

***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

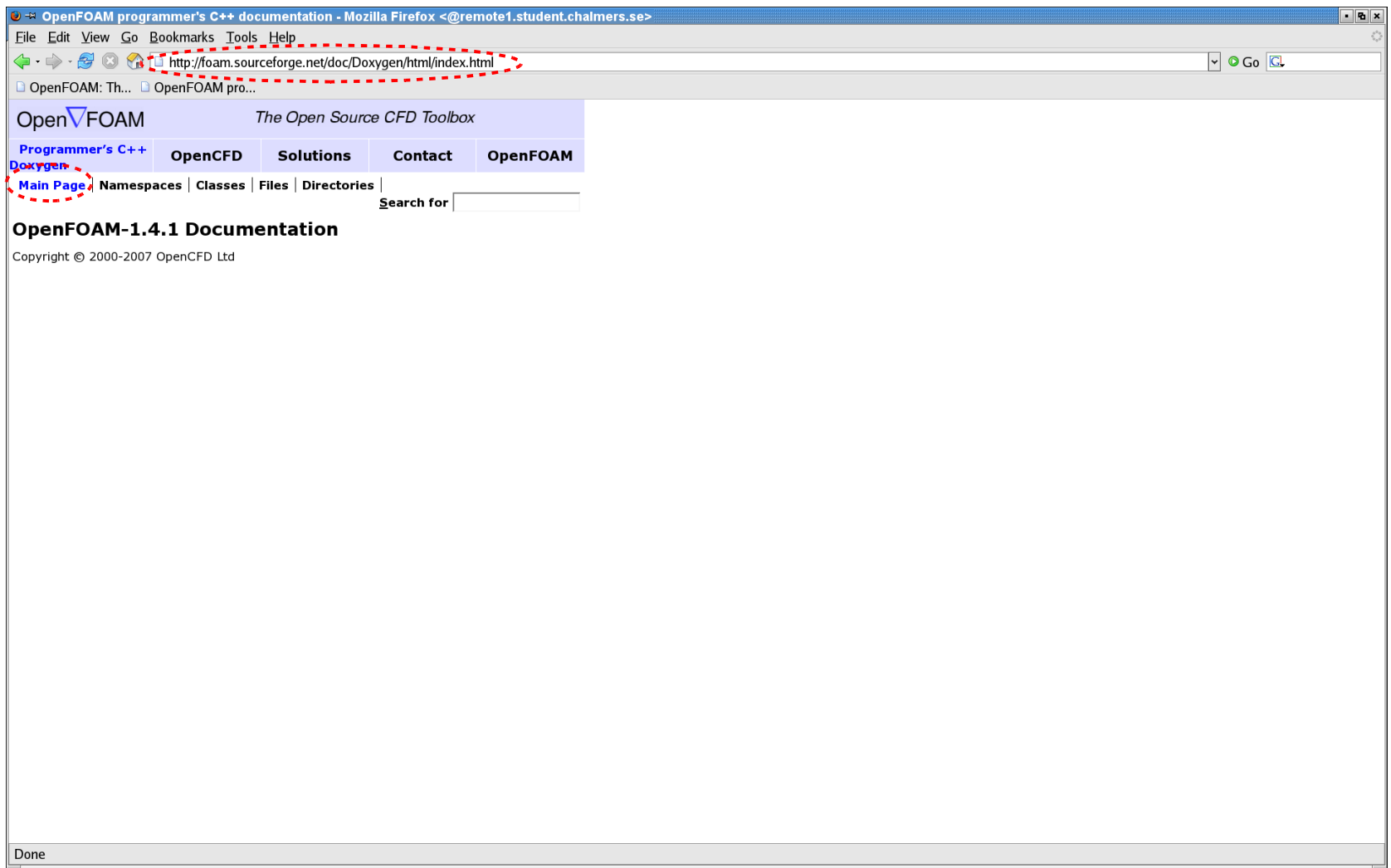
- How to navigate the file revision history using revision control tools.

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)

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/namespaces.html

OpenFOAM: Th... OpenFOAM pro...

OpenFOAM The Open Source CFD Toolbox

Programmer's C++ OpenCFD Solutions Contact OpenFOAM

Doxygen

Main Page Namespaces Classes Files Directories Search for

Namespace List | Namespace Members |

OpenFOAM-1.4.1 Namespace List

Here is a list of all namespaces with brief descriptions:

- Foam
- Foam::compressible
- Foam::compressible::LESmodels
- Foam::compressible::turbulenceModels
- Foam::debug
- Foam::fv
- Foam::fvc
- Foam::fvn
- Foam::LESmodels
- Foam::limitFuncs
- Foam::ListListOps
- Foam::mathematicalConstant
- Foam::meshTools
- Foam::MULES
- Foam::PstreamGlobals
- Foam::resError
- Foam::triSurfaceTools
- Foam::turbulenceModels
- Foam::Unix
- Foam::viscosityModels

Copyright © 2000-2007 OpenCFD Ltd

http://foam.sourceforge.net/doc/Doxygen/html/namespaceFoam_1_1mathematicalConstant.html

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

Alphabetical List Class List Class Hierarchy Class Members

Search for

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	Field (Foam)	meshCutAndRemove (Foam)	removePoints (Foam)
aC10H7CH3 (Foam)	FieldField (Foam)	meshCutSurface (Foam)	repatchPolyTopoChanger (Foam)
accessOp (Foam)	FieldMapper (Foam)	meshCutter (Foam)	reuseTmp (Foam)
addPatchCellLayer (Foam)	fieldToCell (Foam)	meshEdgeCuts (Foam)	reuseTmp< TypeR, TypeR > (Foam)
algebraicPairGAMGAgglomeration (Foam)	FIFOStack (Foam)	MeshObject (Foam)	reuseTmpDimensionedField (Foam)
andEqOp (Foam)	fileDiffusivity (Foam)	meshSearch (Foam)	reuseTmpDimensionedField< TypeR, TypeR, GeoMesh > (Foam)
andEqOp2 (Foam)	fileName (Foam)	meshToMesh (Foam)	reuseTmpFieldField (Foam)
andOp (Foam)	fileStat (Foam)	meshToMesh::patchFieldInterpolator (Foam)	reuseTmpFieldField< Field, TypeR, TypeR > (Foam)
andOp2 (Foam)	filteredLinear2Limiter (Foam)	meshTriangulation (Foam)	reuseTmpGeometricField (Foam)
andOp3 (Foam)	filteredLinear2VLimiter (Foam)	MeshWave (Foam)	reuseTmpGeometricField< TypeR, TypeR, PatchField, GeoMesh > (Foam)
angularOscillatingDisplacementPointPatchVectorField (Foam)	filteredLinearLimiter (Foam)	messageStream (Foam)	reuseTmpTmp (Foam)
angularOscillatingVelocityPointPatchVectorField (Foam)	fixedEnthalpyFvPatchScalarField (Foam)	midPoint (Foam)	reuseTmpTmp< TypeR, Type1, Type12, TypeR > (Foam)
anisotropicFilter (Foam)	fixedFluxBuoyantPressureFvPatchScalarField (Foam)	minEqOp (Foam)	reuseTmpTmp< TypeR, TypeR, TypeR, Type2 > (Foam)
APIdiffCoefFunc (Foam)	fixedFluxPressureFvPatchScalarField (Foam)	minEqOp2 (Foam)	reuseTmpTmp< TypeR, TypeR, TypeR, TypeR > (Foam)
Ar (Foam)	fixedGradientFvPatchField (Foam)	MinmodLimiter (Foam)	reuseTmpTmpDimensionedField (Foam)
argList (Foam)	FixedList (Foam)	minModOp (Foam)	reuseTmpTmpDimensionedField< TypeR, Type1, Type12, TypeR, GeoMesh > (Foam)
argList::initValidTables (Foam)	FixedList::Hash (Foam)	minModOp2 (Foam)	reuseTmpTmpDimensionedField< TypeR, TypeR, TypeR, Type2, GeoMesh > (Foam)
ArrheniusReactionRate (Foam)	fixedNormalSlipFvPatchField (Foam)	minModOp3 (Foam)	reuseTmpTmpDimensionedField< TypeR, TypeR, TypeR, TypeR, GeoMesh > (Foam)
arrow2D (Foam)	fixedUnburntEnthalpyFvPatchScalarField (Foam)	minOp (Foam)	reuseTmpTmpFieldField (Foam)
atomicWeightTable (Foam)	fixedValueFvPatchField (Foam)	minOp2 (Foam)	reuseTmpTmpFieldField< Field, TypeR, Type1, Type12, TypeR > (Foam)
atomicWeightTable::atomicWeight (Foam)	fixedValueFvsPatchField (Foam)	minOp3 (Foam)	reuseTmpTmpFieldField< Field, TypeR, TypeR, TypeR, Type2 > (Foam)
atomizationModel (Foam)	fixedValuePointPatchField (Foam)	minusEqOp (Foam)	reuseTmpTmpFieldField< Field, TypeR, TypeR, TypeR, TypeR > (Foam)
attachDetach (Foam)	flux (Foam)	minusEqOp2 (Foam)	reuseTmpTmpGeometricField (Foam)
attachPolyTopoChanger (Foam)	flux< scalar > (Foam)	minusOp (Foam)	reuseTmpTmpGeometricField< TypeR, Type1, Type12, TypeR, PatchField, GeoMesh > (Foam)

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

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...

OpenFOAM The Open Source CFD Toolbox

Programmer's C++ Doxygen OpenCFD Solutions Contact OpenFOAM

Main Page | Namespaces | Classes | **Files** | Directories |

File List | File Members |

Search for

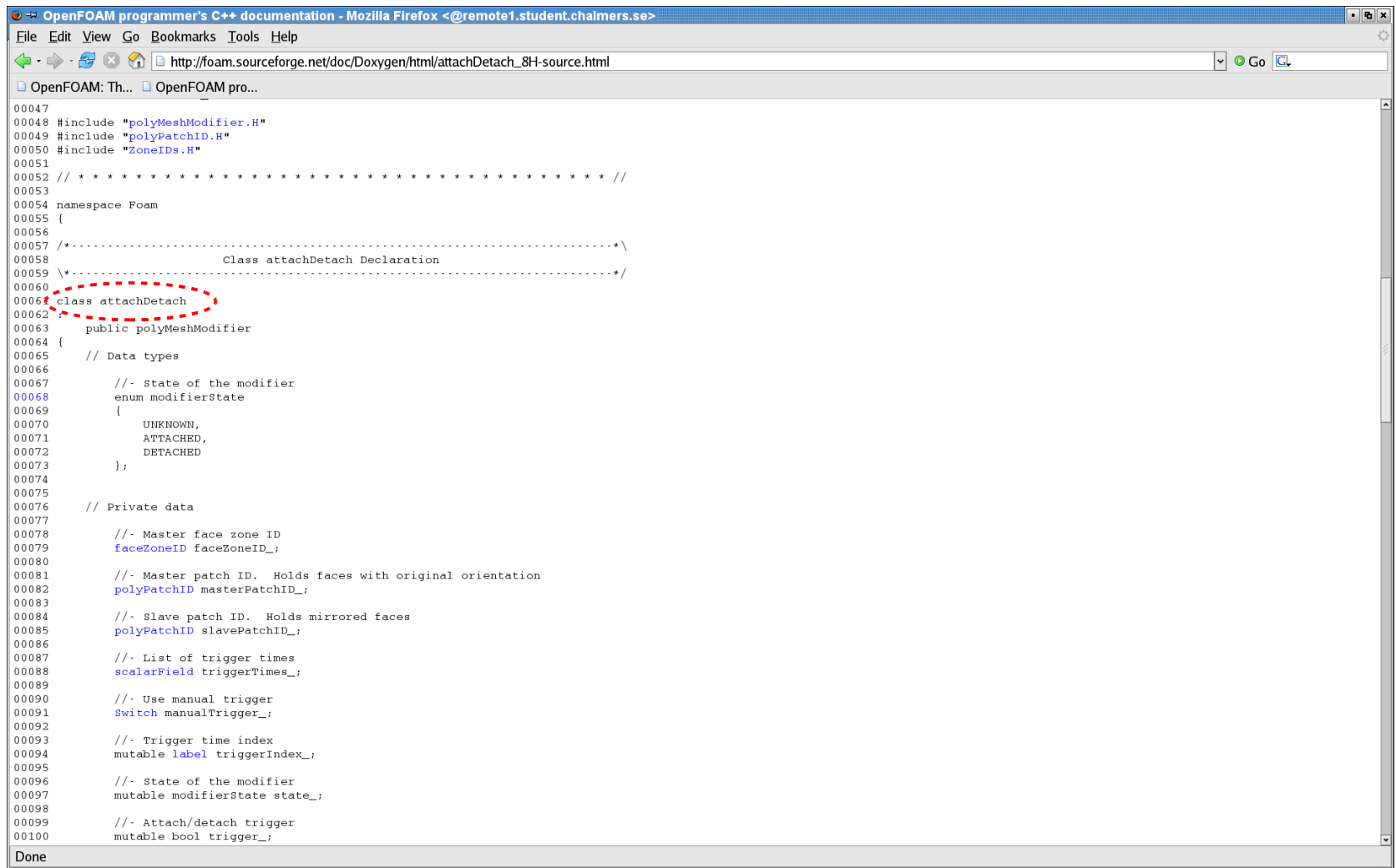
OpenFOAM-1.4.1 File List

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)



```
00047
00048 #include "polyMeshModifier.H"
00049 #include "polyPatchID.H"
00050 #include "zoneIDs.H"
00051
00052 // * * * * *
00053
00054 namespace Foam
00055 {
00056
00057 /*-----\
00058          Class attachDetach Declaration
00059 \*-----*/
00060
00061 class attachDetach
00062 {
00063     public polyMeshModifier
00064     {
00065         // Data types
00066
00067         //- State of the modifier
00068         enum modifierState
00069         {
00070             UNKNOWN,
00071             ATTACHED,
00072             DETACHED
00073         };
00074
00075
00076         // Private data
00077
00078         //- Master face zone ID
00079         faceZoneID faceZoneID_;
00080
00081         //- Master patch ID. Holds faces with original orientation
00082         polyPatchID masterPatchID_;
00083
00084         //- Slave patch ID. Holds mirrored faces
00085         polyPatchID slavePatchID_;
00086
00087         //- List of trigger times
00088         scalarField triggerTimes_;
00089
00090         //- Use manual trigger
00091         Switch manualTrigger_;
00092
00093         //- Trigger time index
00094         mutable label triggerIndex_;
00095
00096         //- State of the modifier
00097         mutable modifierState state_;
00098
00099         //- Attach/detach trigger
00100         mutable bool trigger_;
00101
00102     };
00103
00104 };
00105
00106 }
```

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

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/globals_0x6f.html#index_o

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

File List | File Members

All | Functions | Variables | Typedefs | Enumerations | Enumerator | Defines

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

Here is a list of all file members with links to the files they belong to:

- o -

- [oct\(\)](#) : [IOstream.H](#)
- [oneThirdI](#) : [sphericalTensor.H](#)
- [oneThirdI2D](#) : [sphericalTensor2D.H](#)
- [Op](#) : [ops.H](#)
- [operator &\(\)](#) : [DimensionedFieldFunctions.H](#), [FieldFunctions.H](#), [fvMatrix.H](#), [FieldFunctions.H](#), [DimensionedFieldFunctions.H](#), [dimensionedType.H](#), [GeometricFieldFunctions.H](#), [dimensionedType.H](#), [FieldFieldFunctions.H](#), [complexVectorI.H](#), [FieldFieldFunctions.H](#), [DiagTensorI.H](#), [FieldFieldFunctions.H](#), [DiagTensorI.H](#), [FieldFieldFunctions.H](#), [DiagTensorI.H](#), [FieldFieldFunctions.H](#), [SphericalTensorI.H](#), [SphericalTensor2DI.H](#), [wordI.H](#), [SymmTensorI.H](#), [TensorI.H](#), [Tensor2DI.H](#), [VectorI.H](#), [Vector2DI.H](#)
- [operator &&\(\)](#) : [VectorSpaceI.H](#), [FieldFunctions.H](#), [DimensionedFieldFunctions.H](#), [GeometricFieldFunctions.H](#), [dimensionedType.H](#), [FieldFieldFunctions.H](#), [SphericalTensorI.H](#), [FieldFieldFunctions.H](#), [SymmTensorI.H](#), [TensorI.H](#), [Tensor2DI.H](#)
- [operator *\(\)](#) : [janafThermo.H](#), [DimensionedFieldFunctions.H](#), [VectorSpaceI.H](#), [DimensionedFieldFunctions.H](#), [ZeroI.H](#), [DimensionedFieldFunctions.H](#), [ZeroI.H](#), [DimensionedFieldFunctions.H](#), [fvMatrix.H](#), [perfectGasI.H](#), [dimensionedTensor.H](#), [dimensionedScalar.H](#), [dimensionedSymmTensor.H](#), [dimensionedTensor.H](#), [specieI.H](#), [GeometricFieldFunctions.H](#), [eConstThermo.H](#), [GeometricFieldFunctions.H](#), [eConstThermoI.H](#), [GeometricFieldFunctions.H](#), [hConstThermo.H](#), [GeometricFieldFunctions.H](#), [hConstThermo.H](#), [GeometricFieldFunctions.H](#), [janafThermo.H](#), [GeometricFieldFunctions.H](#), [specieThermo.H](#), [constTransport.H](#), [dimensionedType.H](#), [constTransport.H](#), [sutherlandTransport.H](#), [dimensionedType.H](#), [errorEstimate.H](#), [simpleMatrix.H](#), [dimensionedType.H](#), [point2DI.H](#), [dimensionedType.H](#), [FieldFieldFunctions.H](#), [Matrix.H](#), [FieldFieldFunctions.H](#), [complexI.H](#), [complexVectorI.H](#), [errorEstimate.H](#), [SymmTensorI.H](#), [TensorI.H](#), [FieldFunctions.H](#), [DimensionedFieldFunctions.H](#), [FieldFunctions.H](#), [Tensor2DI.H](#), [FieldFunctions.H](#), [DimensionedFieldFunctions.H](#), [FieldFunctions.H](#), [DimensionedFieldFunctions.H](#), [FieldFunctions.H](#)
- [operator !=\(\)](#) : [VectorSpaceI.H](#), [cellI.H](#), [colourI.H](#), [cellModelI.H](#), [edgeI.H](#), [point2DI.H](#), [triFaceI.H](#), [objectMapI.H](#)
- [operator +\(\)](#) : [ZeroI.H](#), [PtrList.H](#), [fvMatrix.H](#), [errorEstimate.H](#), [dimensionedScalar.H](#), [fvMatrix.H](#), [dimensionedScalar.H](#), [errorEstimate.H](#), [fvMatrix.H](#), [errorEstimate.H](#), [GeometricFieldFunctions.H](#), [fvMatrix.H](#), [GeometricFieldFunctions.H](#), [perfectGasI.H](#)

Done

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

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/dirs.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

OpenFOAM-1.4.1 Directories

This directory hierarchy is sorted roughly, but not completely, alphabetically:

- src
 - dynamicFvMesh
 - dynamicFvMesh
 - dynamicInkJetFvMesh
 - dynamicMotionSolverFvMesh
 - include
 - staticFvMesh
 - dynamicMesh
 - attachDetach
 - boundaryMesh
 - boundaryPatch
 - fvMeshAdder
 - fvMeshDistribute
 - layerAdditionRemoval
 - meshCut
 - cellCuts
 - cellLooper
 - directions
 - directionInfo
 - edgeVertex
 - meshModifiers
 - boundaryCutter
 - meshCutAndRemove
 - meshCutter
 - multiDirRefinement
 - refinementIterator
 - undoableMeshCutter
 - refineCell
 - splitCell
 - wallLayerCells
 - wallNormalInfo
 - motionSmoother
 - polyMeshGeometry
 - motionSolver
 - perfectInterface
 - polyMeshAdder
 - polyTopoChange
 - attachPolyTopoChanger

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

The screenshot shows a Mozilla Firefox browser window with the address bar containing the URL `http://foam.sourceforge.net/doc/Doxygen/html/search.php?query=fvMesh`. The page title is "OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote1.student.chalmers.se>". The browser's menu bar includes "File", "Edit", "View", "Go", "Bookmarks", "Tools", and "Help".

