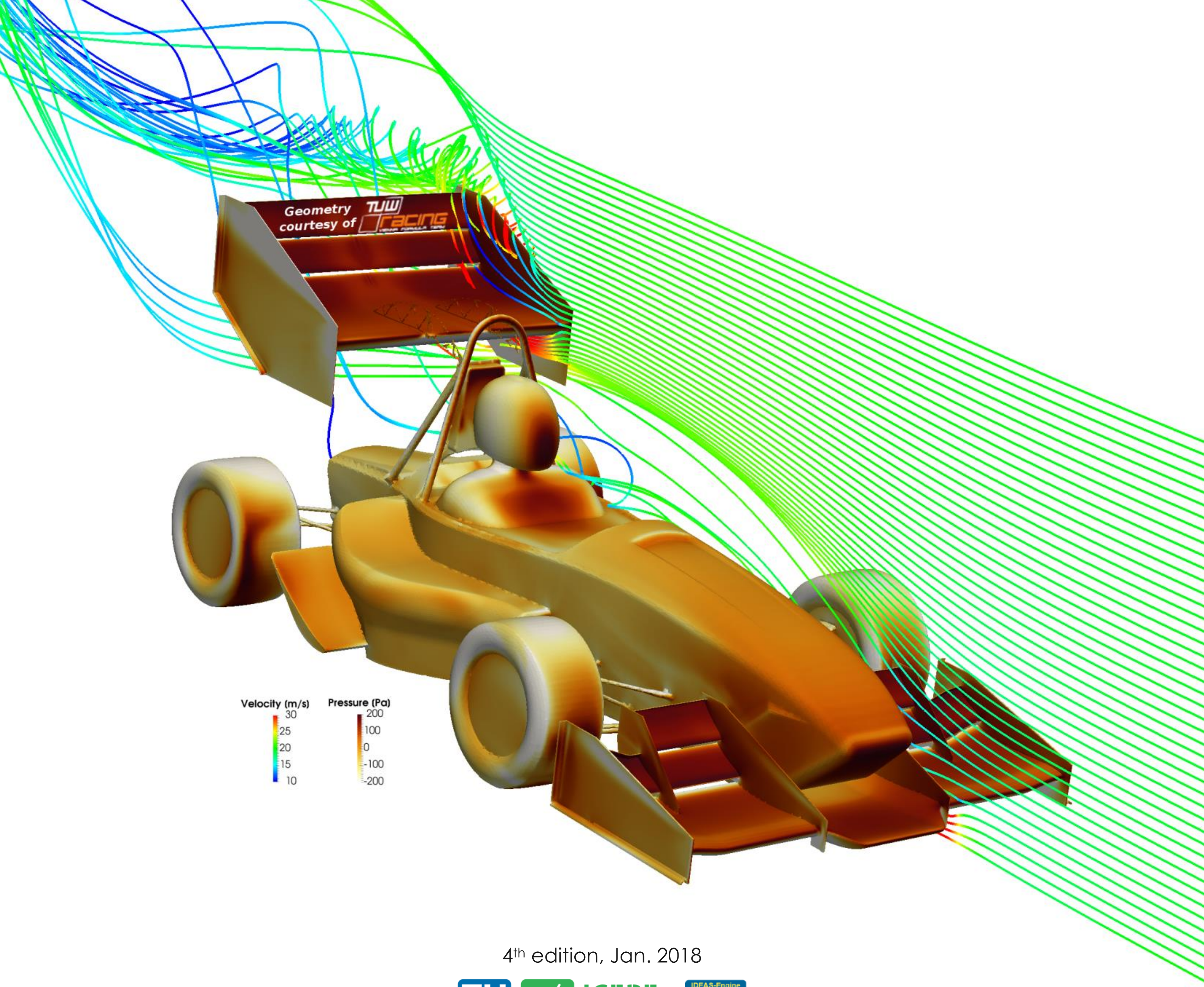
OpenFOAM® Basic Training



4th edition, Jan. 2018

This offering is not approved or endorsed by ESI® Group, ESI-OpenCFD® or the OpenFOAM® Foundation, the producer of the OpenFOAM® software and owner of the OpenFOAM® trademark.

