OpenFOAM 流体構造連成解析

概要：
「OpenFOAM-1.5-devに搭載されている連成解析
ソルバー（icoFsiFoam）と、公開されている片持ち梁の
例題（flappingConsoleSmall）を使って、
それらの使用方法と、（例題の）拡張方法を演習する。」

目次

- はじめに
- プログラム（icoFsiFoam）の説明
- 公開ケース（flappingConsoleSmall）の説明
- DEXCSランチャーの説明
- 解析実習とパラメタ変更要領の概説
- 解析事例の紹介と課題

はじめに（背景）

- OpenFOAMによる流体構造連成解析
  - 公開版（~1.7.0）標準ソルバーは存在しない
  - 拡張版（1.5-dev）にicoFsiFoam有るがtutorialは無い
  - 一般公開情報はいくつか存在

http://bit.ly/cIogWp

CFD Online
www.cfd-online.com

http://www.cfd-online.com/Forums/openfoam-solving/58153-fsi-porous-media-thin-shell.html

May 28, 2008, 06:15 | Hi Hrv, Thanks for your rep | #3

novak
New Member

Novak Elliott
Join Date: Mar 2009
Posts: 6
Rep Power: 3

Hi Hrv,

Thanks for your reply. I found the following thesis report on FSI with OpenFOAM:
http://powerlab.fsb.hr/ped/kturbo/Op...RIVOLA_FSI.pdf
(linked from the OpenFOAM page http://foamcfd.org/resources/theses.html)

The relative merits of the two solvers were compared:
1. IcoFsiFoam:
+ integrated automesh motion capacity to minimise mesh degenration
- only allows for a domain with 2 regions
- difficult to use(?)
2. IcoStructFoam
+ easy to use do to better architecture(?)
- only handles small structural deformations
- only allows for a domain with 2 regions
Generally:
* OpenFOAM structural equations only handle homogeneous structures
* OpenFOAM's ability to handle large displacements for thin-shell structures is limited by small cell sizes and small timestep sizes hence long computation times

Second OpenFOAM Workshop in Zagreb, Croatia

http://bit.ly/a9JYr8

http://www.openfoamworkshop.org/2007/index.php?title=Coupled_Simulations_and_Fluid-Structure_Interaction

article | discussion | edit | history

WIKKI

Coupled Simulations and Fluid-Structure Interaction

A generic Computational Continuum Mechanics library like OpenFOAM is a natural platform for Fluid-Structure Interaction (FSI): both fluids and structural solvers already exist. Furthermore, doing a simulation in a single software simplifies the operation: there is no need for multi-threaded simulations of software to
…lvers and discretisation methods share the base mesh
… mapping tools are already implemented further

…ed at Imperial College in late 1990-s - but it wasn't easy.
…esh-based field registration, FSI in the new version is
…SI-relevant capabilities and examples of application.

[edit]

…eering Applications: Fluid-Structure Interaction
…dar Karac of University College Dublin Abstract

…CO s.r.l. Abstract Slides
…tructures by Thomas Gallinger of Technical University

…ons of flapping wings at low Reynolds numbers by
…Netherlands Abstract Slides
…Željko Tukovic of University of Zagreb, Croatia

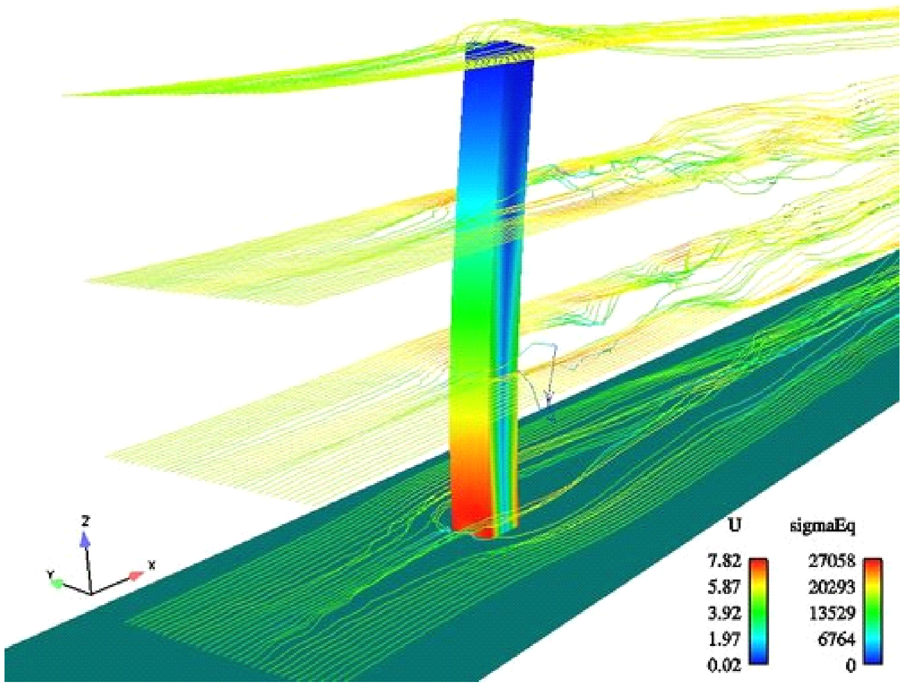
…media by Marianne Mataln of ICE Strömungsforschung

…rithms by Vicente Diaz Casas of Universidade da

…g by Valentine Kanyanta of University College Dublin

FSI results

The FSI solver is tested on the flow past a cantilevered elastic square beam. The frequency of the inlet flow velocity pulsation is equal to the first natural frequency of the beam. The picture below shows streamlines pattern and equivalent Cauchy stress at the beam boundary. This calculation is done for the solid-fluid density ratio 100:1. For lower density ratios loosely coupled algorithm becomes unstable.



icoFsiFoam 情報 CFD Online www.cfd-online.com http://bit.ly/bVzmIs

http://www.cfd-online.com/Forums/openfoam-solving/58588-patch-end-points-mesh-motion-movingwallvelocity.htmlhttp://bit.ly/bVzmIs

March 22, 2008, 02:38 | **Hello Patrick, 1) You have** | #2

hjasak
Senior Member

Hrvoje Jasak
Join Date: Mar 2009
Location: London, England
Posts: 1,580
Rep Power: 11

Hello Patrick,

1) You have a problem with consistency of a boundary condition in mesh motion: a "lower" (eg. slip), will be over-ridden by the "higher" (eg. fixed value).

2) No need: moving wall velocity is an absolutely consistent version of a fixed value velocity. It is done because the code can calculate the wall-normal motion flux better than anything else.

Have a look at the flux field for the boundary - do you have a non-zero flux for it?

3) Yes, we are writing some papers about it. This is the main reason I am reluctant to release the tutorial, 'cause the new code from Zeljko is so much better. This will definitely come out in SVN at some stage, but the Eccomas, Commodia and journal papers are not out yet.

This is what I propose to do: I will give you a tutorial for icoFsiFoam as it stands without checking it in. Please find flappingConsoleSmall_HJ_21Mar2008.tgz in

http://powerlab.fsb.hr/ped/kturbo/OpenFOAM/run/

It runs fo

Enjoy,

Hrv

Hrvoje Ja

powerlab.fsb.hr - /ped/kturbo/OpenFOAM/run/

[To Parent Directory]

20. ožujak 2008	23:53	5835	dropletSplash_HJ_22Nov2007.tgz
21. ožujak 2008	19:32	2233668	flappingConsoleSmall_HJ_21Mar2008.
10. veljača 2006	1:21	81753739	kivaTest_HJ_9Feb2006.tgz
8. srpanj 2006	16:10	12334045	mixer3D_HJ_8Jul2006.tgz
19. veljača 2008	17:43	849490	mixerGgi_HJ_19Feb2008.tgz
30. svibanj 2006	22:55	12021687	pipeInTank.tgz
27. lipanj 2006	15:47	3151504	simpleEngineTutorial.tgz

http://bit.ly/dkDvKS

http://powerlab.fsb.hr/ped/kturbo/OpenFOAM/run/

icoStructFoam 情報

http://bit.ly/ckKlwc

http://openfoamwiki.net/index.php/Contrib_icoStructFoam

page | discussion | view source | history

navigation
- Main Page
- Community portal
- Current events
- Recent changes
- Random page
- FAQ
- Help

search

Go | Search

toolbox
- What links here
- Related changes
- Special pages
- Permanent link
- Browse properties

print/export
- Create a book
- Download as PDF
- Printable version

Contrib icoStructFoam

Valid versions: OF v1.3

Contents [hide]

1 Description

This solver is a hybrid of icoFoam ... is a solid. The pressure of the flu...

This solver is only a demo. It ...

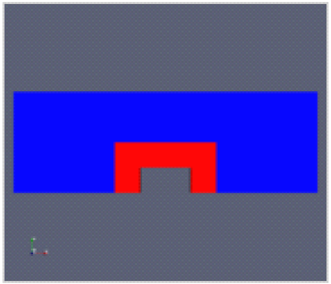
Ah. And by the way: it does not ...

2 Algorithm

At each time-step the following t...

1. the equations in the fluid ...
2. fluid pressures at the fluid ...
3. the equations for the solid ...
4. displacements for the inte... fluid region
5. the grid of the fluid region ...

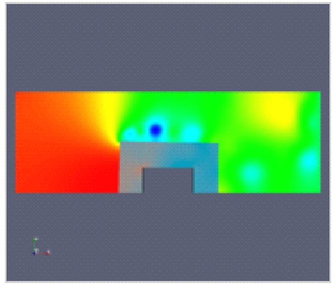
4.2 Soft thing on a stick



The two regions

This case simulates the flow through a channel with an obstacle that consists of two parts: a solid, fixed part around which a deformable material is wrapped.

The two regions are shown in the picture on the left:



Pressure and strain

- Blue: the fluid. On the left-side a velocity inlet, on the right side a pressure outlet. Top is a wall, botto, a symmetry boundary.
- Red: the solid. The bottom edges are symmetry boundaries. The inner edges are fixed (the stick in the title)

The picture on the left is the undeformed geometry at the beginning. The picture on the right is the geometry at the end of the simulation. The solid is colored with the strain, the fluid with the pressure.

In certain parts of Austria this is also known as the Lentos-case.

5 Download

The solver

Case of the deformable channel

Case of the soft thing on a stick

6 History

Category: OpenFOAM Version 1.3

- 10 Nov 2005: Changed version for 1.3

--Bgschaid 13:34, 11 Nov 2005 (CET)

icoStructFoam 情報2

http://bit.ly/cDWBHO

http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2007/

PhD course in CFD with OpenSource software, 2007

Syllabus

The course gives an introduction to the use of OpenSource software for CFD applications. A major project work in OpenFOAM (see the short description below) forms a large part of the course. The project may be defined according to the student's special interests. The result of the project should be a detailed tutorial for a specific application of OpenFOAM. The tutorials will be peer-reviewed and graded by the students, and the tutorials thus form a part of the course. The tutorials will be made available as OpenSource, as a contribution to the OpenFOAM community. To pass the course the student must do the project and peer-review the tutorials from the other projects.

The course homepage is http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2007

Final, peer-reviewed, student-contributed tutorials

These files should at least work for OF-1.4.1 or OF-1.4.1-dev at the student computers in the Mechanical Engineering building at Chalmers, at the time of the third occasion of this course.

- A tutorial on how to use Dynamic Mesh solver IcoDyMFoam, by Pirooz Moradnia:
  Report, Presentation, Case
- Implementing third order compressible flow solver for hexahedral meshes in OpenFoam, by Martin Olausson:
  Report, g3dFoam.tar, shockTube.tar
- icoStructFoam, a Fluid-Structure Interaction Solver, by Philip Evegren:
  Report, Presentation, IcoStructFoam_Rev561.tgz (From openfoam-extend at SourceForge, Revision 561: /trunk/Breeder/solvers/other/IcoStructFoam)
- Different ways to treat rotating geometries, by Olivier Petit:
  Report
- reactingFoam tutorial (simple gas phase reaction), by Andreas Lundstr•:
  Report, Test case
- Free surface tutorial using interFoam and rasInterFoam, by Hassan Hemida:
  Report, Test case, Movie
- Large Eddy Simulation of a Tilt-rotor wing with Active Flow Control, by Mohammad El-Alti:

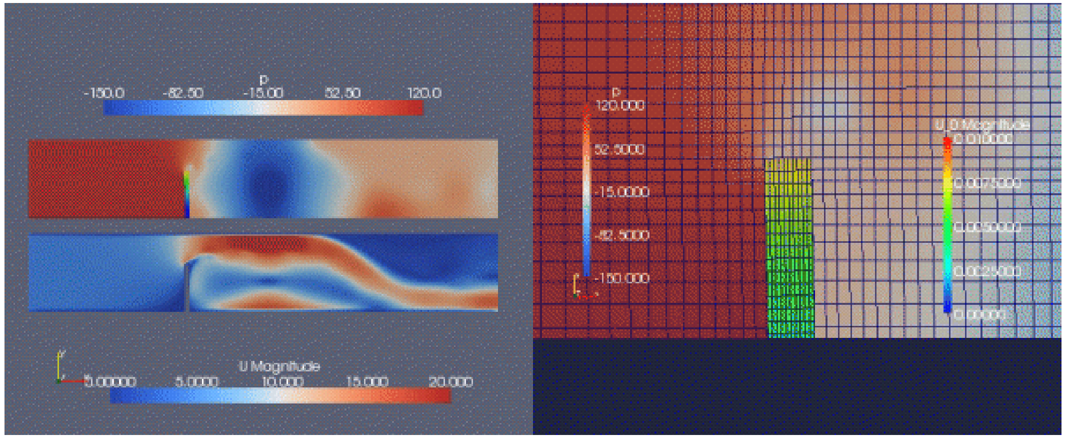
事前調査

3月 8

FSI-flappingConsoleSmall

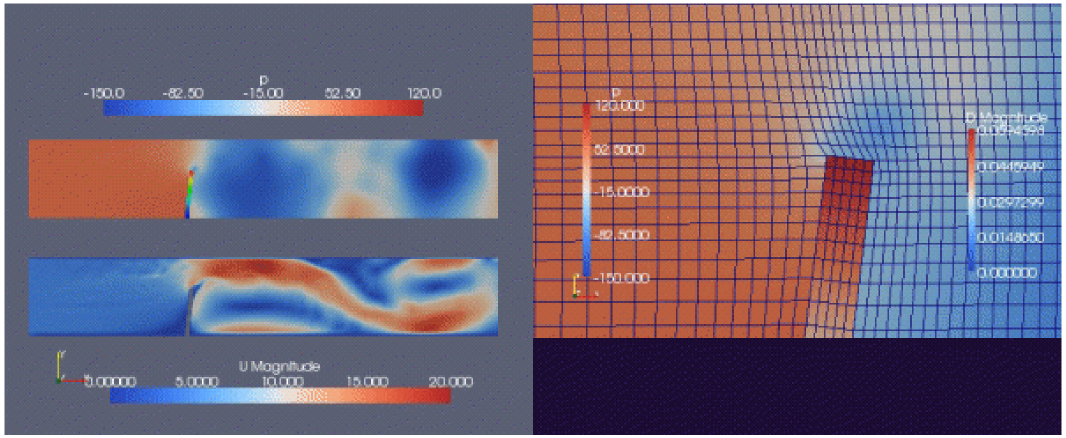
オープンCAE

FSI(Fluid Structure Interface) の、お次のレッスンは、OF-1.5-dev 用のチュートリアル。
まずは、OF-1.5-devで計算したもの。モデルは、
http://powerlab.fsb.hr/ped/kturbo/OpenFOAM/run/flappingConsoleSmall_HJ_21Mar2008.tgz より入手したものをそのまま使用。



左の図は全体像で、上段が圧力コンタ図、下段が流速コンタ図。流入口から全体流路の1/3ほどの位置に、片持ち梁が設置してあり、これが流体の流れを受けて、変形・振動している。右側の図は、この片持ち梁の先端部分を拡大したもの。いずれもクリックすると動画で表示されるが、梁が振動している様子がよくわかる。これは、某原子力発電所のナトリウム漏洩事故の原因になった現象なんだろうな。。。と彷彿させるものがあります。

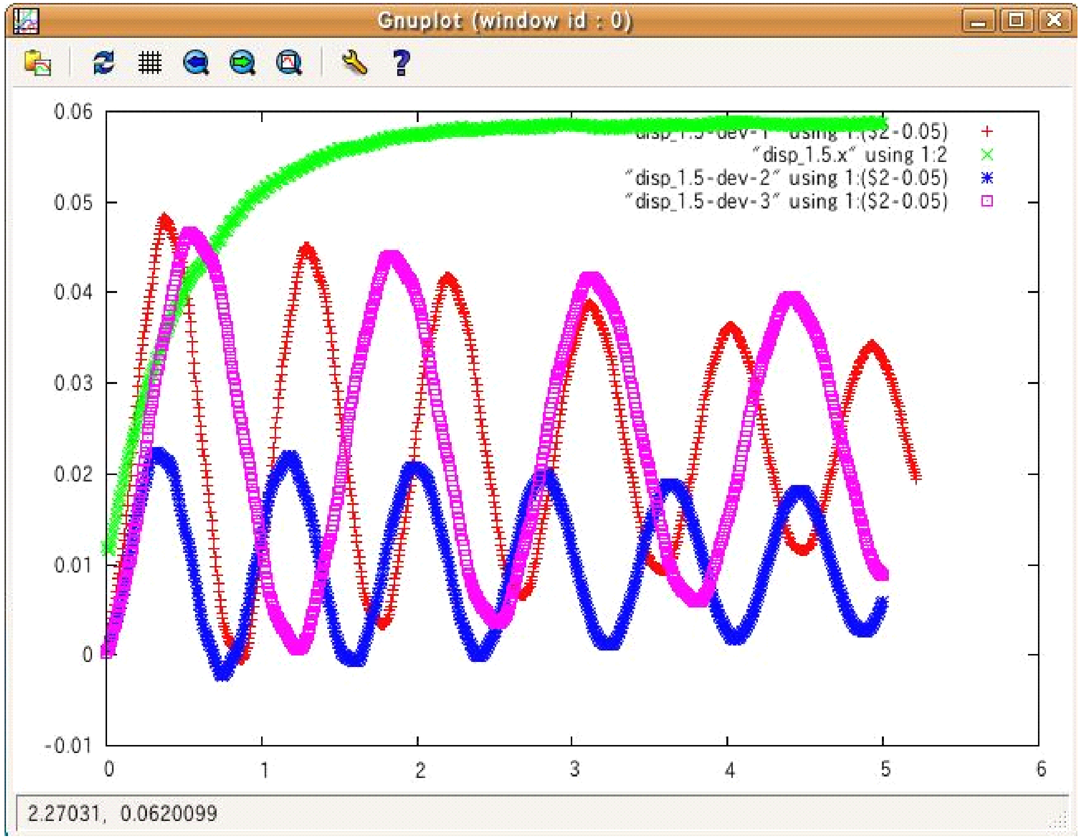
お次は、OF-1.5.xで、icoStructFoam で計算したもの。基本的には、上で使ったモデルをそのまま転用しましたが、そのままでは使えない部分を一部手直ししました。



問題は、、、1.5-devのicoFsiFoam でやった結果とずいぶん異なるということです。

http://bit.ly/djScCr

http://mogura7.zenno.info/~et/wordpress/tag/fsi/



赤のグラフがOpenFOAM-1.5-devのicoFsiFoam を使って計算したものであるのに対し、

緑は、OpenFOAM-1.5.xに追加したicoStructFoam を使って計算したもの。

アニメーションで見られた、振動の有無の違いがよくわかる。

今回、追加して計算したものが2つ。

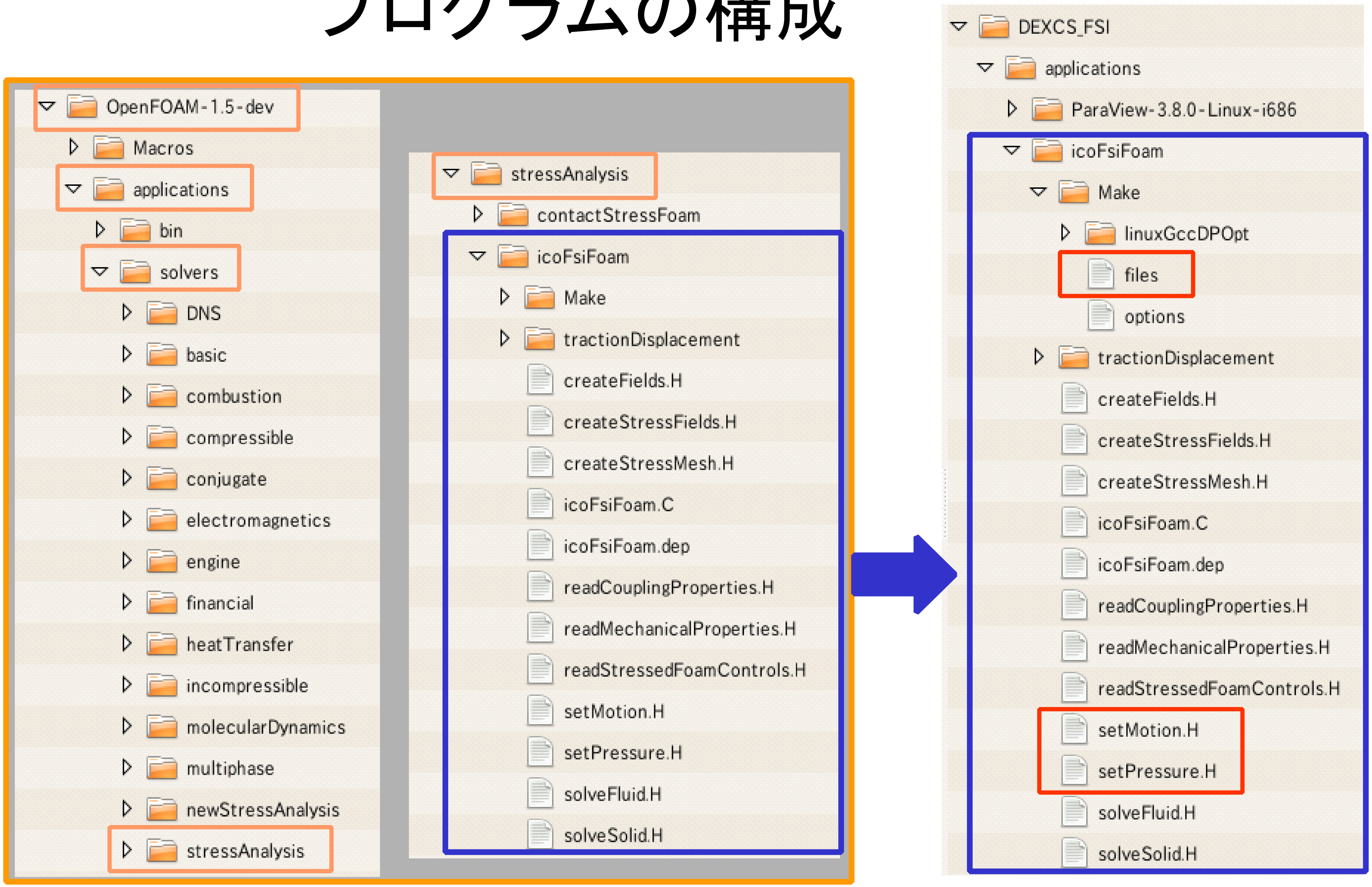
濃青・・・固体部分のメッシュを、icoStructFoamで使ったものと同等にして、icoFsiFoamにて計算

オレンジ・・・icoFsiFoam(赤)に対し固体の密度を2倍にして計算

これらの挙動変化は、およそ予想通りの結果になってくれていたので、、、結論は、**FSIには、OpenFOAM-1.5-devのicoFsiFoamを使いなさい**ということのようです。

プログラム（icoFsiFoam）の説明

プログラムの構成



DEXCS-FSI版では、一部 を改変

改変内容

http://bit.ly/c7OMam

http://www.cfd-online.com/Forums/openfoam-solving/58513-fluid-structure-interaction-using-icofsifoam-problems-3.html#post267620

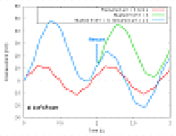
July 16, 2010, 18:25 **icoFsiFoam problem with restarted simulations** #55

7islands
Senior Member

Takuya OSHIMA
Join Date: Mar 2009
Location: Niigata City, Japan
Posts: 439
Blog Entries: 1
Rep Power: 5

Hi,
For future interests in doing FSI simulations, we are taking a look at icoFsiFoam in OF 1.5-dev in conjunction with the flappingConsole case (flappingConsoleSmall_HJ_21Mar2008.tgz), which includes several steps of saved field data up to t = 0.024 [s] and the simulation is setup to start from the latestTime. However, we are having problem in restarted simulations.

Here I attach a figure where the x-directional displacements at the top of the console are plotted for three cases: the red line is a restarted simulation from the saved data at t = 0.024 [s] (which is how one would usually run icoFsiFoam), the green line is a fresh simulation started from t = 0 by deleting the saved data, and the blue line is also a freshly started simulation but stopped at t = 1 [s] and restarted from there. There's no wonder in the difference between the red and the green/blue lines if the saved data and our data are calculated by e.g. different solvers, but what was problem to us was the discrepancy between the green and the red lines after t = 1 [s].



The main cause of the discrepancy, besides other unsignificant ones, was sorted out to be the patch-to-patch interpolation weights between the fluid and the solid moving meshes not being updated per time step basis, hence the interpolation weights calculated at the time of simulation (re)start were constantly used during the simulation run regardless of the mesh movements. So here is what we tried to correct the issue:

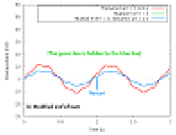
Code:

```
Index: applications/solvers/stressAnalysis/icoFsiFoam/setMotion.H
===================================================================
--- applications/solvers/stressAnalysis/icoFsiFoam/setMotion.H  (revision 1790)
+++ applications/solvers/stressAnalysis/icoFsiFoam/setMotion.H  (working copy)
@@ -11,6 +11,7 @@
     stressMesh.movePoints(newPoints);


+    interpolatorSolidFluid.movePoints();
     vectorField fluidPatchPointsDispl =
         interpolatorSolidFluid.pointInterpolate
         (
Index: applications/solvers/stressAnalysis/icoFsiFoam/setPressure.H
===================================================================
```

改変内容(つづき)

```
================================================================================
--- applications/solvers/stressAnalysis/icoFsiFoam/setPressure.H       (revision 1790)
+++ applications/solvers/stressAnalysis/icoFsiFoam/setPressure.H       (working copy)
@@ -2,6 +2,7 @@
     // Setting pressure on solid patch
     Info << "Setting pressure" << endl;

+    interpolatorFluidSolid.movePoints();
     scalarField solidPatchPressure =
         interpolatorFluidSolid.faceInterpolate
         (
```

And the results with the patched icoFsiFoam are:

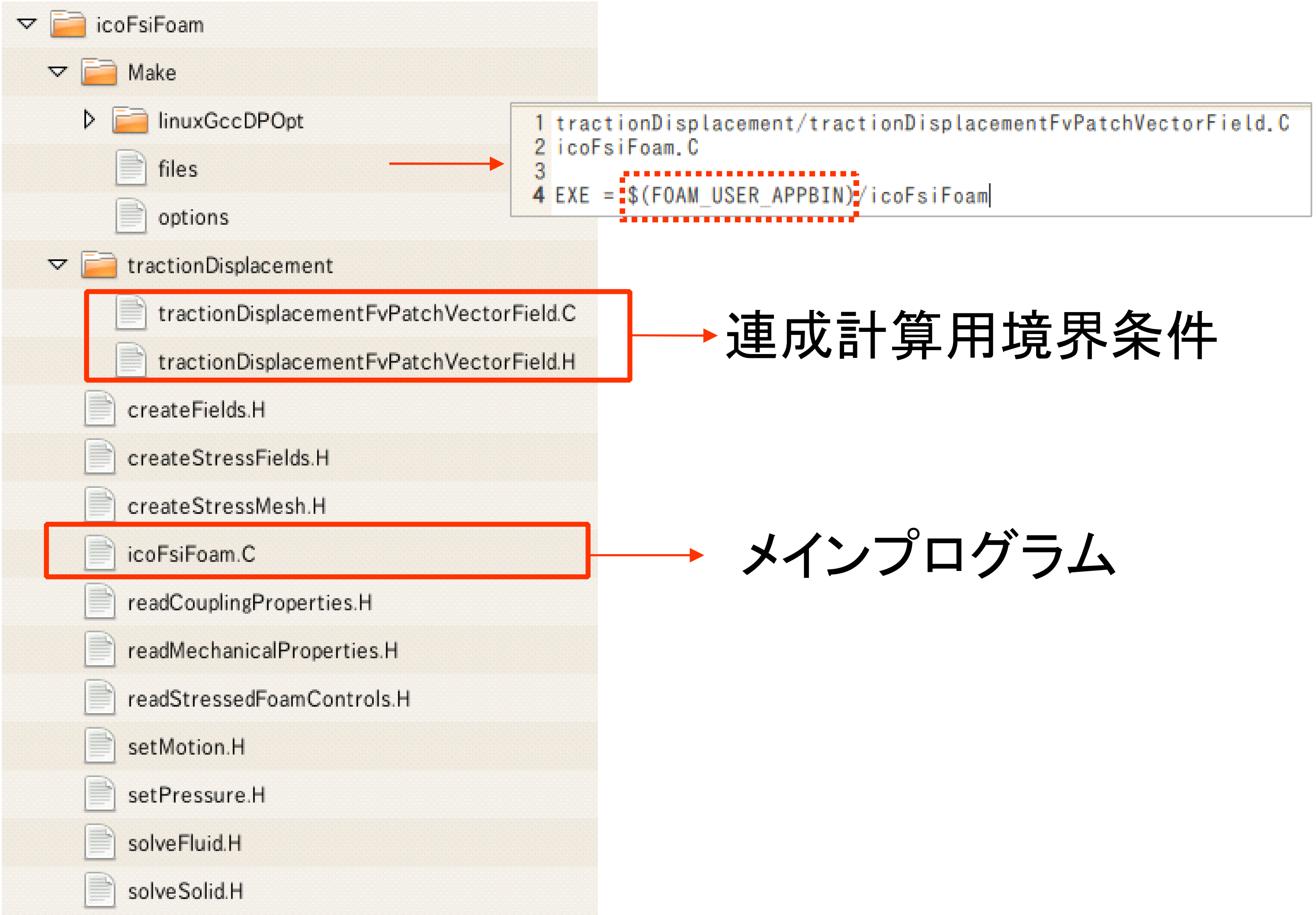


This time the green and the blue line agrees perfectly (up to around four significant digits) and the oscillating frequencies of all cases as well. Besides, combined with the results of other runs we have a general impression that the stability is also better. However, the displacements changed too drastically, to around an order of a magnitude smaller compared to those with the unpatched icoFsiFoam. Thus we are unsure if the omission of movePoints() as shown in the patch is a bug or intended for some reasons.

We'd appreciate any inputs, thoughts, comments. Thanks!

