

Initiation to PAM-RTM

Law of Darcy

Application: central injection of a square plate

Objective

There are two goals for this exercise. It consists first of all in the learning of the software PAM-RTM. For this reason, the explanations relating to the software will be detailed and embellished with numerous screenshots.

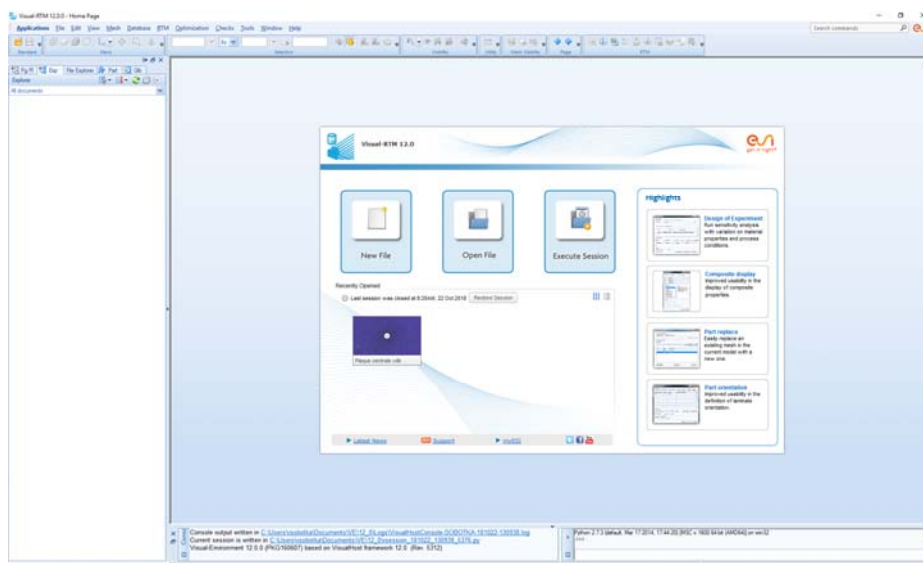
The second objective concerns the study of the law of Darcy and more specifically the comparison between analytical results given by the law and those obtained from simple simulations. To reach this objective, a simple configuration will be studied in this exercise. It will consist of the central filling of a 2D preform. The simple geometry of the piece will make it possible to compare the numerical results with the analytical results deduced from Darcy's law.

Before you start

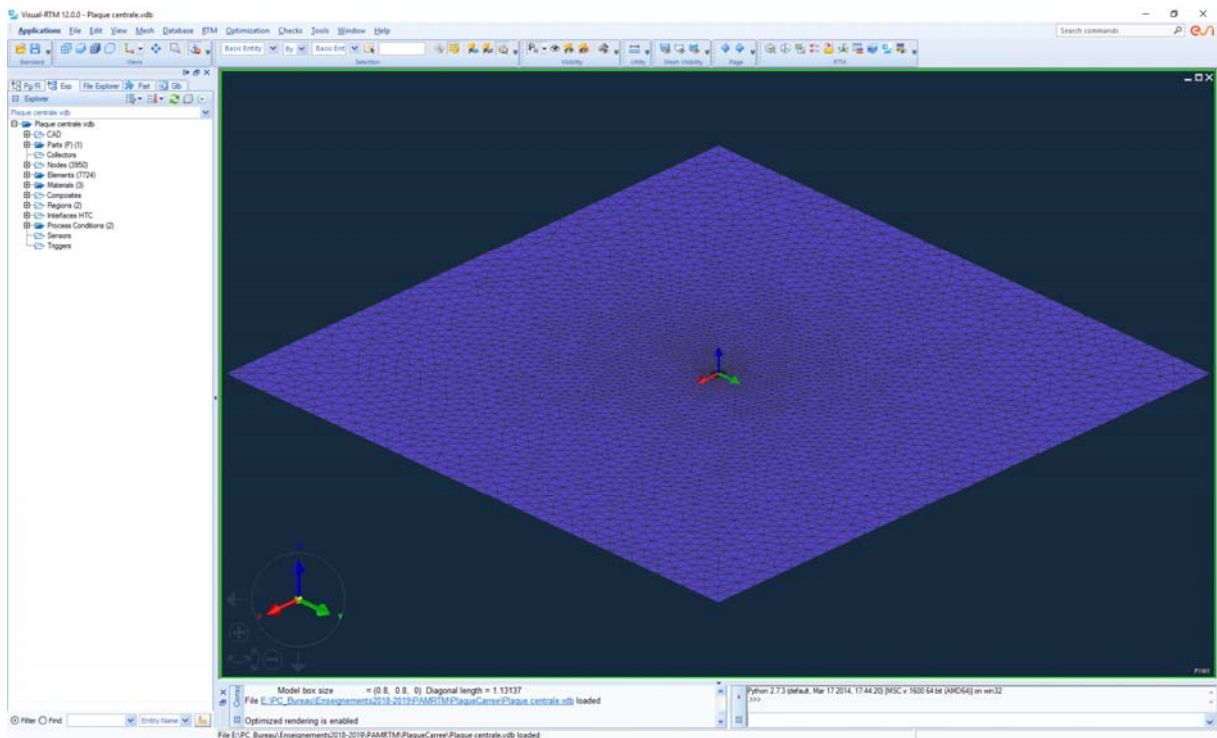
- Create a local folder on C:\. The software won't work if you try to work on Z:\.
- In your file name: no blank or accentuation
- Get the mesh (**Plaque centrale.vdb**) from Madoc and copy it in your directory

