ただで始める流体解析

2.2.xのインストール_Update版

初心者の私でも今回はコンパイルできました？

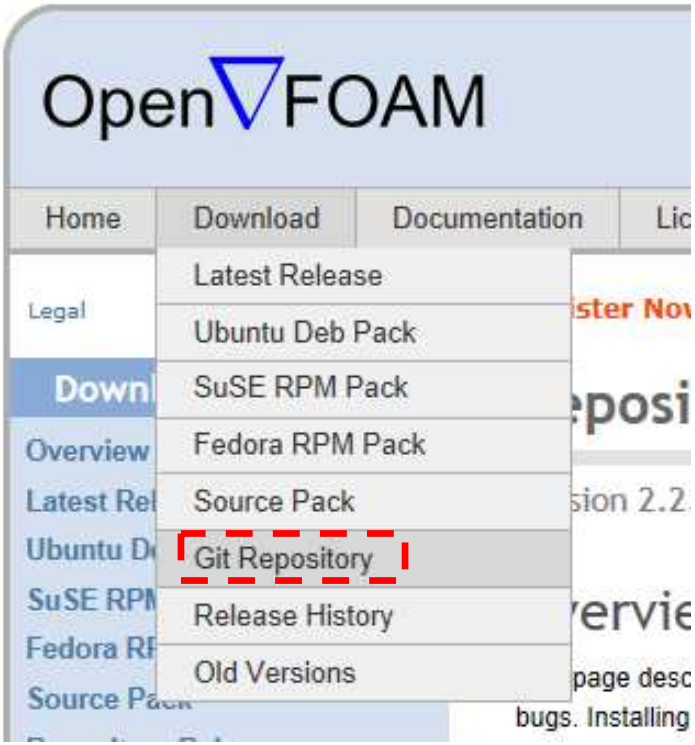
自分の環境：ubuntu 12.04（11.04→11.10さらにアップ）

gccのバージョン

```
sakuramaru@SAKURA-MARU:~/OpenFOAM/OpenFOAM-2.2.x/bin$ gcc --version
gcc (Ubuntu/Linaro 4.6.3-1ubuntu5) 4.6.3
Copyright (C) 2011 Free Software Foundation, Inc.
This is free software; see the source for copying conditions.  There is NO
warranty; not even for MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE.
```

