ภาคผนวก

โปรแกรมเพื่อใช้ในการวิเคราะห์ OpenFOAM version 7 (The OpenFOAM Foundation, 2019)

velocity : U

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform (1 0 0);

boundaryField
{

    inlet_left
    {
        type            uniformFixedValue;
        uniformValue    constant (1 0 0);
    }

    inlet_top_bottom
    {
        type            uniformFixedValue;
        uniformValue    constant (1 0 0);
    }
```



```
    outlet_right
    {
//Hmee      type            uniformFixedValue;
//Hmee      uniformValue    constant (1 0 0);
        type                zeroGradient;
    }

    outlet_top_bottom
    {
//Hmee      type            uniformFixedValue;
//Hmee      uniformValue    constant (1 0 0);
        type                zeroGradient;
    }

    front_left
    {
        type            cyclic;
    }

    front_right
    {
        type            cyclic;
    }

    back_left
    {
        type            cyclic;
    }

    back_right
    {
        type            cyclic;
    }
```



```
    sphere
    {
        type            rotatingWallVelocity;                   //wall ; rotatingWallvelocity;
        origin          (0 0 0);
        axis            (0 0 1);
        omega           constant -0.0;                          //omega  0; //4; spin ratio:2
//          type            noSlip;
    }

}

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


pressure : P

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 0;

boundaryField
{

    inlet_left
    {
                type                    zeroGradient;
//Hmee      type            freestreamPressure;
//Hmee      freestreamValue uniform 0;
        }

        inlet_top_bottom
    {
                type                    zeroGradient;
//Hmee      type            freestreamPressure;
```

```
//Hmee        freestreamValue uniform 0;
        }

    outlet_right
    {
//Hmee        type            fixedValue;
        type            freestreamPressure;
        freestreamValue uniform 0;
//          value           uniform 0;
    }

    outlet_top_bottom
    {
//Hmee        type            fixedValue;
        type            freestreamPressure;
        freestreamValue uniform 0;
//          value           uniform 0;
    }


    front_left
    {
        type            cyclic;
    }

    front_right
    {
        type            cyclic;
    }

    back_left
    {
        type            cyclic;
    }
```



```
    back_right
    {
        type            cyclic;
    }


    sphere
    {
        type            zeroGradient;
    }

//    defaultFaces
//    {
//        type            empty;
//    }
}

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


turbulent kinetic energy : k

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      k;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 1e-4;

boundaryField
{
    inlet_left
    {
        type            fixedValue;
        value           $internalField;
    }

    inlet_top_bottom
    {
        type            fixedValue;
        value           $internalField;
    }
```



```
    outlet_right
    {
        type            zeroGradient;
    }

    outlet_top_bottom
    {
        type            zeroGradient;
    }

    front_left
    {
        type            cyclic;
    }

    front_right
    {
        type            cyclic;
    }

    back_left
    {
        type            cyclic;
    }

    back_right
    {
        type            cyclic;
    }


    sphere
    {
```

```
//              type            zeroGradient;

        type            kqRWallFunction;
//          value           uniform 1e-4;
        value           $internalField;
    }
//  lowerWall
//  {
//      type            kqRWallFunction;
//      value           uniform 1e-4;
//  }

}

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


epsilon

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      epsilon;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 2 -3 0 0 0 0];

internalField   uniform 6.3e-6;

boundaryField
{
    inlet_left
    {
        type            fixedValue;
        value           $internalField;
    }

    inlet_top_bottom
    {
        type            fixedValue;
        value           $internalField;
    }
```



```
    outlet_right
    {
        type            zeroGradient;
    }

    outlet_top_bottom
    {
        type            zeroGradient;
    }

    front_left
    {
        type            cyclic;
    }

    front_right
    {
        type            cyclic;
    }

    back_left
    {
        type            cyclic;
    }

    back_right
    {
        type            cyclic;
    }


    sphere
    {
//          type            zeroGradient;
```



```
        type            epsilonWallFunction;
//          value           uniform 6.3e-6;
        value           $internalField;
    }
//  lowerWall
//  {
//      type            epsilonWallFunction;
//      value           uniform 6.30234e-6;
//  }


