



AUTODESK UNIVERSITY 2015

DE10379

Aerodynamic CFD simulations just got a whole lot better!

Royce Abel
Autodesk Inc.

Learning Objectives

- Learn how to capitalize on SimStudio Tools or Surface Wrapping for Simulation CFD to reach the desired simulation goals
- Discover which advanced turbulence models should be used for aerodynamics analysis
- Discover how best to implement new mesh adaption tools to eliminate the guess work from meshing complex aerodynamics models
- Learn how to extract accurate steady state or transient wall force results efficiently from different areas of a complex model

Description

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.

Your AU Experts

*As the technical support manager at Autodesk, Inc., for the Simulation Team, **Royce Abel** oversees all simulation technical support in the Americas. Before that he spent 8 years supporting Simulation CFD software as the WW knowledge domain expert with a mix of sales, consulting, quality assurance, API development, validation, and technical writing. He developed the Autodesk Help Webinar series, starting with the [Simulation CFD](#) software product line with rave reviews. Before joining Autodesk he worked with companies like Impact Technologies; Lockheed Martin; General Electric; Lutron Electronics, Inc.; and Blue Ridge Numerics, Inc. These experiences gave him a breadth of experience around condition-based monitoring and prognostics of industrial machinery, structural analysis, signal processing, manufacturing, design, carbon nanotubes, electrostatics, and automation. He has a master's degree in mechanical engineering from Rochester Institute of Technology.*

Learn how to capitalize on SimStudio Tools or Surface Wrapping for Simulation CFD to reach the desired simulation goals

Using SimStudio Tools to prepare complex geometry for aerodynamics simulation

All too often complex models need to be brought into a simulation tool that are significantly far from being simulation ready. This means that the CAD files include geometry features that will make meshing inefficient or not valid enough to bring the geometry into the CFD software. Aerodynamic simulations typically require longer runtime and a higher mesh requirement compared to other industries.

Therefore, any geometry changes that can be done to improve mesh inefficiency should be applied. The SimStudio platform has proven to be one of the fastest ways to make these types of geometry changes to be simulation ready. Below is an example of preparing an automotive model to be simulation ready by using SimStudio Tools.

Preparing Autodesk Technicon Automotive model to be simulation ready using SimStudio Tools

In the design process there are multiple levels that geometry has to pass through: sketches, surface geometry, and full production models as some of the most basic checkpoints. The earliest form of this geometry the automotive industry would typically want to simulate airflow around to get design insights originates as surfaces in Autodesk Alias. The problem with files starting with Autodesk Alias is that this is not a typical file format that Autodesk CFD can read, but it is one that SimStudio Tools can easily read and start to prepare to be brought into Autodesk CFD. Even though you will be able to bring the Alias geometry into SimStudio, typically found in the Alias models is a daunting task to prepare the model. The problem at hand is very doable, but when models like this are supplied with the goal of doing simulation it is best to push back on the Alias experts! With some basic guidance they can quickly remove the details that are not necessary within Alias and supply geometry that is much closer to being Simulation ready. In Figure 1, details that you wouldn't need for an external aerodynamics analysis includes all the internal details like the seats, dash, steering wheel, and any other miniscule details. Really, in the end all that is needed external surface that come together cleanly.

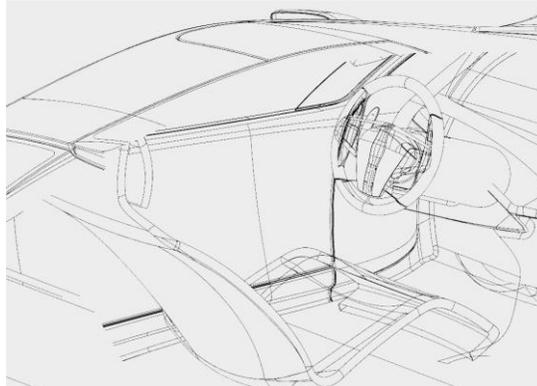


FIGURE 1: RAW AUTODESK ALIAS CAR SURFACE GEOMETRY

Once the Alias expert returns the model, there will most likely be many details that remain that still need to be repaired to get the geometry simulation ready. The major problems with this



specific car geometry that needs to be addressed is the abundance of seams around the entire car (Figure 2), intersecting surfaces (Figure 3), and other fine details or complex surface in areas of minor interest. All of these can be improved or eliminated due the features lack of importance to the overall simulation for improved mesh efficiency.

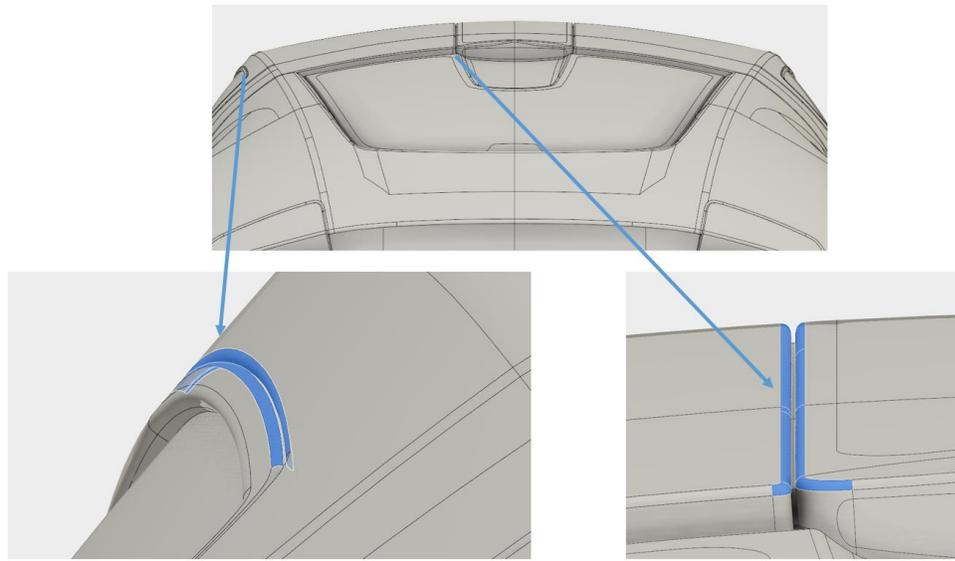


FIGURE 2: EXAMPLE OF SEAMS AROUND THE CAR BODY

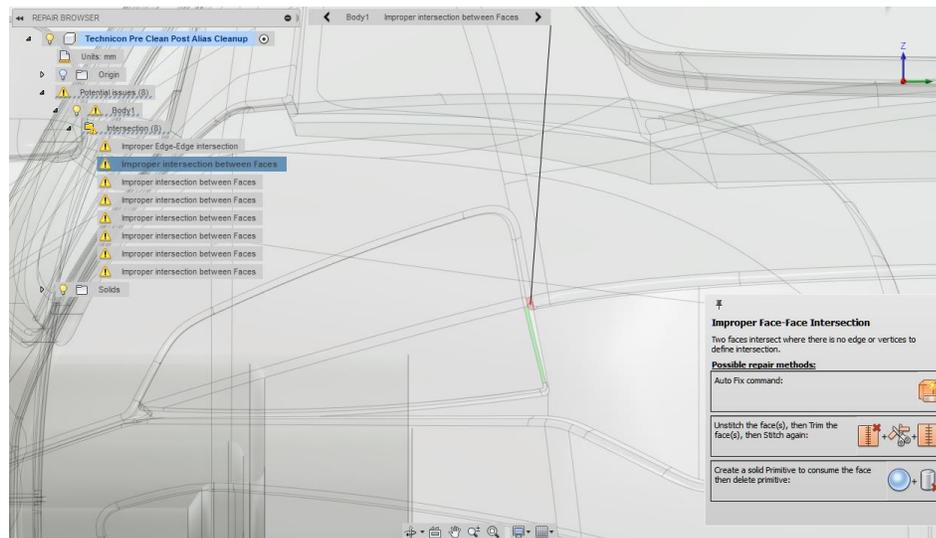


FIGURE 3: INTERSECTING SURFACES ON TAIL LIGHT

Workflow leveraged to prepare the Technicon car for simulation

When working with SimStudio there are many automatic fixing tools that can do wonders for geometry. Other times, like in this example the automatic tools don't have enough knowledge about intent to do most of the heavy lifting. Most of issues with this car geometry are centered

on the seam details that would require too much mesh to capture and using advanced meshing controls to build the necessary mesh would result in an inefficient mesh. With this in mind the seam geometry the best place to start cleaning. This was done throughout the model by [removing the seams](#) (Figure 4) and then [filling the gaps](#) that remain.

