International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 12, Number 3 (2019), pp. 333-336

ASME Section VIII Div. 2 Finite Element Elastic Plastic Analysis Method– A Case Study

A. S. El-Kabbany Ph.D., [1] **and Yu Miao Ph.D.,**[2*]

[1]*Lanmark Engineering Inc., Calgary, T2R0E1, Canada.*

[2]*Lanmark Engineering Inc., Calgary, T2R0E1, Canada.*

**Corresponding author*

Abstract:

A current challenge in the pipeline bulging repair is to install the sleeve assemblies during the pipeline operation; thus to avoid costly shutdown of the entire pipeline. This repair operation is feasible; however the stress validation should be conducted to ensure the integrity of the piping system design. This case study presents the stress assessment case study of pressure containing assembly using the design-by-analysis methodology in accordance with ASME Section VIII Div. 2 Part 5.2.4.

Keyword: bulging, ASME, Plasticity, Stress, FEA

1. INTRODUCTION

The purpose of this case study is to assess the static stresses in a repair sleeve assembly. The repair sleeve is installed on an existing pipeline girth site where Inline Inspection examination indicated a bulge in the pipeline across the wild, see Figure 1. The repair sleeve shall reinforce the main pipeline and shall contain the pressure release in the event of main line rupture at the bulge location. The design of this pressure containing vessel is neither covered by the ASME BPVC design by rule nor by applicable piping design codes, e.g., ASME B31 and CSA Z662. Therefore, ASME BPV Section VIII Div. 2 Part 5, Design by Analysis, criteria is utilized to conduct the assessment herein. Elastic-Plastic Analysis Method is one of the ASME design codes that are applicable to evaluate protection against plastic collapse by determining the plastic load for the pressure containing components. In this method, the allowable loads on the components are established by applying a design factor to the plastic collapse load. The plastic collapse loads are obtained using Finite Element Analysis (FEA) by incorporating an elastic-plastic material model to obtain a solution. The effects of nonlinear geometry (e.g. large deformation and surface contacts) are also considered in the FEA analysis. Elastic-plastic stress analysis provides a more reliable and straightforward assessment of the protection against plastic collapse for the components relative to other ASME Div. 2 design by analysis criteria (e.g. Elastic Stress Analysis Method; Limit-Load Analysis Method) as the actual structural behavior is more closely captured in the simulation. In the following sections, a case study in accordance with the Elastic-Plastic Analysis Method is presented.

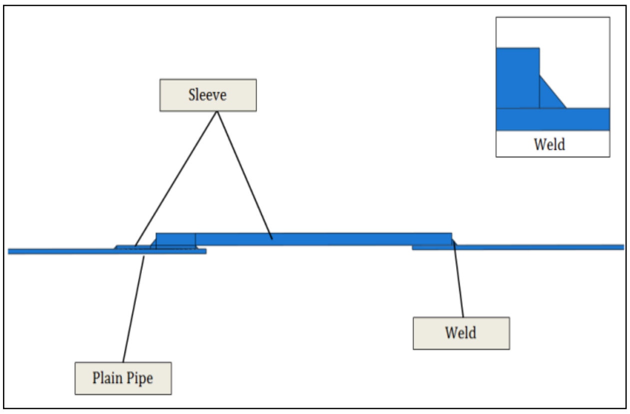

Figure 1: Model – Sleeve Assembly

2. ANALYTICAL METHODOLOGY

In this case study, the sleeve assembly is installed at a bulging section of the buried NPS 42 mainlines to reinforce the pipe at the bulge and contain the pressure in the case of potential rupture. The sleeve assembly model was created as an axisymmetric model in ABAQUS CAE 2018 as shown in Figure 1. The welds were simplified to a full penetration weld and modeled with a sharp notch at the weld toe.

2.1 Material

Material properties used in this analysis are summarized in Table 1. An elastic-plastic material model based on the Ramsberg-Osgood equation and the true-stress true-strain relation is assigned to the sleeve assembly. The material model behaves perfectly-plastic beyond the ultimate tensile limit as illustrated in Figure 2. Welds are assumed to have equivalent material properties to the base material. All materials are homogenous and isotropic.

