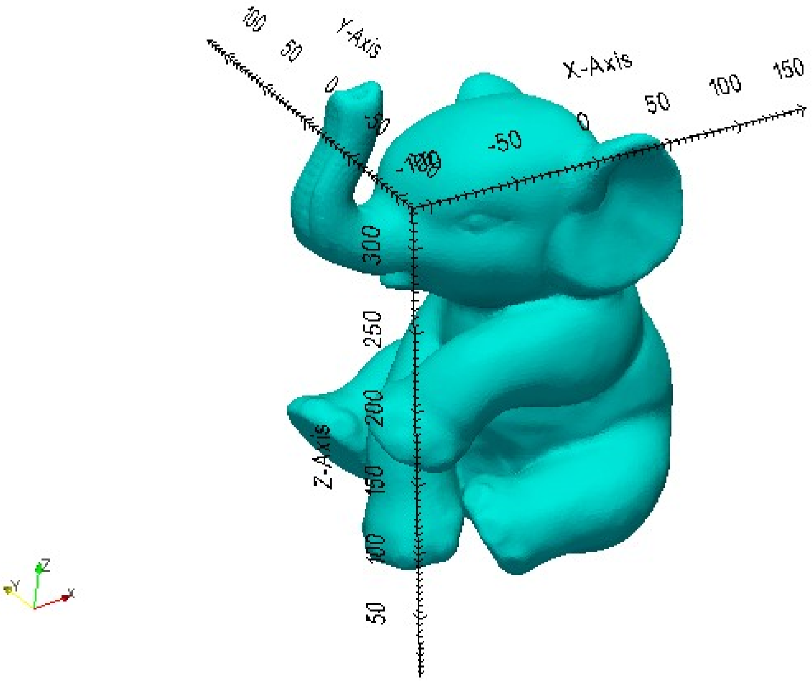
Project Codeword	Document No.	Editors	Date	Revision
Alamshar	201501161836	hk	2015-01-16	00

This document illustrates the use of the Insight Case Builder tool for quickly creating OpenFOAM analysis cases on an example.

The aim is to calculate the external flow around a solid object. The mesh is created using the trimmed mesher snappyHexMesh. Solution of the flow equations is done by the steady, incompressible, single-phase flow solver simpleFoam.

The object is shown below:



Its bounding box has dimensions LxWxH = 263 x 241 x 322 mm.

The object shall now be placed in a rectangular domain of size LxWxH = 2 x 2 x 2 m, sitting on the bottom. The bottom surface will be a wall, the patch at x_{min} will be the inlet with a specified velocity and the patch at x_{max} shall be the outlet with specified pressure. All remaining patches shall be symmetry planes.

