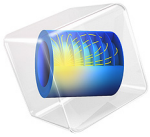


Optimization of a Waveguide Iris Bandpass Filter — Transformation Version

Introduction

RF filters are designed to allow a certain frequency band to pass, while reflecting frequencies outside the band. The design of such filters can be a tedious manual task, so in this example we will demonstrate how to automate the process using shape optimization. The frequencies and overall dimensions are inspired by the model Waveguide Iris Bandpass Filter. The optimization is performed in 2D, but the result is verified in a 3D component.

Model Definition

The physics of this model is identical to that of Waveguide Iris Bandpass Filter, so the main difference is in the setup of the parameterized geometry. The result of the shape optimization is shown in Figure 2, while the initial geometry for this model is shown in Figure 1.

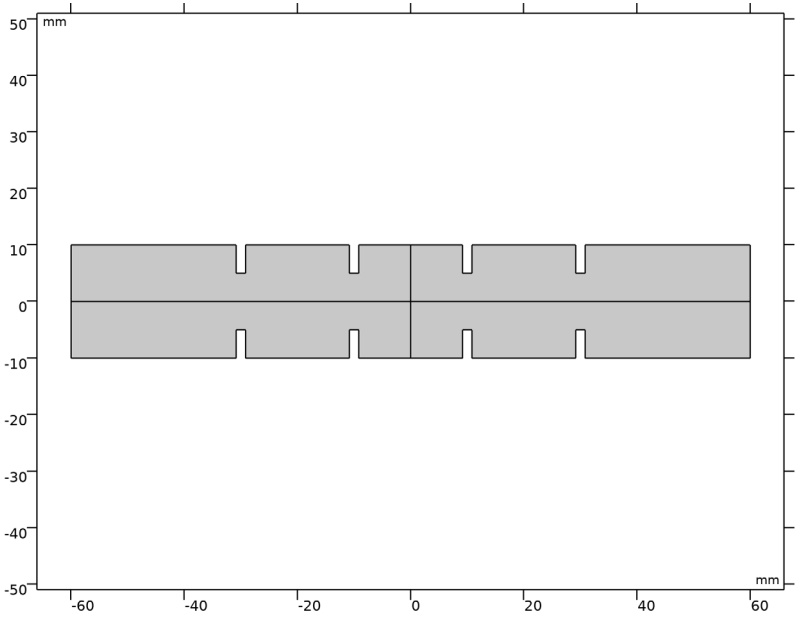


Figure 1: The initial geometry.

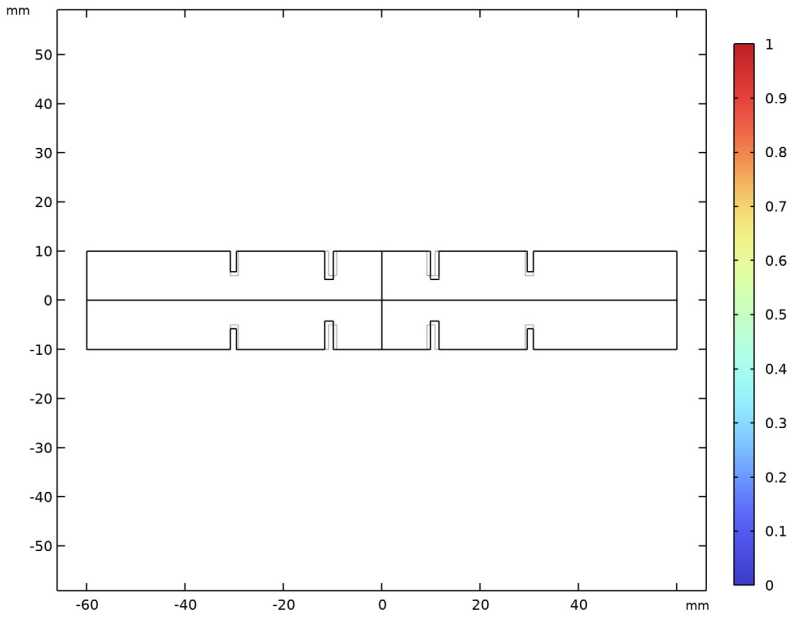


Figure 2: The optimized geometry.

