

Voxel Tutorial

Wave Computation Tech.

Oct., 2017

Content

- Voxel definition
- Voxel editor
- Voxel displaying: 3D & 2D cross-section
- Voxel input data file format
- Voxel demo cases
 - Part I: EM solver
 - Transient response from a dielectric sphere
 - Transient response from multiple rectangular solids, including PEC solids
 - Simulation on a digital breast model
 - Part II: Cartesian EL solver
 - Transient response from a Box in 3 layers background, general isotropic material
 - Transient response from a Box in homogenous background, weak anisotropic material
 - Simulation on 2D Marmousi II model

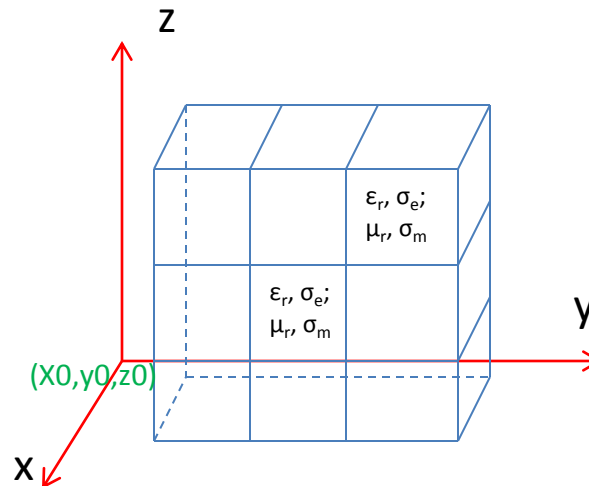
- Simulate 2D slice/3D volume generated from Wavenology mesher
 - Generate mesh files from Wavenology mesher
 - Use the model from mesh files
 - Demo: Simulate a 2D slice cut from STL model

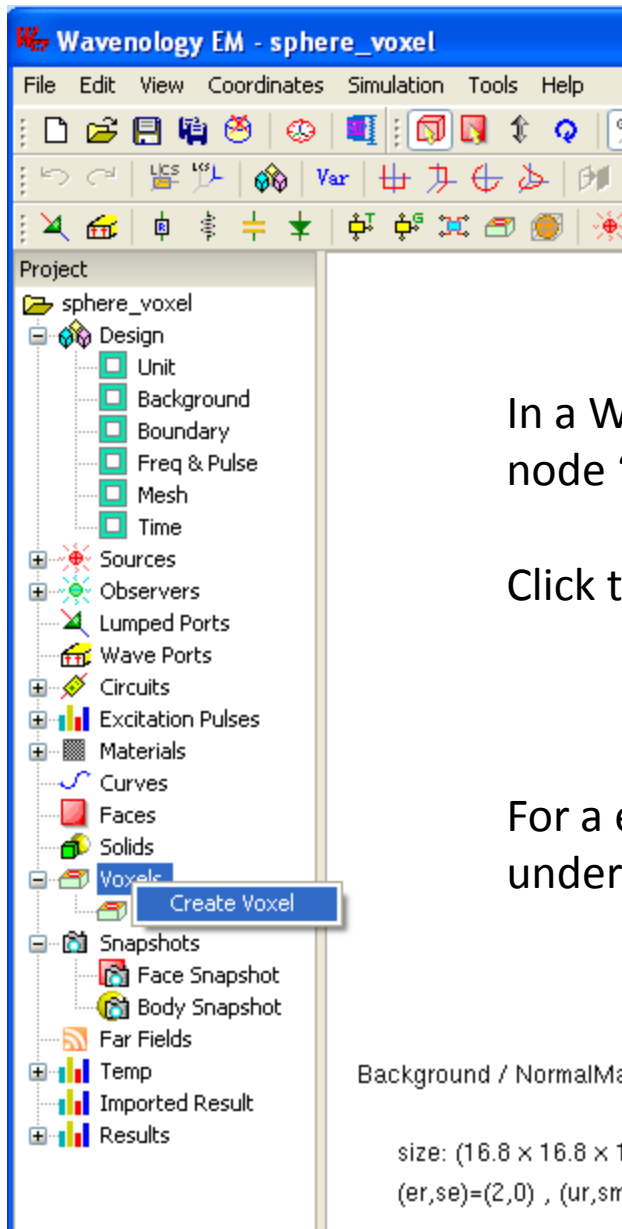
Voxel Definition

In WCT, the voxel is a rectangular volume. User **must** define

- 1) the volume start position,
- 2) uniform resolution in x, y and z direction
- 3) cells number in x, y and z direction
- 4) the material profile for each cell in the volume.

Note: Due to the simulation on voxel is an advance function of WCT, we will not cover the topics of the basic operations of WCT in this tutorial. Please refer to WCT manual and tutorial for these topics.





In a WCT project, right click mouse button on project tree node “Voxels”, a menu “Create Voxel” will be pop upped.

Click this menu, enter the *Voxel Editor*.

For a existing voxel in a project, double click the voxel item under the treenode “Voxels”, user can enter the *Voxel Editor*.

Voxel Editor

Voxel Name

Voxel volume start position

EM property page

EL property pages

Unit for Voxel size & resolution

Voxel resolution

General

Name Unit

Physical Size

Loaded Volume

Volume Start Position in computation domain (x,y,z) Resolution (dx,dy,dz)

Advance

Use the Whole Volume Use sub-volume

Corner 1 (x,y,z)

Corner 2 (x,y,z)

Electrical Parameters | Elastodynamic Parameters | Ela. Weak Aniso. Parameters

Relative Permittivity Bulk Load Mesh size

Elec. Conductivity (S/m) Bulk Load Mesh size

Relative Permeability Bulk Load Mesh size

Mag. Conductivity (W/m) Bulk Load Mesh size

Display

Data Type Color Map Transparency

Help OK Cancel

Set material profile as a constant value for whole voxel

Load material profile from data file

The number of cells in each direction. These values are read from data file.

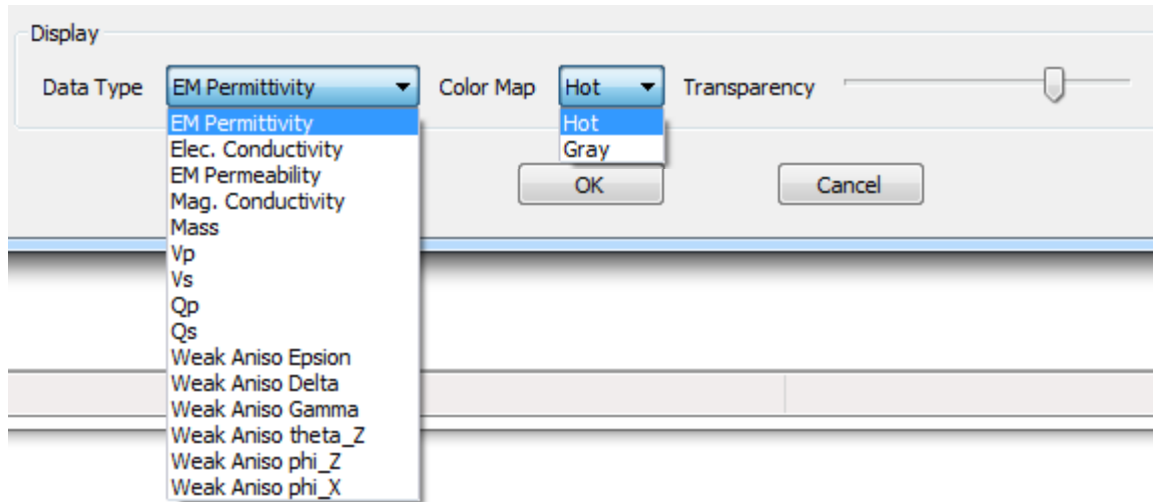
Elastic: Iso/visco-elastic parameters

Electrical Parameters	Elastodynamic Parameters	Ela. Weak Aniso. Parameters
Mass (kg/m ³)	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="1000"/>	<input type="text" value="Mesh Size"/>
Vp (m/s)	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="300"/>	<input type="text" value="Mesh Size"/>
Vs (m/s)	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="0"/>	<input type="text" value="Mesh Size"/>
Qp	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="-1"/>	<input type="text" value="Mesh Size"/>
Qs	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="-1"/>	<input type="text" value="Mesh Size"/>

Elastic: Weak anisotropic parameters

Electrical Parameters	Elastodynamic Parameters	Ela. Weak Aniso. Parameters
Epsilon	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="0.1"/>	<input type="text" value="Mesh Size"/>
Delta	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="-0.2"/>	<input type="text" value="Mesh Size"/>
Gamma	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="0.3"/>	<input type="text" value="Mesh Size"/>
ZX Axis Angles (Unit: degree)	[note: VTI-(0,-,-); HTI-(90, phi_Z,-)]	
theta_Z	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="0"/>	<input type="text" value="Mesh Size"/>
phi_Z	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="0"/>	<input type="text" value="Mesh Size"/>
theta_X	<input checked="" type="radio"/> Bulk	<input type="radio"/> Load
	<input type="text" value="90"/>	<input type="text" value="Mesh Size"/>

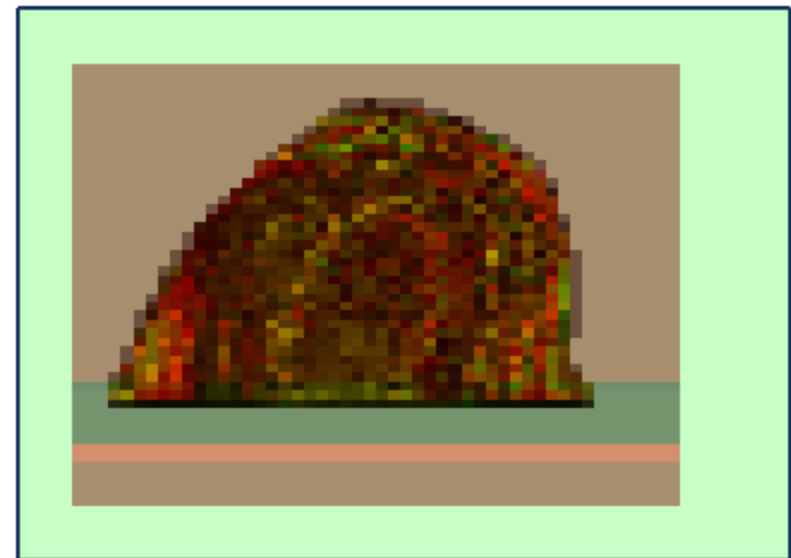
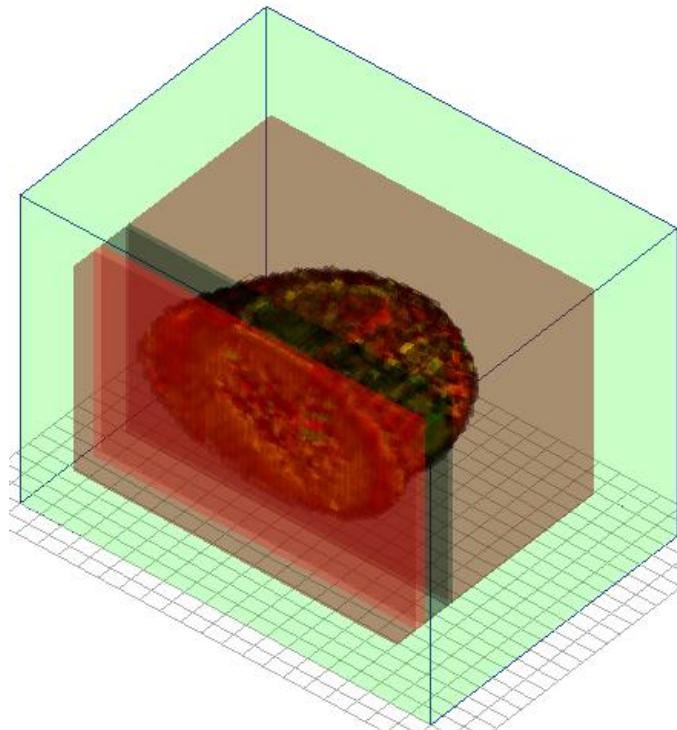
3D brief displaying control for GUI 3D canvas



Displaying of Voxel in WCT GUI

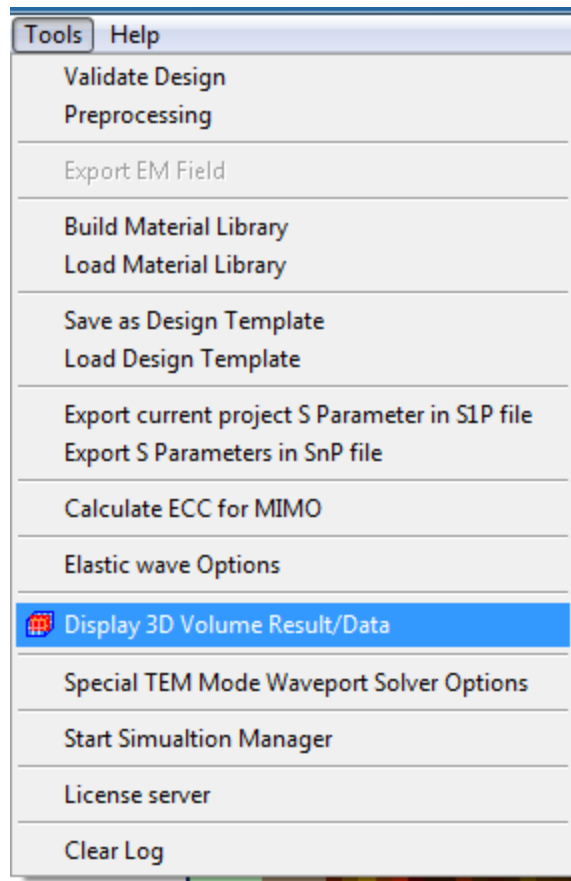
(1) 3D Display

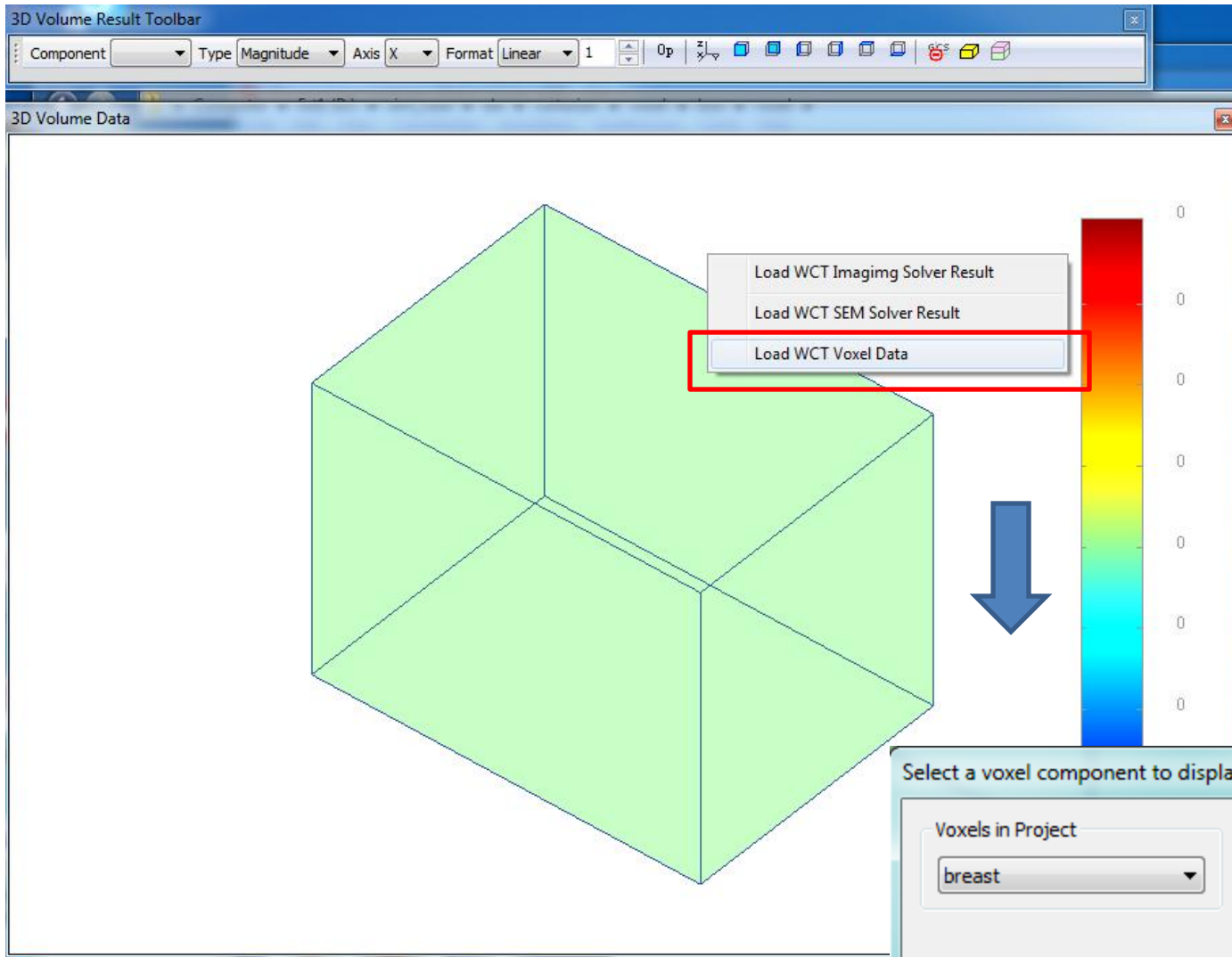
Following is an example of the 3D voxel brief displaying in GUI's 3D canvas. This voxel is a digital breast model from [UWCEM Numerical Breast Phantoms Repository](#)



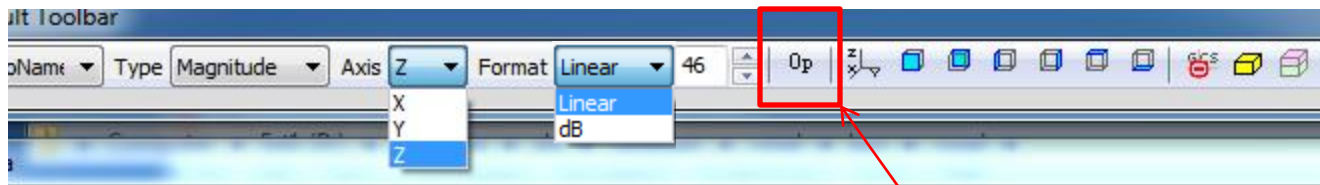
(2) 2D High Resolution View

User can use “Display 3D Volume Result/Data” functionality to show voxel in a high resolution 2D cross-section mode.





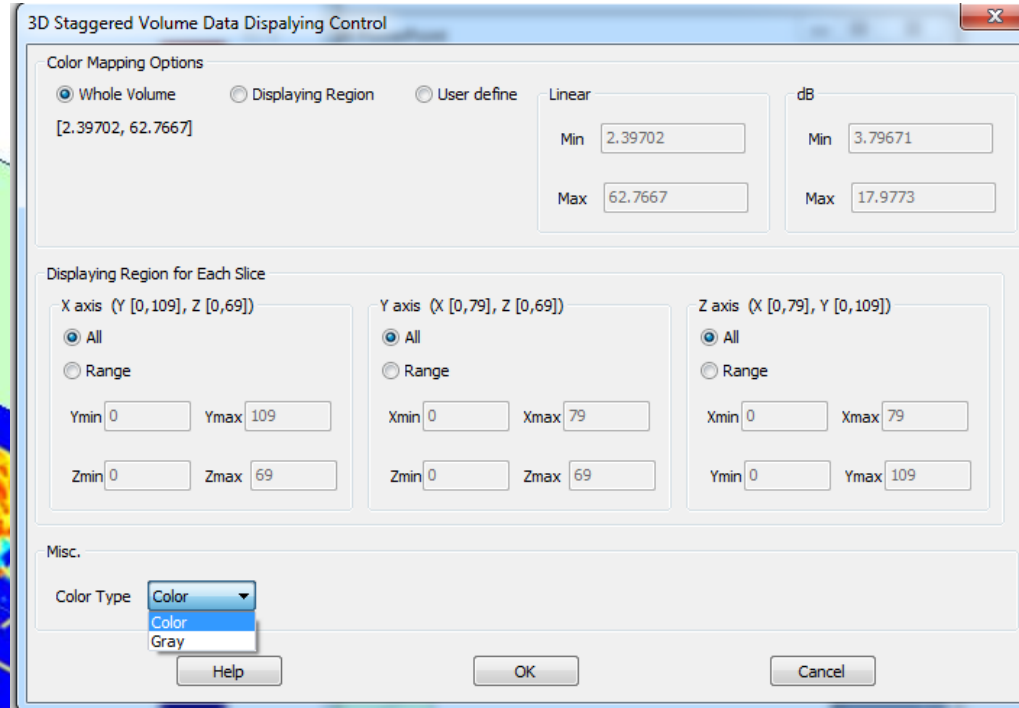
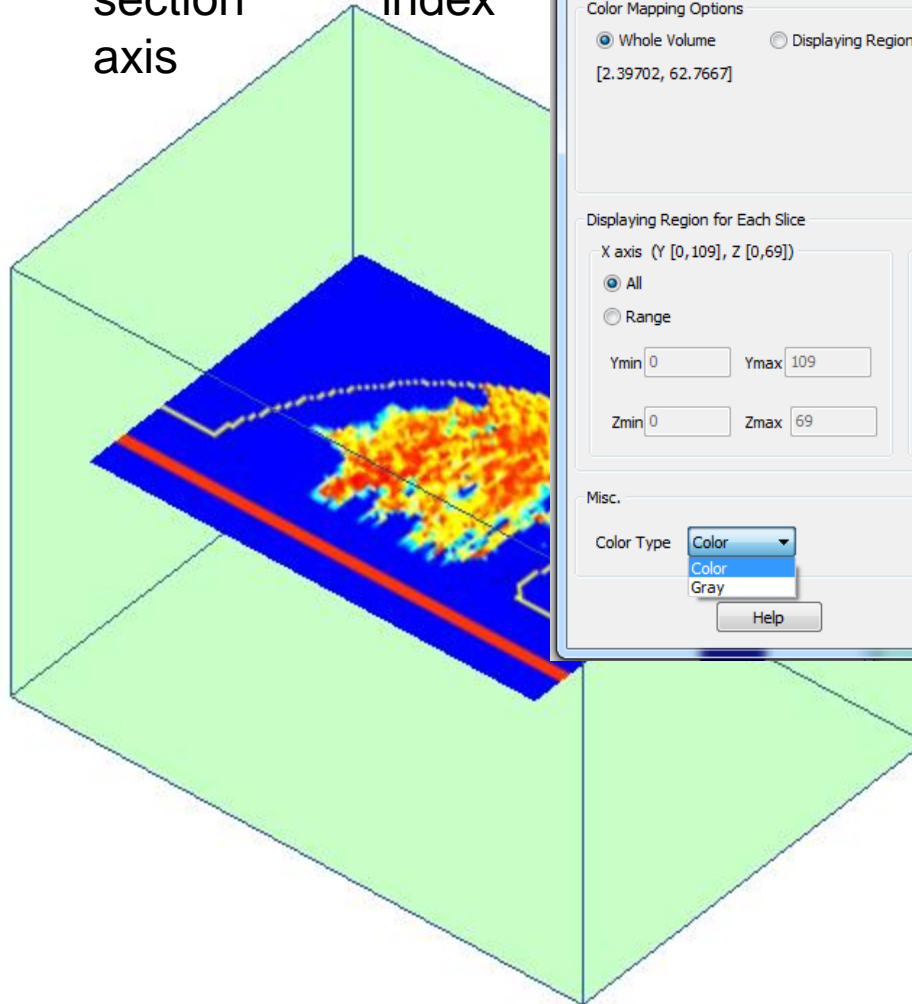
User need to determine which voxel and what kind of parameter will be shown



Displaying control

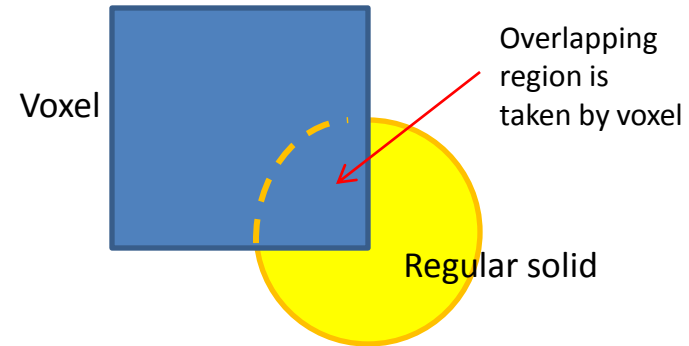
Cross-section axis

Frame index

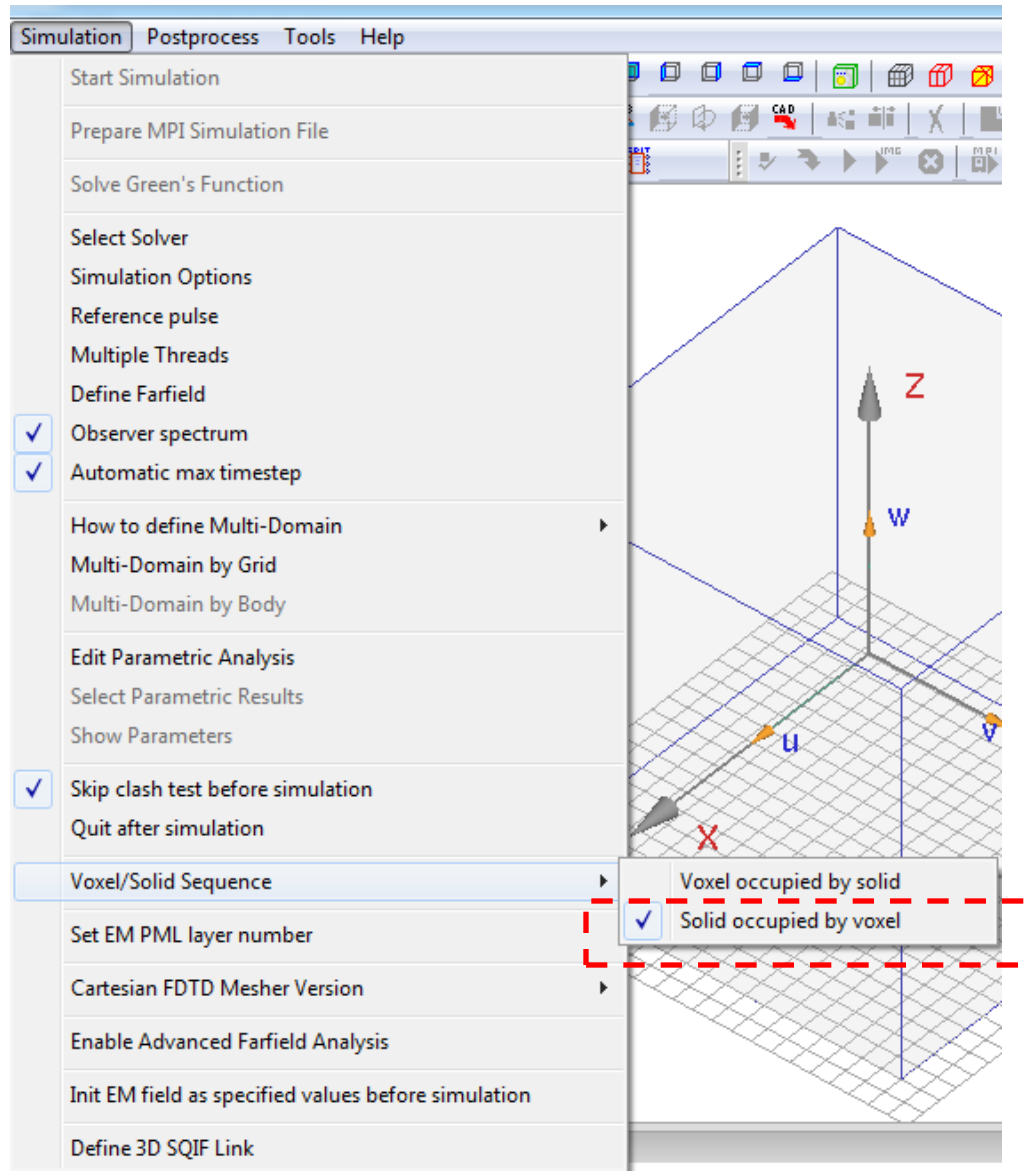


Other Information

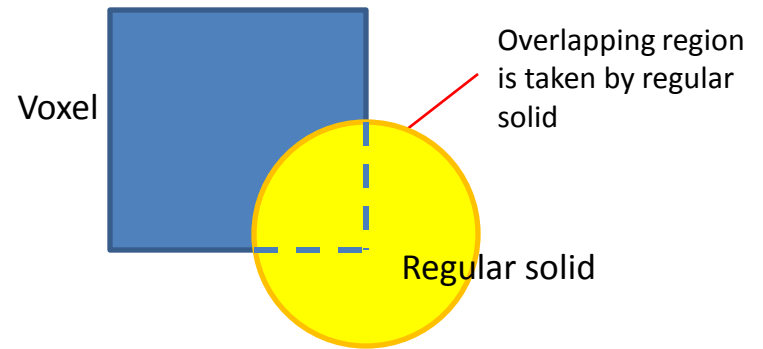
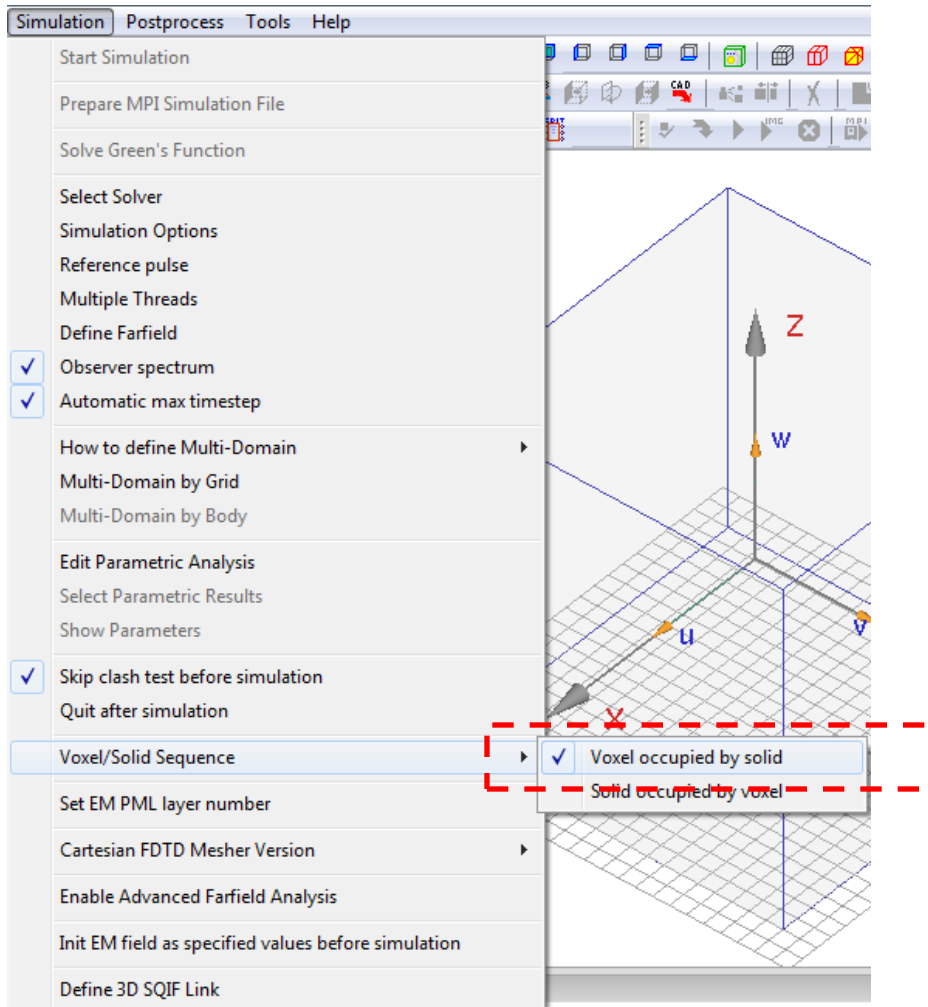
- 1) There can be multiple voxels in a project.
- 2) If a voxel has overlapping region with regular solids,
 - 2.1) By default, WCT will use the material profile from voxel.



This default option is set through menu:
Simulation->Voxel/Solid Sequence
as next page



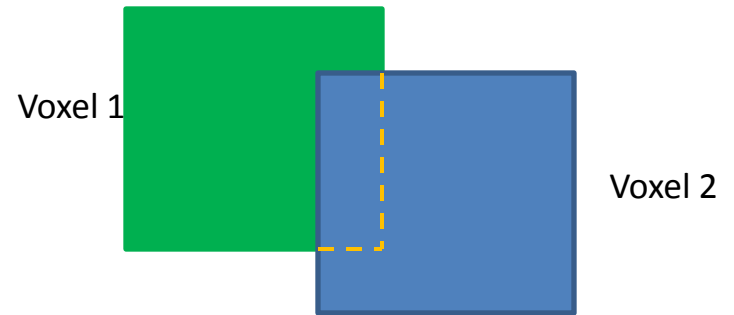
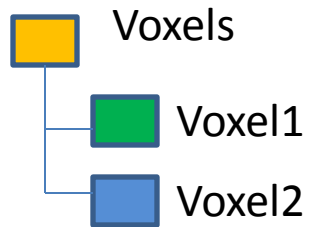
2.2) But user can switch to use the material profile from object. In this situation, it is equivalent to place solid in the voxel region.



Note: this voxel sequence modification capability is enabled in **EM** solver only.

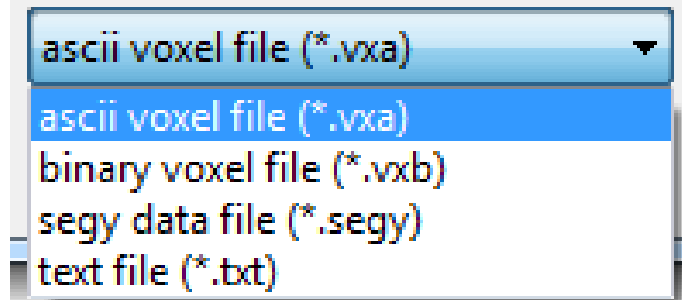
For current **EL** solver, this option will be skipped.

3) If a voxel has overlapping region with another voxel , WCT will use the material profile from the later voxel in the tree.



Voxel Input Data File Format

- 1) The data file for a voxel parameter can be
 - 1) ASCII text file, suffix as “txt”
 - 2) WCT internal Voxel data file, ASCII format, suffix as “vxa”
 - 3) WCT internal Voxel data file, binary format, suffix as “vxb”
 - 4) SEG Y format, suffix as “seg y”



2) For the ASCII text file (txt), the format is:

Line index	Meaning
1	Cell number in x direction (integer)
2	Cell number in y direction (integer)
3	Cell number in z direction (integer)
4	Parameter for cell (0,0,0)
.	Parameter for cell (0,0,1)
.
.	Parameter for cell (0,1,0)
.	Parameter for cell (0,1,1)
.

The *for-loop* to read parameter for each cell is:

```
for x;  
  for y;  
    for z;  
      .....  
    end z;  
  end y;  
end x;
```

3) File examples (from demo case (2))

A data file for ϵ_r

```
4
1
2
-----
2.0
1.0
3.0
1.0
1.0
1.0
1.0
1.0
```

4 x cells, 1 y cell, 2 z cells
Totally 8 cells

Following 8 floating
numbers is the
parameter for each cell

A data file for σ_e

```
4
1
2
-----
0.0
1e20
10.0
0.0
1e20
1e20
1e20
1e20
```

User can insert
empty line
between two lines

Note: for PEC and PMC, ideally they should be a infinite number. In WCT voxel data, user need use a high conductivity value to make an approximation. Here, we set σ_e as $1e^{20}$ to represent PEC material. But this value should be less than $1e^{30}$.

4) Wavenology ASCII voxel file (vxa), the format is:

parts	Meaning
Line 1	comment
Line 2	version
Next 3 lines	nx, ny, nz
Following N lines	$N=nx*ny*nz$; one value for one cell

The *data sequence* in the data block

```
for x;  
  for y;  
    for z;  
      .....  
    end z;  
  end y;  
end x;
```

5) Wavenology binary voxel file (vxb), the format is:

parts	Meaning
80 char	comment
int32	Major version
int32	Minor version
int32	nx
int32	ny
int32	nz
float32[N]	$N=nx*ny*nz$

The **data sequence** in the data block “float[N]”

```
for x;  
  for y;  
    for z;  
      .....  
    end z;  
  end y;  
end x;
```

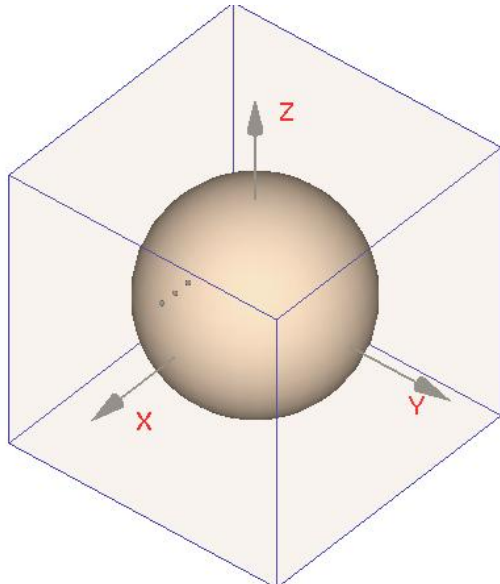
Voxel Demo Cases

Part I: EM Solver

- 1) Transient response from a dielectric sphere
- 2) Transient response from multiple rectangular solids, including PEC solids
- 3) Simulation on a digital breast model

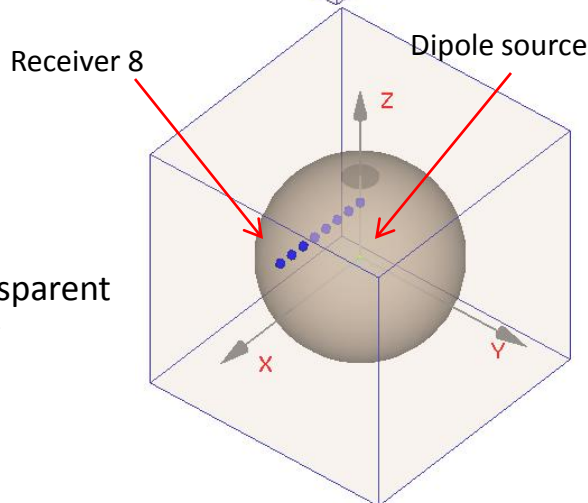
Case (1) Sphere

This case simulates a dielectric sphere, defined by voxel, in homogenous background.



The left Fig. is WCT tutorial case “NormalSphere”, a dielectric sphere in homogenous background.

In this voxel case, we will replace the rectangular volume enclosing sphere by a voxel, and compare the transient response on receiver 8 with analytical solution.



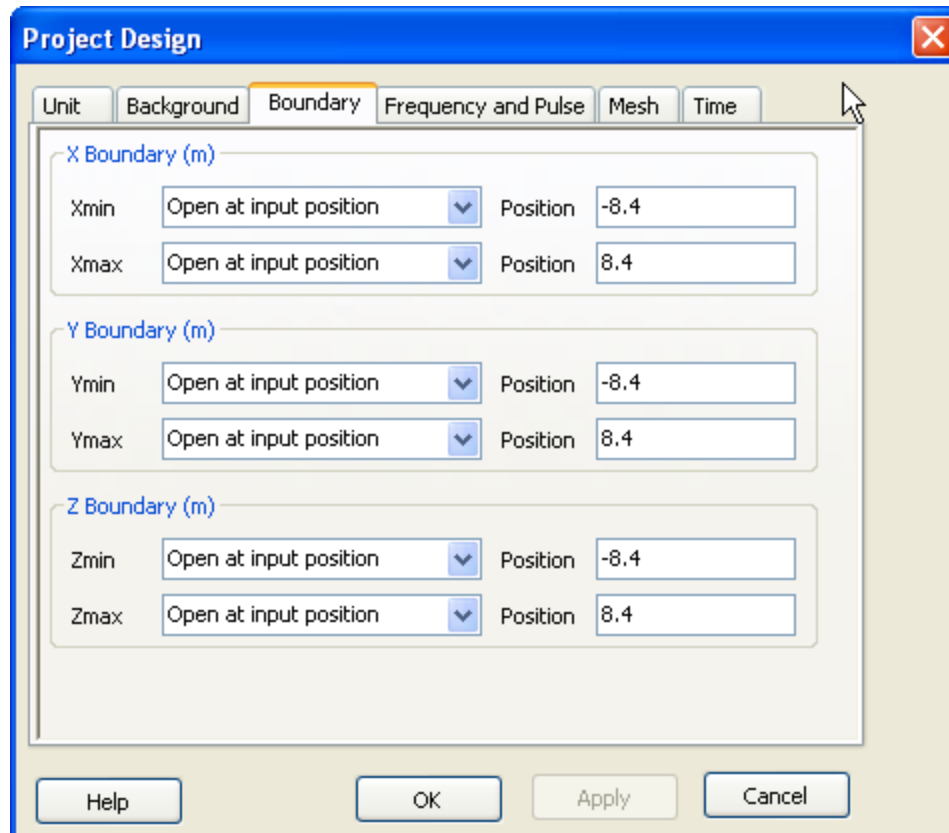
Background: $\epsilon_r=2$, $\sigma_e=0$; $\mu_r=2$, $\sigma_m=0$

Sphere: $\epsilon_r=4$, $\sigma_e=0$; $\mu_r=1$, $\sigma_m=0$; $r=6$ m

Excitation: Ideal E dipole with 45.4K-21.6M Hz BHW pulse, x polarization.

