

1 Creating and Meshing a model with Trelis for SPECSEM2D

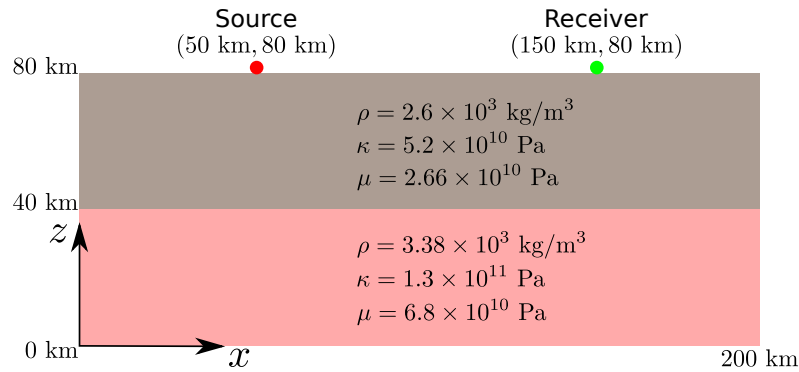


Figure 1: A layered model.

In this tutorial, we will create and mesh the model given in Homework 9 as shown in Figure 1 using the meshing software Trelis (Figure 2). We will use the moment-tensor source for the wave propagation simulation.

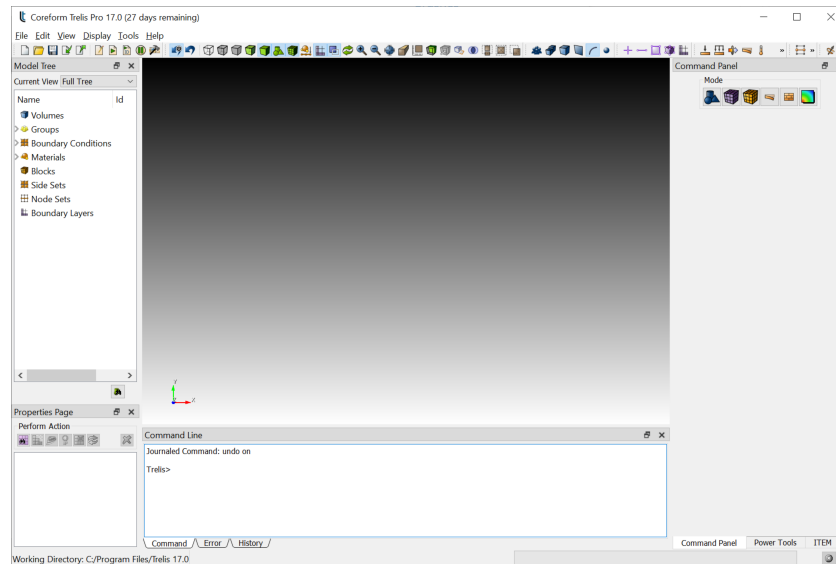
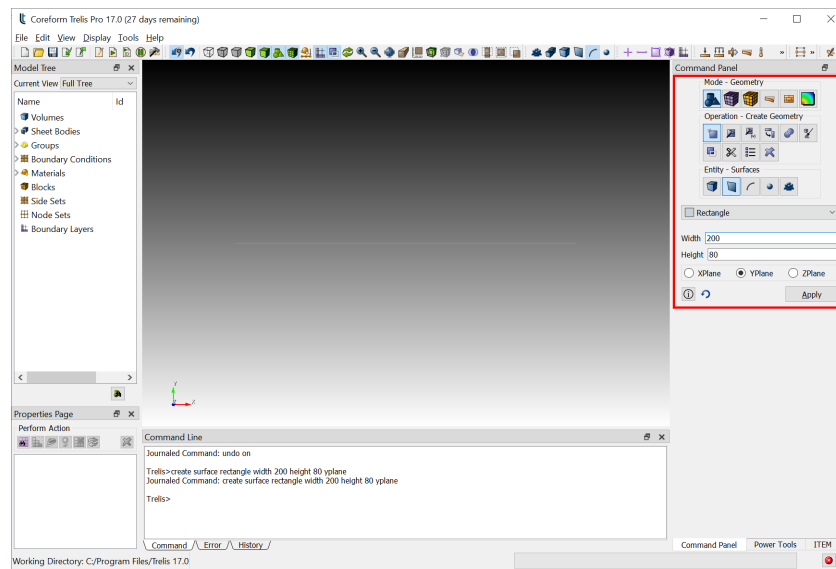
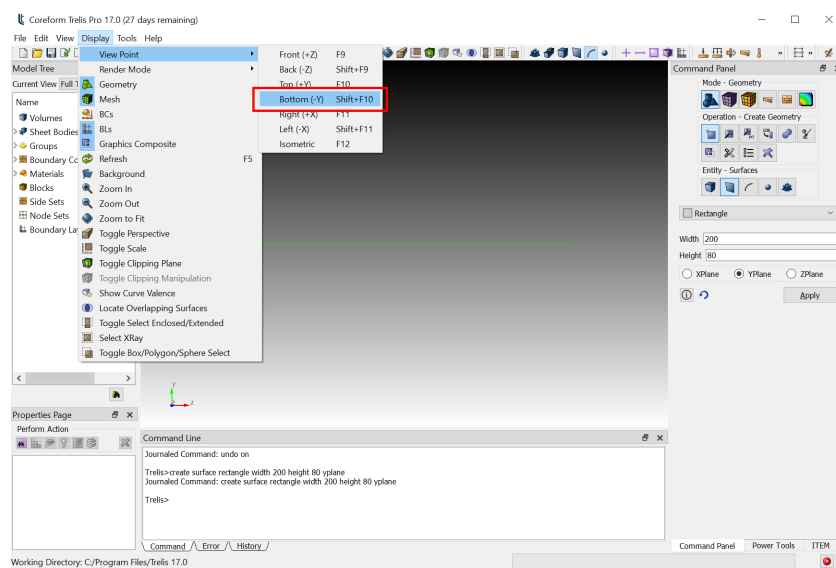