①OpenFOAMの導入

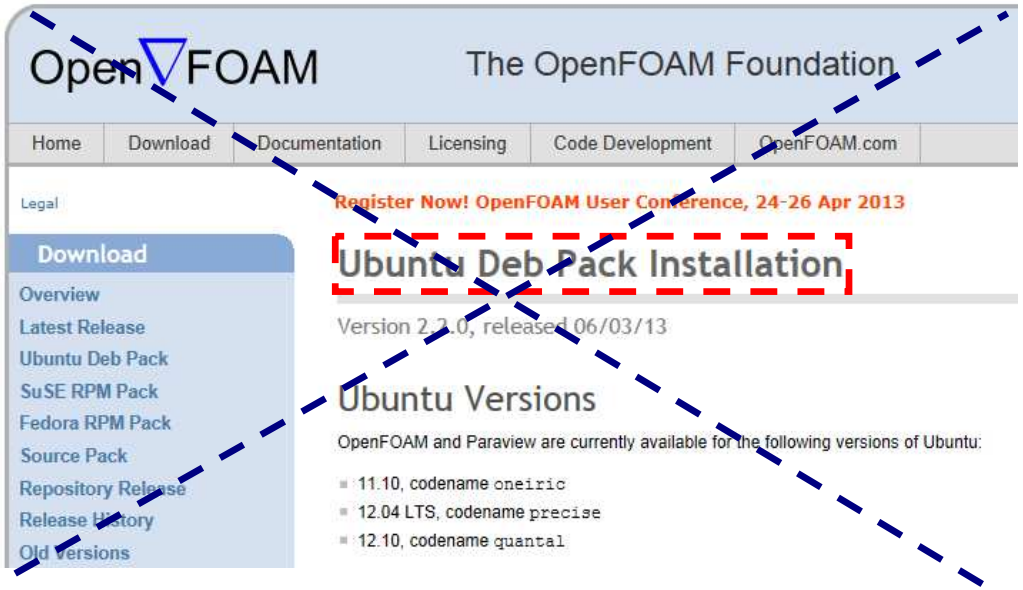
Git Repositoryから持ってきてコンパイルする。



詳細な手順はOpenFOAMのサイトを参照

お手軽なバイナリインストールもあるが，頻繁にアップがあるため，Git Repositoryがお勧め，これがベスト。

毎日ちょこっと差分をコンパイルすれば最新が常に使える。これに限る！



①OpenFOAMの導入

OpenFOAMのサイトから必要なファイルを持ってくる。

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

OpenFOAM-2.2.x

The OpenFOAM-2.2.x repository is available at http://github.com/OpenFOAM/OpenFOAM-2.2.x. From the OpenFOAM installation root directory the repository can be obtained using

EITHER

the git protocol (`git://`), which is very efficient:

- `git clone git://github.com/OpenFOAM/OpenFOAM-2.2.x.git`

Note: the git protocol will not work if your computer is behind a firewall which blocks the relevant TCP port (9418).

OR

the http protocol (`http://`), which is fairly inefficient:

- `git clone http://github.com/OpenFOAM/OpenFOAM-2.2.x.git`

Note: while the TCP port for http is rarely blocked by a firewall, download hangs have been experienced; upgrading to git version 1.7 seems to overcome this problem.

Either of the commands above will create an OpenFOAM-2.2.x directory that the user can subsequently be updated to the latest published copy using

- `cd OpenFOAM-2.2.x`
- `git pull`

①OpenFOAMの導入

OpenFOAMのサイトから必要なファイルを持ってくる。

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

ThirdParty

The source-pack containing additional software required to build and run OpenFOAM-2.2.x is available from SourceForge: Click to download ThirdParty-2.2.0.tgz now. In the OpenFOAM installation root directory unpack the ThirdParty-2.2.x.gtgz file and rename the unpacked directory `ThirdParty-2.2.x`:

```
tar xzf ThirdParty-2.2.0.tgz
mv ThirdParty-2.2.0 ThirdParty-2.2.x
```

②bashrcの書き換え

Setting Environment Variables

The environment variable settings are contained in files in an `OpenFOAM-2.2.x/etc` directory in the OpenFOAM release. e.g. for the case where the installation is in `$HOME/OpenFOAM`, in:

- `$HOME/OpenFOAM/OpenFOAM-2.2.x/etc`

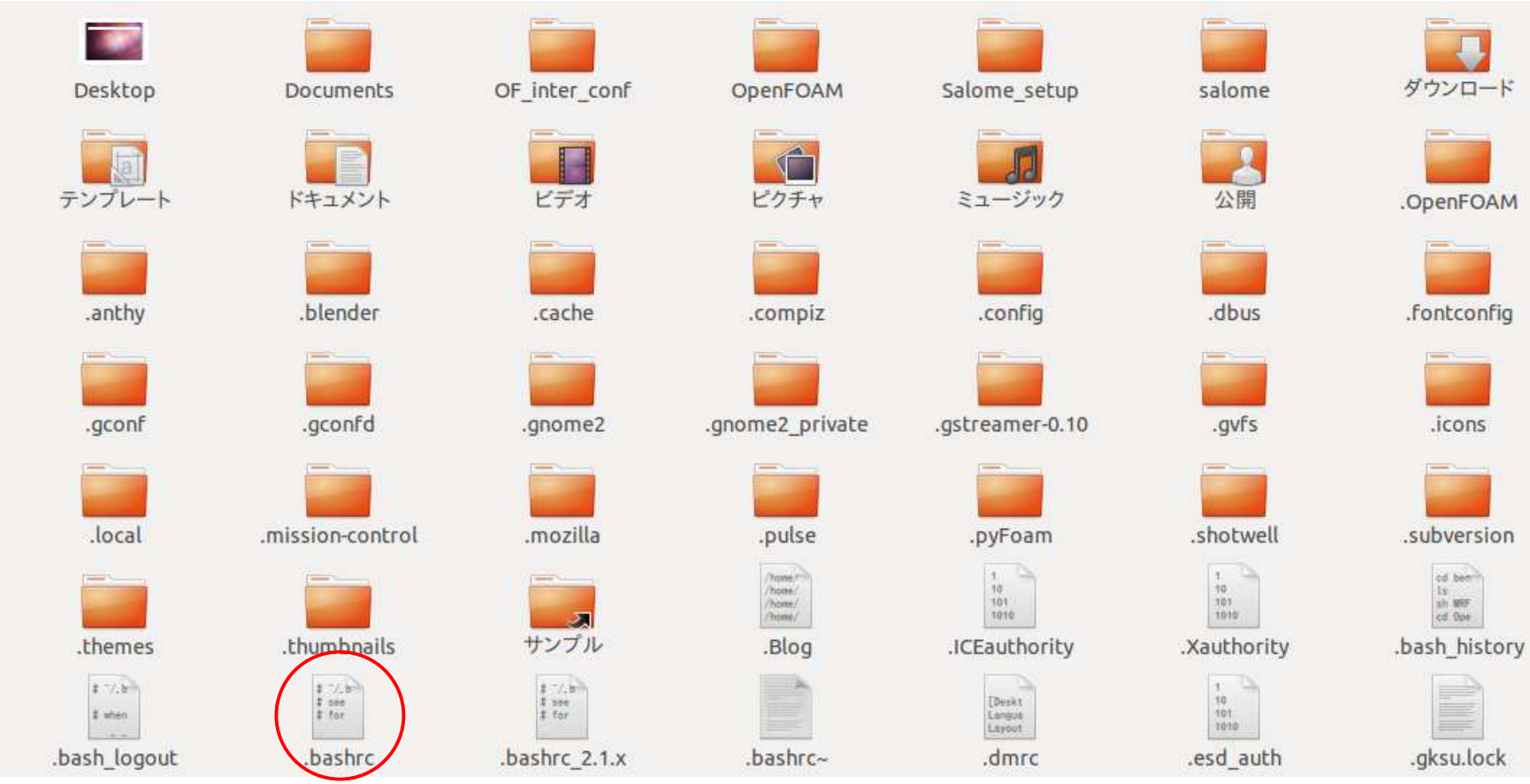
EITHER

if running `bash` or `ksh` (if in doubt type `echo $SHELL`), source the `etc/bashrc` file by adding the following line to the end of your `$HOME/.bashrc` file:

```
source $HOME/OpenFOAM/OpenFOAM-2.2.x/etc/bashrc
```

then type "`source $HOME/.bashrc`" in the current terminal window

②bashrcの書き換え



②bashrcの書き換え

既にあるVer2.1.xの設定をコメントアウトして，Ver2.2.xの設定を追加している。

```
#2.1.x
#source $HOME/OpenFOAM/OpenFOAM-2.1.x/etc/bashrc
#. $WM_PROJECT_DIR/bin/tools/RunFunctions

#export ParaView_DIR=/home/sakuramaru/OpenFOAM/ThirdParty-2.1.x/platforms/linux64Gcc/paraview-3.12.0
#export PATH=$ParaView_DIR/bin:$PATH
#export PV_PLUGIN_PATH=$FOAM_LIBBIN/paraview-3.12
```

```
#2.2.x
source $HOME/OpenFOAM/OpenFOAM-2.2.x/etc/bashrc
. $WM_PROJECT_DIR/bin/tools/RunFunctions

export ParaView_DIR=/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/platforms/linux64Gcc/paraview-3.12.0
export PATH=$ParaView_DIR/bin:$PATH
export PV_PLUGIN_PATH=$FOAM_LIBBIN/paraview-3.12
```