PAM-RTM Software

Launch **Visual-RTM 15.0**



Open the file: **Plaque centrale.vdb**



The part is a square with a **side of 80 cm**.

To perform the simulation, you need:

- ✓ To define material properties.
- ✓ To associate a material with a geometry.
- ✓ To define the location of your boundary conditions
- ✓ To define the values of the boundary conditions.
- ✓ To define the type of simulation you want to perform.

In this first exercise, the goal is to perform an isothermal injection.

Definition of the material properties

You have to create a new **resin** and a new **reinforcement**

Material Database

File Database Unit

All

Search

Public User Model

All

Resin

Fiber

Reinforcement

Ply

Core

Mold

Name	Resin_1			
Owner Name	vsobotka			
Last Modified By	vsobotka			
Last Modified	2018-10-22			
Description				

General Thermal Chemical +/-

Property	Type	Value	Value Unit	F(*) Unit
Density	Const.		kg/m ³	
Newtonian Viscosity	Const.		N-s/m ²	

Material Database

File Database Unit

All

Search

Public User Model

All

Resin

Fiber

Reinforcement

Ply

Core

Mold

Name	Reinforcement1			
Owner Name	vsobotka			
Last Modified By	vsobotka			
Last Modified	2016-11-28			
Description				

General Thermal +/-

Property	Type	Value	Value Unit	F(*) Unit
Density	Const.		kg/m ³	
Permeability, 1st principal axis	Const.		m ²	
Permeability, 2nd principal axis	Const.		m ²	
Permeability, 3rd principal axis	Const.		m ²	
Permeability Angle	Const.		rad	
Compressibility				
Fiber Content	F(Shear Angle, vF0)			

The properties of the material are given below:

Reinforcement

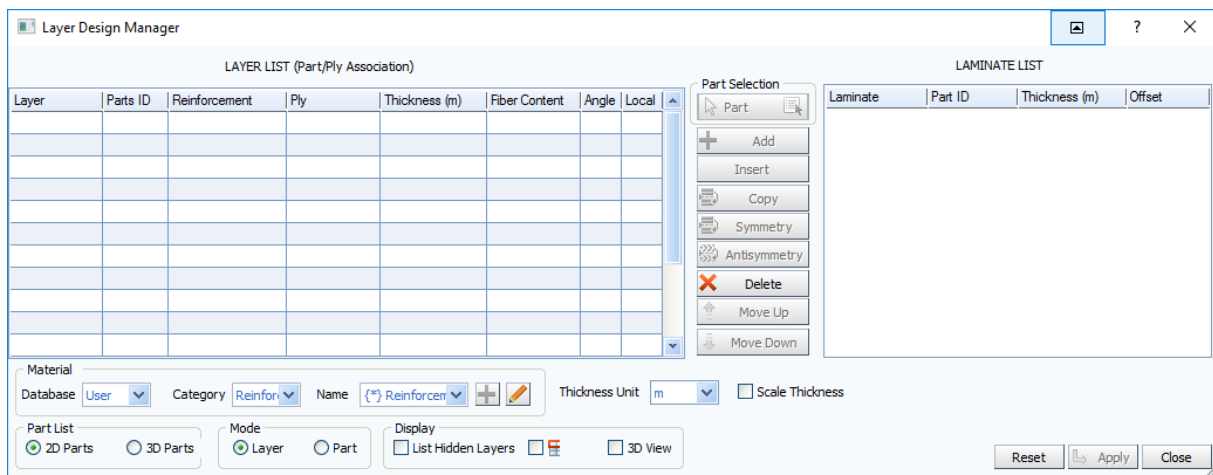
The reinforcement used is a glass fabric whose density is equal to **2560kg.m⁻³**. We consider an isotropic material whose permeability is equal to **K=10⁻¹⁰m²**.

Resin

The density of the resin equal to **1200 kg.m⁻³**. The viscosity of the resin taken constant and equal to **0.12Pa.s**.

Apply reinforcement to geometry

Open *Layer Design Manager*



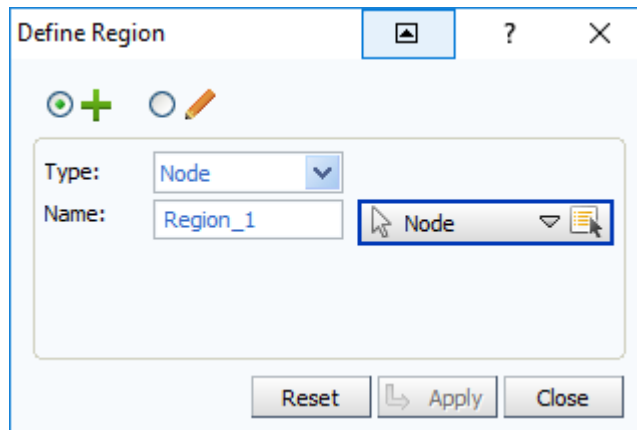
You have to select the part ID associated with the reinforcement you have created.

We will choose a thickness of 5mm for the part and the porosity of 40%.

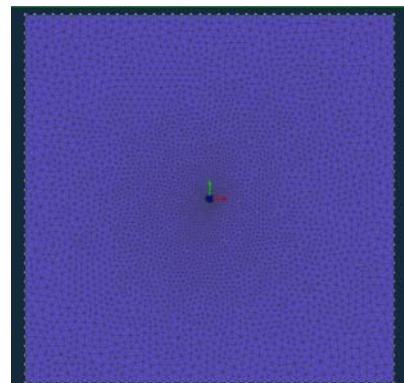
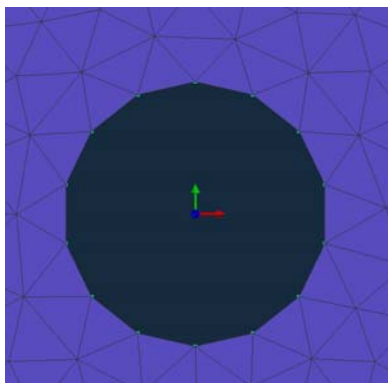
Define of region for boundary conditions

In this part, you need to create 2 regions:

- ✓ 1 for the injection zone
- ✓ 1 for the vent zone



Select the nodes delimiting the region, then click Apply. Repeat the same operation to create a 2nd region for the vent that is located all around the square.



Boundary conditions

In this 1st simulation, we consider an injection at constant pressure of 1.5bar in the center of the part. The vent is put under vacuum.

