LS-DYNA® Aerospace Working Group

*MAT_224 User Guide

A User Guide for *MAT_TABULATED_JOHNSON_COOK in LS-DYNA®

December 14, 2021

Table of Contents

List of Figures

List of Tables

1 Introduction

*MAT_TABULATED_JOHNSON_COOK (*MAT_224) is a new constitutive model for metals that was developed in a collaborative effort of the Federal Aviation Administration (FAA), National Aeronautics and Space Administration (NASA), George Washington University (GWU), Ohio State University (OSU), and George Mason University (GMU). This collaboration resulted in the new physics-based constitutive model for metals that can be used to accurately predict the mode of failure and extent of damage in aerospace impact events. Previous to the development of this model, material failure parameters were tuned to match a specific test with a specific failure mode. In order to obtain accurate results for a different mode of failure, a different set of material parameters was required and, since the failure mode had to be known ahead of time, its use as a predictive tool in design and analysis was precluded.

This new *MAT_TABULATED_JOHNSON_COOK model is an elastic-viscoplastic material model with the option to define strain rate and temperature dependent stress versus strain curves. In addition, plastic failure strain can be defined as a function of any combination of triaxiality and Lode parameter (both are functions of the 3D state of stress), strain rate, temperature, and element size. The model also gives the user the option to use the temperature or strain rate dependencies with or without failure.

This document is a user guide for developing the material model input parameters for *MAT_224. This user guide includes sections on the theoretical overview, LS-DYNA® input parameters, mechanical property input generation, failure surface generation, material model validation, and test program.

As a summary user guide, this document references various technical reports that were written during the development of the material model, all of which are listed in the reference section.

2 Theoretical Overview

In LS-DYNA®, *MAT_TABULATED_JOHNSON_COOK or *MAT_224 is an isotropic elastic thermo-viscoplastic constitutive relation that states stress is a function of strain, strain rate, and temperature:

$$\sigma_{ij} = \sigma_{ij}(\varepsilon_{ij}, \dot{\varepsilon}_{ij}, T)$$

Where σ_{ij} is stress, ε_{ij} is strain, $\dot{\varepsilon}_{ij}$ is strain rate, and T is temperature. Tensile tests conducted at various strain rates and temperatures are used to derive the LS-DYNA® input stress-strain curves and tables.

In the plastic region, the material response is determined by a von Mises type yield surface that expands or contracts due to strain hardening, rate effects, and thermal softening:

$$\sigma_{vm}(\sigma_{ij}) \leq \sigma_y\left(\varepsilon_{ij}^p, \dot{\varepsilon}_{ij}^p, T\right) = \left(\varepsilon_{eff}^p, \dot{\varepsilon}_{eff}^p, T\right)$$

Where σ_{vm} is the von Mises stress, ε_{eff}^p is the equivalent plastic strain, and $\dot{\varepsilon}_{eff}^p$ is the equivalent plastic strain rate.

To model accumulated damage and element failure, *MAT_224 uses four input parameters. The first parameter is either a load curve for use with shell elements or a table of curves for use with solid elements. The load curve defines the plastic failure strain as a function of triaxiality and the table of curves defines the plastic failure strain as a function of triaxiality and Lode parameter, which results in the definition of a failure surface. The second parameter is a load curve that defines the plastic failure strain as a function of plastic strain rate. The third parameter is a load curve that defines the plastic failure strain as a function of temperature. The last parameter is a load curve (or table of curves) that defines that plastic failure strain as a function of element size (and triaxiality).

Triaxiality is defined by the equation:

$$\tau = \frac{p}{\sigma_{vm}}$$

where p is the pressure defined by

$$p = -\frac{\sigma_1 + \sigma_2 + \sigma_3}{3}$$

and σ_{vm} is the Von Mises stress defined by

$$\sigma_{vm} = \sqrt{\frac{3}{2}\left((\sigma_1 + p)^2 + (\sigma_2 + p)^2 + (\sigma_3 + p)^2\right)}$$

The Lode parameter is defined by the equation:

$$\theta_L = \frac{27\ s_1 s_2 s_3}{2\sigma_{vm}^3}$$

where s_1, s_2, and s_3 are the principal deviatoric stresses. The Lode parameter ranges between plus and minus one, which provides a convenient boundary for the definition of the failure surface. The stress triaxiality, however, ranges between plus and minus infinity and, therefore, it is necessary to define reasonable boundaries. A triaxiality range of -1.0 to +1.0 should be sufficient for most impact applications.

The plastic failure strain is defined by:

$$\varepsilon_{pf} = f(\tau, \theta_L)\ g(\dot{\varepsilon}_p)h(T)i(l_c)$$

where τ is the triaxiality, θ_L is the Lode parameter, $\dot{\varepsilon}_p$ is the plastic strain rate, and l_c is the element size. When more than one of the failure parameters are used, the net plastic failure strain is the product of the functions defined in the above equation.

The failure criterion is based on an accumulated damage parameter defined by:

$$F = \int \frac{\dot{\varepsilon}_p}{\varepsilon_{pf}}\,dt$$

where $\dot{\varepsilon}_p$ is the plastic strain rate and ε_{pf} is the plastic failure strain, which is continually changing based on triaxiality and Lode parameter. When this damage parameter is greater than or equal to one, the element has failed and is deleted.

Failure prediction using shell elements requires the plastic strain failure to be defined as a function of only triaxiality and, therefore, a failure curve is defined. Failure prediction using solid elements requires the plastic failure strain to be defined as a function of both triaxiality and Lode parameter and, therefore, a failure surface, specified in a table, is defined.

To determine the failure surface, various types of material specimens are created and tested with each specimen having a unique triaxiality and Lode parameter representing a specific point on the failure surface. To generate an accurate and complete failure surface, therefore, requires as many material specimens as possible. Using the triaxiality, Lode parameter, and failure strain data from the specimen testing, a complete failure surface can then be created using a three-dimensional curve fitting tool.

After a failure surface is generated, the remaining parameter load curves can be created. A strain rate testing series and a temperature testing series must be completed to determine the next two failure parameters. Finally, by varying the size of the elements in the mesh analytically, a load curve can be created that accounts for many different sizes of discretization.

3 LS-DYNA® Input Parameters

The material model input parameters are listed here with the variable name from the LS-DYNA® input manual, the variable symbol used in equations, and a brief description.

It is recommended that when defining tables the *DEFINE_TABLE_2D keyword input be used. The *DEFINE_TABLE_2D keyword input requires a curve ID to be specified for each value defined in the table. This allows the same curve ID to be referenced by multiple tables, and the curves may be defined anywhere in the input file. If the alternate *DEFINE_TABLE is used, curve ids are not used and the set of curves referenced in the table must immediately follow the table and be in the same order as referenced in the table with no other keywords in between.

An additional recommendation to help convergence in the material model plasticity algorithm is to define the LCINT parameter, which defines the number of discretization points for curve interpolation, to 1000 on the material model *DEFINE_CURVE keyword inputs. If this is not done, the value of LCINT is taken from the *CONTROL_SOLUTION keyword input where the default value of 100 has been shown to be insufficient and result in plasticity algorithm non-convergence.

