Elmer

Manually running and editing ElmerSolver cases

Peter Råback
ElmerTeam
CSC – IT Center for Science

Running an existing case

- Elmer case is located in the working directory where you saved the ElmerGUI project
- To run the case manually go to the projector directory and say
  `>ElmerSolver`
- In most environments (in serial) you may use the command file name as an argument
  `>ElmerSolver case.sif`

- Running the case like this is exactly the same as running the ElmerSolver via ElmerGUI

Manually editing the command files

- Only the most important solvers and features are supported by the ElmerGUI
- Minor modifications are most easily done by manual manipulation of the **sif** file
- The tutorials, test cases and documentation all include usable sif file pieces
- Use your favorite text editor (emacs, notepad++,…) and copy-paste new definitions to your .sif file
- If your additiones were sensible you can rerun your case
- Note: you cannot read in the changes made in the .sif file

Using updated mesh files for your case

- The command files refers to the **body** and **boundary** numbering of the mesh files only
  - ”logical mesh”
  - If these remain intact there is no need to modify the command file
- If the mesh defintion process is deterministic in such a way that the numbering stays constant the computations may be easily repeated with different meshes

Using tests as a starting point

- There are ~300 consistancy tests that come with the Elmer distribution
  - The hope is to minimize the propability of new bugs
- The tests are small for speedy computation
- Step-by-step instructions
  1. Go to tests at **`$ELMER_HOME/tests`**
  2. Choose a test case relevant to you (by name, or by grep)
     - Look in Models manual for good search strings
  3. Copy the tests to your working directory
  4. Edit the sif file
     - Activate the output writing: **`Post File`**
     - Make the solver more verbose: **`Max Output Level`**
  5. Run the case (see Makefile for the procedure)
     - Often just: **`ElmerSolver`**
  6. Open the result file to see what you got
  7. Modify the case and rerun etc.

Adding a new solver to an existing sif

- As a starting point we assume a workable sif file
- From models manual look for the solver of interest
- Make desired modifications
  - Add the solver section manually
  - Add the solver to the active solver list
  - Add the materials parameters, if any
  - Add the body forces, if any
  - Add the boundary conditions, if any
- Worth noting
  - New keywords are not always in the SOLVER.KEYWORDS database
    Therefore often a type must be provided for the keyword values
  - Pay attention to the order of the solvers
    - Use ”Exec Solver” if needed i.e. ”Exec Solver = after timestep”
  - If you add new physical equations check the iteration sequences