Outline

1. **Installation of FreeFEM++**

2. **Examples**

Outline

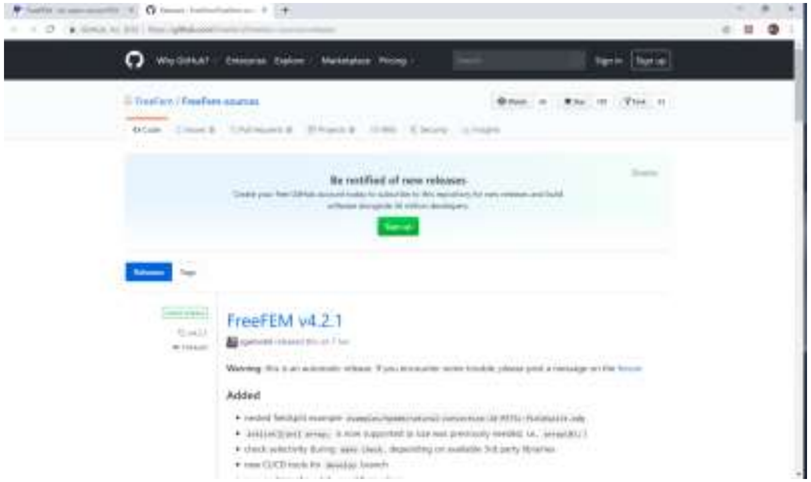
1. **Installation of FreeFEM++**
2. **Examples**

Install FreeFEM++

<u>FreeFEM++</u>: Free software for FEM

- Free
- (Relatively) Easy to implementation
- Easy to connect with C++, Matlab, Paraview, etc···

1. **<u>Go to the main page (</u>**https://freefem.org/**<u>)</u>**
2. **<u>Go to the download page</u>**

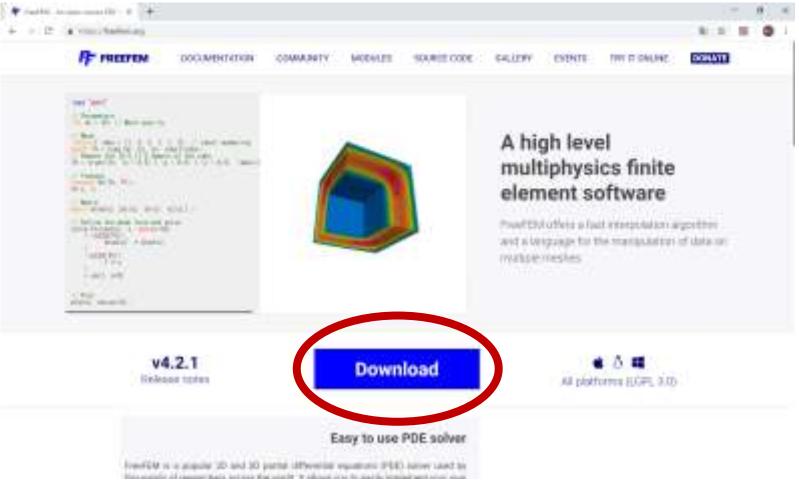


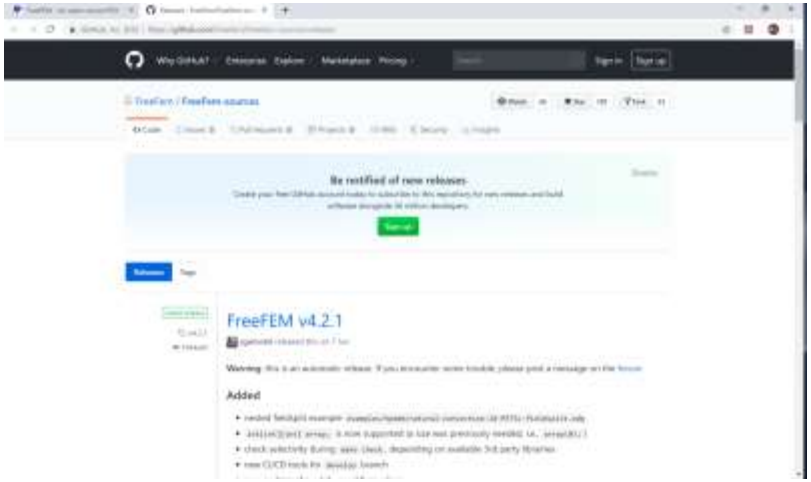
Install FreeFEM++

FreeFEM++: Free software for FEM

- Free
- (Relatively) Easy to implementation
- Easy to connect with C++, Matlab, Paraview, etc···

