Created in COMSOL Multiphysics 5.6

Stress Corrosion

Introduction

Steel pipelines are often subjected to complex stress/strain conditions in oil and gas industry. In addition to stress from internal pressure, pipelines are subjected to significant longitudinal strain due to surrounding soil movement. The effect of elastic and plastic deformation on pipeline corrosion is demonstrated in this model example.

The elasto-plastic stress simulations are performed here using a small strain plasticity model and von-Mises yielding criterion. Iron dissolution (anodic) and hydrogen evolution (cathodic) are considered as electrochemical reactions, using kinetic expressions that account for the effect of elasto-plastic deformations.

The example is based on a paper by L. Y. Xu and Y. F. Cheng (Ref. 1).

Note: This model requires the Structural Mechanics, or the MEMS Module, with the addition of either the Nonlinear Structural Materials or the Geomechanics Module.

Model Definition

The model geometry is shown in Figure 1.

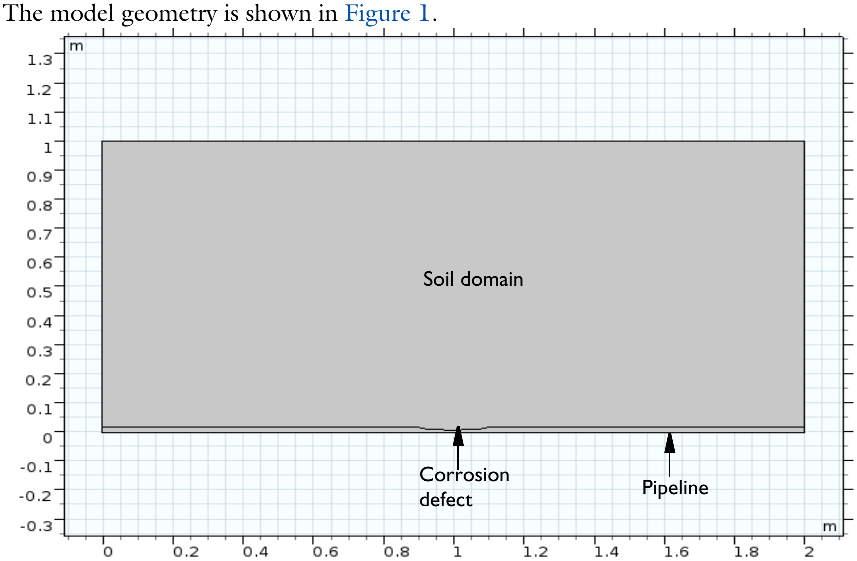

Figure 1: The model geometry consists of a pipeline with corrosion defect and surrounding soil domain.

The model geometry consists of high strength alloy steel pipeline and surrounding soil domain. The pipeline length is 2 m and wall thickness is 19.1 mm. The corrosion defect on the exterior side of the pipeline is elliptical in shape with a length of 200 mm and a depth of 11.46 mm. The electrolyte conductivity of soil domain in 0.096 S/m.

ELASTOPLASTIC STRESS

An elastoplastic stress simulation is performed over pipeline domain using the small strain plasticity model. The user defined isotropic hardening model is used where the hardening function, σ_{yhard}, is defined as:

$$\sigma_{\text{yhard}} = \sigma_{\text{exp}}\left(\varepsilon_{\text{p}} + \frac{\sigma_e}{E}\right) - \sigma_{\text{ys}}$$

where σ_{exp} is the experimental stress-strain curve, ε_{p} is the plastic deformation, σ_e is the von Mises stress, E is the Young's modulus ($207 \cdot 10^9$ Pa), and σ_{ys} is the yield strength of high strength alloy steel($806 \cdot 10^6$ Pa).

The experimental stress-strain curve used in the model is prescribed in terms of a piecewise cubic interpolation function and is taken from Ref. 2.

ELECTROCHEMICAL REACTIONS

The iron dissolution (anodic) and hydrogen evolution (cathodic) reactions are the two electrochemical reactions that occur at the corrosion defect surface of pipelines. The rest of pipeline surfaces are assumed to be electrochemically inactive.

