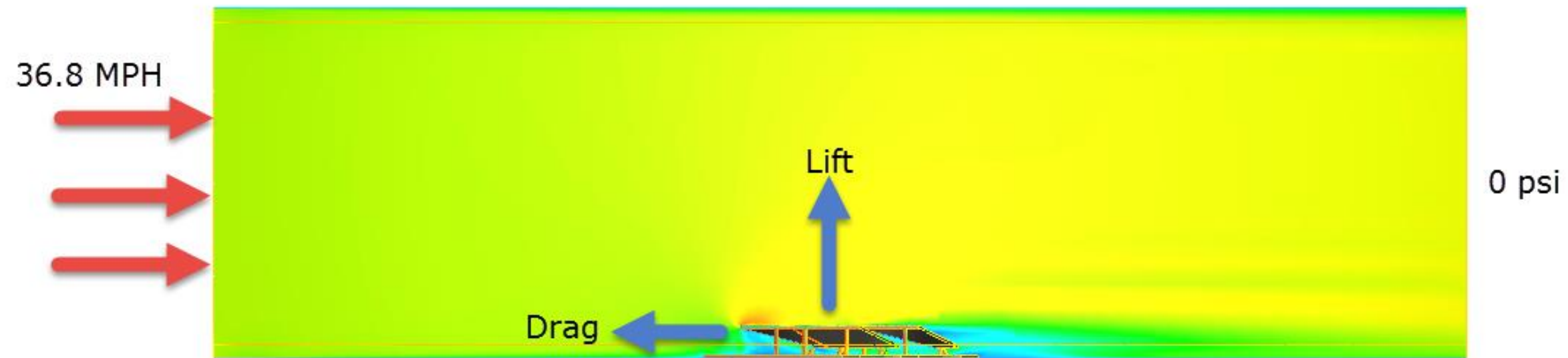


The Importance of Solution Convergence

Leverage 2nd Order Manual Automatic Convergence Assessment



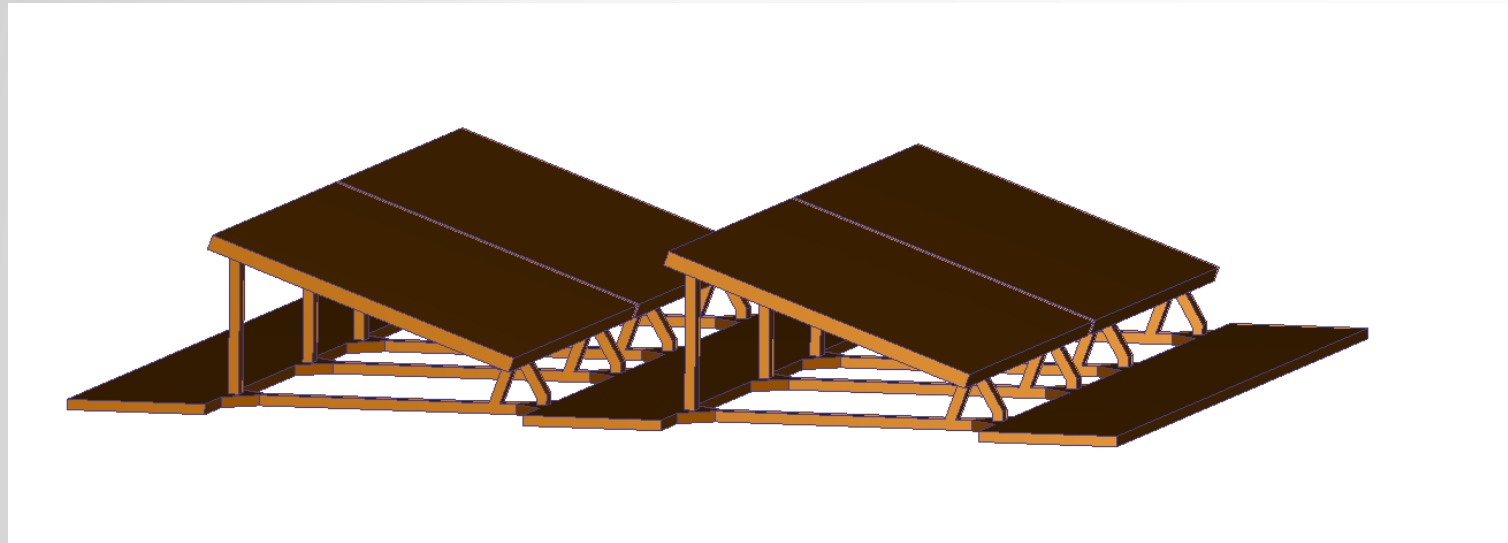
Example: Flow around 2x2 Solar Panel Array



