Development of Automated Model Generation and Analysis of Surface Cracked Plates in Bending for Interpolated Elastic-Plastic J-integral Solutions

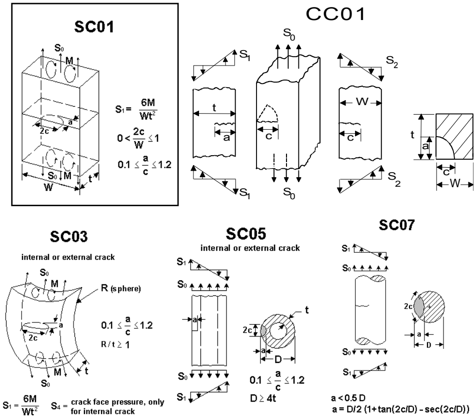
Michael W. Renfro[1] Phillip A. Allen[2] Christopher D. Wilson[1]

May 15, 2019

[1]Tennessee Tech University
[2]NASA Marshall Space Flight Center

Introduction

- Constraint and stress states can get pretty involved.
- Semi-elliptical surface cracks (SC01) are among the simplest part-through crack cases, and are the subject of ASTM E2899.
- Handbook or curve-fit solutions exist for the other geometries, but only for linear elastic materials.
- NASA's TASC program covers elastic-plastic surface cracks in tension, but **not** in bending.



Research goals

- High quality set of elastic-plastic finite element analysis results as a basis for curve-fit or handbook calculations
- Modified TASC program for bending or tension analysis

Literature Review

For a surface crack in bending, stress intensity at a given location:

$$K_\mathrm{I} = (H\sigma_\mathrm{b}F_\mathrm{b})\left(\frac{\pi a}{Z}\right)^{0.5}$$

$$\sigma_\mathrm{b} = \frac{6M}{Wt^2}$$

$$Z = 1 + 1.464\left(\frac{a}{c}\right)^{1.65}$$

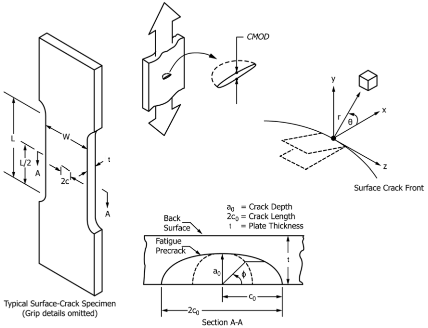
$$F_\mathrm{b} = \left[M_1 + M_2\left(\frac{a}{t}\right)^2 + M_3\left(\frac{a}{t}\right)^4\right] f_\phi f_\mathrm{wb} g$$

For surface cracks in tension, we need “only” 10 equations.

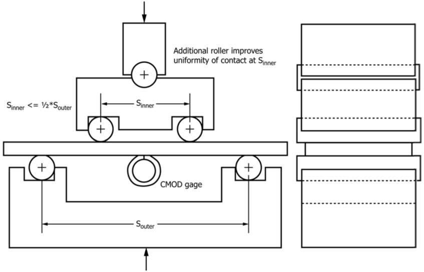
No such curve fit exists for elastic-plastic materials.

...plus another 12 equations, and that’s just for linear elastic materials.

ASTM E2899: Standard Test Method for Measurement of Initiation Toughness in Surface Cracks Under Tension and Bending



Test specimen and crack configurations



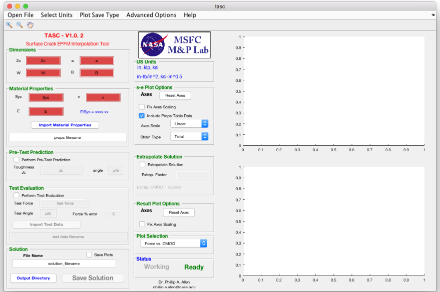
Four-point bend test configuration

- Starter crack machined into flat plate, fatigued to sharpen crack front
- CMOD monitored as tension or bending load increased monotonically
- Either specimen fails or start of stable crack tearing is detected
- Location where crack growth occurs is recorded
- Conditions classified as linear elastic, elastic-plastic, or fully-plastic
- If LEFM or EPFM, calculate constraint from tables
- If LEFM, calculate K from series of provided equations
- If EPFM, **use nonlinear FEA** to calculate J

