

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

OpenFOAM バージョン比較 (1.4 と 1.6): interFoam

June 27, 2011
Shinji Nakagawa, Toyama Pref. Univ.

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

OpenFOAM 1.4	OpenFOAM 1.6
	<p>相違点概略 VOF gamma が alpha1 に。 重力加速度設定ファイル名が environmentalProperties から g に。 圧力が pd から p に。</p>
<p>ケース ディレクトリ構造</p> <pre> / system/ controlDict decomposeParDict fvSchemes fvSolution setFieldsDict constant/ transportProperties environmentalProperties dynamicMeshDict polyMesh/ blockMeshDict boundary faces neighbour owner points 0/ gamma pd U </pre>	<p>ケース ディレクトリ構造</p> <pre> / system/ controlDict decomposeParDict fvSchemes fvSolution setFieldsDict constant/ transportProperties g dynamicMeshDict turbulenceProperties polyMesh/ blockMeshDict boundary faces neighbour owner points 0/ alpha1 p U </pre>
<p>system</p> <pre> /*-----*\ ===== \ \ / F i e l d OpenFOAM: The Open Source CFD Toolbox \ \ / O p e r a t i o n Version: 1.4 \ \ / A n d Web: http://www.openfoam.org ===== */ </pre>	<p>system</p> <pre> /*-----*-- C++ -*-----*\ ===== \ \ / F i e l d OpenFOAM: The Open Source CFD Toolbox \ \ / O p e r a t i o n Version: 1.6 \ \ / A n d Web: www.OpenFOAM.org ===== */ </pre>

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

```

|  \ \ /  M anipulation  |
\*-----*/
FoamFile
{
  version      2.0;
  format       ascii;
  root         "";
  case         "";
  instance     "";
  local        "";
  class        dictionary;
  object       controlDict;
}
// ***** //
application interFoam;
startFrom     startTime;
startTime     0;
stopAt        endTime;
endTime       10;
deltaT        0.0001;
writeControl  adjustableRunTime;
writeInterval 0.05;
purgeWrite    0;
writeFormat   ascii;
writePrecision 6;
writeCompression uncompressed;
timeFormat    general;
timePrecision 6;
runTimeModifiable yes;
adjustTimeStep yes;
maxCo         0.5;
maxDeltaT     1;
// ***** //

```

```

/*-----*/
| ===== |
| \ \ /  /  F ield      | OpenFOAM: The Open Source CFD Toolbox |
| \ \ /  /  O peration  | Version: 1.4 |
| \ \ /  /  A nd         | Web:      http://www.openfoam.org |
| \ \ /  /  M anipulation |
/*-----*/
FoamFile
{
  version      2.0;  format       ascii;

```

```

|  \ \ /  M anipulation  |
\*-----*/
FoamFile
{
  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;
maxDeltaT     1;
// ***** //

```

```

/*-----*- C++ -*-----*/
| ===== |
| \ \ /  /  F ield      | OpenFOAM: The Open Source CFD Toolbox |
| \ \ /  /  O peration  | Version: 1.6 |
| \ \ /  /  A nd         | Web:      www.OpenFOAM.org |
| \ \ /  /  M anipulation |
/*-----*/
FoamFile
{
  version      2.0;

```

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre> root ""; case ""; instance ""; local ""; class dictionary; object fvSchemes; } // ***** // ddtSchemes { default Euler; } gradSchemes { default Gauss linear; grad(U) Gauss linear; grad(gamma) Gauss linear; } divSchemes { div(rho*phi,U) Gauss limitedLinearV 1; div(phi,gamma) Gauss vanLeer; div(phirb,gamma) Gauss interfaceCompression; } laplacianSchemes { default Gauss linear corrected; } interpolationSchemes { default linear; } snGradSchemes { default corrected; } fluxRequired { default no; pd; pcorr; gamma; } // ***** // </pre>	<pre> format ascii; class dictionary; location "system"; object fvSchemes; } // ***** // ddtSchemes { default Euler; } gradSchemes { default Gauss linear; grad(U) Gauss linear; grad(alpha1) 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; pcorr; alpha1; } // ***** // </pre>
<pre> /*-----*\ ===== \\ / Field OpenFOAM: The Open Source CFD Toolbox \\ / Operation Version: 1.4 </pre>	<pre> /*-----* C++ -*-----*\ ===== \\ / Field OpenFOAM: The Open Source CFD Toolbox \\ / Operation Version: 1.6 </pre>

