OpenFOAM バージョン比較（2.1.0 と 2.2.0）： icoUncoupledKinematicParcelFoam

April 26, 2013

Suguru Iriyama
Nakagawa-Lab, Mechanical Engineering Dept., Toyama Pref. Univ.

<table>
<tr><td>hopper/hopperInitialState(ver 2.1.0)</td><td>hopper/hopperInitialState(ver 2.2.0)</td></tr>
<tr><td>ケース ディレクトリ構造
/
system/
controlDict
decomposeParDict
fvSchemes
fvSolultion
constant/
g
kinematicCloudPositions
kinematicCloudProperties
RASProperties
transportProperties
turbulencePropertis
polyMesh/
blockMeshDict
boundary
0/
U</td><td>ケース ディレクトリ構造
/
system/
controlDict
decomposeParDict
fvSchemes
fvSolultion
constant/
g
kinematicCloudPositions
kinematicCloudProperties
RASProperties
transportProperties
turbulencePropertis
polyMesh/
blockMeshDict
boundary
0/
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: 2.1.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	| *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "system"; object controlDict; } // * //</td><td>/*---------------------------------*- C++ -*-----------------------------------*\ | =========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.2.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	| *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "system"; object controlDict; } // * //</td></tr>
</table>

application icoUncoupledKinematicParcelFoam; startFrom startTime; startTime 0; stopAt endTime; endTime 0.25; deltaT 5e-5; writeControl runTime; writeInterval 0.05; purgeWrite 0; writeFormat ascii; writePrecision 6; writeCompression uncompressed; timeFormat general; timePrecision 6; runTimeModifiable yes; // *** //	application icoUncoupledKinematicParcelFoam; startFrom startTime; startTime 0; stopAt endTime; endTime 0.25; deltaT 5e-5; writeControl runTime; writeInterval 0.05; purgeWrite 0; writeFormat ascii; writePrecision 6; writeCompression uncompressed; timeFormat general; timePrecision 6; runTimeModifiable yes; // *** //
/*---------------------------------*- C++ -*-----------------------------------*\ | ========= | |	/*---------------------------------*- C++ -*-----------------------------------*\ | ========= | |

```
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      decomposeParDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

numberOfSubdomains 4;

method          simple;

simpleCoeffs
{
    n               ( 4 1 1 );
    delta           0.001;
}

hierarchicalCoeffs
{
    n               ( 4 1 1 );
    delta           0.001;
    order           xyz;
}

manualCoeffs
{
    dataFile        "";
}
```

```
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.2.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      decomposeParDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

numberOfSubdomains 4;

method          simple;

simpleCoeffs
{
    n               ( 4 1 1 );
    delta           0.001;
}

hierarchicalCoeffs
{
    n               ( 4 1 1 );
    delta           0.001;
    order           xyz;
}

manualCoeffs
{
    dataFile        "";
}
```

<table>
<tr><td>
distributed   no;

roots   ( );

//
*** //
</td><td>
distributed   no;

roots   ( );

//
*** //
</td></tr>
<tr><td>
/*--------------------------------*- C++ -*----------------------------------*\
| ========= | |
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.1.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	| *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "system"; object fvSchemes; } // * // ddtSchemes { default none; } gradSchemes { default none; } divSchemes
</td><td>
/*--------------------------------*- C++ -*----------------------------------*\ | =========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.2.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	| *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "system"; object fvSchemes; } // * // ddtSchemes { default none; } gradSchemes { default none; } divSchemes
</td></tr>
</table>

{ default none; } laplacianSchemes { default none; } interpolationSchemes { default none; } // *** //	{ default none; } laplacianSchemes { default none; } interpolationSchemes { default none; } // *** //
/*--------------------------------*- C++ -*----------------------------------*\\ | ========= | | | \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox | | \\\\ / O peration | Version: 2.1.0 | | \\\\ / A nd | Web: www.OpenFOAM.org | | \\\\/ M anipulation | | *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "system"; object fvSolution; } // * // solvers {	/*--------------------------------*- C++ -*----------------------------------*\\ | ========= | | | \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox | | \\\\ / O peration | Version: 2.2.0 | | \\\\ / A nd | Web: www.OpenFOAM.org | | \\\\/ M anipulation | | *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "system"; object fvSolution; } // * // solvers {

