1 Creating and Meshing a model with Trelis for SPECFEM2D

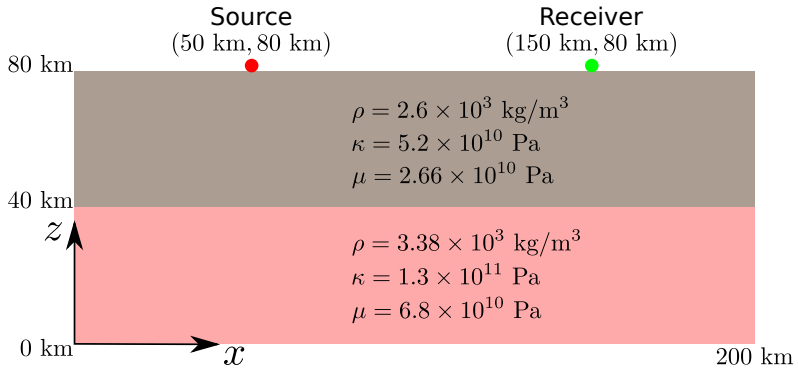


Figure 1: A layered model.

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.

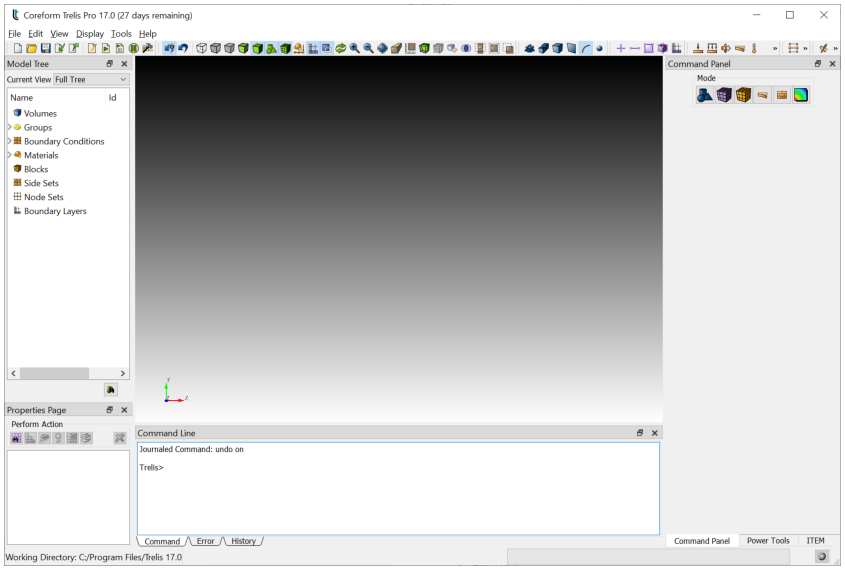
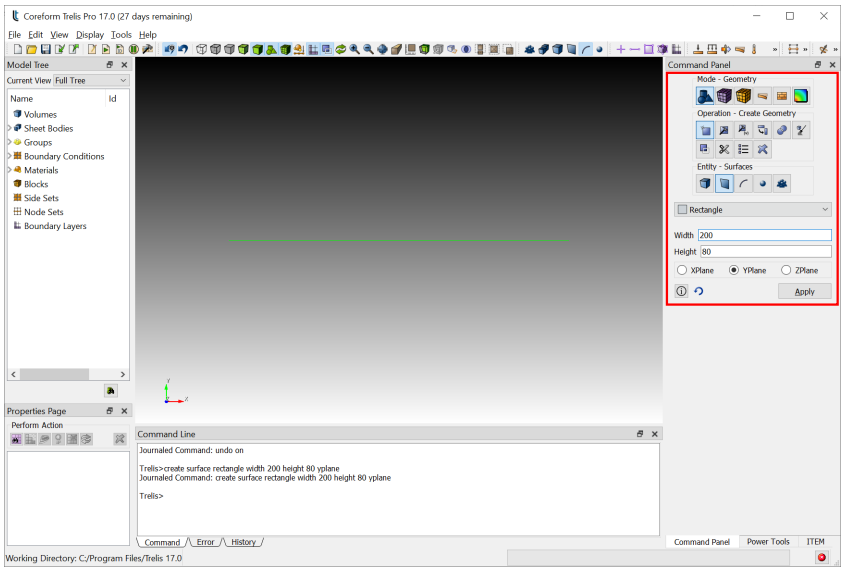
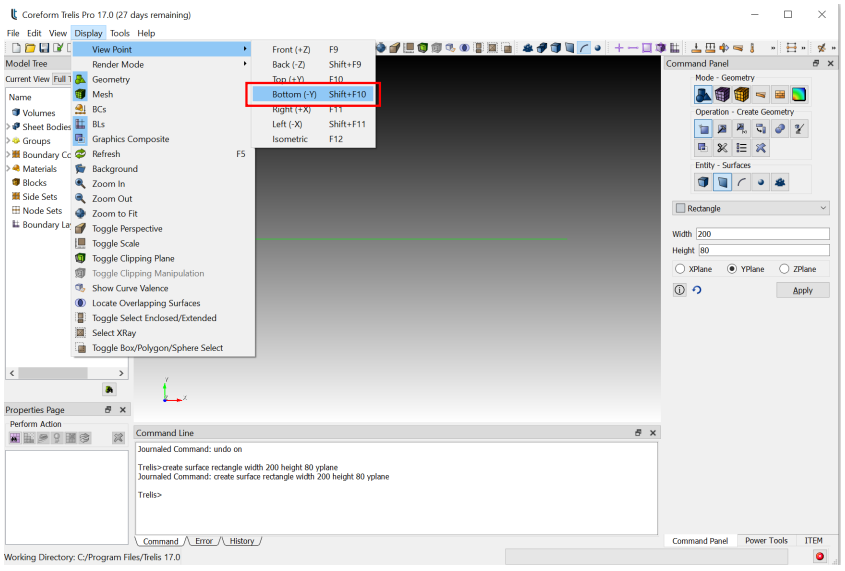


