Imperial College London

IC-FERST Training Course

Novel Reservoir Modelling & Simulation

P. Salinas, C. Jacquemyn, A. Kampitsis, C.C. Pain, M.D. Jackson

Imperial College London

IC-FERST

The code is a “Scientific Development” and not a “Commercial Package”

IC-FERST’s cons:

1) The GUI is not that “User Friendly”
2) Code might stop working if the problem is not set correctly
3) The code is not fully optimised
4) We need a post- process program (Paraview) to visualise results

IC-FERST’s pros:

I. Increased accuracy and physics representation
II. Multiple physical phenomena
III. Advanced computational methods

In this training we keep things simple, however we will discuss some code!

Imperial College London

Setting up IC-FERST

IC-FERST runs with Linux/Ubuntu (16.04 / 18.04)

I. IC-FERST is installed to Department's computers at the computer room X.X

II. Install IC-FERST on your personal computer. General information:

1. IC-FERST lives in github, a Code Repository. You will have to make a github account, to access the IC repositories
2. Download the code, configure code, create executable
3. Use the user interface called Diamond
4. Linux terminal commands to run the code
5. Install post-process visualization program (Paraview)

Imperial College London

Setting up IC-FERST

1) Make the code's executable

Go inside file (MultiFluids_Dev) the code directory, open a new terminal and run the command (that might a couple of minutes):

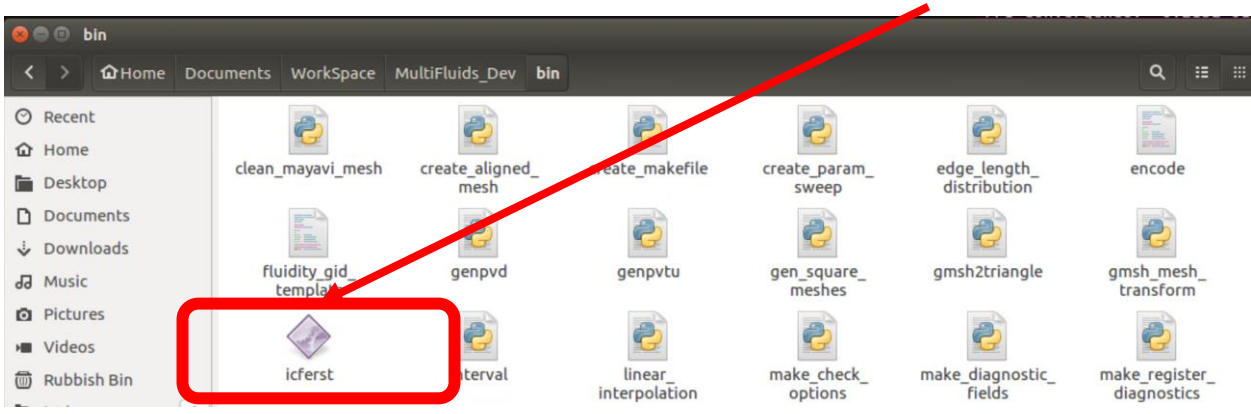
```
./configure --enable-2d-adaptivity && make clean && make mp
```

Create make files

Removes Previous files

Compile the program

Go to folder MultiFluids_Dev/bin and you should see the executable:

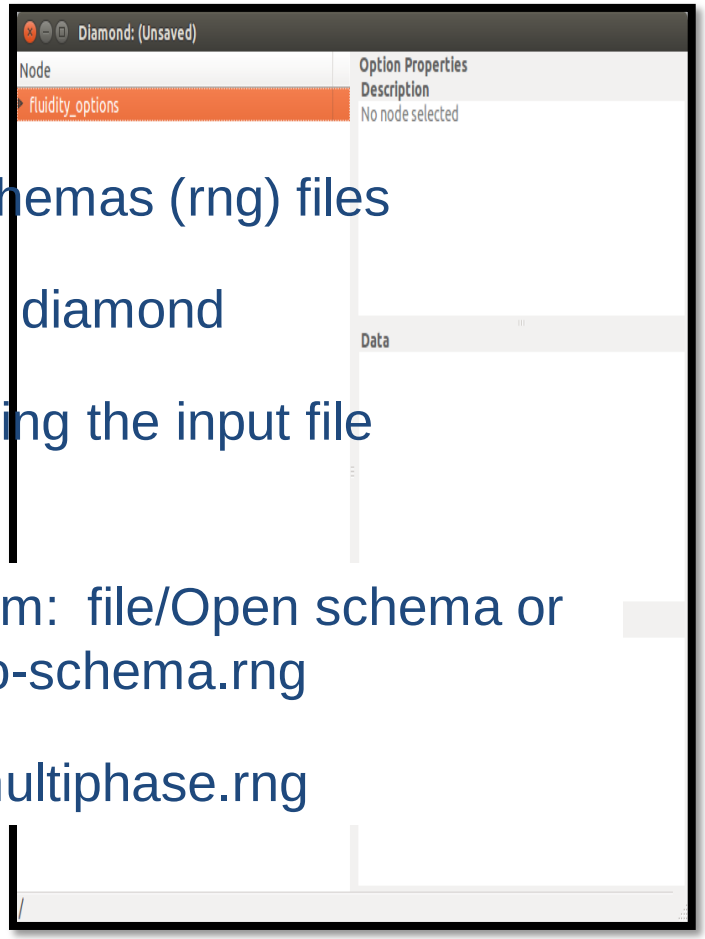


Imperial College London

Setting up IC-FERST

2) User Interface: Diamond

- Diamond is the GUI used to create the inputs for IC-FERST
- It is different for different applications
- The interface is modified by using different schemas (rng) files
- To run the program, open a terminal and type: diamond
- The schemas have to be opened before opening the input file



- Schemas can be opened by using the GUI from: file/Open schema or through the command line: diamond -s path-to-schema.rng
- The schema is in MultiFluids_Dev/schemas/multiphase.rng

Imperial College London

Input file format for IC-FERST

3) User Interface: Diamond

- The file format used in IC-FERST is .mpml
- This file format follows an xml format and it can also be edited using a text editor like gedit, notepad, vim, etc
- Through diamond: file/open/ and set option all files
- Through the terminal: diamond -s path-to-schema.rng path-to-input.mpml

