

# Changelog

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

## Rolling Release (2.1)

### 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.

### Changed

- Moved to FreeFEM 4.7-1.
- Moved to PETSc 3.14.
- New version of top-ii-vol v 1.2 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 Evelyne 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[],PsiMinus[],HistPlus[],HistMinus[],par);.
- 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 13.12.

- 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/3DNotched-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
<a href="#">1.0</a>	8a8ecb2746b7da792073358c60df33bae647f788
<a href="#">1.1</a>	a667e6085ba1f92f8dd619bd40e18f85c593bc0a
<a href="#">1.2</a>	e48b7b3a30c05ad4c343efa6a17fee386031f437
<a href="#">1.3</a>	39f4324550365849852c5264b8d4535aae05e30d
<a href="#">1.4</a>	f51f678630eb9b2fed355e5cedf976ce8b5fa341
<a href="#">1.5</a>	07293ba09a69d3d6a16278220a0b4a7a9f318f96
<a href="#">1.6</a>	f359dd049fb1ddde376e8ad8e5177c663e430418

Version	Git tag
<a href="#">1.7</a>	aee9bfec868a70b3d9974d7692bc19f9739ab7dc
<a href="#">1.8</a>	2f26292636c7248133e31ae912ee58113de2ef71
<a href="#">1.8</a>	6e721edfafa4ebf3aceebce01e292332a27f980a