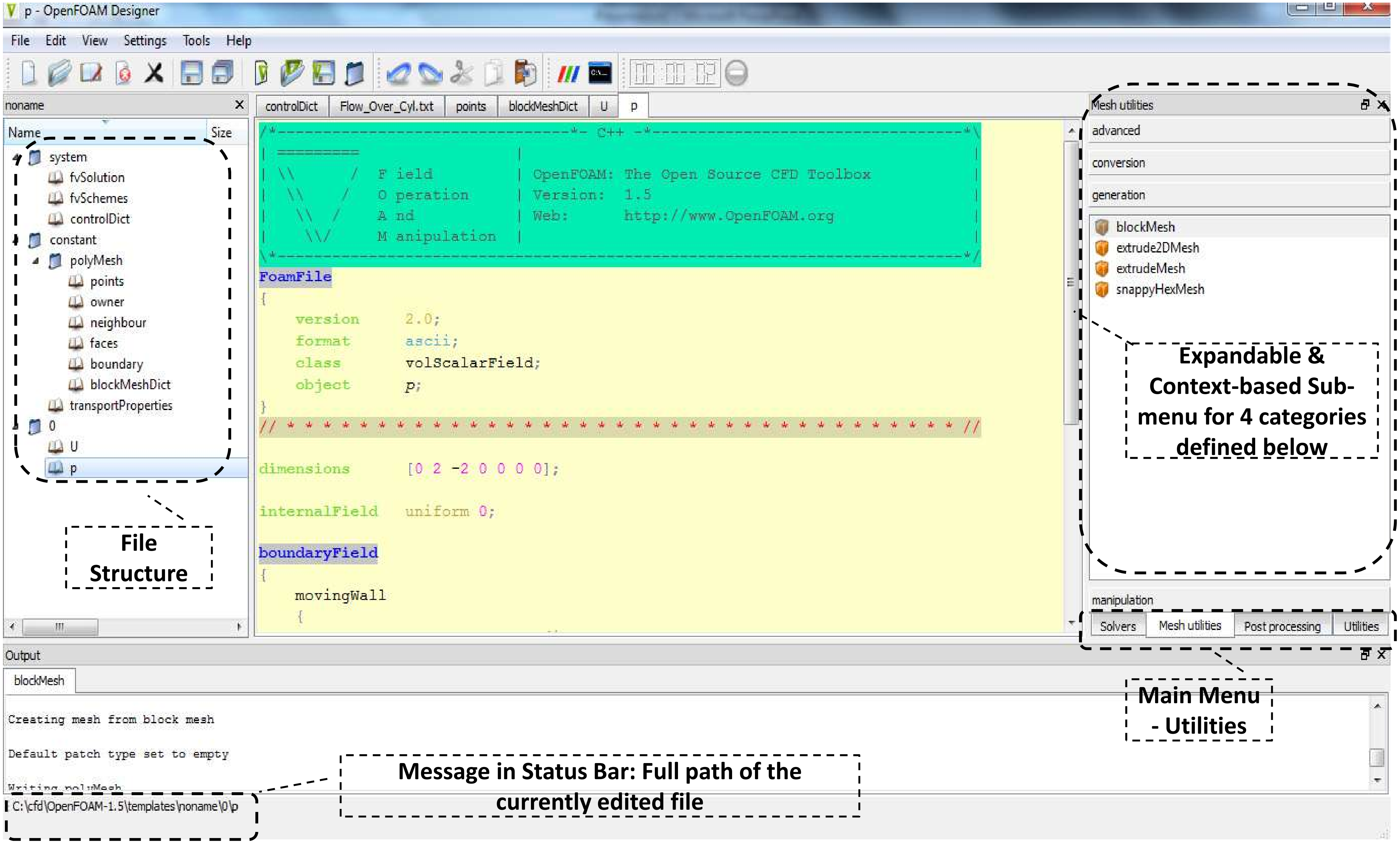
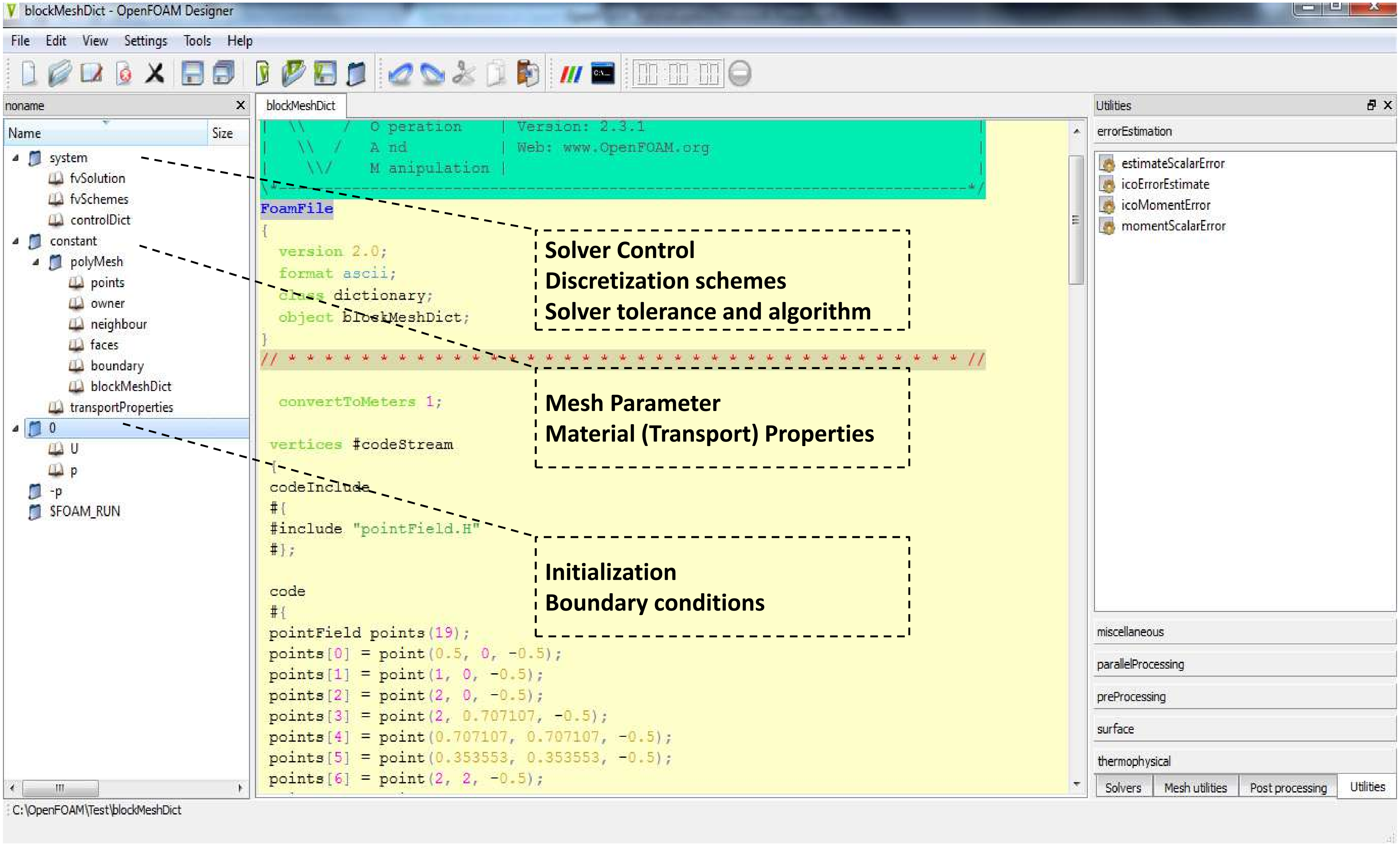
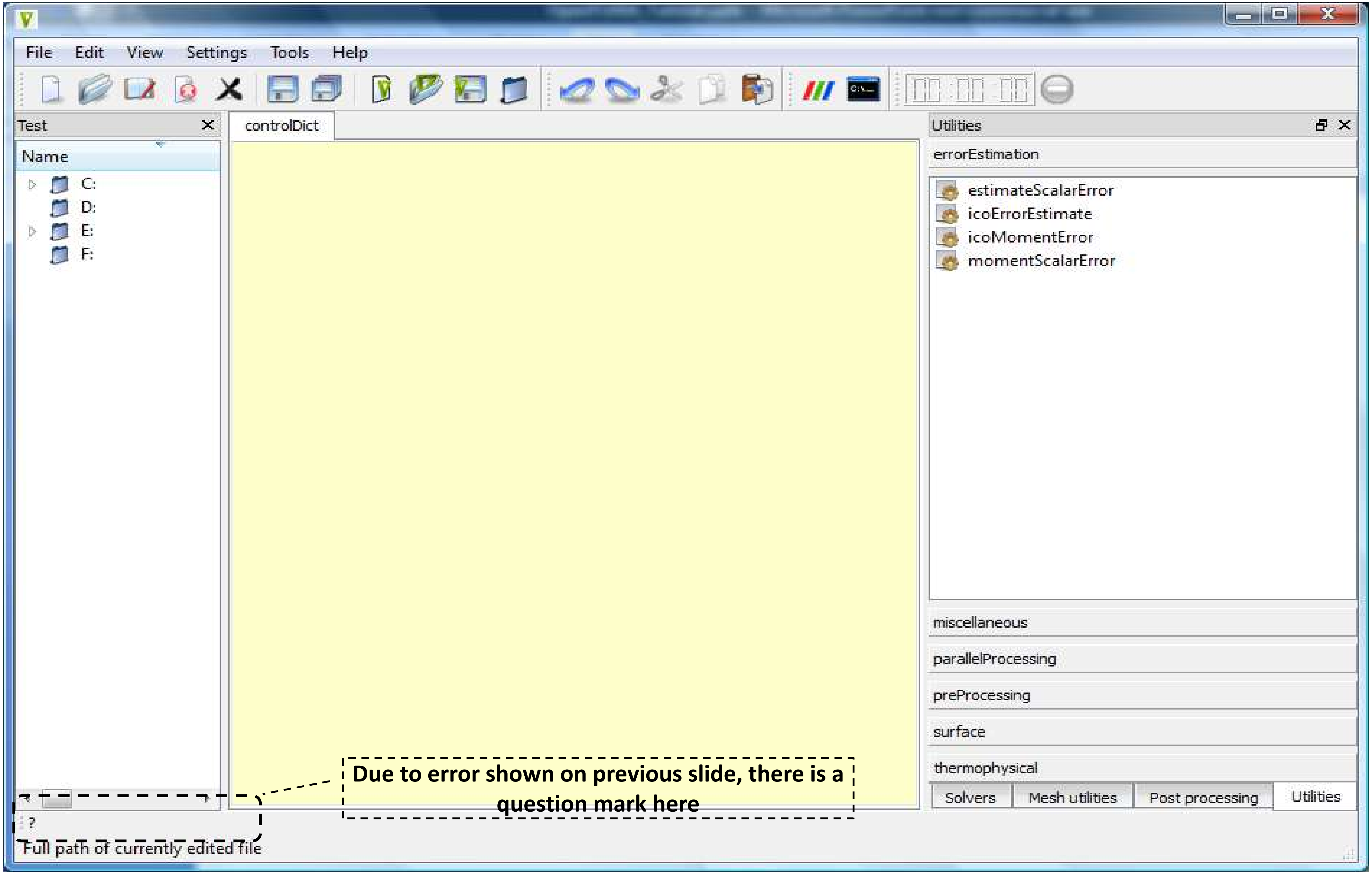
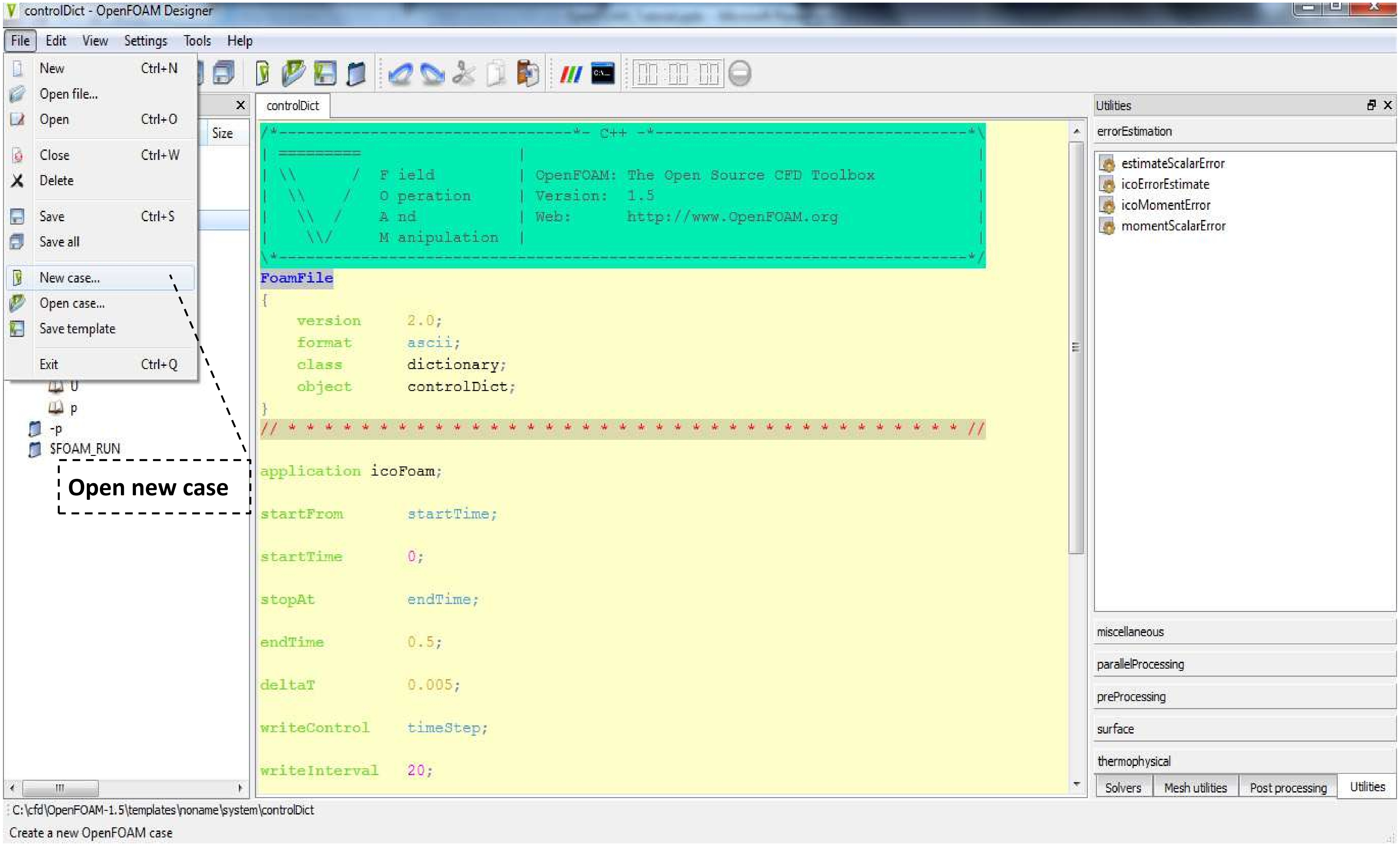
GUI & File Structure in OpenFOAM Designer Version 1

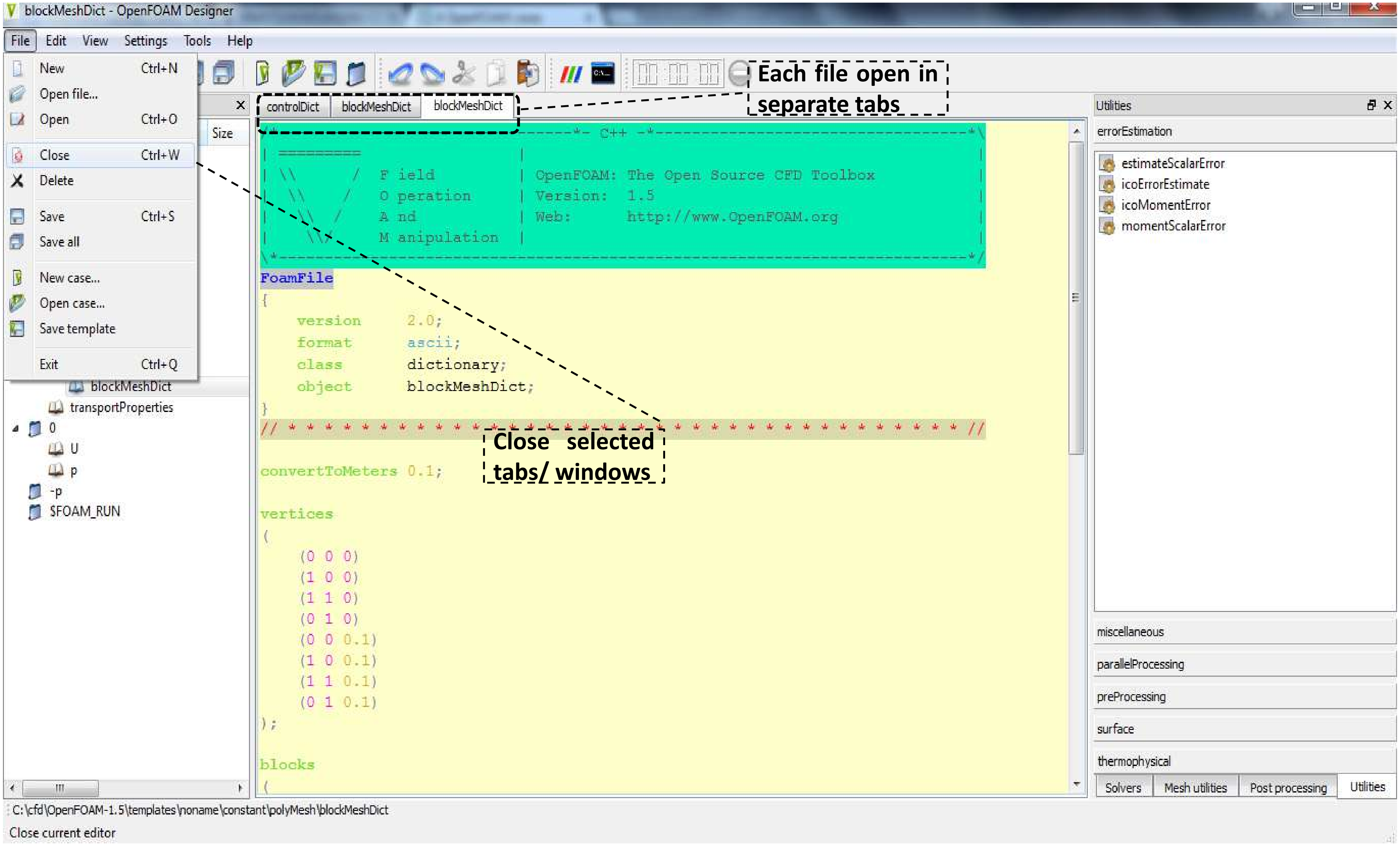


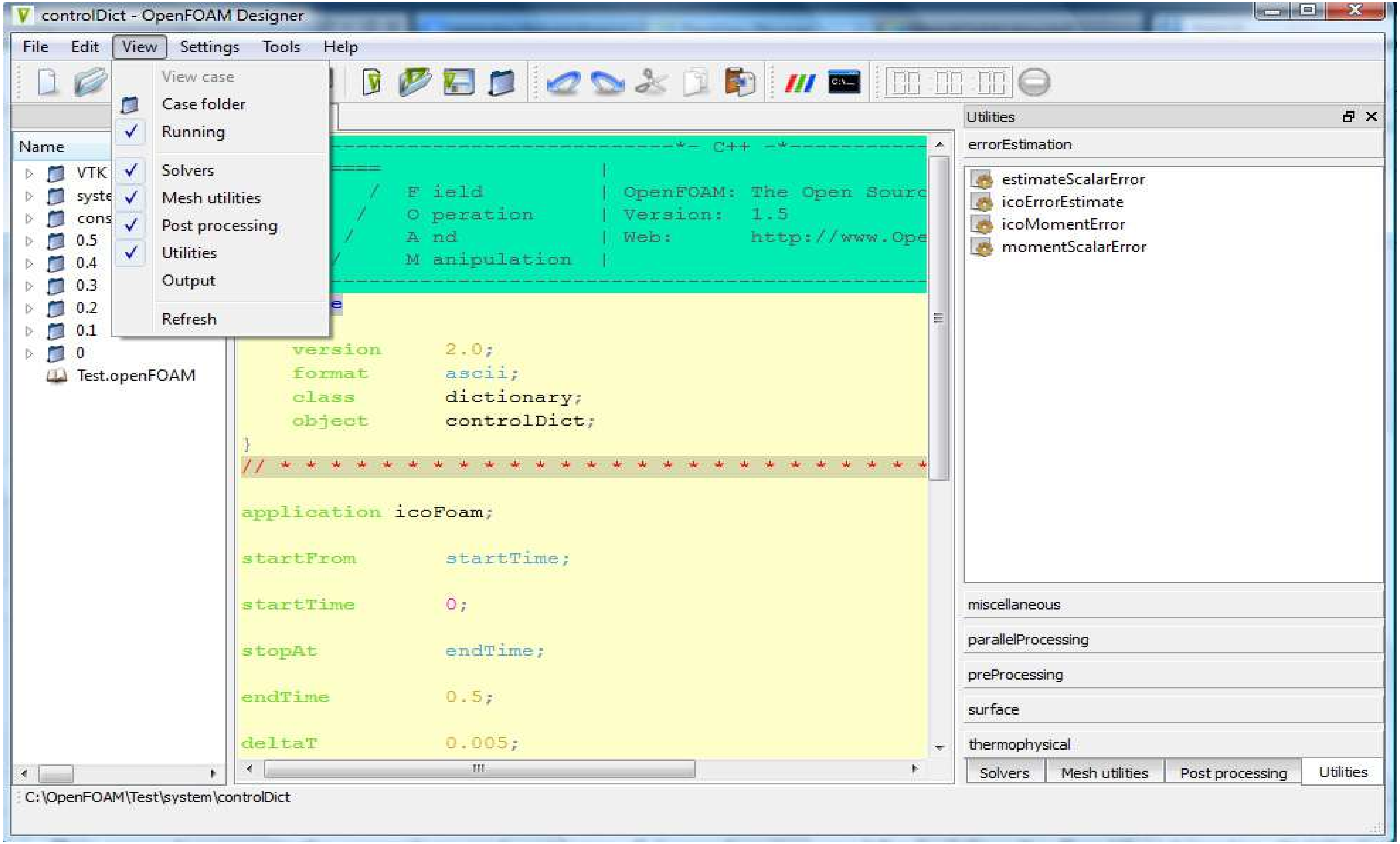
p - OpenFOAM Designer
File Edit View Settings Tools Help
noname
Name Size
system
fvSolution
fvSchemes
controlDict
constant
polyMesh
points
owner
neighbour
faces
boundary
blockMeshDict
transportProperties
0
U
p
File Structure
controlDict Flow_Over_Cyl.txt points blockMeshDict U p
F ield | OpenFOAM: The Open Source CFD Toolbox
O peration | Version: 1.5
A nd | Web: http://www.OpenFOAM.org
M anipulation |
FoamFile
version 2.0;
format ascii;
class volScalarField;
object p;
dimensions [0 2 -2 0 0 0 0];
internalField uniform 0;
boundaryField
movingWall
Mesh utilities
advanced
conversion
generation
blockMesh
extrude2DMesh
extrudeMesh
snappyHexMesh
Expandable & Context-based Sub-menu for 4 categories defined below
manipulation
Solvers Mesh utilities Post processing Utilities
Main Menu - Utilities
Output
blockMesh
Creating mesh from block mesh
Default patch type set to empty
Message in Status Bar: Full path of the currently edited file
C:\cfd\OpenFOAM-1.5\templates\noname\0\p


blockMeshDict - OpenFOAM Designer
File
Edit
View
Settings
Tools
Help
noname
Name
Size
system
fvSolution
fvSchemes
controlDict
constant
polyMesh
points
owner
neighbour
faces
boundary
blockMeshDict
transportProperties
0
U
p
-p
$FOAM_RUN
blockMeshDict
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
miscellaneous
parallelProcessing
preProcessing
surface
thermophysical
Solvers
Mesh utilities
Post processing
Utilities
C:\OpenFOAM\Test\blockMeshDict
Solver Control
Discretization schemes
Solver tolerance and algorithm
Mesh Parameter
Material (Transport) Properties
Initialization
Boundary conditions
| \\ / O peration | Version: 2.3.1 |
| \\ / A nd | Web: www.OpenFOAM.org |
| \\/ M anipulation |
FoamFile
{
version 2.0;
format ascii;
class dictionary;
object blockMeshDict;
}
convertToMeters 1;
vertices #codeStream
{
codeInclude
#{
#include "pointField.H"
#};
code
#{
pointField points(19);
points[0] = point(0.5, 0, -0.5);
points[1] = point(1, 0, -0.5);
points[2] = point(2, 0, -0.5);
points[3] = point(2, 0.707107, -0.5);
points[4] = point(0.707107, 0.707107, -0.5);
points[5] = point(0.353553, 0.353553, -0.5);
points[6] = point(2, 2, -0.5);


