AE 433 COMPUTATIONAL FLUID DYNAMICS TERM PROJECT REPORT

FALL 2021/2022

Student Number – Surname, Name:
Project Title:

1. **Introduction**

   Give a short explanation about the project subject. For example, write if it is chambered or non-chambered wing, application area (aeronautics, sail, wind turbine, etc), laminar or turbulent flow, symmetrical or non-symmetrical. Give real dimensions and also nondimensionalized data of them. Describe both airfoils in separate figures using "Airfoil Nomenclature". Give them all in one page at most. Lastly, in short, give the outline of the report, no longer than half of a page.

2. **Literature Survey**

   - Search through internet (e-books, academic papers, etc) about your subject through scholar.google.com.
   - Find the oldest reference that you can find about your subject: For example, for NACA profiles, you can search through internet for extended name of NACA, which is "National Advisory Committee for Aeronautics". So you will find NACA is a U.S. federal agency founded on March 3, 1915, to undertake, promote, and institutionalize aeronautical research. On October 1, 1958, the agency was dissolved, and its assets and personnel transferred to the newly created National Aeronautics and Space Administration (NASA). NACA Airfoil series are founded and categorized in late 1940s **[1]**….
   - You are expected to find theory and background information relevant for your specific case and summarize your findings in a 2-3 page report (see Assessment Criteria and Suggested Structure at the end of the file for more details).
   - You should give references that can be your case study (ALL SHOULD BE AN EXPERIMENTAL STUDY). Give at least 2 different experimental studies for your study (a total of 4) that you can simulate. Be sure that all inlet, outlet, etc. and boundary conditions are given in those experimental studies.

3. **Purpose**

Write the purpose of the term project in a maximum of one page.

4. **Simulation Design**

Write necessary geometry dimensions in a table (Table 1). Draw the 2D/3D figures of your geometries and the solution domain in ANSYS DESIGN MODELER (Figure 1). Write down boundary conditions explicitly in another table (Table 2).

5. **ANSYS Student Edition Directives**

Give necessary directives that you used inside ANSYS Meshing and ANSYS Fluent, such as "Start > All Programs> ANSYS 2020 R1> Workbench 2020 R1" and support them with some

figures for making the project easy to follow and easy to replicate. You can add sub-titles such as "3.1 Geometry generation" "3.2 Mesh generation" and "3.3 Solution" for better understanding. You should describe the methodology that you have applied: mesh strategy, numerical scheme and solver settings, boundary conditions, etc.

6. Results

After the numerical analysis part of the project is completed, the work should be presented in the form of a technical report (no more than ten pages). A suggested structure for the report is given in the end of this file. Show XY plots (such as residuals, velocity plots, temperature plots, wall shear stress distribution, etc), residuals, graphical outputs (such as contours, vectors, etc), force outputs (such as drag, lift, shear etc), moment outputs whenever necessary.

7. Discussion and Conclusion

Write your outcomes and key findings regarding the subject that you surveyed. As final words, bring forward the name of your study (such as "A comparative study of NACA 0012 and NACA 0015 in a Wind Turbine Application")

8. References

Give references (especially for "Literature Survey" part). Take a look at TÜBİTAK site for better citing: https://www.tubitak.gov.tr/tr/duyuru/bibliyografik-verilerin-duzenlenmesi

*DO NOT EXCEED 20 PAGES IN TOTAL. Give a printed project report at the beginning of the final session (Jan 14th, 9.30 A.M.)

After you submit your project report, the last part of the project is to make an oral presentation of the project and the obtained results.. All students must be in Computer Lab strictly at 9.25 A.M. on Jan 14th.

Students will be randomly called out for their presentation limited to 10 minutes including Q&A. So, do not exceed 7-8 minutes for your powerpoint presentation.

Project Assessment Criteria for grading evaluation

The assessment of the numerical analysis project will be done using the criteria presented below. To get the maximum number of points it is necessary to meet all the marked criteria and participate in the oral presentation session.

Literature survey (part 1)

- Relevant literature for the problem found
- At least five relevant references used for the literature survey report
- The literature survey report gives a summary of ideas and concepts relevant for the chosen case

Numerical analysis work (part 2)

- A sound mesh strategy is used
    - High resolution is used where needed
    - The mesh is not unnecessarily fine in flow regions where only large scales needs to be resolved
    - Mesh stretching is used in a constructive way (In the report, figures showing the mesh for the whole domain and near the investigated object should be included)
- Numerical accuracy is addressed
    - At least three meshes with significant difference in cell count and/or meshing strategy were evaluated
    - At least two numerical schemes have been used (study the influence of numerical scheme on the solution for one case and one grid - order of accuracy, flux type)
- A mesh quality check was performed and the results were correctly interpreted

Project report (part 3)

- The report follows the “suggested structure” or similar (no more than ten pages)
- The report describes the case studied and related effects
- It is possible to redo the analysis using the information in the report (boundary conditions, mesh strategy, solver settings, etc.)
- The report should present details about the analysis work such as convergence criteria (for steady-state cases), chosen numerical scheme, etc.
- Software limitations are addressed
- The literature survey material is used to support the conclusions drawn from the results obtained in the numerical analysis work

Oral presentation (part 4)

- Well-structured presentation
- Case description
    - Geometry described
    - Boundary conditions are described
- Numerical approach described
    - Mesh strategy and mesh quality is discussed
    - Numerical scheme
    - Convergence criteria (if steady-state)
    - Findings from numerical accuracy are presented (if applicable)
- Results and discussion
    - Discussion on physical properties of the flow
    - References to literature (if applicable)
- Presentation finished on time (7-8 minutes per student + 1-2 minutes for questions)

Suggested Report Structure

1. Introduction

Introduction to the studied problem:

- Description of the application, relevance of effects (such as compressible flow) for the application
- References to published work related to your case (these publications can report work done on for example methods related to your work, investigations of related applications (numerical and/or experimental), investigations of related flow phenomena, etc.)

2. Method description

Describe your simulation approach in some detail:

- Mesh strategy (if a mesh sensitivity study has been done, the strategy for the study should be given here but the differences in the obtained results should be presented and discussed in the results and discussion)
- Chosen Numerical scheme (time and space. If an investigation of the effects on the predicted flow field of different numerical schemes has been done that should be reported here but the corresponding effect on the obtained results should be presented and discussed in the results and discussion sections)
- Turbulence model (described chosen approach if applicable)
- Convergence criteria (if steady-state), software limitations

3. Case description

Describe details of your case in terms of:

- Geometrical definition (schematic view of the case geometry, main features, difficulties, etc.)
- Boundary conditions (in-/outflow conditions, wall treatment, etc), hand calculations (if applicable)

4. Results

Present results from the simulations:

- Discuss numerical accuracy (if applicable)
    - Mesh dependency study (discuss differences in results as a consequence of mesh quality)
    - Compare results obtained using different numerical schemes (discuss the effects of numerical scheme on the flow predictions for your specific case)
    - What was the main findings when it comes to choice of mesh quality and numerical schemes
- Present results for best (believed to be most accurate) configuration (mesh, scheme)
- Discuss your results in physical terms (main flow features)

5. Discussion

- Summarize your study. Did you expect to get the result that you got (based on what you have learned about flows in the course and based on literature related to your specific case)?

6. Conclusions

- Short overview of study
- Main findings
- Reflection on results

7. List of References