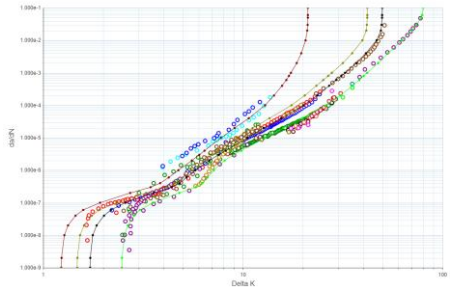
AFGROW User Workshop 2026

Automatic β-Table generation for general model definitions in AFGROW

Adrian Loghin, Simmetrix Inc.

AFGROW: user-friendly and run-time efficient Linear Elastic Fracture Mechanics (LEFM) based crack growth life prediction tool that serves industry -Table tolerance analysis needs



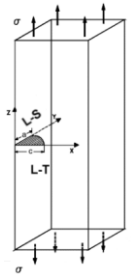
Model based fits to FCGR datasets
(Forman, Walker, NASGRO, Harter-T)

Tabular entries for 100 materials

User defined FCGR

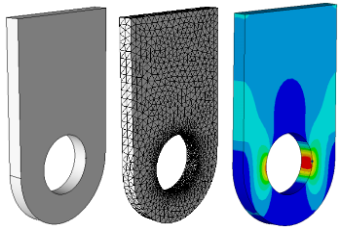
AFMAT

β-Table



1-DoF and 2-DoF models
(precompiled database, analytical formulations, user input - β-Table)

Parametric Multi-DoF models
(automatic 3D FEA based solution directly from AFGROW)



Retardation models

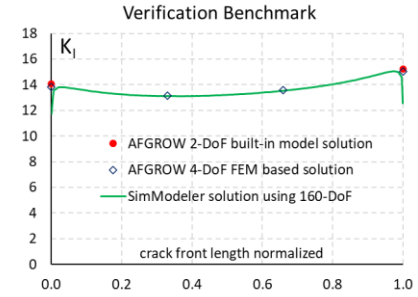
Modeling Capabilities

Crack propagation life solution vs. experimental measurement

K_I solution comparison against other solutions

V&V Benchmarking

Solution convergence

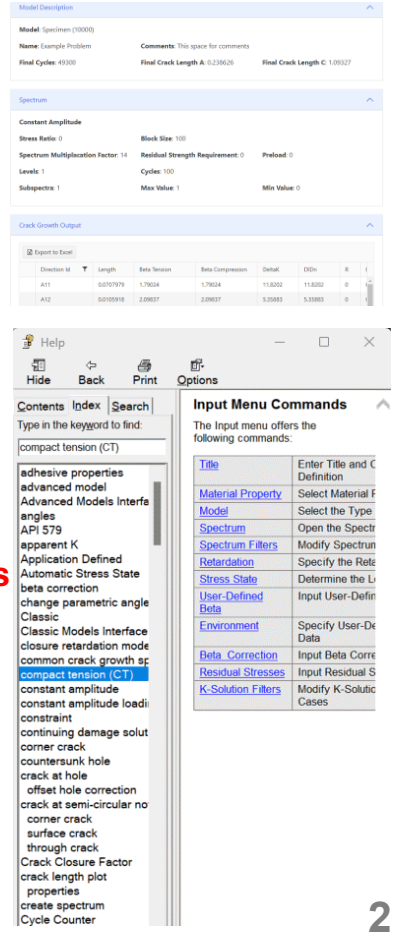


Postprocessor

Output

Plots/Reports

Crack growth evolution graphics

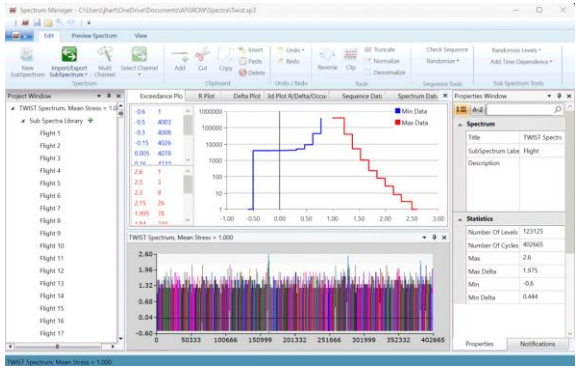


AFGROW

GUI

Spectrum Manager

Flight specific spectrum definition



Documentation

Public access to all Workshop presentations

Examples/Training

Manuals/Context Sensitive Help

COM Interface

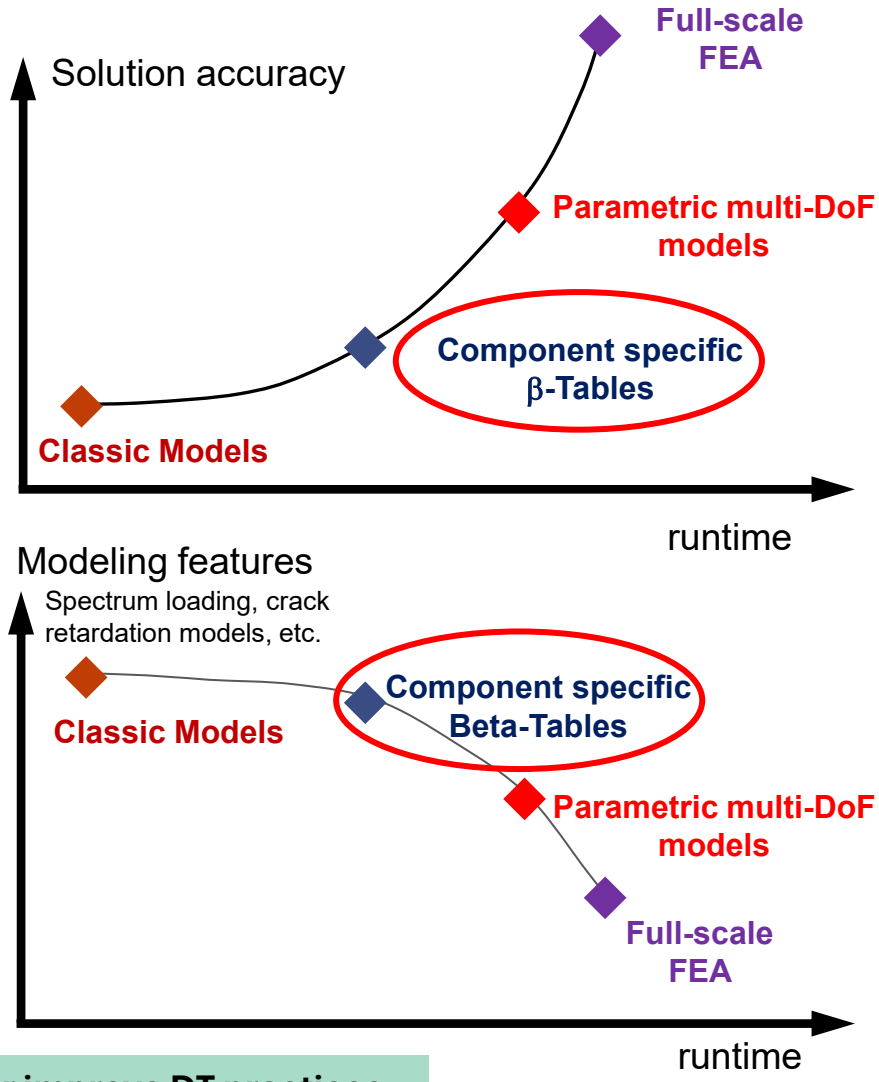
Loading spectrum generation & editing

User Application

Motivation

- **Extend usage of AFGROW capabilities by allowing end-users to perform 3D FEA and automatically generate β-Tables.**
  - ✓ **Modeling procedure is not restricted to specific geometries and loading configurations.**
  - ✓ **K_I accuracy improvements due to usage of local models that are more representative of the structure or even full component level models.**
  - ✓ **By enabling β-Table generation, we allow end users improve solution accuracy and provide a modeling solution for cases that cannot be reduced to a model that is already available in the database.**
  - ✓ **Access to all other capabilities available in AFGROW: spectrum loading, AFMAT, COM interface.**
  - ✓ **Allow end-users to improve damage tolerance analysis practices**
  - ✓ **Allow users make use of existing FE models and offer a platform for generating β-Tables.**

Generic assessment on various modeling options



End-user can pick the suitable model for their application or improve DT practices

Development of β-Table generation for AFGROW

End-user Requirements	Versatile input to address different Damage Tolerance modeling practices in the industry	Automatic and robust fracture mechanics-based modeling process	Demonstrate solution accuracy V&V benchmarking availability
Modeling capabilities	• Component level CAD models, orphan meshes • Geometry and boundary conditions agnostic • Model based identification of crack location and orientation	• Automatic mesh generation and solution for a set of predefined crack definitions • Supported solvers: CalculiX (free FEM software), Ansys, Abaqus. • Solver built-in element formulation, access to all solver capabilities.	• V&V benchmarking distributed with SimModeler • Several V&V documents in public domain • Additional solution verification is included herein
End-user Benefits	• Efficiently utilize existing solver specific FE models and analysis scripts to perform a Damage Tolerance assessment • Improve solution accuracy by using a more representative FE model definition	• Automatic β-Table generation and connectivity with all AFGROW features for crack propagation life assessment • Reduced development time for new fracture mechanics models or extensions to existing models	• Increased confidence in the modeling process • Examples can be provided for a quick learning process • Guidance can be provided for end-user specific models

Fracture mechanics modeling API functionality enables β-Table generation for AFGROW solutions

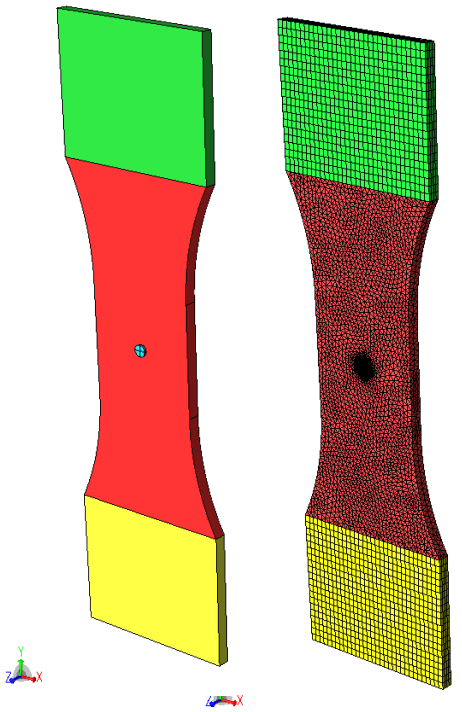
Automatic mesh generation, crack insertion, Plugin capabilities, KI calculation, V&V benchmarking