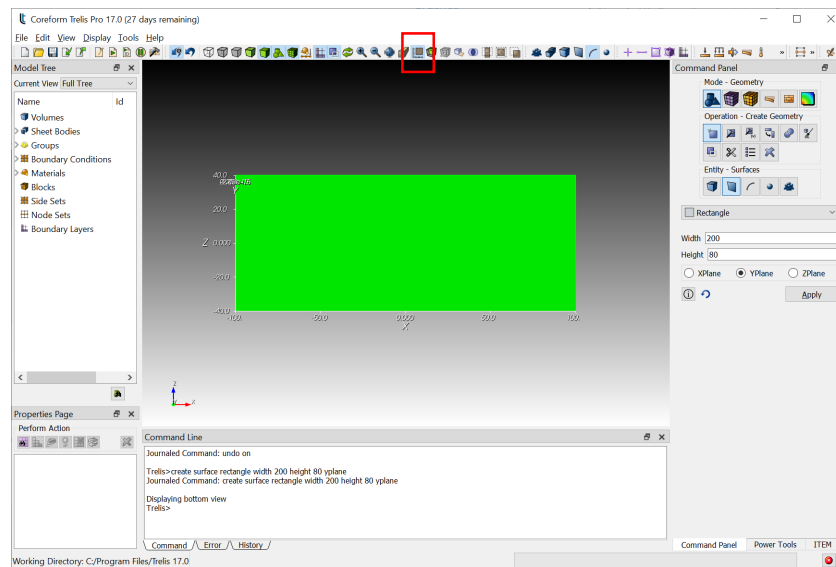
Figure 2: Trelis.



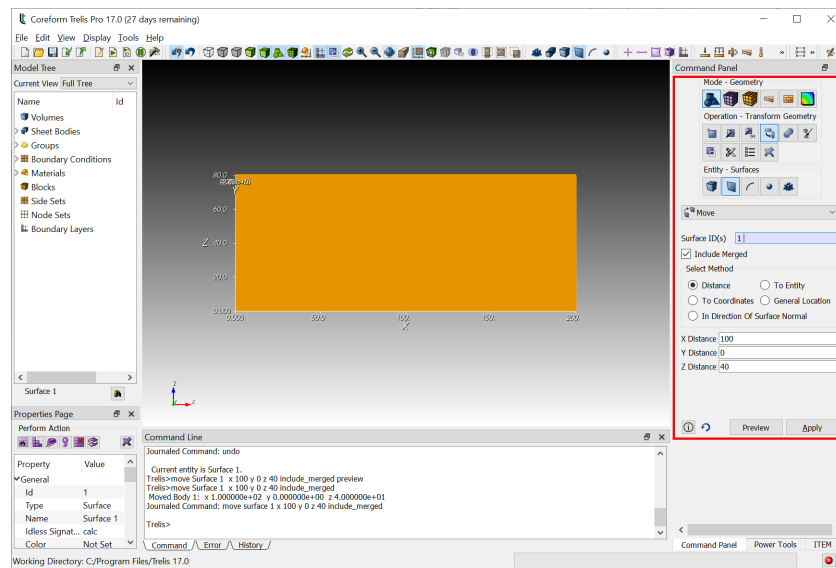
Step 1: Create a rectangle of size 200 km \times 80 km.



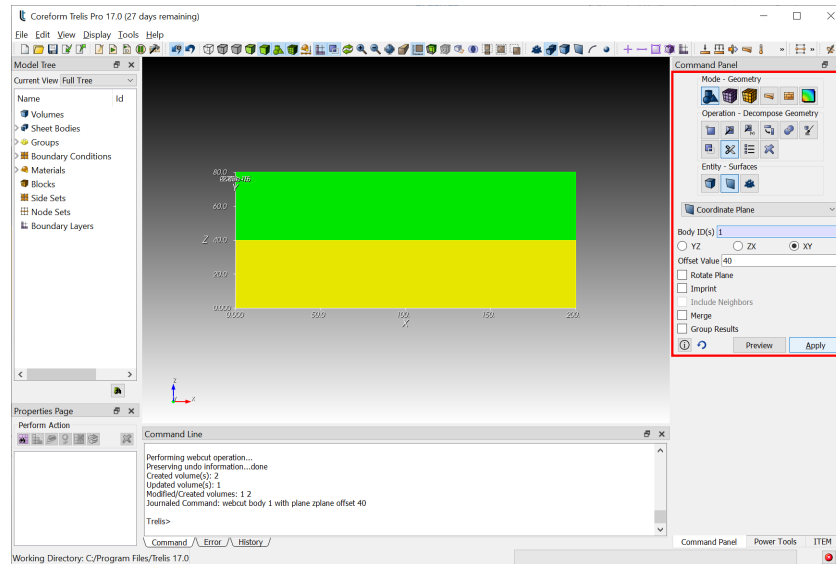
Step 2: Change display view to XZ plane.