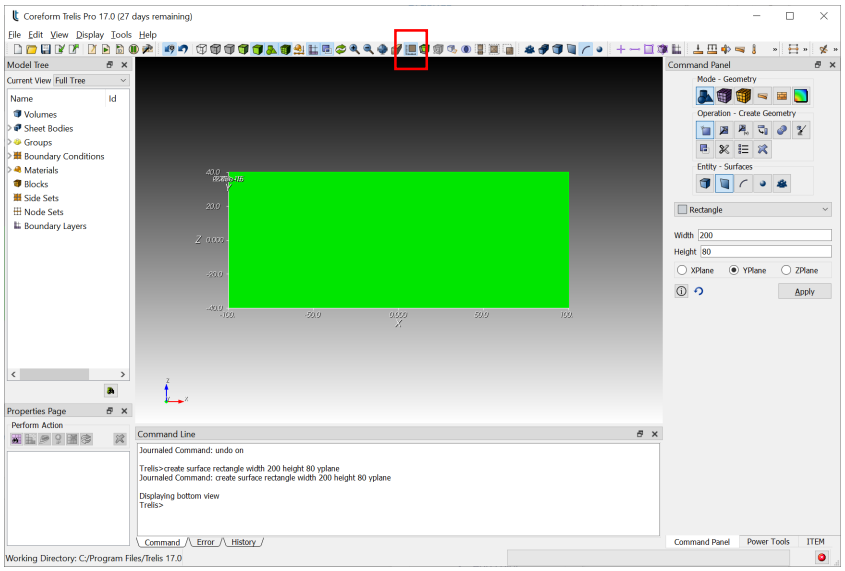
Figure 2: Trelis.



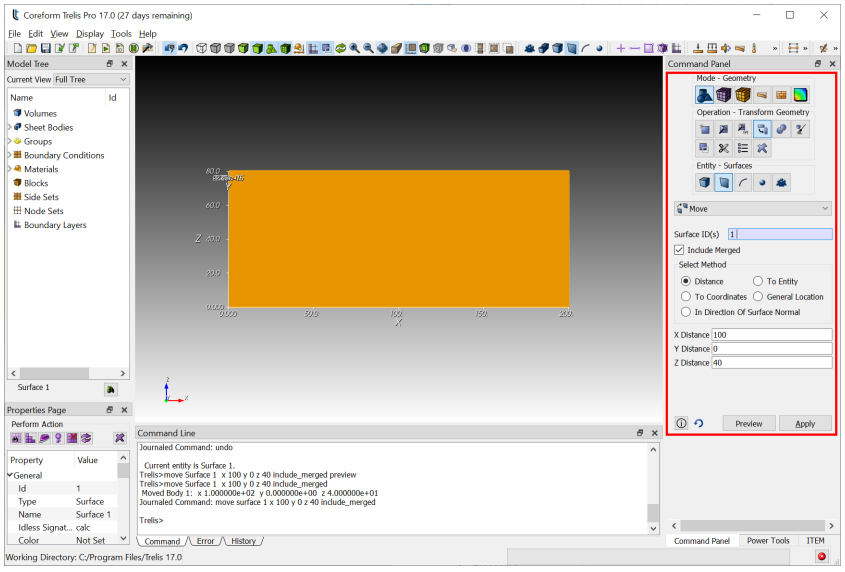
Step 1: Create a rectangle of size 200 km × 80 km.