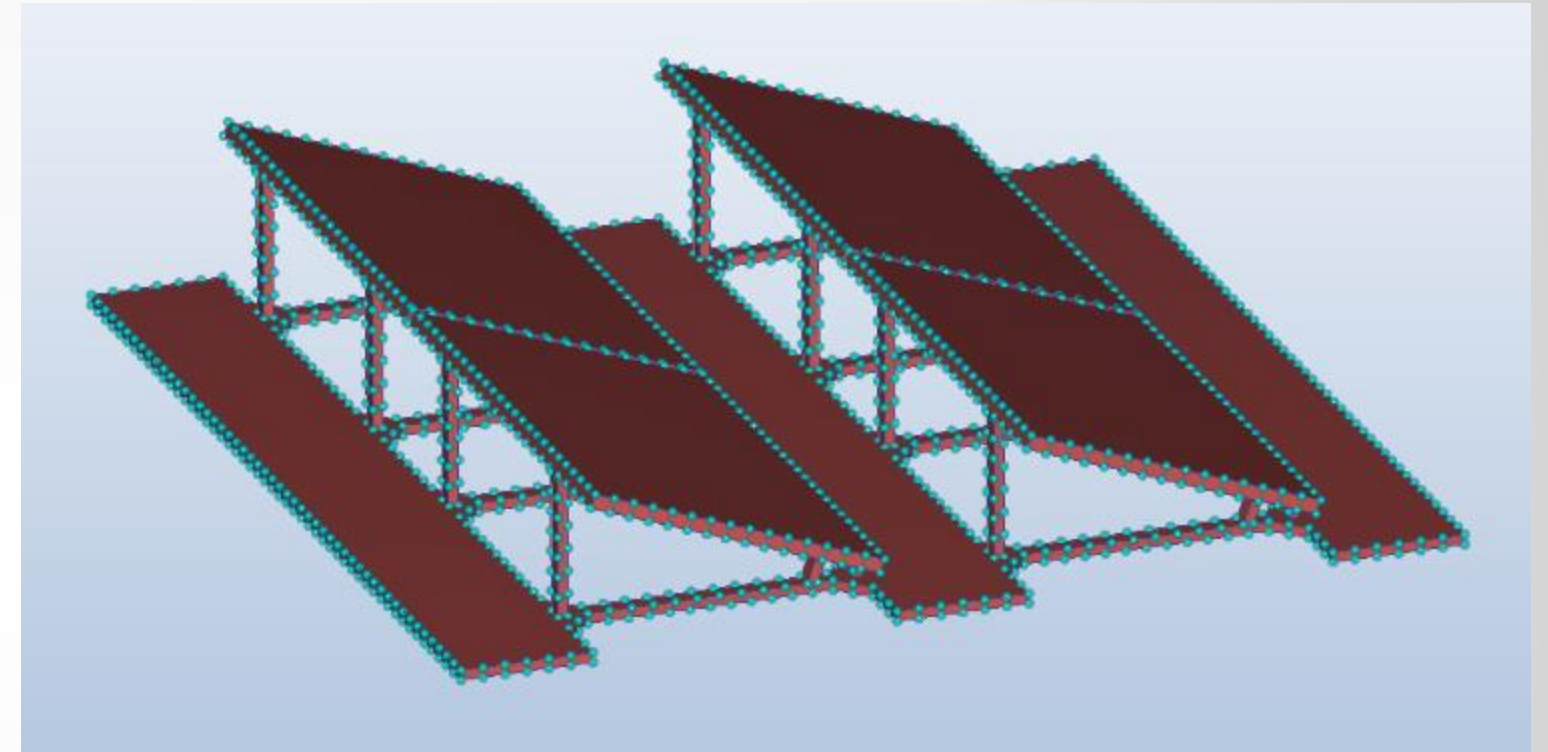
Lift: 0.9 lbs

Drag: 0.3 lbs

Solar Panel Array

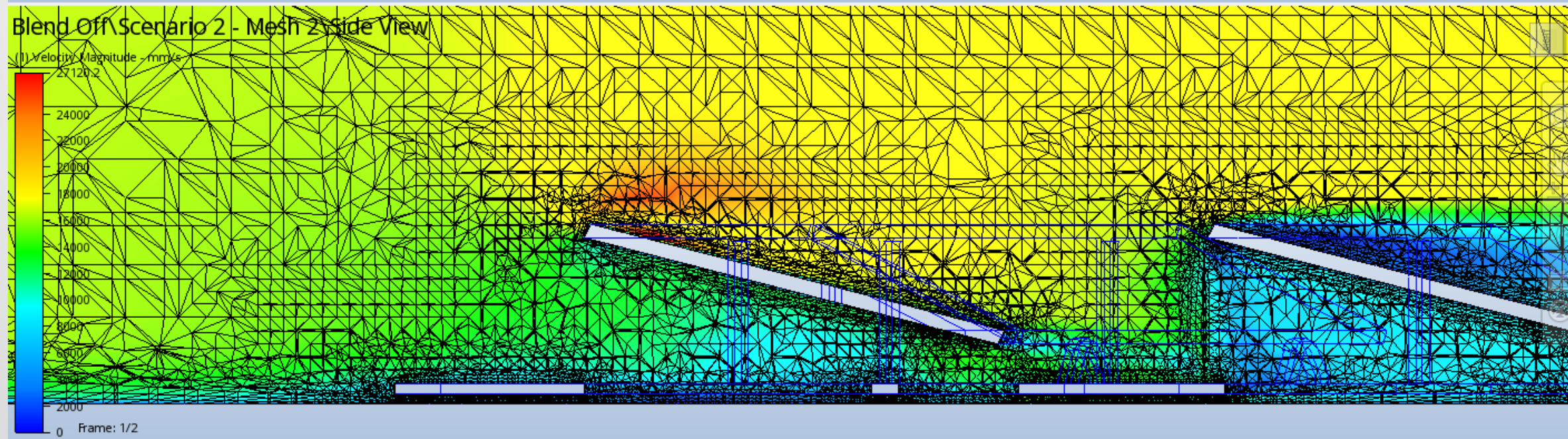
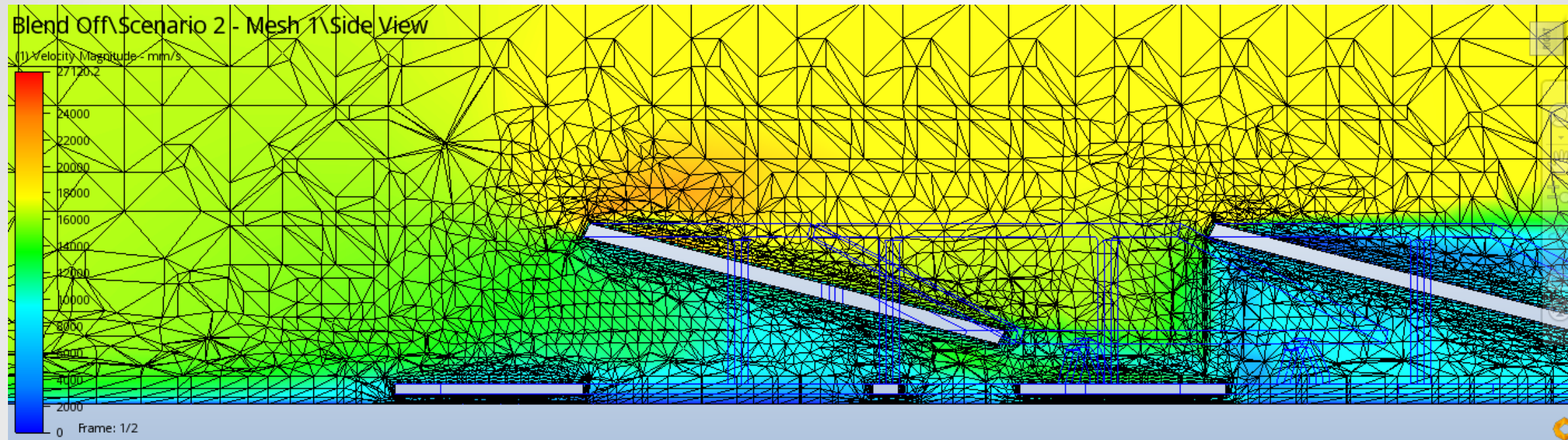


Size: 10" x 10"
Panels: 20°

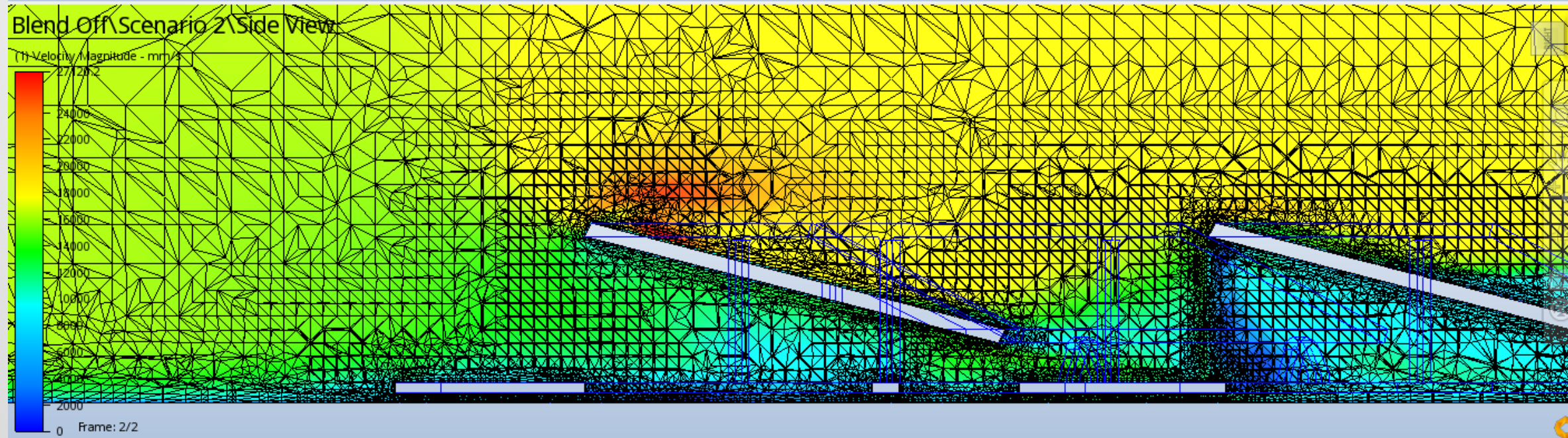
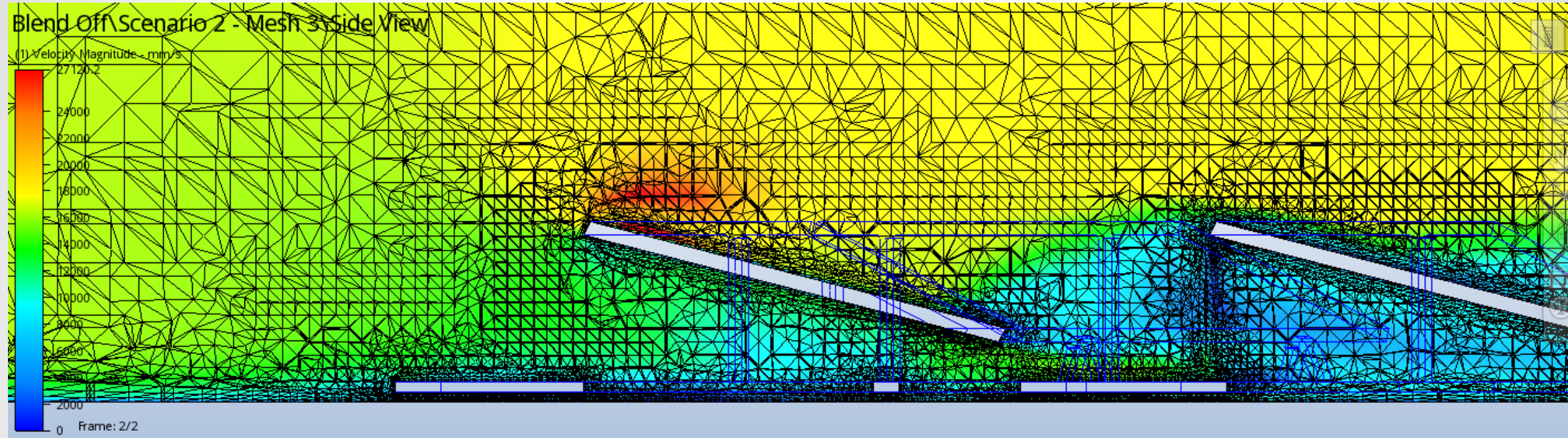


