

FEACrack Version History

Version 3.2.036 – June 2, 2021

Continuing development on FEACrack version 4 (64-bit) GUI; partly working beta version for post-processing, mesh generation, and some Tools menu utilities. Added user-defined geometry definition mesh import and viewing; the selected mesh surface for the crack location is shown with a unique color in the definition mesh view.

Updated Warp3D packets output file to return the element killed status needed to report J-R crack extension results for cell type meshes.

Updated element porosity key word to “killable element status” to read the element porosity results from the Warp3D output file.

Updated the analytic rigid surface normal vector to support the flat plate geometry at the +Z and -Z locations, and to support axial and circumferential cracks in the cylinder geometry; the rigid surface is used in Warp3D to model crack face closure.

Added automation to automatically combine the selected master mesh with the user-defined geometry crack mesh. The master mesh does not need to have the “master mesh keyword” added in the input file before generating the crack mesh; the new automation adds the key words.

Fixed the plastic zone option crack tube radius for SI units using meters with the crack tube radius entered in mm; in the Geometry/Plastic Zone Dimensions form.

Fixed a crack length error check when the user-defined geometry is used with the full-width crack shape. Fixed a mesh flag warning message for the user-defined geometry, since more mesh flags have been added to support new boundary conditions such as load pins.

Added a report text file for the reference stress post-processing; create a unique file name in the same directory with the selected result file by appending “_RefStress.txt” to the selected result file. Activate the report file in the FEA Result form using the “Create an FAD Curve” option.

Increased the maximum size of the array to read the “Node List Data” comment data block to support larger meshes and more pre-defined node sets.

Fixed the maximum node set mesh flag index for geometries other than a flat plate, such as the cylinder when using the tabular temperature input.

Updated the A2 calculator report text file to be written in the same directory as the general log files; refer to the Help/About form for the location of the general log files.

The key hole mesh option supports use of the custom mid side node location using the key word “*use midpoint”; this can be useful to change the default midpoint mid side node location for plasticity analysis to a quarter point location to get a combined $1/r$ plus $1/\sqrt{r}$ singularity

Added “mesh face 1 nodes” to the Node List Data comment block to provide a list of nodes containing the crack ligament and crack front; this node list can be used to quickly select those nodes (in the Select Nodes form) to get a summation of the reaction force in the ligament.

Version 3.2.035 – October 23, 2019

Added information and figures in the user’s manual to use the PRO mesh settings to create the crack mesh to get remote contours to the edge of the model, which can be useful for elastic-plastic analyses with large strain deformations when the J-integral contours near the crack front in the focused mesh region can have path dependent values.

Changed warning messages about small key hole radius values to info messages that are written only to the log file.

Improve the warning message when using the “use view crack face femap files” option with the standard mesh block instead of the PRO mesh.

Updated post processing to read Warp3D version 18.1.8 standard output result file.
Update Warp3D executable to version 18.1.8 in the installer.

Updated a warning message to an info message and to give the sum of the inner plastic zone elements plus outer plastic zone elements for comparison to the total J contours when using the inner plastic zone option; this warning can occur when using more contours than elements in the crack tube region.

Version 4 beta 4 update 7 for 64-bit release (no change to executable or version number), September 4, 2019

In post processing updated the ANSYS library to support reading *.rst result files through the ANSYS 2019 R3 release (i.e. ANSYS version 19.5) when using FEACrack version 4.x.

Added option in the Abaqus input file to request output to the *.odb output database result file; refer to the “*use abaqus odb=yes” key word text in the Home/Notes form.

Version 4 beta supports reading results from the Abaqus *.odb output database result file.

Fixed a problem when writing a large mesh to the Femap neutral file if a previous error has set the number of nodes larger than the current mesh array size.

Version 4 beta 4 update 7 for 64-bit release, April 11, 2019

Updated ANSYS post-processing in version 4 to support version 19.1 and release 2019 R1 (19.3)

Improved the key hole mesh option to support the shifted crack location in a user-defined geometry mesh.

Added an option to omit nodes and elements in the key hole from the crack face, see the “*use keyhole omit” key word syntax in the user’s manual.

Fixed a problem with a small volume user-defined geometry with a shifted crack location causing the final mesh to be smaller than expected; the crack mesh block size is updated to fill the given small volume.

Fixed a problem with the full-width (partial depth) crack shape using the bevel angle in a user-defined geometry.

Fixed a problem when the left crack tip is near the left side of the user-defined geometry volume to set the correct remaining width for the crack mesh in the volume.

Updated the Weibull stress calculator to support reading results from the Warp3D result files: standard output *.out and binary packets file *.bpf

Updated the subfiles with the full file path to the current directory of the main input file, needed to support version 4.

Added “named lists” in the Warp3D input file to contain the crack mesh surface node sets. Can be used to add or modify constraints in the crack mesh, such as holding the crack face closed.

