

---

# **amrex Documentation**

***Release 18.01-dev***

**AMReX Team**

**Dec 07, 2018**



**CONTENTS:**

<b>1</b>	<b>Tutorials/Amr</b>	<b>3</b>
<b>2</b>	<b>Tutorials/Basic</b>	<b>5</b>
<b>3</b>	<b>Tutorials/Blueprint</b>	<b>7</b>
<b>4</b>	<b>Tutorials/CVODE</b>	<b>9</b>
<b>5</b>	<b>Tutorials/EB</b>	<b>11</b>
<b>6</b>	<b>Tutorials/GPU</b>	<b>13</b>
<b>7</b>	<b>Tutorials/LinearSolvers</b>	<b>15</b>
<b>8</b>	<b>Tutorials/MUI</b>	<b>17</b>
<b>9</b>	<b>Tutorials/Particles</b>	<b>19</b>
<b>10</b>	<b>Tutorials/SDC</b>	<b>21</b>
<b>11</b>	<b>Tutorials/SENSEI</b>	<b>23</b>
<b>12</b>	<b>Tutorials/SWFFT</b>	<b>25</b>
<b>13</b>	<b>Indices and tables</b>	<b>27</b>



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**





## 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](mailto:cyrush@llnl.gov)> or Matt Larsen <[larsen30@llnl.gov](mailto: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/GPU**



## 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**



## TUTORIALS/PARTICLES

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

### 9.1 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 using a Struct-of-Arrays layout, with `1 + 2*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`.

### 9.2 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_n1` 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`.





**TUTORIALS/SDC****10.1 IMEX\_Advec\_Diff\_C****10.2 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**



## 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 for a brief explanation of how the SWFFT redistributes data into pencil grids.

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

- `SWFFT` tutorial: The tutorial found in `amrex/Tutorials/SWFFT` 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_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  $2\pi$ , ordering. The results are written to a plot file.

### 12.1 SWFFT

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`. 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*.

## 12.2 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  $2\pi$ , ordering. The results are written to a plot file.

In a similar fashion to the `SWFFT` tutorial:

To build the code, type ‘make’ in `amrex/Tutorials/SWFFT_simple`. This will include code from `amrex/Src/Extern/SWFFT_simple` 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* 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](#).