## Contents

1. **Introduction**  
2. **System Overview**  
3. **Compiling applications**  
4. **Connecting to CTE-POWER**  
   4.1 Password Management  
   4.2 Transferring files  
5. **File Systems**  
   5.1 Root Filesystem  
   5.2 GPFS Filesystem  
   5.3 Local Hard Drive  
   5.4 NVMe  
   5.5 Quotas  
6. **Running Jobs**  
   6.1 Submitting jobs  
   6.2 Interactive Sessions  
   6.3 Job directives  
7. **Software Environment**  
   7.1 C Compilers  
   7.2 FORTRAN Compilers  
   7.3 Modules Environment  
   7.4 BSC Commands  
8. **Getting help**  
   8.1 Frequently Asked Questions (FAQ)  
9. **Appendices**  
   9.1 SSH  
   9.2 Transferring files  
   9.3 Using X11  
   9.4 Using the DDT debugger
1 Introduction

This user’s guide for the CTE IBM Power9 cluster is intended to provide the minimum amount of information needed by a new user of this system. As such, it assumes that the user is familiar with many of the standard features of supercomputing as the Unix operating system.

Here you can find most of the information you need to use our computing resources and the technical documentation about the machine. Please read carefully this document and if any doubt arises do not hesitate to contact us (Getting help (chapter 8)).

2 System Overview

CTE-POWER is a cluster based on IBM Power9 processors, with a Linux Operating System and an Infiniband interconnection network.

It has the following configuration:

- 2 login node and 52 compute nodes, each of them:
  - 2 x IBM Power9 8335-GTG @ 3.00GHz (20 cores and 4 threads/core, total 160 threads per node)
  - 512GB of main memory distributed in 16 dimms x 32GB @ 2666MHz
  - 2 x SSD 1.9TB as local storage
  - 2 x 3.2TB NVME
  - 4 x GPU NVIDIA V100 (Volta) with 16GB HBM2.
  - Single Port Mellanox EDR
  - GPFS via one fiber link 10 GBit

The operating system is Red Hat Enterprise Linux Server 7.4.

3 Compiling applications

For compiling applications the system provides GCC version 4.8.5, IBM XL compilers for C/C++ v13.1.6 and for Fortran v15.1.6.

Via modules you can find other compilers such as:

- GCC/7.3.0
- PGI/18.1

4 Connecting to CTE-POWER

The first thing you should know is your username and password. Once you have a login and its associated password you can get into the cluster through the following login nodes:

- plogin1.bsc.es
- plogin2.bsc.es

This will provide you with a shell in the login node. There you can compile and prepare your applications.

You must use Secure Shell (ssh) tools to login into or transfer files into the cluster. We do not accept incoming connections from protocols like telnet, ftp, plogin, rcp, or rsh commands. Once you have logged into the cluster you cannot make outgoing connections for security reasons.
4.1 Password Management

In order to change the password, you have to login to a different machine (dt01.bsc.es). This connection must be established from your local machine.

```
% ssh -l username dt01.bsc.es
username@dttransfer1:~> passwd
Changing password for username.
Old Password: 
New Password: 
Reenter New Password: 
Password changed.
```

Mind that that the password change takes about 10 minutes to be effective.

4.2 Transferring files

There are two ways to copy files from/to the Cluster:

- Direct scp or sftp to the login nodes
- Using a Data transfer Machine which shares all the GPFS filesystem for transferring large files

**Direct copy to the login nodes.**

As said before no connections are allowed from inside the cluster to the outside world, so all scp and sftp commands have to be executed from your local machines and never from the cluster. The usage examples are in the next section.

On a Windows system, most of the secure shell clients come with a tool to make secure copies or secure ftp's. There are several tools that accomplish the requirements, please refer to the Appendices (chapter 9), where you will find the most common ones and examples of use.