Table 1: Material Physical Properties

Material	Density (kg/m³)	Linear Coefficient of Thermal Expansion (C^{-1})	Elastic Modulus (GPa)	Poisson Ratio
Carbon Steel	7833	12×10^{-6}	207	0.3

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 12, Number 3 (2019), pp. 333-336

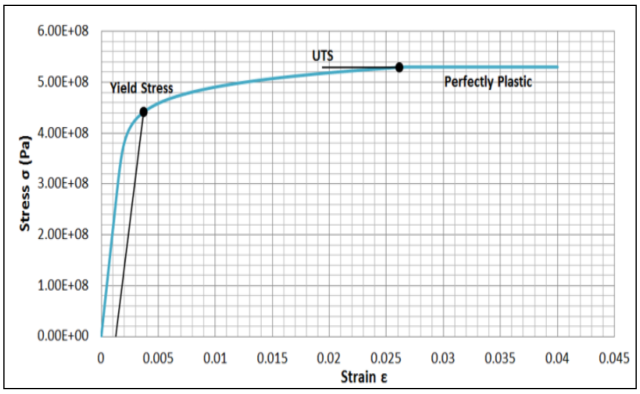

Figure 2: Elastic-Plastic Curve – CSA Z245.1 Grade 448

2.2. Contacts

Two types of surface contacts are included in this analysis as illustrated in Figure 3. 1) Hard contact, where slide and separation between surfaces are allowed. The friction coefficient of 0.4 is assigned in the tangential direction associated with the hard contact surfaces. 2) Bonded contact, where welds restrain any relative movement between associated surfaces is used at the interactions of the welds to the sleeves and/or to the plain pipe. All welded joints are assumed to be perfectly welded and used full penetration welds.

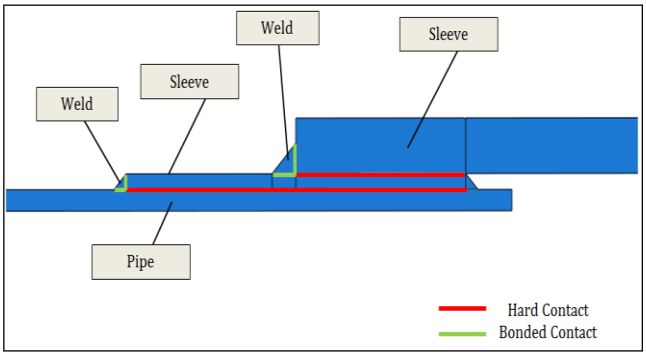

Figure 3: Contacts – Sleeve Assembly

2.3 Preloaded Conditions

The sleeve assemblies are installed while the pipeline is in operation, i.e., pressurized; thus, the installation conditions for the assemblies are the maximum operating conditions. At the installation conditions, the bulged section is under longitudinal compressive stress as it is located at the fully restrained section of the mainline. The calculated pipe longitudinal and hoop stresses are -226 MPa and 490 MPa, respectively. In the initial state, the sleeve assembly are attached to the existing preloaded pipe, as illustrated in Figure 4. The preloading procedure is accomplished by importing the deformed piping results and its associated stress state to the assembly model.

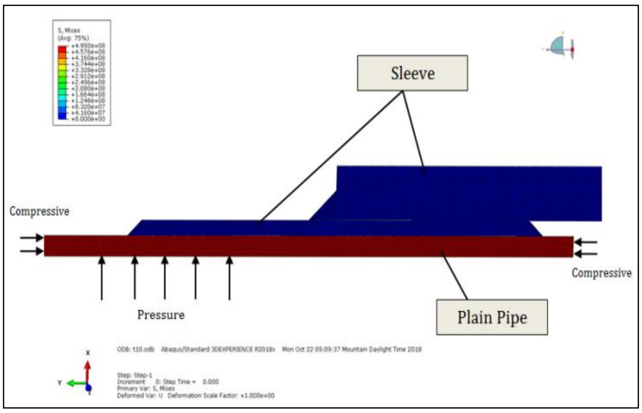

Figure 4: Initial States – Preloaded Plain Pipe (Von Mises: Pa)

