
amrex Documentation

Release 19.02-dev

AMReX Team

Jan 05, 2019

CONTENTS:

1	Tutorials/Amr	3
2	Tutorials/Basic	5
3	Tutorials/Blueprint	7
4	Tutorials/CVODE	9
5	Tutorials/EB	11
6	Tutorials/Forkjoin	13
7	Tutorials/GPU	15
8	Tutorials/LinearSolvers	17
9	Tutorials/MUI	19
10	Tutorials/Particles	21
11	Tutorials/SDC	23
12	Tutorials/SENSEI	25
13	Tutorials/SWFFT	29
14	Indices and tables	31

AMReX is a software framework library containing all the functionality to write massively parallel, block-structured adaptive mesh refinement (AMR) applications. AMReX is freely available at <https://github.com/AMReX-Codes/amrex>.

AMReX Tutorials are a set of small stand-alone example codes that demonstrate how to use different parts of the AMReX functionality.

We are always happy to have users contribute to AMReX Tutorials as well as the AMReX source code. To contribute, issue a pull request against the development branch (details at <https://help.github.com/articles/creating-a-pull-request/>).

The amrex/Tutorials directory is broken into the following categories:

TUTORIALS/AMR

For each of these tutorials, plotfiles are generated that can be viewed with amrvis2d / amrvis3d (CCSE's native vis / spreadsheet tool, downloadable separately from ccse.lbl.gov) or with VisIt.

1.1 Advection_AmrCore

Advection_AmrCore: This tutorial contains an AMR advection code that advects a single scalar field with a velocity field that is specified on faces.

It is an AMReX based code designed to run in parallel using MPI/OMP.

This example uses source code from the amrex/Src/Base, Boundary, and AmrCore directories.

Notably, this example does not use source code from amrex/Src/Amr (see the tutorial Advection_AmrLevel).

The directory Exec/SingleVortex includes a makefile and a sample inputs file.

1.2 Advection_AmrLevel

Advection_AmrLevel: This tutorial contains an AMR advection code that advects a single scalar field with a velocity field that is specified on faces.

It is an AMReX based code designed to run in parallel using MPI/OMP.

This example uses source code from the amrex/Src/Base, Boundary, AmrCore, and Amr directories.

The directories Exec/SingleVortex and Exec/UniformVelocity each include a makefile and a sample inputs file.

1.3 Advection_F

This code advects a single scalar field with a velocity field that is specified on faces.

It is a AMReX based code designed to run in parallel using MPI/OMP. It uses the Fortran interfaces of AMReX.

The directory Exec/SingleVortex includes a makefile and a sample inputs file.

1.4 Advection_octree_F

This code advects a single scalar field with a velocity field that is specified on faces.

It is a AMReX based code designed to run in parallel using MPI/OMP. It uses the Fortran interfaces of AMReX. The grids have an octree structure with a grid size of 8. No subcycling is used.

The directory Exec/SingleVortex includes a makefile and a sample inputs file.

TUTORIALS/BASIC

The tutorials in `amrex/Tutorials/Basic` demonstrate the most fundamental operations supported by AMReX.

2.1 HelloWorld

`HelloWorld_C` and `HelloWorld_F` demonstrate the GNU Make system – with a sample `Make.package` and `GNUmakefile` – and the `amrex::Initialize` and `amrex::Finalize` functions.

In addition, in `HelloWorld_C`, the `amrex::Print()` operation, which only prints from the I/O processor, is used to print out the AMReX version (as defined by `amrex::Version()`) being used.

`HelloWorld_F` is a simple example of how to use the `F_Interface` routines, which are Fortran wrappers for the underlying C++ data structures and iterators. Here, for example, rather than calling `amrex::Print()` in C++, we test on whether `amrex_parallel_ioprocessor()` is true, and if so, invoke the usual Fortran print call.

2.2 main

`main_C` and `main_F` introduce the following:

1. By default, AMReX initializes MPI and uses `MPI_COMM_WORLD` as its communicator. However, applications could choose to initialize MPI themselves and pass in an existing communicator.
2. By default, AMReX treats command line arguments as input parameters. The expected format of `argv` is

executable inputs_file parm=value

Here, *executable* is the filename of the executable, *inputs_file* is the file containing runtime parameters used to build AMReX ParmParse database, and *parm=value* is an input parameter that will override its value in *inputs_file*. Both *inputs_file* and *parm=value* are optional. At most one *inputs_file* is allowed. However, there can be multiple *parm=value* s.

