Created in COMSOL Multiphysics 6.2

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. As a result of the changed surface energies at the pipe surface, the resulting stresses may have an effect on the pipe corrosion rate. The intercoupled effects 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). The model is built in two parts. In the first part, the stationary effect of the elasto-plastic stress on corrosion is investigated by varying the longitudinal strain using a parametric study. In the second part, a transient simulation is performed to investigate the intercoupled effects of a dissolving, and hence dynamically varying, corrosion defect depth on a pipe, subject to a constant strain.

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.

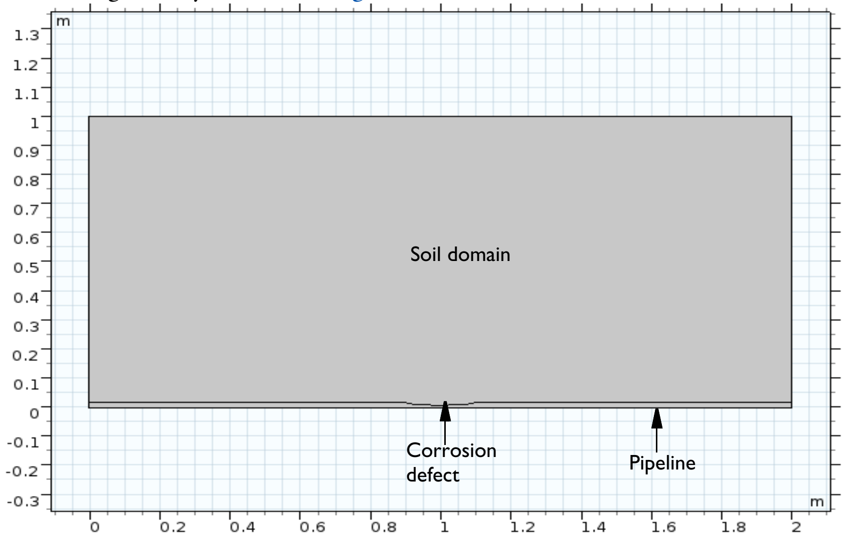

Figure 1: The model geometry consists of a pipeline with corrosion defect and surrounding soil domain.

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:

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

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

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

where $i_{0,\mathrm{a}}$ 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

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

The equilibrium potential for the anodic reaction is calculated from

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

where $E_{\mathrm{eq0,a}}$ 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, $\nu$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 hydrogen evolution reaction, this sets the local cathodic current density to

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

where $i_{0,\mathrm{c}}$ 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

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

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

The exchange current density for the cathodic reaction is calculated from

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

where $i_{0,\mathrm{c,ref}}$ 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).

DEFORMED GEOMETRY

In the second part of the model, dissolution of iron from the corrosion defect is modeled using deformed geometry. The dissolution of iron causes the electrode boundary to move, with a velocity in the normal direction, v (SI unit: m/s), according to

$$v = \frac{i_\mathrm{a}}{2F}\frac{M}{\rho}$$

where i_a (SI unit: A/m^2) is the anodic current density, M is the molar mass (55.845 g/mol) and ρ the density (7870 kg/m^3) of the iron.

Solve the model in a time-dependent study, simulating the corrosion for 20 years of exposure.

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.

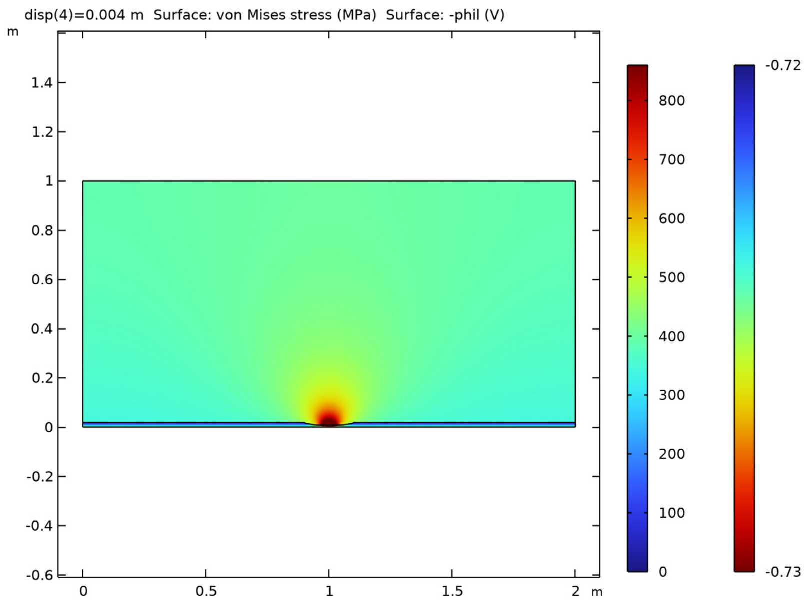

Figure 2: The electrolyte potential distribution over the soil domain and the von Mises stress distribution over the pipeline domain for a prescribed displacement of 4 mm.

