Hydro Québec
Institut de recherche

Exploring OpenFOAM source code:

There and back again.

Martin Beaudoin

IREQ, Hydro-Québec's Research Institute

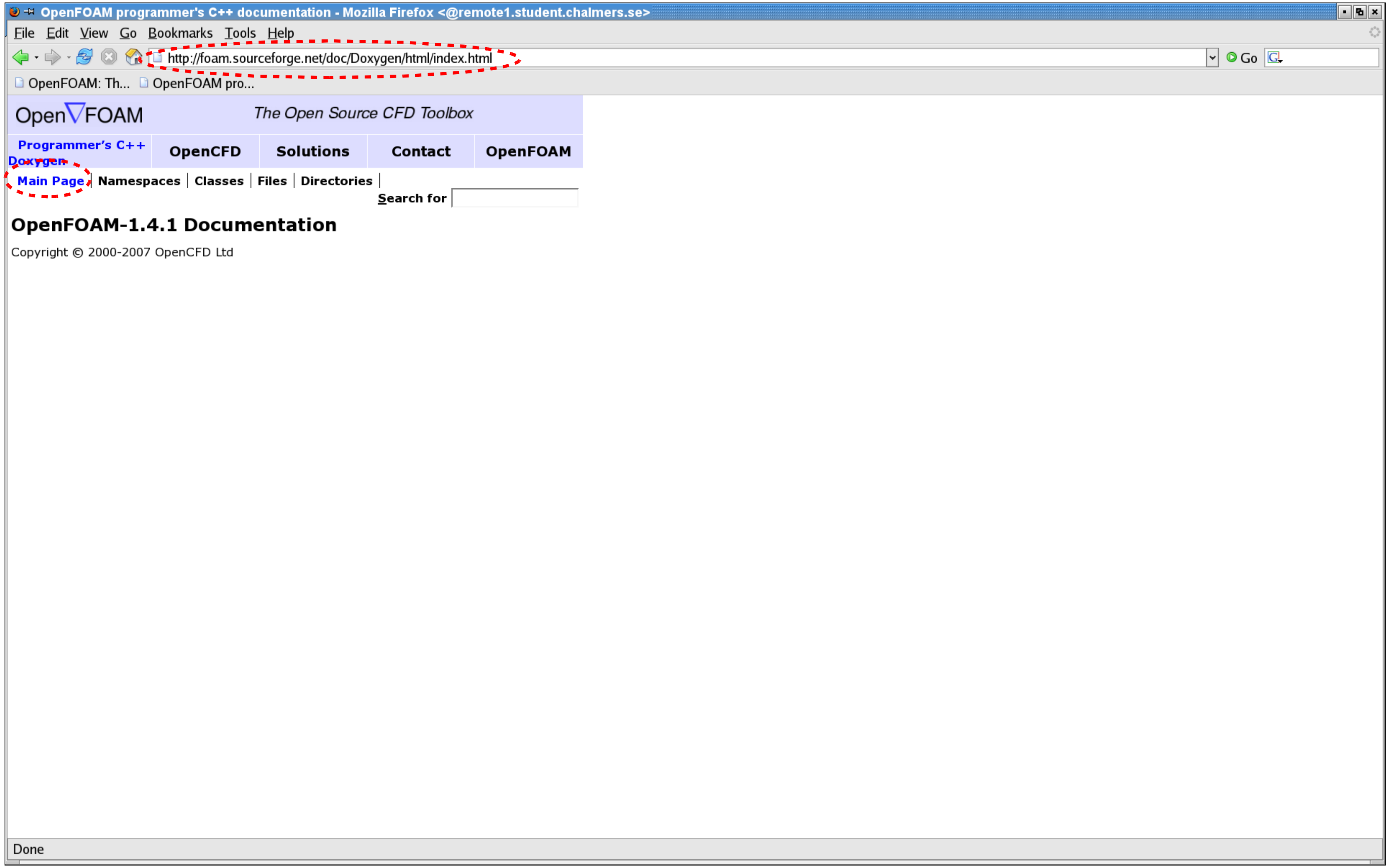
Three different facets of the same information

- **C++ Classes definitions**
  - How to navigate the C++ class definitions using Doxygen
- **Directory and file structure**
  - How to navigate and find information efficiently using Unix commands.
- **Evolution through time**
  - How to navigate the file revision history using revision control tools.

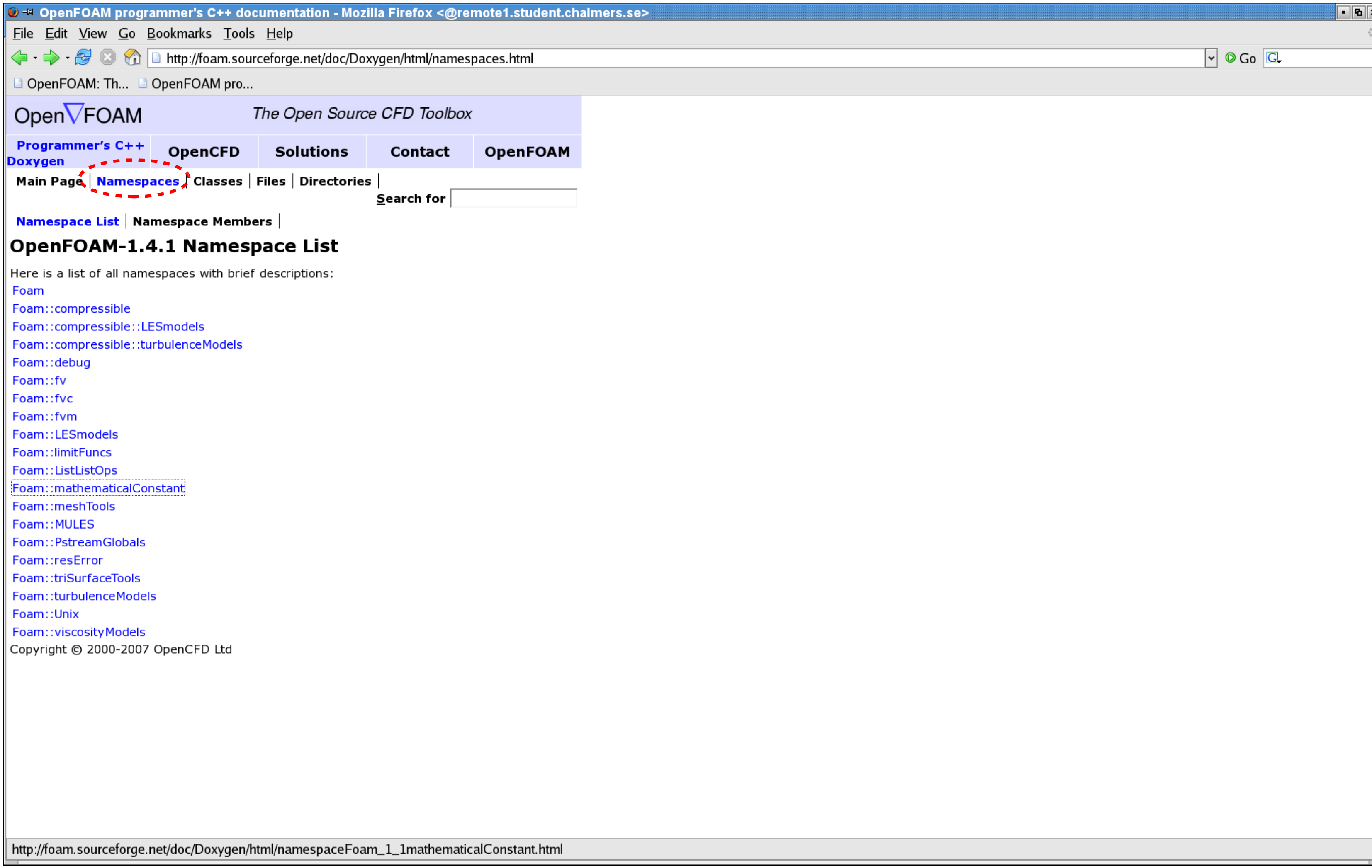
Navigating the OpenFOAM class maze:

- **Doxygen:**
  - Documentation system for C++, Java, Python, C, etc.
  - Generates:
    - On-line documentation in HTML from C++ source code
    - Inheritance Class diagram
    - Collaboration class diagram
    - Hyperlinks so you can navigate swiftly through the class hierarchy
    - Hyperlinks to the actual source code class definition (.H files)
    - Search facility available if using a Web server (PHP script)
  - OpenFOAM 1.4.1: HTML documentation available on-line:
    - http://foam.sourceforge.net/doc/Doxygen/html/
  - The configuration files for generating your own local copy of the Doxygen documentation is also available
    - $WM_PROJECT_DIR/doc/Doxygen/

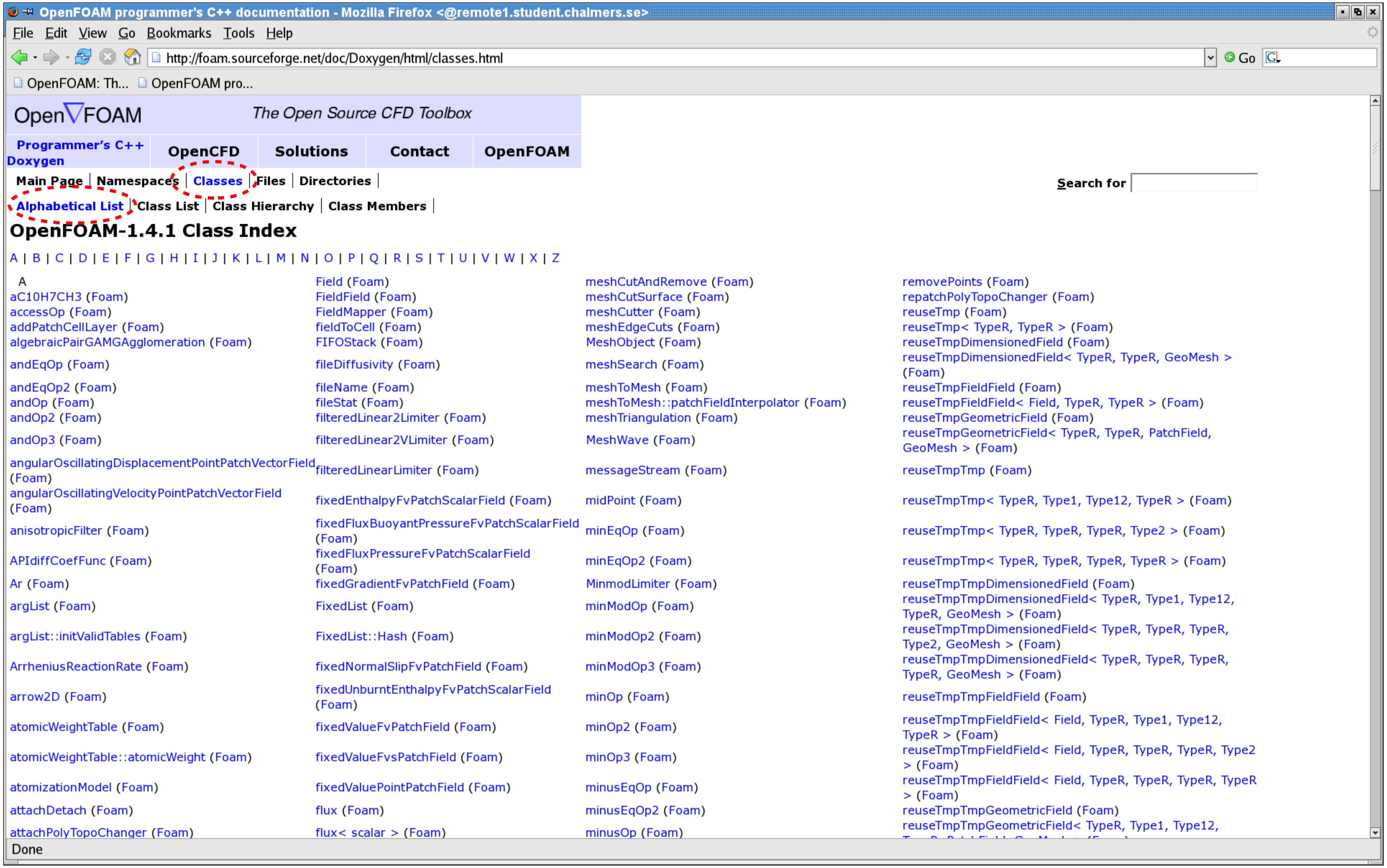
A quick overview of OpenFOAM Doxygen doc: (1/14)



A quick overview of OpenFOAM Doxygen doc: (2/14)



A quick overview of OpenFOAM Doxygen doc: (3/14)


OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>

File Edit View Go Bookmarks Tools Help

http://foam.sourceforge.net/doc/Doxygen/html/classes.html

OpenFOAM: Th... OpenFOAM pro...

OpenFOAM The Open Source CFD Toolbox

Programmer's C++ Doxygen | OpenCFD | Solutions | Contact | OpenFOAM

Main Page | Namespaces | Classes | Files | Directories |

Search for

Alphabetical List | Class List | Class Hierarchy | Class Members |

OpenFOAM-1.4.1 Class Index

A | B | C | D | E | F | G | H | I | J | K | L | M | N | O | P | Q | R | S | T | U | V | W | X | Z

A
aC10H7CH3 (Foam)
accessOp (Foam)
addPatchCellLayer (Foam)
algebraicPairGAMGAgglomeration (Foam)
andEqOp (Foam)
andEqOp2 (Foam)
andOp (Foam)
andOp2 (Foam)
andOp3 (Foam)
angularOscillatingDisplacementPointPatchVectorField (Foam)
angularOscillatingVelocityPointPatchVectorField (Foam)
anisotropicFilter (Foam)
APIdiffCoefFunc (Foam)
Ar (Foam)
argList (Foam)
argList::initValidTables (Foam)
ArrheniusReactionRate (Foam)
arrow2D (Foam)
atomicWeightTable (Foam)
atomicWeightTable::atomicWeight (Foam)
atomizationModel (Foam)
attachDetach (Foam)
attachPolyTopoChanger (Foam)

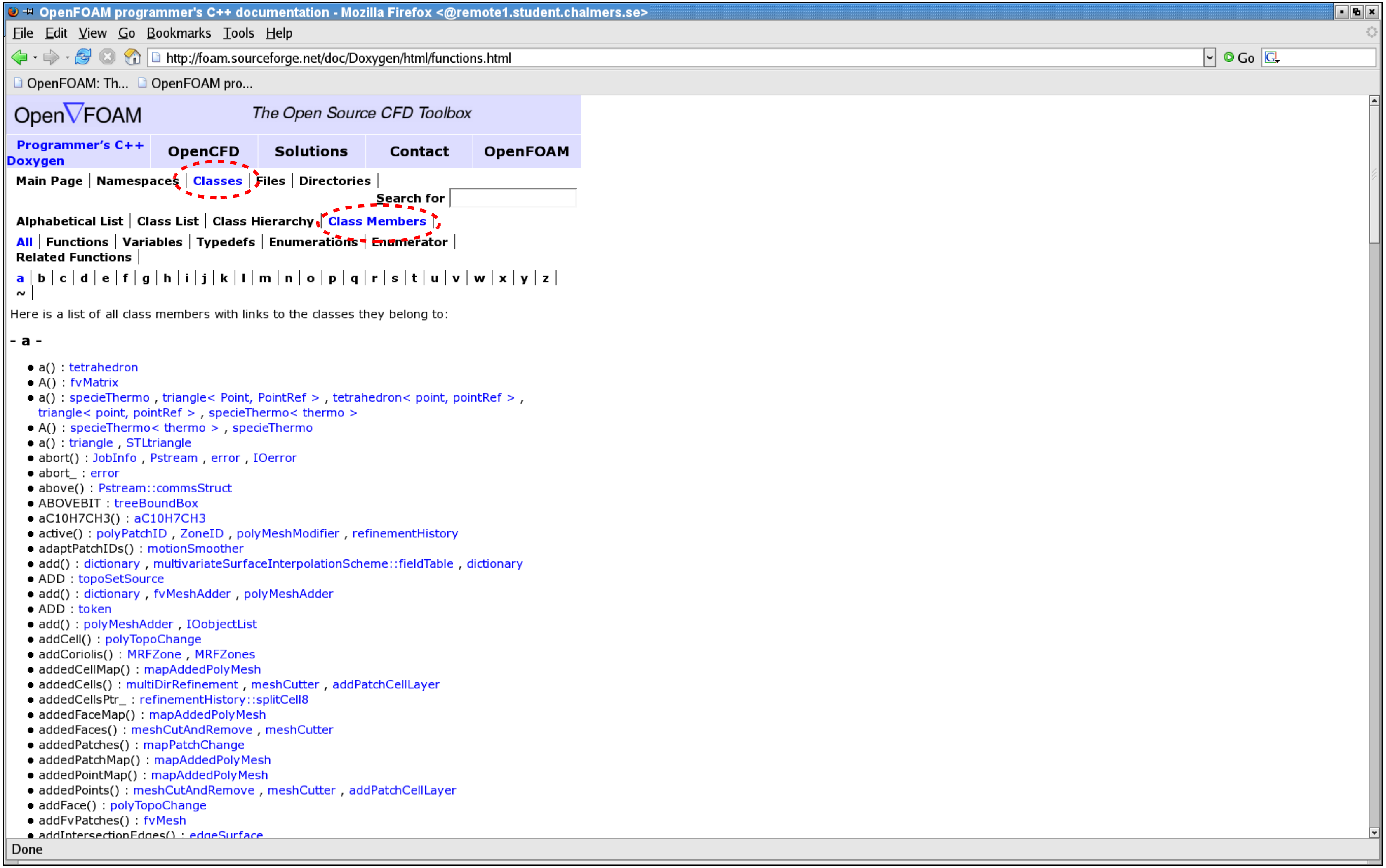
Field (Foam)
FieldField (Foam)
FieldMapper (Foam)
fieldToCell (Foam)
FIFOStack (Foam)
fileDiffusivity (Foam)
fileName (Foam)
fileStat (Foam)
filteredLinear2Limiter (Foam)
filteredLinear2VLimiter (Foam)
filteredLinearLimiter (Foam)
fixedEnthalpyFvPatchScalarField (Foam)
fixedFluxBuoyantPressureFvPatchScalarField (Foam)
fixedFluxPressureFvPatchScalarField (Foam)
fixedGradientFvPatchField (Foam)
FixedList (Foam)
FixedList::Hash (Foam)
fixedNormalSlipFvPatchField (Foam)
fixedUnburntEnthalpyFvPatchScalarField (Foam)
fixedValueFvPatchField (Foam)
fixedValueFvsPatchField (Foam)
fixedValuePointPatchField (Foam)
flux (Foam)
flux< scalar > (Foam)

meshCutAndRemove (Foam)
meshCutSurface (Foam)
meshCutter (Foam)
meshEdgeCuts (Foam)
MeshObject (Foam)
meshSearch (Foam)
meshToMesh (Foam)
meshToMesh::patchFieldInterpolator (Foam)
meshTriangulation (Foam)
MeshWave (Foam)
messageStream (Foam)
midPoint (Foam)
minEqOp (Foam)
minEqOp2 (Foam)
MinmodLimiter (Foam)
minModOp (Foam)
minModOp2 (Foam)
minModOp3 (Foam)
minOp (Foam)
minOp2 (Foam)
minOp3 (Foam)
minusEqOp (Foam)
minusEqOp2 (Foam)
minusOp (Foam)

removePoints (Foam)
repatchPolyTopoChanger (Foam)
reuseTmp (Foam)
reuseTmp< TypeR, TypeR > (Foam)
reuseTmpDimensionedField (Foam)
reuseTmpDimensionedField< TypeR, TypeR, GeoMesh > (Foam)
reuseTmpFieldField (Foam)
reuseTmpFieldField< Field, TypeR, TypeR > (Foam)
reuseTmpGeometricField (Foam)
reuseTmpGeometricField< TypeR, TypeR, PatchField, GeoMesh > (Foam)
reuseTmpTmp (Foam)
reuseTmpTmp< TypeR, Type1, Type12, TypeR > (Foam)
reuseTmpTmp< TypeR, TypeR, TypeR, Type2 > (Foam)
reuseTmpTmp< TypeR, TypeR, TypeR, TypeR > (Foam)
reuseTmpTmpDimensionedField (Foam)
reuseTmpTmpDimensionedField< TypeR, Type1, Type12, TypeR, GeoMesh > (Foam)
reuseTmpTmpDimensionedField< TypeR, TypeR, TypeR, Type2, GeoMesh > (Foam)
reuseTmpTmpDimensionedField< TypeR, TypeR, TypeR, TypeR, GeoMesh > (Foam)
reuseTmpTmpFieldField (Foam)
reuseTmpTmpFieldField< Field, TypeR, Type1, Type12, TypeR > (Foam)
reuseTmpTmpFieldField< Field, TypeR, TypeR, TypeR, Type2 > (Foam)
reuseTmpTmpFieldField< Field, TypeR, TypeR, TypeR, TypeR > (Foam)
reuseTmpTmpGeometricField (Foam)
reuseTmpTmpGeometricField< TypeR, Type1, Type12,

Done

A quick overview of OpenFOAM Doxygen doc: (4/14)


OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>

File Edit View Go Bookmarks Tools Help

http://foam.sourceforge.net/doc/Doxygen/html/functions.html Go

OpenFOAM: Th... OpenFOAM pro...

OpenFOAM The Open Source CFD Toolbox

Programmer's C++ Doxygen | OpenCFD | Solutions | Contact | OpenFOAM

Main Page | Namespaces | Classes | Files | Directories |

Search for

Alphabetical List | Class List | Class Hierarchy | Class Members |

All | Functions | Variables | Typedefs | Enumerations | Enumerator | Related Functions |

a | b | c | d | e | f | g | h | i | j | k | l | m | n | o | p | q | r | s | t | u | v | w | x | y | z | ~ |

Here is a list of all class members with links to the classes they belong to:

- a -

- a() : tetrahedron
- A() : fvMatrix
- a() : specieThermo , triangle< Point, PointRef > , tetrahedron< point, pointRef > , triangle< point, pointRef > , specieThermo< thermo >
- A() : specieThermo< thermo > , specieThermo
- a() : triangle , STLtriangle
- abort() : JobInfo , Pstream , error , IOerror
- abort_ : error
- above() : Pstream::commsStruct
- ABOVEBIT : treeBoundBox
- aC10H7CH3() : aC10H7CH3
- active() : polyPatchID , ZoneID , polyMeshModifier , refinementHistory
- adaptPatchIDs() : motionSmoother
- add() : dictionary , multivariateSurfaceInterpolationScheme::fieldTable , dictionary
- ADD : topoSetSource
- add() : dictionary , fvMeshAdder , polyMeshAdder
- ADD : token
- add() : polyMeshAdder , IOobjectList
- addCell() : polyTopoChange
- addCoriolis() : MRFZone , MRFZones
- addedCellMap() : mapAddedPolyMesh
- addedCells() : multiDirRefinement , meshCutter , addPatchCellLayer
- addedCellsPtr_ : refinementHistory::splitCell8
- addedFaceMap() : mapAddedPolyMesh
- addedFaces() : meshCutAndRemove , meshCutter
- addedPatches() : mapPatchChange
- addedPatchMap() : mapAddedPolyMesh
- addedPointMap() : mapAddedPolyMesh
- addedPoints() : meshCutAndRemove , meshCutter , addPatchCellLayer
- addFace() : polyTopoChange
- addFvPatches() : fvMesh
- addIntersectionEdges() : edgeSurface

Done

A quick overview of OpenFOAM Doxygen doc: (5/14)

OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>

File Edit View Go Bookmarks Tools Help

http://foam.sourceforge.net/doc/Doxygen/html/files.html

OpenFOAM: Th... OpenFOAM pro...

Open∇FOAM *The Open Source CFD Toolbox*

Programmer's C++ Doxygen | **OpenCFD** | **Solutions** | **Contact** | **OpenFOAM**

Main Page | Namespaces | Classes | Files | Directories |

Search for

File List | File Members |

OpenFOAM-1.4.1 File List

Here is a list of all files with brief descriptions:

src/dynamicFvMesh/dynamicFvMesh/dynamicFvMesh.H [code]
src/dynamicFvMesh/dynamicInkJetFvMesh/dynamicInkJetFvMesh.H [code]
src/dynamicFvMesh/dynamicMotionSolverFvMesh/dynamicMotionSolverFvMesh.H [code]
src/dynamicFvMesh/include/createDynamicFvMesh.H [code]
src/dynamicFvMesh/staticFvMesh/staticFvMesh.H [code]
src/dynamicMesh/attachDetach/attachDetach.H [code]
src/dynamicMesh/boundaryMesh/bMesh.H [code]
src/dynamicMesh/boundaryMesh/boundaryMesh.H [code]
src/dynamicMesh/boundaryMesh/octreeDataFaceList.H [code]
src/dynamicMesh/boundaryPatch/boundaryPatch.H [code]
src/dynamicMesh/fvMeshAdder/fvMeshAdder.H [code]
src/dynamicMesh/fvMeshDistribute/fvMeshDistribute.H [code]
src/dynamicMesh/layerAdditionRemoval/layerAdditionRemoval.H [code]
src/dynamicMesh/meshCut/cellCuts/cellCuts.H [code]
src/dynamicMesh/meshCut/cellLooper/cellLooper.H [code]
src/dynamicMesh/meshCut/cellLooper/geomCellLooper.H [code]
src/dynamicMesh/meshCut/cellLooper/hexCellLooper.H [code]
src/dynamicMesh/meshCut/cellLooper/topoCellLooper.H [code]
src/dynamicMesh/meshCut/directions/directions.H [code]
src/dynamicMesh/meshCut/directions/directionInfo/directionInfo.H [code]
src/dynamicMesh/meshCut/directions/directionInfo/directionInfoI.H [code]
src/dynamicMesh/meshCut/edgeVertex/edgeVertex.H [code]
src/dynamicMesh/meshCut/meshModifiers/boundaryCutter/boundaryCutter.H [code]
src/dynamicMesh/meshCut/meshModifiers/meshCutAndRemove/meshCutAndRemove.H [code]
src/dynamicMesh/meshCut/meshModifiers/meshCutter/meshCutter.H [code]
src/dynamicMesh/meshCut/meshModifiers/multiDirRefinement/multiDirRefinement.H [code]
src/dynamicMesh/meshCut/meshModifiers/refinementIterator/refinementIterator.H [code]
src/dynamicMesh/meshCut/meshModifiers/undoableMeshCutter/undoableMeshCutter.H [code]
src/dynamicMesh/meshCut/refineCell/refineCell.H [code]
src/dynamicMesh/meshCut/splitCell/splitCell.H [code]
src/dynamicMesh/meshCut/wallLayerCells/wallLayerCells.H [code]
src/dynamicMesh/meshCut/wallLayerCells/wallNormalInfo/wallNormalInfo.H [code]

Done

A quick overview of OpenFOAM Doxygen doc: (6/14)

OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>

File Edit View Go Bookmarks Tools Help

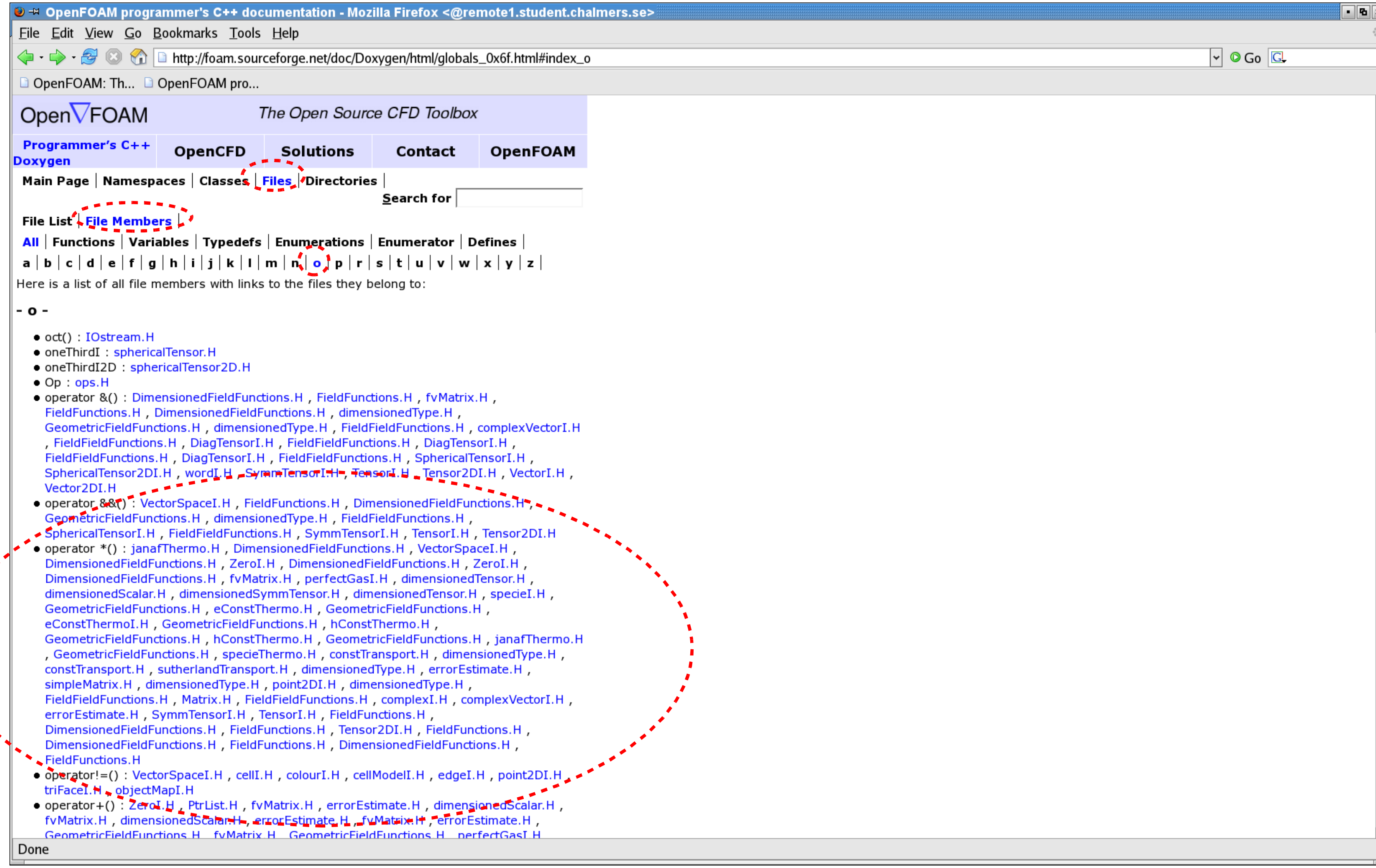
http://foam.sourceforge.net/doc/Doxygen/html/attachDetach_8H-source.html

OpenFOAM: Th... OpenFOAM pro...

