API Ballot id# 6124
SC5 TGTGC

Work Item	3083—Connection FEA Workflow
Type of Distribution [Ballot (vote and comment), Comment-only, Recirculation (comment resolution), Re-ballot, etc.]	Comment-only (review for commenting, no voting)
Impacted Document	RP 5C5, 4th Edition
Other Impacts	None
Revision Key	Current/unchanged content in BLACK; Track Changes as: 1) Additions in underlined BLUE [a] 2) ~~Deletions in stricken RED~~ NOTE The "******" indicates there is un-altered content above/below. [a] For Annex XX, the entirety is new, so it will be all black.

Work Item Charge: Develop a standardized FEA workflow for connection evaluation.

Ballot Rationale: The proposed work will facilitate evaluation of connections (API 5C5 Annex F), which maximizes the number of technically evaluated connections while minimizing testing costs.

NOTE See the ballot email notification for additional information regarding this ballot.

Procedures for Testing Casing and Tubing Connections

API RECOMMENDED PRACTICE 5C5
FOURTH EDITION, JANUARY 2017

ADDENDUM 1, MAY 2021

******

API NOTE: AT LEAST ONE REFERENCE TO THE ANNEX IS TO BE DETERMINED / MADE ELSEWHERE IN THE DOCUMENT.

******

Annex XX

(informative/normative)

Connection Finite Element Analysis Workflow

XX.1 Purpose and Use

This annex provides guidelines for the connection evaluation (sealability and structural) using finite element analysis (FEA), and will focus on the 2-D axisymmetric FEA models. The purpose of standardized connection FEA workflow is to ensure that a consistent analysis approach, modeling assumptions, and procedures are used among different analysts such that consistent results can be obtained.

XX.2 Connection FEA Model Set-up

XX.2.1 General

A typical connection FEA model set-up consists of the following aspects: (1) modeling geometry and dimensions, (2) material constitutive model, (3) finite element mesh, (4) contact interaction, (5) connection make-up, (6) boundary conditions and application of loads, (7) connection evaluation load cases and their corresponding loading steps. The following sub-sections will discuss each of these aspects in detail.

XX.2.2 Model Geometry and Manufacture Tolerance

The connection FEA model shall be constructed based on the latest design drawings.

The model shall use the geometry and dimensions with the worst-case manufacture tolerance for the relevant analysis. For instance, for sealability evaluation, configuration with maximum thread interference and minimum seal interference should be considered. In addition, nominal diameter and wall thickness should be used for the pipe body.

Thread, seal, and taper configurations shall be based on specimen-1 as described in XXX of this standard. Other configurations should also be considered, as desired, for other evaluation objectives.

To avoid the end effect, the modeled length of the nominal pipe section beyond the connection region shall be at least 2xD where D is the API-specified pipe outside diameter (see 3.X).

XX.2.3 Finite Element Mesh

In general, mesh sensitivity and convergence study should be performed for the selection of element-type and mesh density to achieve accurate FEA results.

The following are some recommendations based on industry FEA experience.

a) Element-type: 4-node bi-linear element with full-integration preferred. Whenever possible, triangular elements should be avoided. Care should be taken at the element selection to avoid volumetric locking (recommended elements are those with mixed formulation element with pressure degree of freedom).

This document is not an API Standard; it is under consideration within an API technical committee but has not received all approvals required to become an API Standard. It shall not be reproduced or circulated or quoted, in whole or in part, outside of API committee activities except with the approval of the Chairman of the committee having jurisdiction and staff of the API Standards Dept. Copyright API. All rights reserved.

b) Mesh size:

1) A minimum of 5 elements along the fillet radii at threads and other transition regions.

2) Element sizes on metal-to-metal seal surfaces, torque shoulders, and threads shall not exceed 0.005 in.

3) The mesh size in the uniform section of the pipe beyond the transition region up to half nominal diameter (D) away from the connection should not exceed 0.1 in.; however, this dimension may vary depending on the size of the pipe to be modelled.

4) A minimum of 6 elements through the wall thickness at far-field pipe regions.

5) Structural mesh should be used when possible (e.g., in the far-field pipe sections).

