Geodynamics Tutorial II - Numerical Modeling

Cian Wilson
Peter van Keken

We'll need roughly 5 groups around the room.

Please make sure each group has someone with experience of running models/using a terminal.

http://www.smbc-comics.com/comic/2012-05-01

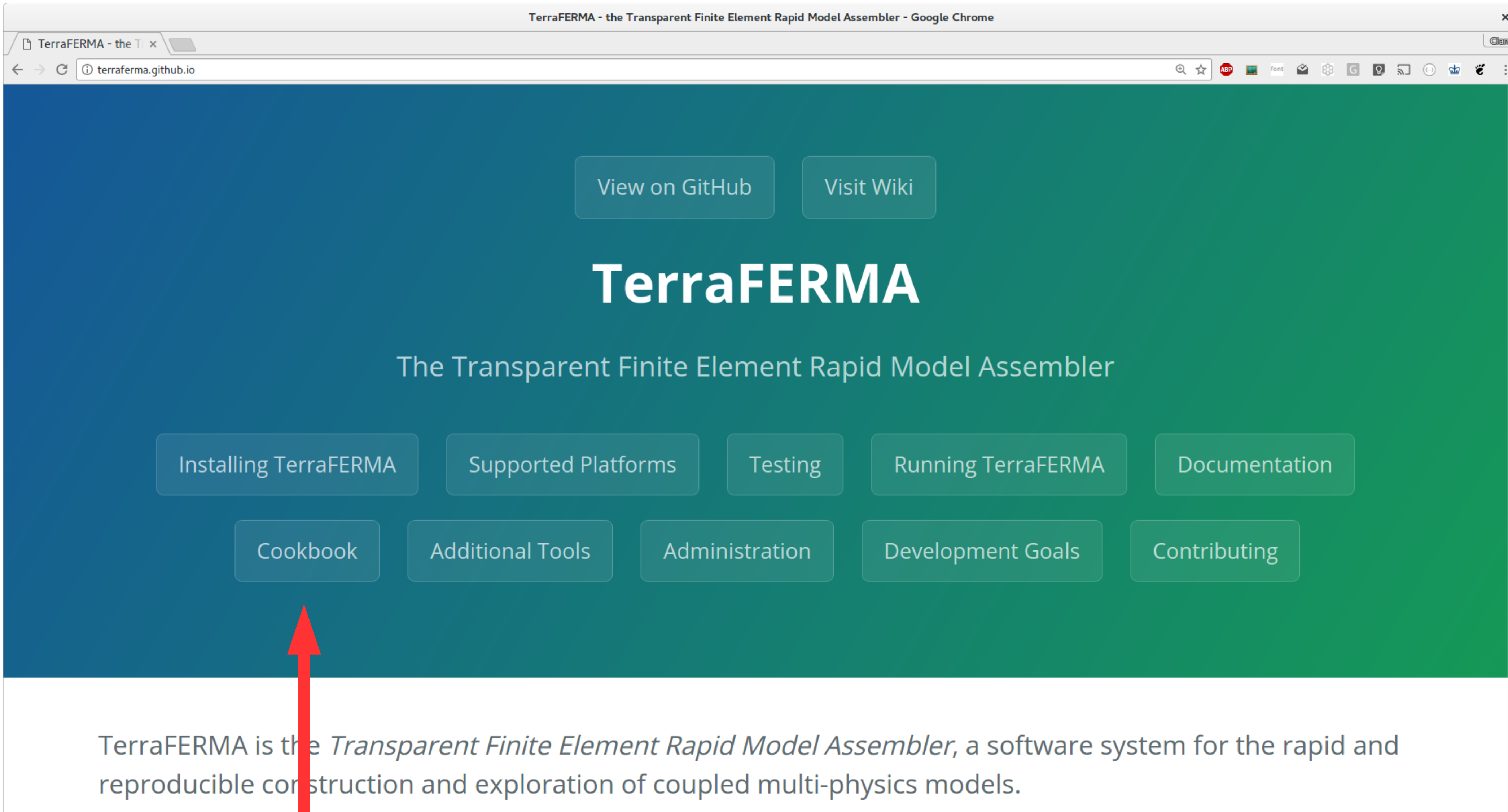
TerraFERMA

Lightweight wrapper exposing functionality from core libraries (among others):

