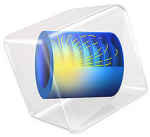


Cathodic Protection with Anode Deformation

Introduction

This tutorial models the cathodic protection of an oil rig structure during a time period of 30 years.

As a result of the consumption of the sacrificial anodes, the protective capabilities of the system are reduced over time.

The anode shape change is defined by using the Level Set interface, coupled to the anode dissolution rate defined by the Cathodic Protection interface.

This tutorial is assuming the reader to already be fairly well-acquainted with corrosion protection modeling in COMSOL Multiphysics. For an introduction, see the Cathodic Protection of Steel in Reinforced Concrete tutorial.

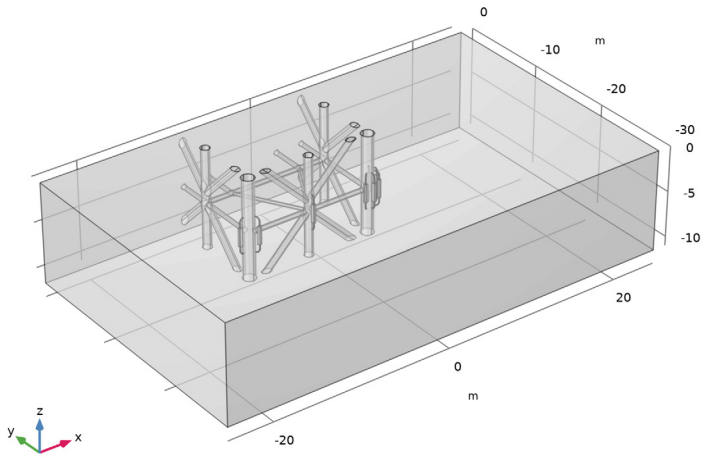


Figure 1: Model geometry, including the electrolyte bounding box, the protected steel structure and the connected sacrificial anodes.

Model Definition

Figure 1 shows the model geometry. A steel structure is immersed into the ocean. To the steel structure, a number of sacrificial anodes have been attached. A close-up of the structure is shown in Figure 2.

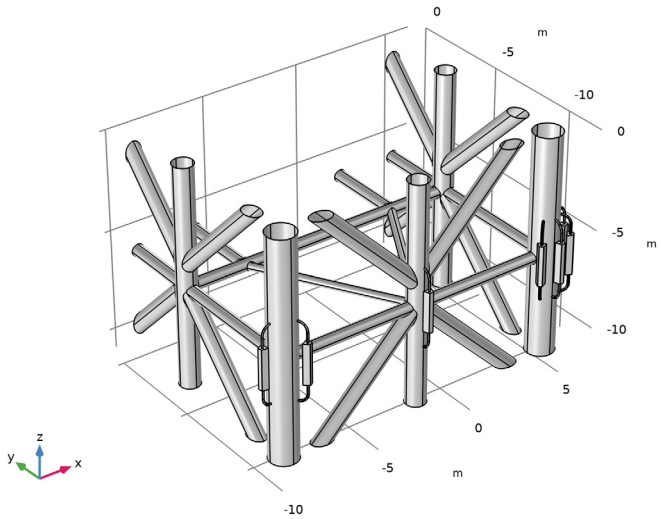


Figure 2: Model geometry close-up of the steel structure and connected sacrificial anodes.

A **Cathodic Protection** (Secondary Current Distribution) interface is used to define the electrochemistry part of the model, where a **Limited linear ramp** function is used to define the oxygen reduction kinetics of the protected steel structure, using a constant (limiting) current density below –0.8 V versus Ag/AgCl and a linear ramp from –0.8 up to –0.6 V versus Ag/AgCl. Linear Butler–Volmer kinetics are used for the sacrificial anodes, with an equilibrium voltage of –1.05 V versus Ag/AgCl. The electrolyte conductivity of the ocean salt water is taken from **Saltwater** in the **Corrosion Material Library**, with the temperature set to 10°C.

A **Level Set** interface is used to model the dissolution of the anodes, with a level set variable of the value 0 representing a solid electrode, and the value 1 representing the pure electrolyte (ocean) phase.