File
Edit
View
Settings
Tools
Help
Test
Name
C:
D:
E:
F:
controlDict
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
miscellaneous
parallelProcessing
preProcessing
surface
thermophysical
Solvers
Mesh utilities
Post processing
Utilities
?
Full path of currently edited file
Due to error shown on previous slide, there is a question mark here


controlDict - OpenFOAM Designer
File Edit View Settings Tools Help
New Ctrl+N
Open file...
Open Ctrl+O
Close Ctrl+W
Delete
Save Ctrl+S
Save all
New case...
Open case...
Save template
Exit Ctrl+Q
Size
U
p
-p
$FOAM_RUN
Open new case
controlDict
/*---------------------------------*- C++ -*----------------------------------*\
=========	
\\ / F ield	OpenFOAM: The Open Source CFD Toolbox
\\ / O peration	Version: 1.5
\\ / A nd	Web: http://www.OpenFOAM.org
\\/ M anipulation	
---/
FoamFile
{
version 2.0;
format ascii;
class dictionary;
object controlDict;
}
// * //
application icoFoam;
startFrom startTime;
startTime 0;
stopAt endTime;
endTime 0.5;
deltaT 0.005;
writeControl timeStep;
writeInterval 20;
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
miscellaneous
parallelProcessing
preProcessing
surface
thermophysical
Solvers
Mesh utilities
Post processing
Utilities
C:\cfd\OpenFOAM-1.5\templates\noname\system\controlDict
Create a new OpenFOAM case


blockMeshDict - OpenFOAM Designer
File Edit View Settings Tools Help
New Ctrl+N
Open file...
Open Ctrl+O
Close Ctrl+W
Delete
Save Ctrl+S
Save all
New case...
Open case...
Save template
Exit Ctrl+Q
Each file open in separate tabs
controlDict
blockMeshDict
blockMeshDict
Size
FoamFile
version 2.0;
format ascii;
class dictionary;
object blockMeshDict;
Close selected tabs/ windows
convertToMeters 0.1;
vertices
blocks
blockMeshDict
transportProperties
0
U
p
-p
$FOAM_RUN
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
miscellaneous
parallelProcessing
preProcessing
surface
thermophysical
Solvers
Mesh utilities
Post processing
Utilities
C:\cfd\OpenFOAM-1.5\templates\noname\constant\polyMesh\blockMeshDict
Close current editor

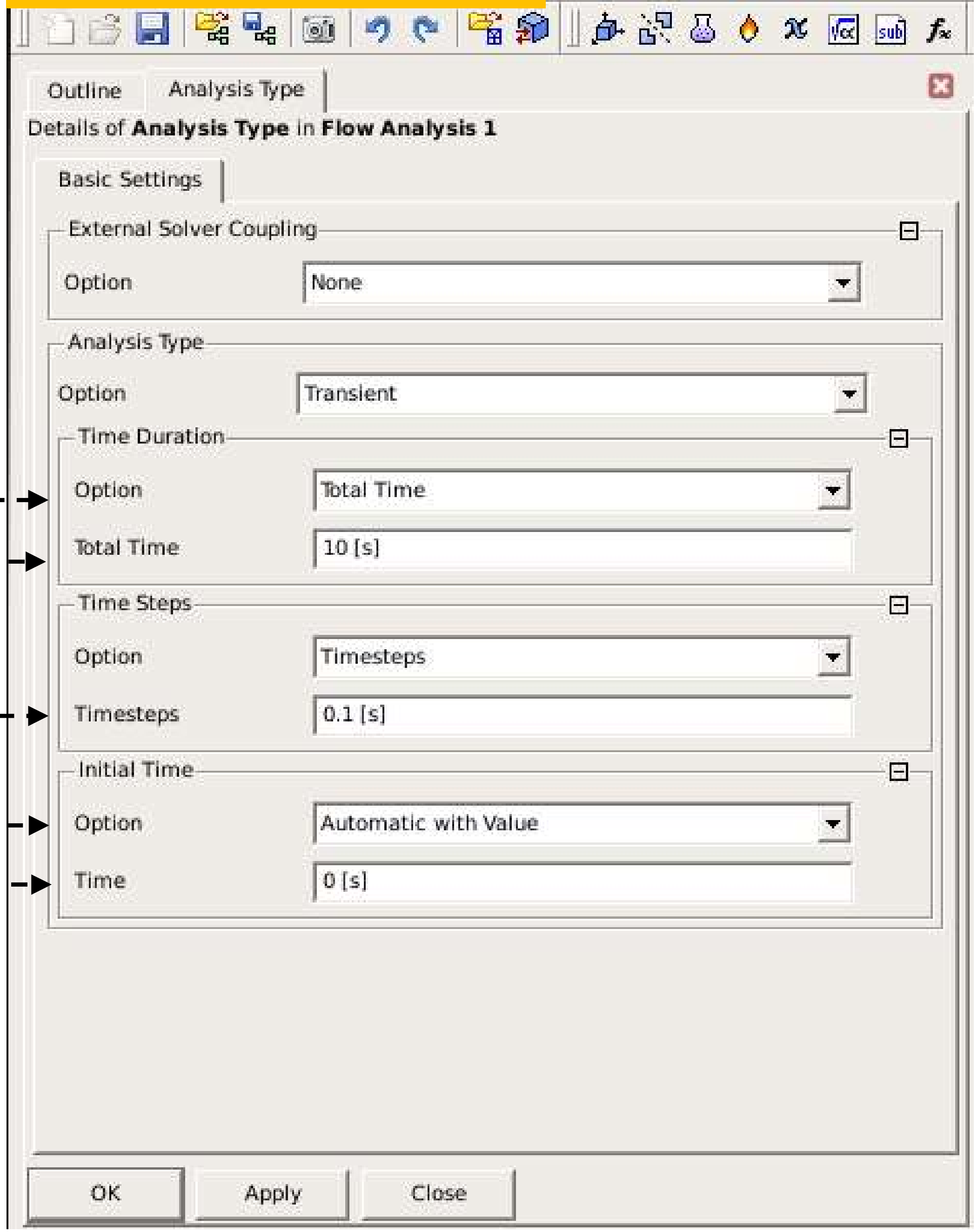

controlDict - OpenFOAM Designer
File
Edit
View
Settings
Tools
Help
View case
Case folder
Running
Solvers
Mesh utilities
Post processing
Utilities
Output
Refresh
Name
VTK
0.5
0.4
0.3
0.2
0.1
0
Test.openFOAM
C++
F ield
O peration
A nd
M anipulation
OpenFOAM: The Open Sourc
Version: 1.5
Web: http://www.Ope
version 2.0;
format ascii;
class dictionary;
object controlDict;
application icoFoam;
startFrom startTime;
startTime 0;
stopAt endTime;
endTime 0.5;
deltaT 0.005;
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
miscellaneous
parallelProcessing
preProcessing
surface
thermophysical
Solvers
Mesh utilities
Post processing
Utilities
C:\OpenFOAM\Test\system\controlDict

Mapping of content in OpenFOAM dictionaries and ANSYS CFX/Fluent

System/controlDict file

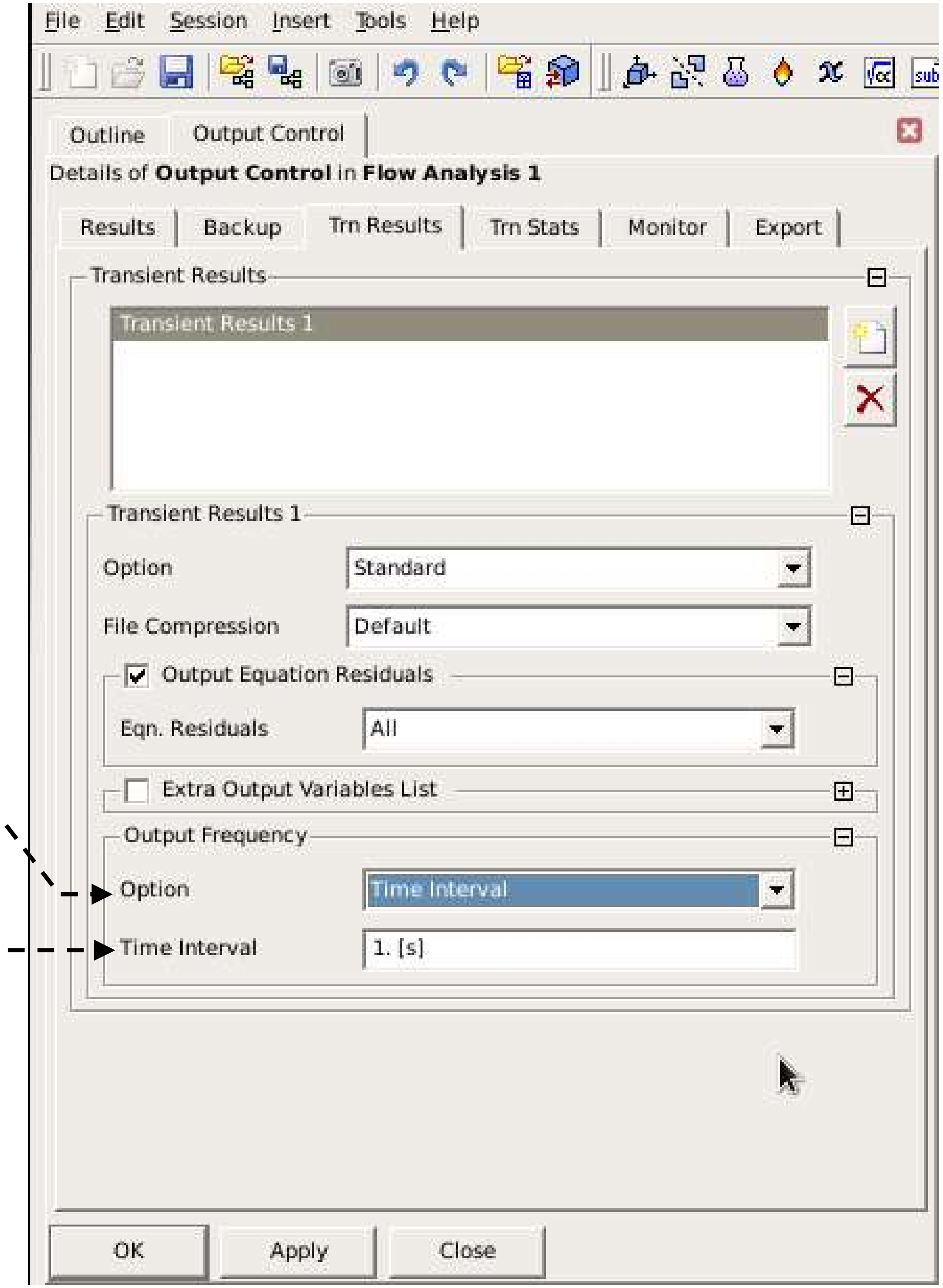
```
application potentialFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         1;

deltaT          1;

writeControl    timeStep;

writeInterval   1;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression uncompressed;

timeFormat      general;

timePrecision   6;

runTimeModifiable yes;
```

CFX Transient Analysis Setting Tab



System/controlDict file

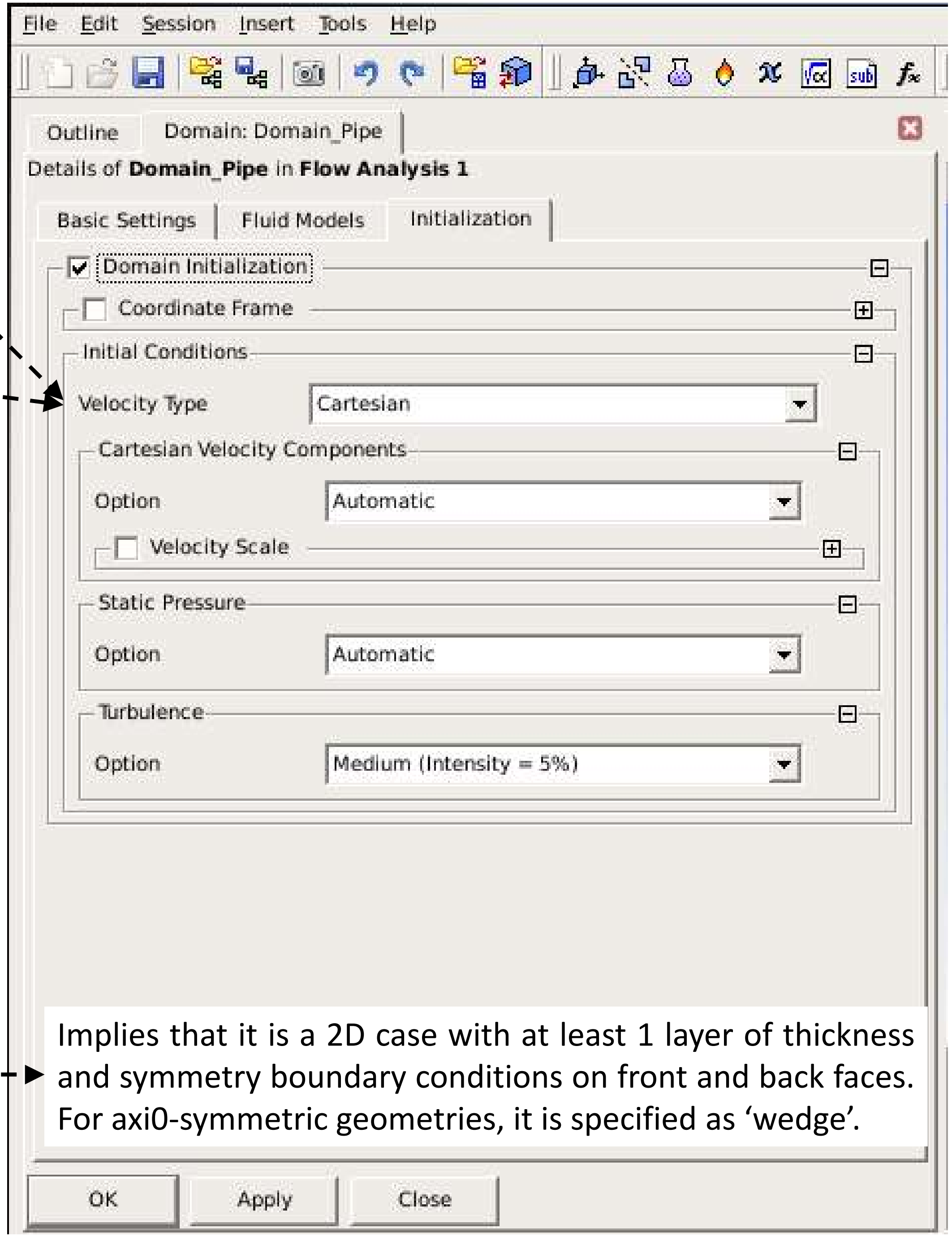
```
application potentialFoam;

startFrom        startTime;

startTime        0;

stopAt           endTime;

endTime          1;

deltaT           1;

writeControl     timeStep;

writeInterval    1;

purgeWrite       0;

writeFormat      ascii;

writePrecision   6;

writeCompression uncompressed;

timeFormat       general;

timePrecision    6;

runTimeModifiable yes;
```



‘Time’: 0/U file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform (0 0 0);

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform (10 0 0);
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    lowerWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    frontAndBack
    {
        type            empty;
    }
}
```

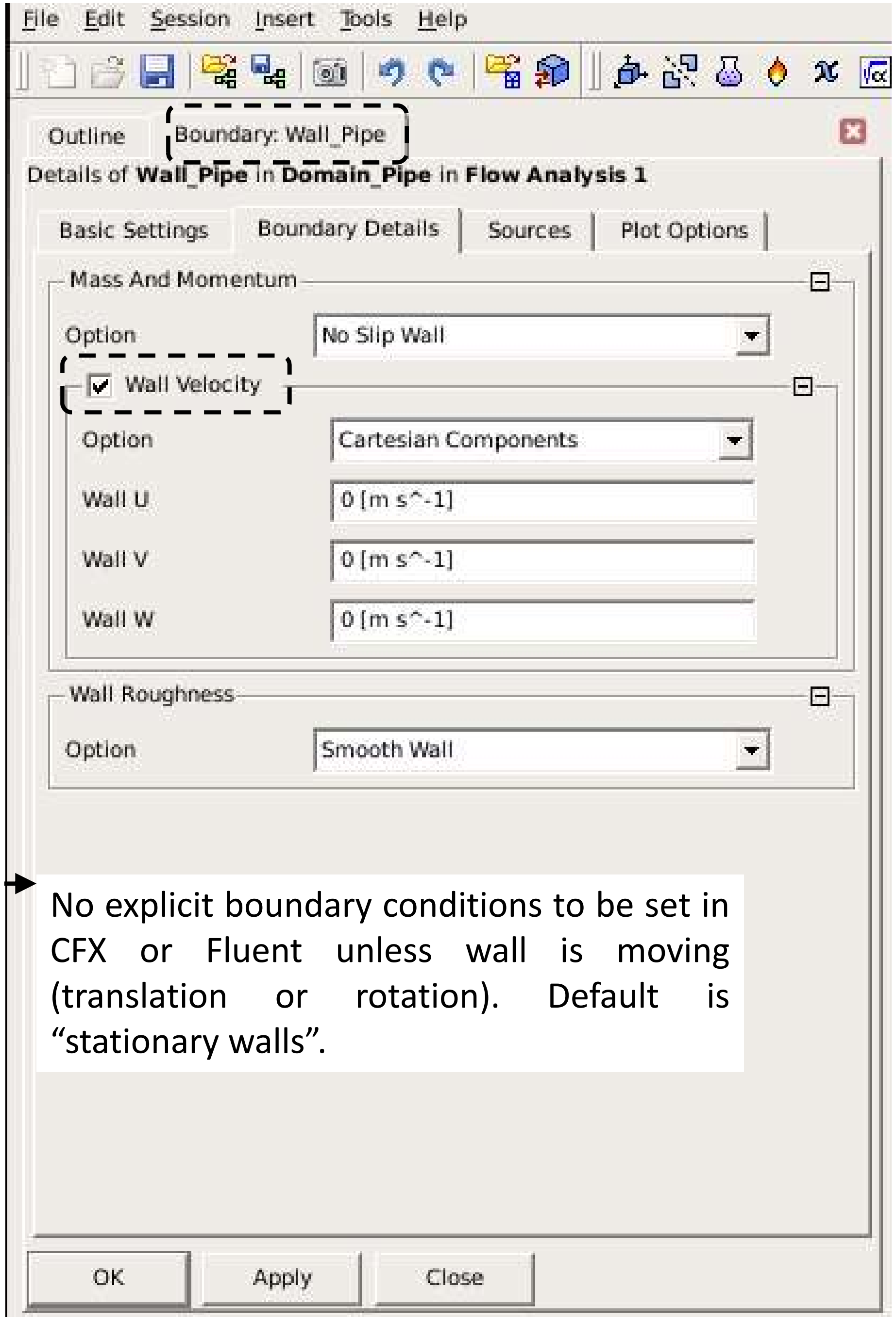
Value of ϕ is specified



Implies that it is a 2D case with at least 1 layer of thickness and symmetry boundary conditions on front and back faces. For axi0-symmetric geometries, it is specified as ‘wedge’.

'Time': 0/U file

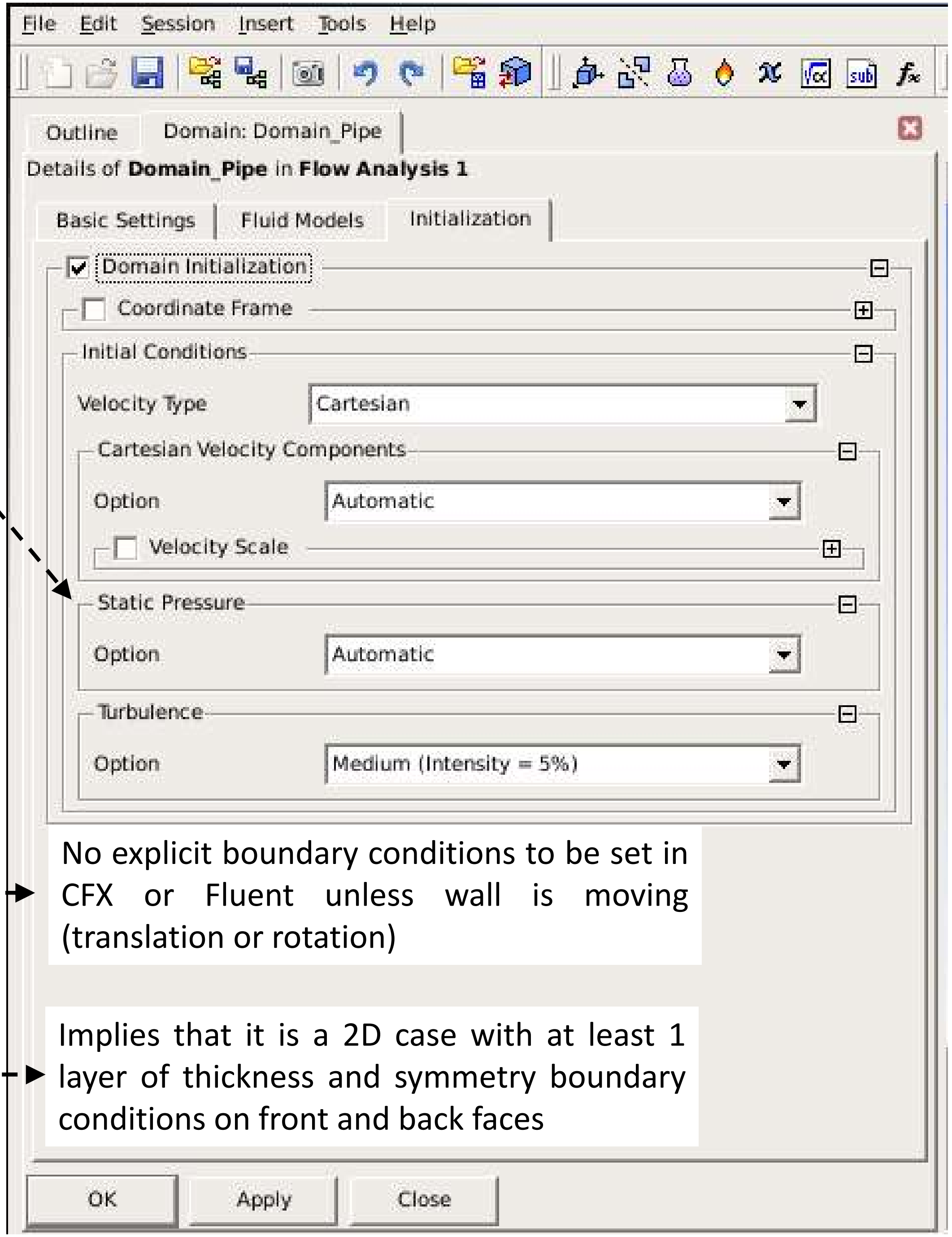
```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform (0 0 0);