Figure 3 shows the von Mises stress distribution along the length of the corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 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 2 mm, 3 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 strain of 1 mm, the entire corrosion defect is observed to be in the elastic deformation range.

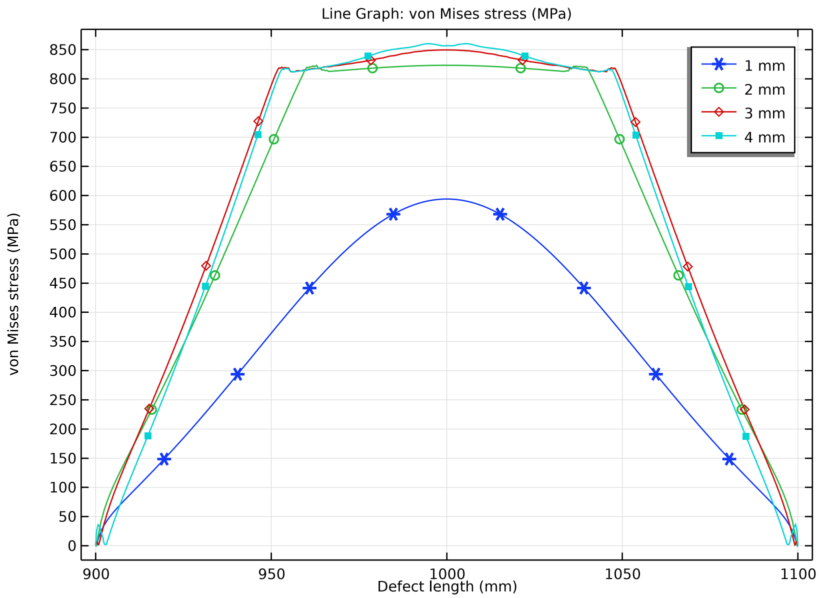

Figure 3: The von Mises stress distribution along the length of the corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 mm, and 4 mm.

Figure 4 shows the corrosion potential distribution along the length of the corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 mm, and 4 mm. For lower tensile strain of 1 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 2 mm, 3 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.

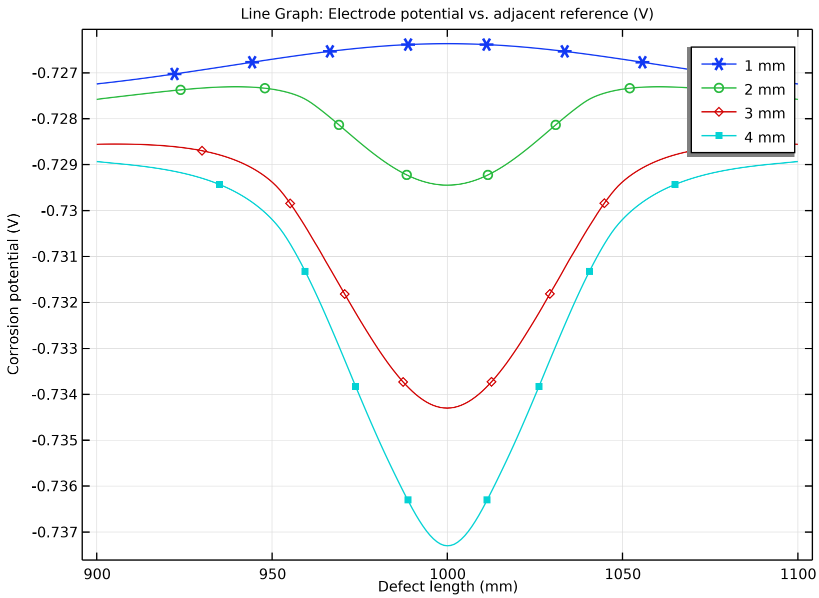

Figure 4: The corrosion potential distribution along the length of the corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 mm, and 4 mm.

Figure 5 shows the anodic current density distribution along the length of corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 mm, and 4 mm. For lower tensile strain of 1 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 2 mm, 3 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 2 mm, 3 mm and 4 mm is attributed to the plastic deformation observed at the center of the corrosion defect (see Figure 3).