Takuya

プログラムの概要


icoFsiFoam
Make
linuxGccDPOpt
files
options
1 tractionDisplacement/tractionDisplacementFvPatchVectorField.C
2 icoFsiFoam.C
3
4 EXE = $(FOAM_USER_APPBIN)/icoFsiFoam
tractionDisplacement
tractionDisplacementFvPatchVectorField.C
tractionDisplacementFvPatchVectorField.H
連成計算用境界条件
createFields.H
createStressFields.H
createStressMesh.H
icoFsiFoam.C
メインプログラム
readCouplingProperties.H
readMechanicalProperties.H
readStressedFoamControls.H
setMotion.H
setPressure.H
solveFluid.H
solveSolid.H

icoFsiFoam.C

メインプログラムの概要(1)

```
#include "fvCFD.H"
#include "dynamicFvMesh.H"
#include "patchToPatchInterpolation.H"
#include "tractionDisplacement/tractionDisplacementFvPatchVectorField.H"
#include "tetFemMatrices.H"
#include "tetPointFields.H"
#include "faceTetPolyPatch.H"
#include "tetPolyPatchInterpolation.H"
#include "fixedValueTetPolyPatchFields.H"

#include "pointMesh.H"
#include "pointFields.H"
#include "volPointInterpolation.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *

int main(int argc, char *argv[])
{
#   include "setRootCase.H"
#   include "createTime.H"
#   include "createDynamicFvMesh.H"
#   include "createStressMesh.H"
#   include "createFields.H"
#   include "createStressFields.H"
#   include "readMechanicalProperties.H"
#   include "readCouplingProperties.H"
#   include "readTimeControls.H"

#   include "initContinuityErrs.H"

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
```

FSI用境界条件
tractionDisplacement

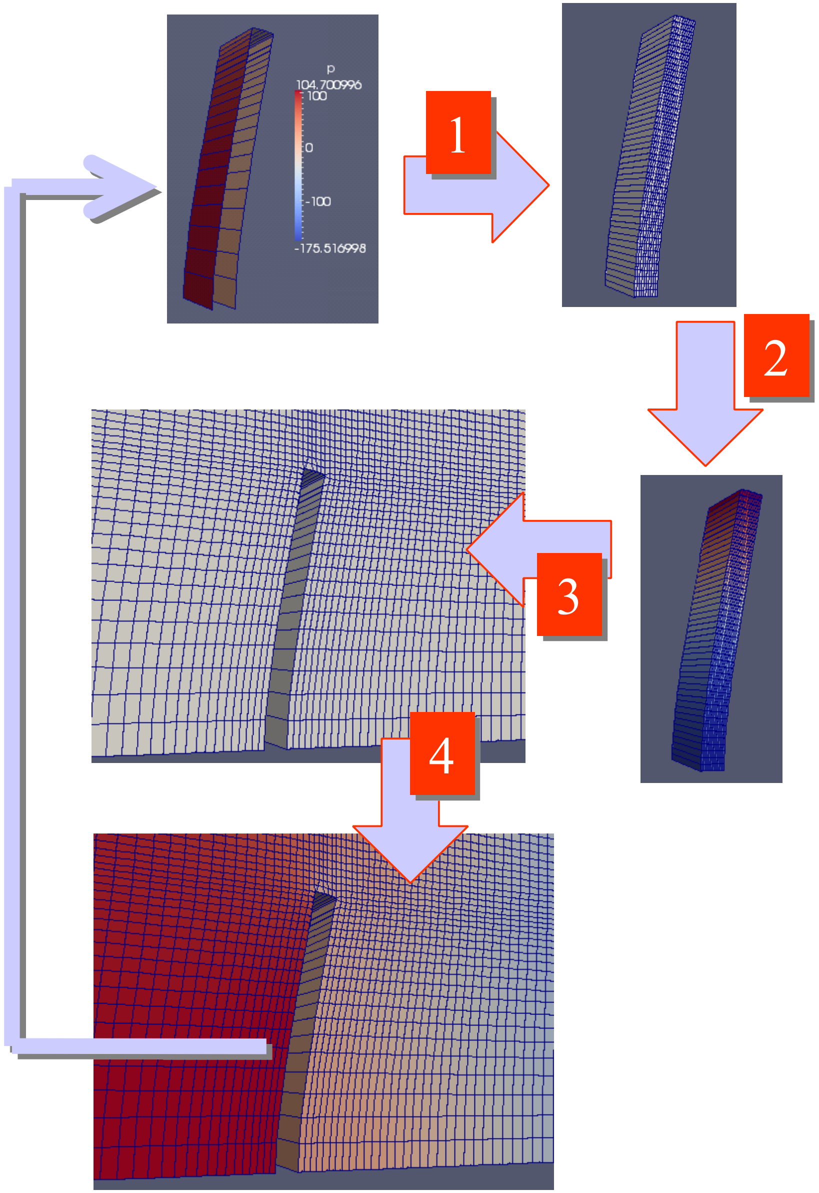
流体、固体部で、
各々メッシュ作成
初期場設定

icoFsiFoam用
カスタマイズ部分

icoFsiFoam.C

メインプログラムの概要(2)

```
    Info<< "\nStarting time loop\n" << endl;

    while (runTime.run())
    {
#       include "readPISOControls.H"
#       include "readTimeControls.H"
#       include "CourantNo.H"
#       include "setDeltaT.H"

        runTime++;

        Info<< "Time = " << runTime.timeNa  << nl << endl;

#       include "setPressure.H"
#       include "solveSolid.H"

#       include "setMotion.H"
#       include "solveFluid.H"

        runTime.write();

        Info<< "ExecutionTime = "
            << runTime.elapsedCpuTime()
            << " s\n\n" << endl;
    }

    Info<< "End\n" << endl;

    return(0);
}
```



setPressure.H

setPressure

```
{
    // Setting pressure on solid patch
    Info << "Setting pressure" << endl;

    interpolatorFluidSolid.movePoints();
    scalarField solidPatchPressure =
        interpolatorFluidSolid.faceInterpolate
        (
            p.boundaryField()[fluidPatchID]
        );

    solidPatchPressure *= rhoFluid.value();

    tForce.pressure() = solidPatchPressure;


    vector totalPressureForce =
        sum
        (
            p.boundaryField()[fluidPatchID]*
            mesh.Sf().boundaryField()[fluidPatchID]
        );


    Info << "Total pressure force = " << totalPressureForce << endl;
}
```

オリジナル
改変(追加)部分

solveSolid.H

solveSolid

```
#   include "readStressedFoamControls.H"

    int iCorr = 0;
    scalar initialResidual = 0;

    do
    {
        volTensorField gradU = fvc::grad(Usolid);

        fvVectorMatrix UEqn
        (
            fvm::d2dt2(Usolid)
         ==
            fvm::laplacian(2*mu + lambda, Usolid, "laplacian(DU,U)")
          + fvc::div
            (
                mu*gradU.T() + lambda*(I*tr(gradU)) - (mu + lambda)*gradU,
                "div(sigma)"
            )
        );

        initialResidual = UEqn.solve().initialResidual();

    } while (initialResidual > convergenceTolerance && ++iCorr < nCorr);
}
```

solidDisplacementFoam.C　とほぼ同じ

setMotion.H

setMotion

```
    // Setting mesh motion

    pointVectorField solidPointsDispl =
        cpi.interpolate(Usolid - Usolid.oldTime());

    vectorField newPoints =
        stressMesh.points()
      + solidPointsDispl.internalField();

    stressMesh.movePoints(newPoints);

    interpolatorSolidFluid.movePoints();

    vectorField fluidPatchPointsDispl =
        interpolatorSolidFluid.pointInterpolate
        (
            solidPointsDispl.boundaryField()[solidPatchID].
            patchInternalField()
        );

    motionUFluidPatch ==
        tppi.pointToPointInterpolate
        (
            fluidPatchPointsDispl/runTime.deltaT().value()
        );

    mesh.update();

#   include "volContinuity.H"

    Info << "Motion magnitude: mean = "
        << average(mag(Usolid.boundaryField()[solidPatchID]))
        << " max = "
        << max(mag(Usolid.boundaryField()[solidPatchID])) << endl;
}
```

オリジナル
改変(追加)部分

solveFluid.H

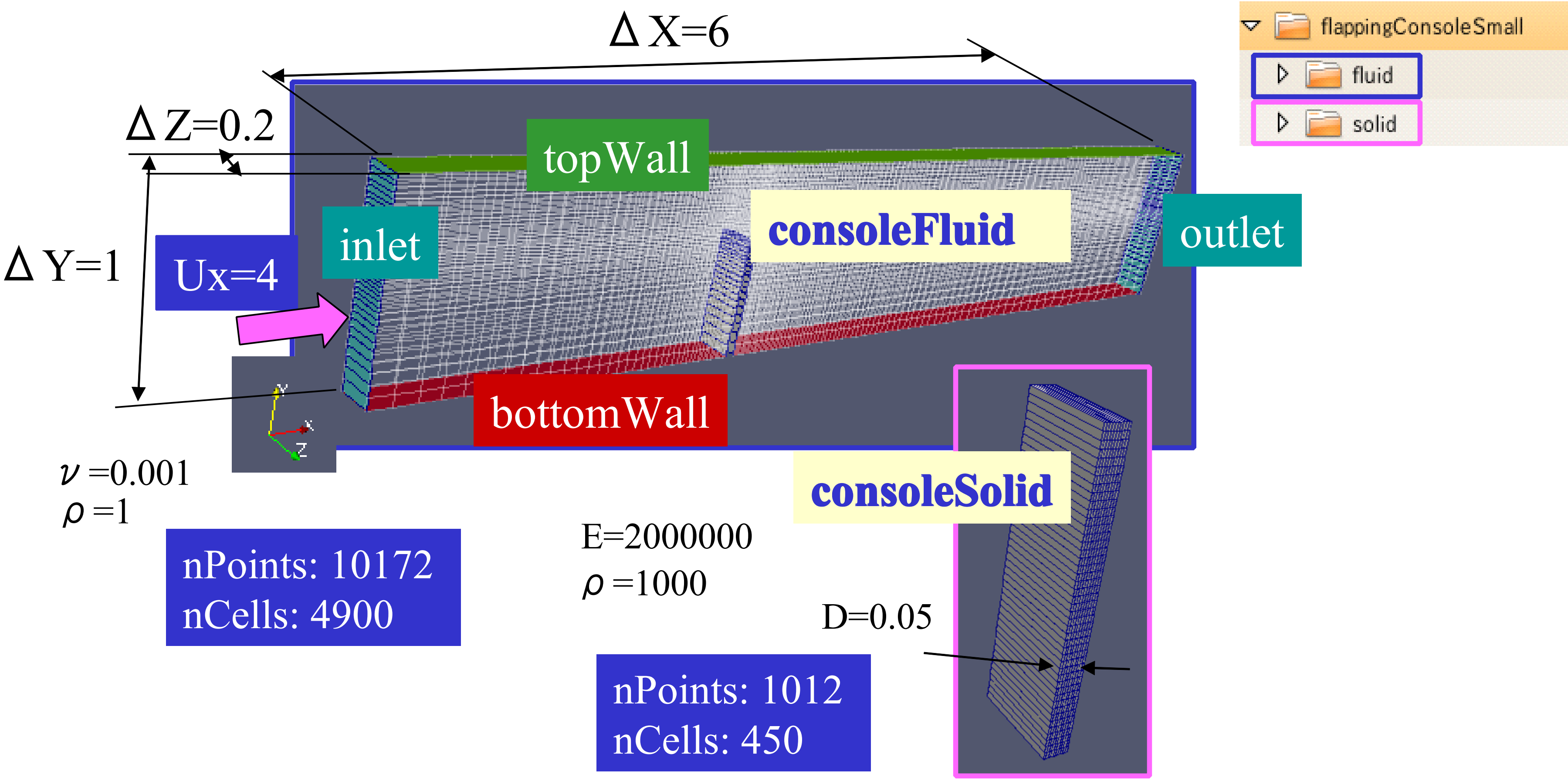
solveFluid

```
    // SIMPLE loop

    for (int corr=0; corr<nCorr; corr++)
    {
        fvVectorMatrix UEqn
        (
            fvm::ddt(U)
          + fvm::div(phi, U)
          - fvm::laplacian(nu, U)
        );

        UEqn.relax();

        solve(UEqn == -fvc::grad(p));

        U = UEqn.H()/UEqn.A();
        U.correctBoundaryConditions();

        adjustPhi(phi, U, p);

        phi = fvc::interpolate(U) & mesh.Sf();

        p.storePrevIter();

        for (int nonOrth=0; nonOrth<=nNonOrthCorr; nonOrth++)
        {
            fvScalarMatrix pEqn
            (
                fvm::laplacian(1.0/UEqn.A(), p) == fvc::div(phi)
            );

            pEqn.setReference(pRefCell, pRefValue);
            pEqn.solve();

            if (nonOrth == nNonOrthCorr)
            {
                phi -= pEqn.flux();
            }
        }

        p.relax();

#       include "movingMeshContinuityErrs.H"

        U -= fvc::grad(p)/UEqn.A();
        U.correctBoundaryConditions();

        // Make the fluxes relative
        phi -= fvc::meshPhi(U);
    }
```

icoFoam.C
（PISO loop）
↓ (?)
SIMPLE loop

公開ケースの説明

http://powerlab.fsb.hr/ped/kturbo/OpenFOAM/run/flappingConsoleSmall_HJ_21Mar2008.tgz



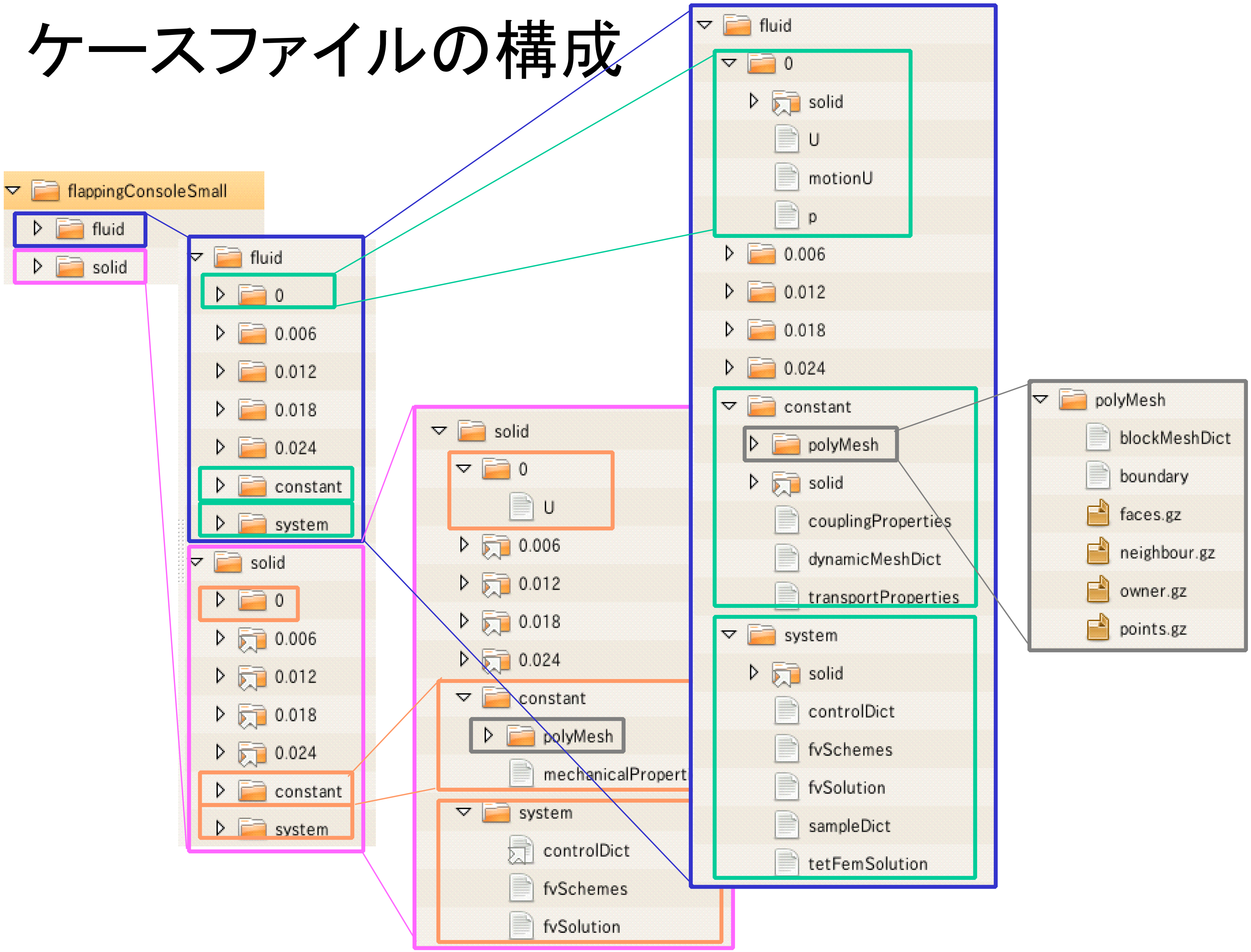
ν=0.001
ρ=1

nPoints: 10172
nCells: 4900

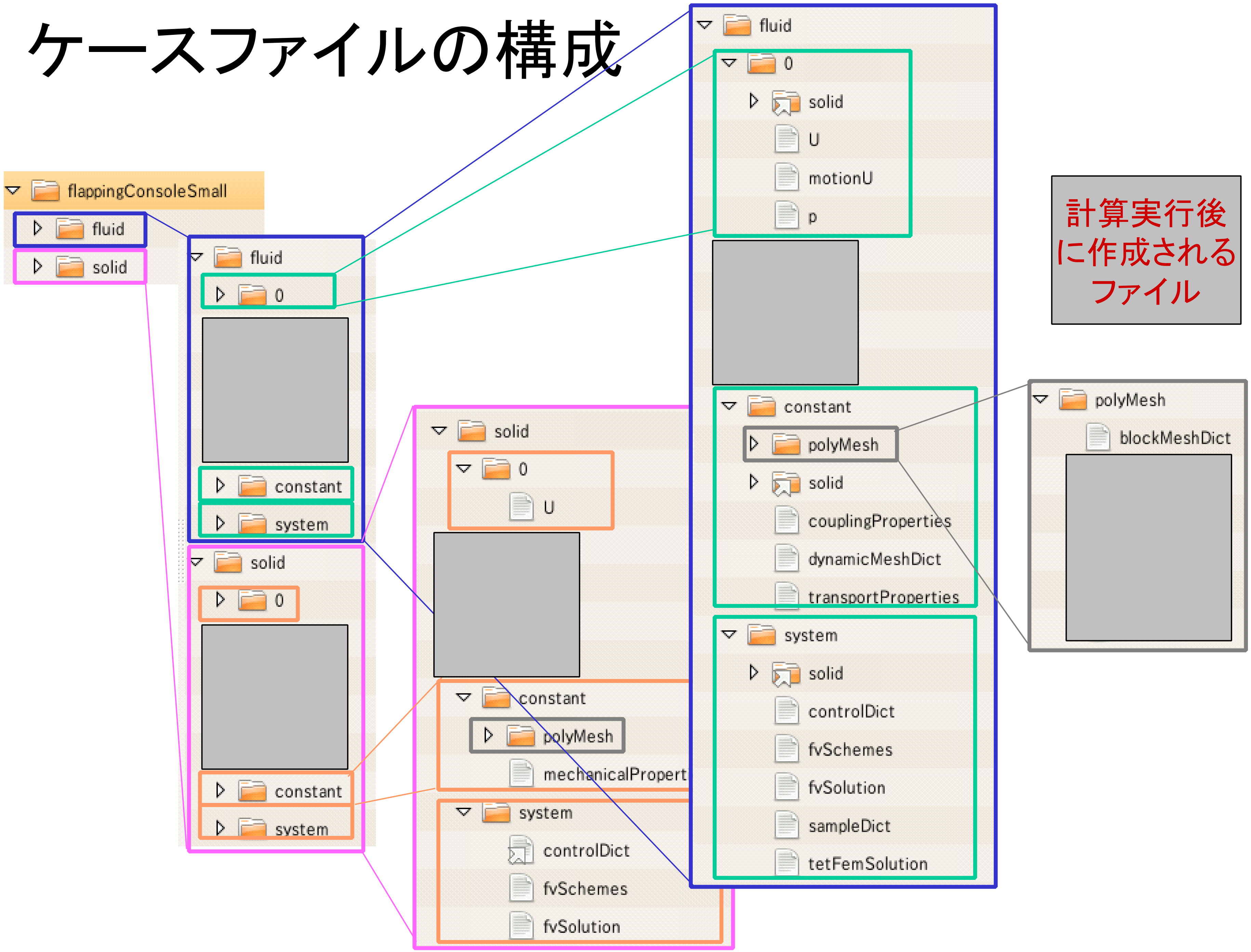
E=2000000
ρ=1000

nPoints: 1012
nCells: 450

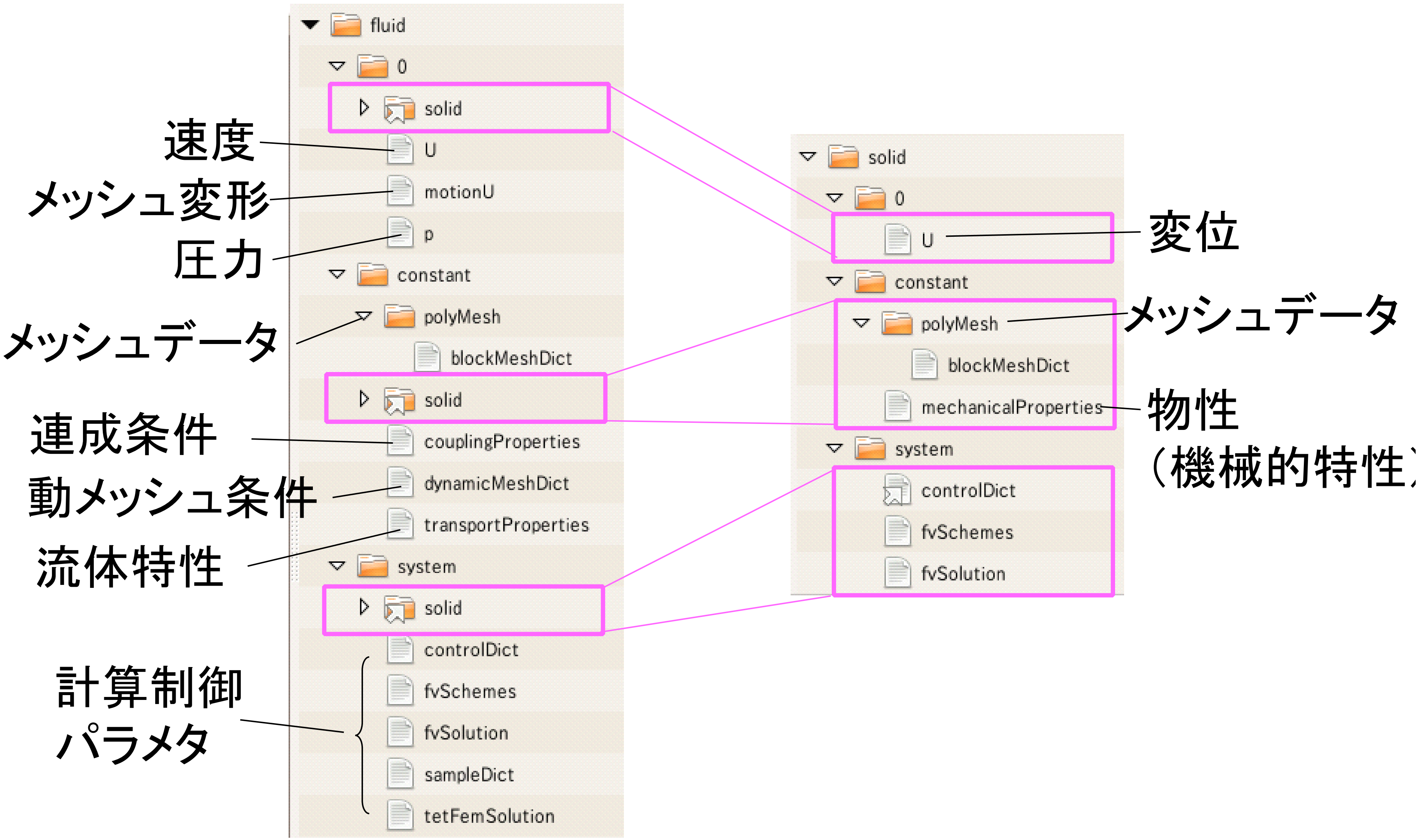
ケースファイルの構成


flappingConsoleSmall
fluid
solid
fluid
0
0.006
0.012
0.018
0.024
constant
system
solid
0
0.006
0.012
0.018
0.024
constant
system
solid
0
U
0.006
0.012
0.018
0.024
constant
polyMesh
mechanicalPropert
system
controlDict
fvSchemes
fvSolution
fluid
0
solid
U
motionU
p
0.006
0.012
0.018
0.024
constant
polyMesh
solid
couplingProperties
dynamicMeshDict
transportProperties
system
solid
controlDict
fvSchemes
fvSolution
sampleDict
tetFemSolution
polyMesh
blockMeshDict
boundary
faces.gz
neighbour.gz
owner.gz
points.gz

ケースファイルの構成

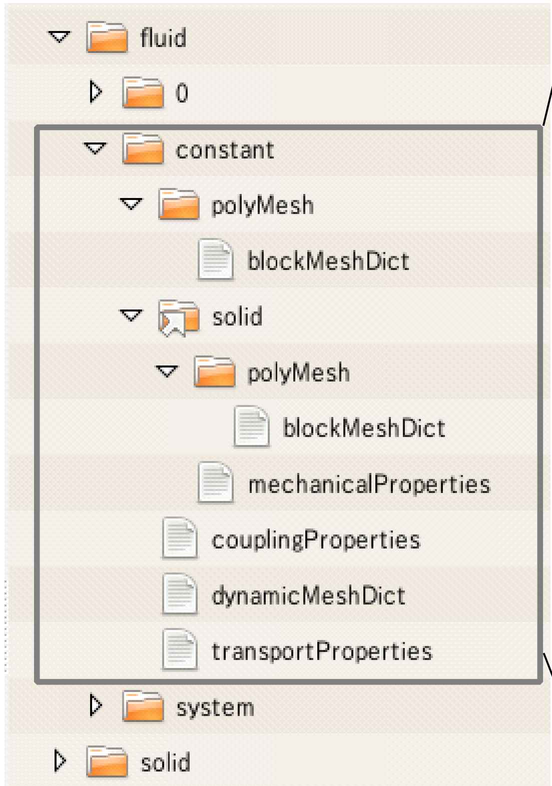

flappingConsoleSmall
fluid
solid
fluid
0
constant
system
solid
0
constant
system
solid
0
U
constant
polyMesh
mechanicalPropert
system
controlDict
fvSchemes
fvSolution
fluid
0
solid
U
motionU
p
constant
polyMesh
solid
couplingProperties
dynamicMeshDict
transportProperties
system
solid
controlDict
fvSchemes
fvSolution
sampleDict
tetFemSolution
polyMesh
blockMeshDict
計算実行後
に作成される
ファイル

