Created in COMSOL Multiphysics 6.1

Submarine Cable 1 — Introduction

Introduction

Numerical analysis of cable systems is an active field of study. It is not only dominated by scientific insight, but also engineering practice and numerical considerations. Production cost plays an important role, and so does durability. The cable industry tends to be on the conservative side — and for good reasons; replacing a faulty submarine cable can be extremely costly[1]. The typical design may last for over forty years. To make sure one gets a good return on investment, the industry has been relying heavily on rules of thumb, safety margins, life-cycle analysis, and international standards such as the IEC 60287 [1].

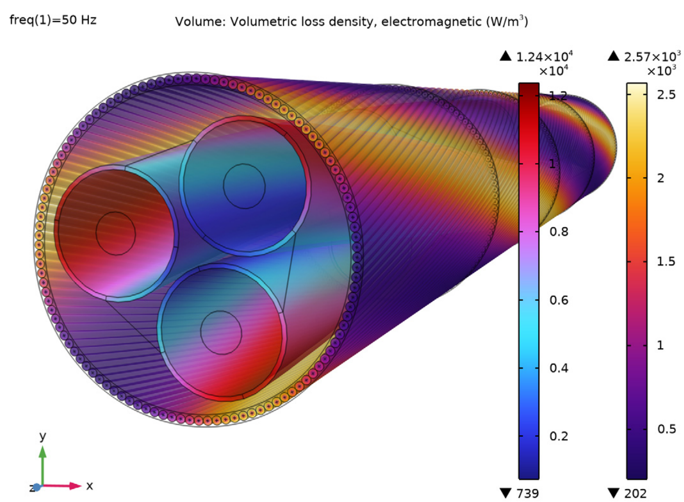


Figure 1: The volumetric loss density in the screens and the armor of a three-core lead sheathed XLPE HVAC submarine cable at a nominal phase temperature of 90°C.

In the meantime, however, computational resources have increased drastically. When investigating research on the matter [2, 6], one cannot help but notice a strong correlation between the year of publishing and the level of detail present in the numerical models. In particular the introduction of *twisted periodicity* [2] and *short-twisted periodicity* [3] has been of great significance: Fully detailed 3D cable modeling is now possible within minutes on relatively cheap hardware (rather than hours, on cluster systems).

As a consequence, traditional methods and safety margins are being questioned: *Numerical research is pushing the industry to find more cost-effective solutions.*

1. Up to several hundred million US dollars for a replacement, and tens of millions of dollars for a repair.

Tutorial Series Overview

This tutorial series discusses the capacitive, inductive, and thermal properties of a three-core, lead sheathed XLPE HVAC[2] submarine cable. It has a main conductor cross section of $500\ \mathrm{mm}^2$, a phase-to-phase voltage of $220\ \mathrm{kV}$, and carries a nominal current of $655\ \mathrm{A}$.

The series is written in eight chapters (as summarized on the following pages), starting with the fundamental principles in 2D. In a systematic fashion it deduces and proofs what cable experts already know — *as a form of validation* — and then continues to increase the complexity to a point where 2D models are no longer sufficient.

Apart from discussing cable related topics such as charging currents, bonding types, armor twists, and temperature dependency, a lot of attention is given to electromagnetism and numerical modeling in general. This includes the importance of certain material properties and geometric features, ill-posed problems, weakly coupled systems, verification, and simplification.

Note: Although this tutorial series investigates a specific kind of *submarine cable*, many of the discussed topics apply to *terrestrial cable* systems just the same. In fact, some of the treated topics (such as *cross bonding*) are included specifically because they are of interest for cable systems in general, not so much for submarine cables in particular.

A NOTE ON EDUCATIONAL VALUE

Although this tutorial will be of particular use for those working within the cable industry, it is not all “just about cables”. This tutorial — and the entire series, for that matter — is about *Electromagnetics*, and *Numerical Analysis*. It is about good engineering practices, about understanding and applying theory, about result validation, and about presenting your results in an attractive and informative way.

The three-phase cable with the magnetic, twisted armor is and ideal device to illustrate and investigate various electromagnetic and numeric phenomena. Since many of these cables are standardized[3], their physical properties are available from literature (allowing for *validation*). At the same time, they are a part of ongoing research. This makes them a suitable tool for industry professionals and academic students alike, to familiarize themselves with the numerical analysis of electromagnetic devices in general.

2. Cross-linked polyethylene, high-voltage alternating current.

3. They are typically based on IEC 60287 or similar standards.

Note: At this point it may be tempting to skip to the 3D part of the tutorial series directly. It should be noted however, that many of the cable's aspects can be understood and investigated perfectly fine with 2D models alone. The computational cost of a 2D model is negligible compared to 3D. *When in doubt, start in 2D.*

SUBMARINE CABLE I — INTRODUCTION

The model that results from this tutorial serves as a template for chapters 2, 4, and 6 (the *Capacitive*, *Inductive*, and *Thermal Effects* tutorials). It contains a detailed 2D geometry of the submarine cable, along with a mesh. Some parameters, selections, and materials are prepared as well. The corresponding instructions start in section Modeling Instructions (Introduction Tutorial).

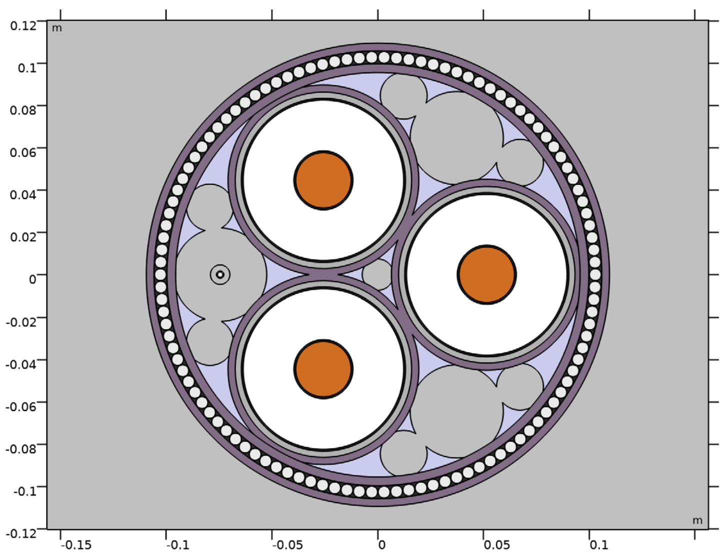


Figure 2: The cable's 2D geometry, including the three phases (with screens) and an armor.

Experienced COMSOL users with little or no interest in these topics — geometry, mesh, selections, and such — may choose to skip this part of the series and continue with one of the following tutorials. When you are new to COMSOL Multiphysics, however, it is worthwhile to take some time for this, as it will help you to familiarize yourself with the basics.

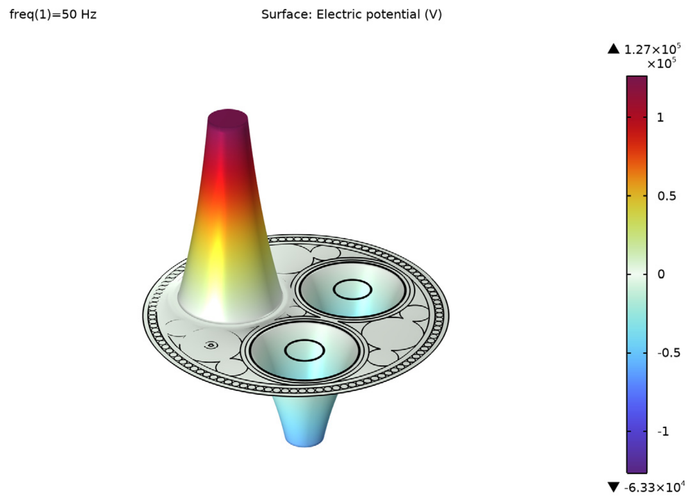


Figure 3: The real part of the electric potential distribution at phase $\varphi = 0°$.

The second chapter uses the 2D geometry from the introduction tutorial to investigate the cable's capacitive properties. It validates the assumption that for the capacitance and the charging currents, analytical approaches are sufficient (verification is included).

The influence of material properties, the cable's length and bonding types is discussed. The model justifies the approach chosen in subsequent tutorials in this series, most notably, the *Bonding Capacitive* and the *Inductive Effects* tutorials (chapters 3, and 4).

Included Topics

- Using the *Electric Currents* interface: **Current Conservation**, **Terminal**, **Ground**... Extracting lumped quantities using the terminal feature.
- Comparing a numerical model to an analytical one. Comparing displacement- and conduction currents. Checking the risk of electromagnetic breakdown. Checking the conditions on the floating end of a 10 km long cable with single-point bonding applied.
- Using expressions such as `exp(-120[deg]*j)` to set a phase difference in the frequency domain. Reflecting on the applicability of the used partial differential equations (whether it is safe to assume the in-plane electric fields are curl-free).
- Reflecting on the relevance of certain details in the model. Discussing the large material contrast (conductors and insulators) and how this affects numerical stability.

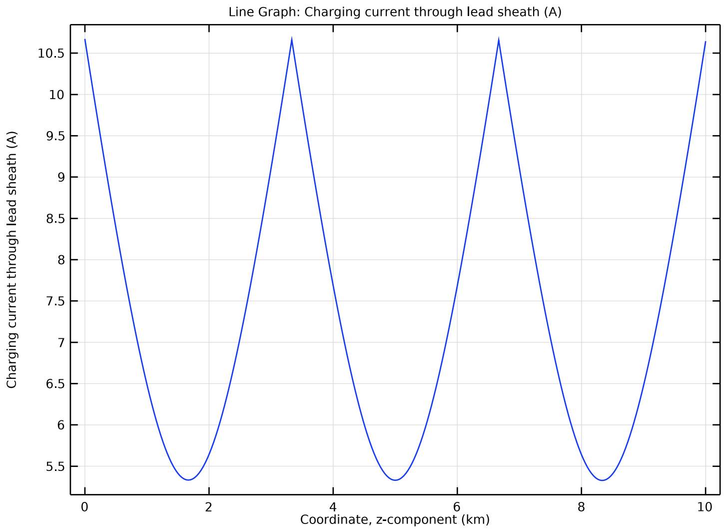


Figure 4: The charging current that builds up in the lead sheath (in case of cross bonding).

Based on the results from the *Capacitive Effects* tutorial, it is justified to neglect the capacitive coupling between the screens and consider one single isolated phase, together with its screen. As opposed to the *Capacitive*, *Inductive*, and *Thermal Effects* tutorials, this tutorial uses a 2D axisymmetric geometry representing the entire 10 km of cable.

For several bonding types, the build-up of charging currents and the corresponding losses in the screen are analyzed (verification is included). The model validates the assumption that the high phase potential induces a uniform charging current — one that barely depends on the screen potential — and so justifies the approach chosen in the *Capacitive* and *Inductive Effects* tutorials (chapters 2, and 4).

Included Topics

- Using the *Electric Currents* interface: **Current Conservation**, **Electric Potential**, **Ground**... Determining the total loss in a system by means of a volume integration.
- Using a 2D axisymmetric geometry. Using scaled coordinate systems to make extremely long geometries manageable. Using an anisotropic conductivity to stabilize a model.
- Discussing several bonding types: *single-point bonding*, *solid bonding*, and *cross bonding*, and reflecting on how they affect the build up of charging currents. Comparing a numerical model to analytical ones (one for each bonding type).

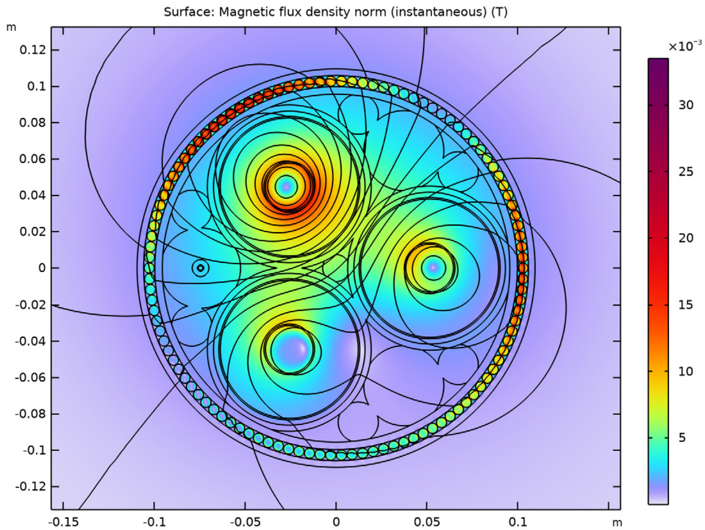


Figure 5: The instantaneous magnetic flux density norm when the armor twist is included.

Results from the *Capacitive Effects* and *Bonding Capacitive* tutorials show there is only a weak coupling between the inductive and capacitive phenomena in the cable. In addition to this, research [2, 6] suggests 2D and 2.5D inductive models are able to provide a good approximation of the cable's lumped quantities, and at only a fraction of the computational cost (as compared to long 3D twist models).

This justifies a 2D/2.5D inductive model that includes out-of-plane currents only. The model demonstrates methods suitable to approximate the armor twist, as well as certain *milliken conductor* designs. It serves as a basis and a reference for the *Thermal Effects* and the *Inductive Effects 3D* tutorials (chapters 6, and 8). Verification is included; the results are compared to the cable's official specifications.