<table>
<tr><td>}

//
****
***** //</td><td>}

//
****
***** //</td></tr>
<tr><td>constant</td><td>constant</td></tr>
<tr><td>/*---------------------------------*- C++ -*-----------------------------------*\
| ========= | |
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.1.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ 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>/*---------------------------------*- C++ -*-----------------------------------*\ | =========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.2.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ 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: 2.1.0
\\ / A nd	Web: www.OpenFOAM.org |</td><td>/*---------------------------------*- C++ -*-----------------------------------*\ | =========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.2.0
\\ / A nd	Web: www.OpenFOAM.org |</td></tr>
</table>

<table>
<tr>
<td>| \\/   M anipulation | |
---/
FoamFile
{
    version   2.0;
    format   ascii;
    class   vectorField;
    object   kinematicCloudPositions;
}
// * //

(
(0.01 0.299 0.0032)
(0.01 0.305 0.0032)
(0.01 0.311 0.0032)
(0.01 0.317 0.0032)
(0.01 0.323 0.0032)
(0.01 0.329 0.0032)
~~~~~~~~~~~~~~~
(0.286 0.473 0.0032)
(0.286 0.479 0.0032)
(0.286 0.485 0.0032)
(0.286 0.491 0.0032)
(0.286 0.497 0.0032)
)

//
****
***** //</td>
<td>| \\/   M anipulation | |
---/
FoamFile
{
    version   2.0;
    format   ascii;
    class   vectorField;
    object   kinematicCloudPositions;
}
// * //

(
(0.01 0.299 0.0032)
(0.01 0.305 0.0032)
(0.01 0.311 0.0032)
(0.01 0.317 0.0032)
(0.01 0.323 0.0032)
(0.01 0.329 0.0032)
~~~~~~~~~~~~~~~
(0.286 0.473 0.0032)
(0.286 0.479 0.0032)
(0.286 0.485 0.0032)
(0.286 0.491 0.0032)
(0.286 0.497 0.0032)
)

//
****
***** //</td>
</tr>
<tr>
<td>/*--------------------------------*- C++ -*----------------------------------*\
| ========= | |
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.1.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	|</td>
<td>/*--------------------------------*- C++ -*----------------------------------*\ | =========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.2.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	|</td>
</tr>
</table>

<table>
<tr>
<td>

```
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      reactingCloud1Properties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

solution
{
    active          true;
    coupled         false;
    transient       yes;
    cellValueSourceCorrection off;

    interpolationSchemes
    {
        rho             cell;
        U               cellPoint;
        mu              cell;
    }

    integrationSchemes
    {
        U               Euler;
    }
}

constantProperties
{
    parcelTypeId    1;

    rhoMin          1e-15;
    minParticleMass 1e-15;
```

</td>
<td>

```
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      reactingCloud1Properties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

solution
{
    active          true;
    coupled         false;
    transient       yes;
    cellValueSourceCorrection off;

    interpolationSchemes
    {
        rho             cell;
        U               cellPoint;
        mu              cell;
    }

    integrationSchemes
    {
        U               Euler;
    }
}

constantProperties
{
```

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

2.1.0	2.2.0
<pre> rho0 964; youngsModulus 6e8; poissonsRatio 0.35; constantVolume false; } subModels { particleForces { sphereDrag; gravity; } injectionModel manualInjection; manualInjectionCoeffs { massTotal 0; parcelBasisType fixed; nParticle 1; SOI 0; positionsFile "kinematicCloudPositions"; U0 (0 0 0); sizeDistribution { type fixedValue; fixedValueDistribution { value 0.006; } } }</pre>	<pre> rho0 964; youngsModulus 6e8; poissonsRatio 0.35; constantVolume false; } subModels { particleForces { sphereDrag; gravity; } injectionModels { model1 { type manualInjection; massTotal 0; parcelBasisType fixed; nParticle 1; SOI 0; positionsFile "kinematicCloudPositions"; U0 (0 0 0); sizeDistribution { type fixedValue; fixedValueDistribution { value 0.006; } } } }</pre>

```
dispersionModel none;

patchInteractionModel standardWallInteraction;

heatTransferModel none;

surfaceFilmModel none;

collisionModel pairCollision;

radiation off;

pairCollisionCoeffs
{
    // Maximum possible particle diameter expected at any time
    maxInteractionDistance  0.006;

    writeReferredParticleCloud no;

    pairModel pairSpringSliderDashpot;

    pairSpringSliderDashpotCoeffs
    {
        useEquivalentSize   no;
        alpha               0.12;
        b                   1.5;
        mu                  0.52;
        cohesionEnergyDensity 0;
        collisionResolutionSteps 12;
    };

    wallModel    wallLocalSpringSliderDashpot;

    wallLocalSpringSliderDashpotCoeffs
    {
        useEquivalentSize no;
```

```
dispersionModel none;

patchInteractionModel standardWallInteraction;

heatTransferModel none;

surfaceFilmModel none;

collisionModel pairCollision;

radiation off;

pairCollisionCoeffs
{
    // Maximum possible particle diameter expected at any time
    maxInteractionDistance  0.006;

    writeReferredParticleCloud no;

    pairModel pairSpringSliderDashpot;

    pairSpringSliderDashpotCoeffs
    {
        useEquivalentSize   no;
        alpha               0.12;
        b                   1.5;
        mu                  0.52;
        cohesionEnergyDensity 0;
        collisionResolutionSteps 12;
    };

    wallModel    wallLocalSpringSliderDashpot;

    wallLocalSpringSliderDashpotCoeffs
    {
        useEquivalentSize no;
```

