Numerical Simulation Methods

Tom Rendall

Lecture 1: Introduction

Unit Information

- The 20 credit unit is split in to two sections
- **Numerical methods (Dr Rendall)** - this is aimed at discretisations, solutions strategies, meshing and hardware specific numerical methods details
- **Aerodynamics (Dr Nicolai)** - this will cover laminar and turbulent boundary layers, general integral boundary layer methods, transition and separation
- This mirrors how aerodynamics was historically split between near field viscous behaviour and mid-field inviscid behaviour

Divide and conquer

- Aerodynamicists like to break things in to manageable chunks that we can understand in separation
- Numerical methods part of unit will describe how you can solve the flow equations
- The boundary layer part will describe how you can understand boundary layer behaviour - which is a fundamental aspect of aerodynamics, whether you are using CFD or experiment
- Boundary layers can then be linked to the compressible/incompressible inviscid models to provide a complete understanding
- Nowadays, it is of course much more common to solver the entire flow with CFD. However, therein lies a problem - **numerical methods are a slow way to learn aerodynamics itself**, and so we will teach you boundary layer behaviour alongside!

Divide and conquer

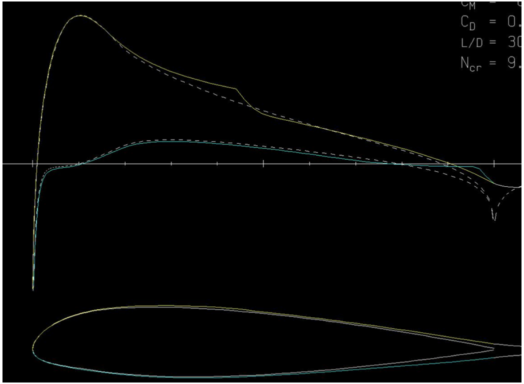


Figure: Example incompressible coupled boundary layer calculation with xfoil. The region within the boundary layer can be solved with integral boundary layer methods, while the region outside can be solved with CFD or other methods.

Divide and conquer

Figure: A transonic (non-linearly compressible) result for RAE2822, which uses a full potential finite difference method **coupled** to an integral boundary layer method. Poetry in computation!

Delivery

- **5 Weekly in-person sessions: 4 of these are lectures, 1 is a question and answer session**
- **Assessed computational exercise (starting week 5)**. This will entail applying the boundary layer methods, coupled to a full-potential model (ESDU VGK), alongside usage of an in-house Euler (inviscid) code, and a commercial CFD tool (Star-CCM+). The exercise will be supported via two computer lab drop-in sessions and will be assessed via one question on the final exam. If you have completed the exercise in full, this will constitute all necessary preparation on the topic; the assessment will cover the same learning content (although, of course, as an exam question)
- **Individual learning activities:** typically 1 per week (*MATLAB app, example question, online video, etc.*)

Introductory Information

Aims of this section of the course:

- What is CFD and where does it fit in aerodynamics ?
- Introduce fundamentals of numerical methods
  Don't need to consider full Navier-Stokes equations to do this.
- How do we apply numerical methods to real flows/geometries ?
  What we need to do to these fundamental methods.
- What effect does computer hardware have on application of these methods ?
- Code simple numerical methods to understand concepts covered.
- Give sufficient background/context to understand the important issues in computational science.

Introductory Information

This part of the course should be sufficiently self-contained that textbooks are not essential. If further information is required, the following books are suggested (all in library)

C. Hirsch *'Numerical Computation of Internal and External Flows'*, Vols 1 and 2.

Ferziger and Peric *'Computational Methods for Fluid Dynamics'*.

J. Anderson *'Computational Fluid Dynamics'*.

Numerical and Simulation Methods content:

- Introduction to CFD and where it fits in aerodynamics
- Fundamentals of numerical methods
- Application to real flows/geometries
- Parallel processing and consideration of computer hardware implications
- + demo lectures.

An Historical Introduction

To Computational Aerodynamics:

Aerodynamics Tools

Finding approximate solutions to 'hard' mathematical problems

What is 'Hard'?