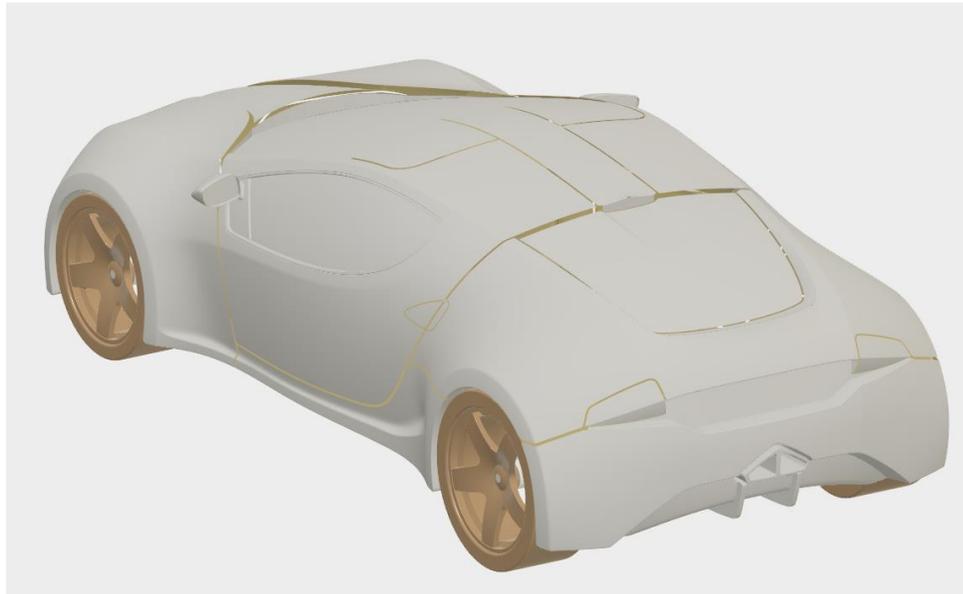


FIGURE 4: SEAMS REMOVED FROM GEOMETRY

The tools inside SimStudio made this very simple and straight forward. While removing the seams the surfaces that need to be selected could be done quickly with the 'Paint' select entity tool without having to click the mouse on every single surface.

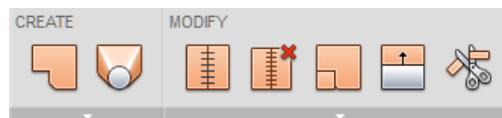


FIGURE 5: CUSTOM TOOL PALLET IN SIMSTUDIO

When filling in the gaps the tools that were used included: patch, loft, stitch, unstitch, merge, extend, and trim (Figure 5). Each were used throughout the model. The most powerful features were the stitch and merge command. The stitch tool has the ability to define what range of gap that you want to close. As you try to stitch surface edges together that aren't fully touching, this will allow you to close those gaps and achieve the perfect contact that is desired. You can even use this to fill in oddly shaped holes so that in the end the small edges that would remain if you had patched them together would be eliminated. The merge command was used to piece together multiple surfaces that really define a larger contour without the extra edges. It is these types of tactics and strategies that will work toward cleaning the geometry as far as possible. Once the geometry is [ready for simulation](#) the seams, surface overlaps, small surfaces/edges, and other unnecessary geometry are eliminated and the geometry is ready to be simulated (Figure 6).

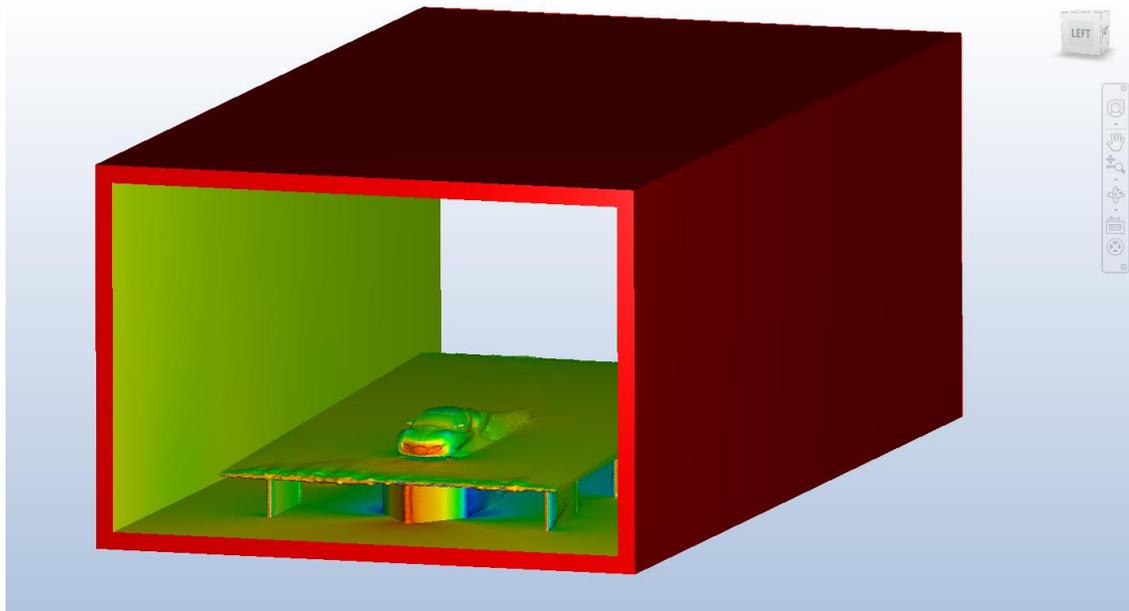


FIGURE 6: CLEANED AUTOMOTIVE CAR MODEL RUNNING IN AUTODESK CFD

Using Surface Wrapping for Autodesk CFD to quickly get aerodynamics simulation running

In the Autodesk CFD 2016 release a new meshing workflow provided the opportunity to skip many forms of geometry cleanup. This would save significant time in the early phase of design when trying to study multiple geometries. The surface wrapping technology will *wrap* the geometry supplied with a much larger surface that will basically smooth over complex details in a model. In the previous geometry discussed, it will lay a surface over all the seam details and eliminate them. Currently, it is developed to be used in an external flow or wind tunnel type of analysis.

Details that are unnecessary for simulations

All the details from the Alias model can be seen from within the Surface Wrapping interface. As an example Figure 7 shows the very small details around the side view mirror.

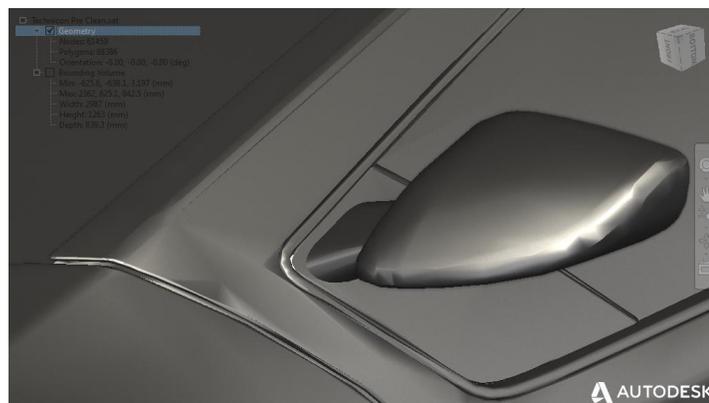


FIGURE 7: SIDE VIEW MIRROR COMPLEX GEOMETRY

Basic Steps for Surface Wrapping

The [workflow for using the surface wrapper](#) are broken into 7 steps.

1. Open Geometry in the Surface Wrapping interface
2. Define the Wind Tunnel Geometry
3. Build the Surface Wrap
4. *Optional:* Define any needed Refinement Regions
5. Generate the Volume Mesh
6. Export a NAS file
7. Inside Autodesk CFD: Create a new design study select the NAS file saved from the Surface Wrapping interface.

