

DE11043-L

Is Your Autodesk CFD Analysis Accurate Or Could You Do Better?

Jon Wilde
Autodesk

Learning Objectives

- A deeper insight into mesh requirements
- Learn about solver controls
- A greater understanding of convergence
- Ability to achieve a higher level of results accuracy

Description

This class will include a competition to set up a model to achieve the best accuracy. It will begin with a discussion and hands-on session involving meshing—basic rules of meshing, mesh sensitivity, and mesh controls; solver settings—Advection Schemes, Nodal aspect ratio, Mach number, and Y+; convergence controls and interpretation; momentum conservation; and energy balance. Finally we will have a competition to apply what was discussed to produce an accurate simulation. Geometry and a description of the test environment will be provided, along with the results required. It is assumed that the user will have a good understanding of the software and can already set up his or her models to a good level.

Your AU Expert

Jon Wilde is a Senior Support Specialist at Autodesk supporting our Simulation products. He is also KDE for Simulation CFD.

His main love and focus is CFD.

Since graduating in 2003, he has worked with various analysis software, initially working within the defence sector, using structural (FEA) analysis while designing flight simulators, moving to both structural and thermal (CFD) analyses on airborne and maritime radomes.

He then moved out of the defence industry and has focused solely on CFD ever since, working with Autodesk CFD for the last 8 years.



His time now is spent supporting customers through 1:1 support and on our forums. He also helps in producing online content and running webinars (which are all recorded and published on YouTube).

A deeper insight into mesh requirements

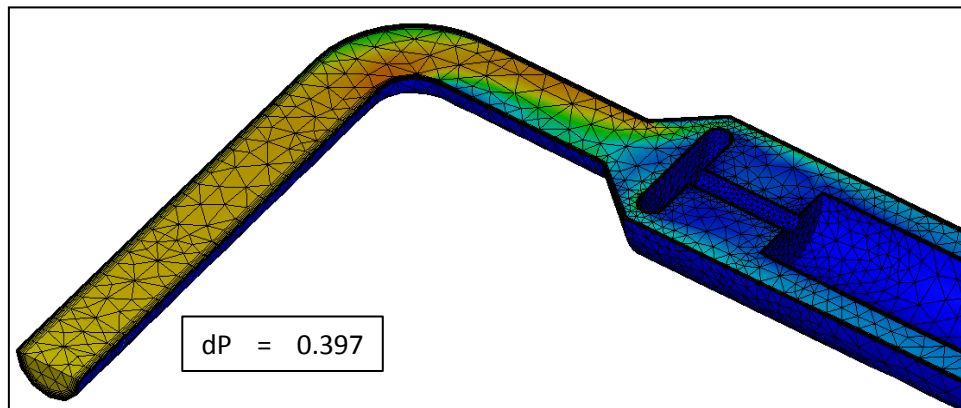
What is the definition of a good mesh?

Ideally, results should be independent of mesh density. This means that if the mesh was refined, the results would not be affected.

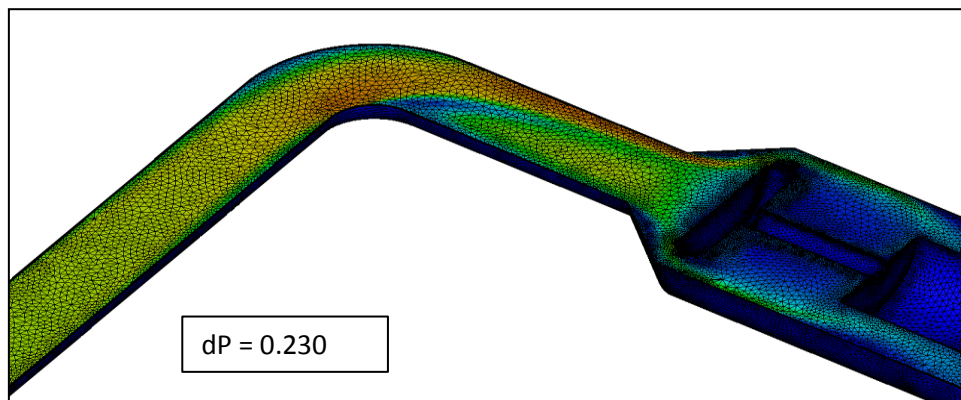
Often, if just using the automatic mesh sizing, this is not the case. By default, the CFD mesher uses edges to define the mesh sizing. This can mean that surfaces and volumes that have areas far from an edge, could be meshed coarser than they should be.

It is also important to note that the mesher does not know the difference between a building or a valve. This means it does not know where or why a refined mesh is required. The user sometimes needs to add some intelligent meshing.

This is an **Automatic Mesh**



This is a mesh that has undergone **refinement**. The difference in results demonstrates the huge effect that a change in mesh can have.



Knowing this, it is recommended that the user refines the mesh before beginning the solution.

