How to get started with **OpenFOAM** at **SHARCNET**

Isaac Ye, High Performance Technical Consultant
SHARCNET, York University
isaac@sharcnet.ca

Outlines

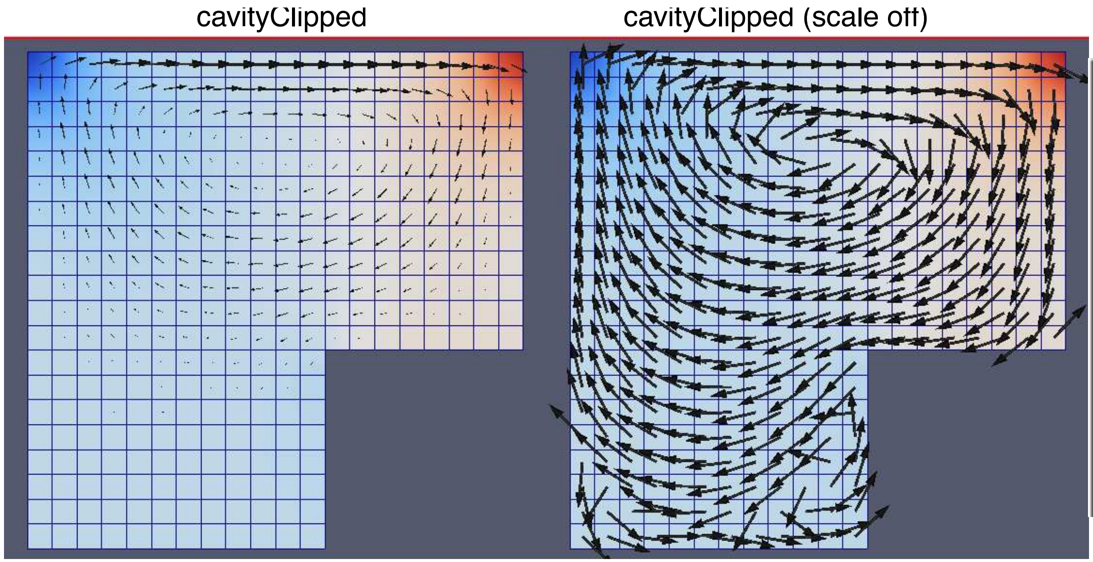
- **Introduction to OpenFOAM**
- **Compilation in SHARCNET**
- **Pre/Post-Processing with Paraview**
- **Running jobs**
- **Compiling user-defined local solver**

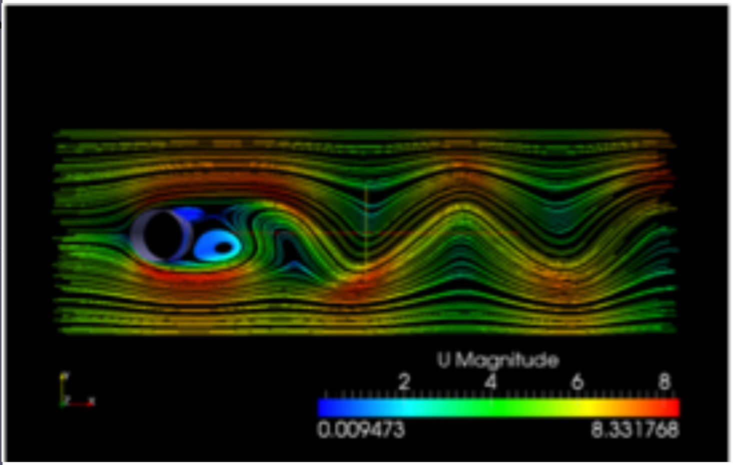
Introduction to OpenFOAM

- WHAT CAN DO?
- PROGRAM STRUCTURE
- THINGS TO KNOW TO RUN IN SHARCNET CLUSTERS

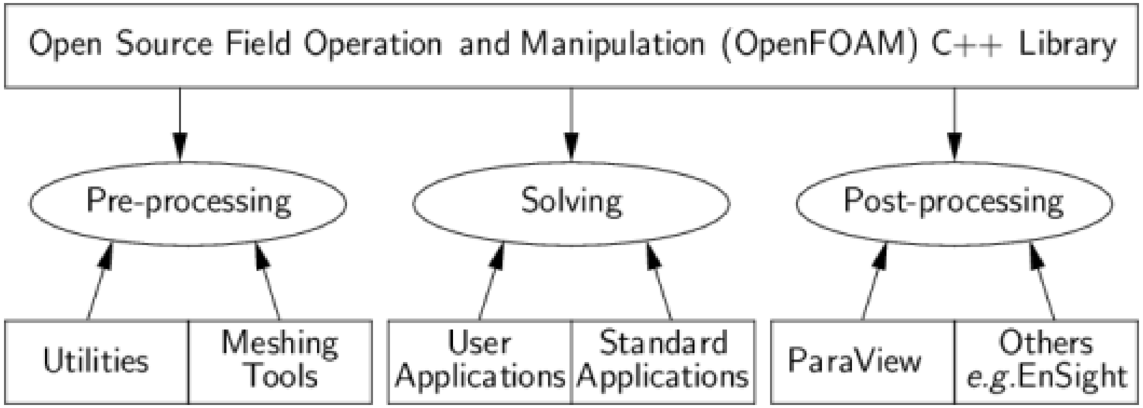
What can do?

OpenFOAM is a C++ toolbox for the development of customized numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems, including computational fluid dynamics (CFD).





