Wavenology Cartesian Elastic Wave Solver Manual with Tutorial Examples

Wave Computation Technologies, Inc.

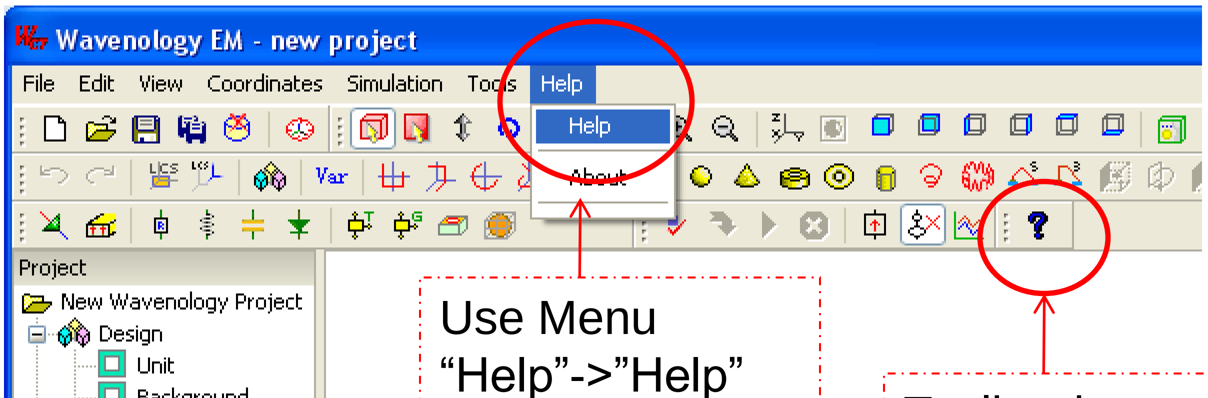
March, 2017

Outline

- GUI layout
- Project setting: Unit, Background, Boundary conditions, Frequency and Excitation Pulse types, Mesh control and Simulation timing.
- Mesh generation and displaying
- Exported mesh data format
- Material definition
  - general isotropic material - support visco-elastic parameter
  - general anisotropic material
  - weak anisotropic (Thomson) material
  - poro-elastic material
- Solid definition and operations
- Special treatment for importing SAT models for elastic wave project
- Variable System and complicated solids
- Define Source: EM source, elastic wave source
- Define point observer
- Define snapshot
- Simulation with Multi-threading
- Batch Simulation through Simulation Manager
- Tutorial cases
  - 1. Point Dipole Source in Homogenous Rock Background, General Isotropic material
  - 2. Point Moment Tensor Source in Layered Background with Soft boundary on Top
  - 3. Point Moment Tensor Source in Layered Background, Poro-elastic material
  - 4. Import 3D Solids from a SAT file

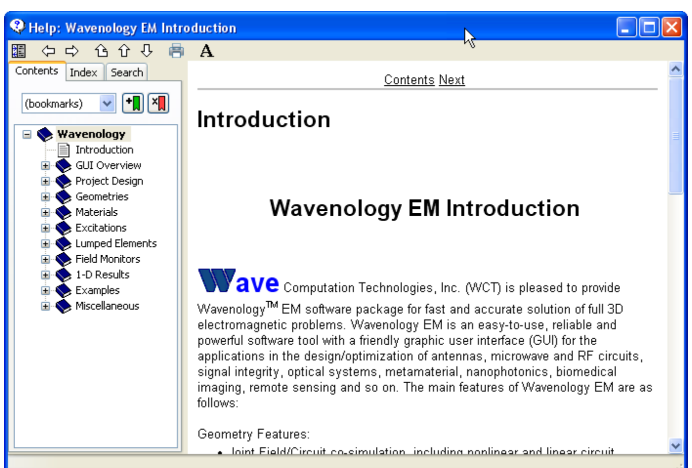
GUI Layout

- For the general GUI information, please refer to Wavenology EM Package manual (“Help”). The elastic wave solver will share most GUI system of Wavenology EM Package.



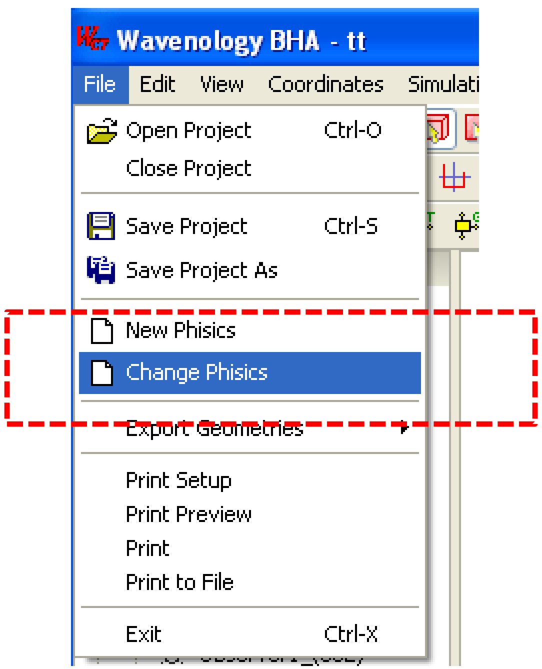
Use Menu “Help”->”Help”

Toolbar button “Help”



Wavenology EM Package manual

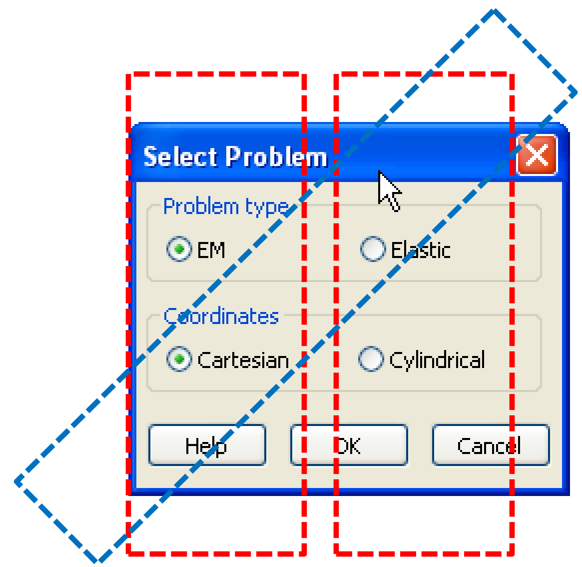
There are two menu items to define a project as a Cylindrical Elastic Wave project, a Cartesian Elastic Wave project, or a Cartesian EM project.



Create a new project

Change a existing project type

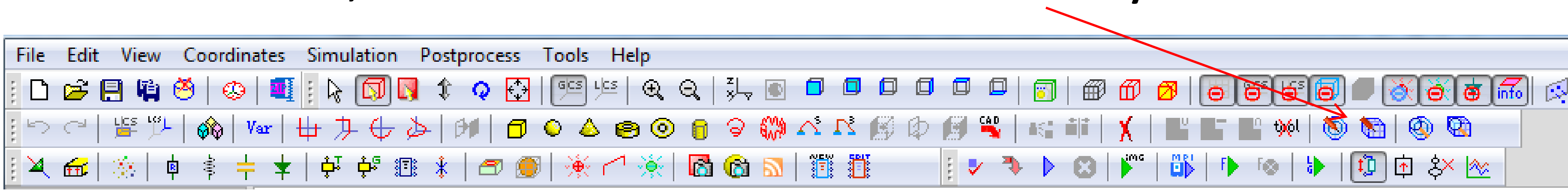
Cartesian Elastic Wave project



Cartesian EM project

Cylindrical Elastic Wave project

- We also support Cylindrical and Cartesian Grid Generator to generate, view and export mesh data.
- We do not have **Menu** for Cylindrical and Cartesian Grid Generator; we have **Toolbar buttons** only



General toolbar layout

Redo & Undo

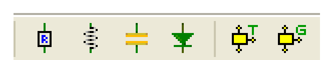
File operation

Solid and face selection

Port in EM solver

View operation

(note: we do not provide pan toolbar button for view, please use keyboard '->', '<-' etc. to pan the view)



Lumped circuit in EM solver

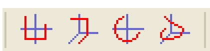
Extended view operation

Simulation control

Curve, solid creation and project setting

Project setting

Variable creation and edit

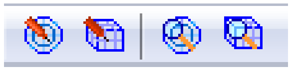


Curve creation

3D solid creation

Insert 3D solid from ***SAT*** file

3D solid BOOLEAN operation, including UNION, SUBTRACTION & INTERSECTION



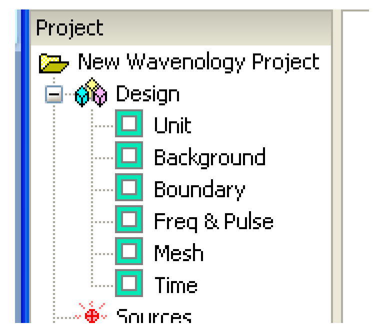
Cylindrical and Cartesian mesh generation and display

Generate cylindrical grid and export the data file

Generate Cartesian grid and export the data file

Display the cylindrical mesh

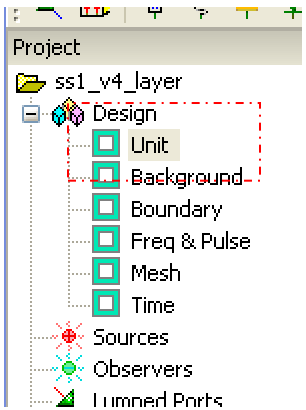
Display the Cartesian mesh



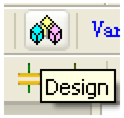
Project Setting

- In Wavenology GUI, we define a problem as “a Project”
- For grid generation purpose, a project includes
  – Units
  – Materials
  – 3D Solids
  – Mesh settings
    - Cylindrical mesh control for cylindrical grid
    - Cartesian mesh control for Cartesian grid

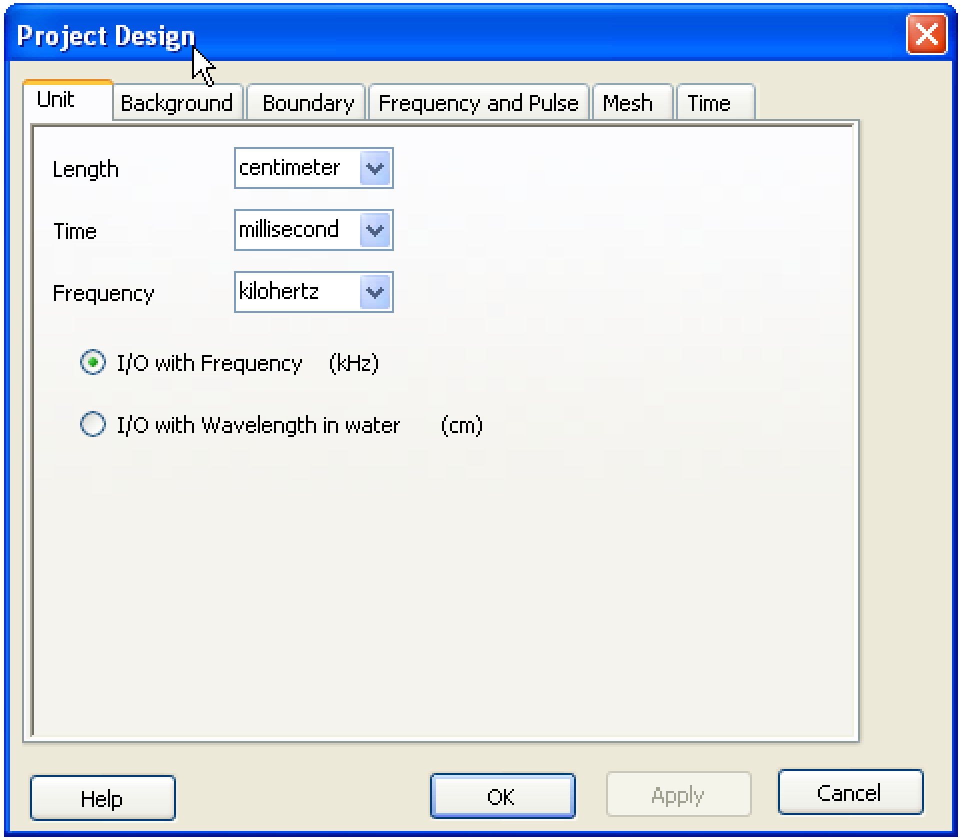
Project Unit



Treenode ***Unit***



toolbar button
"***Design***"



User double click treenode "Unit" or click toolbar button "Design" will enter the ***Unit*** page of ***Project Setting dialog***

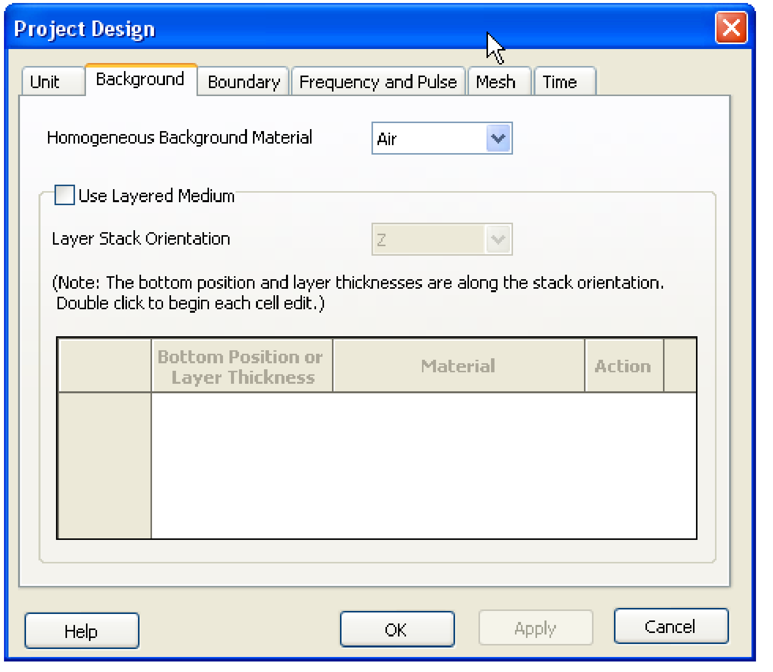
The default frequency unit is **Kilohertz**, for elastic wave problem.

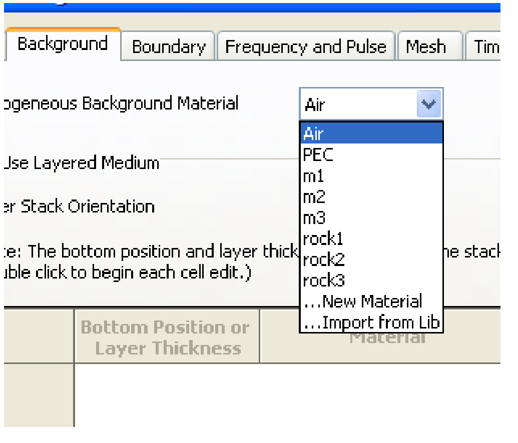
Project Background Material

User needs to set up the background medium of the problem.

The default background for EM problem is a homogeneous medium: Air. For borehole acoustic, the background is water.

User can change it to other medium by using the ***material combobox***



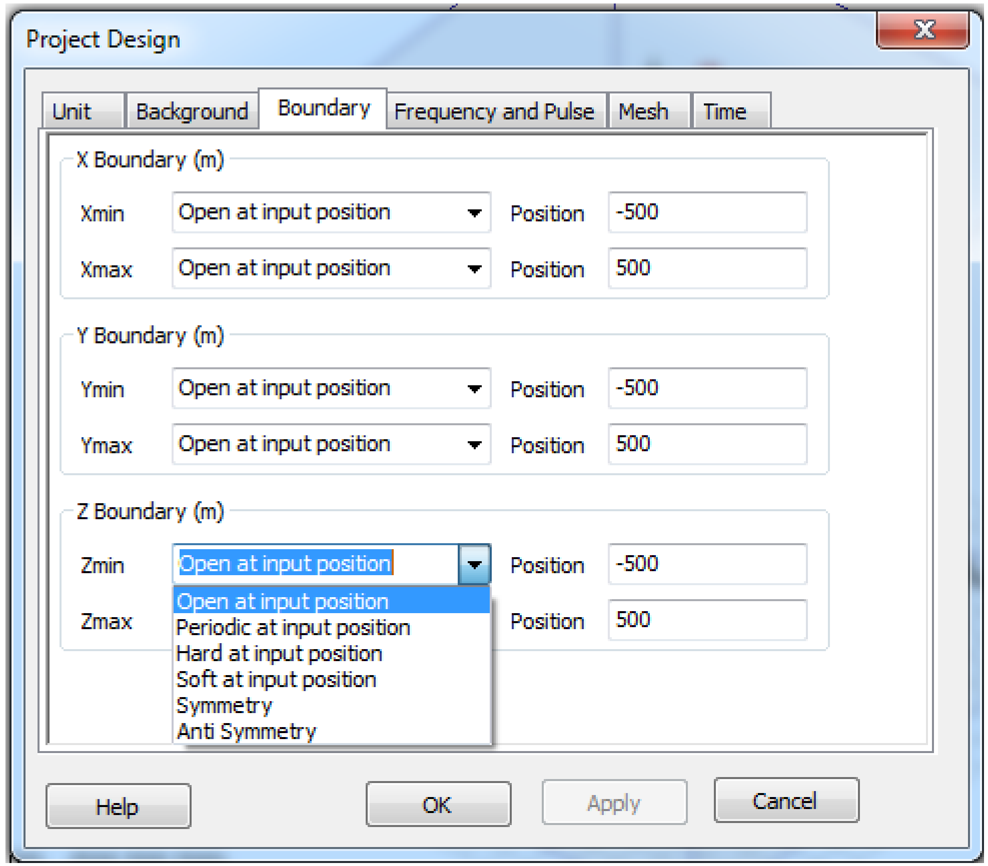


Project Boundary Conditions

User needs to set up the boundary conditions for the problem.

Currently we support absorbing B.C. only (Open at input position), soft & hard B.C. for the Cartesian elastic wave solver.

For Periodic, Symmetry, Anti Symmetry B.C., they are not supported even they are listed.

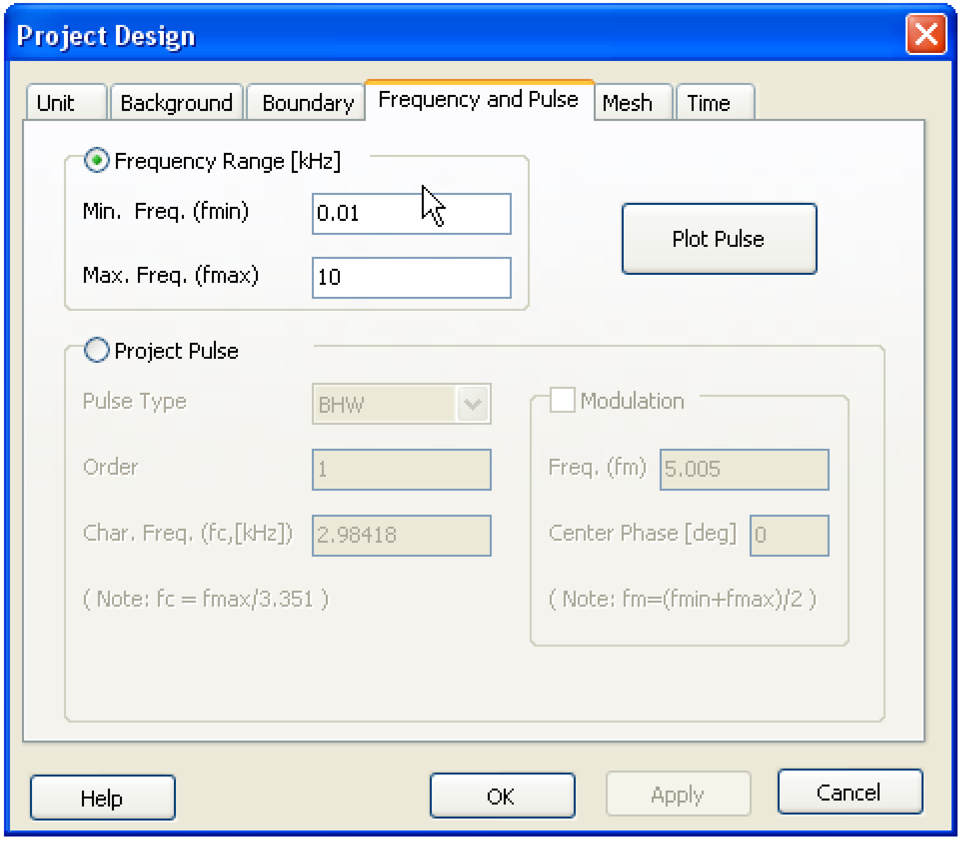


Project Working Frequency and Excitation Pulse Types

User needs to set up the maximum working frequency (fmax) of the problem.
The sampling density of mesh is determined by **fmax**.
Current mesh generator does not use **fmin**, so just input a value such as fmin=0.01***fmax**.

The setting for fmin & fmax is the "**Frequency and Pulse**" page of the ***Project Setting dialog***.

For the excitation pulse types, please refer to the **Wavenology EM Package manual.**



Mesh Setting

In this version, Wavenology GUI supports two types of mesh, Cylindrical mesh and Cartesian mesh.

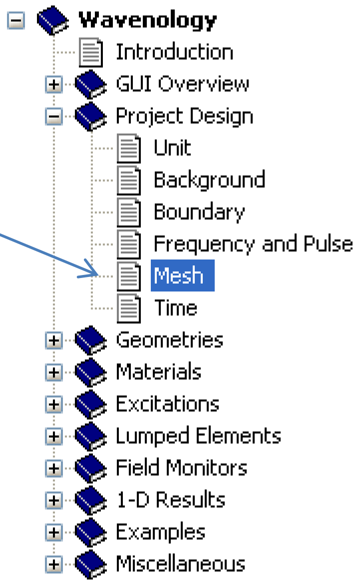
These two meshes are independent, each having its own setting. But they share the same meshing ideas:
1. All need to limit the space size in 3 dimensions.
2. For each dimension, there is an independent sampling density.
3. They use the same fmax in the **Project Setting**.

Before a mesh generation, user needs to set up the corresponding mesh control, but for a particular type of mesh only.

For example, to generate a Cartesian mesh, user only needs to set up Cartesian mesh control; he does not need to set up mesh Cylindrical control.

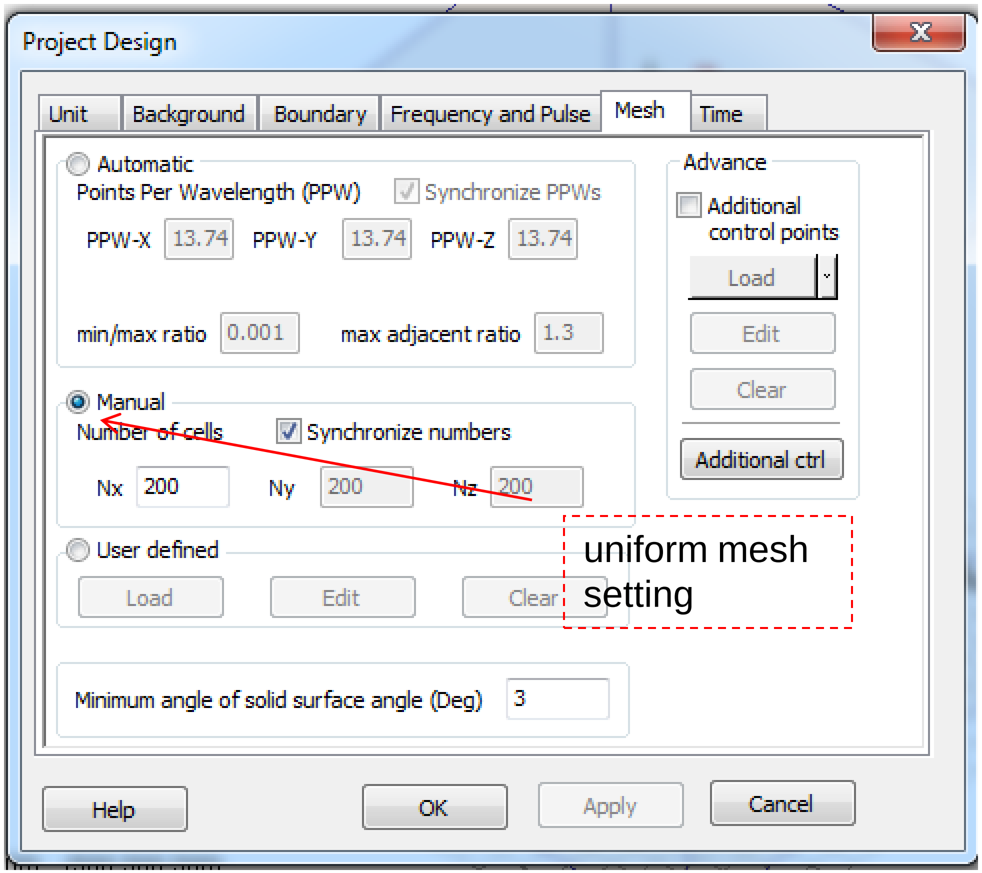
Cartesian Mesh Setting

For the setting of the Cartesian mesh, please refer to the “Mesh” section in the manual of Wavenology EM package



Cartesian Mesh Setting

In this version, for Cartesian elastic wave problem, we support uniform mesh only.



The sampling density in ***X***, ***Y*** and ***Z*** directions.

Cell ratios: please refer to the **Mesh** section of the Wavenology EM package manual.

User needs to input how many uniform cells will be created along ***X***, ***Y*** and ***Z*** direction, respectively.

How to create the triangular mesh for the solid surface for meshing purpose.
The comment is in the next page.

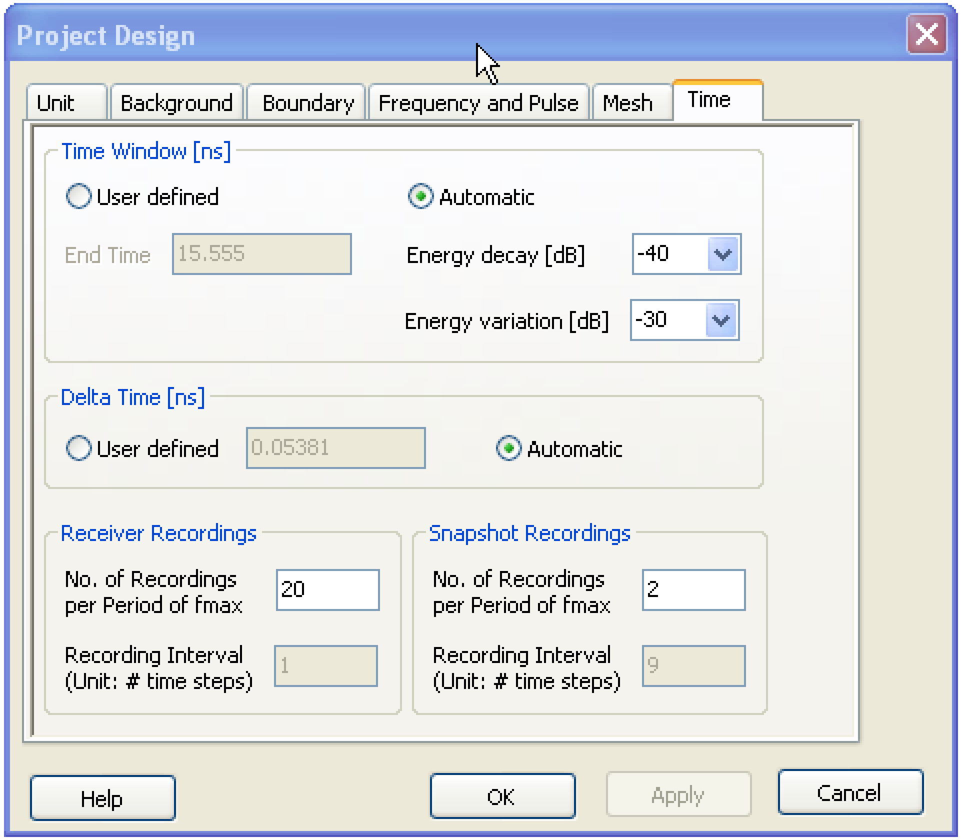
Project Simulation Timing Setup

- There are 3 timing parameters user need to set up for a simulation
  1) Total simulation time windows

     User can specify a time window or using a automatic time window by energy decay level.
  2) Stepping Delta T

     User can specify a Δt or use automatic Δt. User input Δt should be less than automatic Δt. Otherwise, the solver will prevent the simulation.
  3) Receiver and snapshot recording intervals

     User should setup how many time steps to record the data once for receiver and snapshot.



Mesh generation

After the mesh control has been set up correctly, user can generate grid by clicking following toolbar buttons.

Generate cylindrical mesh and export the data file:

cyl_mesh.txt	cell material index, ascii format
material_info.txt	material name and index , ascii format
cyl_mesh_grid.m	grid position, matlab m file

Generate Cartesian mesh and export the data file

cart_mesh.txt	cell material index, ascii format
material_info.txt	material name and index , ascii format
cart_mesh_grid.m	grid position, matlab m file

Note: the exported files has the same names for all cases. The files from a new generation will replace the old data files.
It is suggested to use different folders to save different cases.

Mesh displaying

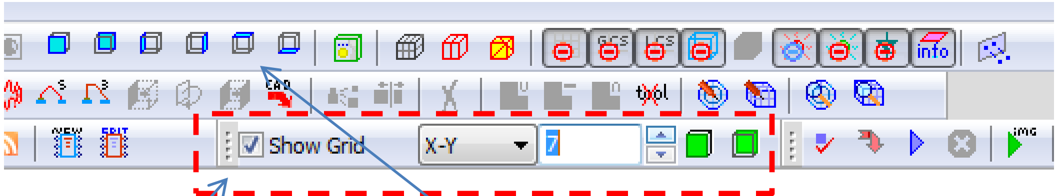
User can use Wavenology GUI to check the mesh result directly by displaying the mesh on the canvas.

 Show the cylindrical mesh

 Show the Cartesian mesh

After click one of the “**Show mesh**” buttons, the main canvas will switch to “**Mesh Displaying**” mode. The two coordinate system modes has similar toolbar system, and only the definition of plane-normal is different.

In following pages, we demonstrate how to display a Cartesian mesh.

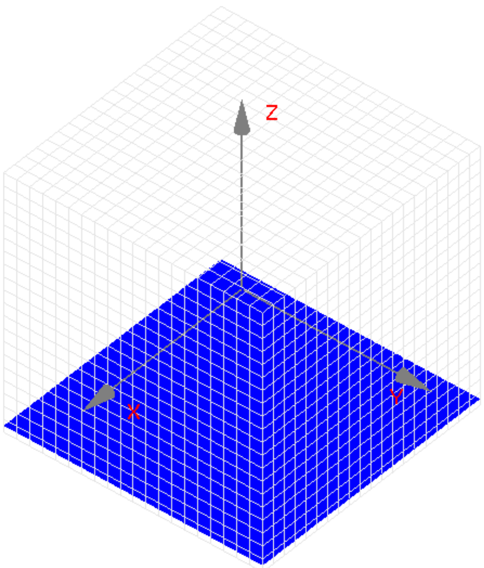


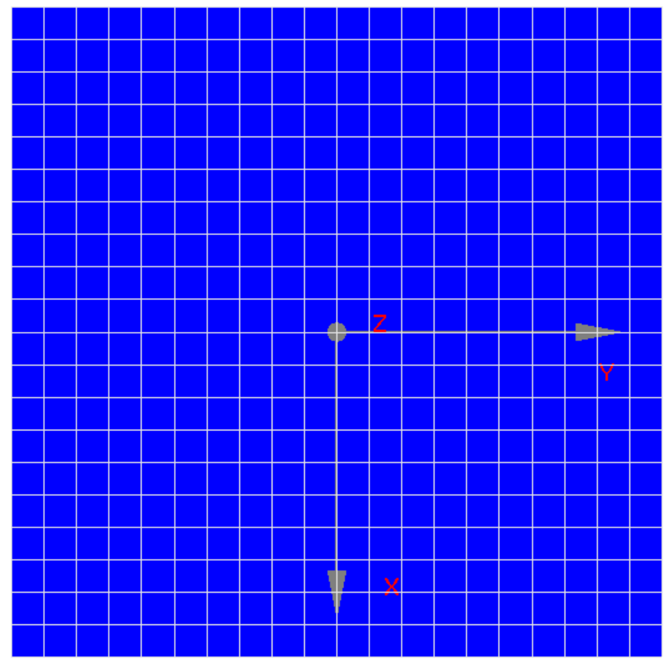
Mesh displaying controls

The grid can be hidden by uncheck

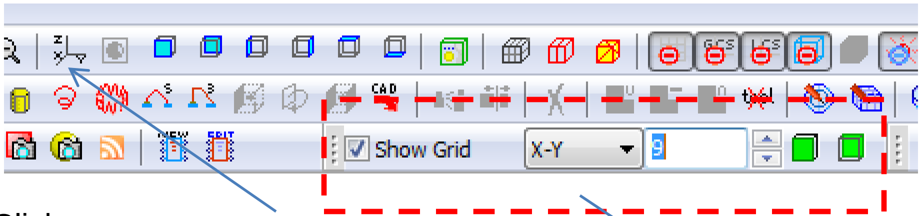
By click top view mode

User can check whether the cell-material is correct or not





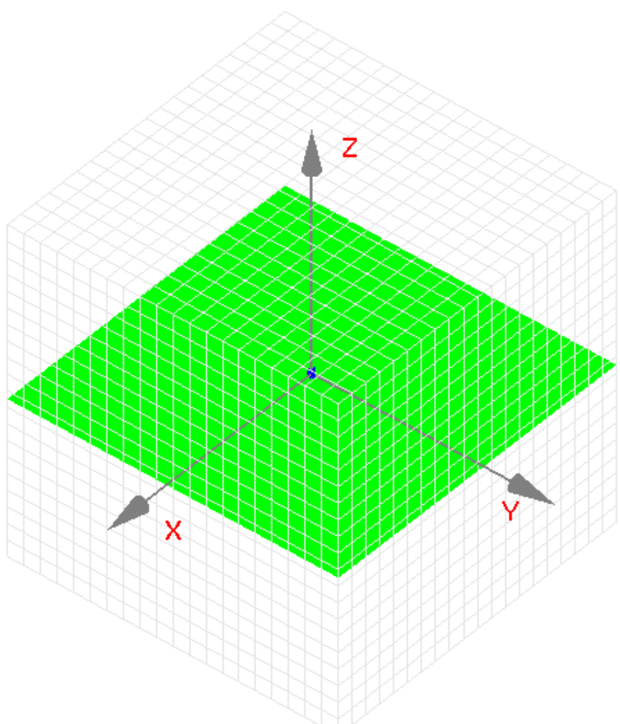
The default displaying mode is lowest Z (X-Y plane) cells with grid line shown.

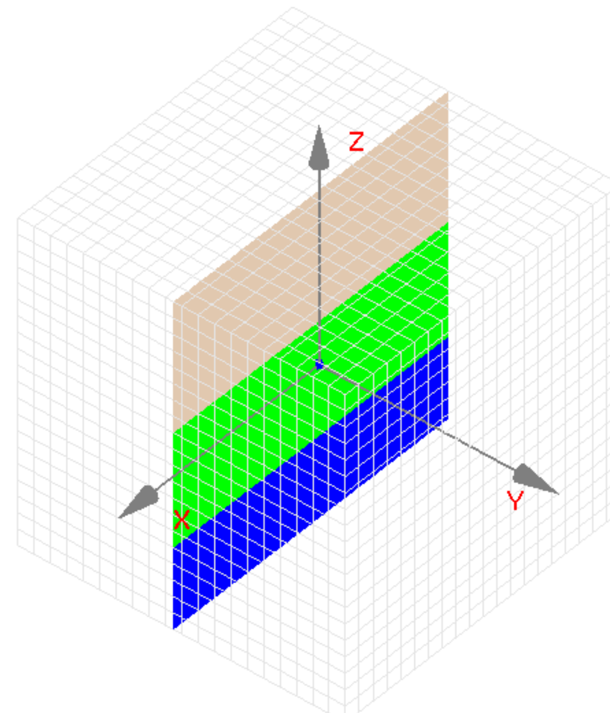


Click
reset view mode 

Change the section index.
A different mesh-section is displayed

The canvas return back the angle view mode.