Process Condition

Definition
Type: Pressure

Process Name: Pressure_1

Selection
Region

Trigger

Database:

Property	Type	Value	Value Unit	F(t) Unit
Pressure	Const.		Pa	
Temperature	Const.	300	K	
State	Const.	1		
Flowrate Corr...				
Tube Length	Const.	0	m	
Tube Radius	Const.	0	m	

Reset Apply Close

Process Condition

Definition
Type: Vent

Process Name: Vent_1

Selection
Region

Trigger

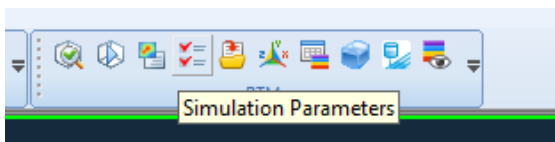
Database:

Property	Type	Value	Value Unit	F(t) Unit
Vent Pressure	Const.	0	Pa	
State	Const.	1		

Reset Apply Close

Simulation parameters

Click on the following button to define the type of simulation as well as the injected resin.



Simulation Parameters

Simulation Type
☐ Preheating
☒ Filling
☐ Heated Filling
☐ Curing

☐ Detect Air Traps

☐ Gravity

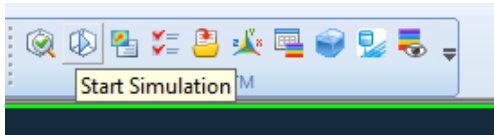
Resin Selection
 Database : User
 Injected Resin : {F} Resin_1

Output Frequency
 Type : % Fill
 Frequency : 2.5

Advanced
 Reset Apply Close

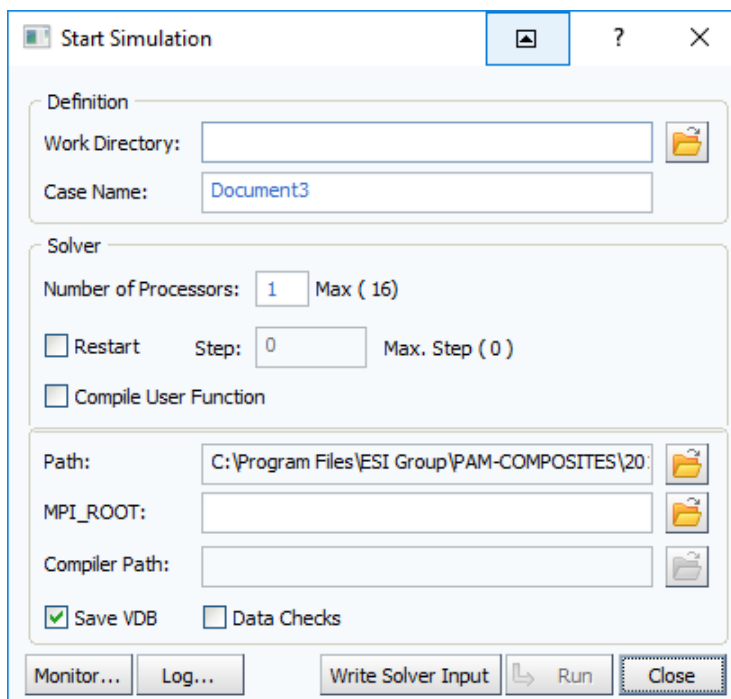
Launch simulation

Click on the following button to open the “start simulation” panel.



Ensure the path used for the solver is well indicated:

C:\Program Files\ESI Group\PAM-COMPOSITES\2019.5\RTMSolver\bin



Then click Run and Log.