6) Bias meshing technique should be used for the mesh transition from the fine mesh regions (e.g., thread, seal, shoulder, and other areas containing geometric change) to the coarse mesh region (e.g., far-field pipe uniform sections).

XX.2.4 Material Modeling

Non-linear elastic-plastic material models shall be used in the connection FEA. It is recommended that non-linear strain hardening behavior is included in the material plasticity modeling. There are numerous plasticity models in literature, for instance, the Ramberg-Osgood model, Needleman plasticity model, and MPC (Material Property Council) stress-strain curve model, and so on. Of these, both the MPC and Ramberg-Osgood models are adopted in API 579-1/ASME FFS-1 [xx] as well as in ASME BPVC Section VIII Division 2 [xx] and Division 3 [xx]. The MPC model has been validated against OCTG steel grades [xx]. Therefore, it is recommended that the MPC model be used in the connection FEA modeling. Detailed stress-strain equations of the MPC model can be found in API 579-1/ASME FFS-1 [xx] as well as in ASME BPVC VIII-2 [xx] and VIII-3 [xx].

Additionally, the isotropic hardening rule should be used for monotonic loadings in most cases. However, the combined kinematic-isotropic hardening rule may be needed when the evaluation involves loading, unloading, and reverse loading, to capture the potential Bauschinger effect.

Finally, tabulated data pairs of true stress versus plastic strain, extracted from the stress-strain equation with sufficient data points to capture the non-linear behavior from proportional limit to beyond the yield point, should be used in the connection FEA material modeling.

In general, both the specified minimum yield strength (SMYS) and specified minimum tensile strength (SMTS) shall be used in the evaluations. If the material is a standard API OCTG-grade, the SMYS and SMTS values in API 5CT shall be used. On the other hand, if the material is a manufacturer's proprietary grade, the manufacturer's SMYS and SMTS values for that material shall be used.

If purpose of the FEA is to compare the FEA results to actual connection testing results, the material stress-strain data/curve from the actual material tensile tests on the coupons cut from the adjacent pup regions of the actual connection test sample should be used. The smoothed actual material stress-strain curve or the curve-fit data using one of the plasticity models may be used. It should be noted that the stress-strain data from a material test is typically presented in engineering stress-strain pairs, which shall be converted to true stress-strain pairs. Then, the true stress versus plastic strain data can be used in the FEA material modeling.

Additionally, the material plasticity curve is capped at true ultimate tensile stress and assumed to be perfectly plastic after that. In other words, the material is allowed to deform freely with no further increase in stress beyond the true ultimate tensile stress point.

See XX.6 for MPC material plasticity curves and tabulated true stress-plastic strain data for standard API OCTG-grades (H40, J55, K55, L80, N80, C90, R95, T95, C110, P110, and Q125). These tabulated data are intended for use in connection evaluation FEA modeling software.

XX.2.5 Contact Interaction

One of the major non-linearities in the connection FEA model is contact, which occurs at the interfaces between pin and box threads, metal-to-metal seals, and torque shoulders. For contact modeling, the following are recommended.

- Surface-to-surface discretization.
- Finite sliding formulation.
- Contact properties including zero coefficient of friction and “hard” contact normal behavior.

It should be noted that the coefficients of friction used to estimate the connection make-up torque, based on the torque output from the FEA program, are different from the ones used in the FEA contact modeling.

In rare situations, where numerical convergence becomes an issue, using a small value for the non-zero friction coefficient may be helpful to resolve the issue and may be allowed. This should be reported in the FEA report.

XX.2.6 Connection Make-up

In 2-D axisymmetric connection FEA modeling, connection make-up is typically performed through interference fits in multiple steps prior to the application of the external loads (e.g., internal/external pressure, axial loads, bending moments, etc.). The make-up steps typically consist of the following:

- Step 1: Resolve the interference fit at the interface between pin and box threads, often described as shrink-fit in the commercial FEA software. If the connection is threaded and coupled (T&C) and the field-end make-up is to be affected by the mill-end make-up, then, the threads on both sides may be resolved in two separate steps.
- Step 2: Resolve the interference fit at the metal-to-metal seal interface, if applicable.
- Step 3: Resolve the interference fit at the torque shoulder, if the connection has a torque shoulder.