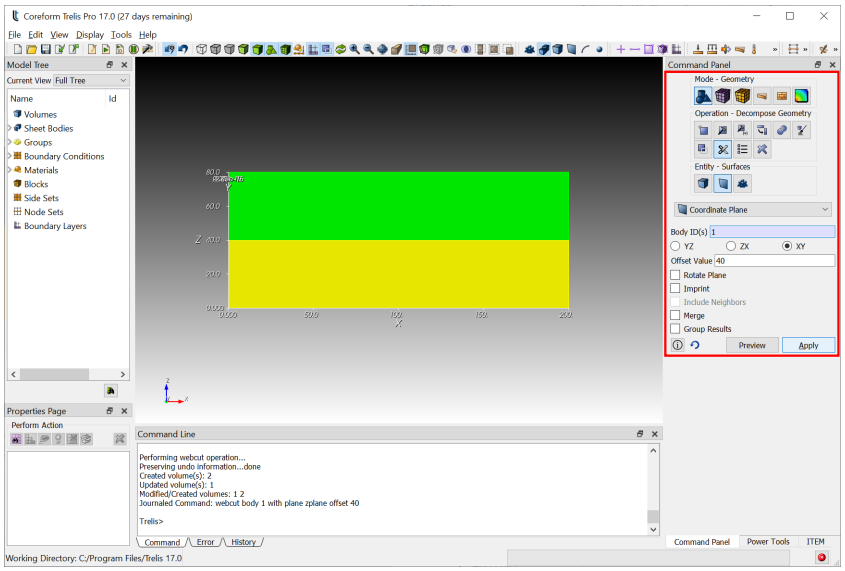
Step 2: Change display view to XZ plane.



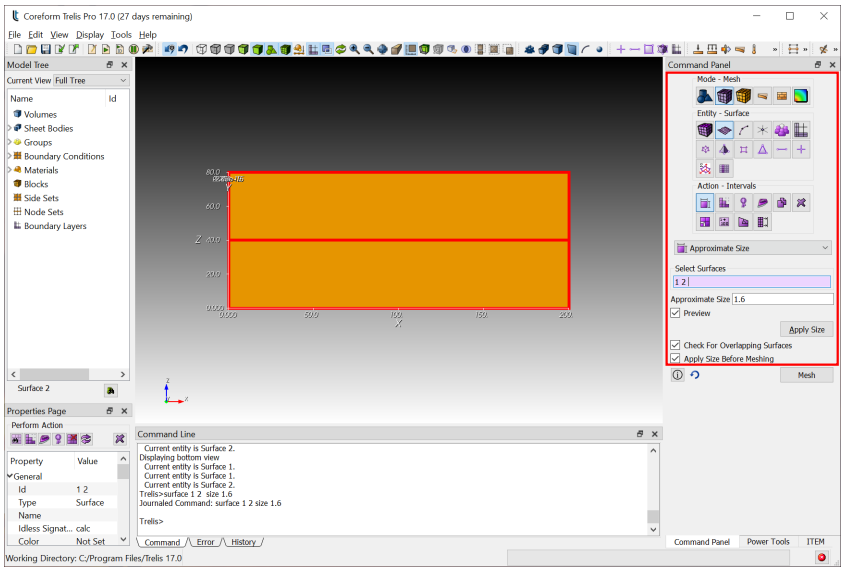
Step 3: Display the axis. Note that the origin is not at the corner of the model.



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



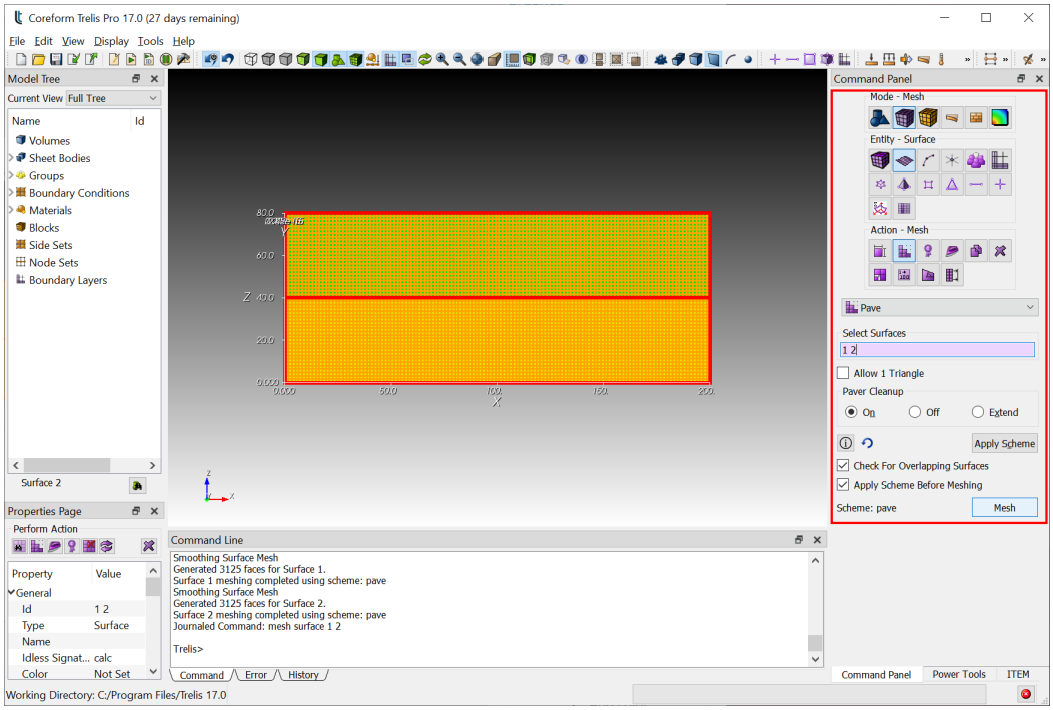
Step 5: Cut the rectangle at z = 40 km to create two layers.