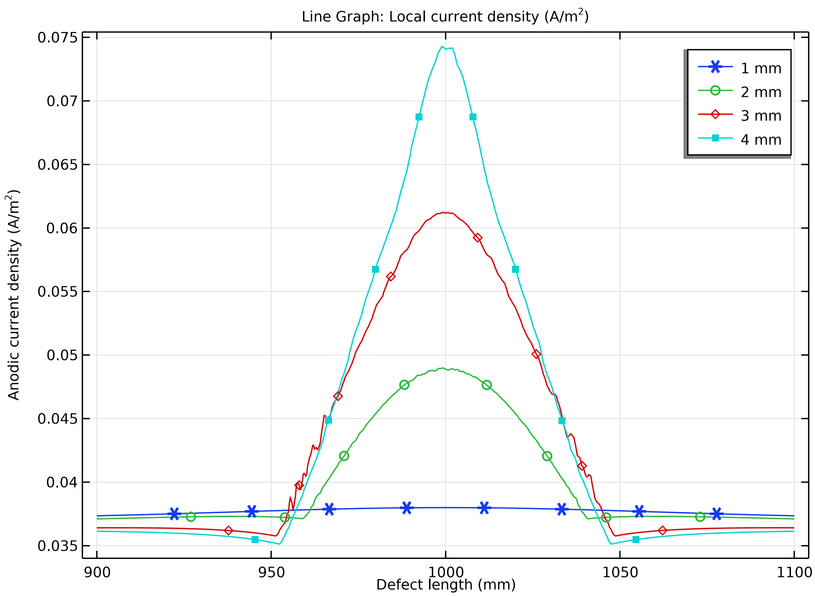

Figure 5: The anodic current density distribution along the length of the corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 mm, and 4 mm.

Figure 6 shows the cathodic current density distribution along the length of the corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 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.

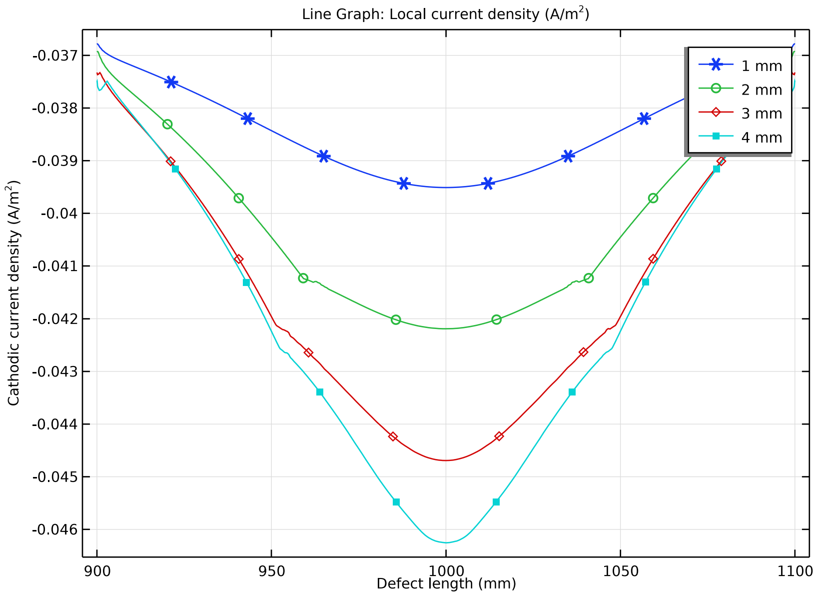

Figure 6: The cathodic current density distribution along the length of the corrosion defect for prescribed displacements of 1 mm, 2 mm, 3 mm, and 4 mm.

[Figure 7](#) shows the corrosion defect profile at time t = 0 a and 20 a. It can be seen that after 20 years, the depth of the corrosion defect is increased particularly at the center of the corrosion defect.