3D modeling technology behind this development

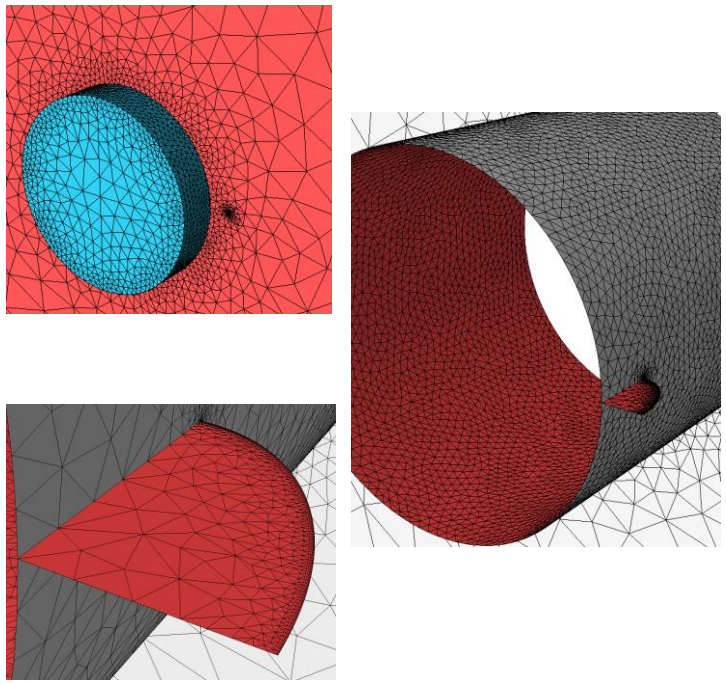
Automatic and robust fracture mechanics-based modeling process

- **Remeshing technology advantages:**
  - ✓ uses classical element formulation that is available in all FEM packages,
  - ✓ local mesh modifications, element compatibility is enforced,
  - ✓ mature technology at production level.

Geometry-Mesh association



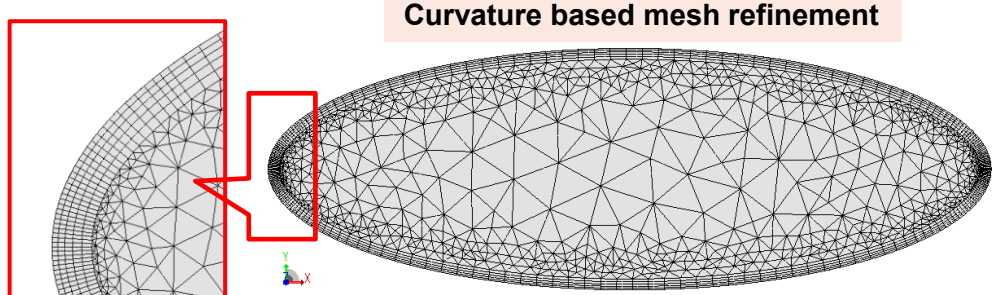
Part-based crack insertion and meshing



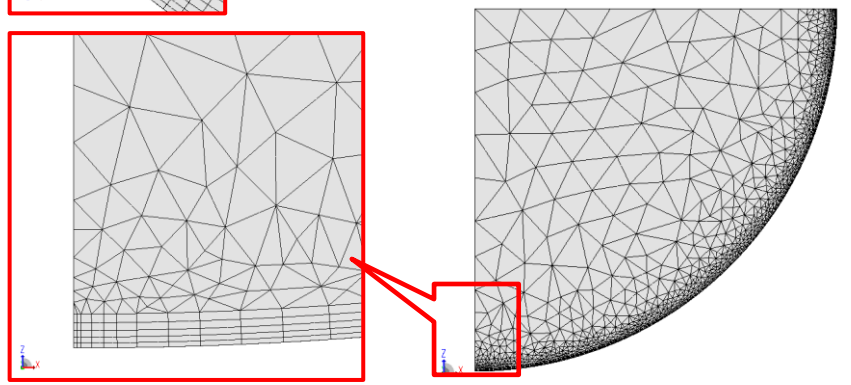
Mesh generation: automatic & geometry based

- **Control over mesh pattern and mesh refinement using crack surface geometry.**

Curvature based mesh refinement



Free boundary mesh refinement



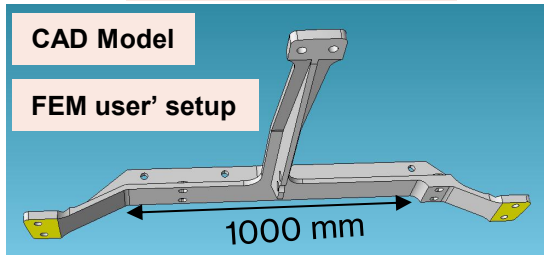
Minimal end-user interaction in the β-Table solution generation

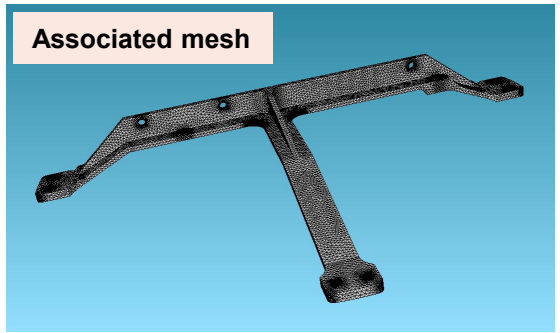
β-Table generation for AFGROW: modeling process flow

- **An initial FEA solution (SimModeler coupled to a solver) is used to identify a critical location for fatigue crack initiation. This solution is used to define crack location and orientation.**
- **A user predefined set of elliptical crack sizes is used to update the model and repeat the analysis to provide K_I values for each crack. These values are normalized to create a β-Table for an AFGROW crack growth solution.**

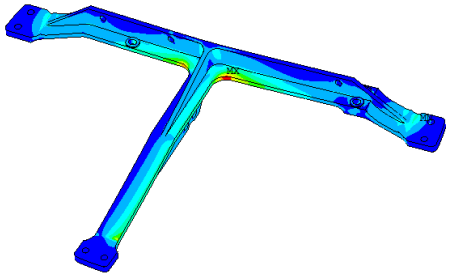
Automatic, parallel job, multi-threaded solution process

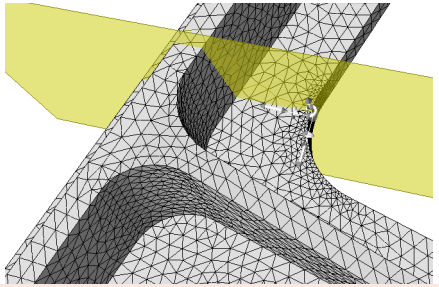
Model Definition





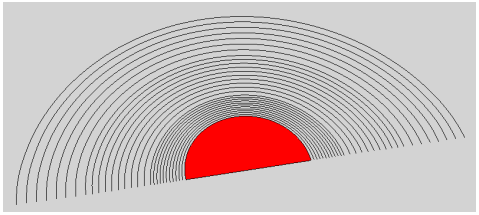
Model solution



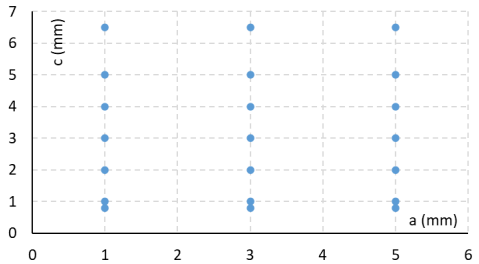


Solution based crack definition

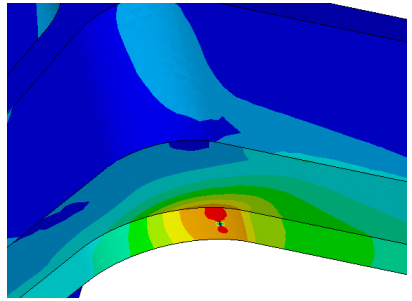
Crack growth bounds

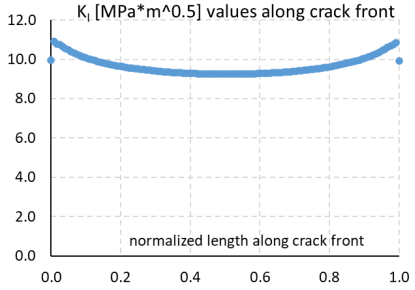


{a, c} crack size-Table definition

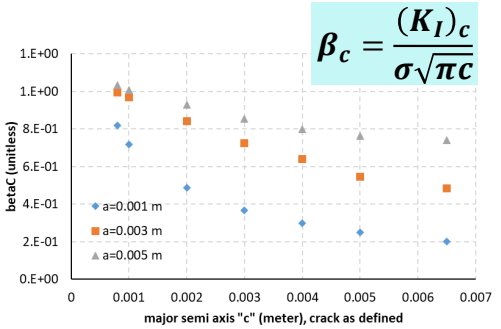


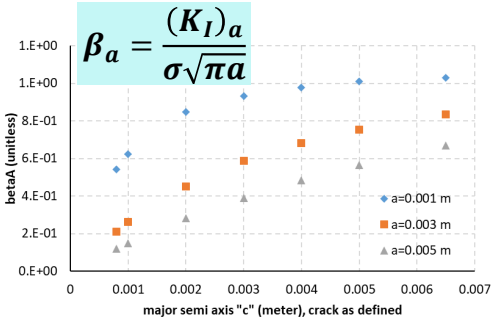
FEM based solutions





Solution postprocessing





Modeling process flow

Verification Benchmarking

- **To provide end-user confidence on the solution accuracy, verification benchmarking is added to this development.**

- **According to Oberkampf & Roy [1], "for a rigorous code verification, an exact solution to the underlying governing equations is required".**
  - ✓ **A 3D exact solution (Kassir and Sih, [2, 3]) will be used to demonstrate solution accuracy and convergence.**
  - ✓ **K_I solution errors are computed for all nodal locations along crack front not only specific locations.**
  - ✓ **10 different aspect ratios (elliptical crack shapes) were considered in this benchmark.**

- **"Code-to-code" comparison is also acceptable when exact solutions do not exist [1]**
  - ✓ **The reference solutions (B. Andersson's) used in K_I accuracy round robin challenge [4][5] are considered as highly accurate.**
  - ✓ **All 7 cases considered in the round robin challenge were considered in this benchmark.**
  - ✓ **K_I solution differences are computed for all nodal locations along crack front.**

