# Acoustics

Bioprocess Laboratory
Department of Chemical Engineering
Chungnam National University

# The Acoustics Application

- The acoustics application mode provides two analysis types.
- Time-harmonic analysis

$$\nabla \cdot \left(-\frac{1}{\rho_0} \nabla p + q\right) - \frac{\omega^2 p}{\rho_0 {c_s}^2} = 0$$

| Coefficient | Description |
|---|---|
| $\omega = 2\pi f$ | Angular frequency |
| $\rho_0$ | Fluid density |
| $c_s$ | Speed of sound |
| $q$ | Dipole source |

- Eigenvalue analysis

$$\nabla \cdot \left(-\frac{1}{\rho_0} \nabla p\right) - \frac{\lambda p}{\rho_0 c^2} = 0$$

| Coefficient | Description |
|---|---|
| $\lambda$ | Eigenvalue |

# Subdomain quantities

| Quantity | Variable | description |
|---|---|---|
| $\rho_0$ | rho0 | Density |
| $c_s$ | cs | Speed of sound |
| $q$ | qx, qy, qz | Dipole source |

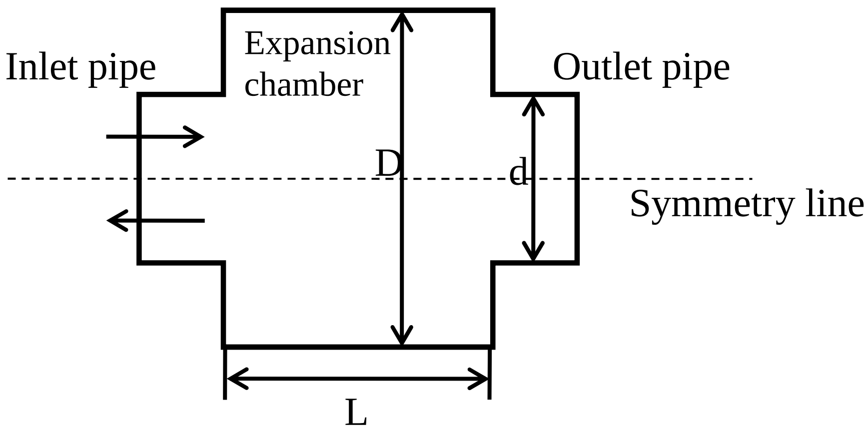
- Density : density is that of the fluid in which the acoustic waves propagate. The default value is 1.024 kg/m3.
- Speed of sound : The default value is 343m/s, the speed of sound in air at approximately 20 ℃.
- Dipole source : Vector with individual components for each space dimension.

# Boundary conditions

- Boundary conditions applied to the different analysis types.
  - Sound hard boundaries (walls)
  - Sound soft boundaries
  - Pressure source
  - Impedance boundary condition
  - Radiation boundary condition
  - Specified normal acceleration
  - Axial symmetry
  - Interface conditions on borders

# Model : Reactive muffler

- This model examines the sound transmission properties of an idealized reactive muffler with long inlet and outlet pipes.




- You need to include only half of the geometry as indicated in the above figure, because this is an axisymmetric model.

# Equations for reactive muffler

- Transmission loss coefficient

$$D_{TL} = 10 \cdot \log(\frac{W_i}{W_t})$$

- Transmission loss coefficient (1D theoretical solution)

$$D_{TL} = 10 \cdot \log(1 + (\frac{S_1}{2 \cdot S_2} - \frac{S_2}{2 \cdot S_1})^2 \cdot (\sin(k \cdot L)^2))$$