The **Highly Conductive Porous Electrode** node is used to define the anode domains in the cathodic protection interface. Defining the anode domains as "porous" in this model, with the porosity mainly being either 0 (solid anode) or 1 (fully dissolved) is a convenient approach since the **Highly Conductive Porous Electrode** node allows for simultaneously

defining the electrolyte charge transfer, with the effective electrolyte conductivity as a function of the porosity, and the electrode kinetics in a **Porous Electrode Reaction** subnode, with the specific surface area set to the level set delta variable.

In the level-set interface, the propagation velocity V_n (m/s) of the electrolyte-electrode interface in the anodes is then set to

$$V_n = \frac{i_{\text{loc}}}{C\rho}$$

where C (Ah/kg), ρ (kg/m^3) is the density of the anodes and i_{loc} is the local current density calculated by the **Porous Electrode Reaction** subnode.

INITIAL VALUES FOR THE LEVEL SET VARIABLE

The initial value of the level set variable is set to 0 in the interior of the anode domains and to 1 at the boundaries facing the electrolyte domain, with the value varying continuously over a distance of 1 cm when moving from the boundary toward the interior. This initialization of the level set variable is accomplished by the use of the `sphavg()` operator, which for each spatial coordinate integrates and calculates the average of an expression over a small sphere of a given radius. The integration expression is formulated to be 2 if the integration coordinates are placed in the same domain as the electrolyte, and 0 if the integration coordinates are located within the anode domains. The radius of the small sphere in the `sphavg()` call is set to 1 cm.

Results and Discussion

Figure 3 to Figure 6 show the potential of the structure and the shape of the anodes at 0, 10, 20, and 30 years of simulated time. Only small changes in the potential levels are seen between 0 to 20 years, whereas the levels are changed significantly between 20 and 30 years. In Figure 6 (after 30 years) some of the anodes are fully dissolved.

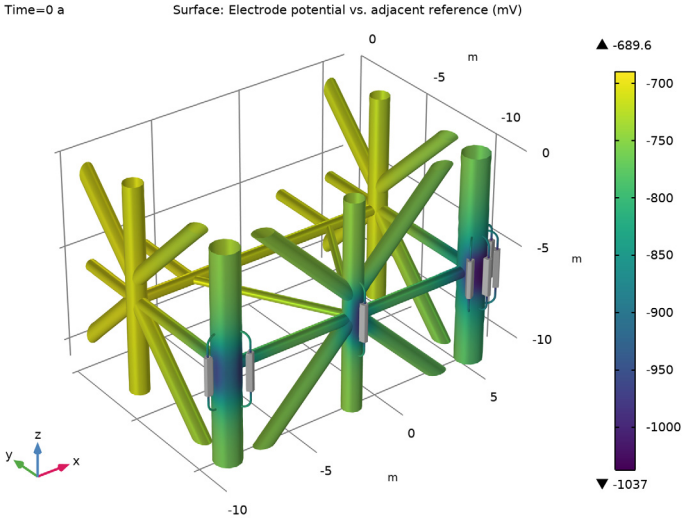


Figure 3: Initial potential versus Ag/AgCl of the protected steel structure (t = 0).

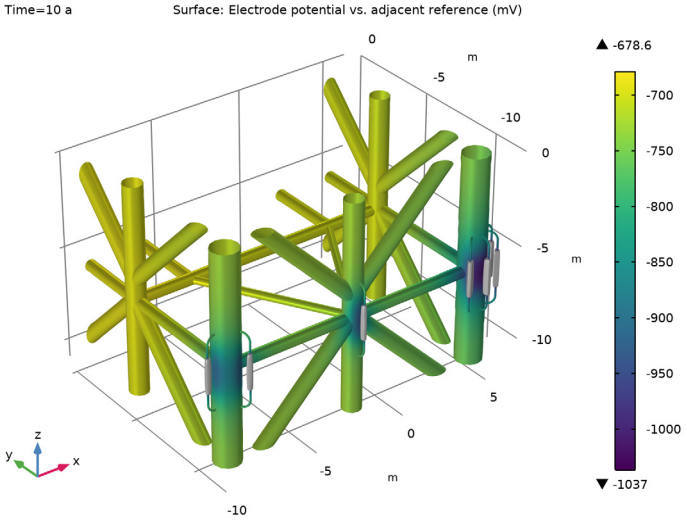


Figure 4: Potential versus Ag/AgCl of the protected steel structure after 10 years.