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform (10 0 0);
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    lowerWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    frontAndBack
    {
        type            empty;
    }
}
```



No explicit boundary conditions to be set in CFX or Fluent unless wall is moving (translation or rotation). Default is "stationary walls".

‘Time’: 0/p file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform 0;

boundaryField
{
    inlet
    {
        type            zeroGradient;
    }

    outlet
    {
        type            fixedValue;
        value           uniform 0;
    }

    upperWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```

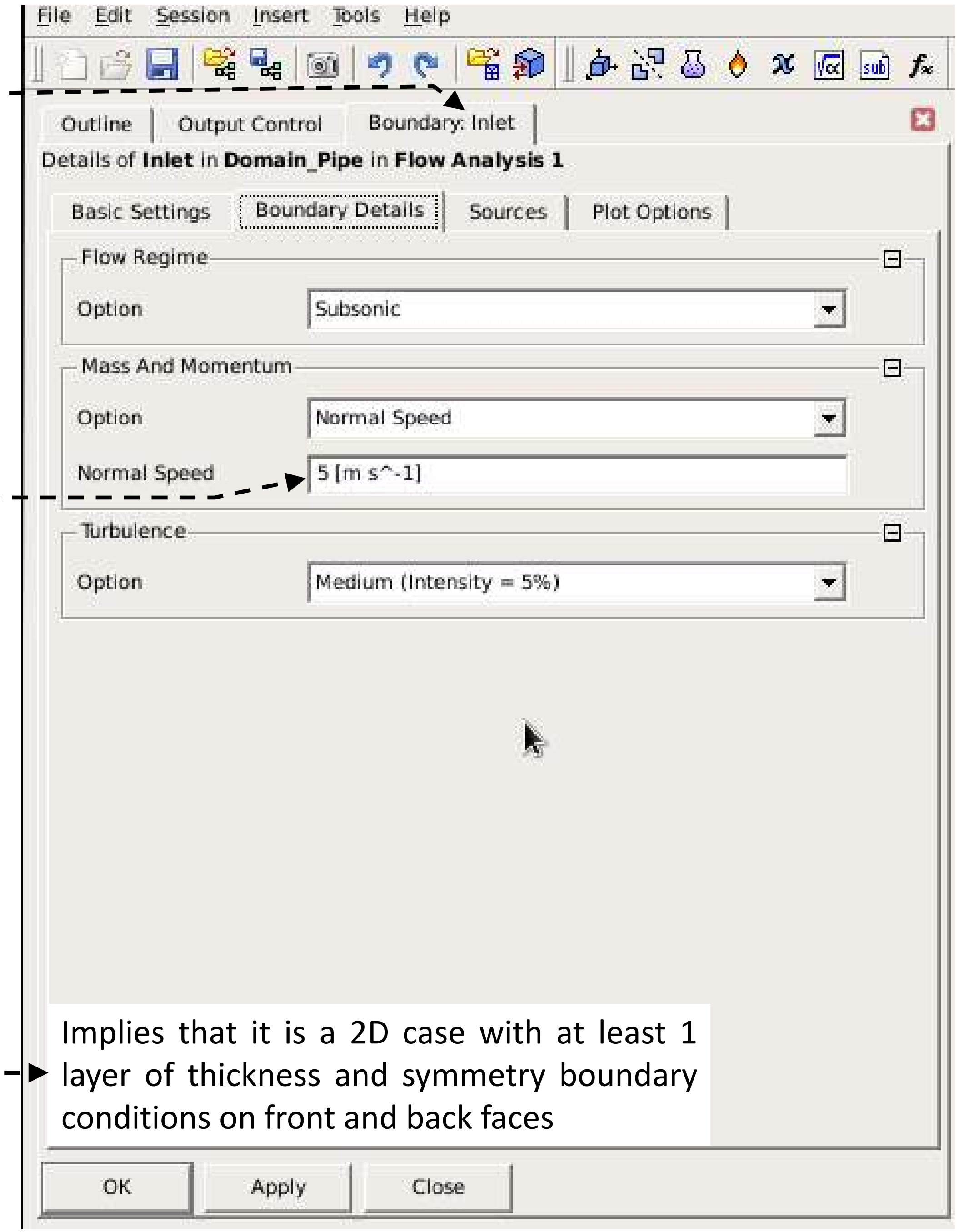


No explicit boundary conditions to be set in CFX or Fluent unless wall is moving (translation or rotation)

Implies that it is a 2D case with at least 1 layer of thickness and symmetry boundary conditions on front and back faces

'Time': 0/U file

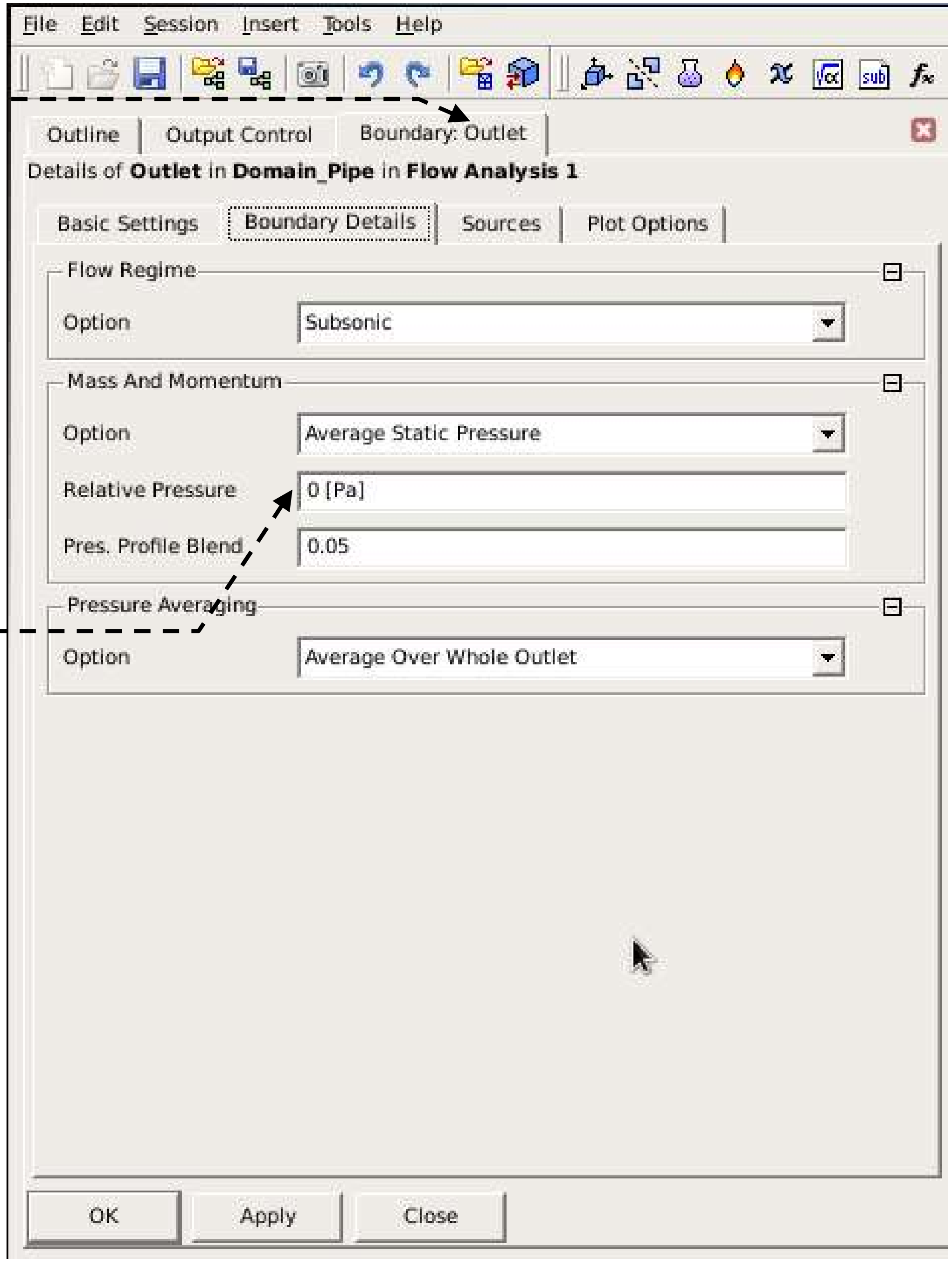
```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform (0 0 0);

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform (10 0 0);
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    lowerWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    frontAndBack
    {
        type            empty;
    }
}
```



Implies that it is a 2D case with at least 1 layer of thickness and symmetry boundary conditions on front and back faces

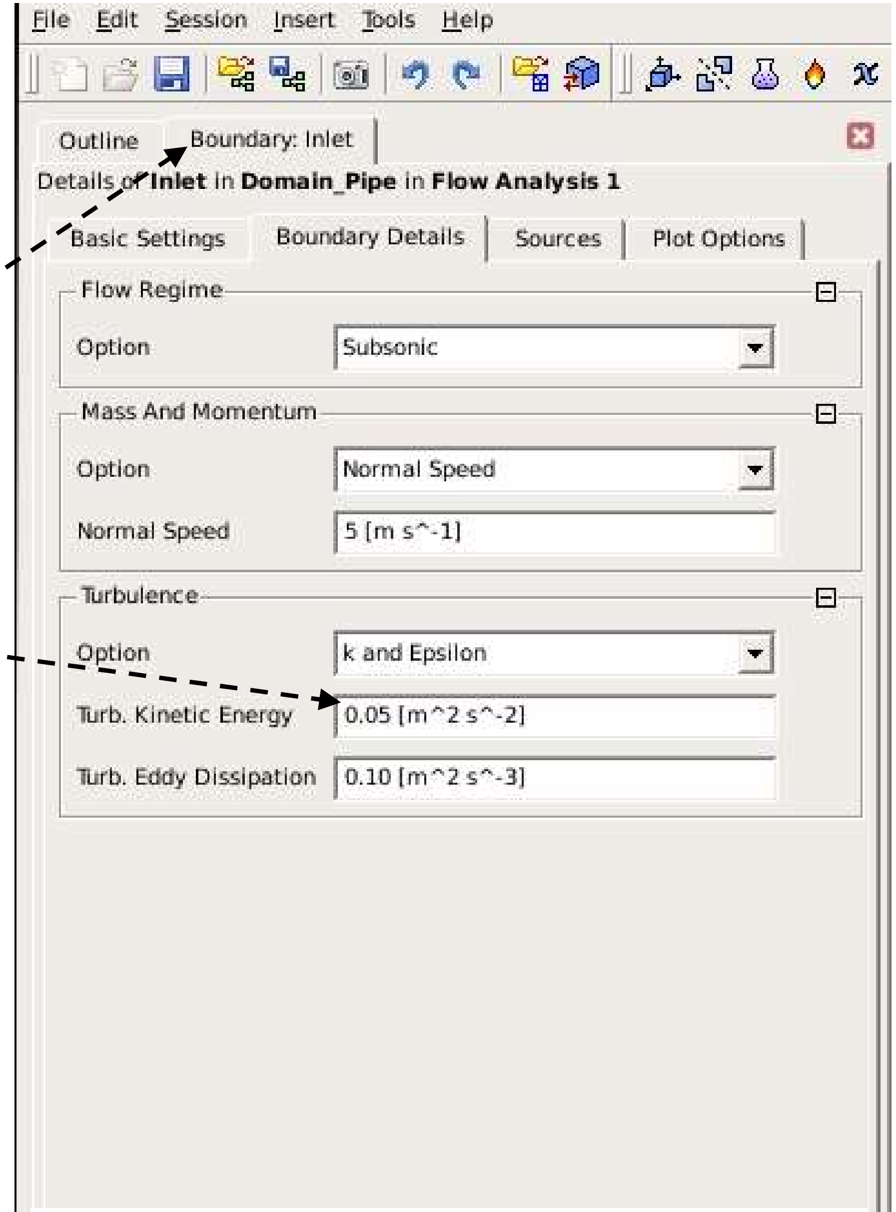
‘Time’: 0/U file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * * * * * * * * * * * * * * * * * * * *

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

internalField   uniform (0 0 0);

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform (10 0 0);
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    lowerWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }

    frontAndBack
    {
        type            empty;
    }
}
```



'Time': 0/k file

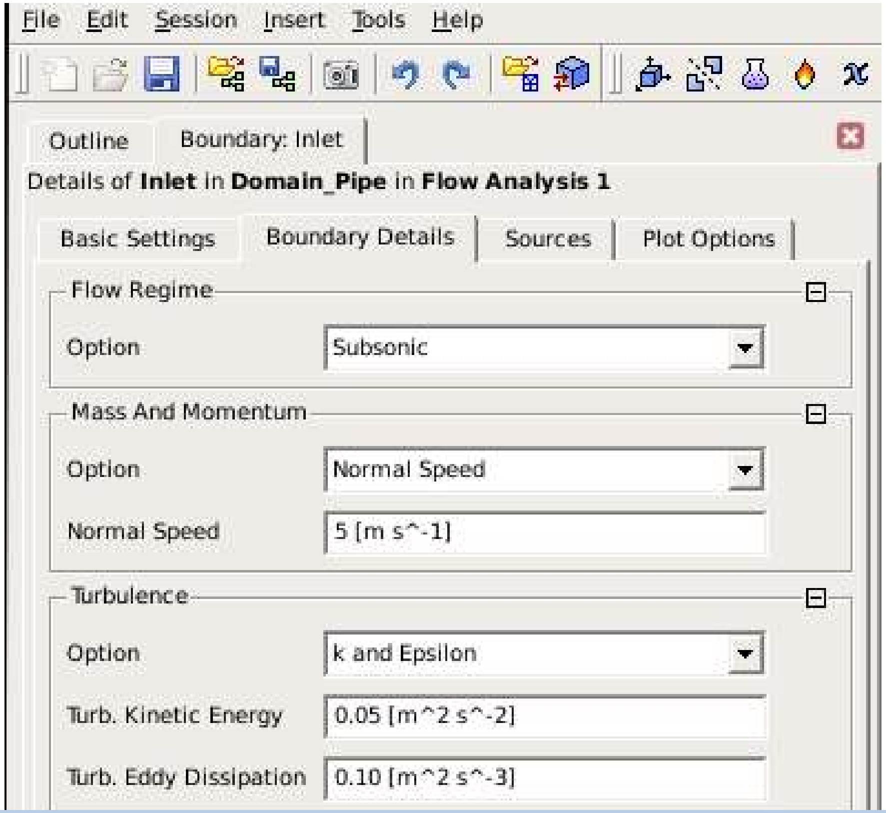
```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      k;
}
// * * * * * * * * * * * * * * * * * * *

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

internalField   uniform 0.375;

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 0.375;
    }
    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```



Implies that it is a 2D case with at least 1 layer of thickness and symmetry boundary conditions on front and back faces

'Time': 0/k file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      k;
}
// * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 0.375;

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 0.375;
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```

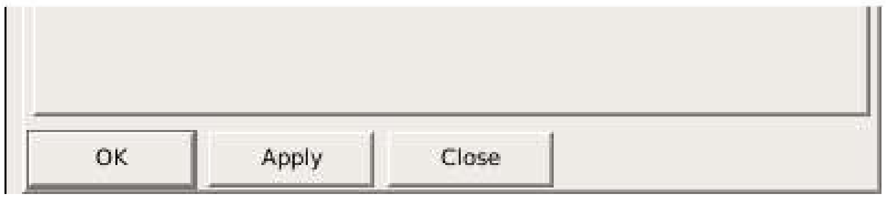


Default setting in CFX and Fluent: Excerpts from Fluent User Manual:

In the k-ε models and in the RSM (if the option to obtain wall boundary conditions from the k equation is enabled), the k equation is solved in the whole domain including the wall-adjacent cells. The boundary condition for k imposed at the wall is

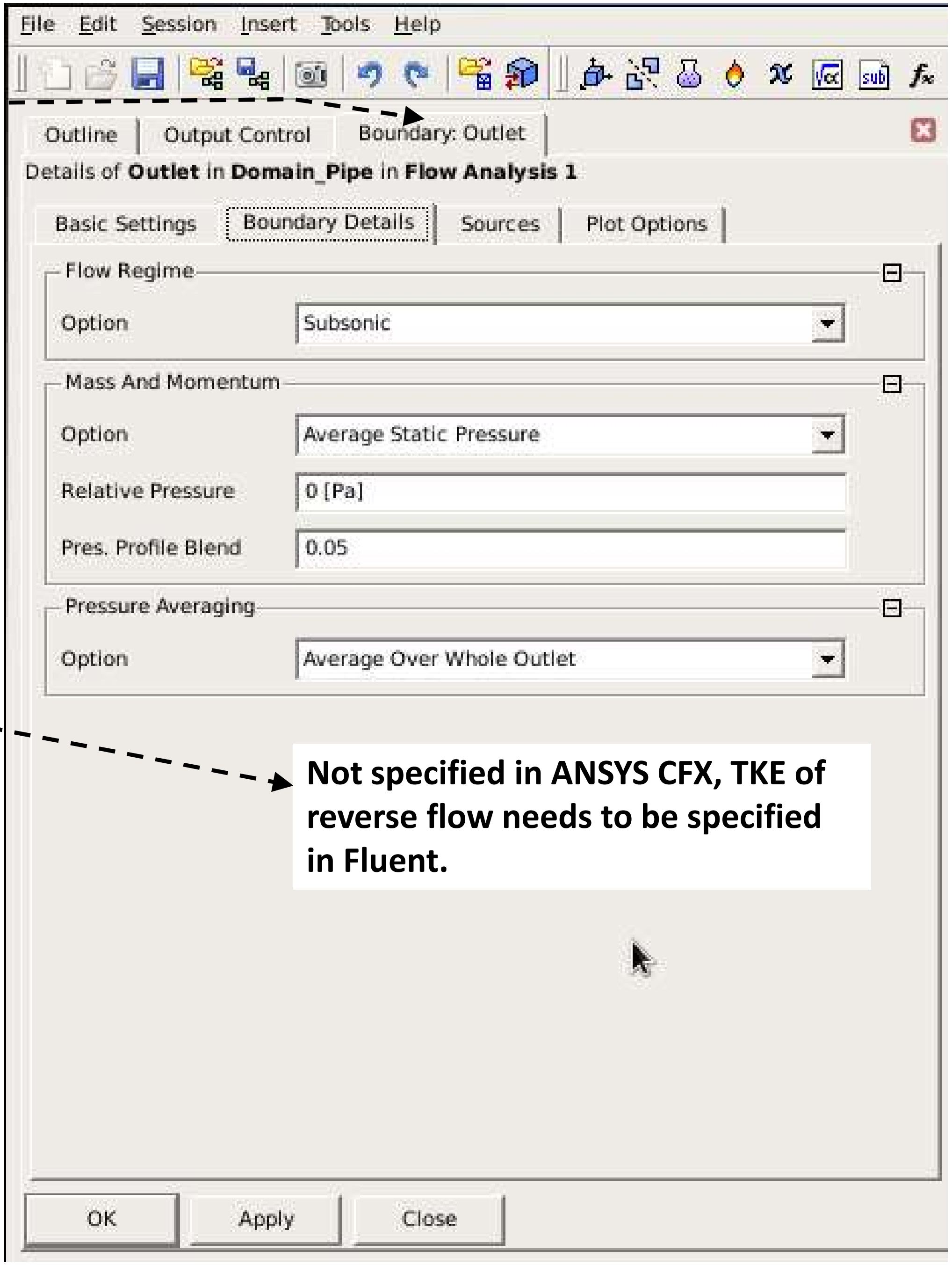
$$\frac{\partial k}{\partial n} = 0$$

where n is the local coordinate normal to the wall.



'Time': 0/k file

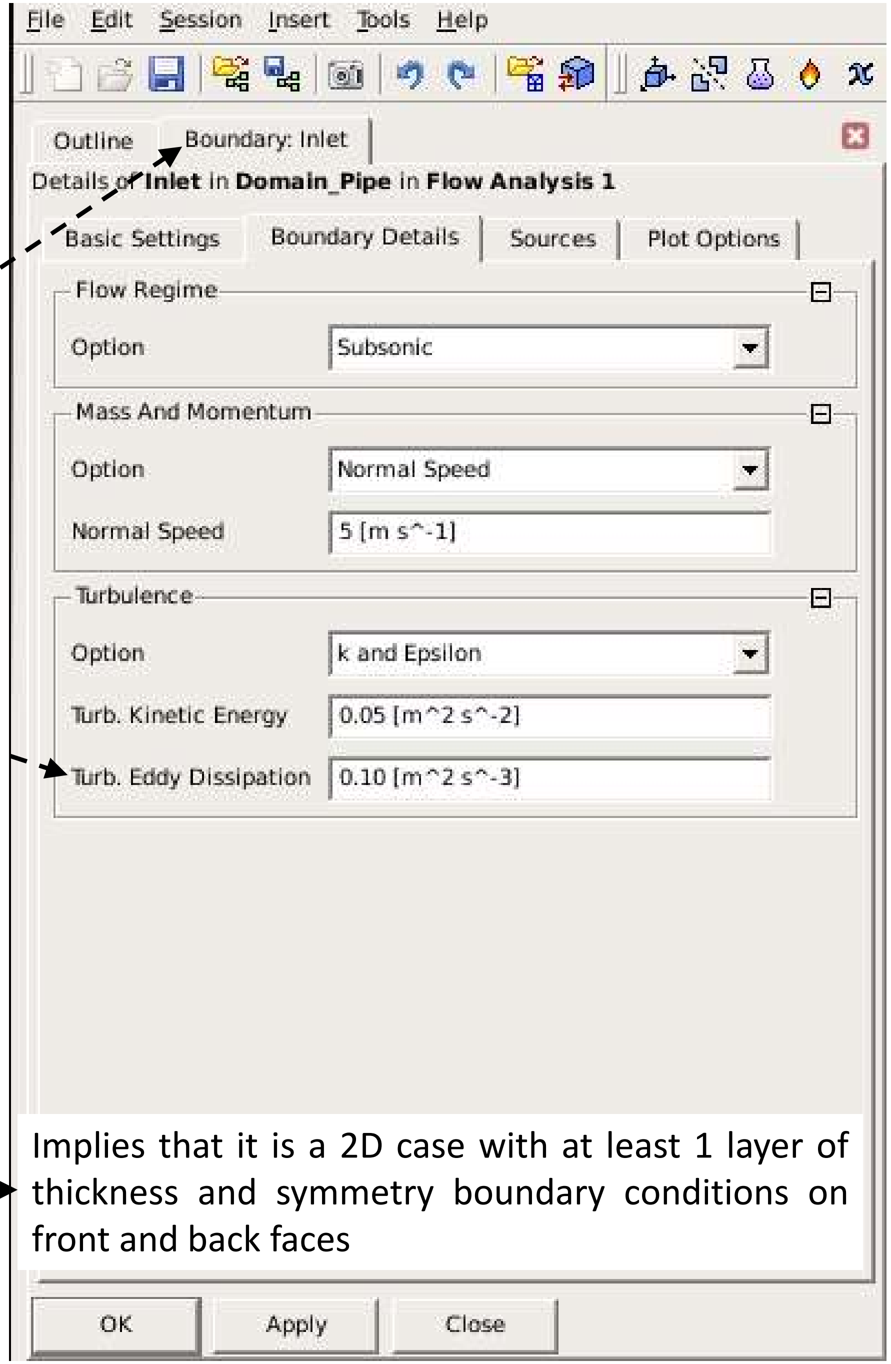
```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      k;
}
// * * * * * * * * * * * * * * * * * * 

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

internalField   uniform 0.375;

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 0.375;
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```



Not specified in ANSYS CFX, TKE of reverse flow needs to be specified in Fluent.