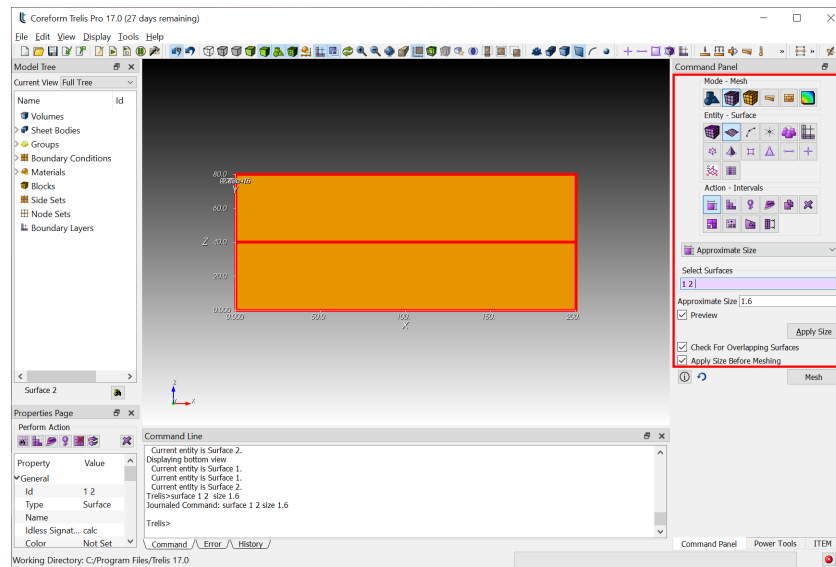
Step 3: Display the axis. Note that the origin is not at the corner of the model.



Step 4: Reset the origin to the bottom left corner of the model.



Step 5: Cut the rectangle at $z = 40$ km to create two layers.



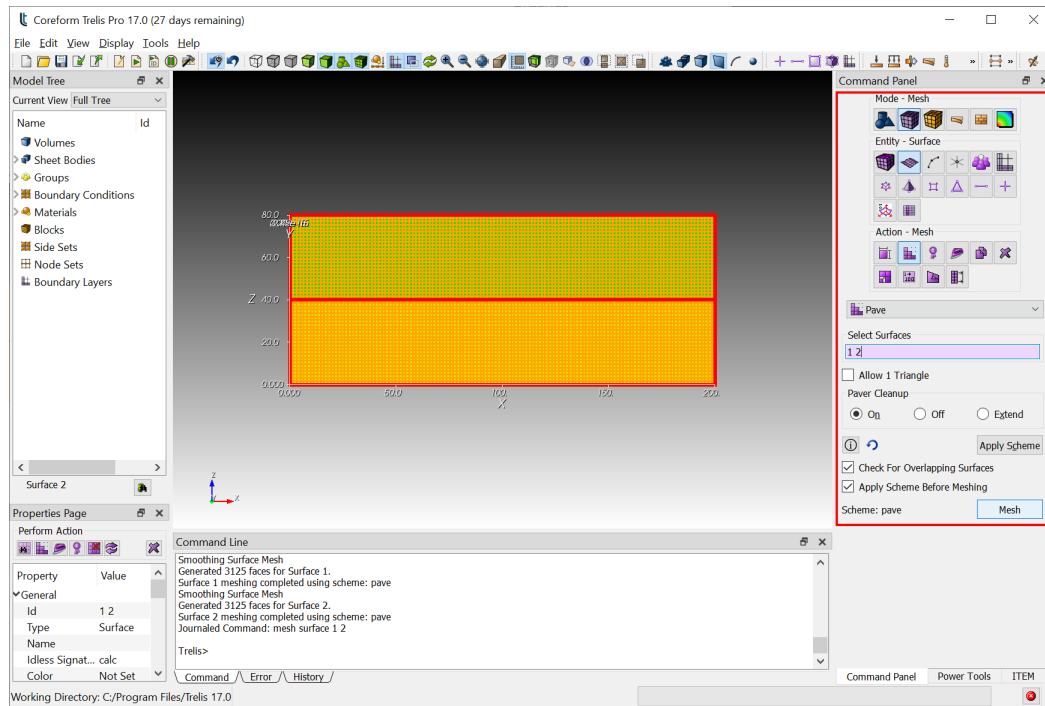
Step 6: First, type the following commands:

imprint all

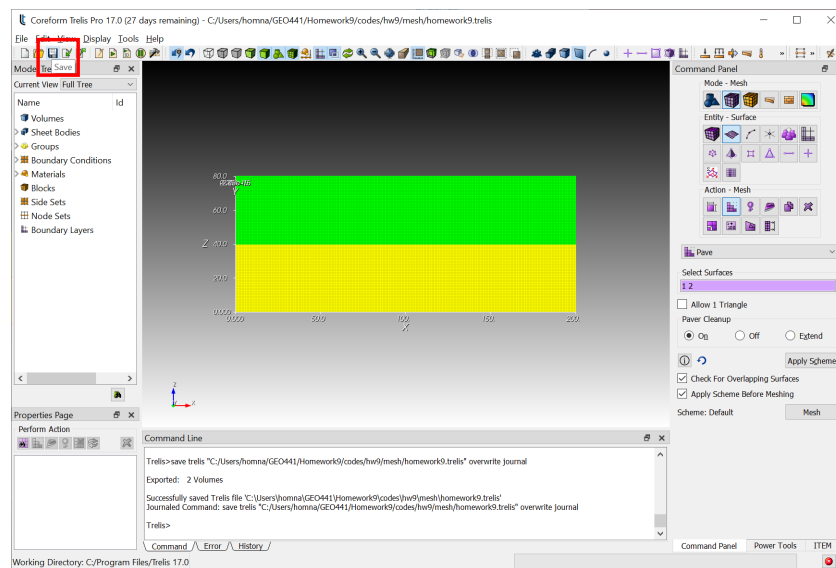
merge all

compress all

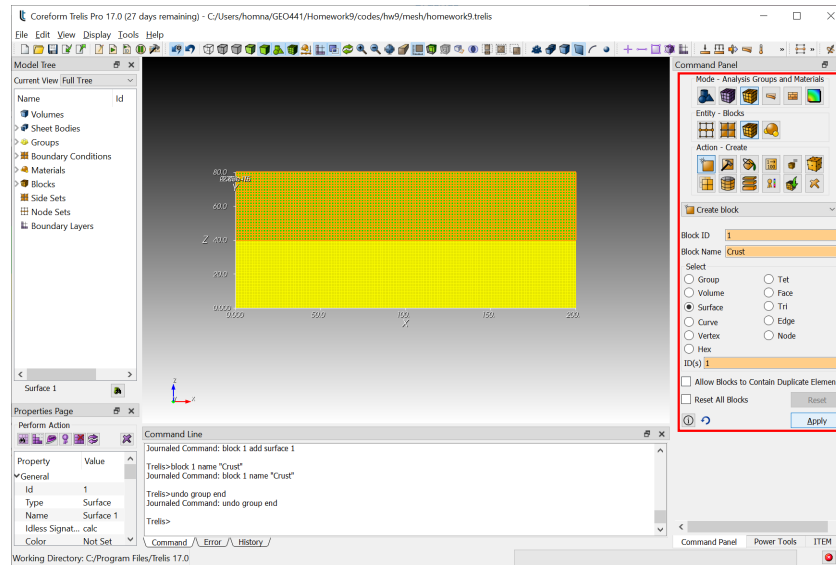
then, define the appropriate mesh size.



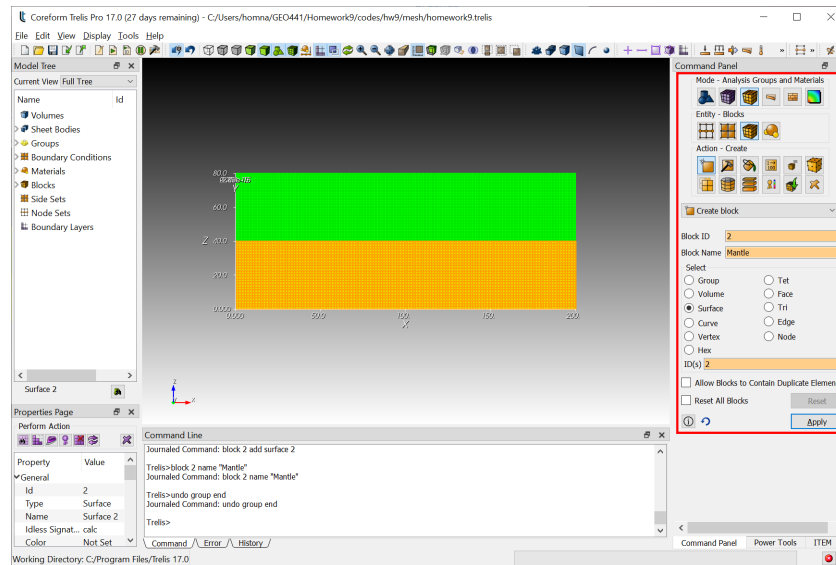
Step 7: Select and apply mesh scheme Pave to all surfaces. Then mesh the model.



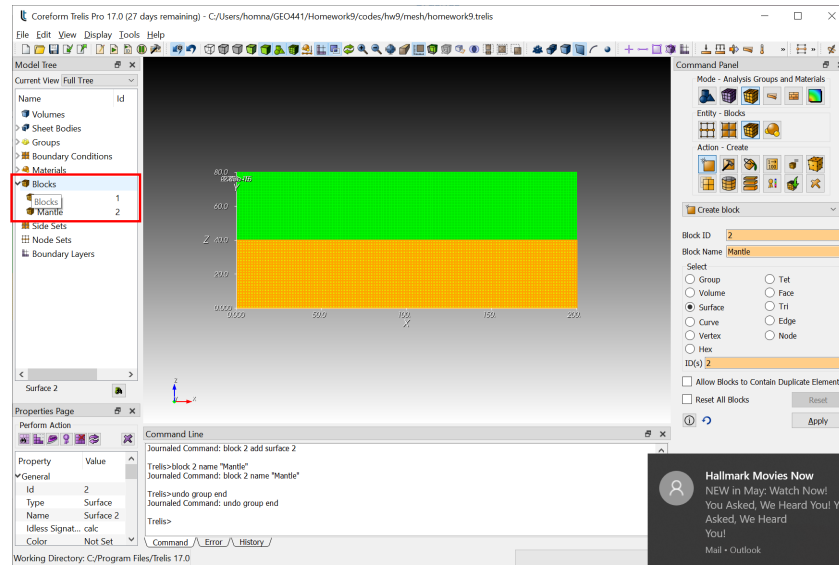
Step 8: Save the file as Homework9.trellis.