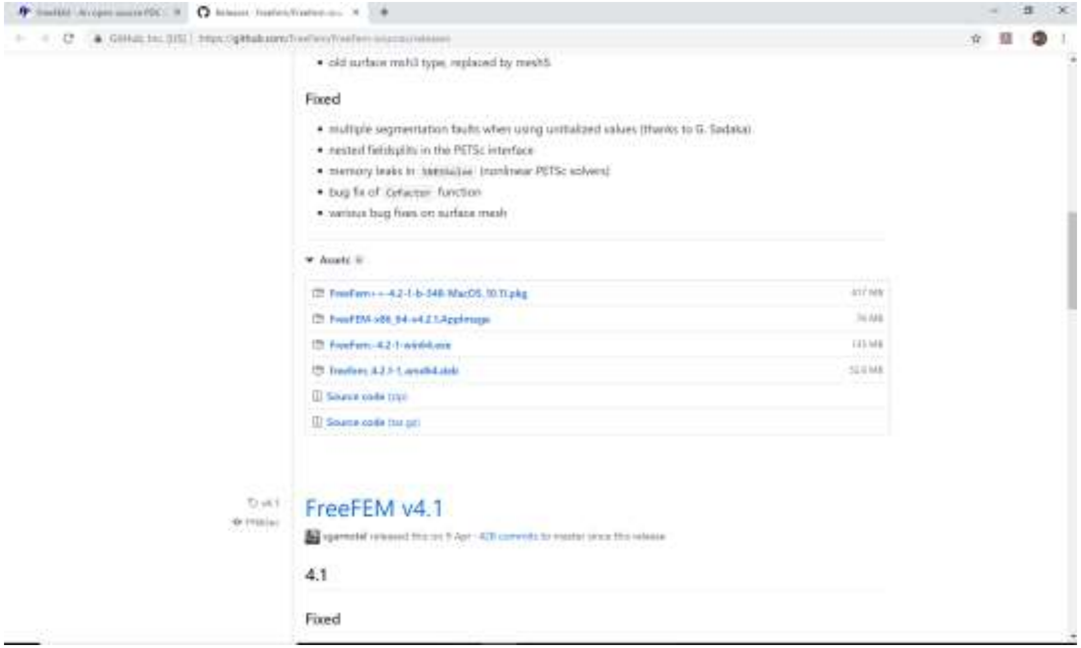
1. **Go to the main page (https://freefem.org/)**
2. **Go to the download page**

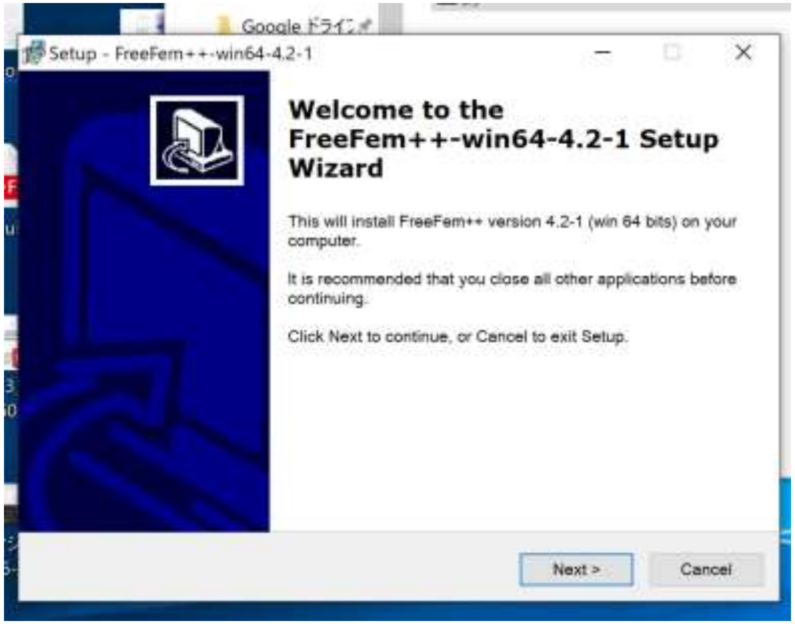




Install FreeFEM++

3. Download a latest version (FreeFEM-4.2.1-win64.exe)
4. Start an installation with .exe file
5. Finish the installation