<table>
<tr>
<td>
<pre>
            collisionResolutionSteps 12;
            walls
            {
                youngsModulus   1e10;
                poissonsRatio   0.23;
                alpha   0.12;
                b   1.5;
                mu   0.43;
                cohesionEnergyDensity 0;
            }
            frontAndBack
            {
                youngsModulus   1e10;
                poissonsRatio   0.23;
                alpha   0.12;
                b   1.5;
                mu   0.1;
                cohesionEnergyDensity 0;
            }
        };
    }

    standardWallInteractionCoeffs
    {
        type   rebound;
    }
}

cloudFunctions
{}

// *** //
</pre>
</td>
<td>
<pre>
            collisionResolutionSteps 12;
            walls
            {
                youngsModulus   1e10;
                poissonsRatio   0.23;
                alpha   0.12;
                b   1.5;
                mu   0.43;
                cohesionEnergyDensity 0;
            }
            frontAndBack
            {
                youngsModulus   1e10;
                poissonsRatio   0.23;
                alpha   0.12;
                b   1.5;
                mu   0.1;
                cohesionEnergyDensity 0;
            }
        };
    }

    standardWallInteractionCoeffs
    {
        type   rebound;
    }
}

cloudFunctions
{}

// *** //
</pre>
</td>
</tr>
</table>

<table>
<tr>
<td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      RASProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

RASModel        laminar;

turbulence      on;

printCoeffs     on;


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

</td>
<td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.2.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      RASProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

RASModel        laminar;

turbulence      on;

printCoeffs     on;


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

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

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
```

</td>
<td>

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.2.0                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
```

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

version 2.0; format ascii; class dictionary; location "constant"; object transportProperties; } // * // rhoInf rhoInf [1 -3 0 0 0 0 0] 1.2; transportModel Newtonian; nu nu [0 2 -1 0 0 0 0] 1e-05; // ** ***** //	version 2.0; format ascii; class dictionary; location "constant"; object transportProperties; } // * // rhoInf rhoInf [1 -3 0 0 0 0 0] 1.2; transportModel Newtonian; nu nu [0 2 -1 0 0 0 0] 1e-05; // ** ***** //
/*---------------------------------*- C++ -*----------------------------------*\\ | ========= | | | \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox | | \\\\ / O peration | Version: 2.1.0 | | \\\\ / A nd | Web: www.OpenFOAM.org | | \\\\/ M anipulation | | *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "constant"; object turbulenceProperties; } // * // simulationType RASModel;	/*---------------------------------*- C++ -*----------------------------------*\\ | ========= | | | \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox | | \\\\ / O peration | Version: 2.2.0 | | \\\\ / A nd | Web: www.OpenFOAM.org | | \\\\/ M anipulation | | *---*/ FoamFile { version 2.0; format ascii; class dictionary; location "constant"; object turbulenceProperties; } // * // simulationType RASModel;