'Time': 0/epsilon file

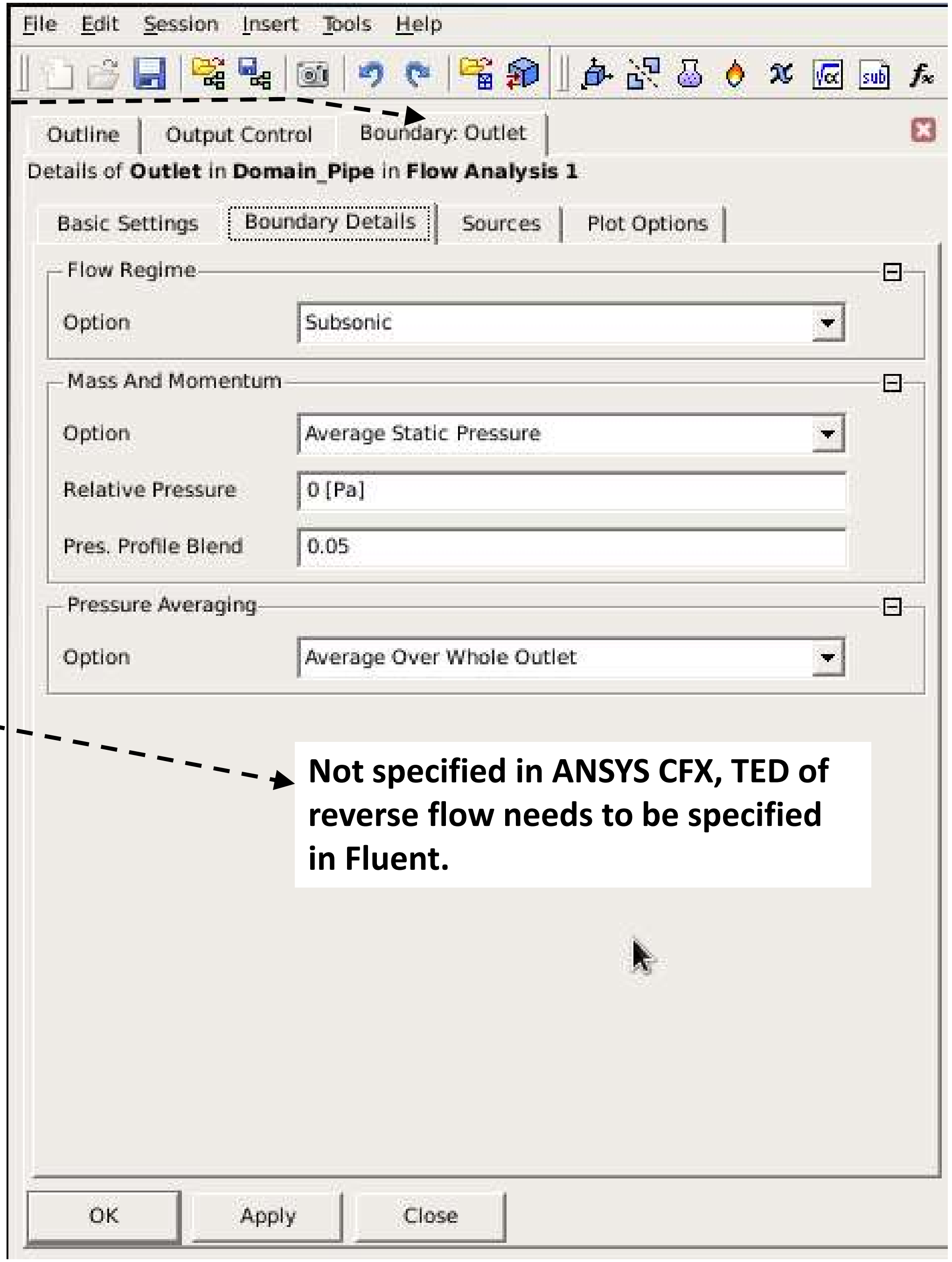
```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      epsilon;
}
// * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 14.855;

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 14.855;
    }
    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```



Implies that it is a 2D case with at least 1 layer of thickness and symmetry boundary conditions on front and back faces

‘Time’: 0/epsilon file

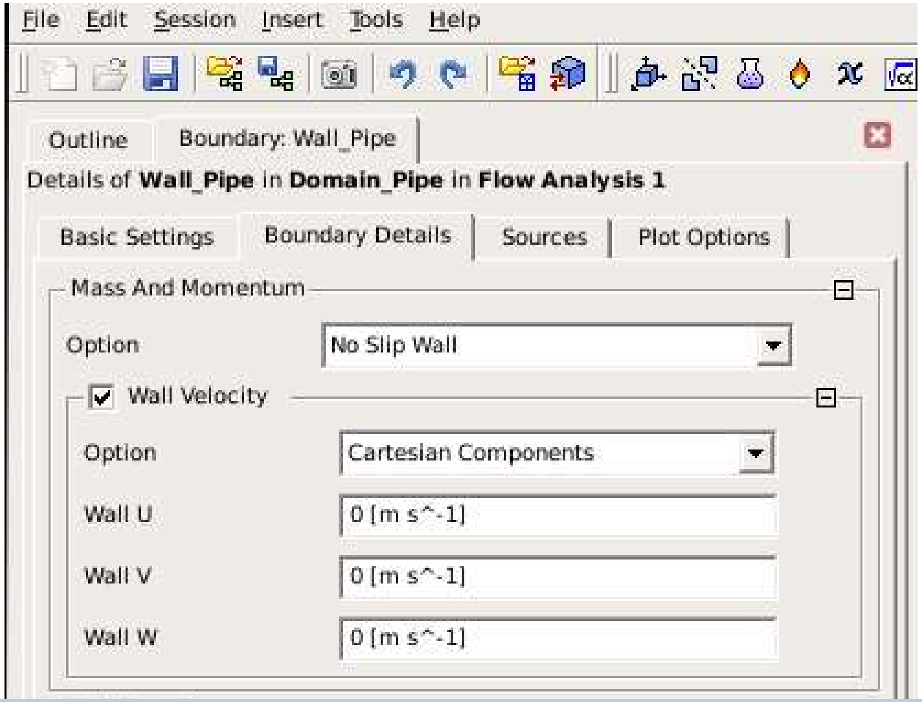
```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      epsilon;
}
// * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 14.855;

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 14.855;
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```



Not specified in ANSYS CFX, TED of reverse flow needs to be specified in Fluent.

'Time': 0/epsilon file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      epsilon;
}
// * * * * * * * * * * * * * * * * * * * * //

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

internalField   uniform 14.855;

boundaryField
{
    inlet
    {
        type            fixedValue;
        value           uniform 14.855;
    }

    outlet
    {
        type            zeroGradient;
    }

    upperWall
    {
        type            zeroGradient;
    }

    lowerWall
    {
        type            zeroGradient;
    }

    frontAndBack
    {
        type            empty;
    }
}
```

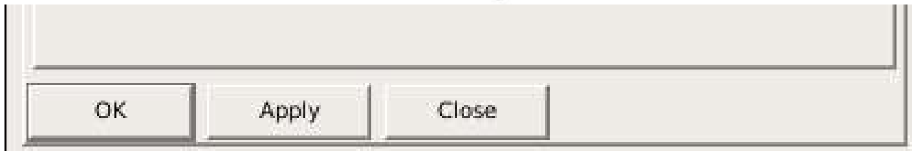


Default setting in CFX and Fluent: Excerpts from Fluent User Manual:

The ε equation is not solved at the wall-adjacent cells, but instead is computed using

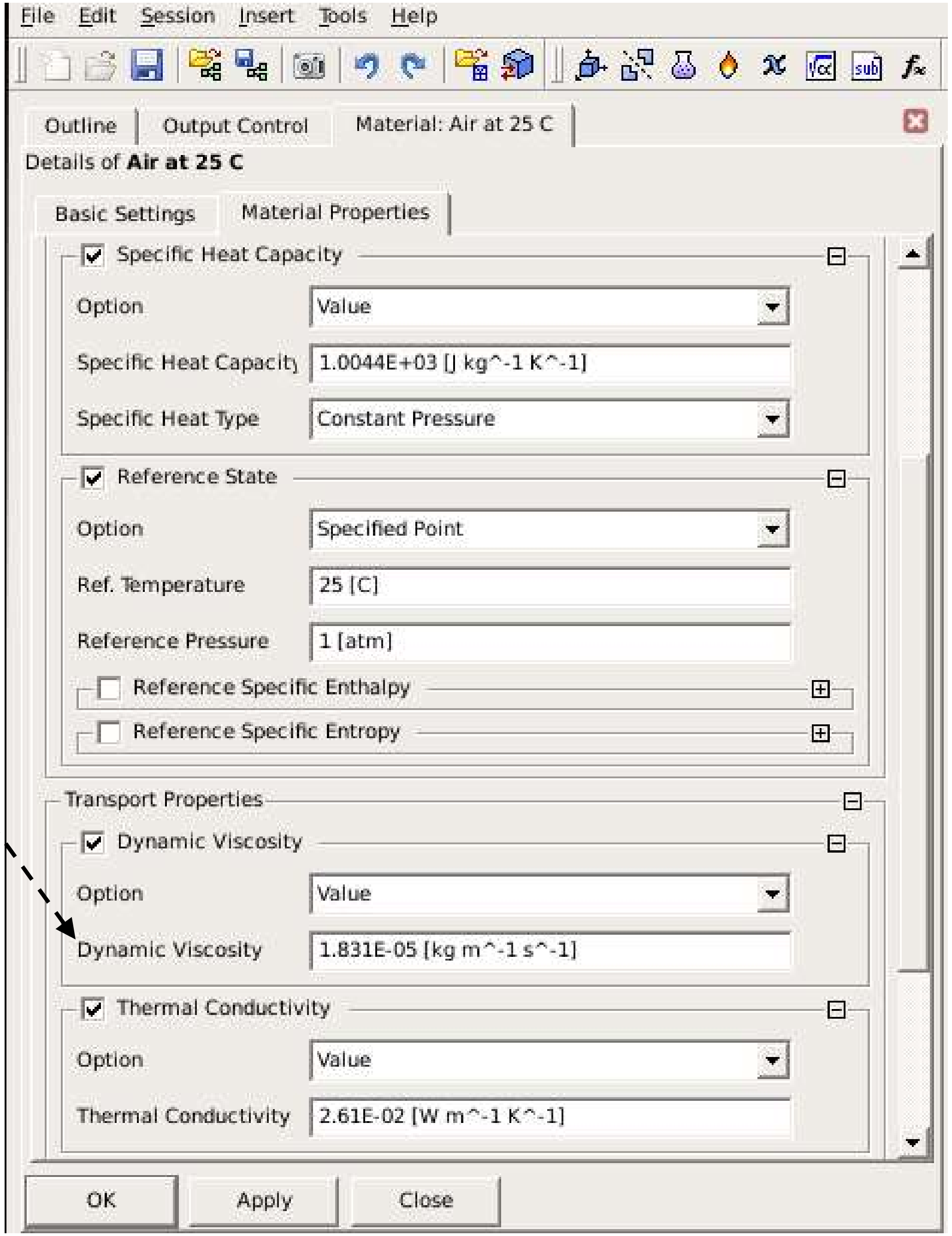
$$\varepsilon_P = \frac{C_\mu^{3/4} k_P^{3/2}}{\kappa y_P}$$

Note that, as shown here, the wall boundary conditions for the solution variables, including mean velocity, temperature, species concentration, k, and ε, are all taken care of by the wall functions. Therefore, you do not need to be concerned about the boundary conditions at the walls.



Constant/transportProperties file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * *

transportModel  Newtonian;

nu              nu [0 2 -1 0 0 0 0] 1e-05;

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] 1;
}

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



system/fvScheme file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSchemes;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * *
```

First and second time derivatives ∂/∂t, $\partial^2/\partial t^2$

```
ddtSchemes
{
    default steadyState;
}
```

Sub-dictionary of 'fvScheme': Gradient ∇

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

Gauss <interpolationScheme>

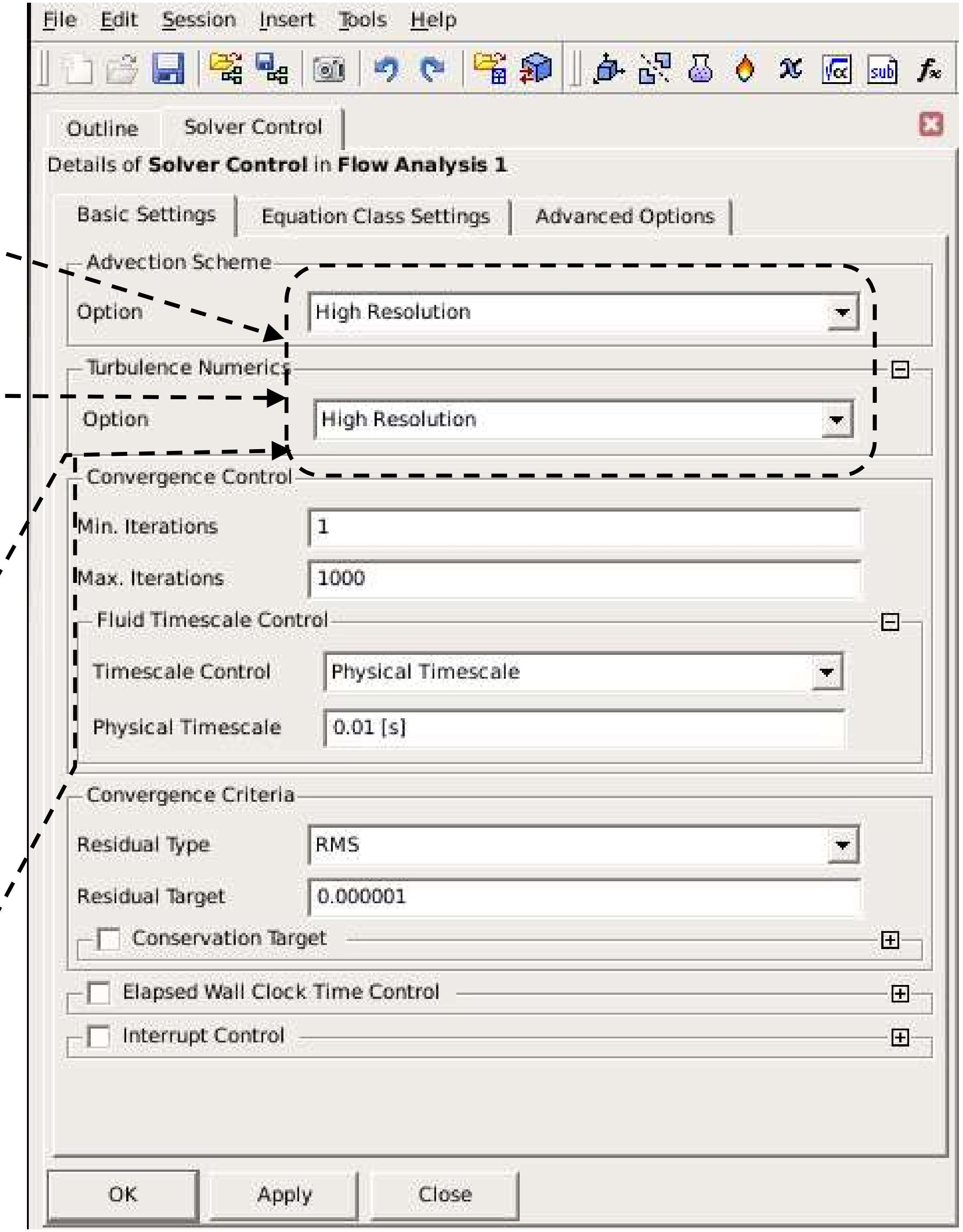
Divergence operator: ∇•

```
divSchemes
{
    default         none;
    div(phi,U)      Gauss upwind;
    div(phi,k)      Gauss upwind;
    div(phi,epsilon) Gauss upwind;
    div(phi,R)      Gauss upwind;
    div(R)          Gauss linear;
    div(phi,nuTilda) Gauss upwind;
    div((nuEff*dev(grad(U).T()))) Gauss linear;
}
```

Gauss <interpolationScheme>

Laplacian operator: $\nabla^2\phi$

```
laplacianSchemes
{
    default         none;
    laplacian(nuEff,U) Gauss linear corrected;
    laplacian((1|A(U)),p) Gauss linear corrected;
    laplacian(DkEff,k) Gauss linear corrected;
    laplacian(DepsilonEff,epsilon) Gauss linear corrected;
    laplacian(DREff,R) Gauss linear corrected;
    laplacian(DnuTildaEff,nuTilda) Gauss linear corrected;
}
```



system/fvScheme file

```
interpolationSchemes
{
    default         linear;
    interpolate(U)  linear;
}
```

The interpolationSchemes sub-dictionary contains terms that are interpolations of values typically from cell centers to face centers.

```
snGradSchemes
{
    default         corrected;
}
```

Component of gradient normal to a cell face: surface normal gradient terms

Explicit non-orthogonal correction

```
fluxRequired
{
    default         no;
    p;
}
```

Fields which require the generation of a flux

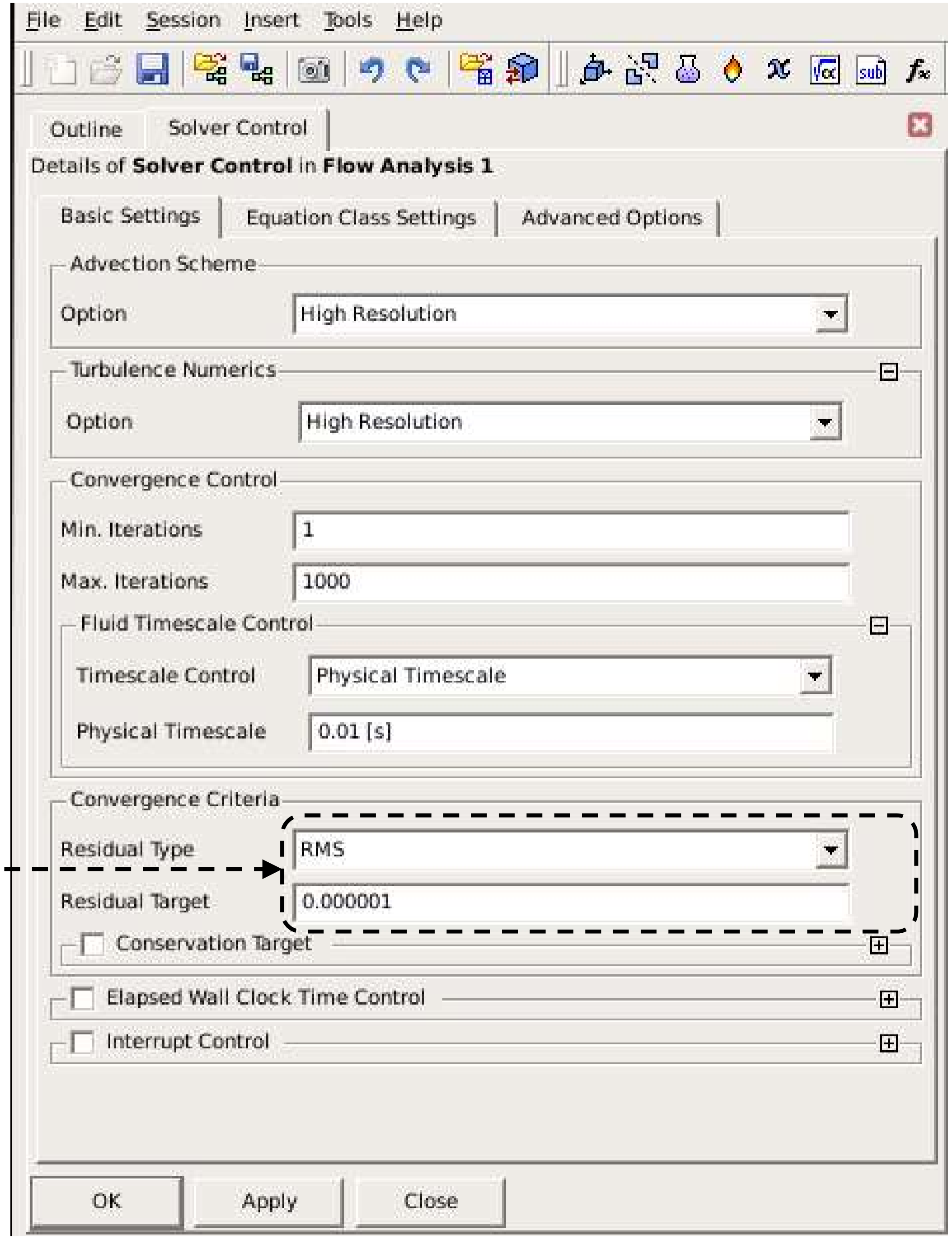
system/fvSolution file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * *

solvers
{
    p PCG
    {
        preconditioner   DIC;
        tolerance        1e-06;
        relTol           0.01;
    };
    U PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
    k PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
    epsilon PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
```

Diagonal Incomplete - Cholesky (symmetric)

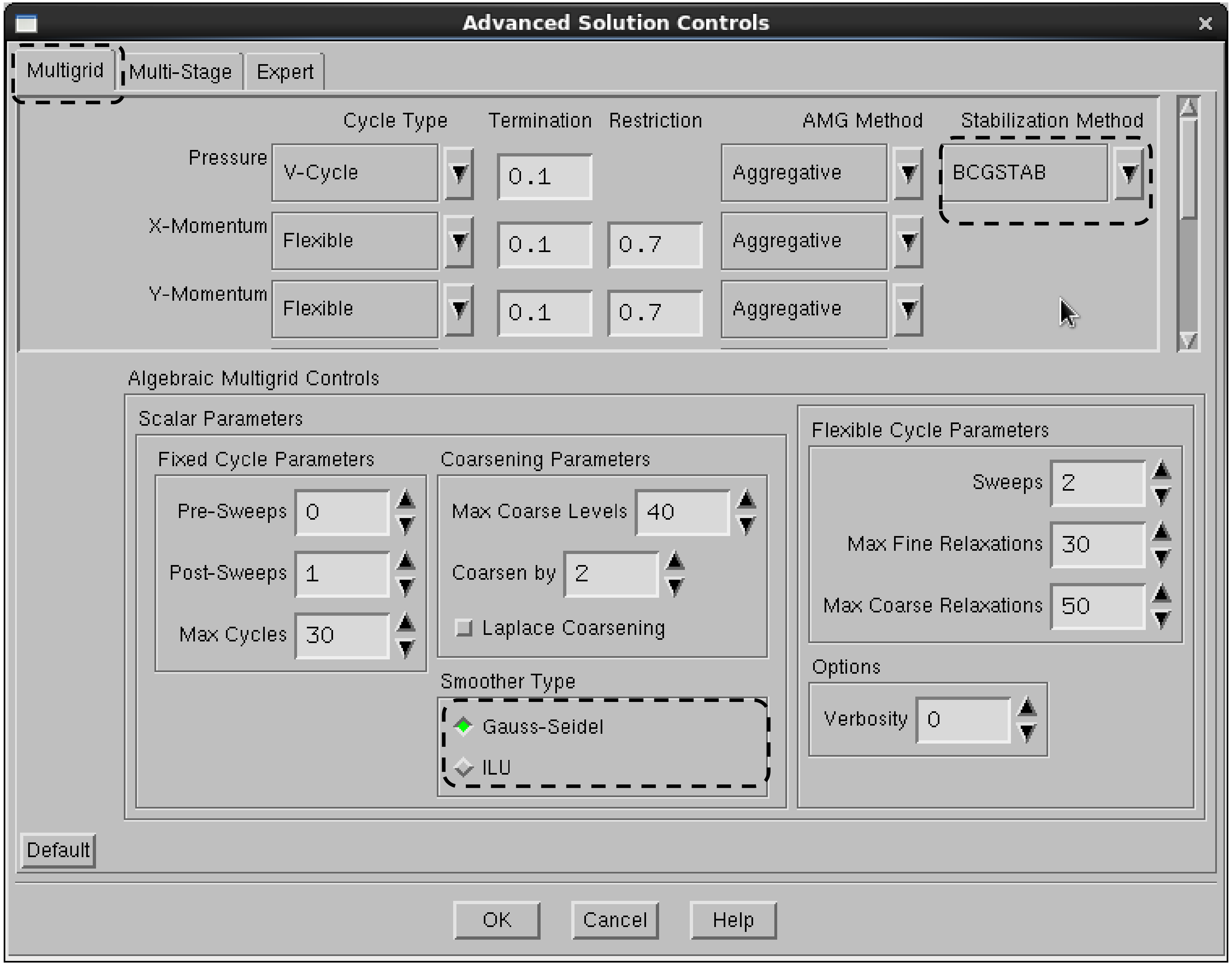
Preconditioned (bi-) conjugate gradient

Diagonal Incomplete -LU (asymmetric)



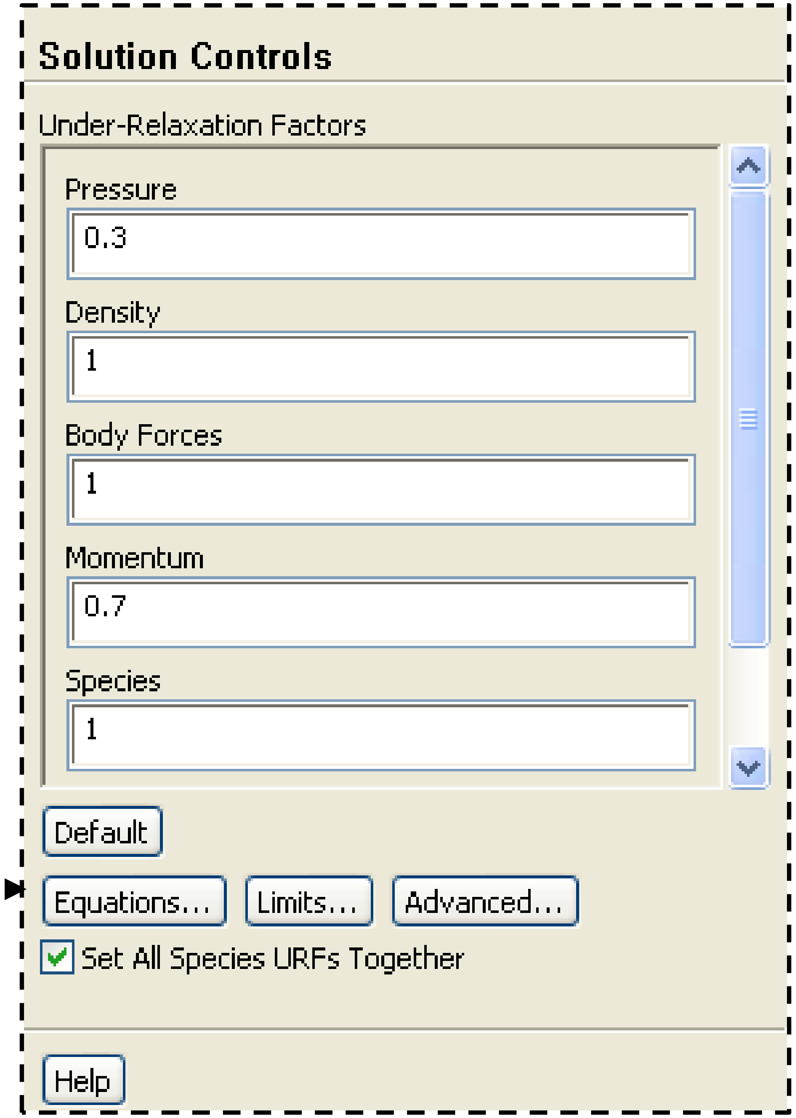
```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      fvSolution;
}
// * * * * * * * * * * * * * * * * *