- Intensity

$$I = \frac{p^2}{2 \cdot \rho_0 \cdot c}$$

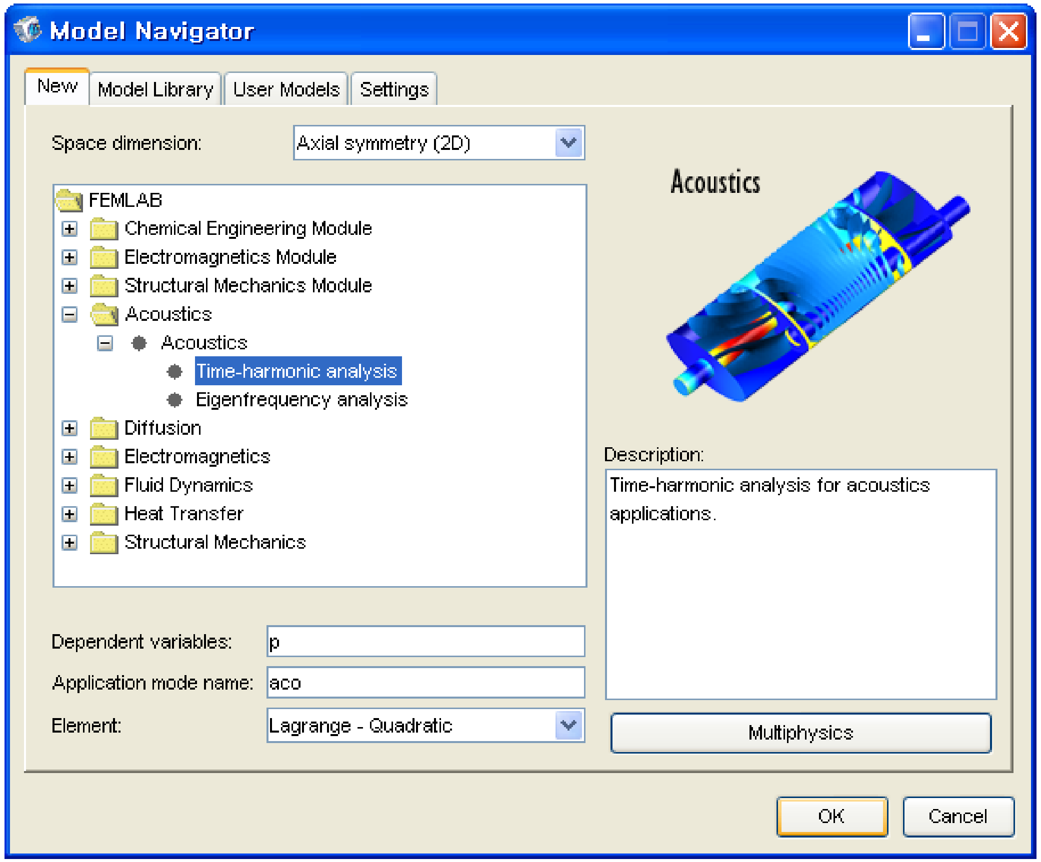
- Sound power

$$W = \int (I \cdot 2 \cdot \pi \cdot r)\, dr$$

- Frequency

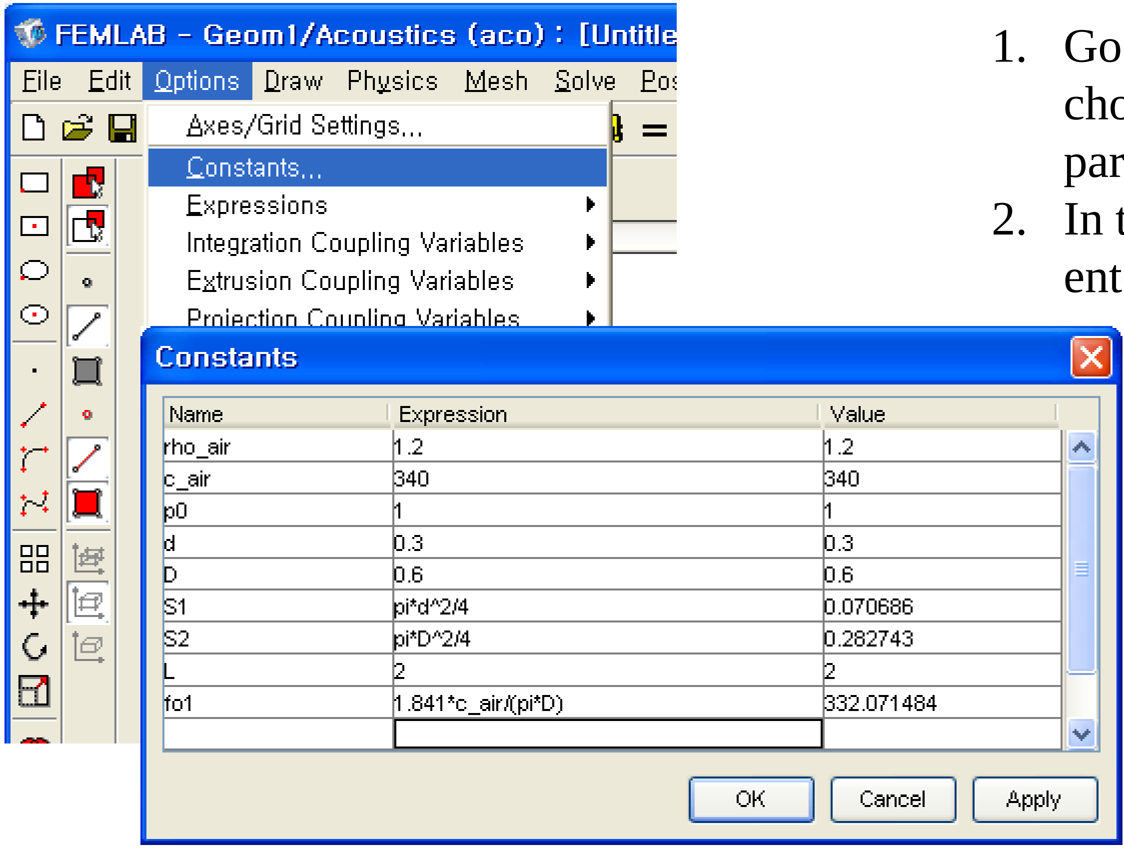
$$f = 1.841 \cdot \frac{c}{\pi \cdot D}$$

# Modeling using the FEMLAB Model navigator



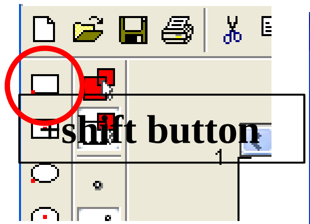

1. Go to the **Model Navigator** and select **Axial symmetry** (2D) in the **Space dimension** list
2. In the list of application modes open the **Acoustics** folder and then **Time-harmonic analysis**.
3. Click **OK**.

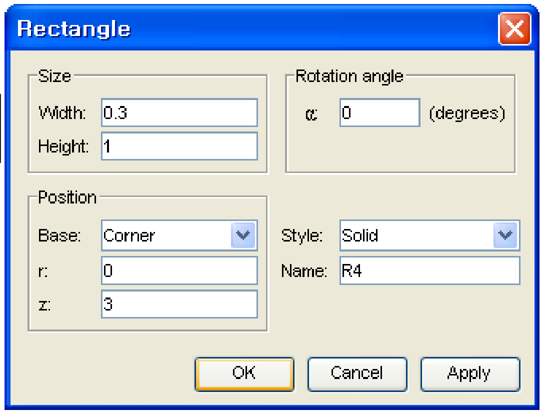
# Modeling using the FEMLAB
# Options and settings