The OpenFOAM website header is visible, featuring the logo "OpenFOAM" and the tagline "The Open Source CFD Toolbox". Navigation tabs include "Programmer's C++ Doxygen", "OpenCFD", "Solutions", "Contact", and "OpenFOAM". A search bar is highlighted with a red dashed circle, containing the text "Search for fvMesh".

The "Search Results" section indicates that 282 documents match the query. The top 14 results are listed as follows:

1. [fvMeshAdder.H](#)
Matches: **fvmeshadder**(1) **fvmeshadder.h**(1) **fvmeshadder.h**(1) **fvmeshes**(1) **fvmeshadder.c**(1) **fvmeshaddertemplates.c**(1)
2. [fvMesh.H](#)
Matches: **fvmesh.h**(1) **fvmesh.h**(1) **fvmesh.c**(1) **fvmeshgeometry.c**(1)
3. [src/dynamicMesh/fvMeshDistribute/fvMeshDistribute.H File Reference](#)
Matches: **fvmeshdistribute.h**(1) **fvmeshdistribute.h**(1) **fvmeshdistribute.c**(1) **fvmeshdistributetemplates.c**(1)
4. [Member Foam::fvMotionSolver::fvMesh_](#)
Matches: **fvmesh**(1) **fvmesh_**(1)
5. [Member Foam::fvMeshMapper::fvMeshMapper](#)
Matches: **fvmesh**(1) **fvmeshmapper**(1)
6. [src/finiteVolume/fvMesh/fvMeshLduAddressing.H File Reference](#)
Matches: **fvmeshlduaddressing.h**(1) **fvmeshlduaddressing.h**(1) **fvmeshlduaddressing.c**(1)
7. [src/finiteVolume/fvMesh/fvMeshSubset/fvMeshSubset.H File Reference](#)
Matches: **fvmeshsubset.h**(1) **fvmeshsubset.h**(1) **fvmeshsubset.c**(1)
8. [fvMeshMapper.H](#)
Matches: **fvmeshmapper.h**(1) **fvmeshmapper.h**(1) **fvmeshmapper.c**(1)
9. [Foam::fvMesh::fvMesh](#)
Matches: **fvmesh**(1)
10. [fvMesh Class Reference](#)
Matches: **fvmesh**(1)
11. [Foam::fvMesh::fvMesh](#)
Matches: **fvmesh**(1)
12. [Foam::fvBoundaryMesh::fvMesh](#)
Matches: **fvmesh**(1)
13. [Foam::fvMesh::fvMesh](#)
Matches: **fvmesh**(1)
14. [fvMeshMapper Class Reference](#)
Matches: **fvmeshmapper**(1)

The status bar at the bottom of the browser window displays "Done".

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

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_1fvBoundaryMesh.html#3fd400a1db107bbcd9e006cf55828e47

OpenFOAM: Th... OpenFOAM pro...

fvBoundaryMesh Class Reference

Inheritance diagram for fvBoundaryMesh:

```
graph BT; fvBoundaryMesh --> fvPatchList;
```

Collaboration diagram for fvBoundaryMesh:

```
graph TD; GeometricField -- magSIPtr_ / phiPtr_ --> fvMesh; polyMesh --> fvMesh; IduMesh --> fvMesh; SlicedGeometricField -- CPtr_ / SFPtr_ / CPtr_ --> fvMesh; fvMeshLduAddressing -- IduPtr_ --> fvMesh; fvMesh -- mesh_ --> fvBoundaryMesh; fvBoundaryMesh -- boundary_ / mesh_ --> fvMesh; fvBoundaryMesh --> fvPatchList; fvPatchList -- ptrs_ --> ListT; fvBoundaryMesh --> fvPatchList;
```

List of all members.

Detailed Description

Definition at line 60 of file `fvBoundaryMesh.H`.

Public Member Functions

- `fvBoundaryMesh (const fvMesh &)`
Construct with zero size.
- `fvBoundaryMesh (const fvMesh &, const polyBoundaryMesh &)`
Construct from `polyBoundaryMesh`.
- `const fvMesh & mesh () const`
Return the mesh reference.

Done

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:

```
graph TD; PrimitivePatchName --> primitivePatch; primitivePatch --> polyPatch; patchIdentifier --> polyPatch; coupledPolyPatch --> polyPatch; directMappedPolyPatch --> polyPatch; emptyPolyPatch --> polyPatch; symmetryPolyPatch --> polyPatch; wallPolyPatch --> polyPatch; wedgePolyPatch --> polyPatch; cyclicPolyPatch --> coupledPolyPatch; processorPolyPatch --> coupledPolyPatch;
```

Collaboration diagram for polyPatch:

```
graph TD; polyPatch --> patchIdentifier; polyPatch --> primitivePatch; polyPatch --> SubList; polyPatch --> polyBoundaryMesh; polyPatch --> boundaryMesh; polyPatch -.-> List;
```

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

```
/*! 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:

```
graph TD
    PublicBase --> Inherited
    ProtectedBase --> Inherited
    PrivateBase --> Inherited
    Templ_int["Templ<int>"] -.-> Inherited
    Templ_int -.-> Templ_T["Templ<T>"]
    Used -.-> Inherited
    Truncated -.-> PublicBase
```

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 for which not all inheritance/containment relations are shown. A graph is truncated if it does not fit within the specified boundaries.

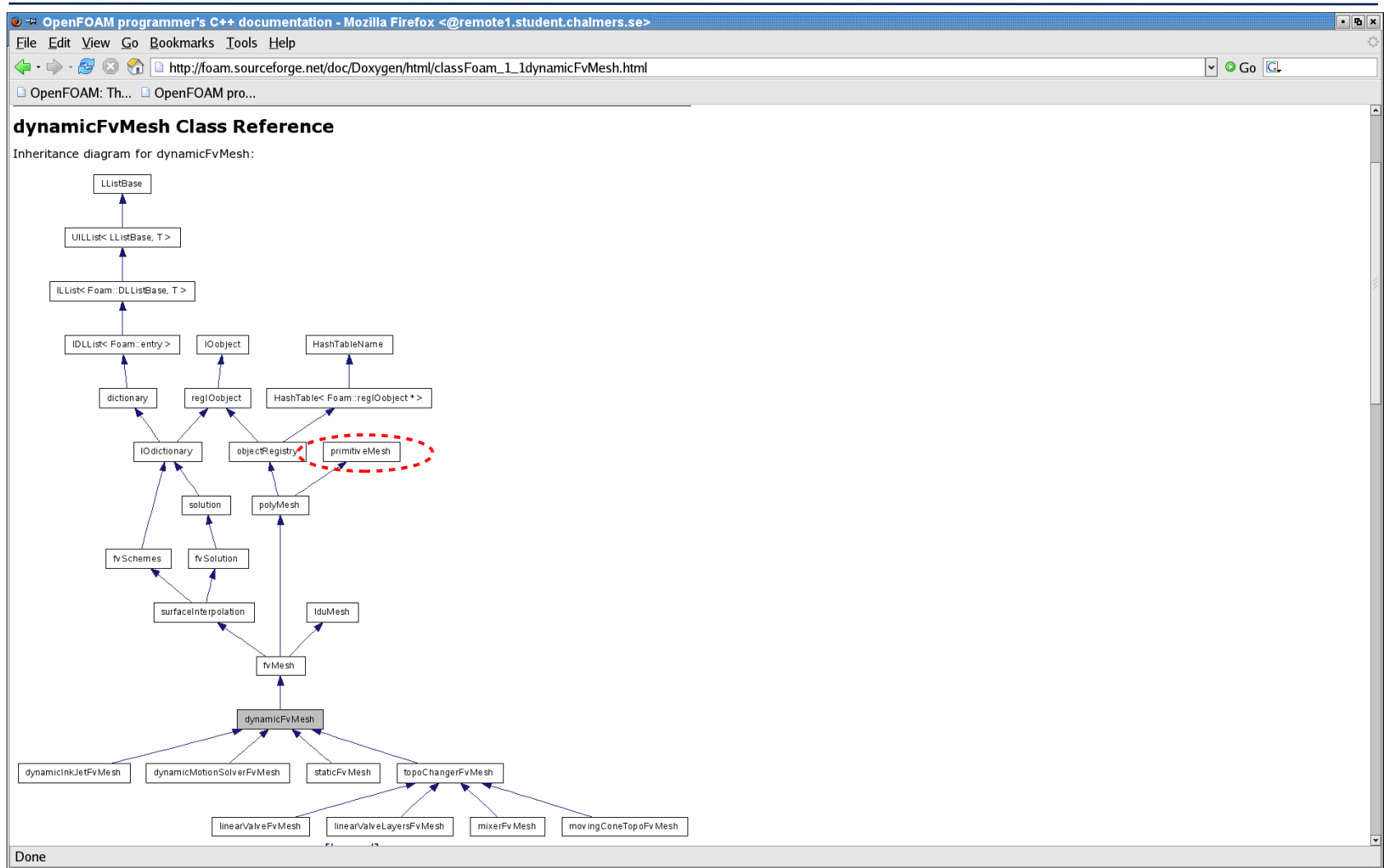
The arrows have the following meaning:

- 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.

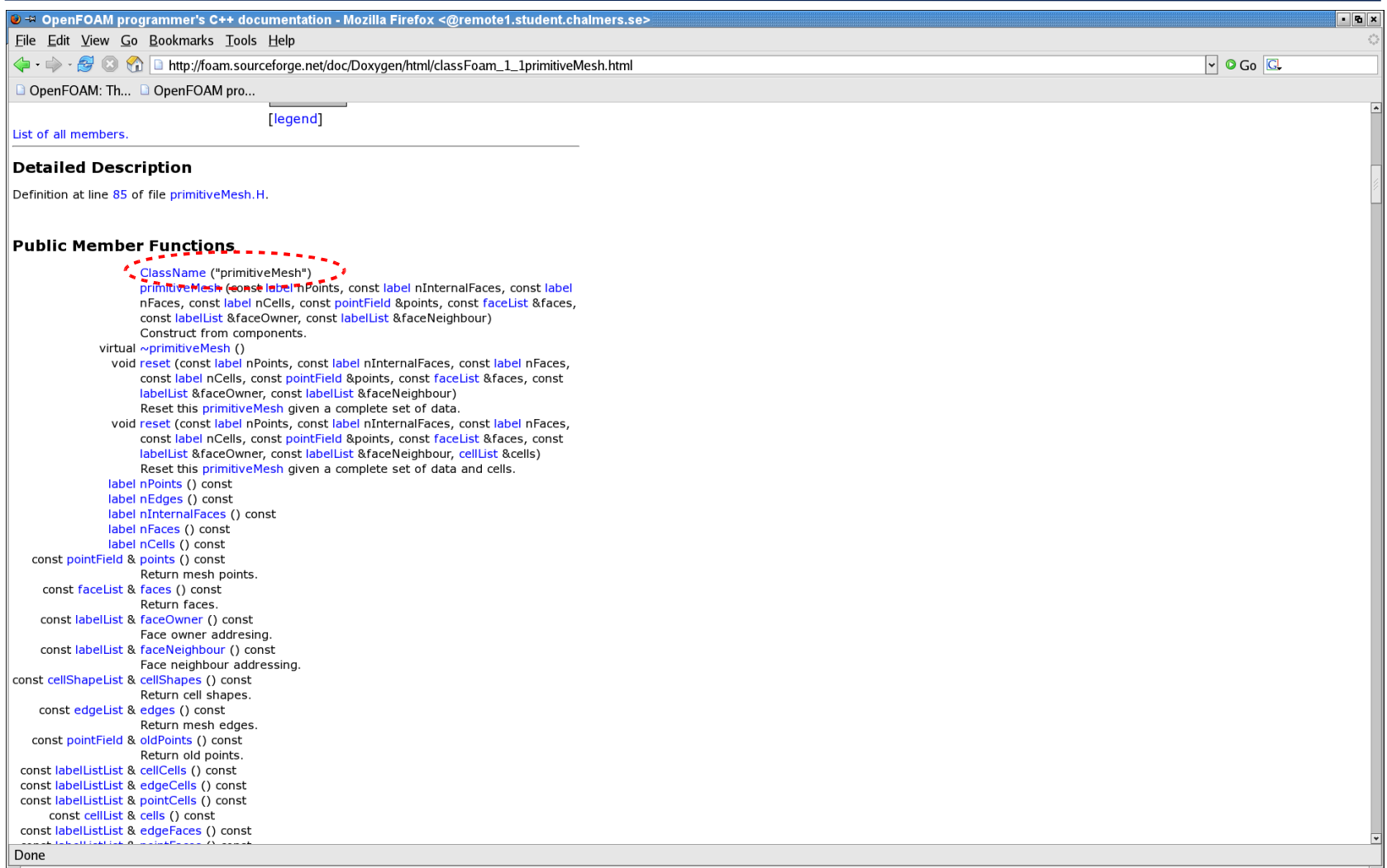
Copyright © 2000-2007 OpenCFD Ltd

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 addressing.
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
```

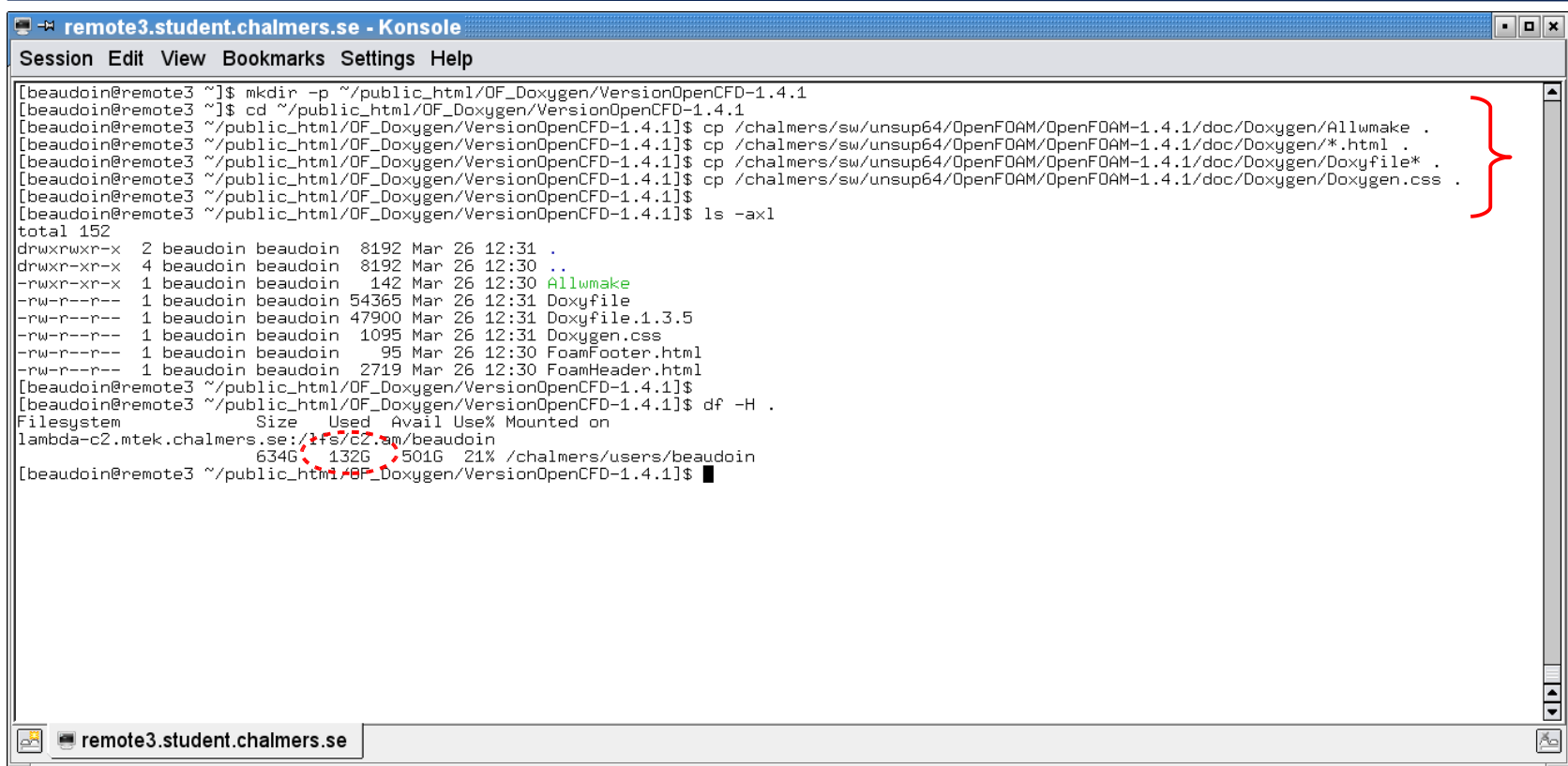
Done

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]$ 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:/fs/c2.ssm/beaudoin
634G  132G  501G  21% /chalmers/users/beaudoin
[beaudoin@remote3 ~/public_html/OF_Doxygen/VersionOpenCFD-1.4.1]$
```

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 -s 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 -s 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

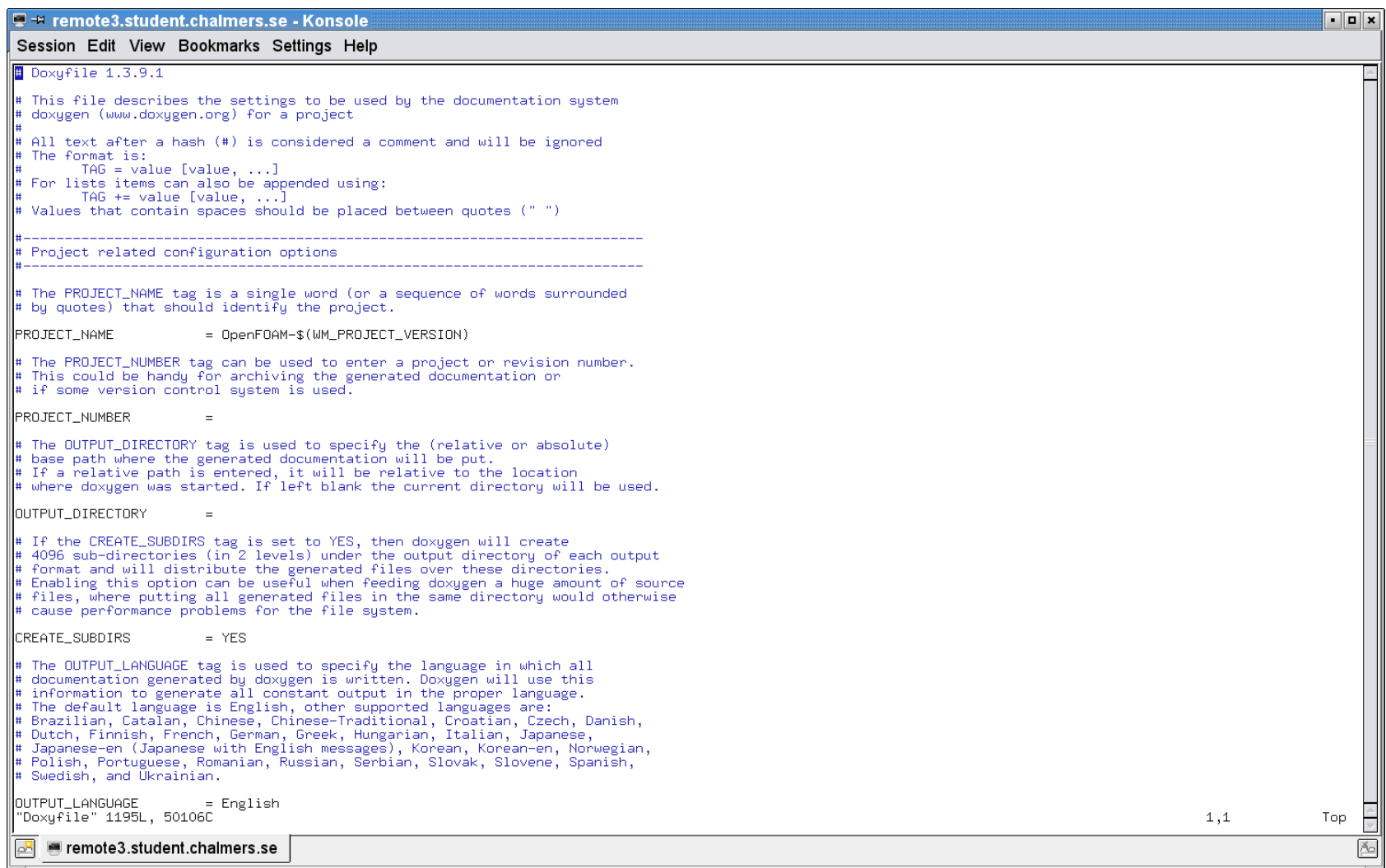
If -s is specified the comments in the config file will be omitted.
If configName is omitted `Doxyfile' will be used as a default.

[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-rw-r--  1 beaudoin beaudoin 47900 Mar 26 12:38 Doxyfile.1.3.5
-rw-rw-r--  1 beaudoin beaudoin 54365 Mar 26 12:40 Doxyfile.1.5.1
-rw-rw-r--  1 beaudoin beaudoin 47900 Mar 26 12:41 Doxyfile.bak
-rw-rw-r--  1 beaudoin beaudoin  1095 Mar 26 12:38 Doxygen.css
-rw-rw-r--  1 beaudoin beaudoin    95 Mar 26 12:38 FoamFooter.html
-rw-rw-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]$ █
```