<table>
<tr><td>//
****
***** //</td><td>//
****
***** //</td></tr>
<tr><td>constant/polyMesh</td><td>constant/polyMesh</td></tr>
<tr><td>/*---------------------------------*- C++ -*----------------------------------*\
| ========= | |
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.1.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	| *---*/ FoamFile { version 2.0; format ascii; class dictionary; object blockMeshDict; } // * // convertToMeters 0.001; vertices ((0 77.9423 6.2) (135 0 6.2) (165 0 6.2) (300 77.9423 6.2) (300 500 6.2) (0 500 6.2) (0 77.9423 0) (135 0 0) (165 0 0) (300 77.9423 0) (300 500 0)</td><td>/*---------------------------------*- C++ -*----------------------------------*\ | =========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.2.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	| *---*/ FoamFile { version 2.0; format ascii; class dictionary; object blockMeshDict; } // * // convertToMeters 0.001; vertices ((0 77.9423 6.2) (135 0 6.2) (165 0 6.2) (300 77.9423 6.2) (300 500 6.2) (0 500 6.2) (0 77.9423 0) (135 0 0) (165 0 0) (300 77.9423 0) (300 500 0)</td></tr>
</table>

<table>
<tr>
<td><pre>
    (0 500 0)
);

blocks
(
    hex (6 9 10 11 0 3 4 5) (20 40 1) simpleGrading (1 1 1)
    hex (7 8 9 6 1 2 3 0) (20 8 1) simpleGrading (1 1 1)
);

boundary
(
    walls
    {
        type wall;
        faces
        (
            (1 7 8 2)
            (0 6 7 1)
            (2 8 9 3)
            (0 5 11 6)
            (3 4 10 9)
            (4 10 11 5)
        );
    }

    frontAndBack
    {
        type wall;
        faces
        (
            (0 3 4 5)
            (1 2 3 0)
            (6 11 10 9)
            (6 9 8 7)
        );
    }
</pre></td>
<td><pre>
    (0 500 0)
);

blocks
(
    hex (6 9 10 11 0 3 4 5) (20 40 1) simpleGrading (1 1 1)
    hex (7 8 9 6 1 2 3 0) (20 8 1) simpleGrading (1 1 1)
);

boundary
(
    walls
    {
        type wall;
        faces
        (
            (1 7 8 2)
            (0 6 7 1)
            (2 8 9 3)
            (0 5 11 6)
            (3 4 10 9)
            (4 10 11 5)
        );
    }

    frontAndBack
    {
        type wall;
        faces
        (
            (0 3 4 5)
            (1 2 3 0)
            (6 11 10 9)
            (6 9 8 7)
        );
    }
</pre></td>
</tr>
</table>

); // ** //	); // ** //
/*---------------------------------*- C++ -*----------------------------------*\\ | ========= | | | \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox | | \\\\ / O peration | Version: 2.1.0 | | \\\\ / A nd | Web: www.OpenFOAM.org | | \\\\/ M anipulation | | *---*/ FoamFile { version 2.0; format ascii; class polyBoundaryMesh; location "constant/polyMesh"; object boundary; } // * // 2 (walls { type wall; nFaces 136; startFace 1852; } frontAndBack { type wall; nFaces 1920; startFace 1988;	/*---------------------------------*- C++ -*----------------------------------*\\ | ========= | | | \\\\ / F ield | OpenFOAM: The Open Source CFD Toolbox | | \\\\ / O peration | Version: 2.2.0 | | \\\\ / A nd | Web: www.OpenFOAM.org | | \\\\/ M anipulation | | *---*/ FoamFile { version 2.0; format ascii; class polyBoundaryMesh; location "constant/polyMesh"; object boundary; } // * // 2 (walls { type wall; nFaces 136; startFace 1852; } frontAndBack { type wall; nFaces 1920; startFace 1988;

<table>
<tr><td><pre>
    }
)

// *** //
</pre></td><td><pre>
    }
)

// *** //
</pre></td></tr>
<tr><td>0</td><td>0</td></tr>
<tr><td><pre>
/*--------------------------------*- C++ -*----------------------------------*\
=========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.1.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	
---/
FoamFile
{
    version   2.0;
    format   ascii;
    class   volVectorField;
    object   U;
}
// * //

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

internalField   uniform (0 0 0);

boundaryField
{
    outlet
    {
        type   zeroGradient;
        value   uniform (0 0 0);
    }
    walls
    {
</pre></td><td><pre>
/*--------------------------------*- C++ -*----------------------------------*\
=========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 2.2.0
\\ / A nd	Web: www.OpenFOAM.org
\\/ M anipulation	
---/
FoamFile
{
    version   2.0;
    format   ascii;
    class   volVectorField;
    object   U;
}
// * //