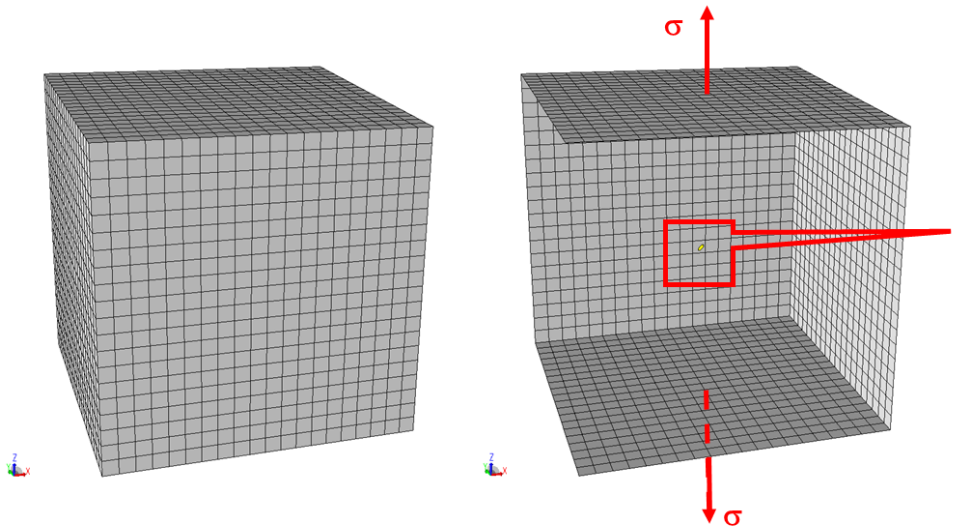
- **Quartiles, median, mean and spread of the relative error or relative difference data is a better choice to demonstrate solution convergence.**

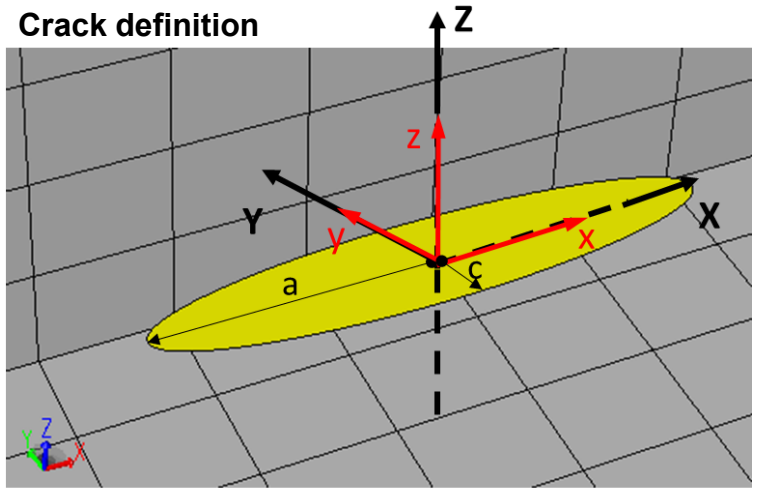
References
[1] Oberkampf W.L., Roy C.J., Verification, Validation, and Uncertainty Quantification in Scientific Computing, 2025.
[2] Kassir and Sih, Three-dimensional stress distribution around an elliptical crack under arbitrary loadings, J. of Applied Mechanics, 1966.
[3] Tada H., Paris P.C., Irwin G.R., The Stress Analysis of Cracks Handbook.
[4] Pilarczyk R., Cross-Comparison of Stress Intensity Factors from Various Sources: The Pathway to Improved SIF solutions, AFGROW Workshop 2021.
[5] Pilarczyk R., Engineered Residual Stress Implementation (ERSI), Stress Intensity Comparisons Round Robin, Report number ERSI-2021-01.

Verification benchmarking against exact Kassir & Sih solution

- **Kassir and Sih (1966): Three-dimensional stress distribution around an elliptical crack under arbitrary loadings. Solutions available in Tada [3].**
- **Only K_I was considered in this verification benchmark since this community is focused on Mode I crack growth conditions.**
- **Displacement correlation technique (singular wedges) is used to compute K_I values along crack front**
- **The 3D equivalent edge crack K_I solutions [6] plane strain conditions are necessary but not sufficient for code verification purposes.**

Cubic domain, 1-meter characteristic length
ν=0.3, E=21e4 MPa
Load: uniform tension, 100 MPa
γ = 90 deg
Elliptical crack, c = 5 mm, a = 1 mm (semi axes length)
K_I [MPa*mm^0.5]





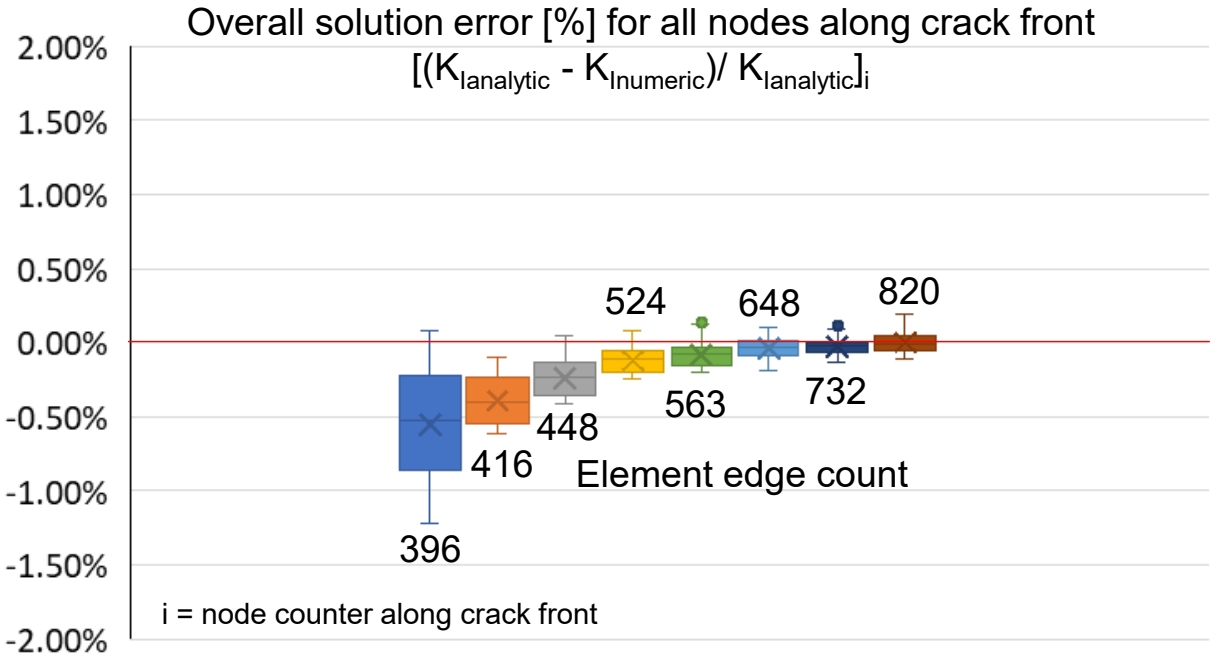
{X, Y, Z} global coordinate system
Crack plane normal z defined by {0, 0, 1}

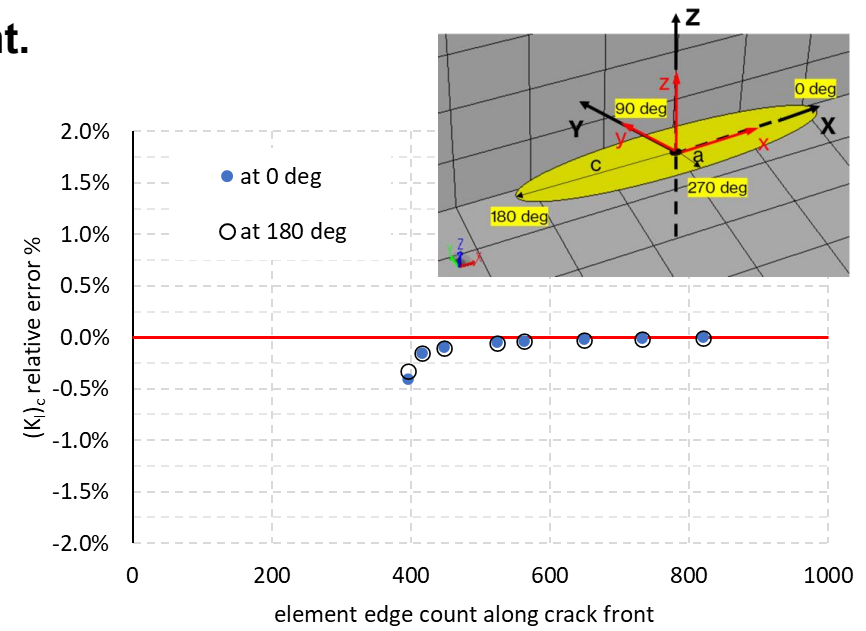
References
[6] Pilarczyk R., Morrison C., Hodges J., Benchmarking problems in Fatigue Crack Growth Analyses, AFGROW Workshop 2016.

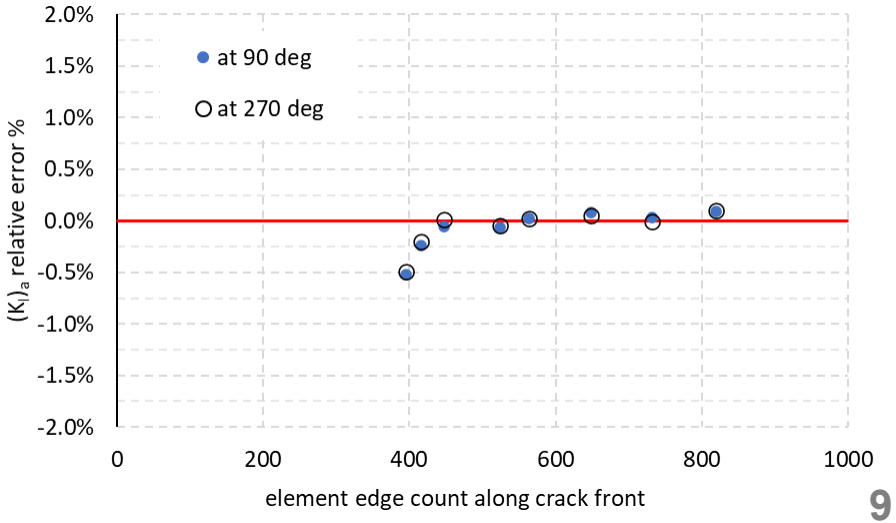
Verification benchmarking against exact Kassir & Sih solution

- **K_I solution convergence is demonstrated using crack front mesh refinement.**