③OpenFOAMのコンパイル

インストール前のチェック

```
sakuramaru@SAKURA-MARU:~/OpenFOAM/OpenFOAM-2.2.x/bin$ foamSystemCheck

Checking basic system...
------------------------------------------------------------------------
Shell:              /bin/bash
Host:               SAKURA-MARU
OS:                 Linux version 3.2.0-38-generic
User:               sakuramaru


System check: PASS
==================
Continue OpenFOAM installation.
```

コンパイルを速くするためパラレルでも出来るが，シリアルコンパイルで行う。ちなみにパラレルコンパイルをした後にfoamInstallationTestを行うと，次のような状態になる。

```
Executing /home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest:


Checking basic setup...
--------------------------------------------------------------------------------
Shell:                bash
Host:                 SAKURA-MARU
OS:                   Linux version 3.2.0-38-generic
--------------------------------------------------------------------------------


Checking main OpenFOAM env variables...
--------------------------------------------------------------------------------
Environment_variable Set_to_file_or_directory                Valid      Crit
--------------------------------------------------------------------------------
$WM_PROJECT_INST_DIR /home/sakuramaru/OpenFOAM                yes        yes
$WM_PROJECT_USER_DIR ...sakuramaru/OpenFOAM/sakuramaru-2.2.x  no         no
$WM_THIRD_PARTY_DIR  ...sakuramaru/OpenFOAM/ThirdParty-2.2.x  yes        yes
--------------------------------------------------------------------------------


Checking the OpenFOAM env variables set on the PATH...
--------------------------------------------------------------------------------
Environment_variable Set_to_file_or_directory                Valid Path Crit
--------------------------------------------------------------------------------
$WM_PROJECT_DIR      /home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x  yes  yes  yes

$FOAM_APPBIN         ...-2.2.x/platforms/linux64GccDPOpt/bin  yes  yes  yes
$FOAM_SITE_APPBIN    .../2.2.x/platforms/linux64GccDPOpt/bin  no        no
$FOAM_USER_APPBIN    ...-2.2.x/platforms/linux64GccDPOpt/bin  no        no
$WM_DIR              ...ramaru/OpenFOAM/OpenFOAM-2.2.x/wmake  yes  yes  yes
--------------------------------------------------------------------------------
```

```
Checking the OpenFOAM env variables set on the LD_LIBRARY_PATH...
-------------------------------------------------------------------------------
Environment_variable Set_to_file_or_directory                Valid Path Crit
-------------------------------------------------------------------------------
$FOAM_LIBBIN         ...-2.2.x/platforms/linux64GccDPOpt/lib  yes  yes  yes
$FOAM_SITE_LIBBIN    .../2.2.x/platforms/linux64GccDPOpt/lib  no        no
$FOAM_USER_LIBBIN    ...-2.2.x/platforms/linux64GccDPOpt/lib  no        no
$MPI_ARCH_PATH       ...x/platforms/linux64Gcc/openmpi-1.6.3  no        yes
-------------------------------------------------------------------------------
```

これが後で
問題になる

```
Third party software
-----------------------------------------------------------------------------
Software Version   Location
-----------------------------------------------------------------------------
flex     2.5.35    /usr/bin/flex
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 267: [: -lt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 269: [: -gt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 274: [: -lt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 276: [: -gt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 281: [: !=: unexpected operator
gcc                /usr/bin/gcc
gzip     1.4       /bin/gzip
tar      1.26      /bin/tar
icoFoam  2.2.x     ...M/OpenFOAM-2.2.x/platforms/linux64GccDPOpt/bin/icoFoam
-----------------------------------------------------------------------------


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

Done
```

