Modelling of TR-XTREME™ ductile iron pipe under direct tension force

Qinglai Zhang[1], Shih-Hung Chiu[2], Tianyu Han[3], Kenichi Soga[4] and Zili Li[5]
[1,5]Civil, Structural and Environmental Engineering, School of Engineering, University College Cork, Ireland
[2,3,4]Civil and Environmental Engineering, University of California, Berkeley
E-mail: 120224300@umail.ucc.ie (corresponding author)

Keywords: Ductile iron pipe, Water pipe, Tension test, Finite-element analysis

1. INTRODUCTION

Pipelines used for water distribution play an important role for civil infrastructure resilience and sustainable living. However, in many countries, deformation and destruction of water pipelines caused by earthquakes have been widely observed. In particular, the performance of the pipelines under earthquake-induced deformation is largely influenced by axial tension load capacity. This paper focuses on the mechanical behavior of TR-XTREMETM ductile iron pipe manufactured by U.S. Pipe under the direct tension force.

In the past, some studies have already determined the characteristics of different water pipelines under the axial tension force conducted by Cornell University. The pullout resistance and failure mechanism of lined DI pipes were tested to consider varying composite interfaces and discontinuities (NEESR group, 2011). As a further study, an earthquake-resistant ductile iron pipe (ERDIP) developed by Kubota was used to test the capacity of the pipe joint in direct tension (C. Pariya-Ekkasut, 2016). More recently, Cornell University evaluated the ability of the Steel Pipe Crossing Faults (SPF) to accommodate axial tension (B.A. Berger, 2018). Nevertheless, in their laboratory tests, the critical experimental data, such as the hoop strains and longitudinal strains, was seldom compared against computed results from finite element model (FEM). While in order to examine the obtained data and also better understand the pipeline behavior in response to the pure axial force, a 3D continuum FE analysis is highly desired.

Therefore, the finite element model is validated by using experimental results obtained from fiber optic sensors at UC Berkeley, which can be used as the reference model for future related pipeline research, including but not limited to parametric study and experimental tests.

1.1 Overview of numerical model

ABAQUS, a commercial finite element modeling software, has been used for the performance of the pipe axial tension tests. The model geometry of the pipe and joint are same as the experimental setup. Three main parts including spigot, bell pipe and three pieces of locking segments are considered in the model as shown in the

Figure 1. FE meshes used for the analysis are shown in

Figure 2. The isotropic 3D solid continuum element (C3D8R) are used to model the pipe and joint. The number of elements and nodes in the finite element mesh are 124,779 and 157,117 respectively.

The loads and boundary conditions are briefly summarized: At the joint area, the surfaces of the pipe and the locking segments are set to be contacted and allowed to slip to each other. The normal behavior of the interaction is set as the hard contact in ABAQUS and the friction coefficient of the tangential behaviour is 0.8. One end of the pipe at the fixed box is constrained in all three directions while the other end of the pipe is allowed to move horizontally with 5 inches displacement. Also, with 50 psi water pressure on the inside surface of the pipeline.

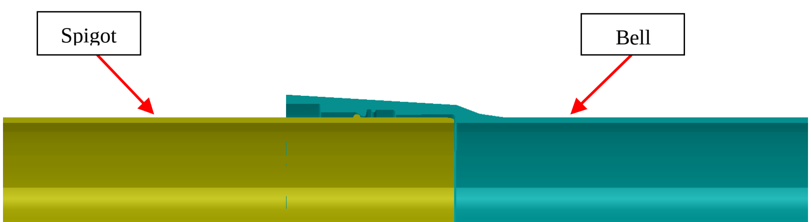


Figure 1.Three main parts in FE model

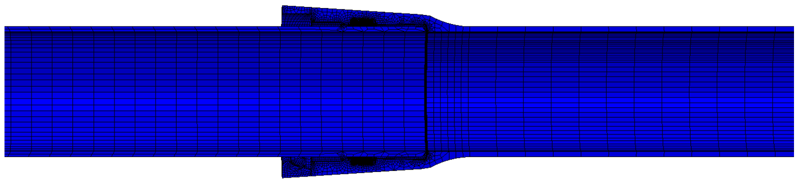


Figure 2.3D FE model mesh for axial tension test

The modeling begins with the spigot fully inserted into the bell. There are three predicted movement stages. First, since the pipe were fully inserted in the bell, a 2.9 inches stroke was provided. Tiny strain can be observed before the weld bead contacts the locking segments. Second, in the FE model, the weld bead and locking segments contact each other when the displacement reaches 2.9 inches, and then the bell side of the pipe at the connection section expands while the 3 pieces of locking segments bear on the weld bead. Lastly, the locking segments slip out of the pipe after the peak displacement was reached and ends up with water leakage.

