RedHat系Linuxへの OpenFOAM ビルド手順まとめ

OpenFOAM-2.4.x を対象に

y.imagawa

15.6.13

RedHat系へのOpenFOAMインストール

www.openfoam.comより

OpenFOAM-2.3.0

Download OpenFOAM®

The OpenFOAM Foundation currently distributes OpenFOAM for Linux operating systems in the following ways. The latest release is version 2.3.0, please see Version 2.3.0 Release Announcement for more information.

Ubuntu Deb Pack

Version 2.3.0, released 17/02/14

The Ubuntu Deb pack is a good option for a user that wants to install OpenFOAM on a personal computer on which they have Ubuntu Linux running either as the native operating system or within a virtual machine. The pack contains both source code and binaries and includes a list of dependent system software which can be installed automatically to reduce the system administration overhead. It is therefore the **simplest installation** option, making it the ideal choice for those people with no experience of Linux. The pack is updated periodically to incorporate new features and bug fixes.
Go to Ubuntu Deb Pack Installation

SuSE RPM Pack

Version 2.3.0, released 17/02/14

OpenFOAM is also released as a source/binary pack for SuSE Linux. It is a little more difficult to install than the Ubuntu pack (above) because not all the dependent software are available as standard SuSE packages, so some additional administration effort is required.
Go to SuSE RPM Pack Installation

Red Hat (RHEL) RPM Pack

Version 2.3.0, released 17/02/14

OpenFOAM is also released as a source/binary pack for Red Hat Enterprise Linux 64 bit.
Go to Red Hat (RHEL) RPM Pack Installation

OpenFOAM-2.4.0

Download OpenFOAM®

The OpenFOAM Foundation currently distributes OpenFOAM for Linux operating systems in the following ways. The latest release is version 2.4.0, please see Version 2.4.0 Release Announcement for more information.

Ubuntu Deb Pack

Version 2.4.0, released 22/05/15

The Ubuntu Deb pack is a good option for a user that wants to install OpenFOAM on a personal computer on which they have Ubuntu Linux running either as the native operating system or within a virtual machine. The pack contains both source code and binaries and includes a list of dependent system software which can be installed automatically to reduce the system administration overhead. It is therefore the **simplest installation** option, making it the ideal choice for those people with no experience of Linux. The pack is updated periodically to incorporate new features and bug fixes.
Go to Ubuntu Deb Pack Installation

Source Pack

Version 2.4.0, released 22/05/15

This source pack distribution allows compilation a wide range of Linux distributions and versions used by OpenFOAM users running on personal computers, clusters, high-performance machines, etc. This installation requires some knowledge of Linux system administration, particularly to ensure that dependent software, e.g. compilers and libraries, are installed correctly. The source pack comes with a pack of third-party software that is used by OpenFOAM.
Go to Source Pack Installation

Git Repository

Version 2.4.x, updated regularly

Those users compiling from source can receive regular updates to OpenFOAM by downloading the Git repository distribution of OpenFOAM. Known as 2.4.x, this version is managed and **updated daily** by the OpenFOAM Foundation and can be conveniently "pulled" using the Git distributed revision control system.
Go to Repository Installation

OpenFOAM ver 2.3.0にRHEL用パッケージがあったが、

ver 2.4.0には（現時点では）存在しない

OpenFOAMコンパイルのススメ

http://www.openfoam.org/download/git.php

Dependent packages required for RHEL 6.5 need updating to newer versions, *e.g.* Gcc needs upgrading from v4.4 to v4.7+. These upgrades may be available under a RHEL support subscription. If not, then upgrades can be obtained from unofficial repositories using the following instructions:

gccを4.4から4.7へのアップグレードが必要

- RHELのサポートで対応してくれるかも
- 非公式のリポジトリからダウンロード

システムに影響するgccやライブラリ一式をupgradeするのは怖い
→システムに(ほとんど)影響を与えない環境を構築

- システム標準のrpmパッケージを使う
- 必要なツールはビルド（野良ビルド）する
- 不要になったらフォルダごと削除できる環境にする

想定環境

- OS　：　CentOS 6.6 (64bit)
  - OSインストール構成：Software Development Workstation
  - 必要なパッケージは、標準リポジトリからyumでインストール
- OpenFOAM　：　ver 2.4.x
  - ソルバやユーティリティのみ作成
  - paraFoamは作らない（ParaViewをダウンロードして使う）
  - 必要なツール(gcc,mpiなど）はOpenFOAMフォルダの中でビルド