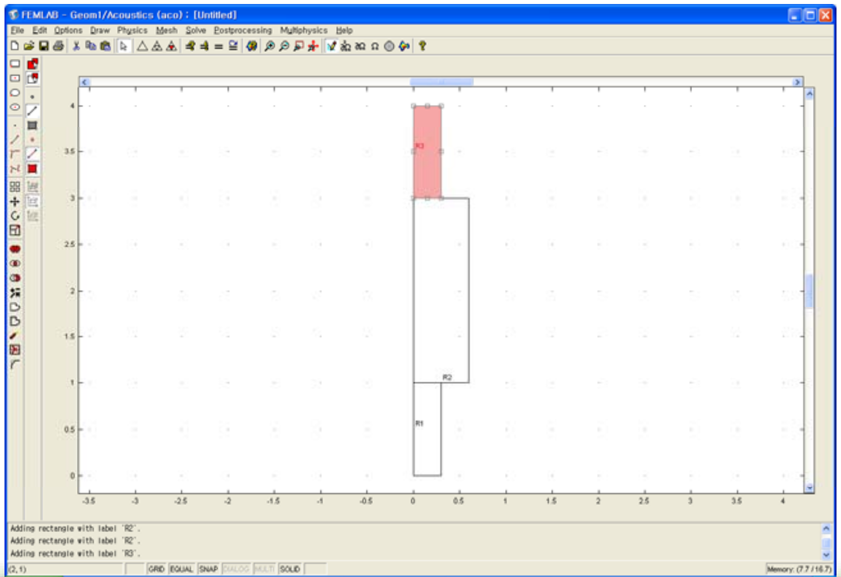



1. Go to the **Options** menu and choose **Constans** to parameterize the model.
2. In the **Constants** dialog box enter the following constants.