The parsing of the command line arguments is performed in `amrex::Initialize`. Applications can choose to skip command line parsing. Applications can also provide a function that adds parameters to AMReX ParmParse database.

2.3 HeatEquation

The `HeatEquation` examples solve a 2D or 3D (determined by how you set `DIM` in the `GNUmakefile`) heat equation explicitly on a domain-decomposed mesh. This example is described in detail in the [Basics](#) chapter of the `amrex` Documentation

TUTORIALS/BLEUPRINT

These tests, `AssignMultiLevelDensity` and `HeatEquation_EX1_C`, demonstrate how to convert AMReX Mesh data into an in-memory Conduit Mesh Blueprint description for consumption by the ALPINE Ascent in situ visualization and analysis tool. These are variants, respectively, of `amrex/Tests/Particles/AssignMultiLevelDensity` and `amrex/Tutorials/Basic/HeatEquation_EX1_C`.

For details about what mesh features are currently supported, see: `amrex/Src/Base/AMReX_Conduit_Blueprint.H`

These tests use the interfaces in `Src/Base/AMReX_Conduit_Blueprint.H`, which are built when `USE_CONDUIT=TRUE`. These tests' GNUmakefiles provide a template of how to enable and link Conduit and Ascent.

For more details about Conduit and Ascent, please see:

Conduit: Repo: <https://github.com/llnl/conduit> Docs <http://llnl-conduit.readthedocs.io/en/latest/> Blueprint Docs: <http://llnl-conduit.readthedocs.io/en/latest/blueprint.html>

Ascent: Ascent Repo: <http://github.com/alpine-dav/ascent> Ascent Docs: <http://ascent.readthedocs.io/en/latest/>

(or ping Cyrus Harrison <cyrush@llnl.gov> or Matt Larsen <larsen30@llnl.gov>)

TUTORIALS/CVODE

There are two CVODE tutorials in the `amrex/Tutorials/CVODE` directory, called EX1 and EX2. EX1 consists of a single ODE that is integrated with CVODE within each cell of a 3-D grid. It demonstrates how to initialize the CVODE solver, how to call the ODE right-hand-side (RHS), and, more importantly, how to *re*-initialize the solver between cells, which avoids allocating and freeing solver memory between each cell (see the call to `FCVReInit()` in the `integrate_ode.f90` file in the EX1 directory.)

The EX2 example demonstrates the slightly more complicated case of integrating a system of coupled ODEs within each cell. Similarly to EX1, it provides an RHS and some solver initialization. However, it also demonstrates the performance effect of providing an analytic Jacobian matrix for the system of ODEs, rather than requiring the solver to compute the Jacobian matrix numerically using a finite-difference approach. The tutorial integrates the same system of ODEs on the same 3-D grid, but in one sweep it instructs CVODE to use the analytic function that computes the Jacobian matrix, and in the other case, it does not, which requires CVODE to compute it manually. One observes a significant performance gain by providing the analytic Jacobian function.

See the [CVODE](#) section of the AMReX documentation for general instructions on how to include CVODE in an AMReX application.

TUTORIALS/FORKJOIN

TUTORIALS/GPU

The tutorials in `amrex/Tutorials/GPU` demonstrate the implementation of AMReX's GPU toolkit as well as provide GPU versions of other CPU tutorials for comparison to help applications convert to GPUs.

7.1 Your first AMReX GPU application

This is a step-by-step guide to preparing, compiling and running your first AMReX GPU program. This guide will use `Tutorials/GPU/HeatEquation_EX1_C`, and instructions will focus on ORNL's Summit and Summitdev systems.

Before compiling, the `pgi` and `cuda` software must be available. On ORNL systems, this is done with `module load pgi cuda`. However, Cuda versions 9.2.x are not compatible with AMReX. So, it is important to check whether your system is using 9.2.x as its default. To check, type `module list`. If Cuda is the wrong version, use `module avail cuda` to find a compatible version and swap using `module swap cuda [good module name]`.