エラー　？

icoFoamはちゃんと動くが

ParaViewのコンパイルでpython, mpiのフラグを付けて実施するとエラーが出る。対策方法は後で説明。

cd $WM_THIRD_PARTY_DIR
export WM_NCOMPPROCS=4
./makeParaView -python -python-lib /usr/lib/libpython2.7.so.1.0 -mpi

```
[  2%] Built target vtkDICOMParser
Scanning dependencies of target vtkproj4
[  2%] Built target ProcessShader
Scanning dependencies of target MapReduceMPI
[  2%] Building CXX object VTK/Utilities/mrmpi/src/CMakeFiles/MapReduceMPI.dir/mapreduce.cpp.o
/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/ParaView-3.12.0/VTK/Utilities/mrmpi/src/mapreduce.cpp:14:17:
致命的エラー: mpi.h: そのようなファイルやディレクトリはありません
コンパイルを停止しました。
make[2]: *** [VTK/Utilities/mrmpi/src/CMakeFiles/MapReduceMPI.dir/mapreduce.cpp.o] エラー 1
make[1]: *** [VTK/Utilities/mrmpi/src/CMakeFiles/MapReduceMPI.dir/all] エラー 2
```

```
[  5%] Built target vtkproj4
Scanning dependencies of target lproj
[  5%] Building C object VTK/Utilities/vtklibproj4/CMakeFiles/lproj.dir/lproj.c.o
Linking C executable ../../../bin/lproj
[  5%] Built target lproj
[  5%] Building CXX object VTK/Utilities/mrmpi/src/CMakeFiles/MapReduceMPI.dir/mapreduce.cpp.o
/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/ParaView-3.12.0/VTK/Utilities/mrmpi/src/mapreduce.cpp:14:17: 致命的エラー:
コンパイルを停止しました。
make[2]: *** [VTK/Utilities/mrmpi/src/CMakeFiles/MapReduceMPI.dir/mapreduce.cpp.o] エラー 1
make[1]: *** [VTK/Utilities/mrmpi/src/CMakeFiles/MapReduceMPI.dir/all] エラー 2
make: *** [all] エラー 2
    ---
    Installation complete for paraview-3.12.0
    Set environment variables:

        export ParaView_DIR=/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/platforms/linux64Gcc/paraview-3.12.0
        export PATH=$ParaView_DIR/bin:$PATH
        export PV_PLUGIN_PATH=$FOAM_LIBBIN/paraview-3.12
```

これで止まってしまう

シリアルコンパイルで実施すると時間はかかるが，
foamInstallationTestを行うと，次のような状態になる。

```
sakuramaru@SAKURA-MARU:~/OpenFOAM/OpenFOAM-2.2.x/bin$ foamInstallationTest
Executing /home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest:


Checking basic setup...
-------------------------------------------------------------------------------
Shell:              bash
Host:               SAKURA-MARU
OS:                 Linux version 3.2.0-39-generic
-------------------------------------------------------------------------------


Checking main OpenFOAM env variables...
-------------------------------------------------------------------------------
Environment_variable Set_to_file_or_directory                Valid      Crit
-------------------------------------------------------------------------------
$WM_PROJECT_INST_DIR /home/sakuramaru/OpenFOAM                yes        yes
$WM_PROJECT_USER_DIR ...sakuramaru/OpenFOAM/sakuramaru-2.2.x  yes        no
$WM_THIRD_PARTY_DIR  ...sakuramaru/OpenFOAM/ThirdParty-2.2.x  yes        yes
-------------------------------------------------------------------------------
```

```
Checking the OpenFOAM env variables set on the PATH...
-------------------------------------------------------------------------------
Environment_variable Set_to_file_or_directory                Valid Path Crit
-------------------------------------------------------------------------------
$WM_PROJECT_DIR      /home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x  yes  yes  yes

$FOAM_APPBIN         ...-2.2.x/platforms/linux64GccDPOpt/bin  yes  yes  yes
$FOAM_SITE_APPBIN    .../2.2.x/platforms/linux64GccDPOpt/bin  no        no
$FOAM_USER_APPBIN    ...-2.2.x/platforms/linux64GccDPOpt/bin  yes  yes  no
$WM_DIR              ...ramaru/OpenFOAM/OpenFOAM-2.2.x/wmake  yes  yes  yes
-------------------------------------------------------------------------------


Checking the OpenFOAM env variables set on the LD_LIBRARY_PATH...
-------------------------------------------------------------------------------
Environment_variable Set_to_file_or_directory                Valid Path Crit
-------------------------------------------------------------------------------
$FOAM_LIBBIN         ...-2.2.x/platforms/linux64GccDPOpt/lib  yes  yes  yes
$FOAM_SITE_LIBBIN    .../2.2.x/platforms/linux64GccDPOpt/lib  no        no
$FOAM_USER_LIBBIN    ...-2.2.x/platforms/linux64GccDPOpt/lib  yes  yes  no
$MPI_ARCH_PATH       ...x/platforms/linux64Gcc/openmpi-1.6.3  yes  yes  yes
-------------------------------------------------------------------------------


Third party software
-------------------------------------------------------------------------------
Software Version   Location
-------------------------------------------------------------------------------
flex     2.5.35    /usr/bin/flex
```

