API Ballot id# 6417
SC5 TG TGC

Work Item Number	3083
Title of Work Item	Standardized FEA workflow for connection Evaluation
Ballot Revision Level	0
Type of Ballot (Initial, Comment, Comment resolution (reference API ballot#), 1st Re-ballot, 2nd Re-ballot, etc.)	Initial Letter Ballot
Submitter Name(s)	David Coe, Anjali Prasad
API Document Modified	API RP 5C5
Revision Key	0

Work Item Charge: The charge of this work item is the development of a standardized FEA workflow for connection evaluation. The proposed work will facilitate evaluation of connections (API RP 5C5 Annex F), which maximizes the number of technically evaluated connections while minimizing testing costs.

Ballot Rationale: The work group has resolved comments from the 2023 Comment Ballot and recommends moving forward with the Letter Ballot.

Annex XX

(informative)

Connection Finite Element Analysis Guidance

XX.1 Purpose and Use

This annex provides guidelines for the connection 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, accurate analysis approach, modeling assumptions, and procedures are used among different analysts such that consistent results can be obtained.

XX.2 Connection FEA Model Setup

XX.2.1 General

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

In general, there are non-linearities involved in a connection FEA including geometrical nonlinearity (finite deformation), material plasticity, and contact interactions at threads, shoulder and seal if exists. All of these nonlinearities should be incorporated in the connection FEA model.

XX.2.2 Model Geometry and Dimensions

Depending on the purpose of the FEA model, the modeled geometry and dimensions can vary.

For the final design analysis and calculations, the connection FEA model shall be constructed based on the final design drawings.The worst-case manufacture tolerances shall be included in 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 sections. 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 3xD where D is the API-specified pipe outside diameter (see 3.3). Total model length should be at least 4xD. Additionally, the modeled pipe length should have a minimum unsupported pup joint length L_{pj} (see Figure 16) that is calculated from Equation (4).

If the purpose of the FEA is to be compared with the actual connection test results, the measured dimensions of the test specimen should be used in the FEA model.

XX.2.3 Finite Element Mesh

In general, a mesh sensitivity and convergence study shall 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). If the mesh sensitivity study demonstrates negligible impact on the analysis results, other element types and mesh density may be applied.

b) Mesh size:
   1) A minimum of 5 elements along the fillet radii at threads and other transition regions is recommended.
   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).
   7) Mesh shall not contain more than 1% distorted elements where distorted element is defined as: Quad-Face Corner Angle less than 30º or greater than 150 º; aspect ratio greater than 5.0. In case, the distorted elements shall not occur at critical regions such as seal, thread, or shoulder.

XX.2.4 Material Modeling

Non-linear elastic-plastic material models should 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 [1] as well as in ASME BPVC Section VIII Division 2 [2] and Division 3 [3]. The MPC model has been validated against OCTG steel grades [4]. 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 [1] as well as in ASME BPVC VIII-2 [2] and VIII-3 [3].

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 analysis 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.

For connection design analysis or evaluation of connection performance properties, , 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 at ambient temperature listed 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 at ambient temperature for that material shall be used.

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 for ambient temperature and are ready to use in connection evaluation FEA software.

if the analysis is for elevated temperature, then the material properties (including young's modulus, yield strength, and ultimate tensile strength) at the corresponding elevated temperature shall be used in the elastic-plastic material modeling.

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, torque shoulders, and elsewhere. Contact pairs shall allow movement and separation between each mating pair to match the expected interface behavior. The surfaces of a connection shall be allowed to re-establish contact as loads vary through the analysis. For contact modeling, the following are recommended.

— Surface-to-surface discretization is recommended unless there is convergence issue, for which, node-to-surface discretization may be used providing that the contact penetration and mesh sizes at master and slave surfaces are checked and validated.

— Finite sliding formulation.

— Contact properties including zero coefficient of friction (frictionless) and "hard" contact normal behavior. When numerical convergence becomes an issue, some small value of coefficient of friction may help. The use of non-zero coefficient of friction and its value shall be documented in the FEA report.

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 Makeup

In 2-D axisymmetric connection FEA modeling, connection make-up is typically performed through interference fits in one or multiple steps prior to the application of 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 results, depends on surface topology, 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 may need to be calibrated separately. 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 axisymmetric models should be calculated as the summation of the multiplication among the contact force at each node, the corresponding radius of that node and the corresponding coefficient of friction along all contact interfaces (threads, seal, and torque shoulder).