- No analytical solution for general real-world problems
- Problem is very large (many thousands of equations/degrees-of-freedom)

Approximate solution

- Calculated with a finite amount of precision (decimal places)
- Calculated on a finite domain with finite fidelity
- Calculated with known error bounds (accuracy)

Applications of Numerical Methods

- Linear algebra
  - Linear systems solution
  - Matrix inverses
  - Eigen decomposition
  - Singular value decomposition
- Function approximation
  - Interpolation, extrapolation, regression
  - Numerical integration (quadrature)
- Non-linear equations
  - Root finding
  - Optimization
- Differential equation
  - Ordinary differential equations
  - Partial differential equations

The Problem

- All fluid flow described by the Navier-Stokes equations

- 3-D, viscous, unsteady, compressible.

- But insoluble analytically, so numerical methods have been sought for over 100 years.

3-D Navier-Stokes Equations in a Fixed Axis System

Velocity vector: $\underline{\mathbf{u}} = [u, v, w]^T$

Body force vector: $\underline{\mathbf{f}} = [f_x, f_y, f_z]^T$

Del operator $\nabla = \left[\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}\right]$

CONTINUITY EQUATION

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \underline{\mathbf{u}}) = 0$$

MOMENTUM EQUATION(S)

$$\frac{\partial \rho \underline{\mathbf{u}}}{\partial t} + \nabla.(\rho \underline{\mathbf{u}}\underline{\mathbf{u}}) = -\nabla P + \rho \underline{\mathbf{f}} + \frac{4}{3}\nabla(\mu \nabla.\underline{\mathbf{u}}) + \nabla(\underline{\mathbf{u}}.\nabla\mu)$$

$$-\underline{\mathbf{u}}\nabla^2\mu + \nabla\mu\times(\nabla \times \underline{\mathbf{u}}) - (\nabla.\underline{\mathbf{u}})\nabla\mu - \nabla\times(\nabla\times\mu\underline{\mathbf{u}})$$

ENERGY EQUATION

$$\frac{\partial \rho e}{\partial t} + \nabla.(P + \rho e)\underline{\mathbf{u}} = \nabla.(K\nabla T) + \Phi$$

Φ = DISSIPATION FUNCTION $= \sum \tau_{ij}\frac{\partial u_i}{\partial x_j}$

Newtonian fluid with constant viscosity, momentum equation reduces to

$$\frac{\partial \rho \underline{\mathbf{u}}}{\partial t} + \nabla.(\rho \underline{\mathbf{u}\mathbf{u}}) = -\nabla P + \rho \underline{\mathbf{f}} + \frac{1}{3}\mu\nabla(\nabla.\underline{\mathbf{u}}) + \mu\nabla^2\underline{\mathbf{u}}$$

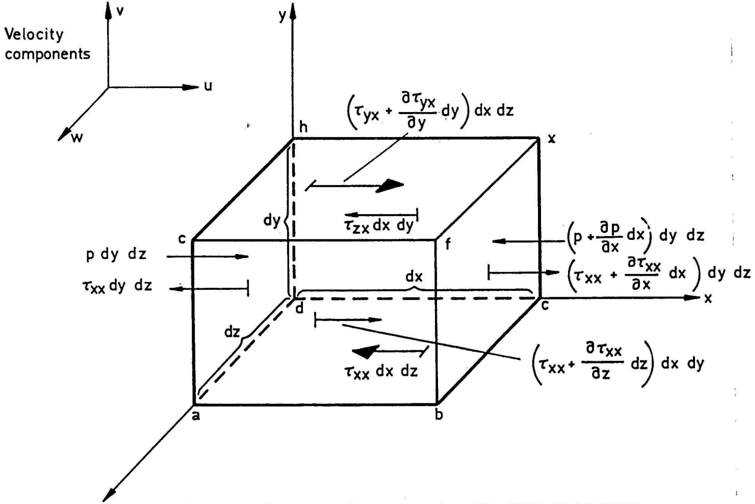
For example
CONTINUITY EQUATION

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0$$

X-MOMENTUM

$$\frac{\partial \rho u}{\partial t} + \frac{\partial \rho u^2}{\partial x} + \frac{\partial \rho uv}{\partial y} + \frac{\partial \rho uw}{\partial z} + \frac{\partial P}{\partial x} - \rho f_x =$$

$$\frac{1}{3}\mu\left\{\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial x \partial y} + \frac{\partial^2 w}{\partial x \partial z}\right\} + \mu\left\{\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right\}$$



