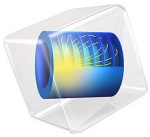

Parameterized Woven Carbon Fibers Geometry

This is a template MPH-file containing the geometry for the model Anisotropic Heat Transfer through Woven Carbon Fibers. For a description of that model, including detailed step-by-step instructions showing how to build it, see Anisotropic Heat Transfer Through Woven Carbon Fibers.

Application Library path: `COMSOL_Multiphysics/Heat_Transfer/carbon_fibers_geom`

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 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
w	4	4	Number of loops
n	2*w-1	7	Number of fibers per axis
l	0.2	0.2	Length of loop
amp	l/4	0.05	Amplitude of loop
q	1	1	Parameter controlling cross section

PART 1

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts> 3D Part**.

2 In the **Settings** window for **Part**, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

4 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Default expression	Value	Description
a	`0.2[mm]`	0.02 cm	`Cross section semiaxis`

Local Parameters

1 Right-click **Part 1** and choose **Local Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
b	a	0.02 cm	`Cross section semiaxis`

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click  **Work Plane**.

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

3 From the **Plane** list, choose **yz-plane**.

4 Click  **Go to Plane Geometry**.

Work Plane 1 (wp1)>Ellipse 1 (e1)

1 In the **Work Plane** toolbar, click  **Ellipse**.

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

3 In the **a-semiaxis** text field, type `a`.

4 In the **b-semiaxis** text field, type `b`.

5 Right-click **Part 1** and choose **Rename**.

6 In the **Rename Part** dialog box, type `Elliptical cross section` in the **New label** text field.

7 Click **OK**.

PART 2

1 In the **Model Builder** window, under **Global Definitions** right-click **Geometry Parts** and choose **3D Part**.

2 In the **Settings** window for **Part**, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

4 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Default expression	Value	Description
a	`0.6[mm]`	0.06 cm	

Local Parameters

1 Right-click **Part 2** and choose **Local Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
b	a/2	0.03 cm	

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click **Work Plane**.

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

3 From the **Plane** list, choose **yz-plane**.

4 Click **Go to Plane Geometry**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

1 In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type a.

4 In the **Height** text field, type b.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

6 In the **Work Plane** toolbar, click **Build All**.

7 Right-click **Part 2** and choose **Rename**.

8 In the **Rename Part** dialog box, type `Rectangular cross section` in the **New label** text field.

9 Click **OK**.

GLOBAL DEFINITIONS

In the **Model Builder** window, collapse the **Global Definitions** node.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

If 1 (if1)

1 In the **Geometry** toolbar, click  **Programming** and choose **If + End If**.

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

3 In the **Condition** text field, type `q==1`.

Elliptical cross section 1 (pi1)

In the **Geometry** toolbar, click  **Part Instance** and choose **Elliptical cross section**.

Else 1 (else1)

In the **Model Builder** window, right-click **Geometry 1** and choose **Programming>Else**.

Rectangular cross section 1 (pi2)

1 In the **Geometry** toolbar, click  **Part Instance** and choose **Rectangular cross section**.

2 In the **Settings** window for **Part Instance**, click  **Build All Objects**.

Box Selection 1 (boxsel1)

1 In the **Geometry** toolbar, click  **Selections** and choose **Box Selection**.

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

3 From the **Level** list, choose **Boundary**.

4 Locate the **Resulting Selection** section. Clear the **Keep selection** check box.

5 Find the **Cumulative selection** subsection. Click **New**.

6 In the **New Cumulative Selection** dialog box, type `Cross section` in the **Name** text field.

7 Click **OK**.

Work Plane 1 (wp1)

1 In the **Geometry** toolbar, click  **Work Plane**.

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

3 Find the **Cumulative selection** subsection. Click **New**.

4 In the **New Cumulative Selection** dialog box, type `Path` in the **Name** text field.

5 Click **OK**.

6 In the **Settings** window for **Work Plane**, click  **Go to Plane Geometry**.

Work Plane 1 (wp1)>Cubic Bézier 1 (cb1)

1 In the **Work Plane** toolbar, click  **More Primitives** and choose **Cubic Bézier**.

2 In the **Settings** window for **Cubic Bézier**, locate the **Control Points** section.

3 In row **2**, set **xw** to `1/4`.

4 In row **3**, set **xw** to `l/4`.

5 In row **4**, set **xw** to `l/2`.

6 In row **3**, set **yw** to `amp`.

7 In row **4**, set **yw** to `amp`.

Work Plane 1 (wp1)>Cubic Bézier 2 (cb2)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Cubic Bézier**.

2 In the **Settings** window for **Cubic Bézier**, locate the **Control Points** section.

3 In row **1**, set **xw** to `l/2`.

4 In row **2**, set **xw** to `3*l/4`.

5 In row **3**, set **xw** to `3*l/4`.

6 In row **4**, set **xw** to `l`.

7 In row **1**, set **yw** to `amp`.

8 In row **2**, set **yw** to `amp`.

Work Plane 1 (wp1)>Convert to Curve 1 (ccur1)

1 In the **Work Plane** toolbar, click **Conversions** and choose **Convert to Curve**.

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Convert to Curve**, click **Build Selected**.