Imperial College London

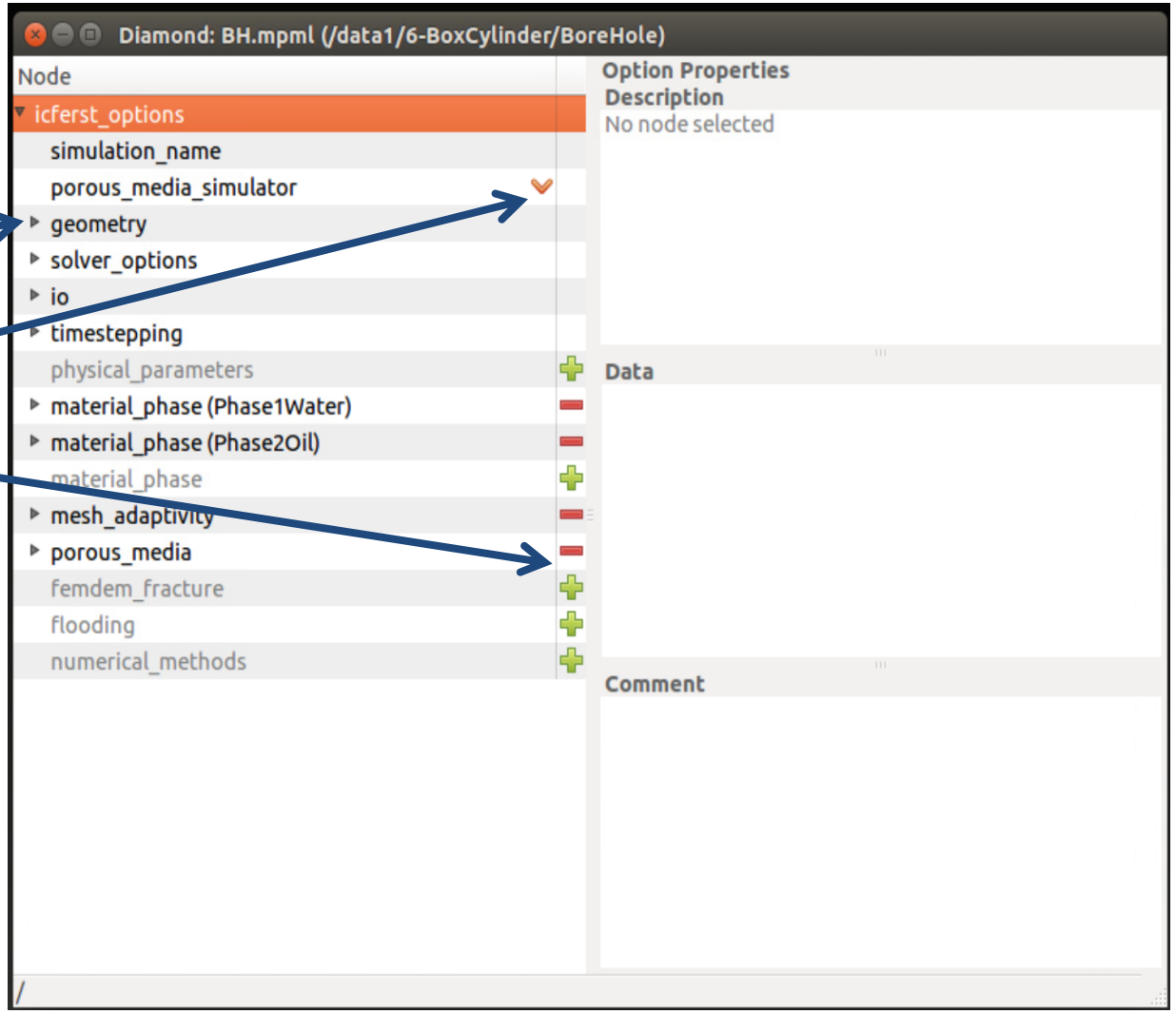
Input file format for IC-FERST

3) Diamond

General operations:

- Open's drop down menu
- Open's drop down options
- Add / Remove

Store Data and Save frequently



Imperial College London

Input file format for IC-FERST

3) Diamond Overview

Name of the output files

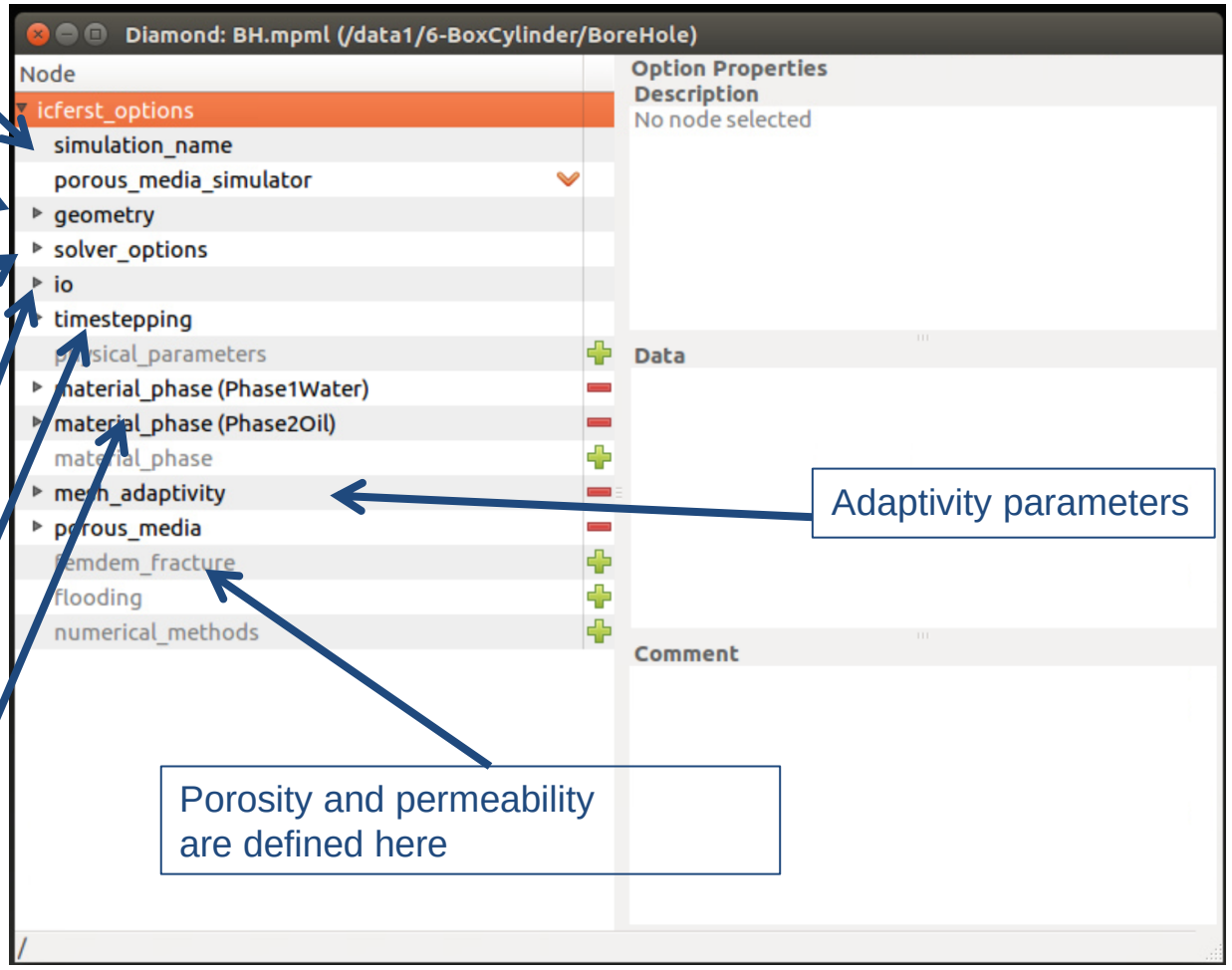
Define the input mesh and computational meshes

Solver parameters
Use default

Define when to create output files and how

Input associated to time

One per phase defines all the physical parameters of the simulation



Adaptivity parameters

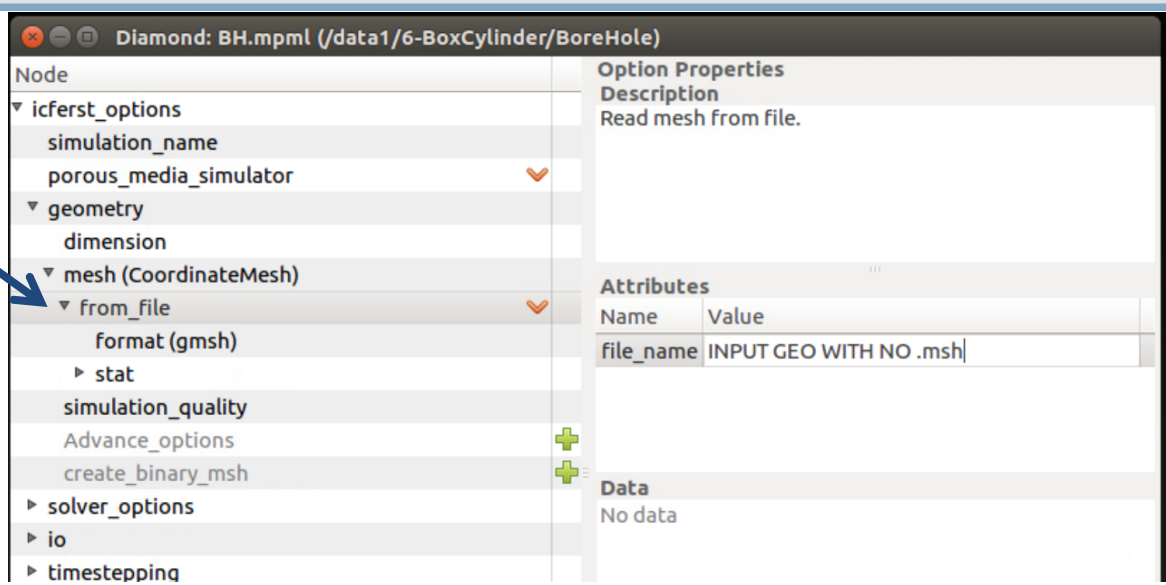
Porosity and permeability are defined here