INFINITESIMALLY SMALL, MOVING FLUID ELEMENT.
ONLY THE FORCES IN THE x DIRECTION ARE SHOWN

Reduction of the Navier-Stokes Equations

- Ignore viscous effects $\Rightarrow$ EULER equations

CONTINUITY EQUATION

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0$$

MOMENTUM

$$\frac{\partial \rho u}{\partial t} + \frac{\partial \rho u^2}{\partial x} + \frac{\partial \rho uv}{\partial y} + \frac{\partial \rho uw}{\partial z} + \frac{\partial P}{\partial x} = 0$$

$$\frac{\partial \rho v}{\partial t} + \frac{\partial \rho uv}{\partial x} + \frac{\partial \rho v^2}{\partial y} + \frac{\partial \rho vw}{\partial z} + \frac{\partial P}{\partial y} = 0$$

$$\frac{\partial \rho w}{\partial t} + \frac{\partial \rho uw}{\partial x} + \frac{\partial \rho vw}{\partial y} + \frac{\partial \rho w^2}{\partial z} + \frac{\partial P}{\partial z} = 0$$

ENERGY

$$\frac{\partial E}{\partial t} + \frac{\partial u(E+P)}{\partial x} + \frac{\partial v(E+P)}{\partial y} + \frac{\partial w(E+P)}{\partial z} = 0$$

- Assume steady flow $\Rightarrow$ steady EULER equations
- Assume irrotational flow $\Rightarrow$ can introduce the velocity potential Φ

$$u = \frac{\partial \Phi}{\partial x}, \qquad v = \frac{\partial \Phi}{\partial y}, \qquad w = \frac{\partial \Phi}{\partial z}$$

- Assume isentropic flow $\Rightarrow$ can derive the full (this is the equation VGK solves) potential equation

$$\left\{ c^2 - \left(\frac{\partial \Phi}{\partial x} \right)^2 \right\} \frac{\partial^2 \Phi}{\partial x^2} + \left\{ c^2 - \left(\frac{\partial \Phi}{\partial y} \right)^2 \right\} \frac{\partial^2 \Phi}{\partial y^2}$$

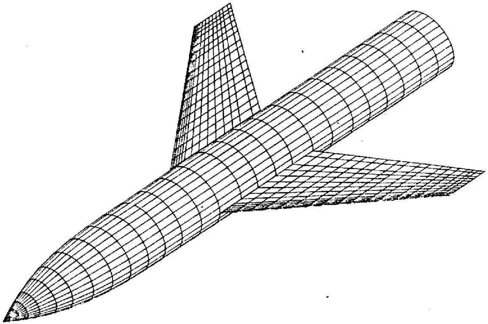
$$+ \left\{ c^2 - \left(\frac{\partial \Phi}{\partial z} \right)^2 \right\} \frac{\partial^2 \Phi}{\partial z^2} - 2 \frac{\partial \Phi}{\partial x} \frac{\partial \Phi}{\partial y} \frac{\partial^2 \Phi}{\partial x \partial y}$$

$$-2 \frac{\partial \Phi}{\partial y} \frac{\partial \Phi}{\partial z} \frac{\partial^2 \Phi}{\partial y \partial z} - 2 \frac{\partial \Phi}{\partial z} \frac{\partial \Phi}{\partial x} \frac{\partial^2 \Phi}{\partial z \partial x} = 0$$

- Assume small perturbations $\Rightarrow$ linear equation (this is what Xfoil or AVL solve)

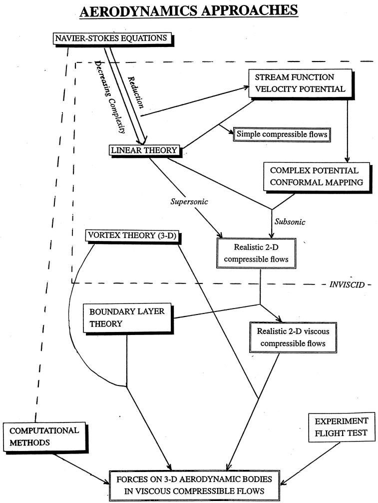
$$\left\{1 - M_\infty^2\right\} \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$

PRANDTL-GLAUERT equation.
Serious restrictions on the use of this.