- FEniCS
  - finite element library allowing the easy representation of partial differential equations
- PETSc
  - scientific computation library providing linear and non-linear solvers
- SPuD
  - options management library

Aims to increase testability, shareability and reproducibility.

http://terraferma.github.io



fully worked examples available here

Terminal Cheat Sheet

Useful commands:

```
cd <directory name>
```
change directory

```
ls
```
list files in current directory

```
rm <file name>
```
remove file

```
eog <image file name>
```
view an image

```
pwd
```
print current directory

To run a command in the background (get control of your terminal back) add a & symbol after the command, or, if it's already running:

```
Ctrl z
bg
```

To kill a command (that's not in the background):

```
Ctrl C
```

We'll be using the command:

```
tfsimulationharness --test <shml file name>
```

or the shorter/easier to type version:

```
tfs --test <shml file name>
```

When we vary parameters, this takes additional arguments:

```
tfs --parameters <name> <value> [<name> <value> ...] --test <shml file name>
```

where you can specify as many `<name> <value>` pairs as you want.

Before we begin...

Start virtual machine.

Open terminal.

Change directory to CIDER tutorials:

```
cd cider_tutorials
```

Get updates:

```
git pull
```

Examples

Instantaneous Heat Diffusion

Driven Lid Convection

Thermal Subduction Zone ★

Buoyancy Driven Wedge Fluid Flow ☺

Compaction (& Buoyancy) Driven Fluid Flow ☺

Subduction Benchmarks

Instantaneous Heat Diffusion

Solve:

$$-\nabla \cdot (k \nabla T) = f$$

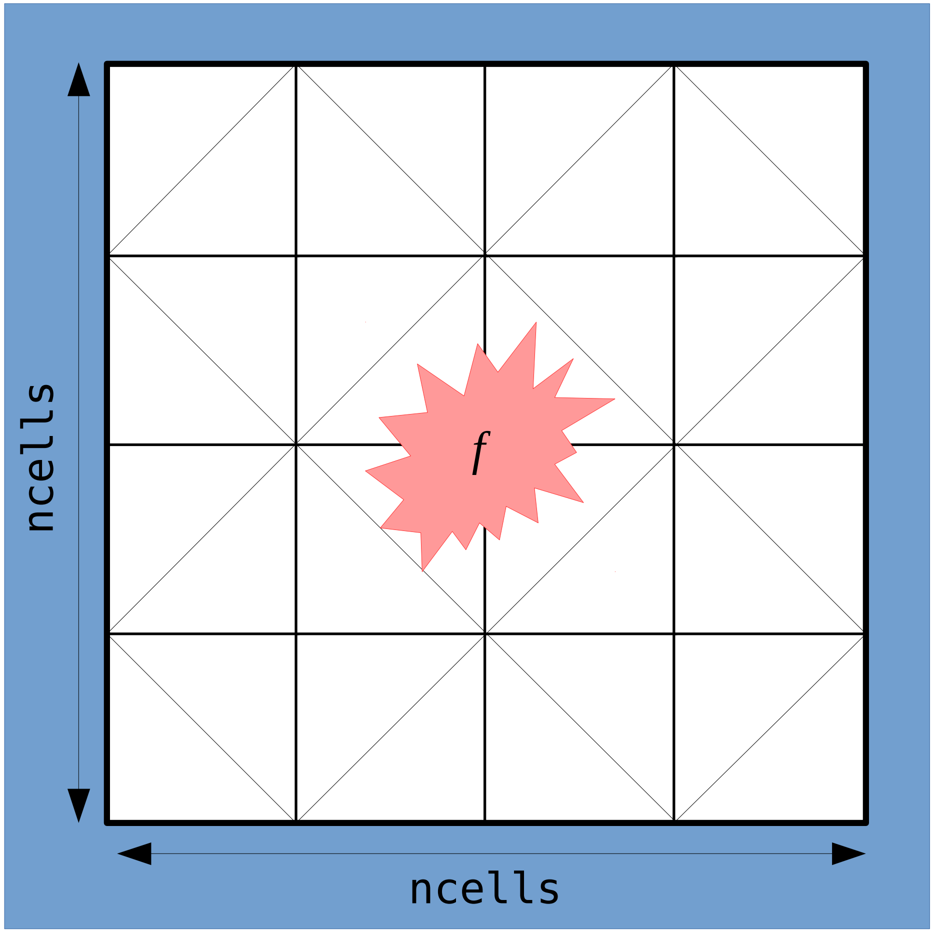
for the temperature, T.