Steps:

- 1) User can open the tutorial case “NormalSphere”, **SaveAs** a new case “SphereVoxel”
- 2) Change project boundary to “OPEN at input position”, as following Fig.



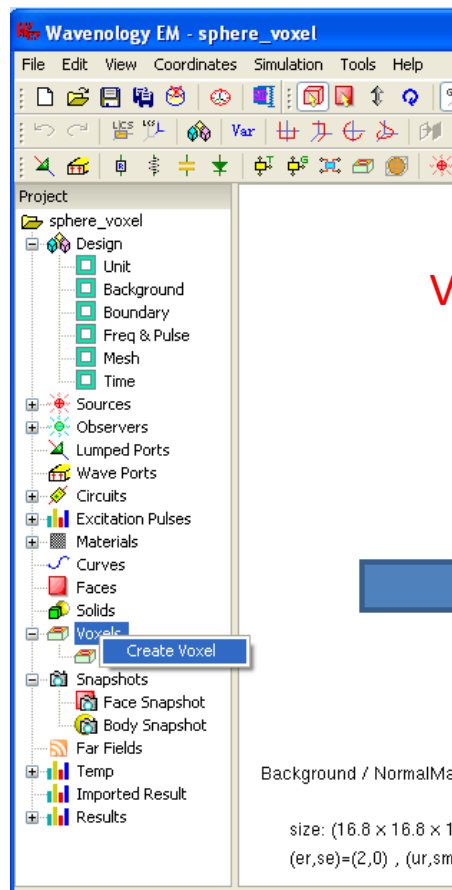
- 3) Delete the sphere.
- 4) Save the project.
- 5) Generate voxel data file for ϵ_r & μ_r .

In demo cases folder: “\sphere

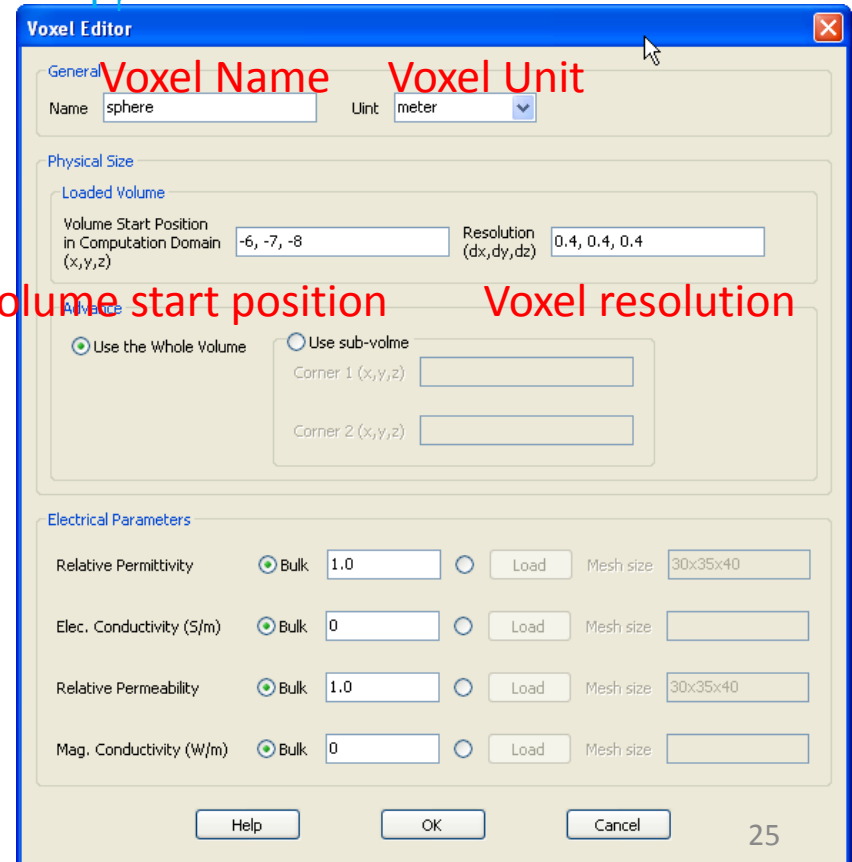
\Sphere_voxel_resolution_0.4_auto_mesh\matlab\”, there is a matlab example code to generate a voxel volume from (-6,-7,-8) to (6,7,8) m with a resolution 0.4 m at x, y and z direction.

The data file for ϵ_r is “eps1.txt”, The data file for μ_r is “mu1.txt”.

- 6) Create a voxel.



Voxel volume start position Voxel resolution



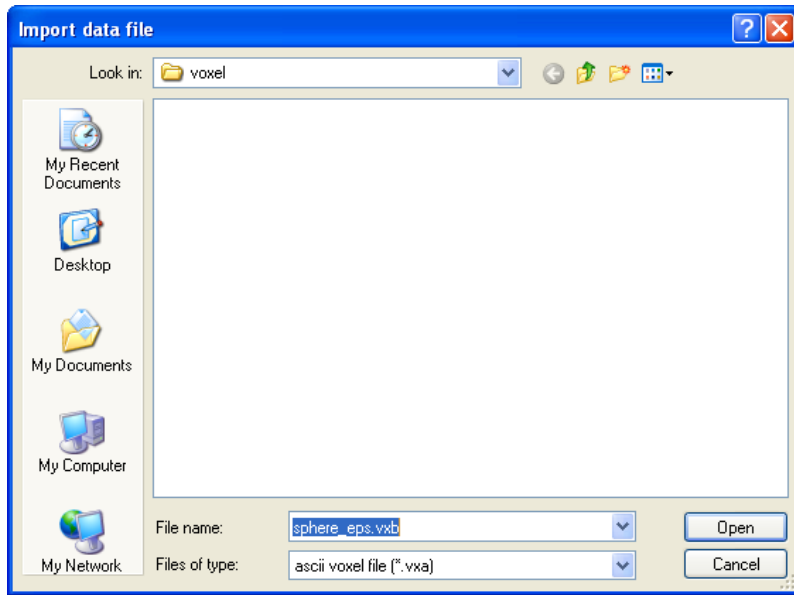
Switch “Relative Permittivity” & “Relative Permeability” to **Load**

The image shows a software dialog box titled "Voxel Editor" with a blue title bar and a close button (X) in the top right corner. The dialog is divided into several sections:

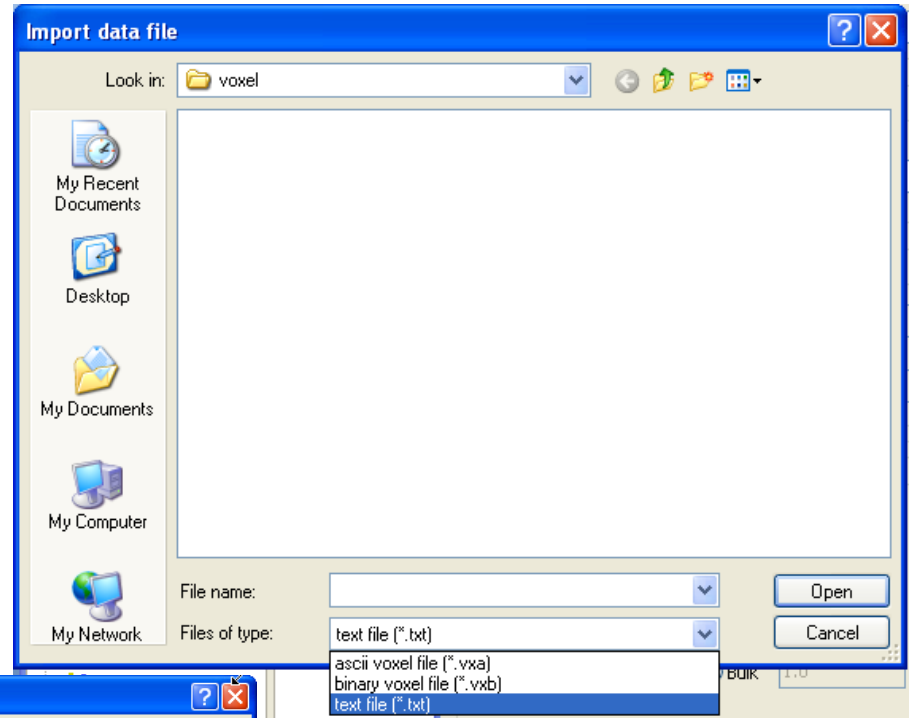
- General:** Name: "sphere", Unit: "meter".
- Physical Size:**
 - Loaded Volume:** Volume Start Position in Computation Domain (x,y,z): "-6, -7, -8", Resolution (dx,dy,dz): "0.4, 0.4, 0.4".
 - Advance:** "Use the Whole Volume" is selected with a radio button. "Use sub-volume" is unselected. Below it are two empty input fields for "Corner 1 (x,y,z)" and "Corner 2 (x,y,z)".
- Electrical Parameters:** This section is highlighted with a blue arrow pointing to the "Load" buttons. It contains four rows:
 - Relative Permittivity:** "Bulk" (1.0) is unselected, "Load" is selected.
 - Elec. Conductivity (S/m):** "Bulk" (0) is selected, "Load" is unselected.
 - Relative Permeability:** "Bulk" (1.0) is unselected, "Load" is selected.
 - Mag. Conductivity (W/m):** "Bulk" (0) is selected, "Load" is unselected.Each row also has a "Mesh size" input field, with "30x35x40" entered for the first and third rows.

At the bottom of the dialog are three buttons: "Help", "OK", and "Cancel".

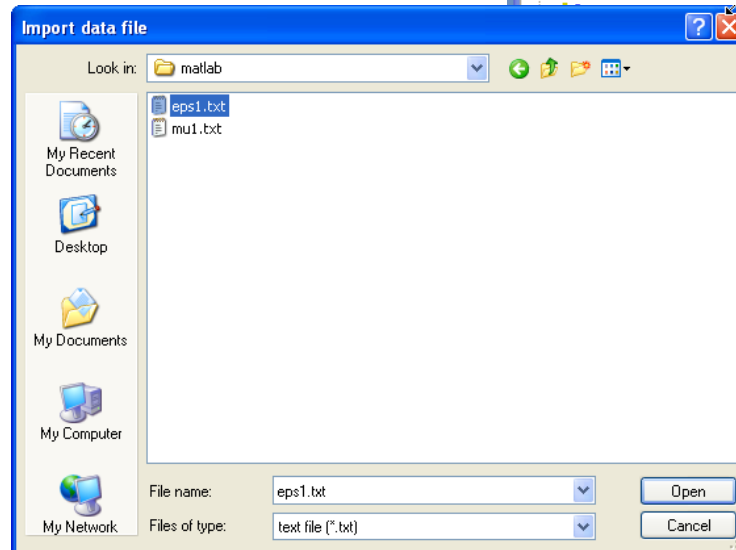
Click **Load** button to load file “eps1.txt” for Relative Permittivity
“mu1.txt” for Relative Permeability, separately.



(1)



(2)



(3)

Voxel Editor

General

Name: Unit:

Physical Size

Loaded Volume

Volume Start Position in Computation Domain (x,y,z): Resolution (dx,dy,dz):

Advance

Use the Whole Volume Use sub-volume

Corner 1 (x,y,z):

Corner 2 (x,y,z):

Electrical Parameters

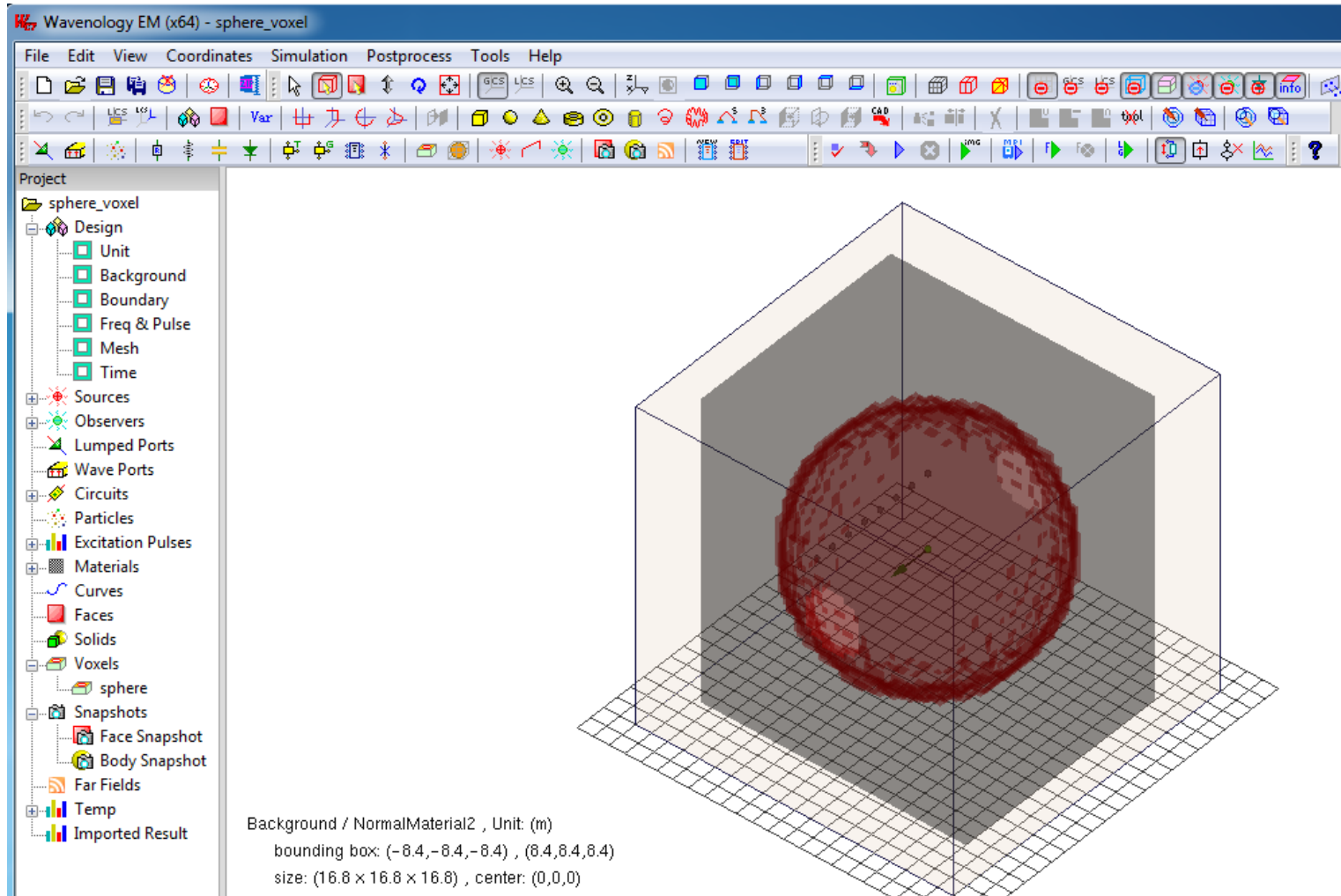
Relative Permittivity	<input type="radio"/> Bulk	<input type="text" value="1.0"/>	<input checked="" type="radio"/> Load	Mech size	<input type="text" value="30x35x40"/>
Elec. Conductivity (S/m)	<input checked="" type="radio"/> Bulk	<input type="text" value="0"/>	<input type="radio"/> Load	Mech size	<input type="text"/>
Relative Permeability	<input type="radio"/> Bulk	<input type="text" value="1.0"/>	<input checked="" type="radio"/> Load	Mech size	<input type="text" value="30x35x40"/>
Mag. Conductivity (W/m)	<input checked="" type="radio"/> Bulk	<input type="text" value="0"/>	<input type="radio"/> Load	Mech size	<input type="text"/>

Buttons: Help, OK, Cancel

WCT will pre-read part of file to verify the data.

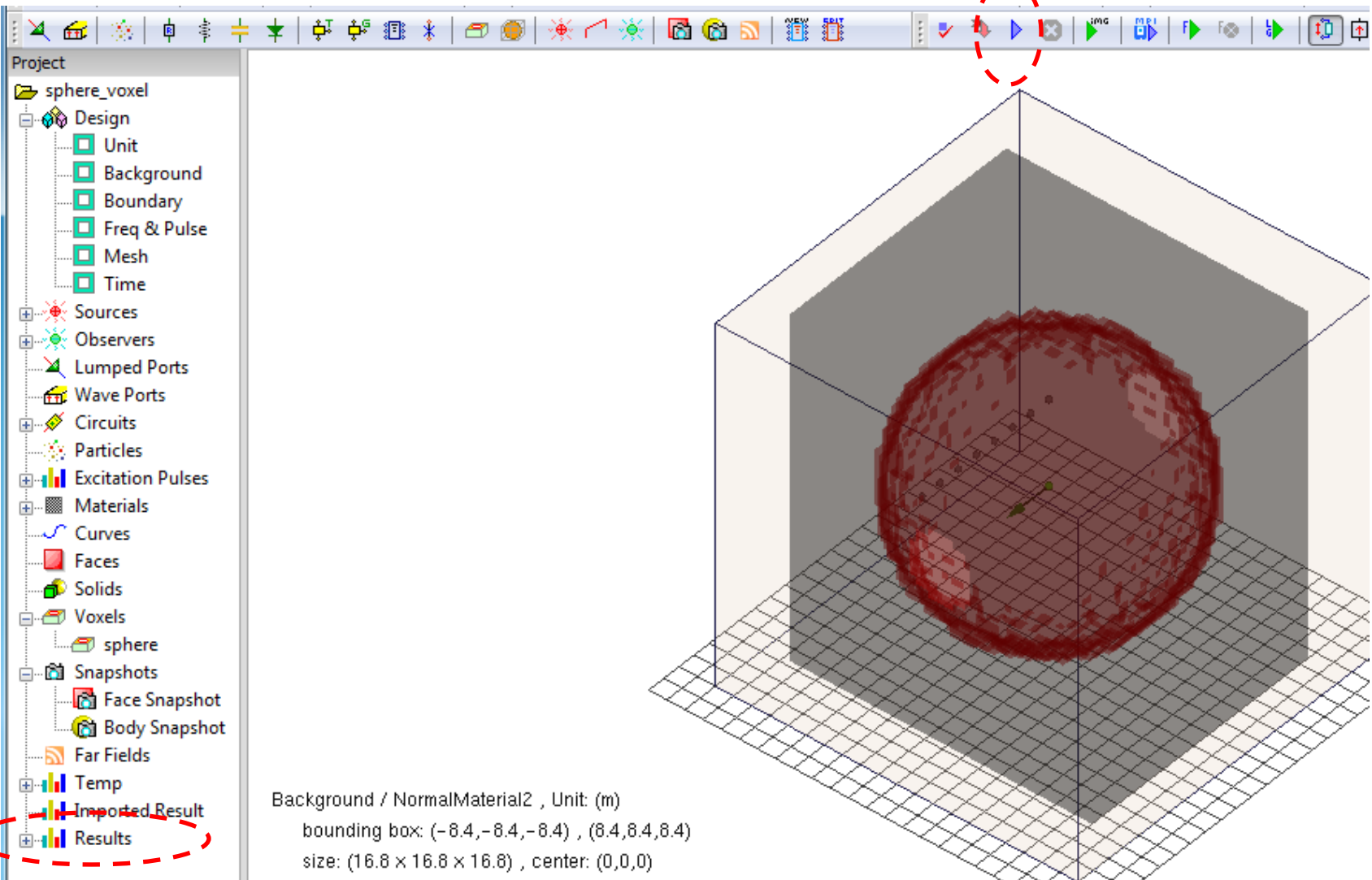
The mesh information is shown here.

7) Press **OK** button to finish voxel creation. Then the voxel is shown in 3D canvas.



8) Simulate the case and get result.

Simulate

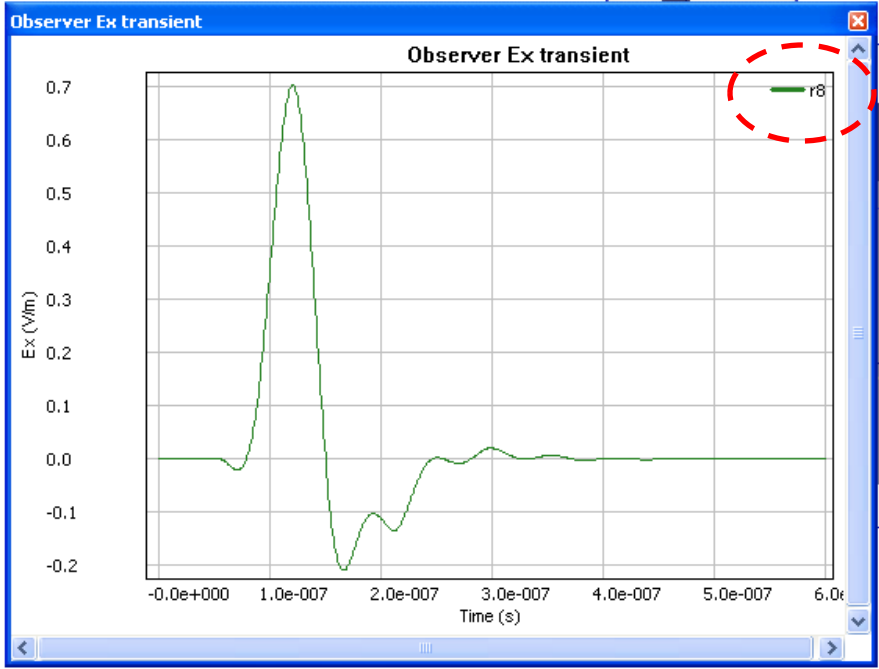
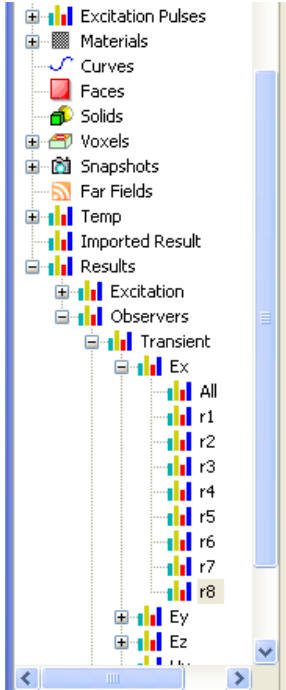
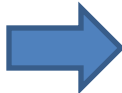
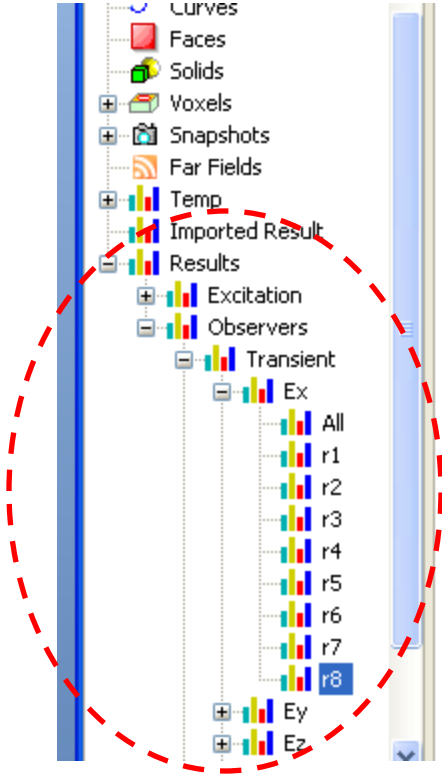


The screenshot displays a simulation software interface. On the left, the 'Project' panel lists various components, with 'Results' circled in red. The main workspace shows a 3D model of a sphere with a red mesh, centered within a gray bounding box. The 'Simulate' button in the top toolbar is also circled in red. Below the 3D view, the following text is displayed:

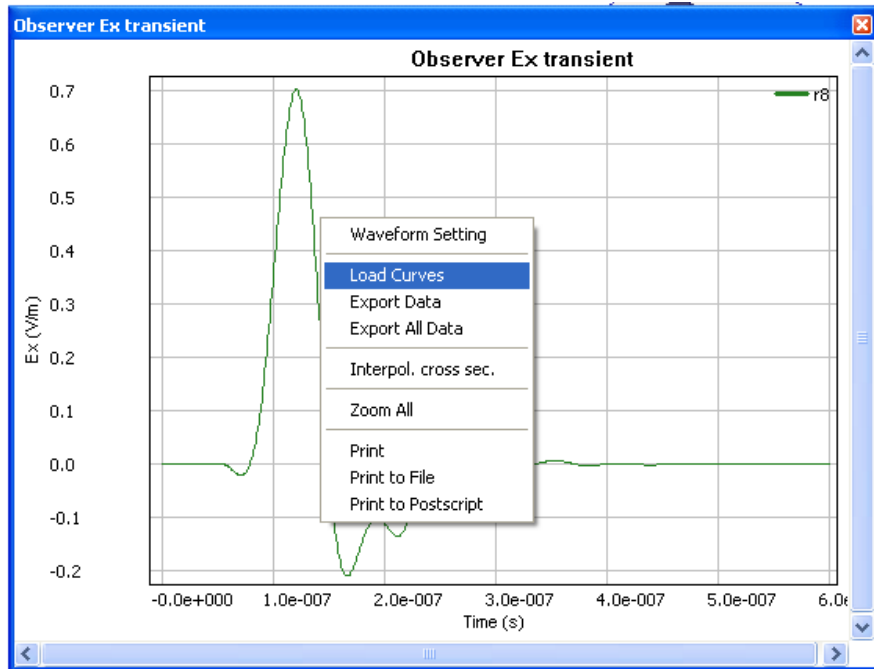
Background / NormalMaterial2 , Unit: (m)
bounding box: (-8.4,-8.4,-8.4) , (8.4,8.4,8.4)
size: (16.8 x 16.8 x 16.8) , center: (0,0,0)

result

9) Compare with Analytical solution

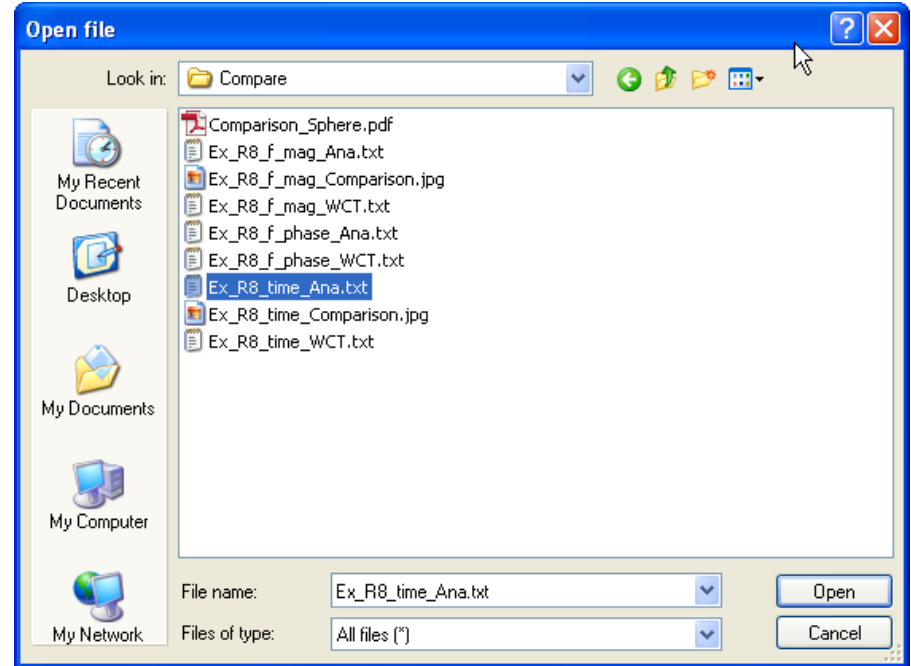


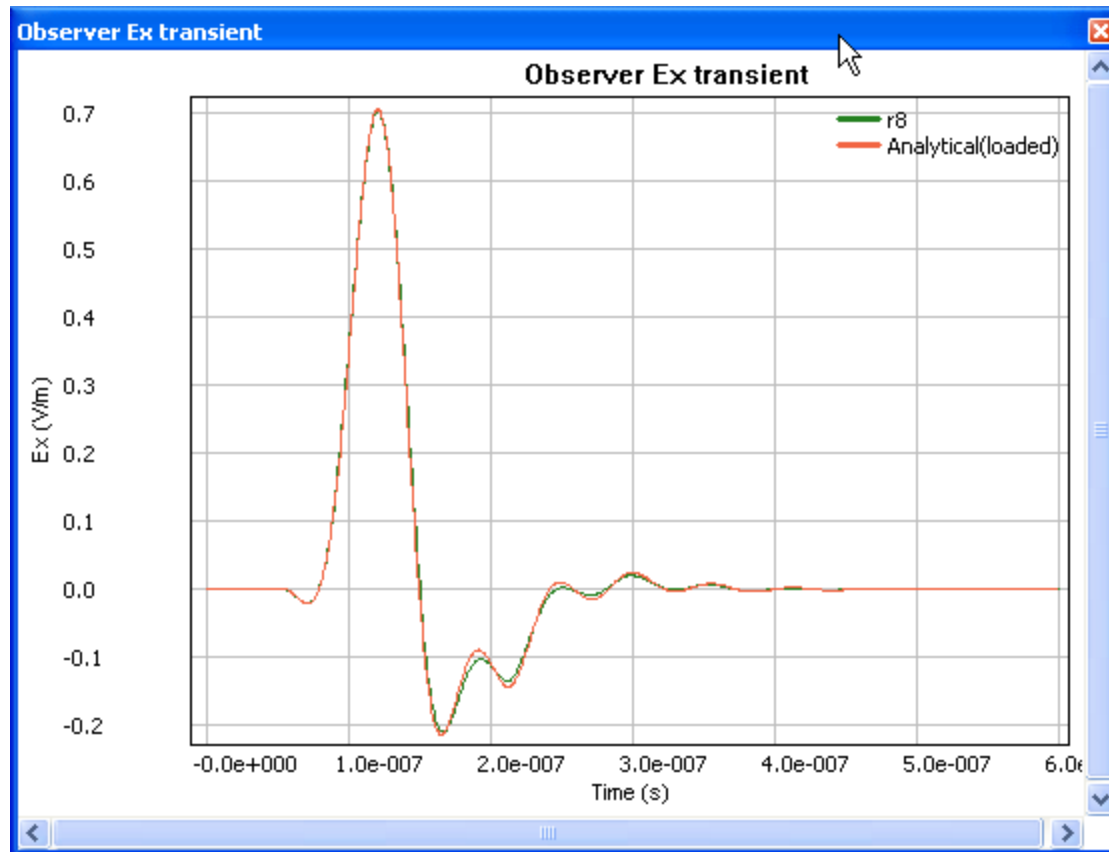
On 2D canvas, **right** click the mouse button, get a menu. Click **Load Curves** item to load analytical solution.



The analytical solution is under the folder:

`\sphere\NormalSphere\Compare\`



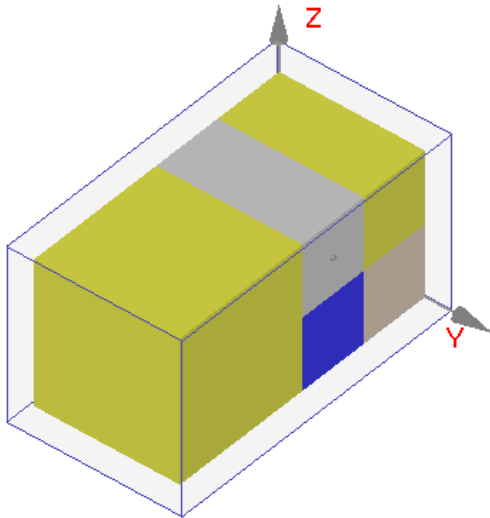


As can be seen, the simulation result is very close to the analytical solution.

The difference comes from the stair casing in discretizing sphere to coarse cubic cell.

Case 2: Multiple Rectangular Solids

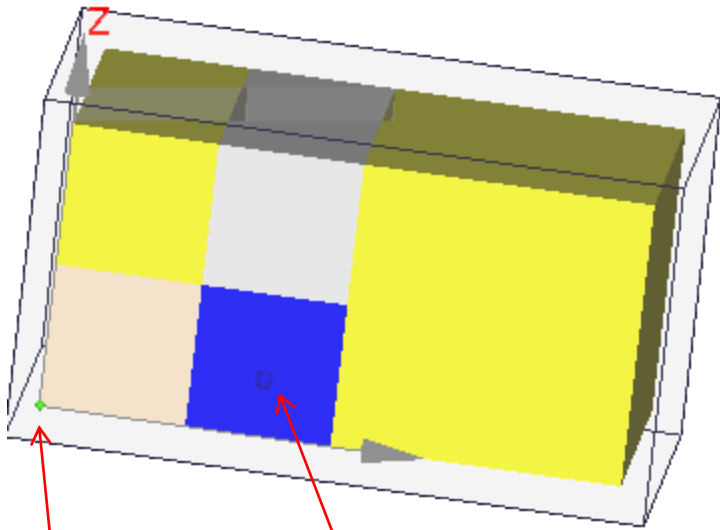
This case demonstrates how to simulate PEC material.



The left Fig. is six solids in homogenous air background, three of them (yellow) are PEC solids

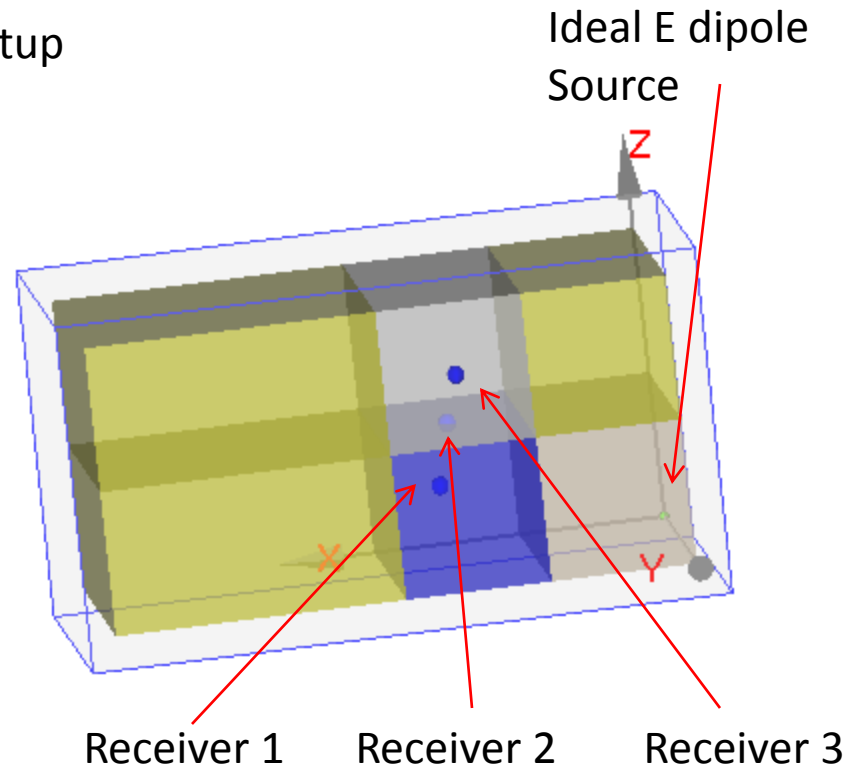
In this voxel case, we will use a voxel to represent these six solids. Because there is not analytical solution for this case. We will compare the result to which comes from six solids case.

Source & Receiver Setup



Ideal E dipole
Source

Receiver 1



Receiver 1

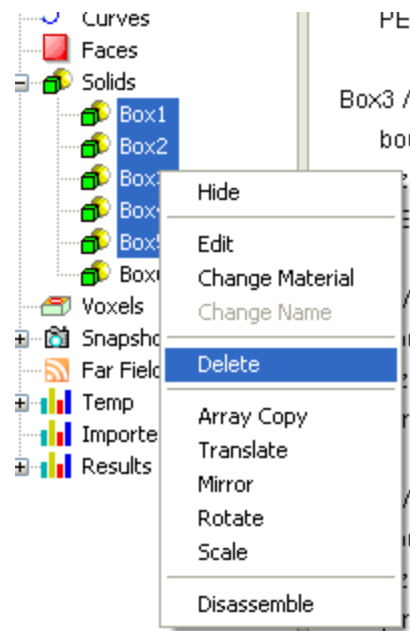
Receiver 2

Receiver 3

Ideal E dipole
Source

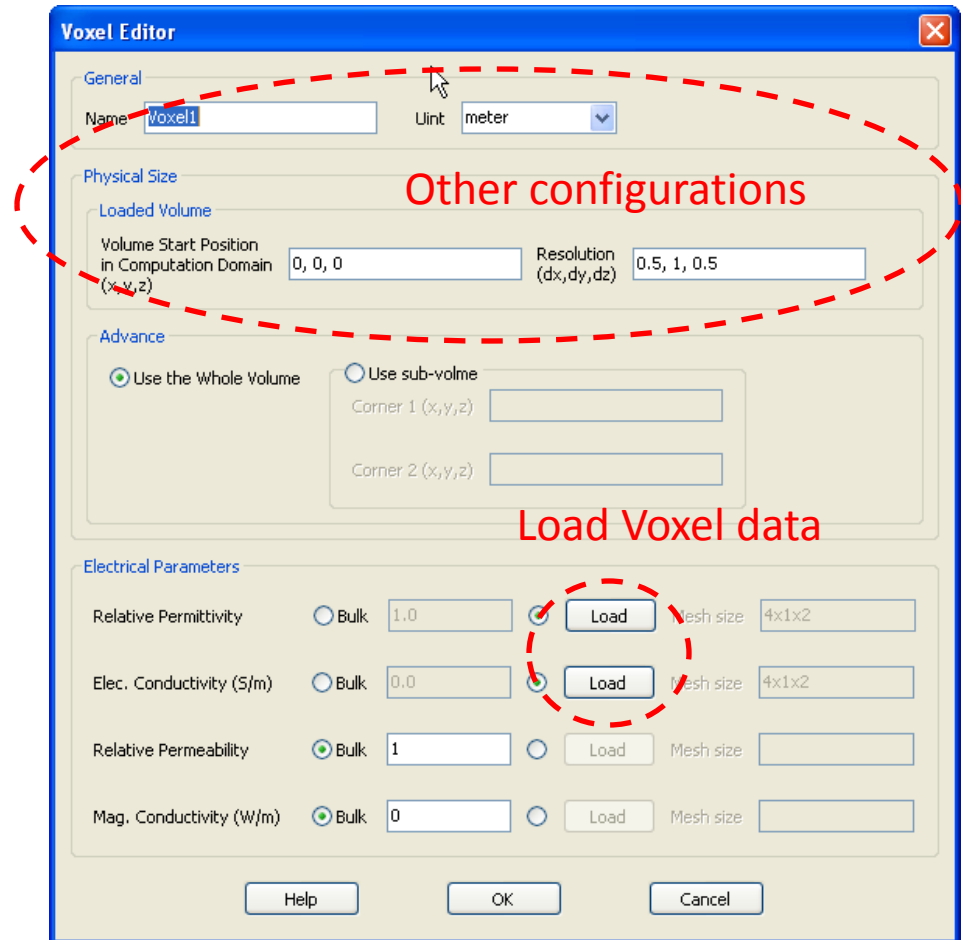
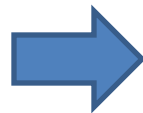
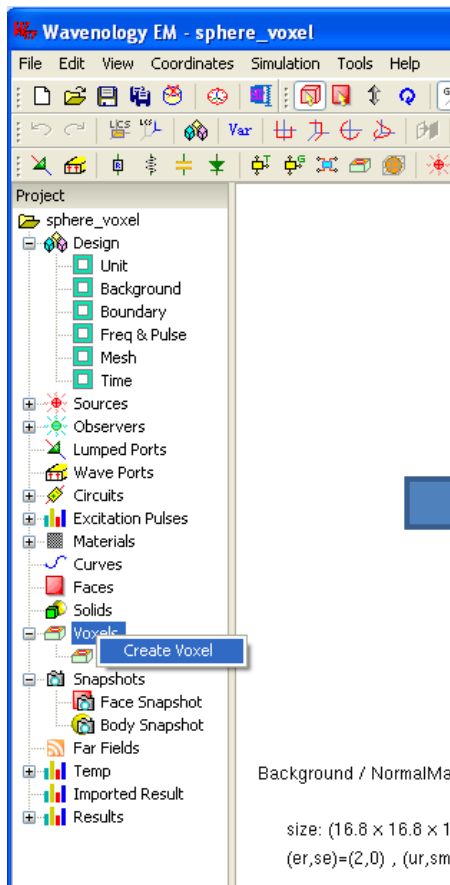
Steps:

- 1) Open “\ Boxes\Normal\ Normal.wnt, a project same as the Fig at page 22 can be viewed, **SaveAs** a new case “Voxel”
- 2) Because all B.C. have been set as “OPEN at input position”, here, delete all solids and save

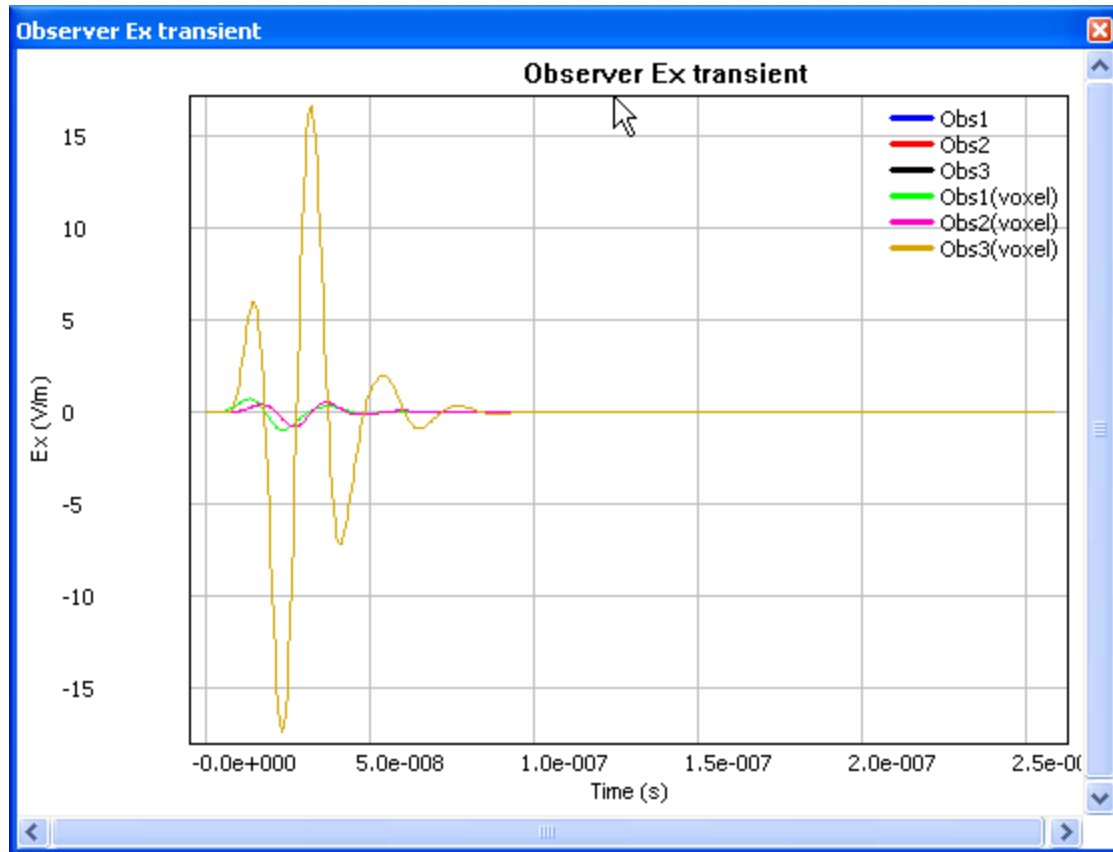


3) Define voxel data files. Because the original solids in this case is rectangular solid, the voxel files is very simple. User can use notepad to create two files as shown in page 9. The resolution of voxel is (0.5, 1, 0.5) m.

