CAAP Annual Report

Date of Report: October 20, 2020

Prepared for: U.S. DOT Pipeline and Hazardous Materials Safety Administration

Contract Number: 693JK31950001CAAP

Project Title: Improved NDT Detection and Probabilistic Failure Prediction for Interacting Pipeline Anomalies

Prepared by: Sijun Niu, William Back, Shruti Trivedi, and Dr. Vikas Srivastava

Contact Information:
Vikas Srivastava
Assistant Professor of Engineering
Brown University
184 Hope Street, Box D, Providence, RI 02912
Email: vikas_srivastava@brown.edu
Phone: 401-863-2863

For quarterly period ending: Sept 30st, 2020

Business and Activity Section

(a) Contract Activity

No modifications were made to the contract.

(b) Status Update of Past Quarter Activities

We have made progress towards successfully stimulating ultrasound testing in three dimensional geometries after showing the applicability of our new crack characterization methodology for a two dimensional plane strain scenario. The details of 2D finite element and machine learning studies and results have been reported in Q1-Q3 reports [1], which we briefly summarize here in addition to reporting the key highlights from the Q4 research.

Key activities and accomplishments in the 1st quarter (Q1):

- A method for simulating ultrasound propagation in a 2D plate geometry with cracks using finite element software Abaqus was developed.

Key activities and accomplishments in the 2nd quarter (Q2):

- A complete methodology for characterizing embedded elliptical cracks in 2D plate using neural network was created, including constructing large preliminary simulation-based database, signal preprocessing, feature extraction and neural network design.

- Trained neural network was tested against test data. High accuracy prediction of **one crack parameter**, either crack size, location or orientation, was achieved with our neural network.

Key activities and accomplishments in the 3rd quarter (Q3):

- Databases with **multiple crack parameters varying** were created using finite element simulations. High accuracy prediction of multiple parameters simultaneously was achieved, for crack size-depth and crack size-orientation pairs.

- Current deterministic models for predicting pipeline burst pressure were identified. ASTM boiler code was selected and studied by applying normal distribution for input variables. Results show probabilistic assessment of failure loads.

Key activities and accomplishment in the 4th quarter (Q4) (recent most quarter):

Finite element simulations for 3D embedded crack in a 3D geometry was successful and currently we are optimizing computational time to make the simulation and hence the methodology computationally more efficient. We have also fabricated test samples for fully embedded cracks of different sizes and orientation for validation experiments. Early experiments of UT testing have started. In parallel, we have implemented the Gurson-Tvergaard-Needleman (GTN) model in Abaqus with an axisymmetric pipeline geometry with a wall-loss type anomaly. Preliminary results demonstrate the failure behavior using GTN model in FEA. We have also been building a framework for probabilistic failure pressure estimation using the deterministic ASME boiler code.

Cost share activity

Partial support for 1 Ph.D. graduate student tuition was provided by Brown University School of Engineering as per the cost share agreement.

Note: *Detailed reports for the first 3 quarter activities have been provided in the Q1 (December 2019), Q2(March 2020) and Q3(June 2020) reports. [1] This annual report focuses on the new quarter 4 activities to minimize repetition and acknowledges the fact that the new results are built on our earlier quarter research findings.*

1. Background and Objectives

1.1 Background

Application of popular ultrasonic non-destructive testing (NDT) technique remains challenging for crack characterization in pipelines as data interpretation is performed by people, which results in significant uncertainty in accurate crack feature predictions.[1][2] We are conducting research to apply finite element modeling and machine learning based solution for crack detection and characterization. [2][3] The geometric parameters are of great importance since they determine the lifespan and failure conditions of pipelines. Once we know the anomaly characteristics, quantitatively predicting failure using appropriate Fracture mechanics modeling is both complicated and stochastic in nature. Cylindrical pressurized vessels such as pipelines are prone to burst if cracks are present in the body. We want to apply more physics based approaches such as GTN model to predict fracture mechanics based failure. Many phenomenological models exist that try to predict the burst pressure of pipes using parameter-based criteria. However, these models although suitable under certain applications are limited to be deterministic and rely heavily on the accuracy of inputs.[4] When it comes to the pipelines transporting dangerous materials, there is no room for error. Thus, in addition to more physical model like GTN based approach, it is important that a probabilistic approach is developed for existing and new models which allows the models to take into account variability of input conditions and provide risk based failure predictions.

