How To Build & Run WRF-Hydro V5 in Standalone Mode

And Create Customized Geographical Inputs & Regrid Forcing Data

Creation Date: 14 April 2018
WRF-Hydro Development Team

Purpose

This document serves two purposes. It is meant to guide you through the WRF-Hydro model build process utilizing an example test case and then guides you through a workflow for creating customized geographical inputs and forcing data for running in standalone mode with the Noah-MP land surface model. For users who want to run the Noah land surface model see the WRF-Hydro V5 Technical Description. There are many ways to set up and run a WRF-Hydro simulation. This guide provides one way of setting up the needed directories and files. For a detailed description of the WRF-Hydro Modeling System see the *WRF-Hydro V5 Technical Description* available from the website https://ral.ucar.edu/projects/wrf_hydro.

WRF-Hydro has been developed for linux-based operating systems including small local clusters and high performance computing systems. Additionally, the model code has also been ported to a selection of virtual machine environments (e.g. "containers") to enable use of small domain cases on many common desktop computing platforms (e.g. Windows and MacOS).

WRF-Hydro is typically set up as a computationally-intensive modeling system. Simple small domains can be configured to run on a desktop platform. Large-domain model runs can require hundreds or thousands of processors. We recommend beginning with example test cases we supply at the WRF-Hydro website https://ral.ucar.edu/projects/wrf_hydro before moving to your domain of interest, particularly if your domain is reasonably large.

Table of Contents

Purpose

Table of Contents

- Requirements
- Building WRF-Hydro
- Running WRF-Hydro
  - WRF-Hydro run time options
  - Run WRF-Hydro with an example test case

WRF-Hydro Standalone Workflow for creating & using customized input for use with the Noah-MP land surface model

Programs you will need to download, install, configure, compile and run for working with the Noah-MP Land Surface Model

Input you will need to provide

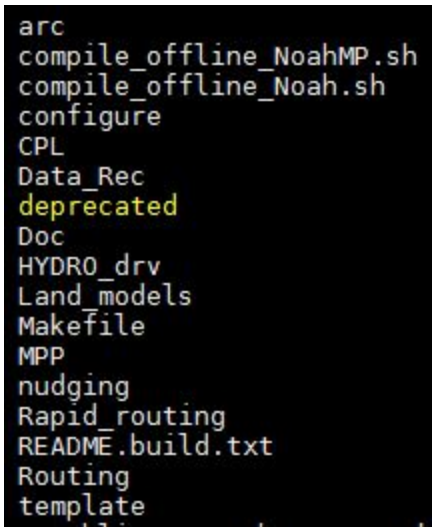
- Define your domain by creating geogrid file(s) using WRF-ARW, WPS, and Global Input Data
- Create your model initial conditions
- Create your full domain file, geospatial metadata, and optional parameter and routing files using the WRF-Hydro GIS Pre-processing Toolkit
- Gather and regrid forcing data
- Directories and their contents
- Run WRF-Hydro with your customized geographic inputs, idealized forcing, and the land surface model (LSM) only
  - Run: Simulation setup and execution
- Run WRF-Hydro again with custom geographic inputs & custom forcing data with the land surface model only
- Run WRF-Hydro with custom geographic inputs, custom forcing data, & minimal routing physics.
- Evaluation
- Considerations for full model simulation

Requirements

- A compiler. WRF-Hydro supported compilers are the Portland Group Fortran compiler, the Intel 'ifort' compiler and the public license GNU Fortran compiler 'gfort' (for use with Linux-based operating systems on desktops and clusters)
- MPICH or OpenMPI
- NetCDF C & Fortran Libraries version 4 or greater available from https://www.unidata.ucar.edu/software/netcdf/ (These libraries need to be compiled with the same compilers as you will use to compile WRF-Hydro)
- WRF-Hydro version 5 model code: This provides physics options, hydrologic routing options, spatial and temporal configurations.
    - https://ral.ucar.edu/projects/wrf_hydro/model-code
    - https://ral.ucar.edu/projects/wrf_hydro/technical-description-user-guide
- Example test case with provided input files, forcing data, and output files to compare against. Used for testing your build and also provides example model configurations
    - https://ral.ucar.edu/projects/wrf_hydro/testcases

WRF-Hydro requires a number of machine and distribution-specific libraries. While we are not able to cover the breadth of all possible configurations, we offer an example of dependency installation for a base Ubuntu Linux distribution to serve as a starting point for other configurations. It is available on our website under FAQs.