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 of the far-field pipe ends or at the center symmetric plane of the coupling for a T&C connection when half of the T&C connection is modeled.

- During the connection make-up steps (i.e., resolving the contact interferences), the other far-field pipe end, 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 should 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 may 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 may be used. For example, the internal pressure may be applied all the way up to the outer-most location of the seal.

- External pressure shall be applied to all exterior surfaces up to the metal-to-metal seal location (including the external shoulder if any 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 may 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 may be used. For example, the external pressure may be applied all the way up to the inner-most location of the seal.

- 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.

XX.2.8 Analysis Cases and Load Combinations

Load schedule considered in the FEA should be chosen from API 5C5 load schedules. Intermediate (transition) load points used in the test execution may be omitted in the FEA. 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 an alternative approach is agreed between manufacturer and the 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) should 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 and analytical values obtained (such as hoop, axial stress, and so on), to ensure that the loads and boundary conditions are applied as intended.

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

XX.4 Analysis Results

The interpretation of the analysis results will be at the discretion of the manufacturer and end-user, but the following FEA results are seen as potentially informative for structural and sealability evaluations of one connection design relative to another connection design which has already undergone an API RP 5C5 test successfully.

- Contact pressure along the seal(s).
- Seal contact length
- Seal contact location relative to a constant location (pin nose, box shoulder, etc.)
- Stress and strain contour plots of the connection model (thread areas, shoulder, contact areas, metal-to-metal seal areas(s), and other critical areas of interest)

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 should 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.

c) Detailed descriptions and figures of FEA model geometry and finite element mesh (element-type, number of elements, mesh distribution, and mesh density).

d) Descriptions (and figures if needed) of contact formulation, surface discretization method, and coefficient(s) of friction used for each contact pair..

e) Material physical properties (i.e., modulus of elasticity, Poisson’s ratio, yield and ultimate tensile strengths; thermal expansion coefficient, and other properties, if used).

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

g) Description of the analysis procedure.

h) Descriptions of modeling techniques used for connection make-up simulation, such as interference fit, approaches to achieve the target make-up torque condition, coefficients of friction used in the connection make-up torque calculation, and so on.

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

j) Figures and descriptions of load and boundary conditions used to address the analysis cases considered in the FEA.

k) Detailed descriptions of load cases and load combinations (e.g. each component of the axial load, total axial load, bending moment if applied, internal pressure, external pressure, and so on) as well as their relevant analysis steps.

l) A summary of the FEA results and evaluations. Detailed analysis results and assessments, including tabulated data and/or graphical plots such as 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. For equivalent stress plots, material yield strength used in the analysis may be used as legend max (scale) while 0.2% (engineering offset for yield strength definition) may be used for equivalent plastic strain plots.

m) Calibration/validation of the FEA model (XX.3).

XX.6 Material Plasticity Model for Standard API OCTG Grades at Ambient Temperature

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.

Stress is capped at the material’s ultimate tensile strength where necking instability starts 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).