Both the symmetry and rectangular nature of the initial geometry is preserved by combining the **Transformation** and the **Sector Symmetry** features. The objective function, φ, is formulated as a MiniMax problem in the S-parameters,

$$\varphi = \min(\max(\varphi_f))$$

$$\varphi_f = \begin{cases} \text{S11dB} & \text{if } |f - f_2| \le \Delta f/2 \\ \text{S21dB} & \text{otherwise} \end{cases}$$

The MMA optimization can solve such problems using gradient based optimization. One frequency is considered on either side of the pass band, while five frequencies are used for the passband itself. In total 7 objective functions are thus considered in every iteration.

Results and Discussion

Figure 3 shows that the optimized design achieves a reflection of 25 dB for the pass band and a transmission of 25 dB for the two out-of-band frequencies. Furthermore, the 3D verification has good agreement with the raw 2D optimization result.

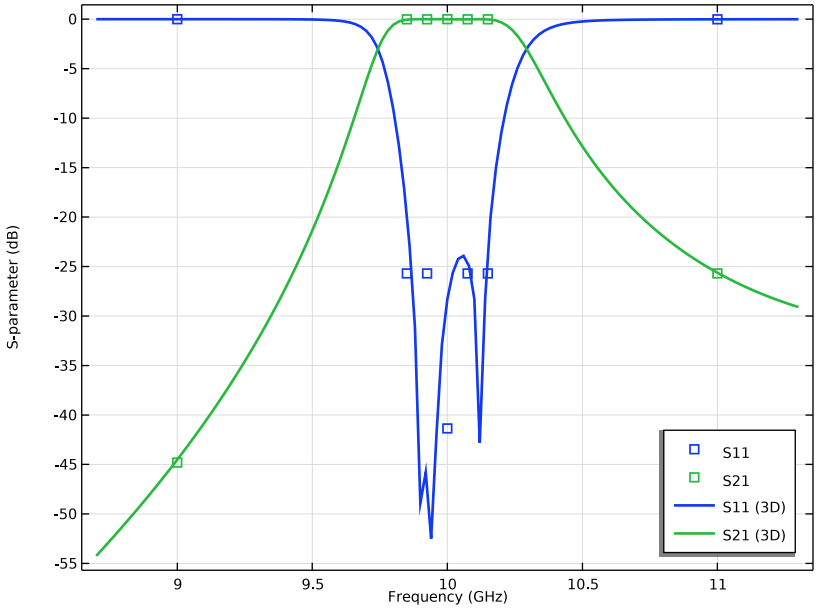


Figure 3: The optimized S-parameters are plotted for the optimization result (points) as well as the 3D verification (lines).

References

1. D.M. Pozar, *Microwave Engineering*, John Wiley & Sons, 1998.

2. R.E. Collin, *Foundation of Microwave Engineering*, McGraw-Hill, 1992.

Application Library path: `RF_Module/Filters/waveguide_filter_optimization_transformation`

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

NEW

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

MODEL WIZARD

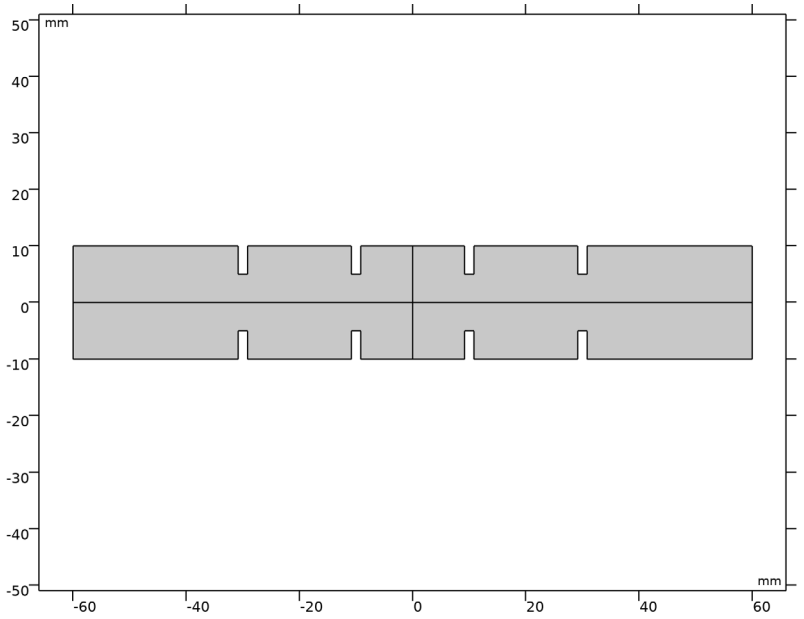
1. In the **Model Wizard** window, click **2D**.
2. In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Frequency Domain**.
6. Click **Done**.

GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

1. In the **Geometry** toolbar, click **Insert Sequence**.
2. Browse to the model's Application Libraries folder and double-click the file `waveguide_filter_optimization_transformation_geom_sequence.mph`.
3. In the **Geometry** toolbar, click **Build All**.
4. Click the **Zoom Extents** button in the **Graphics** toolbar.

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



The geometry should now look like that in Figure 1.

6 In the **Model Builder** window, collapse the **Geometry 1** node.

GLOBAL DEFINITIONS

Geometrical Parameters

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

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

Optimization Parameters

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

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

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

Name	Expression	Value	Description
f1	9[GHz]*2[cm]/w_wg	9E9 Hz	First frequency
f2	f1/9*10	1E10 Hz	Pass frequency

Name	Expression	Value	Description
f3	1.1*f2	1.1E10 Hz	Second frequency
df	0.03*f2	3E8 Hz	Pass frequency bandwidth
dfN	5	5	Frequencies per bandwidth

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 Click **Add to Global Materials** in the window toolbar.

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

MATERIALS

Material Link 1 (matlnk1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Components** section.

2 From the **Electric field components solved for** list, choose **Out-of-plane vector**.

Port 1

1 Right-click **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** and choose **Port**.

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

3 From the **Selection** list, choose **Port 1**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Rectangular**.

Port 2

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

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

3 From the **Selection** list, choose **Port 2**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Rectangular**.

COMPONENT 1 (COMP1)

Free Shape Domain 1

In the **Physics** toolbar, click **Optimization** and choose **Shape Optimization**.

Symmetry/Roller 1

1 In the **Shape Optimization** toolbar, click **Symmetry/Roller**.

2 In the **Settings** window for **Symmetry/Roller**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Symmetry/Roller**.

Transformation 1

1 In the **Shape Optimization** toolbar, click **Transformation**.

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

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

4 From the **Selection** list, choose **Control Boundaries**.

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

	Lock	Lower bound (m)	Upper bound (m)
X		-w_wg/6	w_wg/6

6 Locate the **Scaling** section. From the **Scaling type** list, choose **Anisotropic**.

7 In the table, enter the following settings:

	Lock	Lower bound	Upper bound
X		0.5	2
Y		0.7	1.3

8 Click to expand the **Center of Scaling and Rotation** section. In the table, enter the following settings:

Coordinates	Center type	Center of scaling and rotation (m)
X	Average	0
Y	User defined	w_wg/2

This will allow the rectangles to move in the x direction and scale around a point on the line of the fixed wall.

Sector Symmetry 1

1 In the **Shape Optimization** toolbar, click **Sector Symmetry**.

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

3 From the **Selection** list, choose **Sector Symmetry**.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
obj	if(abs(emw.freq-f2)<df/1.9,emw.S11dB,emw.S21dB)		Objective Function

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

3 In the **Frequencies** text field, type `f1 range(f2-df/2,df/(dfN-1),f2+df/2) f3`.

Shape Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Shape Optimization**.

The default settings work fine, but we can get results faster by reducing the iteration count and disabling the move limits.

2 In the **Settings** window for **Shape Optimization**, locate the **Optimization Solver** section.

3 In the **Maximum number of iterations** text field, type `25`.

4 Clear the **Move limits** check box.

5 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>comp1.obj - Objective Function - 1**.

6 Locate the **Objective Function** section. From the **Solution** list, choose **Maximum of objectives**.

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

8 In the **Settings** window for **Study**, type `Shape Optimization` in the **Label** text field.

9 In the **Study** toolbar, click **Get Initial Value**.

10 In the **Model Builder** window, click **Shape Optimization**.

11 In the **Settings** window for **Shape Optimization**, locate the **Output While Solving** section.

12 Select the **Plot** check box.

13 From the **Plot group** list, choose **Shape Optimization**.

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

RESULTS

Height Expression 1

1 In the **Model Builder** window, expand the **Results>Electric Field (emw)** node.

2 Right-click **Surface** and choose **Height Expression**.

Transfer the optimization results to a 3D component using a **Extrusion 2D** and **Filter** dataset.

