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.

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.