ケースファイルの構成（計算に必要なデータ）

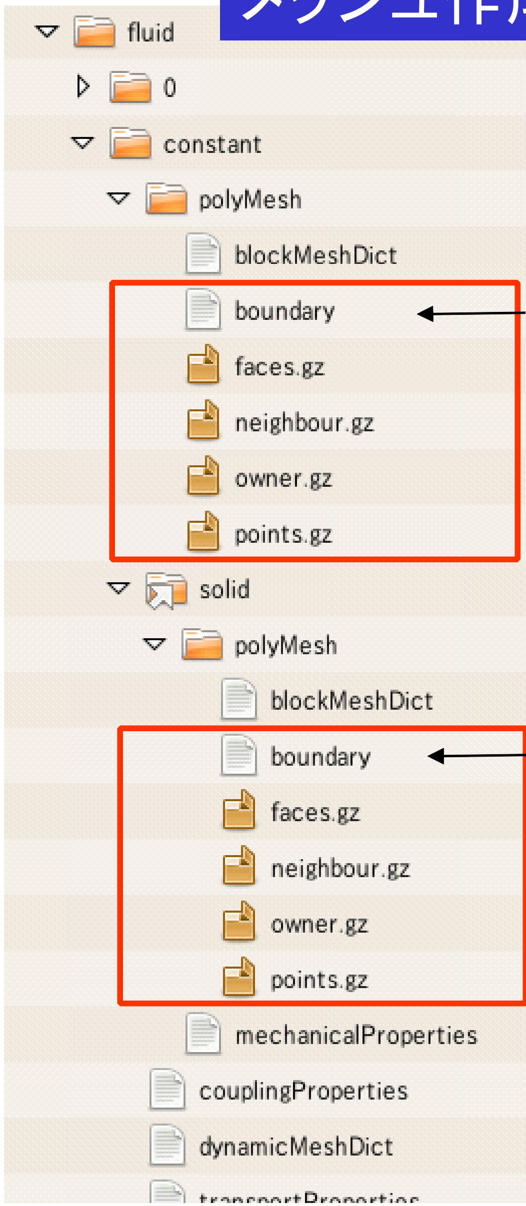


ケースファイルの構成（メッシュ作成時点）

メッシュ作成前



メッシュ作成後



流体部

境界名定義ファイル

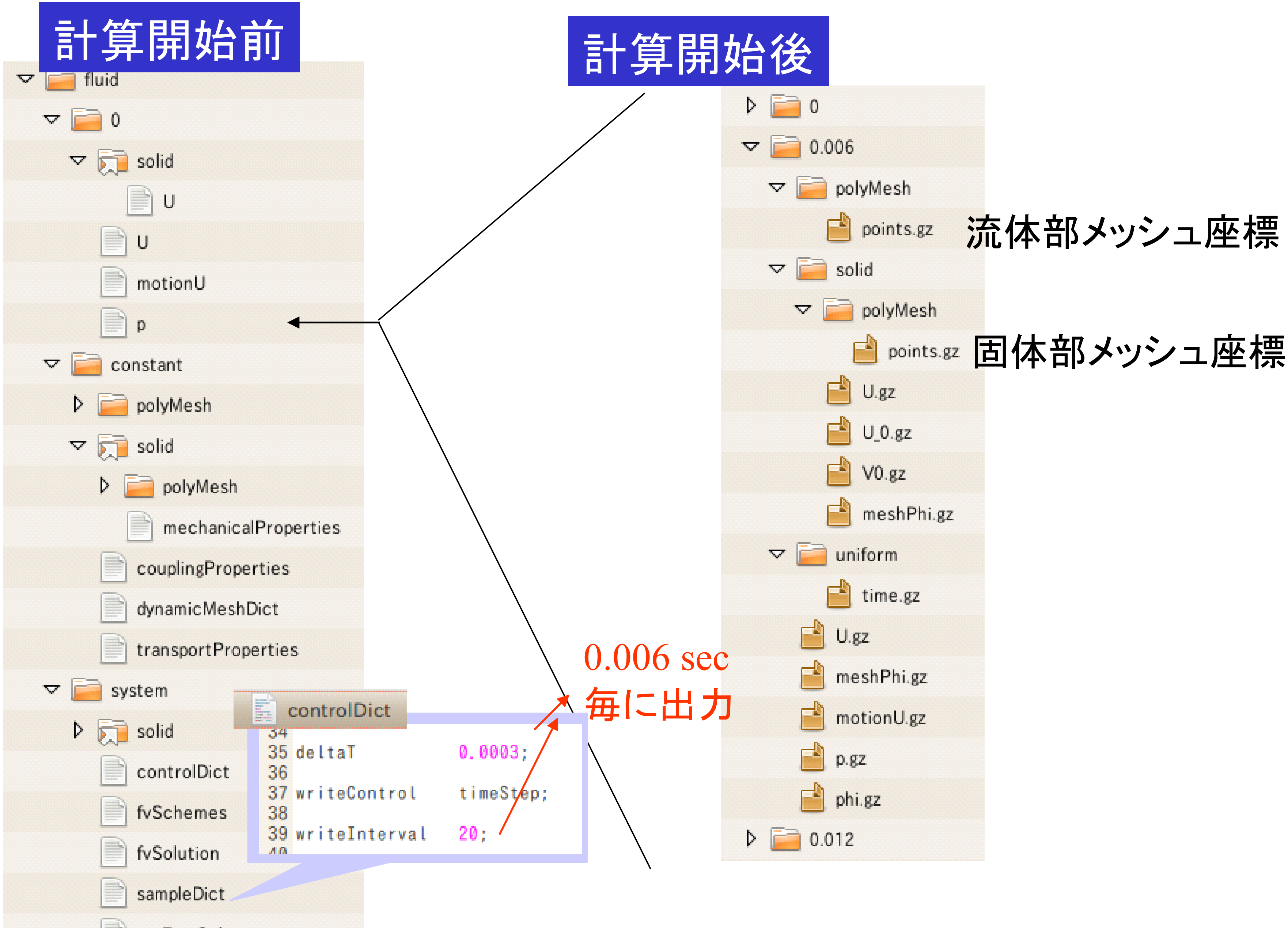
メッシュ定義データ

固体部

境界名定義ファイル

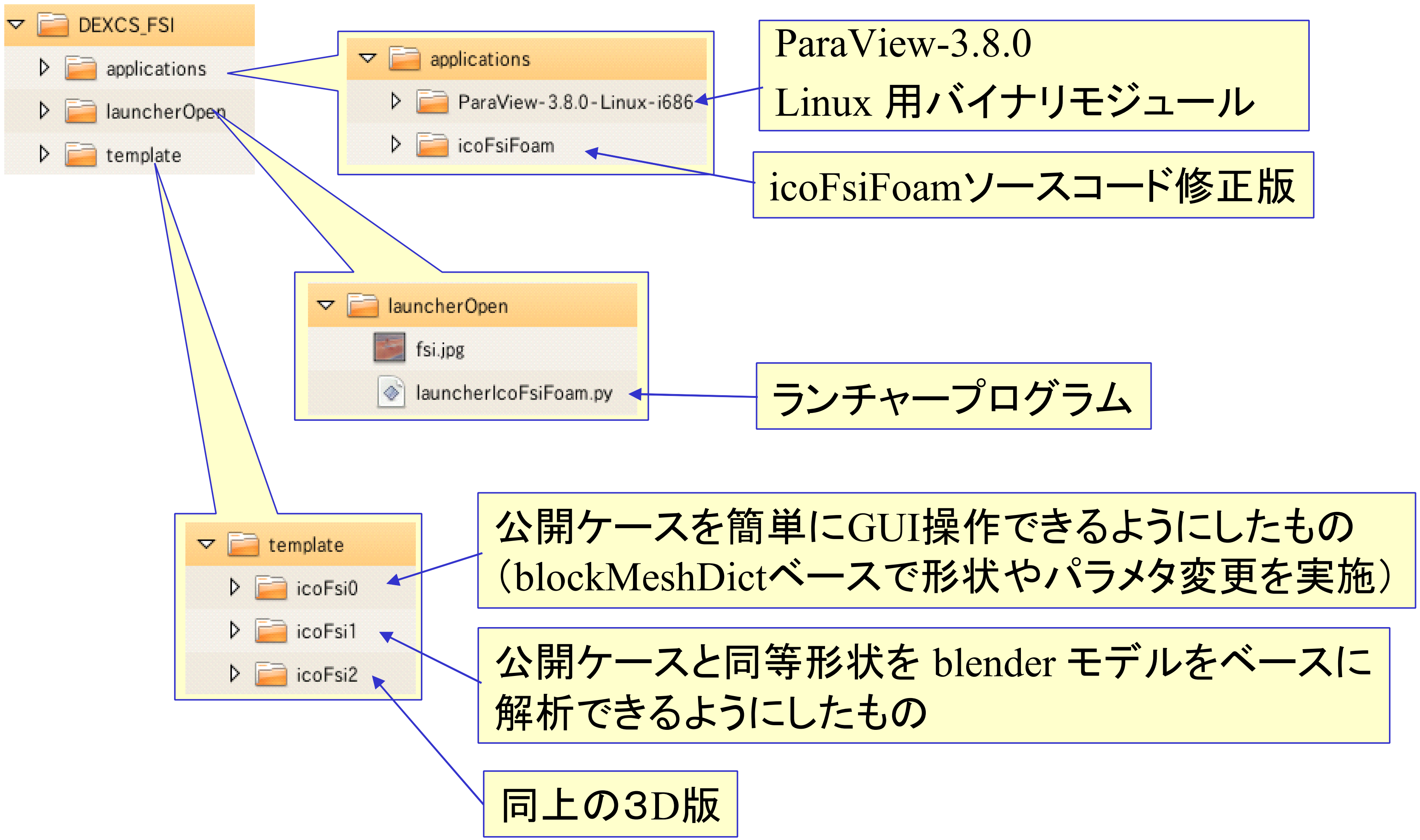
メッシュ定義データ

ケースファイルの構成（計算開始後）

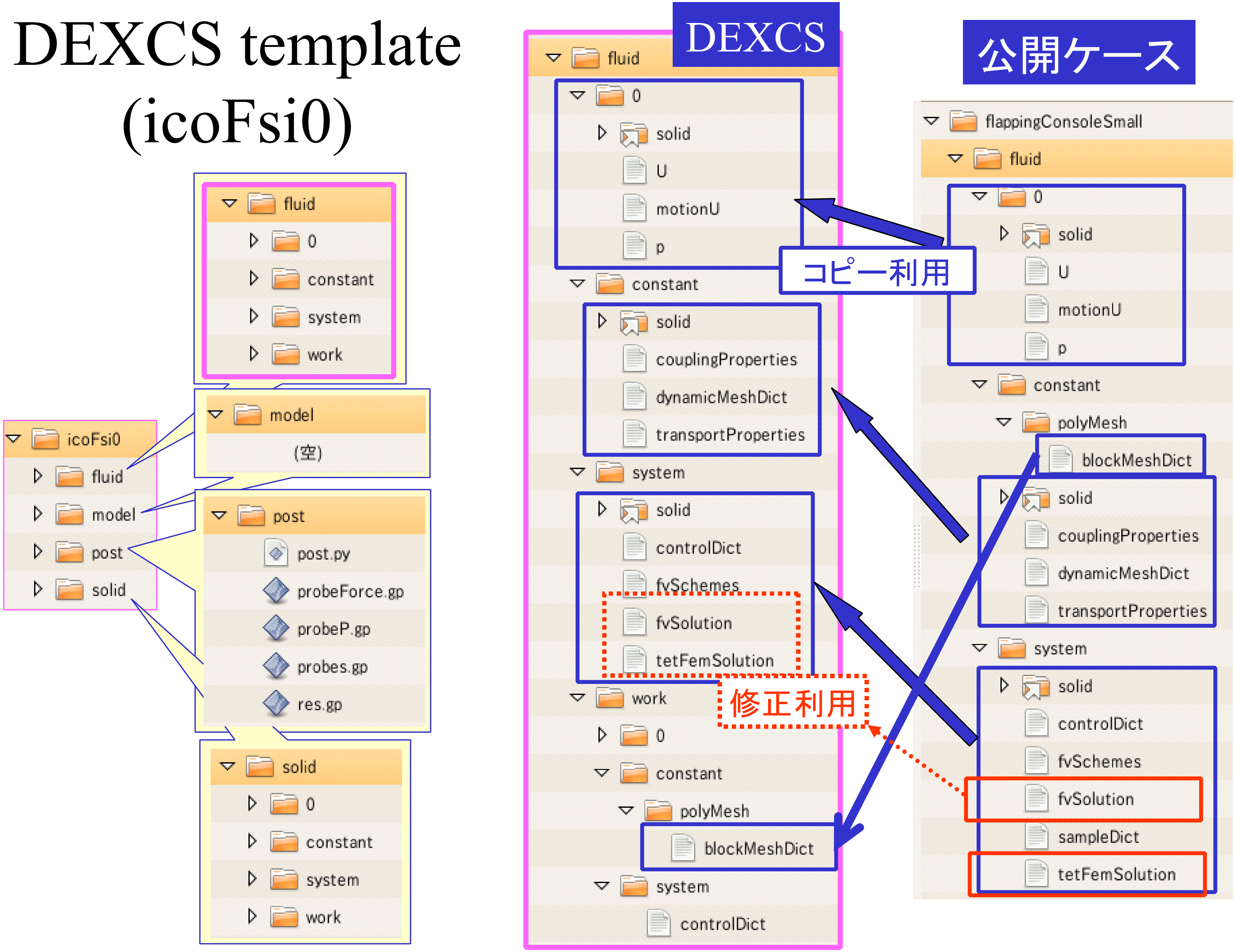


DEXCS_FSI　ランチャーの説明

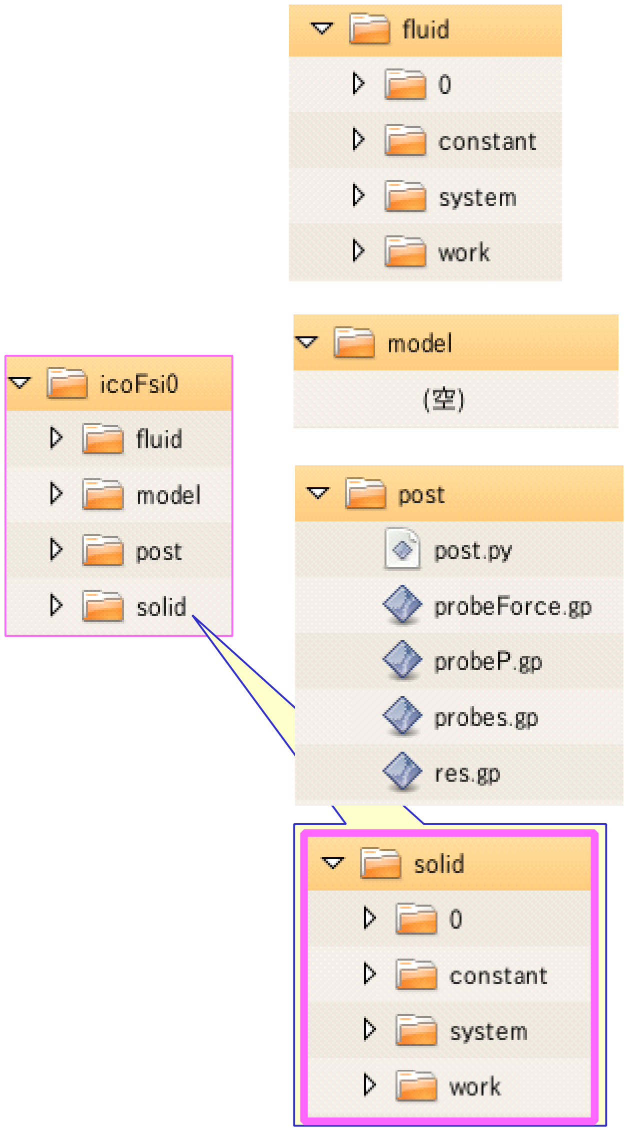
DEXCS-FSI　のファイル構成



DEXCS template (icoFsi0)

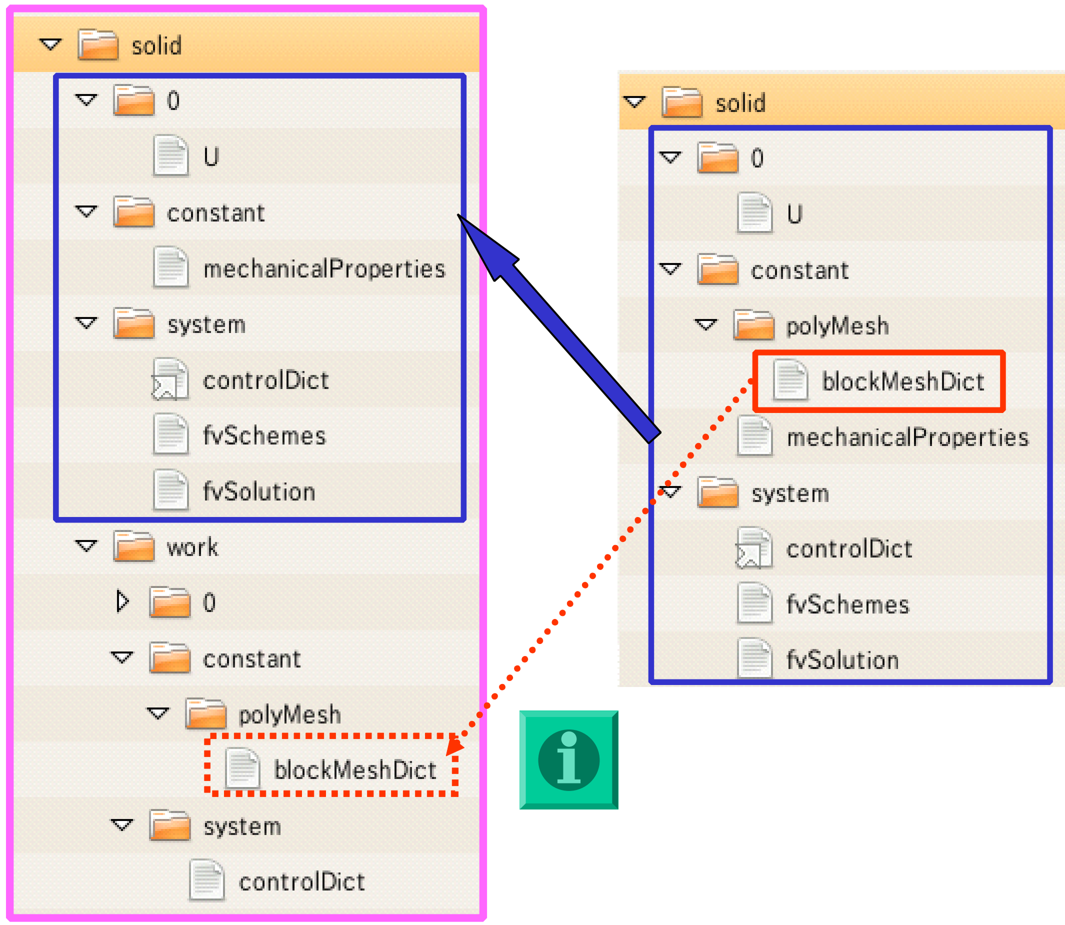


DEXCS template (icoFsi0)



DEXCS

公開ケース

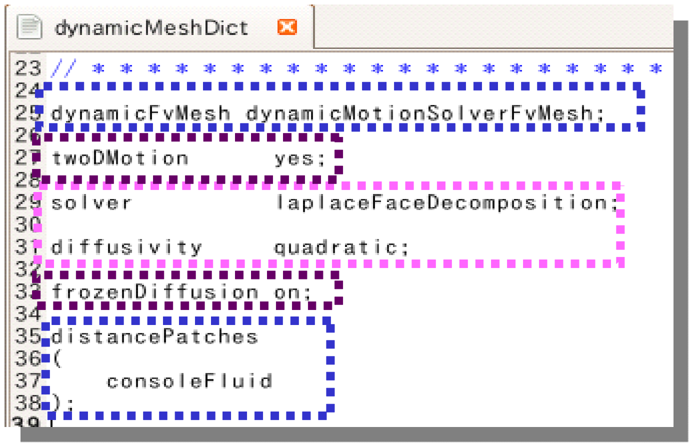


公開ケースのファイル構造を基本的に踏襲
但し、正しく動作しない部分（赤枠部）は修正を実施
workフォルダ下にてメッシュ作成

DEXCS_FSI ランチャーによる解析実行（標準モデルの場合）

公開ケースで使用しているパラメタ		
	そのままでは使えず、変更した	
	そのまま使うしかなさそう	
	他にも選択の余地がありそう	
	ケーススタディ、検証対象になる	
DEXCS固有のカスタマイズパラメタ		

（パラメタ説明図の例）

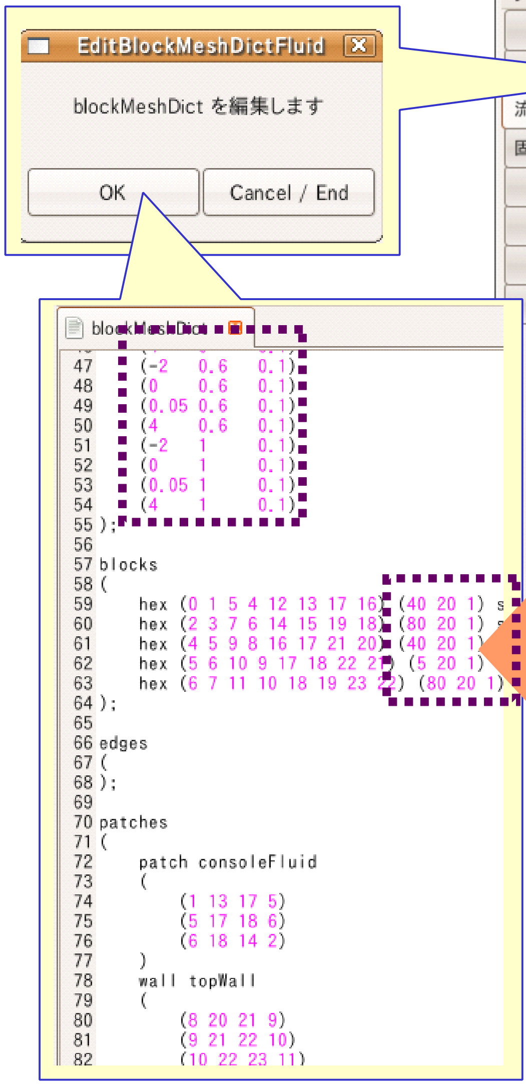


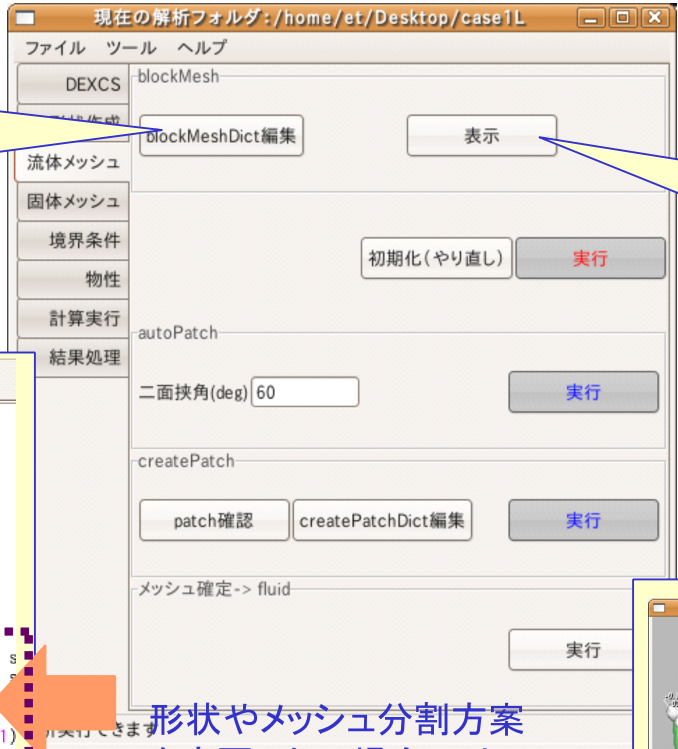
解析フォルダの設定

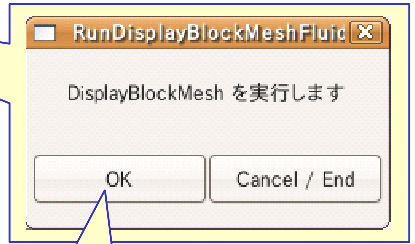

現在の解析フォルダ:/home/et/Desktop/case1L
ファイル ツール ヘルプ
DEXCS
形状作成
流体メッシュ
固体メッシュ
境界条件
物性
計算実行
結果処理
DEXCS icoFsiFoam
DEXCS --- Simple, Just Use It, Have a Fun! ---
解析実行できます

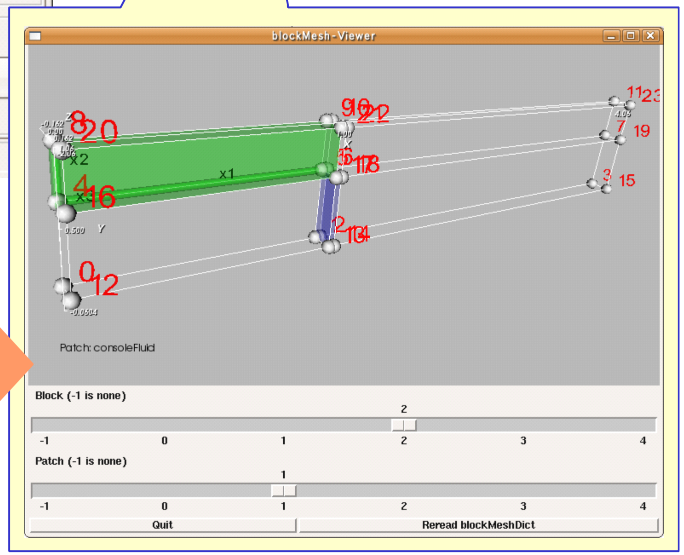

ファイル ツール ヘルプ
解析フォルダ新規作成
解析フォルダを開く
終了
流体メッシュ
固体メッシュ
case0
fluid
model
post
solid
DEXCS_FSI
applications
launcherOpen
template
icoFsi0
fluid
model
post
solid
icoFsi1
icoFsi2
新規作成すると、
テンプレートファイルが
自動でコピーされる
Change Directory
解析フォルダを
/home/et/Desktop/case0
に設定しました。
OK(O)

