APPENDIX A

QUESTIONNAIRE

Contact Name: F-15 Strike Eagle

Collaborators: N/A

Name:

Organization:

Name:

Organization:

Please provide information about the analyses completed:

1. Analysis Software (name and version)
    a. FEA software (if applicable): Ansys solver
    b. Crack growth software: SimModeler Crack

2. FEA Model Setup (if applicable)
    a. Describe the boundary conditions utilized in the FEMs, to include applied loads and constraints
        i. The 3D nominal geometry (Parasolid) of the specimen was created exactly as the drawing in the provided document
        ii. Material properties (E, ν) provided in the document were used in the FE model
        iii. LEFM framework used
        iv. Fastener was represented as a cylinder with diameter of exact value as provided in the document.

v. The overlapping fastener-specimen hole provided the interference fit stress distribution by setting a standard contact type between the two surfaces in Ansys. Each crack front simulation solves for the initial IFF stress state. For this first Ansys load step, the displacements on the top and bottom surfaces are constrained (ux=uy=uz=0)

vi. For the second and the third load step, the displacement constrains are replaced with rigid body constrains and an applied load corresponding to maximum of 27.9 ksi (second load step) and minimum 2.79 ksi (third load step) to reproduce experimental conditions

vii. Initial crack sizes were created at the exact size as provided in the Round Robin Problem document. Initial cracks were inserted using SimModeler. Initial crack shape was assumed elliptical for all three benchmarks. Once the crack is incremented, the crack can take any shape and computed nodal ΔK controls that.

b. Describe the methods to define and control the crack front shape and control meshing along the crack front

i. Stress intensity factors are computed using displacement correlation technique

ii. No crack front shape constraint was used. Crack front increments are based on nodal stress intensity factor ranges computed at each node along crack front.

iii. Each crack increment is meshed with SimModeler Crack and a new input deck is created for an Ansys solution automatically. The entire crack propagation process is run in batch mode for a user defined number of steps (like 50, 100 or 200 steps).

iv. Mesh along crack front is controlled by the user. 50, 100, 200 elements can be used to define number of singular wedges along each crack front increment for the entire analysis.

v. Crack advancement increment is also controlled by the user. The user can decide to simulate 50 increments with a given max advancement for each step.

vi. SimModeler computes stress intensity factor range based on loading cycle and calculates assigned advancement for each node based on ΔK, max advancement and Paris law (constrain to same ΔN)

3. Interference Fit Modeling

a. Describe the methods used to characterize and incorporate the effect of the IFF.

- The diameter of the specimen's hole and the fastener were defined based on description of each benchmark. Parasolid models for each component (fastener and specimen) capture the exact diameter mentioned in the Round Robin problem. The Parasolid models are meshed in SimModeler and an Ansys Plugin prepares the input deck to solve the model. This process is automatic for each crack front increment.

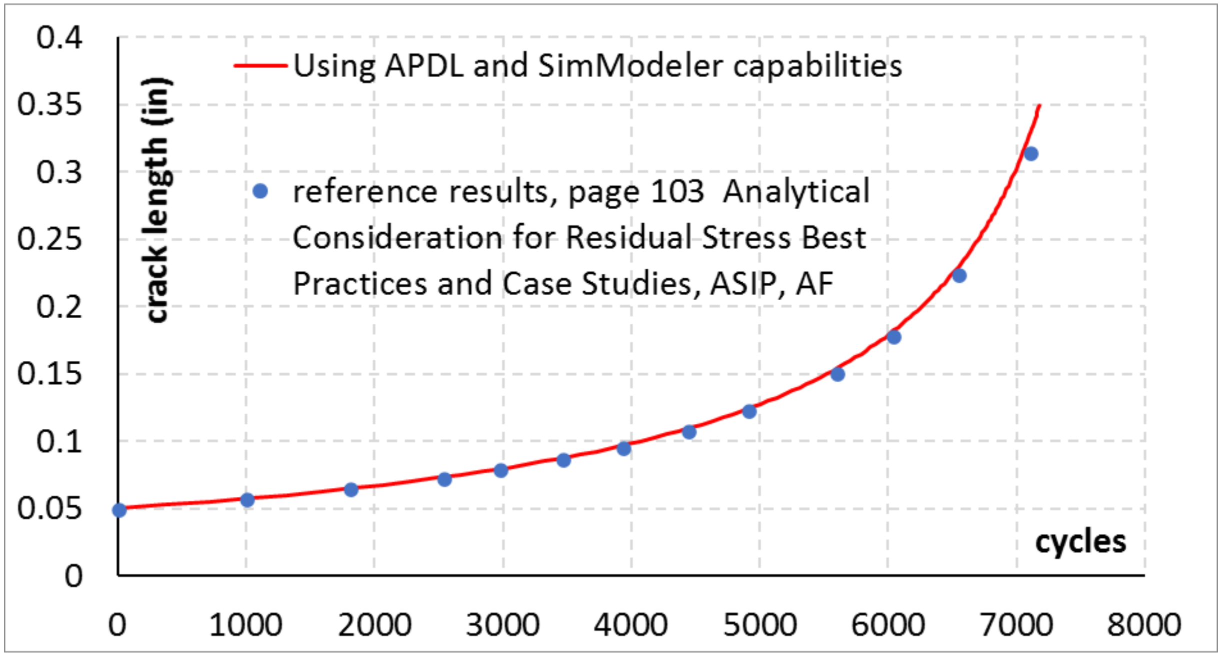
b. If the fastener effect was derived from a closed form solution, what were the assumptions of the solution. Is the solution based on empirical data or FEM correlations?
   i. No "fastener effect" type of simplified modeling was used.