LOG_INTERPOLATION: With this keyword option, the strain rate effect in table LCK1 is interpolated with natural logarithmic interpolation, and the values entered in the table are strain rates (not the natural log of strain rates). Since metals often show a logarithmic function rate dependency, use of this option creates a more representative discretization than the default linear curve discretization, and it may also require fewer discretization points.

RO: Mass density, ρ

E: Young's modulus, E   Temperature dependent Young's moduli may be specified using a load curve using a negative modulus with the absolute value pointing to the specific load curve.

PR: Poisson's ratio, μ

CP: Specific heat, C_p

TR: Room Temperature, T_R

BETA: Fraction of plastic work converted into heat, β (Taylor-Quinney Coefficient)

Temperature increase is caused by plastic work

$$T = T_R + \frac{\beta}{C_p \rho} \int \sigma_y \, \varepsilon_p dt$$

If BETA is input as a positive number the value of β is constant. If BETA is input as a negative number, it is either: a curve ID for strain rate dependence; OR a table ID for strain rate and temperature dependence; or a 3-dimensional table ID for temperature (TABLE_3D), strain rate (TABLE) and plastic strain (CURVE) dependence; OR a 4-dimensional table ID for triaxiality (TABLE_4D), temperature (TABLE_3D), strain rate (TABLE), and plastic strain (CURVE) dependence.

LCK1: A load curve ID that defines the effective stress as a function of effective plastic strain OR a table ID that defines a load curve ID for each plastic strain rate that gives the (isothermal) effective stress versus effective plastic strain for that plastic strain rate. This load curve should be defined at room temperature as specified above with the TR input parameter. If the first strain rate in table LCK1 is negative, all the values in the table are the natural logarithm of the strain rate, and the strain rate effect in table LCK1 is interpolated with natural logarithmic interpolation.

Since metals often show a logarithmic function rate dependency, using logarithmic dependent strain rate curves creates a more representative discretization than the default linear curve discretization, and it may also require fewer discretization points.

LCKT: A table ID that defines a load curve ID for each temperature that gives the effective stress versus effective plastic strain for that temperature. These load curves should be defined under quasi-static conditions.

Plastic failure strain is defined as a function of triaxiality, Lode parameter, plastic strain rate, temperature, and element size by

$$\varepsilon_{pf} = f\left(\frac{p}{\sigma_{vm}}, \frac{27s_1 s_2 s_3}{2\sigma_{vm}^3}\right) g(\dot{\varepsilon}_p) h(T) i(l_o)$$

using the load curves and tables LCF, LCG, LCH, and LCI defined below. If more than one of these four variables LCF, LCG, LCH, and LCI are defined, the net plastic failure strain is the product of the above multiple functions. In that case, only one of the variables LCF, LCG, LCH, and LCI can point to curve(s) with plastic strain along the curve ordinate. The remaining non-zero variables must point to curve(s) that have a dimensionless scaling factor along the curve ordinate. If a particular scaling curve is not used, the scaling factor defaults to a value of 1.0, however, best practices recommend that a scaling curve with ordinates of 1.0 should be defined and referenced.

LCF: A load curve that defines the plastic failure strain or scale factor as a function of triaxiality OR a table ID that defines a load curve ID for each Lode parameter that gives the plastic failure strain versus triaxiality for that Lode parameter. The table option is only available for solid elements, i.e., if shell elements are used, only a curve of plastic failure strain as a function of triaxiality can be input.

LCG: A load curve ID that defines the plastic failure strain or scale factor as a function of plastic strain rate. If the first strain rate in curve LCG is negative, all the values in the table are the natural logarithm of the strain rate, and the strain rate effect in curve LCG is interpolated with natural logarithmic interpolation.

As with the LCK1 table, defining the rate dependency using logarithmic input can create a more representative discretization than the default

linear curve discretization, and it may also require fewer discretization points.

LCH: A load curve ID that defines the plastic failure strain or scale factor as a function of temperature.

LCI: A load curve ID that defines the plastic failure strain or scale factor as a function of element size OR a table ID that defines a load curve ID for each triaxiality that gives plastic strain versus element size for that triaxiality OR a 3-dimensional table ID that defines plastic failure strain as a function of Lode parameter (TABLE_3D), triaxiality (TABLE), and element size (CURVE).

NUMINT: Number of integration points which must fail before the element is deleted.

FAILOPT: Flag for an additional failure criterion F_2.

NUMAVG: Number of time steps for the running average of plastic failure strain in the additional failure criterion.

NCYFAIL: Number of time steps that the additional failure criterion must be met before element deletion.

The default failure criterion for the material model depends on plastic strain evolution $\dot{\varepsilon}_p$ and on plastic failure strain ε_{pf} and is obtained by accumulation over time:

$$ F = \int \frac{\dot{\varepsilon}_p}{\varepsilon_{pf}} \, dt $$

where element erosion takes place when $F \geq 1$, which provides load-path dependent failure.

An additional, load-path independent, failure criterion, F_2, can be invoked by setting FAILOPT=1, where the current state of plastic strain is used:

$$ F_2 = \frac{\varepsilon_p}{\varepsilon_{pf}} $$

Using this failure criterion, two additional parameters are available as countermeasures against stress oscillations. With NUMAVG defined, the plastic strain is averaged over NUMAVG time steps. NCYFAIL defines the number of time steps that $F_2 \geq 1$ must be met before element deletion takes place.

Additional *MAT_224 history variables for post-processing d3plot and d3thdt database files are available using the optional keyword *DATABASE_EXTENT_BINARY where NEIPS and NEIPH indicate the number of additional history variables for shells and solids, respectively. The available history variables are tabulated in Table 1. It should be noted that to obtain the correct triaxiality from *MAT_224, the additional history variables must be requested because the triaxiality calculated in LS-PrePost® using the d3plot files is different from that calculated in *MAT_224.

Table 1 LS-PrePost® History Variables

LS-PrePost history variable #	Shell elements	LS-PrePost history variable #	Solid elements
1	plastic strain rate	5	plastic strain rate
7	plastic work	8	plastic failure strain
8	ratio of plastic strain to plastic failure strain	9	triaxiality
9	element size	10	Lode parameter
10	temperature	11	plastic work
11	plastic failure strain	12	ratio of plastic strain to plastic failure strain
12	triaxiality	13	element size
16	Frac. Plastic work to heat	14	temperature
17	LCPS: critical value	17	LCPS: critical value
		18	Frac. Plastic work to heat

4   Stress-Strain Relation Input Generation

Appendix A contains a flow chart of the stress-strain relationship modeling procedure that is presented this section.