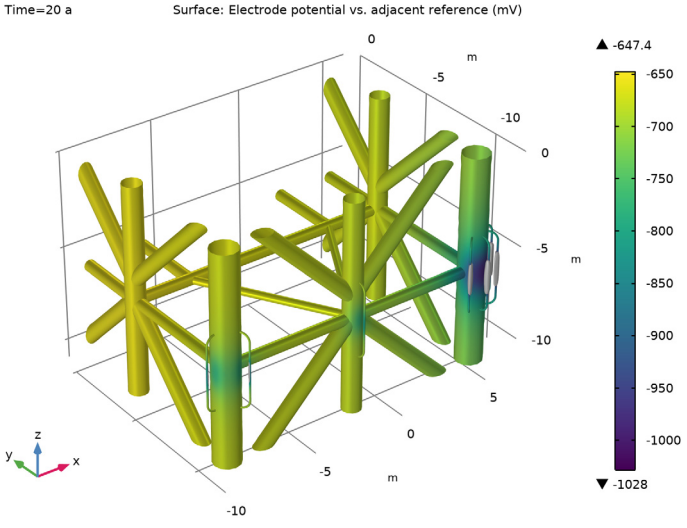


Figure 5: Potential versus Ag/AgCl of the protected steel structure after 20 years.

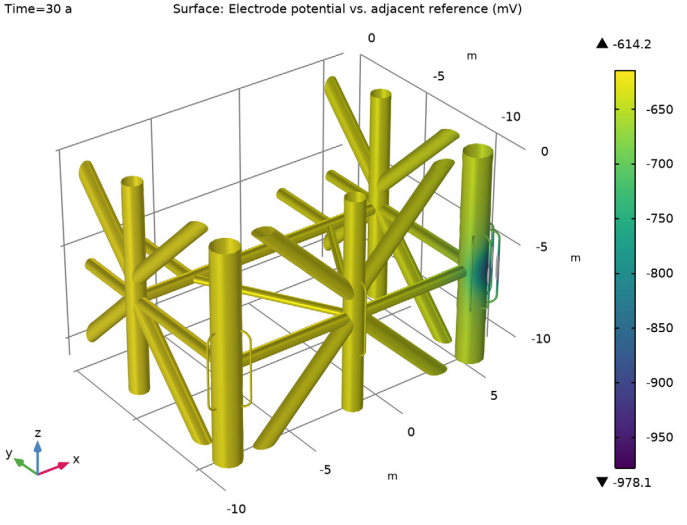


Figure 6: Potential versus Ag/AgCl of the protected steel structure after 30 years.

Notes About the COMSOL Implementation

`max()` and `min()` function calls are used in the variable expression for `epsl` (the electrolyte volume fraction in the anodes). These wrappings improve convergence.

Application Library path: `Corrosion_Module/Cathodic_Protection/cp_with_anode_deformation`

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 **3D**.
2. In the **Select Physics** tree, select **Electrochemistry>Cathodic Protection (cp)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Mathematics>Moving Interface>Level Set (ls)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces> Time Dependent with Initialization**.
8. Click **Done**.

