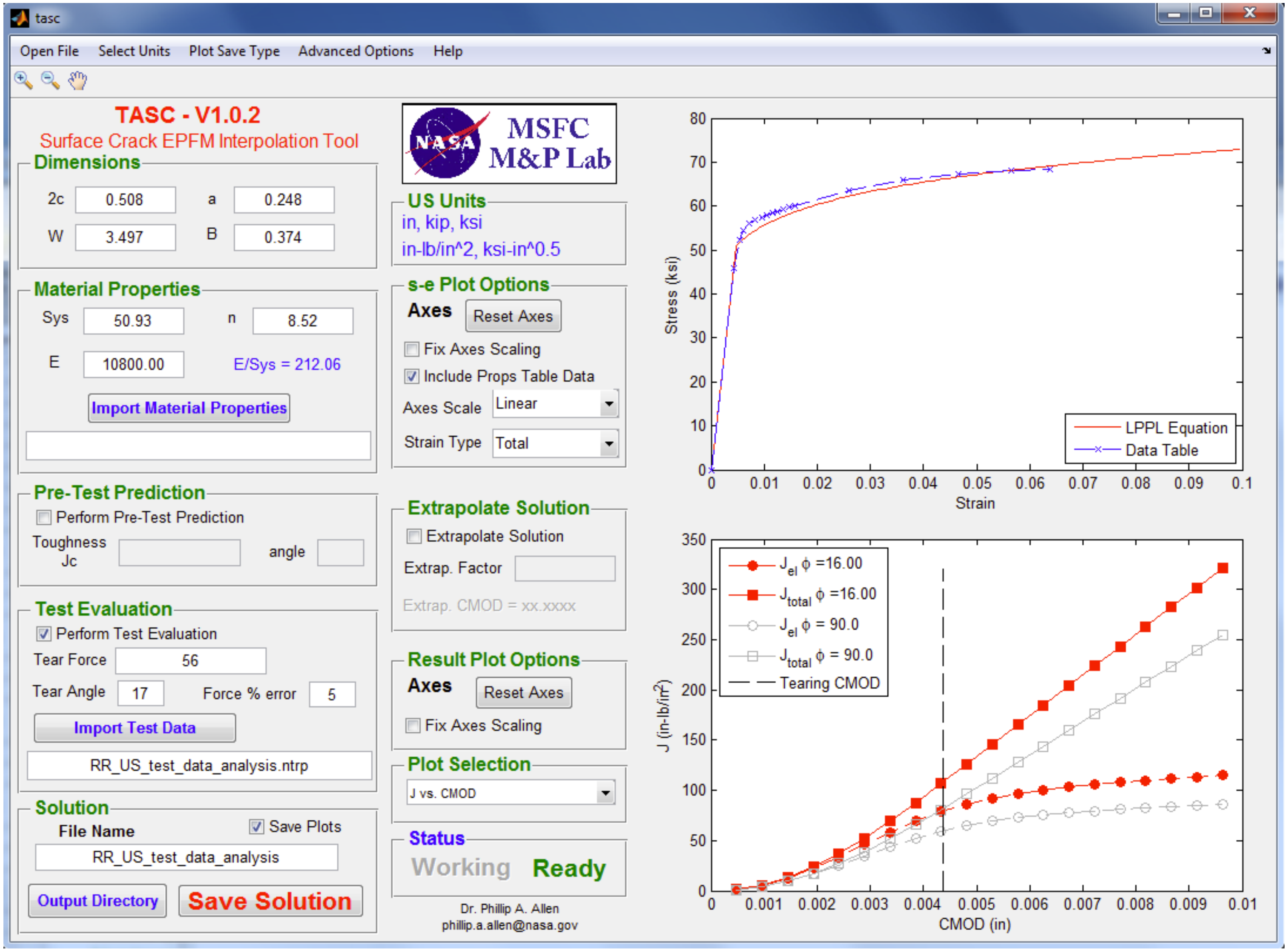
TASC – V1.0.2 (Tool for Analysis of Surface Cracks)

Interpolated Elastic-Plastic J-Integral Solutions

Phillip Allen – NASA MSFC – Phillip.a.allen@nasa.gov

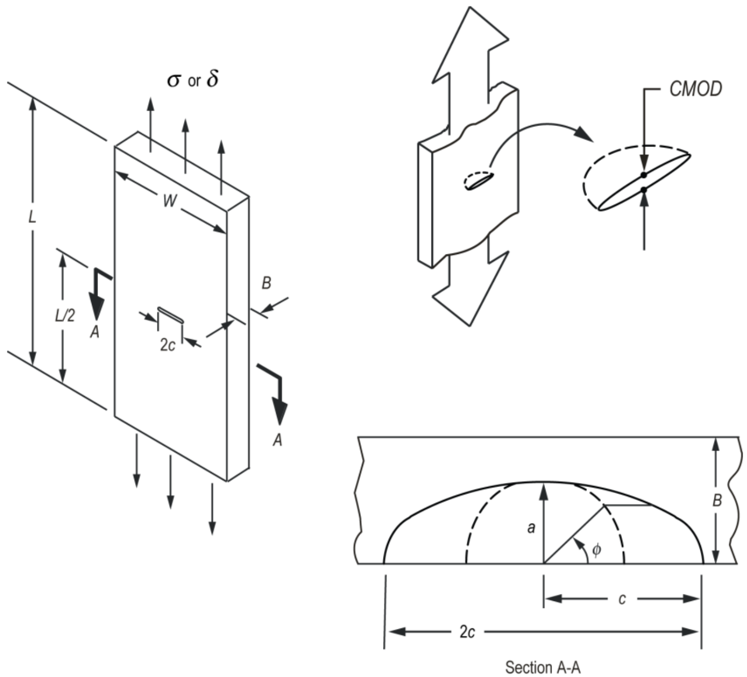
June 2014



Introduction

Tool for Analysis of Surface Cracks (TASC) is a computer program created in MATLAB® to enable easy computation of nonlinear *J*-integral solutions for surface cracked plates in tension by accessing and interpolating between the 600 nonlinear surface crack solutions documented in NASA/TP–2011-217480*. The only required inputs are the surface crack dimensions (2*c* and *a*), plate cross-section dimensions (*W* and *B*), and linear plus power law (LPPL) material properties of elastic modulus, *E*, yield strength, σ_{ys} or Sys, and strain hardening coefficient, *n*. With the geometry and material parameters entered, TASC interpolates to the appropriate $J(\phi)$ vs. crack mouth opening displacement (CMOD) and far-field tension stress, σ, vs. *CMOD* solution, providing the full solution as CMOD ranges from zero out to the CMOD limit of the solution space for the given input parameters. (Note: ϕ is the parametric crack front angle.) TASC provides interpolated solutions over a wide range of crack shapes and depths (shape: $0.2 \le a/c \le 1.0$, depth: $0.2 \le a/B \le 0.8$) and material flow properties (elastic modulus to yield ratio: $100 \le E/\sigma_{ys} \le 1000$, and hardening: $3 \le n \le 20$). With surface crack test design and analysis in mind, TASC also has several other useful features such as:

1. material property import capability with automated material constant fitting,
2. pre-test prediction capabilities based on a critical *J*-integral value and critical ϕ location,
3. test record force, *P*, vs. *CMOD* evaluation and comparison with analysis,
4. the ability to review multiple result plots such as $J(\phi)$, *J* vs. *CMOD*, and deformation limit comparisons, and,
5. the ability to save the solution, input, and plot files.