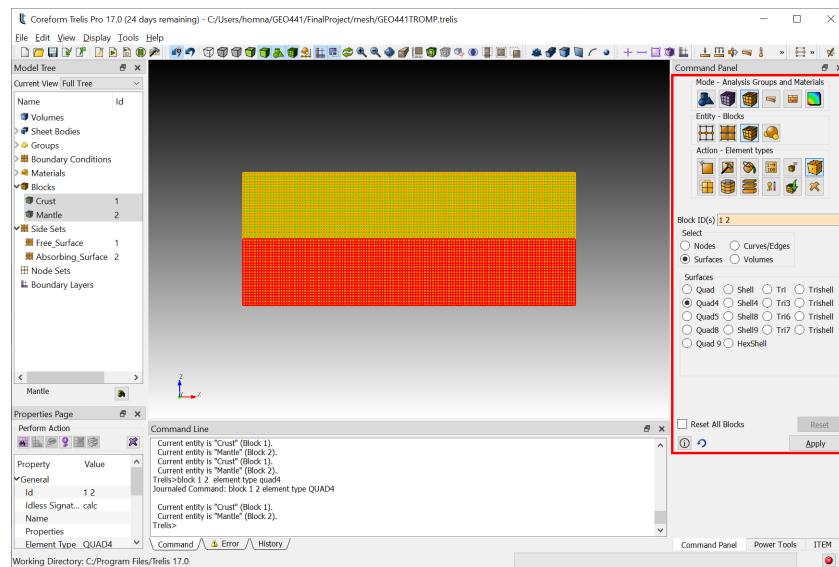
Step 9: Create a block with ID 1 and name **Crust**. Select the top surface in IDs and apply.



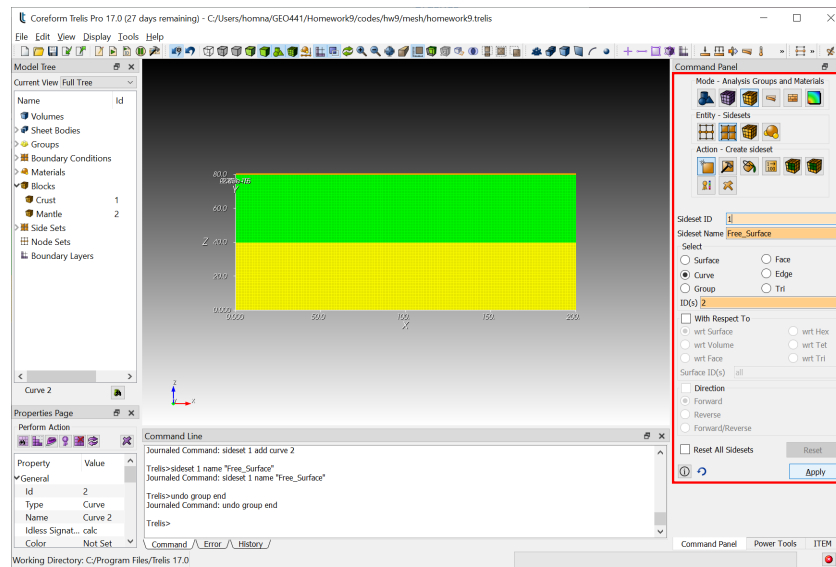
Step 10: Create a block with ID 2 and name **Mantle**. Select the bottom surface in IDs and apply.



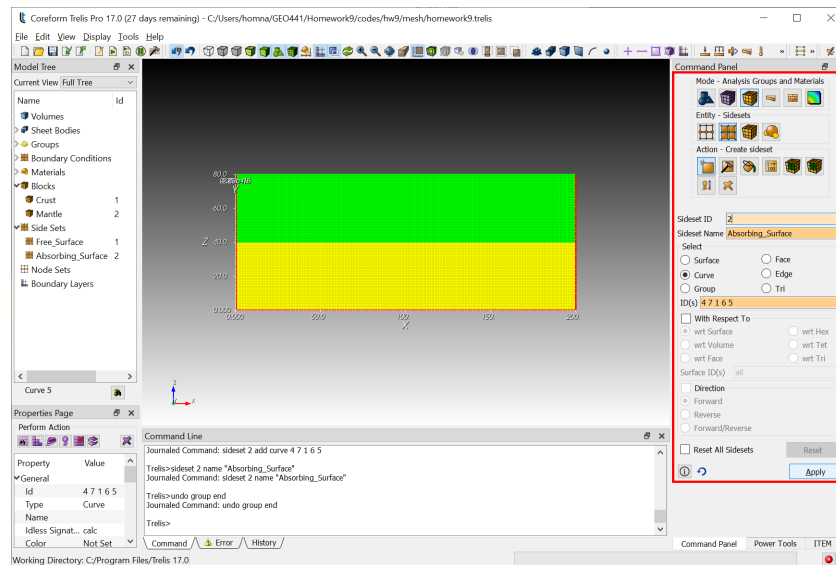
Step 11: Click on **Blocks** on the Model tree panel on the left and make sure that you have properly defined the blocks.



