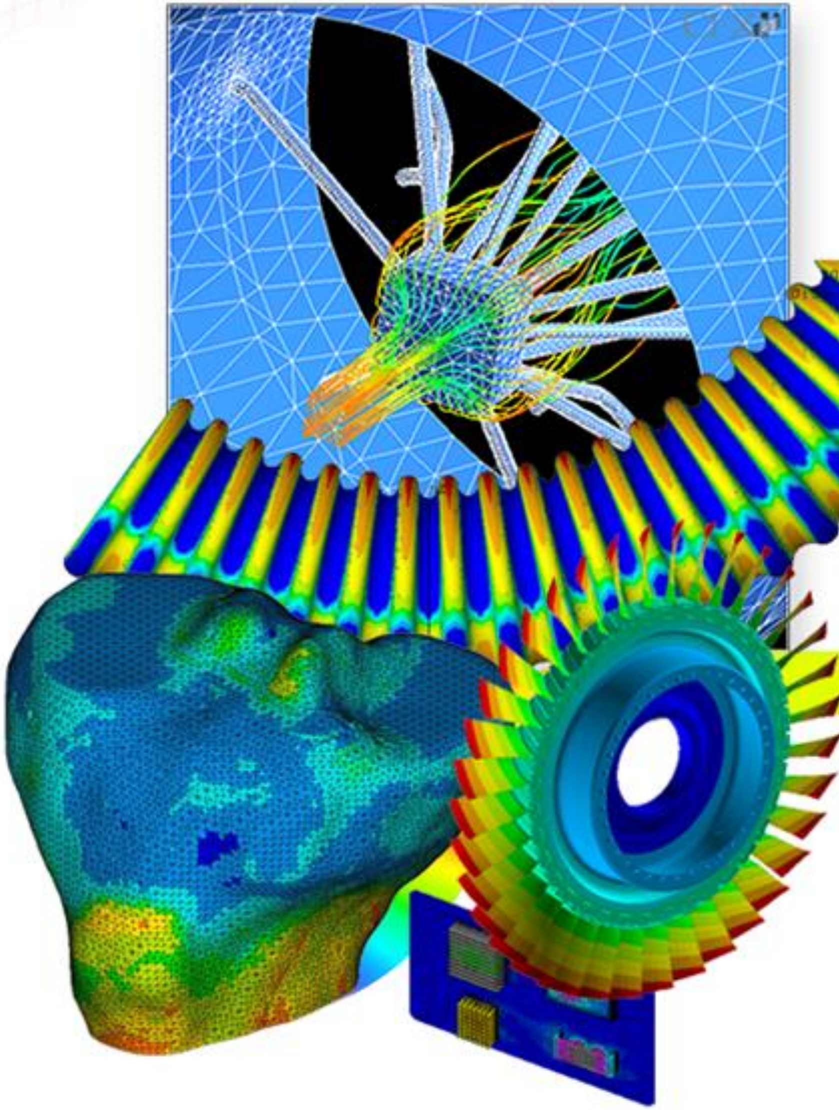
CAE ASSOCIATES

ENGINEERING ADVANTAGE

Mechanical APDL New Capabilities

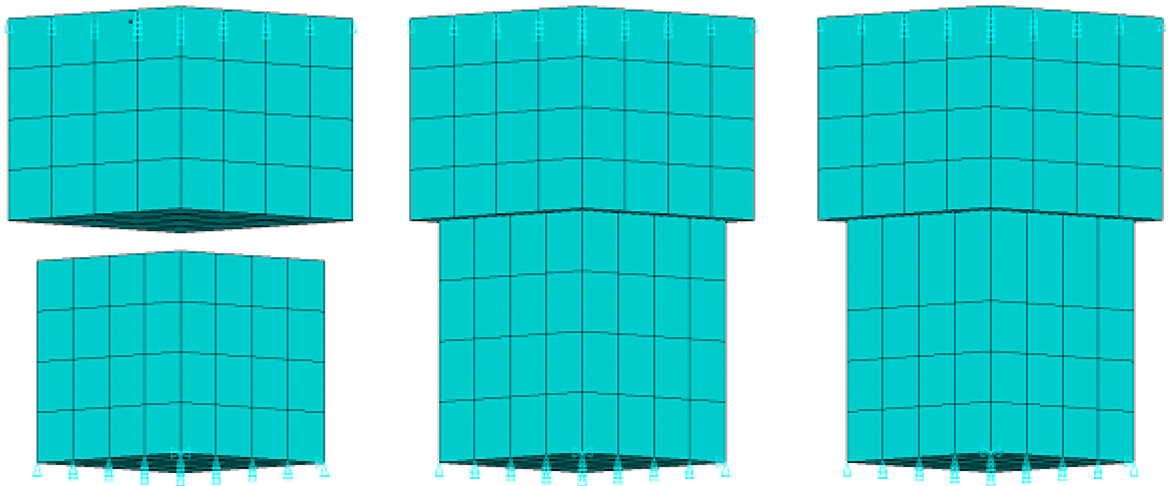


Mike Bak

3/2016

New Enhancements in V17.0

- Enhanced initial interference adjustment with mesh morphing.
  - **CNCHECK,MORPH**