Note: the displaying color for each cell is determined by the material in that cell. The material color is defined as one of the material property in the **Material Editing Dialog**

Exported Mesh Data File Format

material_info.txt material name and index , ascii format

```
% id  name    eps_r    e_cond       mu_r        m_cond    ……
0     Air           1    0    1    0     ….
1     PEC           1    0    1    0
2     m1            2    0    1    0
3     m2            1    0    1    0
4     m3            1    0    1    0
5     rock1         2    0    1    0
6     rock2         4    0    1    0
7     rock3         3    0    1    0
```

1st column is the material index in this case, not matter this material has been used for any solid or background

2nd column is the material name

other columns are electric-profile of the material. We will add more parameters to support elastic wave solver in the future.

We will provide matlab codes to demonstrate how to load these files and display the mesh in matlab.

cyl_mesh.txt & cart_mesh.txt cell material index, ascii format

data	meaning
N	How many cells along R(cylindrical mesh:cyl_mesh.txt) or X(Cartesian mesh:cart_mesh.txt)
next N rows	Cell centers along R or X axis
M	Cells number along Phi or Y axis
next M rows	Cell centers along Phi or Y axis
K	Cells number along Z axis
next K rows	Cell centers along Z axis

next N*M*K rows	Material index for each cell, the index is the same as the file *material_info.txt*
	
	The data is created by the "for-loop"
	For R
	    For Phi
	        For Z
	        ...
	        end Z
	    end Phi
	End R

cyl_mesh_grid.m & *cart_mesh_grid.m* grid position, matlab m file

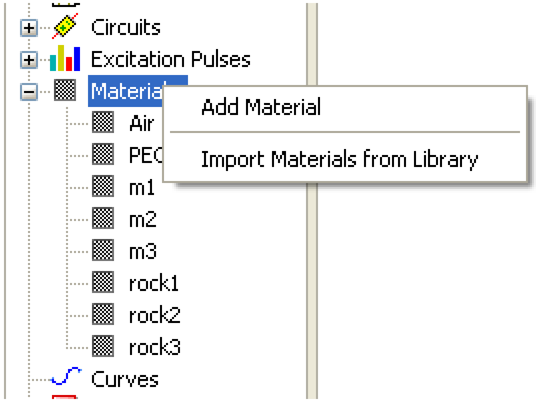
User can directly run these files in matlab. After running, user can get following variables:

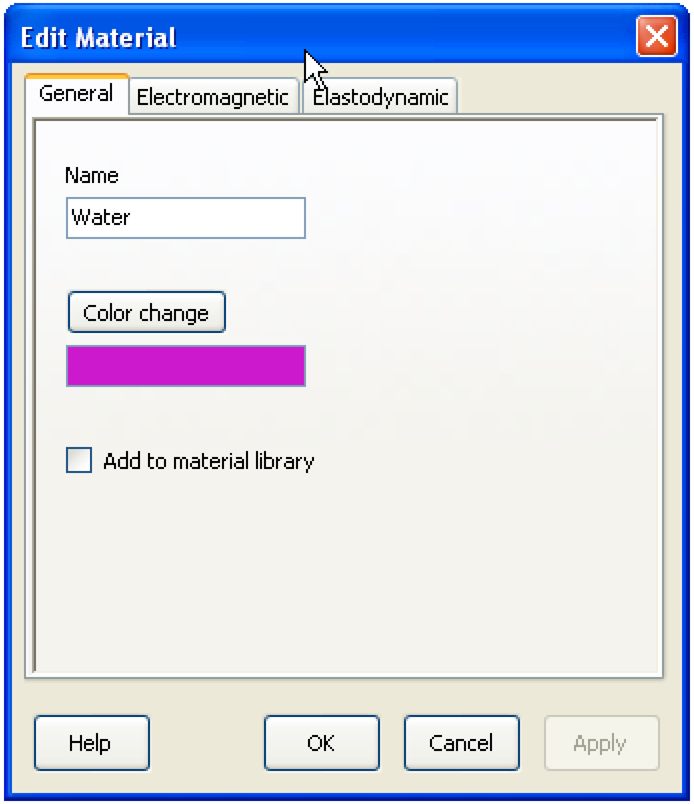
NR or NX:	grid number in R or X axis
NPhi or NY:	grid number in Phi or Y axis
NZ:	grid number in Z axis
Rgrid or Xgrid:	grid position in R or X axis
Phigrid or Ygrid:	grid position in Phi or Y axis
Zgrid:	grid position in Z axis

The content of the file is easy to understand even with Notepad.

Material definition

User can assign different materials to different solids. To create a new material is through the menu attached on the tree-node "Material"

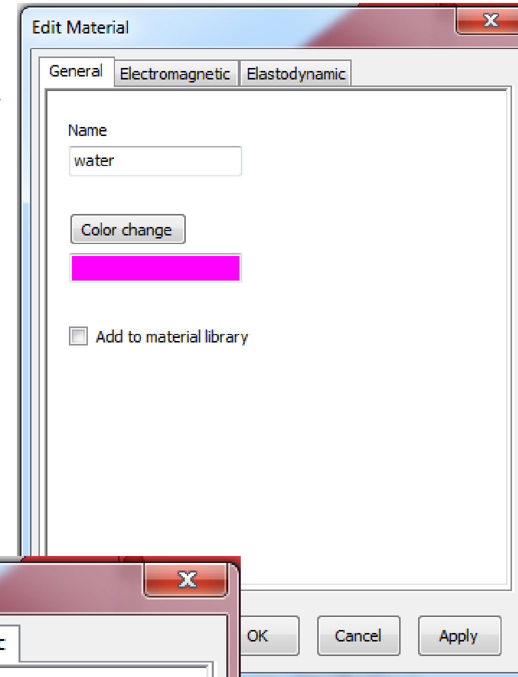


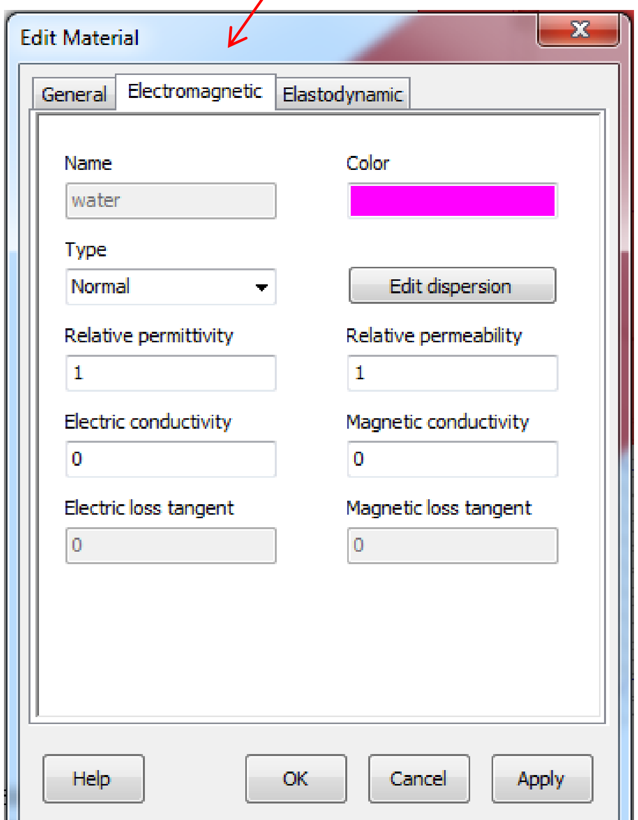


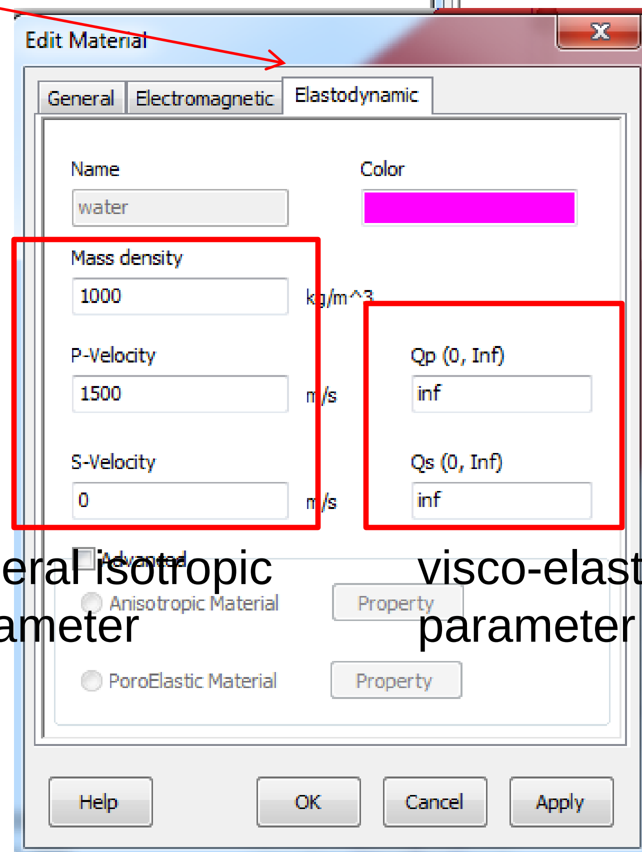
The menu appears by right click mouse button on the treenode **Material**, the material **Edit Dialog** will be popuped by choosing "**Add Material**" menu.

There are three kinds of properties for a material:

1. ***General definition:*** name and displaying color
2. Parameters for **electromagnetic profile**: relative permittivity and electric conductivity, etc.
3. Parameters for **elastic wave**: mass density and velocities, etc.





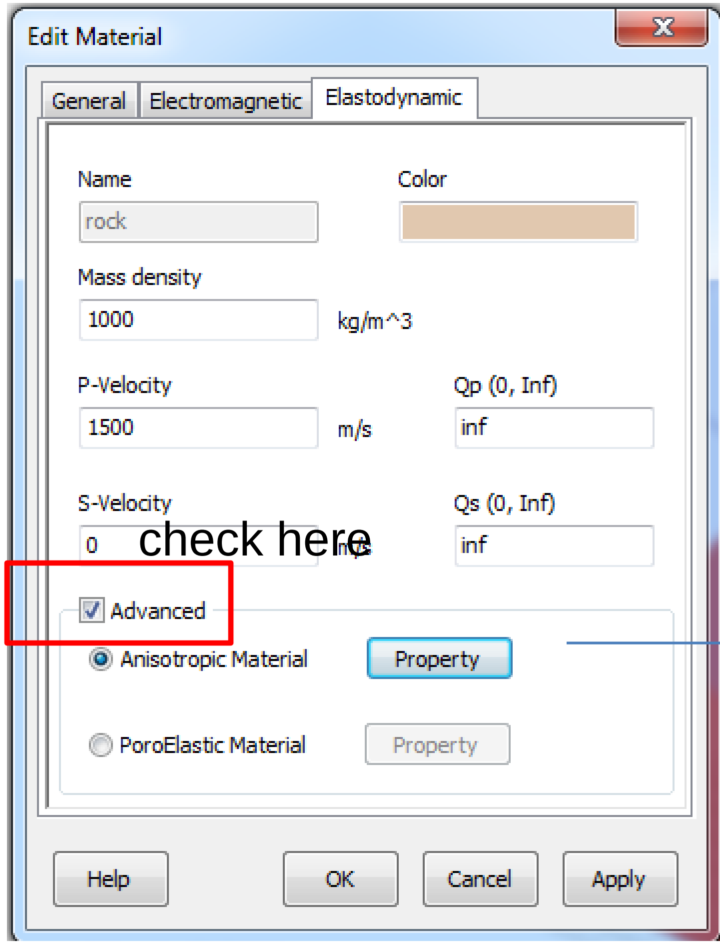


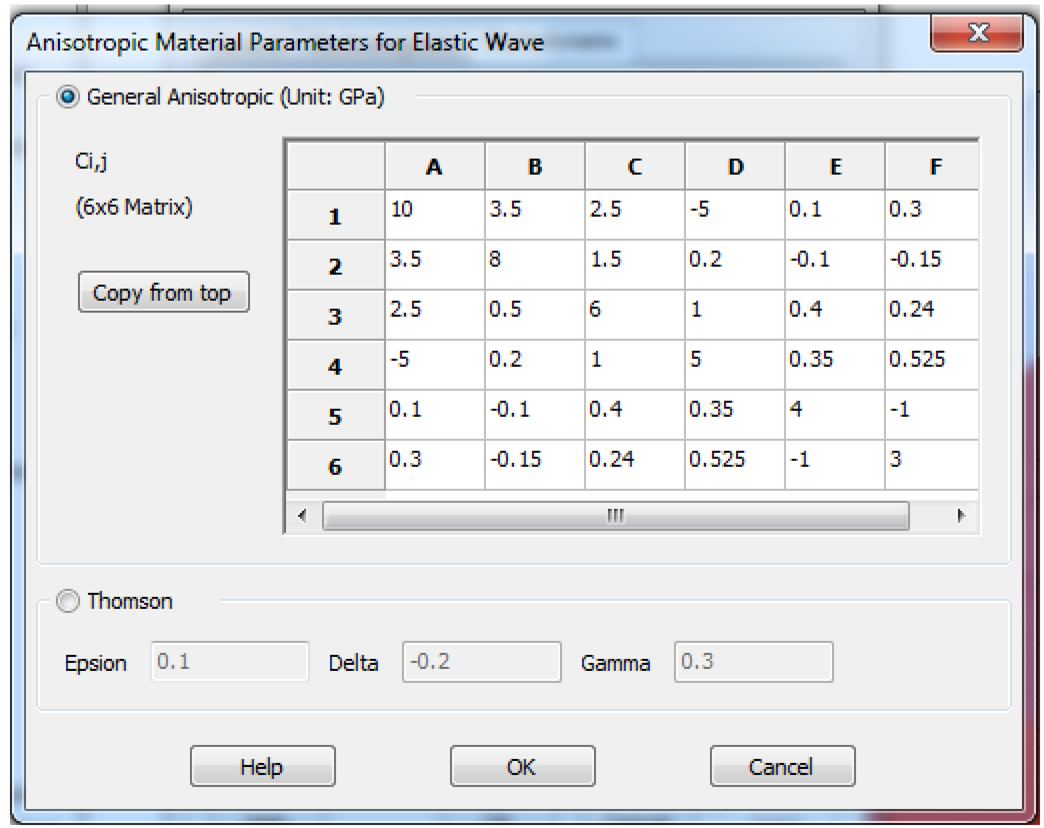
general isotropic parameter

visco-elastic parameter

Advanced Parameters for **Cartesian elastic wave simulation**

1) General Anisotropic Property Cij

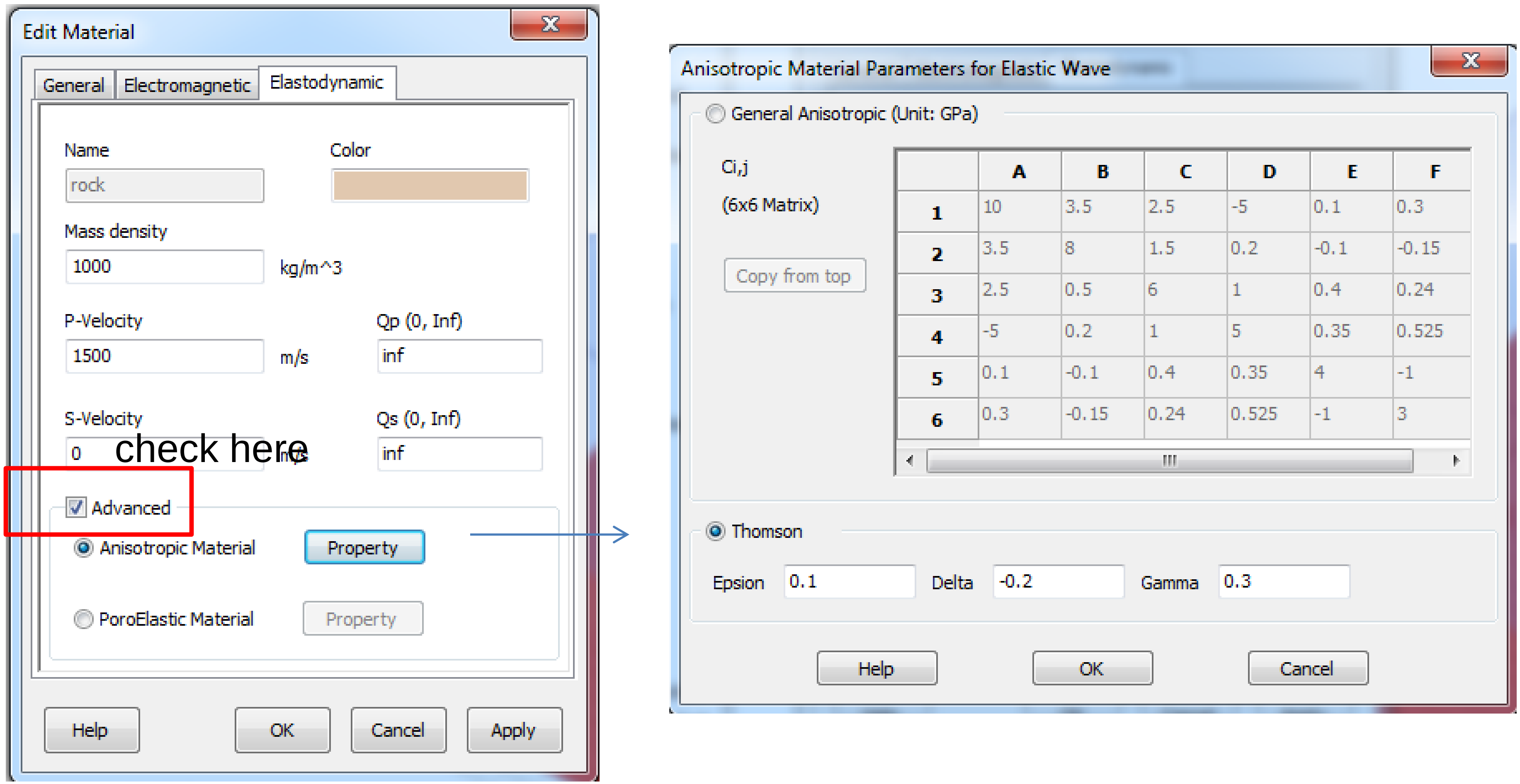




	A	B	C	D	E	F
1	10	3.5	2.5	-5	0.1	0.3
2	3.5	8	1.5	0.2	-0.1	-0.15
3	2.5	0.5	6	1	0.4	0.24
4	-5	0.2	1	5	0.35	0.525
5	0.1	-0.1	0.4	0.35	4	-1
6	0.3	-0.15	0.24	0.525	-1	3

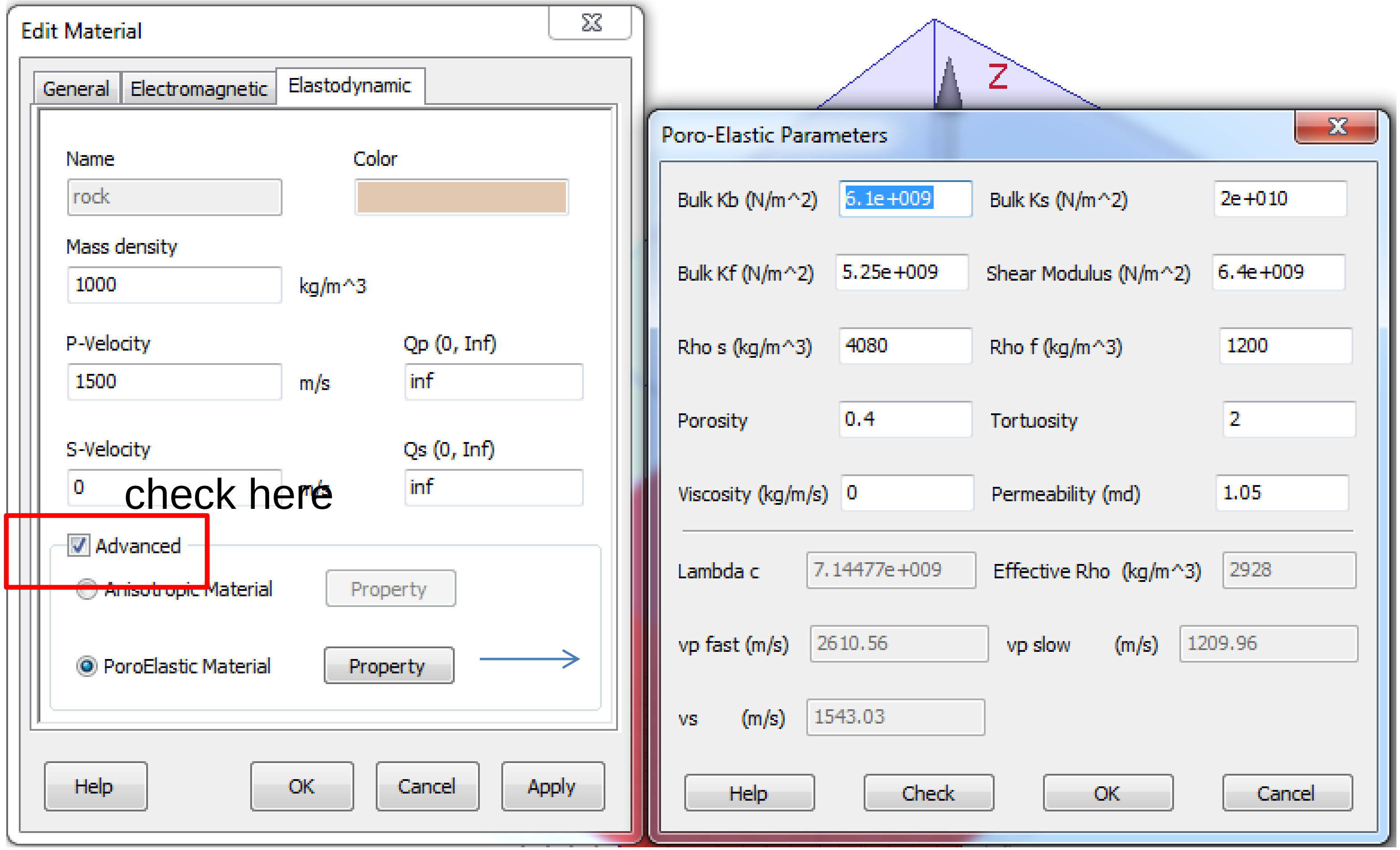
Advanced Parameters for **elastic wave**

2) Weak Anisotropic (Thomson) Property

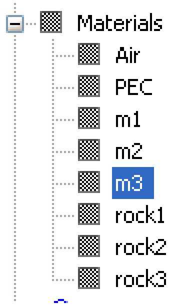


Advanced Parameters for **elastic wave**

3) Poro-Elastic Property



Material Modification

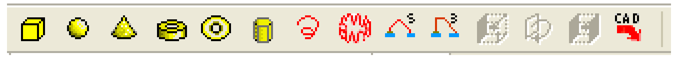


To modify a material's property, user needs to select the material first (as the highlighted material tree-node item), then double click this item. The material **Edit Dialog** will be poped up.

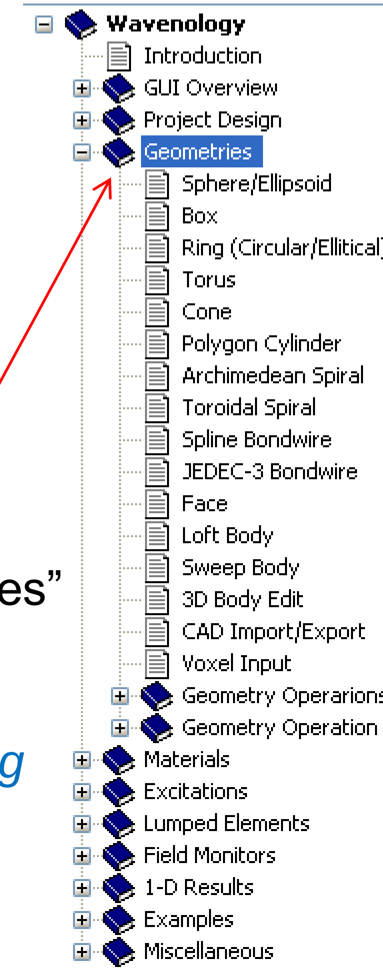
3D Solid Definition

Wavenology GUI supports following basic geometries:

- ***Sphere/Ellipsoid***
- ***Box (Brick)***
- ***Ring (Circular/Elliptical)***
- ***Torus***
- ***Cone***
- ***Polygon***
- ***Cylinder Archimedean***
- ***Spiral***
- ***Toroidal***
- ***Spiral***
- ***Spline***
- ***Bondwire***
- ***JEDEC-3 Bondwire***
- ***CAD solid Import/Export***

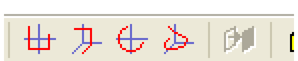


Basic solid creation toolbar buttons



For how to create and edit these solids please refer to the "Geometries" section in the manual of Wavenology EM package.

Wavenology EM package also support complicated solid by sweeping or lofting faces. More details will be covered in the next manual.



2D & 3D curve creation, and cover planar closed curves as a face

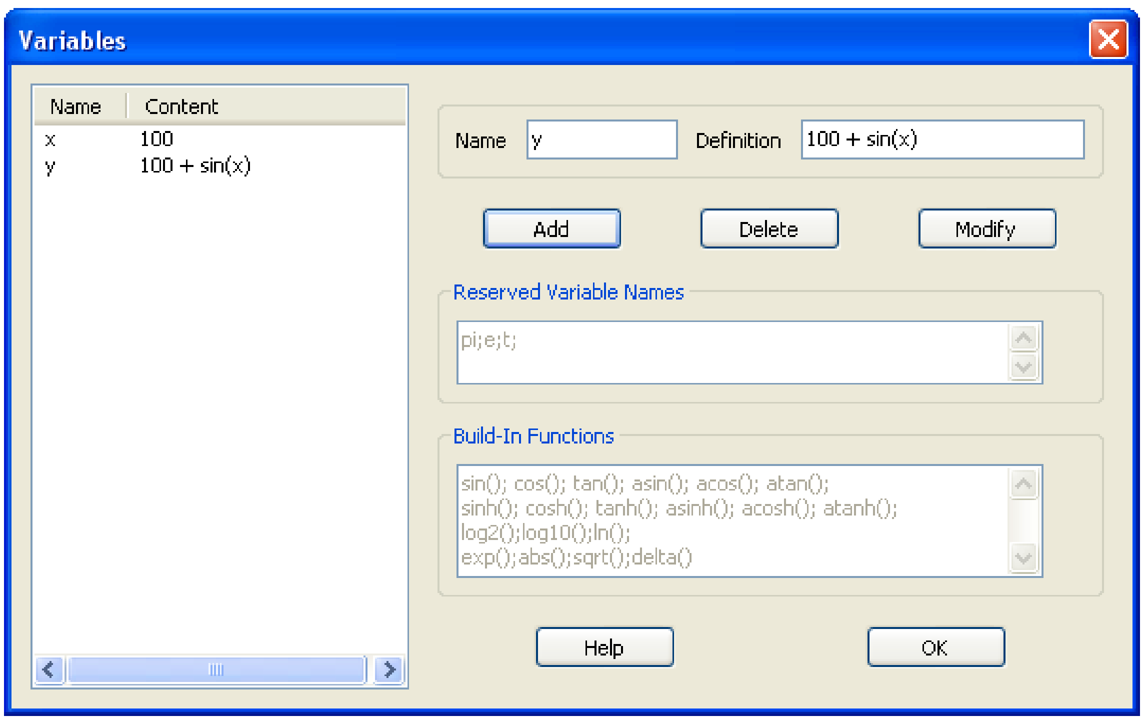
Create solid by sweeping or lofting face

Variable System

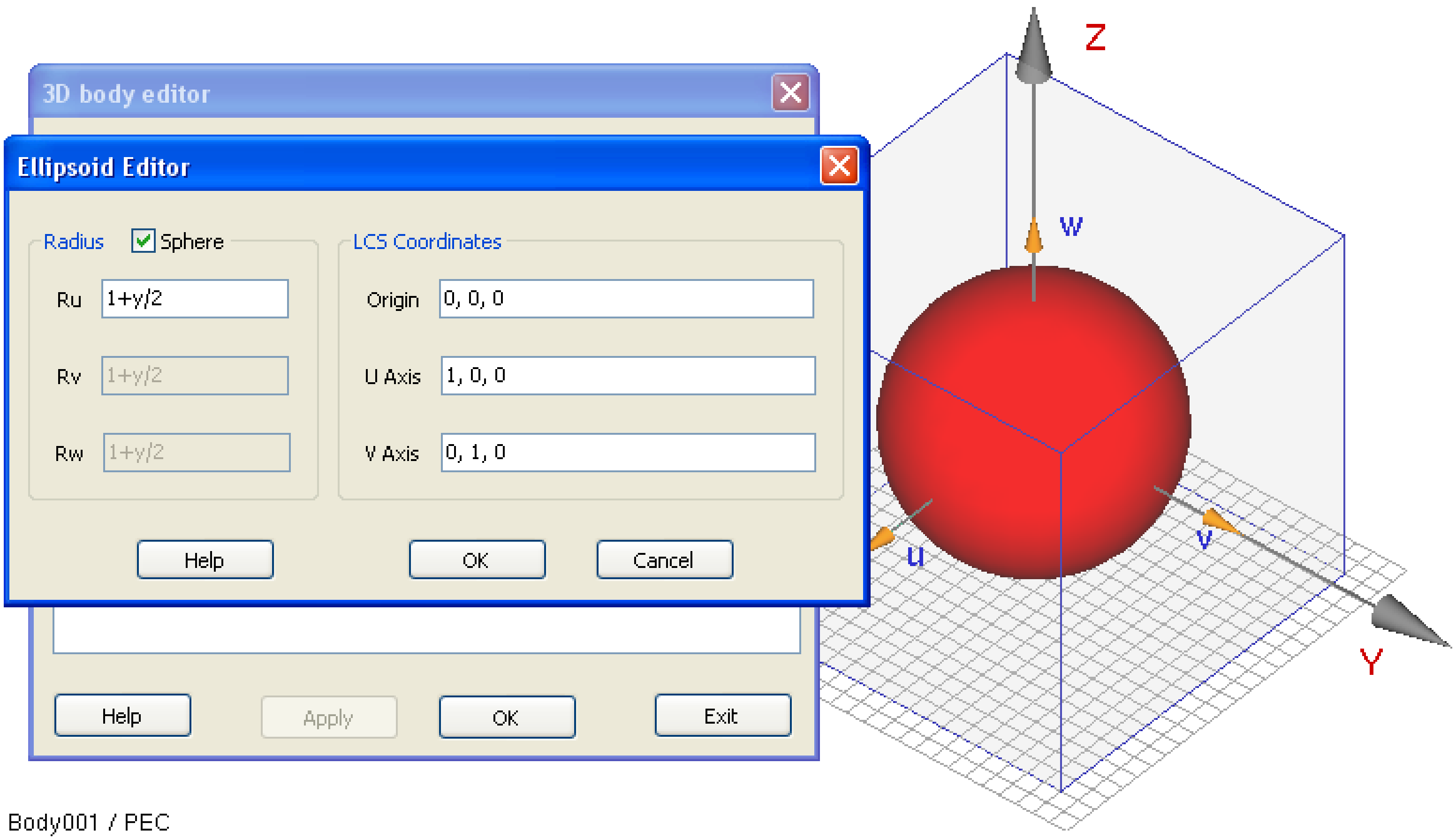
All 3D geometries in Wavenology GUI can be defined by variables. For example, user can define a sphere with radius as variable “x”. The sphere’s shape can change as the value of “x” changes.

User can define and modify a variable by toolbar button 

In following example, we define variable “***x***” as value *100* and variable “***y***” equal to “*100 + sin(x)*”



Then, we can define a sphere with radius as “1+y/2”



Body001 / PEC

bounding box: (-50.7468,-50.7468,-50.7468),(50.7468,50.7468,50.7468)

size: (101.494 x 101.494 x 101.494) , center: (0,0,0)

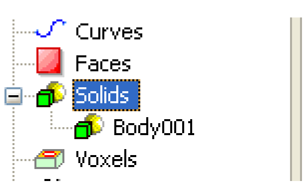
PEC

Solid with Components

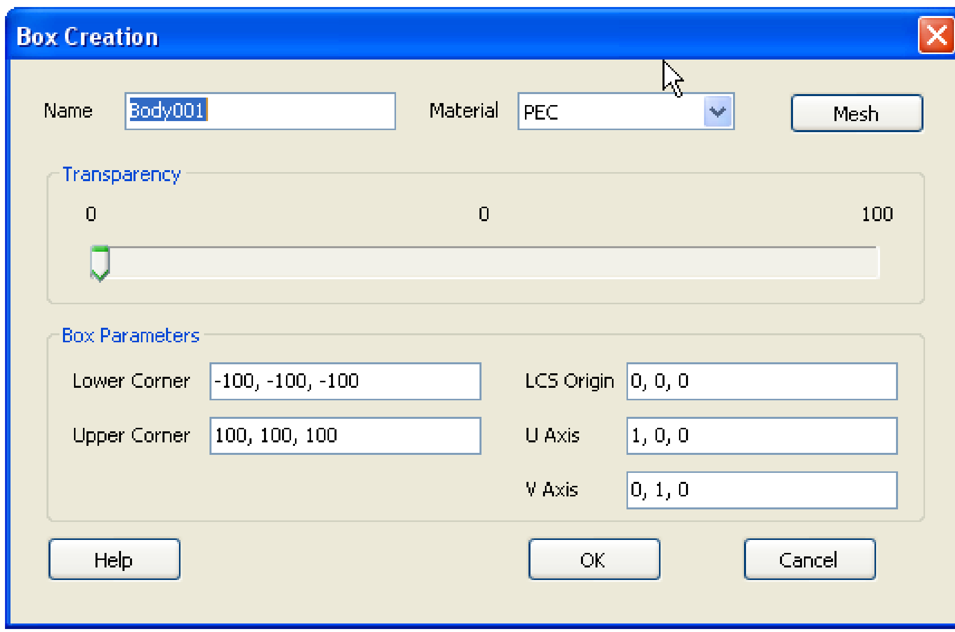
Wavenology GUI supports a solid with multiple components without external BOOLEAN operations.

For example, we would like to create a solid with a box stacking with a half sphere.

