Engineering Data Resources a.s
LEIF TRONSTADS PLASS 4
NO-1337 SANDVIKA
TEL. +47 67 57 21 20
FAX. +47 67 57 21 01

LES Workshop within the Masters Course in Turbulence Modeling at Chalmers

REV.	**DATE**	**REASON FOR ISSUE**	**PREPARED**	**CHECKED**

CLIENT: COMPANY LOGO	DOCUMENT TITLE: LES Workshop within the Masters Course in Turbulence Modeling at Chalmers
	PROJECT NAME: 2011-01_LES Workshop
Chalmers	DOCUMENT NUMBER: 2011-01_LES Workshop_1
NO. of Pages 29 / Rev. 1	PREPARED BY: KLAS JOHANSSON Klas.johansson@edr.no

Table of Contents

1. Introduction

In this project, the flow around a NACA 1012 airfoil will be simulated using LES. The airfoil has a chord length of 34 cm and a span of 10 cm. To be able to run the simulation on a desktop computer the free stream velocity will be limited to 0.5 m/s. Figure 1 shows the airfoil.

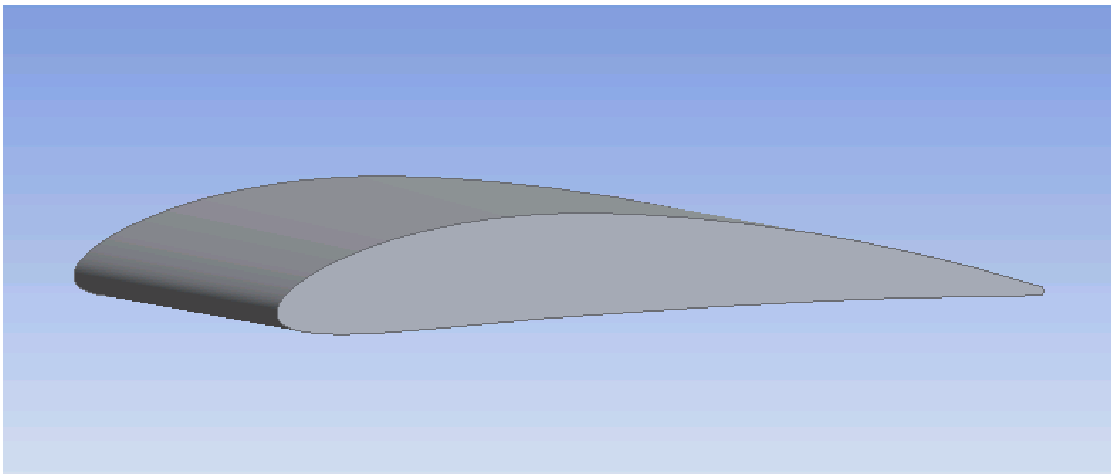


Figure 1: Schematic picture of the engine enclosure.

2. General considerations

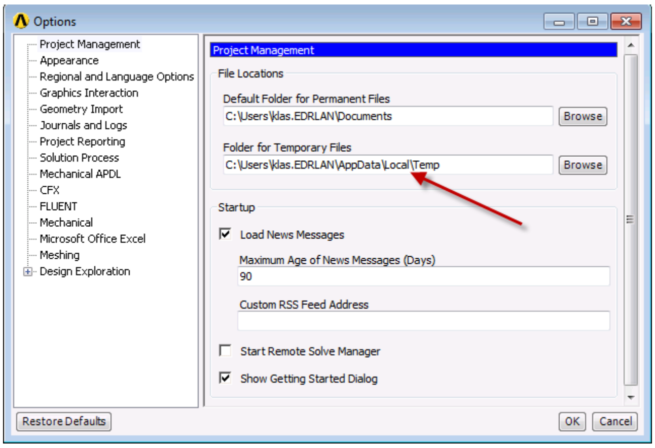
Normally when running a LES simulation an initial flow simulation is provided as an initial guess. In this task we will do this by performing a steady state simulation using a standard two equation model called shear stress transport (SST). The result from this simulation will thereafter be used as an input to our LES simulation. Thus, the first task will be to create a mesh suitable for a SST simulation and thereafter run the steady state flow simulation. The second task will be to create a mesh suitable for the LES simulation and then run the flow simulation.

3. Steady state simulation

3.1. Mesh generation

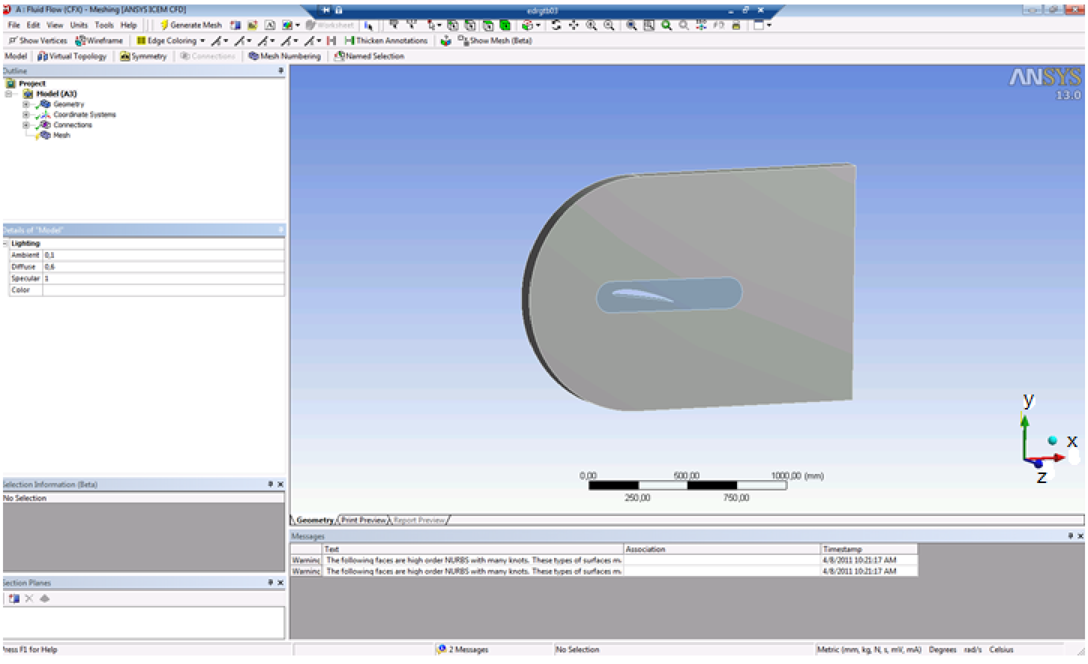
The mesh will be generated within ANSYS meshing.

1. Launch ANSYS Workbench.
2. Change the working directory to a local folder on the system. If not you user account will be filled and the analysis will be stopped due to full disk.

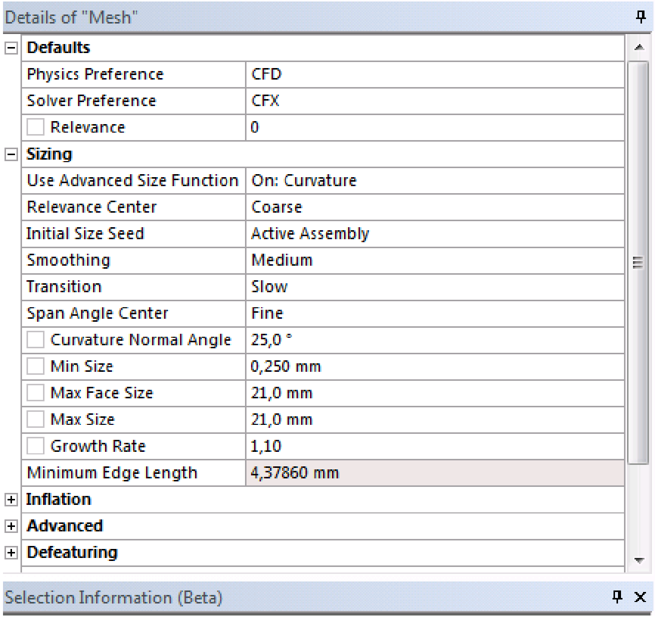


3. Create an analysis system “Fluid Flow CFX” inside the workspace by double clicking in the toolbox or drag and drop.
4. Right click on the geometry cell in the analysis system and select import geometry. Browse to the supplied geometry file NACA1012.agdb.
5. Double click on the mesh cell to start ANSYS Meshing.

The geometry is now automatically imported into ANSYS Meshing. The geometry consists of one multi-body part that consists of two bodies. The inner one with the blade will be meshed with a finer mesh and the outer region will have a coarser mesh. The first step within ANSYS meshing is to create something called “Named Selections”. These will represent our boundary surfaces within ANSYS CFX. The second step is to apply meshing methods and sizing’s to the bodies and surfaces.