流体メッシュ　　blockMeshDictの確認と編集







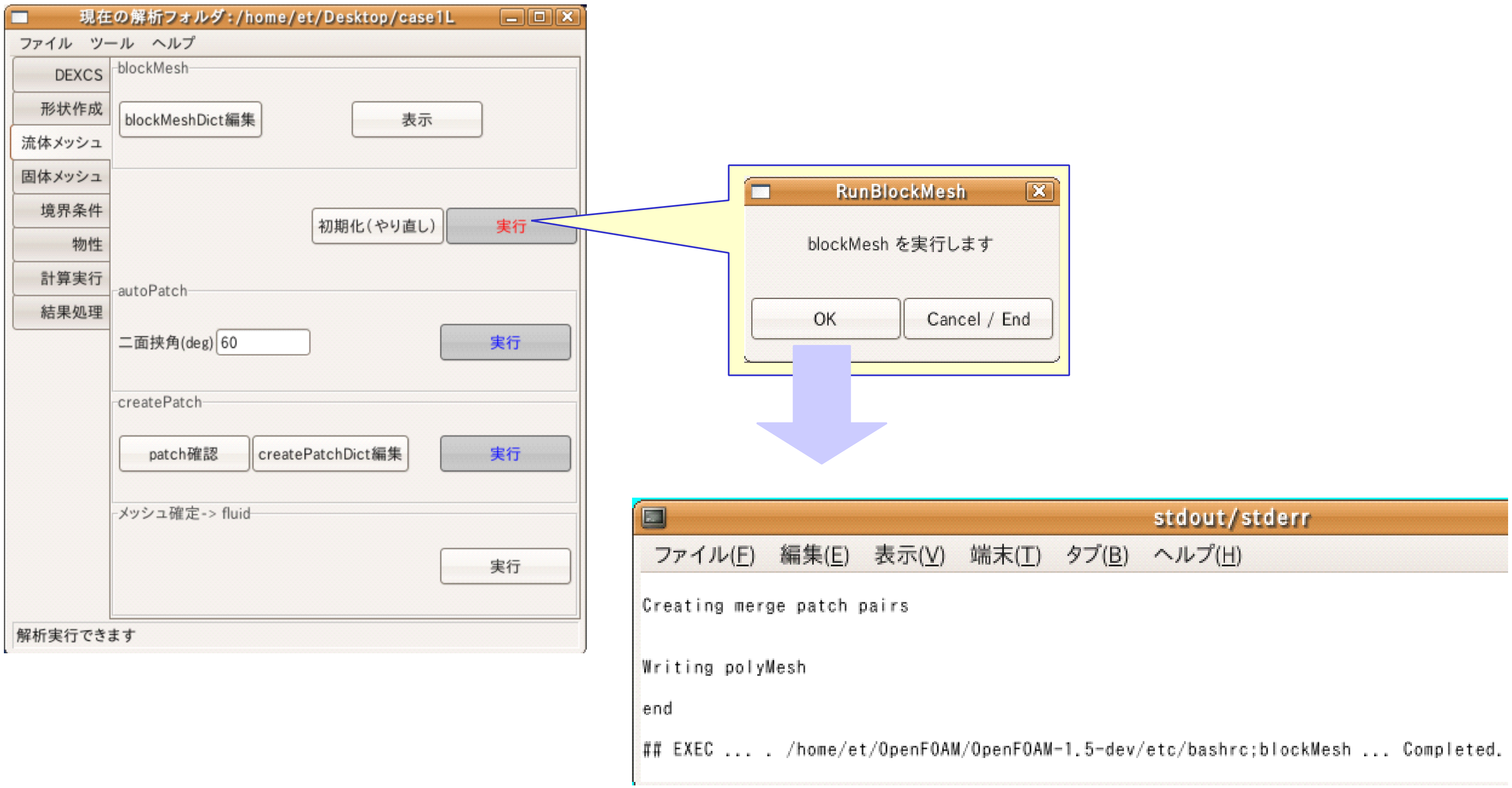


形状やメッシュ分割方案を変更したい場合にはblockMeshDictを直接変更する

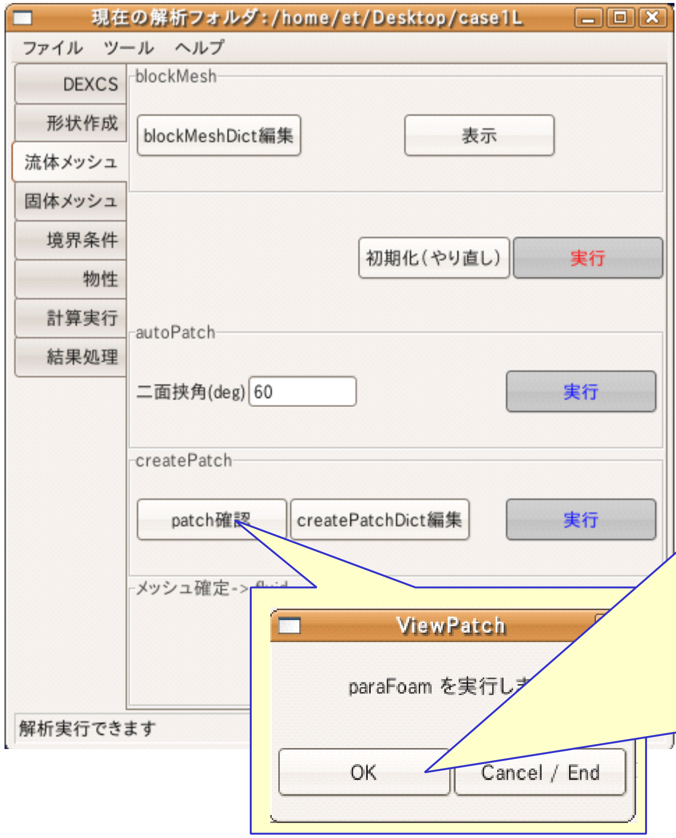
変更の際には、ブロックや節点番号表示を見ながら作業すると良い

流体メッシュ

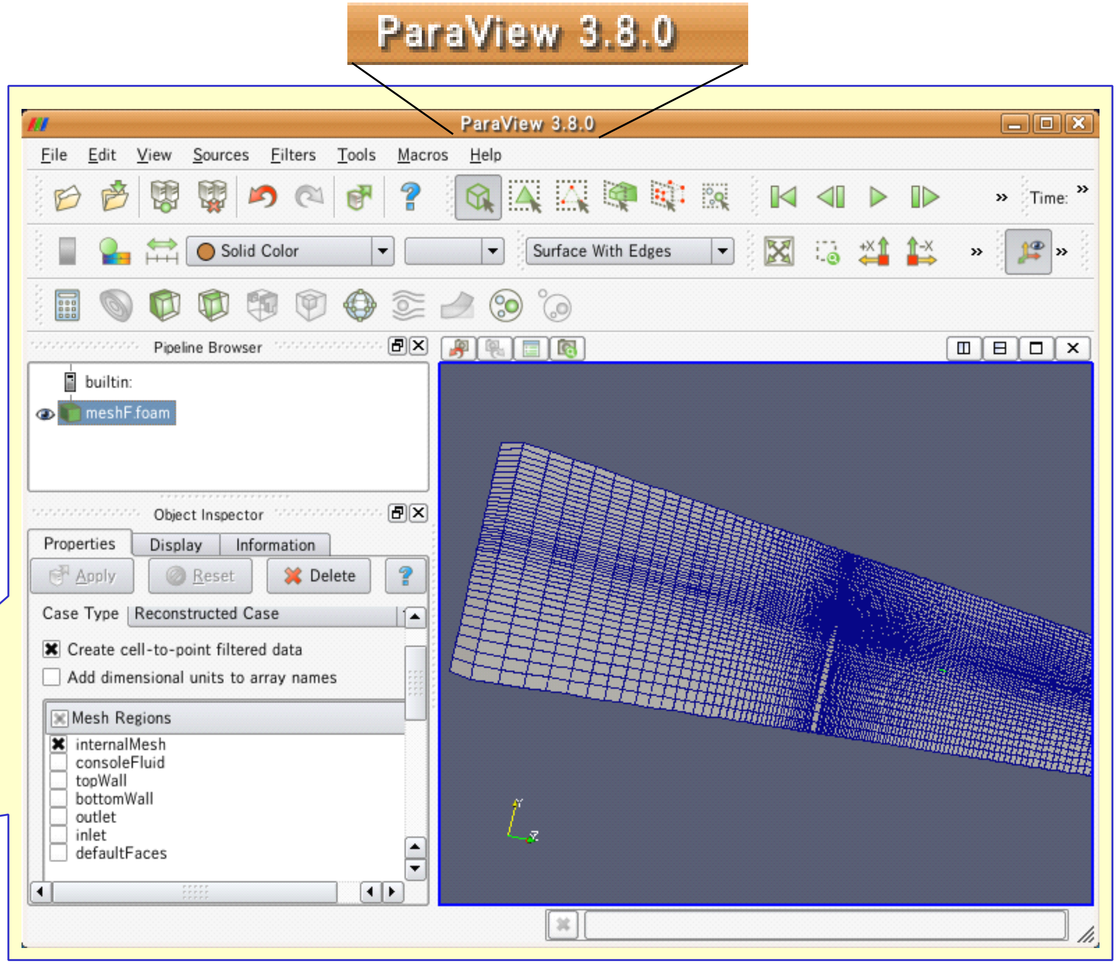
メッシュの作成
（blockMeshの実行）



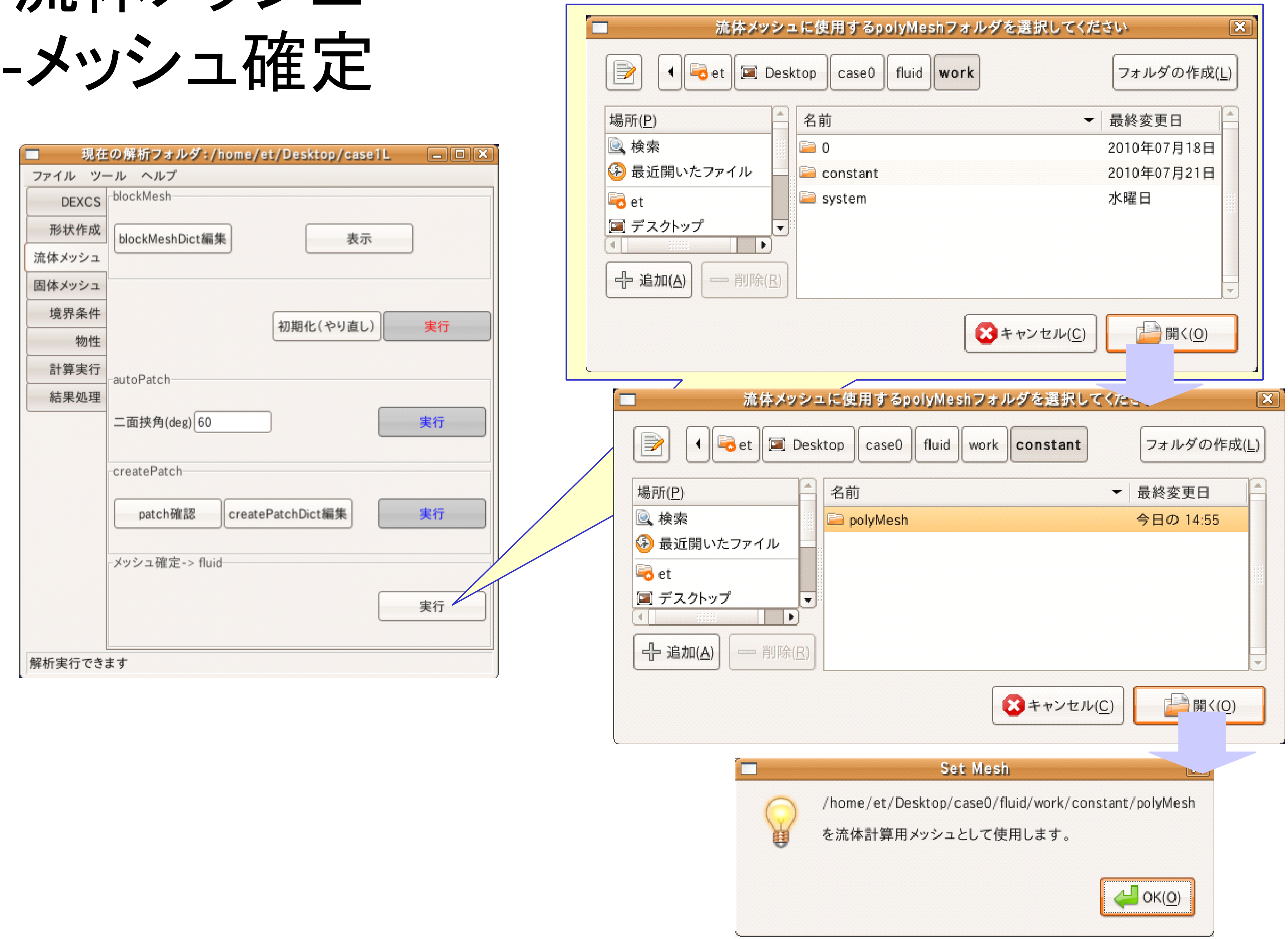
流体メッシュ



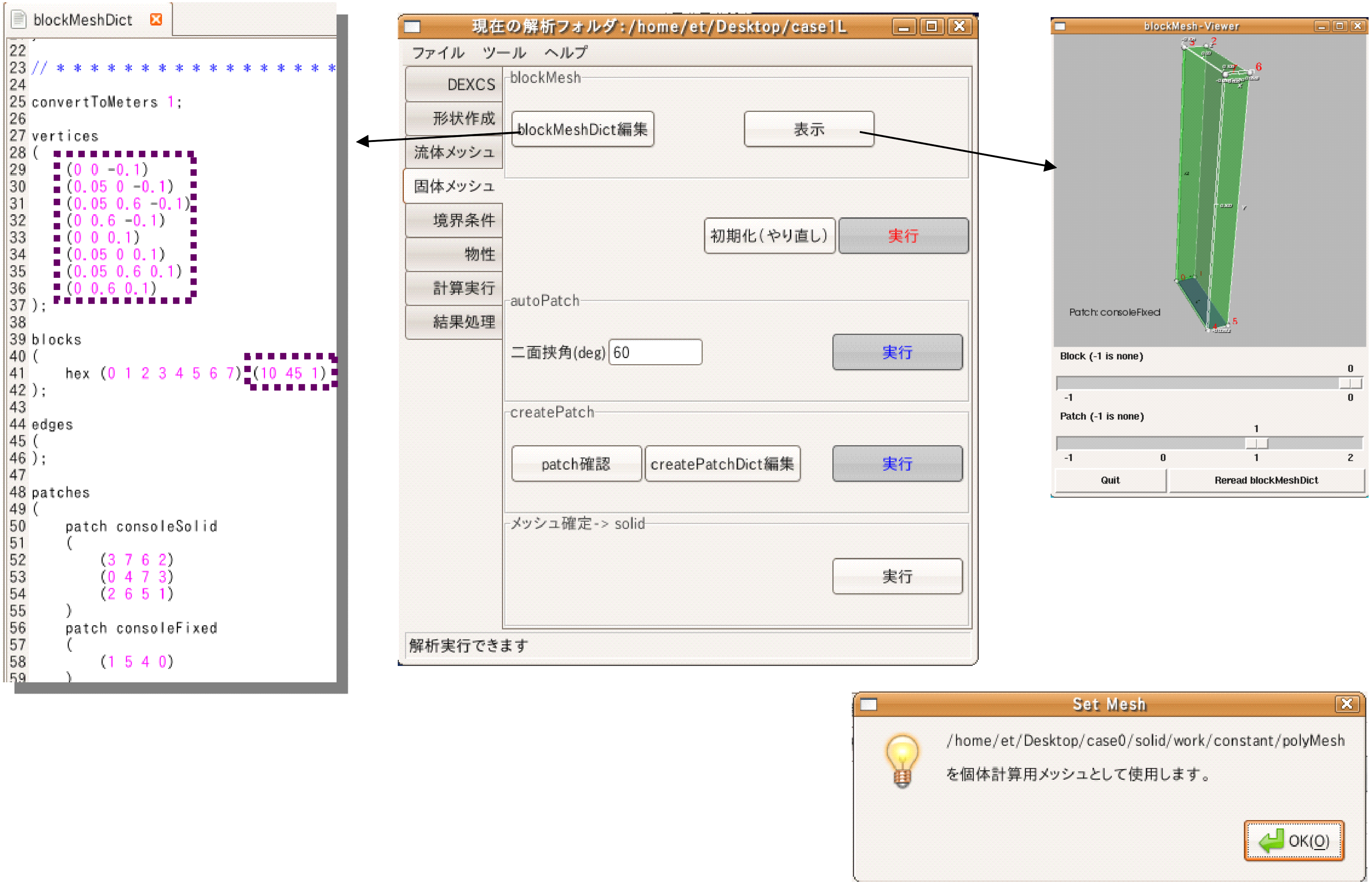
メッシュの確認



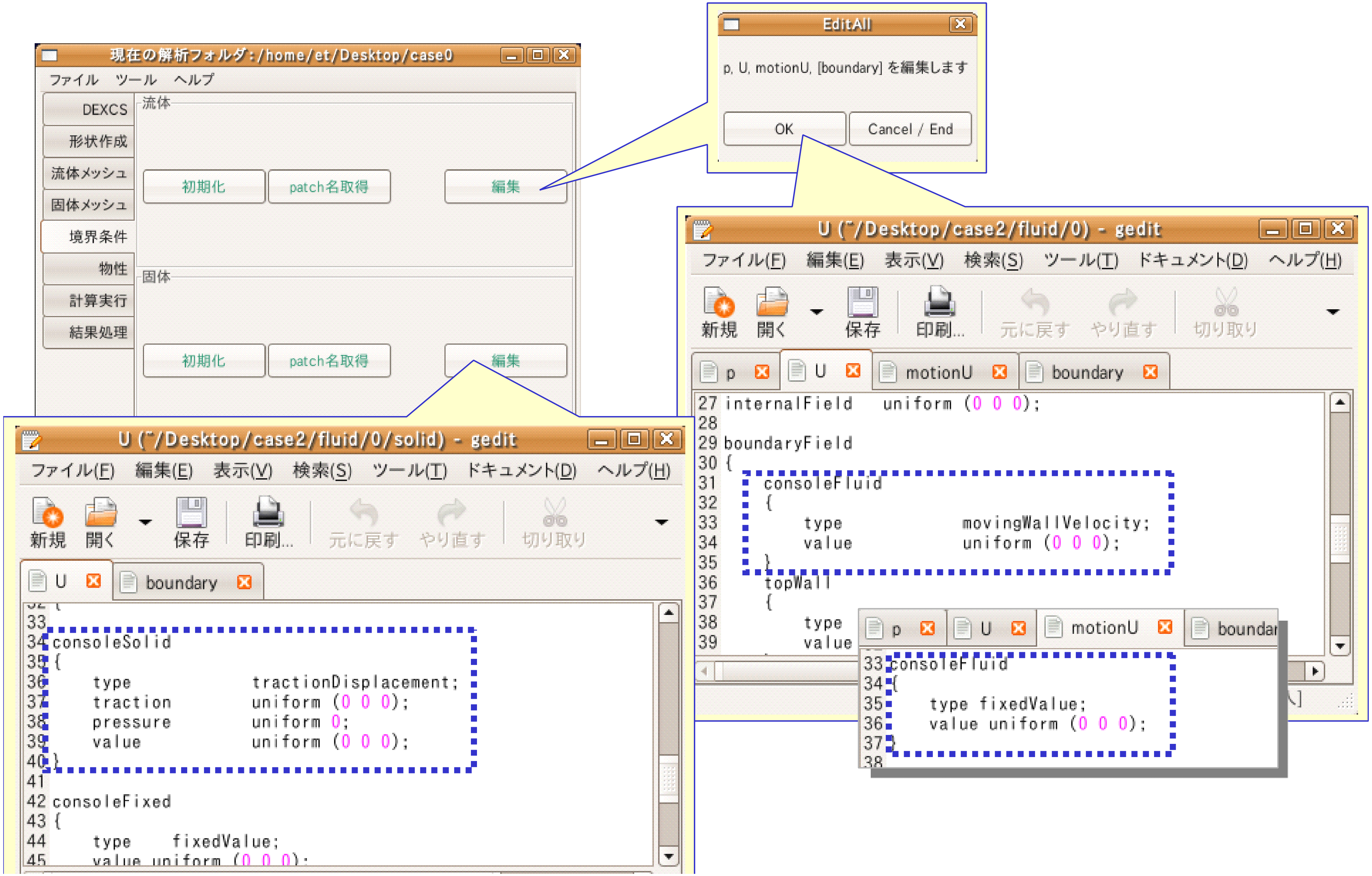
流体メッシュ
-メッシュ確定



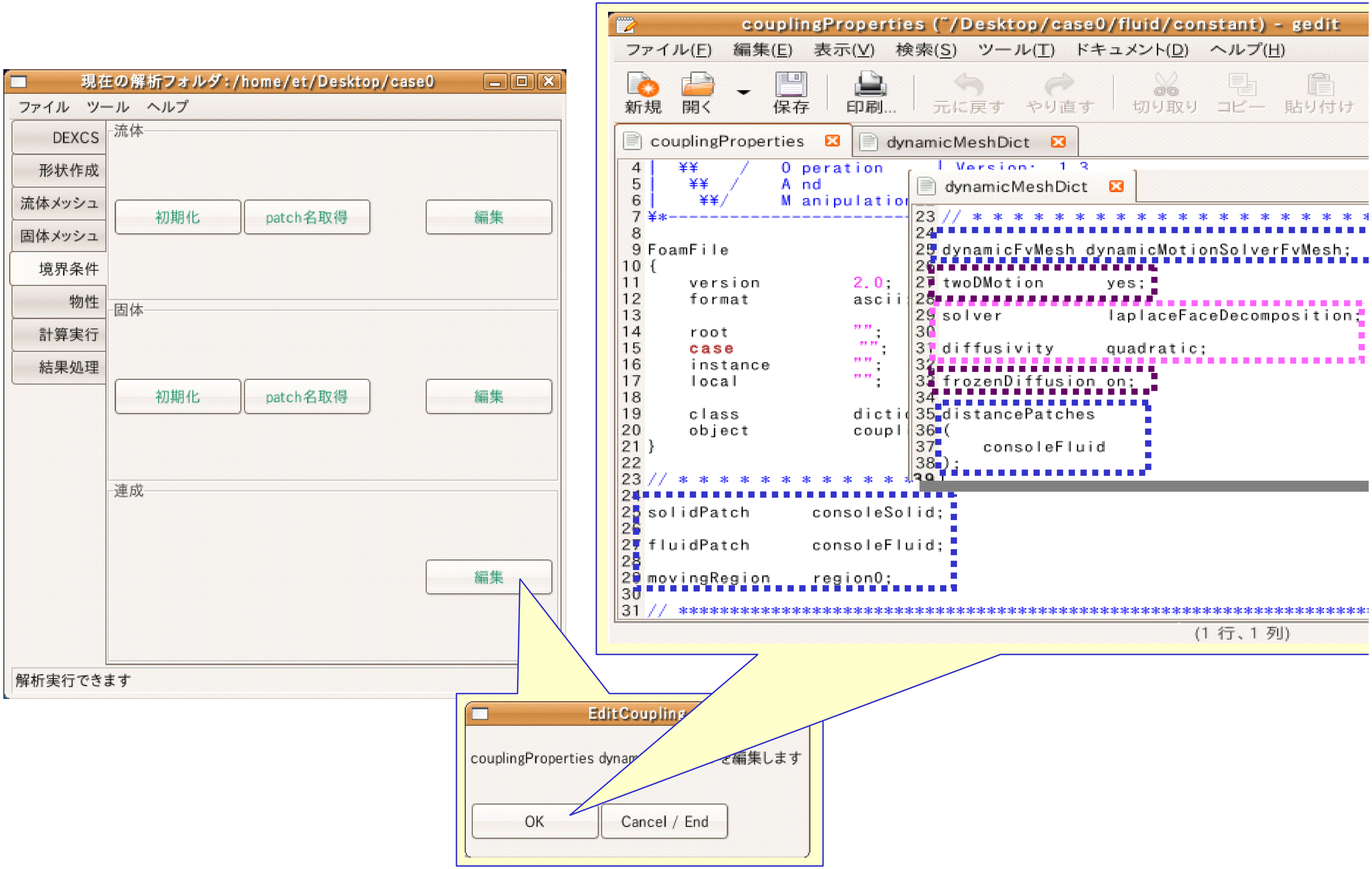
固体メッシュの作成


blockMeshDict
// * * * * * * * * * * * * * * * * *
convertToMeters 1;
vertices
(
(0 0 -0.1)
(0.05 0 -0.1)
(0.05 0.6 -0.1)
(0 0.6 -0.1)
(0 0 0.1)
(0.05 0 0.1)
(0.05 0.6 0.1)
(0 0.6 0.1)
);
blocks
(
hex (0 1 2 3 4 5 6 7) (10 45 1)
);
edges
(
);
patches
(
patch consoleSolid
(
(3 7 6 2)
(0 4 7 3)
(2 6 5 1)
)
patch consoleFixed
(
(1 5 4 0)
)
現在の解析フォルダ：/home/et/Desktop/case1L
ファイル　ツール　ヘルプ
DEXCS
形状作成
流体メッシュ
固体メッシュ
境界条件
物性
計算実行
結果処理
blockMesh
blockMeshDict編集
表示
初期化(やり直し)
実行
autoPatch
二面挟角(deg) 60
実行
createPatch
patch確認
createPatchDict編集
実行
メッシュ確定-> solid
実行
解析実行できます
blockMesh-Viewer
Patch: consoleFixed
Block (-1 is none)
Patch (-1 is none)
Quit
Reread blockMeshDict
Set Mesh
/home/et/Desktop/case0/solid/work/constant/polyMesh
を個体計算用メッシュとして使用します。
OK(O)

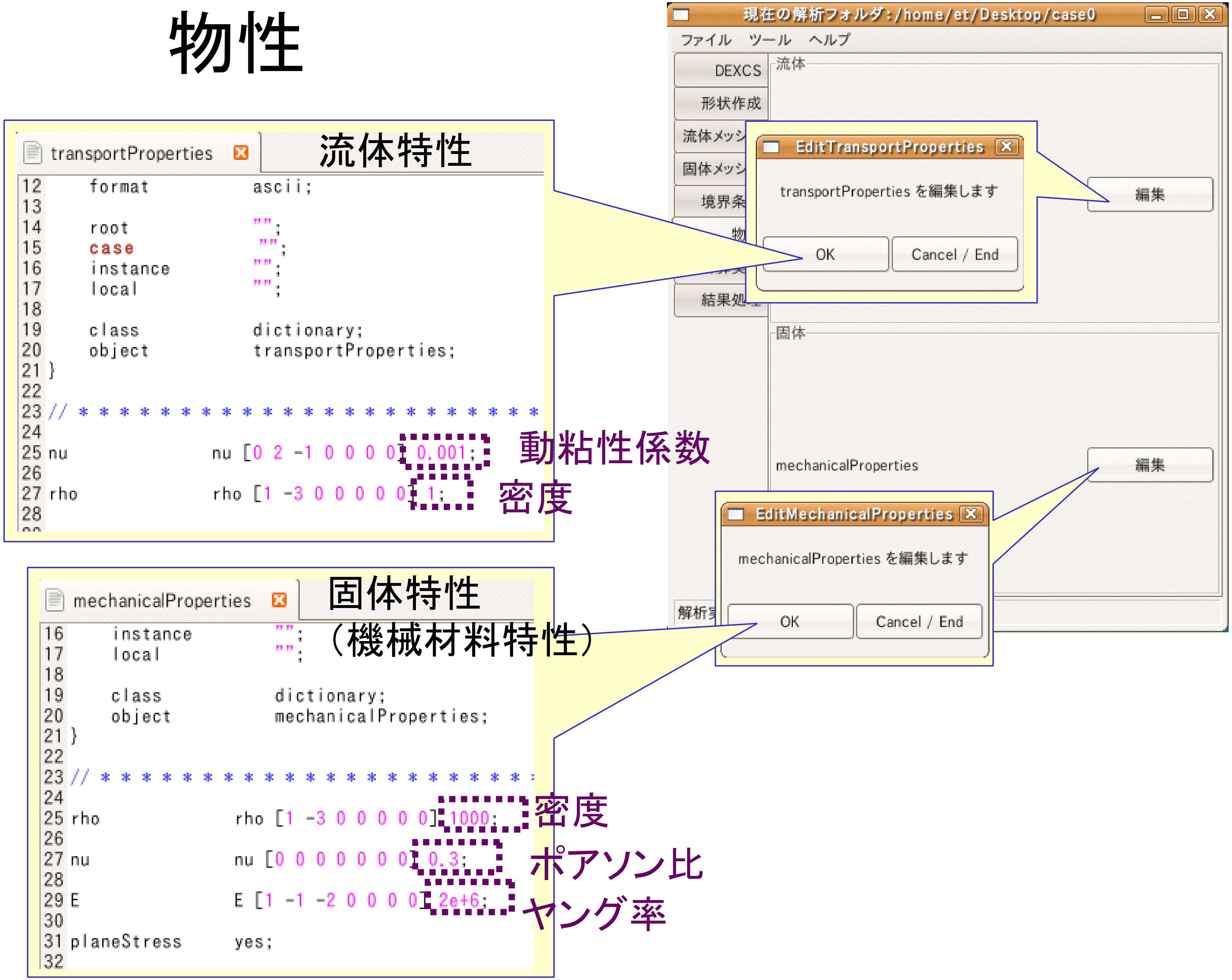
境界条件


現在の解析フォルダ:/home/et/Desktop/case0
ファイル ツール ヘルプ
DEXCS
形状作成
流体メッシュ
固体メッシュ
境界条件
物性
計算実行
結果処理
流体
初期化
patch名取得
編集
固体
初期化
patch名取得
編集
EditAll
p, U, motionU, [boundary] を編集します
OK
Cancel / End
U (~/Desktop/case2/fluid/0) - gedit
ファイル(F) 編集(E) 表示(V) 検索(S) ツール(T) ドキュメント(D) ヘルプ(H)
新規 開く 保存 印刷... 元に戻す やり直す 切り取り
p U motionU boundary
27 internalField uniform (0 0 0);
28
29 boundaryField
30 {
31 consoleFluid
32 {
33 type movingWallVelocity;
34 value uniform (0 0 0);
35 }
36 topWall
37 {
38 type
39 value
p U motionU boundar
33 consoleFluid
34 {
35 type fixedValue;
36 value uniform (0 0 0);
37 }
38
U (~/Desktop/case2/fluid/0/solid) - gedit
ファイル(F) 編集(E) 表示(V) 検索(S) ツール(T) ドキュメント(D) ヘルプ(H)
新規 開く 保存 印刷... 元に戻す やり直す 切り取り
U boundary
33
34 consoleSolid
35 {
36 type tractionDisplacement;
37 traction uniform (0 0 0);
38 pressure uniform 0;
39 value uniform (0 0 0);
40 }
41
42 consoleFixed
43 {
44 type fixedValue;
45 value uniform (0 0 0);

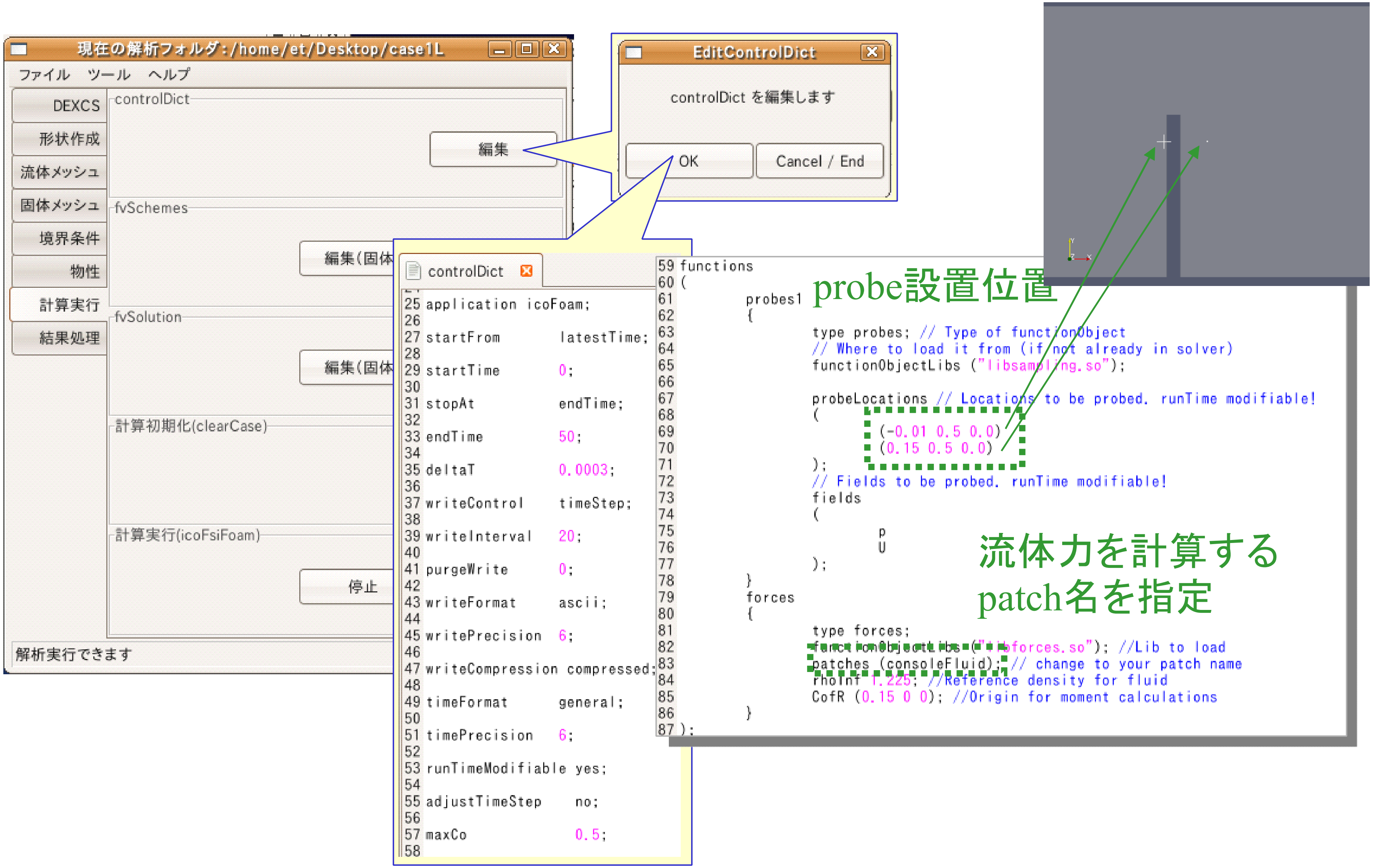
境界条件(連成)


現在の解析フォルダ:/home/et/Desktop/case0
ファイル ツール ヘルプ
DEXCS
形状作成
流体メッシュ
固体メッシュ
境界条件
物性
計算実行
結果処理
流体
初期化
patch名取得
編集
固体
初期化
patch名取得
編集
連成
編集
解析実行できます
EditCoupling
couplingProperties dynam
を編集します
OK
Cancel / End
couplingProperties (~/Desktop/case0/fluid/constant) - gedit
ファイル(F) 編集(E) 表示(V) 検索(S) ツール(T) ドキュメント(D) ヘルプ(H)
新規 開く 保存 印刷... 元に戻す やり直す 切り取り コピー 貼り付け
couplingProperties
dynamicMeshDict
FoamFile
{
version 2.0;
format ascii;
root "";
case "";
instance "";
local "";
class dicti
object coupl
}
solidPatch consoleSolid;
fluidPatch consoleFluid;
movingRegion region0;
dynamicFvMesh dynamicMotionSolverFvMesh;
twoDMotion yes;
solver laplaceFaceDecomposition;
diffusivity quadratic;
frozenDiffusion on;
distancePatches
(
consoleFluid
);
(1 行、1 列)

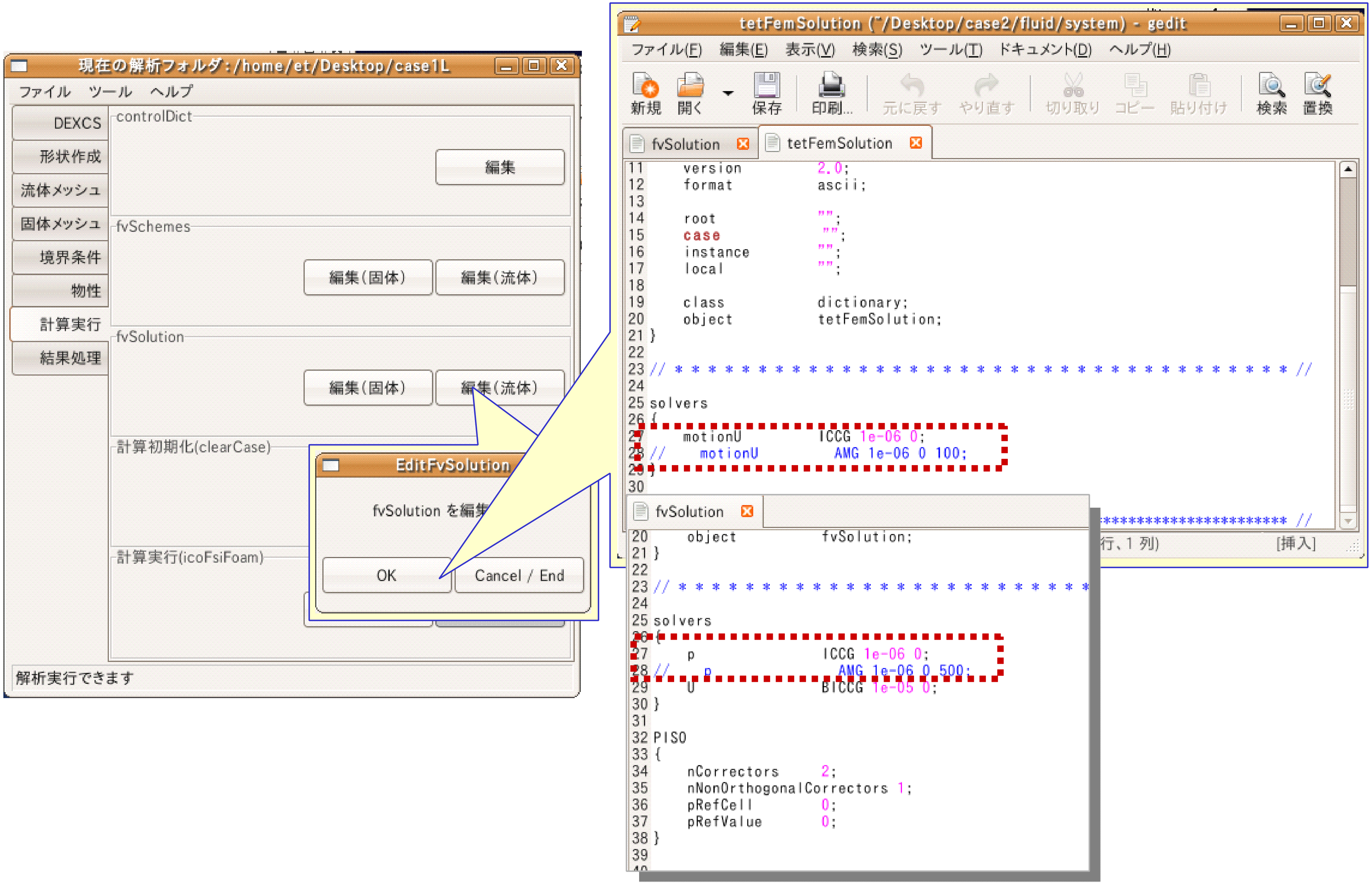
物性


transportProperties
流体特性
12 format ascii;
13
14 root "";
15 case "";
16 instance "";
17 local "";
18
19 class dictionary;
20 object transportProperties;
21 }
22
23 // *
24
25 nu nu [0 2 -1 0 0 0 0] 0.001;
26
27 rho rho [1 -3 0 0 0 0 0] 1;
28
動粘性係数
密度
mechanicalProperties
固体特性
（機械材料特性）
16 instance "";
17 local "";
18
19 class dictionary;
20 object mechanicalProperties;
21 }
22
23 // *
24
25 rho rho [1 -3 0 0 0 0 0] 1000;
26
27 nu nu [0 0 0 0 0 0 0] 0.3;
28
29 E E [1 -1 -2 0 0 0 0] 2e+6;
30
31 planeStress yes;
32
密度
ポアソン比
ヤング率
現在の解析フォルダ：/home/et/Desktop/case0
ファイル ツール ヘルプ
DEXCS
形状作成
流体メッシ
固体メッシ
境界条
物
結果処
解析実
流体
固体
mechanicalProperties
編集
編集
EditTransportProperties
transportProperties を編集します
OK
Cancel / End
EditMechanicalProperties
mechanicalProperties を編集します
OK
Cancel / End