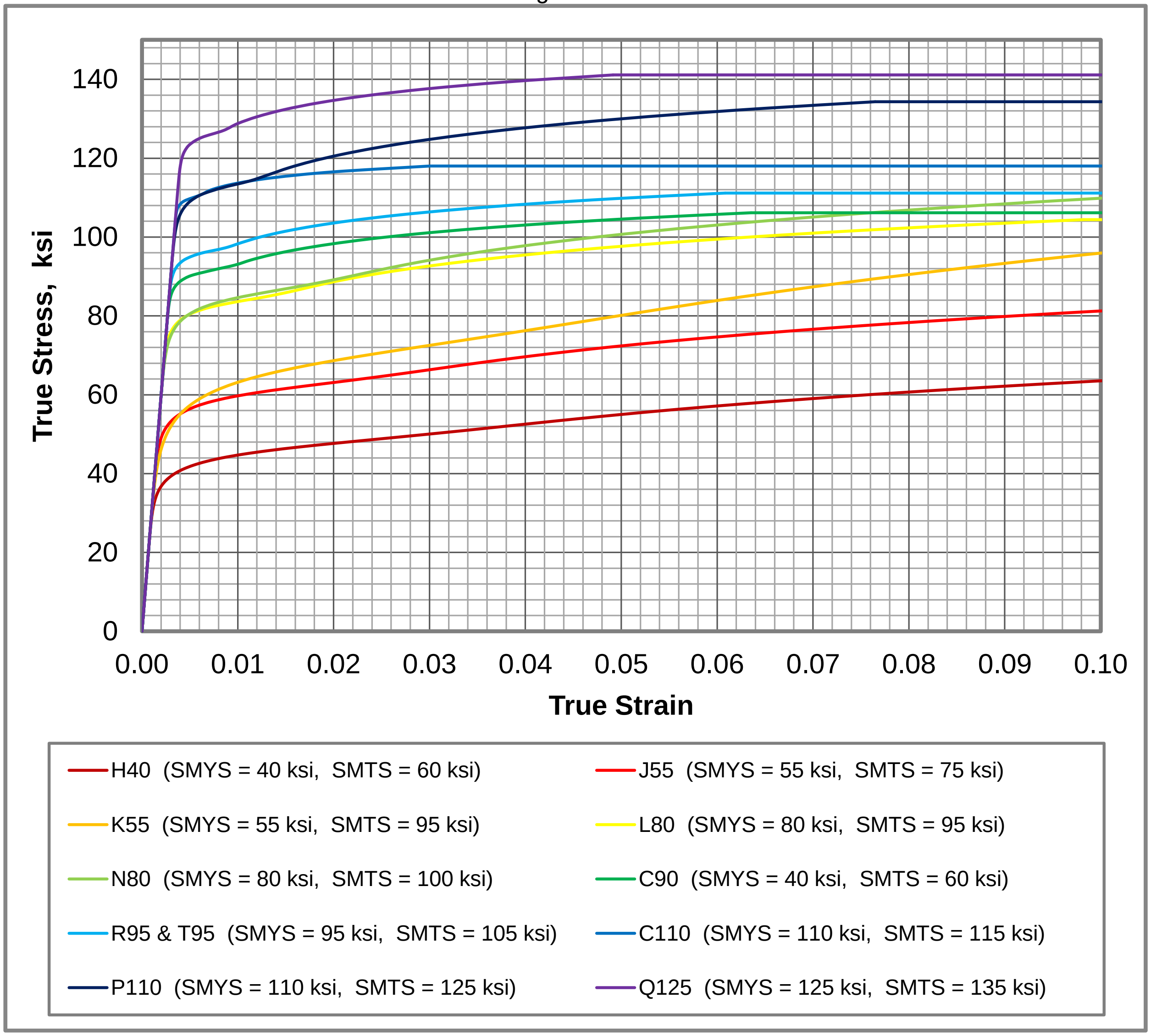

Figure XX.1–True Stress-Strain Curves, Standard API 5CT OCTG Grades, MPC Plasticity Model, Specified Minimum Values at Ambient Temperature