1.2 Objectives in the 4th (recent most) Quarter

During the previous quarter, we successfully trained a fully functional NN that is capable of predicting multiple geometric parameters of cracks simultaneously with high confidence and identified selected useful deterministic burst pressure models and modified them into an early probabilistic framework to evaluate the outcome. In this quarter, we had the following objectives:

- Transition from plane strain simulation to full three-dimensional simulation which represents the real-world situation
- Implementation of GTN model in Abaqus and perform failure analysis
- Study and develop probabilistic framework

2. Experimental and Computational Program for the 4th (recent most) Quarter

2.1 Experimental design

Five test samples with fully embedded 3D penny-shaped cracks were fabricated in this quarter for the purpose of experimental validation studies.

2.2 Computational setup

All computations were conducted on an existing workstation desktop (early computations are relatively smaller sized).

We studied finite element based numerical simulation requirements for sound wave propagation in steel pipelines. All of our numerical study used an ultrasound wave of 5 MHz frequency and wavelength of ~1.2 mm, the numerically stability requirement we obtained that 10-15 meshes per wavelength provides a stable practical element size.

A 3D steel flat pipe section geometry with length and width both being 40 mm and thickness 19 mm (~3/4 inch) was used in our new simulations. A 5 MHz raised-cosine type waveform was applied as boundary condition to the top surface of the simulated pipe. Profile for this waveform is shown in **Figure 1**. Step size is fixed at $\mathbf{2 \times 10^{-9}}\ s$ which corresponds to a 500 MHz sampling rate. Artificial anomalies in the form of embedded penny-shaped cracks are placed in the middle of the pipe. We conducted dynamic numerical simulations in Abaqus/Explicit and analyzed the displacement history profile at the selected point receiver locations on the plate surface.

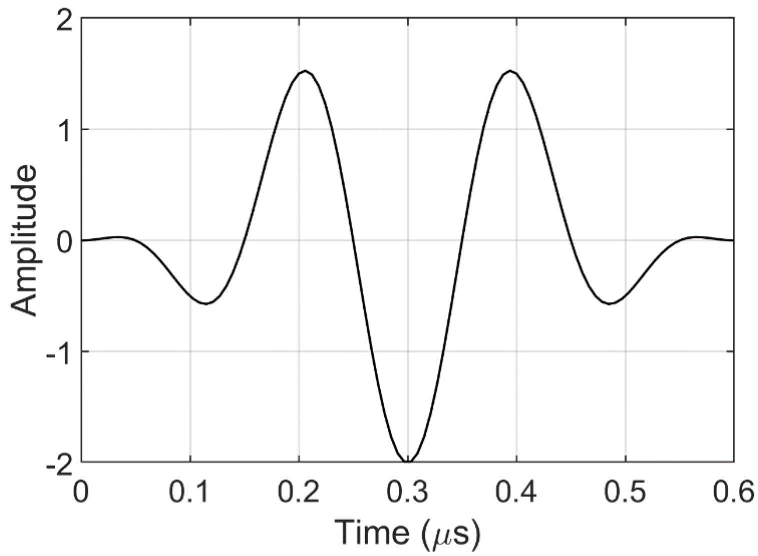

Figure 1. 5MHz, 3 period raised-cosine type pulse signal used in the simulations.

3. Results and discussion

In this annual report, we here summarize the important results for year 1 of the research project.

Key results in the 1st quarter (Q1):

- A method to use finite element simulation to generate ultrasonic signal is proposed and established. Crack echo and back-wall echo can both be seen in the simulated signal as shown in Figure 2.
-

Key results in the 2nd quarter (Q2):

- Important crack parameters (size, location and orientation) and features selection technique (wavelet packet transform) are identified.
- Ultrasonic signal databases with single crack parameter as variable are created through simulations for the 2D case.