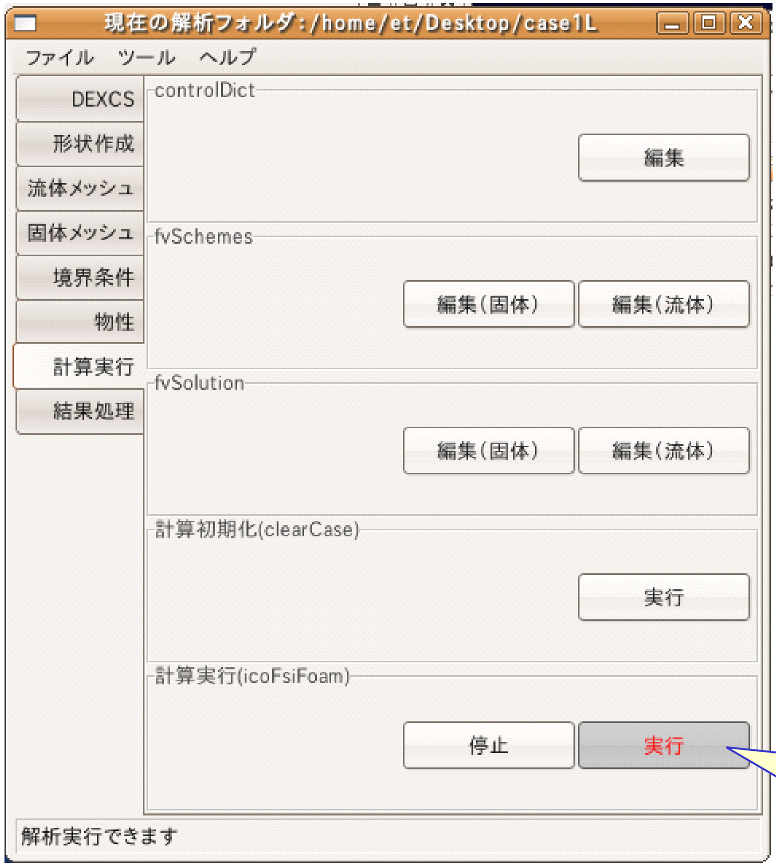
計算実行（controlDict）

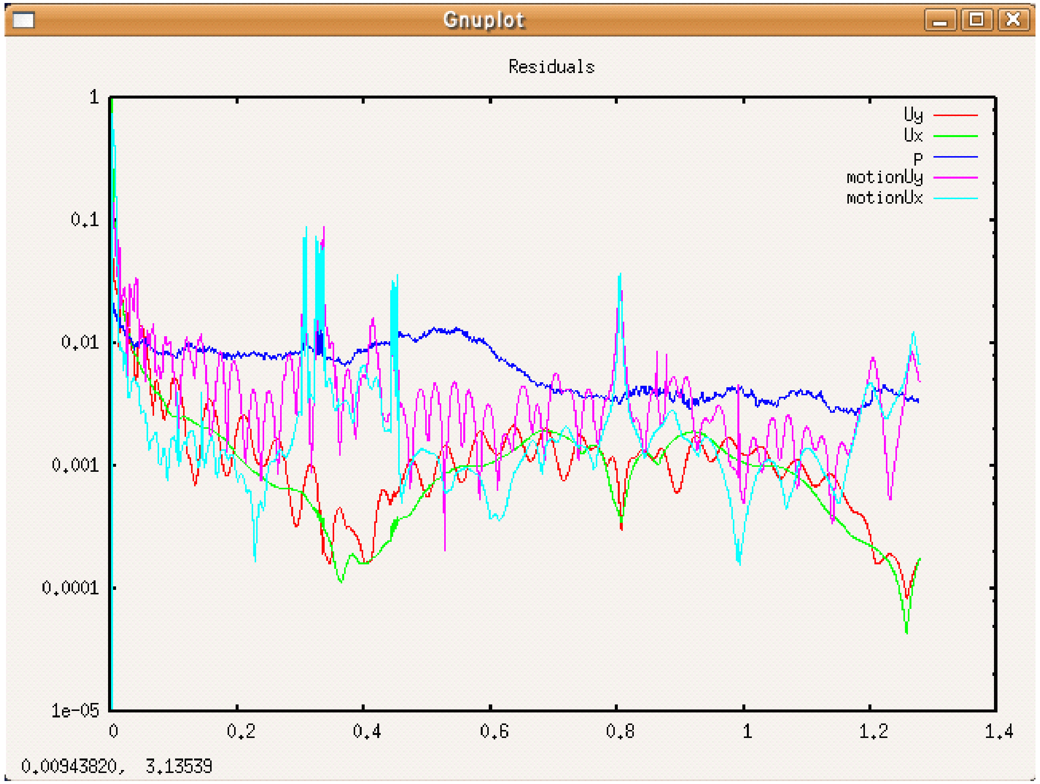


計算実行（fvSolution）

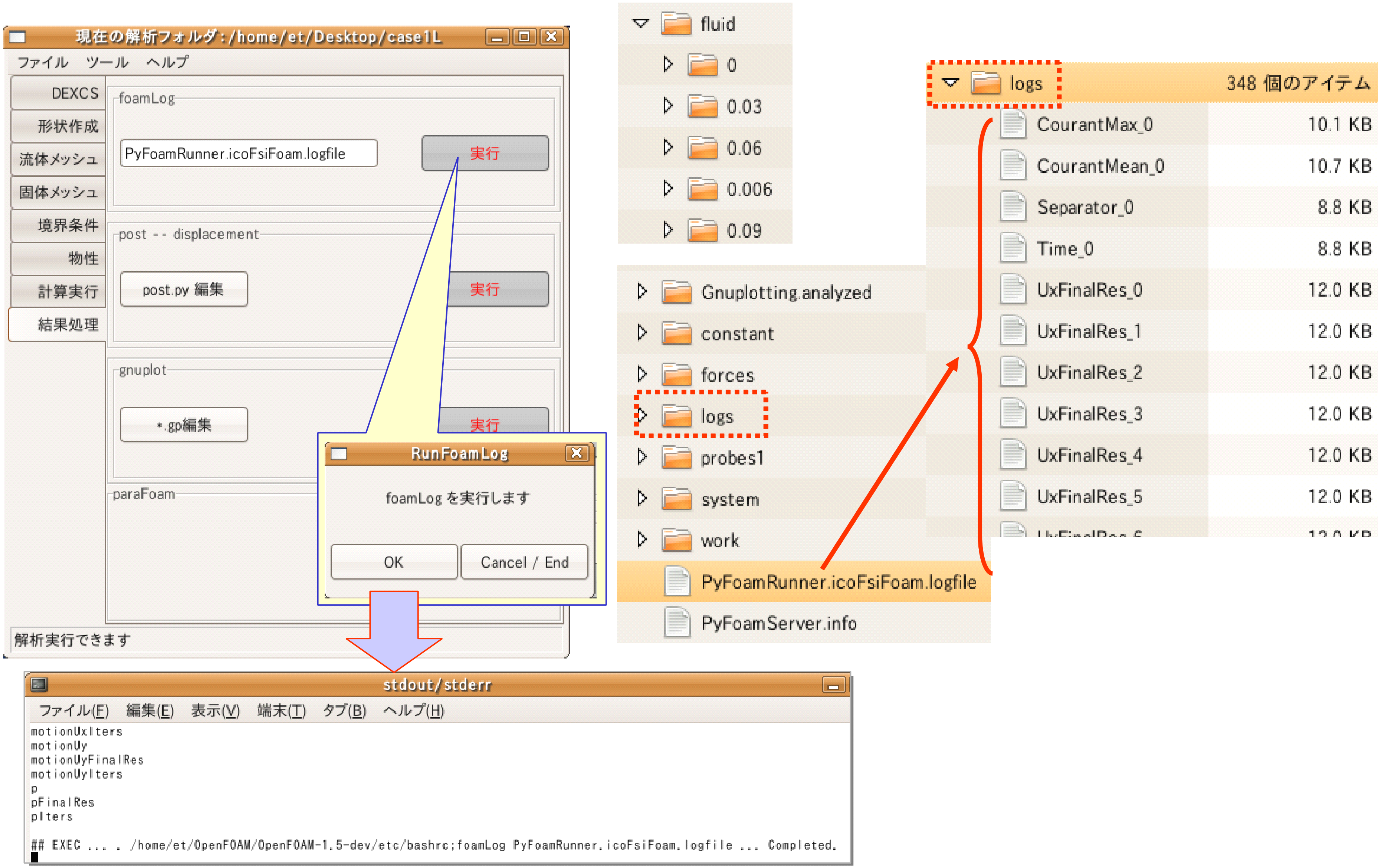


計算実行(icoFsiFoam)

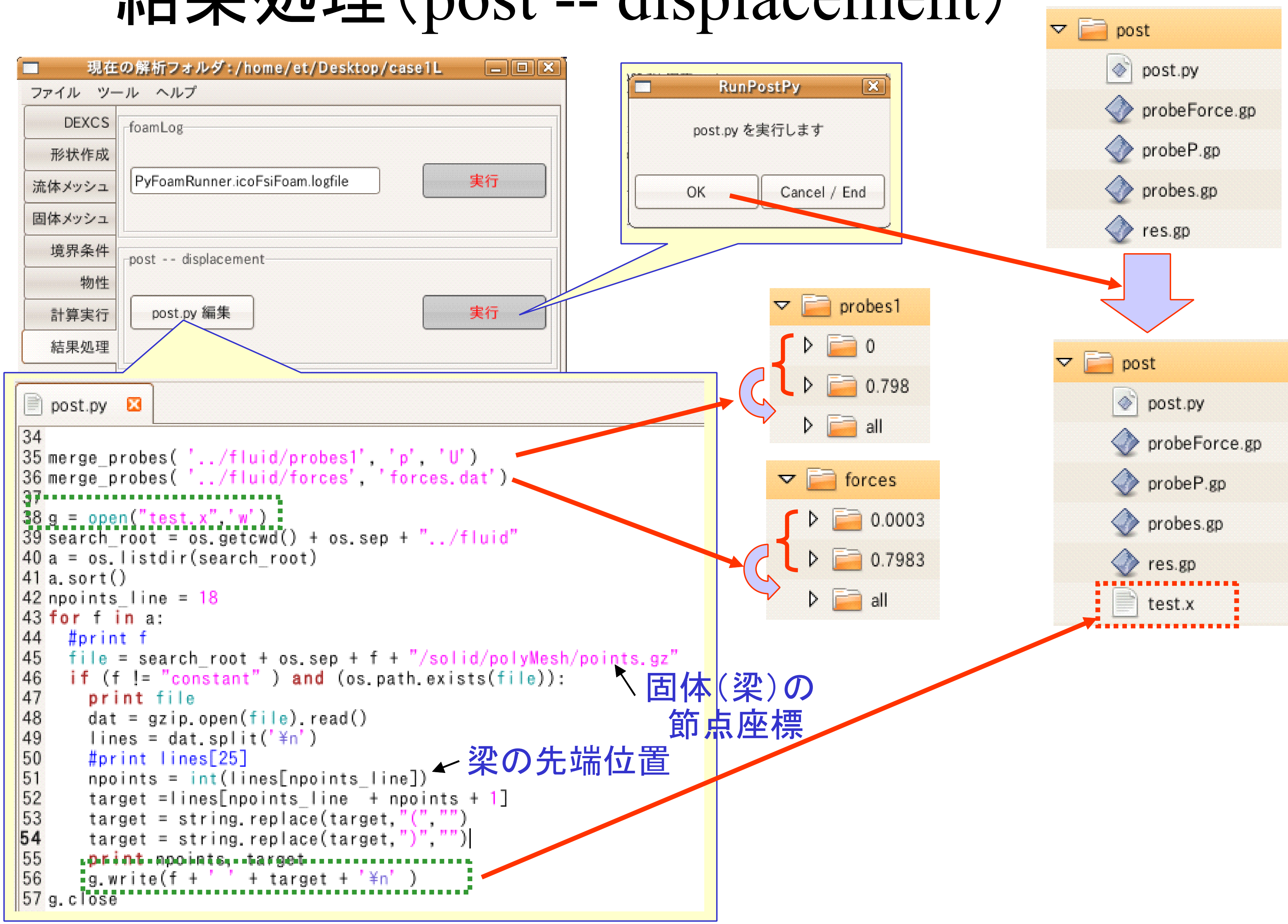




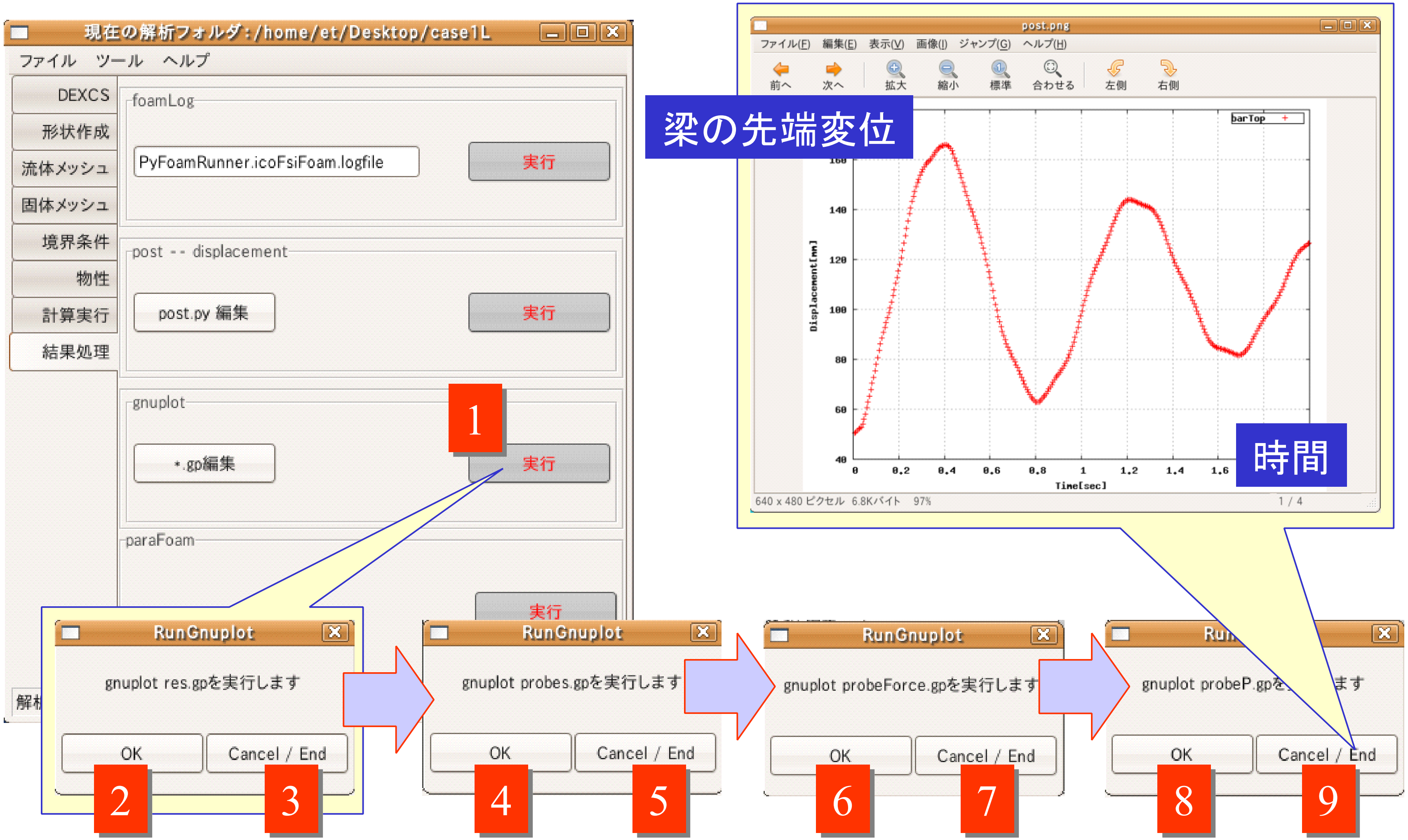
結果処理（foamLog）



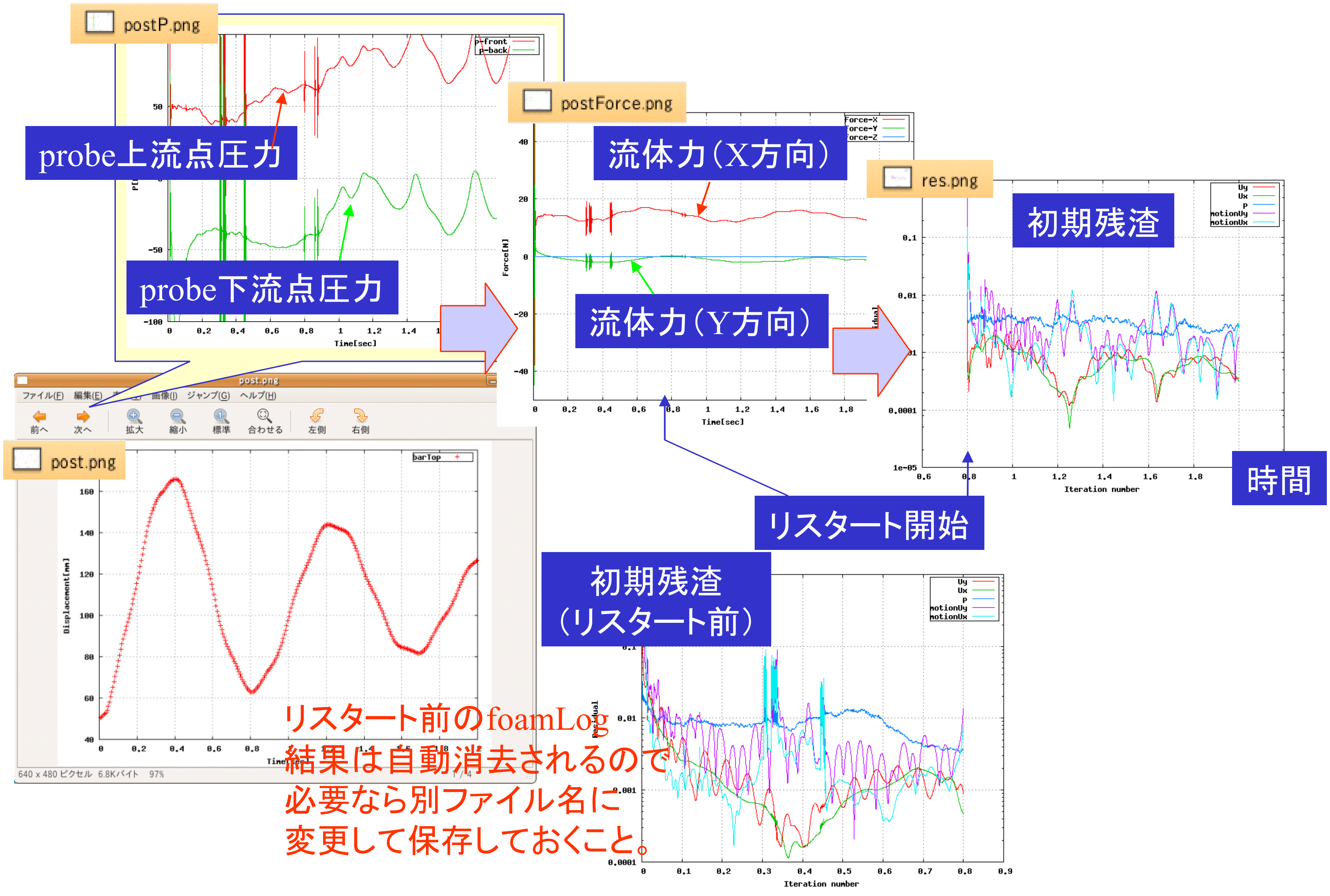
結果処理(post -- displacement)



結果処理(gnuplot)

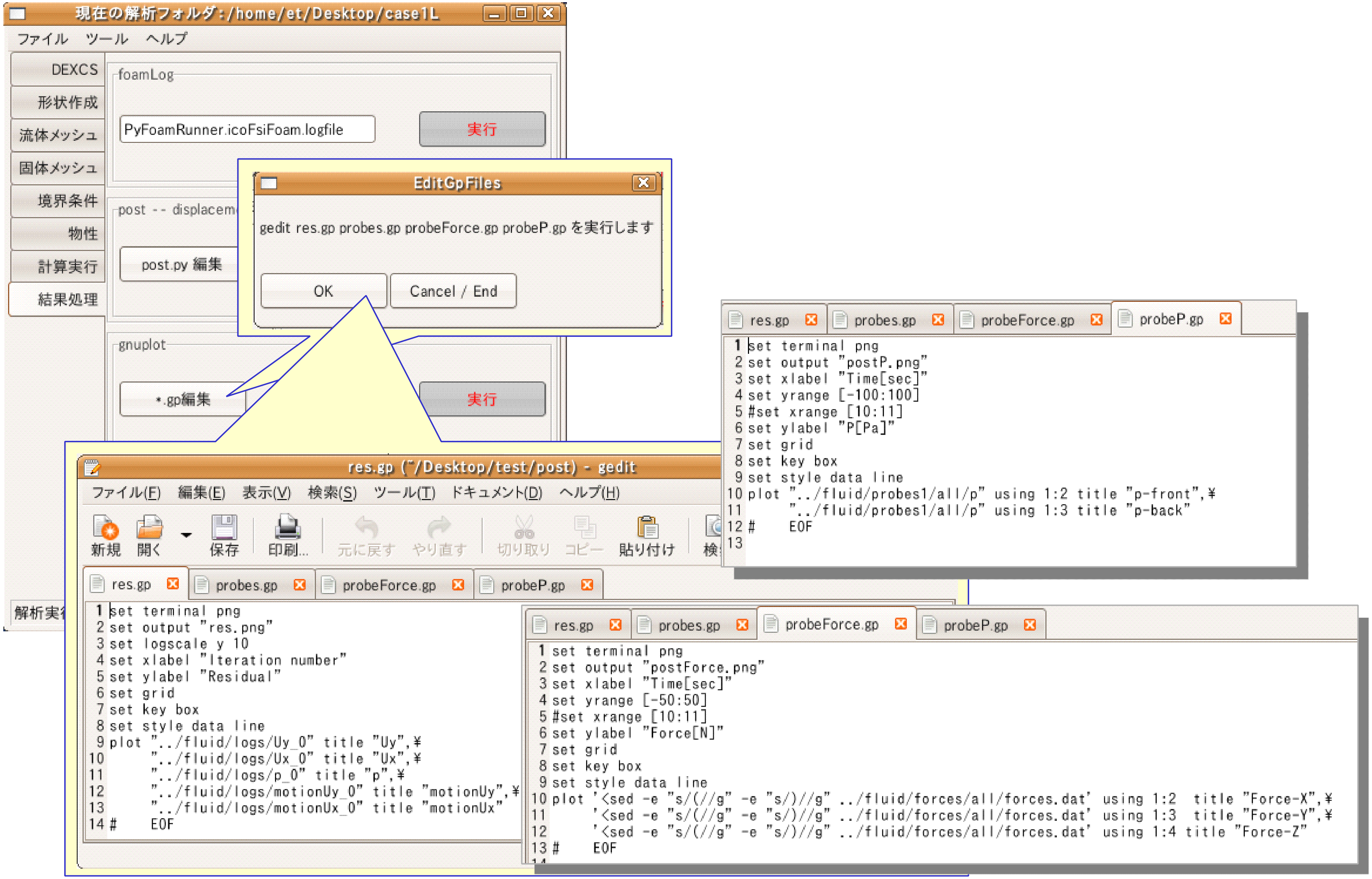


結果処理（gnuplot）続き

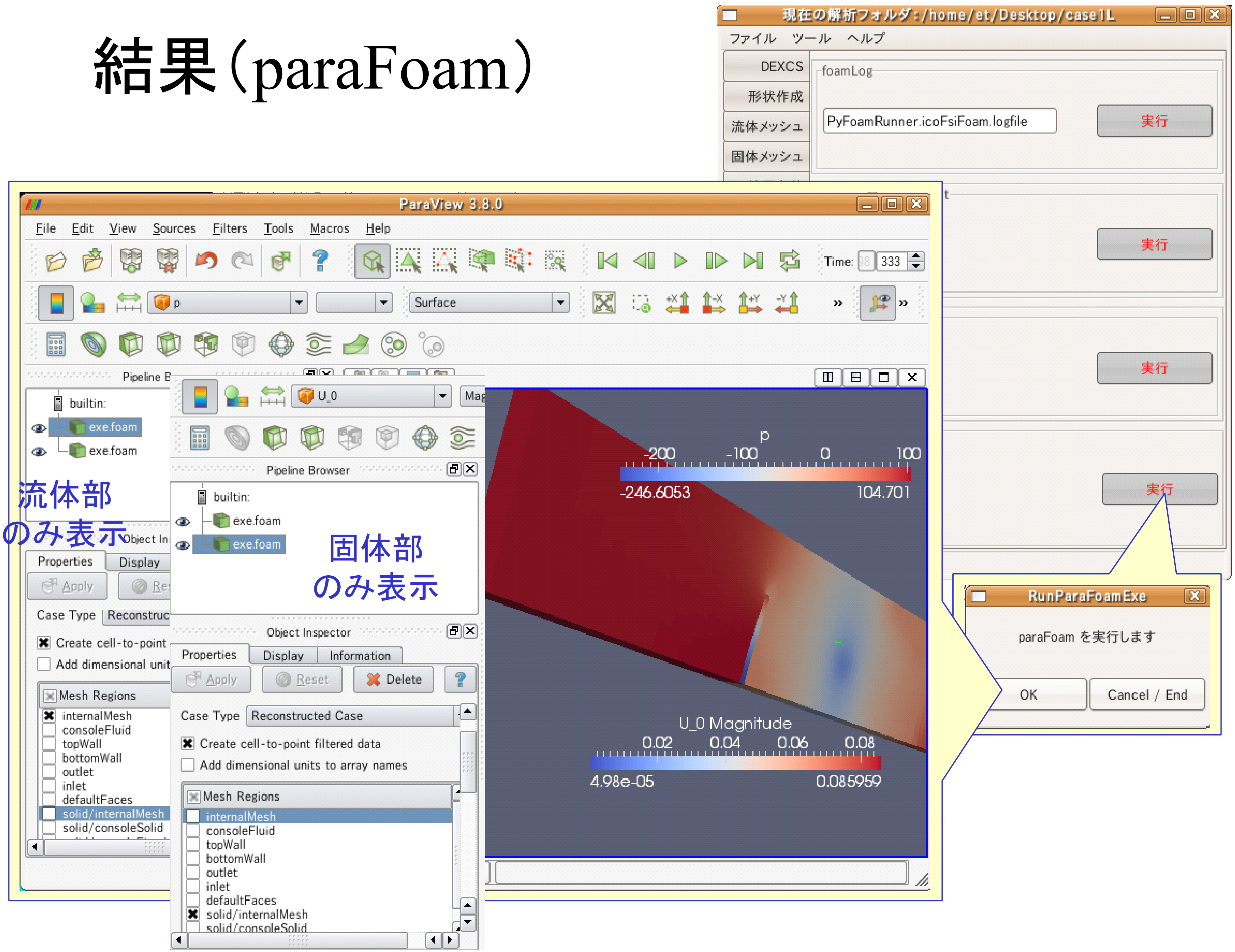


リスタート前のfoamLog
結果は自動消去されるので
必要なら別ファイル名に
変更して保存しておくこと。

結果処理(グラフ表示形式変更)



結果（paraFoam）


現在の解析フォルダ：/home/et/Desktop/case1L
ファイル ツール ヘルプ
DEXCS
形状作成
流体メッシュ
固体メッシュ
foamLog
PyFoamRunner.icoFsiFoam.logfile
実行
実行
実行
実行
ParaView 3.8.0
File Edit View Sources Filters Tools Macros Help
Time: 333
p
Surface
Pipeline Browser
builtin:
exe.foam
exe.foam
流体部
のみ表示
固体部
のみ表示
U_0
Object Inspector
Properties Display Information
Apply Reset Delete
Case Type Reconstructed Case
Create cell-to-point filtered data
Add dimensional units to array names
Mesh Regions
internalMesh
consoleFluid
topWall
bottomWall
outlet
inlet
defaultFaces
solid/internalMesh
solid/consoleSolid
-200 -100 0 100
-246.6053
104.701
U_0 Magnitude
0.02 0.04 0.06 0.08
4.98e-05
0.085959
RunParaFoamExe
paraFoam を実行します
OK
Cancel / End

