

FRANC3D Training Workshop: Part 3

Introduction to FRANC3D

February - 2024

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

Introduction to FRANC3D

- FRANC3D Development History
- FRANC3D Version 8 Software
- What FRANC3D Can Do
- What FRANC3D is NOT
- Global and Local “sub-model” Terminology
- FRANC3D Benchmarking & Validation
- FRANC3D Documentations
- FRANC3D Tutorials

FRANC3D Development History

- 1988 to 1994
 - FRANC3D v1.0 BEM only
- 1994 to 2001
 - FRANC3D v2.0 BEM & Thin Shell FEM
- 2001 to 2005
 - FRANC3D v3.0 BEM & Thin Shell & Solid FEM (ANSYS)
- 2005 to 2009
 - FRANC3D v4.0 Solid FEM only (ANSYS, ABAQUS, NASTRAN)
 - Completely new code written in C++
- 2009 to 2010
 - FRANC3D v5.0 – Command line interface and other enhancements
- 2010 to 2014
 - FRANC3D v6 – Fretting Fatigue, Fatigue Life, Post-processing & other enhancements

FRANC3D Development History

- 2015 to 2020
 - FRANC3D v7.0
 - Built in sub-model generation tool
 - Expanded fatigue crack growth rate library
 - Robust automatic method for fatigue life integration
 - Time dependent and combined time and cycle dependent crack growth models
 - Python user-defined crack growth rules
 - Virtual crack closure technique
 - Partial crack front extension
 - SIF path options to improve paths for crack fronts that split or turn corners
 - Curvilinear elliptical crack geometry added to flaw library
 - Dialog for defining local coordinate systems for crack insertion
 - Preliminary capabilities for computing elasto-plastic J-integral from ABAQUS results
 - Crack face traction (CFT) capability allows a user to add CFT to an existing load step with options for setting temperature and thermal expansion properties

FRANC3D Version 8 Software

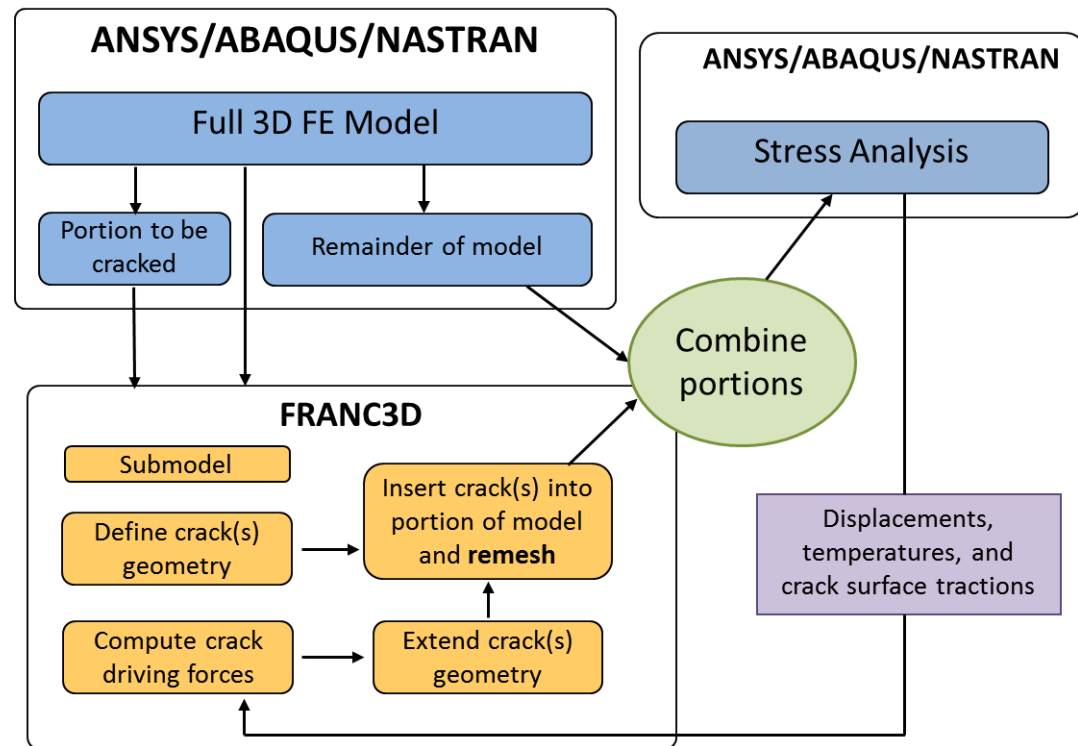
- 2021 to Present
 - FRANC3D v8.0
 - Revised load schedule/events with support for combined LCF/HCF loading & dynamic pairing
 - Revised crack insertion and meshing library with support for cracks crossing bi-material interfaces, cracks embedded in bi-material interfaces, symmetry surface cracks, and finite volume “thick” cracks
 - Computation of single crystal fracture parameters
 - Improved support for partial crack front extension
 - Added user-mesh voids
 - Allowed for crack insertion into a single mesh facet
 - FRANC3D v8.1
 - Improved ANSYS, ABAQUS and NASTRAN interfaces
 - Revised mapping to improve the transfer/mapping of nodal force boundary conditions
 - Improved crack insertion and meshing library
 - Added support for multiple crack growth rate models
 - Added support for inserting multiple symmetry cracks
 - FRANC3D v8.2
 - Added language dictionary to support additional languages for the GUI
 - Updated symmetry crack insertion capabilities
 - Added support for NASTRAN SOL 401 and ANSYS super elements
 - Improved crack surface coarsening/smoothing for partial growth

FRANC3D Version 8 Software

- 2021 to Present
 - FRANC3D v8.3
 - Added support for modal crack face traction (CFT) using .dtp files for stress and temperature
 - Added Python script for pre-NASTRAN analysis
 - Added more crack growth rate models
 - Added label for “passes through load schedule” in fatigue dialog
 - Added electrostatic energy release rate
 - FRANC3D v8.4
 - Revise volume meshing code; old volume meshing is available via Preferences.
 - Revised crack growth data stored in .fdb file.
 - Modified/fixed midside-to-quarter and quarter-to-midside node position code
 - Made blunt brick option available for Ansys and Sierra
 - Add to the Advanced menu to read analysis code contour integral data, with a dialog to plot the CI data.
 - Other enhancements

