

Finite-Element modeling of 3-D Time-Harmonic Structural Acoustic Target Scattering with near-field and far-field codes

Kazufumi Ito ^{*} Zhonghua Qiao [†] Jari Toivanen [‡]

Abstract

Objective of the monograph is twofold. First, it is a user guide for COMSOL Multiphysics 3.3 modeling for 3D undersea acoustics and accurately evaluating the scattering from elastic targets. Second, we develop a code that couples the near-field COMSOL modeling and the fast far-field Fortran solver to evaluate the scattered field with hundreds of wavelength. It is based on an iterative method based on the domain decomposition/embedding method and enables us to evaluate the scattered field for higher frequencies 15-20 kHz. In this frequency range millions of unknowns are required for the finite element modeling.

1 Model problem

The physical problem is the scattering of a wave by an solid elastic cylinder buried in a sediment under the water. It is an exterior problem which have to be truncated into a bounded domain for a finite element discretization. We use a truncation to a rectangular domain. It is shown in Fig. 1.

The partial differential equation model for the pressure p in water and sediment ($\Pi \setminus \Omega$)

^{*}Center for Research in Scientific Computation & Department of Mathematics, North Carolina State University, Raleigh, NC 27695, USA. Email: kito@ncsu.edu

[†]Center for Research in Scientific Computation, North Carolina State University, Raleigh, NC 27695, USA. Email: zqiao@ncsu.edu

[‡]Center for Research in Scientific Computation, North Carolina State University, Raleigh, NC 27695, USA. Email: jatoivan@ncsu.edu

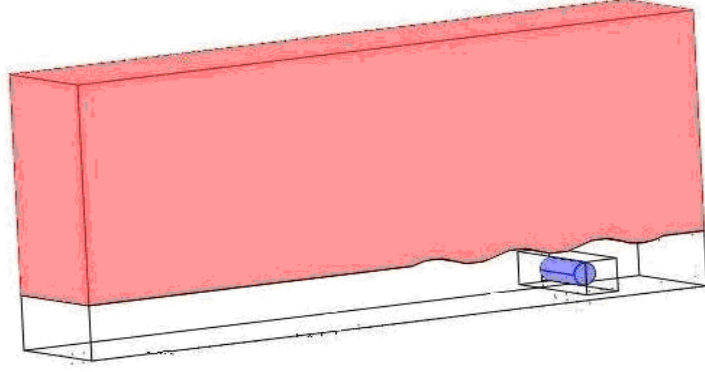


Figure 1: Computational domain.

and the displacement u in the elastic object (Ω) is:

$$\begin{aligned} \nabla \cdot \frac{1}{\rho} \nabla p + \frac{k^2}{\rho} p + g &= 0 \quad \text{in } \Pi \setminus \Omega \\ \frac{1}{\rho} \frac{\partial p}{\partial n} &= \omega^2 u \cdot n, \quad -pn = \sigma(u)n \quad \text{on } \partial\Omega \\ \nabla \cdot \sigma(u) + \omega^2 \rho u &= 0 \quad \text{in } \Omega \\ \mathcal{B}p &= 0 \quad \text{on } \partial\Pi, \end{aligned} \tag{1.1}$$

where ρ is the density, ω is the angular frequency, $k = \omega/c$ is the wavenumber, c is the speed of sound and σ is the stress tensor. f is a time-harmonic forcing term and n is the outward unit normal.

We pose an absorbing boundary condition $\mathcal{B}p = 0$ on $\partial\Pi$. Most often we employ the second-order absorbing boundary condition which usually leads to sufficiently small reflections from the boundaries for practical purposes. The boundary $\partial\Pi$ consists of six rectangular faces, denotes by $\Gamma_{\pm k}$, $k = 1, 2, 3$, whose outward normal directions are given by the coordinate directions $\pm x_k$. The second-order absorbing boundary conditions on these faces are of the form

$$\pm \frac{\partial p}{\partial x_k} - ikp - \frac{i}{2k} \sum_{1 \leq j \neq k \leq 3} \frac{\partial^2 p}{\partial x_j^2} = 0. \tag{1.2}$$

Furthermore, we have the conditions

$$-\frac{3}{2}k^2 p - ik\left(\pm \frac{\partial p}{\partial x_m} \pm \frac{\partial p}{\partial x_j}\right) - \frac{1}{2} \frac{\partial^2 p}{\partial x_k^2} = 0 \tag{1.3}$$

on each edge denoted by $\Gamma_{(\pm m, \pm j)}$ between the faces $\Gamma_{\pm m}$ and $\Gamma_{\pm j}$ and the conditions

$$-2ikp + \sum_{j=1}^3 \pm \frac{\partial p}{\partial x_j} = 0 \tag{1.4}$$

on the eight corners denoted by Ψ of the parallelepiped.

We will describe how to use the COMSOL Multiphysics 3.3/ Structural Acoustic Module to evaluate the scattered field from the elastic target in Section 2. For the large frequency, the large memory is required and COMSOL cannot solve it directly. We describe a coupled code in Section 3 to solve (1.1) for larger frequencies. The problem was solved by an iterative method that utilizes COMSOL code in a small box surrounding the target (near field) and a Fortran code outside the box (far field).

2 Finite-element analysis with COMSOL

In this section, we describe in detail the COMSOL coding for the 3-D time-Harmonic structural acoustic problem (1.1).

For the acoustic system, COMSOL solves the following equation

$$\frac{1}{\omega^2} \nabla \cdot \frac{1}{\rho} \nabla p + \frac{1}{\omega^2} \frac{k^2}{\rho} p + \frac{1}{\omega^2} g = 0, \quad (2.1)$$

which is a Helmholtz equation by multiplying the first equation in (1.1) with $1/\omega^2$. From (2.1) and (1.2)-(1.4), we have the weak formulation: Find

$$p \in V = \{v \in H^1(\Pi \setminus \Omega) : v|_{\partial\Pi} \in H^1(\partial\Pi), v|_{\Gamma_{(m,j)}} \in H^1(\Gamma_{(m,j)}), \forall \Gamma_{(m,j)} \in \Phi\}$$

such that

$$\begin{aligned} & \int_{\Pi \setminus \Omega} \frac{1}{\rho \omega^2} (-\nabla p \cdot \nabla q + k^2 p q) dx + ik \int_{\partial\Pi} \frac{1}{\rho \omega^2} p q ds \\ & - \frac{i}{\omega^2} \sum_{k=1}^3 \sum_{j \neq k} \left(\int_{\Gamma_k} \frac{1}{\rho \omega^2} \frac{\partial p}{\partial x_j} \frac{\partial q}{\partial x_j} ds + \int_{\Gamma_{-k}} \frac{1}{\rho \omega^2} \frac{\partial p}{\partial x_j} \frac{\partial q}{\partial x_j} ds \right) \\ & - \frac{3}{4} \sum_{\Gamma_{(m,j)} \in \Phi} \int_{\Gamma_{(m,j)}} \frac{1}{\rho \omega^2} p q dl + \frac{1}{4\omega^2} \sum_{\Gamma_{(m,j)} \in \Phi} \int_{\Gamma_{(m,j)}} \frac{1}{\rho \omega^2} \frac{\partial p}{\partial x_k} \frac{\partial q}{\partial x_k} |_{k \neq m,j} dl \\ & - \frac{i}{2\omega} \sum_{x \in \Psi} \frac{1}{\rho \omega^2} p(x) q(x) + \frac{1}{\omega^2} \int_{\Pi \setminus \Omega} g q dx = 0. \end{aligned} \quad (2.2)$$

Here Φ is the set of all edges of $\partial\Pi$.

