

IC-FERST Training Course



Novel Reservoir Modelling & Simulation

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



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!



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)



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):

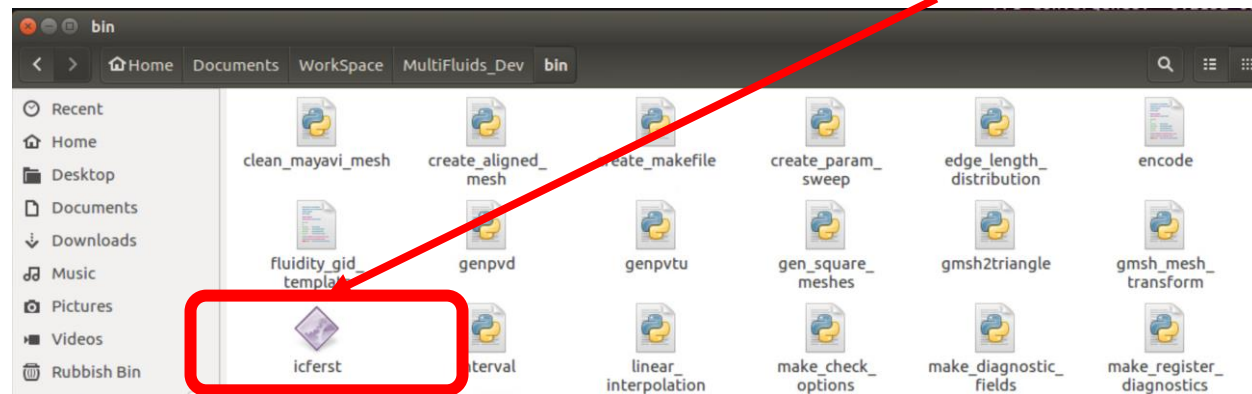
```
./configure --enable-2d-adaptivity && make clean && make mp
```

Create make files

Removes
Previous files

Compile the
program

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

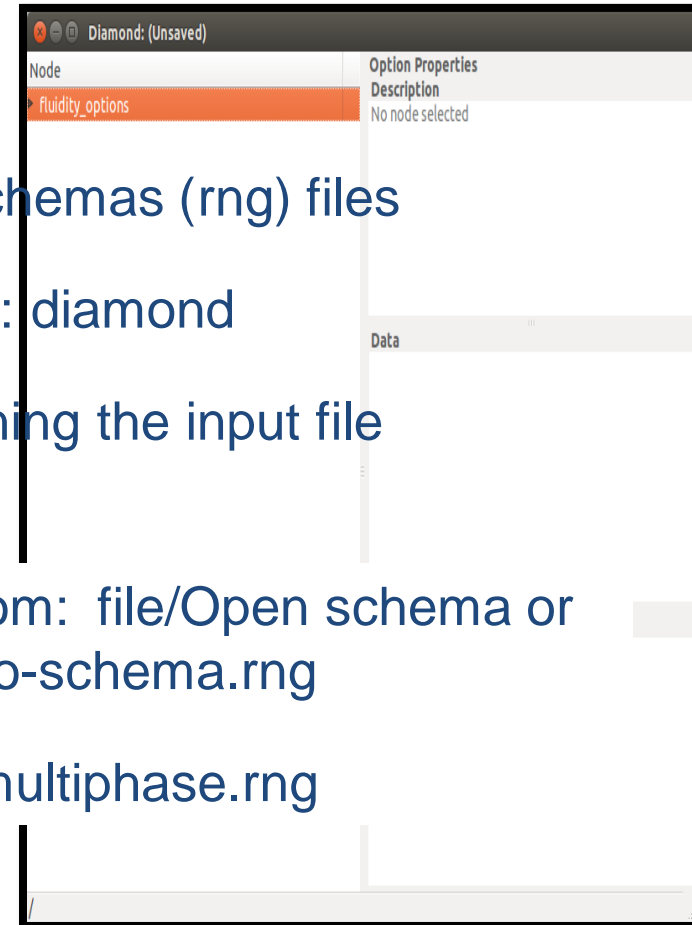




Setting up IC-FERST

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`





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`



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

The screenshot shows the Diamond software interface with the file path `/data1/6-BoxCylinder/BoreHole`. The `icferst_options` node is expanded, showing a list of sub-nodes. Arrows from the text boxes on the left point to specific elements in the interface:

- Open's drop down menu:** Points to the `icferst_options` node.
- Open's drop down options:** Points to the `porous_media_simulator` sub-node.
- Add / Remove:** Points to the `material_phase` sub-node.
- Store Data and Save frequently:** Points to the `numerical_methods` sub-node.

