comsol computes normal stresses from the inputted total force by first dividing the total force by the surface area from the solid mechanics section these are the differing total forces applied to the surfaces the fixed boundary condition implies there is no displacement on the selected surfaces the fixed boundaries are highlighted in green, thanks for your comments it makes me relaxed indeed i believe that in most commercial codes pressure boundary condition has internal or user specified flexibility to adjust pressure values at each boundary node or face, pmc can be seen as the special case for blochs boundary condition periodic boundary condition where the k vector is set to zero in the corresponding direction pec line is the complement of pmc so in figure 10 and figure 11 the pmc line will become the pec line if the wave polarization is changed to tm, boundary condition definition a stated restriction usually in the form of an equation that limits the possible solutions to a differential equation see more boundary condition define boundary condition at dictionary com, impose my boundary conditions in comsol but when it comes to define the flux boundary condition it seems that it is important how the pde was implemented i e in terms of which term is, to reference it from a boundary condition on the domain across the assembly boundary suppose the variable is called v write it as src2dst ap1 v where ap1 is the name of the identity pair that has been created when making an assembly, release notes c omsol multiphysics version 4 3a contains many new functions and additions to the comsol product suite these release notes provide information regarding new functionality in existing products we have strived to achieve backward compatibility with the previous version and to include all functionality that is available there, the initial condition is zero concentration within the domain and the boundary conditions are set to 1 and 0 concentration at the left and right boundary respectively the physical interpretation of this is an initially sharp gradually diffusing front moving in the positive x direction, i am using the pde in comsol 4 4 and i need to set the dirichlet boundary condition on a face but the boundary condition only would be used under certain conditions, suppose if i have a geometry circular for example with triangular mesh and i apply a boundary condition lets say that this is a magnetic field problem can i then access the nodal values for e g the value of
magnetic vector potential at a given node nodes at any given time can i change the value how can the nodes be accessed, now we will set the boundary conditions for the diaphragm 1 all the four edges of the diaphragm are fixed 2 pressure is applied on the top surface hanging or free surface of the diaphragm select the file in the main menu of comsol multiphysics then select save as from the file menu a save window appears type the name of the, hi i have a question about floating potentials and overridden boundary conditions when solving laplace equation my system consists of a few metallic surfaces and some dielectric structures and i solve laplace equation for the surrounding area using dirichlet boundary conditions on the metallic surfaces, hi i m a chemical engineering student with a little problem with comsol multiphysics in practice i have to solve a problem of diffusion in a solid sphere after drawing the domain i have to set a boundary condition on sphere s surfaces this condition for my problem is flux kc cb c and cb bulk concentration is, this video guides you to implement the periodic boundary condition for the periodic structures the periodic condition node adds a periodic boundary condition this periodicity can be continuous, boundary conditions in heat transfer the following boundary conditions can be specified at outward and inner boundaries of the region known temperature boundary condition specifies a known value of temperature t 0 at the vertex or at the edge of the model for example on a liquid cooled surface t 0 value at the edge can be specified as a linear function of coordinates, i deal with pde module in comsol and i would like to simulate poisson s equation actually i need to simulate the distribution of the electric filed created by the spherical probe in the material, transition boundary condition lossy skin depth dependent penetration modeled in 2d conductive electrically much thicker than skin depth comsol can compute the radiated energy far field patterns losses gain directivity impedance and s parameters by solving the linear problem for the e field, periodic boundary conditions pbcs are a set of boundary conditions which are often chosen for approximating a large infinite system by using a small part called a unit cell pbcs are often used in computer simulations and mathematical models the topology of two dimensional pbc is equal to that of a world map of some video games the geometry of the unit cell satisfies perfect two, whenever you are modeling coils with the ac dc module in comsol multiphysics you need to consider what type of boundary conditions to use to truncate your modeling domain in this blog post we will introduce the different boundary conditions that you can use and how to choose between them as we, boundary conditions in fluid dynamics are the set of constraints to boundary
value problems in computational fluid dynamics these boundary conditions include inlet boundary conditions outlet boundary conditions wall boundary conditions constant pressure boundary conditions axisymmetric boundary conditions symmetric boundary conditions and periodic or cyclic boundary conditions, comsol multiphysics speaker impedance boundary condition surface plasmons demo the 5 steps of modeling bloch floquet eigenmode introduction to the wave optics module montana tech high, note the latter type of boundary condition with non zero q is called a mixed or radiation condition or robin condition and the term neumann condition is then reserved for the case q 0 other processes modeled by the comsol equation transversal deflection of membrane, when using a neumann boundary condition one prescribes the gradient normal to the boundary of a variable at the boundary e g \( \nu_x \) constant when using a mixed boundary condition a function of the form \( a \nu_x + b \) \( \nu_x \) constant is applied note that at a given boundary different types of boundary, specifying a rotating boundary condition posted 17 jan 2013 12 31 pm pst fluid computational fluid dynamics cfd modeling tools parameters variables amp functions version 4 2 12 replies, 11 4 2004 boundary conditions on perfect conductors doc 4 5 jim stiles the univ of kansas dept of eecs this more general boundary condition is useful for the dielectric conductor interface since 2 d r0 in the conductor we find that, select boundary settings from the physics menu and enter the boundary conditions according to the following table boundary 1 4 2 3 boundary condition wall no slip inflow outflow velocity outlet pressure no viscous stress \( u_0 \) \( v_0 \) \( p_0 \) 0 this is the selected boundary 1 which is a wall this is where you specify the boundary condition, if i am not mistaken the if statement exists under the geometry branch of the model tree if you are trying to change your geometry with respect to simulation variable t then i would suggest you to use a boundary condition like prescribed displacement, you can define a function e g foo that follows exactly your desired temperature with time profile then in the place where you specify your temperature whether it is a boundary condition or domain condition you insert foo t t being comsol s exclusive variable name for time, electromagnetics modeling in comsol rf module high frequency modeling microwave heating ac dc module statics and low frequency modeling induction heating magnetic shielding boundary condition for very efficient accurate modeling of thin sheets of high permeability materials similar shielding type of boundary, solved with comsol multiphysics 5 1 8 heat sink now define the physical properties of the model start with the fluid domain laminar flow spf on the physics toolbar click heat transfer ht and choose laminar
flow spf the no slip condition is the default boundary condition for the fluid define the inlet, in the first part of this blog series we discussed variational problems and demonstrated how to solve them using the comsol multiphysics software in that case we used simple built in boundary conditions today we will discuss more general boundary conditions and constraints, with boundary condition of dirichlet type \( u = 0 \) at all positions of the circle the analytical solution is given by \( u = 4x^2 + 4y^2 \). in case of quadratic lagrange elements the comsol default setting one obtains the exact solution in the interior the solution is a quadratic, robin boundary condition the robin boundary condition is a type of boundary condition named after victor gustave robin 1855-1897. it consists of a linear combination of the values of the field and its derivatives on the boundary given for example the laplace equation the boundary value problem with the dirichlet b.c. is written as,