LPPL Equations

$$\frac{\varepsilon}{\varepsilon_{ys}} = \frac{\sigma}{\sigma_{ys}} \quad \varepsilon \le \varepsilon_{ys}; \qquad \frac{\varepsilon}{\varepsilon_{ys}} = \left(\frac{\sigma}{\sigma_{ys}}\right)^n \quad \varepsilon > \varepsilon_{ys}$$

* Details of the solution space, interpolation methods, and nomenclature are documented in NASA/TP-2013-217480, *Elastic-Plastic J-Integral Solutions for Surface Cracks in Tension Using an Interpolation Methodology*, which is available for download from the NASA Center for Aerospace Information (CASI) at <http://www.sti.nasa.gov>.

Revision History

June 2014 - Version 1.0.2

- interp_solution_SCGui_CMOD_log_int.m – Corrected error in code that sometimes prevented interpolation to first load step results for very small initial increments of CMOD. In rare cases, this error created “NaN” results for the first load step of the interpolation solution and prevented the pretest prediction from functioning properly.
- plt_crk_front_condition_int.m – Added this new file that creates a plot of the crack front constraint and deformation conditions as a function of loading. This plot corresponds to Figure 8 in ASTM E2899.
- Changed three plot routines to plot the ϕ (90°) results as light gray to differentiate the 90 results from the tearing prediction results. Updated code in: plt_crk_front_condition_int.m, plt_deform_epfm_fea_int.m, and plt_J_CMOD_fea_int.m.

February 2014 - Version 1.0.1

- Released standalone executable versions of TASC V1.0.1 for Windows® 64-bit and Mac OS X® 64-bit, and the TASC source code under the NASA Open Source Agreement. Files are posted the NASA TASC Sourceforge project at http://sourceforge.net/projects/tascnasa/.
- tasc.m – Added a start-up box that requires users to read and accept the NASA Open Source Agreement before using TASC.
- create_summary_table.m – Added additional output in the results summary file including: *E*, *a*, 2*c*, $r_{\phi a}$, and $r_{\phi b}$.
- plt_force_CMOD_interp2_int.m – Added $J(\phi)$ value displayed on plot for test analysis prediction.
- plt_stress_CMOD_interp_int.m – Added $J(\phi)$ value displayed on plot for test analysis prediction.

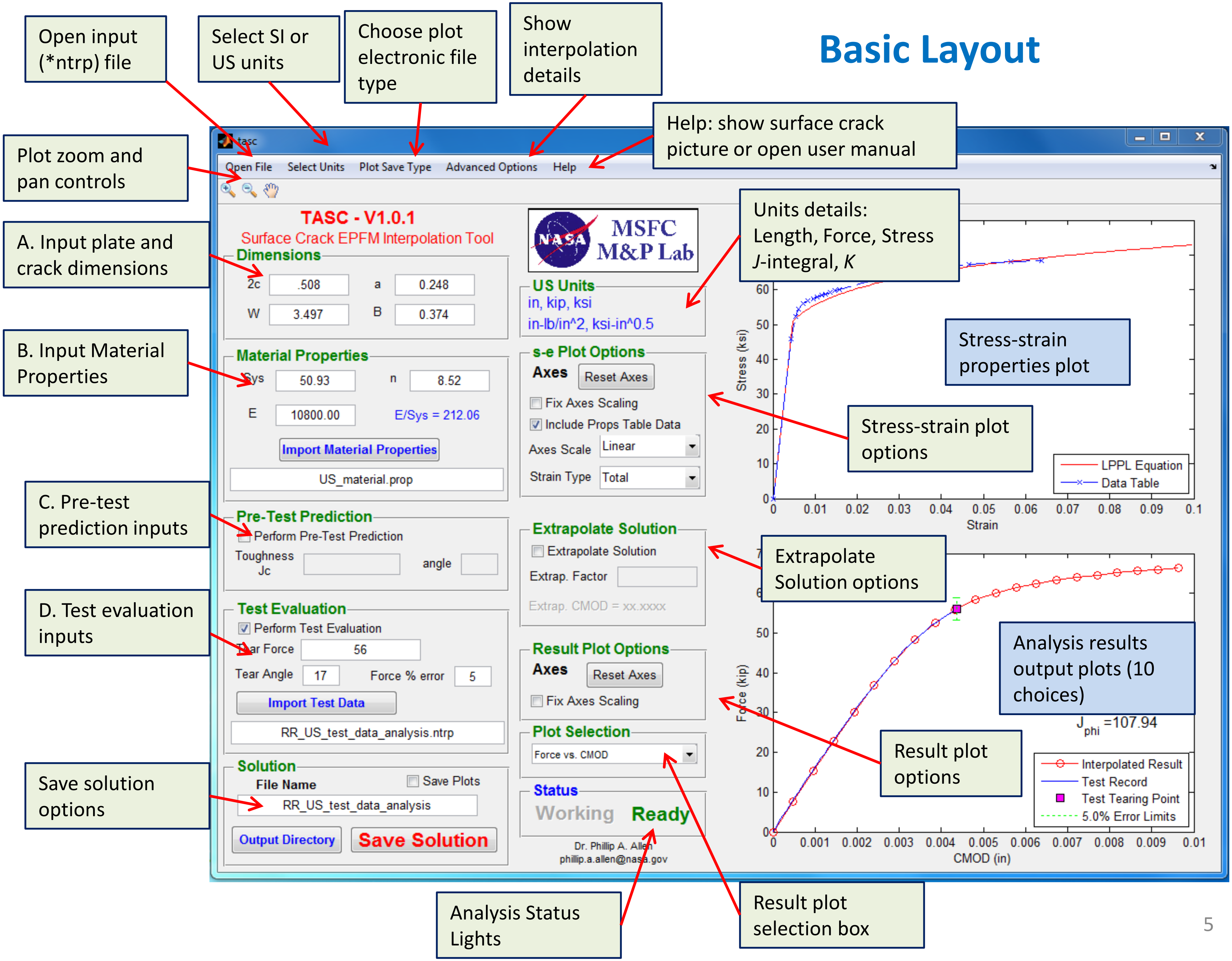
November 2013 - Version 1.0.0

- Released standalone executable versions of TASC V1.0.0 for Windows® 64-bit and Mac OS X® 64-bit under the NASA Public Release Agreement.