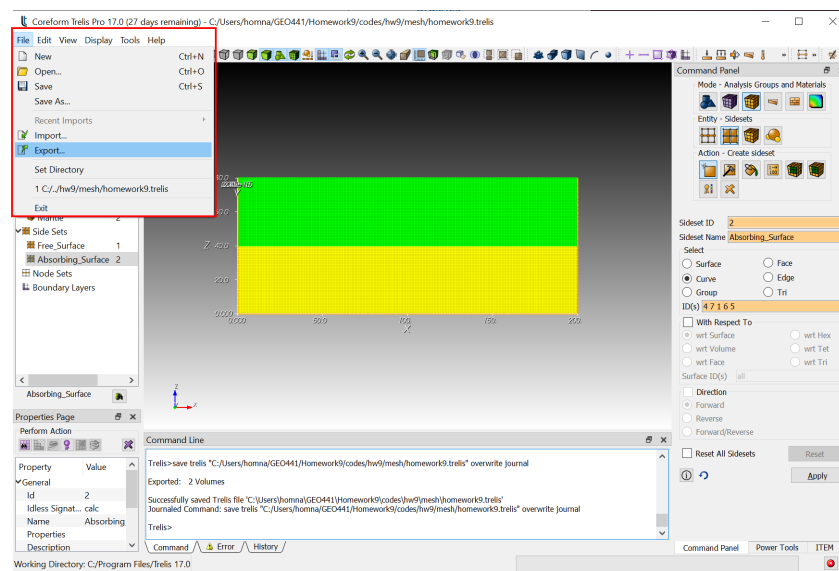
Step 12: Assign element type of all the blocks to **QUAD4**.



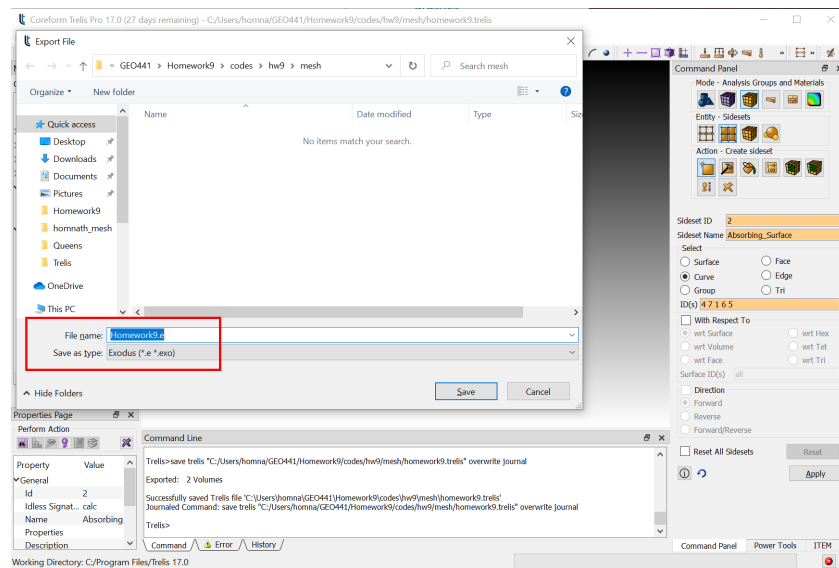
Step 13: Create a sideset with ID 1 and name **Free_Surface**. Select the top curve in IDs and apply.



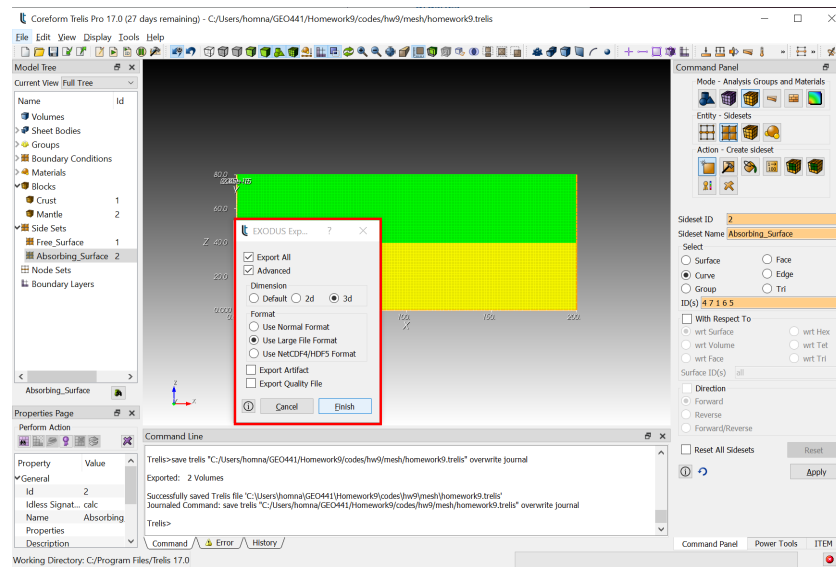
Step 14: Create a sideset with ID 2 and name **Absorbing_Surface**. Select the all left, bottom and right curves in IDs and apply.