Outline

1. Installation of FreeFEM++

2. Examples

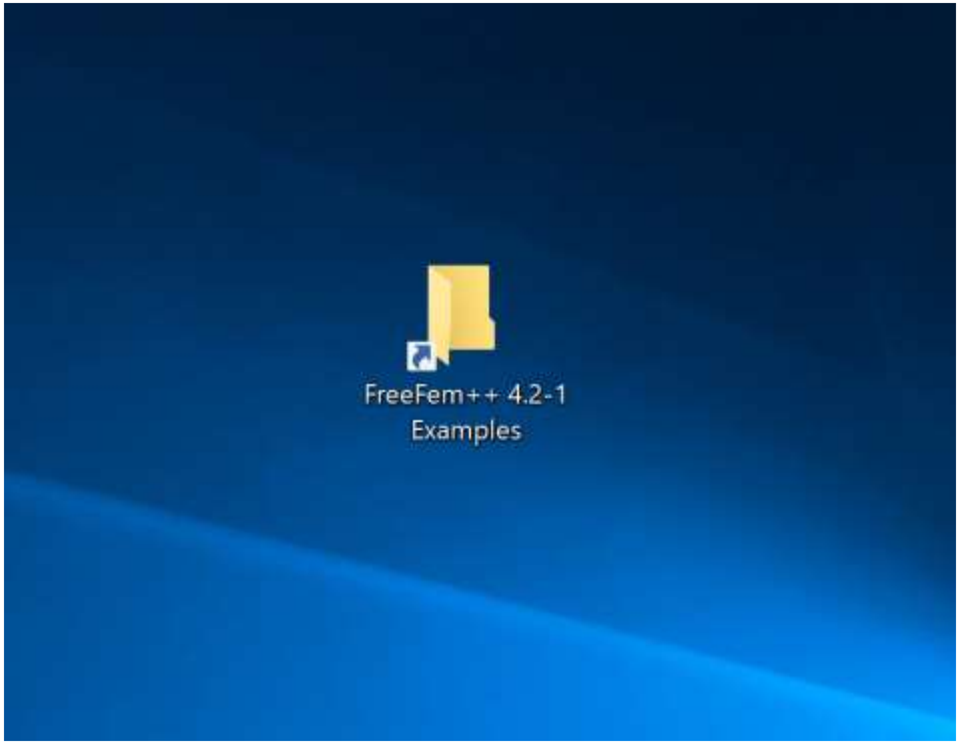
Examples

Try an example

You can find examples in a folder:

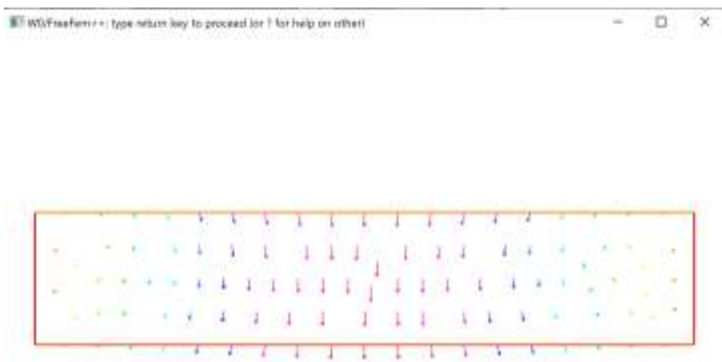
C:¥ProgramData¥Microsoft¥Windows¥Start Menu¥Programs¥FreeFem++¥Examples

You can also jump from the shortcut on your Desktop



Examples

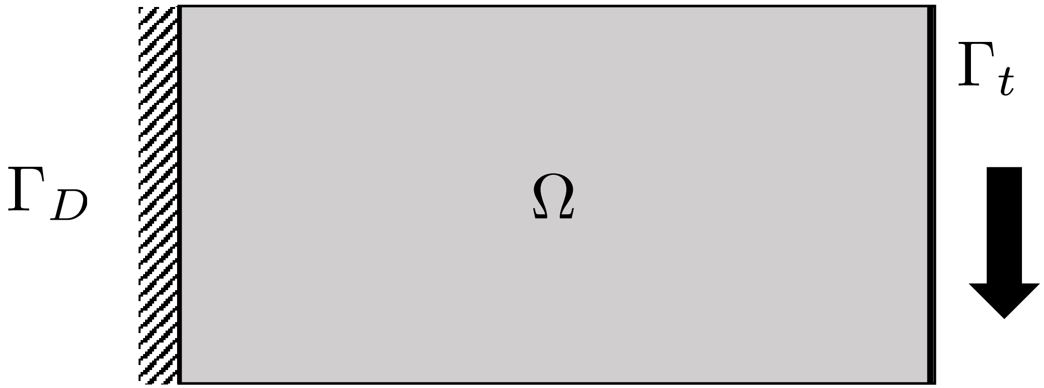
1. Copy ”¥tutorial¥beam.edp” to your Desktop
2. Open your command prompt
3. Change directory to your Desktop
4. Type ”FreeFEM++ beam.edp”



- They provide many examples.

Examples

Solve a linear elasticity problem



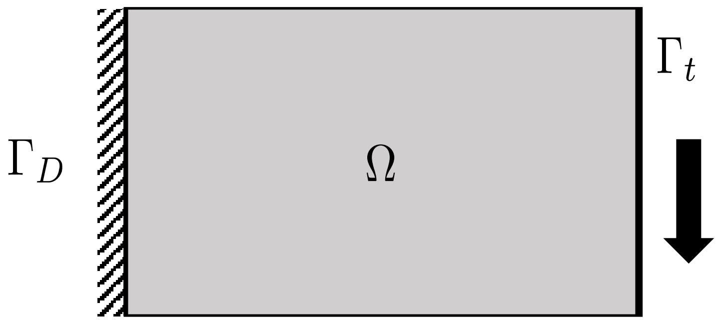
Weak form

$$\int_{\Omega} \sigma(u) : \epsilon(\tilde{u}) \mathrm{d}\Omega = \int_{\Gamma_u} t \cdot \tilde{u} \mathrm{d}\Gamma$$