For elastic-plastic analysis, added the coincident node groups at the crack front to the crack face node list, not including the node at the crack front. Allows all nodes inside the crack front to be listed and used for constraints.

Corrected a small problem with the C(T) specimen with the load pin active and imposed Z-displacement values on the load pin center line.

Added an option to include an amplitude table when the crack mesh does not have imposed loads to automatically create an amplitude table for multiple step analyses; this helps support combining a user-defined geometry crack mesh with a master mesh when only tied contact or symmetry constraints are used on the crack mesh; the combined meshes will use the common amplitude table for each analysis step; refer to the “Common Amplitude Table” section in the user’s manual.

Updated post processing to read Warp3D version 17.9.0 standard output result file.

Version 3.2.034 – July 30, 2018

Created the Linux version *.tar file package to run FEACrack v4 on Linux under Mono (7/26/2018 GVT).

Added key word option to use the Gauss integration 9pt_rule or 14pt_rule in Warp3D. Refer to the user’s manual for the “*use gauss ...” key word syntax. When using the 9 or 14 point rules, the output location needs to be at the centroid or nodes.

Fixed an intermittent problem with the crack front tangent direction vector for the full model through-thickness crack shape when used in a user-defined geometry.

Fixed the modified boundary layer model perimeter imposed displacement at $\theta=\pi$ to give correct z-direction displacement.

Changed the default location for most log files to the system “TEMP” directory. Check in Help/About for the current directory. Or from the Windows CMD command line, type “set” to list the environment variables, including

“TEMP” to get the directory path to the log files. The mesh generator log file is still written to the same directory as the *.elt model file. Refer to information in the user’s manual.

Added the Plasticity Interaction Factor calculator form to v4 Tools menu; use to input values to compute the plasticity interaction factor and obtain Kr ratio; input values: K1 primary, K1 secondary, Lr primary, yield strength, crack size, crack location flag, Kc toughness.

Updated an internal array size to support a cylinder right-half symmetry surface crack with the finite radius key hole.

Updated v4 post-processing to support ANSYS version 19 *.rst results file.

Corrected the weld geometry transformation to support the finite radius crack front with the PRO mesh.

Added Warp3D 17.8.7 and user manual; this version of Warp3D no longer has a fixed upper limit on the number of nodes in the mesh to allow very large meshes; maximum number of elements is 3,000,000.

Added support for the custom surface crack shape for use with the key hole and elastic-plastic analysis when using back half symmetry so that the crack mesh primitives are correctly combined.

Added a check for coincident points in the user defined crack point array.

Version 3.2.033 – December 5, 2017

Fixed a problem in the PRO mesh to assign the weld and HAZ mesh zones to the correct elements when the weld root size is small compared to the top weld size.

Corrected a problem when the PRO mesh N_theta number of elements around the half crack tube is 50 or larger.

Improved C(T) specimen crack mesh to support the finite radius crack front option.

Added support for the finite radius crack front using the PRO mesh in the C(T) specimen to assist with Weibull stress versus J and K calculations.

Added a node set along the front right edge of the plate; use to apply custom constraints and loads.

Added the multipoint macro key word for Warp3D: add warp3d multipoint, used to include custom MPC syntax

Improved post processing error messages for combined Warp3D input file with inconsistent output location.

Added an option to shift the mesh coordinates in the x, y and z directions; enter the “*use shift coord...” key word syntax in the Home/Notes form.

Fixed a typo for the corner node ID numbers listed above Figure 173 in section “Definition Mesh Corner Nodes Comment Data Block:

Extended the material omit option to include omitting the *SOLID SECTION in the Abaqus input file; this allows full customization of the material data for the crack mesh, especially for a user-defined geometry crack mesh combined with a master mesh

Added macro text key word “add warp3d step <n>” to insert custom syntax in specific Warp3D analysis steps.

Improved user-defined geometry definition mesh edge length check to set minimum size for merge tolerance when preparing the mesh transformation objects.

Version 3.2.032 – August 16, 2017

Fixed a geometry transformation problem, fixed the internal crack location, and fixed the crack opening area in post processing for the sphere with a hole geometry.

Updated post processing to support Abaqus version 2017

Fixed a problem for larger finite radius meshes to get more accurate crack front node coordinates.

Fixed a problem in the modified boundary layer model (BLM) for larger finite radius crack front, the outer circle coordinates are on the radius centered about the finite radius crack front.

Fixed a problem for very long surface cracks for plasticity analysis to get the correct crack front node set using the key hole build approach; updated the key hole node search tolerance to use the current reduced key hole build radius.

Fixed a problem for custom cracks with deep to shallow profiles in a plasticity analysis using the key hole option.

Fixed a problem for very long surface cracks using the PRO mesh where some nearly coincident nodes outside the crack tube were not getting merged.

Version 3.2.031 – June 6, 2017

Begin adding OpenMP multi-threading (i.e. parallel programming) to help speed up the crack mesh generator; multi-threading is deactivated by default; to activate use the “*use openmp num threads = <n>” key word in Home/Notes; see the user’s manual for details.