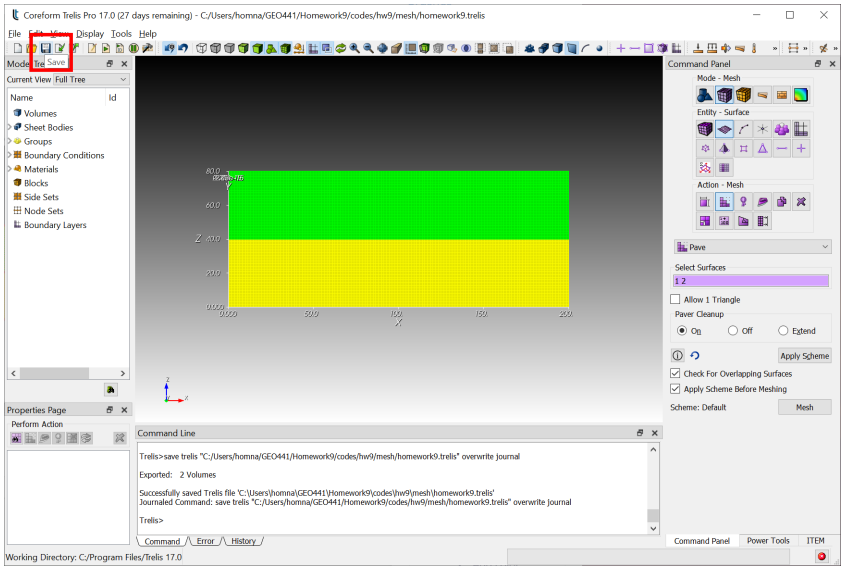
Step 6: First, type the following commands:

```
imprint all
merge all
compress all
```

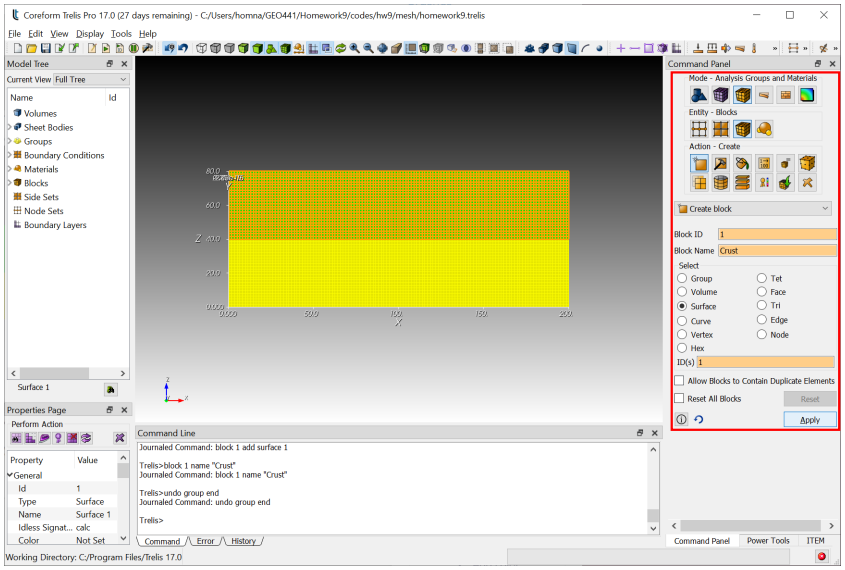
then, define the appropriate mesh size.



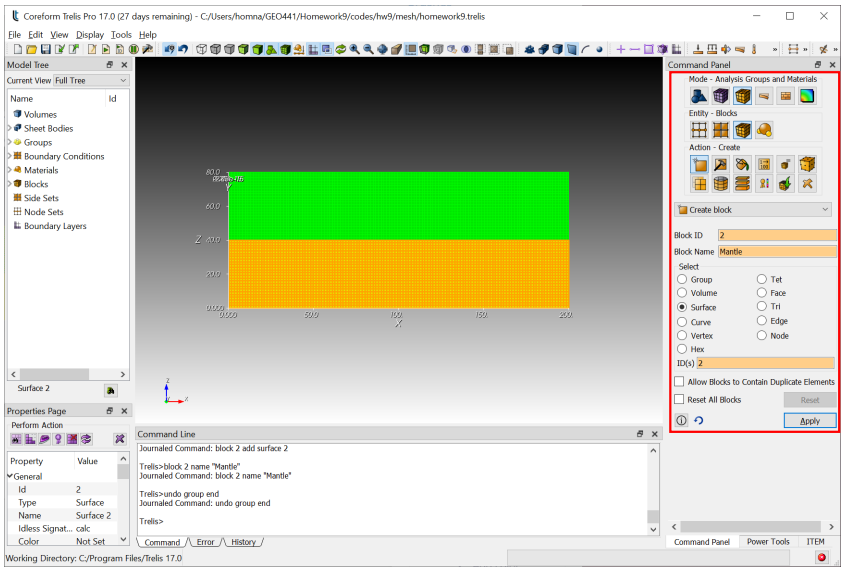
Step 7: Select and apply mesh scheme `Pave` to all surfaces. Then mesh the model.