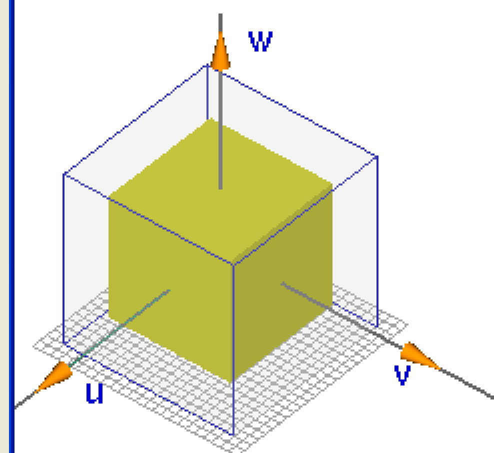
1. Create a box first



box listed in tree
the box has name "Body001"

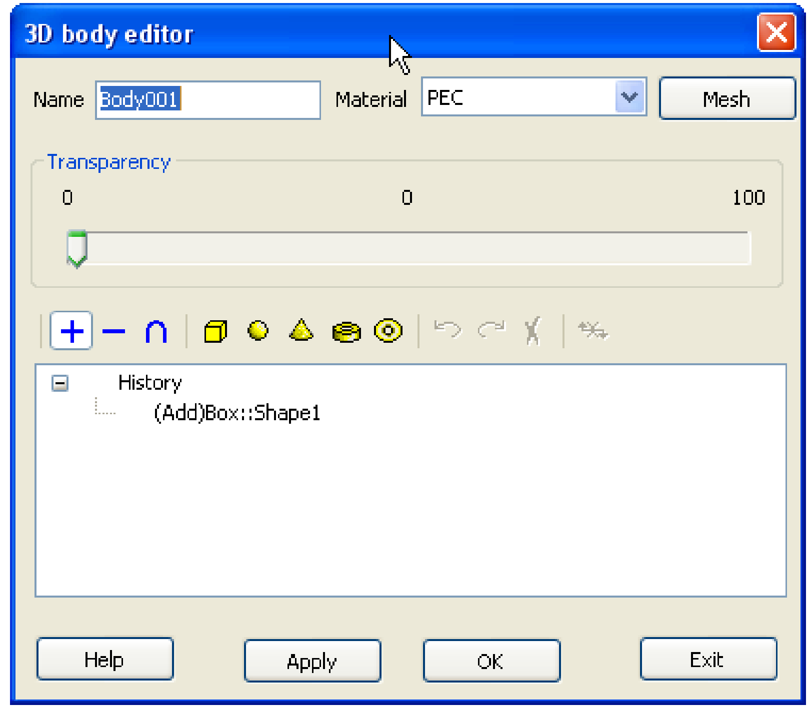


box parameters

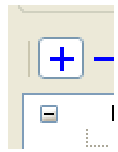


box shown

2. Double click treenode “Body001” to enter solid editor dialog

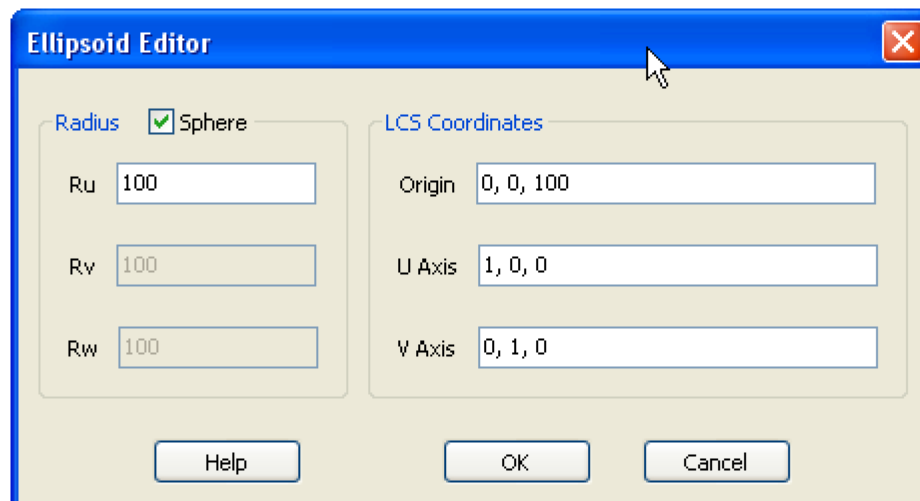


3. Check the “Add” button

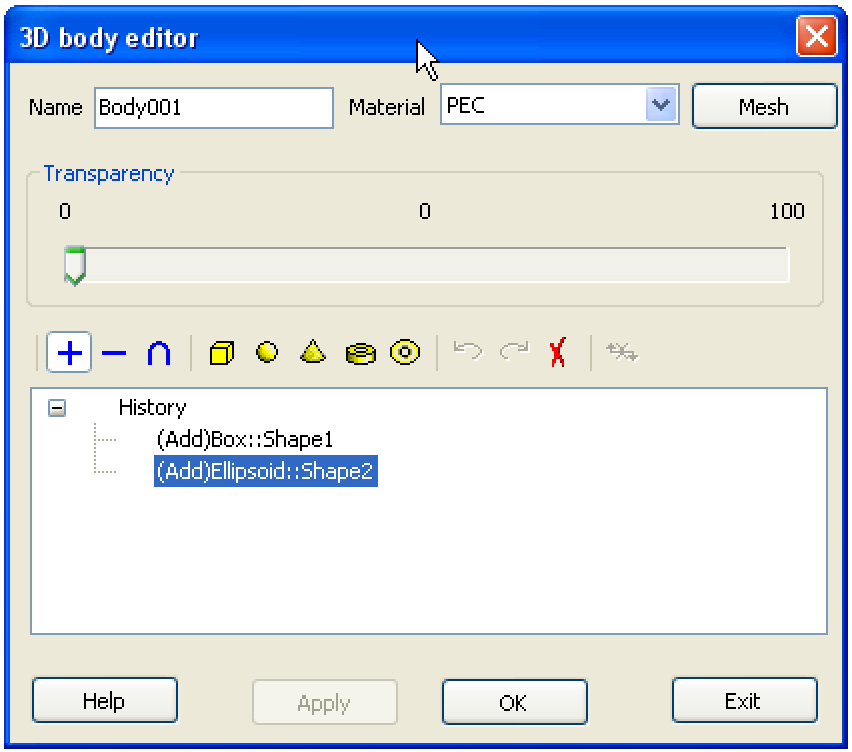


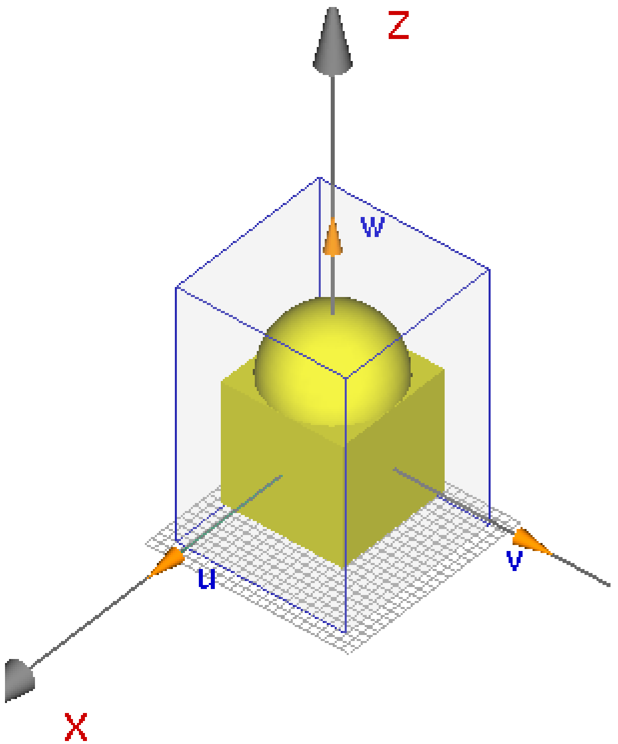
4. Press “sphere” button

to add a sphere on the original box. Here, we make the sphere radius 100 with the center at (0,0,100)

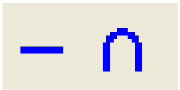


Following is the final shape structure and the displayed figure





Similarly, user can SUBTRACT or INTERSECTION other shape on an existing 3D solid by “subtract” and “intersect” functions

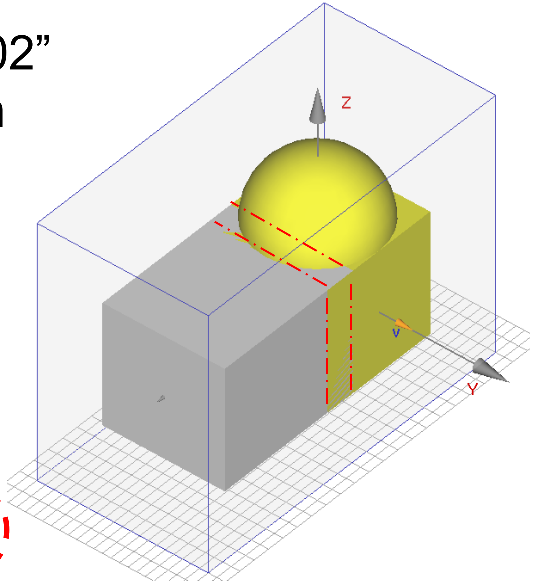


Boolean Operations on 3D Solids

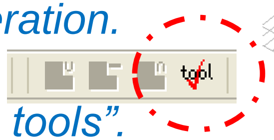
Sometimes, there is clash among several solids. For example, the two solids in following case have space conflict as shown in the red dashed-wire-frame.

Before mesh generation, user needs to decide which solid occupy the clashed region. Otherwise, the mesh could be wrong.

In this example, we decide to let the gray box "Body002" occupy the clashed region. We use Boolean operation to implement it.

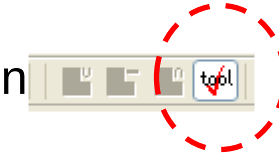


*Note: there must be at least two solids selected to make a Boolean operation. We define the first selected solid as "**Blank**", the other solids as "**Tools**". There are two options for a Boolean operation: whether delete the **Tools** after operation. The options are set by toolbar button:*

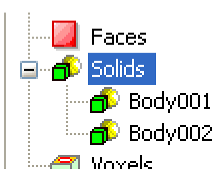


The default setting (unchecked) is "delete the tools". If this button is checked, means "keep the tools"

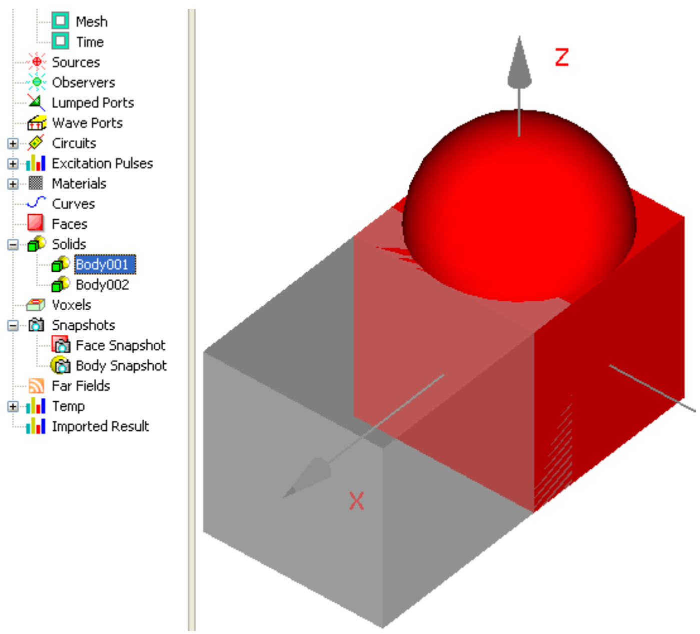
We show two methods to implement a Boolean operation. One is using treenodes to select the target solids, another is using canvas pickup to select the target solids. Both methods use Boolean operation: Body001= Body001-Body002

In these two methods, we will keep two solids. Therefore, make sure button  is checked before the operation.

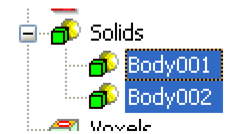
Method 1: operation through treenodes



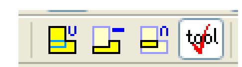
Expand treenode "Solids" to show all solids



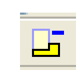
Select treenode "Body001" as **Blank**



Press keyboard "Ctrl" and select "Body002" as **Tool**

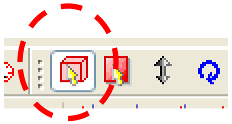


Now, the Boolean operation toolbar buttons become enabled

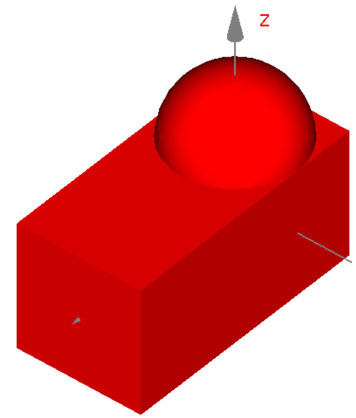


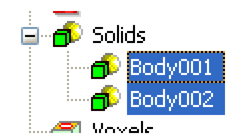
Press "subtract" button. The Boolean operation is done.

Method 2: operation through canvas pickup

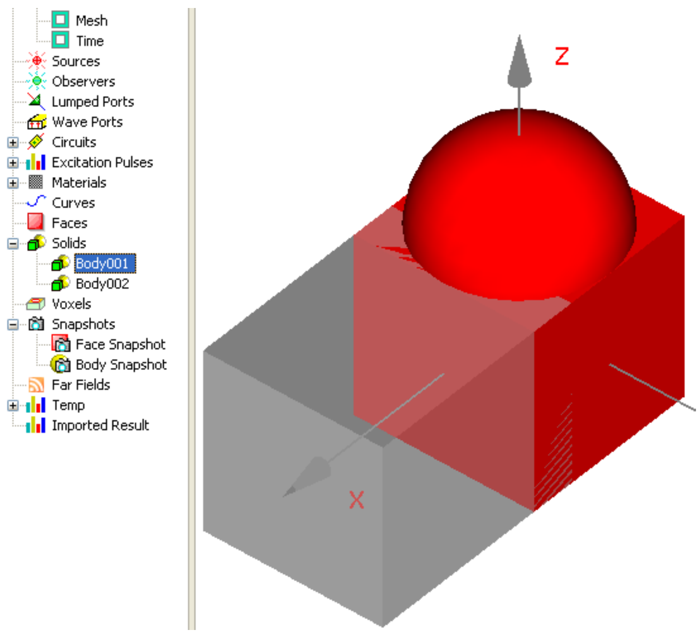


Make sure the display canvas enter the “select body” mode by “Check” this button.

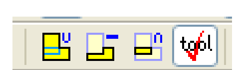




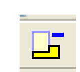
Press keyboard “Ctrl” and use mouse to select “Body002” as **Tool**.
Meanwhile, you can find the “Body002” treenode also selected.



Move mouse arrow on “Body001” and then left click mouse to select it as **Blank**.
Meanwhile, you can find the “Body001” treenode also selected.

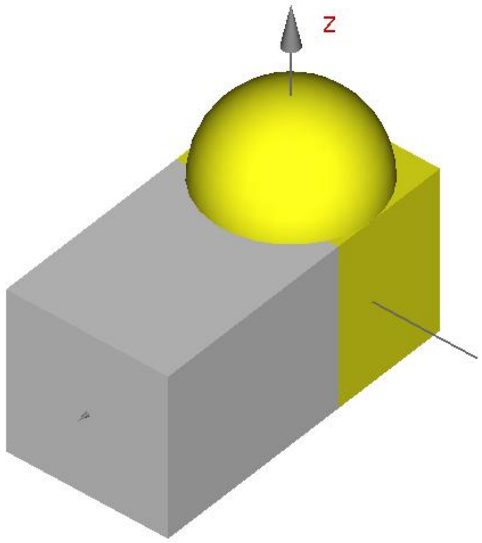


Now, the Boolean operation toolbar buttons become enabled



Press “subtract” button. The Boolean operation is done.

The operation result is shown in following figure



Note: if the operation result is not as user expects, normally it means the operation setting is wrong. User can use “**Undo**” function to recover the system prior to the Boolean operation

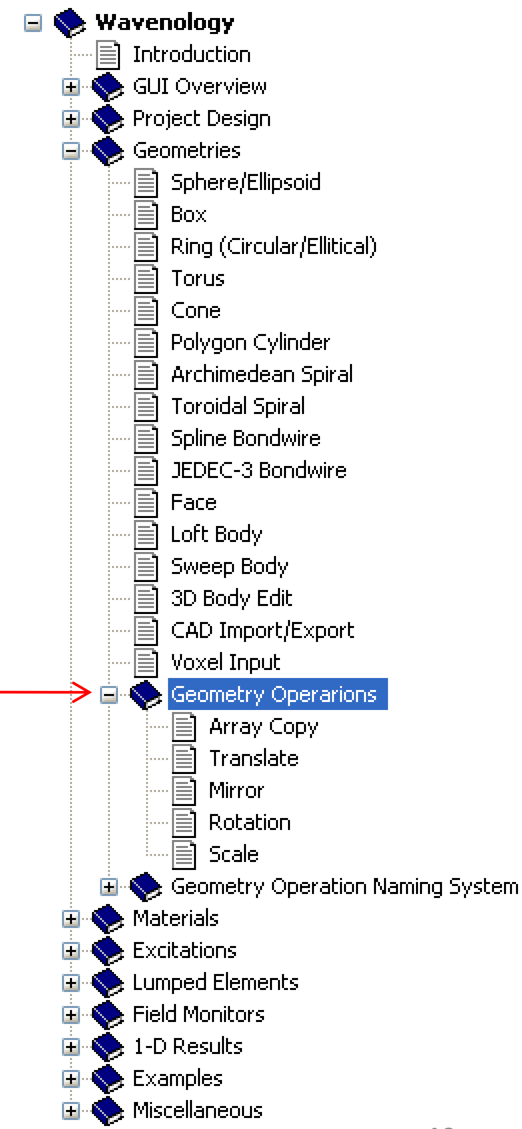
Redo & Undo

Mirror, Translation, Rotation, Scale & Array Copy

Wavenology GUI supports following transform operations on all 3D solids:

Mirror
Translation
Rotation
Scale
Array Copy

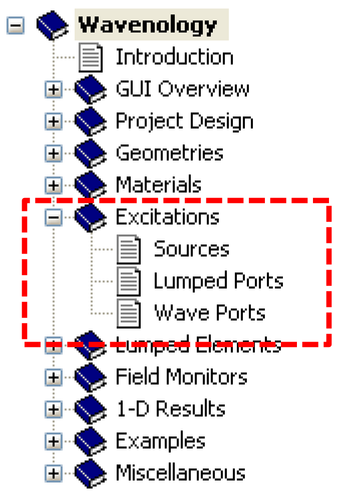
The detail operation please refer to the "**Geometry Operations**" section in the manual of Wavenology EM package.

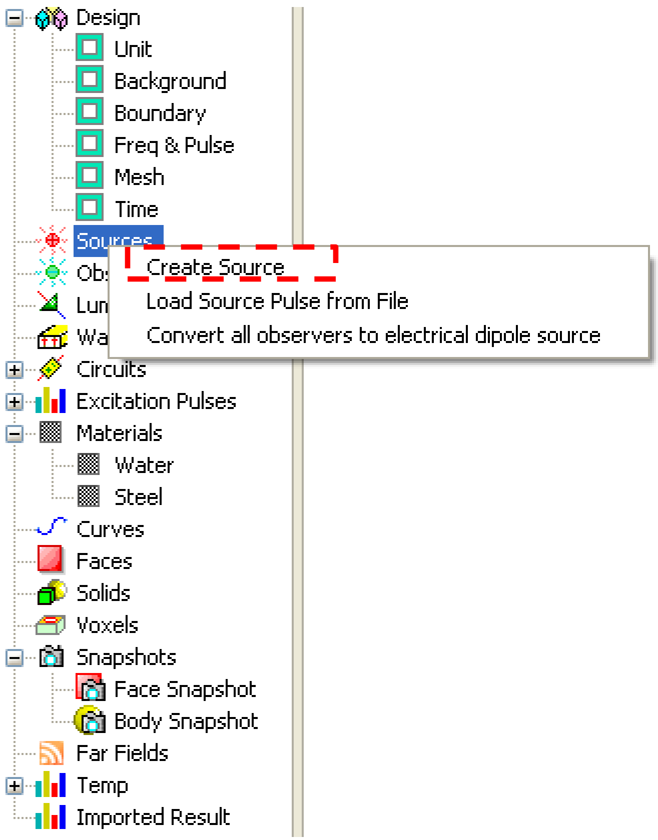


Excitation Source

EM Simulation

For the excitation source in EM simulations, please refer to the “**Excitations**” section in the Wavenology EM Package manual

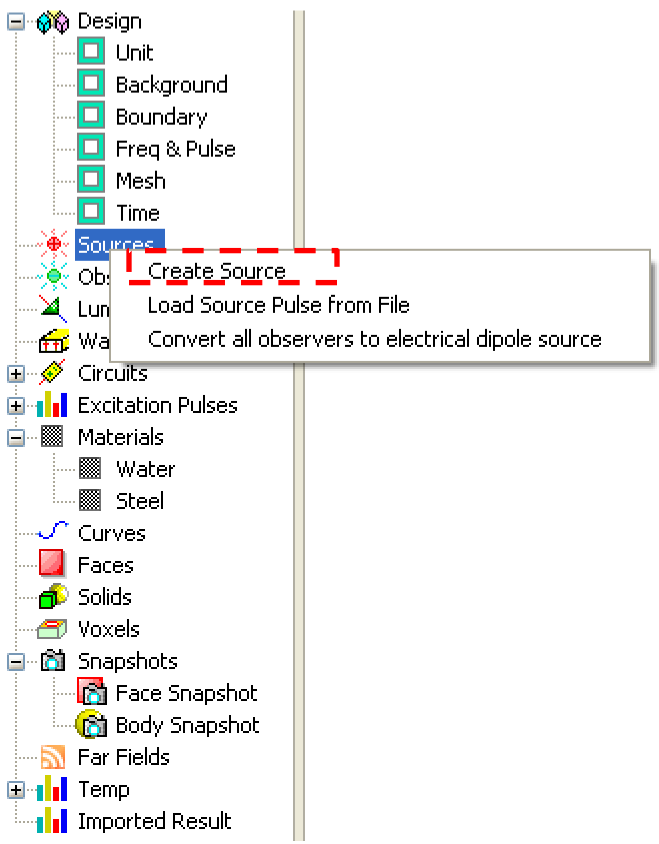


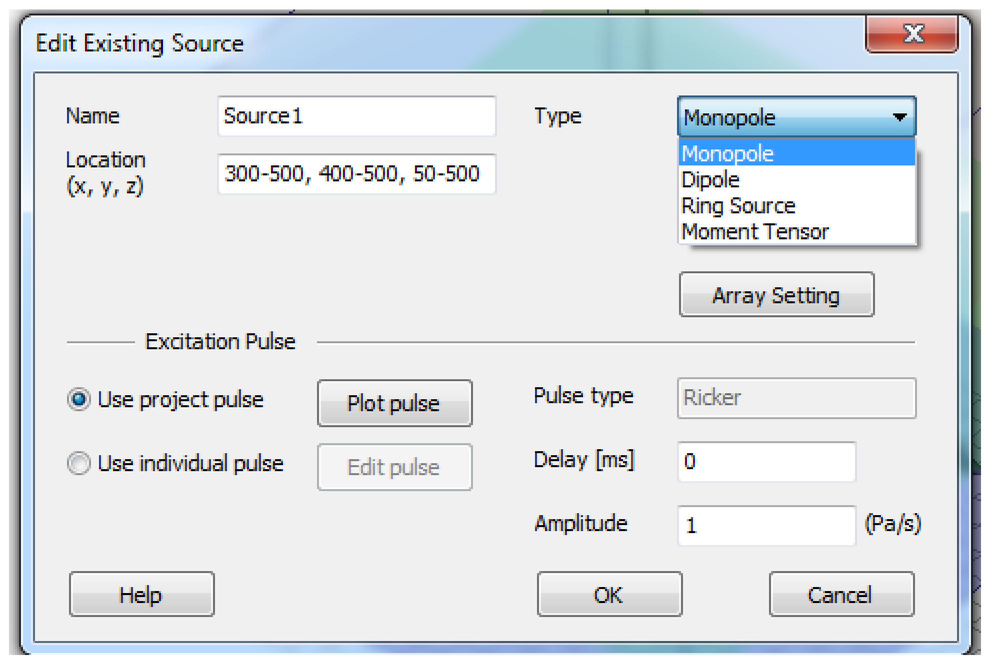


Excitation Source

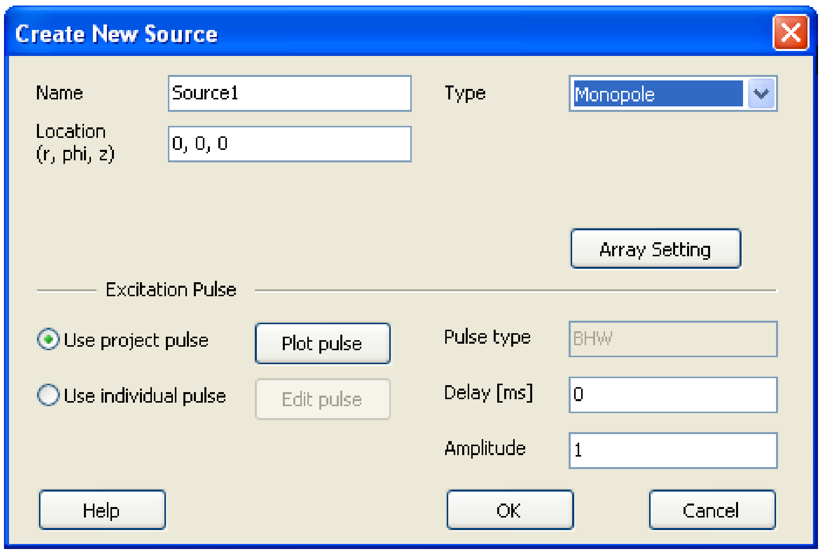
Elastic Wave Simulation

There are 4 types of excitation sources for elastic wave simulations: **ring source** (for Cylindrical Solver only), **point dipole source**, **point monopole source** and **Moment Tensor** (for Cartesian Solver Only)

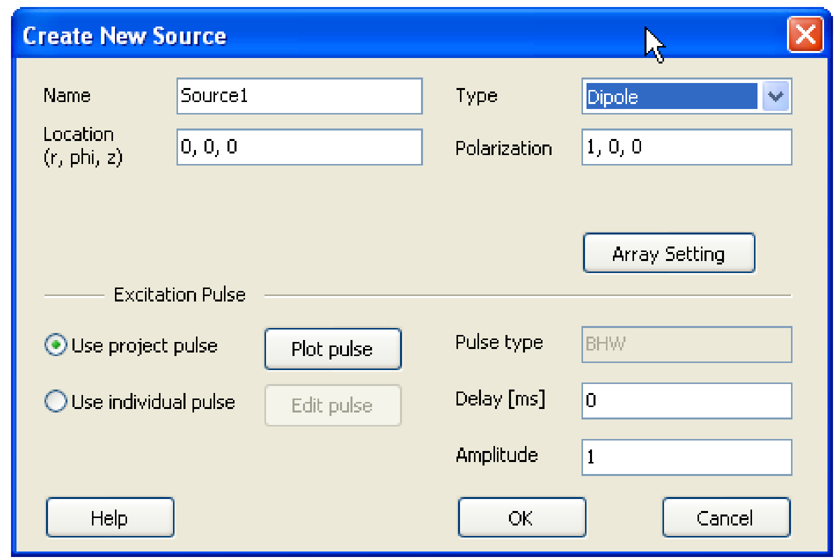


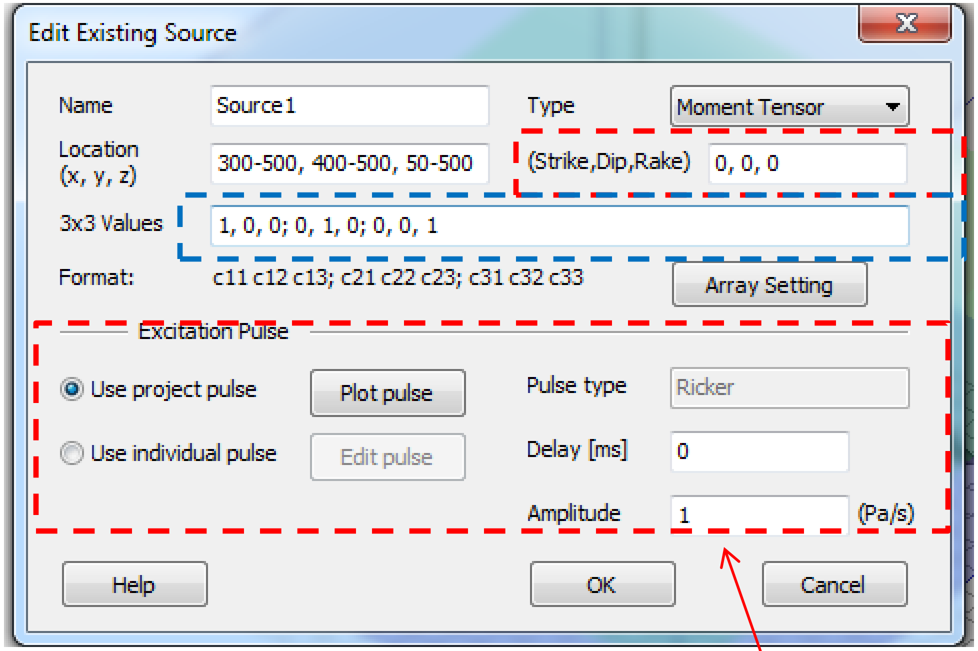


For a point monoploe source, user need to define the position of the source.



For a point diploe source, user need to define the position and the polarization of the source.





Focal Mechanism: Strike, Dip & Rake.
If one of them is not 0, the Focal Mechanism will be enabled. Otherwise, the Focal Mechanism will be disabled.

3x3 moment tensor

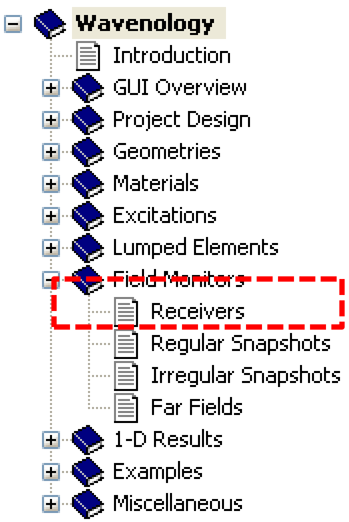
Each source can have individual excitation pulse by setting “Excitation Pulse”.

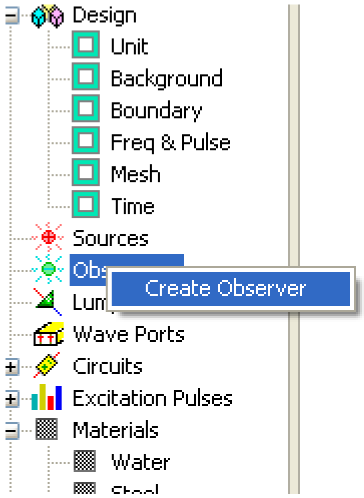
44

Observer

EM Project

User can define observer to record the simulation data.
For the definition of observer in EL, please refer to the “Receivers” section in the manual of Wavenology EM package.



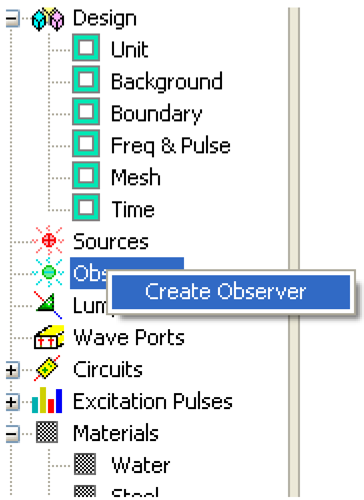


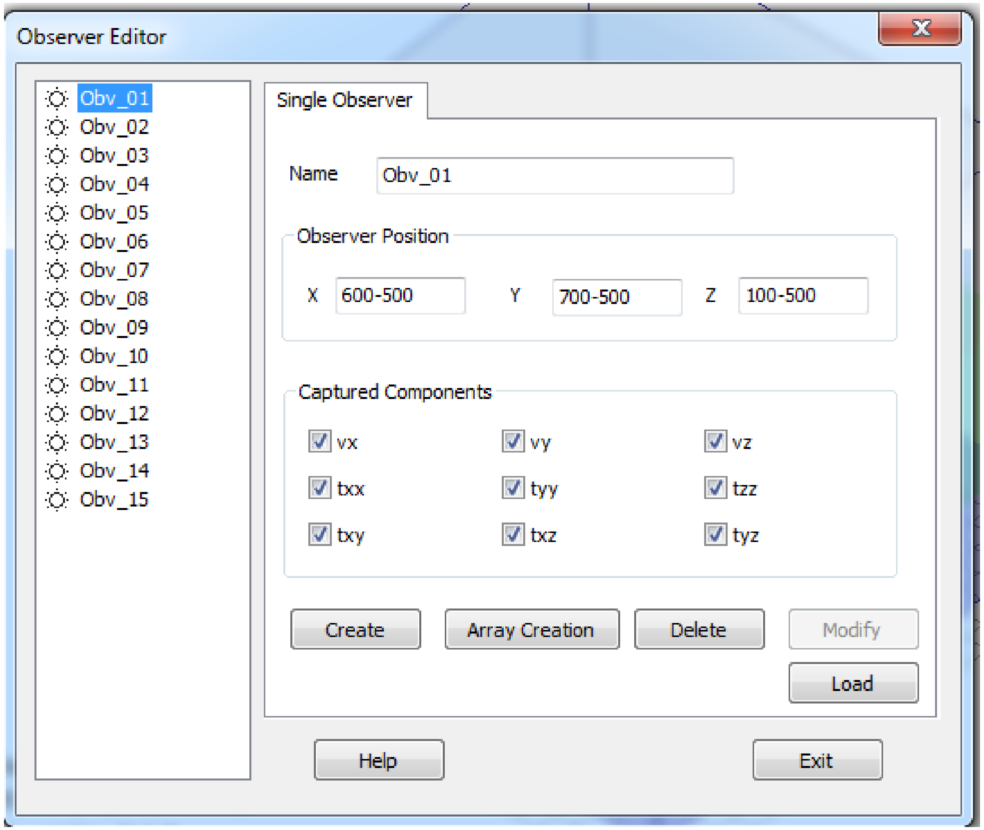
Observer

Elastic Wave Project

User can use the same method as EM project to define and modify observer.

But in the elastic wave project, the recording data are velocity: vx, vy, vz and stress: txx, tyy, tzz, txy, txz, tzz



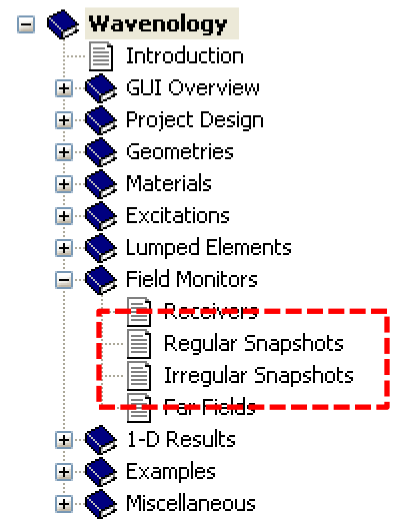


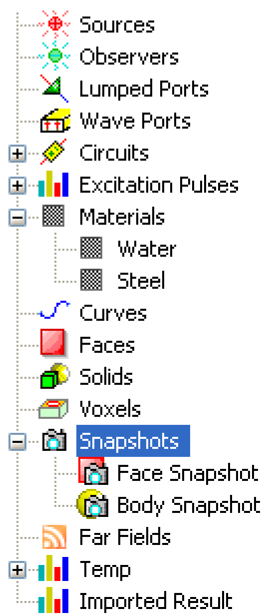
To create observers array by "Array Creation" function, please refer to the "Receivers" section in the manual of Wavenology EM package.