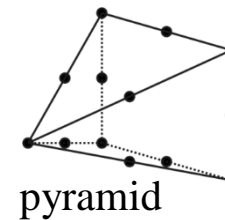
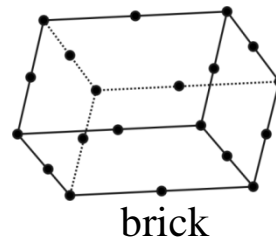
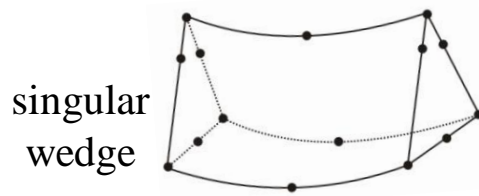
FRANC3D Version 8 Software

- **FR**acture **AN**alysis **CO**de **3-D** uses the finite element (FE) method to simulate crack growth
- Designed to work in conjunction with commercial finite element codes:
 - ANSYS
 - ABAQUS
 - NASTRAN
- Computes stress intensity factors (SIFs) for arbitrary 3D crack geometry



FRANC3D Software

- Adaptively remeshes a 3D FE model to simulate crack growth; uses explicit crack geometry
- Several elements are used at the crack front:



- Written in C++
- Runs on Windows and Linux systems
- Programming interface that is an extension to the Python programming language

What Does FRANC3D Do?

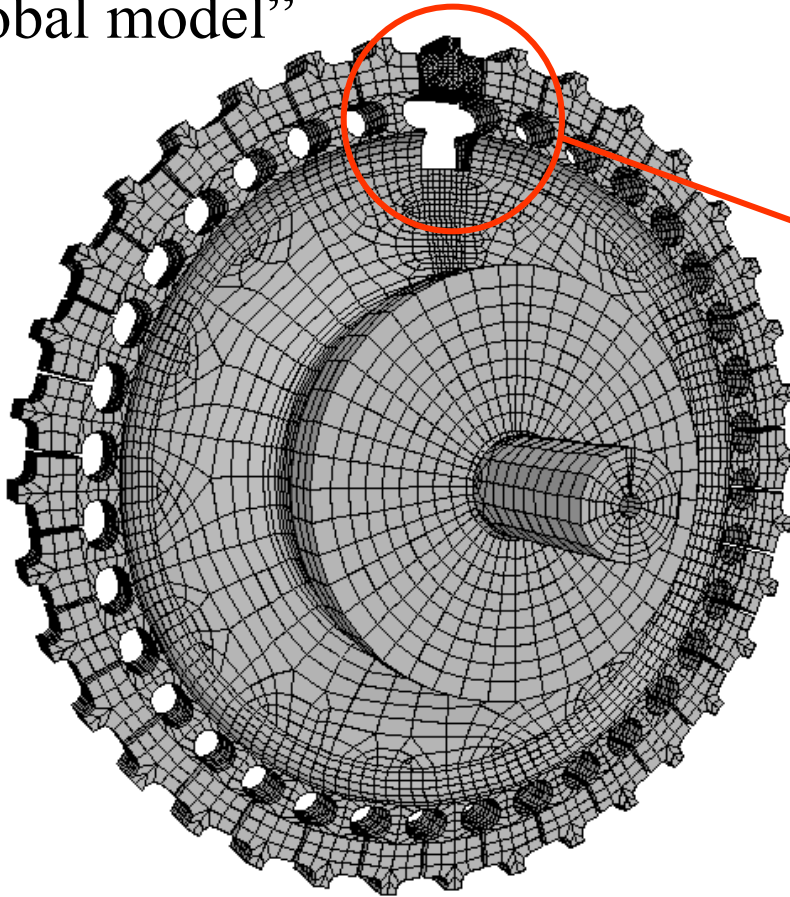
- Inserts a flaw into an *existing* finite element (FE) model and remeshes using singular crack-front elements.
- Computes stress intensity factors (SIF's) for all three “modes” of fracture at FE **mid-side nodes** along crack front for *isotropic* and *anisotropic* materials.
- Predicts how a crack will grow (relative extension and angle) using *engineering* growth criteria, extends the crack geometry and remeshes.

What FRANC3D is NOT

- Not a general FE pre-processor or post-processor: external software is required to build uncracked FE models and to visualize results (FRANC3D can display deformations).
- Not a FE analysis program: an external FE code is required (*e.g.*, ANSYS, ABAQUS or NASTRAN) to perform the stress analysis.
- Not a “fast running” fatigue life prediction code with stored analytical or tabular K solutions (*e.g.*, AFGROW, NASGRO or DARWIN).

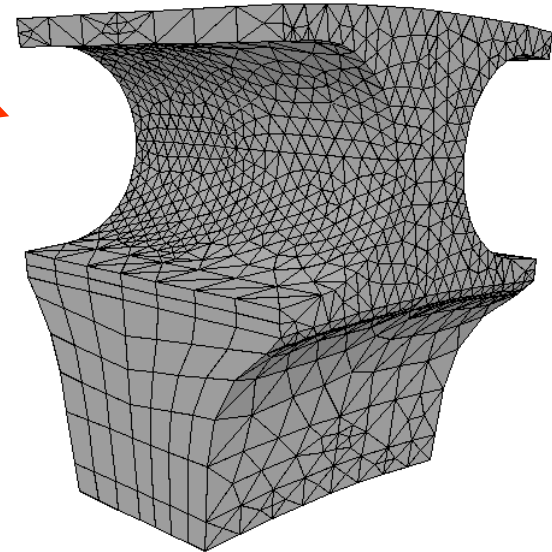
Global and Local “sub-model” Terminology

“global model”



“sub-model”