Available parameters:

`k`
diffusivity (k in equation above),
default value = 1.0

`f`
heat source (f in equation above),
default value = 1.0

`ncells`
number of cells used to discretize each dimension of domain,
default value = 32



Command:

```
tfs --test poisson_simple.shml
```

for the default parameters, or:

```
tfs --parameters <name> <value> <name> <value> --test poisson_simple.shml
```

to vary the parameters, e.g.:

```
tfs --parameters f 10.0 k 0.1 --test poisson_simple.shml
```

Instantaneous Heat Diffusion

How does the maximum and integral of the temperature vary with `f` and `k`? Could we have written our equations better?

How do they vary with `ncells`?

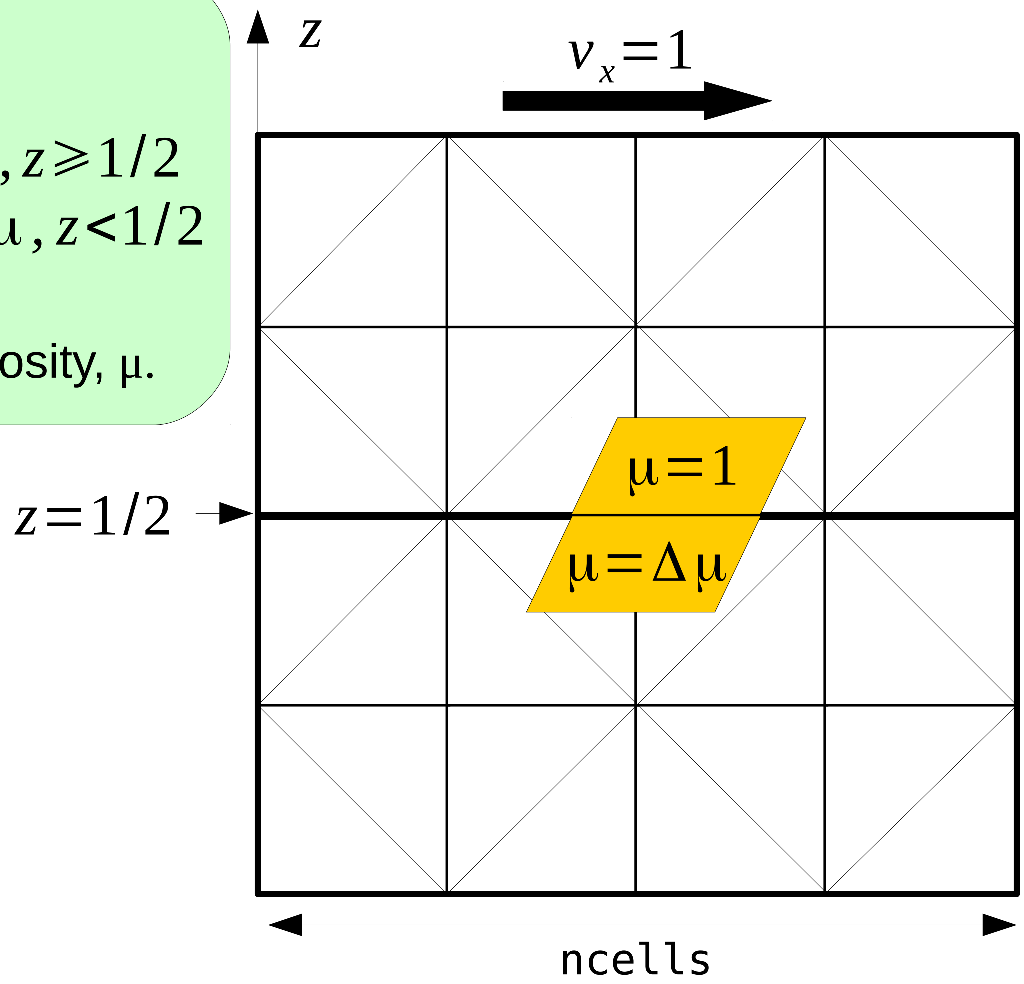
Driven Lid Convection

Solve:

$$-\nabla\cdot\left(2\mu\left(\frac{\nabla v+\nabla v^T}{2}\right)\right)+\nabla p=0 \qquad \mu=\begin{matrix}1, z\geqslant 1/2\\ \Delta\mu, z<1/2\end{matrix}$$

$$\nabla\cdot v=0$$

for the velocity, v, and pressure, p, given viscosity, μ.



Available parameters:

`dmu`
viscosity jump ($\Delta\mu$ in equation above),
default value = 1.0

`ncells`
number of cells used to discretize each
dimension of domain,
default value = 32

Command:

```
tfs [--parameters <name> <value>] --test driven_cavity_freeslip.shml
```

for free-slip sides, or:

```
tfs [--parameters <name> <value>] --test driven_cavity_noslip.shml
```

for no-slip sides.

Driven Lid Convection

What is the effect of varying `dmu`?

What is the effect of switching boundary conditions (different input files)?

How does RMS velocity vary with `ncells`?

★ Thermal Subduction Zone Model

Solve (for 20 Myr):

$$-\nabla\cdot\left(2\mu\left(\frac{\nabla v+\nabla v^T}{2}\right)\right)+\nabla p=0$$