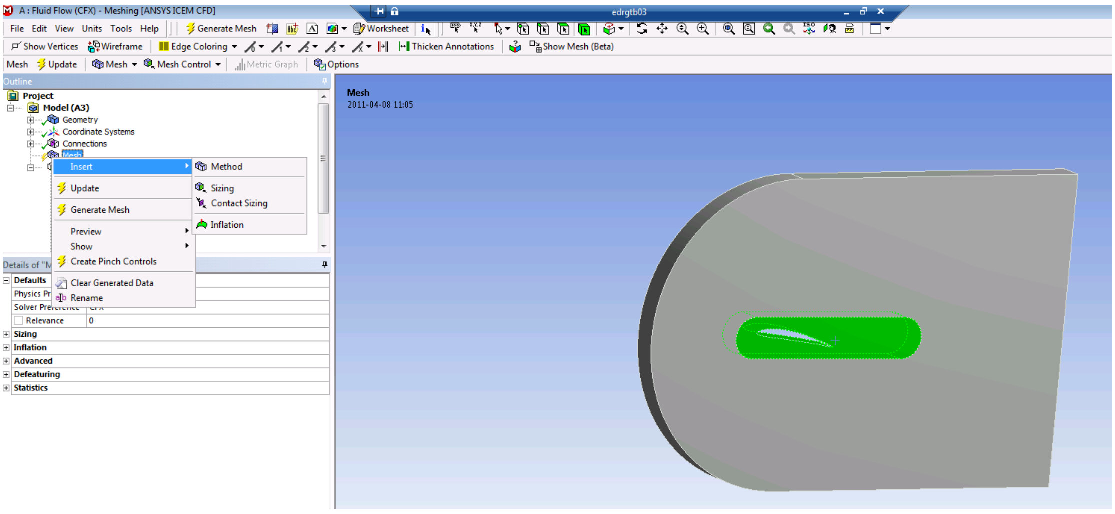
6. Select the surface that represents the inflow. Right click and select “Create Named Selection. Give the selection the name “Inflow”.
7. Repeat this procedure for the outflow and the symmetry surfaces (high and low Z), the entrainment surfaces (high and low Y) and the blade.
8. Set the global mesh sizing by clicking on mesh in the tree and change the sizing in the detail window to the following:


Details of "Mesh"

Defaults	
Physics Preference	CFD
Solver Preference	CFX
Relevance	0
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	25,0 °
Min Size	0,250 mm
Max Face Size	21,0 mm
Max Size	21,0 mm
Growth Rate	1,10
Minimum Edge Length	4,37860 mm
Inflation	
Advanced	
Defeaturing	

Selection Information (Beta)

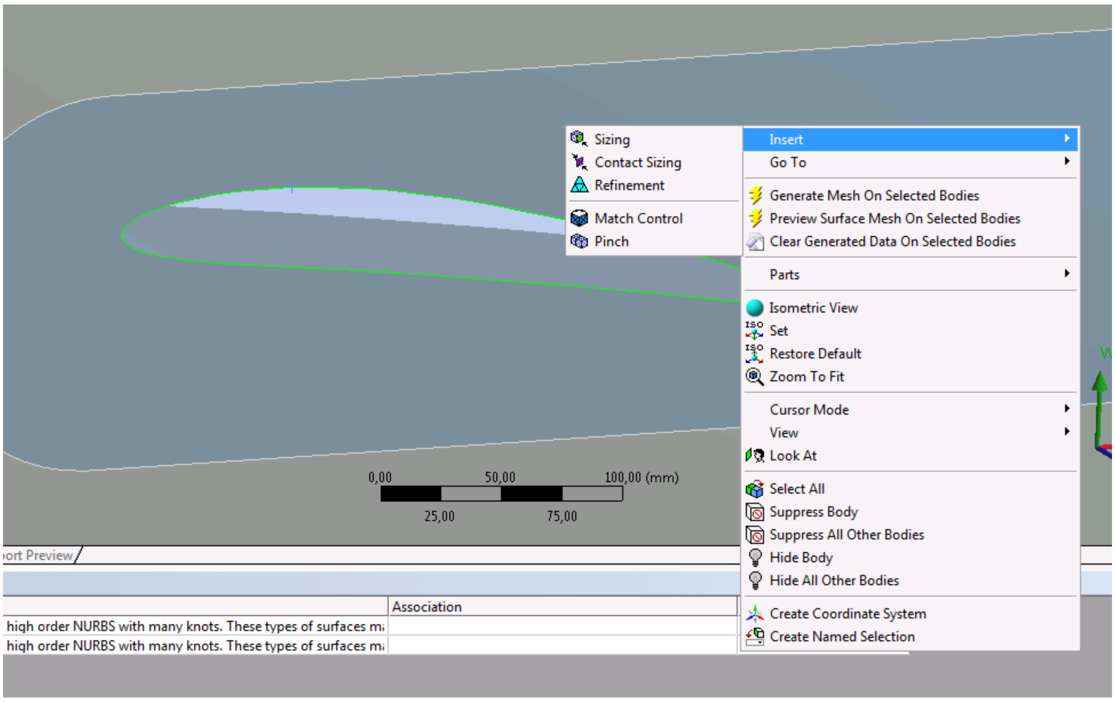
9. Select the body that contain the blade and right click on mesh in the tree and insert method.



10. Change the method to sweep.
11. Change source and target selection to “Manual Source”.
12. Select the surface at high Z and apply.
13. Change face mesh type to “All Tri”.
14. Set the sweep number of division to 20.
15. Select the outer solid and insert a second mesh method.
16. Change the method to tetrahedrons.

These two methods will be the base for the meshing. To preview the mesh so far right click on mesh in the outline tree and update the mesh. The next step in meshing will be to insert some local refinements.

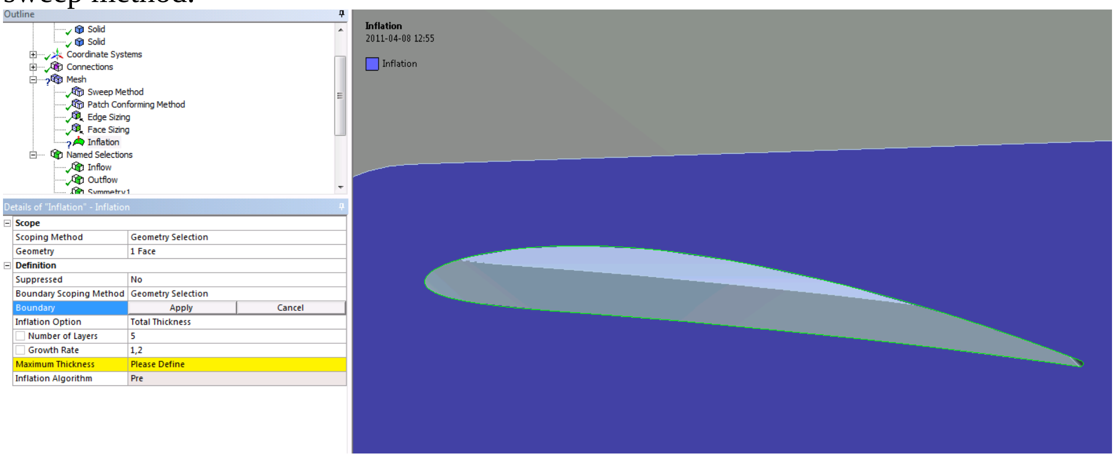
17. Select the edge representing the blade and right click and insert sizing.



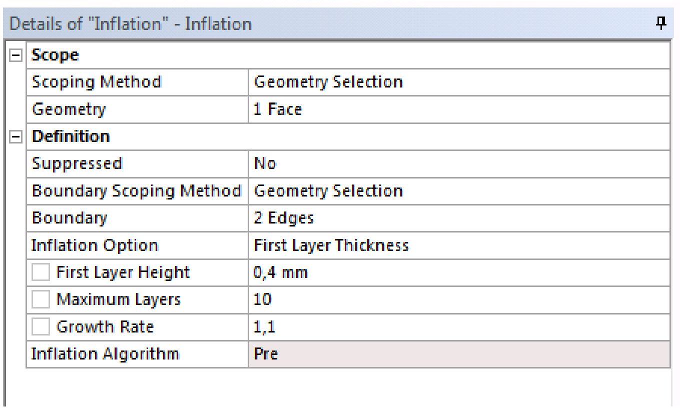
18. Set the mesh size in the detail view to 4 mm.

Finally we need to resolve the boundary layer a lot more to account for the velocity gradient. This is done using an inflation technique where the surface mesh is extruded in the normal direction in order to create thin prism elements.

19. Right click on the sweep method and select “Inflate this method”.
20. Select the edges that represent the blade on the source surface used in the sweep method.



21. Change the inflation option details to:



With these setting a mesh of about 155k nodes will be generated.

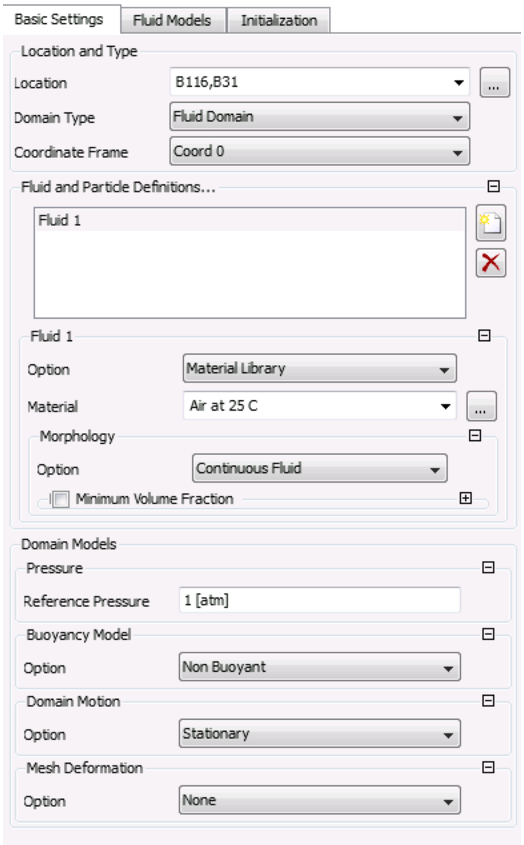
3.2. Creating the CFX case

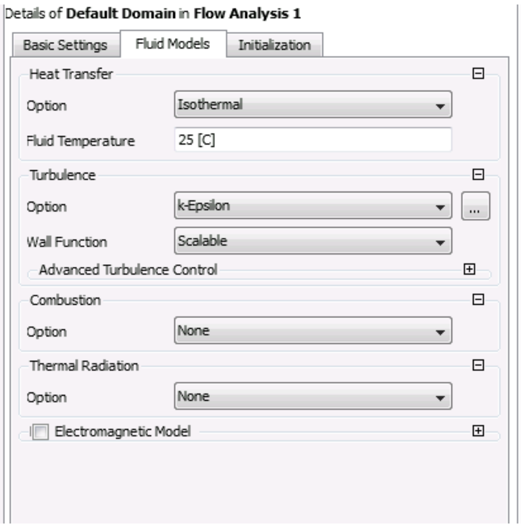
3.2.1. Physics

Now the mesh is completed and if the project isn't saved it is recommended to do so. The next step is to create the CFD simulation. To launch the pre-processor go back to the Workbench project page and right click on the Setup cell in the analyse system and select update. Double click thereafter on the same cell. This will bring up the CFX Pre-processor with the mesh loaded. The pre-processor will apply a set of default physics within the Default Domain.