Included Topics

- Using the *Magnetic Fields* interface: **Ampère's Law**, **Coil (group)**, **Magnetic Insulation**... Exciting a 2D/2.5D inductive model by means of the **Coil** feature.
- Using a **Coil group** to model the *armor twist* (useful for *cross bonding* too). Using the **Homogenized multiturn** setting to approximate a *milliken conductor*.
- Advanced postprocessing. Analyzing and verifying results, investigating the differences between the *Plain 2D*, *2.5D* and *2.5D + milliken* configurations. Reflecting on the difference between the AC- and DC resistance.

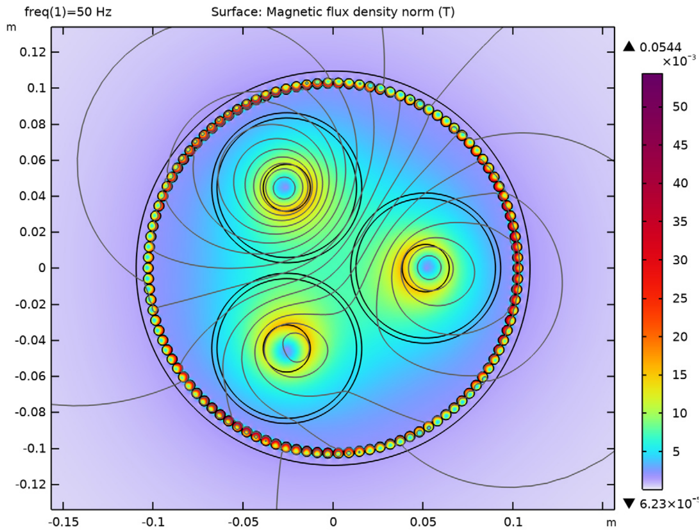


Figure 6: The magnetic flux density norm in a simplified geometry (in case of solid bonding).

In the *Inductive Effects* tutorial, the **Coil group** option is mentioned as a means to mimic the effects of cross bonding, but the exact extent of the validity of this approach is not shown. In order to investigate the different bonding types more closely, the fifth tutorial considers three sections of cable individually, represented by three separate Magnetic Fields interfaces.

The model uses a strongly simplified geometry. Even so, the results correspond very well to those from the *Inductive Effects* tutorial. This justifies both the simplified geometry in this tutorial, as well as the cross-bonding approach suggested in the *Inductive Effects* tutorial. Finally, as opposed to the other inductive models in this series, this one allows for investigating dissimilar section lengths.

Included Topics

- Using the *Magnetic Fields* interface: **Ampère's Law**, **Coil**, **Magnetic Insulation**... Connecting three finite element models in series, using an *Electrical Circuit*.
- Discussing several bonding types: *single-point bonding*, *solid bonding*, and *cross bonding*, and reflecting on how they affect the potential and currents in the screens.
- Validating if cross bonding can be modeled in 2D by means of a **Coil group**. Comparing the losses and lumped parameters to the detailed *plain 2D* model from the *Inductive Effects* tutorial and reflecting on what details are really important.

SUBMARINE CABLE 6 — THERMAL EFFECTS

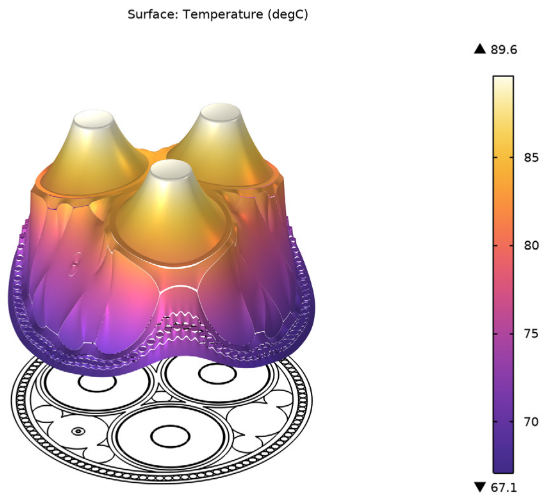


Figure 7: The temperature distribution inside the cable at sea bed temperature T_{ext} = 20°C.

This tutorial uses the *Inductive Effects* model from this series as a basis and adds thermal effects, including a temperature dependent conductivity (through linearized resistivity). It shows how to achieve a multiphysics coupling between electromagnetic fields and heat transfer, using the frequency-stationary study type (induction heating).

The influence of elevated temperatures on losses in the phases, screens, and armor is investigated (verification is included). The obtained temperature values are used in the *Inductive Effects 3D* tutorial, to apply a first-order temperature correction in 3D. Finally, the tutorial demonstrates how to match the resulting AC resistance to the one given by the IEC series of standards.

Included Topics

- Using the *Magnetic Fields* interface, the *Heat Transfer in Solids* interface and the **Electromagnetic Heating** multiphysics coupling to model induction heating.
- Using *linearized resistivity*. Investigating resistive- and magnetic losses. Reflecting on the difference between *current-driven* and *voltage-driven* conductors, and how the heat affects the electromagnetic properties of the cable. Investigating the difference between the AC- and DC resistance, and the validity of stationary-electric reasoning.
- Using a *global ODE* to set a phase conductivity such that the cable's AC resistance matches a certain specified (temperature dependent) value.

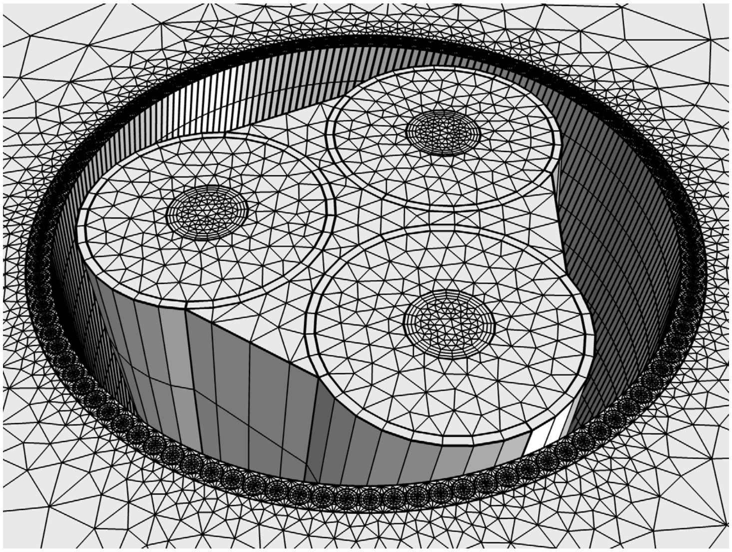


Figure 8: The helically twisted, swept mesh in the phases, the screens, and the armor.

The *Inductive Effects 3D* tutorial requires quite a lot of preparation in terms of geometry handling and meshing. In fact, this is what oftentimes represents a major part of the efforts spent on large 3D FEM models (twisted cable models in particular). In order not to overlook these important topics, they are addressed in a separate tutorial.

The tutorial is organized in four sections. The first part configures the camera settings (ensuring the geometry, mesh, and plots render as intended). Then, the geometry sequence is added, together with the selections. Finally, the mesh is constructed. Some postprocessing is done to investigate the quality of the mesh.

Included Topics

- Using advanced geometry handling and meshing to optimize a 3D mesh to produce a high level of detail, while keeping the *number of degrees of freedom* (DOFs) low.
- Using highly anisotropic meshes. Combining **Swept** meshes, **Free Tetrahedral** meshes, and **Boundary Layer** meshes. Reflecting on the importance of *conforming meshes* for periodicity planes. Demonstrating how mesh conformity can be enforced and checked.
- Improving the accuracy of the model by compensating for geometric effects like slanted cuts and the truncation of circles (linear elements will convert them to *polygons*).
- Using predefined selections of geometric entities to automate the work flow.

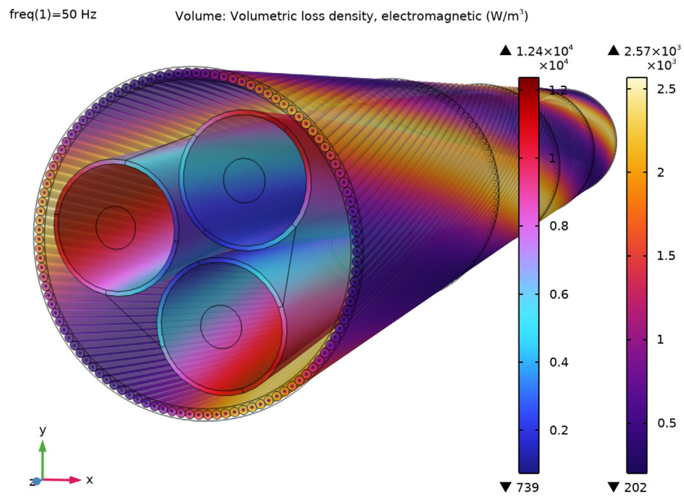


Figure 9: The volumetric loss density in the armor and the screens, for the case where a first-order temperature correction has been applied.

This last tutorial intends to give a "final answer" to 3D cable modeling. It has been developed with feedback from several experts from within the industry, and is on a par with the latest research (2020) when considering both *performance* and *level of detail*. Validation is included: The behavior of the models is analyzed within the context of refs. [2, 4, 5, and 6], and the cable's official specifications.

The tutorial is organized in five sections: First, the cable is modeled without twist and compared to the much more detailed 2D models. Then, the twist is added and the model is solved and verified a second time. A temperature correction is added in a third step and an experimental numerical stabilization method is demonstrated in a fourth step. The tutorial finishes by switching to a different kind of periodicity, reducing the size of the model a hundredfold while still providing a similar accuracy.

Included Topics

- The demonstration of a large (industrial scale) COMSOL model, including a highly optimized mesh, optimized solver settings, advanced postprocessing (with global evaluations, integrations, plots, and animations), and a detailed theoretical treatment of the observed physical phenomena — including references to recent numerical research.

- Using the *Magnetic Fields* interface in 3D: **Ampère's Law**, **Coil**, twisted **Periodicity**... Exiting a 3D inductive model by means of a **Coil** feature (with slanted cut), combined with a **Coil Geometry Analysis** preprocessing step.
- A demonstration of *twisted periodicity* and *short-twisted periodicity*, phase- and armor lay length, first- (and higher) order temperature corrections, and the analysis of resistive and magnetic losses in the phases, the screens, and the armor.
- A detailed comparison between the 3D twist models — with or without *linearized resistivity* included — and 2D/2.5D models (with or without induction heating).
- A detailed analysis of the longitudinal and transverse eddies forming in the armor wires, and the paths taken by the magnetic field lines as they encircle the phase conductors.
- A reflection on numerical stability and the demonstration of a *numerical stabilization method* that manages to improve the accuracy while keeping the solving time low.

References

1. International Electrotechnical Commission, *Electric cables – Calculation of the Current Rating*; IEC 60287; IEC Press: Geneva, Switzerland, 2006.

2. J.C. del-Pino-López, M. Hatlo, and P. Cruz-Romero, "On Simplified 3D Finite Element Simulations of Three-Core Armored Power Cables," *Energies* 2018, 11, 3081.

3. D. de Vries, "3D Cable Modeling in COMSOL Multiphysics®" *IEEE Spectrum*, 2020, https://spectrum.ieee.org/webinar/3d_cable_modeling_in_comsol_multiphysics.

4. M. Hatlo, E. Olsen, R. Stølan, and J. Karlstrand, "Accurate Analytic Formula for Calculation of Losses in Three-Core Submarine Cables," *Proc. 9th Int'l Conf. on Insulated Power Cables* (Jicable'15).

5. D. Willen, C. Thidemann, O. Thyrvin, D. Winkel, and V.M.R. Zermeno, "Fast Modelling of Armour Losses in 3D Validated by Measurements," *Proc. 10th Int'l Conf. on Insulated Power Cables* (Jicable'19).

6. J.J. Bremnes, G. Evenset, R. Stølan, "Power Loss and Inductance of Steel Armoured Multi-Core Cables: Comparison of IEC Values with 2.5D FEA Results and Measurements," (Cigré 2010).

Application Library path: `ACDC_Module/Tutorials,_Cables/submarine_cable_01_introduction`

Modeling Instructions (Introduction Tutorial)

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

NEW

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

MODEL WIZARD

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

In this case, we will not select any physics. This will be done in subsequent tutorials, depending on the analysis performed.

2 Click **Done**.

GEOMETRY 1

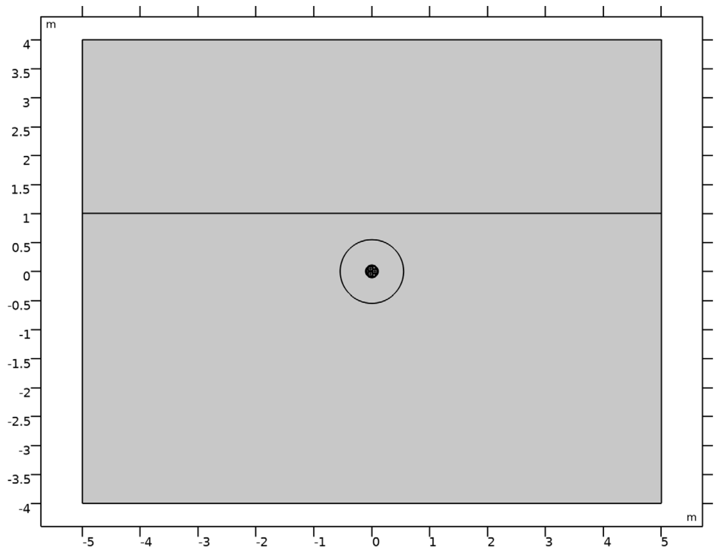
At this point, there are two options. One is to go to section Appendix — Modeling Instructions (Geometry) in this tutorial and follow the instructions for building the geometry there. In case you have limited interest in manually building geometries in COMSOL (because you intend to use CAD software for example), there is a second option. The completed geometry sequence can be inserted from a file:

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `submarine_cable_e_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

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



You now have built the geometry. This step coincides with the last instructions in section Appendix — Modeling Instructions (Geometry). You can continue by preparing some parameters and selections.

