

# OpenFOAM OpenFOAM勉強会 for beginner

2011年8月6日

高橋 功一

(\* )本報告ではVer.2.0.0を使っています

# 本日のお題: functions

controlDictの最後に“Functions”の項目が付く場合がある

右の例では、“outlet”面における  
圧力、流束、流速  
の面積平均値を算出

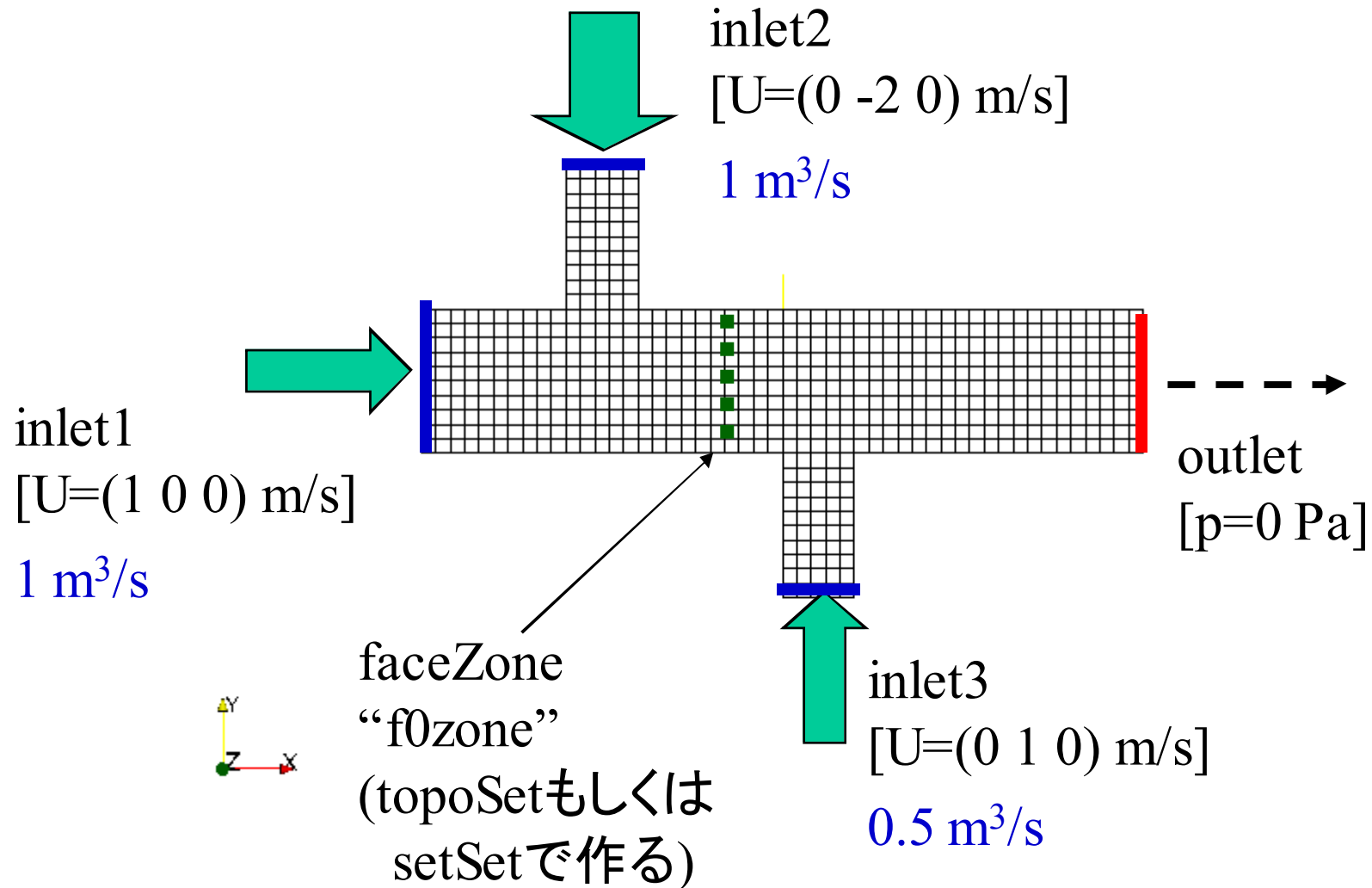
```
functions
{
  faceObj1
  {
    type          faceSource;
    functionObjectLibs ("lib fieldFunctionObjects.so");
    enabled       true;
    outputControl outputTime;
    log           true;
    valueOutput   true;
    source        patch;
    sourceName    outlet;
    operation     areaAverage;
    fields
    (
      p
      phi
      U
    );
  }
}
```

# functionsの使い方

使い方は2通り

- (1)そのままソルバー計算実行  
計算しながら演算値(たとえば出口の平均流速など)を出力する
- (2)ユーティリティ“execFlowFunctionObjects”を実行  
計算終了後のデータ(“0”,“100”,“200”,...)に対して演算値を計算して出力