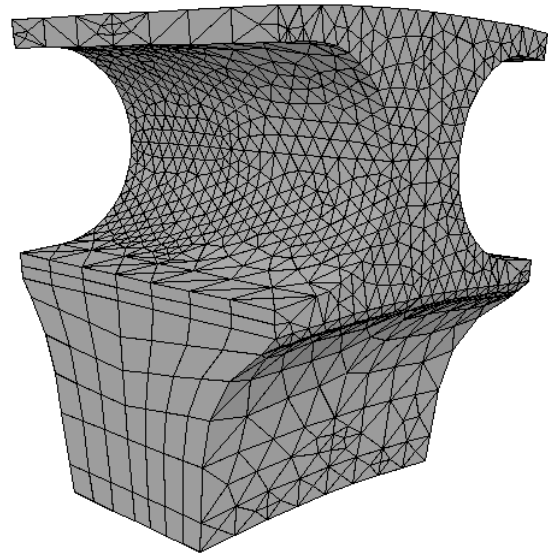
crack growth region



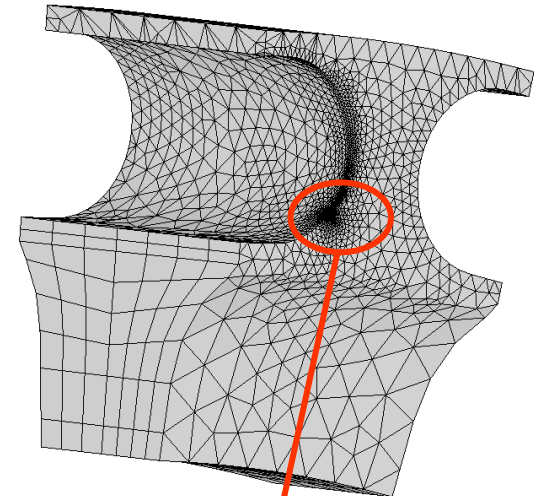
FE software (*e.g.*, ANSYS) or FRANC3D can be used to define the global and sub-model. The sub-model should encompass the expected crack growth with ‘space’ for remeshing.

FRANC3D Modifies the Sub-model

uncracked sub-model

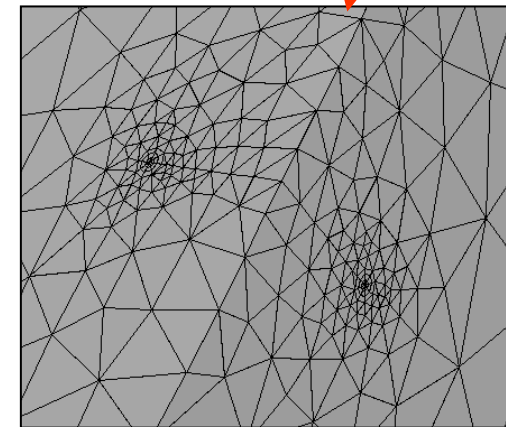


after crack insertion

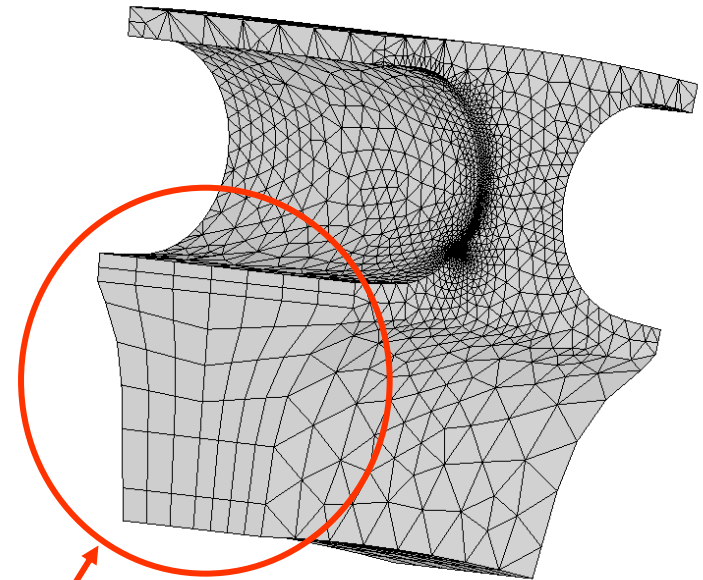
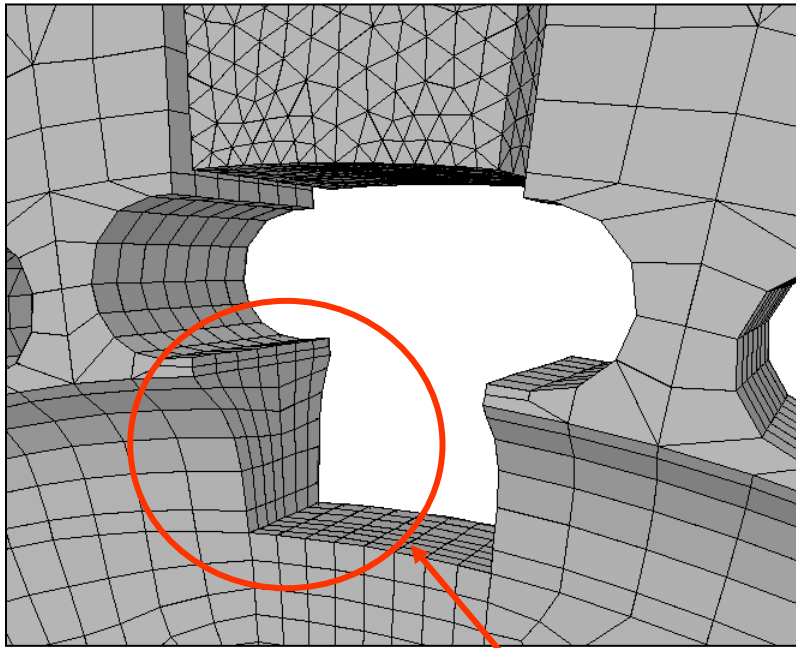


FRANC3D modifies the sub-model by **inserting a crack and remeshing**.

A file that combines the global and remeshed sub-model is output for the FE analysis software.