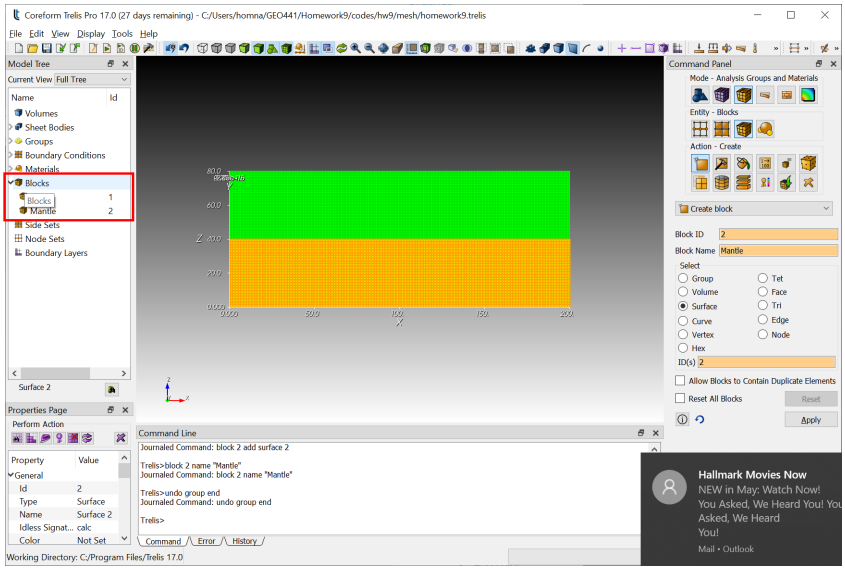
Step 8: Save the file as `Homework9.trelis`.



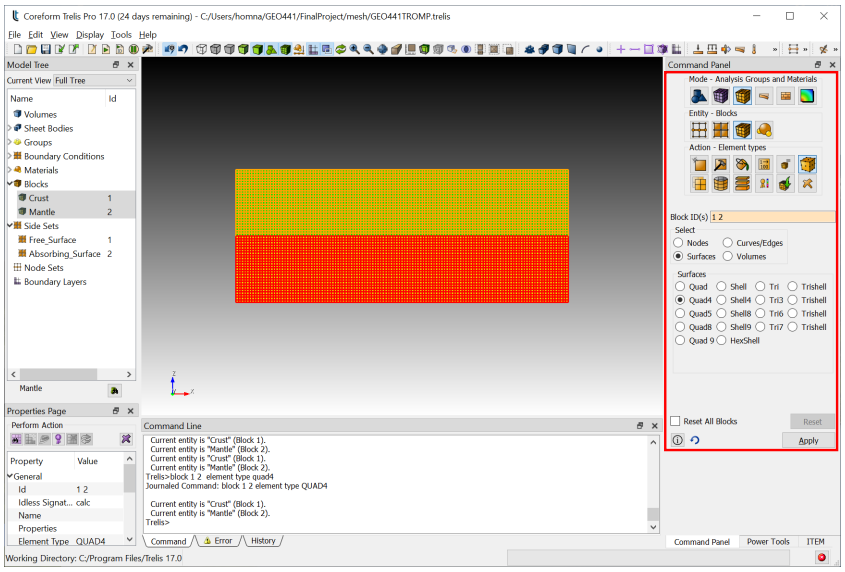
Step 9: Create a block with ID 1 and name `Crust`. Select the top surface in IDs and apply.



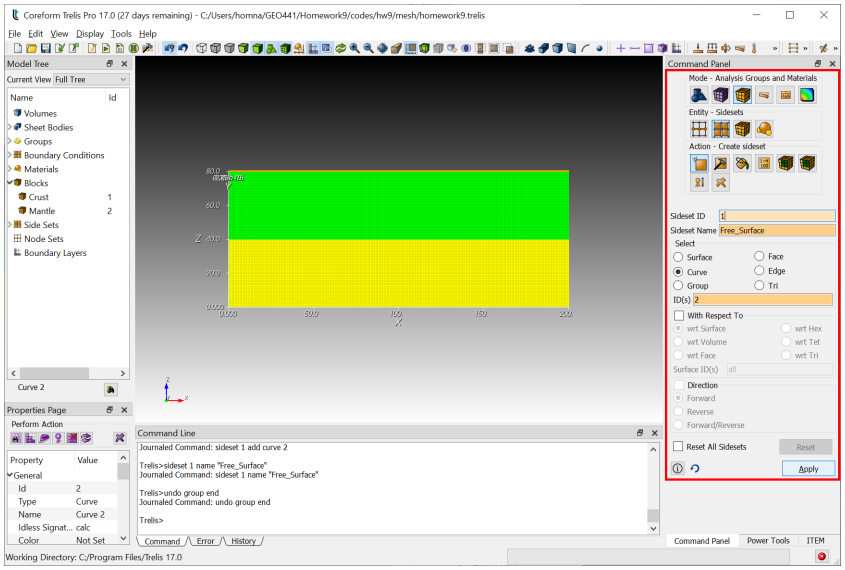
Step 10: Create a block with ID 2 and name `Mantle`. Select the bottom surface in IDs and apply.