Initial mesh with gap

CNCHECK,MORPH

CNCHECK,ADJUST - solid elements at contact interface are stretched

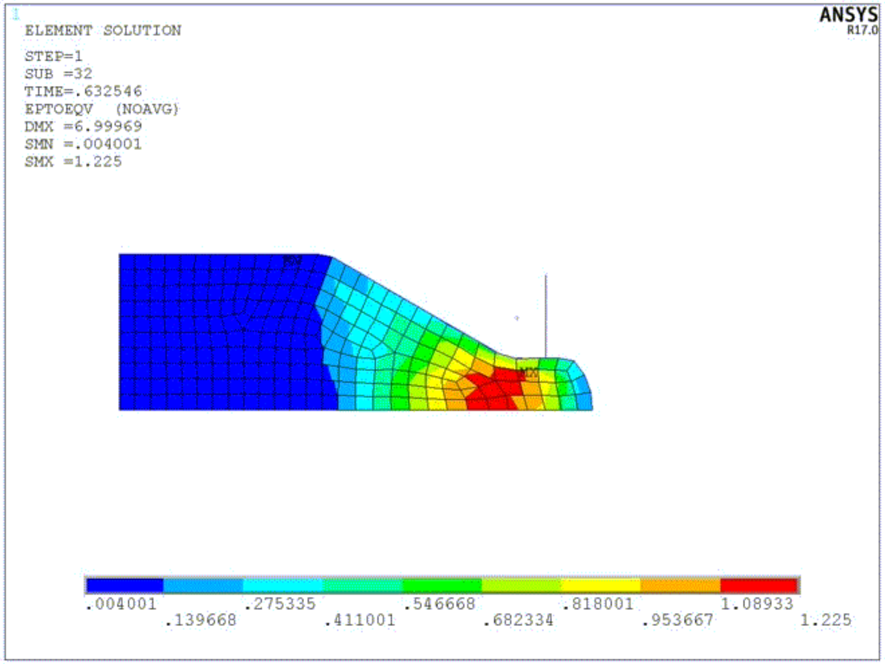
New Enhancements in V17.0

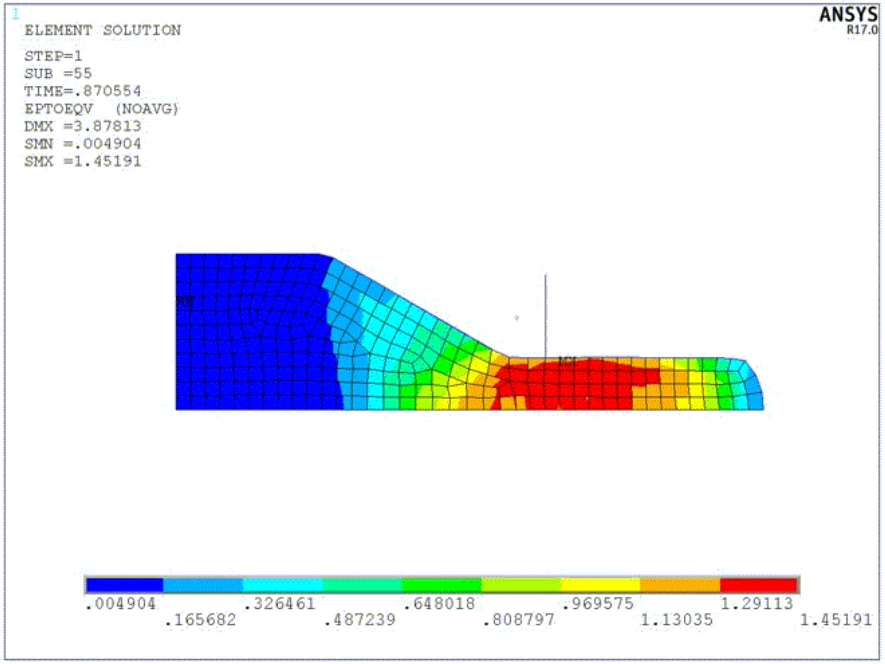
- Thermal contact modeling for thermal shells.
  - The 3D surface to surface contact elements can be used to model thermal contact at the surface of thermal shells SHELL131/SHELL132.
  - TEMP, TBOT, TTOP temperature degrees of freedom can be specified.

KEYOPT(13)	DOF on Contact Surface	DOF on Target Surface
0	TEMP	TEMP
1	TBOT	TBOT
2	TTOP	TTOP
3	TBOT	TEMP
4	TEMP	TBOT
5	TTOP	TEMP
6	TEMP	TTOP
7	TBOT	TTOP
8	TTOP	TBOT