While watching the above linked video, examine the top of the car and notice how after the surface wrap was completed how smooth the top of the car remains. This has the highest potential to eventually generate an efficient mesh.

Running model inside of Autodesk CFD

Looking at Figure 8 it can be seen how the mesh details along the profile of the car have been eliminated and smoothed over by the surface wrap of the car body.

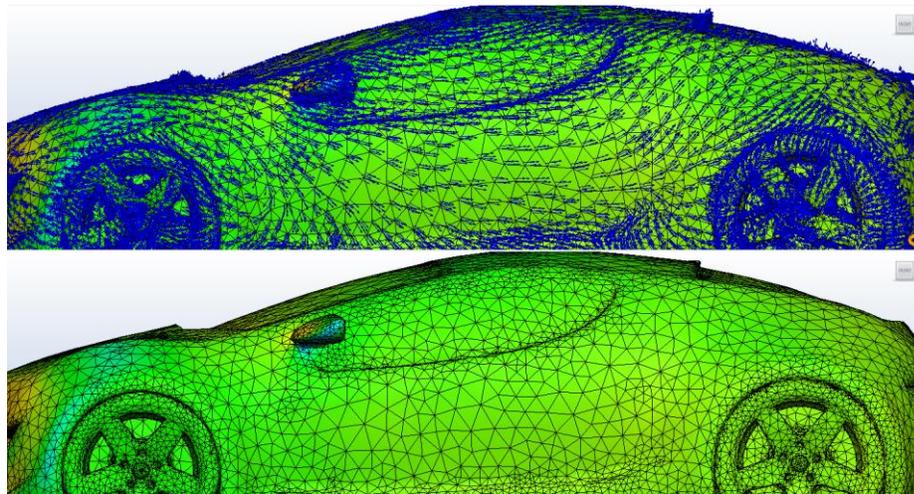


FIGURE 8: SURFACE WRAPPED GEOMETRY RUNNING IN AUTODESK CFD

Which process should be used for your design goals?

Based on your final needs from running an analysis of the car body, there is currently a clear distinction of which method should be used. If your end goals include the following: highly accurate results, forces measured on different surface in the model, complex wind tunnel configurations, time available based on complexity of geometry for repair, and in general a desire to use all the meshing controls available then the workflow that you should use it SimStudio Tools.

If there is pressure to generate results as fast as possible to do comparative studies for rough trending, geometry is just too complex to clean, only visualization is required for non-engineering needs, or you just need basic system level forces you can have success using surface wrapping.



The surface wrapping technology is currently in the first generation in Autodesk CFD. Expect this technology to expand over the next few years such that many of the benefits of going down the SimStudio path can be captured in the surface wrapping process. When this happens maybe extensive geometry preparation will be a thing from the past!

Discover which advanced turbulence models should be used for aerodynamics analysis

Aerodynamics analysis typically push the limits in the ability of turbulence models to accurately predict the results expected from testing. To best predict the results or trends, a working knowledge of the available turbulence models available in Autodesk CFD and when to use them is crucial. It is also important to fully understand the mesh requirement that could be necessary for these models to accurately predict the flow dynamics.

What is Turbulence?

Before discussing what turbulences models should be used, let us take a step back and cover what turbulence is. Turbulence is a violent or unsteady movement of a fluid. When working with CFD software in general we become accustomed to seeing flow moving in a single direction. This is due to the averaging that takes place in Reynolds Averaged Navier-Stokes (RANS) equation used. This averaging scheme will take out the unsteady nature of turbulence that truly exists. In real turbulent flow there are eddies or swirling flow of varying sizes that results in the fluid moving in all directions. In general, the flow does tend to move in a single direction and that is why once averaged these eddies tend to go away. To capture the effect of the turbulence, models are used to allow the energy to dissipate from the turbulence or in the case of heat transfer increase the conductivity.

General Timeline of Turbulence Models

There are many models used in industry and academia that entirely model the turbulence all the way down to direct numerical simulation (DNS) where no models are used and the software attempts to solve for everything. It is good to keep a historical frame of mind when reviewing the various turbulence models available in Autodesk CFD. In Figure 9 a timeline of when each turbulence model was first introduced is shared.

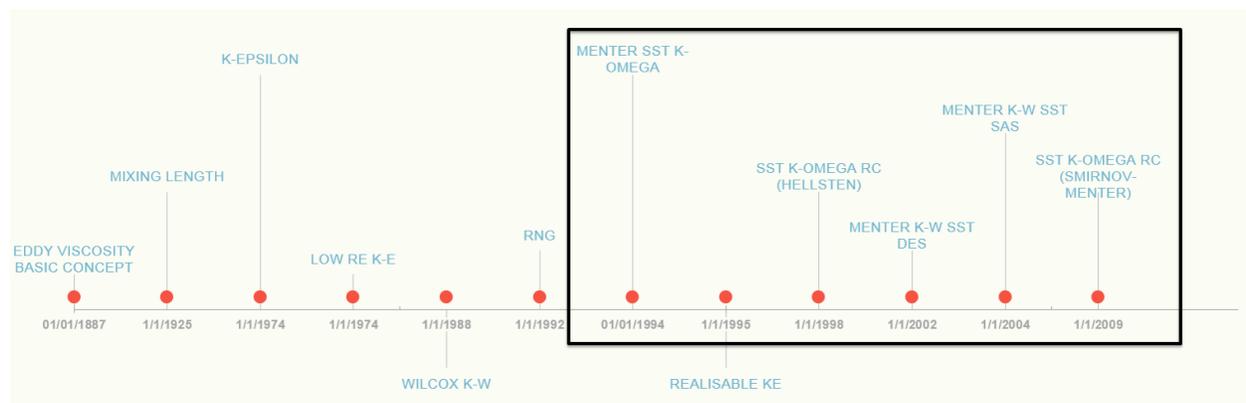


FIGURE 9: TIMELINE OF SELECT TURBULENCE MODELS



On the far left of the timeline back in 1887, the general idea of eddy viscosity or the transfer of momentum. Later in 1974 we see the typical industry standard k-epsilon turbulence model. This model is used throughout many industries, but is not used for sensitive aerodynamic analysis due to its inability to capture adverse pressure gradients and its reliance on wall function to model the boundary layer. In more recent history we start to find more modern aerodynamics turbulence models boxed out in Figure 9. These we will discuss in more details in an applied sense and not mathematical detail.

Overview of Turbulence Models for Aerodynamics Analysis

The first set of turbulence models are very similar having the same foundation of being designed to be used for steady state simulation.

SST k-omega: Two-equation, linear eddy viscosity RANS

Hybrid turbulence model combining the best of k-epsilon and k-omega. The near wall regions are resolved with k-omega which allows for simulations down to the viscous sub-layer. The model will transition to k-epsilon as you get further from the wall. When considering the mesh along the wall the most accurate solution can be achieved when the Y^+ is less than 2 with a goal of 0.3 if achievable. The mesh enhancement should then transition smoothly from the wall into the core mesh inside the model. The model is still fairly robust with Y^+ values form 10-100.

Pros:

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

Cons:

- Can require a fine boundary layer mesh from 10-30+ layers
- Lacks ability to capture high curvature

SST k-omega RC (Hellsten): Two-equation, linear eddy viscosity RANS

The model still originates from the standard SST k-omega model. There is a slight mathematical change to improve flow rotation and curvature. This improvement is captured through a model studied at NASA Langley to review the Coanda Effect. The flow is studied on the back end of a control airfoil to see how well the primary flow stays attached to the curvature and eventually flows downstream. In Figure 10, the test data shows the expected detachment point. Notice the small recirculation region at the arrow x/c . Without the curvature correction available within this model the results would look similar to Figure 11. With the curvature correction inside of Autodesk CFD, very reasonable results can be achieved as shown in figure 12.