Extrusion 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

2 In the **Settings** window for **Extrusion 2D**, locate the **Extrusion** section.

3 In the **z maximum** text field, type `h_wg`.

Filter 1

1 In the **Results** toolbar, click **More Datasets** and choose **Filter**.

2 In the **Settings** window for **Filter**, locate the **Data** section.

3 From the **Dataset** list, choose **Extrusion 2D 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Filter** section. In the **Lower bound** text field, type `0.5`.

6 Right-click **Filter 1** and choose **Create Mesh Part**.

MESH PART 1

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

2 Select the **Use units** check box.

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

Import 1

1 In the **Model Builder** window, under **Global Definitions>Mesh Parts>Mesh Part 1** click **Import 1**.

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

3 From the **Boundary partitioning** list, choose **Detect boundaries**.

4 Click **Import**.

5 Click **Build All**.

6 In the **Model Builder** window, right-click **Mesh Part 1** and choose **Create Geometry**.

GEOMETRY 2

Port 1

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

2 In the **Settings** window for **Box Selection**, type `Port 1` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **x maximum** text field, type `-l_wg-spacing*cavities/2.1`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

6 Right-click **Port 1** and choose **Duplicate**.

Port 2

1 In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** click **Port 1.1 (boxsel2)**.

2 In the **Settings** window for **Box Selection**, type `Port 2` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `l_wg+spacing*cavities/2.1`.

4 In the **x maximum** text field, type `Inf`.

MATERIALS

Material Link 2 (matlnk2)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

ADD PHYSICS

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

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

3 In the tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Shape Optimization**.

5 Click **Add to Component 2** in the window toolbar.

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EMW2)

Port 1

1 Right-click **Component 2 (comp2)>Electromagnetic Waves, Frequency Domain 2 (emw2)** and choose **Port**.

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

3 From the **Selection** list, choose **Port 1**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Rectangular**.

Port 2

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

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

3 From the **Selection** list, choose **Port 2**.

4 Locate the **Port Properties** section. From the **Type of port** list, choose **Rectangular**.

MESH 2

Swept 1

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

2 In the **Settings** window for **Swept**, click to expand the **Sweep Method** section.

3 From the **Face meshing method** list, choose **Triangular (generate prisms)**.

Size

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

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

3 From the **Predefined** list, choose **Extremely fine**.

4 Click **Build All**.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Electromagnetic Waves, Frequency Domain (emw)**.

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

6 In the **Model Builder** window, click the root node.

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

STUDY 2

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `range(f1-df,0.02[GHz],f3+df)`.

3 Click to expand the **Results While Solving** section. From the **Probes** list, choose **None**.

4 Locate the **Physics and Variables Selection** section. In the table, clear the **Solve for** check box for **Deformed geometry (Component 1)**.

5 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output
Electromagnetic Waves, Frequency Domain 2 (emw2)	Selection

6 Click to select row number 2 in the table.

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

8 In the **Add** dialog box, in the **Selections** list, choose **Port 1** and **Port 2**.

9 Click **OK**.

10 In the **Settings** window for **Frequency Domain**, click to expand the **Mesh Selection** section.

11 In the table, enter the following settings:

Component	Mesh
Component 1	No mesh

This reduces the file size of the model.

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

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

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

15 In the **Label** text field, type `Verification (3D)`.

16 In the **Home** toolbar, click **Compute**.

RESULTS

S-parameter (emw)

1 In the **Model Builder** window, expand the **Results>S-parameter (emw)** node, then click **S-parameter (emw)**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **None**.

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

5 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

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

2 In the **Settings** window for **Global**, click to expand the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

Global 2

1 In the **Model Builder** window, right-click **S-parameter (emw)** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Verification (3D)/Solution 2 (3) (sol2)**.

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)>Electromagnetic Waves, Frequency Domain 2> Ports>S-parameter, dB>emw2.S11dB - S11**.