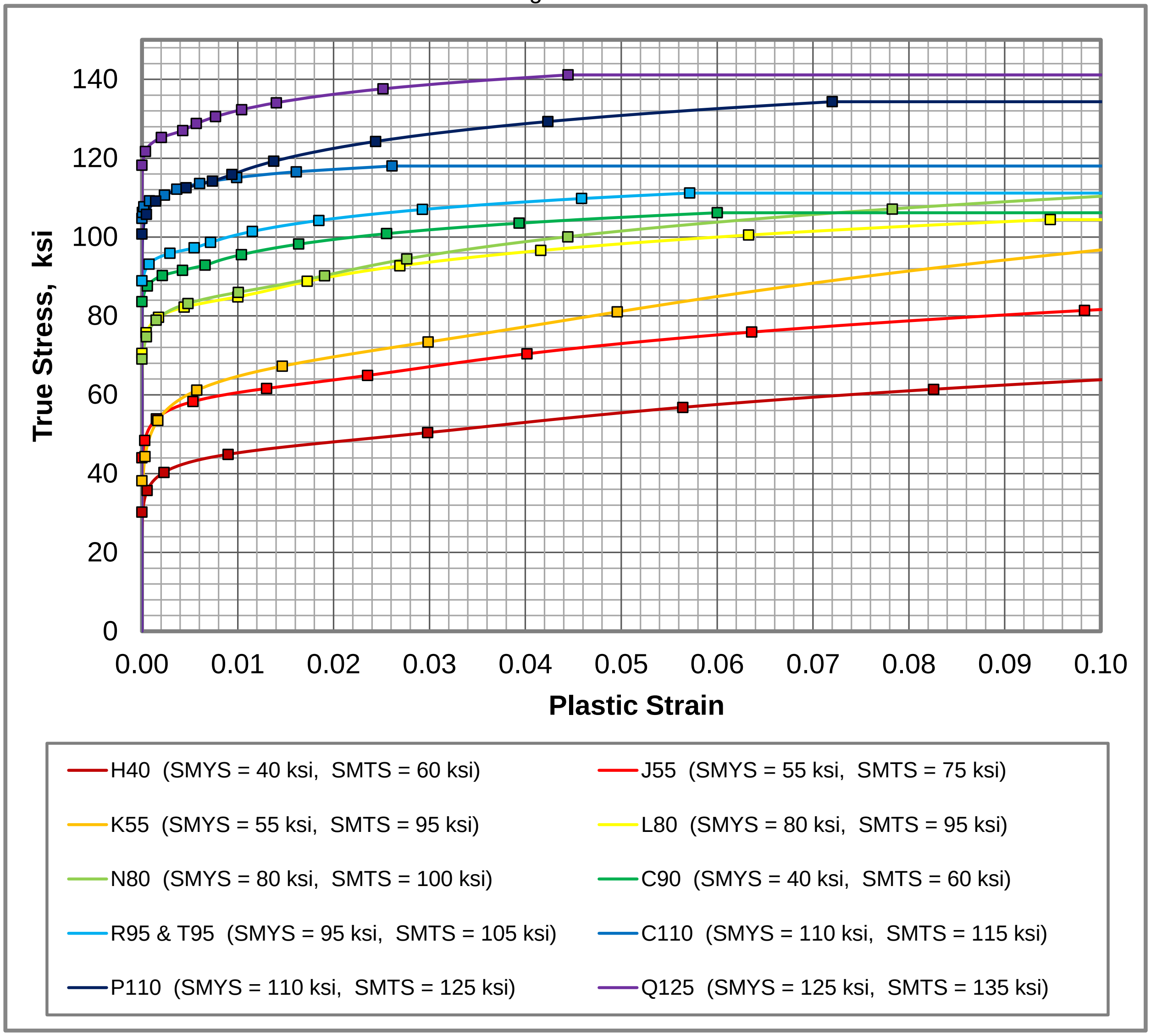

Figure XX.2.–True Stress-Plastic Strain Curves, Standard API 5CT OCTG Grades, MPC Plasticity Model, Specified Minimum Values at Ambient Temperature

Table XX.A.1. Material Plasticity Model Input to FEA Software – True Stress versus Plastic Strain, Standard API 5CT OCTG Grades, MPC Plasticity Model, Specified Minimum Values at Ambient Temperature

H40 (SMYS = 40 ksi & SMTS = 60 ksi)	
True Stress	Plastic Strain
psi	--
3.023E+04	0.000E+00
3.573E+04	5.551E-04
4.031E+04	2.296E-03
4.489E+04	8.999E-03
5.038E+04	2.980E-02
5.680E+04	5.641E-02
6.138E+04	8.257E-02
6.504E+04	1.102E-01
6.962E+04	1.548E-01
7.328E+04	2.000E-01
7.328E+04	1.000E+00

J55 (SMYS = 55 ksi & SMTS = 75 ksi)	
True Stress	Plastic Strain
psi	--
4.401E+04	0.000E+00
4.841E+04	2.969E-04
5.391E+04	1.504E-03
5.831E+04	5.331E-03
6.161E+04	1.300E-02
6.491E+04	2.353E-02
7.041E+04	4.016E-02
7.591E+04	6.360E-02
8.141E+04	9.831E-02
8.801E+04	1.600E-01
8.801E+04	1.000E+00

K55 (SMYS = 55 ksi & SMTS = 95 ksi)	
True Stress	Plastic Strain
psi	--
3.822E+04	0.000E+00
4.434E+04	3.266E-04
5.351E+04	1.658E-03
6.115E+04	5.734E-03
6.727E+04	1.464E-02
7.338E+04	2.985E-02
8.103E+04	4.959E-02
9.784E+04	1.046E-01
1.116E+05	1.758E-01
1.223E+05	2.526E-01
1.223E+05	1.000E+00