GLOBAL DEFINITIONS

Geometric Parameters 1

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

2 In the **Settings** window for **Parameters**, type `Geometric Parameters 1` in the **Label** text field.

If you have inserted the geometry sequence from a file, you might be surprised the table is already populated. This is because the parameters necessary for building the geometry are imported automatically.

Geometric Parameters 2

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Geometric Parameters 2` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

Eight items have been added to the table. The first is the main conductor's true cross section, `Acon`; 500 mm^2. Since these conductors are not actually solid but consist of a group of compacted strands, there is some space in between, and `Acon` is not identical to `pi*(Dcon/2)^2` (the surface area implicitly used in the geometry). This gives rise to the parameter `Ncon`. `Apbs` is pretty straightforward. The last five parameters are related to the cable's length.

DEFINITIONS

Next add some selections to use later on, when assigning physics features for instance. Fifteen domain selections have already been defined by the geometry sequence (as shown at the end of this tutorial). Based on the geometry-induced selections, you can add four *derived selections*: two unions, a difference, and an adjacent selection.

Phases

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

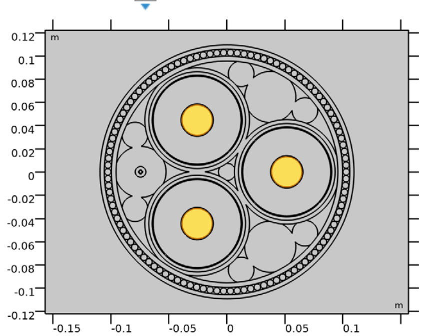
2 In the **Settings** window for **Union**, type `Phases` in the **Label** text field.

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

4 In the **Add** dialog box, in the **Selections to add** list, choose **Phase 1**, **Phase 2**, and **Phase 3**.

5 Click **OK**.

6 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Metals

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

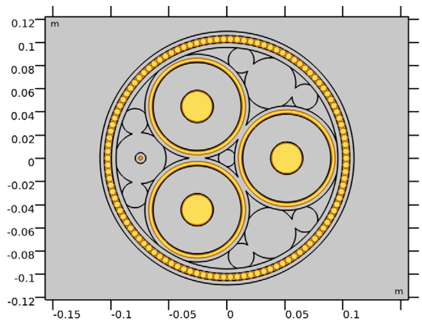
2 In the **Settings** window for **Union**, type `Metals` in the **Label** text field.

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

4 In the **Add** dialog box, in the **Selections to add** list, choose **Phases**, **Screens**, **Steel Helix (Fiber)**, and **Cable Armor**.

5 Click **OK**.

6 Click the **Zoom to Selection** button in the **Graphics** toolbar.



By unifying the phase, screen, fiber and armor selections, you have just found an efficient strategy to determine the **Metals** selection. Notice that selecting the armor wires individually by hand, would have been kind of tedious and error prone. That is why the geometry sequence is used to generate the selection for the armor wires (the same operator in the geometry sequence that builds the wires, also defines their selection). For more details on this, see section Appendix — Modeling Instructions (Geometry).

Insulators (External to Phase)

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

2 In the **Settings** window for **Difference**, type `Insulators (External to Phase)` in the **Label** text field.

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

4 In the **Add** dialog box, select **Cable Domains** in the **Selections to add** list.

5 Click **OK**.

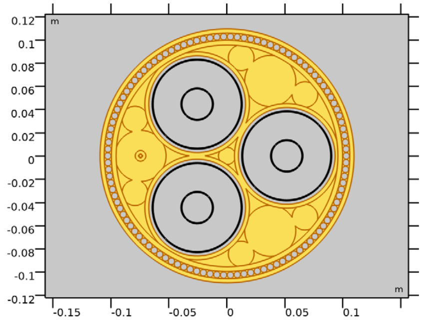
6 In the **Settings** window for **Difference**, locate the **Input Entities** section.

7 Under **Selections to subtract**, click **Add**.

8 In the **Add** dialog box, in the **Selections to subtract** list, choose **Metals**, **Semiconductive Compound**, and **Cross-Linked Polyethylene (XLPE)**.

9 Click **OK**.

10 Click the **Zoom to Selection** button in the **Graphics** toolbar.



Thermal Contact

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

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

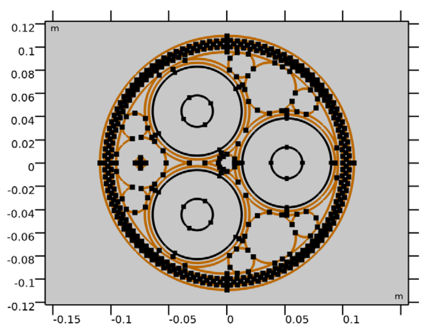
3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog box, select **Insulators (External to Phase)** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent**, locate the **Output Entities** section.

7 Select the **Interior boundaries** check box.



Now that the selections are finished, the materials will be next. To start with, you can modify the **View** to show the material colors. Then, the materials will be added, they will be given an appropriate label (if they do not already have it), a selection and an appearance. At the end of the instructions a table is listed that allows you to recheck the given settings.

View 1

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

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

3 Select the **Show material color and texture** check box.

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>Air**.

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

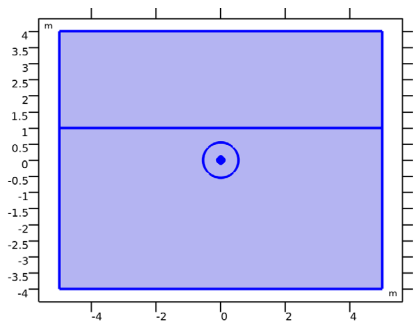
5 In the tree, select **Built-in>Water, liquid**.

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

MATERIALS

Air (mat1)

1 Click the **Zoom to Selection** button in the **Graphics** toolbar.



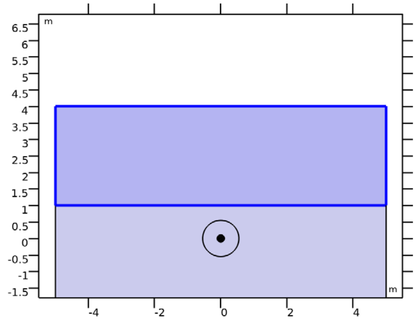
The air is applied to all domains. This is a good thing. You can use it as a default material and override it with other materials where applicable.

Water, liquid (mat2)

1 In the **Model Builder** window, click **Water, liquid (mat2)**.

2 Select Domain 2 only.

3 Click the **Zoom to Selection** button in the **Graphics** toolbar.



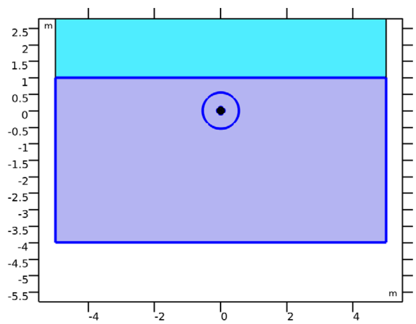
Gravel, saturated

1 In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Gravel, saturated` in the **Label** text field.

3 Select Domains 1 and 7 only.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



5 Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Relative Permeability**.

6 Click **Add to Material**.

7 Repeat these steps for **Electrical Conductivity**, **Relative Permittivity**, **Thermal Conductivity**, **Density**, and **Heat Capacity at Constant Pressure**.

8 Click to collapse the **Material Properties** section. Click to expand the **Appearance** section. From the **Material type** list, choose **Rock**.

Now, you have created a *blank material* with material properties for μ_r, σ, ε_r, k, ρ, and C_p; the material properties used for the electromagnetic and thermal equations respectively.

In a typical modeling scenario you would add the physics first however, before setting up the materials. The physics will then automatically request the material properties required to make the model solve. Setting up the materials in detail without the physics included takes a bit more effort, but it comes with the advantage that subsequent tutorials within this series will have most of the material properties preincluded. This allows the rest of the tutorial series to focus on the material properties that are of special importance to the particular kind of analysis performed.

You can save some time by taking this blank material as a template, and duplicating it a number of times.

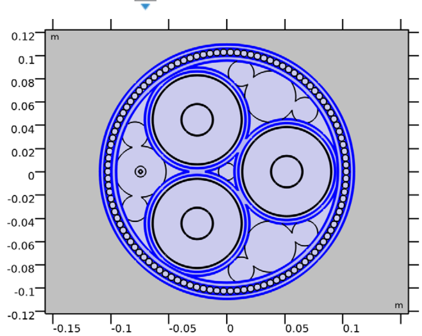
Polyethylene

1 Right-click **Gravel, saturated** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Polyethylene` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Polyethylene**.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



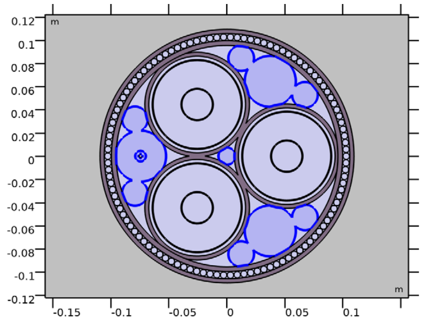
5 Click to expand the **Appearance** section. From the **Material type** list, choose **Oil**. (Admittedly, polyethylene is not “oil”, but the color seems suitable.)

Polypropylene

1 Right-click **Polyethylene** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Polypropylene` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Polypropylene**.



4 Click to expand the **Appearance** section. From the **Material type** list, choose **Plastic**.

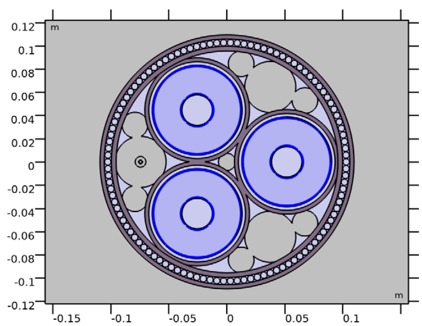
5 From the **Color** list, choose **Gray**.

Cross-linked polyethylene (XLPE)

1 Right-click **Polypropylene** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Cross-linked polyethylene (XLPE)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Cross-Linked Polyethylene (XLPE)**.



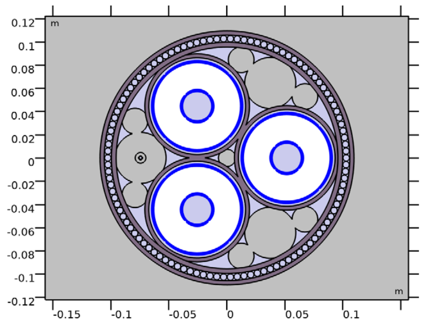
4 Click to expand the **Appearance** section. From the **Color** list, choose **White**.

Semiconductive compound

1 Right-click **Cross-linked polyethylene (XLPE)** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Semiconductive compound` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Semiconductive Compound**.



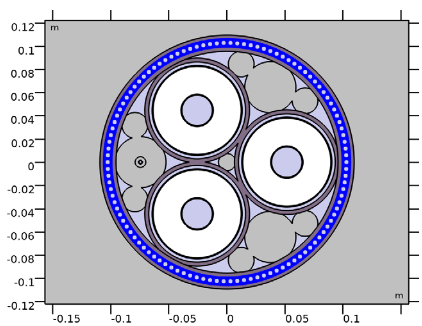
4 Click to expand the **Appearance** section. From the **Color** list, choose **Black**.

Bitumen compound

1 Right-click **Semiconductive compound** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Bitumen compound` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Bitumen Compound**.



The glass is readily available in the material library, you can add it as follows.

ADD MATERIAL

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

2 In the tree, select **Built-in>Silica glass**.

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

MATERIALS

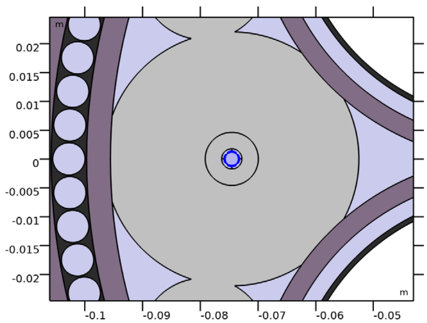
Silica glass (mat9)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silica glass (mat9)**.

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

3 From the **Selection** list, choose **Fiber Optic Core**.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.

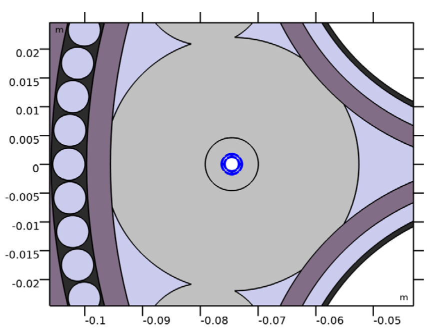


Stainless steel

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Bitumen compound (mat8)** and choose **Duplicate** (*make sure to duplicate the bitumen, not the glass*).

2 In the **Settings** window for **Material**, type `Stainless steel` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Steel Helix (Fiber)**.