4.1   Elastic-Plastic Modeling

To generate the input curves for the effective stress as a function of plastic strain, a series of standard tension tests is performed at various strain rates and temperatures. In the tension tests, a dog-bone specimen under a constant grip is pulled in a tensile machine. During the test, the force versus time is measured with the tensile machine, and the displacement versus time relation is measured by an extensometer fixed to the specimen. The force versus displacement curve is then generated by cross plotting these two curves. Engineering stress and engineering strain are calculated using

$$\sigma_{eng}(t) = \frac{F(t)}{A_0}$$

$$\varepsilon_{eng}(t) = \frac{L(t) - L_0}{L_0} = \frac{D(t)}{L_0}$$

where $\sigma_{eng}(t)$ is the engineering stress, $\varepsilon_{eng}(t)$ is the engineering strain, *F(t)* is the axial force, A_o is the original cross section area, *L(t)* is the instantaneous gauge length, L_0 is the original gauge length, and *D(t)* is the displacement. True stress and true strain are then calculated from

$$\sigma = \sigma_{eng}(1 + \varepsilon_{eng})$$

$$\varepsilon = \ln(\varepsilon_{eng} + 1)$$

Once the stress-strain curve is obtained, Young's Modulus is determined from the slope of the curve in the elastic region and the yield strength, σ_y, is determined at the end of the linear portion of the curve. If several strain rate tests were performed, Young's Modulus should be determined from the lowest strain rate test. It should also be noted that while Young's

modulus does not typically vary with temperature, the ability to input this temperature variation in LS-DYNA® is available by defining a negative modulus with the absolute value of the modulus pointing to a curve defining this variation.   In the plastic region, the plastic strain, ε_p is computed from

$$\varepsilon_p = \varepsilon - \sigma/E$$

and the effective plastic strain, ε_{eff}^p is a monotonically increasing scalar value which is calculated incrementally as a function of the plastic strain rate tensor, $\dot{\varepsilon}_{ij}^p$ as

$$\varepsilon_{eff}^p = \int_{t=0}^{t1} d\varepsilon_{eff}^p = \int_0^t \sqrt{2/3\dot{\varepsilon}_{ij}^p \cdot \dot{\varepsilon}_{ij}^p}\, dt$$

This procedure for determining the elastic-plastic material behavior is valid only to the point of necking of the test specimen. Once necking begins, the test section cross-sectional area is no longer constant in the necked region and a uniaxial stress state no longer exists with the development of transverse stresses in the necked region. As a result, using the previous equations to create the elastic-plastic material input data after necking occurs is invalid and a more accurate method is now described.

4.2 Stress-Strain Relationship after Necking

The recommended method to create the post-necking portion of the stress-strain input curves is a trial-and-error process. Simulations of the tensile tests that explicitly model post-necking behavior must be performed using several different input stress-strain curves. The input stress-strain curve which then most closely matches the test results will be the best approximation of the material's actual stress-strain relationship.

To accurately simulate the tensile test in LS-DYNA® after necking has occurred, start by estimating several post-necking plastic stress-strain curves for input into the tensile test simulation by extrapolating the curve before necking.   Then compare the simulation force vs. displacement curve

output with the test results and pick the input curve that gives the closest match to test. At this stage, the strain rate effect is not considered in the simulation. This process of determining a single input stress-strain curve to match each strain rate tested is repeated for all the tested strain rates, which makes each individual analysis strain rate independent. Because of this strain rate independence, the simulation can be performed with an artificial (higher) loading speed instead of the actual loading speed that was used in the test, allowing for a much shorter simulation time.

The necking point is given by the intersection between the true strain versus true stress curve and its own derivative. As shown in Figure 1 only the part of the strain-stress curve before necking is used for further processing.
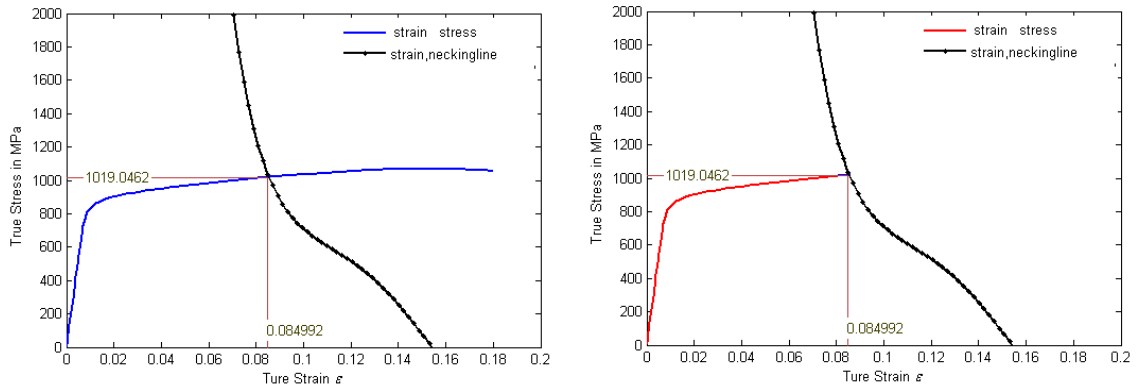
Figure 1 True Stress-True Strain Curve Before and After Trimming

The hardening curves are extrapolated after necking using the formula:

$$\sigma = k\big[\varepsilon_e + \varepsilon_p\big]^n$$

where for a given plastic strain ε_p the stress σ is calculated. The exponent n is expected to vary between 0 and 1 as the hardening curve should be monotonically increasing and have a monotonically decreasing tangent. Thus, given a specific n value, k and ε_e are determined from

$$k = A\left(\frac{An}{C}\right)^{-n} \qquad\qquad \varepsilon_e = \frac{An}{C} - B$$

where *A* is the stress at necking, *B* is the plastic strain at necking, and *C* is the slope (hardening modulus) at necking. After assuming different *n* values, a cluster of curves can be generated as shown in Figure 2.
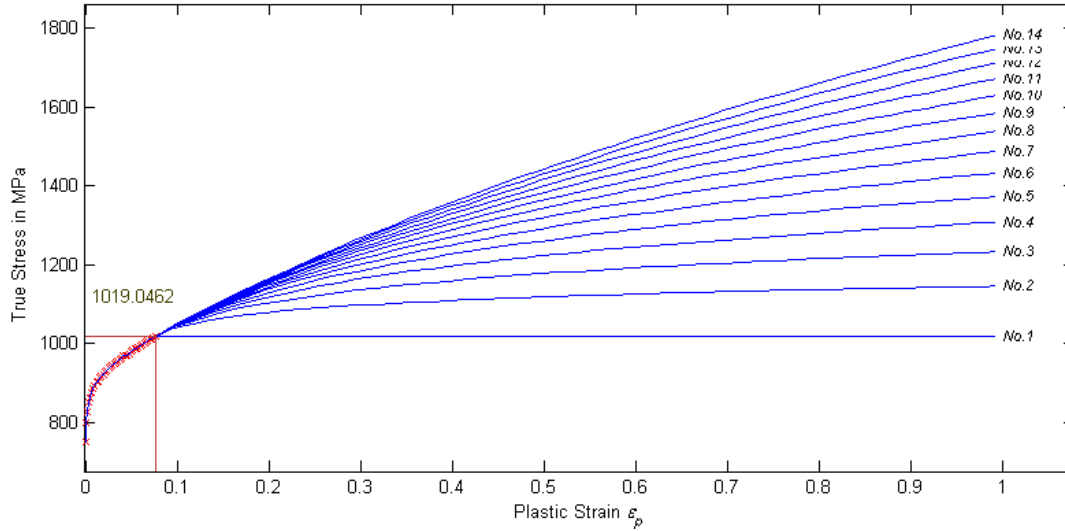
Figure 2 Extrapolated Curves after Necking Point