Some rules of thumb are to have:

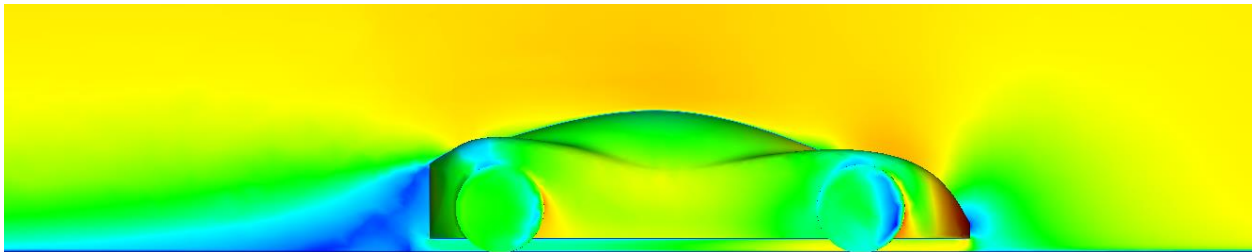
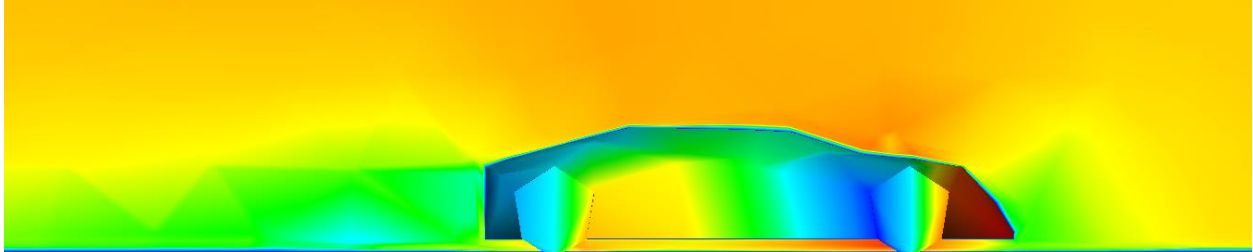
1. A fine enough mesh so that it accurately represents the CAD model
 - a. If a cylinder in CAD looks like a hexagon once meshed in CFD – the results are going to be comparable to flow through or around a hexagon rather than the intended smooth pipe
2. Four to five elements through a small gap or channel as a minimum
 - a. Ideally each gap should have enough elements to capture a flow gradient from a wall, through to the main flow region and back to the opposite wall
 - b. The mesh enhancement (boundary layer) will also help with this but it needs a good starting point (See Y+ on p11 also)
3. Surface and possibly Gap Refinement on
 - a. Surface Refinement will mean that the mesh is no longer just controlled by edges but that it is well distributed over surfaces also
 - b. When using the automated Gap Refinement, testing has shown that three elements is an optimum balance between accuracy and solution time
4. The correct mesh through fans/blower/resistances
 - a. Meshed with a uniform mesh size to include four or five elements from the inlet to outlet face
5. A mesh sensitivity study

Other advanced functionality like Volume Growth Rate can also be enabled. This takes the mesh one level further by controlling the rate at which the mesh coarsens as it spreads out from edges and surfaces. The default is 1.35 (this means that adjacent elements can be no more than 35% different in size) and this can be lowered to ensure that the automatic mesh within a volume is finer than it otherwise might be.

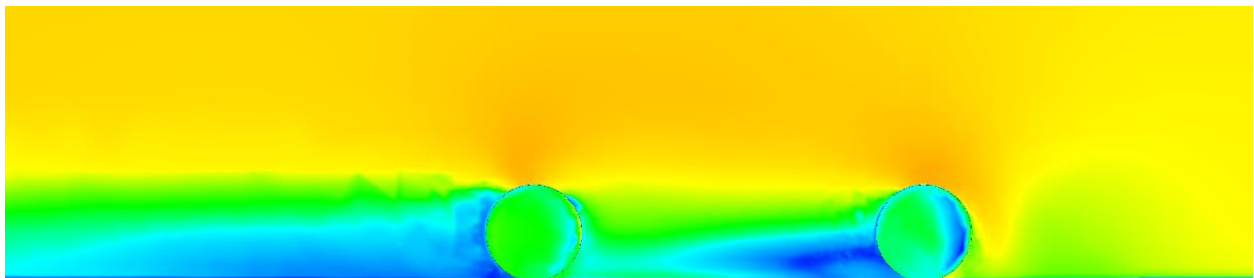
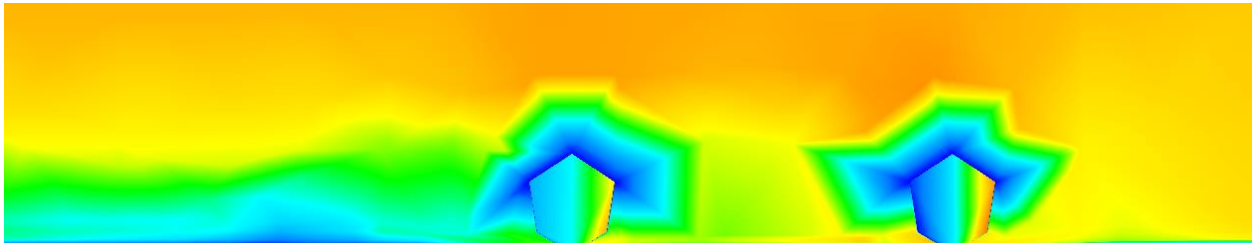


1. A fine enough mesh so that it accurately represents the CAD model

A fine mesh should capture the geometry well enough that differences are hard to spot. The coarse mesh here is clearly not representative of reality. As well as affecting the flow, it is also changing the flow results.