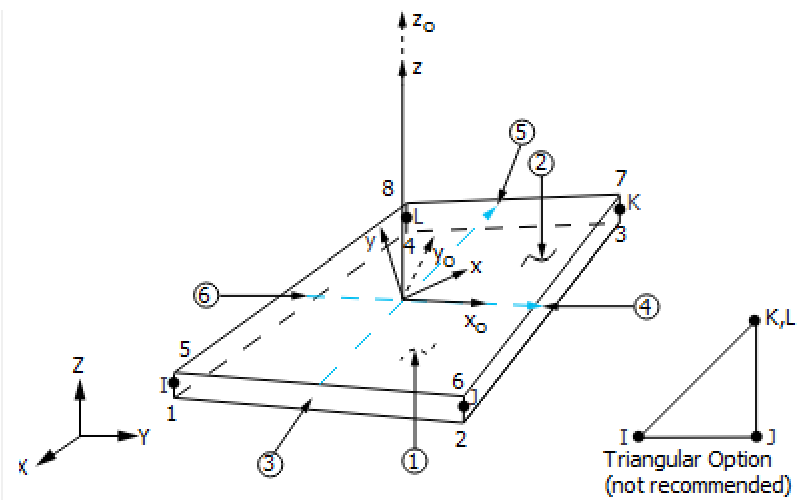
New Enhancements in V17.0

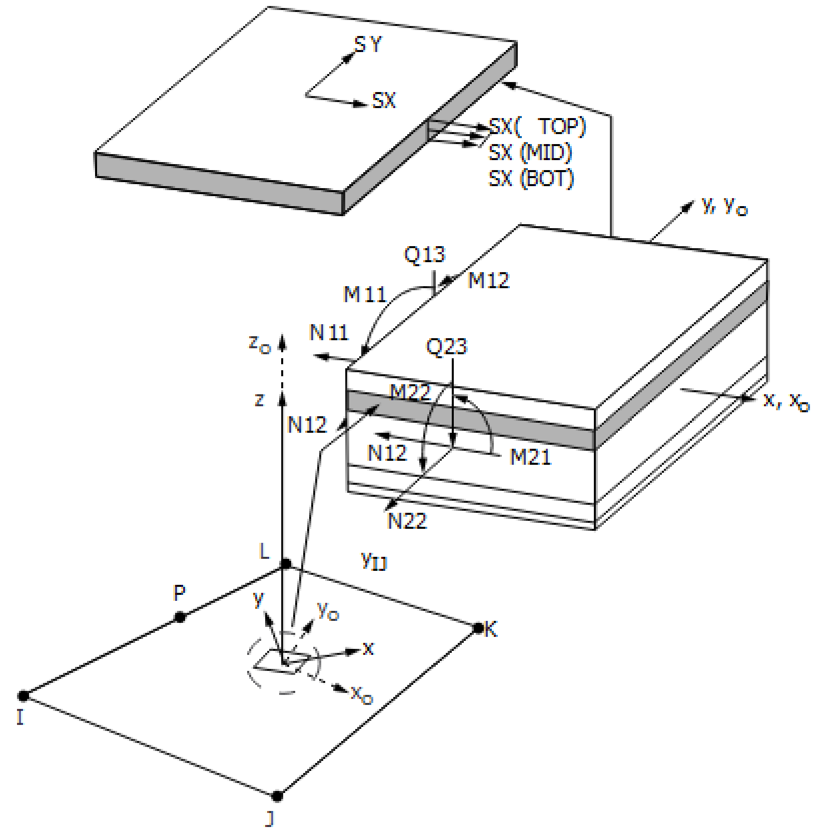
- Improved mesh quality when remeshing in a mesh nonlinear adaptivity analysis.
  - Completely automatic (unlike rezoning).
  - Available for 2D and 3D.
  - Improved mesh quality during solution in v17.0





- Out-of-plane normal stress recovery in shells.
  - New option using KEYOPT(10) for SHELL181 and SHELL281.
  - Outputs normal stress component SZ (Default SZ = 0 per Kirchoff).
  - Independently recovered during the element solution output from the applied pressure load.





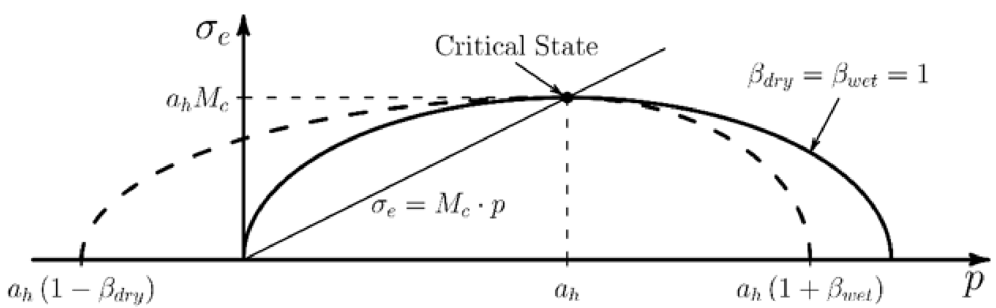
New Enhancements in V17.0

- Mixed u-P formulation available in USERMAT.
  - TBOPT = MXUP on the **TB,USER** command.
  - *Strain* array is the logarithmic strains at the current time.
    - *Strain* array can be redefined within USERMAT.
  - *dStrain* array is zero.
  - *Stress* array is updated for deviatoric part of stress only.
    - Calculated P is passed into USERMAT as *Stress(ncomp+1)*.
  - Only the deviatoric material consistent tangent matrix is output in array *dsdepl*.

- Useful for programming hyperelastic/incompressible material laws.

