Thanks, Dr. Wilson.

Good afternoon, everyone.

The title of my research is Analysis of Surface Cracks in Plates Loaded in Bending under Elastic-Plastic and Fully-Plastic Conditions

I'll start with a introduction to a few fundamental concepts of fracture mechanics to provide some context,

then do a literature review that leads up to the motivation for this research,

followed by an overview of some related work I did earlier to demonstrate the techniques I'd end up using for the main part of the research.

Then, I'll discuss the plan for the new research, the results of that research, and some conclusions and recommendations for future work.

Fracture mechanics of a cracked body can be divided up into three main regimes:

Linear elastic

Elastic-plastic

and fully-plastic, or plastic collapse.

Each of these regimes are characterized by how much material near the crack tip has permanent plastic deformation:

For linear elastic, there's very little material that is plastically deformed.

For fully-plastic, a large region has plastically deformed.

And the elastic-plastic regime is in the middle.

Many high-performance, tough materials fall into the elastic-plastic regime. Materials in the linear elastic regime are often brittle, and materials in the fully-plastic regime severely deform once they reach their yield strength.

Though I've just described some materials as tough, the measured toughness is not purely a material property.

The measured fracture toughness of a sample, or of a real part, is really a property of both the material and the state of stress at the crack tip.

Very thick bodies exhibit a high level of constraint, and can exhibit very high stress for their plastic zone size. This tends to lead to conservative estimates of fracture toughness.

Very thin bodies with lower levels of constraint can show much higher fracture toughness values, and the measured values can vary, even for the same material. Here, fracture toughness is a property of both the material and the state of stress at the crack tip.

The state of stress around some cracks can get pretty complicated.

There's already been a lot of work done on geometries where the cracks go all the way through the part

Among the part-through crack geometries shown here, the semi-elliptical surface crack is one of the simplest, and is the subject of ASTM E2899

Handbook or curve-fit solutions exist for the other crack geometries, but only in the linear elastic regime.

For elastic-plastic materials, the NASA TASC program covers semi-elliptical surface cracks, but only for pure tension, not for bending.

So the primary goals of this research are:

A verified and validated set of bending models comparable to the TASC's tension models

A modified version of TASC that works for both tension and bending

And an investigation of two other estimation techniques for elastic-plastic fracture: EPRI and load separation

The best consensus solution for stress intensity factors for surface cracks in bending is partially given here.

It starts out simply enough, but the various correction factors just pile up on each other

and it's just curve-fit after curve-fit covering 16 equations.

Tension cases only require 10 equations, and neither of these cover elastic-plastic materials.

A failure assessment diagram is one way of predicting when material failure will occur.

A naive fracture model might assume that exceeding either a critical stress horizontally or a critical stress intensity vertically would cause failure.

Materials with plasticity make that more complicated.

And if a material has any strain-hardening properties, or if we have sufficiently low levels of constraint due to geometry or state of stress, the set of curves on the right is more typical.

One way to characterize constraint is a T stress acting in the x direction (that is, the direction of crack growth).

That stress will affect the size of the plastic zone, and varies with geometry and loading condition as seen in the figure on the right.

Specimens cracked on a single edge loaded in tension or bending have the most variation in T stress as cracks grow larger.

The elastic-plastic fracture parameter J was originally defined in terms of a contour integral in 2D, or a surface integral in 3D.

In a finite element analysis code, this can be converted to a version including stress, strain, and deformation values from all the elements surrounding a location along the crack front.

There's a lot more detail in that f function, but that's the high-level version of it.

One way of estimating elastic-plastic behavior is the EPRI method,

where linear-elastic and fully-plastic J components are summed to a total J value.

Under small loads, the linear elastic component dominates.

But as load increases, the fully-plastic component grows more rapidly and eventually contributes the vast majority of the final J value.

The elastic component of J can be related back to the stress intensity factor K, and the plastic component uses a limit load analysis and a power-law material model.

This was originally done for 2D crack geometries, but has been applied for 3D geometries since.

Load separation attempts to separate J into material and geometry components, similar to a separation of variables in differential equations.

It results in a single-specimen technique where J for any crack length can be predicted from a single test record using the same material.

It relies on load-displacement curves converging to fixed ratios versus a given reference curve shown in the left figure,

and being able to fit those ratios to a single curve shown in the right figure.