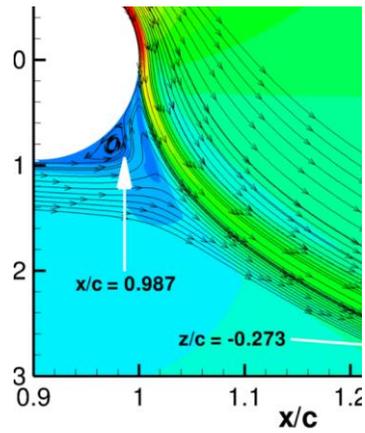


FIGURE 10: NASA LANGLEY TEST DATA SHOWING COANDA EFFECT AROUND AN AIRFOIL

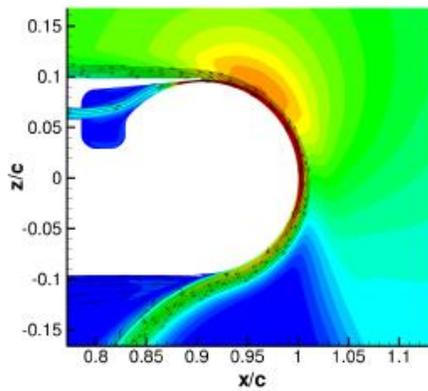


FIGURE 11: NASA RESULTS FOR FLOW AROUND AIRFOIL WITHOUT CURVATURE CORRECTION IN THE SST K-OMEGA TURBULENCE MODEL

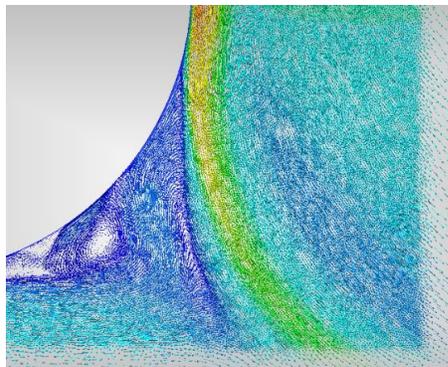


FIGURE 12: AUTODESK CFD RESULTS FOR FLOW AROUND AIRFOIL WITH CURVATURE CORRECTION IN THE SST K-OMEGA TURBULENCE MODEL

Pros:

- Robust across a wide range of flows
 - Accurate for predicting flow separation
 - Ideal for adverse pressure gradients
- Accurate for high Prandtl number (liquid) wall heat flux calculations
- Theoretically, the mathematical change for curvature correction shouldn't impact results when not needed in a simulation
- Added bonus of improved pressure accuracy for cyclones

Cons:

- Can require a fine boundary layer mesh from 10-30+ layers
- Moderate flow profile accuracy for cyclones

SST k-omega RC (Smirnov-Menter): Two-equation, linear eddy viscosity RANS

The model builds upon the Hellsten model with a more complex mathematical model. The differences seen so far for aerodynamics analysis isn't noticeable, but does require more runtime due to increased complexity. The benefit of this model are noticeable for cyclones where pressure and flow profiles are more accurate.

Pros:

- Robust across a wide range of flows
 - Accurate for predicting flow separation
 - Ideal for adverse pressure gradients
- Accurate for high Prandtl number (liquid) wall heat flux calculations
- Mathematically more complex than Hellsten
- Similar pressure drop accuracy for cyclones as Hellsten
- More accurate flow profiles for cyclones

Cons:

- Can require a fine boundary layer mesh from 10-30+ layers



The last two turbulence models are designed to be run for transient analysis, but will still work for steady state. If you are running a steady state analysis there is no benefit of using the following two models.

SST k-omega Detached Eddy Simulations (DES): Two-equation, linear eddy viscosity URANS

This model will simulate turbulent structures in unsteady regions instead of modeling them. This makes it a hybrid between URANS SST k-omega and full LES (Large Eddy Simulation).

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
- More academic versus industrial usage

SST k-omega Scale Adaptive Simulation (SAS): Two-equation, linear eddy viscosity URANS

This model will simulate turbulent structures in unsteady regions instead of modeling them.

Pros:

- State of the art turbulence modeling technique for simulations
- More reasonable for use industrial models when transient analysis is required
- Less mesh dependent compared to full DES while delivering similar accuracy

Cons:

- More computationally intense
- Require fine boundary layer mesh for maximum accuracy
- Transient is best use case which requires longer runtimes

Guidance on which turbulence model to use

Selecting a turbulence model for an analysis can be a difficult task when first approaching a new project. In an attempt to simplify the decision making process, 3 different flow schemes are discussed that outline model setups that are typical in aerodynamics simulations. The flow regimes are attached flow, complex turbulence structures, and Strouhal Number.

The attached flow instance, is very typical when reviewing many streamlined bodies that limit the amount of wake structures beyond the model. The best case scenario would be a highly streamline body like an airfoil in a non-stall configuration. Once the flow has become complex enough where the flow off the body starts to have many recirculation regions or transient bodies are coming off the structure a different approach might be necessary. The Strouhal Number is more of a special case scenario where you are interested in understanding the accurate frequency off the backside of the body. These types of analysis are best done in 2D with the basic profile of the body that you are looking at. When reviewing these guidelines there is conversation around 2 orders of magnitude for convergence. This is accomplished by adding 2 extra zeros for each convergence checking method as shown in Figure 13. This provides a more accurate and predictable convergence point for aerodynamics simulations.



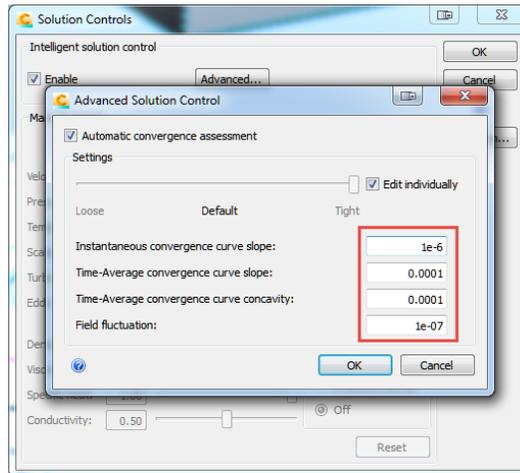


FIGURE 13: CONVERGENCE ASSESSMENT FOR INTELLIGENT SOLUTION CONTROL SET TO AN EXTRA 2 ORDERS OF MAGNITUDE BEYOND THE 'TIGHT' CONVERGENCE OPTION

Attached Flow

Turbulence Model: SST k-omega RC (Hellsten)

Mesh

- 10 -> 15 -> 30 Layers
- Layer Factor: 1.0
- Enhancement Blending Enabled
- Optional Flags:
 - Mesh_enhance_thick: 100-600 (%)
 - Mesh_boundarylayer_blend 1

Solver

- Convergence – Custom with 2 orders of magnitude

Complex Turbulence Structures

Turbulence Model: 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 with 2 orders of magnitude
- **Transient** with testing smaller time steps to test sensitivity

Strouhal Number

Turbulence Model: SST k-omega SAS/DES

Mesh

- Same as **Complex Turbulence Structures**
- Strive for uniform mesh in wake zones to capture eddies

Solver

- Required Flag:
 - Sst_new_iwf 1
- **Transient** – Start with 100 time steps per cycle

Example of using the Attached Flow configuration

The attached flow configuration works best when the flow around your body has minimal flow separations. To capture this workflow with an example we are going to review the NACA 0012 airfoil to demonstrate.

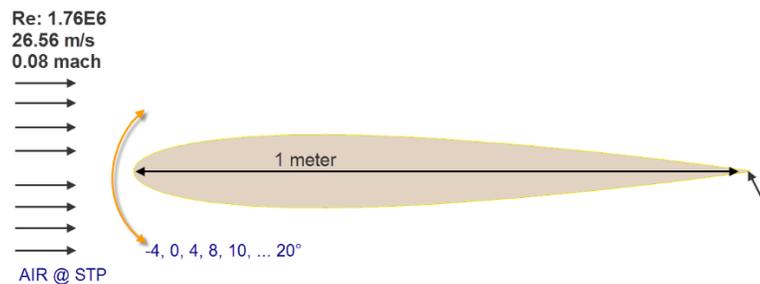


FIGURE 14: BASIC MODEL SETUP FOR NACA 0012

The basic goal for simulating the NACA 0012 is to capture the lift and drag of the airfoil for a given speed with the airfoil at various angles of attack. The goal is to study the shape of this curve and to understand at what angle the wing will stall. The setup of the model is outlined in Figure 14. For this simulation we will leverage the **Attached Flow** configuration previously discussed. Beyond knowing to use this turbulence model there is also meshing to consider.