The coefficient of friction used in the connection make-up torque estimation, based on the FEA torque output (e.g., output variable CTRQ in Abaqus), depends on surface roughness, thread compound, etc., and should be calibrated through testing. In addition, the threads, metal-to-metal seal, and shoulder may have different values for their coefficient of friction and will need to be calibrated separately.

By adjusting the amounts of initial interference, the FEA predicted connection make-up torque should be calculated using the calibrated coefficients of friction and should match the target make-up torque, such as the minimum make-up torque, as specified in the connection design datasheet. The torque in 2-D models should be calculated using the integration of the contact pressures along the different connection contact areas (thread, seal, shoulder) multiplied by the corresponding average radius for each segment and the corresponding coefficient of friction.

One modeling technique to mimic the shoulder make-up in commercial FEA software is to adjust the length of a bolt pre-tension section. This allows the amount of delta torque to be precisely simulated without having to change the length of the pin shoulder region, which can be tedious when several iterations are involved to achieve the accurate target make-up torque.

XX.2.7 Loads and Boundary Conditions

Boundary conditions in a 2-D axisymmetric connection FEA model should consist of the following aspects.

- The connection assembly shall not be externally constrained in the radial direction in any case.

- Restraining the axial displacement at one end of the far-field pipe ends for an integral connection or at the center symmetric plane of the coupling for a T&C connection.
- During the connection make-up steps (i.e., resolving the contact interferences), the other far-field pipe end (integral), or center of the coupling (T&C), may be constrained in the axial direction to prevent rigid body motion mode that may potentially lead to numerical convergence issue. It is important to note that these boundary conditions are temporary and shall be removed in an analysis step right after the connection make-up and prior to the application of any external loads.
- The temperature shall be set at that of the test series being modelled, when applicable.

Once the connection assembly is made-up to the desired target torque, the external loads can then be applied in the subsequent steps as needed. Different load cases involve different axial and pressure load combinations at the desired temperature.

For a premium connection with a metal-to-metal seal, only the metal-to-metal seal shall be considered as the seal element for pressure applications. In other words, the threads with thread compound and the shoulder contact interface shall not be considered as sealing elements for internal and external pressure loads.

The following are some additional recommendations for load application.

- Internal pressure shall be applied to all interior surfaces up to the metal-to-metal seal location including the shoulder contact interface. After that point, pressure penetration should be implemented to accurately capture the pressure penetrating through the interior portion of the seal when the applied internal pressure is high enough to do so. Alternatively, a conservative approach of applying the internal pressure up to the outer-most location of the seal may be used.
- External pressure shall be applied to all exterior surfaces up to the metal-to-metal seal location including the external shoulder and thread surfaces. In the case of connections with external seal, external pressure shall be applied up to the external seal location. After that point, pressure penetration should be implemented to accurately capture the pressure penetrating through the exterior portion of the seal when the applied external pressure is high enough to do so. Alternatively, a conservative approach of applying the external pressure up to the inner-most location of the seal may be used.
- Axial load (tension or compression), and the capped-end pressure end load should be applied as pressure-type loads at the free end of the uniform pipe section. It is noted that the boundary condition of restraining axial displacement is enforced at the other end of the uniform pipe section or center symmetric plane of the coupling. Other modeling techniques may be used to apply the axial load and pressure end load, such as a point load at the pipe center together with a kinematic coupling or surface traction at the free end of the uniform pipe section.

FEA result outputs, including stress, strain, and displacement, shall always be both checked and verified to ensure that the loads and boundary conditions are applied correctly.

XX.2.8 Analysis Cases and Load Combinations

API 5C5 load schedules should be used for the analysis. Alternate connection analysis load schedules may be agreed between the connection manufacturer and the end-user.

Load schedules shall be calculated using nominal pipe body dimensions and SMYS, unless something different is agreed between manufacturer and end-user.

In general, the load cases and schedules include axial load (tension/compression), capped-end or open-end internal pressure, external pressure, and bending moment, at ambient and elevated temperatures, when applicable. In addition, load schedules may be cyclic and include loading, unloading, and reverse loading.