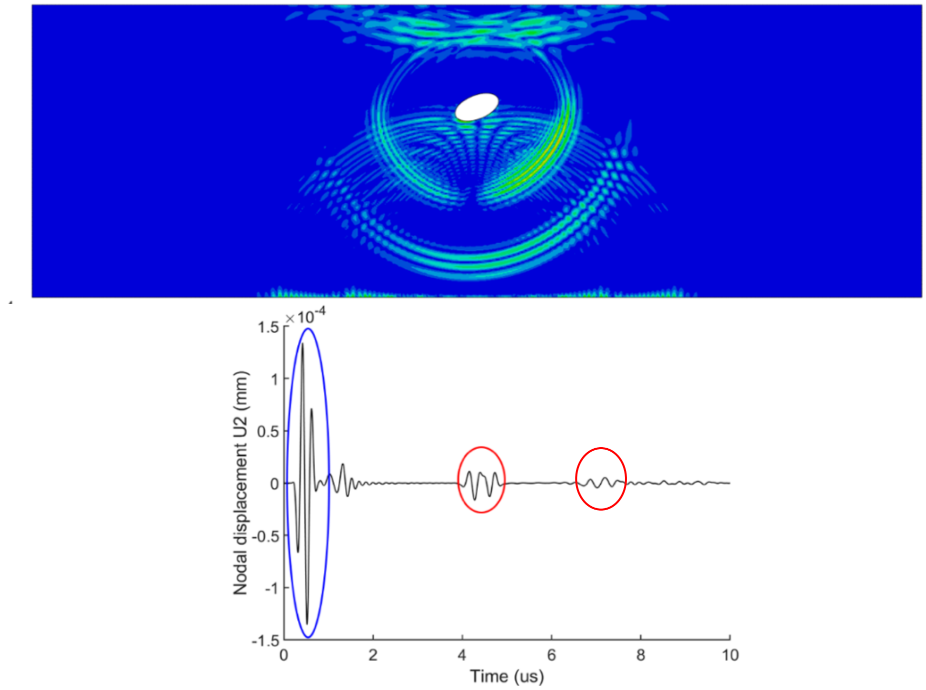

Figure 2. Top: displacement contour plot of ultrasound inside a steel plate; bottom: simulated ultrasonic signal with identified initial pulse (blue) and crack and back-wall echoes (red)

Key results in the 3rd quarter (Q3):

- Large ultrasonic signal databases with combined crack parameters as variables are created through simulations for the 2D case (Table 1).
- Neural network is used with the databases to predict single and combined crack parameters. High accuracy is achieved for both single and combined features for the 2D simulation cases as shown in Figures 3 and 4.

Table 1. Summary of datasets with different parameters, bold numbers indicate a range in which the corresponding parameter varies

	$\boldsymbol{x}$ **(mm)**	$\boldsymbol{y}$ **(mm)**	$\boldsymbol{a}$ **(mm)**	$\boldsymbol{b}$ **(mm)**	θ	**Number of simulations**
Dataset 1	0	15	1	**[1, 3]**	0	479
Dataset 2	0	15	0.5	1.5	**[0,** $\boldsymbol{\pi}$**]**	957
Dataset 3	0	15	1	**[1, 3]**	$\pi/4$	446
Dataset 4	0	**[7, 16]**	1	**[1, 3]**	$\pi/4$	1878
Dataset 5	0	15	1	**[1, 3]**	**[0,** $\boldsymbol{\pi}$**]**	3728