Surface: 5 mm
Air Volume: 30 mm
Attached Flow Strategy w/3 Layers
No Y+ Adaptation

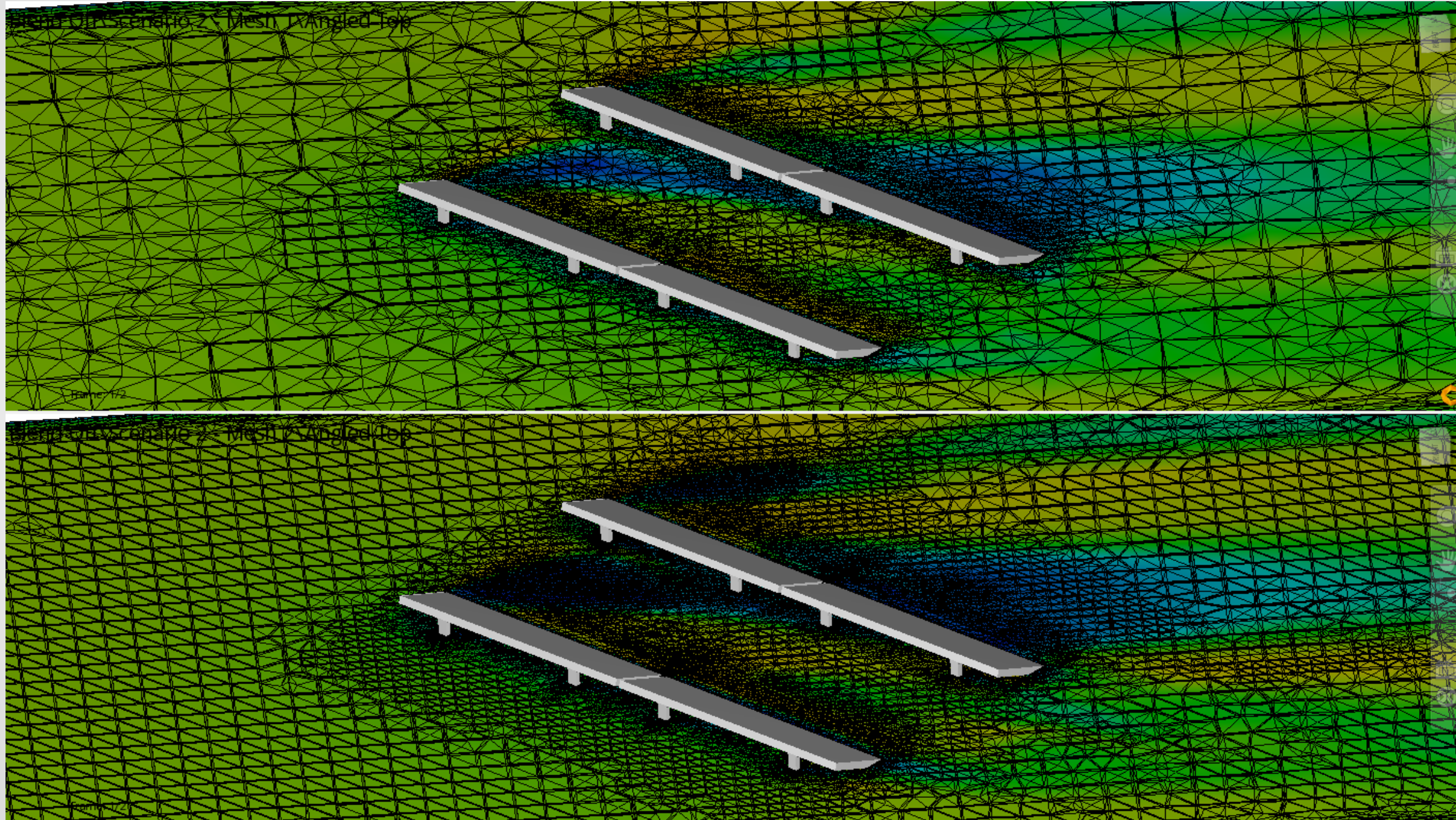
Side View of Mesh Adaptation (1 & 2)



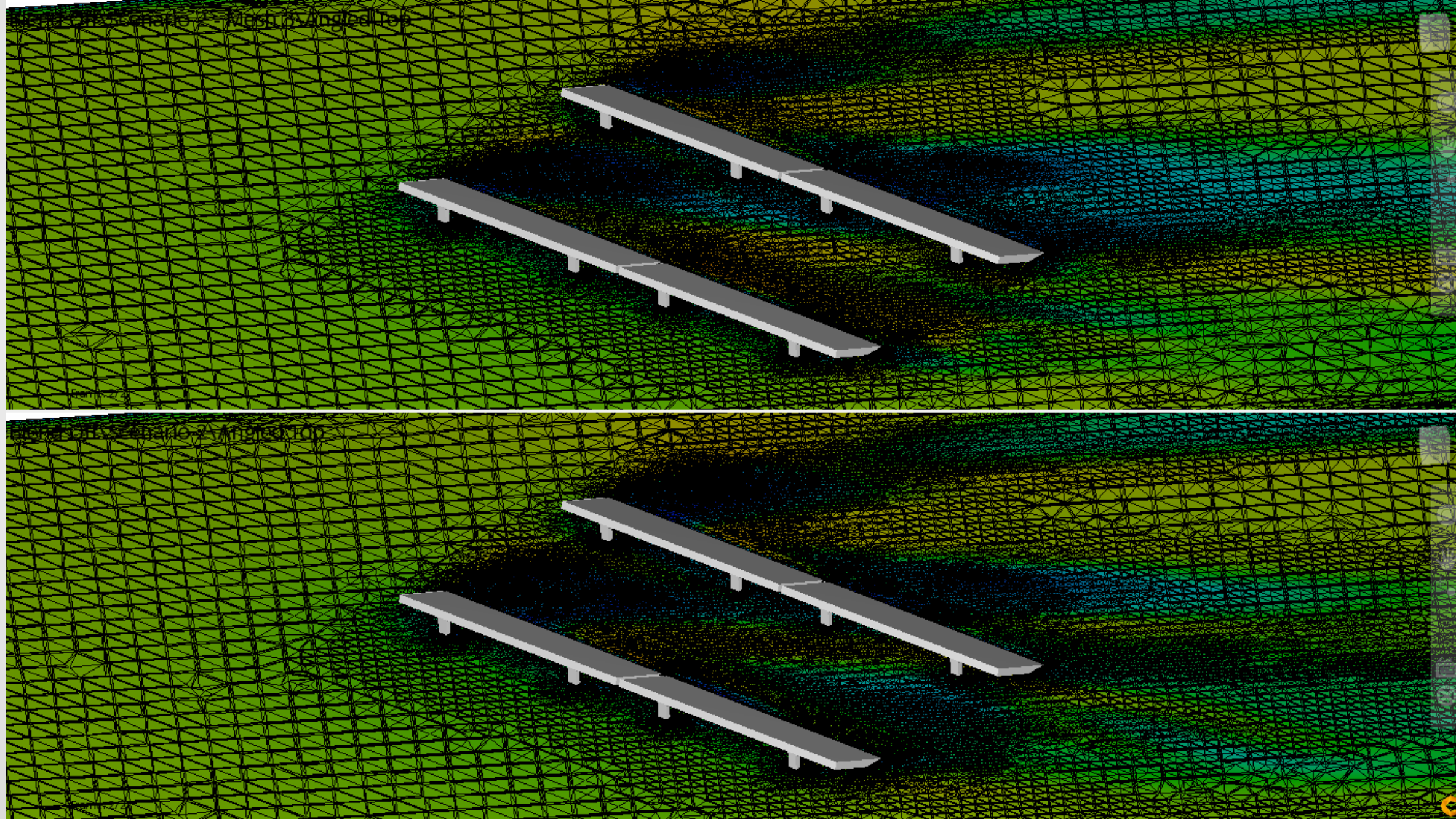
Side View of Mesh Adaptation (3 & 4)