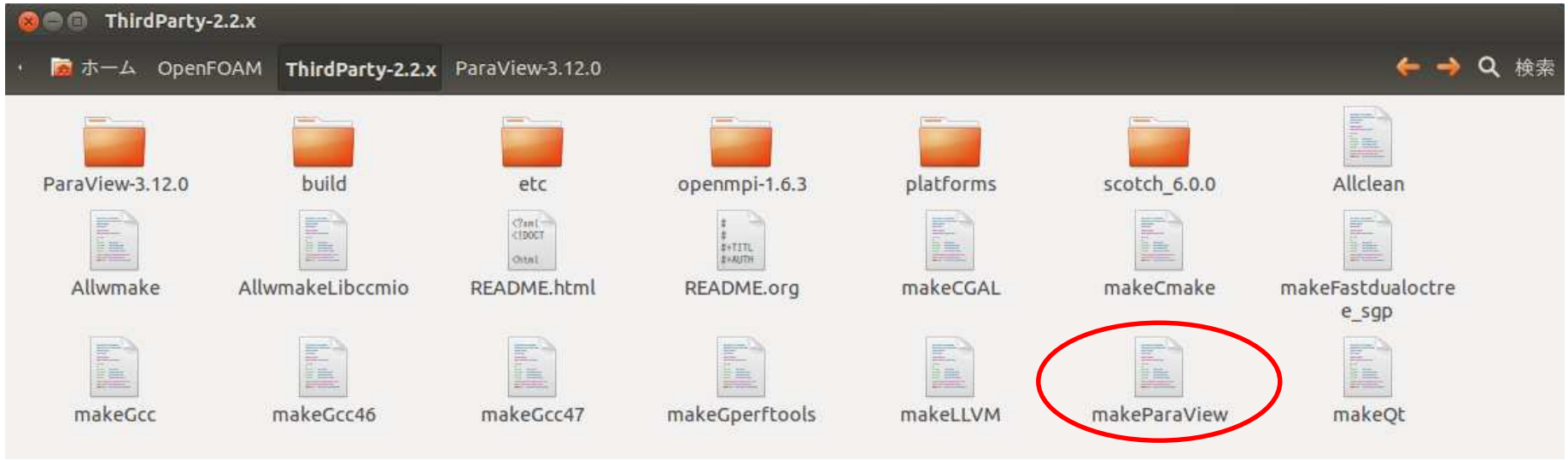
パラレルコンパイルとシリアルコンパイルで異なる

```
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 267: [: -lt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 269: [: -gt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 274: [: -lt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 276: [: -gt: unexpected operator
/home/sakuramaru/OpenFOAM/OpenFOAM-2.2.x/bin/foamInstallationTest: 281: [: !=: unexpected operator
gcc                 /usr/bin/gcc
gzip     1.4        /bin/gzip
tar      1.26       /bin/tar
icoFoam  2.2.x      ...M/OpenFOAM-2.2.x/platforms/linux64GccDPOpt/bin/icoFoam
-----------------------------------------------------------------------------


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

Done
```