Snapshot

EL Project

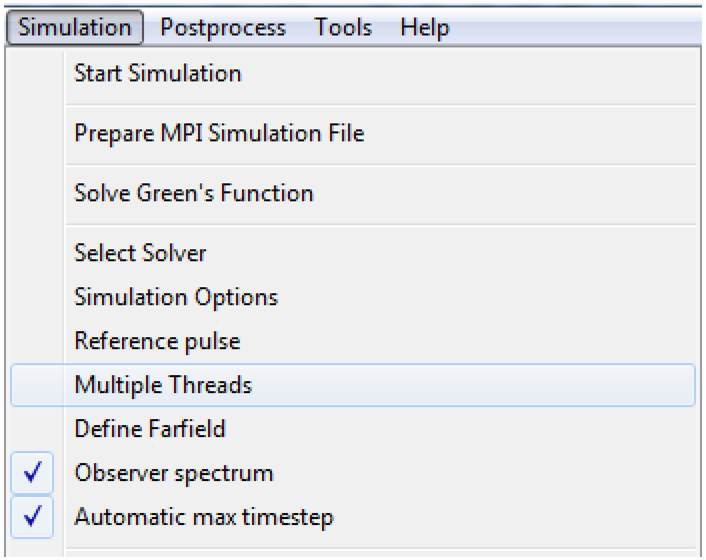
User can define snapshot to record the simulation data on a face or a 3D volume. For the definition of snapshot in EL, please refer to the "Snapshot" section in the manual of Wavenology EM package.



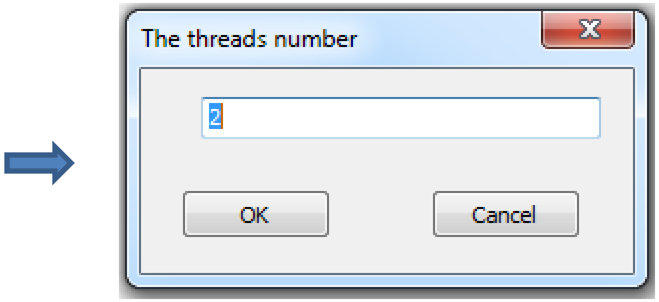


Simulation with Multi-threading

User can use multiple threads to speedup the simulation



Define the thread number



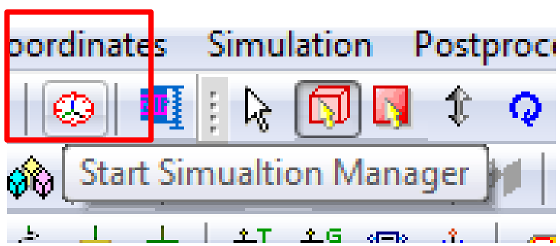
Note: in the WCT Cartesin Elastic Wave solver, the real used threads number will be: min(use_input,computer_CPU_thread_number)

Batch Simulation through Simulation Manager

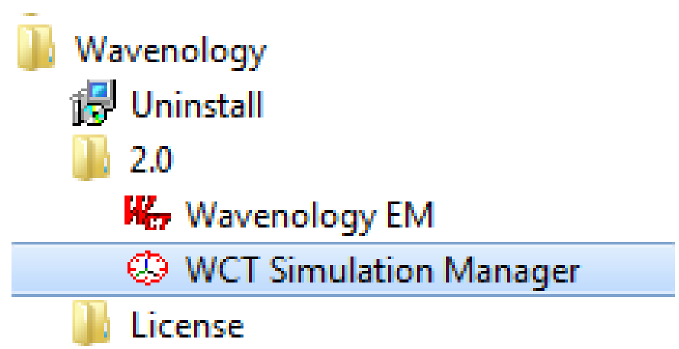
User can simulate multiple projects sequentially through WCT simulation manager

First, a or several WCT Elastic Projects need to be build and stored.

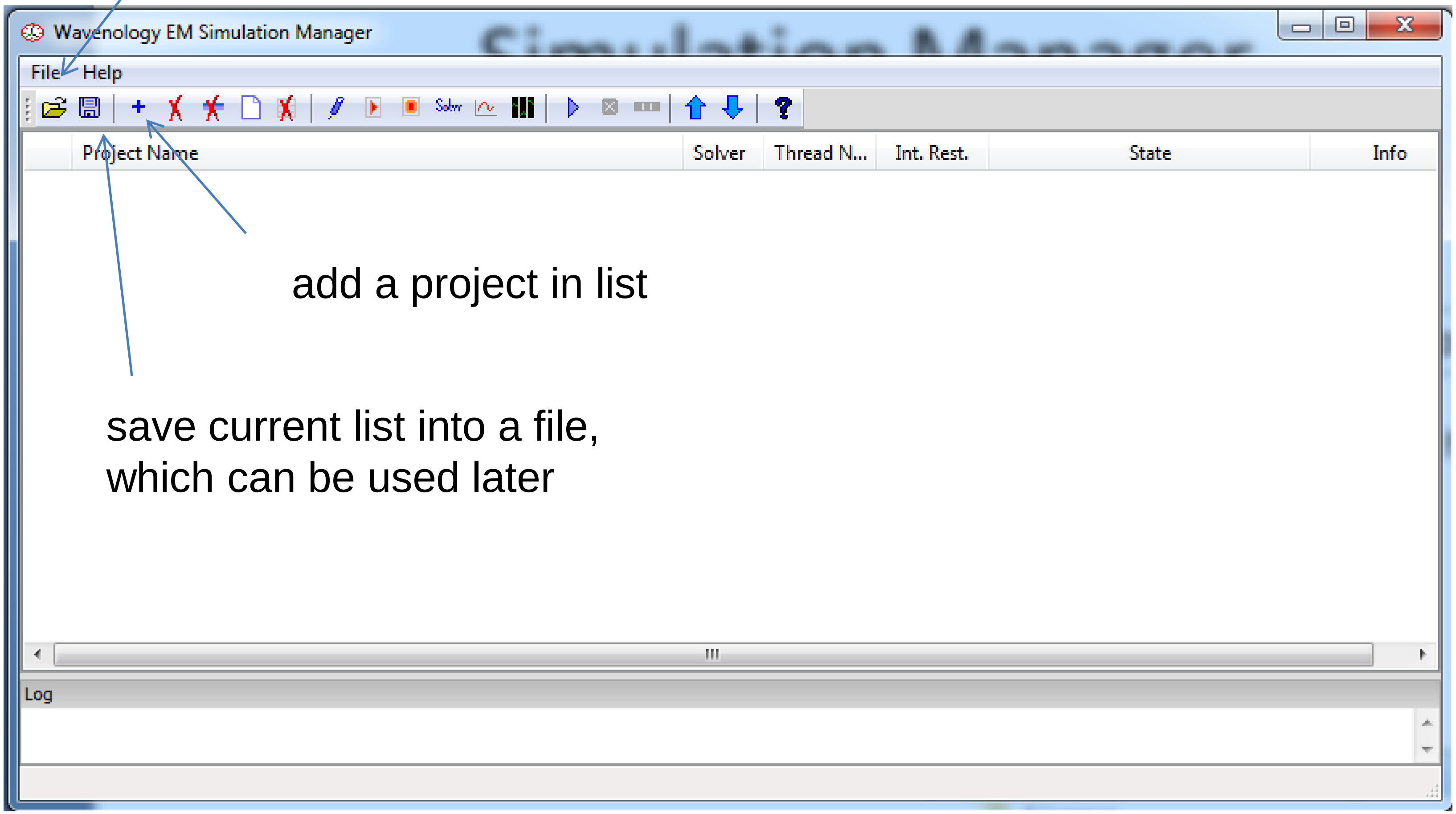
In WCT GUI, use this button to start a simulation manager



Or, in Windows menu, expand here to start a simulation manager

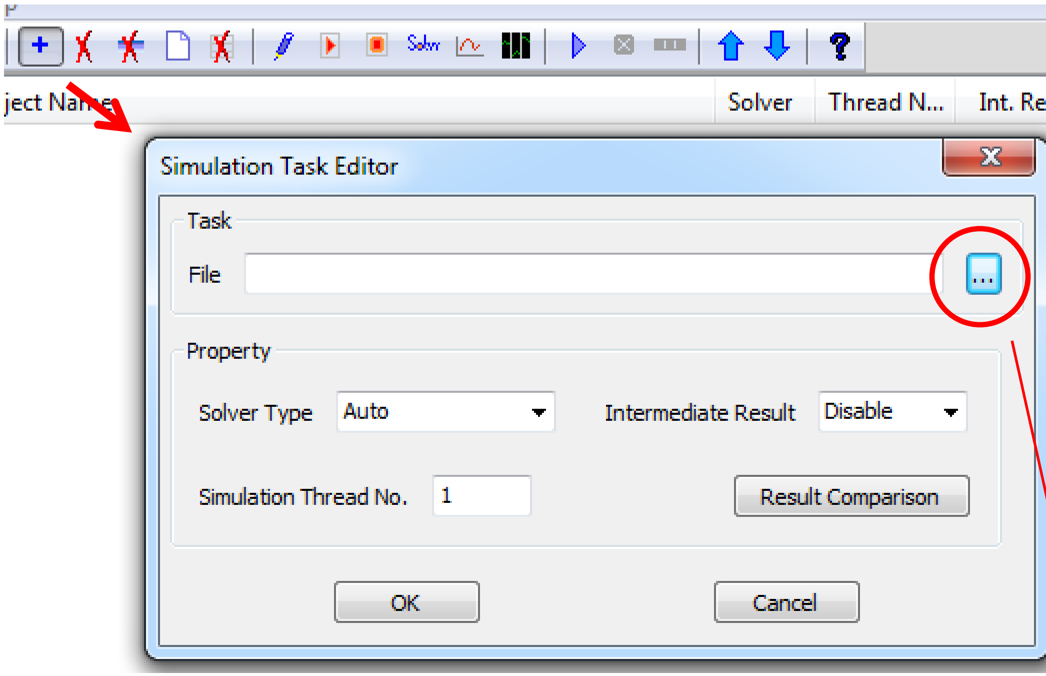


load saved list file or batch-link file

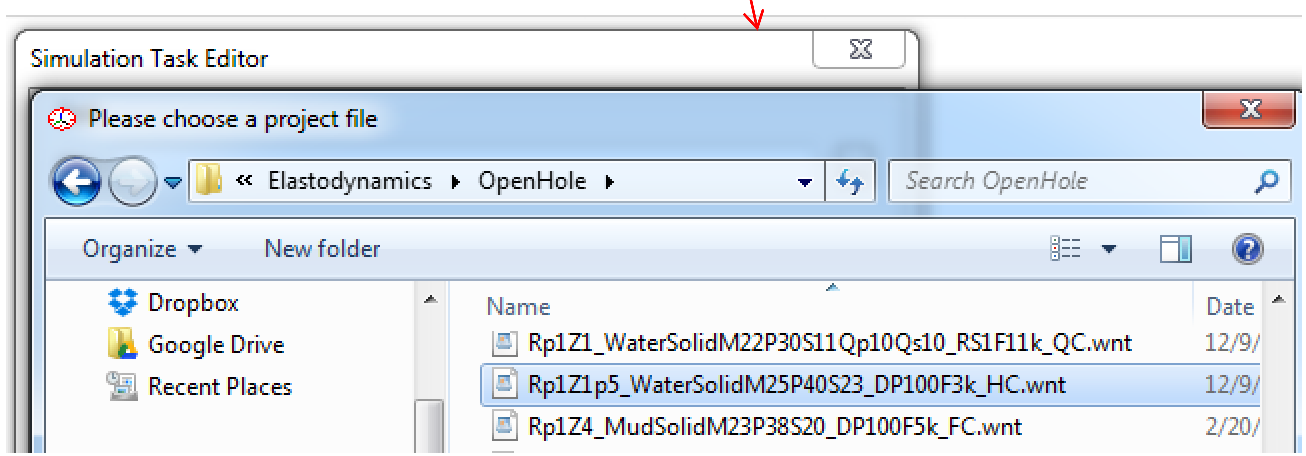


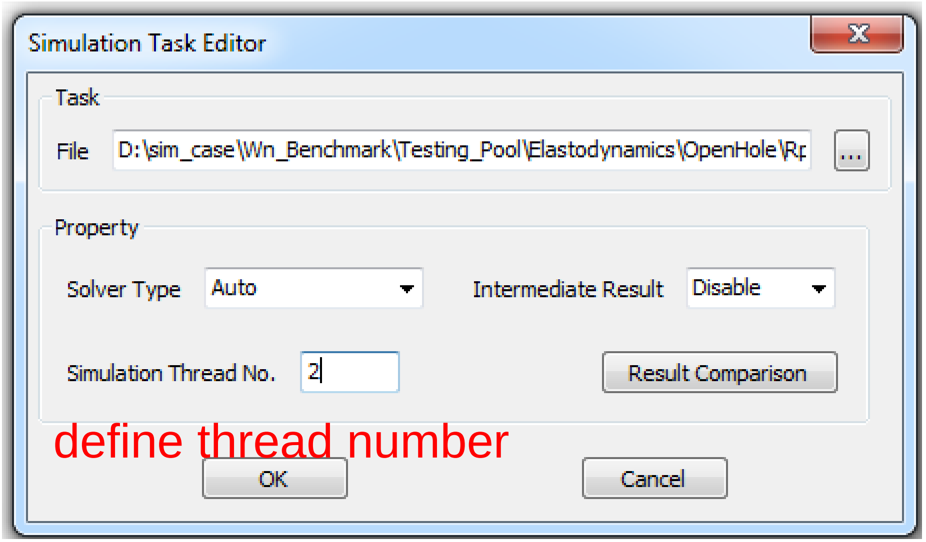
add a project in list

save current list into a file,
which can be used later



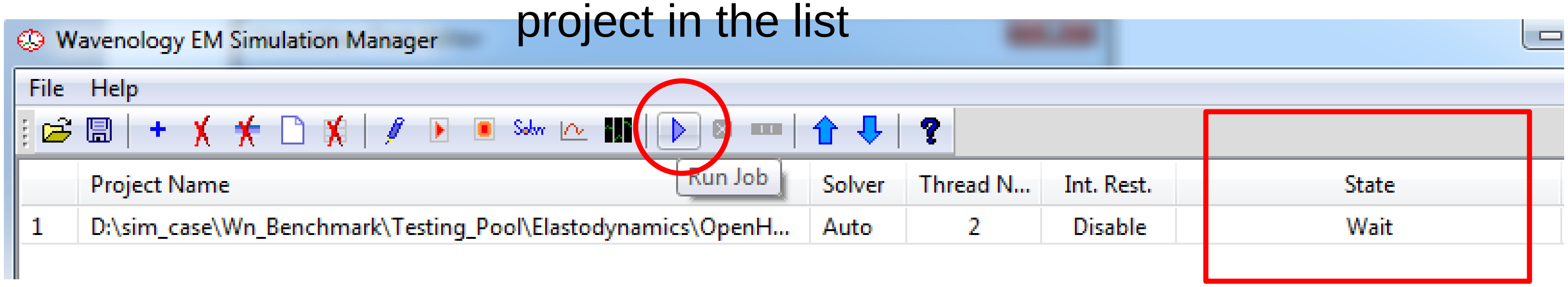
select
project file





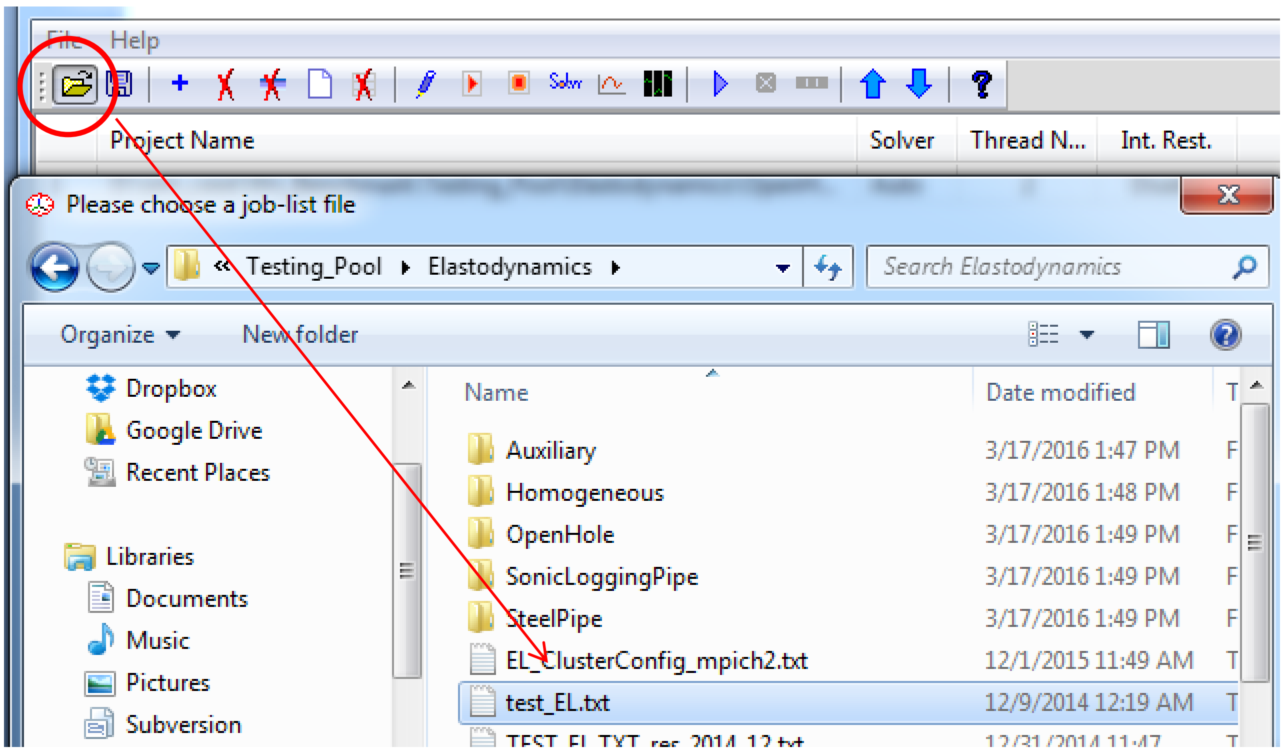
The project status is listed here to shown whether it is finished or not

Click here to run the project in the list



If user have many projects need to simulate and they are in different folders, user can define a link file to include the project.

Then load this link file into WCT simulation manager



WCT Simulation Link file format

Test_EL.txt

```
.include Homogeneous\test_Homogeneous.txt
.include OpenHole\test_OpenHole.txt
.include SteelPipe\test_SteelPipe.txt
.include SonicLoggingPipe\test_SonicLoggingPipe.txt
##########
```

comment line start from '#'

test_Homogeneous.txt

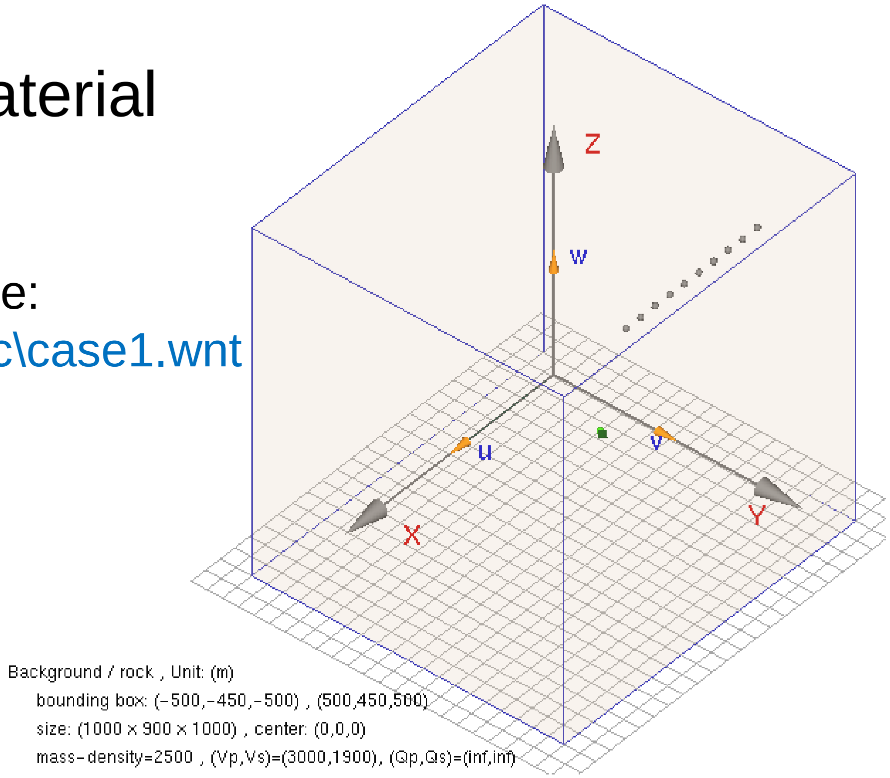
```
"Rp1Z1_WaterSolidM22P30S11_RS0F3k_FC.wnt" SIMTYPE=auto  SIMTHREAD=1 INTRESULT=0 STATUS=wait
"Rp1Z1_WaterSolidM22P30S11_RS0F3k_HC.wnt" SIMTYPE=auto  SIMTHREAD=1 INTRESULT=0 STATUS=wait
"Rp1Z1_WaterSolidM22P30S11_RS0F3k_QC.wnt" SIMTYPE=auto  SIMTHREAD=1 INTRESULT=0 STATUS=wait
```

Note: WCT simulation manager can also load "test_Homogeneous.txt" directly without goes through "Test_EL.txt"

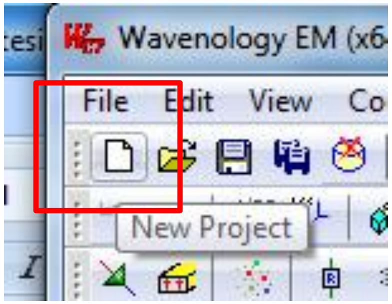
Tutorial Case I
Point Dipole Source in Homogenous Rock Background

General isotropic material

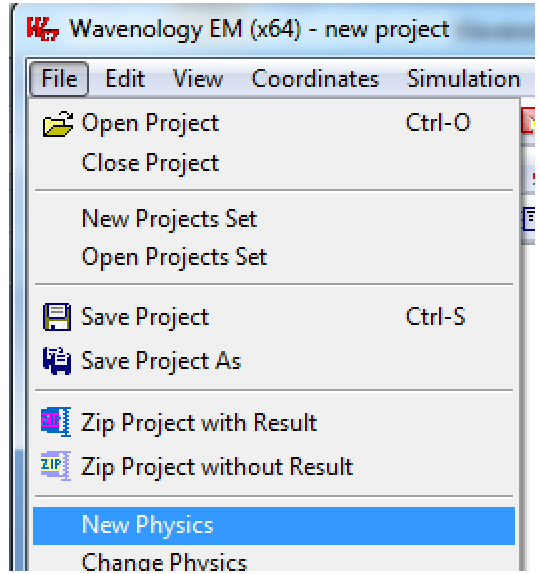
This case is in the demo package:
elastic\homogeneous_dipole_src\case1.wnt

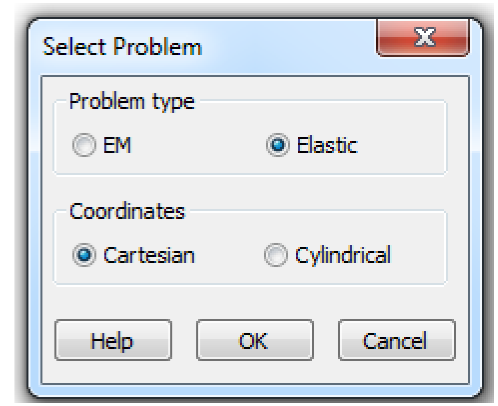


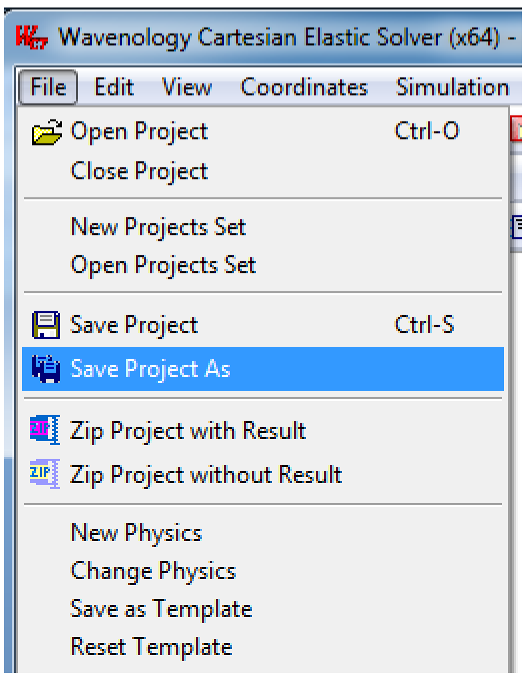
In WCT GUI, "New" a Project



change Physics to Cartesian Elastic Wave Solver

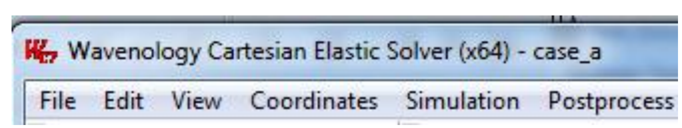


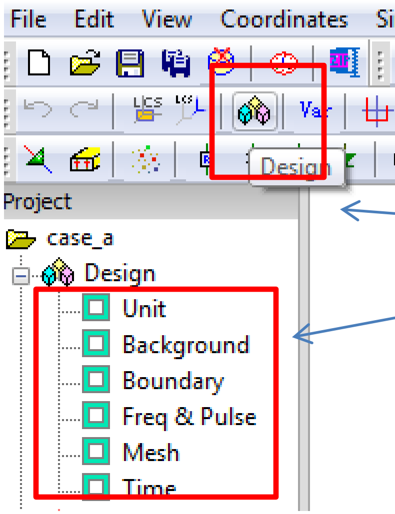




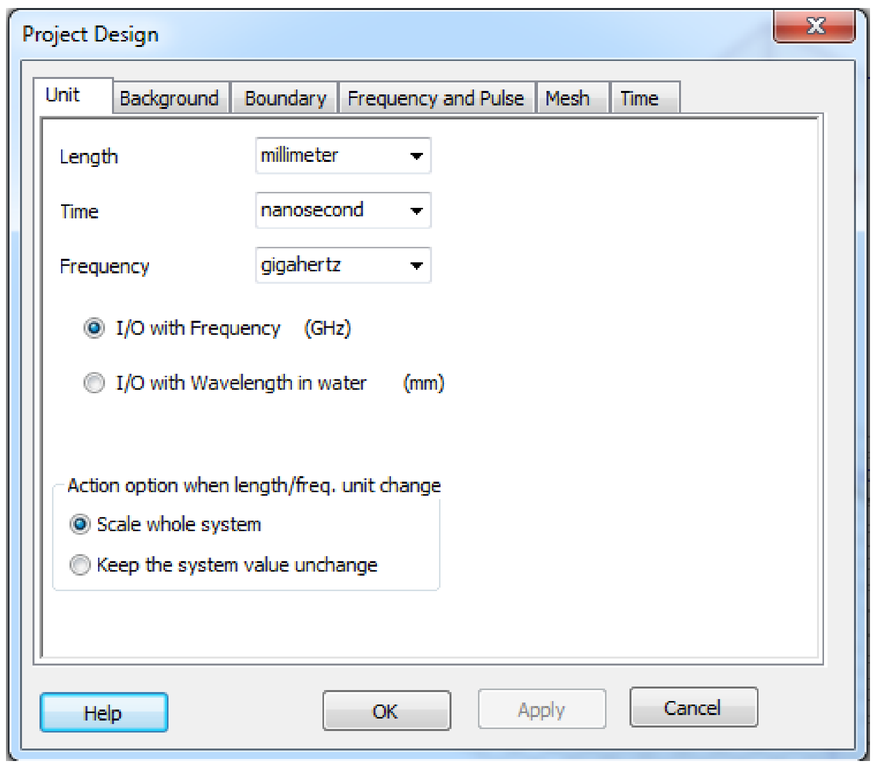
Save as "case_a" project

The WCT GUI title will change to this

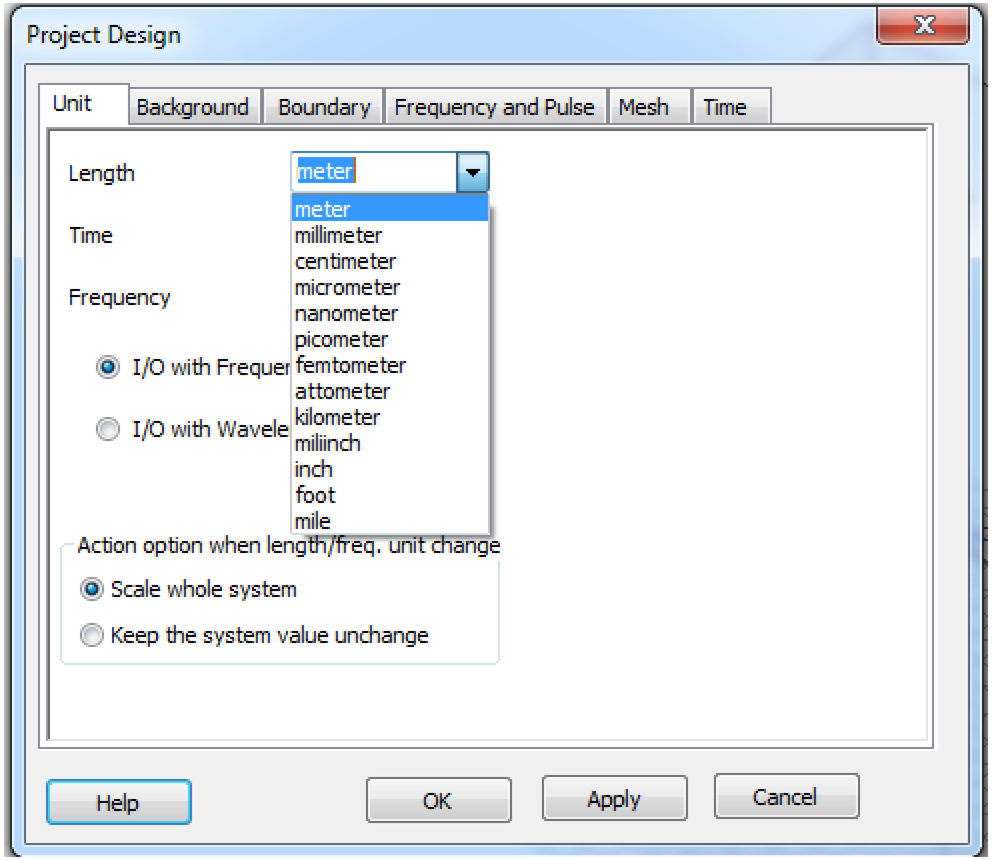




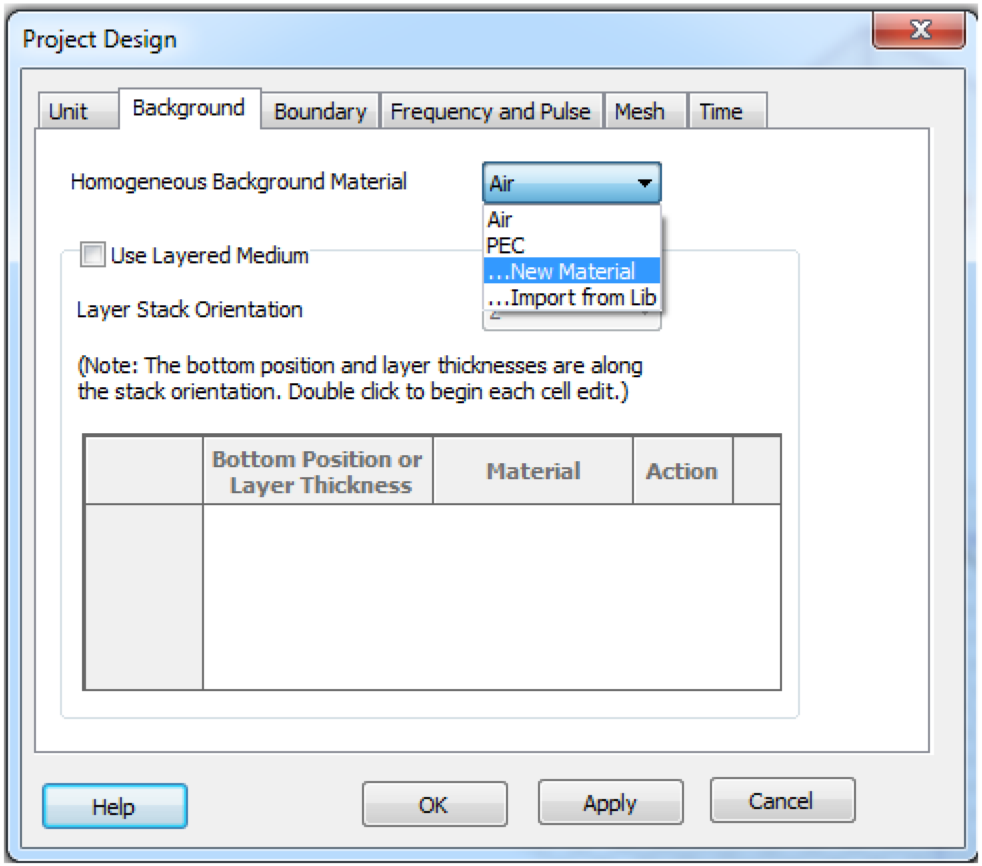
click here,
or double click one of these
to enter project setting dialogue

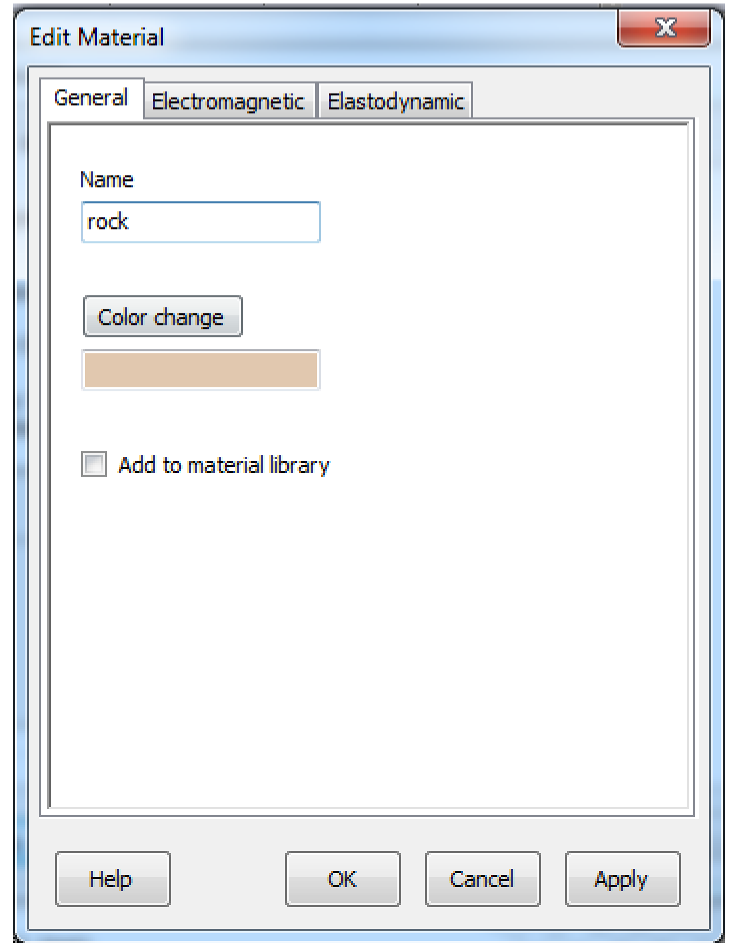


Here, we change length unit to "Meter"