2.4 Loading and Combinations

The load case combinations are created in this analysis following the guideline of ASME BPVC Section VIII Div. 2 Part 5 as summarized in Table 2. Since high number of cyclic loading cycles is not anticipated, this analysis is conducted in a static, steady-state condition where no cyclic loading is presented. The loading sequence for this nonlinear analysis is carried out in the following steps, representing the realistic unloading conditions on the assemblies:

At the operation:

1) The initial state, the sleeve assemblies attached to the preloaded plain pipe
2) The internal pressure applied to the sleeves, imitating a rupture in the main line and pressure release in the sleeve assembly; the sleeve assembly is the pressure containing vessel, and
3) The plain pipe fixed to the initial states, i.e., the main pipe is preloaded to its operating conditions hoop and longitudinal stresses.

At the shutdown:

1) The internal pressure deactivated on the sleeves,
2) The plain pipe released from the initial states, i.e., the main pipe operating condition's hoop, and longitudinal stresses are reversed, and
3) Thermal differentials induced, from 49°C to -5°C.

2.5 ASME Section VIII Div. 2 Criterions

The acceptability of protection against plastic collapse is to perform the elastic-plastic analysis for the load case combination listed in Table 2. If the convergence is achieved then structure is stable under the applied loads. Otherwise, the applied load should be reduced and the analysis should be repeated until convergence is achieved.

Table 2: Elastic Plastic Analysis Load Combinations

Criteria	Design Combinations
Local	1.7 x (P + D)

3. RESULTS AND DISCUSSION

At the operation, Figure 5 shows the resulting von Mises stress distributions in the model. It can be seen that the maximum von Mises stress was 502 MPa and located at the weld toe. At the shutdown, Figure 6 shows the resulting von Mises stress distributions in the model. The maximum von Mises stress is 496 MPa and occurs at the weld root location.

Stress Classification Lines are established at these locations to further investigate the stress components as summarized in Table 3. At the operation, the high membrane stress is found due to the compressive stress and hoop stress that is preloaded to the plain pipe. At the shutdown, the depressurization effecting on the adjacent pipe generates a high local bending stress and peak stress around the weld root. Peak stress exists in both load cases and caused by the geometric discontinuities at the welds.

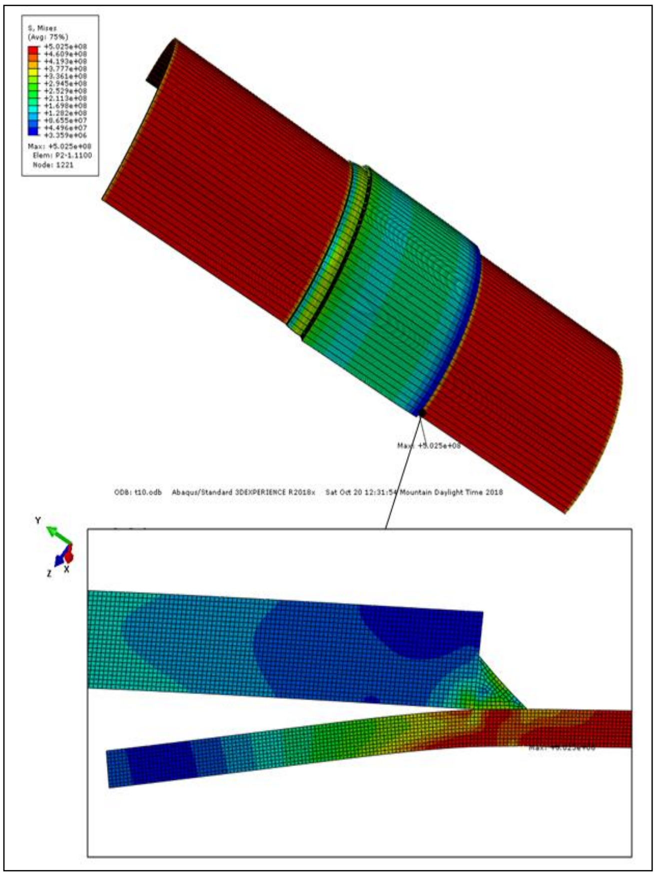

Figure 5: Von Mises Stress Plot – Operation (Pa)