この部分のエラーはいっしょだが，後日会社のPCでコンパイルした時には出なかった。

④ParaViewのコンパイル

makeParaViewを見るとpython，mpiはfalseになっている。




```
# MPI support:
withMPI=false
MPI_MAX_PROCS=32

# Python support:
# note: script will try to determine the appropriate python library.
#       If it fails, specify the path using the PYTHON_LIBRARY variable
withPYTHON=false
PYTHON_LIBRARY=""
# PYTHON_LIBRARY="/usr/lib64/libpython2.6.so.1.0"
```

古い

python，mpiを有効にするため，引数をつけてコンパイルする。
ただし，mpiの設定ではリンク設定が必要になる。
リンク設定なしでコンパイルするとエラーになる。

cd $WM_THIRD_PARTY_DIR
export WM_NCOMPPROCS=4
./makeParaView -python -python-lib /usr/lib/libpython2.7.so.1.0 -mpi

```
sakuramaru@SAKURA-MARU:~/OpenFOAM/ThirdParty-2.2.x$ ./makeParaView -python -python-lib /usr/lib/libpy
----
Python information:
    executable     : /usr/bin/python
    version        : 2.7
    include path   : /usr/include/python2.7
    library        : /usr/lib/libpython2.7.so.1.0

ParaView_SOURCE_DIR=/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/ParaView-3.12.0
ParaView_BINARY_DIR=/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/build/linux64Gcc/paraview-3.12.0
ParaView_DIR=/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/platforms/linux64Gcc/paraview-3.12.0
date-stamp: 2011-07-26

Build stages selected
---------------------
    -config   true
    -make     true
    -install  true
---------------------
```

コンパイルの途中での表示

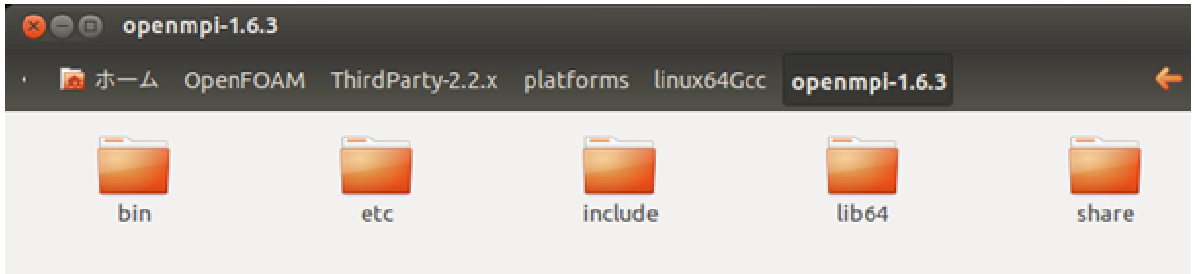
```
Features selected
    mesa       false
    mpi        true
    python     true
    qt         true
---------------------
Version information
    qt         4.8.1
    version    3.12.0
    major      3.12
MISMATCH!
    specified 3.12.0
    found
---------------------
removing old build directory
    /home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/build/linux64Gcc/paraview-3.12.0
----
Configuring paraview-3.12.0 (major version: 3.12)
    MPI    support : true
    Python support : true
    MESA   support : false
    Qt dev support : true
    Source         : /home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/ParaView-3.12.0
    Build          : /home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/build/linux64Gcc/paraview-3.12.0
    Target         : /home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/platforms/linux64Gcc/paraview-3.12.0
```

```
[  4%] Built target vtkMaterialLibraryConfiguredFiles
[  5%] Built target vtkproj4
Scanning dependencies of target lproj
[  5%] Building C object VTK/Utilities/vtklibproj4/CMakeFiles/lproj.dir/lproj.c.o
Linking C executable ../../../bin/lproj
[  5%] Built target lproj
make[2]: *** `bin/libMapReduceMPI.so.pv3.12' に必要なターゲット `/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/platforms/linux64Gcc/openmpi-1.6.3/lib/libmpi.so' を make するル
ールがありません．中止．
make[1]: *** [VTK/Utilities/mrmpi/src/CMakeFiles/MapReduceMPI.dir/all] エラー 2
make: *** [all] エラー 2
    ---
    Installation complete for paraview-3.12.0
    Set environment variables:

        export ParaView_DIR=/home/sakuramaru/OpenFOAM/ThirdParty-2.2.x/platforms/linux64Gcc/paraview-3.12.0
        export PATH=$ParaView_DIR/bin:$PATH
        export PV_PLUGIN_PATH=$FOAM_LIBBIN/paraview-3.12
    ---

Done
```