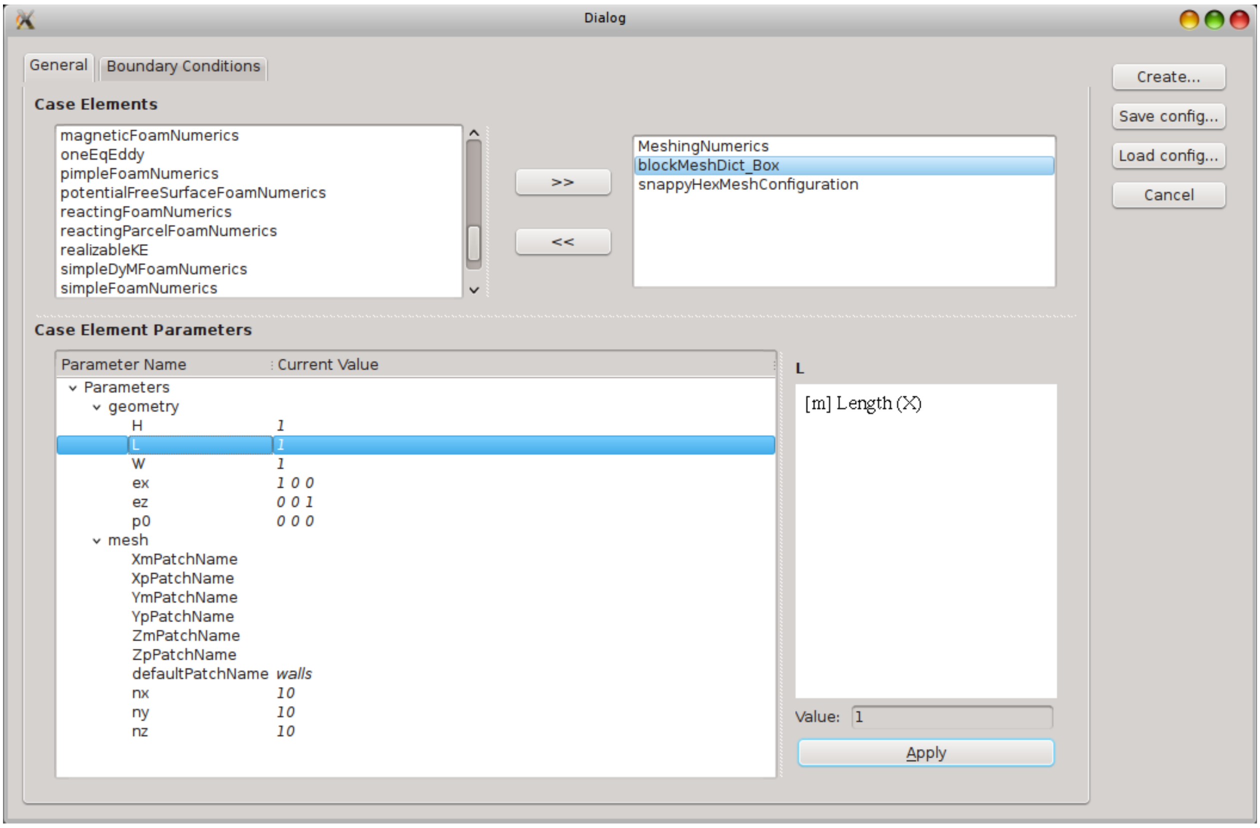
The case will be set up in two steps. First, a case for meshing the geometry with snappyHexMesh will be created. Afterwards, a separate case for running the solver will be created in another directory with the mesh copied from the former case.

1 Mesh Creation Case

Create a new directory for the case. Make it the working directory and start the case builder program:

```
$ cd casedir
$ isofCaseBuilder
```

The window looks like this:



Execute the following steps:

1. Add the following case elements to the case by selecting them in the list "Case Elements" and clicking the button ">>":

   (a) MeshingNumerics
   This will populate a minimum configuration in the system/ directory (controlDict, empty fvSchemes and fvSolutions file).

(b) blockMeshDict_Box
Adds a blockMeshDict for a plain rectangular mesh. This is intended for the template mesh of snappyHexMesh

(c) snappyHexMeshConfiguration
This case element creates a snappyHexMeshDict for OpenFOAM's trimmed mesher.

2. Change the configuration of the case elements. The parameter editor appears below the case element list after selecting an element in the case content list.

   (a) blockMeshDict_Box (template mesh)

      i. geometry/H: Height of the domain: 2
      ii. geometry/L: Length of the domain: 2
      iii. geometry/W: Width of the domain: 2
      iv. geometry/p0: Location of minimum corner of the domain: -1 -1 0
      v. mesh/XmPatchName: inlet
      vi. mesh/XpPatchName: outlet
      vii. mesh/ZmPatchName: ground
      viii. mesh/defaultPatchName: sides

   (b) snappyHexMeshConfiguration (trimmed mesher setup)

      i. PiM/[0]: -0.9 0 0.001
      (A point slightly downstream of the inlet)
      ii. features: click on "+ Add new", then change feature/[0] to "Geometry"
      iii. features/[0]/fileName: select "Elephant.stlb"
      iv. features/[0]/name: select "elephant"
      v. features/[0]/scale: 1e-3 1e-3 1e-3
      (STL is in mm, case shall be in meters => scale the geometry)