Several candidate curves are then chosen from the generated plastic strain vs. stress curves and these curves are input into the material model and tensile test simulations are completed. Force-displacement curves obtained from the simulations are then overlaid with the actual tensile test results to select the curve that is the best estimation of the plastic strain vs. stress relation. The input data developed using this process will match the actual test data as closely as possible while removing high frequency oscillations and negative slope areas that could cause convergence problems.

Matching each test individually is required because each specimen has a unique geometry. The exact geometry from measurement is used in the finite element model. If the exact test geometry is not used in the analysis, significant errors can result. Although the input curve, because it is a function of stress and strain, can be created from an average of all the test results, only a single geometry can be used in each simulation. The force-displacement results from the simulations will vary from each other, while the stress-strain results will not.

13

This procedure is valid for all constant strain rate (including quasi-static) isothermal uniaxial tensile tests, independent of the temperature at which they were performed. If the quasi-static tensile tests are performed at low enough strain rates, the process is isothermal and no thermo-mechanical coupling occurs during the test. As a result, the yield curves obtained using this procedure can be allocated to the specific temperature at which the test was conducted. A table of temperature dependent yield curves can then be created by individually simulating all temperature tensile tests for a given strain rate without considering any thermal coupling in the simulations.

The series of generated stress-strain input curves from the constant strain rate tensile tests can then be bundled together as tabulated input for *MAT_224. Using these temperature and rate dependent yield curves allows for the simulation of dynamic experiments where thermal and rate effects occur simultaneously and interact with each other. However, the procedure described above cannot be used to simulate many higher strain tests, because they are not isothermal and do not maintain a constant strain rate.

4.3 High Strain Rate Sensitivity

Depending on the material being tested, in many higher strain rate tests the strain rate is not constant. This is because after necking occurs the strain localizes, the strain rate increases in the necked area, and both the strain and the strain rate are no longer uniform across the specimen. In addition, since the duration of higher rate tests is short, the heat produced by the plastic work may not conduct enough to maintain a constant temperature. As a result, the procedure required to create isorate and isothermal stress-strain curves requires an additional trial-and-error process.

At extremely high strain rates, the strain rate in the region of localization may reach values significantly above the nominal strain rate for the specimen and it is also not isothermal. As a result, the stress-strain input generated using isothermal and isorate tension tests may be supplemented

with synthetic isothermal curves generated using rate sensitivity trends from compression tests. Compression tests maintain a more constant strain rate than tension tests due to relatively small localization. Compression tests can also reach much higher strain rates than the tension tests, but complete stress-strain curves are difficult to derive from them. As a result, synthetic curves can be created using a combination of information from compression and tension tests. These synthetic curves can then be combined with the stress-strain curves derived directly from isorate and isothermal tension tests.

The synthetic curves can be generated from log-log plots and curve fits of the stress at a particular strain of the compression tests for various strain rates as shown in Figure 3. Typically, strain rate sensitivity conforms to the theory that the yield strength increases as a logarithmic function of strain rate between rates of ~10^{-4} sec^{-1} and ~10^3 sec^{-1}, and as a linear function of the of strain rate at strain rates greater than ~10^3 sec^{-1} [6]. Higher strain rate curves are created by translating a previously created isorate and isothermal stress-strain curve so that their stresses pass through the stress values at the defined strain and strain rate.

Additional trial-and-error iterations will likely be required before the outcome is a close match between the test and the analysis results. Typically, there is a spread in high strain rate test data, as can be seen in Figure 3. As a result, defining the correct stress level is difficult, particularly in the transition region of strain rate sensitivity at ~10^3 sec^{-1}. In addition, strain rate curves must be created to cover the range of strain rates that occur in the particular analysis of interest and, as a result, some extrapolation beyond the tested strain rates may be required, which is frequently the case with ballistic impact simulations.
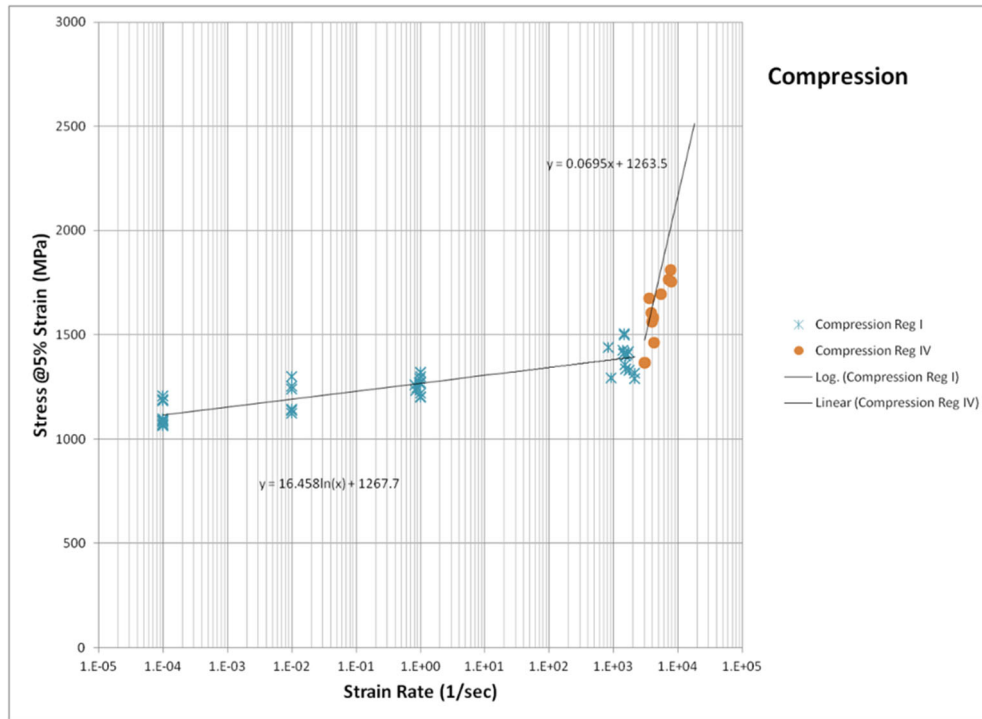
Figure 3 Compression Test Stress at 5% Strain

4.4 Conversion of Plastic Work into Heat

During high strain rate events in the localized necking region, there is not sufficient time for conduction to carry away the heat generated by the plastic deformation, and, as a result, the process becomes adiabatic. This adiabatic process causes a significant increase in the temperature in the specimen locally and, as a result, the simulation of the tension test is sensitive to the amount of energy generated by the plastic work that is converted into thermal energy. The percentage of plastic work that is converted into thermal energy is defined by the Taylor-Quinney coefficient, β. This coefficient is not a constant, but is a function of both strain rate and plastic strain.