1. Double click on the default domain. This will bring up the details of the physics that are applied.

The details show that the default fluid applied here is Air at 25 degrees C. This is what we want in this case so leave that as it is. The other settings on the basic tab are also correct for the type of simulation we will perform. The second tab is used to specify the physical models that we want to solve in this fluid domain.

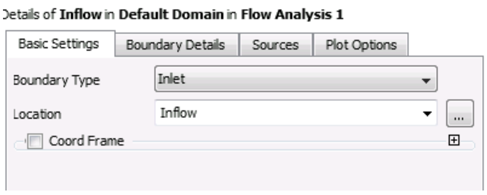


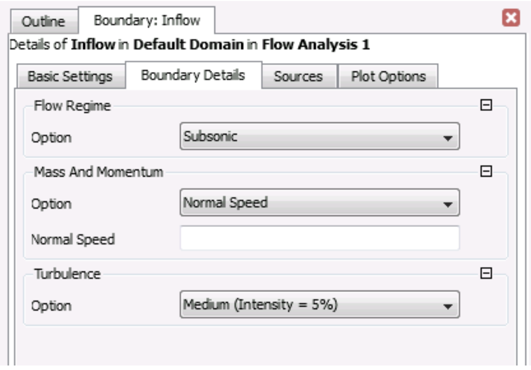


2. Change the turbulence model to shear stress transport (SST).

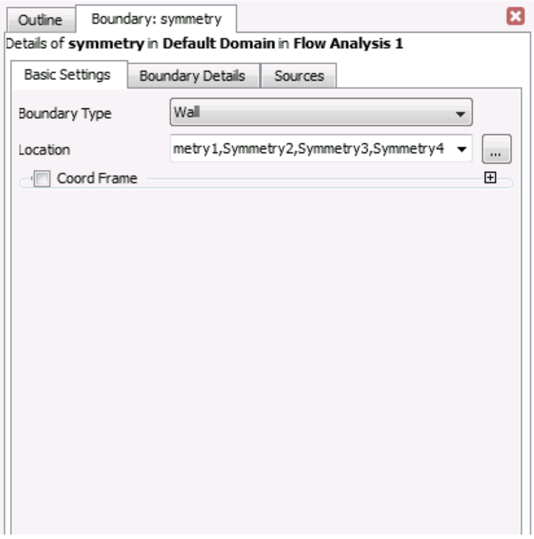
3.2.2. Boundary conditions

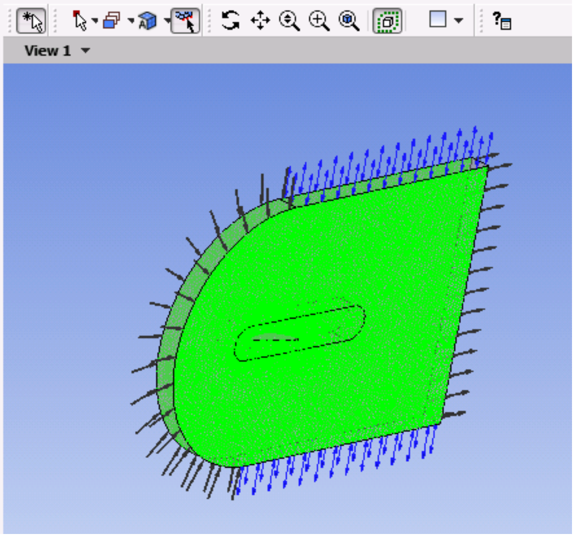
Now the different boundary conditions needs to be defined. Initially all surfaces are assigned a wall boundary condition with a no slip condition. When we apply a new condition to a surface this surface will automatically be removed from the default wall boundary so when we are done with the boundary all remaining surfaces are given a default wall condition.





1. Right click on the default domain and select insert boundary. Give the boundary Inflow as name. Since we have already given a named selection this name it will per default pick this surface as the surface for this boundary condition. If not chance the location to the correct surface or click in the graphics.
2. Change the mass and momentum option to Cartesian velocity component and set the velocity to 0.5 m/s in the x-direction and the turbulence to zero gradient.
3. Create a new boundary for the outflow. This time change the boundary type to outlet.
4. Set the relative pressure to 0 Pa using the "average static pressure option".
5. Create a new boundary condition for the entrainment surfaces. Change the boundary type this time to opening.
6. Specify the mass and momentum option to entrainment and set the relative pressure to 0 Pa and set the turbulence to zero gradient.
7. Create a new boundary condition and set the type to wall. Select the surfaces that describe the symmetry surfaces.



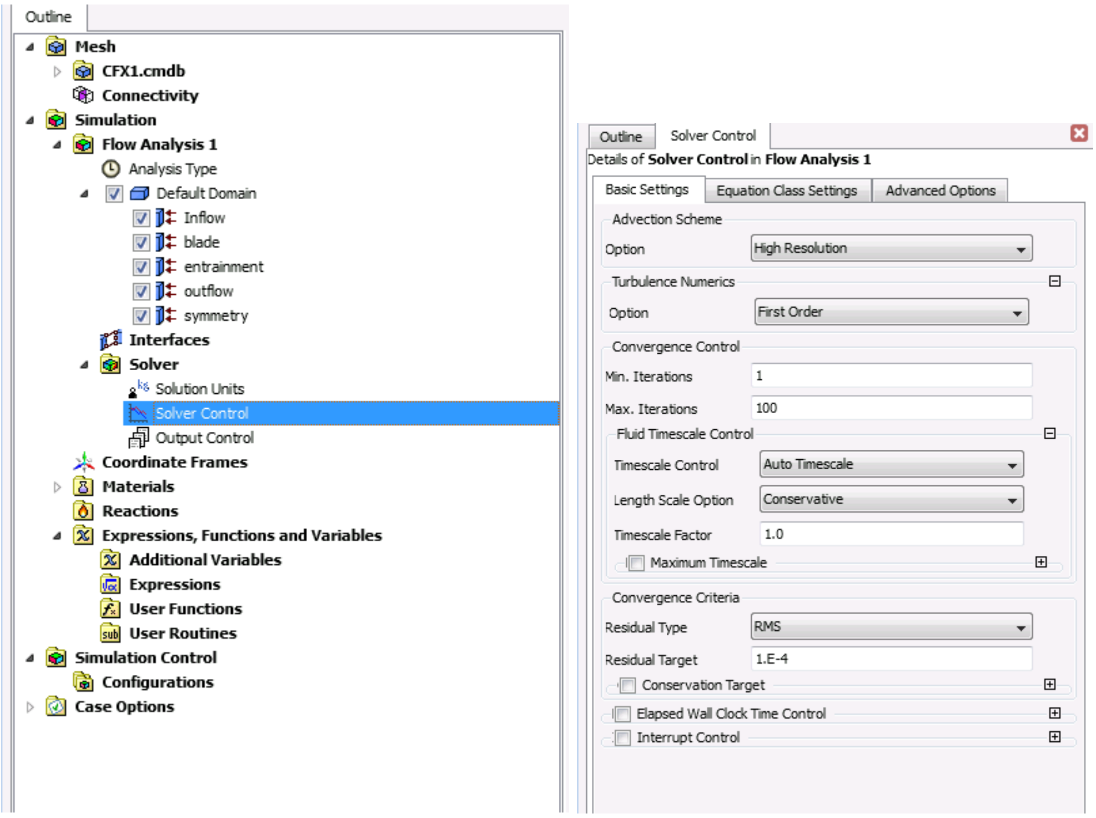


8. On the details setting change to free slip wall.
9. Finally create a condition with the name blade and specify it to the blade surface. Set the boundary type to wall and make sure it's a no slip wall.

Now all surfaces are specified and the default domain default condition is removed since it doesn't contain any surfaces.

3.2.3. Solver control

Normally the default solver controls are good for a standard case. Here we want to increase the convergence criteria and also the maximum number of iterations.