コンパイルの流れ

1. 必要なrpmパッケージの準備
2. 必要なソースのダウンロード & 配置
3. 設定ファイルの編集
4. コンパイラ、ツールのコンパイル
5. Allwmake
6. テスト(foamInstallationTest)

1.必要なrpmパッケージ

1.必要なrpmパッケージ

CentOSのOSインストール構成が Server の場合、
以下のパッケージが追加で必要

- gcc
- gcc-c++
- bison
- flex
- m4
- glibc-devel
- glibc-devel.i686
- zlib-devel
- git

管理者権限で以下を実行すると、インストールされる（要外部ネットワーク接続）

% yum install gcc ~~gcc-g+~~+ bison flex m4 glibc-devel glibc-devel.i686 zlib-devel
gcc-c++

→Software Development Workstationは、Serverよりも多くのパッケージが入っているため、
上記のパッケージをyumコマンドで追加する

2. 必要なソースのダウンロード

OpenFOAM source

% git clone https://github.com/OpenFOAM/OpenFOAM-2.4.x.git

OpenFOAM ThirdParty source

http://downloads.sourceforge.net/foam/ThirdParty-2.4.0.tgz?use_mirror=mesh

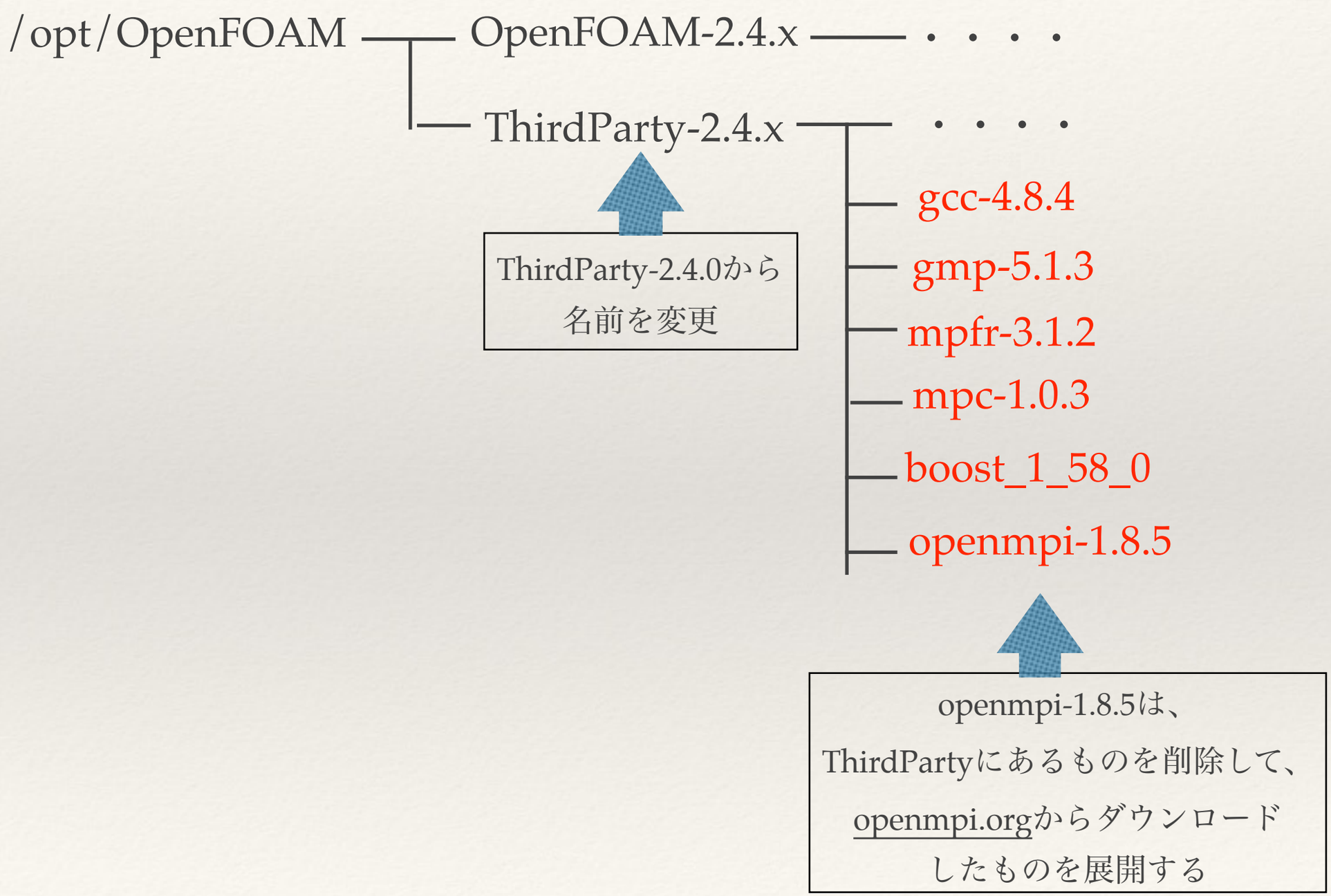
OpenFOAM用コンパイラ等

http://ftp.jaist.ac.jp/pub/GNU/gcc/gcc-4.8.4.tar.bz2
http://ftp.jaist.ac.jp/pub/GNU/mpfr/mpfr-3.1.2.tar.bz2
http://ftp.jaist.ac.jp/pub/GNU/gmp/gmp-5.1.3.tar.bz2
http://ftp.jaist.ac.jp/pub/GNU/mpc/mpc-1.0.3.tar.gz

http://sourceforge.net/projects/boost/files/boost/1.58.0/boost_1_58_0.tar.bz2/download

http://www.open-mpi.org/software/ompi/v1.8/openmpi-1.8.5.tar.gz

2.ファイルの配置


/opt/OpenFOAM
OpenFOAM-2.4.x
・・・・
ThirdParty-2.4.x
・・・・
gcc-4.8.4
gmp-5.1.3
mpfr-3.1.2
mpc-1.0.3
boost_1_58_0
openmpi-1.8.5
ThirdParty-2.4.0から
名前を変更
openmpi-1.8.5は、
ThirdPartyにあるものを削除して、
openmpi.orgからダウンロード
したものを展開する

3. 設定ファイルの編集

OpenFOAM/ThirdParty-2.4.x/makeGcc47の編集

OpenFOAM/OpenFOAM-2.4.x/etc/bashrcの編集

OpenFOAM/OpenFOAM-2.4.x/etc/config/settings.shの編集

OpenFOAM/ThirdParty-2.4.x/Allwmakeの編集

OpenFOAM/ThirdParty-2.4.x/makeCGALの編集

OpenFOAM/OpenFOAM-2.4.x/etc/config/CGAL.shの編集

OpenFOAM/ThirdParty-2.4.x/Allwmakeの編集