FRANC3D Maintains Mesh Compatibility

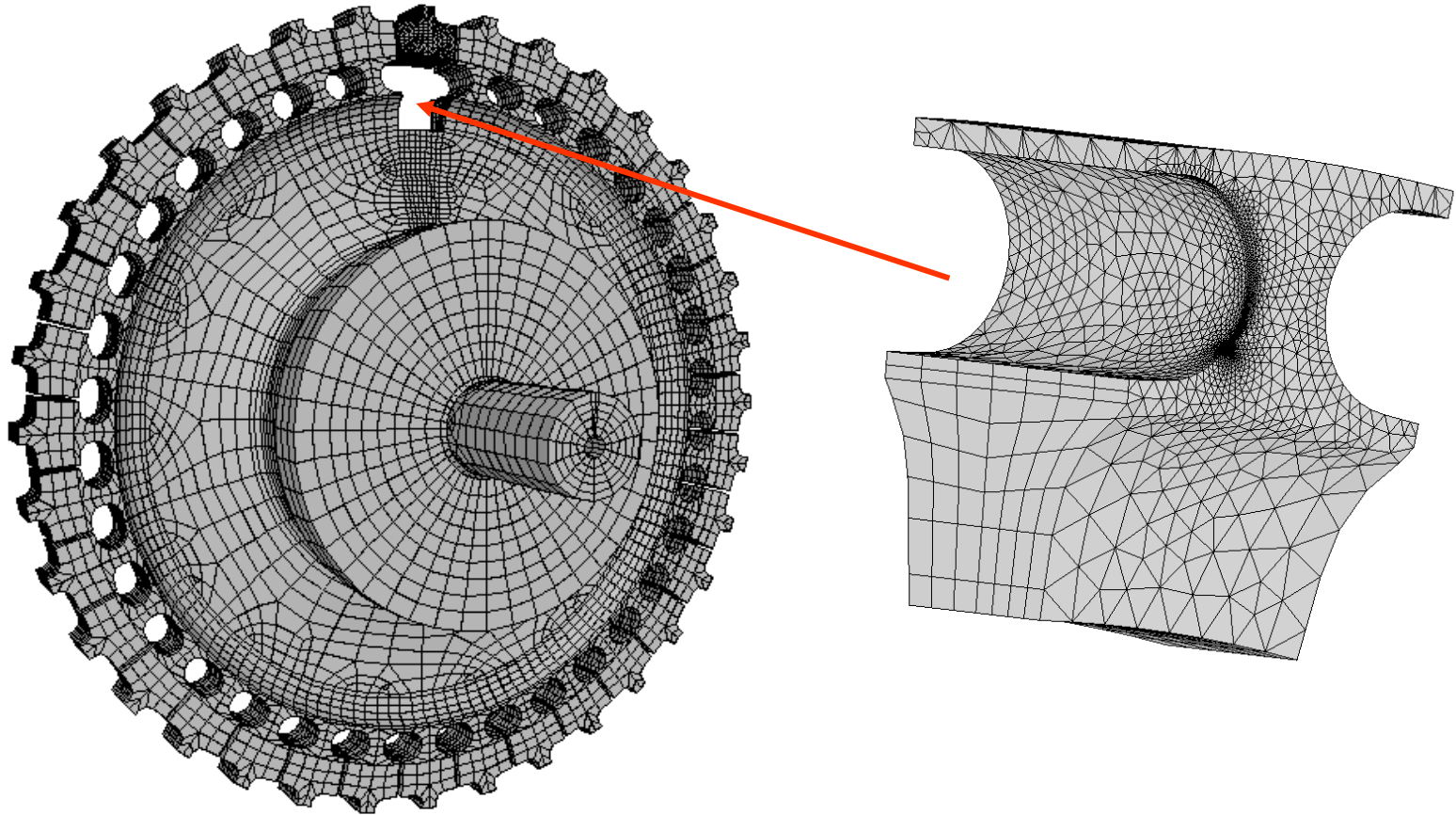


mesh compatibility

FRANC3D retains surface meshes on “cut” surfaces for compatibility between the global and sub-model – the best and preferred approach.

Note that FRANC3D can instruct the FE software to use constraint or contact on these surfaces.

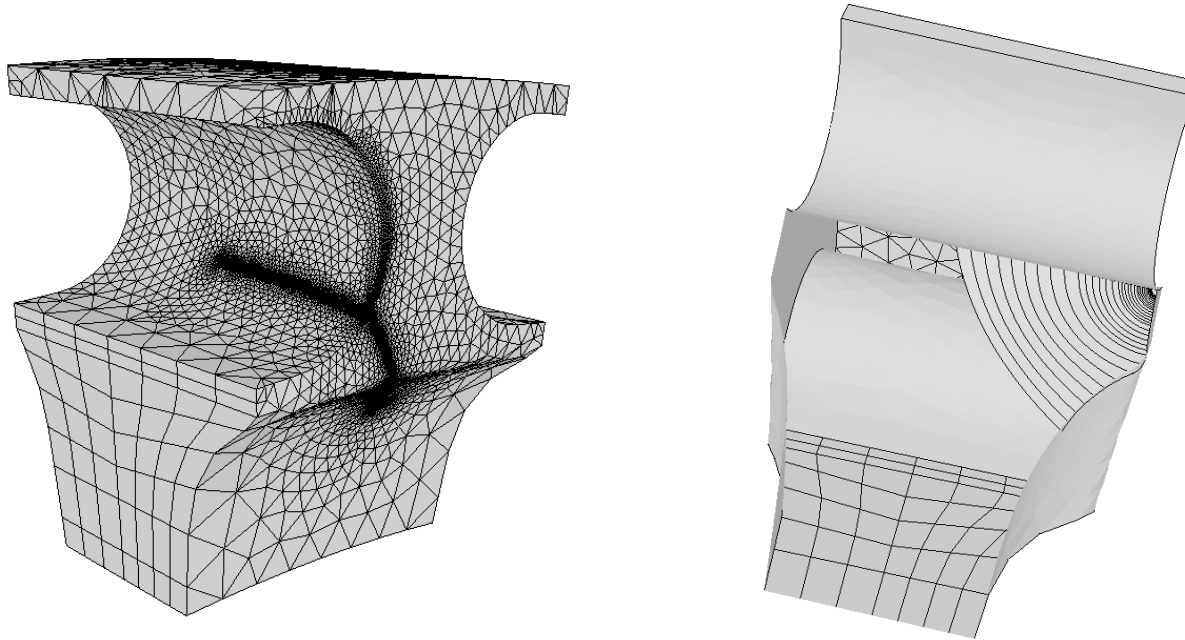
Re-Combined (Full Model) Analysis



FRANC3D does **not** use “global/local” solution approach (although it is possible); FE analysis is performed with the **full** re-combined model.