- 30 to 60 layers
- Mesh_enhance_thick (%): 400
- Layer Factor: 0.8
- Airfoil Surface Mesh: 2.5 mm
- Airfoil Tail mesh: 0.5 mm
- First region beyond airfoil: 5 mm

Once the model was setup with the above options, the model was run with enough iterations that the automatic convergence assessment was hit before the number of iterations defined. The final results trend are shown in Figure 15 as the red dots. The other trends plotted show various wind tunnel facilities and different amounts of leading edge surface roughness. The plots show how the CFD results fit within reason the test data trends.

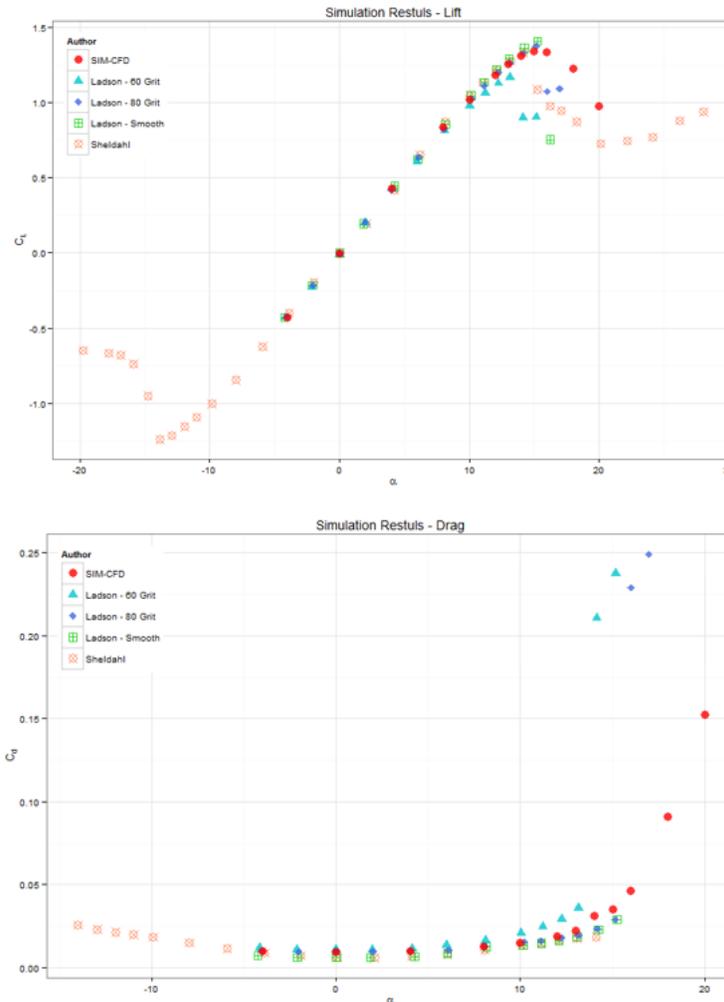


FIGURE 15: NACA 0012 AIRFOIL LIFT AND DRAG RESULTS

Discover how best to implement new mesh adaption tools to eliminate the guess work from meshing complex aerodynamics models

Having the most efficient mesh that captures the required physics is key to running fast and accurate aerodynamics models. Especially for those simulation that require significant runtime to achieve full convergence. Achieving both efficiency and accuracy can be very difficult for anyone trying to define their own mesh and in the end at times almost seems like educated guesswork. This is where mesh adaptation algorithms can eliminate the educated guesswork and make it easier to define a consistent workflow to provide more confidence in trending and final accuracy of the results.

Mesh Adaptation Workflow

Leveraging mesh adaptation does not require significant amount of increased expertise in using the software. Turning it on requires only clicking on a single checkbox, but when looking to leverage mesh adaptation for 3D aerodynamics simulation a few key points should be followed. The basic workflow for the mesh adaptation implementation is outlined in Figure 16. The user will first define a mesh and send the model to the solver to run. At arrow 1 in Figure 16, a key aspect that needs to be adhered to for mesh adaptation is that the solution needs to be converged before the mesh is adapted to. If this is not adhered to then the mesh could potentially adapt to transient flow features that would eventually disappear or move in a steady state analysis. Also at arrow 1 mesh independence indicators will be outputted in the message window. These indicators are not absolute indicators meaning that if the values are 99%, it doesn't mean that the answer is 99% accurate. It just means that the new results after the previous solution is 99% the same as before. In other words the indicators are just relative to previous results. As these values trend towards 100%, running more adaptation cycles will typically prove unfruitful. After the solver finishes running, the solution will be interrogated and new meshing length scales will be calculated and then sent to the mesher to generate a new mesh based on the new requirements. At arrow 2 is where the Autodesk CFD implementation of mesh adaptation is unique. Most mesh adaptation implementations will only re-mesh the areas in the mesh that require changes. It was found that these implementations were not as robust as required in practice. Therefore, the Autodesk CFD implementation will re-mesh the entire model again. This drastically improves the robustness of the mesher with the downside of it is less efficient. In the end though any model that can be meshed can be adapted. Once the new mesh is generated, the old results are interpolated onto the new mesh and eventually sent back to the solver. This loop then repeats based on the number of times defined in the mesh adaptation configuration window, Figure 17.

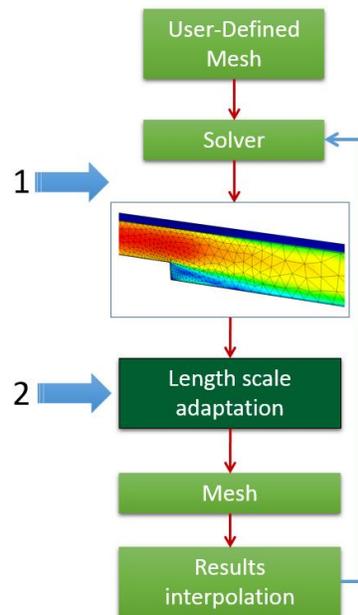


FIGURE 16: MESH ADAPTATION IMPLEMENTATION WORKFLOW

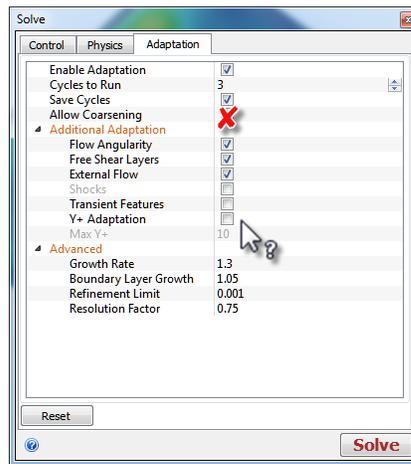


FIGURE 17: MESH ADAPTATION CONFIGURATION WINDOW

The configuration window for mesh adaptation only requires enabling a few extra options for aerodynamic simulations. These options that should be turned on include: Flow Angularity, Free Shear Layers, and External Flow. It is suggested to keep the number of cycles set to 3 and if you are interested in seeing the progression in the adaptation process then enable Save Cycles. Otherwise, it is rarely suggested to enable Allow Coarsening. This option can sometimes cause problems and if you are following best practices in regards to convergence and the process outlined below it shouldn't be necessary. The last feature that you might want to consider using is the Y+ Adaptation. This is not suggested to be used from the start because it does sometimes cause the mesh enhancement to collapse in certain areas of the model. It is best to increase the number of layers and review the Y+ when the simulation is done to make sure that in the areas of interest the Y+ values are in the appropriate range. Or, once the analysis is completed run one extra cycle with Y+ turned on to make some final adjustments. Many times, earlier in the adaptation cycle the mesh is so rough in areas that the Y+ adaptation just doesn't make sense to be leveraged. Once more mesh is built around the model and the layers in general become thinner naturally, this is a great time to do some final adjustments using the Y+ option. Otherwise, all other settings and advanced features are left at defaults.

Example of using the Complex Turbulence Structures configuration

When designing and installing solar arrays on roofs of buildings it is important to understand the lift forces being generated from the wind. This is crucial so the loads on the building if they are mounted are understood and if unmounted due to cost and building codes to make sure that enough weight is on the solar panels such that they don't fly off the roofs!