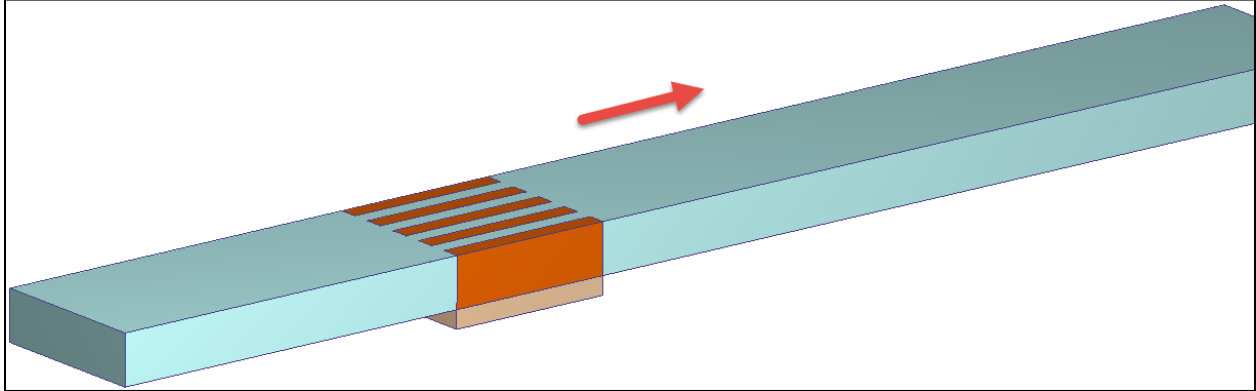


The coarse mesh is causing the wheels to appear pentagonal.



2. Four to five elements through a small gap or channel is a sensible minimum

This is a simple heat sink with a heat load beneath it.



In each scenario, the mesh was refined further from the automatic mesh. The images below show the flow profile through the gaps between the fins of the heat sink as well as the temperature of the chip. It can be seen how the results improve as the mesh becomes finer.

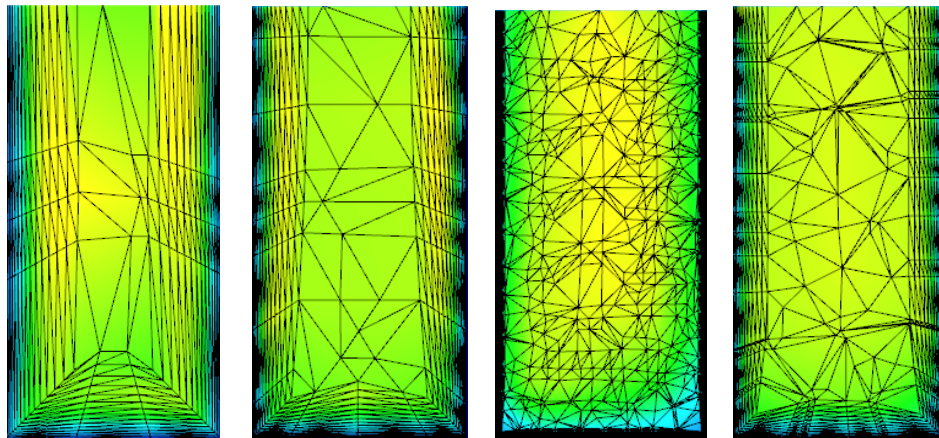
There are a few points to consider when meshing this model

1. Capturing the flow profile through the gap
 2. Sufficient mesh through the inlet and outlet channels
 3. Having at least two elements through the thickness of a solid is also beneficial as it allows CFD to plot a thermal gradient. With just one element this is not possible. Again this can be automated by using the 'Thin Solid Elements' option.
- With a heat sink, the fin temperature tends to be the same from one side to the other so this is not considered here.

This is a comparison between the results of each of the scenarios.

The images show the difference in mesh between two fins as it was refined.