2.1 Starting COMSOL Multiphysics

We solve the two problems of with and without the elastic target simultaneously for the ease of postprocessing, e.g., evaluating the scatter field. Open COMSOL Multiphysics 3.3. In the Model Navigator window Fig. 2, we define the Multiphysics model for our problem as follows.

- Click Mutiphysics.
- Click Add Geometry. Choose 3D for Space dimension in the pop-up Add Geometry window and click OK. Then we have added a 3D geometry (Geom1).
- Click Add Geometry. Choose 2D for Space dimension in the pop-up Add Geometry window and click OK. Then we have added a 2D geometry (Geom2).
- Click the icon 'Geom1(3D)' in the small window on the right, and then in the large window on the left, under Application Modes, select COMSOL Multiphysics > Acoustic > Acoustics, click Time-harmonic analysis. In Element (bottom left corner), select Lagrange-Linear and click Add (top of the right window). Now we have one acoustic system 'aco'.
- Click the icon 'Geom1(3D)' in the small window on the right, and then in the large window on the left, under Application Modes, select COMSOL Multiphysics > Structural Mechanics Module > Solid, Stress-Strain, click Frequency response analysis. In Element, select Lagrange-Linear and in Dependent variables, change the variables to 'u,v,w,p2', then click Add. Now we have the structural mechanics module 'smsld'.
- Click the icon 'Geom1(3D)' in the small window on the right, and then in the large window on the left, under Application Modes, select COMSOL Multiphysics > Acoustic > Acoustics, click Time-harmonic analysis. In Element, select Lagrange-Linear and click Add. Now we have one acoustic system 'aco2'.
- Click OK on the Model Navigator window.

We now have COMSOL Multiphysics window.

Remark: Of course, we can solve the both problems separately. In fact we only solve the problem with target alone for the coupled code in Section 3, i.e, we only solve the system with 'aco' and 'smsld'.

2.2 Material properties definition

First, we define the material properties of the water and the sediment by

- Click the COMSOL menu Options > Constants, and fill Constants window as Fig. 3.

Secondly, we define the application scalar variables by

- Click the COMSOL menu Physics > Scalar Variables, just change the frequency in the Application Scalar Variables window as Fig. 4.

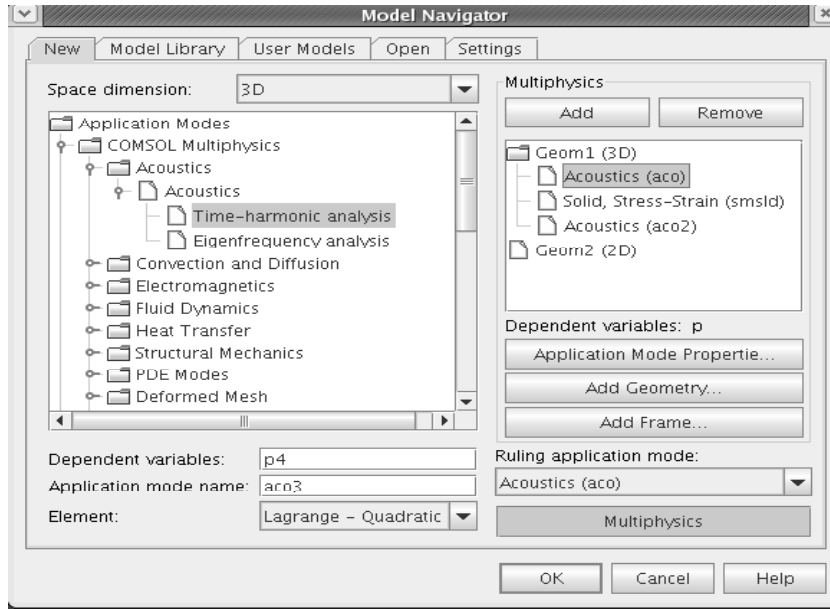


Figure 2: Model Navigator.

Then COMSOL automatically defines c , ω and k as:

$cs_aco = c1$ or $c2$ (depend on the different subdomains),

$\omega_{aco} = 2 * \pi * freq_aco$,

$k_aco = \omega_{aco} / cs_aco$.

2.3 Geometry

In the COMSOL Multiphysics window, first we need define a 2D working plane, and then we extrude it to the 3D geometry. The 2D geometry is defined as follows.

- Select 'Geom2' from the two tabs 'Geom1' and 'Geom2'.
- Click Draw > Specify Objects > Rectangular, and on the pop-up window specify the box as in Fig. 5 and click OK (as a result we have Box R1).
- Click Draw > Specify Objects > Rectangular and on the pop-up window (just like the Fig. 5) specify the box. We just change the Width and Height and the Position in the window. Click OK (as a result we have Box R2).
- Click Draw > Draw Objects > 2nd Degree Bezier Curve and click at the point $(-2.2625, 0.4354)$ and connect to the second point $(-2.075, 0.5354)$, click at the point. Then connect to the third point $(-1.8875, 0.4354)$, click. Right click to finish a curve (as a result we have a domain CO1). Click Copy symbol on the menu to copy CO1. Click CO1 and press Ctrl key, then click R2 (now we selected the two domains). Click

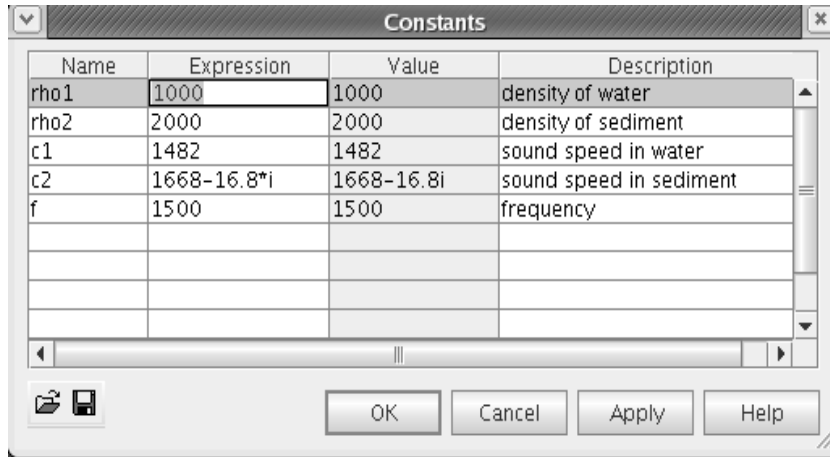


Figure 3: Constants.