Some Important Standalone Executable Details

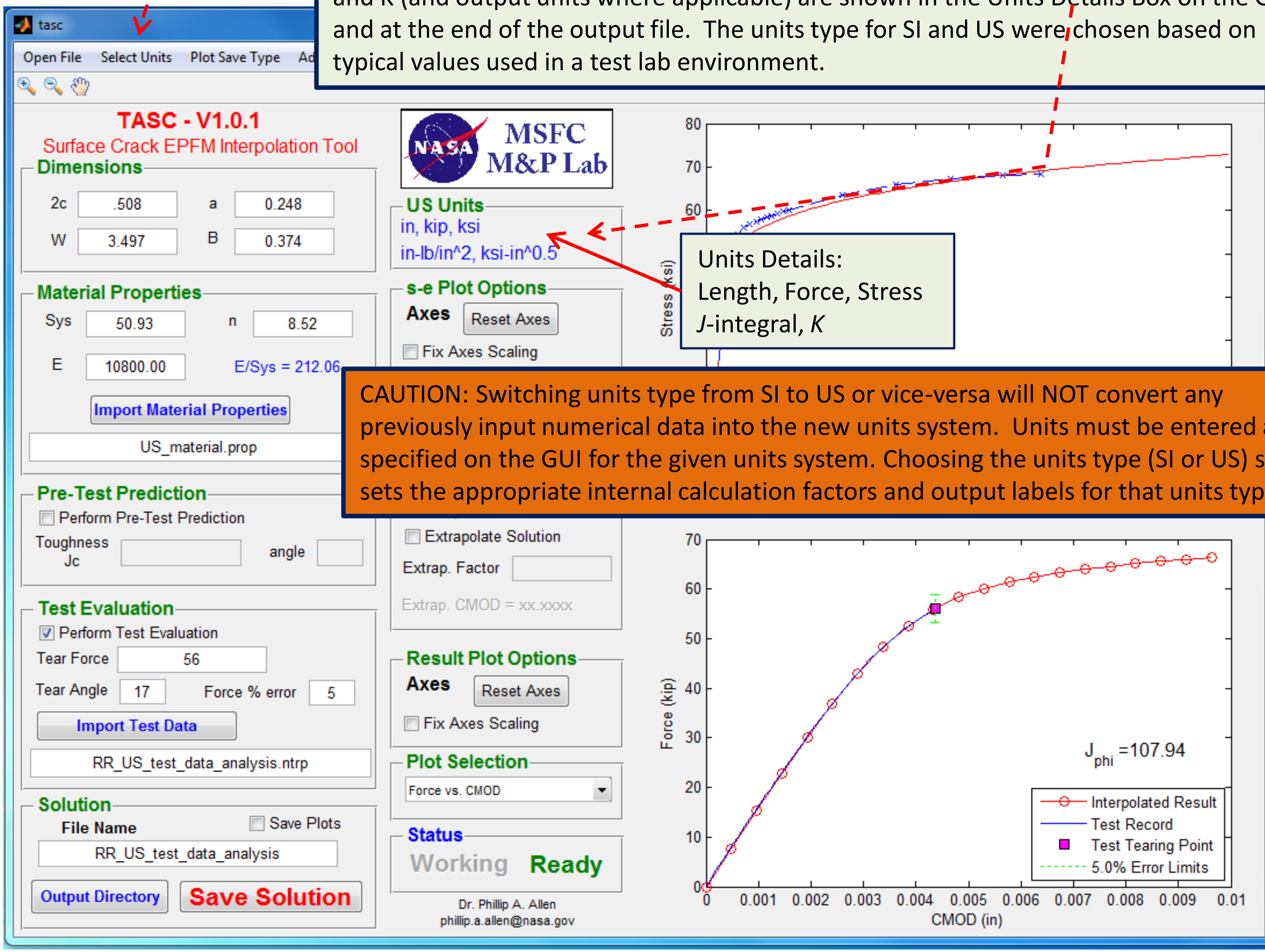
- TASC is a standalone executable program available for Windows® 64-bit and Mac OS X® 64-bit. Individual users of the standalone executables do not need a MATLAB license due to the royalty-free MATLAB Complier Runtime (MCR) distribution provided with the program installation package.
- Installation details are in the *readme.txt* file included in the distribution package. The correct version of the MCR package must first be installed on each computer on which you want to run TASC. The correct MCR installation file (*MCRInstaller.exe*) is included in the distribution package. The TASC standalone executable file (TASC_V1P0P2.exe for Windows and TASC_V1P0P2_mac.app for Mac) can be located in any convenient place on your computer. Double click *TASC_V1P0P2.exe* for Windows or execute *run_TASC_V1P0P2_mac.sh* from the terminal for Mac to start TASC.
- TASC takes 10 to 60 seconds to start, depending on your computer because the MCR files are essentially starting a version of Matlab in the background. Unfortunately using the Matlab graphical user interface (GUI) system, there is no practical way to have a start up splash screen to show you the program is starting; please be patient. Once the TASC GUI is running, each elastic-plastic solution is computed in seconds.
- The first time you run TASC, a start up window will appear stating "*I have read and accept the terms of the NASA Open Source Agreement included in the TASC distribution package*." You must select "Yes" to run TASC. Selecting "Yes" creates a small text file, TASC_lic.txt" in your executable directory. You will not be asked to accept the agreement again unless you move or copy the executable file to another directory.
- Example TASC input files (*.ntrp), material property files (*.prop), and test data files (*.txt) for both US and SI units are included in the *Distribution_Example_Files* directory. The use and description of these files are explained later in this manual.
- The GUI was created and formatted on the Windows platform. As a result, on the Mac platform the GUI appearance for items such as font sizes are not optimized. In addition the Mac version does not have the capability to save an Excel summary of the results in *.xlsx format. The results are output in text format and drop cleanly into Excel if so desired. Also the Mac version does not have the ability to save plot images in the *.emf format so instead defaults to the *.tiff format.
- Background information on surface crack tension testing including deformation limits and critical ϕ calculation methods is documented in ASTM E2899, *Standard Test Method for Measurement of Initiation Toughness in Surface Cracks Under Tension and Bending*, which is available from ASTM, <http://www.astm.org>.

Basic Layout



Important Units Considerations

The underlying solution database is dimensionless. Therefore TASC relies on the user to input a consistent set of units to get output in the expected and desired units. The user must first decide on SI or US units by using the Select Units menu or by specifying the units type in the *.ntrp file. The expected units type for length, force, stress, J-integral and K (and output units where applicable) are shown in the Units Details Box on the GUI and at the end of the output file. The units type for SI and US were chosen based on typical values used in a test lab environment.