Imperial College London

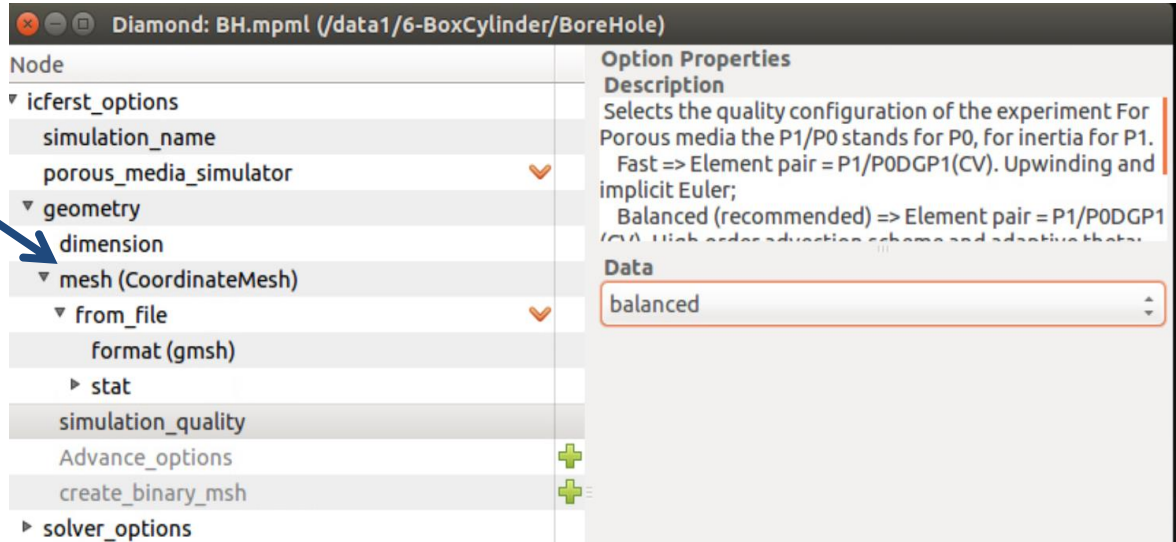
Input file format for IC-FERST

3) Geometry

Name of the input mesh file
with NO .msh
The file should be in the same folder
with the exe



Element pair.
Most of the times use balanced



Imperial College London

Input file format for IC-FERST

3) IO and Time

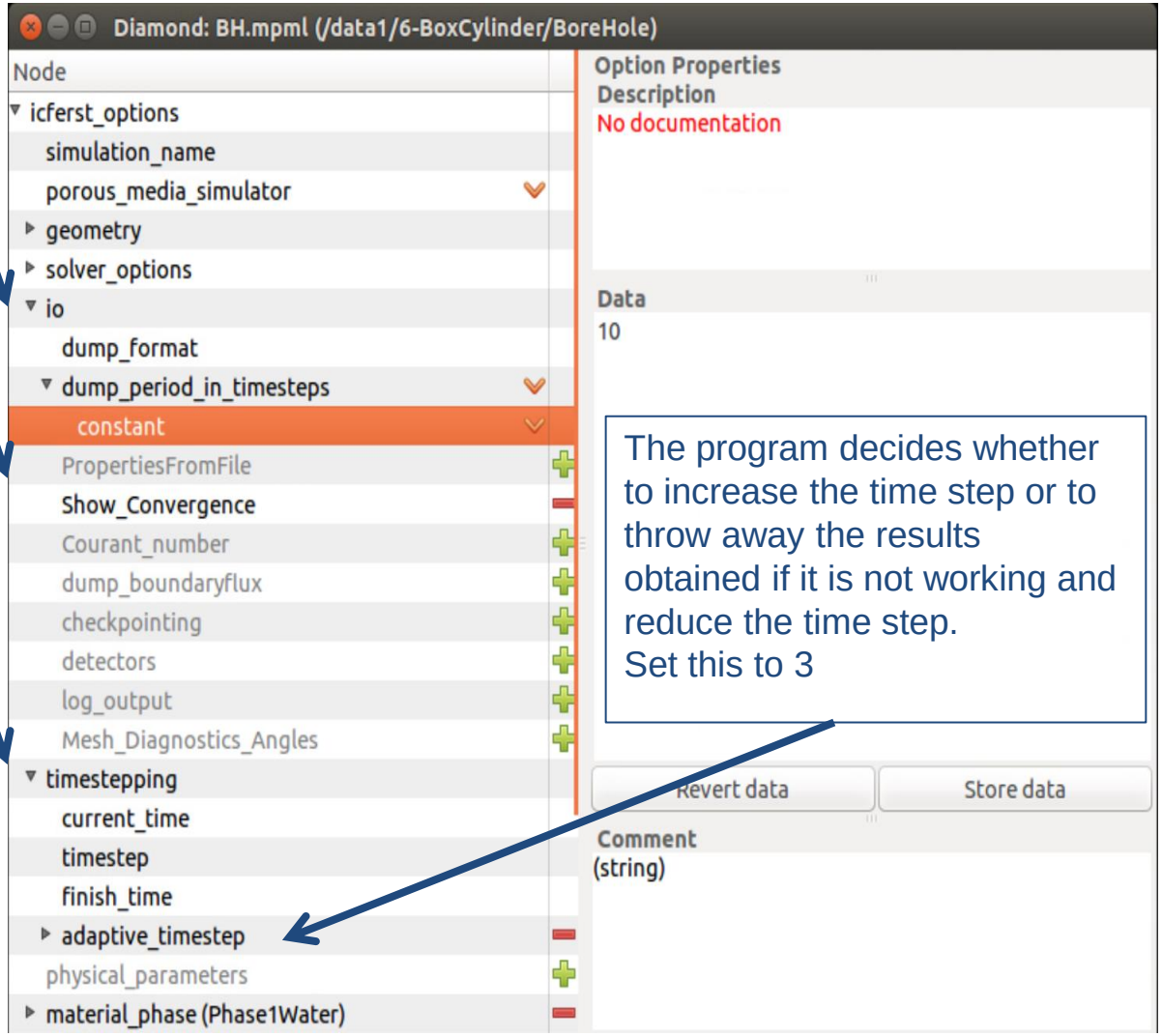
Output frequency
1) On time-steps
2) On actual time

Show simulation data on terminal

T_0 : Simulation start time, 0 is fine

dt: Temporal discretization jump size
Simulations are very sensitive
to this parameter it depends on
many aspects of your simulation.
If your simulation fails decrease this

T_{end} : Simulation finish time.
Note that you get results without
reaching this value



The program decides whether to increase the time step or to throw away the results obtained if it is not working and reduce the time step.
Set this to 3