Go to `Tutorials/GPU/HeatEquation_EX1_C/Exec/SingleVortex` to compile the executable. Compile with `USE_CUDA=TRUE`, `COMP=pgi`, `USE_MPI=TRUE` and `USE_OMP=FALSE`. This should result in an executable: `main3d.pgi.MPI.CUDA.ex`.

On ORNL's systems, this executable can be ran by using one of the run scripts found in `Tutorials/GPU`. `run.script` can be used to run on Summitdev, and `run.summit` can be used for Summit. To change the number of ranks and GPUs used in the simulation, change the number of resource sets, `n` in the `jsrun` line*i*. Also, set the inputs line to take the appopriate input file based on the dimensionality of your build.

When ready, submit the job (on ORNL: `bsub run.script`). Congratulations! You've accelerated AMReX using GPUs!

7.2 Launch

7.3 HeatEquation_EX1_C

7.4 CNS

7.5 AmrCore

TUTORIALS/LINEARSOLVERS

There are three examples in the Tutorials/LinearSolvers directory.

`ABecLaplacian_C` demonstrates how to solve with cell-centered data in a C++ framework. This example shows how to use either hypre or PETSc as a bottom-solver (or to solve the equation at the finest level if you set the “max coarsening level” to 0).

`ABecLaplacian_F` demonstrates how to solve with cell-centered data using the Fortran interfaces.

`NodalPoisson` demonstrates how to solve with nodal data using the C++ framework.

TUTORIALS/MUI

The goal of this tutorial is to incorporate the MxUI/MUI (Multiscale Universal Interface) framework into AMReX. This framework allows two separate executables to communicate with one another in parallel using MPI. In addition, this framework is adaptable for different geometries, in which the bounds of data one would like to send and/or receive can be specified using the `announce_send_span()` and `announce_recv_span()` commands.

In this tutorial, two different C++ codes are built separately. Each has different spatial dimensions: one is built in 3D (`AMREX_SPACEDIM = 3`), and the other in 2D (`AMREX_SPACEDIM = 2`). Each code is compiled separately within their respective “exec” directories `Exec_01` & `Exec_02`, after which the two executables are run together using the following command, specifying the number of MPI processes to designate to each executable:

```
$ mpirun -np N1 ../Exec_01/main3d.gnu.MPI.ex inputs  
: -np n2 ../Exec_02/main2d.gnu.MPI.ex inputs
```

on a single line within the `Exec_coupled` directory. `N1` and `n2` are the number of MPI ranks designated for each executable, respectively. Each executable is given the same `inputs` file within `Exec_coupled`. Input variables `max_grid_size_3d` and `max_grid_size_2d` determine the respective grid sizes for 3D and 2D. As far as I am aware, the code works for any AMReX grid structure. Details of how to build and run the code are contained in the script `cmd_mpirun`.

The figure below shows one possible grid structure of the 2D (red grid) and 3D (multicolored blocks) setup.



MUI interface: 2D and 3D grid setup

The 3D code initializes a 3D MultiFab (Note: with no ghost cells), and sends a 2D slice of this data at the $k = 0$ location to the 2D executable, which stores the data in a 2D MultiFab, multiplies the data by a constant, and sends the modified platter back to the 3D executable. Finally, the 3D executable receives the modified data and places it back into the 3D MultiFab, at $k = 0$.

The 2D, original 3D, and modified 3D data are all written to separate plot files, which can be visualized using software such as Amrvis.

Although our code does not include this, it would be possible to pair an AMReX code with code that is outside of the AMReX framework, because each code is compiled separately. For example, using the `announce_send_span()` and `announce_recv_span()` commands, MUI would be able to determine the overlap between the two regions to correctly exchange the data, even if the two grid structures differ.

TUTORIALS/PARTICLES

There are several tutorials in `amrex/Tutorials/Particles` that demonstrate the basic usage of AMReX's particle data structures.

10.1 ElectrostaticPIC

This tutorial demonstrates how to perform an electrostatic Particle-in-Cell calculation using AMReX. The code initializes a single particle in a conducting box (i.e. Dirichlet zero boundary conditions) that is slightly off-center in one direction. Because of the boundary conditions, the particle sees an image charge and is accelerated in this direction.