It will be necessary to untar and uncompress the model code on your machine. Detailed information regarding the model code is provided in the *WRF-Hydro V5 Technical Description* available from https://ral.ucar.edu/projects/wrf_hydro. Below is an example of listed model code directories under the `trunk/NDHMS/` level.



Building WRF-Hydro

The following steps configure your machine to run the uncoupled WRF-Hydro model. The example below is written in bash. Other shells will require different, yet similar syntax.

1. Change directories to `trunk/NDHMS`

2. Make a copy of the template environment variable file
   `cp template/setEnvar.sh .`

3. Edit the WRF-Hydro environment variables (compile time options) in the file `setEnvar.sh` with a text editor. Users are responsible for understanding these options and selecting the compile time options to suit their needs.

Compile time options are choices about the model structure which are determined when the model is compiled. Most users generally choose the compile options which suit their needs and seldom explore others. Compile time options are listed in the `trunk/NDHMS/template/setEnvar.sh script.` The information in that script is annotated here. As the path implies, this script is a template file. The compileconfigure script in `trunk/NDHMS/compile*shconfigure` can accept the variables listed in this file if they are set in your environment. You can also optionaly supply a file containing the variables as the first argument to the script. However this is not recommended if you intend to use `make` with the code.

Below is one example configuration. Descriptions are indicated with <<-- and blue text.

```
#!/bin/bash

# WRF-Hydro compile time options

# Always set WRF_HYDRO to 1
export WRF_HYDRO=1

# Enhanced diagnostic output for debugging: 0=Off, 1=On.
export HYDRO_D=0

# Spatially distributed parameters for NoahMP: 0=Off, 1=On.
export SPATIAL_SOIL=1 <<-- This allows NoahMP to use spatially distributed parameters for
the land surface model rather than parameter based upon soil class and land use category look up
tables. See section

# RAPID model: 0=Off, 1=On.
export WRF_HYDRO_RAPID=0 <<-- Coupling with the RAPID routing model. This option is not
currently supported.

# Large netcdf file support: 0=Off, 1=On.
export WRFIO_NCD_LARGE_FILE_SUPPORT=1 <<-- Allow netCDF I/O of files larger than 2GiB.

# WCOSS file units: 0=Off, 1=On.
export NCEP_WCOSS=0 <<- Do not use unless working on the WCOSS machines.

# Streamflow nudging: 0=Off, 1=On.
export WRF_HYDRO_NUDGING=0 <<-- Enable the streamflow nudging routines for
Muskingum-Cunge Routing. See the WRF-Hydro Technical Description for more information.
export NCEP_WCOSS=0 <<- Do not use unless working on the WCOSS machines.
```

4. NetCDF Environment
   The compiler needs to know where to find your netCDF 4+ installation.
   If `nc-config` is in your path (or you can find it and run it, replacing it with its full path), simply run the following (bash example)
   `export NETCDF=`nc-config --prefix``
   If `nc-config` is not in your path then, export the following environment variables.
   `export NETCDF_INC="/netCdfPathOnYourMachine/include"`
   `export NETCDF_LIB="/netCdfPathOnYourMachine/lib"`
   Note that we do not currently support parallel netCDF.
5. Run the configure script.
   The configure script establishes environment variables and compile flags for your build.
   The configuration step chooses the compiler to be used for compiling the model. Enter the following:
   `./configure`

   The following compiler list will appear on you screen:

```
Please select from following supported options.
   1. Linux PGI compiler dmpar
   2. Linux gfort compiler dmpar
   3. Linux ifort compiler dmpar
   4. Linux ifort compiler dmpar (WCOSS Luna)
   0. exit only

Enter selection [0-4]:
```

   Select the compiler you wish to use. This generates the `macros` and the `Makefile.mpp` files in the working directory from available options in `arc/`.
   At this point you should verify that all of the variables and pathways that need to be specified in the 'macros' file are properly setup or linked (e.g. COMPILER90, CPP, NETCDFINC, NETCDFLIB). Most compile-time errors or issues relate to the settings in the `macros` file. As noted above, as of version 5, we require a minimum of netCDF 4.0. The configure script establishes environment variables and compile flags for your build. It is hoped that the setup matches your needs, however some additional configuration of the appropriate file in the `arc/` directory may be necessary.
   Compile options are embedded in the `macros` file. Running configure again will clobber any changes to these files in the working directory.