Imperial College London

Input file format for IC-FERST

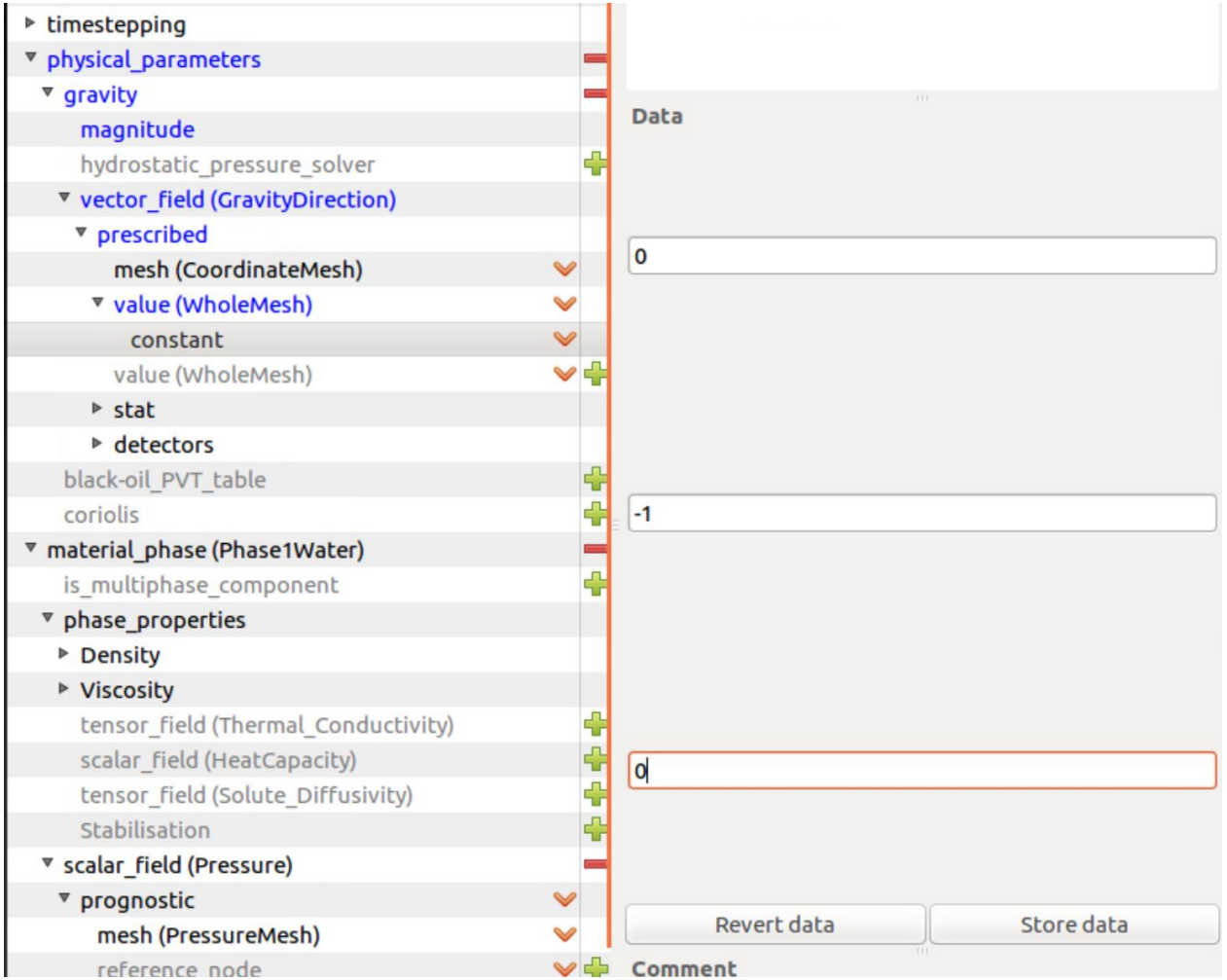
3) Physical parameters

Section to define physical parameters

9.8 in the SI

Direction of the gravity

Direction of the gravity
set to –y (i.e: pointing down)