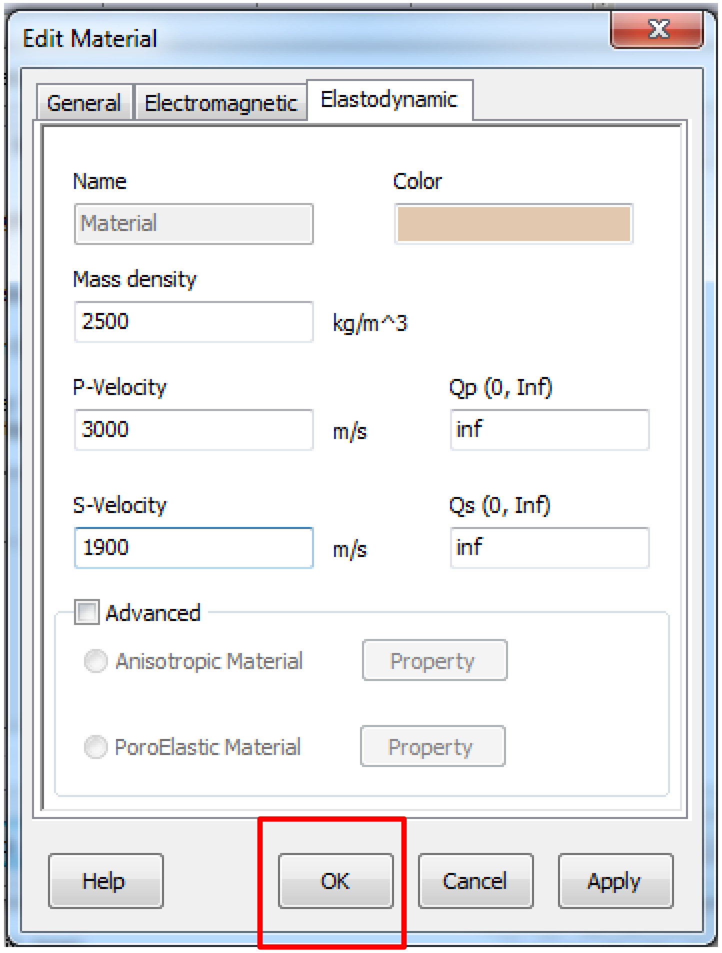


In background page, we define a new material as background

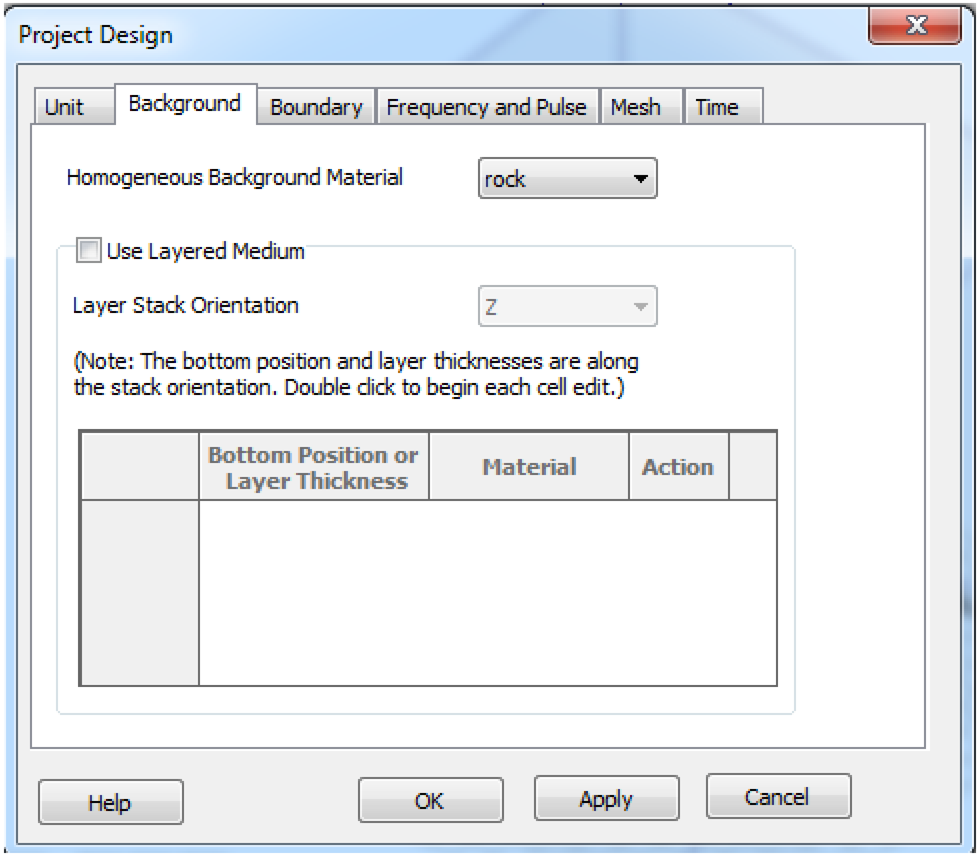




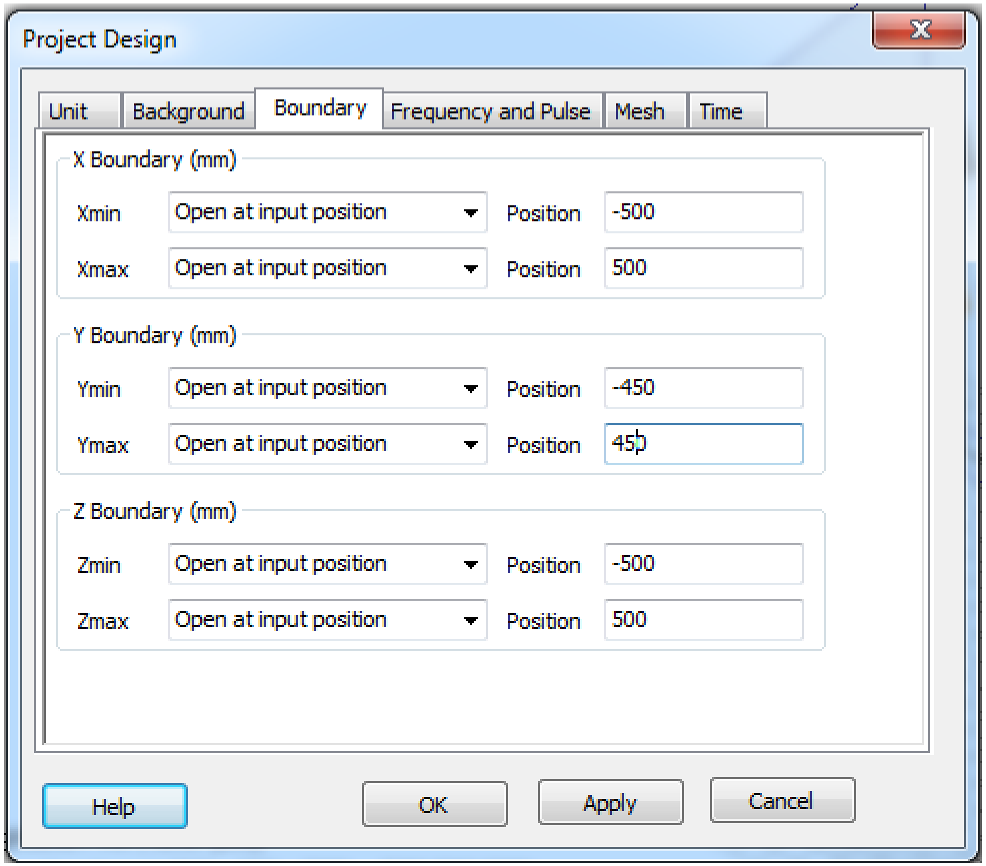
Because it is a elastic simulation, we skip EM parameter, enter "Elastodynamic" page, enter following content. Then "OK"

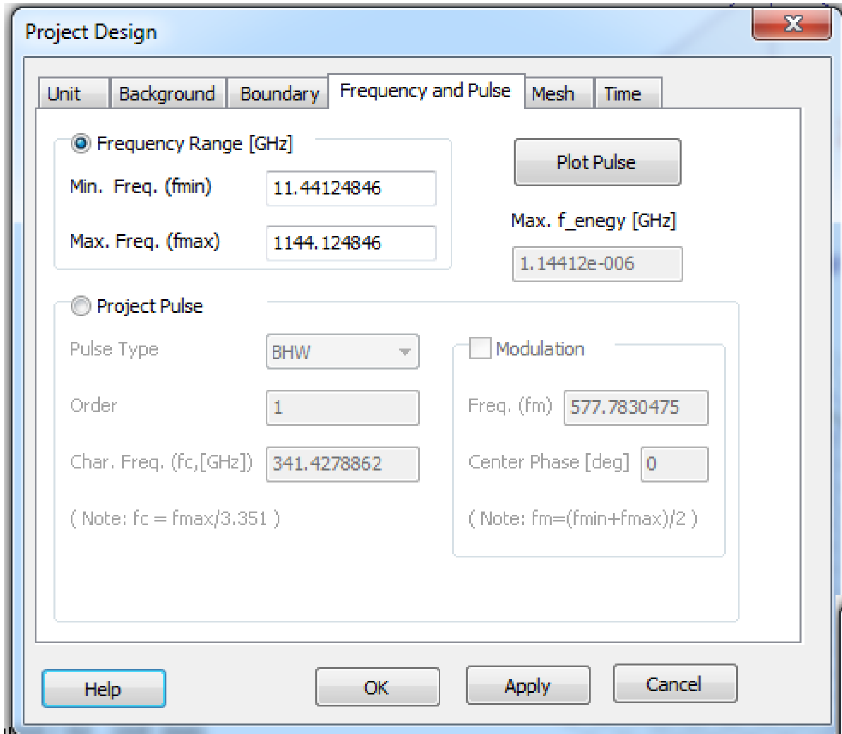


The background will be the new material "rock"

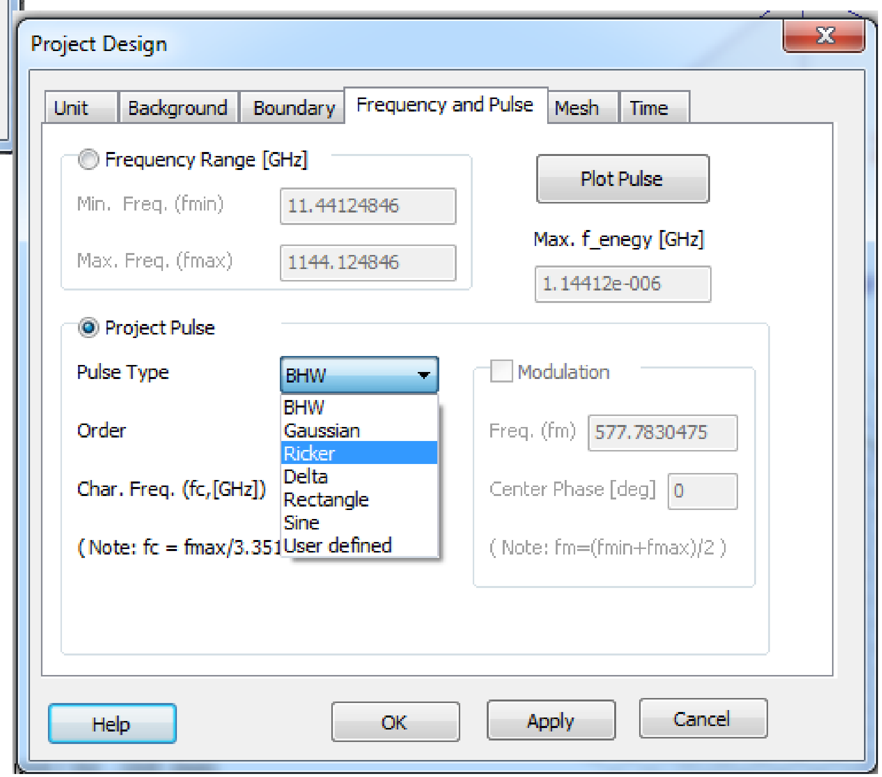


Set Project size as following, all direction use **OPEN** boundary conditions

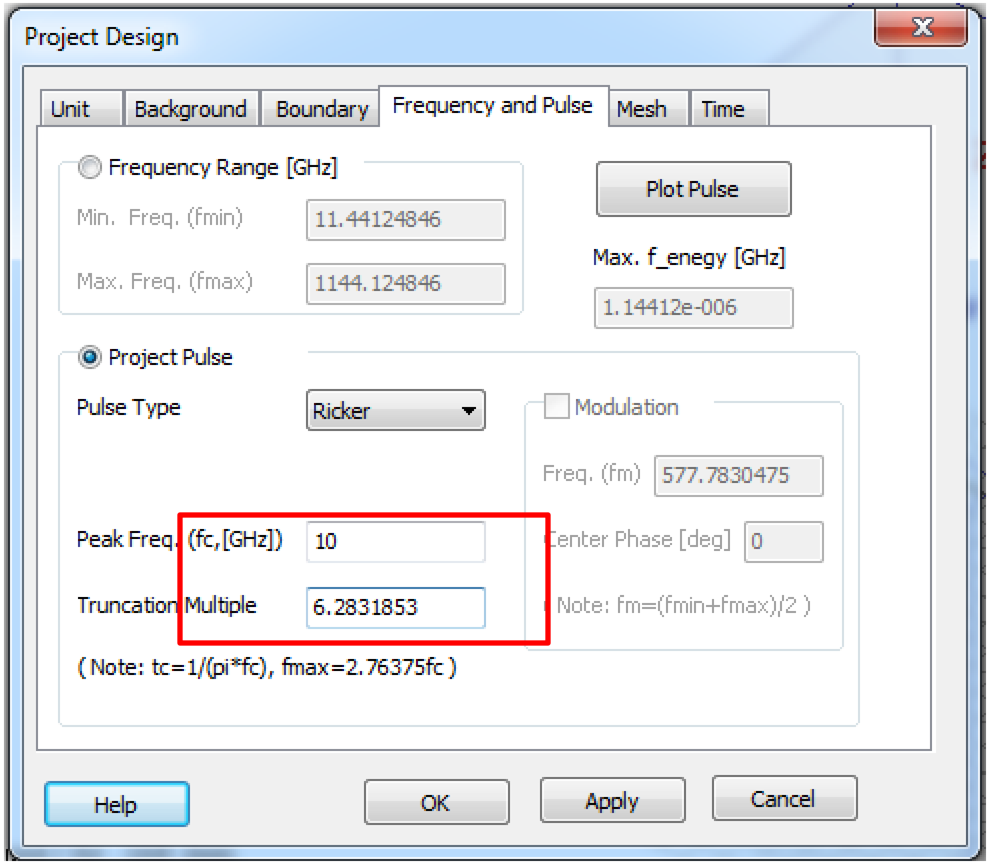


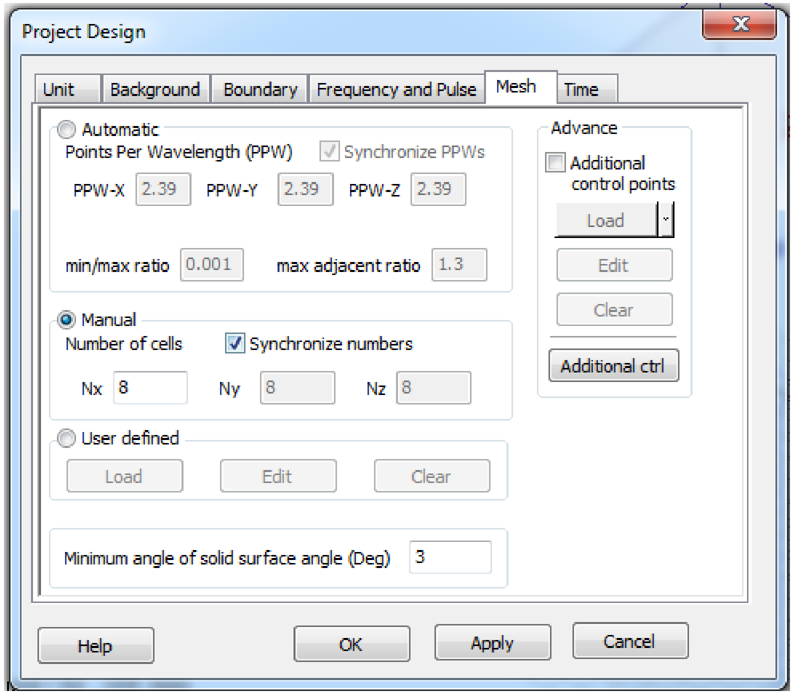


Project pulse, we change to Ricker wave

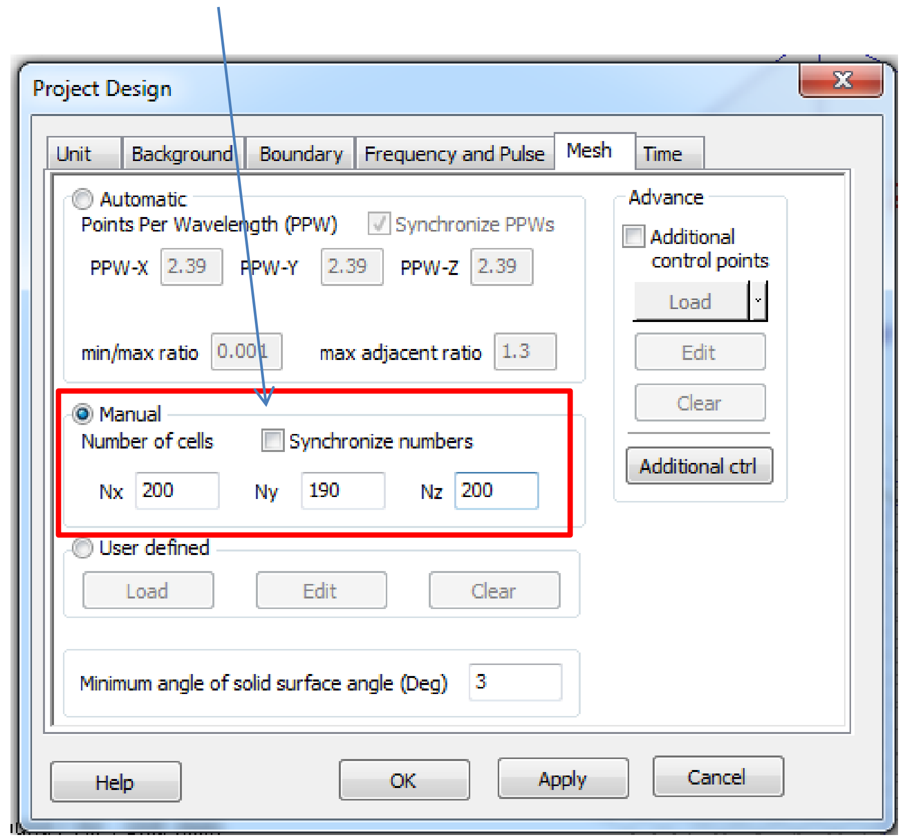


Finally, define project pulse as this



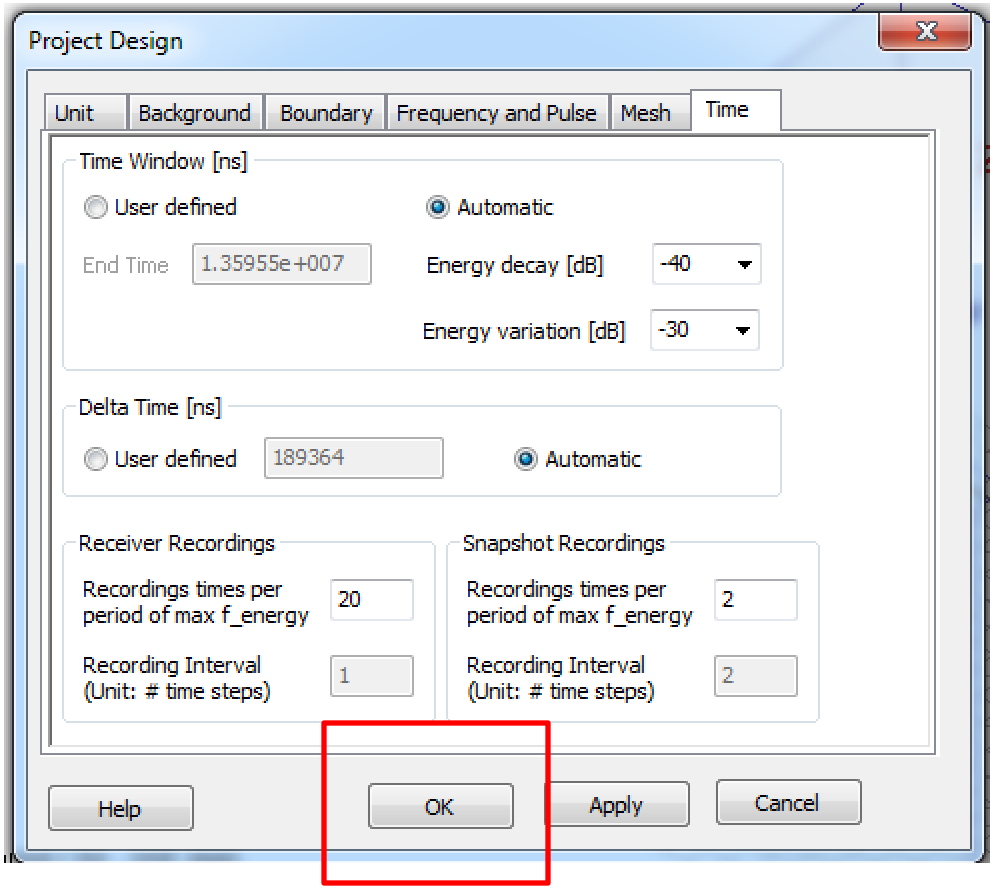


Uncheck this



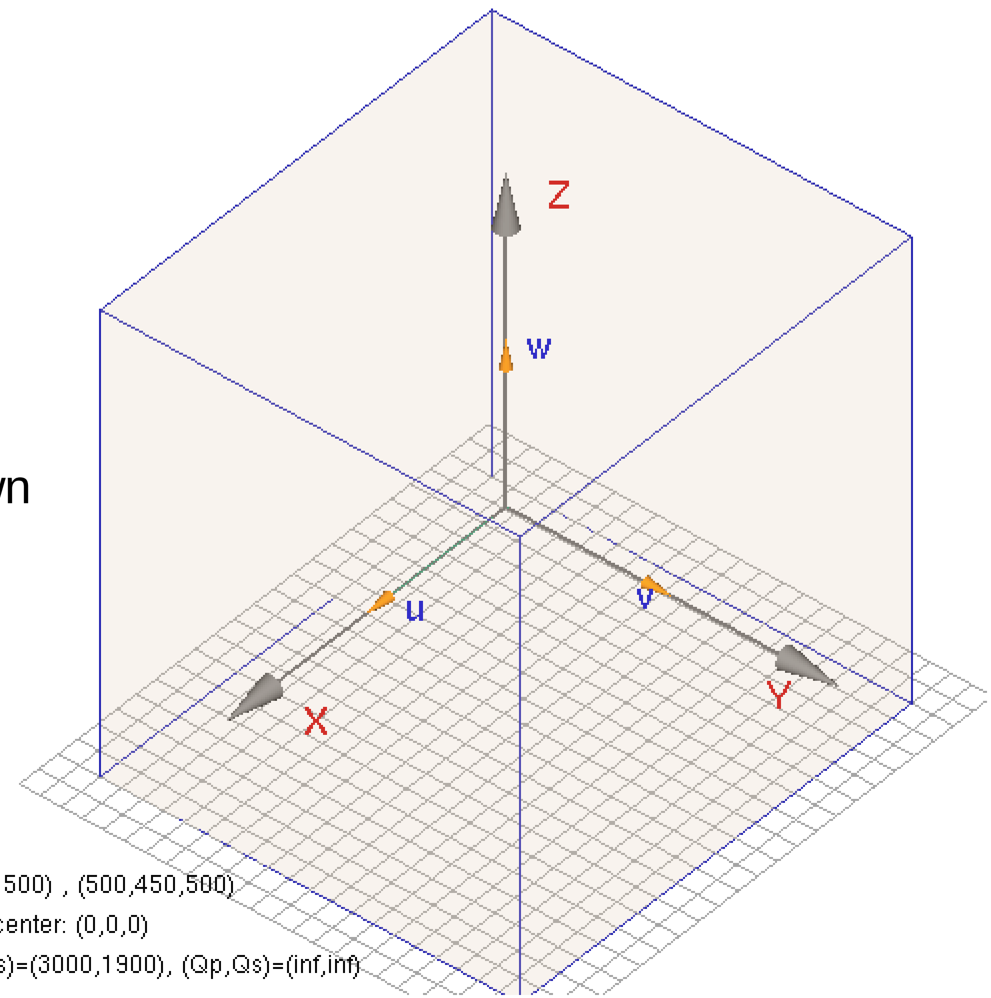
Input cell numbers in each axis

In the **Time** page, all use default input

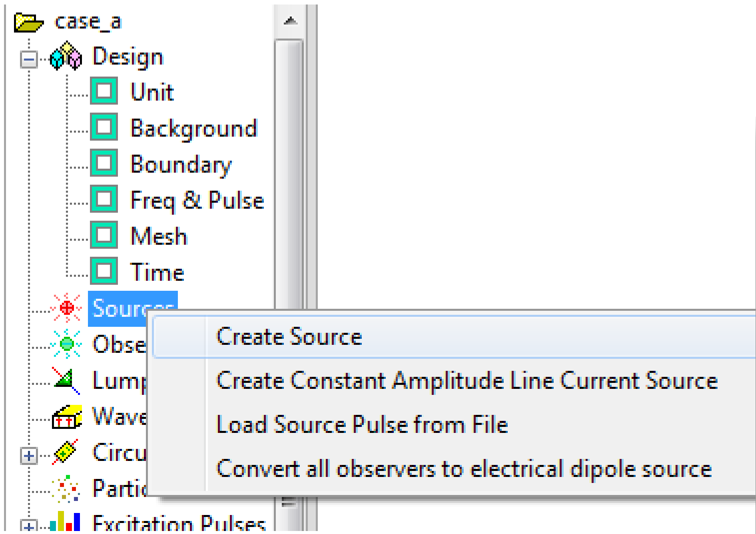


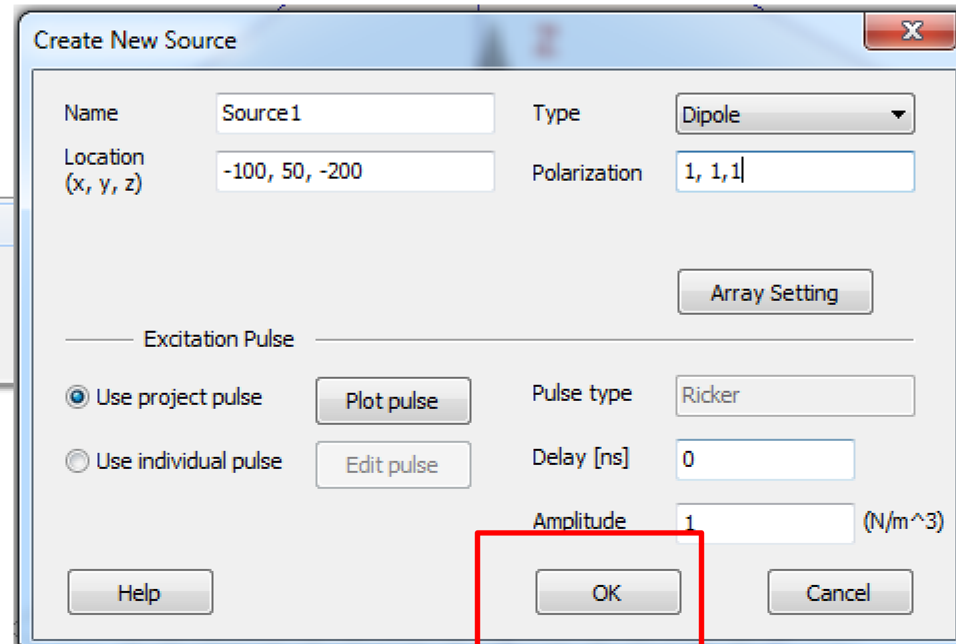
Press **OK** to finish project design

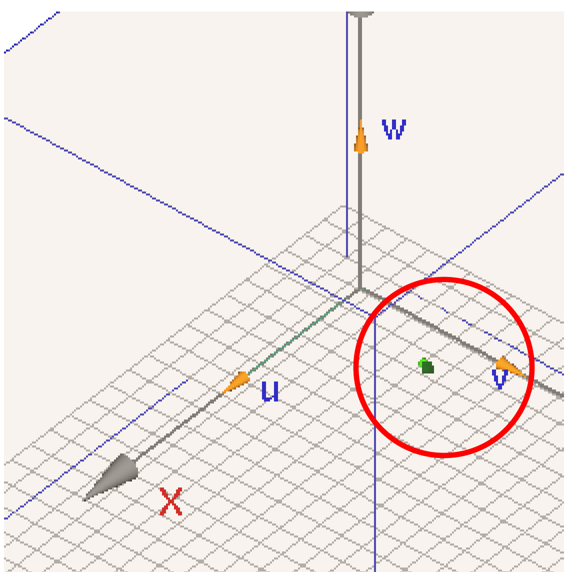
Then the project shown in 3D canvas will be



Define source



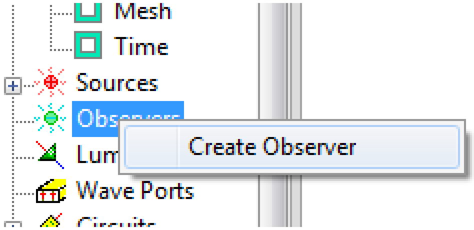


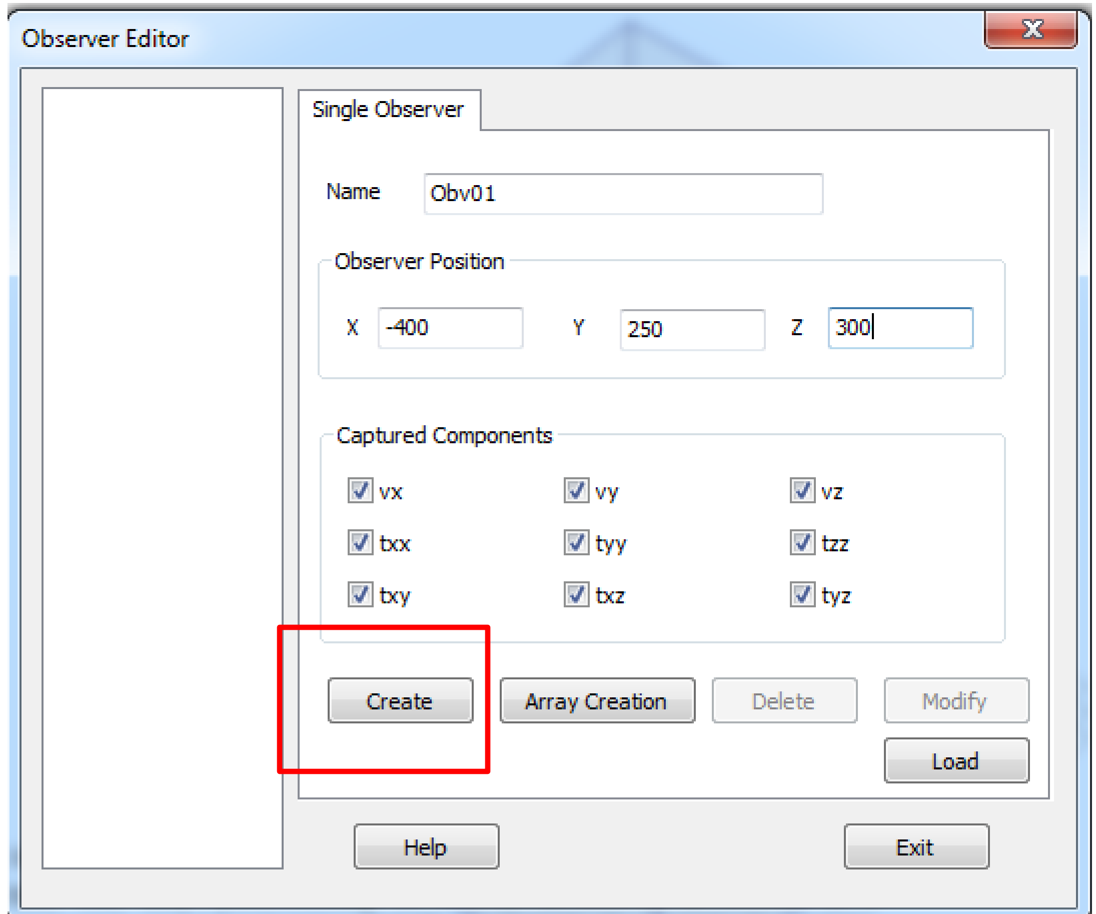


Press **OK**

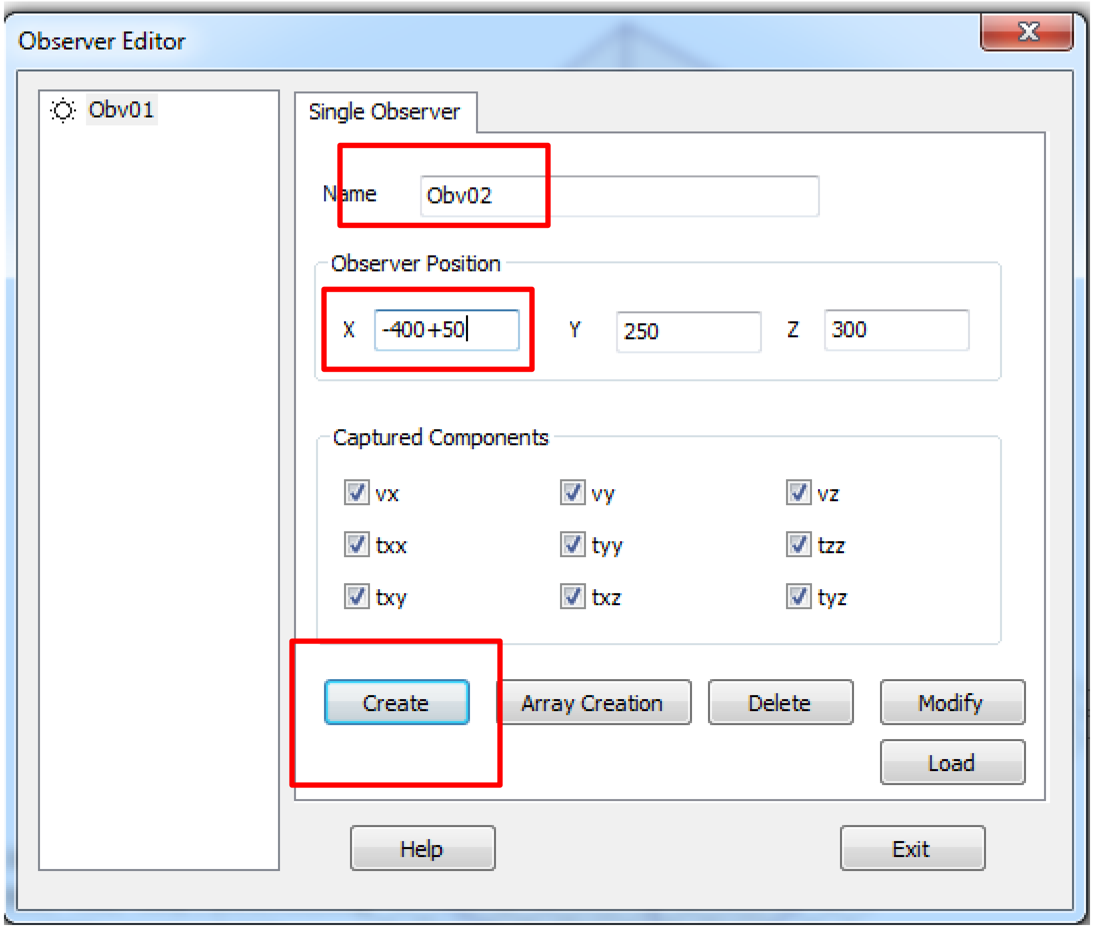
A source is added in main canvas

Define Observer

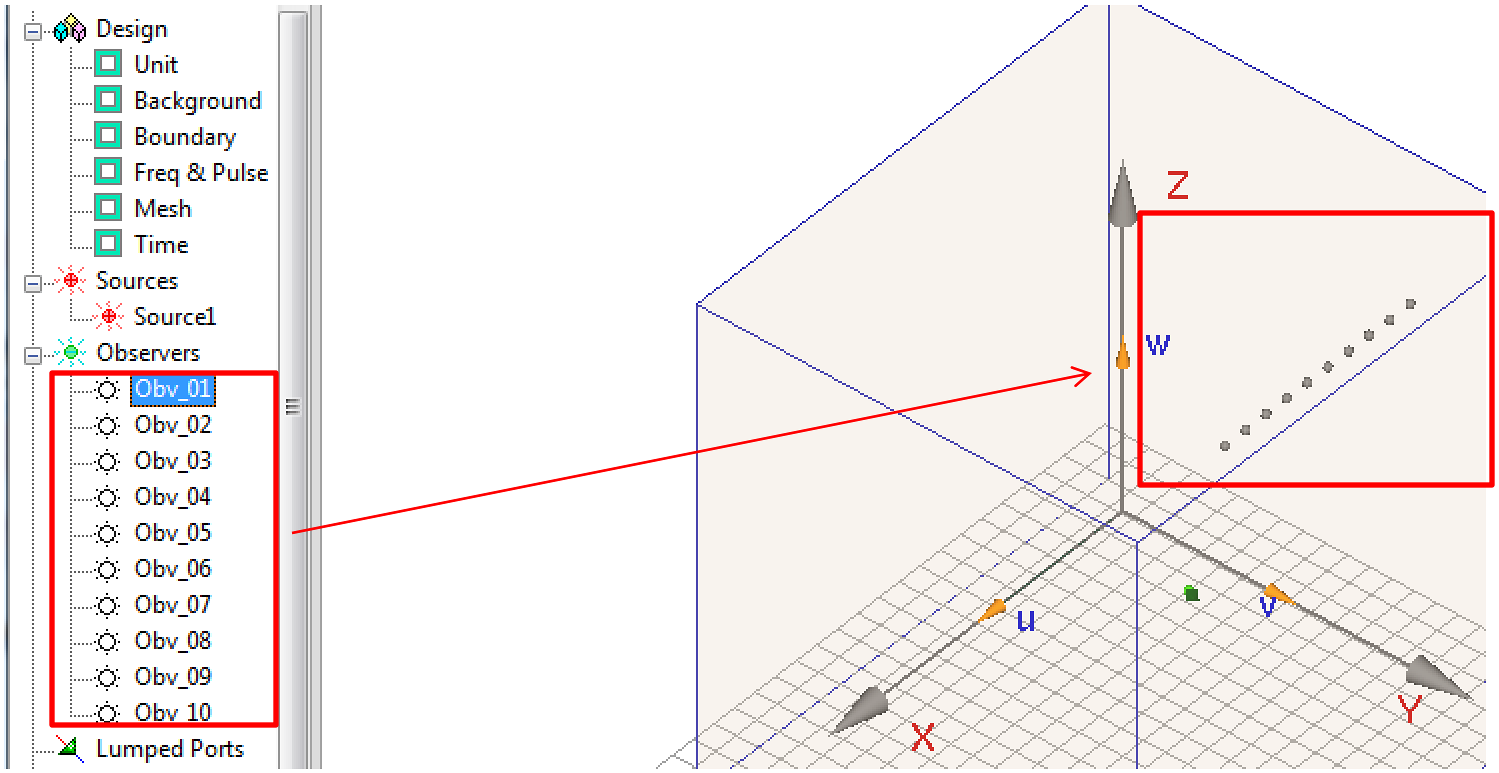




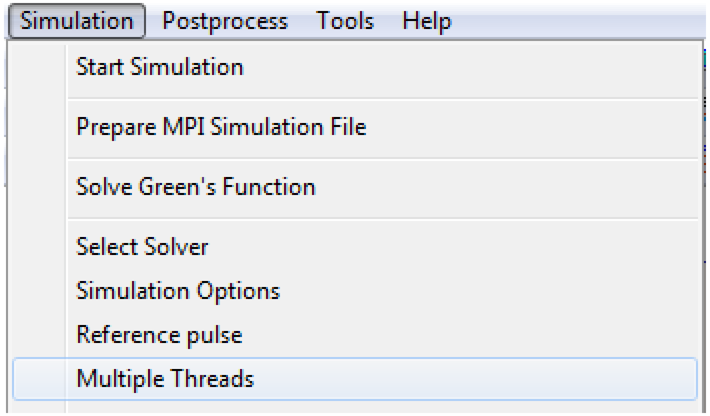
Then create 2nd observer with a distance of 50 m in X axis