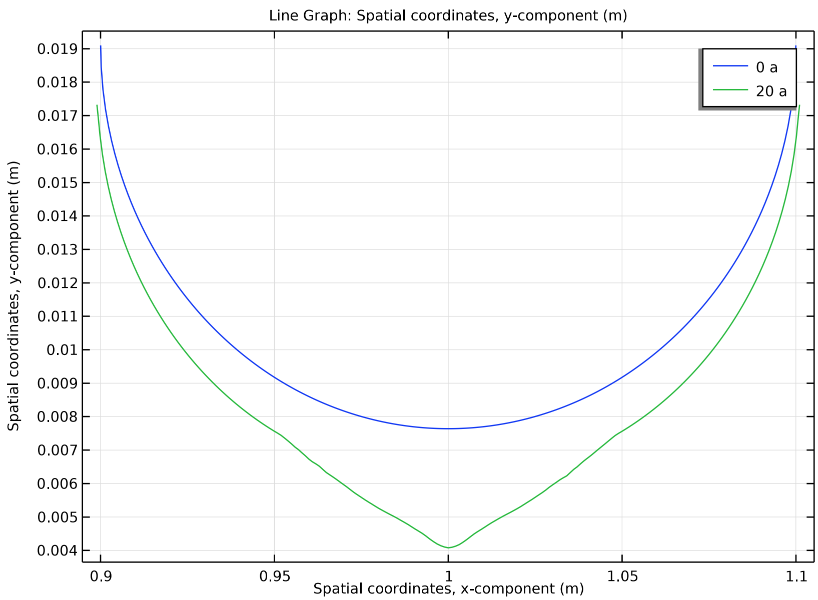

Figure 7: The corrosion defect profile at time t = 0 a and 20 a.

Figure 8 shows the von Mises stress distribution along the length of the corrosion defect at time t = 0 a and 20 a. It can be seen that after 20 years, the von Mises stress is higher at the center of the corrosion defect than that at both the sides of the corrosion defect.

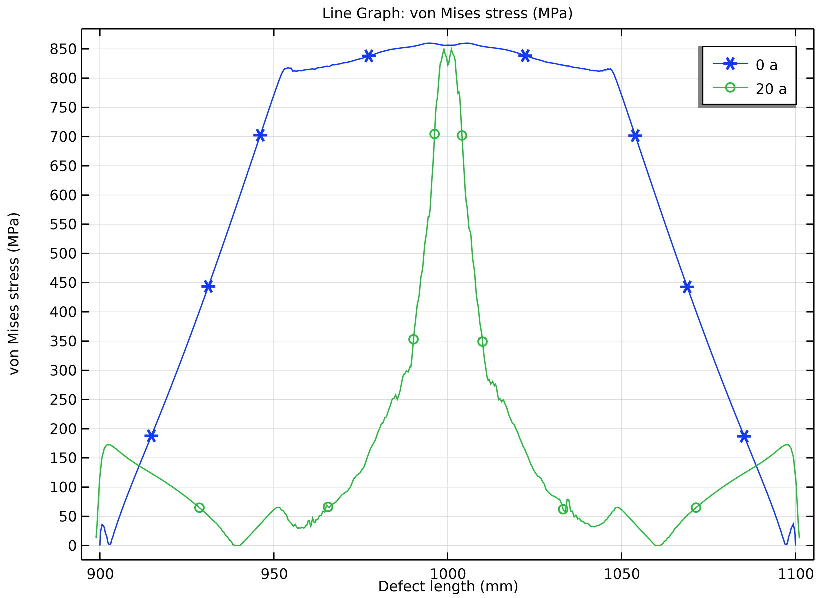

Figure 8: The von Mises stress distribution along the length of the corrosion defect at time t = 0 a and 20 a.

Figure 9 shows the corrosion potential distribution along the length of the corrosion defect at time t = 0 a and 20 a. It can be seen that after 20 years, the corrosion potential is more negative throughout the length of the corrosion defect.

