App Documentation for ANSYS Fluent Post-Process

Mehmet Arslan

February 20, 2025

1 Introduction

This document provides an overview of the MATLAB application for post-process of CFD results optained from ANSYS Fluent, including its purpose, features, and how to use it. App conssits of three mai tabs. First offers the curve plotting option which is usaully used for plotting profile or distribution of quantites such as temperature, velocity, pressure etc. The second provides the contour option for scalar and vector fields. The last part is the TUI command generation which offers a simple way to create the out put files for a two types of domains.

2 Installation

2.1 Prerequisites

- Ensure that MATLAB is installed on your system.
- The *MainWindow.mlapp* file can be opened and run in MATLAB App Designer (available in MATLAB R2016a and later).

2.2 Installation Steps

- Ensure you have the required dependencies installed.
- Clone the repository: `git clone https://github.com/arslan-mehmet/Post-Process-4-CFD`
- Open MATLAB on your computer.
- In the MATLAB Home tab, click on App Designer.
- Click Open, then browse to select the downloaded *MainWindow.mlapp* file.
- Click Run to launch the app.

3 Functions

3.1 TUI Command

This section provides an easy and optimized data exportation that is most compatible for the design of the code. The usage of these section and the TUI command it presents is crucial especially for *Contour Plotting* section. The code enables user to generate two different domain types; namely, NACA Arifoil and Rectangle. These options serve different purposes and advanteages depending on the case.

3.1.1 Domain Parameters for NACA 4 Digit Airfoil

This option is most useful for the two-dimensional airfoil testings. It generates a domian that wraps around the airfoil. For the time being, only symmetric 4-Digit NACA airfoils can be generated. Figure 1 shows the parameters used to define the region.

- *Airfoil Thickness Ratio* is the the maximum thickness divided by the chord length of the airfoil. This number should be between $0 - 1$ and has no unit.
- *Airfoil Chord Length* is the distance between leading and trailling edge. Only chord length has a dimension and it is in *meters*.
- *Height-To-Chord Ratio* is the ratio of the length of the each line to the chord length of the airfoil. In Figure 2, the length is depicted for clear understanding.
- *Wake-To-Chord Ratio* defines the distance between trailing edge and left-most point of the domain (*Wake Length* in Figure 2) as factors of the chord. However, it is noted that the algorithm does not allocate a line at this location instead it sets a limit for the lines to go "until".
- The *Growth Rate* is defined as

$$k = \frac{d_{i+1}}{d_i} \tag{1}$$

  Therefore, each line is separated by a factor of the growth rate of distance between the previous point yielding an increasing distance to maximize the number of lines near the leading edge to capture sharp gradients.
- *Point Number* defines the number of surface on airfoil. This also indirectly defines the total number of lines.
- *Number of Points on Line* determines how many points should be located on each line. This inherityl defines the row number of the domain. Therefore, avoid using great numbers if the height is small or the grid used for the numerical analysis already has few elements in the region.
- The code assumes the leading edge is at $x = 0$ and $y = 0$. Therefore, in cases where the geometry in the user's analysis palces airfoil at a different location, the points should be adjusted according to that exact location. Thus, if necessary user can specify this point in *Leading Edge x-* and *y-coordinates* options.

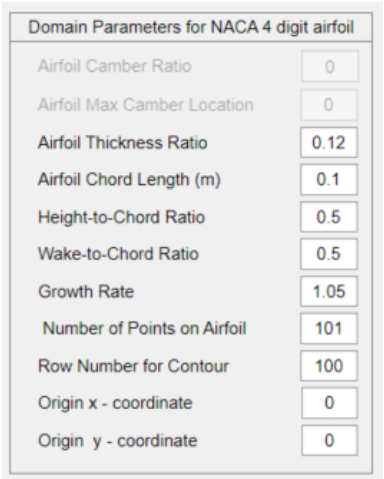

Figure 1: Domain parameters panel for cases that includes airfoil.

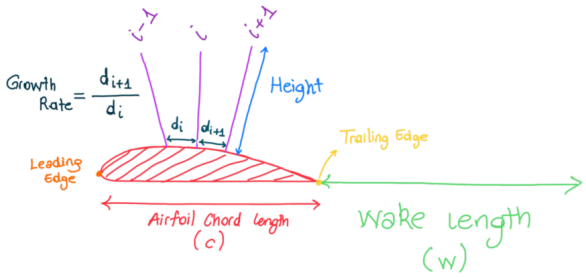

Figure 2: Sketch for the Airfoil based domain.

3.1.2 Domain Parameters for Rectangular Area

This option generates a rectangular domain which is usefull for many cases. The parameters are straight forward as shown in Figure 3.