Units Details:
Length, Force, Stress
J-integral, *K*

CAUTION: Switching units type from SI to US or vice-versa will NOT convert any previously input numerical data into the new units system. Units must be entered as specified on the GUI for the given units system. Choosing the units type (SI or US) simply sets the appropriate internal calculation factors and output labels for that units type.

Get started and perform analysis

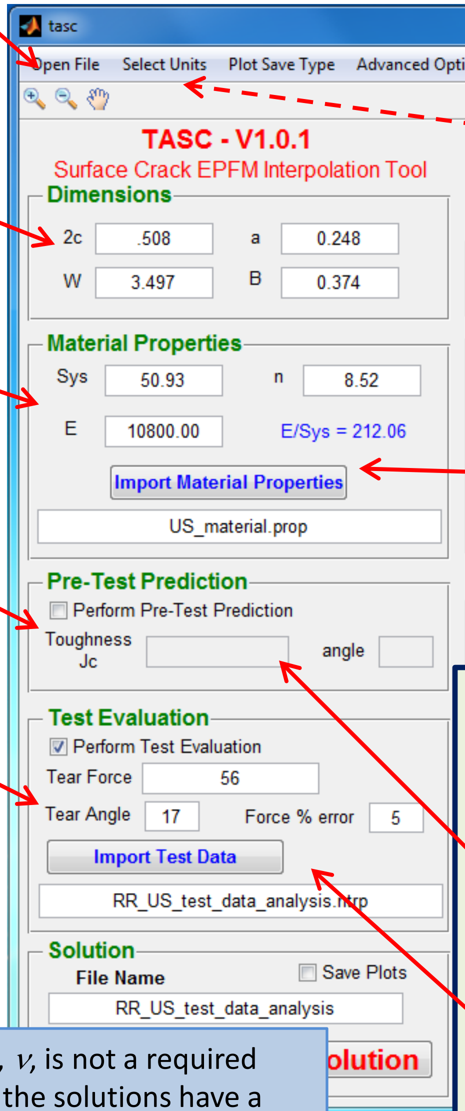
Open input (*ntrp) file

A. Input plate and crack dimensions

B. Input Material Properties

C. Pre-test prediction inputs

D. Test evaluation inputs



Two choices to begin analysis:

1) Open a *.ntrp file. Data boxes will be filled in and analysis performed based on information given in *.ntrp file
2) First select units system, and then manually type in values in boxes (A) and (B). Once the values are entered and no errors are found, the analysis will be performed.

Material properties for box (B) can be imported using the "Import material properties button." The material properties file can contain:

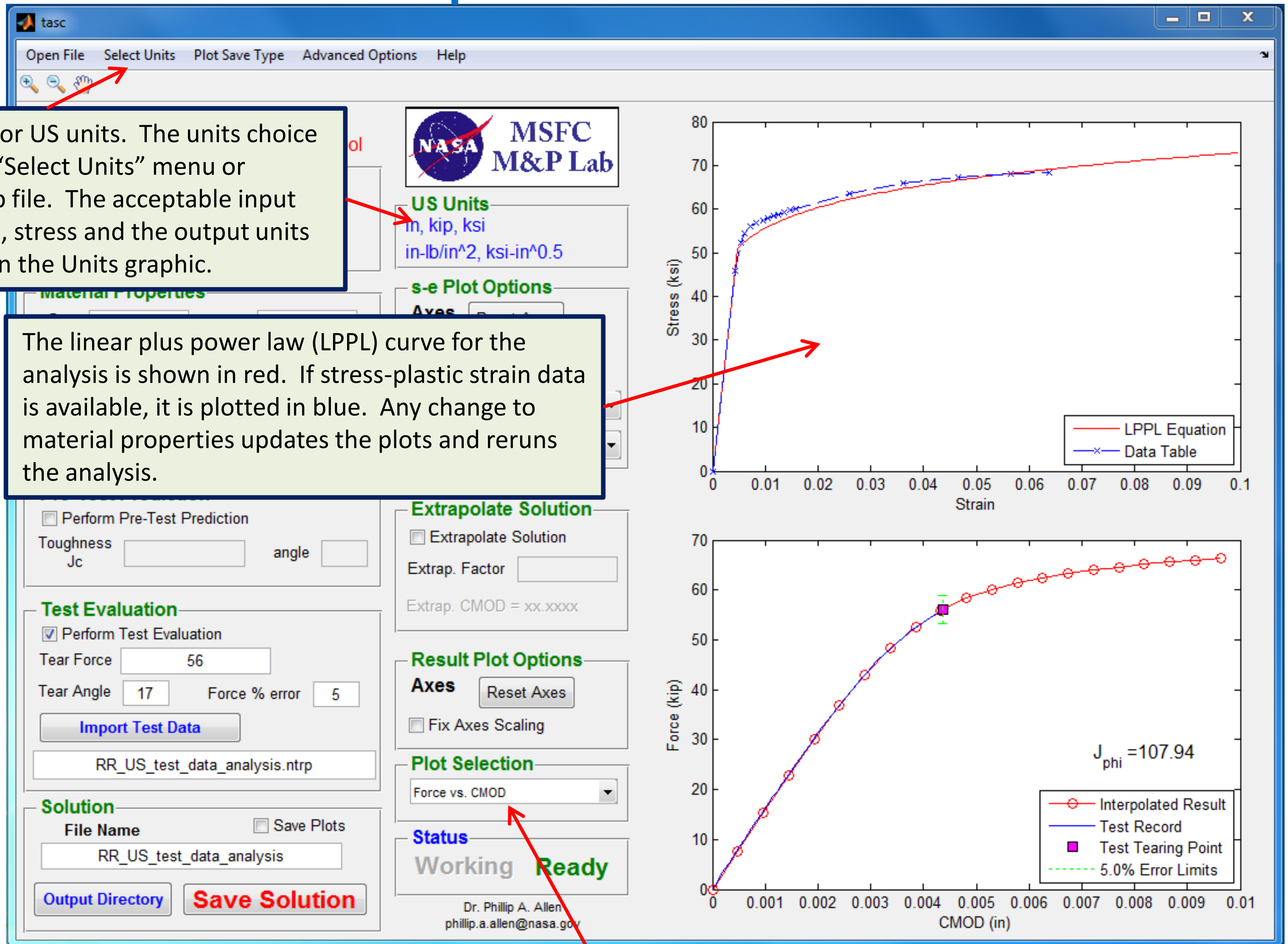
1. Sys, n and E, or
2. the file can contain E and a stress vs. plastic strain table. If a table is present, the code will set Sys as the average of the first 3 stress values in the table and determine a "n" value by fitting the stress-plastic strain curve. This provides starting approximations for Sys and n that can then be adjusted as necessary by the user.

NOTE: Poisson's ratio, ν, is not a required material input. All of the solutions have a fixed value of $\nu = 0.30$ from the interpolated solutions data set.