}

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

turbulent viscosity : nut

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    location    "0";
    object      nut;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 0;

boundaryField
{
    inlet_left
    {
        type            calculated;
        value           $internalField;
    }

    inlet_top_bottom
    {
        type            calculated;
        value           $internalField;
    }
```



```
    outlet_right
    {
        type            zeroGradient;
    }

    outlet_top_bottom
    {
        type            zeroGradient;
    }

    front_left
    {
        type            cyclic;
    }

    front_right
    {
        type            cyclic;
    }

    back_left
    {
        type            cyclic;
    }

    back_right
    {
        type            cyclic;
    }


    sphere
    {
//      type            zeroGradient;
```



```
        type            nutUWallFunction;
//      value           uniform 0;
        value           $internalField;
    }

}


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


Reynolds Stress Magnitude : R

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volSymmTensorField;
    location    "0";
    object      R;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

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

internalField   (0.5e-4 0 0 0.5e-4 0 1e-4);                    //1.5873e-3

boundaryField
{
    inlet_left
    {
        type            fixedValue;
//      type            zeroGradient;
        value           $internalField;
    }

    inlet_top_bottom
    {
        type            fixedValue;
//      type            zeroGradient;
```

```
    value           $internalField;;
  }

  outlet_right
  {
    type            zeroGradient;
  }

  outlet_top_bottom
  {
    type            zeroGradient;
  }

  front_left
  {
    type            cyclic;
  }

  front_right
  {
    type            cyclic;
  }

  back_left
  {
    type            cyclic;
  }

  back_right
  {
    type            cyclic;
  }
```



```
  sphere
  {
//              type            zeroGradient;

    type          kqRWallFunction;
//      value           uniform 1e-4;
    value         $internalField;
  }
//  lowerWall
//  {
//    type          kqRWallFunction;
//    value         uniform 1e-4;
//  }

}

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


blockMeshDict

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
  version     2.0;
  format      ascii;
  class       dictionary;
  object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 1.0;                    //metre

vertices
(
    (0  0   -5)                         //point 0
    (0  5   -5)                         //point 1
    (-5 5   -5)                         //point 2
    (-5 0   -5)                         //point 3
    (-5 -5  -5)                         //point 4
    (0  -5  -5)                         //point 5
    (5  -5  -5)                         //point 6
    (5  0   -5)                         //point 7
    (5  5   -5)                         //point 8

    (0  0    0)                         //point 9
    (0  5    0)                         //point 10
```



```
    (-5 5    0)                         //point 11
    (-5 0    0)                         //point 12
    (-5 -5   0)                         //point 13
    (0  -5   0)                         //point 14
    (5  -5   0)                         //point 15
    (5  0    0)                         //point 16
    (5  5    0)                         //point 17

    (0  0    5)                         //point 18
    (0  5    5)                         //point 19
    (-5 5    5)                         //point 20
    (-5 0    5)                         //point 21
    (-5 -5   5)                         //point 22
    (0  -5   5)                         //point 23
    (5  -5   5)                         //point 24
    (5  0    5)                         //point 25
    (5  5    5)                         //point 26

);

blocks
(
  hex (0 1 2 3 9 10 11 12) (50  50  50) simpleGrading (1 1 1)      // B0 Top-Left    back   Block Q2 -+  (10  10  0.1)
  hex (0 3 4 5 9 12 13 14) (50  50  50) simpleGrading (1 1 1)      // B1 Bottom-Left back   Block Q3 --  (10  10  0.1)
  hex (0 5 6 7 9 14 15 16) (50  50  50) simpleGrading (1 1 1)      // B2 Bottom-Right back  Block Q4 +-  (10  10
0.1)
  hex (0 7 8 1 9 16 17 10) (50  50  50) simpleGrading (1 1 1)      // B3 Top-Right   back   Block Q1 ++  (10  10
0.1)
```