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

```

|  \ \ /  A nd      | Web:   http://www.openfoam.org      |
|  \ \ /  M anipulation |                               |
\*-----*//

FoamFile
{
  version      2.0;      format      ascii;
  root         "";      case        "";
  instance     "";      local      "";
  class        dictionary;
  object       fvSolution;
}
// *****

solvers
{
  pcorr PCG
  {
    preconditioner DIC;
    tolerance      1e-10;
    relTol         0;
  };

  pd PCG
  {
    preconditioner DIC;
    tolerance      1e-7;
    relTol         0.05;
  };

  pdFinal PCG
  {
    preconditioner DIC;
    tolerance      1e-7;
    relTol         0;
  };
  U PBiCG
  {
    preconditioner DILU;
    tolerance      1e-06;
    relTol         0;
  };
}

```

```

|  \ \ /  A nd      | Web:   www.OpenFOAM.org      |
|  \ \ /  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
  {
    solver       PCG;
    preconditioner DIC;
    tolerance     1e-07;
    relTol       0.05;
  }
  pFinal
  {
    solver       PCG;
    preconditioner DIC;
    tolerance     1e-07;
    relTol       0;
  }
  U
  {
    solver       PBiCG;
    preconditioner DILU;
    tolerance     1e-06;
    relTol       0;
  }
}

```

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre> } PISO { momentumPredictor no; nCorrectors 3; nNonOrthogonalCorrectors 0; nGammaCorr 1; nGammaSubCycles 2; cGamma 1; } // ***** // </pre>	<pre> } PISO { momentumPredictor no; nCorrectors 3; nNonOrthogonalCorrectors 0; nAlphaCorr 1; nAlphaSubCycles 2; cAlpha 1; } // ***** // </pre>
<pre> /*-----* ===== \ \ / Field OpenFOAM: The Open Source CFD Toolbox \ \ / Operation Version: 1.4 \ \ / And Web: http://www.openfoam.org \ \ / Manipulation /*-----* FoamFile { version 2.0; format ascii; root ""; case ""; instance ""; local ""; class dictionary; object setFieldsDict; } // ***** // defaultFieldValues (volScalarFieldValue gamma 0 volVectorFieldValue U (0 0 0)); regions (boxToCell { box (0 0 -1) (0.1461 0.292 1); fieldValues (volScalarFieldValue gamma 1 </pre>	<pre> /*-----* C++ -*-----* ===== \ \ / Field OpenFOAM: The Open Source CFD Toolbox \ \ / Operation Version: 1.6 \ \ / And Web: www.OpenFOAM.org \ \ / Manipulation /*-----* 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 </pre>

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre>); }); // ***** // </pre>	<pre>); }); // ***** // // gamma -> alpha1 </pre>
<p>Constant/polyMesh</p>	
<pre> /*-----*\ ===== \\ / F i e l d OpenFOAM: The Open Source CFD Toolbox \\ / O p e r a t i o n Version: 1.4 \\ / A n d Web: http://www.openfoam.org \\ \ M a n i p u l a t i o n *-----*/ FoamFile { version 2.0; format ascii; root "/home/p08161/OpenFOAM/p08161-1.4/run/tutorials/interFoam"; case "damBreak"; instance "constant"; local "polyMesh"; class polyBoundaryMesh; object boundary; } // ***** // 5 (leftWall { type wall; physicalType wall; nFaces 50; startFace 4432; } rightWall { type wall; physicalType wall; nFaces 50; startFace 4482; } lowerWall { type wall; </pre>	<pre> /*-----*\ ===== \\ / F i e l d OpenFOAM: The Open Source CFD Toolbox \\ / O p e r a t i o n Version: 1.6 \\ / A n d Web: www.OpenFOAM.org \\ \ M a n i p u l a t i o n *-----*/ 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; </pre>