An anodic Tafel expression is used to model the iron dissolution reaction, with a local anodic current density defined as

$$i_{\text{a}} = i_{0,\text{a}} 10^{\frac{\eta_{\text{a}}}{A_{\text{a}}}}$$

where $i_{0,\text{a}}$ is the exchange current density ($2.353 \cdot 10^{-3}$ A/m^2), A_{a} is the Tafel slope (0.118 V) and the overpotential η_{a} for the anodic reaction is calculated from

$$\eta_{\text{a}} = \phi_s - \phi_l - E_{\text{eq,a}}$$

The equilibrium potential for the anodic reaction is calculated from

$$E_{\text{eq,a}} = E_{\text{eq0,a}} - \frac{\Delta P_{\text{m}} V_{\text{m}}}{zF} - \frac{TR}{zF}\ln\left(\frac{\nu\alpha}{N_0}\varepsilon_p + 1\right)$$

where $E_{\text{eq0,a}}$ is the standard equilibrium potential for the anodic reaction (–0.859 V), ΔP_m is the excess pressure to elastic deformation ($2.687 \cdot 10^8$ Pa), V_{m} is the molar volume of steel ($7.13 \cdot 10^{-6}$ m^3/mol), z is the charge number for steel (2), F is the Faraday's constant, T is the absolute temperature (298.15 K), R is the ideal gas constant, ν is an orientation dependent factor (0.45), α is a coefficient ($1.67 \cdot 10^{15}$ m^{-2}) and N_0 is the initial dislocation density ($1 \cdot 10^{12}$ m^{-2}).

A cathodic Tafel expression is used to model the iron dissolution reaction, this sets the local cathodic current density to

$$i_{\text{c}} = i_{0,\text{c}} 10^{\frac{\eta_{\text{c}}}{A_{\text{c}}}}$$

where $i_{0,\text{c}}$ is the exchange current density, A_{c} is the Tafel slope (–0.207 V) and the overpotential η_{c} (SI unit: V) for the cathodic reaction is calculated from

$$\eta_{\text{c}} = \phi_s - \phi_l - E_{\text{eq0,c}}$$

where $E_{\text{eq0,c}}$ is the standard equilibrium potential for the cathodic reaction (–0.644 V)

The exchange current density for the cathodic reaction is calculated from

$$i_{0,\text{c}} = i_{0,\text{c,ref}} 10^{\frac{\sigma_e V_{\text{m}}}{6F(-A_{\text{c}})}}$$

where $i_{0,\text{c,ref}}$ is the reference exchange current density for the cathodic reaction in the absence of external stress/strain ($1.457 \cdot 10^{-2}$ A/m^2).

Results and Discussion

Figure 2 shows the electrolyte potential distribution (V) over the soil domain and the von Mises stress distribution (MPa) over the pipe domain, as indicated by the color bars for a prescribed displacement of 4 mm in the x direction. It can be seen that the local stresses are significantly higher near the corrosion defect than that at the rest of the pipeline. A nonuniform electrolyte potential distribution near the corrosion defect is also evident in Figure 2, as indicated by a semi-circular area.

