

A note on different versions, a historical view

First some history, based on the relation between the 1.5 versions.

- **OpenFOAM-1.5** is distributed by OpenCFD, at www.openfoam.org (now www.openfoam.com). Static pre-compiled version without bug fixes.
- **OpenFOAM-1.5.x** is a patched (bug-fixed) version of OpenFOAM-1.5, and is also distributed by OpenCFD at www.openfoam.org (now www.openfoam.com). Not pre-compiled. Download and update using Git (version control system).
- **OpenFOAM-1.5-dev** is an extended version, distributed by Hrvoje Jasak with contributions from the community, and can be found at:
<http://openfoam-extend.wiki.sourceforge.net/>
It is regularly extended and bug-fixed. Not pre-compiled. Download and update using Subversion (version control system – the dev version was first!).
- **User contributions** at <http://openfoam-extend.wiki.sourceforge.net/>, with corresponding descriptions at <http://openfoamwiki.net>

A note on different versions, today

This is what it looks like today:

- **OpenFOAM-1.7.1, Ubuntu/Debian pack** Is installed using a packet manager, which means that you must trust a packet repository that is not supported by Ubuntu. Is updated monthly by OpenCFD.
- **OpenFOAM-1.7.1, Source pack** The source of the original 1.7.1 distribution. Must be compiled. Is not updated with bug fixes.
- **OpenFOAM-1.7.x** is the patched (bug-fixed) version of OpenFOAM-1.7. It is updated regularly with bug fixes etc.
- **OpenFOAM-1.5-dev** is still the most recent dev version, contributed by the community. It contains lots of implementations that is not available in the OpenCFD versions. It is regularly extended and bug-fixed.
- **OpenFOAM-dev-1.5, Ubuntu/Debian pack** Similar to the same for 1.7.1 (but this one is maintained by the community, and it was first!). See instructions at:
`http://openfoamwiki.net/index.php/Installation#OpenFOAM-dev-1.5_under_Ubuntu_Lucid_10.04_LTS`
- **User contributions** at `http://openfoam-extend.wiki.sourceforge.net/`, with corresponding descriptions at `http://openfoamwiki.net`

Some more details on OpenFOAM-1.5-dev

- OpenFOAM-1.5-dev is maintained by Professor Hrvoje Jasak.
- Hosted at SourceForge (<http://openfoam-extend.wiki.sourceforge.net/>)

- Exact location:

<http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/Core/OpenFOAM-1.5-dev/>

- Has all the features in OpenFOAM-1.5.x, but also many extensions.
- Check out source code using the Svn version control system (instructions at the link above)

Basically:

```
svn checkout <theLinkAboveButWith'svnroot' InsteadOf 'viewvc' >
```

- No binaries distributed - you have to compile everything!
- Use the same Third-Party products as OpenFOAM-1.5
- Update your installation using Svn, and re-compile to have the latest version.

Update by going to the OpenFOAM-1.5-dev directory, and type:

```
svn update
```

Some more details on user contributions

- The OpenFOAM-extend project at SourceForge:
<http://openfoam-extend.wiki.sourceforge.net/> was developed by OpenFOAM users to allow user contributions.
- See:
<http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/>
- The main contribution is OpenFOAM-1.5-dev, click on Core
- Many plug-in libraries and applications can be found, click on Breeder-1.5
- The OpenFOAM Wiki, <http://openfoamwiki.net/>, is the place for descriptions of the code in the OpenFOAM-extend project.
- OpenFOAM Working Groups share files and information at OpenFOAM-extend and in the OpenFOAM Wiki, see:
http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/Breeder_1.5/OSIG/
http://openfoamwiki.net/index.php/Main_Special_Interest_Groups