ASTM E2899 is unusual in its analysis requirements

- requires results of elastic-plastic finite element analysis
- other standards require much simpler calculations or graphical constructions
- NASA TASC program satisfies requirements, but only for tension

Mechanics of bending is more complex than for tension (constraint, crack closure)

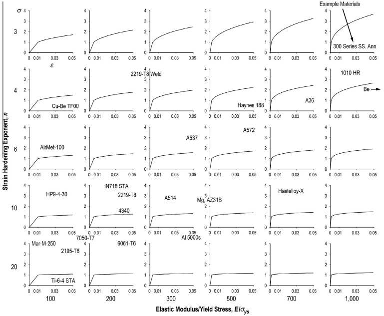


NASA TASC program

TASC: interpolation code using database of 600 EPFM results for flat plates in tension

20 normalized geometries:

$0.2 \leq \frac{a}{c} \leq 1.0, \ 0.2 \leq \frac{a}{t} \leq 0.8$

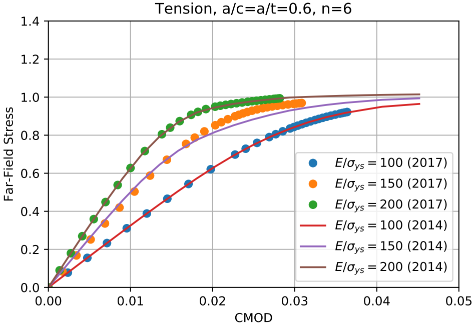


30 normalized materials:

$100 \leq \frac{E}{S_{ys}} \leq 1000, \ 3 \leq n \leq 20$

Modeling Preparation for Research Tasks

Verification of two TASC cases, applying procedure to a new material ($\frac{E}{\sigma_{ys}} = 150$)



Comparison of normalized FEA results, interpolated result, and TASC raw data

Research Plan for Bending Models and Modified TASC Program

$$t = 1 \quad 0.2 \leq \frac{a}{c} \leq 1.0 \quad 0.2 \leq \frac{a}{t} \leq 0.8$$
$$W = 5\max(c, t) \quad S_{\text{inner}} = W \quad S_{\text{outer}} = 2W \quad L = 1.1 S_{\text{outer}}$$

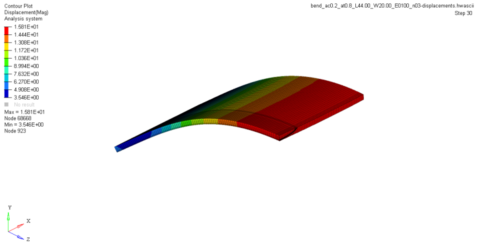


Plate model, $\frac{a}{c} = 0.2$, $\frac{a}{t} = 0.8$

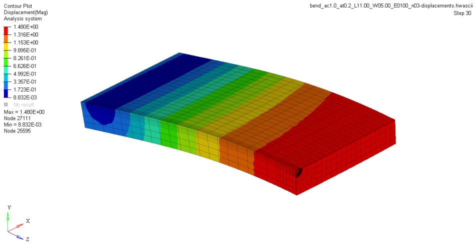
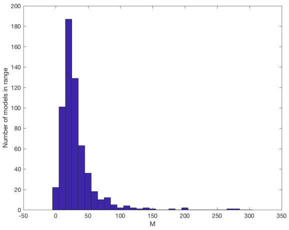
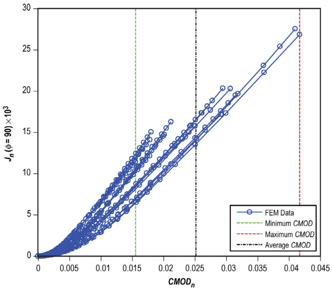