5 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)>Electromagnetic Waves, Frequency Domain 2> Ports>S-parameter, dB>emw2.S21dB - S21**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
emw2.S11dB	1	S11 (3D)
emw2.S21dB	1	S21 (3D)

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

8 From the **Width** list, choose **2**.

9 In the **S-parameter (emw)** toolbar, click **Plot**.

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

Electric Field (emw)

1 In the **Model Builder** window, under **Results** click **Electric Field (emw)**.

2 In the **Electric Field (emw)** toolbar, click  **Plot**.

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

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

5 From the **Parameter value (freq (GHz))** list, choose **10**.

6 In the **Electric Field (emw)** toolbar, click  **Plot**.

Shape Optimization

1 In the **Model Builder** window, click **Shape Optimization**.

2 In the **Shape Optimization** toolbar, click  **Plot**.

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

Geometry Modeling Instructions

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

NEW

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

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
h_wg	1[cm]	0.01 m	Waveguide width
w_wg	2*h_wg	0.02 m	Waveguide width
spacing	w_wg	0.02 m	Cavity spacing
cavities	3	3	Number of cavities
l_wg	1.5*w_wg	0.03 m	Port distance to cavities

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D**.

GEOMETRY 1

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

2 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `2*l_wg+cavities*spacing`.

4 In the **Height** text field, type `w_wg/2`.

5 Locate the **Position** section. In the **x** text field, type `-l_wg-cavities/2*spacing`.

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

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `w_wg/12`.

4 In the **Height** text field, type `w_wg/4`.

5 Locate the **Position** section. In the **x** text field, type `-l_wg-w_wg/24`.

6 In the **y** text field, type `w_wg/4`.

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

Array 1 (arr1)

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

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

3 From the **Input objects** list, choose **Rectangle 2**.

4 Locate the **Size** section. In the **x size** text field, type `cavities+1`.

5 Locate the **Displacement** section. In the **x** text field, type `spacing`.

Difference 1 (dif1)

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

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

3 From the **Objects to add** list, choose **Rectangle 1**.

4 From the **Objects to subtract** list, choose **Rectangle 2**.

Symmetry

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

2 In the **Settings** window for **Box Selection**, type `Symmetry` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **y minimum** text field, type `w_wg/2.1`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

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 **Rectangle 1**.

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

5 Locate the **Normal Vector to Line of Reflection** section. In the **x** text field, type `0`.

6 In the **y** text field, type `1`.

Union 1 (uni1)

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

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

3 From the **Input objects** list, choose **Rectangle 1**.

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

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 In the **y** text field, type `-w_wg/2`.

5 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

6 In the **y** text field, type `w_wg/2`.

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

Exterior Boundaries

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

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

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Union 1** in the **Input selections** list.

5 Click **OK**.

Control Boundaries

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

2 In the **Settings** window for **Box Selection**, type `Control Boundaries` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-spacing/10`.

5 In the **x maximum** text field, type `l_wg+cavities*spacing/2.1`.

6 In the **y minimum** text field, type `0`.

7 In the **y maximum** text field, type `w_wg/2.1`.

8 Locate the **Input Entities** section. From the **Entities** list, choose **From selections**.

9 Click **Add**.

10 In the **Add** dialog box, select **Exterior Boundaries** in the **Selections** list.

11 Click **OK**.

Port 1

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

2 In the **Settings** window for **Box Selection**, type `Port 1` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **x maximum** text field, type `-l_wg-cavities*spacing/2.1`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

6 Right-click **Port 1** and choose **Duplicate**.

Port 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Port 1.1 (boxsel4)**.

2 In the **Settings** window for **Box Selection**, type `Port 2` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `l_wg+cavities*spacing/2.1`.

4 In the **x maximum** text field, type `Inf`.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Free Shape Domain

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

2 In the **Settings** window for **Box Selection**, type `Free Shape Domain` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `-w_wg*0.001`.

4 In the **y minimum** text field, type `-w_wg*0.001`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Sector Symmetry

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

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

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog box, select **Free Shape Domain** in the **Selections to invert** list.

5 Click **OK**.

Center Line

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

2 In the **Settings** window for **Box Selection**, type `Center Line` in the **Label** text field.

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

4 Locate the **Box Limits** section. In the **y minimum** text field, type `-0.001*w_wg`.

5 In the **y maximum** text field, type `0.001*w_wg`.

6 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Symmetry/Roller

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

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

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

4 In the **Label** text field, type `Symmetry/Roller`.

5 Locate the **Input Entities** section. Click **Add**.

6 In the **Add** dialog box, in the **Selections to add** list, choose **Symmetry**, **Line Segment 1**, and **Center Line**.

7 Click **OK**.

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

The model geometry is now complete.