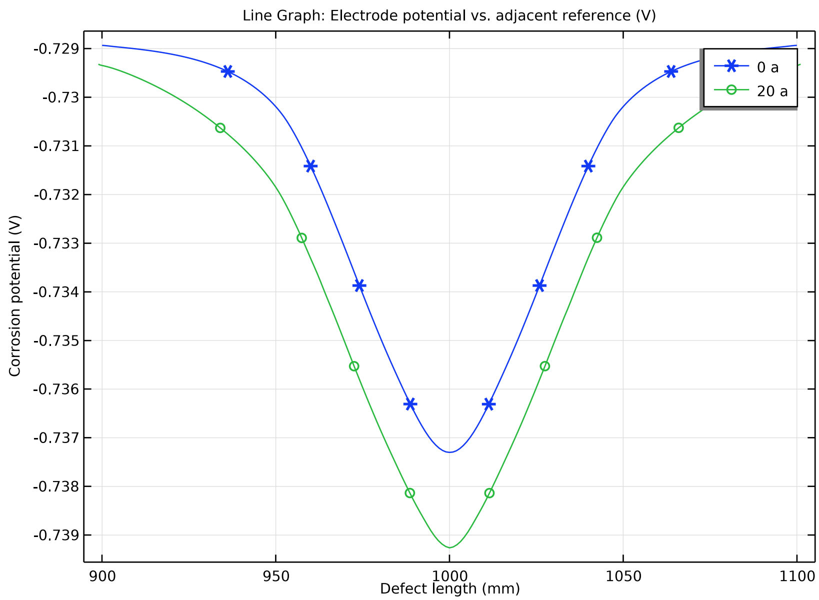

Figure 9: The corrosion potential distribution along the length of the corrosion defect at time t = 0 a and 20 a.

Figure 10 shows the anodic current density distribution along the length of corrosion defect at time t = 0 a and 20 a. It can be seen that after 20 years, the anodic current density is marginally higher at the center of the corrosion defect when compared with that at time t = 0 a.

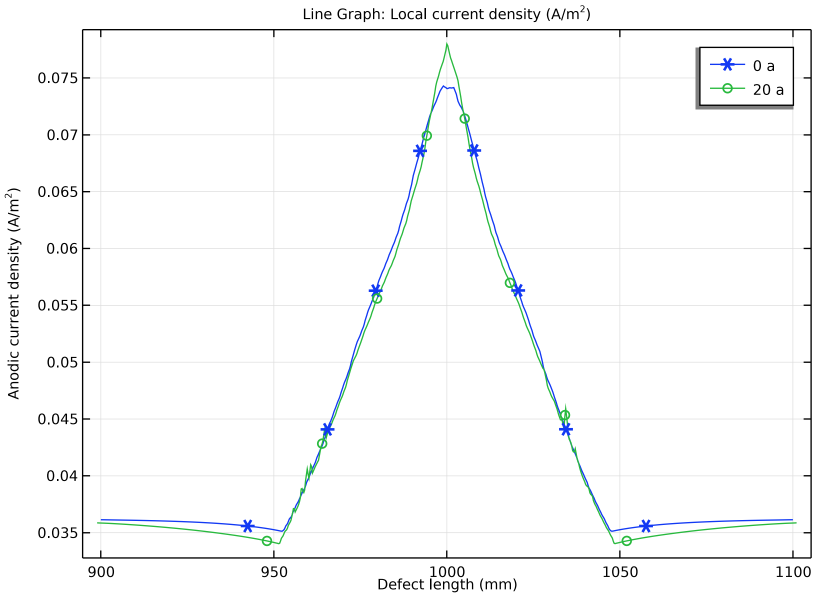

Figure 10: The anodic current density distribution along the length of the corrosion defect at time t = 0 a and 20 a.

Figure 11 shows the cathodic current density distribution along the length of the corrosion defect at time t = 0 a and 20 a. It can be seen that after 20 years, the cathodic current density is more negative at the center of the corrosion defect than that at both the sides of the corrosion defect. The cathodic current density is also found to be marginally more negative at the center of the corrosion defect when compared with that at time t = 0 a.

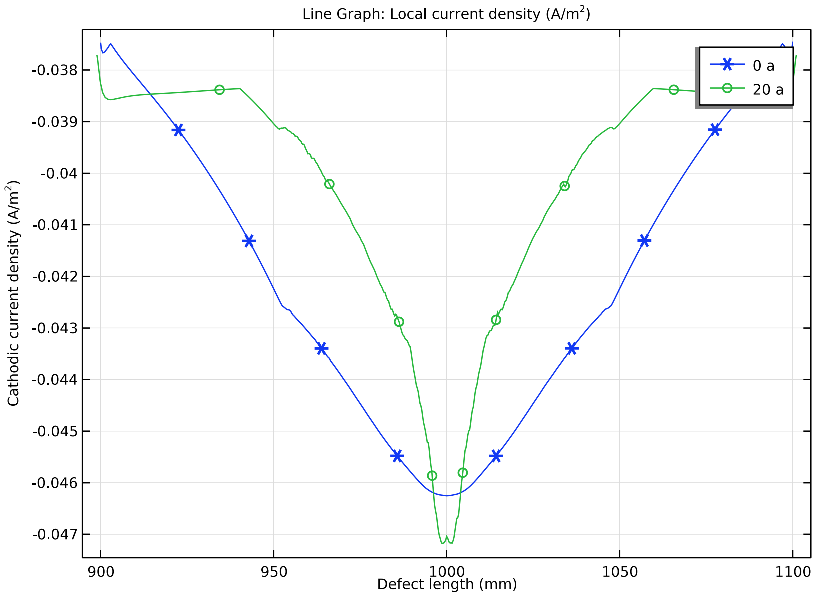

Figure 11: The cathodic current density distribution along the length of the corrosion defect at time t = 0 a and 20 a.