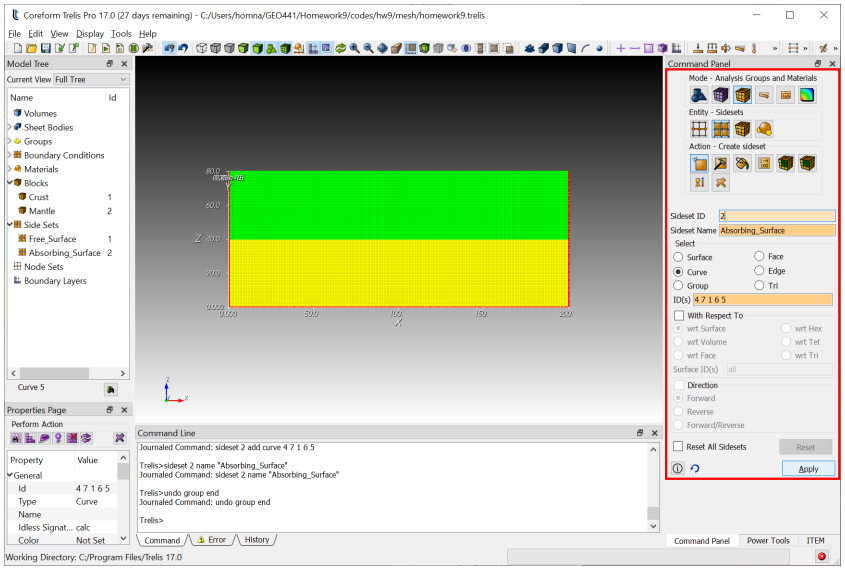
Step 11: Click on `Blocks` on the `Model tree` panel on the left and make sure that you have properly defined the blocks.



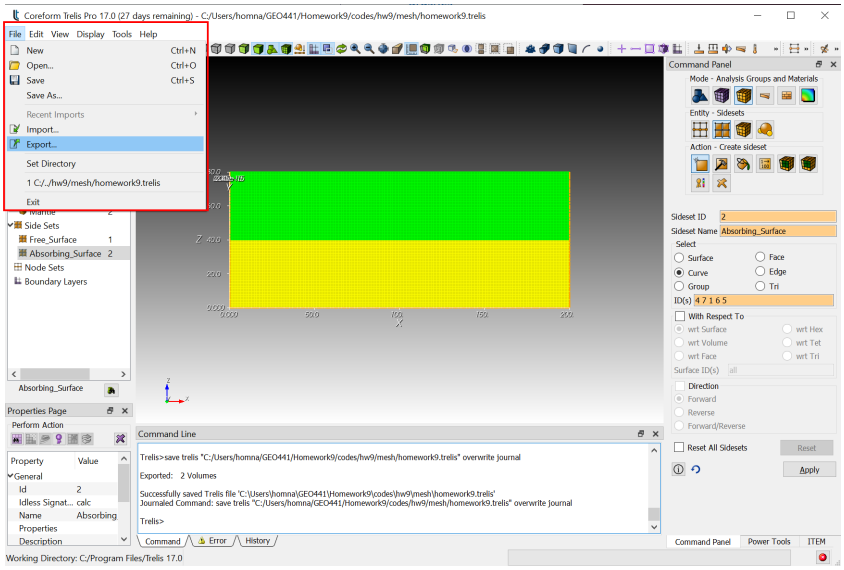
Step 12: Assign element type of all the blocks to `QUAD4`.



