Lund University
Department of Energy Sciences
Division of Fluid Mechanics

Turbulence - Theory and Modelling

Computer exercise 4

Compressible turbulent flow

In this computer exercise you will explore the performance of several RANS based turbulence models for high speed compressible flow. The set-up is the same as was used in the previous exercises, i.e. flow around a cylinder. You should look at some (at least 3) of the following models: the standard k-ε model, realizable k-ε, Spalart-Almaras 1-equation model, SST k-ω and RSM. You should do this for three inlet Mach numbers: 0.4, 0.7 and 2.0. Please hand in a short (max 4 pages) report on your findings latest 20 December by email to johan.revstedt@energy.lth.se. The report should include comparisons of the models in accordance with the task below.

Preparations

- Look at the lecture notes on compressible RANS models and the short introduction to compressible flows and shock waves that you will find here

Instructions

- Download the case-file from the course web-page
- The case is initially set to the standard k-e model. Make sure Viscous heating is turned on. To change model goto **Models→Viscous**.
- Simulate using the models and Mach numbers above (You do not need to run all models for each Mach number, but at least all Mach numbers for one model and three models for one Mach number). Don't forget to save your cases and data. Hint: It is recommended that you re-initialize using **Hybrid initialization** when you change model except for the RSM where it is better to start from a previous solution (e.g. k-ε). Also, it is recommended that you start with first order accuracy for all variables and the try to go to second order (**Solution methods**).You might need to adjust the near wall resolution. Check the y+ values. If you need to adjust go to **Adapt→Yplus/Ystar**
- You will be using the density based solver for these simulations, and I recommend that you enable **Solution Steering** and change **Flow Type** depending on Mach number, i.e. Subsonic for Ma=0.4, Transonic for Ma=0.7 and Supersonic for Ma=2. This will improve convergence.
- Compare your results from the models. A first step would be to look at contour plots of some variables such as speed, Mach number, turbulent kinetic energy etc. Also, it is interesting to look at the production of k and how it varies with Mach number. For a more detailed analysis you should look at parameters such as length of the recirculation, separation angle, drag and distribution of k along lines. Apart from model differences, how do these vary with Mach number? Can you think of explanations?

- **Practicalities:**
  - To create a line goto the tab **Postprocessing,** on the upper left chose **Create→Line/Rake** and type in the coordinates of the end points and name the line. NB, the cylinder is centred at (0,0) and the radius is 0.5. I recommend that you at least create the following lines:
    - 1 line from x=-5 to x=10 at y=0
    - 3 lines at x=1, 3, 5 from y=-3 to y=3
  - To plot data along a line goto **Plots→XY Plots**. If you want to plot along one of you lines make sure that you mark **'Position on X Axis'** under **Options**, mark the line you

want to use and make sure that **‘Plot Direction’** is set along your line. If you want to save your line data mark the option **Write to File**. This is good when you want to compare cases, then just press **Load File** and choose the cases you want to look at.

- To find the separation point a good way is to plot the wall shear stress or the skin friction coefficient. To plot as a function of angle you will have to use a **Custom Field Function**. To plot you should unmark **‘Position on X Axis’** and use your function ‘angle’ on the x-axis. You will find it under the group Custom Field Functions, i.e. under **X Axis Function** replace **‘Pressure’** with **‘Custom Field Function’.** Don’t forget to mark ‘cylinderwall’ under **Surfaces**.
- To calculate the drag goto the tab **Postprocessing** and choose **Forces**, check the direction vector and press **Print.**