

LES Workshop within the Masters Course in Turbulence Modeling at Chalmers

REV.	DATE]	REASON F	OR ISSUE	PREPARED	CHECKED	
CLIENT	•			DOCUMENT TITLE:			
COMPA	NY LOGO)					
				LES Workshop within the	e Masters C	ourse in	
				Turbulence Modeling at Chalmers			
				ε			
				PROJECT NAME:			
			2011-01_LES Workshop				
				DOCUMENT NUMBER:			
C 1. 1.				2011 01 LEC Werlichen 1			
Chalmers				2011-01_LES Workshop_1			
NO. of P	ages	Rev.		ΡΚΕΡΑΚΕΟ ΒΥ:			
2	9		1	KLAS JOHANSSON			
_	-		-	Vlas ichongen Øedra			
				<u>mas.jonansson@edr.no</u>			

6		
2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

Table of Contents

1.	INTRODUCTION	2
2.	GENERAL CONSIDERATIONS	2
3.	STEADY STATE SIMULATION	3
3.1	1. Mesh generation	3
3.2	2. CREATING THE CFX CASE	7
	<i>3.2.1. Physics</i>	7
	3.2.2. Boundary conditions	8
	3.2.3. Solver control	10
	3.2.4. Solver monitoring	11
3.3	3. RUN THE SOLVER.	14
3.4	4. INVESTIGATE THE RESULTS	16
4.	LES SIMULATION	16
4.1	1. Mesh generation	17
4.2	2. MODIFYING THE CASE	19
4.3	3. Run the solver.	24

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

Engin	eering Data Resources a.s		
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

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.

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

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.

() Options		- • •
Project Management Appearance Regional and Language Options Graphics Interaction Geometry Import Journals and Logs Project Reporting Solution Process Mechanical APDL CFX FLUENT Merchanical Microsoft Office Excel Meshing Design Exploration	Project Management File Locations Default Folder for Permanent Files C: \Users \Users EDRLAN\Documents Folder for Temporary Files C: \Users \Users \Users \Users EDRLAN\Documents Startup Image: Comparent of News Messages Maximum Age of News Messages (Days) 90 Custom RSS Feed Address Image: Start Remote Solve Manager Image: Show Getting Started Dialog	Browse
Restore Defaults		OK Cancel

- 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.

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		



- 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:

De	tails of "Mesh"		
-	Defaults		*
	Physics Preference	CFD	
	Solver Preference	CFX	
	Relevance	0	
Ξ	Sizing		
	Use Advanced Size Function	On: Curvature	
	Relevance Center	Coarse	
	Initial Size Seed	Active Assembly	
	Smoothing	Medium	Ξ
	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		-
Se	lection Information (Beta)	Ţ	×

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

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.

-			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		



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.



2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

21. Change the inflation option details to:

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	·
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	2 Edges
Inflation Option	First Layer Thickness
First Layer Height	0,4 mm
Maximum Layers	10
Growth Rate	1,1
Inflation Algorithm	Pre

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.

-			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

Basic Settings Flu	uid Models Initialization				
-Location and Type-					
Location	B116,B31	▼			
Domain Type	Fluid Domain	•			
Coordinate Frame	Coord 0	•			
-Fluid and Particle De	finitions				
Fluid 1					
Fluid 1			Details of Default Doma	ain in Flow Analysis 1	
Option	Material Library	•	Basic Settings Fluid	d Models Initialization	
Material	Air at 25 C	▼	Heat Transfer		
Morphology			Option	Isothermal 👻	
Option	Continuous Fluid	•	Fluid Temperature	25 [C]	
I Minimum Vo	lume Fraction	€	Turbulence		
Domain Models			Option	k-Epsilon 👻	
Pressure			Wall Function	Scalable 👻	
Reference Pressure	1 [atm]		Advanced Turbulence	e Control	⊞
Buoyancy Model			Combustion		
Option	Non Buoyant	•	Option	None	
Domain Motion			Thermal Radiation		
Option	Stationary	-	Option	None	
Mesh Deformation			Electromagnetic N	Model	•
Option	None	•			

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.