**Data Transfer Machine**

We provide special machines for file transfer (required for large amounts of data). These machines are dedicated to Data Transfer and are accessible through ssh with the same account credentials as the cluster. They are:

- dt01.bsc.es
- dt02.bsc.es

These machines share the GPFS filesystem with all other BSC HPC machines. Besides scp and sftp, they allow some other useful transfer protocols:

- **scp**

```
localsystem$ scp localfile username@dt01.bsc.es:
username's password:
```

```
localsystem$ sftp username@dt01.bsc.es
username's password:
sftp> put localfile
```

- **sftp**

```
localsystem$ scp username@dt01.bsc.es:remotefile localdir
username's password:
```

```
localsystem$ sftp username@dt01.bsc.es
username's password:
sftp> get remotefile
```
5 File Systems

**IMPORTANT:** It is your responsibility as a user of our facilities to backup all your critical data. We only guarantee a daily backup of user data under /gpfs/home and /gpfs/projects.

Each user has several areas of disk space for storing files. These areas may have size or time limits, please read carefully all this section to know about the policy of usage of each of these filesystems. There are 4 different types of storage available inside a node:

- **Root filesystem:** Is the filesystem where the operating system resides
- **GPFS filesystems:** GPFS is a distributed networked filesystem which can be accessed from all the nodes and Data Transfer Machine (section 4.2)
- **Local hard drive:** Every node has an internal hard drive
- **NVMe:** Non-Volatile Memory Express local partition in every node.

5.1 Root Filesystem

The root file system, where the operating system is stored has its own partition.

There is a separate partition of the local hard drive mounted on /tmp that can be used for storing user data as you can read in Local Hard Drive (section 5.3).

5.2 GPFS Filesystem

The IBM General Parallel File System (GPFS) is a high-performance shared-disk file system providing fast, reliable data access from all nodes of the cluster to a global filesystem. GPFS allows parallel applications simultaneous access to a set of files (even a single file) from any node that has the GPFS file system mounted while providing a high level of control over all file system operations. In addition, GPFS can read or write large blocks of data in a single I/O operation, thereby minimizing overhead.

An incremental backup will be performed daily only for /gpfs/home and /gpfs/projects (not for /gpfs/scratch).

These are the GPFS filesystems available in the machine from all nodes:

- **/apps:** Over this filesystem will reside the applications and libraries that have already been installed on the machine. Take a look at the directories to know the applications available for general use.

- **/gpfs/home:** This filesystem has the home directories of all the users, and when you log in you start in your home directory by default. Every user will have their own home directory to store own developed sources and their personal data. A default quota (section 5.5) will be enforced on all users to limit the amount of data stored there. Also, it is highly discouraged to run jobs from this filesystem. **Please run your jobs on your group’s /gpfs/projects or /gpfs/scratch instead.**
• `/gpfs/projects`: In addition to the home directory, there is a directory in `/gpfs/projects` for each group of users. For instance, the group bsc01 will have a `/gpfs/projects/bsc01` directory ready to use. This space is intended to store data that needs to be shared between the users of the same group or project. A quota (section 5.5) per group will be enforced depending on the space assigned by Access Committee. It is the project's manager responsibility to determine and coordinate the better use of this space, and how it is distributed or shared between their users.

• `/gpfs/scratch`: Each user will have a directory over `/gpfs/scratch`. Its intended use is to store temporary files of your jobs during their execution. A quota (section 5.5) per group will be enforced depending on the space assigned.

5.3 Local Hard Drive

Every node has a local solid-state drive that can be used as a local scratch space to store temporary files during executions of one of your jobs. This space is mounted over `/scratch/tmp/$JOBID` directory and pointed out by `$TMPDIR` environment variable. The amount of space within the `/scratch` filesystem is about 1.6 TB. All data stored in the local hard drives at the compute nodes will not be available from the login nodes. You should use the directory referred to by `$TMPDIR` to save your temporary files during job executions. This directory will automatically be cleaned after the job finishes.

5.4 NVMe

Every node has a local non-volatile memory express drive that can be used as a local working directory to speed-up the code by reducing notably the access time to disk.

5.5 Quotas

The quotas are the amount of storage available for a user or a groups' users. You can picture it as a small disk readily available to you. A default value is applied to all users and groups and cannot be outgrown.

You can inspect your quota anytime you want using the following command from inside each filesystem:

```
% bsc_quota
```

The command provides a readable output for the quota.

If you need more disk space in this filesystem or in any other of the GPFS filesystems, the responsible for your project has to make a request for the extra space needed, specifying the requested space and the reasons why it is needed. For more information or requests you can Contact Us (chapter 8).

