

# 4A2 Interim Report

Peter Robinson  
Trinity College  
pr428@cam.ac.uk

November 14, 2019

## 1 Introduction

A simple flow solver has been created, which aims to find the solution to a flow through various sections using a Lax-Friedrichs method [1]. This method makes use of a central difference formula in space, which is inherently unstable, and so is stabilised by the addition of artificial smoothing. A uniform grid is used throughout for simplicity of implementation, and the produced results are analysed below.

## 2 Test Case Results

The results of test cases 0 and 1 are shown below, with contours of the Mach number of the local flow and the CFD convergence given. Both mach number distributions shown an expected result, with some flaws. Considering test case 0 (Figure 1), the flow accelerates as expected through the narrow section, and there is a high pressure region at the front of the bump, with a low pressure region behind. The main issue with the results of the simulation is the fact that the stagnation pressure is not constant throughout the flow, which is expected when the flow is isentropic. This is likely due to the fact that a large amount of artificial smoothing has been applied, which reduces the stagnation pressure slowly through the section. In some meshes, another source of accuracy is a non-smooth change in the grid size. In this case however, element sizes are similar throughout and change slowly, so there will be minimal loss of accuracy from this. The Lax-Friedrichs scheme used in the simulation is first order accurate in time and space, and given the minimum element size for test case 0:  $d_{min} \approx 0.13$ , this means a possible loss of accuracy of 0.0169. Due to the artificial viscosity, for cases where there are discontinuities in the flow (for example a shock) there scheme will show a considerable loss in accuracy as the edges of a solution will be overly smoothed. Further improvements of the solver should aim to reduce the necessary smoothing to keep the solution stable.

Test case 1 (1600 iterations, Figure 4) shows a slightly faster convergence than test case 0 (2800 iterations Figure 2), but again there is a small loss in stagnation pressure, caused by the addition of artificial smoothing to stabilise the (usually unstable) central difference scheme.

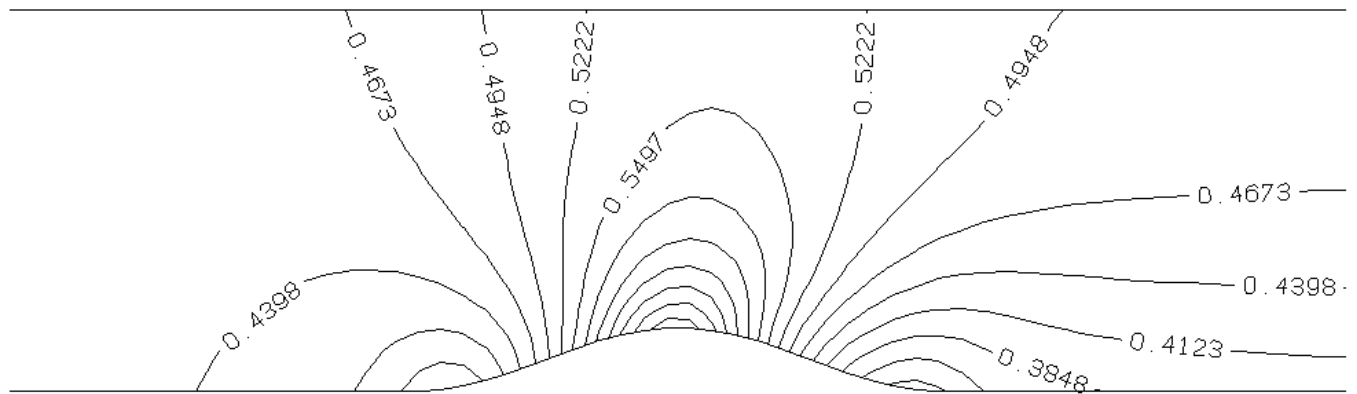
## 3 CFD Mesh Types [1, 2]

### 3.1 Structured Mesh

A structured mesh, as the name suggests, is a mesh which follows a strict spacial structure, and for a 2D problem can be represented in a 2D array. Elements within this mesh are quadrilaterals (2D) or polyhedra with 6 faces (3D). This comes with the benefit of being easy to create and store in computer memory, and makes the implementation of the conservation equations for each element far easier, as nested loops can be used with very little complication except at boundaries.