Users have several options for defining β including a single constant value for β, effectively a rate independent β; or a curve of β vs plastic strain for various strain rates; or a table ID for strain rate and temperature dependence of β; or a 3-dimensional table ID for temperature, strain rate,

16

and plastic strain dependence of β; or a 4-dimensional table ID for triaxiality, temperature, strain rate, and plastic strain dependence of β. With this capability in *MAT_224, a fully coupled thermal solution that simulates the conduction of thermal energy away from highly strained elements can be performed. Without including this conduction, temperatures will rise more in the simulations than in tests, leading to non-physical analytical results. This non-physical result will be very small in high rate, short duration simulations, and can be safely ignored. However, in simulations where both high and low rate loading takes place simultaneously and significant strains are introduced into elements (e.g., longer duration simulations, such as full engine blade loss events) including this conduction may be required for accurate analysis.

5 Failure Surface Generation

To develop the failure surface, various material specimens, each with a unique triaxiality and Lode parameter, are created and tested to failure. To generate an accurate and complete failure surface, there should be as many material specimens as possible covering a broad range of triaxiality and Lode parameter. Using the specimen test results, a three-dimensional curve fitting tool can then be used to create a complete failure surface. The fundamental strategy of the curve fitting tool is to use points representing triaxiality, Lode parameter, and failure strain from the specimens tested to develop a complete failure surface. MathWorks MATLAB software was used in the development of the failure surface that is presented in this example.

Suppose a material testing program provides twenty different material tests with these states of stress:
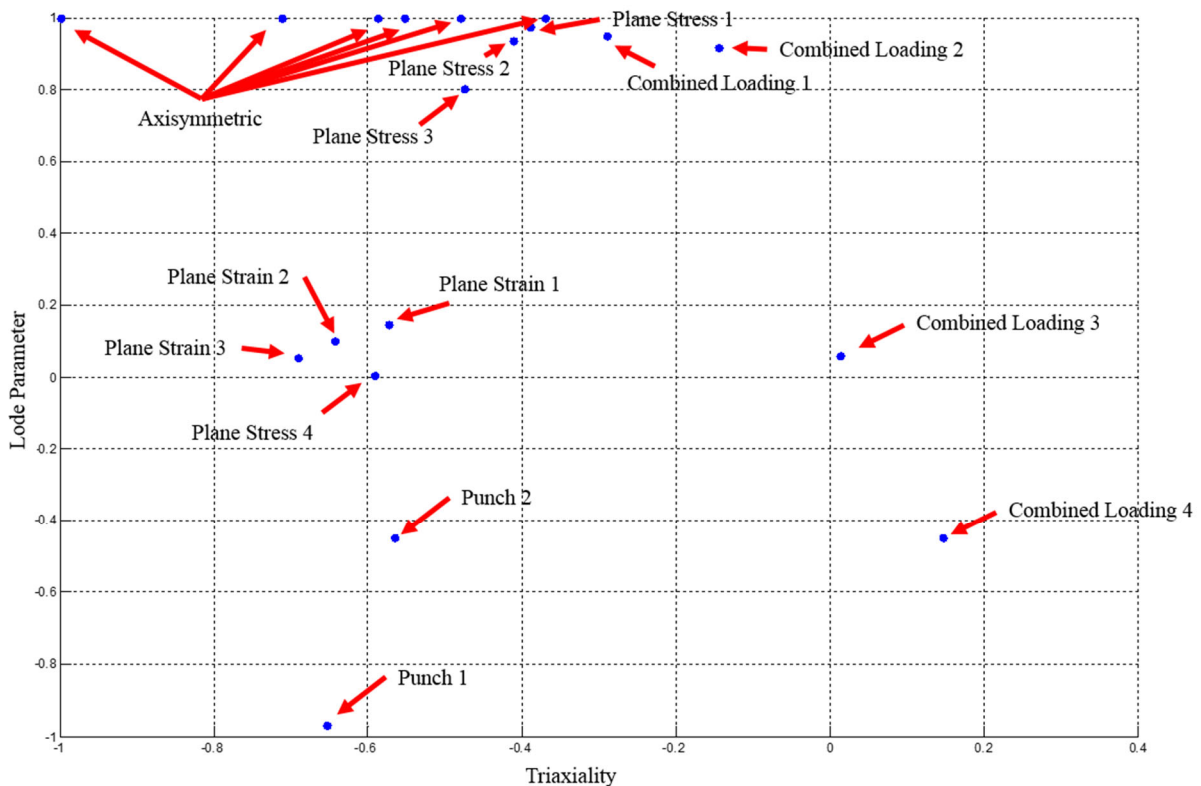
Figure 4 Stress States for a Material Testing Program

Since it is challenging to design specimens representing a broad range of triaxiality and Lode parameters, there are areas of the failure surface that are not covered by physical tests and, as a result, a "grid and control point strategy" was developed to define the entire failure surface. It should also be noted that the complete failure surface does not have to be defined, as long as the range of triaxiality and Lode parameter covers the range for the application in which the material model is being developed.

An orthogonal grid structure can be overlaid onto the existing plot. While these grid lines can be arbitrary, they should be based on existing physical tests where possible. If there are several specimens that are close together, one grid line can represent those points. Each test point or group of group of test points creates a grid line. An example of a grid structure for the data in Figure 4 is shown in Figure 5.
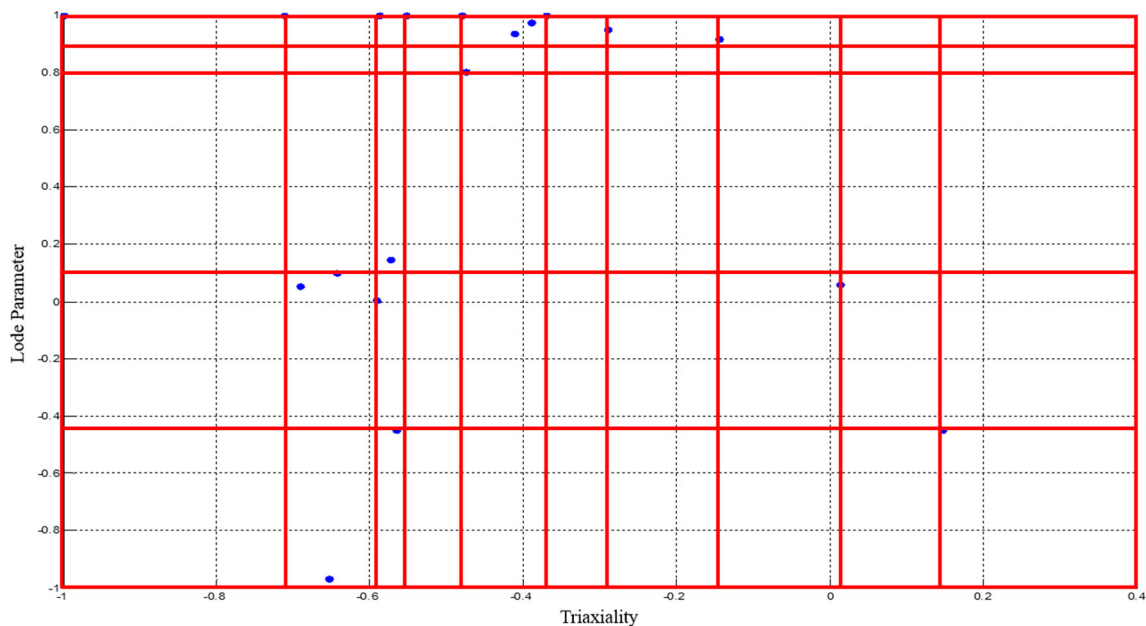
Figure 5 Grid and Control Point Example

