Applications and libraries

Table of Contents

Applications on the clusters

An important number of applications and libraries are available on Baobab and Yggdrasil, and we often offer multiple versions.

The magic which allows that is `lmod` with the `module` command. This is the recommended way to load any application or libraries on the clusters.

On this page, we will give the most common `module` usage and give some example for a selection of applications.

Module - lmod

The recommended way to load an application on the clusters is to use the `module` command. By using `module`, you don't need to know where the software and libraries are physically located (the full path), but instead you can just type the application name as if its path was in your PATH (this is indeed what `module` does).

The `module` command can also set other important environment variables for an application, so it is always recommended to use it.

How to use 'module'

These are the most common options you will use with `module`.

To get a **complete list of applications available** with `module`

```
module spider
```

To find the available versions of a certain application :

```
module spider <app_name>
```

To load one (or more) application :

```
module load <app_name_1> <app_name_2> ...
```

To load a specific version of an application :

```
module load <app_name/version>
```

Hint : Module version. By choosing a module without specifying a version, you will always get the latest version available. However, we always recommend to specify the version, as your code might produce different results if you are using another version. If **reproducibility of results** is important for you, you should definitely used a fixed version.

You can see the help for a particular module (it must be loaded first):

```
module help R
```

See the list of currently loaded modules :

```
module list
```

To unload all currently loaded modules :

```
module purge
```

Hint : If you are in a hurry you can also use `ml` instead of `module` for any of the above mentioned commands :

```
ml spider
```

For more information, use the manpage `man module`.

What do I do when an application is not available via 'module' ?

If the application you need or the exact version is not available via `module` :

- First drop us an email at hpc@unige.ch and explain with as much detail as possible what you need (provide scripts, links, etc.). If we can install what you need, we will do it, most of the time via `EasyBuild/module` (you can check the list of available software).
- If we cannot, you can compile binaries in your `$HOME` directory and use them on any node of the cluster (since your `$HOME` is accessible from any node). Make sure you load the compiler with module first!
  - Read more in the section Compile and install a software in your /home
- Another interesting option is to use Singularity which allows you to run containers on the clusters.

Detailed example of using 'module'

Loading 'R'

Let's go through an example of loading R.

First, let's find all available version of R :

```
[brero@login2 ~]$ module spider R

----------------------------------------------------------------------------
----------------------------------------------
  R:
----------------------------------------------------------------------------
----------------------------------------------
    Description:
      R is a free software environment for statistical computing and
graphics.

     Versions:
        R/3.2.3
        R/3.3.1
        R/3.3.2
        R/3.5.1
        R/3.6.0
        R/3.6.2
        R/4.0.0
     Other possible modules matches:
        APR  APR-util  BioPerl  Bismark  Blender  CellProfiler  CellRanger
CoordgenLibs  DISCOVARdenovo  DendroPy  ...
[...]
```

Now some people just need the latest version available and can simply load it with `module load R` ; whitout specifying a version, you will always get the latest version.
But sometimes, you need to use the same version everytime and we recommend it. This is very important as your code might produce different results if you are using another version. If reproducibility of results is important for you, you should definitely used a fixed version.

To load a specific version, in this case R version 3.6.2 :

```
[brero@login2 ~]$ module load R/3.6.2
Lmod has detected the following error:  These module(s) or extension(s)
exist but cannot be loaded as requested:
"R/3.6.2"
   Try: "module spider R/3.6.2" to see how to load the module(s).
```

This fails as the module “cannot be loaded as requested”. This is usually because you are missing dependencies.
The message also suggest to try the following command :

```
[brero@login2 ~]$ module spider R/3.6.2

----------------------------------------------------------------------------
----------------------------------------------------------------------------
----------------------------------------------------
  R: R/3.6.2
----------------------------------------------------------------------------
```

```
------------------------------------------------------------------------------
-------------------------------------------------------
    Description:
      R is a free software environment for statistical computing and
graphics.


    You will need to load all module(s) on any one of the lines below before
the "R/3.6.2" module is available to load.

      GCC/8.3.0  OpenMPI/3.1.4

    Help:

      Description
      ===========
      R is a free software environment for statistical computing
       and graphics.


      More information
      ================
       - Homepage: https://www.r-project.org/


      Included extensions
      ===================
      abc-2.1, abc.data-1.0, abe-3.0.1, abind-1.4-5, acepack-1.4.1,
adabag-4.2,
      ade4-1.7-13, ADGofTest-0.3, aggregation-1.0.1, akima-0.6-2,
AlgDesign-1.2.0,
      animation-2.6, aod-1.3.1, ape-5.3, arm-1.10-1, askpass-1.1,
asnipe-1.1.12,
      assertthat-0.2.1, AUC-0.3.0, audio-0.1-6, b-a, backports-1.1.5,
bacr-1.0.1,
      bartMachine-1.2.4.2, bartMachineJARs-1.1, base64-2.0, base64enc-0.1-3,
      BatchJobs-1.8, BayesianTools-0.1.7, bayesm-3.1-4, BayesPen-1.0,
BB-2019.10-1,
      BBmisc-1.11, bbmle-1.0.20, BCEE-1.2, BDgraph-2.62, bdsmatrix-1.3-3,
      beanplot-1.2, beeswarm-0.2.3, BH-1.69.0-1, BiasedUrn-1.07,
bibtex-0.4.2,
      bigmemory-4.5.33, bigmemory.sri-0.1.3, bindr-0.1.1, bindrcpp-0.2.2,
      bio3d-2.4-0, biom-0.3.12, bit-1.1-14, bit64-0.9-7, bitops-1.0-6,
blob-1.2.0
[...]
```

As the message explains, you **need** to load 2 dependencies `GCC/8.3.0` and `OpenMPI/3.1.4` before you can load R.

You can then simply execute the following command :

```
[brero@login2 ~]$ module load GCC/8.3.0 OpenMPI/3.1.4 R/3.6.2
```

Remember that GCC and OpenMPI are packaged together in the foss module. You can then load the corresponding foss module instead which is shorter:

```
[brero@login2 ~]$ module load foss/2019b R/3.6.2
```

Then you can just invoke R by typing R in the terminal (instead of using the full path). Of course you are still required to use Slurm and an sbatch script to launch your software.

Hint : To automatically load some modules at login, you can add something like this in your `$HOME/.bashrc`:

```
if [ -z "$BASHRC_READ" ]; then
  export BASHRC_READ=1
  # Place any module commands here
  module load GCC/8.3.0 OpenMPI/3.1.4 R/3.6.2
fi
```

Choosing the compiler toolchain

You have the choice between FOSS toolchain or Intel toolchain (license required).

If you want to compile your software against MPI, it is very important not to compile using directly `gcc`, `icc` or similar commands, but rather rely on the wrappers `mpicc`, `mpic++`, `mpicxx` or similar ones provided by module.

All the newer versions of `MPI` will be available through the use of module.

FOSS toolchain

module	compiler	mpi
foss/2016a	gcc 4.9.3	openmpi 1.10.2
foss/2016b	gcc 5.4.0	openmpi 1.10.3
foss/2017a	gcc 6.3.0	openmpi 2.0.2
foss/2017b	gcc 6.4.0	openmpi 2.1.1
foss/2018a	gcc 6.4.0	openmpi 2.1.2
foss/2018b	gcc 7.3.0	openmpi 3.1.1
foss/2019a	gcc 8.2.0	openmpi 3.1.3
foss/2019b	gcc 8.3.0	openmpi 3.1.4
foss/2020a	gcc 9.3.0	openmpi 4.0.3
foss/2020b	gcc 10.2.0	openmpi 4.0.5
foss/2021a	gcc 10.3.0	openmpi 4.1.1
foss/2021b	gcc 11.2.0	openmpi 4.1.1
foss/2022a	gcc 11.3.0	openmpi 4.1.4
foss/2022b	gcc 12.2.0	openmpi 4.1.4
foss/2023a	gcc 12.3.0	openmpi 4.1.5

module	compiler	mpi
foss/2023b	gcc 13.2.0	openmpi 4.1.6
foss/2024a	gcc 13.3.0	openmpi 5.0.3

Example for the latest version of `gcc`:

```
module load foss
```

If needed, you can stick to a particular (or legacy) version:

```
module load foss/2021b
```

You can see the details of what is loaded in foss with:

```
module list
```

Intel toolchain

module	compiler	mpi
intel/2016a	icc 16.0.1	impi 5.1.2
intel/2016b	icc 16.0.3	impi 5.1.3
intel/2017a	icc 17.0.1	impi 2017 Update 1
intel/2018a	icc 18.0.1	impi 2018.1.163
intel/2019a	icc 19.0.1	impi 2018.4.274
intel/2020a	icc 19.1.1.217	impi 2019.7.217
intel/2021a	icc 2021.2.0	impi/2021.2.0
intel/2021b	icc 2021.4.0	impi/2021.4.0
intel/2022a	icc 2022.1.0	impi/2021.6.0

Example for the latest version of icc from Intel:

```
module load intel
```

If needed, you can stick to a particular (or legacy) version:

```
module load intel/2021a
```

You can see the details of what is loaded in intel with:

```
module list
```

If you want to use the intel compiler for mpi job, you need to export the variable:

```
export I_MPI_PMI_LIBRARY=/usr/lib64/libpmi.so
```

intel mpi with slurm

Intel compiler licenses

If you want to use an **old** Intel compiler (before 2021a), you need to have your own Intel license compiler.

Once you get the license you should copy it to your home directory in a directory named `Licenses` ($HOME/Licenses).

It isn't anymore possible to get this Intel license. Instead you are required to use Intel oneAPI which doesn't requires a license. Use intel/2021a or newer.

fosscuda toolchain

Cuda is provided through fosscuda or directly through CUDA module. The difference is that fosscuda is a bundle of software and CUDA only provide the minimum.

module	CUDA	GCC	OpenMPI
fosscuda/2018b	9.2.88	7.3.0	3.1.1
fosscuda/2019a	10.1.105	8.2.0	3.1.3
fosscuda/2019b	10.1.243	8.3.0	3.1.4
fosscuda/2020a	11.0.2	9.3.0	4.0.3
fosscuda/2020b	11.1.1	10.2.0	4.0.5

Examples for selected applications

We are providing a bunch of sbatch examples on our GitLab repository here. Feel free to clone the repository and provide other examples or fixes through pull request.

OpenMPI

Specify MCA parameters through ''srun''

Normally you would pass those parameters to `mpirun` but as you are using `srun` with Slurm, this is not possible directly.

You should have a dedicated file with parameters located here: `$HOME/.openmpi/mca-params.conf`.

Or you may use environment variable with `OMPI_MCA_` prefix. here for more details.

Conda

How to Create a Conda Environment in a Container

Using **Conda** directly on HPC systems or shared servers can cause performance issues and storage overload because Conda environments create thousands of small files. This often results in:

- Slow job startup times
- Filesystem limitations being hit
- High I/O load on the cluster
- Complex environment management

A better solution is to **encapsulate Conda environments inside a container**. This way, the entire environment is packaged into a single file (such as a `.sif` image used by Apptainer/Singularity), which can be easily deployed, shared, and reused without polluting clusters.

Benefits

Using this method offers multiple advantages:

1. **Fewer files**: Your environment is stored in a single `.sif` file
2. **Portability**: Share the container easily with collaborators
3. **Reproducibility**: Your environment stays consistent across systems
4. **Isolation**: No risk of interfering with other environment
5. **Stability**: Cluster updates will not break your environment

Limitations

1. ⚠ The container is static; to update packages, you need to rebuild the image

This guide explains how to build such a container using cotainr, a tool that simplifies container creation.

Step 1 – Define the Conda Environment

Create a file `env.yml` that contains the definition of your environment: (As exemple we will use `bioenv.yml`)

```
name: bioenv
channels:
  - bioconda
  - conda-forge
  - defaults
dependencies:
  - blast=2.16.0
  - diamond=2.1.11
```

```
  - exonerate=2.4.0
  - spades=4.1.0
  - mafft=7.525
  - trimal=1.5.0
  - numpy
  - joblib
  - scipy
[...]

prefix:/home/users/a/alberta/env      # <==== To delete/edit
```

You can generate this file using the following commands:

```
# 1. (optional) create your environment (or not if you already have one)
$ conda create -n bioenv -c bioconda -c conda-forge spades exonerate diamond
blast mafft trimal numpy joblib scipy -y

# 2. Activate your environment
$ conda activate bioenv

# 3. Export the settings of your environment
# It's recommended to manually remove the `prefix:` line at the bottom of
the file before using it with cotainr.
$ conda env export > bioenv.yml
```

Step 2 – Build the Container

Now use `cotainr` to create the image:

```
$ module load GCCcore/13.3.0 cotainr
# Ex: cotainr build <env.sif> --base-image=docker://<WhatEverYouWant>:latest
--accept-licenses --conda-env=<env.yml>
$ cotainr build bioenv.sif --base-image=docker://ubuntu:latest --accept-
licenses --conda-env=bioenv.yml
```

You can replace `ubuntu:latest` with any other base image, such as `rockylinux:latest`.

Step 3 – Use the Container

You can now run commands inside the container as follows:

```
$ apptainer exec bioenv.sif python3 -c "import numpy;
print(numpy.__version__)"
```

Or launch any program inside the container just like you would in a normal environment.

Conda environment management

Use it

```
module load Anaconda3
```

Create

```
conda create --name environment_name
```

As the conda env itself contains a lot of files, it may be a good idea to store it on the scratch folder for example.

```
conda create --prefix $HOME/scratch/test-env --name environment_name
```

List

```
conda env list
```

Activate

```
conda activate <env name>
```

Deactivate

```
conda deactivate
```

Package management

List packages in a given environment:

```
conda list
```

ADF

You can use SCM ADF on one ore more nodes of the cluster. Please note that you must use `module` (see Module - lmod) in you sbatch script to set the variables correctly.

Please see ADF example on GitLab for some example and scripts related to ADF.

Do not launch ADF using `srun`. ADF is a wrapper which uses `srun` internally.

ADF needs a fast local scratch space. On Baobab, the local scratch of each node is only about 180GB. If you need more space, you need to find another solution (do the

calculation on more nodes, do not use local scratch, buy us new hard disks)

By default, the local scratch space is defined in the SCM_TMPDIR environment variable, /scratch as set by module load ADF . This default value will gives error when calling ADFview on the login nodes, given that /scratch does not exist there. You can overcome it by using the Linux default /tmp :

```
capello@login2:~$ module load ADF/2019.104
capello@login2:~$ SCM_TMPDIR=/tmp adfview
```

Gaussian

You can use Gaussian g09 on one node of the cluster. Please note that you must use module (see Module - lmod) in you sbatch script to set the variables correctly. When using the module, it will set the variable GAUSS_SCRDIR to /scratch of the local hard disk of the allocated node. This should lower the calculation time and as well lower the usage of the shared filesystem. See below for other optimizations.

There are two versions of g09 on the cluster. Revision c01 and d01.

Please see Gaussian example on GitLab for some examples and scripts related to Gaussian.

To optimize the run, you can add some lines in your job file. If you need more than 190 GB of scratch space, you should add the line (adapt /home/yourusername to your own path):

```
%RWF=/scratch/,170GB,/home/yourusername/scratch/,-1
```

You may as well specify how much memory you want to use. By default, Gaussian will use 250MB of ram. You can try with 50GB for example:

```
%Mem=50GB
```

You need to specify as well how many CPU cores you want to use:

```
%NProcShared=16
```

Git

To use git on the cluster you need to do the following:

Add that to your ${HOME}/.gitconfig:

```
[core]
createObject = rename
```

To invoke git:

```
git clone --no-hardlinks
```

NVIDIA HPC SDK Installation in Home Directory on a Heterogeneous Cluster

https://developer.nvidia.com/hpc-sdk

Prerequisites

- Download and extract ONLY the tarball by following the procedure: https://developer.nvidia.com/hpc-sdk-downloads
- Ensure you have the latest version of GCC loaded. You can check and load the module using the following commands:

```
(base) (yggdrasil)-[alberta@login1 ~]$ ml GCC
(base) (yggdrasil)-[alberta@login1 ~]$ ml

Currently Loaded Modules:

GCCcore/13.2.0 2) zlib/1.2.13 3) binutils/2.40 4) GCC/13.2.0
```

Installation Steps

1. **Start the NVIDIA HPC SDK Installer:**

As it'writte press **enter** to continue

```
(base) (yggdrasil)-[alberta@login1 ~]$
nvhpc_2024_245_Linux_x86_64_cuda_12.4/install

Welcome to the NVIDIA HPC SDK Linux installer!

You are installing NVIDIA HPC SDK 2024 version 24.5 for Linux_x86_64.
Please note that all Trademarks and Marks are the properties
of their respective owners.

Press enter to continue...
```

2. **Select the Installation config:** Since we are setting up for a **heterogeneous** cluster, select the **network installation** option:

```
A single system installation is appropriate for a single system or a
homogeneous cluster.  A network installation should be selected for a
heterogeneous cluster.  For either a single system or network installation,
the HPC SDK configuration (localrc) is created at install time and saved
in the installation directory.
```

```
An auto installation is appropriate for any scenario.  The HPC SDK
configuration (localrc) is created at first use and stored in each user's
home directory.

1  Single system install
2  Network install
3  Auto install

Please choose install option:
2
```

3. **Specify the full path you want to install the software:**

```


Please specify the directory path under which the software will be
installed.
The default directory is /opt/nvidia/hpc_sdk, but you may install anywhere
you wish, assuming you have permission to do so.

Installation directory? [/opt/nvidia/hpc_sdk]
/home/users/a/alberta/HPC_sdk

Installing NVIDIA HPC SDK version 24.5 into /home/users/a/alberta/HPC_sdk
Making symbolic link in /home/users/a/alberta/HPC_sdk/Linux_x86_64

generating environment modules for NV HPC SDK 24.5 ... done.
Installation complete.
Please run add_network_host to create host specific localrc files:

/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/compilers/bin/localrc.$host

on all other hosts you wish to run NVIDIA HPC SDK compilers.

For 64-bit NVIDIA HPC SDK compilers on 64-bit Linux systems, do the
following:
/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/compilers/bin/add_network_ho
st

HPC SDK successfully installed into /home/users/a/alberta/HPC_sdk

If you use the Environment Modules package, that is, the module load
command, the NVIDIA HPC SDK includes a script to set up the
appropriate module files.

% module load /home/users/a/alberta/HPC_sdk/modulefiles/nvhpc/24.5
% module load nvhpc/24.5

Alternatively, the shell environment may be initialized to use the HPC SDK.

In csh, use these commands:
```

```
% set path = (/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/compilers/bin
$path)
% setenv MANPATH
/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/compilers/man:"$MANPATH"

To use MPI, also set:

% set path =
(/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/comm_libs/mpi/bin $path)

In bash, sh, or ksh, use these commands:

$ export
PATH=/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/compilers/bin:$PATH
$ export
MANPATH=/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/compilers/man:$MANPA
TH

To use MPI, also set:

$ export
PATH=/home/users/a/alberta/HPC_sdk/Linux_x86_64/24.5/comm_libs/mpi/bin:$PATH

Please check https://developer.nvidia.com for documentation,
use of NVIDIA HPC SDK software, and other questions.
```

Now the installation is complete **BUT** the module method does not work and you **MUST** use the export command.

If you haven't read the installation instructions carefully, do so now. Missing steps could lead to unforeseen adventures in troubleshooting! (An angry admin may RTFM you)

Verify Installation

Create and compile a Test program

```
(yggdrasil)-[alberta@login1 ~]$ cat test.cpp
#include <ranges>

int main(){
  return 0;
}
```

```
(yggdrasil)-[alberta@login1 ~]$ cat sbatch_nv
#!/bin/bash
```

```
#SBATCH --job-name=compile_test
#SBATCH --output=compile_test.out
#SBATCH --error=compile_test.err
#SBATCH --time=00:01:00
#SBATCH --partition=debug-cpu

# Load the necessary modules
ml GCC

# Set up the environment for NVIDIA HPC SDK
export HPC_SDK_DIR=/home/users/a/alberta/HPC_sdk
export PATH=$HPC_SDK_DIR/Linux_x86_64/24.5/compilers/bin:$PATH
export MANPATH=$HPC_SDK_DIR/Linux_x86_64/24.5/compilers/man:$MANPATH
export PATH=$HPC_SDK_DIR/Linux_x86_64/24.5/comm_libs/mpi/bin:$PATH

# Run add_network_host to create host-specific localrc files
$HPC_SDK_DIR/Linux_x86_64/24.5/compilers/bin/add_network_host


# Compile the test.cpp file
nvc++ test.cpp -o test
```

```
(yggdrasil)-[alberta@login1 ~]$ sbatch sbatch_nv
Submitted batch job 33866816
```

- Ensure to run the `add_network_host` command on all nodes in your heterogeneous cluster to create the necessary `localrc` files.
- Make sure the environment variables are set in your shell configuration files (e.g., `.bashrc`, `.cshrc`) for persistent settings across sessions.
- For more details and further documentation, visit the [NVIDIA HPC SDK website](https://developer.nvidia.com).

Gurobi

NB , the following instructions come from <https://hpc-community.unige.ch/t/gurobi-solver-license-issue/459>.

We provide an internal Gurobi token server, here how to use it:

```
capello@login2:~$ module load Gurobi
capello@login2:~$ gurobi_cl --tokens

Checking status of Gurobi token server 'master.cluster'...
```

```
Token server functioning normally.
Maximum allowed uses: 4096, current: 0

capello@login2:~$
```

Jupyter notebook and Jupyter lab

Jupyter notebook can run on our clusters, however we do not have a dedicated server for Jupyter. This means you need to submit a job and request some resources before you can connect to your instance of Jupyter notebook.

The easiest way is to launch a session (on Baobab only for now) using OpenOnDemand

If it isn't working for you, check our git repo which as some example scripts to launch Jupyter as Slurm job.

You can also read : [tutorial] Jupyter notebook

or

updated Tutorial done for HPC-Lunch

For interactive programs such as Jupyter notebook, you might want to use the `public-interactive-cpu` partition (Which partition for my job)

Mathematica

If you would like to use a different license server (by default mathlm.unige.ch), you can specify its URL in the `${HOME}/.Mathematica/Licensing/mathpass` file, prependend by an exclamation mark:

```
capello@login2:~$ MATHEMATICA_LICENSE_SERVER_URL=mathlm.unige.ch
capello@login2:~$ mkdir -p ~/.Mathematica/Licensing/
capello@login2:~$ cat <<EOF >~/.Mathematica/Licensing/mathpass
!${MATHEMATICA_LICENSE_SERVER_URL}
EOF
capello@login2:~$ cat ${HOME}/.Mathematica/Licensing/mathpass
!mathlm.unige.ch
capello@login2:~$
```

See http://support.wolfram.com/kb/25655 and http://support.wolfram.com/kb/112 for more information.

Matlab

Matlab is available in Baobab in different versions:

```
$ module spider matlab

----------------------------------------------------------------------------
-------
  matlab:
----------------------------------------------------------------------------
-------
     Versions:
        MATLAB/2021a
        MATLAB/2021b
        MATLAB/2022a
```

Keep in mind that it's a licensed program, and that the licenses are shared with the whole university. To be fair with the other users, we have set up a limitation on the number of licenses you can use. We kindly ask you to specify in your `sbatch` file that you are using Matlab in order to keep the limitation effective. If you are using some licensed toolbox, like Wavelet_Toolbox, you need to specify it as well. If you don't, your job may be killed without further notice in case we are out of licenses.

Example to specify that you need `Matlab`:

```
#SBATCH --licenses=matlab@matlablm.unige.ch
```

Example to specify that you need the `Wavelet_Toolbox`:

```
#SBATCH --licenses=wavelet-toolbox@matlablm.unige.ch
```

See the licenses available on Baobab:

```
scontrol show lic
```

If you need a license not listed here, please ask us at **hpc [at] unige [dot] ch**.

You need to specify at LEAST the Matlab license, and zero or more toolbox.

To run Matlab in batch mode, you can create a batch file like this one:

```
#!/bin/bash

#SBATCH --cpus-per-task=1
#SBATCH --ntasks=1
#SBATCH --licenses=matlab@matlablm.unige.ch

module load MATLAB/2021b
```

```
BASE_MFILE_NAME=hello

unset DISPLAY

echo "Running ${BASE_MFILE_NAME}.m on $(hostname)"

srun matlab -nodesktop -nosplash -nodisplay -r ${BASE_MFILE_NAME}
```

In this example, you need to have your code in the file `hello.m`.

You submit the Matlab job like a normal sbatch SLURM job:

```
sbatch ./yourBatch
```

Parallel with Matlab

Since version 2014, you are not limited to 12 CPU cores anymore.

Please see Matlab on gitlab for some examples and scripts related to Matlab parallel

As we are talking about parallel and not distributed Matlab, you can consided Matlab as a multithread application.

See how to submit Multithreaded jobs.

If you are a parallel or distributed Matlab specialist and you have some hints, you are very welcome to contact us!

Pass sbatch arguments to Matlab

You can pass arguments from sbatch to Matlab as described below.

Example of sbatch file (sbatch.sh)

```
[sbatch part as usual]

BASE_MFILE_NAME=test
MATLAB_MFILE=${BASE_MFILE_NAME}.m

unset DISPLAY

module load MATLAB/2021b

#the variable you want to pass to matlab
job_array_index=${SLURM_ARRAY_TASK_ID}

echo "Starting at $(date)"
```

```
echo "Running ${MATLAB_MFILE} on $(hostname)"


# we call the matlab function (see the parenthesis around the argument) and
the argument type will be integer.
srun matlab -nodesktop -nosplash -nodisplay -r
"${BASE_MFILE_NAME}($job_array_index)"
echo "Finished at $(date)"
```

Example of Matlab file (test.m)

```
function test(job_array_index)

fprintf('array index: %d\n', job_array_index)
```

See arguments with Matlab on gitlab for more examples.

Compile your Matlab code

Thanks to Philippe Esling for his contribution to this procedure.

The idea of compiling Matlab code is to save on licenses usage. Indeed, once compiled, a Matlab code can be run without using any license.

The Matlab compiler is named MCC

First load the needed modules:

```
module load foss/2016a matlab/2016b
```

Let's say you want to compile this `.m` file:

```
function hello(name)
   strcat({'Hello '}, name)
end
```

This operation compiles it (this takes some time) :

```
DISPLAY="" mcc -m -v -R '-nojvm, -nodisplay' -o hello hello.m
```

If you have some other `.m` files that you need to include, you need to explicitly specify their location as follows:

```
mcc -m -v -R '-nojvm, -nodisplay' -I /path/to/functions/ -I
/path/to/other/functions/ [...]
```

The resulting files are a text file named `readme.txt`, a script named `run_hello.sh` and an executable named `hello`.

You can then launch the executable hello like any other executable using a sbatch script:

```
#!/bin/bash

#SBATCH --partition=debug-cpu
#SBATCH --ntasks=1

module load foss/2016a matlab/2016b

srun ./hello Philippe
```

In this case, Philippe is an argument for the function *hello*. Be careful, arguments are always passed to Matlab as strings.

You do NOT need to specify the Matlab license once compiled, and you are not restricted by the number of available licenses.

Please see compile Matlab on gitlab for some examples and scripts related to Matlab compilation.

Matlab PATH

If you need to add a directory to the Matlab path, for example to use a toolbox installed by you, please proceed as follow.

Add the needed path recursively to matlab path:

```
addpath(genpath('/home/sagon/tests/matlab/dtb/decoding_toolbox_v3.991'))
```

Save the current matlab path to the default matlab definition path:

```
savepath('/home/sagon/pathdef.m')
```

You can now access any file from the toolbox directly:

```
demo2_simpletoydata
```

Matlab java.opts

If you are getting erros such as the one listed here when using Matlab through X2go, you can try the following changes to mitigate the issue.

Create a file named java.opts in the default startup folder which is the folder from which you started MATLAB, *i.e.* the folder where you type the `matlab` command (usually your `${HOME}` base folder).

Put the following content in it:

```
-Dsun.java2d.xrender=false
```

And restart Matlab. This will probably reduce the graphical performance but should not impact the computation time.

CHROMIUM mailbox/texture errors

If you are getting erros such as the one listed here and here when using Matlab through X2go, unfortunately we have not found a *once-and-for-all* solution yet.

Please contact us providing the MATLAB version as well as the `salloc` full command.

Wavelab

To use the Wavelab library with Matlab, load Matlab 2014 :

```
module load matlab/2014b
```

Launch Matlab as usual and type this command to go to the Wavelab library:

```
cd /opt/wavelab/Wavelab850/
Wavepath (answer /opt/wavelab/Wavelab850)
```

OpenCL

You can use OpenCL on CPU. To compile your software, please proceed as following:

```
gcc  -I/opt/intel/opencl-1.2-sdk-6.3.0.1904/include/ -
L/opt/intel/opencl-1.2-sdk-6.3.0.1904/lib64/ -Wl,-
rpath,/opt/intel/opencl/lib64/ -lOpenCL -o hello hello.c
```

Distant Paraview

Thanks to Orestis for this tutorial.

Warning :

- Do not use at any point X11 forwarding it will be done by paraview itself.
- You must have the SAME version of paraview on your local machine and on baobab (5.3.0).

Baobab connection:

```
ssh login2.baobab.hpc.unige.ch
```

Get some resources in interactive mode (4 cores here, you can add a lot more options here if you want). You can even ask for GPU nodes (more on this afterwards):

```
salloc -n 4
```

Determine on which node your resources are affected (here node001):

```
echo $SLURM_JOB_NODELIST

SLURM_JOB_NODELIST=node001
```

On another terminal (in your pc) open an ssh tunnel to your NODE (in this case node001):

```
ssh -L 11150:node001:11111 login2.baobab.hpc.unige.ch
```

On the first terminal load paraview (if not loaded by default) and launch pserver:

```
module load foss/2016b ParaView/5.3.0-mpi
srun pvserver --server-port=11111
```

If you asked for GPU nodes the command is slightly different:

```
srun pvserver --server-port=11111 -display :0.0 --use-offscreen-rendering
```

On your local machine launch paraview and click on Connect. There you should find the menu to add a server. Put the name you want in Name, leave Client/Server as Server Type, and `localhost` in Host. The only very important configuration is the port which should be 11150 (the same number as in `11150:node001` from before). Save the configuration (click on Configure and Save, leave startup as Manual) and the click on Connect. The remote paraview session should start immediately. There should be an error message: *Display is not accessible on the server side. Remote rendering will be disabled*. This message is normal.

Python

Default Python version on the cluster is Python 2.7.5.

You can have access to modern Python through Module - lmod

The Python version provided by module come with a lot of packages already installed.

You can check with module spider what are the packages provided. Example:

```
ml spider Python/3.7.4
[...]
Included extensions
===================
alabaster-0.7.12, asn1crypto-0.24.0, atomicwrites-1.3.0, attrs-19.1.0,
Babel-2.7.0, bcrypt-3.1.7, bitstring-3.1.6, blist-1.3.6, certifi-2019.9.11,
cffi-1.12.3, chardet-3.0.4, Click-7.0, cryptography-2.7, Cython-0.29.13,
```

```
deap-1.3.0, decorator-4.4.0, docopt-0.6.2, docutils-0.15.2, ecdsa-0.13.2,
future-0.17.1, idna-2.8, imagesize-1.1.0, importlib_metadata-0.22,
ipaddress-1.0.22, Jinja2-2.10.1, joblib-0.13.2, liac-arff-2.4.0,
MarkupSafe-1.1.1, mock-3.0.5, more-itertools-7.2.0, netaddr-0.7.19,
netifaces-0.10.9, nose-1.3.7, packaging-19.1, paramiko-2.6.0,
pathlib2-2.3.4,
paycheck-1.0.2, pbr-5.4.3, pip-19.2.3, pluggy-0.13.0, psutil-5.6.3,
py-1.8.0,
py_expression_eval-0.3.9, pyasn1-0.4.7, pycparser-2.19, pycrypto-2.6.1,
Pygments-2.4.2, PyNaCl-1.3.0, pyparsing-2.4.2, pytest-5.1.2, python-
dateutil-2.8.0, pytz-2019.2, requests-2.22.0, scandir-1.10.0,
setuptools-41.2.0, setuptools_scm-3.3.3, six-1.12.0, snowballstemmer-1.9.1,
Sphinx-2.2.0, sphinxcontrib-applehelp-1.0.1, sphinxcontrib-devhelp-1.0.1,
sphinxcontrib-htmlhelp-1.0.2, sphinxcontrib-jsmath-1.0.1, sphinxcontrib-
qthelp-1.0.2, sphinxcontrib-serializinghtml-1.1.3, sphinxcontrib-
websupport-1.1.2, tabulate-0.8.3, ujson-1.35, urllib3-1.25.3,
virtualenv-16.7.5, wcwidth-0.1.7, wheel-0.33.6, xlrd-1.2.0, zipp-0.6.0
```

If you need numpy, SciPy, pandas, mpi4py, they are provided by the SciPy-bundle module.

Example to load a recent version of Python with SciPy:

```
ml GCC/8.2.0-2.31.1 OpenMPI/3.1.3 Python/3.7.2 SciPy-bundle/2019.03
```

Custom Python lib

If you need to install a Python library or a different version of the ones already installed, **virtualenv** is the solution.

Python-virtualenv is installed on Baobab: http://www.virtualenv.org/en/latest/

Begin by loading a version of Python using the module system (see above).

Before creating your virtual environment, load the required modules. You must load **Python** and any additional Python packages you need (for example, SciPy-bundle):

```
ml GCC/14.3.0 Python/3.11.6 virtualenv/20.32.0 SciPy-bundle/2025.07
```

Create a new virtual environment (you can choose the location and name).

It is recommended to use the option `--system-site-packages` so that you can benefit from the Python modules provided by the loaded modules.

```
virtualenv --system-site-packages ~/baobab_python_env
```

This will create a directory named **baobab_python_env** in your home directory.

Every time you want to use your virtual environment, you should activate it first:

```
. ~/baobab_python_env/bin/activate
```

Before reusing this virtual environment, you must reload the same modules you used when creating it (e.g., Python and SciPy-bundle). Otherwise, some packages may not be available.

Install all the needed packages in the environment:

```
~/baobab_python_env/bin/pip install mpi4py
```

Use your new environment:

```
~/baobab_python_env/bin/python
```

Pip install from source

By default when you use `pip` to install a library, it will download a binary of `.whl` file instead of building the module from source. This may be an issue if the module itself depend on a custom `libc` version or is optimized for a given kind of CPU. In this case, you can force `pip` to install a module by building it from source.

Example to build hpy from source. The `h5py` argument to `–no-binary` is to specify that you want to build from source only `h5py`.

```
pip install --no-binary h5py h5py
```

R project and RStudio

The latest version of R (if it's not the latest, you can ask us to install it) is installed on the cluster.

Please see here for an sbatch example with R. You will find as well an exemple using the R package parallel.

External helper to create sbatch scripts for R golembash

RStudio

RStudio is IDE (Integrated Development Environment) for R, basically a more user friendly version than plain R.

With the Baobab upgrade to CentOS 7 (cf.

https://hpc-community.unige.ch/t/baobab-migration-from-centos6-to-centos7/361) we do not provide anymore a central RStudio.

Instead, we provide Rstudio on OpenOnDemand.

R packages

You can install R packages as a user. Just follow once the steps given below:

Create a file named `.Rprofile` (note the dot in front of the file) in your home directory with the following content:

```
local({
r = getOption("repos") # hard code the Switzerland repo for CRAN
r["CRAN"] = "https://stat.ethz.ch/CRAN/"
options(repos = r)
})
```

Create a file named `.Renviron` (note the dot in front of the file) in your home directory with the following content:

```
(yggdrasil)-[alberta@login1 ~]$ cat ~/.Renviron
R_LIBS=~/Rpackages/
```

Create a directory where to store the installed R packages:

```
(yggdrasil)-[alberta@login1 ~]$ mkdir ~/Rpackages
```

Once done, make sure you have loaded R with module. Then, from a R command line you can install a R package.

For R version 3.5 and below :

```
install.packages("ggplot2")
```

For R version 3.6 or above :

```
Sys.setenv(R_INSTALL_STAGED = FALSE)
install.packages("ggplot2")
```

Use your newly installed package:

```
library(ggplot2)
```

Example:

```
(yggdrasil)-[alberta@login1 scratch]$ cat sbatch.sh
#!/bin/sh
```

```
#SBATCH --partition=shared-cpu
#SBATCH --time=0:05:00
module load GCC/11.3.0 OpenMPI/4.1.4 R/4.2.1

srun R CMD BATCH test.r
```

```
(yggdrasil)-[alberta@login1 scratch]$ cat test.r
##### Set the working directory #####
#setwd("/srv/beegfs/scratch/users/a/alberta")

##### Clear the global environment #####
rm(list = ls())

Sys.setenv(R_INSTALL_STAGED = FALSE)

##### Install and load packages #####
install.packages("rootSolve")
library(rootSolve)
```

```
(yggdrasil)-[alberta@login1 scratch]$ sbatch sbatch.sh
sbatch sbatch.sh
Submitted batch job 25456882
(yggdrasil)-[alberta@login1 scratch]$ sac -j 25456882
          JobID    JobName    Account      User        NodeList   NTasks
Start                 End      State
--------------- ---------- ---------- --------- --------------- -------- ---
---------------- ------------------- ----------
       25456882  sbatch.sh      burgi   alberta          cpu149
2023-07-14T10:25:13 2023-07-14T10:25:29  COMPLETED
```

The log file is test.r.Rout and you should see everything is working :)

Variant Effect Predictor (VEP)

This tutorials is inspired from VEP documentation:
https://www.ensembl.org/info/docs/tools/vep/script/vep_download.html#singularity

According vep maintainers (https://github.com/Ensembl/ensembl-vep/issues/1515) each users should have one instance of vep. Please edit destination use the following sbatch to install your instance.

```
To install species edit the following option in  the install cmd line:
"-a cfp -s <cpecies_name>  "
```

```
(baobab)-[alberta@login2 ~]$ mkidr $HOME/vep_singularity
(baobab)-[alberta@login2 vep_singularity]$ cd $HOME/vep_singularity
(baobab)-[alberta@login2 vep_singularity]$ vim install_vep.sh

#!/bin/bash
```

```
#SBATCH --job-name=install_vep
#SBATCH --output=output.log
#SBATCH --partition=shared-cpu
#SBATCH --time=01:00:00
#SBATCH --cpus-per-task 12
#SBATCH --mem=12GB
#SBATCH --chdir=/scratch
#SBATCH --export=All



srun singularity pull --name vep.sif docker://ensemblorg/ensembl-vep
srun mkdir vep_data
srun singularity exec vep.sif INSTALL.pl -c vep_data -a p -g all -r vep_data
srun mkdir -p $HOME/vep_singularity/install_dir
srun mv * $HOME/vep_singularity/install_dir
```