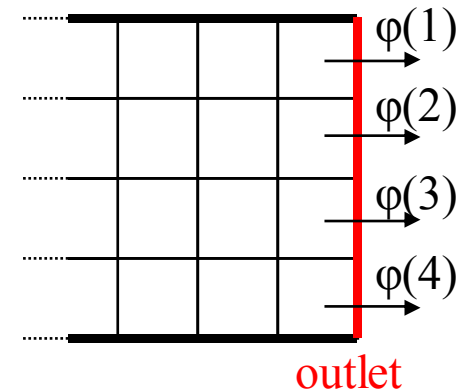
# 例題:2次元混合管



## 例題: controlDictの記述

```
...
...
functions
{
  faceObj1
  {
    type          faceSource;    //faceに対する演算
    functionObjectLibs ("libfieldFunctionObjects.so");
    enabled       true;
    outputControl timeStep; //timestepごとに出力
    log           true;
    valueOutput   false;
    source        patch;
    sourceName    outlet; //outletパッチ
    operation     sum; //総和
    fields
    (
      phi //フェイスを通る流量[m³/s]
    );
  }
}
```

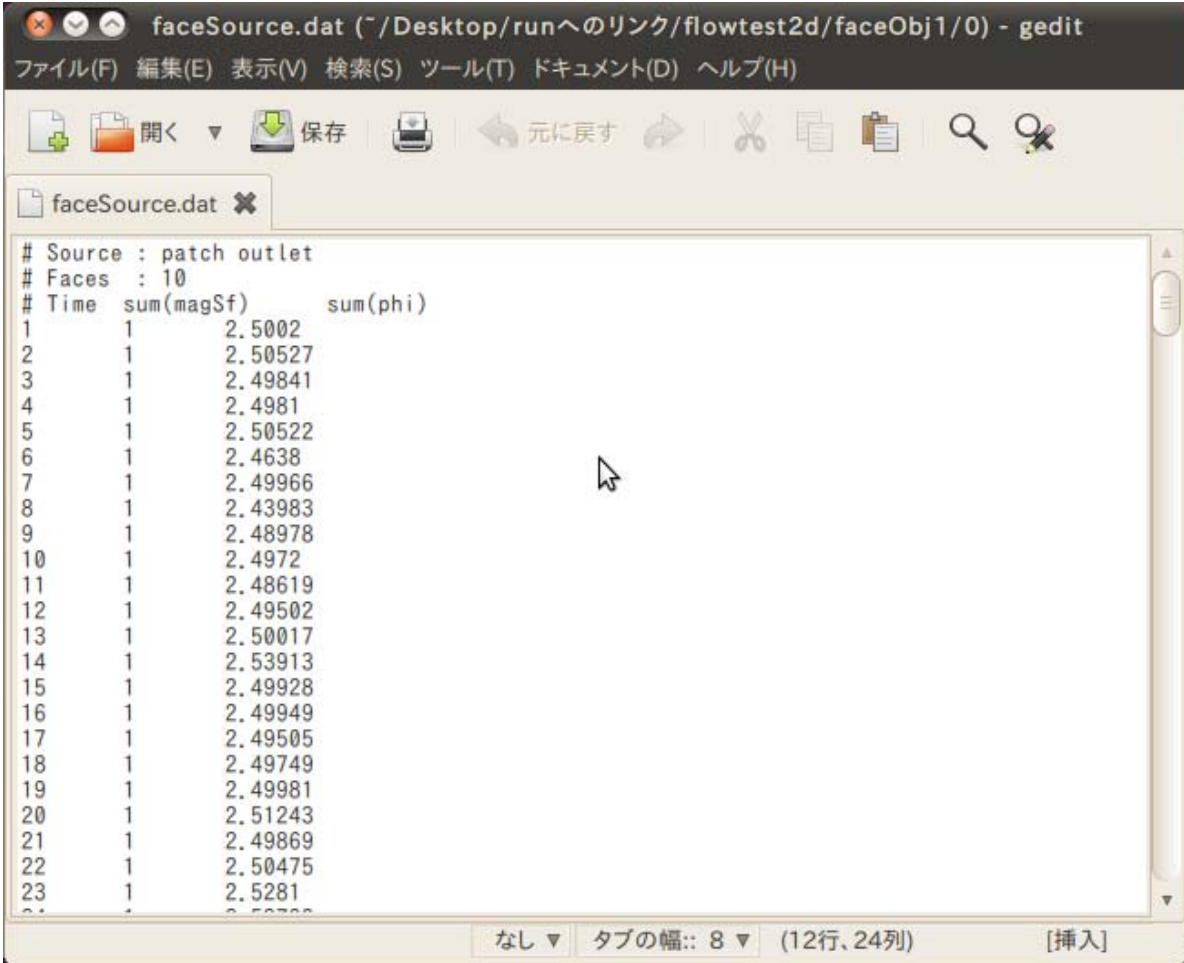
$$\phi(i) = \mathbf{U}_f(i) \cdot d\mathbf{S}(i)$$



Sum(phi)  
=phi(1)+ phi(2)+ phi(3)+ phi(4)  
=outletを通る流量[m³/s]