This was also originally applied to 2D geometries, but has later been used for 3D cracks.

So now we get to the current ASTM standard for measuring fracture toughness for flat plates with surface cracks, E2899.

On the left is an example specimen including a cross section of its semi-elliptical surface crack, and on the right is a specimen loaded into a four-point bending configuration.

In ASTM E2899, the first few steps to calculating stress conditions at the crack tip are purely related to experimental measurement.

For linear elastic conditions, a series of equations derived from experimental curve fits can be used to calculate K.

But for elastic-plastic conditions, no such series of equations exists to calculate J, and finite element analysis is required in the standard.

This last step is a major departure compared to other standards, and would normally require construction of models for every tested geometry and material.

By comparison, other ASTM standards like E8, E399, and E1820 require much less analysis, and are fundamentally experimental with some handbook, tabular, or curve-fitted equations to calculate.

Elastic-plastic finite element analysis is a huge departure from what people had expected from earlier standards.

The vast majority of ASTM standards focus on experimental techniques, with some basic graphical constructions or simple equations to verify the test was done properly

But ASTM E2899 requires finite element analysis for elastic-plastic materials, and this can be a burden on those performing these kinds of tests.

The NASA TASC program satisfies those analysis requirements, but only works for tension, not in bending.

Bending analysis is more complex than tension analysis: cracks can close, and the state of stress around the crack front makes for variable constraint conditions.

But in 2014, Phillip Allen at NASA released TASC, the tool for analysis of surface cracks.

TASC includes elastic-plastic results from 600 finite element analyses: all combinations of 20 normalized geometries and 30 normalized materials shown here.

It interpolates results for intermediate values, but only includes models for plates in tension.

No bending results are included, so work remained on satisfying the entire E2899 standard.

We always want our models or experiments to predict real-world behavior as closely as possible.

But there's always going to be some uncertainty and differences between the prediction and the actual behavior.

The root cause of differences can generally be lumped into one of two categories:

Either a measurement or calculation has limited repeatability or accuracy,

or we aren't using the right model or right procedure to include all the relevant physical effects.

Fixing problems with repeatability or accuracy is improving verification,

and fixing problems with selecting the right models and procedures is improving validation.

Some of the literature on verification and validation includes **favanesi1994** who checked NASCRAC against FLAGRO, FRANC2D, plus a range of experimental data,

wilson1995 identifying errors in NASCRAC and FLAGRO,

and **mcclung2012** verifying a K solution among FADD3D, FEACrack, and FLAGRO, and finding some anomalies at the free surface with FEACrack.

Time check: 10–12 minutes here.

It made sense to develop my modeling techniques for surface cracks in bending on existing results, and that's what's shown in this part.

I started at first with EPRI h_1 estimates for plates in tension.

This spans the same range of crack geometries as TASC, and followed the M.S. research for Eric Quillen, where his results consistently showed some odd anomalies in J and h_1 values at three locations along the crack front.

At the time, it was simplest to reconstruct Eric's models in a newer version of Abaqus using Python to create the models, and all the anomalous h_1 results disappeared.

I wasn't sure if the root cause of the anomalies was the version of Abaqus or something in Eric's input files.

But I did have improved methods for building reproducible FEA models by writing Python scripts instead of constructing purpose-built models interactively.

More recently, I adapted a few TASC geometries similar to those used by Sharobeam and Landes for load separation in surface cracks.

The left graph shows the stress versus the plastic component of CMOD.

Taking the most compliant model as the reference curve, the graph on the right shows the separation parameters calculated from dividing each curve by the reference curve,

and they show the constant values expected from having enough plastic deformation in the models

Plotting the separation parameters against the effective length of the uncracked ligament, the results on the right figure lie near a linear fit similar to the one in the left figure.

Then I moved on to digging into the internals of the TASC model database,

seeing if I could make models that would match two of the ones from the database,

and a third model to test the limits of the interpolation method.

On this figure, the lines represent TASC results for $E = 100$ and $E = 200$, with an interpolated result for $E = 150$.

The circles represent my model results at each modulus, and you can see that the results are identical for $E = 100$ and $E = 200$, and that the interpolated TASC result is a few percent more compliant than the actual results from the same modulus.

Time check: be at the 15–17 minute mark here.

So now we get to the plan for the new research work

The dimensions on each plate follow the same rules.