Examples

Algorithm

1. Define parameters
2. Make a geometry and mesh
3. Define a functional space
4. Define variables
5. Define a linear elastic problem
6. Solve a problem
7. Visualization of your result



Implementation

- Any editors are available (NotePad, Atom, etc···)
  Some editors have a package (Atom, etc···)
- The language is based on C++

Examples

Algorithm

Define parameters

```
/*=========================
      Parameters
 ==========================*/
real E; E = 200.e9; // Young's module
real nu; nu = 0.3; // Poisson's ratio
real lambda; lambda = nu*1./(1+nu)/(1-2.*nu); // Lame coefficient
real mu; mu = 1./2./(1+nu); // Lame coefficient
cout << "mu=" << mu <<endl;
```

Make a geometry and mesh

```
/*=========================
       Make a geometry
  =========================*/
border a(t=0,10){x=t; y=0; label=1;} //bottom
border b(t=0,5){x=10; y=t; label=2;} //right
border c(t=0,10){x=10-t;y=5;label=3;} //top
border d(t=0,5){x=0;y=5-t;label=4;} //left
mesh Sh = buildmesh(a(40)+b(20)+c(40)+d(20));
plot(Sh,wait=1);
```

Examples

Algorithm

Define a functional space

```
// Define functional space
fespace Vh(Sh,[P1,P1]);
```

Define variables and macros

```
//Define Solid Mechanics
macro u[u1,u2]  //EOM displacement vector
macro tu[tu1,tu2] //EOM test function
macro e(u) [dx(u[0]),dy(u[1]),(dx(u[1]) + dy(u[0])  )] //EOM strain tensor
macro D[[2.*mu + lambda, lambda,0],[lambda,2.*mu+lambda,0],[0,0,2.*mu]] //EOM elastic tensor
macro g[0,-1.e3] //EOM traction vector
Vh      u,tu;
```

Define a linear elastic problem

```
//governing equation
problem gov(u,tu) = int1d(Sh,2)(g'*tu) - int2d(Sh)((D*e(u))'*e(tu)*E) + on(4,u1=0,u2=0);
```

Examples

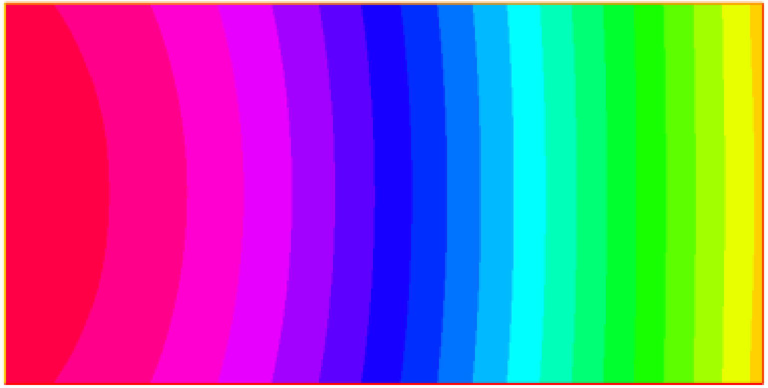
Algorithm

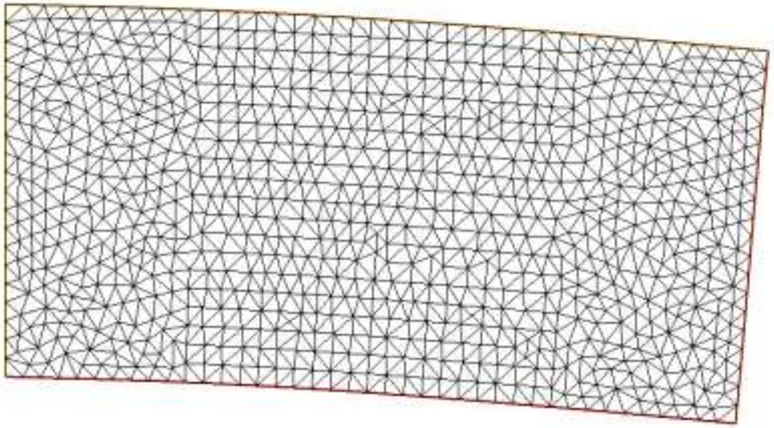
Solve and visualization

```
//SOLVE
gov;
plot(u2,fill=true);

mesh Sh1 = movemesh(Sh,[x+1000*u1,y+1000*u2]);
plot(Sh1,wait=1);
```

Calculation





Examples

Visualization of your result with Paraview

Implement following a code at the top of your edp file.

```
/*=============================
  Initial setting for visualization
  =============================*/
load "iovtk"   // use vtk file for paraview
```

Implement following codes after a calculation.