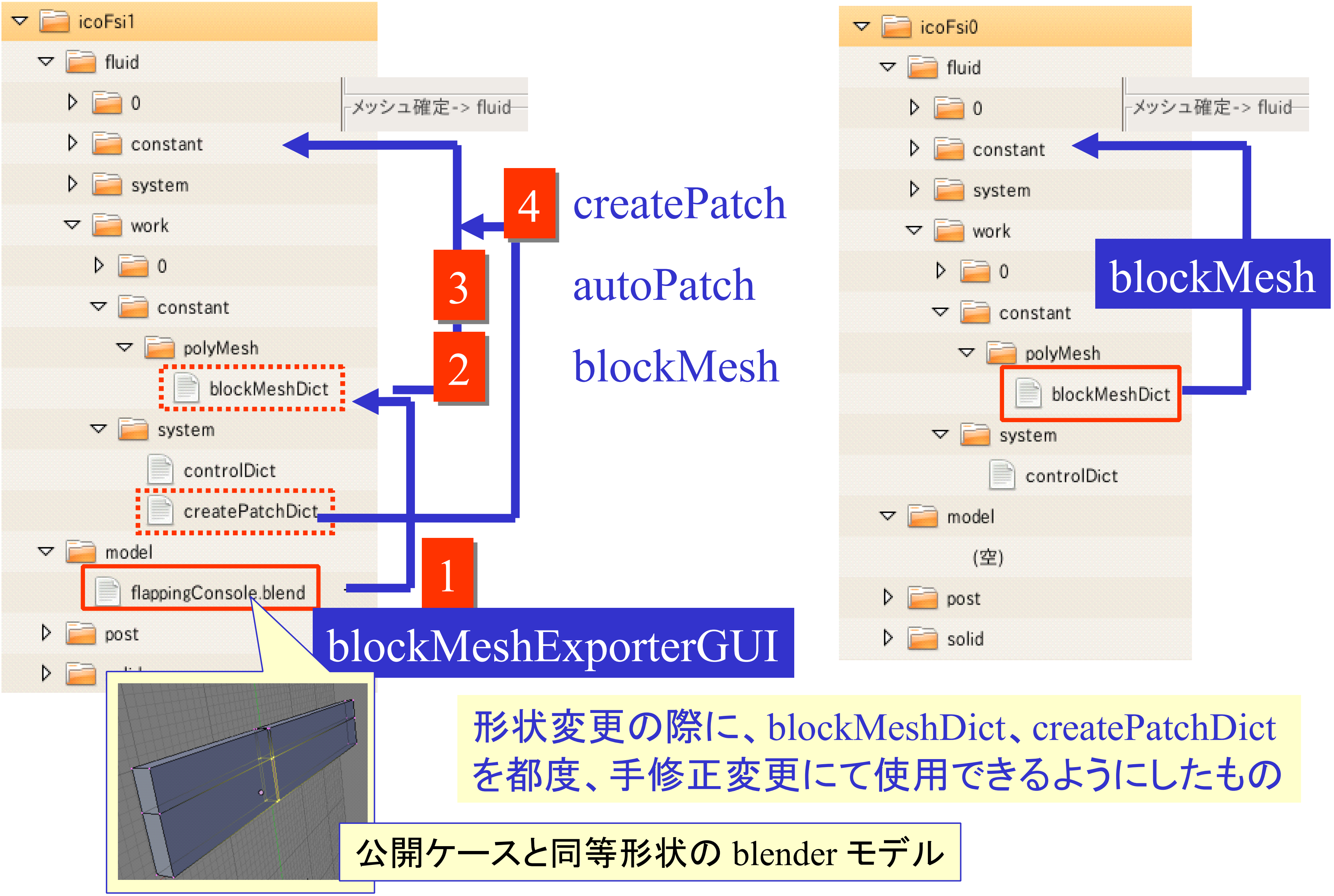
DEXCS_FSI ランチャーによる解析実行
（標準モデルの形状変更）

blender形状モデル

↓

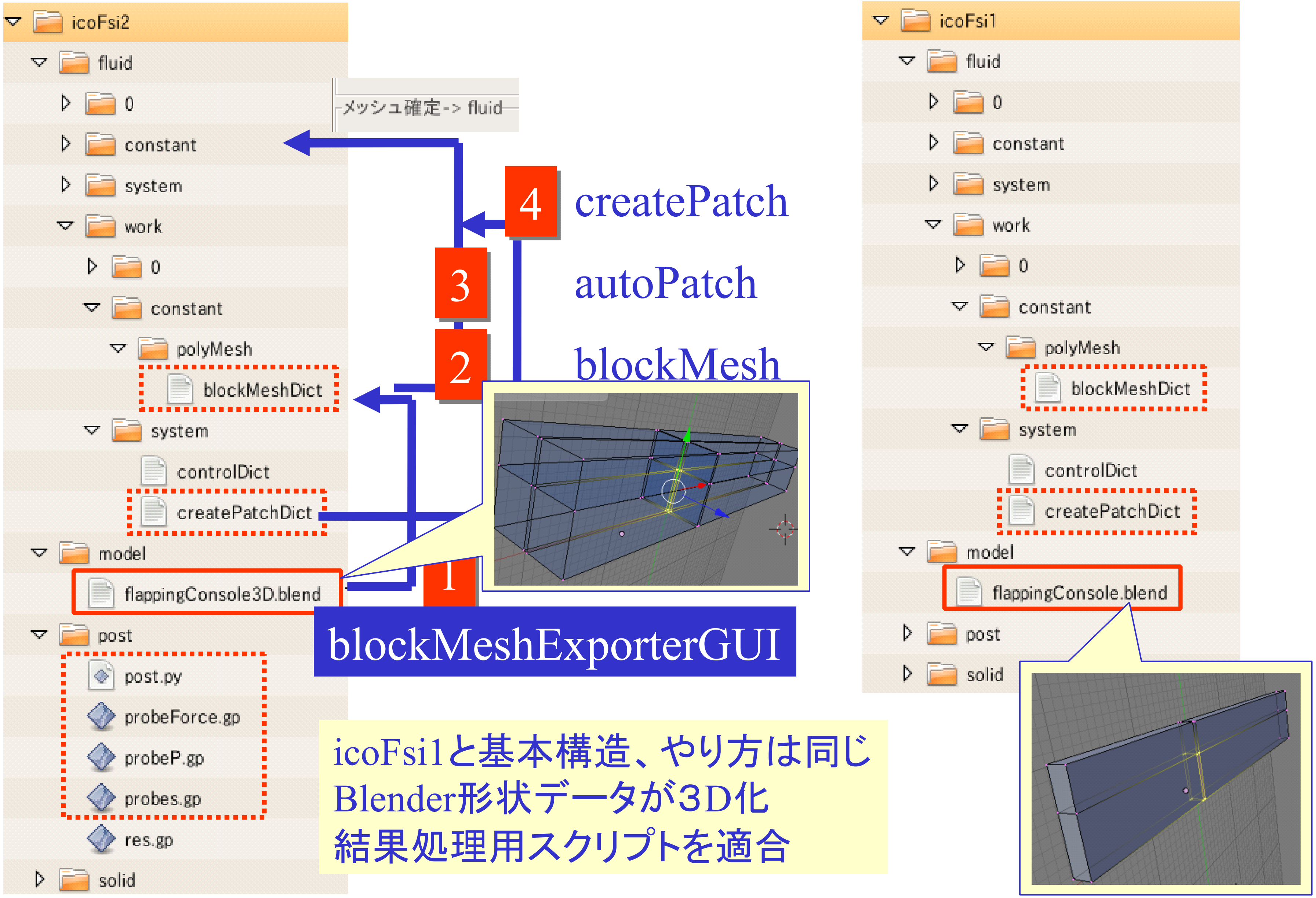
blockMeshExporter GUI

DEXCS template 2（icoFsi1）



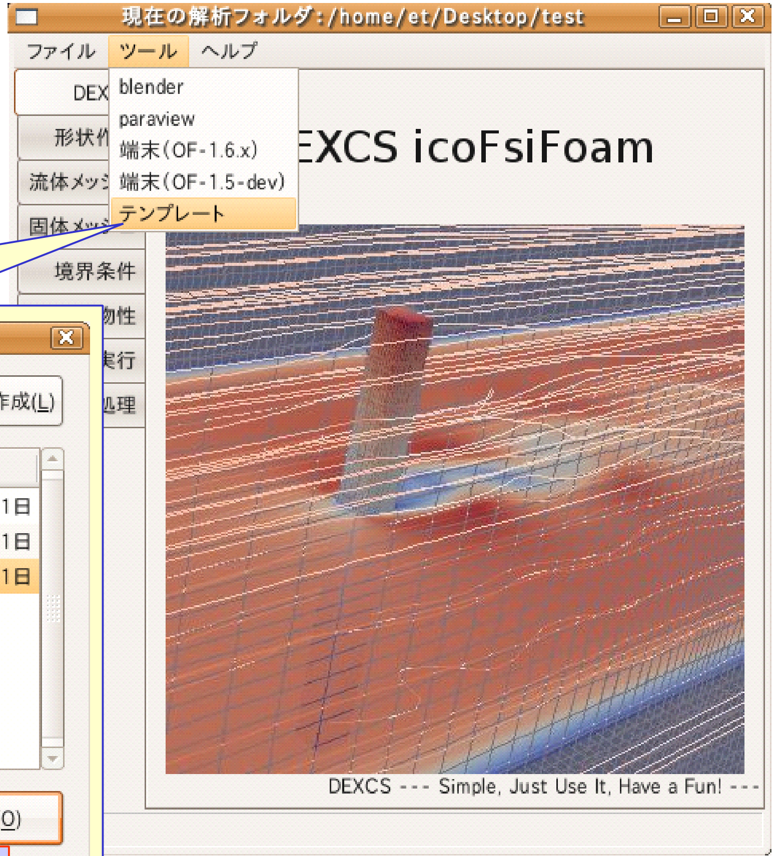
形状変更の際に、blockMeshDict、createPatchDict
を都度、手修正変更にて使用できるようにしたもの

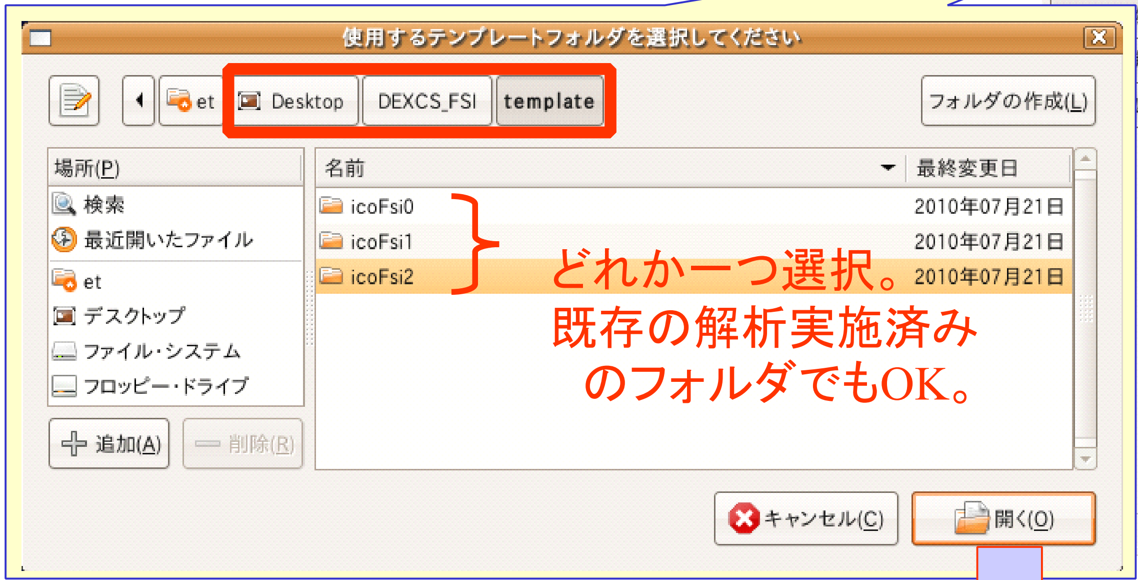
DEXCS template 3（icoFsi2）



icoFsi1と基本構造、やり方は同じ
Blender形状データが3D化
結果処理用スクリプトを適合

テンプレートの変更方法

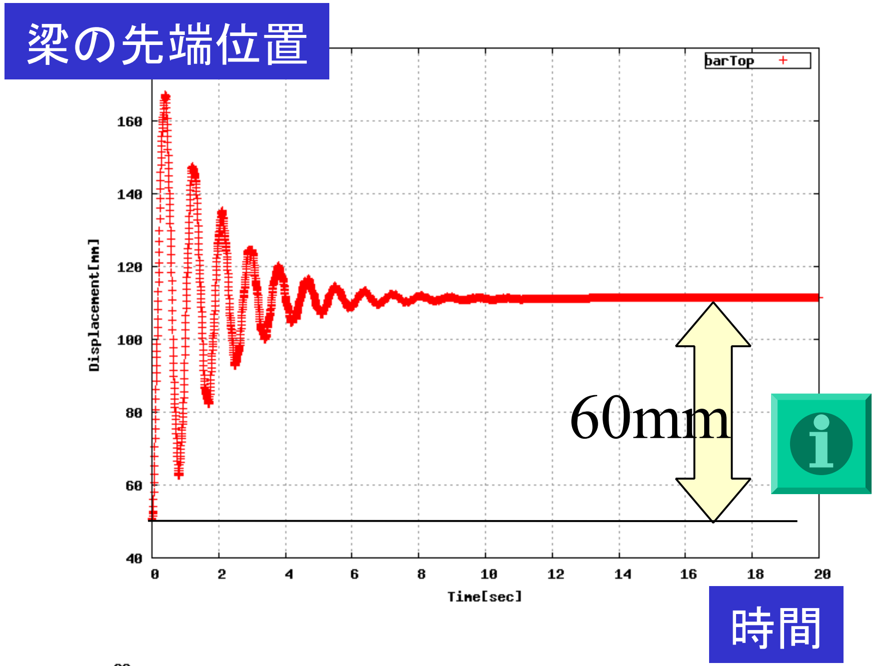




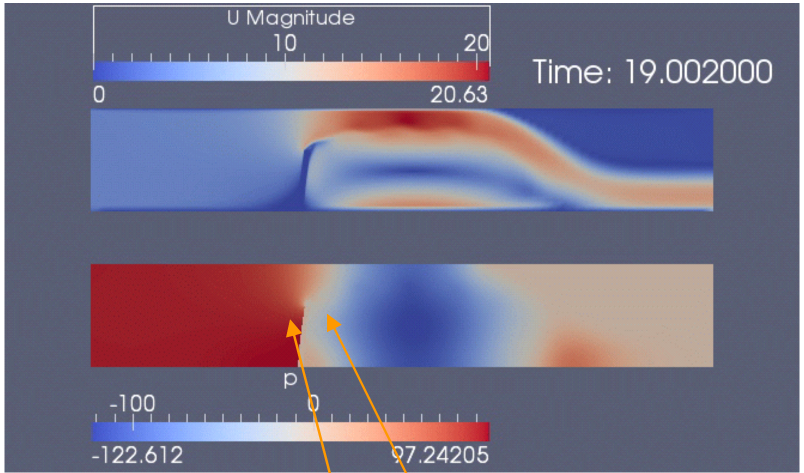
- この設定以降に作成される新規解析フォルダに適用される。
- 但し、ランチャーを終了すれば本設定変更はリセットされる。
- 作成済みの解析フォルダには影響しない。

解析例の紹介

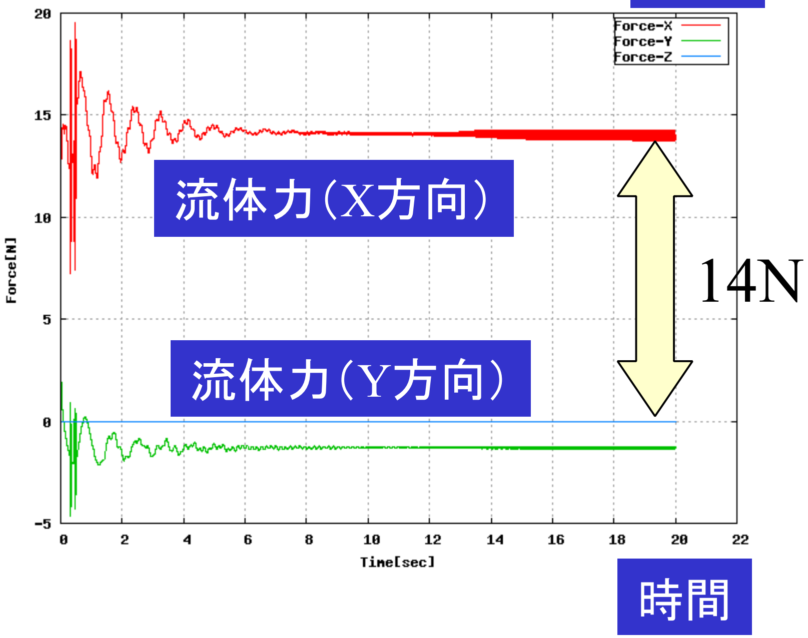
解析例1（公開ケース）



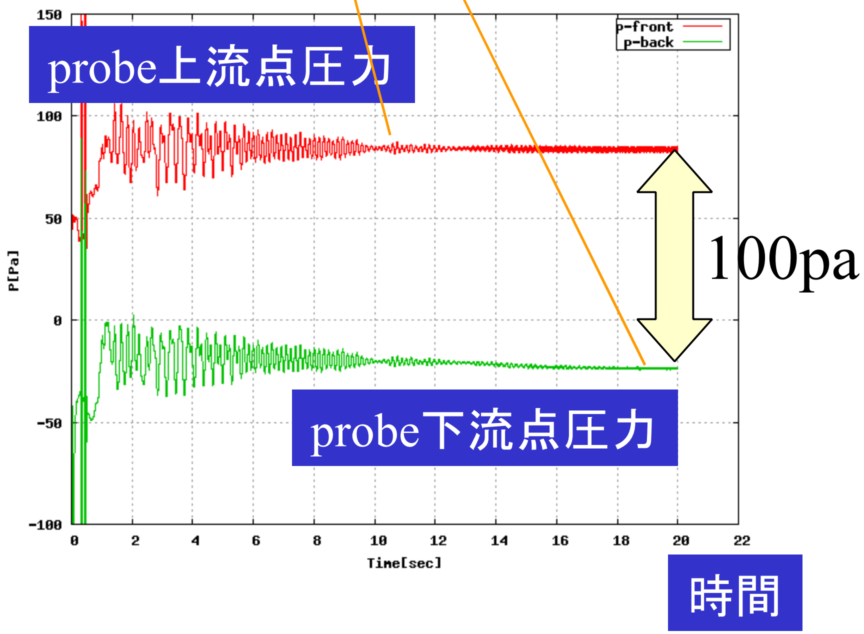
梁の先端位置
barTop
Displacement[mm]
Time[sec]
60mm
時間



U Magnitude
10
20
0
20.63
Time: 19.002000
p
-100
0
-122.612
97.24205



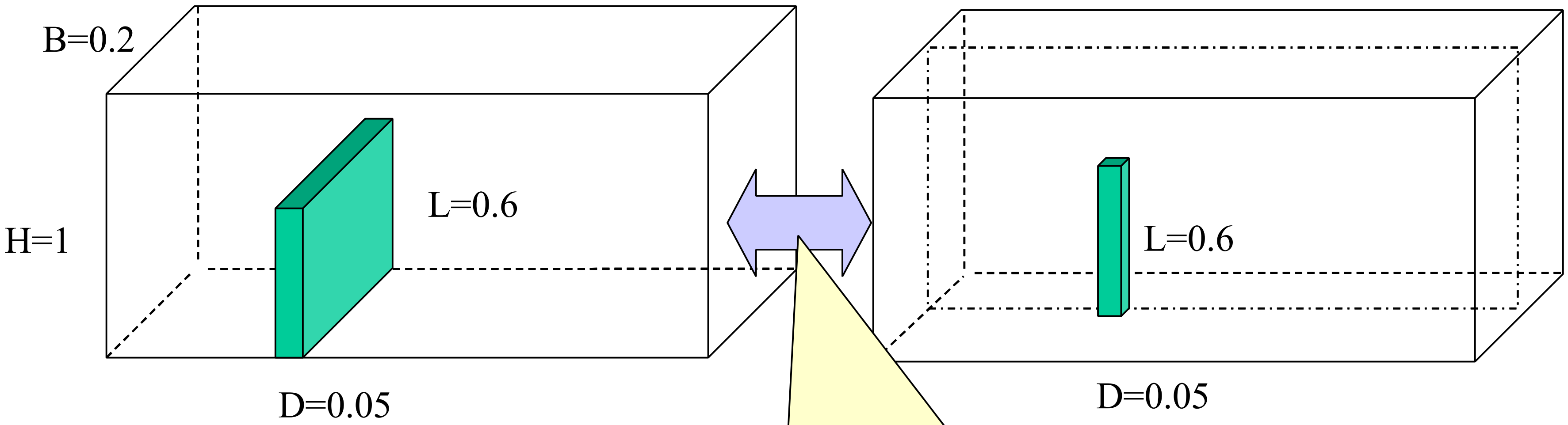
Force-X
Force-Y
Force-Z
Force[N]
Time[sec]
流体力（X方向）
流体力（Y方向）
14N
時間



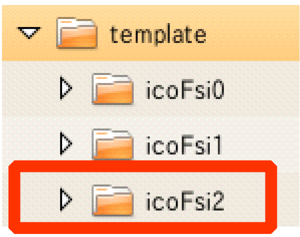
p-front
p-back
P[Pa]
Time[sec]
probe上流点圧力
probe下流点圧力
100pa
時間

考察1

公開ケース（2D）　　　解析したい事象？（3D）

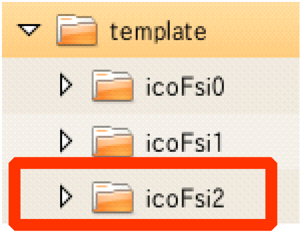


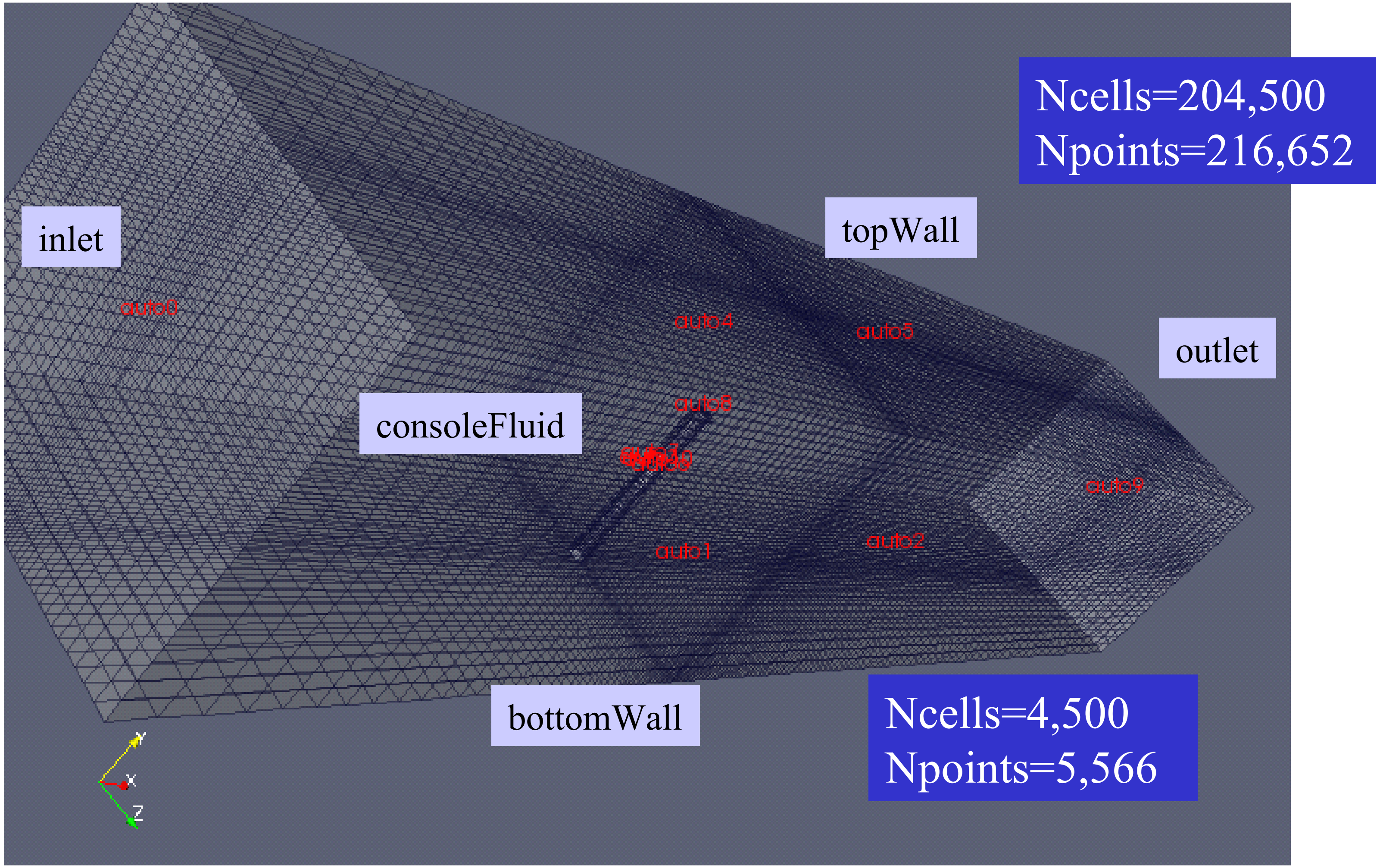
曲げの固有振動数は同一
しかし、流れ場が大きく異なる



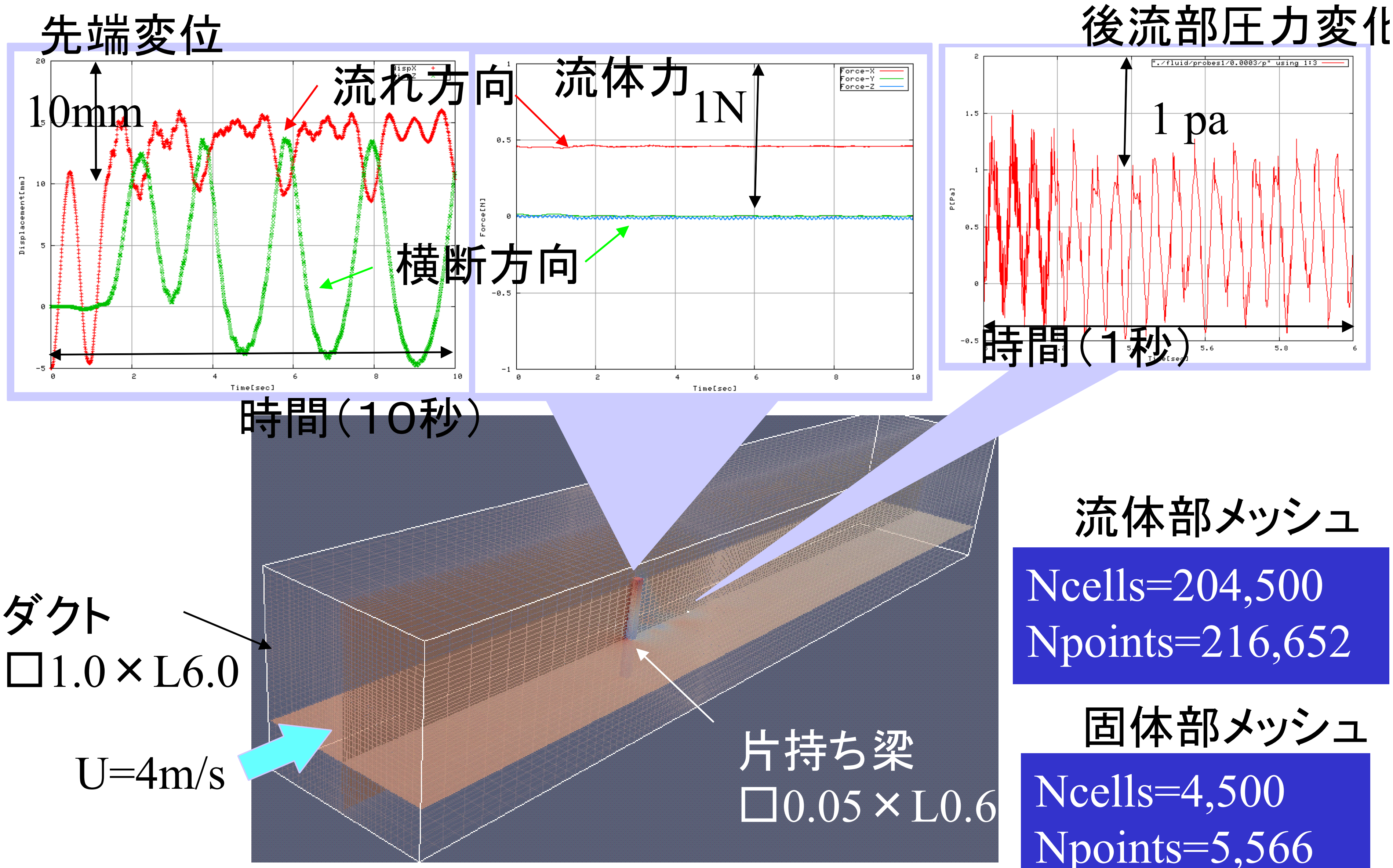
3Dで計算してみよう！

解析例2（3D）


template
icoFsi0
icoFsi1
icoFsi2


Ncells=204,500
Npoints=216,652
inlet
topWall
outlet
consoleFluid
bottomWall
Ncells=4,500
Npoints=5,566
auto0
auto4
auto5
auto8
auto9
auto1
auto2

解析例2（3D）（参考）



流体部メッシュ

Ncells=204,500
Npoints=216,652

固体部メッシュ

Ncells=4,500
Npoints=5,566

3D計算の補足

mechanicalProperties

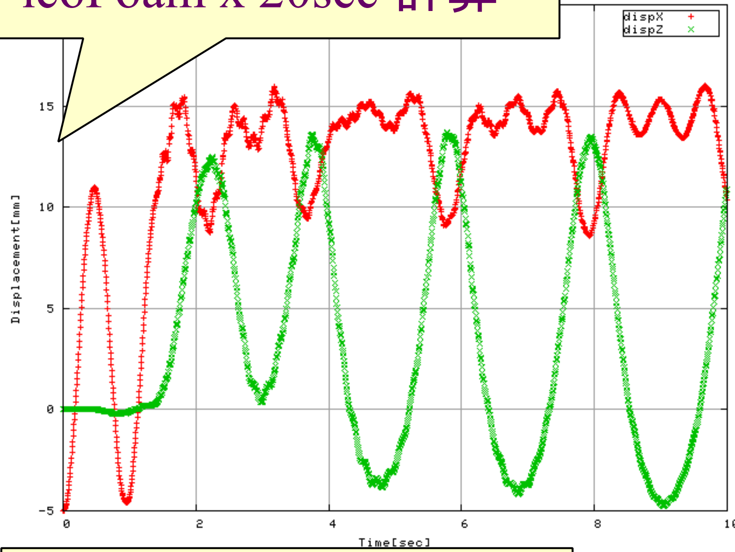
```
    instance         "";
    local            "";

    class            dictionary;
    object           mechanicalProperties;
}

// * * * * * * * * * * * * * * * * * * * * * * *

rho              rho [1 -3 0 0 0 0 0] 1000;

nu               nu [0 0 0 0 0 0 0] 0.3;

E                E [1 -1 -2 0 0 0 0] 2e+6;

planeStress      yes;

```

dynamicMeshDict

```
// * * * * * * * * * * * * * * * * * * * * * * *

dynamicFvMesh dynamicMotionSolverFvMesh;

twoDMotion       yes;

solver           laplaceFaceDecomposition;

diffusivity      quadratic;

frozenDiffusion on;

distancePatches
(
    consoleFluid
);
```