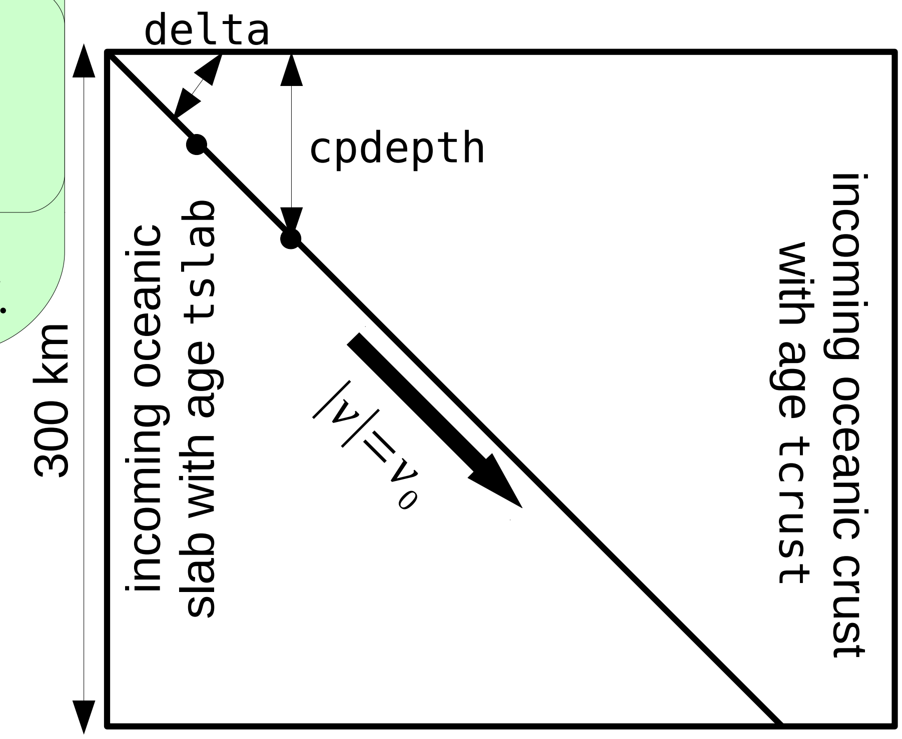
$$\nabla\cdot v=0$$

$$\frac{\partial T}{\partial t}+v\cdot\nabla T-\frac{1}{Pe}\nabla^2 T=0$$

for the velocity, v, and pressure, p, & temperature, T.

$$\mu=\left(\frac{1}{\mu_{diff}}+\frac{1}{\mu_{max}}\right)^{-1},\ \mu_{diff}=\exp\left(\frac{Q}{RT_0}\frac{(T_0-T_s)(1-T)}{T_s+(T_0-T_s)T}\right)$$

$$Pe=\frac{hv_0}{\kappa}$$



Available parameters:

- `delta`
  - slab dip, default value = 45º
- `tslab`
  - age of slab, default value = 50 Myr
- `tcrust`
  - age of over-riding crust, default value = 50 Myr
- `vslab`
  - convergence rate (v_0 above), default value = 0.05 m/yr
- `cpdepth`
  - full mechanical coupling depth, default value = 80 km

Also available:

- `mindx`
  - resolution in wedge corner, default value = 3 km

