

Changelog

All notable changes to PSD solver will be documented in this file.

Rolling Release (2.2)

Added

Changed

- Moved to GitLab for hosting the repository.

Removed

2.1 - 2021-01-27

Added

- New accurate force calculations via matrix-vector product: new flag `-getreactionforce`.
- New flag `-reactionforce` variational-based | stress-based to get reaction force on a surface.
- New flag `-plotreactionforce` to activate real time pipe plotting using GnuPlot.
- More verbos info on `-help`
- New flag `-mesh` to provide the name of mesh to `PSD_Solve`.
- New flag `-probe` to postprocess FE variables at a point.
- New flag `-crackdirichletcondition` to include a pre-cracked Dirichlet in damage models.
- New tests for more advance top-ii-vol partitioning.
- New flag `-constrainHPF` to enable constrain conditions in hybrid phase-field (WIP).
- New developments in parallel interpolations.
- Tutorials added, use `make tutorials` to install

Changed

- Moved to FreeFEM 4.7-1.
- Moved to PETSc 3.14.
- New version of top-ii-vol v 1.3 support for exascale computing (includes new 2D 3D partitioning)
- `-fastmethod` now replaced by `-withmaterialtensor` (this is now inverse of `-fastmethod`)

Removed

- Flag `-pipegnu` not supported for damage mechanics (to be further deprecated from `elsto/soildynamics`)

Bugs removed

- Stain vector inccorect numbering in split fuction of GFP

(see hash [5ec7b882494f71984d07f468b518ec886e942d32](#))

- Hybrid phase-field with constrain with wrong update

(see has [20ebfbc3cc58b9f1407658543bf3b239e74bd089](#))

2.0 - 2020-08-18

Added

- New processing via C++, `PSD_PreProcess` binary (MAJOR CHANGE).
- New solving via shell wrapper `PSD_Solve` instead of `FreeFem++` or `FreeFem++-mpi`.
- New examples of using the solver.
- New Pdf documentation containing tutorials, function definitions, verbos on PSD C++ library.
- New option `-nonlinearmethod` (Picard|Newton-Raphsons).
- New time discretization option `-timediscretization`.
- New point boundary conditions.
- New dummy city mesh and analysis 2D for soil dynamics.
- Automatic identification of FreeFEM and Gmsh during `./configure`.
- New flags for `--with-FreeFEM=` and `--with-Gmsh=` during `./configure`.
- New flag `-bodyforceconditions` [int] to include body force.
- New flag `-problem` linear-elasticity | damage | elastodynamics | soildynamics to define physics.
- New flag `-model` to set approximation for damage mechanics hybrid-phase-field | Mazar | pseudo-nonlinear | Hujeux.
- Better energy splitting included Hybrid phase-field compressibility vs tensile energy condition.
- Introduce boundary conditions via `-dirichletconditions` [int] flag.
- Introduce point boundary conditions via `-dirichletpointcondition` [int] flag.
- Introduce traction boundary conditions via `-tractionconditions` [int] flag.
- New folder `tests` containing unit-tests for modules.
- New Hujeux law (nonlinear soil law) interface using C++ class (Thanks to Evelyn Foerster).
- New pseudo-nonlinear model for solving elastodynamics and soildynamics with nonlinear Newton-Raphsons.
- Introduced double couple earthquake source boundary condition for soildynamics.
- New flag `-doublecouple` force-based | displacement-based to use double couple source for soildynamics.
- New top-ii-vol parallel meshing via `-top2vol-meshing` flag (compatible with soildynamics).

Changed

- Moved to FreeFEM 4.6.
- Moved to PETSc 13.13.
- Moved to C++ for preprocessing.
- Moved to shell wrapper `PSD_Solve` for solving.
- Changes to GFP energydecomposition plugin 'DecompEnergy_Op', now includes compressibility history.
- Replaced GFPDecompEnergy2D/GFPDecompEnergy3D by a generic 2D/3D function `GFPsplitEnergy(Eps1[],PsiPlus[],PsiMin[])`.
- Postprocessing flag `-postprocess` options now support `u` , `v` , `a` , `uv` , `ua` , `av` or `uav`.

Removed

- No more `FFINSTALLDIR` and `GMSH` variables during `make` and `make check`.
- No more `-plot` flag now handled by `-postprocess`.
- No more `-nonlinear` flag now handled by `-problem` and `-model`.
- No more `-bodyforce` flag now handled by an int valued `-bodyforceconditions`.
- No more `-dynamic` flag now handled by `-problem` and `-model`.
- No more `-soildynamic` flag now handled by `-problem` and `-model`.
- No more `-quasistatic` flag now handled by `-problem` and `-model`.
- No more `-dirichletbc` flag now handled by `-dirichletconditions`.

Bugs removed

- Bug in 3D paraxial loading (see hash [8fcbe7390e526423cd24b5f0ab1c06899b36c67f](#))

1.8 - 2020-01-21

Added

- New soil dynamic module `-soildynamics`
- New paraxial element support in 2D.
- New timeplotting support `-timepvd`
- New `-postprocess` option for postprocessing `u` , `v` , `a` , or `uav`.