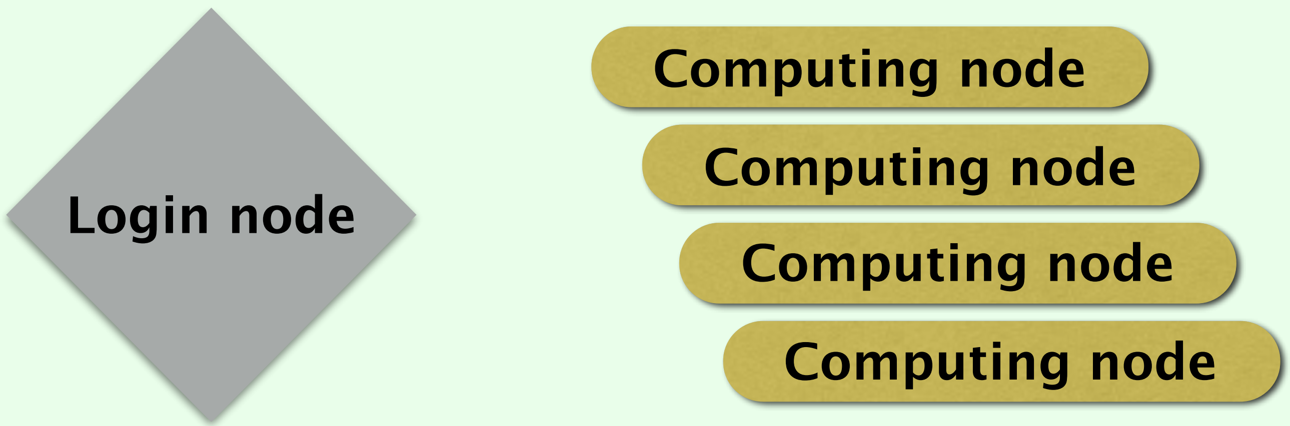
OpenFOAM Structures




```
[isaac@orc-login2:..solvers] ls
basic      compressible    DNS          financial     incompressible  multiphase
combustion  discreteMethods  electromagnetics  heatTransfer  lagrangian
stressAnalysis
```

http://www.openfoam.org/features/standard-solvers.php

SHARCNET CLUSTERS



Login node is not for running a job
'ParaFoam' is not compilable

Should use 'sqsub' – scheduler
Paraview must be used

Installation in SHARCNET

- DOWNLOAD
- SETUP THE ENVIRONMENT
- COMPILATION

Download the current release

```
1. Create the main project directory

[1] mkdir /work/$USER/OpenFOAM

2. Change directory and download using 'wget'

[2] cd /work/$USER/OpenFOAM
[3] wget http://downloads.sourceforge.net/foam/OpenFOAM-3.0.1.tgz?use_mirror=mesh
[4] wget http://downloads.sourceforge.net/foam/ThirdParty-3.0.1.tgz?use_mirror=mesh

3. Extract the downloaded files

[5] tar zxvf OpenFOAM-3.0.1.tgz
[6] tar zxvf ThirdParty-3.0.1.tgz

4. Download 'boost'
(you can download it on you PC and upload it to your account)

[7] cd /work/$USER/ThirdParty-3.0.1
[8] cp /home/isaac/boost_1_55_0.tar.gz .
[9] tar zxvf boost_1_55_0.tar.gz
```

Setup the environment (bashrc)

```
1. bashrc

[1] cd /work/$USER/OpenFOAM/OpenFOAM-3.0.1/etc
[2] vi bashrc

        module unload intel
        module unload openmpi
        module load gcc/4.9.2
        module load openmpi/gcc-4.9.2/std/1.8.7

        export NEWHOME=/work/isaac

and change all ‘HOME’ into ‘NEWHOME’ in bashrc

Note: Please do not set ‘HOME=/home/$USER’ which will slow down all
performance.
```

Setup the environment (boost)

```
1. CGAL.sh

[1] cd /work/$USER/OpenFOAM/OpenFOAM-3.0.1/etc/config
[2] sed -i -e ‘s=boost-system=boost_1_55_0=' CGAL.sh

2. makeCGAL

[1] cd /work/$USER/OpenFOAM/ThirdParty-3.0.1
[2] sed -i -e 's=boost-system=boost_1_55_0=' makeCGAL
```

Job running environment

1. modules (OpenFOAM is optimized with Gcc)

```
module load gcc/4.9.2
module load openmpi/gcc-4.9.2/std/1.8.7
```

2. of_301 script (setting up the right environment for OpenFOAM 3.0.1)

It is better to make an alias to execute the OpenFOAM environment

```
[1] vi ~/.bashrc
[2] " alias of301='source /work/$USER/OpenFOAM/OpenFOAM-3.0.1/etc/bashrc' "
```

Setup the environment

```
[isaac@orc-login2:~] module list
Currently Loaded Modulefiles:
  1) torque/2.5.13          3) intel/12.1.3           5) mkl/10.3.9
7) ldwrapper/1.1
  2) moab/7.1.1             4) openmpi/intel/1.6.2    6) sq-tm/2.5
8) user-environment/2.0.1

[isaac@orc-login2:~] of301

[isaac@orc-login2:~] module list
Currently Loaded Modulefiles:
  1) torque/2.5.13               3) mkl/10.3.9                        5) ldwrapper/
1.1                  7) gcc/4.9.2
  2) moab/7.1.1                  4) sq-tm/2.5                         6) user-
environment/2.0.1        8) openmpi/gcc-4.9.2/std/1.8.7
```

Setup the environment (Checking!)

