

FRANC3D Training Workshop: Part 7

Static Crack Analysis & SIF Computation

August - 2026

Drs. Paul “Wash” Wawrzynek, Bruce Carter,
Tony Ingraffea and Omar Ibrahim

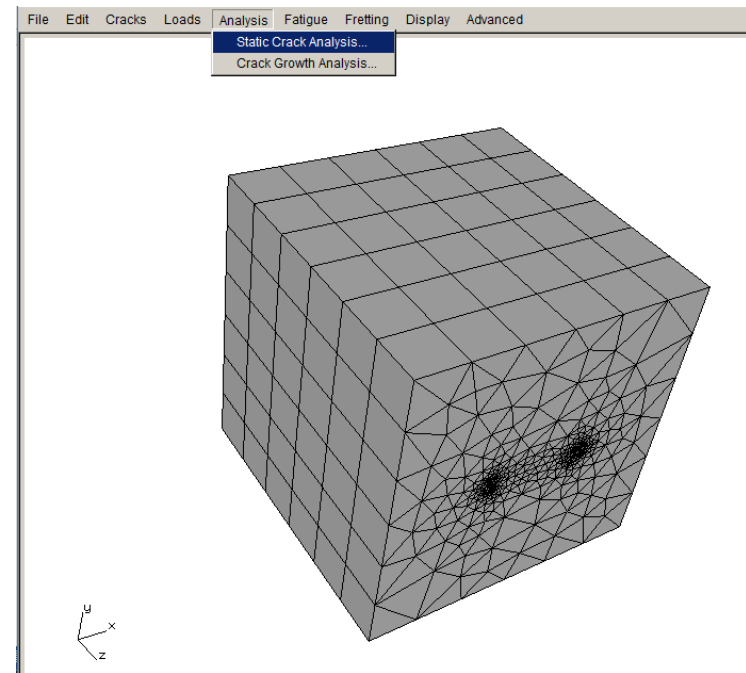
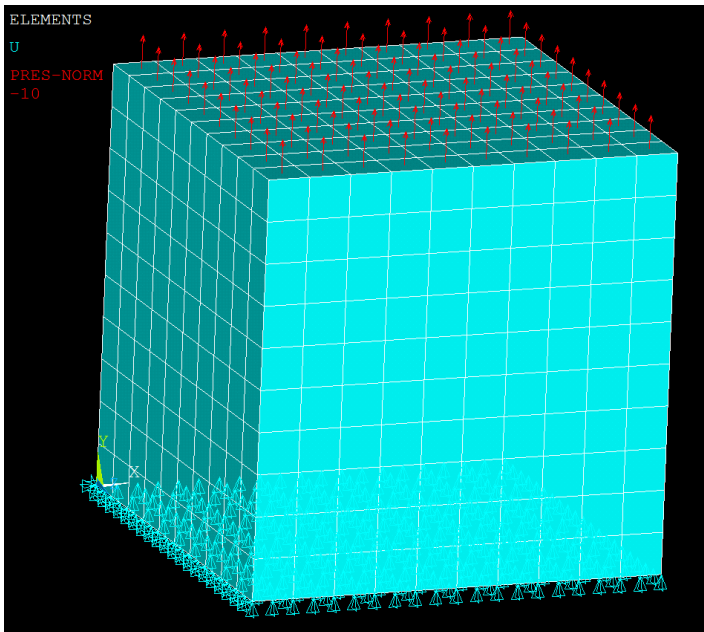
Workshop Agenda

- Part 1: Introduction to Fatigue and Damage Tolerance
- Part 2: Introduction to Fracture Mechanics Analysis
- Part 3: Introduction to FRANC3D
- Part 4: FRANC3D User Interface
- Part 5: Finite Element (FE) Model Import
- Part 6: Crack Insertion
- **Part 7: Static Crack Analysis & SIF Computation**
- Part 8: SIFs from FE Analysis
- Part 9: Crack Growth
- Part 10: SIF History & Fatigue Life
- Part 11: Miscellaneous Topics