There are three independent analysis options:

1) A elastic-plastic solution with no test data evaluation is performed by filling in (A)& (B) and not marking (C) and (D) checkboxes. This gives you esentially all the outputs you would get from running a nonlinear FEM.
2) Choose (C) checkbox and fill in estimations for the critical toughness, Jc, and the critical crack front angle to get a pre-test prediction of the tearing force, CMOD, and crack front conditions. This option is intended for pre-test planning and design.
3) Choose (D) to perform a test analysis. Fill in the tearing force and tearing crack front angle and import the force-CMOD test record. The code will attempt to find the test tearing CMOD value corresponding to the tear force that is input. If the analysis reaction force corresponding to the tearing CMOD is within 5% of the test tearing force, the code will perform several calculations and report out tearing analysis results for your test. The % error allowed can be changed in the Force % error box.

Explore results

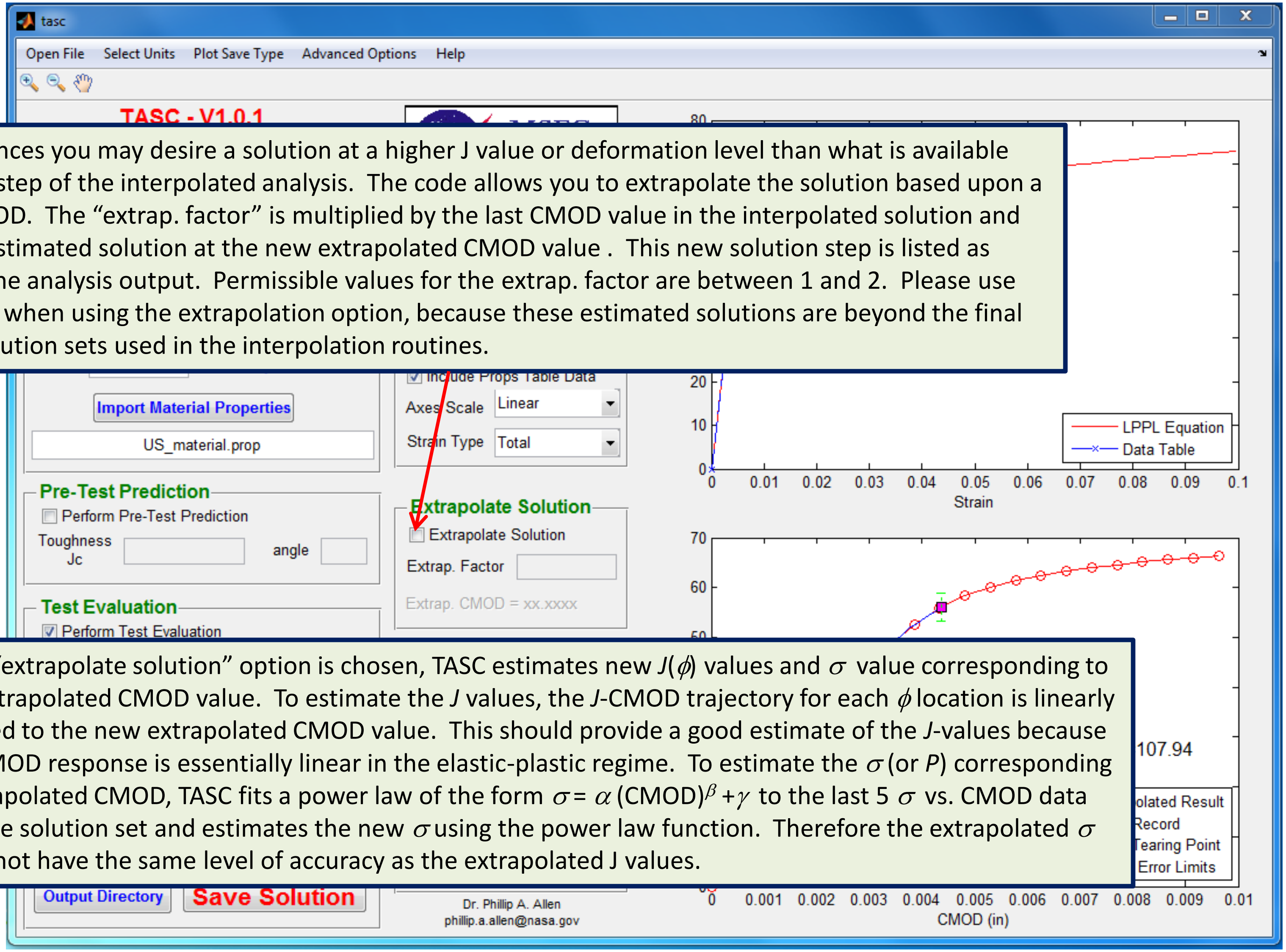


The code supports SI or US units. The units choice is made through the "Select Units" menu or specified in the *.ntrp file. The acceptable input units for length, force, stress and the output units for K and J are given in the Units graphic.

The linear plus power law (LPPL) curve for the analysis is shown in red. If stress-plastic strain data is available, it is plotted in blue. Any change to material properties updates the plots and reruns the analysis.

One of 10 plots can be picked from this list. Any change to the GUI values reruns the analysis and updates the plots.

