FEBio 2.3. Release Notes

FEBio 2.3 - Release Notes

=================================

- The pressure boundary load now implements a non-symmetric stiffness matrix in addition to the symmetric formulation. Although the symmetric formulation often results in good convergence behavior, it is not consistent with the pressure residual unless the boundary of the pressure load surface is constrained. It this latter condition is not satisfied, the consistent non-symmetric formulation can result in better convergence behavior. To use the non-symmetric formulation, add the following to the pressure load definition.

<symmetric\_stiffness>1</symmetric\_stiffness>

- Elements sets, surface, and node sets can now be defined in the input file. These can be used to define boundary conditions, loads, and contact definitions more concisely. See sections 3.8.4 of the FEBio user's manual and following for an in-depth discussion.

- A new Parameters section was added to FEBio file format, where users can define parameters that can be used in most variable definitions. Parameters are defined by name/value pairs in the Parameters section.

<Parameters>  
 <param name="var1">1.0</param>  
</Parameter>

Then, parameters can be used as the values of most xml tags by preceding the parameter name with the '@' symbol.

<material id="1" type="neo-Hookean">  
 <E>@var</E>  
</material>

- Added new Include section to the FEBio file format, which will include another .feb file verbatim. This can be used (in addition to the "from" attribute) to split the model definition across multiple files.

- A new reaction rate was implemented where the reaction rate is proportional to exponential of strain energy density.

- A loadcurve can now be defined for vector parameters. The loadcurve will scale the vector.

- Improved contact detection for several contact implementations (sliding\_with\_gaps, facet-to-facet sliding, tied, facet-to-facet tied, sticky). Certain special cases in the contact projection are now handled correctly. This results in fewer contact detection failures which could cause stability issues in some contact problems.

- A volume constraint is added that can be used to preserve the volume of an enclosed space. This could be used to model the containment of an incompressible fluid inside a solid.

- A point-to-point distance constraint was implemented which enforces a constant distance between two points.

- Conewise linear elastic materials were implemented.

- Support was added for node relocation on initial contact for all contact interfaces. This feature forces nodes that have an initial penetration to lie on the contact surface. To use this feature add <node\_reloc>1</node\_reloc> to the contact definition.

- The augmentation flag of most contact interfaces now accepts a loadcurve attribute which can be used to control when augmentations are performed. This can be used for instance to enforce augmentation only on the last time step.

- Output of negative Jacobians can now be controlled via a flag in the control file. Set the <output\_negative\_jacobians> flag to 0 to turn off printing of all the negative jacobians. (By default, FEBio prints all the negative Jacobians). When set to 1 FEBio will only print a single warning when the time step has to restarted due to negative Jacobians.

- FEBio will now retry the time step when NANs are encountered. (Previously, FEBio simply terminated.)

- The sticky interface (a variation of tied which allows for initial separation) now has two new parameters: the <max\_traction> parameter can be used to release the tie when a the normal traction exceeds this value. The <snap> parameter sets an initial distance of penetration before the slave node is tied to the master surface.

- A new output variable was added to output the Euler-Lagrange strain. Add <var type="Lagrange strain"/> to the plotfile section to output the Lagrange strain directly to the plotfile.

- Constraints can now be repeated in multiple steps. This can be useful when a constraint has to persist across multiple steps. Assuming a constraint was defined in some step,

<constraint name="my\_constraint" type="volume">  
 ...  
</constraint>

to repeat the constraint in a subsequent step, simply reference it by name:

<constraint name="my\_constraint"/>

- Several issues with the restart feature were fixed.

- Augmented Lagrangian formulation for incompressibility is fixed.

- A bug was fixed in the export of contact tractions of tied interface.

- Periodic boundary conditions now also output the contact gap and contact traction. (The contact gap is the deviation from periodicity).

- Several issues were fixed with using constraints in multi-step analyses.