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre> physicalType wall; nFaces 62; startFace 4532; } atmosphere { type patch; physicalType atmosphere; nFaces 46; startFace 4594; } defaultFaces { type empty; nFaces 4536; startFace 4640; }) // ***** // </pre>	<pre> nFaces 62; startFace 4532; } atmosphere { type patch; nFaces 46; startFace 4594; } defaultFaces { type empty; nFaces 4536; startFace 4640; }) // ***** // </pre>
<pre> constant /*-----*\ ===== \ \ / F i e l d OpenFOAM: The Open Source CFD Toolbox \ \ / O p e r a t i o n Version: 1.4 \ \ / A n d Web: http://www.openfoam.org \ \ M a n i p u l a t i o n *-----*/ FoamFile { version 2.0; format ascii; root ""; case ""; instance ""; local ""; class dictionary; object environmentalProperties; } // ***** // g g [0 1 -2 0 0 0] (0 -9.81 0); // ***** // </pre>	<pre> constant /*-----* C++ -*-----*\ ===== \ \ / F i e l d OpenFOAM: The Open Source CFD Toolbox \ \ / O p e r a t i o n Version: 1.6 \ \ / A n d Web: www.OpenFOAM.org \ \ M a n i p u l a t i o n *-----*/ FoamFile { version 2.0; format ascii; class uniformDimensionedVectorField; location "constant"; object g; } // ***** // dimensions [0 1 -2 0 0 0]; value (0 -9.81 0); // ***** // </pre>
<pre> /*-----*\ ===== \ \ / F i e l d OpenFOAM: The Open Source CFD Toolbox </pre>	<pre> /*-----* C++ -*-----*\ ===== \ \ / F i e l d OpenFOAM: The Open Source CFD Toolbox </pre>

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

```

|  \ \ /  O peration | Version:  1.4 |
|  \ \ /  A nd       | Web:      http://www.openfoam.org |
|  \ \ /  M anipulation | |
|-----*-----*/
FoamFile
{
  version      2.0;      format      ascii;
  root         "";      case        "";
  instance     "";      local      "";
  class        dictionary;
  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;
    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

```

```

|  \ \ /  O peration | Version:  1.6 |
|  \ \ /  A nd       | Web:      www.OpenFOAM.org |
|  \ \ /  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;
    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

```

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre> { nu0 nu0 [0 2 -1 0 0 0] 1e-06; nuInf nuInf [0 2 -1 0 0 0] 1e-06; m m [0 0 1 0 0 0] 1; n n [0 0 0 0 0 0] 0; } BirdCarreauCoeffs { nu0 nu0 [0 2 -1 0 0 0] 0.0142515; nuInf nuInf [0 2 -1 0 0 0] 1e-06; k k [0 0 1 0 0 0] 99.6; n n [0 0 0 0 0 0] 0.1003; } } sigma sigma [1 0 -2 0 0 0] 0.07; // ***** // </pre>	<pre> { nu0 nu0 [0 2 -1 0 0 0] 1e-06; nuInf nuInf [0 2 -1 0 0 0] 1e-06; m m [0 0 1 0 0 0] 1; n n [0 0 0 0 0 0] 0; } BirdCarreauCoeffs { nu0 nu0 [0 2 -1 0 0 0] 0.0142515; nuInf nuInf [0 2 -1 0 0 0] 1e-06; k k [0 0 1 0 0 0] 99.6; n n [0 0 0 0 0 0] 0.1003; } } sigma sigma [1 0 -2 0 0 0] 0.07; // ***** // </pre>
<pre> 0 </pre>	<pre> /*-----* C++ *-----*\ ===== \\ / Field OpenFOAM: The Open Source CFD Toolbox \\ / Operation Version: 1.6 \\ / And Web: www.OpenFOAM.org \\ / Manipulation *-----*\ FoamFile { version 2.0; format ascii; class dictionary; location "constant"; object turbulenceProperties; } // ***** // simulationType laminar; // ***** // </pre>
<pre> /*-----* C++ *-----*\ ===== \\ / Field OpenFOAM: The Open Source CFD Toolbox \\ / Operation Version: 1.4 </pre>	<pre> /*-----* C++ *-----*\ ===== \\ / Field OpenFOAM: The Open Source CFD Toolbox \\ / Operation Version: 1.6 </pre>