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)

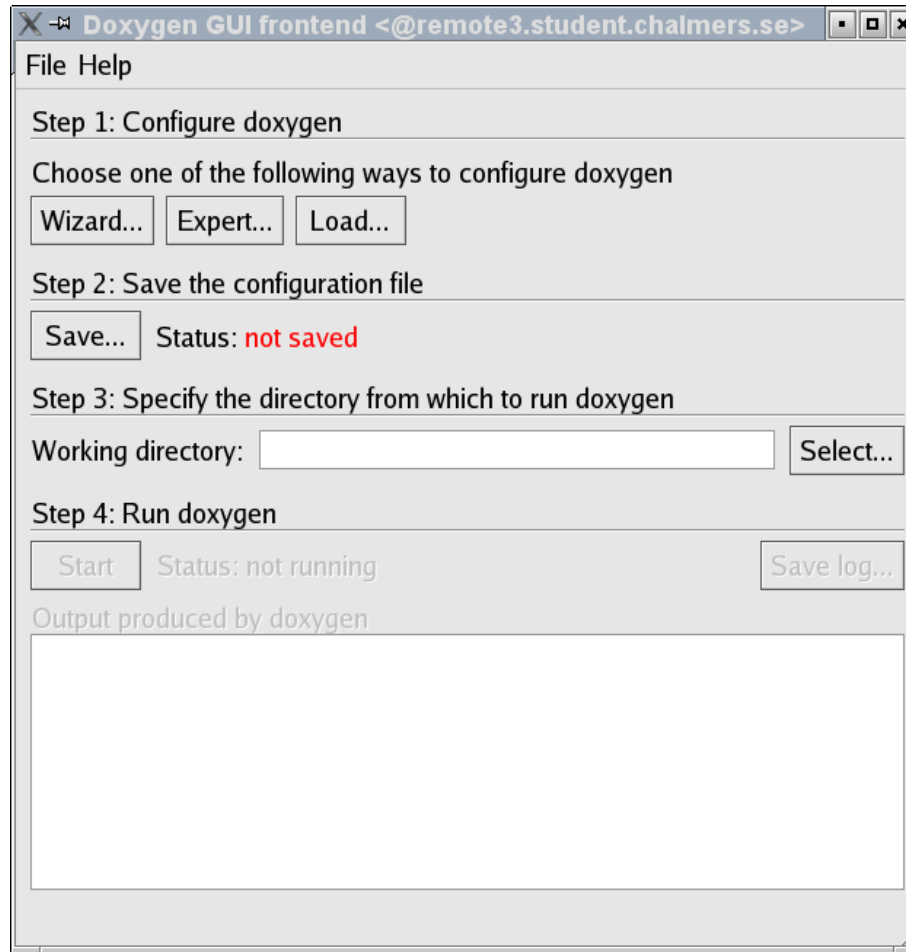
# The PROJECT_NUMBER tag can be used to enter a project or revision number.
# This could be handy for archiving the generated documentation or
# if some version control system is used.
PROJECT_NUMBER         =