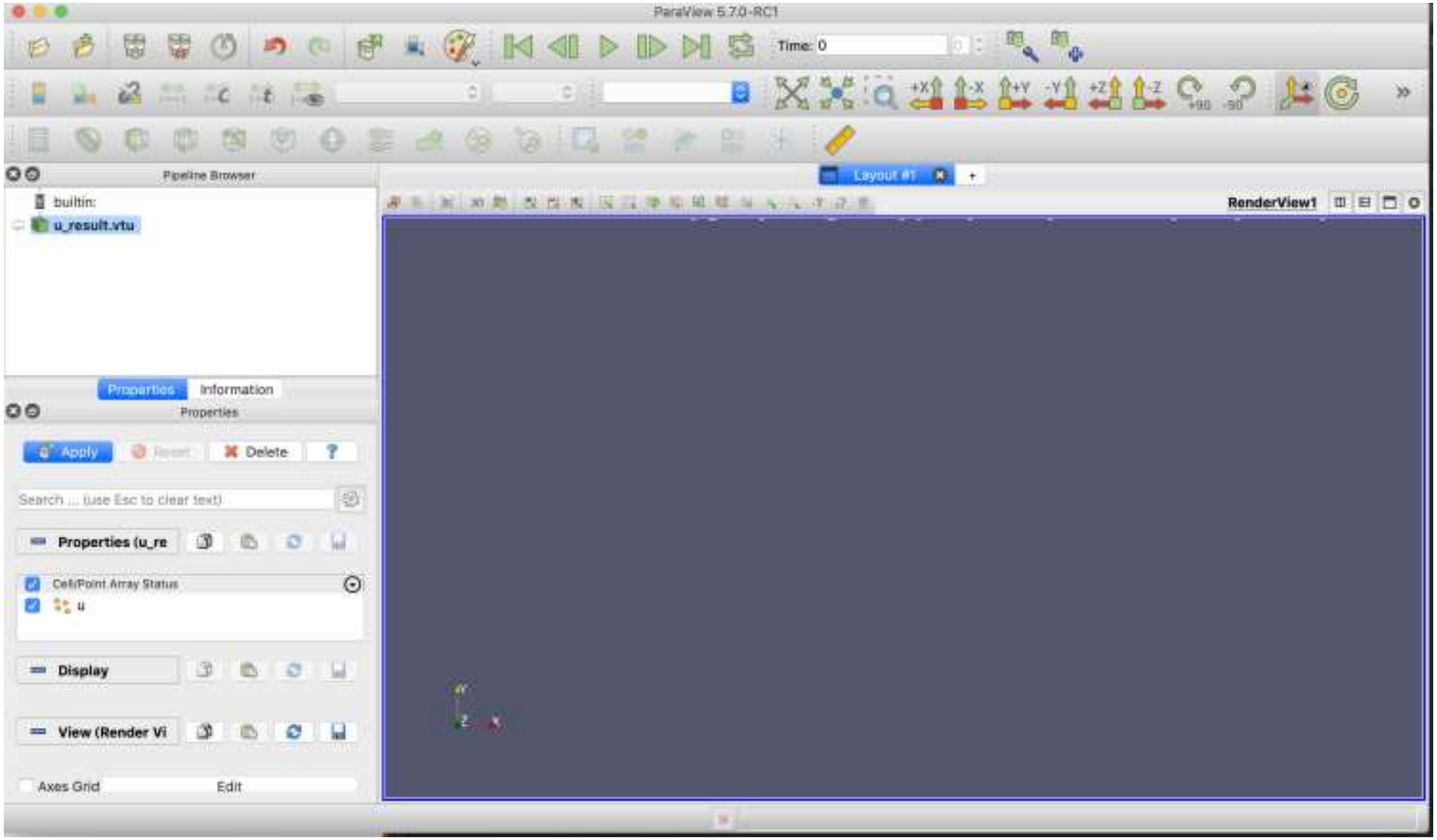
```
//For visualization with Paraview
int[int] Order = [1];
string DataName = "u"; // define a variable for save as .vtu file
savevtk("u_result.vtu", Sh, [u[0],u[1],0], dataname=DataName, order=Order);
```

Now, you save your result as “u_result.vtu”.

Examples

Visualization of your result with Paraview

Open your Paraview and open "u_result.vtu”.