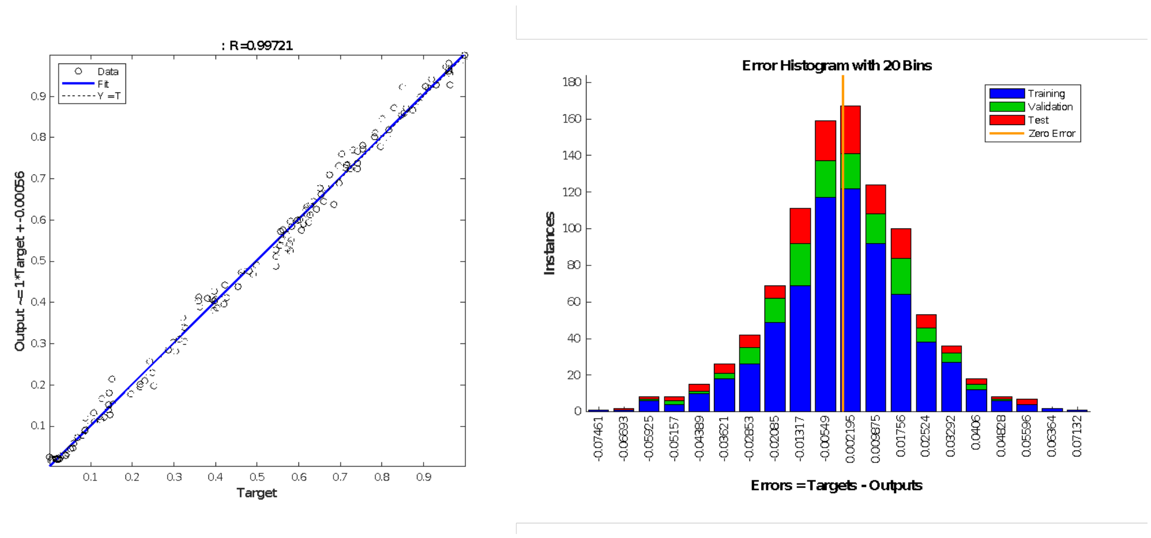

Figure 3. MATLAB NN regression plot (left) and error histogram (right) for predicting single crack parameter – orientation

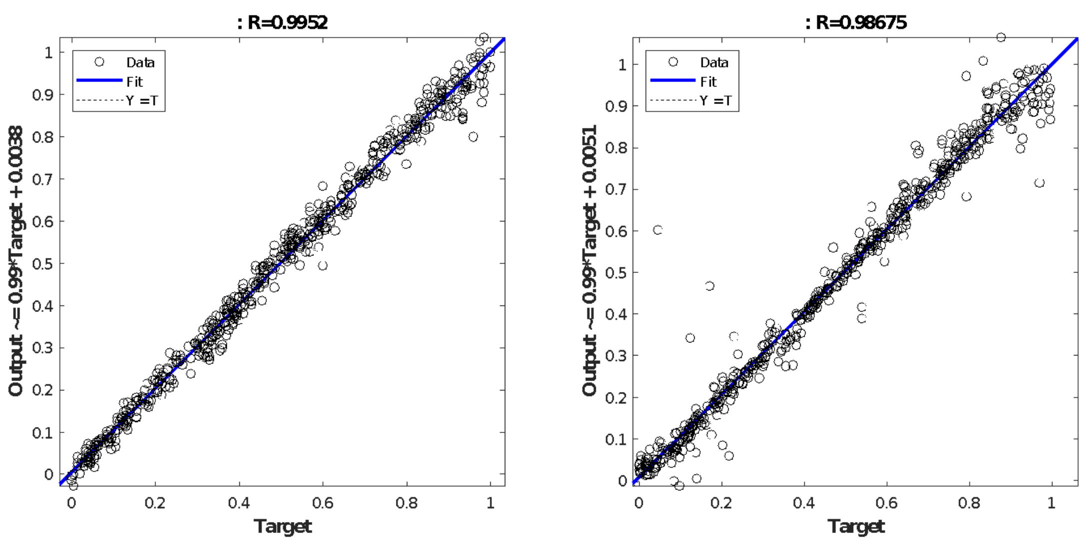

Figure 4. MATLAB NN regression plot for predicting combined crack parameters - long axis (left) and orientation (right) simultaneously

As described in the object section, we have identified three important objectives in this quarter, and the results are presented and discussed below.

3.1 Technical approach and result for the 4th quarter (Q4) in the research project year 1

We have been using two dimensional plane strain simulations so far to create databases for our neural network and the results have been very promising. However, three dimensional geometry of pipes is a more realistic scenario which we aim to progress in our modeling.