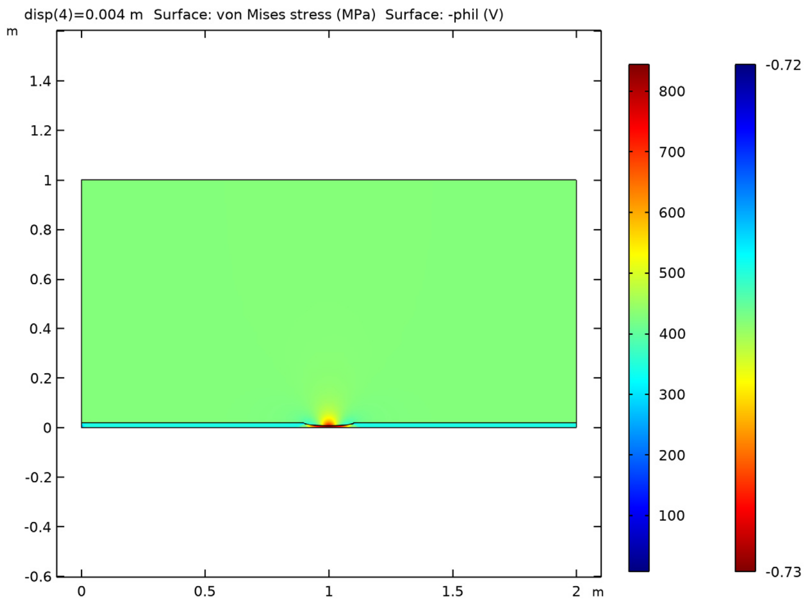

Figure 2: The electrolyte potential distribution over the soil domain and the von Mises stress distribution over the pipeline domain for a prescribed displacement of 4 mm.

Figure 3 shows the von Mises stress distribution along the length of the corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm, respectively. The von Mises stress increases with an increase in the tensile strain and it is found to be maximum at the center of corrosion defect. For the tensile strain of 3.75 mm and 4 mm, it is observed that the local stress, particularly at the center of corrosion defect, exceeds the yield strength of high strength alloy steel ($806 \cdot 10^6$ Pa). This results in the plastic deformation at the center of corrosion defect while deformation in the remaining area of corrosion defect remains in the elastic range. For the lower tensile strains of 1.375 mm, and 2.75 mm, the entire corrosion defect is observed to be in the elastic deformation range.

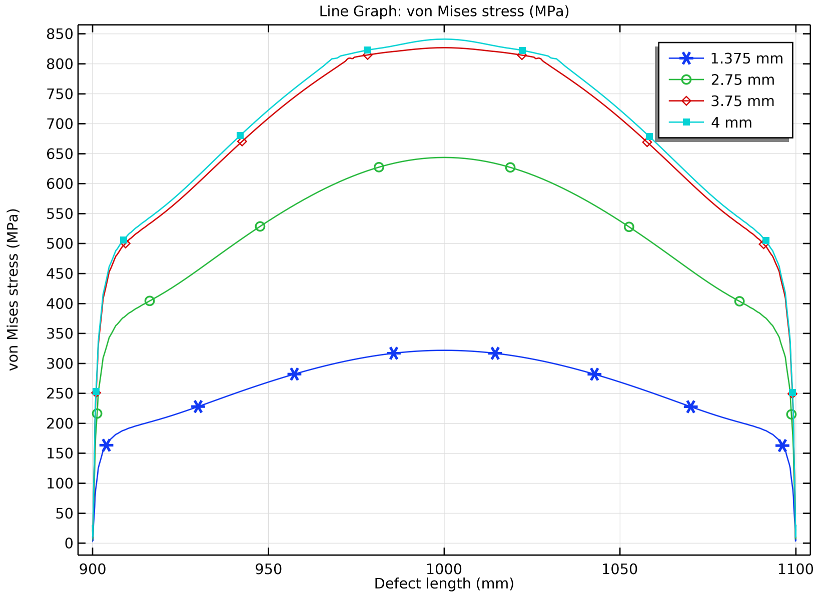

Figure 3: The von Mises stress distribution along the length of the corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm.

Figure 4 shows the corrosion potential distribution along the length of the corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm. For lower tensile strains of 1.375 mm and 2.75 mm, the variation in the corrosion potential is found to be uniform along the length of the corrosion defect. However, for higher tensile strains of 3.75 mm and 4 mm, the variation in the corrosion potential is nonuniform with the more negative corrosion potential at the center of the corrosion defect than that at both the sides of the corrosion defect.