# 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      =

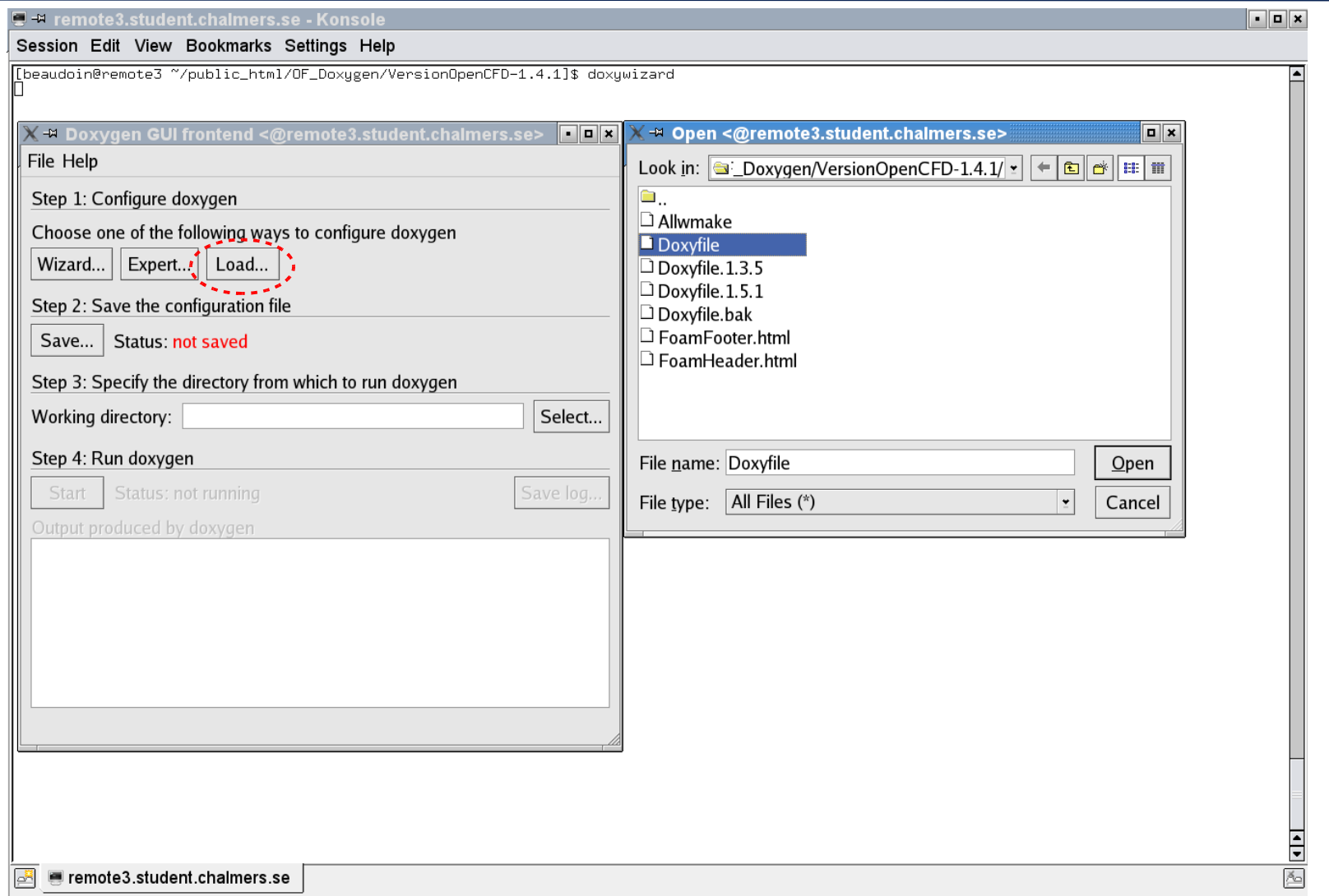
# If the CREATE_SUBDIRS tag is set to YES, then doxygen will create
# 4096 sub-directories (in 2 levels) under the output directory of each output
# format and will distribute the generated files over these directories.
# Enabling this option can be useful when feeding doxygen a huge amount of source
# 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
```

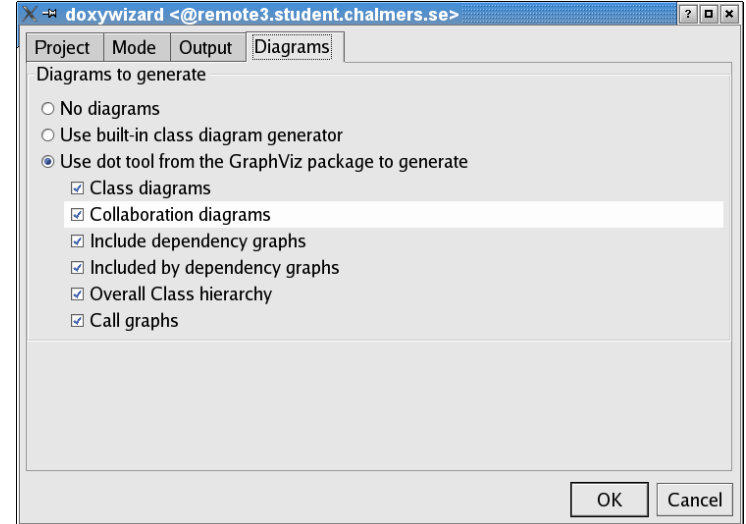
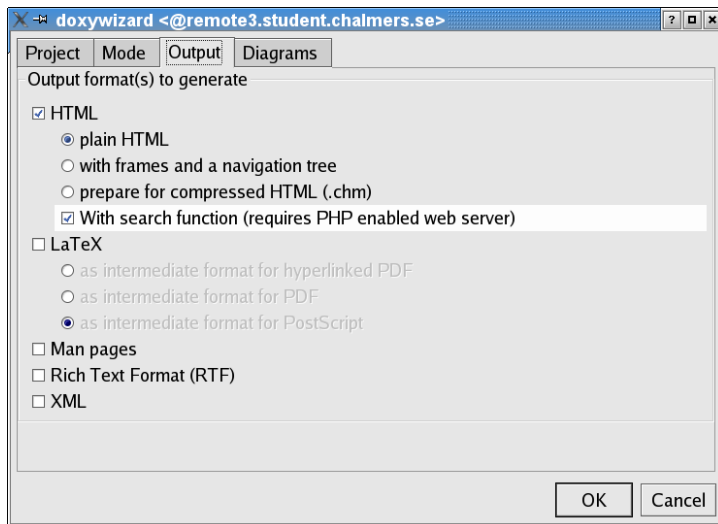
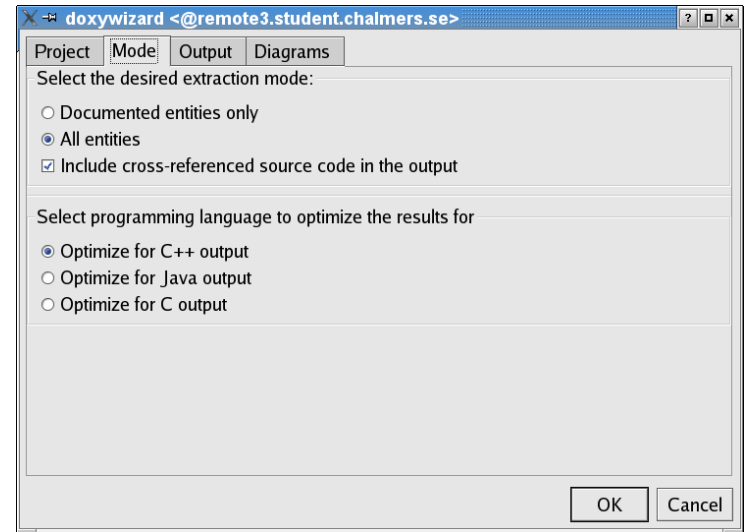
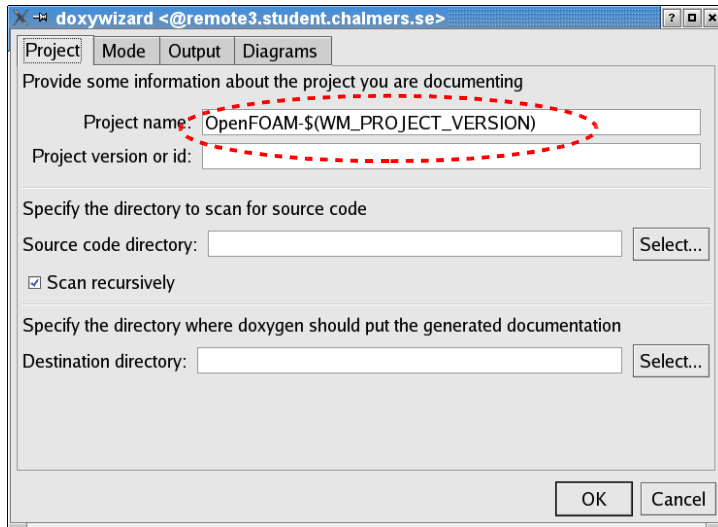
Modifying Doxyfile : using doxywizard



Doxywizard : loading a configuration file



Doxywizard : Wizard mode



Doxywizard : Expert mode (1/9)

doxywizard <@remote3.student.chalmers.se>

Project Build Messages Input Source Browser Index HTML LaTeX RTF Man XML DEF PerlMod Preprocessor External Dot Search

PROJECT_NAME OpenFOAM-\$(WM_PROJECT_VERSION)

PROJECT_NUMBER

OUTPUT_DIRECTORY Folder...

CREATE_SUBDIRS

OUTPUT_LANGUAGE English

USE_WINDOWS_ENCODING

BRIEF_MEMBER_DESC

REPEAT_BRIEF

ABBREVIATE_BRIEF + - *

ALWAYS_DETAILED_SEC

INLINE_INHERITED_MEMB

FULL_PATH_NAMES

STRIP_FROM_PATH + - *

\$(WM_PROJECT_DIR)

STRIP_FROM_INC_PATH + - *

SHORT_NAMES

JAVADOC_AUTOBRIEF

MULTILINE_CPP_IS_BRIEF

DETAILS_AT_TOP

INHERIT_DOCS

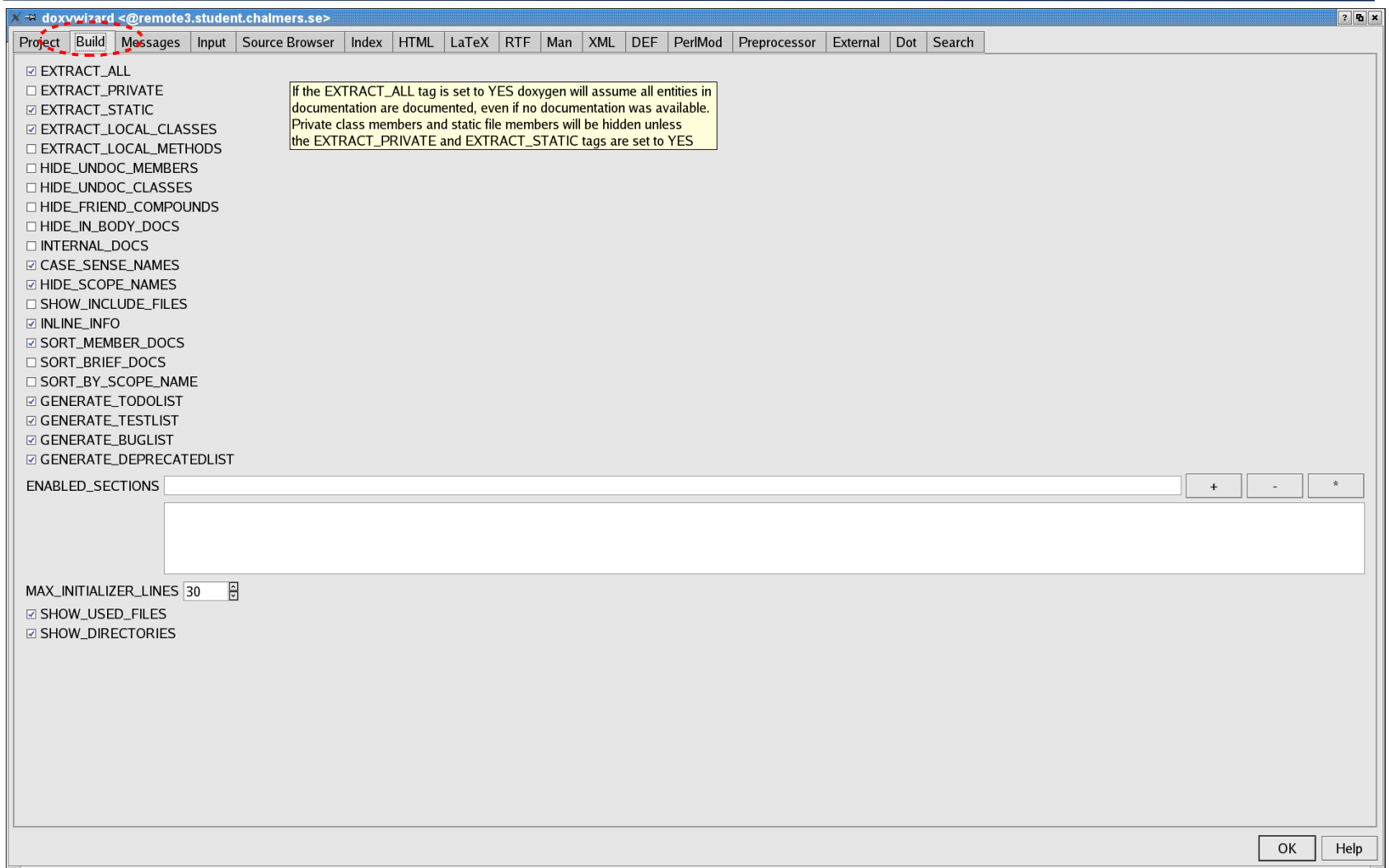