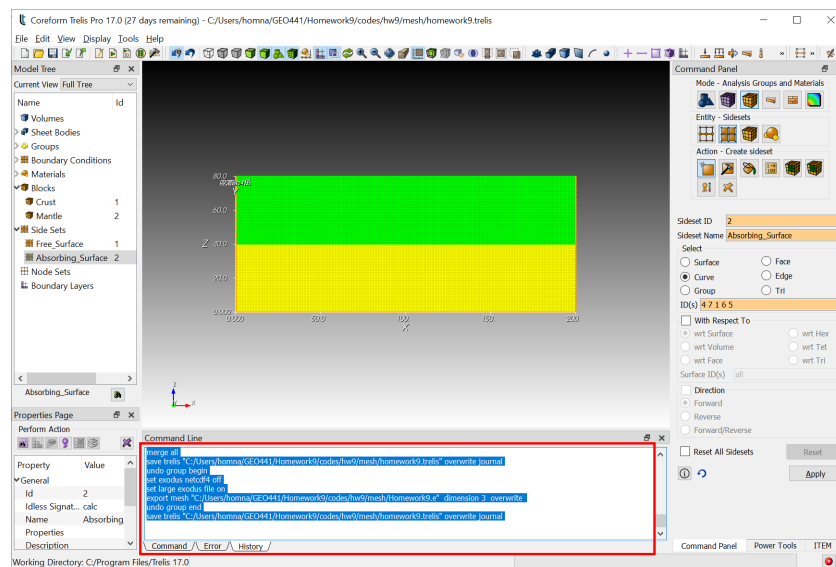
Step 15: Click File ⇒ Export.



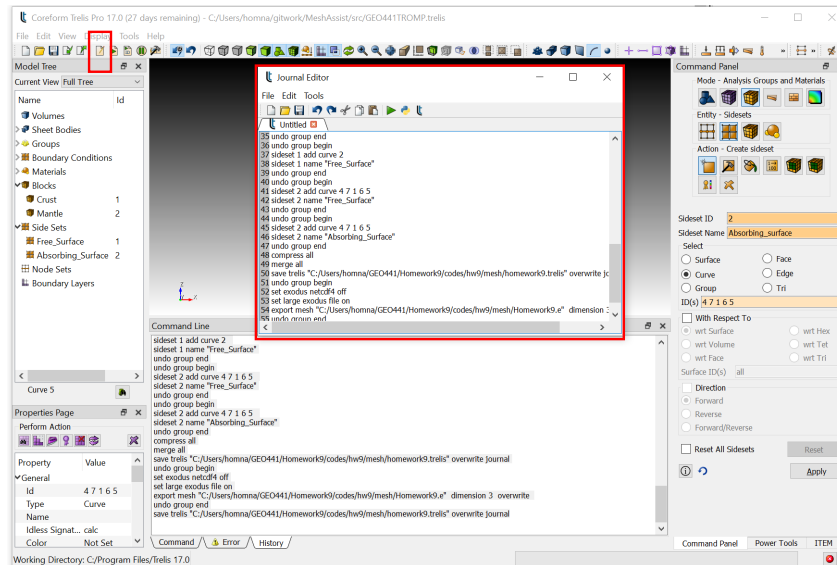
Step 16: Choose the file name Homework9.e and select the type Exodus.



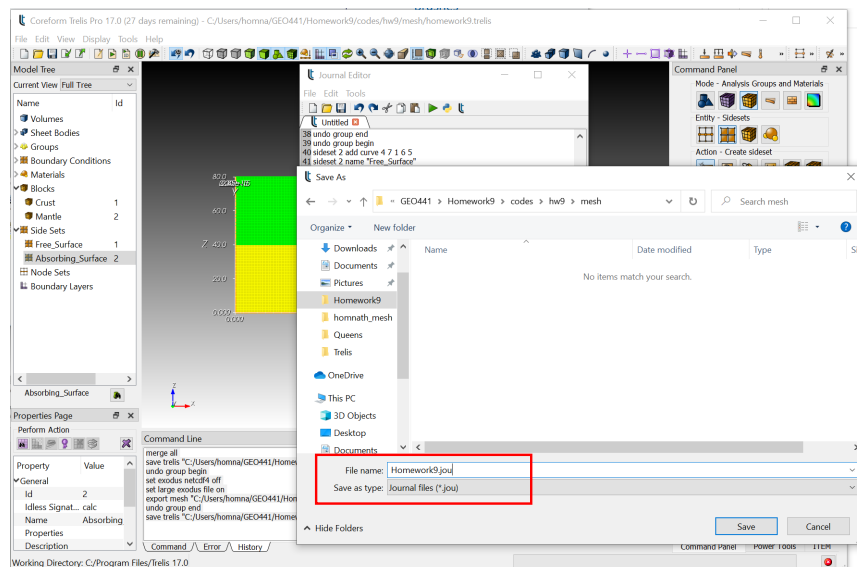
Step 17: Click Advanced and select the dimension 3d and the format Use Large File Format. Click Finish.