Command (default parameters take ~5-10 minutes on my laptop):

```
tfs [--parameters <name> <value>] --test subduction_solid.shml
```

Thermal Subduction Zone Model

Each group should vary one of:

```
delta
tslab
tcrust
vslab
cpdepth
```

What is the effect of varying each on the slab temperature at 100km depth and the maximum temperature in the wedge above that point? (These values are printed out to the terminal.)

Compile results in spreadsheet: http://tinyurl.com/ycoeqm4p
(One tab per parameter.)

NOTE: simulations will take different amounts of time depending on parameters chosen, CPU speed etc. but you should expect to wait at least 5-10 minutes for each run.

Buoyancy Driven Wedge Fluid Flow

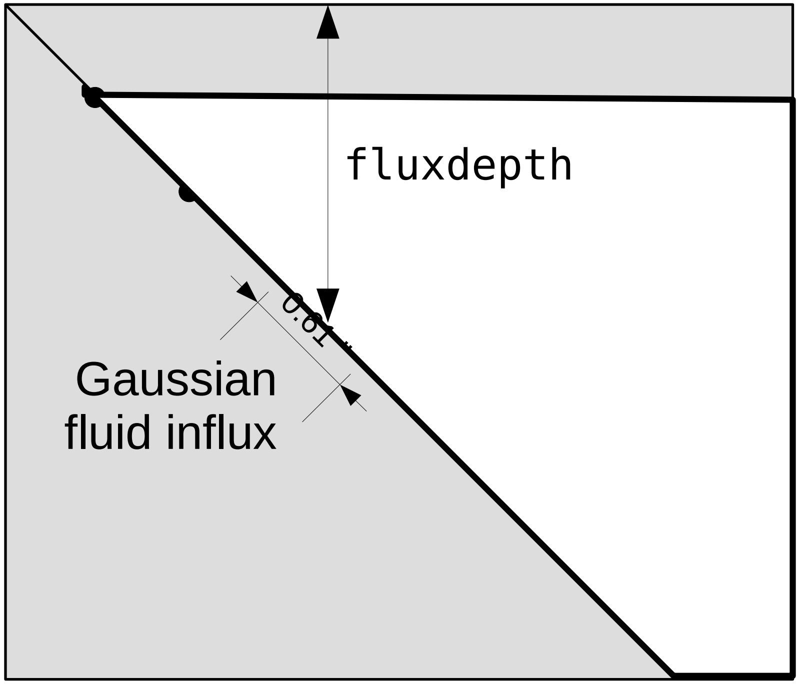
Given solid velocity, v, solve:

$$\frac{\partial\phi}{\partial t}+v\cdot\nabla\phi+\nabla\cdot(Kk)-\nabla\cdot(\kappa_\phi\nabla\phi)=0$$

artificial diffusion

$$K=\phi^n$$

for the porosity, ϕ. given permeability, K.



Available parameters:

all the parameters from the thermal model plus...

`fluxdepth`

depth of fluid release from slab,

default value = 150 km

`w0onv0`

reference fluid velocity to slab velocity (non-dimensional parameter incorporating effects of grain size, fluid viscosity, source magnitude),

default value = 50

Command (default parameters take ~2 minutes on my laptop):

```
tfs --parameters <name> <value> --test subduction_fluidbuoyancy.shml
```

include the parameters you used for the thermal calculation as well as those for the fluid simulation or else a new thermal simulation will be automatically run as well (taking longer!)

Compaction (& Buoyancy) Driven Fluid Flow

Given solid velocity, v, and temperature, T, solve:

$$\frac{\partial \phi}{\partial t} + v \cdot \nabla \phi - \frac{h^2}{\delta^2} \frac{P}{\tilde{\xi}} = 0$$

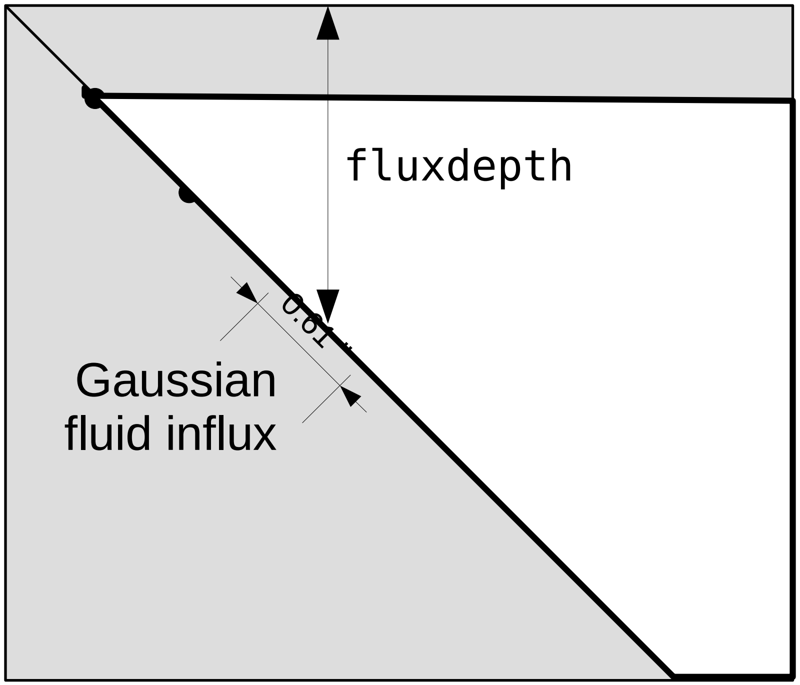
$$-\nabla \cdot \left(\tilde{K} \nabla P \right) + \frac{h^2}{\delta^2} \frac{P}{\tilde{\xi}} + \nabla \cdot \left(K k \right) = 0$$

$$\tilde{K} = \left(\phi + \phi_\epsilon \right)^n$$

regularization

$$\tilde{\xi} = \mu / \left(\phi + \phi_\epsilon \right)$$

for the porosity, ϕ. and pore/compaction pressure, P.