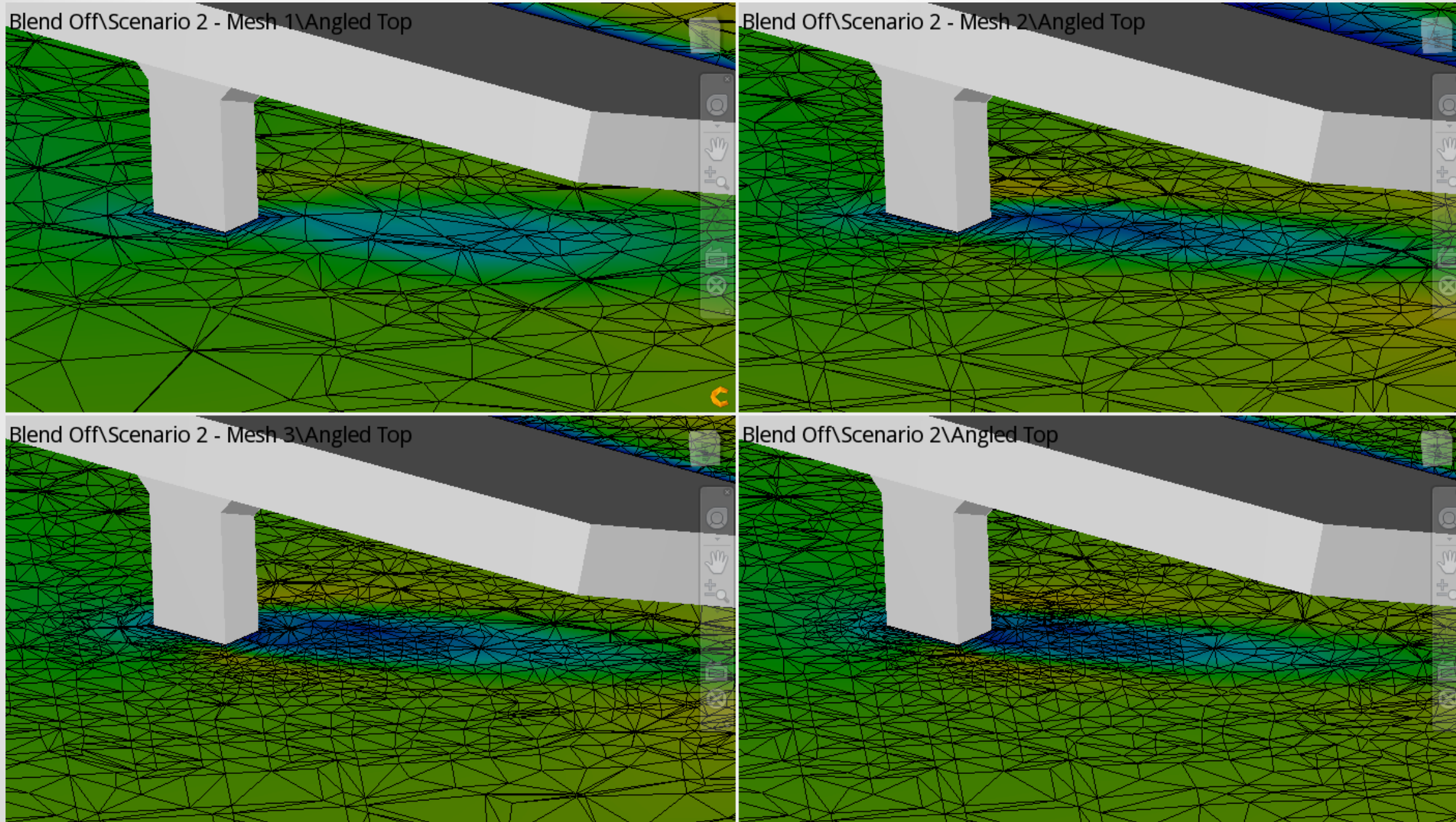
Top View of Mesh Adaptation (1 & 2)



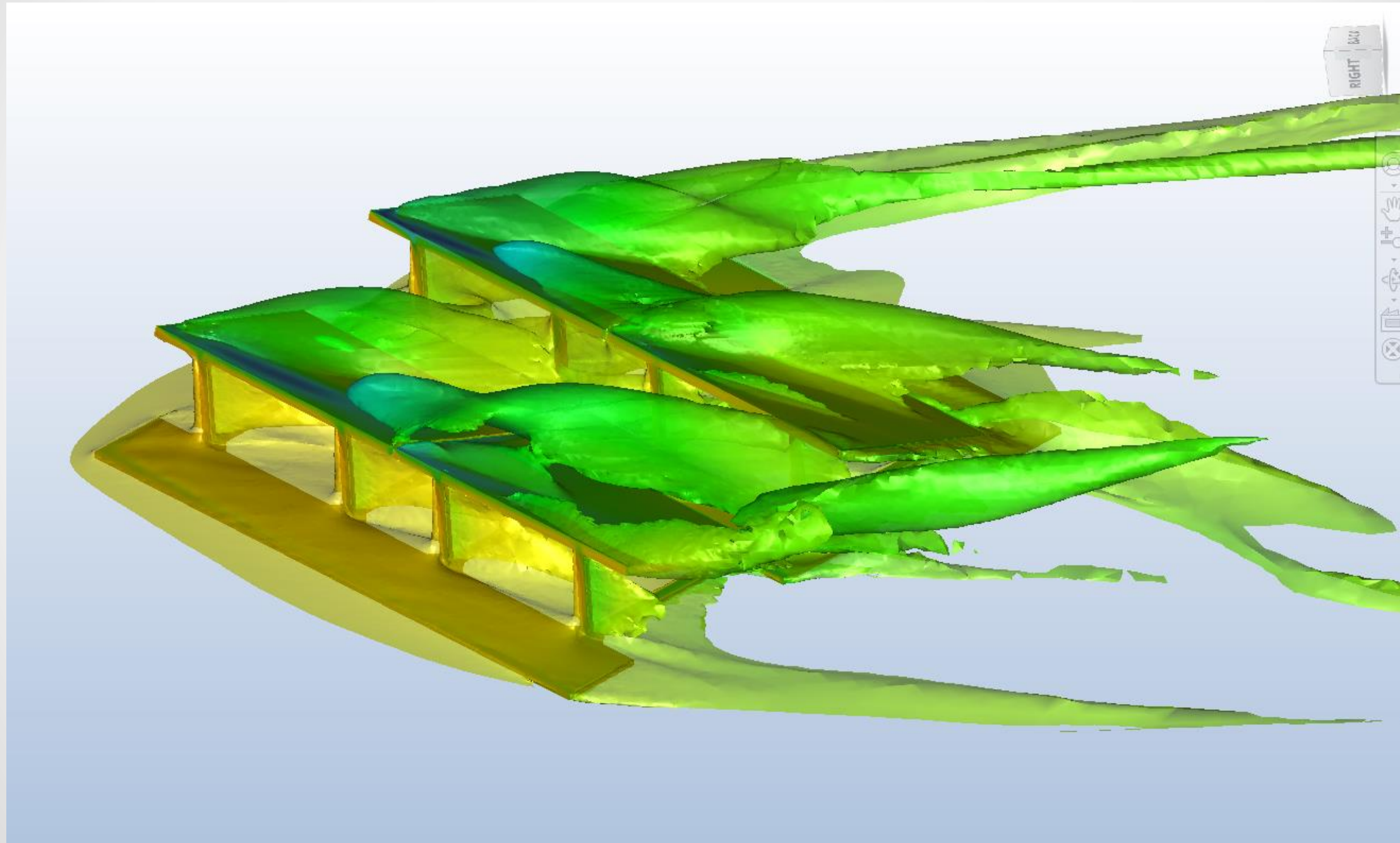
Top View of Mesh Adaptation (3 & 4)