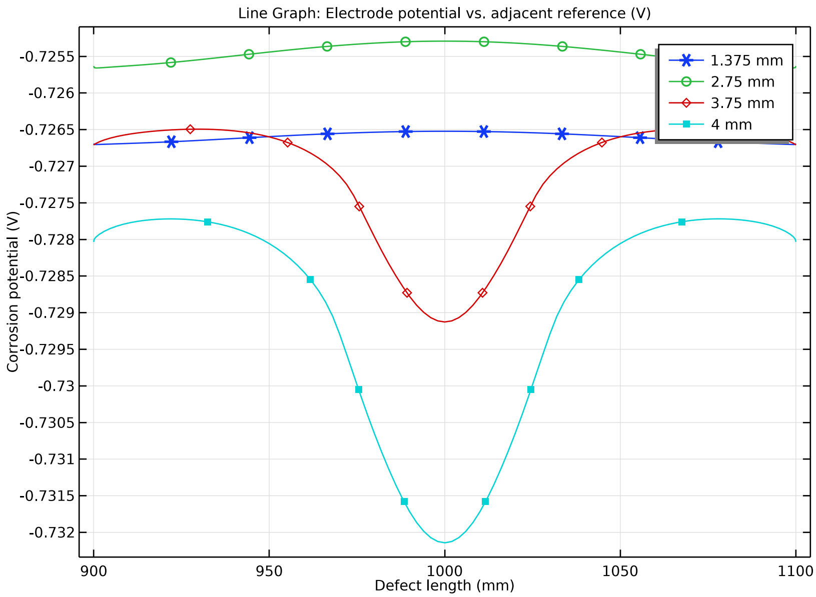

Figure 4: The corrosion potential distribution along the length of the corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm.

Figure 5 shows the anodic current density distribution along the length of corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm. For lower tensile strains of 1.375 mm and 2.75 mm, the variation in the anodic current density is found to be uniform along the length of the corrosion defect, similar to the corrosion potential behavior. However, for higher tensile strains of 3.75 mm and 4 mm, the variation in the anodic current density is significantly nonuniform, particularly at the center of the corrosion defect. It can be seen that the anodic current density increases significantly at the center of the corrosion defect whereas it decreases slightly at both the sides of the corrosion defect for higher tensile strains. The increase in the anodic current density for tensile strains of 3.75 mm and 4 mm is attributed to the plastic deformation observed at the center of the corrosion defect (see Figure 3).

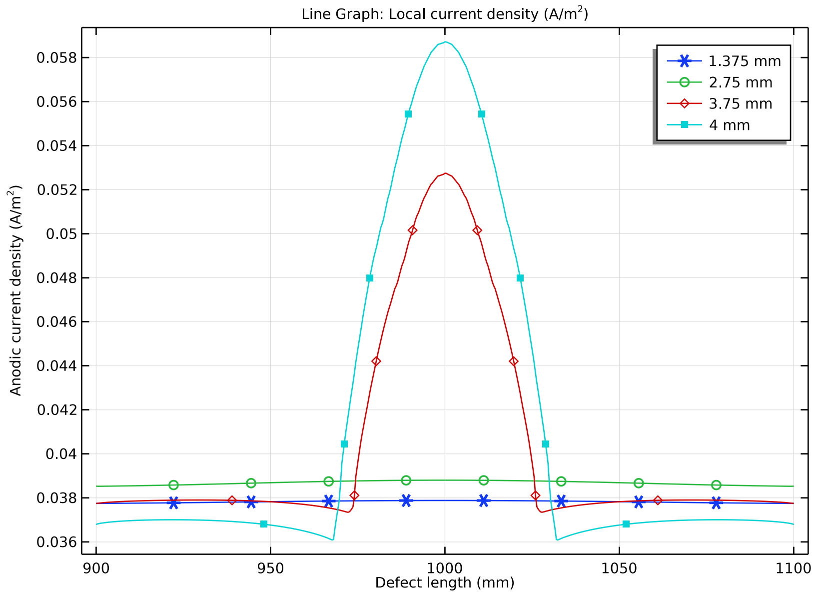

Figure 5: The anodic current density distribution along the length of the corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm.

Figure 6 shows the cathodic current density distribution along the length of the corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm. It can be seen that the cathodic current density increases negatively with an increase in the tensile strain and it is found to be the most negative at the center of the corrosion defect. The nonuniformity in the cathodic current density is also found to increase with an increase in the tensile strain. Thus, the cathodic current density distribution is found to be the most nonuniform for a tensile strain of 4 mm.

