ASPECT Tutorial

John Naliboff, Rene Gassmoeller, Juliane Dannberg

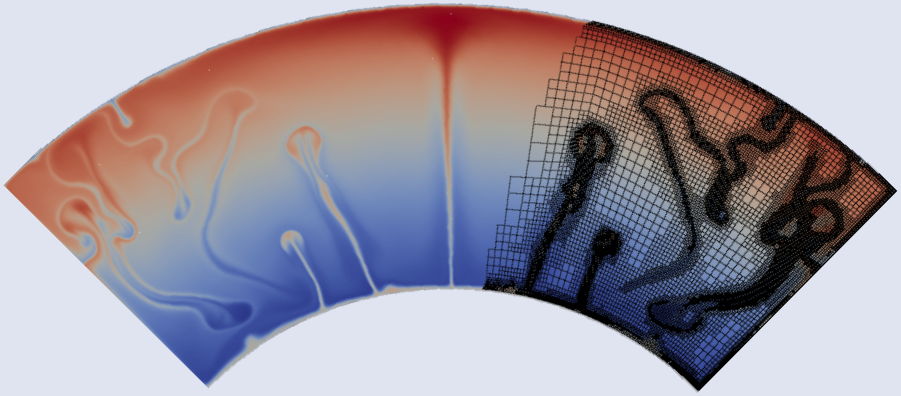
With material from:
Wolfgang Bangerth, Timo Heister, and many ASPECT contributors

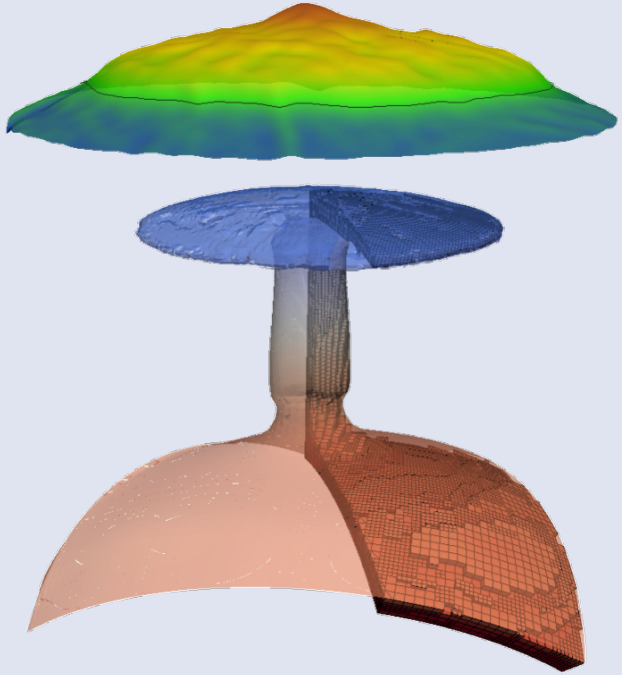
What is Aspect?

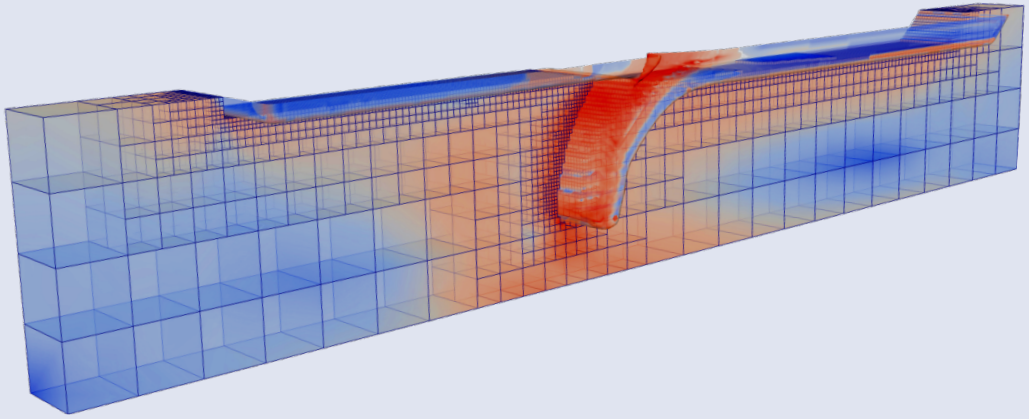
CIG COMPUTATIONAL INFRASTRUCTURE for GEODYNAMICS

ASPECT

- Advanced Solver for Problems in Earth's ConvecTion -







ASPECT: Methods

- **Modern numerical methods:**
  adaptive mesh refinement, linear and nonlinear solvers, higher-order discretizations, stabilization schemes
- **Usability and extensibility:**
  manual: 450+ pages, ~40 cookbooks/examples
  plugin architecture
- **Parallel scalability**
- **Building on others' work:**
  tested foundation, smaller codebase, automatic improvements
- **Community:**
  GPL, developed in the open
  Encourage contributions, be welcoming