OpenFOAM/ThirdParty-2.4.x/makeGcc の編集

```
# Note
#     gcc 4.3.2 is known to miscompile GMP 4.3.x on 64-bit machines
#
#------------------------------------------------------------------------
gmpPACKAGE=gmp-5.1.3
mpfrPACKAGE=mpfr-3.1.2
mpcPACKAGE=mpc-1.0.3
gccPACKAGE=gcc-4.8.4
```

OpenFOAM/OpenFOAM-2.4.x/etc/bashrc の編集

```
#- Compiler location:
#    foamCompiler= system | ThirdParty (OpenFOAM)
#foamCompiler=system
foamCompiler=ThirdParty


~~~~~~~~~~~~~~~~~~

#- MPI implementation:
#    WM_MPLIB = SYSTEMOPENMPI | OPENMPI | SYSTEMMPI | MPICH |
MPICH-GM | HPMPI
#              | MPI | QSMPI | SGIMPI
# export WM_MPLIB=SYSTEMOPENMPI
export WM_MPLIB=OPENMPI
```

OpenFOAM/OpenFOAM-2.4.x/etc/config/settings.sh の編集

```
case "${foamCompiler}" in
OpenFOAM | ThirdParty)
    case "$WM_COMPILER" in
    Gcc | Gcc++0x | Gcc48 | Gcc48++0x)
        gcc_version=gcc-4.8.4
        gmp_version=gmp-5.1.3
        mpfr_version=mpfr-3.1.2
        mpc_version=mpc-1.0.3
        ;;
```

OpenFOAM/ThirdParty-2.4.x/makeCGAL の編集

```
#     Mixing system and ThirdParty for gmp/mpfr is not supported.
#
#-----------------------------------------------------------------------
set -x

cgalPACKAGE=CGAL-4.6
boostPACKAGE=boost_1_58_0
gmpPACKAGE=gmp-5.1.3
mpfrPACKAGE=mpfr-3.1.2
```

OpenFOAM/OpenFOAM-2.4.x/etc/config/CGAL.shの編集

```
# File
#     config/CGAL.sh
#
# Description
#     Setup file for CGAL (& boost) include/libraries.
#     Sourced from OpenFOAM-<VERSION>/etc/bashrc
#------------------------------------------------------------------------

boost_version=boost_1_58_0
cgal_version=CGAL-4.6
```

OpenFOAM/ThirdParty-2.4.x/Allwmake の編集