3. Save the case configuration to a file for later reuse: click "Save config..."

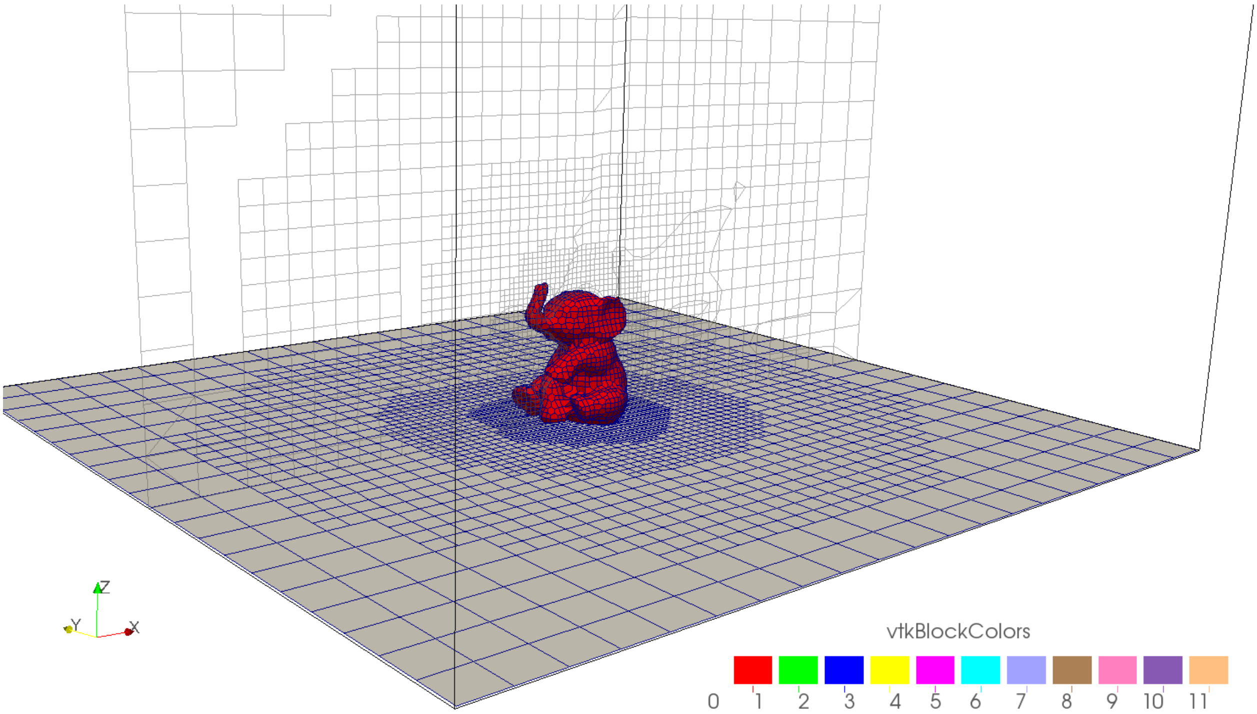
4. Finally create the OpenFOAM case on disk: click "Create..."
The dictionaries will be written to current directory and the geometry file will be scaled and copied into the "constant/triSurface" directory.

Execute the meshing process:

```
$ blockMesh
$ snappyHexMesh
```

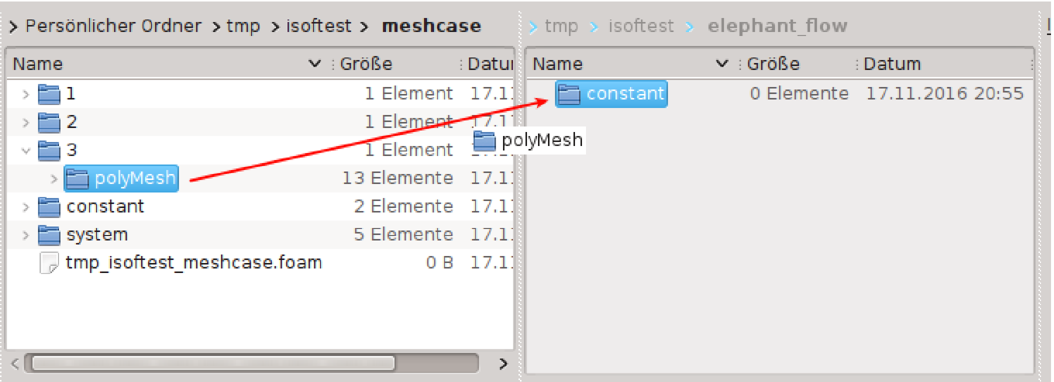
After snappyHexMesh has finished, the result can be inspected in ParaView:

```
$ isPV.py
```