4) Define voxel in project.



8) Simulate the case and compare result.



As can be seen, the results from voxel simulation match that from the simulation with six solids

Case 3: Simulation on A Digital Breast Model

The digital breast model is obtained from:

UWCEM Numerical Breast Phantoms Repository, <http://uwcem.ece.wisc.edu/MRI/database/>.

The manual of the database is “Database of 3D Grid-Based Numerical Breast Phantoms for use in Computational Electromagnetics Simulations:

<http://uwcem.ece.wisc.edu/MRI/database/InstructionManual.pdf>.

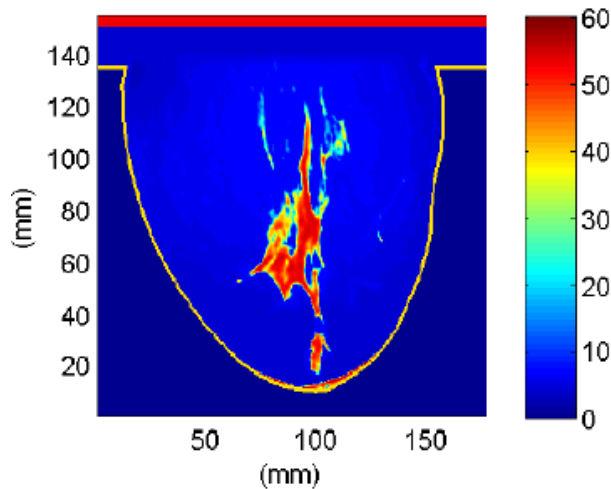
There are four models in this repository, corresponding to four categories of breast (the American College of Radiology categories).

In this case, we use model “ACR CLASS 1 - Mostly Fatty, Phantom 1”. The original model is MRI contrast data, we use “MRI contrast-EM profile conversion” method to obtain the ϵ_r & σ_e . Please refer to following papers to get the conversion method:

- 1) M. Lazebnik, S. C. Hagne, “A large-scale study of the ultrawide band microwave dielectric properties of normal breast tissue obtained from reduction surgeries,” *Phys. Med. Biol.*, vol. 52, pp. 2637-2656, 2007.
- 2) M. Lazebnik, S. C. Hagness, “A large-scale study of the ultrawideband microwave dielectric properties of normal, benign, and malignant breast tissues obtained from cancer surgeries,” *Phys. Med. Biol.*, vol. 52, pp. 6093-6115, Oct. 2007.

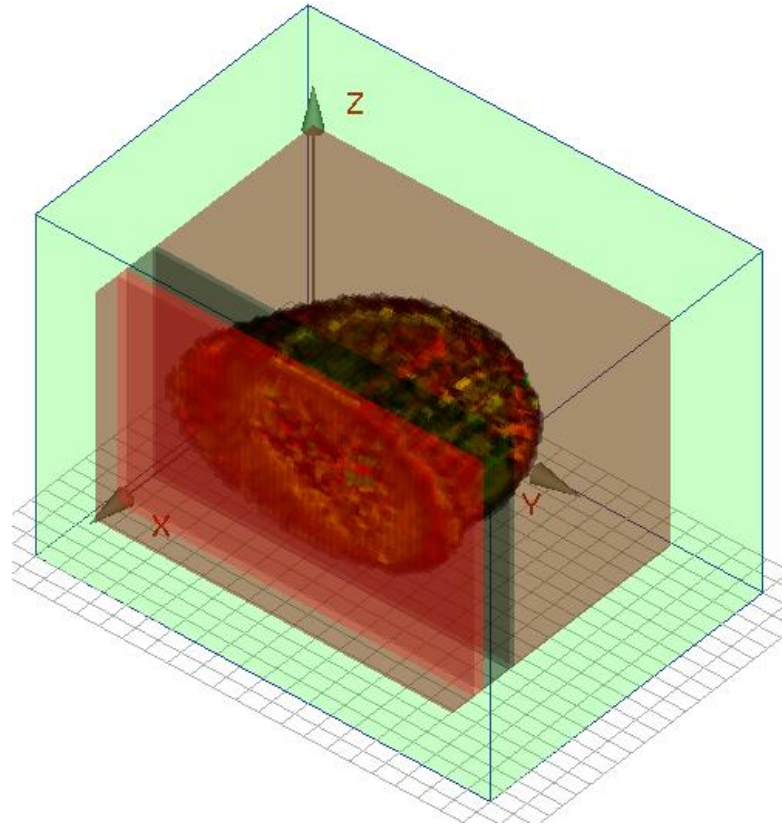
Due to the high resolution of original model, and the for-loop is different from WCT voxel definition. We down sample this model to a resolution of $1.5 \times 1.5 \times 1.5 \text{ mm}^3$; and convert data sequence as *for x:y:z loop*.

The final data file in WCT voxel format is *breast_eps.vxb* & *breast_econd.vxb*.



One cross-section of the converted model is shown in left Fig.

The loaded 3D breast model in the GUI looks like



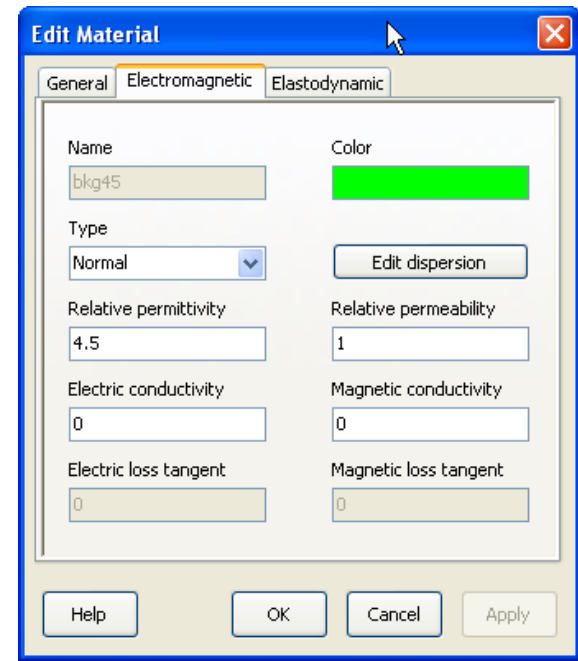
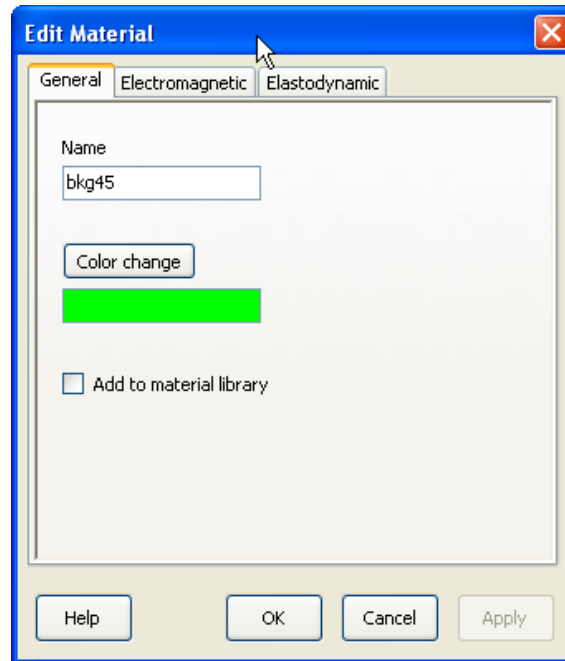
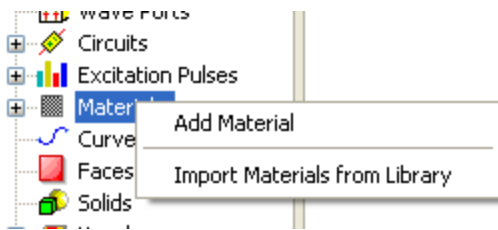
The size of the voxel volume is $12 \times 16.5 \times 10.5 \text{ cm}^3$, with $80 \times 110 \times 70$ cells in x, y and z direction, respectively.

For this model, the material outside the breast has $\epsilon_r = 4.5$, $\sigma_e = 0$.

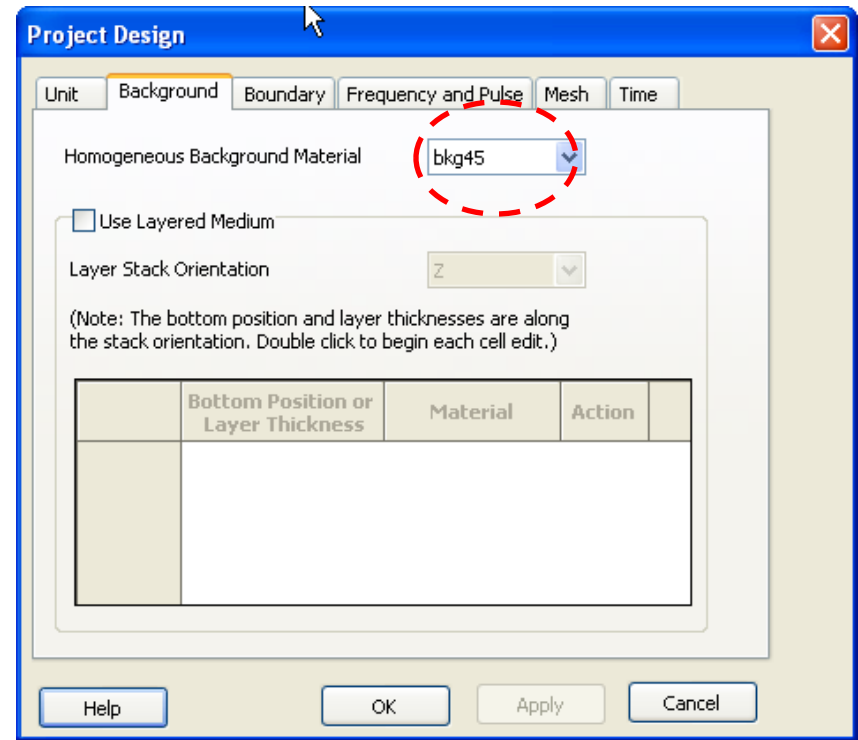
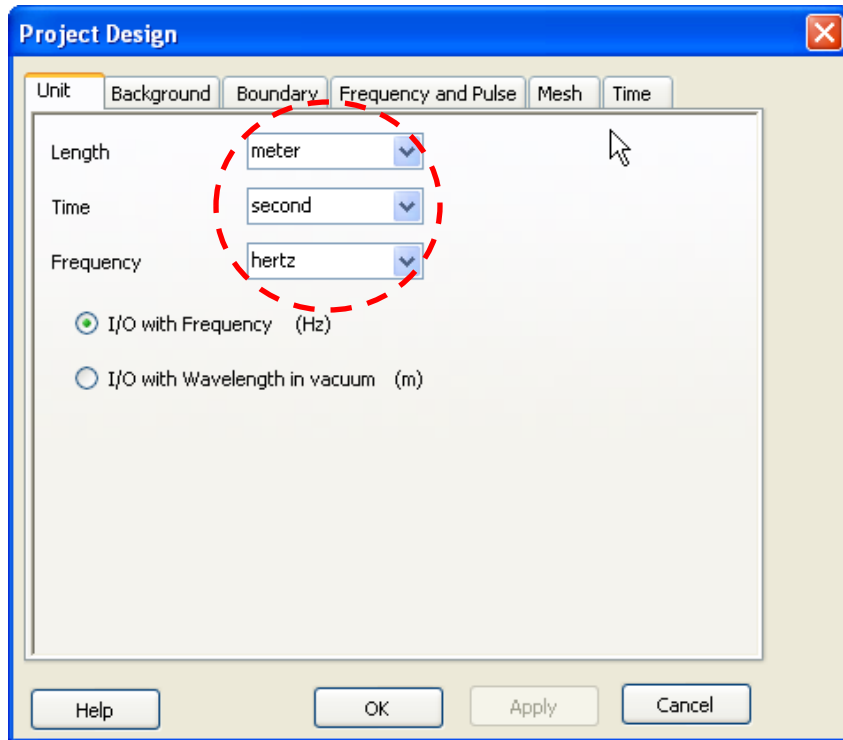
In this case, we will show how to use face-snapshot to monitor the propagation of EM wave in the breast. The source is an ideal E dipole outside the breast model. The breast model is immersed into a matching material with $\epsilon_r=4.5$, $\sigma_e=0$ to reduce the reflection from the breast.

Steps:

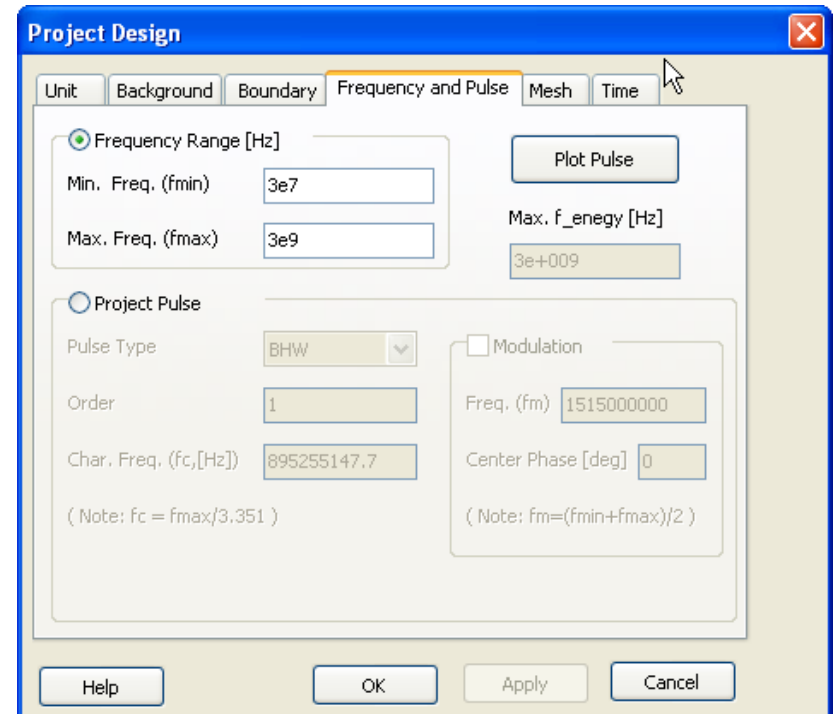
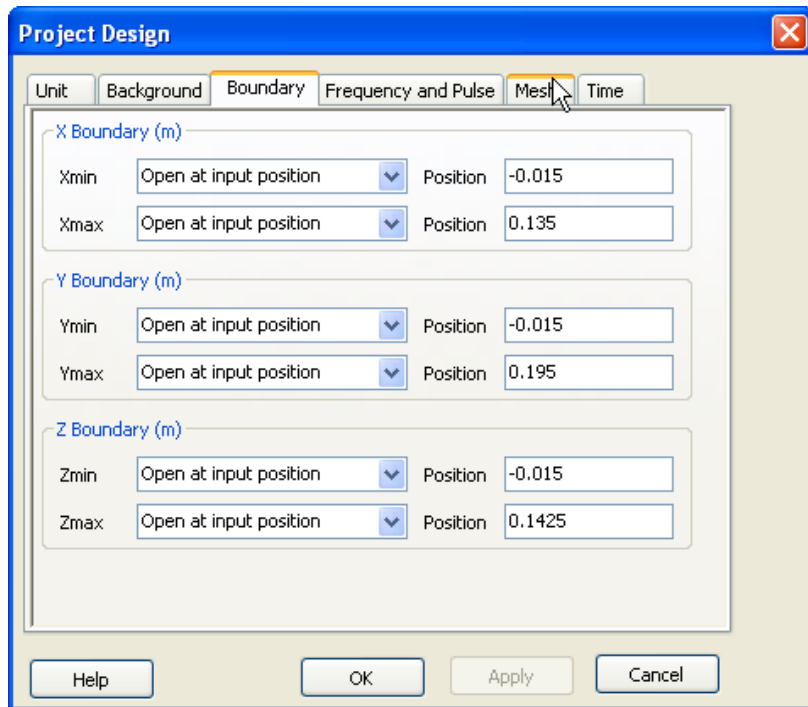
- (1) Create a new case and save with name “tpl”
- (2) Create matching material “bkg45”



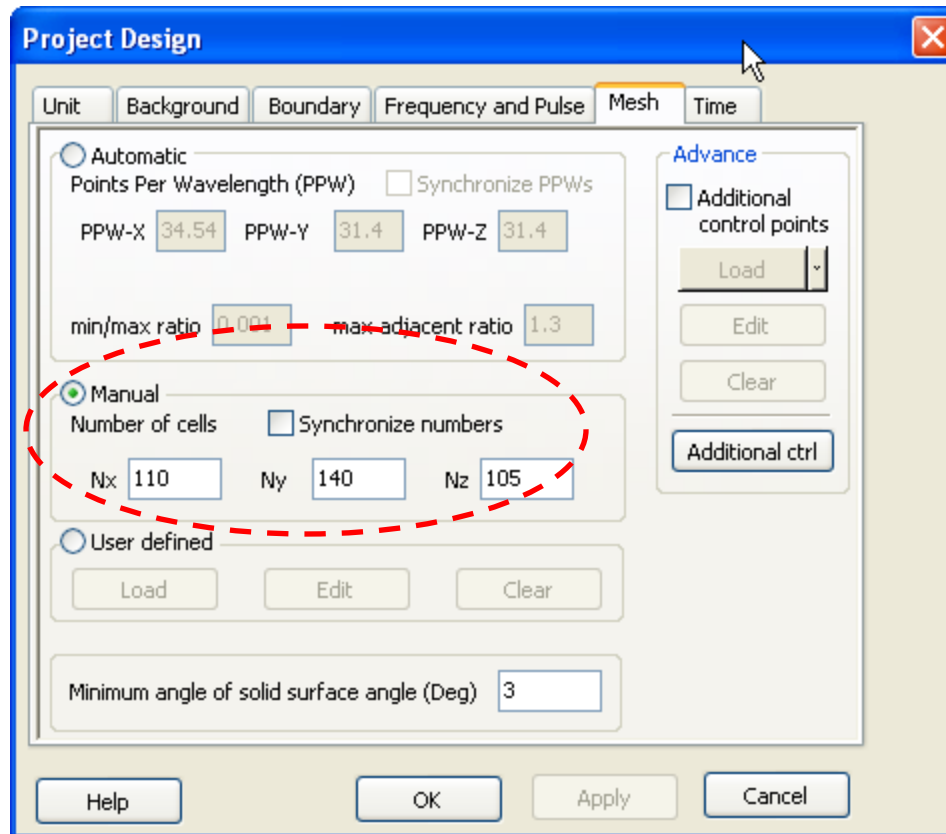
(3) Setup project unit as **meter, second & hertz**, background as “**Bkg45**”.



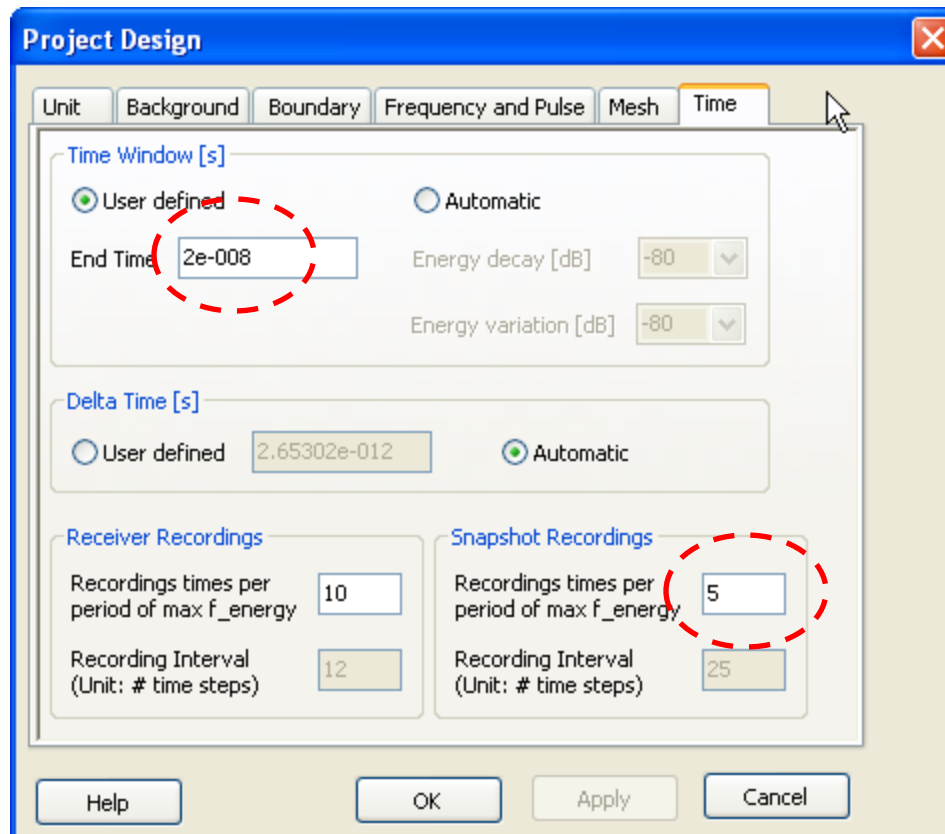
set computation domain B.C. as following, and set simulation frequency as $3e^7$ - $3e^9$ Hz



Because we want to capture the signal in all voxel cell, we use **manual uniform mesh** as following. (user can use **Automatic** mesh also)



Because we only want to capture the transient signal on face-snapshot, we don't need to analyze the frequency response. Here, we use “**User defined**” time window, 8 ns.



In order to make the picture on the snapshot more smooth, we increase “Snapshot Recordings” density to 5 points per wavelength at f_{\max} ($3e^9\text{Hz}$).

4) Add dipole source in the project.

Edit Existing Source

Name: Type:

Location (x, y, z): Polarization:

Excitation Pulse

Use project pulse

Use individual pulse

5) Define voxel

Voxel Editor

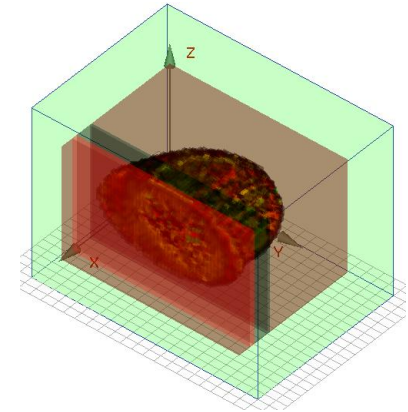
General
Name: Unit:

Physical Size
Loaded Volume
Volume Start Position in Computation Domain (x,y,z): Resolution (dx,dy,dz):

Advance
 Use the Whole Volume Use sub-volme
Corner 1 (x,y,z):
Corner 2 (x,y,z):

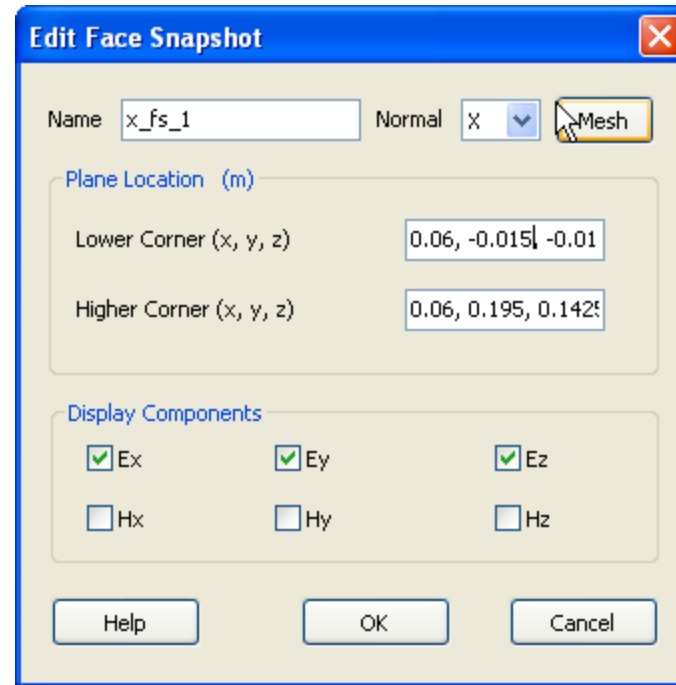
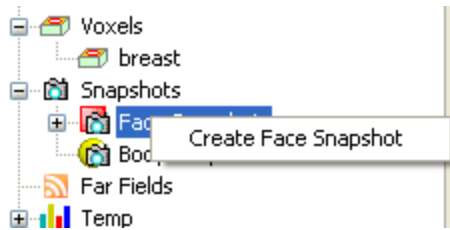
Electrical Parameters
Relative Permittivity: Bulk Load
Elec. Conductivity (S/m): Bulk Load
Relative Permeability: Bulk Load
Mag. Conductivity (W/m): Bulk Load

Help OK Cancel

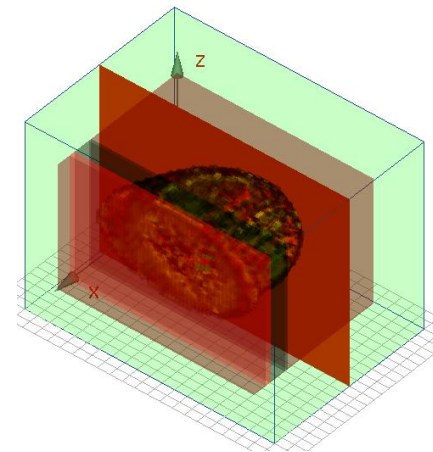


The data file is *breast_eps.vxb* & *breast_econd.vxb* under folder *Digital_breast_model*.

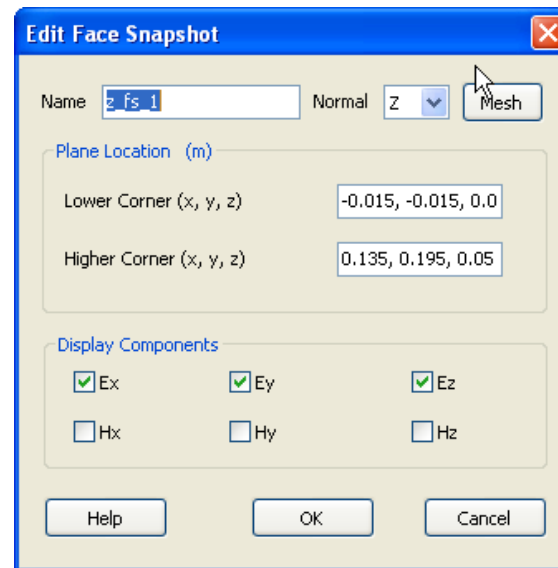
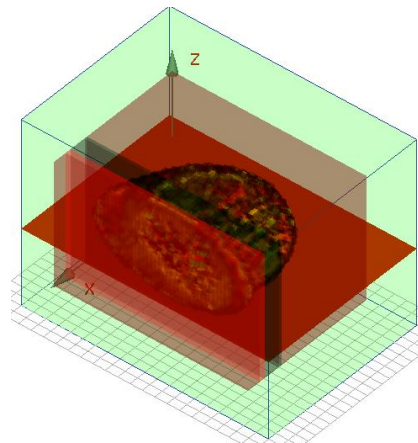
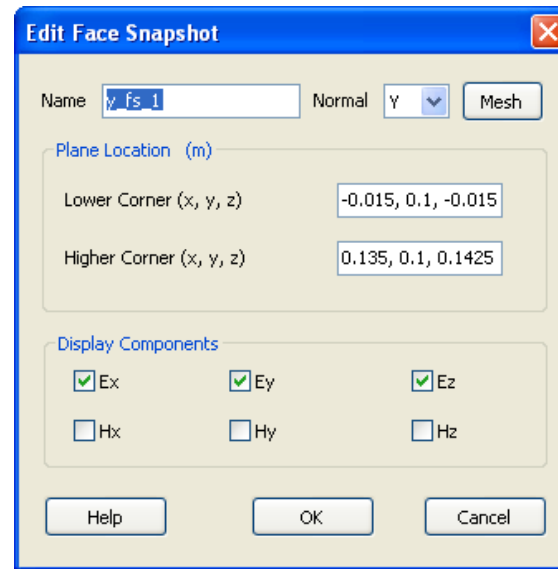
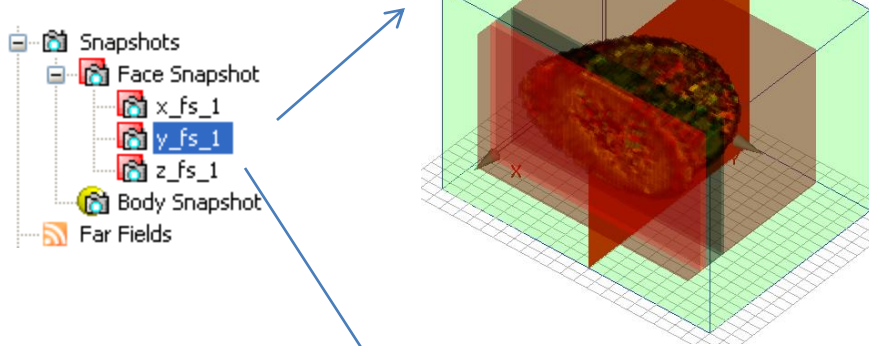
5) Define face-snapshots



X normal snapshot,
capture E field only



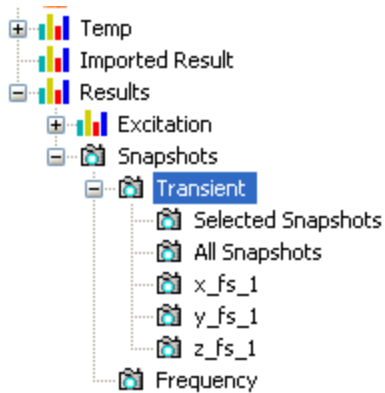
Y normal snapshot



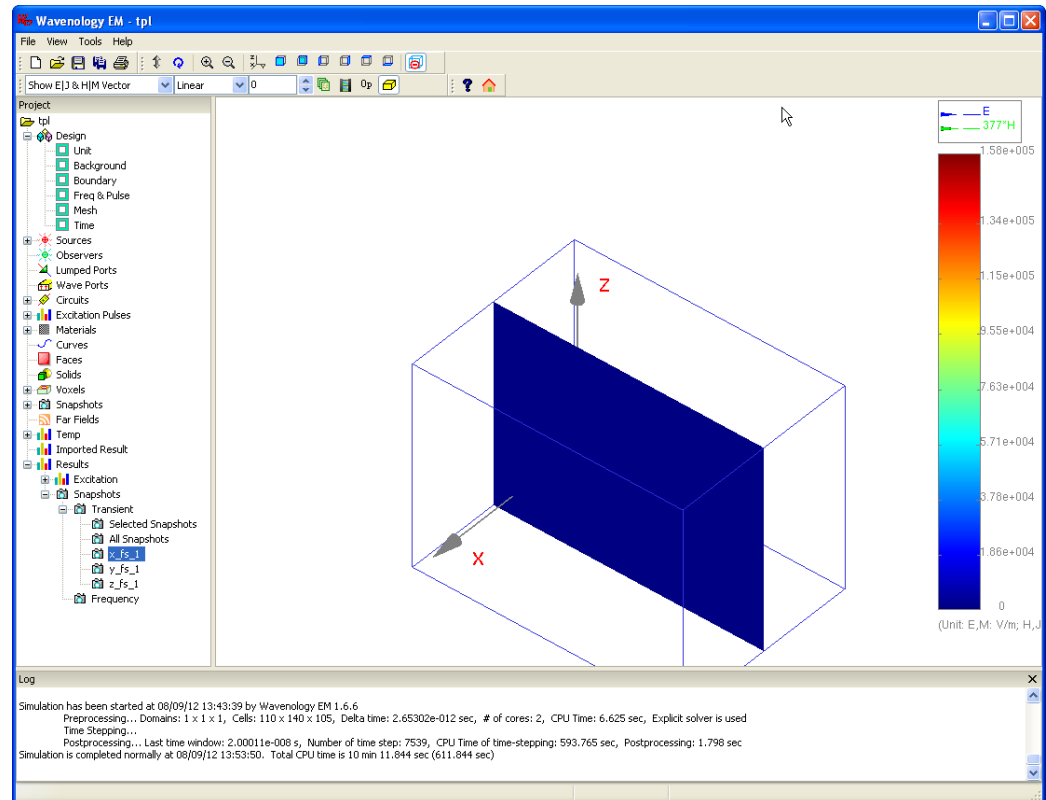
Z normal snapshot

6) Simulate the case and check result.

