

Q Search or go to...

Development / openfoam / Wiki / page access code

Project

openfoam

Manage

Plan

Issues

Issue boards

Milestones

Wiki

Code

Deploy

Analyze

## page access code

Last edited by Mark Olesen 3 years ago



OpenFOAM repositories are hosted on the [develop.openfoam.com](https://develop.openfoam.com) site. Many are **public-access** and can be downloaded or cloned directly; to access other internal repositories and take advantage of advanced features (such as reporting issues), users must create an account.

Repository	Issue tracker
OpenFOAM	issues
ThirdParty build scripts	issues
Additional modules	individual
packaging/containers	-

- Organization of git branches
- Additional Information
  - Exploring projects
    - Terminal mode (cloning)
      - Get a snapshot
      - Shallow cloning
      - Update existing snapshot
    - Registration
    - Adding a SSH key

### Organization of git branches

OpenFOAM follows a [branching workflow](#) with the following main branches:

Branch name	Description
master	Bugfix updates since the last release, should always be stable and runnable (suitable for production)
develop	Current developments, slated for the next release. The code in develop is generally quite stable, but minor regressions may occasionally occur.
maintenance-v1912 ...	Bugfix branches for previous releases, should always be stable and runnable (suitable for production)
feature-XYZ	Work-in-progress feature branch (primarily internal use, no guarantees)

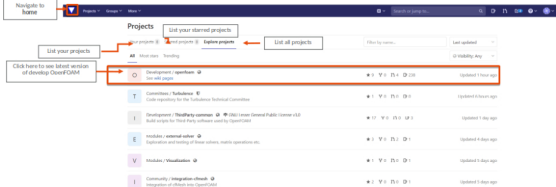
Most people will use the **master** branch to have up-to-date bugfixes of the current release. When a new code version is released, **master** is renamed **maintenance--(prevRelease)** and a new **master** is created at release.

The **feature** branches (as denoted by their prefix), have no guarantee of being stable and are liable to change at any time. Since they will also occasionally be rebased onto the latest **develop**, there is **no** guarantee of continuity.

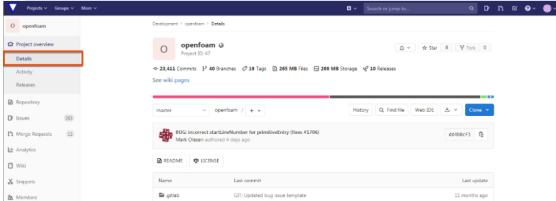
### Additional Information

#### Exploring projects

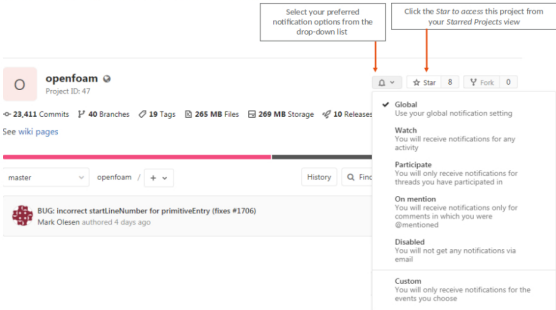
- From the projects menu you can navigate through various projects as shown below.



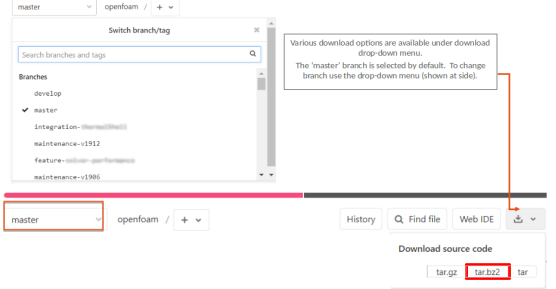
- Selecting a project, e.g. **openfoam** or **ThirdParty:common**, provides further details. Each project has a similar structure; the main OpenFOAM repository is shown below.



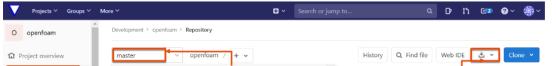
- Options for managing notifications, marking favourites (starred projects), or accessing git clone features are located on the right side of webpage as shown below. See the [Cloning](#) section for more information.



- If you are looking for a one-off download for any particular branch (with no git history), browse to the required branch and select from the download options:



- Alternatively, browse a repository for files, branches, tags, etc. via the **Repository** section:

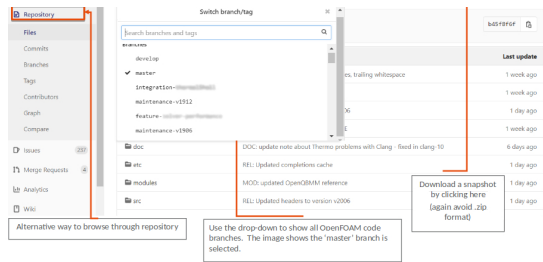


- On this page
- Organization of git branches
  - Additional Information
  - Exploring projects
  - Terminal mode (cloning)
  - Get a snapshot
  - Shallow cloning
  - Update existing snapshot
  - Registration
  - Adding a SSH key