\[
\text{Learn how to apply conditional boundary conditions for part of a boundary or only for certain instances in your comsol multiphysics model. How to make boundary conditions conditional in your simulation? In these cases among other instances, you may want to apply a boundary condition to only part of the geometrical boundary or only under, re comsol you can give dirichlet, neumann, or mixed bcs directly from physics gt boundary depending on your application mode or then you can go about it via physics gt equation systems and work with the boundary terms for the pde you're working with or define something specific for a boundary in question. If you need something more general, the ultraweak element will be available in comsol 4.0a. Generalized neumann boundary condition the q coefficient is not available in version 4.0. Combine the expressions for the q coefficient with the dependent variable for example use \( g = \frac{1}{1} u_1, \frac{1}{2} u_2 \) for the case of two dependent variables \( u_1 \) and \( u_2 \). Units, in transient analysis every boundary condition is active all the time unless you decide it should be otherwise. You can define any function to control your boundary condition. Heat flux is available under that name in comsol's heat transfer in fluids physics module. Subject: comsol users re open boundary condition lets see if i can explain this a bit better this time. The problem consists of an infinitely high water column in which an oil bubble is rising. The computational domain consists of a section of the water column of height \( h \) in which the bubble originates near the base and rises upward. Port boundary condition port bcs are very important and useful when working with total field. User guide doesn't provide us with very useful information what a surprise and above all, many of users just avoid reading them and try out iteratively to set parameters until they figure it out, the boundary
condition for the poisson equation is $v_0$ but this boundary condition is calculated by the solution of a very simple circuit equation $v_0 = v_{sup} i r i$ is the current that arises from the solution of continuity and poisson $r$ is the resistance and $v_{sup}$ a constant voltage if you still need help with comsol and have an on, comsol multiphysics cfd module the euler euler model turbulent flow interface a discontinuous galerkin formulation is applied for the dispersed phase gas boundary condition in order to switch from a dispersed phase gas outlet condition to a dispersed phase gas concentration condition on the parts of the outlet where backflow occurs, one way to create an acoustic source is to use an acceleration boundary condition which will produce sound through the vibration of the boundary you could set this to be a small boundary and set up its normal acceleration to get a desired simple source strength or make a baffled piston etc, boundary condition n mathematics an equation that specifies the behavior of the solution to a system of differential equations at the boundary of its domain thesaurus antonyms related wordssynonyms legend switch to new thesaurus noun 1 boundary condition mathematics a condition specified for the solution to a set of differential equations math, from what i understand the stress free boundary condition must be shear stress free boundary condition which can be symmetry or free slip wall analogically since there is no change in velocity perpendicular to the boundary unlike wall where we have significant velocity gradient the shear stress value should be zero negligible, imposing such constraints comsol multiphysics looks for a solution that satisfies the acoustical dynamics and the boundary condition impedance models are really models of a full flow that is a condition imposed simultaneously on the acoustic pressure and acoustic velocity defining a given relation between the two elements, i don t think you are restricted to only one boundary condition you can specify boundary conditions through constraint thereby you can set as many as you need on the other hand your problem seems to fit ac dc module in comsol why not try it
February 18th, 2019 - COMSOL computes normal stresses from the inputted total force by first dividing the total force by the surface area. From the solid mechanics section, these are the differing total forces applied to the surfaces. The fixed boundary condition implies there is no displacement on the selected surfaces. The fixed boundaries are highlighted in green.

