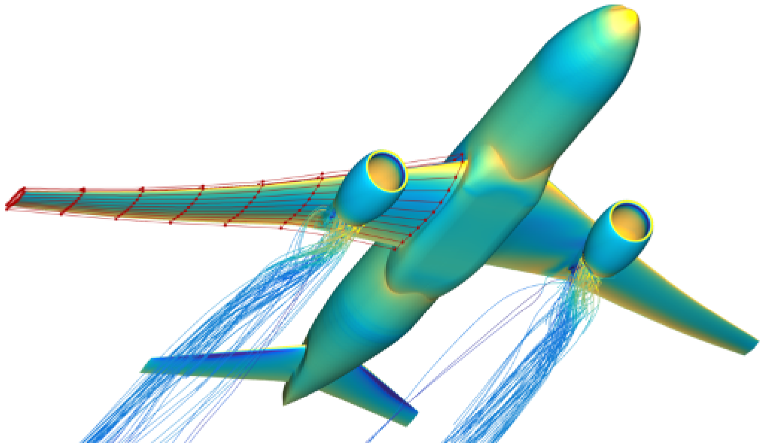
DAFoam Documentation

Release v1812

Ping He

Jun 18, 2019

Contents



CHAPTER 1

Test: Discrete Adjoint with OpenFoam

Test contains a suite of discrete adjoint solvers for OpenFOAM. These adjoint solvers run as standalone executives to compute derivatives. Test also has a Python interface that allows the adjoint solvers to interact with external modules for high-fidelity design optimization. Test has the following features:

- It implements an efficient discrete adjoint approach with competitive speed, scalability, accuracy, and compatibility.
- It allows rapid discrete adjoint development for any steady-state OpenFOAM solvers with modifying only O(100) lines of source codes.
- It supports design optimizations for a wide range of disciplines such as aerodynamics, heat transfer, structures, hydrodynamics, and radiation.

The Test repository comprises of five main directories:

- applications: adjoint solvers and utilities
- doc: documentation
- python: python interface to other optimization packages
- src: the core Test libraries
- tutorials: sample optimization setup for each adjoint solver

Contents:

1.1 Installation

Test runs on Linux systems and is based on **OpenFOAM-v1812**. You must install **OpenFOAM** and verify that it is working correctly. You also need to install the 3rd party and **MDOLab** packages before using **Test** for optimization. Other dependencies include:

- C/C++ compilers (gcc/g++ or icc/icpc)
- Fortran compiler (gfortran or ifort)

- MPI software (openmpi, mvapich2, or impi)
- Swig
- cmake

To compile the documentation, you also need:

1.2 Tutorials

1.2.1 simpleFoam

OK this is simpleFoam

1.2.2 rhoSimpleFoam

OK this is rhoSimpleFoam