Added an option to omit the automatic crack face pressure load; use the “*use omit auto crack pressure” key word in Home/Notes; see the user’s manual for details.

Added an option to omit the automatic axial pressure load; use the “*use omit auto axial pressure” key word in Home/Notes; see the user’s manual for details.

Added Warp3D version 17.8.0 executables and new manual.

Moved the Warp3D executables to the Warp3D_run subdirectory and updated the Run FEA wizard to use the exe files in that directory; this move supports the x86 32-bit DLLs with OpenMP in the install directory, and allows use of the x64 Warp3D exe for analysis.

Added an error check on the maximum number of elements for very long surface cracks to correct a problem.

Corrected a problem with the macro text option for an ANSYS input file.

Added an option to omit the standard material data syntax from the input file. The material data can be customized by replacing the omitted syntax using the master mesh file option or for an Abaqus input file using the macro text option.

Added the millimeter to m length unit conversion needed for fatigue analysis C coefficient for the N-m-Pa unit system. Corrected the normal stress constraint factor unit conversion for the N-m-Pa unit system. The length values are entered in mm in the N-m-Pa unit system for convenience and need to be converted to meter values for the input file.

Corrected the nodal strain result flag to skip reading nodal strain results when not available in the result file.

Corrected the buried crack for fatigue analysis, report correct K at $\phi=270\text{deg}$ results to Fatigue/Results summary table, add error check on constant depth buried crack shape (not yet supported in fatigue, available for single cracks).

V4 beta: added more model inputs, supports opening a model file and building the crack mesh.

Version 3.2.030 – May 13, 2016

Added Warp3D version 17.7.1

V4: updated the Abaqus *.fil post processing to allow for an incomplete set of J-integral results for an analysis step that does not converge; allows all the available output steps to load and provide J results.

Added the crack front nodes as a node sub set to the ANSYS input file; each crack front is written as a separate node subset with a unique “crack_front_nodes_N” label; intended to help work flow using the crack mesh in ANSYS Workbench.

Added option to define the characteristic stress for post processing; in Boundary Conditions activate the “User Defined Load History”, and in the UD Load tab use the label “charstress” for one of the user defined load histories; if another characteristic stress is not found the given charstress value will be used.

April 25, 2016

Added the “definition mesh corner nodes” comment data block option to import a definition mesh when the default import tolerance angle approach is not adequate; the 8 corner nodes are listed in the definition mesh file with the comment data block and used to find the edge nodes.

April 14, 2016

Updated the post processing to get the Abaqus version number from the Abaqus 2016 version *.dat output data file.

April 5, 2016

Modified the crack face pressure load input to be available for multiple analysis steps; the crack face pressure uses the common amplitude table for the same history as the other loads.

Added warning message if step 1 output is deactivated and Abaqus *CONTOUR command is written only to step 1, which results in all J-integral values being zero.

Write the crack mesh surface summary table a second time in the Abaqus input file just before the *TIE information for convenience when determining the tied contact surface pairs.

March 30, 2016

Extended the automatic master mesh combining option for Warp3D and ANSYS; added more “master mesh” key words to use in the master mesh input file; write a list of available master mesh key words to the top of the input file for reference.

Version 3.2.029 – March 14, 2016

Add the automatic master mesh combining option to combine a user-defined geometry crack mesh with the larger master mesh model; key words are added to the master mesh to identify each section of data to be copied to the combined mesh input file during crack mesh generation; the Abaqus input file is supported for this new option, and support for Warp3D and ANSYS should be available soon; refer to the user’s manual for details.

Improved the message from v3.2.x that supports reading ANSYS results through ANSYS 15

February 9, 2016

V4: release beta 3 update 6 2016.0209

Updated v4 with component upgrades: Microsoft .NET 4.6.1, DevExpress 15.2, NUnit 3, Warp3D 17.6.0

February 4, 2016

V4: update the post processing to read ANSYS 17 *.rst result file; since ANSYS 16 and 17 provide a programmable library only for 64-bit applications the FEACrack v4 (x64) supports the post processing.

Fixed file path string conversion calling SRT_Jintegral.dll when total file path is more than 128 characters.

February 1, 2016

Add Warp3D version 17.7.0 executables and user manual.

Version 3.2.028 – January 28, 2016

Expanded the “macro text” option in the Tools/Model Options/Macros tab to add data to Abaqus and Warp3D input files; key words in the macro text identify each section of data to add; especially useful for user-defined geometry meshes where adding data such as constraints or analytic rigid surface load pins allows the model to run as a stand-alone analysis within the Fatigue module; the macro text can still be used to replace the ANSYS “solve” command with model data.

Fixed the analytic rigid surface reference node ID for Abaqus when the mesh is renumbered.

Fixed the single node y-constraint for the flat plate when load pins are used.

Add Warp3D version 17.6.0 executables and user manual.