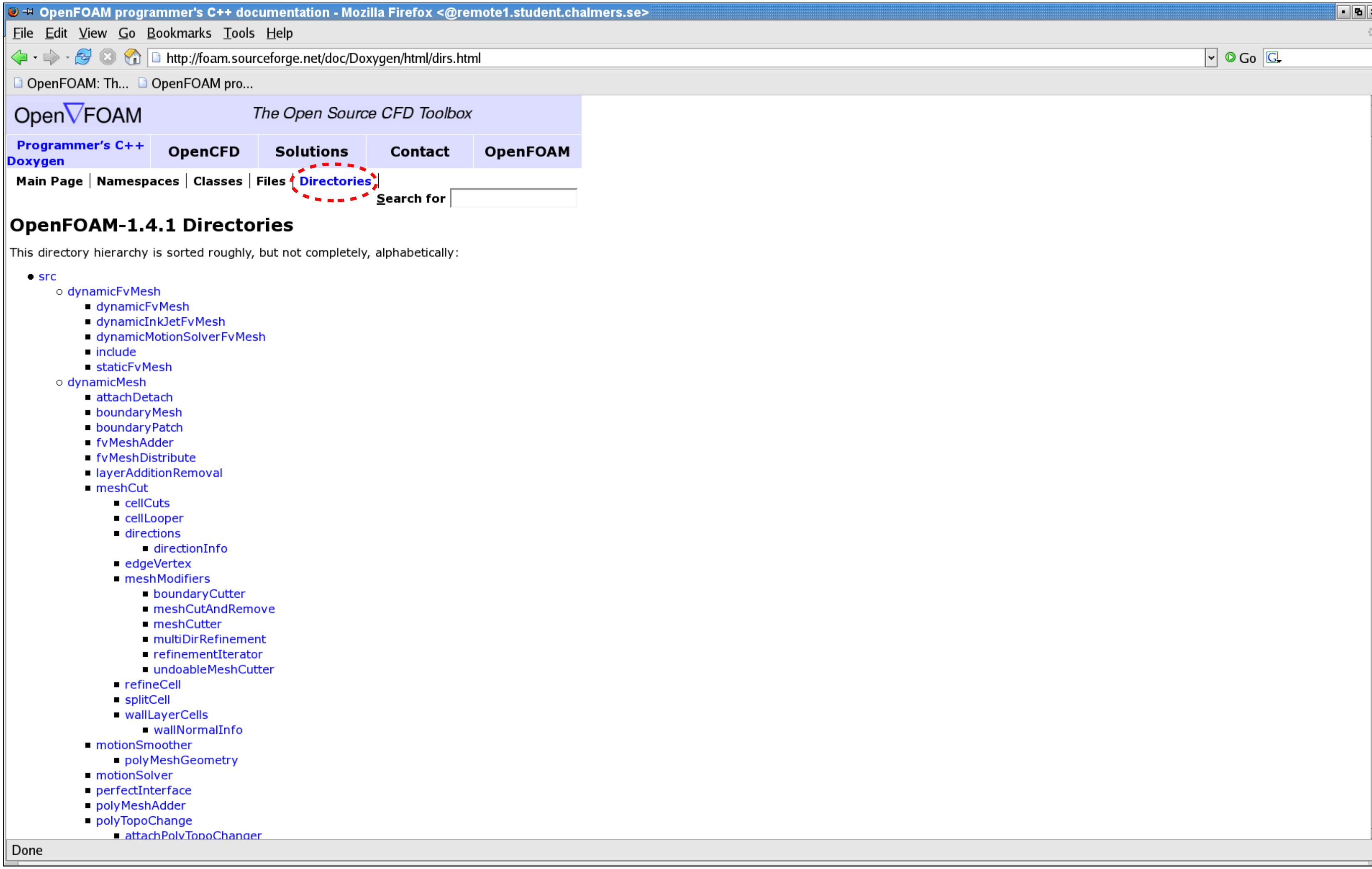
```

#include "polyMeshModifier.H"
#include "polyPatchID.H"
#include "ZoneIDs.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

namespace Foam
{

/*---------------------------------------------------------------------------*\
                       Class attachDetach Declaration
\*---------------------------------------------------------------------------*/

class attachDetach
:
    public polyMeshModifier
{
    // Data types

        //- State of the modifier
        enum modifierState
        {
            UNKNOWN,
            ATTACHED,
            DETACHED
        };


    // Private data

        //- Master face zone ID
        faceZoneID faceZoneID_;

        //- Master patch ID.  Holds faces with original orientation
        polyPatchID masterPatchID_;

        //- Slave patch ID.  Holds mirrored faces
        polyPatchID slavePatchID_;

        //- List of trigger times
        scalarField triggerTimes_;

        //- Use manual trigger
        Switch manualTrigger_;

        //- Trigger time index
        mutable label triggerIndex_;

        //- State of the modifier
        mutable modifierState state_;

        //- Attach/detach trigger
        mutable bool trigger_;
```

Done

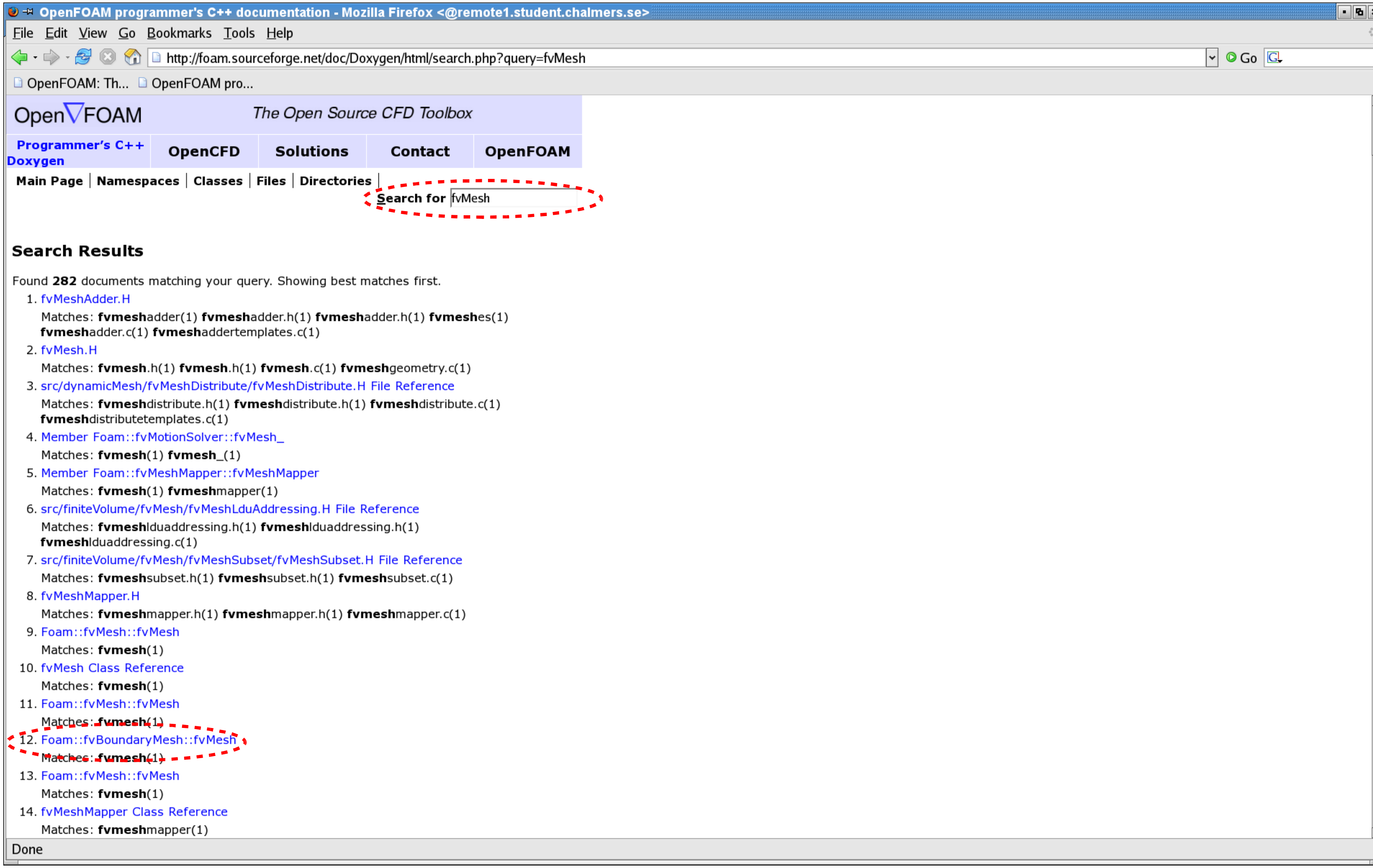
A quick overview of OpenFOAM Doxygen doc: (7/14)



A quick overview of OpenFOAM Doxygen doc: (8/14)



A quick overview of OpenFOAM Doxygen doc: (9/14)

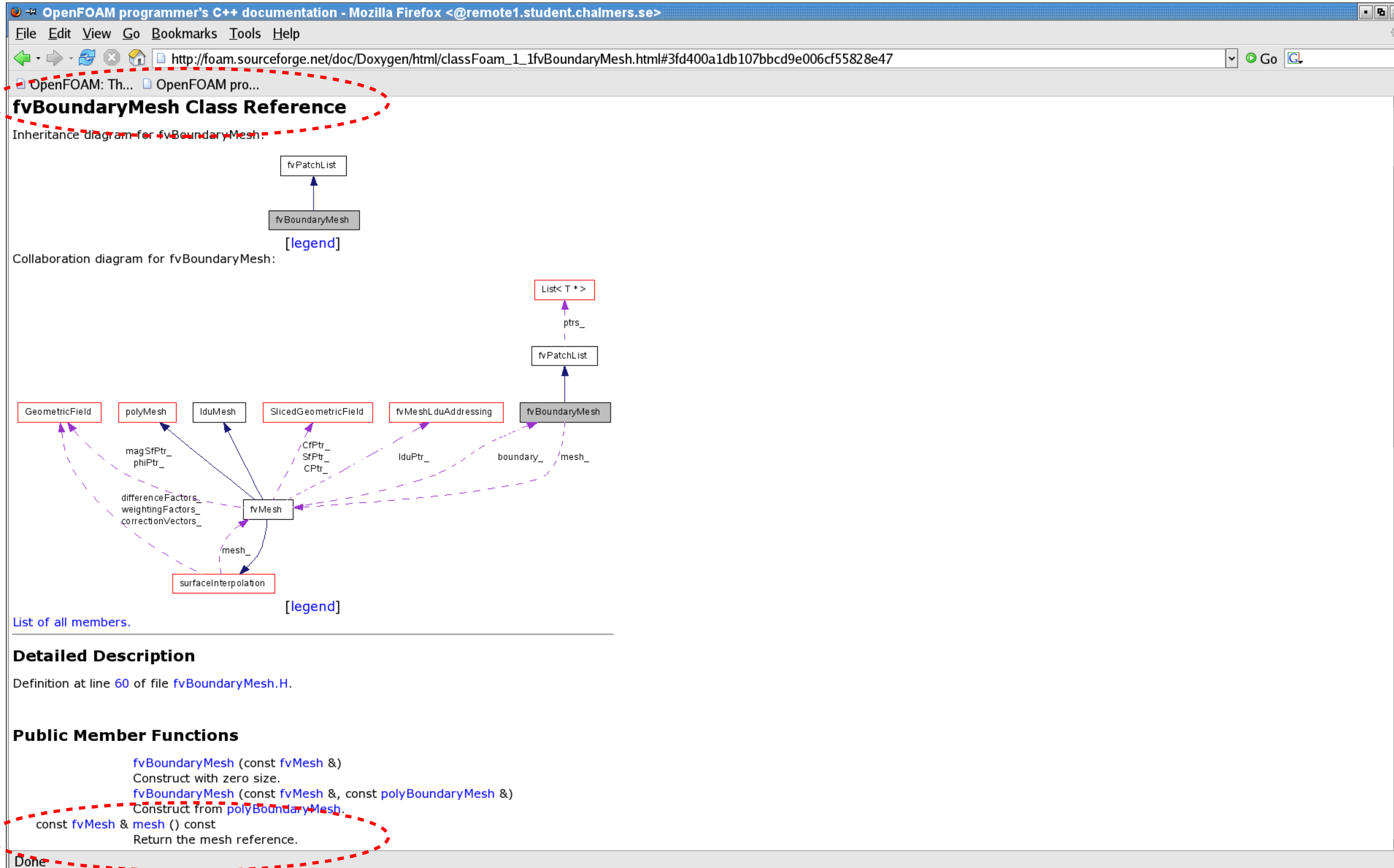


Hydro Québec
Institut de recherche

Chalmers OpenFOAM course: Guest presentation

March 28th 2008

A quick overview of OpenFOAM Doxygen doc: (10/14)



Hydro Québec
Institut de recherche

A quick overview of OpenFOAM Doxygen doc: (11/14)

OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>

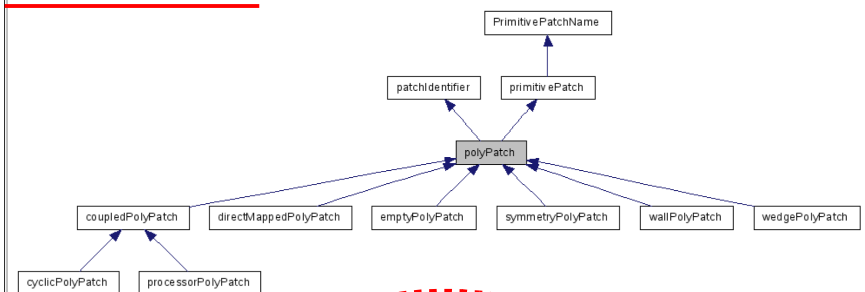
File Edit View Go Bookmarks Tools Help

http://foam.sourceforge.net/doc/Doxygen/html/classFoam_1_1polyPatch.html

OpenFOAM: Th... OpenFOAM pro...

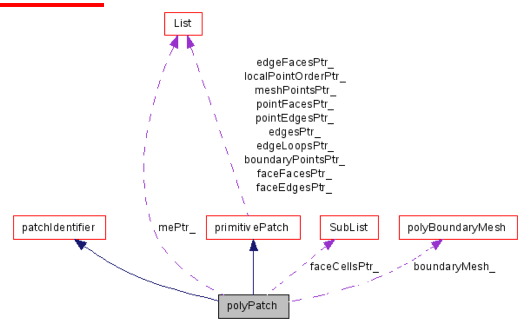
polyPatch Class Reference

Inheritance diagram for polyPatch:



[legend]

Collaboration diagram for polyPatch:



[legend]

List of all members.

Detailed Description

Definition at line 75 of file polyPatch.H.

Done

A quick overview of OpenFOAM Doxygen doc: (12/14)

OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>

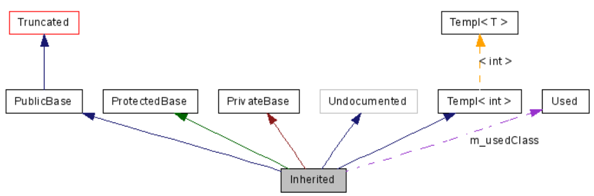
File Edit View Go Bookmarks Tools Help

http://foam.sourceforge.net/doc/Doxygen/html/graph_legend.html

OpenFOAM: Th... OpenFOAM pro...

```
/*! Class that is inherited using private inheritance */
class PrivateBase { };

/*! Class that is used by the Inherited class */
class Used { };

/*! Super class that inherits a number of other classes */
class Inherited : public PublicBase,
                  protected ProtectedBase,
                  private PrivateBase,
                  public Undocumented
                  public Templ<int>
{
  private:
    Used *m_usedClass;
};
```

If the `MAX_DOT_GRAPH_HEIGHT` tag in the configuration file is set to 240 this will result in the following graph:



The boxes in the above graph have the following meaning:

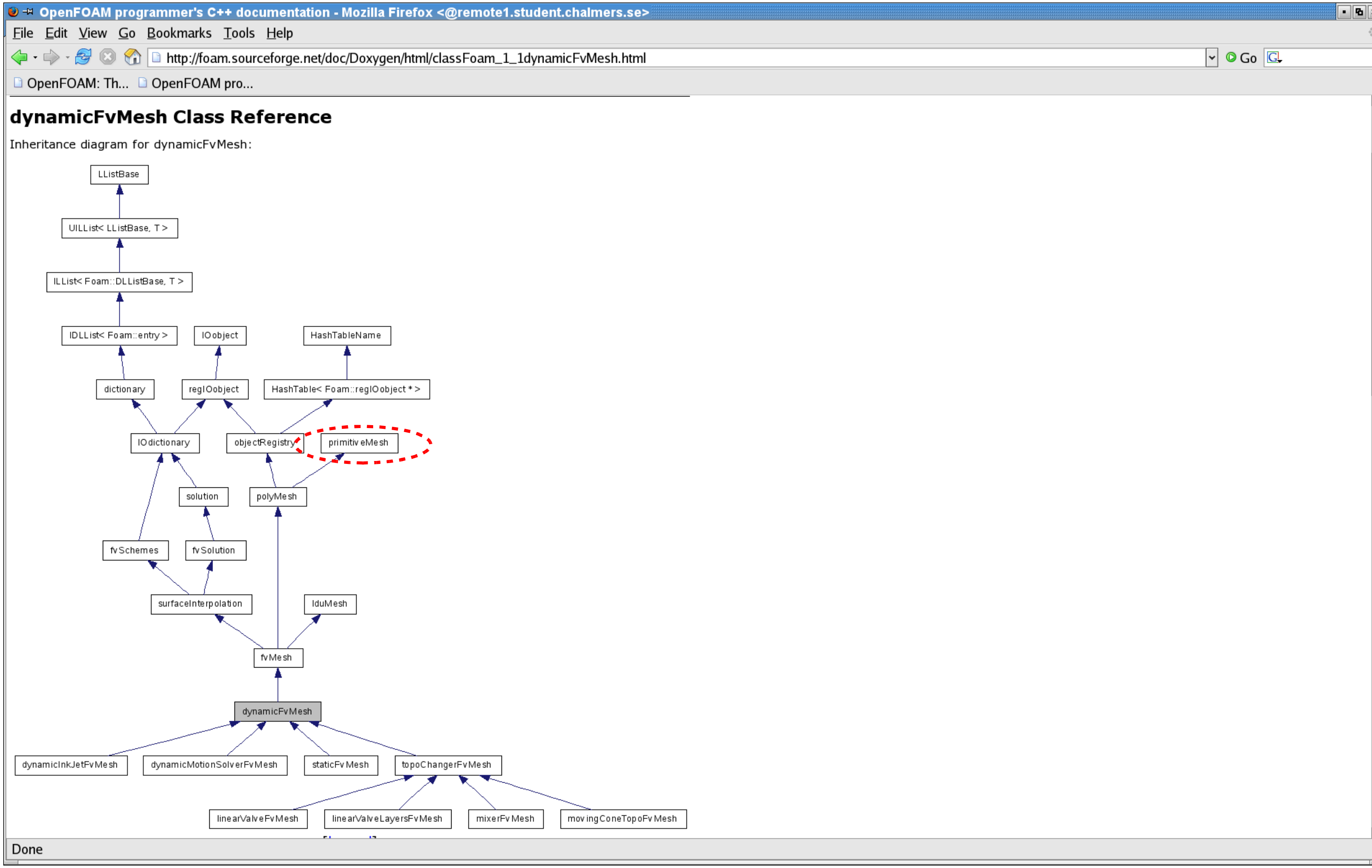
- A filled black box represents the struct or class for which the graph is generated.
- A box with a black border denotes a documented struct or class.
- A box with a grey border denotes an undocumented struct or class.
- A box with a red border denotes a documented struct or class forwhich not all inheritance/containment relations are shown. A graph is truncated if it does not fit within the specified boundaries.

The arrows have the following meaning:

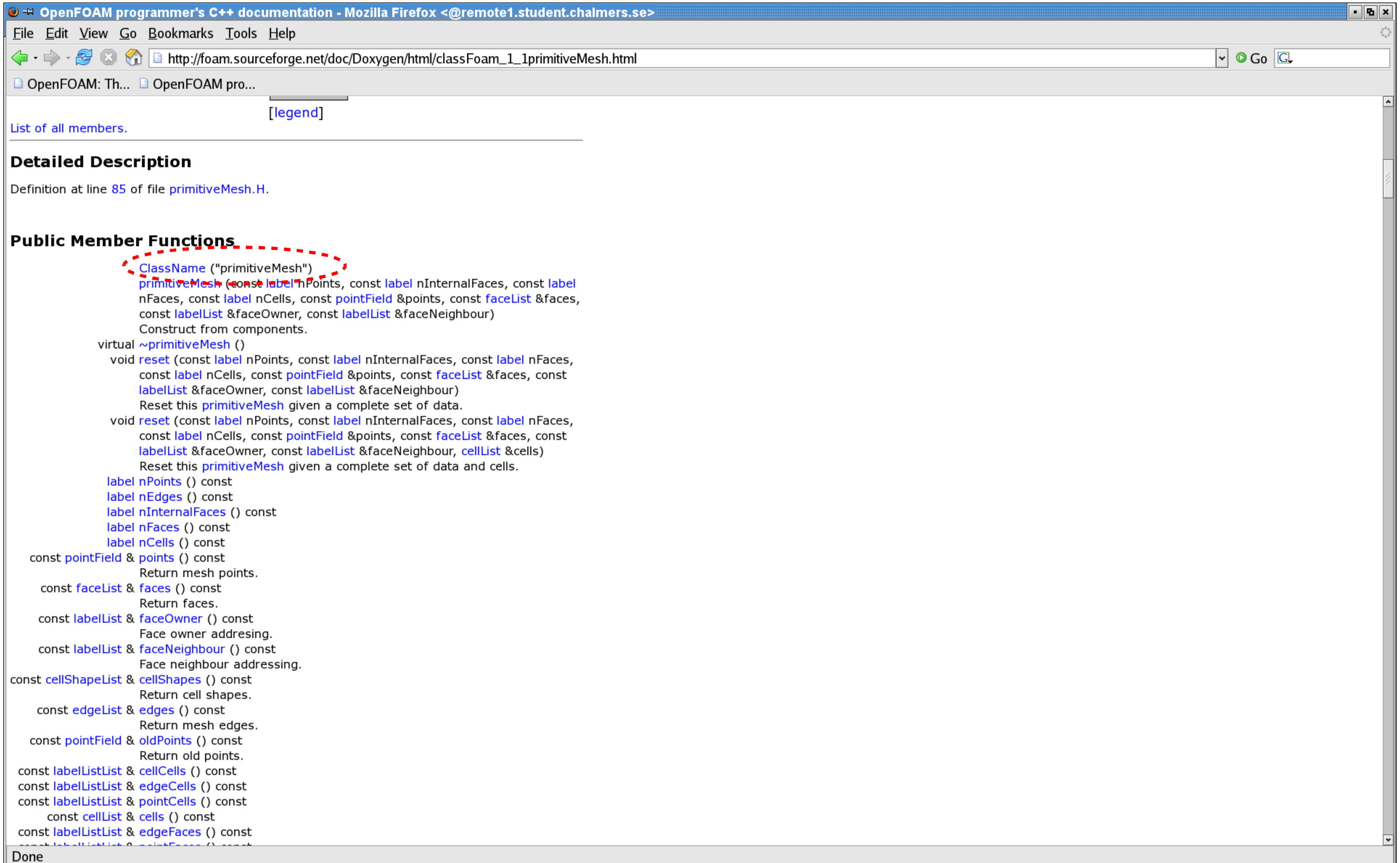
- A dark blue arrow is used to visualize a public inheritance relation between two classes.
- A dark green arrow is used for protected inheritance.
- A dark red arrow is used for private inheritance.
- A purple dashed arrow is used if a class is contained or used by another class. The arrow is labeled with the variable(s) through which the pointed class or struct is accessible.
- A yellow dashed arrow denotes a relation between a template instance and the template class it was instantiated from. The arrow is labeled with the template parameters of the instance.

Done

A quick overview of OpenFOAM Doxygen doc: (13/14)



A quick overview of OpenFOAM Doxygen doc: (14/14)



OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>

File Edit View Go Bookmarks Tools Help

http://foam.sourceforge.net/doc/Doxygen/html/classFoam_1_1primitiveMesh.html

OpenFOAM: Th... OpenFOAM pro...