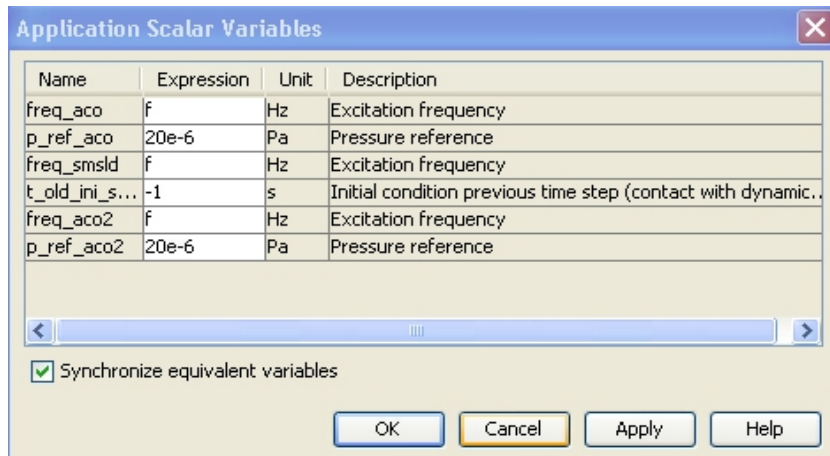


Figure 4: Model Navigator.

the vertical menu Union and click the third icon from the bottom of the vertical menu Delete interior boundaries, click Paste symbol on the menu and click OK (as a result we have domain CO2). Click CO2 and press Ctrl, then click R1. Click Difference icon on the vertical menu (as a result we have two subdomains CO3 and CO1).

- Click Draw > Draw Objects > 2nd Degree Bezier Curve and click at the point (-1.8875,0.4354) and connect to the second point (-1.7,0.3354), click at the point. Then connect to the third point (-1.5125,0.4354),click. Right click to finish a curve (as a result we have a domain CO2). Click Copy symbol on the menu to copy CO2. Click CO2 and press Ctrl key, then click CO3 (now we selected the two domains). Click the vertical menu Union and click the third icon from the bottom of the vertical menu Delete interior boundaries, click Paste symbol on the menu and click OK (as a result we have domain CO3). Click CO3 and press Ctrl key, then click CO1. Click Difference icon on the vertical menu (as a result we have two subdomains CO2 and

CO4). Now we have two subdomains CO2 and CO4 with one wave of the wavy interface.

- Click Draw > Specify Objects > Square and on the pop-up window specify the left top corner box as in Fig. 6. Click OK (as a result we have a box R1). The coordinate of the right lower corner of 'R1' is $(-10, 4.4354)$ in the 2D working plane (we wish that the source point $(-10, 4.4354, 0.25)$ is located on a mesh point, so we define 'R1'.)
- Click R1, Click Copy, press Ctrl key and click CO2, click Difference Icon, click Paste and then OK (as a result we have R1, CO1, CO4).
- Click Draw > Specify Objects > Circle and on the pop-up window specify the circle as in Fig. 7. Click OK (as a result we have target cylinder C1).
- Click C1, Click Copy, press Ctrl key and Click CO4, Click Difference Icon, click Paste and then OK (as a result we have R1, CO1, CO2, E1).

In order to have the wavy interface, we need to repeat the two steps 4 and 5 at different points. After the wavy interface is done, we have completed the 2D working plane as shown in Fig. 8.

Then we extrude the 2D geometry to 3D geometry by doing the following steps.

- Click Draw > Extrude > Select all objects CO1, CO2, R1 and E1 and specify the distance (0.25) and click OK (as a result temporary 3D geometry).
- Click Cut symbol to cut this temporary 3D geometry. Click Paste symbol, in the pop-up window, change the Displacements on z to -0.25. Click OK.
- Click Paste symbol, in the pop-up window, change the Displacements on z to 0.5. Click OK.
- Click Draw > Extrude > Select all objects CO1, CO2, R1 and E1 and specify the distance (0.5) and click OK.

Now we have the 3D geometry Fig. 9.

2.4 Mesh

For the large domain computation, we prefer not to use the tetrahedral mesh. It will have more freedoms than prism mesh with the same mesh size. For a 1500 frequency, we need the mesh size smaller than 0.1m. Now we will construct a prism mesh on the 3D geometry Fig. 9.

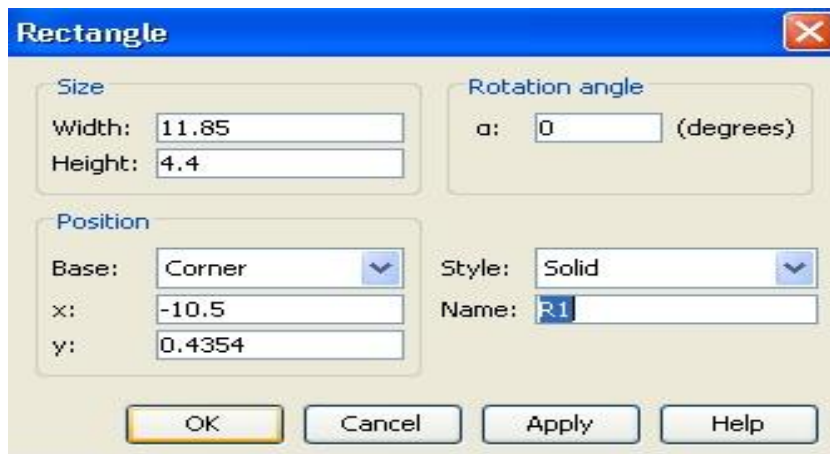


Figure 5: Rectangular defining.

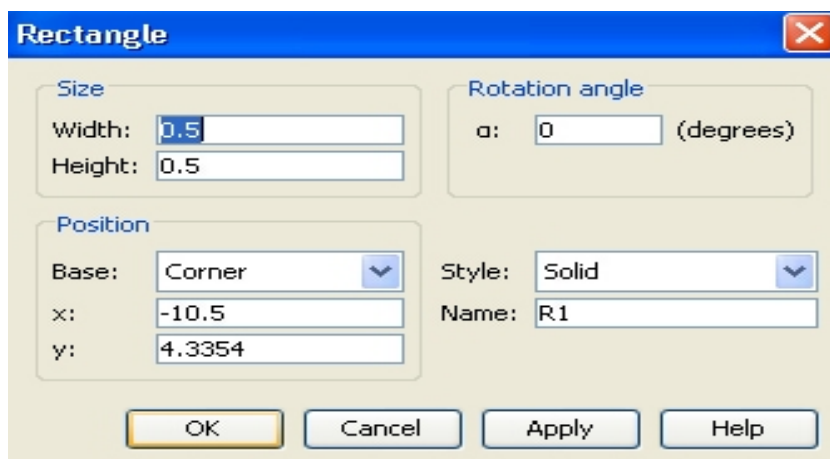


Figure 6: Corner box defining.