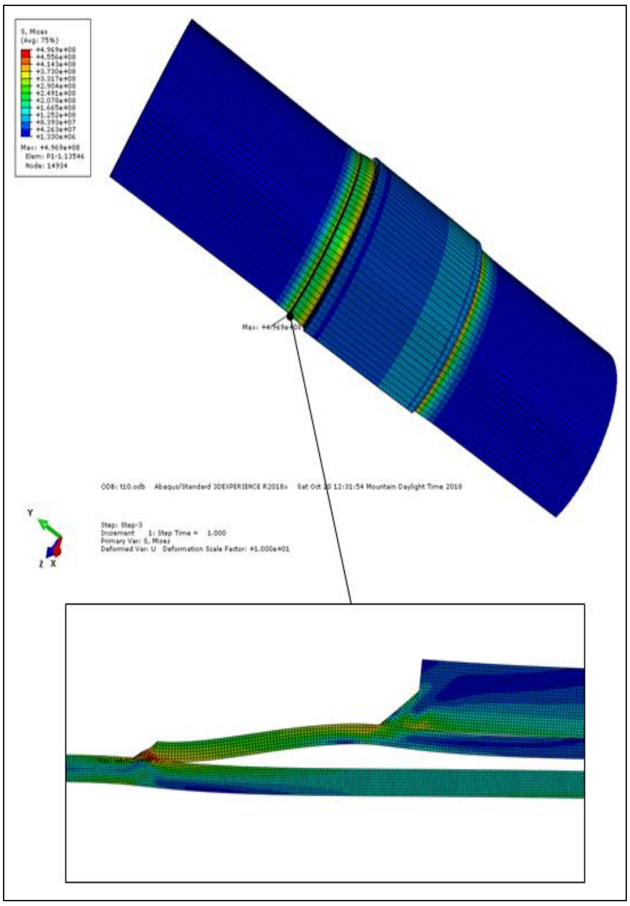

Figure 6: Von Mises Stress Plot- - Shutdown (Pa)

Table 3: Linearization VM Stress Results (MPa)

Case #	Membrane	Membrane Plus Bending (Outer Surface)	Membrane Plus Bending (Inner Surface)	Peak (Outer Surface)	Peak (Outer Surface)
Operation	438	445	475	101	107
Shutdown	91	265	88	350	153

4. CONCLUSION

This article presents a case study of the sleeve assembly design and validation that are conducted in accordance with ASME Section VIII Div. 2 Part 5.2.4. The convergence is achieved by subjecting the assemblies to the load case listed in Table 2. The assemblies appear to be stable under the applied loads. The assemblies are shown that the maximum von Mises stresses are 502 MPa and 496 MPa at the operation and the shutdown, respectively. The maximum von Mises is found at the location of the weld toes and/or roots due to geometric discontinuities.

Nomenclature

P = Internal Pressure

D = Dead Weight

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 12, Number 3 (2019), pp. 333-336

REFERENCE

[1] American Lifelines Alliance, Guidelines for the Design of Buried Steel Pipe, 2005

[2] CSA Z662-15 Oil and Gas Pipeline Systems, 2015

[3] CSA Z245.1 -14 Steel Pipe, 2014

[4] ASME, Boiler and Pressure Vessel Code Section II Material, Part D, 2015

[5] ASME, Boiler and Pressure Vessel Code Section VIII Division 2, 2015 Edition, New York, The American Society of Mechanical Engineers

[6] Soil Mechanics For Pipeline Stress Analysis, James M.Oswell, 2016

[7] Applied Soil Mechanics with Abaqus Applications, S. Helwany, Hoboken, 2007