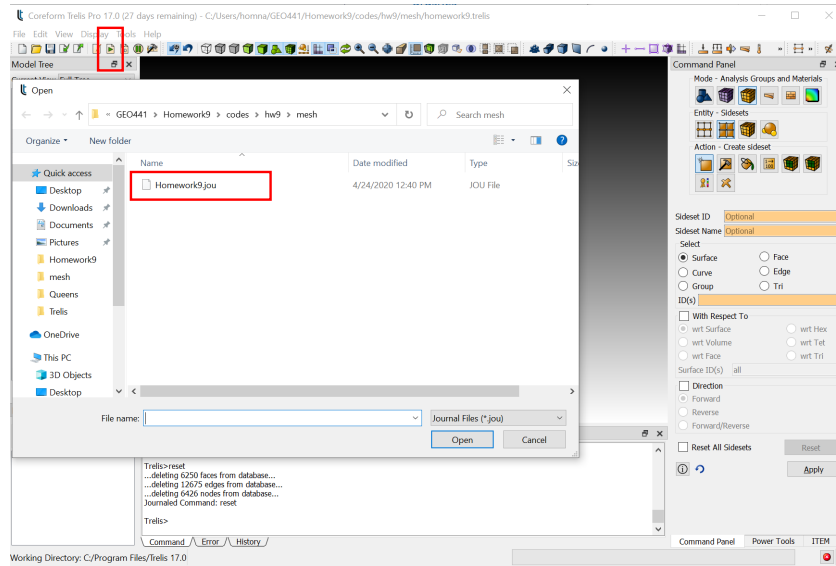
Step 18: For future use, click on History on the Command panel on the bottom. Select the entire command history and copy.



Step 19: Click on the Journal Editor icon on the top tool bar to open new journal file. Paste the entire command history from the previous step.



Step 20: Save the journal file with the file name Homework9.jou and select the type Journal files.



Step 21: To recreate the same model and mesh you can always play the journal file. To play the journal file, click on the **Play Journal File** icon on the top tool bar and select the file **Homework9.jou**. You can also modify the journal file if necessary and play again to create the modified model and mesh. **Please make sure that the file paths are properly defined in the journal file if any.**

2 Converting mesh to SPECSEM2D mesh files

Step 1: Download MeshAssist package from GitHub:

```
git clone --recursive https://github.com/homnath/MeshAssist.git
```

Step 2: Go to the MeshAssist folder. If necessary, change the compilers in Makefile and type:

```
make all
```

Step 3: Type the command:

```
exodus2specsem2d Homework9.e -bin=1 -fac=1000 -head=1
```

This command will generate several files with file names **Homework9_***.

NOTE 1: You need a NetCDF tool called **ncdump** on the path.

NOTE 2: Both **exodus2specsem2d** and **Homework9.e** have to be in the same folder, otherwise, you need to provide the proper paths.

NOTE 3: If you get “Negative Jacobian” error in SPECSEM2D, try the following command:

```
exodus2specsem2d Homework9.e -bin=1 -fac=1000 -order=1 -head=1
```

3 Preparing complete input files for SPECFEM2D

Step 1: Create a directory `Homework9` inside the `EXAMPLES` folder of `SPECFEM2D`.

Step 2: Go to `Homework9` directory. Create a new folder `MESH`. Copy all files created by `exodus2specfem2d` in `MESH`.

Step 3: Create a new folder `DATA`.

Step 4: Copy two input files `Par_file` and `SOURCE` from one of the examples to the folder `DATA`.

For example,

```
cp ../salt_dome_CUBIT_mesh/Stacey_homogeneous/Par_file DATA/
cp ../salt_dome_CUBIT_mesh/Stacey_homogeneous/SOURCE DATA/
```

Step 5: Prepare the `DATA/Par_file`

Open `DATA/Par_file` with any text editor. Find and modify the following parameters:

```
title   = Homework 9
:
nt       = 20000
deltat   = 5.0e-3
:
nrec     = 1           # number of receivers
xdeb     = 150000.     # first receiver x in meters
zdeb     = 80000.      # first receiver z in meters
xfin     = 150000.     # last receiver x in meters (ignored if only one receiver)
zfin     = 80000.      # last receiver z in meters (ignored if only one receiver)
:
nbmodels = 2          # number of models
:
1  1  2600.d0  5800.088417d0  3198.557367d0  0  0  9999  9999  0  0  0  0  0  0
2  1  3380.d0  8079.975003d0  4485.347611d0  0  0  9999  9999  0  0  0  0  0  0
:
read_external_mesh = .true.
:
```

```

mesh_file          = ./MESH/Homework9_connectivity # mesh connectivity
nodes_coords_file  = ./MESH/Homework9_coordinates # nodes coordinates
materials_file      = ./MESH/Homework9_materials   # material IDs
free_surface_file   = ./MESH/Homework9_Free_Surface # free surface
:
absorbing_surface_file = ./MESH/Homework9_Absorbing_Surface # absorbing surface

```

Step 6: Prepare DATA/SOURCE file

Open DATA/SOURCE with any text editor. Find and modify the following parameters:

```

source_surf        = .false. # source inside the medium or at the surface
xs                 = 50000.   # source location x in meters
zs                 = 80000.   # source location z in meters
source_type        = 2        # moment tensor source = 2
time_function_type = 1        # Ricker = 1
:
f0                 = 10.      # dominant source frequency (Hz) if not Dirac or Heaviside
tshift             = 0.0      # time shift when multi sources (if one source, must be zero)
:
Mxx                = 0.       # Mxx component (for a moment tensor source only)
Mzz                = 0.       # Mzz component (for a moment tensor source only)
Mxz                = 1.       # Mxz component (for a moment tensor source only)
factor             = 1.d5     # amplification factor

```

4 Running the program

Step 1: Create a output directory `OUTPUT_FILES`.

Step 2: Create symbolic link to the `bin/` folder:

```
ln -s ../../bin/
```

Step 3: Run the mesher:

```
./bin/xmeshfem2D
```

Step 4: Run the solver:

```
./bin/xspecfem2D
```