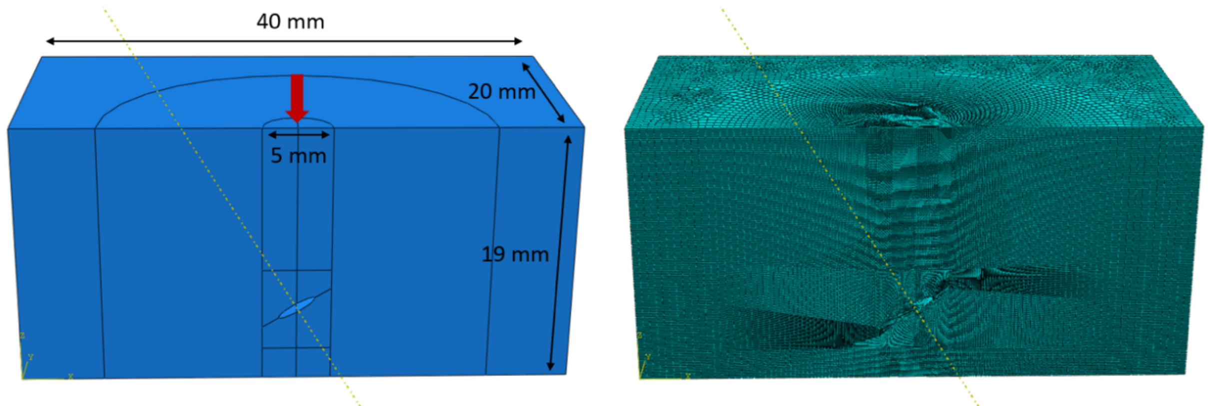

Figure 5. Left: geometry and partition of a 3D flat pipe section that is being modeled; Right: finite element mesh of the pipe section.

Taking advantage of the symmetry, we simulated half of a pipe section with one embedded penny-shaped crack located in the center, as illustrated in **Figure 5**. Finer mesh is used near the crack to ensure the accuracy of the simulation and coarser mesh is used in regions far from the crack to reduce the simulation time. The region where the ultrasonic pulse initiates and is received is partitioned to be a semi-circle on the top surface, which is indicated by the red arrow. A sample A-scan created by the simulation is shown in **Figure 6**. It is very consistent with our earlier 2D simulation studies, displaying very clear peaks for the crack and the back wall. This confirms the viability of transitioning from 2D to 3D geometry. A 3D finite element simulation with fine mesh is computationally very expensive. We have made good progress in optimizing the finite element model for faster computations with reasonable success.

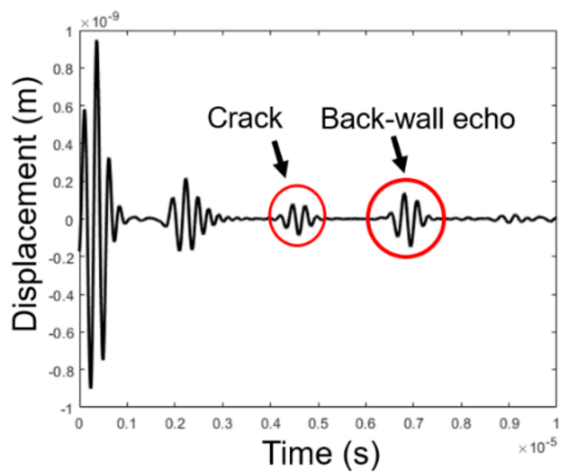

Figure 6. Results from a representative 3D finite element simulation showing ultrasound signals expected from A-scan signal

Five ultrasonic test samples were made during the past quarter. The test samples have three dimensional embedded cracks of different sizes and orientation. Four of five samples are identical to the simulation geometry with each containing one penny shaped crack in the middle. One is slightly different, with the length to be 60 mm instead of 40 mm and has two cracks separated by 20 mm. The samples are shown in **Figure 7**, and the crack parameter information is summarized in **Table 2**. As a reminder, *a* is the minor axis, *b* is the major axis, *d* is the depth of the crack and $\boldsymbol{\theta}$ is the orientation (angle). These samples will be tested in the next quarter as a validation of our simulation-driven neural network.

Figure 7. Test samples with embedded penny-shaped cracks.

Table 2. Designed crack parameters for the test samples

Sample #	***a* (mm)**	***b* (mm)**	***d* (mm)**	$\boldsymbol{\theta}$
1	0.6	4	12	0
2	0.6	3.5	11	30
3	0.6	1.5	9	0
4	0.6	2.5	14	45
5	0.6	3.5	11	0
	0.6	2.5	14	15

Regarding Aim 2 in the proposal, the GTN model in the finite element program Abaqus is defined as follows through a yield function $\boldsymbol{\Phi}$:

$$\boldsymbol{\Phi} = \left(\frac{\sigma_e}{\sigma_0}\right)^2 + 2q_1 f^* \cosh\left(-q_2 \frac{3\sigma_m}{2\sigma_0}\right) - (1 + q_3 f^{*2}) = 0 \quad \textbf{(1)}$$

where $\boldsymbol{\sigma_e}$ is the von-Mises equivalent stress; $\boldsymbol{\sigma_0}$ is the yield strength; $\boldsymbol{\sigma_m}$ is the mean normal stress; and the q's the porous material parameters. We have selected 316L stainless steel as our test material and have found corresponding GTN parameters in literature which are summarized in Table 2 [5].