- *Draw a rectangular region* notifies the system that the rectangular domain option is activated. This enables the options whicha re intitially disabled.
- *Horizontal max*, *Horizontal min*, *Vertical max*, and *Vertical min* determines the the vertices of the rectangular domian as show in Figure 4
- *Num of Columns* defines the number of lines. the lines are equally spaced
- *Num of Rows* defines the number of points on each line.

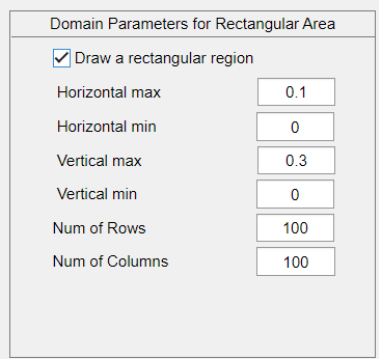

Figure 3: Domain parameters panel for rectangular domains.

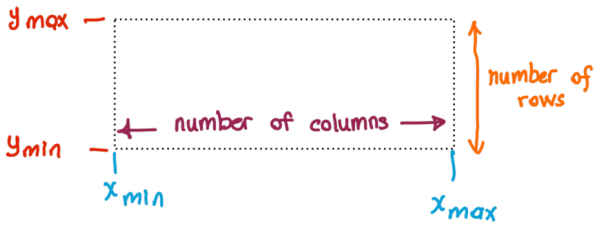

Figure 4: The rectangular domain sketch

3.1.3 Data Options

The flow quantites that may be exported are listed here as shown in Figure 5. The system exports only the quantites whose box is checked.

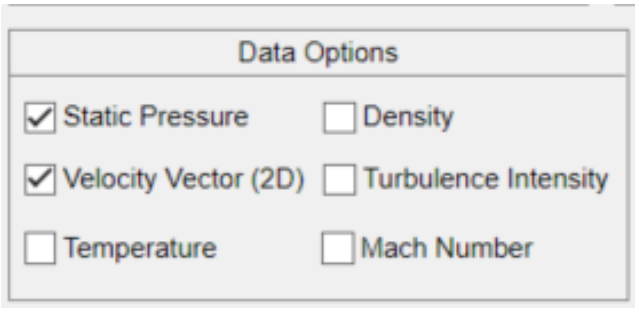

Figure 5: Quantity options that can be exported.

3.1.4 Saving Options

The Saving options are as given in Figure 6These options set the general properties of the Fluent's exportation process and are must be filled.

- *Save Folder* is the folder that Fluent will export the data files in.
- *Case Name* is the name that will be given to each file. The set naming rule is "*CaseName*-typeC-*Abbrevation of the Quantity*-*Position*.xy". An example for a file name is "TestCase-typeC-p-x.xy"; this case includes pressure data along with x coordinates at each point. The abbrevations that are in use are as follows:

  - **p**: Pressure
  - **V**: Velocity
  - **M**: Mach number
  - **rho**: Density
  - **TI**: Turbulence Intensity
  - **T**: Temperature

- *Node Values* this ensures that the Fluent interpolates the data for the point and creates smoother contours.

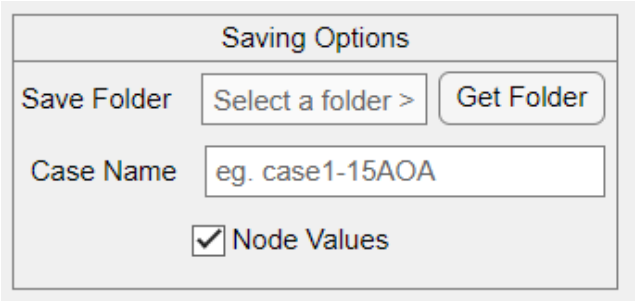

Figure 6: Saving options for ANSYS.

3.1.5 Buttons

There are tow buttons *Draw Domain* button plots the domain according to the settings into the "Figure" panel. This button allows users to change the settings and observe the change in the domain for further adjustments. It also provides a visualization for the domain. The figure can be saved through *Export* menu on the menu bar on top. *Write Commands* generates the TUI command and writes them into the *TUI Commands* text area. The text area prohibits an easy access to commands. Changes made here are alowed but not adivised. User is strongly urged to make the changes in the respective panel and regenerate the comands.

3.1.6 How To Use

Step 1: Decide which domain option is suitable for the case.

Step 2: Adjust parameters and check via "Draw Domain" button.

Step 3: Select the quantites which you want to export data for in "Data Options" panel.

Step 4: Adjust "Save Folder" and "Case Name"

Step 5: Click "Write Commands" button to write commands to "TUI Commands" area