Plate model, $\frac{a}{c} = 1.0$, $\frac{a}{t} = 0.2$

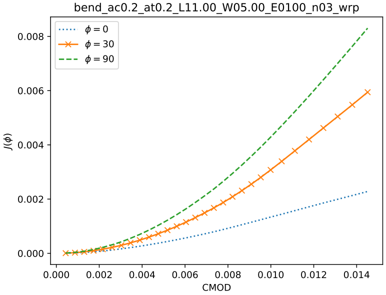
Allen and Wells (2014) reported $M = \frac{r_\phi \sigma_{ys}}{J} < 25$ for tension



Histogram of M results from TASC tension model database



J-CMOD graph used for extrapolation



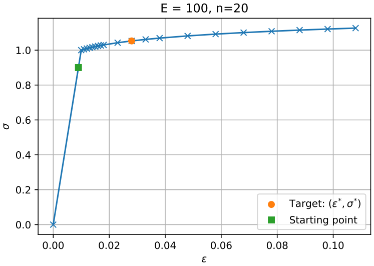
$\frac{a}{c} = 0.2,\ \frac{a}{t} = 0.2,\ E = 100,\ n = 3$

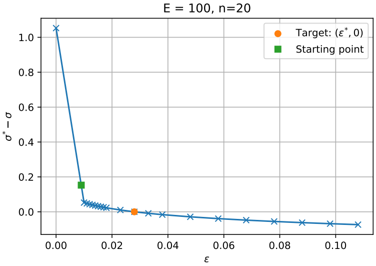
Adjust boundary conditions until

- slope of last 20% of J-CMOD curve is $20\times$ larger than initial slope
- slope of last 20% of J-CMOD curve is $< 10\%$ different than slope of previous 20%

at $\phi = 30^\circ$

Displacement control for tension models makes optimization easier

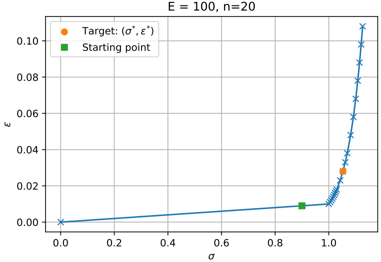




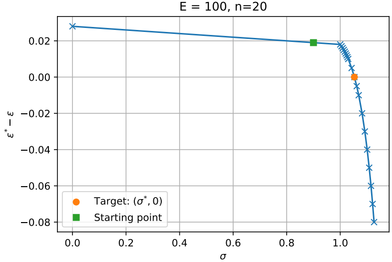
Example stress-strain curve using linear plus power law (LPPL) formulation

Transformed to find required strain level

Load control for bending models makes optimization more difficult

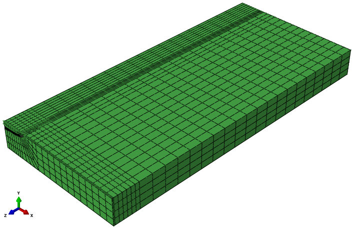


Example stress-strain curve using LPPL formulation, transformed to stress-controlled



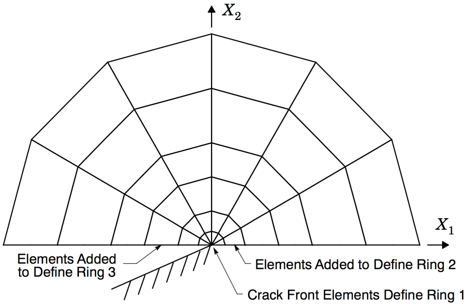
Transformed to find required stress level