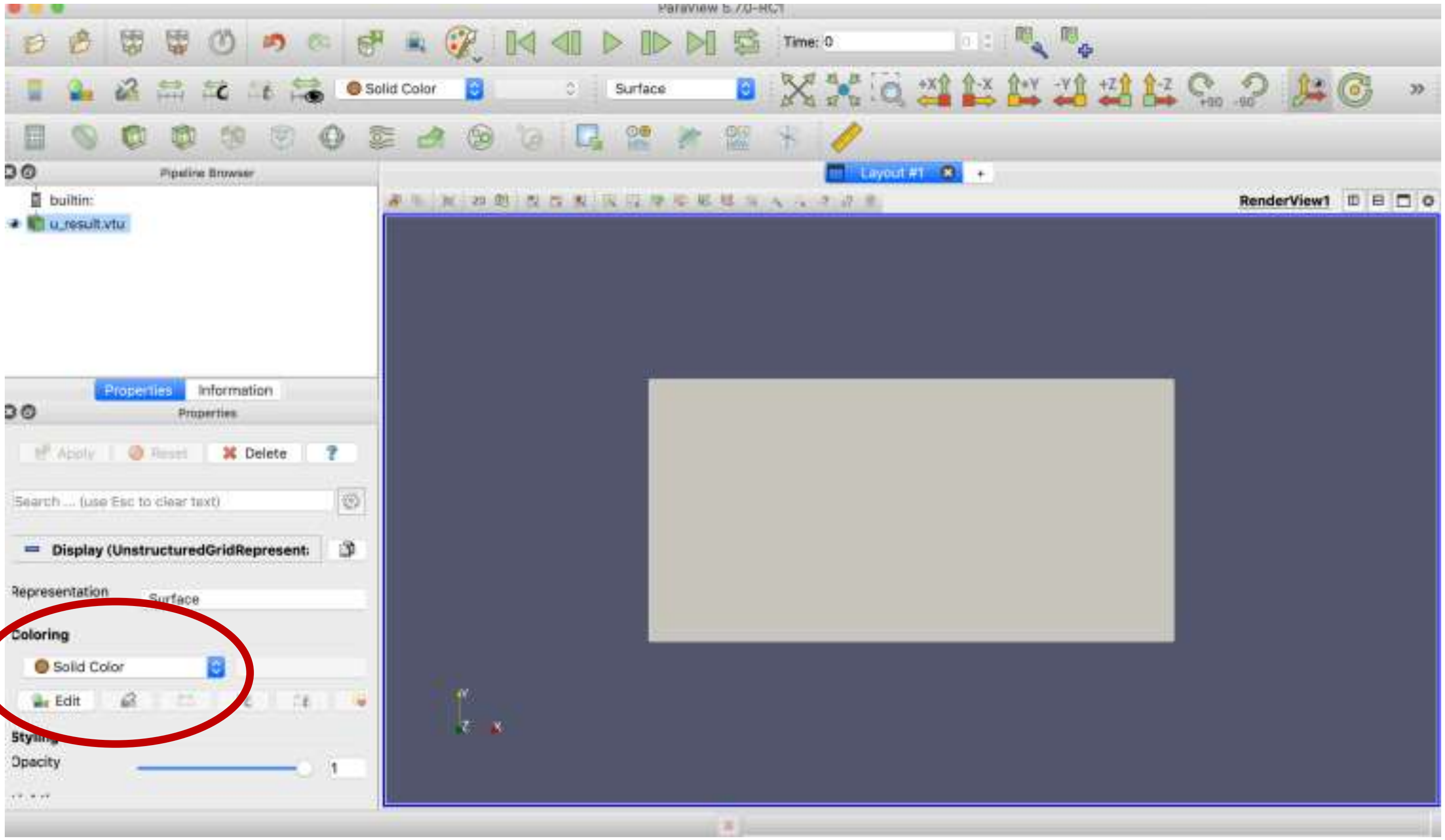


Select the file and click a tab, “Apply”.

Examples

Visualization of your result with Paraview

Change “Solid Color” to “u” in the left window.