We support compiling the standalone WRF-Hydro model with both the Noah or NoahMP land surface models.
To compile WRF-Hydro offline with Noah land surface model, the following command is used:
`./compile_offline_Noah.sh setEnvar.sh`
To compile WRF-Hydro offline with the NoahMP land surface model, the following script is used:
`./compile_offline_NoahMP.sh setEnvar.sh`
Both scripts execute a similar process for their respective land surface models. This process is described below for the NoahMP model. Note that the workflow instructions for customizing your model geographical inputs and forcing data that come later in this guide are for NoahMP. For information on working with Noah please see the *WRF-Hydro V5 Technical Description*.

- Environment variables in the `setEnvar.sh` file will be sourced by the compile script.
- In the working directory, `LandModel/ and LandModel_cpl` directories are replaced with those for NoahMP. The `Makefile` is also taken from the NoahMP source directory and placed in the current directory.
- The script runs: `make clean`

- The script runs: `make`
  Note that after initially invoking `./compile_offline_NoahMP.sh` to ensure correct setup of of the model, one may simply run make in the working directory to only rebuild the changed dependency files.
- The return value of `make` is queried to print a success/failure message about the compilation. Unfortunately, the make return code does not seem to be 100% correlated with a successful `make`. If the make was not successful, the script exits at this point.
- If `make` (compilation) is successful, the executable file created for the uncoupled WRF-Hydro model is created for NoahMP in the `Run/` directory:
  `wrf_hydro.exe`
- Then, in `Run/`, `wrf_hydro.exe` is copied to `wrf_hydro_NoahMP.exe` and then `wrf_hydro_NoahMP.exe` is symlinked to `wrf_hydro.exe`.
  Note: if running `make` after previously running the `compile_offline_NoahMP.sh` script, only the `wrf_hydro.exe` file is updated. To avoid confusion use a `make clean` before each compile.
- Finally, the environment variables used in the compile are printed.

If compilation was unsuccessful, oftentimes compilation error messages will be provided with module names and line numbers. Note that these module line numbers are ONLY relevant to lowercase (*.f) files and NOT uppercase (*.F) source files. The lowercase (*.f) files are the result of the C-preprocessing set applied to the uppercase, source files (*.F). The *.f files are scrubbed within the Makefile upon compilation they are not available to view. To not scrub the .f files one needs to make the appropriate edit to the appropriate Makefile. This means determining which make file needs to be edited and commenting out the 'rm *.f' line under the appropriate module.

When compiling with both Noah and NoahMP, the contents of the new `Run/` directory contain example namelists and the correct lookup (tabular) parameter files for running the compiled executable. It is recommended to copy these files to a separate location where you will perform your modeling experiments.

Running WRF-Hydro

WRF-Hydro run time options

There are two namelist files that users must edit in order to successfully execute the WRF-Hydro system in a standalone mode or 'uncoupled' to WRF. One of these namelist files is the `hydro.namelist` file and in it are the various settings for operating all of the routing components of the WRF-Hydro system. The `hydro.namelist` file is internally commented so that it should be clear as to what is needed for each setting. A full annotated example of the 'hydro.namelist' file is provided in Appendix 5 of the WRF-Hydro Technical Description.

The second namelist is the namelist which specifies the land surface model options to be used. This namelist can change depending on which land model is to be used in conjunction with the WRF-Hydro routing components. When the standalone WRF-Hydro model is compiled the appropriate `namelist.hrldas` template file is copied over to the `Run` directory based upon the specified land surface model.

In WRF-Hydro v5.0, the Noah and Noah-MP land surface models are the main land surface model options when WRF-Hydro is run in an uncoupled mode. Both Noah and Noah-MP use a namelist file called `namelist.hrldas`, which will contain different settings for the two different land surface models. Full annotated example `namelist.hrldas` files for Noah and Noah-MP are provided in the *WRF-Hydro V5 Technical Description.*

The example test cases provided on our website have namelists configured for different routing options. The next step will test your build using an example test case.