Abaqus validation



Example Abaqus bending model from FEACrack

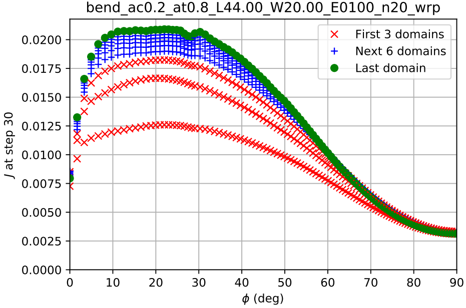
J convergence



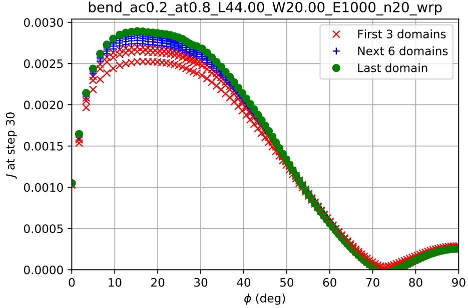
Elements used in WARP3D J calculations

- Don't break anything already working for tension
- Make a `results_bending` database alongside the existing `results` database for tension
- Identify any equations only valid for tension models
- Replace with conditionals checking for model type, then use tension or bending equations as required
- Interpolation method should need no changes
- Validate a load-CMOD curve against existing bending experimental data

Results and Discussion

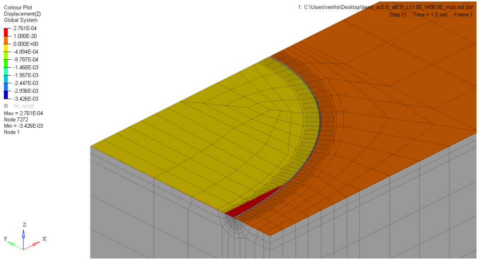


Convergence of J across 10 domains

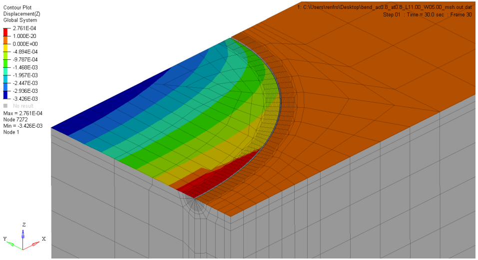


Anomalous J convergence graph

Why is J higher at $\phi = 90$ in some cases? What happens to plate deflection?



First load step



Last load step

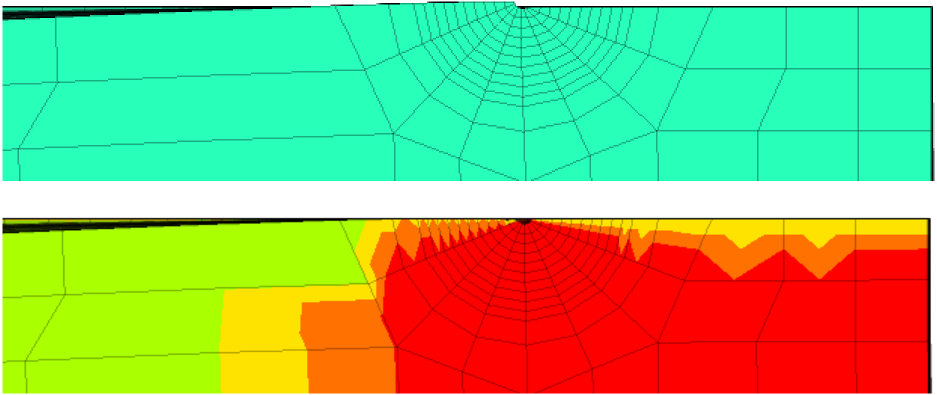
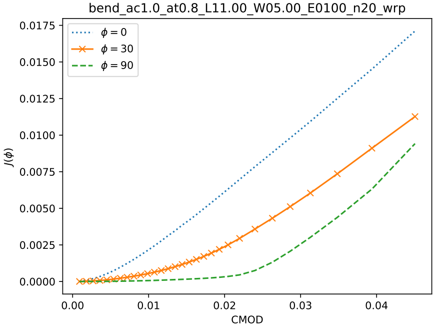
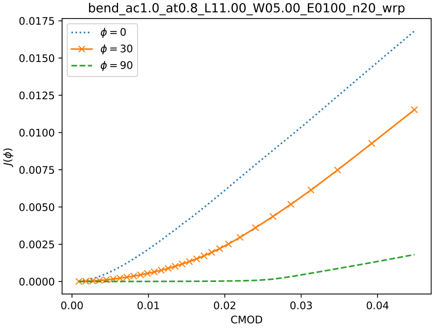