Expand Results treenode till you can see **Snapshot** -> **Transient**



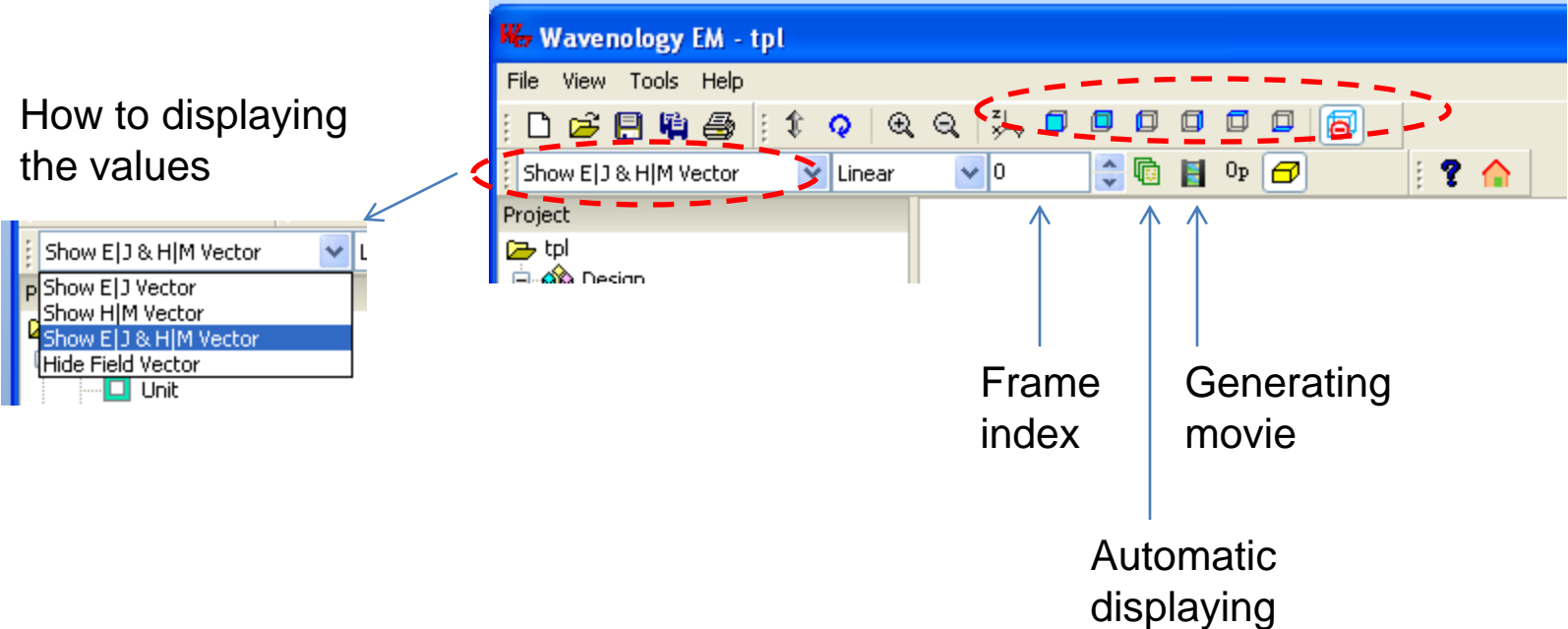
Click “x_fs_1” snapshot to enter snapshot displaying mode



User can use this toolbar to control the snapshot displaying

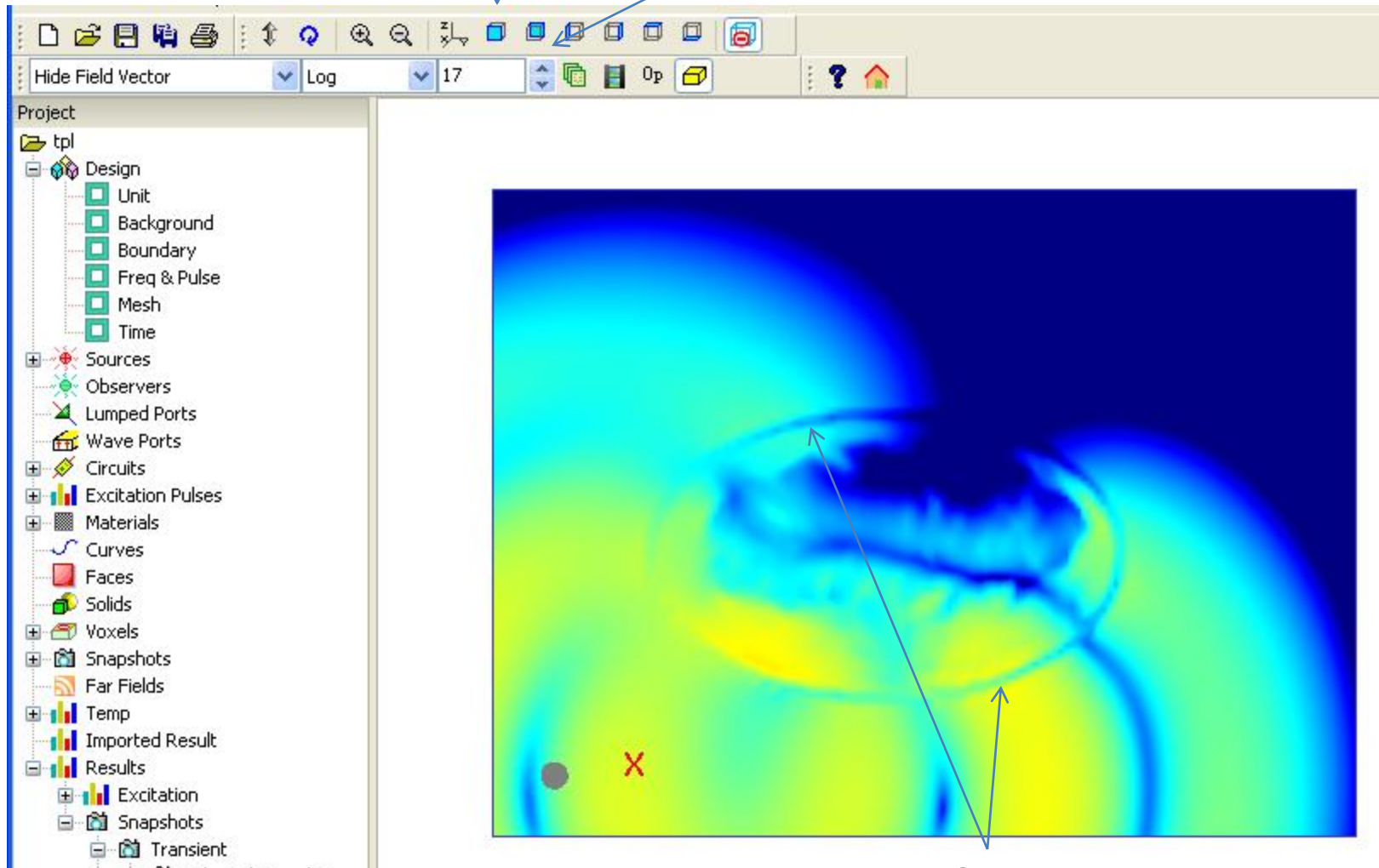
View angle

How to displaying the values



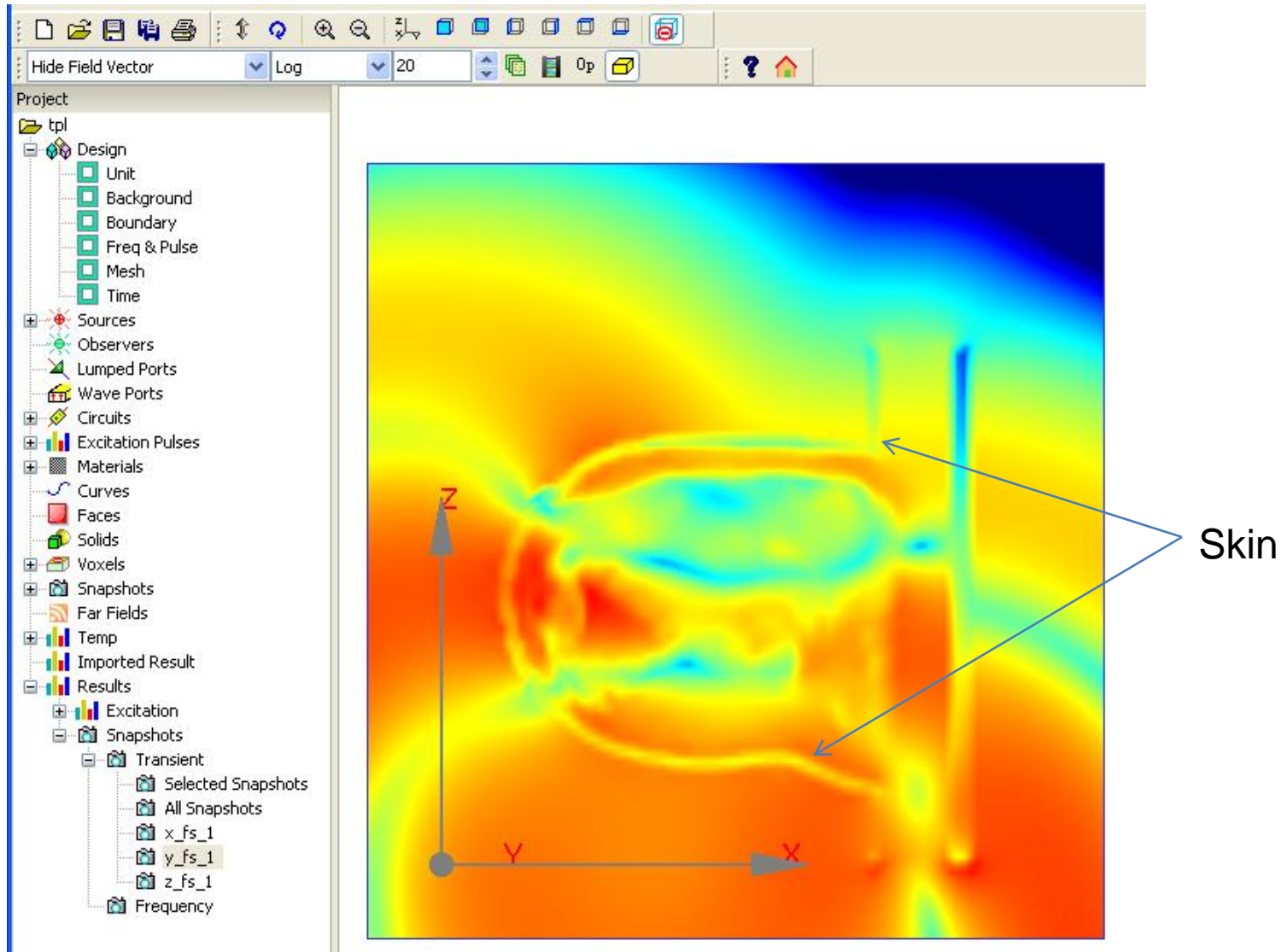
Here, we choose “front view” + “Hide Field Vector” + “Log” mode

then use the spin button to change displayed frame index. The E field propagation around breast can be observed clearly.

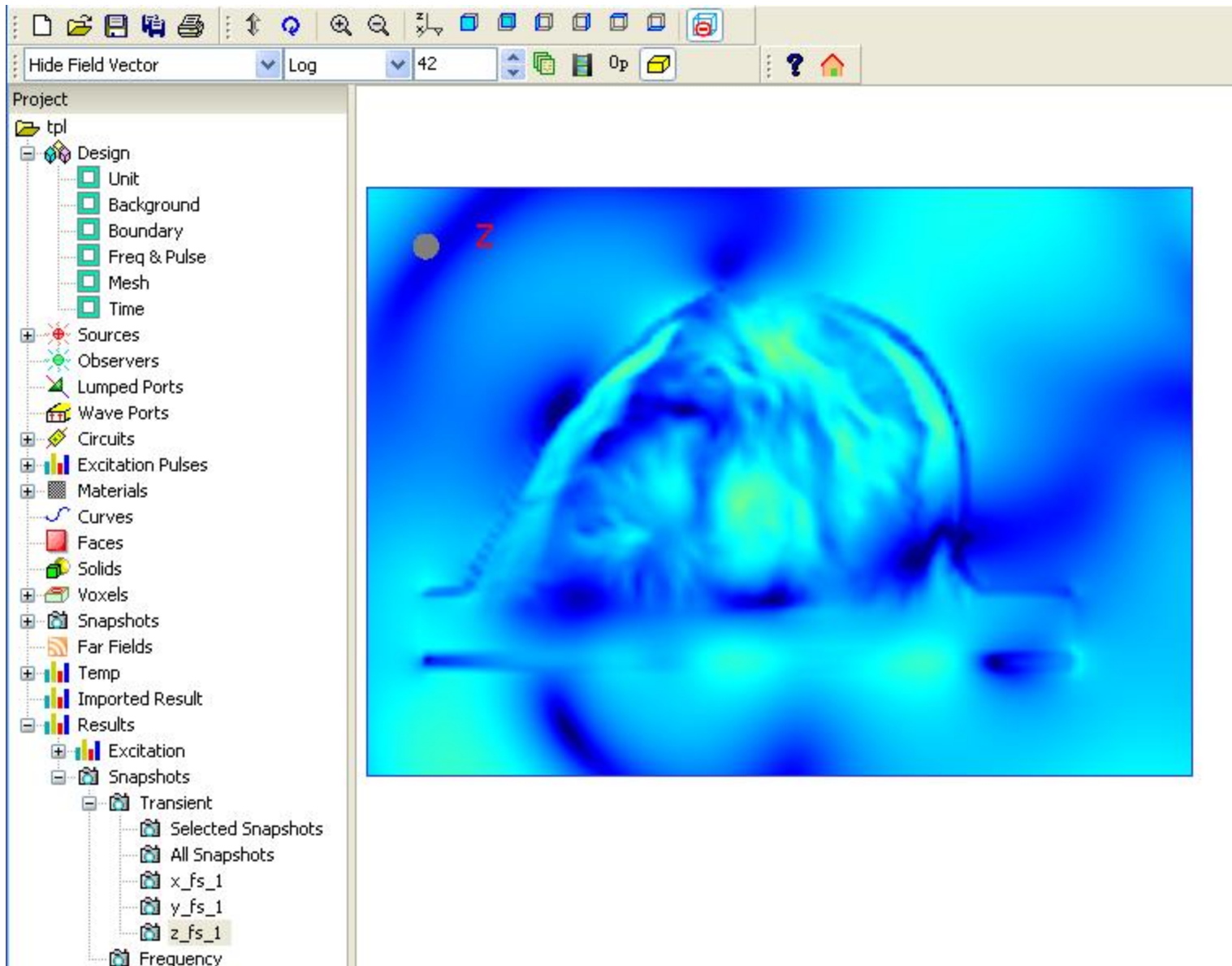


Skin

Similarly, for y plane snapshot “y_sf_1”, we choose “left view” + “Hide Field Vector” + “Log” mode to display



Similarly, for z plane snapshot “z_sf_1”, we choose “top view” + “Hide Field Vector” + “Log” mode to display



Voxel Demo Cases

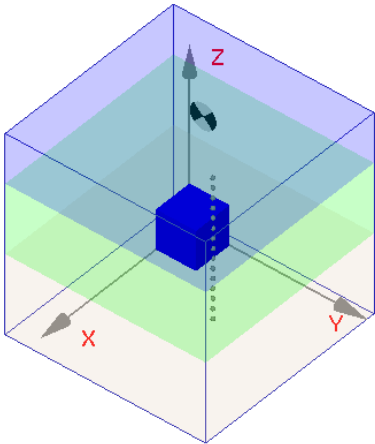
Part II: EL Solver

- 1) Transient response from a Box in 3layered background, general isotropic material
- 2) Transient response from a Box in homogenous background, weak anisotropic material
- 3) Simulation on 2D Marmousi II model
- 4) Simulation on SEG_EAGE_3DOverThrustModel

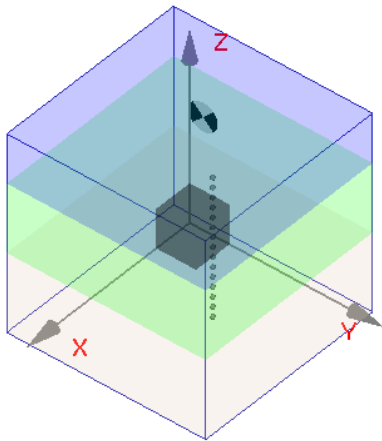
Case (1) a Box in 3 layers Background

general isotropic material

This case simulates a box in a 3 layers background. We will show how to define this box by the voxel way and compare the simulation result with the result by defining a box by a normal way.



The left Fig. is the reference case, a regular box in a 3 layers background.



In the voxel case, we will replace the box region by a voxel with 2x2x2 cells, and then compare the transient response on receiver in two cases.

Case setup: Materials

Rock

The 'Edit Material' dialog box for the 'rock' material is shown. It has three tabs: 'General', 'Electromagnetic', and 'Elastodynamic'. The 'Elastodynamic' tab is selected. The 'Name' field contains 'rock' and the 'Color' field shows a brownish-tan color. The 'Mass density' is set to 2200 kg/m³. The 'P-Velocity' is 2500 m/s and the 'Qp (0, Inf)' is 'inf'. The 'S-Velocity' is 1700 m/s and the 'Qs (0, Inf)' is 'inf'. There is an 'Advanced' section with two radio buttons: 'Anisotropic Material' and 'PoroElastic Material', each with a 'Property' button next to it. At the bottom are 'Help', 'OK', 'Cancel', and 'Apply' buttons.

for top layer

Bottom

The 'Edit Material' dialog box for the 'bottom' material is shown. It has three tabs: 'General', 'Electromagnetic', and 'Elastodynamic'. The 'Elastodynamic' tab is selected. The 'Name' field contains 'bottom' and the 'Color' field shows a blue color. The 'Mass density' is set to 2500 kg/m³. The 'P-Velocity' is 3000 m/s and the 'Qp (0, Inf)' is 'inf'. The 'S-Velocity' is 1900 m/s and the 'Qs (0, Inf)' is 'inf'. There is an 'Advanced' section with two radio buttons: 'Anisotropic Material' and 'PoroElastic Material', each with a 'Property' button next to it. At the bottom are 'Help', 'OK', 'Cancel', and 'Apply' buttons.

for bottom layer

middle

The 'Edit Material' dialog box for the material 'middle' is shown. It has three tabs: 'General', 'Electromagnetic', and 'Elastodynamic'. The 'Elastodynamic' tab is selected. The 'Name' field contains 'middle' and the 'Color' field is a bright green rectangle. The 'Mass density' is 2800 kg/m³. The 'P-Velocity' is 3800 m/s and 'Qp (0, Inf)' is inf. The 'S-Velocity' is 2300 m/s and 'Qs (0, Inf)' is inf. The 'Advanced' section is collapsed, showing radio buttons for 'Anisotropic Material' and 'PoroElastic Material', each with a 'Property' button.

for middle layer

f2

The 'Edit Material' dialog box for the material 'f2' is shown. It has three tabs: 'General', 'Electromagnetic', and 'Elastodynamic'. The 'Elastodynamic' tab is selected. The 'Name' field contains 'f2' and the 'Color' field is a blue rectangle. The 'Mass density' is 4000 kg/m³. The 'P-Velocity' is 3500 m/s and 'Qp (0, Inf)' is inf. The 'S-Velocity' is 2200 m/s and 'Qs (0, Inf)' is inf. The 'Advanced' section is collapsed, showing radio buttons for 'Anisotropic Material' and 'PoroElastic Material', each with a 'Property' button.

for box

Case setup

Project setting: unit

The screenshot shows the 'Project Design' dialog box with the 'Unit' tab selected. The 'Length' dropdown is set to 'meter', 'Time' to 'millisecond', and 'Frequency' to 'hertz'. There are two radio button options: 'I/O with Frequency (Hz)' (selected) and 'I/O with Wavelength in water (m)'. Below these is a section for 'Action option when length/freq. unit change' with two radio buttons: 'Scale whole system' (selected) and 'Keep the system value unchange'. At the bottom are 'Help', 'OK', 'Apply', and 'Cancel' buttons.

Project size & B.C.

The screenshot shows the 'Project Design' dialog box with the 'Boundary' tab selected. It displays boundary conditions for X, Y, and Z axes. For each axis, there are 'Xmin' and 'Xmax' (or 'Ymin', 'Ymax', 'Zmin', 'Zmax') settings. Each setting consists of a dropdown menu (set to 'Open at input position') and a 'Position' text box. For X and Y axes, the positions are -500 and 500. For the Z axis, 'Zmin' is 'Open at input position' with position -500, and 'Zmax' is 'Soft at input position' with position 500. At the bottom are 'Help', 'OK', 'Apply', and 'Cancel' buttons.

Project background

Project Design

Unit Background Boundary Frequency and Pulse Mesh Time

Homogeneous Background Material f2

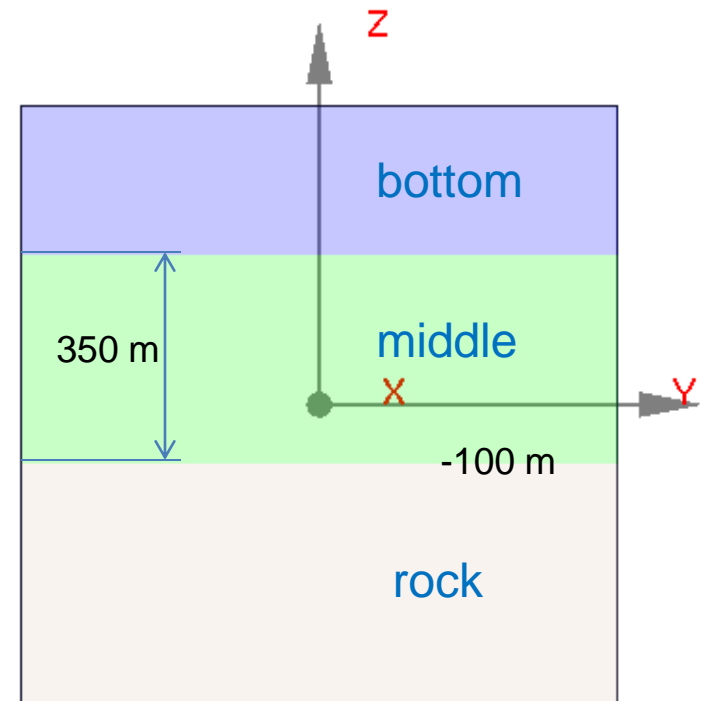
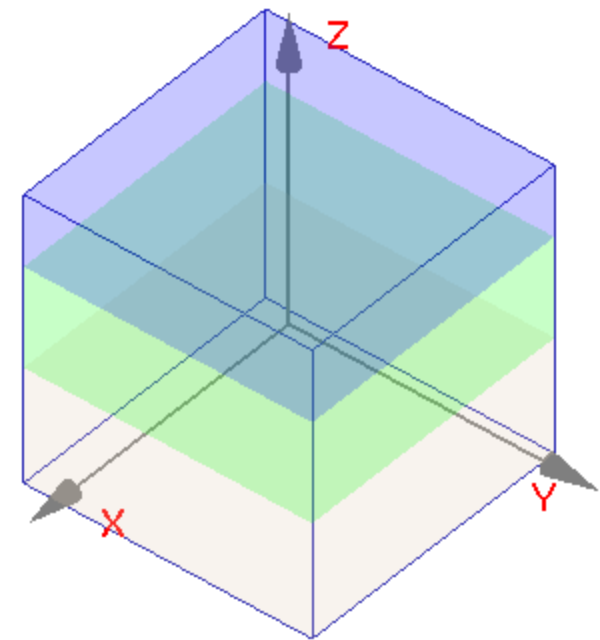
Use Layered Medium

Layer Stack Orientation Z

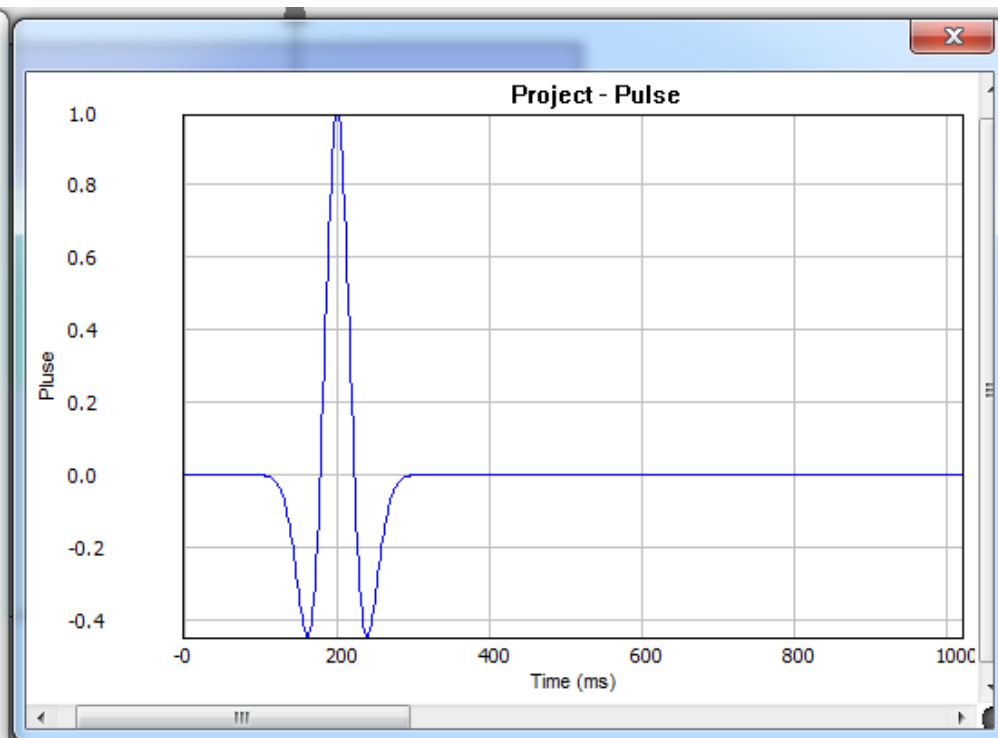
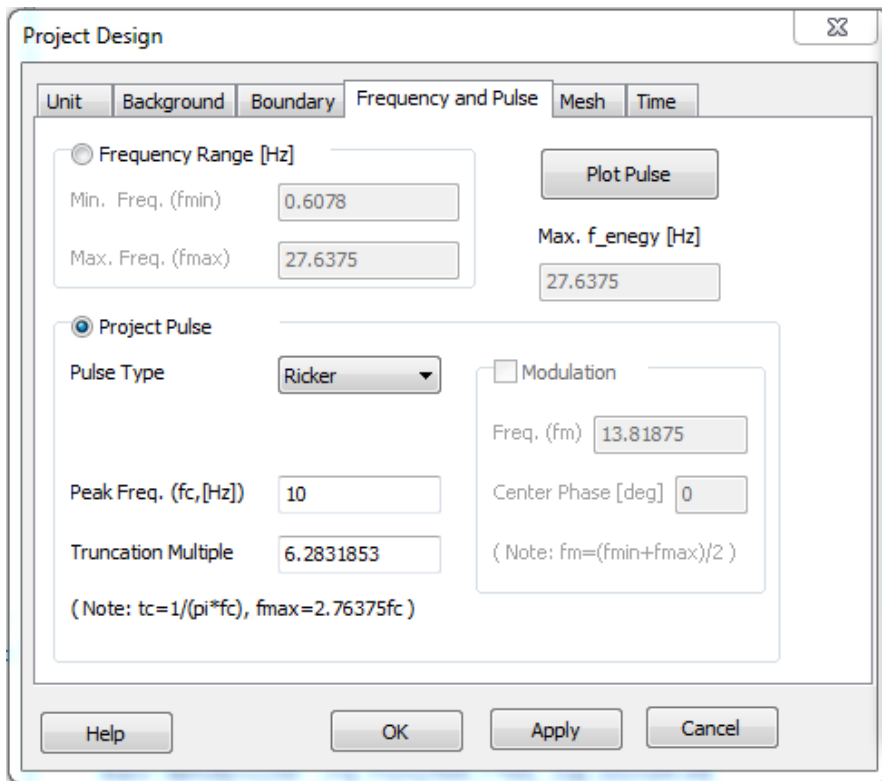
(Note: The bottom position and layer thicknesses are along the stack orientation. Double click to begin each cell edit.)

	Bottom Position or Layer Thickness	Material	Action
Top		bottom	
			Add
1	350	middle	Delete
Bottom	-100	rock	

Help OK Apply Cancel

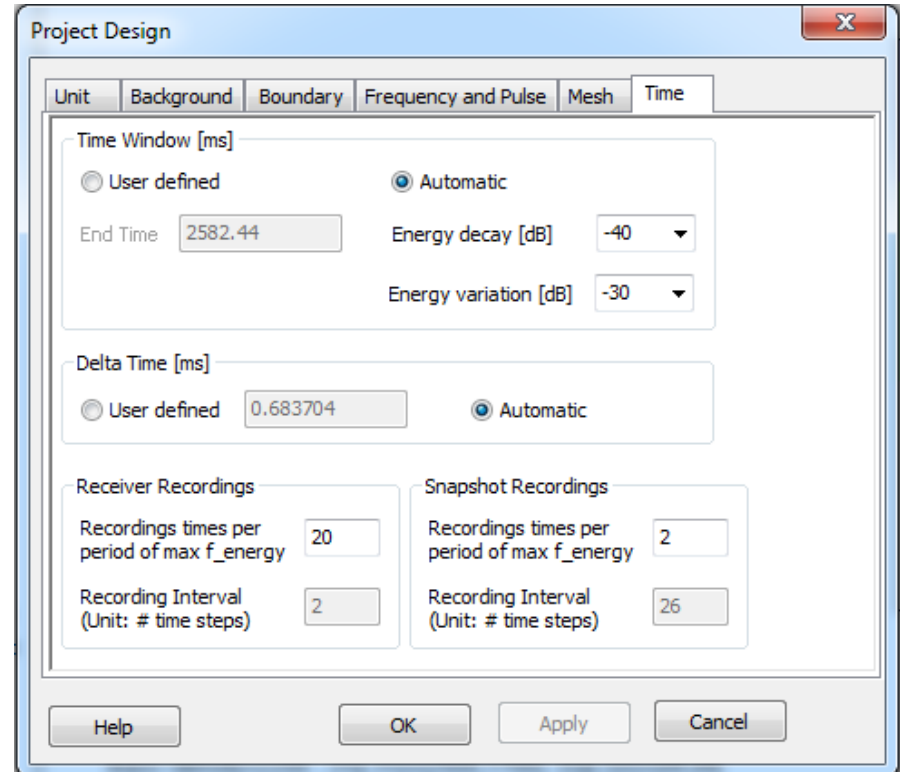
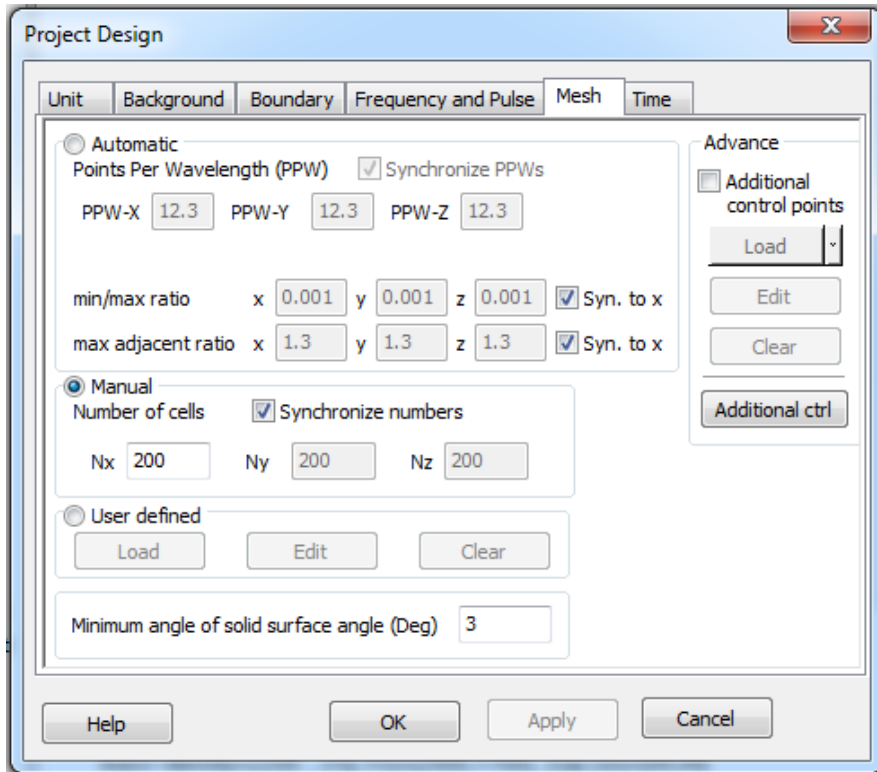


Project pulse: $f_c=10$ Hz Ricker wave



Project mesh: 200 cell in x, y & z

Project time setting: default time window & automatic Δt



Source: moment tensor at (-200, -100, -350)

Edit Existing Source [X]

Name: Type:

Location (x, y, z): (Strike,Dip,Rake)

3x3 Values:

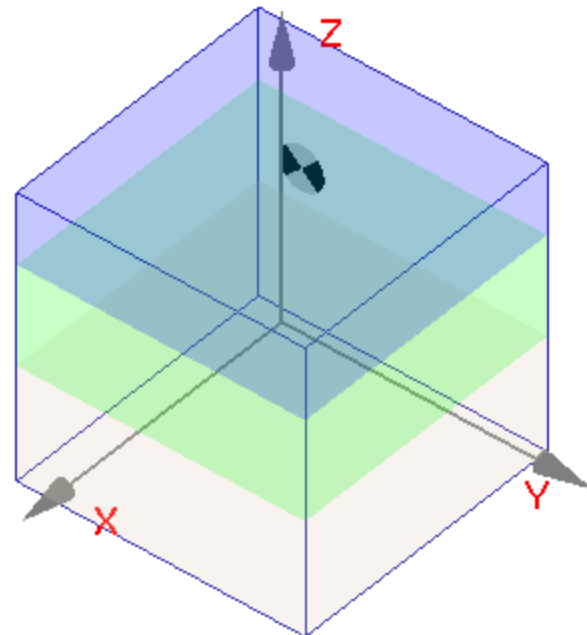
Format:

Excitation Pulse

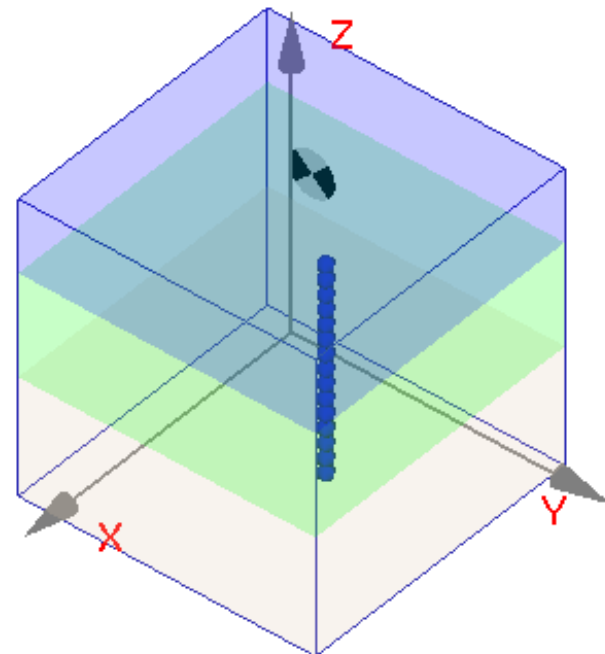
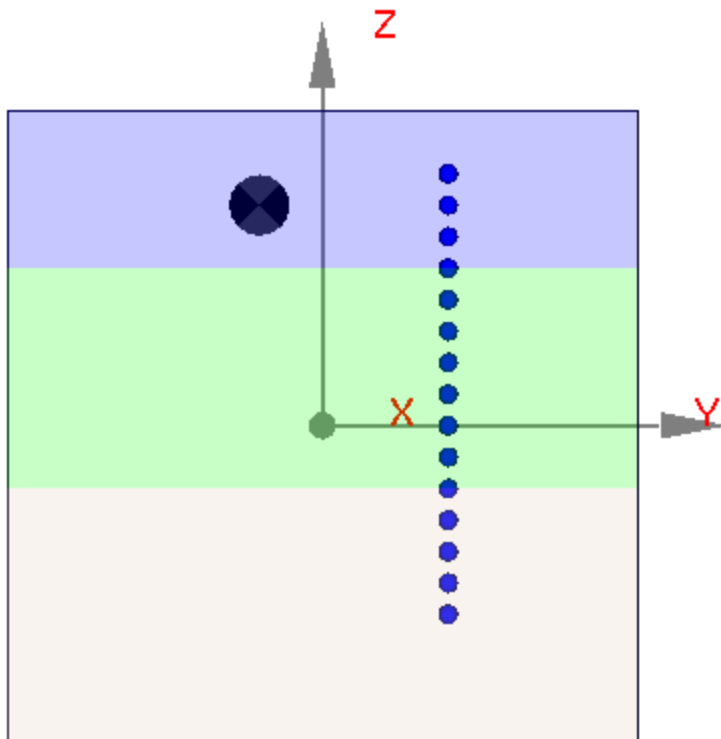
Use project pulse Pulse type:

Use individual pulse Delay [ms]:

Amplitude: (Pa/s)



15 observers along z: $x=100$, $y=200$; z positions: [400:50:-350]



Box region

As right figure.

Size: $(0,0,0)$ - $(200,200,200)$

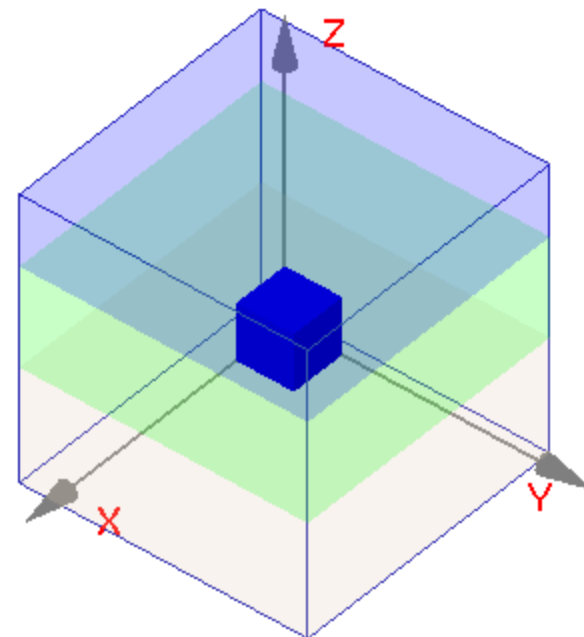
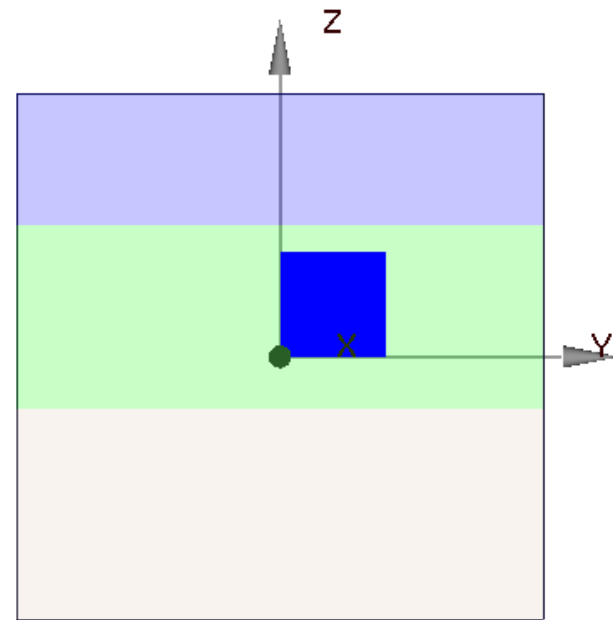
Material: f2

mass: 4000

vp: 3500

vs: 2200

But in this case, we will
use a voxel to represent
this: $(0,0,0)$ -
 $(200,200,200)$
rectangular region
instead of a regular box



Prepare voxel data files

This is a rectangular region with regular isotropic EL material, so, we need only 3 inputs: mass, vp, vs.

Assuming this voxel is composed by: $2 \times 2 \times 2 = 8$ uniform cells (cell size is $100 \times 100 \times 100 \text{ m}^3$), each cell's material is defined by file (ASCII format), we can define 3 files for each input, as following,

Mass.txt

```
2
2
2

4000
4000

4000
4000

4000
4000

4000
4000
```

Vp.txt

```
2
2
2

3500
3500

3500
3500

3500
3500

3500
3500
```

Vs.txt

```
2
2
2

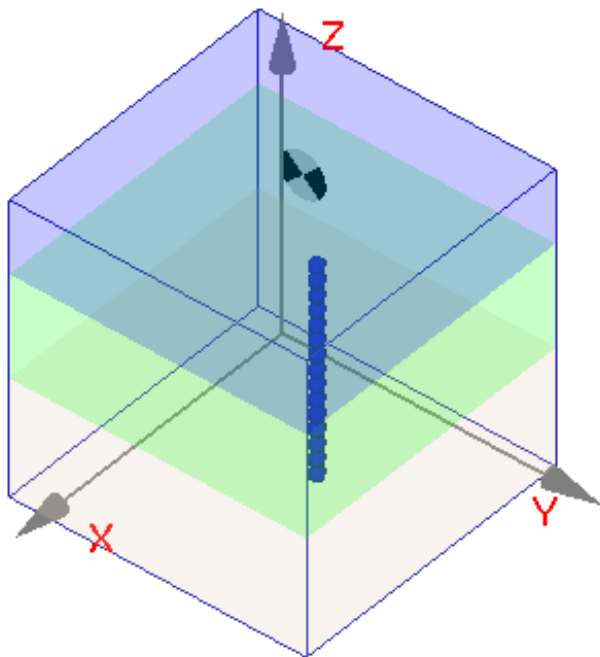
2200
2200

2200
2200

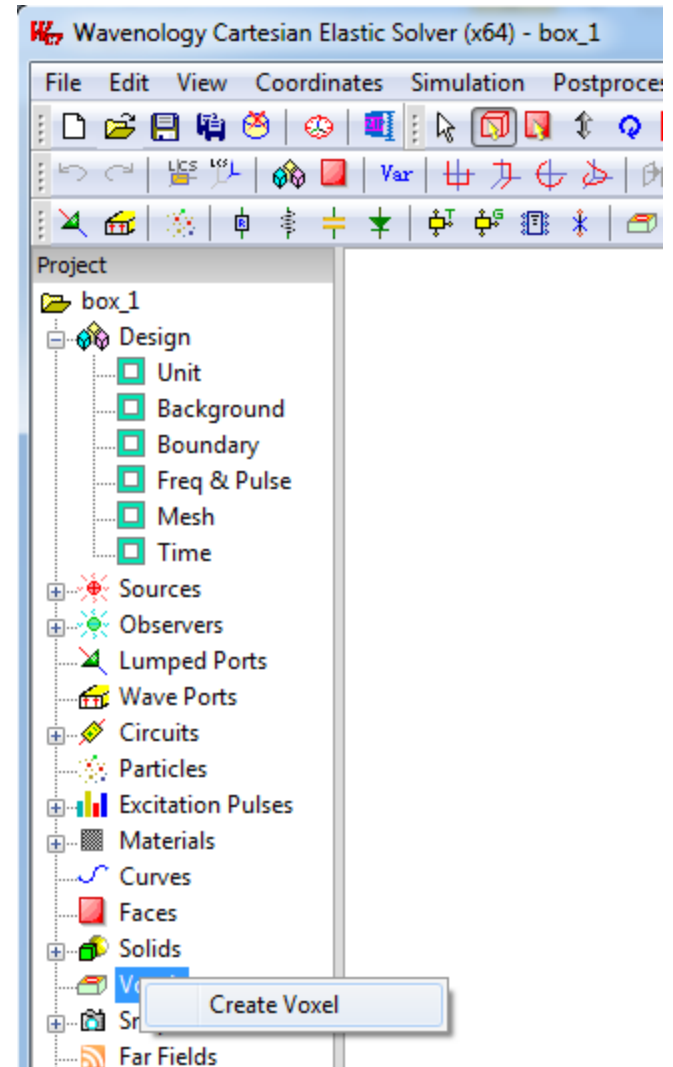
2200
2200

2200
2200
```

After project setup, source and receiver, voxel data files are ready, we can see the project as following



Then, create voxel



Voxel name

The image shows a 'Voxel Editor' dialog box with several sections and highlighted fields:

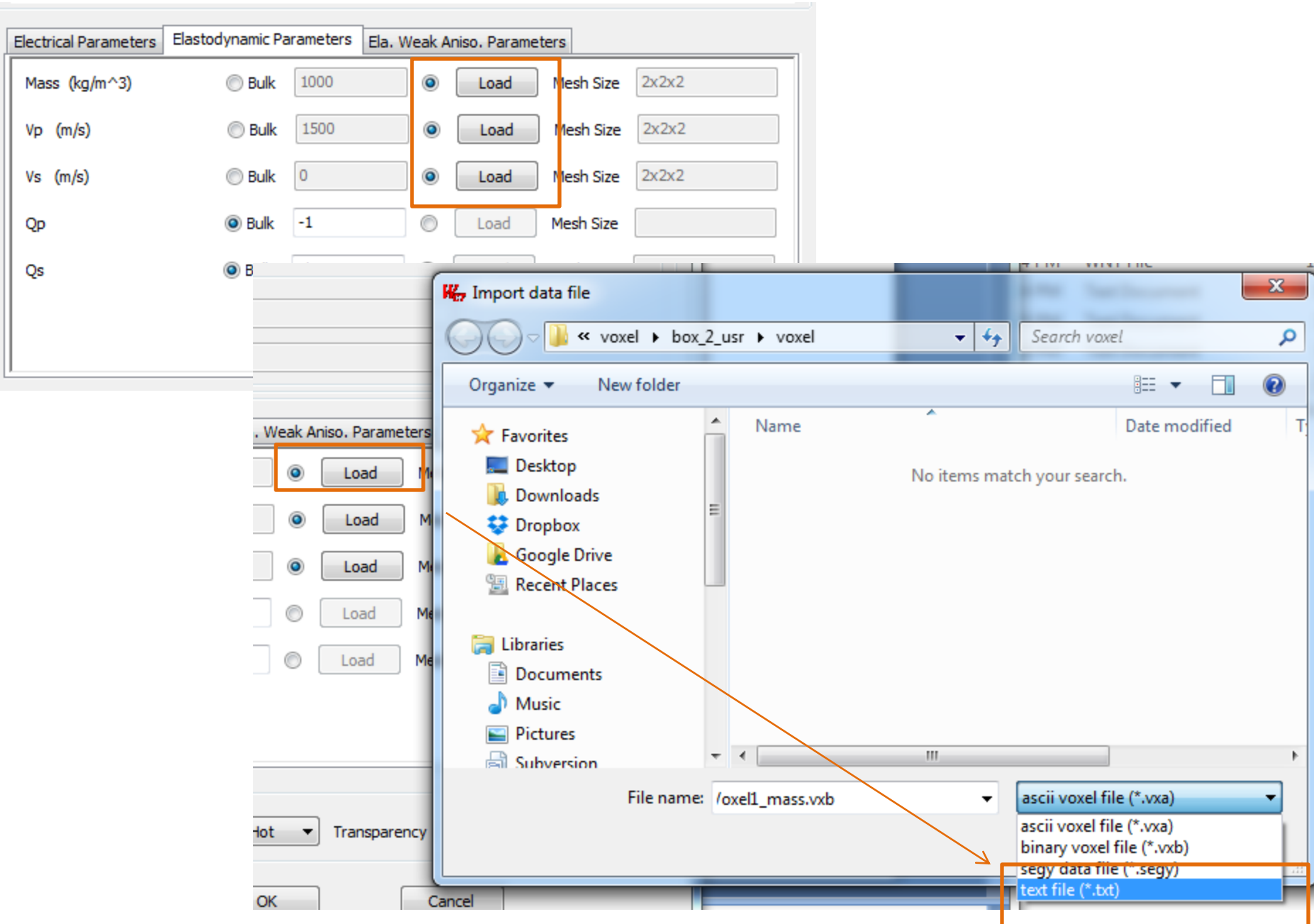
- General:** A blue box highlights the 'Name' field containing 'Voxel1'. An orange box highlights the 'Unit' dropdown menu set to 'meter'.
- Physical Size:** An orange box highlights the 'Loaded Volume' section, which includes 'Volume Start Position in computation domain (x,y,z)' set to '0, 0, 0' and 'Resolution (dx,dy,dz)' set to '100, 100, 100'.
- Advance:** The 'Use the Whole Volume' radio button is selected.
- Electrical Parameters:** An orange box highlights the 'Electrical Parameters' tab. Within this tab, another orange box highlights the 'Load' button next to the 'Relative Permittivity' field.
- Other Parameters:** The 'Ela. Weak Aniso. Parameters' tab is also visible.
- Display:** The 'Data Type' is set to 'Mass', 'Color Map' is 'Hot', and there is a 'Transparency' slider.