Problem Statement for the 2x2 solar array

The 2x2 solar array is placed inside of a wind tunnel at a 45° angle such that the leading corner of the solar panel array is creating a damn so that the lift and drag forces should be at the worst case scenario. From the single test data point the lift was measured to be approximately 0.9 lbs and the drag was measured to be 0.3 lbs.

Strategy for Solving

The model is such that we expect very complex turbulence structures to be formed behind the solar panel which would require running transient with SAS. With that in mind we also want to



leverage mesh adaptation because the flow is going to be very complex around all the bodies of the solar panel array. We can't do both at the same time because the transient solver does not support mesh adaptation. Therefore, our strategy will be to first setup the model as an Attached Flow with adaptation turned on followed by a final run of switching to the Complex Turbulence Structures configuration and the transient solver without adaptation. This will allow a very refined mesh to be generated from the attached flow configuration that can be reused with the transient solver.

Initial User-Defined Mesh

For aerodynamics models that are going to be adapted it is best to start with a very rough uniform manual mesh. This will make sure that in the large zones upstream and downstream of the model the mesh isn't too rough from using the automatic meshing approach and the boundary layers through the model start out very consistent. Otherwise, you might also see very bloated boundary layers that will make adaptation calculations more complex and take longer. After a rough mesh is defined for the volume of the flow, a tighter surface mesh is applied to the body being studied. In this example a rough volume mesh of 30 mm was applied with a surface mesh on the solar panels of 5 mm. The number of layers was 3 and this was used due to time constraints. It would have been desirable to start with 10 layers.

Mesh Adaptation Results

At this time the solar panel array is run for 3 cycles of adaptation which means a total of 4 simulations were performed. After the fourth simulation a transient analysis is also run based on the last mesh generated. The overall mesh adaptation progression can be reviewed in Figures 18 – 22. A few highlights to recognize in the results.

- The boundary layers naturally get thinner to maintain a smooth transition from the overall tightening of the mesh.
- The Enhancement Blending adds a lot of extra mesh to help transition the mesh from the boundary layers to the core mesh. This is a desirable outcome.
- As the mesh becomes more refined, the leading edge velocity is captured allowing for higher peak velocities.
- Along the rear panel, as the mesh becomes more refined it can be seen how the recirculation profile shifts further down the panel.
- From the top view the External Flow option helps capture the wake structure downstream of the solar panel. This type of localized refinement would be impossible with user-defined mesh controls.
- Figure 22 really captures that how with even a rough mesh around the leg of the array, the adaptation does capture more of the wake structure and improves the overall resolution around the circumference.
- Figure 23 and Figure 24 show how the transient solver captures the complex turbulence structures in the wake region of the solar panel.



