

3D data visualization tutorial

Step 0: Run the script:

```
meep source-in-vacuum.ctl
```

The CTL file runs a source in vacuum and saves the Ez field values in two different ways:

- 1) To one HDF5 file per time slice. Data format: XYZ
- 2) To a single HDF5 file for all time slices. Data format: XYZT

Three different ways of visualizing the data are presented here:

- 1) Converting each time slice HDF5 file to VTK and then opening them as a group in [Paraview](#) after renaming them (another VTK viewer is [MayaVI](#)).
- 2) Doing the same, but using the single XYZT HDF5 file.
- 3) Creating images and a GIF from the single XYZT HDF5 file using a slice through the central Z layer.

Method 1:

- 1) Convert the separate .h5 files (**source-in-vacuum-ez-*.h5**) to .vtk files:

```
h5tovtk source-in-vacuum-ez-*.h5
```

- 2) Rename them by removing the "." in the time slice numbering:
 - a. source-in-vacuum-ez-000000.10.vtk -> source-in-vacuum-ez-00000010.vtk
 - b. source-in-vacuum-ez-000000.20.vtk -> source-in-vacuum-ez-00000020.vtk
 - c. ...
 - d. source-in-vacuum-ez-000005.00.vtk -> source-in-vacuum-ez-00000500.vtk

On Windows, you can use the *PowerRename* tool to do this quite easily:

<https://docs.microsoft.com/en-us/windows/powertoys/>

- 3) Open those .vtk files as a group in Paraview.

Method 2 and 3:

- 1) For methods 2 and 3, run the *postprocess-single-h5.sh* script:

```
bash postprocess-single-h5.sh
```

- a. It will create .vtk and .png files from the single HDF5 file output **source-in-vacuum-fields.h5** .
- b. It will also create a .gif from the .h5 files.
- c. It will create separate VTK files, that you can directly open as a group in Paraview, as in method 1.