Run WRF-Hydro with an example test case

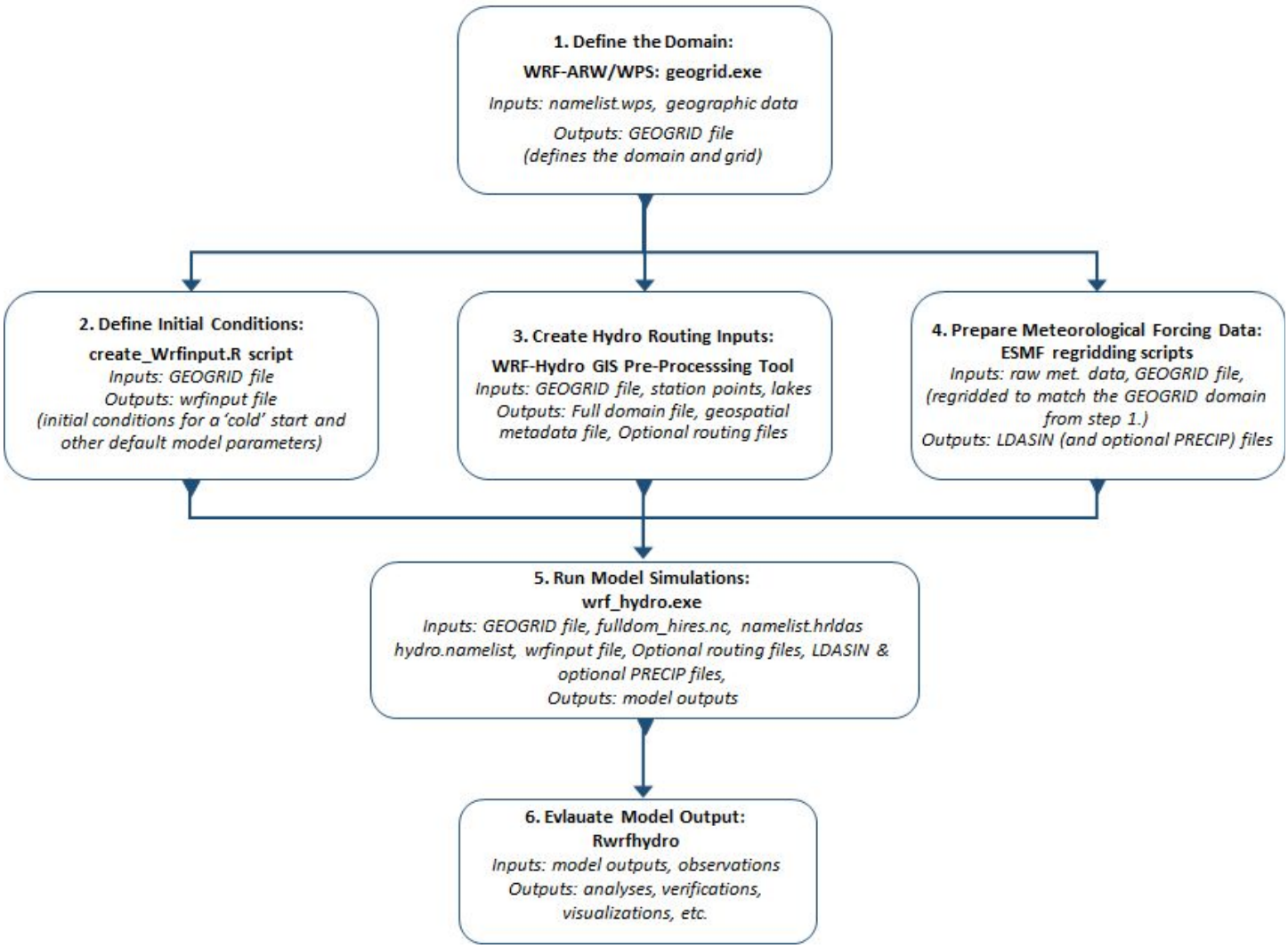
Once you have successfully built the WRF-Hydro model, we recommend that you download and run our example test case in order to familiarize yourself with the process of running the model and ensure that the model is functioning properly as it is installed on your system. Example test cases contain a prepared geographical domain, the required inputs, preconfigured namelists for different routing configurations, and prepared forcing data. Download a test case from https://ral.ucar.edu/projects/wrf_hydro/testcases and follow the *Test Case User Guide* and `Readme.txt` file. The *Test Case User Guide* includes information on how to check if your simulation ran by checking the diagnostic files, and comparing the restart files from the simulation run with those provided.