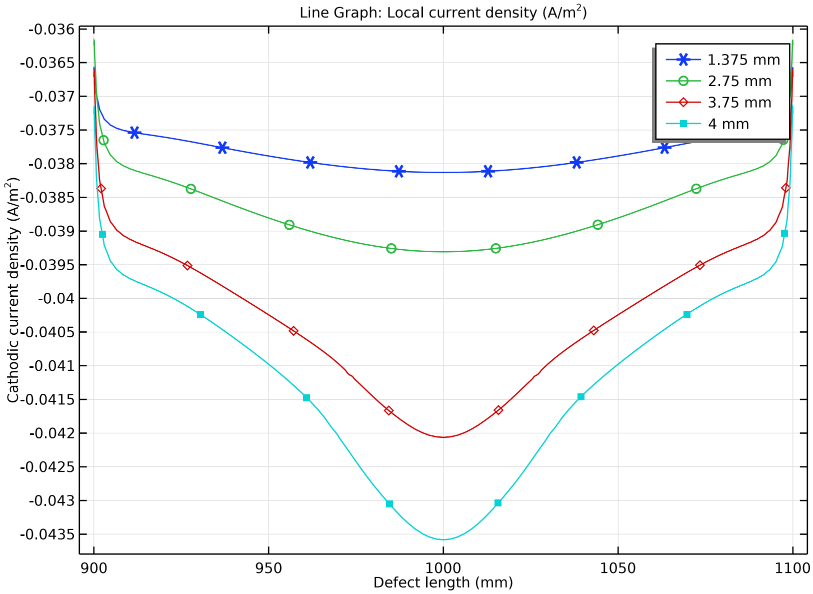

Figure 6: The cathodic current density distribution along the length of the corrosion defect for prescribed displacements of 1.375 mm, 2.75 mm, 3.75 mm, and 4 mm.

Notes About the COMSOL Implementation

The model is implemented using the Solid Mechanics interface and the Secondary Current Distribution interface. Since the Solid Mechanics interface does not depend upon the Secondary Current Distribution interface, we use a sequential solver set up with a Parametric Sweep to study the impact of elastoplastic deformations on electrochemical reactions.

References

1. L. Y. Xu and Y. F. Cheng, “Development of a finite element model for simulation and prediction of mechanoelectrochemical effect of pipeline corrosion”, *Corrosion Science*, vol. 73, pp. 150–160, 2013.

2. L. Xu, “Assessment of corrosion defects on high-strength steel pipelines”, *PhD thesis*, Department of mechanical and manufacturing engineering, University of Calgary, Alberta, August 2013.

Application Library path: `Corrosion_Module/Galvanic_Corrosion/stress_corrosion`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Electrochemistry> Primary and Secondary Current Distribution>Secondary Current Distribution (cd)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 Click **Done**.

GEOMETRY 1

Draw the geometry for pipe with corrosion defect and surrounding soil domain.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `2`.

4 In the **Height** text field, type `19.1[mm]`.

Ellipse 1 (e1)

1 In the **Geometry** toolbar, click **Ellipse**.

2 In the **Settings** window for **Ellipse**, locate the **Size and Shape** section.

3 In the **a-semiaxis** text field, type `100[mm]`.

4 In the **b-semiaxis** text field, type `11.46[mm]`.

5 Locate the **Position** section. In the **x** text field, type `1`.

6 In the **y** text field, type `19.1[mm]`.

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the object **e1** only.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `2`.

4 Click **Build All Objects**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Your geometry should look like Figure 1.

GLOBAL DEFINITIONS

Parameters 1

Load the model parameters.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `stress_corrosion_parameters.txt`.

DEFINITIONS

Load the model variables.

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, locate the **Variables** section.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `stress_corrosion_variables.txt`.

Interpolation 1 (int1)

Load the stress strain interpolation data from a text file.

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `stress_strain_curve`.