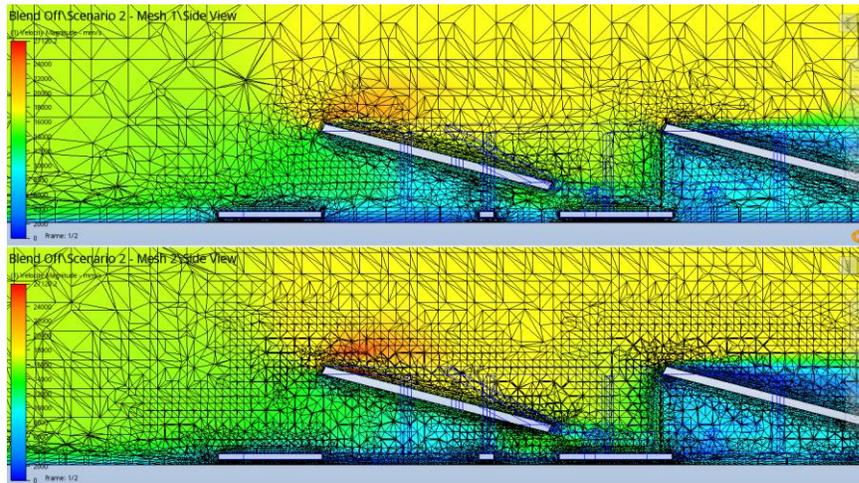


FIGURE 18: SIDE VIEW OF MESH 1 & MESH 2

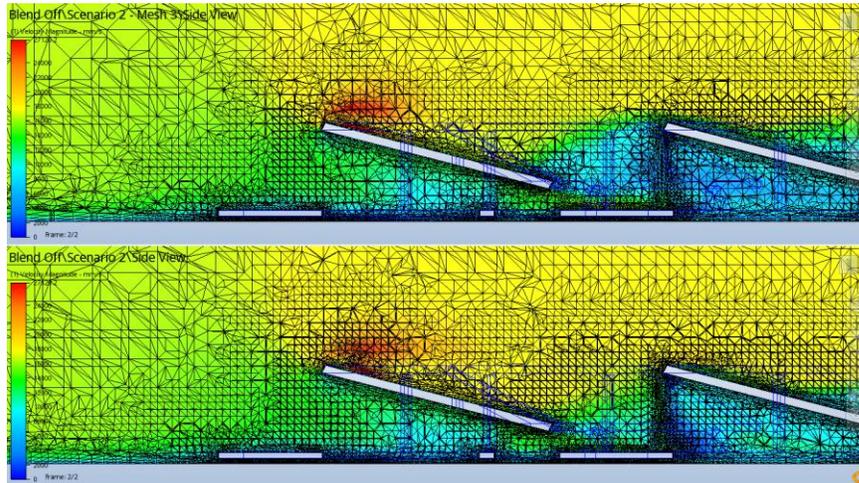


FIGURE 19: SIDE VIEW OF MESH 3 & MESH 4

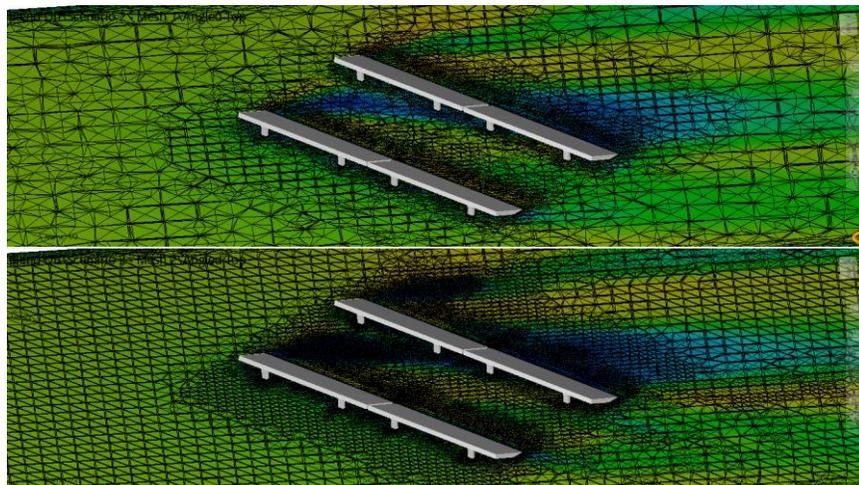


FIGURE 20: TOP VIEW OF MESH 1 & MESH 2

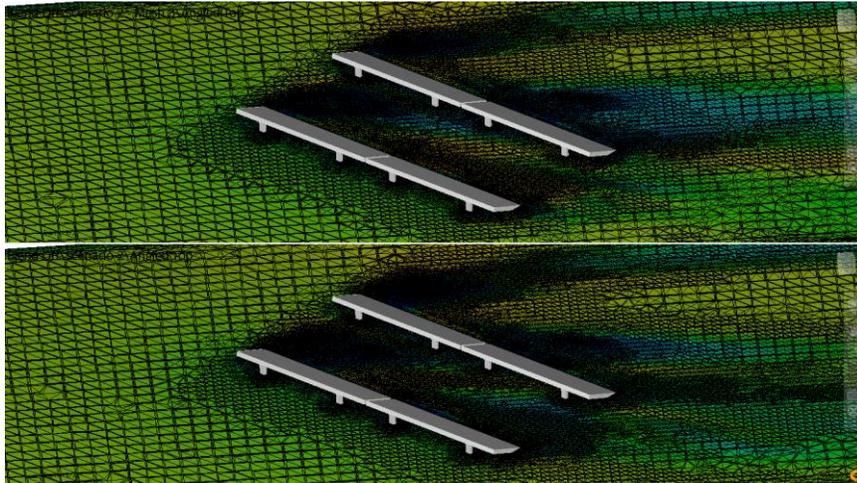


FIGURE 21: TOP VIEW OF MESH 3 & MESH 4

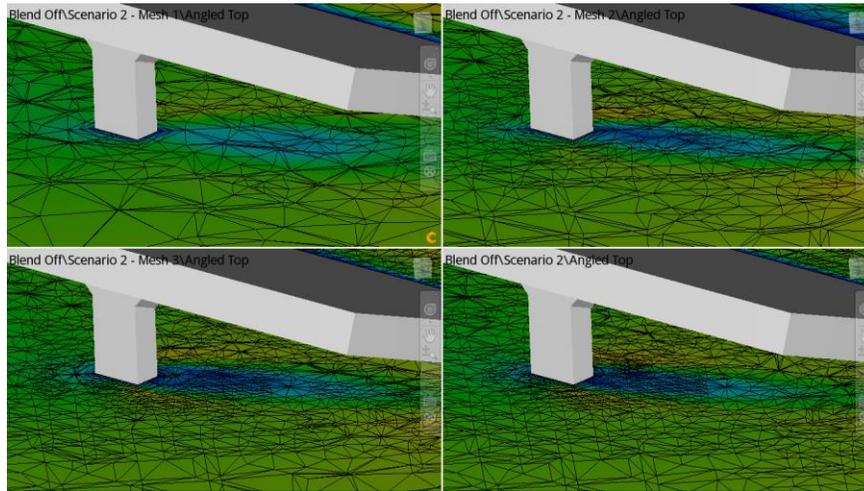


FIGURE 22: DETAILED VIEW OF LEG OF SOLAR PANEL

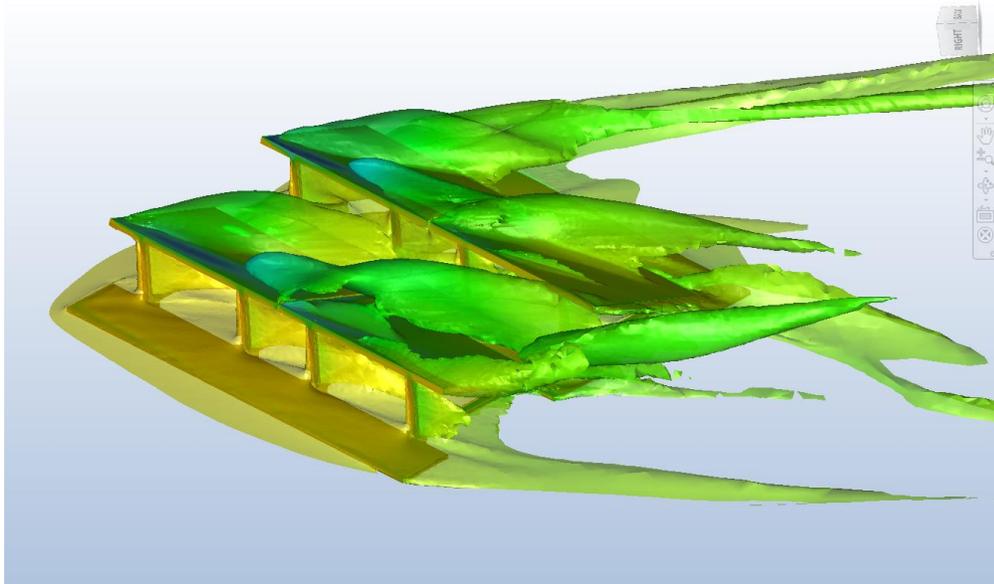


FIGURE 23: 20 MPH ISO-SURFACE OVER SOLAR PANEL SOLVED STEADY STATE

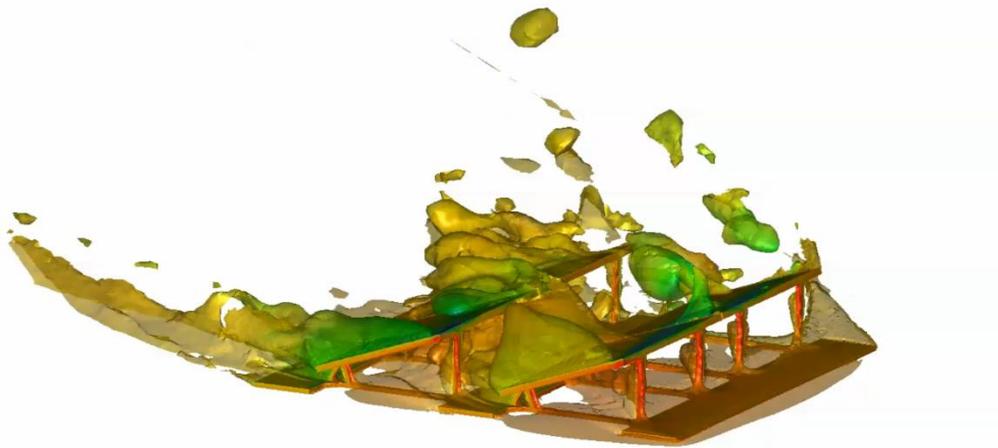


FIGURE 24: 20 MPH ISO-SURFACE OVER SOLAR PANEL WITH TRANSIENT SOLVER

Summary of results compared to test data

The target lift and drag for this solar panel from a wind tunnel test setup was 0.9 and 0.3 lbs respectively. To get an idea of the quality of the mesh adaptation we can study the measured lift and drag for each mesh cycle and then finally for the transient simulation. This is organized in Chart 1.

	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%

CHART 1: SOLAR PANEL ARRAY LIFT & DRAG SUMMARY OF RESULTS

Overall, the results are reasonable since less than 20% error for lift and drag even from the start would be enough for many to base design decisions. Using the mesh adaptation during steady state mostly improved the lift values, but the drag values didn't change significantly. After the first cycle the error did shift from being low to being high. This is actually beneficial to know for the designer as now they know the values are generally conservative. There is some speculation that the Mesh 2 run wasn't near convergence as it maxed out at the defined maximum number of iterations, 5000. With that in mind one should not consider the 1% error in lift with the Mesh 2 cycle as where we should have stopped to begin with.

What was exciting to see was the improvement in the drag results for the transient analysis. This simulation did require significant amount of time (12,295 iterations). Therefore, seeing the drag error cut half gives confidence in the tactic used in regards to running the transient analysis from the steady state mesh adaptation.

Learn how to extract accurate steady state or transient wall force results efficiently from different areas of a complex model

A very common difficulty for many users is to accurately capture steady state or transient wall forces. When customers are running their steady state analysis they typically do not run out to full convergence nor have a good understand if the wall forces they want to study are starting to become asymptotic vs general flow convergence. For the transient studies it is impossible for customers to clearly review their wall forces over many iterations with the software out of the box. Without being able to do this with a transient analysis becomes a burden to try to determine the average wall force or even the frequency if that is desired.



When it comes to basic task of using the wall calculator in Autodesk CFD to measure the wall forces for a fully converged aerodynamic simulation. One should not use the default wall calculator algorithm. Instead, the newer method should be enabled through the resid_bdry_force_calc flag. Figure 25 shows the settings that need to be included within the Flag Manager before the simulation is run to be able to leverage the newer wall calculator formulation. This method will provide accurate results during run-time and in general provides a more robust and accurate formulation compared to the older force calculator

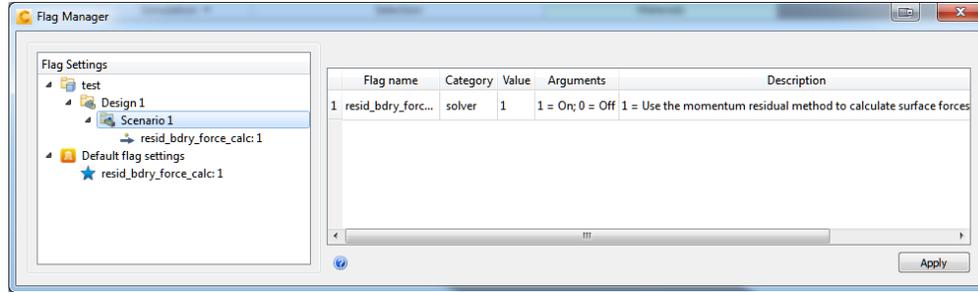


FIGURE 25: FLAG MANAGER CONFIGURATION FOR NEWER WALL CALCULATOR FORMULATION

Wall Force History Tool

To try to solve the customer frustration of understanding if the wall forces have converged or the average wall force of a transient analysis. I created an add-in for Autodesk CFD which will help determine these plus a few other calculations and automation tools.

