Abstract 3

List of figures 4

List of tables 5

Acronyms 6

1. Introduction 7

Large Eddy Simulation 8

Grundidee 8

Heat Transfer 8

Methods 9

Resources 9

Used Software 9

Used Hardware 9

Geometry 9

Mesh generation with Ansys ICEM 14 9

Y+ 10

xxx 10

Simulation Setup in Ansys CFX-Pre 15 11

Static Simulation 11

Domain 11

Boundary Conditions 11

Transient Simulation 11

Domain 11

Boundary Conditions 11

Initial Conditions 11

Solution with Ansys CFX-Solver-Manager 15.0 11

Results 12

# Abstract

Turbulence is a phenomenon that occurs more or less in almost every natural flow.

This leads to great ambitions in terms of calculating turbulent flows in order to predict their behavior.

The objective of this work is the investigation of the heat transfer on a NACA 0012 airfoil by means of the Large Eddy Simulation.

The LES Simulation has not yet become standard for industrial application, due to its high demand on resources.

Large Eddy Simulation, a subdomain of Computational Fluid Dynamics, is recently experiencing an increased attention, due to increasing capabilities of the necessary hardware, in detail CPU and memory. In most sectors it is not yet industrial standard, because of its high demand in terms of resources, but it will become an important tool for investigation of complex flow prob- lems in near future.

Therefore the aim of the Bachelor project is the execution of a high-resolution simula- tion of the heat transfer on a wing surface in three dimensions. The given geometry for this task is a NACA 0012 airfoil and the software used will be Ansys ICEM and Ansys CFX. Subsequent the achieved results shall be compared to results obtained from RANS- simulations, which are nowadays standard for industrial application.

Due to the complexity of the Large Eddy Simulation a majority of the work will be studying the theoretical basics as well as performing LES in practice in order to achieve the necessary skills.

# List of figures

# List of tables

# Acronyms

|  |  |
| --- | --- |
| DNS | Direct Nummerical Simulation |
| LES | Large Eddy Simulation |
| VLES | Very Large Eddy Simulation |
| CFD | Computional Fluid Dynamics |
| CAD | Computer Aided Design |
| NS | Navier-Stokes |
| RANS | Raynoldsgemittelte Navier-Stokes Gleichungen |
| URANS | instationäre RANS |
| GS | Grobstruktur |
| FS | Feinstruktur |
| WR-LES | Wall-resoving LES |
| DES | Detached Eddy Simulation |
| SGS | subgrid-scale viscosity |

# 1. Introduction

# Large Eddy Simulation

## Grundidee

Turbulences appear in a great range of shapes and sizes. The basic idea behind the Large Eddy Simulation is to resolve the large eddies numerically, while smaller ones are modeled with dedicated functions.

## Heat Transfer

# Methods

## Resources

### Used Software

### Used Hardware

### Geometry

## Mesh generation with Ansys ICEM 14

The meshed NACA 0012 airfoil was provided as 2D mesh by Dr. Wolfgang Hassler with a total of 219.000 elements. It is meshed with hexahedral elements and features a total of 219.000 elements. The domain shows physical meassurements of 7m by 5m while the wing profile inside the domain shows a chord length of 1m due to the nature of the profile a maximum thickness of 12%, which would therefore be 0,12m in total values. On the left side is located the inlet, on the right the outlet and the upper and lower border are defined as walls, as you can see in figure xxx.

Due to the three dimensional characteristics of the Large Eddies this two dimensional mesh is not sufficient, but has to be extended in a third direction, in order to be capable of providing convincing results. This was achieved by simply extending the given mesh in the third direction by 30 elements. This leads to a total of 6.570.000 elements. The properties of the final mesh, as it was exported from Ansys ICEM can be seen in table xxx.

|  |  |
| --- | --- |
| Domain length | 7m |
| domain height | 5m |
| domain width |  |
| profile chord length | 1m |
| profile maximum thickness | 0,12m |

### Y+

For the Large Eddy Simulation it is crucial to score a Y+ value at around 1. There exist formulas for estimating the first cell height in order to achieve a desired Y+ value.

The definition of the Y+ value is:

where the friction velocity UT is:

The wall shear stress, Tw can be obtained by the following formula:

The value for Cf needs to be taken from empirical estimations. For this calculation the value provided on … has been used, which numbers the Cf with

for internal flows.

Although the Y+ value is dependend from time and location for simple geometries and flows, such as the one used for this simulation, this correlation is highly accurate.

For the calculation of the deltay1 value a short MATLAB script has been applied, which yielded a result of xx. An investigation of the given geometry in Ansys ICEM showed that the height of the cell closest to the wing surfaces features a cell height of xx, which is already beneath the desired value and therefore an alteration of the 2D mesh was unnecessary.

*Picture from ICEM mesh closeup*

### xxx

Due to the three dimensional nature of the Large Eddy Simulation the given mesh was expanded by 30 cells in the z-direction with the Ansys ICEM function xxx, in order to receive a satisfying 3d mesh.

## Simulation Setup in Ansys CFX-Pre 15

### Static Simulation

#### Domain

#### Boundary Conditions

### Transient Simulation

#### Domain

The fluid model for the transient simulation remains the same, apart from the turbulence model. For the transient simulation the LES Smagorinsky model has been appliend, which is capable of dealing with Large Eddy Turbulences.

To model the subgrid-scale viscosity the Smagorinsky model has been applied. This method deals with the assumption that energy production and dissipation of small scales is in equilibrium.

#### Analysis Type

For the transient analysis a number of time steps and a value for the time steps themselves have to be considered. The so-called Courant number is a good measurement for the accuracy. In order to provide reliable and stable results an average Courant number in the range of 0.5-1 is demanded [1]. There are also stable results possible with higher Courant number, but the turbulences may be damped.

According to the Documentation [2] “1,000 – 10,000 timesteps are typically required for getting converged statistics.” Since the simulation is based on the results of a static simulation with a developed flow field and time, as well as resources were limited, a total of 2.000 timesteps was chosen for this simulation.

The value for the timestep was set as 1ms, which leads to a Courant number of … .

#### Boundary Conditions

#### Initial Conditions

#### Solver Control Settings

For the Advection Scheme was chosen Central Difference and for the Transient Scheme the Second Order Backward Euler. This was done due to recommendations at the CFX Documentation [1], where it was stated that the Central Difference Scheme is less dissipative and has provided superior results than the High Resolution Scheme and therefore it is the better choice for turbulent flows.

“The implicit coupled solver used in CFX requires the equations to be converged within each timestep to guarantee conservation. The number of coefficient loops required to achieve this is a function of the timestep size. With CFL numbers of order 0.5-1, convergence within each timestep should be achieved quickly. It is advisable to test the sensitivity of the solution to the number of coefficient loops, to avoid using more coefficient loops (and hence longer run times) than necessary.” – [3]

As an initial … try the number of maximum coefficient loops has been set to 10. However if the size of the timestep requires more than tree to five coefficient loops the result can be considered as inaccurate [3]. After starting with this initial value and reviewing the solver output the value was adjusted to … .

As convergence criteria a root mean square of below 1e-6 of the residual target has been demanded.

<https://www.sharcnet.ca/Software/Fluent14/help/cfx_mod/i1303019.html>

#### Output Control

-Trn Results

-Trn Stats

-Backup

## Solution with Ansys CFX-Solver-Manager 15.0

# Results