GEOMETRY 1

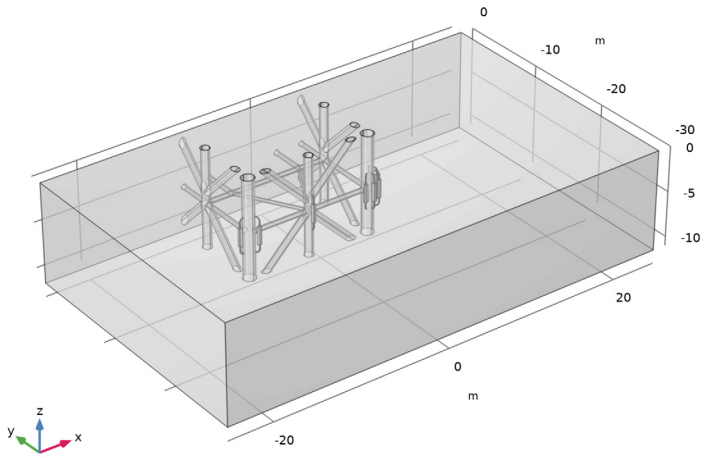
Import 1 (imp1)

Import the model geometry from a file.

1. In the **Home** toolbar, click **Import**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. Click **Browse**.
4. Browse to the model's Application Libraries folder and double-click the file `cp_with_anode_deformation.mphbin`.

5 Click **Build All Objects**.

6 Click the **Transparency** button in the **Graphics** toolbar.



GLOBAL DEFINITIONS

Parameters 1

Import also some model parameters and variables from text files.

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 `cp_with_anode_deformation_parameters.txt`.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

Some of the imported variable expressions will be marked in orange, indicating missing variables. This will be corrected when setting up the physics.

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

3 Click **Load from File**.

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

Electrolyte

Add a number of named selections to the model. This will facilitate setting up the physics, meshing and results later on.

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Electrolyte` in the **Label** text field.

3 Select Domain 1 only.

Anodes

1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Anodes` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

4 In the **Add** dialog box, select **Electrolyte** in the **Selections to invert** list.

5 Click **OK**.

Electrolyte Boundaries

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Electrolyte Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Electrolyte** in the **Input selections** list.

5 Click **OK**.

6 Right-click **Electrolyte Boundaries** and choose **Duplicate**.

Anode Boundaries

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions>Selections** click **Electrolyte Boundaries 1**.

2 In the **Settings** window for **Adjacent**, type `Anode Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. In the **Input selections** list, select **Electrolyte**.

4 Under **Input selections**, click **Delete**.

5 Under **Input selections**, click **Add**.

6 In the **Add** dialog box, select **Anodes** in the **Input selections** list.

7 Click **OK**.

Electrolyte-Anode Boundaries

1 In the **Definitions** toolbar, click **Intersection**.

2 In the **Settings** window for **Intersection**, type `Electrolyte-Anode Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to intersect**, click **Add**.

5 In the **Add** dialog box, in the **Selections to intersect** list, choose **Electrolyte Boundaries** and **Anode Boundaries**.

6 Click **OK**.

Outer Electrolyte Boundaries

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Outer Electrolyte Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

Select the six outer boundaries of the electrolyte block.

4 Select Boundaries 1–5 and 394 only.

Complement to Outer Electrolyte Boundaries

1 In the **Definitions** toolbar, click **Complement**.

2 In the **Settings** window for **Complement**, type `Complement to Outer Electrolyte Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to invert**, click **Add**.

5 In the **Add** dialog box, select **Outer Electrolyte Boundaries** in the **Selections to invert** list.

6 Click **OK**.

Structure

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `Structure` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

5 In the **Add** dialog box, select **Complement to Outer Electrolyte Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **+ Add**.

9 In the **Add** dialog box, select **Electrolyte-Anode Boundaries** in the **Selections to subtract** list.

10 Click **OK**.

Protected Steel Boundaries

1 In the **Definitions** toolbar, click **Difference**.

2 In the **Settings** window for **Difference**, type `Protected Steel Boundaries` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Under **Selections to add**, click **+ Add**.

5 In the **Add** dialog box, select **Complement to Outer Electrolyte Boundaries** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference**, locate the **Input Entities** section.

8 Under **Selections to subtract**, click **+ Add**.

9 In the **Add** dialog box, select **Anode Boundaries** in the **Selections to subtract** list.

10 Click **OK**.

MATERIALS

In this model the electrolyte conductivity is defined in the **Materials** node.

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 **Corrosion>Electrolytes>Seawater**.

4 Right-click and choose **Add to Component 1 (comp1)**.

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

CATHODIC PROTECTION (CP)

Lower the discretization order of the electrolyte phase potential. This will lower the memory and solver time requirements.