January 15, 2016

V4: Fixed a problem reading an ANSYS *.CDB file to import a user-defined geometry definition mesh, and added more checking on the file import

August 26, 2015

Version 4 beta 3 update 3 x64, 2015.0826, added error trapping on the crack results form, add an option to choose the x-y-z axes tripod, added dependencies to the installer

August 20, 2015

Corrected a warning message for the material zones in the PRO mesh with a weld and HAZ, particularly for a back-half cylinder with axial surface crack

August 19, 2015

Improved the crack front node sorting index for the custom crack shape (tabular crack point input)

August 13, 2015

Add two PRO mesh options for the finite radius crack front. The inner crack tube length can be set, and the length of the inner mesh box can be set. Refer to the user's manual sections "Inner Crack Tube Length Option" and "Inner Mesh Box Multiplier Option".

Fixed a problem with the Warp3D renumbered mesh when multiple material groups are used.

June 2, 2015

Added the ILNumerics 4.8 update to FEACrack v4.

May 6, 2015

Add the automatic element blocking option for Warp3D, enter keyword text in the Home/Notes form: "*use warp3d auto blocking".

Fixed a problem with the C(T) specimen load pin node constraint selection.

Version 4 beta 2 – April 27, 2015

Installer executable: FEACrack4_Setup.exe, mesh graphics support output at all 3 locations: centroids, Gauss integration points, nodes, additional crack result charts.

Version 3.2.027 – April 14, 2015

Add Warp3D version 17.5.8, the maximum mesh sizes are 1,000,000 nodes and elements.

Fixed the nodal stress and strain result averaging for the Warp3D packets results file.

Fixed a problem with the crack front direction vector for the through-thickness crack in a cylinder with back half or full symmetry.

Version 3.2.026 – January 29, 2015

Update release with cumulative changes.

January 27, 2015

Added a missing automatic node constraint for the cylinder with an axial crack and back-half symmetry.

December 10, 2014

Fixed a problem in the J-integral module with the crack normal vector calculation when crack front is nearly straight; typically used with ANSYS analysis and post processing.

November 21, 2014

Added ANSYS submodel analysis options; add key word options in the Home/Notes form to set the global model file and the global model results step number.

Version 4 beta 1 – September 30, 2014

Post-processing features working in the new graphical user interface (GUI); release for beta use and comments

Version 3.2.025 – June 16, 2014

Fixed a small problem with the crack front direction vector in the post-processor J module used with ANSYS stress and strain results to compute J.

April 7, 2014

The 64-bit Warp3D executables are installed by default in the FEACrack install directory; 32-bit Warp3D executables are available in the “Warp3D_files_32” subfolder if needed for a Windows x86 32-bit computer; see the “Readme_Warp3D_32bit.txt” file in that subfolder for more details.

ANSYS version 15.0 is supported for mesh generation and post processing to read the *.rst results file; updated to the newer brick element types: solid185 and solid186.

The ANSYS contour integral “CINT” command is now supported in the input file and during post processing to get the J-integral values computed by ANSYS; however the CINT syntax is omitted by default; to activate the CINT syntax in the ANSYS input file use the “*use ansys write cint” keyword in the Home/Notes form.

Added a check to give a warning if both the Warp3D packet file and sub-file options are used, since the post processor cannot yet read the results with both options active.

Updated the user’s manual to remove a figure image from the table of contents; updated notes about Warp3D executables.

February 25, 2014

Added the crack length to mesh width check for the cylinder geometry; the crack tube radius is automatically adjusted if the crack length is close to the edge of the mesh size: cylinder length for axial cracks and cylinder circumference for circumferential cracks.

December 17, 2013

Corrected a problem when reading an Abaqus format input file into the Tools/Show Stress Interpolation form to view a mesh; needed to update reading the *contour integral syntax.

December 4, 2013

Added a node set in the “Node List Data” comment block for the nodes through the ligament ahead of the surface crack at the $\phi=90$ location, and a node list for the nodes across the crack face at $\phi=90$; added for surface cracks in cylinders, plates, spheres, and elbows; the node lists are sorted by the y-coordinate from the crack front node; the node sets help to plot stress through the ligament and plot the crack opening displacement using the “Select Nodes” option or the extracted crack results file.

Added nodal stress values to the tables in the extracted crack results file (*_res.out) for the node lists in the “Node List Data” comment block; the sorted node lists are used; include the “*use node list data” in the Home/Notes form to activate.

November 18, 2013

Added J-integral contour versus load step values in the extracted crack results file (*_res.out) in the “J versus load step for all crack nodes” section; the step index is still the first column and the average J is still the second column; the contour specific J values are in the remaining appended columns. The J versus step tables are given for each crack front node.

October 11, 2013

In the user-defined geometry, use all the definition mesh elements to find the minimum edge length to set the mid face node merge tolerance; allows for a definition mesh with very large and very small element sizes.

Version 3.2.024 – September 19, 2013

Corrected the flat plate geometry with a buried (embedded) crack mesh position so that the bending stress can be correctly applied as equivalent nodal forces to the back face of the mesh.