# Modeling using the FEMLAB Geometry modeling

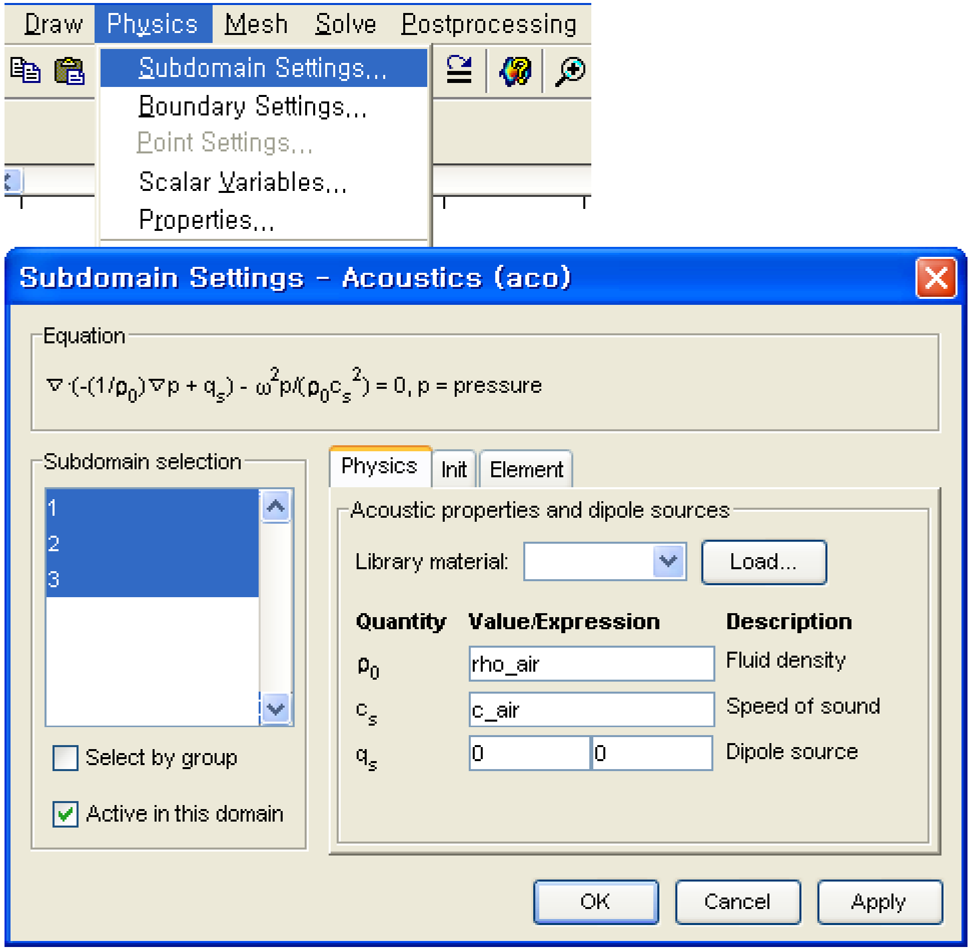


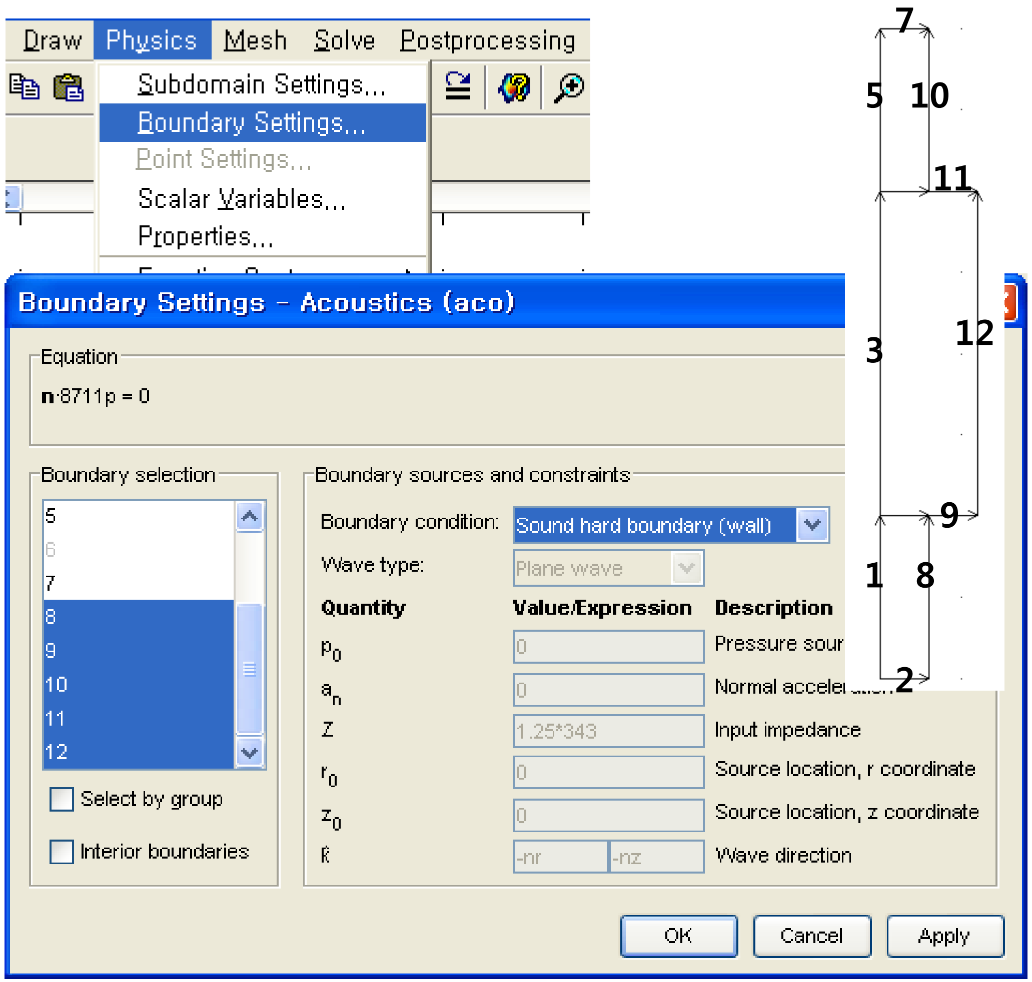
1. Shift-click the **Rectangle/Square** button to specify a rectangle.
2. Enter 0.3, 0.6, 0.3 in the **Width** edit field and 1, 2, 1 in the **height** field and 0, 1, 3 in the **z** edit field respectively.

# Modeling using the FEMLAB Physics settings (subdomain)



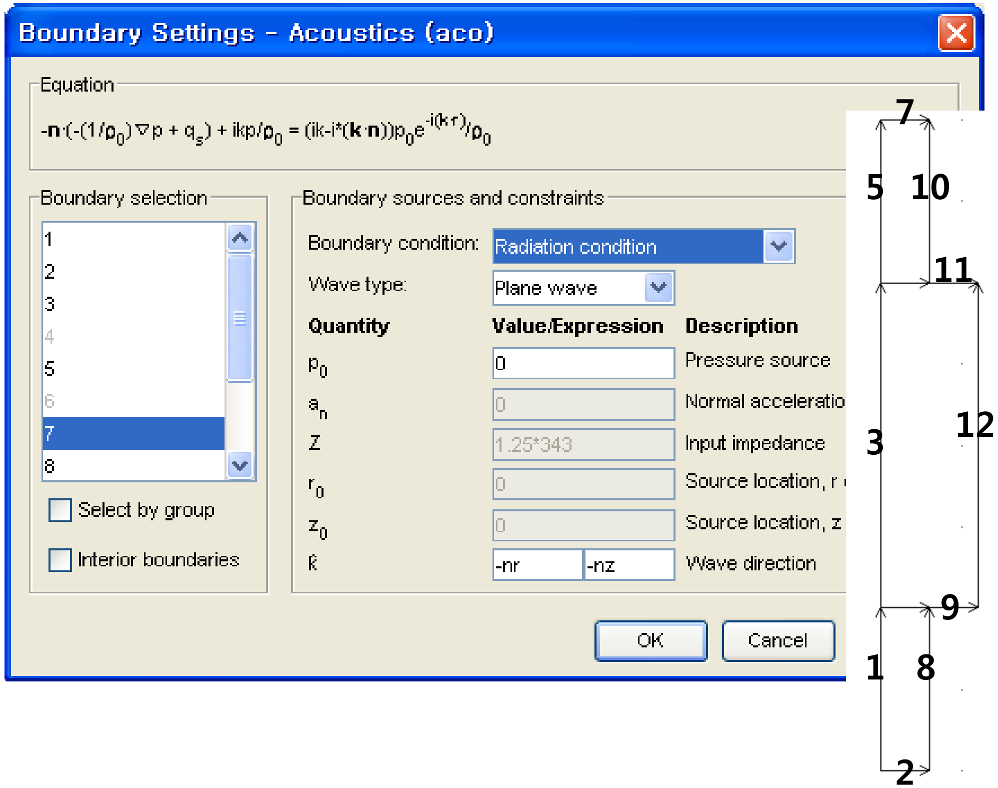

1. Go to the **Subdomain** menu and choose **Subdomain Settings**.
2. Select all subdomains from the **Subdomain selection** list.
3. Enter rho_air in the **Fluid Density** edit field.
4. Enter c_air in the **Speed of sound** edit field.
5. Leave the default settings (0) for the **Dipole source**.