Check if right modules and environment are set **whenever you log in.**

```
[isaac@orc-login1:..] module list
Currently Loaded Modulefiles:
  1) torque/2.5.13              4) sq-tm/2.5                    7) gcc/4.9.2
  2) moab/7.1.1                 5) ldwrapper/1.1                8) openmpi/gcc-4.9.2/std/1.8.7
  3) mkl/10.3.9                 6) user-environment/2.0.1


[isaac@orc-login1:..] echo $WM_PROJECT_DIR
/work/isaac/OpenFOAM/OpenFOAM-3.0.1

[isaac@orc-login1:..] echo $FOAM_INST_DIR
/work/isaac/OpenFOAM
```

Submitting a compilation job

```
[1] cd /work/$USER/OpenFOAM/OpenFOAM-3.0.1
[2] sqsub -r 2d -o log -e error /work/$USER/OpenFOAM/OpenFOAM-3.0.1/Allwmake
```

Check 'log' and 'error' to check the status and errors

Pre/Post Processing

- MESH GENERATION
- PARAVIEW

Tutorial test

1. Lid-driven cavity case
(http://cfd.direct/openfoam/user-guide/cavity/#x5-40002.1)

```
[1] mkdir -p $FOAM_RUN
[2] cp -r $FOAM_TUTORIALS $FOAM_RUN
[3] cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity
```

```
[isaac@orc-login1:..] ls -lrt
total 12
drwxrwxr-x 2 isaac isaac 4096 Mar 29 00:31 0
drwxrwxr-x 2 isaac isaac 4096 Mar 29 00:31 system
drwxrwxr-x 2 isaac isaac 4096 Mar 29 00:31 constant
```

[4] **blockMesh**
- Generating mesh
- Needs to run in the case directory

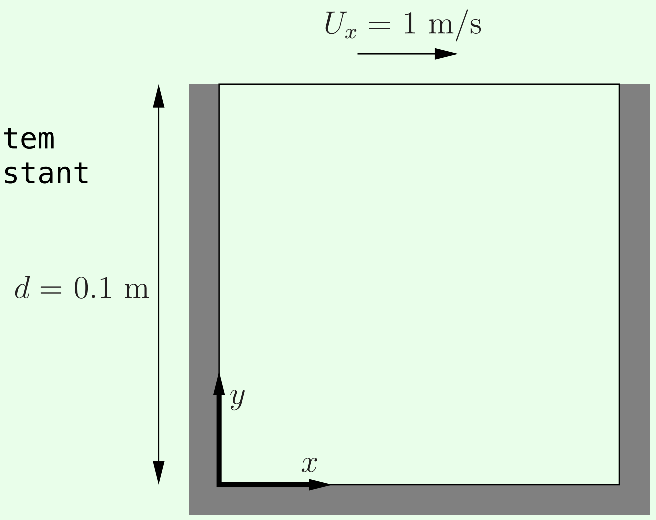


Figure 2.1: Geometry of the lid driven cavity.

Basic case structure

0

constant

system

Contains initial values and boundary conditions for all variables

Specifies what methods and schemes to use and controls for the simulation

- Timestep
- Numerical schemes
- SIMPLE/PISO controls
- Simulation start/end time
- Additional runtime calculations

Contains things that are constant

- Mesh
- Material properties
- Environmental constants

‘blockMesh’ result

```
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  3.0.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : 3.0.1-6b88e07ba67e
Exec   : blockMesh

     Basic statistics
          Number of internal faces : 0
          Number of boundary faces : 6
          Number of defined boundary faces : 6
          Number of undefined boundary faces : 0
     Checking patch -> block consistency

Creating points with scale 0.1
    Block 0 cell size :
        i : 0.005 .. 0.005
        j : 0.005 .. 0.005
        k : 0.01 .. 0.01
----------------
Mesh Information
----------------
  boundingBox: (0 0 0) (0.1 0.1 0.01)
  nPoints: 882
  nCells: 400
  nFaces: 1640
  nInternalFaces: 760
----------------
Patches
----------------
  patch 0 (start: 760 size: 20) name: movingWall
  patch 1 (start: 780 size: 60) name: fixedWalls
  patch 2 (start: 840 size: 800) name: frontAndBack

End
```

Mesh generation

BlockMesh
Generate simple structured meshes based on blocks http://www.openfoam.org/docs/user/blockMesh.php