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

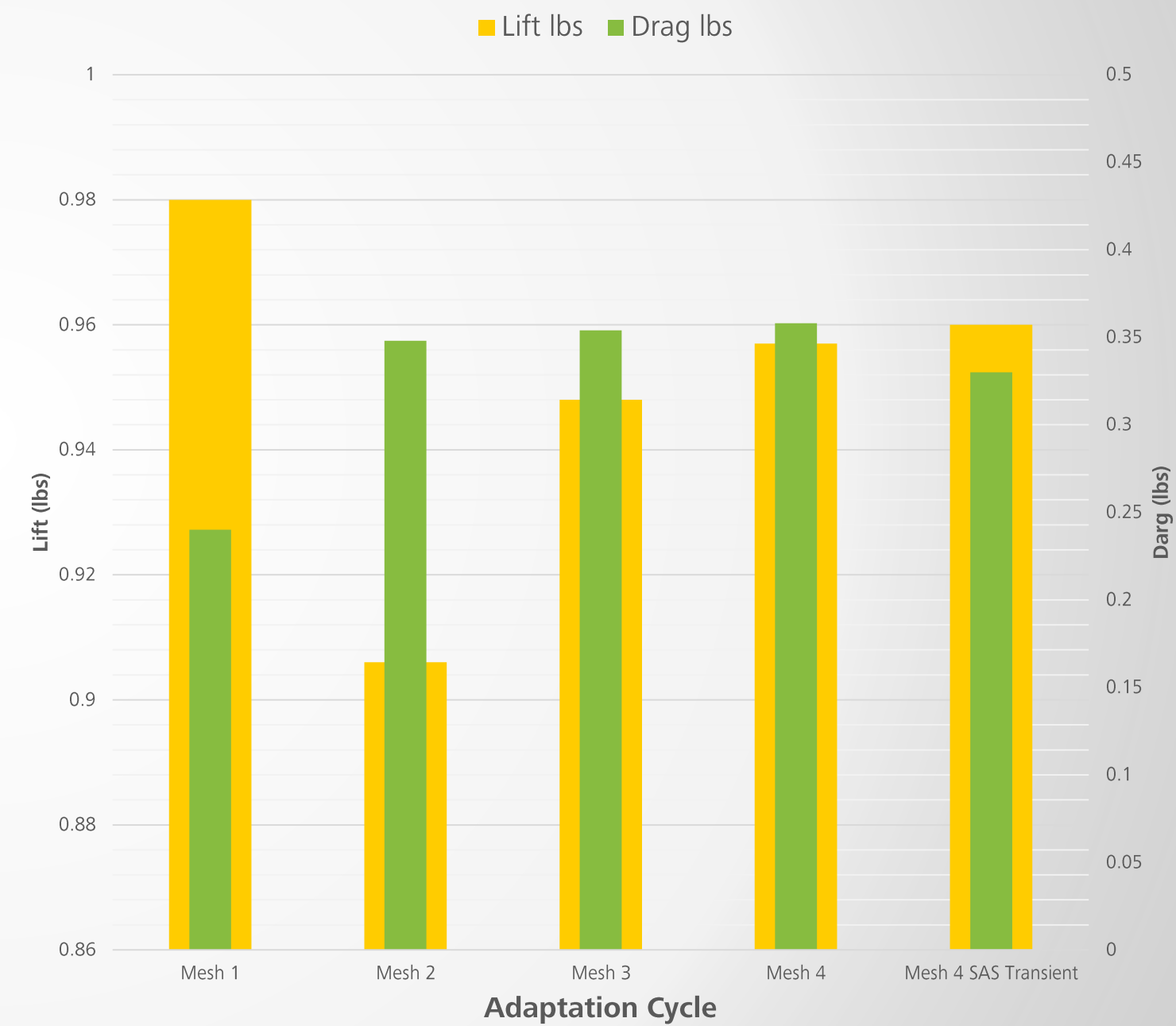
Complex Turbulence Structures

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

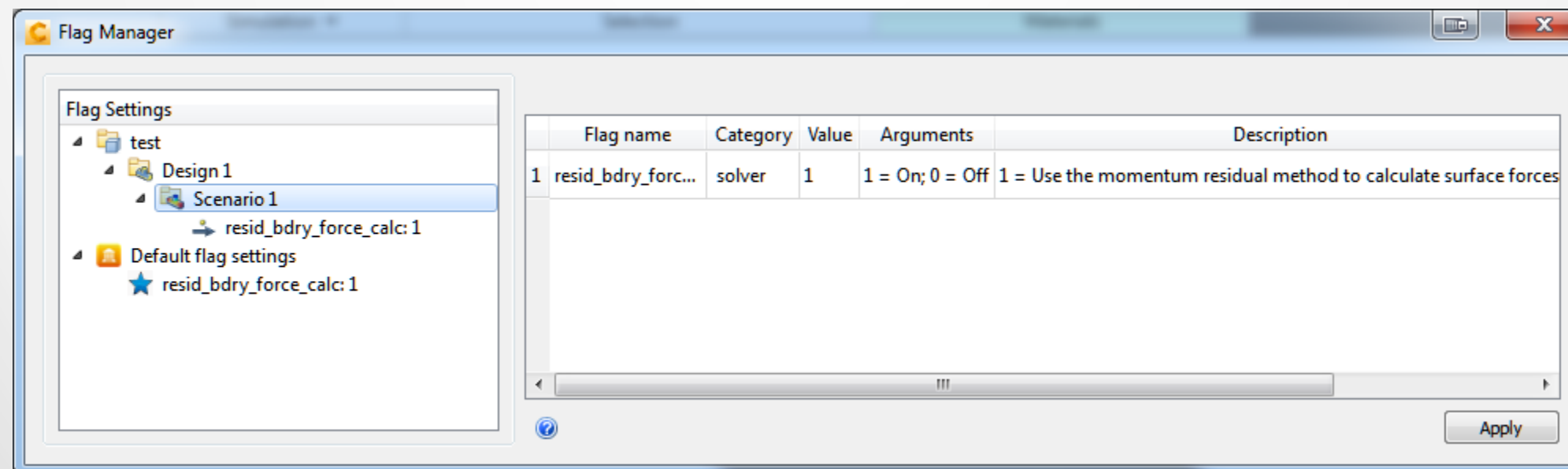
Newer Wall Calculator algorithm

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

Wall Results

Selection and Result **Output**

Model entity selection

☐ Volume ☐ Surface ☒ Edge

Group operation:

1

☒ Force ☐ Cutoff pressure ☐ Pressure ☐ Temperature ☐ Heat flux ☐ Film coefficient ☐ Torque