## 例題: 出力1 ログ

faceObj1(controlDictに書いた名前)のディレクトリができる  
中の“faceSource.dat”ファイルに演算値が出力される



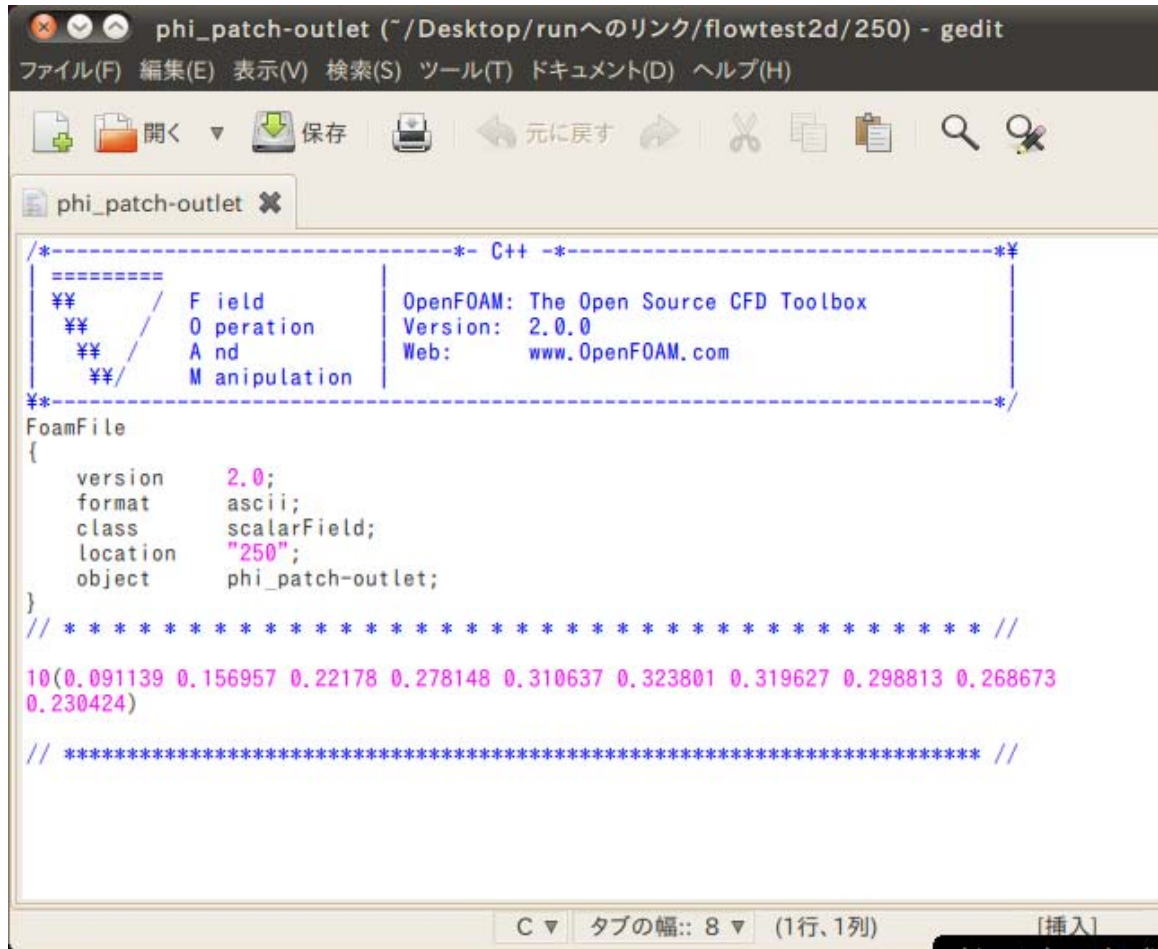
```
# Source : patch outlet
# Faces : 10
# Time  sum(magSf)      sum(phi)
1      1      2.5002
2      1      2.50527
3      1      2.49841
4      1      2.4981
5      1      2.50522
6      1      2.4638
7      1      2.49966
8      1      2.43983
9      1      2.48978
10     1      2.4972
11     1      2.48619
12     1      2.49502
13     1      2.50017
14     1      2.53913
15     1      2.49928
16     1      2.49949
17     1      2.49505
18     1      2.49749
19     1      2.49981
20     1      2.51243
21     1      2.49869
22     1      2.50475
23     1      2.5281
```