```
    ./configure \
      --prefix=$MPI_ARCH_PATH \
      --disable-orterun-prefix-by-default \
      --enable-shared --disable-static \
      --libdir=$MPI_ARCH_PATH/lib${WM_COMPILER_LIB_ARCH} \
#       --disable-mpi-f77 \
#       --disable-mpi-f90 \
    --enable-mpi-fortran= no \
      --disable-mpi-profile \
      $configOpt \
      ;
```

openmpi-1.8.5のconfigureでは、

—disable-mpi-f77, —disable-mpi-f90が利用できないため、

—enable-mpi-fortran=noを指定する

makeGcc

OpenFOAM用のコンパイラを作る

```
% . /opt/OpenFOAM/OpenFOAM-2.4.x/etc/bashrc（OpenFOAM環境変数読み込み）
（以下のようなWarningが出るが、気にしない）

Warning in /opt/OpenFOAM/OpenFOAM-2.4.x/etc/config/settings.sh:
   Cannot find /opt/OpenFOAM/ThirdParty-2.4.x/platforms/linux64/gcc-4.8.4 installation.
   Please install this compiler version or if you wish to use the system compiler,
   change the 'foamCompiler' setting to 'system'

% cd /opt/OpenFOAM/ThirdParty-2.4.x
% ./makeGcc
（コンパイル実行）
% exit （OpenFOAM環境から抜ける）
```

makeCmake

ParaViewをコンパイルするのに必要なcmakeを作成

```
% . /opt/OpenFOAM/OpenFOAM-2.4.x/etc/bashrc（OpenFOAM環境変数読み込み）
(コンパイラ環境ができていれば、Warningは出ない)
% cd /opt/OpenFOAM/ThirdParty-2.4.x
% ./makeCmake
 （コンパイル実行）
% exit （OpenFOAM環境から抜ける）
```

makeCmake実行直後は、cmakeにPATHが通ってない

OpenFOAM環境変数を読み直す

makeCGAL

```
% . /opt/OpenFOAM/OpenFOAM-2.4.x/etc/bashrc（OpenFOAM環境変数読み込み）
(コンパイラ環境ができていれば、Warningは出ない)
% cd /opt/OpenFOAM/ThirdParty-2.4.x
% ./makeCGAL
（コンパイル実行）
% exit （OpenFOAM環境から抜ける）
```

makeCGAL実行直後は、CGALにPATHが通ってない

OpenFOAM環境変数を読み直すため、OpenFOAM環境から抜ける

Allwmake

mpi, scotch, OpenFOAM library, applicationをコンパイルする

```
% . /opt/OpenFOAM/OpenFOAM-2.4.x/etc/bashrc（OpenFOAM環境変数読み込み）
% export WM_NCOMPPROCS=4 (４並列でコンパイル実行)
% cd /opt/OpenFOAM/OpenFOAM-2.4.x
% ./Allwmake
（コンパイル実行）
% exit （OpenFOAM環境から抜ける）
```

テスト

```
% foamInstallationTest

    . . . . . .

Summary
-------------------------------------------------------------------------
Base configuration ok.
Critical systems ok.

Done
```

おまけ：paraFoam

想定環境

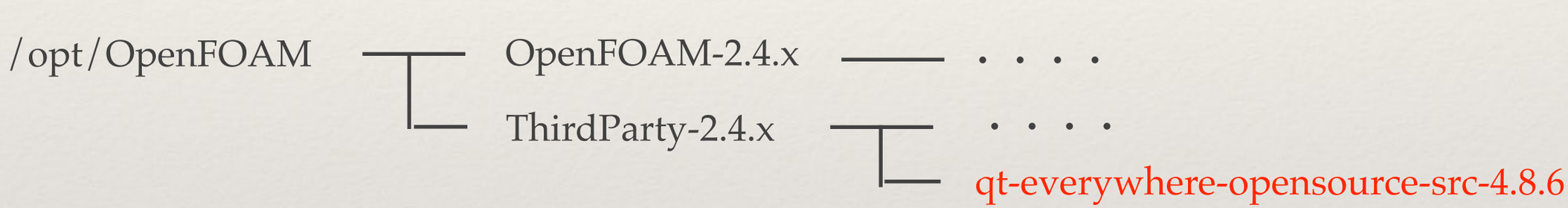
- OS　：　CentOS 6.6 (64bit)
  - インストール環境：Software Development Workstation
  - 必要なパッケージは、標準リポジトリからyumでインストール
- OpenFOAM　：　ver 2.4.x
  - ソルバやユーティリティのみ作成
  - paraFoamは作らない→**作ってみた**
  - 必要なツール(gcc,mpiなど）はOpenFOAMフォルダの中でビルド

導入準備

Qt ダウンロード

http://download.qt.io/archive/qt/4.8/4.8.6/qt-everywhere-opensource-src-4.8.6.tar.gz

Qt の配置



OpenFOAM/ThirdParty-2.4.x/makeQtの編集