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

Work Plane 1 (wp1)>Array 1 (arr1)

1 In the **Work Plane** toolbar, click **Transforms** and choose **Array**.

2 Select the object **ccur1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **xw size** text field, type `w`.

5 Locate the **Displacement** section. In the **xw** text field, type `l`.

6 In the **Work Plane** toolbar, click **Build All**.

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

Sweep 1 (swe1)

1 Right-click **Geometry 1** and choose **Sweep**.

2 In the **Settings** window for **Sweep**, locate the **Cross Section** section.

3 From the **Entities to sweep** list, choose **Cross section**.

4 Locate the **Spine Curve** section. Click to select the **Activate Selection** toggle button for **Edges to follow**.

5 From the **Edges to follow** list, choose **Path**.

6 Locate the **Input Object Handling** section. Clear the **Keep input objects** check box.

7 Click **Build All Objects**.

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

Extrude 1 (ext1)

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

2 In the **Settings** window for **Extrude**, locate the **General** section.

3 From the **Extrude from** list, choose **Faces**.

4 On the object **swe1**, select Boundary 1 only.

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

Distances (cm)
1/2

6 Right-click **Extrude 1 (ext1)** and choose **Duplicate**.

Extrude 2 (ext2)

1 In the **Model Builder** window, click **Extrude 2 (ext2)**.

2 In the **Settings** window for **Extrude**, locate the **General** section.

3 Click to select the **Activate Selection** toggle button for **Input faces**.

4 On the object **ext1**, select Boundary 46 only.

5 Click **Build All Objects**.

Copy 1 (copy1)

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

2 Select the object **ext2** only.

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

4 In the **z** text field, type `1/2`.

Array 1 (arr1)

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

2 Select the object **ext2** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **z size** text field, type `ceil(n/2)`.

5 Locate the **Displacement** section. In the **z** text field, type `1`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Fiber array 1` in the **Name** text field.

8 Click **OK**.

Array 2 (arr2)

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

2 Select the object **copy1** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **z size** text field, type `floor(n/2)`.

5 Locate the **Displacement** section. In the **z** text field, type `1`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog box, type `Fiber array 2` in the **Name** text field.

8 Click **OK**.

9 In the **Settings** window for **Array**, click  **Build Selected**.

Mirror 1 (mir1)

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

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

3 From the **Input objects** list, choose **Fiber array 2**.

4 Locate the **Point on Plane of Reflection** section. In the **y** text field, type `1/8`.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `1`.

6 In the **z** text field, type `0`.

7 Click  **Build Selected**.

Rotate 1 (rot1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

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

4 Select the **Keep input objects** check box.

5 Locate the **Rotation** section. In the **Angle** text field, type `90`.

6 Locate the **Point on Axis of Rotation** section. In the **x** text field, type `w*1/2`.

7 In the **z** text field, type `n/2*l/2-l/4`.

8 Locate the **Rotation** section. From the **Axis type** list, choose **y-axis**.

9 Click **Build All Objects**.

Mirror 2 (mir2)

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

2 Select the objects **arr1(1,1,1)**, **arr1(1,1,2)**, **arr1(1,1,3)**, **arr1(1,1,4)**, **mir1(1)**, **mir1(2)**, and **mir1(3)** only.

3 In the **Settings** window for **Mirror**, locate the **Point on Plane of Reflection** section.

4 In the **y** text field, type `l/8`.

5 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `1`.

6 In the **z** text field, type `0`.

7 Click **Build Selected**.

Union 1 (uni1)

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

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Union**, locate the **Selections of Resulting Entities** section.

4 Find the **Cumulative selection** subsection. Click **New**.

5 In the **New Cumulative Selection** dialog box, type `Fibers` in the **Name** text field.

6 Click **OK**.

Block 1 (blk1)

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

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

3 In the **Width** text field, type `l*(w+1)`.

4 In the **Depth** text field, type `5/8*l`.

5 In the **Height** text field, type `l*(w+1)`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 In the **x** text field, type `(n+1)*l/4`.

8 In the **y** text field, type `l/8`.

9 In the **z** text field, type `(n-1)*l/4`.

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

Layer name	Thickness (cm)
Layer 1	1/2

11 Find the **Layer position** subsection. Select the **Left** check box.

12 Select the **Right** check box.

13 Select the **Top** check box.

Union 2 (uni2)

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

2 Click in the **Graphics** window and then press Ctrl+A to select both objects.

3 In the **Settings** window for **Union**, click  **Build All Objects**.

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

GEOMETRY 1

In the **Model Builder** window, collapse the **Component 1 (comp1)>Geometry 1** node.

DEFINITIONS

Infinite Element Domains

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

2 Right-click **Definitions** and choose **Selections>Explicit**.

3 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.

4 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

5 Click  **Paste Selection**.

6 In the **Paste Selection** dialog box, type `1-11, 13, 21, 29, 46-47, 55, 63, 80-81, 89, 97, 114-115, 123, 131, 140-149` in the **Selection** text field.