Available parameters:

all the parameters from the thermal model plus...

`fluxdepth`

depth of fluid release from slab,
default value = 150 km

`w0onv0`

reference fluid velocity to slab velocity (non-dimensional parameter incorporating effects of grain size, fluid viscosity, source magnitude), sets compaction length, δ, above,
default value = 50

Command (default parameters take ~8 minutes on my laptop):

```
tfs --parameters <name> <value> --test subduction_fluidcompaction.shml
```

include the parameters you used for the thermal calculation as well as those for the fluid simulation or else a new thermal simulation will be automatically run as well (taking longer!)

Compaction/Buoyancy Driven Fluid Flow

What is the effect of varying `w0onv0`?

What is the effect of adding compaction effects to flow?

Thermal Subduction Zone Benchmarks

Solve various combinations of:

$$-\nabla\cdot\left(2\mu\left(\frac{\nabla v+\nabla v^T}{2}\right)\right)+\nabla p=0$$

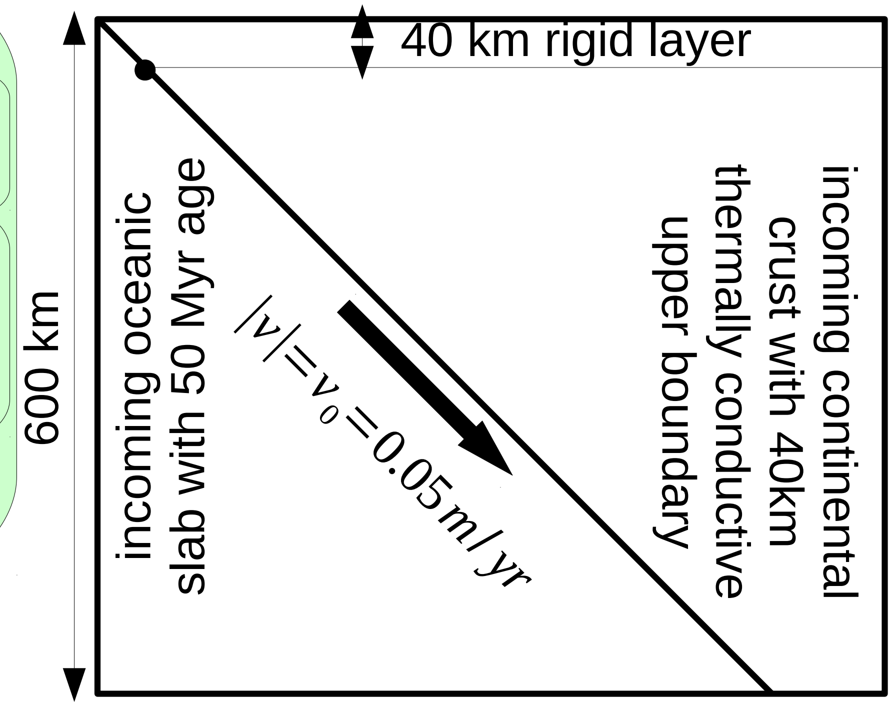
$$\nabla\cdot v=0$$

$$v\cdot\nabla T-\frac{1}{Pe}\nabla^2 T=0$$

various μ

$$Pe=\frac{h v_0}{\kappa}$$

for the velocity, v, and pressure, p, & temperature, T.



