

Vaango Installation Guide

Version 20.5.7

May 7, 2020

Biswajit Banerjee

and

The Uintah team



Copyright © 2015-2020 Biswajit Banerjee

The contents of this manual can and will change significantly over time. Please make sure that all the information is up to date.



Contents

1	Overview of Vaango	5
1.1	Obtaining the Code	5
1.2	Developer version	6
1.2.1	Getting the latest version, adding files etc.	6
1.3	Installation Overview	7
1.3.1	Explicit time integration only	7
1.3.2	Implicit and explicit time integration	7
2	Prerequisites and library dependencies	9
2.1	Prerequisites for explicit time integration only	9
2.1.1	Cmake	9
2.1.2	Compilers	9
2.1.3	MPI and XML libraries	9
2.1.4	Eigen3 library	10
2.1.5	HDF5 library	10
2.1.6	Other libraries	10
2.1.7	Optional libraries	10
2.2	Extra prerequisites for implicit time integration	10
2.2.1	PETSc Installation	11
2.2.2	Hypre Installation	11
3	Configuring and Building Vaango	13
3.1	Basic instructions	13
3.1.1	The Vaango repository	13
3.1.2	Out-of-source optimized build	13
3.1.3	Out-of-source debug build	13
3.1.4	Unit tests	13
3.1.5	Clang compiler:	14
3.1.6	Visit build:	14

3.2	Compiling the code:	14
4	Installing VisIt	15
4.1	Directory structure	15
4.2	Using the VisIt install script	15
4.3	Using the VisIt build script	15
4.3.1	Remote visualization	16
4.3.2	Host Profile	16



1 — Overview of Vaango

The VAANGO library is a fork of UINTAH (<http://uintah.utah.edu>) intended for solid mechanics and coupled solid-fluid simulations.

The main components of VAANGO are a material point method (MPM) code, a compressible fluid dynamics code (ICE), a coupled MPM-ICE code, and an experimental Peridynamics code. Efforts are underway to incorporate a Discrete Element Method (DEM) code and a Smoothed Particle Hydrodynamics (SPH) code into the framework.

The library framework in UINTAH was designed for solving PDEs on structured AMR grids on large parallel supercomputers. In VAANGO, we mainly use the grid for searches and scratch-pad calculations.

VisIt (<https://wci.llnl.gov/simulation/computer-codes/visit/>) is used to visualize data generated from VAANGO.

1.1 Obtaining the Code

VAANGO can be obtained either as a zipped archive (approx. 550 Mb) from

```
https://github.com/bbanerjee/ParSim/archive/master.zip
```

or cloned from Github using either HTTP

```
git clone --recursive https://github.com/bbanerjee/ParSim.git
```

or SSH

```
git clone --recursive git@github.com:bbanerjee/ParSim.git
```

The above commands check out the PARSIM source tree and installs it into a directory called **ParSim** in the user's home directory. The PARSIM code contains several research codes. The main VAANGO code can be found in the **ParSim/Vaango/src** directory. Manuals for VAANGO are in **ParSim/Vaango/Manuals**.

You may also need to install **git** on your machine if you want to clone the repository from **Github**:

```
sudo apt-get install git-core
```

The **--recursive** flag is needed in order to make sure that the submodules **googletest** and **json** are also populated during the download process.

1.2 Developer version

If you are a developer, we will suggest that you use the following longer process.

1. Create a **GitHub** account

```
https://github.com/signup/free
```

2. Install **git** on your machine

```
sudo apt-get install git-core
```

3. Setup git configuration

```
git config --global user.email "your_email@your_address.com"
git config --global user.name "your_github_username"
```

4. Email your **github** username to the owner of PARSIM.

5. Create a clone of the ParSim repository

```
git clone https://github.com/bbanerjee/ParSim
```

6. Check branches/master

```
git branch -a
```

7. Configure **ssh** for secure access (if needed)

```
cd ~/.ssh
mkdir backup
cp * backup/
rm id_rsa*
ssh-keygen -t rsa -C "your_email@your_address.com"
sudo apt-get install xclip
cd ~/ParSim/
xclip -sel clip < ~/.ssh/id_rsa.pub
ssh -T git@github.com
ssh-add
ssh -T git@github.com
```

8. For password-less access with **ssh**, use

```
git remote set-url origin git@github.com:bbanerjee/ParSim.git
```

9. To change the default message editor, do

```
git config --global core.editor "vim"
```

If you are new to VAANGO, we strongly suggest that you fork the code from GitHub and submit your changes via pull requests.

1.2.1 Getting the latest version, adding files etc.

1. Check status

```
git status
git diff origin/master
```

2. Update local repository

```
git pull
```

3. Add file to your local repository

```
git add README
git commit -m 'Added README file for git'
```

4. Update main repository with your changes

```
git push origin master
```

If you are new to VAANGO, we strongly suggest that you fork the code from GitHub and submit your changes via pull requests.