solvers
{
    p PCG
    {
        preconditioner   DIC;
        tolerance        1e-06;
        relTol           0.01;
    };
    U PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
    k PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
    epsilon PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
```

system/fvSolution file



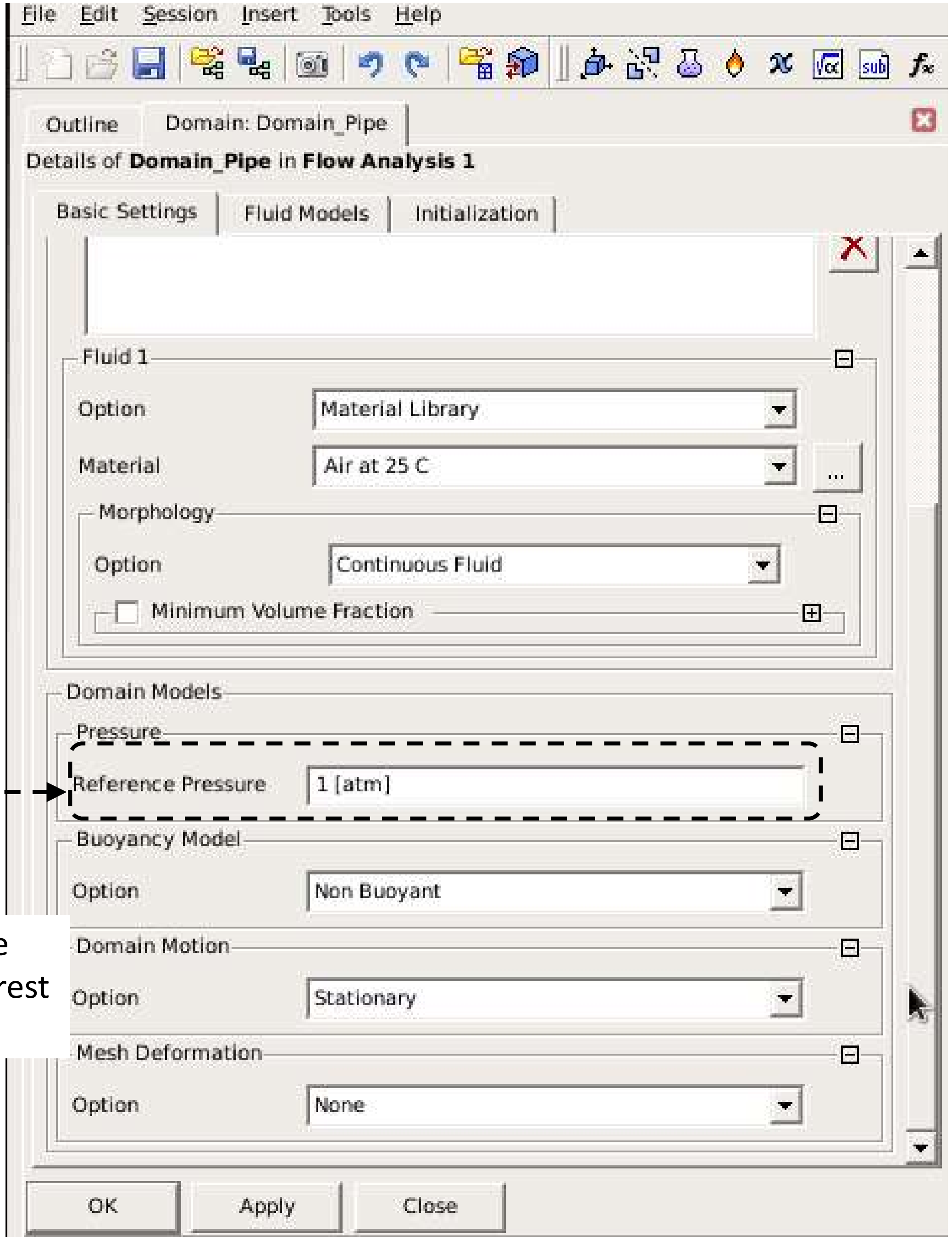
system/fvSolution file

```
    R PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
    nuTilda PBiCG
    {
        preconditioner   DILU;
        tolerance        1e-05;
        relTol           0.1;
    };
}

SIMPLE
{
    nNonOrthogonalCorrectors 0;
}

relaxationFactors
{
    p               0.3;
    U               0.7;
    k               0.7;
    epsilon         0.7;
    R               0.7;
    nuTilda         0.7;
}
```



pRefValue

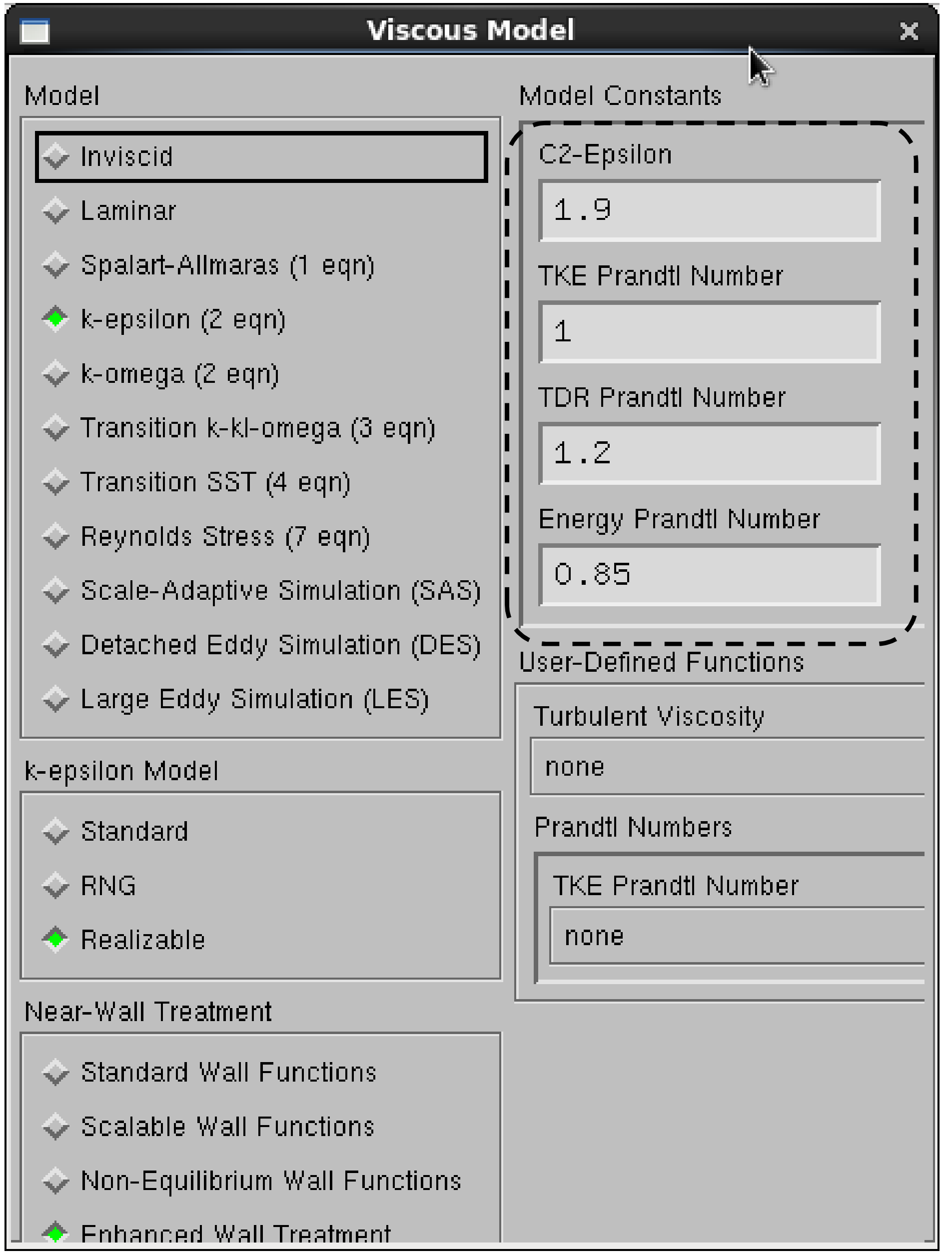
pRefCell



Fluent: Default location of reference pressure in Fluent is cell center nearest to origin (0, 0, 0)

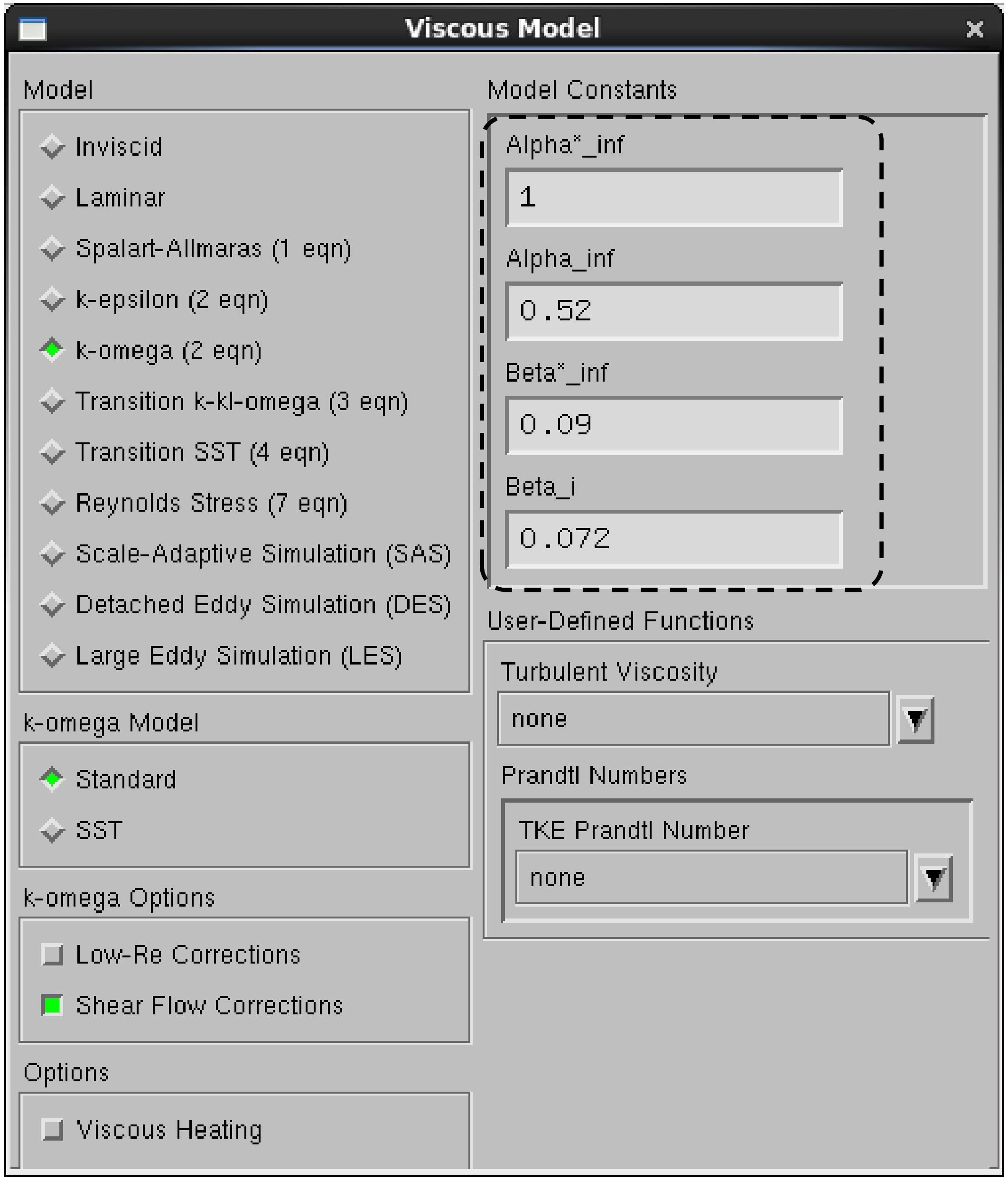
constant/ RASProperties file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      RASProperties;
}
// * * * * * * * * * * * * * * * * * * *

RASModel            kEpsilon;

turbulence          on;

printCoeffs         on;

laminarCoeffs
{
}

kEpsilonCoeffs
{
    Cmu              0.09;
    C1               1.44;
    C2               1.92;
    alphaEps         0.76923;
}

RNGkEpsilonCoeffs
{
    Cmu              0.0845;
    C1               1.42;
    C2               1.68;
    alphak           1.39;
    alphaEps         1.39;
    eta0             4.38;
    beta             0.012;
}
```



constant/ RASProperties file

```
realizableKECoeffs
{
    Cmu             0.09;
    A0              4.0;
    C2              1.9;
    alphak          1;
    alphaEps        0.833333;
}

kOmegaSSTCoeffs
{
    alphaK1         0.85034;
    alphaK2         1.0;
    alphaOmega1     0.5;
    alphaOmega2     0.85616;
    gamma1          0.5532;
    gamma2          0.4403;
    beta1           0.0750;
    beta2           0.0828;
    betaStar        0.09;
    a1              0.31;
    c1              10;

    Cmu             0.09;
}
```



Constant/RASProperties file

```
NonlinearKEShihCoeffs
{
    Cmu             0.09;
    C1              1.44;
    C2              1.92;
    alphak          1;
    alphaEps        0.76932;
    A1              1.25;
    A2              1000;
    Ctau1           -4;
    Ctau2           13;
    Ctau3           -2;
    alphaKsi        0.9;
}

LienCubicKECoeffs
{
    C1              1.44;
    C2              1.92;
    alphak          1;
    alphaEps        0.76923;
    A1              1.25;
    A2              1000;
    Ctau1           -4;
    Ctau2           13;
    Ctau3           -2;
    alphaKsi        0.9;
}
```

constant/ RASProperties file

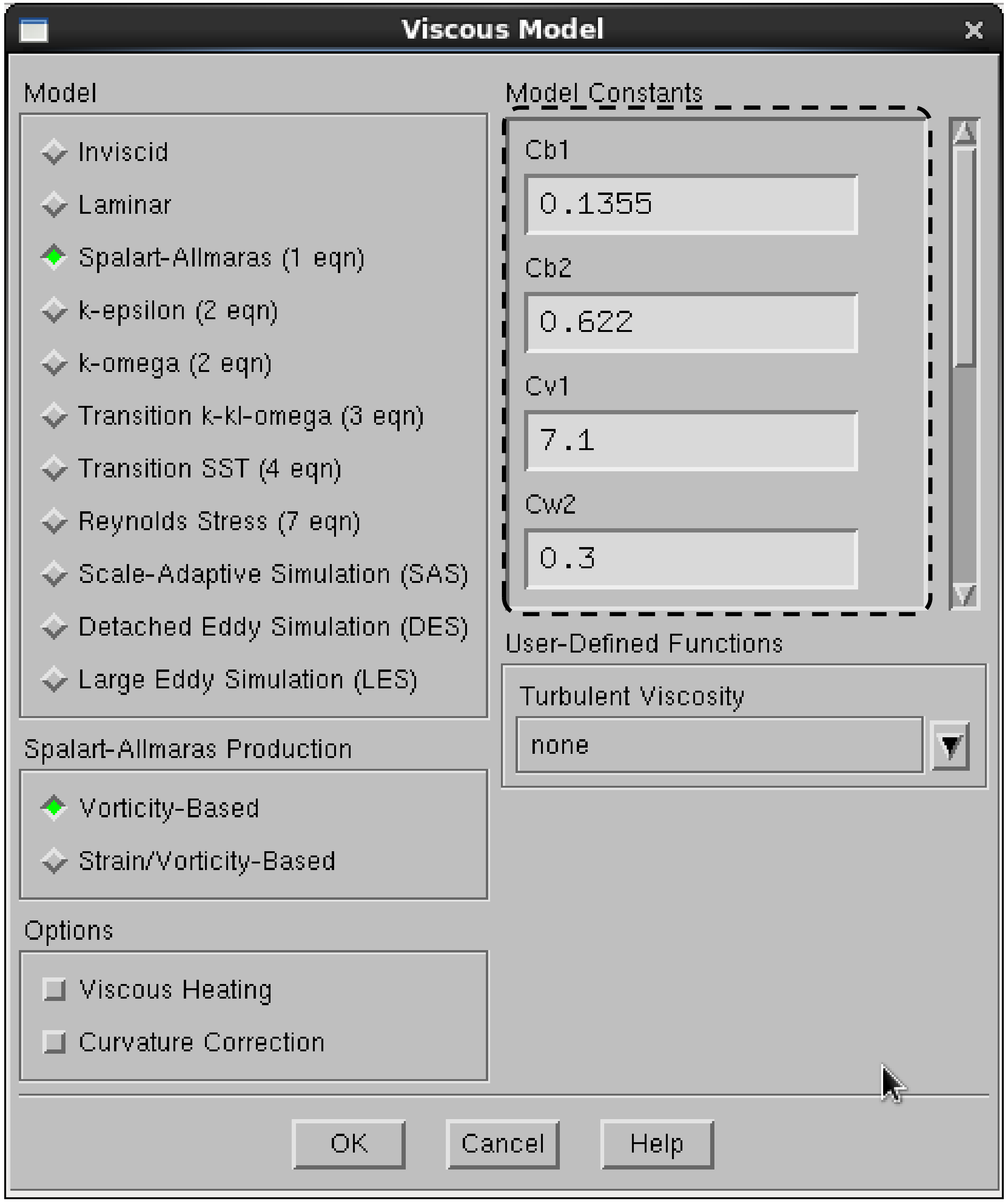
```
QZetaCoeffs
{
    Cmu                 0.09;
    C1                  1.44;
    C2                  1.92;
    alphaZeta           0.76923;
    anisotropic        no;
}

LaunderSharmaKECoeffs
{
    Cmu                 0.09;
    C1                  1.44;
    C2                  1.92;
    alphaEps            0.76923;
}

LamBremhorstKECoeffs
{
    Cmu                 0.09;
    C1                  1.44;
    C2                  1.92;
    alphaEps            0.76923;
}
```

constant/ RASProperties file

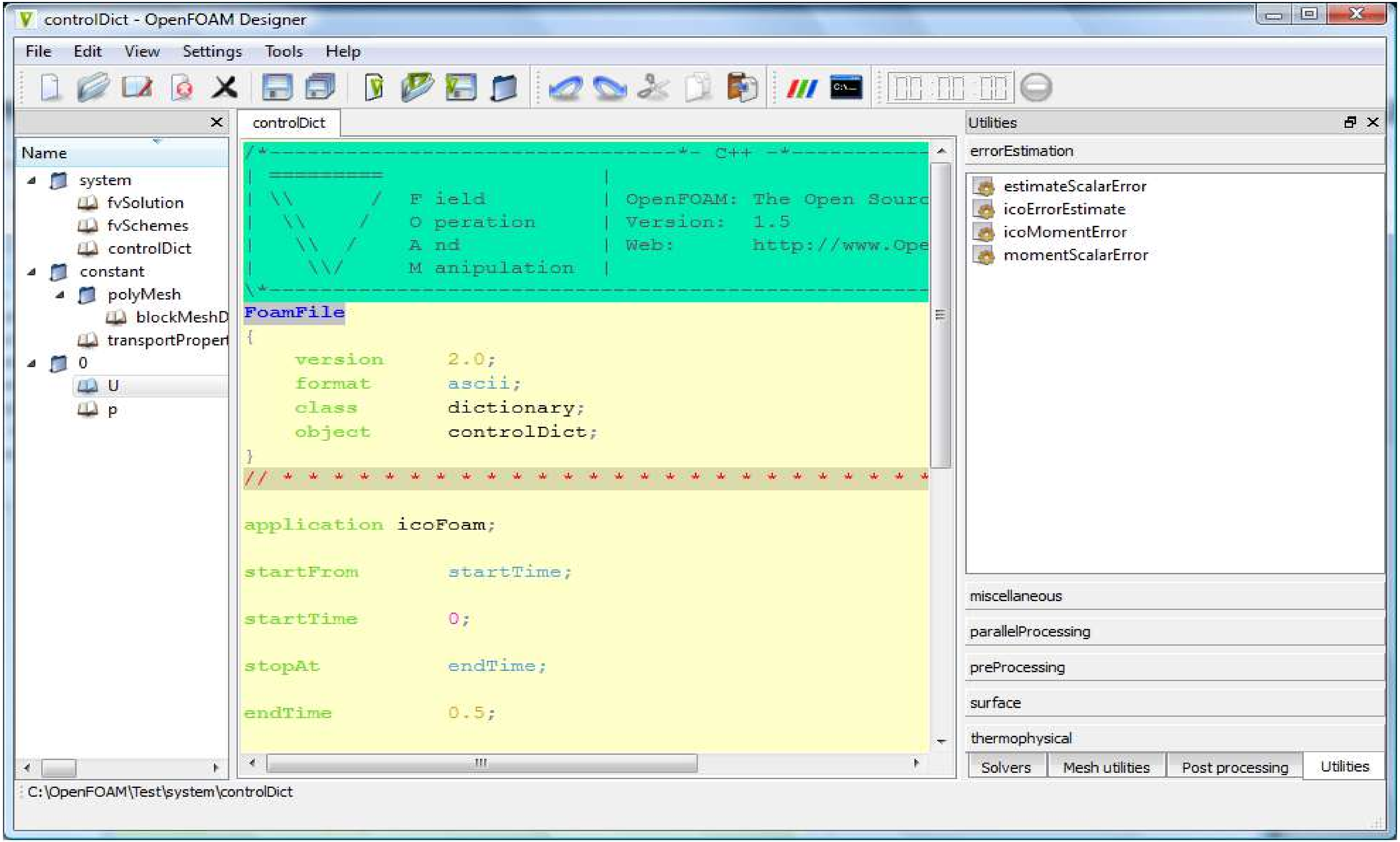
```
LienCubicKELowReCoeffs
{
    Cmu                 0.09;
    C1                  1.44;
    C2                  1.92;
    alphak              1;
    alphaEps            0.76923;
    A1                  1.25;
    A2                  1000;
    Ctau1               -4;
    Ctau2               13;
    Ctau3               -2;
    alphaKsi            0.9;
    Am                  0.016;
    Aepsilon            0.263;
    Amu                 0.00222;
}

LienLeschzinerLowReCoeffs
{
    Cmu                 0.09;
    C1                  1.44;
    C2                  1.92;
    alphak              1;
    alphaEps            0.76923;
    Am                  0.016;
    Aepsilon            0.263;
    Amu                 0.00222;
}
```

