



Using Periodic Boundary Conditions

Use *periodic boundary conditions*, *periodic edge conditions*, and *periodic point conditions* to define a constraint that makes two quantities equal on two different (but usually equally shaped) domains.

To implement a periodic boundary condition in FEMLAB, go through the following steps:

1. Use a homogenous Neumann boundary condition on all the boundaries that you want to apply the periodicity to. This boundary condition normally corresponds to zero flux or insulation. When you make the boundaries periodic, this will make the inflow through one of the boundaries equal the outflow through the other.
2. On the **Physics** menu, point to **Periodic Condition**, and then click the domain type: **Boundary**, **Edge**, or **Point**.
3. In the dialog box for the periodic condition, start by specifying the source geometry on the **Source** page. Type the expression that you wish to use in the periodic condition for the source. Refer to each constraint in the periodic boundary condition by a name. A default name appears automatically for the constraint that implements the periodic condition. To change the default name of the constraint while working on the source page, type another name in the **Constraint name** column.

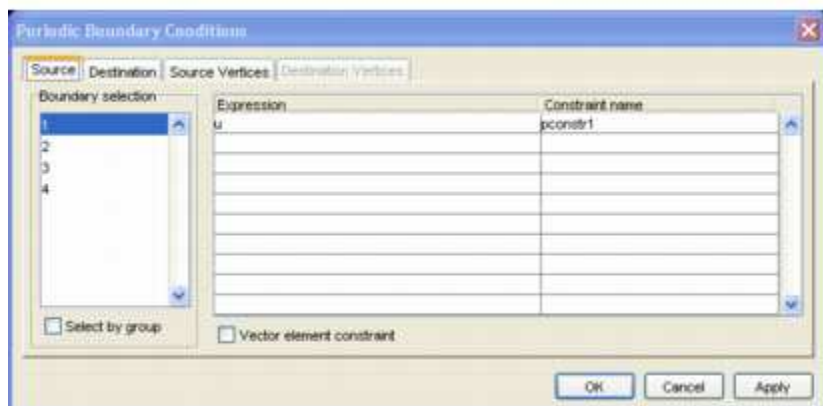


Figure 4-24: The source page in the Periodic Boundary Conditions dialog box.

1. Click the **Destination** tab to define the destination domains. Select the geometry (if there are more than one geometry in the model) in the **Geometry** list and then click to select the boundaries that should be the destination boundaries. You are free to use several boundaries and more than one geometry. You can also select boundaries by clicking in the geometry and then click to select the **Use selected boundaries as destination** check box. Continue by selecting the constraint that you want to specify in the **Constraint name** list. In the **Expression** edit field, type the expression that should be equal to the expression on the source domain for the corresponding constraint; for a strictly periodic boundary condition, this is the same as the expression on the **Source** page. Set the constraint order to the same order as the elements. This is the default setting, except when you are using application modes with different element orders

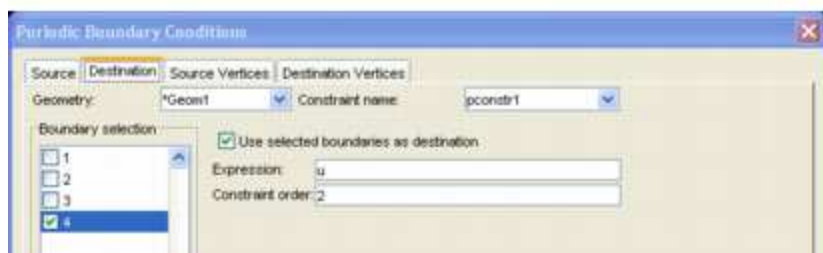




Figure 4-25: The destination page in the Periodic Boundary Conditions dialog box.

1. Click the **Source Vertices** tab to specify the transformation between the source and destination domains. Select 1, 2, 3, or 4 vertices that span the subspace constituted by the source domain. The number of vertices depends on the dimension of the source domain. For a periodic boundary, the number of vertices should be the same as the dimension of the geometry. In a 2D simulation, the source vertices are typically the end points of the boundary.
2. Finally click the **Destination Vertices** tab and select 1, 2, 3, or 4 vertices that span the subspace constituted by the destination domain; the number depends on the dimension of the destination domain, but you must provide the same number of points in both the source and destination domains. The source expression will be projected onto this subspace. Its value at the first source vertex is projected to the first destination vertex, and so on. It is therefore important to assign the vertices in the correct order. Together the vertices on the source geometry and destination geometry define a linear transformation. See the following section for more information about the linear transformation from source geometry to destination geometry.

Note: When using periodic boundary conditions to connect boundaries or edges that do not have equivalent meshes, you may need to use the sparse null-space function, particularly in 3D. To do so, open the **Solver Parameters** dialog box, click the **Advanced** tab, and select **Sparse** in the **Null-space function** list.