Crack Growth Process

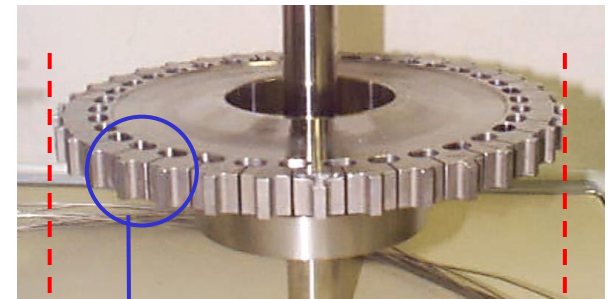
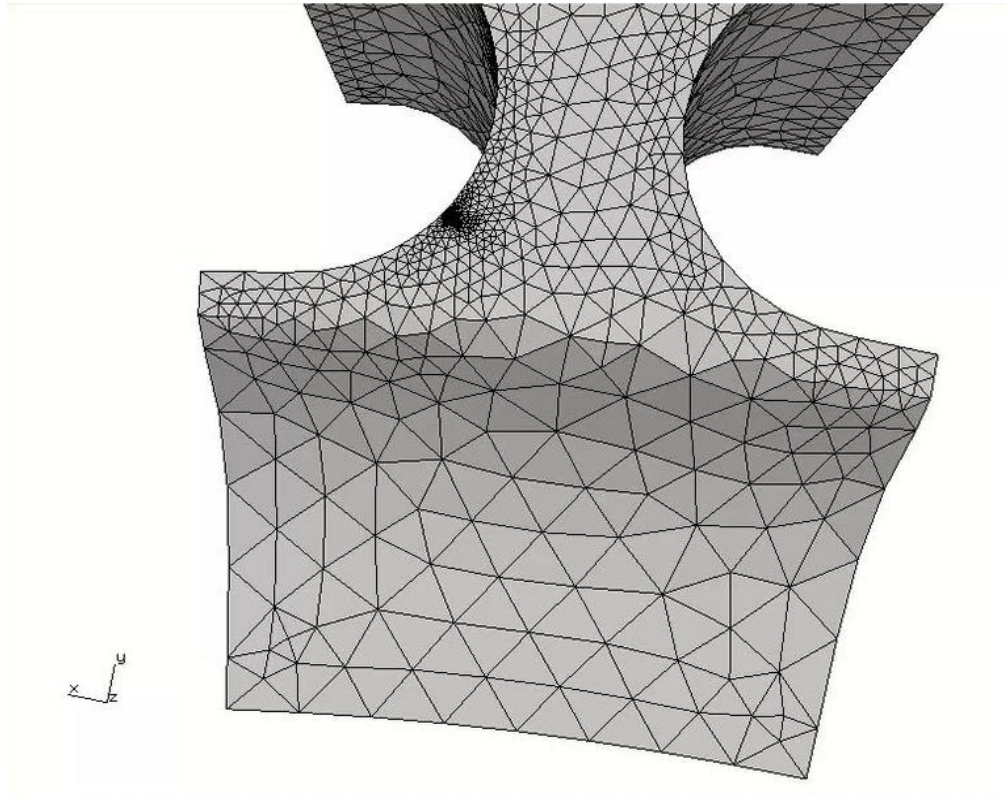
after 33 steps of crack growth



Crack growth is simulated by repeatedly reading and modifying the initial uncracked sub-model; *do not remove or overwrite uncracked FE model files.*

At each step, the global and modified (local) sub-model are re-combined and the full model is analyzed.

Mini-turbine disk crack growth simulation



← 264 mm (10.4 in) →



Mini-turbine disk (Russ et al, 2004)

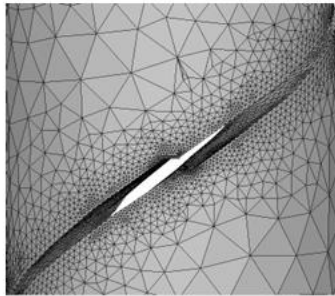
FRANC3D Benchmarking & Validation:

- in-house
- industry examples

FRANC3D Benchmarks

FRANC3D

Benchmark Examples



Version 8.3

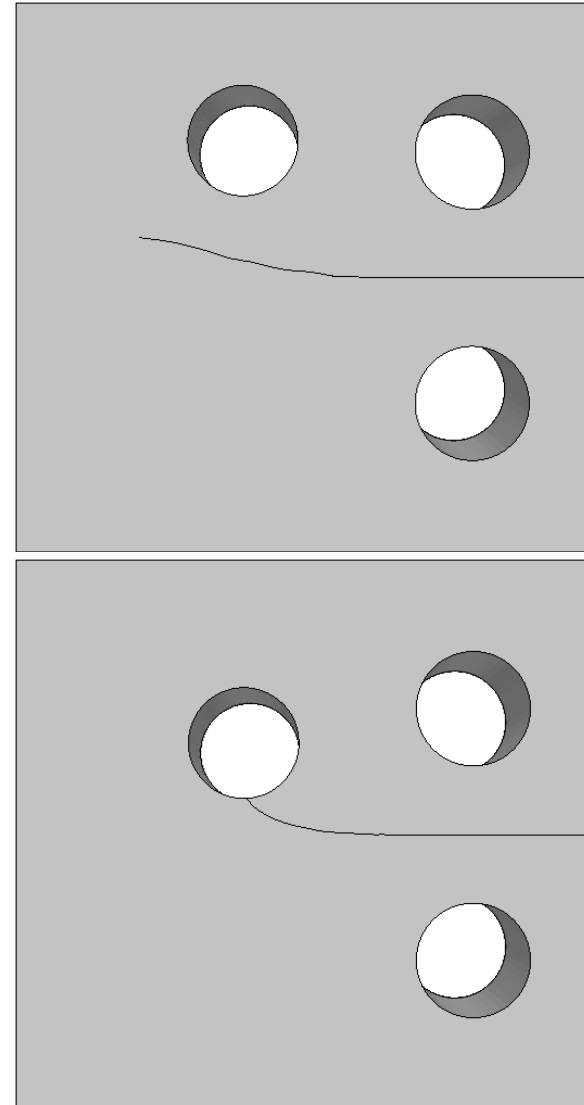
2 Interior Penny Shaped Crack in a Rectilinear Bar (Sneddon Solution)	3
2.1 Crack Face Pressure (Traction)	7
3 Internal, Inclined (45 degrees), Penny-Shaped Crack	9
4 Surface Ellipse Crack in a Plate – Raju-Newman Finite Element Solution	12
5 Through-Thickness Crack in a Plate	15
5.1 Through-Thickness Edge Crack	16
5.2 Middle-Through-Thickness (Center) Crack	19
6 Thick Plate with Middle Through Crack and Anisotropic Material Properties	21
6.1 Uniform Tension	23
6.2 Pure Shear	24
7 Corner Crack in a Plate with a Hole	26
8 Compact Tension Specimen	30

<http://www.fracanalysis.com/software.html>

Other Simulations/Examples

- Compact Tension (CT) is a standard fracture test specimen, which is a 2-D problem modeled as 3-D.
- If a third hole is bored in the specimen, this causes the crack to curve; depending on where the third hole is placed, the crack either grows into or bypasses the hole.