-			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

						Outline	Boun	ndary: Inf	low				×
						Details of I	Inflow in	n Defaul	t Domain in	Flow Anal	ysis 1	_	
Details of Inflow in	n Default Domain in	Flow Analy	/sis 1			Basic Se	ettings	Bound	ary Details	Sources	Plot Options		
Basic Settings	Boundary Details	Sources	Plot Options			Flow R	egime						Ξ
Boundary Type	Inlet			•		Option			Subsonic			•	
Location	Inflow			•		Mass /	And Mom	nentum					Ξ
Coord Fran	ne				Ð	Option			Normal Spee	d		•	
						Normal	Speed						
						Turbul	ence						-8
						Option			Medium (Int	ensity = 5%)	•	

- 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.

Outine Boundary: symmetry X Details of symmetry in Default Domain in Flow Analysis 1 1	ĨŊ Ŋ Ĩ ヽ ゚ ヽ ゚ ヽ ゚゚゚゚゚ヽ ゚゚゚゚゚゚゚゚゚゚゚゚゚゚゚゚
Basic Settings Boundary Details Sources	
Boundary Type Wall Location metry1,Symmetry2,Symmetry3,Symmetry4 Coord Frame Image: Coord Frame	
	The second secon

¥		
2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

- 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.

Outline		
🔺 🞯 Mesh		
🖻 🙆 CFX1.cmdb		
Connectivity		
4 👰 Simulation		
4 🔞 Flow Analysis 1	Outline Solver Control	
Analysis Type	Details of Solver Control in Flow Analysis 1	
4 📝 🗇 Default Domain	Basic Settings Equation Class Settings Advanced Options	
Inflow	Advection Scheme	
🔽 🕽 🏝 blade	Option High Resolution	
🔽 🕽 🗱 entrainment		
✓ J‡ outflow	Turbulence Numerics	
📝 🕽 🎝 symmetry	Option First Order 👻	
Interfaces	Convergence Control	
4 👰 Solver	Min Iterations 1	-
2 ^{ks} Solution Units		
Solver Control	Max. Iterations 100	
Dutput Control	Fluid Timescale Control	-8-
🎠 Coordinate Frames	Timescale Control Auto Timescale 👻	
🖻 🙆 Materials		
🙆 Reactions	Conservative V	
Expressions, Functions and Variables	Timescale Factor 1.0	
🔀 Additional Variables	Maximum Timescale	- ±
👼 Expressions	Conversioner Oritaria	
🔀 User Functions		- I
📠 User Routines	Residual Type RMS	
4 😰 Simulation Control	Residual Target 1.E-4	
Configurations	Conservation Target	÷
Case Options	Elapsed Wall Clock Time Control	Đ
	Interrupt Control	Đ

- 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.

Engin	eering Data Resources a.s		
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

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).

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

Expressions	
4 👿 Expressions	
va airdensity	1. 185 [kg m^-3]
va area	span*chord
va chord	0.34 [m]
va span	0.1 [m]
velocity	0.5 [m s^-1]

- 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.

				-	
Details of cd					area
Definition	Plot	Evaluate			areaAve
					areaInt
					ave
	f.	Functions	•	Lines	count
		Evoressions		User	countTrue
	<u>va</u>	Variablee		Locator-based	force
	~	variables	Į,	CEL	▶ inside
		Please Locators			mass
	-	Physics Locators			massAve
Apply	C	Constants	·	Reset	massFlow
	-	Edit	•		massFlowAve
					massFlowAveA
					massFlowInt
					massInt
					maxVal
					minVal
					probe
					rbstate
					rmsAve
					sum
					torque
					volume
					volume volumeAve

- 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.