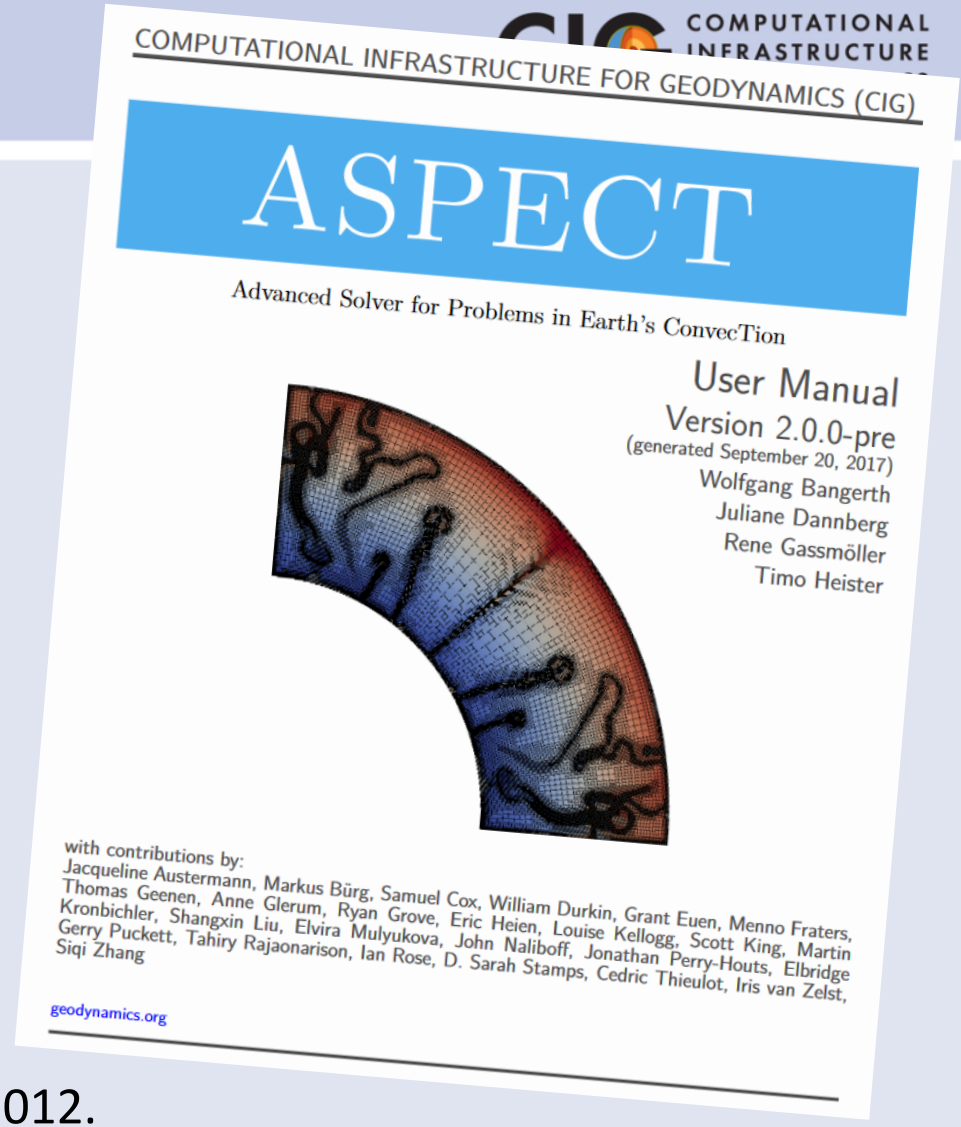
Credits

Website and manual:
aspect.geodynamics.org

Maintainers:

Wolfgang Bangerth,
Juliane Dannberg,
Timo Heister, Rene Gassmöller

Contributors: many more (~45)

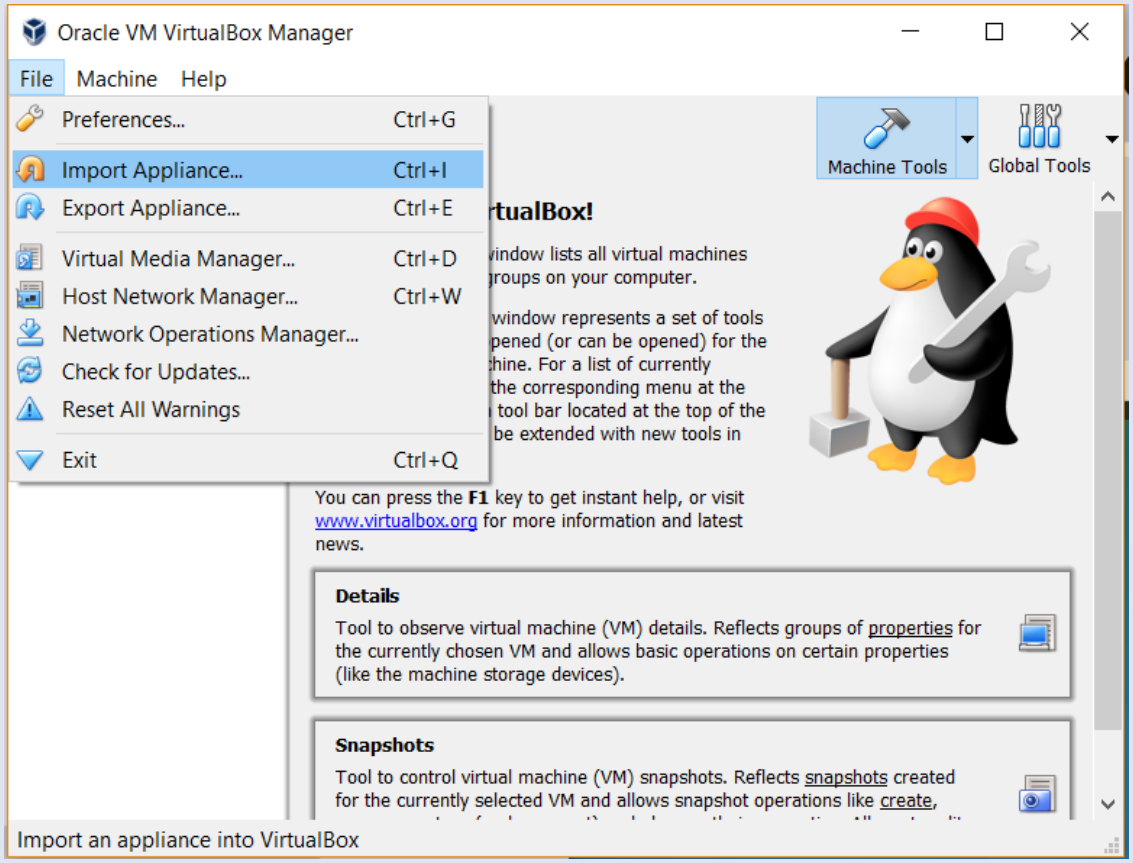


Publications: (~30)