http://www.fracanalysis.com/simulations/ct_specimen.html



Industry Example

Cracking of a Gear Tooth

Simulated

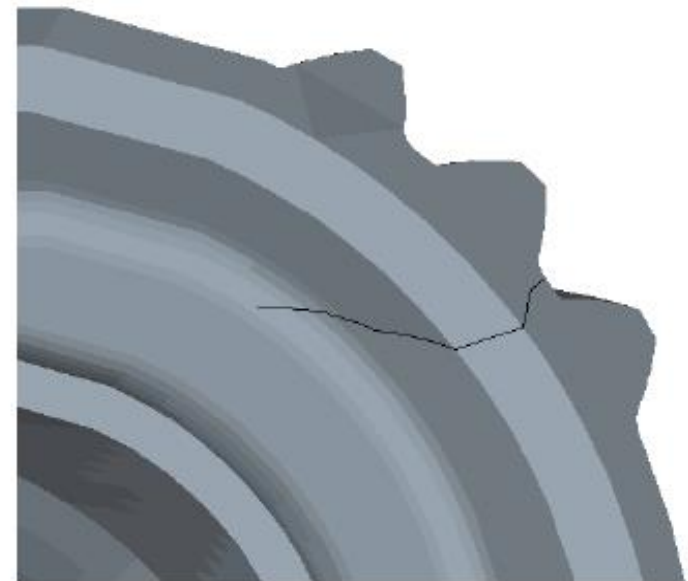
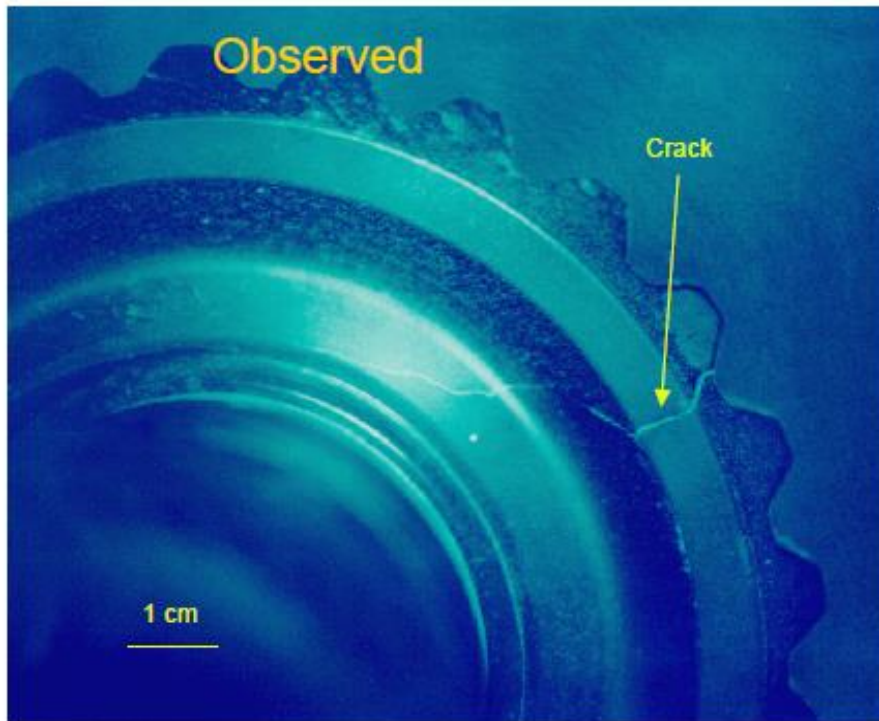