Notes About the COMSOL Implementation

The model is implemented using the Solid Mechanics interface and the Secondary Current Distribution interface. In the first study, the Solid Mechanics model does not depend upon the results of the Secondary Current Distribution model, and we use a sequential solver set up with a Parametric Sweep to study the impact of elastoplastic deformations on electrochemical reactions. In the second part of the model, a transient analysis is performed using Deformed Geometry to analyze the effect of varying depth, due to dissolution, on stress as well as corrosion.

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/General_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 pipe geometry with a corrosion defect and a 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. Click to select the **Activate Selection** toggle button for **Objects to subtract**.
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 that in 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 **Argument** table, enter the following settings:

Argument	Unit
t	1

8 In the **Function** table, enter the following settings:

Function	Unit
stress_strain_curve	MPa

SOLID MECHANICS (SOLID)

Start setting up the physics. First, set the elastoplastic deformation at the Linear Elastic Material node by adding a 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 a high-strength alloy steel material for the 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.

5 Locate the **Material Contents** section. In the table, set the following property values:

Property	Variable	Value	Unit	Property group
Initial yield stress	sigmags	`806[MPa]`	Pa	Elastoplastic material model
Hardening function	sigmagh	`hardening`	Pa	Elastoplastic material model

Property	Variable	Value	Unit	Property group
Young's modulus	E	207[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.33	1	Young's modulus and Poisson's ratio

SOLID MECHANICS (SOLID)

Next, set the initial value for the displacement field and then proceed to the setup of boundary conditions for the Solid Mechanics 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 From the **Displacement in x direction** list, choose **Prescribed**.

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 1 only.

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

4 From the **Displacement in x direction** list, choose **Prescribed**.

Fixed Constraint 2

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

2 Select Point 1 only.

SECONDARY CURRENT DISTRIBUTION (CD)

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

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`.

Current Conductor 1

1 In the **Physics** toolbar, click **Domains** and choose **Current Conductor**.

2 Select Domain 1 only.

3 In the **Settings** window for **Current Conductor**, locate the **Electrode** section.

4 From the σ_s list, choose **User defined**. In the associated text field, type `sigmas`.

Internal Electrode Surface 1

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

2 Select Boundaries 4, 6, 9, and 10 only.

Electrode Reaction 1

1 In the **Model Builder** window, 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`.

Internal Electrode Surface 1

In the **Model Builder** window, click **Internal 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`.

Electric Ground 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Electric Ground**.

2 Select Boundary 1 only.

MESH 1

Set a finer mesh near the corrosion defect surface of pipe.

Free Triangular 1

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

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 **Domain**.

4 Select Domain 1 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.01`.

Size 2

1 In the **Model Builder** window, 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 9 and 10 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.001`.

8 Click **Build All**.

STUDY: STATIONARY PARAMETRIC

Next, set the solver settings. Since the Solid Mechanics interface is independent of the electrochemical reactions, use a sequential solver setup with a parametric sweep.

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

2 In the **Settings** window for **Study**, type `Study: Stationary Parametric` in the **Label** text field.

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)	`0.001 0.002 0.003 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)**.

Step 2: 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 the **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`.

5 In the **Model Builder** window, click **Study: Stationary Parametric**.

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

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

The model is now ready to be solved.

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: Stationary Parametric/ 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.misesGp - 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. From the **Color table transformation** list, choose **Reverse**.

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 the one in Figure 2. Zoom in on a region close to the corrosion defect using the **Zoom Box** button in the **Graphics** window.

von Mises Stress, Parametric

Plot the 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, Parametric` in the **Label** text field.

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Defect length (mm)`.

Line Graph 1

1 In the **von Mises Stress, Parametric** 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.misesGp - 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 From the **Positioning** list, choose **Interpolated**.

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

11 From the **Legends** list, choose **Evaluated**.

12 In the **Legend** text field, type `eval(disp,mm) mm`.

13 In the **von Mises Stress, Parametric** toolbar, click **Plot**.

The plot should look like Figure 3.

von Mises Stress, Parametric

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

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

Corrosion Potential, Parametric

1 In the **Model Builder** window, under **Results** click **von Mises Stress, Parametric 1**.

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Corrosion potential (V)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Corrosion Potential, Parametric** 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, Parametric** toolbar, click **Plot**.