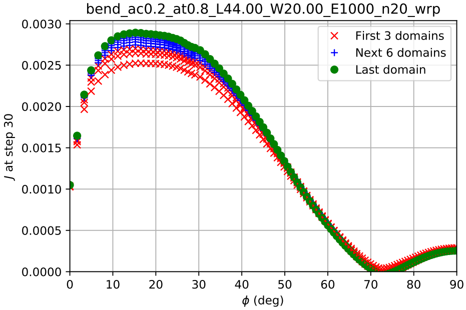
Plate deflection before and after addition of elastic boundary



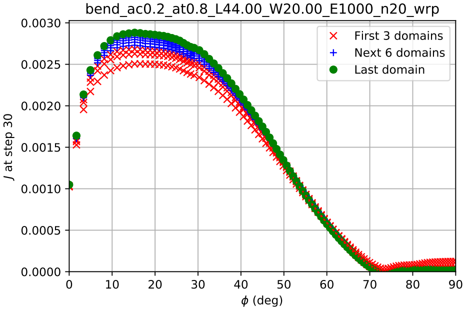
Before addition of elastic boundary



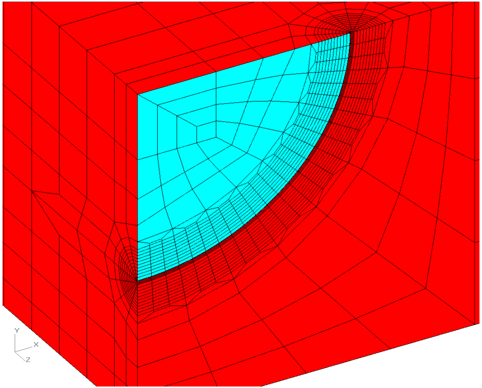
After addition of elastic boundary



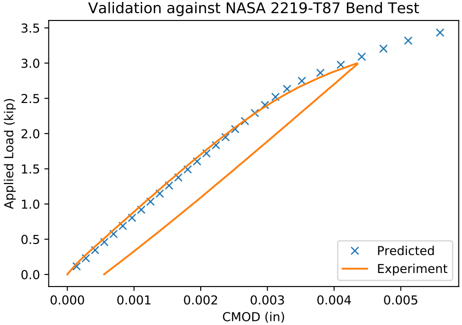
Before addition of elastic boundary



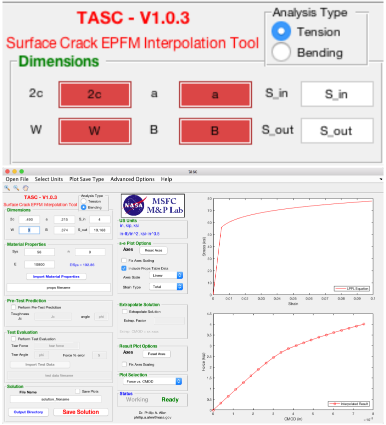
After addition of elastic boundary



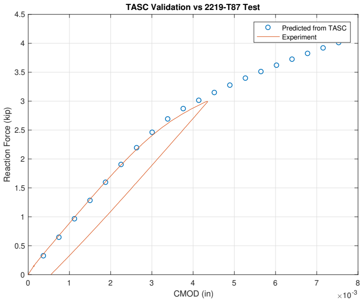
Crack front mesh of purpose-built experimental validation model



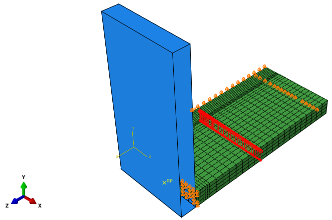
Comparison of predicted load and CMOD between purpose-built model and experiment