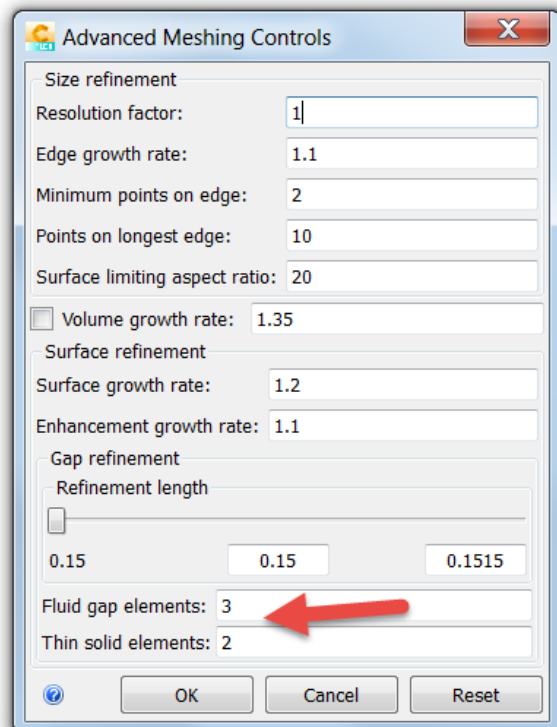
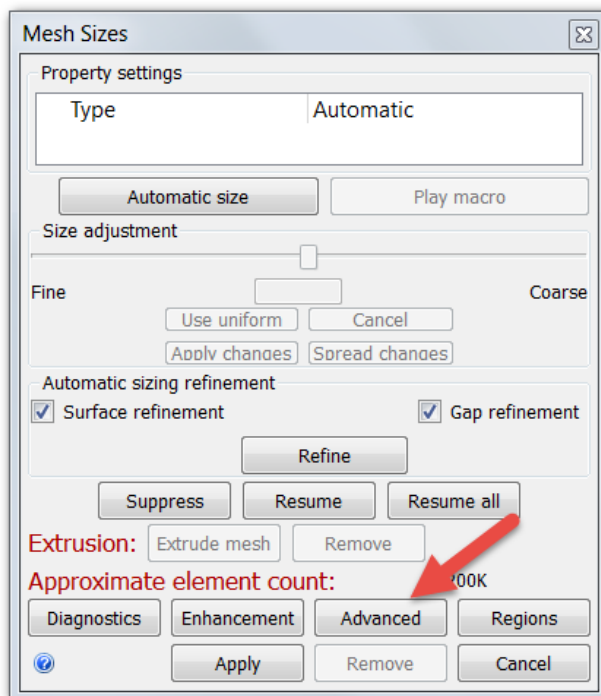
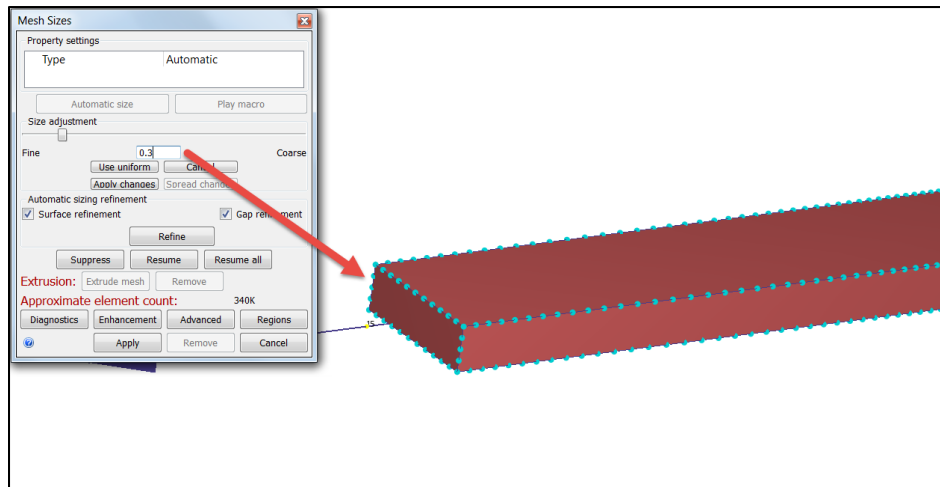
Scenario	Automatic Mesh	Mesh Refined to 0.5	Mesh Refined to 0.1	Surface and Gap Refinement
Elements	13,996	54,316	2,363,514	228,946
Solve Time (s)	296	787	15,228	2201
Maximum Temperature (T)	32.78	41.33	41.27	43.75



3. Surface and Gap Refinement On

The model below was refined globally to 0.3 (so that there was sufficient mesh across the inlet and outlet

thicknesses) and with both Surface Refinement and Gap Refinement on. Notice that three gap elements were used.