SimModeler/Python scripting/solver: CalculiX





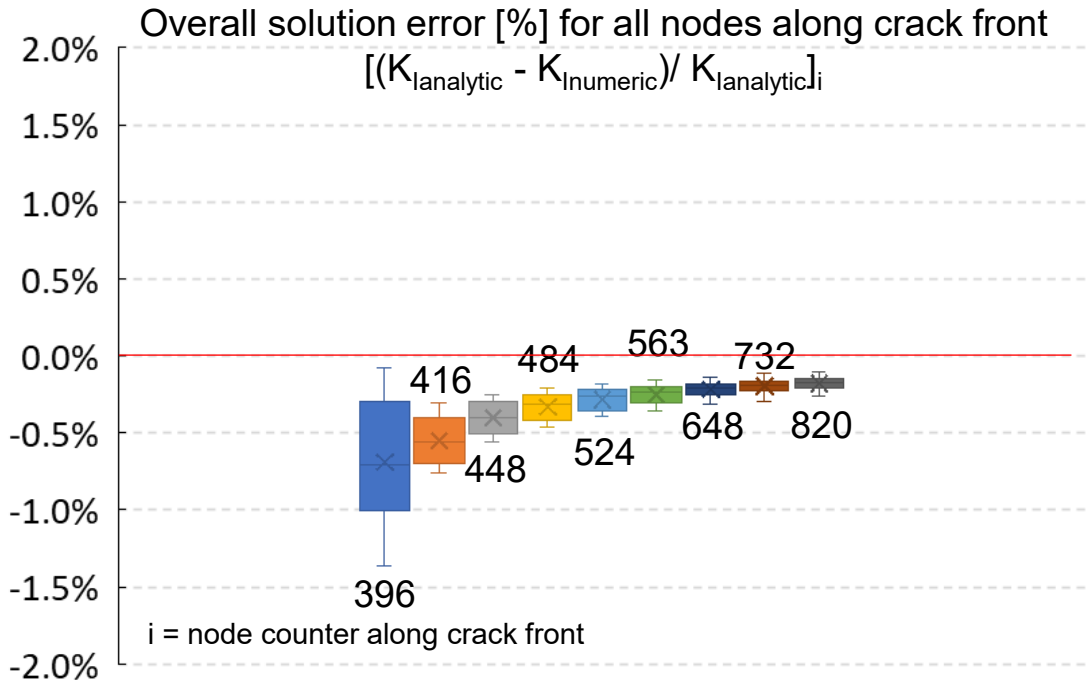


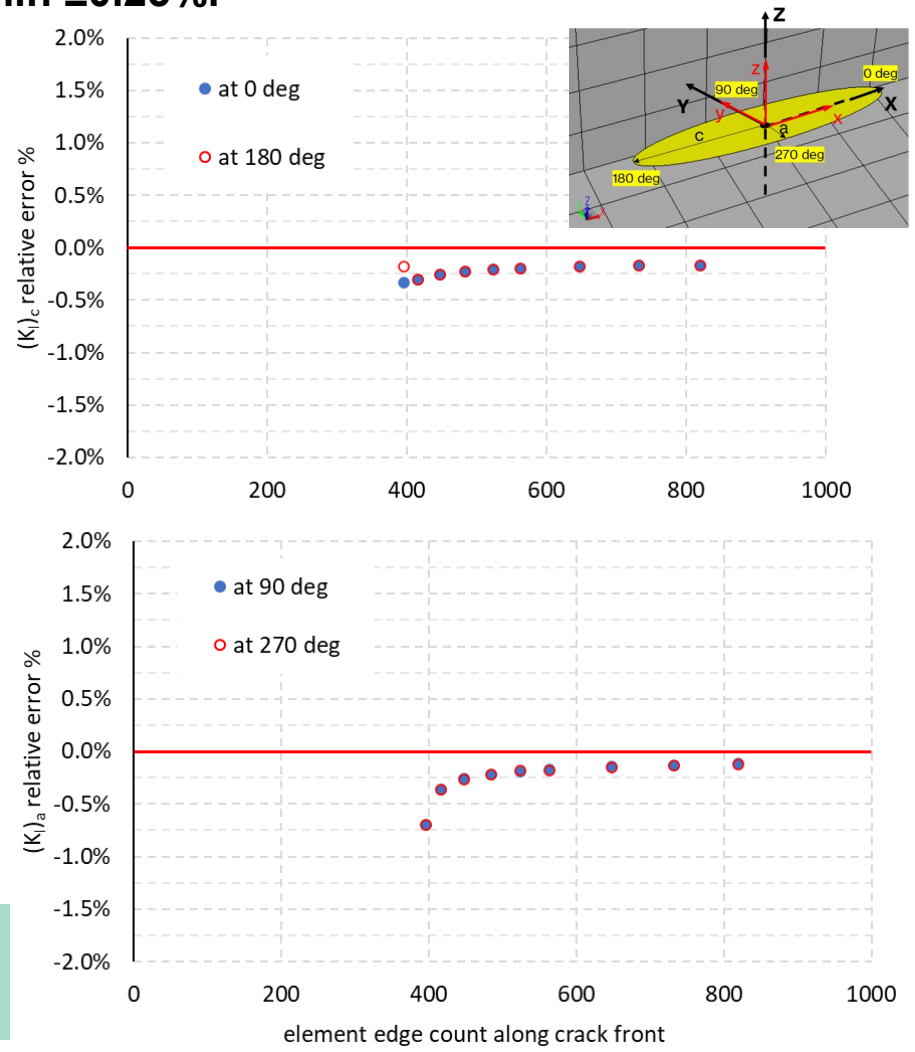
$(K_I)_i$ solutions converge within (min, max)=(-0.11%, 0.20%)
$(K_I)_a$ and $(K_I)_c$ solutions converge within ±0.1% error bounds

Verification benchmarking against exact Kassir & Sih solution

- **The same K_I solution accuracy and convergence assessment is carried out using Ansys as a solver. This assessment uses the same meshes as the CalculiX based solutions. For other mesh patterns, Ansys solution shows higher accuracy than the CalculiX equivalent however, the relative errors are within ±0.25%.**

SimModeler/Python scripting/solver: Ansys



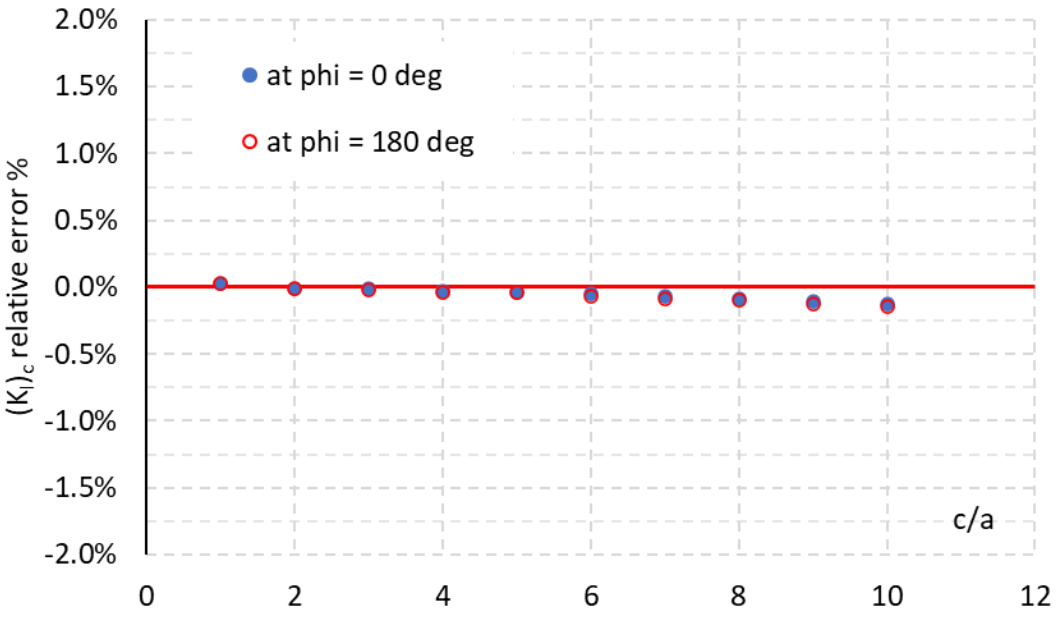


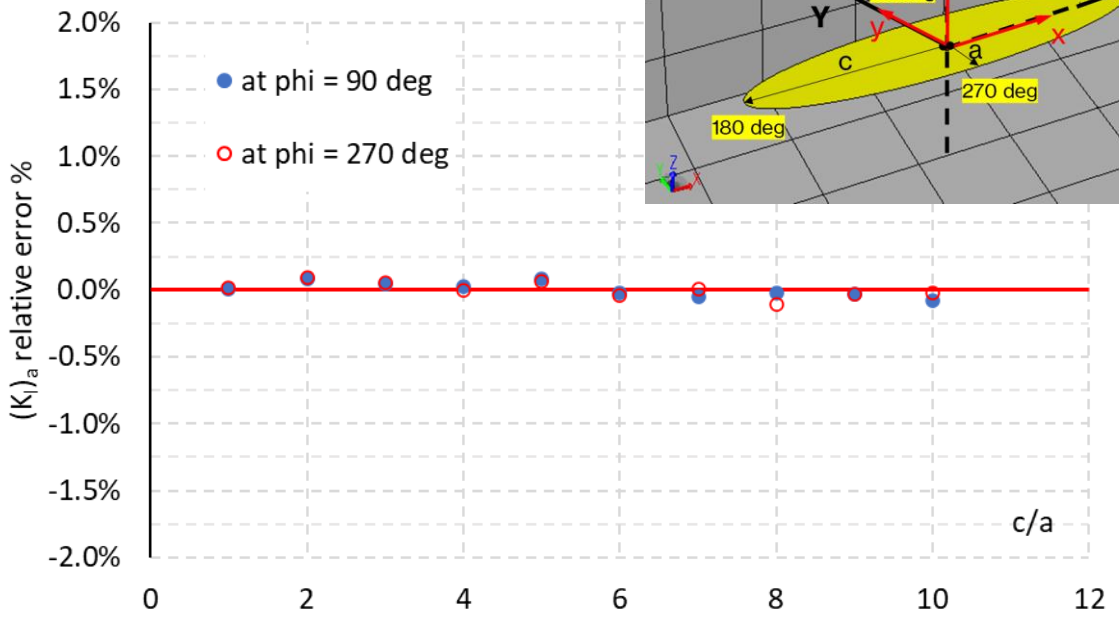
$(K_I)_i$ solutions converge within (min, max)=(-0.26%, -0.1%)
$(K_I)_a$ and $(K_I)_c$ solutions converge within (-0.25%, -0.1%) error bounds

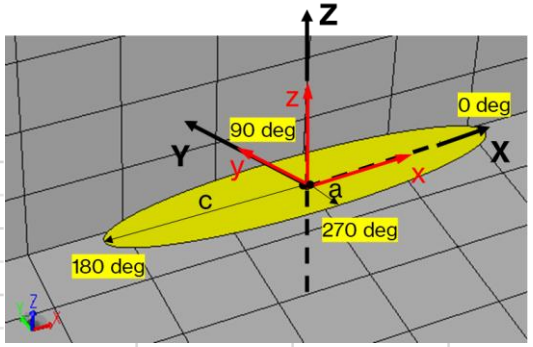
Verification benchmarking against exact Kassir & Sih solution

