

TURBOMACHINERY AND PROPULSION

PROJECT

Due Date: December 6 at 23:55

Instructions:

- Individual work only
- You must submit your solution in the form of a mini-report – Please see below on what to include in your report.
- Mini-report format (7 to 20 pages maximum):
 - Introduction and Literature Review
 - Problem Statement
 - Methodology
 - Results and Discussions
 - Conclusions
 - Appendices
- Write down your name and student ID clearly on the front page of the report.
- On Moodle, submit two files: one PDF file AND one CFX file. Use your name and student ID as the name of the file you submit, e.g. name_idnumber.pdf
- Make sure to submit your project well before the deadline to avoid potential internet problems.
- Only projects submitted on Moodle will be considered.

Problem statement:

Computational Fluid Dynamics (CFD) is an essential tool to the design and analysis of gas turbines and compressors. The objective of this project is to become familiar with one of the many commercially available CFD software.

The VKI turbine blade is a typical 1st stage gas turbine blade that is cylindrical in design (i.e. it consists of the same 2D cross section from hub to tip). The goal of this exercise is to set-up and simulate a linear cascade of the VKI blade using ANSYS CFX. The numerical solution will be post-processed in order to extract typical turbine performance parameters.

Follow the steps outlined below to successfully complete this assignment:

1. Generate the geometry of the VKI turbine blade. A .xlsx file is available on Moodle with coordinate points of the blade profile and flow path geometries. You can use the coordinate points to generate the 3D geometry using any CAD software of your choice.
2. Create a reasonably sized mesh (around 40,000 nodes) using ICEM CFD or ANSYS Workbench meshing applet. Discuss your mesh generation technique your mesh size.
3. Using ANSYS CFX, perform a CFD simulation of the meshed turbine cascade using the boundary conditions listed below and the $k-\varepsilon$ turbulence model.
4. Demonstrate graphically that your numerical simulation has converged by showing physical convergence of absolute values and numerical convergence. (please reference proper literature that supports your attainment of a convergent solution).
5. Post process your numerical solution, and calculate turbine performance parameters:
 - a) Calculate the isentropic Mach number at the inlet and exit of the cascade.
 - b) Plot the isentropic Mach number distribution over the suction and pressure surfaces of the blade.
 - c) Calculate the total pressure loss across the blade.
 - d) Provide colored contours of the velocity, pressure, and temperature across the cascade. Use a reasonable scale and provide units in your legend.
6. Discuss your results.

ANSYS CFX can be found in the following general labs:

In Hall Building:

H0811-00,H0821-00,H0825-00,H0843-00,H0961-01,H0961-02,H0961-03,H0961-04,H0961-06,H0961-07,H0961-10,H0961-11,H0961-13,H0961-14,H0961-15,H0961-17,H0961-19,H0961-21,H0961-23,H0961-25,H0961-26,H0961-27,H0961-28,H0961-29,H0961-31,H0961-33,H0964-00,H1067-00,

In EV Building :

EV008-101,EV009-139,EV009-245,EV012.161

ICEM CFD can be found:

- On Windows: in

H0815-00, H0819-00, H0821-00, H0921-00, H1067-00, EV008-101, EV008-165, EV009-139, EV009-245, EV012.161

- On Linux: in all labs, just run icemcfd from the command line.

Data Needed:

VKI blade profile and flow path geometries: A .xlsx file is available on Moodle with the coordinate points of the profile geometry of the blade.

