

TimeVaryingMappedFixedValue

interpolates from a set of supplied points in space and time

source code:

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

tutorial: `$FOAM_TUTORIALS/incompressible/simpleFoam/pitzDailyExptInlet/`

Example: You have experimental data you want to use as inflow boundary condition for your case, sampled data from a different OpenFOAM case, ...

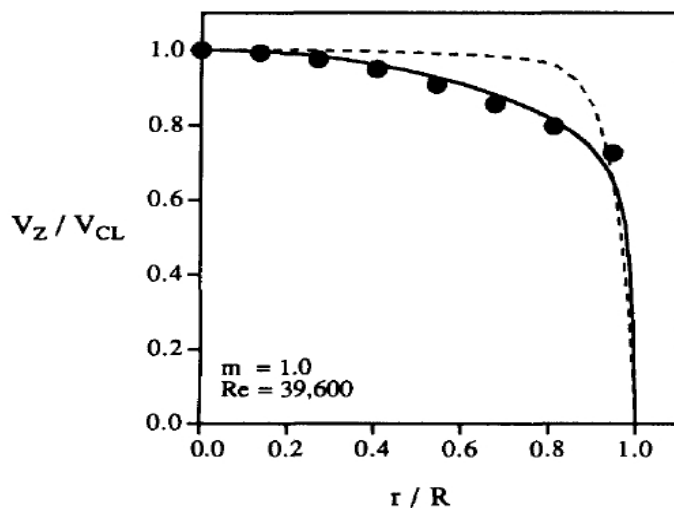


Fig.7(a) from Bolio et al, AIChE (1995)

r/R	v_z/v_{CL}
0	1
0.06	0.99
0.13	0.99
0.19	0.99
0.24	0.98
0.31	0.97
0.37	0.97
0.44	0.95
0.51	0.94
0.57	0.92
0.61	0.9
...	...

```
case
-constant
  -boundaryData
    -PATCH
      -TIME1
        -VARIABLE1
        -VARIABLE2
      -TIME2
      -TIME3
      -points
```

set up a folder named "PATCH" with the name of the patch you want to map the data on, in the case/constant/boundaryData folder;

This "PATCH"-folder contains different time directories for which you want to define mapped values and a file named "points", containing the geometric information;

- if only `TIME1=startTime_of_simulation` is set, these values are used for all time steps;
- if data is not set for all time steps, linear interpolation is done between the data sets of the smaller and larger time steps;

constant/boundaryData/PATCH/points

```
(
// min z
(0 0 0)
(0.000748 0 0)
(0.001705 0 0)
(0.002495 0 0)
...
// max z
(0 0 0.1)
(0.000748 0 0.1)
(0.001705 0 0.1)
(0.002495 0 0.1)
...
)
```

- contains data points
(≥ 3 points have to be defined on a plane - if experimental data is only 1D, define it for two different z-values -> needed for triangulation of points)

- data has to be in standard OpenFOAM units [m] but need not to be ordered

how geometry is evaluated:
- take first point of "point"-file (=p1)
- search for farthest point from p1 (=p2)
- search point that is farthest from p2-p1 (=p3)
- linear interpolation for every face center of chosen patch (weighted to 3 vertices)

constant/boundaryData/PATCH/TIME1/VARIABLE1

```
// Average
(0 0 0)

// Data on points
60
(
//min z
(0 18.9 0)
(0 18.78 0)
(0 18.79 0)
(0 18.67 0)
...
// max z
(0 18.9 0)
(0 18.78 0)
(0 18.79 0)
(0 18.67 0)
...
)
```

- contains variable data

- one entry for each defined point (data entry 5 belongs to point 5...)

- data has to be in standard OpenFOAM units [velocity in m/s]

The boundary condition is defined as following in the 0/VARIABLE1 -file:

```
boundaryField
{
    PATCH
    {
        type                timeVaryingMappedFixedValue;
        setAverage          off; //adjust mapped field to maintain average value
        // perturb          0.0; //change perturbation (default 1e-5) to avoid any
                           ties in triangulating regular geometries
        //fieldName NAME; //use NAME instead of patchname for location of data
    }
}
```