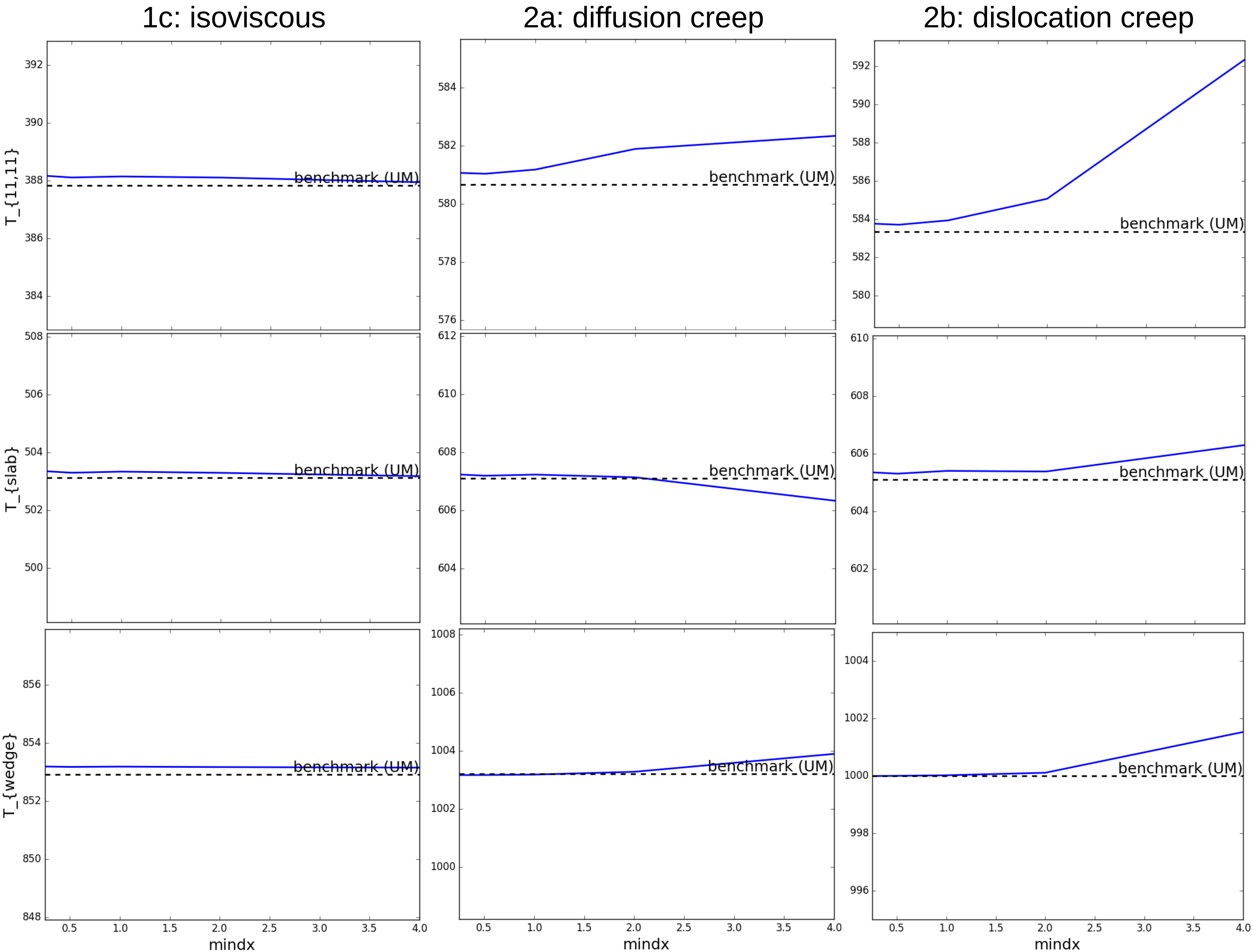
Available parameters:

`mindx`

resolution in wedge corner, default value = 3 km

Commands:

```
tfs [--parameters mindx <value>] --test subduction_benchmark1a.shml
tfs [--parameters mindx <value>] --test subduction_benchmark1b.shml
tfs [--parameters mindx <value>] --test subduction_benchmark1c.shml
tfs [--parameters mindx <value>] --test subduction_benchmark2a.shml
tfs [--parameters mindx <value>] --test subduction_benchmark2b.shml
```


1c: isoviscous
2a: diffusion creep
2b: dislocation creep
T_{11,11}
T_{slab}
T_{wedge}
benchmark (UM)
mindx