OpenFOAM バージョン比較（1.7.1 と 2.0.0）: interFoam

June 27, 2011

Shinji Nakagawa, Toyama Pref. Univ.

<table>
<tr><th>OpenFOAM 1.7.1</th><th>OpenFOAM 2.0.0</th></tr>
<tr><td></td><td>相違点概略
PISO から PIMPLE へ
boundary ファイルの書き方に変更あり。U 等と統一的な記述方法になった。

大きな変化はない？</td></tr>
<tr><td>ケース ディレクトリ構造
/
system/
controlDict
decomposeParDict
fvSchemes
fvSolultion
setFieldsDict
constant/
transportProperties
g
dynamicMeshDict
turbulencePropertis
polyMesh/
blockMeshDict
boundary
faces
neighbour
owner
points
0/
alpha1
p_rgh
U</td><td>ケース ディレクトリ構造
/
system/
controlDict
decomposeParDict
fvSchemes
fvSolultion
setFieldsDict
constant/
transportProperties
g
dynamicMeshDict
turbulencePropertis
polyMesh/
blockMeshDict
boundary
faces
neighbour
owner
points
0/
alpha1
p
U</td></tr>
<tr><td>system</td><td>system</td></tr>
<tr><td>/*---------------------------------*- C++ -*----------------------------------*\
| ========= | |
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 1.7.1
\\ / A nd	Web: www.OpenFOAM.com
\\/ M anipulation	| *---*/ FoamFile</td><td>/*---------------------------------*- C++ -*----------------------------------*\ | =========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.0.0
\\ / A nd	Web: www.OpenFOAM.com
\\/ M anipulation	| *---*/ FoamFile</td></tr>
</table>

<table>
<tr>
<td>

```
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
application     interFoam;
startFrom       startTime;
startTime       0;
stopAt          endTime;
endTime         1;
deltaT          0.001;
writeControl    adjustableRunTime;
writeInterval   0.05;
purgeWrite      0;
writeFormat     ascii;
writePrecision  6;
writeCompression uncompressed;
timeFormat      general;
timePrecision   6;
runTimeModifiable yes;
adjustTimeStep  yes;
maxCo           0.5;
maxAlphaCo      0.5;
maxDeltaT       1;
// ************************************************************************* //
```

</td>
<td>

```
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
application     interFoam;
startFrom       startTime;
startTime       0;
stopAt          endTime;
endTime         1;
deltaT          0.001;
writeControl    adjustableRunTime;
writeInterval   0.05;
purgeWrite      0;
writeFormat     ascii;
writePrecision  6;
writeCompression uncompressed;
timeFormat      general;
timePrecision   6;
runTimeModifiable yes;
adjustTimeStep  yes;
maxCo           0.5;
maxAlphaCo      0.5;
maxDeltaT       1;
// ************************************************************************* //
```

</td>
</tr>
<tr>
<td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSchemes;
```

</td>
<td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSchemes;
```