The right panel shows the `Option Properties` section with a description of the selected node. The `Data` section is also visible.



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

Diamond: BH.mpml (/data1/6-BoxCylinder/BoreHole)

Node	Option Properties	Description
icferst_options		No node selected
simulation_name		
porous_media_simulator	▼	
▶ geometry		
▶ solver_options		
▶ io		
timestepping		
physical_parameters	+	Data
▶ material_phase (Phase1Water)	-	
▶ material_phase (Phase2Oil)	-	
material_phase	+	
▶ mesh_adaptivity	-	
▶ porous_media	-	
femdem_fracture	+	
flooding	+	
numerical_methods	+	Comment

Adaptivity parameters

Porosity and permeability are defined here



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

Diamond: BH.mpml (/data1/6-BoxCylinder/BoreHole)

Node	Option Properties
icferst_options	Description
simulation_name	Read mesh from file.
porous_media_simulator	
geometry	
dimension	
mesh (CoordinateMesh)	
from_file	
format (gmsh)	
stat	
simulation_quality	
Advance_options	
create_binary_msh	
solver_options	
io	
timestepping	

Attributes	
Name	Value
file_name	INPUT GEO WITH NO .msh

Data
No data

Element pair.
Most of the times use balanced

Diamond: BH.mpml (/data1/6-BoxCylinder/BoreHole)

Node	Option Properties
icferst_options	Description
simulation_name	Selects the quality configuration of the experiment For Porous media the P1/P0 stands for P0, for inertia for P1. Fast => Element pair = P1/P0DGP1(CV). Upwinding and implicit Euler; Balanced (recommended) => Element pair = P1/P0DGP1(CV). High order advection schemes and adaptive theta.
porous_media_simulator	
geometry	
dimension	
mesh (CoordinateMesh)	
from_file	
format (gmsh)	
stat	
simulation_quality	
Advance_options	
create_binary_msh	
solver_options	

Data
balanced



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 screenshot shows the 'Diamond' software interface. The left pane displays a tree view of parameters under the 'Node' column. The 'icferst_options' node is expanded, showing sub-nodes like 'simulation_name', 'porous_media_simulator', 'geometry', 'solver_options', 'io', 'dump_format', 'dump_period_in_timesteps', 'constant', 'PropertiesFromFile', 'Show_Convergence', 'Courant_number', 'dump_boundaryflux', 'checkpointing', 'detectors', 'log_output', 'Mesh_Diagnostics_Angles', 'timestepping', 'current_time', 'timestep', 'finish_time', 'adaptive_timestep', 'physical_parameters', and 'material_phase (Phase1Water)'. The 'constant' node under 'dump_period_in_timesteps' is selected and highlighted in orange. The right pane shows the 'Option Properties' for the selected node, with a 'Description' of 'No documentation' and a 'Data' value of '10'. Below the right pane are buttons for 'Revert data' and 'Store data', and a 'Comment (string)' field. Arrows from the text boxes on the left point to specific parameters in the tree: one to 'dump_format', one to 'dump_period_in_timesteps', one to 'constant', and one to 'adaptive_timestep'.