August 7, 2013

Improved labels and added J_{total}/J_{elastic} output at debug level=3 to the FEACrack_RefStress.log file; used when post processing elastic-plastic results with the compute FAD option activated in the FEA Results/File Selection form.

Version 3.2.023 – July 30, 2013

Added a subfolder "Warp3D_files_64" with the Warp3D version 17.4.0 files and manual that can be used on 64-bit computers. Refer to the "Readme_Warp3D_64.txt" file in the subfolder for instructions.

July 15, 2013

Added an option to compute the crack face area for the flat plate and user-defined geometry; add the key word text: "*use crack face area compute on" in the Home/Notes form; also see the user's manual; the crack face area is written to the log file.

June 27, 2013

Added the ANSYS brick element types SOLID185 and SOLID186 to the post processor to help reading combined meshes.

May 24, 2013

Improved the SE(B) ram load pin force; apply the given x-direction force as element face pressure loading on the mesh instead of the previous distributed nodal forces; added a "*use" key word for the Home/Notes to use the previous nodal forces for the ram load pin.

Version 3.2.022 – May 13, 2013

Improved the reference stress and FAD curve post processing calculations; the calculations will continue to the next crack front location if there is insufficient crack front plasticity to compute the nominal load and reference stress; the if not all load steps are converged the partial results will be used to obtain a maximum J-integral value to compute the reference stress; added the FEACrack_RefStress.log file in the program directory to give any warning messages and a summary of the reference stress and FAD curve results for log flag = 1, and more details for log flag = 5; the log file contains the reference stress and FAD curves for all crack front nodes.

Improved the constant depth "canoe" shaped surface crack to allow short and deep cracks when using the PRO mesh block.

Fixed a problem with the cell type boundary layer model when there is 1 element in the thickness.

Fixed a problem with the SE(B) geometry for a small model using meter units.

Version 3.2.021 – May 3, 2013 – testing release

May 3, 2013

Improved the reference stress and FAD curve post processing; the reference stress calculations can use partial elastic-plastic J-integral results that are not converged for all analysis steps; the reference stress can be computed if the J_{total}/J_{elastic} ratio is above the material specific ratio for either phi=0 or phi=90 crack locations.

Version 3.2.020 – no release, internal test build

April 10, 2013

Updated the user's manual with information about the boundary layer model cell type mesh, the tabular temperature for a custom thermal gradient, and the tabular stress for the flat plate.

Version 3.2.019 – March 21, 2013

Add the cell type mesh to the boundary layer model (BLM) using Home/Notes keywords for the options, used for node release analysis.

For tabular crack face pressure, crack face elements with a load value of zero will not be written to the FEA input file to avoid post processing problems. Allows part of the crack face to have a zero pressure value.

Updated the Quest Integrity web page and e-mail address in the GUI.

February 20, 2013 – corrected a problem in the asymmetric buried crack when the elastic-plastic analysis is selected.

February 19, 2013 – corrected a problem in the ANSYS post processing module when reading a model containing bonded contact and target element data; since that data is not used in the result post processing it is currently skipped

February 2, 2013 – updated the warning message for crack opening area calculations not available for a quarter symmetric user defined crack mesh.

Version 3.2.018 – no build or release

Version 3.2.017 – June 29, 2012

Investigated incorrect crack face constraints for a PRO mesh constant depth surface crack; recompiled CrackGen.dll corrected the mesh constraints.

Version 3.2.016 – March 2, 2012

Updated the post processing module to support ANSYS version 14 *.rst result file; previous version of the ANSYS result file are still supported.

Added support for the C(T) geometry with a custom curved crack front using the PRO mesh.

Version 3.2.015 – November 22, 2011

Added prototype features for some boundary conditions, request the PDF information files for the details:

1. tabular temperature gradient in the plate and cylinder geometries
2. tabular stress gradient in the plate geometry
3. tabular crack face pressure to apply crack face pressure for multi-step analysis

Version 3.2.014 – August 11, 2011

Updated the post processing module to correctly read the “Crack Free Surface Node List” comment data block for larger models so that the node list is included in the extracted crack results files (*_res.out).

Updated the Boundary Conditions/Constraints form, bottom surface menu for the flat plate, to correctly retain the custom displacements choice.

Version 3.2.013 – July 27, 2011

Updated the Abaqus “*contour integral” command syntax to use two node sets at each crack front position.

Write the Abaqus “*contour integral” command in just the first analysis step data to reduce the Abaqus preprocessing; results are available from all analysis steps.

A key word option is available to control these two changes in the Home/Notes form; refer to the user’s manual for details on these two changes.

Version 3.2.012 – January 27, 2011

The crack front J-integral results for the free surface crack tip node for a surface crack is updated when one of the extrapolation options is selected even if the J path-dependence index is below the 0.05 cut-off; this helps the FAD calculation at the crack tip $\phi=0$ location for plasticity results.