</td>
</tr>
</table>

```
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
ddtSchemes
{
    default         Euler;
}
gradSchemes
{
    default         Gauss linear;
}
divSchemes
{
    div(rho*phi,U)  Gauss limitedLinearV 1;
    div(phi,alpha)  Gauss vanLeer;
    div(phirb,alpha) Gauss interfaceCompression;
}
laplacianSchemes
{
    default         Gauss linear corrected;
}
interpolationSchemes
{
    default         linear;
}
snGradSchemes
{
    default         corrected;
}
fluxRequired
{
    default         no;
    p_rgh;
    pcorr;
    alpha1;
}
// ************************************************************************* //
```

```
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
ddtSchemes
{
    default         Euler;
}
gradSchemes
{
    default         Gauss linear;
}
divSchemes
{
    div(rho*phi,U)  Gauss limitedLinearV 1;
    div(phi,alpha)  Gauss vanLeer;
    div(phirb,alpha) Gauss interfaceCompression;
}
laplacianSchemes
{
    default         Gauss linear corrected;
}
interpolationSchemes
{
    default         linear;
}
snGradSchemes
{
    default         corrected;
}
fluxRequired
{
    default         no;
    p_rgh;
    pcorr;
    alpha1;
}
// ************************************************************************* //
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
```

```
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
solvers
{
    pcorr
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-10;
        relTol          0;
    }
    p_rgh
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-07;
        relTol          0.05;
    }
    p_rghFinal
    {
        $p_rgh;
        tolerance       1e-07;
        relTol          0;
    }
    U
    {
        solver          PBiCG;
        preconditioner  DILU;
        tolerance       1e-06;
        relTol          0;
    }
}
PISO
{
    momentumPredictor no;
    nCorrectors     3;
```

```
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
solvers
{
    pcorr
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-10;
        relTol          0;
    }
    p_rgh
    {
        solver          PCG;
        preconditioner  DIC;
        tolerance       1e-07;
        relTol          0.05;
    }
    p_rghFinal
    {
        $p_rgh;
        tolerance       1e-07;
        relTol          0;
    }
    U
    {
        solver          PBiCG;
        preconditioner  DILU;
        tolerance       1e-06;
        relTol          0;
    }
}
PIMPLE
{
    momentumPredictor no;
    nCorrectors     3;
```

```
    nNonOrthogonalCorrectors 0;
    nAlphaCorr      1;
    nAlphaSubCycles 2;
    cAlpha          1;
}
// ************************************************************************* //
```

```
    nNonOrthogonalCorrectors 0;
    nAlphaCorr      1;
    nAlphaSubCycles 2;
    cAlpha          1;
}
// ************************************************************************* //
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      setFieldsDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

defaultFieldValues
(
    volScalarFieldValue alpha1 0
);

regions
(
    boxToCell
    {
        box (0 0 -1) (0.1461 0.292 1);
        fieldValues
        (
            volScalarFieldValue alpha1 1
        );
    }
);

// ************************************************************************* //
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      setFieldsDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

defaultFieldValues
(
    volScalarFieldValue alpha1 0
);

regions
(
    boxToCell
    {
        box (0 0 -1) (0.1461 0.292 1);
        fieldValues
        (
            volScalarFieldValue alpha1 1
        );
    }
);
// ************************************************************************* //
```

<table>
<tr><td>Constant/polyMesh/</td><td></td></tr>
<tr><td><pre>
/*---------------------------------*- C++ -*-----------------------------------*\
=========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 1.7.1
\\ / A nd	Web: www.OpenFOAM.com
\\/ M anipulation	
---/
FoamFile
{
    version   2.0;
    format   ascii;
    class   polyBoundaryMesh;
    location   "constant/polyMesh";
    object   boundary;
}
// * //

5
(
    leftWall
    {
        type   wall;
        nFaces   50;
        startFace   4432;
    }
    rightWall
    {
        type   wall;
        nFaces   50;
        startFace   4482;
    }
    lowerWall
    {
        type   wall;
        nFaces   62;
        startFace   4532;
    }
    atmosphere
    {
        type   patch;
        nFaces   46;
</pre></td><td><pre>
/*---------------------------------*- C++ -*-----------------------------------*\
=========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.0.0
\\ / A nd	Web: www.OpenFOAM.com
\\/ M anipulation	
---/
FoamFile
{
    version   2.0;
    format   ascii;
    class   polyBoundaryMesh;
    location   "constant/polyMesh";
    object   boundary;
}
// * //

5
(
    leftWall
    {
        type   wall;
        nFaces   50;
        startFace   4432;
    }
    rightWall
    {
        type   wall;
        nFaces   50;
        startFace   4482;
    }
    lowerWall
    {
        type   wall;
        nFaces   62;
        startFace   4532;
    }
    atmosphere
    {
        type   patch;
        nFaces   46;
</pre></td></tr>
</table>

```
        startFace       4594;
    }
    defaultFaces
    {
        type            empty;
        nFaces          4536;
        startFace       4640;
    }
)
// ************************************************************************* //
```

```
        startFace       4594;
    }
    defaultFaces
    {
        type            empty;
        nFaces          4536;
        startFace       4640;
    }
)
// ************************************************************************* //
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 0.146;

vertices
(
    (0 0 0)
    (2 0 0)
    (2.16438 0 0)
    (4 0 0)
    (0 0.32876 0)
    (2 0.32876 0)
    (2.16438 0.32876 0)
    (4 0.32876 0)
    (0 4 0)
    (2 4 0)
    (2.16438 4 0)
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 0.146;

vertices
(
    (0 0 0)
    (2 0 0)
    (2.16438 0 0)
    (4 0 0)
    (0 0.32876 0)
    (2 0.32876 0)
    (2.16438 0.32876 0)
    (4 0.32876 0)
    (0 4 0)
    (2 4 0)
    (2.16438 4 0)
```