- **Evaluate $(K_I)_a$ and $(K_I)_c$ numerical error for a set of aspect ratios: c/a = {1, 2, 3, 4, 5, 6, 7, 8, 9, 10}.**
- **Use Python scripting, parallel processing, to obtain all these solutions.**
- **Mesh was automatically adjusted to the curvature of the elliptical crack front shape.**

SimModeler/Python scripting/CalculiX solver







$(K_I)_a$ and $(K_I)_c$ solutions are within (-0.15%, 0.1%) error bounds

Verification benchmarking against exact Kassir & Sih solution

Partial Conclusions:

- **Crack front mesh size assignment based on local curvature allows automatic mesh refinement which leads to a high solution accuracy with minimal user intervention.**

- **Built-in finite element formulations and classical displacement correlation technique are used to compute K_I values along the crack front. Using two solvers, CalculiX and Ansys, K_I solution accuracy and convergence are demonstrated using an exact solution as a reference.**

  - ✓ **The recorded modeling errors provide modeling confidence when the techniques are applied to any other geometry and loading configuration.**

  - ✓ **Practically, any FEA solver can be used in the modeling process since no special element formulation is needed.**

Verification benchmarking against reference solutions [4, 5]

- **A K_I accuracy round robin challenge was held in 2021 [4, 5].**
- **More recently, a K_I surrogate model was trained using a solution database [7]. Accuracy of the surrogate model for five of the verification benchmarks is stated to be higher when compared with other FEM based solutions in the $3° \leq \theta \leq 87°$ range.**
- **A solution accuracy assessment is carried out to determine relative errors between reference solutions in [4, 5] considered best-in-class and, SimModeler solutions that rely on built-in element formulations (available in many FEM tools) and classical displacement correlation technique to compute K_I solutions along the crack front.**

Reference [5]

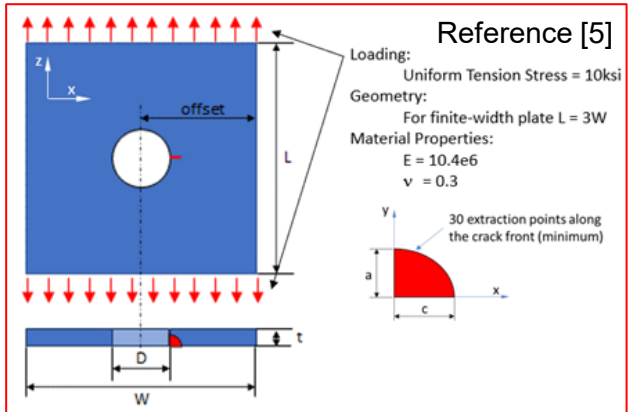


Figure 1: Typical round robin corner crack at a hole configuration

Table 2: Round robin matrix of cases

Case	Surface Crack Length (c) (inch)	Bore Crack Length (a) (inch)	Corner Crack Configuration	Width (inch)	Thickness (inch)	Hole Diameter (inch)	Hole Offset (inch)
1	0.050	0.050	Double Symmetric	100.00	0.25	0.50	50.00
2	0.050	0.050	Single	100.00	0.25	0.50	50.00
3	0.050	0.050	Single	4.00	0.25	0.50	2.00
4	0.050	0.050	Single	4.00	0.25	0.50	0.60
5	0.050	0.050	Single	1.20	0.25	0.50	0.60
6	0.050	0.075	Single	100.00	0.25	0.50	50.00
7	0.100	0.050	Single	100.00	0.25	0.50	50.00

Reference
[5] Pilarczyk R., Engineered Residual Stress Implementation (ERSI), Stress Intensity Comparisons Round Robin, Report number ERSI-2021-01.

Summary of error range in K solutions

7 Case studies identified in a round robin exercise. **Cases 2, 3, 5, 6, 7 are in the Castle database.**
Reference is a set of highly refined hp finite element solutions.†

Case	Other FEM [%]	AFGROW [%]	NASGRO [%]	NR-Solution [%]	UTSA-UTAH ML ($3° \leq \theta \leq 87°$) [%]
1	-2 to 2	~0	0 to 0.5	-1 to 2	-
2	-1 to 2	~0	-0.5 to 0	-0.5 to 2	-0.30 to 0.24
3	-1 to 2	-0.6	-0.5 to -0.2	-1 to 2	-0.20 to 0.04
4	-2 to 2	-4.5 to -2	3 to 3.5	-6 to -3	-
5	-1 to 2	-8 to -7	-2	-8 to -5	-0.10 to 0.04
6	-2 to 2	~0	-2 to 0	-2 to 2	-0.16 to 0.10
7	-2 to 3	~0	~0	-2 to 1	-0.32 to 0.21

Reference
[7] Millwater H., et. al., Machine learning-based development of stress intensity factor solutions for a corner crack at a hole using DeepONet technology, ASIP 2025.

Verification benchmarking against reference solutions [4, 5]

- A more rigorous range of $1° \leq \theta \leq 89°$ is used to report the differences between the two FEA based solutions
- Overall, relative differences are less than abs(0.38%). At this K_I accuracy level, the focus should be on other uncertainties (modeling, experimental) with a larger impact on the crack growth solution that need to be considered in a fatigue crack growth life assessment.
- A multi-variable surrogate model development needs preexisting FEA solutions (thousands to million training points used in [7]) - a large upfront time investment. A β-Table can be generated right away using existing FEA expertise and models. A β-Table solution also has the advantage of using component level representative models which leads to a higher accuracy when compared to generic models that only resemble component cracking locations.

	$3° < \theta < 87°$	$1° < \theta < 89°$				
Condition	USTA-UTAH ML	FEM: SimModeler, CalculiX solver (min error, max error) %	solution runtime (min)	FEM: SimModeler, Ansys solver (min error, max error) %	solution runtime (min)	DoF
1	-	(0.00%, 0.21%)	6.5	(0.14%, 0.35%)	4.5	4.4E+06
2	(-0.30%, 0.24%)	(-0.02%, 0.16%)	4	(0.03%, 0.19%)	3	2.3E+06
3	(-0.20%, 0.04%)	(-0.04%, 0.19%)	4	(0.02%, 0.23%)	3	2.3E+06
4	-	(-0.19%, 0.21%)	2	(-0.13%, 0.25%)	2	1.4E+06
5	(-0.10%, 0.04%)	(-0.25%, 0.24%)	1	(-0.09%, 0.38%)	1	7.2E+05
6	(-0.16%, 0.10%)	(-0.01%, 0.30%)	6	(0.04%, 0.35%)	4	4.1E+06
7	(-0.32%, 0.21%)	(-0.19%, 0.29%)	6	(-0.15%, 0.33%)	4	4.1E+06

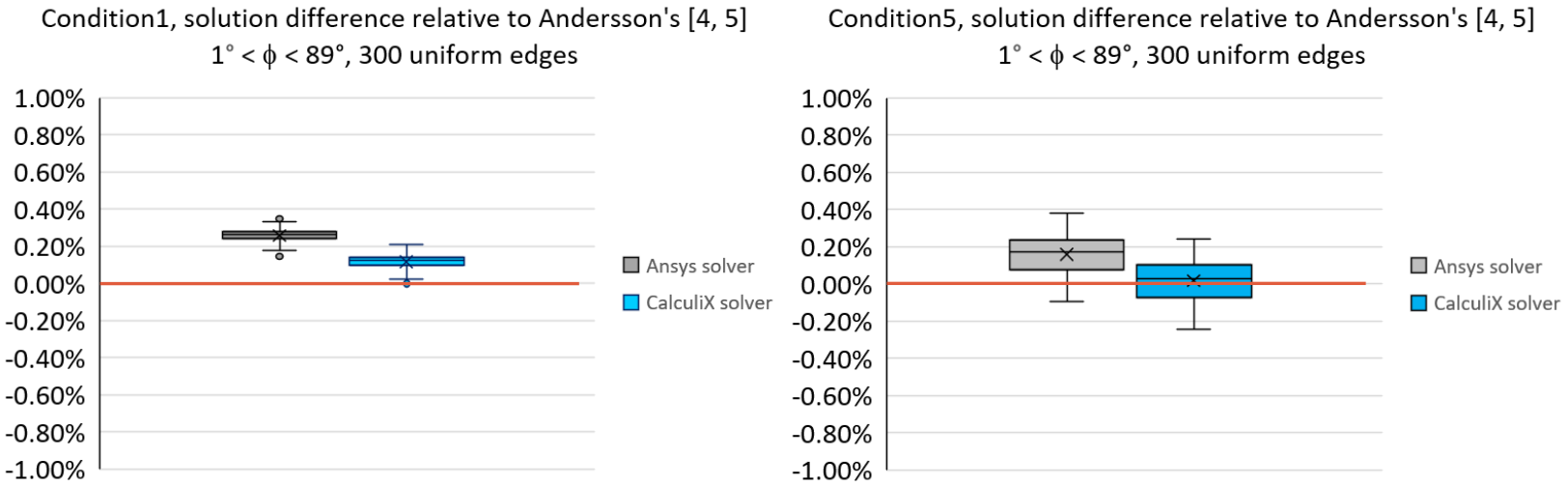
The same mesh was used to solve the model in Ansys and CalculiX

For a typical and effortless 300 element edges along the crack front, (K_I) solutions are within (-0.25%, 0.38%) relative difference bounds.