4 Click to expand the **Appearance** section. From the **Material type** list, choose **Steel**.

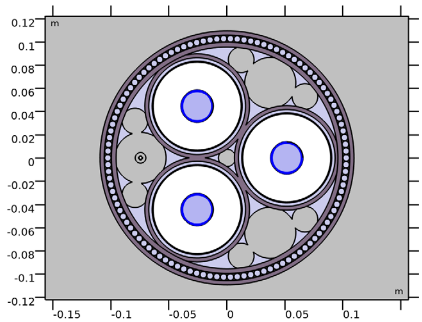
Copper

1 Right-click **Stainless steel** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Copper` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Phases**.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



For the phases, the screens and the armor, we will investigate the effect of elevated temperatures on the cable's electromagnetic properties in a fully coupled induction heating model (see the *Thermal Effects* tutorial). To this end, add *linearized resistivity* to the copper (and indirectly, to the lead and the steel).

5 Locate the **Material Properties** section. In the **Material properties** tree, select **Electromagnetic Models>Linearized Resistivity**.

6 Click **Add to Material**.

7 Click to collapse the **Material Properties** section. Click to expand the **Appearance** section. From the **Material type** list, choose **Copper**.

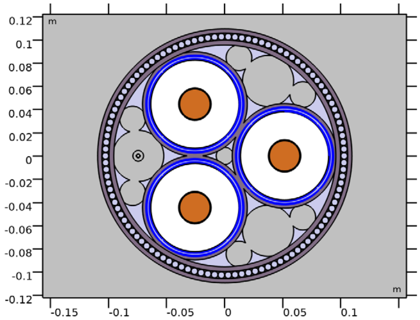
Lead

1 Right-click **Copper** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Lead` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Screens**.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



5 Click to expand the **Appearance** section. From the **Material type** list, choose **Lead**.

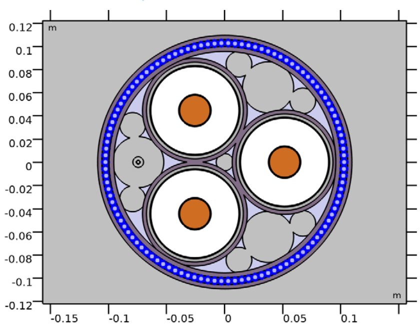
Galvanized steel

1 Right-click **Lead** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Galvanized steel` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Cable Armor**.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



5 Click to expand the **Appearance** section. From the **Material type** list, choose **Steel**.

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

Now that the materials are in place, it is a good practice to recheck their basic settings (as presented in the following table).

7 In the **Model Builder** window, under **Component 1 (comp1)>Materials**, check whether the following materials are present:

	Label	Selection	Appearance
mat1	`Air`	All domains (partially overridden)	Air
mat2	`Water, liquid`	Domain 2	Water
mat3	`Gravel, saturated`	Domains 1 and 7	Concrete
mat4	`Polyethylene`	Polyethylene	Plastic
mat5	`Polypropylene`	Polypropylene	Plastic
mat6	`Cross-linked polyethylene (XLPE)`	Cross-Linked Polyethylene (XLPE)	Steel
mat7	`Semiconductive compound`	Semiconductive Compound	Plastic
mat8	`Bitumen compound`	Bitumen Compound	Plastic
mat9	`Silica glass`	Fiber Optic Core	Custom
mat10	`Stainless steel`	Steel Helix (Fiber)	Steel
mat11	`Copper`	Phases	Copper
mat12	`Lead`	Screens	Lead
mat13	`Galvanized steel`	Cable Armor	Steel

Please see to it that the materials are actually put in **Component 1 (comp1)>Materials**, rather than **Global Definitions>Materials**.

MATERIALS

What remains are the material properties. The properties that should be added are listed in the following table. Please check all of them for the correct value, even the ones that are already filled in. Note that for cases like this, *a convenient option is to copy-paste the values directly from this *.pdf file to COMSOL.*

Furthermore, note that material properties marked with an "`x`" are not used at all (they may be present, but their value is irrelevant). Material properties marked with a "`-`", are deliberately left blank. They will be given a value in subsequent tutorials. Common practice is to include the unit when typing: "`1e-14[S/m]`", although strictly speaking that should not be necessary in this case. The entire tutorial series assumes standard SI units.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials**, add the following material properties:

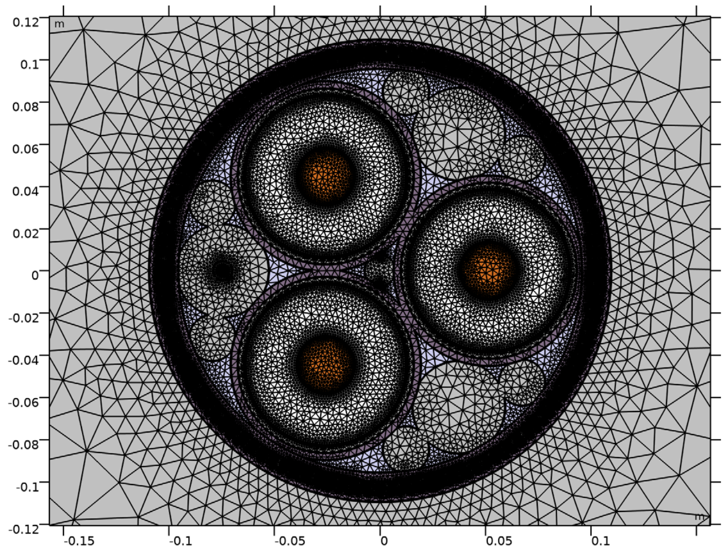
	mur	sigma [S/m]	epsr	k [W/(m*K)]	rho [kg/m^3]	Cp [J/(kg*K)]
mat1	1	1e-14	1	k(T)	rho(pA,T)	Cp(T)
mat2	x	x	x	k(T)	rho(T)	Cp(T)
mat3	1	1	28	1	2020	2512
mat4	1	1e-18	2.25	0.46	935	2302
mat5	1	1e-18	2.36	0.25	946	1920
mat6	1	1e-18	-	0.46	930	2302
mat7	1	2	2.25	10	1055	2405
mat8	1	3.2e-9	3.16	0.17	1062	1885
mat9	1	1e-14	3.75	1.38	2203	703
mat10	1	1.46e6	1	17.5	7920	475
mat11	-	-	1	-	8940	385
mat12	-	-	1	35.3	11340	127
mat13	-	-	1	58	7850	475

MESH 1

The last part of this tutorial consists of checking the mesh. COMSOL includes a whole range of tools for advanced meshing. In this case however, we are dealing with a 2D geometry of little to modest complexity: Even with a coarse mesh, the model will solve quite accurately. In the results, the difference between **Extra fine** and **Coarse** will show up somewhere in the 4th or 5th significant digit. *Feel free to check this in subsequent tutorials.*

So instead of spending time on intelligent mesh optimization schemes, we will choose a pragmatic approach and just use the element size setting **Normal** (the default).

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.



This will be our mesh for the *Capacitive*, *Inductive* and *Thermal Effects* tutorials. it is more than fine enough to capture skin effects and small geometric features.

You have now completed this tutorial. The result is a COMSOL Multiphysics file with geometry, materials and mesh, that will be used as a template for a multitude of models investigating capacitive, inductive and thermal properties. Subsequent tutorials will refer to this file as `submarine_cable_01_introduction.mph`. The next tutorial in this series will include a detailed capacitive analysis.

1 From the **File** menu, choose **Save As**.

2 Browse to a suitable folder and type the filename `submarine_cable_01_introduction.mph`.

The geometry in this tutorial series is based on parameters. This is not strictly necessary in COMSOL, rather it accommodates quick adjustments and keeps things consistent. Most of the parameters are based directly upon the mayor national or international standards (it is a standardized cable). This particular case is a 500 mm^2, 220 kV cable. By adjusting the parameters, other types can easily be made. In order to speed up this part of the tutorial, the parameters will be loaded from a file.

GLOBAL DEFINITIONS

Geometric Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, type `Geometric Parameters 1` in the **Label** text field.
3. Locate the **Parameters** section. Click **Load from File**.
4. Browse to the model's Application Libraries folder and double-click the file `submarine_cable_a_geom_parameters.txt`.

GEOMETRY 1

The geometry is constructed by means of a geometry sequence. The advantage of the sequence is that it not only saves the final result, but the entire process (the recipe, if you like). This way, it is easy to modify or parameterize the geometry. Additionally, some selections are being made. These selections will be used later on, when assigning material properties for instance.

You can start with the three phases. By setting their labels, and enabling the **Resulting objects selection**, you automatically set a domain selection that coincides with the shape you create. The selections will then propagate through the sequence (*when a shape is copied, its selection will cover both the original and the copy*). At the end of these modeling instructions, a blank material is created allowing you to check all geometry-induced selections once more.

Phase 1

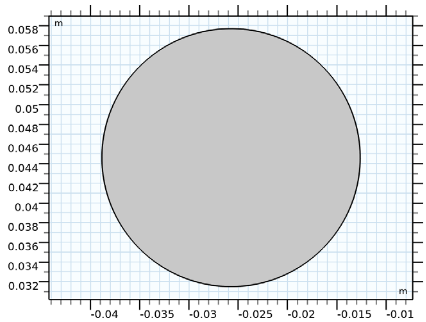
1. In the **Geometry** toolbar, click **Circle**.
2. In the **Settings** window for **Circle**, type `Phase 1` in the **Label** text field.
3. Locate the **Size and Shape** section. In the **Radius** text field, type `Dcon/2`.
4. Locate the **Position** section. In the **x** text field, type `-(Dpha/2)/sqrt(3)`.
5. In the **y** text field, type `Dpha/2`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `120`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 Click **Build Selected**.

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



Phase 2

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

2 In the **Settings** window for **Circle**, type `Phase 2` in the **Label** text field.

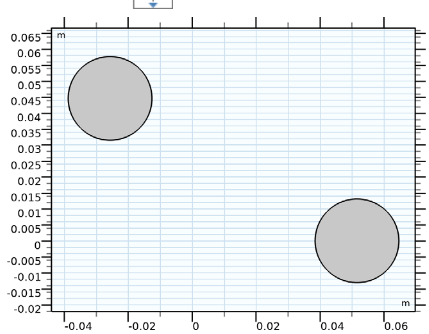
3 Locate the **Size and Shape** section. In the **Radius** text field, type `Dcon/2`.

4 Locate the **Position** section. In the **x** text field, type `Dpha/sqrt(3)`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

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



Phase 3

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

2 In the **Settings** window for **Circle**, type `Phase 3` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `Dcon/2`.

4 Locate the **Position** section. In the **x** text field, type `-(Dpha/2)/sqrt(3)`.

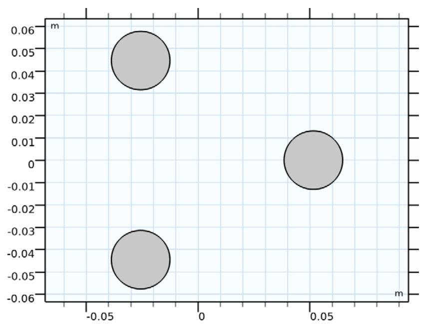
5 In the **y** text field, type `-Dpha/2`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-120`.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 Click **Build Selected**.

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



Circle 4 (c4)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

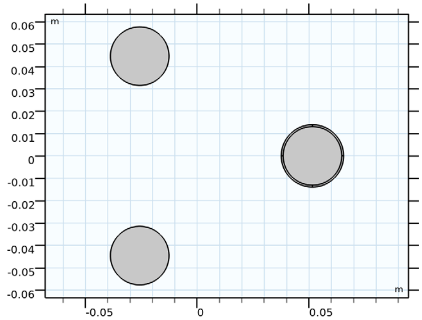
4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dcon/2+Tscc`.

5 Locate the **Position** section. In the **x** text field, type `Dpha/sqrt(3)`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Tscc

7 Click **Build Selected**.



Circle 5 (c5)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dins/2-Tscc`.

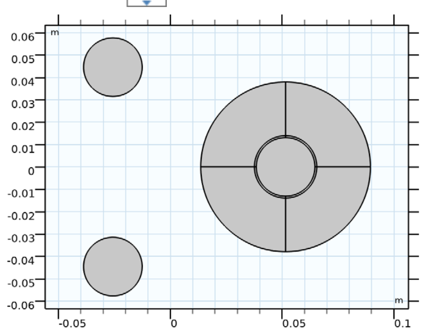
5 Locate the **Position** section. In the **x** text field, type `Dpha/sqrt(3)`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Tins

7 Click **Build Selected**.

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



Circle 6 (c6)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

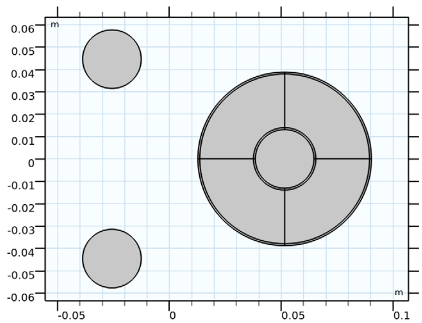
4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dins/2`.

5 Locate the **Position** section. In the **x** text field, type `Dpha/sqrt(3)`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Tscc

7 Click **Build Selected**.



Circle 7 (c7)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

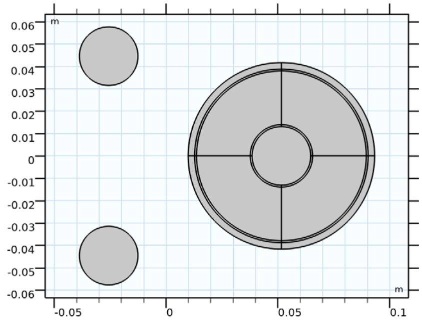
4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dins/2+Tpbs`.