Each model represents 1/4 of the full experimental plate, and has thickness of 1 inch.

Model width is 5 times larger than either the crack half width or the plate thickness, whichever is larger.

The plane where the inner rollers would appear will be a distance W from the center, and the line representing the outer rollers will be spaced at twice that distance.

The whole plate will extent 10% past the outer roller location.

All these models will start from a single input file for the FEACrack program, and a Python script will run FEACrack with command-line parameters to adjust dimensions and write a new input file for WARP3D for each of 20 geometries.

After that, the same Python script can adjust material properties in the 20 WARP3D input files and write a new input file for each of 30 materials.

The TASC literature indicated models were loaded until J values were high enough to drive this dimensionless deformation formula below 25,

but looking at the full set of M values recorded in the TASC database shown in the left figure, it's clear many of the models didn't reach that deformation level.

The real criteria for loading is to load the model heavily enough to straighten the J-CMOD curves shown in the right figure so we can accurately extrapolate beyond the applied load.

As a result, since we have a full set of J-ϕ-CMOD data after running a model, my plan was to increase loads until the two criteria shown here were met at $\phi = 30^\circ$.

This ensures that the last 40% of the J-CMOD curve is consistently straight, and any major curvature is confined to the first 60% of the range.

Early versions of satisfying this load criteria for tension models used displacement boundary conditions, and some convenient optimizations could take place.

Because we're effectively adjusting strain to match a target stress, the left stress-strain curve can be transformed to the form on the right, and we can do a secant method to find the strain required to drive the transformed stress value to zero.

Just move on a tangent at wherever we are on the transformed curve, and we quickly end up at the desired strain value.

Unfortunately, the bending models have to operate with traction boundary conditions, where we effectively adjust applied stress to match a target strain.

The secant method won't work, as our first step would run us way off the right side of the transformed graph.

So we just have to take small steps and stop whenever we see the curve has straightened out enough.

Since we're going to spend a lot of time finding J values for the models, it makes sense to do some checks to ensure the J values are accurate.

First, a subset of the models will be run in Abaqus with identical meshes.

Second, as J values can be calculated for any set of elements surrounding the crack front, those J values should converge to a final result as we add more elements and capture more of the strain energy and work done inside the plastic zone.

For modifications to the TASC program, the goal is to make the smallest set of changes required to support bending.

We can't break tension results, and most of the code assumes a certain arrangement of data in the database.

So we'll make a results_bending database alongside the existing results database.

Any equations that are only applicable for tension will be replaced with a check for which type of analysis we're running, and then picking the right equation

There shouldn't need to be any changes to the interpolation parts of TASC

and ideally, we'll be able to reproduce an experimental load-CMOD curve from a four-point bend experiment with good accuracy.

Additionally, I want to do some investigation into both EPRI h_1 and load separation.

For EPRI, we've got some prior results in the literature for 2D geometries and surface cracks in tension, but not in bending.

Same for load separation. Having 20 different crack geometries for each material means I've got plenty of data points for the linear fit.

So how did all of this turn out compared to the original plan?

During the J convergence study, some of the models had local minima in the J values at weird locations, like the one on the right.

They did converge well as we used more elements around the crack front, but converged to unexpected values sometimes.

Looking at how the plate deformed,

where orange is basically zero deflection,

red is positive deflection, and

yellow and below are negative deflections,

it's clear that regions deep in the crack are pushing out past the symmetry plane, which is physically impossible.

This never happens on tension models, as the plate cross section is always being stretched. But in bending, the underside of the plate is effectively being compressed, and it's pushing some of the cracked material along with it.

So an elastic boundary was added to the symmetry plane.

A rigid boundary plane would be more common, but WARP3D doesn't support those.

The effect of the boundary on displacement can be seen here, where now all the plate material stays below the symmetry plane.

In terms of J values, the plane has no effect toward the free surface (the solid and dotted lines),

but greatly reduces the J values seen deep in the crack (the dashed line)

Looking at the effect on J around the crack front, it's clear this removed the local minimum for J that was the original symptom of the problem.

Next up is making sure that FEACrack and WARP3D can reproduce a load-displacement curve from a four-point bend experiment.

A plate model with the right material properties and a crack of the correct depth and aspect ratio was created, and the blue x marks show its load-CMOD curve.