Verification benchmarking against reference solutions [4, 5]

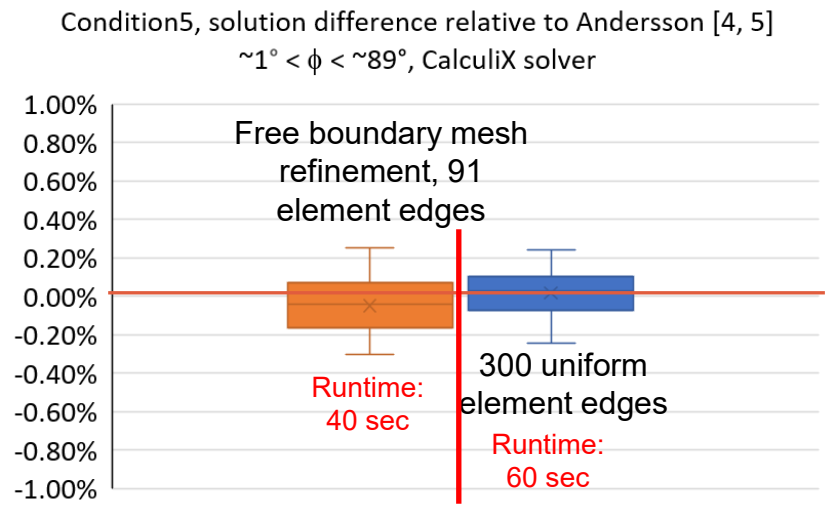
- **Relative solution differences using "condition1" and "condition5" benchmarks are reported. For the other cases, similar conclusion was reached.**
- **For "condition5" case, two solutions are compared: from a uniform mesh and, using a refinement at the free boundary along crack front edge. Similar solution accuracy is obtained.**

Uniform mesh along crack front, 300 element edges



The same mesh was used to solve the model in Ansys and CalculiX

Solution comparison between a uniform mesh and free boundary mesh refinement along crack front



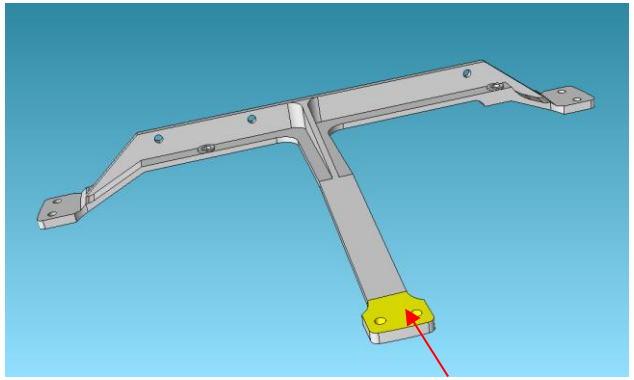
These results provide modeling confidence when the techniques are applied to any other geometry and loading configuration.

β-Table generation for AFGROW: component level example

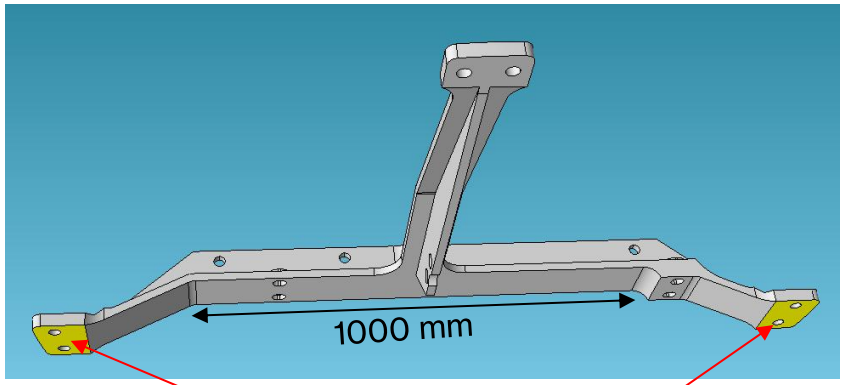
- The β-Table solution generation is demonstrated on a generic model. The same modeling process is applicable for any other geometry and loading configuration (Mode I only).
- FEA solution of the undamaged structure is used to identify location and orientation of the crack that will be considered for β-Table solution generation.
- Two FEA solvers were considered in this β-Table solution generation example: Ansys and CalculiX.
- β-Table solution generation is fully automatic.
- β-Table solution is verified against a multi-DoF solution (crack front shape is not constrained to an elliptical representation).

Generic bracket example: model definition

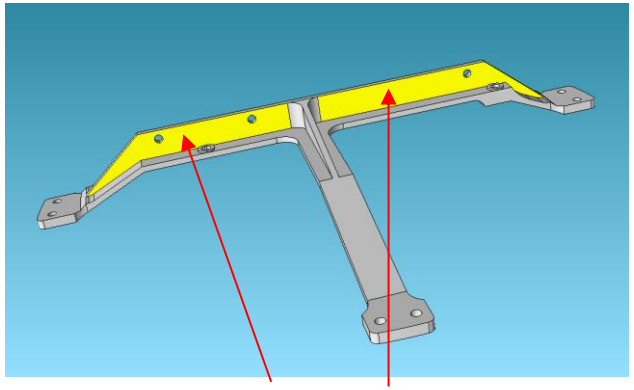
- **A 3D Parasolid representation of a bracket is used in this demonstration, mesh and FE Model completely defined in SimModeler, solver: CalculiX or Ansys.**
- **E = 21e4 MPa, ν = 0.3, Paris relationship used in the assessment, C = 6e-12, n = 2.5 (FE model units: N, mm).**
- **Constant amplitude loading, R = 0.**



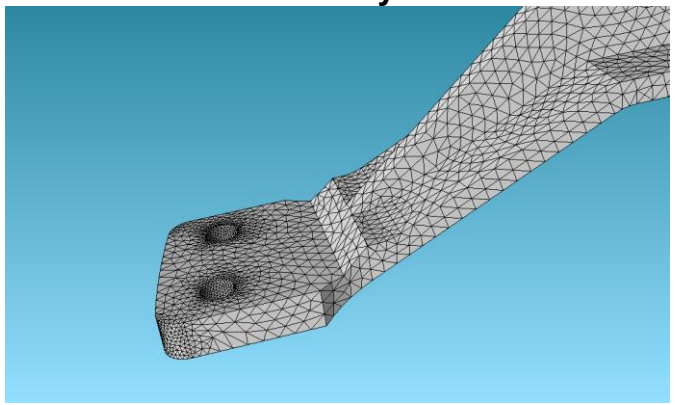
End of the arm is fully constrained



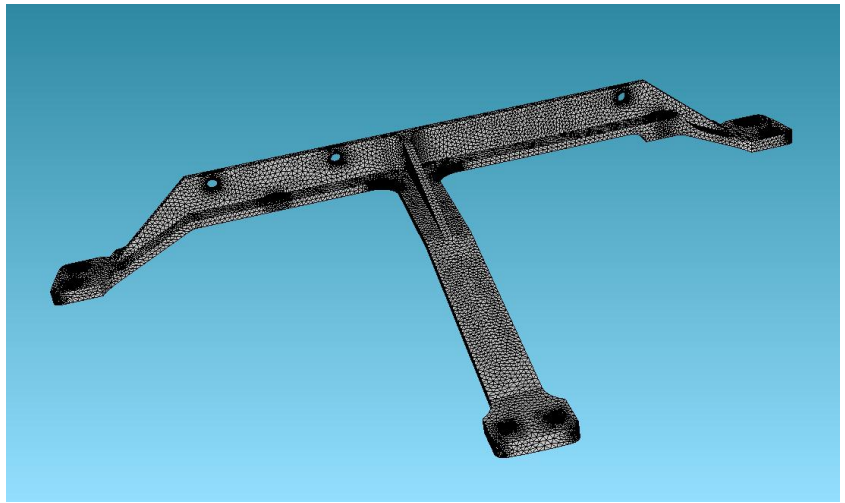
Simple support for the two other arms



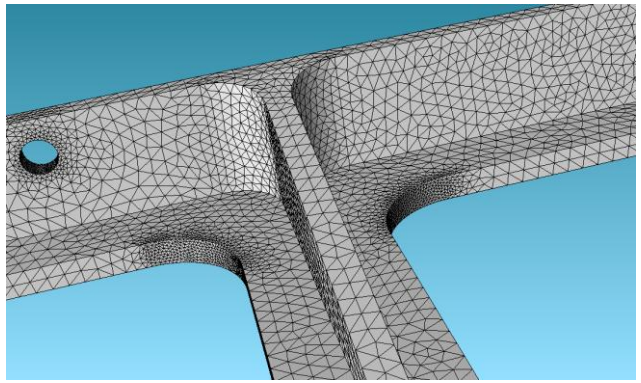
Distributed load of 0.5 MPa



Mesh detail



Overall mesh used in the simulation

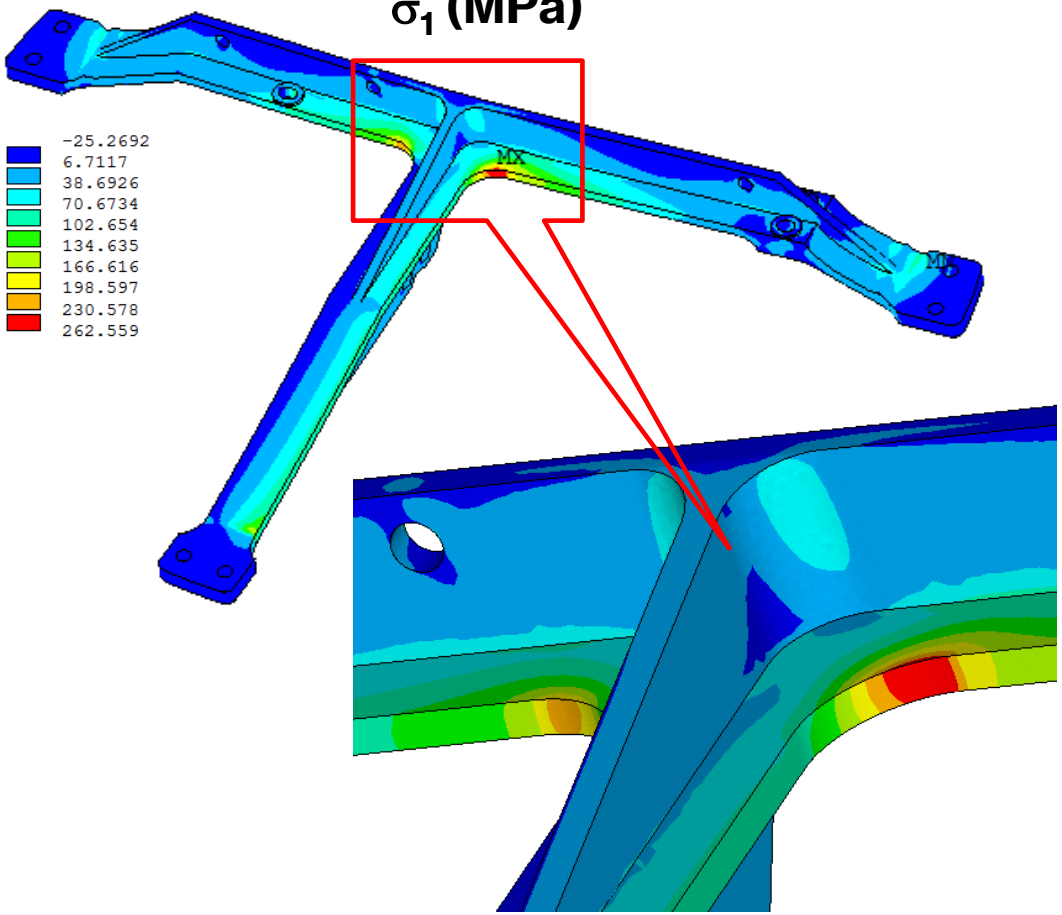


Mesh refinement at the location where stress concentration is expected

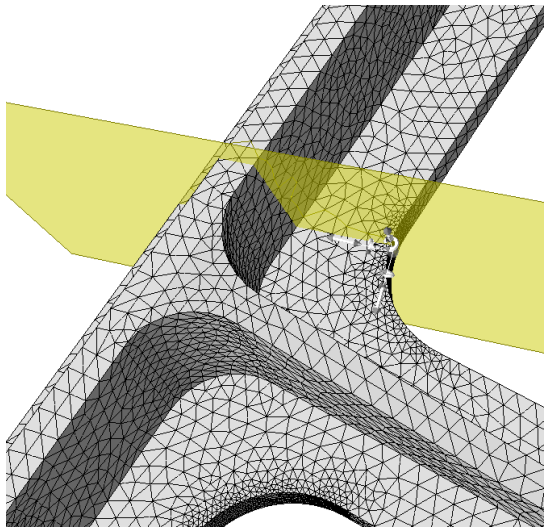
Generic bracket example: model definition