Imperial College London

Input file format for IC-FERST

3) Phase Properties for each Phase

Define constant density for phase 1

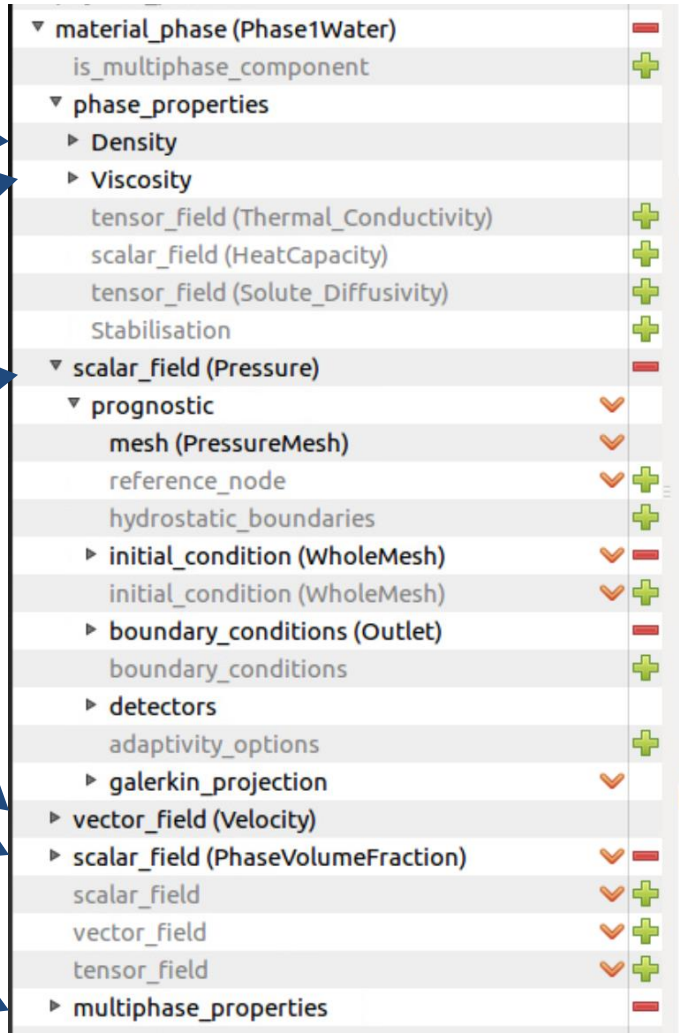
Define Viscosity

Define Pressure field

Define Velocity field

Define Phase Volume Fraction (Saturation) field

Multiphase



Imperial College London

Input file format for IC-FERST

3) Pressure

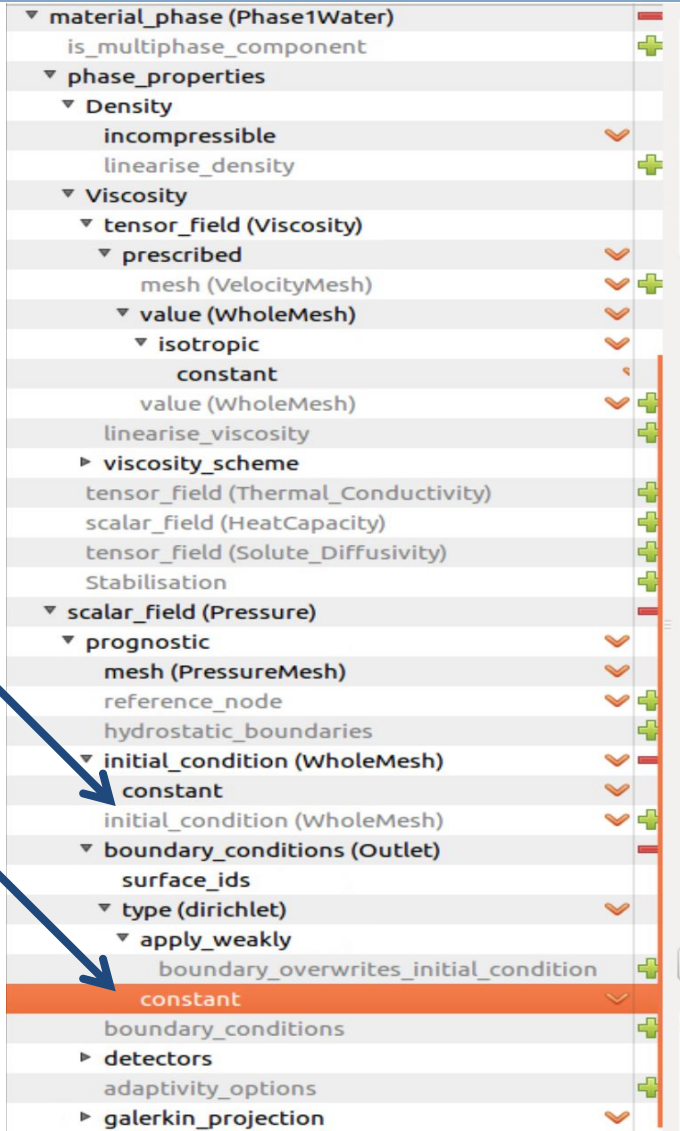
Define pressure BC and IC

Value of this field at the beginning, for pressure 0 is fine.

Surface id is the physical line/surface in which we are going to apply this boundary condition

Define Continuous Galerkin projection for pressure

For other phases the pressure field has to be set to aliased



Imperial College London

Input file format for IC-FERST

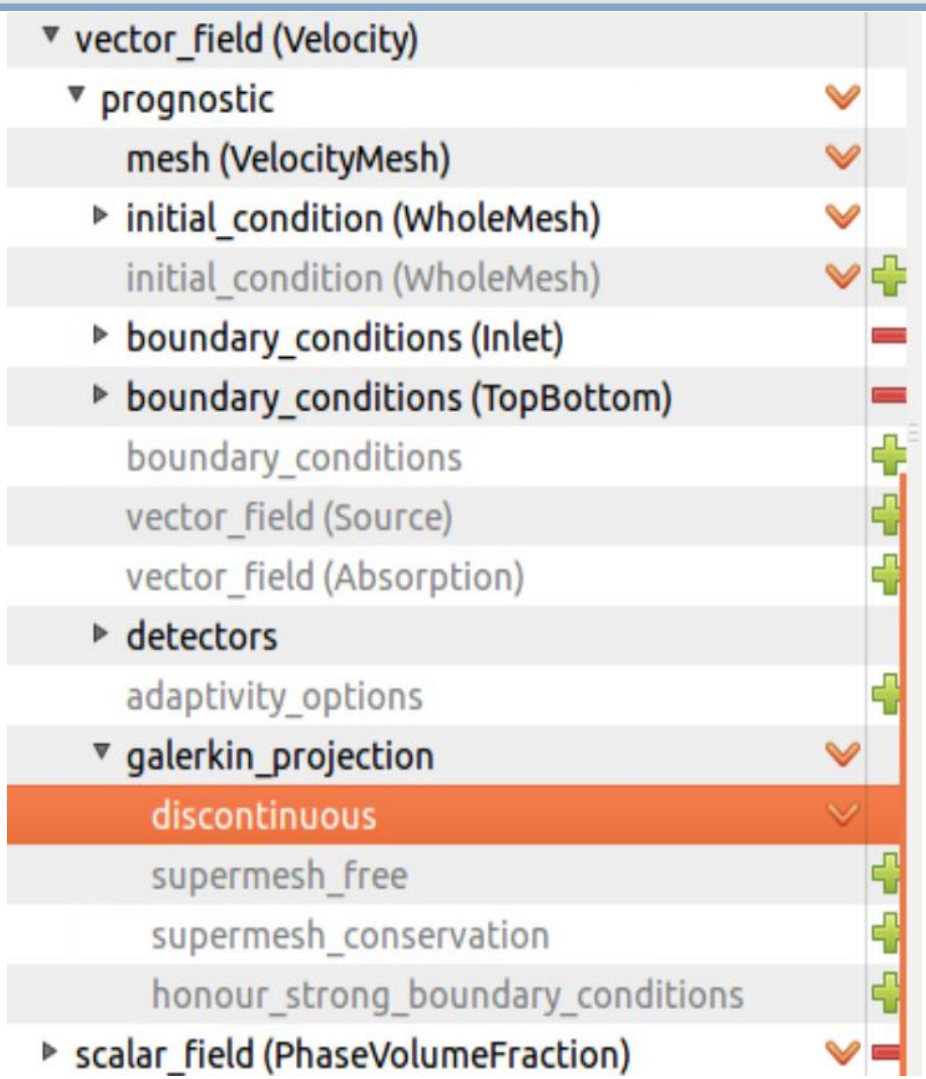
3) Velocity

Define velocity BC and IC

Value of this field at the beginning.

Surface id is the physical line/surface in which we are going to apply this boundary condition

Define Discontinuous Galerkin projection for velocity



Imperial College London

Input file format for IC-FERST

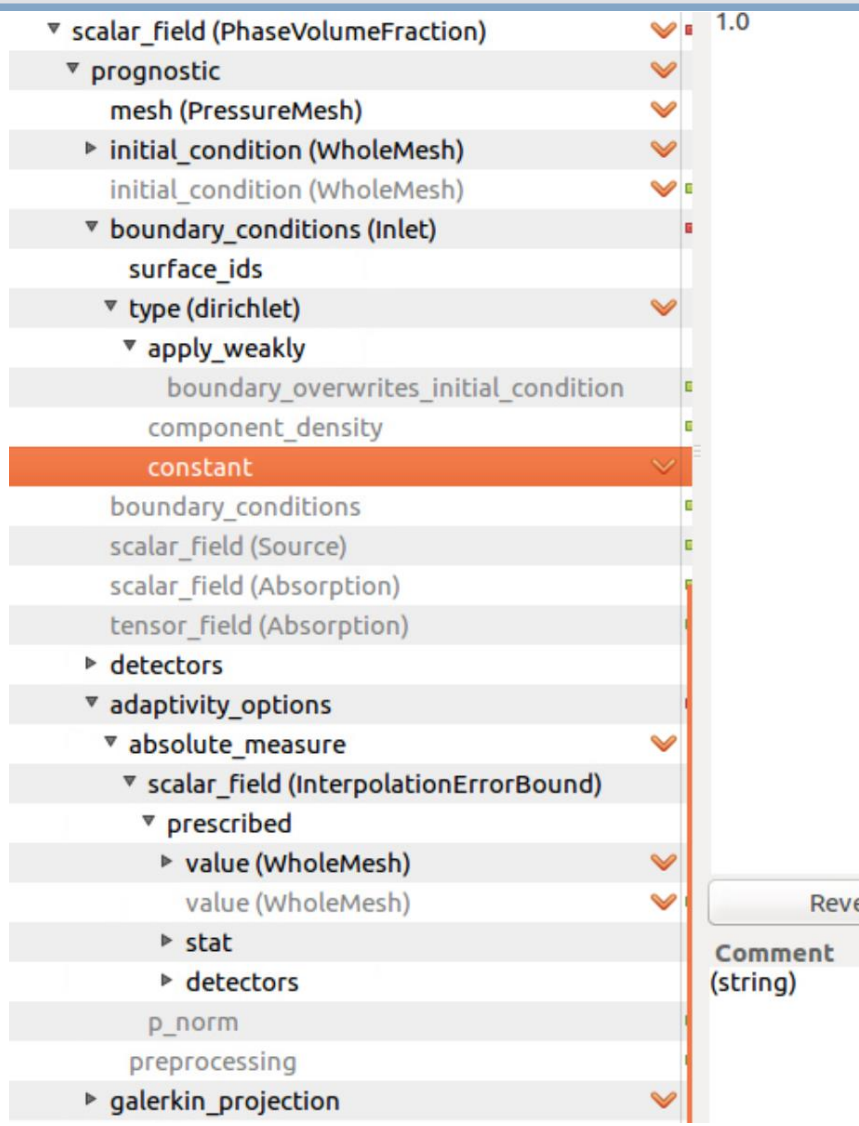
3) Saturation

Define velocity BC and IC

….Similar to the Pressure…

Define in the inlet BC, Phase Volume Fraction equal to 1

Define Continuous Galerkin projection



Imperial College London

Input file format for IC-FERST

3) Multiphase properties

The relative permeability is calculate using the Brooks-Corey model for 2 phases or the Stone model for 3 phases

