

May 14 2018

FRANC3D 7.1.2. fixes the following:

- 1) add temperature setting to CFT (crack face traction) dialog
- 2) modify analysis code solution when temperature set for CFT
- 3) fix Nastran PCH file stress reader (for CFT based on external stress file)
- 4) automatically adjust SIF computation settings if CFT or CFC (crack face contact) turned on
- 5) fix Ansys CFC pressure output macro
- 6) add to Abaqus reader for contact output data (for fretting)
- 7) add support for analysis load substeps in fretting module
- 8) update to crack front merging
- 9) minor bug fixes in SIF display and life integration display
- 10) status update info line added for time-consuming processes

Mar 28 2018

FRANC3D 7.1.1. fixes the following:

- 1) changed some recursive function calls to use heap memory to avoid crashes in MSWindows
- 2) add "small sliding" to Abaqus contact options
- 3) allow user to switch 'master' and 'slave' surfaces for Abaqus constraint and contact connections
- 1) between local and global model portions
- 4) fix the extra connection names for local and global connection
- 5) remove interface when importing meshes of brick elements directly connected to tet elements

Mar 28 2018

FRANC3D 7.1.0.9 version number skipped.

Mar 19 2018

FRANC3D 7.1.0.8 fixes the following:

- 1) add \*visco support for ABAQUS
- 2) fix a Win7 compiler issue for 2D residual CFT
- 3) fix issue with add / delete of CFT
- 4) make sure local crack model load steps are consistent with global

Mar 11 2018

FRANC3D 7.1.0.7 fixes the following:

- 1) fix tie constraint and contact separation in local & global model portions

- 2) fix crash when using ANSYS volume meshing
- 3) fix dload op= type string for ABAQUS
- 4) fix ABAQUS odb file crack face tractions
- 5) fix crash in SIFs For All Fronts when analysis results are not present
- 6) fix crack growth when user turns off growth for a crack front
- 7) set better initial guess for nonlinear fretting data fitting
- 8) fix NASGRO4 xml input
- 9) fix documentation file names
- 10) check sign of equivalent K as a function of square root of all three modes
- 11) fix anisotropic toughness GUI entry

Feb 23 2018

FRANC3D 7.1.0.6 fixes the following:

- 1) ANSYS reader modified so load steps are not accidentally deleted from the global portion
- 2) SIF computation parameters no longer cleared when doing Static Analyses
- 3) additional numerical overflow checks in fatigue module
- 4) kink angle set to zero if crack extension is zero
- 5) check for consistent temperature dependent load schedule and growth rate model
- 6) fix GUI for NASGRO temperature dependent data entry

Feb 9 2018

FRANC3D 7.1.0.5 fixes:

- 1) searchTree modified to help identify points on the model surface - fixes some issues with crack growth
- 2) modified ABAQUS .fil and .dtp results reader to process more contact surface data
- 3) fretting module updates: (i) ABAQUS model/results, (ii) contour label set to fretting model,
- 4) and (iii) surface names displayed instead of "master" and "slave"
- 5) temperature dependent NASGRO data entry fixed
- 6) spectrum integration bug fix for multiple spectra
- 7) fatigue schedule load step multiplier read/write fixed

Jan 30 2018

FRANC3D 7.1.0.4 fixes:

- 1) fix crack face traction when using local + global models with no other loads
- 2) for Abaqus "\*Include" INP= added; and "ENDSTEP" without space supported
- 3) material id and coordinate system id used to identify regions and boundaries
- 4) set defaults for substep and growth type in Fatigue dialog if unset

Jan 22 2018

FRANC3D 7.1.0.3 fixes:

- 1) a bug in fatigue life computation when using spectrum loading
- 2) allow multiple \*ModelChange lines for Abaqus
- 3) allow for nset and elset with and without generate flag for same set for Abaqus
- 4) revised MSWindows startup folder
- 5) add tolerance when checking ends of extrapolated crack front points to make sure they are outside
- 6) add flag to Preferences to control amount of Ansys results output to the .dtp file
- 7) add flag to Preferences to increase button and text size for high resolution displays

Jan 9 2018

FRANC3D 7.1.0.2 fixes:

- 1) bug in the kink angle computation when using fatigue load schedules
- 2) Abaqus "\*\*Boundary, submodel, step=n" data now output correctly
- 3) Registry settings moved to a new location in the User's home folder

Dec 15 2017

FRANC3D 7.1.0.1 fixes:

- 1) Transient load schedules can be defined with one load step with multiple substeps
- 2) Abaqus results output frequency is enabled in Preferences
- 3) Abaqus absolute exterior tolerance supported for \*submodel command
- 4) Ansys variable surface pressure mapping is fixed
- 5) Crack growth & merging made more robust

Dec 1, 2017

Changes to FRANC3D from Version 7.0.9 to 7.1.0

- 1) Added Display menu, which includes View Response and Create Animation.  
View Response was moved from the Advanced menu.  
Create Animation was developed starting from the stand-alone GrowthSequence program, which is no longer distributed.
- 2) A Capture button was added to the main Window to provide .png or .jpg images of the current model.
- 3) Fatigue crack growth dialogs have been revised to allow time or cycle or combined time+cycle dependent crack growth. Multiple load steps and load substeps are supported.

- 4) View/Edit Growth Parameters button was added to the Fatigue menu to allow a user to edit the fatigue growth model parameters.
- 5) User-defined Python functions for crack growth are now supported. A Read User Extensions menu button was added to the Advanced menu.
- 6) The VCCT (virtual crack closure technique) was added for computing SIFs. This is intended primarily for bi-material interface cracks, which will be supported in the next release (Ver 7.2). VCCT works for isotropic and anisotropic materials.
- 7) Displacement Correlation SIFs for orthotropic materials is now supported.
- 8) A Large Rotations check box was added for the M-integral SIFs for analyses with large rigid body motions, such as rotating gears.
- 9) The Fretting module interface was modified to support multiple load step analyses.
- 10) Documentation and tutorial example files have been updated.

July 2017

Changes to FRANC3D from Version 7.0.8 to 7.0.9

- 11) Fix issue with ANSYS contact material ids
- 12) Allow multiple ANSYS pilot nodes (and associated contact surfaces).
- 13) Fix ABAQUS Python script to get all frames of results (first frame was missing).
- 14) Add support for ABAQUS analytical (revolution and cylinder) surfaces.
- 15) Add support for ABAQUS ROTA dload.
- 16) Add support for "adjust=nset" in ABAQUS contact pair.
- 17) Fix issues processing ANSYS surf154 elements – specifically for elements with missing or condensed nodes.
- 18) Ensure the ANSYS local cracked .cdb file includes the TREF value.
- 19) Fixed some bugs in surface meshing.
- 20) Allow multiple crack front group labels for user-mesh cracks.
- 21) Added check box to turn on/off curvature correction for M-integral SIFs (for some cases with high Mode II and negligible Mode I, the correction produced incorrect Mode II SIFs).
- 22) Ortho-view icon updated in Submodel dialog when "Rubberband" tool is selected.
- 23) Fixed ANSYS contact "keyopt" default values.

Apr 2017

Changes to FRANC3D from Version 7.0.7 to 7.0.8

- 1) Fixed a problem with unit conversion between FE model and crack growth data, when mixed units are used.
- 2) Fatigue cycles are now represented by int64 data type to allow for larger numbers.

Feb 2017

Changes to FRANC3D from Version 7.0.6 to 7.0.7

- 1) Changes to FE readers to check for empty first load cases. This eliminates solve time for an empty load case after crack insertion and thus removes set the resulting SIFs that are all zero.
  - i) NASTRAN load step added for .bdf files without a SUBCASE.
- 2) Changes to convert crack face traction from pressures or shear tractions to equivalent nodal forces, which simplifies the ANSYS interface considerably as SURF154 elements are not needed on the crack faces to apply shear. (See additional documentation.)
- 3) ABAQUS .rpt file reader edited to make somewhat more robust. Report (.rpt) files are used to apply mesh-based stress as crack face traction.
- 4) ABAQUS heat transfer element types read and converted to structural elements.
- 5) ABAQUS element type string in session log and command language interpreter fixed.
- 6) ABAQUS first pass reading of \*Nset and \*Elset data defined after \*Boundary or \*Dload data that references the sets.
- 7) ABAQUS \*Submodel defined within \*Assembly processed.
- 8) ANSYS SURF154 element pressure mapped to solid element faces for local model.
- 9) ANSYS load steps include time and additional boundary condition data passed through.
- 10) WriteCOD dialog in the Advanced menu modified to allow specification of the load step.
- 11) Crack growth dialog for the case of Implicit R fixed so that all user edits to the dialog are collected.
- 12) User-defined crack growth increment table access fixed; index into the table was off by 1.
- 13) Reference temperature set as the default temperature (instead of 0.0) when computing SIFs using M-integral if user turns off the "do thermal" flag.
- 14) A number of bug fixes have been made to the crack insertion and meshing library also.