# Modeling using the FEMLAB Physics settings (boundary)



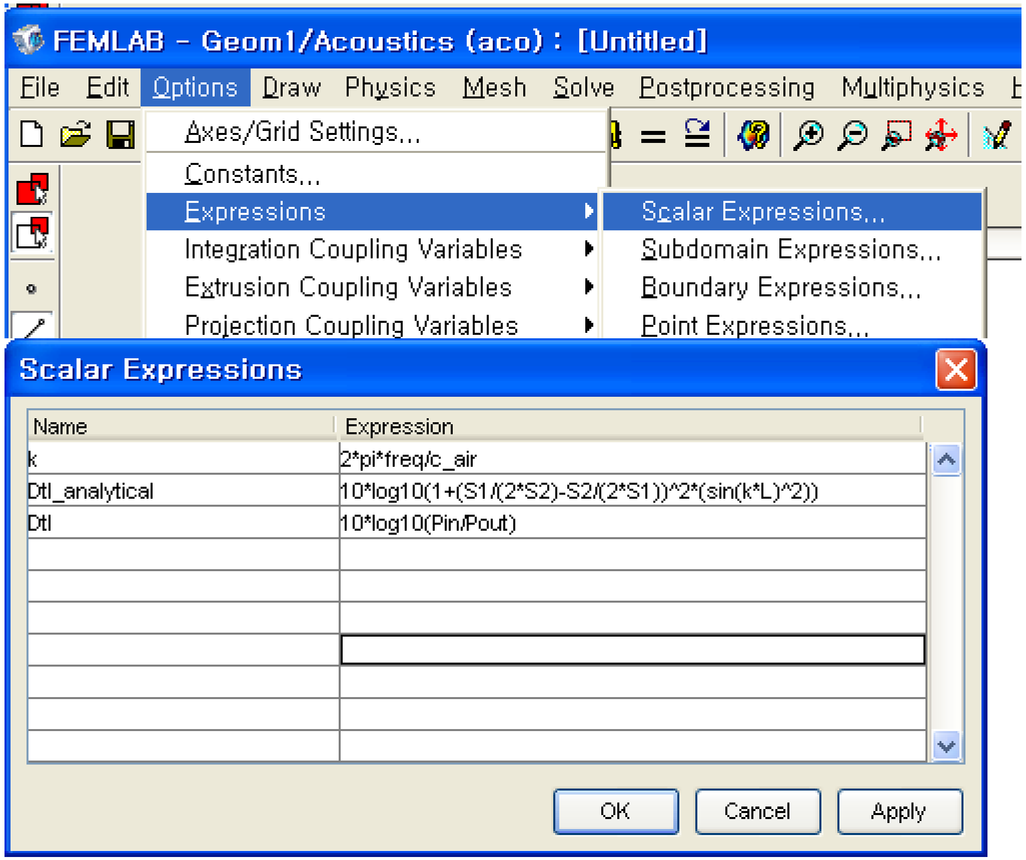

1. Go to the **Boundary** menu and choose **Boundary Settings**.
2. In the **Boundary Conditions** dialog box enter the following boundary coefficients
3. Select boundaries 1, 3 and 5 in the **Boundary selection** list.
4. Select Axial symmetry in the **Boundary condition** list.
5. Select boundaries 8 to 12 in the **Boundary selection** list.
6. Select Sound hard boundary (wall) in the **Boundary condition** list.