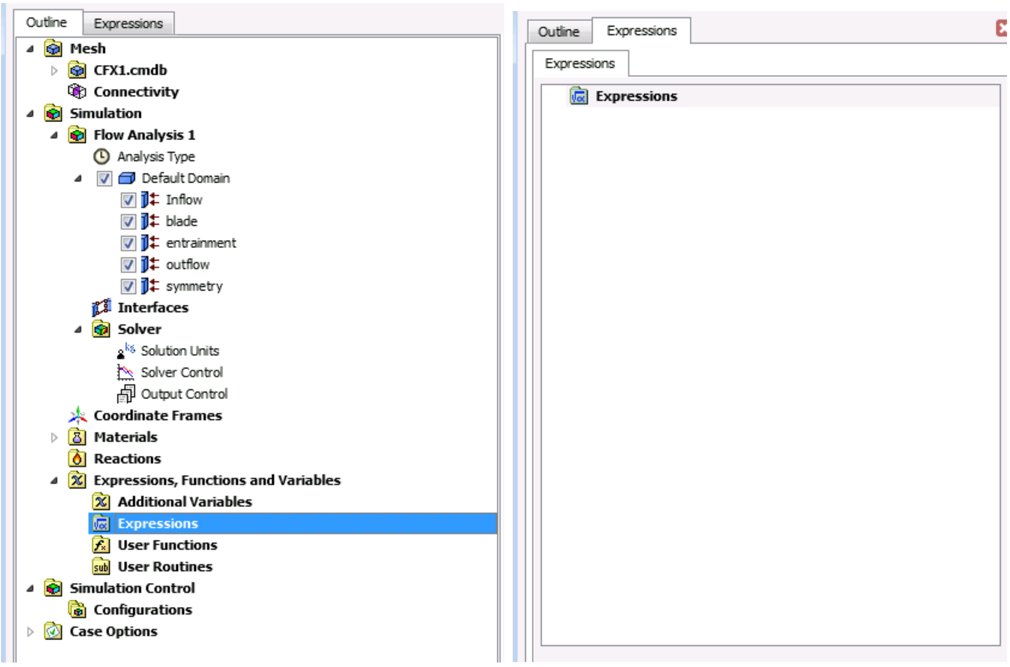


1. Double click on the solver control in the tree view. This brings up the solver settings.
2. Verify that the high resolution scheme is used.
3. Increase the maximum number of iterations to 200.
4. Change the convergence criteria to 1e-5.

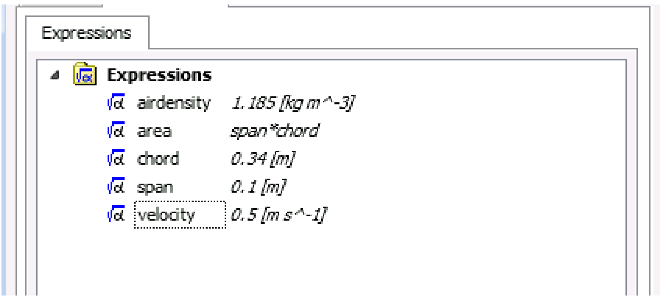
3.2.4. Solver monitoring.

Before the solver is started we want to specify some monitoring points. This will help us verify that the flow case is converged after reaching the convergence criteria. In this case the Cd and Cl are of interest so we want to use the definition of those and monitor them during the simulation. The way we do this is to create a few expressions that define the parameters of interest. Thereafter, these expressions are set as monitor points.

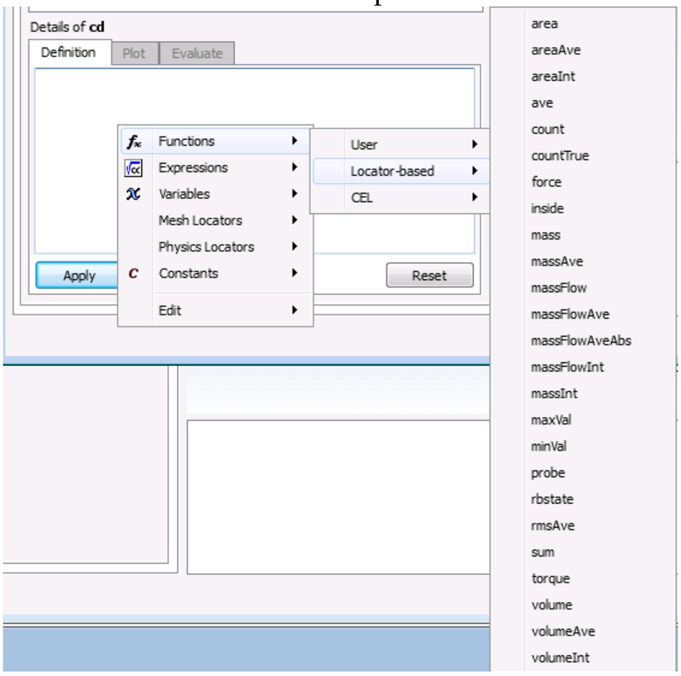
1. Double click on the expressions. This will bring up a blank field called the expression editor.



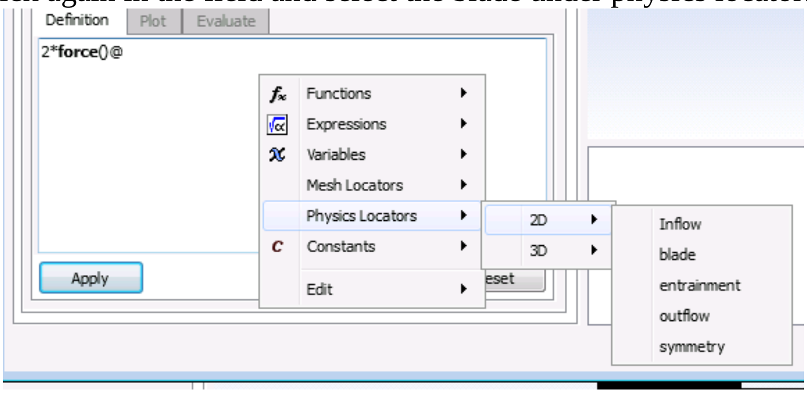
2. Right click on expressions in the expression editor and insert a new expression called span. Give the expression the value 0.1 [m]. The unit is very important since CFX keeps track of all units.
3. Continue to create an expression for chord, velocity, area and density (1.185 kg/m3).



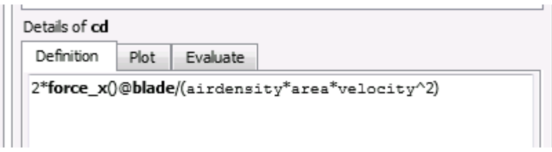
4. The Cd expression involves the free stream, velocity, area the force drag force on the blade. Right click on the expression to insert a new expression called cd.
5. Right click in the white field where the expressions are entered.



6. Select the force. This is a pre-defined expression already built in that gives us the total force on a surface we specify.
7. Right click again in the field and select the blade under physics locator.



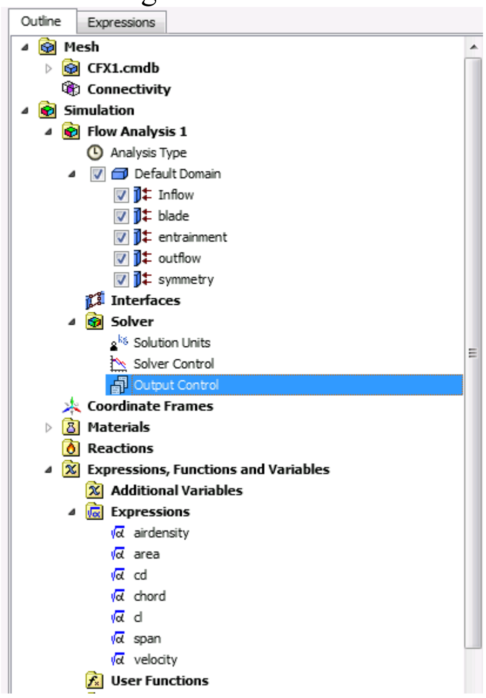
8. To get the force in the flow direction (i.e. x-direction) modify the expression to force_x()@blade.
9. Add the remaining variables that define cd. It should look something like this:

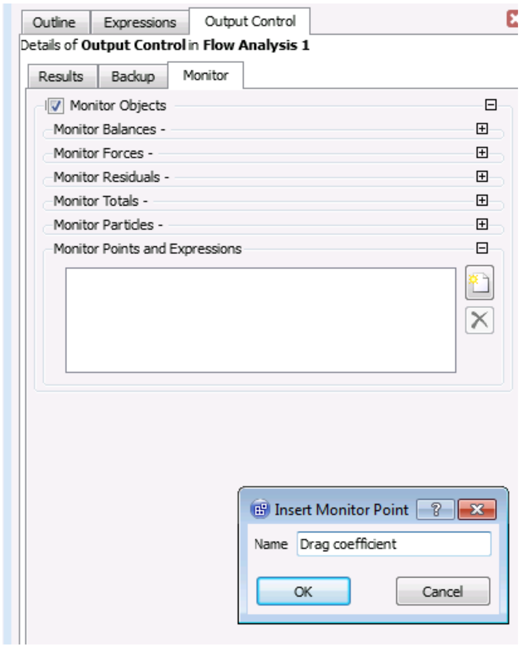


10. Create a similar expression for cl using the y component of the force.

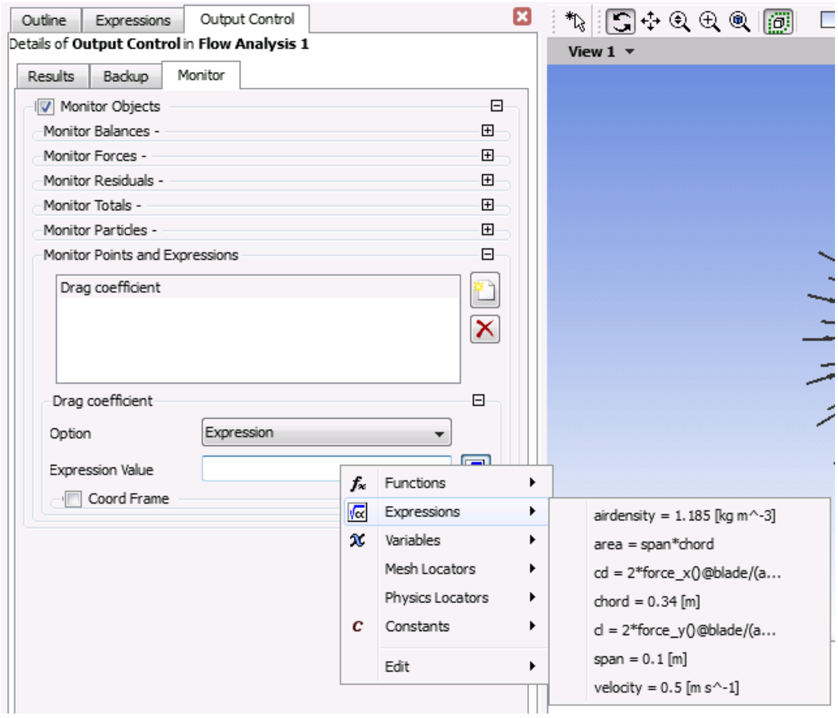
With these expressions in place we can now use the cl and cd expression as monitors during the simulation.