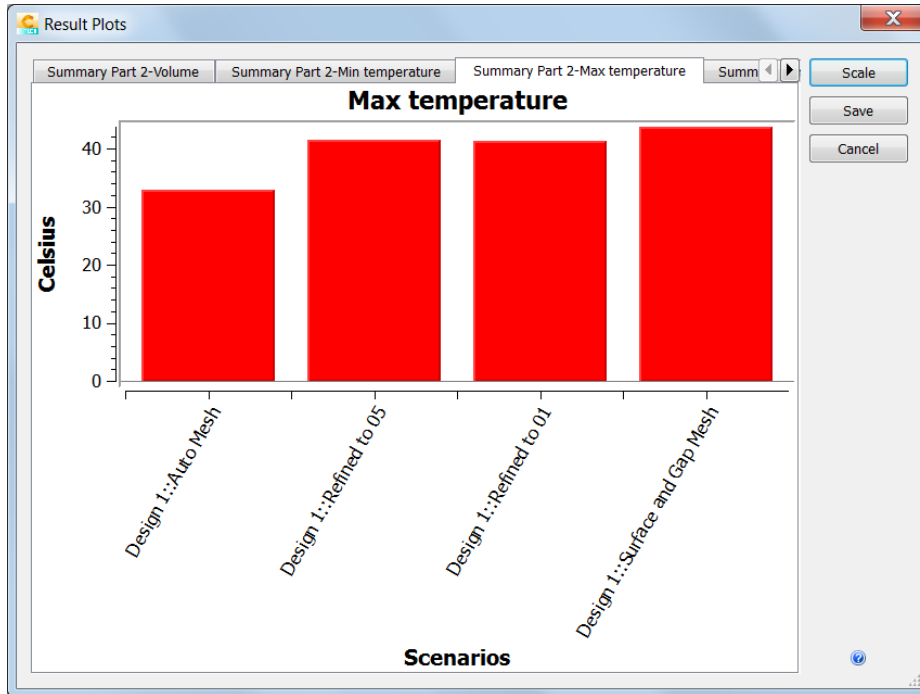


What this demonstrates is that using the advanced meshing tools in CFD, the mesh through small gaps is properly captured with minimal effort. This is a great way to begin an adaptive mesh cycle.



Temperature Comparison

This shows the maximum chip temperature between each of the designs.



4. The correct mesh through fans/blower/resistances

These images are to demonstrate what can happen if the mesh is insufficient. What will typically be found is that the flow rate and/or pressure drop are incorrectly predicted.

With a poor mesh and if a fan curve or head capacity curve were assigned, it is possible that the item could operate at a point off of the curve, which is not realistic.

5. Mesh Sensitivity Study

Running a manual study can be a quick way to verify that the results are independent of the mesh.

To do this:

- Pick some critical values from the results of the study
 - Making these summary points/parts is useful at this stage
- Clone the scenario
- Refine the mesh by 30% (select everything and refine to 0.7)
- Compare the results between scenarios (utilize the Decision Centre)
- If the results vary by less than 5%, it is safe to assume that the results of the previous study were independent of the mesh settings. If not, repeat the process until this is achieved



Users might utilise Adaptive Meshing based off of an initial automatically sized mesh. This is not recommended.

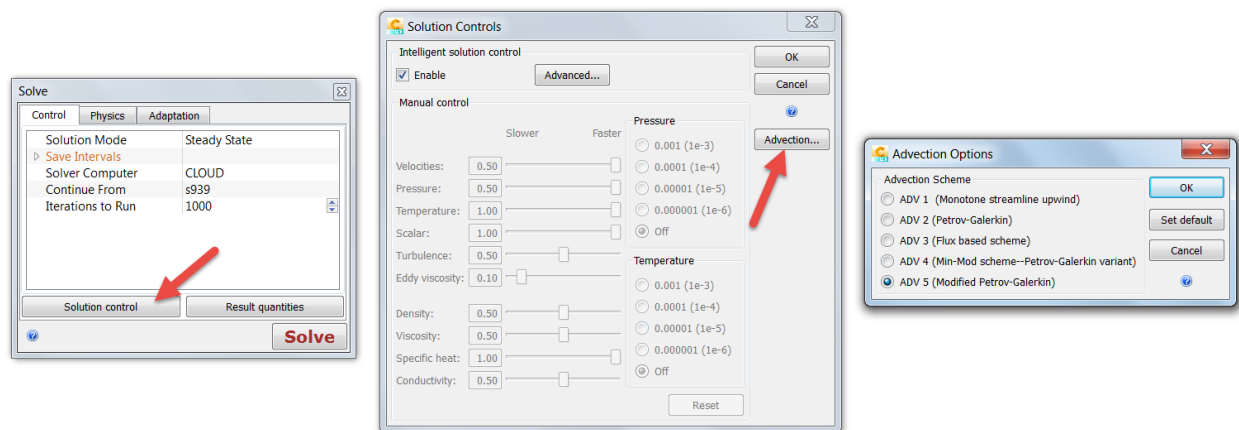
Instead, apply the above settings as a starting point and allow CFD to automatically refine the mesh further once a converged solution is obtained.



Solver Controls

Advection Schemes

Advection is the numerical mechanism of transporting a quantity (velocity, temperature, etc.) through the solution domain. There are five options available within CFD.



- Advection Scheme 1
 - Used to be the workhorse and is still the default but is now superseded by ADV5 for almost everything except:
 - Surface Resistances (inaccuracy with other ADV schemes)
 - Free Surface Analyses (where ADV1 is the default setting)
- Advection Scheme 2
 - Rarely used (ADV5 has superseded this entirely)
- Advection Scheme 3
 - Rarely used. It used to be used for external aerodynamics but again, ADV5 tends to be better
- Advection Scheme 4
 - Long skinny ducts
- Advection Scheme 5
 - **The new king**
 - Recommended for almost all types of analyses and turbulence models. If in doubt though, log a case and request some assistance from our engineers



Turbulence Models

There are many options here although typically one or two are suitable for almost all applications.

The default is **k-epsilon** and works well for most studies.

The other that is becoming more common now (since it has recently been added to Autodesk CFD) is **SST –k-omega**. We might change to this when looking at external aerodynamics, if we have problems with predicting internal pressure drop or are struggling to properly capture a thermal gradient at the wall.