**Pressure boundary condition CFD Online Discussion Forums**
April 15th, 2019 - Thanks for your comments; it makes me relaxed. Indeed, I believe that in most commercial codes, the pressure boundary condition has internal or user-specified flexibility to adjust pressure values at each boundary node or face.

**PMC PEC Boundary Conditions and Plane Wave Simulation**
April 16th, 2019 - PMC can be seen as the special case for Bloch’s boundary condition periodic boundary condition where the k vector is set to zero in the corresponding direction. PEC line is the complement of PMC. So in Figure 10 and Figure 11 the PMC line will become the PEC line if the wave polarization is changed to TM.

**Boundary condition Define Boundary dictionary com**
April 18th, 2019 - Boundary condition definition: a stated restriction usually in the form of an equation that limits the possible solutions to a differential equation. See more Boundary condition Define Boundary condition at Dictionary com.

**Impose my boundary conditions in COMSOL ResearchGate**
April 17th, 2019 - Imposing my boundary conditions in COMSOL. But when it comes to define the flux boundary condition, it seems that it is important how the PDE was implemented i.e., in terms of which term is involved.

**Top nine hints for working with Comsol evgeni.org**
April 17th, 2019 - To reference it from a boundary condition on the domain across the assembly boundary, suppose the variable is called V. Write it as src2dst ap1 V where ap1 is the name of the identity pair that has been created when making an assembly.

**COMSOL Multiphysics Release Notes**
April 15th, 2019 - Release Notes: COMSOL Multiphysics version 4.3a contains many new functions and additions to the COMSOL product suite. These Release Notes provide information regarding new functionality in existing products. We have strived to achieve backward compatibility with the previous version and to include all functionality that is available there.

**How to avoid negative concentrations COMSOL**
April 4th, 2019 - The initial condition is zero concentration within the domain and the boundary conditions are set to 1 and 0 concentration at the left and right boundary respectively. The physical interpretation of this is an initially sharp gradually diffusing front moving in the positive x direction.

**How can I set the if condition in COMSOL PDE boundary**
April 18th, 2019 - I am using the PDE in COMSOL 4.4. And I need to set the Dirichlet boundary condition on a face. But the boundary condition only would be used under certain conditions.

**Boundary Condition mesh nodes comsol fi**
April 5th, 2019 - Suppose if I have a geometry circular for example with triangular mesh and I apply a boundary condition. Let's say that this is a magnetic field problem. Can I then access the nodal values for e.g., the value of the magnetic vector potential at a given node nodes at any given time? Can I change the value? How can the nodes be accessed?