New Enhancements in V17.0

- New geomechanics capabilities.
  - New soil analysis including geostatic stress equilibrium or consolidation.
    - Valid for structural or fluid-pore-pressure degrees of freedom.
  - New porous elasticity model to represent soils or polymer foams displaying nonlinear elastic behavior caused by the effect of voids.
  - New Cam-clay model.
  - New Mohr-Coulomb model to represent aggregate materials.
  - New jointed rock material model to represent geologic and aggregate materials with inhomogeneous behavior.
  - New Drucker-Prager concrete model to represent large differences in tensile and compressive behavior of concrete.
    - Can also be combined with Mohr-Coulomb to define joints in concrete.



Modified Cam-clay Yield Surface in the q-p Plane

New Enhancements in V17.0

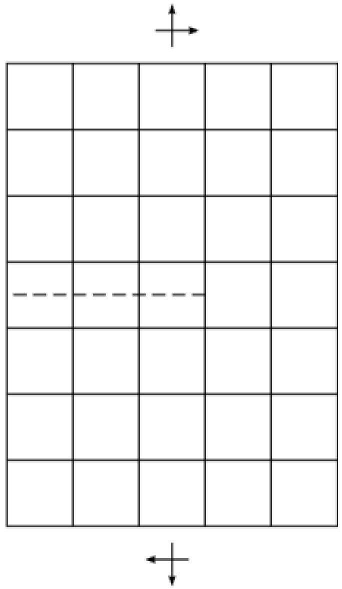
- Remote modal files usage for spectrum analyses.
  - Remote read-only modal files usage is now supported.
  - Especially useful when running several spectrum analyses based on the same modal analysis.
  - New **MODDIR** command.

New Enhancements in V17.0

- User-defined thermal material enhancements.
  - User-defined thermal material subroutine: USERMATTH.
  - The user-defined thermal material is now available for the coupled thermal-fluid pipe element, FLUID116.
  - State variables for layered thermal solid elements can now be stored for all layers.

XFEM-Based Crack Growth Simulation

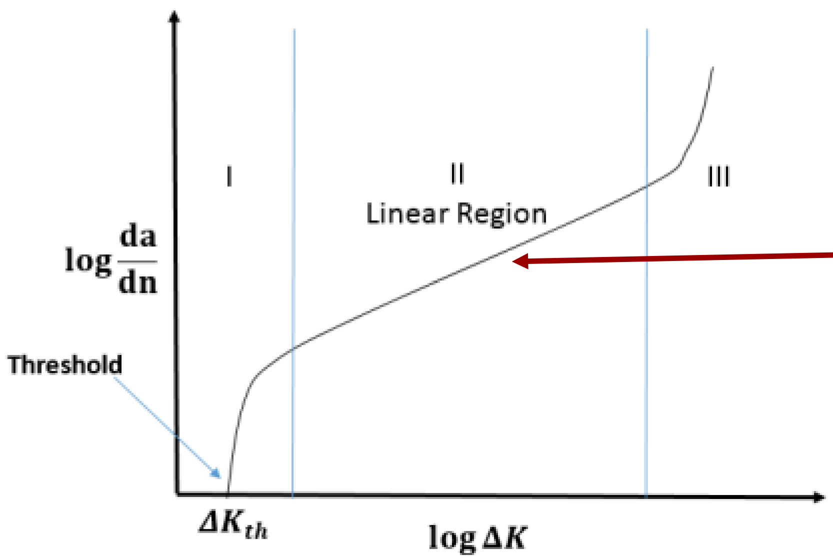
- The e**X**tended **F**inite **E**lement **M**ethod (XFEM):
  - Used to model crack-growth simulations.
  - Eliminates the necessity of re-meshing crack tip regions.
  - Enriches the DOFs in the model with additional displacement functions that account for the jump in displacements across discontinuities.



New Features in ANSYS XFEM in V17.0

- XFEM-based crack growth simulation can now be used to simulate fatigue crack growth.
  - Uses the new singularity-based XFEM.
  - Based on Paris Law.
  - Supports 2D fatigue crack growth simulation.
    - PLANE182 with B-bar integration only.
  - Linear elastic isotropic materials only.
  - Ignores large deflection and finite rotation effects, crack tip plasticity effects, and crack tip closure and compression effects.

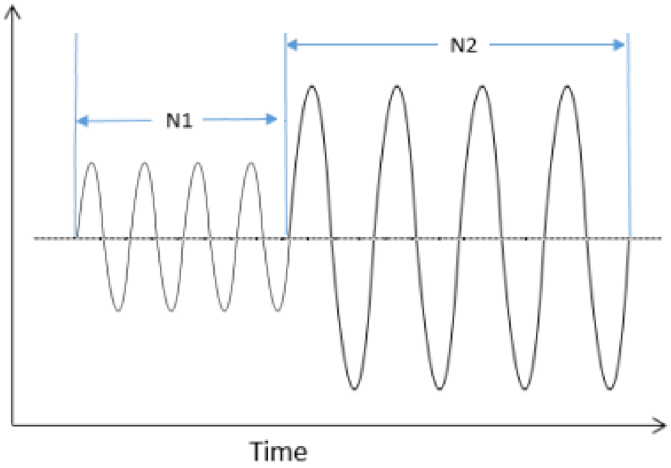
- Classic linear elastic fracture mechanics (LEFM) concepts are used to model fatigue crack growth in Mechanical APDL.
  - Region II of the crack growth rate versus ΔK plot, which is typically described by the Paris Law, is simulated.
    - a = crack length
    - n = number of cycles
    - $\Delta K = K_{max} - K_{min}$, where K's are stress intensity factors.



