

Tutorial 2-Velocity Boundary Conditions

Tutorial 2: Velocity Boundary Conditions

Problem

This example solves for thermal convection with velocity boundary conditions imposed on the top surface within a given region of a sphere. This requires using the regional version of CitcomS.py.

Solution

You will use `cookbook2.cfg`. The parameters specify the number of time steps (61), the prefix of the output filenames (`cookbook2`), and how often outputs will be created (30):

```
steps = 61
    monitoringFrequency = 30
    datafile = cookbook2
```

The solver.mesher facility has several properties involved in the generation of the computational mesh. Continue to use the default values for the physical portion of the domain in which you are interested. However, try modifying the layout of the mesh as shown:

```
nprocx = 2
    nprocy = 2
    nodex = 17
    nodey = 17
    nodez = 9
```

The solver.bc facility allows you to impose a uniform velocity across the top surface. You have a velocity which is purely in the colatitude direction with a non-dimensional velocity of 100 (see Equation 1.6 in the CitcomS User Manual for how to dimensionalize it):

```
topvbc = 1
    topvbxval = 100
```

TUTORIAL 2-VELOCITY BOUNDARY CONDITIONS

In addition, the initial temperature field has a linear conductive profile. The amplitude of initial temperature perturbation is set to zero using the solver.ic facility

```
num_perturbations = 1
    perturbmag      = 0.0
```

Example: Velocity Boundary Conditions, cookbook2.cfg:

```
[CitcomS]
    steps = 61                ; number of time steps

[CitcomS.controller]
    monitoringFrequency = 30    ; how often outputs are created

[CitcomS.solver]
    datafile = cookbook2       ; prefix of output filenames

# Modify the layout of the mesh.
[CitcomS.solver.mesher]
    nprocx = 2
    nprocy = 2
    nodex  = 17
    nodey  = 17
    nodez  = 9

# Impose a uniform velocity across the top surface.
[CitcomS.solver.bc]
    topvbc      = 1
    topvbxval = 100

# Modify the layout of the mesh.
[CitcomS.solver.mesher]
    nprocx = 2
    nprocy = 2
    nodex  = 17
    nodey  = 17
    nodez  = 9

# Impose a uniform velocity across the top surface.
[CitcomS.solver.bc]
    topvbc      = 1
    topvbxval = 100
```

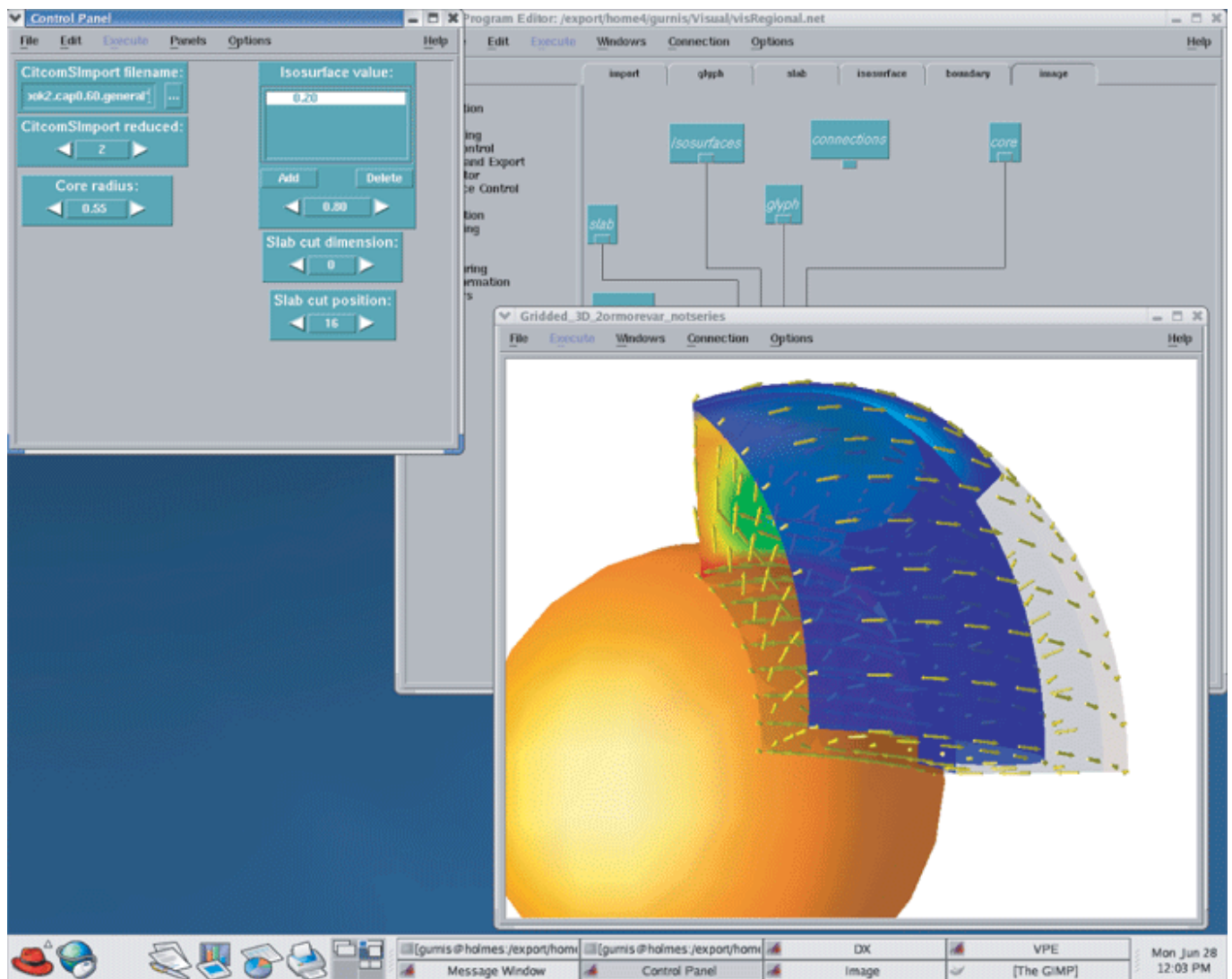
TUTORIAL 2-VELOCITY BOUNDARY CONDITIONS

```
topvbyval = 0
```

```
# In addition, set the initial temperature perturbation to zero.  
[CitcomS.solver.ic]  
num_perturbations = 1  
perturbmag = 0.0
```

Discussion

Using OpenDX to visualize the results (Figure 3 below), this model allows you to create a plate-driven convection in which there is a thermal upwelling on one wall, a thermal downwelling on another, and uniform horizontal velocity across the top. The downwelling is not exactly subduction because the default boundary conditions are close to zero shear stress on the boundaries. This means that there is a symmetrical downwelling in a vertical domain on the other side.



TUTORIAL 2-VELOCITY BOUNDARY CONDITIONS

More tutorials are found in Chapter 6 of the CitcomS User Manual.