MATLAB Activity: 2D Unstructured Inviscid Solver

L. Kedward, November 2020

1   Getting started

There are two MATLAB apps provided in the folder: '`mesh_generator`' and '`run_solver`'. These apps provide graphical interfaces to Tom Rendall's unstructured cutcell mesh generator and unstructured inviscid flow solver.

To run either app, make sure your MATLAB instance is navigated to the folder containing the apps, then type the name of the app in the command window followed by the <ENTER> key.
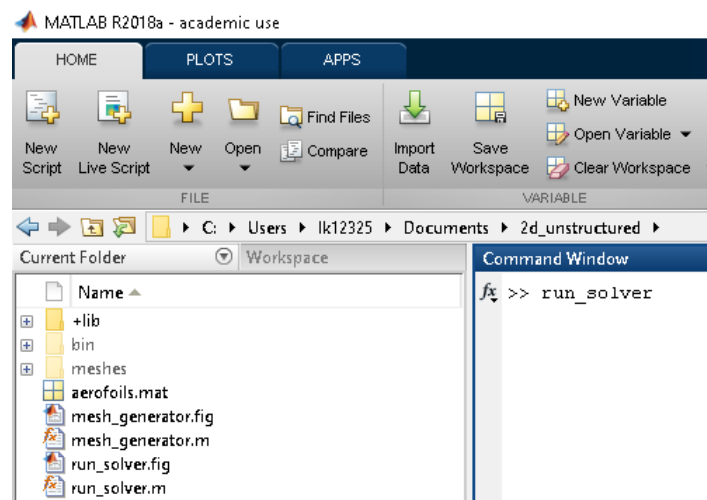
Note: The mesh generator and flow solver are provided as binary executables and **only support 64bit Windows or Linux**.

1.1   About the flow solver

The compressible Euler equations are solved by a cell-centred finite volume method using the face-based methodology of Eliasson for arbitrary shape elements. Compact central JST terms with artificial dissipation are used for the discretisation with steady-state integration performed explicitly by four-stage Runge-Kutta. Convergence acceleration is performed using local time-stepping and non-linear agglomeration multigrid.

References
Eliasson, P., "EDGE, a Navier-Stokes Solver for Unstructured Grids", Tech. repo., FOI, 2002.

Jameson, A., Schmidt, W., Turkel, E., "Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes", 14th Fluid and Plasma Dynamics Conferences, 1981.

Mavriplis, D. J., "Multigrid Techniques for unstructured Meshes", 26th Computational Fluid Dynamics Lecture Series Program of the von Karmon Institute (VKI) for Fluid Dynamics, 1995.

Moulitsas, I. and Karypis, G., "Multilevel Algorithms for Generating Coarse Grids for Multigrid Methods", Proceedings of the 2001 ACM/IEEE Conferences on Supercomputing, ACM, New York, USA, 2001.

1.2 Running the flow solver

After launching the '`run_solver`' app, you are presented with an interface from which you can choose a mesh file (see the '`meshes`' folder) and configure the settings for the inviscid flow solver. After running the solver, click 'Load results' followed by 'Plot' to view the results.
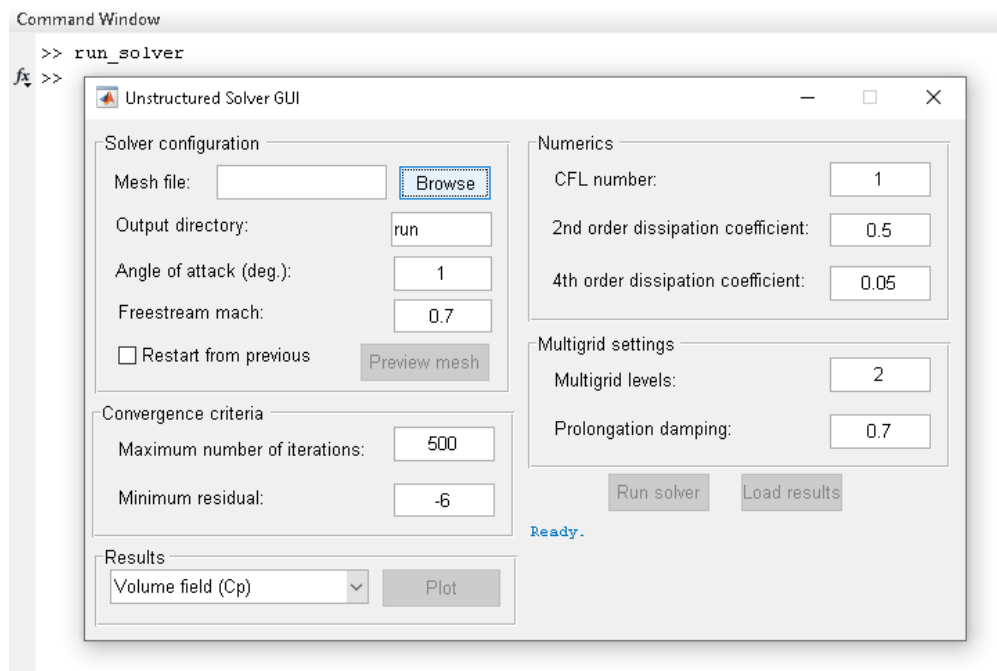