Option Properties
Description
No documentation

Data
10

Revert data Store data

Comment (string)

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

Input file format for IC-FERST

3) Physical parameters

Section to define physical parameters

9.8 in the SI

Direction of the gravity

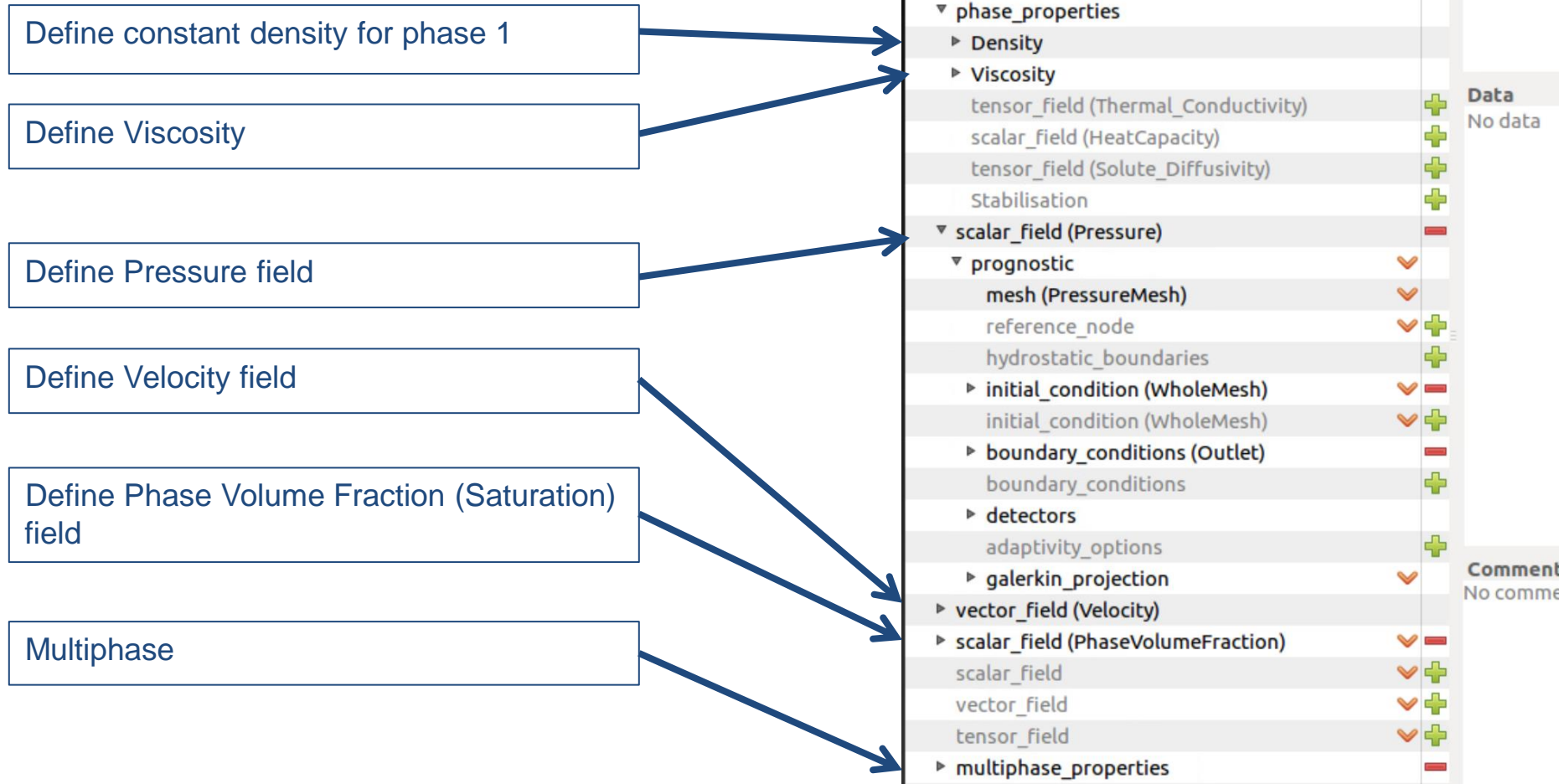
Direction of the gravity
set to $-y$ (i.e: pointing down)