Using these grid lines, a new set of points is defined based on the points of grid line intersection. For each grid line intersection, a point on the triaxiality-Lode parameter plane is created. In this example, there are a total of 66 "control points" and, as a result, three 66 point 1-D vectors can be created using: 1) the triaxiality of the control point, 2) the Lode parameter of the control point, and 3) the failure strain of the control

point. The failure strain is determined manually by the user of the tool. At control point locations where there is a single physical test, that specimen's failure strain defines the control point failure strain. If there are a group of points that represent one control point, those points are averaged using equal weights. If there is no physical test for a given control point, the nearest point is used, or the average of many nearby points. Using this method, all three of the 66 point 1-D vectors are full of values.

After the input vectors are defined, the program can be executed. MATLAB begins by implementing a subroutine that was developed using the Curve Fitting Toolbox, which uses a cubic interpolation routine that takes the three 66 point 1-D vectors and generates a fit 3-D surface. The user is then shown two plots of the surface as those shown in Figure 6.
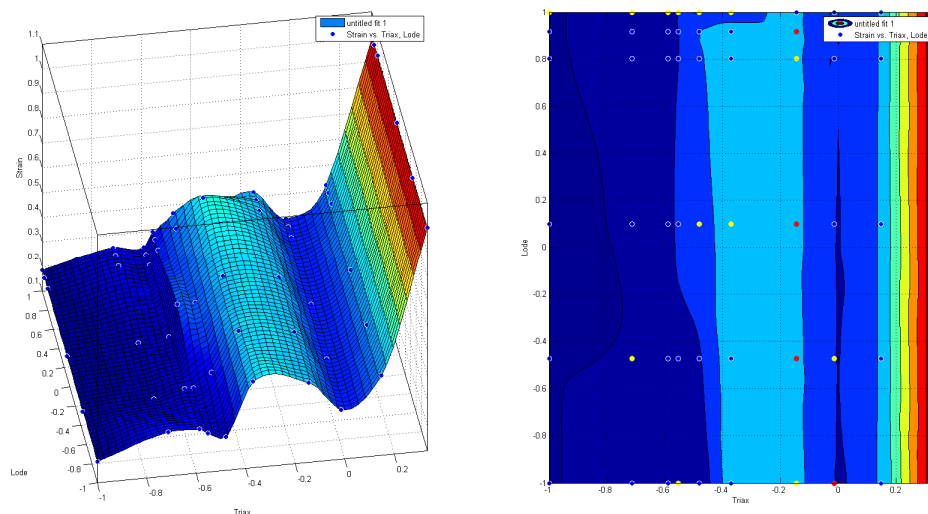
Figure 6 Surface Plots Generated by MATLAB Subroutine

The next step of the failure surface generation tool is to discretize the 3-D surface into individual points for input into LS-DYNA®. The 3-D surface is sampled into 2814 total points (134 triaxiality points by 21 Lode angle points). The tool will also show the user a visual representation of the discretization of the 3-D surface as shown in Figure 7.
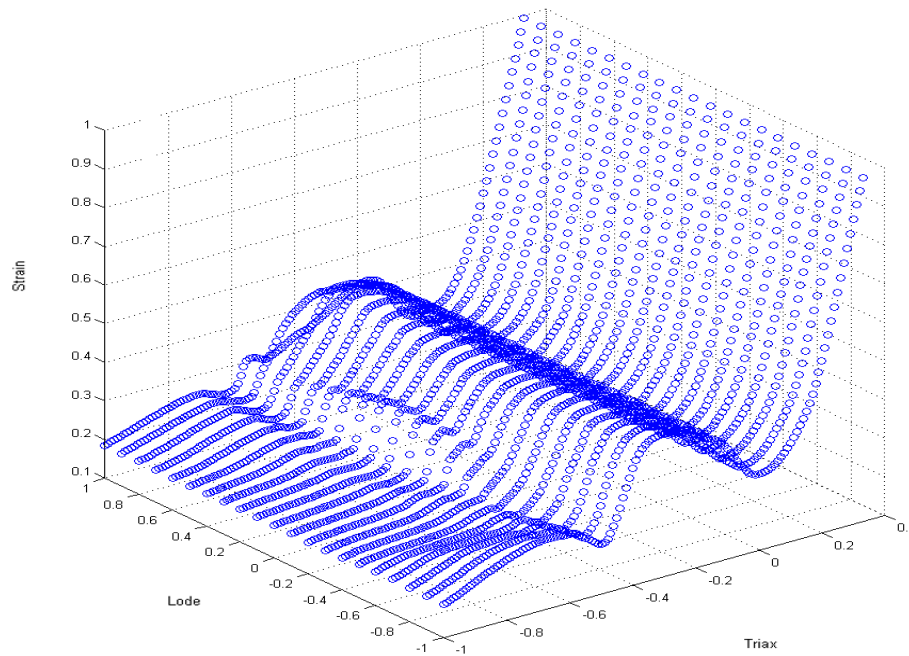
Figure 7 Discretized 3D Failure Surface for LS-DYNA®

The final step for the failure surface generation tool is to automatically generate a keyword file to use in LS-DYNA®. This file includes a *DEFINE_TABLE which has 21 values from -1 to 1 that represent the line for each Lode parameter shown in Figure 7. Also included is the 21 *DEFINE_CURVE keywords that represent the failure strain as a function of the triaxiality. Each one of these curves has 134 total points. This keyword file can then be directly included into the simulation model. This is very valuable to the analyst because there are typically many iterations of failure surface generation that must be completed to get a representative surface for a given material. After each specimen is simulated, the displacement at failure is checked against the experimental test result. If the simulation reaches the failure criteria too late (or too early) in the simulation, the control point(s) near that specimen's state of stress can be adjusted and the surface can be immediately regenerated.