2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

- 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"

Outline Expressions			
🔺 🚱 Mesh	*		
FX1.cmdb		Outline Expressions Output Control	E
Connectivity		Details of Output Control in Flow Analysis 1	
A 😨 Simulation			
4 😧 Flow Analysis 1		Results Backup Monitor	
Analysis Type		Monitor Objects	
4 📝 🗇 Default Domain		Monitor Balances -	Ŧ
🔽 🕽 🏝 Inflow		Monitor Forces -	Ŧ
V 1t blade		Monitor Residuals -	Ξ
✓ J‡ entrainment		Monitor Tetalo	
✓ J‡ outflow		Maritas Pasilalas	
V D symmetry		Monitor Particles -	±
11 Interfaces		Monitor Points and Expressions	
▲ Solver			
2ºº Solution Units	E		
Solver Control			
Coordinate rrames			
Reactions A Strategiese Functions and Variables			
Additional Variables			
airdensity			
d area		(a	
v cd		1 Insert Mon	itor Point 🔐 🔜
d chord		Name Drag co	efficient
va d			
√α span		ОК	Cancel
velocity			
🔀 User Functions			

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

_			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

ails of Output Control in Flow A	Control			*\$; `S ↔ Q, ⊕, @, [<u>@</u> View 1, ▼	J
Results Backup Monitor					
Monitor Objects		Ξ			
Monitor Balances -		Ŧ			
Monitor Forces -		Ŧ			
Monitor Residuals -		+			
Monitor Totals -		ŧ			
Monitor Particles -		ŧ			
-Monitor Points and Expressions-					
Drag coefficient					
		X			
Dura suefficient					
Drag coefficient					
Drag coefficient Option Expres	sion				
Drag coefficient Option Expression Value	sion				
Drag coefficient Option Express Expression Value Coord Frame	sion f×	Functions	•		
Drag coefficient Option Express Expression Value Coord Frame	sion f~ lec	Functions Expressions	•	airdensity = 1.185 [kg m^-3]	
Drag coefficient Option Expression Value Coord Frame	sion fr kc X	Functions Expressions Variables	• •	airdensity = 1.185 [kg m^-3] area = span*chord	
Drag coefficient Option Expression Value Coord Frame	sion fr C X	Functions Expressions Variables Mesh Locators	• • •	airdensity = 1.185 [kg m^-3] area = span*chord cd = 2*force_x()@blade/(a	
Drag coefficient Option Expression Value Coord Frame	sion f~ cc X	Functions Expressions Variables Mesh Locators Physics Locators	> > > > >	airdensity = 1.185 [kg m^-3] area = span*chord cd = 2*force_x()@blade/(a chord = 0.34 [m]	
Drag coefficient Option Express Expression Value Coord Frame	sion fr k k c	Functions Expressions Variables Mesh Locators Physics Locators Constants	> > > > > > >	airdensity = 1.185 [kg m^-3] area = span*chord cd = 2*force_x0@blade/(a chord = 0.34 [m] d = 2*force_y0@blade/(a	
Drag coefficient Option Express Expression Value Coord Frame	sion fr C	Functions Expressions Variables Mesh Locators Physics Locators Constants Edit	> > > > > > > >	airdensity = 1.185 [kg m^-3] area = span*chord cd = 2*force_x()@blade/(a chord = 0.34 [m] d = 2*force_y()@blade/(a span = 0.1 [m]	

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.

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

💿 Define Run	? 🗙
Solver Input File	hop_files\dp0\CFX\Fluid Flow CFX.def
Global Run Settin	igs
Run Definition	Partitioner Solver Interpolator
Initialization Opt	ion Current Solution Data (if possible) Use Specification
Type of Run	Full
Double Preci Parallel Enviro	nment 🛛
Run Mode	HP MPI Local Parallel 🗸
Host Name	Partitions
EDRGTB03	4 +
V Show Advar	iced Controls
Start Run Sa	ve Settings Cancel

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.



Engin	eering Data Resources a.s		
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

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.

-			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		



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.