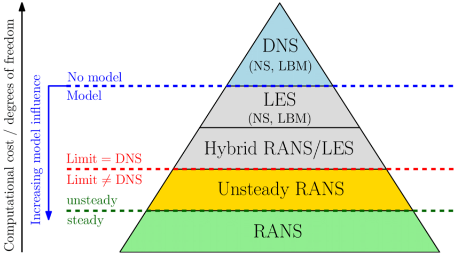


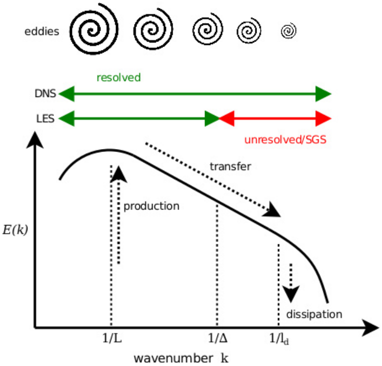
Numerical 'Hierarchy':

Ideal flow: Joukowski. (u, v, P)
Panel methods, vortex lattice.
Potential methods.
Coupled Potential/VLM - boundary layer.
Euler methods.
Coupled Euler - boundary layer.
RANS methods.
DES/LES/DNS methods.



Forms of Full Simulation





- DNS: Direct Numerical Simulation (*Resolves* ALL turbulence scales)
- LES: Large Eddy Simulation (*Resolves* MOST of turbulence scales)
- RANS: Reynolds-Averaged Navier Stokes (Captures only dominant turbulence scale, fully *model* turbulence)

Analytical methods could only be derived for vastly reduced forms of the N-S equations.

In fact sweeping limiting assumptions had to be made about the flow and the set of equations had to be reduced to a single linear equation before exact solutions could be obtained.

This formed the basis of classical aerodynamics.

Numerical methods were required to solve the complete equations

CLASSICAL AERODYNAMICS - Exact solution of Approximate equations.

COMPUTATIONAL AERODYNAMICS - Approximate solution of Exact equations.

Why CFD ?

• Increasing flight envelopes have led to much more complex designs of aircraft
• Principally, this was increases in Mach number, driven by the change to (initially) turbojets and (subsequently) turbofans
• Increases in Mach number introduce transonic flow, where normal shocks appear and there are strong changes in lift, moment and drag coefficients. A failure to understand and design behaviour cause a wide range of aircraft problems, and led to a perceived (but nonsensical) 'sound barrier'. Resolving these changes required
⇒ high demands on wind-tunnel testing and the development of new, transonic tunnels (actually, this was what initially helped solve things, not CFD!)
⇒ lengthy and expensive development times
• But rapidly increasing CPU power, and maturity of CFD methods means computational solutions became much faster and a viable alternative to tunnel tests in many (but not all) situations

Why not CFD ?

• Over last 20 years CFD has become standard in industry, and used extensively in main design stages. It's no panacea, though, as there are many areas where it's capabilities are still somewhat poor (sssh, don't tell everyone)
• Generally, attached flows at up to supersonic speeds are well predicted. Any separated flows are still an area of difficulty, with higher uncertainties. A key point is that CFD will very often deliver a result, but the accuracy is up to you to assess. Just because the pizza arrives on time does not mean it is any good.
• Don't underestimate what good engineering and common sense achieve - these remain the cornerstones of aircraft design. There's one thing that CFD doesn't directly deliver, which is an understanding of aerodynamics, so keep that in mind!
• Most of the aircraft ever built were designed entirely without CFD.
• Despite those caveats, I love CFD, and it's really useful!

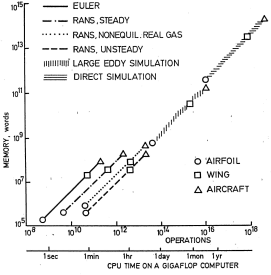
A Measure of Computer Power

- Equations solved determined initially by numerical methods available
- Now limited by available computer power
- First solutions performed by rooms of people

- 7 billion people on the planet
- Give each one a calculator
- 7 seconds to multiply two 16 digit numbers