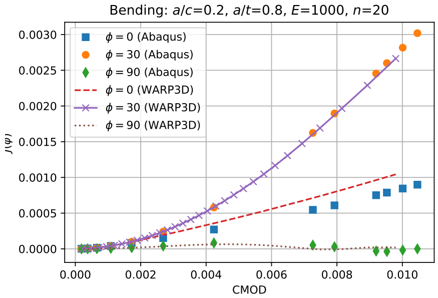
Modified TASC Program



Comparison of TASC and experiment



Sample Abaqus validation model

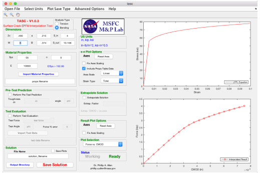


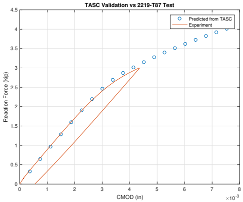
Sample J comparison between Abaqus and WARP3D

Conclusions and Recommendations for Future Work

Database of 600 elastic-plastic finite element results for surface cracks in bending

- More challenging than tension models
- Subset of results verified and validated against Abaqus and experimental data
- Modified TASC program
- Possible to satisfy requirements of ASTM E2899 for tension **and** bending without constructing purpose-built EPFM models
- Greatly reduces analytical burden on anyone doing ASTM E2899 tests





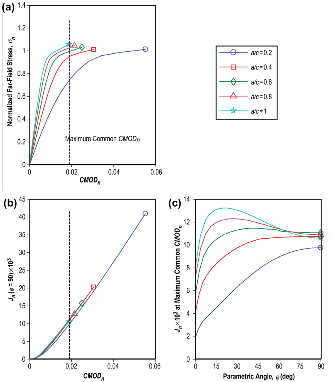
1. TASC and E2899
   - Finish integrating bend data into TASC, beyond load-CMOD validation
   - Additional values for material and/or crack geometry
   - Investigate other interpolation methods (piecewise cubic?)
   - Replace traction boundary conditions with rigid rollers and contact modeling
2. **More experimental data needed for these.**

Thanks to:

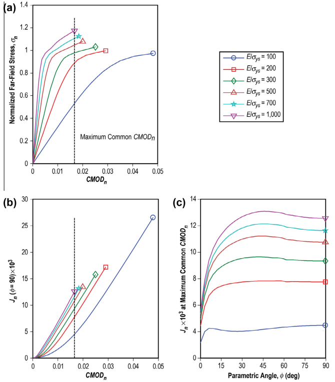
- Tennessee Tech University's:
  - Center for Manufacturing Research (Computer Aided Engineering)
  - Information Technology Services (High Performance Computing)
- Quest Integrity's FEACrack
- UIUC's WARP3D
- Dassault's Abaqus
- Anaconda's Python distribution

Appendix

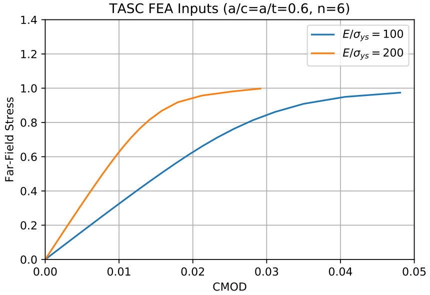
Initial Verification of Two Tension Cases from Allen and Wells (2014)