XX.3 Model Check and Validation

As a rule of thumb, FEA results (such as component stresses, strains, and displacements) shall always be checked to make sure that they make sound engineering sense and agree with engineering first principles. The first thing is to check the far-field component stresses in the uniform pipe sections against the theoretical hand calculations (such as hoop, axial stress, and so on), to ensure that the loads and boundary conditions are applied as desired.

XX.4 Analysis Results

The analysis of results will be at the discretion of the end-user.

If stabilization is used for convergence, then the maximum ratio of stabilization energy to internal energy should not exceed 5 %.

XX.5 Reporting

The content of the report should be agreed upon between end-user and analyst. At a minimum, the connection FEA evaluation report shall include the following contents:

a) Final as-modeled schematic, including the drawing number and revision used for the analysis.

b) Name and version of the FEA software used in the model construction, analysis runs, and result post-processing; software tool (plug-in) used to automate the processes, if any.

c) Detailed descriptions and figures of model geometry according to the API 5C5-specimen selected for the analysis and finite element mesh (element-type, mesh distribution, and mesh density).

d) Descriptions and figures of contact formulation, surface discretization method, and coefficients of friction used for thread engagement, the seal interface, shoulder contact, and so on.

e) Material physical properties (i.e., modulus of elasticity, Poisson’s ratio, yield and ultimate tensile strengths; mass density, thermal expansion coefficient, and other properties, if used) and non-linear plasticity model utilized in the analysis.

f) Descriptions of material constitutive model, plasticity model (tabulated material property data and plot of true stress-strain curve) used in the analysis.

g) Description of the analysis procedure (e.g., static analysis, dynamic analysis, and thermal analysis, etc.) and if geometrical non-linearity is invoked.

h) Descriptions of modeling techniques used for connection make-up simulation, such as interference fit, bolt pre-tension section for shoulder make-up, and so on.

i) Description of modeling technique used for bending simulation if applicable.

j) Loading and boundary conditions used to address the analysis cases considered in the FEA.

k) Detailed descriptions of load cases and load combinations as well as their relevant analysis steps.

l) A summary of the FEA results and evaluations

m) Analysis results and assessments, including tabulated data and graphical contour plots.

    1) Contact pressure distribution along the metal-to-metal seal surface.

    2) Stress and strain (including equivalent plastic strain) contour plots of the entire model, thread areas, shoulder contact area, metal-to-metal seal area, and other critical areas of interest.

n) Calibration/validation of the FEA model.

XX.6 Material Plasticity Model for Standard API Grades

The non-linear stress-strain curves of API grades provided in this appendix are constructed using the MPC plasticity model with the SMYS and SMTS as provided in the API 5CT (tables for *Tensile and Hardness Requirements*). It is noted that these values are the 5CT-spec minimum required values at ambient temperature.

NOTE The temperature degradation equations on Young's modulus, yield strength, and ultimate tensile strength for API OCTG-grades are pending development.

Stress is capped at the material's ultimate tensile strength where necking instability occurs in a uniaxial material tensile test. Beyond this point (ultimate tensile strength), the material is assumed to be perfectly plastic (i.e., the stress will remain constant at the ultimate tensile strength for any further increase in strain).