The code is currently set up to use one level of static mesh refinement. The charge density, electric field, and electrostatic potential are all defined on the mesh nodes. To solve Poisson's equation, we use AMReX's Fortran-based multigrid solver. The Fortran routines for performing charge deposition, field gathering, and the particle push are all defined in `electrostatic_pic_2d.f90` and `electrostatic_pic_3d.f90` for 2D and 3D, respectively.

The particle container in this example uses a Struct-of-Arrays layout, with $1 + 2 \times \text{BL_SPACEDIM}$ real components to store the particle weight, velocity, and the electric field interpolated to the particle position. To see how to set up such a particle container, see `ElectrostaticParticleContainer.H`.

10.2 ElectromagneticPIC

This tutorial shows how to perform an electromagnetic particle-in-cell calculation using AMReX. Essentially, this is a mini-app version of the WarpX application code. The electric fields, magnetic fields, and current densities are stored using the staggered Yee grid, and it solves Maxwell's Equations using the finite-difference time domain method.

This tutorial also demonstrates how to offload calculations involving particle data onto the GPU using OpenACC. To compile with GPU support, use the `pgi` compiler, and set `USE_ACC = TRUE`, and `USE_CUDA = TRUE`, `USE_OMP = FALSE`.

You can choose between two problem types by toggling the `problem_type` parameter in the provided inputs file. Choosing the uniform plasma setup provides a nearly perfectly load balanced problem setup that is useful for performance testing. Choosing the Langmuir wave problem will automatically compare the simulated fields to the exact solution.

Currently, this tutorial does not use mesh refinement.

10.3 NeighborList

This tutorial demonstrates how to have AMReX's particles undergo short-range collisions with each other. To facilitate this, a neighbor list data structure is created, in which all of the partners that could potentially collide with a given

particle are pre-computed. This is done by first constructing a cell-linked list, and then looping over all 27 neighbor cells to test for potential collision partners. The Fortran subroutine `amrex_compute_forces_nl` defined in `neighbor_list_2d.f90` and `neighbor_list_3d.f90` demonstrates how to loop over the resulting data structure.

The particles in this example store velocity and acceleration in addition to the default components. They are initially placed at cell centers and given random velocities. When a particle reaches the domain boundary, it is specularly reflected back into the domain. To see how the particle data structures are set up, see `NeighborListParticleContainer.cpp`.

The file called `inputs` can be used to run this tutorial with a single level, and `inputs.mr` sets up a run with static mesh refinement.

10.4 CellSortedParticles

Sometimes, it's useful to sort particles at a finer granularity than grids or tiles. In this Tutorial, each cell contains a list of particle indices that tell you which particles belong to that cell. This is useful, for example, in Direct Simulation Monte Carlo calculations, where you want to potentially interact particles that are in the same cell as each other. Every time the particles move, we check to see whether it's still in the same cell or not. If it isn't, we mark the particle as unsorted. We then call `Redistribute()` as normal, and then insert the unsorted particles into the proper cells. Care is taken so that, if the `Redistribute` call changes the order of the particles in the Container, the indices in the cell lists are updated accordingly.

This Tutorial is currently single-level only.

11.1 IMEX_Advec_Diff_C

11.2 MISDC_ADR_2d

This tutorial presents an example of using a “multi-implicit” spectral deferred corrections (MISDC) integrator to solve a simple scalar advection-diffusion-reaction equation in two dimensions. Both diffusion and reaction terms are treated implicitly but solved for independently in an operator splitting fashion. The advection is treated explicitly. The relative strengths of the three terms can be adjusted by changing the coefficients a , d , and r in `inputs_2d`.

The advection operator is a 4th-order centered difference in flux form. The diffusion operator is a 2nd order discretization of the Laplacian, and the implicit diffusion solve is done using multigrid. The “reaction” term here is just a simple linear damping hence the implicit solve is trivial. See the routines in `functions_2d.f90` for the code that evaluates the rhs terms.

The simple form of the equation allows for an exact solution of the PDE in a periodic geometry. There is a flag called “`plot_err`” in `main.cpp`, which if set equal 1 will cause the code to output the error in the solution for plotting. If the advection term is omitted ($a=0$), then an exact solution to the method of lines ODE is computed and used to compute the error. Hence the error in this case will scale in dt with the order of the time integrator.

This code can also be run as an IMEX advection-diffusion example simply by setting `Npieces=2` in `main.cpp`. This should also be equivalent to setting $r=0$.

TUTORIALS/SENSEI

SENSEI is a middleware that allows one to send data to various visualization and analysis back ends through a uniform interface. It's data model and API enable one to chose the desired visualization and analysis back end for a given task with out limiting ones options, as the back ends can be inter-changed at run time via a text based config file.

12.1 Configuring the environment at NERSC

First select the desired SENSEI install. Each install will support different set of backends. This is necessary because not all of the back ends are compatible with each other.

For instance to use SENSEI with ParaView Catalyst:

```
module load sensei/2.1.0-catalyst
```

To use SENSEI with VisIt Libsim:

```
module load sensei/2.1.0-libsim
```

SENSEI features in AMReX are conditionally compiled when the Make file variable `USE_SENSEI_INSITU` is set. When this variable is set, the Make file will query environment variables to determine the list of include directories and link libraries needed to compile with SENSEI.

The `sensei_config` tool that is installed with SENSEI will set the environment variables that are used in the make files.

With a SENSEI module loaded, in a bash shell:

```
source sensei_config
```

There are two SENSEI tutorials included with AMReX, for for use with `AmrCore`, and one for `AmrLevel`.

12.2 Compiling and Running the `Advection_AmrCore` tutorial

This example uses source code from the `amrex/Src/Base`, `Boundary`, and `AmrCore` directories. Notably, this example does not use source code from `amrex/Src/Amr` (see the tutorial `Advection_AmrLevel`).

The directory `Exec/SingleVortex` in `Tutorials/SENSEI/Advection_AmrCore` includes a makefile and a sample inputs file. Plotfiles are generated that can be viewed with `amrvis2d` / `amrvis3d` (CCSE's native vis / spreadsheet tool, downloadable separately from ccse.lbl.gov) or with VisIt.

Edit the file `Exec/SingleVortex/GNUMakefile`, set

```
USE_SENSEI_INSITU = TRUE
```

Build the tutorial

```
make -j4
```

To use SENSEI in AMReX one needs to enable it via ParmParse input file. Additionally one needs to provide a SENSEI XML configuration that selects and configures the desired SENSEI backend.

Example XML configs are included in Exec/SingleVortex/SENSEI.

Edit the file Exec/SingleVortex/inputs

Running with ParaView Catalyst:

```
sensei.enabled = 1           # turn SENSEI in situ on/off
sensei.config = SENSEI/render_catalyst.xml # render simulation data with ParaView_
↪Catalyst
sensei.frequency = 1         # number of level 0 steps between in situ_
↪processing
```

Running with VisIt Libsim:

```
sensei.enabled = 1           # turn SENSEI in situ on/off
sensei.config = SENSEI/render_libsim.xml  # render simulation data with VisIt Libsim
sensei.frequency = 1         # number of level 0 steps between in situ_
↪processing
```

Once the inputs files has been edited, run the executable as usual

```
mpiexec -np 4 ./main2d.gnu.MPI.ex inputs
```

12.3 Compiling and Running the Advection_AmrLevel tutorial

This example uses source code from the amrex/Src/Base, Boundary, Amrlevel, and Amr directories.

The directories Exec/SingleVortex and Exec/UniformVelocity in Tutorials/SENSEI/Advection_AmrLevel each include a makefile and a sample inputs file. Plotfiles are generated that can be viewed with amrvis2d / amrvis3d (CCSE's native vis / spreadsheet tool, downloadable separately from ccse.lbl.gov) or with VisIt.

Edit the file Exec/SingleVortex/GNUMakefile, set

```
USE_SENSEI_INSITU = TRUE
```

Finally, make the tutorial

```
make -j4
```

Running ## To use SENSEI in AMReX one needs to enable it via ParmParse input file. Additionally one needs to provide a SENSEI XML configuration that selects and configures the desired SENSEI backend.

Example XML configs are included in Exec/SingleVortex/SENSEI.