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

```

|  \ \ /  A nd      | Web:   http://www.openfoam.org      |
|  \ \ /  M anipulation | |
\*-----*/
FoamFile
{
  version      2.0;      format      ascii;
  root         "";      case       "";
  instance     "";      local      "";
  class        volScalarField;
  object       gamma;
}
// ***** //
dimensions     [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;
  }
}
// ***** //

```

```

|  \ \ /  A nd      | Web:   http://www.OpenFOAM.org      |
|  \ \ /  M anipulation | |
\*-----*/
FoamFile
{
  version      2.0;
  format       ascii;
  class        volScalarField;
  object       alpha;
}
// ***** //
dimensions     [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;
  }
}
// ***** //

```

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre> /*-----*/ ===== \ / Field OpenFOAM: The Open Source CFD Toolbox \ / Operation Version: 1.4 \ / And Web: http://www.openfoam.org \ / Manipulation -----*/ FoamFile { version 2.0; format ascii; root ""; case ""; instance ""; local ""; class volScalarField; object pd; } // ***** dimensions [1 -1 -2 0 0 0]; internalField uniform 0; boundaryField { leftWall { type zeroGradient; } rightWall { type zeroGradient; } lowerWall { type zeroGradient; } atmosphere { type totalPressure; p0 uniform 0; } } </pre>	<pre> /*-----*- C++ -*-----*/ ===== \ / Field OpenFOAM: The Open Source CFD Toolbox \ / Operation Version: 1.6 \ / And Web: http://www.OpenFOAM.org \ / Manipulation -----*/ FoamFile { version 2.0; format ascii; class volScalarField; object p; } // ***** dimensions [1 -1 -2 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; } } </pre>
--	---

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre> U U; phi phi; rho none; psi none; gamma 1; value uniform 0; } defaultFaces { type empty; } } // ***** // </pre>	<pre> U U; phi phi; rho rho; psi none; gamma 1; value uniform 0; } defaultFaces { type empty; } } // ***** // </pre>
<pre> /*-----*\ ===== \\ / Field OpenFOAM: The Open Source CFD Toolbox \\ / Operation Version: 1.4 \\ / And Web: http://www.openfoam.org \\ / Manipulation *-----*/ FoamFile { version 2.0; format ascii; root ""; case ""; instance ""; local ""; class volVectorField; object U; } // ***** // dimensions [0 1 -1 0 0 0]; internalField uniform (0 0 0); boundaryField { leftWall { type fixedValue; value uniform (0 0 0); } rightWall </pre>	<pre> /*-----*\ ===== \\ / Field OpenFOAM: The Open Source CFD Toolbox \\ / Operation Version: 1.6 \\ / And Web: http://www.OpenFOAM.org \\ / Manipulation *-----*/ FoamFile { version 2.0; format ascii; class volVectorField; location "0"; object U; } // ***** // dimensions [0 1 -1 0 0 0]; internalField uniform (0 0 0); boundaryField { leftWall { type fixedValue; value uniform (0 0 0); } rightWall </pre>

OpenFOAM バージョン比較 (1.4 と 1.6) : interFoam

<pre>{ type fixedValue; value uniform (0 0 0); } lowerWall { type fixedValue; value uniform (0 0 0); } atmosphere { type pressureInletOutletVelocity; phi phi; value uniform (0 0 0); } defaultFaces { type empty; } } // ***** //</pre>	<pre>{ type fixedValue; value uniform (0 0 0); } lowerWall { type fixedValue; value uniform (0 0 0); } atmosphere { type pressureInletOutletVelocity; value uniform (0 0 0); } defaultFaces { type empty; } } // ***** //</pre>
--	---