1 In the **Settings** window for **Cathodic Protection**, click to expand the **Discretization** section.

2 From the **Electrolyte potential** list, choose **Linear**.

DEFINITIONS

Add a second **View** that hides the outer electrolyte boundaries. You may then toggle between the views when defining the physics.

View 2

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

Hide for Physics 1

1 In the **Model Builder** window, right-click **View 2** and choose **Hide for Physics**.

2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Outer Electrolyte Boundaries**.

View 2

1 In the **Model Builder** window, click **View 2**.

2 In the **Settings** window for **View**, click to expand the **Transparency** section.

3 Clear the **Transparency** check box.

View 1

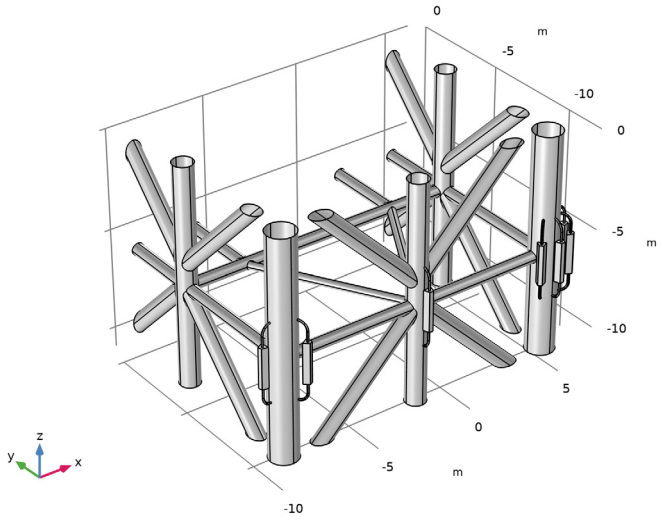
1 In the **Model Builder** window, click **View 1**.

2 In the **Settings** window for **View**, locate the **Transparency** section.

3 Clear the **Transparency** check box.

View 2

In the **Model Builder** window, click **View 2**.



CATHODIC PROTECTION (CP)

In this tutorial we will use the **Highly Conductive Porous Electrode** node to define the deforming anode domains. The porosity variable, defined under **Variables**, depends on the level-set variable.

Highly Conductive Porous Electrode - Anodes

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Cathodic Protection (cp)** and choose **Highly Conductive Porous Electrode**.
2. In the **Settings** window for **Highly Conductive Porous Electrode**, type `Highly Conductive Porous Electrode - Anodes` in the **Label** text field.
3. Locate the **Domain Selection** section. From the **Selection** list, choose **Anodes**.
4. Locate the **Electrolyte Current Conduction** section. In the ε_l text field, type `epsl`.
5. From the **Effective conductivity correction** list, choose **Tortuosity**.

Porous Electrode Reaction 1

1. In the **Model Builder** window, click **Porous Electrode Reaction 1**.
2. In the **Settings** window for **Porous Electrode Reaction**, locate the **Equilibrium Potential** section.
3. In the E_{eq} text field, type `Eeq_anodes`.

4 Locate the **Electrode Kinetics** section. In the i_0 text field, type `i0_anodes`.

Set the specific surface area to equal the level set delta variable. In this way the electrode reaction will only be active at the phase boundary between the electrode and electrolyte phases.

5 Locate the **Active Specific Surface Area** section. In the a_v text field, type `ls.delta`.

DEFINITIONS

Variables 1

The variable expression for `Vn`, using the local current density variable defined by the **Porous Electrode Reaction** node you just defined, should now have turned black.

CATHODIC PROTECTION (CP)

Protected Metal Surface 1

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

2 In the **Settings** window for **Protected Metal Surface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Protected Steel Boundaries**.

4 Locate the **Oxygen Reduction Current Density** section. From the **Expression type** list, choose **Limited linear ramp**.