### 3.2 Unstructured Mesh

An unstructured mesh is one which does not follow a regular pattern or structure, and therefore is far more difficult to store and utilise within computer memory and programs. Each element geometry must be stored, as well as its connections to neighboring elements, making the memory requirements significantly larger. Triangular (2D) or tetrahedral (3D) element shapes are often used, as these give the smallest partition of a shape. An unstructured mesh has the advantage of being able to be far finer (and so more accurate) in specific areas where fluid properties change more rapidly, and so can give a more accurate solution to many problems.



Nozzle

time step number 2740

MACH NUMBER

MERIDIONAL SURFACE NUMBER 1  
LOCAL/GLOBAL MAX. = 0.731/ 0.731  
LOCAL/GLOBAL MIN. = 0.319/ 0.319  
INCREMENT = 0.027

Figure 1: Mach number plot of Test Case 0, showing (incorrect) lack of symmetry in flow

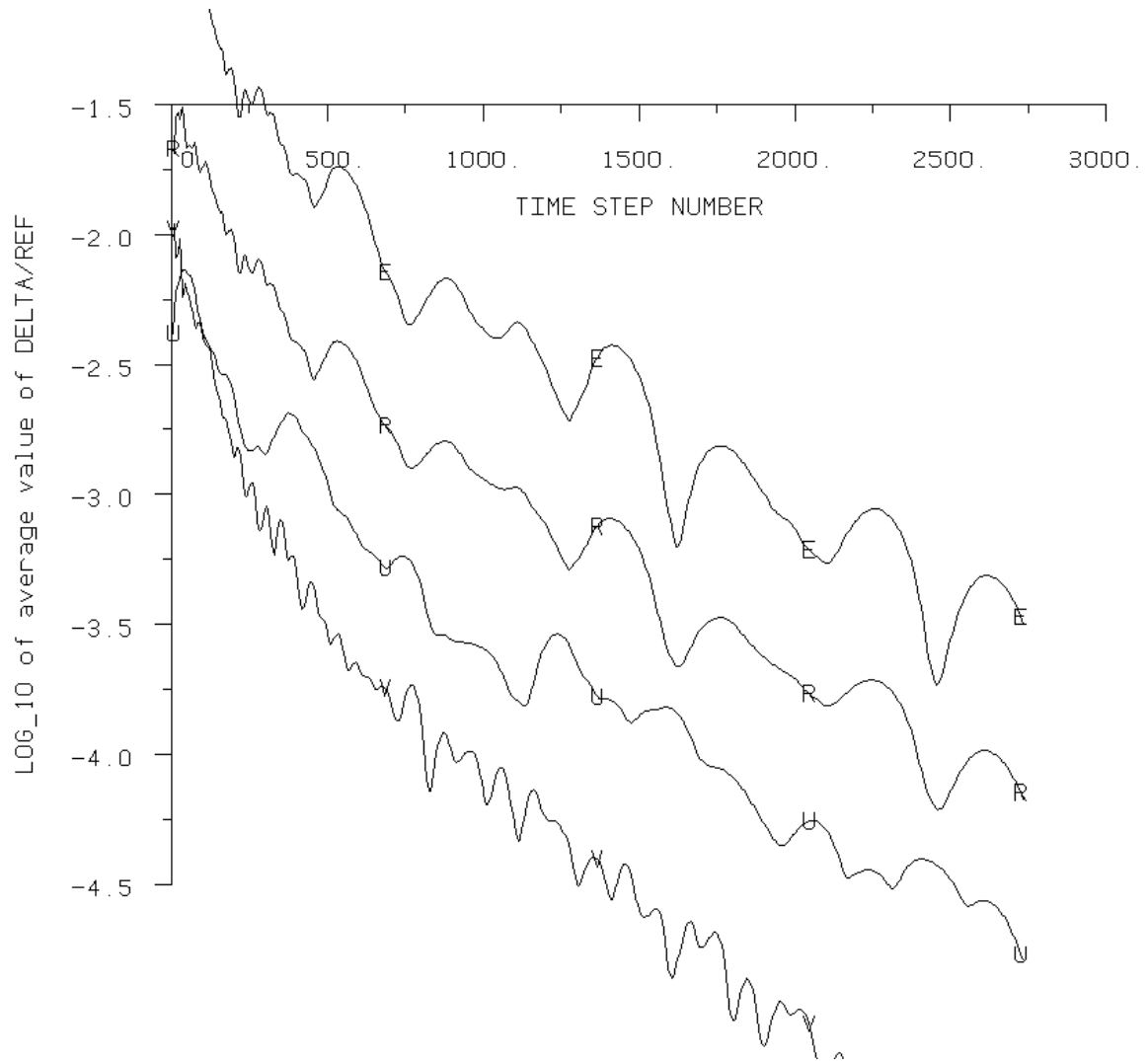
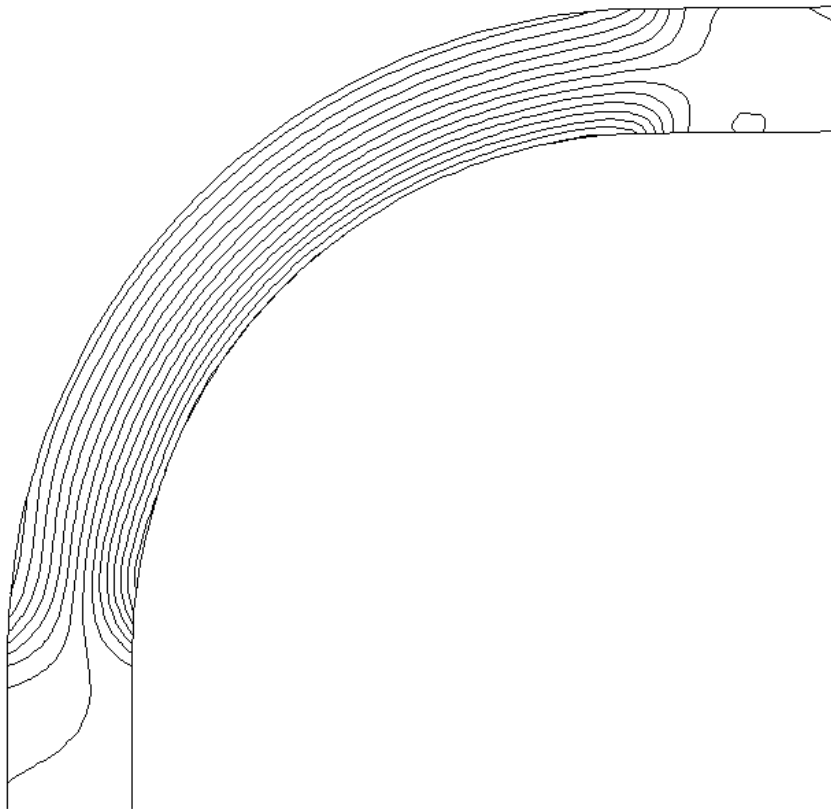