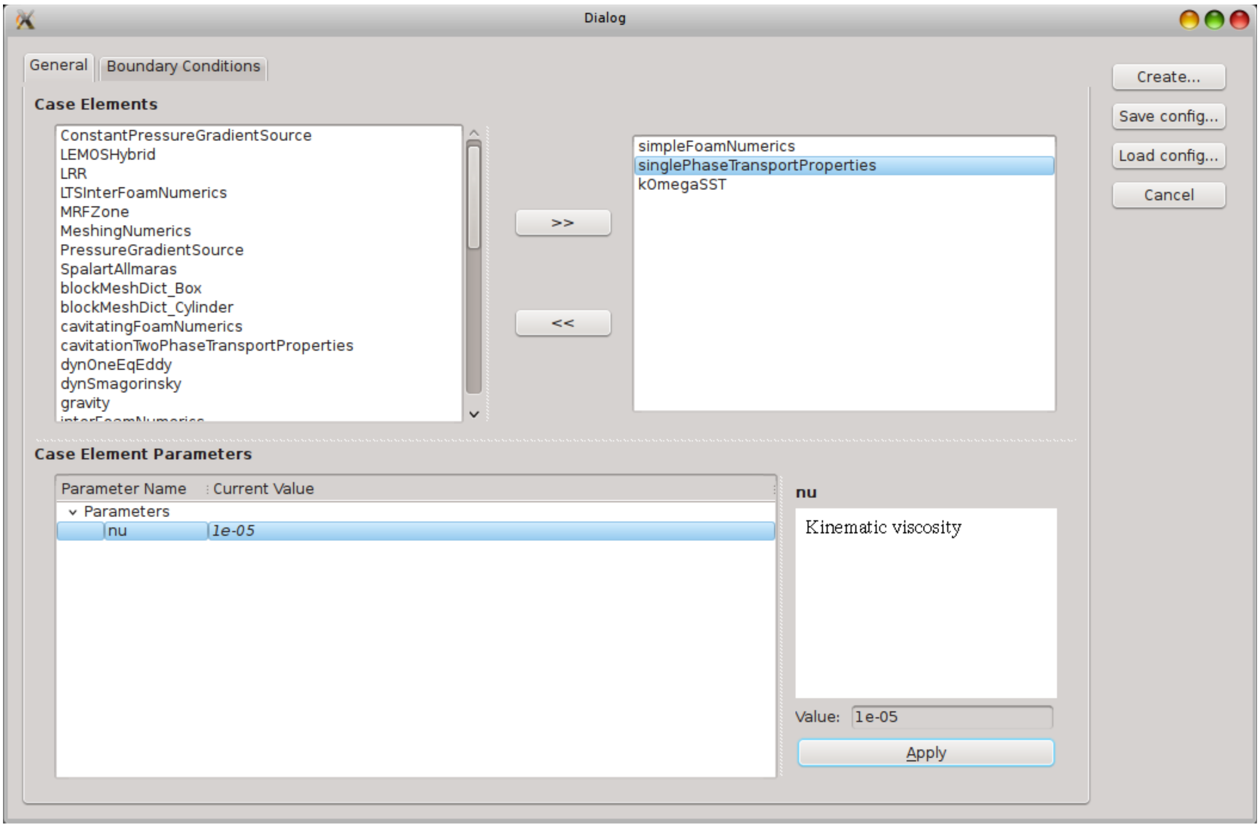
2 Solution Case

Create another empty directory for the solution case. In this directory, create a folder named "constant". From the mesh case, copy the folder "3/polyMesh" into this newly created "constant" folder:



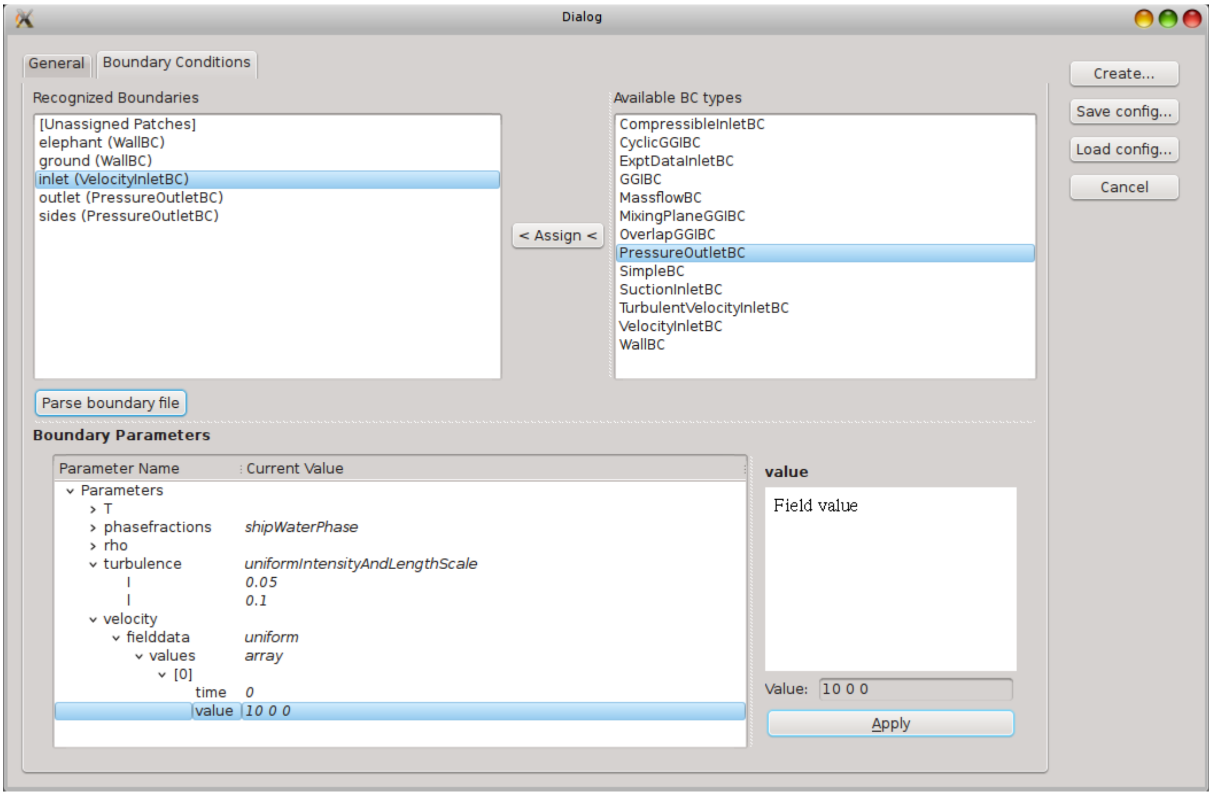
Make the solution case directory the active working directory and start the case builder program again:

```
$ cd elephant_flow
$ isofCaseBuilder
```



Execute these steps:

1. Add the following case elements to the case:
   (a) simpleFoamNumerics
   This will populate the system/ directory with appropriate settings for simpleFoam (controlDict, fvSchemes and fvSolutions file).
   (b) singlePhaseTransportProperties
   (c) kOmegaSST
2. Change the configuration of the case elements:
   (a) singlePhaseTransportProperties
      i. nu: Kinematic viscosity: 1e-5
      (air)
3. Configure boundaries: change to the tab "Boundary Conditions":



(a) Update the list of available patches by clicking "Parse boundary file".

(b) Assign BCs to the patches by selecting the patch, selecting the BC type and then click "< Assign <"

(c) Set the BC parameters for each patch. The parameter editor appears below the patch list after selecting a patch in "Recognized Boundaries" which already has an assigned BC type.
Assign the types as in the figure above. Leave the default settings, except:

   i. inlet: velocity/fielddata/values/[0]/value: 10 0 0
   (A velocity along the x-axis of 10m/s, constant in time)

4. Save the case configuration to a file for later reuse: click "Save config..."

5. Finally create the OpenFOAM case on disk: click "Create..."
The dictionaries will be written to current directory.

After the dictionaries have been created, start the solver:

```
$ simpleFoam
```

When the solver has finished, the flow is best visualized by ParaView. Again, you may start it by the wrapper script:

```
$ isPV.py
```

A pronounced flow separation at the back of the elephant is expected: