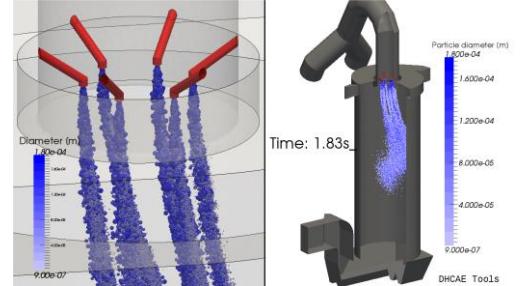


Release notes for CastNet 4.2 and runGui 3.0

Major new features:

The new release of CastNet/runGui:

- allows now to run cases in the cloud directly from runGui under Windows or Linux
- supports OpenFOAM® 3.0
- supports reacting and Lagrangian solvers and
- supports CalculiX much better with a direct Windows executable (shipped with runGui) and a data translator. Therefore, there is no need to install the bconverged version of CalculiX or to work with cgx any more.



OpenFOAM®'s rocket solvers support templates in CastNet: The figures show evaporating particles with strong interactions in a quench

New Features in detail:

- CastNet now supports
 - Lagrangian solvers: reactingParcelFoam, simpleReactingParcelFoam, LTSReactingParcelFOAM
 - Combustion solvers: reactingFoam, LTSReactingFOAM
 - the new version of multiPhaseEulerFOAM.

A number of additional case setup features were included:

- Multiple thermophysical materials
- Improved scalar definition for case setup and boundary conditions
- Chemistry and Combustion-Dicts
- Reacting particle setup

All new solvers are shipped with templates for an easier case setup.

- CastNet now supports OpenFOAM® 3.*
 - Including: new features such as "consistent-SIMPLE"

Small improvements, bug fixes and adaptions for OpenFOAM® keyword-changes:

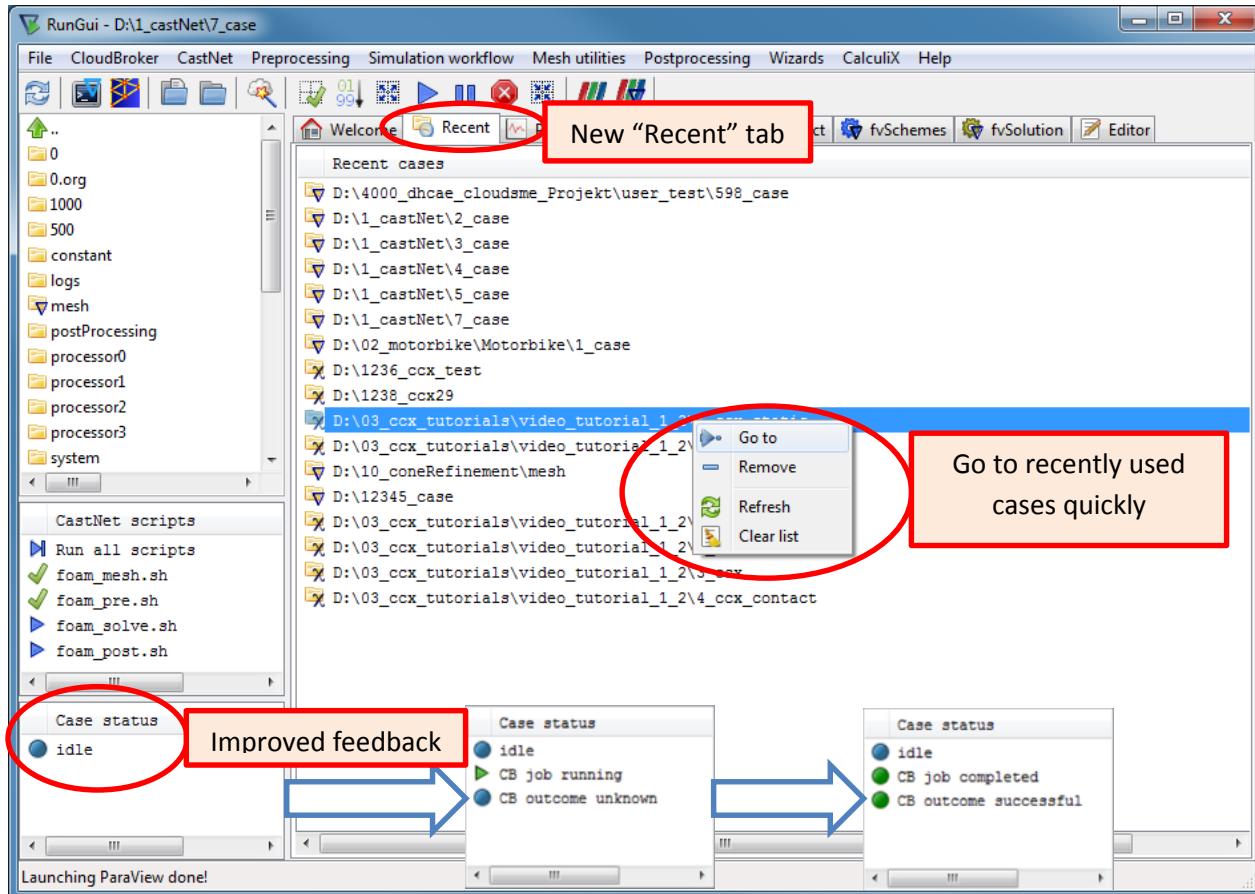
CastNet:

- Name corrections for VOF
- Time-dependent BCs
- snappyHexMesh boundary layer definition
- LES-case setup corrections
- Turbulence modelling update for OpenFOAM® 3.*
- Better support of Abaqus

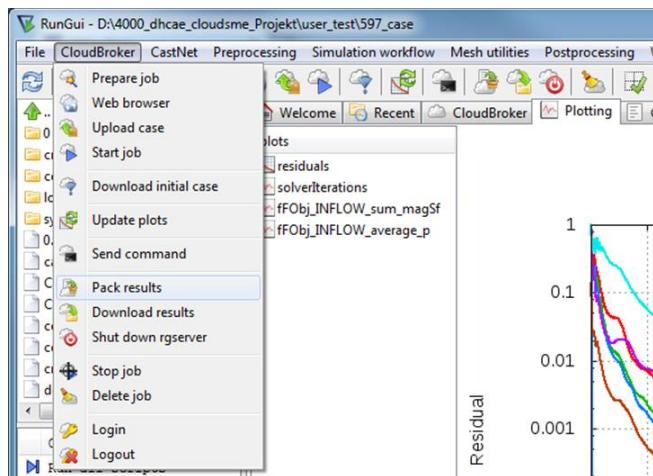
runGui:

- runGui now supports all OpenFOAM® versions up to OpenFOAM® 3.*
- patchAverage and patchIntegrate both show parallel and serial results now.
- Toolbar now changes the buttons, depending on the current case directory and its content.
- Improved settings for the rho-optimizer: set maxIter to zero for hEqn and TEqn to prevent solver crash in the initial stages of convergence.
- File → Clone case now changes to the cloned case directory and refreshes the GUI.
- The plotting functionality now handles new OpenFOAM® syntax for probes, forces etc.
- twoPhaseEulerFoam is now supported by the plotting functionality.
- Mesh Utilities → Extrude mesh now works more intuitively.

New runGui features



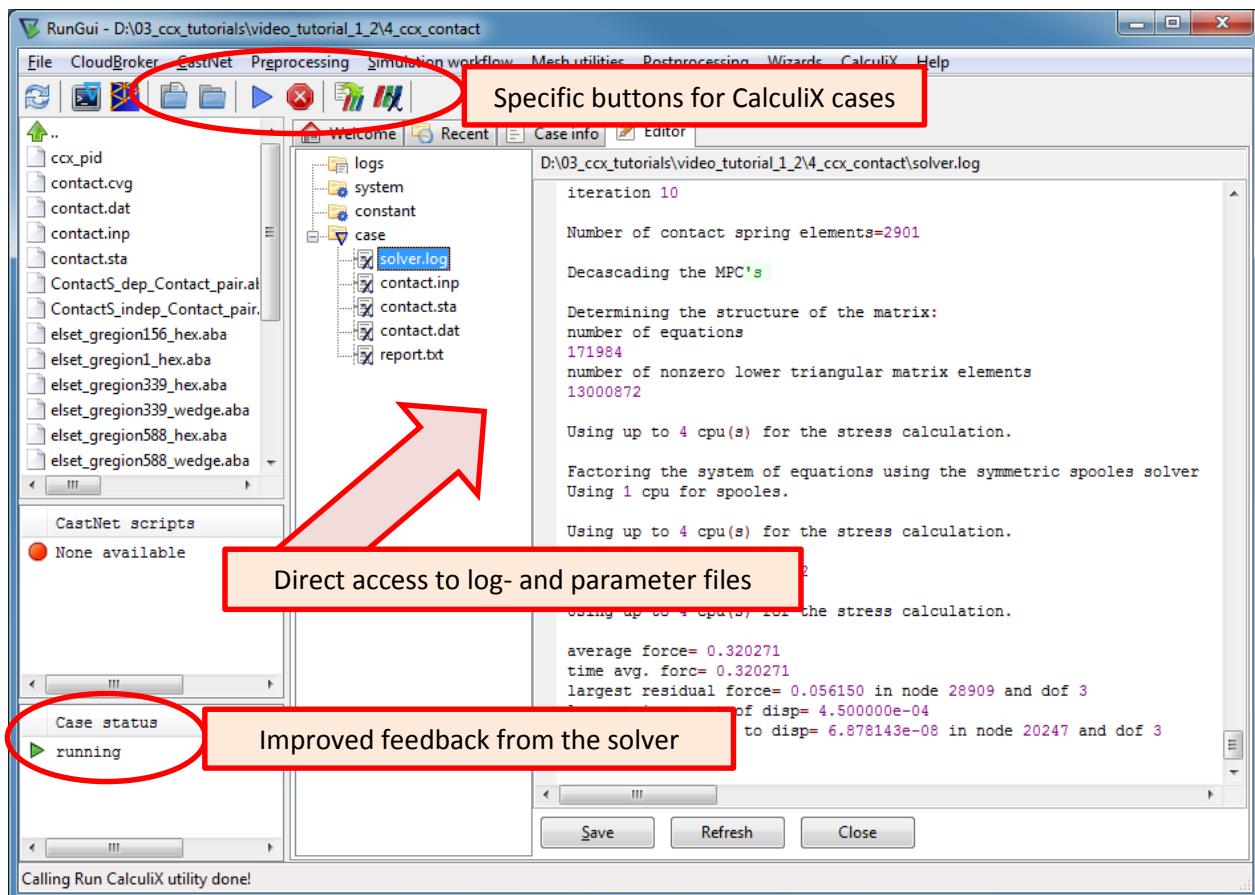
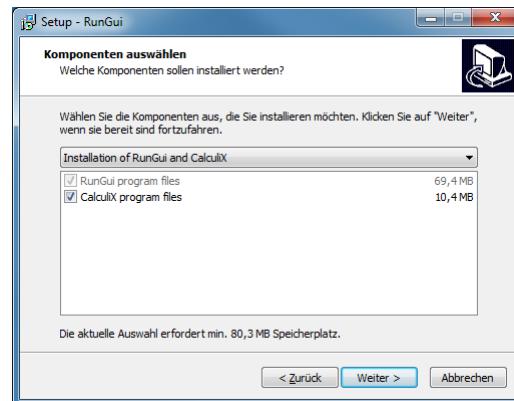
- A new “Recent” tab was introduced. You can jump to your recently used case directories quickly.
- A new “Case status” section was introduced. Here you can find information regarding the state of your current case.



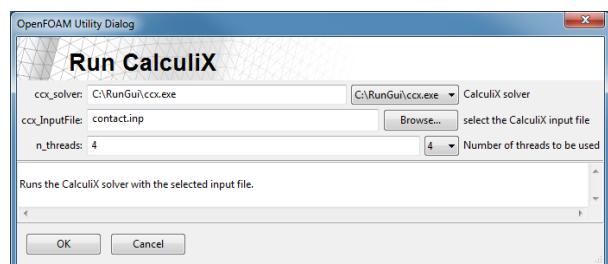
- Optional usage of Cloud HPC resources via the CloudBroker Platform was added.
- OpenFOAM® 2.3.1 and CalculiX cases can be run in the cloud.
- Various HPC Cloud providers are available.

Improved runGui support for CalculiX

- The runGui installer for Windows provides optional installation of the CalculiX executable.

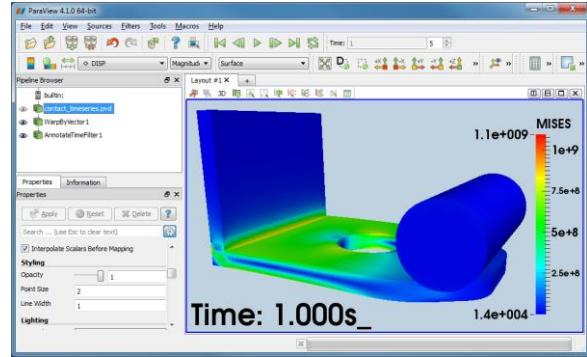
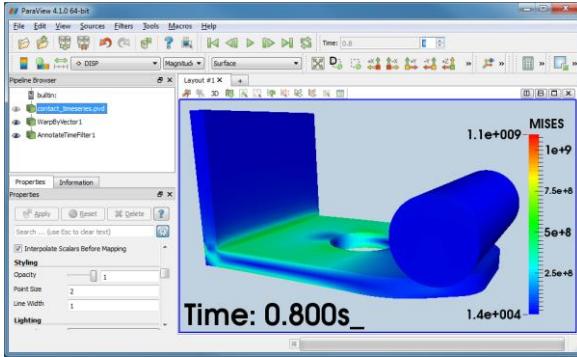
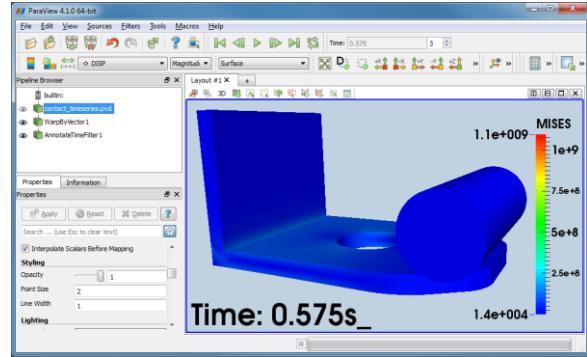
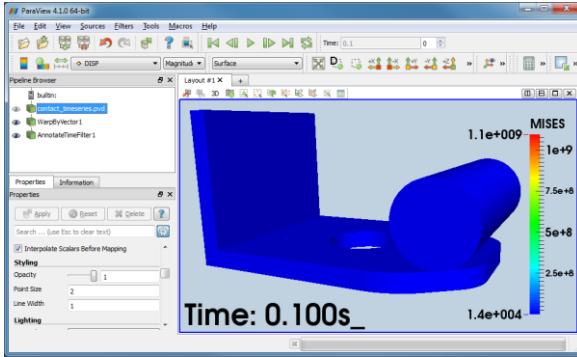
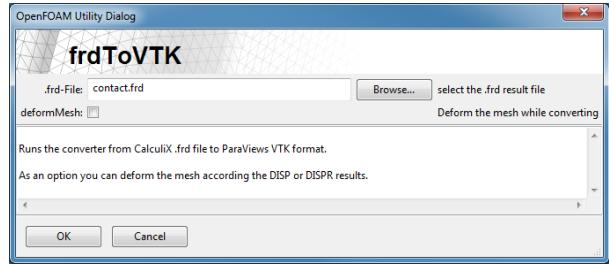


- The CalculiX solver can be started from runGui directly.
- The number of CPU cores to be used can be defined easily.



New CalculiX to ParaView-VTK converter

- A new converter for CalculiX results is available in runGui.
- The frdToVTK converter translates the CalculiX results to ParaView's VTK format.
- The time steps are available in ParaView.
- Optional deformation of the mesh according to DISP results is available.



- Patches with boundary conditions and loadings are available in ParaView and can be extracted with the "Extract Block" filter.