7 Click **OK**.

8 In the **Settings** window for **Explicit**, type `Infinite Element Domains` in the **Label** text field.

Core Domains

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

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

3 Under **Selections to invert**, click  **Add**.

4 In the **Add** dialog box, select **Infinite Element Domains** in the **Selections to invert** list.

5 Click **OK**.

6 In the **Settings** window for **Complement**, type `Core Domains` in the **Label** text field.

Core Boundaries

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

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

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

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

5 Click **OK**.

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

Epoxy (Core)

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

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

3 Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **Core 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, select **Fibers** in the **Selections to subtract** list.

9 Click **OK**.

10 In the **Settings** window for **Difference**, type `Epoxy (Core)` in the **Label** text field.

Fibers (Core)

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

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

3 Under **Selections to add**, click **Add**.

4 In the **Add** dialog box, select **Fibers** 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, select **Infinite Element Domains** in the **Selections to subtract** list.

9 Click **OK**.

10 In the **Settings** window for **Difference**, type `Fibers (Core)` in the **Label** text field.

Fiber Boundaries (Core)

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

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

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

4 In the **Add** dialog box, select **Fibers (Core)** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent**, type `Fiber Boundaries (Core)` in the **Label** text field.

Heat Source Boundary

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

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

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

4 Select Boundary 53 only.

5 In the **Label** text field, type `Heat Source Boundary`.

Cooling Boundaries

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

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

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

4 Select Boundaries 53 and 95 only.

5 In the **Label** text field, type `Cooling Boundaries`.

Plane 1

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

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

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

4 Select the **Group by continuous tangent** check box.

5 Select Boundaries 9, 49, 52, 55, 57, 59, 62, 65, 74, 77, 80, 83, 122, 212, 294, 384, 466, 556, 638, and 707 only.

6 In the **Label** text field, type `Plane 1`.

Plane 2

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

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

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

4 Select Boundary 6 only.

5 Select the **Group by continuous tangent** check box.

6 Select Boundaries 6, 54, 102, 206, 274, 378, 446, 550, 618, 699, 702, 704, 705, 709, 712, 715, 724, 727, 730, and 733 only.

7 In the **Label** text field, type `Plane 2`.

Intersection 1

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

2 In the **Settings** window for **Intersection**, locate the **Geometric Entity Level** section.

3 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 **Core Boundaries** and **Fiber Boundaries (Core)**.

6 Click **OK**.

Inlets

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

2 In the **Settings** window for **Difference**, locate the **Geometric Entity Level** section.

3 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 **Intersection 1** 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 **Plane 1** in the **Selections to subtract** list.

10 Click **OK**.

11 In the **Settings** window for **Difference**, type `Inlets` in the **Label** text field.

Outlets

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

2 In the **Settings** window for **Difference**, locate the **Geometric Entity Level** section.

3 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 **Intersection 1** 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 **Plane 2** in the **Selections to subtract** list.

10 Click **OK**.

11 In the **Settings** window for **Difference**, type `Outlets` in the **Label** text field.

Epoxy Boundaries (Core)

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

2 In the **Settings** window for **Difference**, locate the **Geometric Entity Level** section.

3 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 **Core 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, in the **Selections to subtract** list, choose **Inlets** and **Outlets**.

10 Click **OK**.

11 In the **Settings** window for **Difference**, type `Epoxy Boundaries (Core)` in the **Label** text field.

Temperature Boundaries

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

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

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

4 Select the **Group by continuous tangent** check box.

5 Select Boundaries 1, 3, 4, 7, 10, 11, 14, 17, 26, 29, 32, 35, 51, 58, 99, 123, 205, 213, 271, 295, 377, 385, 443, 467, 549, 557, 615, 639, 701, 708, and 747–756 only.

6 In the **Label** text field, type `Temperature Boundaries`.

Fiber Walls

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

2 In the **Settings** window for **Difference**, locate the **Geometric Entity Level** section.

3 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 **Fiber Boundaries (Core)** 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, in the **Selections to subtract** list, choose **Inlets** and **Outlets**.

10 Click **OK**.

11 In the **Settings** window for **Difference**, type `Fiber Walls` in the **Label** text field.

Inlet Edges

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

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

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

4 Under **Input selections**, click + **Add**.

5 In the **Add** dialog box, select **Inlets** in the **Input selections** list.

6 Click **OK**.

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

8 From the **Geometric entity level** list, choose **Adjacent edges**.

9 In the **Label** text field, type `Inlet Edges`.

Fibers (Infinite Element Domain)

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

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

3 Under **Selections to intersect**, click + **Add**.

4 In the **Add** dialog box, in the **Selections to intersect** list, choose **Infinite Element Domains** and **Fibers**.

5 Click **OK**.

6 In the **Settings** window for **Intersection**, type `Fibers (Infinite Element Domain)` in the **Label** text field.

COMPONENT 1 (COMP1)

In the **Model Builder** window, collapse the **Component 1 (comp1)** node.