- Kronbichler, Heister, Bangerth: "High Accuracy Mantle Convection Simulation through Modern Numerical Methods". *Geophysical Journal International*, 2012.
- Heister, Dannberg, Gassmoeller, Bangerth: "High Accuracy Mantle Convection Simulation through Modern Numerical Methods. II: Realistic Models and Problems". *Geophysical Journal International,* 2017.

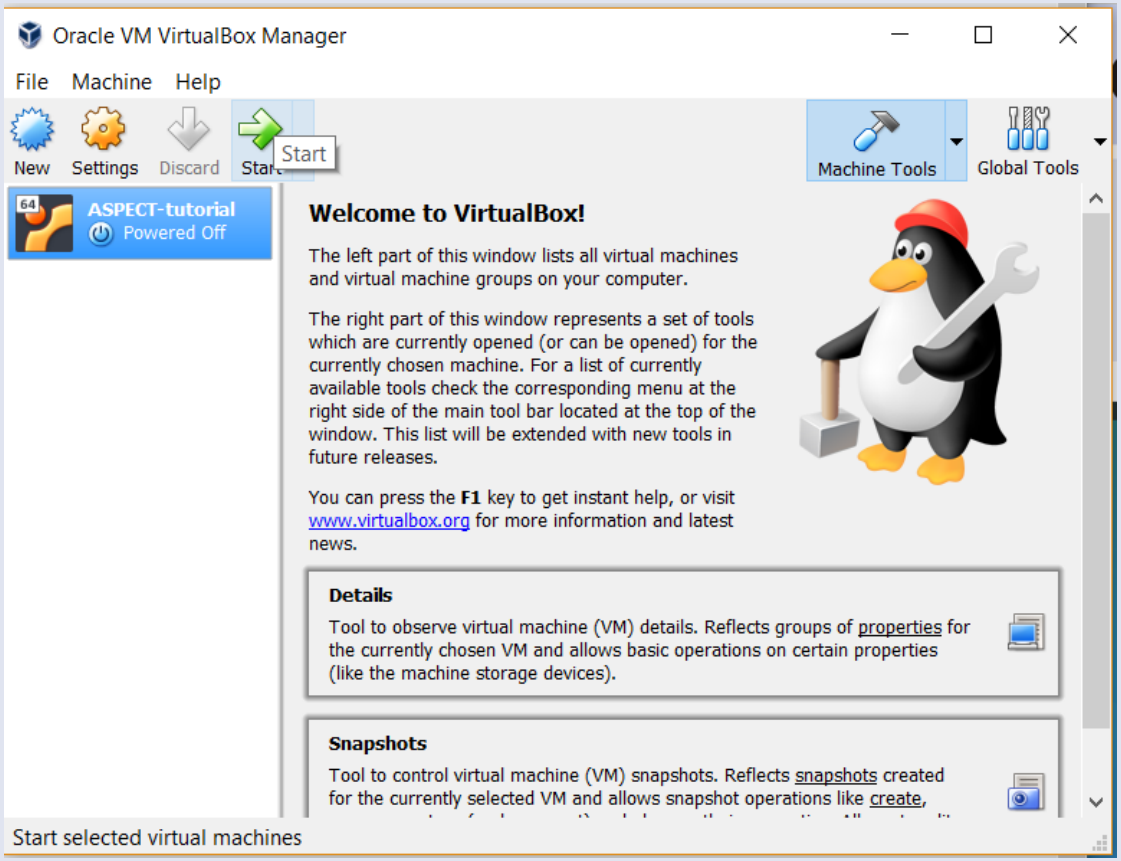
VirtualBox

- Start Oracle VM VirtualBox and import VM



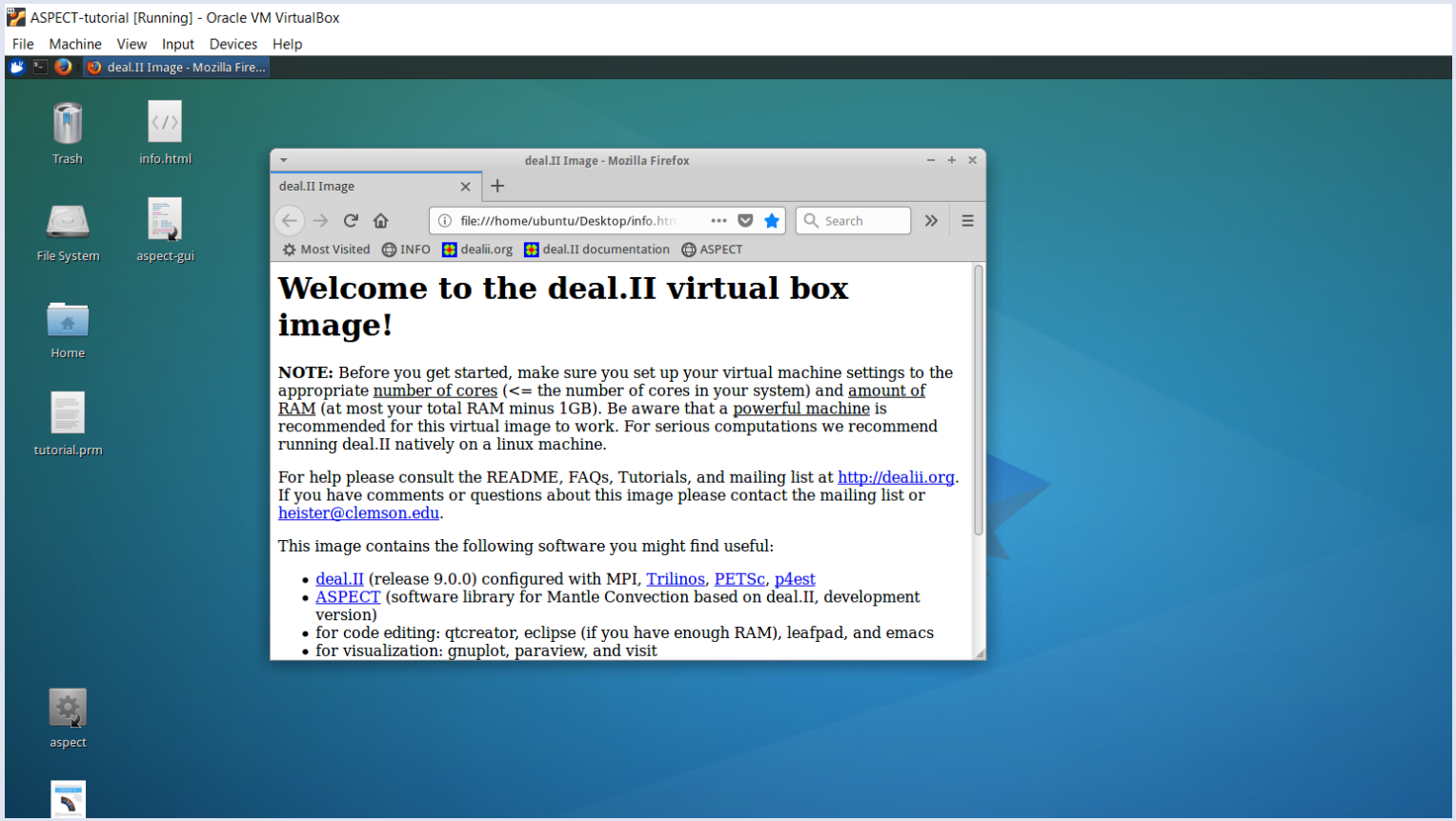
VirtualBox

- Start ASPECT-tutorial virtual machine



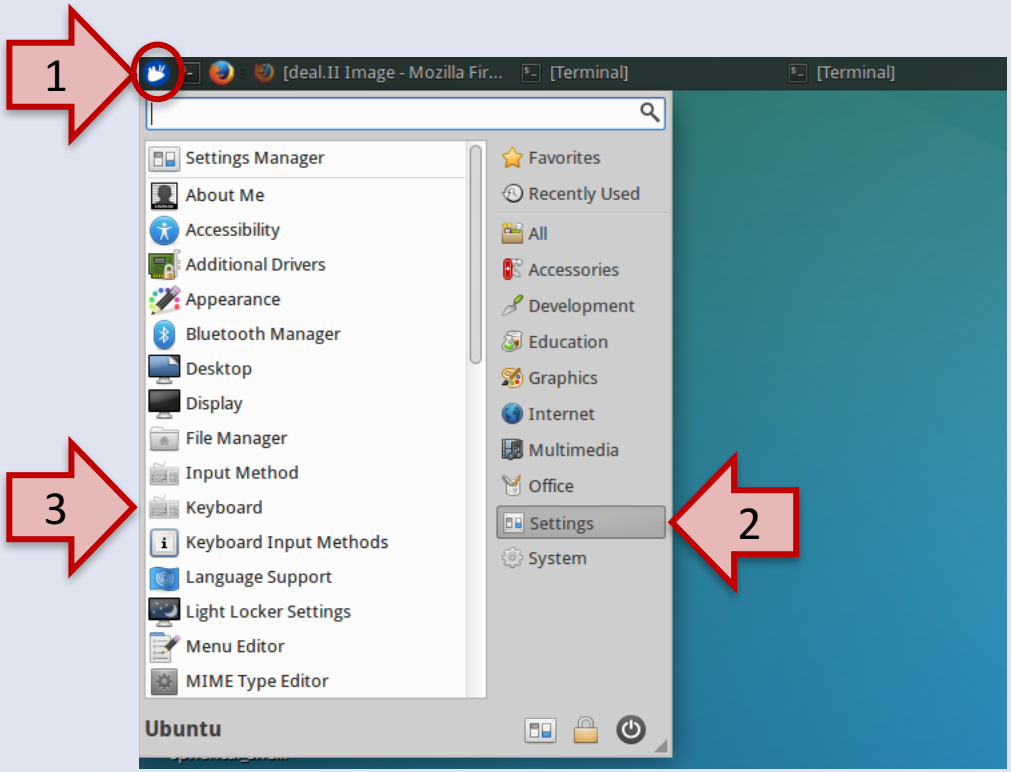
VirtualBox

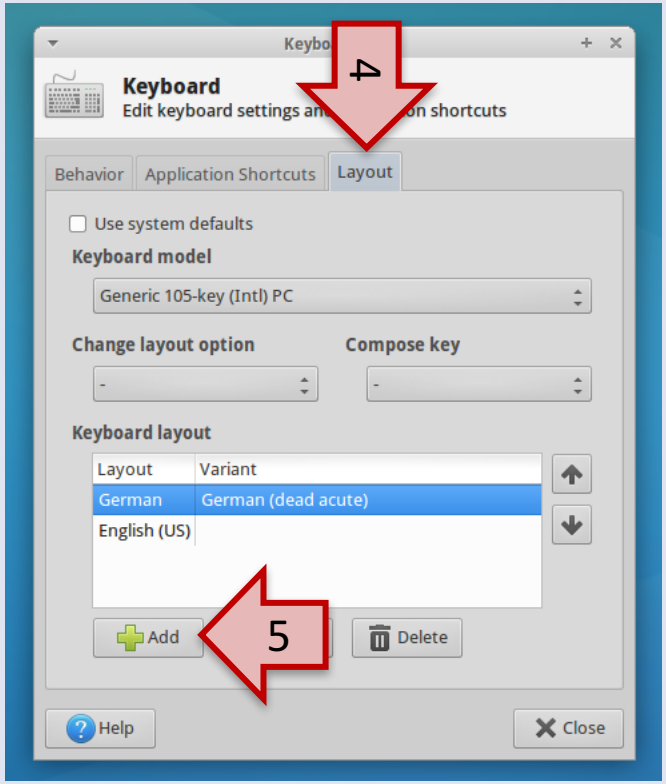
- You should see a screen like this:



VirtualBox: Keyboard layout

- If you have a different keyboard layout than the standard English(US)…





Using ASPECT

- Basic usage of ASPECT is specified through a parameter file (e.g. tutorial.prm)
- The parameter file is used by the simulation to determine the discretization, parameters, initial conditions, boundary conditions, etc.
- By the end of this tutorial, you should be able to:
  1. Run aspect from the command line.
  2. Understand the basic layout of the parameter files that are used to control Aspect simulations.
  3. Be able to visualize the generated output in ParaView.

Using ASPECT

- We will begin by running ASPECT in the Terminal (ctrl + alt + t)

1. Change to the appropriate directory

```
cd Desktop
```

2. Run ASPECT with the tutorial parameter file and print the output
(this will take about 20 seconds)

```
./aspect tutorial.prm
```