For the plug weld geometry, improved the automatic tied contact surface selection between the plug zone and transverse weld zones.

Added an error message that the plug weld geometry is not yet available in the fatigue analysis module.

Version 3.2.011 – November 19, 2010

Fixed a problem for the finite radius crack tip in the full mesh and back half symmetry crack mesh for a surface crack to allow the full crack mesh to correctly generate.

Added support for reading special and international characters from the notes field in the input file; helps improve the post processing module.

Version 3.2.010 – November 5, 2010

Improved the shallow crack mesh additional thickness mesh block for shallow cracks in the plug weld geometry.

Added support to read Warp3D binary packets and text packets result file when running from the command line.

Fixed the load history incremental calculation mode option applied to the boundary condition values when running from the command line.

Improved the error trapping for the subset mesh option when a small region size gives no element or no nodes in the subset; report a warning message to adjust the region size, but continue writing the FEA input file.

Version 3.2.009 – September 24, 2010

Added SE(B) specimen specific PRO mesh controls for other mesh zones within the SE(B); new PRO mesh controls used by key word text in the Home/Notes form; see the user's manual for more details.

Changed the default parameter file setting to off so that the radial result post processing does not always automatically run.

Version 3.2.008 – August 30, 2010

Added the /ParamFile and file name command line options to activate the parameter text file for the radial result interpolation during post processing.

Added the through-thickness crack front node weighting values used to compute the average crack front J value to the extracted crack results *_res.out file.

Improved the final crack dimension calculation using the crack face edge nodes to better support an elastic-plastic mesh with untied crack front nodes.

Version 3.2.007 - July 19, 2010

Warp3D version 16.2.7 updated to input optional crack front node tangent vector, which is used at the crack tip and symmetry plane crack front nodes to reduce the J-integral path dependence; refer to the Warp3D manual section 4.4.6.

Updated the routine that writes the Warp3D input file created by the mesh generator to write the crack front tangent vector at the first and last crack front node to get the accurate J results.

Version 3.2.006 - June 22, 2010

Improved the elbow geometry (curved tube) transformation for better accuracy of the final crack length.

Added an option to the PRO mesh to get vertical mesh lines below the surface crack; useful for long surface cracks to avoid skewed elements in the ligament below the crack front; currently activated by key word text in the Home/Notes form.

Added crack dimension calculation using the crack face edge node lists; the computed crack dimensions and node list tables are written to the model log file; useful for user-defined geometry model to compare the given crack dimensions to the final crack dimensions in the crack mesh.

Version 3.2.005 - May 13, 2010

Added the Visual Studio 2008 C++ redistributable package to the installer.

Increased the crack tube maximum size for the through-thickness crack shape when using the refined mesh block; the user defined crack tube diameter can be up to $0.45 \times \text{thickness}$.

Improved the error message for invalid node ID data in the user-defined geometry definition mesh.

Added a suggestion to the MEMORY ERROR message to reduce the maximum number of nodes and elements in the Tools/Model Options/Crack Analysis form.

Version 3.2.004 - April 6, 2010

Added a side groove option to the SE(B) model; use key word text in the Home/Notes form to activate and set the size of the side groove.

Changed the default Intel MKL solver out-of-core memory size to 1000 MB; solver parameter for Warp3D analysis; the memory size can also be set using key word text in the Home/Notes form.

Version 3.2.003 - February 18, 2010

Added the side groove option for the SE(B) geometry; a new prototype feature for quarter symmetry SE(B) geometry.

Added Intel MKL solver out-of-core options for Warp3D analysis; the options are set with key word text in the Home/Notes form.

Version 3.2.001 - January 28, 2010

Added the "node constraint release" option (prototype) for the SE(B) geometry; use key word syntax in the Home/Notes form to activate the option and set the zone size.

Version 3.2.0 - December 4, 2009

Updated the "FEACrack User's Manual v3_2.pdf".

Improved the default ANSYS *.node file name, when the option is used.

Added the corner crack shifted location in the flat plate geometry.

Changed the default (restore) Warp3D element block size to 128 in the Tools/Program Options/FEA form; element block size of 64 or 128 give the best cpu time performance on most PCs.

Corrected a problem saving elastic-plastic material stress-strain data in the Materials form; old model files can be updated by changing the value of the "m_ElasticPlastic" variable in the *.elt file (FALSE for elastic, TRUE for elastic-plastic), or update the option in each active material form and resave the model.

Version 3.1.80 - October 19, 2009

Corrected a problem writing the Warp3D input file with the finite radius crack tip for 20-node brick elements.

Added the "Stress intensity K from J-integral results" section in the extracted crack results file; lists the K results in the FEA Results/Crack Results/Data form.

Version 3.1.79 - October 12, 2009

Warp3D version 16.2 executables added to the install, uses the improved multi-core Intel MKL sparse matrix solver with out-of-core capability; 64-bit executables are available on request.