The solid line represents a NASA four-point bend experiment, and matches the prediction within a few percent at most.

Finally, in terms of TASC work, the very minimal user interface changes can be seen in the top left,

where a radio button switching between tension and bending analysis has been added,

as well as two text entries for roller spans.

After entering in the correct crack geometry, plate dimensions, and material properties, the bottom left figure shows the predicted load-CMOD curve.

The same curve is shown on the right and compared to the original experimental data.

Again, the prediction is within percentage points all along the experimental curve.

For validating J values from WARP3D against Abaqus, you can see a representative model on the left figure.

A rigid plane has been added similar to the elastic boundary added in WARP3D.

Looking at some of the J values from Abaqus shown with solid markers, versus the WARP3D results shown with lines,

the J values are coincident except at the free surface where $\phi = 0$, and even those are pretty close.

For the EPRI h_1 work using Abaqus' automatic check for fully-plastic behavior,

we restrict the fully-plastic check to just a group of elements around the crack front.

Tension models often would use the uncracked portion of the symmetry plane, but in bending, too many of these elements would have very low bending stress applied.

The J values from Abaqus have the same trend as the ones from WARP3D, but they're about twice as large due to extra load being applied to satisfy the fully-plastic check.

The h_1 values scale with J as expected.

In theory, h_1 values should not be dependent on E, but the WARP3D h_1 values vary across E values to some degree.

In Abaqus, the applied stress values increase with E, and that helps drive its h_1 values progressively lower.

On the left is one set of separation parameters for cracks with $\frac{a}{t} = 0.6$.

They're nearly constant across the range of CMOD values.

On the right is all 20 separation parameters plotted against the effective uncracked ligament length used previously.

They obviously don't lie on a single line. At best, they're self-similar at a given depth, but not across depths.

Looking at 14 tension models including at least two examples of each depths and aspect ratio, they show the same trend:

Each group of points at a given depth follow a trend, but trends do not extend across depths.

Looking at the points closest to the original **sharobeamlandes1994** results, it may be that their data sets were too sparse to see the more complete trend.

Time check: be at 30–35 minute mark here.

So what conclusions can we draw, and where do we go from here?

First and foremost, we now have a database of 600 elastic-plastic FEA results for surface cracks in bending.

These models had some extra challenges compared to tension models, and those were overcome.

A subset of the results have been verified and validated against Abaqus models and against experimental data

The TASC program has been modified to support bending, at least as far as predicting load-CMOD curves.

All of this means it's now possible to satisfy all the requirements of ASTM E2899 in both tension and bending, without building custom models for each test,

and this greatly reduces the analytical burden on anyone performing tests under ASTM E2899.

In the area of EPRI h_1, it looks like there's some dependence on E which was unexpected.

The check for fully-plastic conditions in Abaqus is not magic.

And in the area of load separation, examining a broader set of models indicated that a single-specimen technique is more complicated than first thought,

as separation parameters don't lie on a single curve fit.

For future work related to this research,
there's still some work to be done in TASC modifications and testing, as I didn't modify anything beyond what was necessary to validate a load-CMOD curve
some additional data points for crack geometry and material models may be worthwhile,
as well as seeing if an alternative interpolation method will provide better agreement with experimental results.
As WARP3D supports contact modeling, it may be worth replacing the traction conditions used in bending models with contact modeling of rigid rollers to more closely match experimental conditions
For EPRI h_1 estimation, we need some more detailed examination of its limitations, or where I may have a mistake in my current models
Limitations could be related to crack shape, tension versus bending, or material properties.
Finally, for load separation, there may be an alternative length measure other than the one from R6 that will collapse the $S_{i,j}$ values onto a single curve.
Now for either of the EPRI h_1 or the load separation items, each of those could be someone's dissertation on their own, especially if experiments are involved.

That's it for the presentation.

I'd like to thank Dr. Chris Wilson and my committee members: Dr. Brian O'Connor, Dr. Sally Pardue, Dr. Guillermo Ramirez, and Dr. Dale Wilson.

Facilities from both the Center for Manufacturing Research and Information Technology Services were instrumental at different phases of this research.

Dr. Phillip Allen and ASTM Committee E08 provided a foundation for my research, and hopefully a permanent home for its results.

And finally, my friends and family who have been consistently encouraging through this whole thing.

Any questions?