1.2 Determination of pipe parameters

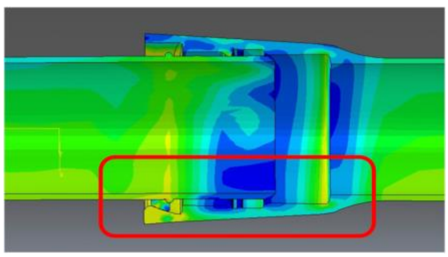
Table 1 presents the material properties of the ductile iron pipe and locking segments used in the tension test. The plastic properties are included in the simulation to accommodate some parts of the pipe reaching the yielding stress of the material, resulting in plastic deformation status.

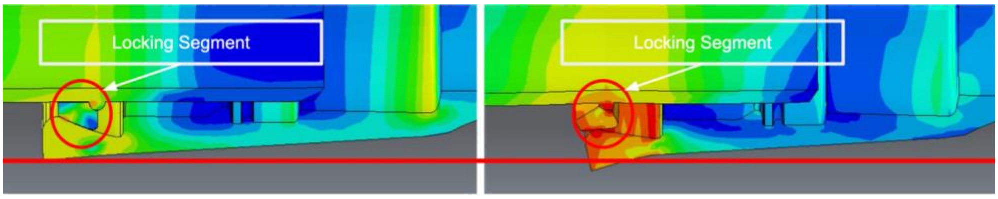
Table 1 Ductile Iron Pipe properties

Part	Density (ib/in3)	Young's Modulus (psi)	Poisson's Ratio	Yield Strength (psi)	Ultimate Strength (psi)	Elongation
Ductile Iron Pipe (plastic)	0.28	23,500,000	0.29	42,000	60,000	10%
Locking segments (plastic)	0.3	24,000,000	0.26	42,000	60,000	10%

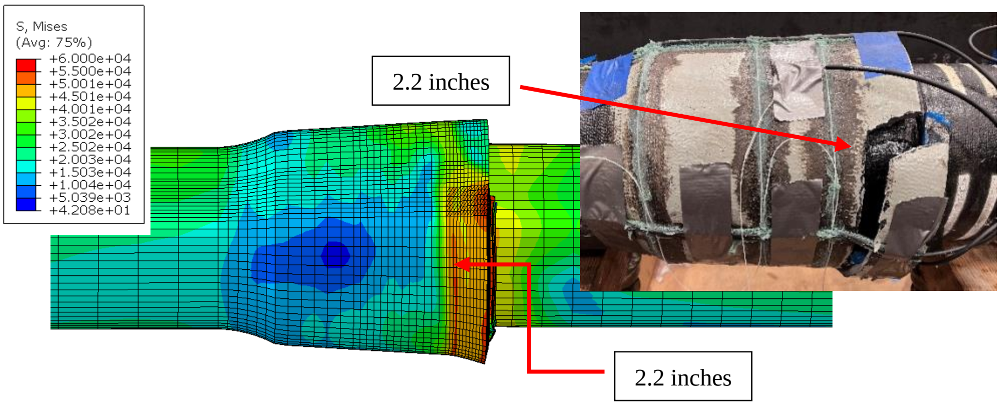
2. FEM RESULTS

2.1 Bell failure mechanism





(a)



(b)

Figure 3. Damage area prediction. (a) bell sketch (b) comparison of damage location