6 Running Jobs

Slurm is the utility used for batch processing support, so all jobs must be run through it. This section provides information for getting started with job execution at the Cluster.

6.1 Submitting jobs

The method for submitting jobs is to use the SLURM `sbatch` directives directly.

A job is the execution unit for SLURM. A job is defined by a text file containing a set of directives describing the job's requirements, and the commands to execute.

In order to ensure the proper scheduling of jobs, there are execution limitations in the number of nodes and cpus that can be used at the same time by a group. You may check those limits using command `bsc_queues`. If you need to run an execution bigger than the limits already granted, you may contact support@bsc.es.
SBATCH commands

These are the basic directives to submit jobs with `sbatch`:

```
sbatch <job_script>
```

submits a “job script” to the queue system (see Job directives [section 6.3]).

```
squeue
```

shows all the submitted jobs.

```
scancel <job_id>
```

remove the job from the queue system, canceling the execution of the processes, if they were still running.

6.2 Interactive Sessions

Allocation of an interactive session in the debug partition has to be done through SLURM:

- Interactive session, 64 cores:

  ```
salloc -t 00:10:00 -n 1 -c 64 -J debug srun --pty /bin/bash
  
  You may add -c <ncpus> to allocate n CPUs.
  ```

6.3 Job directives

A job must contain a series of directives to inform the batch system about the characteristics of the job. These directives appear as comments in the job script and have to conform to either the `sbatch` syntaxes.

`sbatch` syntax is of the form:

```
#SBATCH --directive=value
```

Additionally, the job script may contain a set of commands to execute. If not, an external script may be provided with the ‘executable’ directive. Here you may find the most common directives for both syntaxes:

```
# sbatch
#SBATCH --qos=debug

This partition is only intended for small tests.

```
# sbatch
#SBATCH --time=HH:MM:SS

The limit of wall clock time. This is a mandatory field and you must set it to a value greater than real execution time for your application and smaller than the time limits granted to the user. Notice that your job will be killed after the time has passed.

```
# sbatch
#SBATCH -D pathname

The working directory of your job (i.e. where the job will run). If not specified, it is the current working directory at the time the job was submitted.

```
# sbatch
#SBATCH --error=file
```
The name of the file to collect the standard error output (stderr) of the job.

```
# sbatch
#SBATCH --output=file
```

The name of the file to collect the standard output (stdout) of the job.

```
# sbatch
#SBATCH --output
```

The number of processes to start.
Optionally, you can specify how many threads each process would open with the directive:

```
# sbatch
#SBATCH --cpus-per-task=number
```

The number of cpus assigned to the job will be the total_tasks number * cpus_per_task number.

```
# sbatch
#SBATCH --ntasks-per-node=number
```

The number of tasks assigned to a node.

```
# sbatch
#SBATCH --gres=gpu:number
```

The number of GPU assigned to a node.

```
#SBATCH --exclusive
```

To request an exclusive use of a compute node without sharing the resources with other users.

```
# sbatch
#SBATCH --switches=number@timeout
```

By default, Slurm schedules a job in order to use the minimum amount of switches. However, a user can request a specific network topology in order to run his job. Slurm will try to schedule the job for timeout minutes. If it is not possible to request number switches (from 1 to 14) after timeout minutes, Slurm will schedule the job by default.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>SLURM_JOBID</td>
<td>Specifies the job ID of the executing job</td>
</tr>
<tr>
<td>SLURM_NPROCS</td>
<td>Specifies the total number of processes in the job</td>
</tr>
<tr>
<td>SLURM_NNODES</td>
<td>Is the actual number of nodes assigned to run your job</td>
</tr>
<tr>
<td>SLURM_PROCID</td>
<td>Specifies the MPI rank (or relative process ID) for the current process. The range is from 0-(SLURM_NPROCS-1)</td>
</tr>
<tr>
<td>SLURM_NODEID</td>
<td>Specifies relative node ID of the current job. The range is from 0-(SLURM_NNODES-1)</td>
</tr>
<tr>
<td>SLURM_LOCALID</td>
<td>Specifies the node-local task ID for the process within a job</td>
</tr>
</tbody>
</table>
Examples

\textit{sbatch} examples

Example for a sequential job:

```bash
#!/bin/bash
#SBATCH --job-name="test_serial"
#SBATCH -D .
#SBATCH --output=serial_%j.out
#SBATCH --error=serial_%j.err
#SBATCH --n tasks=1
#SBATCH --time=00:02:00
./serial_binary> serial.out
```

The job would be submitted using:

```bash
> sbatch ptest.cmd
```

Example for a parallel job:

```bash
#!/bin/bash
#SBATCH --job-name=test_parallel
#SBATCH -D .
#SBATCH --output=mpi_%j.out
#SBATCH --error=mpi_%j.err
#SBATCH --n tasks=16
#SBATCH --cpus-per-task=4
#SBATCH --gres=gpu:2
mpirun ./parallel_binary> parallel.output
```

7 Software Environment

All software and numerical libraries available at the cluster can be found at /apps/. If you need something that is not there please contact us to get it installed (see Getting Help (chapter 8)).

7.1 C Compilers

In the cluster you can find these C/C++ compilers:

\texttt{xlc /xlc++ -> IBM C/C++ Compilers}

```bash
% xlc --help
% xlc++ --help
```

\texttt{gcc /g++ -> GNU Compilers for C/C++}

```bash
% man gcc
% man g++
```

\texttt{pgcc /pgc++ -> Portland Group Compilers for C/C++}

```bash
% module load pgi
% man gcc
% man g++
```

All invocations of the C or C++ compilers follow these suffix conventions for input files:

- .C, .cc, .cpp, or .cxx -> C++ source file.
- .c -> C source file
- .i -> preprocessed C source file
- .so -> shared object file
- .o -> object file for ld command
- .s -> assembler source file

By default, the preprocessor is run on both C and C++ source files.

These are the default sizes of the standard C/C++ datatypes on the machine:
Table 1: Default datatype sizes on the machine

<table>
<thead>
<tr>
<th>Type</th>
<th>Length (bytes)</th>
</tr>
</thead>
<tbody>
<tr>
<td>bool (c++ only)</td>
<td>1</td>
</tr>
<tr>
<td>char</td>
<td>1</td>
</tr>
<tr>
<td>wchar_t</td>
<td>4</td>
</tr>
<tr>
<td>short</td>
<td>2</td>
</tr>
<tr>
<td>int</td>
<td>4</td>
</tr>
<tr>
<td>long</td>
<td>8</td>
</tr>
<tr>
<td>float</td>
<td>4</td>
</tr>
<tr>
<td>double</td>
<td>8</td>
</tr>
<tr>
<td>long double</td>
<td>16</td>
</tr>
</tbody>
</table>

Distributed Memory Parallelism

To compile MPI programs it is recommended to use the following handy wrappers: mpicc, mpicxx for C and C++ source code. You need to choose the Parallel environment first: module load openmpi/module load ibm_mpi. These wrappers will include all the necessary libraries to build MPI applications without having to specify all the details by hand.

```
% mpicc a.c -o a.exe
% mpicxx a.C -o a.exe
```

Shared Memory Parallelism

OpenMP directives are supported by the GNU/PGI C and C++ compilers. To use it, the flag -fopenmp/-mp must be added to the compile line.

```
% gcc -fopenmp -o exename filename.c
% g++ -fopenmp -o exename filename.C
```

You can also mix MPI + OPENMP code using -fopenmp/-mp with the mpi wrappers mentioned above.

7.2 FORTRAN Compilers

In the cluster you can find these compilers :

xlf -> IBM Fortran Compiler

```
% xlf -qhelp
```

gfortran -> GNU Compilers for FORTRAN

```
% man gfortran
```

pgfortran -> Portland Group Compilers for Fortran

```
% module load pgi
% man pgfortran
```

By default, the compilers expect all FORTRAN source files to have the extension “.f”, and all FORTRAN source files that require preprocessing to have the extension “.F”. The same applies to FORTRAN 90 source files with extensions “.f90” and “.F90”.

Distributed Memory Parallelism

In order to use MPI, again you can use the wrappers mpi77 or mpi90 depending on the source code type. You can always man mpi77 to see a detailed list of options to configure the wrappers, ie: change the default compiler.