Observed



Industry Example

Comparison: Simulated vs Observed Crack Trace on Gear Hub



Industry Example

Observed crack growth

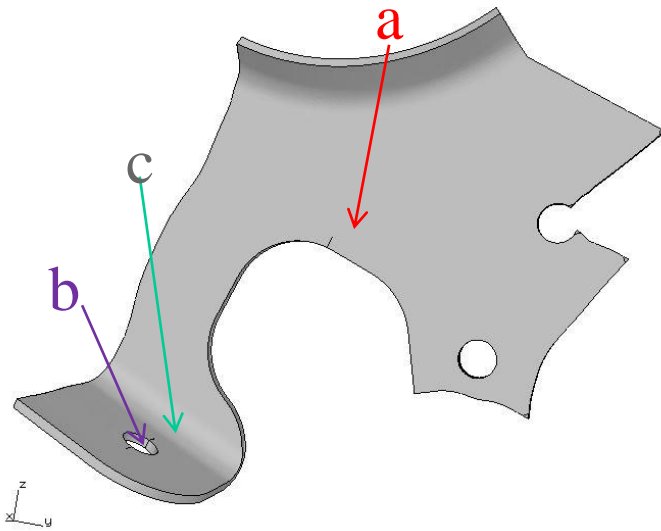


FRANC3D blind prediction

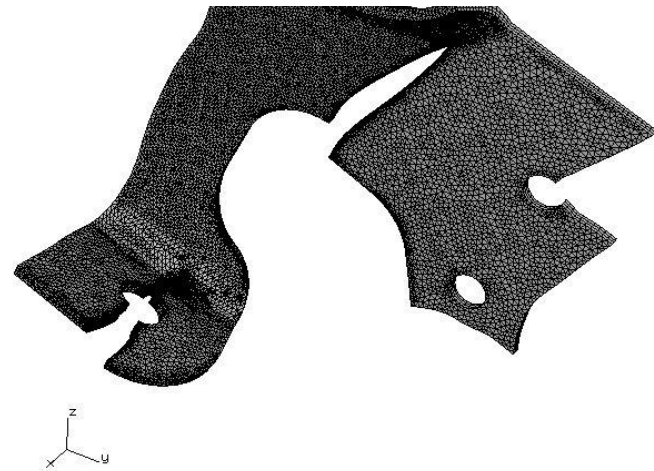


Industry Example

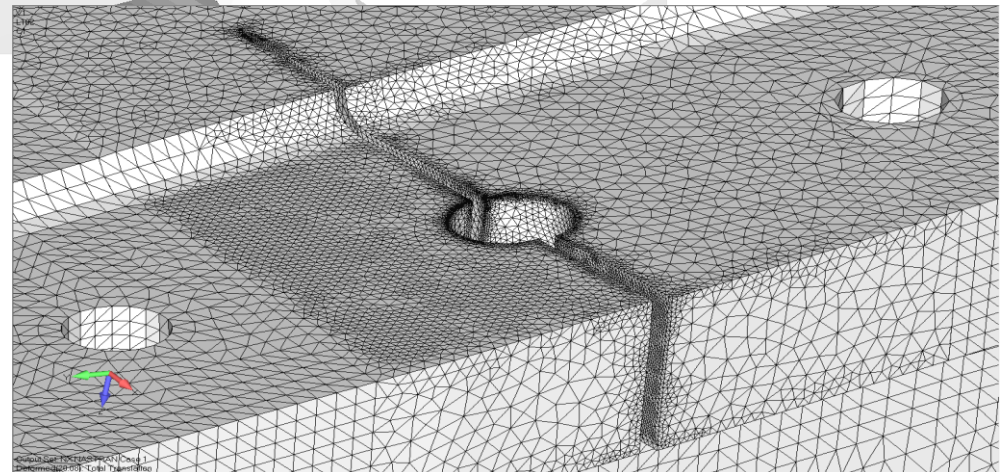
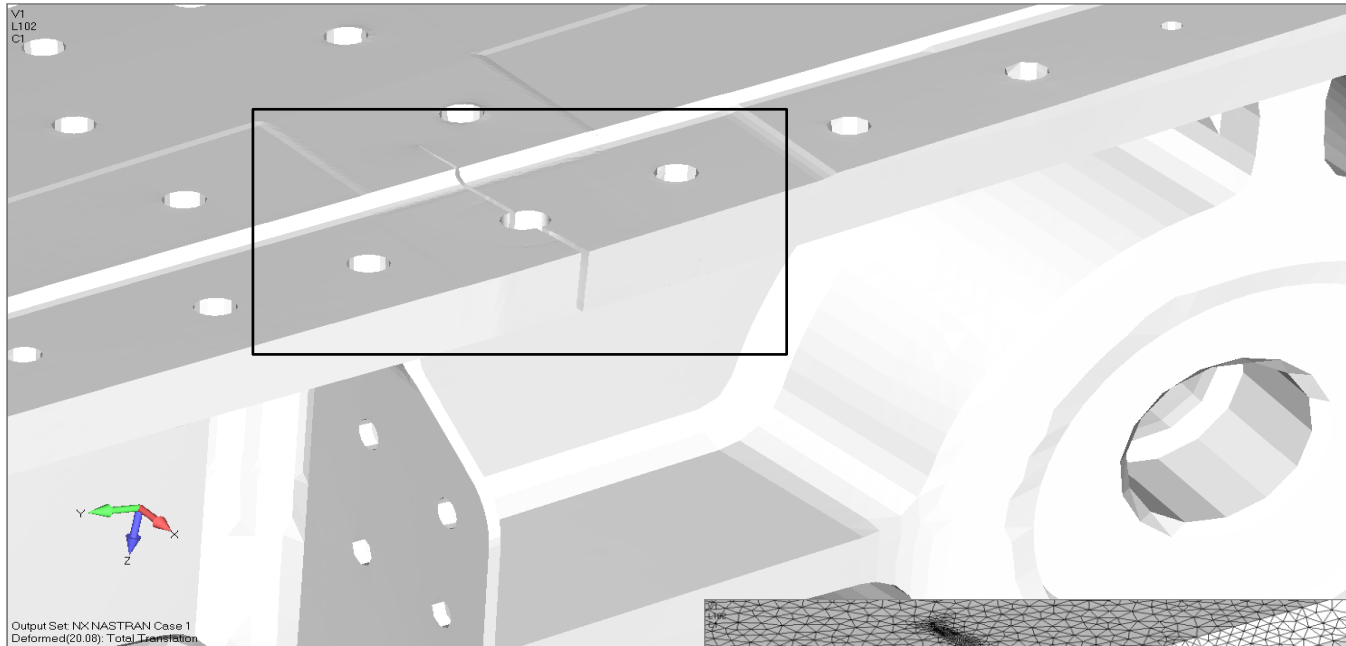
Three cracks inserted and propagated



45 steps of crack growth

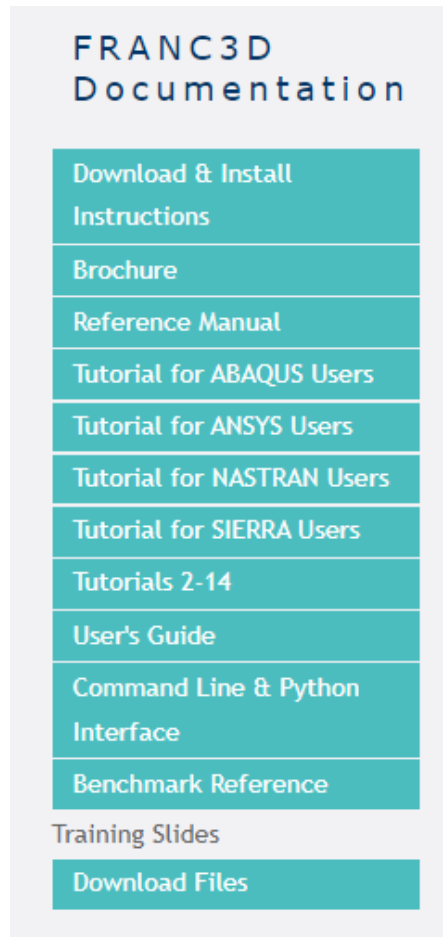


Industry Example



FRANC3D Documentation

<http://www.fracanalysis.com/software.html>



The image shows a vertical navigation menu for the FRANC3D Documentation website. The menu items are listed in a light gray box with teal-colored buttons. The items are: Download & Install Instructions, Brochure, Reference Manual, Tutorial for ABAQUS Users, Tutorial for ANSYS Users, Tutorial for NASTRAN Users, Tutorial for SIERRA Users, Tutorials 2-14, User's Guide, Command Line & Python Interface, Benchmark Reference, Training Slides, and Download Files.