# Modeling using the FEMLAB Physics settings (boundary)



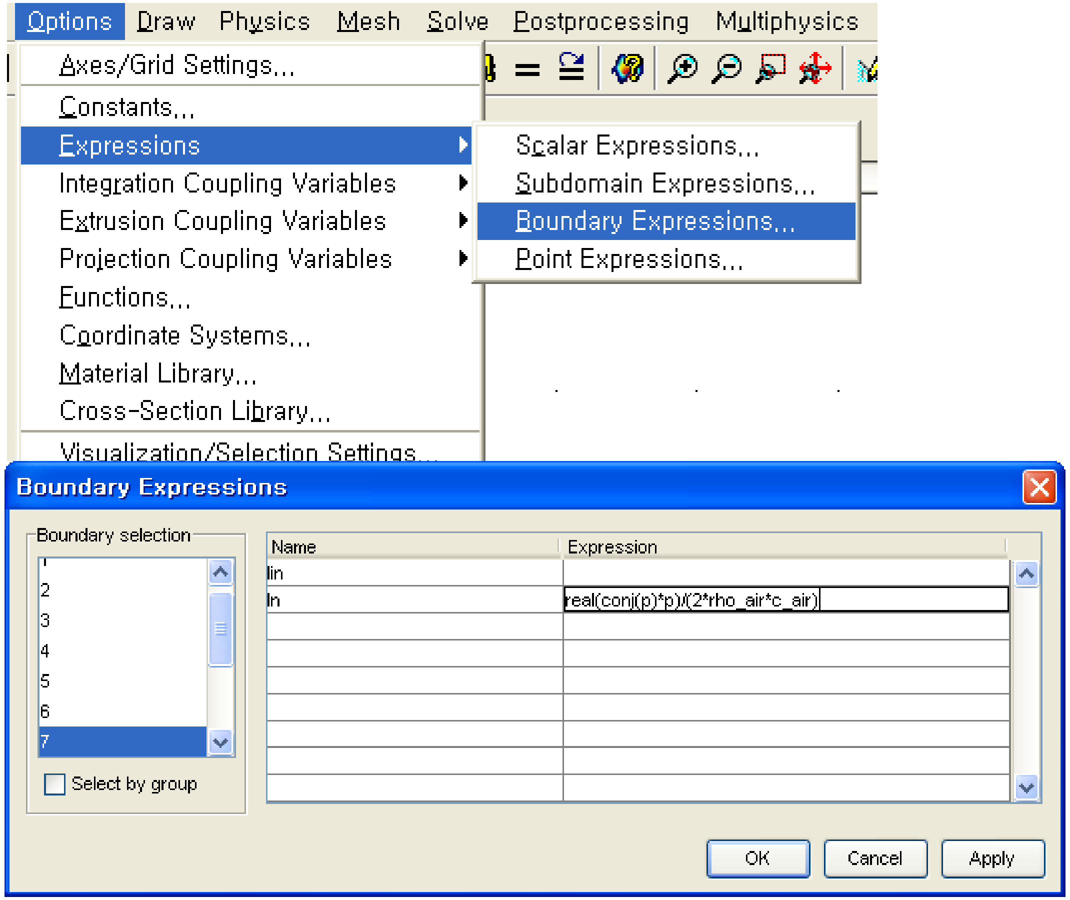

7. Select boundary 2 in the **Boundary selection** list.
8. Select **Radiation condition** in the **Boundary condition** list.
9. Type 1 in the **Pressure source** edit field.
10. Finally select boundary 7 in the **Boundary selection** list.
11. Select **Radiation condition** in the **Boundary condition** list.
12. Click **OK**.

# Modeling using the FEMLAB
# Physics settings (expression variables)



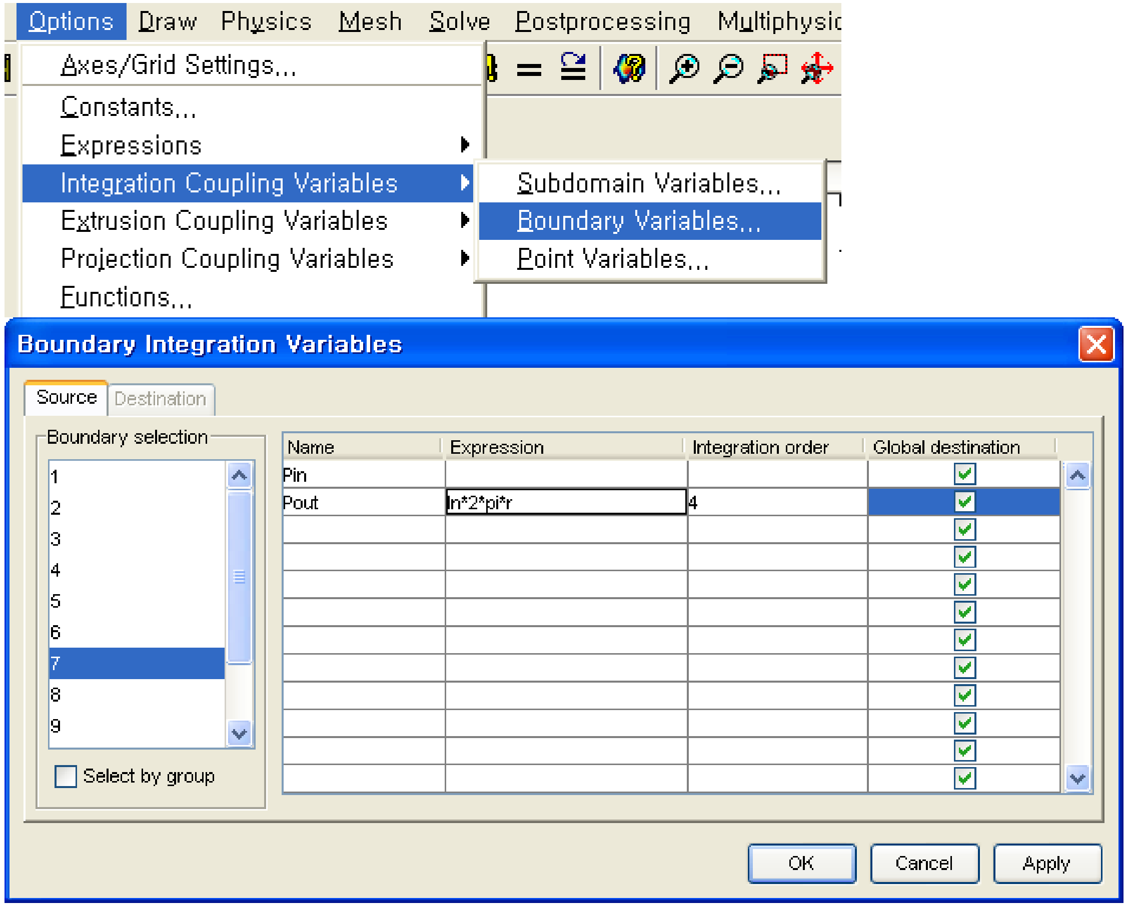

1. On the **Options** menu, point to **Expressions**, and then click **Scalar Expressions**.
2. In the **Scalar Expression** dialog box enter the following.
3. Click **OK**.