Paris Law: $$\frac{da}{dn} = C(\Delta K)^m$$

Cyclic Fatigue Loading

- Cyclic loadings of constant amplitudes are allowed.
  - Each set of n cycles with the same load amplitude should be modeled separately as a load step.
    - User applies maximum condition and specifies R-ratio (min stress/max stress).
  - Loading is assumed step applied (KBC,1).
  - Each substep yields a solution where the solution parameters are calculated and the crack is propagated. Fixed time-stepping is recommended.

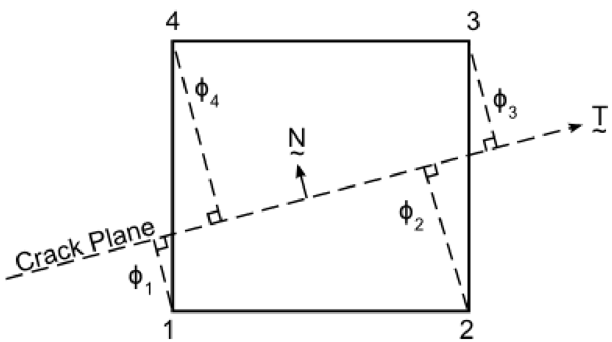


Fatigue Crack Growth Methods

- Two fatigue crack growth methods are available:
  - Life Cycle (LC) method:
    - Used with constant amplitude cyclic loads.
    - User defines crack extension increment da, and number of cycles dn is calculated in each step.
    - Crack will propagate one element at a time.
  - Cycle by Cycle (CBC) method:
    - Suitable for variable amplitude cyclic loadings and overload simulations.
    - User defines incremental cycles dn, and the crack extension increment da is calculated in each step.
    - Crack will adjust to end incremental propagation at element edge.
  - Both approaches assume the crack is straight within an element.
    - The crack propagation angle is constant until an element if fully cut.

Defining Initial Crack

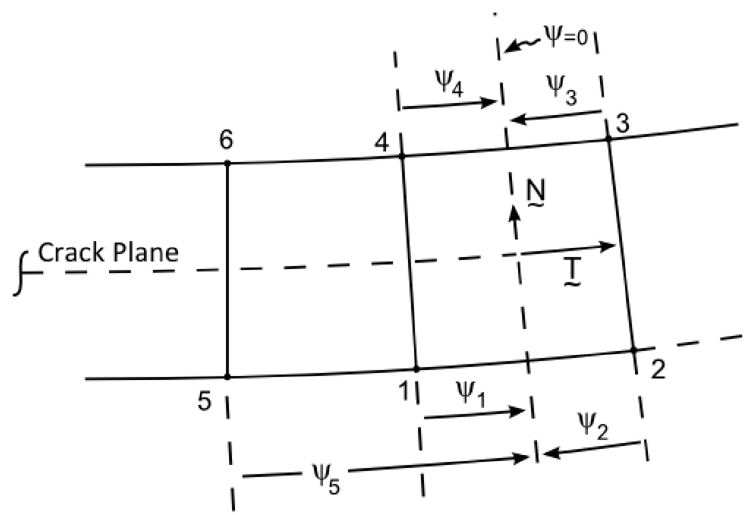
- The level set method is used to define the location of the crack in the model.
  - Crack defined by specifying signed distance functions from the crack surface and from the crack front.
  - Must define ϕ and ψ for each node that the initial crack runs through.
  - Best achieved by creating a local coordinate system at crack tip and obtaining values in that coordinate system.



N = Normal to crack plane

T = Tangent to crack plane

$\phi_1, \phi_2, \phi_3, \phi_4$ = Signed normal distances of nodes distances of nodes 1, 2, 3, 4 from the crack plane



T = Tangent to crack plane

N = Normal to crack plane

$\psi_1, \psi_2, \psi_3, \psi_{etc.}$ = Signed normal distance of nodes 1, 2, 3, etc. from the ψ =0 plane

Fatigue Crack Growth Post-Processing

- The XFEM crack growth procedure will extend the crack in each substep of loading.

- The singularity-based XFEM will calculate the ΔK at each crack increment, along with the length of the crack increment Δa.

- The Paris Law is used to solve for the number of cycles Δn to propagate the crack in each increment.