#### Pages

Q Search pages

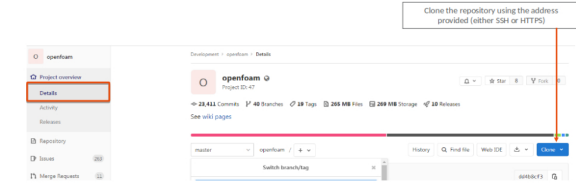
- building
  - cross compile mingw
- coding
  - git workflow
- patterns
  - dictionary
  - HashTable
  - memory
  - parallel
  - patterns
  - precision
  - registry
  - selectors
  - strings
  - scripts
    - scripts
  - style
    - filenames
    - style
- configuring
- Home
- icons
- info
- modules
  - visualization
- packaging
  - debian
  - Locations
  - README
- sourceforge
  - README
- suse
  - Locations
- page access code
- page build code
- page feature requests
- precompiled
  - apptainer
  - docker
  - docker old
  - redhat
  - suse
  - windows
- running
  - openfoam selector
  - shell session
  - Submitting issues
- tuning
- upgrade
  - upgrade
  - v3 Developer Upgrade Guide
  - v3 User Upgrade Guide
  - v1606 Developer Upgrade Guide
  - v1606 User Upgrade Guide
  - v1612 Developer Upgrade Guide
  - v1612 User Upgrade Guide
  - v1612 utility postProcess
  - v1706 Developer Upgrade Guide
  - v1712 Developer Upgrade Guide
  - v1712 User Upgrade Guide
  - v1806 Developer Upgrade Guide
  - v1806 User Upgrade Guide
  - v1812 Developer Upgrade Guide
  - v1812 User Upgrade Guide
  - v1906 Developer Upgrade Guide
  - v1906 User Upgrade Guide
  - v1912 Developer Upgrade Guide
  - v1912 User Upgrade Guide



## Terminal mode (cloning)

### Get a snapshot

- Cloning allows you to synchronize your local code with the latest developments. The address for cloning is available on the right hand side of Project overview -> Details slider of the repository:



- Clone the default **master** branch via HTTPS using

```
git clone https://develop.openfoam.com/Development/openfoam.git
```

- Optional:** You may specify a target branch directly using

```
git clone -b <local branch name> <git url> <name of resulting directory>
```

e.g. to clone the **develop** branch to your local **OpenFOAM-com** directory via HTTPS:

```
git clone -b develop https://develop.openfoam.com/Development/openfoam.git OpenFOAM-com
```

- Since OpenFOAM-v1712, community contributions may be included using the **git submodule system**. To initialise the submodules located in the **\$WM\_PROJECT\_DIR/modules** directory:

```
git submodule init
```

The respective modules/README contains more information.

### Shallow cloning

- Shallow cloning is a clone of the last repository state without its entire history. This makes for a smaller repository and faster cloning.

```
git clone --depth=1 https://develop.openfoam.com/Development/openfoam.git
cd openfoam
git submodule init && git submodule update --depth=1
```

If you want more than just the very last commit, adjust the clone depth accordingly. For example, to obtain the repository with the last 8 commits:

```
git clone --depth=8 https://develop.openfoam.com/Development/openfoam.git
```

### Update existing snapshot

- The repositories are updated regularly to include bug fixes and new features. Retrieve the updates using:

```
git pull
```

- To update the additional community repositories, issue the command:

```
git submodule update
```

- Note:** If you experience merge conflicts during the pull due to your own local code changes, you will need to resolve them manually/reset the state of your repository and re-pull

Quite often, it can be useful to monitor changes without actually updating your local files:

```
git fetch
```

## Registration

- Go to <https://develop.openfoam.com> and create a new user account
- A confirmation email will be sent to the registered address (check the spam folder if no emails are received after a few minutes)
- Follow the link to complete the sign-up process

### OpenFOAM Development

Join the OpenFOAM community!

Looking for the latest version of OpenFOAM? Take a look at <https://develop.openfoam.com/Development/openfoam>  
Register/login in to access all the Community repositories!

Click here to register

Fill all the required details

Check the box and proceed as directed

v2006 Developer Upgra...  
v2006 User Upgrade Gui...  
v2012 Developer Upgrad...  
v2012 User Upgrade Gui...  
v2106 Developer Upgrad...  
v2106 User Upgrade Gui...  
v2112 Developer Upgrad...  
v2112 User Upgrade Guide  
v2206 Developer Upgra...  
v2206 User Upgrade Gui...  
v2212 Developer Upgrad...  
v2212 User Upgrade Guide  
v2306 Developer Upgra...  
v2306 User Upgrade Gui...  
v2312 Developer Upgrad...  
v2312 User Upgrade Gui...  
v2406 Developer Upgra...  
v2406 User Upgrade Gui...  
v2412 Developer Upgrad...  
v2412 User Upgrade Guide

[View all pages](#)