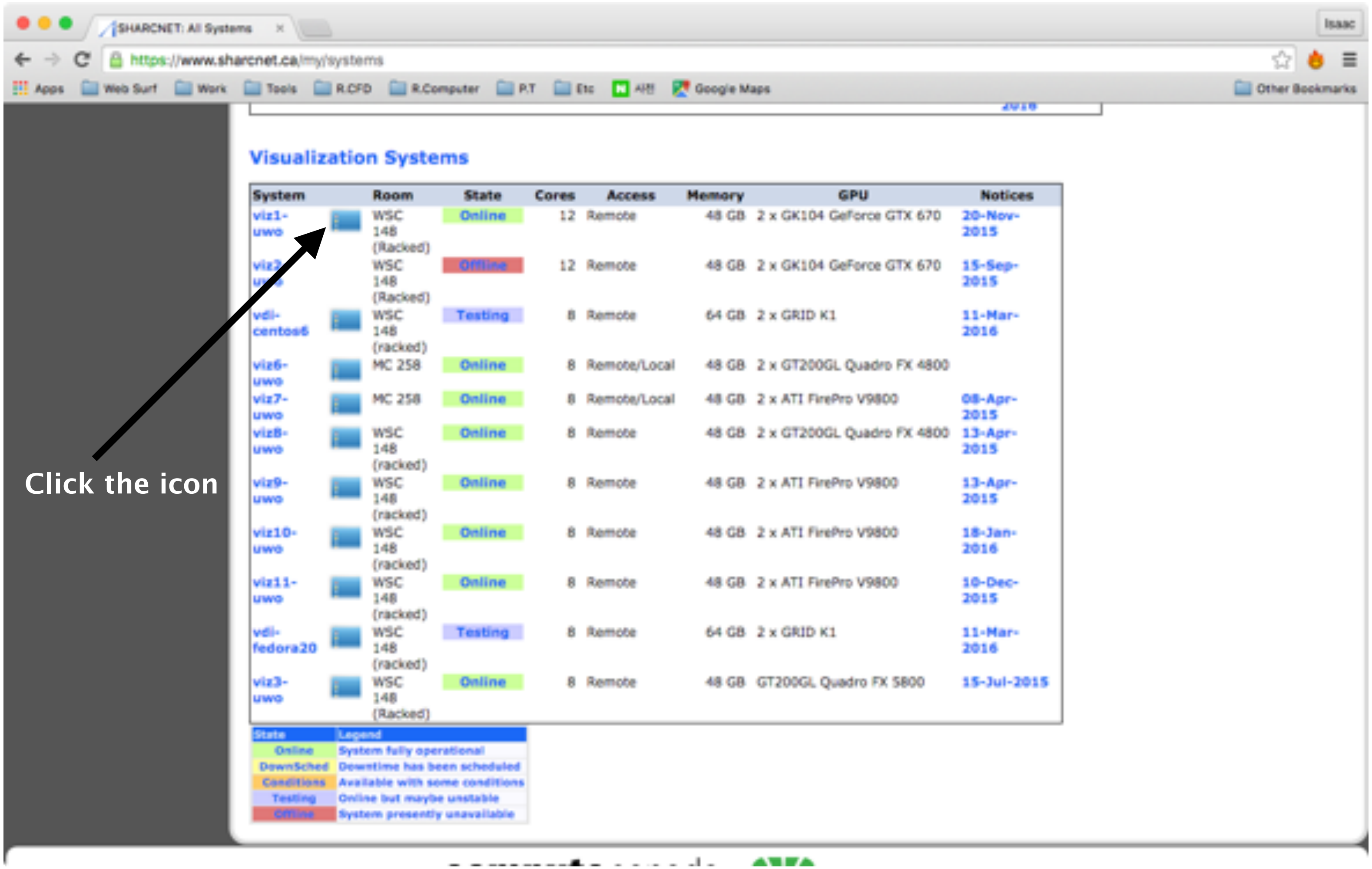
➔ SnappyHexMesh
Create unstructured meshes based on
complex surface geometries (e.g stl) http://www.openfoam.org/docs/user/snappyHexMesh.php

➔ Other meshers
conversion to OpenFOAM format from most popular formats possible

Prepare a ‘case’ for Paraview

```
[1] cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity

[2] touch cavity.openFOAM

[3] Open the file cavity.openFOAM in Paraview

 (select file type "all files" and them type “openFOAM”)

[4] Choose ‘OpenFOAM’ for the reader choice


Note: ParaFoam is not supported on SHARCNET login node.
```