Improved the ANSYS file import when there extra unused element types in the model file; the "mesh200" element is skipped.

Added predefined node lists with displacement results to the extracted crack results file; the predefined node lists are created by the mesh generator.

Improved the key hole mesh option for the full model when the crack location is at a user specified, shifted, location.

Version 3.1.78 - September 28, 2009

Corrected a crash on exit error when using the command line syntax; additional initializations have been added when "/noquit" is omitted.

Version 3.1.76 - September 25, 2009

Updated the Warp3D executables to correct a crack face pressure loading warning.

ANSYS version 12.0 results (*.rst file) supported by the post processor.

Added "*use ansys node file" to the Home/Notes form as a prototype to test use of the ANSYS *.NODE file in the submodel cut-boundary analysis method.

Version 3.1.75 - September 10, 2009

Added an optional position for the C(T) specimen side groove height "H_g"; improved the C(T) specimen mesh generation and input labels.

Added command line syntax options to modify model parameters and run a complete analysis from data input to mesh generation to FEA to post processing.

Limited the length of subfile names in the Warp3D input file to avoid long file names getting truncated, which causes an error during the Warp3D analysis.

The material properties forms now close one at a time, or all forms can be closed using the "OK close all" button on the main form.

Added "*use warp3d function type=d" to the Home/Notes form as a prototype to test use of the through-thickness crack J average value in Warp3D; updated the post-processing to set all the crack front J values to the same single value from function d.

Added "*use midpoint = <value>" to the Home/Notes form as a prototype to test locating the crack front element midside node at locations other than the default quarter point.

Version 3.1.72 - August 14, 2009

Added the Mises and Cyclic material option for Warp3D input files in the Material Properties form

Added the plasticity hardening option for Abaqus input files

Improved the "assume elastic behavior" and "assume elastic-plastic" behavior option for plasticity material; check material data for old models

Added the subfile option for Abaqus and Warp3D; the input file references subfiles for the mesh data

Improved the friction stir plug weld mesh options input form, labels, and parameter pictures

The crack initially closed option is available for the plug weld with Abaqus

Improved the Status window in the Run FEA form to better echo the FEA solution status file during analysis

Corrected the custom through-thickness crack PRO mesh to have an independent number of crack front elements from the crack front profile points

Corrected the crack front node sorting index for some shallow surface cracks

Corrected a problem reading older *.elt model files for the boundary conditions form data

Corrected a problem in the C(T) model dimensions form for value h_2 being reset

Improved the C(T) geometry mesh zones to the right of the crack front

Improved the increment calc. mode menu on the Boundary Conditions form

The batch messages option in Tools/Program Options/Warp3D is now set with the Restore button

Version 3.1.60 - July 1, 2009

Added analytic rigid cylinder surface load pins to the flat plate boundary conditions.

Added an option to set the ANSYS command line syntax to use redirect "<" or input/output "-i" and "-o" flags.

Improved the Abaqus FEA input file to allow larger node ID and element ID numbers to 99 million for large meshes or renumbering.

Corrected a problem using multiple crack sizes in the plug weld geometry.

Corrected a problem with the radial node list, in some surface cracks the left crack tip was missing from the list, now included.

Version 3.1.56 - June 10, 2009

Corrected a problem saving a user-defined geometry to the model *.elt file; a work-around is available to correctly load the .elt file, please contact technical support for more information.

Version 3.1.55 - June 5, 2009

Added the boundary layer model (BLM) plane-stress perimeter displacement option

Corrected in the crack result x-y plot to autosize the x-axis

Version 3.1.45 - May 5, 2009

Corrected a problem with $K = 0$ for the fatigue analysis module using custom crack shapes; the analysis will now continue by setting $K = 0$ at specific crack front nodes.

Version 3.1.42 - April 9, 2009

Added the friction stir plug weld model as a new geometry choice.

Improved the node coordinate accuracy on the inside and outside surfaces of the cylinder geometry.

*INFO: The FEAFLaw_Artwork.dll was modified, but a problem with this update install does not correctly place the updated dll in the install directory; work-around is to remove the dll, then run the install update.

Version 3.1.33 - March 2, 2009

Improved the cylinder with a custom crack surface crack to support fatigue crack growth analysis.

The Pressure loading tab is now always active for appropriate geometries (cylinder, elbow, sphere, sphere with hole, nozzle on shell or head, perforated cylinder); this allows both the Pressure and Crack Face Pressure tabs to be active; the crack face pressure value will take precedence over the internal pressure and the load values are not cumulative.

Version 3.1.29 - January 8, 2009

Corrected a problem with the nozzle mesh transformation.

Version 3.1.28 - December 15, 2008

Improved the custom surface crack for the full crack front (back half symmetry for example); if a custom crack front point is not at $c=0.0$ interpolation is used to update the nearest custom point; this allows for more general custom crack growth.

Improved the error trapping for initial stress interpolation with Abaqus results.