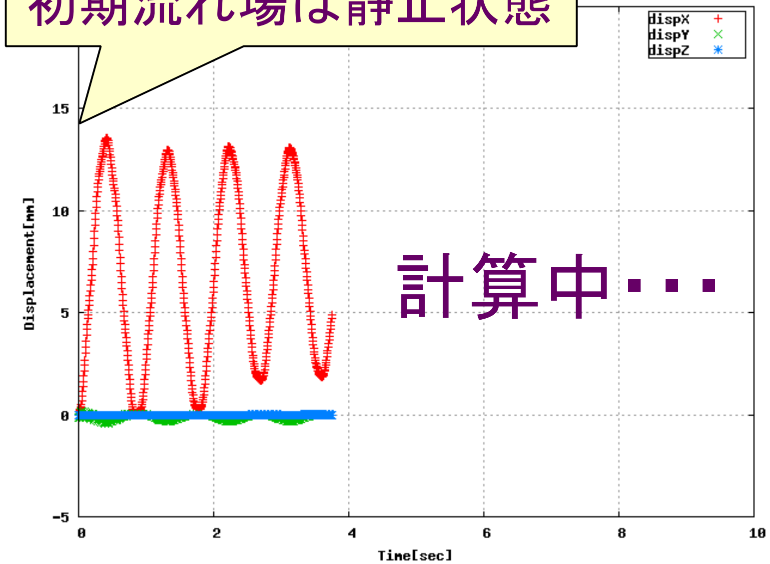
yes

初期流れ場は
icoFoam x 20sec 計算



no

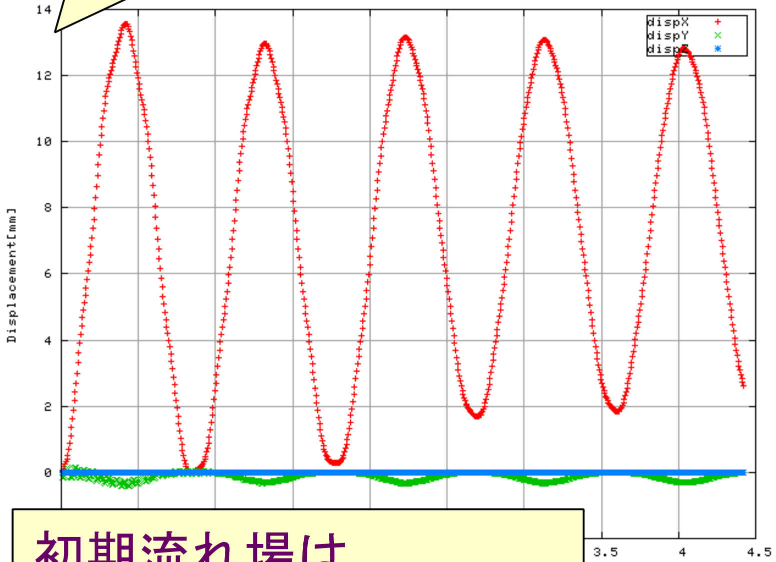
初期流れ場は静止状態

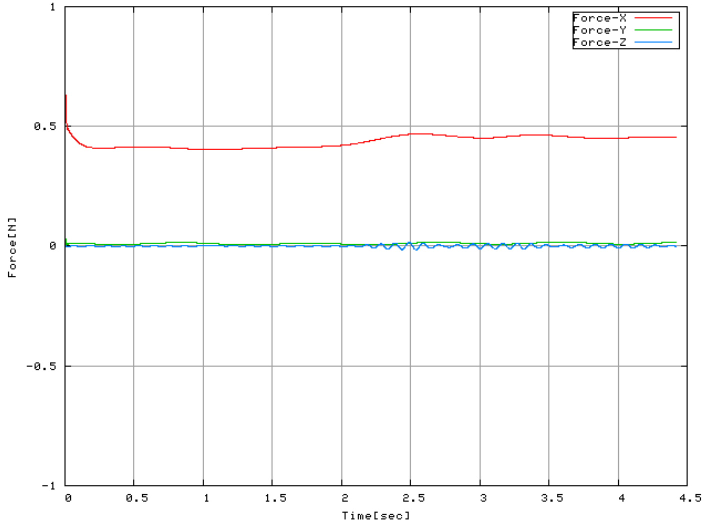


計算中・・・

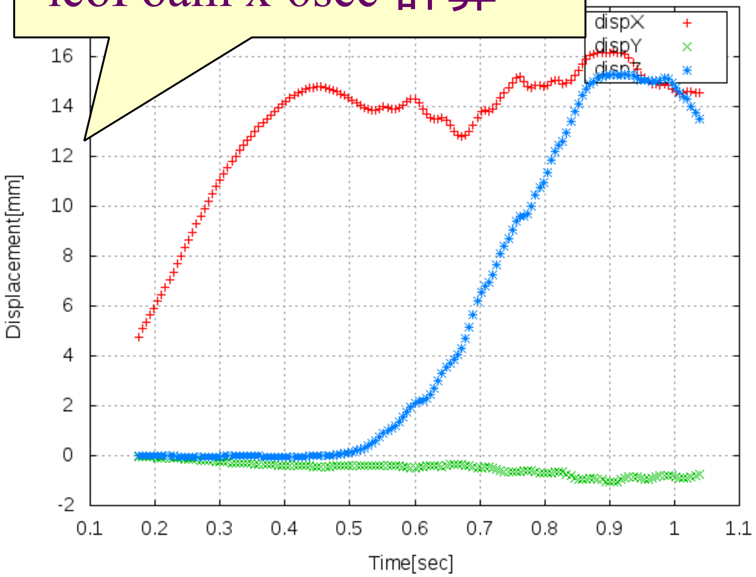
再計算の状況（3D）

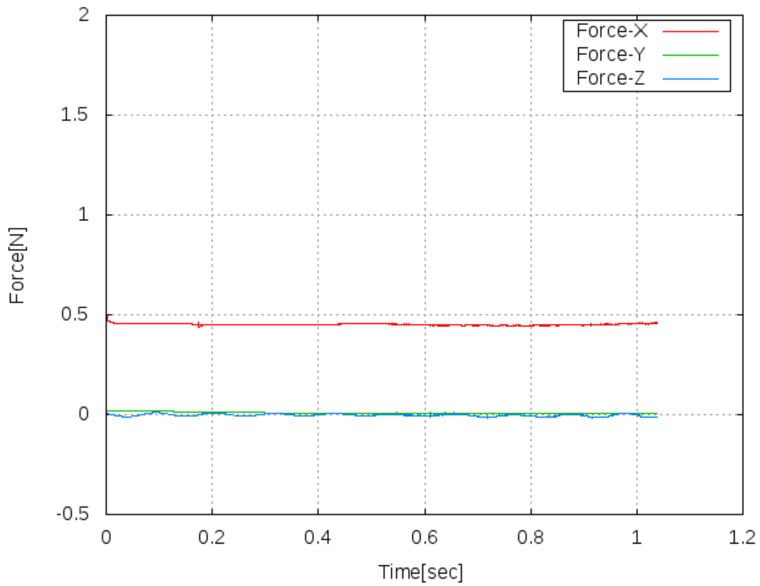
初期流れ場は静止状態





初期流れ場は
icoFoam x 6sec 計算

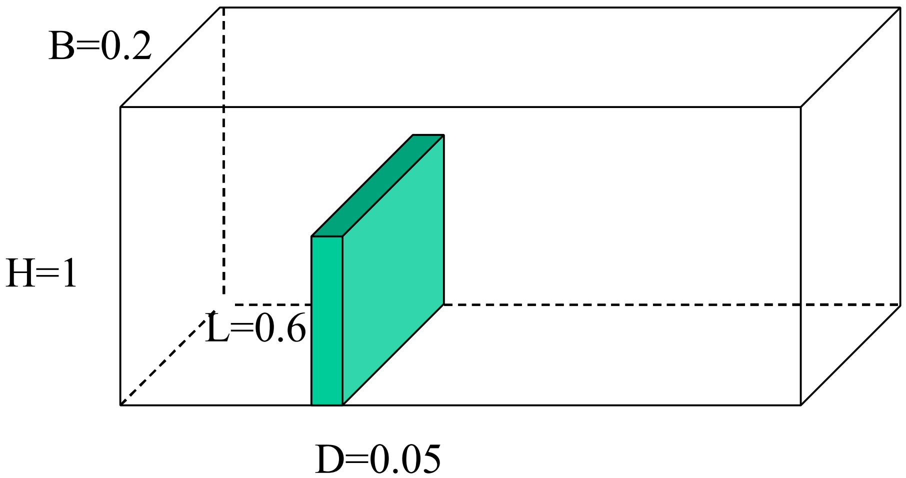


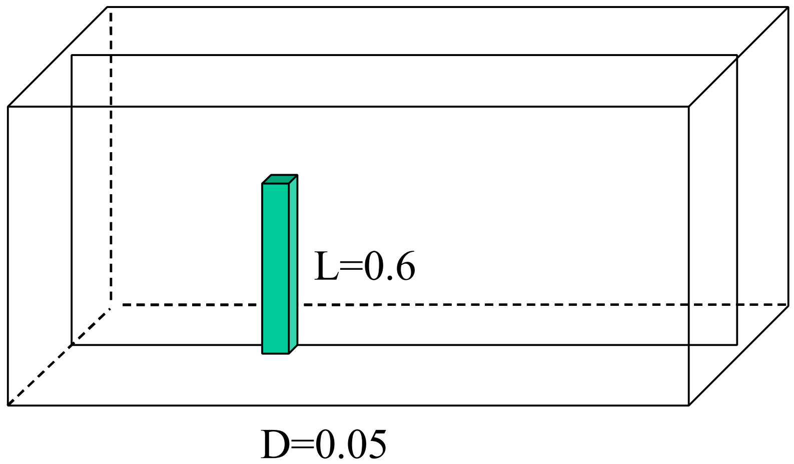


考察2 （2D計算で何とかできないか？）

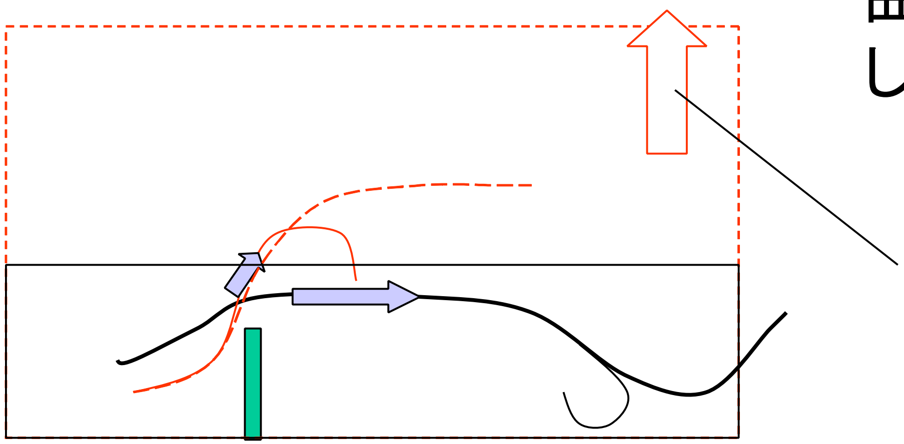
公開ケース（2D）

解析したい事象？（3D）





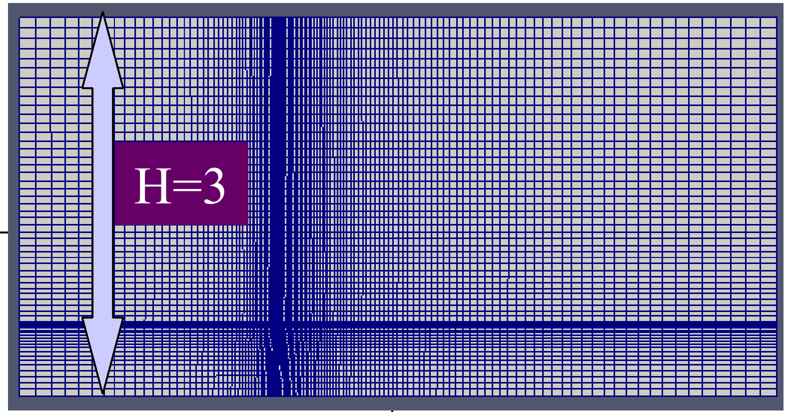
曲げの固有振動数は同一
しかし、流れ場が大きく異なる



解析領域の拡大はどうか？

解析例3　blockMesh

40分割



blockMeshDict

```
vertices
(
    (-2    0    -0.1)
    (0     0    -0.1)
    (0.05  0    -0.1)
    (4     0    -0.1)
    (-2    0.6  -0.1)
    (0     0.6  -0.1)
    (0.05  0.6  -0.1)
    (4     0.6  -0.1)
    (-2    3    -0.1)
    (0     3    -0.1)
    (0.05  3    -0.1)
    (4     3    -0.1)

    (-2    0     0.1)
    (0     0     0.1)
    (0.05  0     0.1)
    (4     0     0.1)
    (-2    0.6   0.1)
    (0     0.6   0.1)
    (0.05  0.6   0.1)
    (4     0.6   0.1)
    (-2    3     0.1)
    (0     3     0.1)
    (0.05  3     0.1)
    (4     3     0.1)
);
```

```
blocks
(
    hex (0 1 5 4 12 13 17 16) (40 20 1) simpleGrading (0.1 0.2 1)
    hex (2 3 7 6 14 15 19 18) (80 20 1) simpleGrading (10 0.2 1)
    hex (4 5 9 8 16 17 21 20) (40 40 1) simpleGrading (0.1 2 1)
    hex (5 6 10 9 17 18 22 21) (5 40 1) simpleGrading (1 2 1)
    hex (6 7 11 10 18 19 23 22) (80 40 1) simpleGrading (10 2 1)
);
```

□部を変更

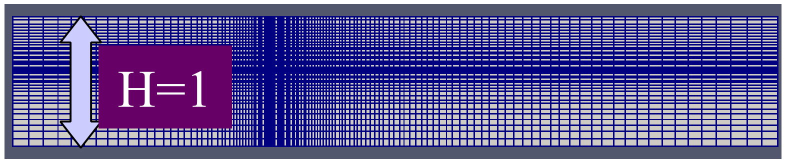
blockMeshDict

```
vertices
(
    (-2    0    -0.1)
    (0     0    -0.1)
    (0.05  0    -0.1)
    (4     0    -0.1)
    (-2    0.6  -0.1)
    (0     0.6  -0.1)
    (0.05  0.6  -0.1)
    (4     0.6  -0.1)
    (-2    1    -0.1)
    (0     1    -0.1)
    (0.05  1    -0.1)
    (4     1    -0.1)

    (-2    0     0.1)
    (0     0     0.1)
    (0.05  0     0.1)
    (4     0     0.1)
    (-2    0.6   0.1)
    (0     0.6   0.1)
    (0.05  0.6   0.1)
    (4     0.6   0.1)
```

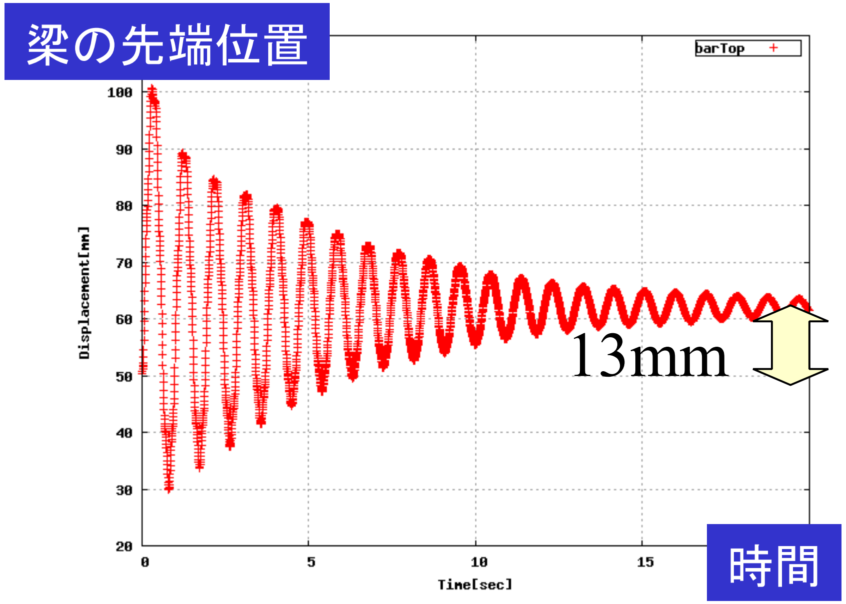
```
blocks
(
    hex (0 1 5 4 12 13 17 16) (40 20 1) simpleGrading (0.1 0.2 1)
    hex (2 3 7 6 14 15 19 18) (80 20 1) simpleGrading (10 0.2 1)
    hex (4 5 9 8 16 17 21 20) (40 20 1) simpleGrading (0.1 2 1)
    hex (5 6 10 9 17 18 22 21) (5 20 1) simpleGrading (1 2 1)
    hex (6 7 11 10 18 19 23 22) (80 20 1) simpleGrading (10 2 1)
);
```

（解析例1）

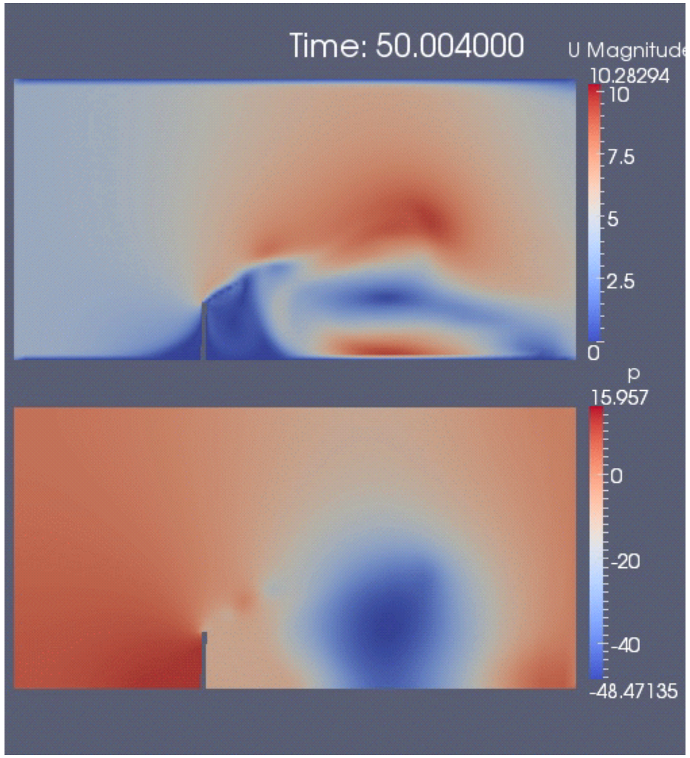
20分割



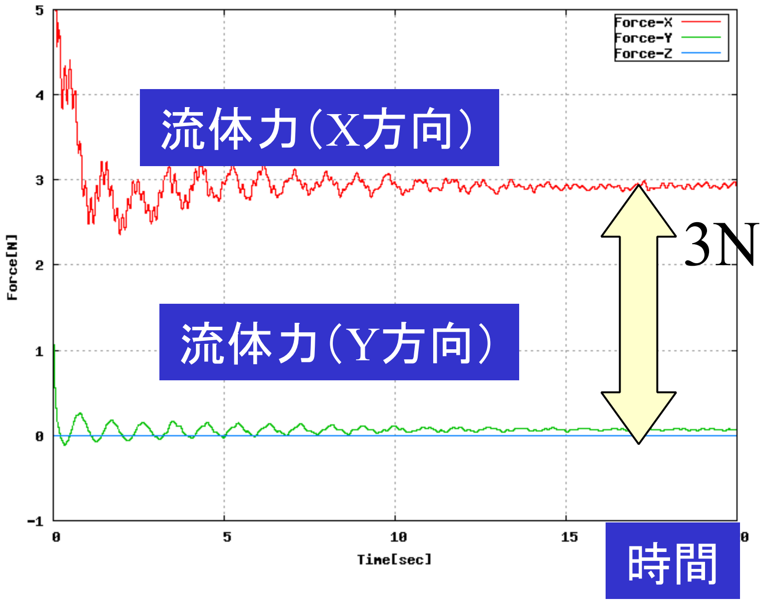
解析例3



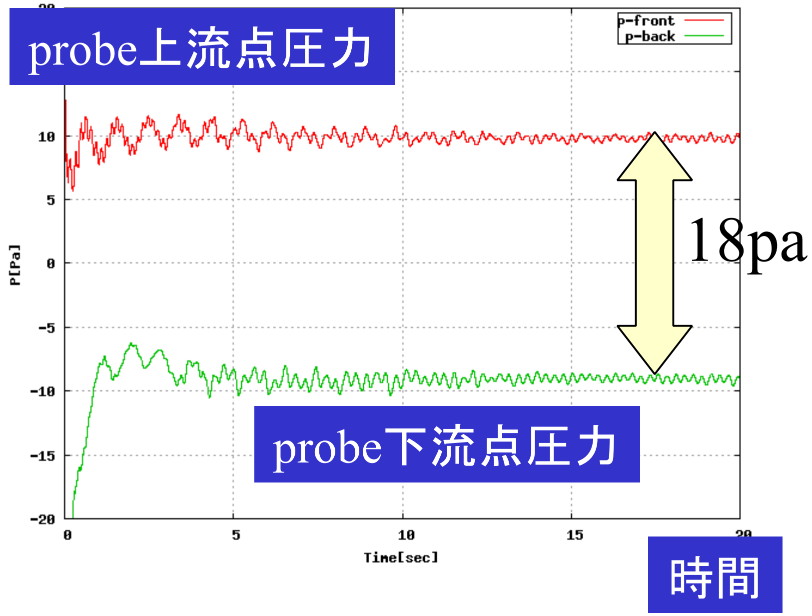
梁の先端位置
barTop
Displacement[mm]
Time[sec]
13mm
時間



Time: 50.004000
U Magnitude
10.28294
0
p
15.957
-48.47135

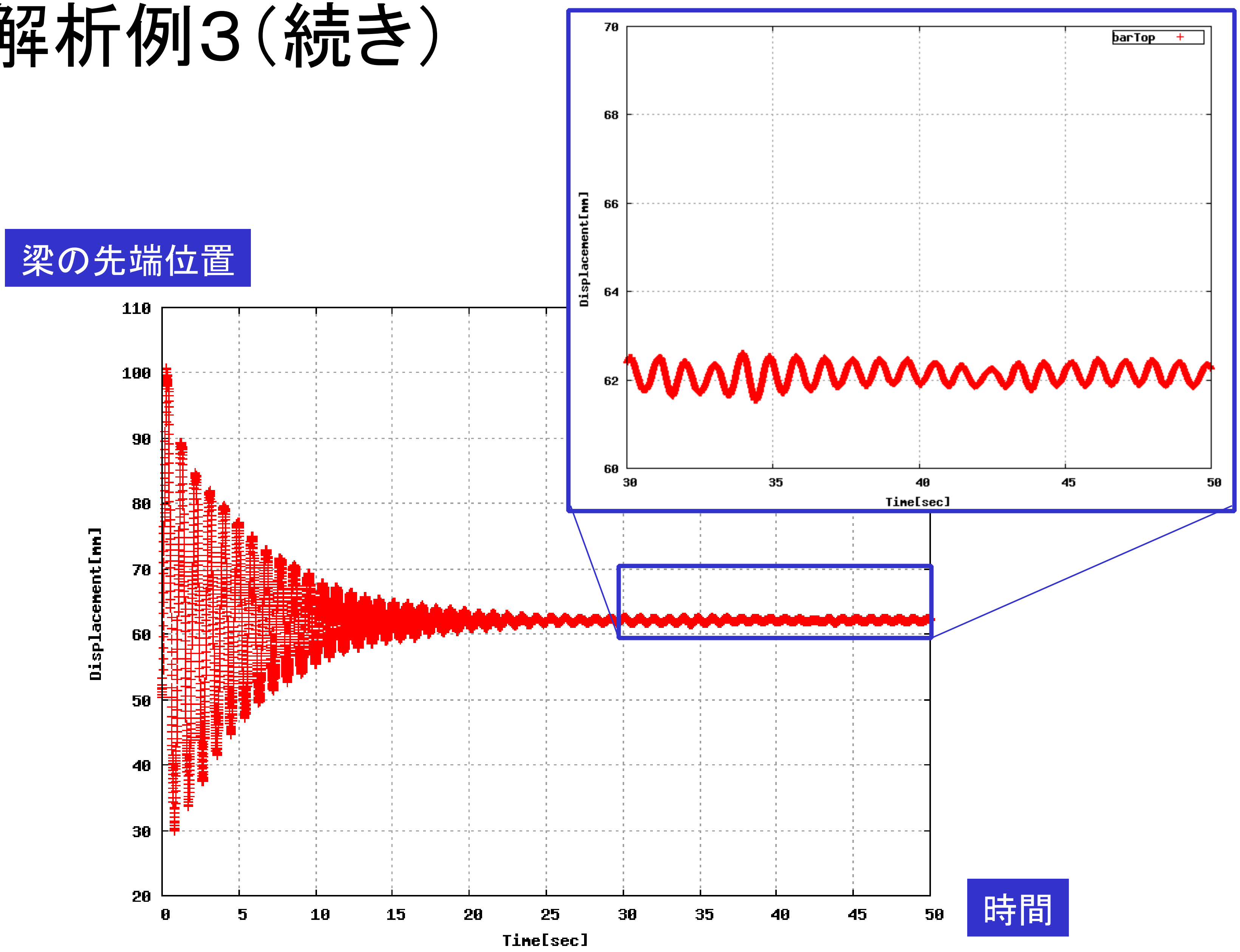


Force-X
Force-Y
Force-Z
流体力（X方向）
流体力（Y方向）
3N
Force[N]
Time[sec]
時間



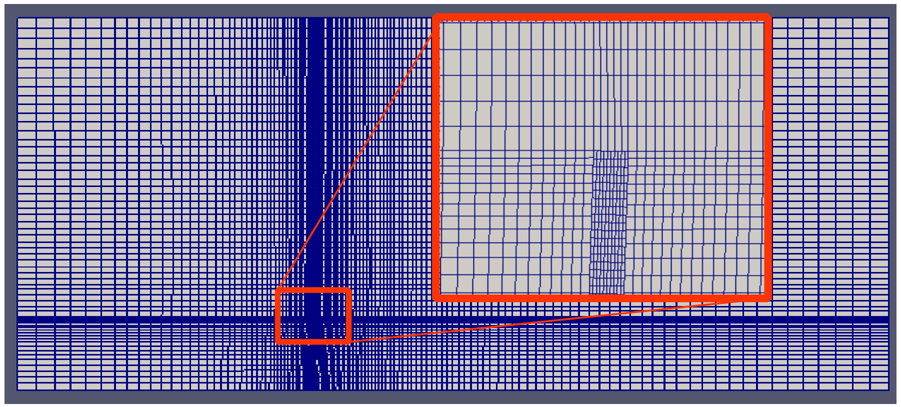
probe上流点圧力
p-front
p-back
18pa
P[Pa]
probe下流点圧力
Time[sec]
時間

解析例3(続き)



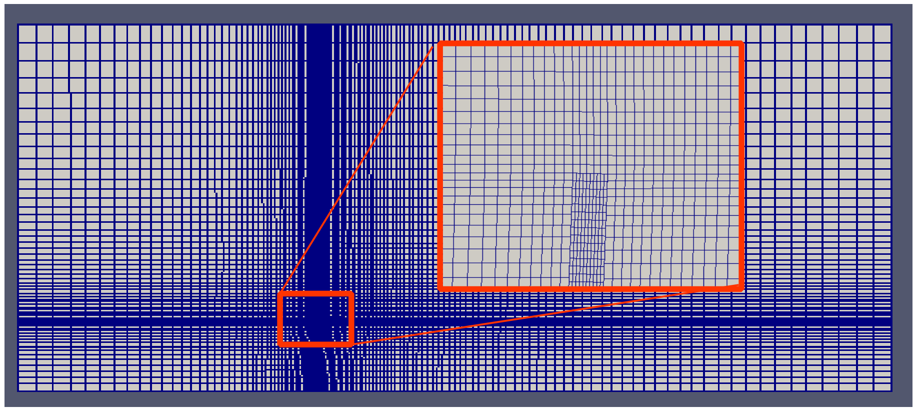
解析例3

```
blocks
(
    hex (0 1 5 4 12 13 17 16) (40 20 1) simpleGrading (0.1 0.2 1)
    hex (2 3 7 6 14 15 19 18) (80 20 1) simpleGrading (10 0.2 1)
    hex (4 5 9 8 16 17 21 20) (40 20 1) simpleGrading (0.1 2 1)
    hex (5 6 10 9 17 18 22 21) (5 20 1) simpleGrading (1 2 1)
    hex (6 7 11 10 18 19 23 22) (80 20 1) simpleGrading (10 2 1)
);
```

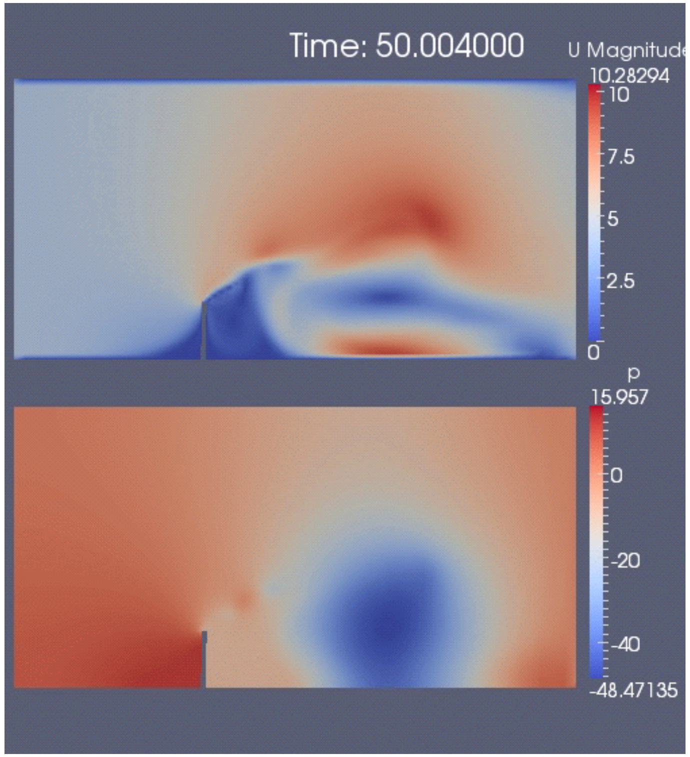


解析例3改

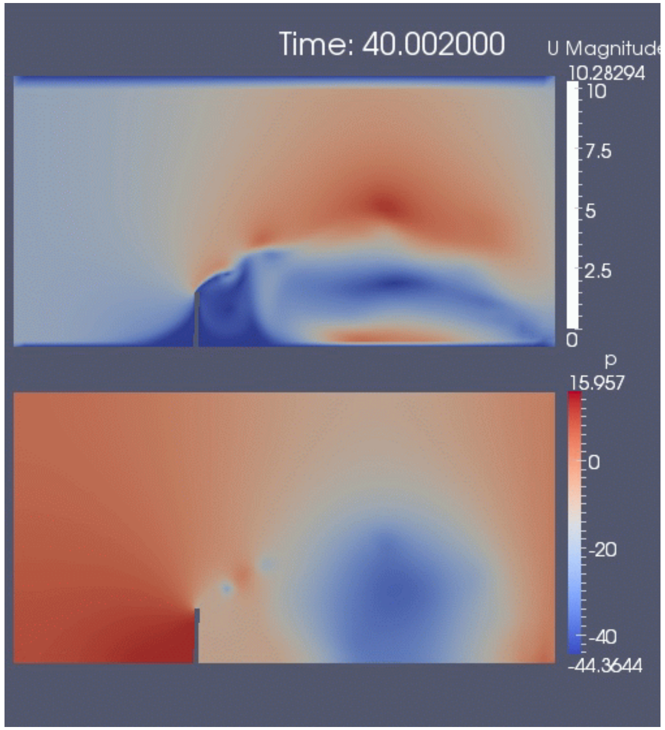
```
ks

ex (0 1 5 4 12 13 17 16) (40 20 1) simpleGrading (0.1 0.2 1)
ex (2 3 7 6 14 15 19 18) (80 20 1) simpleGrading (10 0.2 1)
ex (4 5 9 8 16 17 21 20) (40 40 1) simpleGrading (0.1 10 1)
ex (5 6 10 9 17 18 22 21) (5 40 1) simpleGrading (1 10 1)
ex (6 7 11 10 18 19 23 22) (80 40 1) simpleGrading (10 10 1)
```