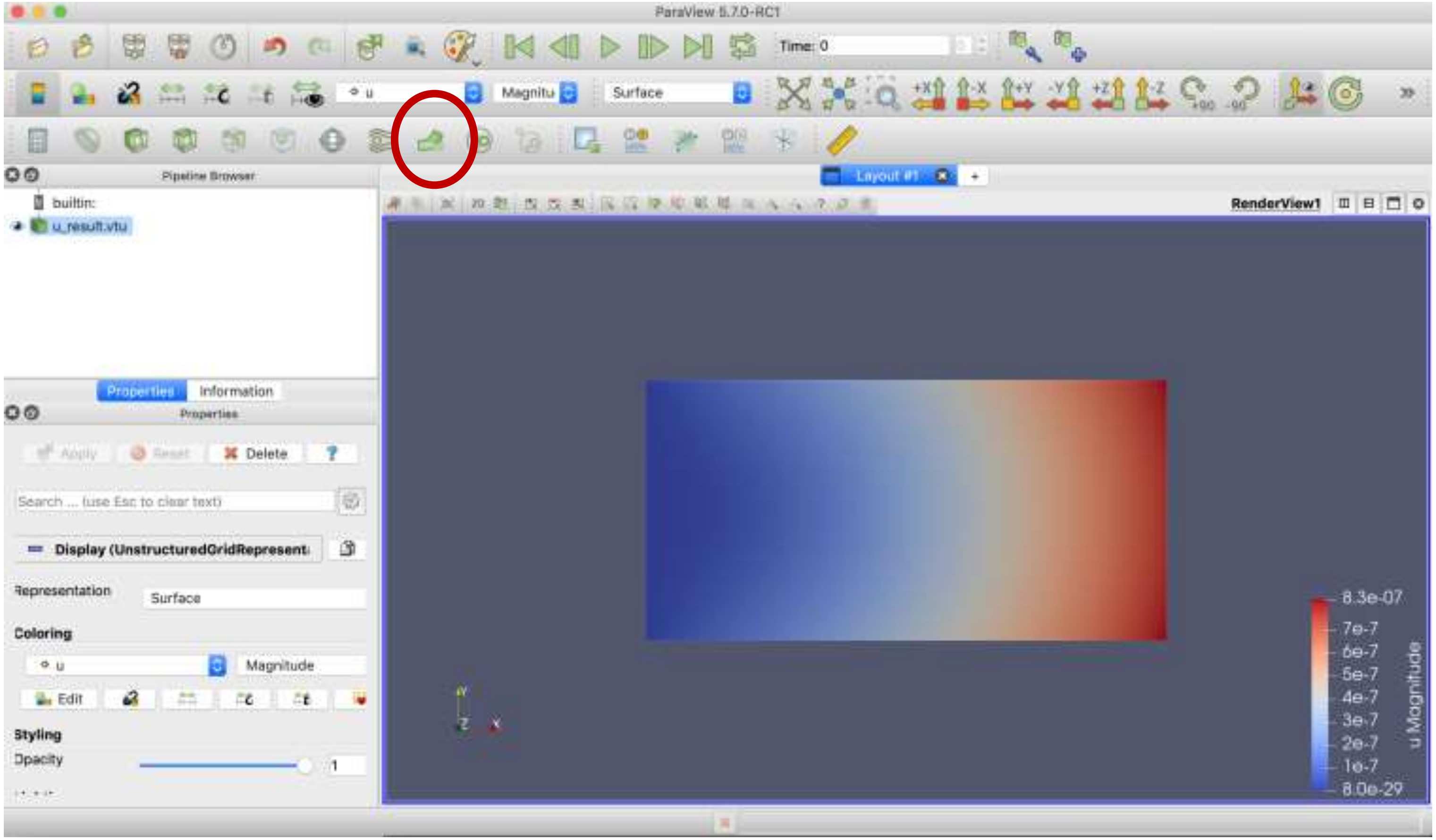


Click a tab, “Apply”.

Examples

Visualization of your result with Paraview

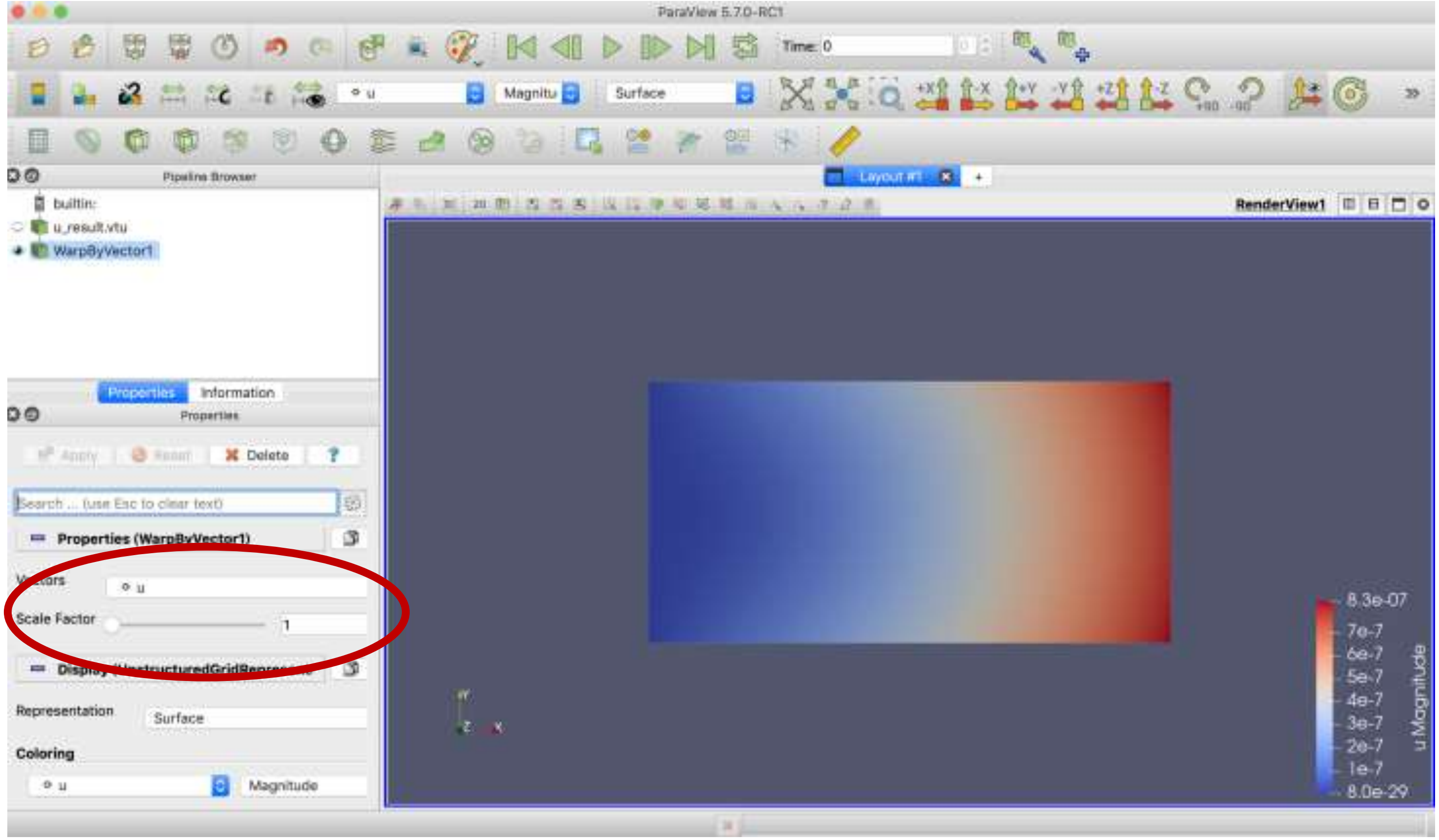
Click the deformation icon as the following figure.