You can also utilize the example test case to explore the namelists of the different routing configurations.

After you have WRF-Hydro built and running successfully, the next step is to create customized input for your geographic region of interest and your research interests. The next sections of this document will walk you through that process.

WRF-Hydro Standalone Workflow for creating & using customized input for use with the Noah-MP land surface model

The above workflow is for utilization with the NoahMP land surface model only. For information and guidance on running with the Noah land surface model see the *WRF-Hydro V5 Technical Description*.



If you have not already done so create the following directories to hold your customized input.

- **DATA:** Will contain the WRF Preprocessing System (WPS) Geographical Input Data
- **DOMAIN:** Will contain your geospatial and hydrographic input files
- **FORCING:** Will contain all of the meteorological forcing data you wish to use

Programs you will need to download, install, configure, compile and run for working with the Noah-MP Land Surface Model

- **WRF-ARW** (latest version)**:** http://www2.mmm.ucar.edu/wrf/users/download/get_source.html
  - In the stand alone offline mode, WRF-ARW is only needed to compile WPS
  - http://www2.mmm.ucar.edu/wrf/OnLineTutorial/compilation_tutorial.php
- **WRF Preprocessing System (WPS)** (latest version): This will provide a 'namelist.wps' file that you will need to edit, and a geogrid executable which defines model domains and interpolates static geographical data to grids. The output are GEOGRID netCDF files.
  - http://www2.mmm.ucar.edu/wrf/users/download/get_source.html
  - http://www2.mmm.ucar.edu/wrf/OnLineTutorial/compilation_tutorial.php
  - http://www2.mmm.ucar.edu/wrf/users/docs/user_guide_V3/users_guide_chap3.htm
- **create_wrfinput_file.R script**
  - This is a script to create a very basic WRF-Hydro initialization file (wrfinput) from a WRF GEOGRID file and a set of user-specified conditions. The script produces fields of spatially uniform initial model states of soil moisture, soil temperature, soil liquid water content and skin temperature among a few other variables necessary for model cold-start initialization
  - https://ral.ucar.edu/projects/wrf_hydro/pre-processing-tools
- **WRF-Hydro GIS Pre-processing Tool**: This tool will create the data layers for terrestrial overland flow, subsurface flow and channel routing processes required by WRF Hydro. The outputs are geospatial and tabular data layers.
  - This will use the GEOGRID netCDF files created by WPS, your DEM and other optional inputs to create a high resolution full domain file '`Fulldom_hires.nc`' which will be used as input into WRF-Hydro. It will also output optional parameter, spatial metadata, and routing files.
  - https://ral.ucar.edu/projects/wrf_hydro/pre-processing-tools
  - https://github.com/NCAR/wrf_hydro_arcgis_preprocessor
- **(Optional) ESMF Regridding scripts:**
  - Script packages to regrid meteorological forcing data to a smaller, completely encompassed WRF-Hydro domain which will match your GEOGRID domain.
  - https://ral.ucar.edu/projects/wrf_hydro/regridding-scripts
- **(Optional) Rwrfhydro:** There are many ways to visualize and evaluate model output. We provide Rwrfhydro, an R package for evaluating model output.
  - https://github.com/NCAR/rwrfhydro
  - https://github.com/NCAR/rwrfhydro/blob/master/README.Rmd

Other Software & Libraries you will need:

Windows OS, ArcGIS 10.3.1, Spatial Analyst extension, R, NCO, the NetCDF4 R package, NCL 6.1 or later.