$\Rightarrow$ one billion operations per second

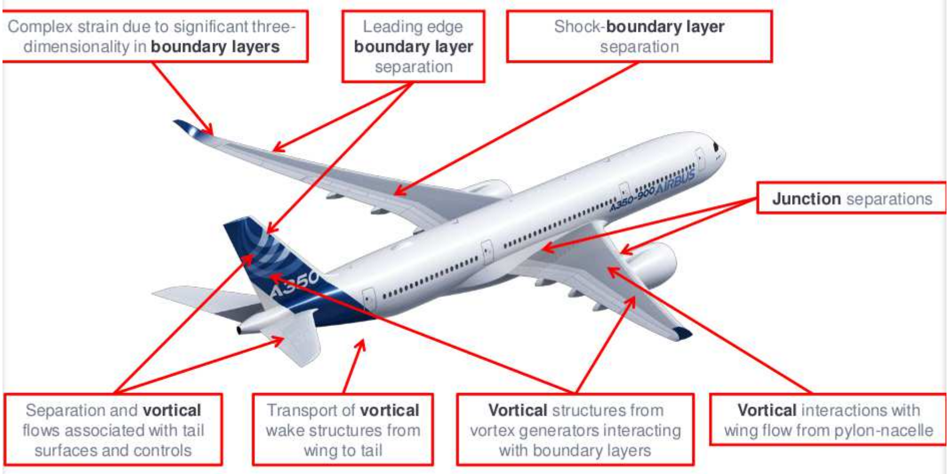
- 'flop' defined as one floating point operation.
- Intel i7 quad/hexa core 4.0GHz: 100-200Gflop/s.



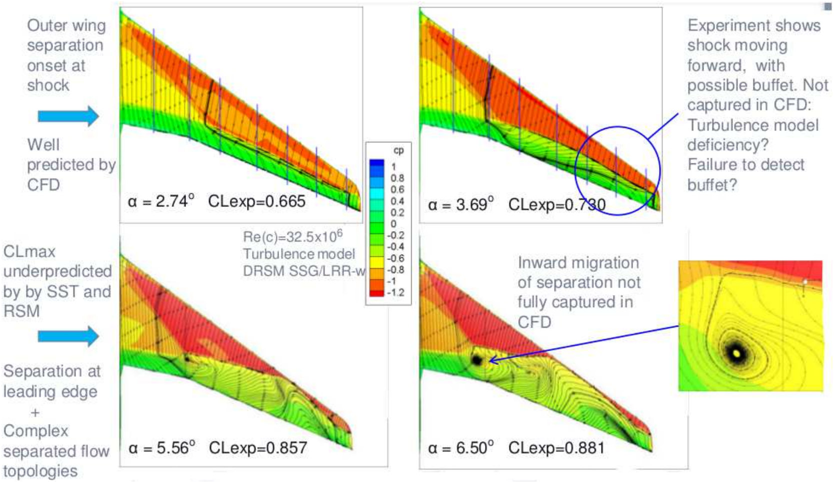
ESTIMATED MEMORY AND EXECUTION TIME REQUIREMENTS FOR SEVERAL DIFFERENT CFD FORMULATIONS

(Coarse meshes)

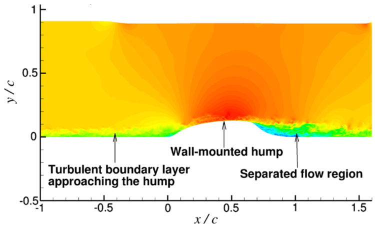
Industrial challenges for CFD application.

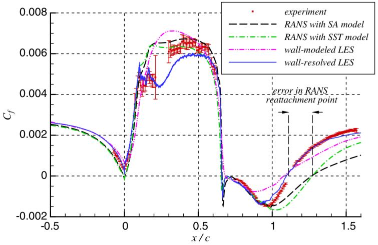


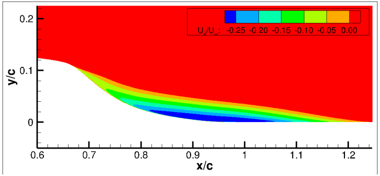
Example: physics modelling.

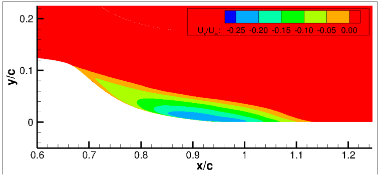


Example: physics modelling.