Examples

Visualization of your result with Paraview

Apply “WarpByVector1” as you did for ”u_result.vtu”.

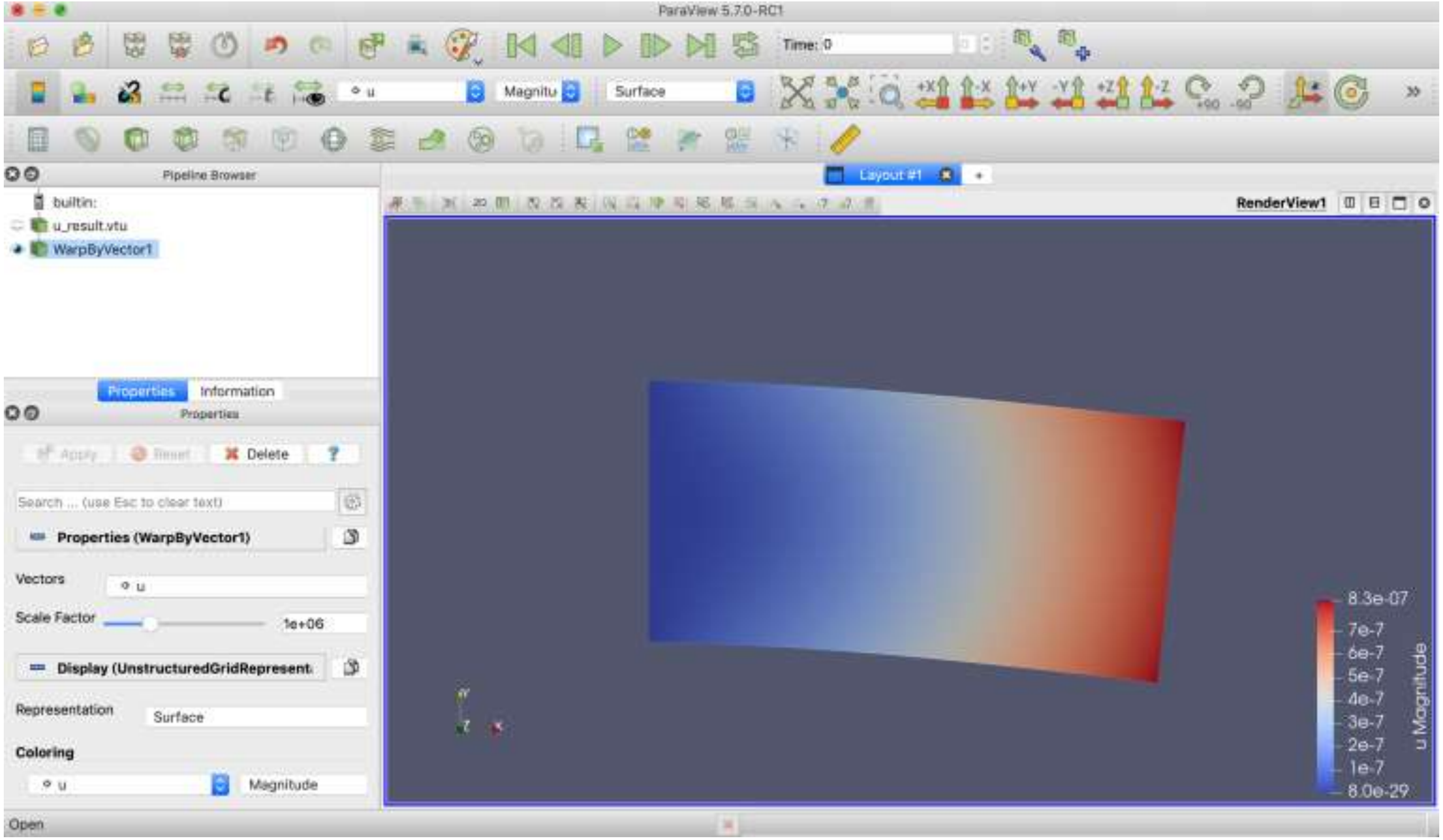


You can change a deformation scale by “Scale Factor”.

Examples

Visualization of your result with Paraview

Click “Apply” again and finish the visualization.



You can find many introductions for Paraview on the internet.