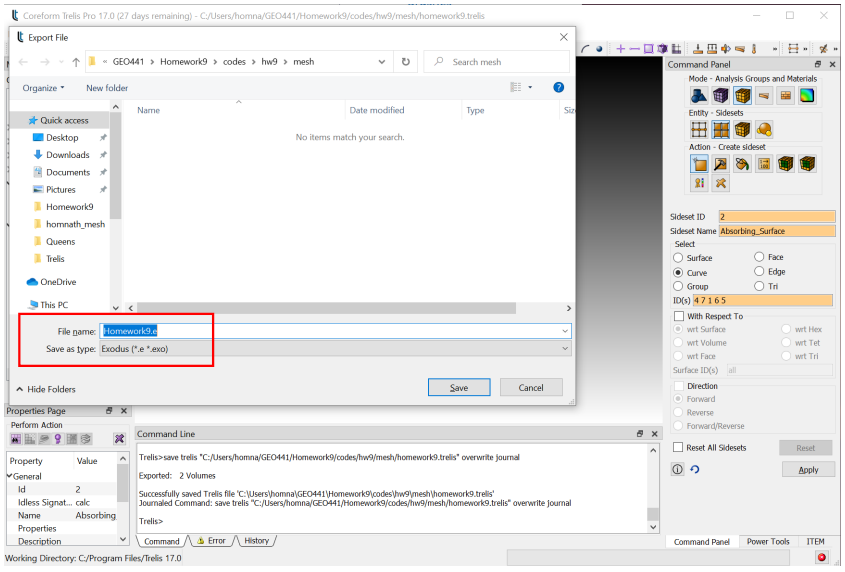
Step 13: Create a sideset with ID 1 and name `Free_Surface`. Select the top curve in IDs and apply.



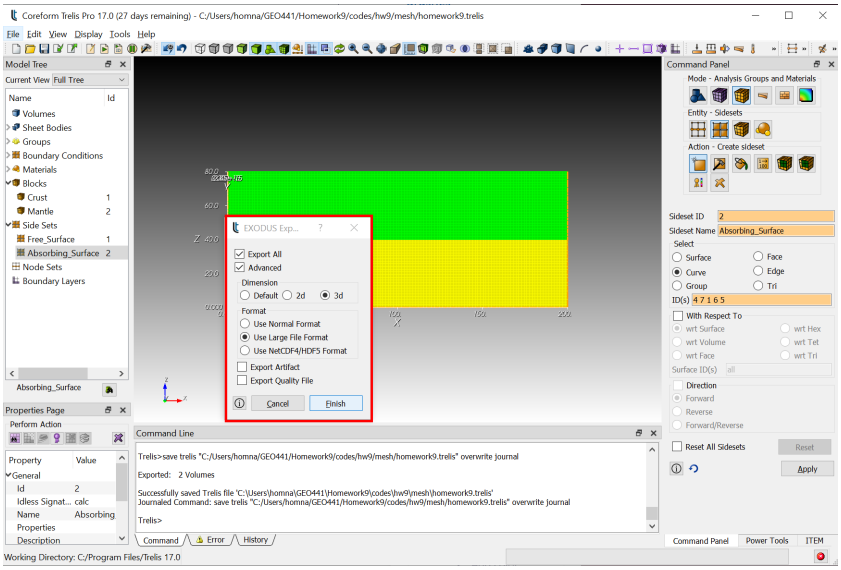
Step 14: Create a sideset with ID 2 and name `Absorbing_Surface`. Select the all left, bottom and right curves in IDs and apply.



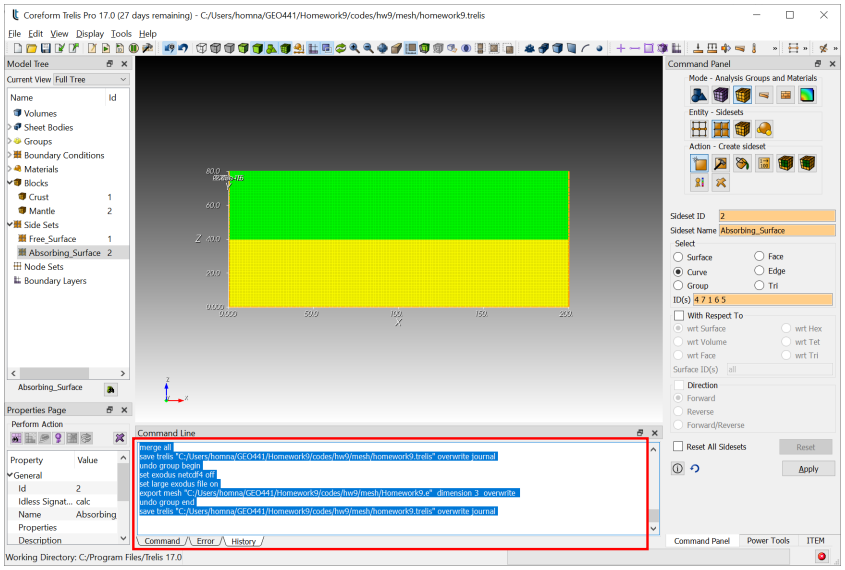
Step 15: Click `File` ⇒ `Export`.