Input you will need to provide

- **Geographical input data**
  WRF Preprocessing System (WPS) Geographical Input Data Downloads
  http://www2.mmm.ucar.edu/wrf/users/download/get_sources_wps_geog.html
  Global input data to build your domain :the domain's physical location on the globe and the static information for that location must be created. This requires a data set that includes such fields as topography and land use categories.
    - The data expands to approximately 10 GB. This data allows a user to run the `geogrid.exe` program within WPS.
    - `geogrid.exe` will look for this data and if not there will not execute
    -

- **Domain Coordinates:** These will be used in the WRF Preprocessing System (WPS) steps as input in the file 'namelist.wps' which will then help produce GEOGRID files which will then be used by WRF-Hydro.
  If you do not already have coordinates, you can obtain them from Google Earth or a standard GIS system. Another tool that is available is the WRF Domain Wizard
  https://esrl.noaa.gov/gsd/wrfportal/DomainWizard.html

- **Digital Elevation Model/ Elevation Data:** This will be used as input in the WRF-Hydro GIS Pre-processing Tool.
  If you do not already have a DEM or elevation data you can obtain them from
    - USGS HydroSHEDS https://hydrosheds.cr.usgs.gov/dataavail.php
    - National Hydrography Dataset Plus, Version 2 http://www.horizon-systems.com/nhdplus/NHDPlusV2_home.php
    - EU-DEM http://www.eea.europa.eu/data-and-maps/data/eu-dem

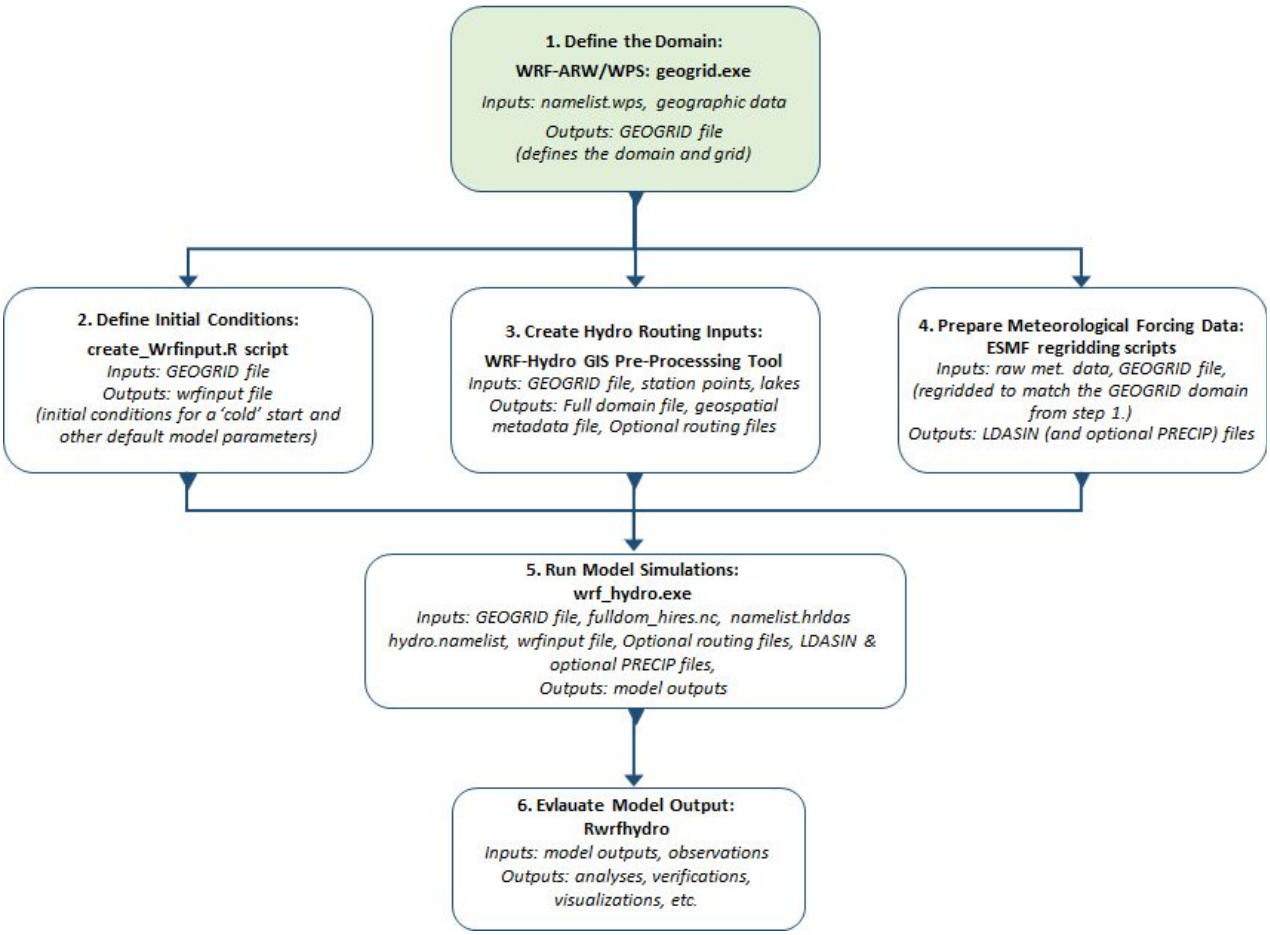
  The DEM/elevation data must be in a raster format, readable by ArcGIS such as GeoTIFF and ESRI GRID.
  The DEM/elevation data must have a valid coordinate reference system, meaning that the elevation values are georeferenced.
  The DEM/elevation data must cover the entire extent of your GEOGRID domain. It is best to have a slightly larger area as a buffer.
  Elevation units must be converted to meters (m)
  It is also suggested that the DEM/elevation data be hydrologically corrected (i.e. processed to ensure that rivers and drainage divides are in the right places)
    - Not required but this helps with channel placement, hydro enforcement, etc.