DISTRIBUTE_GROUP_DOC

TAB_SIZE 4

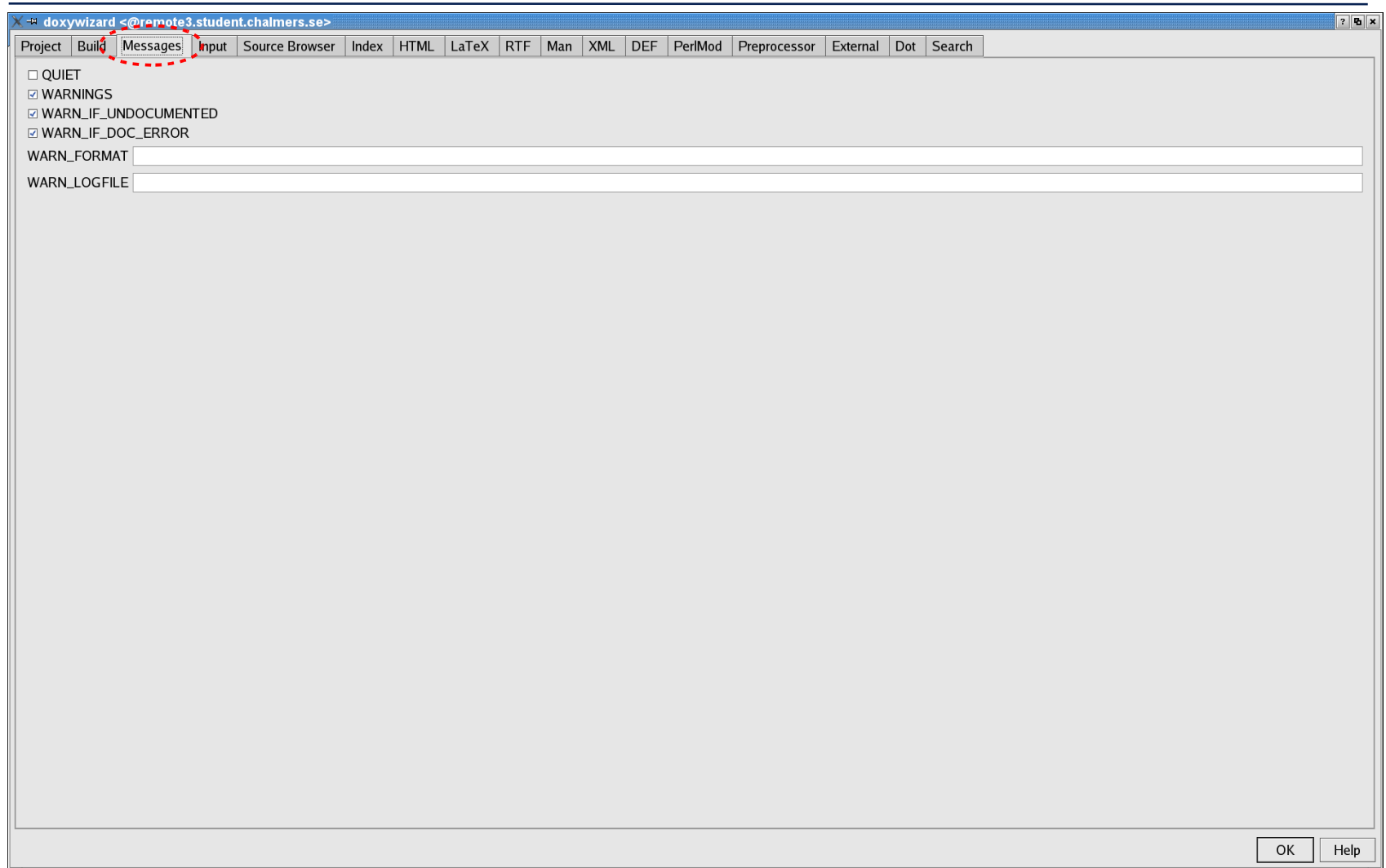
ALIASES + - *

OK Help

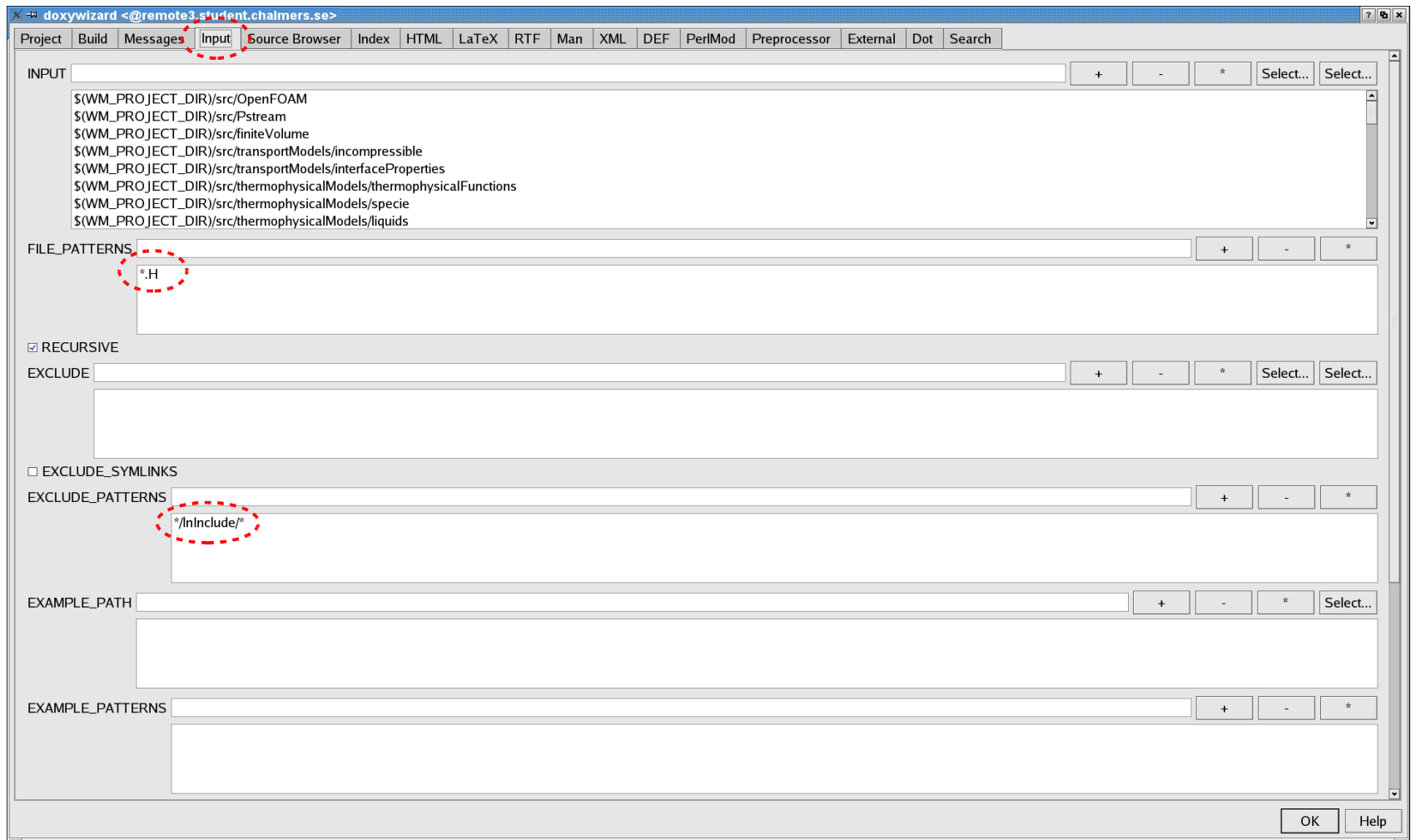
Doxywizard : Expert mode (2/9)



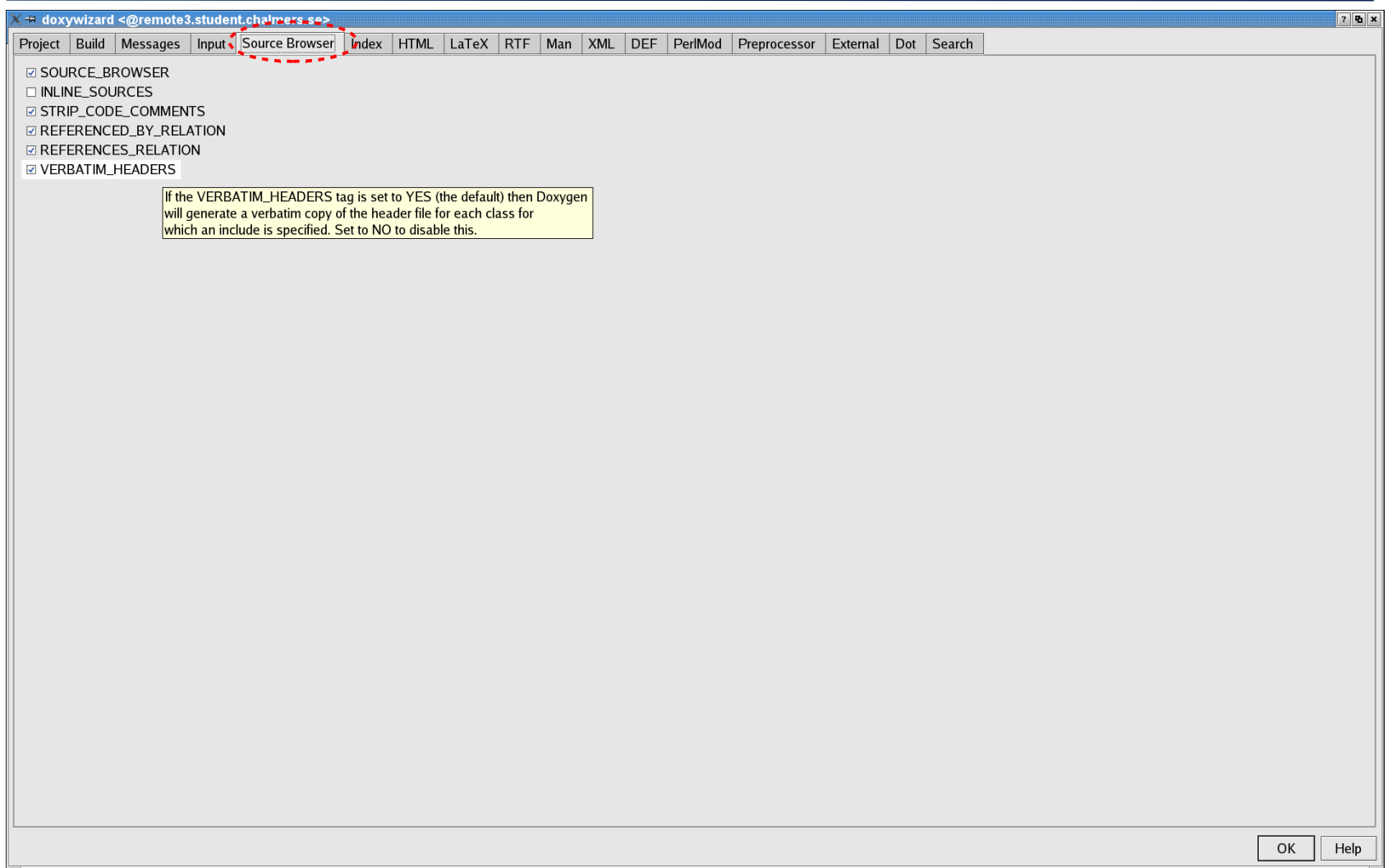
Doxywizard : Expert mode (3/9)



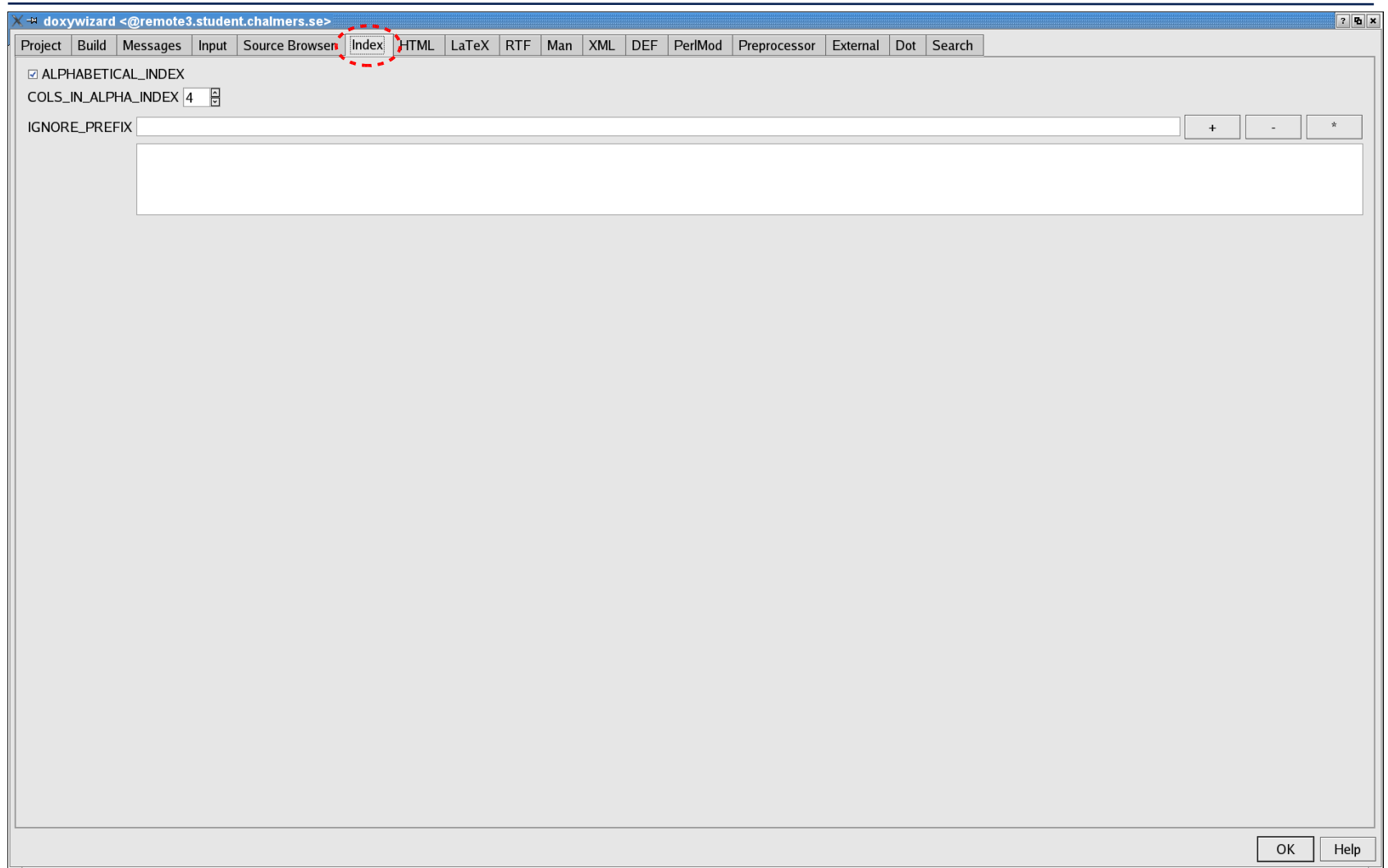
Doxywizard : Expert mode (4/9)



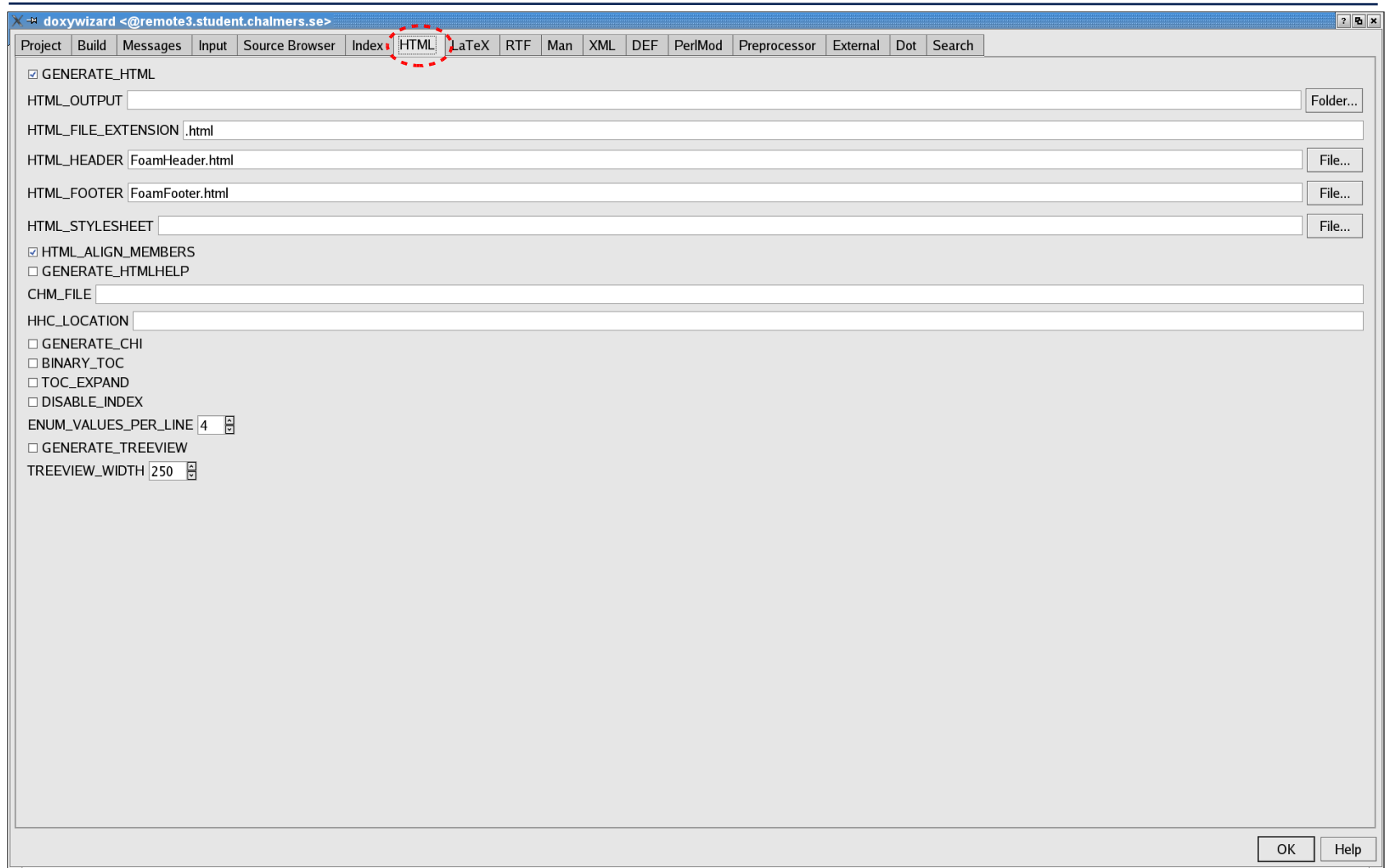
Doxywizard : Expert mode (5/9)



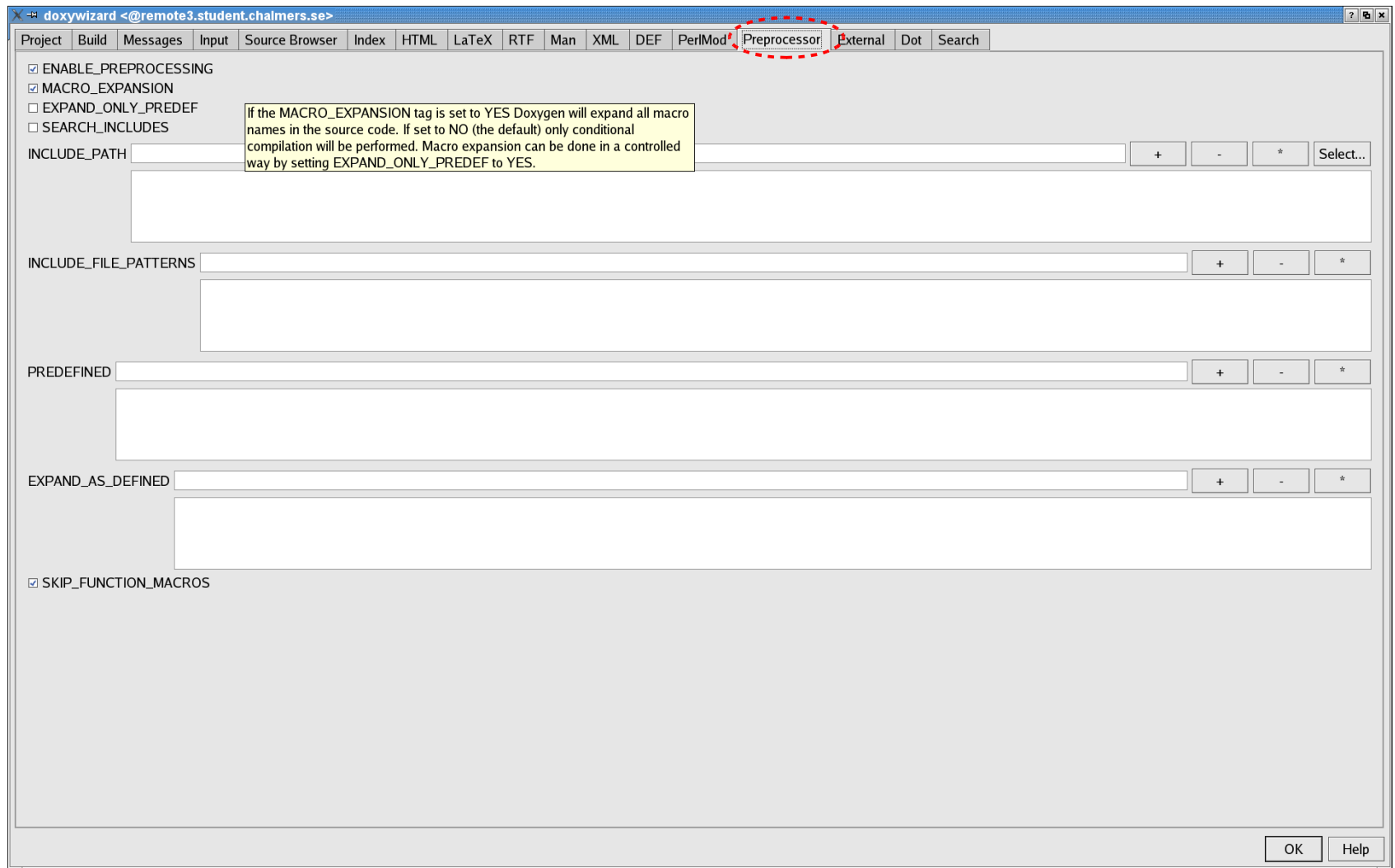
Doxywizard : Expert mode (6/9)



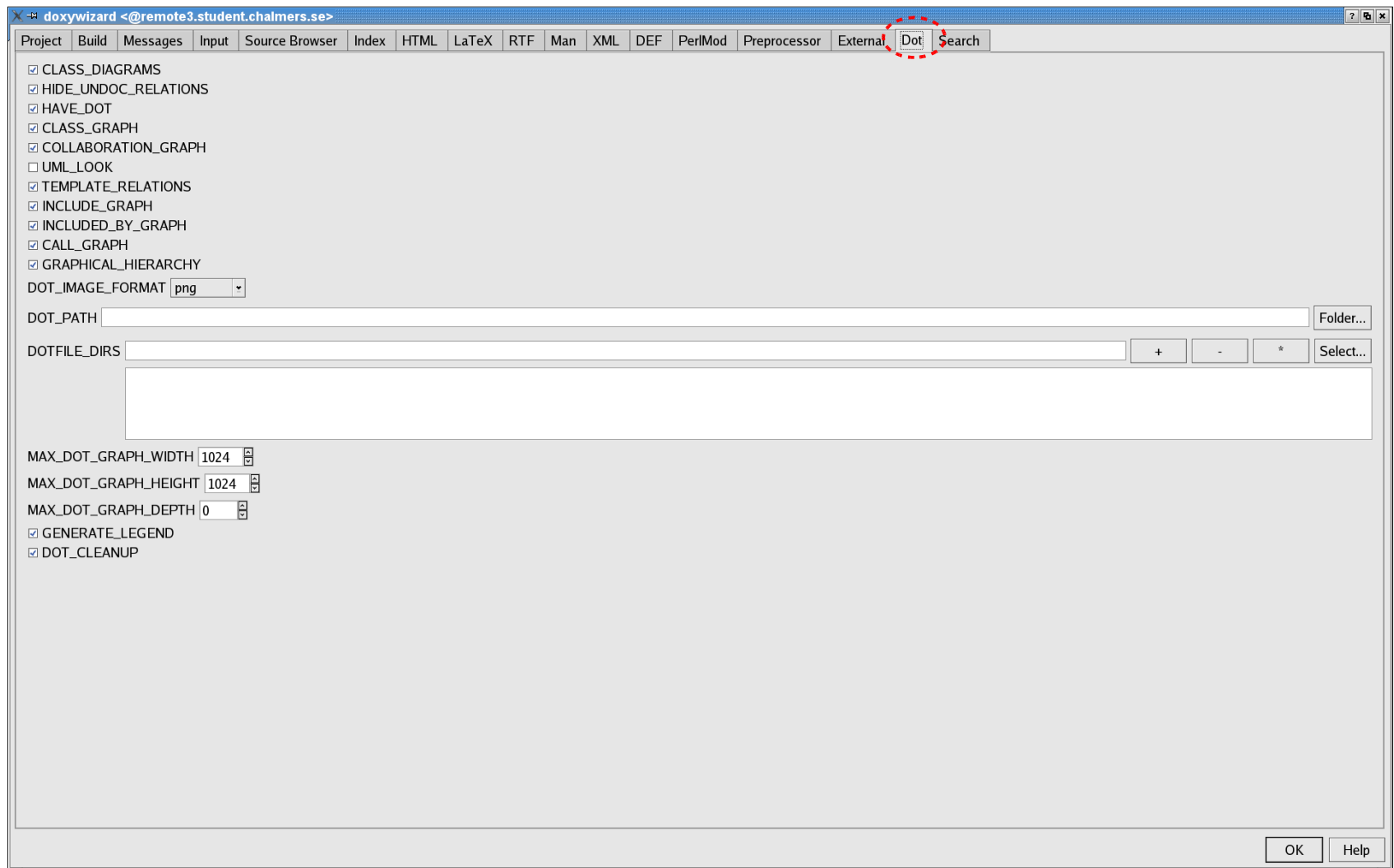
Doxywizard : Expert mode (7/9)



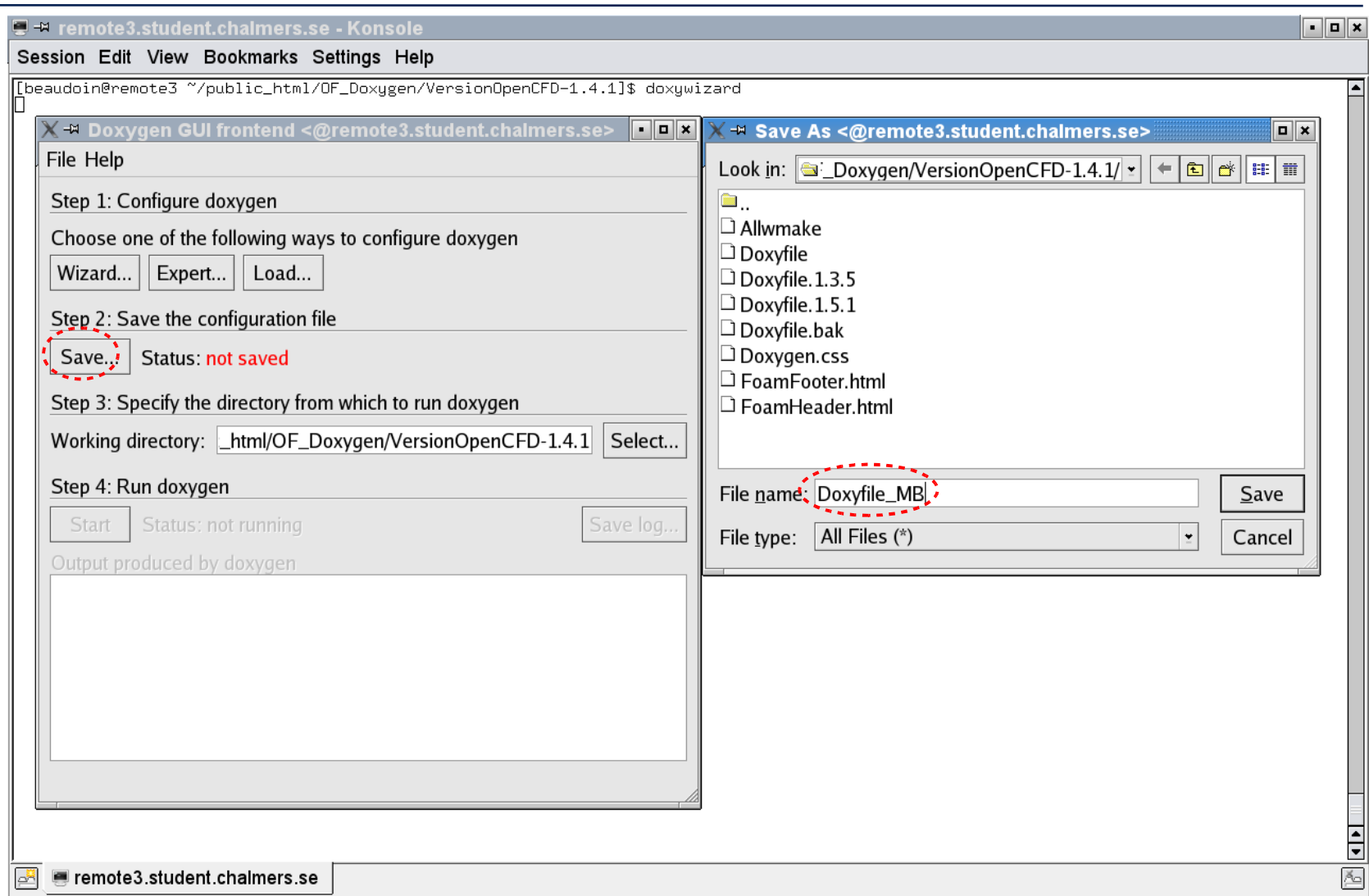
Doxywizard : Expert mode (8/9)



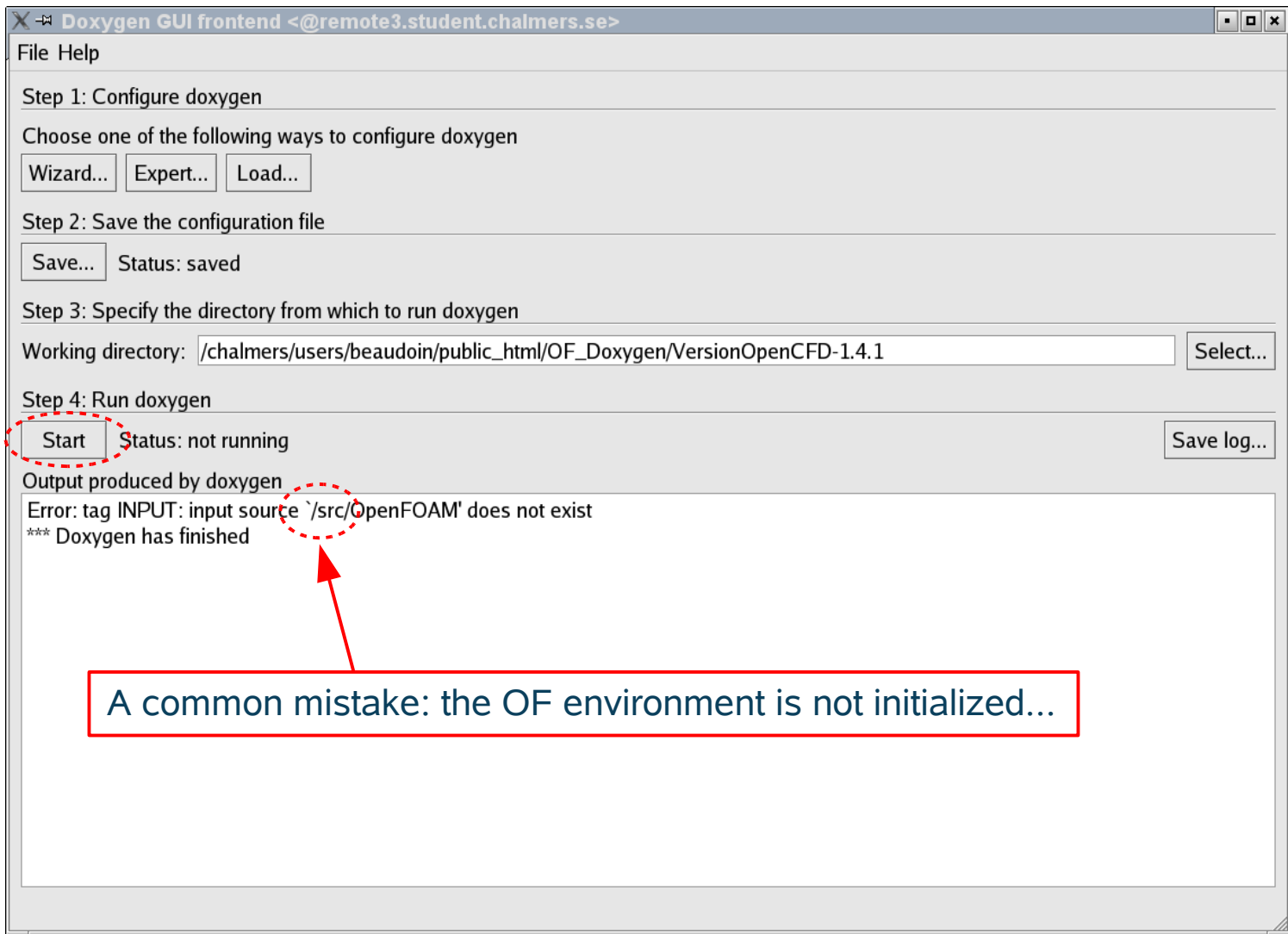
Doxywizard : Expert mode (9/9)



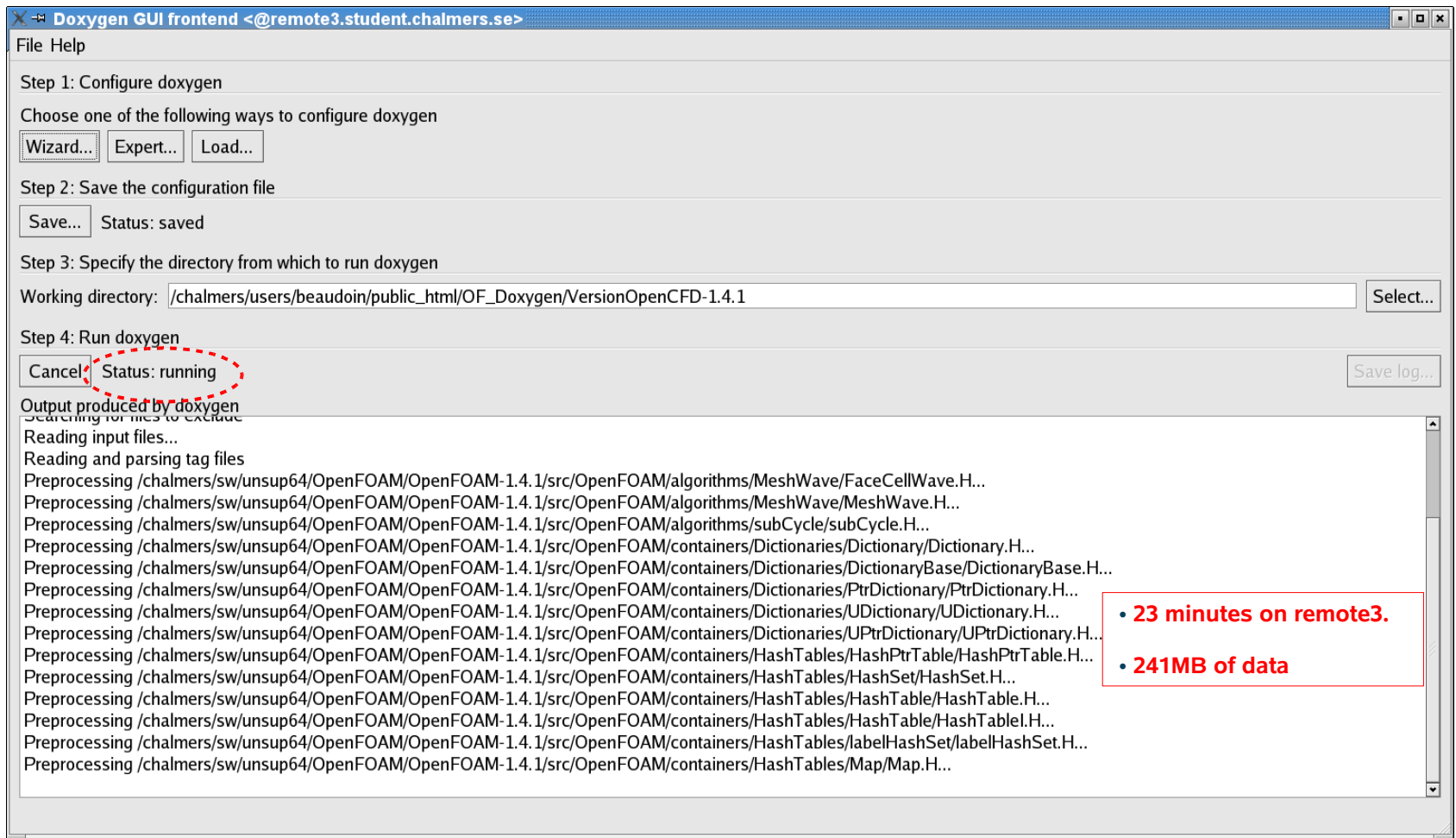
Doxywizard : Saving your modifications



Doxywizard : Generating the HTML files



Doxywizard : Generating the HTML files (take 2)



The screenshot shows the Doxywizard GUI frontend with the following steps and status:

- Step 1: Configure doxygen (Wizard... Expert... Load...)
- Step 2: Save the configuration file (Save... Status: saved)
- Step 3: Specify the directory from which to run doxygen (Working directory: /chalmers/users/beaudoin/public_html/OF_Doxygen/VersionOpenCFD-1.4.1)
- Step 4: Run doxygen (Cancel Status: running)

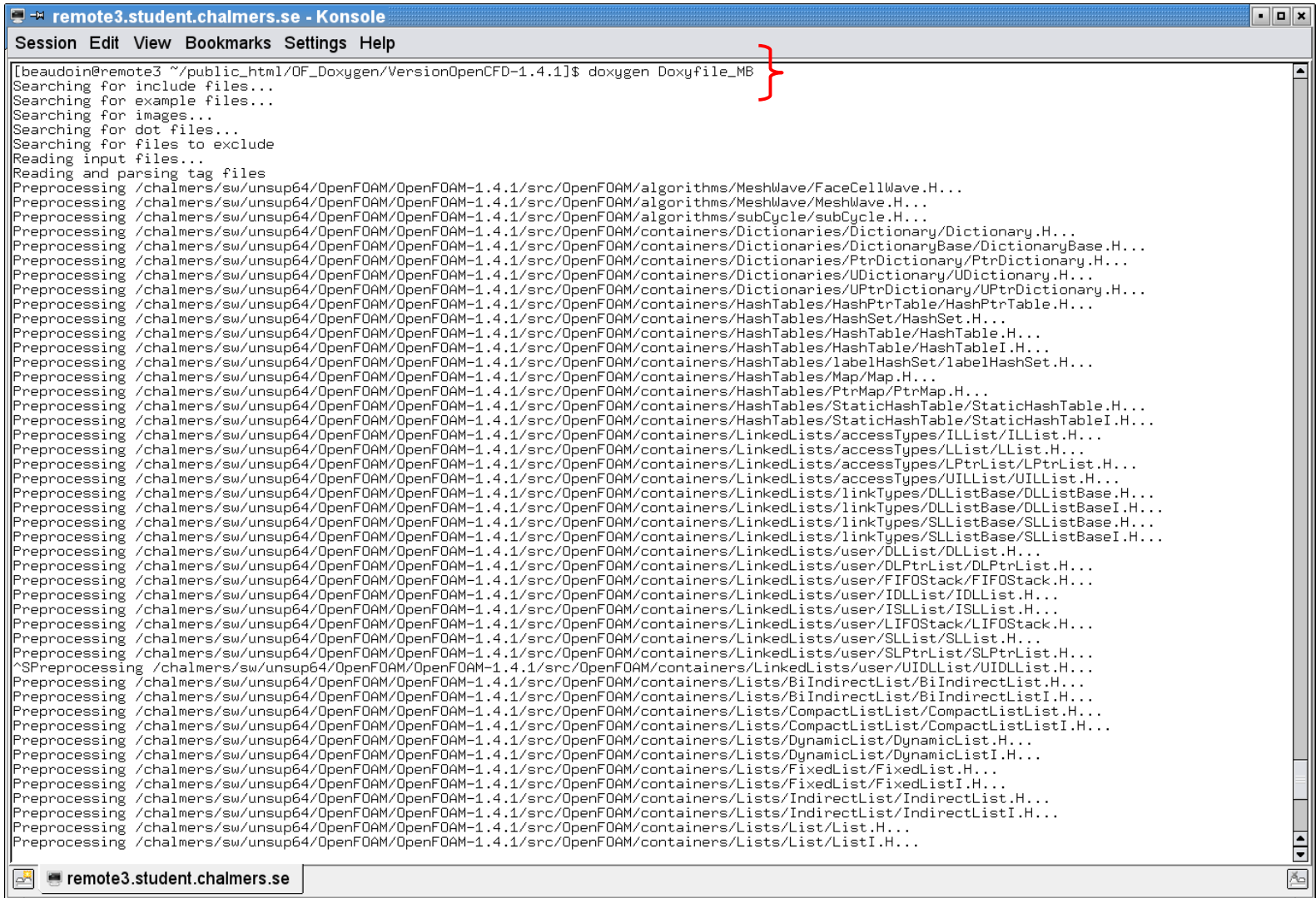
The output produced by doxygen is displayed in a scrollable area:

```
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/HashTable.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...
```

A red box highlights the following statistics:

- 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/Dictionary/Dictionary.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/DictionaryBase/DictionaryBase.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionary/PtrDictionary.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionary/UDictionary.H...
Preprocessing /chalmers/sw/unsup64/OpenFOAM/OpenFOAM-1.4.1/src/OpenFOAM/containers/Dictionary/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/UList/UList.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...
```

Doxygen: Browsing the end result (Class Index)

OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote3.student.chalmers.se> <2>

File Edit View Go Bookmarks Tools Help

file:///chalmers/users/beaudoin/public_html/OF_Doxygen/VersionOpenCFD-1.4.1/html/classes.html

OpenFOAM: The Open Source CFD Toolbox

Programmer's C++ Doxygen [OpenCFD](#) [Solutions](#) [Contact](#) [OpenFOAM](#)

Main Page | [Namespace List](#) | [Class Hierarchy](#) | [Alphabetical List](#) | [Class List](#) | [Directories](#) | [File List](#) | [Namespace Members](#) | [Class Members](#) | [File Members](#) | Search for

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)	Field (Foam)	MB (Foam)	removeParcel (Foam)
accessOp (Foam)	FieldField (Foam)	MCLimiter (Foam)	removePoints (Foam)
addPatchCellLayer (Foam)	FieldMapper (Foam)	meshCutAndRemove (Foam)	repatchPolyTopoChanger (Foam)
algebraicPairGAMGAgglomeration (Foam)	fieldToCell (Foam)	meshCutSurface (Foam)	reuseTmp (Foam)
andEqOp (Foam)	FIFOStack (Foam)	meshCutter (Foam)	reuseTmp< TypeR, TypeR > (Foam)
andEqOp2 (Foam)	fileDiffusivity (Foam)	meshEdgeCuts (Foam)	reuseTmpDimensionedField (Foam)
andOp (Foam)	fileName (Foam)	MeshObject (Foam)	reuseTmpDimensionedField< TypeR, TypeR, GeoM (Foam)
andOp2 (Foam)	fileStat (Foam)	meshSearch (Foam)	reuseTmpFieldField (Foam)
andOp3 (Foam)	filteredLinear2Limiter (Foam)	meshToMesh (Foam)	reuseTmpFieldField< Field, TypeR, TypeR > (Foam)
angularOscillatingDisplacementPointPatchVectorField (Foam)	filteredLinear2VLimiter (Foam)	meshToMesh::patchFieldInterpolator (Foam)	reuseTmpGeometricField (Foam)
angularOscillatingVelocityPointPatchVectorField (Foam)	filteredLinearLimiter (Foam)	meshTriangulation (Foam)	reuseTmpGeometricField< TypeR, TypeR, PatchFie GeoMesh > (Foam)
anisotropicFilter (Foam)	fixedEnthalpyFvPatchScalarField (Foam)	MeshWave (Foam)	reuseTmpTmp (Foam)
APLdiffCoeffFunc (Foam)	fixedFluxBuoyantPressureFvPatchScalarField (Foam)	messageStream (Foam)	reuseTmpTmp< TypeR, Type1, Type12, TypeR > (Foam)
Ar (Foam)	fixedFluxPressureFvPatchScalarField (Foam)	midPoint (Foam)	reuseTmpTmp< TypeR, TypeR, TypeR, Type2 > (Foam)
argList (Foam)	fixedGradientFvPatchField (Foam)	minEqOp (Foam)	reuseTmpTmp< TypeR, TypeR, TypeR, TypeR > (Foam)
argList::initValidTables (Foam)	FixedList (Foam)	minEqOp2 (Foam)	reuseTmpTmpDimensionedField (Foam)
	FixedList::Hash (Foam)	MinmodLimiter (Foam)	reuseTmpTmpDimensionedField< TypeR, Type1, Ty TypeR, GeoMesh > (Foam)
			reuseTmpTmpDimensionedField< TypeR, TypeR, T

Done

Doxygen: Browsing the end result (Class Reference)

OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote3.student.chalmers.se> <3>

File Edit View Go Bookmarks Tools Help

file:///chalmers/users/beaudoin/public_html/OF_Doxygen/VersionOpenCFD-1.4.1/html/d9/d1f/classFoam_1_1fvMesh.html

OpenFOAM: Th... OpenFOAM prd...

OpenFOAM logo The Open Source CFD Toolbox

[Programmer's C++ Doxygen](#) [OpenCFD](#) [Solutions](#) [Contact](#) [OpenFOAM](#)

[Main Page](#) | [Namespace List](#) | [Class Hierarchy](#) | [Alphabetical List](#) | [Class List](#) | [Directories](#) | [File List](#) | [Namespace Members](#) | [Class Members](#) | [File Members](#) | Search for

fvMesh Class Reference

Inheritance diagram for fvMesh:

```
graph TD; fvMesh --> polyMesh; fvMesh --> kduMesh; fvMesh --> surfaceInterpolation; fvMesh --> dynamicFvMesh; fvMesh --> engineMesh;
```

Collaboration diagram for fvMesh:

```
graph TD; fvMesh --> polyMesh; fvMesh --> kduMesh; fvMesh --> surfaceInterpolation; fvMesh --> fvBoundaryMesh; fvMesh --> GeometricField; fvMesh --> SlicedGeometricField; fvMesh --> DimensionedField; fvMesh --> fvMeshLduAddressing; fvMesh --> magSIPtr; fvMesh --> phiPtr; fvMesh --> differenceFactors; fvMesh --> weightingFactors; fvMesh --> CPtr; fvMesh --> OPtr; fvMesh --> SIPtr; fvMesh --> VooPtr; fvMesh --> VOPtr; fvMesh --> kduPtr;
```

[List of all members.](#)

Done

Doxygen: Browsing the end result: one limitation

OpenFOAM programmer's C++ documentation - Mozilla Firefox <@remote3.student.chalmers.se> <3>

file:///chalmers/users/beaudoin/public_html/OF_Doxygen/VersionOpenCFD-1.4.1/html/classes.html

OpenFOAM logo The Open Source CFD Toolbox

Programmer's C++ Doxygen OpenCFD Solutions Contact OpenFOAM

Main Page | Namespace List | Class Hierarchy | Alphabetical List | Class List | Directories | File List | Namespace Members | Class Members | File Members | Search for fMesh

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) Field (Foam) MB (Foam)

accessOp (Foam) FieldField (Foam)

addPatchCellLayer (Foam) FieldMapper (Foam)

algebraicPairGAMGAgglomeration (Foam) fieldToCell (Foam)

andEqOp (Foam) FIFOStack (Foam)

andEqOp2 (Foam) fileDiffusivity (Foam)

andOp (Foam) fileName (Foam)

andOp2 (Foam) fileStat (Foam)

andOp3 (Foam) filteredLinear2Limiter (Foam)

angularOscillatingDisplacementPointPatchVectorField (Foam) filteredLinear2VLimiter (Foam)

angularOscillatingVelocityPointPatchVectorField (Foam) filteredLinearLimiter (Foam)

anisotropicFilter (Foam) fixedEnthalpyFvPatchScalarField (Foam)

APIdiffCoeffFunc (Foam) fixedFluxBuoyantPressureFvPatchScalarField (Foam)

Ar (Foam) fixedFluxPressureFvPatchScalarField (Foam)

argList (Foam) fixedGradientFvPatchField (Foam)

argList::initValidTables (Foam) FixedList (Foam)

ArrheniusReactionRate (Foam) FixedList::Hash (Foam)

arrow2D (Foam) fixedNormalSlipFvPatchField (Foam)

atomicWeightTable (Foam) fixedUnburntEnthalpyFvPatchScalarField (Foam)

atomicWeightTable::atomicWeight (Foam) fixedValueFvPatchField (Foam)

atomizationModel (Foam) fixedValueFvsPatchField (Foam)

attachDetach (Foam) fixedValuePointPatchField (Foam)

attachPolyTopoChanger (Foam) flux (Foam)

autoPtr (Foam) flux < scalar > (Foam)

AverageIOField (Foam) fluxCorrectedVelocityFvPatchVectorField (Foam)

Done foamChemistryReader (Foam)

removeParcel (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)

reuseTmpTmpField (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, TypeR, PatchField, GeoMesh > (Foam)

reuseTmpTmpGeometricField < TypeR, TypeR, TypeR, Type2, PatchField, GeoMesh > (Foam)