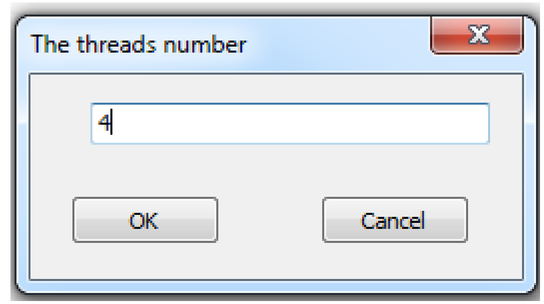


Finally, we create 10 observers in a line

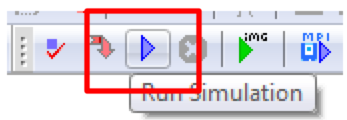


Define 4 threads to run simulation

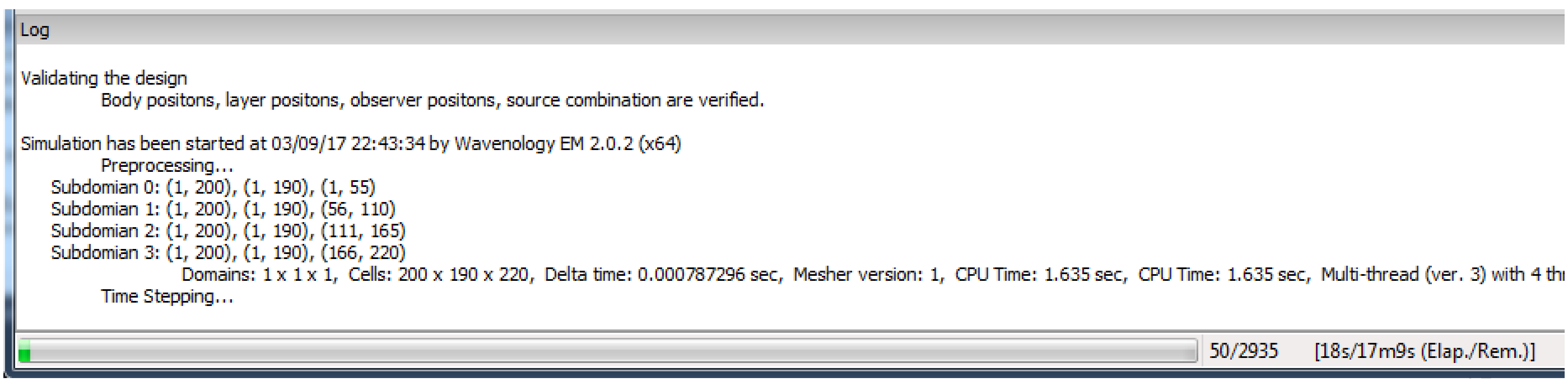




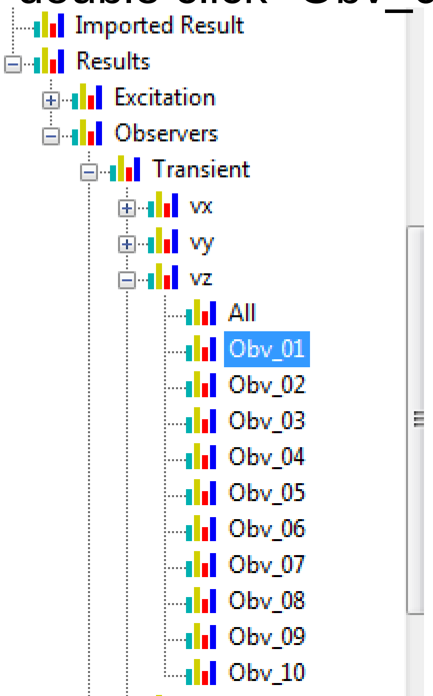
Now, we are ready for this case, "Start" simulation



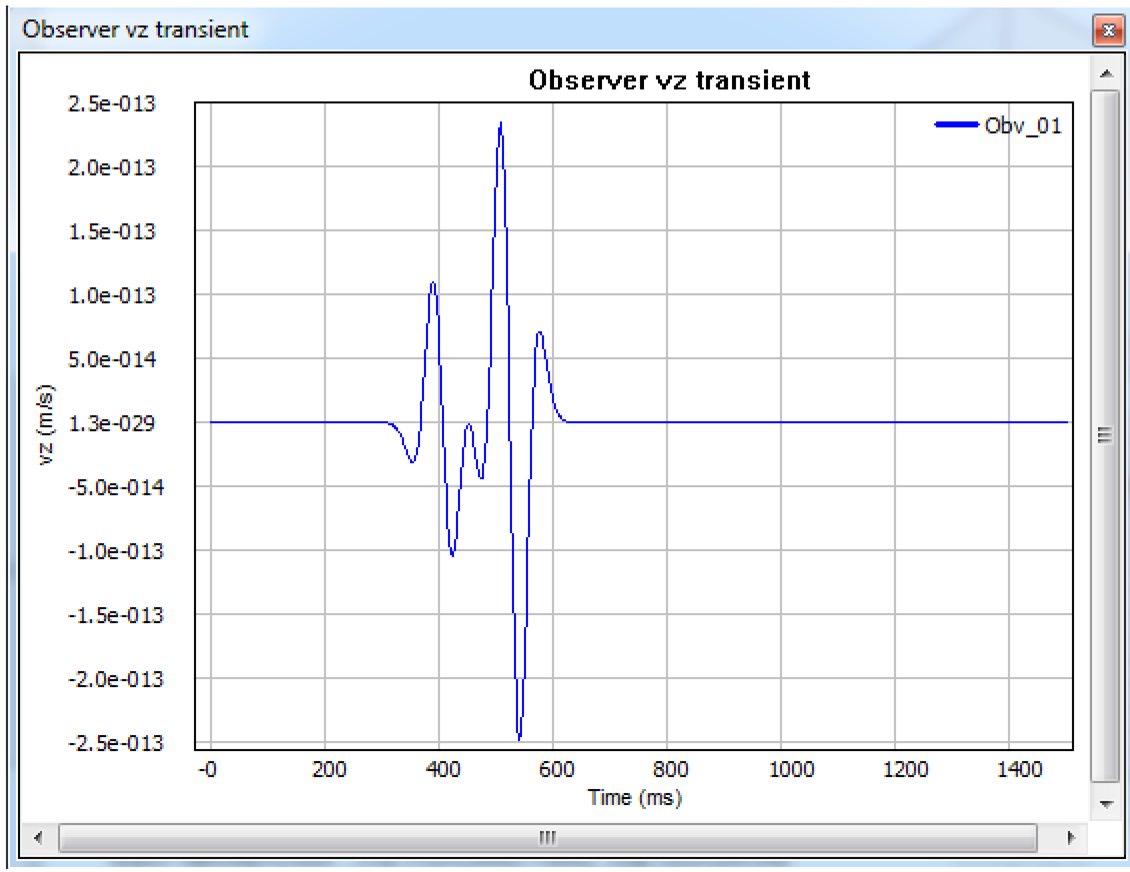
Here is the simulation progress report



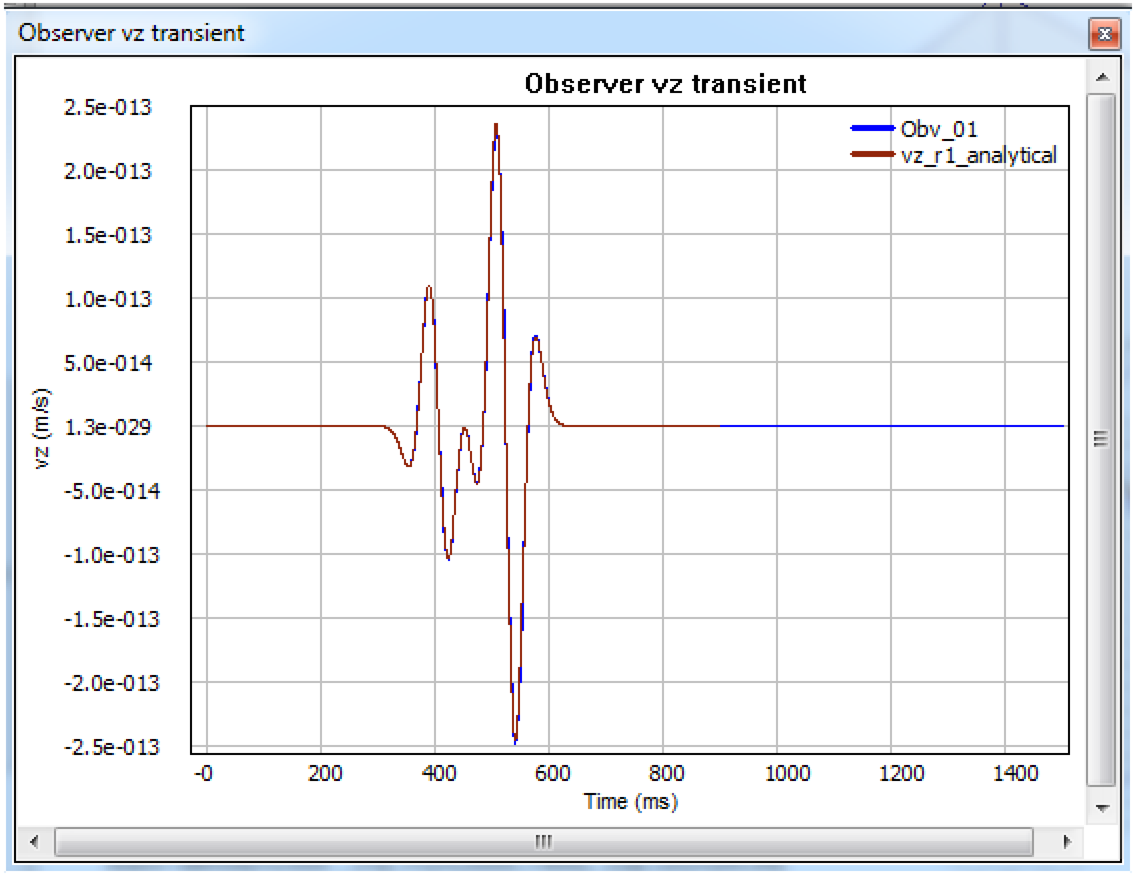
After simulation finish, expand "Results"->Observers->vz, double click "Obv_01"



This is the vz on observer "Obv_01"

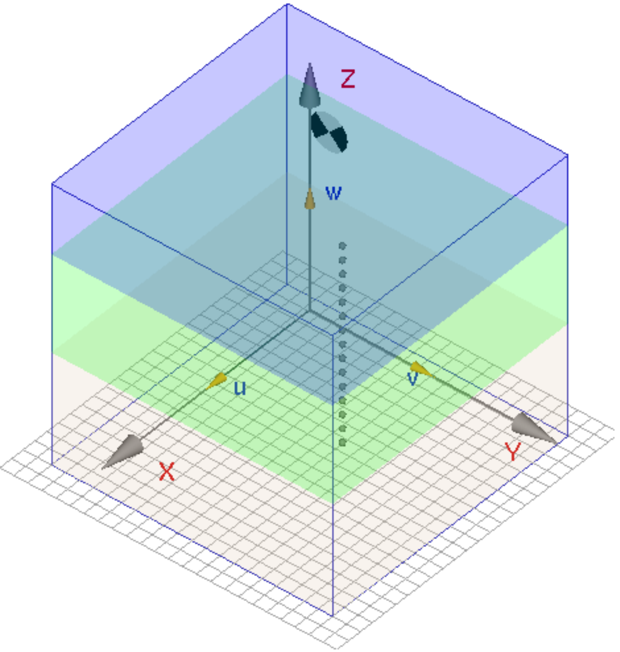


WCT simulation result compared with analytical solution



Tutorial Case II
Point Moment Tensor Source in Layered Background with Soft boundary on Top

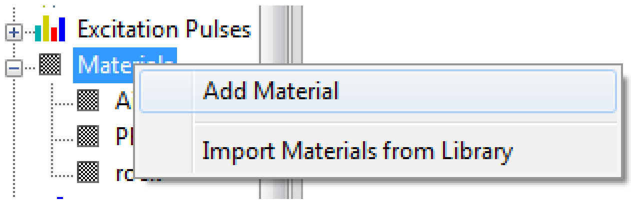
General isotropic material

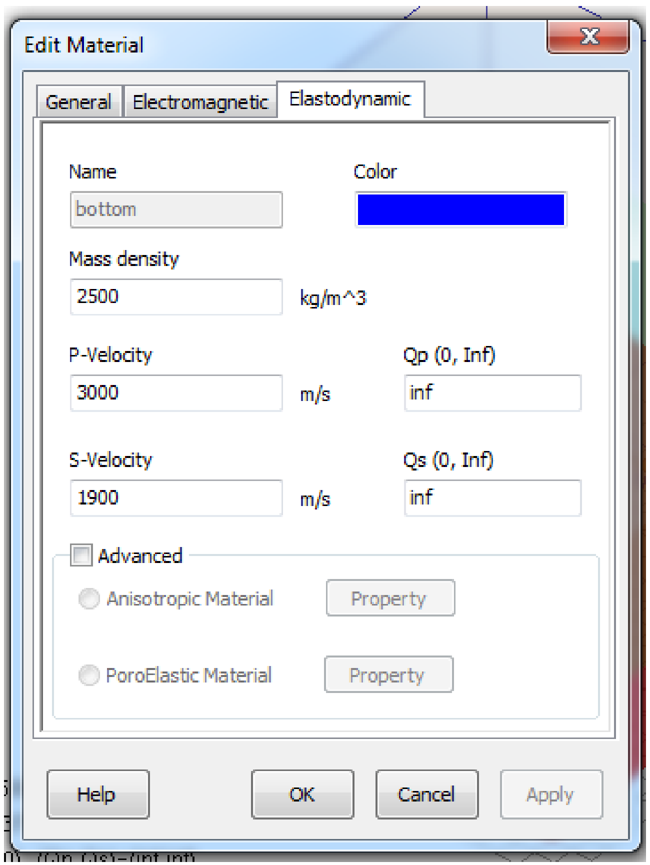


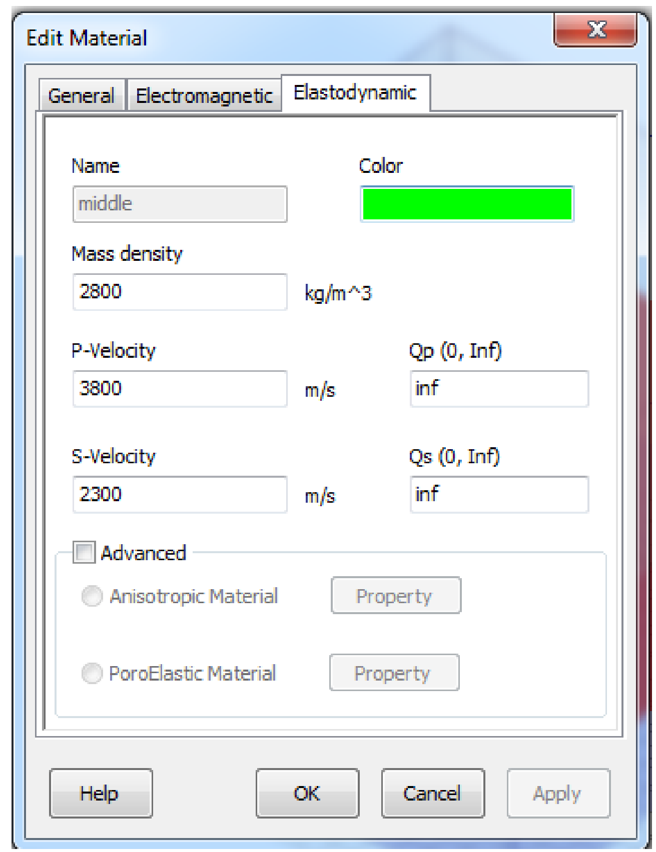
This case is in the demo package:
elastic\ layered\case_2_monopole.wnt

Save tutorial case 1 as "case_2_monopole"

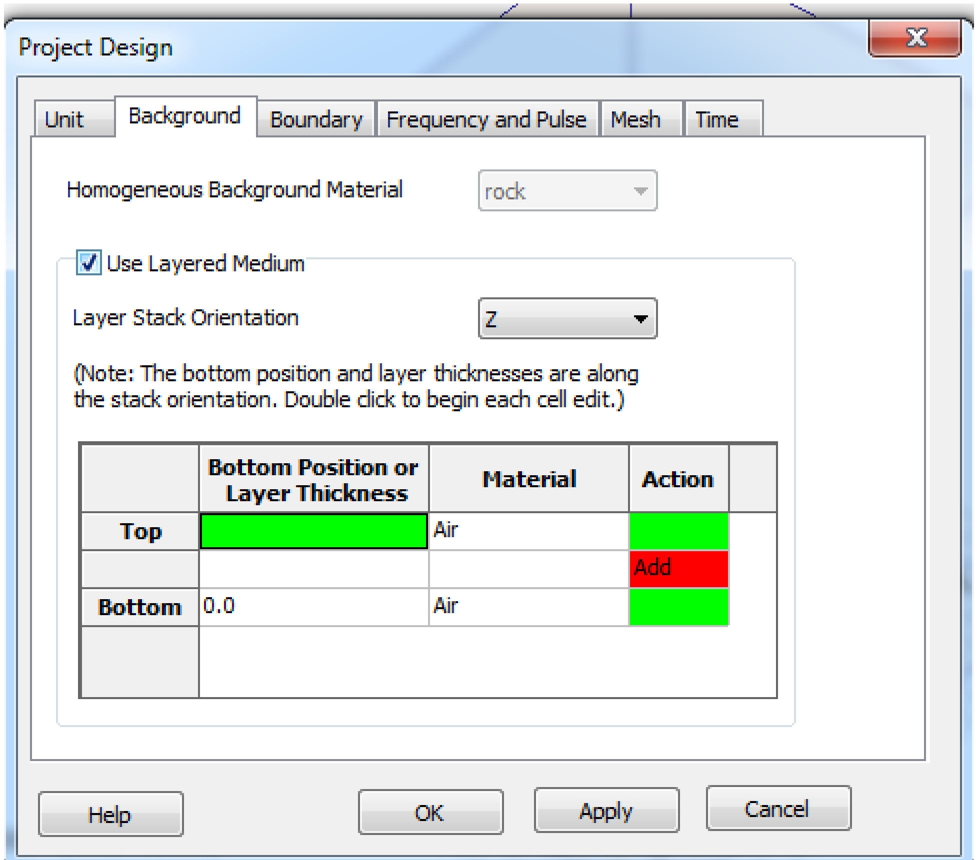
Then create new material "bottom" & "middle" for layered background



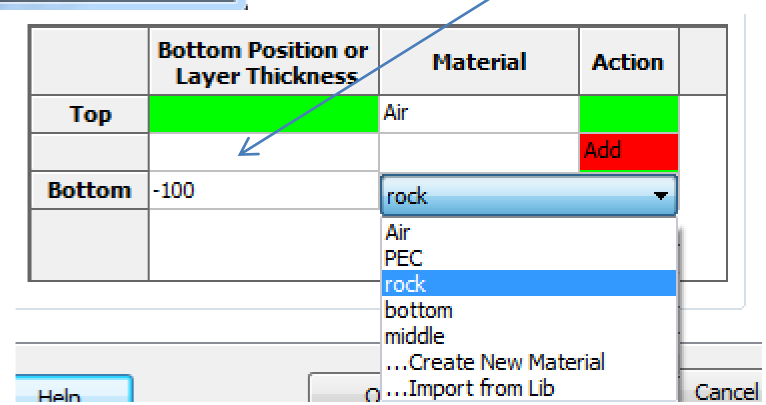




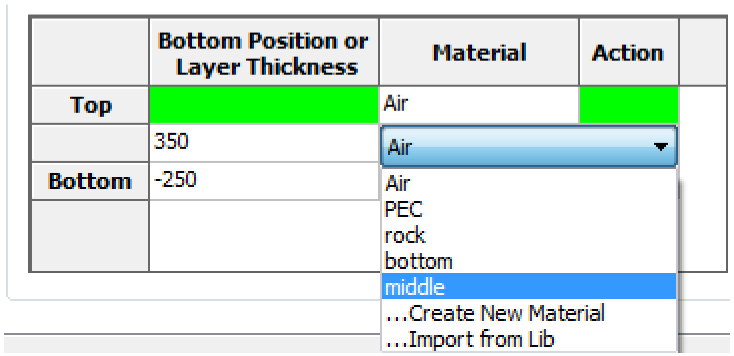
Change background to layered medium

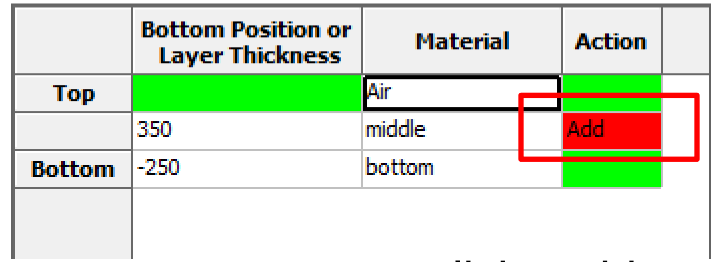


set bottom layer as “rock" material, it's range is below -100 m



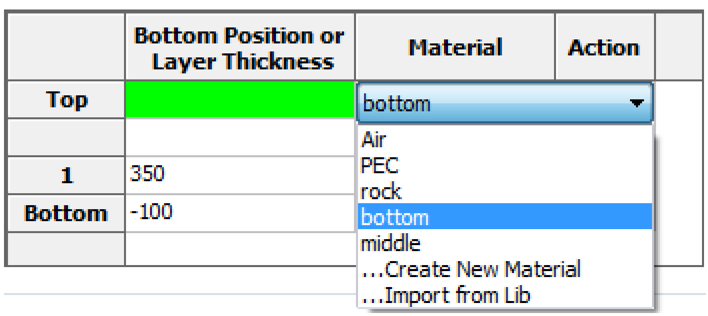
Then define the 2nd layer has a thickness as 350 m, material is "middle"



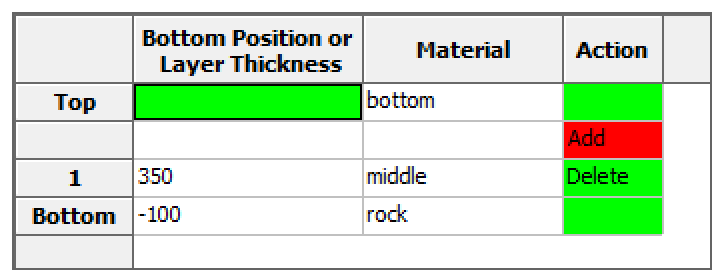


click "Add" to add 2nd layer

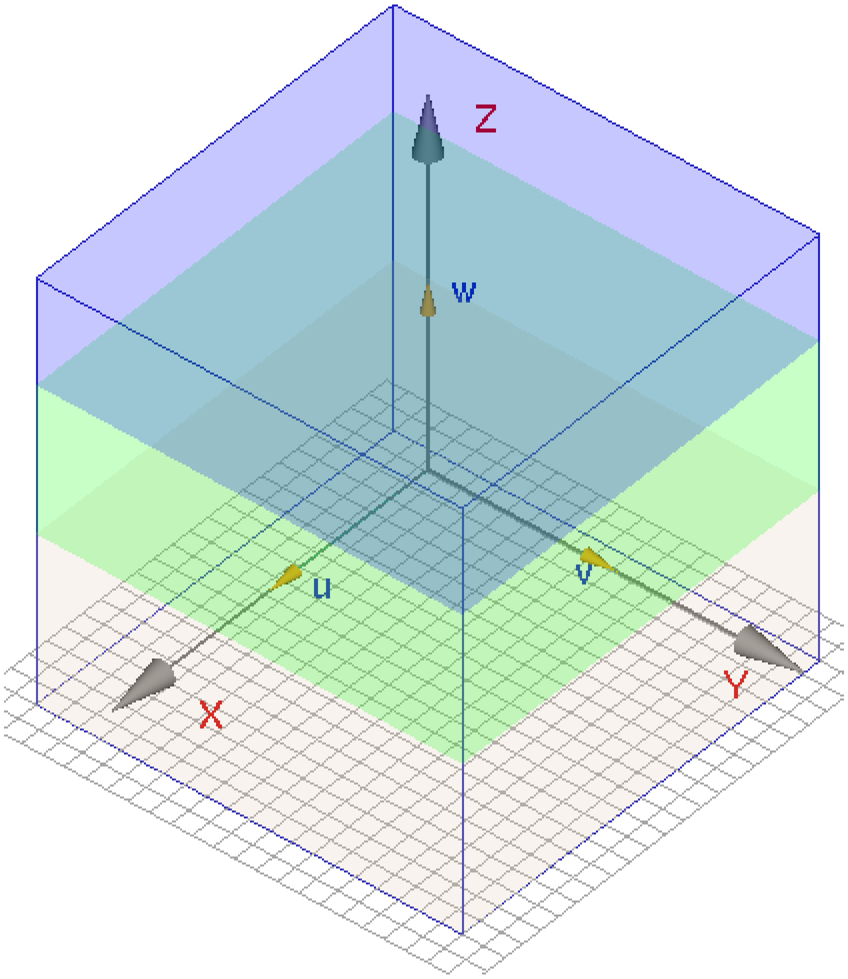
Set the top layer as material “bottom"



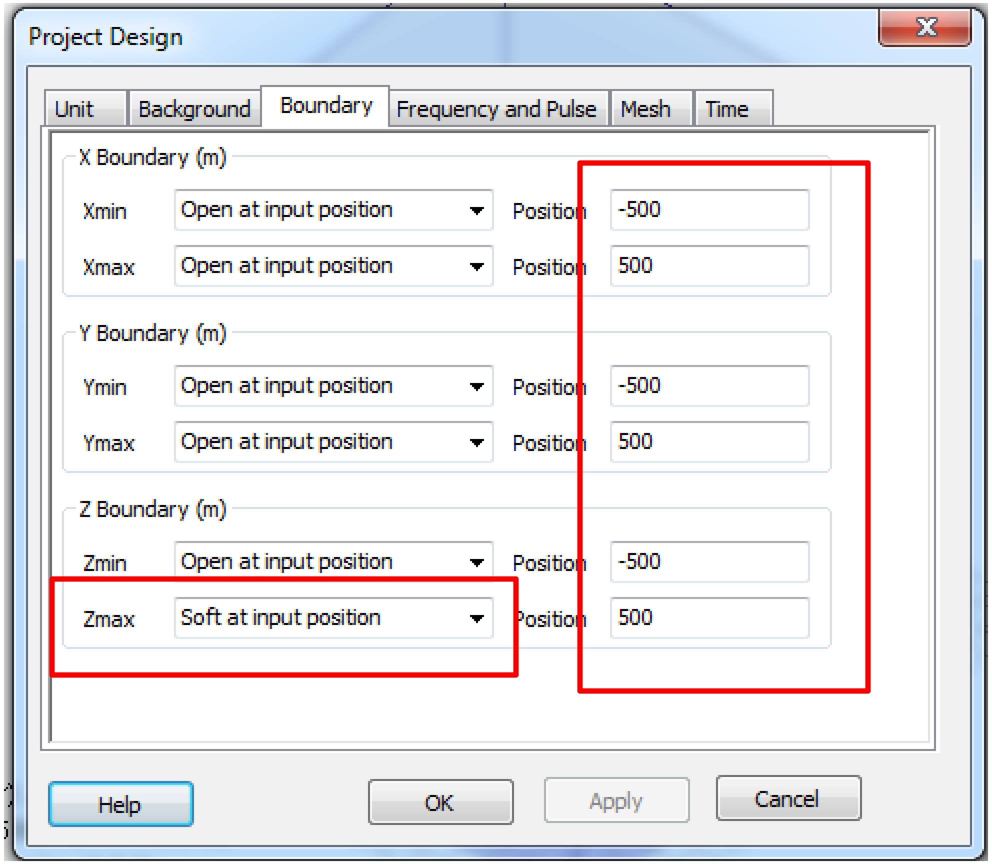
After 3 layers are set, it will be as this