For natural convection models, we might also utilize the **Mixing Length** model, which seems to produce more accurate results and often in a shorter time than other models.

Intelligent Wall Formation

This is enabled automatically if SST is enabled.

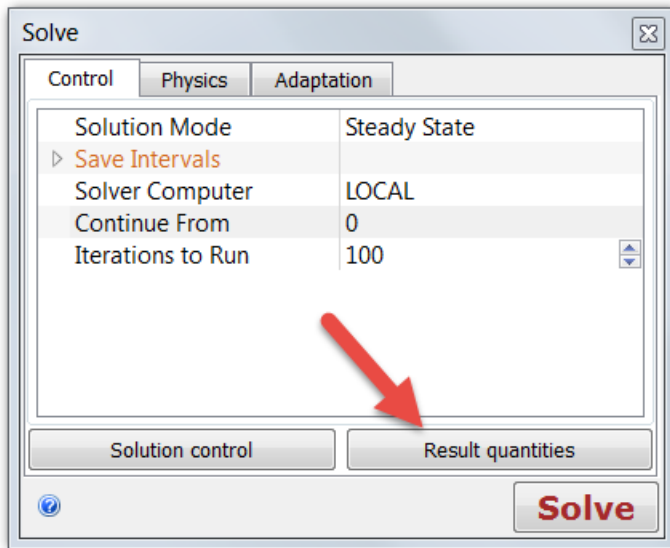
It is often wise to enable this for the k-epsilon model if refining the mesh. It means that the sensitivity to the Y+ value is removed, which can be useful if this value drops below 35.



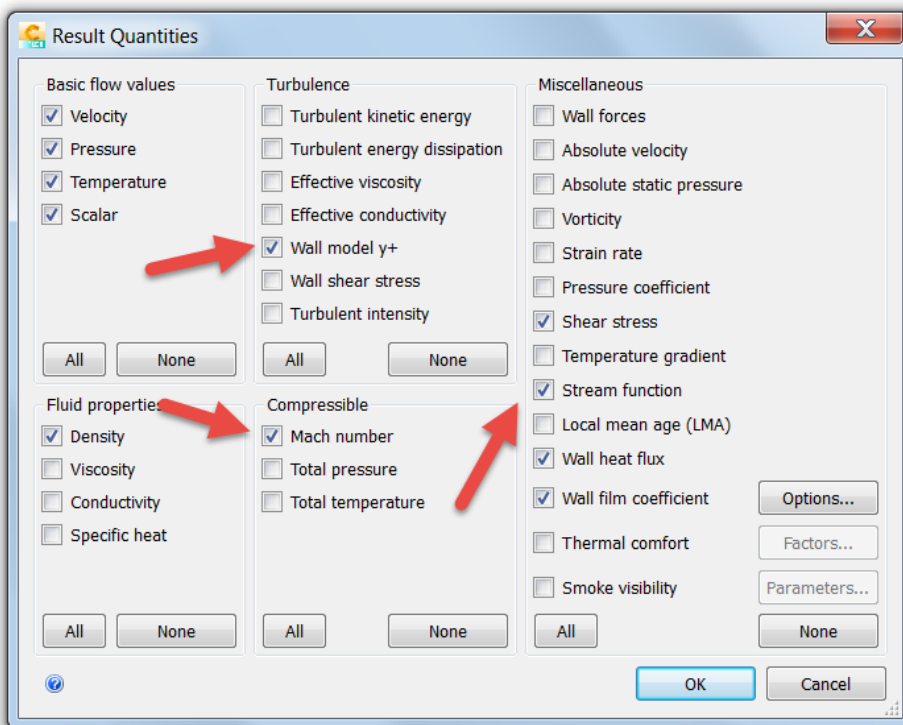
Results Quantities

There are some very useful values within this window and they are off by default.

To access these options:



And to turn on the three shown here, which will be discussed below:



Mach Number



- If the analysis is compressible and the user was unaware, the set up will almost certainly be wrong
 - Values above 0.3 are subsonic and might show some compressibility
 - Values above 0.7 are likely to contain the beginnings of shockwaves – these need to be run compressible
- If the Mach number is showing that an analysis is not solely incompressible, a different setup will be needed. The online [CFD help](#) is useful here but so might be logging a support request for model verification

Y+

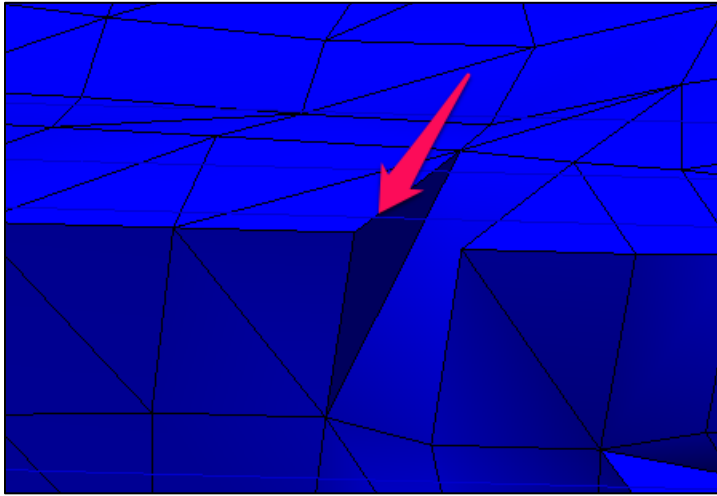
- This can be a really useful tool to identify mesh quality and the required value will vary depending on the chosen turbulence model, As an example for the two most commonly used models in Autodesk CFD:
 - SST $Y+ < 1$
 - ke $35 < Y+ < 300$