End-point relative permeability value

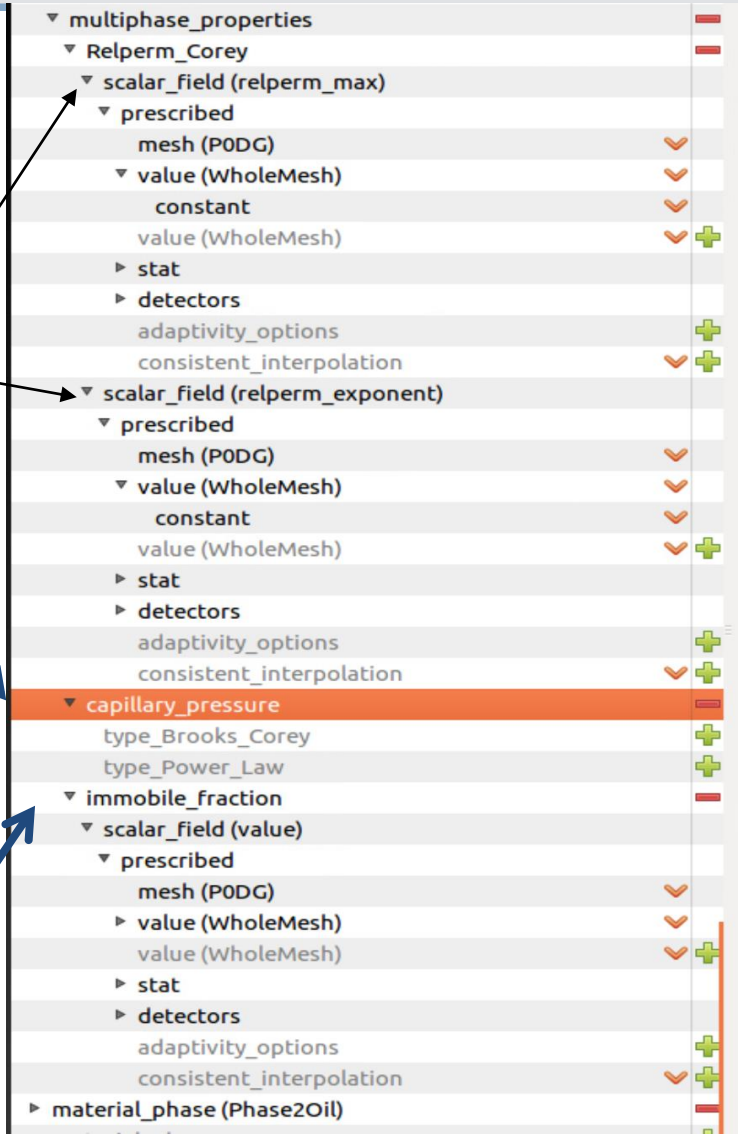
Exponent of the formula

The capillary pressure is calculate using the Brooks-Corey model
It has to be defined ONLY for the wetting phase

The entry pressure can be defined for different regions differently, use the P0DG mesh

The exponent is defined like the entry pressure, use the P0DG mesh

Irreducible phase saturation



Imperial College London

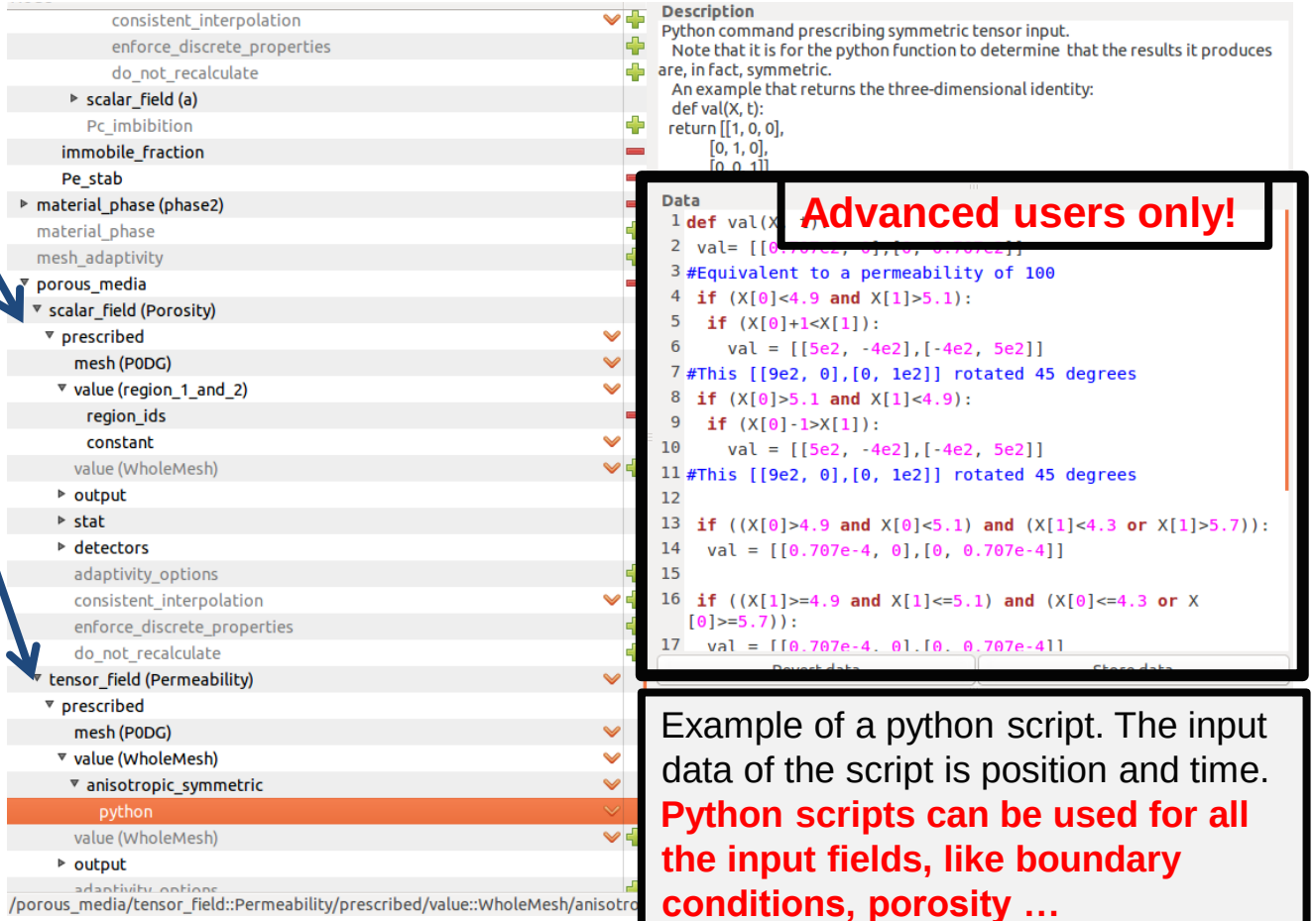
Input file format for IC-FERST

3) Porous Media properties.
Porosity and Permeability have to be Defined using the P0DG mesh

The porosity and permeability can be defined different for different regions using either python functions or surfaces ids (physical surfaces or physical volumes)

The permeability can be defined as a scalar field or as a tensor field

Python scripts can be used to introduced the values.