3. Open log.txt and look at the output

```
gedit output-tutorial/log.txt
```

Visualizing Results with ParaView

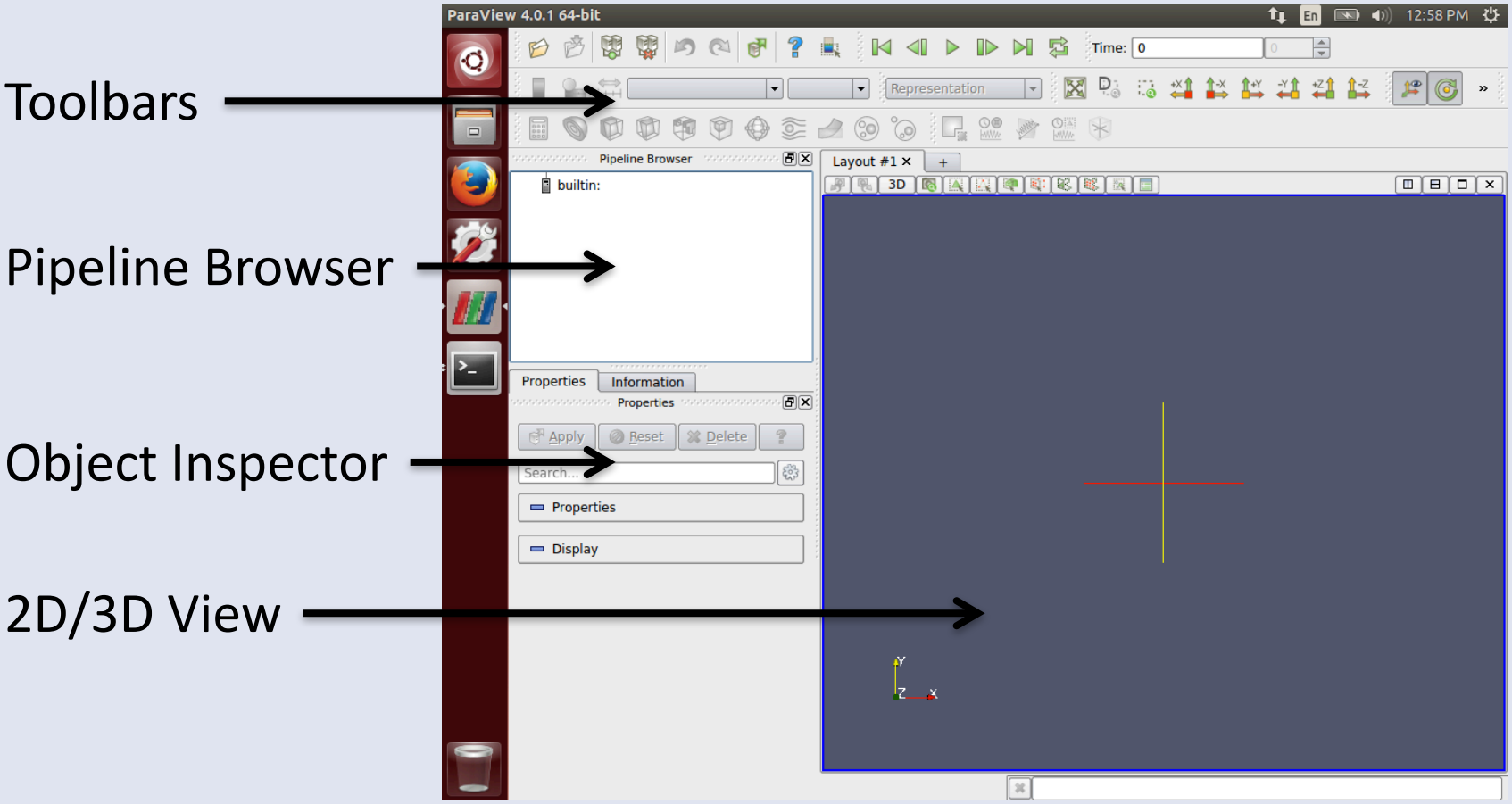
Visualization with ParaView

CIG COMPUTATIONAL INFRASTRUCTURE for GEODYNAMICS

- To visualize the simulation results, we will use ParaView

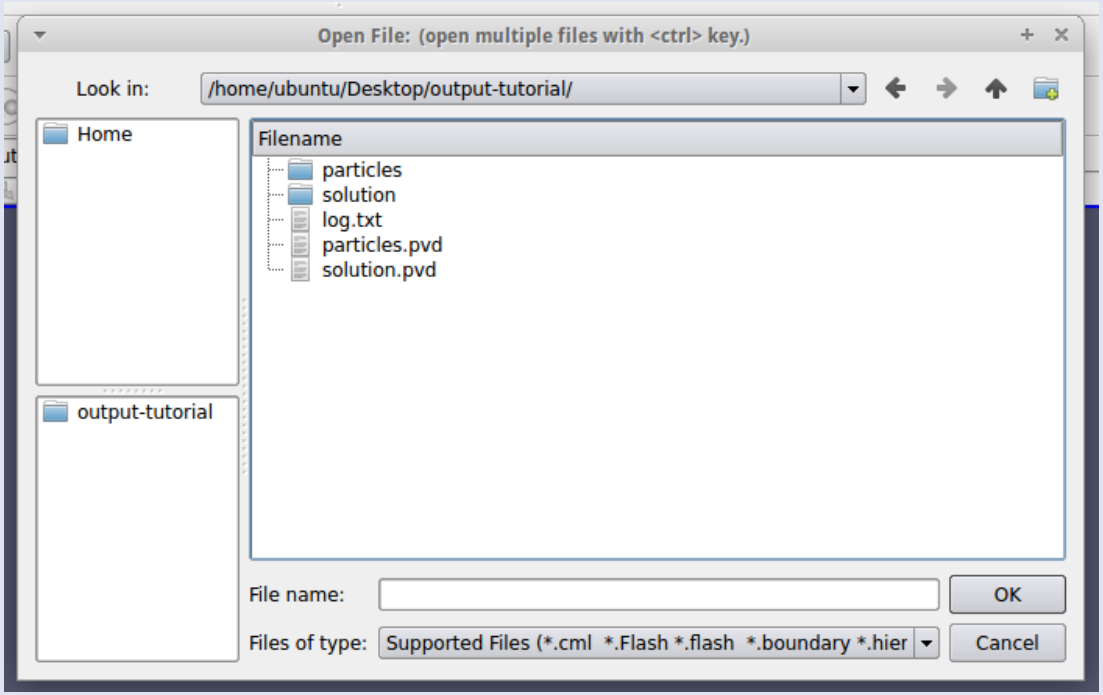
- ParaView is a program for visualization of large data sets
- It is already installed on the virtual machine, open it now by typing “paraview” in a terminal
- ParaView supports visualization tools such as isosurfaces, slices, streamlines, volume rendering, and other complex visualization techniques