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

internalField   uniform (0 0 0);

boundaryField
{
    outlet
    {
        type   zeroGradient;
        value   uniform (0 0 0);
    }
    walls
    {
</pre></td></tr>
</table>

<table>
<tr>
<td>
<pre>
        type   fixedValue;
        value   uniform (0 0 0);
    }
    frontAndBack
    {
        type   fixedValue;
        value   uniform (0 0 0);
    }
}

//
****
***** //
</pre>
</td>
<td>
<pre>
        type   fixedValue;
        value   uniform (0 0 0);
    }
    frontAndBack
    {
        type   fixedValue;
        value   uniform (0 0 0);
    }
}

//
****
***** //
</pre>
</td>
</tr>
<tr>
<td>ソースコード</td>
<td>ソースコード</td>
</tr>
<tr>
<td>
<pre>
/*---*\
  ========= |
  \\   / F ield | OpenFOAM: The Open Source CFD Toolbox
   \\   /   O peration |
    \\ /   A nd | Copyright (C) 2011 OpenFOAM Foundation
     \\/   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.
</pre>
</td>
<td>
<pre>
/*---*\
  ========= |
  \\   / F ield | OpenFOAM: The Open Source CFD Toolbox
   \\   /   O peration |
    \\ /   A nd | Copyright (C) 2011-2012 OpenFOAM Foundation
     \\/   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.
</pre>
</td>
</tr>
</table>

2.1.0	2.2.0
You should have received a copy of the GNU General Public License along with OpenFOAM. If not, see <http://www.gnu.org/licenses/>. Application uncoupledKinematicParcelFoam Description Transient solver for the passive transport of a single kinematic particle could. Uses a pre-calculated velocity field to evolve the cloud. *---*/ #include "fvCFD.H" #include "singlePhaseTransportModel.H" #include "turbulenceModel.H" #include "basicKinematicCollidingCloud.H" // * // int main(int argc, char *argv[]) { argList::addOption ("cloudName", "name", "specify alternative cloud name. default is 'kinematicCloud'"); #include "setRootCase.H" #include "createTime.H" #include "createMesh.H" #include "readGravitationalAcceleration.H" #include "createFields.H"	You should have received a copy of the GNU General Public License along with OpenFOAM. If not, see <http://www.gnu.org/licenses/>. Application uncoupledKinematicParcelFoam Description Transient solver for the passive transport of a single kinematic particle cloud. Uses a pre-calculated velocity field to evolve the cloud. *---*/ #include "fvCFD.H" #include "singlePhaseTransportModel.H" #include "turbulenceModel.H" #include "basicKinematicCollidingCloud.H" // * // int main(int argc, char *argv[]) { argList::addOption ("cloudName", "name", "specify alternative cloud name. default is 'kinematicCloud'"); #include "setRootCase.H" #include "createTime.H" #include "createMesh.H" #include "readGravitationalAcceleration.H" #include "createFields.H"

<pre> // * // Info<< "\nStarting time loop\n" << endl; while (runTime.loop()) { Info<< "Time = " << runTime.timeName() << nl << endl; Info<< "Evolving " << kinematicCloud.name() << endl; laminarTransport.correct(); mu = nu*rhoInfValue; kinematicCloud.evolve(); runTime.write(); Info<< "ExecutionTime = " << runTime.elapsedCpuTime() << " s" << " ClockTime = " << runTime.elapsedClockTime() << " s" << nl << endl; } Info<< "End\n" << endl; return 0; } // *** //</pre>	<pre> // * // Info<< "\nStarting time loop\n" << endl; while (runTime.loop()) { Info<< "Time = " << runTime.timeName() << nl << endl; Info<< "Evolving " << kinematicCloud.name() << endl; laminarTransport.correct(); mu = laminarTransport.nu()*rhoInfValue; kinematicCloud.evolve(); runTime.write(); Info<< "ExecutionTime = " << runTime.elapsedCpuTime() << " s" << " ClockTime = " << runTime.elapsedClockTime() << " s" << nl << endl; } Info<< "End\n" << endl; return 0; } // *** //</pre>