5 In the E_0 text field, type `E_zero_steel`.

6 In the E_{lim} text field, type `E_lim_O2`.

7 In the $i_{O2,lim}$ text field, type `i_lim_O2`.

GLOBAL DEFINITIONS

Default Model Inputs

Set the temperature to be used by all physics nodes as follows:

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

2 In the **Settings** window for **Default Model Inputs**, locate the **Browse Model Inputs** section.

3 In the tree, select **General>Temperature (K) - minput.T**.

4 Find the **Expression for remaining selection** subsection. In the **Temperature** text field, type `T`.

LEVEL SET (LS)

Now set up the Level Set interface. It should only be active in the anode domains.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Level Set (ls)**.

2 In the **Settings** window for **Level Set**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Anodes**.

Level Set Model 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Level Set (ls)** click **Level Set Model 1**.

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

3 In the γ text field, type `max(Vn,eps)`.

4 In the ε_{ls} text field, type `h_interface`.

5 Locate the **Convection** section. Specify the $\mathbf{u}$ vector as

`Vn*ls.intnormx`	`x`
`Vn*ls.intnormy`	`y`
`Vn*ls.intnormz`	`z`

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 From the **Level set variable** list, choose **User defined**.

4 In the ϕ text field, type `phils_init`.

Initial Values, Fluid 2

Since we are using a user-defined initial value for the level-set variable in the first **Initial Values** node, there is no need for this second node.

1 In the **Model Builder** window, right-click **Initial Values, Fluid 2** and choose **Disable**.

Inlet 1

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

2 In the **Settings** window for **Inlet**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Electrolyte-Anode Boundaries**.

4 Locate the **Level Set Condition** section. From the list, choose **Fluid 2 (φ = 1)**.

The physics settings are now complete. The next step is to set up the mesh.

MESH 1

Use a swept mesh for the anode domains.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

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

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

4 From the **Selection** list, choose **Anodes**.

Size 1

1 Right-click **Swept 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

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

5 Select the **Maximum element size** check box. In the associated text field, type `0.05`.

Size

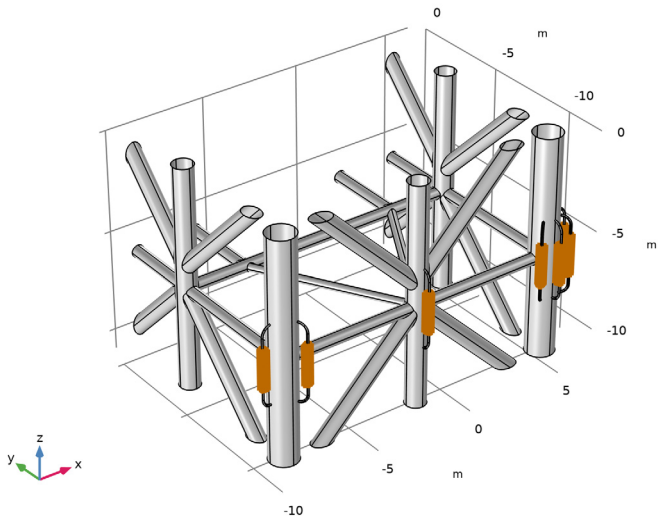
1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

3 In the **Minimum element size** text field, type `0.05`.

Swept 1

In the **Model Builder** window, right-click **Swept 1** and choose **Build Selected**.



Boundary Layers 1

Add a boundary layer inside the anode domains. This will improve the resolution of the initial level-set interface, and the solution accuracy toward the final depletion of the anodes.

1 In the **Mesh** toolbar, click  **Boundary Layers**.

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Anodes**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

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

4 Locate the **Layers** section. In the **Number of layers** text field, type `1`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `h_interface`.

7 Click  **Build Selected**.

Free Tetrahedral 1

Use a free tetrahedral mesh for the remaining electrolyte domain.