Stream Function

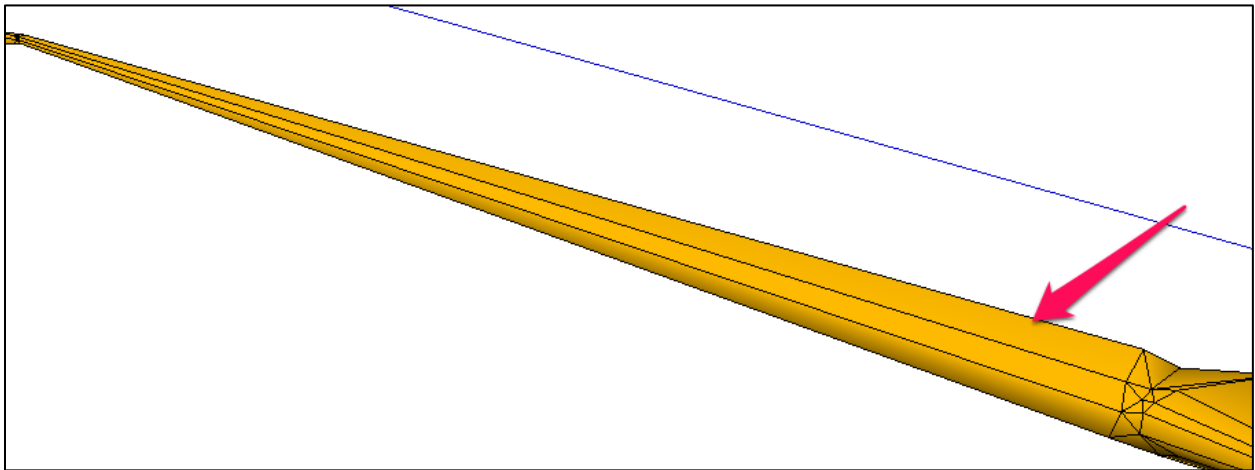
- This will enable Nodal Aspect Ratio (NAR), which is the ratio of the longest to shortest edge of any single element. This is an indication of how equilateral or skewed each element is. The closer to 1 this value is, the more equilateral the element and the better the mesh quality
 - The lower the NAR, the more accurate the analysis tends to be
 - The ideal scenario is to have most elements beneath an NAR of 100-150
 - It is acceptable to have the NAR up to 500,000 in regions that are of no interest to the user, although this is likely to increase the time taken to solve an analysis
 - If the NAR is close to 1, CFD will use closer to 1GB of RAM for each million elements. This will be closer to 2.5 GB for most meshes



This is an element with a NAR of 1:



This is an element with a NAR of 50 (still acceptable)



Imagine an element with an NAR of 1000 (this is very skewed. Picture this capturing flow through a narrow path – it would not produce an accurate representation of reality).



Using NAR can be a hugely useful tool if the meshing fails during volumetric meshing or the optimizing stage.

Often, CFD is unable to create the boundary layer due to a small gap or angle in the model. If two inward facing boundary layers overlap, the mesher will run into issues.

To identify where this is occurring:

- Turn off Mesh Enhancement (so there will be no boundary layer created)
- Mesh the model (no need to run any iterations, just generate the mesh)
- Plot NAR as an ISO surface
 - Typically there will be regions of high NAR and these are often where small gaps or infinite angles exist within the fluid. Infinite angles can occur if a radii meets a planar surface. This kind of shape is difficult to mesh with tetrahedrons
- Return to CAD and close up gaps and/or remove infinite angles from in the fluid (where an fillet meets a planar surface is a good example of this)
 - Alternatively, the mesh on small surfaces can be refined but this is only recommended if these regions are realistic. Taking a short-cut here just to get a mesh is unwise as the run time with a poor mesh will usually take longer than fixing the CAD and can still leave the user with poor results
 - As we know from before, the higher the NAR, the more RAM is required to mesh the model and typically the longer the run time