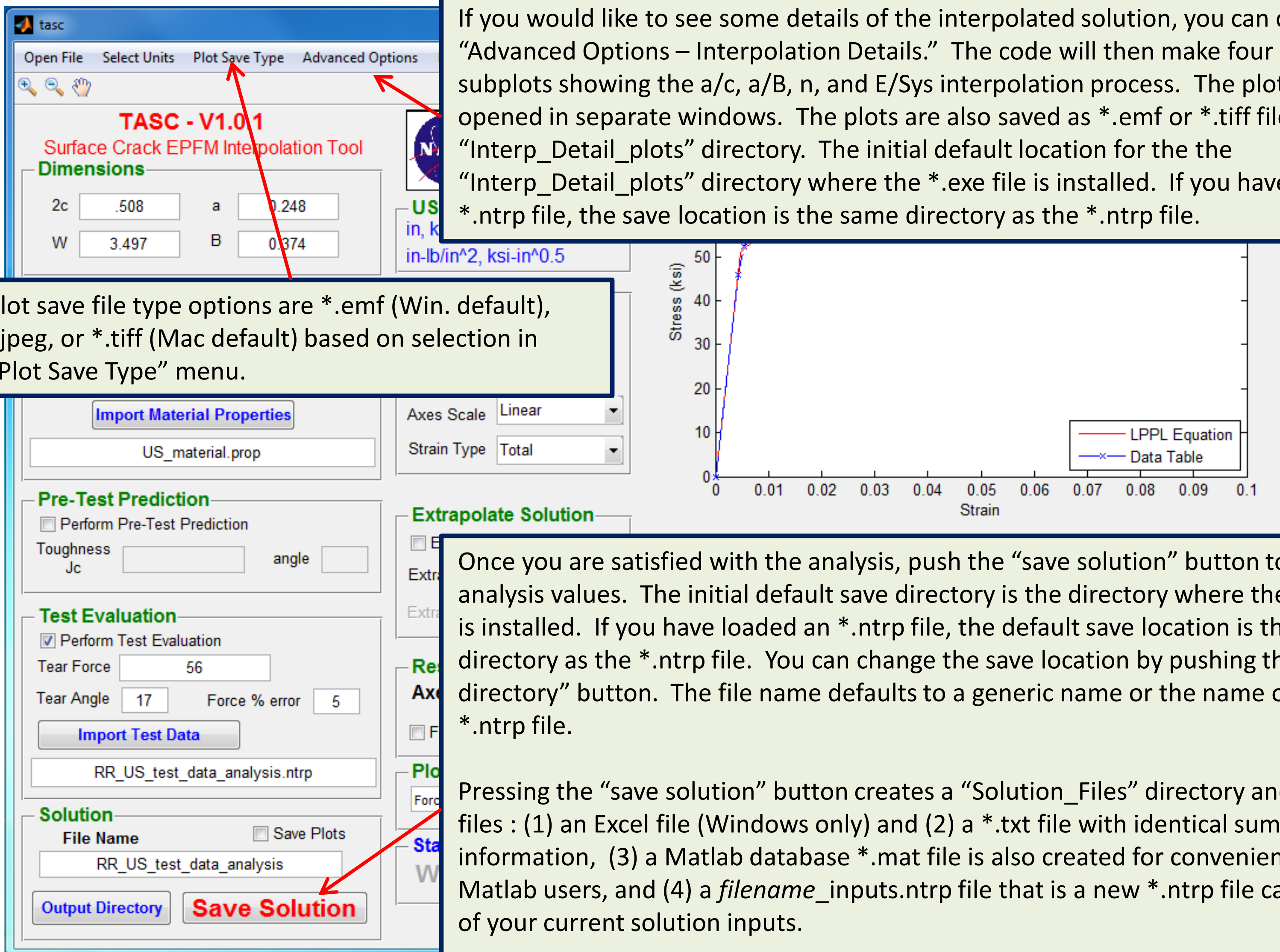
Extrapolate Solution Option



In some instances you may desire a solution at a higher J value or deformation level than what is available from the last step of the interpolated analysis. The code allows you to extrapolate the solution based upon a factor on CMOD. The “extrap. factor” is multiplied by the last CMOD value in the interpolated solution and provides an estimated solution at the new extrapolated CMOD value . This new solution step is listed as “step 21” in the analysis output. Permissible values for the extrap. factor are between 1 and 2. Please use extra scrutiny when using the extrapolation option, because these estimated solutions are beyond the final converged solution sets used in the interpolation routines.

When the “extrapolate solution” option is chosen, TASC estimates new $J(\phi)$ values and σ value corresponding to the new extrapolated CMOD value. To estimate the *J* values, the *J*-CMOD trajectory for each ϕ location is linearly extrapolated to the new extrapolated CMOD value. This should provide a good estimate of the *J*-values because the *J* vs. CMOD response is essentially linear in the elastic-plastic regime. To estimate the σ (or *P*) corresponding to the extrapolated CMOD, TASC fits a power law of the form $\sigma = \alpha\,(\text{CMOD})^{\beta} + \gamma$ to the last 5 σ vs. CMOD data points in the solution set and estimates the new σ using the power law function. Therefore the extrapolated σ value may not have the same level of accuracy as the extrapolated J values.

Save results



If you would like to see some details of the interpolated solution, you can choose "Advanced Options – Interpolation Details." The code will then make four sets of subplots showing the a/c, a/B, n, and E/Sys interpolation process. The plots are opened in separate windows. The plots are also saved as *.emf or *.tiff files in a "Interp_Detail_plots" directory. The initial default location for the the "Interp_Detail_plots" directory where the *.exe file is installed. If you have loaded an *.ntrp file, the save location is the same directory as the *.ntrp file.

Plot save file type options are *.emf (Win. default), *jpeg, or *.tiff (Mac default) based on selection in "Plot Save Type" menu.

Once you are satisfied with the analysis, push the "save solution" button to save the analysis values. The initial default save directory is the directory where the *.exe file is installed. If you have loaded an *.ntrp file, the default save location is the same directory as the *.ntrp file. You can change the save location by pushing the "output directory" button. The file name defaults to a generic name or the name of the *.ntrp file.

Pressing the "save solution" button creates a "Solution_Files" directory and stores 4 files : (1) an Excel file (Windows only) and (2) a *.txt file with identical summary information, (3) a Matlab database *.mat file is also created for convenience for Matlab users, and (4) a *filename*_inputs.ntrp file that is a new *.ntrp file capturing all of your current solution inputs.

If "Save Plots" is checked, the code also creates a "Plot_Files" directory and saves files for all of the plots in the analysis.
The filenames all have the solution filename as a prefix.