[legend]

List of all members.

Detailed Description

Definition at line 85 of file primitiveMesh.H.

Public Member Functions

```
                                ClassName ("primitiveMesh")
                                primitiveMesh (const label nPoints, const label nInternalFaces, const label
                                nFaces, const label nCells, const pointField &points, const faceList &faces,
                                const labelList &faceOwner, const labelList &faceNeighbour)
                                Construct from components.
                        virtual ~primitiveMesh ()
                           void reset (const label nPoints, const label nInternalFaces, const label nFaces,
                                const label nCells, const pointField &points, const faceList &faces, const
                                labelList &faceOwner, const labelList &faceNeighbour)
                                Reset this primitiveMesh given a complete set of data.
                           void reset (const label nPoints, const label nInternalFaces, const label nFaces,
                                const label nCells, const pointField &points, const faceList &faces, const
                                labelList &faceOwner, const labelList &faceNeighbour, cellList &cells)
                                Reset this primitiveMesh given a complete set of data and cells.
                          label nPoints () const
                          label nEdges () const
                          label nInternalFaces () const
                          label nFaces () const
                          label nCells () const
             const pointField & points () const
                                Return mesh points.
               const faceList & faces () const
                                Return faces.
              const labelList & faceOwner () const
                                Face owner addresing.
              const labelList & faceNeighbour () const
                                Face neighbour addressing.
          const cellShapeList & cellShapes () const
                                Return cell shapes.
               const edgeList & edges () const
                                Return mesh edges.
             const pointField & oldPoints () const
                                Return old points.
          const labelListList & cellCells () const
          const labelListList & edgeCells () const
          const labelListList & pointCells () const
               const cellList & cells () const
          const labelListList & edgeFaces () const
```

Done

Hydro Québec
Institut de recherche

Modifying the generated documentation:

- It is possible to customize the Doxygen documentation:
  - Changing the default settings chosen by OpenCFD
  - Adding new information (side panel, etc.)
  - Adding your own C++ classes

- These are the basic tools you need:
  - Files from $WM_PROJECT_DIR/doc/Doxygen:
    - Allwmake
    - Doxyfile (for doxygen 1.5.1)
    - Doxyfile.1.3.5 (for doxygen 1.3.5)
    - FoamHeader.html, FoamFooter.html, Doxygen.css
  - Unix commands:
    - doxygen, doxywizard, dot
  - Enough disk space:
    - ~300 MB

Preparing our new documentation sandbox

remote3.student.chalmers.se - Konsole

Session Edit View Bookmarks Settings Help

```
[beaudoin@remote3 ~]$ mkdir -p ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1
[beaudoin@remote3 ~]$ cd ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ cp /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/doc/Doxygen/Allwmake .
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ cp /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/doc/Doxygen/*.html .
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ cp /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/doc/Doxygen/Doxyfile* .
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ cp /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/doc/Doxygen/Doxygen.css .
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ ls -axl
total 152
drwxrwxr-x  2 beaudoin beaudoin  8192 Mar 26 12:31 .
drwxr-xr-x  4 beaudoin beaudoin  8192 Mar 26 12:30 ..
-rwxr-xr-x  1 beaudoin beaudoin   142 Mar 26 12:30 Allwmake
-rw-r--r--  1 beaudoin beaudoin 54365 Mar 26 12:31 Doxyfile
-rw-r--r--  1 beaudoin beaudoin 47900 Mar 26 12:31 Doxyfile.1.3.5
-rw-r--r--  1 beaudoin beaudoin  1095 Mar 26 12:31 Doxygen.css
-rw-r--r--  1 beaudoin beaudoin    95 Mar 26 12:30 FoamFooter.html
-rw-r--r--  1 beaudoin beaudoin  2719 Mar 26 12:30 FoamHeader.html
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ df -H .
Filesystem             Size   Used  Avail Use% Mounted on
lambda-c2.mtek.chalmers.se:/lfs/c2.am/beaudoin
                       634G   132G   501G  21% /chalmers/users/beaudoin
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$
```

remote3.student.chalmers.se

Adapting for the available version of doxygen

remote3.student.chalmers.se - Konsole

Session Edit View Bookmarks Settings Help

```
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ head -2 Doxyfile
# Doxyfile 1.5.1

[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ cp Doxyfile Doxyfile.1.5.1
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ doxygen --help
Doxygen version 1.3.9.1
Copyright Dimitri van Heesch 1997-2004

You can use doxygen in a number of ways:

1) Use doxygen to generate a template configuration file:
    doxygen [-s] -g [configName]

    If - is used for configName doxygen will write to standard output.

2) Use doxygen to update an old configuration file:
    doxygen [-s] -u [configName]

3) Use doxygen to generate documentation using an existing configuration file:
    doxygen [configName]

    If - is used for configName doxygen will read from standard input.

4) Use doxygen to generate a template style sheet file for RTF, HTML or Latex.
    RTF:   doxygen -w rtf styleSheetFile
    HTML:  doxygen -w html headerFile footerFile styleSheetFile [configFile]
    LaTeX: doxygen -w latex headerFile styleSheetFile [configFile]

5) Use doxygen to generate an rtf extensions file
    RTF:   doxygen -e rtf extensionsFile

If -s is specified the comments in the config file will be omitted.
If configName is omitted `Doxyfile' will be used as a default.

[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ cp Doxyfile.1.3.5 Doxyfile
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ doxygen -u


Configuration file `Doxyfile' updated.

[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ ls -axl
total 256
drwxrwxr-x  2 beaudoin beaudoin  8192 Mar 26 12:41 .
drwxr-xr-x  4 beaudoin beaudoin  8192 Mar 26 12:38 ..
-rwxr-xr-x  1 beaudoin beaudoin   142 Mar 26 12:38 Allwmake
-rw-rw-r--  1 beaudoin beaudoin 50106 Mar 26 12:41 Doxyfile
-rw-r--r--  1 beaudoin beaudoin 47900 Mar 26 12:38 Doxyfile.1.3.5
-rw-r--r--  1 beaudoin beaudoin 54365 Mar 26 12:40 Doxyfile.1.5.1
-rw-r--r--  1 beaudoin beaudoin 47900 Mar 26 12:41 Doxyfile.bak
-rw-r--r--  1 beaudoin beaudoin  1095 Mar 26 12:38 Doxygen.css
-rw-r--r--  1 beaudoin beaudoin    95 Mar 26 12:38 FoamFooter.html
-rw-r--r--  1 beaudoin beaudoin  2719 Mar 26 12:38 FoamHeader.html
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ head -2 Doxyfile
# Doxyfile 1.3.9.1

[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$
```

remote3.student.chalmers.se

Modifying Doxyfile : the hard way

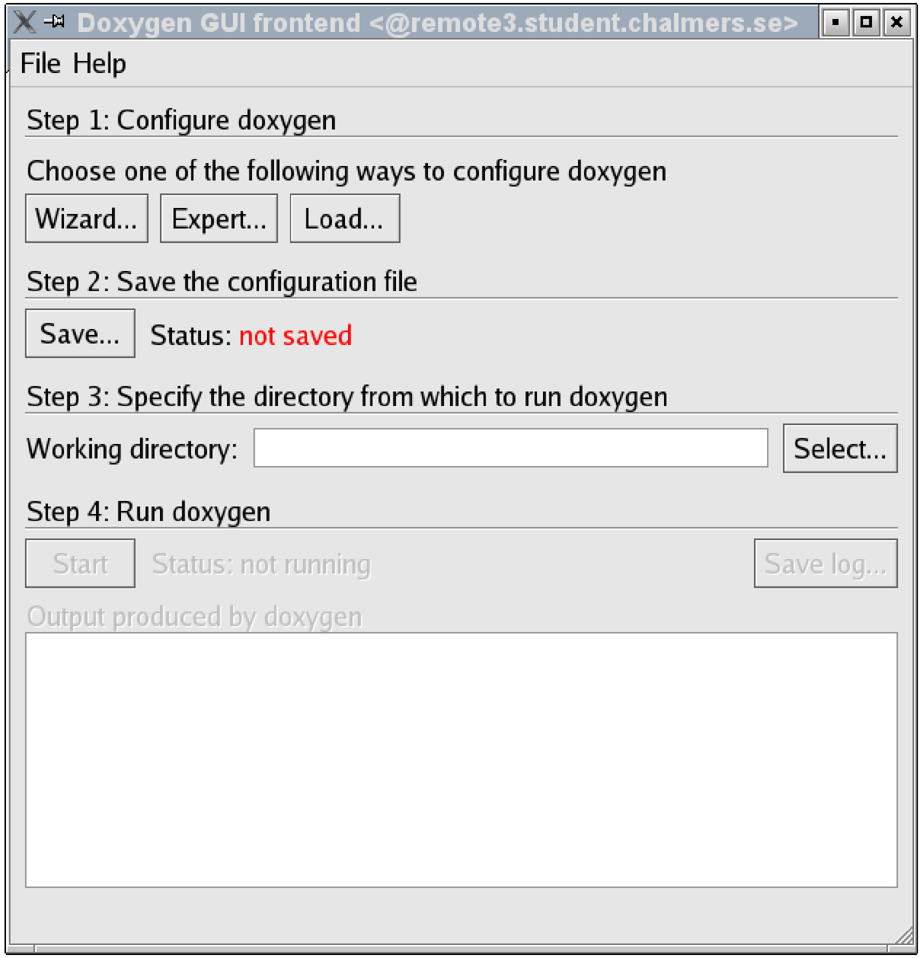
remote3.student.chalmers.se - Konsole

Session Edit View Bookmarks Settings Help

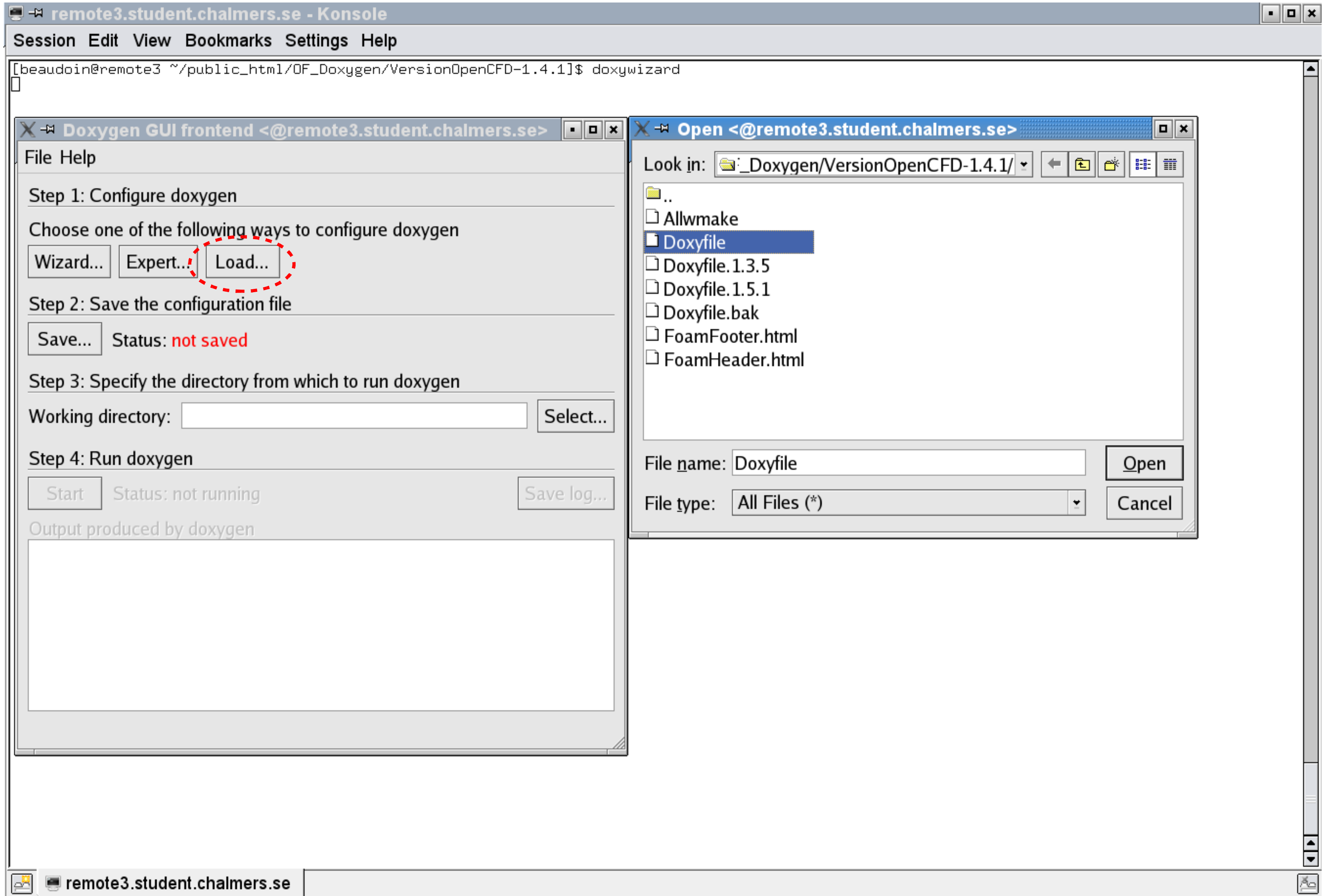
```
# Doxyfile 1.3.9.1

# This file describes the settings to be used by the documentation system
# doxygen (www.doxygen.org) for a project
#
# All text after a hash (#) is considered a comment and will be ignored
# The format is:
#       TAG = value [value, ...]
# For lists items can also be appended using:
#       TAG += value [value, ...]
# Values that contain spaces should be placed between quotes (" ")

#---------------------------------------------------------------------------
# Project related configuration options
#---------------------------------------------------------------------------

# The PROJECT_NAME tag is a single word (or a sequence of words surrounded
# by quotes) that should identify the project.

PROJECT_NAME           = OpenFOAM-$(WM_PROJECT_VERSION)

# The PROJECT_NUMBER tag can be used to enter a project or revision number.
# This could be handy for archiving the generated documentation or
# if some version control system is used.

PROJECT_NUMBER         =

# The OUTPUT_DIRECTORY tag is used to specify the (relative or absolute)
# base path where the generated documentation will be put.
# If a relative path is entered, it will be relative to the location
# where doxygen was started. If left blank the current directory will be used.

OUTPUT_DIRECTORY       =

# If the CREATE_SUBDIRS tag is set to YES, then doxygen will create
# 4096 sub-directories (in 2 levels) under the output directory of each output
# format and will distribute the generated files over these directories.
# Enabling this option can be useful when feeding doxygen a huge amount of source
# files, where putting all generated files in the same directory would otherwise
# cause performance problems for the file system.

CREATE_SUBDIRS         = YES

# The OUTPUT_LANGUAGE tag is used to specify the language in which all
# documentation generated by doxygen is written. Doxygen will use this
# information to generate all constant output in the proper language.
# The default language is English, other supported languages are:
# Brazilian, Catalan, Chinese, Chinese-Traditional, Croatian, Czech, Danish,
# Dutch, Finnish, French, German, Greek, Hungarian, Italian, Japanese,
# Japanese-en (Japanese with English messages), Korean, Korean-en, Norwegian,
# Polish, Portuguese, Romanian, Russian, Serbian, Slovak, Slovene, Spanish,
# Swedish, and Ukrainian.

OUTPUT_LANGUAGE        = English
"Doxyfile" 1195L, 50106C                                        1,1           Top
```

remote3.student.chalmers.se

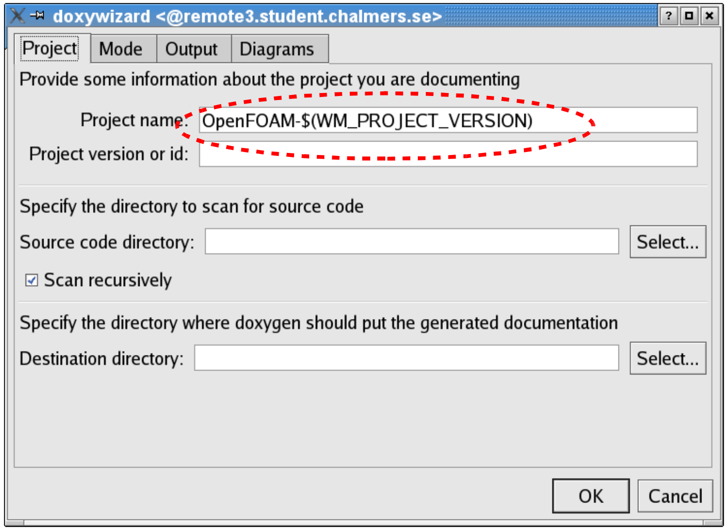
Modifying Doxyfile : using doxywizard

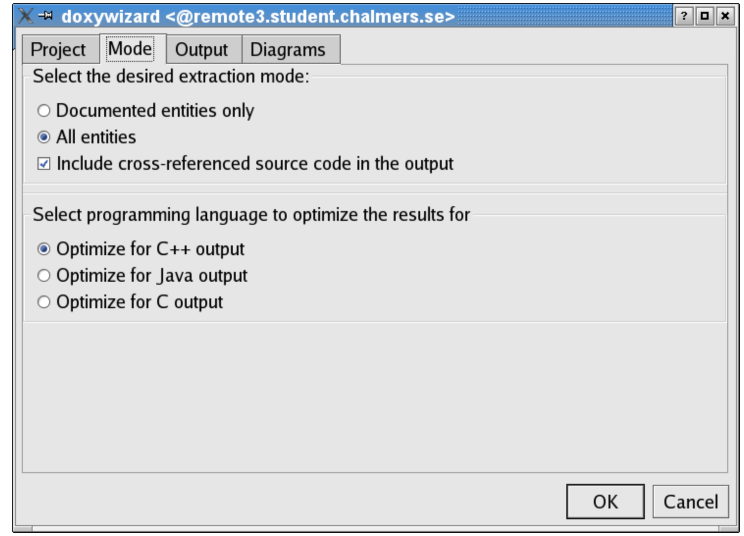


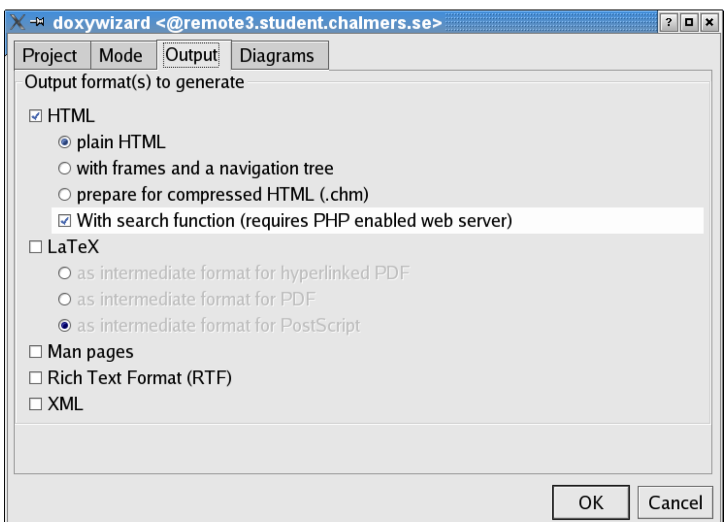
Doxywizard : loading a configuration file