5 Locate the **Position** section. In the **x** text field, type `Dpha/sqrt(3)`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	Tpbs

7 Click **Build Selected**.

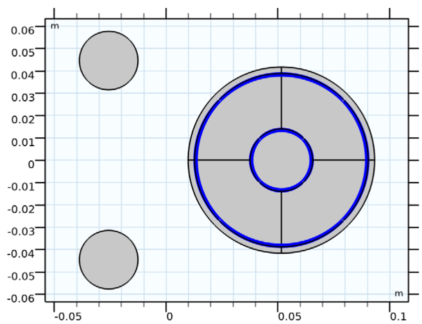


Semiconductive Compound

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

2 In the **Settings** window for **Union**, type `Semiconductive Compound` in the **Label** text field.

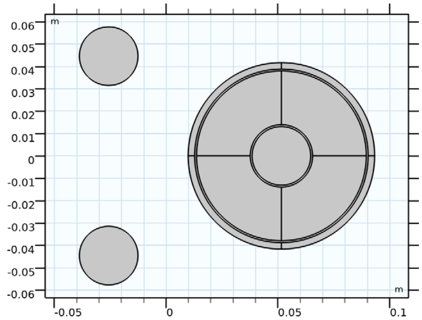
3 Select the objects **c4** and **c6** only.



4 Locate the **Union** section. Clear the **Keep interior boundaries** check box.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

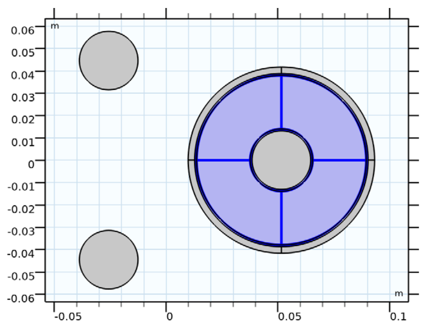


Cross-Linked Polyethylene (XLPE)

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

2 In the **Settings** window for **Union**, type `Cross-Linked Polyethylene (XLPE)` in the **Label** text field.

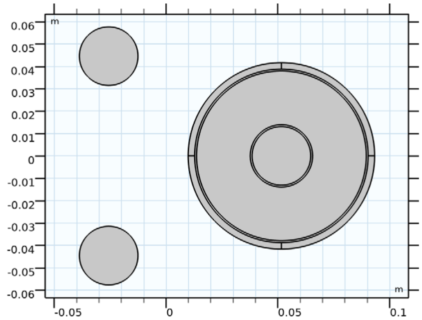
3 Select the object **c5** only.



4 Locate the **Union** section. Clear the **Keep interior boundaries** check box.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

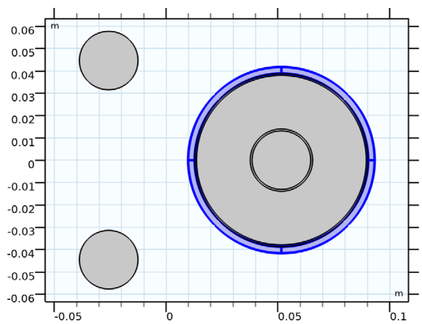


Screens

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

2 In the **Settings** window for **Union**, type `Screens` in the **Label** text field.

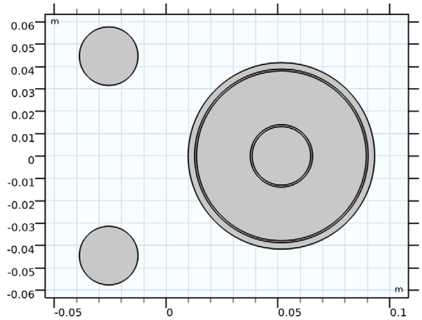
3 Select the object **c7** only.



4 Locate the **Union** section. Clear the **Keep interior boundaries** check box.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.



Next, will be the polyethylene parts. In order to simplify the geometry and accommodate meshing, a deliberate overlap between the circles is created (using a parameter called `mfil`). This avoids having infinitely narrow regions between circles touching. The sequence can be adjusted if necessary, to support a more complex combination of polymers.

Circle 8 (c8)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

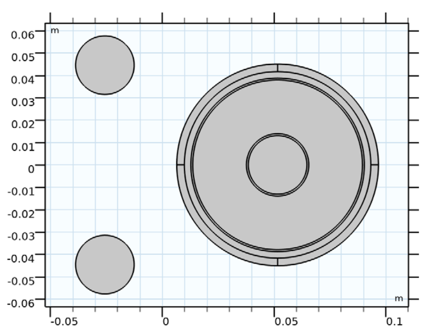
4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dpha/2+mfil`.

5 Locate the **Position** section. In the **x** text field, type `Dpha/sqrt(3)`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Tpe+mfil`

7 Click **Build Selected**.



Circular Arc 1 (ca1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Radius** section.

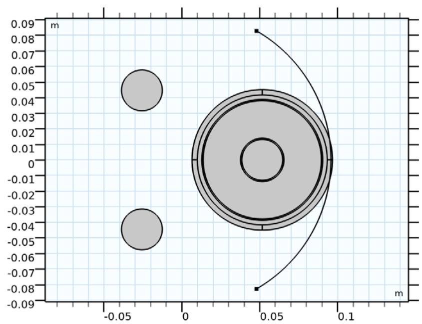
3 In the **Radius** text field, type `Dpha3/2-mfil`.

4 Locate the **Angles** section. In the **Start angle** text field, type `-60`.

5 In the **End angle** text field, type `60`.

6 Click **Build Selected**.

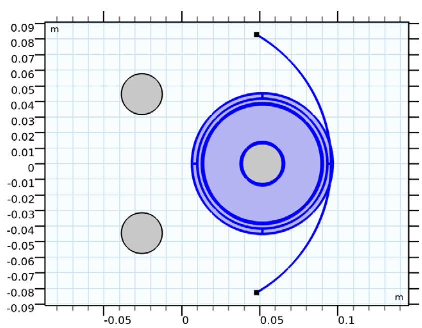
7 Click the **Zoom Extents** button in the **Graphics** toolbar.



Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **c8**, **ca1**, **uni1**, **uni2**, and **uni3** only.

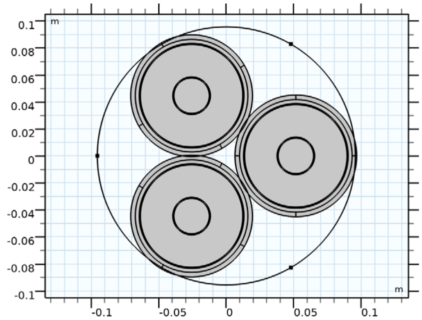


3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `0[deg], 120[deg], 240[deg]`.

5 Click **Build Selected**.

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



Here, three different angles are used to create three copies of the same input objects (*while propagating their selections*).

Circle 9 (c9)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dcab/2`.

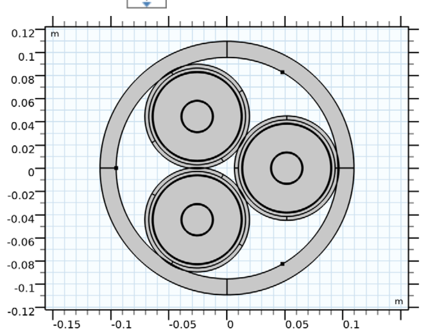
5 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Dcab/2-(Dpha3/2-mfil)`

Note that for longer expressions like this one, *the easiest way to go, is to copy-paste them directly from this *.pdf file to COMSOL.*

6 Click **Build Selected**.

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



Circle 10 (c10)

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

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

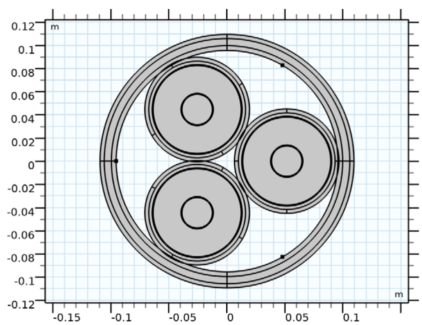
3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `Darm/2+Tarm/2+marm`.

5 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Tarm+2*marm`

6 Click **Build Selected**.

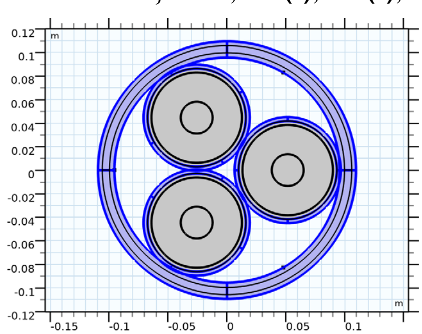


Polyethylene

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

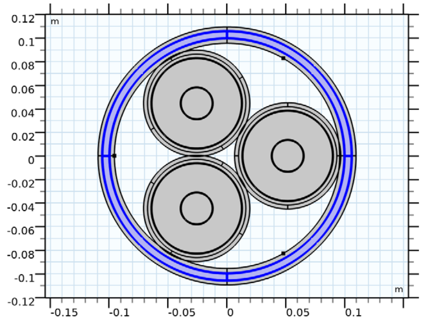
2 In the **Settings** window for **Difference**, type `Polyethylene` in the **Label** text field.

3 Select the objects **c9**, **rot1(1)**, **rot1(2)**, and **rot1(3)** only.



4 Locate the **Difference** section. Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

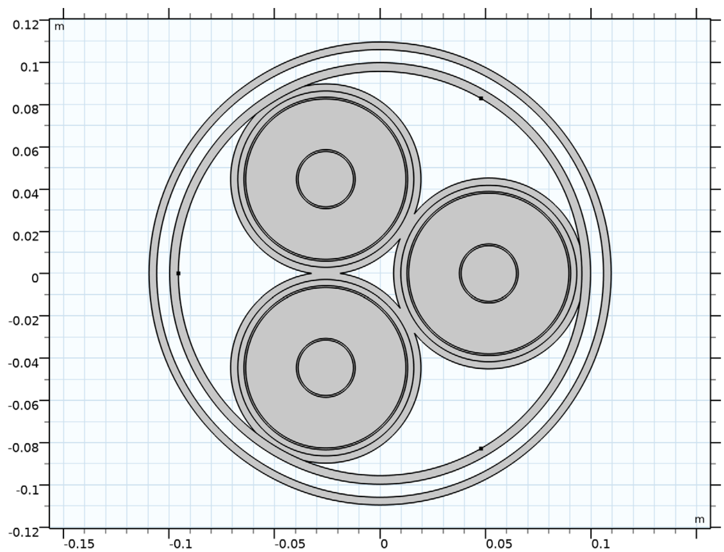
5 Select the object **c10** only.



6 Clear the **Keep interior boundaries** check box.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 Click **Build Selected**.



You have now created the polyethylene parts. What is more, you have created a *domain selection* that refers to these parts. Next, let us have a look at the polypropylene parts.

Circle 11 (c11)

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

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

3 In the **Radius** text field, type `Dpha*0.241335+mfil`.

4 Locate the **Position** section. In the **x** text field, type `Dpha*-0.836015`.

5 Click **Build Selected**.

Circle 12 (c12)

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

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

3 In the **Radius** text field, type `Dpha*0.118595+mfil`.

4 Locate the **Position** section. In the **x** text field, type `Dpha*-0.890205`.

5 In the **y** text field, type `Dpha*-0.355815`.

6 Click **Build Selected**.

Circle 13 (c13)

1 Right-click **Circle 12 (c12)** and choose **Duplicate**.

2 In the **Settings** window for **Circle**, locate the **Position** section.

3 In the **y** text field, type `Dpha*0.355815`, that is, *remove the minus sign*.

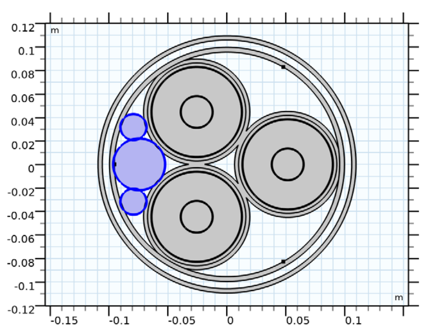
4 Click **Build Selected**.

Perhaps you are wondering about the numbers used here. These are the result of geometrical expressions that make the circles fit (apart from the overlap margin used). The size and position of the circles scale with `Dpha`, and the expressions have been simplified accordingly.

Rotate 2 (rot2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the objects **c11**, **c12**, and **c13** only.



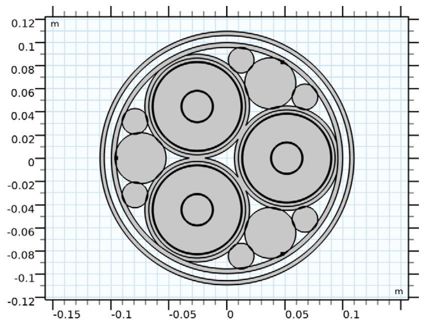
3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `0[deg], 120[deg], 240[deg]`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 From the **Show in physics** list, choose **Off**.

7 Click **Build Selected**.



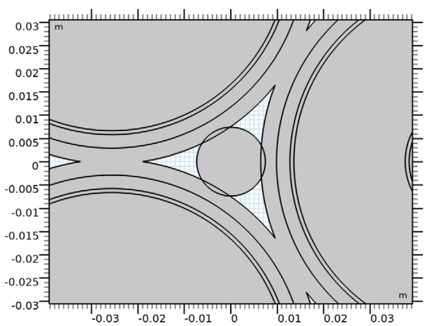
Circle 14 (c14)

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

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

3 In the **Radius** text field, type `Dpha*(1/sqrt(3)-1/2)+mfil`.

4 Click **Build Selected**.



The selection for **Rotate 2** is not shown in the physics, because it is only meaningful within the geometry sequence itself (it is used in **Difference 2**). The polypropylene domains are now created by taking the result from **Rotate 2** and **c14**, and subtracting a hole for the fiber, together with a copy of the polyethylene.