4 Click **Load from File**.

5 Browse to the model's Application Libraries folder and double-click the file `stress_corrosion_stress_strain_curve_interpolation.txt`.

6 Locate the **Interpolation and Extrapolation** section. From the **Interpolation** list, choose **Piecewise cubic**.

7 Locate the **Units** section. In the **Arguments** text field, type `1`.

8 In the **Function** text field, type `MPa`.

SOLID MECHANICS (SOLID)

Start setting up the physics. First, set the elastoplastic deformation at the Linear Elastic Material node by adding Plasticity node.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 In the list, select **2**.

4 Click **Remove from Selection**.

5 Select Domain 1 only.

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Plasticity**.

2 In the **Settings** window for **Plasticity**, locate the **Plasticity Model** section.

3 Find the **Isotropic hardening model** subsection. From the list, choose **Hardening function**.

MATERIALS

Now, add high-strength alloy steel material for pipe and set the values for initial yield stress, hardening function, Young's modulus and Poisson's ratio.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>High-strength alloy steel**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

High-strength alloy steel (mat1)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 In the list, select **2**.

3 Click **Remove from Selection**.

4 Select Domain 1 only.

In the table under **Material Contents**, set the value of Initial yield stress to 806e6 [Pa] and Hardening function to `hardening`. Also, change the value of Young's modulus to 207e9 [Pa] and Poisson's ratio to 0.33.

SOLID MECHANICS (SOLID)

Now, set the initial value for displacement field and then proceed to setting up boundary conditions for Solid Mechanics physics interface.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 Specify the **u** vector as

0.0001*X	X
0	Y

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 1 only.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 In the u_{0x} text field, type `disp`.

Prescribed Displacement 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 2 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

SECONDARY CURRENT DISTRIBUTION (CD)

Now, set up the physics for electrochemical reactions. First, set electrolyte conductivity, initial value for electrolyte potential and then set both anodic and cathodic reactions at the corrosion defect surface of pipe.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Secondary Current Distribution (cd)**.

2 In the **Settings** window for **Secondary Current Distribution**, locate the **Domain Selection** section.

3 In the list, select **1**.

4 Click **Remove from Selection**.

5 Select Domain 2 only.

Electrolyte 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Secondary Current Distribution (cd)** click **Electrolyte 1**.

2 In the **Settings** window for **Electrolyte**, locate the **Electrolyte** section.

3 From the σ_l list, choose **User defined**. In the associated text field, type `sigmal`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *phil* text field, type `-(Eeq0a+Eeq0c)/2`.

Electrode Surface 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Electrode Surface**.

2 Select Boundaries 9 and 10 only.

Electrode Reaction 1

1 In the **Model Builder** window, expand the **Electrode Surface 1** node, then click **Electrode Reaction 1**.

2 In the **Settings** window for **Electrode Reaction**, locate the **Equilibrium Potential** section.

3 In the E_{eq} text field, type `Eeqa`.

4 Locate the **Electrode Kinetics** section. From the **Kinetics expression type** list, choose **Anodic Tafel equation**.

5 In the i_0 text field, type `i0a`.

6 In the A_a text field, type `ba`.

Electrode Surface 1

In the **Model Builder** window, click **Electrode Surface 1**.

Electrode Reaction 2

1 In the **Physics** toolbar, click **Attributes** and choose **Electrode Reaction**.

2 In the **Settings** window for **Electrode Reaction**, locate the **Equilibrium Potential** section.

3 In the E_{eq} text field, type `Eeq0c`.

4 Locate the **Electrode Kinetics** section. From the **Kinetics expression type** list, choose **Cathodic Tafel equation**.

5 In the i_0 text field, type `ic`.

6 In the A_c text field, type `bc`.

MESH 1

Set the finer mesh near corrosion defect surface of pipe.

Free Triangular 1

1 In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **All domains**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 2, 9, and 10 only.
5 Locate the **Element Size** section. Click the **Custom** button.
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7 In the associated text field, type `0.002`.
8 Click **Build All**.

