Changelog

PSD has been maturing and evolving with time, following subsections present the highlights of some key changes made to each PSD version.

Rolling release (2.5)

Added

Changed

Removed

2.4 - 2022-07-20

Added

- SALOME `.med` file support. For conversion on disk is done upon reading a `.med`
- New `load "medio"` enables loading of medio library for PSD
- Function `savemedmesh()` used for saving med file (see unit test `Test_1.edp`)
- Function `loadmedmesh()` used for saving med file (see unit test `Test_2.edp`)
- Function `getMedLabelsOnGroup()` used for getting the label tag for a group (see unit test `Test_2.edp`)
- Additional tests for Elastodynamics, Damage Mechanics.
- Additional validation tests in `tests/validation` folder.
- New argument `-getenergies` works in soildynamics and elastodynamics, can be used to get K.E, E.E, D.E, T.T.
- New flag `-activeplot` to activate Gnuplot plotting via piping, note this replaces the `-piegnu`.
- New flag `-validation` to produce valdiation test case code, for now only accepts `iwan`.
- Support for Dirichlet Point conditions in sequential for fracture mechanics.
- New tests for checking ponint BC.
- New Iwan law for PSD-MFront coupling.
- PSD installation support for MacOS.
- New macro `perfromRCMreordering(meshObject)` to perform mesh level Reverse Cuthill-Mackee algo.
- New `PsdMfrontPrintBehaviourStats()` to probe the mfront file and know the input/output.
- New variables added for `PsdMfrontHandler()` function `mfrontStrainTensor, mfrontStateVariable,`.
- New variables added for `PsdMfrontHandler()` function `mfrontExternalStateVariableNames, mfrontExternalStateVariabl`
- More unit tests for mfront plugin.
- New elasto-plastic von Mises non-linear model using mfornt. Via argument `-problem elasto_plastic` and `-model von_mises`
- New elasto-plastic tutorial.
- Ablity to calculate reaction forces for linear-elasticity model (use `-getreactionforce -reactionforce .....`)
- New automake variable `--with-dependencies=yes` for installing all dependencies (FreeFEM, PETSc, Gmsh, Mfront, MGIS)
- Dependencies can now be installed by PSD.

Changed

- `startProcedure()` and `endProcedure()` macros replace `timerBegin()` and `timerEnd()` macros.
- Moved to FreeFEM 4.10
- Moved to PETSc 3.16.1
- Mfront material handler has a new name: `mfrontElasticityHandler` is now `PsdMfrontHandler`

Removed

- Removed the `-supercomp` argument, no longer needed.
- Removed the `-pipegnu` flag.

Bug

- Bug in `-plotreactionforce` when used with stress-based method has been removed.

2.3 - 2021-09-13

Added

- New argument `-useMfront` now activates the Mfront–PSD coupling.
- More tests for each plugin and physics, now added in their respective folders.
- Support for Mfront and Mgis interface for non-linear material laws
- New plugin `mfront` can now be loaded in PSD.
- `-withmaterialtensor` now uses Quadrature finite elements to build the material tensor.
- New variational formulation for handling Linear mechanics problems, Quadrature point wise material tensor is built.
- `pseudo_nonlinear` implementation of Linear Elasticity now works.
- New `Notes` section added in the source repo, to help with mathematical and algorithmic reasoning.

Bug

- Bug in top-ii-vol meshing due to MPI communication removed.
- Bug in `pseudo_nonlinear` model for Elastodynamics/soildynamics fixed.

2.2 - 2021-07-28

Added

- New and more verbose tutorials on fracture mechanics, soil-dynamics.
- Fast and parallel post processing is now performed using `pvtu` files.
- New PETSc interface in plugins that supports pvtu output.
- Error mechanism to signify wrong PSD flags.

Changed

- Flag values now donot contain hyphens '-' use underscore instead '_', e.g, `linear-elasticity` is now `linear_elasticity`
- 4 CPU procs are now used for `make check`, user can control this by `make check NP=USER_PROCS`.
- Moved to FreeFEM 4.9 and PETSc 3.15.0 .
- Moved to GitLab for hosting the repository.
- New checks for wrong flag. Now if wrong flag or values is entered PSD will give error.
- Boolean flags now also accept value 1|0|yes|no|on|off|true|false for turning on or off.

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 incorrect numbering in split function 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 energydecopostion plugin 'DecompEnergy_Op', now includes compressibility history.
- Replaced GFPDecompEnergy2D/GFPDecompEnergy3D by a generic 2D/3D function GFPSplitEnergy(Eps1[],PsiPlus[],PsiMinu
- 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-filed `-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	39f43245503658498 52c5264b8d4535aae05e30d
1.4	f51f678630eb9b2fed355e5cedf976ce8b5fa341
1.5	07293ba09a69d3d6a16278220a0b4a7a9f318f96
1.6	f359dd049fb1ddde376e8ad8e5177c663e430418
1.7	aee9bfec868a70b3d9974d7692bc19f9739ab7dc
1.8	2f26292636c7248133e31ae912ee58113de2ef71
2.0	1e1a4d7f10df30d106b52eba1c5caf69e8bc0f36
2.1	8b9d84f25aedbd684eb0f06cdd4ffbbf9a60a0e2
2.2	5e0368f990d505d3bf1960122cb99a23e08567b0
2.3	0744b19fbe7da6d523754092e92f3882b57f0760