Figure 2: Convergence of Test Case 0 (using crude guess)



Circular Bend with extensions and bump  
time step number 1660

MACH NUMBER

MERIDIONAL SURFACE NUMBER 1  
LOCAL/GLOBAL MAX. = 1.173/ 1.173  
LOCAL/GLOBAL MIN. = 0.878/ 0.878  
INCREMENT = 0.020

Figure 3: Mach number plot of Test Case 1

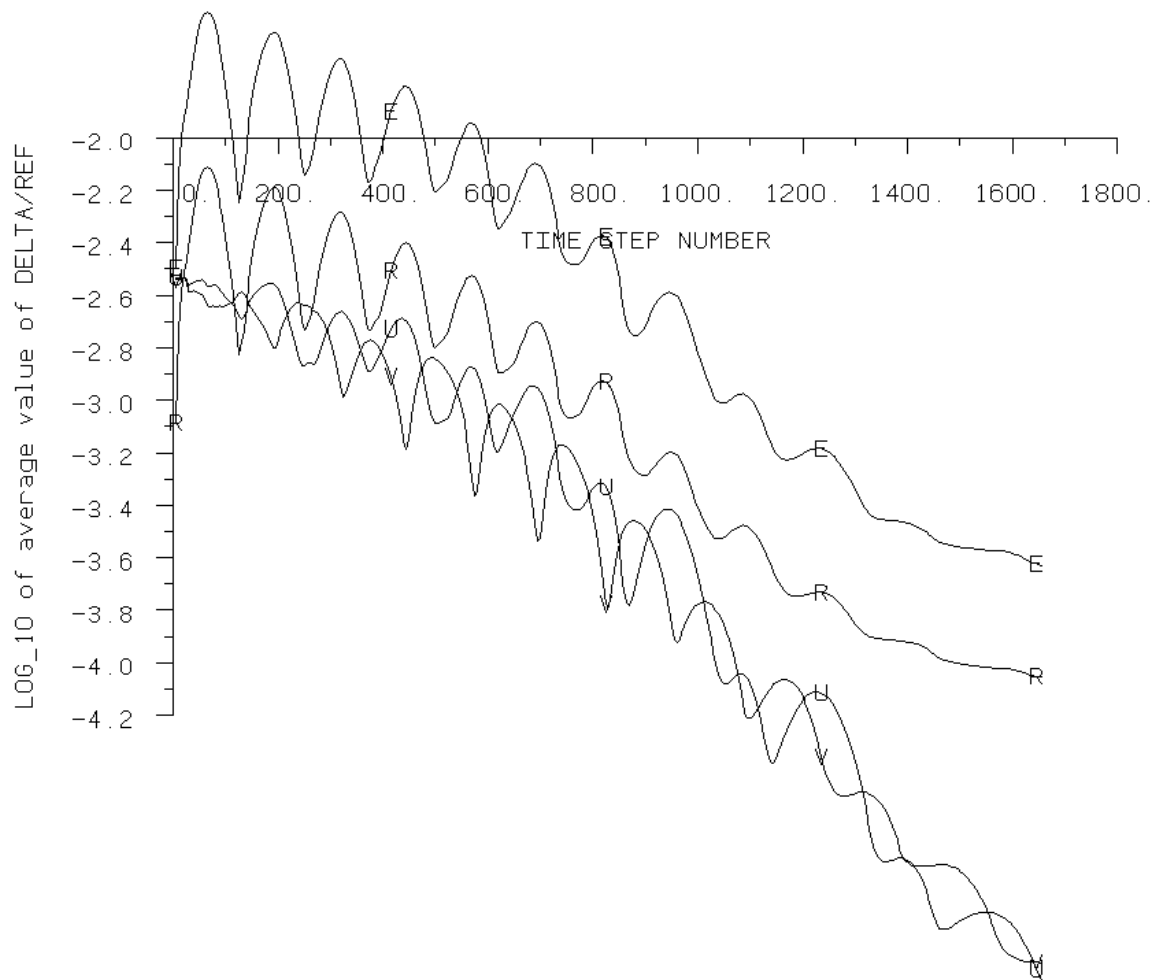


Figure 4: Convergence of Test Case 1 (using crude guess)

### 3.3 Hybrid Mesh

A hybrid mesh takes the benefits from both unstructured and structured meshes, being structured in some areas, and unstructured (and often finer) in other areas. This results in a lower memory cost to a fully unstructured mesh, but does not compromise on solution accuracy in regions of interest where fluid properties are changing quickly.

## 4 CFD Post Processing Techniques

Having calculated the various flow quantities at every grid point, the final step is to present the data in an intuitive and understandable way. This is done in a variety of ways, each with certain benefits for showing certain aspects of the flow.

A commonly used technique is to plot contours of a flow quantity, such as pressure or mach number (as seen above). This is a very good way to display the overall distribution of a quantity within a flow, highlighting maxima and minima. However, this does have the disadvantage of not showing the quantity at any specific point, and so should not be used when this data is necessary to know. Colour contours can be used to give a very intuitive visual representation of this contour plot. Contour plots work well in 2D, but are often difficult to interpret in 3D, so often only a slice of a 3D result is shown when displaying it as a contour plot.

Vector plots are a very good way to display vector quantities within the flow, the most common of these being the flow velocity. They allow for the display of flow direction as well as speed, and can make sense in 3D. However, often if there is a large density of grid points, only a subset of grid point vectors are shown, meaning sudden discontinuities can sometimes be missed if the subset is too small. An extension of a vector plot is to show the streamlines of a flow, which is often used with a 3D flow. These can also have colour to represent the flow speed, and can give a great deal of information about the flow in a single image.

## References

- [1] Jie Li 2019, CUED 4A2 Course notes
- [2] André Bakker 2002, <http://www.bakker.org/dartmouth06/engs150/07-mesh.pdf>