11. Double click on the output control. On the monitor tab create a monitor point called “drag coefficient”





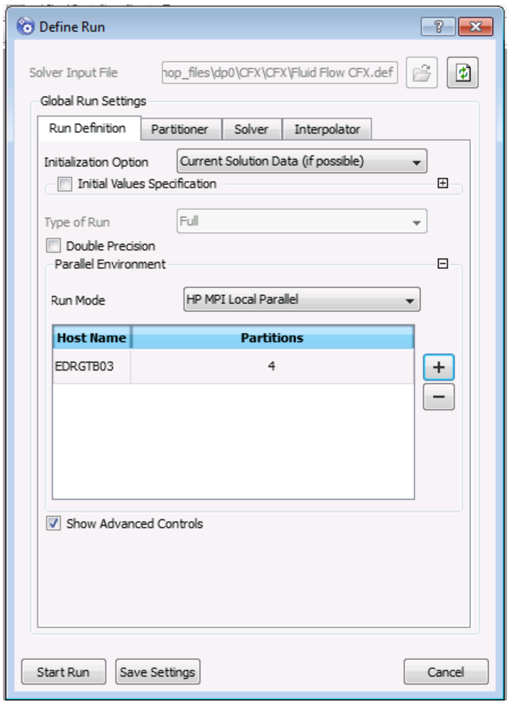
12. Specify the monitor point to use and expression. Right click in the white field and select the cd expression that we created above.



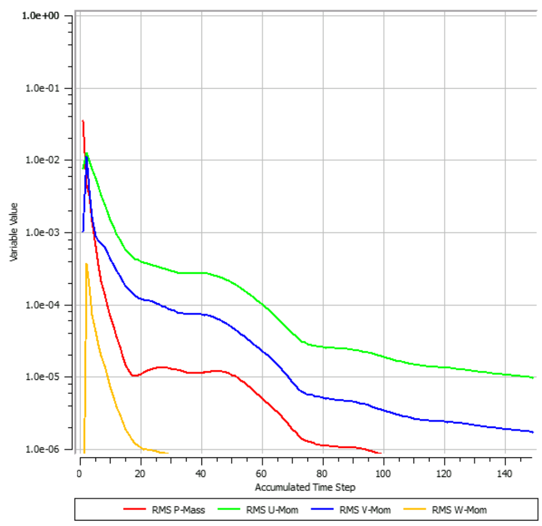
13. Create an addition monitor point for the lift coefficient.

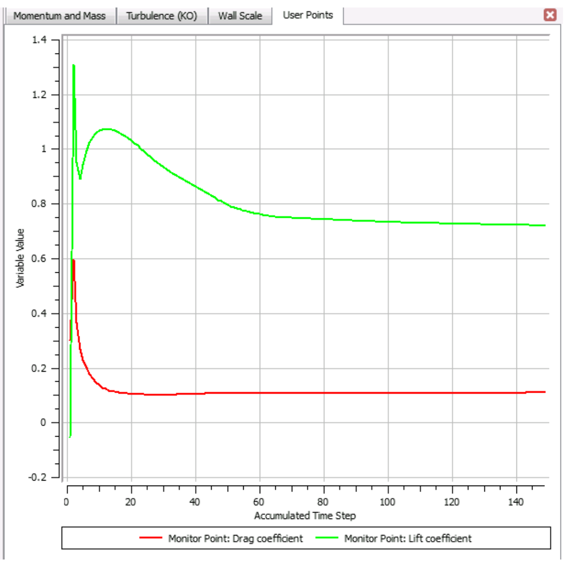
3.3. Run the solver.

Now everything is set up for this case. Save the project and change back to the workbench project page. To start the solver right click on the Solution cell in the Fluid Flow analyse and select edit. This will bring up the solver manager that submits the case to the solver. The solver manager helps us to specify the number of CPU cores we want to use and/or if we want to run locally or distributed over the network. Here we will run the case locally so change the start method to “HP MPI Local Parallel” and specify to the number of cores available on the computer.



Start the run. The solver residuals will show convergence after 140 iterations with the condition that we have set. However, if we look at the monitor points that we added we still see that the lift coefficient is not fully converged. This shows the importance to not only look at residuals when judging convergence. For many cases it is enough to look at residuals but it should in general be combined with monitor points.



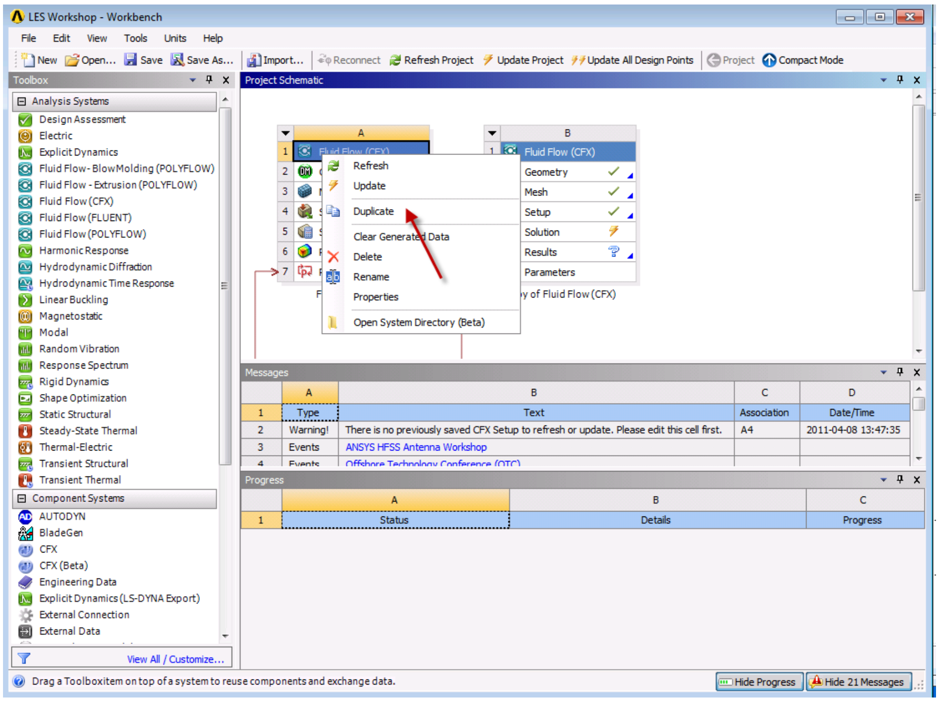


3.4. Investigate the results.

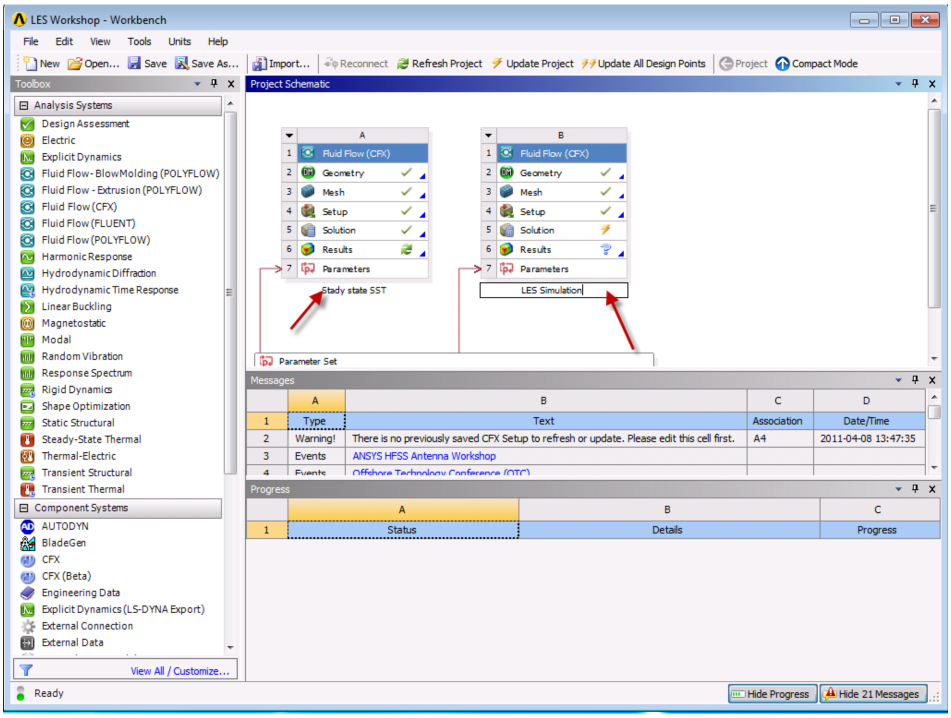
Switch back to the workbench project page and double click on the results cell in the analysis system. Try to add a plane through the domain and plot for example pressure and velocity. Add velocity vectors to the plane as well and perhaps some streamlines through the domain.