The following windows opens. Explain the meaning of the different information.

```

[Icon] Plaque centrale [16.1 KB] - WinTail

mpirun: IBM Platform MPI Community Edition:
09.01.02.01W RTM [11530] Windows 32

(C) Copyright Platform Computing Inc., an IBM Company 1997-2014.
US Government Users Restricted Rights:
- Use, duplication or disclosure restricted by
  GSA ADP Schedule Contract with IBM Corp.
Compatible IBM Platform MPI Remote Launch Service version V02.00.00

WARNING: No cached password or password provided.
         use '-pass' or '-cache' to provide password
Intel(R) C++ 64 Compiler XE for applications running on Intel(R) 64, Version 12.1 us

PAM-RTM Version 2016.0, Parallel, Compiled on Jun  7 2016 at 03:43:50
Copyright ESI Group (France) 2004-2016

Executable adapted for MPI-Wrapper MPM!

Running on 16 processors

Calculation is running. It can be aborted by closing this window

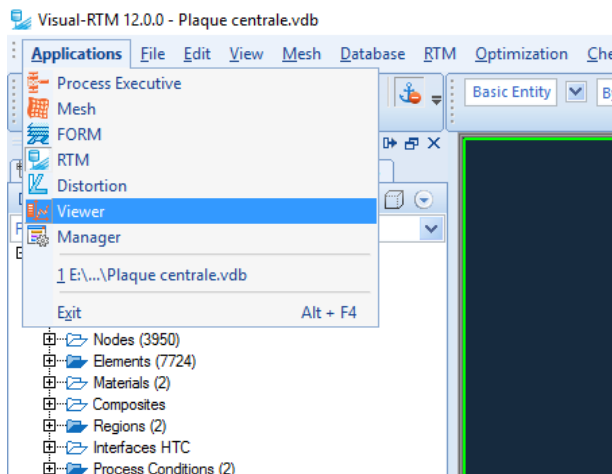
step =1, time=1.000000e-04, DT=1.000000e-04, filled=0, percent= 0.00%
step =2, time=1.102228e-04, DT=1.022280e-05, filled=0, percent= 0.00%
step =3, time=1.161597e-04, DT=5.936859e-06, filled=0, percent= 0.00%
step =4, time=1.220965e-04, DT=5.936859e-06, filled=3, percent= 0.04%
step =5, time=1.323193e-04, DT=1.022280e-05, filled=13, percent= 0.17%
step =6, time=1.453442e-04, DT=1.302489e-05, filled=24, percent= 0.31%
step =7, time=1.595505e-04, DT=1.420624e-05, filled=28, percent= 0.36%
step =8, time=1.781608e-04, DT=1.861035e-05, filled=34, percent= 0.44%