STUDY 1

Now, set the solver settings. Since solid mechanics physics is not dependent upon electrochemical reactions, we use a sequential solver setup with a parametric sweep.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.
2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
3 Click **Add**.
4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
disp (Displacement)	0.001375 0.00275 0.00375 0.004	m

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, clear the **Solve for** check box for **Secondary Current Distribution (cd)**.

Stationary 2

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

Solution 1 (sol1)

Lower the relative tolerance for Secondary Current Distribution study step.

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Stationary Solver 2**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `0.00001`.

Clear the Generate default plots check box. The model is now ready to be solved.

5 In the **Model Builder** window, click **Study 1**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 In the **Study** toolbar, click **Compute**.

RESULTS

Corrosion potential and von Mises stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Corrosion potential and von Mises stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

Surface 1

1 In the **Corrosion potential and von Mises stress** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics> Stress>solid.mises - von Mises stress - N/m²**.

3 Locate the **Expression** section. From the **Unit** list, choose **MPa**.

Corrosion potential and von Mises stress

In the **Model Builder** window, click **Corrosion potential and von Mises stress**.

Surface 2

1 In the **Corrosion potential and von Mises stress** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `-phil`.

4 Click to expand the **Range** section. Select the **Manual color range** check box.

5 In the **Minimum** text field, type `-0.733`.

6 In the **Maximum** text field, type `-0.724`.

7 Locate the **Coloring and Style** section. Select the **Reverse color table** check box.

8 In the **Corrosion potential and von Mises stress** toolbar, click **Plot**.

9 Click the **Zoom Extents** button in the **Graphics** toolbar.

The plot should like Figure 2. One can zoom in a region close to corrosion defect using Zoom Box icon in the **Graphics** window.

von Mises Stress

Plot von Mises stress along the corrosion defect for different values of prescribed displacements.

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `von Mises Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (sol3)**.

4 Locate the **Plot Settings** section. Select the **x-axis label** check box.

5 In the associated text field, type `Defect length (mm)`.

Line Graph 1

1 In the **von Mises Stress** toolbar, click **Line Graph**.

2 Select Boundaries 9 and 10 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Stress>solid.mises - von Mises stress - N/m²**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 From the **Unit** list, choose **mm**.

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 Click to expand the **Legends** section. Select the **Show legends** check box.

10 From the **Legends** list, choose **Manual**.

11 In the table, enter the following settings:

Legends
`1.375 mm`
`2.75 mm`

Legends
`3.75 mm`
`4 mm`

12 In the **von Mises Stress** toolbar, click **Plot**.

The plot should look like Figure 3.

von Mises Stress

Now, plot corrosion potential along defect length for different prerscribed displacements.

Corrosion potential

1 In the **Model Builder** window, right-click **von Mises Stress** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Corrosion potential` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **y-axis label** check box.

4 In the associated text field, type `Corrosion potential (V)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Corrosion potential** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)> Secondary Current Distribution>cd.Evsref - Electrode potential vs. adjacent reference - V**.

3 In the **Corrosion potential** toolbar, click **Plot**.

The plot should look like Figure 4.

Corrosion potential

Plot the anodic current density along the corrosion defect.

Anodic current density

1 In the **Model Builder** window, right-click **Corrosion potential** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type Anodic current density in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type Anodic current density (A/m²).

Line Graph 1

1 In the **Model Builder** window, expand the **Anodic current density** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Secondary Current Distribution>Electrode kinetics>cd.iloc_er1 - Local current density - A/m^2**.

3 In the **Anodic current density** toolbar, click **Plot**.

The plot should look like Figure 5.

Anodic current density

Plot the cathodic current density along the corrosion defect.

Cathodic current density

1 In the **Model Builder** window, right-click **Anodic current density** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type Cathodic current density in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type Cathodic current density (A/m²).

Line Graph 1

1 In the **Model Builder** window, expand the **Cathodic current density** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Secondary Current Distribution>Electrode kinetics>cd.iloc_er2 - Local current density - A/m^2**.

3 In the **Cathodic current density** toolbar, click **Plot**.

The plot should look like Figure 6.