**COMSOL Example Diaphragm Start COMSOL Multiphysics 1**
April 14th, 2019 - Now we will set the boundary conditions for the diaphragm. 1. All the four edges of the diaphragm are fixed. 2. Pressure is applied on the top surface hanging or free surface of the diaphragm. Select the File in the main menu of COMSOL Multiphysics. Then select Save as from the file menu. A save window appears. Type the name of the
Floating potentials when solving Laplace equation comsol fr

April 12th, 2019 - Hi I have a question about floating potentials and overridden boundary conditions when solving Laplace equation My system consists of a few metallic surfaces and some dielectric structures and I solve Laplace equation for the surrounding area using Dirichlet Boundary conditions on the metallic surfaces

Problem with a boundary condition in Comsol Physics Forums

February 21st, 2009 - hi i m a chemical engineering student with a little problem with Comsol multiphysics in practice i have to solve a problem of diffusion in a solid sphere after drawing the domain i have to set a boundary condition on sphere s surfaces this condition for my problem is FLUX Kc Cb C and Cb bulk concentration is

Implementation of Periodic Boundary Condition COMSOL Multiphysics Tutorial 7

April 3rd, 2019 - This video guides you to implement the periodic boundary condition for the periodic structures The Periodic Condition node adds a periodic boundary condition This periodicity can be continuous

Boundary conditions in Heat transfer quickfield com

April 17th, 2019 - Boundary conditions in Heat transfer The following boundary conditions can be specified at outward and inner boundaries of the region Known temperature boundary condition specifies a known value of temperature T 0 at the vertex or at the edge of the model for example on a liquid cooled surface T 0 value at the edge can be specified as a linear function of coordinates

boundary condition in COMSOL Stack Exchange

March 20th, 2019 - I deal with PDE module in COMSOL and I would like to simulate Poisson s equation Actually I need to simulate the distribution of the electric filed created by the spherical probe in the material

RF simulations with COMSOL icps2017 it

April 12th, 2019 - -- Transition Boundary Condition lossy skin depth dependent penetration modeled in 2D • Conductive electrically much thicker than skin depth COMSOL can compute the radiated energy far field patterns losses gain directivity impedance and S parameters by solving the linear problem for the E field

Periodic boundary conditions Wikipedia

April 18th, 2019 - Periodic boundary conditions PBCs are a set of boundary conditions which are often chosen for approximating a large infinite system by using a small part called a unit cell PBCs are often used in computer simulations and mathematical models The topology of two dimensional PBC is equal to that of a world map of some video games the geometry of the unit cell satisfies perfect two

How to Choose Between Boundary Conditions for Coil

March 21st, 2016 - Whenever you are modeling coils with the AC DC Module in COMSOL Multiphysics you need to consider what type of boundary conditions to use to truncate your modeling domain In this blog post we will introduce the different boundary conditions that you can use and how to choose between them As we

Boundary conditions in fluid dynamics Wikipedia

April 14th, 2019 - Boundary conditions in fluid dynamics are the set of constraints to boundary value problems in computational fluid dynamics These boundary conditions include inlet boundary conditions outlet boundary conditions wall boundary conditions constant pressure boundary conditions axisymmetric boundary conditions symmetric boundary conditions and periodic or cyclic boundary conditions

Fresnel Equations COMSOL

April 6th, 2019 - COMSOL Multiphysics Speaker Impedance Boundary Condition Surface Plasmons Demo The 5 Steps of Modeling ÆBloch Floquet Eigenmode Introduction to the Wave Optics Module Montana Tech High

Introduction to PDE with Comsol

April 8th, 2019 - Note The latter type of boundary condition with non zero q is called a mixed or radiation condition or Robin condition and the term Neumann condition is then reserved for the case q 0 Other processes modeled by the Comsol equation Transversal deflection of membrane
Lecture 6 Boundary Conditions Applied Computational
April 18th, 2019 - When using a Neumann boundary condition one prescribes the gradient normal to the boundary of a variable at the boundary e.g. $\nabla u \times$ constant. When using a mixed boundary condition a function of the form $au + b \nabla u \times$ constant is applied. Note that at a given boundary different types of boundary conditions can be used.