エラー発生

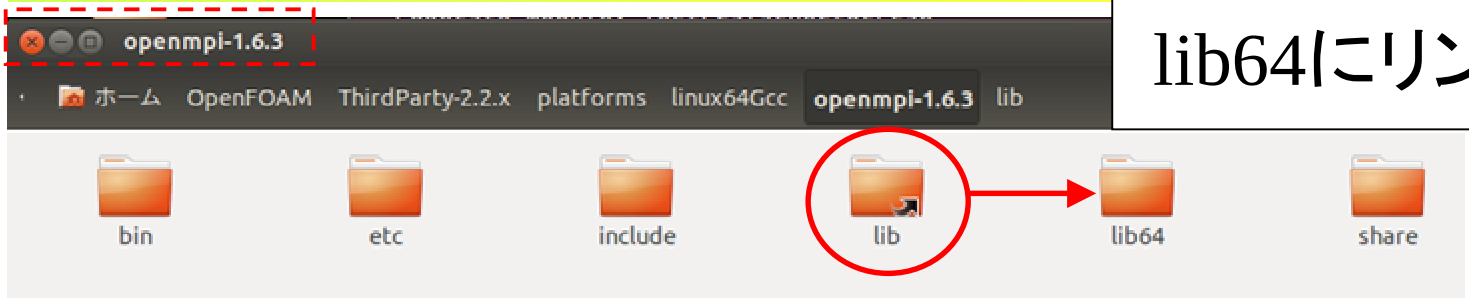
```
/ThirdParty-2.2.x/platforms/linux64Gcc/openmpi-1.6.3/lib/libmpi.so' を make
```

ホルダをチェックするとlibがない？

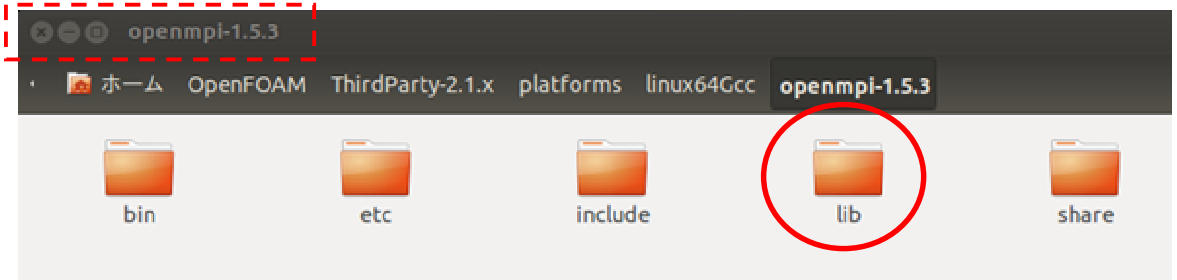


./makeParaViewだけならエラーは出ない

lib64にリンクをはる



OpenFOAM 2.1の時はlibであった



今回はめでたくコンパイル成功！