1.7.1

```
    (4 4 0)
    (0 0 0.1)
    (2 0 0.1)
    (2.16438 0 0.1)
    (4 0 0.1)
    (0 0.32876 0.1)
    (2 0.32876 0.1)
    (2.16438 0.32876 0.1)
    (4 0.32876 0.1)
    (0 4 0.1)
    (2 4 0.1)
    (2.16438 4 0.1)
    (4 4 0.1)
);

blocks
(
    hex (0 1 5 4 12 13 17 16) (23 8 1) simpleGrading (1 1 1)
    hex (2 3 7 6 14 15 19 18) (19 8 1) simpleGrading (1 1 1)
    hex (4 5 9 8 16 17 21 20) (23 42 1) simpleGrading (1 1 1)
    hex (5 6 10 9 17 18 22 21) (4 42 1) simpleGrading (1 1 1)
    hex (6 7 11 10 18 19 23 22) (19 42 1) simpleGrading (1 1 1)
);

edges
(
);

patches
(
    wall leftWall
    (
        (0 12 16 4)
        (4 16 20 8)
    )
    wall rightWall
    (
        (7 19 15 3)
```

2.0.0

```
    (4 4 0)
    (0 0 0.1)
    (2 0 0.1)
    (2.16438 0 0.1)
    (4 0 0.1)
    (0 0.32876 0.1)
    (2 0.32876 0.1)
    (2.16438 0.32876 0.1)
    (4 0.32876 0.1)
    (0 4 0.1)
    (2 4 0.1)
    (2.16438 4 0.1)
    (4 4 0.1)
);

blocks
(
    hex (0 1 5 4 12 13 17 16) (23 8 1) simpleGrading (1 1 1)
    hex (2 3 7 6 14 15 19 18) (19 8 1) simpleGrading (1 1 1)
    hex (4 5 9 8 16 17 21 20) (23 42 1) simpleGrading (1 1 1)
    hex (5 6 10 9 17 18 22 21) (4 42 1) simpleGrading (1 1 1)
    hex (6 7 11 10 18 19 23 22) (19 42 1) simpleGrading (1 1 1)
);

edges
(
);

boundary
(
    leftWall
    {
        type wall;
        faces
        (
            (0 12 16 4)
            (4 16 20 8)
        );
    }
    rightWall
    {
        type wall;
```

<pre>   (11 23 19 7)
)
 	
    wall lowerWall
    (
        (0 1 13 12)
        (1 5 17 13)
        (5 6 18 17)
        (2 14 18 6)
        (2 3 15 14)
)
 	
    patch atmosphere
    (
        (8 20 21 9)
        (9 21 22 10)
        (10 22 23 11)
)
);
 	
mergePatchPairs
(
);
 	
// *** //</pre>
	<pre> faces
	        (
	            (7 19 15 3)
	            (11 23 19 7)
	        );
	    }
	    lowerWall
	    {
	        type wall;
	        faces
	        (
	            (0 1 13 12)
	            (1 5 17 13)
	            (5 6 18 17)
	            (2 14 18 6)
	            (2 3 15 14)
	        );
	    }
	    atmosphere
	    {
	        type patch;
	        faces
	        (
	            (8 20 21 9)
	            (9 21 22 10)
	            (10 22 23 11)
	        );
	    }
	);
	
	mergePatchPairs
	(
	);
	