1.3 Installation Overview

VAANGO can be installed individually and in conjunction with the visualization tool, VisIt. It is recommended for first time users to install VAANGO first and then install VisIt second after you have a working build of VAANGO.

VisIt recognizes files in the UINTAH format automatically. VAANGO uses an older version of the UINTAH data file format that may not be recognized by the latest version of VisIt. We have found that VisIt 2.10 works with the VAANGO code.

For older versions of VisIt, the `cmake` command used to build VAANGO may have to be modified to include the location of the VisIt installation directory.

The VisIt versions that we have tested require `Python 2.7`. However, most modern Linux installations use `Python 3` by default. For VisIt to work with `Python 3` installed, you will have to create a virtual environment for `Python 2.7` using

```
virtualenv visit-env
```

To activate the environment, run

```
source visit-env/bin/activate
```

After that install packages that may be needed by VisIt. To exit the environment use:

```
deactivate
```

1.3.1 Explicit time integration only

Most of the material models in VAANGO support only explicit time integration. If you want to use only these models, the installation process is relatively straightforward:

1. Install basic dependencies: MPI, etc..
2. Configure and compile VAANGO
3. Install visualization tools

1.3.2 Implicit and explicit time integration

A few VAANGO models need implicit time integration. If you need to use these models, the installation procedure follows the basic outline:

1. Install basic dependencies: MPI, Blas, Lapack, Hypre, PETSc, etc.
2. Configure and compile Vaango
3. Install visualization tools (optional if installing on a supercomputer or first time users) and reconfigure Vaango.



2 — Prerequisites and library dependencies

VAANGO depends on several tools and libraries that are commonly available or easily installed on various Linux or Unix like OS distributions.

The following instructions are tailored for installations on Ubuntu/Debian-like systems.

2.1 Prerequisites for explicit time integration only

2.1.1 Cmake

You will probably need to install **Cmake** to control the software compilation process. To do that:

```
sudo apt-get install cmake
```

2.1.2 Compilers

- You will need a C++14 compliant compiler, either **g++** or **clang**:

```
sudo apt-get install g++
```

```
sudo apt-get install clang-3.8
```

- and a C Compiler such as **gcc**:

```
sudo apt-get install gcc
```

- You also need to have **gfortran**:

```
sudo apt-get install gfortran
```

- You may need **Boost** for parts of the code (and also some of the unit tests) to compile.

```
sudo apt-get install libboost-all-dev
```

We are trying to remove **Boost** dependencies from the code. Try to build the code without **Boost**, and only install the library if the build fails.

2.1.3 MPI and XML libraries

- Install the **OpenMPI** libraries:

```
sudo apt-get install mpi-default-dev
```

- and `libxml2`:

```
sudo apt-get install libxml2
sudo apt-get install libxml2-dev
```

2.1.4 Eigen3 library

The Vaango MPM code has transitioned to `Eigen3` from version 18.01 onwards. The peridynamics code also uses the `Eigen3` library. You will have to install this library if you don't have it in your system.

```
sudo apt-get install libeigen3-dev
```

2.1.5 HDF5 library

From version 19.03, the Vaango MPM code uses the `HDF5` library to read files generated with `Tensorflow`. Install the development library using

```
sudo apt-get install libhdf5-dev
```

2.1.6 Other libraries

- You will also need to install the development version of `zlib`.

```
sudo apt-get install zlib1g zlib1g-dev
```

- The `googletest` libraries are cloned into the repository when you use the `-recursive` flag. You don't need any further installation.
- The `json` header-only library is also cloned into the repository when you use the `-recursive` flag. You don't need any further installation.

2.1.7 Optional libraries

- In order for related code like `MPM3D_xx` etc. to work, you will also need the `VTK` libraries. Use

```
sudo apt-get install libvtk5-dev
sudo apt-get install python-vtk tcl-vtk libvtk-java libvtk5-qt4-dev
```

This dependency is not needed for most installations.

- In some older versions of `VAANGO`, `libpcl` is used read point data file. In order to get PCL(The Point Cloud Library) you need to run these three commands:

```
sudo add-apt-repository ppa:v-launchpad-jochen-sprickerhof-de/pcl
sudo apt-get update
sudo apt-get install libpcl-all
```

This dependency is not needed for most installations.

2.2 Extra prerequisites for implicit time integration

Required libraries:

- blas
- lapack
- Hypre v2.8.ob (<https://computation.llnl.gov/casc/hypre/software.html>)
- PETSc v3.3 (<http://www.mcs.anl.gov/petsc/petsc-as/download/index.html>)

Note, we suggest using Hypre v2.8.ob and PETSc v3.3-p3.