L80 (SMYS = 80 ksi & SMTS = 95 ksi)	
True Stress	Plastic Strain
psi	--
7.050E+04	0.000E+00
7.572E+04	4.425E-04
7.964E+04	1.727E-03
8.225E+04	4.421E-03
8.486E+04	1.001E-02
8.877E+04	1.723E-02
9.269E+04	2.690E-02
9.661E+04	4.161E-02
1.005E+05	6.329E-02
1.044E+05	9.474E-02
1.044E+05	1.000E+00

N80 (SMYS = 80 ksi & SMTS = 100 ksi)	
True Stress	Plastic Strain
psi	--
6.906E+04	0.000E+00
7.470E+04	4.741E-04
7.892E+04	1.499E-03
8.315E+04	4.810E-03
8.597E+04	1.006E-02
9.020E+04	1.905E-02
9.443E+04	2.763E-02
1.001E+05	4.443E-02
1.071E+05	7.826E-02
1.127E+05	1.200E-01
1.127E+05	1.000E+00

C90 (SMYS = 90 ksi & SMTS = 100 ksi)	
True Stress	Plastic Strain
psi	--
8.362E+04	0.000E+00
8.760E+04	5.760E-04
9.026E+04	2.128E-03
9.158E+04	4.226E-03
9.291E+04	6.591E-03
9.557E+04	1.038E-02
9.822E+04	1.636E-02
1.009E+05	2.552E-02
1.035E+05	3.935E-02
1.062E+05	6.000E-02

R95 & T95 (SMYS = 95 ksi & SMTS = 105 ksi)	
True Stress	Plastic Strain
psi	--
8.894E+04	0.000E+00
9.311E+04	7.378E-04
9.589E+04	2.930E-03
9.728E+04	5.450E-03
9.867E+04	7.171E-03
1.014E+05	1.151E-02
1.042E+05	1.847E-02
1.070E+05	2.927E-02
1.098E+05	4.585E-02
1.112E+05	5.714E-02

C110 (SMYS = 110 ksi & SMTS = 115 ksi)	
True Stress	Plastic Strain
psi	--
1.048E+05	0.000E+00
1.062E+05	5.188E-05
1.077E+05	2.037E-04
1.092E+05	7.851E-04
1.107E+05	2.363E-03
1.121E+05	3.652E-03
1.136E+05	6.027E-03
1.151E+05	9.884E-03
1.166E+05	1.611E-02
1.180E+05	2.609E-02

P110 (SMYS = 110 ksi & SMTS = 125 ksi)	
True Stress	Plastic Strain
psi	--
1.007E+05	0.000E+00
1.058E+05	4.661E-04
1.091E+05	1.429E-03
1.125E+05	4.608E-03
1.142E+05	7.358E-03
1.159E+05	9.396E-03
1.192E+05	1.375E-02
1.243E+05	2.438E-02
1.293E+05	4.234E-02
1.343E+05	7.200E-02

Q125 (SMYS = 125 ksi & SMTS = 135 ksi)	
True Stress	Plastic Strain
psi	--
1.182E+05	0.000E+00
1.217E+05	3.788E-04
1.253E+05	2.040E-03
1.270E+05	4.251E-03
1.288E+05	5.682E-03
1.306E+05	7.692E-03
1.323E+05	1.040E-02
1.341E+05	1.402E-02
1.376E+05	2.514E-02
1.411E+05	4.444E-02

1.062E+05	1.000E+00

1.112E+05	1.000E+00

1.180E+05	1.000E+00

1.343E+05	1.000E+00

1.411E+05	1.000E+00

XX.7 References

1. API 579-1/ASME FFS-1, Fitness-for-Service, 2021, Washington D.C.: API; New York, NY: ASME.
2. ASME Boiler and Pressure Vessel Code, Section VIII Division 2, Rules for Construction of Pressure Vessels, Alternative Rules, 2023, New York, NY, USA: ASME.
3. ASME Boiler and Pressure Vessel Code, Section VIII Division 3, Rules for Construction of Pressure Vessels, Alternative Rules for Construction of High Pressure Vessels, 2023, New York, NY, USA: ASME.