Specifying a rotating boundary condition COMSOL
April 18th, 2019 - Specifying a rotating boundary condition. Posted 17 Jan 2013 12:31 pm PST Fluid Computational Fluid Dynamics CFD Modeling Tools Parameters Variables amp Functions Version 4.2 12 Replies

Boundary Conditions on Perfect Conductors KU ITTC
April 13th, 2019 - 11:42 2004 Boundary Conditions on Perfect Conductors doc 4.5 Jim Stiles The Univ of Kansas Dept of EECS. This more general boundary condition is useful for the dielectric conductor interface. Since 2D r0 in the conductor we find that

Introduction to COMSOL Rowan University
April 9th, 2019 - Select Boundary Settings from the Physics menu and enter the boundary conditions according to the following table:

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall</td>
<td>No slip</td>
</tr>
<tr>
<td>Inflow</td>
<td>Outlet</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure</td>
</tr>
</tbody>
</table>

This is the selected boundary 1 which is a wall. This is where you specify the boundary condition.

If Statement COMSOL
April 17th, 2019 - If I am not mistaken the If statement exists under the Geometry branch of the model tree. If you are trying to change your geometry with respect to simulation variable t then I would suggest you to use a boundary condition like Prescribed Displacement.

How I could make a temperature sweep in comsol Stack
April 14th, 2019 - You can define a function e.g. foo that follows exactly your desired temperature with time profile. Then in the place where you specify your temperature whether it is a boundary condition or domain condition you insert foo(t) being COMSOL’s exclusive variable name for time.

Electromagnetics Modeling in COMSOL Multiphysics
April 15th, 2019 - Electromagnetics Modeling in COMSOL • RF Module – High frequency modeling – Microwave Heating • AC DC Module – Statics and low frequency modeling – Induction Heating • Magnetic shielding boundary condition for very efficient accurate modeling of thin sheets of high permeability materials • Similar shielding type of boundary

Exercise 1 Intro to COMSOL Olin College
April 16th, 2019 - Solved with COMSOL Multiphysics 5.1.8 HEAT SINK. Now define the physical properties of the model. Start with the fluid domain LAMINAR FLOW SPF. On the Physics toolbar click Heat Transfer ht and choose Laminar Flow spf. The no slip condition is the default boundary condition for the fluid. Define the inlet.

Specifying Boundary Conditions and Constraints comsitol
April 15th, 2019 - In the first part of this blog series we discussed variational problems and demonstrated how to solve them using the COMSOL Multiphysics® software. In that case we used simple built in boundary conditions. Today we will discuss more general boundary conditions and constraints.

On the Convergence Order of COMSOL Solutions
April 10th, 2019 - with boundary condition of Dirichlet type $u = 0$ at all positions of the circle. The analytical solution is given by $u(x, y) = 1 + 4x^2 + 2y^2$. In case of quadratic Lagrange elements the COMSOL default setting one obtains the exact solution in the interior the solution is a quadratic.

What are Boundary Conditions — SimScale Documentation
April 17th, 2019 - Robin boundary condition. The Robin boundary condition is a type of boundary condition named after...
It consists of a linear combination of the values of the field and its derivatives on the boundary. Given for example the Laplace equation the boundary value problem with the Dirichlet b.c is written as

**How to Make Boundary Conditions Conditional in Your Simulation**

June 26th, 2016 - Learn how to apply conditional boundary conditions for part of a boundary or only for certain instances in your COMSOL Multiphysics model. How to Make Boundary Conditions Conditional in Your Simulation. In these cases among other instances you may want to apply a boundary condition to only part of the geometrical boundary or only under

**Set these boundary conditions in COMSOL 3.4 Physics Forums**

May 25th, 2008 - Re Comsol You can give Dirichlet Neumann or mixed BCs directly from physics gt boundary depending on your application mode or then you can go about it via physics gt equation systems and work with the boundary terms for the PDE you re working with or define something specific for a boundary in question if you need something more general

**COMSOL Users Open Boundary Condition**

September 14th, 2018 - Subject COMSOL Users Re Open Boundary Condition. Lets see if I can explain this a bit better this time. The problem consists of an infinitely high water column in which an oil bubble is rising. The computational domain consists of a section of the water column of height h in which the bubble originates near the base and rises upward