	Data	
timestepping		
physical_parameters		
gravity		
magnitude		
hydrostatic_pressure_solver	+	
vector_field (GravityDirection)		
prescribed		
mesh (CoordinateMesh)	∨	0
value (WholeMesh)	∨	
constant	∨	
value (WholeMesh)	∨ +	
stat		
detectors		
black-oil_PVT_table	+	
coriolis	+	-1
material_phase (Phase1Water)		
is_multiphase_component	+	
phase_properties		
Density		
Viscosity		
tensor_field (Thermal_Conductivity)	+	
scalar_field (HeatCapacity)	+	d
tensor_field (Solute_Diffusivity)	+	
Stabilisation	+	
scalar_field (Pressure)		
prognostic	∨	
mesh (PressureMesh)	∨	
reference_node	∨ +	



Input file format for IC-FERST

3) Phase Properties for each Phase





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

▼ material_phase (Phase1Water)	
is_multiphase_component	+
▼ phase_properties	
▼ Density	
incompressible	▼
linearise_density	+
▼ Viscosity	
▼ tensor_field (Viscosity)	
▼ prescribed	▼
mesh (VelocityMesh)	▼
▼ value (WholeMesh)	▼
▼ isotropic	▼
constant	▼
value (WholeMesh)	▼
linearise_viscosity	+
► viscosity_scheme	
tensor_field (Thermal_Conductivity)	+
scalar_field (HeatCapacity)	+
tensor_field (Solute_Diffusivity)	+
Stabilisation	+
▼ scalar_field (Pressure)	
▼ prognostic	▼
mesh (PressureMesh)	▼
reference_node	▼
hydrostatic_boundaries	+
▼ initial_condition (WholeMesh)	▼
constant	▼
initial_condition (WholeMesh)	▼
▼ boundary_conditions (Outlet)	
surface_ids	
▼ type (dirichlet)	▼
▼ apply_weakly	
boundary_overwrites_initial_condition	+
constant	▼
boundary_conditions	+
► detectors	
adaptivity_options	+
► galerkin_projection	▼



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

▼ vector_field (Velocity)	
▼ prognostic	▼
mesh (VelocityMesh)	▼
▶ initial_condition (WholeMesh)	▼
initial_condition (WholeMesh)	▼ +
▶ boundary_conditions (Inlet)	—
▶ boundary_conditions (TopBottom)	—
boundary_conditions	+
vector_field (Source)	+
vector_field (Absorption)	+
▶ detectors	
adaptivity_options	+
▼ galerkin_projection	▼
discontinuous	▼
supermesh_free	+
supermesh_conservation	+
honour_strong_boundary_conditions	+
▶ scalar_field (PhaseVolumeFraction)	▼ —



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

▼ scalar_field (PhaseVolumeFraction)	▼	1.0
▼ prognostic	▼	
mesh (PressureMesh)	▼	
► initial_condition (WholeMesh)	▼	
initial_condition (WholeMesh)	▼	
▼ boundary_conditions (Inlet)	▼	
surface_ids		
▼ type (dirichlet)	▼	
▼ apply_weakly	▼	
boundary_overwrites_initial_condition		
component_density		
constant	▼	
boundary_conditions		
scalar_field (Source)		
scalar_field (Absorption)		
tensor_field (Absorption)		
► detectors		
▼ adaptivity_options	▼	
▼ absolute_measure	▼	
▼ scalar_field (InterpolationErrorBound)	▼	
▼ prescribed	▼	
► value (WholeMesh)	▼	
value (WholeMesh)	▼	
► stat		
► detectors		
p_norm		
preprocessing		
► galerkin_projection	▼	

Reve

Comment
(string)



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

▼ multiphase_properties	
▼ Relperm_Corey	
▼ scalar_field (relperm_max)	
▼ prescribed	
mesh (P0DG)	▼
▼ value (WholeMesh)	▼
constant	▼
value (WholeMesh)	▼ +
▶ stat	
▶ detectors	
adaptivity_options	+
consistent_interpolation	▼ +
▼ scalar_field (relperm_exponent)	
▼ prescribed	
mesh (P0DG)	▼
▼ value (WholeMesh)	▼
constant	▼
value (WholeMesh)	▼ +
▶ stat	
▶ detectors	
adaptivity_options	+
consistent_interpolation	▼ +
▼ capillary_pressure	
type_Brooks_Corey	+
type_Power_Law	+
▼ immobile_fraction	
▼ scalar_field (value)	
▼ prescribed	
mesh (P0DG)	▼
▶ value (WholeMesh)	▼
value (WholeMesh)	▼ +
▶ stat	
▶ detectors	
adaptivity_options	+
consistent_interpolation	▼ +
▶ material_phase (Phase2Oil)	



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.