## 例題:出力2 ログ

```
k-takahashi@k-takahashi-desktop: ~/OpenFOAM/k-takahashi-2.0.0/  
ファイル(F) 編集(E) 表示(V) 端末(T) ヘルプ(H)  
Time = 1000  
DILUPBiCG: Solving for Ux, Initial residual = 6.92806e-06, Final residual = 6.9  
2806e-06, No Iterations 0  
DILUPBiCG: Solving for Uy, Initial residual = 3.94993e-06, Final residual = 3.9  
4993e-06, No Iterations 0  
DICPCG: Solving for p, Initial residual = 1.97791e-05, Final residual = 7.62157  
e-07, No Iterations 29  
time step continuity errors : sum local = 7.56521e-07, global = -2.34158e-09, cu  
mulative = -0.00355484  
DILUPBiCG: Solving for C, Initial residual = 1.45972e-05, Final residual = 4.16  
032e-06, No Iterations 1  
DILUPBiCG: Solving for epsilon, Initial residual = 1.04866e-05, Final residual  
= 3.26767e-07, No Iterations 1  
DILUPBiCG: Solving for k, Initial residual = 5.65613e-05, Final residual = 1.07  
055e-06, No Iterations 1  
ExecutionTime = 2.0 s ClockTime = 4 s  
faceSource faceObj1 output:  
sum(outlet) for phi = 2.5  
End  
k-takahashi@k-takahashi-desktop:~/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d$
```

logをtrueにすると、terminalに演算値が表示される

## 例題: 出力3 データディレクトリ



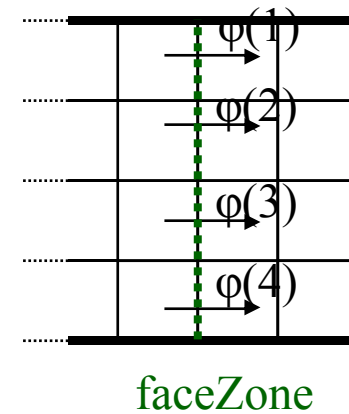
```
phi_patch-outlet (~ /Desktop/runへのリンク/flowtest2d/250) - gedit
ファイル(F) 編集(E) 表示(V) 検索(S) ツール(T) ドキュメント(D) ヘルプ(H)
開く 保存 元に戻す
phi_patch-outlet
/*-----* C++ *-----*/
====
¥¥ / Field OpenFOAM: The Open Source CFD Toolbox
¥¥ / Operation Version: 2.0.0
¥¥ / And Web: www.OpenFOAM.com
¥¥ / Manipulation
/*-----*/
FoamFile
{
  version      2.0;
  format       ascii;
  class        scalarField;
  location     "250";
  object       phi_patch-outlet;
}
// ***** //
10(0.091139 0.156957 0.22178 0.278148 0.310637 0.323801 0.319627 0.298813 0.268673
0.230424)
// ***** //
```

`valueOutput`をtrueにすると、各時刻のデータディレクトリ (“0”, “100”, “200”, ...)内に演算値が格納される  
この例では、“phi\_patch-outlet”