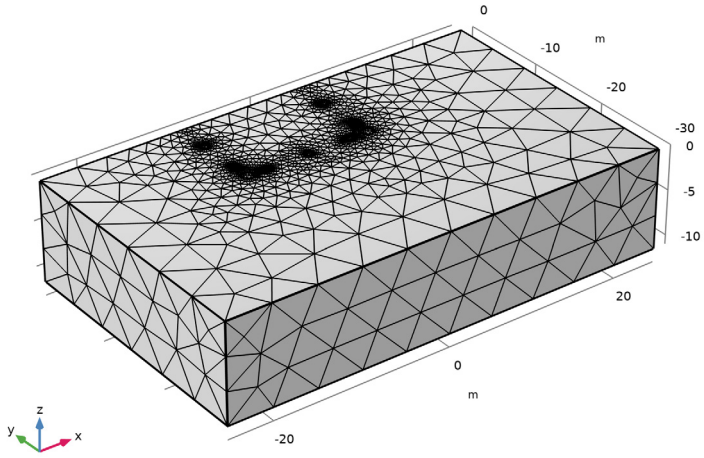
1 In the **Mesh** toolbar, click  **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click  **Build All**.

MESH 1

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

2 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.



The model is now ready for solving.

STUDY 1

Step 1: Current Distribution Initialization

Use a Secondary current distribution for the initialization step. This will improve the accuracy of the initial values used for the following time-dependent step.

1 In the **Model Builder** window, under **Study 1** click **Step 1: Current Distribution Initialization**.

2 In the **Settings** window for **Current Distribution Initialization**, locate the **Study Settings** section.

3 From the **Current distribution type** list, choose **Secondary**.

Step 2: Time Dependent

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

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

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

Set the simulation time to range from 0 up to 30 years, storing the result every year.

4 In the **Output times** text field, type `range(0,1,30)`.

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 **Home** toolbar, click **Compute**.

The solution will take some 10-30 minutes to solve, depending on the computer.

RESULTS

Reproduce the plots from the Results and Discussion section as follows:

Electrode Potential vs Adjacent Ref

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Electrode Potential vs Adjacent Ref` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

4 Click to expand the **Color Legend** section. Select the **Show maximum and minimum values** check box.

5 Click to expand the **Number Format** section. Select the **Manual color legend settings** check box.

6 In the **Precision** text field, type `4`.

Surface 1

1 Right-click **Electrode Potential vs Adjacent Ref** and choose **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)> Cathodic Protection>cp.Evsref - Electrode potential vs. adjacent reference - V**.

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

4 Locate the **Coloring and Style** section. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>Viridis** in the tree.

6 Click **OK**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Structure**.

Isosurface 1

1 In the **Model Builder** window, right-click **Electrode Potential vs Adjacent Ref** and choose **Isosurface**.

2 In the **Settings** window for **Isosurface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Level Set> ls.Vf2 - Volume fraction of fluid 2 - 1**.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

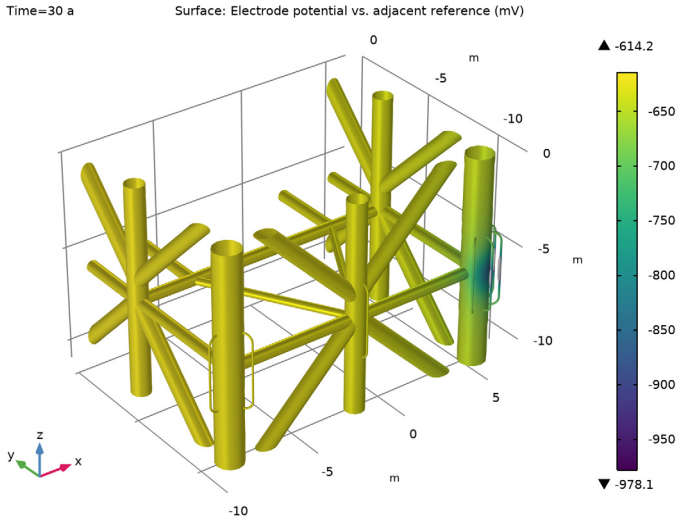
4 Locate the **Levels** section. In the **Total levels** text field, type 1.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