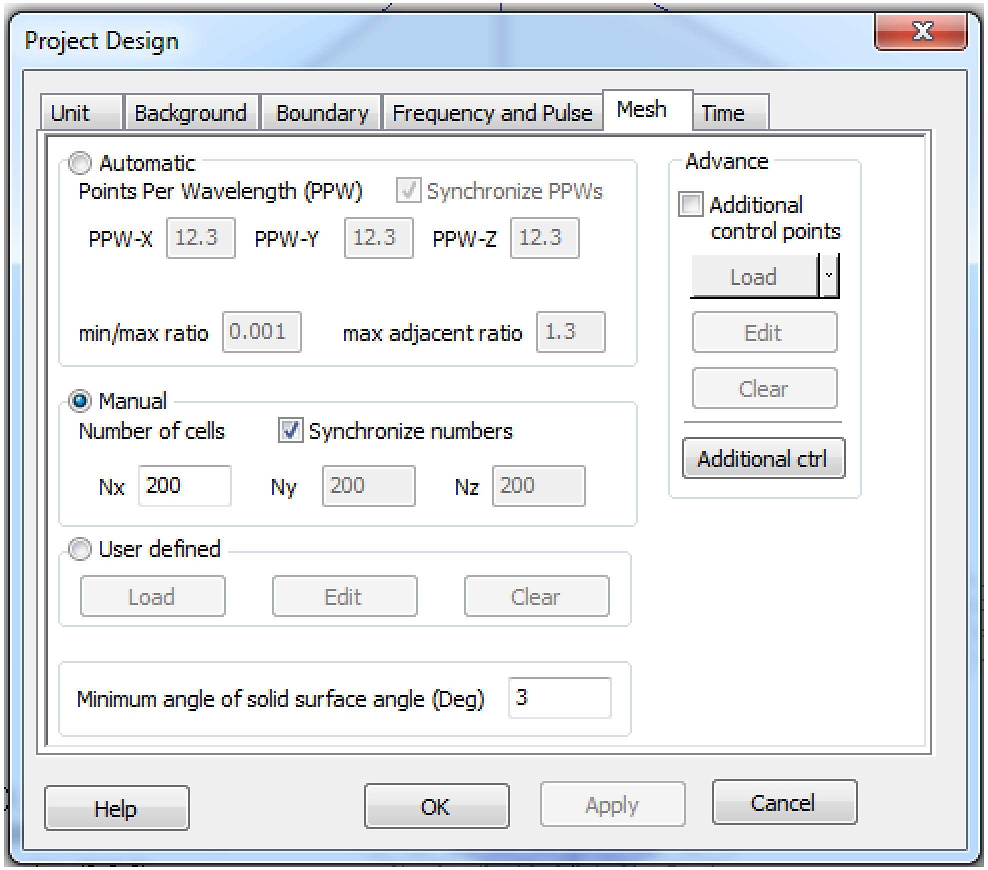
The 3 layers background
will be as this



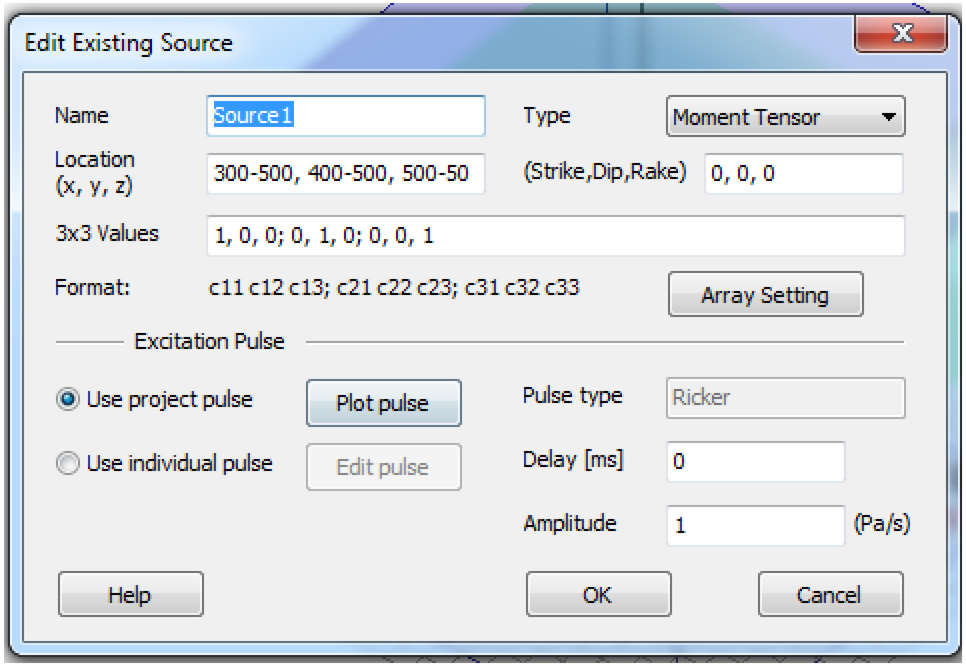
Change boundary size and top B.C. type



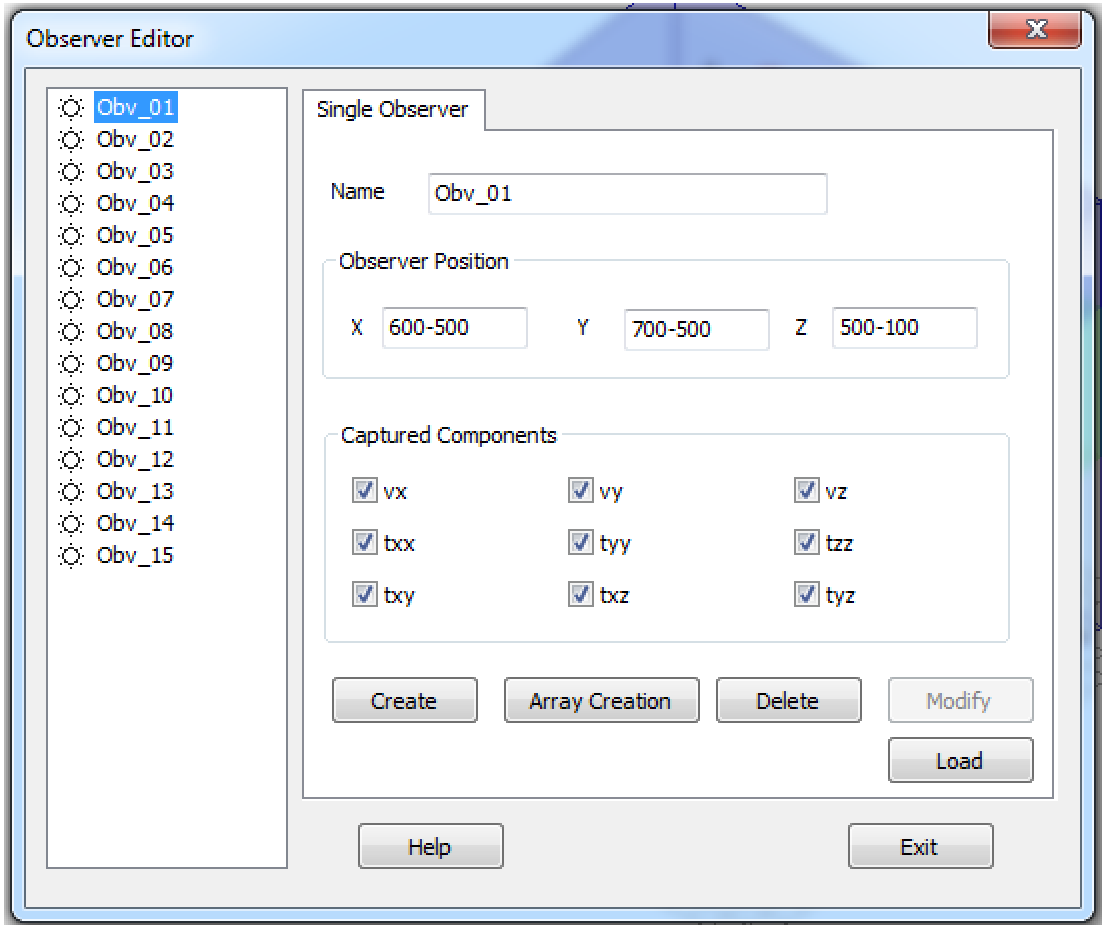
change cell numbers to
200 both in X, Y & Z axis



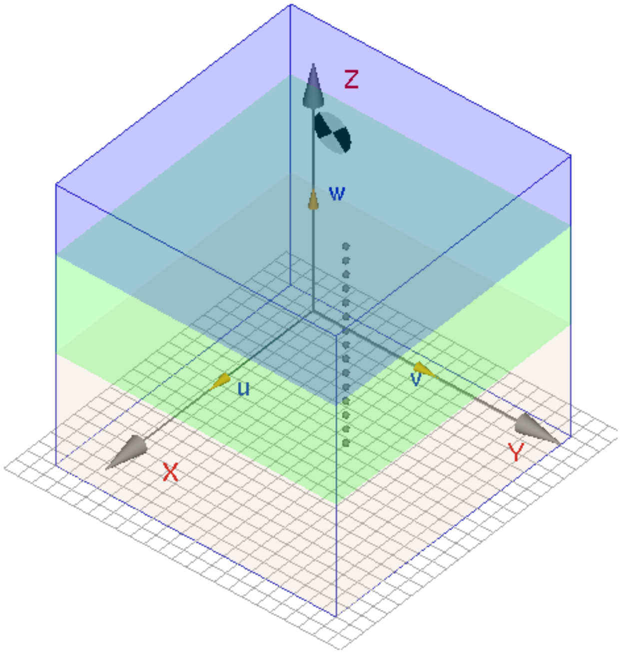
change source to "Moment Tensor", property as following



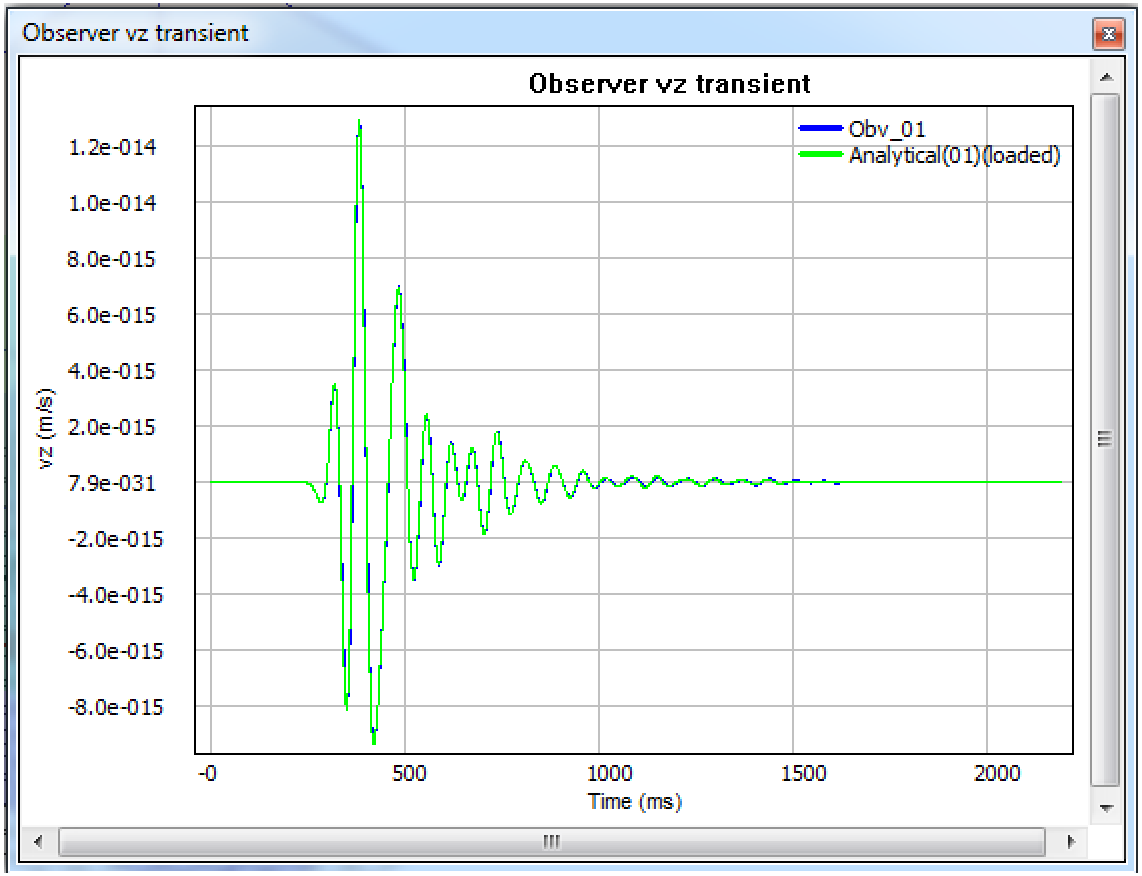
Make 15 observers along Z, Z positions are (400:-50:-300)



Finally, the project is as this, start simulation

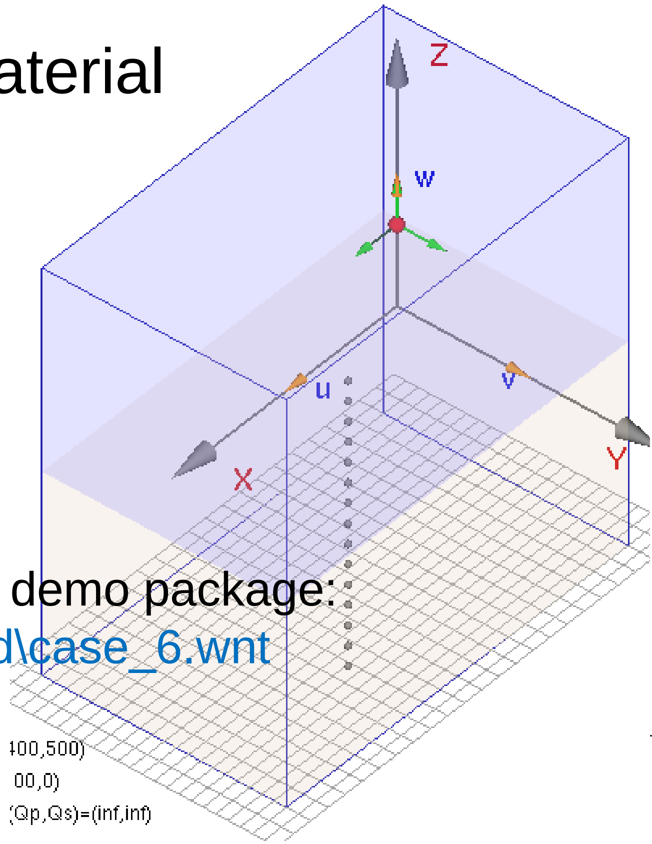


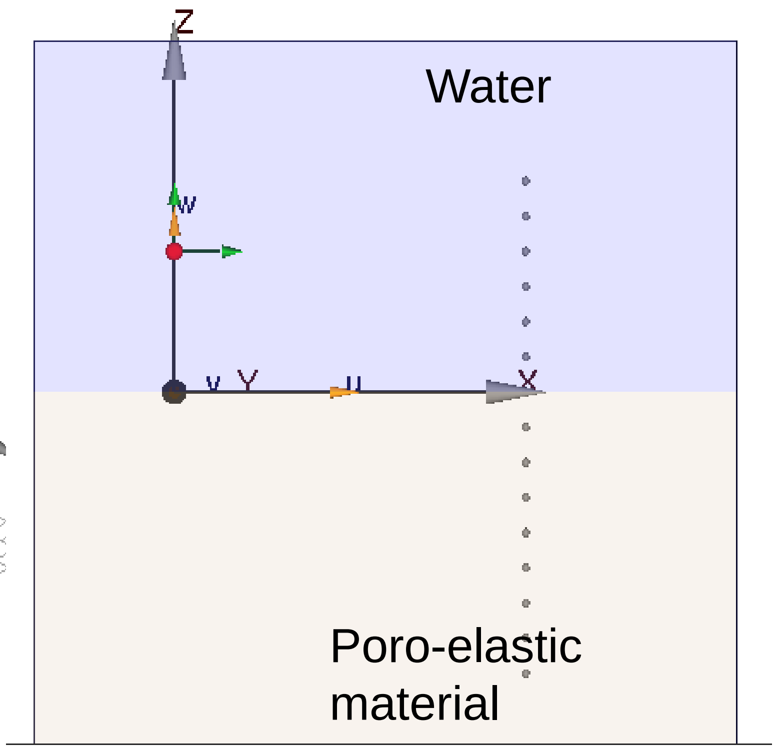
WCT simulation result compared
with analytical solution



Tutorial Case III
Point Moment Tensor Source in Layered Background

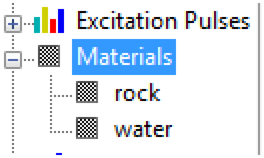
Poro-elastic material



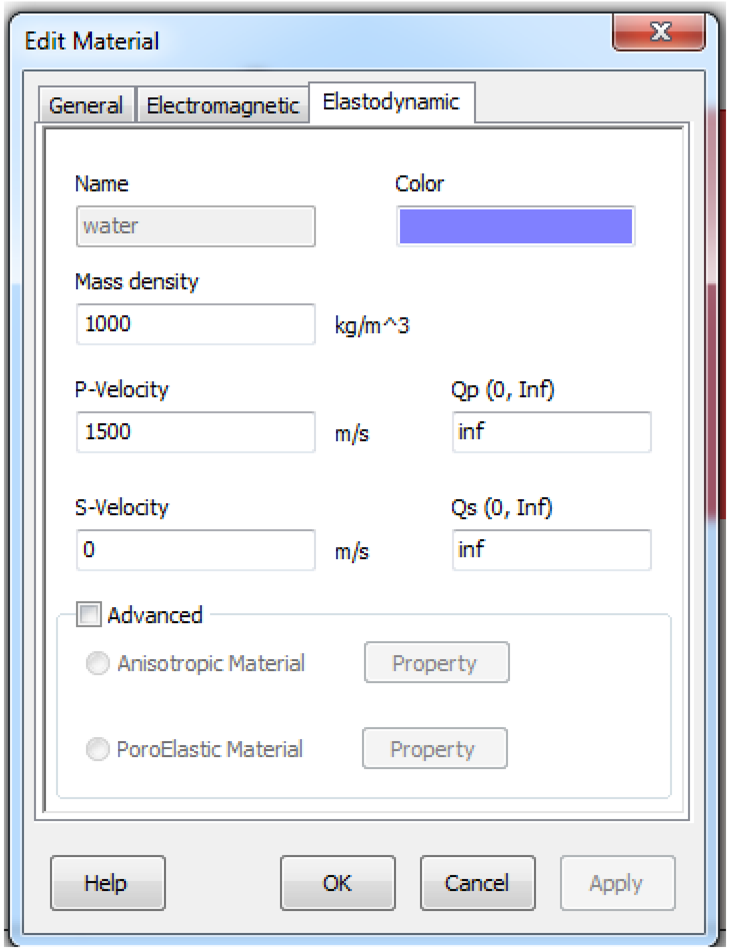


This case is in the demo package:
poroelastic\layered\case_6.wnt

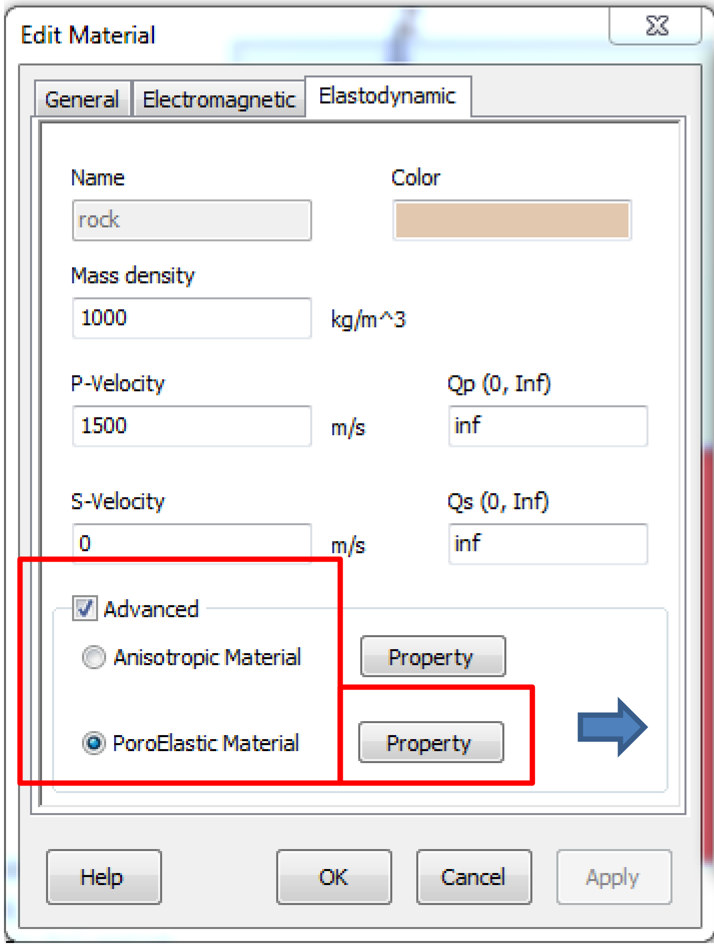
We define two materials

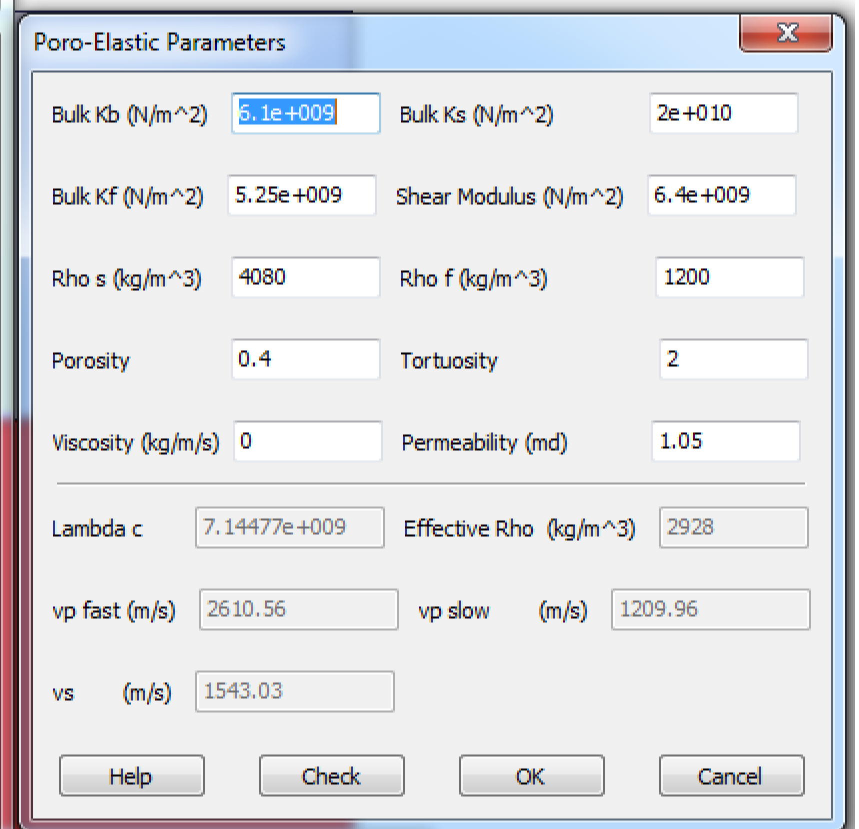


Water parameter

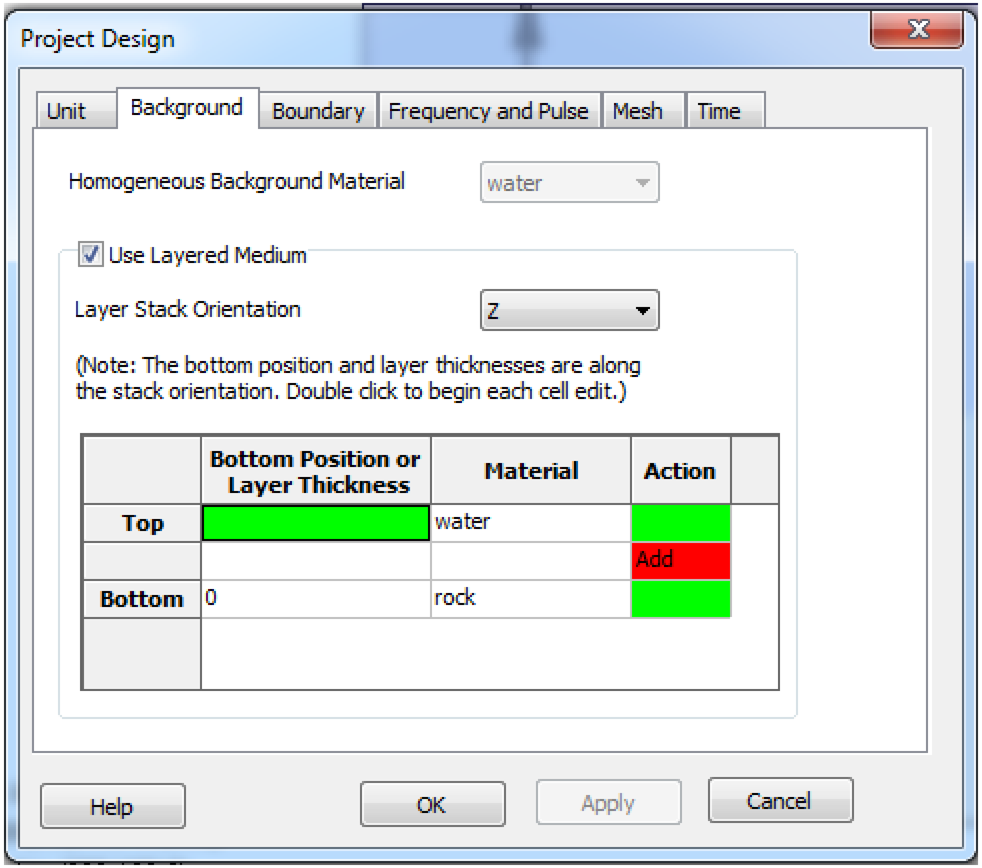


rock has a poro-elastc parameter as

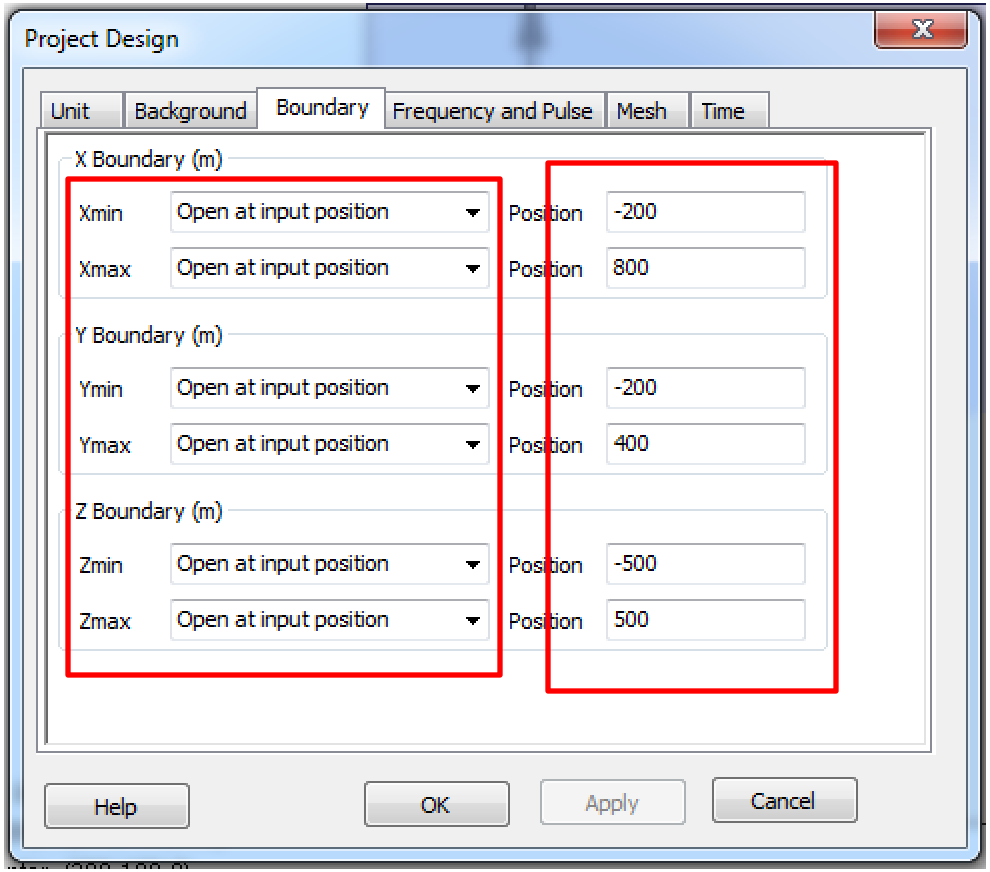




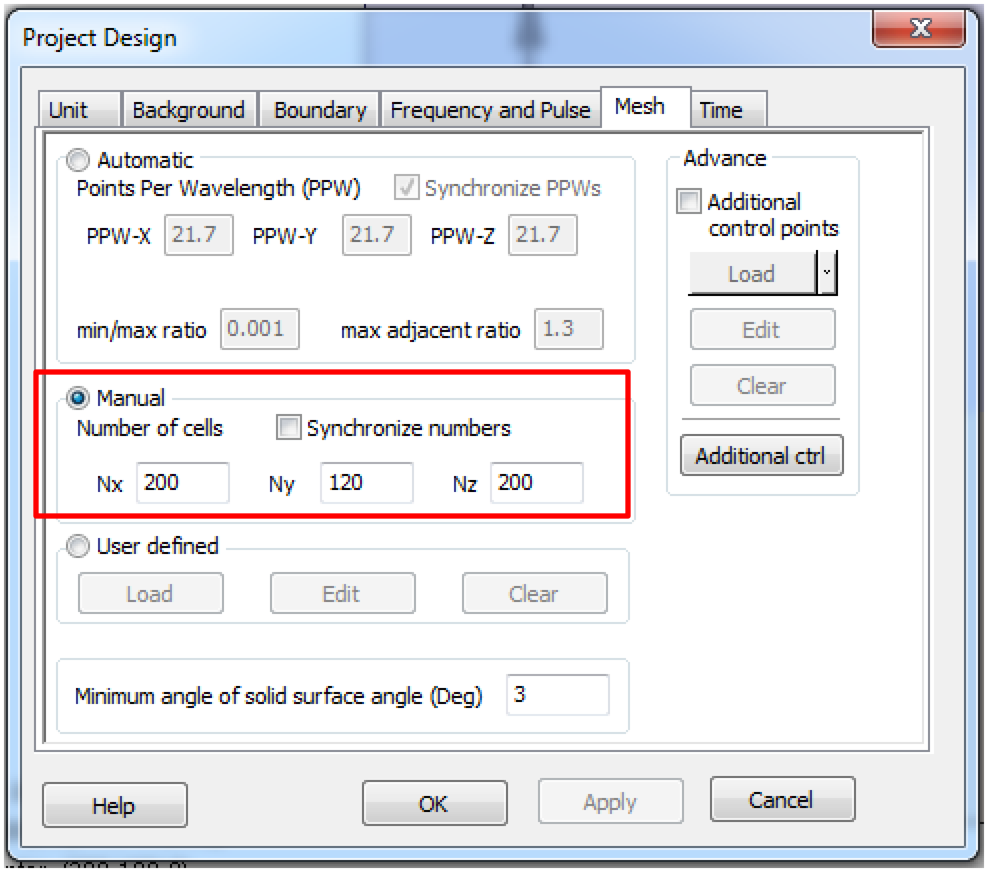
Background as



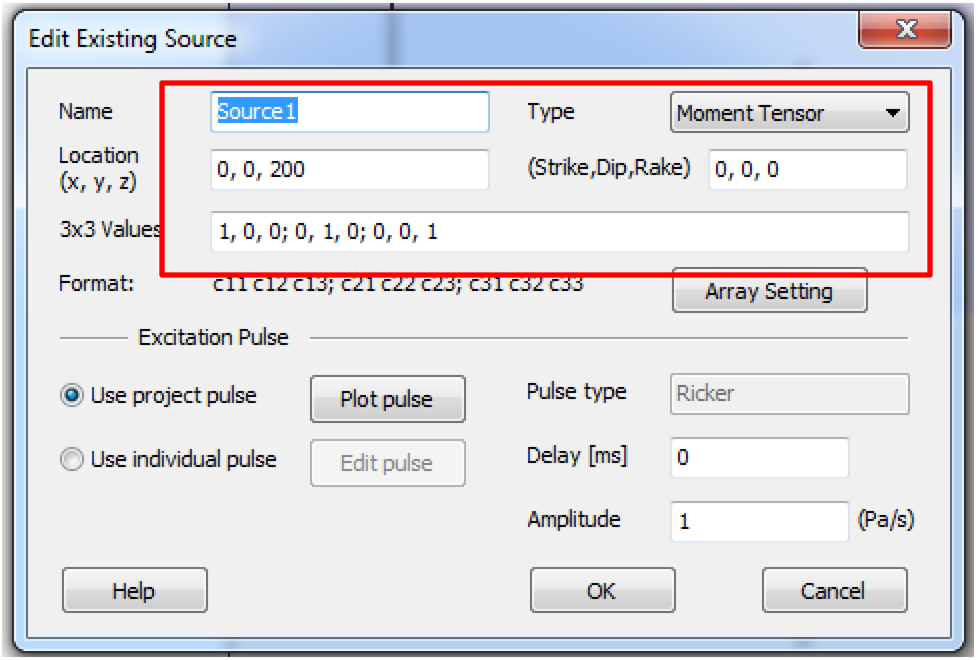
Change boundary size and top B.C. type



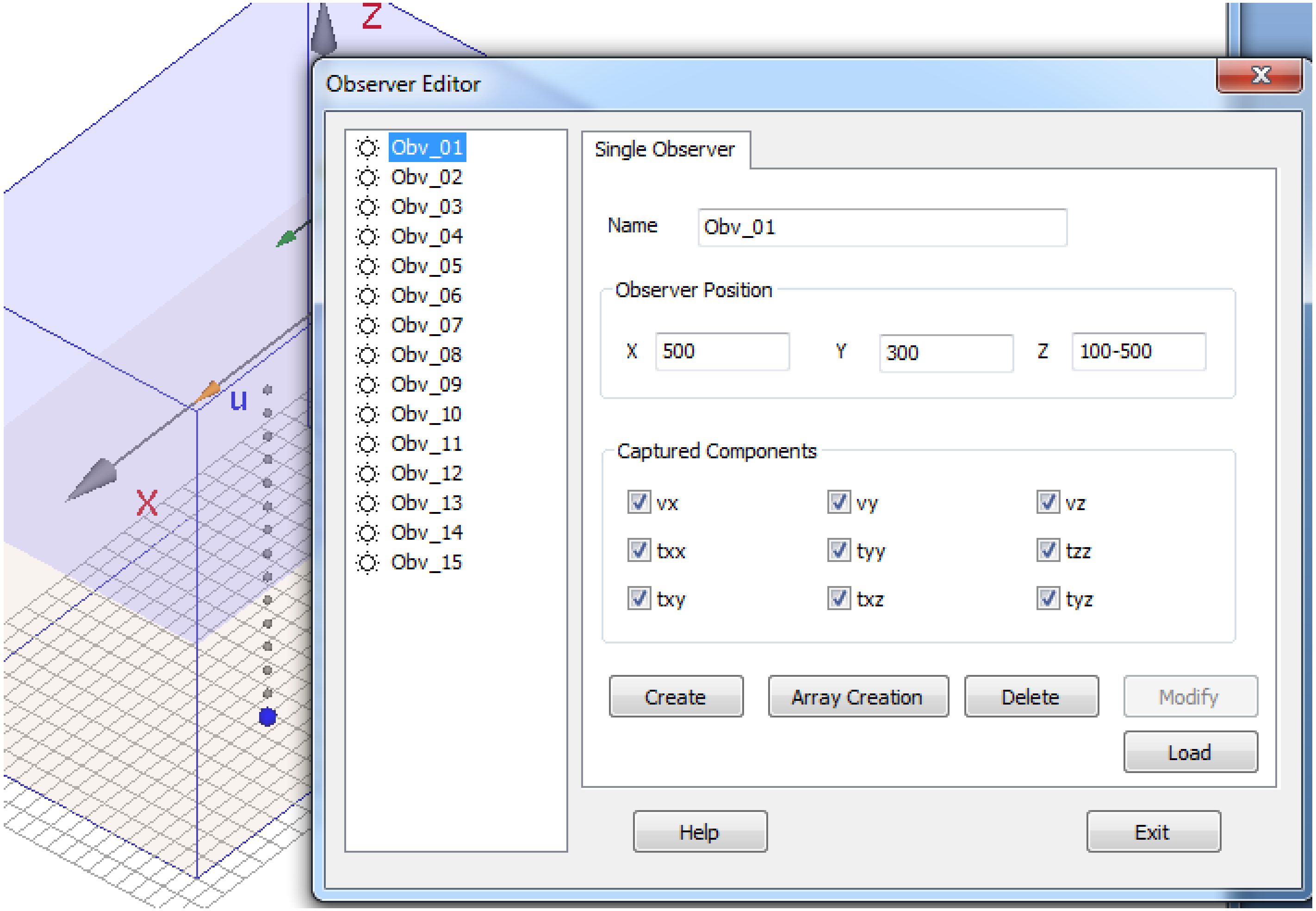
cell numbers is (200, 100, 200)
in X, Y & Z axis, respectively



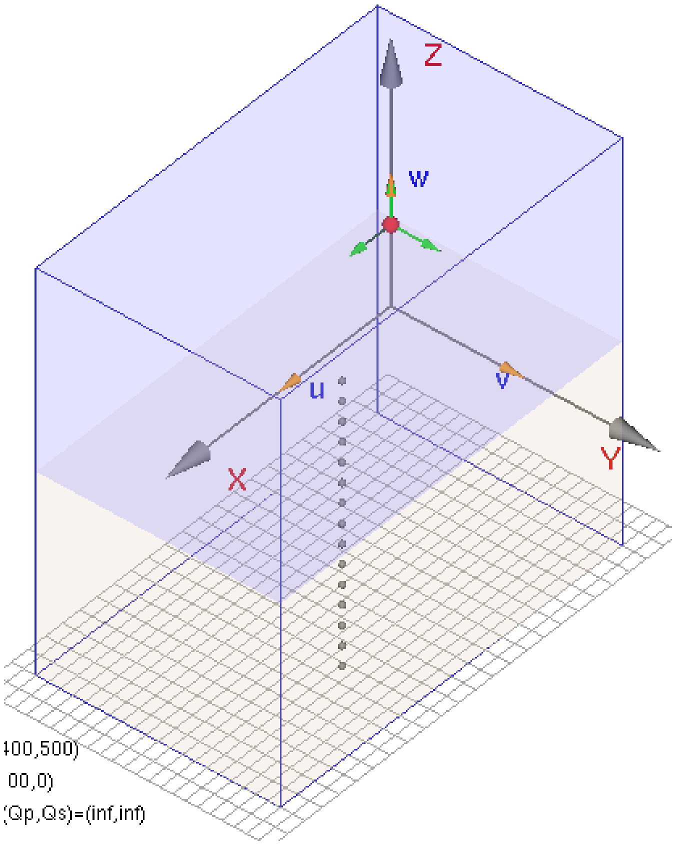
"Moment Tensor" property as following



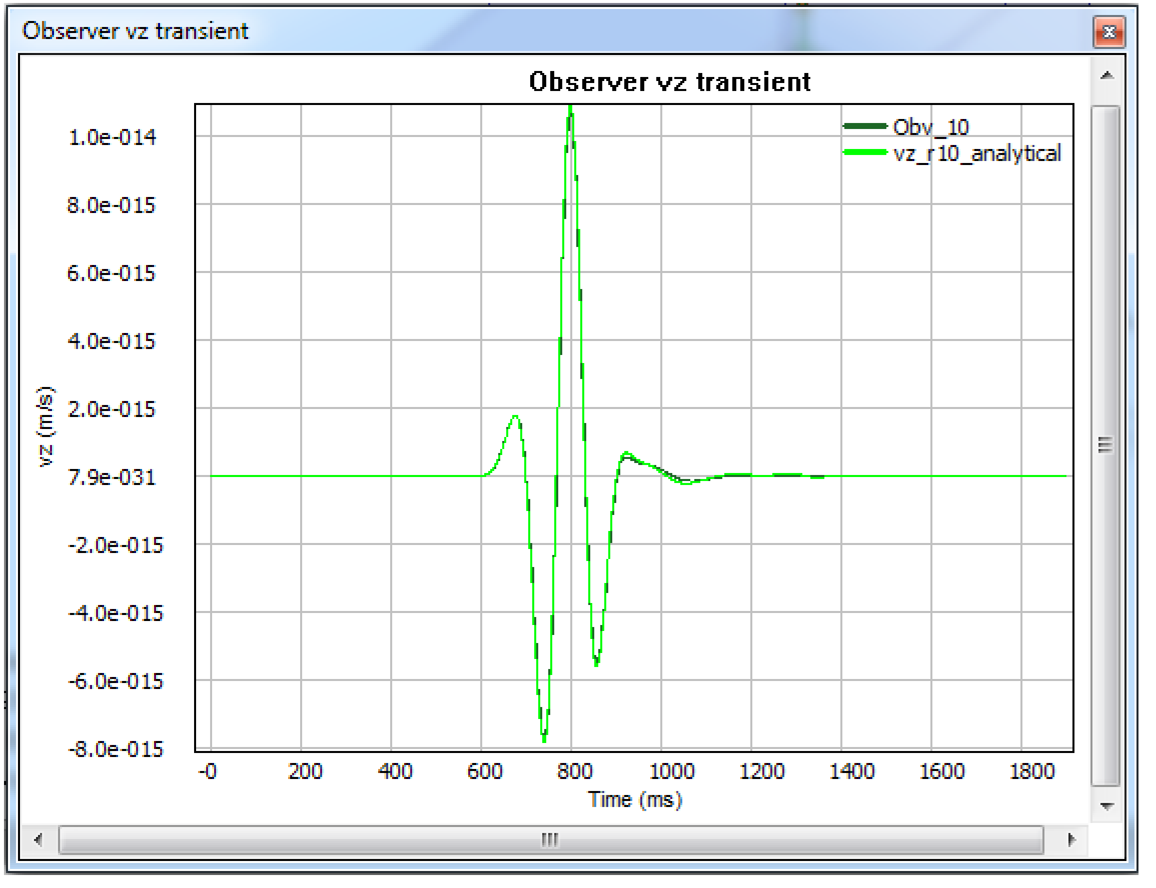
Make 15 observers along Z, Z positions are (-400:50:300)



Finally, the project is as this, start simulation



WCT simulation result compared
with analytical solution



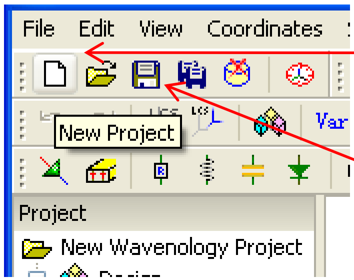
Special Treatment for Importing SAT Models for Elastic Wave Project

For a elastic wave project, there is a special treatment to let user can import 3D models from SAT file with material name and profiles.