Table 3. GTN parameters for 316L stainless steel

Porous material parameters			
$\boldsymbol{q_1}$	1.25	$\boldsymbol{q_2}$	1
$\boldsymbol{q_3}$	1.5625		
Void nucleation parameters			
$\boldsymbol{e_N}$	0.3	$\boldsymbol{s_N}$	0.1
$\boldsymbol{f_N}$	0.0005	$\boldsymbol{f_0}$	0.00005
Porous failure criteria			
$\boldsymbol{f_c}$	0.01	$\boldsymbol{f_F}$	0.15

In order to test the embedded GTN model in Abaqus, we have performed a failure analysis simulation on a test geometry. The test geometry is a plane-strain notched plate, with a width of 30 mm and length of 100 mm. The notch is 5 mm deep in the width direction and has a 0.1 mm radius at the tip. The bottom of the plate is fixed with no displacement through the whole simulation, and the top has a constant velocity of 0.5 mm/s. A dynamic, explicit step of total 30 s is simulated. The plate is meshed with CPE4R elements and very fine mesh is used from the notch tip and along the crack extension direction. Failed elements are deleted from the simulation and results are shown in **Figure 8**.

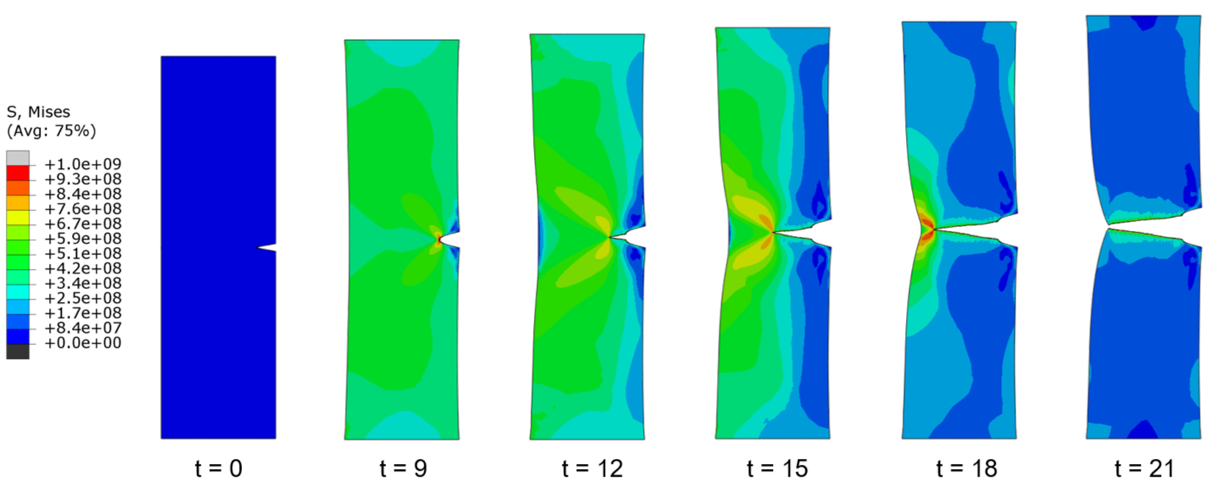

Figure 8. Mises stress and failure progression prediction from finite element simulation using the GTN model for a plate with a notch.

It can be visually seen that the crack tip opens followed by the initiation of fracture until complete failure. To investigate the stress history and the growth of void, one element near the notch tip is selected and plotted in **Figure 9**.