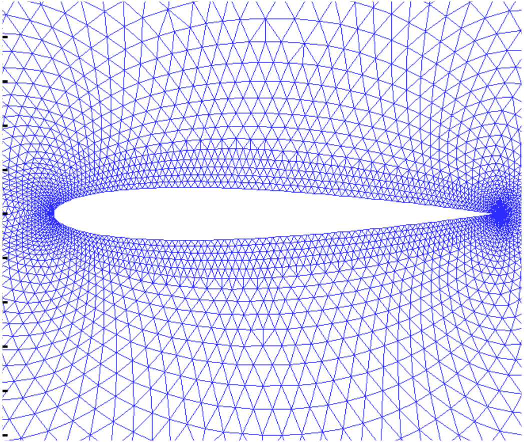




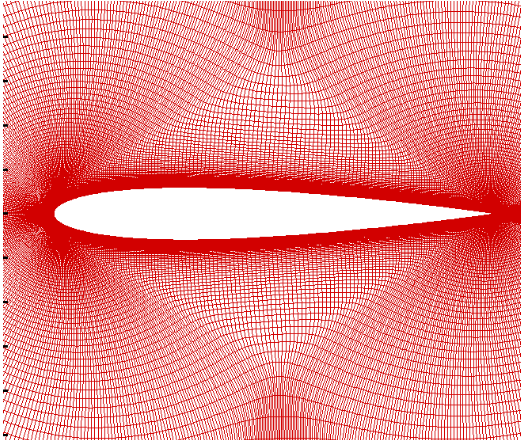


Wall-Resolved LES ($N_p = 4 \times 10^9$) RANS ($N_p = 5 \times 10^5$)