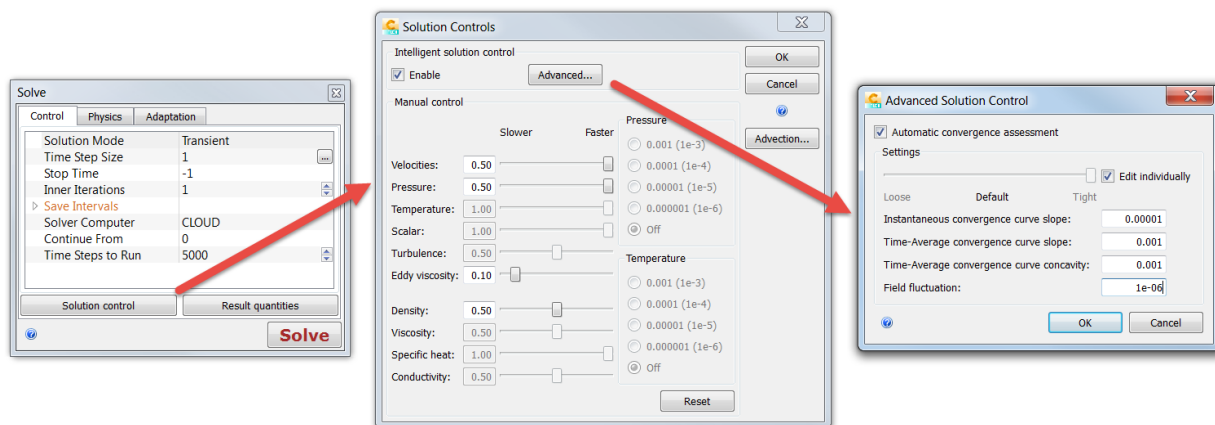
A greater understanding of convergence

Adjusting the Convergence Criteria

Sometimes, CFD stops at a solution before the analysis is truly converged.

This can happen if there are very minor changes between iterations or during a transient analysis. A finer mesh will almost certainly mean that less changes from one iteration to the next, so in a drive for a more accurate solution, one must also consider the convergence criteria.

Generally, changing this from the default to a 'Tight' setting is sufficient. Although it is also acceptable to manually tweak the convergence values further still. A further factor of ten is what is usually used if very tight convergence is required, although the model could run for a very long time with this setting.



There are additional controls here that can help to smooth convergence but will also slow the solution process.

Ability to achieve a higher level of results accuracy

Momentum Conservation

It is possible that due to a few factors, the flow entering the model does not equal the flow exiting. This can be verified by using the bulk calculator. Although CFD can stop and proclaim convergence once there is minimal change between iterations, it is sensible to check that the flow is entering and leaving as expected.

The most common issue here is that the outlet extensions are too short, causing recirculation over what is typically a $P=0$. Simply assigning a velocity at the outlet and a pressure at the inlet to work around this is not a practical solution as the analysis will still be chopping off part of what is happening in reality.

A solution is to add outlet extensions to the CAD model. These tend to be ten times their diameter (or width) in length and should allow the completion of any recirculation before the flow exits the model. If this recirculation is not completed or occurs over the outlet, the pressure drop through the system will likely not be accurate. Refer back to the image on p5 for an example of this.

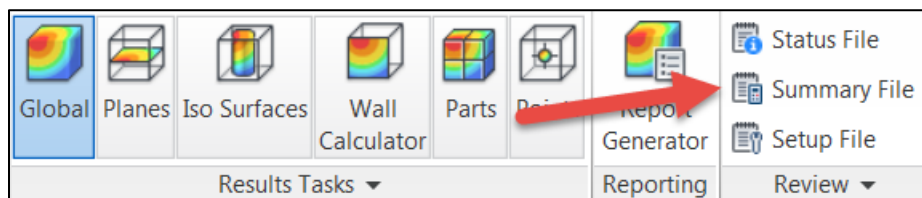
An easy way to see if this is occurring is to add Global Vectors to the model. These always originate at inlets and outlets (and fans/resistances). They should help the user to quickly understand if the flow is exiting the outlets smoothly or if there is recirculation.

It is also to extend the inlet extrusions to about five times the diameter (or width) of the duct. This helps flow become more developed before entering the region under scrutiny. Without this, we could end up with a poor result. If there are suppressed air ducts, pumping flow to an internal section of the model – recess the inlet boundary conditions a little inside the ducts to allow the flow to develop before entering the room, just like a standard inlet.

Energy Balance

If flow is re-entering the outlet, CFD is not going to be able to predict what temperature it should be and therefore there will be a poor energy balance. Although it is likely divergence will occur here, it is possible that the model could converge with a poor thermal balance, or at least a poor thermal solution.

Utilise the **Summary File** (.sol). It details how much energy is entering and leaving the model. This is a great starting point for validating the energy balance of a model.



This is taken from the Summary file and shows the Energy Out – Energy In value is small (close to zero is expected).

It is also useful to check that any energy we know was inputted to the model has been captured and passed to the fluid domain.

```
*** Fluid Energy Balance Information:
MdotIn x Cp x (TOut - TIn) = 0.014709 Btu/s
(Numerical) Energy Out - Energy In = -0.00011754 Btu/s
Heat Transfer from Wall To Fluid = 0.0094694 Btu/s
Heat Transfer Due to Sources In Fluid = 0 Btu/s

*** Solid Energy Balance Information:
Heat Transfer from Exterior To Solid = 0 Btu/s
Heat Transfer Due to Sources In Solid = 0.0094697 Btu/s
Heat Transfer From Fluid To Solid = -0.0094699 Btu/s
```

The combination of all of these factors should yield improved results accuracy.

In reality, all need to be carefully considered during a CFD project.