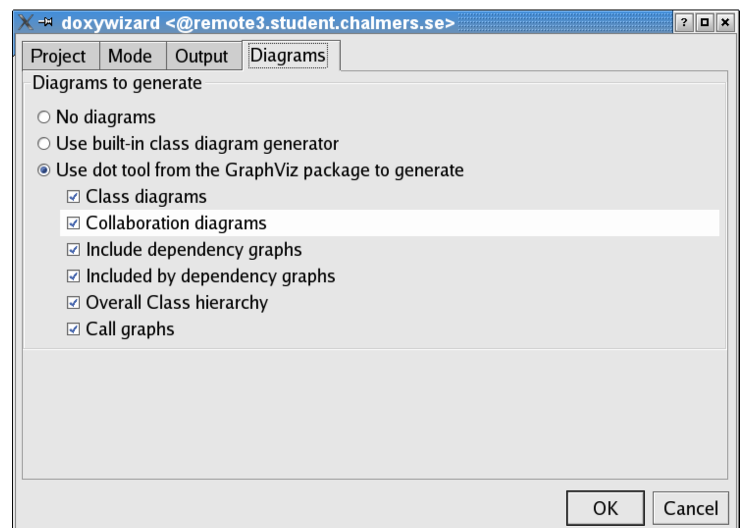


Doxywizard : Wizard mode

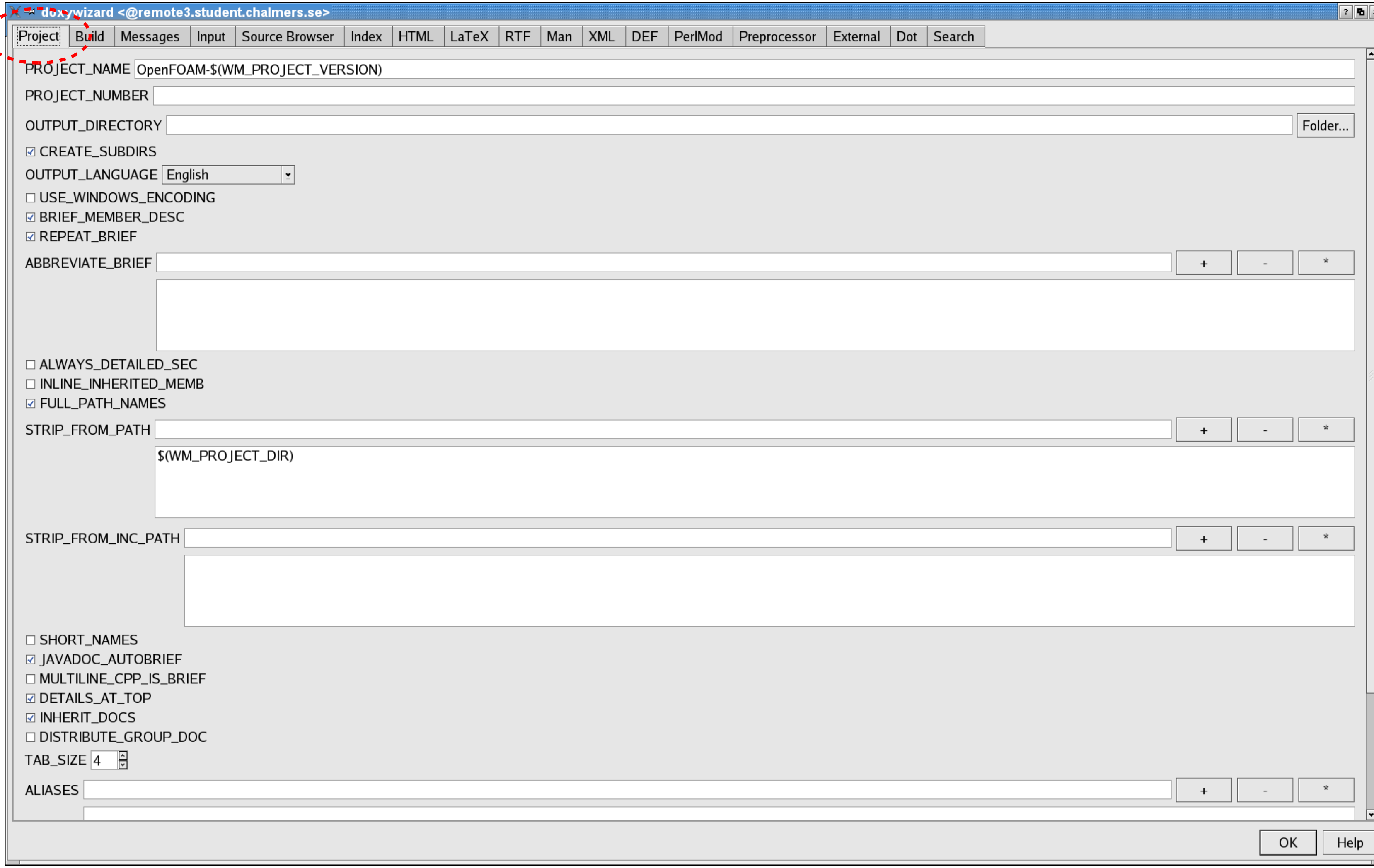




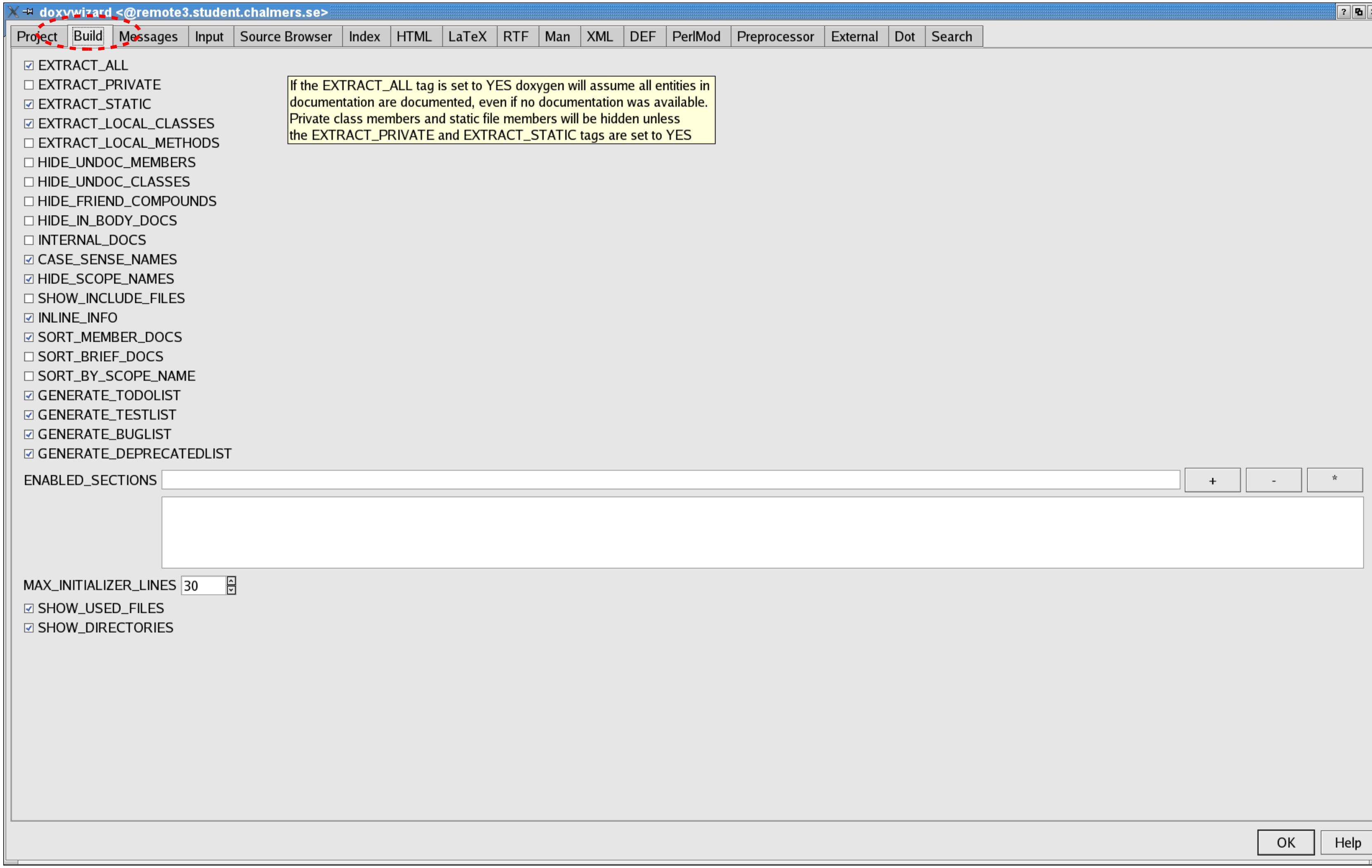




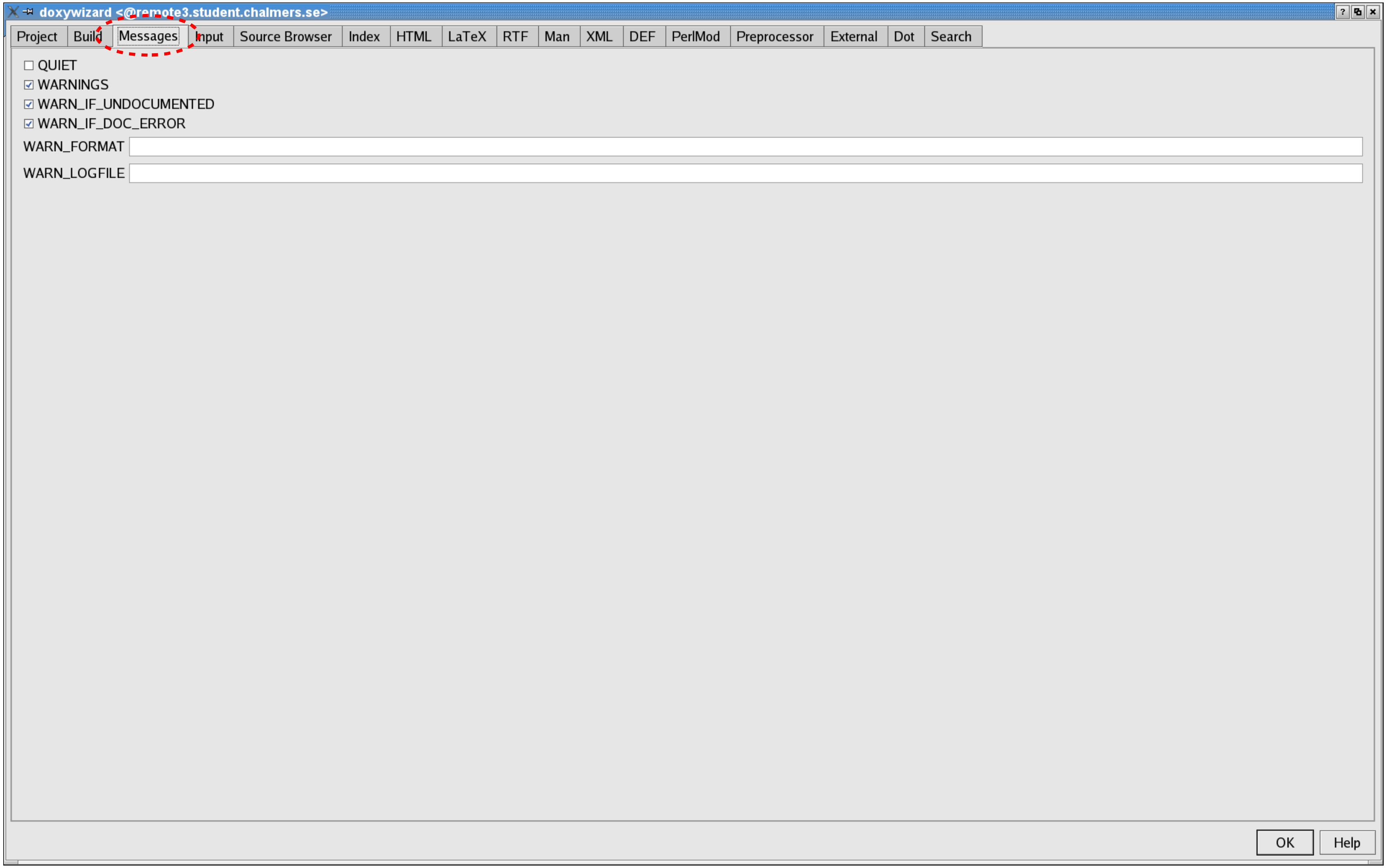
Doxywizard : Expert mode (1/9)



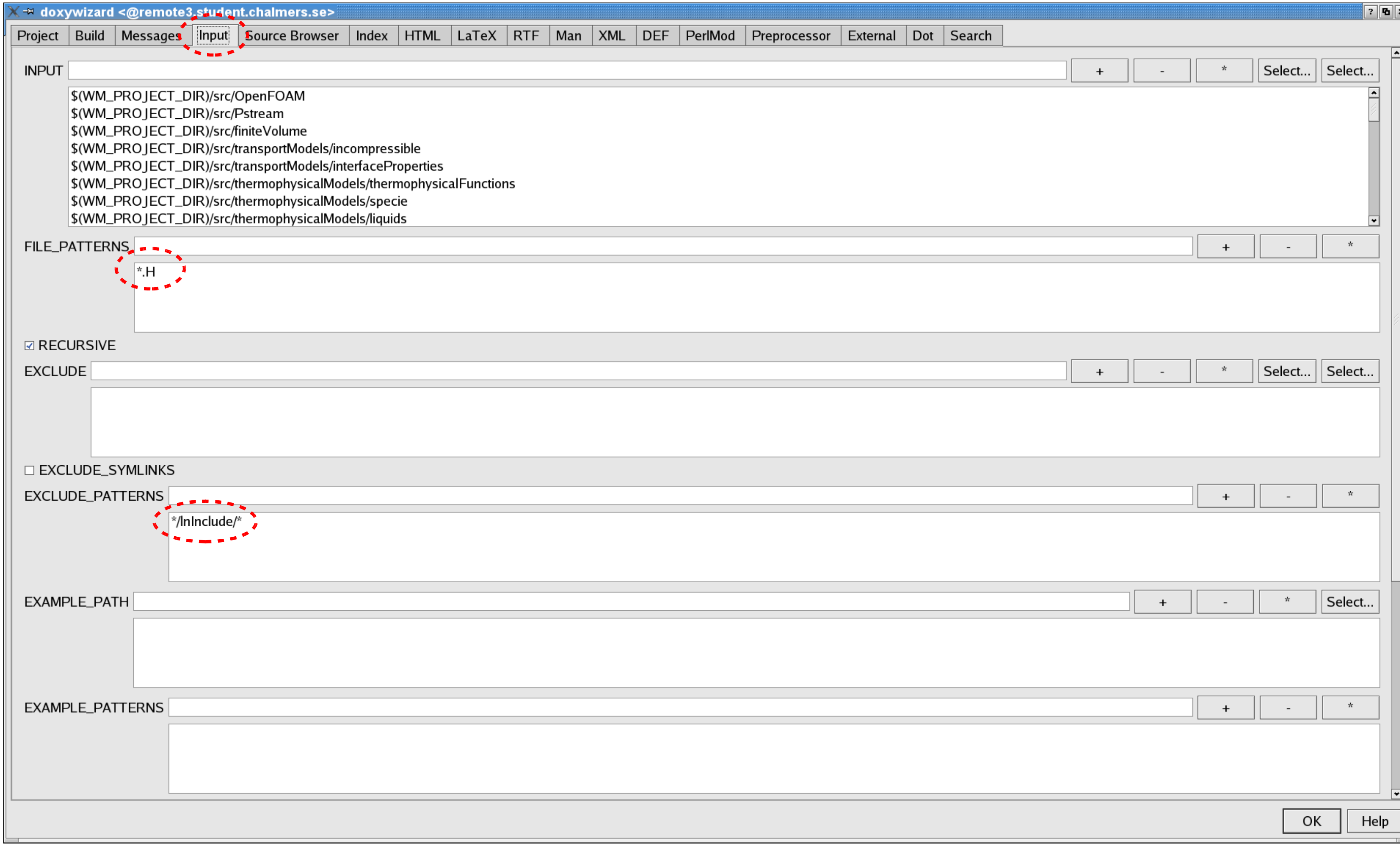
Doxywizard : Expert mode (2/9)



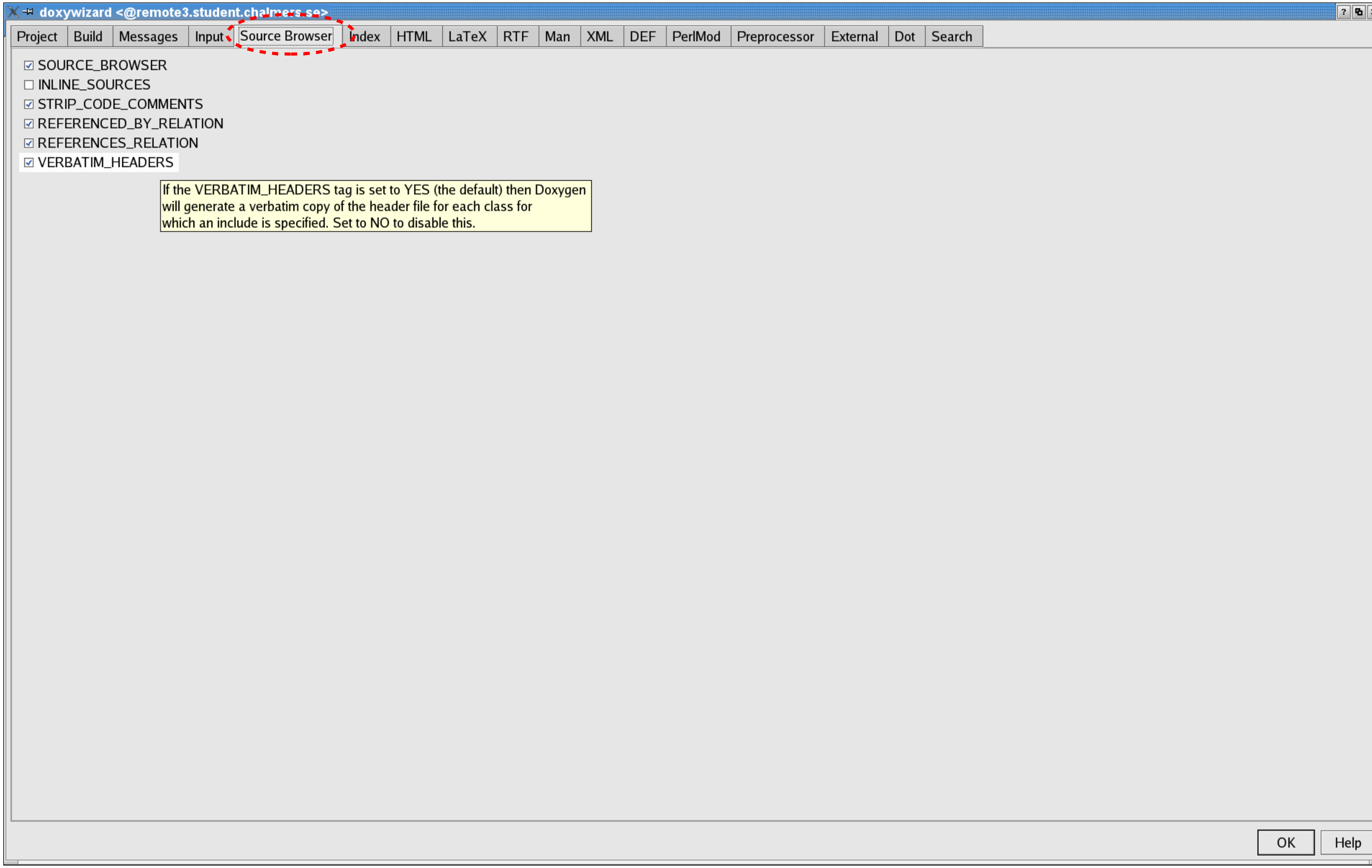
Doxywizard : Expert mode (3/9)



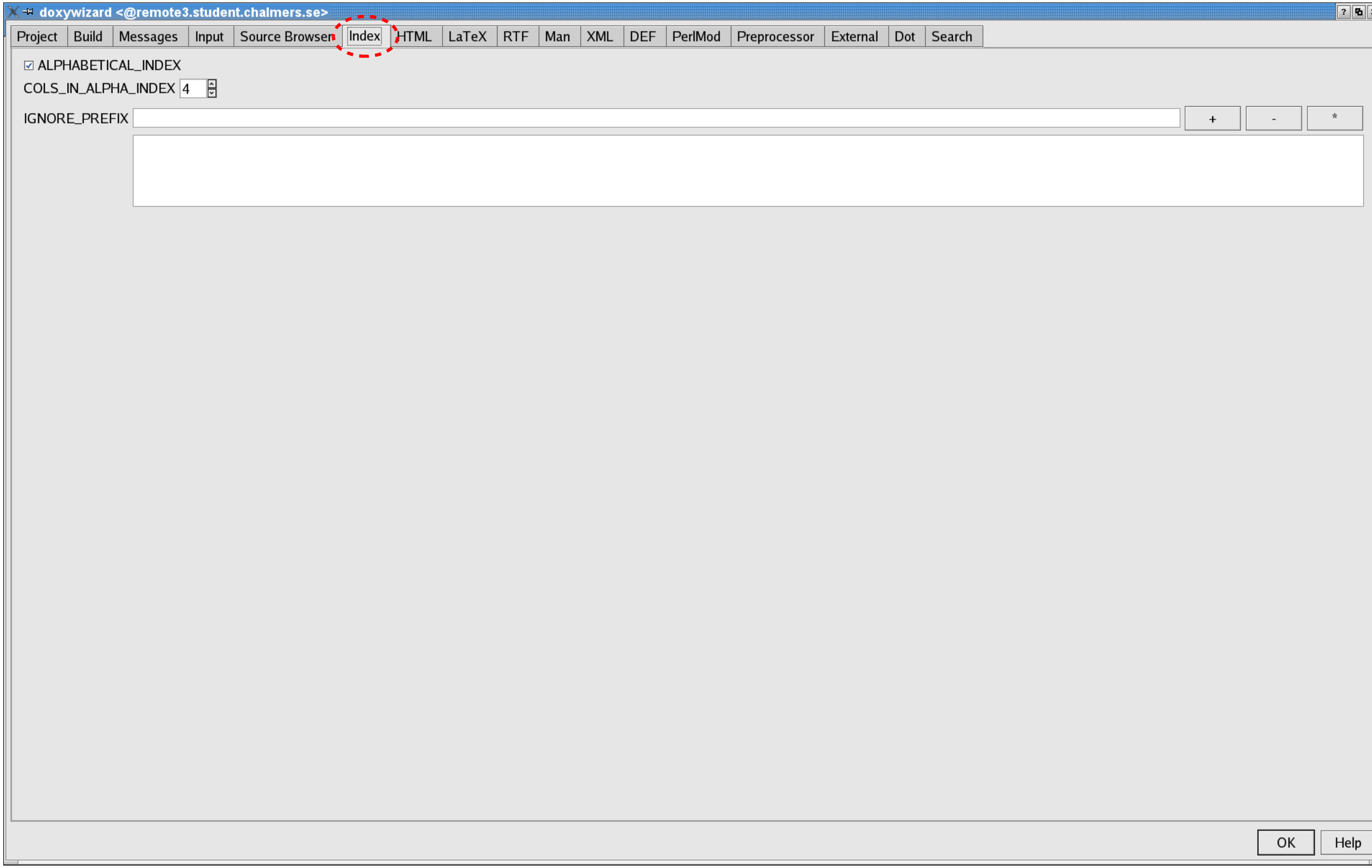
Doxywizard : Expert mode (4/9)



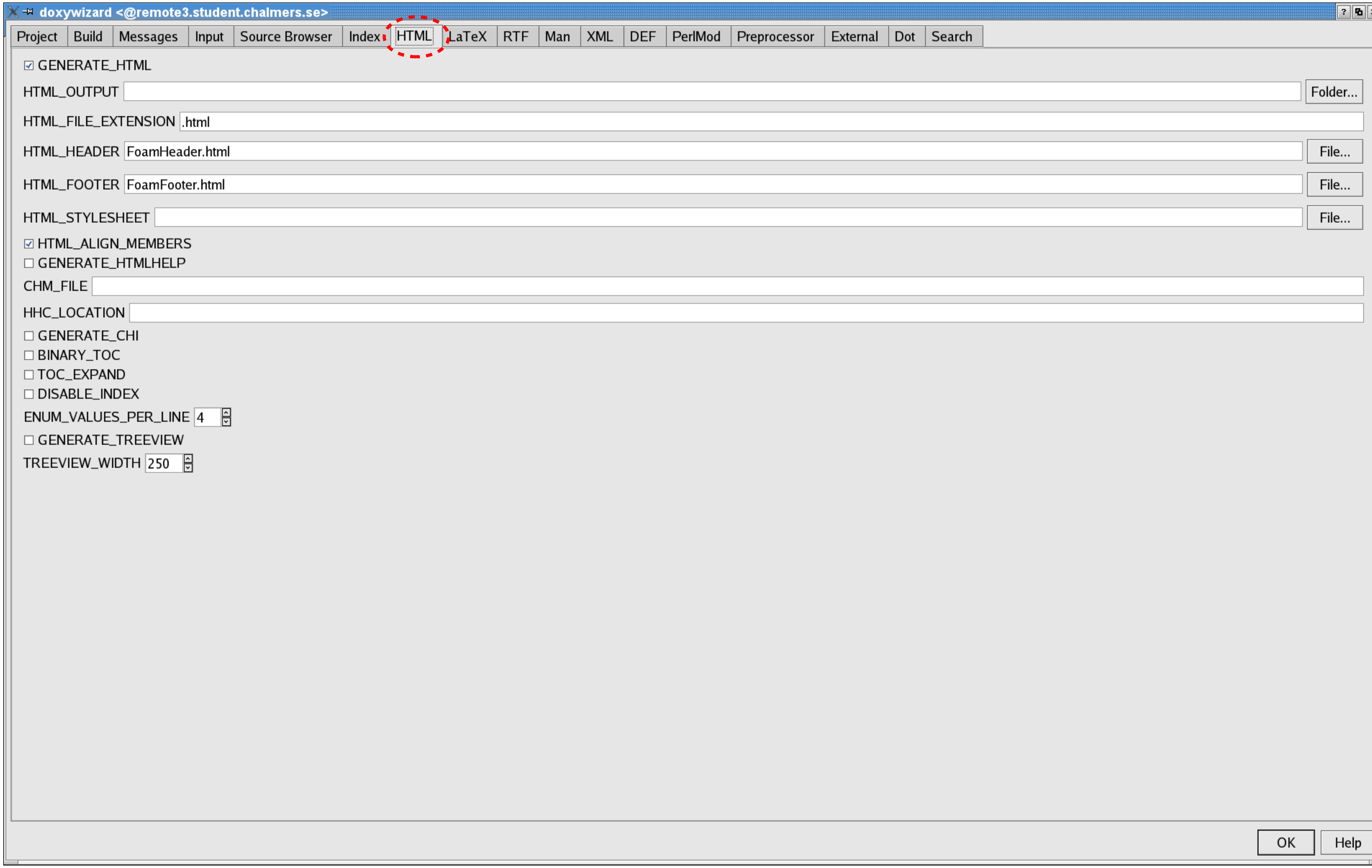
Doxywizard : Expert mode (5/9)



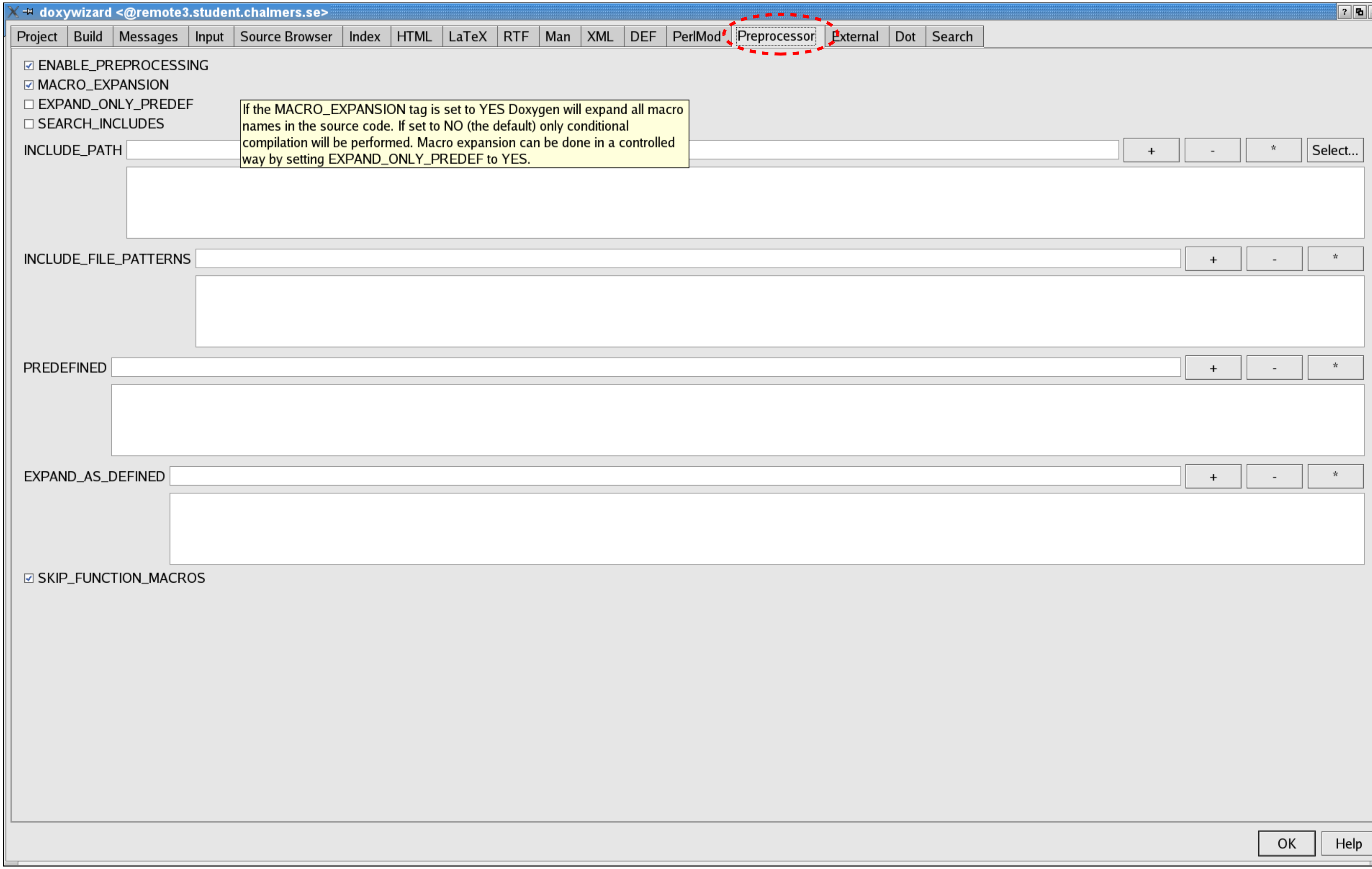
Doxywizard : Expert mode (6/9)



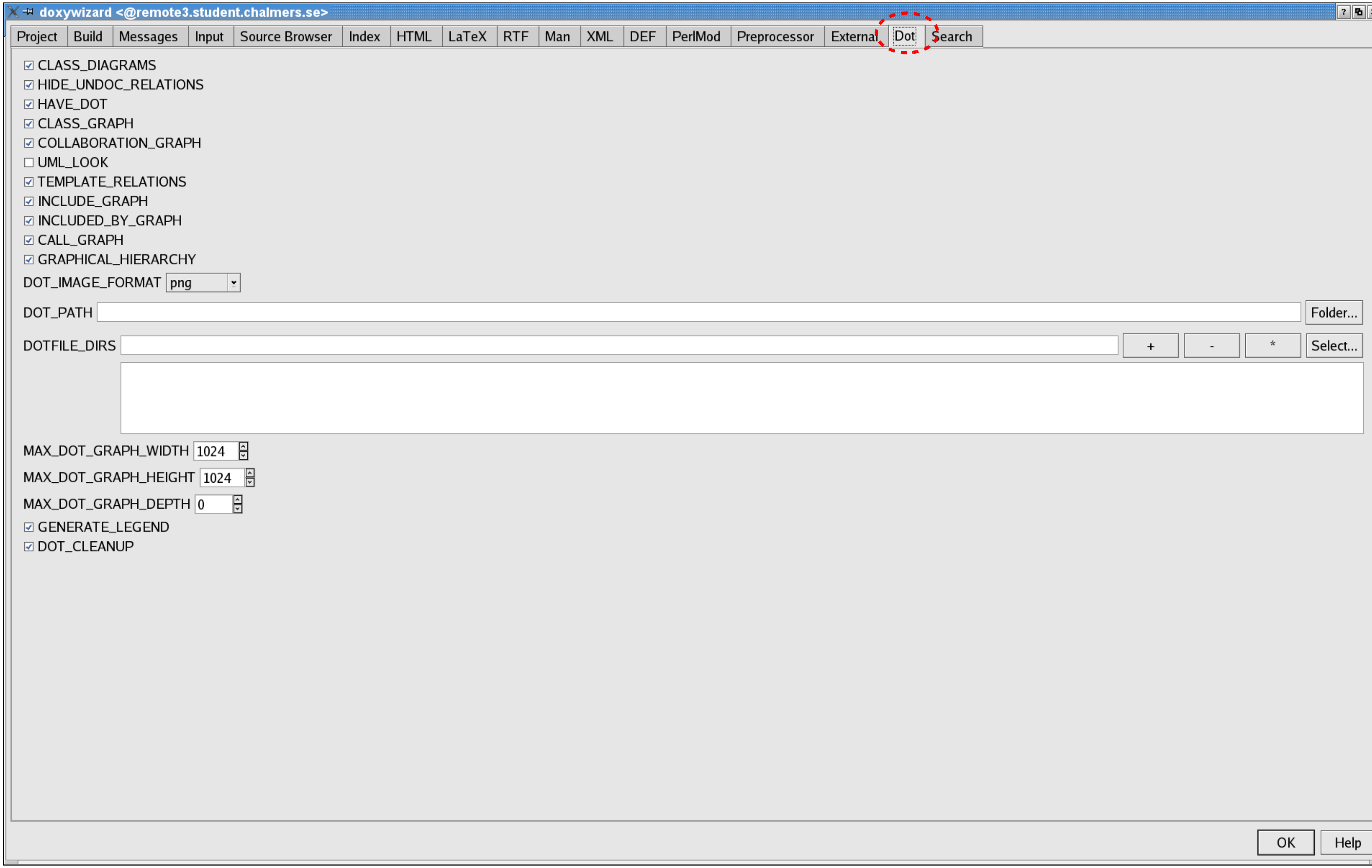
Doxywizard : Expert mode (7/9)