- Forces extracted to excel for every iteration
- Coefficient of Drag/Lift calculated
- Frequency Response determined for each global direction
- Multiple zones can be studied by using Groups of surfaces or edges
- In excel all data is cleanly plotted and labeled automatically
- Steady State and Transient supported and the output to excel will adjust automatically
- Necessary material and far-field flow values are calculated automatically to reduce inputs

An example of this output is shown in Figure 26. This output is for a transient response and also captures the FFT output on the far right of the figure. All of these data in the figure are automatically generated from the tool. This is a very useful result when needing to calculate the wake frequency. If you are looking to [leverage this tool](#) you can [download](#) it and use it at your leisure. If you have any questions bring it to the CFD forums and we can help out there!

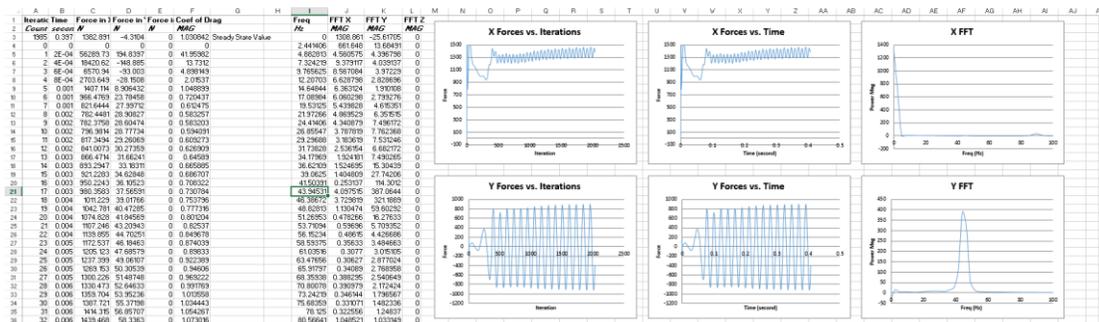


FIGURE 26: WALL FORCE HISTORY TOOL EXCEL FILE OUTPUT

Example of using the Strouhal Number configuration

The last example and configuration that needs to be reviewed is the Strouhal number or more simply solving for high resolution frequency calculation. To study this example, the basic flow over a cylinder is an excellent starting point. It is simple enough to understand, but in the end the physics being solved for are complex and challenging when looking at the full sweep of potential Reynolds numbers flow conditions that could be approaching the cylinder.

Problem Statement for flow over a cylinder

For this implementation of flow over a cylinder we will be using a generic oil material flowing at 30 ft/s in a 2D model. The cylinder itself will be 1.5 in diameter. The mesh defined is very simple with a 0.001 m edge mesh on the cylinder and a 0.004 m mesh in the core flow or surface. There is no usage of refinement regions.

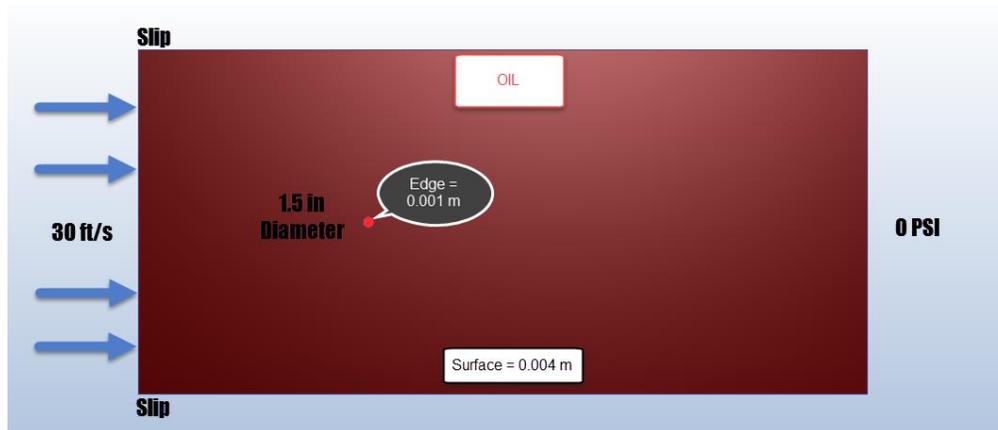


FIGURE 27: FLOW OVER A CYLINDER PROBLEM SETUP

Expected Results

Flow over a cylinder has been studied in detail for years and there is many sources of reference. When looking to study the frequency response of the flow over the cylinder first you need to find a high quality Strouhal (St) vs Reynolds (Re) number chart. An example of this type of data is shown in Figure 28. For the model setup defined previously, the Reynolds number is calculated to be approximately $2.5E4$. This indicates that we would expect a St of 0.19 or a frequency of 45.94 Hz. The drag is also estimated to be 1620N based on a 1 m long cylinder.

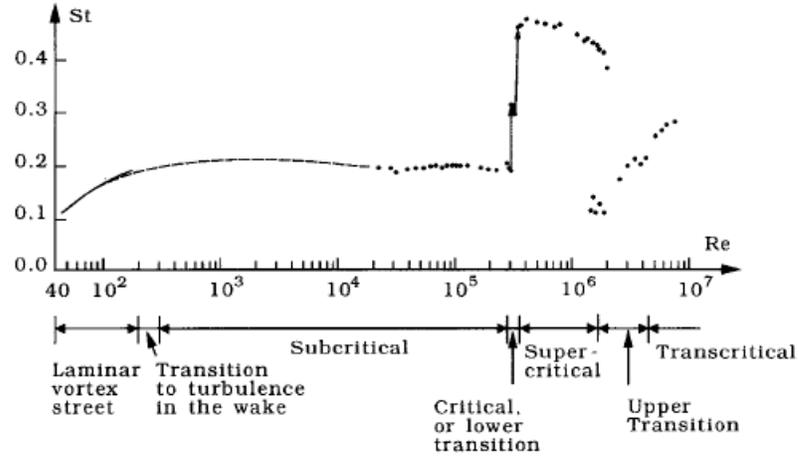


FIGURE 28: STROUHAL CHART FOR FLOW OVER A SMOOTH CYLINDER

Simulation Results

Knowing an estimated frequency response guides us to use a time step that would have at minimum of 100 time steps per wake cycle. Using this as a guide the results are a St value of 0.18 or a frequency value of 43.03 Hz (Figure 29). This is a 6% error. The error could be improved by reducing the time step and increasing the run time and eventually achieve an error of less than 1%. On the drag side the results are measured at 1430N (12% error) with the average value of the oscillations easily determined by using the Wall Force History app previously discussed.

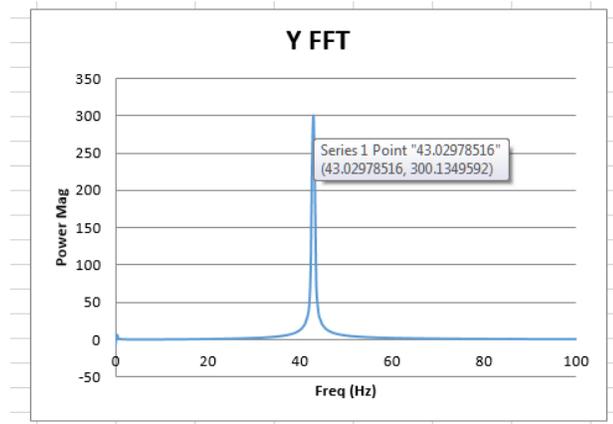


FIGURE 29: FFT OUTPUT FROM WALL FORCE HISTORY APPLICATION

Conclusion

From this handout and the discussion from class 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.

As you start to gain more experience with running aerodynamics simulations bring your questions and share your successes at our [Autodesk CFD forum](#).