	// *** //</pre>

constant	

<pre>/*--------------------------------*- C++ -*----------------------------------*\
| ========= | |
| \\   / F ield | OpenFOAM: The Open Source CFD Toolbox |
| \\   /   O peration | Version: 1.7.1 |
| \\ /   A nd | Web: www.OpenFOAM.com |</pre>
	<pre>/*--------------------------------*- C++ -*----------------------------------*\
	| ========= | |
	| \\ / F ield | OpenFOAM: The Open Source CFD Toolbox |
	| \\ / O peration | Version: 2.0.0 |
	| \\ / A nd | Web: www.OpenFOAM.com |</pre>

<table>
<tr>
<td>

```
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       uniformDimensionedVectorField;
    location    "constant";
    object      g;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 1 -2 0 0 0 0];
value           ( 0 -9.81 0 );

// ************************************************************************* //
```

</td>
<td>

```
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       uniformDimensionedVectorField;
    location    "constant";
    object      g;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 1 -2 0 0 0 0];
value           ( 0 -9.81 0 );
// ************************************************************************* //
```

</td>
</tr>
<tr>
<td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

phase1
{
    transportModel  Newtonian;
    nu              nu [ 0 2 -1 0 0 0 0 ] 1e-06;
    rho             rho [ 1 -3 0 0 0 0 0 ] 1000;
    CrossPowerLawCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 1e-06;
```

</td>
<td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

phase1
{
    transportModel  Newtonian;
    nu              nu [ 0 2 -1 0 0 0 0 ] 1e-06;
    rho             rho [ 1 -3 0 0 0 0 0 ] 1000;
    CrossPowerLawCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 1e-06;
```

</td>
</tr>
</table>

```
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        m               m [ 0 0 1 0 0 0 0 ] 1;
        n               n [ 0 0 0 0 0 0 0 ] 0;
    }

    BirdCarreauCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 0.0142515;
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        k               k [ 0 0 1 0 0 0 0 ] 99.6;
        n               n [ 0 0 0 0 0 0 0 ] 0.1003;
    }
}

phase2
{
    transportModel  Newtonian;
    nu              nu [ 0 2 -1 0 0 0 0 ] 1.48e-05;
    rho             rho [ 1 -3 0 0 0 0 0 ] 1;
    CrossPowerLawCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 1e-06;
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        m               m [ 0 0 1 0 0 0 0 ] 1;
        n               n [ 0 0 0 0 0 0 0 ] 0;
    }

    BirdCarreauCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 0.0142515;
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        k               k [ 0 0 1 0 0 0 0 ] 99.6;
        n               n [ 0 0 0 0 0 0 0 ] 0.1003;
    }
}

sigma           sigma [ 1 0 -2 0 0 0 0 ] 0.07;
// ************************************************************************* //
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
```

```
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        m               m [ 0 0 1 0 0 0 0 ] 1;
        n               n [ 0 0 0 0 0 0 0 ] 0;
    }

    BirdCarreauCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 0.0142515;
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        k               k [ 0 0 1 0 0 0 0 ] 99.6;
        n               n [ 0 0 0 0 0 0 0 ] 0.1003;
    }
}

phase2
{
    transportModel  Newtonian;
    nu              nu [ 0 2 -1 0 0 0 0 ] 1.48e-05;
    rho             rho [ 1 -3 0 0 0 0 0 ] 1;
    CrossPowerLawCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 1e-06;
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        m               m [ 0 0 1 0 0 0 0 ] 1;
        n               n [ 0 0 0 0 0 0 0 ] 0;
    }

    BirdCarreauCoeffs
    {
        nu0             nu0 [ 0 2 -1 0 0 0 0 ] 0.0142515;
        nuInf           nuInf [ 0 2 -1 0 0 0 0 ] 1e-06;
        k               k [ 0 0 1 0 0 0 0 ] 99.6;
        n               n [ 0 0 0 0 0 0 0 ] 0.1003;
    }
}

sigma           sigma [ 1 0 -2 0 0 0 0 ] 0.07;
// ************************************************************************* //
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
```

```
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      turbulenceProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

simulationType  laminar;

// ************************************************************************* //
```

```
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      turbulenceProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

simulationType  laminar;

// ************************************************************************* //
```

/0/

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      alpha;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 0 0 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    leftWall
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      alpha;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 0 0 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    leftWall
```