- **(Optional) Forecast points:** at least a .csv file that includes FID, latitude, longitude, and station name
  These are optional input in the WRF-Hydro GIS Pre-processing Tool

- **(Optional) Lake polygons:** a polygon feature class or .shp file.
  These are optional input in the WRF-Hydro GIS Pre-processing Tool

- **Meteorological Forcing Data:** Data that will "force" your model run. Obtain data for your simulation period and regrid it to your domain so that the model can use it.

Define your domain by creating geogrid file(s) using WRF-ARW, WPS, and Global Input Data



1. Obtain/Download global WRF Preprocessing System (WPS) Geographical Input Data

- http://www2.mmm.ucar.edu/wrf/users/download/get_sources_wps_geog.html
- *http://www2.mmm.ucar.edu/wrf/OnLineTutorial/compilation_tutorial.php*
- Keep in mind that if you are downloading the complete dataset, the file is very large. If you are sharing space on a cluster, you may want to consider placing this in a central location so that everyone can use it, and it's not necessary to download for each person.
- See http://www2.mmm.ucar.edu/wrf/users/docs/user_guide_V3/users_guide_chap3.htm for information on selecting between USGS and MODIS land use Data

2. Uncompress and un-tar the file. Add the data to the **DATA** directory that you created.

3. Download, Compile and Run WRF-ARW. Follow the online guide to install, and compile WRF-ARW http://www2.mmm.ucar.edu/wrf/OnLineTutorial/compilation_tutorial.php. See the tutorial for a list of requirements and dependencies.

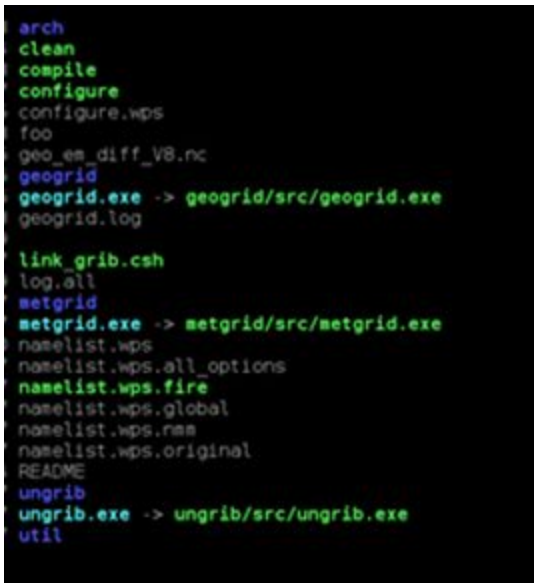
The WRF-ARW directory should contain these files and directories after installation:

```
arch
clean
compile
configure
configure.wrf
configure.wrf.backup
dyn_em
dyn_exp
dyn_nmm
external
frame
inc
log.compile
main
Makefile
phys
README
README.DA
README.hybrid_vert_coord
README.hydro
README.io_config
README.NMM
README.rsl_output
README.SSIB
README_test_cases
README.windturbine
Registry
run
share
test
tools
```