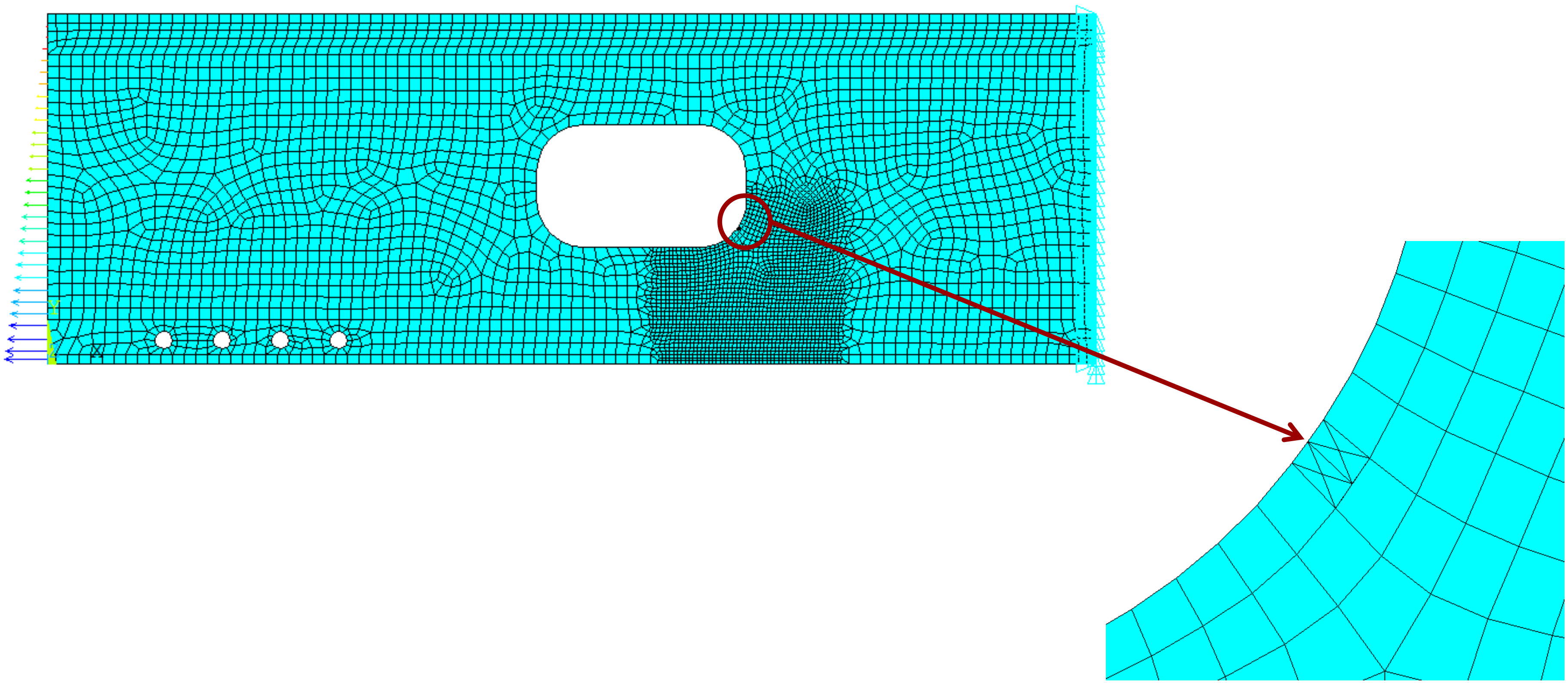
$$\frac{da}{dn} = C(\Delta K)^m$$

$$\frac{\Delta a}{\Delta n} = C(\Delta K)^m$$

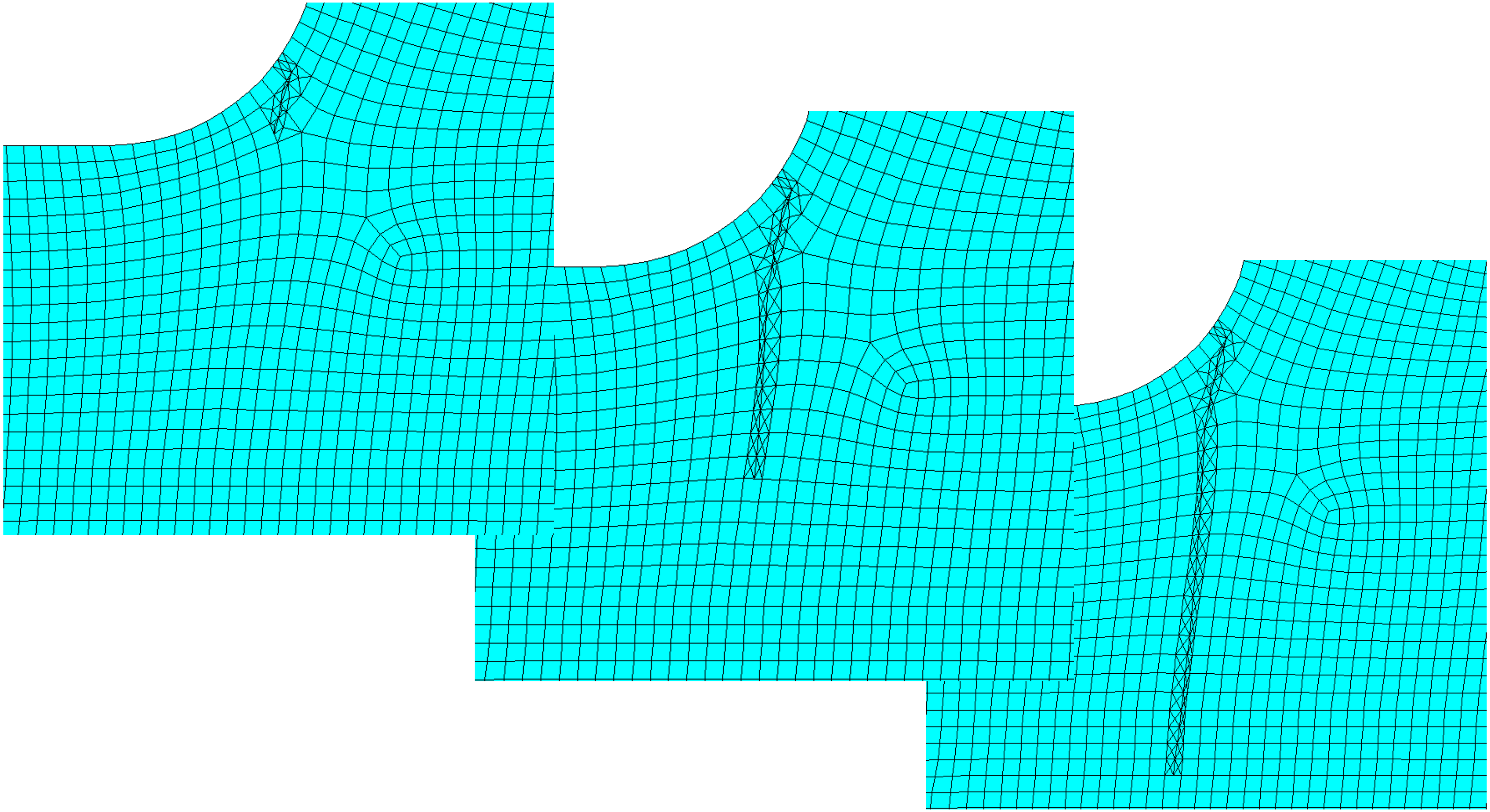
$$\Delta n = \frac{\Delta a}{C(\Delta K)^m}$$