Output file

Input Summary

Continues every 2 degrees to phi = 90

index	a/B	a/c	n	E/Sys	W	B	Sys	E	a	2c						
0.66_0.98_9_10800	0.66	0.98	8.52	212.1	3.497	0.374	50.93	10800	0.248	0.508						
**	**	phi	0	2	4	6	8	10	12	14	16	18	20	22	24	26
step	stress	CMOD	J0	J2	J4	J6	J8	J10	J12	J14	J16	J18	J20	J22	J24	J26
1	5.85E+00	4.81E-04	1.70E+00	1.72E+00	1.73E+00	1.68E+00	1.64E+00	1.59E+00	1.55E+00	1.52E+00	1.49E+00	1.46E+00	1.43E+00	1.41E+00	1.39E+00	1.37E+00
2	1.17E+01	9.63E-04	6.33E+00	6.59E+00	6.79E+00	6.62E+00	6.44E+00	6.27E+00	6.11E+00	5.98E+00	5.85E+00	5.74E+00	5.63E+00	5.53E+00	5.45E+00	5.37E+00
3	1.74E+01	1.44E-03	1.30E+01	1.41E+01	1.50E+01	1.48E+01	1.45E+01	1.41E+01	1.38E+01	1.35E+01	1.32E+01	1.29E+01	1.27E+01	1.24E+01	1.22E+01	1.21E+01
4	2.29E+01	1.93E-03	2.11E+01	2.35E+01	2.56E+01	2.57E+01	2.56E+01	2.51E+01	2.45E+01	2.40E+01	2.35E+01	2.31E+01	2.26E+01	2.22E+01	2.19E+01	2.15E+01
5	2.82E+01	2.41E-03	3.01E+01	3.42E+01	3.78E+01	3.86E+01	3.91E+01	3.86E+01	3.80E+01	3.73E+01	3.67E+01	3.60E+01	3.54E+01	3.48E+01	3.42E+01	3.37E+01
6	3.29E+01	2.89E-03	3.98E+01	4.57E+01	5.12E+01	5.29E+01	5.42E+01	5.40E+01	5.36E+01	5.29E+01	5.21E+01	5.13E+01	5.06E+01	4.98E+01	4.91E+01	4.84E+01
7	3.69E+01	3.37E-03	5.00E+01	5.79E+01	6.52E+01	6.82E+01	7.04E+01	7.07E+01	7.06E+01	7.00E+01	6.93E+01	6.84E+01	6.76E+01	6.67E+01	6.59E+01	6.51E+01
8	4.02E+01	3.85E-03	6.05E+01	7.05E+01	7.97E+01	8.40E+01	8.73E+01	8.82E+01	8.85E+01	8.81E+01	8.75E+01	8.68E+01	8.59E+01	8.49E+01	8.40E+01	8.31E+01
9	4.28E+01	4.33E-03	7.13E+01	8.34E+01	9.46E+01	1.00E+02	1.05E+02	1.06E+02	1.07E+02	1.07E+02	1.07E+02	1.06E+02	1.05E+02	1.04E+02	1.03E+02	1.02E+02
10	4.46E+01	4.81E-03	8.22E+01	9.64E+01	1.10E+02	1.16E+02	1.22E+02	1.24E+02	1.26E+02	1.26E+02	1.26E+02	1.26E+02	1.25E+02	1.24E+02	1.23E+02	1.22E+02
11	4.60E+01	5.30E-03	9.30E+01	1.09E+02	1.25E+02	1.33E+02	1.40E+02	1.43E+02	1.45E+02	1.45E+02	1.46E+02	1.45E+02	1.45E+02	1.44E+02	1.43E+02	1.42E+02
12	4.70E+01	5.78E-03	1.04E+02	1.22E+02	1.40E+02	1.49E+02	1.57E+02	1.61E+02	1.64E+02	1.65E+02	1.65E+02	1.65E+02	1.65E+02	1.64E+02	1.63E+02	1.62E+02
13	4.78E+01	6.26E-03	1.14E+02	1.35E+02	1.55E+02	1.65E+02	1.74E+02	1.79E+02	1.82E+02	1.84E+02	1.85E+02	1.85E+02	1.85E+02	1.84E+02	1.83E+02	1.82E+02
14	4.84E+01	6.74E-03	1.24E+02	1.48E+02	1.70E+02	1.82E+02	1.92E+02	1.97E+02	2.01E+02	2.03E+02	2.04E+02	2.05E+02	2.05E+02	2.04E+02	2.03E+02	2.02E+02
15	4.90E+01	7.22E-03	1.35E+02	1.61E+02	1.85E+02	1.98E+02	2.09E+02	2.15E+02	2.20E+02	2.22E+02	2.24E+02	2.24E+02	2.24E+02	2.24E+02	2.23E+02	2.22E+02
16	4.94E+01	7.70E-03	1.45E+02	1.73E+02	2.00E+02	2.14E+02	2.27E+02	2.33E+02	2.39E+02	2.41E+02	2.43E+02	2.44E+02	2.44E+02	2.44E+02	2.43E+02	2.42E+02
17	4.98E+01	8.18E-03	1.55E+02	1.86E+02	2.15E+02	2.31E+02	2.44E+02	2.51E+02	2.57E+02	2.61E+02	2.63E+02	2.64E+02	2.64E+02	2.64E+02	2.63E+02	2.63E+02
18	5.01E+01	8.67E-03	1.65E+02	1.98E+02	2.30E+02	2.47E+02	2.62E+02	2.70E+02	2.76E+02	2.80E+02	2.82E+02	2.84E+02	2.84E+02	2.84E+02	2.83E+02	2.83E+02
19	5.05E+01	9.15E-03	1.75E+02	2.11E+02	2.44E+02	2.63E+02	2.79E+02	2.88E+02	2.95E+02	2.99E+02	3.02E+02	3.04E+02	3.04E+02	3.04E+02	3.04E+02	3.03E+02
20	5.07E+01	9.63E-03	1.84E+02	2.23E+02	2.59E+02	2.79E+02	2.97E+02	3.06E+02	3.14E+02	3.18E+02	3.22E+02	3.23E+02	3.24E+02	3.24E+02	3.24E+02	3.23E+02
end																
**	**	phi	0	2	4	6	8	10	12	14	16	18	20	22	24	26
		T/Sigma	-7.32E-01	-7.32E-01	-7.32E-01	-7.32E-01	-6.73E-01	-6.36E-01	-6.14E-01	-6.02E-01	-5.93E-01	-5.84E-01	-5.76E-01	-5.69E-01	-5.64E-01	-5.59E-01
Tearing	Point	Summary	Values	**	**	**										
		J_tear	7.21E+01	8.43E+01	9.56E+01	1.01E+02	1.06E+02	1.07E+02	1.08E+02	1.08E+02	1.08E+02	1.07E+02	1.06E+02	1.05E+02	1.05E+02	1.04E+02
		T/Sys	-6.16E-01	-6.16E-01	-6.16E-01	-6.16E-01	-5.67E-01	-5.36E-01	-5.17E-01	-5.07E-01	-4.99E-01	-4.92E-01	-4.85E-01	-4.80E-01	-4.75E-01	-4.71E-01
		K_Jel	3.29E+01	3.30E+01	3.30E+01	3.26E+01	3.22E+01	3.17E+01	3.13E+01	3.10E+01	3.07E+01	3.04E+01	3.01E+01	2.98E+01	2.96E+01	2.94E+01
		K_Jtotal	2.92E+01	3.16E+01	3.37E+01	3.47E+01	3.54E+01	3.57E+01	3.59E+01	3.59E+01	3.58E+01	3.57E+01	3.55E+01	3.54E+01	3.52E+01	3.51E+01
Tearing	Point	Values	At	Tearing	Phi	Location										
Stress	Force	CMOD	Phi	J	T/Sys	K_Jel	K_Jtotal	Sigma/Sys	Ma	Mb	r_phi_a	r_phi_b				
4.29E+01	5.61E+01	4.37E-03	17	1.08E+02	-4.99E-01	3.07E+01	3.58E+01	0.84	114.46	511.78	0.243	1.085				
end_summary																
US Units																
length (in)																
Force (kip)																
Stress (ksi)																
J (in-lb/in^{2})																
K (ksi-in^{0.5})																
Phi (deg)																
end_units																

Values per load step and phi location

Normalized T-Stress = T/σ

Summary of tearing point values if available

Output Units

Note: The normalized elastic T-stress values are interpolated from the T-stress tables in Annex A2 of ASTM E2899

Analysis only *.ntrp ("interp") file

```
%-----Interpolation Analysis Input File-----%
%words proceeded with * are keywords for the
%text scanning program
%---Set Units----% (options are SI or US)
*units SI
%---Geometry Values----%
*2c 12.70
*a  6.17
*W  88.82
*B  9.5
%
%---Material Properties----%
*material
*E 74.460e3
*stress pl_strain
317.02,0
361.08,0.00055
375.35,0.00105
385.97,0.00203
391.83,0.00309
396.17,0.00422
399.55,0.0052
402.86,0.00618
405.07,0.00703
408.51,0.00813
411.41,0.00928
414.31,0.01022
437.40,0.02013
454.36,0.03003
464.84,0.04042
469.81,0.05014
472.22,0.05743
*end_material
```

Units

Geometry Values

Material properties
E
stress, plastic strain
.
.
.
.
.
.
.
.
.
.
.
end

NOTE: The *keywords are strings that are searched for in the code and must appear as shown in this example. The numeric values next to or below the keywords are changed to create new analysis input files.