LINEAR TRANSFORMATION FROM SOURCE TO DESTINATION

You specify a linear transformation by giving a set of points in the source geometry and in the destination geometry. The order of the vertices is significant. FEMLAB constructs a linear transformation from the source domain to the destination domain using the subspaces spanned by the vertices. Denote the transformation rank by n , denote the source vertices by x_0, x_1, \dots, x_n , and denote the destination vertices by x'_0, x'_1, \dots, x'_n . After padding the source and destination vertices' vectors with zeros as necessary, the software solves the following matrix equation for a *transformation matrix* T and a *translation vector* V :

$$x'_0 = Tx_0 + V$$

$$x'_1 - x'_0 = T(x_1 - x_0)$$

$$\dots$$

$$x'_n - x'_0 = T(x_n - x_0)$$

Choose n such that the source domain can be embedded a subspace of dimension $n-1$. For example, when the source domain is a straight line it suffices to have two vertices; when it is a curve that can be embedded in a plane, you need three vertices. Four vertices are needed for a source domain that is a curved surface or a curve that can only be embedded in 3D.

Implementation of the Periodic Boundary Conditions

FEMLAB implements a periodic boundary condition as a special case of the extrusion coupling variable. In particular, it adds a variable name equal to the constraint name on the destination domains with a value equal to the constrained expression. The periodic boundary constraint works as an *ideal constraint*, that is, the derivatives of the constraint with respect to the solution help from the reaction forces. If you need more freedom in specifying how the reaction forces are applied, use the extrusion coupling variable

directly and specify the reaction force and constraint as weak contributions.

Periodic Boundary Condition Example

This example demonstrates how to implement a simple periodic boundary condition.

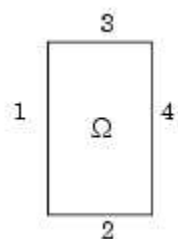
MODEL DEFINITION

Consider an eigenvalue equation on a rectangle of dimension $1 \times \sqrt{\pi}$.

$$-\Delta u = \lambda u \quad \text{on domain } \Omega$$

$$u(x_1, y) = u(x_4, y) \quad \text{on boundaries 1 and 4}$$

$$u(x, y_2) = u(x, y_3) \quad \text{on boundaries 2 and 3}$$



RESULTS AND DISCUSSION

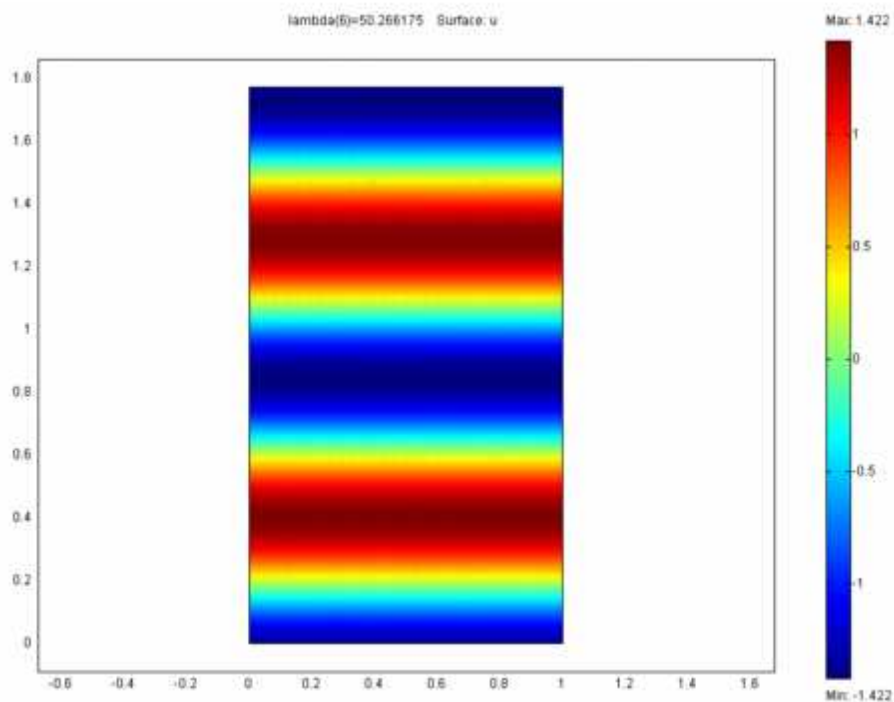


Figure 4-26: The periodic pattern in the solution (6th eigenmode).

The wavelength of the sinusoidal solutions is the rectangle side length divided by integers. Each integer corresponds to an eigensolution. The positionings of the maxima are random because the absolute value of the solution is not constrained anywhere.

MODELING USING THE GRAPHICAL USER INTERFACE

Model Navigator

1. Start FEMLAB.
2. In the **Model Navigator** select **2D** in the **Space dimension** list.
3. In the **PDE Modes** folder, select **PDE, Coefficient Form** and then **Eigenvalue analysis**. Make sure that **Lagrange - Quadratic** is selected in the **Element** list.
4. Click **OK**.