```
    hex (9 10 11 12 18 19 20 21) (50  50  50)  simpleGrading (1 1 1)     // B4 Top-Left     front   Block Q2  -+   (10  10
10)
    hex (9 12 13 14 18 21 22 23) (50  50  50)  simpleGrading (1 1 1)     // B5 Bottom-Left  front   Block Q3  --   (10  10
10)
    hex (9 14 15 16 18 23 24 25) (50  50  50)  simpleGrading (1 1 1)     // B6 Bottom-Right front   Block Q4  +-   (10  10
10)
    hex (9 16 17 10 18 25 26 19) (50  50  50)  simpleGrading (1 1 1)     // B7 Top-Right    front   Block Q1  ++   (10  10
10)


);

edges
(

);

boundary
(

    inlet_left
    {
    type patch ;
    faces
     (
     (3 2 11 12)
     (4 3 12 13)
     (12 11 20 21)
     (13 12 21 22)
     );
    }
```



```
    inlet_top_bottom
    {
    type patch ;
    faces
     (
     (2 1 10 11)
     (11 10 19 20)
     (4 13 14 5)
     (13 22 23 14)
     );
    }

    outlet_right
    {
    type patch ;
    faces
     (
     (6 15 16 7)
     (7 16 17 8)
     (15 24 25 16)
     (16 25 26 17)
     );
    }

    outlet_top_bottom
    {
    type patch ;
    faces
     (
     (1 8 17 10)
     (10 17 26 19)
     (5 14 15 6)
```



```
     (14 23 24 15)
     );
    }

    front_left
    {
//        type patch ;
//     type symmetryPlane;
    type cyclic;
    neighbourPatch back_left;
    faces
     (
     (21 20 19 18)
     (21 18 23 22)
     );
    }

    back_left
    {
//     type patch;
//    type symmetryPlane;
    type cyclic;
    neighbourPatch front_left;
    faces
     (
     (0 1 2 3)
     (0 3 4 5)
     );
    }

    front_right
    {
```



```
//        type patch;
//     type symmetryPlane;
    type cyclic;
    neighbourPatch back_right;
    faces
     (
     (19 26 25 18)
     (18 25 24 23)
     );
    }

    back_right
    {
//        type patch;
//     type symmetryPlane;
    type cyclic;
    neighbourPatch front_right;
    faces
     (
     (0 7 8 1)
     (5 6 7 0)
     );
    }

);

mergePatchPairs
(

);

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

meshQualityDict

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      meshQualityDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

// Include defaults parameters from master dictionary
#includeEtc "caseDicts/mesh/generation/meshQualityDict"

//- minFaceWeight (0 -> 0.5)
minFaceWeight 0.02;


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


controlDict

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     simpleFoam;

startFrom       startTime;                              //startTime;  latestTime;

startTime       0;
//latestTime      70;

stopAt          endTime;

endTime         200;

deltaT          0.05;

writeControl    timeStep;

writeInterval   2000;
```



```
purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;

functions
    {forceCoeffs1
    {
     // Mandatory entries
     type            forceCoeffs;
     libs            ("libforces.so");
     patches         (sphere);                              //wall


     // Optional entries

     // Field names
     p               p;
     U               U;
     rho             rhoInf;
     rhoInf          1;

     // Reference pressure [Pa]
     pRef            0;
```

```
    // Include porosity effects?
    porosity        no;

    // Store and write volume field representations of forces and moments
    writeFields     yes;

    // Centre of rotation for moment calculations
    CofR            (0 0 0);

    // Lift direction
    liftDir         (0 1 0);                                //(0 0 1)

    // Drag direction
    dragDir         (1 0 0);

    // Pitch axis
    pitchAxis       (0 0 1);                                //(0 1 0)

    // Freestream velocity magnitude [m/s]
    magUInf         1;

    // Reference length [m]
    lRef            1;                                      //for sphere l = D

    // Reference area [m2]
    Aref            0.7857142857;

    // Spatial data binning
    // - extents given by the bounds of the input geometry
    binData
    {
```