Circle 15 (c15)

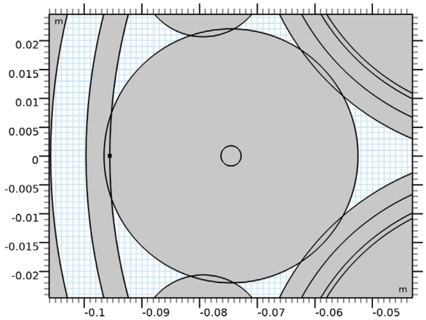
1 In the **Geometry** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `Dfic/2+Tfih`.

4 Locate the **Position** section. In the **x** text field, type `Dpha*-0.836015`.

5 Click **Build Selected**.

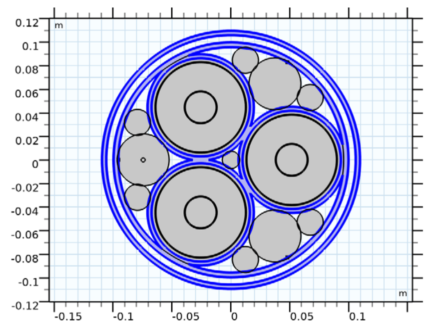


Copy 1 (copy1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 In the **Settings** window for **Copy**, locate the **Input** section.

3 From the **Input objects** list, choose **Polyethylene**.



4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

5 From the **Show in physics** list, choose **Off**.

6 Click **Build Selected**.

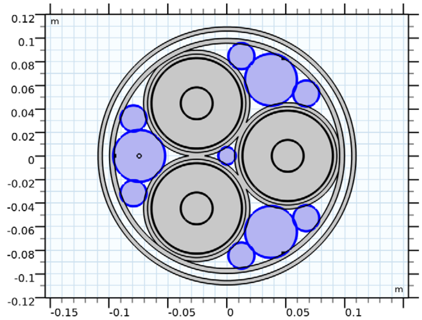
Polypropylene

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

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

3 Locate the **Difference** section. From the **Objects to add** list, choose **Rotate 2**.

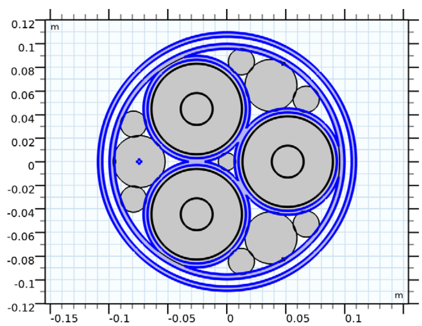
4 In addition to **Rotate 2**, select the object **c14**; the *center filler* domain.



5 Find the **Objects to subtract** subsection. Click to select the **Activate Selection** toggle button.

6 From the **Objects to subtract** list, choose **Copy 1**.

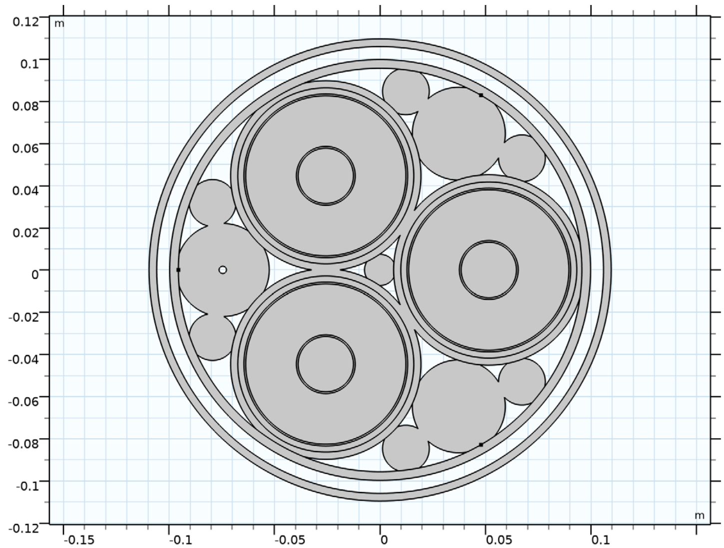
7 In addition to **Copy 1**, select the object **c15**; the small hole for the fiber on the left.



8 Clear the **Keep interior boundaries** check box.

9 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

10 Click **Build Selected**.



Now that the polypropylene filler material is complete, let us add some details for the fiber. Although a single circle object would suffice, the helix and the core are made as separate objects. This will result in two separate geometry-induced material selections.

Circle 16 (c16)

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

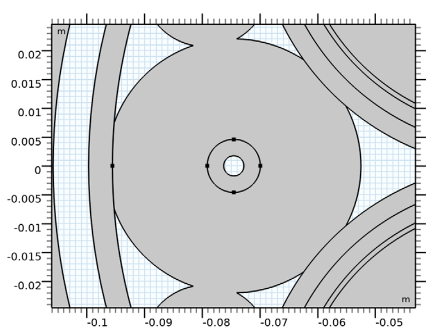
2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dfib/2`.

5 Locate the **Position** section. In the **x** text field, type `Dpha*-0.836015`.

6 Click **Build Selected**.



Steel Helix (Fiber)

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

2 In the **Settings** window for **Circle**, type `Steel Helix (Fiber)` in the **Label** text field.

3 Locate the **Object Type** section. From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `Dfic/2+Tfih`.

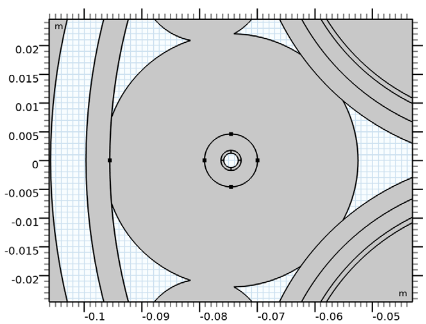
5 Locate the **Position** section. In the **x** text field, type `Dpha*-0.836015`.

6 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
`Layer 1`	`Tfih`

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 Click **Build Selected**.



Fiber Optic Core

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

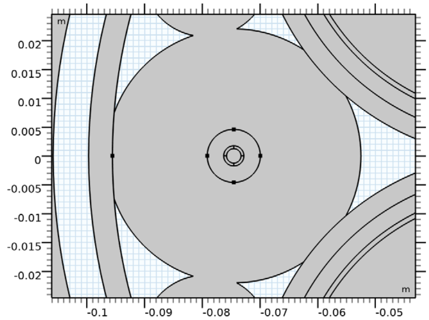
2 In the **Settings** window for **Circle**, type `Fiber Optic Core` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `Dfic/2`.

4 Locate the **Position** section. In the **x** text field, type `Dpha*-0.836015`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.



Next, are the armor wires. For the armor, we have the restriction that there should be an *integer amount* of wires in the cable's circumference. This number, `Narm`, is limited by the armor wire's diameter, and the overall size of the cable. Using the **Rotate** transform and the `range()` operator, the wires are equally distributed in the interval [0,360] degrees. In-between the armor wires, some space is left open to allow for electric insulation and a *low permeability gap* for the magnetic flux density.

Cable Armor

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

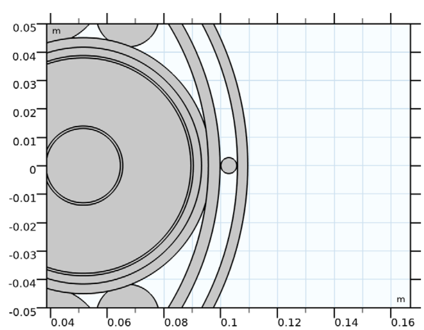
2 In the **Settings** window for **Circle**, type `Cable Armor` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `Tarm/2`.

4 Locate the **Position** section. In the **x** text field, type `Darm/2`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

6 Click **Build Selected**.

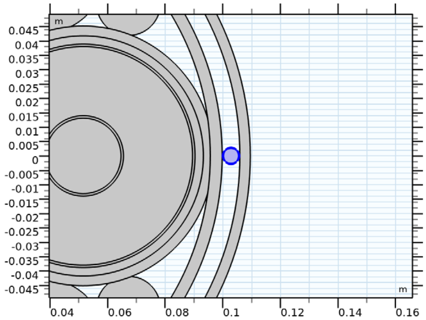


Rotate 3 (rot3)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 In the **Settings** window for **Rotate**, locate the **Input** section.

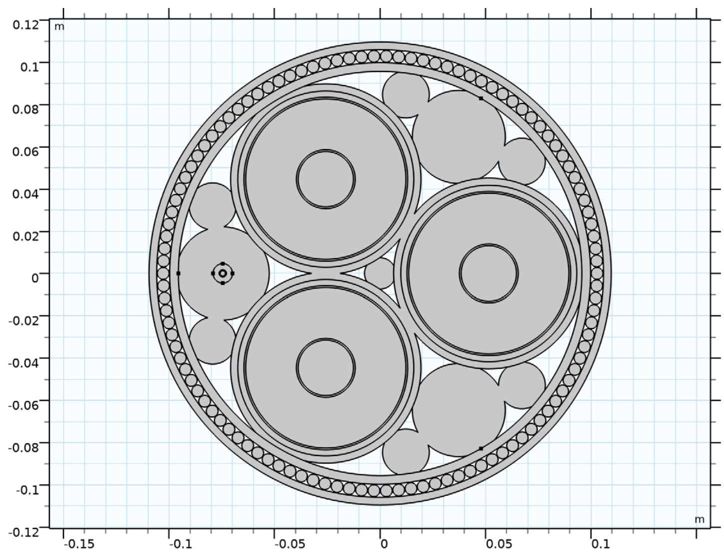
3 From the **Input objects** list, choose **Cable Armor**.



4 In the **Settings** window for **Rotate**, locate the **Rotation** section.

5 In the **Angle** text field, type `360[deg]*range(1/Narm,1/Narm,1)`.

6 Click **Build Selected**.



This finishes the cable armor. The domain for the bitumen compound is now created by duplicating object **c10**, and subtracting from that, a copy of the armor.

Circle 20 (c20)

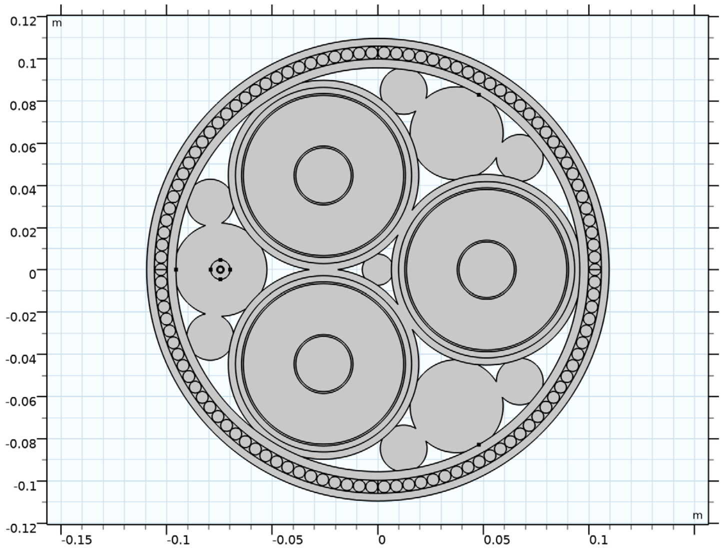
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** right-click **Circle 10 (c10)** and choose **Duplicate**.

2 In the **Settings** window for **Circle**, locate the **Selections of Resulting Entities** section.

3 Select the **Resulting objects selection** check box.

4 From the **Show in physics** list, choose **Off**.

5 Click **Build Selected**.

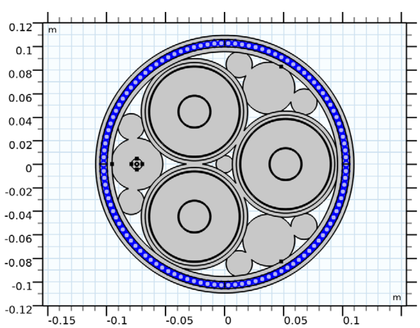


Copy 2 (copy2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Copy**.

2 In the **Settings** window for **Copy**, locate the **Input** section.

3 From the **Input objects** list, choose **Cable Armor**.



4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

5 From the **Show in physics** list, choose **Off**.

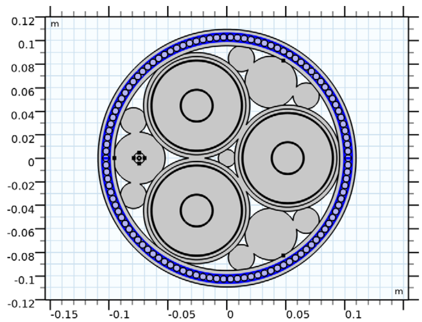
6 Click **Build Selected**.

Bitumen Compound

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

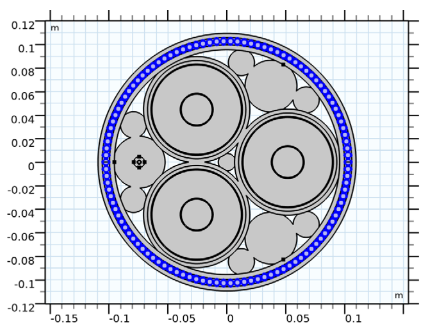
2 In the **Settings** window for **Difference**, type `Bitumen Compound` in the **Label** text field.