- Select 'geom1' from the two tab 'geom1' and 'geom2'.
- Click COMSOL menu Mesh > Free mesh parameters. Select 'boundary' tab in the pop-up Free Mesh Parameters window.
- Select all the faces on the front x-y plane of the 3D geometry. Type in 0.1 in the Maximum element size.
- Click the Mesh Selected button, we obtain a triangular mesh distribution on this face.
- Click the COMSOL menu Mesh > Swept mesh parameters. In the pop-up window, select the subdomain 3 from the Subdomain selection window on the left. Select the Element Layers tab, check Manual specification of element layers and Number of element layers, and type in 5. Select Sweep Direction, check Manual specification

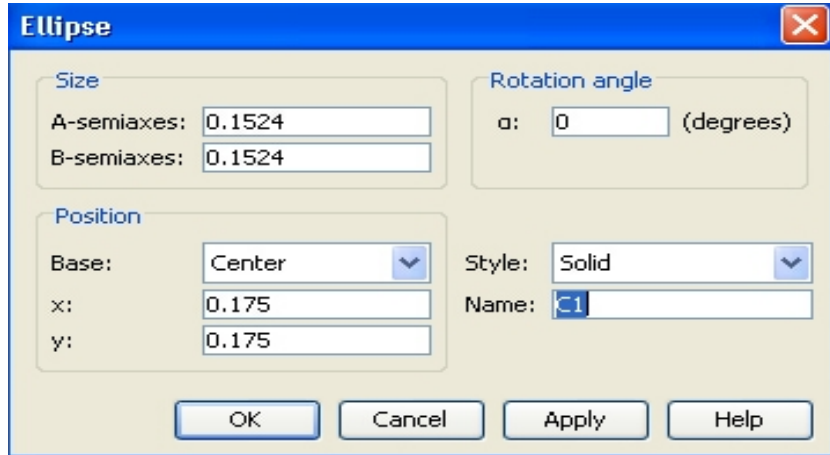


Figure 7: Cylinder target defining.



Figure 8: Cross section of the 3D geometry.

of sweep direction. Type in 10 in the source face and 9 in the target source. Click Mesh Selected. Then we have the prism mesh distribution in the subdomain 3.

- Repeat the step 5 in each subdomain from the front to the behind.

Finally, we obtain a 3D mesh distribution for the whole domain. See Fig. 10.

2.5 Subdomain settings

For the subdomain settings, first we set up for the acoustic part.

- Select Mutiphysics > Acoustic (aco), and click Physics > Subdomain settings.
- Select subdomain for the elastic target, and uncheck 'Active in this subdomain'.
- Select all the other subdomains, on the Physics Tab, enter corresponding ρ_0 (with rho1,rho2), c_s (with c1,c2), q_s (with 0,0,0) and Q (with 0 for the subdomains without source, with

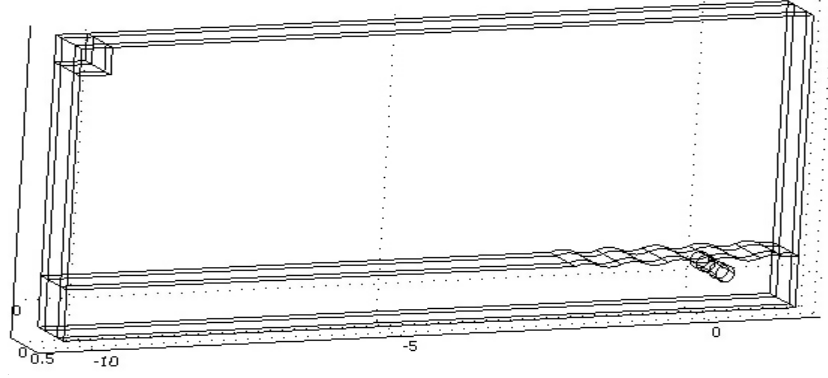


Figure 9: The 3D geometry.

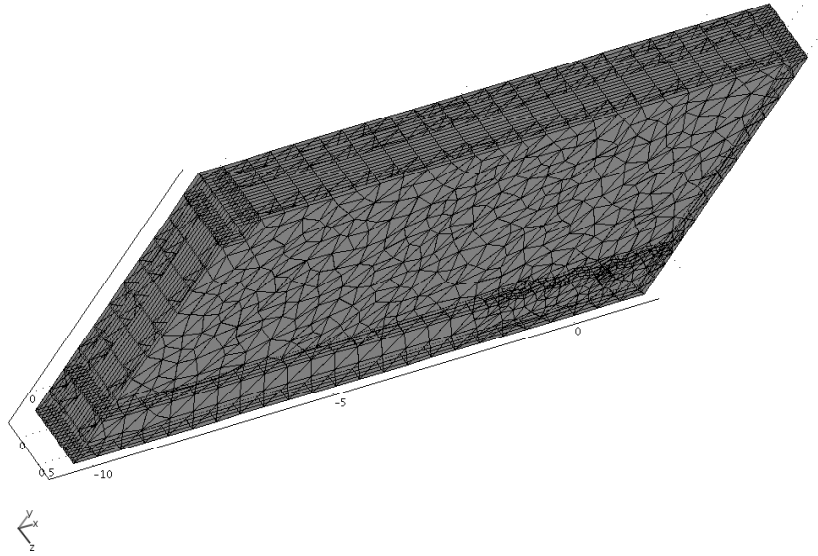


Figure 10: Mesh.

$$'sech(40*\sqrt{((x+10)^2+(y-4.4354)^2+(z-0.25)^2)^8})'$$

for the subdomains with source, which is shown in Fig. 11.

Then for another acoustic system 'aco2', every setting up is the same as 'aco' except that there is no elastic target in this system, i.e., we do not uncheck 'Active in this subdomain' in the domain for the cylinder target.

Finally for the elastic target system,

- Select Mutiphysics > Solid, stress-strain (smsld), and click Physics > Subdomain settings.
- Select all the subdomains for the acoustic part, and uncheck 'Active in this subdomain'.