- **Structural analysis of the bracket is used to identify stress concentration areas prone to crack initiation.**
- **A crack is defined in SimModeler using location and orientation of max(σ_1) from post-processing initial structural analysis solution performed in Ansys (uncracked structure).**

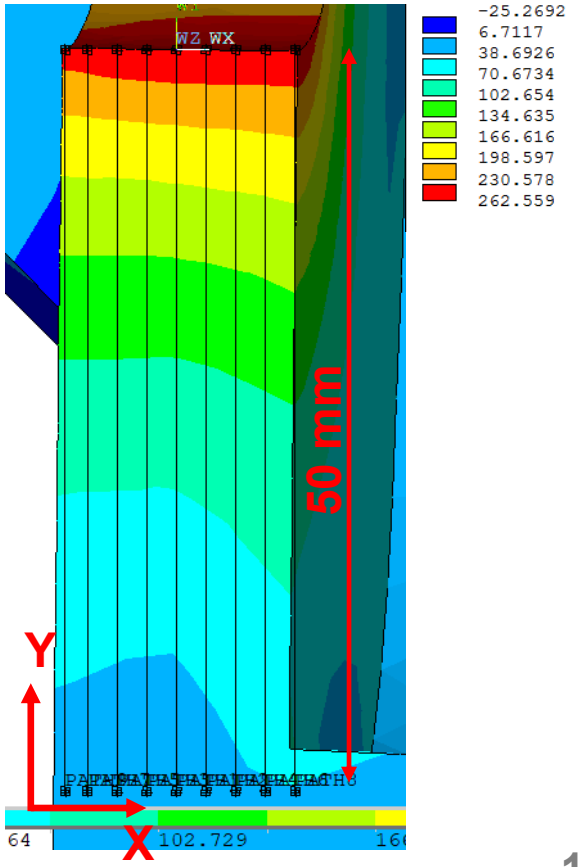
σ_1 (MPa)



Crack plane definition

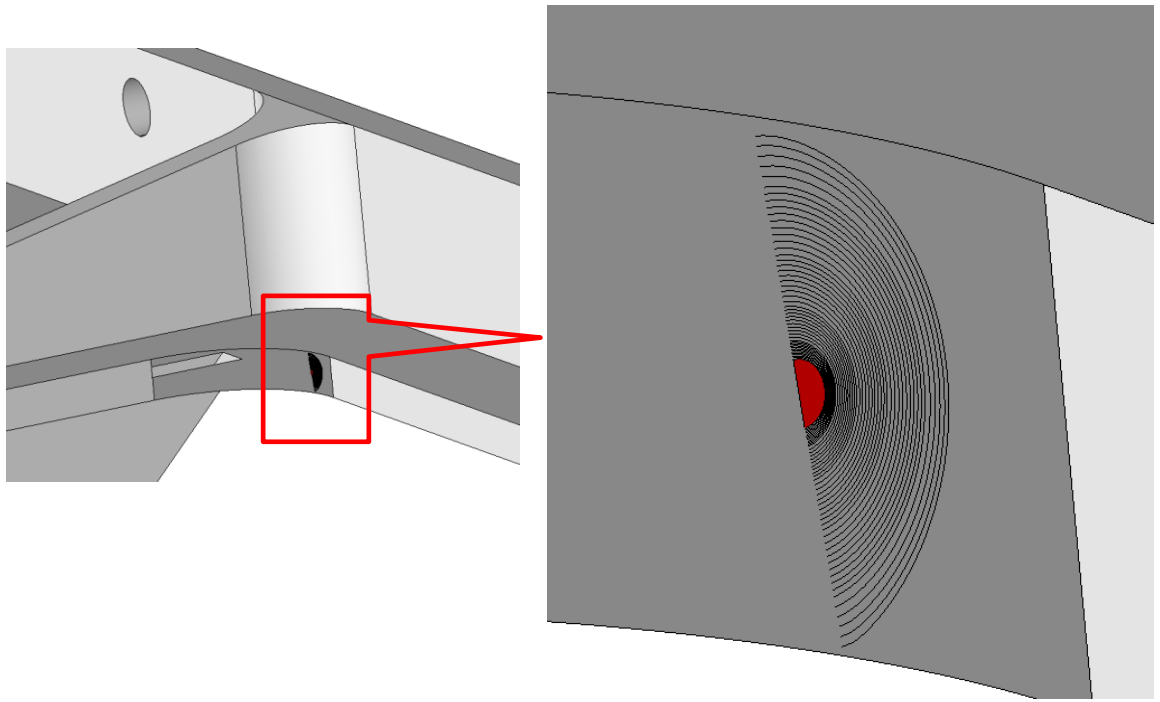


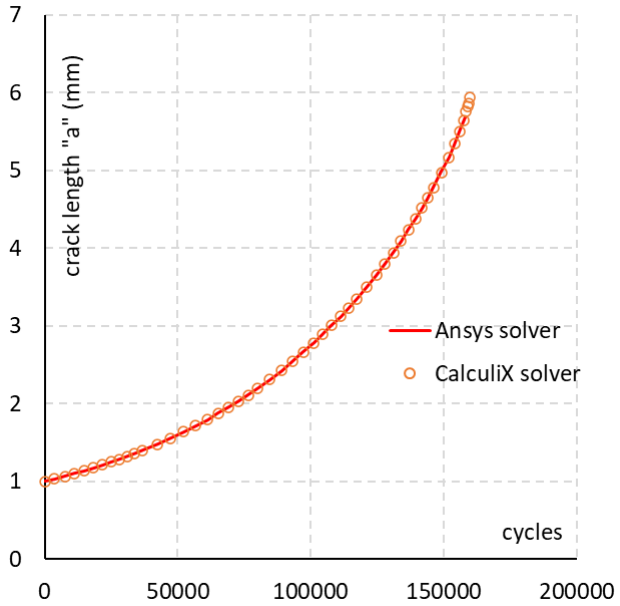
Normal stress gradient acting on the crack plane (σ_z in local CSYS)

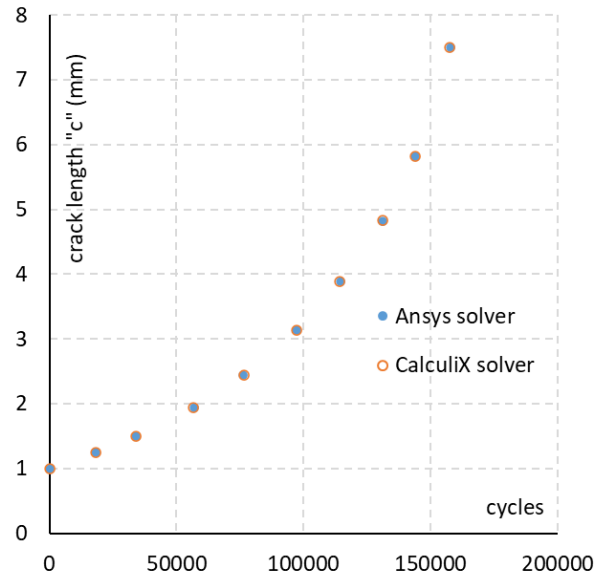


Multi-DoF solution: generic bracket example

- **First solution is generated using a multi-DoF (crack front shape is not constrained to an elliptical shape), component level FEA using two solvers: Ansys and CalculiX.**
- **This solution serves as reference for the β-Table solution (for verification purposes).**





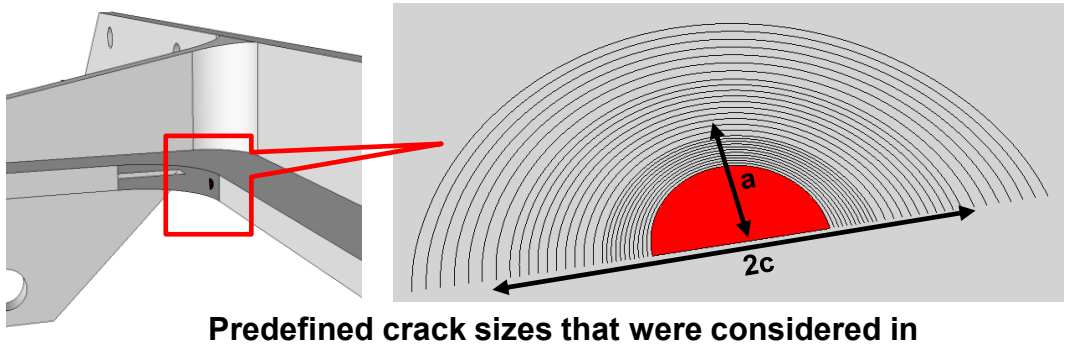


No difference between the two multi-DoF solutions: Ansys or CalculiX -based solvers

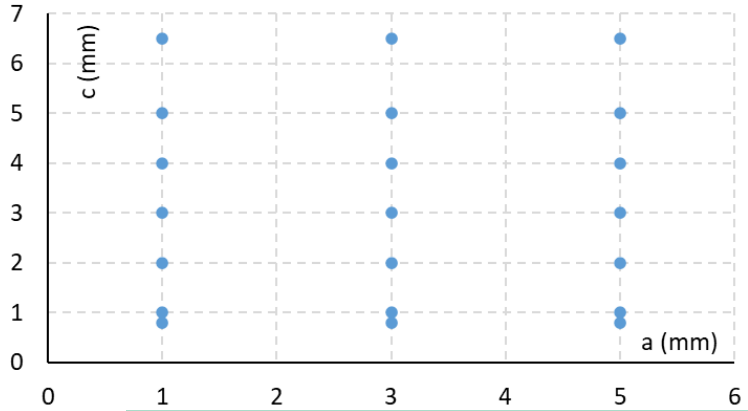
β-Table generation for AFGROW: generic bracket example