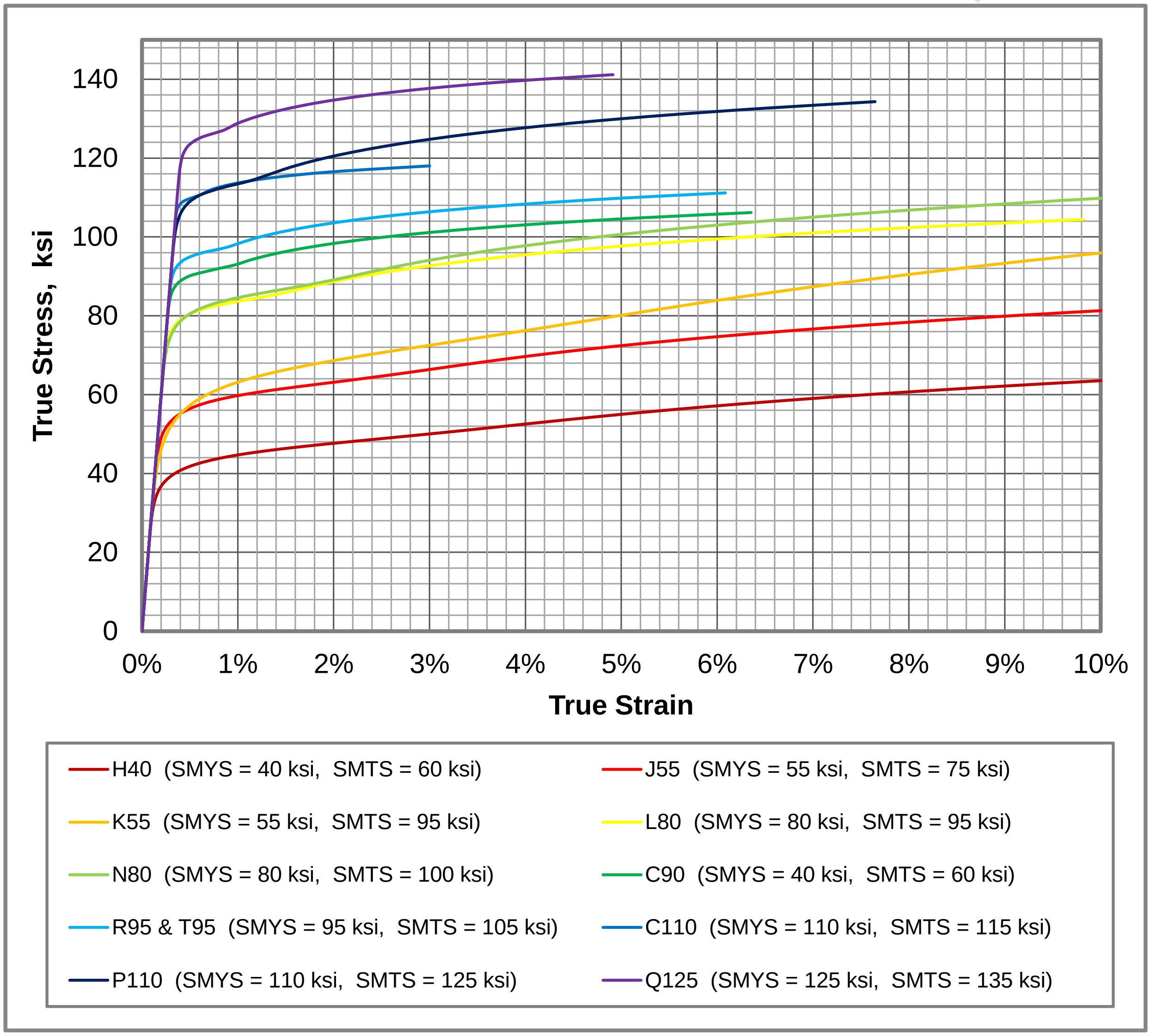

Figure XX.1—True Stress-Strain Curves, Standard API 5CT OCTG-Grades, MPC Plasticity Model