Description

Python command prescribing symmetric tensor input.
Note that it is for the python function to determine that the results it produces are, in fact, symmetric.
An example that returns the three-dimensional identity:

```
def val(X, t):
    return [[1, 0, 0],
            [0, 1, 0],
            [0, 0, 1]]
```

Data

```
1 def val(X, t):
2     val= [[0.707e-4, 0],[0, 0.707e-4]]
3     #Equivalent to a permeability of 100
4     if (X[0]<4.9 and X[1]>5.1):
5         if (X[0]+1<X[1]):
6             val = [[5e2, -4e2],[-4e2, 5e2]]
7         #This [[9e2, 0],[0, 1e2]] rotated 45 degrees
8     if (X[0]>5.1 and X[1]<4.9):
9         if (X[0]-1>X[1]):
10            val = [[5e2, -4e2],[-4e2, 5e2]]
11        #This [[9e2, 0],[0, 1e2]] rotated 45 degrees
12
13    if ((X[0]>4.9 and X[0]<5.1) and (X[1]<4.3 or X[1]>5.7)):
14        val = [[0.707e-4, 0],[0, 0.707e-4]]
15
16    if ((X[1]>=4.9 and X[1]<=5.1) and (X[0]<=4.3 or X[0]>=5.7)):
17        val = [[0.707e-4, 0],[0, 0.707e-4]]
```

Advanced users only!

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 ...



Input file format for IC-FERST

3) Adaptivity

The screenshot shows the input file format for IC-FERST, specifically the adaptivity section. The interface is a tree view of parameters with corresponding input fields on the right. Annotations with arrows point to specific parameters and their values:

- Adapt the mesh every this time steps:** Points to the `period_in_timesteps` parameter, which has a value of `0.0004`.
- Limit the number of nodes, set it to 100000:** Points to the `maximum_number_of_nodes` parameter, which has a value of `0`.
- Maximum variation in size between neighbouring triangles. Default 1.5:** Points to the `gradation_parameter` parameter, which has a value of `0.0004`.
- Maximum and minimum lengths of the elements:** Points to the `aspect_ratio_bound` parameter, which has a value of `0.0004`.
- Limit the aspect ratio of the triangles, 1 means equilateral triangles. Use from 5 to 25:** Points to the `adapt_at_first_timestep` parameter, which has a value of `0.0004`.
- Delay adaptivity. Set value in real time.** Points to the `t_adapt_delay` parameter, which has a value of `0.0004`.

