

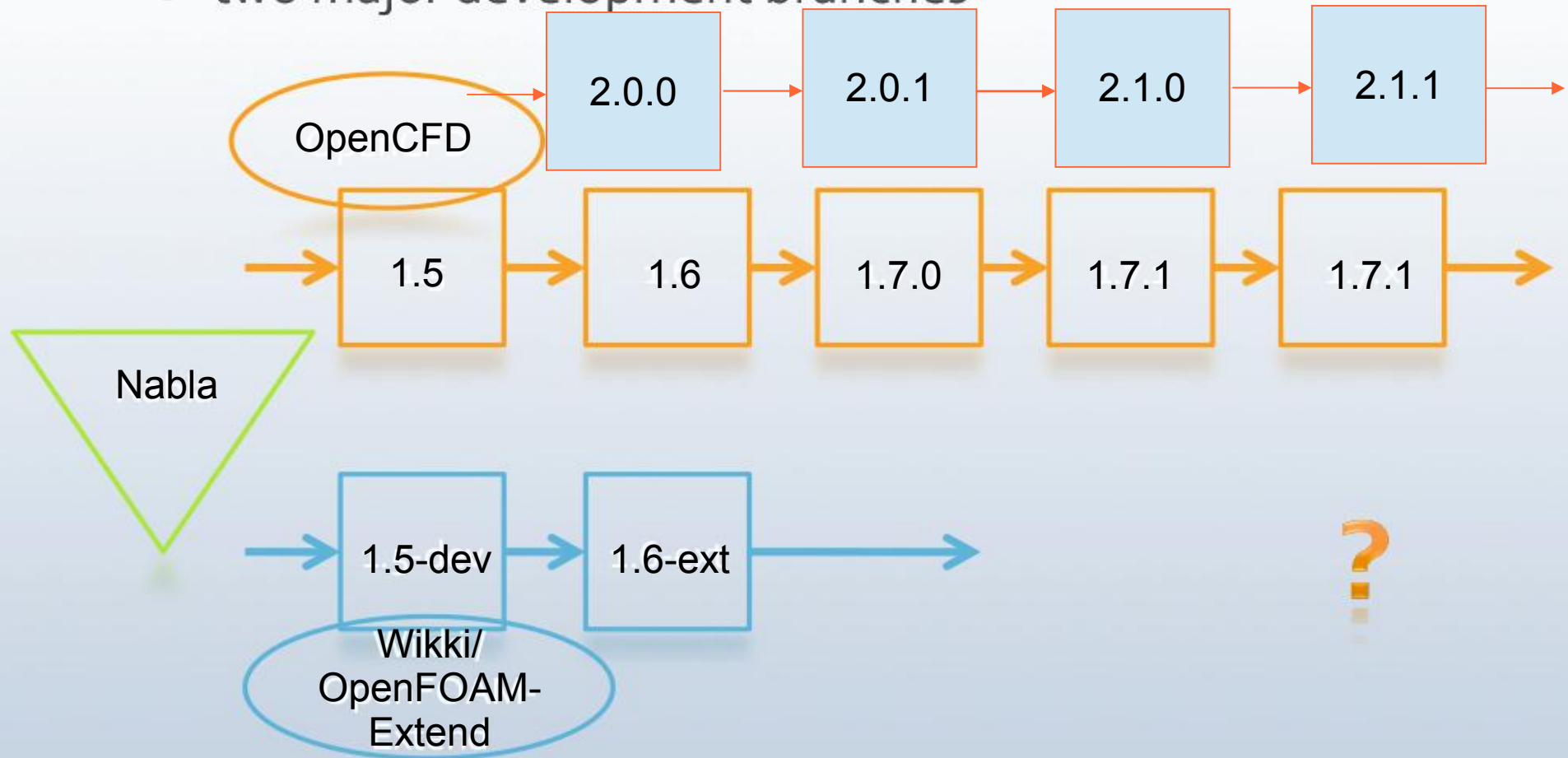
First Israel OpenFOAM® user group meeting

30.7.2012

9:40-9:50	Prof. Aviv Rosen, Technion Aerospace faculty, opening remarks
9:50-10:00	Hanan Einav-Levy, Technion Aerospace faculty, overview of OpenFOAM world community
10:00-10:20	Philipp Hofemeier, Technion biofluids laboratory
10:20-10:30	Dr. Seror Stephane – Head IAI CFD Dept - Why OpenFOAM as a R&D CFD platform Strategy and Risk Evaluation
10:30-10:40	Alex Landsberg, Eshkol Ltd./IAI - installation and compilation of OF on Infiniband Intel i7 HPC Cluster
10:40-11:00	Marom Shlomo, IAI CFD Dep. - application and comprehensive evaluation of OF for Aerospace Industrial full aircraft configurations - insight on bad drag accuracy of OF solvers
11:00 – 11:20	Eli Friedman, IAI CFD Dept - towards accurate OpenFOAM RANS flow solver - recent improvement of OF by IAI C++ implementation of the 2nd order accurate scheme AeroFoam
11:20 – 11:40	Lev Magazinnik, Ben Gurion University of the Negev
11:40 – 11:50	Coffee break
11:50 – 12:30	Discussion - How to organize the Israeli OF user group, agendas and interests

what are the versions of OpenFOAM?