Demo/Hands On (Homework): Static Crack Analysis and SIF Computations

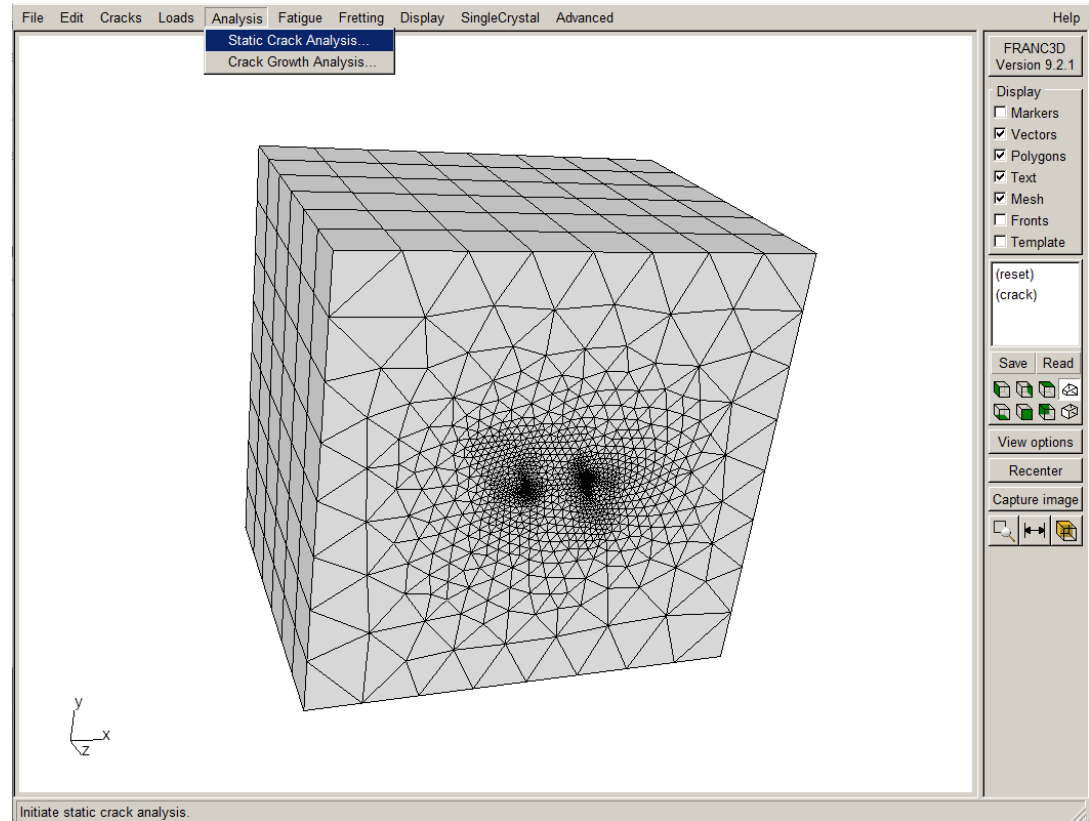
Tutorial #1:

Static Crack Analysis of a crack in a cube and SIF computation.