NOTE: Poisson's ratio, ν, is not a required material input. All of the solutions have a fixed value of $\nu = 0.30$ from the interpolated solutions data set.

Analysis only *.ntrp file with simple material definition

Units

Geometry Values

Material properties
E
Yield Stress
n (hardening)
end

```
%-----Interpolation Analysis Input File-----%
%words proceeded with * are keywords for the
%text scanning program
%---Set Units----% (options are SI or US)
*units SI
%---Geometry Values----%
*2c 12.70
*a  6.17
*W  88.82
*B  9.5
%
%---Material Properties----%
*material
*E 74.460e3
*Sys 351.6
*n 8.5
*end_material
```

Pre-Test prediction *.ntrp

Units

Geometry Values

Material properties Can be entered in table or short format

Critical Toughness Critical angle

```
%-----Interpolation Analysis Input File-----%
%words proceeded with * are keywords for the
%text scanning program
%---Set Units----% (options are SI or US)
*units SI
%---Geometry Values----%
*2c 12.70
*a  6.17
*W  88.82
*B  9.5
%
%---Material Properties----%
*material
*E 74.460e3
*stress pl_strain
317.02,0
361.08,0.00055
375.35,0.00105
385.97,0.00203
391.83,0.00309
396.17,0.00422
399.55,0.0052
402.86,0.00618
405.07,0.00703
408.51,0.00813
411.41,0.00928
414.31,0.01022
437.40,0.02013
454.36,0.03003
464.84,0.04042
469.81,0.05014
472.22,0.05743
*end_material
%
%
%---Pre-test Prediction Values----%
*pretest
*Jc 21.0
*phi_crit 17
```

Test Analysis *.ntrp file

Units

Geometry Values

Material properties Can be entered in table or short format

Test Tearing force
Tearing angle
CMOD, Force Table
.
.
.
.
.
.
.
.
.

End_test_data

```
%-----Interpolation Analysis Input File-----%
%words proceeded with * are keywords for the
%text scanning program
%---Set Units----% (options are SI or US)
*units SI
%---Geometry Values----%
*2c 12.70
*a  6.17
*W  88.82
*B  9.5
%
%---Material Properties----%
*material
*E 74.460e3
*Sys 351.6
*n 8.5
*end_material
%
%---Test Evaluation----%
*test_eval
*tear_force 251.8
*tear_phi 17
*CMOD            Force
0.0000,0.095
0.0000,0.149
0.0005,1.195
0.0013,3.638
0.0023,6.788
0.0036,10.616
0.0048,14.756
0.0064,19.195
.
.
.
0.1123,250.593
0.1135,251.814
*end_test_data
```

Example material property *.prop files –
Use with the "Import Material Properties Button"

File containing stress-plastic_strain table

Material properties
E
stress, plastic strain
.
.
.
.
.
.
.
.
.
.
.
end

```
%---Material Properties----%
*material
*E 10.8e3
*stress pl_strain
45.98,0.00000
52.37,0.00055
54.44,0.00105
55.98,0.00203
56.83,0.00309
57.46,0.00422
57.95,0.00520
58.43,0.00618
58.75,0.00703
59.25,0.00813
59.67,0.00928
60.09,0.01022
63.44,0.02013
65.90,0.03003
67.42,0.04042
68.14,0.05014
68.49,0.05743
*end_material
```

NOTE: Poisson's ratio, ν, is not a required material input. All of the solutions have a fixed value of $\nu = 0.30$ from the interpolated solutions data set.

Short format file with no stress-plastic_strain table

```
%---Material Properties----%
*material
*E 10.8e3
*Sys 51.0
*n 8.5
*end_material
```

NOTE: The *keywords are strings that are searched for in the code and must appear as shown in this example. The numeric values next to or below the keywords are changed to create new analysis input files.

Stress – plastic strain data can be tab, space, or comma delimited

Example test data*.txt files –
Use with the “Import Test Data” button

```
%Unlimited header lines at top
%ensure that "*CMOD" keyword precedes the data
%and "*end_test_data" is at the end of the data
%Example test data in US units
%
*CMOD          Force
0              0.0214
0              0.0336
0.00002        0.2686
0.00005        0.8179
0.00009        1.5259
0.00014        2.3865
0.00019        3.3173
0.00025        4.3152
.
.
.
0.00406        54.0283
0.00424        55.2063
0.00442        56.3355
0.00447        56.6101
*end_test_data
```

NOTE: The *keywords are strings that are searched for in the code and must appear as shown in this example. The numeric values next to or below the keywords are changed to create new analysis input files.

CMOD and force data can be tab, space, or comma delimited

Source File Information

- If so desired, TASC can be run from the source files available from the NASA TASC Sourceforge project at http://sourceforge.net/projects/tascnasa/. Running TASC from the source files requires a current Matlab license along with a Matlab Curve Fitting Toolbox license and a Matlab Image Processing Toolbox License.
- Type '"tasc" from the Matlab command prompt to start TASC.
- The first time you run TASC, a start up window will appear stating "*I have read and accept the terms of the NASA Open Source Agreement included in the TASC distribution package*." You must select "Yes" to run TASC. Selecting "Yes" creates a small text file, TASC_lic.txt" in your executable directory. You will not be asked to accept the agreement again unless you move or copy the executable file to another directory.
- The interpolation routines rely on the interpolated solutions database file, interp_solution_database.mat, located in the source file directory. The *.mat database is a 4-D Matlab structure. The solution database is arranged in a *result*(I,J,K,L).*fea* structure where the *result* indices are *result*(a/B, a/c, n ,E/σ_{ys}). Details of the solution space, interpolation methods, and nomenclature are documented in NASA/TP-2013-217480, *Elastic-Plastic J-Integral Solutions for Surface Cracks in Tension Using an Interpolation Methodology*, which is available for download from the NASA Center for Aerospace Information (CASI) at <http://www.sti.nasa.gov>.