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]$ 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]$
```

Doxygen: Configuration for OpenFoamTurbo (1/5)

The screenshot shows a terminal window titled "remote3.student.chalmers.se - Konsole" with the command `doxywizard` executed. Two windows are overlaid on the terminal:

- Doxygen GUI frontend**: A window titled "Doxygen GUI frontend <@remote3.student.chalmers.se>". It contains four steps:
 - Step 1: Configure doxygen**: "Choose one of the following ways to configure doxygen" with buttons for "Wizard...", "Expert...", and "Load...".
 - Step 2: Save the configuration file**: A "Save..." button and a status indicator "Status: not saved".
 - Step 3: Specify the directory from which to run doxygen**: A "Working directory:" label, an empty text input field, and a "Select..." button.
 - Step 4: Run doxygen**: A "Start" button, a status indicator "Status: not running", and a "Save log..." button.Below the steps is a text area labeled "Output produced by doxygen".
- Open dialog**: A window titled "Open <@remote3.student.chalmers.se>". The "Look in:" field shows the path `xygen/openfoam-extend-TurboWG/`. The file list includes:
 - ..
 - Allwmake
 - Doxyfile** (selected)
 - Doxyfile.1.3.5
 - Doxyfile.1.5.1
 - Doxyfile.bak
 - Doxygen.css
 - FoamFooter.html
 - FoamHeader.htmlThe "File name:" field contains "Doxyfile" and the "File type:" dropdown is set to "All Files (*)". "Open" and "Cancel" buttons are at the bottom right.

Doxygen: Configuration for OpenFoamTurbo (2/5)

doxywizard -@remote3.student.chalmers.se>

Project Build Messages Input Source Browser Index HTML LaTeX RTF Man XML DEF PerlMod Preprocessor External Dot Search

PROJECT_NAME OpenFOAM-\$(WM_PROJECT_VERSION)-TurboWG

PROJECT_NUMBER

OUTPUT_DIRECTORY Folder...

CREATE_SUBDIRS

OUTPUT_LANGUAGE English

USE_WINDOWS_ENCODING

BRIEF_MEMBER_DESC

REPEAT_BRIEF

ABBREVIATE_BRIEF + - *

ALWAYS_DETAILED_SEC

INLINE_INHERITED_MEMB

FULL_PATH_NAMES

STRIP_FROM_PATH + - *

\$(WM_PROJECT_DIR)

STRIP_FROM_INC_PATH + - *

SHORT_NAMES

JAVADOC_AUTOBRIEF

MULTILINE_CPP_IS_BRIEF

DETAILS_AT_TOP

INHERIT_DOCS

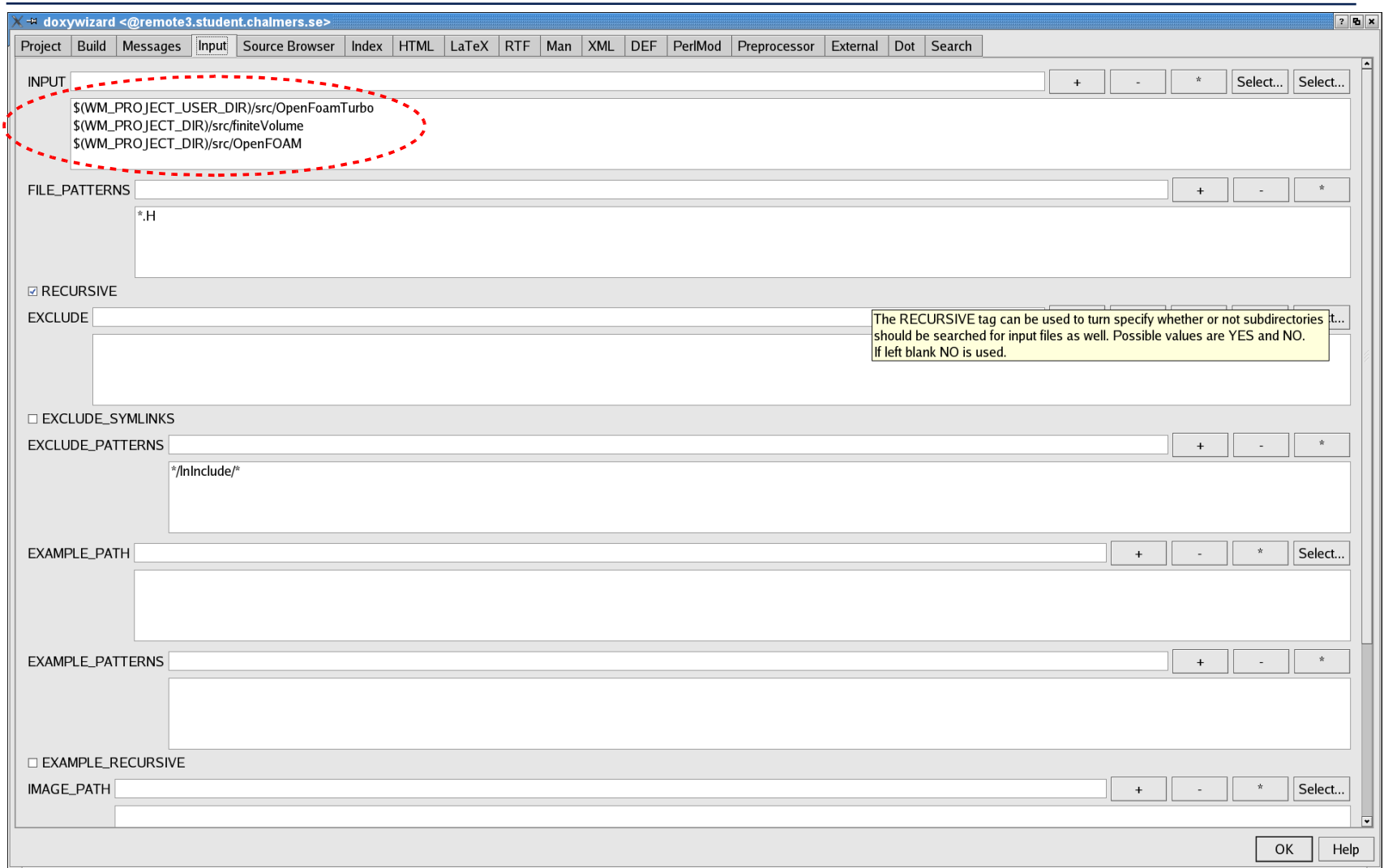
DISTRIBUTE_GROUP_DOC

TAB_SIZE 4

ALIASES + - *

OK Help

Doxygen: Configuration for OpenFoamTurbo (3/5)

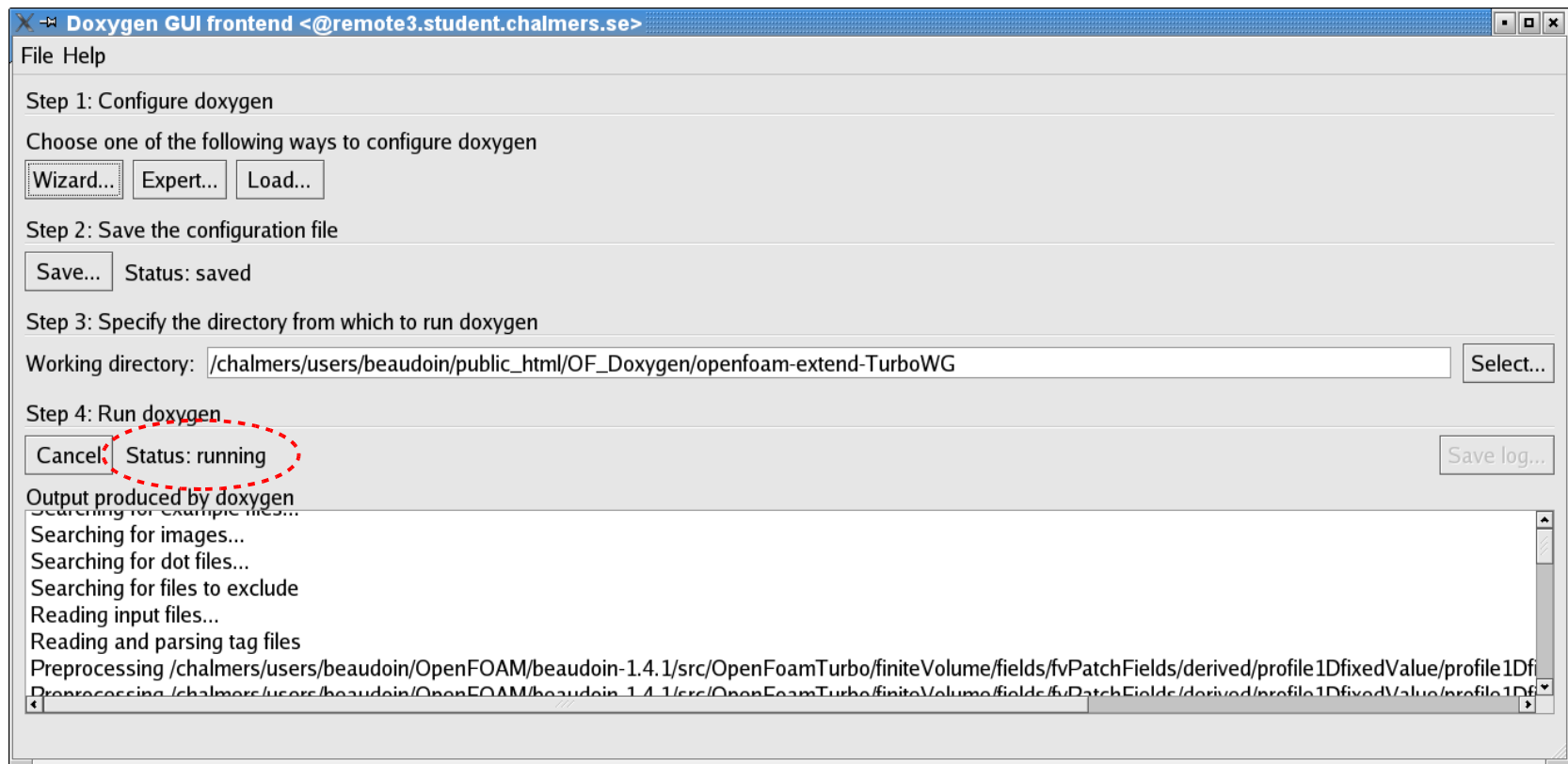


Doxygen: Configuration for OpenFoamTurbo (4/5)

The screenshot shows a terminal window titled "remote3.student.chalmers.se - Konsole" with a menu bar (Session, Edit, View, Bookmarks, Settings, Help). The terminal prompt is `[beaudoin@remote3 ~/public_html/OF_Doxygen/openfoam-extend-TurboWG]$ doxywizard`. Two windows are overlaid on the terminal:

- Doxygen GUI frontend <@remote3.student.chalmers.se>**:
 - File Help
 - Step 1: Configure doxygen
 - Choose one of the following ways to configure doxygen
 - Wizard... Expert... Load...
 - Step 2: Save the configuration file
 - Save... Status: not saved
 - Step 3: Specify the directory from which to run doxygen
 - Working directory: `~/OF_Doxygen/openfoam-extend-TurboWG` Select...
 - Step 4: Run doxygen
 - Start Status: not running Save log...
 - Output produced by doxygen
- Save As <@remote3.student.chalmers.se>**:
 - Look in: `~/xygen/openfoam-extend-TurboWG/`
 - Files: `FoamFooter.html`, `FoamHeader.html`, `Allwmake`, `Doxyfile`, `Doxyfile.1.3.5`, `Doxyfile.1.5.1`, `Doxyfile.bak`, `Doxyfile_MB`, `Doxyfile_MB_TurboWG` (selected), `Doxygen.css`
 - File name: `Doxyfile_MB_TurboWG`
 - File type: `All Files (*)`

Doxygen: Configuration for OpenFoamTurbo (5/5)



OpenFOAM Turbo: Browsing the result (1/3)

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

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

OpenFOAM: Th... OpenFOAM pro...

DynamicList (Foam)	uduMatrix (Foam)	(Foam)	Cmpt > > (Foam)
E	IduMatrix::preconditioner (Foam)	pressureInletOutletVelocityFvPatchVectorField (Foam)	typeOfSum< SphericalTensor< Cmpt >, Tensor< Cmpt > > (Foam)
edge (Foam)	IduMatrix::smoother (Foam)	pressureInletUniformVelocityFvPatchVectorField (Foam)	typeOfSum< SymmTensor< Cmpt >, SphericalTensor< Cmpt > > (Foam)
empty (Foam)	IduMatrix::solver (Foam)	pressureInletVelocityFvPatchVectorField (Foam)	typeOfSum< SymmTensor< Cmpt >, Tensor< Cmpt > > (Foam)
emptyFvPatch (Foam)	IduMatrix::solverPerformance (Foam)	primitiveEntry (Foam)	typeOfSum< Tensor2D< Cmpt >, SphericalTensor2D< Cmpt > > (Foam)
emptyFvPatchField (Foam)	IduMesh (Foam)	primitiveMesh (Foam)	typeOfSum< Tensor< Cmpt >, SphericalTensor< Cmpt > > (Foam)
emptyFvsPatchField (Foam)	IduPrimitiveMesh (Foam)	PrimitivePatch (Foam)	typeOfSum< Tensor< Cmpt >, SymmTensor< Cmpt > > (Foam)
emptyPointPatch (Foam)	IduScheduleEntry (Foam)	PrimitivePatchInterpolation (Foam)	U
emptyPointPatchField (Foam)	leastSquaresGrad (Foam::fv)	PrimitivePatchName	UDictionary (Foam)
emptyPolyPatch (Foam)	leastSquaresVectors (Foam)	prismMatcher (Foam)	UIDList (Foam)
entry (Foam)	LIFOStack (Foam)	processorFvPatch (Foam)	UILLList (Foam)
eqEqOp (Foam)	Limited01Limiter (Foam)	processorFvPatchField (Foam)	UILLList::const_iterator (Foam)
eqEqOp2 (Foam)	limitedCubicLimiter (Foam)	processorFvsPatchField (Foam)	UILLList::iterator (Foam)
eqEqOp3 (Foam)	limitedCubicVLimiter (Foam)	processorGAMGInterface (Foam)	UList (Foam)
eqMagOp (Foam)	LimitedLimiter (Foam)	processorGAMGInterfaceField (Foam)	UMISTLimiter (Foam)
eqMagOp2 (Foam)	limitedLinearLimiter (Foam)	processorLduInterface (Foam)	uncorrectedSnGrad (Foam::fv)
eqMinusOp (Foam)	LimitedScheme (Foam)	processorLduInterfaceField (Foam)	uniformFixedValueFvPatchField (Foam)
eqMinusOp2 (Foam)	limitedSnGrad (Foam::fv)	processorPointPatch (Foam)	UPtrDictionary (Foam)
eqOp (Foam)	limitedSurfaceInterpolationScheme (Foam)	processorPointPatchField (Foam)	UPtrList (Foam)
eqOp2 (Foam)	line (Foam)	processorPolyPatch (Foam)	UPtrList::iterator (Foam)
error (Foam)	linear (Foam)	ProcessorTopology (Foam)	upwind (Foam)
errorManip (Foam)	linearUpwind (Foam)	procLduInterface (Foam)	V
errorManipArg (Foam)	linearUpwindV (Foam)	procLduInterfaceField (Foam)	valuePointPatchField (Foam)
EulerD2dt2Scheme (Foam::fv)	List (Foam)	profile1DfixedValueFvPatchField (Foam)	vanAlbadaLimiter (Foam)
EulerDdtScheme (Foam::fv)	LList (Foam)	profile1DRawData (Foam)	vanLeerLimiter (Foam)
expDirectionMixedFvPatchField (Foam)	LList::const_iterator (Foam)	stream (Foam)	Vector (Foam)
extendedLeastSquaresGrad (Foam::fv)	LList::iterator (Foam)	stream::commsStruct (Foam)	Vector2D (Foam)
extendedLeastSquaresVectors (Foam)	LList::link (Foam)	pTraits (Foam)	VectorSpace (Foam)
F	LListBase	pTraits< bool > (Foam)	VectorSpaceOps
face (Foam)	localBlended (Foam)	pTraits< label > (Foam)	VectorSpaceOps< 0, 0 >
faceAreaPairGAMGAgglomeration (Foam)	localMax (Foam)	pTraits< Scalar >	volMesh (Foam)
FaceCellWave (Foam)	localMin (Foam)	PtrDictionary (Foam)	volPointInterpolation (Foam)
faceDecompCuts (Foam)	LPtrList (Foam)	PtrList (Foam)	W
faceDecompIsoSurfCuts (Foam)	LPtrList::const_iterator (Foam)	PtrList::iterator (Foam)	walkPatch (Foam)
faceEdge (Foam)	LPtrList::iterator (Foam)	PtrMap (Foam)	wallDist (Foam)
faceEdge::faceEdgeHash (Foam)	LUScalarMatrix (Foam)	pyramid (Foam)	wallDistData (Foam)
faceLimitedGrad (Foam::fv)	M	pyramidEdge (Foam)	wallDistReflection (Foam)
FaceList	magSqr (Foam)::limitFuncs	pyramidEdge::pyramidEdgeHash (Foam)	wallFvPatch (Foam)
faceMapper (Foam)	Map (Foam)	pyrMatcher (Foam)	wallPointPatch (Foam)
faceMDLimitedGrad (Foam::fv)	mapAddedPolyMesh (Foam)	Q	wallPointYPlus (Foam)
facePointPatch (Foam)	mapDistributePolyMesh (Foam)	QUICKLimiter (Foam)	wallPolyPatch (Foam)
faceZone (Foam)	MapInternalField (Foam)	QUICKVLimiter (Foam)	waveTransmissiveFvPatchField (Foam)
EDICPreconditioner (Foam)	MapInternalField< Type, MeshMapper,	R	wedgeFvPatch (Foam)
Done			

OpenFoamTurbo: Browsing the result (2/3)

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

file:///chalmers/users/beaudoin/public_html/OF_Doxygen/openfoam-extend-TurboWG/html/d3/d3b/classFoam_1_1profile1DfixedValueFvPatchField.html

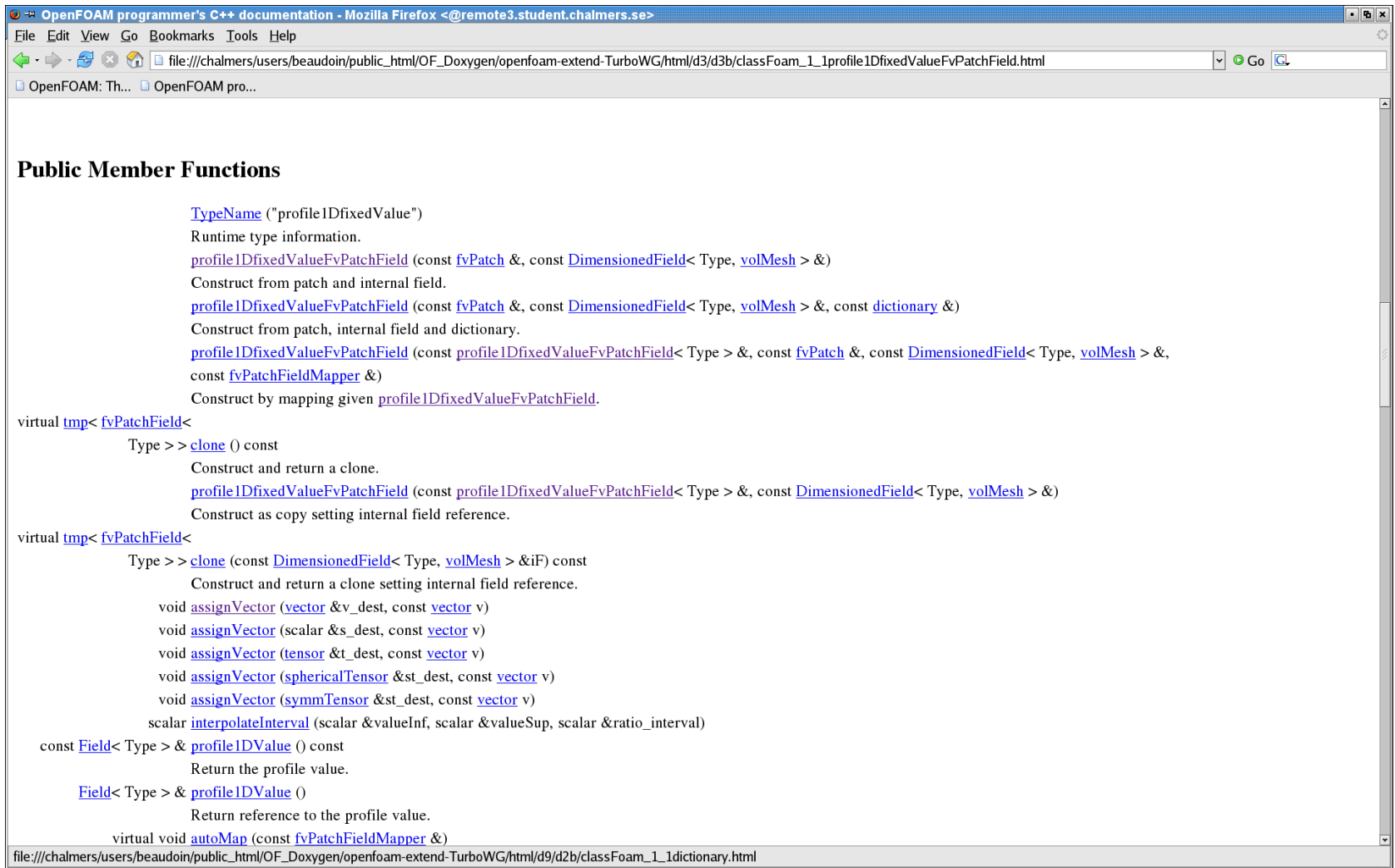
profile1DfixedValueFvPatchField Class Template Reference

Inheritance diagram for profile1DfixedValueFvPatchField:

Collaboration diagram for profile1DfixedValueFvPatchField:

Done

OpenFoamTurbo: Browsing the result (3/3)



```
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/d3/d3b/classFoam_1_1profileIDfixedValueFvPatchField.html
OpenFOAM: Th... OpenFOAM pro...

Public Member Functions

TypeName ("profileIDfixedValue")
Runtime type information.
profileIDfixedValueFvPatchField (const fvPatch &, const DimensionedField< Type, volMesh > &)
Construct from patch and internal field.
profileIDfixedValueFvPatchField (const fvPatch &, const DimensionedField< Type, volMesh > &, const dictionary &)
Construct from patch, internal field and dictionary.
profileIDfixedValueFvPatchField (const profileIDfixedValueFvPatchField< Type > &, const fvPatch &, const DimensionedField< Type, volMesh > &, const fvPatchFieldMapper &)
Construct by mapping given profileIDfixedValueFvPatchField.

virtual tmp< fvPatchField<
Type >> clone () const
Construct and return a clone.
profileIDfixedValueFvPatchField (const profileIDfixedValueFvPatchField< Type > &, const DimensionedField< Type, volMesh > &)
Construct as copy setting internal field reference.

virtual tmp< fvPatchField<
Type >> clone (const DimensionedField< Type, volMesh > &iF) const
Construct and return a clone setting internal field reference.
void assignVector (vector &v_dest, const vector v)
void assignVector (scalar &s_dest, const vector v)
void assignVector (tensor &t_dest, const vector v)
void assignVector (sphericalTensor &st_dest, const vector v)
void assignVector (symmTensor &st_dest, const vector v)
scalar interpolateInterval (scalar &valueInf, scalar &valueSup, scalar &ratio_interval)

const Field< Type > & profileIDValue () const
Return the profile value.
Field< Type > & profileIDValue ()
Return reference to the profile value.
virtual void autoMap (const fvPatchFieldMapper &)