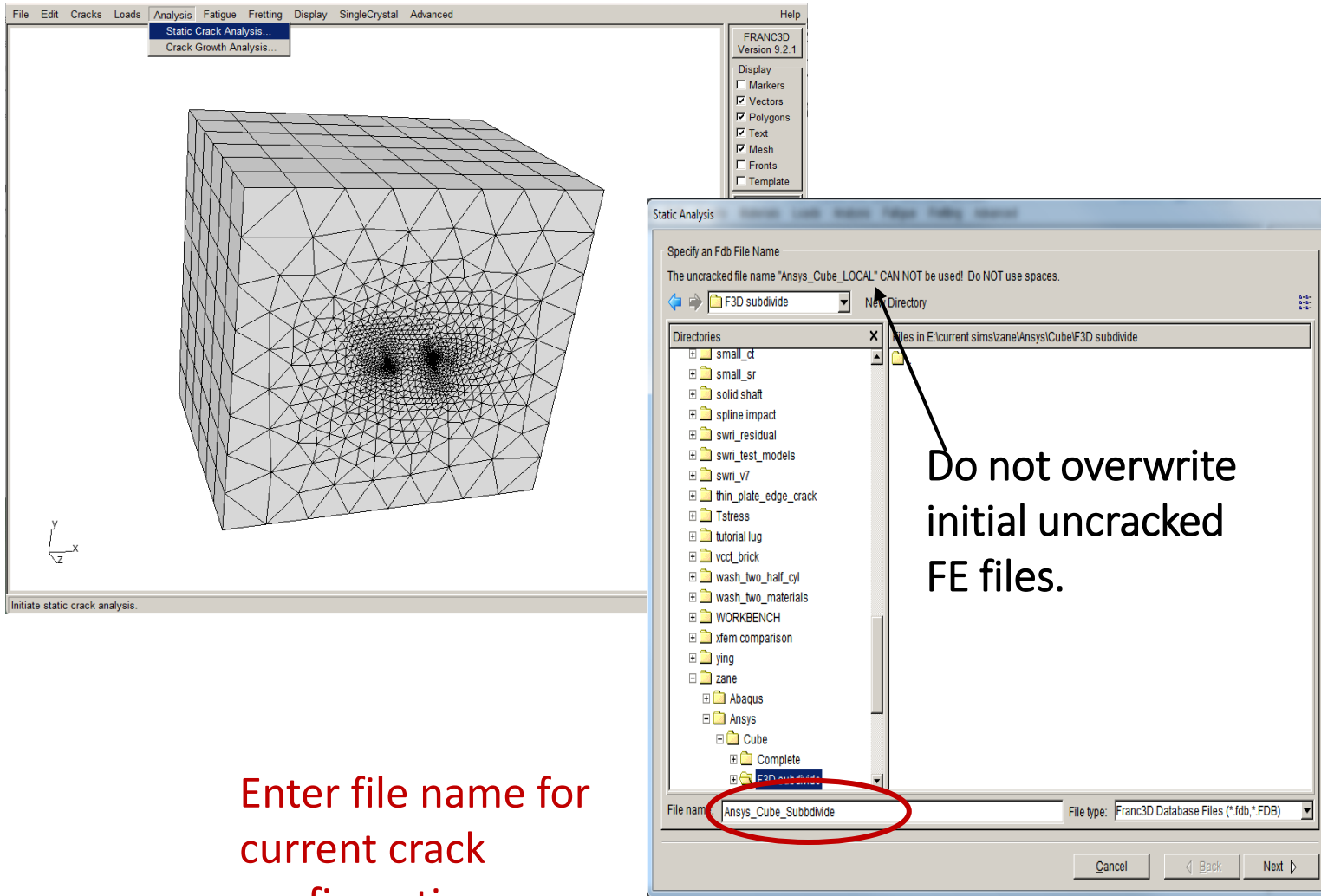


Static Crack Analysis

- Perform FE analysis of current crack configuration to get displacement (and other) results to compute SIFs



Static Crack Analysis



Static Crack Analysis

Static Analysis

Ansysis Options

Ansysis run time:

Ansysis Executable: Browse License:

Local model output:

Global model:

☒ Connect to global model filename: Browse

Boundary conditions:

☐ Apply crack face tractions ☐ Define crack face contact

Ansysis command:

☐ Write files but DO NOT run analysis

Connect to global is set if the user imports and divides or imports an already-divided model.

Write files but Do Not run if you need to run analysis on a different computer.

Executable (and license info) should be pre-set using **Edit - Preferences**.