Edit the file Exec/SingleVortex/inputs

Running with ParaView Catalyst:

```
sensei.enabled = 1           # turn SENSEI in situ on/off
sensei.config = SENSEI/render_catalyst.xml # render simulation data with ParaView_
↪Catalyst
sensei.frequency = 1         # number of level 0 steps between in situ_
↪processing
```

Running with VisIt Libsim:

```
sensei.enabled = 1           # turn SENSEI in situ on/off
sensei.config = SENSEI/render_libsim.xml  # render simulation data with VisIt Libsim
sensei.frequency = 1         # number of level 0 steps between in situ_
↪processing
```

Once the inputs files has been edited, run the execcutable as usual

```
mpiexec -np 4 ./main2d.gnu.MPI.ex inputs
```


TUTORIALS/SWFFT

This Tutorial demonstrates how to call the SWFFT wrapper to the FFTW3 solver.

Note that the SWFFT source code was developed by Adrian Pope and colleagues and is available at:

<https://xgitlab.cels.anl.gov/hacc/SWFFT>

Please refer to the AMReX documentation at https://amrex-codes.github.io/amrex/docs_html/SWFFT.html for a brief explanation of how the SWFFT redistributes data into pencil grids.

AMReX contains two SWFFT tutorials, `SWFFT_poisson` and `SWFFT_simple`:

- `SWFFT_poisson` tutorial: The tutorial found in `amrex/Tutorials/SWFFT/SWFFT_poisson` solves a Poisson equation with periodic boundary conditions. In it, both a forward FFT and reverse FFT are called to solve the equation, however, no reordering of the DFT data in k-space is performed.
- `SWFFT_simple` tutorial: This tutorial: `amrex/Tutorials/SWFFT/SWFFT_simple`, is useful if the objective is to simply take a forward FFT of data, and the DFT's ordering in k-space matters to the user. This tutorial initializes a 3D or 2D `MultiFab`, takes a forward FFT, and then redistributes the data in k-space back to the "correct," 0 to 2π , ordering. The results are written to a plot file.

13.1 SWFFT_poisson

In this test case we set up a right hand side (rhs), call the forward transform, modify the coefficients, then call the backward solver and output the solution to the discrete Poisson equation.

To build the code, type 'make' in `amrex/Tutorials/SWFFT/SWFFT_poisson`. This will include code from `amrex/Src/Extern/SWFFT` and you will need to link to the FFT solvers themselves (on NERSC's Cori machine, for example, you would need to "module load fft")

To run the code, type 'main3d.gnu.MPI.ex inputs' in this directory

To visualize the output, set the bool `write_data` to true, then use `amrvis3d` (source available at <https://github.com/AMReX-Codes/Amrvis>):

```
amrvis3d -mf RHS SOL_EXACT SOL_COMP
```

to visualize the rhs, the exact solution and the computed solution.

The max norm of the difference between the exact and computed solution is also printed.

For instructions on how to take a forward FFT only using SWFFT, please refer to *SWFFT_simple*.

13.2 SWFFT_simple

This tutorial initializes a 3D or 2D `MultiFab`, takes a forward FFT, and then redistributes the data in k-space back to the “correct,” 0 to 2π , ordering. The results are written to a plot file.

In a similar fashion to the `SWFFT_poisson` tutorial:

To build the code, type ‘make’ in `amrex/Tutorials/SWFFT/SWFFT_simple`. This will include code from `amrex/Src/Extern/SWFFT` and you will need to link to the FFT solvers themselves (on NERSC’s Cori machine, for example, you would need to “module load fft”)

To run the code, type ‘main*.ex inputs.oneGrid’ in this directory to run the code in serial. To run the code in parallel, type ‘mpirun -n \$N main*.ex inputs.multipleGrids’ instead, where `N` holds the number of MPI processes (equal to the number of grids). `run_me_2d` and `run_me_3d` also provide examples of how to run the code.

Use `amrvis2d` or `amrvis3d` to visualize the output (source available at <https://github.com/AMReX-Codes/Amrvis>):

```
amrvis${dims}d plt_fft*
```

where `dims` specifies `AMREX_SPACEDIM`. The DFT of the data and the original data are labeled as `FFT_of_phi` and `phi` within the plot file.

The `SWFFT_poisson` tutorial provides an example of solving a Poisson equation using a discrete spectral method, in which a forward and reverse FFT of a `MultiFab` are computed.

INDICES AND TABLES

- [genindex](#)
- [modindex](#)
- [search](#)

The copyright notice of AMReX is included in the AMReX home directory as README.txt.

Your use of this software is under a 3-clause BSD license with additional modification – the license agreement is included in the AMReX home directory as license.txt.

For a pdf version of this documentation, click [here](#).