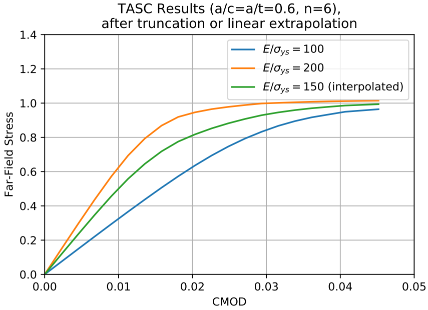


Gap in results for widest aspect ratios



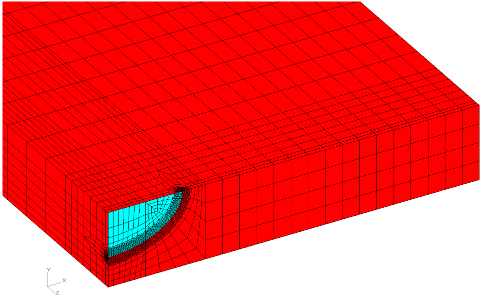
Gap in results for lowest E values



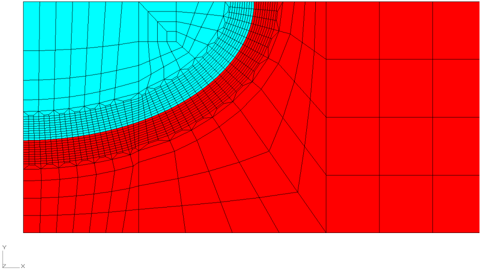


Raw FEA results used in TASC

Interpolated FEA results displayed by TASC



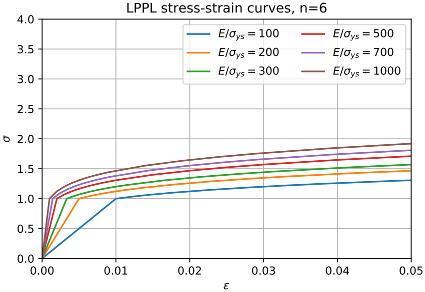
Isometric view of overall mesh



Detailed view of crack front

$$\frac{\epsilon}{\epsilon_{\text{ys}}} = \begin{cases} \dfrac{\sigma}{\sigma_{\text{ys}}}, & \epsilon \leq \epsilon_{\text{ys}} \\ \left(\dfrac{\sigma}{\sigma_{\text{ys}}}\right)^{n}, & \epsilon > \epsilon_{\text{ys}} \end{cases}$$

where $\epsilon_{\text{ys}} = \frac{\sigma_{\text{ys}}}{E}$.



Set of LPPL stress-strain curves

$$M = \frac{r_\phi \sigma_{\text{ys}}}{J}$$

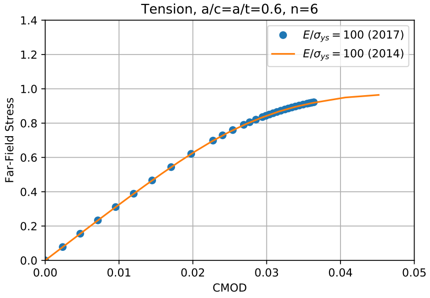
Applied displacement values for verification models

$\frac{E}{\sigma_{\text{ys}}}$	Displacement	ϕ	M using $r_{\phi a}$	M using $r_{\phi b}$
100	0.1028	30°	15.9833	36.4241
		90°	22.6234	15.0822
200	0.0550	30°	24.7288	56.3542
		90°	34.9604	23.3069

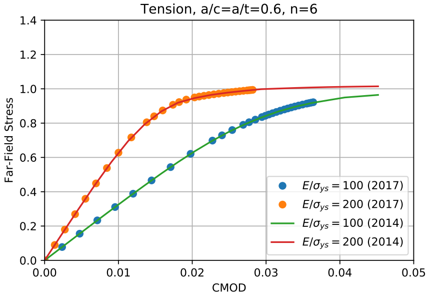
- 30 load steps
- `warp3d < file.inp > file.out`
- 21.6 minutes to solve on laptop, 2.2 minutes on HPC node

Python program

- run `packet_reader` to export displacements, forces
- extract node 1 z displacement, double to get CMOD
- identify nodes on $z = 0$ from input file
- extract z reactions from all identified nodes, sum to reaction force
- divide reaction force by plate cross section area to get stress



Verification of stress and CMOD relationship for first model



Verification of stress and CMOD relationship for second model