

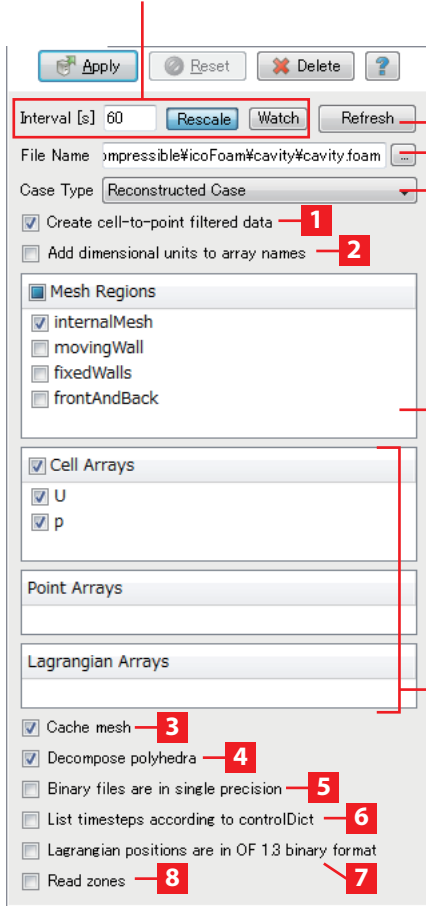
Parallelized Native OpenFOAM® Reader for ParaView: UI Reference

for version 20090920

Takuya OSHIMA <oshima@eng.niigata-u.ac.jp>

Controls marked by * are available only when the GPLed plugin is installed.

*These are used for watching a running case and refreshing the render view whenever a new timestep is detected. The **Watch** button toggles the case watching on or off. The number entered in the **Interval [s]** field specifies the interval in which new time directories are picked up. If **Rescale** is on, automatic rescaling of color mapping to data range is applied every time new timestep data is rendered. One may occasionally see reader errors due to race conditions where the reader reads up until the end of file while the file is being written.



The screenshot shows the OpenFOAM Reader panel in ParaView. It includes buttons for Apply, Reset, Delete, and a help icon. Below these are fields for Interval [s] (set to 60), Rescale, Watch, and Refresh. The File Name field contains 'compressible#icoFoam#cavity#cavity.foam'. The Case Type is set to 'Reconstructed Case'. There are checkboxes for 'Create cell-to-point filtered data' (marked 1), 'Add dimensional units to array names' (marked 2), and 'Cache mesh' (marked 3). The Mesh Regions section has checkboxes for 'internalMesh', 'movingWall', 'fixedWalls', and 'frontAndBack'. The Cell Arrays section has checkboxes for 'U' and 'p'. The Point Arrays and Lagrangian Arrays sections are empty. At the bottom, there are checkboxes for 'Decompose polyhedra' (marked 4), 'Binary files are in single precision' (marked 5), 'List timesteps according to controlDict' (marked 6), 'Lagrangian positions are in OF 1.3 binary format' (marked 7), and 'Read zones' (marked 8).

*Instruct the reader to reset everything except for the selection statuses of the reader panel, rescan timesteps and reload mesh.

Switch to another case while keeping the reader instance intact. This is especially useful when one wanted to quickly inspect another case using already constructed complex visualization pipeline.

Select whether the case to be read is a reconstructed case (a serial case) or a decomposed case (a case decomposed into *processorX* subdirectories). If ParaView is running in client/server mode and the servers (*pvserver*s) are running in parallel, the decomposed case is read in parallel. Each *processorX* subdirectory is assigned to a server process in an interleaved way. For example, if you have 5 processor subdirectories and running 2 *pvserver*s, the *pvserver* process 0 reads *processor0*, *processor2* and *processor4* subdirectories while *pvserver* process 1 reads *processor1* and *processor3* subdirectories.

Selections of internal mesh, patches and lagrangian regions. By default, internal mesh is selected. For multi-region case, internal mesh in each region except the default region is selected.

Selections of vol/point/lagrangian fields.

1. Create cell-to-point filtered data for volFields. Unlike the volPoint interpolator in OpenFOAM, the filter does not do inverse distance weighting, hence is faster but less accurate. Turning the option off allows the reader to save computational loads and memory for the filtering.

2. Read *dimensions* entries from field files, convert to human readable dimensional unit representations and suffixes array names with the units. This does not work for lagrangian fields because they do not have *dimensions* entries.

3. Cache mesh so that the reader does not have to read mesh each timestep. You usually would want to keep this checkbox checked.

4. Decompose polyhedral cells into tetrahedra and pyramids. Otherwise the reader uses type *CONVEX_POINT_SET* in order to represent polyhedra. You usually would want to turn it on, but keep in mind the option could drastically increase data sizes of polyhedral meshes (decomposition of a polyhedron usually takes 10 or more tetrahedra and pyramids).

5. Check the box in order to read a case in single

precision binary format.

6. When the checkbox is turned on and the values of the entries (*adjustTimeStep*, *writeControl*) in *controlDict* are either (*yes*, *adjustableRunTime*) or (*no*, *timeStep*) (i. e. writing interval is supposed to be constant in simulated time), the reader lists time instances according to *startTime*, *endTime*, *deltaT*, *writeInterval*, etc. Otherwise, the reader lists all valid time directories.

7. Check the box when reading lagrangian field files in binary format created by OpenFOAM 1.3 (or possibly earlier). They have different format than ones created by OpenFOAM 1.4 or later.

8. Read mesh zone information and add its geometry to the reader output. Currently the reader only handles geometry, and does not associate field data with the geometry.