Install species

To download species after the installation run the following cmd line(it assumes you have the same directory arch as above):
It ran the install command on debug-cpu slurm partition assuming the install required less than 15 min otherwise use another partition with –time=HH:mm:ss

```
(baobab)-[alberta@login2 ~]$ cd
/home/users/a/alberta/vep_singularity/install_dir
(baobab)-[alberta@login2 ~]$ srun singularity exec vep.sif INSTALL.pl -c
vep_data -a cf -s <species_name>
```

Apptainer (was Singularity)

Intro

Apptainer is a Docker like for HPC. It is available directly on the OS, not through module.
As you don't have root access on Baobab, you cannot build recipe file on Baobab. If you need to do this, you may want to build the image on your own machine and transfer the image to Baobab.
You can download existing images from shub (singularity hub) or from docker (docker public or private registry).
The image will be converted to a read-only squashfs on Baobab disk. It's not possible to have writable images as user. Instead you should build the image as sandbox.

Apptainer images are immutable, but it is possible to append an overlay, see below.

Pull an existing image

Do not run the following commands on login node !! It's using too much cpu

Exemple with Rstudio:

Create a project directory and pull the image from compute node:

```
(baobab)-[alberta@login2 ~]$ MYPROJECT="rocker"
(baobab)-[alberta@login2 ~]$ mkdir -p singularity/$MYPROJECT
(baobab)-[alberta@login2 ~]$ cd !$
(baobab)-[alberta@login2 rocker]$ salloc --partition=shared-cpu --
time=00:30:00 --cpus-per-task 12
(baobab)-[alberta@cpu300 rocker]$ apptainer pull docker://rocker/rstudio:4.2
(baobab)-[alberta@cpu300 rocker]$ ls
rstudio_4.2.sif
```

And Voila; I get my sif image rstudio_4.2.sif

Convert a Docker image

Example to download an existing docker image and build a squashfs image from it:

```
apptainer build lolcow.simg docker://godlovedc/lolcow
```

Then you will end-up with an Apptainer image named `lolcow.simg`.

Example to download an image from a custom registry:

```
apptainer build docker://registry.gitlab.com/flowkit/webservice/compute
```

By default, Apptainer will build your image in `/tmp`. This is an issue on the login node because `tmp` is small. You should instead specify an alternate `tmp` https://apptainer.org/docs/user/main/build_env.html#temporary-folders

Run a container

Once you have a singularity image, you can do various things:

- run a container (the script named singularity at the root of the image, proceed as follow

```
apptainer run lolcow.simg
```

When you import a docker image, the ENTRYPOINT is used to create the singularity run script.

- exec to execute a single command inside the container.

- shell to launch a container and to spawn a shell (/bin/bash)

Modify the image (not persistent)

If you need for example to install a new rpm inside the image, you can use an ephemeral overlay. In the example below, we are using `–fakeroot` to behave as if we were root inside the container.

```
apptainer exec --fakeroot --writable-tmpfs [...]
```

Here you can install an rpm for example, but as soon as you close the container, it is “reset”.

if you are on a compute node, the variable $TMPDIR is set to /scratch. You should either unset it or mount /scratch

Modify the image (persistent)

The way to go with apptainer is to add a writable overlay. This is an ext3 fs. You can create for example a sparse 10G image like that:

```
(baobab)-[sagon@cpu065 ~]$ apptainer overlay create --fakeroot --sparse --
size  10000 my_overlay.img
```

Notice we added the `–fakeroot` flag. If you don't, you won't be able to use the image when apptainer is started with `–fakeroot`

You can then start apptainer

```
(baobab)-[sagon@cpu065 ~]$  apptainer exec --fakeroot --overlay
my_overlay.img [...]
```

References

For more examples, please check the following posts :

- [tutorial] launch openpose with GPU support through Singularity
- Help needed running MPI/Palabos software using Singularity
- Quick guide to Docker, singularity and shifter

Stata

Stata versions

* 14 mp 24 cores * 16 mp 32 cores * 17 mp 32 cores

are availables on the cluster.

To use it, you need to add load Stata using module. Example to load Stata 17:

```
module load Stata/17
```

The Stata binaries are `stata-mp` or `xstata-mp` for the graphical interface.

If you need a graphical interactive session, please proceed as follows from x2go for example:

```
salloc -n1 -c 16 --partition=interactive-cpu --time=15:00 --x11 srun -n1 -N1
--pty $SHELL
xstata-mp
```

Doing so will launch a graphical Stata on a debug node with 16 cores for a 15 minutes session. See on this document for other partition/time limits.

Please keep in mind that the cluster may be full and that you will have to wait until the resources are allocated to you. It's best to launch stata in batch mode instead of using it interactively.

To launch Stata in batch mode, see the Multithreaded jobs section and specify that you want one task and n cpus.

Please see here for an sbatch example with Stata.

TensorFlow

Please see TensorFlow on gitlab for some examples and scripts related to TensorFlow.

ATTENTION , the module:TensorFlow we provide are compiled with GPU support (see https://www.tensorflow.org/install/source#gpu_support), thus they must be used on a GPU partition (cf. Gpu resources and Gpu jobs section).

You can also read this post :

- Getting started with TensorFlow on Baobab (ImportError: libcuda.so.1)

Compile and install a software in your /home

The following posts can inspire you if you need to compile or install a software in your $HOME directory :

- [HOWTO] compile a software in your home directory
- [howto] install and run cudimot
- Remember, you **do not** have sudo rights. Rules and etiquette