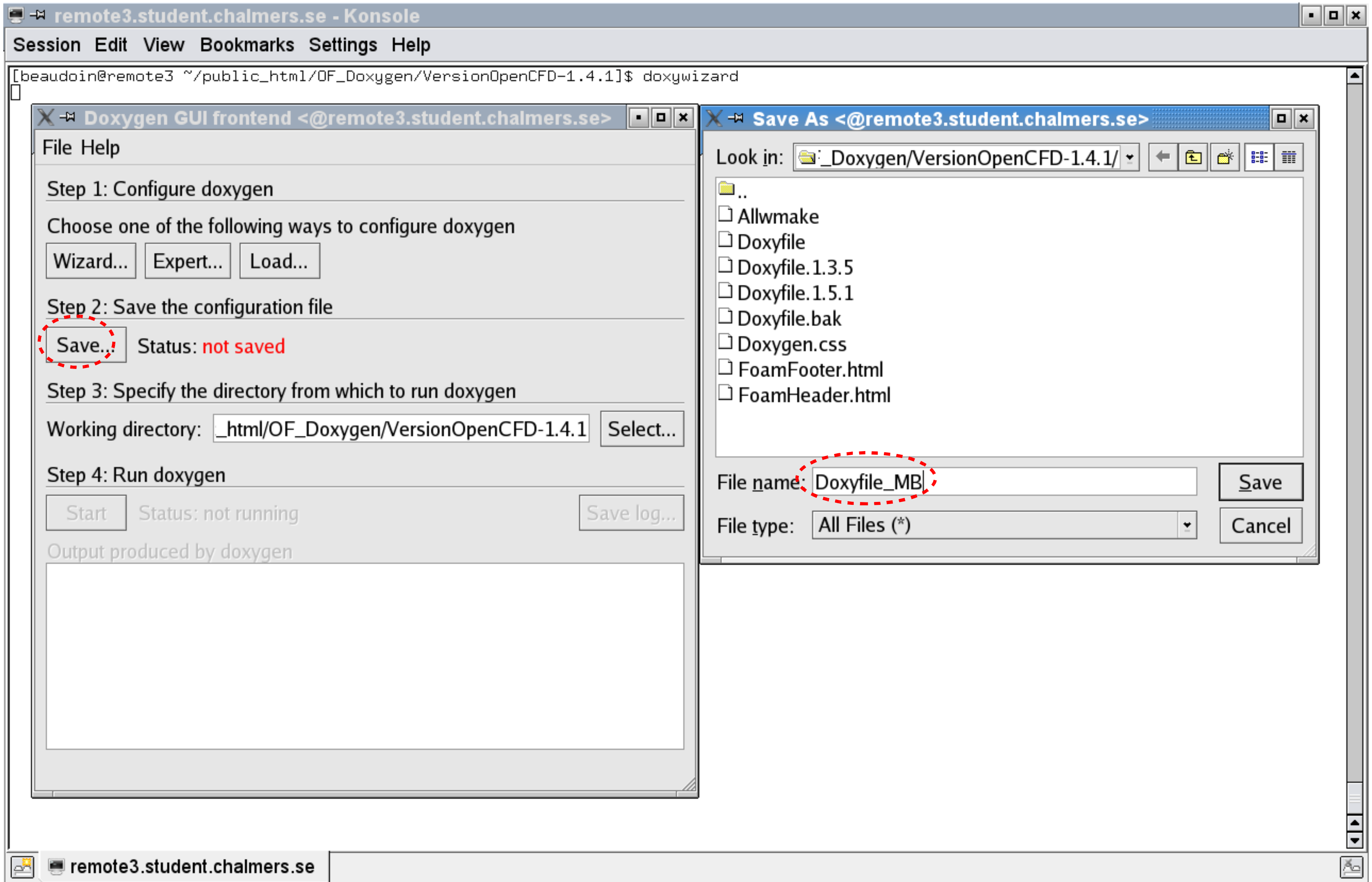
Doxywizard : Expert mode (8/9)



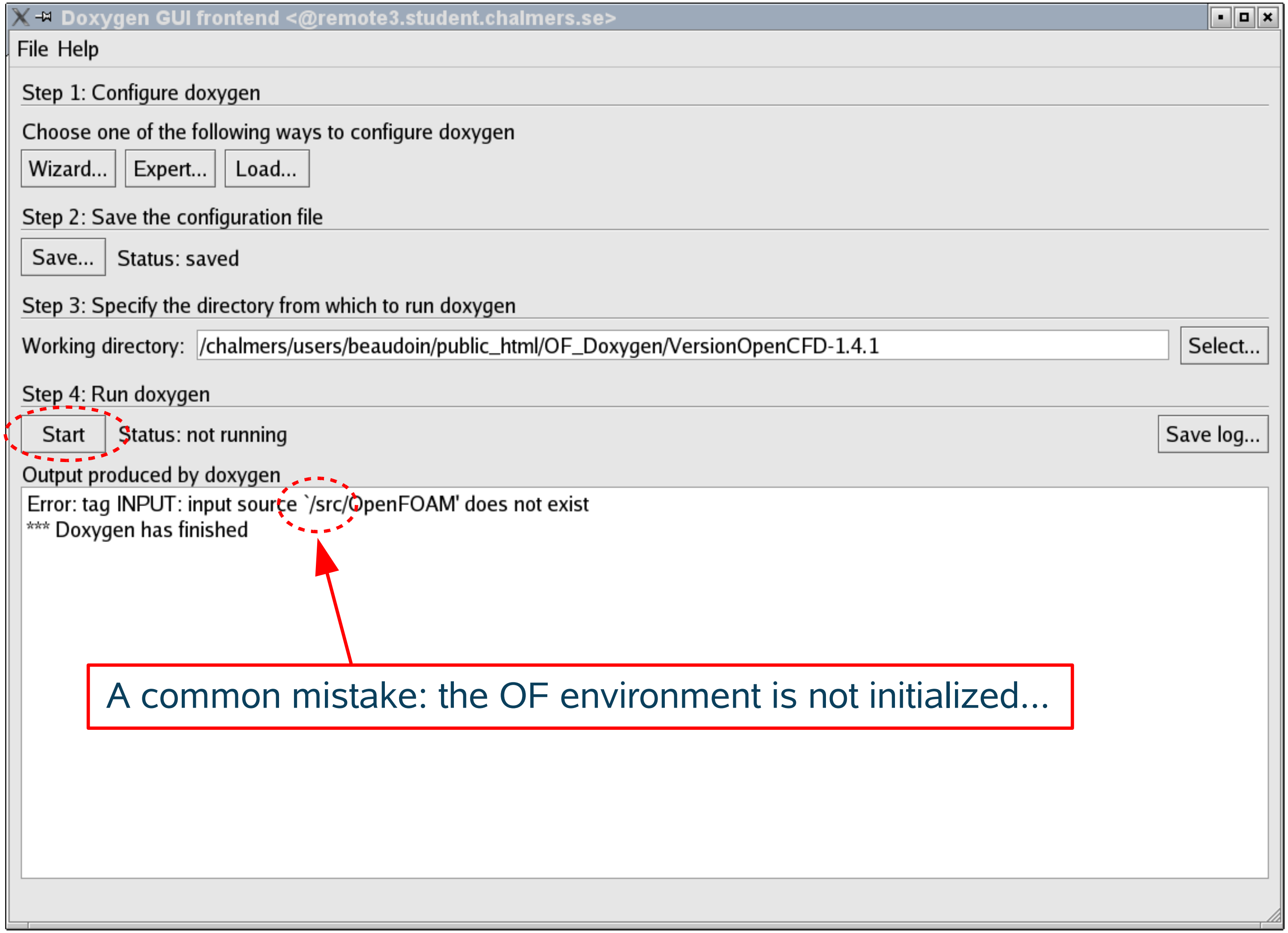
Doxywizard : Expert mode (9/9)



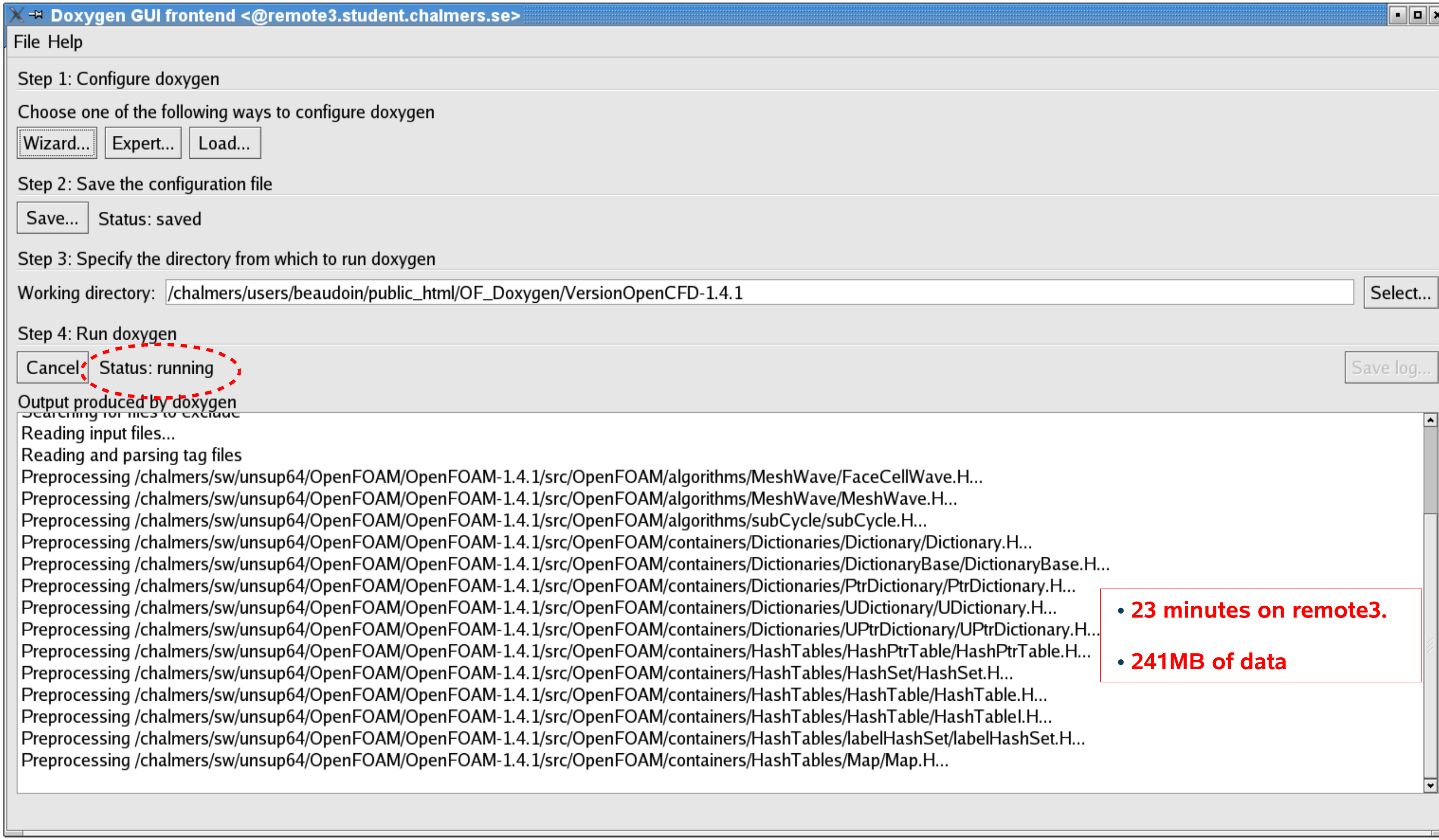
Doxywizard : Saving your modifications



Doxywizard : Generating the HTML files



Doxywizard : Generating the HTML files (take 2)



- **23 minutes on remote3.**
- **241MB of data**

Doxygen: Generating from the command line

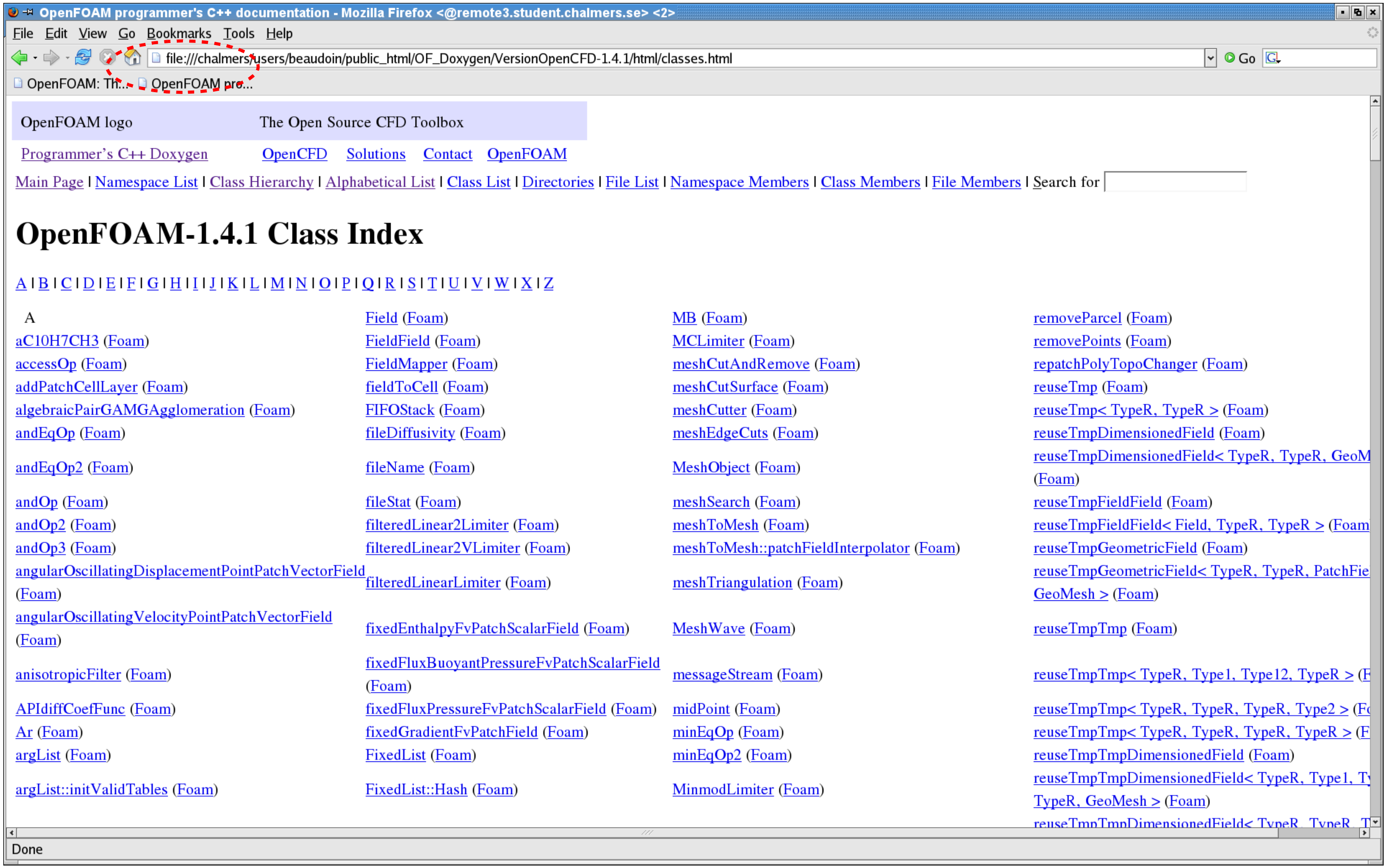
remote3.student.chalmers.se - Konsole

Session Edit View Bookmarks Settings Help

```
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$ doxygen Doxyfile_MB
Searching for include files...
Searching for example files...
Searching for images...
Searching for dot files...
Searching for files to exclude
Reading input files...
Reading and parsing tag files
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/algorithms/MeshWave/FaceCellWave.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/algorithms/MeshWave/MeshWave.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/algorithms/subCycle/subCycle.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionaries/Dictionary/Dictionary.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionaries/DictionaryBase/DictionaryBase.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionaries/PtrDictionary/PtrDictionary.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionaries/UDictionary/UDictionary.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionaries/UPtrDictionary/UPtrDictionary.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/HashPtrTable/HashPtrTable.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/HashSet/HashSet.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/HashTable/HashTable.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/HashTable/HashTableI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/labelHashSet/labelHashSet.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/Map/Map.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/PtrMap/PtrMap.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/StaticHashTable/StaticHashTable.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/HashTables/StaticHashTable/StaticHashTableI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/accessTypes/ILList/ILList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/accessTypes/LList/LList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/accessTypes/LPtrList/LPtrList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/accessTypes/UILList/UILList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/linkTypes/DLListBase/DLListBase.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/linkTypes/DLListBase/DLListBaseI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/linkTypes/SLListBase/SLListBase.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/linkTypes/SLListBase/SLListBaseI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/DLList/DLList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/DLPtrList/DLPtrList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/FIFOStack/FIFOStack.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/IDLList/IDLList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/ISLList/ISLList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/LIFOStack/LIFOStack.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/SLList/SLList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/SLPtrList/SLPtrList.H...
^SPreprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/LinkedLists/user/UIDLList/UIDLList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/BiIndirectList/BiIndirectList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/BiIndirectList/BiIndirectListI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/CompactListList/CompactListList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/CompactListList/CompactListListI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/DynamicList/DynamicList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/DynamicList/DynamicListI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/FixedList/FixedList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/FixedList/FixedListI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/IndirectList/IndirectList.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/IndirectList/IndirectListI.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/List/List.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Lists/List/ListI.H...
```

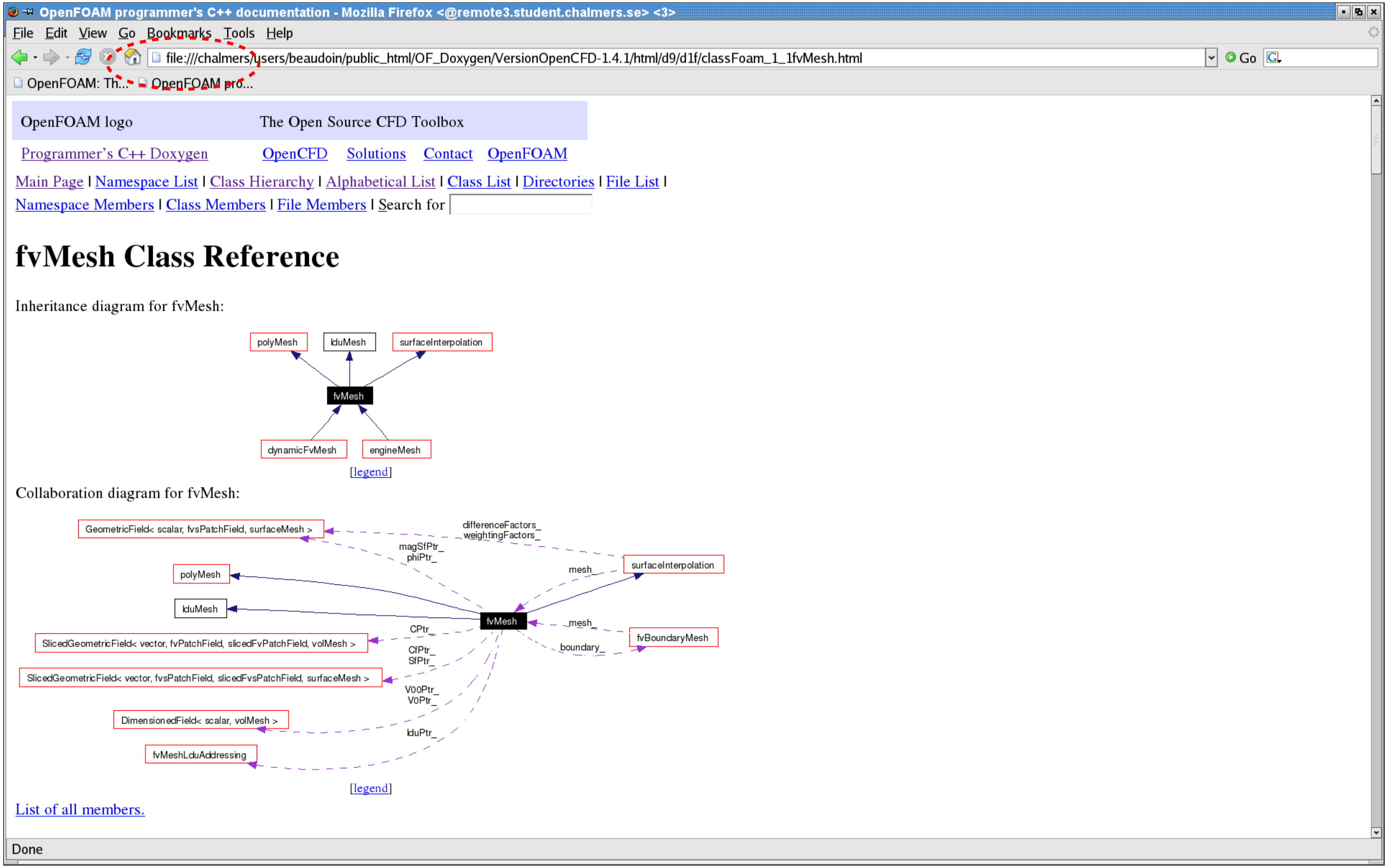
remote3.student.chalmers.se

Doxygen: Browsing the end result (Class Index)

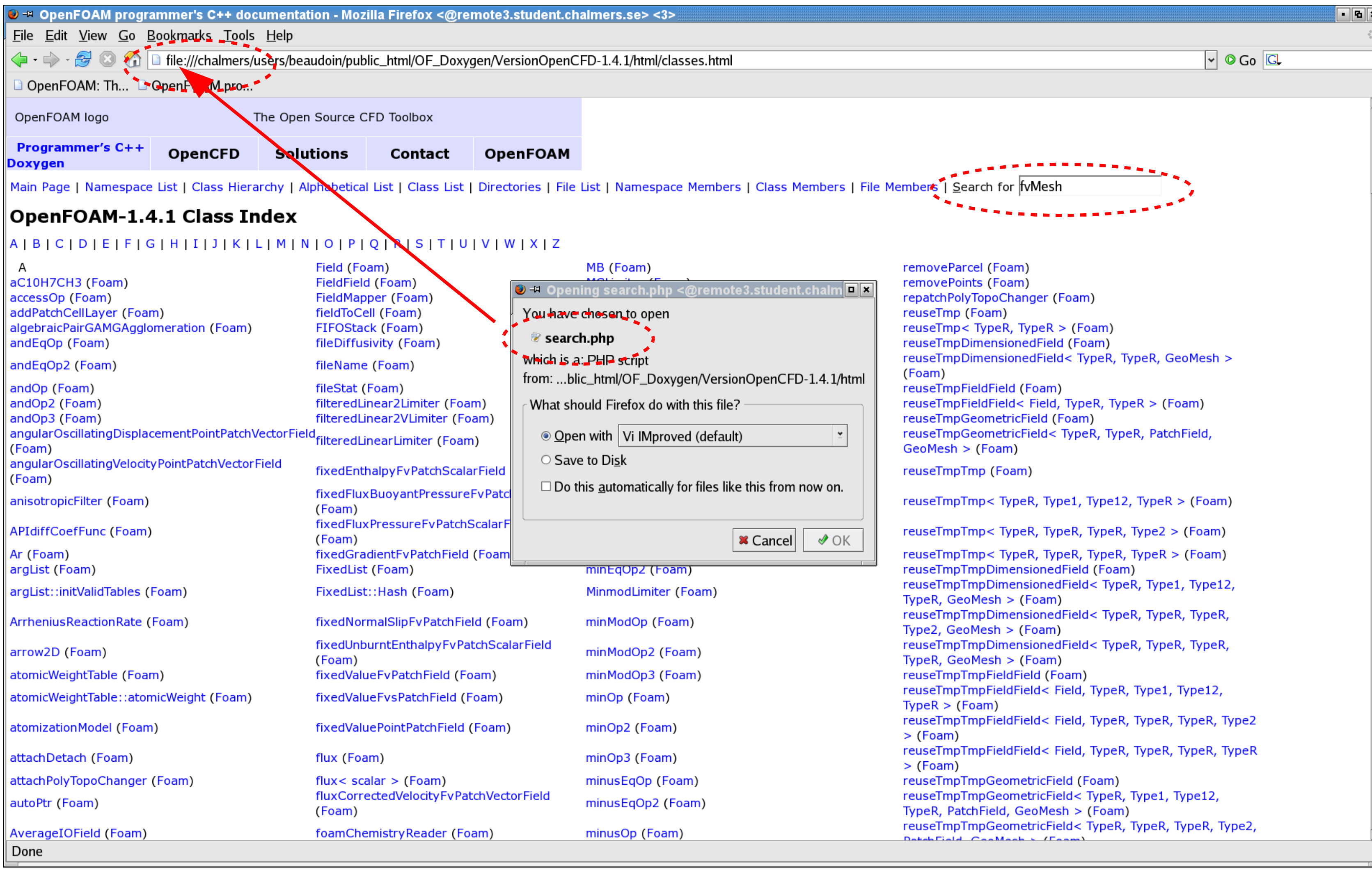


Hydro Québec
Institut de recherche

Doxygen: Browsing the end result (Class Reference)



Doxygen: Browsing the end result: one limitation



Using Doxygen for your own development

- Create a separate directory for the new documentation
- Start from a pre-initialized Doxyfile
- Customize the Doxygen configuration:
  - Change the name of the project
  - Add the path to your library source code
  - Keep only the libraries from OpenFOAM that you need
    - Check your Make/options file for the list of needed libraries
  - Customize as you see fit

- An example:
  - Library OpenFoamTurbo from openfoam-extend
  - http://openfoamwiki.net/index.php/Sig_Turbomachinery_Library_OpenFoamTurbo