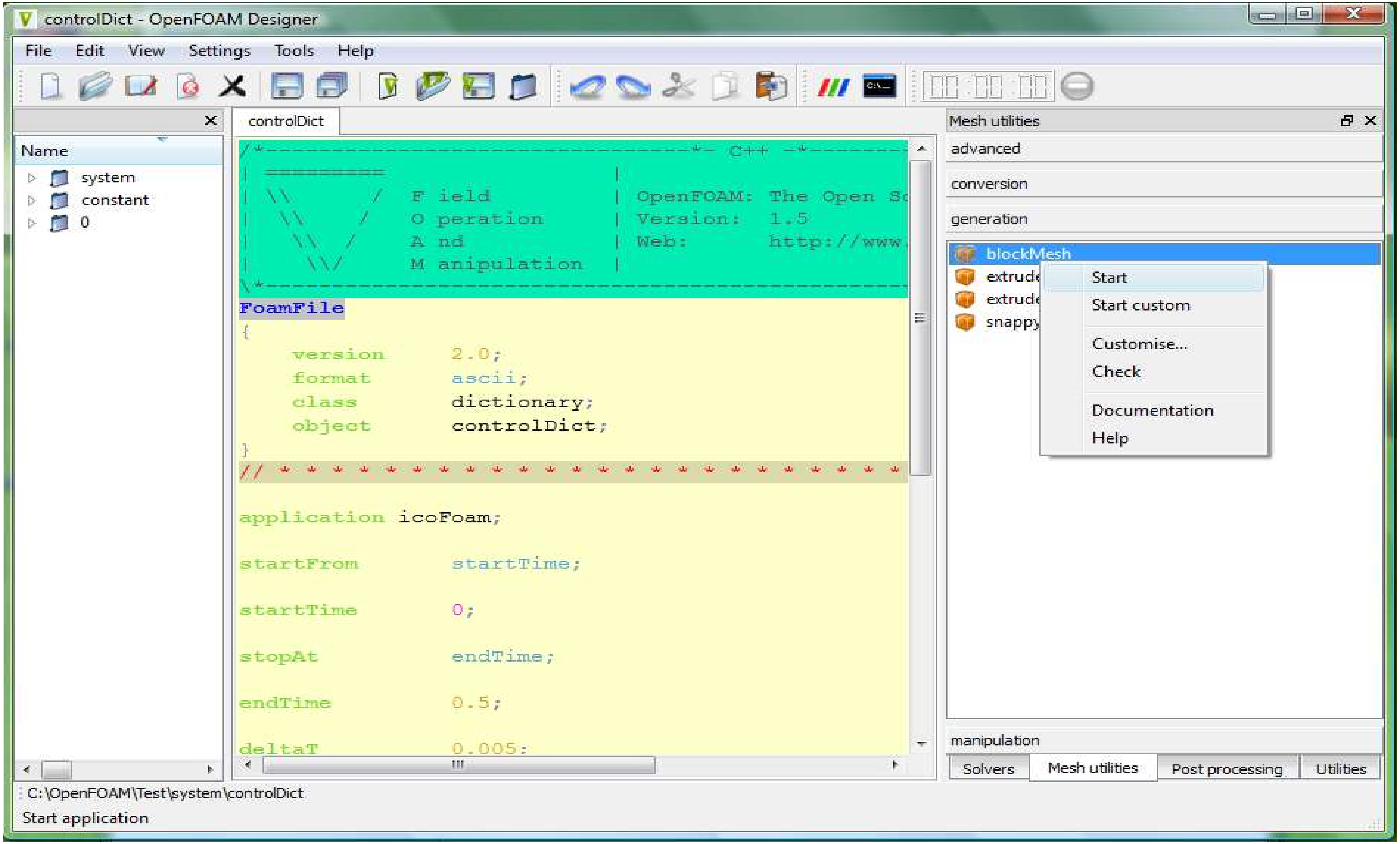
constant/ RASProperties file

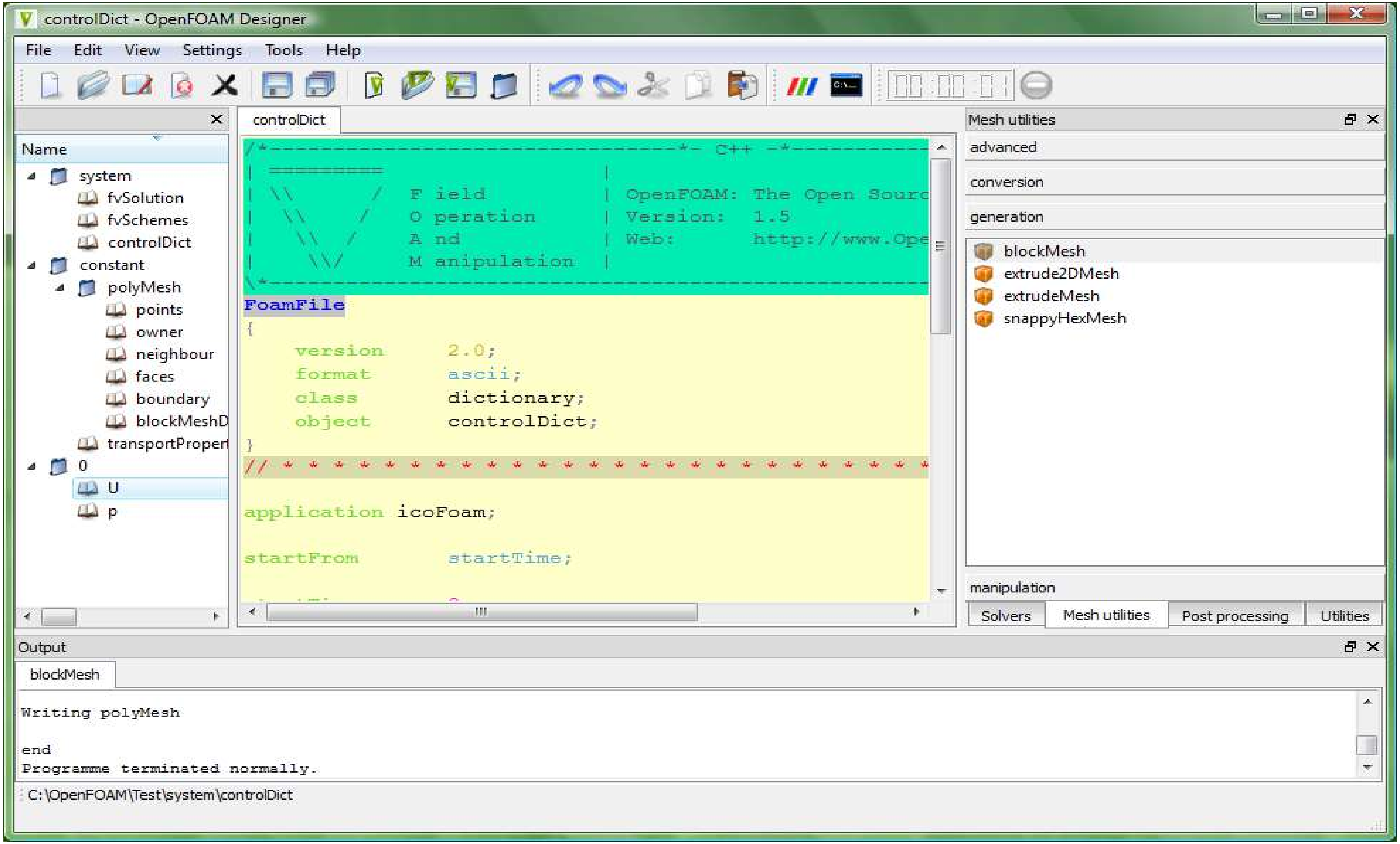
```
LRRCoeffs
{
    Cmu             0.09;
    Clrr1           1.8;
    Clrr2           0.6;
    C1              1.44;
    C2              1.92;
    Cs              0.25;
    Ceps            0.15;
    alphaEps        0.76923;
}

LaunderGibsonRSTMCoeffs
{
    Cmu             0.09;
    Clg1            1.8;
    Clg2            0.6;
    C1              1.44;
    C2              1.92;
    C1Ref           0.5;
    C2Ref           0.3;
    Cs              0.25;
    Ceps            0.15;
    alphaEps        0.76923;
    alphaR          1.22;
}

SpalartAllmarasCoeffs
{
    alphaNut        1.5;
    Cb1             0.1355;
    Cb2             0.622;
    Cw2             0.3;
    Cw3             2;
    Cv1             7.1;
    Cv2             5.0;
}

wallFunctionCoeffs
{
    kappa           0.4187;
    E               9;
}
```

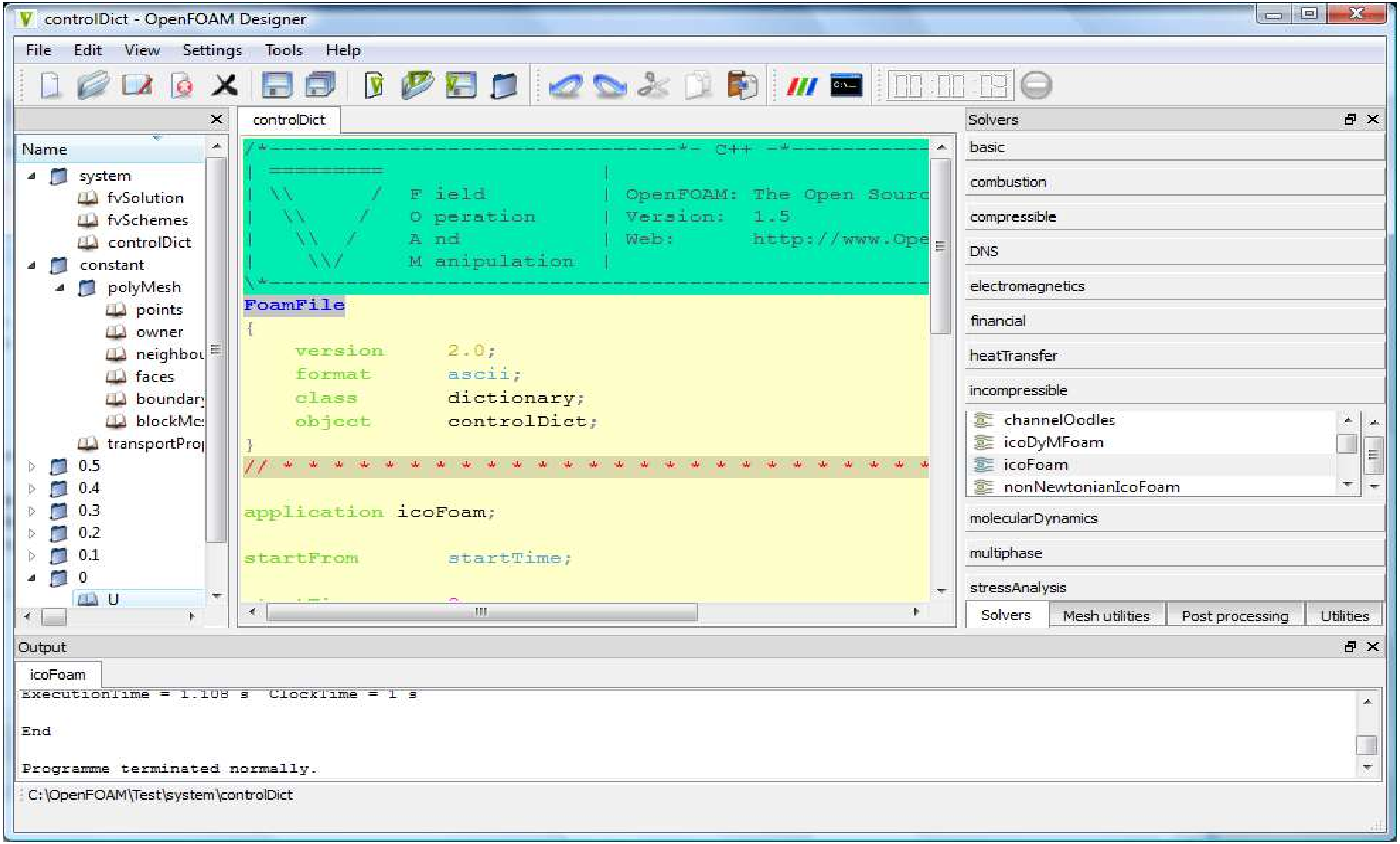


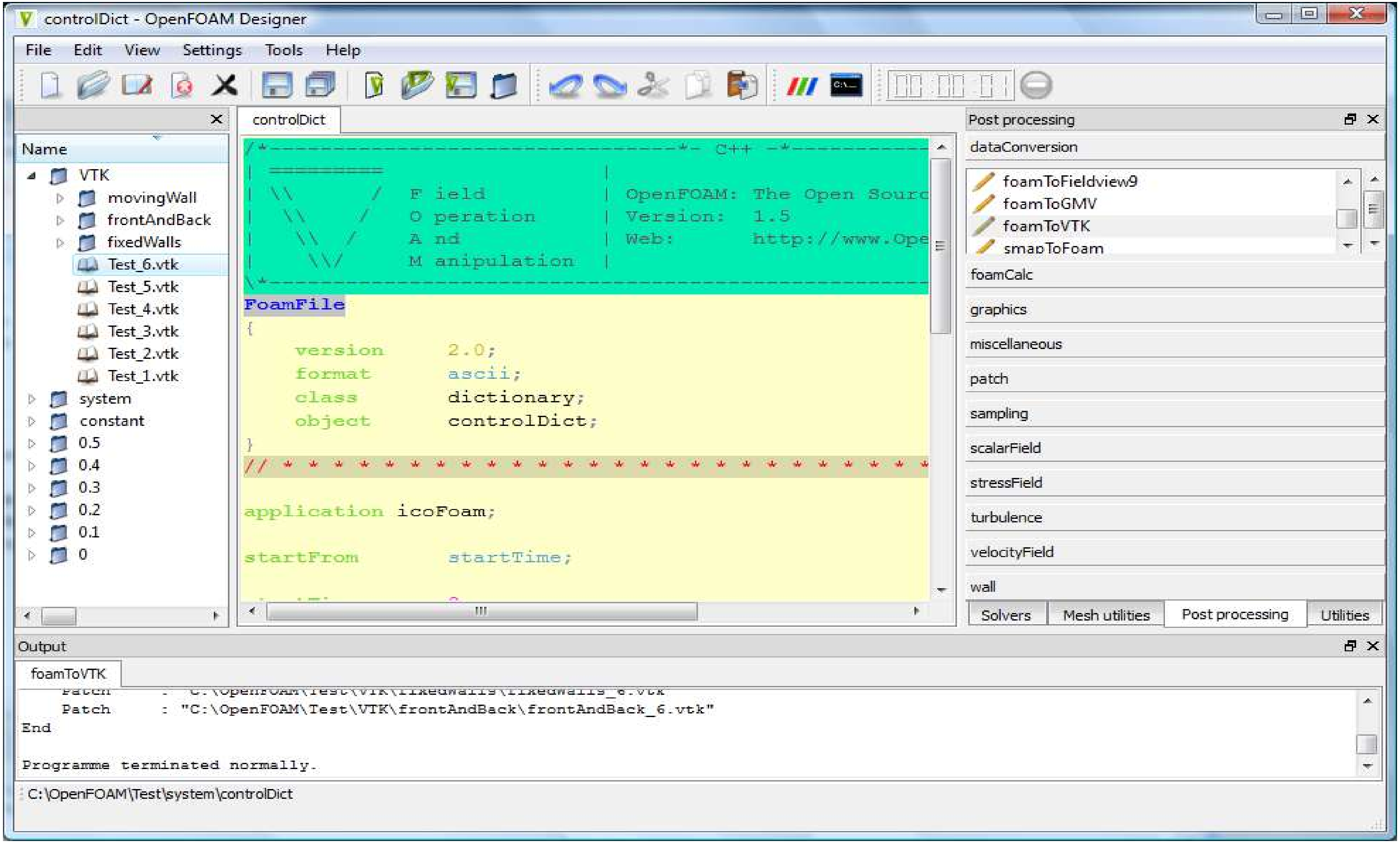
Running cavity template in icoFoam OpenFOAM designer version 1.0 on Windows Vista


controlDict - OpenFOAM Designer
File
Edit
View
Settings
Tools
Help
controlDict
Name
system
fvSolution
fvSchemes
controlDict
constant
polyMesh
blockMeshD
transportPropert
0
U
p
/*---------------------------------*- C++ -*-----------
| ========= |
| \\   / F ield | OpenFOAM: The Open Sourc
| \\   /   O peration | Version: 1.5
| \\ /   A nd | Web: http://www.Ope
| \\/   M anipulation |
*---
FoamFile
{
version 2.0;
format ascii;
class dictionary;
object controlDict;
}
// *
application icoFoam;
startFrom startTime;
startTime 0;
stopAt endTime;
endTime 0.5;
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
miscellaneous
parallelProcessing
preProcessing
surface
thermophysical
Solvers
Mesh utilities
Post processing
Utilities
C:\OpenFOAM\Test\system\controlDict

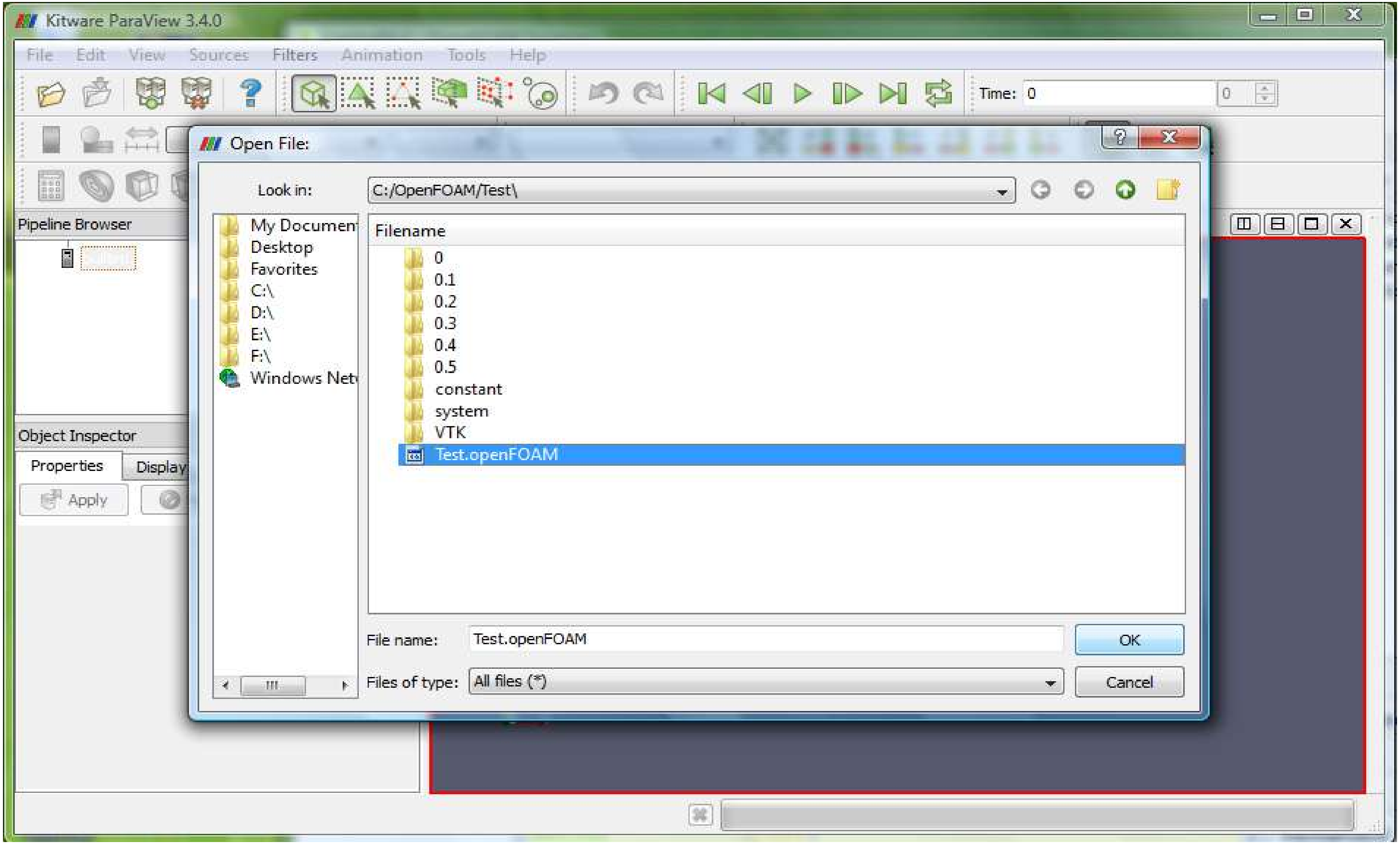

controlDict - OpenFOAM Designer
File Edit View Settings Tools Help
Name
system
constant
0
controlDict
/*---------------------------------*- C++ -*----------
| ========= |
| \\   / F ield | OpenFOAM: The Open So
| \\   /   O peration | Version: 1.5
| \\ /   A nd | Web: http://www.
| \\/   M anipulation |
*--
FoamFile
{
version 2.0;
format ascii;
class dictionary;
object controlDict;
}
// *
application icoFoam;
startFrom startTime;
startTime 0;
stopAt endTime;
endTime 0.5;
deltaT 0.005;
Mesh utilities
advanced
conversion
generation
blockMesh
extrude
extrude
snappy
Start
Start custom
Customise...
Check
Documentation
Help
manipulation
Solvers
Mesh utilities
Post processing
Utilities
C:\OpenFOAM\Test\system\controlDict
Start application