Tips:

- Convergence may stall if you run a purely symmetric case (symmetric aerofoil at 0 degrees)
    - Increasing 4th order dissipation to 0.1 may help, otherwise simply add a small angle of attack
- If convergence is stalling:
    - First try reducing multigrid prolongation down to 0.6
    - Then try reducing the CFL number to 1
    - Then try reducing the number of multigrid levels
    - Finally, try small adjustments to the 4th order dissipation coefficient: for coarse grids, reduce to 0.025; for fine grids increase to 0.075 or 0.1
- You can fine-tune solver settings by selecting 'Restart from previous' and running only ~100 iterations at a time; you can then adjust settings if the solver stalls midway

1.3 Using the Cutcell mesh generator

The 'mesh_generator' app allows you to arrange one or more aerofoils and generate a cutcell mesh around them. The workflow is as follows:

1. Select a configuration from the configuration list or Click 'New' to create a new one (you must click on the configuration to activate it)
2. Click 'Add' to add a new aerofoil to the configuration
3. Click on a body in the body list to select it for transformation
   o Use the sliders to scale, rotate and translate the body
4. Select a refinement level and click 'Generate Mesh' to run the mesh generator
   o The generated mesh will be saved in the 'meshes' folder
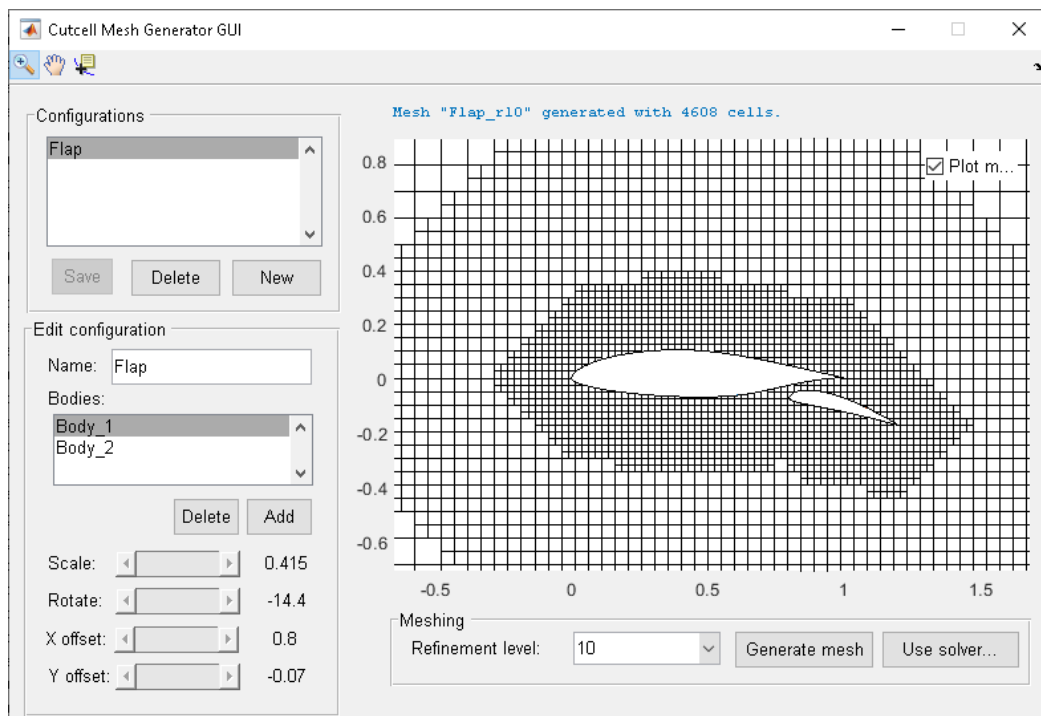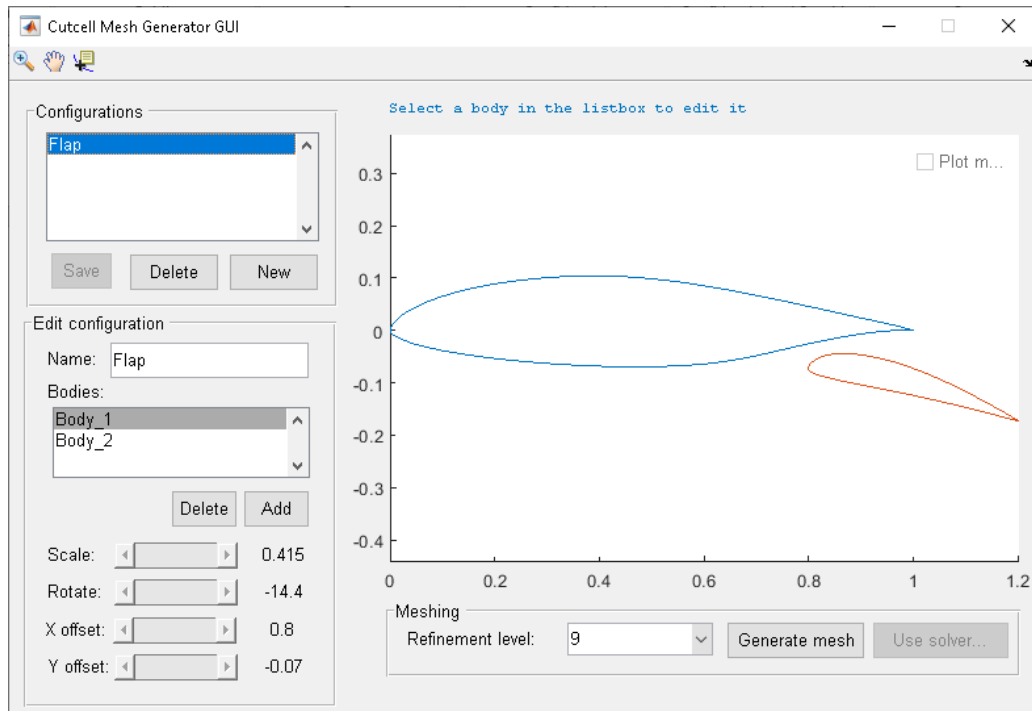
2   Tasks

