

Doxygen in the cylindricalInletVelocity boundary condition

The `cylindricalInletVelocity` boundary condition is implemented in

`$FOAM_SRC/finiteVolume/fields/fvPatchFields/derived/cylindricalInletVelocity`

Let's have a closer look at how the Doxygen documentation is generated ...

Doxygen documentation of classes, in Modules (1/3)

In e.g. `cylindricalInletVelocityFvPatchVectorField.H`, we see a commented header. After the License description, it continues with:

```
Class
    Foam::cylindricalInletVelocityFvPatchVectorField

Group
    grpInletBoundaryConditions
```

The group is defined for Doxygen in:

```
$FOAM_SRC/finiteVolume/fields/fvPatchFields/doc/fvPatchFieldDoc.H
```

as:

```
\defgroup grpInletBoundaryConditions Inlet
@{
    \ingroup grpBoundaryConditions
    This group contains inlet boundary condition types
@}
```

We see a group name `Inlet`, a general description of the group, and that the group is a subgroup to `grpBoundaryConditions`, which is defined in the same file as:

```
\defgroup grpBoundaryConditions Boundary conditions
@{ This group contains OpenFOAM boundary condition types. ... @}
}
```

Doxygen documentation of classes, in Modules (2/3)

The groups are used by Doxygen to organize the information in Modules at

<https://www.openfoam.com/documentation/guides/latest/doc/index.html>

under (e.g.)

OpenFOAM API / Modules / Boundary conditions

at (e.g.)

https://www.openfoam.com/documentation/guides/latest/api/group__grpBoundaryConditions.html

Here we can see the "Detailed description" of boundary conditions, specified in the file at the previous slide.

The page gives an alphabetic list of the sub-modules/groups that are defined in the file in the previous slide. A couple of them are not found in that file. Find them by:

```
grep -r "\ingroup grpBoundaryConditions" $FOAM_SRC
```

yielding the additional files

```
$FOAM_SRC/regionModels/doc/regionBoundaryConditionsDoc.H
```

```
$FOAM_SRC/thermophysicalModels/basic/derivedFvPatchFields/doc/thermophysicalBoundaryConditionsDoc.H
```



Doxygen documentation of classes, in Modules (3/3)

If we go further to the sub-module/group `Inlet`, we end up at

https://www.openfoam.com/documentation/guides/latest/api/group__grpInletBoundaryConditions.html

Here we see all the classes that have noted that they belong to that group, and at the end we see the "Detailed description" of that group.

If we click on `cylindricalInletVelocityFvPatchVectorField`, we end up at (identify class `Foam::cyli...` in the URL):

https://www.openfoam.com/documentation/guides/latest/api/classFoam_1_1cylindricalInletVelocityFvPatchVectorField.html

At that page we can see the Detailed description of that boundary condition, given in the rest of the commented header of the `*.H` file of that class (click on `More`).

Without checking in detail (it is not very important), it is likely that the URL of the above page is related to the entry in the commented header of the class:

Class

```
Foam::cylindricalInletVelocityFvPatchVectorField
```

Thus, a class can make itself part of the `Modules` structure in Doxygen (if Doxygen is recompiled).



Doxygen documentation of member functions in classes

We see at

https://www.openfoam.com/documentation/guides/latest/api/classFoam_1_1cylindricalInletVelocityFvPatchVectorField.html

that each member function has a short description. Those are taken from the code, where `//–` is found, e.g.:

```
//– Construct from patch, internal field and dictionary  
cylindricalInletVelocityFvPatchVectorField  
(  
    const fvPatch&  
    const DimensionedField<vector, volMesh>&  
    const dictionary&  
);
```