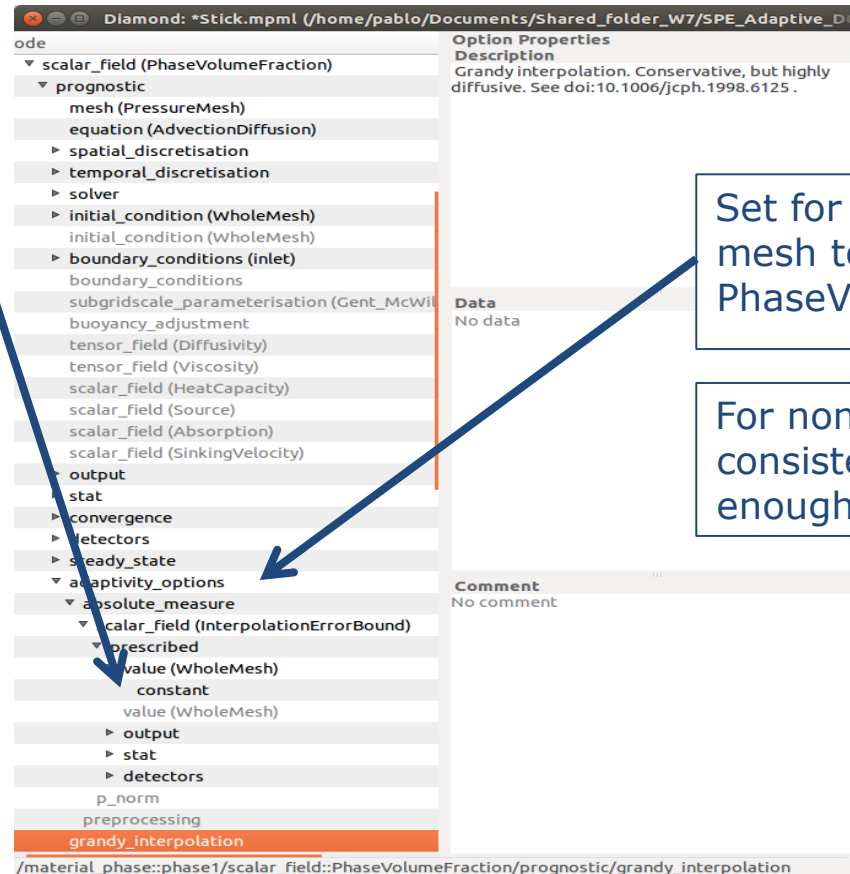
The interface also includes a `Store data` button at the bottom right.



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

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

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

For non-conservative fields the consistent interpolation is enough



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:

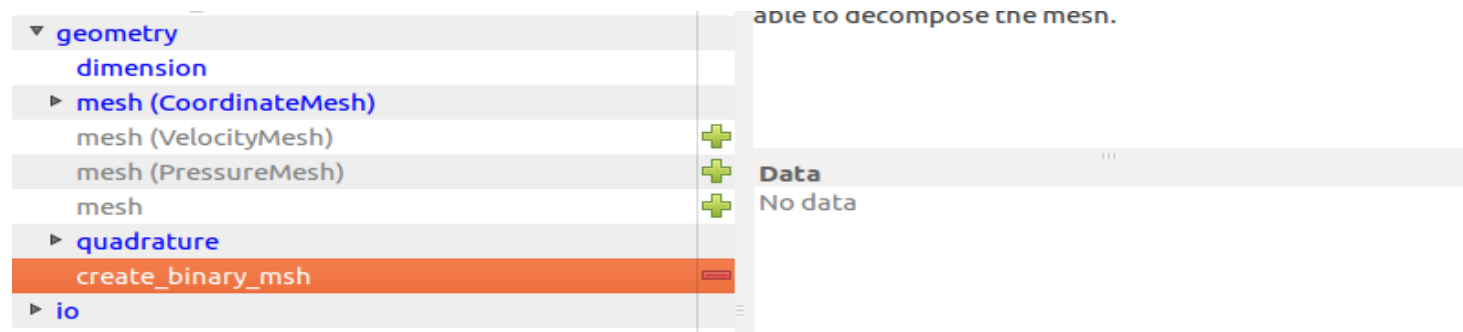
```
akampits@ese-amcgmdjack: /data1/6-BoxCylinder/BoreHole
akampits@ese-amcgmdjack:/data1/6-BoxCylinder/BoreHole$ ./icferst INPUT.mpml
```