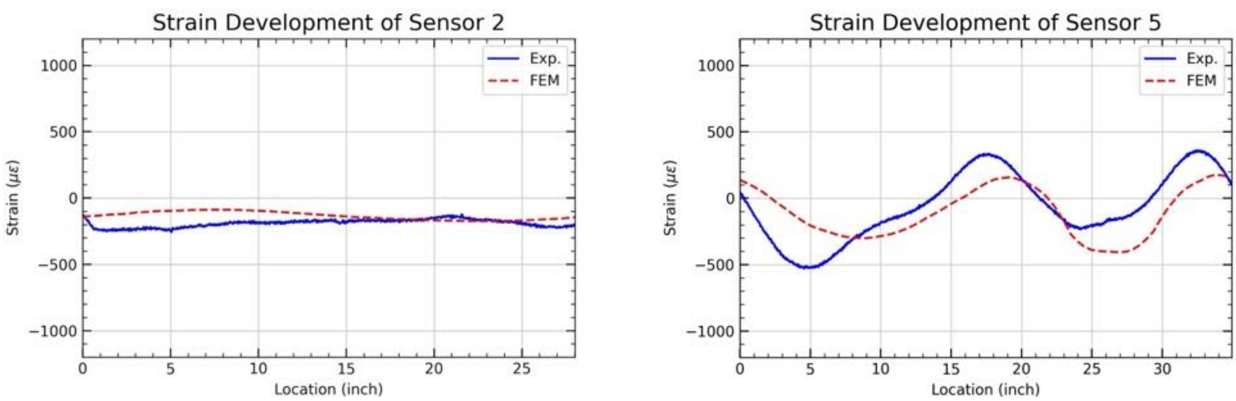
(b)

Figure 3 shows the FEM results. The maximum Mises stress happens at the areas within 2.2 inches from the edge of the bell, matching the result obtained from lab test. Theoretically, the damage to the bell should be symmetric, as shown in Figure 3. However, the experimental result shows that the location of the crack starts at 45 degrees from the top and extends all the way through the bottom. However, no crack is observed on the east side of the pipe. This might be the reason that the material properties of the bell are not uniform.

2.2 Comparison of the FE model and experimental data

In this study, numerical simulation is conducted to investigate the response of the pipeline subjected to the pulling conditions. The comparison strain results from FE simulation and experiment are plot at 170-kips loading, which closed to the maximum tension capacity of the pipe. Figure 4 and Figure 5 show some of the circumferential and longitudinal strains from FEM, which can match well with the corresponding lab test results. It can be seen that comparing to the experimental result, FE analysis shows a flatter trend when the locations are away from the bell. On the other hand, while viewing the locations close to the bell, the strains vary.

Circumferential strains at 170-kips loading



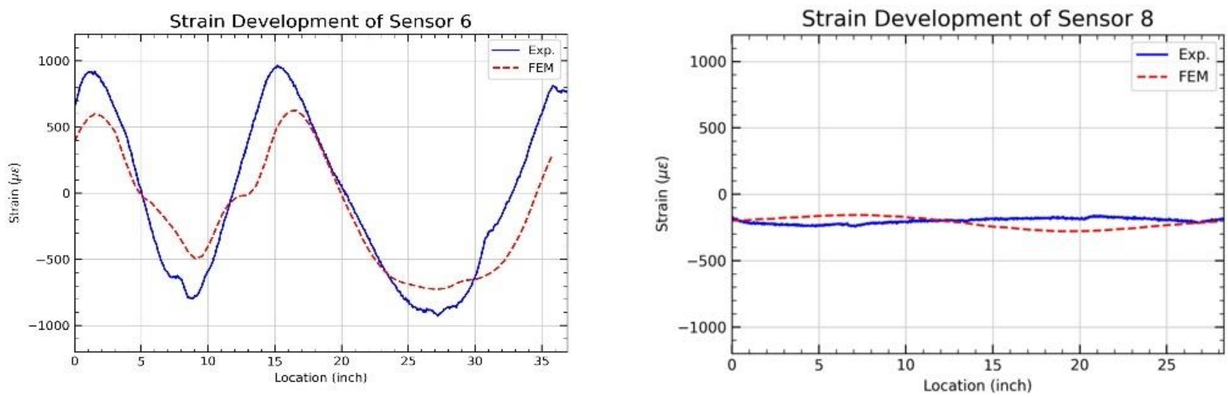


Figure 4.Comparison of circumferential strain results from FE simulation and experiment