4. LES simulation.

The mesh for the LES simulation will be similar to the mesh used in the steady state simulation. We will refine slightly around the blade and refine the boundary layer. Apart from that we will use identical settings. Thus, to simplify the creation we start by duplicating the analyse system from the first case and make use of the work we have already put into the model. Navigate to the workbench project page and right click on the first cell (A) of the steady state system. Select duplicate to create an identical analyse.



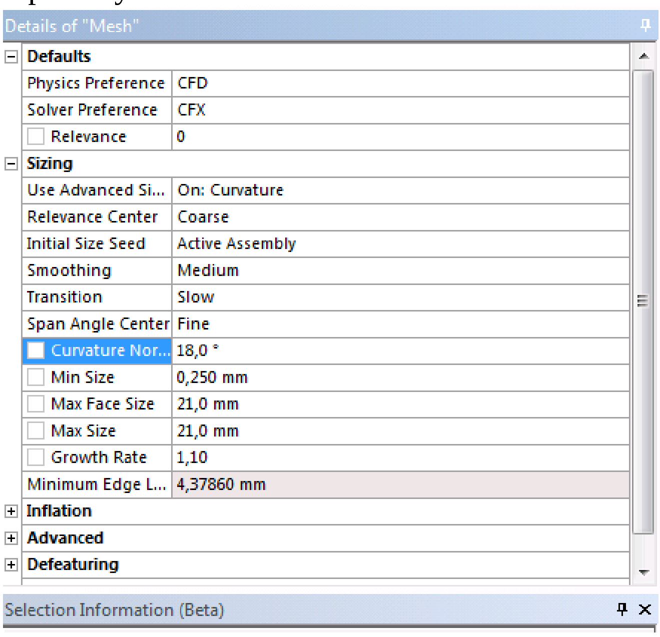
In order to remember which of the boxes that correspond to the steady state and the new one that we will use for LES we can rename the systems. This is more of a cosmetic matter.



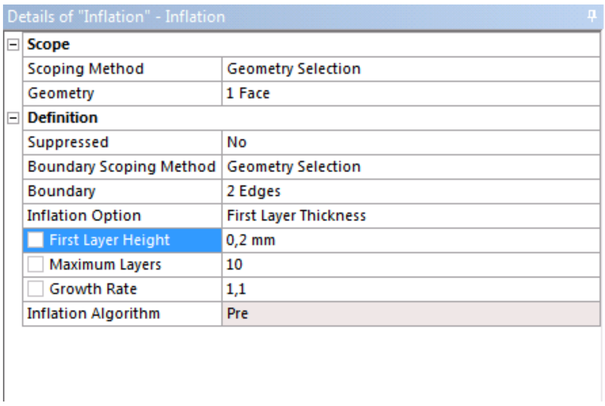
4.1. Mesh generation

Double click the meshing cell of the new system to start the meshing application.

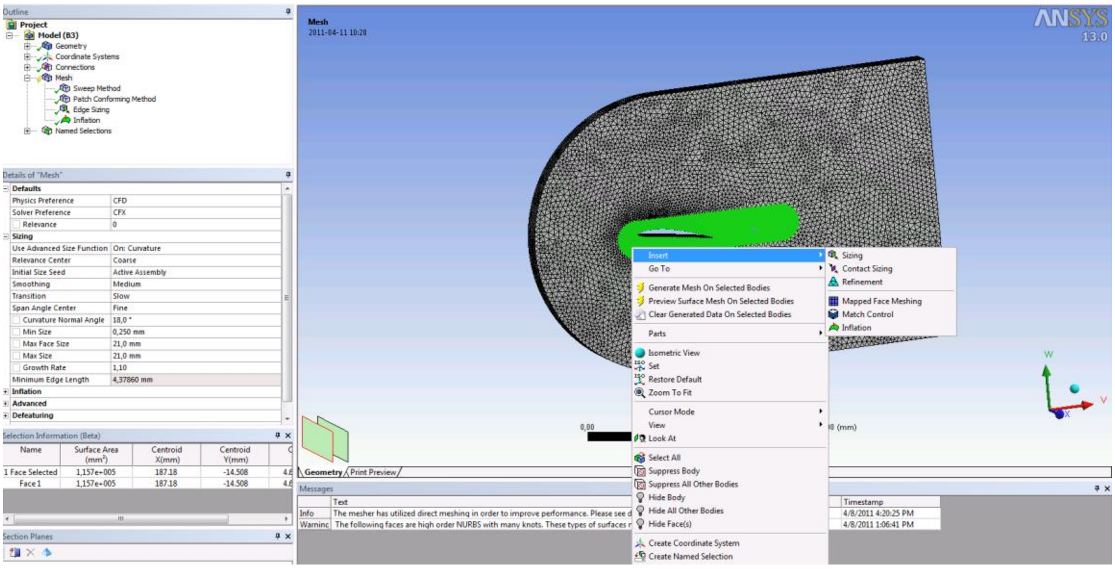
1. The first change we will do is to change the angular resolution on curved surfaces from 25 degrees to 18 degrees. A lower value here will resolve curved surfaces more and we will thus create more elements around the training edge especially.



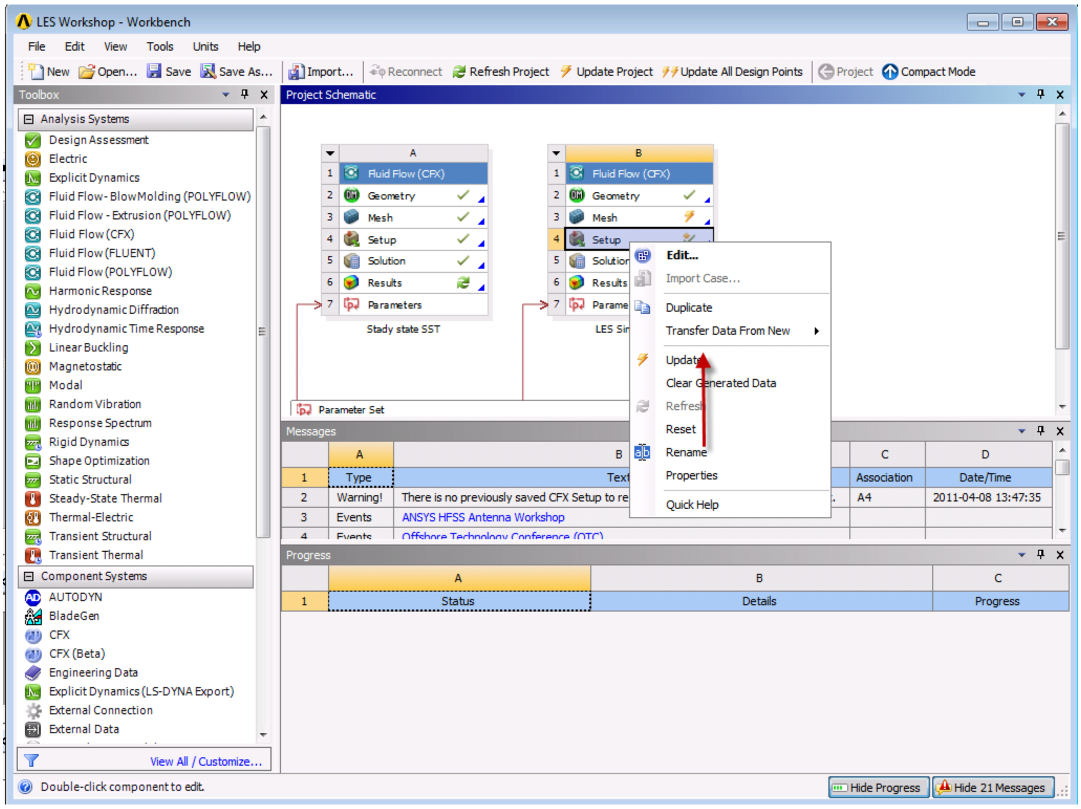
2. Change the first layer thickness of the inflation layers to 0.2 mm.



3. The third change we will do is to add a face sizing on the face of the cut out section. Set the element size to 5 mm



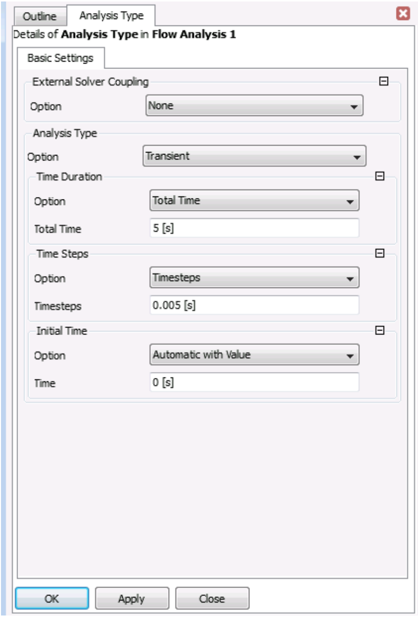
4. Jump back to the workbench project page and right click on the setup line and select update. This will regenerate the mesh and reload the mesh into the CFX pre-processor. Save the case.