7 Clear the **Color legend** check box.

8 In the **Electrode Potential vs Adjacent Ref** toolbar, click **Plot**.



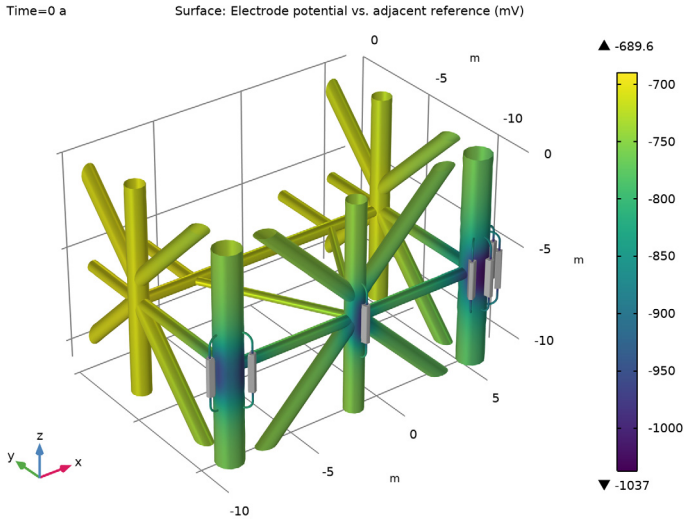
Electrode Potential vs Adjacent Ref

1 In the **Model Builder** window, click **Electrode Potential vs Adjacent Ref**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

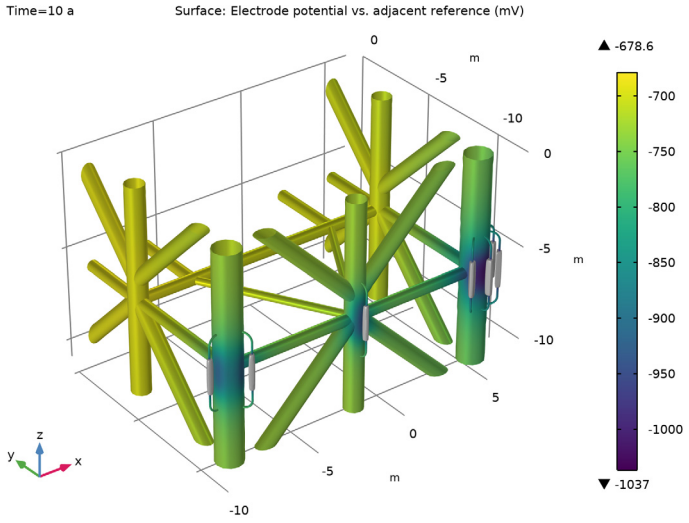
3 From the **Time (a)** list, choose **0**.

4 In the **Electrode Potential vs Adjacent Ref** toolbar, click **Plot**.



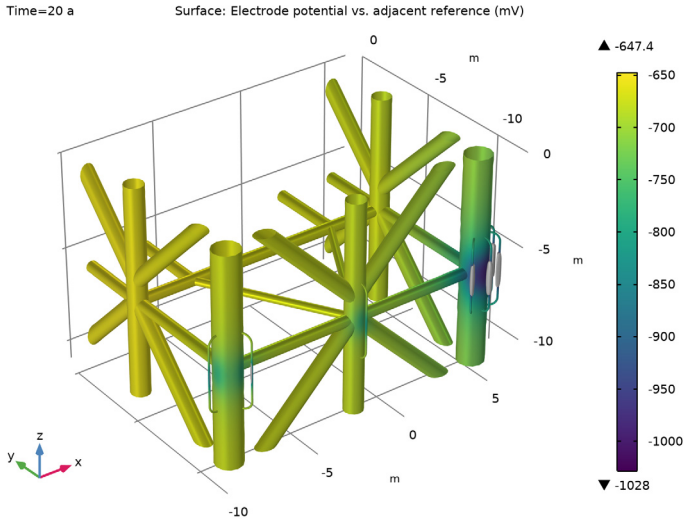
5 From the **Time (a)** list, choose **10**.

6 In the **Electrode Potential vs Adjacent Ref** toolbar, click **Plot**.



7 From the **Time (a)** list, choose **20**.

8 In the **Electrode Potential vs Adjacent Ref** toolbar, click **Plot**.



9 From the **Time (a)** list, choose **30**.