Newton * Pa * Celsius * W/m2/K * N-m *

Ref. temperature: 300 Celsius *

☒ Use near-wall temperatures

Torque axis

Point on axis 0.4445, 0, 0

Direction 0,0,1

Calculate

Wall Results

Selection and Result **Output**

BOUNDARY ID, 1

Area, 0.119681, m²

sum FX, 1448.75, Newton

sum FY, 1516.52, Newton

sum FZ, 0, Newton

Center of Force about X-Axis (Y-Z), 0, 0, m

Center of Force about Y-Axis (X-Z), 0, 0, m

Center of Force about Z-Axis (X-Y), 0.00821884, 0.0

Summary

Total area, 0.119681, m²

TOTAL FX, 1448.75, Newton

TOTAL FY, 1516.52, Newton

TOTAL FZ, 0, Newton

Center of Force about X-Axis (Y-Z), 0, 0, m

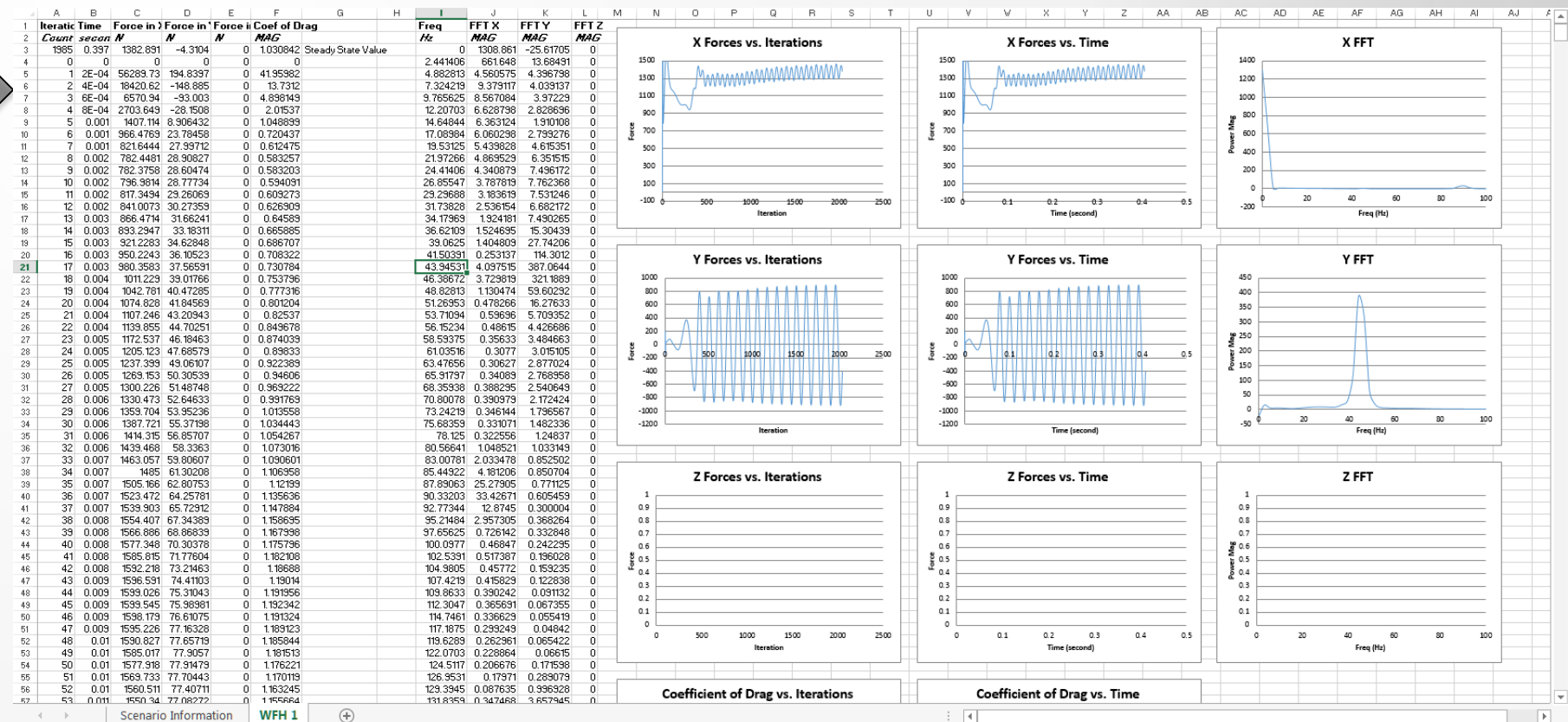
Center of Force about Y-Axis (X-Z), 0, 0, m

Center of Force about Z-Axis (X-Y), 0.00821884, 0.0

Write to file...

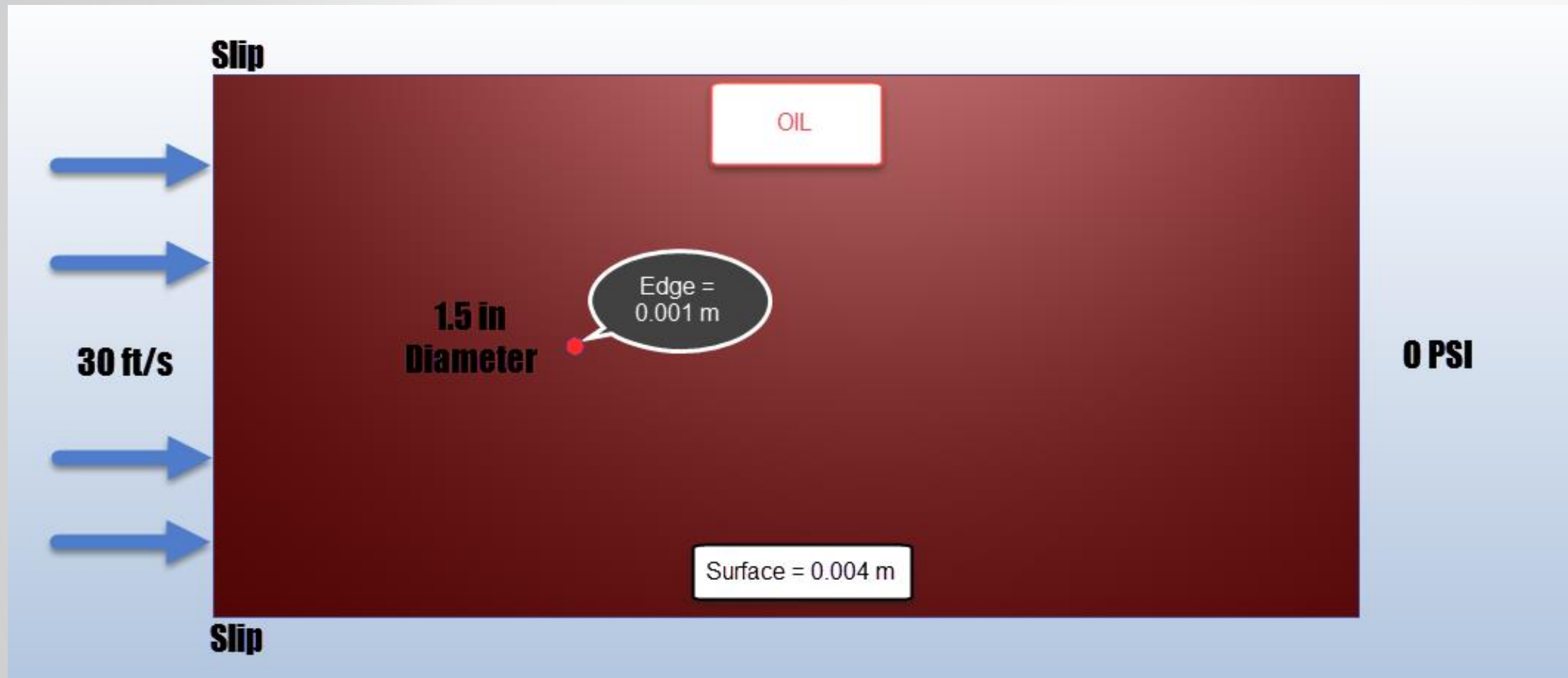
View file...