Longitudinal strain at 170-kips loading

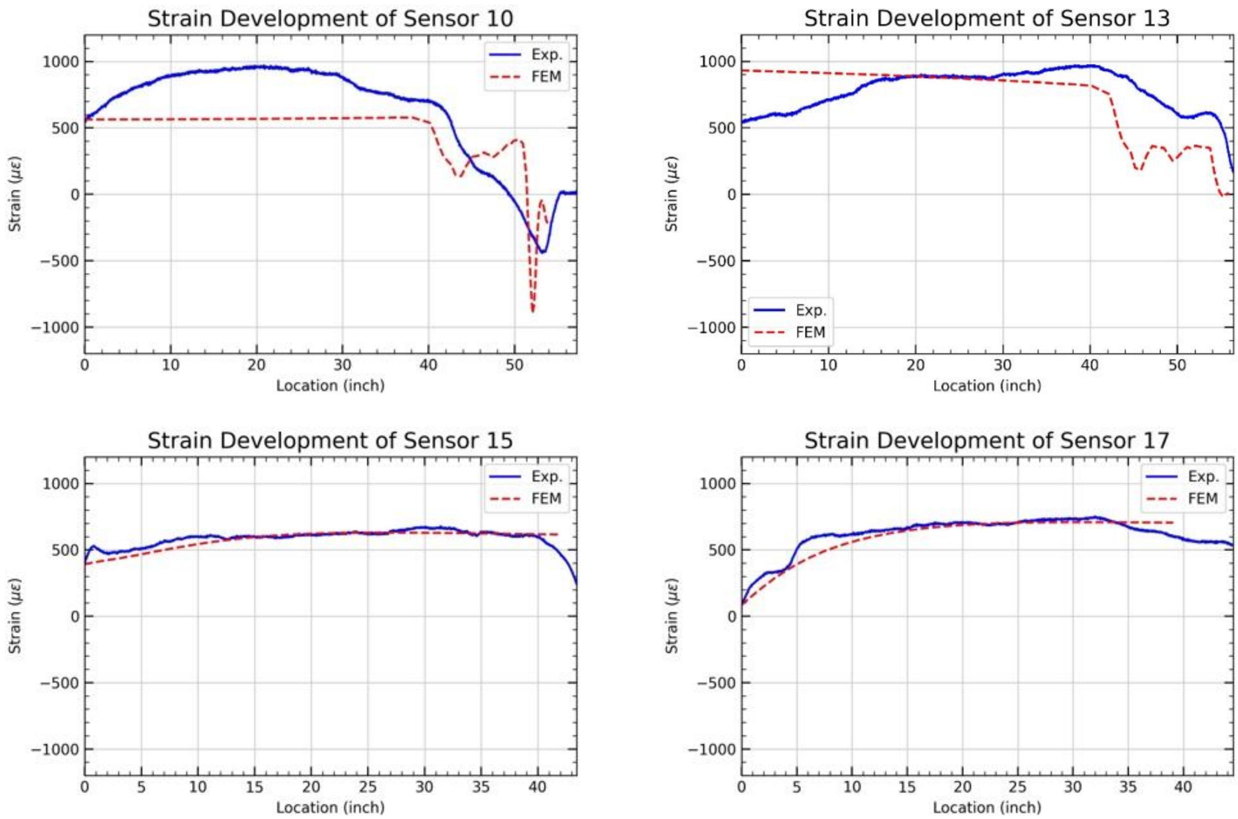


Figure 5.Comparison of longitudinal strain results from FE simulation and experiment

3 CONCLUSION

The experimental results were compared to the finite element model. The strain distribution and patterns match well between the experiments and the simulation, indicating that the finite element model possesses the ability to predict the behavior of the bell-spigot joint under tensile forces. The proposed model can be used in future parametric studies and as a reference for following lab tests (biaxial tension test, four-point bending test, split basin test, etc.) which will be conducted soon.

4 REFERENCE

B.A. Berger, B. T. (2018). Direct Tension and Cyclic Testing of JFE SPF Wave Feature.

C. Pariya-Ekkasut, H. E. (2016). Direct Tension and Split Basin Testing of 6-in. (150-mm)-Diameter Kubota Earthquake Resistant Ductile Iron Pipe.

NEESR group, C. U. (2011). Axial Pull Tests on Insituform IMain Liner.