- two major development branches



From David Boger's slides, OFW6

what is OpenCFD Ltd.?

- Established in 2004 by Henry Weller, coinciding with the release of OpenFOAM under general public license
- From www.openfoam.com/about: “OpenCFD® produces the OpenFOAM® open source CFD toolbox and documentation and distributes it through this web site. OpenCFD provides contracted **developments, support and training** for users of OpenFOAM”

From David Boger's slides, OFW6

what is the Extend Project?

- Originated by Wikki, Ltd. (Hrv Jasak et al.) in 2004
- Core group
 - Hrv Jasak, Hakan Nilsson, Bernhard Gschaider, Henrik Rusche, Holger Marschall, Martin Beaudoin
- From www.extend-project.de: "The goal of the Extend-Project is to open the OpenFOAM® CFD toolbox to **community contributed extensions** in the spirit of the OpenSource development model."
- Organizers of the **OpenFOAM workshops**

From David Boger's slides, OFW6

OF workshop's overview

- Training day
- Lectures (1-2 days)
- Special interest groups
 - Turbo-machinery
 - Turbulence
 - Wind energy
 - Multi-phase
 - ...



OF workshop's overview

Monday, June 25, 2012

Training Courses on OpenFOAM® Technology

	Track 1 Basic Level 1 (A/B/C)	Track 2 Advanced Level 2 (E)	Track 3 Advanced Level 2 (D)	Track 4 Advanced Level 2 (D)
8:30*-10:30	Running my First Case with OpenFOAM; I. Buntić-Ogor (room: hassium) (slides material)	A comprehensive tour of snappyHexMesh; E. de Villiers (room: helium2) (slides)	Thin liquid film flow simulation with OpenFOAM; Z. Tuković (room: titanium3) (slides material)	Turbomachinery; H. Nilsson (room: germanium) (slides material)
10:30-10:45	Coffee Break			
10:45-12:45	Generic external aerodynamic simulation training; T. Grahs (room: hassium) (slides material)	C++ in protectOpenFOAM - Object orientation, generic programming and polymorphism by examples ; T. Marić (room: titanium3) (slides material)	A deeper look into Lagrangian spray source code; P. Keller (room: germanium) (slides)	Multiphysics Simulation - Inter-Region and Inter-Equation Coupling in OpenFOAM; H. Jasak (room: helium2) (slides)
12:45-13:30	Lunch			
13:30-15:30	How to add a Scalar Transport Equation to scalarTransportFoam; H. Rusche (room: hassium) (slides material)	Programming Model Libraries from Scratch; K. Kissling (room: titanium3) (slides material)	Immersed Boundary Method in OpenFOAM; Z. Tuković (room: helium2) (slides material)	Marine Hydrodynamics; E. Patterson (room: germanium) (slides material)
15:30-15:45	Coffee Break			
15:45-17:45	Automatic testing of solvers using PyFOAM; B.F.W. Gschaider (room: helium2) (slides material)	A brief Introduction to Function Objects; J. Höpken (room: hassium) (slides)	Adaptive Tetrahedral Re-Meshing for Deforming Domain Simulations; K. Mooney (room: germanium) (slides material)	Fluid Structure Interaction Simulations with OpenFOAM; M. Lombardi (room: titanium3) (slides material)

OF workshop's overview

Monday, June 25, 2012

Training Courses on OpenFOAM® Technology

Track 1	Track 2	Track 3	Track 4
Basic Level 1 (A/B/C)	Advanced Level 2 (E)	Advanced Level 2 (D)	Advanced Level 2 (D)

The 8-th OpenFOAM® Workshop will be held in South Korea, Jeju from June 11 (Tue) to 14 (Fri), 2013. Venue is the International Convention Center (ICC).

10:45-12:45	Generic external aerodynamic simulation training; T. Grahs (room: hassium) (slides material)	Object orientation, generic programming and	A deeper look into Lagrangian spray source code; P. Keller	Multiphysics Simulation - Inter-Region and Inter-Equation Coupling in OpenFOAM; H. Jasak (room: helium2) (slides)
12:45-13:30				
13:30-15:30	How to add a Scalar Transport Equation to scalarTransportFoam; H. Rusch (room: hassium) (slides material)			Method in material
15:30-15:45				
15:45-17:45	Automatic testing of solvers using PyFOAM; B.F.W. Gschaider (room: helium2) (slides material)			Fluid Structure Interaction Simulations with OpenFOAM; M. Lombardi (room: titanium3) (slides material)



Where to get and give help

- OpenCFD
 - www.openfoam.com/docs
- The Forum
 - www.cfd-online.com/forums/openfoam
- The wiki
 - http://openfoamwiki.net/index.php/main_FAQ
- The OpenFOAM® Extend Project
 - <http://www.extend-project.de>
 - SIG
 - User groups
- Previous workshop presentations and training
 - <http://www.openfoamworkshop.org/2012/downloads/>
 - <http://www.openfoamworkshop.org/2011>
 - ...



Israel OF user group - discussion

Agenda ideas

- Exchange information
- Organize user group meetings
- Organize hands-on training meetings

Tools

- Discussion forum – extend-project.de/
- Wiki page – openfoamwiki.net/