2.2.1 PETSc Installation

The PETSc library is needed to run implicit MPM. Pre-compiled binaries are available for download. However, they typically work only with MPICH and not OpenMPI. You will need to download PETSc 3.12.5 or newer. As of version 20.05, the recommended download page is <https://www.mcs.anl.gov/petsc/download/index.html> and the recommended version is 3.12.5. Alternatively, you can download the latest release from GitLab.

We recommend building the debug version of PETSc from source using

```
./configure
make all test
```

For an optimized build use

```
./configure --with-debugging=0
make all test
```

You will also need to set the `PETSC_ARCH` and `PETSC_DIR` variables in your `.bashrc` file. For example,

```
PETSC_ARCH=arch-linux-c-debug
export PETSC_ARCH
PETSC_DIR=/home/banerjee/Packages/petsc-3.12.5
export PETSC_DIR
```

2.2.2 Hydre Installation

Hydre can be installed by executing the instructions in the HYPRE site. The ICE-Hydre implementation has not been tested extensively and should be used with care.

Download hydre-2.8.ob from <https://computation.llnl.gov/casc/hypre/software.html>



3 — Configuring and Building Vaango

3.1 Basic instructions

3.1.1 The Vaango repository

After you get the code from GitHub, follow these steps:

- Go to the **Vaango** directory:

```
cd ParSim/Vaango
```

- The source code is in the directory **src**.

3.1.2 Out-of-source optimized build

- For the optimized build, create a new directory called **opt** under **Vaango** :

```
mkdir opt
```

- followed by:

```
cd opt
```

- To create the make files do:

```
cmake ../src
```

3.1.3 Out-of-source debug build

- For the debug build, create a directory **dbg** under **Vaango** :

```
mkdir dbg  
cd dbg
```

- To create the makefiles for the debug build, use

```
cmake -DCMAKE_BUILD_TYPE=Debug ../src
```

3.1.4 Unit tests

If you want the units tests to be compiled, use the alternative command

```
cmake ../src -DBUILD_UNITS_TESTS=1
```

3.1.5 Clang compiler:

If you want to use the **clang** compiler instead of **gcc** :

```
cmake ../src -DUSE_CLANG=1
```

3.1.6 Visit build:

Older versions of **Visit** required the following extra step if you want to make sure that Visit is able to read Uintah output format files (also called UDA files) you will need to use

```
cmake ../src -DVISIT_DIR=/path/to/Visit
```

3.2 Compiling the code:

Next you need to compile the files from **src** . So go to the **opt** directory and then type :

```
make -j4
```

After this compilation you have all the executable files in your **opt** directory. The same process can be used for the debug build.



4 — Installing VisIt

Visualization of VAANGO data is currently possible using VisIt. The VisIt package from LLNL is general purpose visualization software that offers all of the usual capabilities for rendering scientific data. It is still developed and maintained by LLNL staff, and its interface to VAANGO data is supported by the Uintah team.

4.1 Directory structure

The following directory structure is suggested for building VisIt:

```
/myPath/VisIt  
/myPath/VisIt/thirdparty  
/myPath/VisIt/2.10.2
```

4.2 Using the VisIt install script

Download the build script from http://portal.nersc.gov/project/visit/releases/2.10.2/visit-install2_10_2. Follow the instructions at http://portal.nersc.gov/project/visit/releases/2.10.2/INSTALL_NOTES. VAANGO writes its output in the Uintah file format. If VisIt does not recognize the Uintah file format, you may have to follow the detailed instructions below.

4.3 Using the VisIt build script

1. Get the `build_visit` script from http://portal.nersc.gov/project/visit/releases/2.10.2/build_visit2_10_2.
2. Set the MPI compiler to be used (must be the same version used to build Uintah):

```
export PAR_COMPILER='which mpic++'  
export PAR_COMPILER_CXX='which mpicxx'  
export PAR_INCLUDE="-I/usr/lib/openmpi/include/"
```

3. Use the `build_visit` script to download, build, and install the thirdparty libraries required by VisIt.
4. Go to the VisIt thirdparty directory:

```
cd /myPath/VisIt/thirdparty
```

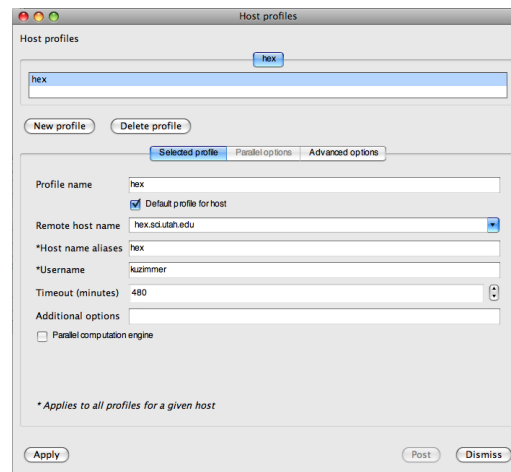


Figure 4.1: Setting up Host Profile

5. Run `build_visit` :

```
/myPath/VisIt/2.10.2/build_visit --console --thirdparty-path /myPath/VisIt/
thirdparty --no-visit --parallel --fortran --uintah
```

To add additional libraries to enable other database readers using the `build_visit` script you will have to add the appropriate flags when you run `build_visit`.