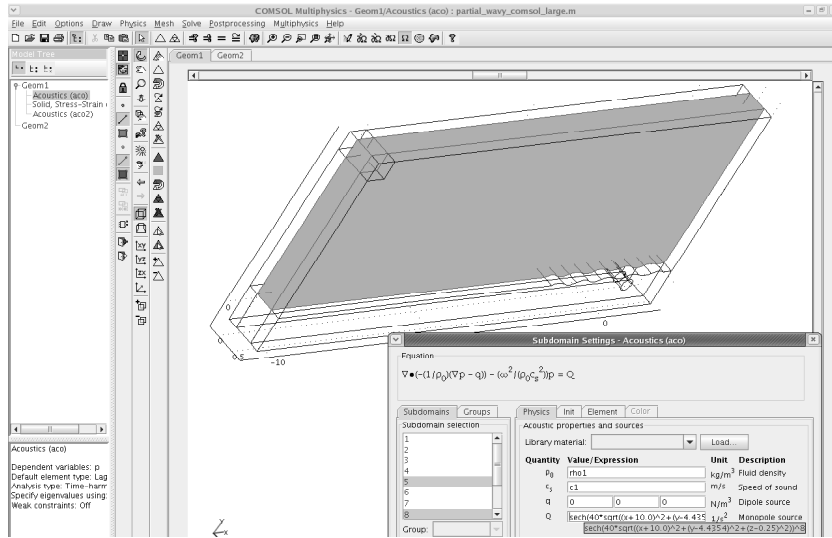


Figure 11: Subdomain settings for acoustic system.

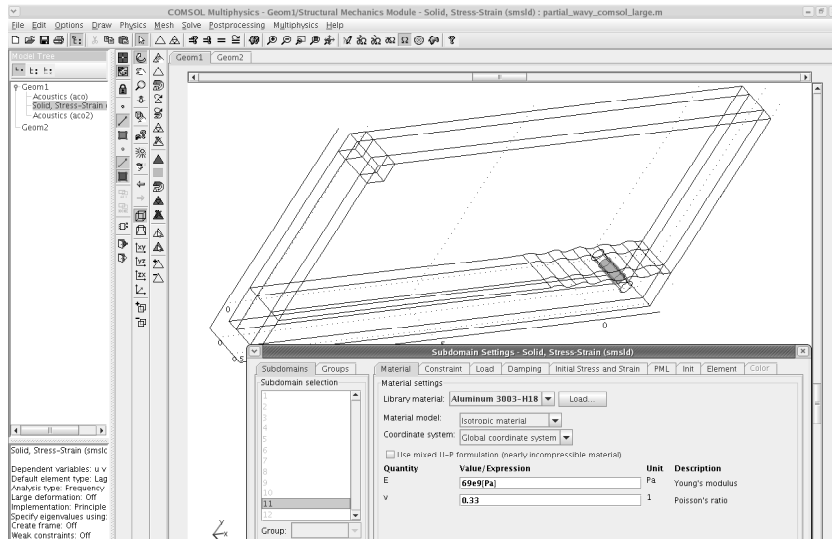


Figure 12: Subdomain settings for elastic target.

- On the 'Material settings' tab, click the 'load' button, and load the 'Aluminum 3003-H18' model.

Then we have the subdomain settings for the elastic target part as shown in Fig. 12.

2.6 Interface settings

First, we define the interface condition for the elastic target. This interface condition couples the acoustic pressure p from acoustic system 'aco' to the velocity (u, v, w) of the Structural Mechanics Module 'smsld'. It gives the continuity property of normal component of stress vector on the interface (the surface between elastic target and the sediment).

- From COMSOL menu Multiphysics, select Solid, Stress-Strain (smsld). Click COMSOL menu Physics > Boundary settings.
- In the pop-up window, select all the boundaries of the elastic target. On the tab 'Constraint', select 'Free' for the constraint condition.
- On the tab 'Load', select 'Global coordinate system'.
 - Type in '**-p*nx_smsld**' for F_x , '**-p*ny_smsld**' for F_y , '**-p*nz_smsld**' for F_z .
 - Type in **1** for F_{xAmp} , F_{yAmp} , F_{zAmp} .
 - Type in **0** for F_{xPh} , F_{yPh} , F_{zPh} .

See Fig. 13.

Secondly, for the acoustics system 'aco', we need define the continuity property of normal displacement of p on the interface.

- From COMSOL menu Multiphysics, select Acoustics (aco).
- Click COMSOL menu Physics > Boundary settings. In the pop-up window, select all the boundaries of the solid target.
- Select 'Normal acceleration' for the boundary condition.
Type in '**-omega_aco^2*(u*nx_smsld+v*ny_smsld+w*nz_smsld)**' for the acceleration value.

See Fig. 14.

2.7 Boundary settings

We need define the absorbing boundary conditions on the outside boundary $\partial\Pi$. Here we use the second order ABC. We need set up the faces, the edges and the corners on the boundary.

- For the faces on the boundary, from the COMSOL menu Physics, select Equation System, and then click Boundary Settings, in the pop-up window, select the tab 'Weak'. And then type in the weak form for the boundary faces. For example, we choose the boundary faces on the y-z plane, and then type in

$$\text{'-}(i*k_aco*p*p_test + i*(1/(2*k_aco))$$

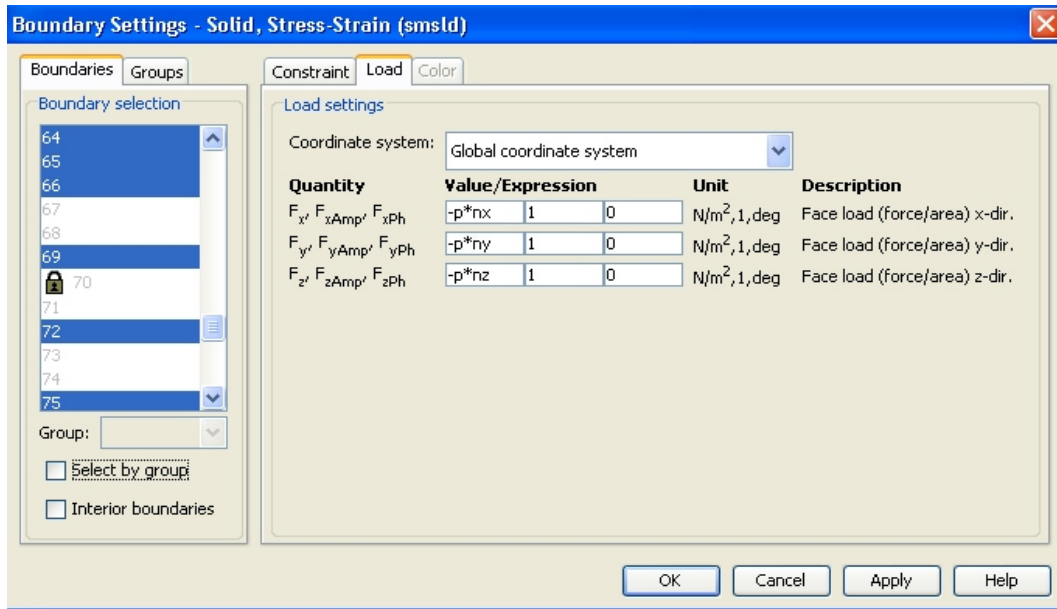


Figure 13: Interface condition setting up for elastic target.

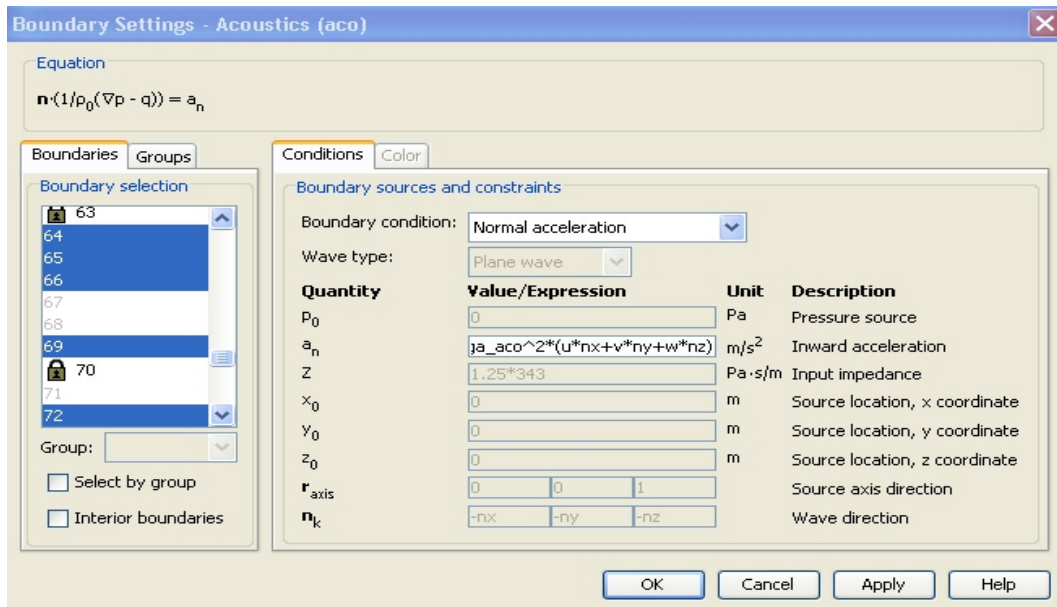


Figure 14: Interface condition setting up for acoustic system 'aco'.

$$*(pTy*test(pTy)+pTz*test(pTz)))/(rho_aco*omega_aco^2)'$$

in the first line of the pop-up six lines. For the sixth line of the pop-up six lines, type in

$$'-(i*k_aco^2*p3*p3_test+i*(1/(2*k_aco^2))$$

$$*(p3Ty*test(p3Ty)+p3Tz*test(p3Tz)))/(rho_aco^2*omega_aco^2^2)'$$

See Fig. 15.

- For the edges on the boundaries, from the COMSOL menu Physics, select Equation System, and then click Edge settings, in the pop-up window, select the tab 'Weak'. And

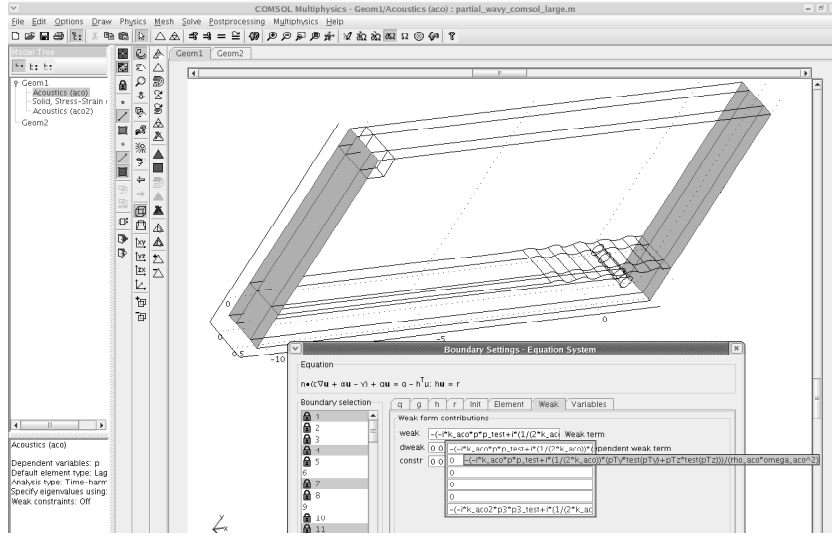


Figure 15: Boundary settings for the faces on y-z plane.

then type in the weak form for the boundary edges. For example, we choose the boundary edges on x-direction, and then type in

$$\begin{aligned} &'((1/(2*k_aco)^2)*pTx*test(pTx) \\ &- (3.0/4)*p*test(p))/(rho_aco*omega_aco^2)' \end{aligned}$$

in the first line of the pop-up six lines. For the sixth line of the pop-up six lines, type in

$$\begin{aligned} &'((1/(2*k_aco2)^2)*p3Tx*test(p3Tx) \\ &- (3.0/4)*p3*test(p3))/(rho_aco2*omega_aco2^2)' \end{aligned}$$

See Fig. 16.

- For the points at the domain corners, from the COMSOL menu Physics, select Equation System, and then click Points settings, in the pop-up window, select the tab 'Weak'. And then type in the weak form

$$'-(i/(2*k_aco))*p*test(p)/(rho_aco*omega_aco^2)'$$

in the first line of the pop-up six lines. For the sixth line of the pop-up six lines, type in

$$'-(i/(2*k_aco2))*p3*test(p3)/(rho_aco2*omega_aco2^2)'$$

See Fig. 17.

2.8 Solution

COMSOL gives us a wide selection of direct and indirect solvers. For this Time-harmonic problem, we use the iteration solver 'GMRES' with the 'Geometric Multigrid' preconditioner.