Doxygen: Setup for library OpenFoamTurbo

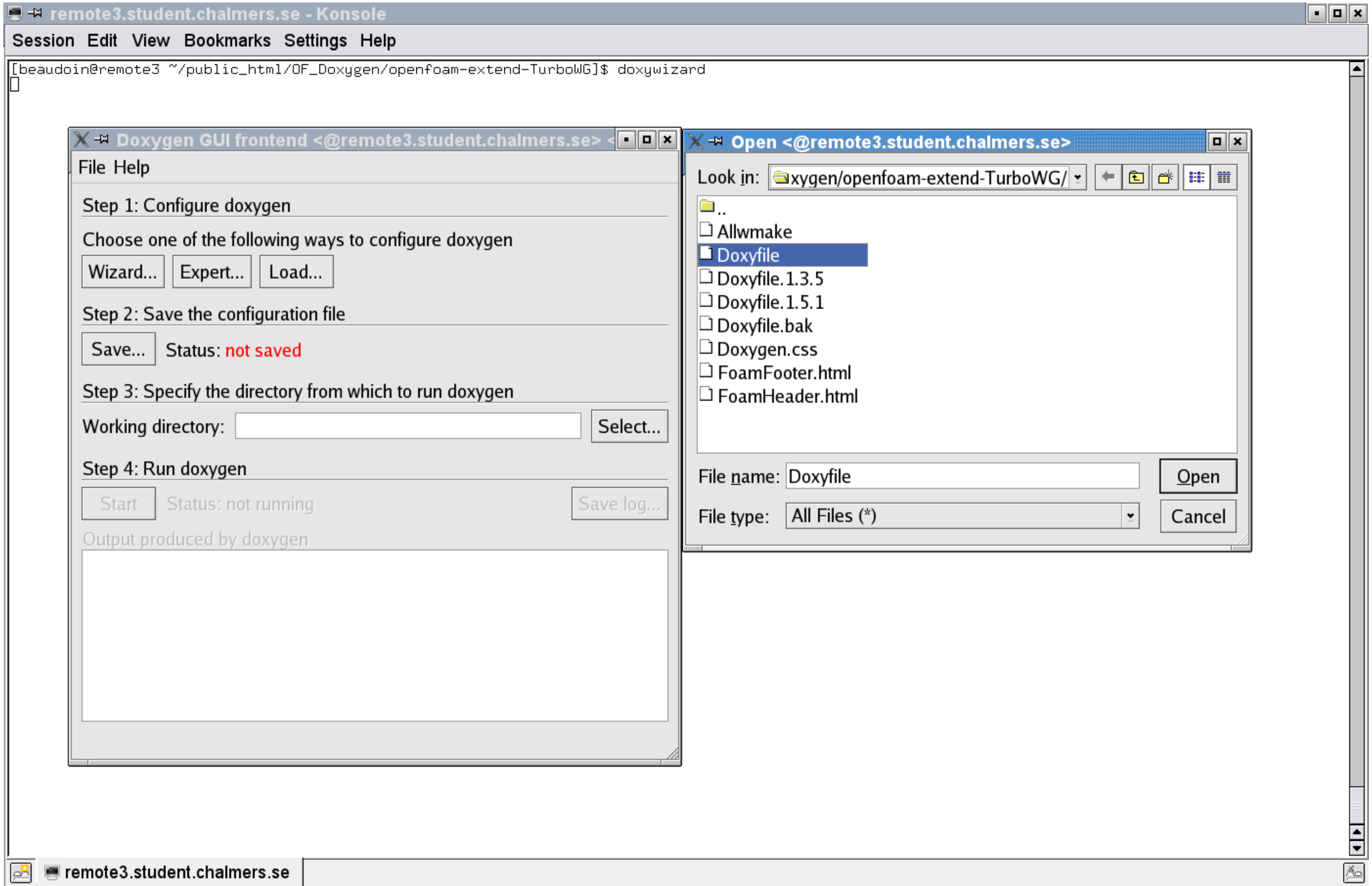
remote3.student.chalmers.se - Konsole

Session Edit View Bookmarks Settings Help

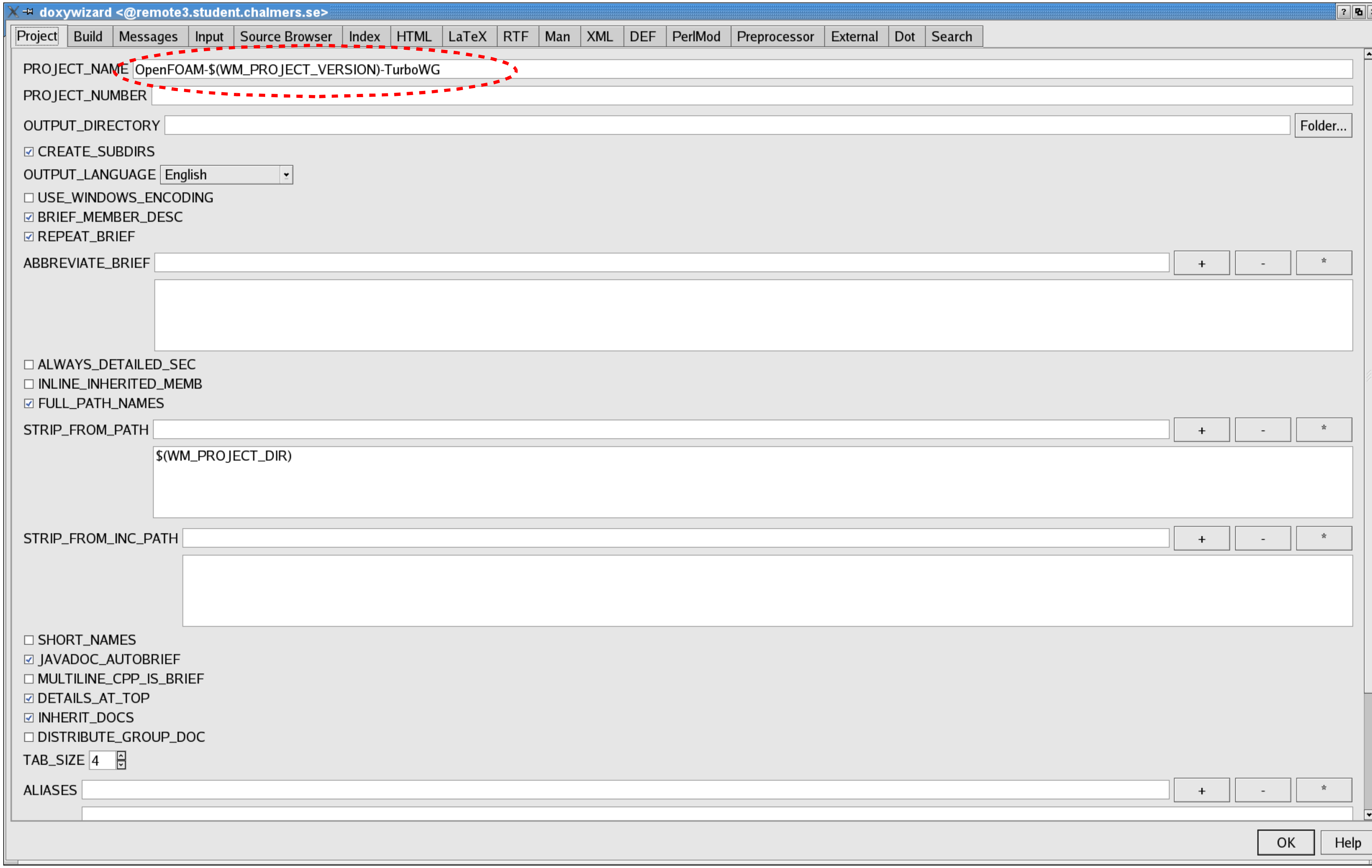
```
[beaudoin@remote3 ~]$ mkdir -p ~/public_html/OF_Doxygen/openfoam-extend-TurboWG
[beaudoin@remote3 ~]$ cd ~/public_html/OF_Doxygen/openfoam-extend-TurboWG
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ cp /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/doc/Doxygen/Allwmake .
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ cp /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/doc/Doxygen/*.html .
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ cp /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/doc/Doxygen/Doxy* .
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ ls -axl
total 152
drwxrwxr-x  2 beaudoin beaudoin  8192 Mar 26 14:31 .
drwxr-xr-x  5 beaudoin beaudoin  8192 Mar 26 14:29 ..
-rwxr-xr-x  1 beaudoin beaudoin   142 Mar 26 14:31 Allwmake
-rw-r--r--  1 beaudoin beaudoin 54365 Mar 26 14:31 Doxyfile
-rw-r--r--  1 beaudoin beaudoin 47900 Mar 26 14:31 Doxyfile.1.3.5
-rw-r--r--  1 beaudoin beaudoin  1095 Mar 26 14:31 Doxygen.css
-rw-r--r--  1 beaudoin beaudoin    95 Mar 26 14:31 FoamFooter.html
-rw-r--r--  1 beaudoin beaudoin  2719 Mar 26 14:31 FoamHeader.html
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ cp Doxyfile Doxyfile.1.5.1
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ cp Doxyfile.1.3.5 Doxyfile
[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ doxygen -u


Configuration file `Doxyfile' updated.

[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ head -2 Doxyfile
# Doxyfile 1.3.9.1

[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ 
```

remote3.student.chalmers.se

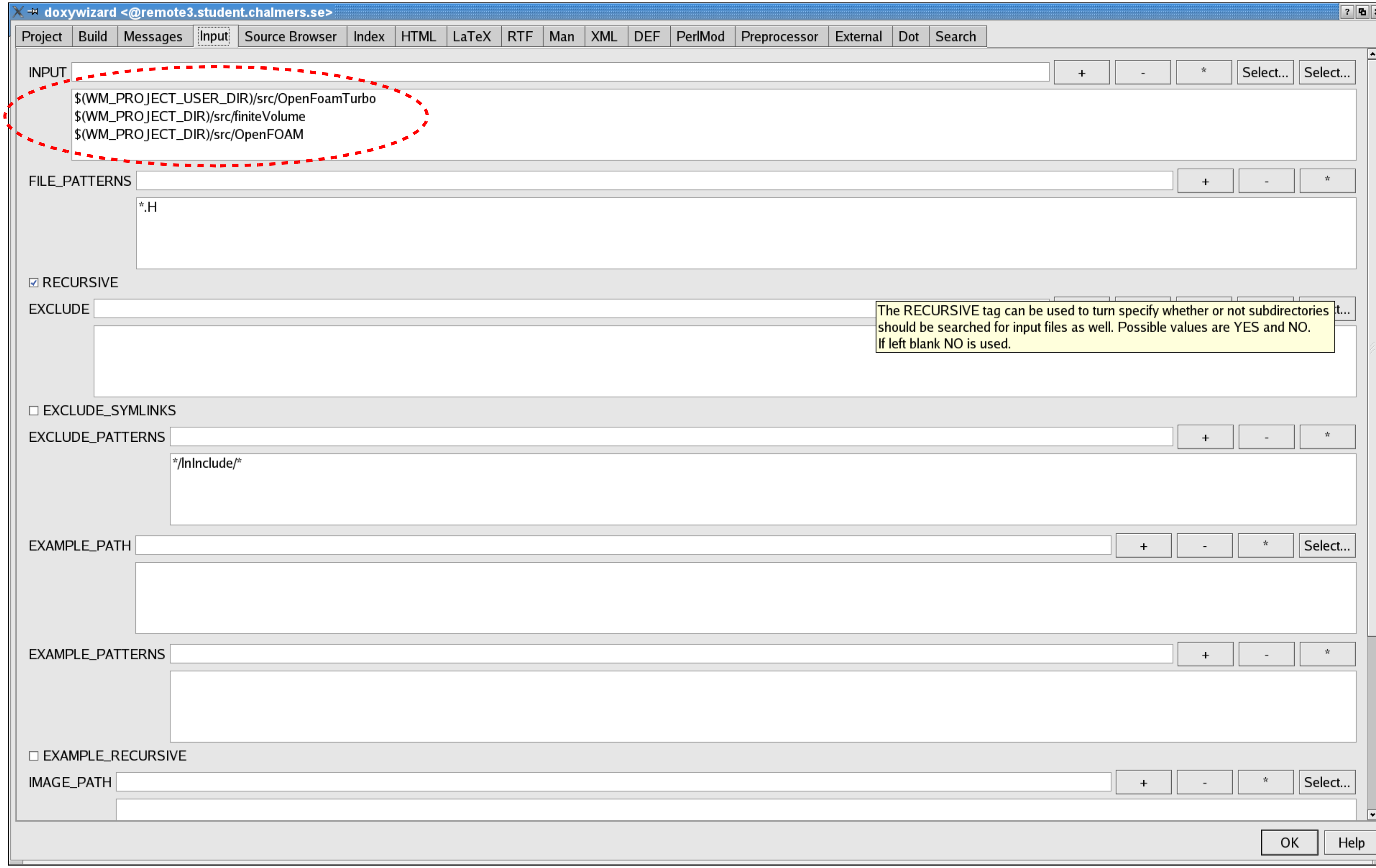
Doxygen: Configuration for OpenFoamTurbo (1/5)



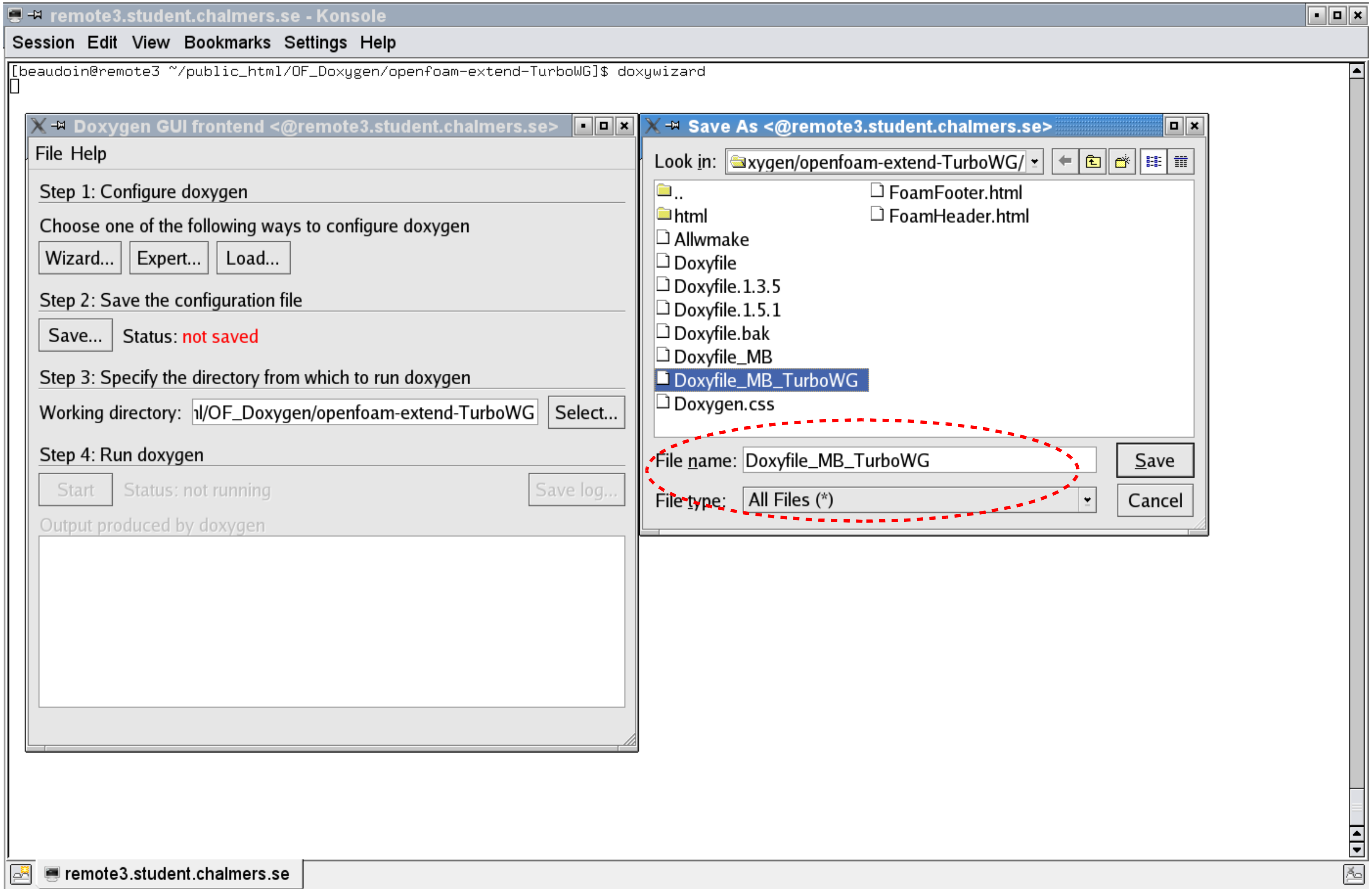
Doxygen: Configuration for OpenFoamTurbo (2/5)



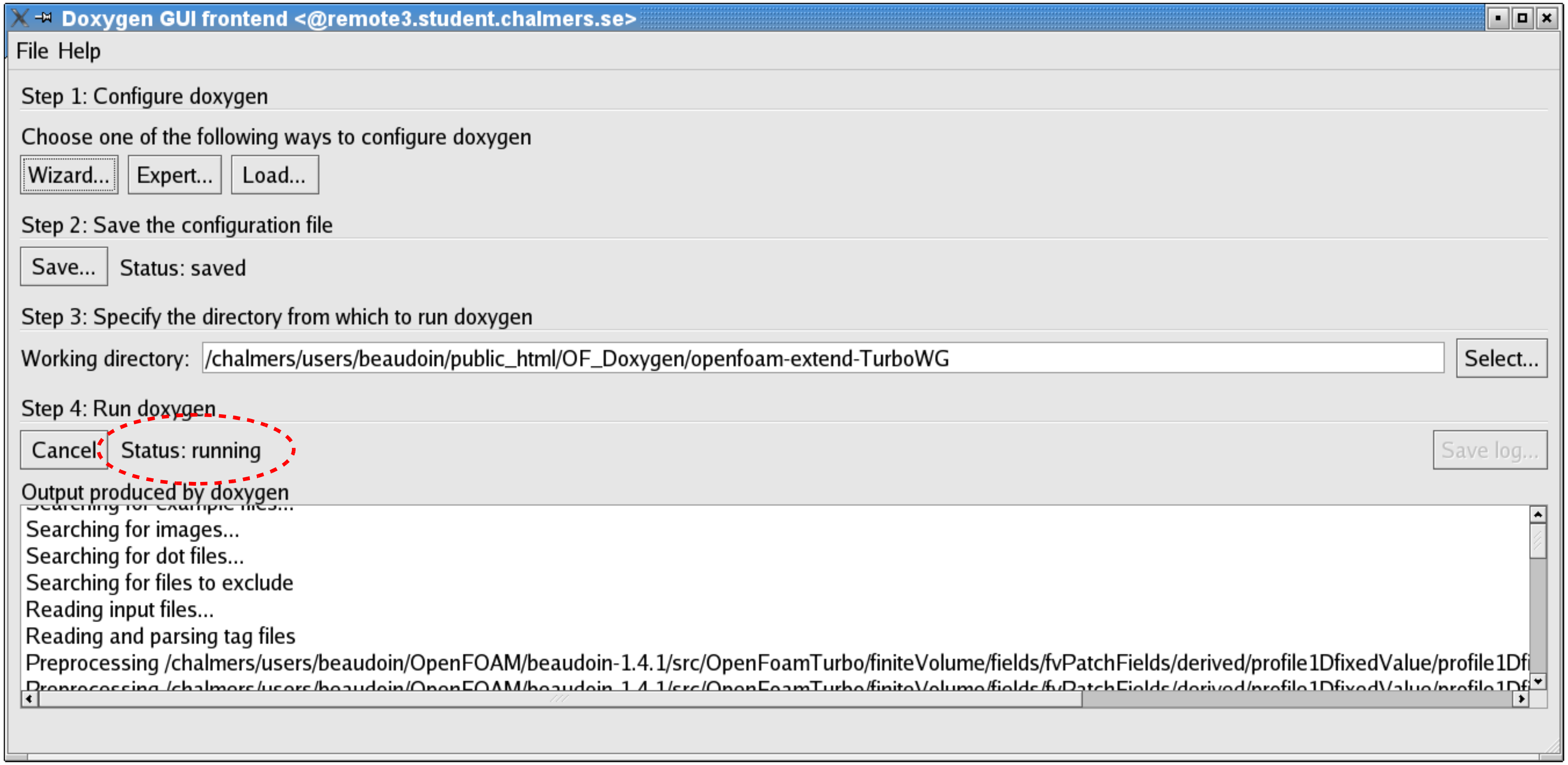
Doxygen: Configuration for OpenFoamTurbo (3/5)



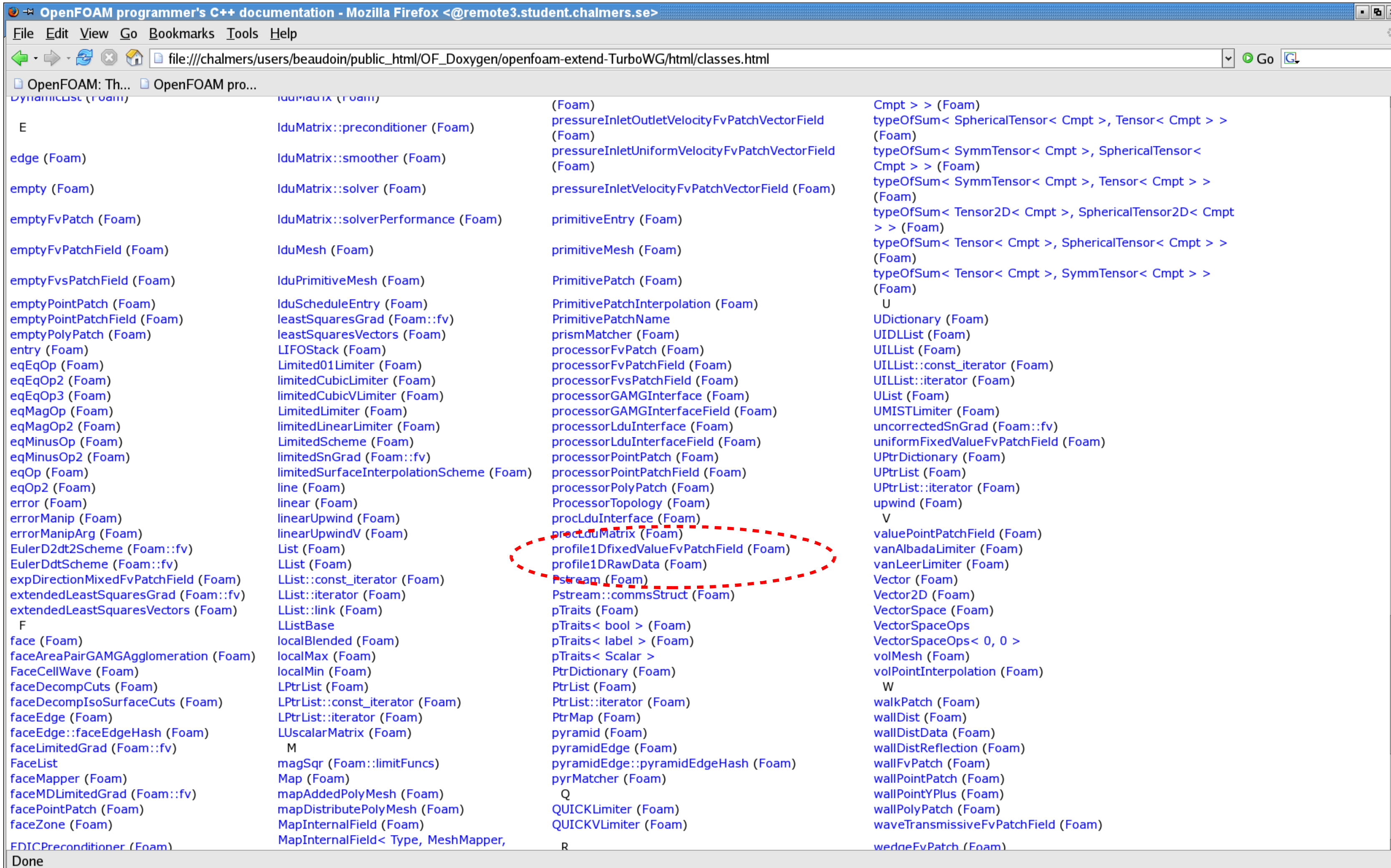
Doxygen: Configuration for OpenFoamTurbo (4/5)



Doxygen: Configuration for OpenFoamTurbo (5/5)



OpenFoamTurbo: Browsing the result (1/3)



OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote3.student.chalmers.se>

File Edit View Go Bookmarks Tools Help

file:///chalmers/users/beaudoin/public_html/OF_Doxygen/openfoam-extend-TurboWG/html/classes.html Go

OpenFOAM: Th... OpenFOAM pro...

DynamicList (Foam)
E
edge (Foam)
empty (Foam)
emptyFvPatch (Foam)
emptyFvPatchField (Foam)
emptyFvsPatchField (Foam)
emptyPointPatch (Foam)
emptyPointPatchField (Foam)
emptyPolyPatch (Foam)
entry (Foam)
eqEqOp (Foam)
eqEqOp2 (Foam)
eqEqOp3 (Foam)
eqMagOp (Foam)
eqMagOp2 (Foam)
eqMinusOp (Foam)
eqMinusOp2 (Foam)
eqOp (Foam)
eqOp2 (Foam)
error (Foam)
errorManip (Foam)
errorManipArg (Foam)
EulerD2dt2Scheme (Foam::fv)
EulerDdtScheme (Foam::fv)
expDirectionMixedFvPatchField (Foam)
extendedLeastSquaresGrad (Foam::fv)
extendedLeastSquaresVectors (Foam)
F
face (Foam)
faceAreaPairGAMGAgglomeration (Foam)
FaceCellWave (Foam)
faceDecompCuts (Foam)
faceDecompIsoSurfaceCuts (Foam)
faceEdge (Foam)
faceEdge::faceEdgeHash (Foam)
faceLimitedGrad (Foam::fv)
FaceList
faceMapper (Foam)
faceMDLimitedGrad (Foam::fv)
facePointPatch (Foam)
faceZone (Foam)
FDICPreconditioner (Foam)

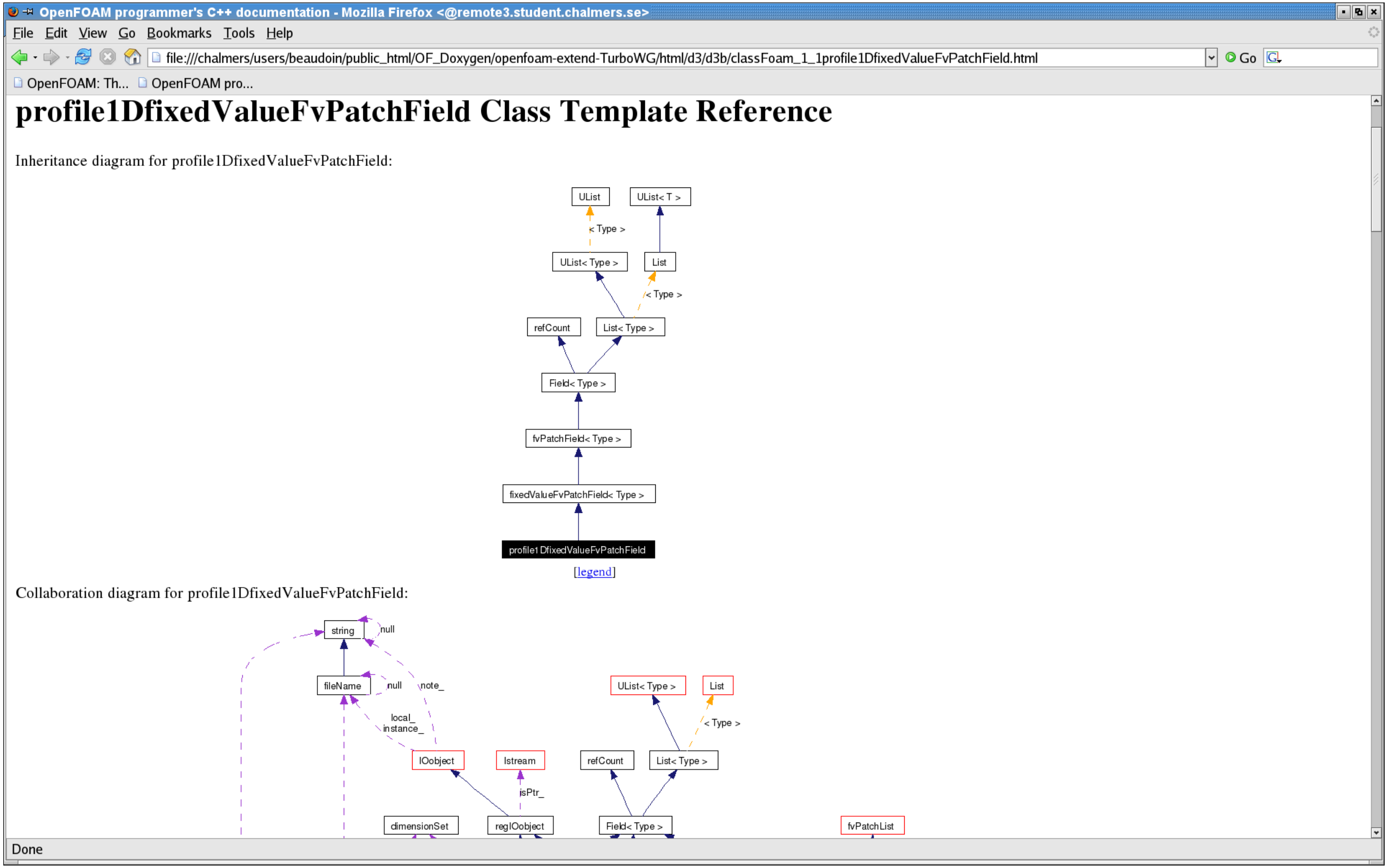
lduMatrix (Foam)
lduMatrix::preconditioner (Foam)
lduMatrix::smoother (Foam)
lduMatrix::solver (Foam)
lduMatrix::solverPerformance (Foam)
lduMesh (Foam)
lduPrimitiveMesh (Foam)
lduScheduleEntry (Foam)
leastSquaresGrad (Foam::fv)
leastSquaresVectors (Foam)
LIFOStack (Foam)
Limited01Limiter (Foam)
limitedCubicLimiter (Foam)
limitedCubicVLimiter (Foam)
LimitedLimiter (Foam)
limitedLinearLimiter (Foam)
LimitedScheme (Foam)
limitedSnGrad (Foam::fv)
limitedSurfaceInterpolationScheme (Foam)
line (Foam)
linear (Foam)
linearUpwind (Foam)
linearUpwindV (Foam)
List (Foam)
LList (Foam)
LList::const_iterator (Foam)
LList::iterator (Foam)
LList::link (Foam)
LListBase
localBlended (Foam)
localMax (Foam)
localMin (Foam)
LPtrList (Foam)
LPtrList::const_iterator (Foam)
LPtrList::iterator (Foam)
LUscalarMatrix (Foam)
M
magSqr (Foam::limitFuncs)
Map (Foam)
mapAddedPolyMesh (Foam)
mapDistributePolyMesh (Foam)
MapInternalField (Foam)
MapInternalField< Type, MeshMapper,