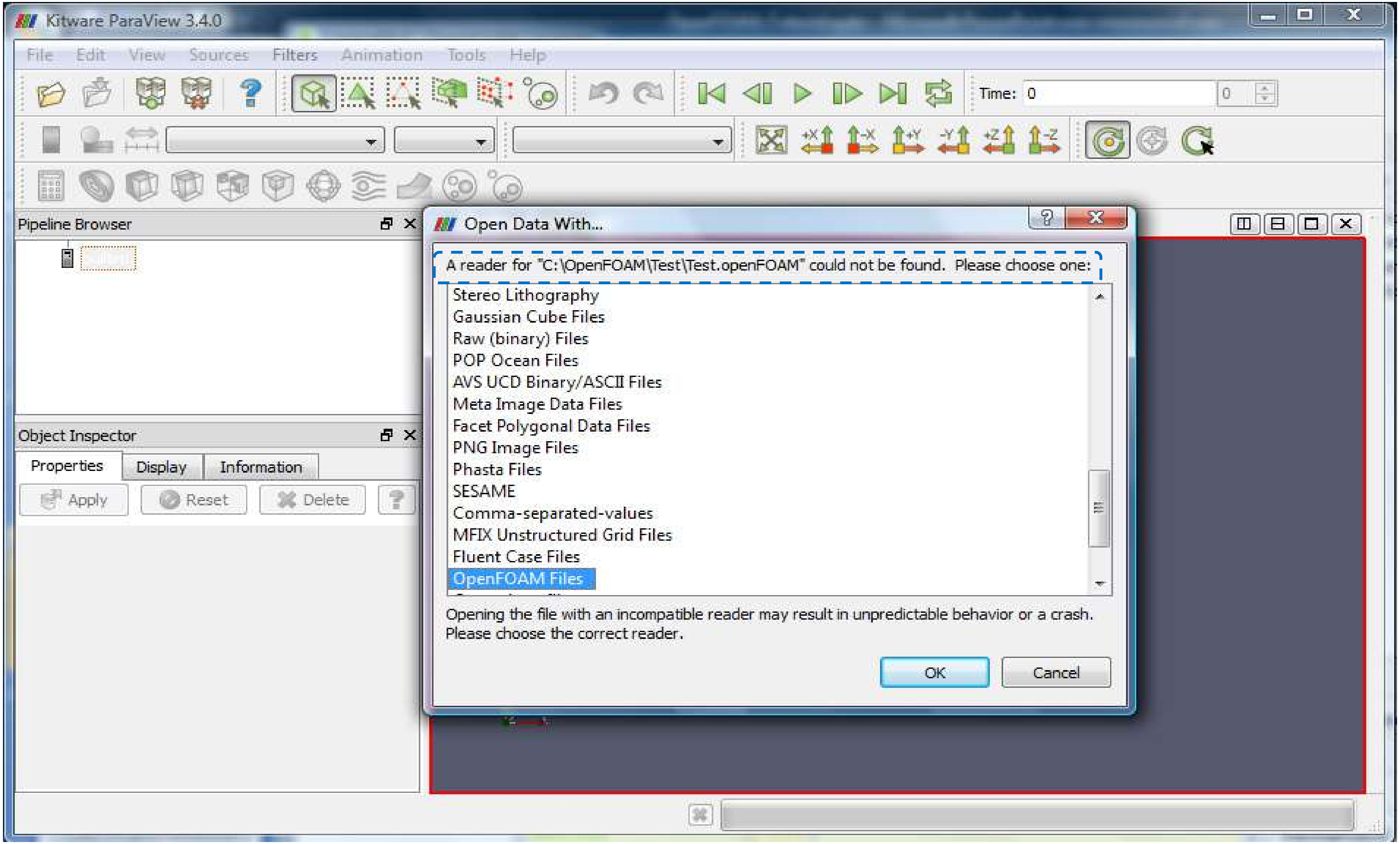

controlDict - OpenFOAM Designer
File
Edit
View
Settings
Tools
Help
Name
system
fvSolution
fvSchemes
controlDict
constant
polyMesh
points
owner
neighbour
faces
boundary
blockMeshD
transportPropert
0
U
p
controlDict
/*---------------------------------*- C++ -*-----------
| ========= |
| \\   / F ield | OpenFOAM: The Open Sourc
| \\   /   O peration | Version: 1.5
| \\ /   A nd | Web: http://www.Ope
| \\/   M anipulation |
*---
FoamFile
{
version   2.0;
format   ascii;
class   dictionary;
object   controlDict;
}
// *
application icoFoam;
startFrom   startTime;
Mesh utilities
advanced
conversion
generation
blockMesh
extrude2DMesh
extrudeMesh
snappyHexMesh
manipulation
Solvers
Mesh utilities
Post processing
Utilities
Output
blockMesh
Writing polyMesh
end
Programme terminated normally.
C:\OpenFOAM\Test\system\controlDict


controlDict - OpenFOAM Designer
File Edit View Settings Tools Help
Name
system
fvSolution
fvSchemes
controlDict
constant
polyMesh
points
owner
neighbou
faces
boundar
blockMe
transportPro
0.5
0.4
0.3
0.2
0.1
0
U
controlDict
F ield
O peration
A nd
M anipulation
OpenFOAM: The Open Sourc
Version: 1.5
Web: http://www.Ope
FoamFile
version 2.0;
format ascii;
class dictionary;
object controlDict;
application icoFoam;
startFrom startTime;
Solvers
basic
combustion
compressible
DNS
electromagnetics
financial
heatTransfer
incompressible
channelOodles
icoDyMFoam
icoFoam
nonNewtonianIcoFoam
molecularDynamics
multiphase
stressAnalysis
Solvers
Mesh utilities
Post processing
Utilities
Output
icoFoam
End
Programme terminated normally.
C:\OpenFOAM\Test\system\controlDict

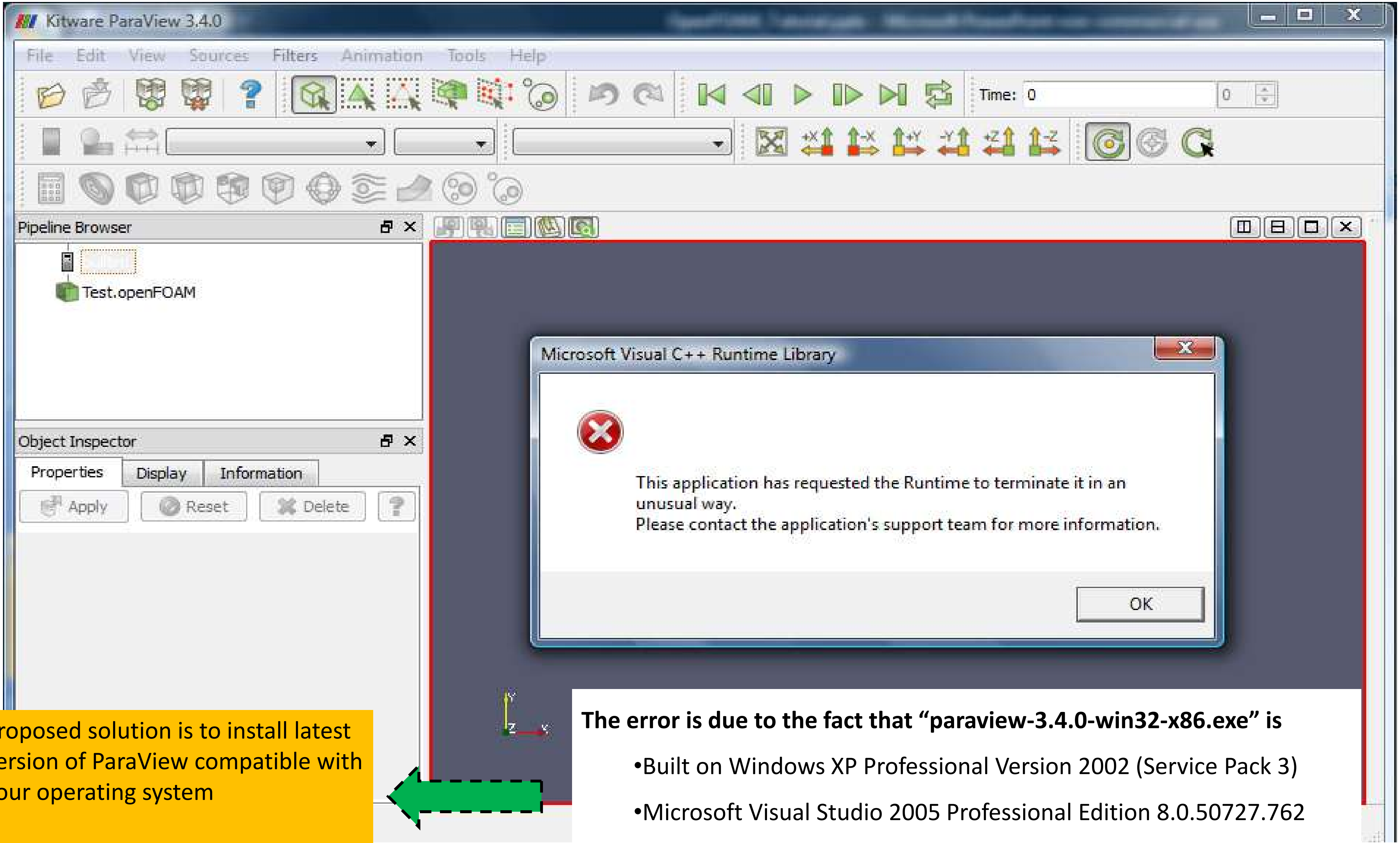

controlDict - OpenFOAM Designer
File Edit View Settings Tools Help
Name
VTK
movingWall
frontAndBack
fixedWalls
Test_6.vtk
Test_5.vtk
Test_4.vtk
Test_3.vtk
Test_2.vtk
Test_1.vtk
system
constant
0.5
0.4
0.3
0.2
0.1
0
controlDict
/*---------------------------------*- C++ -*-----------
| ========= |
| \\   / F ield | OpenFOAM: The Open Sourc
| \\   /   O peration | Version: 1.5
| \\ /   A nd | Web: http://www.Ope
| \\/   M anipulation |
*---
FoamFile
{
    version   2.0;
    format   ascii;
    class   dictionary;
    object   controlDict;
}
// *
application icoFoam;
startFrom   startTime;
Post processing
dataConversion
foamToFieldview9
foamToGMV
foamToVTK
smapToFoam
foamCalc
graphics
miscellaneous
patch
sampling
scalarField
stressField
turbulence
velocityField
wall
Solvers
Mesh utilities
Post processing
Utilities
Output
foamToVTK
Patch   : "C:\OpenFOAM\Test\VTK\frontAndBack\frontAndBack_6.vtk"
End
Programme terminated normally.
C:\OpenFOAM\Test\system\controlDict

Post-processing in ParaView 3.4.0 available in OpenFOAM designer version 1.0 in Windows Vista


Kitware ParaView 3.4.0
File Edit View Sources Filters Animation Tools Help
Time: 0
Pipeline Browser
Object Inspector
Properties
Display
Apply
Open File:
Look in:
C:/OpenFOAM/Test\
My Documen
Desktop
Favorites
C:\
D:\
E:\
F:\
Windows Net
Filename
0
0.1
0.2
0.3
0.4
0.5
constant
system
VTK
Test.openFOAM
File name:
Test.openFOAM
OK
Files of type:
All files (*)
Cancel


Kitware ParaView 3.4.0
File
Edit
View
Sources
Filters
Animation
Tools
Help
Time: 0
Pipeline Browser
Object Inspector
Properties
Display
Information
Apply
Reset
Delete
Open Data With...
A reader for "C:\OpenFOAM\Test\Test.openFOAM" could not be found. Please choose one:
Stereo Lithography
Gaussian Cube Files
Raw (binary) Files
POP Ocean Files
AVS UCD Binary/ASCII Files
Meta Image Data Files
Facet Polygonal Data Files
PNG Image Files
Phasta Files
SESAME
Comma-separated-values
MFIX Unstructured Grid Files
Fluent Case Files
OpenFOAM Files
Opening the file with an incompatible reader may result in unpredictable behavior or a crash.
Please choose the correct reader.
OK
Cancel

Issues observed in OpenFOAM designer version 1.0 and proposed solution for Windows Vista

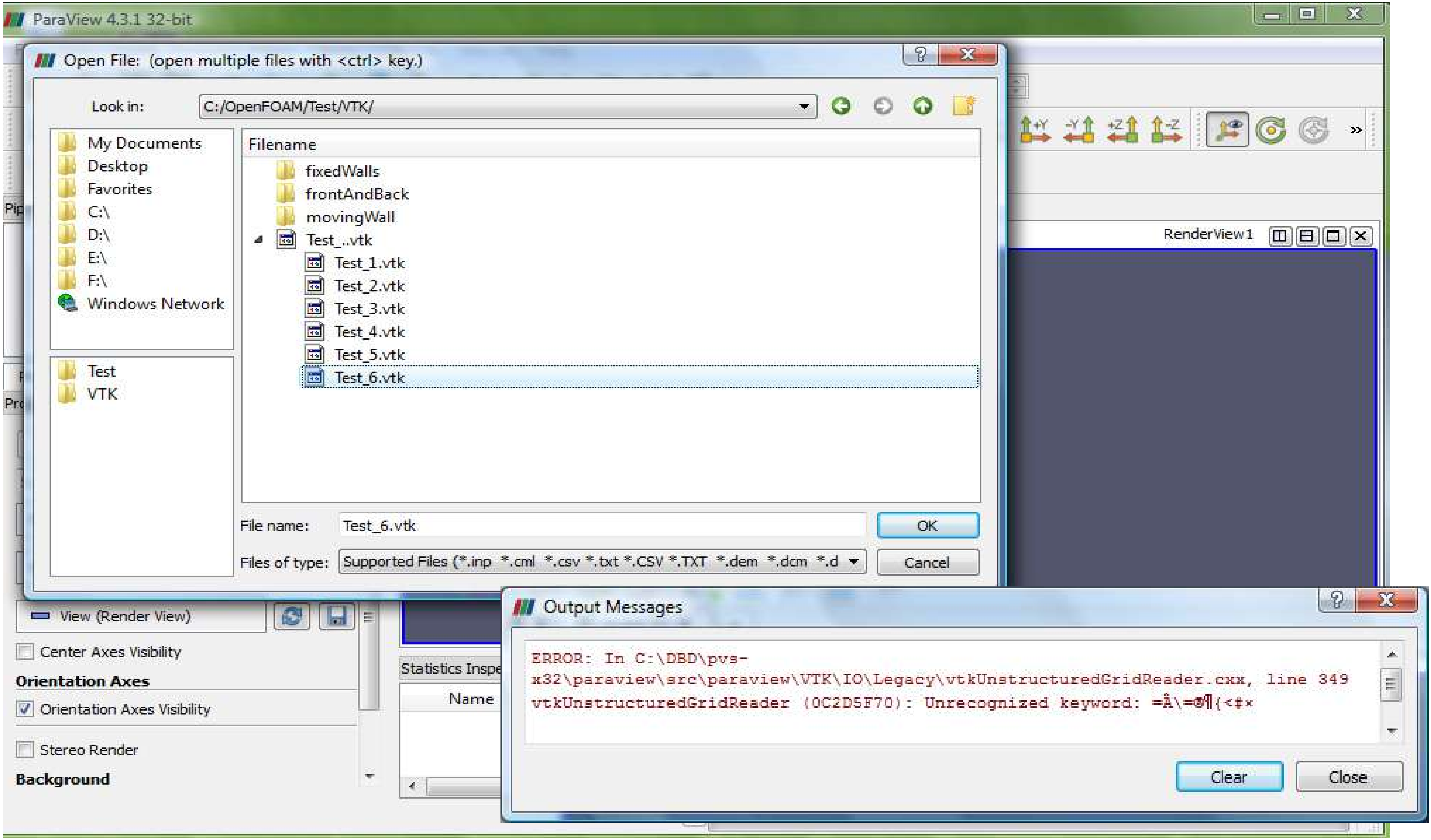


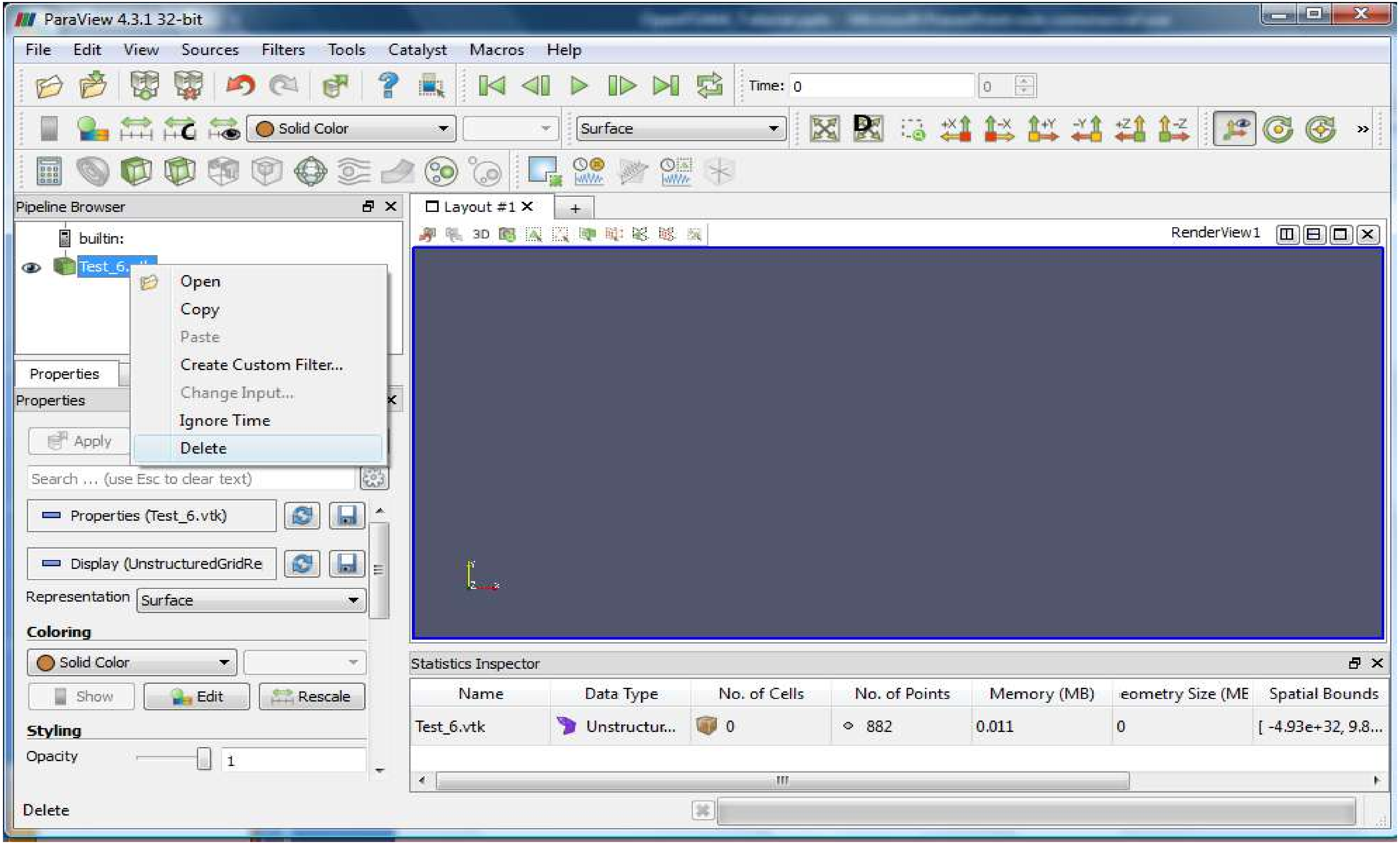
Proposed solution is to install latest version of ParaView compatible with your operating system

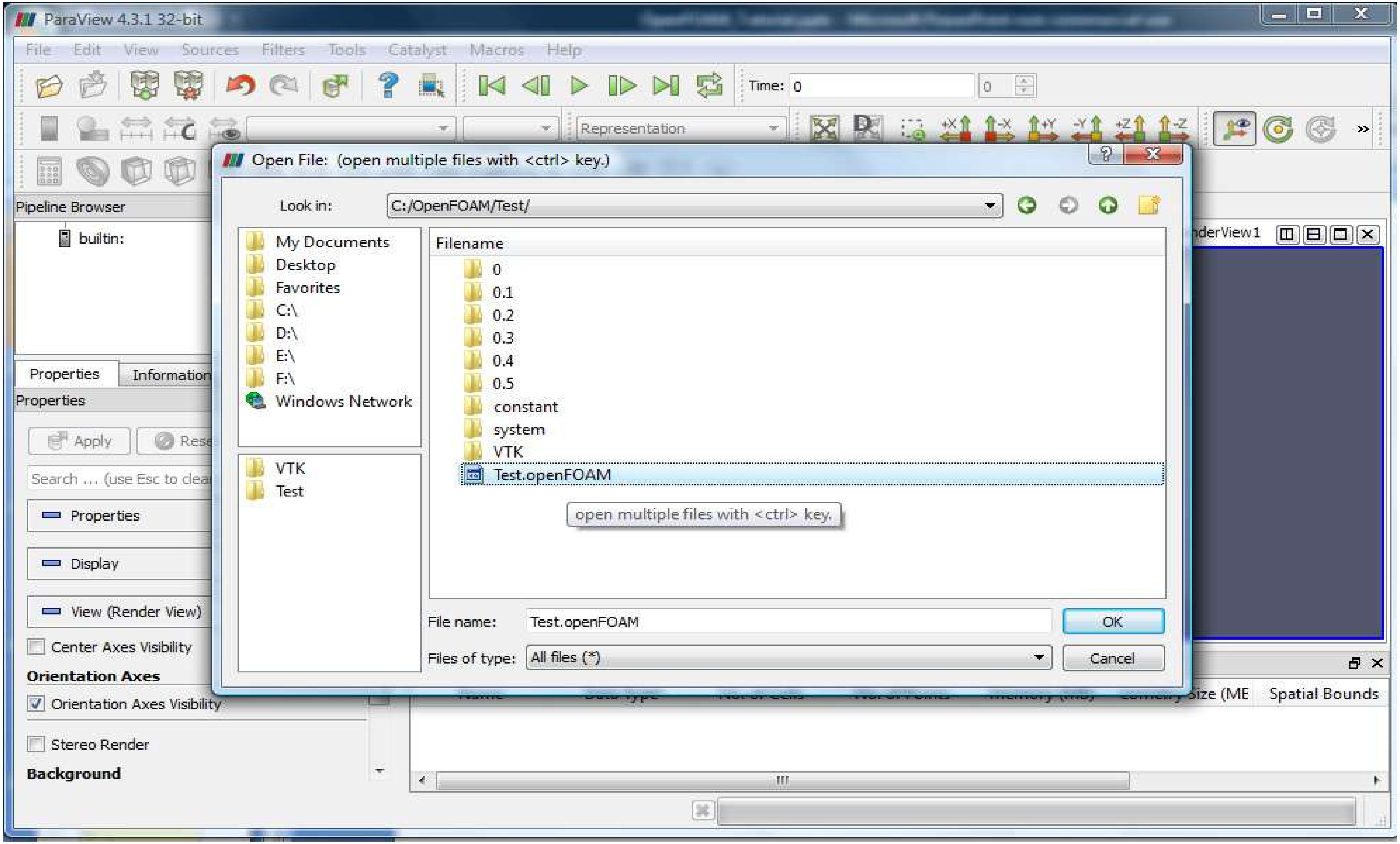
The error is due to the fact that "paraview-3.4.0-win32-x86.exe" is

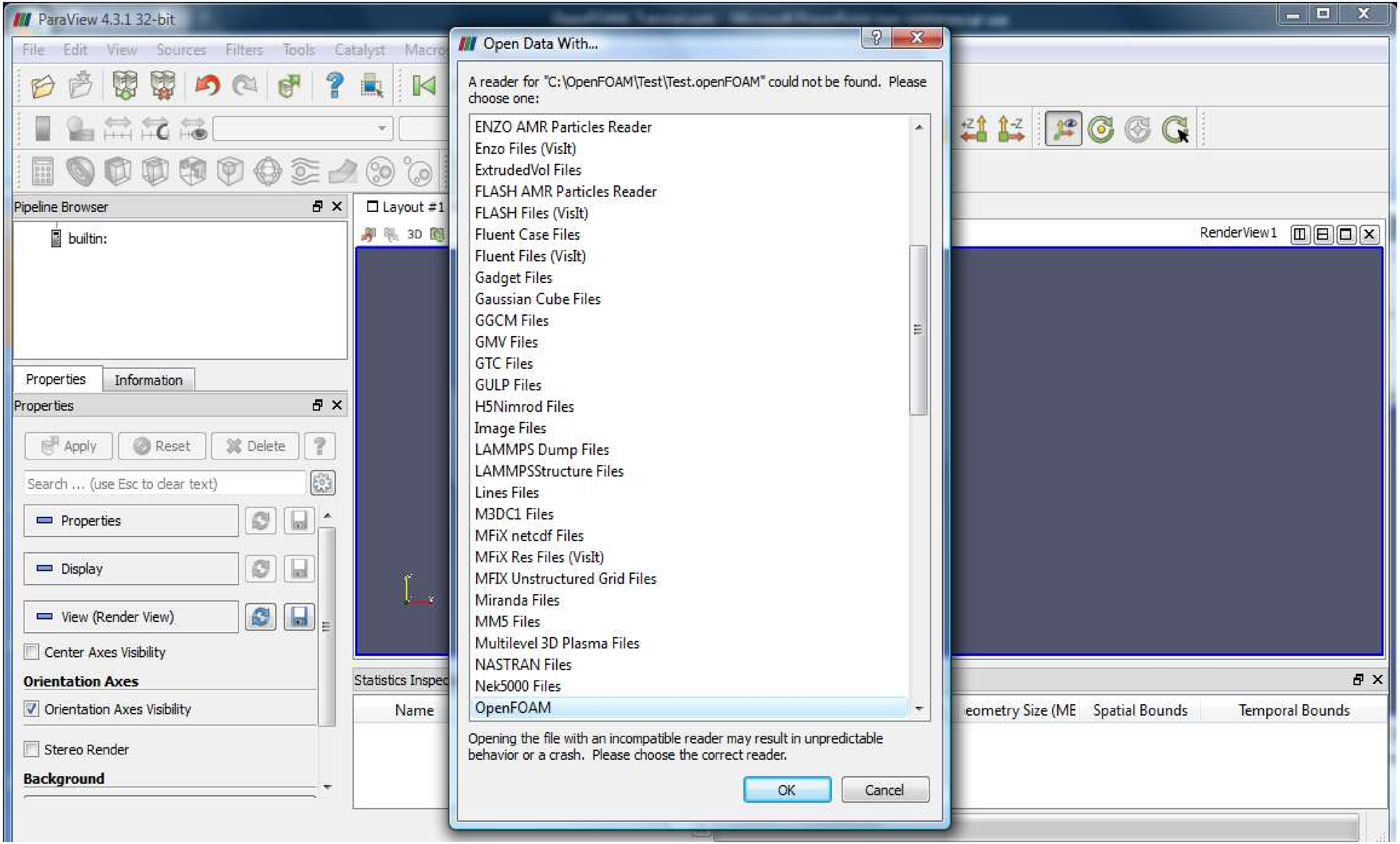
- Built on Windows XP Professional Version 2002 (Service Pack 3)
- Microsoft Visual Studio 2005 Professional Edition 8.0.50727.762