It should be noted that these failure surface iterations are required because the triaxiality and Lode parameter vary during the tests and, as a result, their exact point location in Figure 4 is not definitive. In addition, the

relatively empty areas in Figure 4 where specimen tests have not been defined are used in the simulations that are compared to dynamic impact tests and, therefore, those points require adjustments to produce a good match.

5.1 Temperature Scaling Function Creation

The temperature scaling factors for the failure surface are derived from a series of tension tests at various temperatures for a constant strain rate. It is assumed that temperature scaling is independent of the strain rate because in the theoretical development the strain rate and temperature scaling are multiplicative. As a result, the effect of a change in temperature is the same at every strain rate.

The scaling function is created by separately running each temperature yield curve without failure to determine the failure strain when the analysis specimen reaches the displacement at which the test specimen failed. The scaling factors are then calculated by normalizing the failure strain to the room temperature (RT) failure strain as

$$Scaling\ Factor = \frac{\varepsilon_{pf}}{\varepsilon_{pf\ RT}}$$

Using the scaling factor and the single room temperature yield curve, the simulations are then re-run using the failure surface to make sure the analysis specimens fail at the proper displacement with minor adjustments made to the scaling factors as necessary. The material melting point temperature should also be added at the end of the curve so that high temperatures will not result in extremely high scaling factors due to extrapolation.

5.2 Strain Rate Scaling Function Creation

If a series of tests was performed at various strain rates, a strain rate scaling function for the failure surface can be developed. The analytical specimens should be simulated using the displacement-time curves from

the actual specimen tests with the exact specimen geometries. A scaling curve for strain rate can be developed and adjusted until all the specimens fail within the experimental spread of the test data.

5.3   Regularization Curve Creation

Failure strain as a function of element size is input into the material model using a mesh size regularization scaling function that is input as a load curve or table. The element size is calculated as the element volume divided by the maximum side area. The regularization curve is generated by analytical simulations of one of the test specimens using several different mesh sizes. These simulations should be performed with the single strain rate yield curve for the specific test specimen and the failure surface with no other scaling factors.

To generate the regularization scaling curve, several mesh sizes for the test specimen should be run initially with a regularization scale factor of 1.0. If the analytical simulation fails before the displacement from the physical test specimen is reached (at a smaller displacement), the scale factor is increased; and, if the analytical simulations fails after the displacement from the physical test specimen is reached (at a larger displacement), the scale factor is reduced.   Several iterations of the scale factor may be required until the analytical simulation fails at the displacement from the physical test. Additional data points should then be created so that there is no LS-DYNA® load function extrapolation for very small and very large mesh sizes.

It should be noted that this procedure is based on a single test specimen with a specific triaxiality. The regularization curve can also be input as a table where a separate load curve giving the plastic failure strain versus element size for each triaxiality is specified.   Since there is much more sensitivity to mesh size in tension failures versus shear failures, this option can improve results crossing many failure modes.

6   Material Model Validation

To validate the effectiveness of the material model input parameters including both the failure surface and scaling curves, dynamic impact tests using a rigid projectile should be completed. Results of the dynamic impact tests can then be compared directly to analytical simulations of those tests to evaluate the material model robustness and serve as validation for the material model inputs. Both dynamic punch tests and ballistic impact tests are recommended so that a range of strain rates will be evaluated for material model validation.

7 Test Program

The test program needed to develop the LS-DYNA® inputs for *MAT_224 requires mechanical property tensile tests to generate temperature and strain rate dependent yield curves, specimen failure tests for various triaxiality and Lode parameters to generate the failure surface, and dynamic impact tests to validate the model.

7.1 Stress-Strain Behavior Tests

Tensile tests need to be performed at various strain rates with a constant temperature and at various temperatures with a constant strain rate. The range of strain rates and temperatures should encompass the areas of interest for the specific impact applications being modeled. For the aluminum 2024 model that is posted on the Aerospace Working Group Webpage under the LS-DYNA® Material Parameter Sets Menu the strain rates tested included room temperature, 1E-4 sec^{-1}, 1E-2 sec^{-1}, 1.0 sec^{-1}, 500 sec^{-1}, and 1500 sec^{-1} and the temperatures tested included -50 °C, 150 °C, 300 °C, and 450 °C. For the titanium Ti-6Al-4V that is in development, the strain rates tested included 1E-4 sec^{-1}, 1E-2 sec^{-1}, 1.0 sec^{-1}, 500 sec^{-1}, and 1000 sec^{-1} and the temperatures tested included room temperature, -50 °C, 200 °C, 400 °C, and 600 °C.

7.2 Specimen Failure Tests

A set of twenty specimens have been designed to generate an effective failure surface. These specimens were designed to cover as broad a range of triaxiality and Lode parameter as possible in order to develop the most complete failure surface possible. Efforts are also continuing to design specimens that will enable the full failure surface to be developed. The specimen designs used in the aluminum 2024 tests are documented in Appendix B of Reference 2 and the specimen designs used in the titanium Ti-6Al-4V tests are documented in Section 5 of Reference 4. While it may not be feasible to complete all twenty tests, it should be recognized that

the more failure tests that are completed, the more accurate and general the failure surface will be.

7.3   Validation Tests

Dynamic impact tests such as dynamic punch tests and ballistic tests using rigid projectiles on the target material are required for final validation of the input parameters, failure surface, and scaling curves. These tests allow for a direct comparison of measured test parameters with analysis results to validate the effectiveness of the material model.

8 References

[1] Development of a New Material Model in LS-DYNA®, Part 1: FAA, NASA and Industry Collaboration Background. DOT/FAA/TC-13/25, P1. William Emmerling, Donald Altobelli, Kelly Carney, and Mike Pereira. April 2014.

[2] Development of a New Material Model in LS-DYNA®, Part 2: Development of a Tabulated Thermo-Viscoplastic Material Model with Regularized Failure for Dynamic Ductile Failure Prediction of Structures Under Impact Loading. DOT/FAA/TC-13/25, P2. Murat Buyuk. July 2014.

[3] Development of a New Material Model in LS-DYNA®, Part 3: Plastic Deformation and Ductile Fracture of 2024 Aluminum Under Various Loading Conditions. DOT/FAA/TC-13/25, P3. Jeremy Seidt. April 2014.

[4] Development of a Titanium Alloy Ti-6Al-4V Material Model Used in LS-DYNA®. DOT/FAA/AR-15/11. Sean Haight. May 26, 2015.

[5] LS-DYNA® Keyword User Manual Volume II - Material Models. LS-DYNA® R11 Revision 12960, June 22, 2020. Livermore Software Corporation.

[6] Carney K., Pereira M., Revilock D., and Matheny P., "Jet Engine Fan Blade Containment using an Alternate Geometry and the Effect of Very High Strain Rate Material Behavior," *International Journal of Impact Engineering,* 36 (2009) pp 720-728.