Connecting to Visualization machine



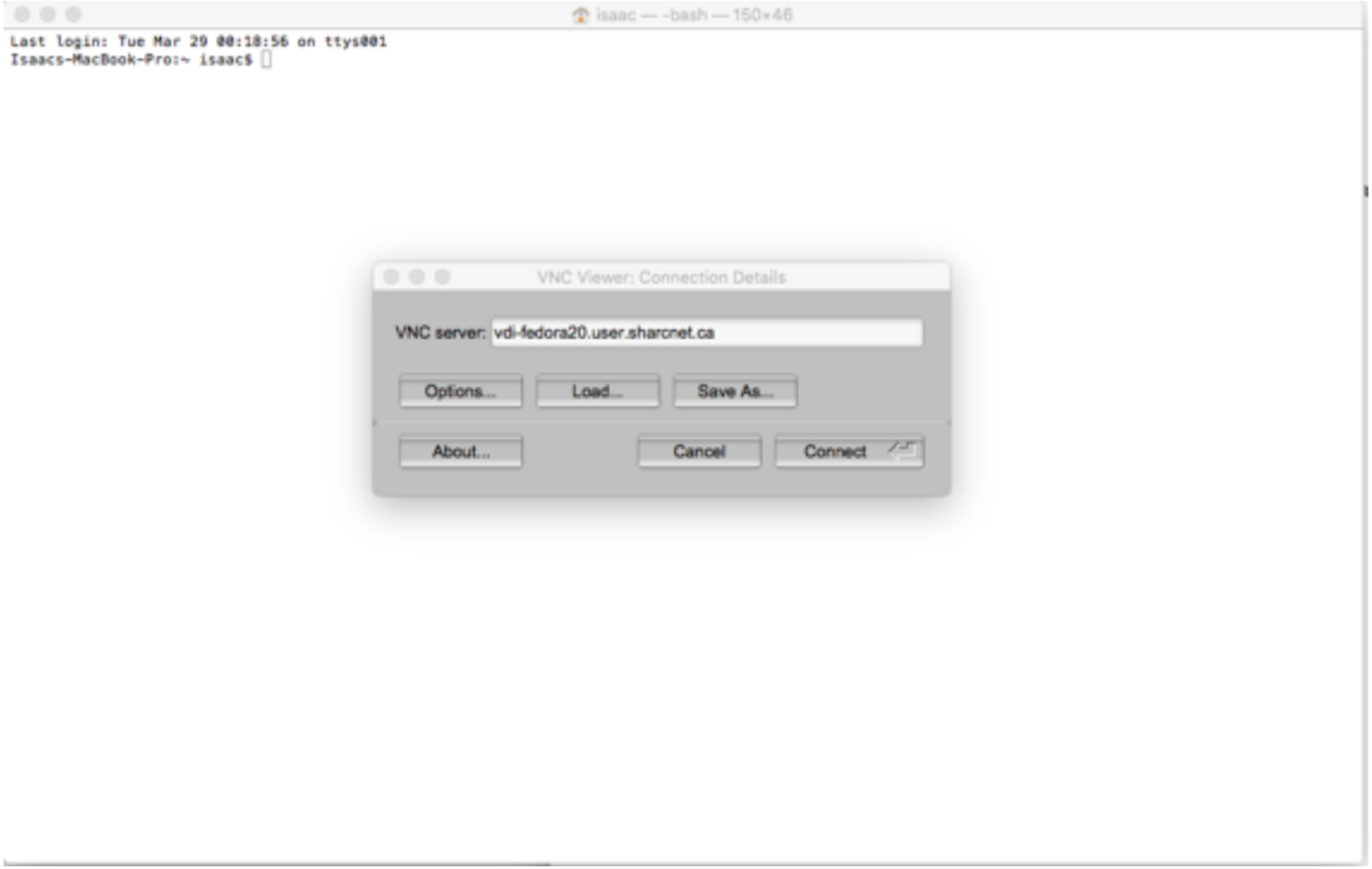
Visualization Systems

System		Room	State	Cores	Access	Memory	GPU	Notices
viz1-uwo	[icon]	WSC 148 (Racked)	Online	12	Remote	48 GB	2 x GK104 GeForce GTX 670	20-Nov-2015
viz2-uwo		WSC 148 (Racked)	Offline	12	Remote	48 GB	2 x GK104 GeForce GTX 670	15-Sep-2015
vdi-centos6	[icon]	WSC 148 (racked)	Testing	8	Remote	64 GB	2 x GRID K1	11-Mar-2016
viz6-uwo	[icon]	MC 258	Online	8	Remote/Local	48 GB	2 x GT200GL Quadro FX 4800	
viz7-uwo	[icon]	MC 258	Online	8	Remote/Local	48 GB	2 x ATI FirePro V9800	08-Apr-2015
viz8-uwo	[icon]	WSC 148 (racked)	Online	8	Remote	48 GB	2 x GT200GL Quadro FX 4800	13-Apr-2015
viz9-uwo	[icon]	WSC 148 (racked)	Online	8	Remote	48 GB	2 x ATI FirePro V9800	13-Apr-2015
viz10-uwo	[icon]	WSC 148 (racked)	Online	8	Remote	48 GB	2 x ATI FirePro V9800	18-Jan-2016
viz11-uwo	[icon]	WSC 148 (racked)	Online	8	Remote	48 GB	2 x ATI FirePro V9800	10-Dec-2015
vdi-fedora20	[icon]	WSC 148 (racked)	Testing	8	Remote	64 GB	2 x GRID K1	11-Mar-2016
viz3-uwo	[icon]	WSC 148 (Racked)	Online	8	Remote	48 GB	GT200GL Quadro FX 5800	15-Jul-2015

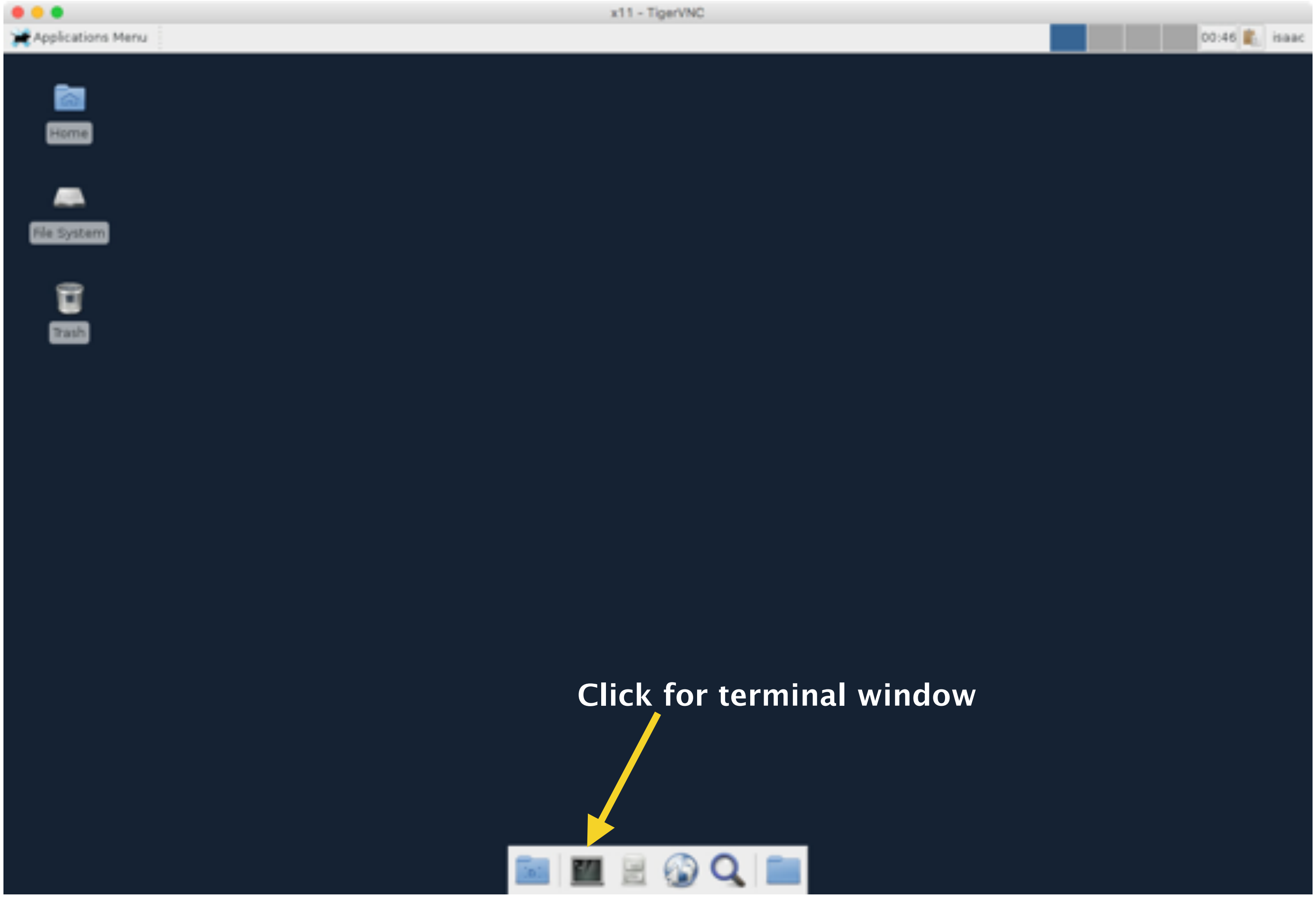
State	Legend
Online	System fully operational
DownSched	Downtime has been scheduled
Conditions	Available with some conditions
Testing	Online but maybe unstable
Offline	System presently unavailable