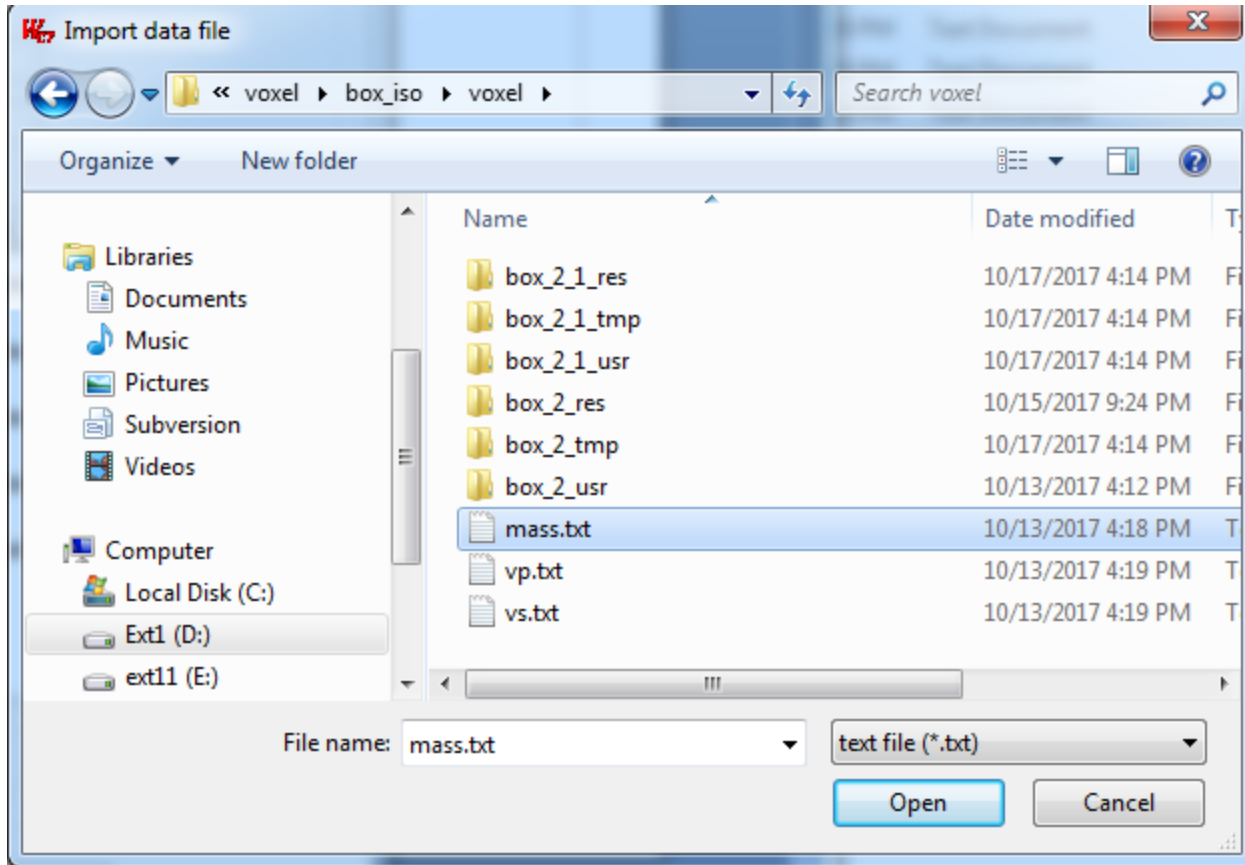
Voxel unit

Voxel start position & resolution

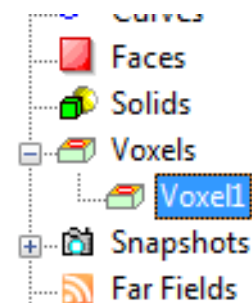
Skip pages:
Electrical parameters & Ela. Weak Aniso. Parameters

In the Elastodynamic Parameters page, load mass, vp, vs from TXT data files, respectively.

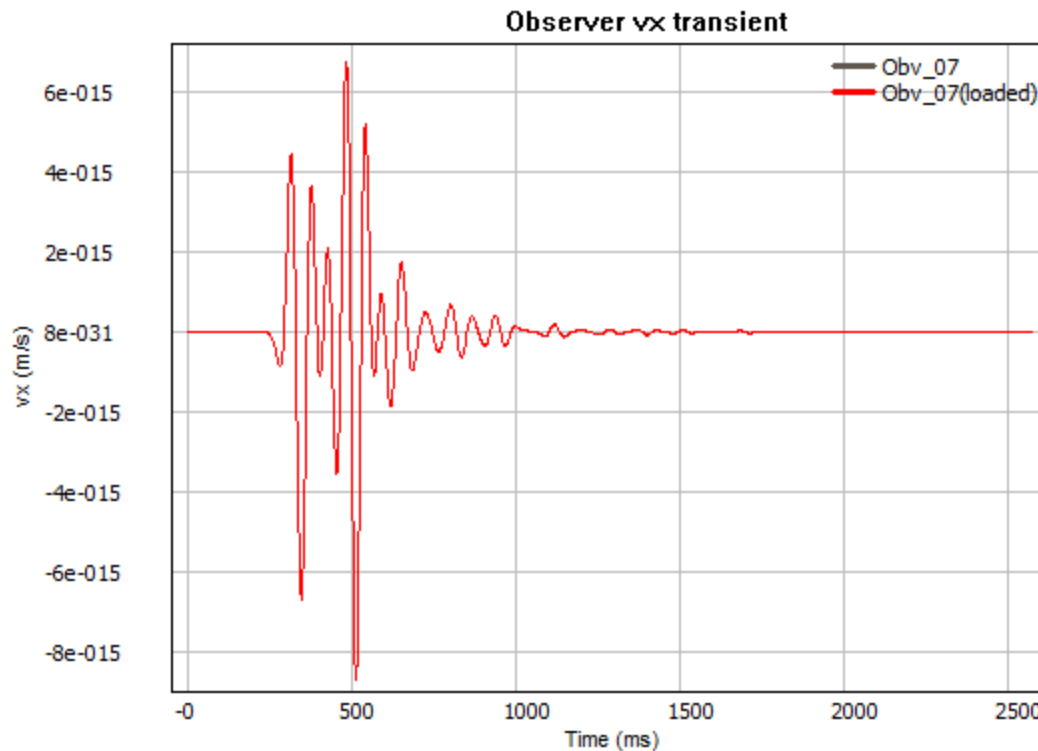




After all voxel parameters are set correctly, then “OK” to add this voxel in the project, as right figure



Then we can simulate this project, and compare the simulation result with a regular box setting. We can see two sets of data overlap.

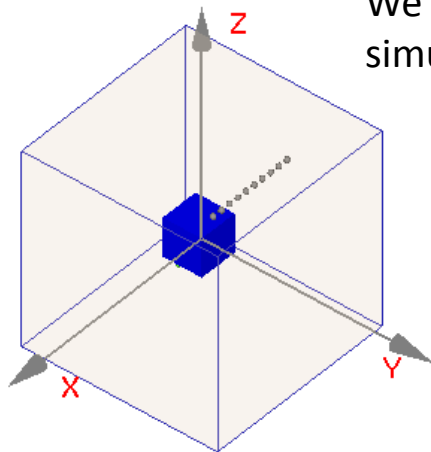


- Voxel
- Regular box

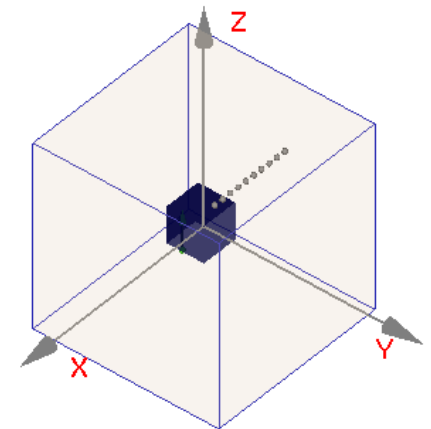
Case (2) a Box in Homogenous Background

weak anisotropic material

This case simulates a box with anisotropic material in a homogenous background. We will show how to define this box by the voxel way and compare the simulation result with the result by defining a box by a normal way.



The left Fig. is the reference case, a regular box in a homogenous background.



In the voxel case, we will replace the box region by a voxel with 2x2x2 cells, and then compare the transient response on receiver in two cases.

Case setup: Materials

Rock

Edit Material

General Electromagnetic Elastodynamic

Name: rock Color: [Color swatch]

Mass density: 1900 kg/m³

P-Velocity: 3000 m/s Qp (0, Inf): inf

S-Velocity: 1900 m/s Qs (0, Inf): inf

Advanced

Anisotropic Material

PoroElastic Material

Help OK Cancel Apply

Anisotropic Material Parameters for Elastic Wave

General Anisotropic (Unit: GPa)

Cl,j (6x6 Matrix)

	A	B	C	D	E	F
1	16.248	1.48	2.4	-1.152	0.416	-0.561
2	1.48	11.88	1.032	0.912	1.608	1.248
3	2.4	1.032	12.216	-0.6724	0.216	-0.216
4	-1.152	0.912	-0.6724	5.64	2.16	0.5
5	0.416	1.608	0.216	2.16	5.88	0.58
6	-0.561	1.248	-0.216	0.5	0.58	6.912

Thomson

Epsilon: 0.1 Delta: -0.2 Gamma: 0.3

Anlges (Unit: Degree)

ZX Axis Angles theta_Z: 0 phi_Z: 0 theta_X: 90

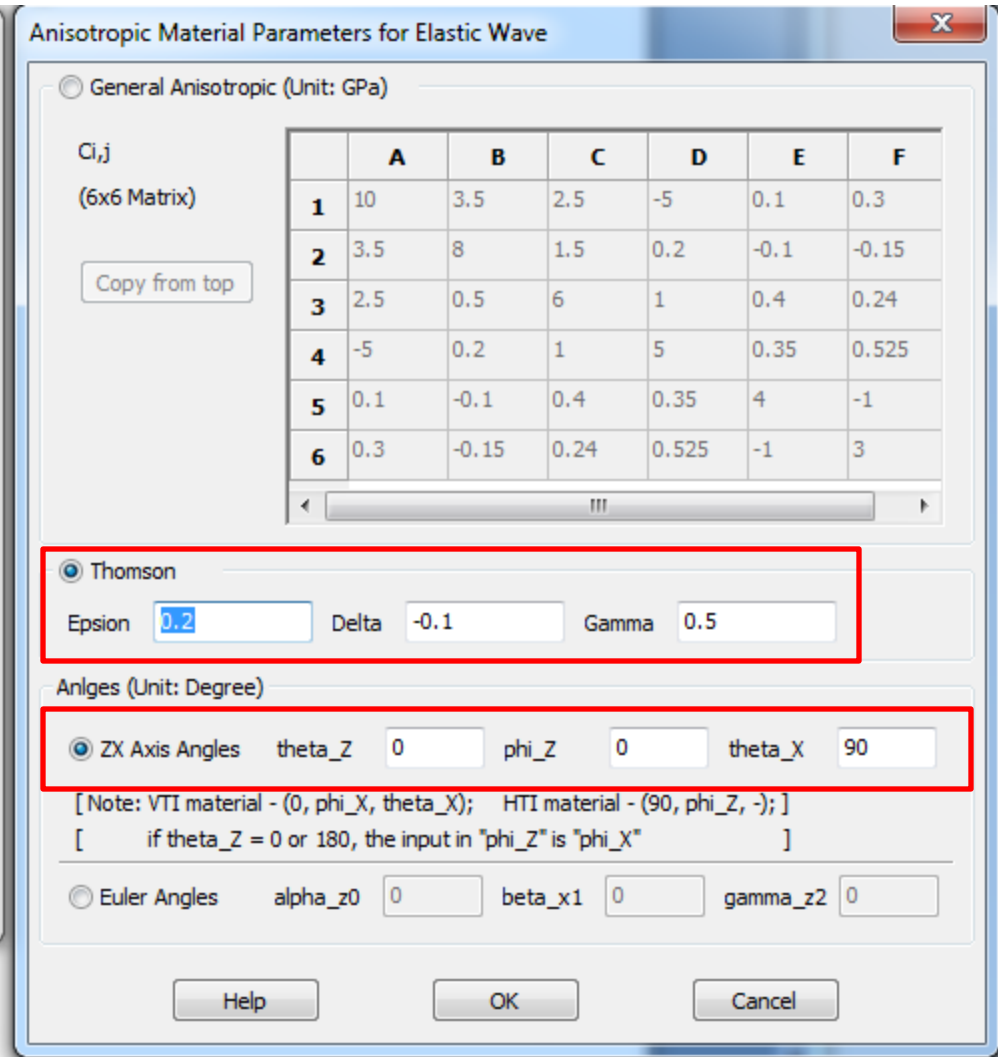
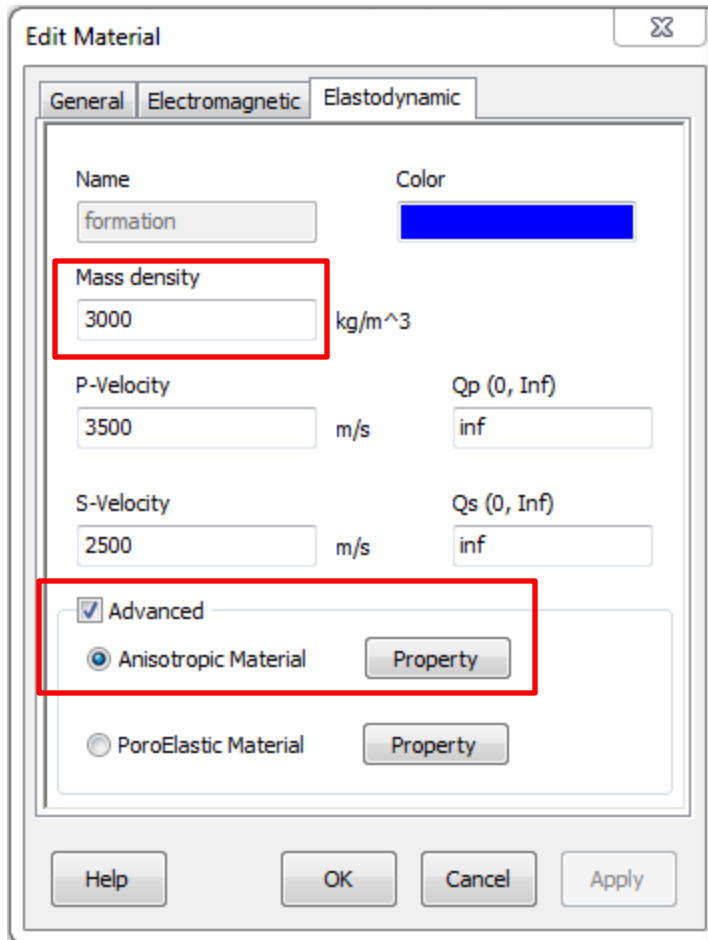
[Note: VTI material - (0, phi_X, theta_X); HTI material - (90, phi_Z, -);]
[if theta_Z = 0 or 180, the input in "phi_Z" is "phi_X"]

Euler Angles alpha_z0: 0 beta_x1: 0 gamma_z2: 0

Help OK Cancel

General anisotropic material,
for background

The VTI anisotropic material for the voxel has following properties



This property is for information purpose only , user can skip the definition of this material in the project setup

Case setup

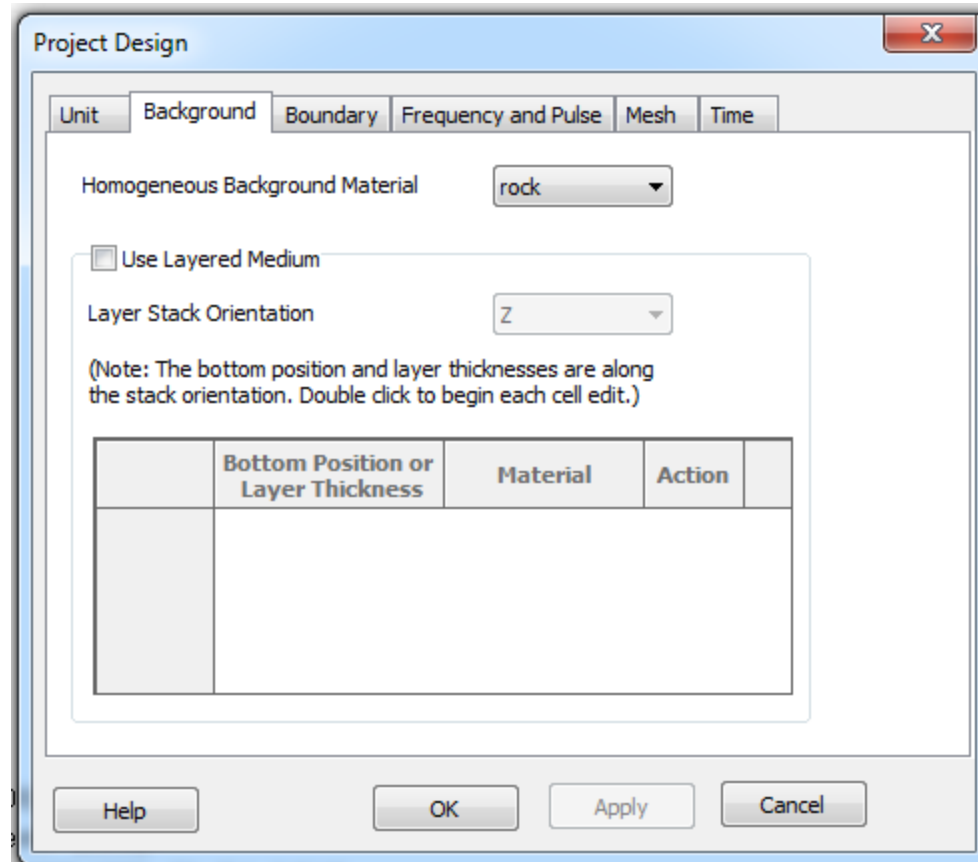
Project setting: unit

The screenshot shows the 'Project Design' dialog box with the 'Unit' tab selected. The 'Length' dropdown is set to 'meter', 'Time' to 'millisecond', and 'Frequency' to 'hertz'. There are two radio button options: 'I/O with Frequency (Hz)' (selected) and 'I/O with Wavelength in water (m)'. Below these is a section for 'Action option when length/freq. unit change' with two radio buttons: 'Scale whole system' (selected) and 'Keep the system value unchange'. At the bottom are 'Help', 'OK', 'Apply', and 'Cancel' buttons.

Project size & B.C.

The screenshot shows the 'Project Design' dialog box with the 'Boundary' tab selected. It displays settings for X, Y, and Z boundaries. For each axis (X, Y, Z), there are two rows: 'Xmin', 'Ymin', and 'Zmin' are all set to 'Open at input position' with a 'Position' of '-500'; 'Xmax', 'Ymax', and 'Zmax' are all set to 'Open at input position' with a 'Position' of '500'. The 'Zmax' dropdown is set to 'Soft at input position'. At the bottom are 'Help', 'OK', 'Apply', and 'Cancel' buttons.

Project background



Project pulse: $f_c=10$ Hz Ricker wave

Project Design

Unit Background Boundary Frequency and Pulse Mesh Time

Frequency Range [Hz]

Min. Freq. (fmin) 0.6078

Max. Freq. (fmax) 27.6375

Plot Pulse

Max. f_eneqy [Hz] 27.6375

Project Pulse

Pulse Type Ricker

Modulation

Freq. (fm) 13.81875

Peak Freq. (fc,[Hz]) 10

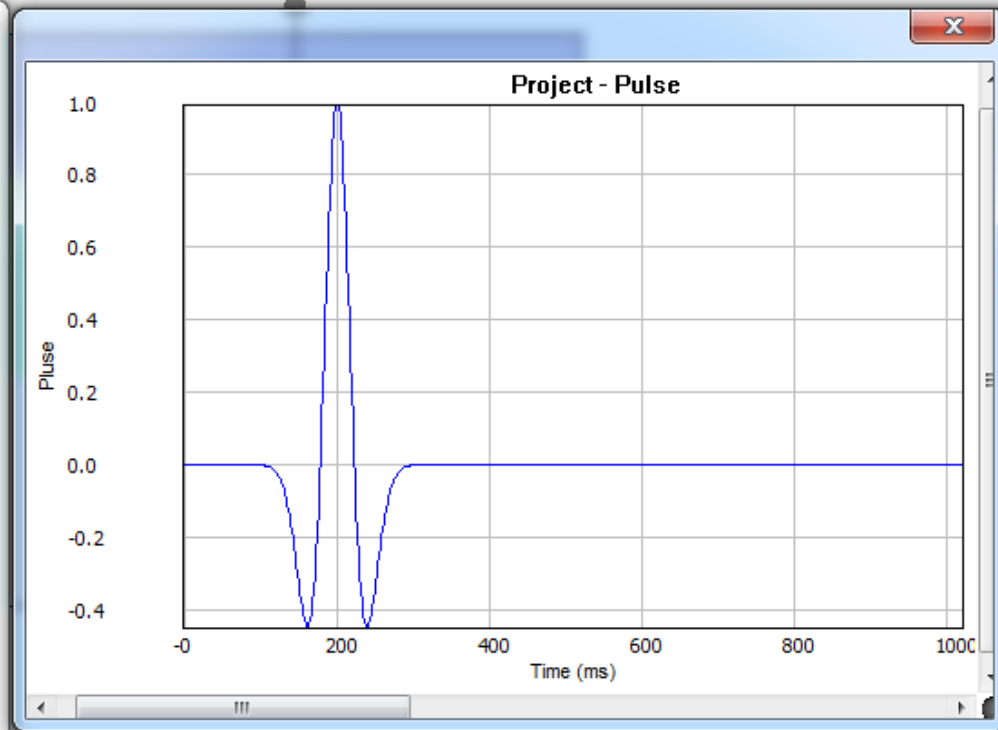
Center Phase [deg] 0

Truncation Multiple 6.2831853

(Note: $f_m=(f_{min}+f_{max})/2$)

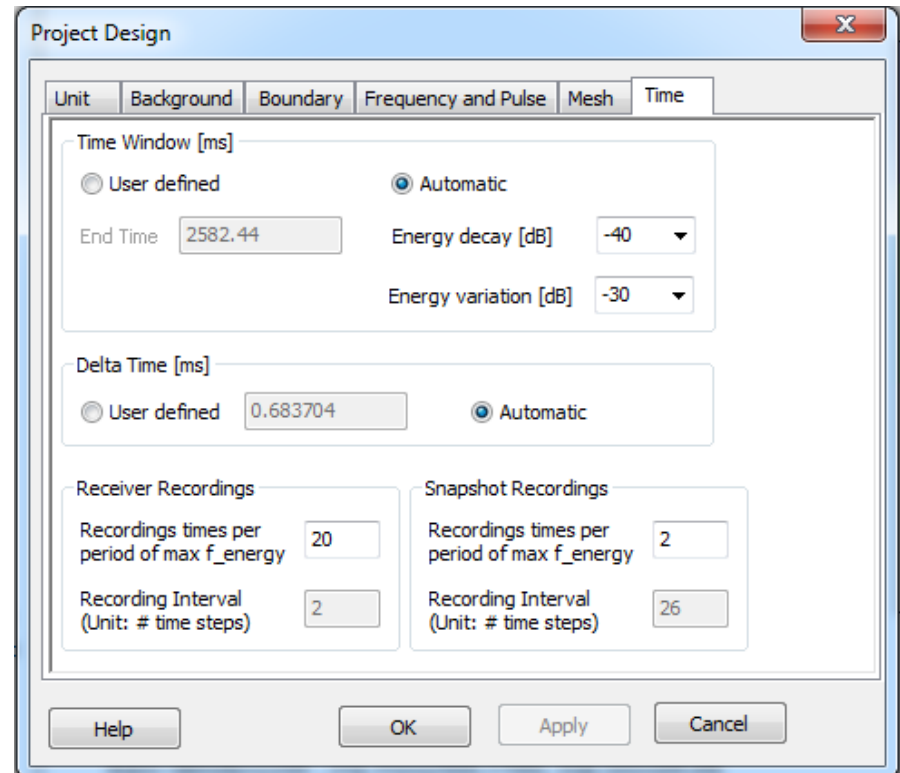
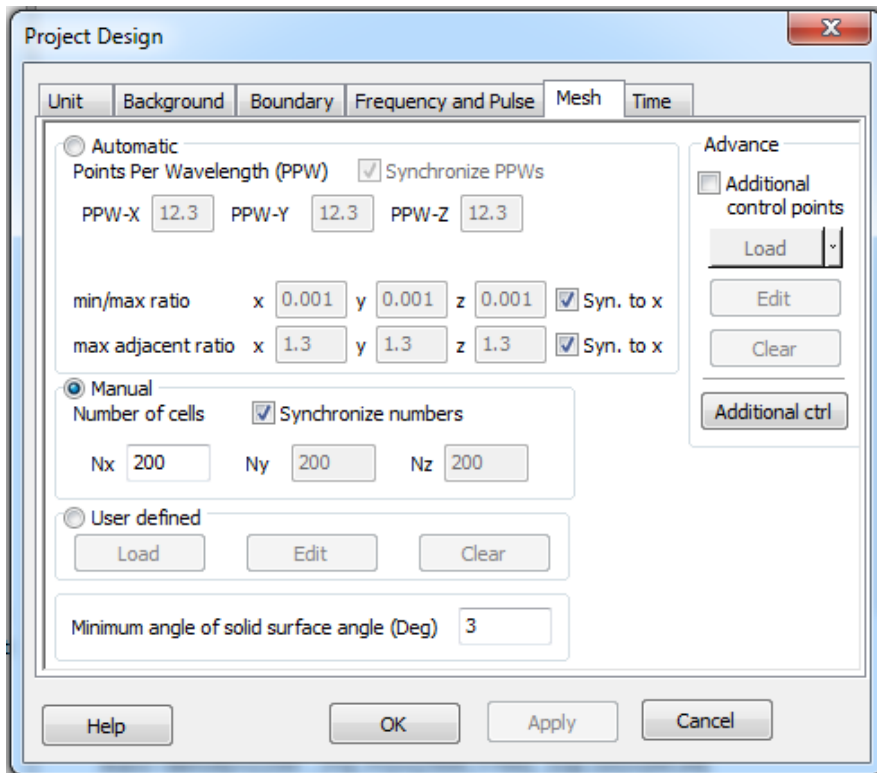
(Note: $t_c=1/(\pi*f_c)$, $f_{max}=2.76375f_c$)

Help OK Apply Cancel

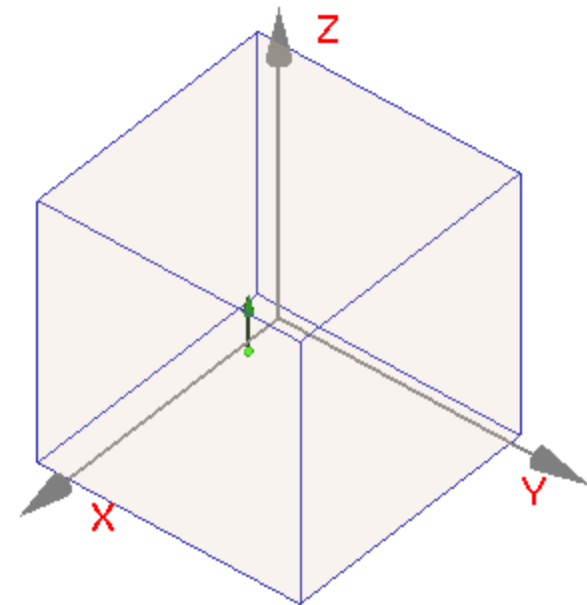
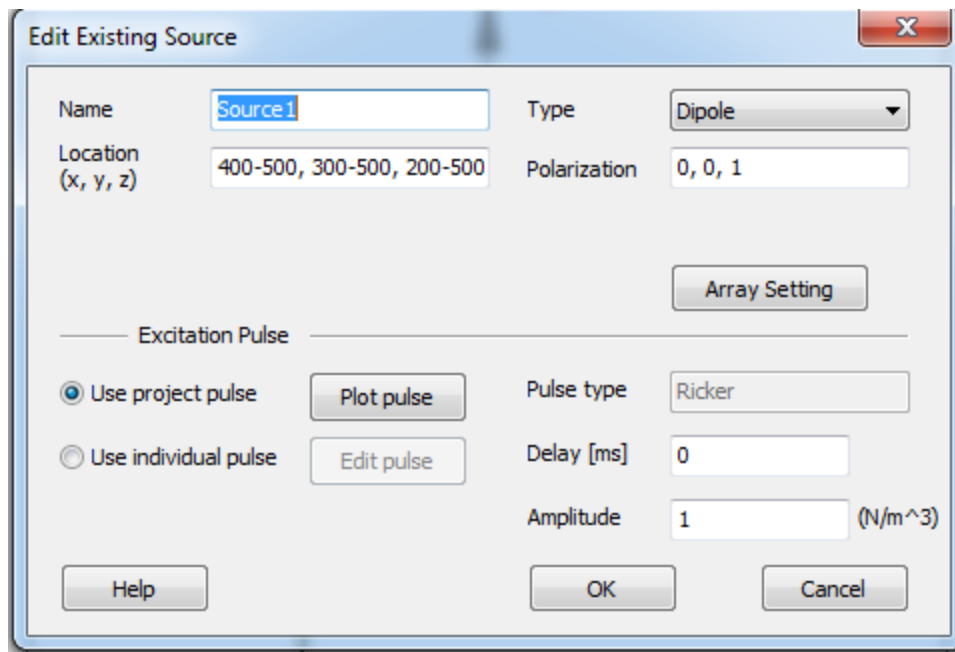


Project mesh: 200 cell in x, y & z

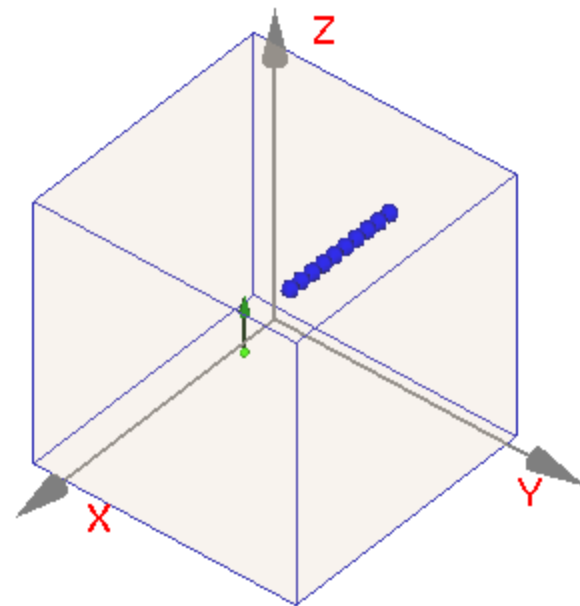
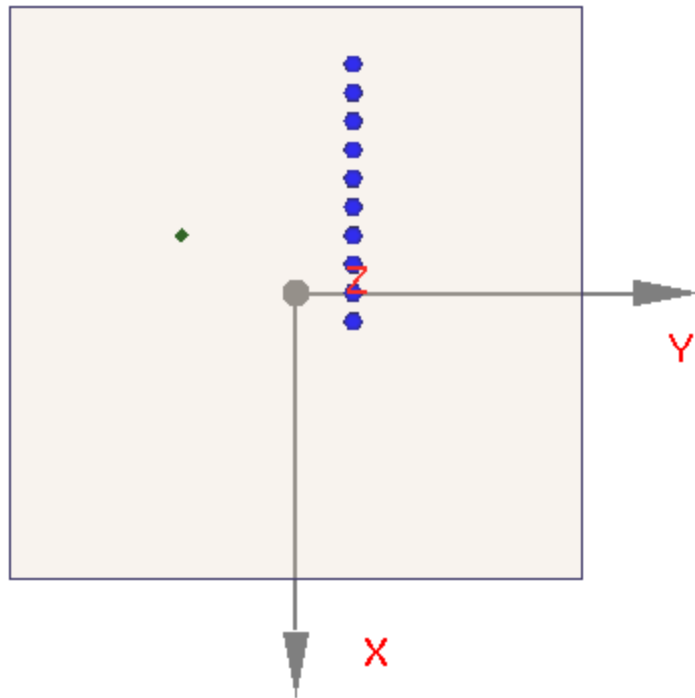
Project time setting: default time window & automatic Δt



Source: Z polarization point dipole source at (-100, -200, -300)



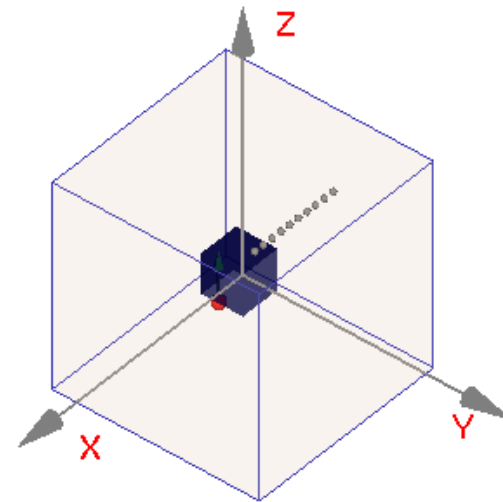
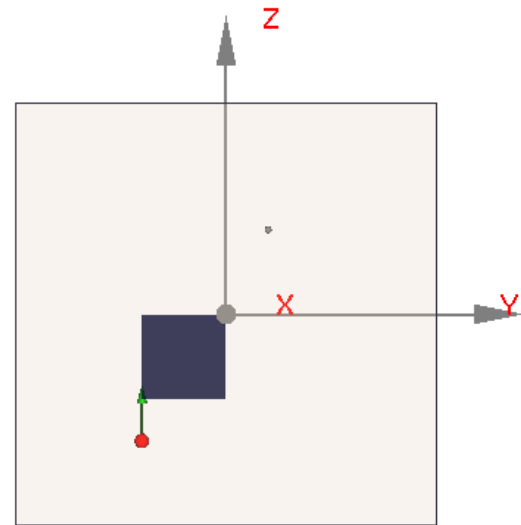
10 observers along x: y=100, z=200; x positions: [-400:50:50]



Box region

As right figure.

Size: $(200,-200,-200) - (0,0,0)$
Material is the one we shown
in [the project material page](#)



Prepare voxel data files

This is a rectangular region with weak anisotropic EL material (VTI material, Thomson type). Here, we need only 3 inputs: epsilon, delta, gamma. The material rotation angle for each cell is bulk value.