```
    {
        type            zeroGradient;
    }

    rightWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    atmosphere
    {
        type            inletOutlet;
        inletValue      uniform 0;
        value           uniform 0;
    }

    defaultFaces
    {
        type            empty;
    }
}
// ************************************************************************* //
```

```
    {
        type            zeroGradient;
    }

    rightWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    atmosphere
    {
        type            inletOutlet;
        inletValue      uniform 0;
        value           uniform 0;
    }

    defaultFaces
    {
        type            empty;
    }
}
// ************************************************************************* //
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p_rgh;
}
```

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p_rgh;
}
```

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [1 -1 -2 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    leftWall
    {
        type            buoyantPressure;
        value           uniform 0;
    }

    rightWall
    {
        type            buoyantPressure;
        value           uniform 0;
    }

    lowerWall
    {
        type            buoyantPressure;
        value           uniform 0;
    }

    atmosphere
    {
        type            totalPressure;
        p0              uniform 0;
        U               U;
        phi             phi;
        rho             rho;
        psi             none;
        gamma           1;
        value           uniform 0;
    }

    defaultFaces
    {
        type            empty;
    }
```

```
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [1 -1 -2 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    leftWall
    {
        type            buoyantPressure;
        value           uniform 0;
    }

    rightWall
    {
        type            buoyantPressure;
        value           uniform 0;
    }

    lowerWall
    {
        type            buoyantPressure;
        value           uniform 0;
    }

    atmosphere
    {
        type            totalPressure;
        p0              uniform 0;
        U               U;
        phi             phi;
        rho             rho;
        psi             none;
        gamma           1;
        value           uniform 0;
    }

    defaultFaces
    {
        type            empty;
    }
```

<table>
<tr><td>

```
}

// ************************************************************************* //
```

</td><td>

```
}

// ************************************************************************* //
```

</td></tr>
<tr><td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  1.7.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    location    "0";
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    leftWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    rightWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    lowerWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    atmosphere
```

</td><td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.0.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    location    "0";
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    leftWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    rightWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    lowerWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    atmosphere
```

</td></tr>
</table>

<table>
<tr>
<td>

```
    {
        type            pressureInletOutletVelocity;
        value           uniform (0 0 0);
    }
    defaultFaces
    {
        type            empty;
    }
}


// ************************************************************************* //
```

</td>
<td>

```
    {
        type            pressureInletOutletVelocity;
        value           uniform (0 0 0);
    }
    defaultFaces
    {
        type            empty;
    }
}


// ************************************************************************* //
```

</td>
</tr>
<tr><td></td><td></td></tr>
<tr><td>ソースコード</td><td>ソースコード</td></tr>
<tr><td>interFoam.C</td><td>interFoam.C</td></tr>
<tr>
<td>

```
/*---------------------------------------------------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     |
    \\  /    A nd           | Copyright (C) 1991-2010 OpenCFD Ltd.
     \\/     M anipulation  |
-------------------------------------------------------------------------------
License
    This file is part of OpenFOAM.

    OpenFOAM is free software: you can redistribute it and/or modify it
    under the terms of the GNU General Public License as published by
    the Free Software Foundation, either version 3 of the License, or
    (at your option) any later version.

    OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
    ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
    FITNESS FOR A PARTICULAR PURPOSE.  See the GNU General Public License
    for more details.

    You should have received a copy of the GNU General Public License
    along with OpenFOAM.  If not, see <http://www.gnu.org/licenses/>.

Application
    interFoam

Description
```

</td>
<td>

```
/*---------------------------------------------------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     |
    \\  /    A nd           | Copyright (C) 2004-2011 OpenCFD Ltd.
     \\/     M anipulation  |
-------------------------------------------------------------------------------
License
    This file is part of OpenFOAM.

    OpenFOAM is free software: you can redistribute it and/or modify it
    under the terms of the GNU General Public License as published by
    the Free Software Foundation, either version 3 of the License, or
    (at your option) any later version.

    OpenFOAM is distributed in the hope that it will be useful, but WITHOUT
    ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or
    FITNESS FOR A PARTICULAR PURPOSE.  See the GNU General Public License
    for more details.

    You should have received a copy of the GNU General Public License
    along with OpenFOAM.  If not, see <http://www.gnu.org/licenses/>.