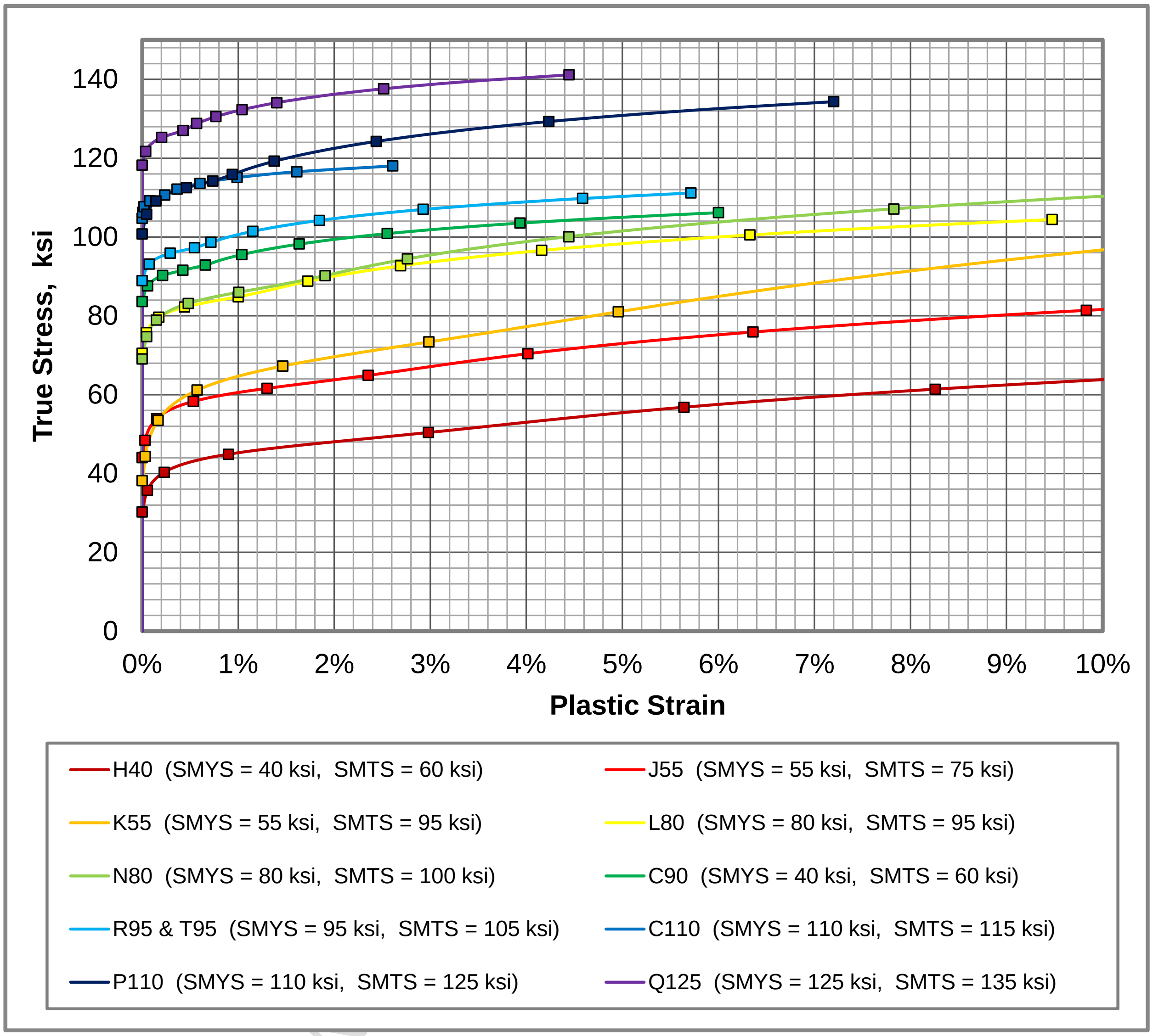

Figure XX.2—True Stress-Plastic Strain Curves, Standard API 5CT OCTG-Grades, MPC Plasticity Model

Table XX.1—Material Plasticity Input to FEA Software: True Stress vs. Plastic Strain, Standard API 5CT OCTG-Grades