Visualization with ParaView



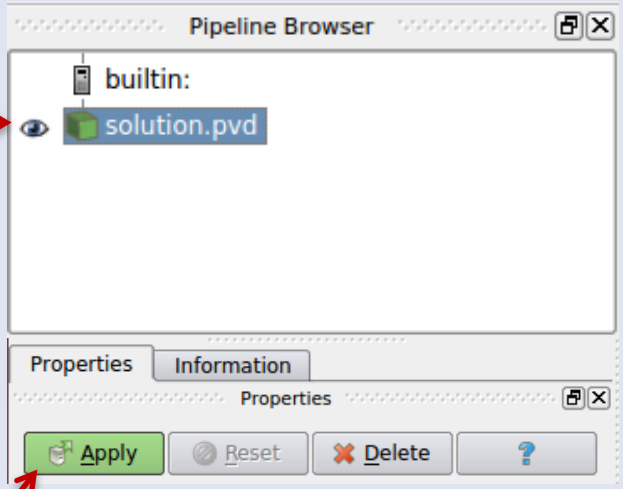
Visualization with ParaView

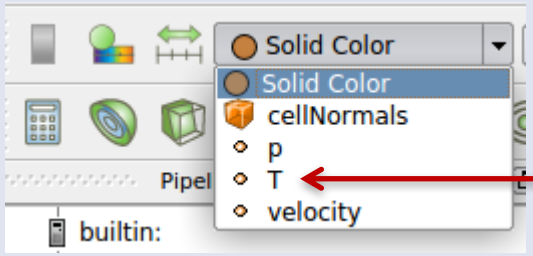
- Start by opening solution.pvd which was created by running ASPECT
- You can choose “Open” from the File menu or use the Open icon  in the toolbar
- The file is in /home/ubuntu/Desktop/output-tutorial/

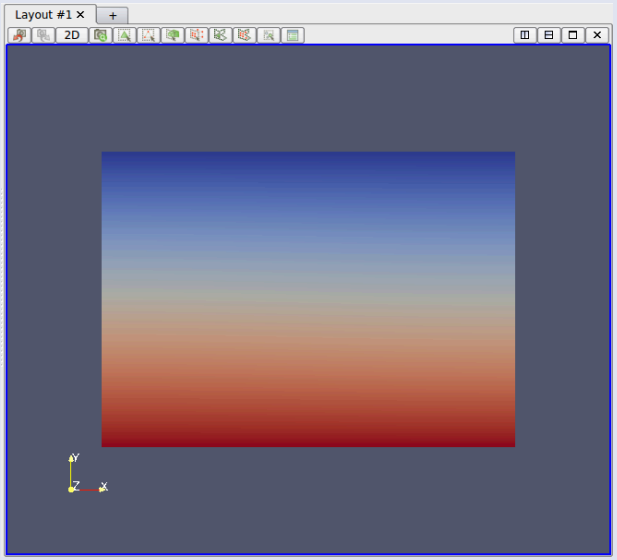


Visualization with ParaView

- The file will appear in the pipeline browser
  - Make sure this is solution.pvd
- Click “Apply” to show the field in the view area
  - By default, no field is shown
  - Select “T” in the toolbar to show the temperature field