(Foam)
pressureInletOutletVelocityFvPatchVectorField (Foam)
pressureInletUniformVelocityFvPatchVectorField (Foam)
pressureInletVelocityFvPatchVectorField (Foam)
primitiveEntry (Foam)
primitiveMesh (Foam)
PrimitivePatch (Foam)
PrimitivePatchInterpolation (Foam)
PrimitivePatchName
prismMatcher (Foam)
processorFvPatch (Foam)
processorFvPatchField (Foam)
processorFvsPatchField (Foam)
processorGAMGInterface (Foam)
processorGAMGInterfaceField (Foam)
processorLduInterface (Foam)
processorLduInterfaceField (Foam)
processorPointPatch (Foam)
processorPointPatchField (Foam)
processorPolyPatch (Foam)
ProcessorTopology (Foam)
procLduInterface (Foam)
procLduMatrix (Foam)
profile1DfixedValueFvPatchField (Foam)
profile1DRawData (Foam)
Pstream (Foam)
Pstream::commsStruct (Foam)
pTraits (Foam)
pTraits< bool > (Foam)
pTraits< label > (Foam)
pTraits< Scalar >
PtrDictionary (Foam)
PtrList (Foam)
PtrList::iterator (Foam)
PtrMap (Foam)
pyramid (Foam)
pyramidEdge (Foam)
pyramidEdge::pyramidEdgeHash (Foam)
pyrMatcher (Foam)
Q
QUICKLimiter (Foam)
QUICKVLimiter (Foam)
R

Cmpt > > (Foam)
typeOfSum< SphericalTensor< Cmpt >, Tensor< Cmpt > > (Foam)
typeOfSum< SymmTensor< Cmpt >, SphericalTensor< Cmpt > > (Foam)
typeOfSum< SymmTensor< Cmpt >, Tensor< Cmpt > > (Foam)
typeOfSum< Tensor2D< Cmpt >, SphericalTensor2D< Cmpt > > (Foam)
typeOfSum< Tensor< Cmpt >, SphericalTensor< Cmpt > > (Foam)
typeOfSum< Tensor< Cmpt >, SymmTensor< Cmpt > > (Foam)
U
UDictionary (Foam)
UIDLList (Foam)
UILList (Foam)
UILList::const_iterator (Foam)
UILList::iterator (Foam)
UList (Foam)
UMISTLimiter (Foam)
uncorrectedSnGrad (Foam::fv)
uniformFixedValueFvPatchField (Foam)
UPtrDictionary (Foam)
UPtrList (Foam)
UPtrList::iterator (Foam)
upwind (Foam)
V
valuePointPatchField (Foam)
vanAlbadaLimiter (Foam)
vanLeerLimiter (Foam)
Vector (Foam)
Vector2D (Foam)
VectorSpace (Foam)
VectorSpaceOps
VectorSpaceOps< 0, 0 >
volMesh (Foam)
volPointInterpolation (Foam)
W
walkPatch (Foam)
wallDist (Foam)
wallDistData (Foam)
wallDistReflection (Foam)
wallFvPatch (Foam)
wallPointPatch (Foam)
wallPointYPlus (Foam)
wallPolyPatch (Foam)
waveTransmissiveFvPatchField (Foam)
wedgeFvPatch (Foam)

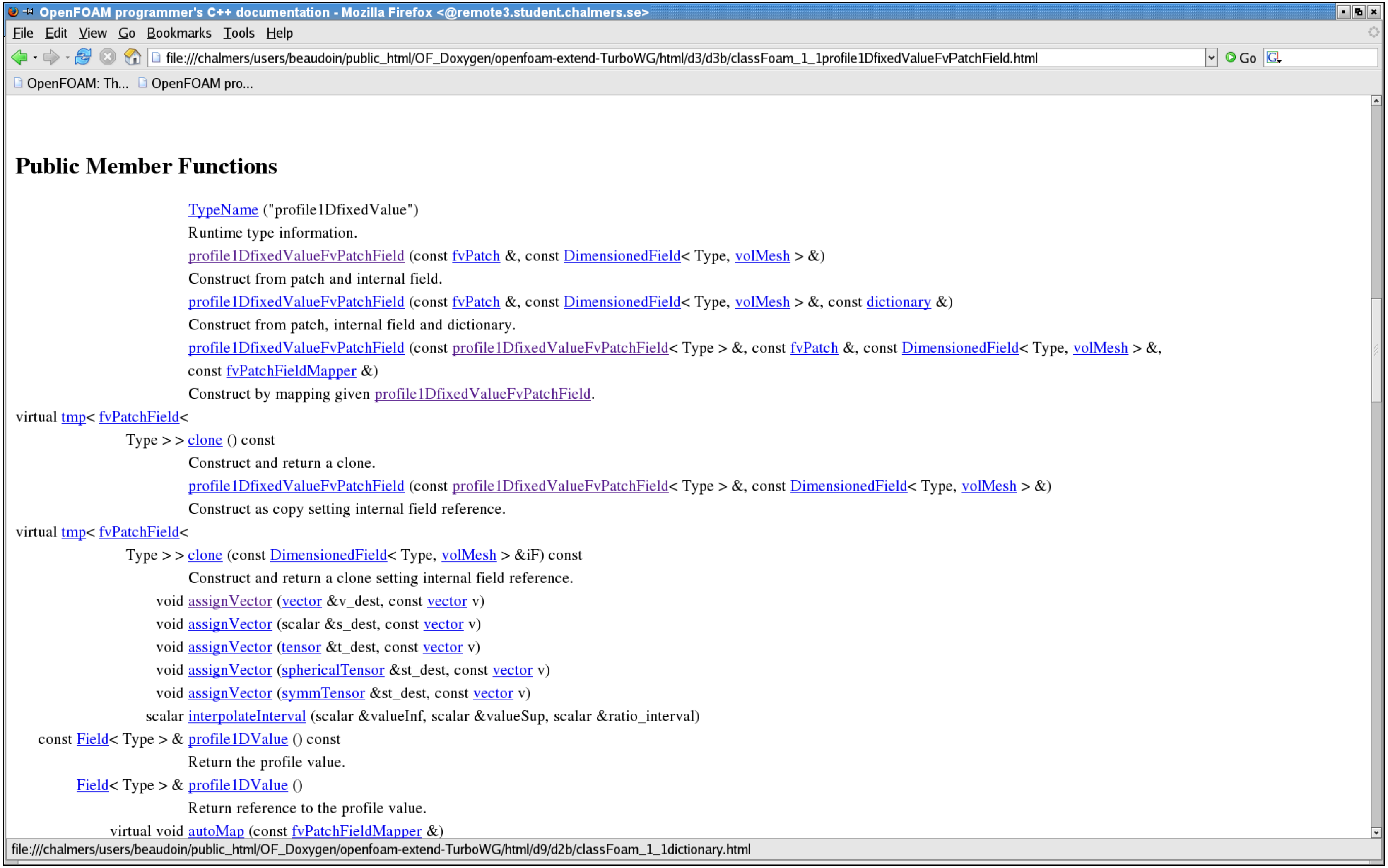
Done

OpenFoamTurbo: Browsing the result (2/3)

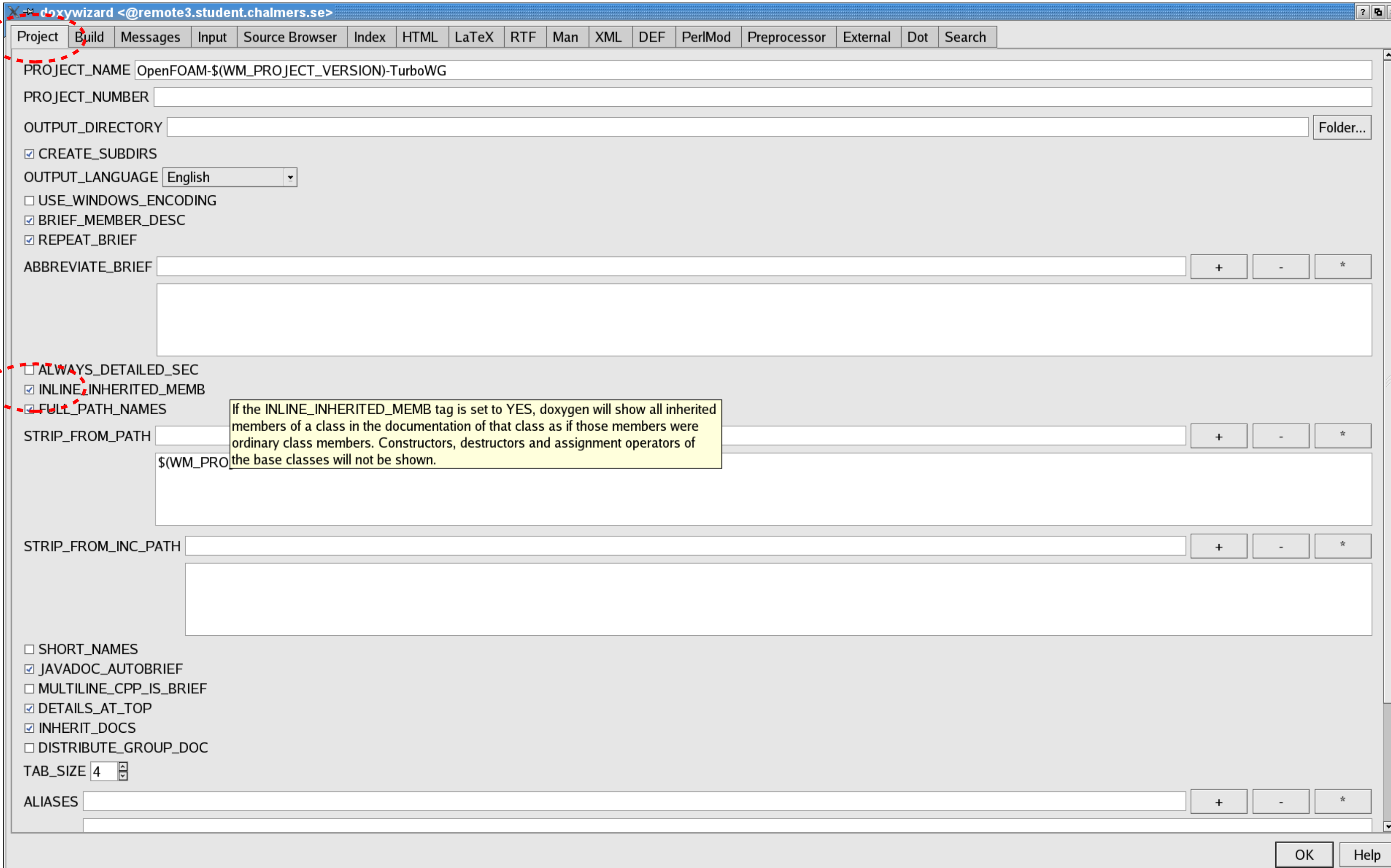


Hydro Québec Institut de recherche

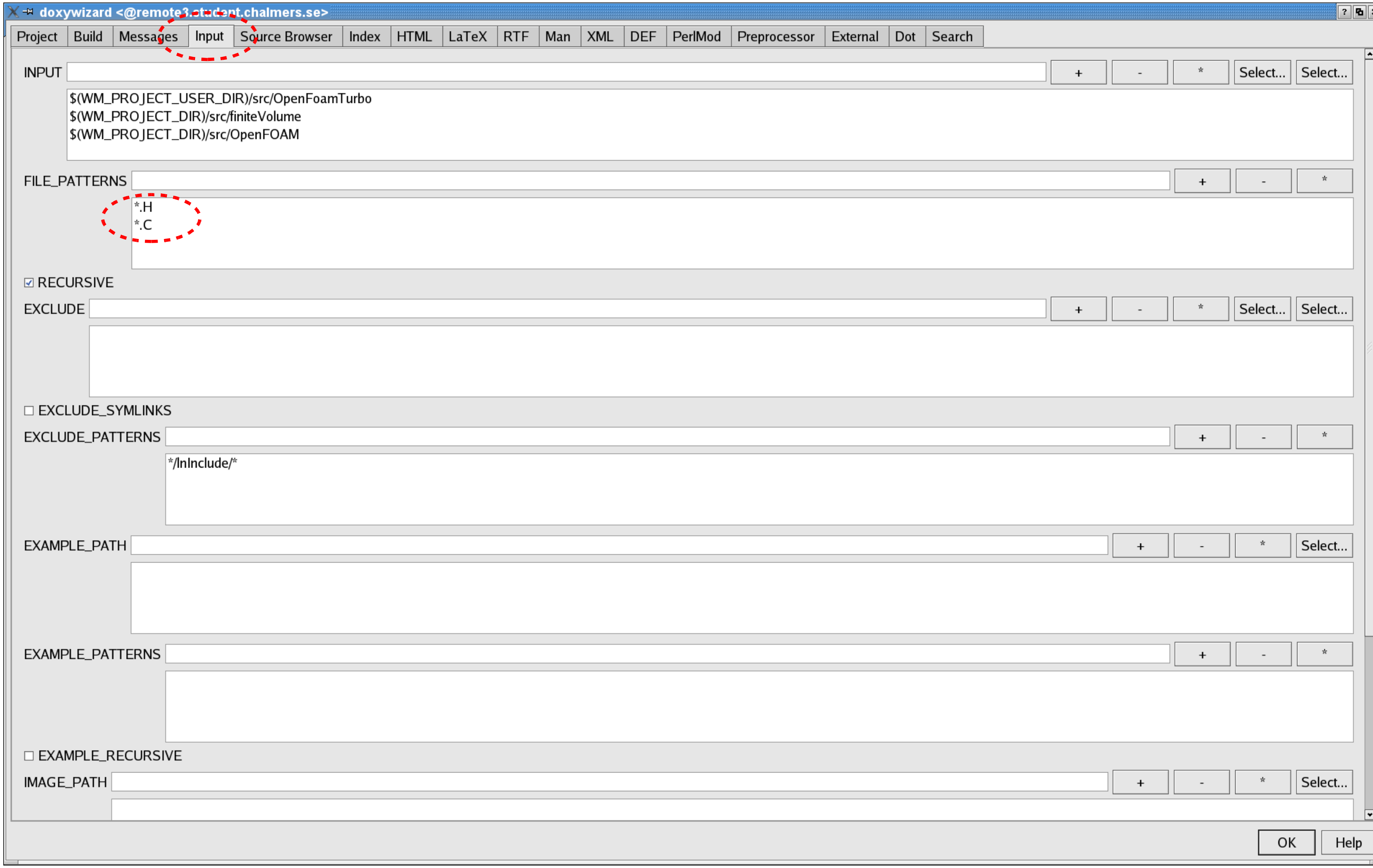
OpenFoamTurbo: Browsing the result (3/3)



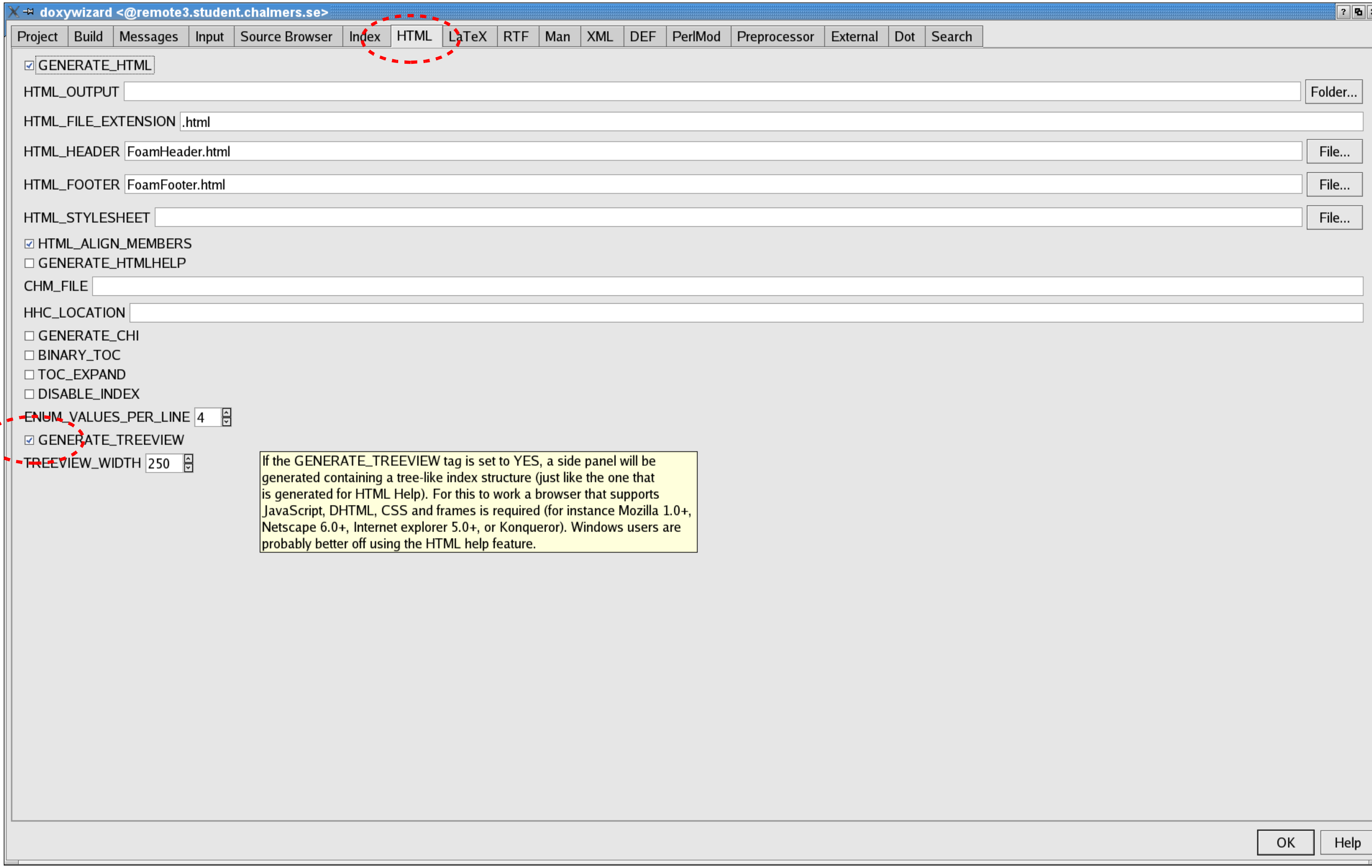
Doxygen: some possible improvements (1/5)



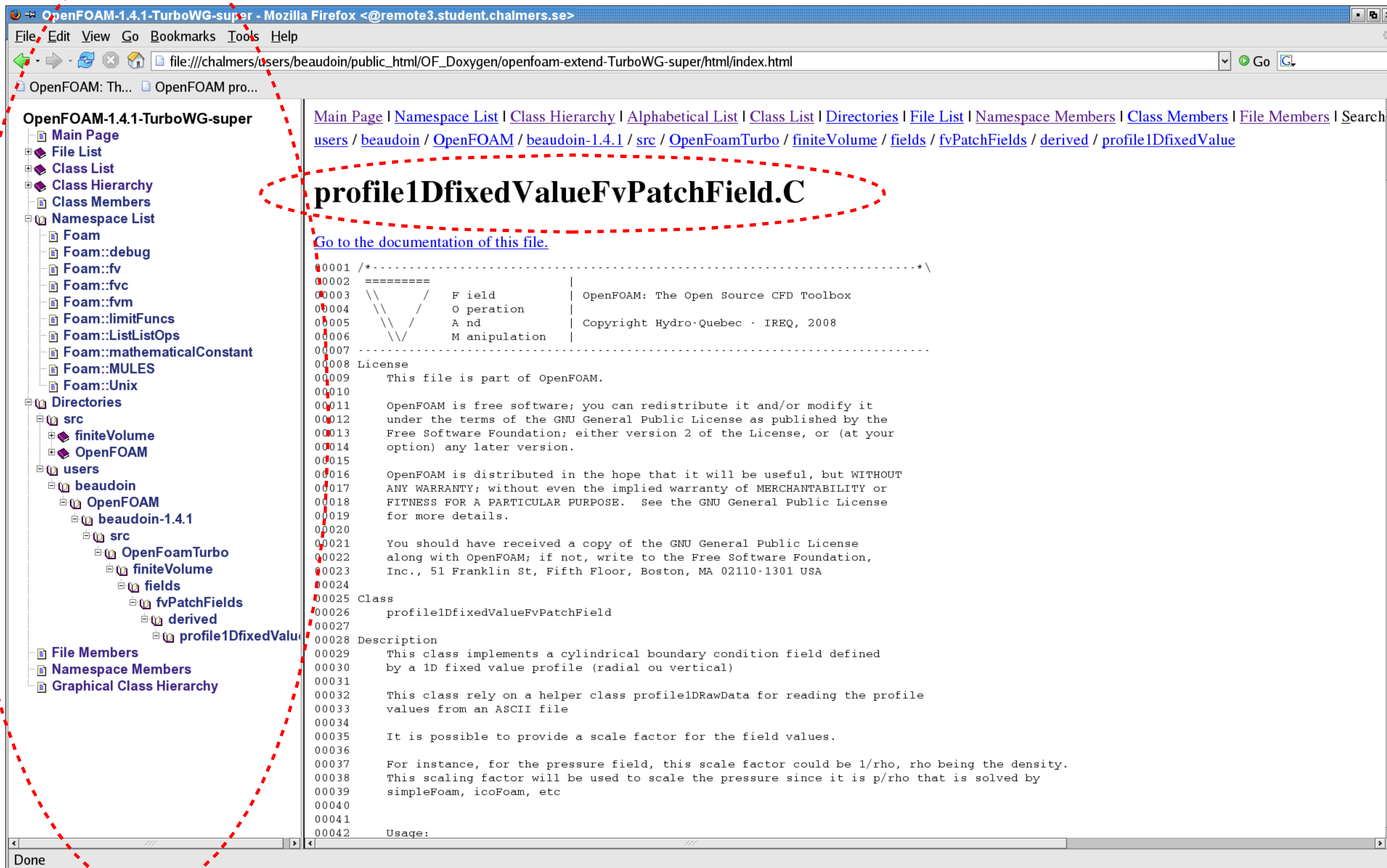
Doxygen: some possible improvements (2/5)



Doxygen: some possible improvements (3/5)

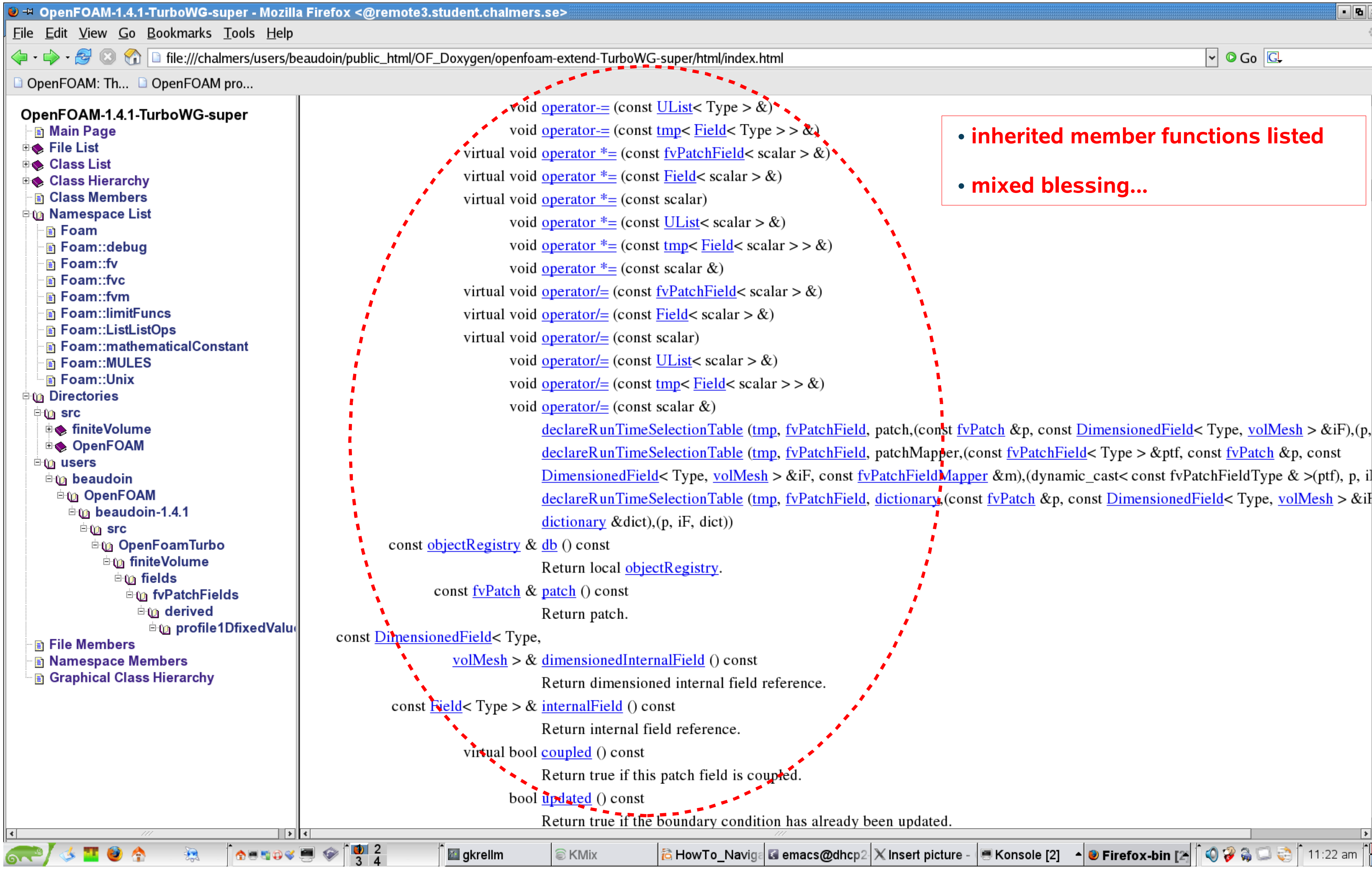


Doxygen: some possible improvements (4/5)



Hydro Québec
Institut de recherche

Doxygen: some possible improvements (5/5)



- **inherited member functions listed**
- **mixed blessing…**

Doxygen: some useful pointers

- **Main Web site:**
  - http://www.stack.nl/~dimitri/doxygen/index.html
- **The Doxygen manual:**
  - http://www.stack.nl/~dimitri/doxygen/manual.html
- **Helping Doxygen grab the comments in your source code:**
  - http://www.stack.nl/~dimitri/doxygen/docblocks.html

OpenFOAM overall directory structure

- $WM_PROJECT_DIR
  - applications/
  - bin/
  - doc/
  - lib/
  - src/
  - tutorials/
  - wmake/

- $WM_PROJECT_USER_DIR
  - applications/
  - lib/
  - run/

OpenFOAM useful shell aliases

- OpenFOAM provides useful predefined shell aliases:
  - alias foam='cd $WM_PROJECT_DIR'
  - alias run='cd $FOAM_RUN'
  - alias tut='cd $FOAM_TUTORIALS'
  - alias app='cd $FOAM_APP'
  - alias src='cd $FOAM_SRC'
  - etc., etc.