The plot should look like Figure 4.

Corrosion Potential, Parametric

Plot the anodic current density along the corrosion defect.

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

Anodic Current Density, Parametric

1 In the **Model Builder** window, under **Results** click **Corrosion Potential, Parametric 1**.

2 In the **Settings** window for **1D Plot Group**, type `Anodic Current Density, Parametric` in the **Label** text field.

3 Locate the **Plot Settings** section. In the **y-axis label** text field, type `Anodic current density (A/m<sup>2</sup>)`.

Line Graph 1

1 In the **Model Builder** window, expand the **Anodic Current Density, Parametric** 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, Parametric** toolbar, click **Plot**.

The plot should look like Figure 5.

Anodic Current Density, Parametric

Plot the cathodic current density along the corrosion defect.

1 In the **Model Builder** window, right-click **Anodic Current Density, Parametric** and choose **Duplicate**.

Cathodic Current Density, Parametric

1 In the **Model Builder** window, under **Results** click **Anodic Current Density, Parametric 1**.

2 In the **Settings** window for **1D Plot Group**, type Cathodic Current Density, Parametric 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, Parametric** 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, Parametric** toolbar, click **Plot**.

The plot should look like Figure 6.

COMPONENT 1 (COMP1)

In the second part of the model, we add a Deformed Geometry node to investigate the impact of deformation due to dissolution on the stress distribution and hence on the corrosion.

Deforming Domain 1

1 In the **Physics** toolbar, click **Deformed Geometry** and choose **Free Deformation**.

2 In the **Settings** window for **Deforming Domain**, locate the **Smoothing** section.

3 From the **Mesh smoothing type** list, choose **Hyperelastic**.

SECONDARY CURRENT DISTRIBUTION (CD)

Next, add a dissolving-depositing species to capture dissolution at the electrode surface.

Internal Electrode Surface 1

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Secondary Current Distribution (cd)** click **Internal Electrode Surface 1**.
2 In the **Settings** window for **Internal Electrode Surface**, click to expand the **Dissolving-Depositing Species** section.
3 Click **Add**.
4 In the table, enter the following settings:

Species	Density (kg/m^3)	Molar mass (kg/mol)
s1	rho_Fe	M_Fe

Electrode Reaction 1

1 In the **Model Builder** window, click **Electrode Reaction 1**.
2 In the **Settings** window for **Electrode Reaction**, locate the **Stoichiometric Coefficients** section.
3 In the **Stoichiometric coefficients for dissolving-depositing species:** table, enter the following settings:

Species	Stoichiometric coefficient (1)
s1	1

MULTIPHYSICS

Next, set multiphysics features for deformed geometry.

Nondeforming Boundary 1 (ndbdg1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>** **Nondeforming Boundary**.
2 In the **Settings** window for **Nondeforming Boundary**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 Locate the **Nondeforming Boundary** section. From the **Boundary condition** list, choose **Zero normal displacement**.

Deforming Electrode Surface 1 (desdg1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>** **Deforming Electrode Surface**.

2 In the **Settings** window for **Deforming Electrode Surface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

STUDY: STATIONARY PARAMETRIC

Before setting up a transient study, disable the appropriate nodes in the Stationary Parametric study for completeness.

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Stationary Parametric** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Deformed geometry (Component 1)		Automatic

4 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Nondeforming Boundary 1 (ndbdg1)		Automatic (Stationary)
Deforming Electrode Surface 1 (desdg1)		Automatic (Stationary)

Step 2: Stationary 2

1 In the **Model Builder** window, click **Step 2: Stationary 2**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Deformed geometry (Component 1)		Automatic

4 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Nondeforming Boundary 1 (ndbdg1)		Automatic (Stationary)
Deforming Electrode Surface 1 (desdg1)		Automatic (Stationary)

ROOT

Finally, add a new time-dependent study to capture the deformation.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Time Dependent**.

4 Click **Add Study** in the window toolbar.

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

STUDY 2

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

2 From the **Time unit** list, choose **a**.

3 In the **Output times** text field, type `range(0,1,20)`.

4 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study: Stationary Parametric, Stationary 2**.

