The current OpenFOAM landscape

Johan Rønby

DANSIS OpenFOAM Seminar
27 March 2019
What is OpenFOAM?

• OpenFOAM is an open source CFD software toolkit written in C++
• Has gone "viral" within a wide range of engineering fields
• A platform for collaboration on use and development of CFD in industry and academia

Example:
Primary diesel jet atomization
Credits: Dr. Vuko Vukčević
Note: Preliminary work
Registration is now open for the 7th OpenFOAMConference on 15-17 October 2019 in Berlin, Germany. Click on the link for details!

---

Group rules

- All posts to this Group must be clearly related to OpenFOAM.
- Try to minimise advertising of events, products and services to encourage discussions.

We delete posts that ignore these rules.

---

Group owner

Jenya Collings - 1st
Co-founder, Operations Director at CFD Direct Limited
A brief OpenFOAM history

- 1990-2000: **FOAM** initially created at Imperial College by Henry Weller developed with Hrvoje Jasak as the other main developer
- 2004: Weller+ founds OpenCFD releasing **OpenFOAM** and Jasak founds Wikki releasing OpenFOAM-extend (now foam-extend)
- 2004: Jasak+ organizes 1st OpenFOAM Workshop which becomes backbone a successful community around the foam-extend.
- 2011: SGI acquires OpenCFD and the OpenFOAM Foundation is founded
- 2012: ESI acquired OpenCFD from SGI
- 2013: OpenCFD started their annual OpenFOAM Conferences.
- 2015: Weller+ leaves OpenCFD and founds CFDDirect
- Status quo: OpenCFD, OpenFOAM Foundation and foam-extend community release each their OpenFOAM version

Main sources: openfoam.com/history, wikipedia.org/wiki/OpenFOAM
The current OpenFOAM main players

1. **OpenFOAM**
   - Website: www.OpenFOAM.org
   - Led by Henry Weller
   - Holds code copyright
   - Current release: OpenFOAM-6
   - OpenFOAM Open Day

2. **CFD Direct**
   - Logo
   - Website: www.OpenFOAM.org
   - Led by Henry Weller
   - Holds code copyright
   - Current release: OpenFOAM-6
   - OpenFOAM Open Day

3. **esi**
   - Logo
   - Website: www.OpenFOAM.com
   - Led by Fred Mendonca
   - Holds OpenFOAM trademark
   - Current release: OpenFOAM-v1812
   - OpenFOAM Conference

4. **FSB**
   - Logo
   - Website: www.sourceforge.net/projects/foam-extend/
   - Led by Hrvoje Jasak
   - "Holds" the community
   - Current release: foam-extend-4.0
   - OpenFOAM Conference

5. **WIKI**
   - Logo
   - Website: www.sourceforge.net/projects/foam-extend/
   - Led by Hrvoje Jasak
   - "Holds" the community
   - Current release: foam-extend-4.0
   - OpenFOAM Workshop