Assuming this voxel is composed by: $2 \times 2 \times 2 = 8$ uniform cells (cell size is $100 \times 100 \times 100 \text{ m}^3$), each cell's material is defined by file (ASCII format), we can define 3 files for each input, as following,

epsilon.txt

```
2
2
2

0.2
0.2

0.2
0.2

0.2
0.2

0.2
0.2
```

delta.txt

```
2
2
2

-0.1
-0.1

-0.1
-0.1

-0.1
-0.1

-0.1
-0.1
```

gamma.txt

```
2
2
2

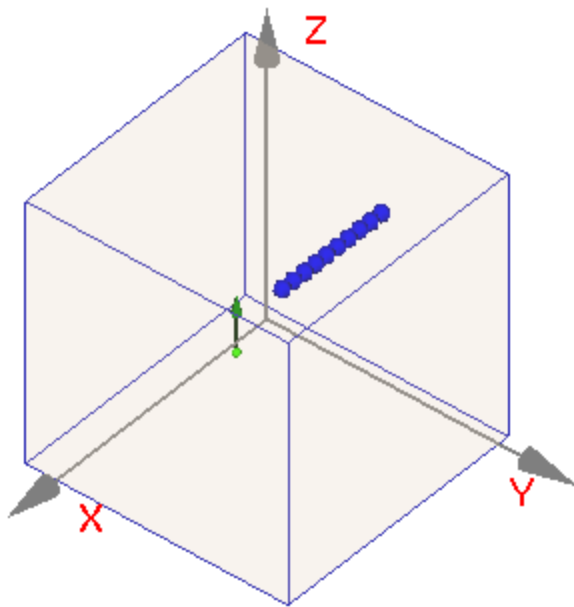
0.5
0.5

0.5
0.5

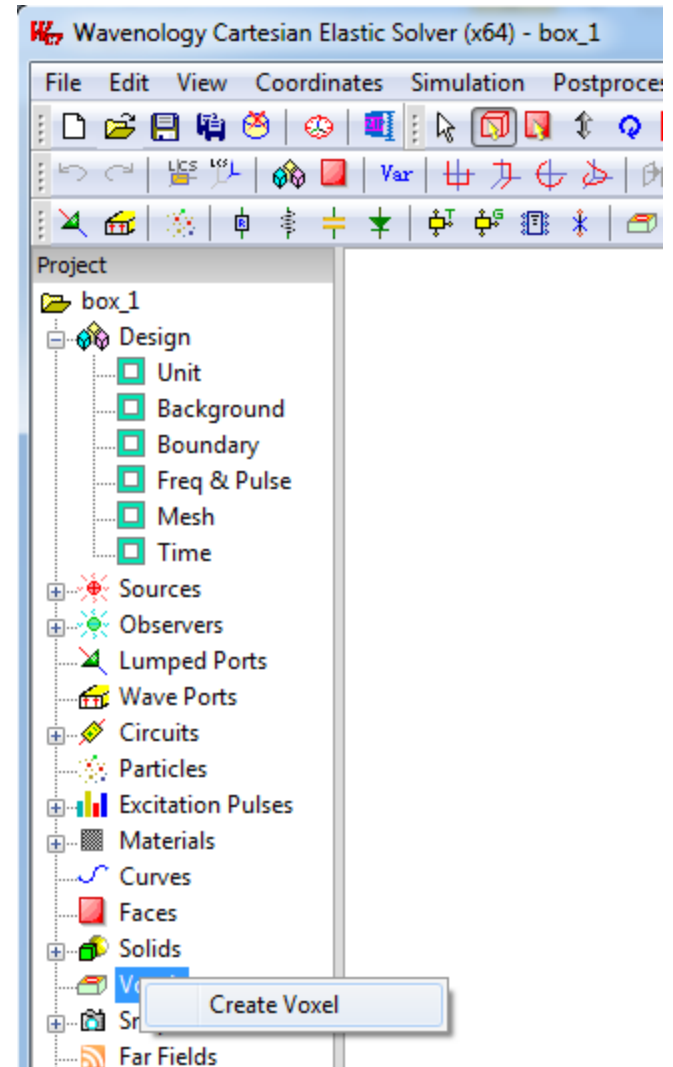
0.5
0.5

0.5
0.5
```

After project setup, source and receiver, voxel data files are ready, we can see the project as following



Then, create voxel



Voxel name

The image shows a 'Voxel Editor' dialog box with several sections and fields:

- General:** Contains a 'Name' field with the value 'Voxel1' and a 'Unit' dropdown menu set to 'meter'. These two fields are enclosed in a blue box.
- Physical Size:** Contains a 'Loaded Volume' section with 'Volume Start Position in computation domain (x,y,z)' set to '0, 0, 0' and 'Resolution (dx,dy,dz)' set to '100, 100, 100'. This entire section is enclosed in an orange box.
- Advance:** Contains radio buttons for 'Use the Whole Volume' (selected) and 'Use sub-volume'. The 'Use sub-volume' section includes 'Corner 1 (x,y,z)' and 'Corner 2 (x,y,z)' input fields.
- Electrical Parameters:** This tab is selected and highlighted with an orange box. It contains four rows of parameters, each with a radio button for 'Bulk' (selected) and a 'Load' button, followed by a 'Mesh size' input field:
 - Relative Permittivity: Bulk 1
 - Elec. Conductivity (S/m): Bulk 0
 - Relative Permeability: Bulk 1
 - Mag. Conductivity (W/m): Bulk 0
- Display:** Contains a 'Data Type' dropdown set to 'Mass', a 'Color Map' dropdown set to 'Hot', and a 'Transparency' slider.
- Buttons:** 'Help', 'OK', and 'Cancel' buttons are located at the bottom.

Voxel unit

Voxel start position & resolution

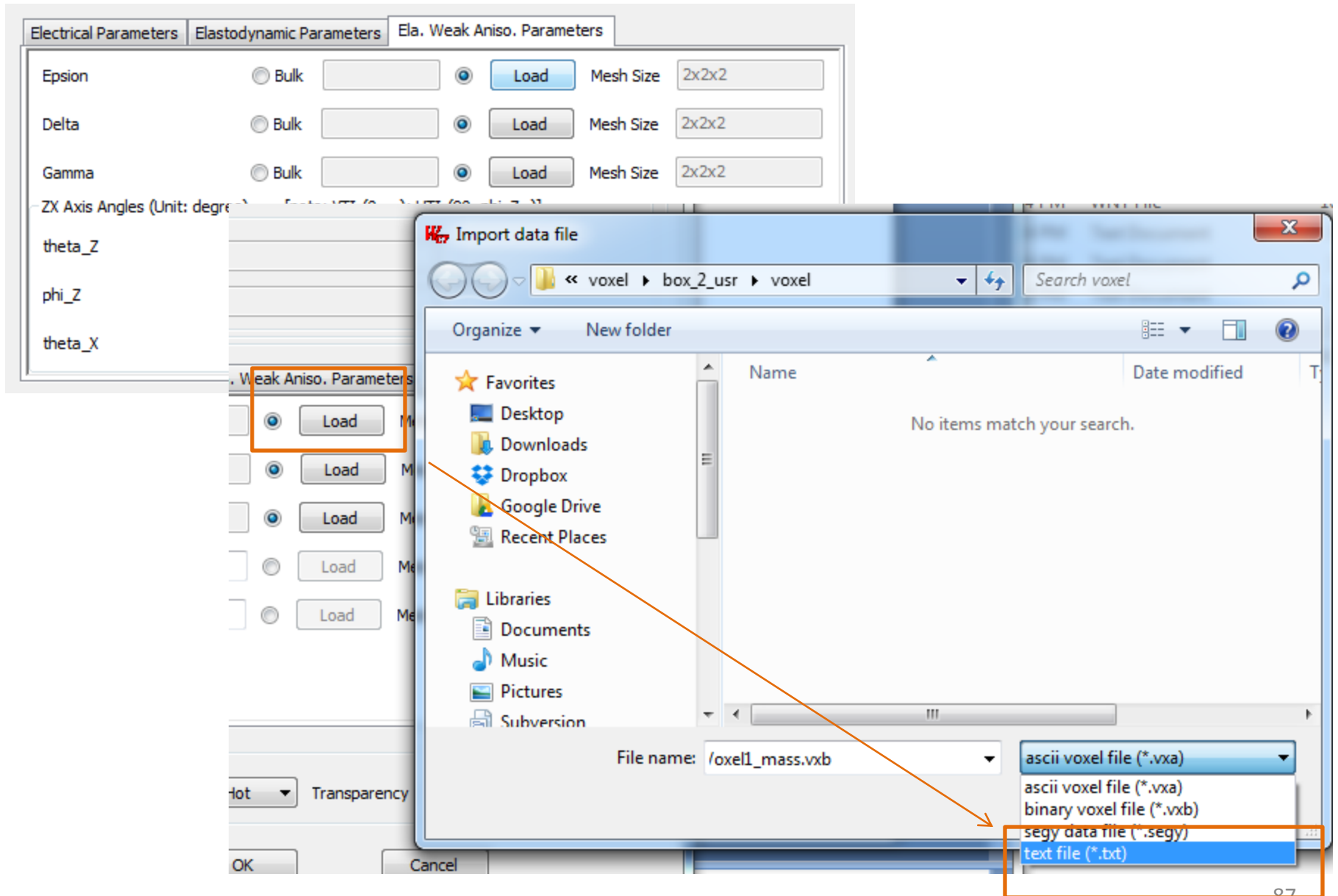
Skip page:
Electrical parameters

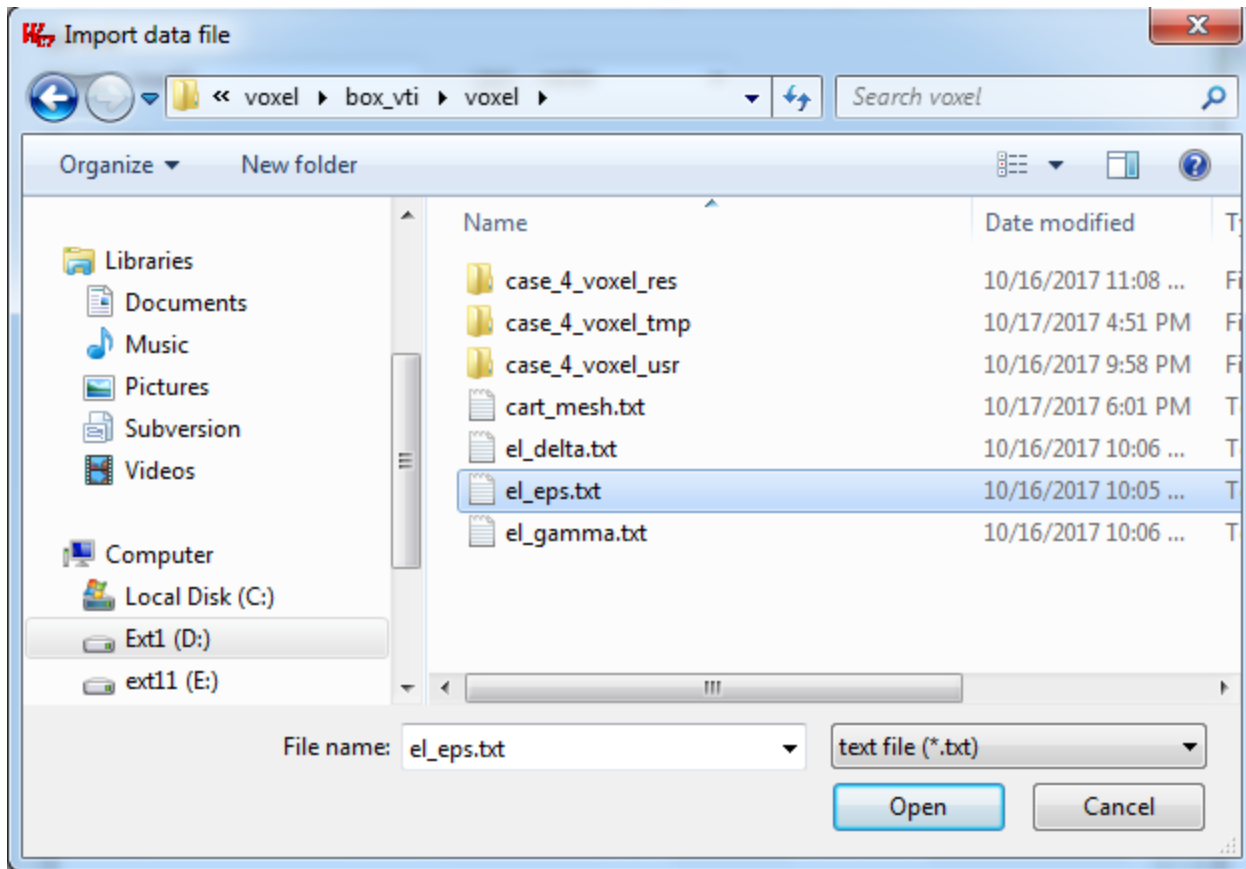
In the Elastodynamic Parameters page, input bulk mass, vp, vs, respectively.

Actually, input for vp & vs can be skipped in this case, because they will not be used in the simulation.

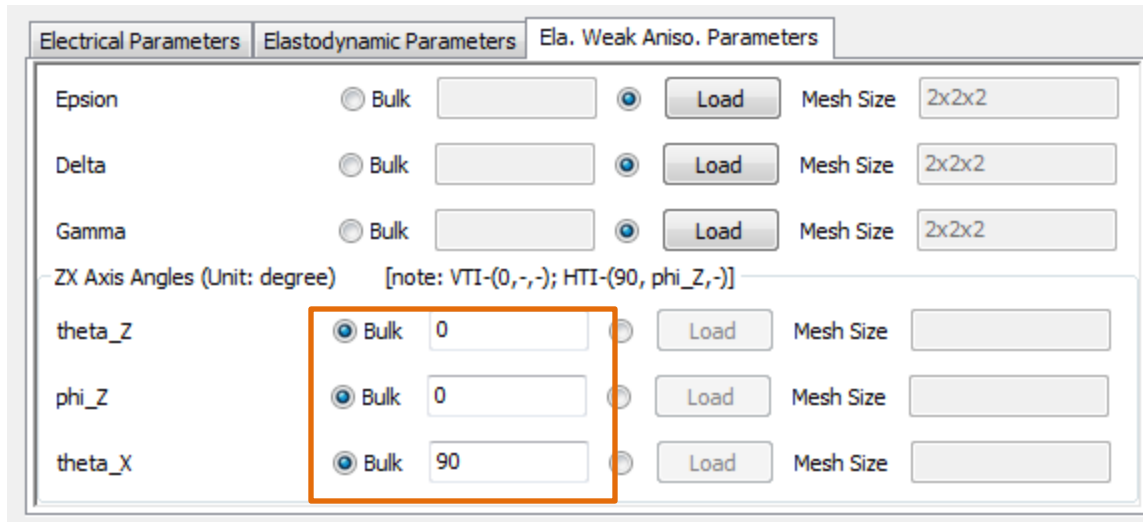
Parameter	Radio Button	Value	Load	Mesh Size
Mass (kg/m ³)	<input checked="" type="radio"/> Bulk	3000	<input type="radio"/> Load	<input type="text"/>
Vp (m/s)	<input checked="" type="radio"/> Bulk	3500	<input type="radio"/> Load	<input type="text"/>
Vs (m/s)	<input checked="" type="radio"/> Bulk	2500	<input type="radio"/> Load	<input type="text"/>
Qp	<input checked="" type="radio"/> Bulk	-1	<input type="radio"/> Load	<input type="text"/>
Qs	<input checked="" type="radio"/> Bulk	-1	<input type="radio"/> Load	<input type="text"/>

In the Weak Aniso. Parameters page, input bulk mass, vp, vs, respectively.

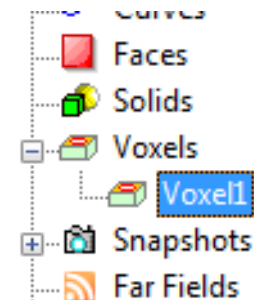




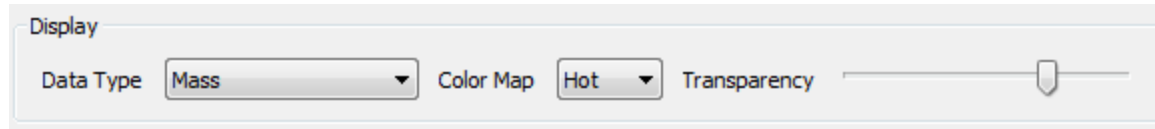
For the material rotation angle, due to it is VTI material, we keep these input as default values: (0, 0, 90°)



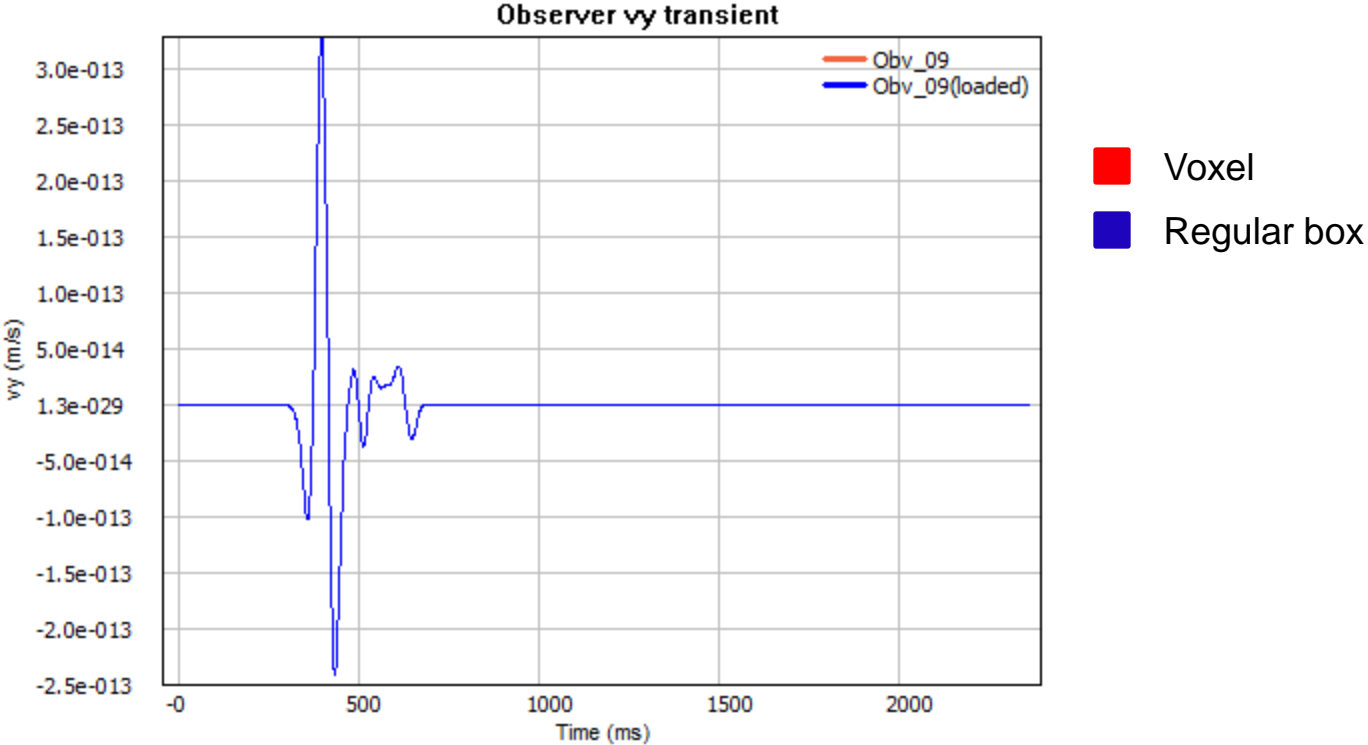
After all voxel parameters are set correctly, then “OK” to add this voxel in the project, as right figure



After data files are defined, let the GUI show the Mass in the voxel region, setup the transparency also, as following.

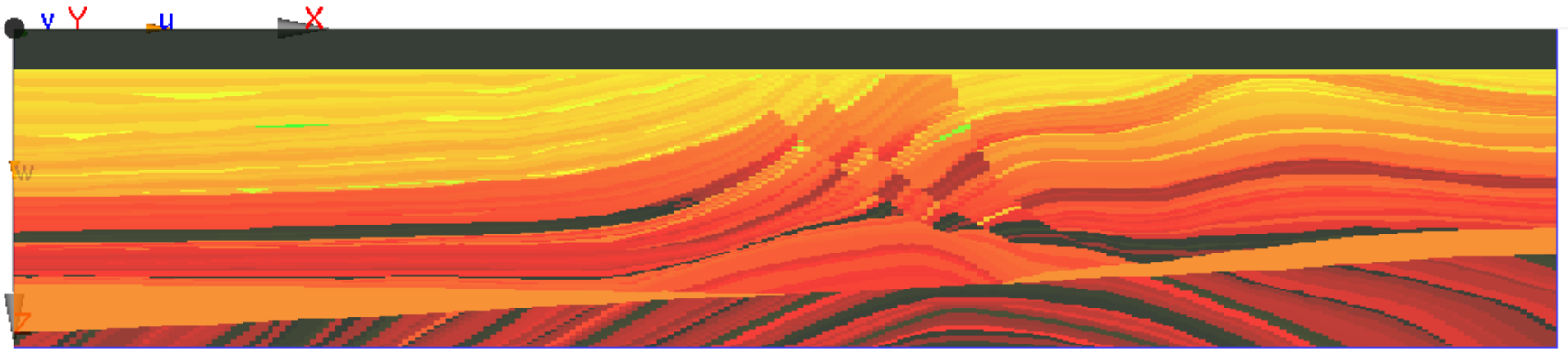


Then we can simulate this project, and compare the simulation result with a regular box setting. We can see two sets of data overlap.



Case (3) Simulation on 2D Marmousi II model

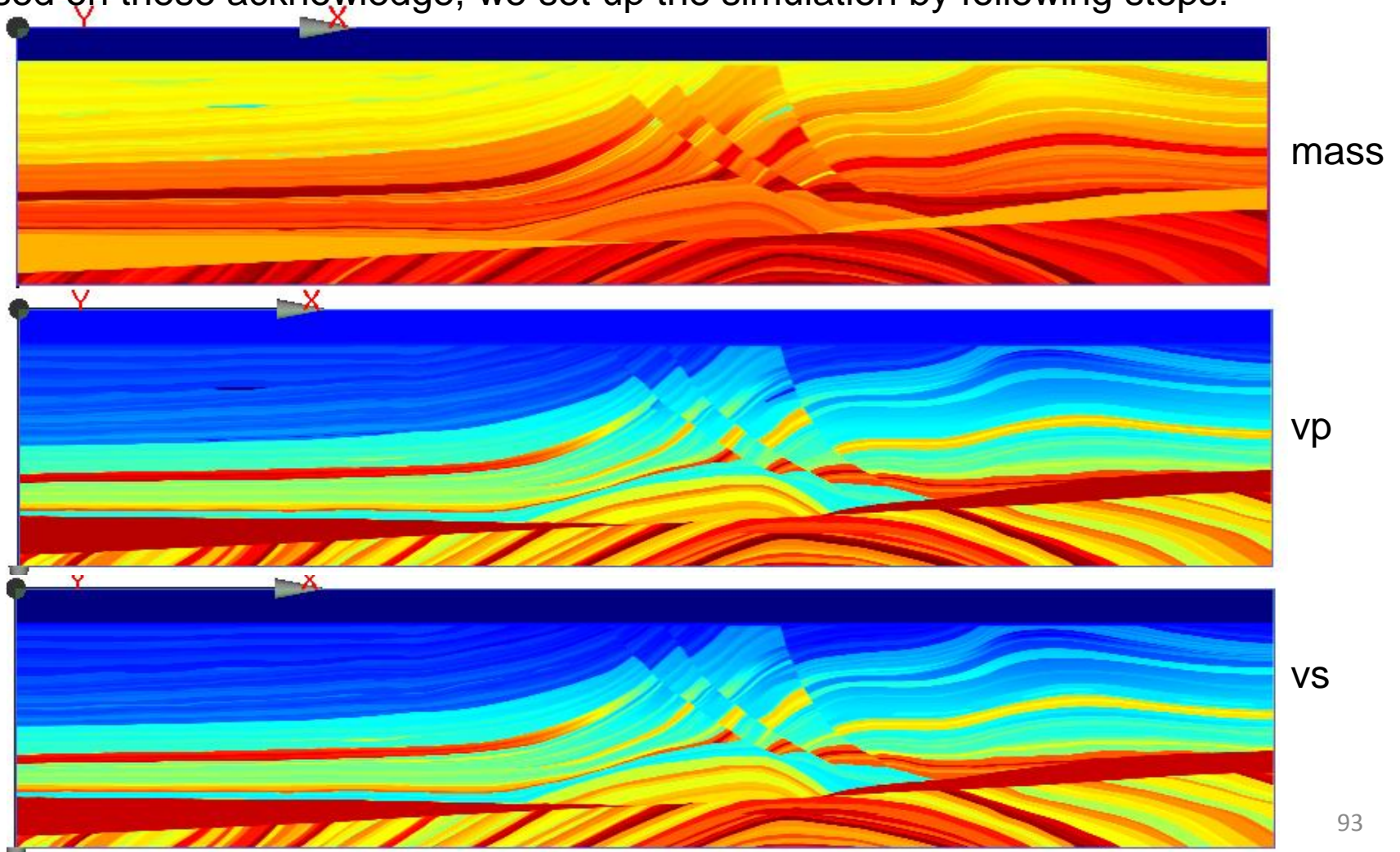
This case simulates the Marmousi ii model, to show how the wave propagate in the computational domain by snapshot



Marmousi ii dataset

The Marmousi ii dataset used in the simulation include mass, vp and vs properties. All are imported from segy files.

The resolution of dataset is 1.25m in x, y and z. The cell number is 13601x1x2801. Based on these acknowledge, we set up the simulation by following steps.

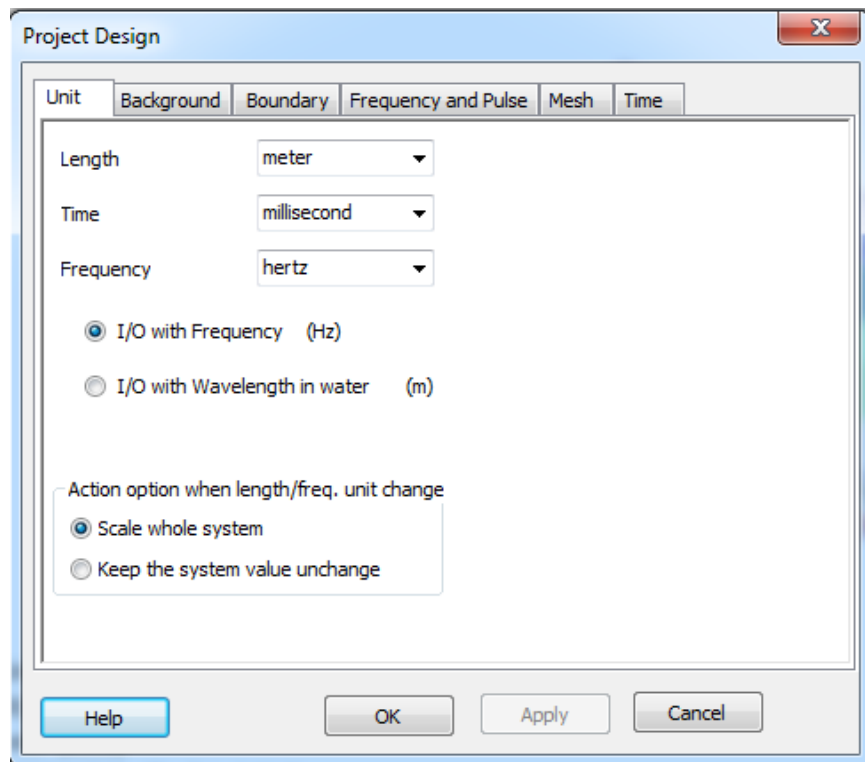


Case setup: Materials

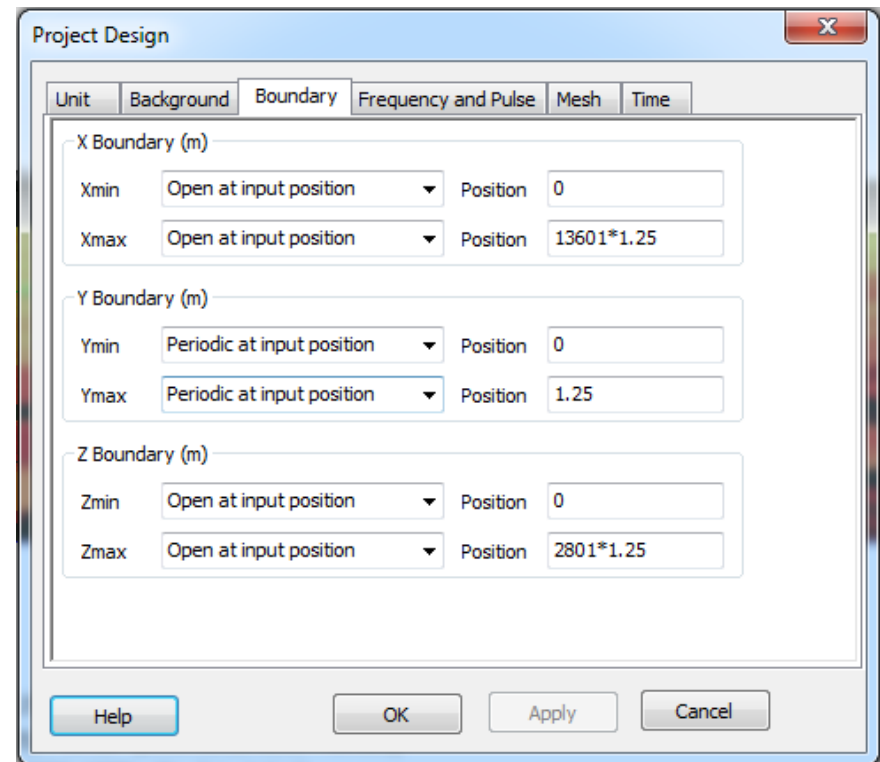
In this simulation, the whole computational domain will be occupied by Marmousi II model. So, we don't need to setup any addition material, just let the project use the default material.

Case setup

Project setting: unit

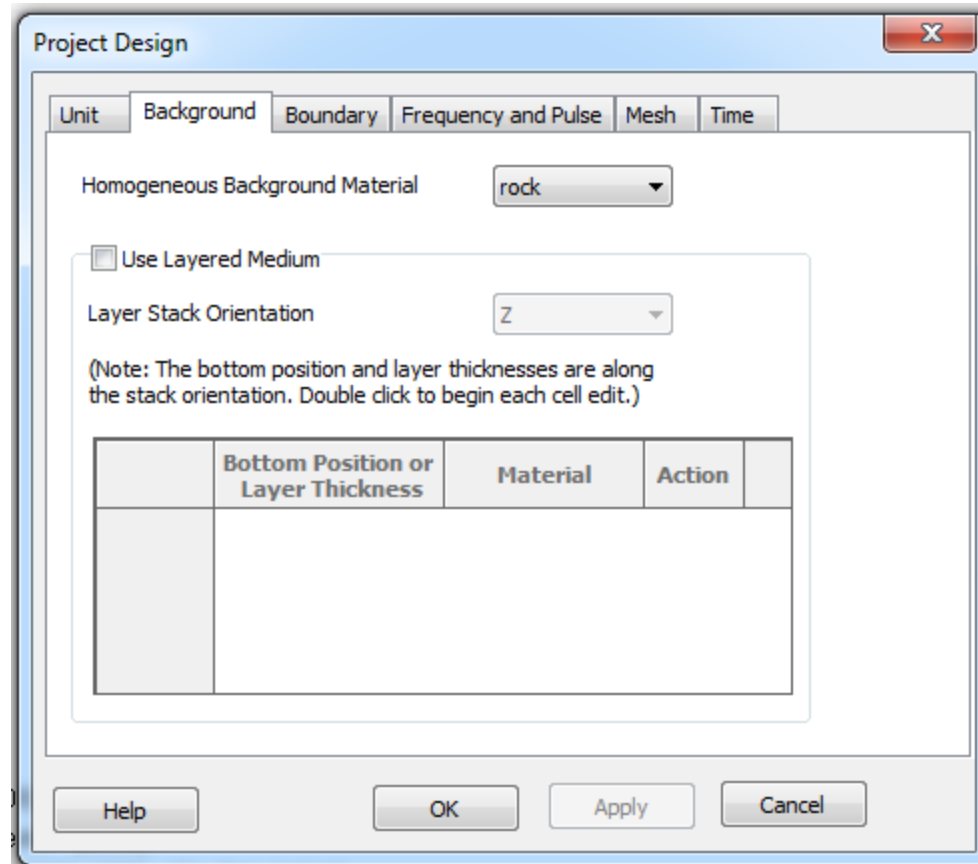


Project size & B.C.

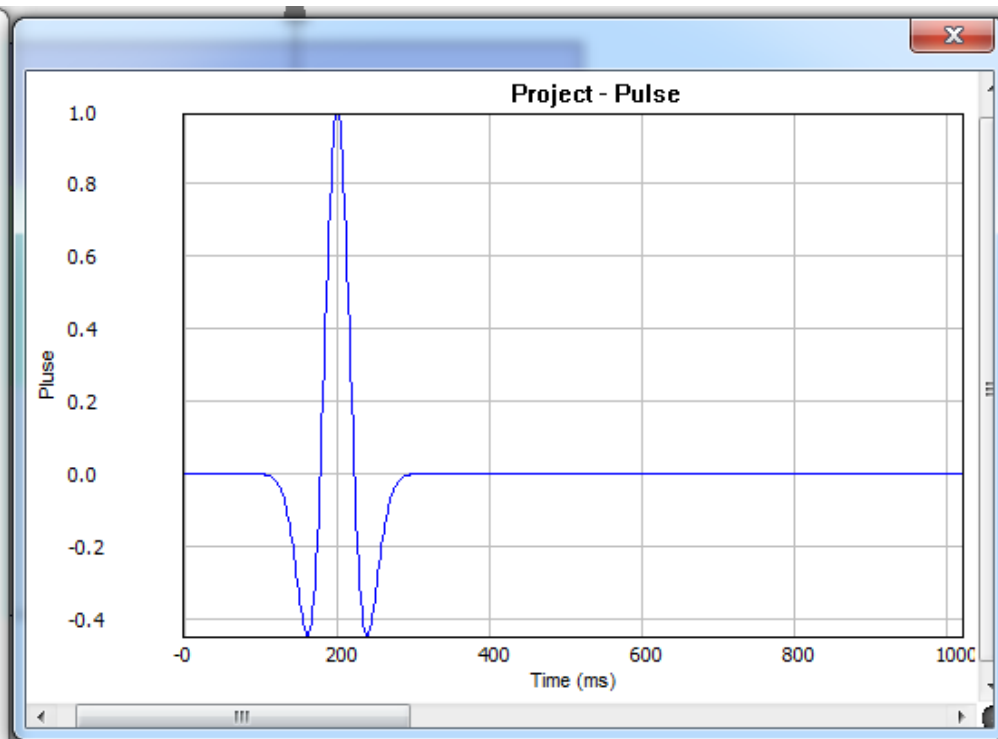
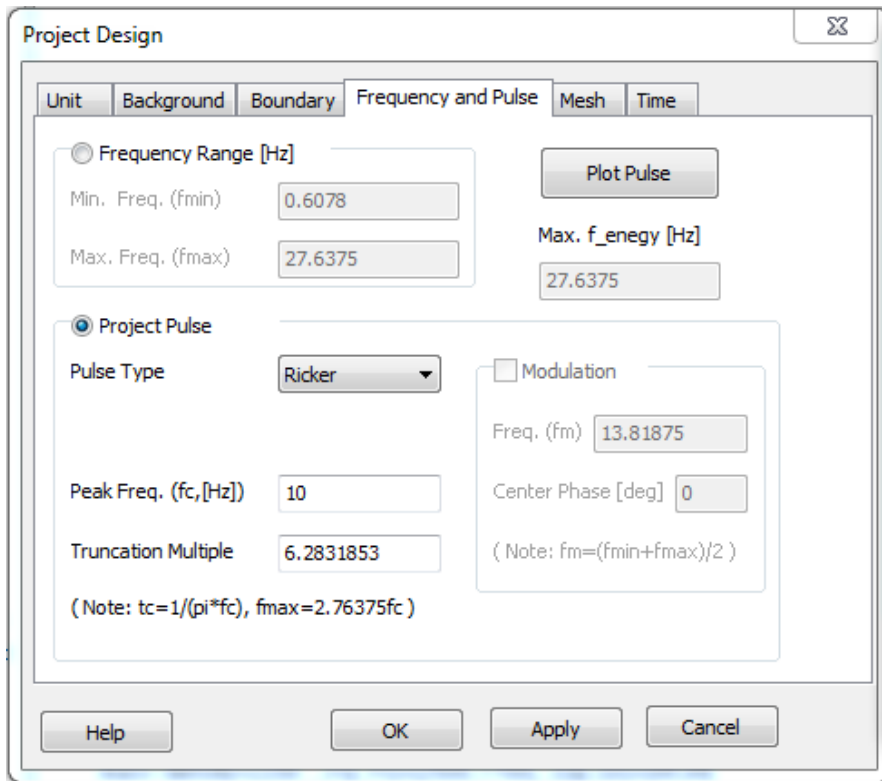


Set project size as: (13601*1.25,
1.25, 2801*1.25)

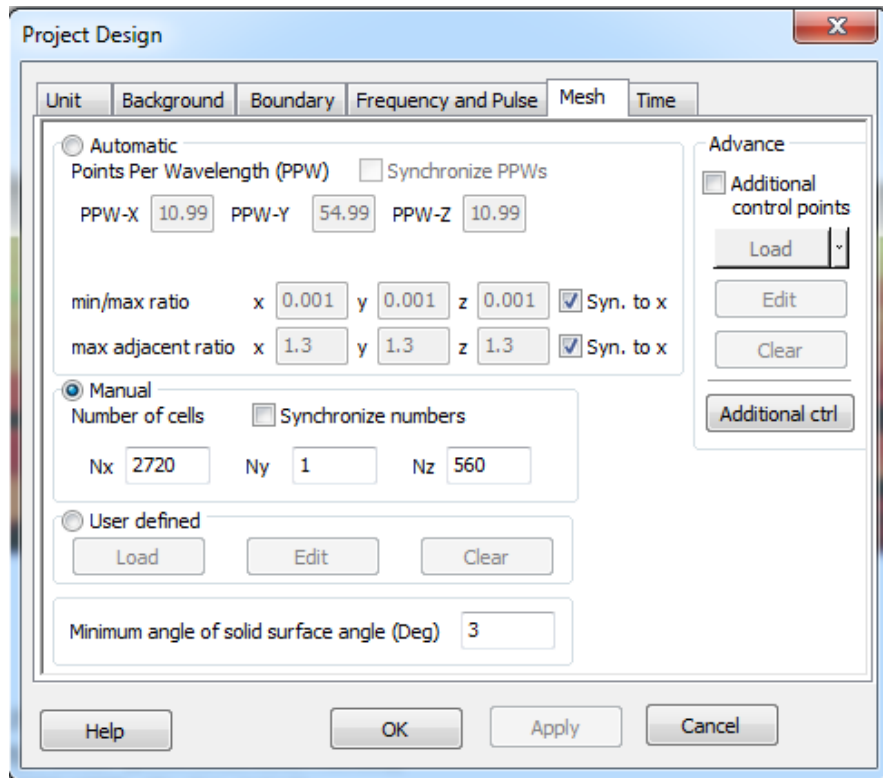
Project background



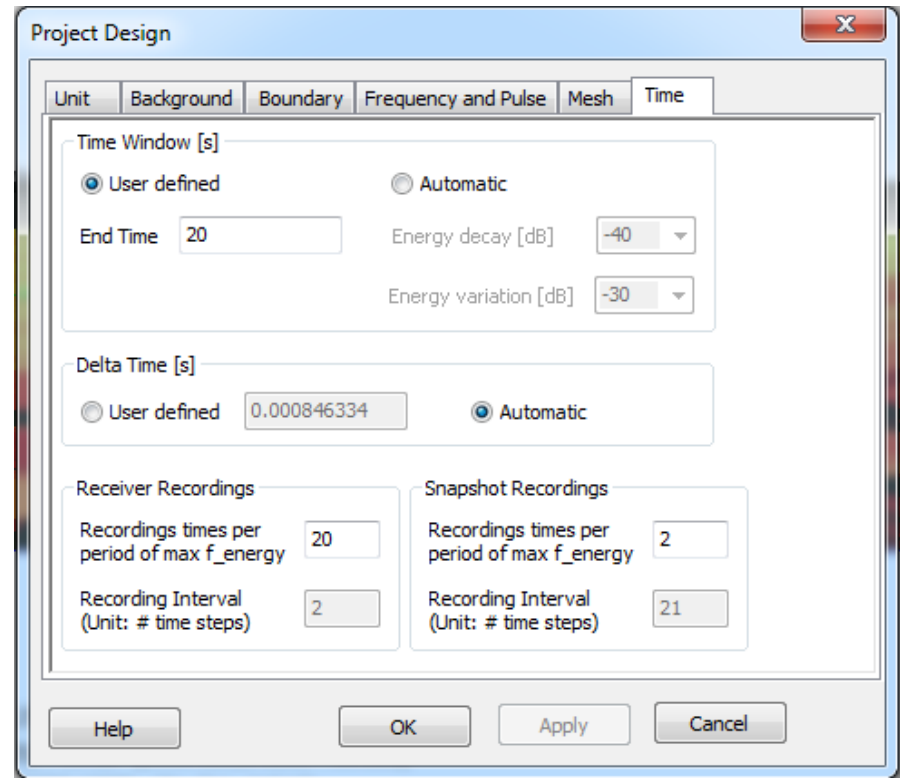
Project pulse: $f_c=10$ Hz Ricker wave



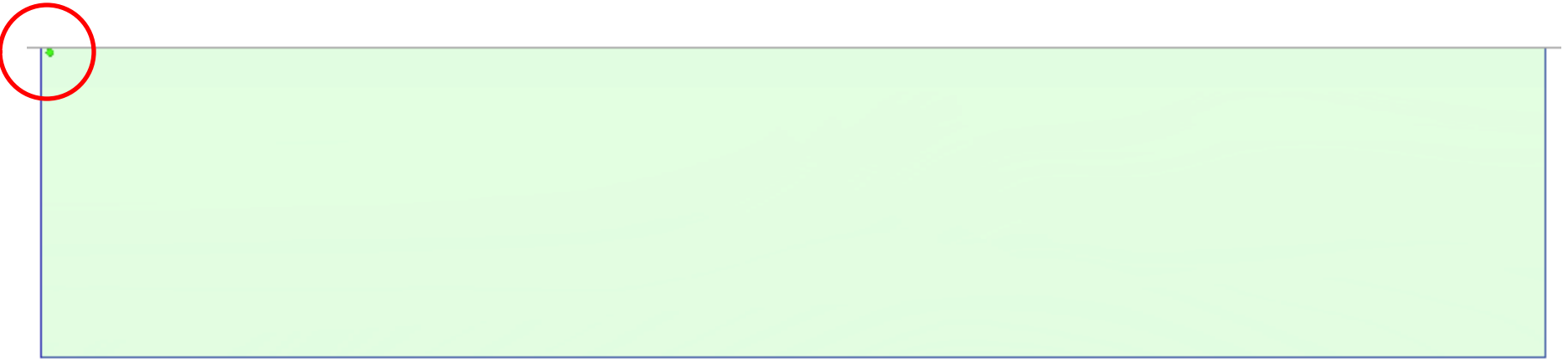
Project mesh: 2720x1x560, so, we can make sure the FDTD sampling density is bigger than 10 points per wave length



Project time setting: 20s time window & automatic Δt . Snapshot recording interval is 2 points per period.