Example: CFD Meshes

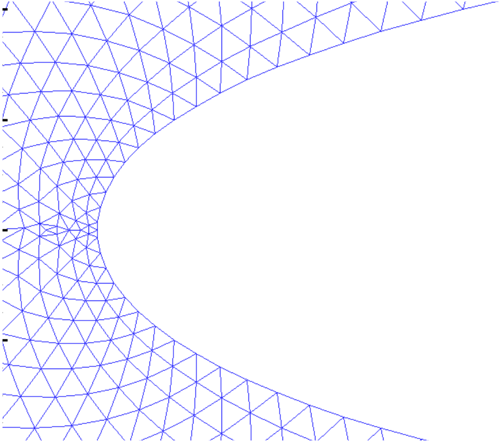


Euler

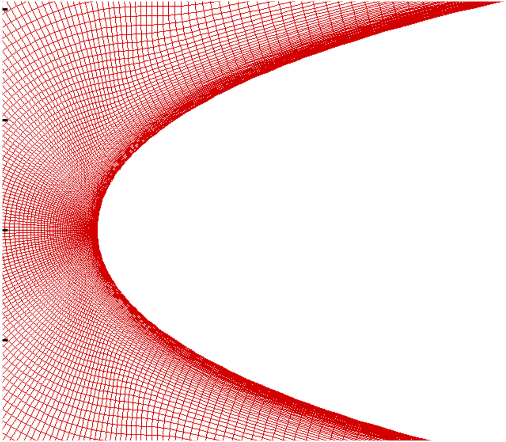


RANS

Example: CFD Meshes

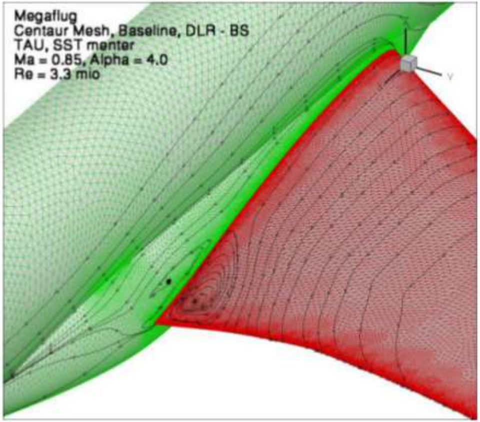


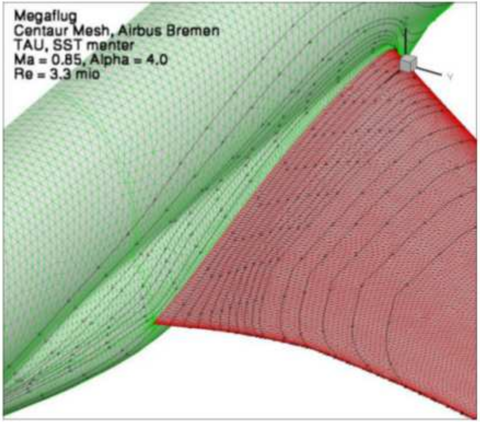
Euler



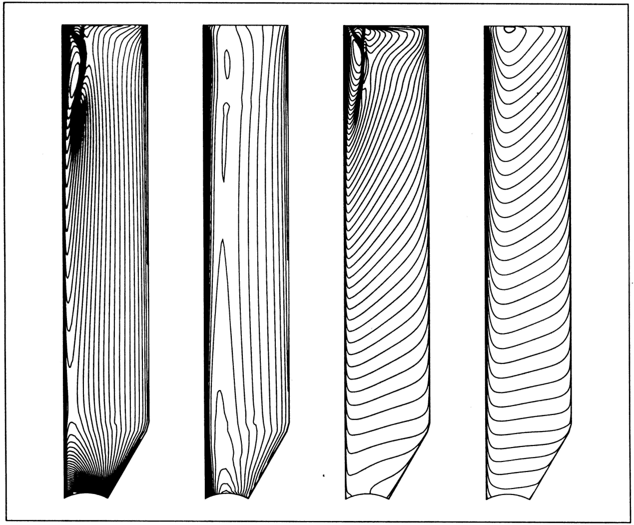
RANS

Example: mesh dependence.

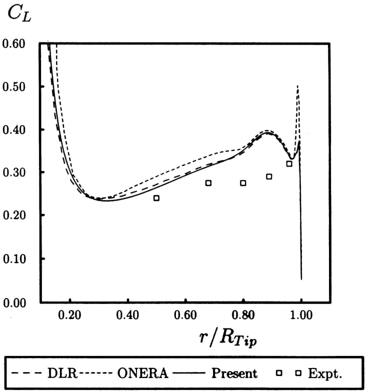




O-H Grid 7×10^5 points: C_P and Mach contours



Blade loading



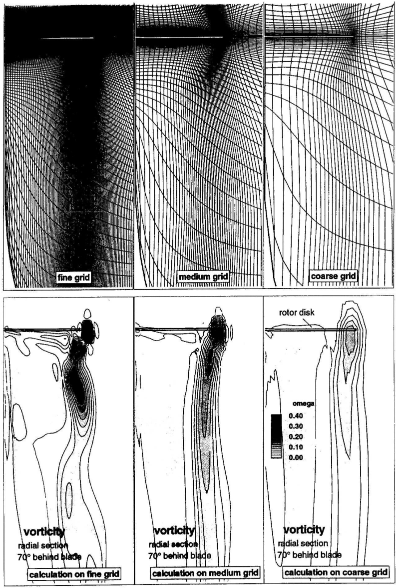


Figure 10: Grid refinement study at a radial position 70° behind the rotor blade, Caradonna-Tung model rotor in hover: $\theta_{0.7} = 8°$, $M_{\omega R} = 0.794$

Aims of Course

This half of the course is primarily a numerical/computational science. You do not *need* any aerodynamics or fluids background, but it will of course help, so we suggest you remind yourselves of the content from years 1 and 2 (year 2 being especially relevant in terms of shockwaves).

At the end of this section of the course you will understand:

FUNDAMENTAL/THEORETICAL ASPECTS ("Numerical" part):

- Fundamentals of deriving and analysing numerical methods.

APPLIED ASPECTS ("Simulation" part):

- How these methods are applied to real flows/geometries.
- Problems involved in mesh generation.
- Effects of computer developments on simulation methods.

Assessment:

100% exam

Course Contents

- Derivation of Navier-Stokes equations
- Forms of the equations - conservative, non-conservative
- Simple 1-D model equation
- Discretisation - approximation of time and space derivatives
- Accuracy, consistency, stability - Fourier analysis
- Different numerical finite-difference schemes
- Explicit and implicit schemes stability limits
- Extension to 2-D and 3-D equations
- Finite-volume schemes
- Consideration of unsteady flows
- Mesh generation - model complex geometries
- Computer architectures - parallel processing
- Recent state-of-the-art results