H40 (SMYS = 40 ksi & SMTS = 60 ksi)		**J55 (SMYS = 55 ksi & SMTS = 75 ksi)**		**K55 (SMYS = 55 ksi & SMTS = 95 ksi)**		**L80 (SMYS = 80 ksi & SMTS = 95 ksi)**		**N80 (SMYS = 80 ksi & SMTS = 100 ksi)**	
True Stress	**Plastic Strain**	**True Stress**	**Plastic Strain**	**True Stress**	**Plastic Strain**	**True Stress**	**Plastic Strain**	**True Stress**	**Plastic Strain**
psi	**--**	**psi**	**--**	**psi**	**--**	**psi**	**--**	**psi**	**--**
3.023E+04	0.000E+00	4.401E+04	0.000E+00	3.822E+04	0.000E+00	7.050E+04	0.000E+00	6.906E+04	0.000E+00
3.573E+04	5.551E-04	4.841E+04	2.969E-04	4.434E+04	3.266E-04	7.572E+04	4.425E-04	7.470E+04	4.741E-04
4.031E+04	2.296E-03	5.391E+04	1.504E-03	5.351E+04	1.658E-03	7.964E+04	1.727E-03	7.892E+04	1.499E-03
4.489E+04	8.999E-03	5.831E+04	5.331E-03	6.115E+04	5.734E-03	8.225E+04	4.421E-03	8.315E+04	4.810E-03
5.038E+04	2.980E-02	6.161E+04	1.300E-02	6.727E+04	1.464E-02	8.486E+04	1.001E-02	8.597E+04	1.006E-02
5.680E+04	5.641E-02	6.491E+04	2.353E-02	7.338E+04	2.985E-02	8.877E+04	1.723E-02	9.020E+04	1.905E-02
6.138E+04	8.257E-02	7.041E+04	4.016E-02	8.103E+04	4.959E-02	9.269E+04	2.690E-02	9.443E+04	2.763E-02
6.504E+04	1.102E-01	7.591E+04	6.360E-02	9.784E+04	1.046E-01	9.661E+04	4.161E-02	1.001E+05	4.443E-02
6.962E+04	1.548E-01	8.141E+04	9.831E-02	1.116E+05	1.758E-01	1.005E+05	6.329E-02	1.071E+05	7.826E-02
7.328E+04	2.000E-01	8.801E+04	1.600E-01	1.223E+05	2.526E-01	1.044E+05	9.474E-02	1.127E+05	1.200E-01

C90 (SMYS = 90 ksi & SMTS = 100 ksi)		**R95 & T95 (SMYS = 95 ksi & SMTS = 105 ksi)**		**C110 (SMYS = 110 ksi & SMTS = 115 ksi)**		**P110 (SMYS = 110 ksi & SMTS = 125 ksi)**		**Q125 (SMYS = 125 ksi & SMTS = 135 ksi)**	
True Stress	**Plastic Strain**	**True Stress**	**Plastic Strain**	**True Stress**	**Plastic Strain**	**True Stress**	**Plastic Strain**	**True Stress**	**Plastic Strain**
psi	**--**	**psi**	**--**	**psi**	**--**	**psi**	**--**	**psi**	**--**
8.362E+04	0.000E+00	8.894E+04	0.000E+00	1.048E+05	0.000E+00	1.007E+05	0.000E+00	1.182E+05	0.000E+00
8.760E+04	5.760E-04	9.311E+04	7.378E-04	1.062E+05	5.188E-05	1.058E+05	4.661E-04	1.217E+05	3.788E-04
9.026E+04	2.128E-03	9.589E+04	2.930E-03	1.077E+05	2.037E-04	1.091E+05	1.429E-03	1.253E+05	2.040E-03
9.158E+04	4.226E-03	9.728E+04	5.450E-03	1.092E+05	7.851E-04	1.125E+05	4.608E-03	1.270E+05	4.251E-03
9.291E+04	6.591E-03	9.867E+04	7.171E-03	1.107E+05	2.363E-03	1.142E+05	7.358E-03	1.288E+05	5.682E-03
9.557E+04	1.038E-02	1.014E+05	1.151E-02	1.121E+05	3.652E-03	1.159E+05	9.396E-03	1.306E+05	7.692E-03
9.822E+04	1.636E-02	1.042E+05	1.847E-02	1.136E+05	6.027E-03	1.192E+05	1.375E-02	1.323E+05	1.040E-02
1.009E+05	2.552E-02	1.070E+05	2.927E-02	1.151E+05	9.884E-03	1.243E+05	2.438E-02	1.341E+05	1.402E-02
1.035E+05	3.935E-02	1.098E+05	4.585E-02	1.166E+05	1.611E-02	1.293E+05	4.234E-02	1.376E+05	2.514E-02
1.062E+05	6.000E-02	1.112E+05	5.714E-02	1.180E+05	2.609E-02	1.343E+05	7.200E-02	1.411E+05	4.444E-02