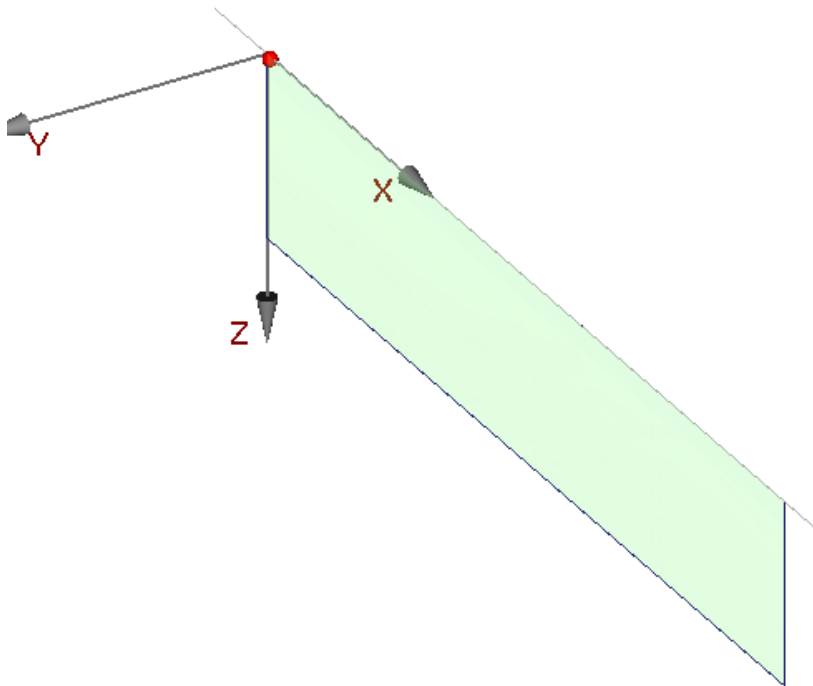


Source: point monopole source at (100, 0.625, 50)

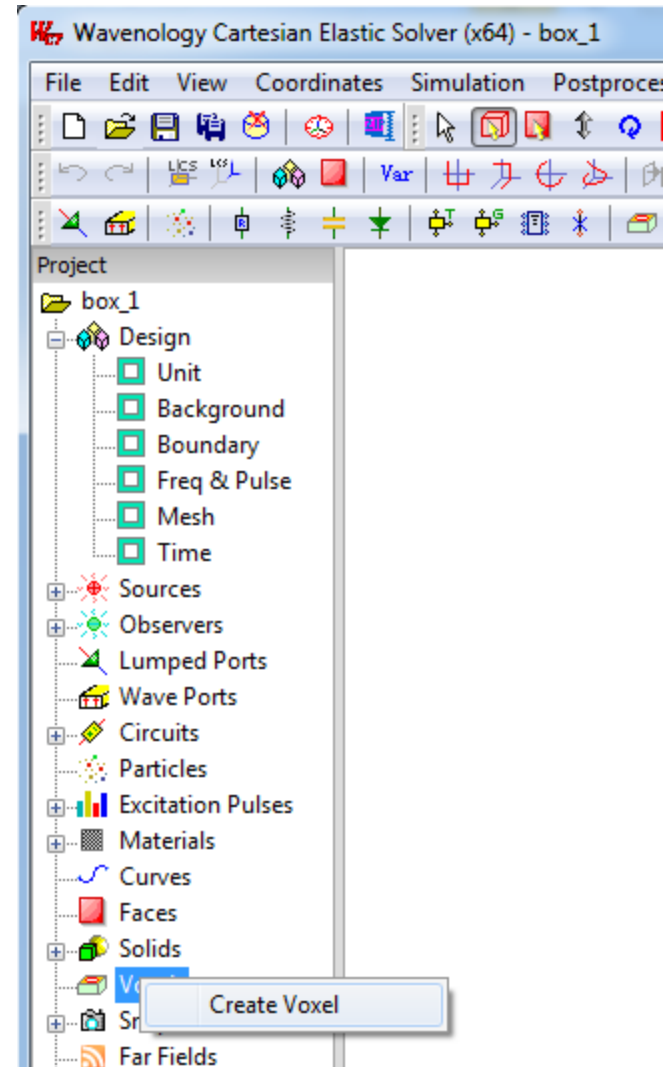


There is not receiver in this case, we will use snapshot to check the field propagation.

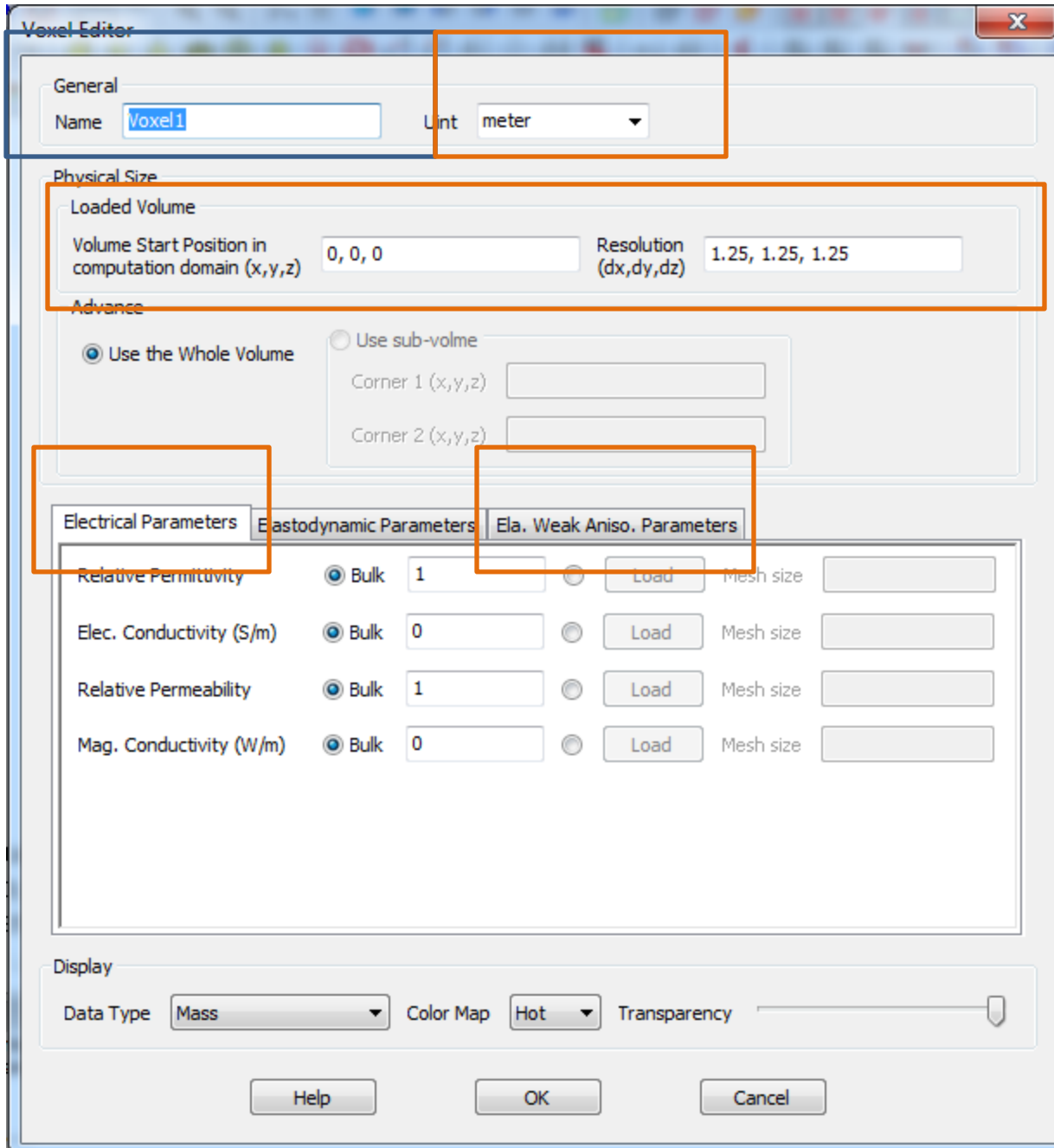
After project setup, source and voxel data files are ready, we can see the project as following



Then, create voxel



Voxel name



Voxel unit

Voxel start position & resolution

Skip pages:
Electrical parameters & Ela. Weak Aniso. Parameters

In the Elastodynamic Parameters page, load mass, vp, vs from SEGY data files, respectively.

The image shows a software interface with two main components. The top component is the 'Elastodynamic Parameters' page, which has three tabs: 'Electrical Parameters', 'Elastodynamic Parameters', and 'Ela. Weak Aniso. Parameters'. The 'Elastodynamic Parameters' tab is active. It contains several rows of parameters, each with a radio button for 'Bulk' and a 'Load' button. The parameters are: Mass (kg/m³) with a value of 1000, Vp (m/s) with a value of 1500, Vs (m/s) with a value of 0, Qp with a value of -1, and Qs with a value of -1. Each parameter also has a 'Mesh Size' field. The 'Load' buttons for Mass, Vp, and Vs are highlighted with a blue box. An orange arrow points from this box to the 'Import data file' dialog box below. The dialog box shows a file explorer view of a folder named 'MODEL_DENSITY_1.25m.se...'. It contains a single file named 'MODEL_DENSITY_1.25m.segy' with a date modified of '6/29/2005 8:17 PM'. The 'File name' field is empty, and a dropdown menu is open, showing file type options: 'segy data file (*.segy)', 'ascii voxel file (*.vxa)', 'binary voxel file (*.vxb)', 'segy data file (*.segy)', and 'text file (*.txt)'. The 'segy data file (*.segy)' option is selected and highlighted.

Electrical Parameters Elastodynamic Parameters Ela. Weak Aniso. Parameters

Mass (kg/m³) Bulk 1000 Load Mesh Size 13601x1x2801

Vp (m/s) Bulk 1500 Load Mesh Size 13601x1x2801

Vs (m/s) Bulk 0 Load Mesh Size 13601x1x2801

Qp Bulk -1 Load Mesh Size

Qs Bulk -1 Load Mesh Size

Import data file

2D MODEL_DENSITY_1.25m.se... Search MODEL_DENSITY_1.25...

Organize New folder

Libraries

- Documents
- Music
- Pictures
- Subversion
- Videos

Computer

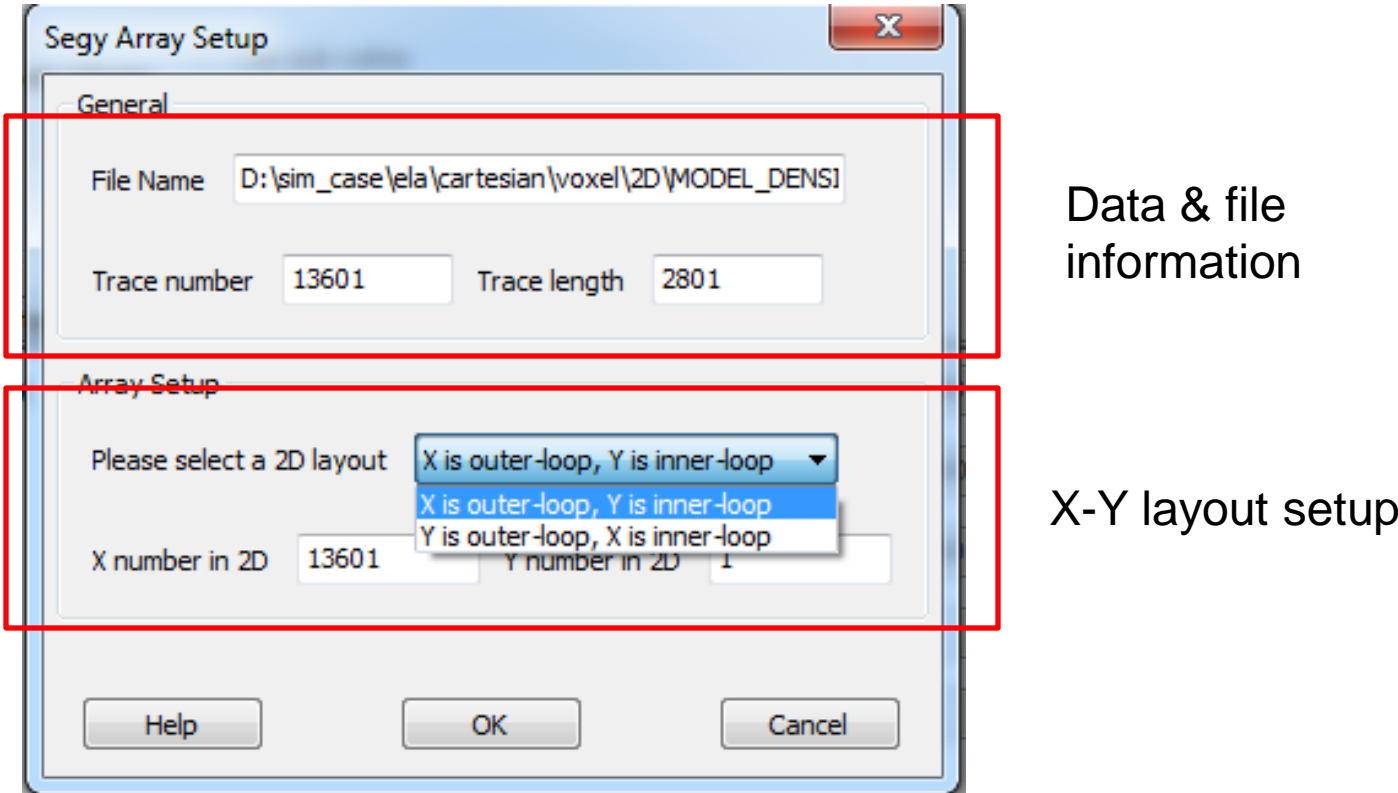
- Local Disk (C:)
- Ext1 (D:)
- ext11 (E:)

Name	Date modified	Type
MODEL_DENSITY_1.25m.segy	6/29/2005 8:17 PM	SI

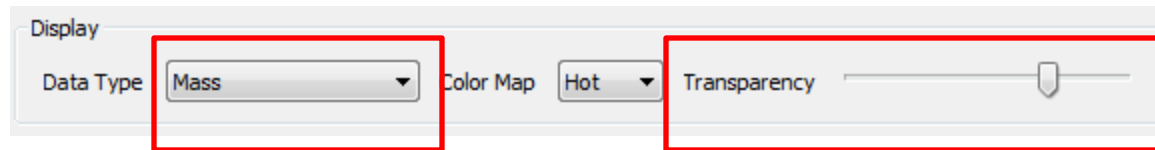
File name: segy data file (*.segy) ascii voxel file (*.vxa) binary voxel file (*.vxb) segy data file (*.segy) text file (*.txt)

In a SEG-Y data file, there are N traces with a length of M . We will treat M as the number of cell in Z direction. N traces will be with a 2D layout in X - Y plane as $nX \times nY$ traces.

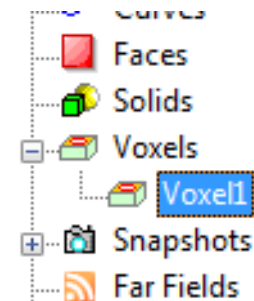
User need to define how to organize the X - Y layout.



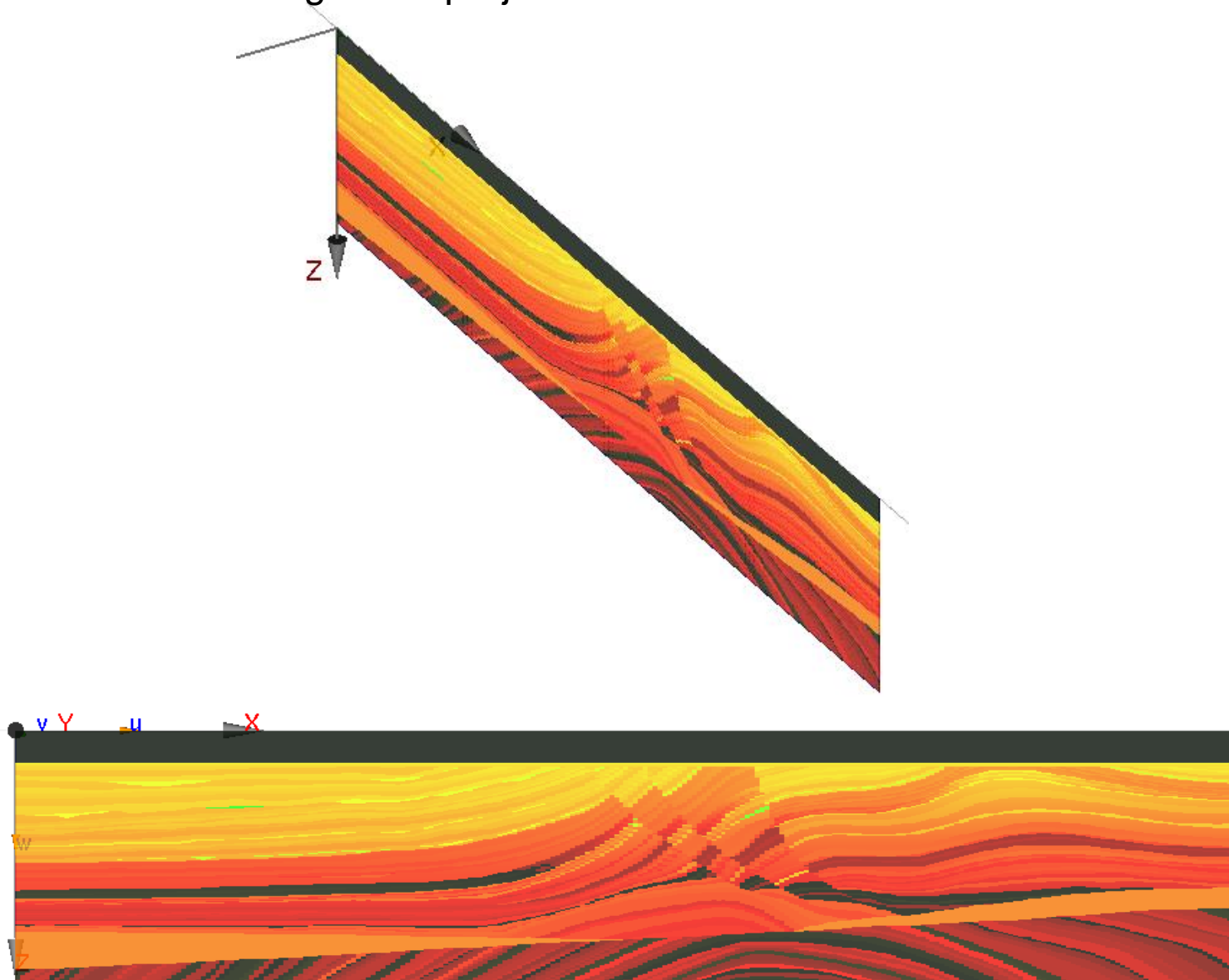
After data files are defined, let the GUI show the Mass in the voxel region, setup the transparency also, as following.



After all voxel parameters are set correctly, then “OK” to add this voxel in the project, as right figure

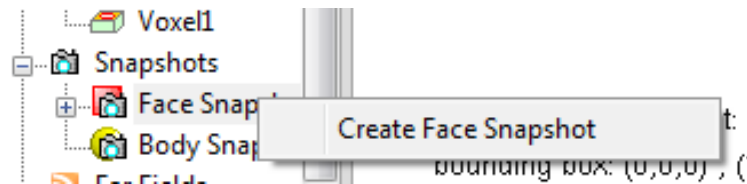


Following is the project in GUI after voxel is set.

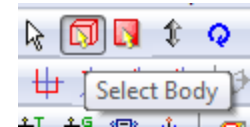


Then, set up a 2D snapshot to capture the field

Right click on treenode: “Snapshot->Facesnapshot” to
“Create Face Snapshot”

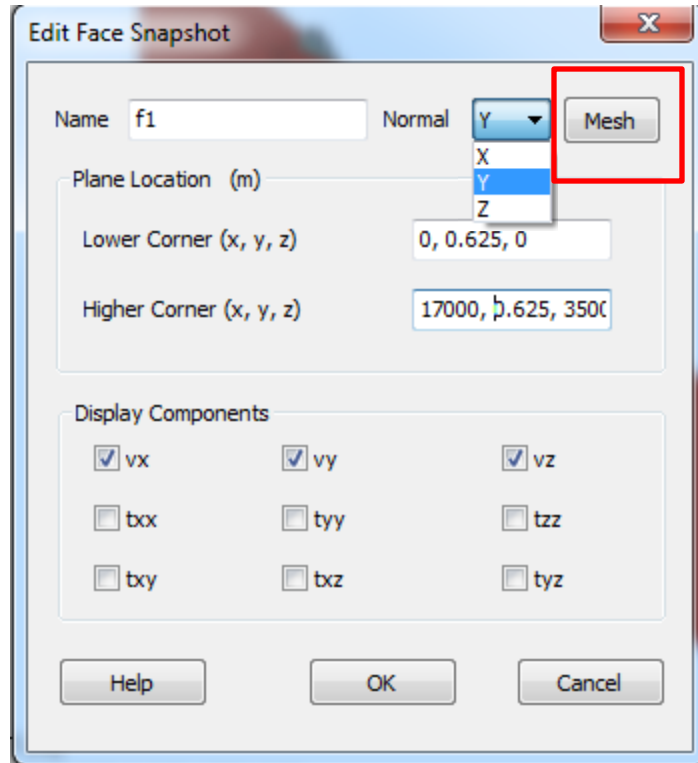


If you find the menu “Create Face Snapshot” is disabled (gray), please make sure the GUI is in the “Select Body” mode.

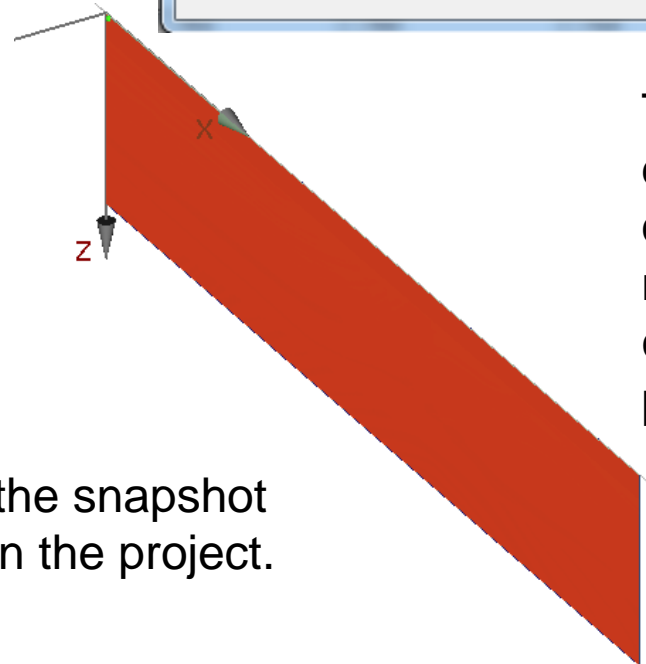
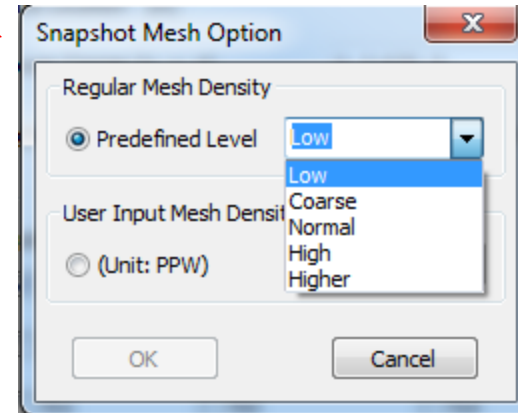


And then click on 3D canvas once, then popup this “Create Face Snapshot” menu again, it will be enabled.

Set up this snapshot in the Y center of domain, and capture all velocity components



Due to this snapshot covers the whole computational domain, the data in snapshot will be very large, we need to use **low** quality discretization for this snapshot



The default discretization quality is “**Normal**”, means that each cell has one sample point.

This is the snapshot region in the project.

Then we can simulate this project.
Following is the simulation result in the snapshot.

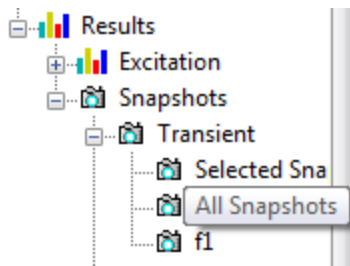
frame: 60



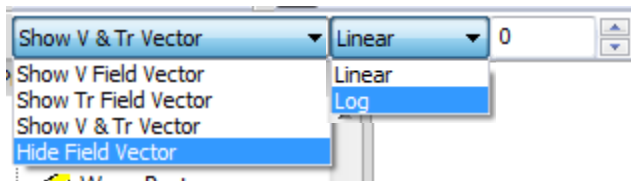
frame: 120



frame: 180

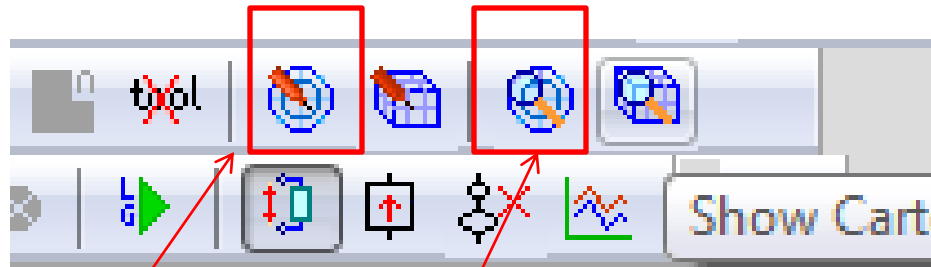


Hide field vector & use **log** mode



How to set up the displaying control, please refer to [here](#).

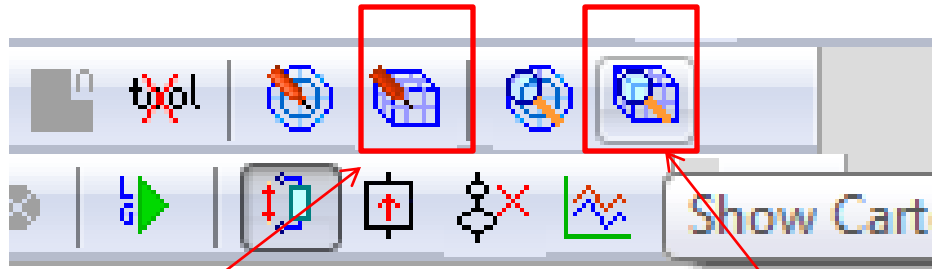
Generate Mesh Files from Wavenology Mesher



Generate
Cylindrical
mesh

View Cylindrical
mesh

Note: for Cylindrical mesh, please refer to Wavenology BHA manual



Generate Cartesian mesh for **whole simulation space**, data files as following:

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

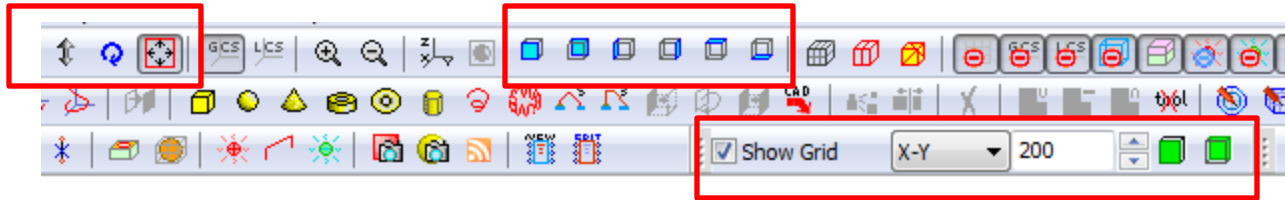
View Cartesian mesh

It will enter a 3D mesh view mode, which provides a functionality to export 2D/3D sub-volume mesh from the whole mesh

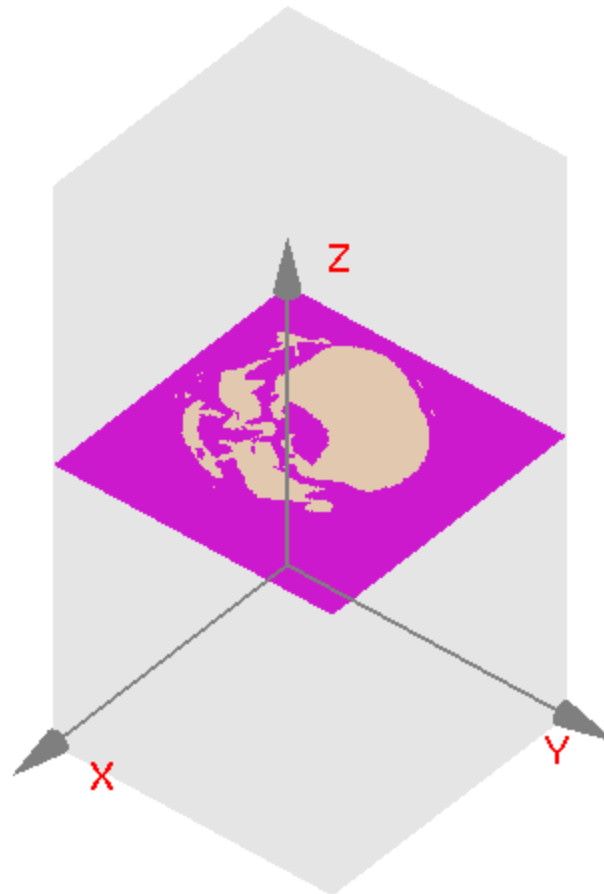
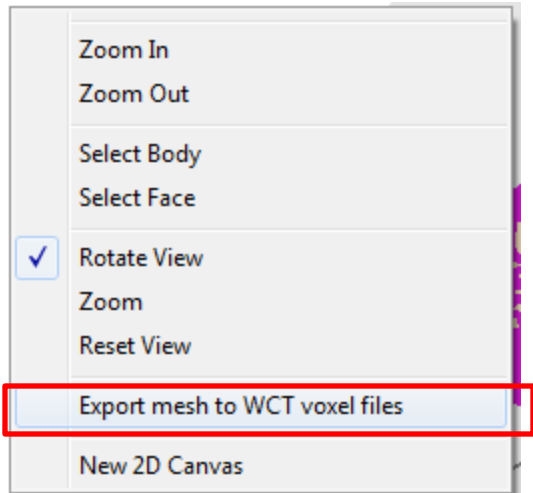
In the 3D mesh view mode

Mouse mode

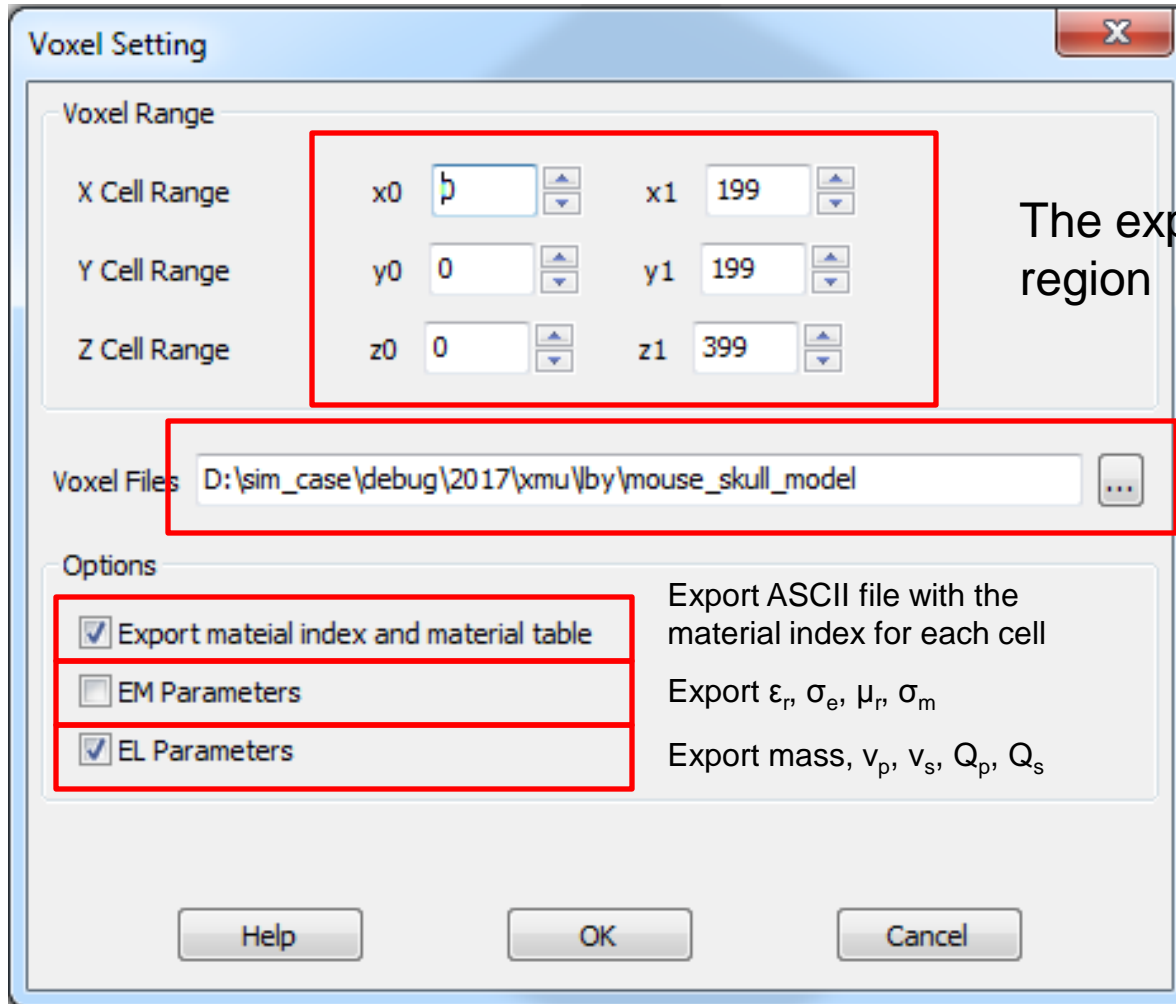
View angle



Cross-section control



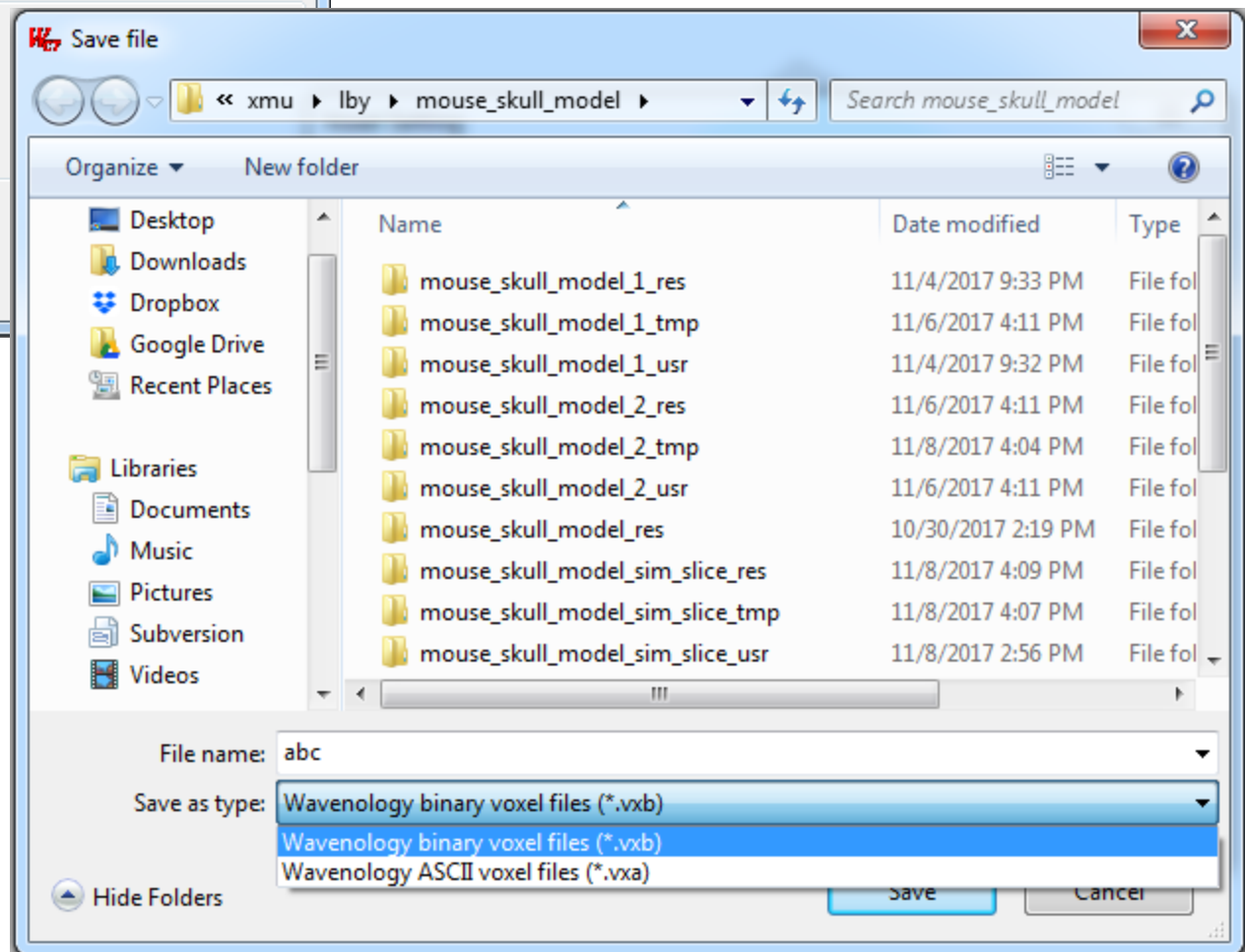
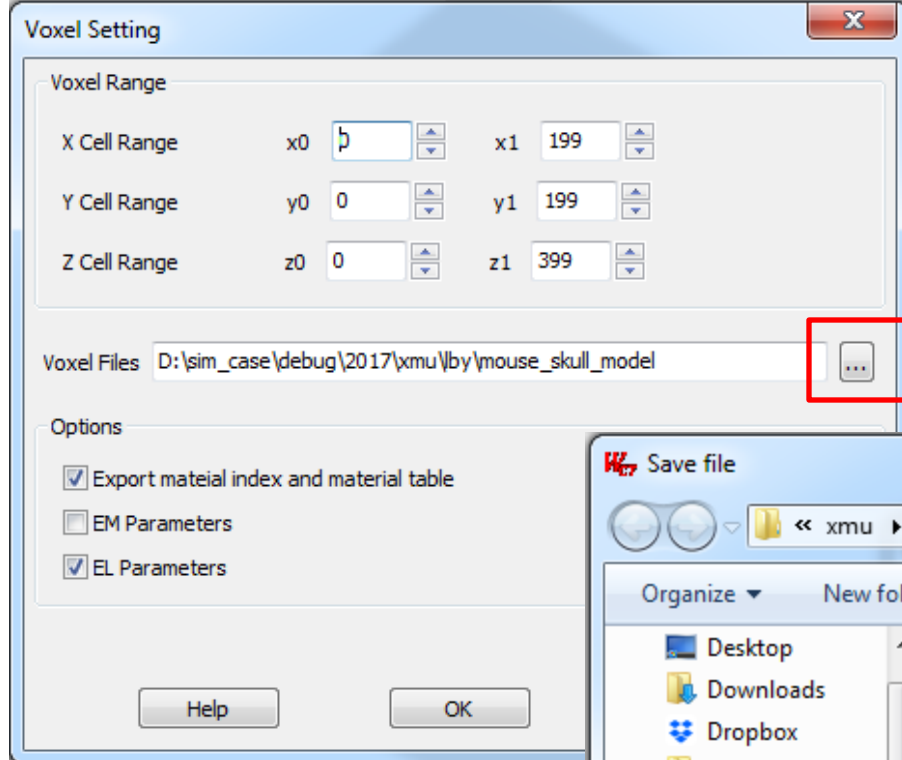
Right click on this canvas, a menu with **“Export mesh to WCT voxel files”** item will appear



The exported region

The target folder with the root of file name for exported files.