file:///chalmers/users/beaudoin/public_html/OF_Doxygen/openfoam-extend-TurboWG/html/d9/d2b/classFoam_1_1dictionary.html
```

Doxygen: some possible improvements (1/5)

doxygenwizard <@remote3.student.chalmers.se>

Project Build Messages Input Source Browser Index HTML LaTeX RTF Man XML DEF PerlMod Preprocessor External Dot Search

PROJECT_NAME OpenFOAM-\$(WM_PROJECT_VERSION)-TurboWG

PROJECT_NUMBER

OUTPUT_DIRECTORY Folder...

CREATE_SUBDIRS

OUTPUT_LANGUAGE English

USE_WINDOWS_ENCODING

BRIEF_MEMBER_DESC

REPEAT_BRIEF

ABBREVIATE_BRIEF + - *

ALWAYS_DETAILED_SEC

INLINE_INHERITED_MEMB

FULL_PATH_NAMES

STRIP_FROM_PATH + - *

\$(WM_PRO

STRIP_FROM_INC_PATH + - *

SHORT_NAMES

JAVADOC_AUTOBRIEF

MULTILINE_CPP_IS_BRIEF

DETAILS_AT_TOP

INHERIT_DOCS

DISTRIBUTE_GROUP_DOC

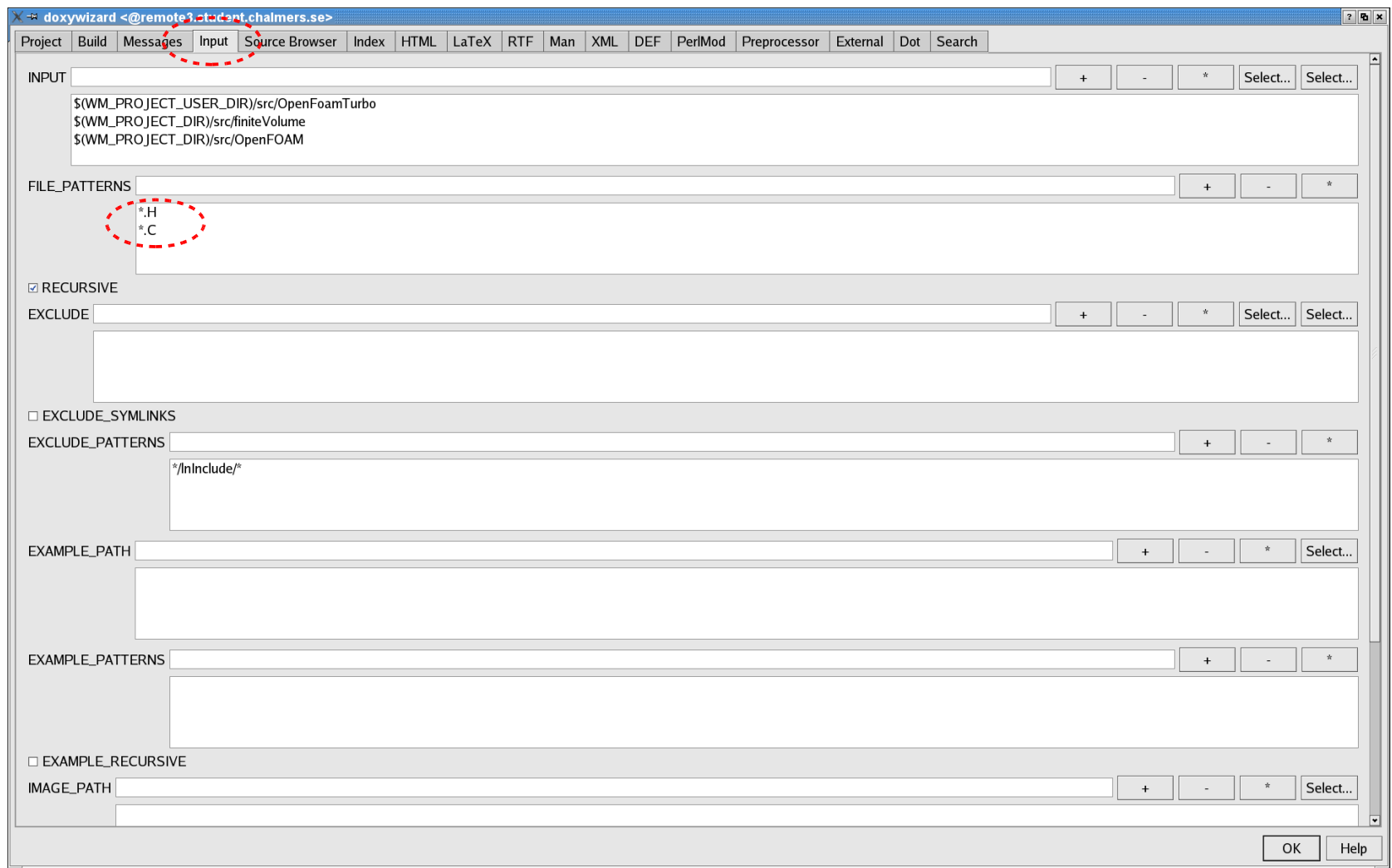
TAB_SIZE 4

ALIASES + - *

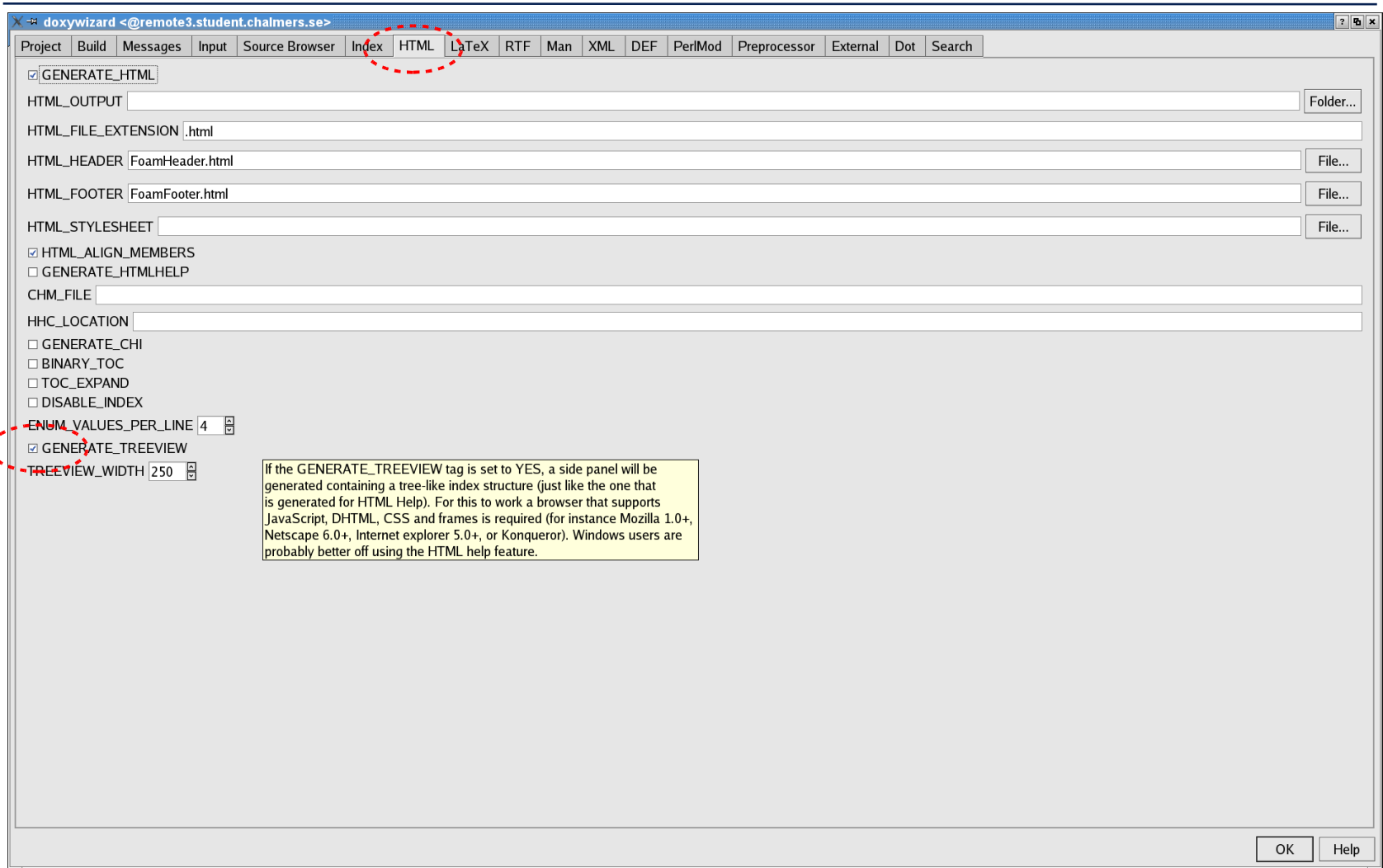
OK Help

If the `INLINE_INHERITED_MEMB` tag is set to YES, doxygen will show all inherited members of a class in the documentation of that class as if those members were ordinary class members. Constructors, destructors and assignment operators of the base classes will not be shown.

Doxygen: some possible improvements (2/5)



Doxygen: some possible improvements (3/5)



Doxygen: some possible improvements (4/5)

OpenFOAM-1.4.1-TurboWG-super - Mozilla Firefox <@remote3.student.chalmers.se>

file:///chalmers/users/beaudoin/public_html/OF_Doxygen/openfoam-extend-TurboWG-super/html/index.html

OpenFOAM: Th... OpenFOAM pro...

OpenFOAM-1.4.1-TurboWG-super

- Main Page
- File List
- Class List
- Class Hierarchy
- Class Members
- Namespace List
 - Foam
 - Foam::debug
 - Foam::fv
 - Foam::fvc
 - Foam::fvn
 - Foam::limitFuncs
 - Foam::ListListOps
 - Foam::mathematicalConstant
 - Foam::MULES
 - Foam::Unix
- Directories
 - src
 - finiteVolume
 - OpenFOAM
 - users
 - beaudoin
 - OpenFOAM
 - beaudoin-1.4.1
 - src
 - OpenFoamTurbo
 - finiteVolume
 - fields
 - fvPatchFields
 - derived
 - profile1DfixedValueFvPatchField
 - File Members
 - Namespace Members
 - Graphical Class Hierarchy

[Main Page](#) | [Namespace List](#) | [Class Hierarchy](#) | [Alphabetical List](#) | [Class List](#) | [Directories](#) | [File List](#) | [Namespace Members](#) | [Class Members](#) | [File Members](#) | [Search](#)

[users / beaudoin / OpenFOAM / beaudoin-1.4.1 / src / OpenFoamTurbo / finiteVolume / fields / fvPatchFields / derived / profile1DfixedValueFvPatchField.C](#)

profile1DfixedValueFvPatchField.C

[Go to the documentation of this file.](#)

```
00001 /*-----*/
00002 =====
00003 \      /      F i e l d      OpenFOAM: The Open Source CFD Toolbox
00004  \      /      O p e r a t i o n
00005   \    /      A n d      Copyright Hydro-Quebec - IREQ, 2008
00006    \  /      M a n i p u l a t i o n
00007     \ /
00008 -----
00009 License
00010 This file is part of OpenFOAM.
00011
00012 OpenFOAM is free software; you can redistribute it and/or modify it
00013 under the terms of the GNU General Public License as published by the
00014 Free Software Foundation; either version 2 of the License, or (at your
00015 option) any later version.
00016
00017 OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
00018 ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
00019 FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License
00020 for more details.
00021
00022 You should have received a copy of the GNU General Public License
00023 along with OpenFOAM; if not, write to the Free Software Foundation,
00024 Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA
00025
00026 Class
00027 profile1DfixedValueFvPatchField
00028
00029 Description
00030 This class implements a cylindrical boundary condition field defined
00031 by a 1D fixed value profile (radial ou vertical)
00032
00033 This class rely on a helper class profile1DrowData for reading the profile
00034 values from an ASCII file
00035
00036 It is possible to provide a scale factor for the field values.
00037
00038 For instance, for the pressure field, this scale factor could be 1/rho, rho being the density.
00039 This scaling factor will be used to scale the pressure since it is p/rho that is solved by
00040 simpleFoam, icoFoam, etc
00041
00042 Usage:
```

Doxygen: some possible improvements (5/5)

void [operator=](#) (const [UList](#)< Type > &)
void [operator=](#) (const [tmp](#)< [Field](#)< Type > > &)
virtual void [operator *=](#) (const [fvPatchField](#)< scalar > &)
virtual void [operator *=](#) (const [Field](#)< scalar > &)
virtual void [operator *=](#) (const scalar)
void [operator *=](#) (const [UList](#)< scalar > &)
void [operator *=](#) (const [tmp](#)< [Field](#)< scalar > > &)
void [operator *=](#) (const scalar &)
virtual void [operator/=](#) (const [fvPatchField](#)< scalar > &)
virtual void [operator/=](#) (const [Field](#)< scalar > &)
virtual void [operator/=](#) (const scalar)
void [operator/=](#) (const [UList](#)< scalar > &)
void [operator/=](#) (const [tmp](#)< [Field](#)< scalar > > &)
void [operator/=](#) (const scalar &)
[declareRunTimeSelectionTable](#) ([tmp](#), [fvPatchField](#), [patch](#), (const [fvPatch](#) &p, const [DimensionedField](#)< Type, [volMesh](#) > &iF), (p, iF, dict))
[declareRunTimeSelectionTable](#) ([tmp](#), [fvPatchField](#), [patchMapper](#), (const [fvPatchField](#)< Type > &ptf, const [fvPatch](#) &p, const [DimensionedField](#)< Type, [volMesh](#) > &iF, const [fvPatchFieldMapper](#) &m), (dynamic_cast< const [fvPatchFieldType](#) &>(ptf), p, iF, dict))
[declareRunTimeSelectionTable](#) ([tmp](#), [fvPatchField](#), [dictionary](#), (const [fvPatch](#) &p, const [DimensionedField](#)< Type, [volMesh](#) > &iF, [dictionary](#) &dict), (p, iF, dict))
const [objectRegistry](#) & [db](#) () const
Return local [objectRegistry](#).
const [fvPatch](#) & [patch](#) () const
Return [patch](#).
const [DimensionedField](#)< Type, [volMesh](#) > & [dimensionedInternalField](#) () const
Return dimensioned internal field reference.
const [Field](#)< Type > & [internalField](#) () const
Return internal field reference.
virtual bool [coupled](#) () const
Return true if this patch field is coupled.
bool [updated](#) () const
Return true if the boundary condition has already been updated.

- inherited member functions listed
- mixed blessing...

Doxygen: some useful pointers

➤ **Main Web site:**

- <http://www.stack.nl/~dimitri/doxygen/index.html>

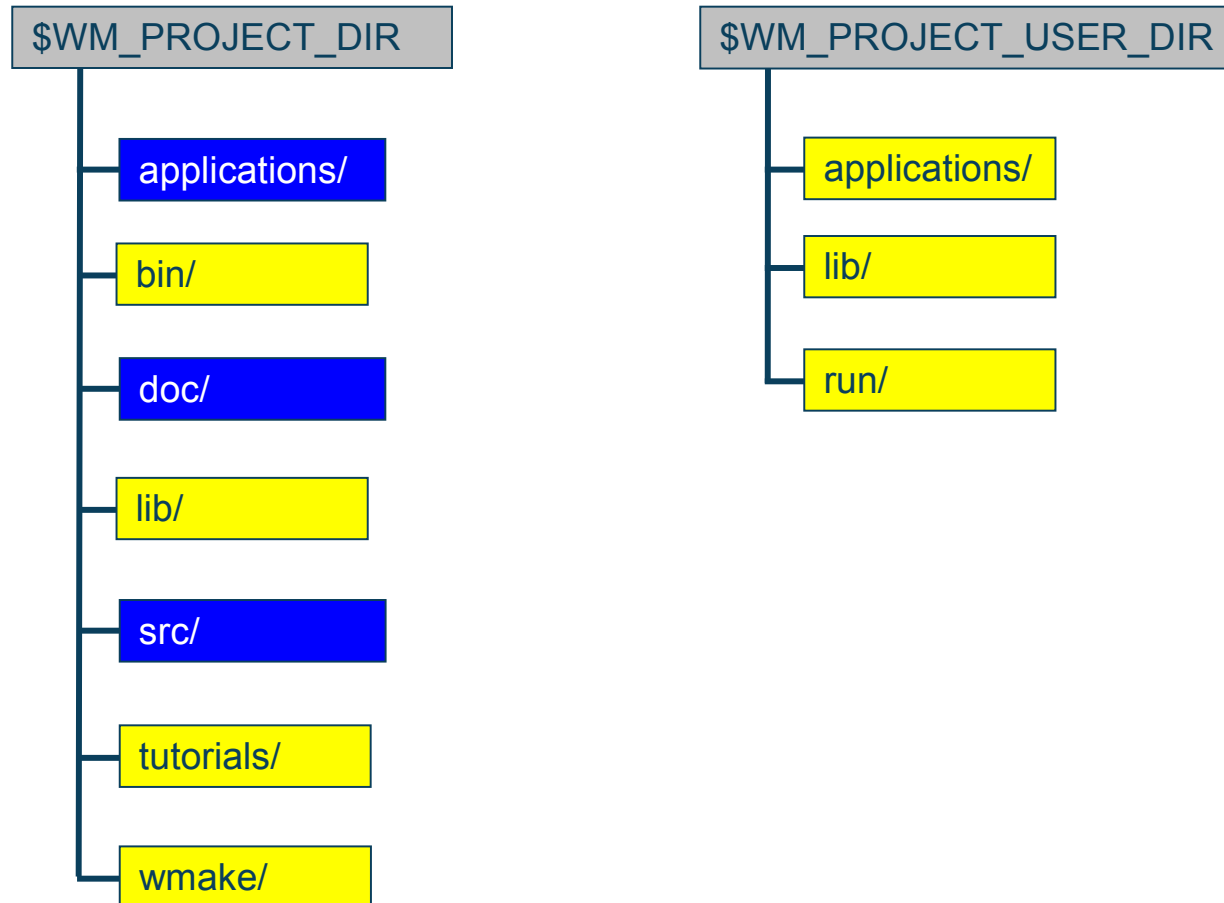
➤ **The Doxygen manual:**

- <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



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:

➤ **InInclude** 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/InInclude/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_\(software\)](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)

SourceForge.net: openfoam-extend » home - Mozilla Firefox

File Edit View History Bookmarks Tools Help

http://openfoam-extend.wiki.sourceforge.net/

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

SOURCEFORGE.NET®

Home Browse Software Marketplace Community Create Project Jobs

Software Search Advanced

PM Office Software
Develop a Successful Project Mgmt Office. Download the Toolkit Today.
www.Daptiv.com

CFdesign - Upfront CFD
Fluid flow and thermal simulations for mechanical engineers
www.valeng.com

Browser CRM - Free CRM
web based customer management CRM, Email & Collaboration
www.browsercrm.com

SF.net » Projects » openfoam-extend » Wiki

OpenFOAM extensions

Project Tracker Mailing Lists Forums Code Services Download Documentation Tasks Wiki

Wiki Navigation home page | discuss SVN SVN Statistics SVN | backlinks

Recent Changes Manage Space

This Wiki Home

Documentation

- General guidelines for using the OpenFOAM-extend Subversion archive

Useful Links

- OpenCFD OpenFOAM
- OpenFOAM Forum
- OpenFOAM Wiki

OpenFOAM-extend: Extensions to the OpenFOAM CFD Toolbox

The definition of this project, as stated on the main openfoam-extend project page is:

"The goal of this project is to open the OpenFOAM CFD toolbox to community contributed extensions", in the spirit of the OpenSource development model.

The primary mechanism made available for facilitating this collaboration is a Subversion repository for user developments related to OpenFOAM.

The Subversion repository is also a convenient mechanism for downloading patches to the OpenCFD distribution of OpenFOAM.

The Wiki Navigation panel on the left-side of this page will lead you to the information on how to take (or be) part of the OpenFOAM-extend project.

Protected

wikispaces Wikis by Wikispaces®

http://sourceforge.net/svn/?group_id=194890

Browsing revisions from a Subversion repository (2/5)

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

File Edit View History Bookmarks Tools Help

http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/

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

[openfoam-extend] /trunk

Index of /trunk

FILES SHOWN: 0

Directory revision: 561 (of 561)

Sticky Revision: Set

| File | Rev. | Age | Author | Last log entry |
|----------------------------------|------|------------|-----------|--|
| Parent Directory | | | | |
| Breeder/ | 553 | 4 days | mbeaudoin | Uncompressed files in 0_orig for both Case0 and Case1 |
| Core/ | 561 | 60 minutes | hjasak | Forum bug fix |
| Forge/ | 42 | 7 months | bgschaid | Set properties for directories so that Include etc are ignored |

[Back to SourceForge.net](#)

Powered by [ViewVC 1.0.3](#)

[ViewVC and Help](#)

Done

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 persondata... OpenFOAM Message B... OpenFOAMWiki SourceForge.net: openf... OF_Performance

Log of /trunk/Core

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 symTensor 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)

SourceForge.net Repository - [openfoam-extend] Revision 549 - Mozilla Firefox <@rdle02220mdw>

File Edit View History Bookmarks Tools Help

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

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

[openfoam-extend]

Revision 549

SOURCEFORGE.NET®

Jump to revision: Go ↵ ↵

Author: mbeaudoin

Date: Thu Mar 20 12:15:34 2008 UTC (5 days, 2 hours ago)

Log Message: Corrected infinite loop problem from meshSearch::intersections

Changed paths:

| Path | Details |
|---|---|
| trunk/Core/OpenFOAM-1.4.1-dev/src/meshTools/meshSearch/meshSearch.C | modified , text changed |

[Back to SourceForge.net](#)

Powered by [ViewVC 1.0.3](#)

[ViewVC and Help](#)

Done

Browsing revisions from a Subversion repository (5/5)

SourceForge.net Repository - [openfoam-extend] Diff of /trunk/Core/OpenFOAM-1.4.1-dev/src/meshTools/meshSearch/meshSearch.C - Mozilla Firefox <@rdle02220mdw>

File Edit View History Bookmarks Tools Help

http://openfoam-extend.svn.sourceforge.net/viewvc/openfoam-extend/trunk/Core/OpenFOAM-1.4.1-dev/src/meshTools/meshSearch/meshSearch.C?r1=5

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

[openfoam-extend] / trunk / Core / OpenFOAM-1.4.1-dev / src / meshTools / meshSearch / meshSearch.C

Diff of /trunk/Core/OpenFOAM-1.4.1-dev/src/meshTools/meshSearch/meshSearch.C SOURCEFORGE.NET

Parent Directory | Revision Log | Patch

revision 548, Thu Mar 20 10:03:44 2008 UTC

revision 549, Thu Mar 20 12:15:34 2008 UTC

```
# Line 698
698
699     if (bHit.hit())
700     {
701
702
703
704
705
706
707
708
709
710
711
712
713
714
715
716     hits.append(bHit);
717
718     const vector& area = mesh_faceAreas()[bHit.index()];

```

```
# Line 698
698     if (bHit.hit())
699     {
700         // Verify if this hit point has not been visited before
701         // Otherwise, we are entering an infinite loop.
702         bool alreadyVisitedPoint = false;
703         forAll(hits, hi)
704         {
705             if (bHit.hitPoint() == hits[hi].hitPoint())
706             {
707                 alreadyVisitedPoint = true;
708                 break;
709             }
710         }
711         // Start of infinite loop?
712         if (alreadyVisitedPoint)
713             break;
714
715     hits.append(bHit);
716
717     const vector& area = mesh_faceAreas()[bHit.index()];

```

Colored Diff Show Legend:
Removed from v. 548
changed lines
Added in v. 549

[Back to SourceForge.net](#) [ViewVC and Help](#)

Powered by [ViewVC 1.0.3](#)

Done

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