Connecting to the Visualization machine

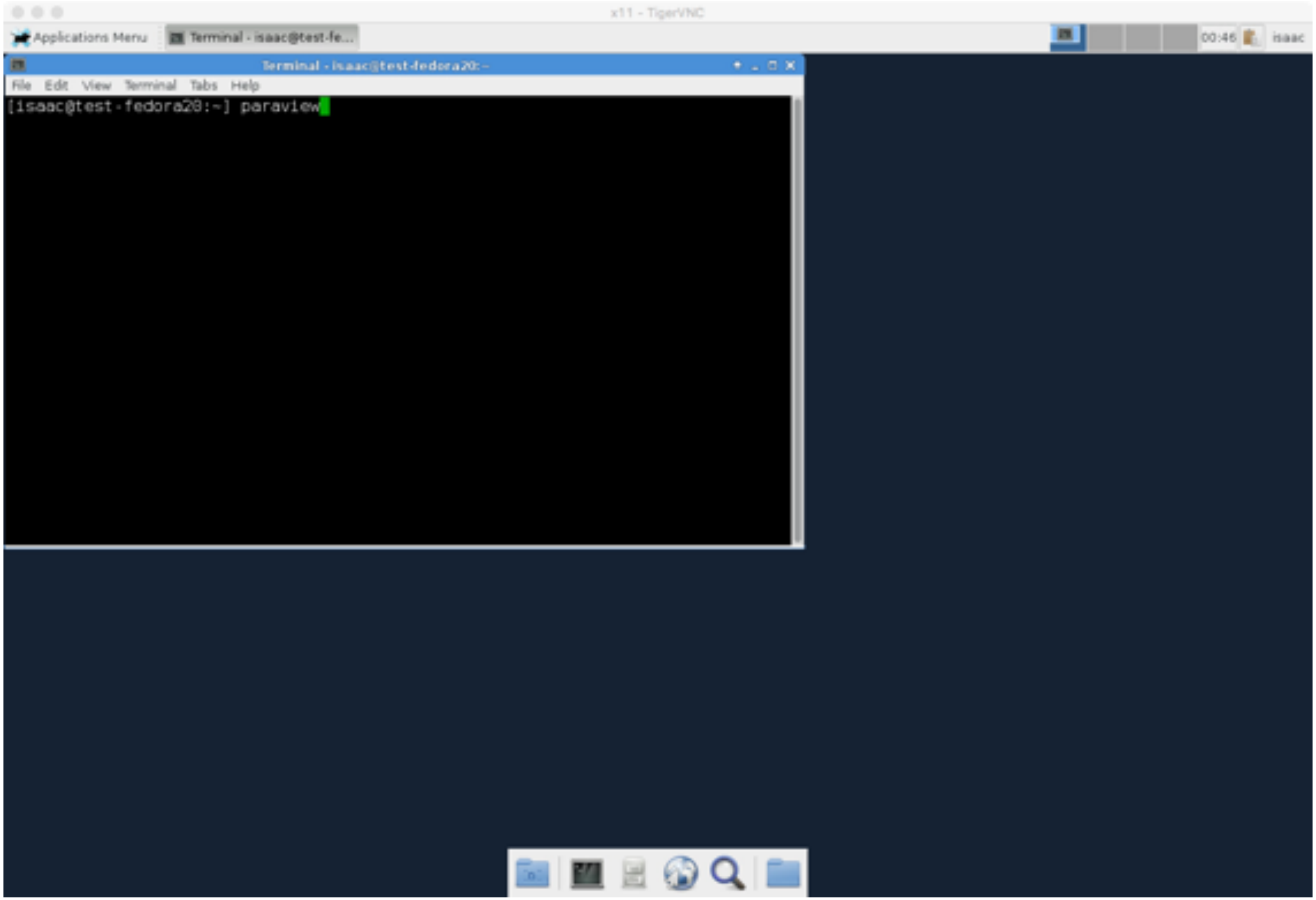
https://www.sharcnet.ca/help/index.php/Remote_Graphical_Connections