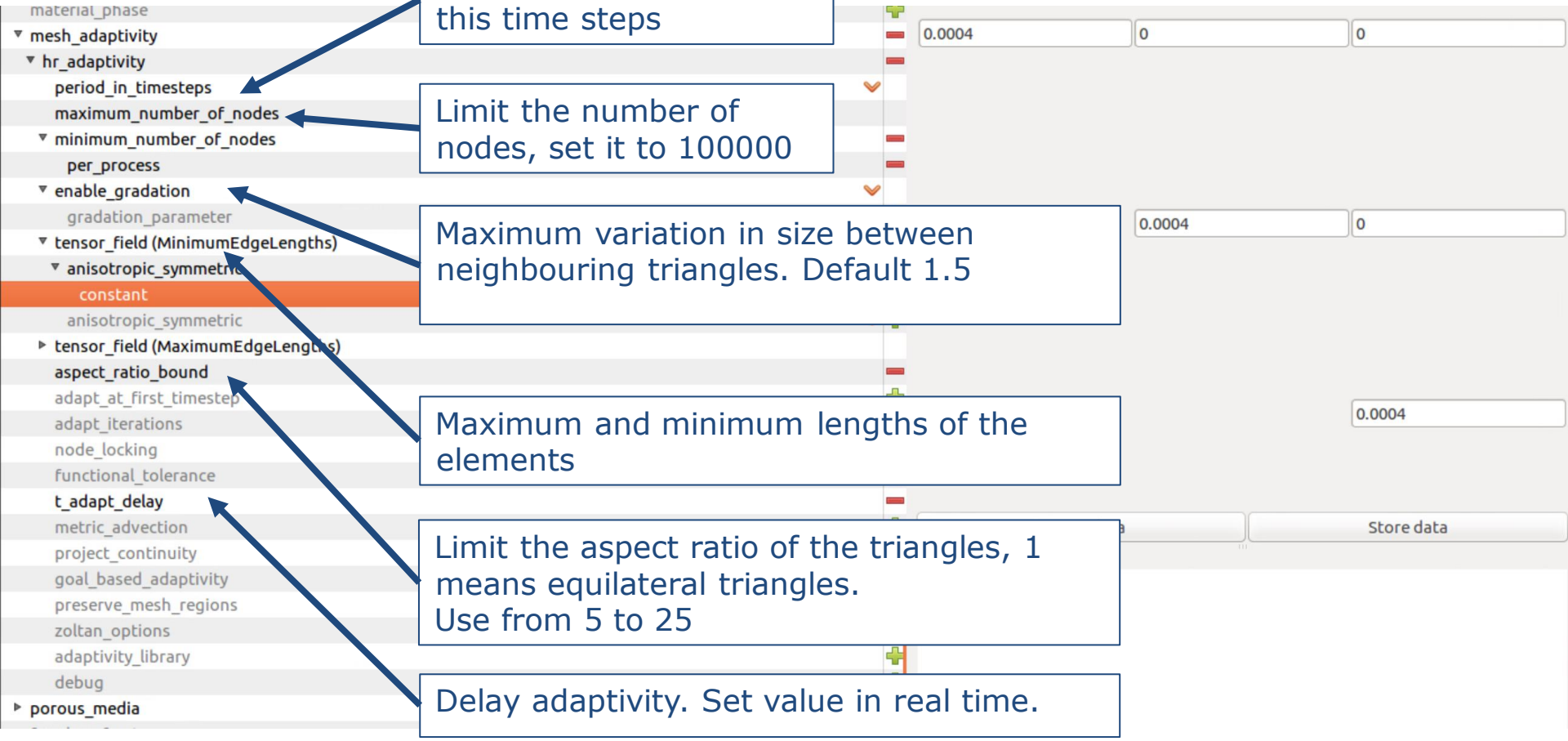
Example of a python script. The input data of the script is position and time.

Python scripts can be used for all the input fields, like boundary conditions, porosity …

Imperial College London

Input file format for IC-FERST

3) Adaptivity



Adapt the mesh every this time steps

Limit the number of nodes, set it to 100000

Maximum variation in size between neighbouring triangles. Default 1.5

Maximum and minimum lengths of the elements

Limit the aspect ratio of the triangles, 1 means equilateral triangles.
Use from 5 to 25

Delay adaptivity. Set value in real time.

Imperial College London

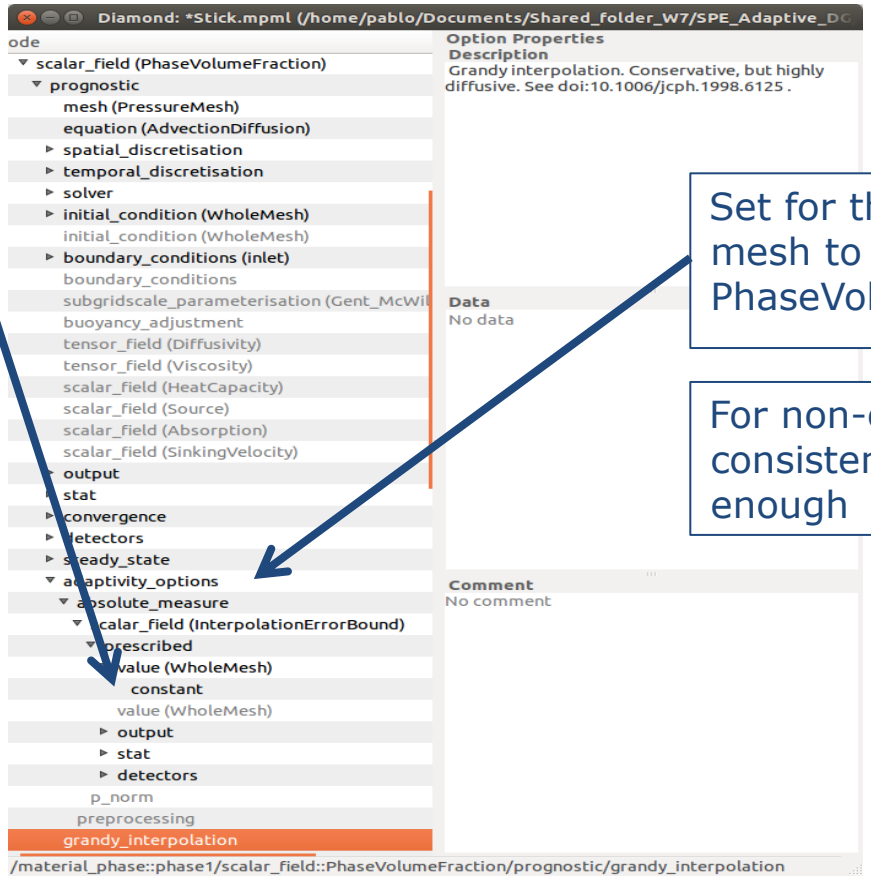
Input file format for IC-FERST

3) Adaptivity

Along with the adaptivity options, you should also specify in which field you want the mesh to adapt to. For example adapt to PVF field:

Error the adaptivity mesh has to try to reach. Smaller values means more elements

This option has to be set for all the fields.
For discontinuous pressure the PhaseVolumeFraction needs Grandy, otherwise use Galerkin Projection



Set for the field you want the mesh to adapt to. Usually PhaseVolumeFraction of phase 1