3 Locate the **Difference** section. From the **Objects to add** list, choose **Circle 20**.



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

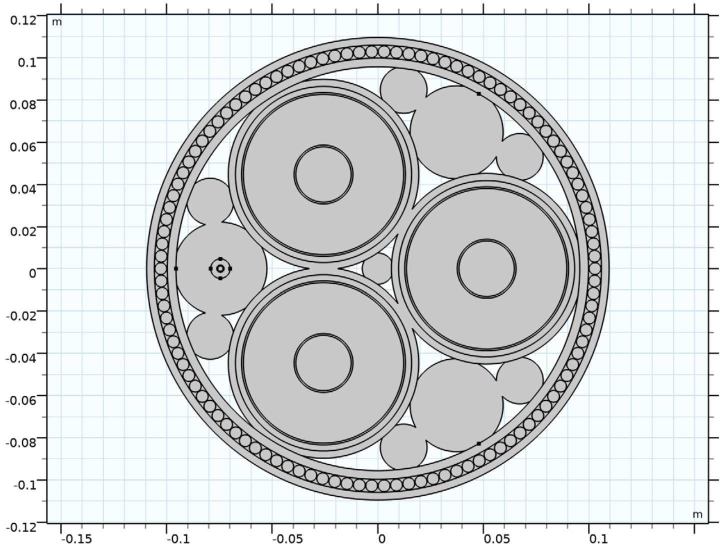
5 From the **Objects to subtract** list, choose **Copy 2**.



6 Clear the **Keep interior boundaries** check box.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

8 Click **Build Selected**.



Now that the bitumen is finished, add a circle that covers the whole cable. This is primarily to get the **Cable Domains** selection.

Cable Domains

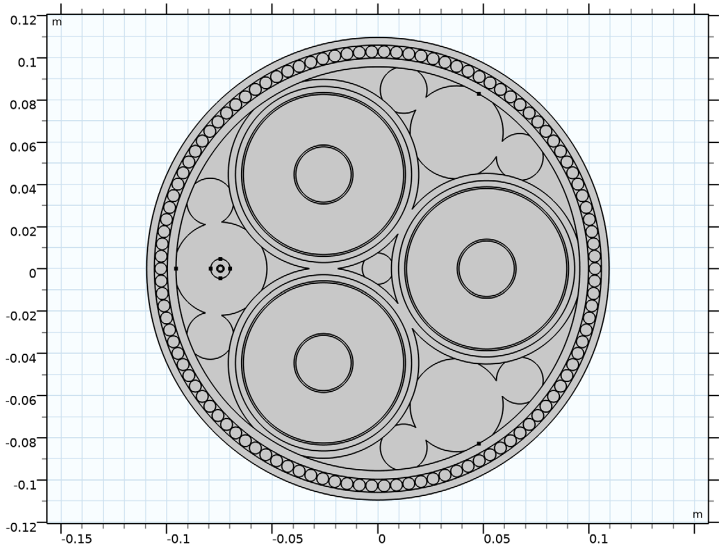
1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, type `Cable Domains` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `Dcab/2`.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

5 Click **Build Selected**.



What remains are the domains exterior to the cable. Part of this, is the circle that covers the electromagnetic domains. Outside this circle, you will only be solving for heat transfer (electromagnetic fields are assumed to have dropped to zero). Rectangular domains are added for the sea bed and the water.

Electromagnetic Domains

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

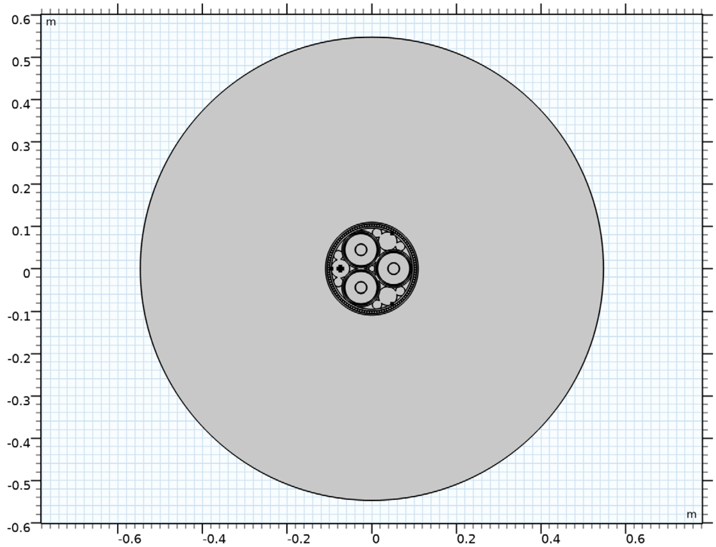
2 In the **Settings** window for **Circle**, type `Electromagnetic Domains` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Radius** text field, type `5*Dcab/2`.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

5 Click **Build Selected**.

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



Thermal Domains

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

2 In the **Settings** window for **Rectangle**, type `Thermal Domains` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `10[m]`.

4 In the **Height** text field, type `8[m]`.

5 Locate the **Position** section. In the **x** text field, type `-5[m]`.

6 In the **y** text field, type `-4[m]`.

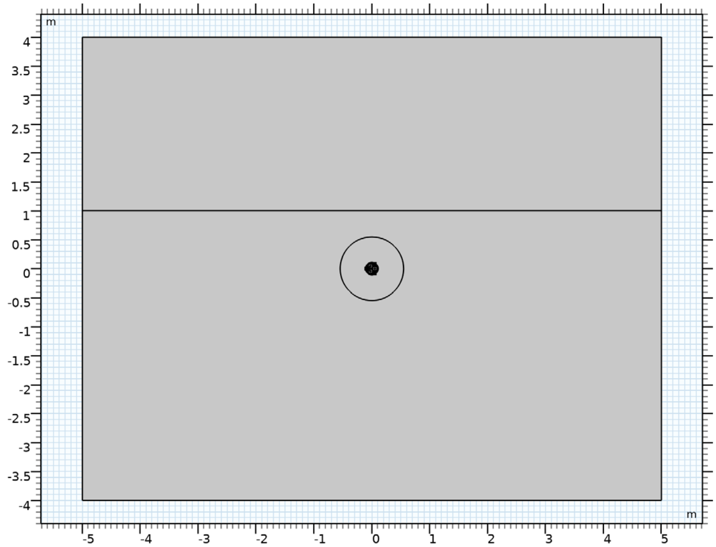
7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	5[m]

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.

9 Click **Build Selected**.

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



This completes the geometry. Notice that the size of the rectangle does not depend on the cable's parameters. A wise choice for the size of the thermal domains strongly depends on the thermal properties and conditions. For a discussion on this, see the *Thermal Effects* tutorial.

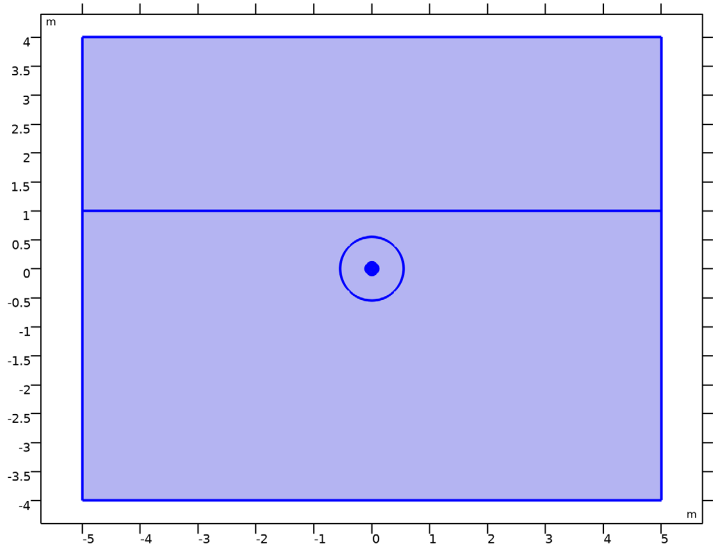
Before finishing, feel free to check that the geometry-induced selections exists, and are according to your expectations. You can do so by adding a *blank material*.

MATERIALS

Material 1 (mat1)

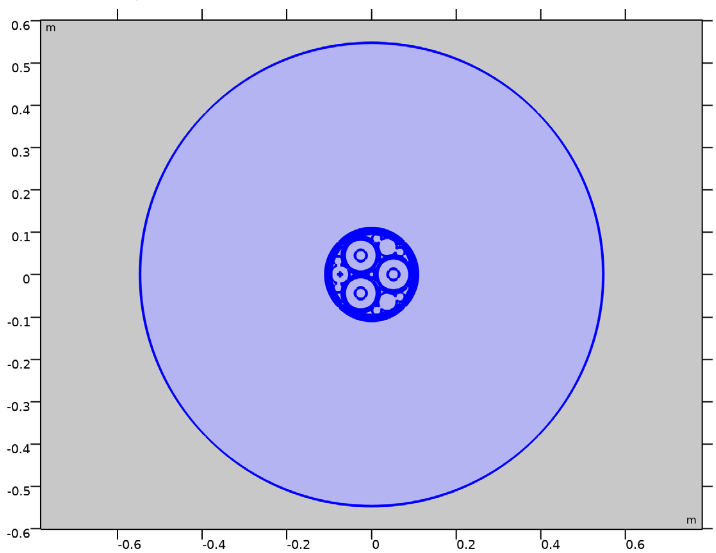
1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Thermal Domains**.

4 Click the **Zoom to Selection** button in the **Graphics** toolbar.



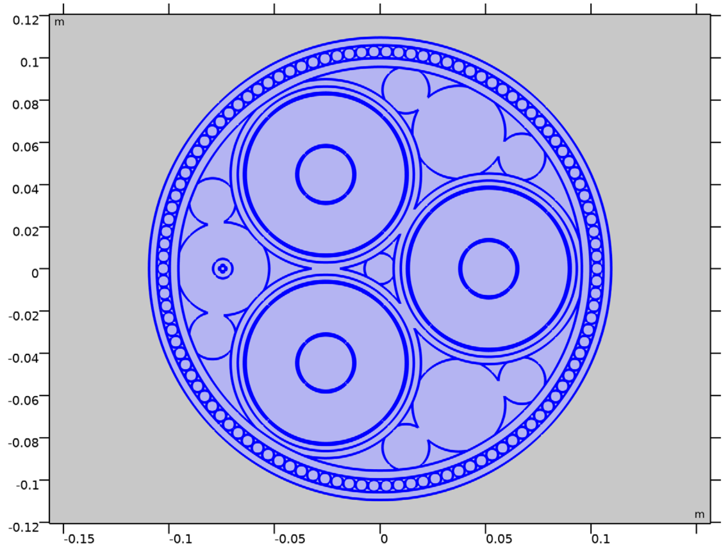
5 From the **Selection** list, choose **Electromagnetic Domains**.

6 Click the **Zoom to Selection** button in the **Graphics** toolbar.

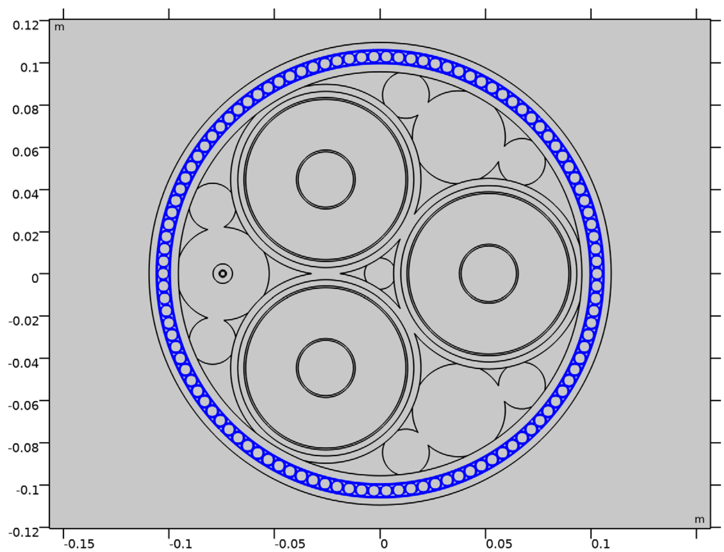


7 From the **Selection** list, choose **Cable Domains**.

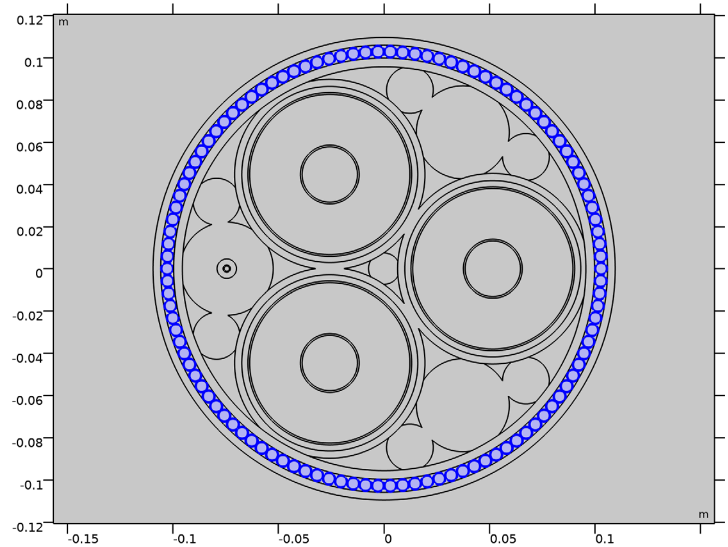
8 Click the **Zoom to Selection** button in the **Graphics** toolbar.