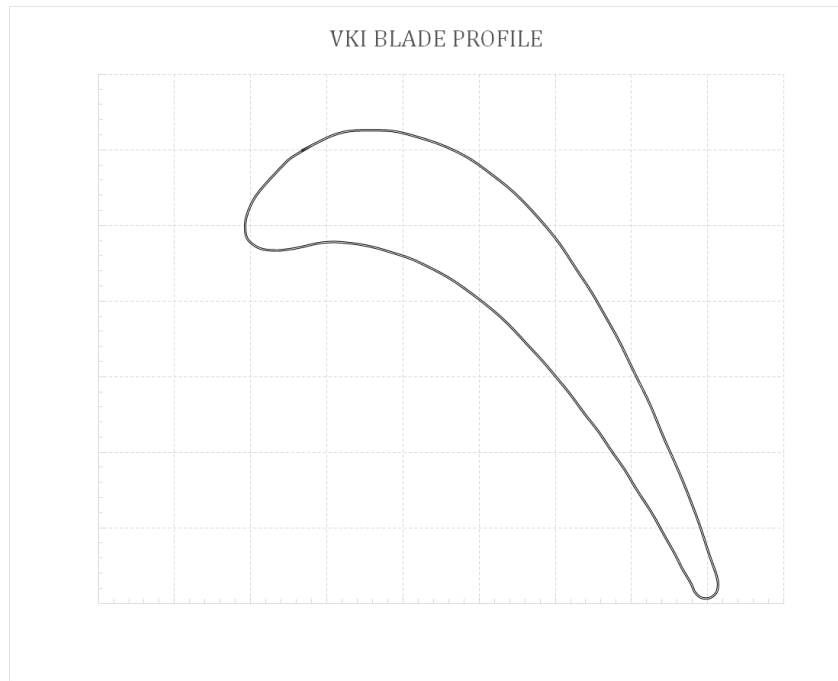


Figure 1: VKI blade profile

VKI cascade geometry:

Table 1: The VKI cascade geometric parameters

True chord length [mm]	80
Span (or height) [mm]	10-20
Stagger angle	38.5°
Inlet blade angle	30°
Inlet flow angle	30°
Outlet flow angle	69.5°
Pitch [mm]	113.5

Boundary conditions to the CFD domain:

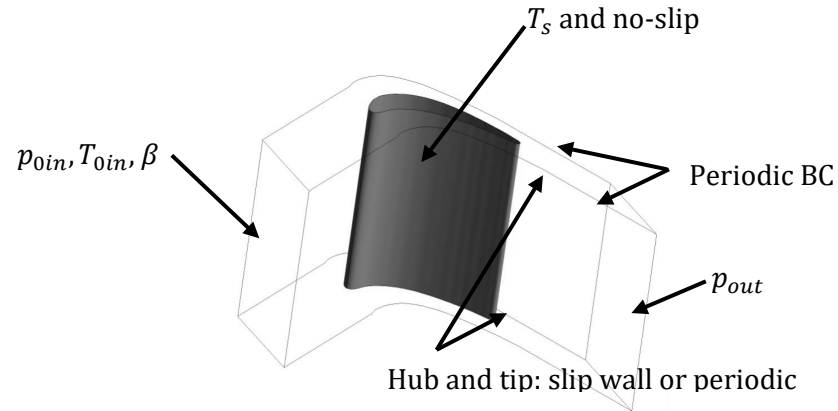


Figure 2: The three dimensional CFD domain

Table 2: Boundary conditions and other operating parameters

Inlet total temperature T_{0in}	415 K
Inlet total pressure p_{0in}	288 kPa
Turbulence	Medium Intensity
Inlet flow angle β	30°
Exit static pressure p_{out}	165.5 kPa
Blade wall temperature T_s	290 K
Re_c (Reynolds number based on chord)	8.5×10^5

What should be in your mini-report?

1. A figure showing your CFD domain, and boundary conditions
2. A figure showing your mesh, and explanation on how you generated it
3. Demonstration that your solution is mesh-independent
4. A figure showing the numerical convergence of your solution – refer to appropriate sources explaining what numerical convergence is
5. Post-processing:
 - a. Isentropic Mach number values at the inlet and exit of the cascade and how you calculated them
 - b. A figure of the isentropic Mach number distribution over the suction and pressure surfaces of the blade and an explanation of how you plotted it
 - c. The total pressure loss value across the blade and how you calculated it
 - d. Contours of velocity, pressure, and temperature across the cascade. Use a reasonable scale and provide units in your legend and discuss those figures