6. The `build_visit` script will take a while to finish. After the script finishes it will produce a machine specific `cmake` file called `<your_machine_name>.cmake`. This file will be used when compiling visit.
7. Move the `<your_machine_name>.cmake` file to the VisIt source `config-site` directory:

```
mv <your_machine_name>.cmake /myPath/VisIt/2.10.2/src/config-site
```

8. Go to the VisIt source directory:

```
cd /myPath/VisIt/2.10.2/src
```

9. Run `cmake` using the `cmake` that was built by the `build_visit` script

```
/myPath/VisIt/thirdparty/cmake/<version>/<OS-gcc_version>bin/cmake .
```

10. Finally make VisIt :

```
make -j 8
```

One can install VisIt or run it directly from the bin directory.

4.3.1 Remote visualization

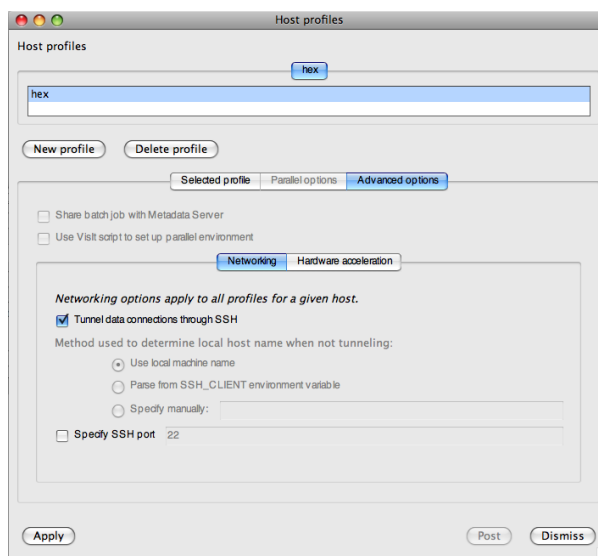
In order to read data on a remote machine, you will need to have a version of VisIt and the UINTAH data format reader plugin installed on that machine. You will also need to make sure that the VisIt executable is in your path. You may need to edit your `.<shell>rc` file so that the path to VisIt is included in your shell's `PATH` variable.

The version of VisIt installed on your local desktop and the remote machine should be the same.

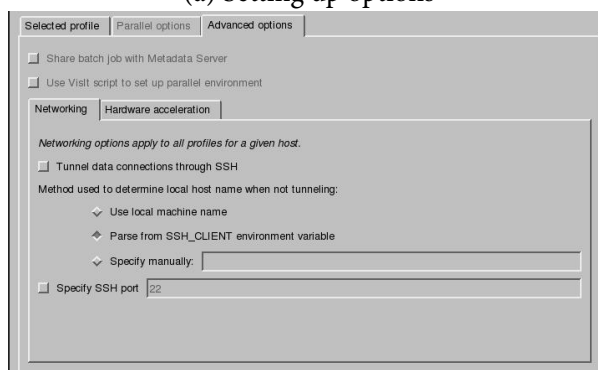
4.3.2 Host Profile

Next you will want to set up a Host Profile for your remote machine. Select `Host Profiles` from the `Options` menu and set up a new profile as shown in figure 4.1.

After filling in the remote machine information, select the **Advanced options** tab, then **Networking** and check the **Tunnel data connections through SSH** option. This is illustrated in figure 4.2(a). Click on **Apply** and then do a **Save Settings** in the **Options** menu.



(a) Setting up options



(b) Setting up advanced options

Figure 4.2: VisIt remote visualization options

For the remote visualization option to work, you must have ports 5600 - 5609 open. You can try this by running the following on your local desktop (on Linux distributions),

```
traceroute -p 560[0-9] <remote machine>
```

If the tunneling option doesn't work, try the option as shown in figure 4.2(b).

Now you should be able to select **Open** from the VisIt **File** menu. After selecting your host from the host entry list, you will be prompted for a password on the remote machine (unless you have set up password-less ssh access).

Once the ssh login has completed, you should see the directory listing. You can then change directories to your UDA and load the data.