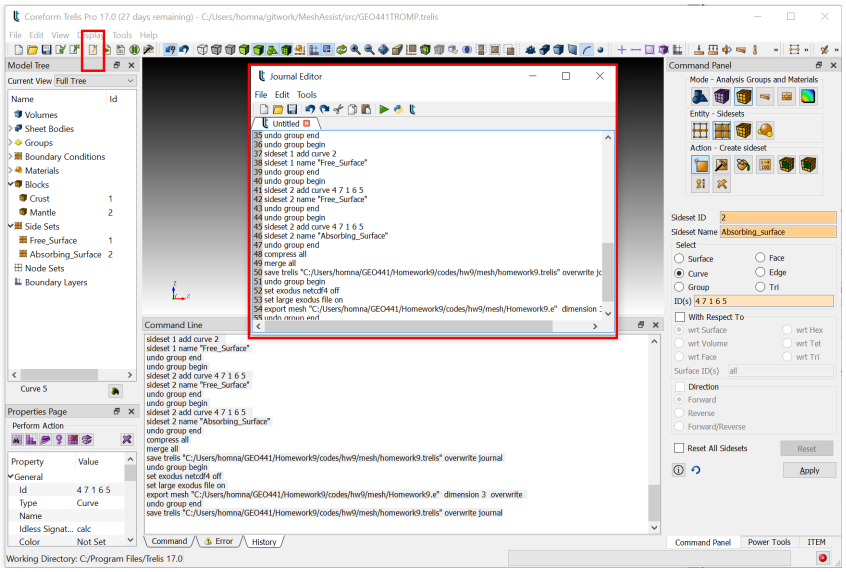
Step 16: Choose the file name `Homework9.e` and select the type `Exodus`.



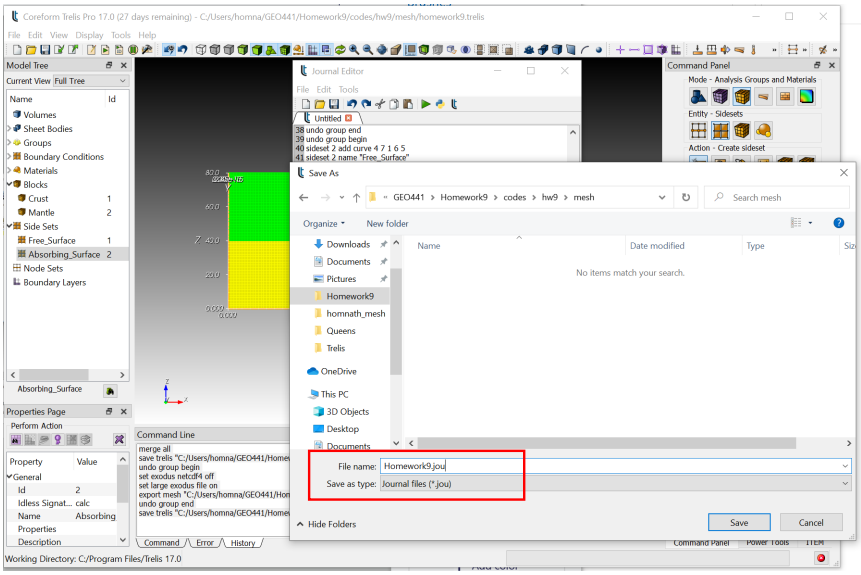
Step 17: Click `Advanced` and select the dimension `3d` and the format `Use Large File Format`. Click `Finish`.



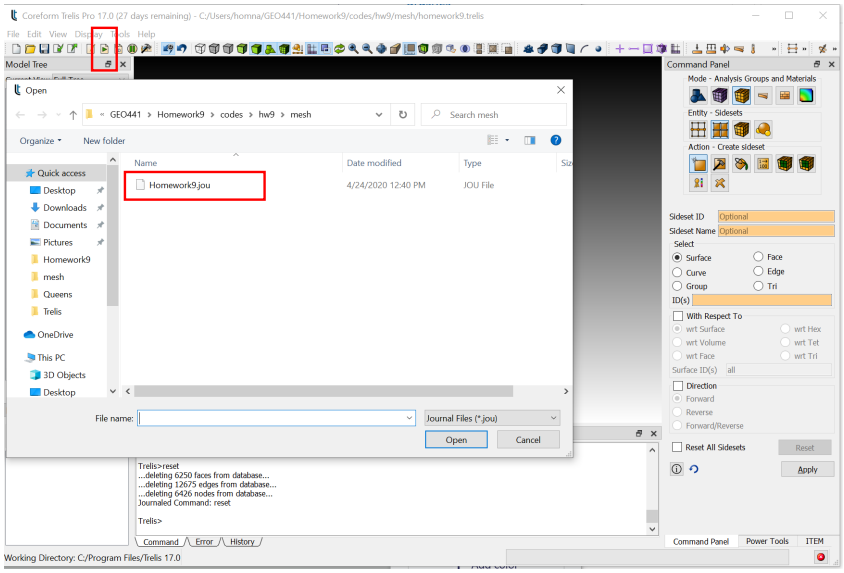
Step 18: For future use, click on `History` on the `Command` panel on the bottom. Select the entire command history and copy.



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 SPECFEM2D 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:

```
exodus2specfem2d 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 `exodus2specfem2d` 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 `SPECFEM2D`, try the following command:

```
exodus2specfem2d 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
```