For non-conservative fields the consistent interpolation is enough

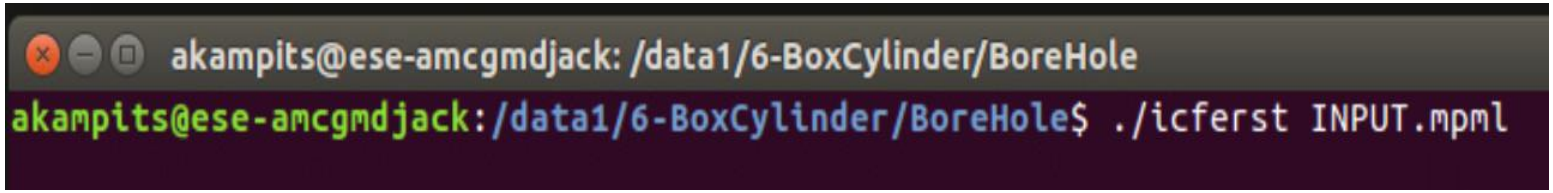
Imperial College London

Run IC-FERST

4) Linux terminal commands to run the code

Run code in serial:

Make directory including .exe, .mpml, and .msh. Open new terminal and run:

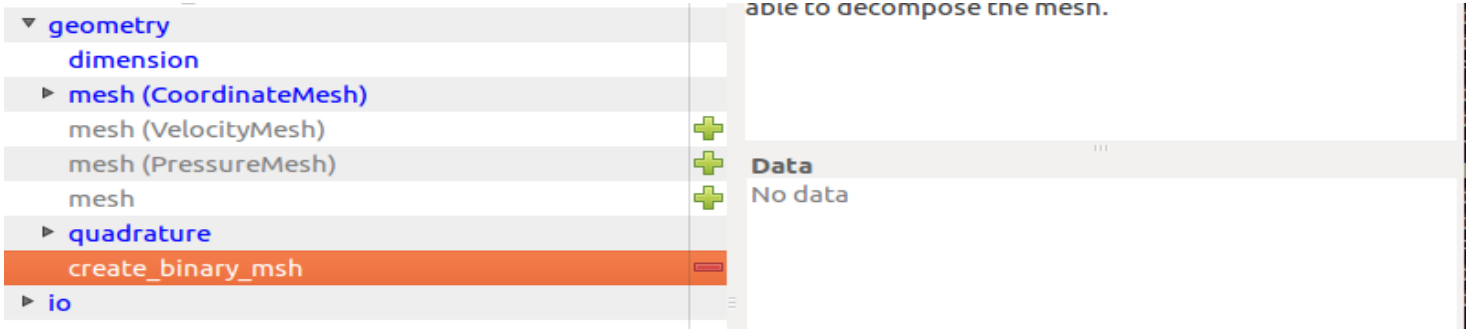


Imperial College London

Run IC-FERST

4) Linux terminal commands to run the code

Run in Parallel: The idea is to split the mesh to subdomains and run them in parallel. You need the mesh in binary format. Convert the mesh to binary mesh using Diamond and IC-FERTS. Read mesh and convert it. Then kill the job. NOTE: the mesh will be overwritten...!



Then use the binary .msh and decompose it to multiple smaller domains

```
fldecomp -n 2 -m gmsh DOMAIN
```

Number of cores you are going to use Mesh file format (without the .msh)

Then run IC-FERST using

```
mpirun -n 4 icferst ./filename.mpml
```

Imperial College London

Visualise Results

5) Paraview

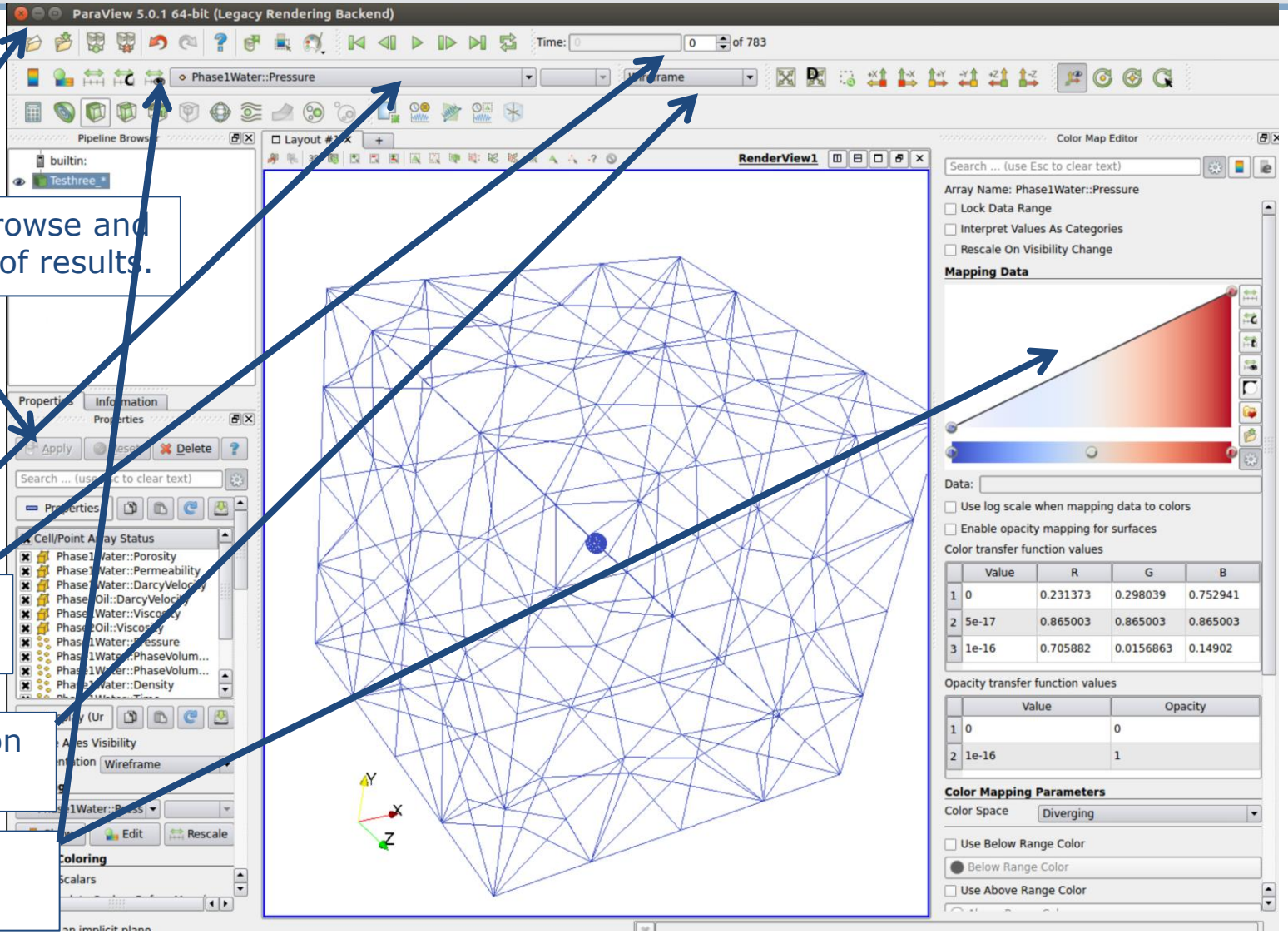
- Results are created in the vtu format file (or pvtu in parallel runs)
- Can be opened using Paraview (www.paraview.org/)
- Phase volume fraction, pressure, velocity, velocity vectors, permeability map, porosity and time are some parameters that can be visualized from the vtu files

  NOTE: The velocity shown in Paraview is not the actual velocity.
- Videos, pictures and plots over lines can be easily obtained using Paraview

Imperial College London

Visualise Results

5) Paraview



Open your file, browse and find your vtu set of results.

Press Apply to load the files.

Select what to visualize

Select the time to visualize

Select visualization options

Contour options

Thanks for your attention.