```
#-----------------------------------------------------------------------
qtVERSION=4.8.6
qtTYPE=qt-everywhere-opensource-src
#-----------------------------------------------------------------------
```

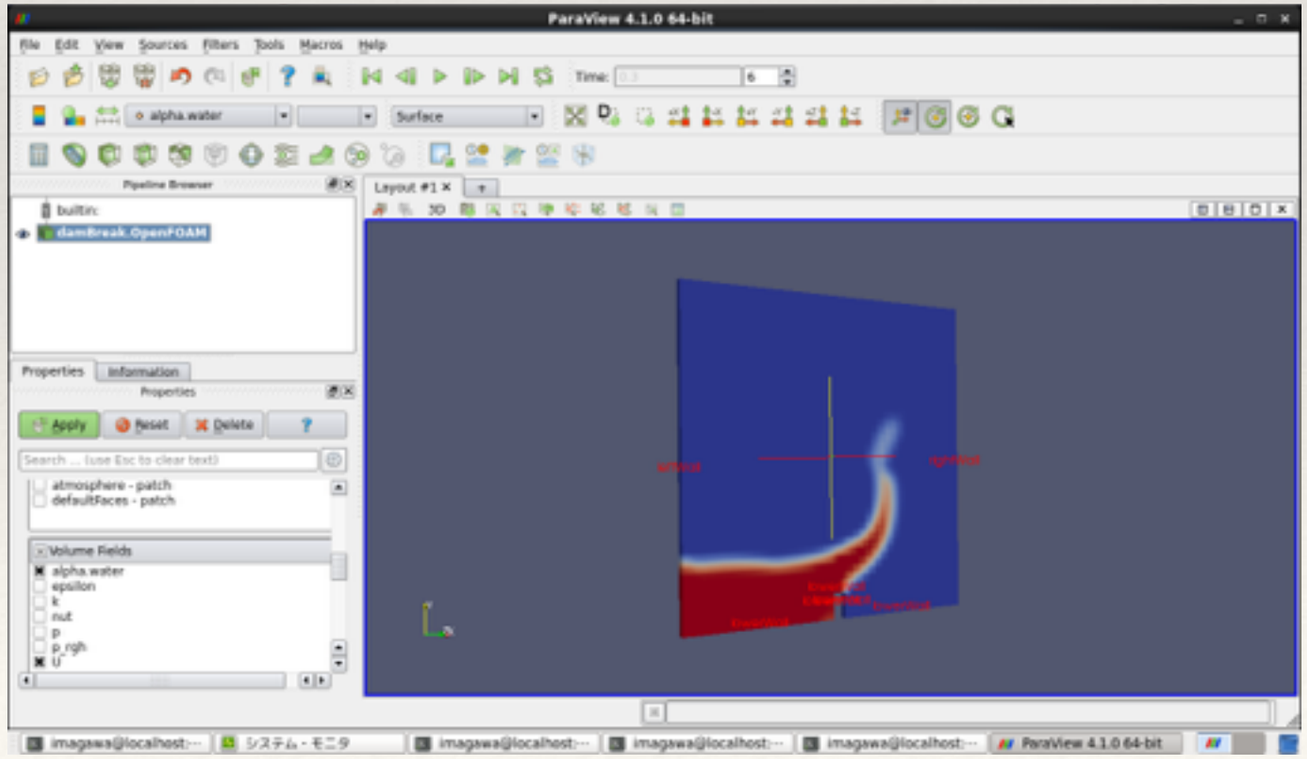
必要なrpmパッケージの導入(rootにて)

```
% yum install gstreamer-plugins-*
```

ビルドと結果

% . /opt/OpenFOAM/OpenFOAM-2.4.x/etc/bashrc（OpenFOAM環境変数読み込み）

% cd /opt/OpenFOAM/ThirdParty-2.4.x
% ./makeQt　（Qtの作成）

% ./makeParaView4（ParaView4の作成）

% cd $FOAM_UTILITIES/postProcessing/graphics/PV4Readers
% ./Allwclean
% ./Allwmake（PV4Readerのビルド）