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 gmsht 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
```



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



Visualise Results

5) Paraview

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

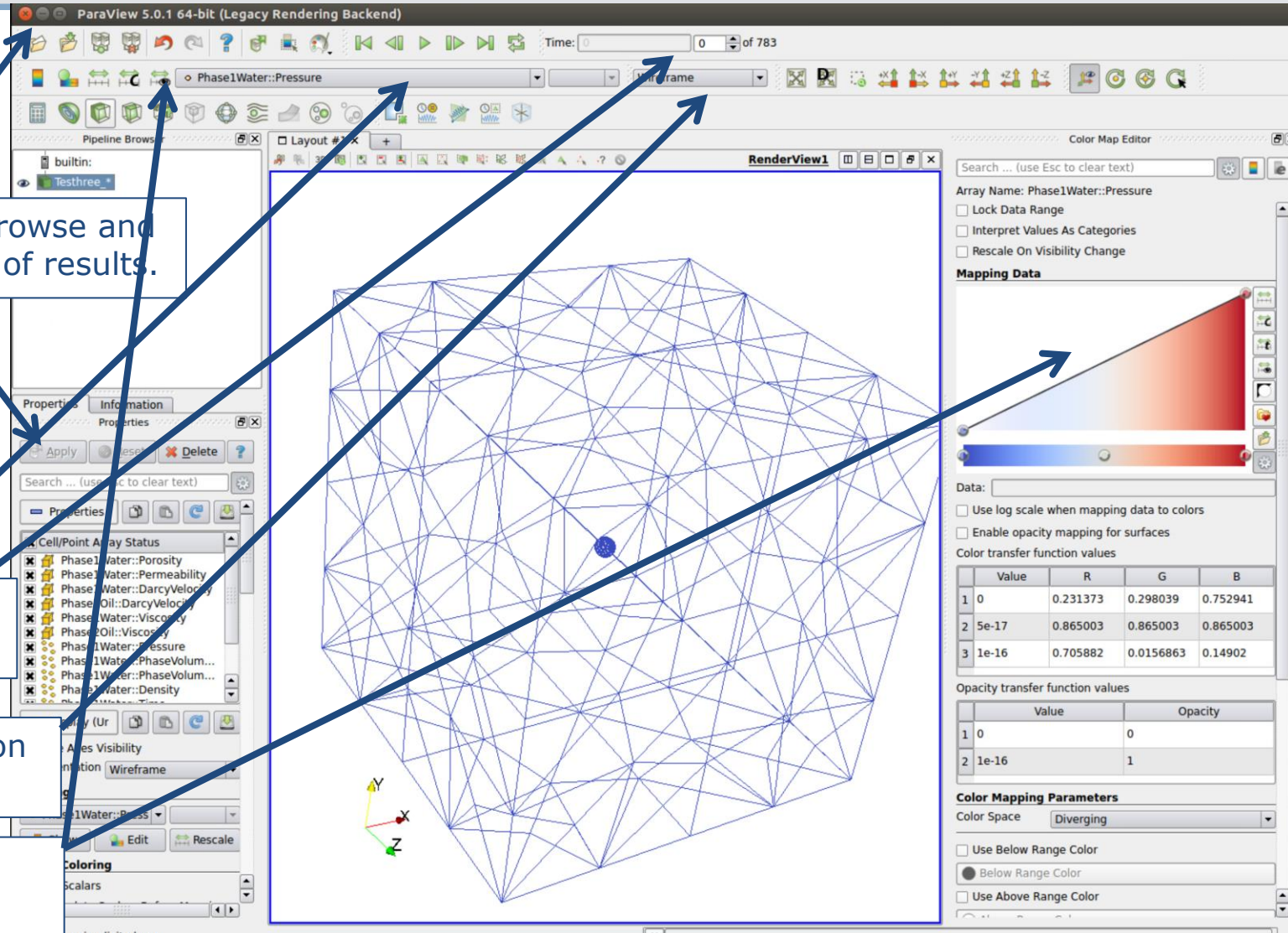
Press Apply to load the files.

Select what to visualize

Select the time to visualize

Select visualization options

Contour options



Color Map Editor

Search ... (use Esc to clear text)

Array Name: Phase1Water::Pressure

☐ Lock Data Range

☐ Interpret Values As Categories

☐ Rescale On Visibility Change

Mapping Data

☐ Use log scale when mapping data to colors

☐ Enable opacity mapping for surfaces

Color transfer function values

	Value	R	G	B
1	0	0.231373	0.298039	0.752941
2	5e-17	0.865003	0.865003	0.865003
3	1e-16	0.705882	0.0156863	0.14902

Opacity transfer function values

	Value	Opacity
1	0	0
2	1e-16	1

Color Mapping Parameters

Color Space: Diverging

☐ Use Below Range Color

☒ Below Range Color

☐ Use Above Range Color



IC-FERST

Thanks for your attention.