Changed

- Moved to FreeFEM 4.4.2.
- Moved to PETSc 3.12.
- New simpler way of plotting `savevtk` in parallel with `append` flag for iterative solutions.
- VTU files get stored with a date and time stamp.
- New way of maintaining a logfile for all simulations (date,time,case,..) in `simulation-log.csv`.

1.7 - 2019-11-08

Added

- New mesh reordering via Reverse Cuthill-Mackee via `-useRCM`.
- New quasi-static parallel solver (Extension of B.Masseron & G.Rastiello sequential version).
- New GFP plugin for Mazar's damage update for 2D/3D problems `GFPMazarsDamageUpdate(...)`.
- New MPI plotting routine `plotJustMeshMPI()`.
- New option `-fastmethod` to switch back to default variational formulation.
- New make flag for compiling on supercomputer.

Changed

- Changed variational formulation now using $\epsilon(u) : A : \epsilon(v)$.
- Using GFP becomes optional `-useGFP`.
- Better documentation via `.md` and `.html` files.
- Better plotting support for `PlotMPI()`.
- Moved to FreeFEM 4.4.

1.6 - 2019-06-11

Added

- Dynamic linear solver in 2D and 3D parallel/sequential.
- New finite element variable for partition of unity– for fixing integrals.

Changed

- Better documentation via `.md` and `.html` files.
- Correct quadrature order for faster computations.
- Major changes/splitting of `.script` files.

Removed

- Removed the `BoundaryAndSourceConditions.script` merged with `ControlParameters.script`.

Bugs

- Bug in integrals fixed.

1.5 - 2019-05-29

Added

- Dynamic linear solver in 2D and 3D sequential.
- New meshes for dynamics tests `bar-dynamic.msh`.
- Checking modules `make check`.
- Faster sparsity pattern calculations.

Changed

- Better documentation via `.md` and `.html` files.
- Major restructuring of the codes.
- Moved to `automake` for solver installation.
- Mesh building via `make`.

Removed

- Removed the manufactured solution codes.

1.4 - 2019-05-14

Added

- Fully vectorial finite element solver for phase-field `-vectorial`.
- New `-supercomp` for avoiding xterm issues on super computers.
- New `MatViz()` function for matrix sparsity visualization.
- Introduced `GFP` plugin support (Go Fast Plugins).

Changed

- Elastic energy decomposition is now optional `-energydecomp`.
- Force calculation using integrals (Thanks to G.Rastiello).

1.3 - 2019-04-08

Added

- New meshes in 2D/3D `Notched-plate`, `square-crack`, etc.
- New fracture mechanics module.
- New `-nonlinear` flag to activate phase-field model for brittle fracture.
- New `-timelog` for time logging the solver.
- New `-pipegnu` for GNUplot piping.

Changed

- Scripting now performed using `.script` files:
 - `BoundaryAndSourceConditions.script`
 - `LinearFormBuilderAndSolver.script`
 - `Macros.script`
 - `Main.script`
 - `VariationalFormulation.script`
 -
- Move to FreeFEM version 4.0.
- Move to PETSc version 3.11.

1.2 - 2019-03-18

Added

- Support for Gmsh's `.msh` or Medit's `.mesh` meshes in folder `Meshes`.
- Advance to 3D physics.
- New MPI based parallel solver linear elasticity.
- New approach for solver generation via scripting (PhD thesis MA Badri) with `scriptGenerator.edp`.
- Integrated Domain decomposition macro (PhD thesis MA Badri).
- Customized `.vtk` support for ParaView post-processing.
- New point boundary condition macro `pointbc(Real[int], fespace, matrix)`.
- New flags for communicating with the solver: `-dimension`, `-plot`, `-bodyforce`, `-lagrange`, etc.

Changed

- More advance README.MD.
- Sequential solver now merged within scripting via flag `-sequential`.
- Move to FreeFEM version 3.62.
- Moved manufactured solutions to `validation-test` folder.

1.1 - 2019-03-04

Added

- Initial FreeFEM files for sequential linear elasticity in 2D (case of constrained bar).
- More cases of manufactured solution for linear elasticity in 2D.
- Added README.MD for explaining the solver.
- ParaView plotting activated.

Changed

- Moved to Tuleap git hosting from CEA.
- Separate folder of manufactured solutions and the linear elastic solver.
- Move to FreeFEM version 3.61.

1.0 - 2019-02-15

Added

- Initial FreeFEM files Method of manufactured solution for linear elasticity in 2D.

Version git tags

Version	Git tag
1.0	8a8ecb2746b7da792073358c60df33bae647f788
1.1	a667e6085ba1f92f8dd619bd40e18f85c593bc0a
1.2	e48b7b3a30c05ad4c343efa6a17fee386031f437
1.3	39f4324550365849852c5264b8d4535aae05e30d
1.4	f51f678630eb9b2fed355e5cedf976ce8b5fa341
1.5	07293ba09a69d3d6a16278220a0b4a7a9f318f96
1.6	f359dd049fb1ddde376e8ad8e5177c663e430418
1.7	ae9bfec868a70b3d9974d7692bc19f9739ab7dc
1.8	2f26292636c7248133e31ae912ee58113de2ef71
2.0	1e1a4d7f10df30d106b52eba1c5caf69e8bc0f36
2.1	8b9d84f25aedbd684eb0f06cdd4ffbbf9a60a0e2