```
        nBin        20;
        direction   (1 0 0);
        cumulative  yes;
    }
    }
    }

functions
    {forces1
    {
    // Mandatory entries
    type            forces;
    libs            ("libforces.so");
    patches         (sphere);                               //wall


    // Optional entries

    // Field names
    p               p;
    U               U;
    rho             rhoInf;
    rhoInf          1;

    // Reference pressure [Pa]
    pRef            0;

    // Include porosity effects?
    porosity        no;

    // Store and write volume field representations of forces and moments
    writeFields     yes;
```



```
    // Centre of rotation for moment calculations
    CofR            (0 0 0);

    // Spatial data binning
    // - extents given by the bounds of the input geometry
    binData
    {
        nBin        20;
        direction   (1 0 0);
        cumulative  yes;
    }
    }
    }

functions
    {fieldAverage1
    {
        type            fieldAverage;
        libs            ("libfieldFunctionObjects.so");
        writeControl    writeTime;

        fields
        (
            U
            {
                mean        on;
                prime2Mean  on;
                base        time;
            }

            p
            {
                mean        on;
                prime2Mean  on;
```



```
                base        time;
            }

            k
            {
                mean        on;
                prime2Mean  on;
                base        time;
            }

            epsilon
            {
                mean        on;
                prime2Mean  on;
                base        time;
            }

            nut
            {
                mean        on;
                prime2Mean  on;
                base        time;
            }

            phi
            {
                mean        on;
                prime2Mean  on;
                base        time;
            }
        );
    }
    }
```

```
    //rho        rhoInf;
    //rhoInf     1;


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


snappyHexMeshDict

```
/*--------------------------------*- 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      snappyHexMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

// Which of the steps to run
castellatedMesh true;
snap            true;
addLayers       true;


geometry
{

    sphere
    {
        type searchableSphere;
        centre  (0 0 0);
        radius  0.5;
        name sphere;                                    //Hmee
    }

//Hmee    refinementBox
```



```
//    {
//        type searchableBox;
//        min (-1.0 -1.0 -1.0);
//        max ( 1.0  1.0  1.0);
//Hmee    }
};

castellatedMeshControls
{
    maxLocalCells 1000000;
    maxGlobalCells 2000000;
    minRefinementCells 0;
    maxLoadUnbalance 0.10;
    nCellsBetweenLevels 1;
    features
    (
    );
    refinementSurfaces
    {
        sphere
        {
            level (3 3);                                // (5 5)
         /*     regions
            {
                secondSolid
                {
                    level (5 5);
                }
            }
         */
            patchInfo
            {
```



```
                type wall;
            }
        }

    }

    resolveFeatureAngle 30;
    refinementRegions
    {
     /*  sphere
        {
                mode outside;
                levels ((1.0 1));
        }
     */
    }


    locationInMesh (1.1 0 0);
    allowFreeStandingZoneFaces true;
}


snapControls
{
    nSmoothPatch 3;
    tolerance 1.0;                                  //typically 2
    nSolveIter 30;
    nRelaxIter 5;
}

addLayersControls
{
    relativeSizes false;
```

```
    layers
    {
        sphere_region0                                          //Hmee
        {
            nSurfaceLayers 5;                                   //Hmee 2 , 5*

        }

    }

    expansionRatio 1.0;
//      finalLayerThickness 1e-9;
    finalLayerThickness 0.1;
    finalLayerRatio 0.01;
    minThickness 0.002;
    nGrow 0;
    featureAngle 30;
    nRelaxIter 10;
    nSmoothSurfaceNormals 1;
    nSmoothNormals 3;
    nSmoothThickness 10;
    maxFaceThicknessRatio 0.5;
    maxThicknessToMedialRatio 0.3;
    minMedianAxisAngle 40;
    nBufferCellsNoExtrude 0;
    nLayerIter 100;
    nRelaxedIter 20;
}

meshQualityControls
{
```



```
    maxNonOrtho 65;
    maxBoundarySkewness 20;
    maxInternalSkewness 4;
    maxConcave 80;
    minVol 1e-200;
    minTetQuality 1e-9;
    minArea -1;
    minTwist 0.05;
    minDeterminant 0.001;
    minFaceWeight 0.05;
    minVolRatio 0.01;
    minTriangleTwist -1;

    nSmoothScale 4;
    errorReduction 0.75;

    relaxed
    {
        maxNonOrtho 75;
    }
}

debug 0;
mergeTolerance 1e-6;

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


transportProperties

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

transportModel  Newtonian;

nu              [0 2 -1 0 0 0 0] 1.42804e-05;                   //Reynolds number = 70,026, D = 1m

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


turbulenceProperties

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "constant";
    object      turbulenceProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

simulationType RAS;

RAS
{
    RASModel        LRR;
//      RASModel        SSG;

    turbulence      on;

    printCoeffs     on;
}

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

fvSolution

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

solvers
{
        Phi
    {
        solver          GAMG;                           //GAMG;
        smoother        DIC;

        tolerance       1e-30;                          //1e-6 //1e-8
        relTol          0.00001;
    }

    p
    {
        solver          GAMG;                           //GAMG;
        tolerance       1e-30;                          //1e-6
        relTol          0.00001;
        smoother        GaussSeidel;
    }
```



```
    }

    "(U|k|epsilon|omega|f|v2)"
    {
        solver          smoothSolver;
        smoother        symGaussSeidel;
        tolerance       1e-30;                          //1e-6
        relTol          0.00001;
    }

    R
    {
        solver          GAMG;                           //GAMG;
        tolerance       1e-30;                          //1e-6
        relTol          0.00001;
        smoother        GaussSeidel;
    }
}

SIMPLE                                                  //SIMPLE
{
    nNonOrthogonalCorrectors 3;
    residualControl
    {
        p               1e-9;
        U               1e-9;
        nuTilda         1e-9;
        "(k|epsilon|omega|f|v2|R)"  1e-9
        pRefCell        0;
        pRefValue       0;
    }

}
```



```
relaxationFactors
{
    fields                          //Hmee
    {
        p               0.1;            //Hmee              //0.3    not sure this cause can make good convergence for RSMs
    }
    equations
    {
        U               0.2; // 0.9 is more stable but 0.95 more convergent  //0.4    not sure this cause can make good convergence for
RSMs
        k               0.2; //  Hmee
        ".*"            0.2; // 0.9 is more stable but 0.95 more convergent
    }
}


potentialFlow
{
    nNonOrthogonalCorrectors 3;
}


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


fvSchemes