step =9, time=1.997092e-04, DT=2.154844e-05, filled=39, percent= 0.50%
step =10, time=2.268113e-04, DT=2.710212e-05, filled=52, percent= 0.67%
step =11, time=2.567713e-04, DT=2.996001e-05, filled=65, percent= 0.84%
step =12, time=2.941196e-04, DT=3.734832e-05, filled=76, percent= 0.98%
step =13, time=3.377694e-04, DT=4.364972e-05, filled=85, percent= 1.10%
step =14, time=3.863375e-04, DT=4.856813e-05, filled=96, percent= 1.24%
step =15, time=4.398997e-04, DT=5.356217e-05, filled=114, percent= 1.48%
step =16, time=4.964806e-04, DT=5.658090e-05, filled=122, percent= 1.58%
step =17, time=5.651433e-04, DT=6.866273e-05, filled=136, percent= 1.76%
step =18, time=6.476884e-04, DT=8.254512e-05, filled=150, percent= 1.94%
step =19, time=7.428658e-04, DT=9.517745e-05, filled=161, percent= 2.08%
step =20, time=8.389189e-04, DT=9.605305e-05, filled=186, percent= 2.41%
step =21, time=9.547737e-04, DT=1.158548e-04, filled=199, percent= 2.58%
step =22, time=1.082713e-03, DT=1.279397e-04, filled=208, percent= 2.69%
step =23, time=1.219009e-03, DT=1.362951e-04, filled=226, percent= 2.93%
step =24, time=1.369404e-03, DT=1.503951e-04, filled=247, percent= 3.20%
step =25, time=1.553363e-03, DT=1.839593e-04, filled=262, percent= 3.39%
step =26, time=1.742847e-03, DT=1.894844e-04, filled=281, percent= 3.64%
step =27, time=1.945439e-03, DT=2.025922e-04, filled=300, percent= 3.88%
step =28, time=2.156883e-03, DT=2.114439e-04, filled=324, percent= 4.19%
step =29, time=2.369759e-03, DT=2.128753e-04, filled=335, percent= 4.34%

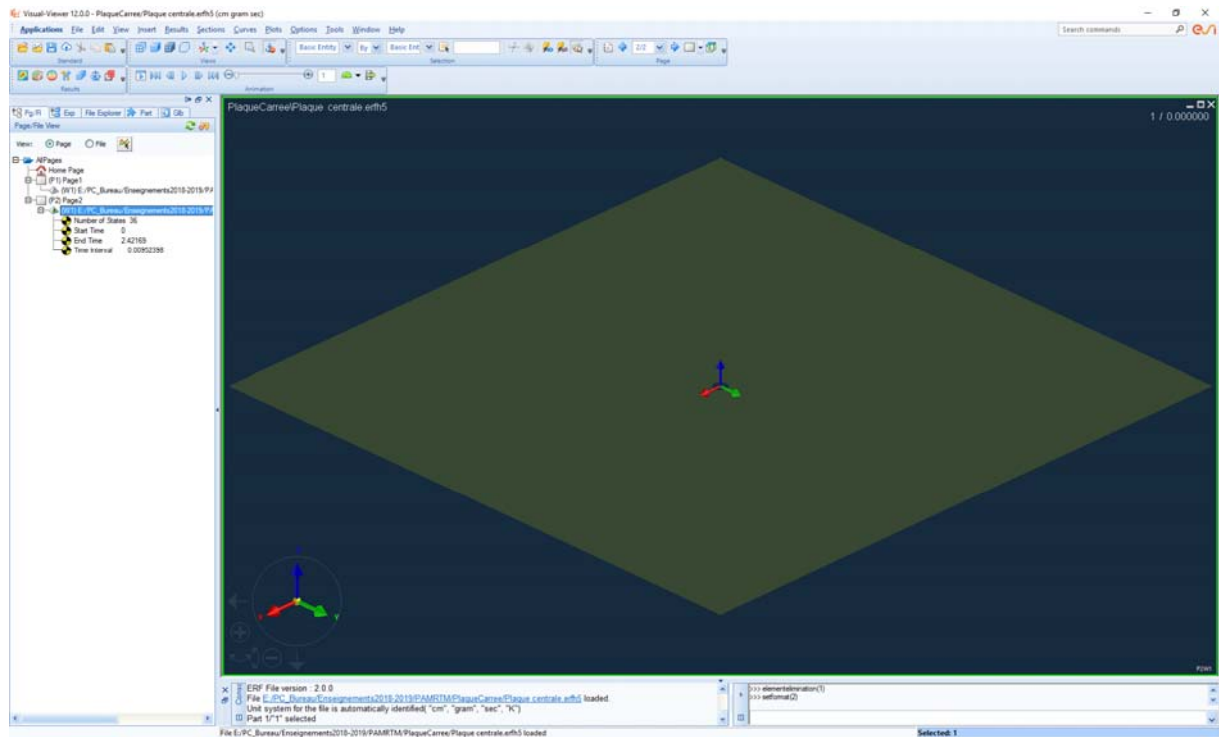
```