Version 3.1.27(B) - November 25, 2008 - not posted to the web site

Improved post processing for ANSYS results with mixed element types; added the 10-node tetrahedron element to the ANSYS post processing module.

Corrected a file path problem when reading an Abaqus result file.

Version 3.1.27 - November 6, 2008

Run FEA form - QR File Control Wizard; update with option to echo the FEA status file to the bottom status window in the form; use to monitor the analysis progress.

Update the Geometry/Crack/Dimensions/Auto Calc form to compute crack front points for the half ellipse curve, for the full model and back-half symmetry.

Improved the user-defined geometry import for ANSYS format input files.

Improved tabular crack face pressure warning message.

Version 3.1.22 - September 24, 2008

SE(B) specimen model; update the roller diameter, D, and load ram radius, R, to allow zero size; a single line of nodes across the thickness is constrained.

SE(B) specimen model; add an optional second material group to set material values at the constraint and load locations; allows elastic material at the constraints in a plasticity analysis.

Updated the material form to support elastic or plasticity material type per material group.

Version 3.1.19 - September 17, 2008

Corrected a problem for the custom through-thickness crack shape for use in the fatigue analysis module.

Version 3.1.17 - September 10, 2008

Updated the fatigue analysis module for the surface crack; an update to the fatigue equations in the user's manual will follow.

Updated the plate force loading to get equivalent traction stress loads on mesh surfaces for the various symmetry choices.

Updated the definition mesh import to improve reading an Abaqus input file created by CAE; changed an error to warning message when the number of detected surfaces is not equal to 6, allows the imported mesh to be viewed.

Version 3.1.15 - August 11, 2008

Added the custom surface crack full mesh and back-half symmetry mesh options.

Corrected a problem for ANSYS post processing for a large number of crack front nodes.

Version 3.1.14 - August 1, 2008

Corrected a problem in the ANSYS post processing module when the crack plane rigid surface elements are used and when reporting model data to the log file.

Corrected a problem for the PRO mesh surface crack.

Version 3.1.13 - July 23, 2008

Boundary layer model, set the crack tube radius equal to the model radius to avoid an error.

Version 3.1.12 - June 13, 2008

Corrected pressure loading near the crack front

Corrected custom crack fatigue analysis

Version 3.1.8 - May 6, 2008

Reduced the memory required for a user-defined geometry mesh transformation; allows use of larger definition meshes.

Ansys version 11 now supported by the post-processing module.

The submodel boundary nodes are written to a single node set for Abaqus.

Version 3.1.7 - April 25, 2008

Corrected the custom through-wall crack shape mesh transformation.

Improved the crack mesh block size control for the standard mesh blocks to reduce the crack block size.
Updated the crack rotation transformation; can now use the theta_Y value with the PRO mesh.
Improved the PRO mesh for very small elements at the crack front.

Version 3.1.6 - April 21, 2008

The name, label, and notes text from the Home form is passed through to the extracted crack results file.
Added user-defined material support for the Abaqus UMAT routine.

Version 3.1.3 - March 25, 2008

Corrected the asymmetric buried crack in a single-V weld

Version 3.1.2 - March 18, 2008

Corrected the C(T) geometry bottom load pin constraint.
Improved the GUI form controller for dual monitor use.
Corrected the sphere-with-hole geometry transformation.
Corrected the sphere geometry transformation when changing from 20 node to 8 node brick mesh.

Version 3.1.0 - March 12, 2008

Release of new features:

Inner plastic zone option for PRO meshes
Flat plate analytic rigid cylinder surfaces to model load pins, 3-point bending
File translator to convert an Abaqus input file to a Warp3D input file
Expanded analysis parameter controls for Warp3D and Abaqus
Updated features:
Update the Patran neutral file format for nodal forces (packet 7)

FEACrack Version 3.0 Updates

Version 3.0.35 - February 22, 2008

Update for the definition mesh file importing module for a user-defined geometry

Version 3.0.34 - January 31, 2008

Added use of the 20-node brick element mesh to the custom crack fatigue analysis module.

Version 3.0.33 - January 29, 2008

Corrected a problem with the cylinder boundary conditions that had been caused by recent additions to the program.

Version 3.0.32 - January 25, 2008

Corrected a problem with the material group ID for elements in a back-half symmetry mesh.

Version 3.0.31 - January 16, 2008

Corrected a problem with the rotated and shifted crack in a user defined geometry.

Version 3.0.30 - January 15, 2008

Corrected a problem with the unique material groups on each crack face when the shallow crack option is used.
Corrected a problem with the SE(B) geometry to allow longer cracks in the model.

Version 3.0.29 - January 10, 2008

Added load pin boundary conditions for the flat plate to model 3-point bending; the load pin can be modeled as an analytic rigid cylindrical surface that contacts the mesh, or by applying constraints or forces to the mesh at the load pin locations.

Version 3.0.25 - December 20, 2007

Added analysis parameter controls for the FEA input file analysis options.

Added load history function choices to automatically fill the boundary conditions load tables.