1. The “ProductId” part of the SAT file must be the string “ElasticWave”.
2. There must be two accompany files to define the material usage information and material profile.
   For example, the SAT file is A.sat, there are 5 models in the file.
   2.1 User must define a material usage file as name A. SatMat. The file is an ASCII text format. Each line lists the material name for a model. For example:
   water
   steel s1
   …
   2.2 User must define a material profile file as name A. MatDef. The file is a ASCII text format. Each line lists the profile for a material. The name, velocities are separated by ‘,’. For example,
   water, 1000,1500, 0
   steel s1, 7890, 5790, 3235
   The first column is the material name; 2nd is the mass density (kg/m3); 3rd is the P speed and 4^{th} is the S speed (m/s).

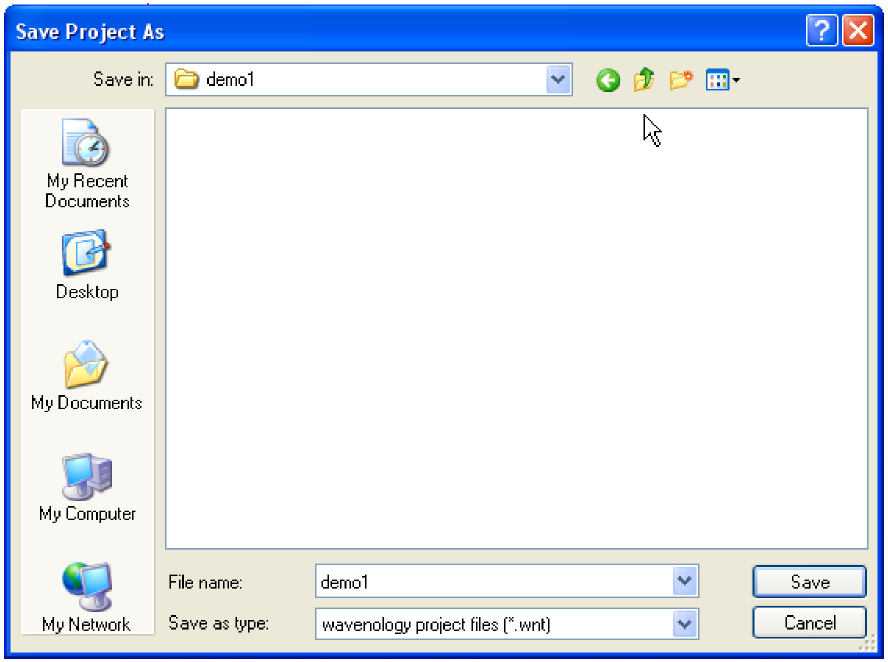
Tutorial Case
Import 3D Solids from a SAT file

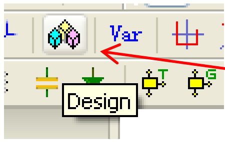
(Here, we use Cylindrical Elastic Solver, but for Cartesian Elastic Solver, it is the same procedure)



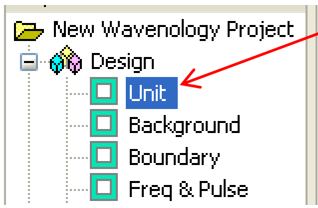
Use "New Project" button create a new project

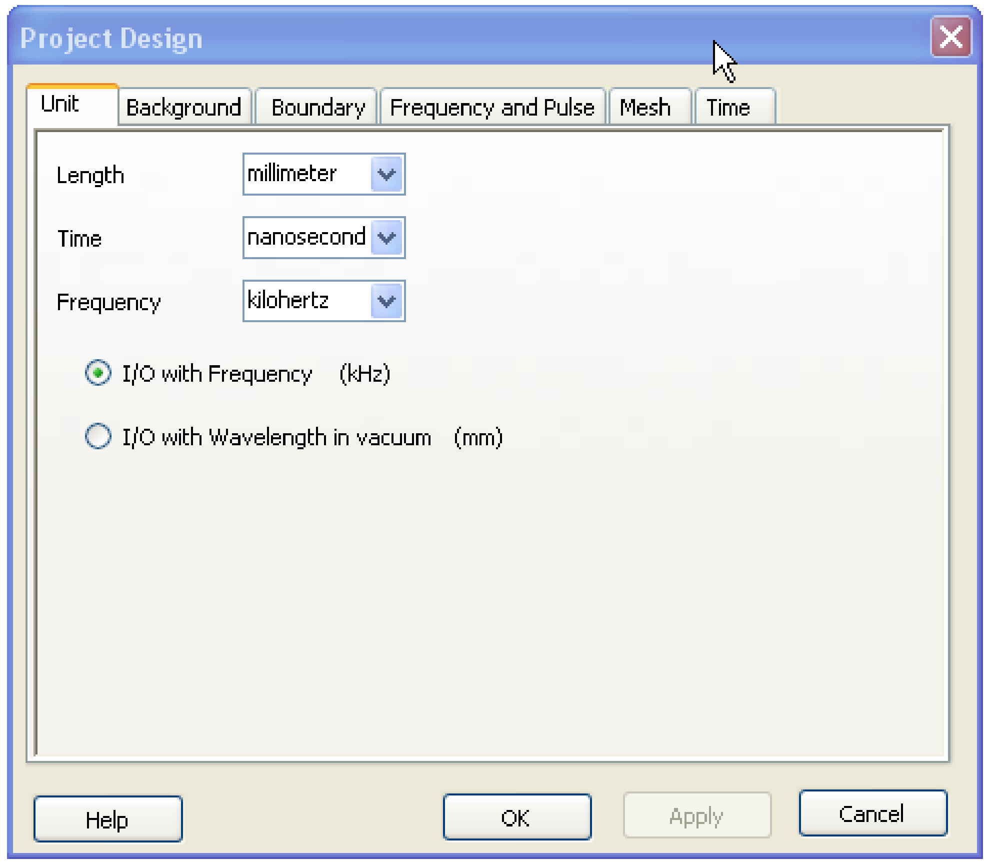
It is better, then, to save the project with a name under the target folder. Here, we define the project name as "demo1" under folder "demo1". Because in order to generate the mesh data file, the software need a predefined folder to export them. Without this predefined folder, the data file will be saved in some system default folder and will be hard to find.





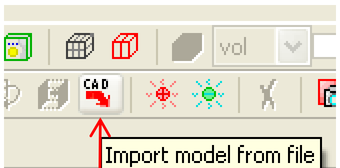
Use “**Project Design**” button or “**Unit**” treenode to modify project unit



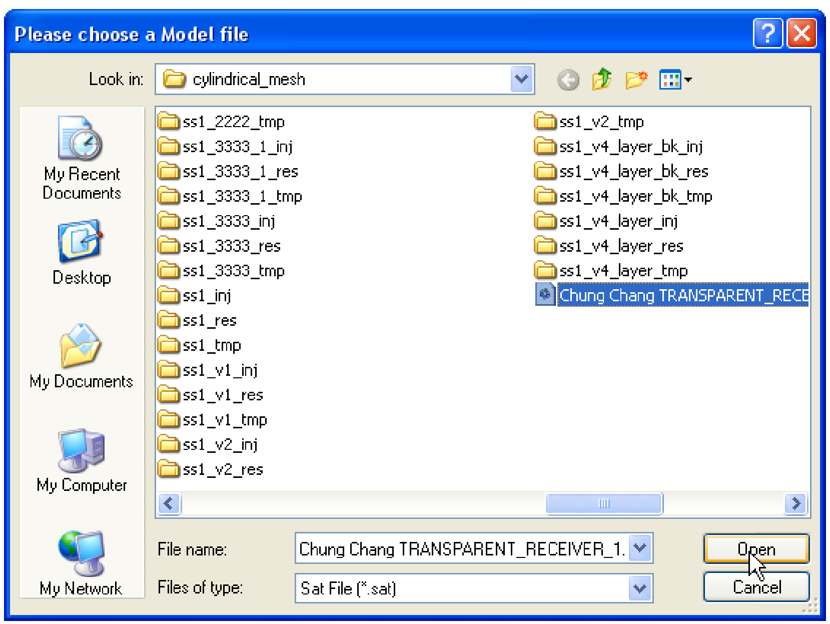


Define project unit as “mm”, “ns” and “KHz” . Other settings use ***Default*** values

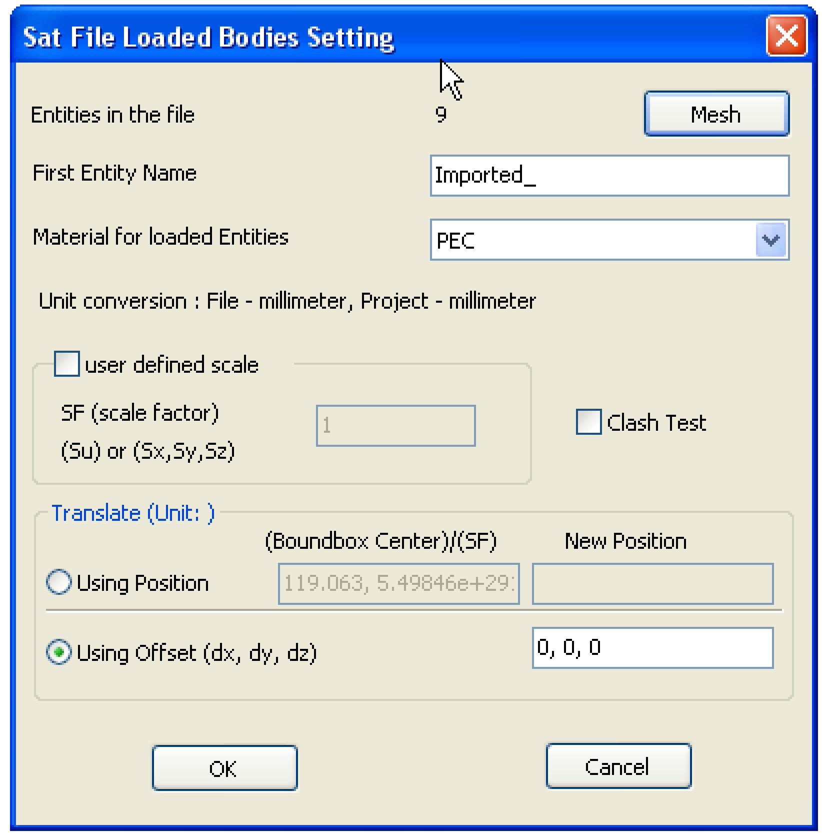
Insert 3D solids from an SAT file. For example, "TRANSPARENT_RECEIVER_1.SAT"



Use this Toolbar button



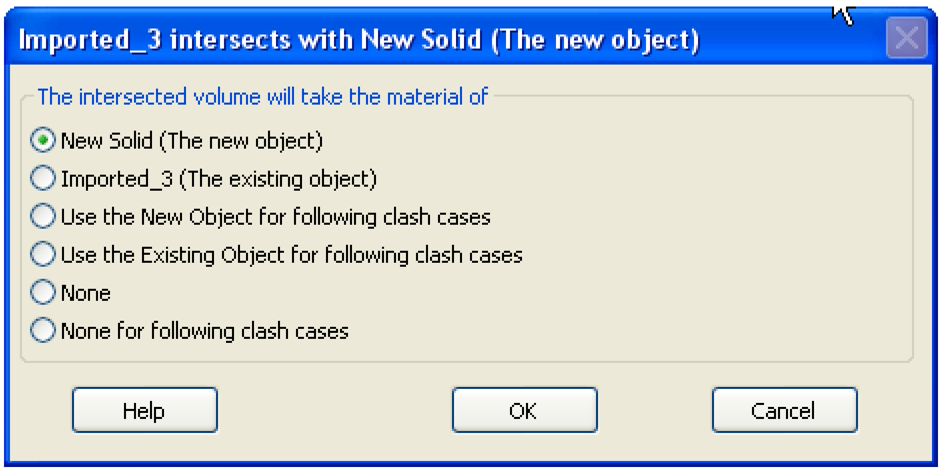
Select the SAT file



Define the name, default material and position for all imported solids

Due to the following reasons:

1. Different 3D modeling software has different tolerance. Not-intersect solids could be treated as intersect in porting among different software packages
2. The solids are actually intersect due to the original 3D modeling software that permits such an intersection.

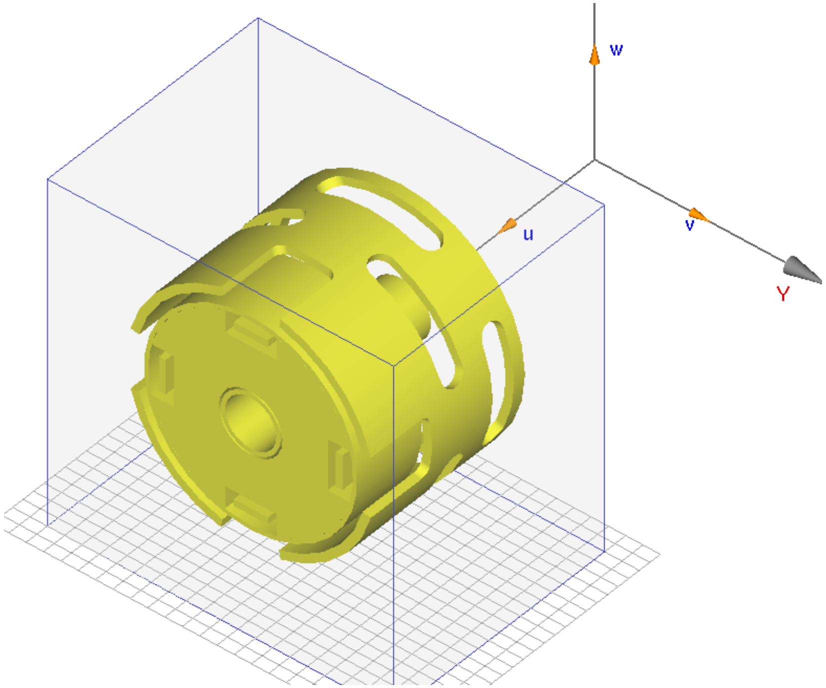


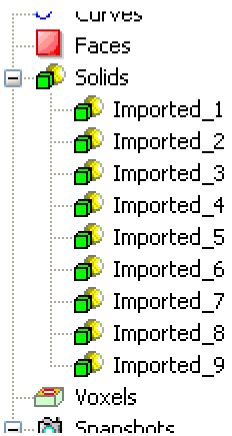
In importing this file, Wavenology finds there is an intersection among some solids and reports a warning.

User needs to decide how to handle intersections. Here, we let the solid ***Imported_3*** occupy the intersected region.

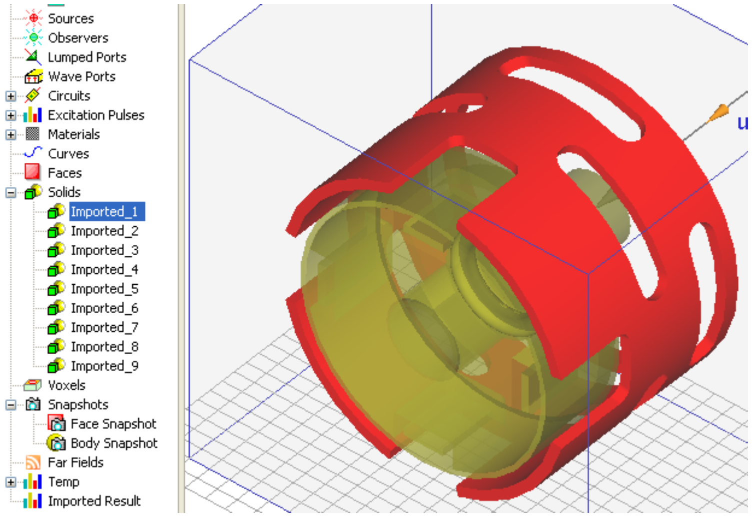
Note: WCT provide a special treatment for the cylindrical mesh generation and elastic wave simulation. If user consider the imported models should not be reported as clashing and these clashes will not effect the final mesh and simulation. User can choose "***None for following clash cases***" to let WCT skip clash test.

Following is the shape of the imported solids and their names

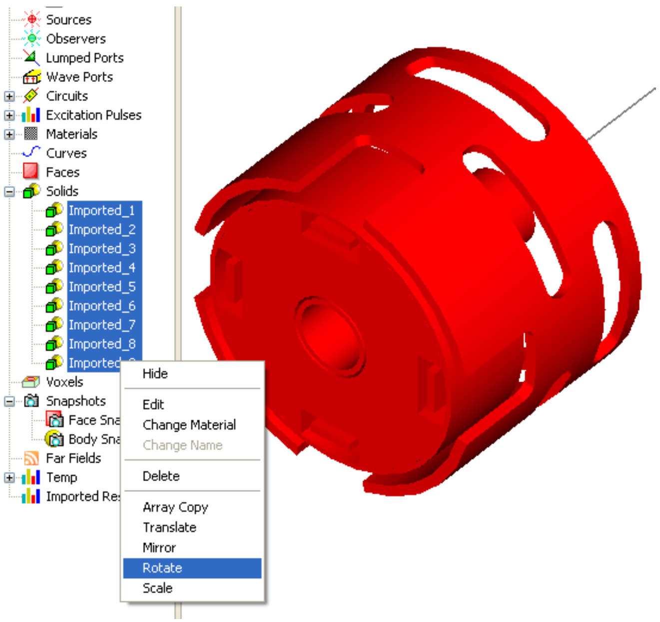


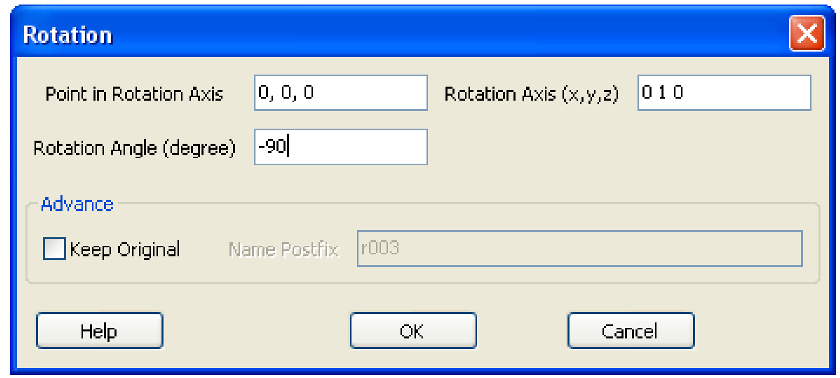


User can check the name-shape mapping by select a solid in the tree.

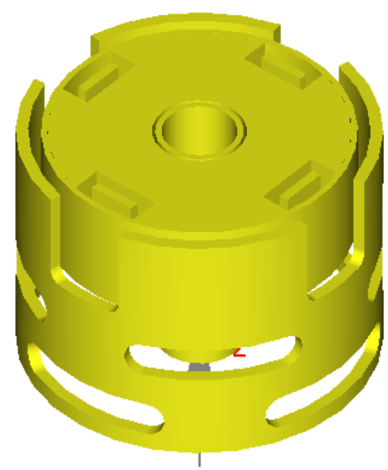


The current Cylindrical mesh generator only supports Z axis as the rotation axis, but the imported solids can be X aligned. Therefore, we need to align all solids to Z axis by **Rotation**.





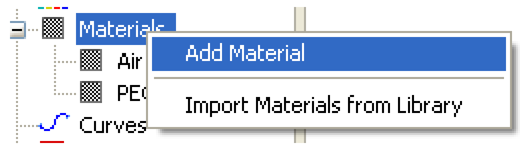
Let all solids rotate -90 degree along Y axis.
The input is shown in above dialog.
Note: CCW is positive degree, CW is negative degree.

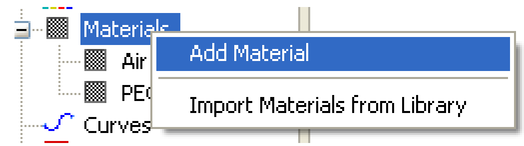


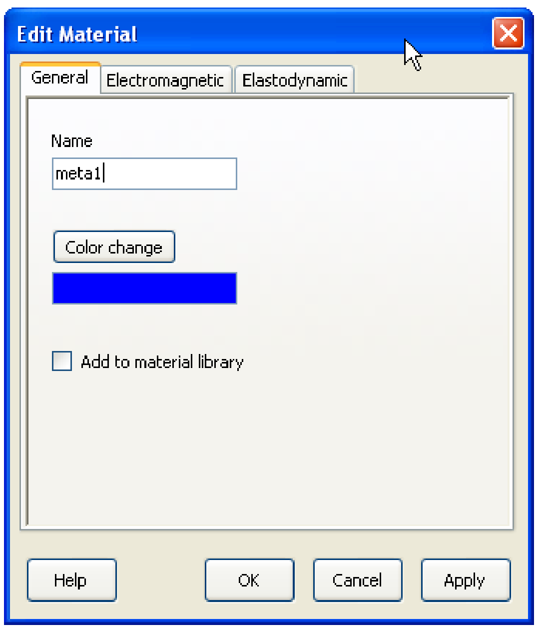
Press “ctrl” key and use mouse left button to select all solids, then press mouse right button to popup ***Solid Transform Menu***.
Press “**Rotate**” menu.

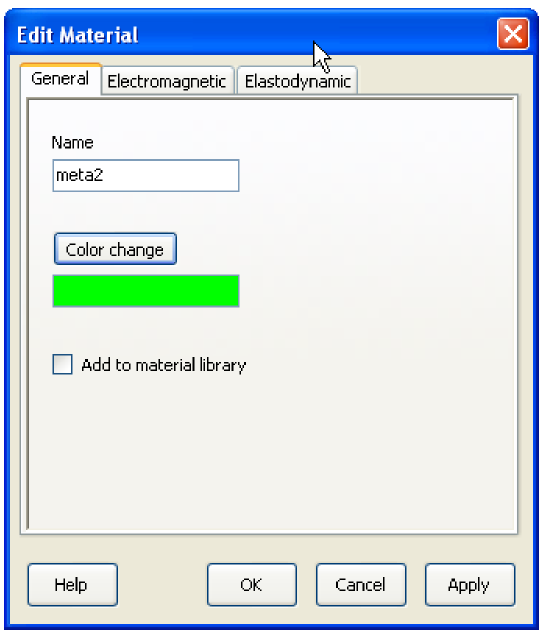
Define different materials for different components.

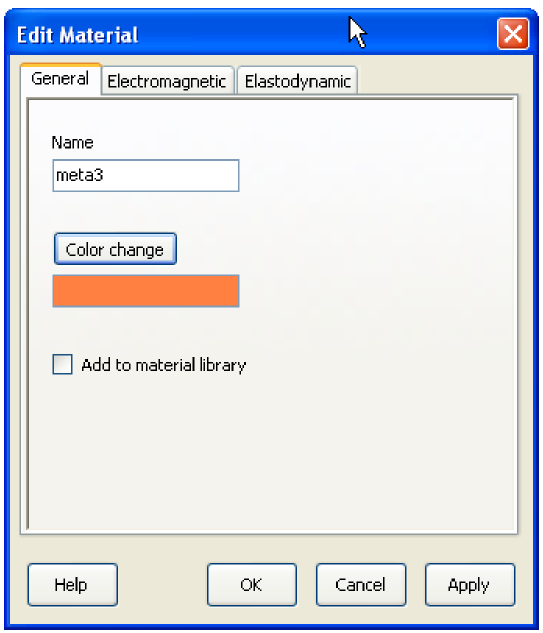
Here, we define metal1, metal2 and metal3.







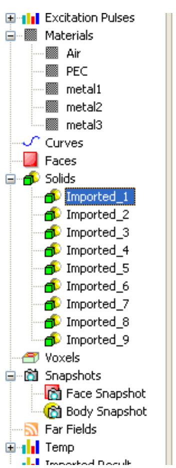


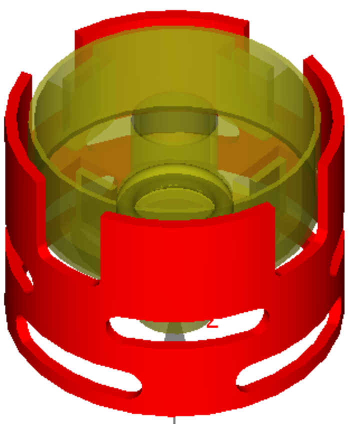


Assign different components by different materials

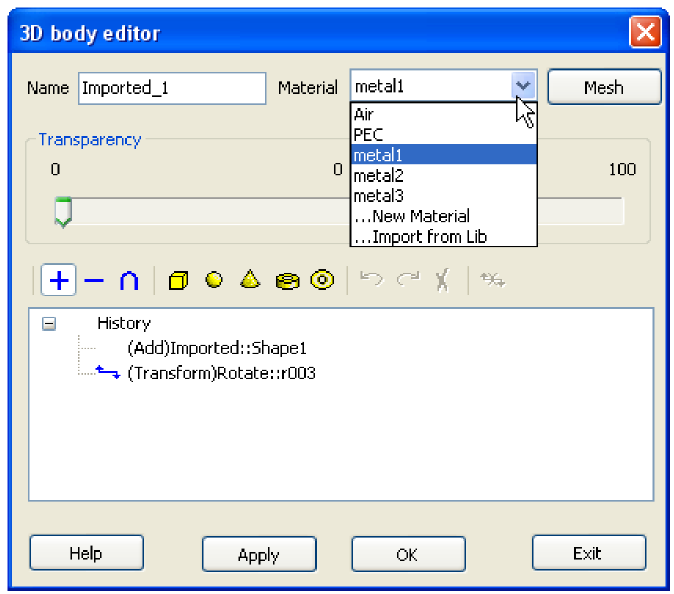
For example, assign component “Imported_1” by material metal1

double click treenode “Imported_1”

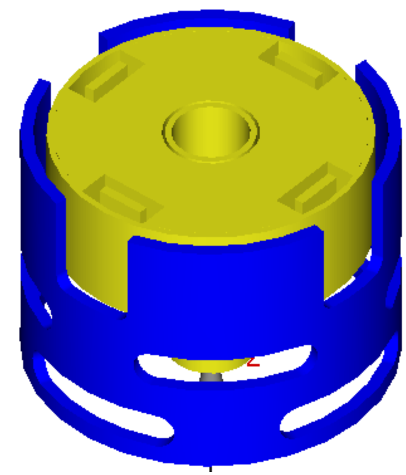




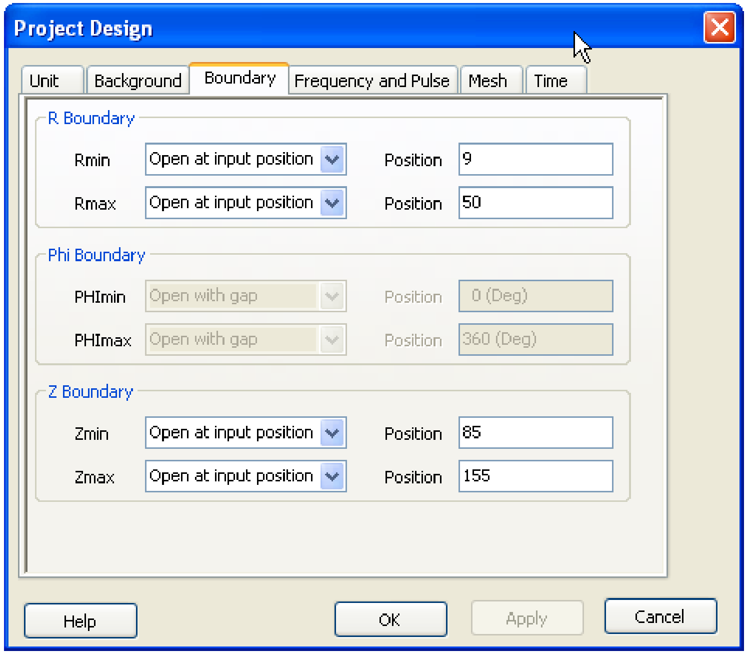
Modify material to metal1, then press “OK”



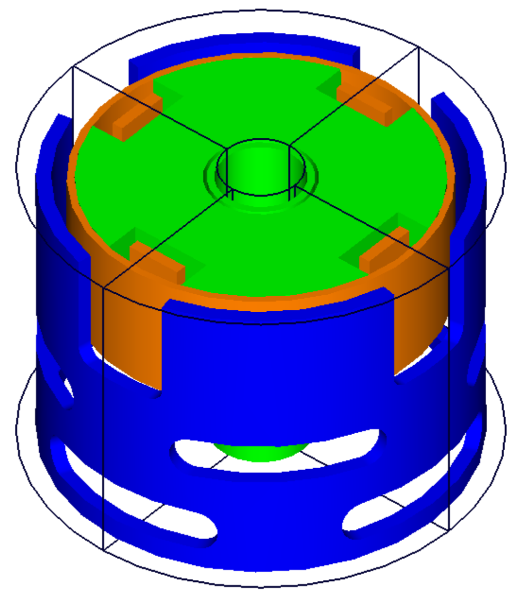
The result

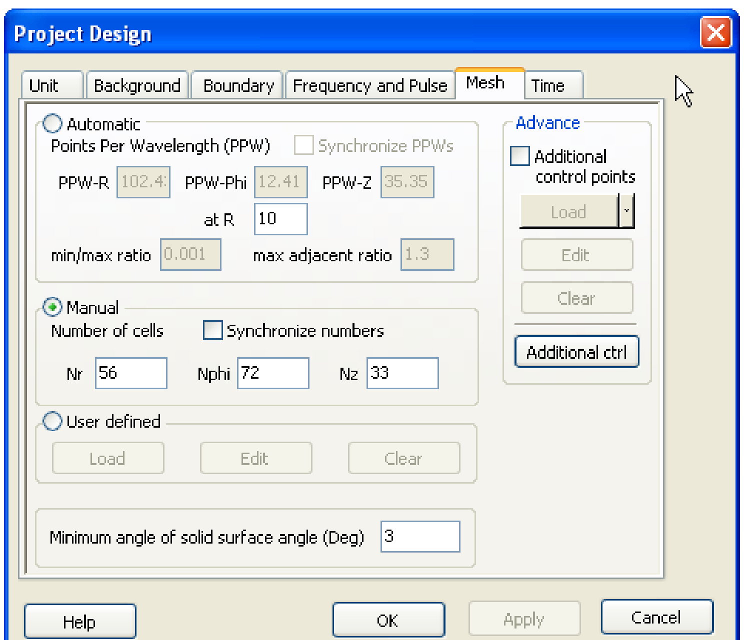


Set up cylindrical mesh control



Computation domain boundary and size





Mesh setup

After setting up cylindrical mesh control,
User can export or display cylindrical mesh by

Generate cylindrical mesh and export the data file

Begin to show the cylindrical mesh