# Modeling using the FEMLAB
# Physics settings (expression variables)



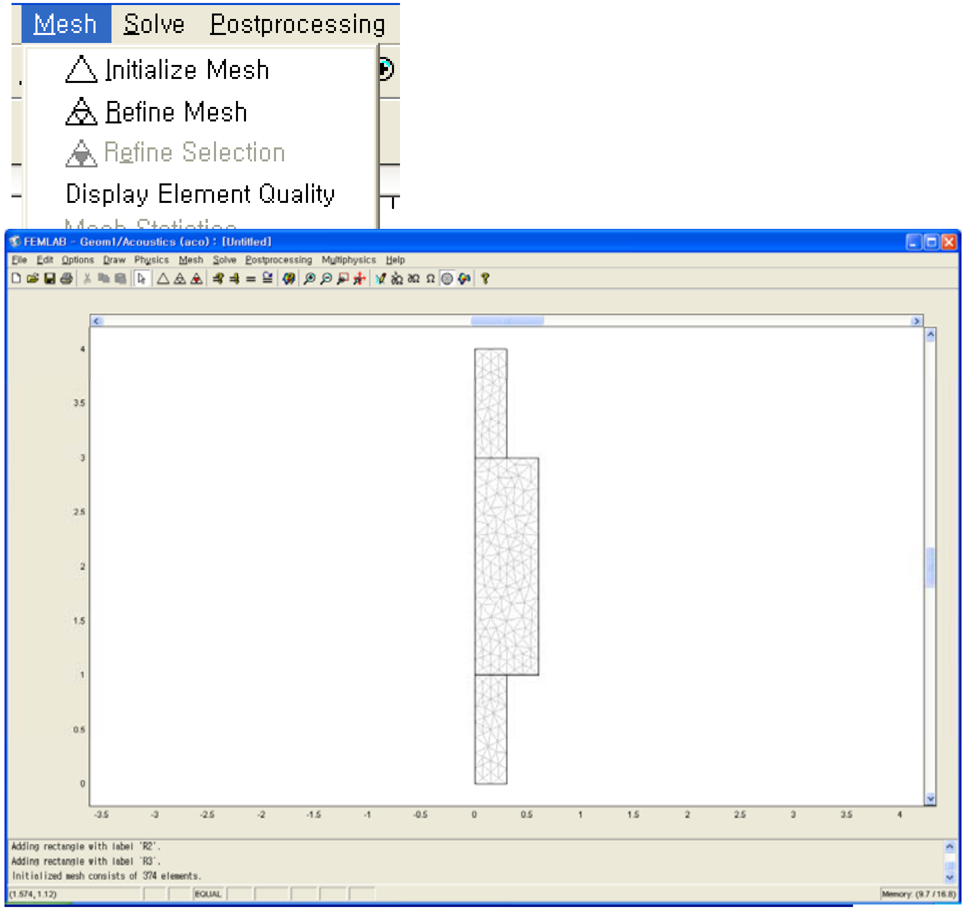

4. Go the **Options** menu and choose **Expressions** and then **Boundary Expressions**.
5. In the **Boundary Expressions** dialog box select boundary 2 from the **Boundary selection** list and enter the following expression.
6. In the **Boundary Expression** dialog box select boundary 7 from the **Boundary selection list** and enter the following boundary integration variables
7. Click **OK**.

# Modeling using the FEMLAB Physics settings (scalar coupling variables)



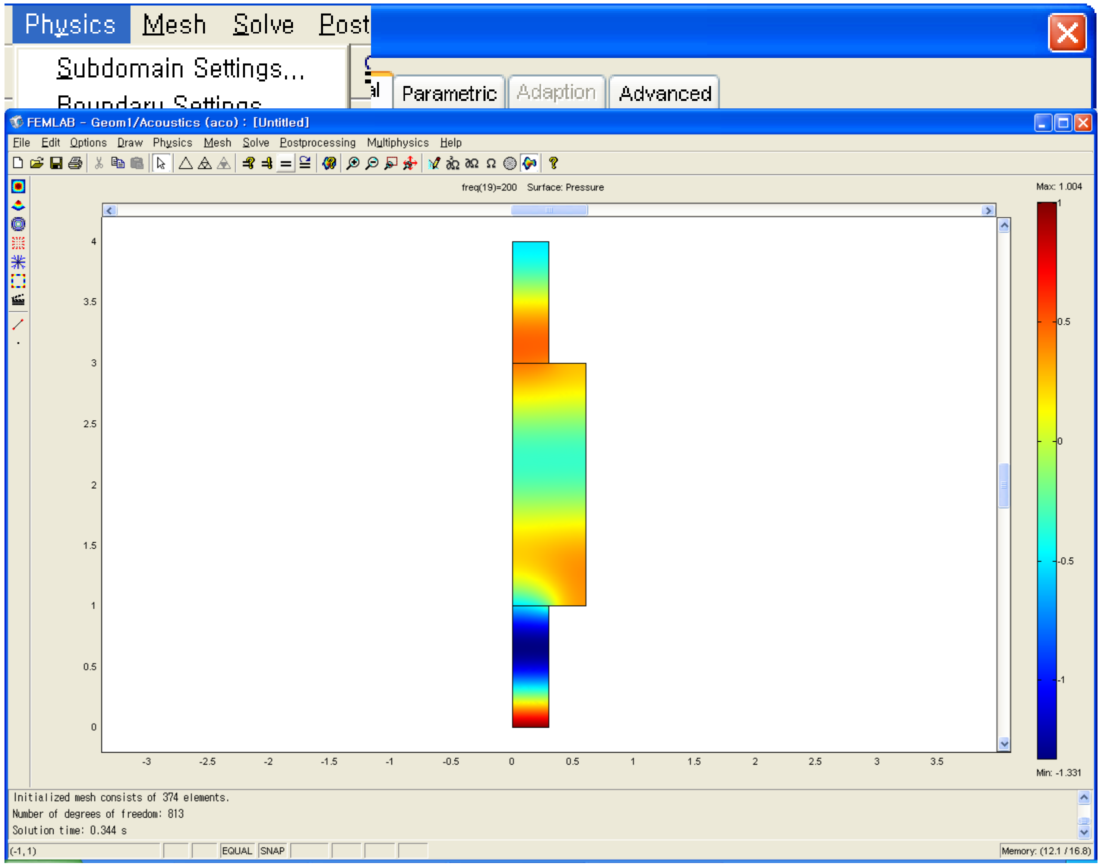

1. Go to the **Options** menu and choose **Scalar Coupling Variables** and the **Boundary Variables**
2. In the **Boundary Integration Variables** dialog box select boundary 2 and the enter the following boundary integration expression.
3. In the **Boundary Integration Variables** dialog box select boundary 7 and enter the following boundary integration expression.
4. Click **OK**.