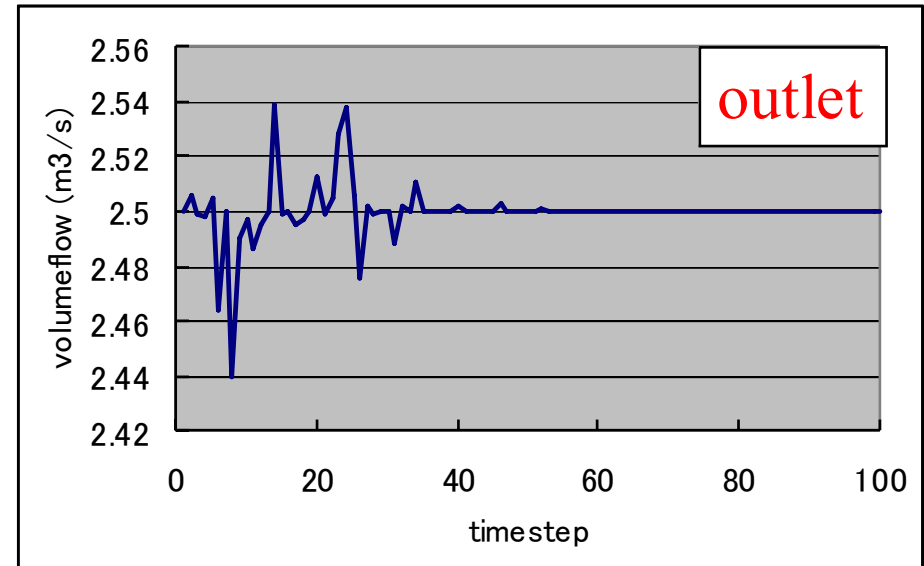
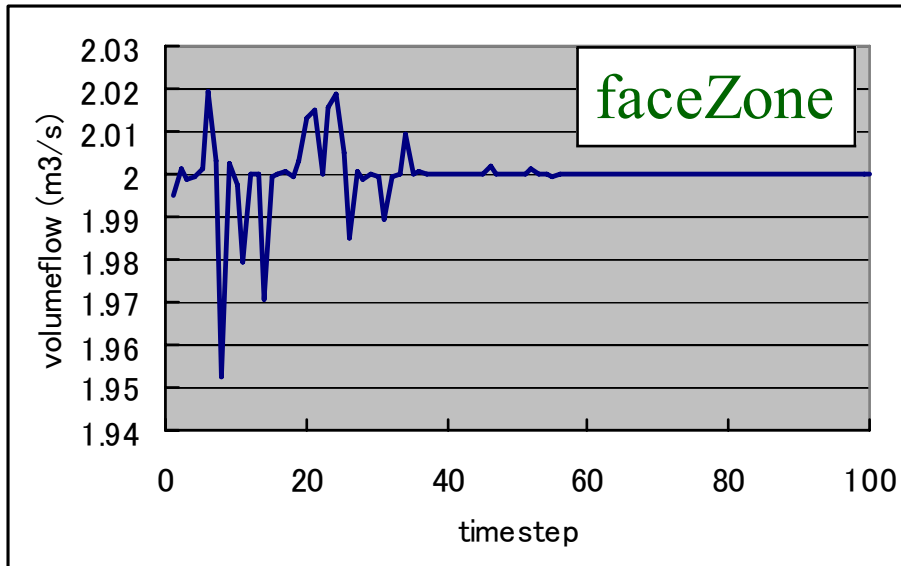
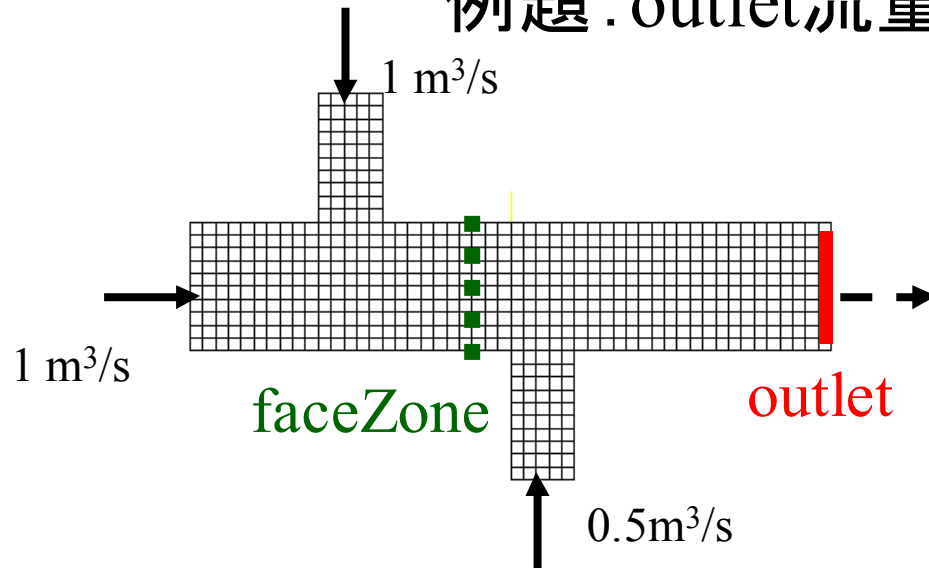
## 例題: contolDictの記述 その2

```
...
...
functions
{
  faceObj2
  {
    type          faceSource;
    functionObjectLibs ("libfieldFunctionObjects.so");
    enabled       true;
    outputControl  timeStep;
    log           true;
    valueOutput   false;
    source        faceZone;
    sourceName    f0zone; //zone名
    operation     sum;    //総和
    fields
    (
      phi        //フェイスゾーンを通る流量[m3/s]
    );
  }
}
```



faceZoneを使えば  
patch(境界)だけではなく  
内部faceでも可

# 例題: outlet流量の推移



40stepくらいで流量が収束(マスバランスの確認)

## もっと詳しくfunctionsを知るために(type)

```
functions
{
  faceObj1
  {
    type      faceSource1; //わざと間違えてみる
    functionObjectLibs ("libfieldFunctionObjects.so");
    enabled   true;
    outputControl  outputTime;
    log       true;
    valueOutput  true;
    source     patch;
    sourceName  outlet;
    operation  areaAverage;
    fields
    (
      phi
    );
  }
}
```

## もっと詳しくfunctionsを知るために(type)

Valid functions are :

12

(

cellSource

//cellの最大最小・平均・総和演算

faceSource

//faceの最大最小・平均・総和演算

fieldAverage

//Field値(U,pなど)の時間平均

fieldMinMax

//Field値(U,pなど)の最大・最小値

nearWallFields

//例題なし

patchProbes

//例題なし

probes

//指定点のField値

readFields

//例題あり・未調査

sets

//例題なし

streamLine

//指定点から流線を出力

surfaceInterpolateFields

//例題なし

surfaces

//任意の面で演算？(難しそう)

)