- Click the COMSOL menu Solve → Solver Parameters, from the pop-up window, on the 'General' Tab, choose 'Gmres' for the linear system solver, and 'Geometric Multigrid'

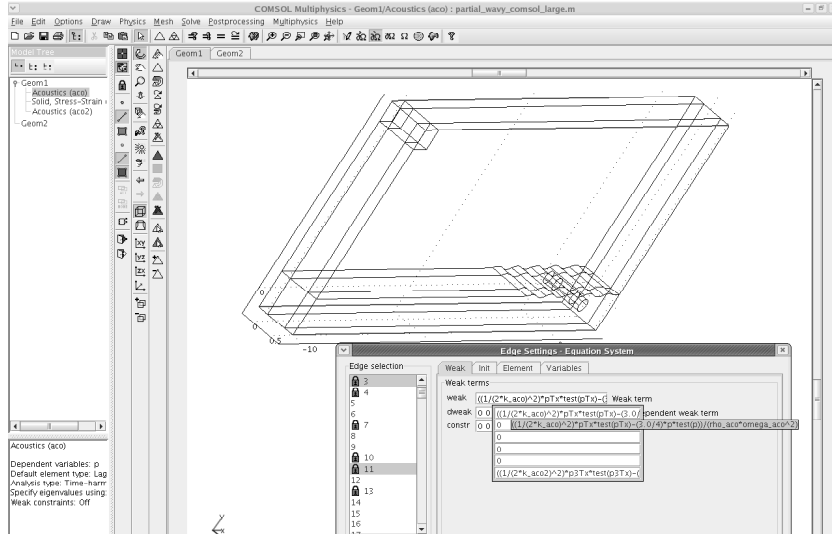


Figure 16: Boundary settings for the edges on x-direction.

for the preconditioner.

- Click the COMSOL menu Solve –> Solve Problem.

Then the system for the coupled acoustic and elastic system (p , u , v , w , p_2) has been solved. At the same time the acoustic system without the target (p_3) also has been solved.

2.9 Postprocessing

From the COMSOL menu 'Postprocessing', we can plot the total field (p) or the scatter field ($p-p_3$). They will plot the real part. If we wish to plot the imagine part or absolute value, just type in ' $\text{imag}(p)$ ' and ' $\text{abs}(p)$ ' for the total field, ' $\text{imag}(p-p_3)$ ' and ' $\text{abs}(p-p_3)$ ' for the scatter field. And because it is a 3D problem, if we wish to plot any cross-section of the 3D domain, just click Postprocessing –> Cross-Section plot parameters. On the pop-up window, select the 'slice' tab, select the 'slice plot' and type in the expression we wish to plot and also the coordinates for the three points on the plane which we wish to plot.

3 Coupling of Fortran exterior code and COMSOL structure acoustic code

With higher frequencies a finite element discretization leads to very large systems of linear equations. Often two-dimensional problems have millions of unknowns. It might be pos-

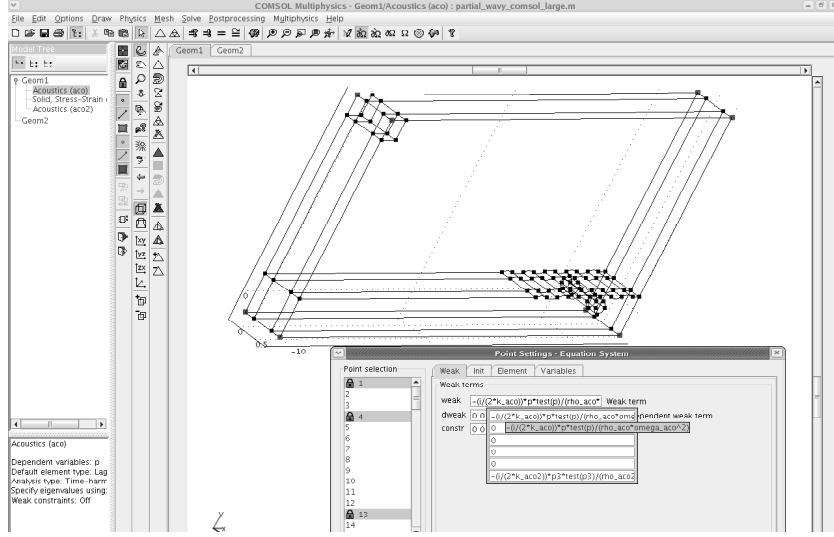


Figure 17: Boundary settings for the points at the domain corners.

sible to solve these problems using a LU factorization with a nested dissection reordering of unknowns, but this approach cannot be used for three-dimensional problems which can have billions of unknowns. For this reason, we consider the iterative solution of these problems. In the right preconditioned GMRES method we employ a domain decomposition preconditioner based on a domain embedding approach. A splorable preconditioner based on discretization of the perfectly vertically layered without an object is used. We discretize the time harmonic wave equation using linear finite elements on uniform rectangular meshes which are locally adapted to the wavy sediment interface and the surface of the object. In our solution procedure we employ a domain decomposition in which the near field subdomain is the interior of the dashed box in Figure 1 and the far field subdomain is the rest of the rectangle Π . Since the media is vertically layered with the wavy interface, our preconditioner coincides with the system matrix except the rows corresponding to unknowns near-by the interfaces. Thus, we can reduce iterations on a small sparse subspace. This reduction makes our iterative method extremely efficient and enables us to solve systems with billions of unknowns. Using the domain decomposition the resulting system of linear equations has the block form

$$Ax = \begin{pmatrix} A_{11} & A_{12} \\ A_{21} & A_{22} \end{pmatrix} \begin{pmatrix} x_1 \\ x_2 \end{pmatrix} = \begin{pmatrix} b_1 \\ b_2 \end{pmatrix} = b, \quad (3.1)$$

where the first block row corresponds the near field domain inside the dashed box in Figure 1 and the second block row corresponds the rest of Π . The matrix block A_{11} corresponds to an elastic object and its near field and it is formed using Comsol Multiphysics software. The matrix block A_{22} corresponds to the exterior scattering field discretized via the locally adapted finite element method. The matrix A has entries with imaginary part and it is

non-Hermitian. We take an advantage of the following facts. First, due to the small size of the near field subdomain linear systems with the matrix block A_{11} is solved by a Comsol linear equation solver. Second, let C be the corresponding matrix to of the perfectly vertically layered without an object. The direct inverse method is solve $C^{-1}b$ is developed in [2,3] and it requires $O(N \log N)$ floating point operations where N is the dimension of C .

With a right preconditioner B system (3.1) is equivalent to

$$AB^{-1}v = b. \quad (3.2)$$

And (3.2) can be written as

$$v + (A - B)B^{-1}v = b. \quad (3.3)$$

After solving this system the solution of the original problem is $x = B^{-1}v$. Here, our preconditioner has the upper block triangular form

$$B = \begin{pmatrix} A_{11} & A_{12} \\ 0 & C_{22} - C_{21}C_{11}^{-1}C_{12} \end{pmatrix}. \quad (3.4)$$