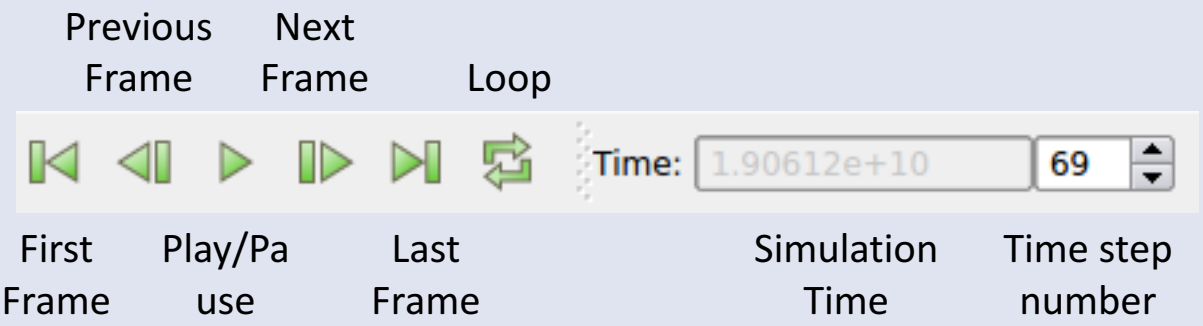


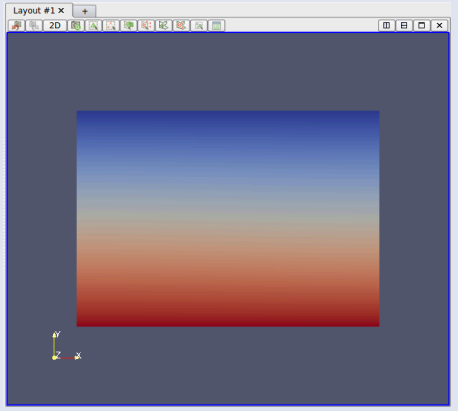




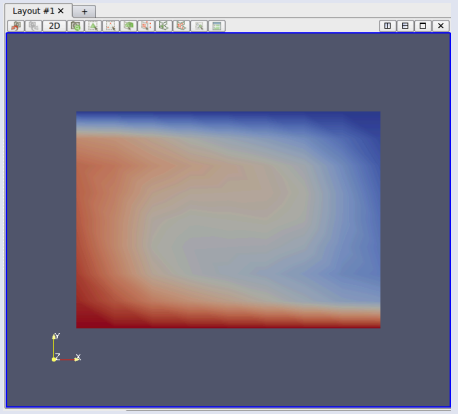
Visualization with ParaView

- The top toolbar has buttons to change the time, shown below
  - Click the play button and watch how the temperature field changes
  - Near the end, is the temperature field static? Is the velocity field static? Is material moving?





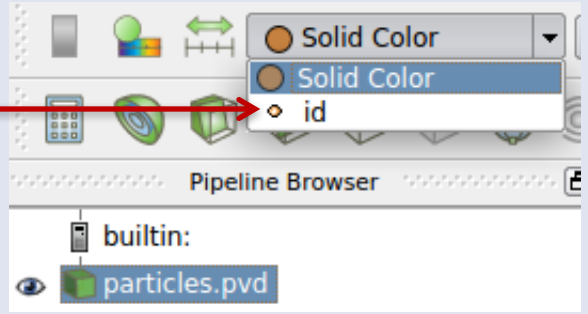
Frame 0



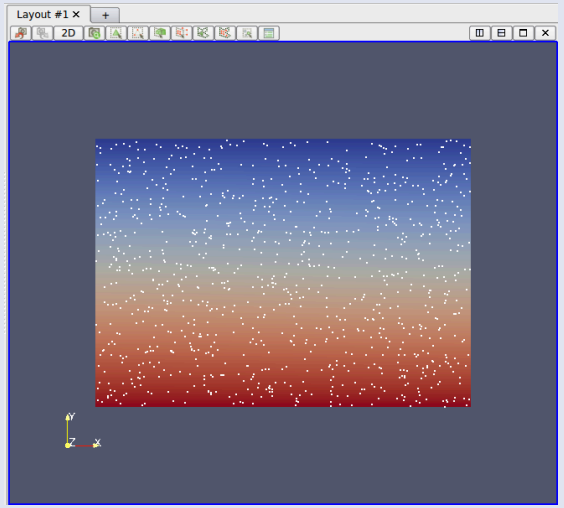
Frame 231

Visualization with ParaView

- Open the file particles.pvd and click “Apply”
  - The tracer particles from the simulation now appear on the temperature field
  - By default they are uniformly colored
  - Change the coloring scheme to “id” to show each particle in a different color
  - Click play again to see how material is flowing with the tracer particles
  - Even when the temperature field is static, is material flowing?
  - How would you characterize this flow pattern? Where is the upwelling material? The downwelling material?