- 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

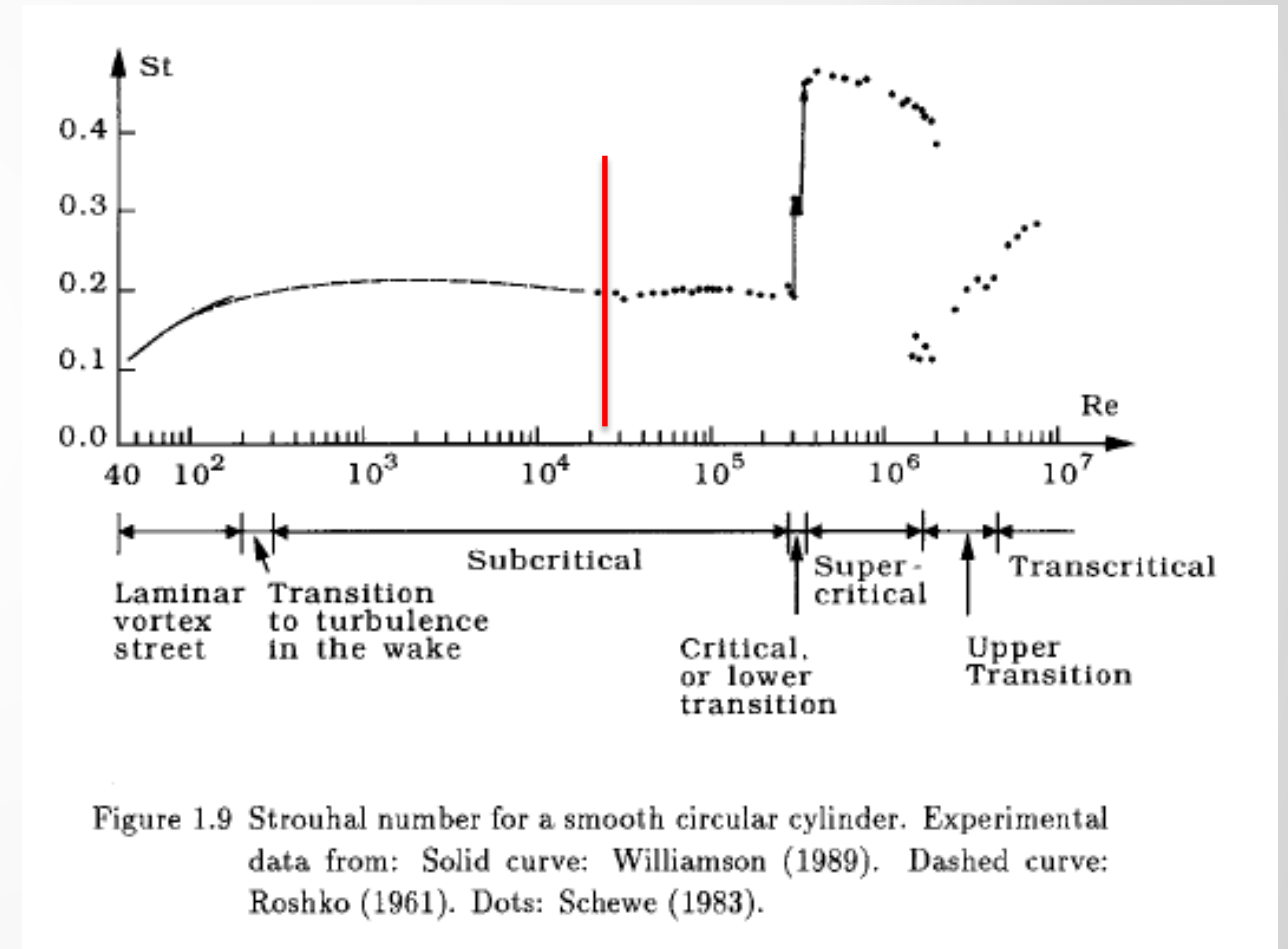
- sst_new_iwf 1
- **Transient - Start with 100 timesteps per cycle**
 - 0.0002 s
- ISC = Off

Flow over a Cylinder – Expected Results

St = 0.19

Frequency = 45.94 Hz

Drag = 1619.55 N



Demo of Extracting Results

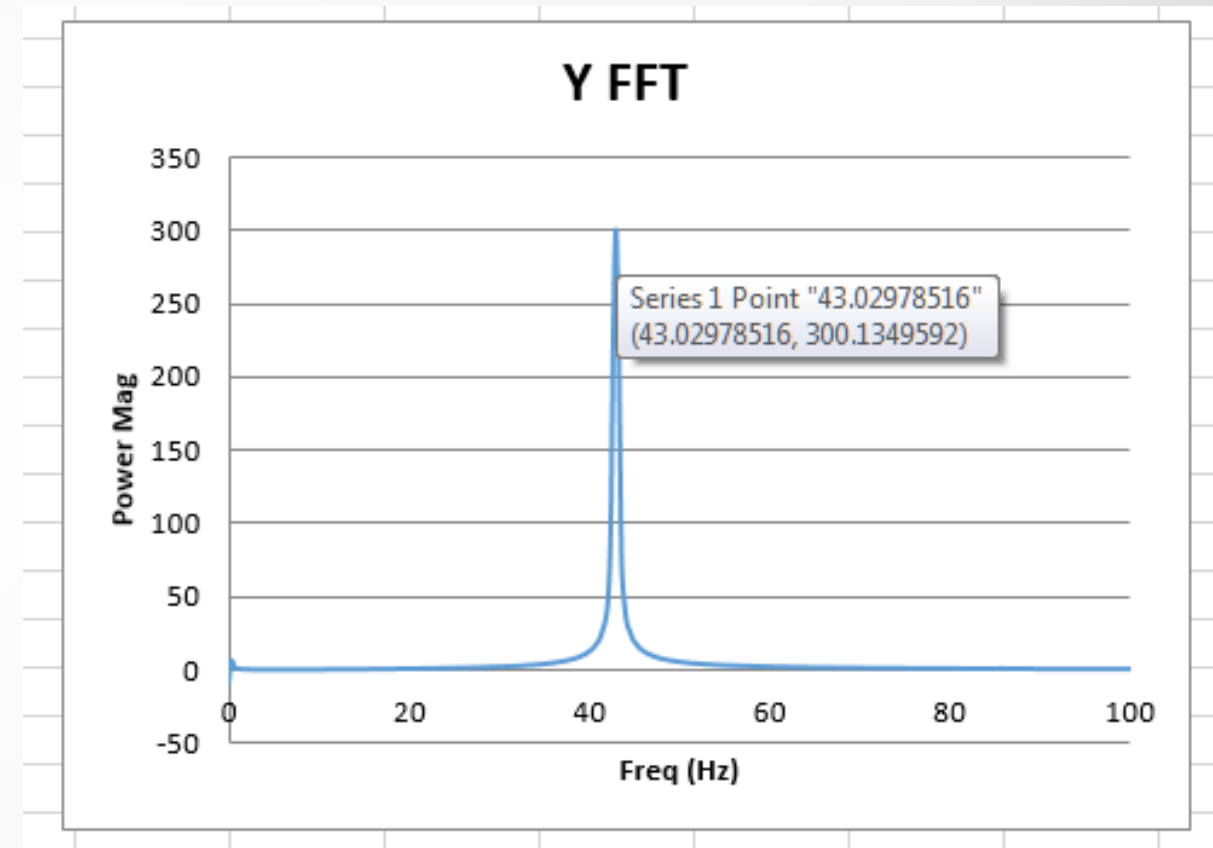
Flow over a Cylinder – Results Summary

Strouhal Number Setup

St = 0.18

Frequency = 43.03 Hz (6%)

Drag = 1433 N (12%)