- **The model definition from the multi-DoF solution can be used right away in the β-Table solution generation.**
- **All modeling steps are automatic: pre-processing, solution and post-processing for all predefined cracks.**
- **These solutions can be performed in parallel; each solution can be caried out in a multi-threaded mode.**
- **Solver: CalculiX (for this example).**

Definition of the 2-DoF used in the β-Table solution



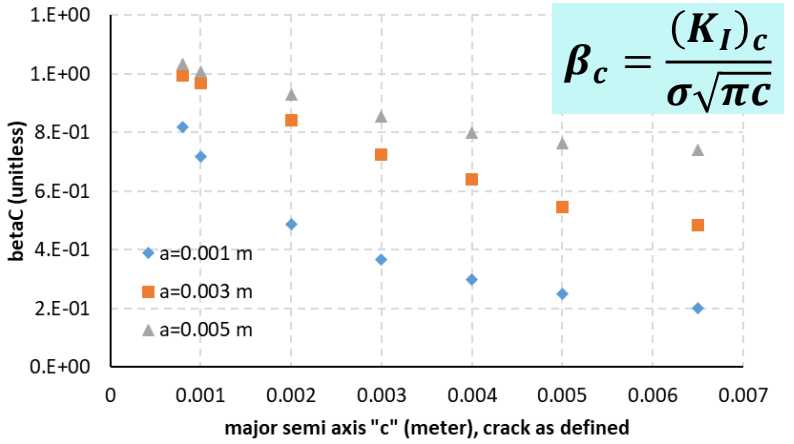
Predefined crack sizes that were considered in the β-Table solution generation

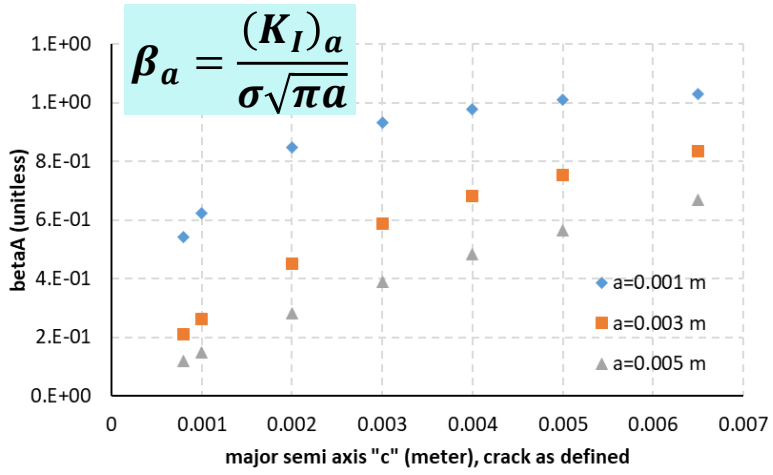


BetaA		alength1	alength2	alength3
		0.0010	0.0030	0.0050
clength1	0.0008	0.5417	0.2110	0.1181
clength2	0.0010	0.6240	0.2613	0.1480
clength3	0.002	0.8471	0.4512	0.2800
clength4	0.0030	0.9327	0.5889	0.3887
clength5	0.004	0.9788	0.6831	0.4828
clength6	0.0050	1.0104	0.7535	0.5640
clength7	0.0065	1.0304	0.8353	0.6684

BetaC		alength1	alength2	alength3
		0.0010	0.0030	0.0050
clength1	0.0008	0.8193	0.9952	1.0336
clength2	0.0010	0.7193	0.9695	1.0064
clength3	0.002	0.4855	0.8409	0.9291
clength4	0.0030	0.3682	0.7234	0.8560
clength5	0.004	0.2986	0.6412	0.8001
clength6	0.0050	0.2492	0.5447	0.7642
clength7	0.0065	0.2006	0.4838	0.7419

BetaC1		alength1	alength2	alength3
		0.0010	0.0030	0.0050
clength1	0.0008	0.7753	0.9937	1.0388
clength2	0.0010	0.7345	0.9500	1.0161
clength3	0.0020	0.4828	0.8245	0.9343
clength4	0.0030	0.3705	0.7135	0.8493
clength5	0.0040	0.2924	0.6415	0.7910
clength6	0.0050	0.2413	0.5373	0.7614
clength7	0.0065	0.1957	0.4830	0.7560



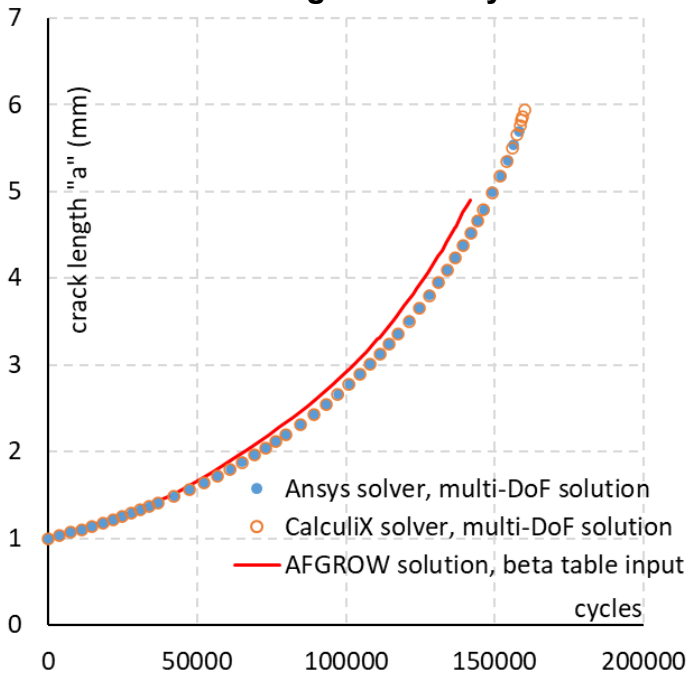


These 3D FEM based K_I solutions were used as an input in AFGROW for a crack propagation solution

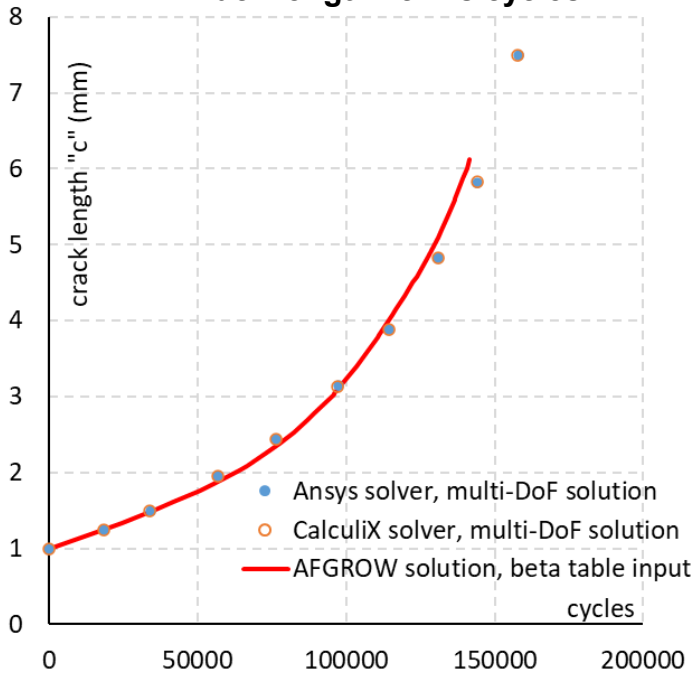
β-Table generation for AFGROW: generic bracket example

- The β-Table was used as an input for an AFGROW based solution consistent with the model definition.
- β-Table based solution was trimmed to the predefined solution space.

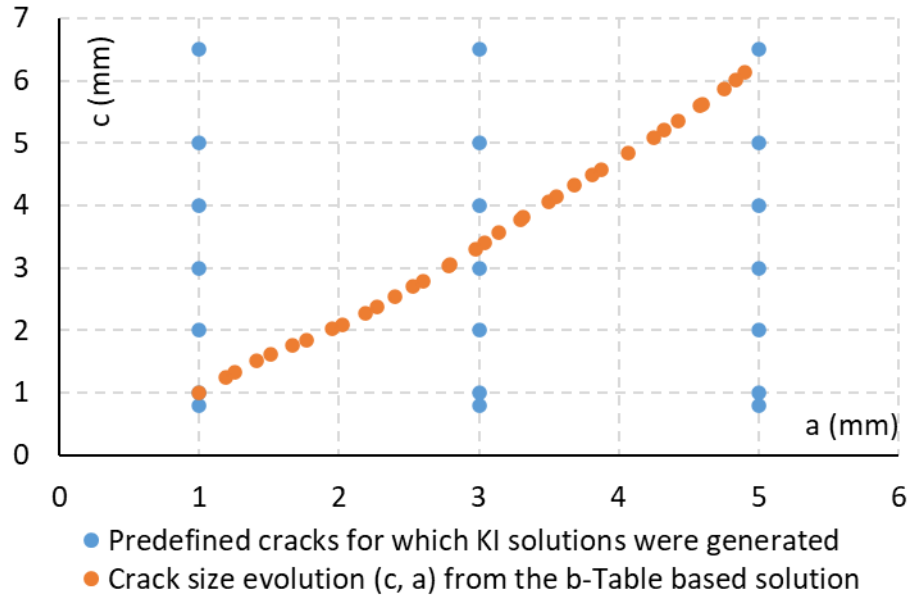
Crack length "a" vs cycles



Crack length "c" vs cycles



Predefined crack sizes along with the (a, c) values generated in the β-Table based solution



Runtime multi-DoF automatic solution (Ansys solver): 1.5 min/step, 50 steps. Total runtime: 75 min.
Runtime multi-DoF automatic solution (CalculiX solver): 2 min/step (4 CPU), 50 steps. Total runtime: 100 min.

Runtime β-Table generation (21 jobs, 3 jobs in parallel, 4-CPU, CalculiX): 30 min

Conclusions

- ✓ **A β-Table generation FEM modeling procedure is accessible to AFGROW via Python scripting. The modeling process is not restrictive to predefined geometries and loading conditions.**
- ✓ **K_I solution convergence and accuracy (verification benchmarking) is demonstrated using one exact 3D solution and 7 semi-analytical 3D solutions.**
- ✓ **An β-Table generation example is provided along with the AFGROW fatigue crack growth solution and a comparison against a multi-DoF solution.**
- ✓ **Based on the verification benchmarks, modeling confidence is reached. Many other benchmarks and examples can be created.**
- ✓ **For the level of errors and solution differences reported herein, it can be stated that this modeling process is quite accurate without any need to further investigate how a lower error can be achieved. The overall fatigue crack growth modeling process has many other sources of uncertainty with a larger impact on accuracy.**