7 From the **Solution** list, choose **Parametric Solutions 1 (sol3)**.

8 In the **Model Builder** window, click **Study 2**.

9 In the **Settings** window for **Study**, type `Study: Transient, Deformed Geometry` in the **Label** text field.

10 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Solution 8 (sol8)

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

2 In the **Model Builder** window, expand the **Solution 8 (sol8)** node.

3 In the **Model Builder** window, expand the **Study: Transient, Deformed Geometry>Solver Configurations>Solution 8 (sol8)>Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

5 In the **Maximum number of iterations** text field, type `12`.

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

RESULTS

Corrosion Defect Profile

Now, plot the corrosion defect profile for the first and last time steps.

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Transient, Deformed Geometry/Solution 8 (sol8)**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (a)** list, choose **0** and **20**.

Line Graph 1

1 In the **Corrosion Defect Profile** toolbar, click **Line Graph**.

2 Select Boundaries 9 and 10 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `y`.

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

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

7 Locate the **Legends** section. Select the **Show legends** check box.

8 In the **Corrosion Defect Profile** toolbar, click **Plot**.

The plot should look like Figure 7.

von Mises Stress, Parametric

Plot the von Mises stress along the corrosion defect for the first and last time steps.

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

von Mises Stress, Deformed Geometry

1 In the **Model Builder** window, under **Results** click **von Mises Stress, Parametric 1**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Transient, Deformed Geometry/Solution 8 (sol8)**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (a)** list, choose **0** and **20**.

Line Graph 1

1 In the **Model Builder** window, expand the **von Mises Stress, Deformed Geometry** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **Legends** section.

3 From the **Legends** list, choose **Automatic**.

4 In the **von Mises Stress, Deformed Geometry** toolbar, click **Plot**.

The plot should look like Figure 8.

Corrosion Potential, Parametric

Plot the corrosion potential along the corrosion defect for the first and last time steps.

In the **Model Builder** window, under **Results** right-click **Corrosion Potential, Parametric** and choose **Duplicate**.

Corrosion Potential, Deformed Geometry

1 In the **Model Builder** window, under **Results** click **Corrosion Potential, Parametric 1**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Transient, Deformed Geometry/Solution 8 (sol8)**.

4 From the **Time selection** list, choose **From list**.

5 In the **Times (a)** list, choose **0** and **20**.

Line Graph 1

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

2 In the **Settings** window for **Line Graph**, locate the **Legends** section.

3 From the **Legends** list, choose **Automatic**.

4 In the **Corrosion Potential, Deformed Geometry** toolbar, click **Plot**.

The plot should look like Figure 9.

Anodic Current Density, Parametric

Plot the anodic current density along the corrosion defect for the first and last time steps.

In the **Model Builder** window, under **Results** right-click **Anodic Current Density, Parametric** and choose **Duplicate**.

Anodic Current Density, Deformed Geometry

1 In the **Model Builder** window, under **Results** click **Anodic Current Density, Parametric 1**.
2 In the **Settings** window for **1D Plot Group**, type `Anodic Current Density, Deformed Geometry` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study: Transient, Deformed Geometry/Solution 8 (sol8)**.
4 From the **Time selection** list, choose **From list**.
5 In the **Times (a)** list, choose **0** and **20**.

Line Graph 1

1 In the **Model Builder** window, expand the **Anodic Current Density, Deformed Geometry** node, then click **Line Graph 1**.
2 In the **Settings** window for **Line Graph**, locate the **Legends** section.
3 From the **Legends** list, choose **Automatic**.
4 In the **Anodic Current Density, Deformed Geometry** toolbar, click **Plot**.

   The plot should look like Figure 10.

Cathodic Current Density, Parametric

Plot the cathodic current density along the corrosion defect for the first and last time steps.

In the **Model Builder** window, under **Results** right-click **Cathodic Current Density, Parametric** and choose **Duplicate**.

Cathodic Current Density, Deformed Geometry

1 In the **Model Builder** window, under **Results** click **Cathodic Current Density, Parametric 1**.
2 In the **Settings** window for **1D Plot Group**, type `Cathodic Current Density, Deformed Geometry` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study: Transient, Deformed Geometry/Solution 8 (sol8)**.
4 From the **Time selection** list, choose **From list**.
5 In the **Times (a)** list, choose **0** and **20**.

Line Graph 1

1 In the **Model Builder** window, expand the **Cathodic Current Density, Deformed Geometry** node, then click **Line Graph 1**.

2 In the **Settings** window for **Line Graph**, locate the **Legends** section.

3 From the **Legends** list, choose **Automatic**.

4 In the **Cathodic Current Density, Deformed Geometry** toolbar, click **Plot**.

The plot should look like Figure 11.