**PORT Boundary Condition 2D Introduction to nanoparticle**

April 4th, 2019 - PORT Boundary Condition. Port BCs are very important and useful when working with total field. User guide doesn’t provide us with very useful information. What a surprise and above all many of users just avoid reading them and try out iteratively to set parameters until they figure it out

**Boundary condition problem in comsol 3.5a**

April 15th, 2019 - The boundary condition for the Poisson equation is V=0. But this boundary condition is calculated by the solution of a very simple circuit equation V=0 Vsup I R. I is the current that arises from the solution of continuity and Poisson. R is the resistance and Vsup a constant voltage. If you still need help with COMSOL and have an on

**CFD Module COMSOL 5.1 Release Highlights**

April 13th, 2019 - COMSOL Multiphysics. CFD Module. The Euler Euler Model Turbulent Flow Interface. A Discontinuous Galerkin formulation is applied for the Dispersed Phase Gas Boundary Condition in order to switch from a Dispersed phase Gas outlet condition to a Dispersed phase Gas concentration condition on the parts of the outlet where backflow occurs

**Tutorial Created in Comsol 4.3 2012**

April 9th, 2019 - One way to create an acoustic source is to use an acceleration boundary condition which will produce sound through the vibration of the boundary. You could set this to be a small boundary and set up its normal acceleration to get a desired simple source strength or make a baffled piston etc

**Boundary condition definition of boundary condition by**

April 7th, 2019 - boundary condition n Mathematics. An equation that specifies the behavior of the solution to a system of
What is the meaning of stress free boundary condition
April 16th, 2019 - From what I understand the stress free boundary condition must be shear stress free boundary condition which can be symmetry or free slip wall Analogically since there is no change in velocity perpendicular to the boundary unlike wall where we have significant velocity gradient the shear stress value should be zero negligible

New Impedance Boundary Conditions for Acoustics
April 16th, 2019 - Imposing such constraints COMSOL Multiphysics looks for a solution that satisfies the acoustical dynamics and the boundary condition Impedance models are really models of a “full flow” that is a condition imposed simultaneously on the acoustic pressure and acoustic velocity defining a given relation between the two elements

boundary condition in COMSOL Stack Exchange
April 15th, 2019 - I don’t think you are restricted to only one boundary condition You can specify boundary conditions through constraint thereby you can set as many as you need On the other hand your problem seems to fit AC DC module in COMSOL Why not try it
comsol bme401 sec1 team4 2015 sites google com, pressure boundary condition cfd online discussion forums, pmc pec boundary conditions and plane wave simulation, boundary condition define boundary dictionary com, impose my boundary conditions in comsol researchgate, top nine hints for working with comsol evgeni org, comsol multiphysics release notes, how to avoid negative concentrations comsol, how can i set the if condition in comsol pde boundary, boundary condition mesh nodes comsol fi, comsol example diaphragm start comsol multiphysics 1, floating potentials when solving laplace equation comsol fr, problem with a boundary condition in comsol physics forums, implementation of periodic boundary condition comsol multiphysics tutorial 7, boundary conditions in heat transfer quickfield com, boundary condition in comsol stack exchange, rf simulations with comsol icps2017 it, periodic boundary conditions wikipedia, how to choose between boundary conditions for coil, boundary conditions in fluid dynamics wikipedia, fresnel equations comsol, introduction to pde with comsol, lecture 6 boundary conditions applied computational, specifying
a rotating boundary condition comsol, boundary conditions on perfect conductors
ku ittc, introduction to comsol rowan university, if statement comsol, how i could
make a temperature sweep in comsol stack, electromagnetics modeling in comsol
multiphysics, exercise 1 intro to comsol olin college, specifying boundary
conditions and constraints comsol it, on the convergence order of comsol
solutions, what are boundary conditions simscale documentation, how to make
boundary conditions conditional in your, set these boundary conditions in comsol
3 4 physics forums, comsol release notes comsol multiphysics, how to implement
constant heat flux boundary condition in, comsol users open boundary condition,
port boundary condition 2d introduction to nanoparticle, boundary condition
problem in comsol 3 5a, cfd module comsol 5 1 release highlights, tutorial created
in comsol 4 3 2012, boundary condition definition of boundary condition by, what
is the meaning of stress free boundary condition, new impedance boundary
conditions for acoustics, boundary condition in comsol stack exchange