## もっと詳しくfunctionsを知るために(operation)

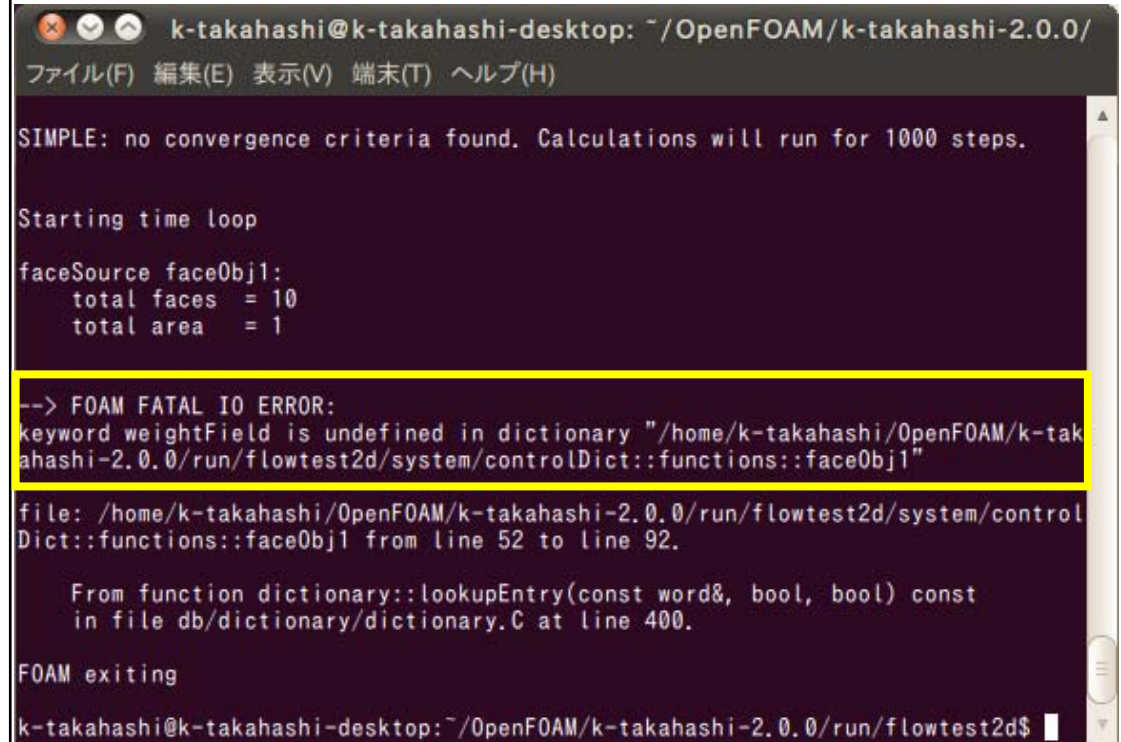
```
functions
{
  faceObj1
  {
    ...
    ...
    source      patch;
    sourceName  outlet;
    operation   summ;
                //わざと間違えてみる

    fields
    (
      phi
    );
  }
}
```

```
summ is not in enumeration:
7
(
  areaAverage //面平均
  areaIntegrate //面積分
  max //最大値
  min //最小値
  none //?
  sum //総和
  weightedAverage
           //重みつき平均
)
```

# もっと詳しくfunctionsを知るために(operation)

```
functions
{
  faceObj1
  {
    ...
    ...
    sourceName    outlet;
    operation weightedAverage;
    fields
    (
      U
    );
  }
}
```



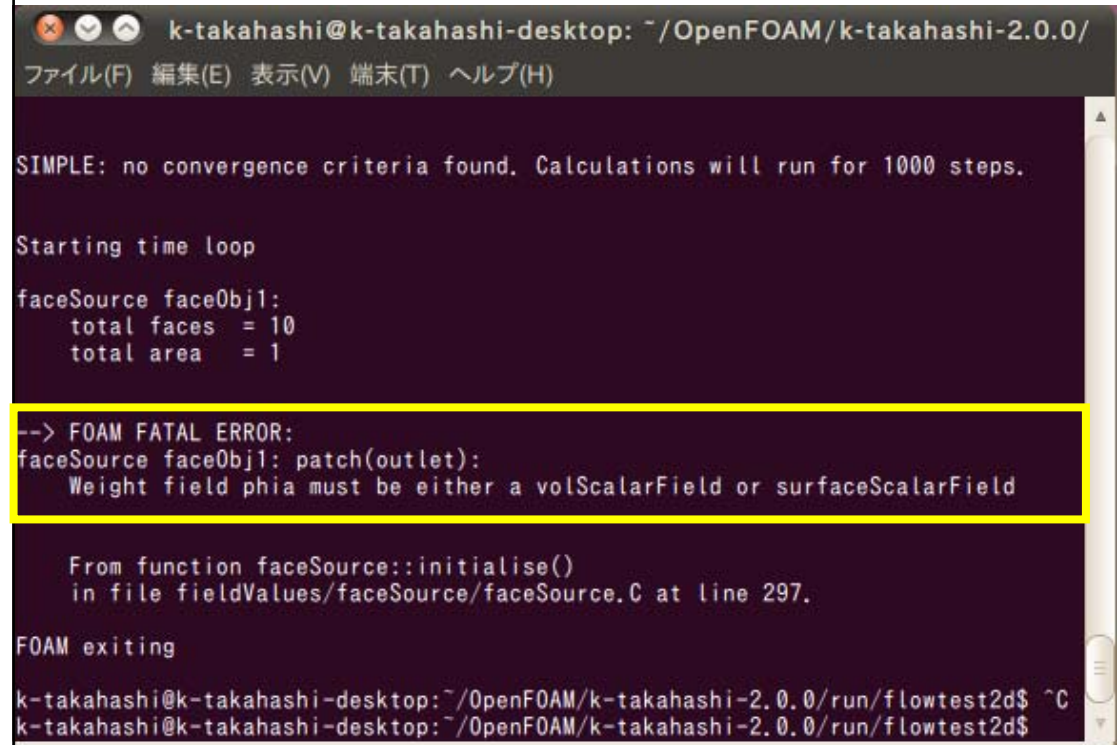
A terminal window showing the execution of an OpenFOAM simulation. The window title is "k-takahashi@k-takahashi-desktop: ~/OpenFOAM/k-takahashi-2.0.0/". The terminal output includes the following text:

```
ファイル(F) 編集(E) 表示(V) 端末(T) ヘルプ(H)
SIMPLE: no convergence criteria found. Calculations will run for 1000 steps.
Starting time loop
faceSource faceObj1:
  total faces = 10
  total area = 1
--> FOAM FATAL IO ERROR:
keyword weightField is undefined in dictionary "/home/k-takahashi/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d/system/controlDict::functions::faceObj1"
file: /home/k-takahashi/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d/system/controlDict::functions::faceObj1 from line 52 to line 92.
From function dictionary::lookupEntry(const word&, bool, bool) const
in file db/dictionary/dictionary.C at line 400.
FOAM exiting
k-takahashi@k-takahashi-desktop:~/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d$
```