The voxel data file will be “vxb” or “vxa” only



Data Files and Format

Export material index and material table

- The ASCII files with the material index for each cell
 - this option will export 2 files
 1. xxx_lmat.txt : mesh file
 2. xxx_material.txt : a table with material parameters

An example of “xxx_material.txt”

Note: the index start from 0

% id	name	eps_r	e_cond	mu_r	m_cond	mass_density	Vp	Vs	
0	Water		78	0	1	0	1000	1500	0
1	Steel		1	0	1000	0	7890	5790	3235
2	Bone		1	0	1	0	1700	4800	1400

An example of “xxx_lmat.txt”

```
% Wavenology lmat file, version 1.0
% nx
20
% ny
10
% nz
6
% material index for each cell, for-loop as:
% for x
%   for y
%     for z
0
0
0
0
0
0
0
0
0
0
0
0
0
0
0
0
```

Note: the material index start from 0

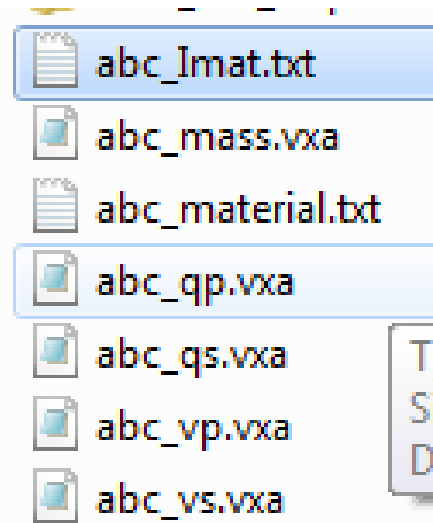
EM Parameters

- Export 4 data files for EM parameters
 1. xxx_eps_r.vxb/vxa : voxel data file for relative permittivity
 2. xxx_sigma_e.vxb/vxa : voxel data file for electrical conductivity
 3. xxx_mu_r.vxb/vxa : voxel data file for relative permeability
 4. xxx_sigma_m.vxb/vxa : voxel data file for magnetic conductivity

EL Parameters

- Export 5 data files for EL parameters
 1. xxx_massvxb/vxa : voxel data file for mass
 2. xxx_vp.vxb/vxa : voxel data file for vp
 3. xxx_vs.vxb/vxa : voxel data file for vs
 4. xxx_qp.vxb/vxa : voxel data file for qp
 5. xxx_qs.vxb/vxa : voxel data file for qs

Following is an example of exported data files



Note for mesh data files

The exported data files only include the material information of each cell. It does not include any grid information and unit. So, with these data files only, whether they are generated from a non-uniform or uniform grid is unknown. User need to obtain more information about grid in using these data files.

In order to make a good mesh data file, it is suggested to use a uniform grid.

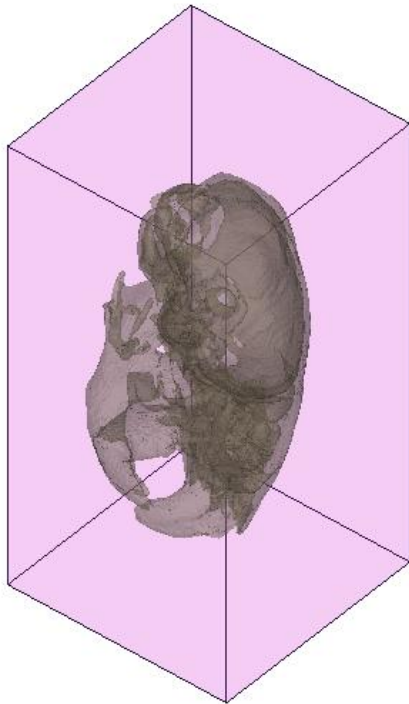
Use the Model from Mesh Files

- For **vxa** or **vxb** files, they can be loaded to a voxel
- For **Imat** file, user can use his own code to convert it to **vxa** or **vxb** files with other material property mapping method, then load to a voxel

Demo

Simulate a 2D Slice Cut from a STL model

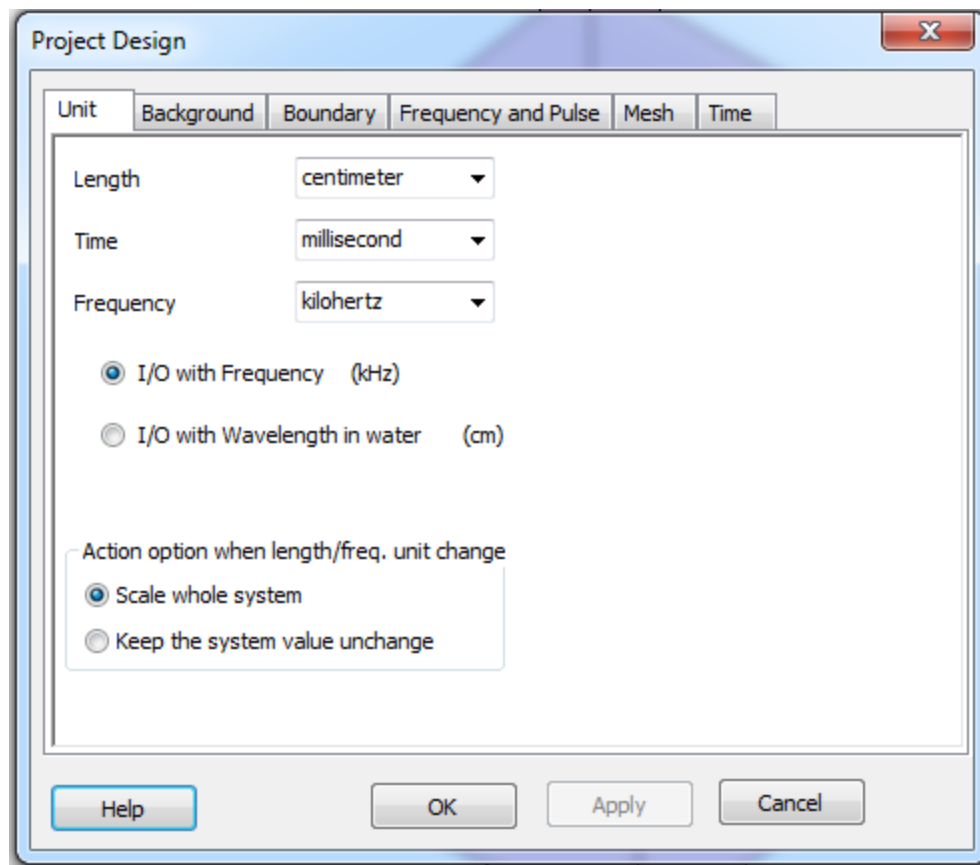
- we have a STL file to describe a mouse skull
- we want to simulate one 2D slice of this model by Cartesian EL solver



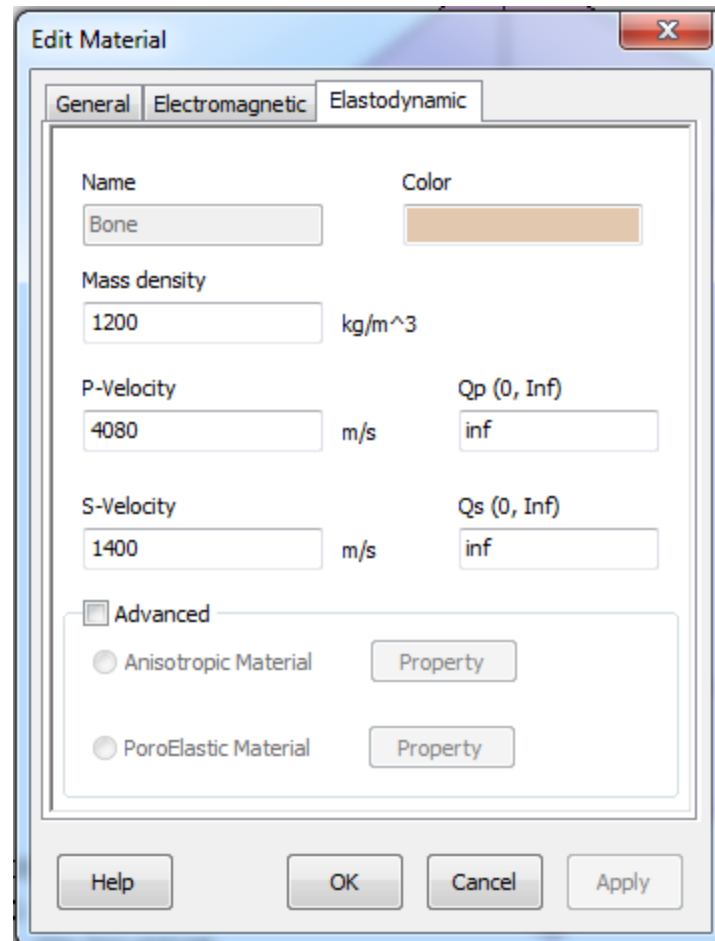
- Due to a STL file can not be guaranteed as a closed 3D solid model. Therefore, it can not be treated as a 3D geometry and be operated by boolean operation. A 2D slice of this model can be only generated by the voxel way.

Step I: use a WCT Cartesian EL project to generate the 2D slice we need

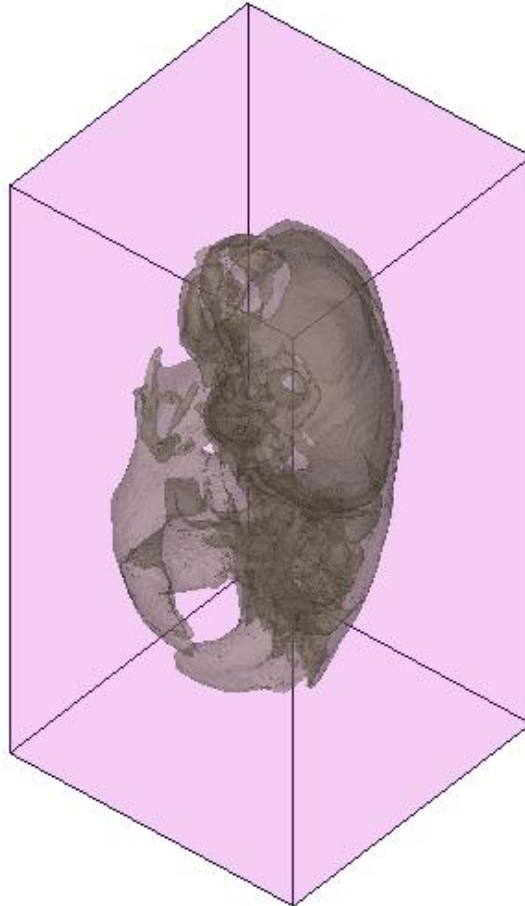
Build a empty Cartesian EL project with **CM** as unit



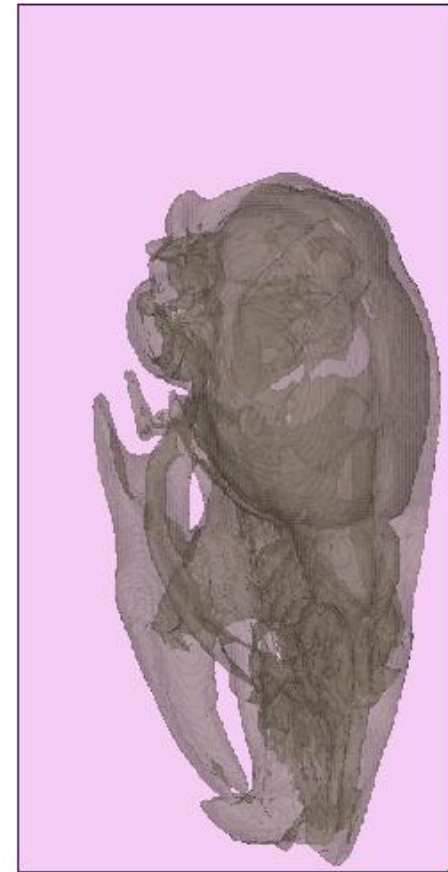
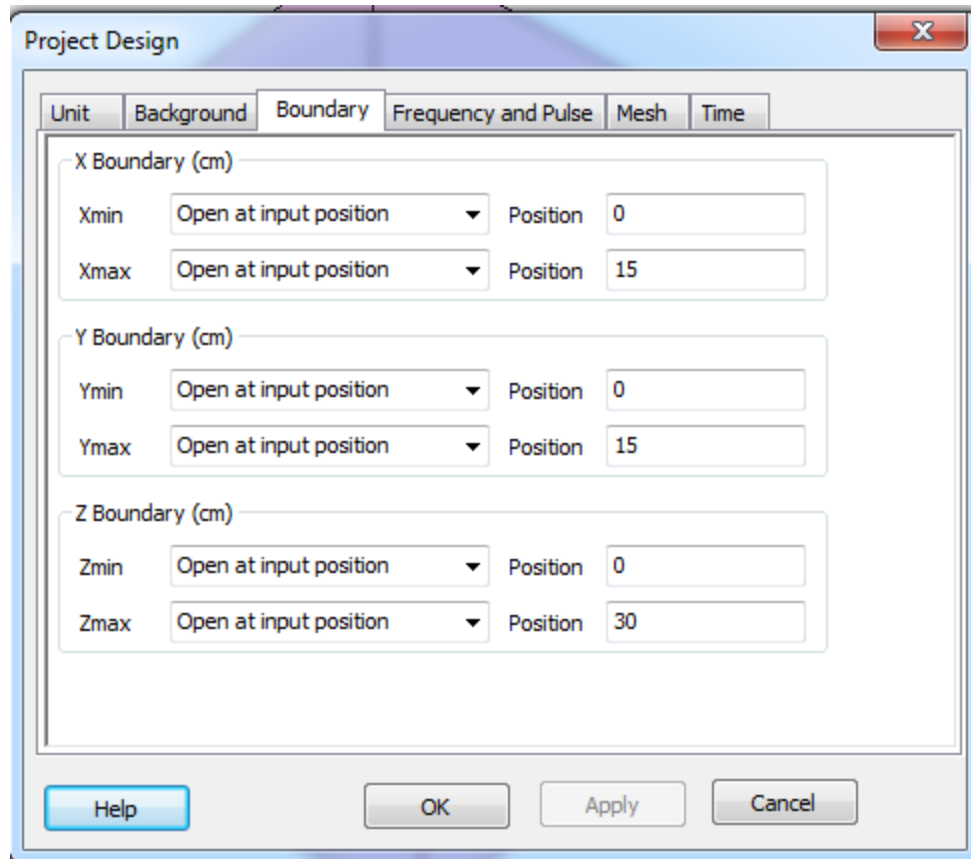
Set project background as **water** and define a new material: **bone**



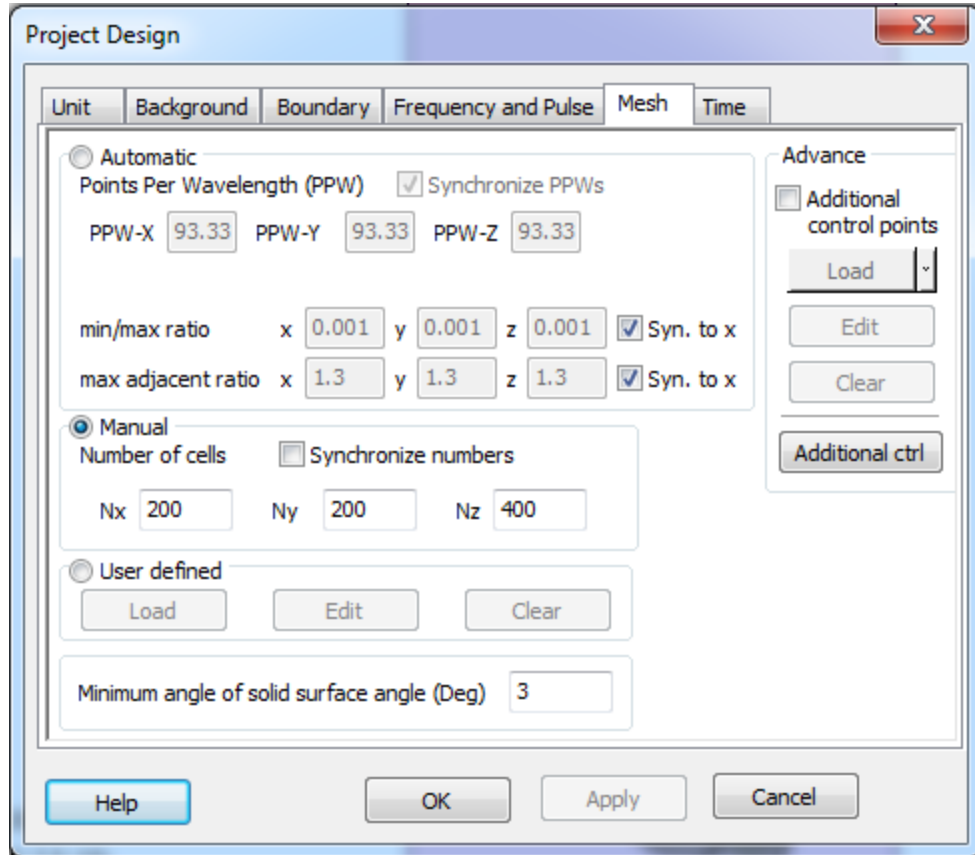
Insert STL model in this project and use material: bone.
Set the model with 50% transparency, it will be looked as,



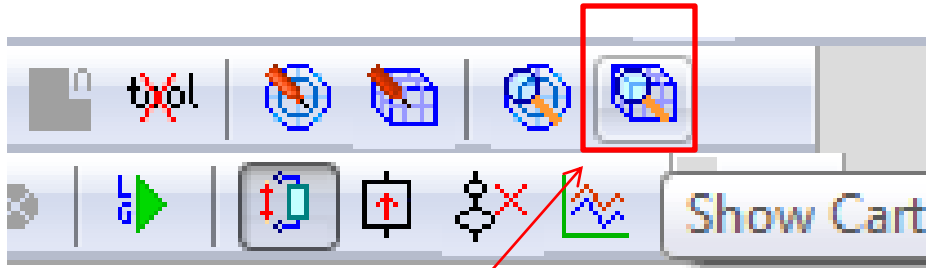
Then we define the domain size as following to close the model



Then set the mesh as a uniform grid as following



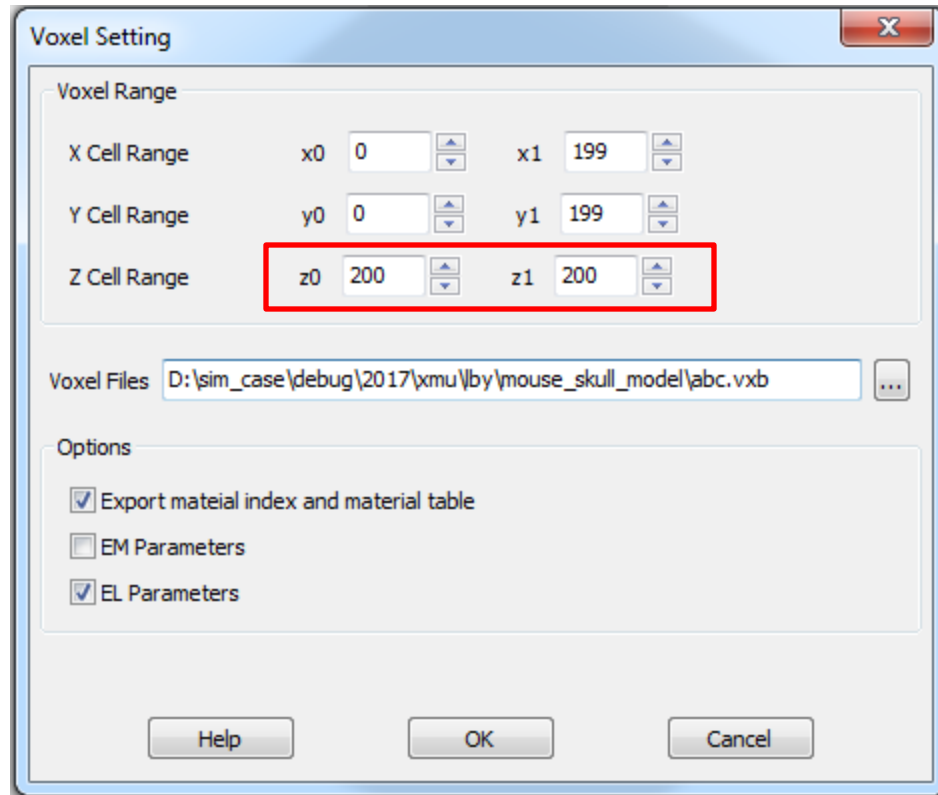
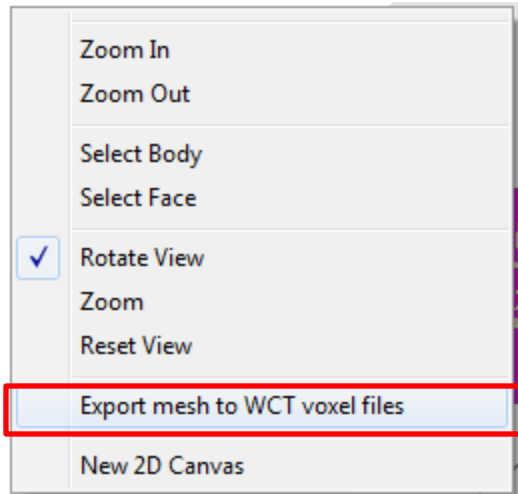
With this setting, we know $[\Delta x, \Delta y, \Delta z]=[0.075, 0.075, 0.075]$ cm



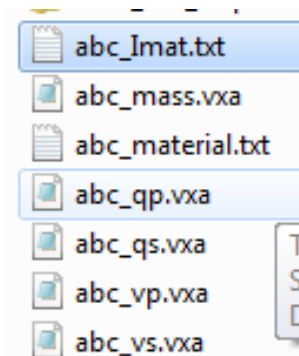
Enter View Cartesian Mesh Mode



To export the 200th Z slice, which is at Z=15 cm



With data file root name as “abc”, format as vxb, the EL parameter files is

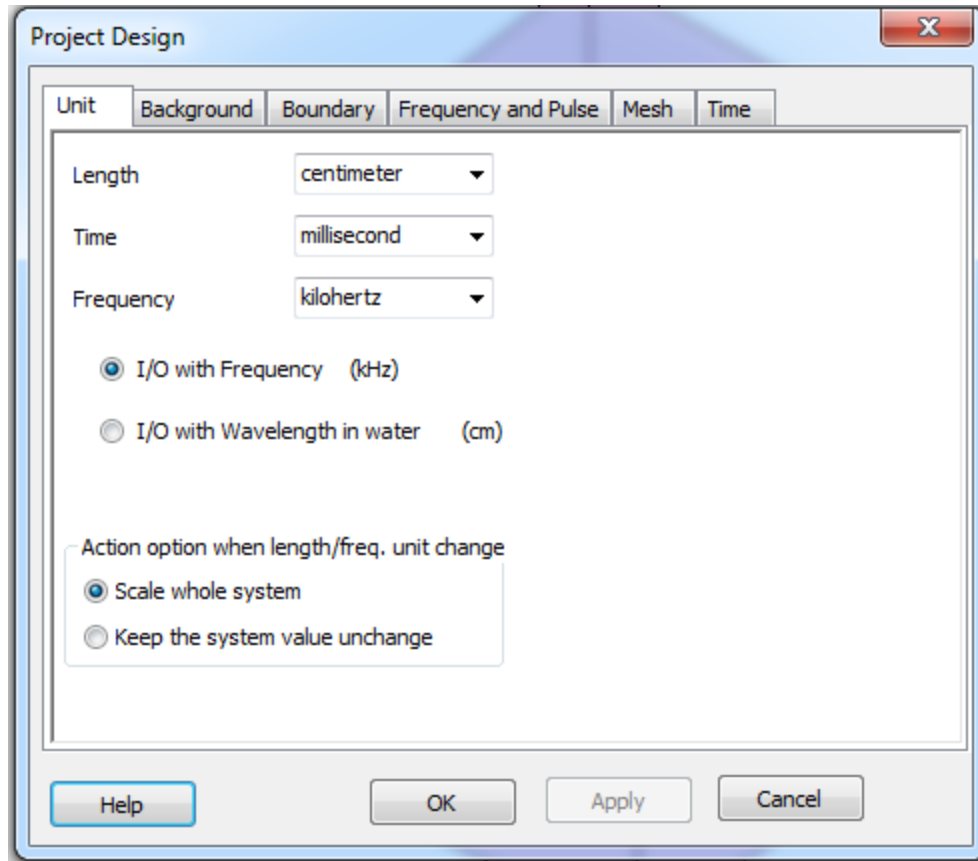


Step II: Insert this 2D slice as voxel in the project we want to simulate

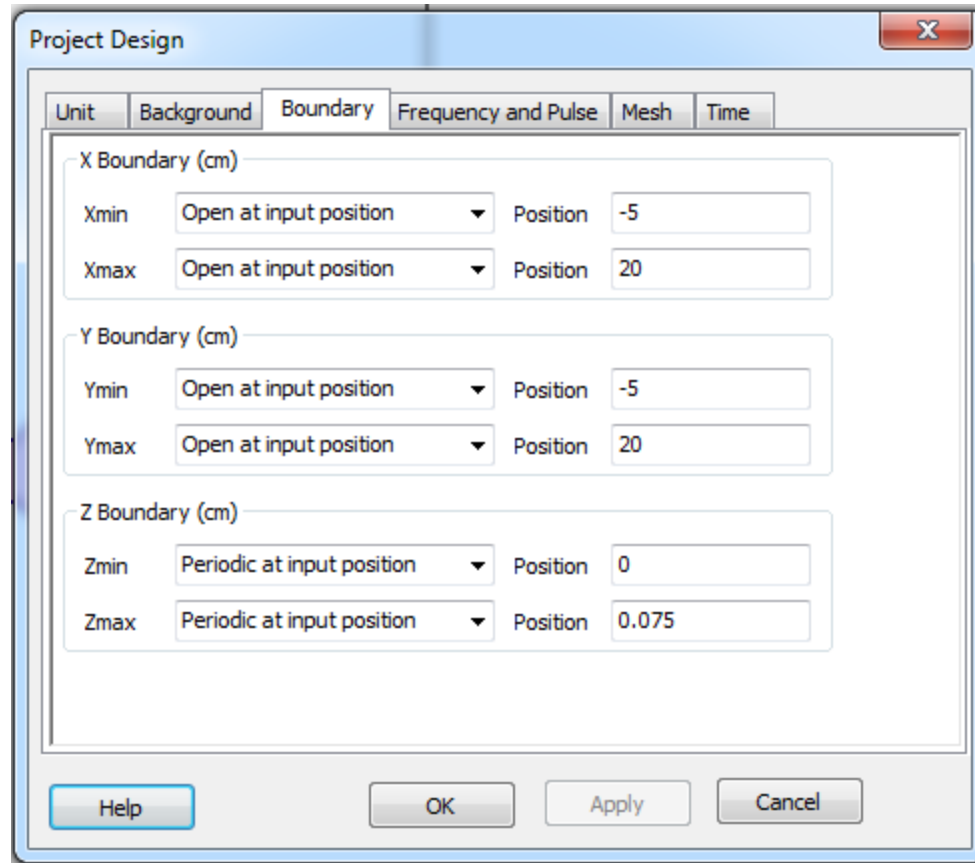
- We know the slice with following information
 - 200x200x1 cells, the cell size is 0.075 cm, both in x, y and z.
 - But we can re-scale them in voxel definition.
 - the skull is embedded in water



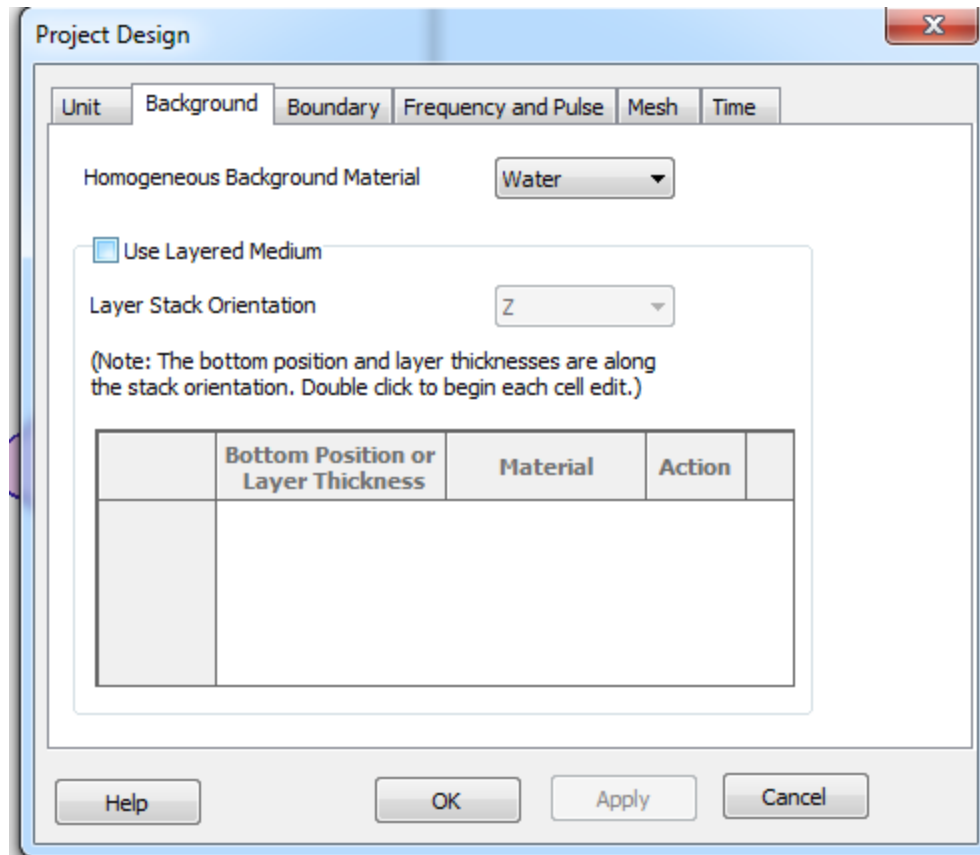
We setup the simulation project with **CM** as unit



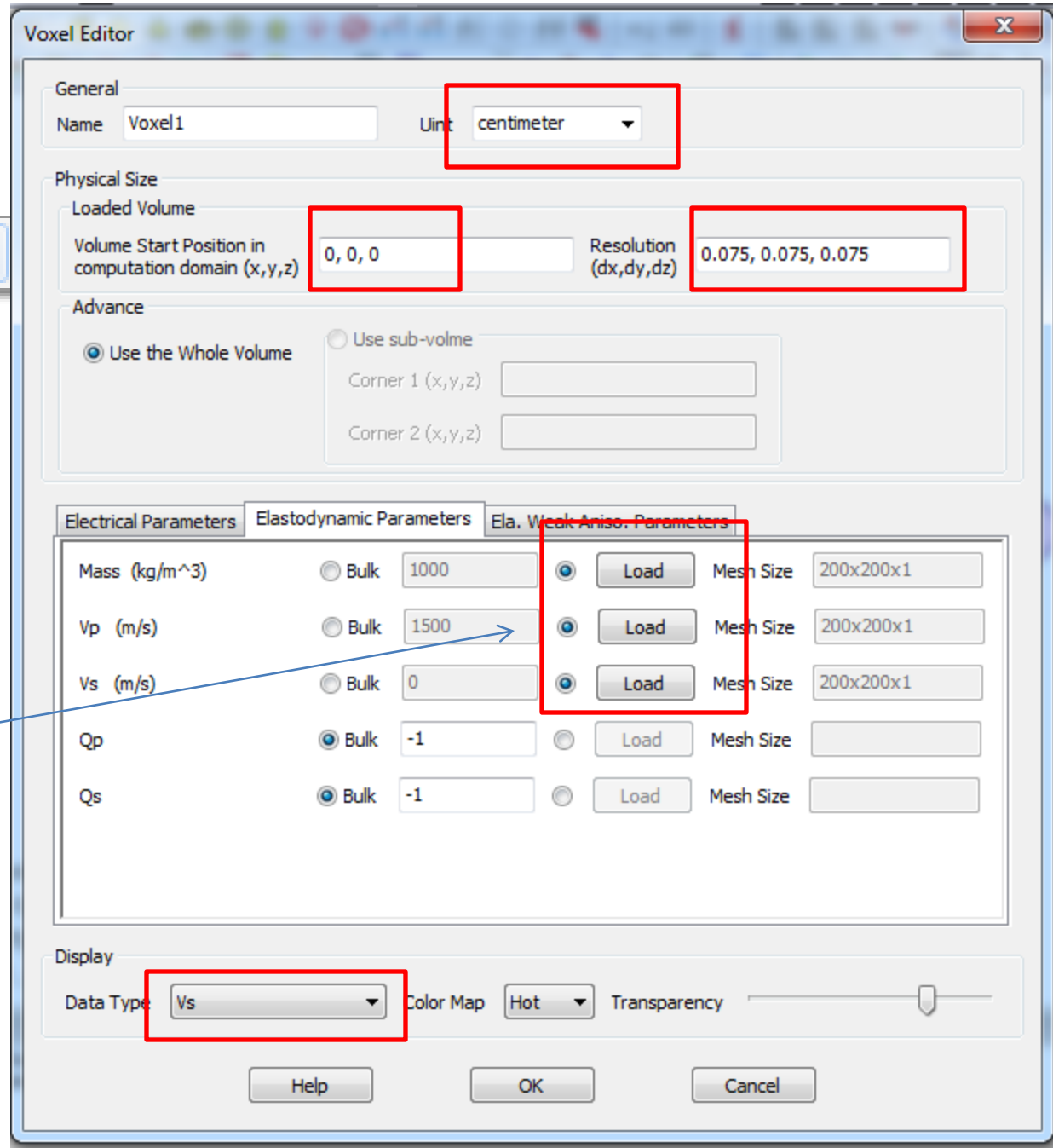
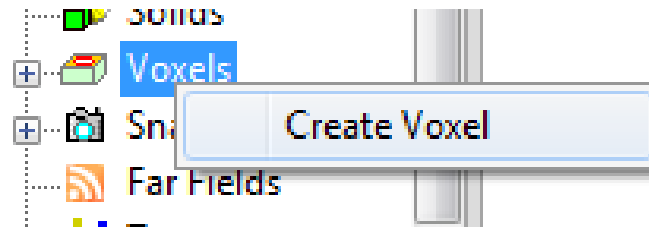
Set project size as following



Set project background as **water**

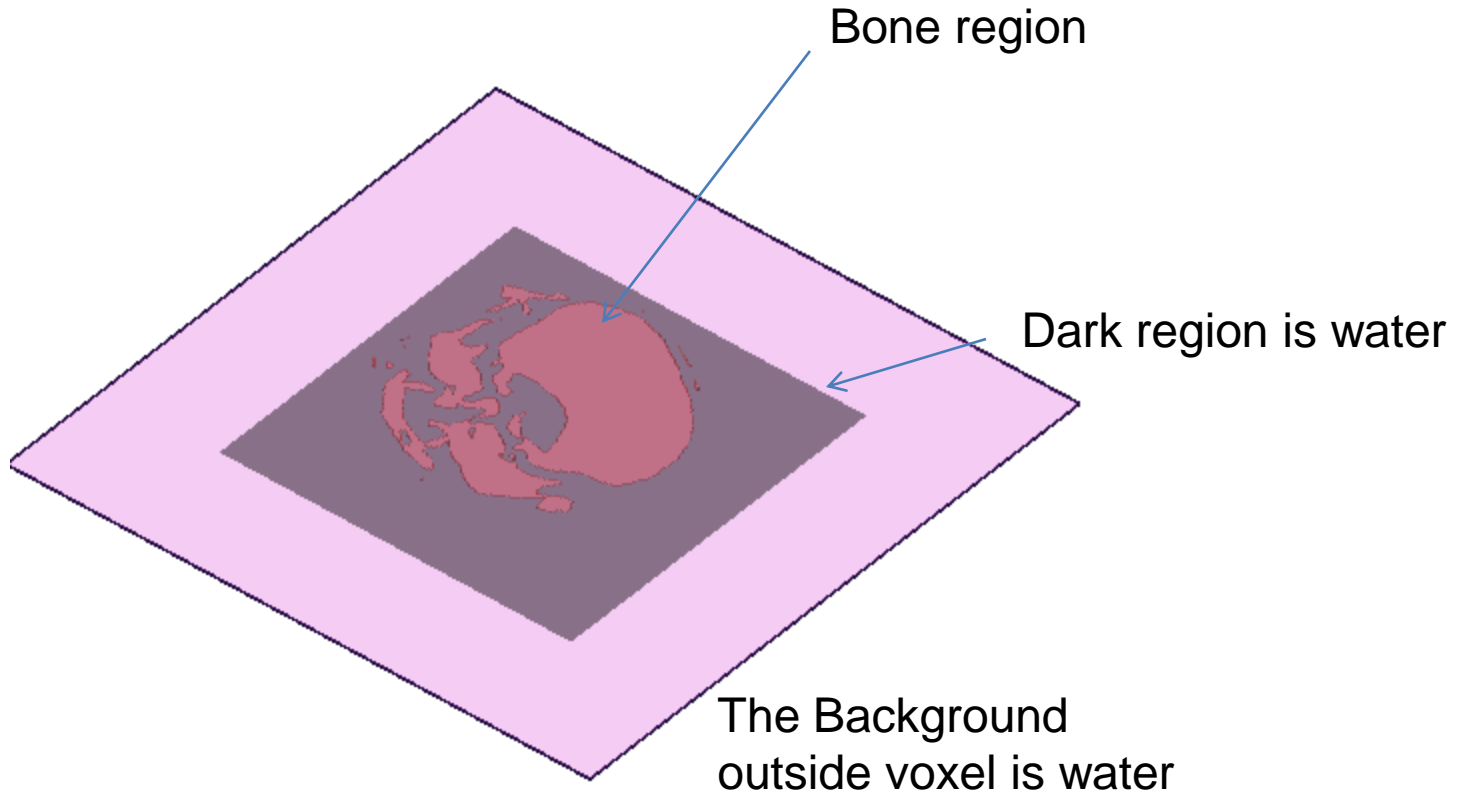


Then load voxel files

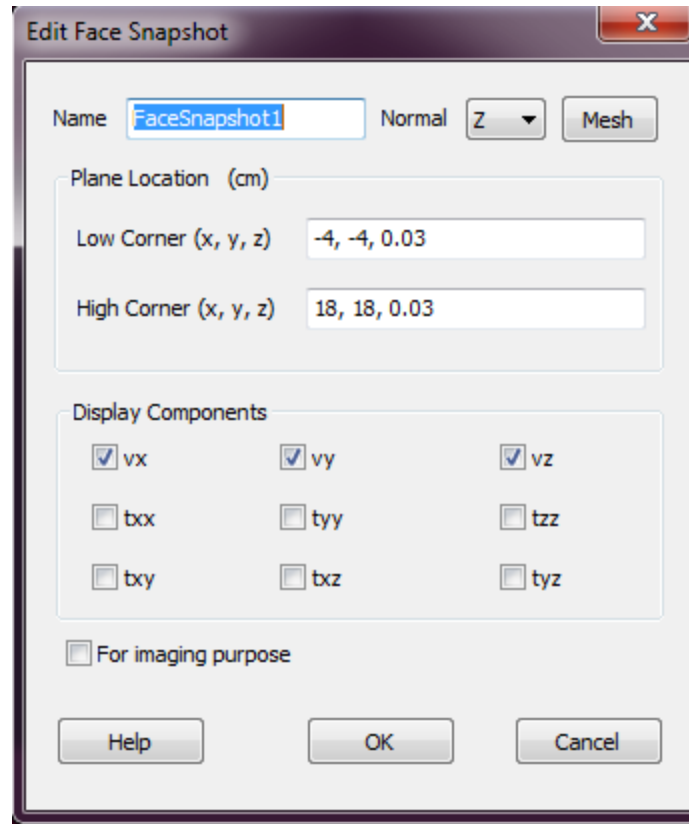


Load
"abc_mass.vxb",
"abc_vp.vxb",
"abc_vs.vxb"

After "OK", we get



Then, define a monopole source at (0,0,0). And a 2D snapshot to capture (vx, vy, vz) as following:



Then, we can simulate the project,
and check the snapshot