Visualization of results

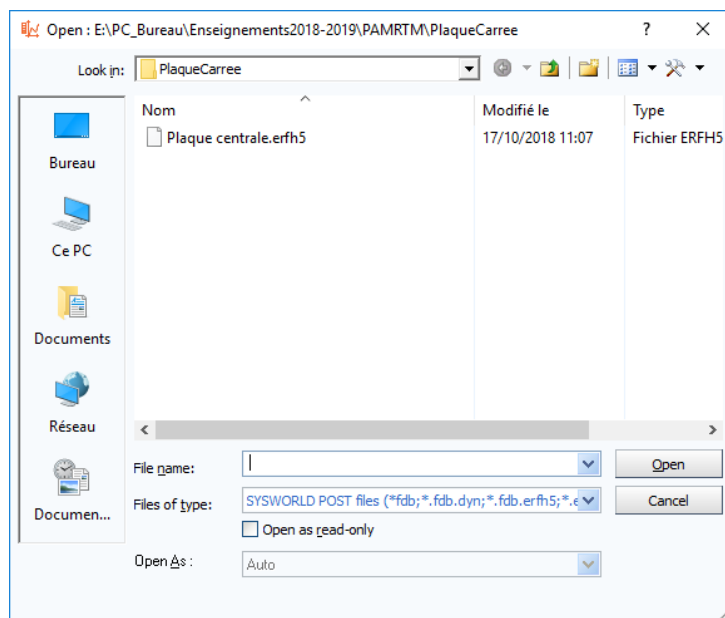
After the end of the simulation, you can check the results.

You have to open the Viewer

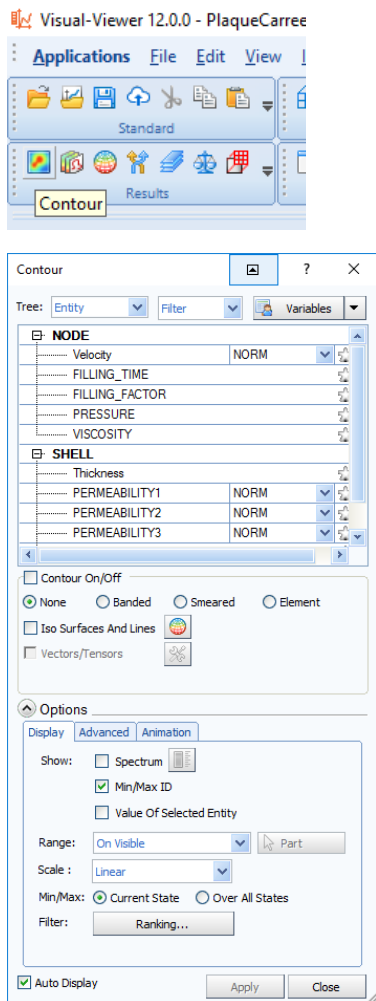




Load your result file: its extension is *.erfh5*



Then click on the *contour* button and visualize the different fields.



Post-processing files

In your work directory open with Excel (or any spreadsheet software) the two files:

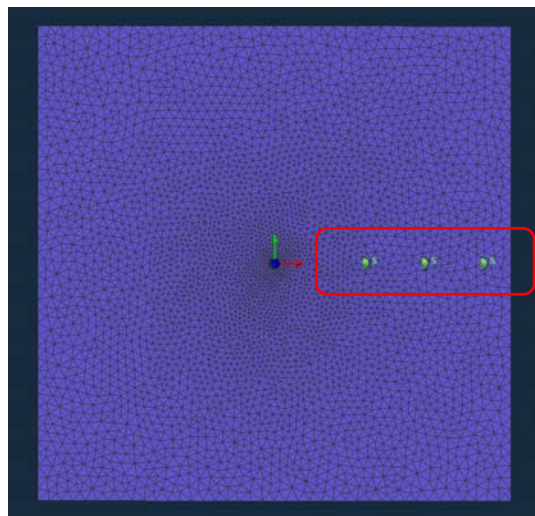
filename_BC1.out and *filename_BC1.out*

These files gather information on the boundary conditions of your simulation.

Plot the different information and comment.

Work to do

1. Second simulation: start over your simulation with closed Vents. Comment the results and post-processing files.
2. Third simulation: start over the simulation by positioning sensors in your mesh.



- Compare the position of the flow front with the one given analytically by the law of Darcy.
3. Repeat the same simulation with an injection at constant flow rate at 1L/min.
 - Check the position of the flow front according the time and compare it with analytical solution
 - Compare the evolution of the pressure in the injection region