weightedAverageを使うとエラーが出る  
“keyword **weightField** is undefined ...”

## もっと詳しくfunctionsを知るために(operation)

```
functions
{
  faceObj1
  {
    ...
    ...
    sourceName    outlet;
    operation weightedAverage;
    weightField aa; //適当に
    fields
    (
      U
    );
  }
}
```

A terminal window showing the execution of a FOAM simulation. The window title is "k-takahashi@k-takahashi-desktop: ~/OpenFOAM/k-takahashi-2.0.0/". The terminal output includes: "SIMPLE: no convergence criteria found. Calculations will run for 1000 steps.", "Starting time loop", "faceSource faceObj1:", "total faces = 10", "total area = 1", and a highlighted error message: "--> FOAM FATAL ERROR: faceSource faceObj1: patch(outlet): Weight field phia must be either a volScalarField or surfaceScalarField". Below the error, it says "From function faceSource::initialise() in file fieldValues/faceSource/faceSource.C at line 297." and "FOAM exiting". The prompt shows the user is in a directory "~/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d\$".

```
k-takahashi@k-takahashi-desktop: ~/OpenFOAM/k-takahashi-2.0.0/
ファイル(F) 編集(E) 表示(V) 端末(T) ヘルプ(H)

SIMPLE: no convergence criteria found. Calculations will run for 1000 steps.

Starting time loop
faceSource faceObj1:
total faces = 10
total area = 1

--> FOAM FATAL ERROR:
faceSource faceObj1: patch(outlet):
Weight field phia must be either a volScalarField or surfaceScalarField

From function faceSource::initialise()
in file fieldValues/faceSource/faceSource.C at line 297.

FOAM exiting

k-takahashi@k-takahashi-desktop:~/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d$ ^C
k-takahashi@k-takahashi-desktop:~/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d$
```

適当に書いたので、当然エラーが出る

“Weight field aa must be either a **volScalarField** or **surfaceScalarField**”

# もっと詳しくfunctionsを知るために(operation)

```
functions
{
  faceObj1
  {
    ...
    ...
    sourceName    outlet;
    operation      weightedAverage;
    weightField    phi;
    //phiはsurfaceScalarFieldの値
    fields
    (
      U
    );
  }
}
```

```
k-takahashi@k-takahashi-desktop: ~/OpenFOAM/k-takahashi-2.0.0/
ファイル(F) 編集(E) 表示(V) 端末(T) ヘルプ(H)
DILUPBiCG: Solving for Ux, Initial residual = 6.92806e-06, Final residual = 6.9
2806e-06, No Iterations 0
DILUPBiCG: Solving for Uy, Initial residual = 3.94993e-06, Final residual = 3.9
4993e-06, No Iterations 0
DICPCG: Solving for p, Initial residual = 1.97791e-05, Final residual = 7.62157
e-07, No Iterations 29
time step continuity errors : sum local = 7.56521e-07, global = -2.34158e-09, cu
mulative = -0.00355484
DILUPBiCG: Solving for C, Initial residual = 1.45972e-05, Final residual = 4.16
032e-06, No Iterations 1
DILUPBiCG: Solving for epsilon, Initial residual = 1.04866e-05, Final residual
= 3.26767e-07, No Iterations 1
DILUPBiCG: Solving for k, Initial residual = 5.65613e-05, Final residual = 1.07
055e-06, No Iterations 1
ExecutionTime = 2.67 s  ClockTime = 4 s

faceSource faceObj1 output:
  weightedAverage(outlet) for U = (2.71081 -0.0182181 -1.39929e-23)
End

k-takahashi@k-takahashi-desktop:~/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d$ ^C
k-takahashi@k-takahashi-desktop:~/OpenFOAM/k-takahashi-2.0.0/run/flowtest2d$
```

めでたく成功

重み付け平均 
$$\bar{U} = \frac{\sum_i U_f(i) \cdot \phi(i)}{\sum_i \phi(i)}$$

疑問:  $U_f(i)$  はどこの値なの  
か...