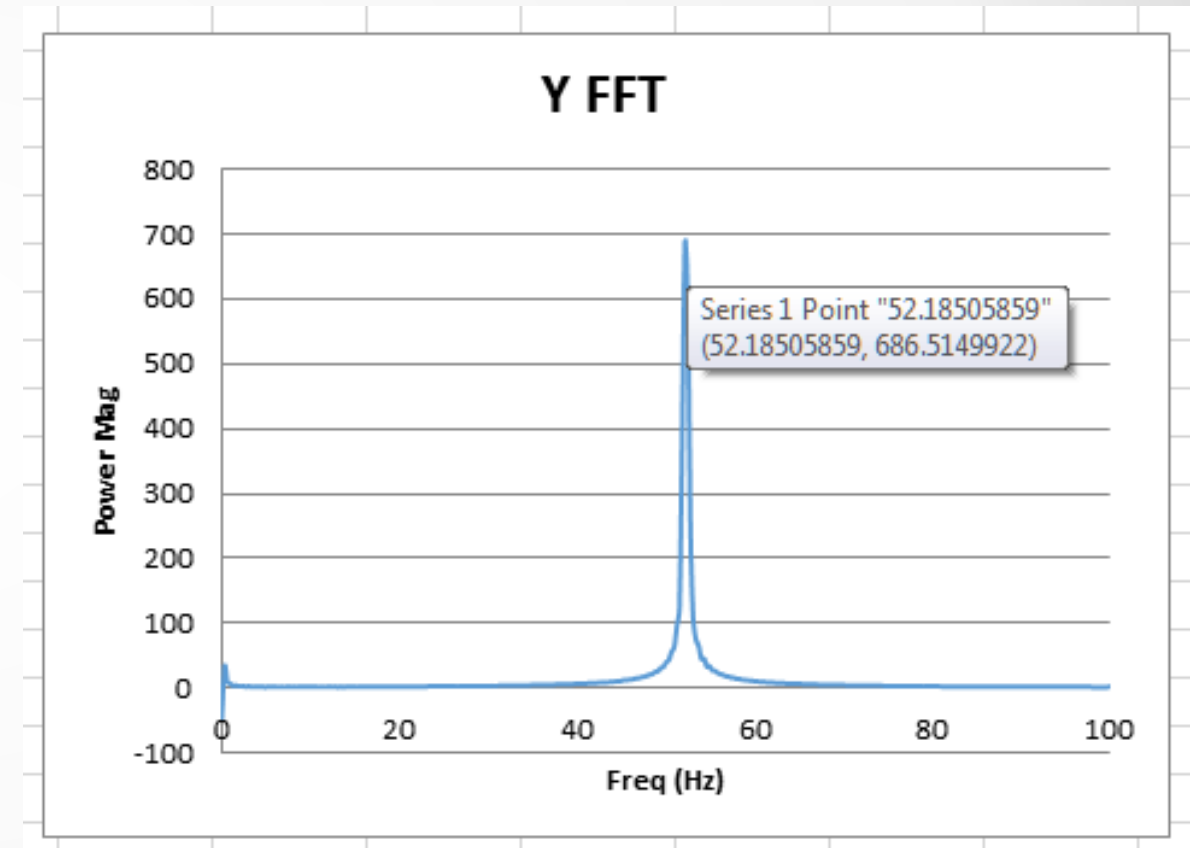
Flow over a Cylinder – Results Summary

Attached Flow Setup

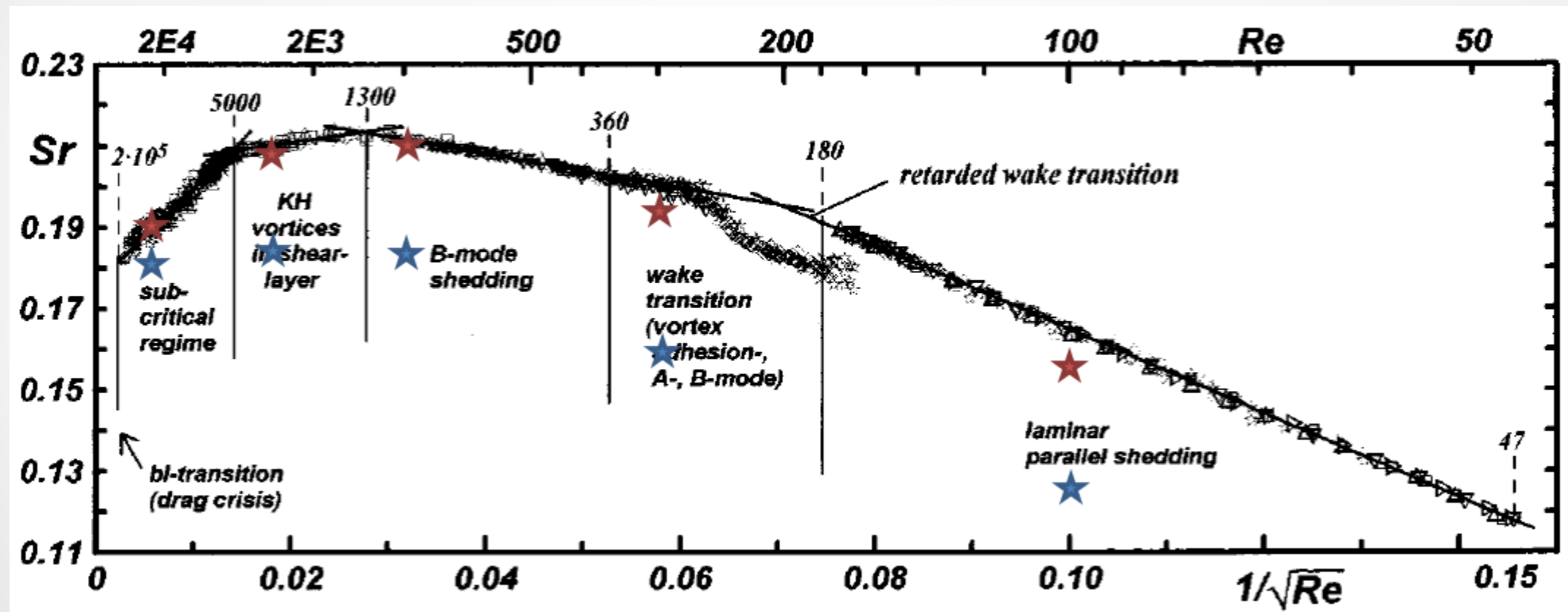
St = 0.22

Frequency = 52.2 Hz (14%)

Drag = 1502 N (7%)



Flow over a Cylinder – Re Sweep



Where to get the Wall Force History Tool?

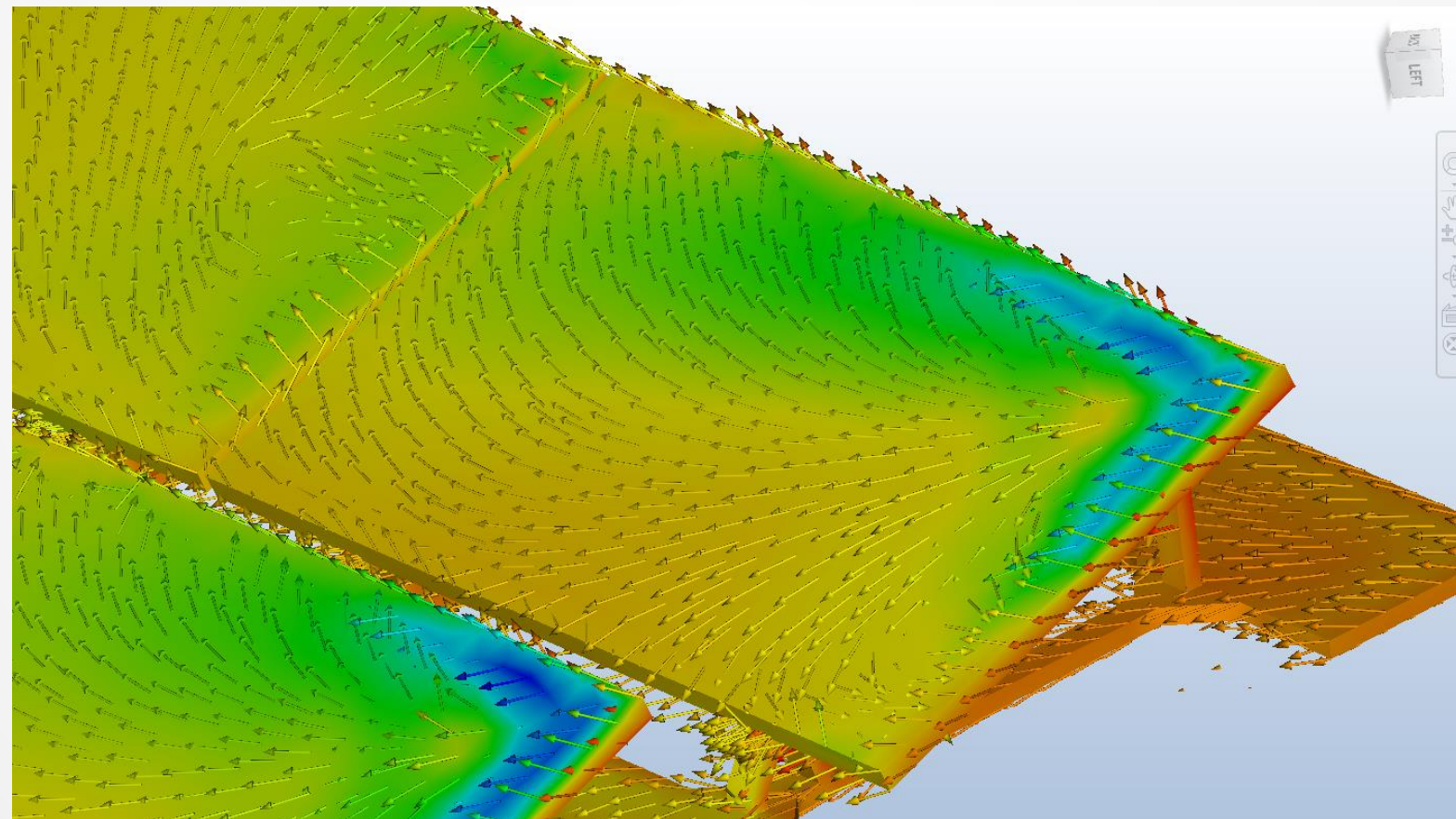
- Download

- 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.