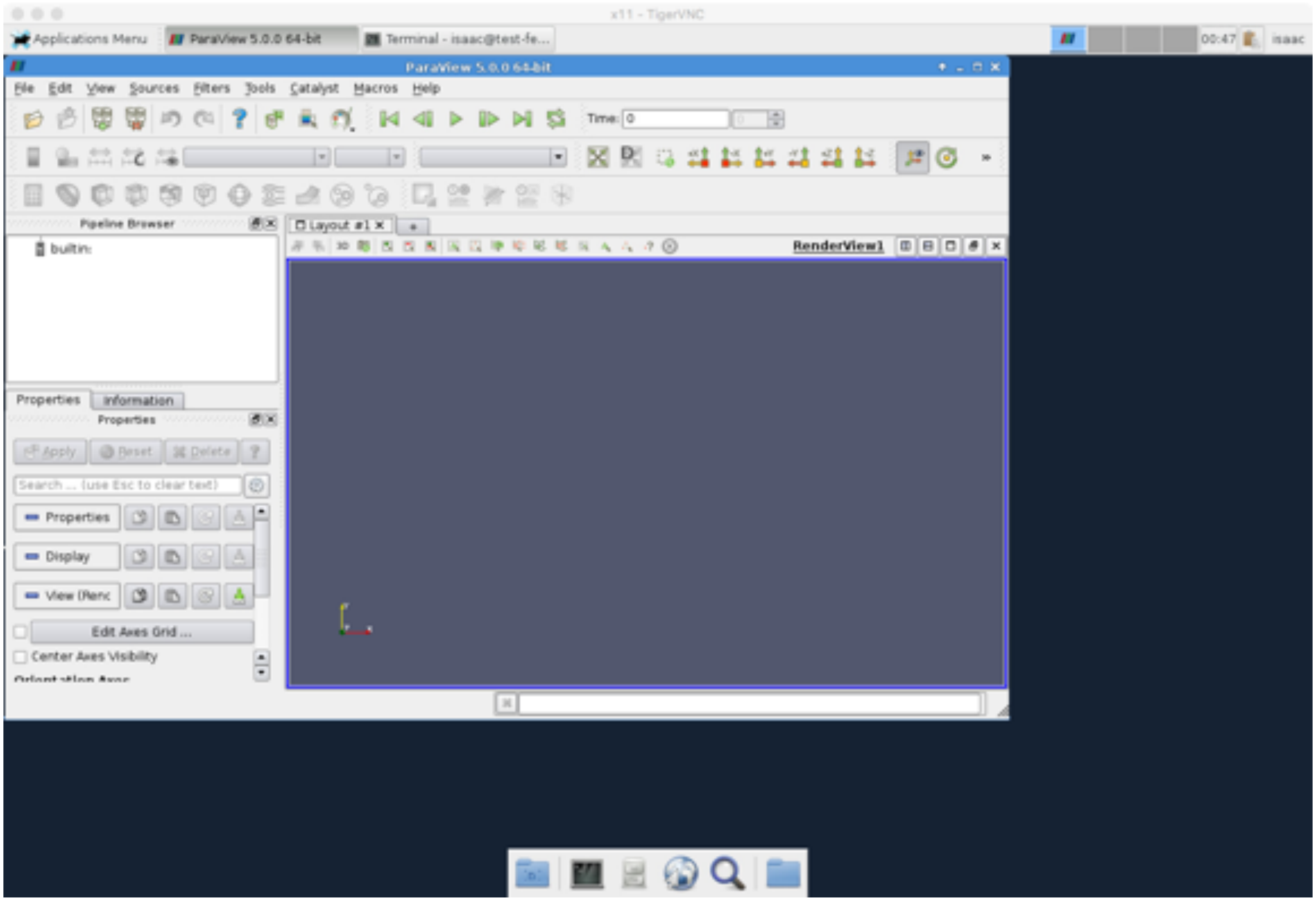
Connecting to the Visualization machine



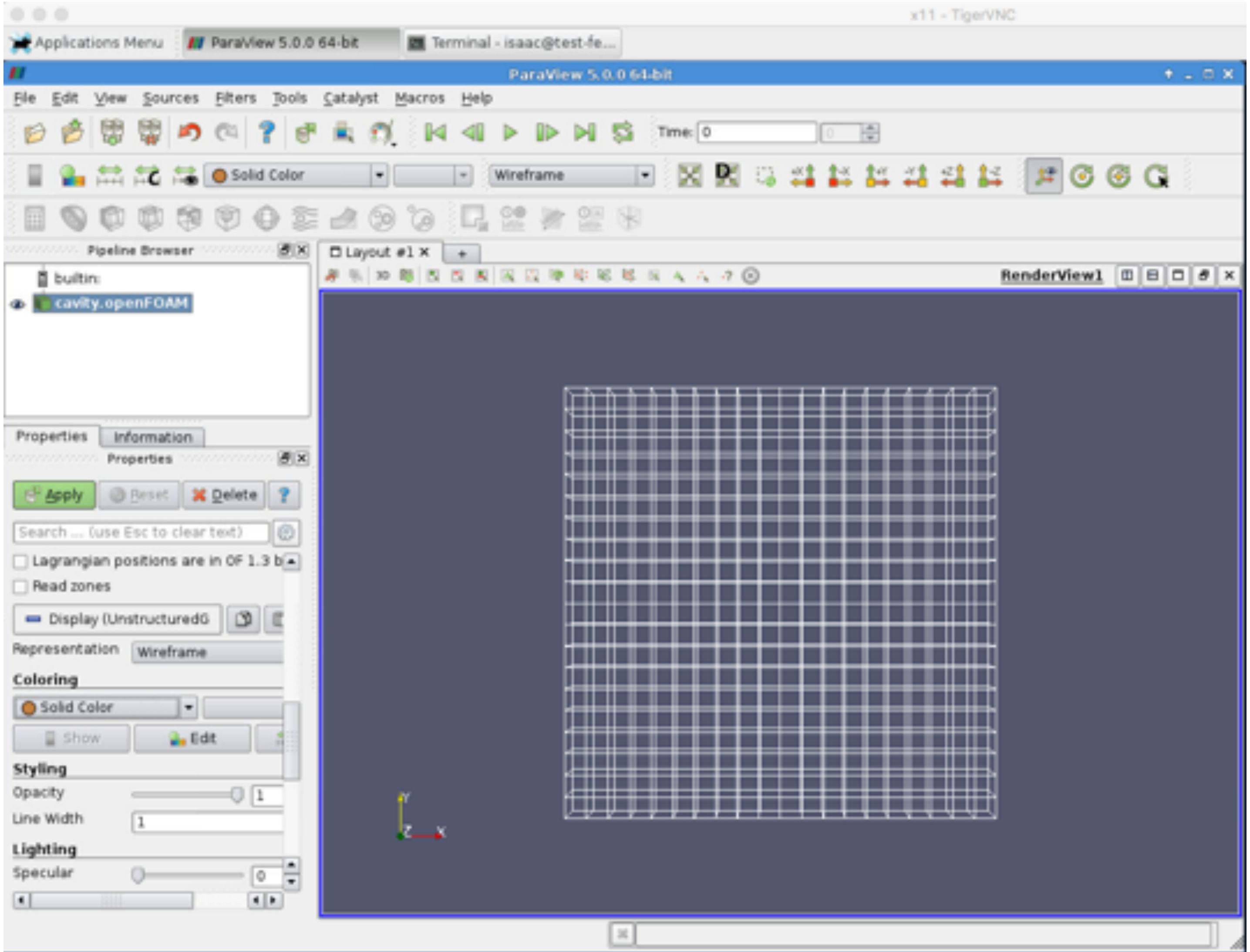
Starting Paraview



Starting ParaView



Mesh in Paraview



Running OpenFOAM

– DAMBREAKFINE CASE

– SUBMITTING A JOB IN SERIAL AND PARALLEL

Running a serial job

[1] of301

(call the right environment using the alias or command line directly)

[2] cd $FOAM_RUN/tutorials/incompressible/icoFoam/cavity

Lid-driven cavity case,

http://cfd.direct/openfoam/user-guide/cavity/#x5-40002.1)

[3] **sqsub -r 10m -o log -e error icoFoam**

```
[isaac@orc-login1:..cavity] ls -lrt
total 162
drwxrwxr-x 2 isaac isaac  4096 Mar 29 00:31 0
drwxrwxr-x 2 isaac isaac  4096 Mar 29 00:31 system
drwxrwxr-x 3 isaac isaac  4096 Mar 29 00:35 constant
-rw-rw-r-- 1 isaac isaac     0 Mar 29 01:06 cavity.openFOAM
-rw-rw-r-- 1 isaac isaac     0 Mar 29 01:34 error
drwxrwxr-x 3 isaac isaac  4096 Mar 29 01:34 0.3
drwxrwxr-x 3 isaac isaac  4096 Mar 29 01:34 0.2
drwxrwxr-x 3 isaac isaac  4096 Mar 29 01:34 0.1
drwxrwxr-x 3 isaac isaac  4096 Mar 29 01:34 0.5
drwxrwxr-x 3 isaac isaac  4096 Mar 29 01:34 0.4
-rw-rw-r-- 1 isaac isaac 74232 Mar 29 01:35 log
```

Running a parallel job

1. Dambreak case (damBreakFine)

(http://cfd.direct/openfoam/user-guide/damBreak/#x7-610002.3.11)

2. Input conditions and boundaries must be set properly

(see the OpenFoam user guide: blockMeshDist, setFields, decomposePar)

[1] of301

[2] cd $FOAM_RUN/tutorials/multiphase/interFoam/laminar/damBreakFine

[3] **sqsub -r 1h -q mpi -n 4 -o log -e error interFoam -parallel**

```
[isaac@orc-login1:..damBreakFine] sqjobs
  jobid  queue state ncpus                nodes  time command
------- ------ ----- ----- ------------------- ----- -------
..
7785272    mpi     R     4 orc[307-308,310-311]  350s interFoam -parallel
```

```
[isaac@orc-login2:..damBreakFine] ls -lrt
total 674
drwxrwxr-x  2 isaac isaac   4096 Mar 29 10:36 0
drwxrwxr-x  3 isaac isaac   4096 Mar 29 10:36 constant