Included in the 'meshes' folder are two meshes for the NACA0012 aerofoil:

1. naca0012_structured: generated with Prof. Allen's structured mesh generator and converted to unstructured format
2. naca0012_cutcell: generated with Dr Rendall's cutcell mesher

2.1   Compare different mesh topologies
Run both NACA0012 meshes using the settings shown in Table 1 and compare the resulting surface pressure distributions.

Table 1: Test case settings

Mach no.	0.75
Angle of attack	1 degree
Multigrid levels	3

In what way do the pressure distributions differ and why is this?

Hint: plot and compare the surface coordinates from each mesh.

2.2   Effect of the dissipation coefficients
Run the naca0012_structured mesh again with the settings shown in Table 1 and plot the surface pressure distribution.

Notice the slight oscillations around the upper surface shock.

Which dissipation coefficient should you increase to reduce these oscillations? Test this by increasing the coefficient by a factor of four and comparing the new pressure distribution.

2.3   Maximum CFL number
Jameson's time-stepping scheme increases the maximum theoretical CFL number to $2\sqrt{2}$.

Using the naca0012_structured mesh what is the maximum CFL number for which you can achieve a residual of -5 using the settings in Table 1. How else can you speedup convergence?

2.4   Design tasks (optional)

- **Find the critical mach number for an aerofoil**: perform a sweep in Mach number and check the surface Mach plot to see when sonic flow first occurs;
- **Quantify compressibility effects in 2D**: perform a sweep in angle-of-attack to obtain the lift-curve slope with compressibility effects included. Use the drag coefficient as a measure of compressible wave drag;
- **Investigate multi-element configurations**. *e.g.* measure the lift coefficient for an aerofoil with flap at low Mach and low angle of attack or visualise transonic shock locations for a 2D wing-nacelle configuration