# Modeling using the FEMLAB Mesh generation



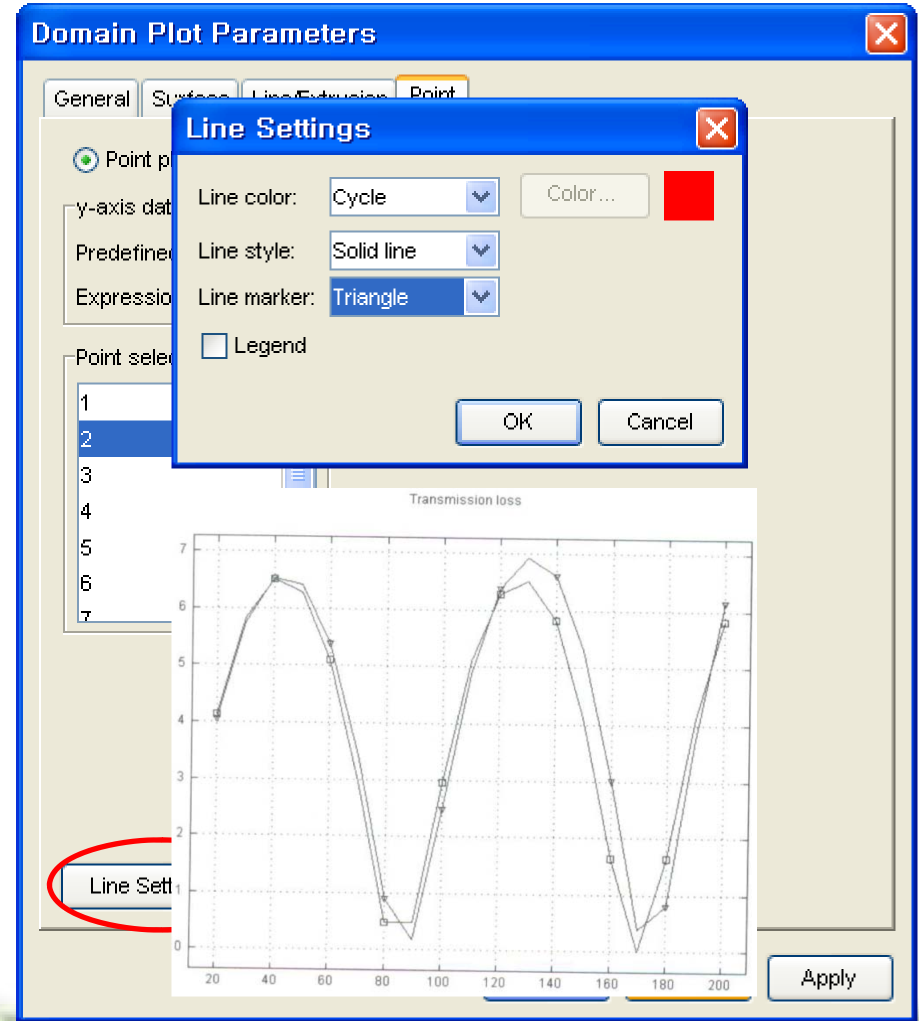

1. Go to the **Mesh** menu and choose **Mesh Parameters**.
2. In the **Mesh Parameters** dialog box select **Finer** from the **Predefined mesh size** list.
3. Click **Remesh**.
4. Click **OK**.

# Modeling using the FEMLAB
# Computing the solution




1. From the **Solve** menu, choose **Solver Parameters**.
2. In the **Solver Parameters** dialog box select **Parametric linear** from the solver selection list.
3. Enter **freq** in the **Name of Parameter** edit field.
4. Enter **20:10:200** in the **List of parameter** values edit field.
5. Click **OK**.
6. Go to the **Physics** menu and choose **Scalar Variables**.
7. Enter **freq** in the frequency edit field.
8. Click **OK**.
9. Click the **Solve** button to start the simulation.

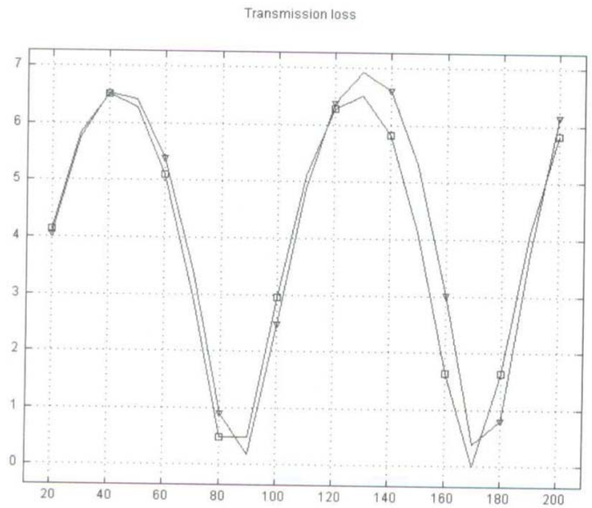
# Modeling using the FEMLAB Postprocessing and visualization




1. Go to the **Postprocessing** menu and select **Domain Plot Parameters**.
2. Click the **Keep current plot** page.
3. Click the **Title/Axis** button and enter **Transmission loss** in the **Title** edit field.
4. Click **OK**.
5. Select **Point plot** in **Domain Plot Parameters** dialog box and click on the **Point tab**.
6. Select point **1** from the **Point selection** list.
7. Enter **Dtl** in the **Expression** edit field.
8. Click the **Line Settings** button and select **Triangle** in the **Line marker** list.
9. Click **OK**.
10. Click **Apply** in the **Domain Plot Parameters** dialog box
11. **Dtl_analytical** plot method is also same

# Conclusions

- The discrepancy increases with frequency between the 1D theoretical model and a 3D analysis.
- In the lower frequency range, there is good agreement between the theoretical solution and the FEM solution.




Dtl analysis : triangle
Dtl theoretical model : square