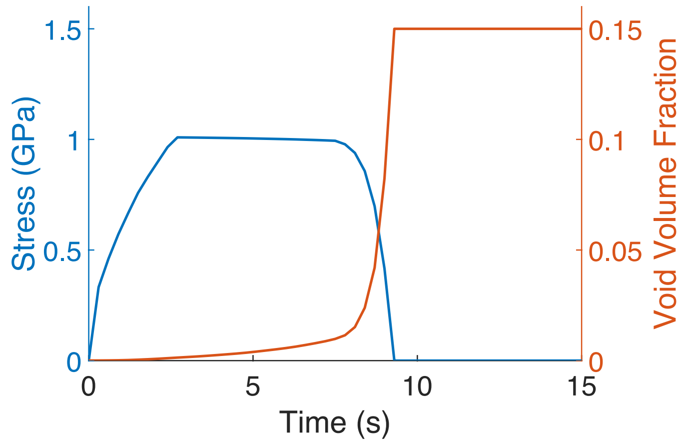

Figure 9. Mises stress and void volume fraction for one element near the notch tip.

The deformation is initially elastic, followed by the plastic strain hardening up to 3 seconds. During the plastic deformation stage, the void starts to grow slowly. Then as the void grows, the stress carrying capability of the element starts to reduce, with the stress slowly decreasing up until the void volume fraction reaches the critical void volume fraction $\boldsymbol{f_c = 0.01}$. Then the void volume fraction grows rapidly to the final $\boldsymbol{f_F = 0.15}$, representing total failure of the element, during which the stress also decreases rapidly to 0. The failure analysis shows the capability of the built-in GTN model in Abaqus that we can use to simulate pipelines with different interactive anomalies. By varying the anomaly geometry and model inputs in a systematic way, we will be able to create a failure database which contains numerous failure loads for different geometries and load conditions.

In this quarter, we continued developing our probabilistic framework. We ran it using the ASME boiler code model [4] for defect-free pipelines as it is a more simple model that allows us to more easily construct the framework at this stage. This model has three input parameters: $\boldsymbol{t}$ is the thickness of the pipe and $\boldsymbol{D_O}$ is the outer diameter of the pipe and $\boldsymbol{\sigma_{UTS}}$ is the ultimate tensile stress. The burst pressure is then given by the deterministic equation:

$$\boldsymbol{P_b = \sigma_{UTS}\left(\frac{2t}{D_O - 0.8t}\right)} \qquad \textbf{(2)}$$

The deterministic method plugs the input data into the model in order to compute the burst pressure. For a mill-expanded X-80 pipe, the deterministic input data is: $\boldsymbol{\sigma_{UTS}}$ = 677 MPa, $\boldsymbol{t}$ = 7 mm, and $\boldsymbol{D_O}$ = 357 mm. The deterministic result for burst pressure is then $\boldsymbol{P_b}$ = 27 MPa. Inserting the deterministic ASME boiler code model into our probabilistic framework we were able to get a normal distribution of burst pressure illustrated in **Figure 10**.