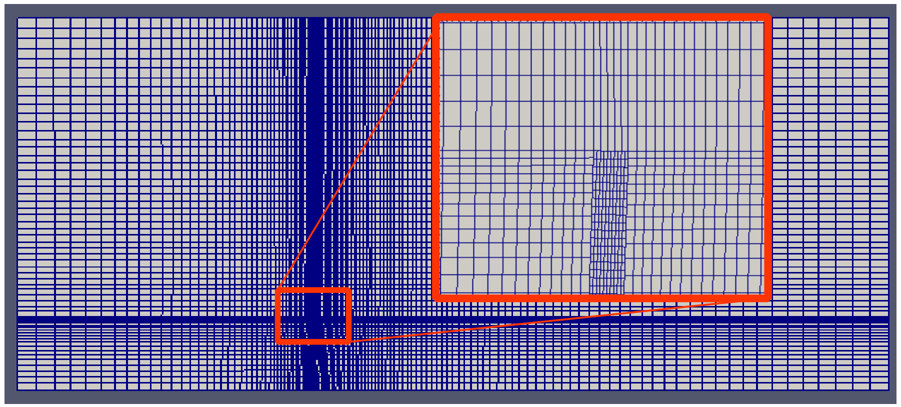


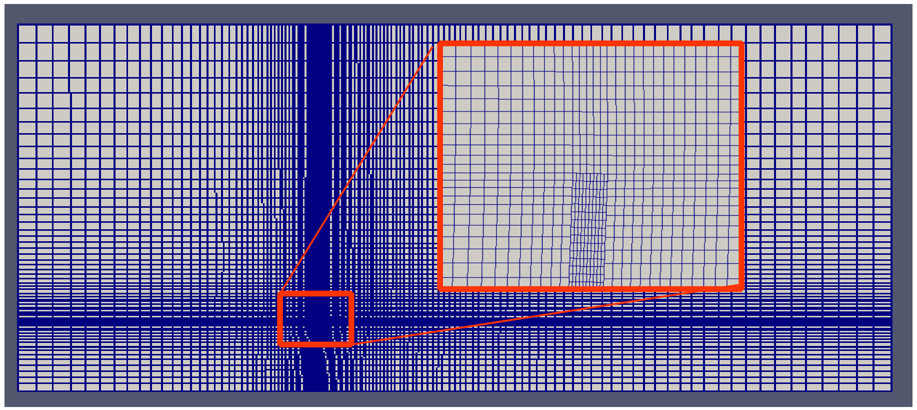
解析例3


Time: 50.004000
U Magnitude
10.28294
10
7.5
5
2.5
0
p
15.957
0
-20
-40
-48.47135

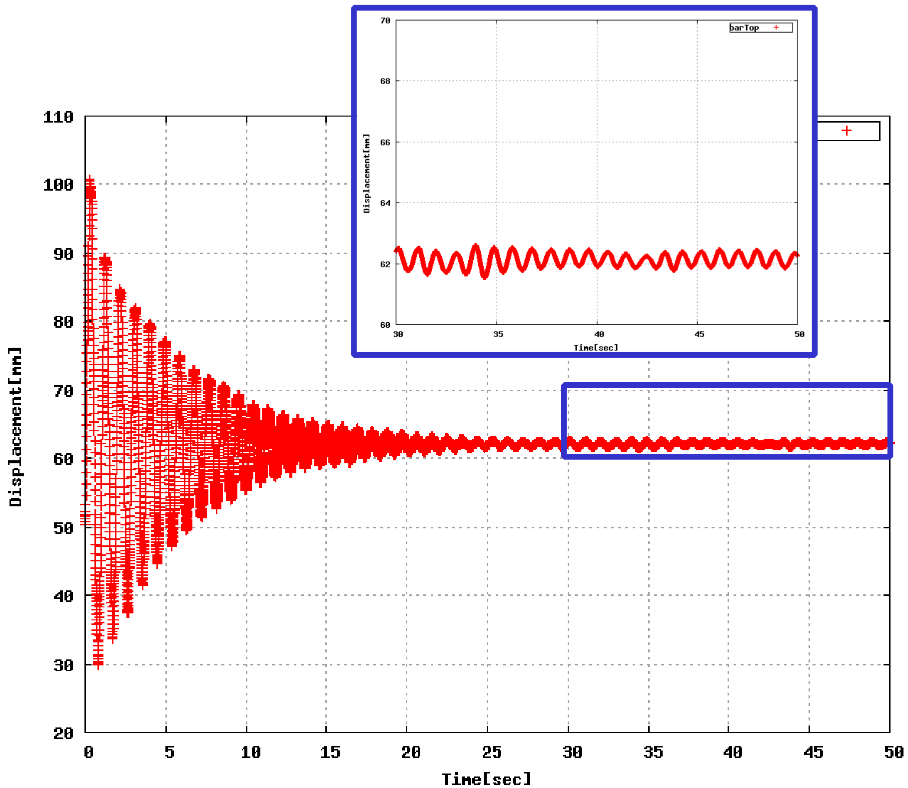
解析例3改


Time: 40.002000
U Magnitude
10.28294
10
7.5
5
2.5
0
p
15.957
0
-20
-40
-44.3644

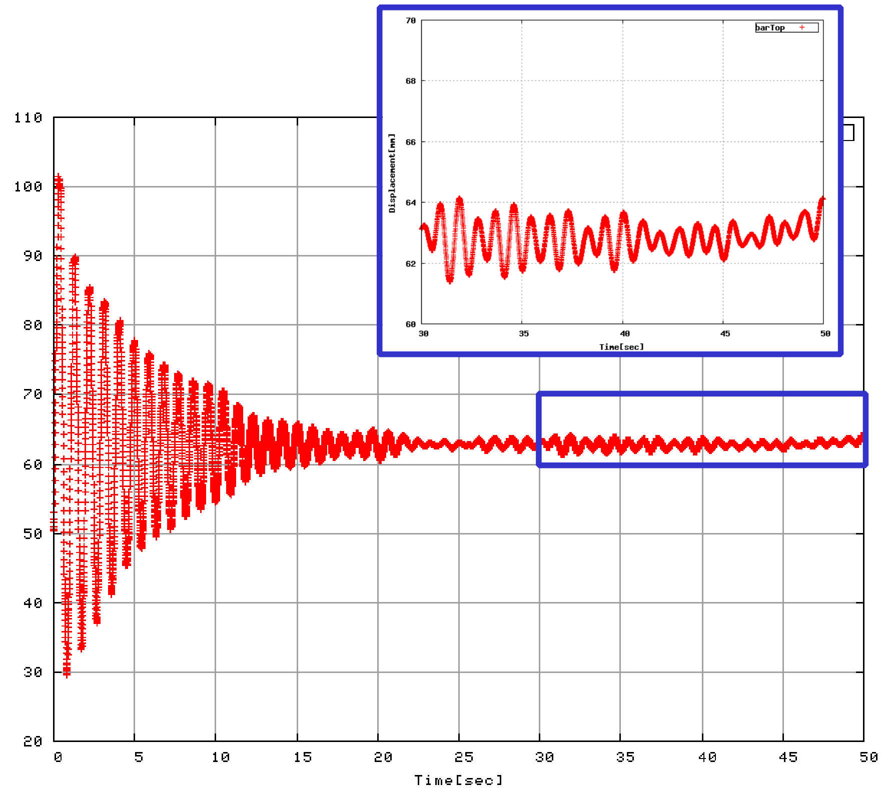




解析例3


barTop
Displacement[mm]
Time[sec]

解析例3改

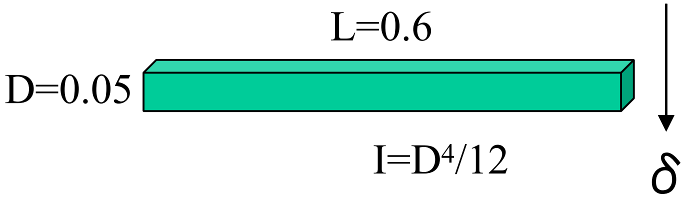

barTop
Displacement[mm]
Time[sec]

まとめ

1. OpenFOAM-1.5-dev のicoFsiFoam を使って、流体構造連成解析を実施した
2. 一般公開情報をそのまま使った解析では問題があり、さらなる工夫が必要であった
3. これまでの実施例では、概ね合理的な計算結果が得られているが、十分な検証が出来ているとは言い難い
4. 非定常計算には長大な計算時間がかかるが、icoFsiFoamは並列計算に対応できておらず、実用面では問題になる
5. 今回取り上げなかった icoStructFoam（OF-1.6/1.7で動作OK）もソルバーの改変や使い方の工夫で適用可能性はありそう

参考資料

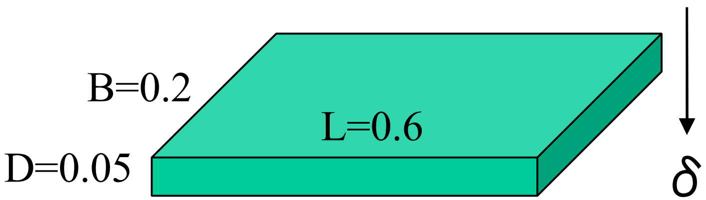
片持ちはりの変形



I=D⁴/12

$$\delta = WL^4/8EI$$
$$= \frac{0.6^4}{8\times 2000000\times 0.05^4/12}\times W$$
$$= 0.0155\times W$$

E=2000000



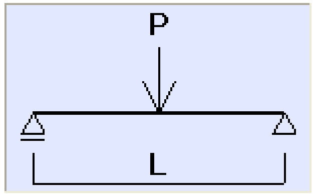
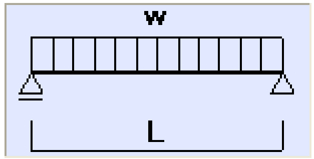
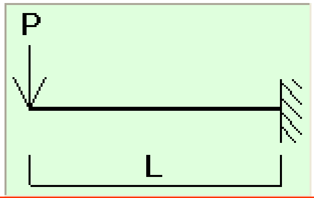
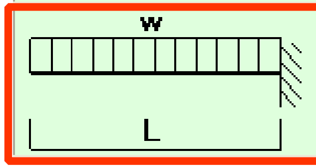
I=BD³/12

$$\delta = WL^4/8EI$$
$$= \frac{0.6^4}{8\times 2000000\times 0.2\times 0.05^3/12}\times W$$
$$= 0.0039\times W$$

E=2000000

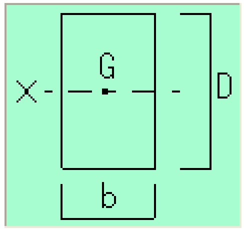
http://bit.ly/aJ8aX4

http://kozo.milkcafe.to/rikigaku2/henkei.html

梁	最大たわみ角θ	最大たわみδ
	$\frac{PL^2}{16EI}$	$\frac{PL^3}{48EI}$
	$\frac{WL^3}{24EI}$	$\frac{5WL^4}{384EI}$
	$-\frac{PL^2}{2EI}$	$\frac{PL^3}{3EI}$
	$-\frac{WL^3}{6EI}$	$\frac{WL^4}{8EI}$

http://bit.ly/aOYWG2

http://kozo.milkcafe.to/rikigaku2/seinou.html

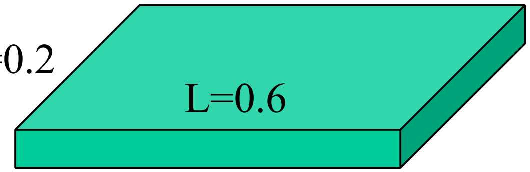
断面形状	断面2次モーメント
	$Ix=\frac{bD^3}{12}$

http://bit.ly/cGX49o

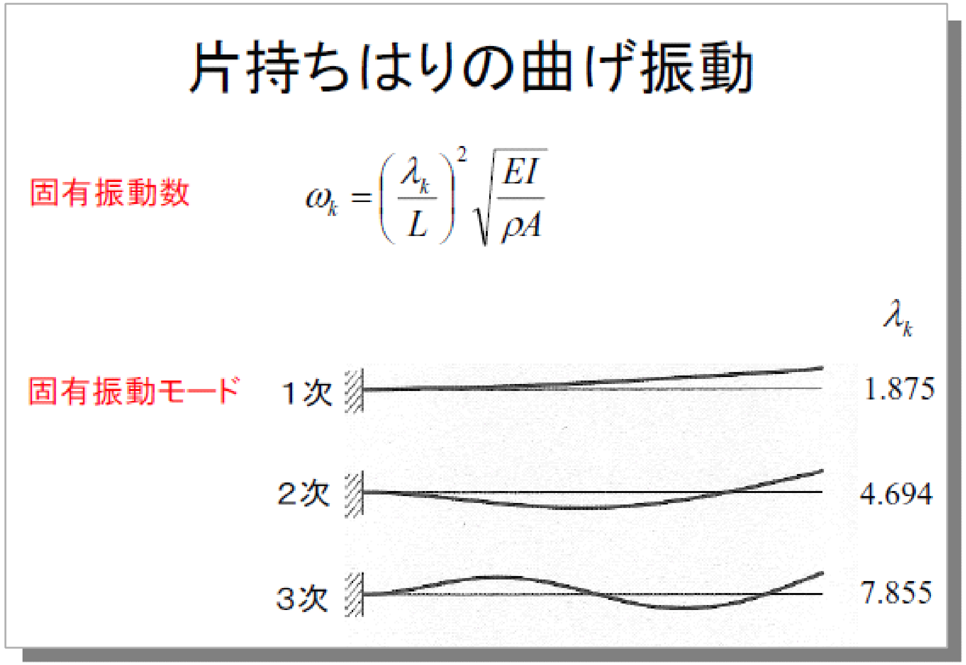
片持ちはりの曲げ振動

http://www.mech.usp.ac.jp/~hnw/model/kogi/2006/kiriki2_12.pdf

$A = D^2$, $I = D^4/12$ } $I/A = D^2/12$



$A = BD$, $I = BD^3/12$ } $I/A = D^2/12$



$$\omega = \left(\frac{\lambda}{L}\right)^2 \sqrt{\frac{EI}{\rho A}} = \left(\frac{\lambda}{L}\right)^2 \sqrt{\frac{ED^2}{12\rho}}$$

$$= \left(\frac{1.875}{0.6}\right)^2 \sqrt{\frac{2000000 \cdot 0.05^2}{12 \cdot 1000}} = 6.67 \quad \text{rad/sec}$$

$E = 2000000$

$\rho = 1000$

カルマン渦の放出周波数

http://bit.ly/8XrmKS

http://www.inss.co.jp/seika/pdf/7/201.pdf

この渦の放出周波数f_sはUとDによって次式により無次元化され，ストローハル数Stと呼ぶ.

$$St = \frac{f_s D}{U} \qquad (1)$$

St数は一般にRe数の関数として実験的に求められる．図２に実験値を示す.

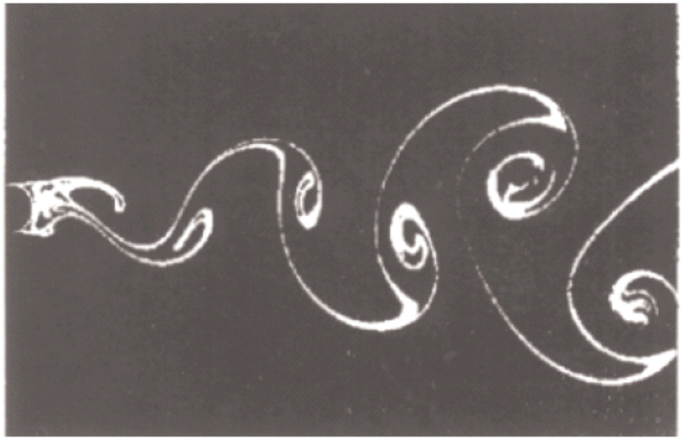


図１　カルマン渦 [4]

(4)　種子田定俊：「画像から学ぶ流体力学」，朝倉書店　(1988)．

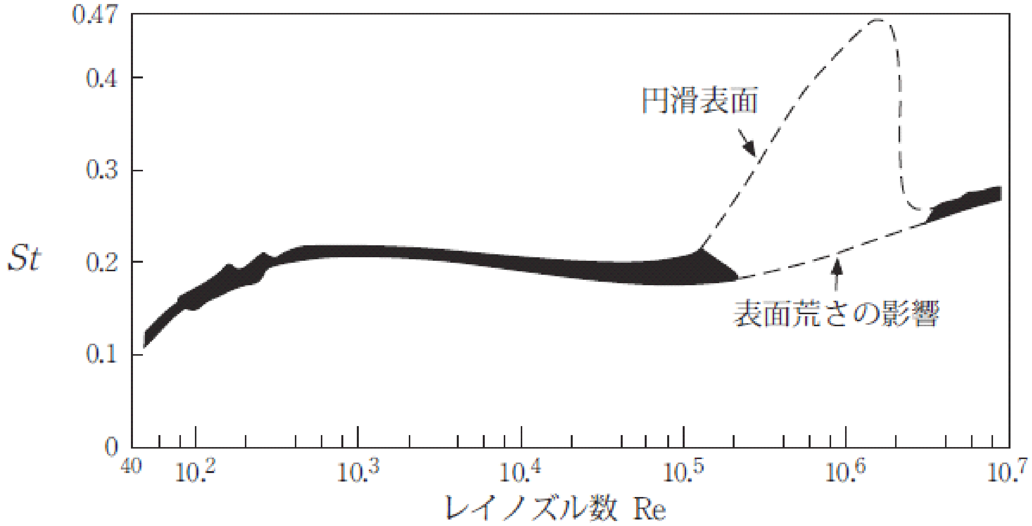


図2　レイノルズ数に対するストローハル数 [5]

(5)　R. D. Blevins: "Flow-induced vibration", Krieger publishing company　(1990)．

D=0.05
ν =0.001
U=4
Re=200

St=0.2

fs=St･U/D=16

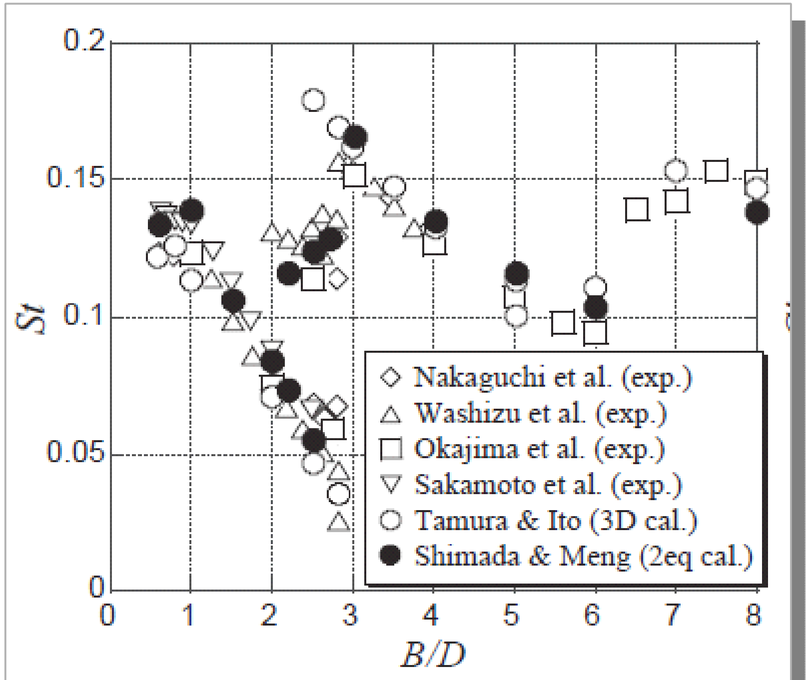
ストローハル数（角柱の場合）

http://bit.ly/b86pJE

http://ci.nii.ac.jp/els/110002399378.pdf?id=ART0002680678&type=pdf&lang=jp&host=cinii&order_no=&ppv_type=0&lang_sw=&no=1280810119&cp=

http://bit.ly/bsNgzx

http://www.nagare.or.jp/download/noauth.html?d=22-1-t01.pdf&dir=36



(b) Strouhal数

辺長比B/D（B：角柱の幅，D：奥行き）

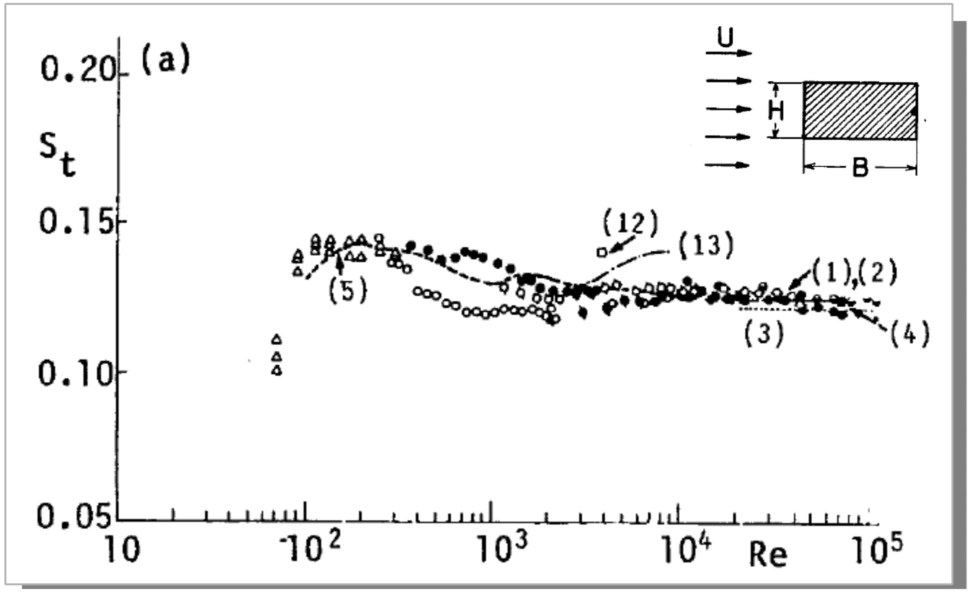


図 2　$B/H=1.0$ 正方形断面柱の（a）　S_t 数

D=0.05

ν =0.001

U=4

Re=200

St=0.13

fs=St・U/D=10.4

「もんじゅ」事故と原因究明の現状･･･より

http://www.jaea.go.jp/04/monju/category05/mj_accirep/mj_accirep17.html

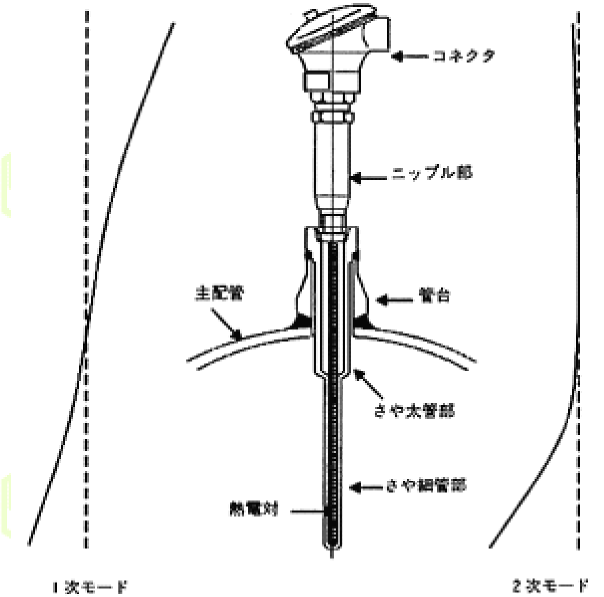


図4-2 温度計の固有振動モード(ナトリウム中、200度の例)

温度	1次固有振動数(Hz)		2次固有振動数(Hz)	
	流れ直角方向	流れ方向	流れ直角方向	流れ方向
200℃	163	173	257	257
325℃	160	170	251	251
485℃	157	164	241	241

表4-1 温度計の固有振動数

二次主冷却系の温度計は，主配管の横腹に設けられた管台に溶接され，温度計さやが配管内に約185mm突き出した構造となっている。このうち，さやの先端約150mmの部分は，直径が10mmと細くなっている（図４－２参照）。

http://www.jaea.go.jp/04/monju/category05/mj_accirep/mj_accirep18.html

		100%流量試験（200℃等温）	100%流量試験（325℃等温）	40%流量試験（485℃）
Na流速	v	5.2 m/s	5.2 m/s	2.2 m/s
Na密度	ρ	904 kg/m^3	874 kg/m^3	836 kg/m^3
レイノルズ数	Re	1.0×10^5	1.4×10^5	7.2×10^4
固有振動数（流体質量効果を考慮）	f	272Hz (257Hz)	265Hz (251Hz)	254Hz (241Hz)

表4-2 プラントの運転状態

http://www.jaea.go.jp/04/monju/category05/mj_accirep/mj_accirep19.html

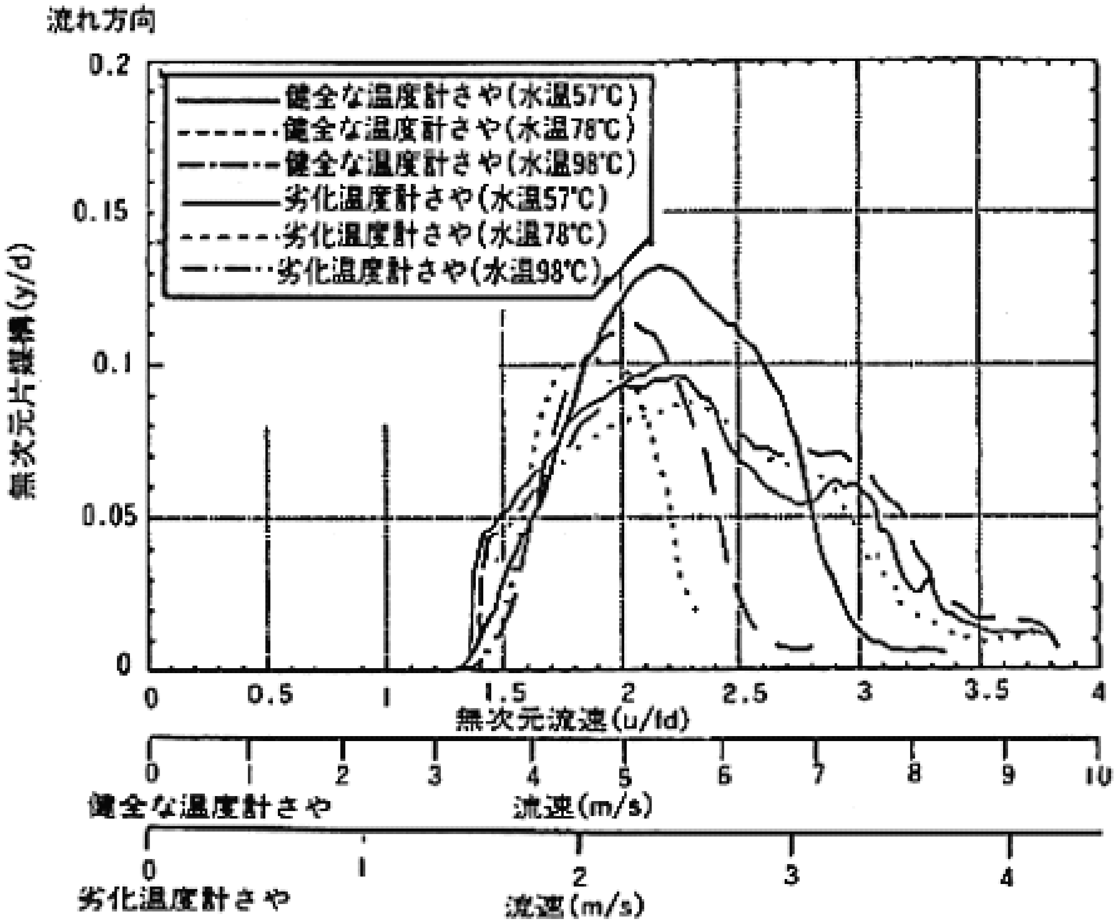
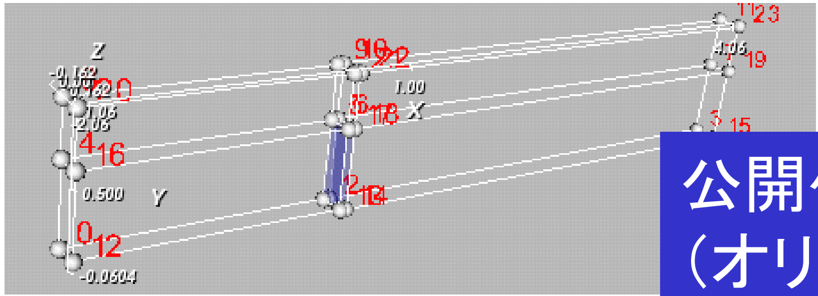
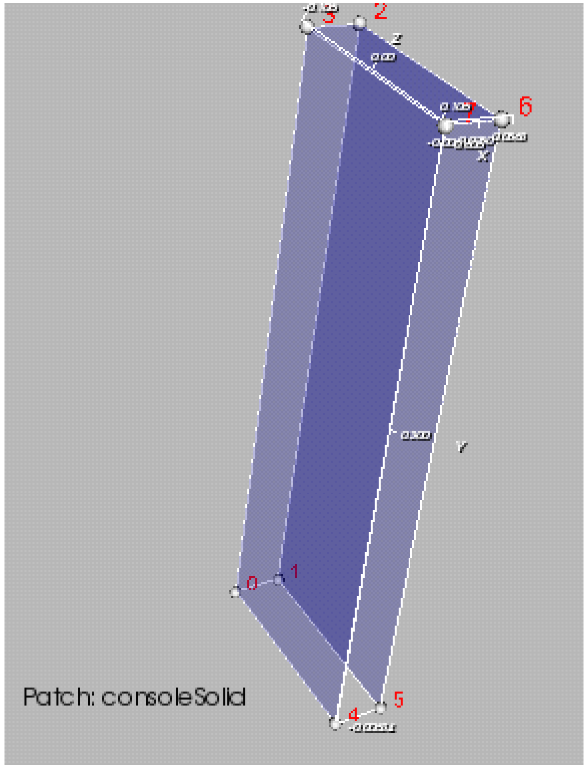


図4-9 温度計さや先端変位振幅と無次元流速の関係

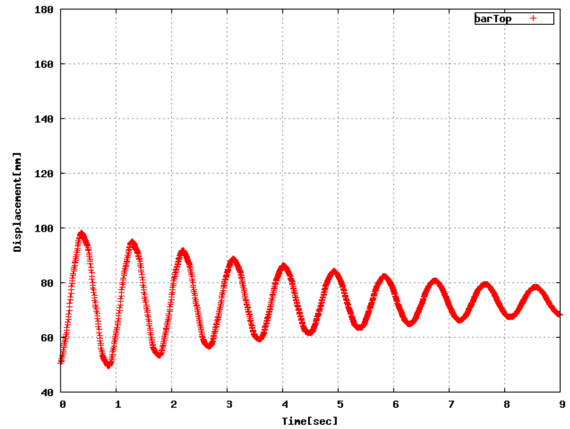
FSI境界面定義方法の問題





公開ケース（オリジナル）

```
patches
(
    patch consoleSolid
    (
        (3 7 6 2)
        (0 4 7 3)
        (2 6 5 1)
    )
```



DEXCS

```
patches
(
    patch consoleSolid
    (
        (0 4 7 3)
        (2 6 5 1)
        (3 7 6 2)
    )
```

autoPatch
⇒createPatch