Example Problem

- Example problem of a thin plane stress structure with a 45-degree initial crack at the large hole.
  - Finer mesh in expected region of crack propagation.
  - Initial crack defined by level set method in one element.
  - Cyclic loading assumes R-ratio = 0.

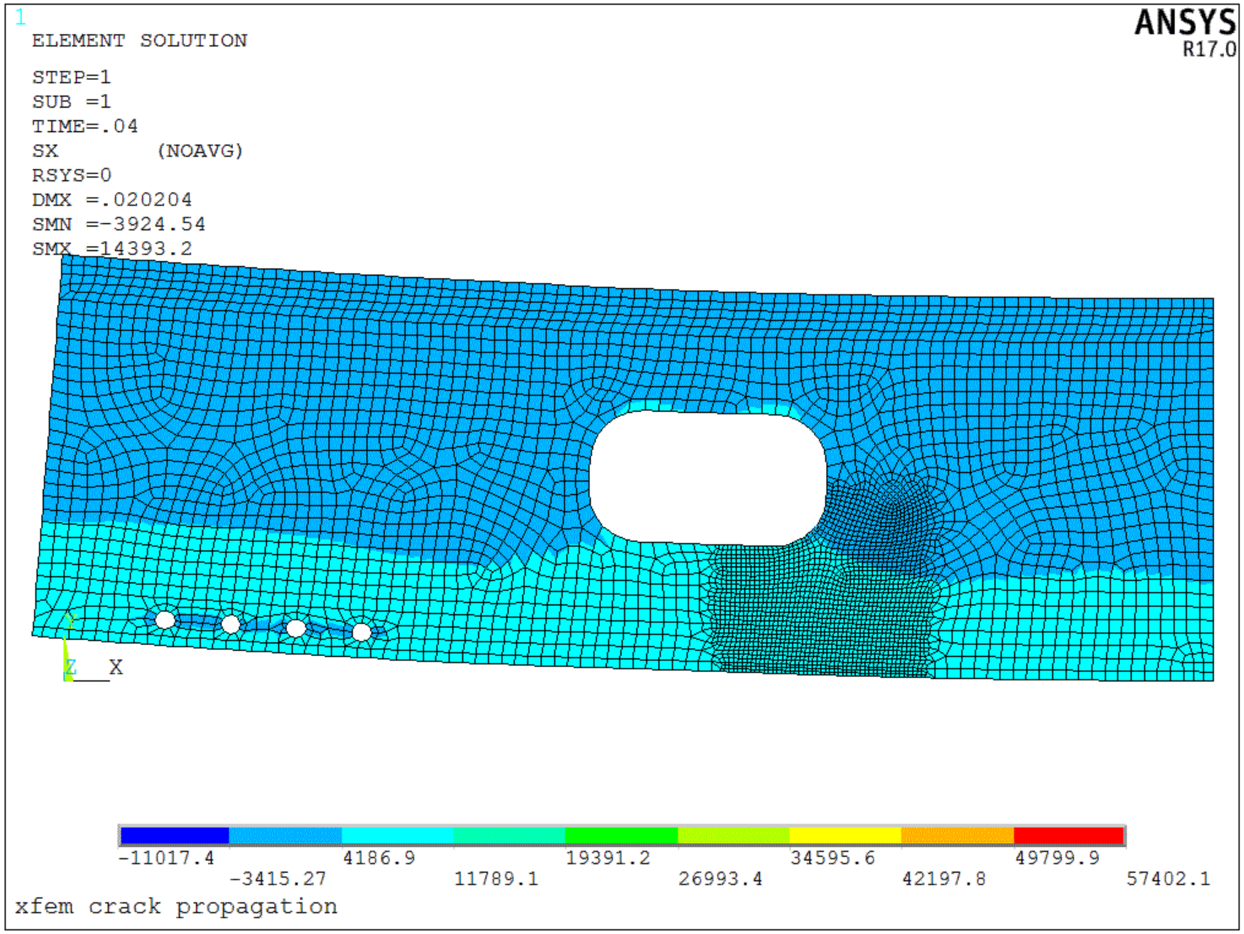


- Crack visualization is shown on mesh plot if miscellaneous items are written to the results file (OUTRES,ALL,ALL).