drwxrwxr-x  2 isaac isaac   4096 Mar 29 10:41 system
-rw-rw-r--  1 isaac isaac      0 Mar 29 11:00 error
drwxrwxr-x 24 isaac isaac   4096 Mar 29 11:02 processor3
drwxrwxr-x 24 isaac isaac   4096 Mar 29 11:02 processor2
drwxrwxr-x 24 isaac isaac   4096 Mar 29 11:02 processor1
drwxrwxr-x 24 isaac isaac   4096 Mar 29 11:02 processor0
-rw-rw-r--  1 isaac isaac 629325 Mar 29 11:03 log

[isaac@orc-login2:..damBreakFine] more log
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  3.0.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : 3.0.1-6b88e07ba67e
Exec   : interFoam -parallel
Date   : Mar 29 2016
Time   : 11:02:34
Host   : "orc307"
PID    : 30332
Case   : /gwork/isaac/OpenFOAM/isaac-3.0.1/run/tutorials/multiphase/interFoam/laminar/
damBreakFine
nProcs : 4
```

Compiling local solver

– USER-DEFINED SOLVER

Example: myFoam

There is a need to compile your own version of solver attaching to the OpenFOAM. Here is an example study about how to.
(Note: the OpenFOAM environment must be loaded before this procedure.)
1] Make a local solver folder

```
mkdir $WM_PROJECT_DIR/solvers
```

2] Copy the existing solver

```
cd $WM_PROJECT_DIR/solvers
cp -r $FOAM_SOLVERS/incompressible/pisoFoam myFoam
```

It is a better practice to copy the existing solver structure and modify it.

Example: myFoam

3] Modify the name and etc accordingly.

```
cd $WM_PROJECT_DIR/solvers/myFoam
mv pisoFoam.C myFoam.C
```

4] Edit some description and application name in myFoam.C

I

```
vi myFoam.C
                    :
                    Application
                      myFoam
                    :
                    Description
                      my solver
                    :
```

Example: myFoam

5] Modify the makefile list of files

```
vi Make/files

myFoam.C

EXE=$(FOAM_USER_APPBIN)/myFoam
```

6] Compilation

```
wclean
wmake

[isaac@orc-login1:..myFoam] which myFoam
/scratch/isaac/OpenFOAM/OpenFOAM-3.0.1/platforms/linux64GccDPInt32Opt/bin/myFoam
```

compute canada | calcul canada

Thank you!