Geometry Modeling

1. Go to **Draw>Specify Objects>Rectangle** to open the **Rectangle** dialog box. Type **1** in the **Width** edit field and $\text{sqrt}(\pi)$ in the **Height** edit field.
2. Click **OK**.
3. Click the **Zoom Extents** toolbar button.

Physics Settings

Boundary Conditions

1. Open the **Boundary Settings** dialog box.
2. Select all boundaries and choose Neumann boundary conditions. Leave the default values.
3. Click **OK**.

Subdomain Settings

Use the default settings.

Periodic Boundary Conditions

1. On the **Physics** menu, point to **Periodic Conditions**, and then click **Periodic Boundary Conditions**.
2. On the **Source** page, select boundary 1. Type u in the **Expression** edit field and press **Enter**.
3. Click the **Destination** tab. Select the check box for boundary 4. Type u in the **Expression** edit field.
4. Click the **Source Vertices** tab. Add vertices 1 and 2 to the **Source vertices** list.
5. Click the **Destination Vertices** tab. Add vertices 3 and 4 to the **Destination vertices** list.

This concludes the periodicity across the vertical boundaries. Proceed to the next steps to make the solution periodic across the horizontal boundaries.

6. Click the **Source** tab and select boundary 2. On the second row, type u in the **Expression** edit field and press **Enter**.
7. Click the **Destination** tab and select the check box for boundary 3. Type u in the **Expression** edit field.
8. Click the **Source Vertices** tab and add vertices 1 and 3 to the **Source vertices** list.
9. Click the **Destination Vertices** tab and add vertices 2 and 4 to the **Destination vertices** list.
10. Click **OK**.

Mesh Generation

1. Click the **Initialize Mesh** button.
2. Click the **Refine Mesh** button once.

Computing the Solution

The eigenvalue algorithm does not work reliably when the parameter that you specify in the **Search for eigenvalues about** edit field is equal to an eigenvalue. This particular problem has a zero eigenvalue so you must change the value of this parameter.

1. Open the **Solver Parameters** dialog box.
2. Type 10 in the **Search for eigenvalues around** edit field.
3. Click the **Solve** button to compute the solution.

Postprocessing and Visualization

By default FEMLAB plots the first eigenmode, which in this case is a flat solution associated with a zero eigenvalue. The higher eigenmodes are more interesting.

1. Open the **Plot Parameters** dialog box.
2. Look at the solution for different eigenvalues by making a corresponding selection in the **Eigenvalue** list. Start with the last eigenvalue.
3. Click **Apply** to plot for each eigenvalue.

Click **OK** when finished.

1. Click **OK**.

Subdomain Settings

Use the default settings.

Periodic Boundary Conditions

1. On the **Physics** menu, point to **Periodic Conditions**, and then click **Periodic Boundary Conditions**.
2. On the **Source** page, select boundary 1. Type u in the **Expression** edit field and press **Enter**.
3. Click the **Destination** tab. Click to select the check box for boundary 4. Type u in the **Expression** edit field.
4. Click the **Source Vertices** tab. Add the vertices 1 and 2 to the **Source vertices** list.
5. Click the **Destination Vertices** tab. Add the vertices 3 and 4 to the **Destination vertices** list.

This concludes the periodicity across the vertical boundaries. Proceed to make the solution periodic across the horizontal boundaries:

6. Click the **Source** tab and select boundary 2. On the second line, type u in the **Expression** edit field and press **Enter**.
7. Click the **Destination** tab and then click to select the check box for boundary 3. Type u in the **Expression** edit field.
8. Click the **Source Vertices** tab and then add the vertices 1 and 3 to the **Source vertices** list.
9. Click the **Destination Vertices** tab and then add the vertices 2 and 4 to the **Destination vertices** list.

10. Click **OK**.

Mesh Generation

1. Click the **Initialize Mesh** button to initialize the mesh.
2. Click the **Refine Mesh** button once to refine the mesh.

Computing the Solution

1. Open the **Solver Parameters** dialog box.
2. Type 10 in the **Search for eigenvalues around** edit field.
3. Click the **Solve** button to compute the solution.

The eigenvalue algorithm does not work reliably when the parameter that you specify in the **Search for eigenvalues about** edit field is equal to an eigenvalue. This problem has a zero eigenvalue. This is why you need to change the default.

Postprocessing and Visualization

1. Open the **Plot Parameters** dialog box.
2. Look at the solution for different eigenvalues by making a corresponding selection in the **Eigenvalue** list. Start with the last eigenvalue.
3. Click **Apply** to plot for each eigenvalue.
4. Click **OK** when finished.

[Specifying Point and Edge Settings](#)



[Using Identity Conditions](#)