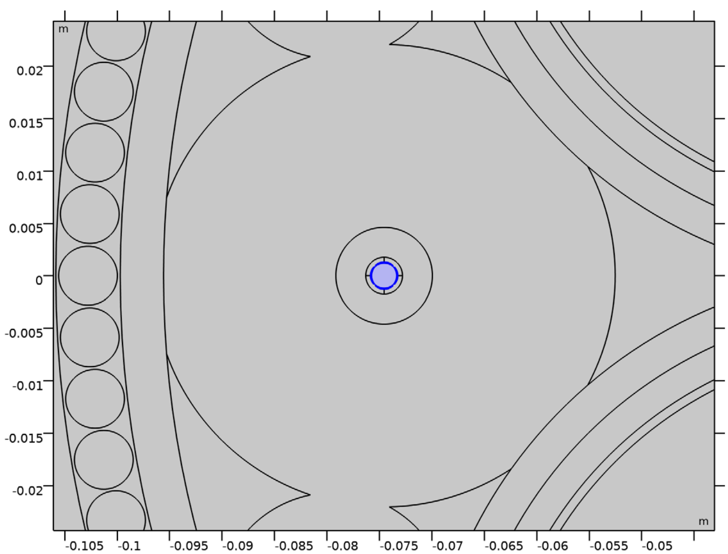
9 From the **Selection** list, choose **Bitumen Compound**.



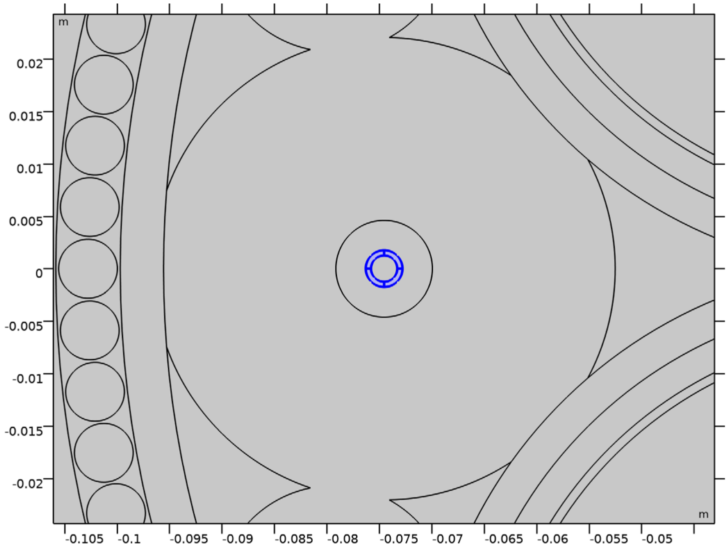
10 From the **Selection** list, choose **Cable Armor**.



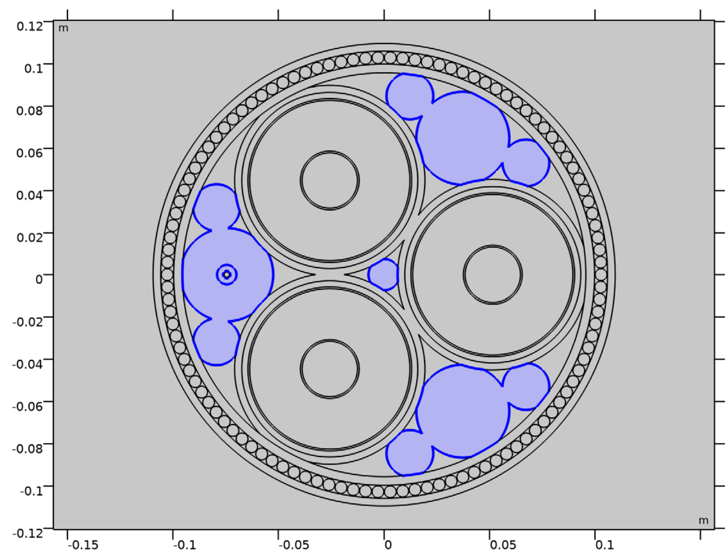
11 From the **Selection** list, choose **Fiber Optic Core**.



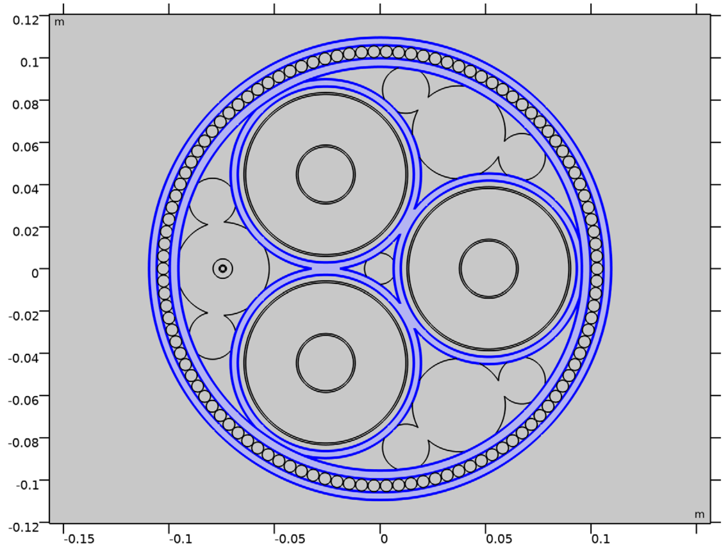
12 From the **Selection** list, choose **Steel Helix (Fiber)**.



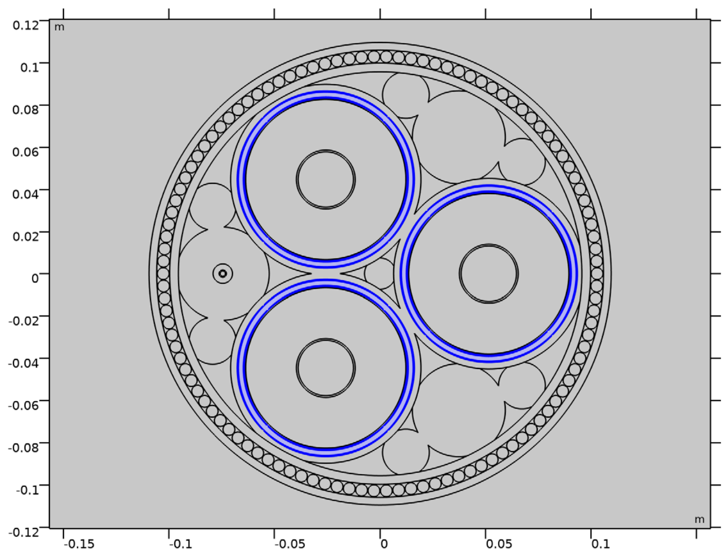
13 From the **Selection** list, choose **Polypropylene**.



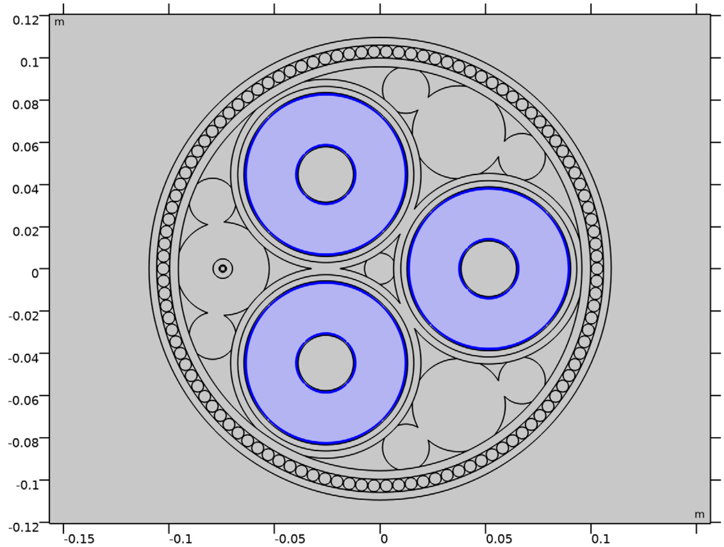
14 From the **Selection** list, choose **Polyethylene**.



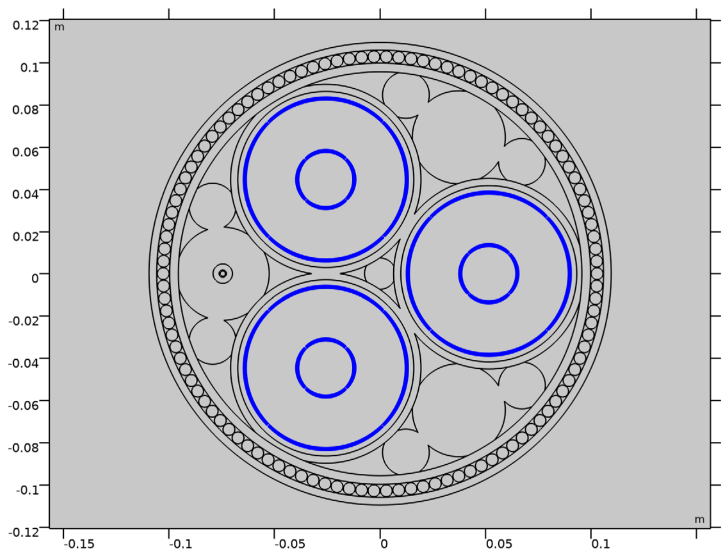
15 From the **Selection** list, choose **Screens**.



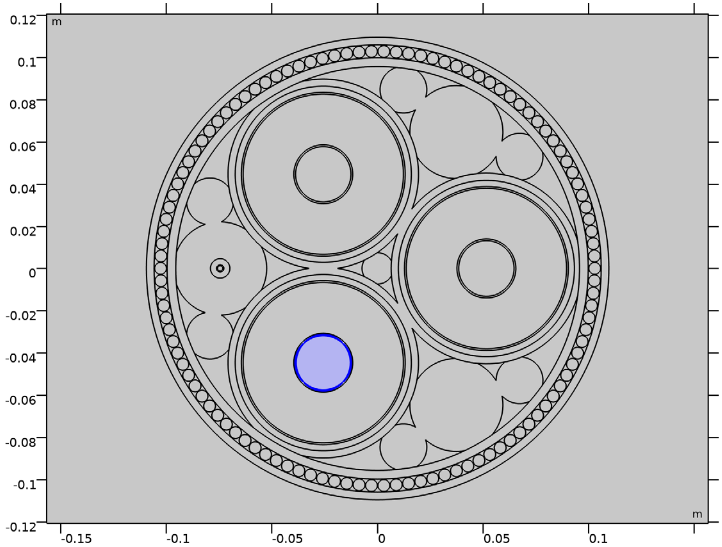
16 From the **Selection** list, choose **Cross-Linked Polyethylene (XLPE)**.



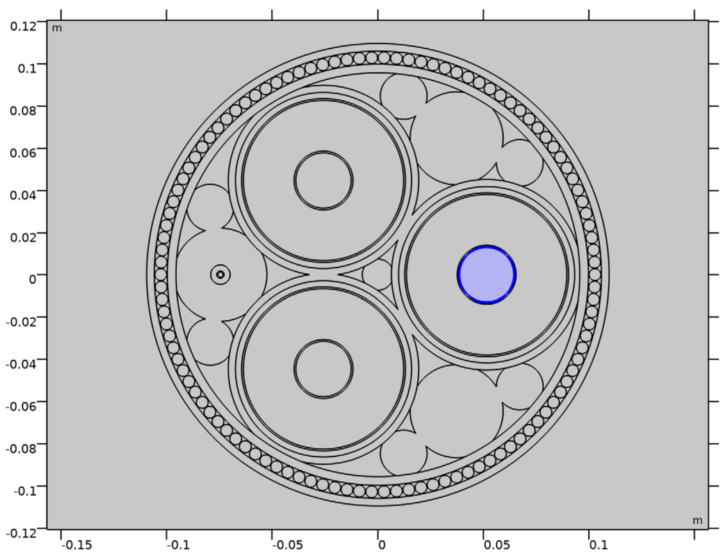
17 From the **Selection** list, choose **Semiconductive Compound**.



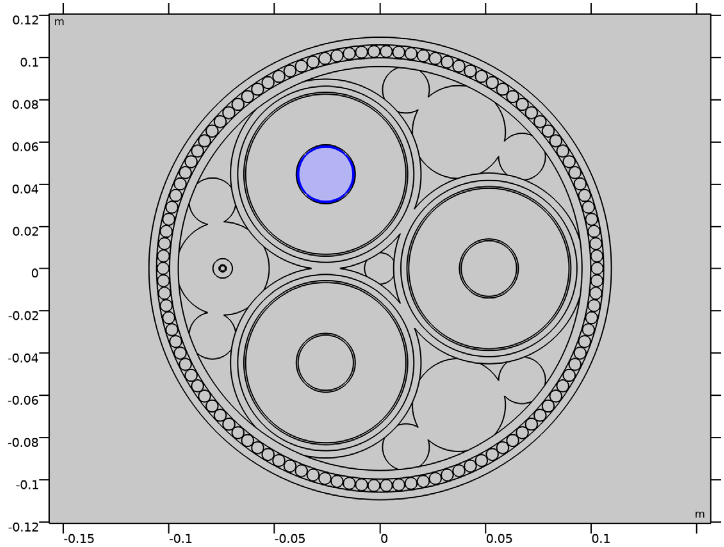
18 From the **Selection** list, choose **Phase 3**.



19 From the **Selection** list, choose **Phase 2**.



20 From the **Selection** list, choose **Phase 1**.



If anything seems out of order, please retrace your steps. Alternatively, check out the completed geometry sequence included in this tutorial series as a reference (`submarine_cable_e_geom_sequence.mph`).

21 Right-click **Material 1 (mat1)** and choose **Delete**.