Change the coloring scheme to “id”



Temperature field with tracer particles

Setup of the numerical model

- Numerical models generally consist of several key components:
  1. The rules (e.g. equations) for the model
  2. The discretization of the model (the mesh)
  3. Model parameters
  4. Dependent and independent variables
  5. The initial state of the model
  6. The boundary conditions
- We will go through the parameter file and look at these components

```
aspect-gui tutorial.prm
```

or

```
gedit tutorial.prm
```

Input file in **gedit**

COMPUTATIONAL INFRASTRUCTURE for GEODYNAMICS

tutorial.prm (~/Desktop) - gedit

File Edit View Search Tools Documents Help

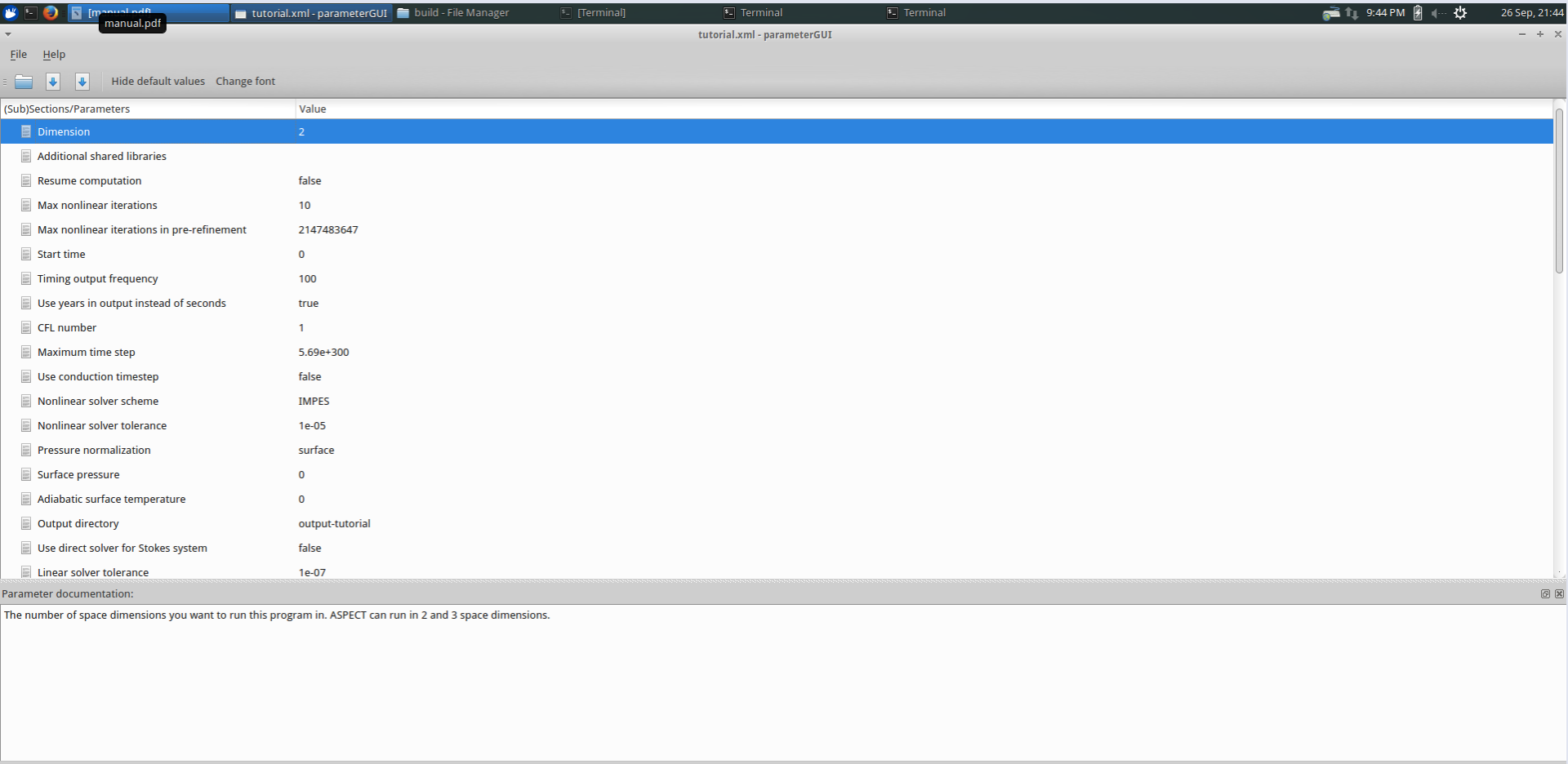
Open Save Undo

tutorial.prm

```
# At the top, we define the number of space dimensions we would like to
# work in:
set Dimension                              = 2

# There are several global variables that have to do with what
# time system we want to work in and what the end time is. We
# also designate an output directory.
set Use years in output instead of seconds = true
set End time                               = 5e10
set Output directory                       = output-tutorial


# Then come a number of sections that deal with the setup
# of the problem to solve. The first one deals with the
# geometry of the domain within which we want to solve.
# The sections that follow all have the same basic setup
# where we select the name of a particular model (here,
# the box geometry) and then, in a further subsection,
# set the parameters that are specific to this particular
# model.
subsection Geometry model
  set Model name = box
  subsection Box
    set X extent = 4.2e6
    set Y extent = 3e6
  end
end


# The following section deals with the discretization of
# this problem, namely the kind of mesh we want to compute
# on. We here use a globally refined mesh without
# adaptive mesh refinement.
subsection Mesh refinement
  set Initial global refinement                = 3
  set Initial adaptive refinement              = 0
  set Time steps between mesh refinement       = 0
end


# The following two sections describe first the
# direction (vertical) and magnitude of gravity and the
# material model (i.e., density, viscosity, etc).
subsection Gravity model
  set Model name = vertical
  subsection Vertical
    set Magnitude = 9.81
  end
end
```

Plain Text

Parameter GUI



- Hide default values…