```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  7
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSchemes;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

ddtSchemes
{
    default         CrankNicolson 0.9;                          //steadyState;  //Euler; //CrankNicolson 0.9;
}

gradSchemes
{
    default         Gauss linear;
    grad(p)             Gauss linear;
}

divSchemes
{
    default         none;
    div(phi,U)      Gauss linearUpwind grad(U);                 //bounded Gauss linearUpwind grad(U);
    div(phi,k)      Gauss upwind;                             //bounded Gauss linearUpwind grad(k);
    div(phi,epsilon) Gauss upwind;                              //bounded Gauss linearUpwind grad(epsilon);
```

```
//     div(phi,alpha)  Gauss vanLeer;
//     div(phirb,alpha)  Gauss vanLeer;

    div(R)          Gauss linear ;
    div(phi,R)      Gauss upwind;                                   //Gauss linearUpwind grad(R); not sure this cause can make good convergence
for RSMs
    div(phi,omega)  bounded Gauss limitedLinear 1;
    div(phi,v2)      bounded Gauss limitedLinear 1;
//     div((nuEff*dev2(T(grad(U))))) Gauss linear;                  //add rho
    div((nu*dev2(T(grad(U))))) Gauss linear;
    div(nonlinearStress) Gauss linear;
}

laplacianSchemes
{
    default         Gauss linear corrected;
}

interpolationSchemes
{
    default         linear;
}

snGradSchemes
{
    default         corrected;                                  //corrected;
}

wallDist
{
    method meshWave;
//     method Poisson;
}
// ************************************************************************* //
```