4.2. Modifying the case.

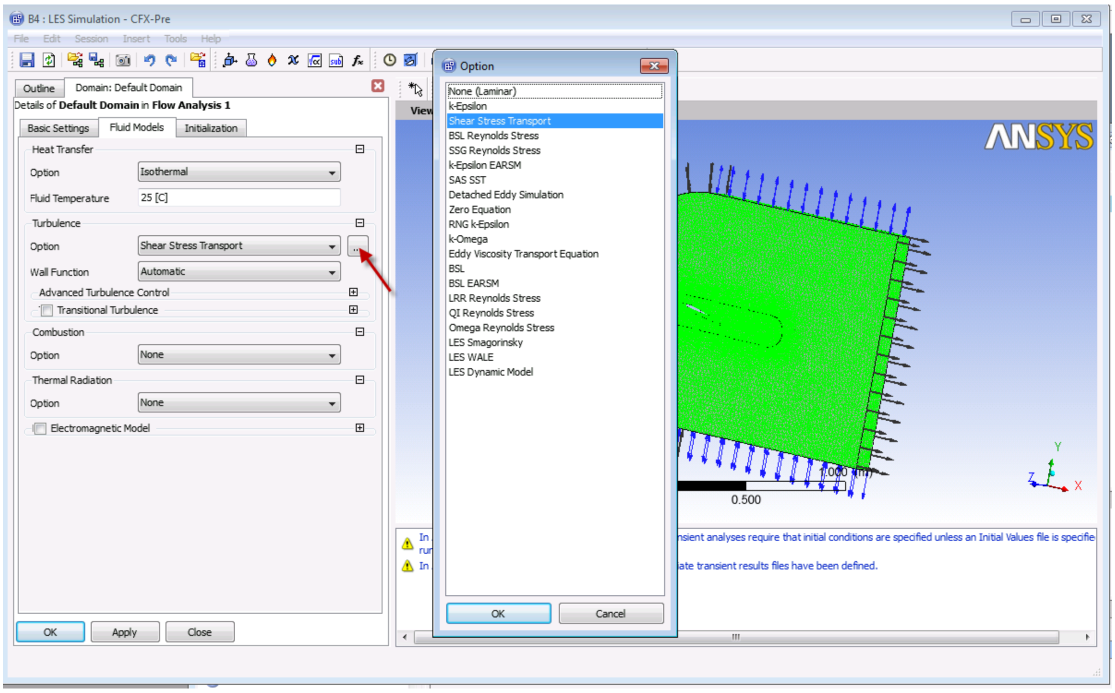
Start the CFX pre-processor of the new analysis system. This will bring up the exact same case as the steady state case but with the new refined mesh created. The LES technique is by default transient so we need to change the simulation into a transient run.

1. Right click on “Analysis Type” in the tree and select edit. This will bring up the following formula. Change the setting to the following and apply.



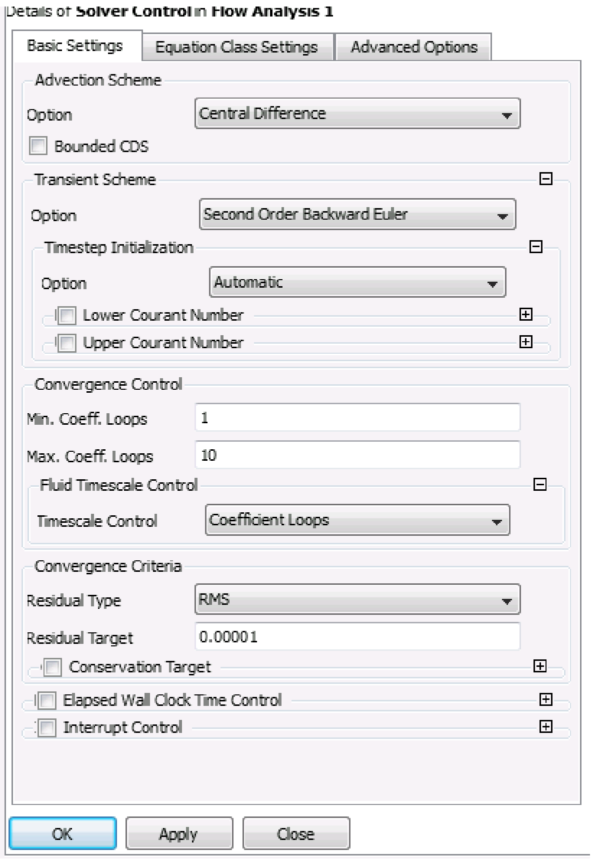
2. Now we need to change the turbulence model to the Dynamic LES model. Double click on the domain and change to the Fluid Models tab where the physics are defined.

3. Click on the button with "…" to bring up a dialog box with all available turbulence models. Select the LES Dynamic Model and press ok. Confirm the physics changes with ok.



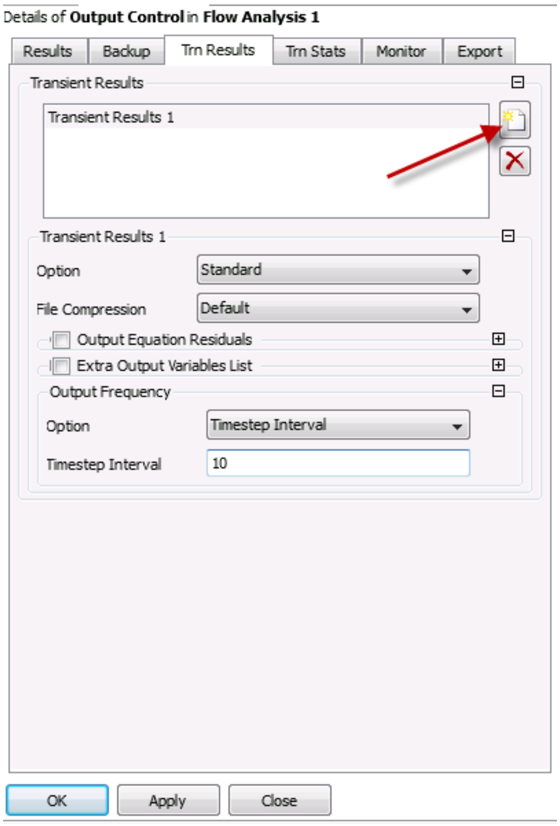
The change in turbulence model results in some warnings on the inflow and entrainment boundary. The reason for this is that the two equation model had some turbulence conditions that are not needed for the LES model. Thus, we need to update the inflow and entrainment condition. Double click on the inflow condition and click ok directly without doing anything. Repeat this for the entrainment condition and bot warnings will be taken care of.

When running LES cases central differencing schemes are normally used and CFX also gives us a recommendation to change numerical scheme. Double click on the solver control and change to central differencing.



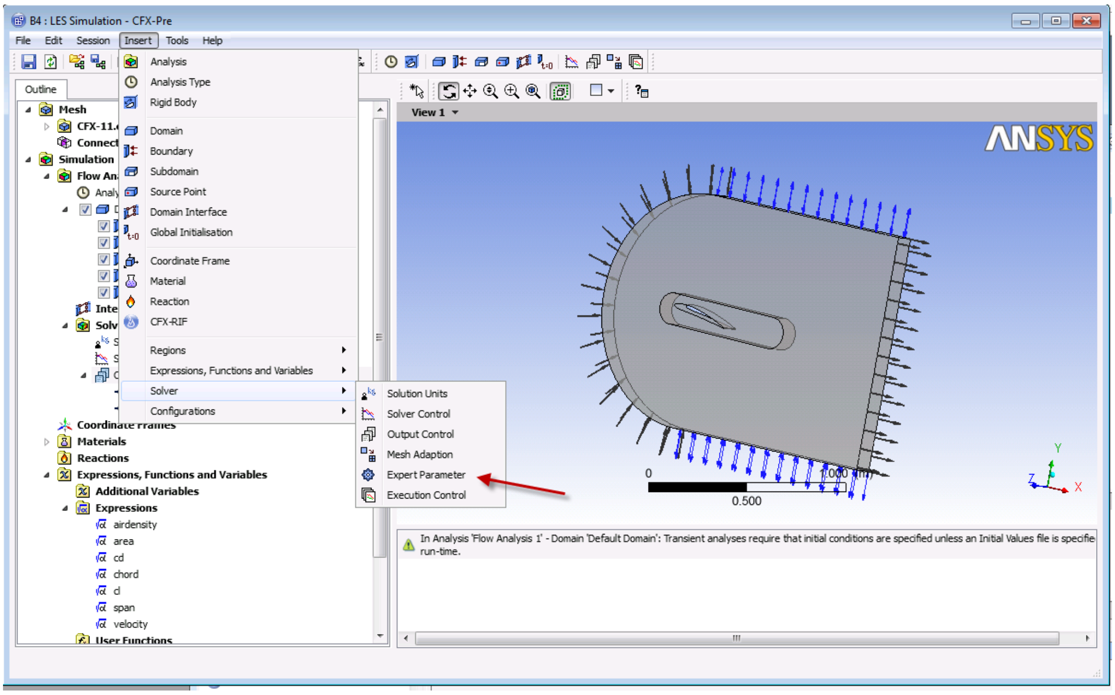
Additionally, we are encouraged to add some intermediate result files. This is useful if we would like to animate the results. Also, if the simulation for some reason crashed due to full hard drive or license failure it is always useful to be able to restart the analysis.

1. Double click on output control and change to the Trn Results tab.
2. Click on the new button and set the time step interval to 10 and press ok.