FRANC3D Documentation
Download & Install Instructions
Brochure
Reference Manual
Tutorial for ABAQUS Users
Tutorial for ANSYS Users
Tutorial for NASTRAN Users
Tutorial for SIERRA Users
Tutorials 2-14
User's Guide
Command Line & Python Interface
Benchmark Reference
Training Slides
Download Files

FRANC3D Tutorials

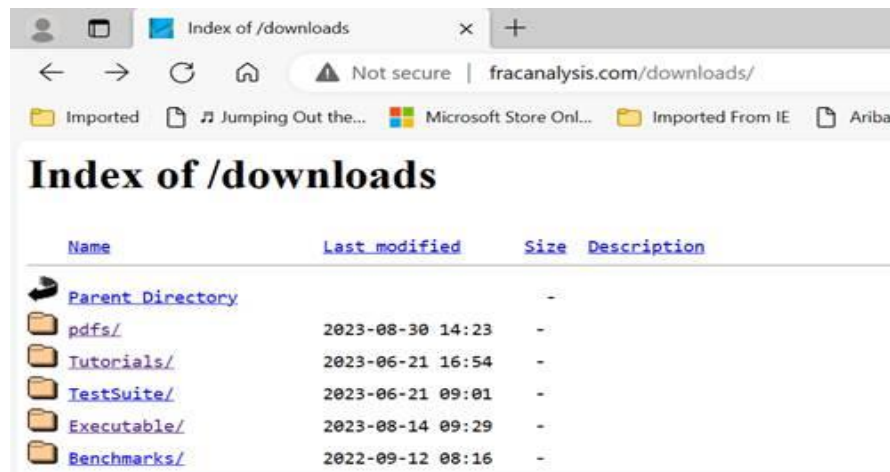
To download the ANSYS tutorials, please click on the link below and follow the steps:

<http://fracanalysis.com/downloads/>

You will be prompted to enter username and password.

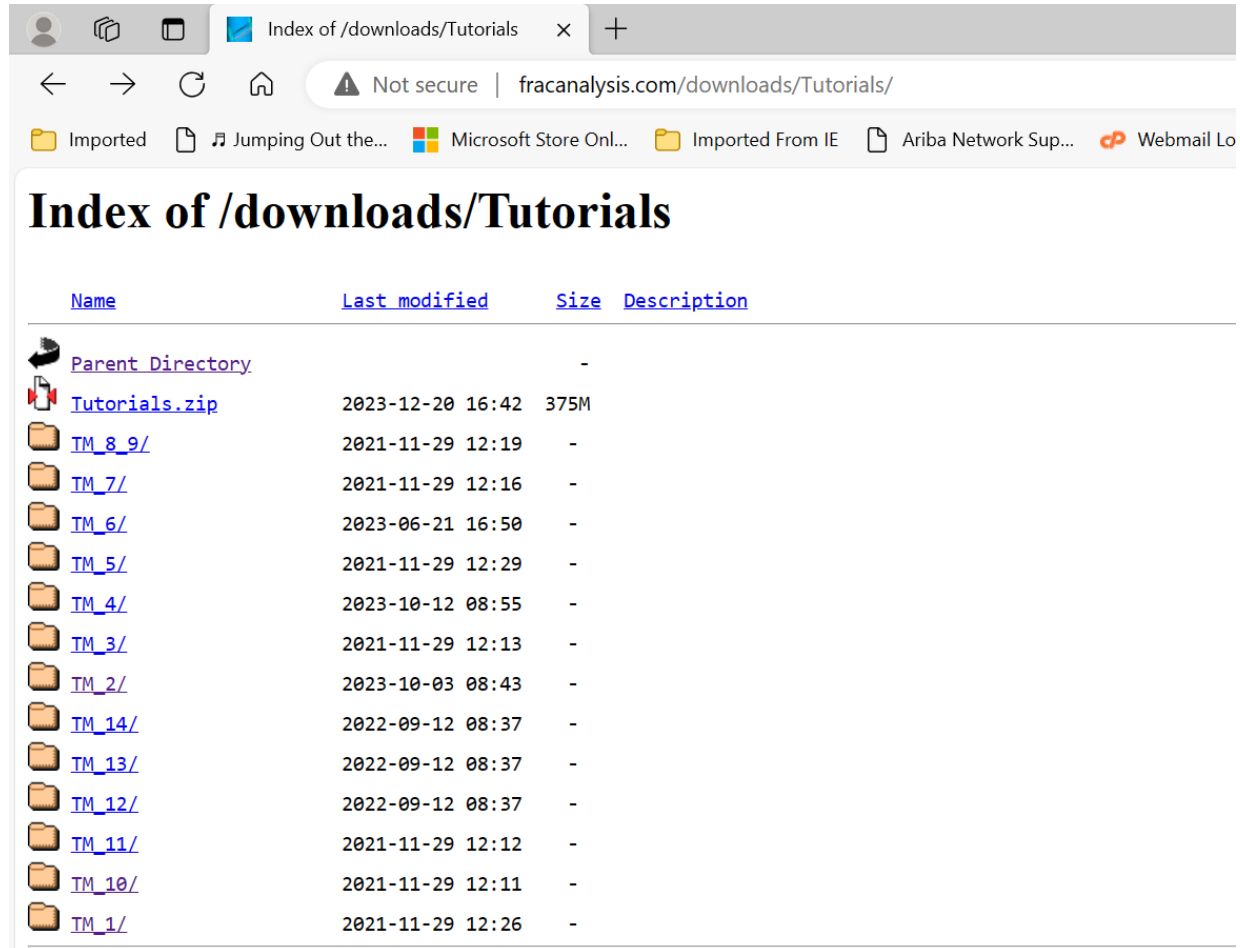
Current username: franc

Current password:



Click on  [Tutorials/](#)

FRANC3D Tutorials



The screenshot shows a web browser window with the address bar displaying "Index of /downloads/Tutorials/". The page title is "Index of /downloads/Tutorials/". The browser's address bar shows "Not secure" and the URL "fracanalysis.com/downloads/Tutorials/". The browser's taskbar shows several open tabs: "Imported", "Jumping Out the...", "Microsoft Store Onl...", "Imported From IE", "Ariba Network Sup...", and "Webmail Lo".

The main content of the page is a directory listing table with the following columns: Name, Last modified, Size, and Description. The table lists the following items:

Name	Last modified	Size	Description
Parent Directory		-	
Tutorials.zip	2023-12-20 16:42	375M	
TM_8_9/	2021-11-29 12:19	-	
TM_7/	2021-11-29 12:16	-	
TM_6/	2023-06-21 16:50	-	
TM_5/	2021-11-29 12:29	-	
TM_4/	2023-10-12 08:55	-	
TM_3/	2021-11-29 12:13	-	
TM_2/	2023-10-03 08:43	-	
TM_14/	2022-09-12 08:37	-	
TM_13/	2022-09-12 08:37	-	
TM_12/	2022-09-12 08:37	-	
TM_11/	2021-11-29 12:12	-	
TM_10/	2021-11-29 12:11	-	
TM_1/	2021-11-29 12:26	-	

End Part 3