Note that $C_{22} - C_{21}C_{11}^{-1}C_{12}$ is the Schur complement of C on the second block and thus $(z_1, z_2) = B^{-1}v$ is carried out by performing

$$(*, z_2) = C^{-1}(0, v_2)$$

and then

$$z_1 = A_{11}^{-1}(v_1 - A_{12}z_2).$$

Now we describe the implement of the algorithm in detail. Hereafter, we use the different labels for the unknowns, i.e., we let the subscripts 1, 2, and Γ denote the unknowns on the near-field subdomain, the far-field subdomain, and the interface between them, respectively. Then the equation (3.1) is equivalent to be written as the following equation:

$$Ay = \begin{pmatrix} A_{11} & A_{1\Gamma} \\ A_{\Gamma 1} & A_{\Gamma\Gamma} & A_{\Gamma 2} \\ & A_{2\Gamma} & A_{22} \end{pmatrix} \begin{pmatrix} y_1 \\ y_\Gamma \\ y_2 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ f \end{pmatrix},$$

where f is due to the acoustic source in the far field domain. The matrix block $A_{\Gamma\Gamma}$ can be expressed as a sum

$$A_{\Gamma\Gamma} = A_{\Gamma\Gamma}^{(1)} + A_{\Gamma\Gamma}^{(2)},$$

where $A_{\Gamma\Gamma}^{(1)}$ and $A_{\Gamma\Gamma}^{(2)}$ contain the contributions from the near-field subdomain and the far-field subdomain, respectively. The right preconditioner (3.4) B and the separable preconditioner C can be written as the following block forms

$$B = \begin{pmatrix} A_{11} & A_{1\Gamma} \\ & B_{\Gamma\Gamma} & A_{\Gamma 2} \\ & A_{2\Gamma} & A_{22} \end{pmatrix}, \quad C = \begin{pmatrix} C_{11} & C_{1\Gamma} \\ C_{\Gamma 1} & A_{\Gamma\Gamma}^{(2)} & A_{\Gamma 2} \\ & A_{2\Gamma} & A_{22} \end{pmatrix}.$$

We solve (3.3) using the GMRES method and each GMRES iterate we need to perform multiplications $g = v + (A - B)B^{-1}v = AB^{-1}v$ with changing vectors v which will have the first block v_1 zero. In the first step of the iteration, we let $v = b$. Then each multiplication can be done in two steps:

1. Solve

$$C \begin{pmatrix} * \\ y_\Gamma \\ y_2 \end{pmatrix} = \begin{pmatrix} 0 \\ v_\Gamma \\ v_2 \end{pmatrix}$$

using a (partial solution) Fortran code.

2. Solve

$$A_{11}y_1 = -A_{1\Gamma}y_\Gamma \quad (3.5)$$

using the Comsol with the Dirichlet boundary condition value y_Γ .

It can be shown that only the Γ block of g is non-zero. It is given by

$$g_\Gamma = A_{\Gamma 1}y_1 + A_{\Gamma \Gamma}y_\Gamma + A_{\Gamma 2}y_2. \quad (3.6)$$

Now we describe how to solve (3.5) and how to carry out (3.6). The step (3.5) is done by Comsol as

```
fem.sol=femstatic(fem,'solcomp','w','u','p','v',  
'outcomp','w','u','p','v',... 'linsolver','gmres',..., 'rhob',1.0,  
'prefun','gmg','mcase',[0,1]),
```

where fem.sol is the solution inside Ω along with the boundary value y_Γ . Because the solution is not ordered, in order to compute g_Γ , we must perform the following sequence Comsol calls:

```
K = assemble(fem,'Out',{ 'K' }),
```

which returns the stiffness matrix K .

```
[Ke,Le,Null,ud] = femlin(fem),
```

which returns the matrices K_e and $Null$ which are

$$K_e = A_{11} \quad \text{and} \quad K_e = Null^T K Null,$$

where the block structure of $Null$ is compatible with the one of K .

```
Null2'*K*fem.sol.u;
```

computes

$$A_{\Gamma 1}y_1 + A_{\Gamma \Gamma}^{(1)}y_\Gamma,$$

and $Null2$ is the nonzero diagonal blocks of $Null*Null^T$.

4 Comparison of solutions of Coupled Code and COMSOL Code

We compared the solutions of the coupled code and to the COMSOL Multiphysics solution for a low frequency 1.5 kHz. See Fig. 18 for the comparison for the real and imagine part plot of the scatter fields. See Fig. 19 for the comparison for the absolute value plot of the scatter fields.

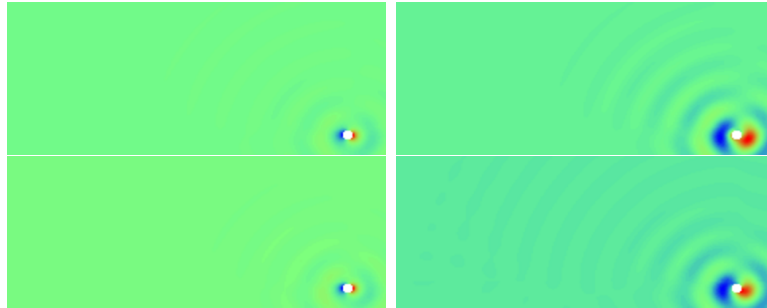


Figure 18: Real(left) and imagine(right) part of the scatter field from the COMSOL computation(up) and the coupled code(down).

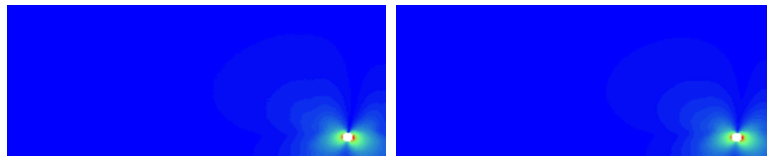


Figure 19: Absolute value of the scatter field from the COMSOL computation(left) and the coupled code(right).

And using this coupled code, we also solved (1.1) with frequencies 15 kHz and 20 kHz, which COMSOL cannot solve it. See Fig. 20 for the absolute value plot of the scatter fields.

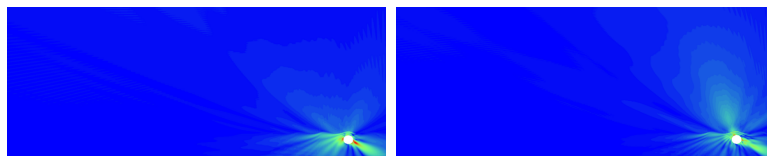


Figure 20: Absolute value of the scatter field from the coupled code with the frequency 15 kHz (left) and 20 kHz (right).

References

- [1] A. Bamberger, P. Joly and J. E. Roberts. Second-order absorbing boundary conditions for the wave equation: A solution for the corner problem. *SIAM J. Numer. Anal.*, 27 (1990), pp. 323-352.
- [2] E. Heikkola, T. Rossi and J. Toivanen. A parallel fictitious domain method for the three-dimension Helmholtz equation. *SIAM J. Sci. Comput.*, 24 (2003), pp. 1567-1588.
- [3] K. Ito and J. Toivanen. A fast iterative solver for scattering by elastic objects in layered media. *Applied Numerical Mathematics*, 57 (2007), pp. 811-820.
- [4] K. Ito and J. Toivanen. Preconditioned Iterative Methods on Sparse Subspaces. *Applied Mathematics Letters*, 19 (2006), pp. 1191-1197.