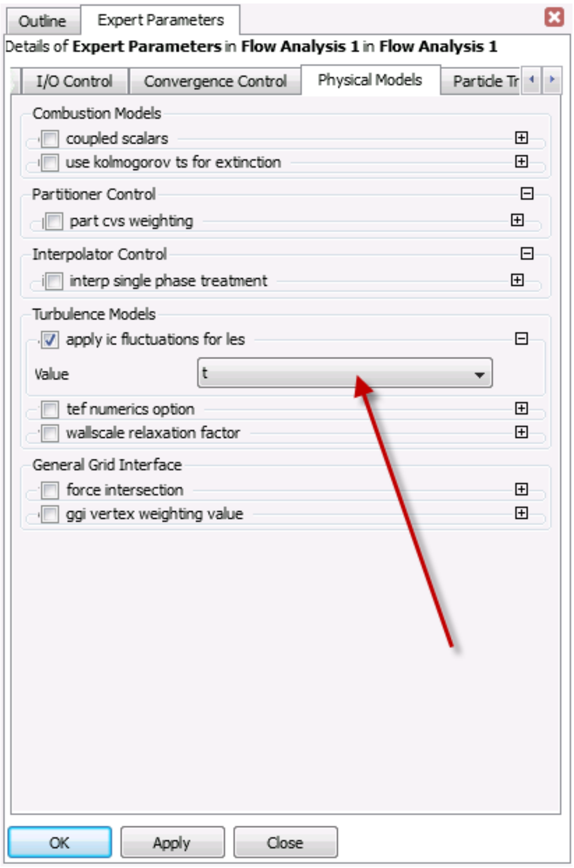


For this simulation we will use the results from the steady state simulation as input and thus we don't need to add any initial conditions so we can ignore that warning. In order to initiate disturbances to the flow and trigger some turbulence we want to apply some random fluctuations to the initial results that we will be using. This is an expert option that needs to be added.

1. Click insert in the top meny and select expert parameter in the solver section. This will bring up some of the advanced features that are available within the pre-processor.



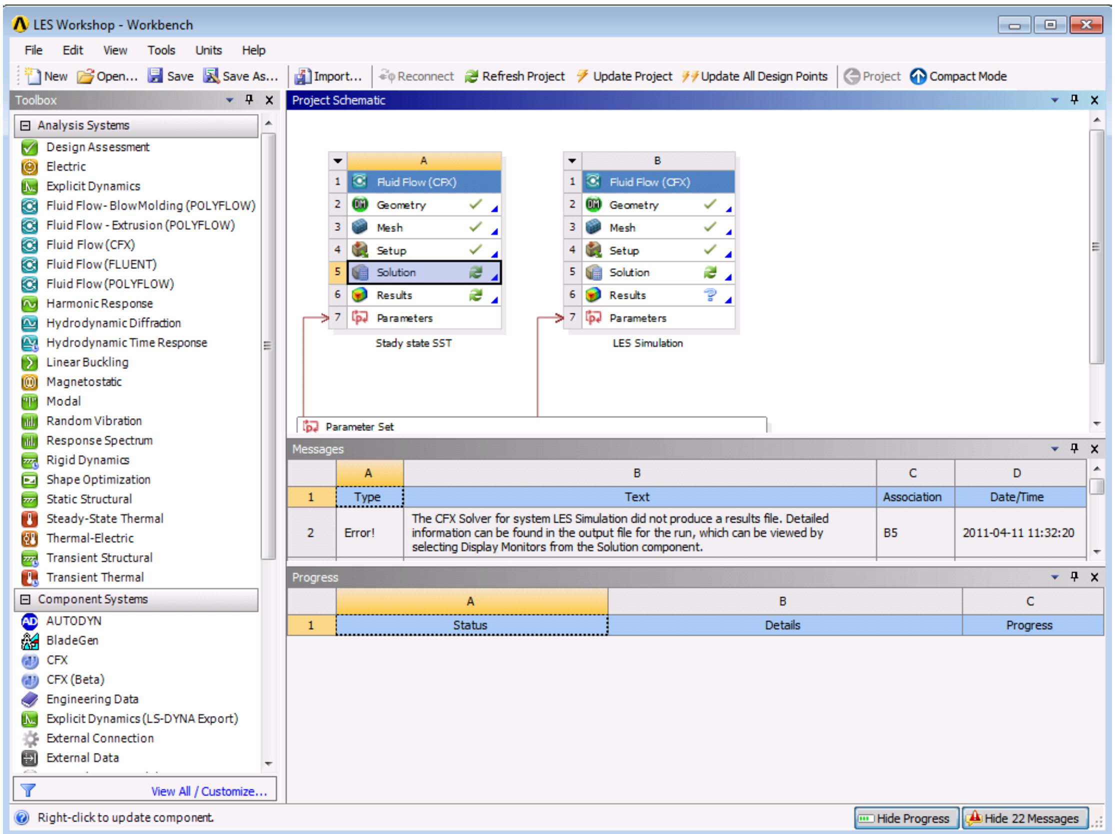
2. Change to the “Physical Models” tab. Hit the check box in from of the option “apply ic fluctuations for les” and set this value to true (t) and confirm with ok.



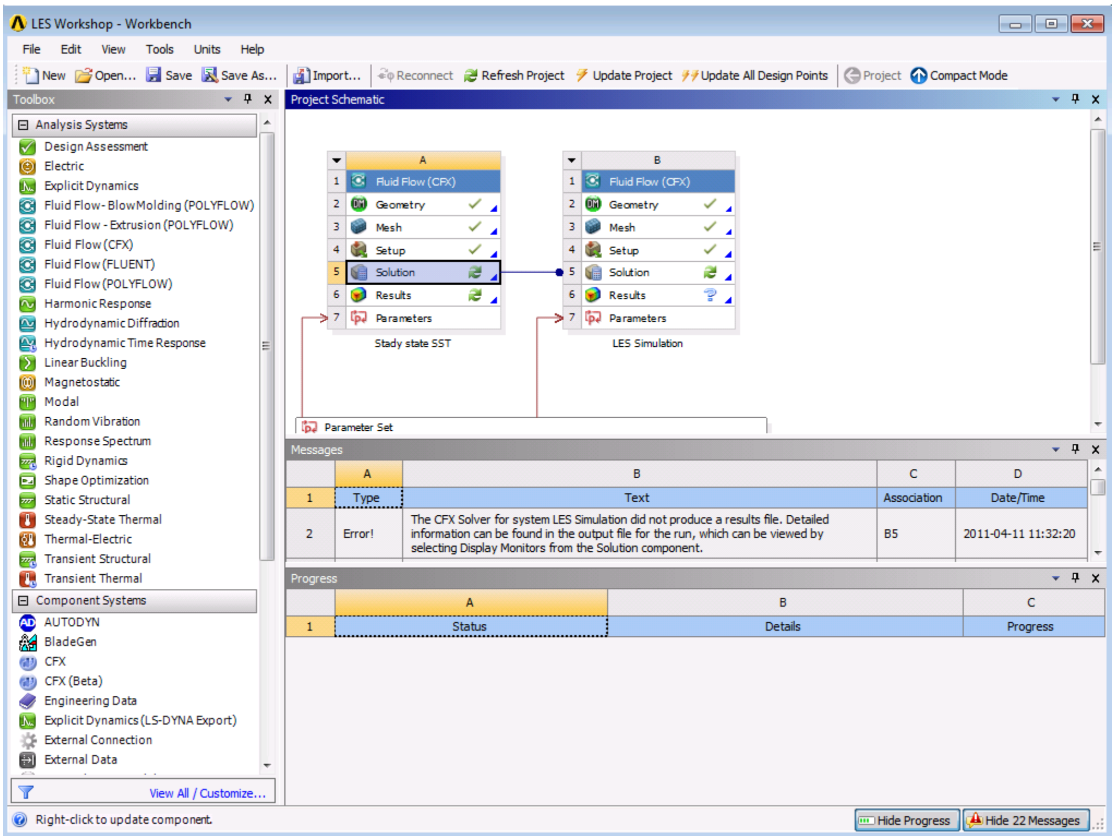
3. Save the project.

4.3. Run the solver.

Now the LES-simulation is set up and is ready for execution. In order to restart form the previous run, we need to couple these analysis systems together. This is done by clicking and holding the left mouse button on the “Solution” cell of the first system. While holding the mouse button, drop the solution cell on top of the solution cell within the new analysis system. This will create a link that informs ANSYS that the results in system A will be used as input in system B.

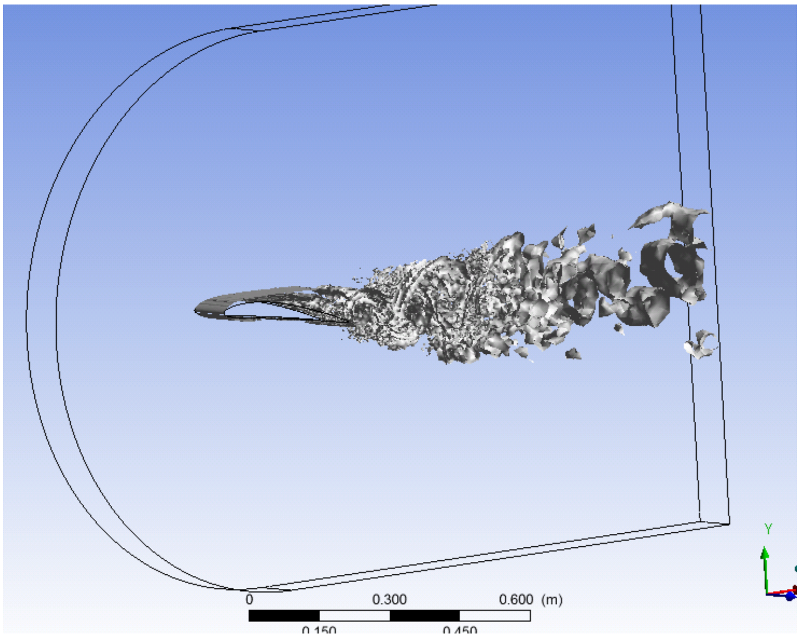


The workbench page should look something like this after this operation:



Right click on the "Solution" cell in system B and select edit to start the solver manager. Notice that workbench automatically have selected the results from the previous analysis as input. Change the parallel setting to resemble the system you are in front of and hit start solver. This run will take some time ….

When the simulation has started you can read in intermediate results to investigate how the solution is progressing. When the solution is done enter the post processor to look at the results.


0
0.300
0.600 (m)
0.150
0.450
Y