- You can also define your own aliases:
  - alias doc='cd $WM_PROJECT_DIR/doc'
  - alias of_1.4.1='. /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/.OpenFOAM-1.4.1/bashrc'
  - alias of_1.4.1-dev='. /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1-dev/.OpenFOAM-1.4.1-dev/bashrc'

- We can now move directly and quickly to some of the major OpenFOAM directories.

Navigating a Unix file tree efficiently :

- **Moving around OF file tree can be quite a typing exercise:**
  - cd src/finiteVolume/interpolation/surfaceInterpolation/multivariateSchemes/multivariateSurfaceInterpolationScheme

- **bash/tcsh auto-completion helps a lot:**
  - cd s[TAB]fi[TAB]in[TAB]su[TAB]mu[TAB]mu[TAB]u[TAB]

- **Moving around while leaving a trail:**
  - **pushd** : change directory, but remember where you came from
  - **popd**: move back to previous directory memorized by pushd
  - **dirs**: display the list of remembered directories

  - popd/pushd/dirs implement a **FILO** (First In Last Out) **stack of directories** that help you navigate efficiently in and out of a deep file tree
  - *man tcsh* or *man bash* for more details

Navigating efficiently : pushd/popd examples

Commands:	Current Directory:	Directory stack:
$> foam $> dirs	$WM_PROJECT_DIR	$WM_PROJECT_DIR
$> pushd /tmp $> dirs	/tmp	$WM_PROJECT_DIR /tmp
$> pushd ~ $> dirs	$HOME	$WM_PROJECT_DIR /tmp $HOME
$> popd $> dirs	/tmp	$WM_PROJECT_DIR /tmp
$> popd $> dirs	$WM_PROJECT_DIR	$WM_PROJECT_DIR

Navigating efficiently : pushd and aliases??

- **A little quiz...**
  - How do you combine aliases like 'src' or 'app' with pushd/popd?

Navigating efficiently : pushd and aliases??

- **A little quiz...**
  - How do you combine aliases like 'src' or 'app' with pushd/popd?

```
$> pushd .                          # memorize where we are
$> src                              # move to $FOAM_SRC
$> cd finiteVolume
$> .... do some browsing, etc. ...
$> popd                             # back where we were before pushd
```

Finding files:

- **find:**
  - find $FOAM_SRC -iname *fixedValue*.[CH]

- **locate:**
  - useful command to search through predefined file systems on a Unix workstation.
    - pre-digested, file system wide, “find” results stored in a database
    - very quick search time
      - example: locate -i doxygen
    - *man locate* for more details

Finding files, one library at time:

- **`lnInclude` directory:**
  - Available for every OpenFOAM library
  - Created and updated automatically by wmake
  - Provides, under a single roof, a quick access to all the files needed to compile a given library
  - Basic Unix commands do apply to symbolic links too:
    - grep, find, more, etc.
  - Editing a symbolic link == editing the actual file
    - vi $FOAM_SRC/finiteVolume/lnInclude/cyclicFvPatch.H
    - vi $FOAM_SRC/finiteVolume/fvMesh/fvPatches/constraint/cyclic/cyclicFvPatch.H

Searching inside files:

- **Example: where is the definition of VSMALL**
- **grep:**
  - `grep -i vsmall $FOAM_SRC/finiteVolume`
    - nothing interesting...
- **find and grep:**
  - `find $FOAM_SRC -iname \*.[CH] | xargs grep VSMALL`
    - too much information...
- **find and grep, but filtering out some information:**
  - `find $FOAM_SRC -iname \*.[CH] | xargs grep VSMALL | grep -v lnInclude | grep const`

1:Same as previous command

2:Exclude results from lnInclude

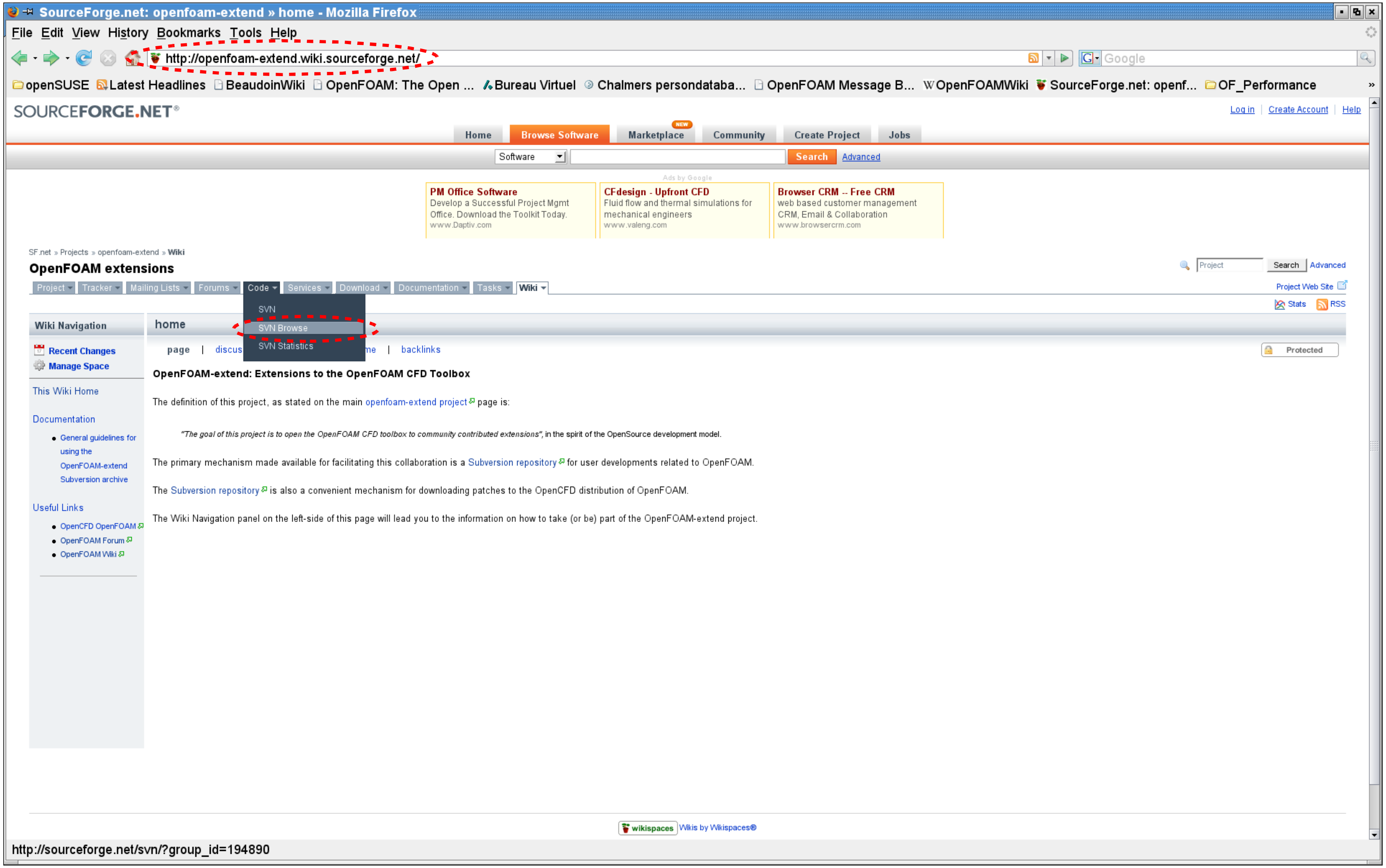
3: Show only the constant definitions

NB: xargs: very useful command

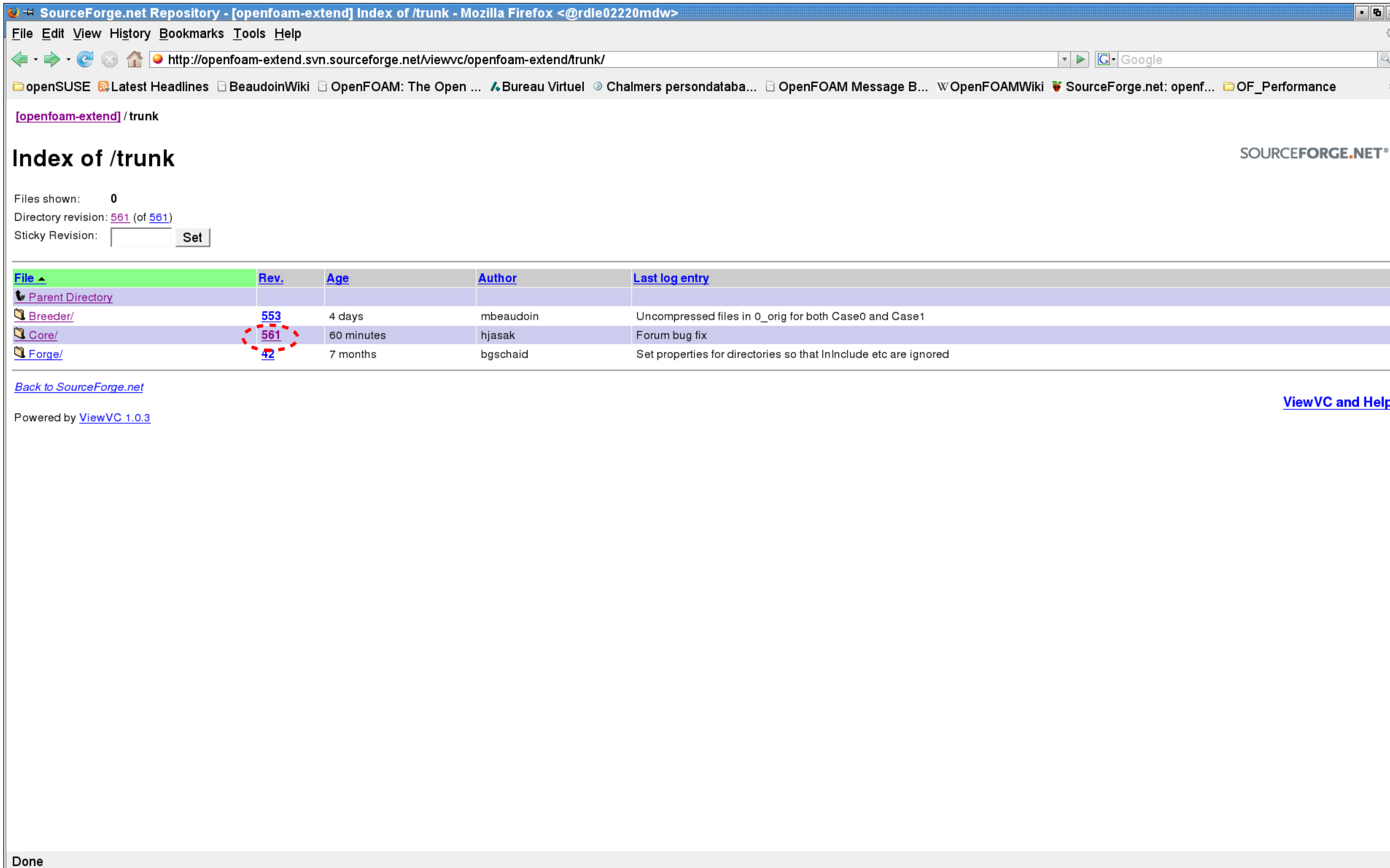
Modifying files: keeping track of versions

- **Using a Revision Control System helps you keep track of your file modifications through time.**
  - copy file1.C file1.C.backup
    - obvious limitations, but better than nothing
  - RCS:
    - Usually available and pre-installed on most Unix systems.
    - Basic functionality. Per directory archive
    - ci, co, rcsdiff
    - A nice introduction:
      - https://agave.garden.org/~aaronh/rcs/tichy1985rcs/html/
  - Subversion for team collaboration
    - http://en.wikipedia.org/wiki/Subversion_(software)
  - International collaboration: openfoam-extend on SourceForge.net :
    - http://openfoam-extend.wiki.sourceforge.net/
    - http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/

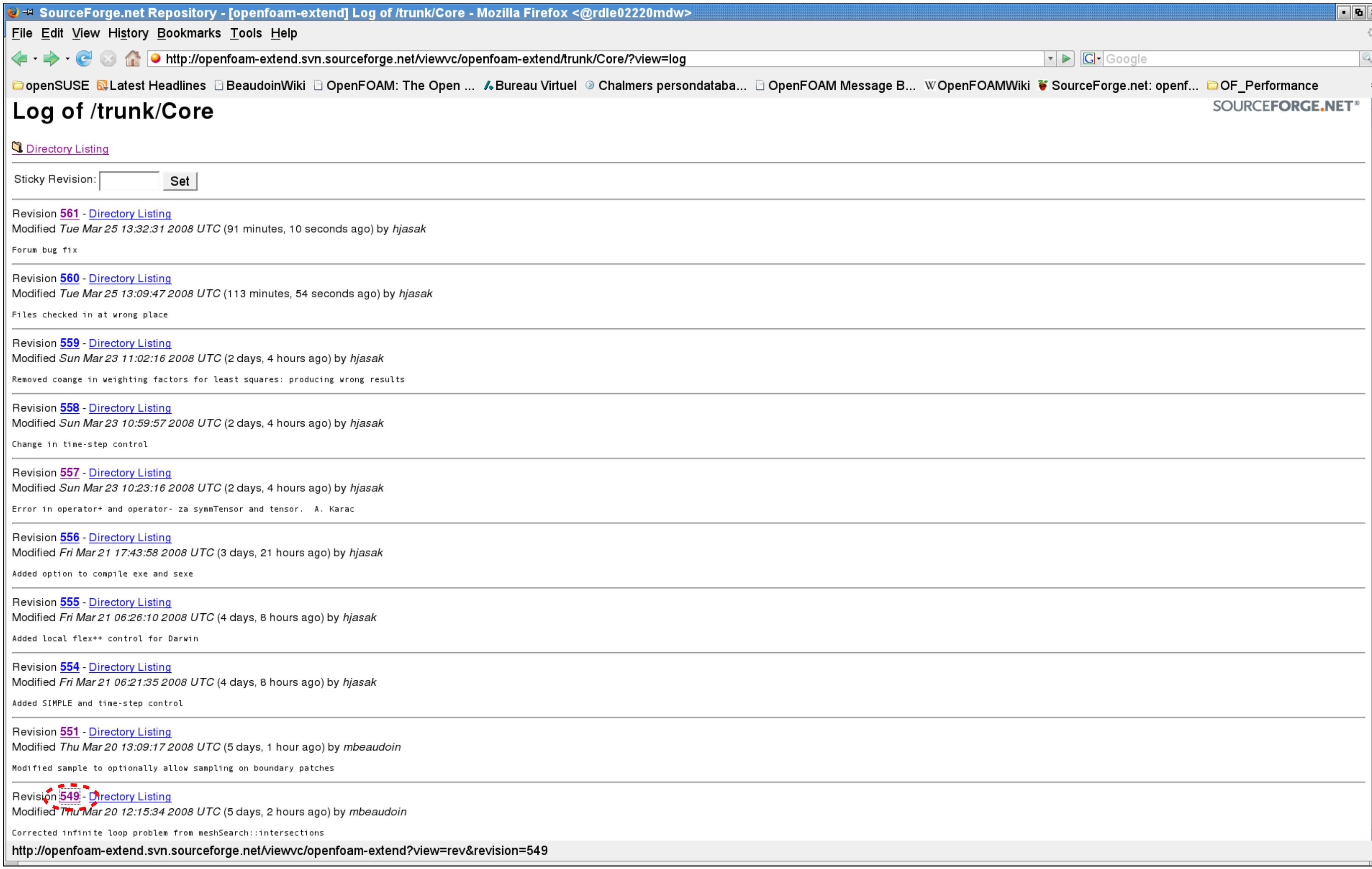
Browsing revisions from a Subversion repository (1/5)



Browsing revisions from a Subversion repository (2/5)



Browsing revisions from a Subversion repository (3/5)


SourceForge.net Repository - [openfoam-extend] Log of /trunk/Core - Mozilla Firefox <@rdle02220mdw>

File Edit View History Bookmarks Tools Help

http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/Core/?view=log

openSUSE Latest Headlines BeaudoinWiki OpenFOAM: The Open ... Bureau Virtuel Chalmers persondataba... OpenFOAM Message B... OpenFOAMWiki SourceForge.net: openf... OF_Performance

Log of /trunk/Core

SOURCEFORGE.NET®

Directory Listing

Sticky Revision: Set

Revision 561 - Directory Listing
Modified *Tue Mar 25 13:32:31 2008 UTC* (91 minutes, 10 seconds ago) by *hjasak*

`Forum bug fix`

Revision 560 - Directory Listing
Modified *Tue Mar 25 13:09:47 2008 UTC* (113 minutes, 54 seconds ago) by *hjasak*

`Files checked in at wrong place`

Revision 559 - Directory Listing
Modified *Sun Mar 23 11:02:16 2008 UTC* (2 days, 4 hours ago) by *hjasak*

`Removed coange in weighting factors for least squares: producing wrong results`

Revision 558 - Directory Listing
Modified *Sun Mar 23 10:59:57 2008 UTC* (2 days, 4 hours ago) by *hjasak*

`Change in time-step control`

Revision 557 - Directory Listing
Modified *Sun Mar 23 10:23:16 2008 UTC* (2 days, 4 hours ago) by *hjasak*

`Error in operator+ and operator- za symmTensor and tensor.  A. Karac`

Revision 556 - Directory Listing
Modified *Fri Mar 21 17:43:58 2008 UTC* (3 days, 21 hours ago) by *hjasak*

`Added option to compile exe and sexe`

Revision 555 - Directory Listing
Modified *Fri Mar 21 06:26:10 2008 UTC* (4 days, 8 hours ago) by *hjasak*

`Added local flex++ control for Darwin`

Revision 554 - Directory Listing
Modified *Fri Mar 21 06:21:35 2008 UTC* (4 days, 8 hours ago) by *hjasak*

`Added SIMPLE and time-step control`

Revision 551 - Directory Listing
Modified *Thu Mar 20 13:09:17 2008 UTC* (5 days, 1 hour ago) by *mbeaudoin*

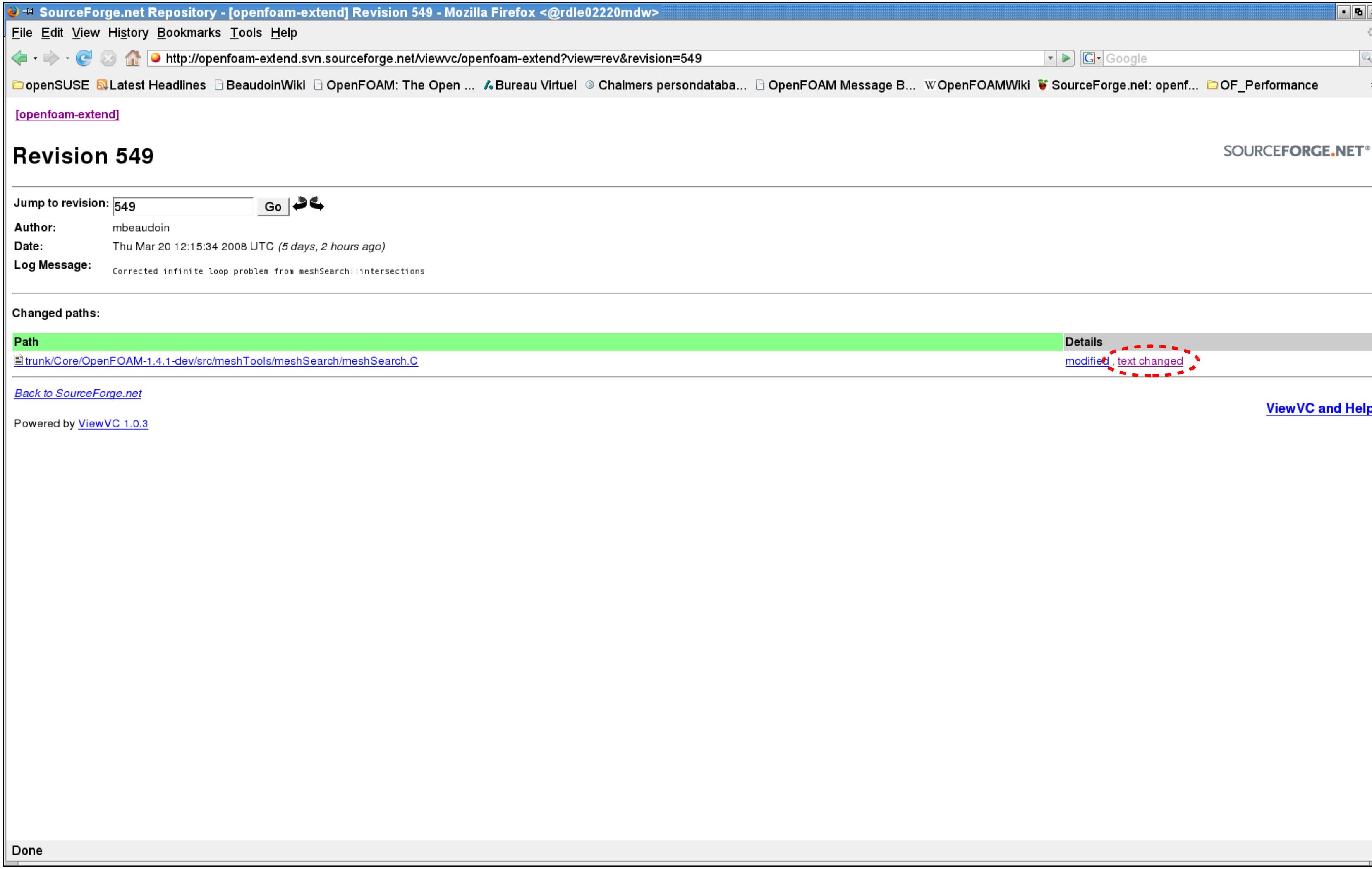
`Modified sample to optionally allow sampling on boundary patches`

Revision 549 - Directory Listing
Modified *Thu Mar 20 12:15:34 2008 UTC* (5 days, 2 hours ago) by *mbeaudoin*

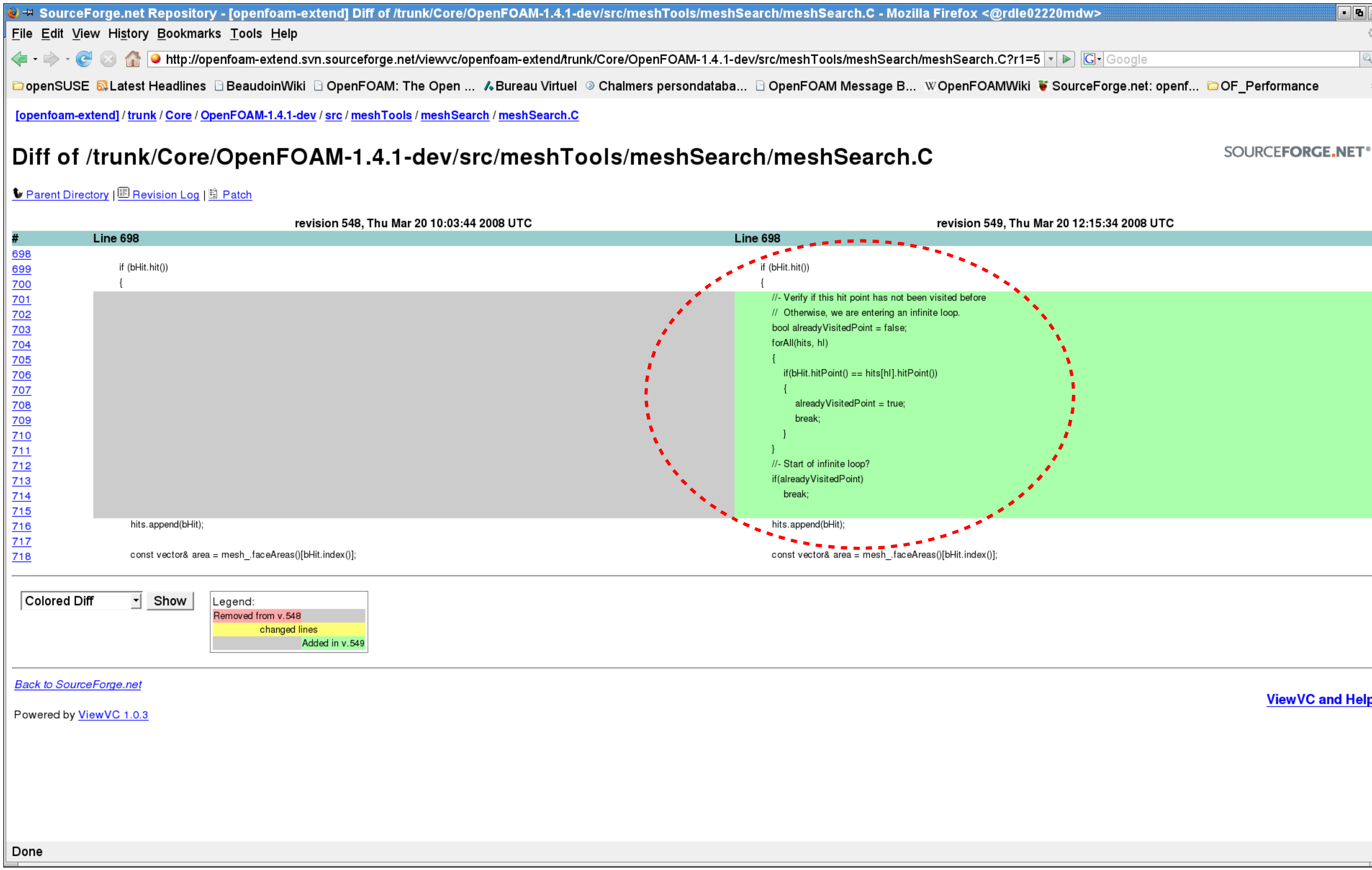
`Corrected infinite loop problem from meshSearch::intersections`

http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend?view=rev&revision=549

Browsing revisions from a Subversion repository (4/5)



Browsing revisions from a Subversion repository (5/5)



How to get help

- **OpenFOAM Forum**
  - http://openfoam.cfd-online.com/cgi-bin/forum/discus.cgi
  - Try searching the Forum first!
    - Your answer might already be there

- **OpenFOAM Wiki**
  - http://openfoamwiki.net/index.php/Main_Page
  - How-To do things:
    - http://openfoamwiki.net/index.php/Main_HowTos