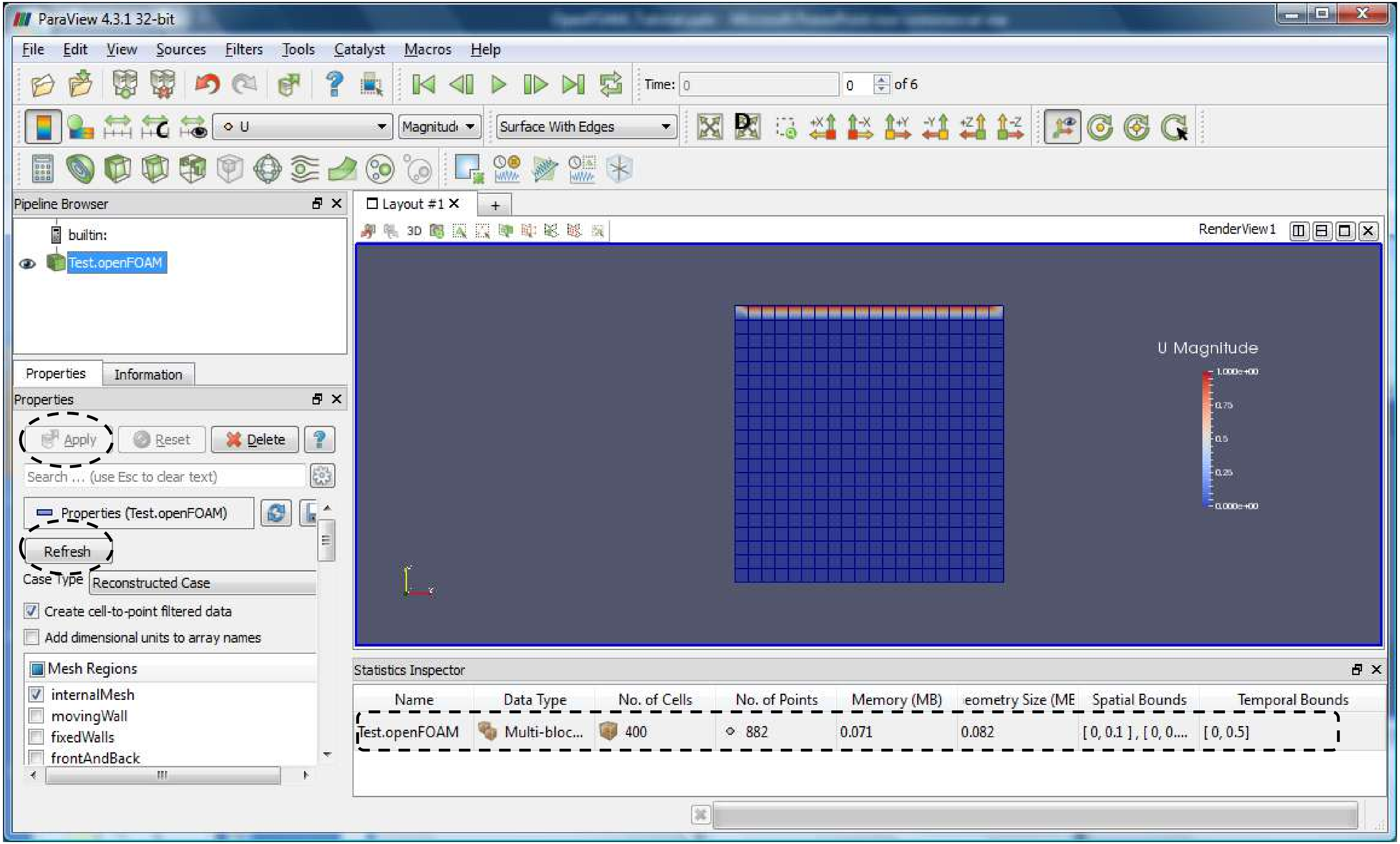
And hence will not have forward compatibility with Windows Vista.


ParaView 4.3.1 32-bit
Open File: (open multiple files with <ctrl> key.)
Look in:
C:/OpenFOAM/Test/VTK/
My Documents
Desktop
Favorites
C:\
D:\
E:\
F:\
Windows Network
Test
VTK
Filename
fixedWalls
frontAndBack
movingWall
Test_..vtk
Test_1.vtk
Test_2.vtk
Test_3.vtk
Test_4.vtk
Test_5.vtk
Test_6.vtk
File name:
Test_6.vtk
OK
Files of type:
Supported Files (*.inp *.cml *.csv *.txt *.CSV *.TXT *.dem *.dcm *.d
Cancel
RenderView1
View (Render View)
Center Axes Visibility
Orientation Axes
Orientation Axes Visibility
Stereo Render
Background
Statistics Inspe
Name
Output Messages
ERROR: In C:\DBD\pvs-x32\paraview\src\paraview\VTK\IO\Legacy\vtkUnstructuredGridReader.cxx, line 349
vtkUnstructuredGridReader (0C2D5F70): Unrecognized keyword:
Clear
Close

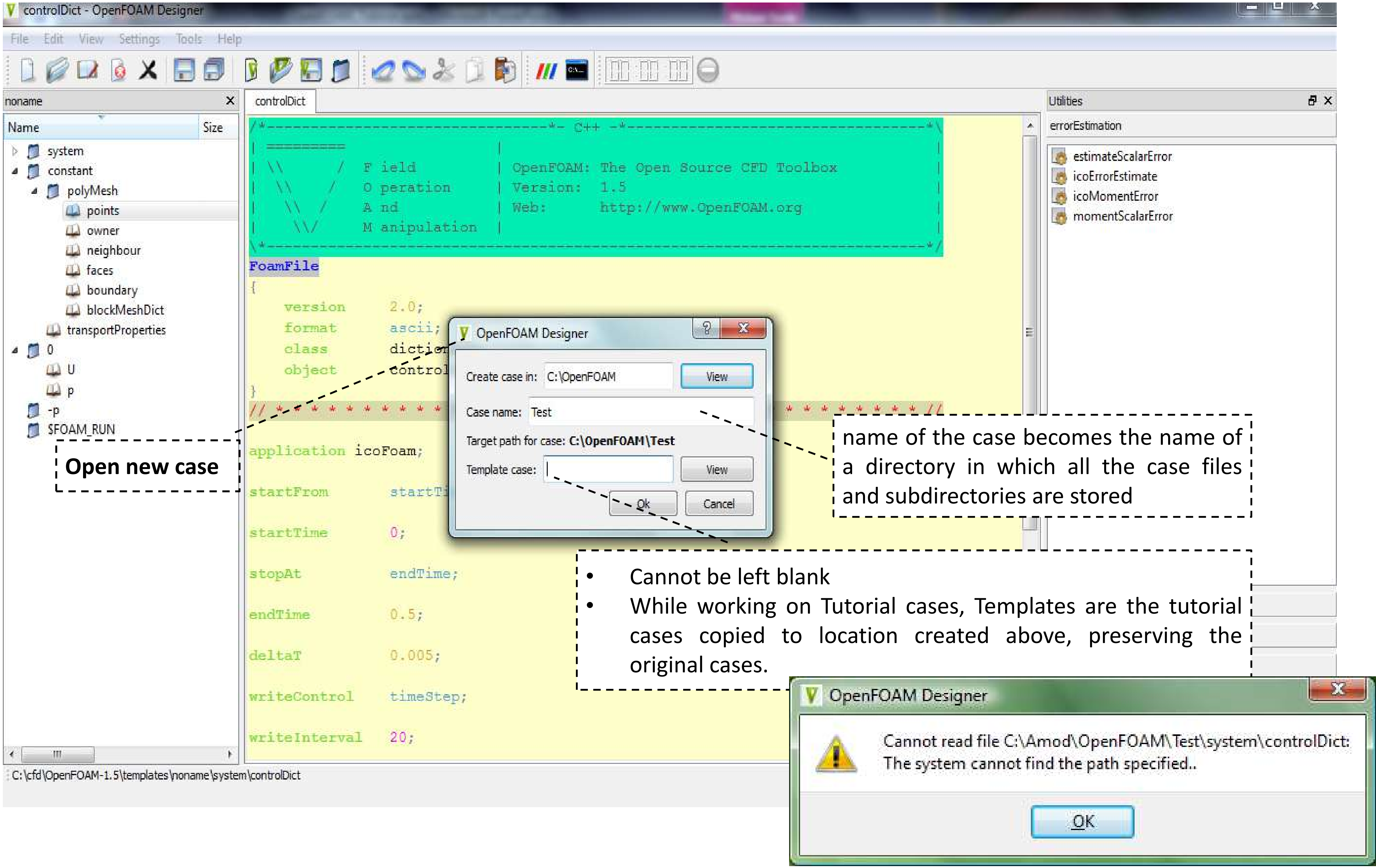

ParaView 4.3.1 32-bit
File Edit View Sources Filters Tools Catalyst Macros Help
Time: 0
Solid Color
Surface
Pipeline Browser
builtin:
Test_6.
Open
Copy
Paste
Create Custom Filter...
Change Input...
Ignore Time
Delete
Properties
Apply
Search ... (use Esc to clear text)
Properties (Test_6.vtk)
Display (UnstructuredGridRe
Representation
Surface
Coloring
Solid Color
Show
Edit
Rescale
Styling
Opacity
1
Layout #1
RenderView1
3D
Statistics Inspector
Name
Data Type
No. of Cells
No. of Points
Memory (MB)
eometry Size (ME
Spatial Bounds
Test_6.vtk
Unstructur...
0
882
0.011
0
[-4.93e+32, 9.8...
Delete

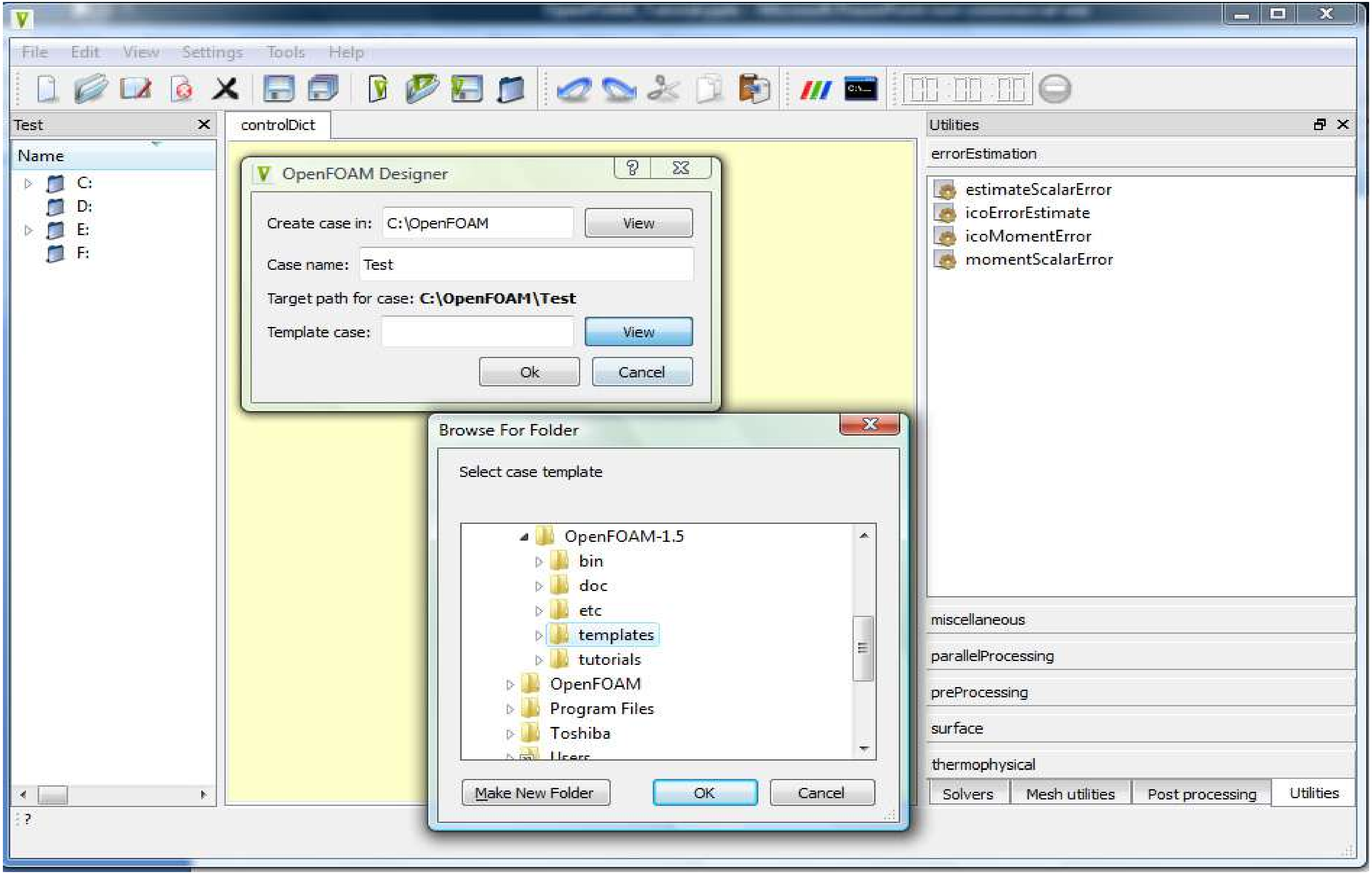

ParaView 4.3.1 32-bit
File
Edit
View
Sources
Filters
Tools
Catalyst
Macros
Help
Time: 0
Representation
Pipeline Browser
builtin:
Properties
Information
Apply
Search ... (use Esc to clea
Display
View (Render View)
Center Axes Visibility
Orientation Axes
Orientation Axes Visibility
Stereo Render
Background
Open File: (open multiple files with <ctrl> key.)
Look in:
C:/OpenFOAM/Test/
My Documents
Desktop
Favorites
C:\
D:\
E:\
F:\
Windows Network
VTK
Test
Filename
0
0.1
0.2
0.3
0.4
0.5
constant
system
VTK
Test.openFOAM
open multiple files with <ctrl> key.
File name:
Test.openFOAM
Files of type:
All files (*)
OK
Cancel
Spatial Bounds

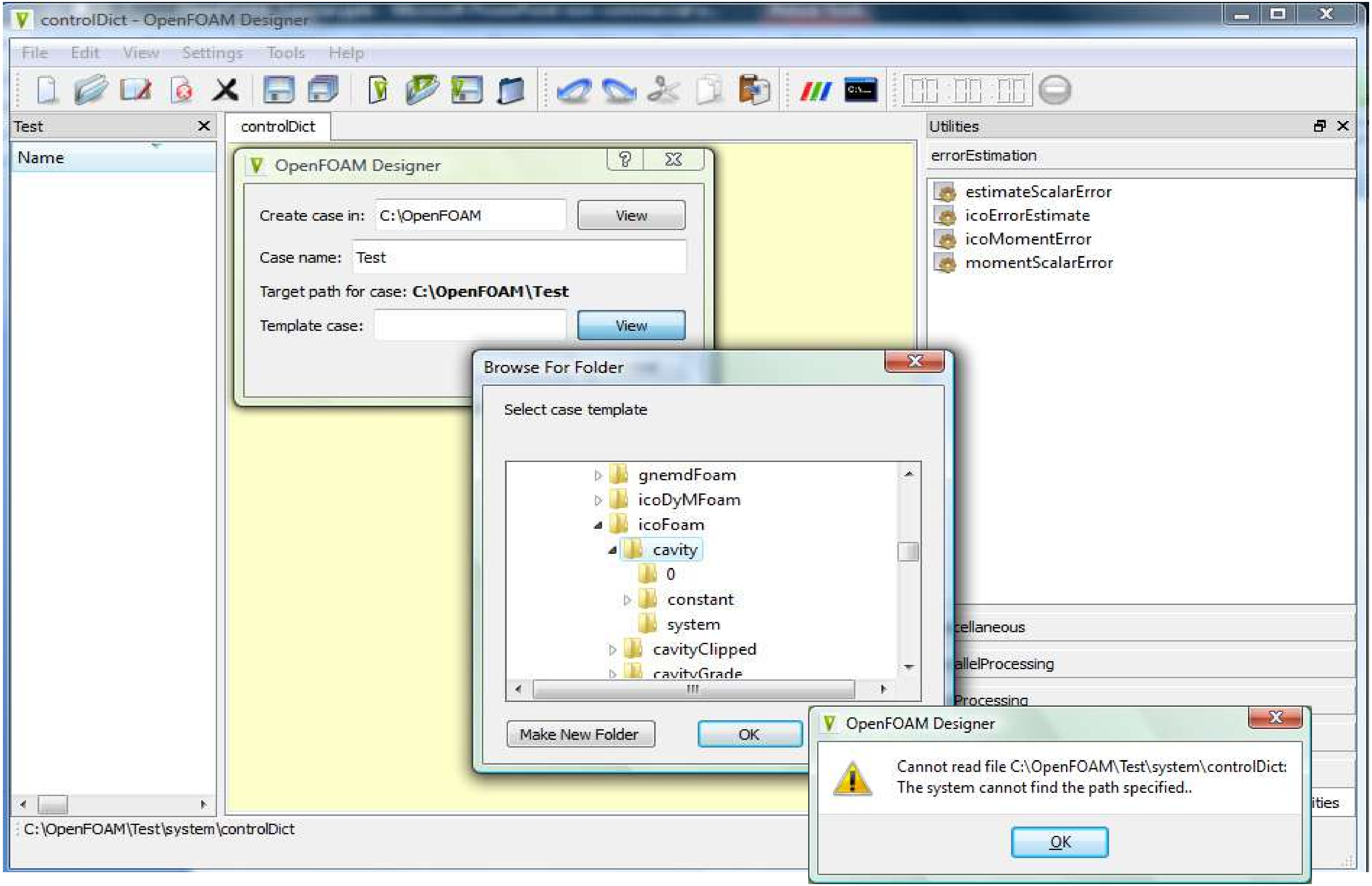

ParaView 4.3.1 32-bit
File Edit View Sources Filters Tools Catalyst Macros
Pipeline Browser
builtin:
Properties Information
Properties
Apply Reset Delete
Search ... (use Esc to clear text)
Properties
Display
View (Render View)
Center Axes Visibility
Orientation Axes
Orientation Axes Visibility
Stereo Render
Background
Layout #1
RenderView1
Statistics Inspector
Name
eometry Size (ME
Spatial Bounds
Temporal Bounds
Open Data With...
A reader for "C:\OpenFOAM\Test\Test.openFOAM" could not be found. Please choose one:
ENZO AMR Particles Reader
Enzo Files (VisIt)
ExtrudedVol Files
FLASH AMR Particles Reader
FLASH Files (VisIt)
Fluent Case Files
Fluent Files (VisIt)
Gadget Files
Gaussian Cube Files
GGCM Files
GMV Files
GTC Files
GULP Files
H5Nimrod Files
Image Files
LAMMPS Dump Files
LAMMPSStructure Files
Lines Files
M3DC1 Files
MFiX netcdf Files
MFiX Res Files (VisIt)
MFIX Unstructured Grid Files
Miranda Files
MM5 Files
Multilevel 3D Plasma Files
NASTRAN Files
Nek5000 Files
OpenFOAM
Opening the file with an incompatible reader may result in unpredictable behavior or a crash. Please choose the correct reader.
OK
Cancel


ParaView 4.3.1 32-bit
File Edit View Sources Filters Tools Catalyst Macros Help
Time: 0
0 of 6
U
Magnitude
Surface With Edges
Pipeline Browser
builtin:
Test.openFOAM
Layout #1
RenderView1
U Magnitude
1.000e+00
0.75
0.5
0.25
0.000e+00
Properties
Information
Apply
Reset
Delete
Search ... (use Esc to clear text)
Properties (Test.openFOAM)
Refresh
Case Type
Reconstructed Case
Create cell-to-point filtered data
Add dimensional units to array names
Mesh Regions
internalMesh
movingWall
fixedWalls
frontAndBack
Statistics Inspector
Name
Data Type
No. of Cells
No. of Points
Memory (MB)
eometry Size (ME
Spatial Bounds
Temporal Bounds
Test.openFOAM
Multi-bloc...
400
882
0.071
0.082
[0, 0.1], [0, 0....
[0, 0.5]

Issues observed in OpenFOAM designer version 1.0 but **no** solutions yet for Windows Vista


controlDict - OpenFOAM Designer
File Edit View Settings Tools Help
noname
Name Size
system
constant
polyMesh
points
owner
neighbour
faces
boundary
blockMeshDict
transportProperties
0
U
p
-p
$FOAM_RUN
controlDict
OpenFOAM: The Open Source CFD Toolbox
Version: 1.5
Web: http://www.OpenFOAM.org
FoamFile
version 2.0;
format ascii;
application icoFoam;
startFrom
startTime 0;
stopAt endTime;
endTime 0.5;
deltaT 0.005;
writeControl timeStep;
writeInterval 20;
C:\cfd\OpenFOAM-1.5\templates\noname\system\controlDict
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
OpenFOAM Designer
Create case in: C:\OpenFOAM
View
Case name: Test
Target path for case: C:\OpenFOAM\Test
Template case:
View
Ok
Cancel
Open new case
name of the case becomes the name of a directory in which all the case files and subdirectories are stored
• Cannot be left blank
• While working on Tutorial cases, Templates are the tutorial cases copied to location created above, preserving the original cases.
OpenFOAM Designer
Cannot read file C:\Amod\OpenFOAM\Test\system\controlDict:
The system cannot find the path specified..
OK


File Edit View Settings Tools Help
Test
Name
C:
D:
E:
F:
controlDict
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
miscellaneous
parallelProcessing
preProcessing
surface
thermophysical
Solvers
Mesh utilities
Post processing
Utilities
OpenFOAM Designer
Create case in:
C:\OpenFOAM
View
Case name:
Test
Target path for case: C:\OpenFOAM\Test
Template case:
View
Ok
Cancel
Browse For Folder
Select case template
OpenFOAM-1.5
bin
doc
etc
templates
tutorials
OpenFOAM
Program Files
Toshiba
Make New Folder
OK
Cancel


controlDict - OpenFOAM Designer
File Edit View Settings Tools Help
Test
Name
controlDict
Utilities
errorEstimation
estimateScalarError
icoErrorEstimate
icoMomentError
momentScalarError
OpenFOAM Designer
Create case in: C:\OpenFOAM
View
Case name: Test
Target path for case: C:\OpenFOAM\Test
Template case:
View
Browse For Folder
Select case template
gnemdFoam
icoDyMFoam
icoFoam
cavity
0
constant
system
cavityClipped
cavityGrade
Make New Folder
OK
OpenFOAM Designer
Cannot read file C:\OpenFOAM\Test\system\controlDict:
The system cannot find the path specified..
OK
C:\OpenFOAM\Test\system\controlDict