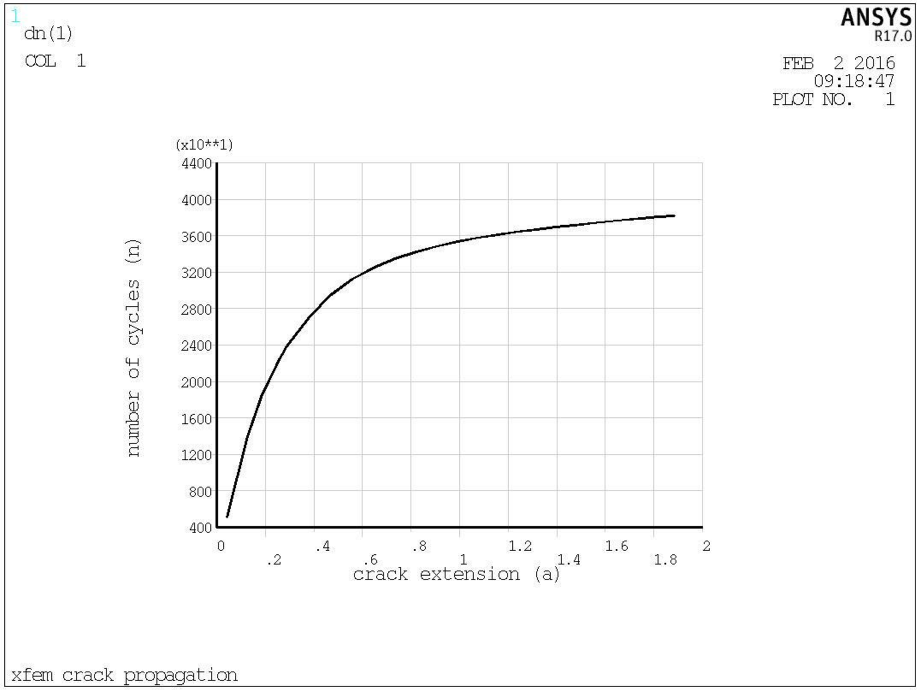
Example Problem

- All results quantities are available as usual.
  - Animation of SX shows presence of crack.
  - The crack tip singularity appears due to the singularity-based XFEM.

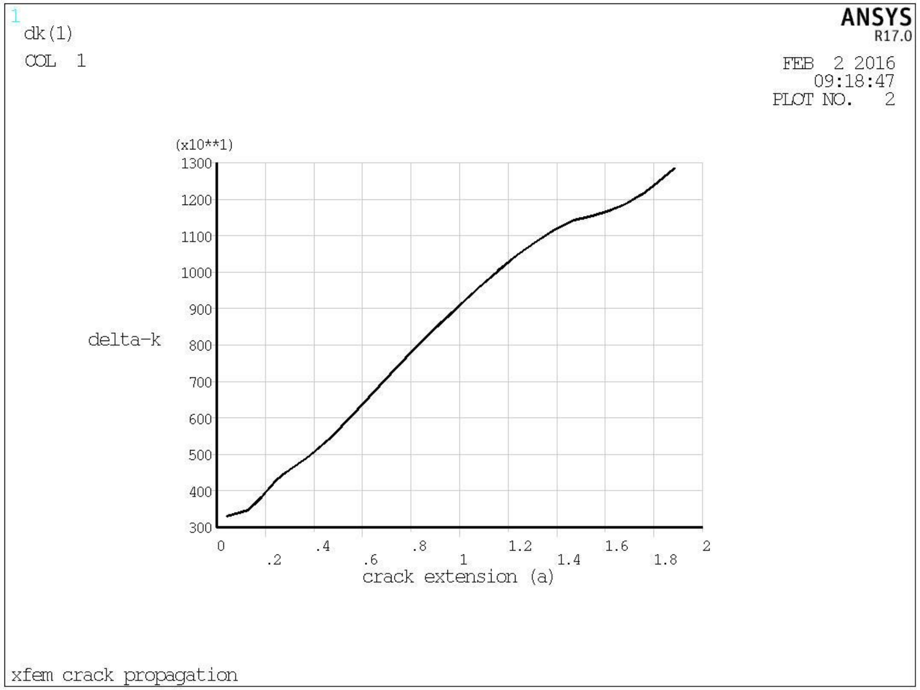


Example Problem

- Fatigue crack propagation results:



Cycles vs. crack length



ΔK vs. crack length