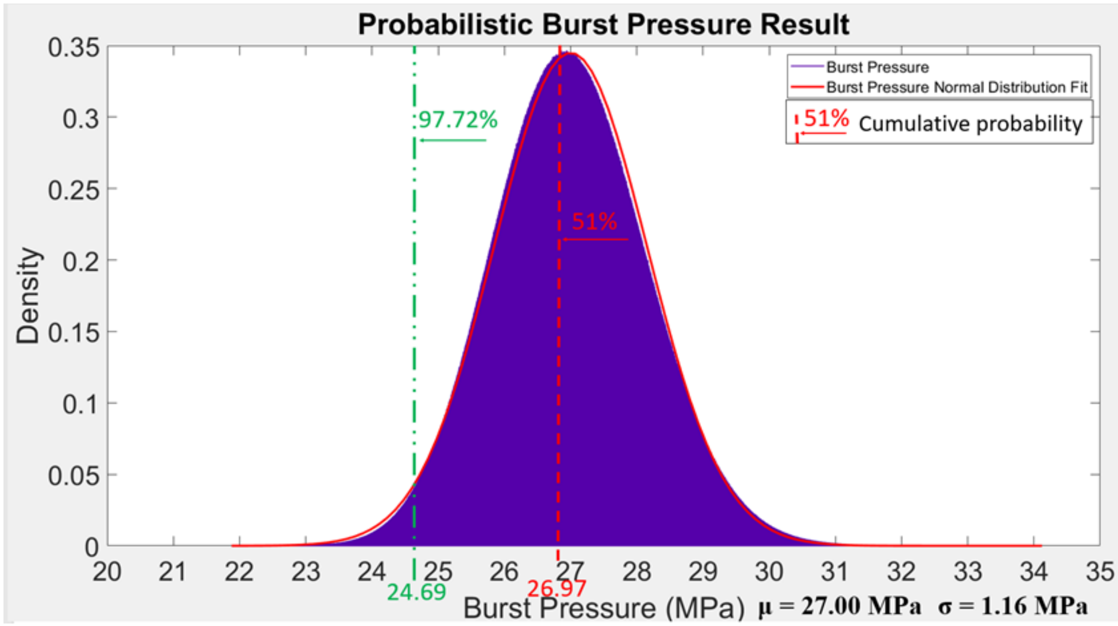

Figure 10: Probabilistic Burst Pressure Result provided by the probabilistic modeling framework. Cumulative probabilities were calculated and plotted.

This probabilistic burst pressure result was able to provide us with much more information than the deterministic result. Instead of providing us a single value for burst pressure without any level of confidence as to how accurate the prediction is, we can determine the cumulative probability of any burst pressure. For instance, from **Figure 10** we can say that the pipe will not burst at or below 24.7 MPa with 97.7% confidence. Depending on the desired level of confidence, the corresponding upper bound for burst pressure can therefore be determined.

3.2 Discussion

In the recent quarter, we successfully accomplished several research tasks described and discussed above. In addition to what is reported here, during the first year of this research, we have been able to successfully demonstrate accurate characterization of embedded cracks considering a 2D geometry. Details for 2D studies are discussed in the previous quarter reports. [1]

4. Future work

Although we are able to simulate ultrasonic A-scan three dimensional pipe with cracks, one difficulty that persists is the computation time required for accurate 3D simulations. To create a sufficiently large database, the overall computation time required could be significantly large. Our immediate next step is to optimize the meshing technique to reduce the simulation time to a manageable amount. Then databases can be created for our neural network training and testing. Also, some of the validation experiments will be carried out in the next quarter. We aim to validate finite element simulation prediction against experimental signals to study the applicability range of our simulation-driven neural network.

We will also apply the probabilistic framework to the GTN model. We will identify suitable values of GTN parameters that represent physical pipeline materials. We also aim to study the role of Bayesian statistics and conditional probability in our future probabilistic framework.

In summary, during the first year of this research project, we have developed a methodology consisting of finite element simulation database trained neural network for predicting features of crack-type anomalies. We have also started to study failure models. Our next goal is to establish a failure model and a probabilistic failure load assessment methodology. We will start with the GTN model as a basis to develop our failure studies, model(s) and probabilistic assessment method(s). *Note,* our plan is to conduct a *comprehensive and accurate methodology development for the single anomaly scenario and dedicate the required significant time on this case first as it will form the very foundation of our future interacting anomaly studies.* Once we have fully established the single anomaly based methodology, we will apply and extend our methodology for other types of anomalies such as corrosion wall loss and for the cases involving more than one anomalies interacting with each other. Corrosion wall loss with embedded crack and a pair of embedded cracks will be two important interacting anomaly cases that we will be focus of our research.

References

[1] U.S. DOT Pipeline and Hazardous Materials Safety Administration CAAP Project839, Contract 693JK31950001CAAP, PI: Vikas Srivastava, Brown University, Quarterly Reports: Dec. 2019, March 2020, June 2020.

[2]. Jiaxing Ye et al., "Computerized Ultrasonic Imaging Inspection: From Shallow to Deep Learning", *Sensors,* 18, 3820, 2018.

[3]. Daniel Bacioiu et al., "Automated defect classification of Aluminium 5083 TIG welding using HDR camera and neural networks", *Journal of Manufacturing Processes,* 45, 603–613, 2019.

[4]. Zhu, Xian-Kui et al., "Evaluation of burst pressure prediction models for line pipes." *International Journal of Pressure Vessels and Piping,* 89, 85-97, 2012.

[5]. J. Chambert, Ph. Bressolette and A. Vergne, "Implementation of coalescence criteria into the GTN model - Application to work-hardening ductile materials.", *Conference: 13th European Conference on Fracture (ECF 13) - Fracture Mechanics: Applications and Challenges, 2000.*