Except where otherwise noted, this work is licensed under
http://creativecommons.org/licenses/by-nc-sa/3.0/

Editorial board:

- Bahram Haddadi
- Christian Jordan
- Michael Harasek

Technische Universität Wien
Institute of Chemical, Environmental & Bioscience Engineering

Compatibility:

- OpenFOAM® 5.0
- OpenFOAM® v1712

Cover picture from:

- Bahram Haddadi. Special thanks to Philipp Schretter and TUW-Racing.

Except where otherwise noted, this work is licensed under
http://creativecommons.org/licenses/by-nc-sa/3.0/

Attribution-NonCommercial-ShareAlike 3.0 Unported (CC BY-NC-SA 3.0)

This is a human-readable summary of the Legal Code (the full license).
Disclaimer
You are free:

- to Share — to copy, distribute and transmit the work
- to Remix — to adapt the work

Under the following conditions:

- Attribution — You must attribute the work in the manner specified by the author or licensor (but not in any way that suggests that they endorse you or your use of the work).
- Noncommercial — You may not use this work for commercial purposes.
- Share Alike — If you alter, transform, or build upon this work, you may distribute the resulting work only under the same or similar license to this one.

With the understanding that:

- Waiver — Any of the above conditions can be waived if you get permission from the copyright holder.
- Public Domain — Where the work or any of its elements is in the public domain under applicable law, that status is in no way affected by the license.
- Other Rights — In no way are any of the following rights affected by the license:
- Your fair dealing or fair use rights, or other applicable copyright exceptions and limitations;
- The author's moral rights;
- Rights other persons may have either in the work itself or in how the work is used, such as publicity or privacy rights.
- Notice — For any reuse or distribution, you must make clear to others the license terms of this work. The best way to do this is with a link to this web page.

This book has been used as a basis for preparing a series of video lectures
on youtube by Jozsef Nagy (JKU Linz):
www.youtube.com/channel/UCjdgpuxuAxH9BqheyE82Vvw
(Search for: Jozsef Nagy OpenFOAM at youtube.com)

In this OpenFOAM® tutorial series, we have prepared fourteen case examples that are designed to help users to learn the key utilities and features within OpenFOAM®, including mesh generation, multiphase modeling, turbulence modeling, parallel processing and reaction modeling. The base tutorial examples can be imported directly from the OpenFOAM® installation directory.

The tutorials should be primarily used for OpenFOAM® versions 5.0 and v1712, with differences in the running procedure between v1712 and 5.0 highlighted in blue boxes. So, simply ignore the blue boxes if you are running in version 5.0! The structure of each case example follow the below general structure:

- **Background**: an introduction about the key topics explored in the tutorial and the relevant CFD theory

1. **Pre-processing**: instructions on how to set up the correct case structure for a given problem using base case tutorials, with explanations on relevant dictionaries
2. **Running simulation**: instructions on running the solver and its associated commands
3. **Post-processing**: examining the results in OpenFOAM®'s post-processing application, ParaView V5.4.0

Tutorial One: **Basic Case Setup**

Solver: icoFoam
Geometry: 2-dimensional
Tutorial: elbow

Tutorial Two: **Built in Mesh**

Solver: sonicFoam
Geometry: 2-dimensional
Tutorial: forwardStep

Tutorial Three: **Patching Fields**

Solver: sonicFoam
Geometry: 1-dimensional
Tutorial: shockTube

Tutorial Four: **Discretization – Part 1**

Solver: scalarTransportFoam
Geometry: 1-dimensional
Tutorial: shockTube

Tutorial Five: **Discretization – Part 2**

Solver: scalarTransportFoam
Geometry: 2-dimensional
Tutorial: circle

Tutorial Six: **Turbulence, Steady state**

Solver: simpleFoam
Geometry: 2-dimensional
Tutorial: pitzDaily

Tutorial Seven: **Turbulence, Transient**

Solver: pisoFoam
Geometry: 2-dimensional
Tutorial: pitzDaily

Tutorial Eight: **Multiphase**

Solver: interFoam
Geometry: 2-dimensional
Tutorial: damBreak