Static Analysis

Ansysis local/global model connection:

☒ Merge nodes ☐ Constraint equations ☐ Contact conditions

Merge tolerance: Local connection midsides: ☒ retain ☐ remove

Local component(s) to merge/constrain:

☒ AUTO_CUT_SURF

Global component(s) to merge/constrain:

☒ GLOBAL_CONNECT_SURF

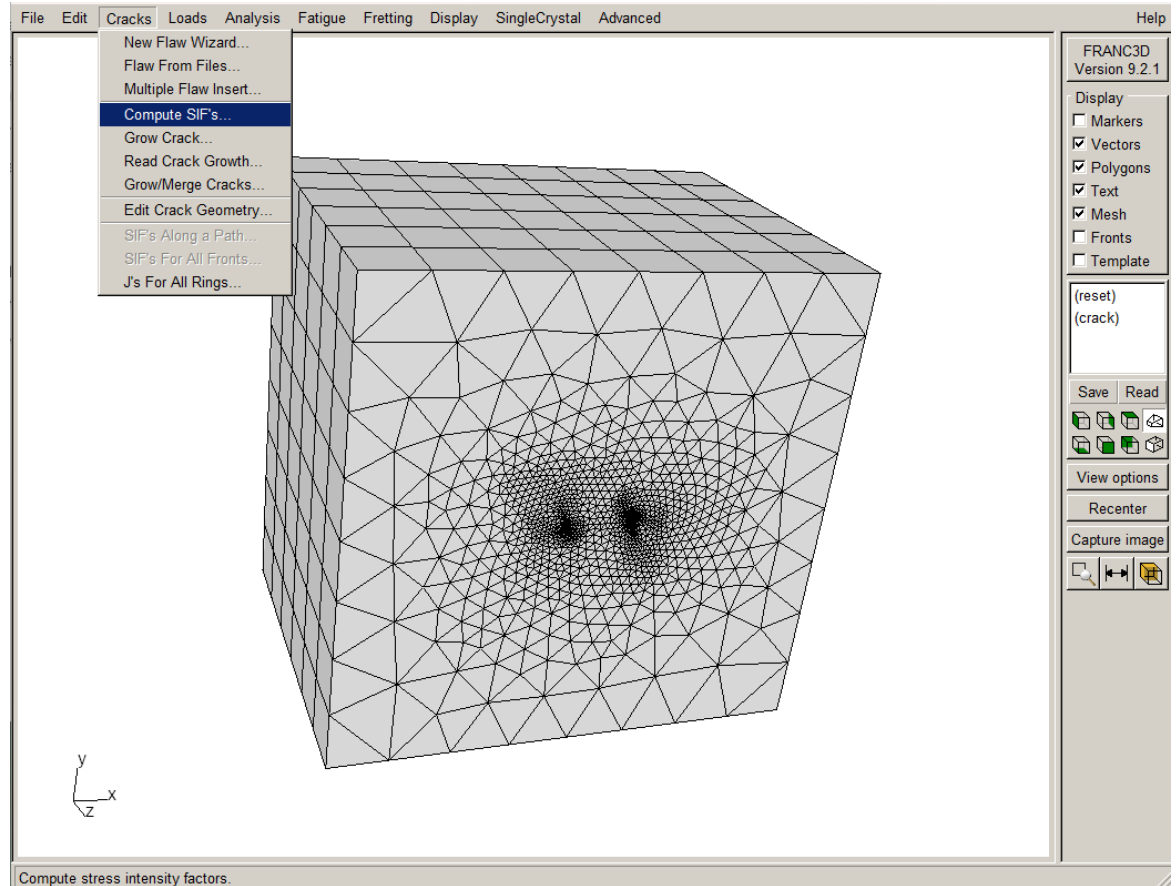
Additional local/global connections:

Ansysis command and script:

☐ Write files but DO NOT run analysis

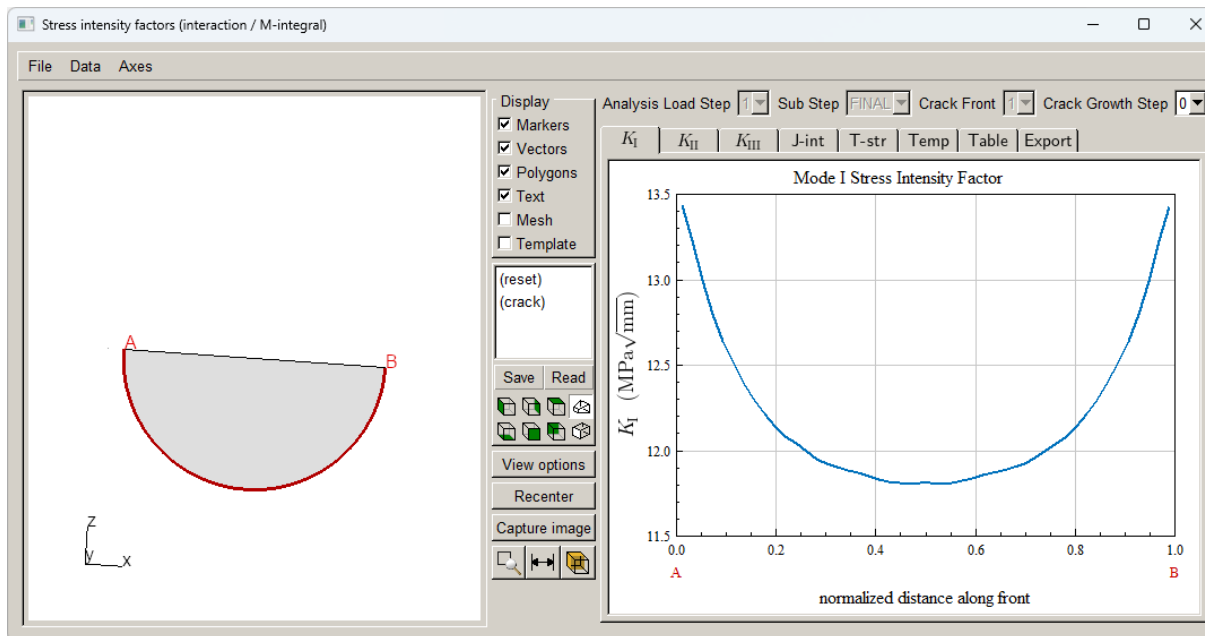
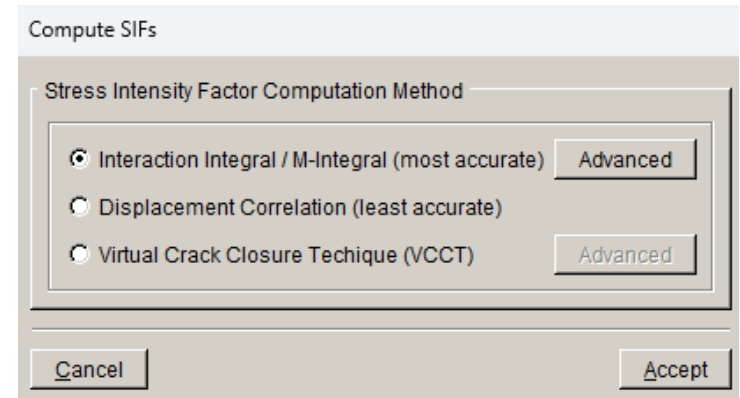
Compute SIFs

- If the FE analysis ran successfully, the Compute SIFs menu option will be active.



Computing SIFs

- M-Integral is the default method
- Thermal and crack face traction terms will be used if needed
- Displacement correlation can be used to verify the M-integral extra terms are included correctly
- Virtual Crack Closure – for cracks in bi-material interfaces



Plot SIF modes.

Export SIF data.

Switch between load steps, crack fronts, and steps of growth.

SIF Computation from FE Analysis Described in Part 8

with further description of the
FRANC3D Compute SIFs dialog.

End Part 7