```
% mpi77 a.f -o a.exe
```
Shared Memory Parallelism

OpenMP directives are supported by the GNU/PGI Fortran compiler when the option -fopenmp/-mp is set:

```
% gfortran -fopenmp
% pgfortran -mp
```

7.3 Modules Environment

The Environment Modules package [http://modules.sourceforge.net/](http://modules.sourceforge.net/) provides a dynamic modification of a user’s environment via modulefiles. Each modulefile contains the information needed to configure the shell for an application or a compilation. Modules can be loaded and unloaded dynamically, in a clean fashion. All popular shells are supported, including bash, ksh, zsh, sh, csh, tcsh, as well as some scripting languages such as perl.

Installed software packages are divided into five categories:

- **Environment**: modulefiles dedicated to prepare the environment, for example, get all necessary variables to use openmpi to compile or run programs
- **Tools**: useful tools which can be used at any time (php, perl, . . .)
- **Applications**: High Performance Computers programs (GROMACS, . . .)
- **Libraries**: Those are typically loaded at a compilation time, they load into the environment the correct compiler and linker flags (FFTW, LAPACK, . . .)
- **Compilers**: Compiler suites available for the system (intel, gcc, . . .)

Modules tool usage

Modules can be invoked in two ways: by name alone or by name and version. Invoking them by name implies loading the default module version. This is usually the most recent version that has been tested to be stable (recommended) or the only version available.

```
% module load pgi
```

Invoking by version loads the version specified of the application. As of this writing, the previous command and the following one load the same module.

```
% module load pgi/18.1
```

The most important commands for modules are these:

- `module list` shows all the loaded modules
- `module avail` shows all the modules the user is able to load
- `module purge` removes all the loaded modules
- `module load <modulename>` loads the necessary environment variables for the selected modulefile (PATH, MANPATH, LD_LIBRARY_PATH . . .)
- `module unload <modulename>` removes all environment changes made by module load command
- `module switch <oldmodule> <newmodule>` unloads the first module (oldmodule) and loads the second module (newmodule)

You can run “module help” any time to check the command’s usage and options or check the module(1) manpage for further information.
7.4 BSC Commands

The Support team at BSC has provided some commands useful for user’s awareness and ease of use in our HPC machines. A short summary of these commands follows:

- **bsc_queues**: Show the queues the user has access to and their time/resources limits.
- **bsc_quota**: Show a comprehensible quota usage summary for all accessible filesystems.
- **bsc_load**: Displays job load information across all related computing nodes.

You can check more information about these commands through any of the following manpages:

```shell
% man bsc_commands
```

8 Getting help

BSC provides users with excellent consulting assistance. User support consultants are available during normal business hours, Monday to Friday, 09 a.m. to 18 p.m. (CEST time).

User questions and support are handled at: **support@bsc.es**

If you need assistance, please supply us with the nature of the problem, the date and time that the problem occurred, and the location of any other relevant information, such as output files. Please contact BSC if you have any questions or comments regarding policies or procedures.

Our address is:

Barcelona Supercomputing Center - Centro Nacional de Supercomputación
C/ Jordi Girona, 31, Edificio Capilla 08034 Barcelona

8.1 Frequently Asked Questions (FAQ)

You can check the answers to most common questions at BSC’s Support Knowledge Center. There you will find online and updated versions of our documentation, including this guide, and a listing with deeper answers to the most common questions we receive as well as advanced specific questions unfit for a general-purpose user guide.

9 Appendices

9.1 SSH

SSH is a program that enables secure logins over an insecure network. It encrypts all the data passing both ways, so that if it is intercepted it cannot be read. It also replaces the old insecure tools like telnet, rlogin, rcp, ftp, etc. SSH is a client-server software. Both machines must have ssh installed for it to work.

We have already installed a ssh server in our machines. You must have installed an ssh client in your local machine. SSH is available without charge for almost all versions of UNIX (including Linux and MacOS X). For UNIX and derivatives, we recommend using the OpenSSH client, downloadable from [http://www.openssh.org](http://www.openssh.org), and for Windows users we recommend using Putty, a free SSH client that can be downloaded from [http://www.putty.org](http://www.putty.org). Otherwise, any client compatible with SSH version 2 can be used.

This section describes installing, configuring and using the client on Windows machines. No matter your client, you will need to specify the following information:

- Select SSH as default protocol
- Select port 22
- Specify the remote machine and username
For example with putty client:

This is the first window that you will see at putty startup. Once finished, press the **Open** button. If it is your first connection to the machine, you will get a **Warning** telling you that the host key from the server is unknown, and will ask you if you are agree to cache the new host key, press Yes.

**IMPORTANT:** If you see this warning another time and you haven’t modified or reinstalled the ssh client, please do not log in, and contact us as soon as possible (see Getting Help (chapter 8)).

Finally, a new window will appear asking for your login and password:

### 9.2 Transferring files

To transfer files to or from the cluster you need a secure ftp (sftp) or secure copy (scp) client. There are several different clients, but as previously mentioned, we recommend using of Putty clients for trans-

ferring files: **psftp** and **pscp**. You can find it at the same web page as Putty (http://www.putty.org/). Some other possible tools for users requiring graphical file transfers could be:

- **WinSCP**: Freeware Sftp and Scp client for Windows (http://www.winscp.net)
- **SSH**: Not free. (http://www.ssh.org)

### Using PSFTP

You will need a command window to execute psftp (press start button, click run and type cmd). The program first asks for the machine name (mn1.bsc.es), and then for the username and password. Once you are connected, it’s like a Unix command line.

With command **help** you will obtain a list of all possible commands. But the most useful are:

- **get file_name**: To transfer from the cluster to your local machine.
- **put file_name**: To transfer a file from your local machine to the cluster.
- **cd directory**: To change remote working directory.
- **dir**: To list contents of a remote directory.
- **lcd directory**: To change local working directory.
- **!dir**: To list contents of a local directory.

You will be able to copy files from your local machine to the cluster, and from the cluster to your local machine. The syntax is the same that cp command except that for remote files you need to specify the remote machine:

**Copy a file from the cluster**:

```
> pscp.exe username@mn1.bsc.es:remote_file local_file
```

**Copy a file to the cluster**:

```
> pscp.exe local_file username@mn1.bsc.es:remote_file
```
9.3 Using X11

In order to start remote X applications you need and X-Server running in your local machine. Here is a list of most common X-servers for windows:

- Cygwin/X: [http://x.cygwin.com](http://x.cygwin.com)
- X-Win32: [http://www.starnet.com](http://www.starnet.com)
- WinaXe: [http://labf.com](http://labf.com)
- XconnectPro: [http://www.labtam-inc.com](http://www.labtam-inc.com)
- Exceed: [http://www.hummingbird.com](http://www.hummingbird.com)

The only Open Source X-server listed here is Cygwin/X, you need to pay for the others.

Once the X-Server is running run putty with X11 forwarding enabled:

![Putty X11 configuration](image)

Figure 4: Putty X11 configuration

9.4 Using the DDT debugger

Introduction to debugging with DDT

Debugging programs that run on MPI can be fairly cumbersome without the right tools, so we have provided our systems with the DDT program.

DDT is a debugger initially developed by Allinea, now property of ARM. The debugger is specifically designed to be used in HPC environments, as its purpose is to keep track of the state of the program in every MPI node/task it uses.

With DDT you can (but not limited to): * Interactively track and debug program crashes that may occur on certain nodes. * Track memory related problems in your programs. * Use offline (non-interactive) debugging for long running jobs. * Get more information about crashes.

We’ll begin explaining how to set up your environment and job scripts for a simple debugging session.
Basic interactive debugging with DDT

To debug with DDT using an interactive session (as if it was a typical debugger), you need to do some things: you need to compile your program with a debugging flag and then modify your job script so your program is launched with the option to connect to the debugger (note that this is only one of the ways you use DDT).

Compiling your program

To compile your program for debugging purposes, you need to add the following flags to your compiler:
* -g (enabling executable debugging) * -O0 (do not apply optimizations)

For example, the compiling line would be rewritten in the following manner:

```
$ mpicc application.c -o application.exe
$ mpicc -O0 -g application.c -o application.exe
```

Modifying your job script

Your job script needs to be modified so it can launch DDT when the job enters execution in the queue. To do that, you need to specify that you want to connect with the DDT debugger when the job is launched, loading the DDT module and adding these parameters to the line that launches the program:

```
module load DDT
mpirun ./application.exe ddt --connect mpirun ./application.exe
```

Launching the job script

Finally, to launch the job script with the debugger you need to load the DDT module, start the program in background mode and then launch the modified job script:

```
$ module load DDT
$ ddt &
$ sbatch your_modified_jobscript
```

We have provided a capture of a real modified job script as an example:

The DDT main screen will appear, but you have to wait until the job enters execution. To do interactive debugging, we strongly recommend using the debug queue as it normally has a shorter waiting time, but remember that you will have limited resources.

When the job enters execution, you will be prompted with the option to accept the incoming connection. It will give you some options before loading the debugger, mainly so it can know if you use OpenMP, CUDA or some sort of memory debugging.

Once the desired options are selected and the program is loaded, you will see the main GUI for the debugger.

Quick look of common utilities and general usage

DDT largely operates the same way than most classical debuggers for serial applications, with the distinct difference that it can effectively track the state of the execution of every MPI process involved. We've made a general legend of the different utilities present on the main GUI:

1. General debugging actions.
2. Process selector. You can also focus on single processes or threads of a process.
3. Project tree.
5. Variable and stack monitoring window.
6. Input/Output and general tracking utility window.
7. Evaluate window (used to view values for arbitrary expressions and global variables).

Outside the process selector, everything is like a normal debugger and is used in a similar way.
```bash
#!/bin/bash

#SBATCH --job-name=armddt
#SBATCH --nodes=1
#SBATCH --ntasks=8
#SBATCH --time=00:30:00
#SBATCH --output=armddt-%j.log
#SBATCH --error=armddt-%j.err
#SBATCH --qos=benchmark

module load impi
module load DDT

ddt --connect mpirun ./mmult1_c.exe 1024
```

Figure 5: Example of a job script

![Debugging options](image)

Figure 6: Debugging options
Figure 7: Main GUI

Figure 8: Utility legend
Offline debugging

As we know, jobs can take a while to complete or even get into an execution state, so an interactive debugging session may not be the best solution if we expect them to take some time. DDT offers the possibility of offline debugging, allowing us to come back whenever the execution finishes. The execution will generate a file (either a .html or .txt) where you can check the parameters of the execution and the problems that it may have encountered.

To do it, you need to follow the next steps.

Compiling the application for debugging

For this step, you have to compile the application applying the same changes we did in the previous chapter:

```bash
$ mpicc application.c -o application.exe  
$ mpicc -O0 -g application.c -o application.exe
```

Modifying your job script

Make sure that your script loads the ddt module:

```bash
module load DDT
```

And now, modify your launching adding ddt and your desired flags. Note that you have the option to choose between generating a .txt or a .html. We will generate a .html in this example:

```bash
ddt --offline --output=report.html mpirun ./your_application.exe
```

Launching the job

To launch the job, you just need to launch it as if you were launching it normally. Once the execution finishes, the report file will be generated. If it was a .txt, you can check it on the login node itself. The HTML version is more user-friendly and interactive, but needs a web browser to display it, so you will need to transfer it to your local machine.

Here’s an example of a report:

It may have caught your eye that there’s a “Memory Leak Report” tab. DDT allows memory debugging with different granularities, which can be really helpful. Let’s talk more about that in the following chapter.

Enabling memory debugging with DDT

DDT can track down memory related issues like invalid pointers, abnormal memory allocation, memory leaks and more. You can enable memory debugging using two different methods, one for interactive debugging and the other for offline debugging.

Interactive debugging

You don’t have to modify anything for this. When your job requests a connection to DDT, you can check the “Memory Debugging” (which can be seen in Fig. 2) option with the desired parameters.

Offline debugging

For offline debugging you will need to add a simple flag to the execution line inside your job script. Using the line we used for the offline debugging chapter as an example, add this flag:

```bash
$ ddt --offline --mem-debug --output=report.html mpirun ./your_application.exe
```

With this, you should be able to have memory-related information inside your report.
Example of a debugging, step by step

To end this manual, we will provide you a code and we will debug it using DDT. You can follow the same procedures that we will show by yourself. You can get the source code here (copy it to your home folder and extract it):

```bash
# Assuming you're inside your home folder
$ cp /apps/DDT/SRC/DDT_example.tar.gz ~
$ tar xvf DDT_example.tar.gz
```

Inside the generated folder you will see some source code files (one in C and the other one in Fortran, we'll use the C version), a job script and a makefile alongside a solutions folder.

Compiling

Our job is to find and fix what is wrong with the source code, so the first step will be compiling our application using our makefile (feel free to check the contents). This makefile has an option to add the required compiling flags for debugging, so we'll take advantage of it:

```bash
$ make DEBUG=1
```

This will generate the required executable files for when we launch our job script.

Adapting the job script

The job script provided is functional as it is, but we will be doing an interactive debugging session, so you could be waiting for a while. To alleviate that, we will be using the debug queue, which shouldn't have too many waiting jobs. To achieve that, add this line to your job script:

```bash
#SBATCH --qos=debug
```

We're almost ready to launch it!
Launching the program and the debugger

First we need to load our DDT module:

```
$ module load DDT
```

Once we’ve done this, we can launch DDT as a background process:

```
$ ddt &
```

As you read before, the DDT window will appear, but ignore it for now. Now it’s time to launch our job script:

```
$ sbatch job.sub
```

It may take a while, but eventually your job will enter execution and DDT will prompt you with a little window telling you there’s an incoming connection. Accept it. In the next window you don’t need to check any box, just press “Run”.

Locating the issue with the debugger

First of all, let’s talk a bit about the program we are launching. It’s a matrix multiplication implemented with MPI, following this algorithm:

1. Master initializes matrices A, B and C.
2. Master slices the matrices A and C, sends them to slaves.
3. Master and slaves perform the multiplication.
4. Slaves send their results back to master.
5. Master writes the result matrix C in an output file.

Here you have a diagram showing the data distribution:

![Data distribution diagram](image)

Figure 10: Data distribution

Reading the code you can see the detailed implementation. To see if the program works, we can just execute it without any break point. Let’s do that:

If everything works as expected (which is, that it isn’t really working), we should see that DDT prompts us with a notification that our program received a signal (SIGFPE, arithmetic exception) and stopped.
DDT will give us some hints. The first one is the nature of the problem, in this case an integer division by zero. Not only that, it also tells (and shows) the line of code that launched the error. We can deduce that there’s something wrong with the operation “\texttt{size/nslices}”.

Using the window to our right, we can check the values of all variables affected by the current line of code, and we can see that the problem resides in the variable “\texttt{nslices}”, having 0 as its value.

The variable “\texttt{nslices}” is a parameter given to the function “\texttt{mmult}” and it’s not changed anywhere inside it. That means that the value provided to our function is incorrect and we should check how the function was called. Looking through the code, we locate it:

We can see the arguments that this call provided. Specifically, we’re interested in the “\texttt{mr}” variable, which in theory should be the one defining the number of slices used to divide the partition the data of the matrices.

Inspecting the code, we can see that the “\texttt{mr}” variable is not what we thought it was. Why? Because we can see that in reality is the variable that holds the identifier of our MPI rank. Our conclusion is that the error is just putting a wrong variable as a function parameter.

This explains why only process 0 is the only that gives us this problem, as it will be the only one where “\texttt{mr}” equals zero. We also know that the right variable is defined in the code, so we only need to find it and put it as the argument inside the “\texttt{mmult}” call.
Knowing that this program distributes the data into N slices of the matrices (one for each process), we can use the variable “nproc” shown above for that purpose. The only thing left to do is to apply the change to the function call:

```
mmult(size, mr, mat_a, mat_b, mat_c); mmult(size, nproc, mat_a, mat_b, mat_c);
```

And with this, the program should work now. Let’s recompile it and launch it again following the same steps we did for the first version, compiler and all. Once DDT is up and running, we can directly click the continue button. This time, DDT shouldn’t give us any problems and the execution should end normally, as shown see here:

And this is it. We’ve debugged our first application! Although it is a rather simple application and fix, it’s a good exercise to grasp the methodology to use with DDT. We hope you find it useful in future debugging sessions.

**Where can I know more?**

If you need more information about DDT and how to use it, check the reference manual: [link](https://developer.arm.com/docs/101136/0701/ddt)