-	1 2		
De	etails of "Mesh"		ф
-	Defaults		*
	Physics Preference	CFD	
	Solver Preference	CFX	
	Relevance	0	
-	Sizing		
	Use Advanced Si	On: Curvature	
	Relevance Center	Coarse	
	Initial Size Seed	Active Assembly	
	Smoothing	Medium	
	Transition	Slow	Ξ
	Span Angle Center	Fine	
	📃 Curvature Nor	18,0 °	
	Min Size	0,250 mm	
	Max Face Size	21,0 mm	
	Max Size	21,0 mm	
	Growth Rate	1,10	
	Minimum Edge L	4,37860 mm	
+	Inflation		
+	Advanced		
+	Defeaturing		Ŧ
~			
26	lection information	t (Beta) 4	×

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

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

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	^
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	2 Edges
Inflation Option	First Layer Thickness
📃 First Layer Height	0,2 mm
Maximum Layers	10
Growth Rate	1,1
Inflation Algorithm	Pre

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.

<u> </u>			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		



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.

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

tails of Analysis Ty	pe in Flow Analysis 1	
Basic Settings		
External Solver Co	upling	Ξ
Option	None	•
Analysis Type		
Option	Transient	•
Time Duration		
Option	Total Time	•
Total Time	5 [s]	
Time Steps		E
Ontion	Timestens	
	0.005 []	•
Timesteps	0.005 [S]	
Initial Time		
Option	Automatic with Value	•
Time	0 [s]	

- 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.



Engineering Data Resources a.s	Engineering Data Resources a.s	
--------------------------------	--------------------------------	--

2011-01_LES Workshop	Rev.	1
LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
Chalmers		

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.

Basic Settings	Equation Class Settings Adva	nced Options
Advection Scher	ne	
Option	Central Difference	•
Bounded CDS	3	
Transient Schem	e	Ξ
Option	Second Order Backward Eu	Jer →
-Timestep Initia	lization	
Option	Automatic	•
-I Lower Co	ourant Number	
-I Upper Co	ourant Number	Đ
Convergence Cr	natral	
Min. Coeff. Loops	3 1	
Max. Coeff. Loop	is 10	
-Fluid Timescale	Control	Ξ
Timescale Contr	ol Coefficient Loops	-
Conversion Co	itaria	
Convergence Cr	literia (auto	
Residual Type	RMS	•
Residual Target	0.00001	
Conservati	on Target	Đ
Elapsed Wa	I Clock Time Control	Đ
Interrupt Co	ontrol	Œ

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.

-			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

esults E	Backup	Trn Results	Trn Stats	Monitor	Export	
Transient Re	esults					Ξ
Transient	t Results 1					
						×
Transient l	Results 1				E]
Option		Standard			•	
File Compre	ession	Default			•	
_ Outp	out Equatio	n Residuals			±	
Extra	a Output Va	ariables List			Ŧ	
Output F	requency				Ξ	
Option		Timestep	Interval		•	
Timestep	Interval	10				

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.

<u> </u>			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		



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.

Outline	Expert	Parameters			×				
Details of Expert Parameters in Flow Analysis 1 in Flow Analysis 1									
I/O Con	trol (Convergence	Control	Physical Models	Particle Tr 🔹 🕨				
Combust	ion Mode	els							
cou	coupled scalars								
use kolmogorov ts for extinction									
Partitioner Control									
	rt cvs we	ighting							
Interpola	ator Cont	trol							
_i inte	erp single	e phase treatr	nent		€				
Turbulen	ce Model	s							
- 🔽 app	ly ic fluct	tuations for le	s						
Value		t			-				
e 🔳 tef	numerics	option		1					
🗁 🔲 wa	lscale rela	axation factor	•						
General	Grid Inte	rface							
_ forc	e interse	ection							
99	vertex w	eighting value	2						
				· · · · ·					
					`				
ОК		Apply	Close						

Engine	eering Data Resources a.s		
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

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:

<u> </u>			
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		



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		
	2011-01_LES Workshop	Rev.	1
	LES Workshop within the Masters Course in Turbulence Modeling at	Date:	3/1/2012
	Chalmers		