Appendix A Stress-Strain Relationship Modeling Procedure Flow Chart

```
                    ╭─────────────────────────────╮
                    │   Isothermal Tensile Test    │
                    │            Data              │
                    │      •  -50 °C               │
                    │      •  Room Temp            │
                    │      •  200 °C               │
                    │      •  400 °C               │
                    ╰─────────────────────────────╯
                                 │
                                 ▼
                  ╱─────────────────────────────╲
                  │      Force vs. Time           │
                  │    Displacement vs. Time      │
                  │   Quasi-Static Strain Rate    │
                  ╲─────────────────────────────╱
                                 │
                                 ▼
                  ┌─────────────────────────────┐
                  │   Plastic Strain vs. Stress  │
                  └─────────────────────────────┘
                                 │
                                 ▼
                  ┌─────────────────────────────┐
                  │ Find the Proper Artificial Loading │
                  └─────────────────────────────┘
                                 │
                                 ▼
                     ┌────────────────────┐
                     │  Find Necking Point │
                     └────────────────────┘
                                 │
                                 ▼
                         ◇ Cut at Necking ◇ ─────────────┐
                                 │                         ▼
                                 │              ┌────────────────────┐
                                 ▼              │  Before Necking:    │
              ┌─────────────────────────────┐  │   Keep the Curve    │
              │ After Necking: Extrapolate a │  └────────────────────┘
              │   Candidate Plastic Curve    │
              └─────────────────────────────┘
                                 │
                                 ▼
                     ┌────────────────────┐
                     │ Simulate the Tensile Test │
                     └────────────────────┘
                                 │
                                 ▼
              ◇ Force vs. Displacement and       ◇
         N ──◇  Strain Field in Simulation        ◇
              ◇    Close to that of test?         ◇
                                 │ Y
                                 ▼
```

┌─────────────────────────────┐
│   *Cont'd on next page*   │
└─────────────────────────────┘

28

The Set of Isothermal Input Curves

Input Displacement vs. Time as B.C. for all Simulations of Different Strain Rates

Tensile Test Data
- Strain Rate = 1E-4
- Strain Rate = 1E-2
- Strain Rate = 500
- Strain Rate = 1000

Force vs. Time
Displacement vs. Time
Quasi-Static Strain Rate

Plastic Strain vs. Stress

Find the Proper Artificial Loading

Find Necking Point

Cut at Necking

Before Necking: Keep the Curve

After Necking: Extrapolate a Candidate Plastic Curve

Simulate the Tensile Test

Force vs. Displacement and Strain Field in Simulation Close to that of test?

N

Y

The Set of Curves at Different Strain

Create Strain Rate Sensitivity Curve Using Compression and Tension Test

Create Synthetic Higher Rate Stress-Strain Curves by Translating a Lower Rate Curve (Curves must be defined higher than the highest rate seen in testing)

Assemble All the Curves Together

Assemble isothermal curves of different temperatures

Asemble isorate curves of different strain rates

Modify Strain Rate Sensitivity Curves and Beta

Simulate and Compare the Force vs. Displacement Curve between Test and Simulation

Does the Simulation Force and Strain Rate vs. Displacement Give Close Match to all Tests?
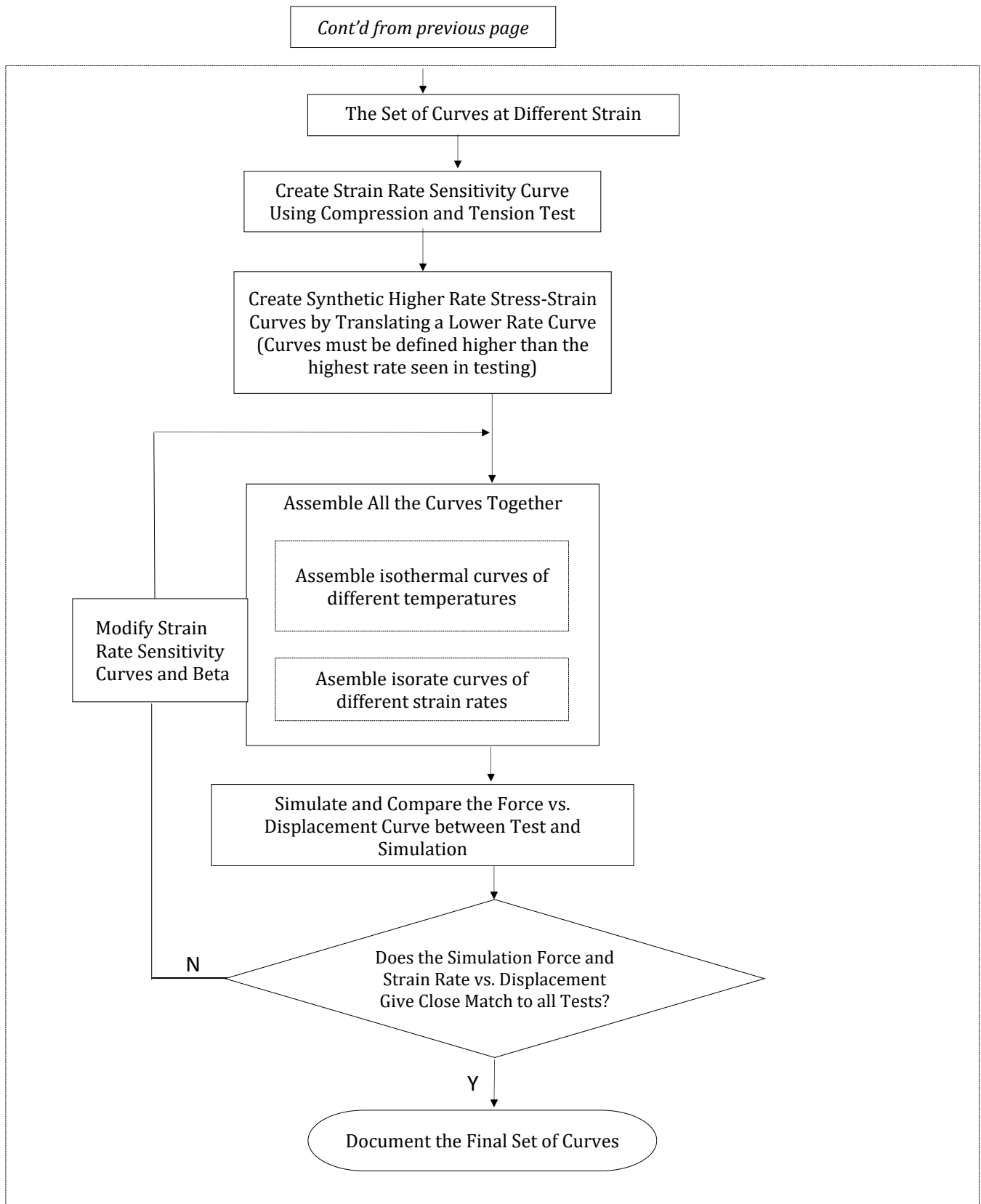
N

Y

Document the Final Set of Curves

Figure 8 Flow Chart of Stress-Strain Relationship Modeling Procedure