近接セル値? 補間値?



# もっと詳しくfunctionsを知るために Functionsが書かれたControlDictの例題を探す

```
Find $WM_PROJECT_DIR -name controlDict | xargs grep functions
```

```
k-takahashi@k-takahashi-desktop:~$ find $WM_PROJECT_DIR -name controlDict | xargs grep functions
/opt/openfoam200/tutorials/compressible/rhoPimpleFoam/les/pitzDaily/system/controlDict.functions
/opt/openfoam200/tutorials/compressible/sonicFoam/ras/nacaAirfoil/system/controlDict.functions
/opt/openfoam200/tutorials/multiphase/twoPhaseEulerFoam/bed/system/controlDict.functions
/opt/openfoam200/tutorials/multiphase/twoPhaseEulerFoam/bubbleColumn/system/controlDict.functions
/opt/openfoam200/tutorials/multiphase/twoPhaseEulerFoam/bed2/system/controlDict.functions
/opt/openfoam200/tutorials/multiphase/interDyMFoam/ras/sloshingTank3D3DoF/system/controlDict.functions
/opt/openfoam200/tutorials/multiphase/interDyMFoam/ras/sloshingTank3D/sy stem/controlDict.functions
/opt/openfoam200/tutorials/multiphase/interDyMFoam/ras/sloshingTank2D3DoF/sy stem/controlDict.functions
/opt/openfoam200/tutorials/multiphase/interDyMFoam/ras/sloshingTank2D/sy stem/controlDict.functions
/opt/openfoam200/tutorials/multiphase/interDyMFoam/ras/sloshingTank3D6DoF/sy stem/controlDict.functions
/opt/openfoam200/tutorials/multiphase/cavitatingFoam/les/throttle3D/sy stem/controlDict.functions
/opt/openfoam200/tutorials/multiphase/cavitatingFoam/les/throttle/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/pimpleDyMFoam/wingMotion/wingMotion2D_simpleFoam/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/pimpleDyMFoam/wingMotion/wingMotion2D_pimpleDyMFoam/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/pimpleFoam/TJunction/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/pimpleFoam/TJunctionFan/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/pisoFoam/les/pitzDaily/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/pisoFoam/les/pitzDaily/Dir ectMappe d/system/controlDict.functions
/opt/openfoam200/tutorials/incompressible/simpleFoam/pitzDaily/system/controlDict.functions
/opt/openfoam200/tutorials/incompressible/simpleFoam/motorBike/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/simpleFoam/pitzDaily Expthlet/sy stem/controlDict.functions
/opt/openfoam200/tutorials/incompressible/channelFoam/channel395/system/controlDict.functions
/opt/openfoam200/tutorials/discreteMethods/dsmcFoam/supersonicCorner/system/controlDict.functions
/opt/openfoam200/tutorials/discreteMethods/dsmcFoam/freeSpacePeriodic/system/controlDict.functions
/opt/openfoam200/tutorials/discreteMethods/dsmcFoam/wedge15Ma5/sy stem/controlDict.functions
/opt/openfoam200/tutorials/discreteMethods/dsmcFoam/freeSpaceStream/system/controlDict.functions
/opt/openfoam200/tutorials/combustion/XiFoam/les/pitzDaily/sy stem/controlDict.functions
/opt/openfoam200/tutorials/combustion/XiFoam/les/pitzDaily 3D/sy stem/controlDict.functions
/opt/openfoam200/tutorials/lagrangian/LTSReactingParcelFoam/verticalChannel/system/controlDict.functions
/opt/openfoam200/tutorials/lagrangian/porousExplicitSourceReactingParcelFoam/verticalChannel/system/controlDict.functions
/opt/openfoam200/tutorials/basic/potentialFoam/cylinder/system/controlDict.functions
/opt/openfoam200/etc/controlDict:// NB: the #functions do not work here
/opt/openfoam200/src/postProcessing/functionObjects/field/fieldValues/controlDict.functions
/opt/openfoam200/src/postProcessing/functionObjects/field/fieldMinMax/controlDict.functions
/opt/openfoam200/src/postProcessing/functionObjects/field/fieldAverage/controlDict.functions
/opt/openfoam200/src/postProcessing/functionObjects/utilities/timeActivatedFileUpdate/controlDict.functions
/opt/openfoam200/src/postProcessing/functionObjects/IO/controlDict.functions
/opt/openfoam200/src/postProcessing/functionObjects/systemCall/controlDict.functions
```

38個  
(tutorial:31, source:7)

もっと詳しくfunctionsを知るために

ソース・チュートリアルの例文にはこのほかのtypeの  
パターンもあり

coded

dsmcFields

forcecoeffs

force

partialWrite

writeRegisteredObject

systemCall

timeActivatedFileUpdate

v2.0.0の新機能

モンテカルロ専用？

抵抗係数？

抵抗力？

？

no\_writeのFieldを強制出力？

？

？

内容は未調査。必要に応じて今後調べます