c. If the fastener was modeled using FEA, does the model consider non-linear effects? Was multi-body contact used? If contact was used, what friction related assumptions were made?
   i. Multi-body contact was used. The fastener was modeled in the 3D FEA simulation for each crack front increment. Contact definition between fastener and specimen hole captured stress gradient at the hole as an initial solution (loading step). The following two loading solution steps captures far field loading in addition to interference fit.
   ii. The Ansys contact type used throughout Benchmark 2 and 3 is "Standard". Coefficient of friction = 0.3.

4. Stress Intensity Calculations
   a. Describe the methods used to extract and calculate the stress intensities for applied remote loads
      i. Each crack front increment was modeled explicitly using 3D FEA.
      ii. Stress intensity factors were computed on nodes along crack front with an Ansys apdl script.
      iii. The APDL script computes stress intensity factors along crack front for IFF+max load and IFF+min load to determine ΔK responsible for the crack growth
      iv. ΔK is used in Paris relationship to determine ΔN for an incremental crack front growth
      v. Each node has a different assigned advancement based on nodal ΔK and an R ratio
   b. Describe the methods used to incorporate the stress intensities into the crack growth code (superposition, etc.)
      i. The stress intensity factor **range** used to predict number of loading cycles between two consecutive crack increment is computed based on second solution step and the third. The second solution step is IFF+Max loading condition while the third is IFF+Min loading condition.

5. Crack Growth Predictions
   a. Describe the material model approach used for the crack growth predictions (NASGRO, tabular, etc.) and the assumptions/approach used for "threshold", stress ratio (R) shift, and negative R behavior.
      i. The entire fatigue crack growth rate data provided in the Round Robin document was used to interpolate the fatigue crack growth rate for each node along the crack front.
      ii. For each node along a given crack front increment a stress intensity factor range and an R ratio is computed based on stress analysis results for IFF+Max load and IFF+Min load (basically the two loading conditions superimposed to IFF)
      iii. Each crack front advancement is simulated using 3D FEA.
   b. What growth increment was utilized between stress intensity calculations?
      i. I used different growth increments. For increments starting from initial crack I used … As the crack propagates and becomes larger, I increased the max advancement increment allowed along each crack front to ….

6. Provide any additional details that may be pertinent to the analyses completed
   - I verified the same procedure used for the three benchmarks (APDL and SimModeler Crack) against solution presented in "Analytical Consideration for Residual Stress Best Practices and Case Studies, ASIP, AF", center crack specimen in plane strain conditions, pages: 100-103, document#: AD1084445.
   - "Reference results" in the plot below are my digitized values correspondent to "Franc3D, NASGRO, Beasy, AFGROW" values reported in Figure 60.
   - The reason for pursuing this verification assessment is to point out that the stress intensity factors are computed accurately. The same APDL script is used for all three benchmarks. Since I am using full 3D representation of the geometry, there is no error introduced due to geometric simplifications associated with handbook solutions i.e "corner crack in a rectangular plate". It might take longer to solve but, the accuracy of a 3D model is superior.



- For all three Benchmarks, I reported values along the bore and along front face of the specimen. I could report data along different directions but that would take too much time.
- I did not run a convergence study. I just gave it a good shot. The difference between the three loading cases (no IFF, two different IFF values) is captured.
- For Benchmark 1 and 2, I stopped reporting data once the crack reaches the length of the bore. Number of cycles was very low (high ΔK values)
- For Benchmark #3 I am not able to finish up in time i.e. propagate the crack to the other side of the bore. I will continue running the model to finish up the comparison beyond the June 1st deadline. Maybe I can use my results for a presentation in one of the Working Group meetings/Tcons.