Application
    interFoam

Description
```

</td>
</tr>
</table>

```
    Solver for 2 incompressible, isothermal immiscible fluids using a VOF
    (volume of fluid) phase-fraction based interface capturing approach.

    The momentum and other fluid properties are of the "mixture" and a single
    momentum equation is solved.

    Turbulence modelling is generic, i.e.  laminar, RAS or LES may be selected.

    For a two-fluid approach see twoPhaseEulerFoam.

\*---------------------------------------------------------------------------*/

#include "fvCFD.H"
#include "MULES.H"
#include "subCycle.H"
#include "interfaceProperties.H"
#include "twoPhaseMixture.H"
#include "turbulenceModel.H"
#include "interpolationTable.H"


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

int main(int argc, char *argv[])
{
    #include "setRootCase.H"
    #include "createTime.H"
    #include "createMesh.H"

    #include "readPISOControls.H"

    #include "initContinuityErrs.H"
    #include "createFields.H"
    #include "readTimeControls.H"
    #include "correctPhi.H"
    #include "CourantNo.H"
    #include "setInitialDeltaT.H"

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

    Info<< "\nStarting time loop\n" << endl;
```

```
    Solver for 2 incompressible, isothermal immiscible fluids using a VOF
    (volume of fluid) phase-fraction based interface capturing approach.

    The momentum and other fluid properties are of the "mixture" and a single
    momentum equation is solved.

    Turbulence modelling is generic, i.e. laminar, RAS or LES may be selected.

    For a two-fluid approach see twoPhaseEulerFoam.

\*---------------------------------------------------------------------------*/

#include "fvCFD.H"
#include "MULES.H"
#include "subCycle.H"
#include "interfaceProperties.H"
#include "twoPhaseMixture.H"
#include "turbulenceModel.H"
#include "interpolationTable.H"
#include "pimpleControl.H"

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

int main(int argc, char *argv[])
{
    #include "setRootCase.H"
    #include "createTime.H"
    #include "createMesh.H"

    pimpleControl pimple(mesh);

    #include "initContinuityErrs.H"
    #include "createFields.H"
    #include "readTimeControls.H"
    #include "correctPhi.H"
    #include "CourantNo.H"
    #include "setInitialDeltaT.H"

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

    Info<< "\nStarting time loop\n" << endl;
```

```
    while (runTime.run())
    {
        #include "readPISOControls.H"
        #include "readTimeControls.H"
        #include "CourantNo.H"
        #include "alphaCourantNo.H"
        #include "setDeltaT.H"

        runTime++;

        Info<< "Time = " << runTime.timeName() << nl << endl;

        twoPhaseProperties.correct();

        #include "alphaEqnSubCycle.H"




        #include "UEqn.H"

        // --- PISO loop
        for (int corr=0; corr<nCorr; corr++)
        {
            #include "pEqn.H"
        }



        turbulence->correct();



        runTime.write();

        Info<< "ExecutionTime = " << runTime.elapsedCpuTime() << " s"
            << "  ClockTime = " << runTime.elapsedClockTime() << " s"
            << nl << endl;
    }

    Info<< "End\n" << endl;
```

```
    while (runTime.run())
    {

        #include "readTimeControls.H"
        #include "CourantNo.H"
        #include "alphaCourantNo.H"
        #include "setDeltaT.H"

        runTime++;

        Info<< "Time = " << runTime.timeName() << nl << endl;

        twoPhaseProperties.correct();

        #include "alphaEqnSubCycle.H"

        // --- Pressure-velocity PIMPLE corrector loop
        for (pimple.start(); pimple.loop(); pimple++)
        {
            #include "UEqn.H"

            // --- PISO loop
            for (int corr=0; corr<pimple.nCorr(); corr++)
            {
                #include "pEqn.H"
            }

            if (pimple.turbCorr())
            {
                turbulence->correct();
            }
        }

        runTime.write();

        Info<< "ExecutionTime = " << runTime.elapsedCpuTime() << " s"
            << "  ClockTime = " << runTime.elapsedClockTime() << " s"
            << nl << endl;
    }

    Info<< "End\n" << endl;
```

return 0; } // *** //	return 0; } // *** //