If compiled correctly it should contain these files and directories:

```
aerosol.formatted
aerosol_lat.formatted
aerosol_lon.formatted
aerosol_plev.formatted
bulkdens.asc_s_0_03_0_9
bulkradii.asc_s_0_03_0_9
CAM_ABS_DATA
CAM_AEROPT_DATA
CAMtr_volume_mixing_ratio.A1B
CAMtr_volume_mixing_ratio.A2
CAMtr_volume_mixing_ratio.RCP4.5
CAMtr_volume_mixing_ratio.RCP6
CAMtr_volume_mixing_ratio.RCP8.5
capacity.asc
CCN_ACTIVATE.BIN
CLM_ALB_ICE_DFS_DATA
CLM_ALB_ICE_DRC_DATA
CLM_ASM_ICE_DFS_DATA
CLM_ASM_ICE_DRC_DATA
CLM_DRDSDT0_DATA
CLM_EXT_ICE_DFS_DATA
CLM_EXT_ICE_DRC_DATA
CLM_KAPPA_DATA
CLM_TAU_DATA
co2_trans
coeff_p.asc
coeff_q.asc
constants.asc
create_p3_lookupTable_1.f90
ETAMPNEW_DATA
ETAMPNEW_DATA_DBL
ETAMPNEW_DATA.expanded_rain
ETAMPNEW_DATA.expanded_rain_DBL
GENPARM.TBL
grib2map.tbl
gribmap.txt
kernels.asc_s_0_03_0_9
kernels_z.asc
LANDUSE.TBL
masses.asc
MPTABLE.TBL
namelist.input
ndown.exe -> ../main/ndown.exe
ozone.formatted
ozone_lat.formatted
ozone_plev.formatted
p3_lookup_table_1.dat
README.namelist
README.rasm_diag
README.tslist
real.exe -> ../main/real.exe
RRTM_DATA
RRTM_DATA_DBL
RRTMG_LW_DATA
RRTMG_LW_DATA_DBL
RRTMG_SW_DATA
RRTMG_SW_DATA_DBL
SOILPARM.TBL
tc.exe -> ../main/tc.exe
termvels.asc
tr49t67
tr49t85
tr67t85
URBPARM.TBL
URBPARM_UZE.TBL
VEGPARM.TBL
wind-turbine-1.tbl
wrf.exe -> ../main/wrf.exe
```

4. Install and compile WPS. Follow the online guide http://www2.mmm.ucar.edu/wrf/OnLineTutorial/compilation_tutorial.php.
If compiled correctly, WPS should contain these files and directories:



The two things that you will be utilizing in the WPS directory are the `namelist.wps` and the `geogrid.exe` program. You will be executing `geogrid.exe` which will define model domains and interpolate static geographical data to the grids.

5. In the WPS directory you will need to open and edit the `namelist.wps` text file with your favorite text editor.

- Variables in the "geogrid" section primarily define the size and location of all model domains, and where the static geographical data are found.
- Edit the "geogrid" section of with your domain information.
- Specify the path to your geographical input data.
- Edit the "geog_data_res" section to specify land use classifications.
  See http://www2.mmm.ucar.edu/wrf/users/docs/user_guide_V3/users_guide_chap3.htm for an explanation on selecting between USGS and MODIS land use classifications.
  Note: For first time model set up it is recommended to choose a relatively small domain size (e.g. 16 km^2). Processing time is heavily machine and hardware dependent. However, in general, a smaller domain will process faster and you can check that your model is setup and running correctly before moving on to a larger domain.

Key components and their definitions

`max_dom` : number of domains (there can be multiple, but it is recommended to start out with 1.)
`map_proj, truelat1, truelat2, stand_lon, ref_lat, ref_lon`: map projections definitions
`dx, dy`: model resolution (for lambert, in meters)
`e_we, e_sn`: model dimensions (how many grid points)
`parent_grid_ratio`: for nested domain, how many nested grids per parent grid
`i_parent_start, j_parent_start`: where the lower left corner of nest is relative to parent

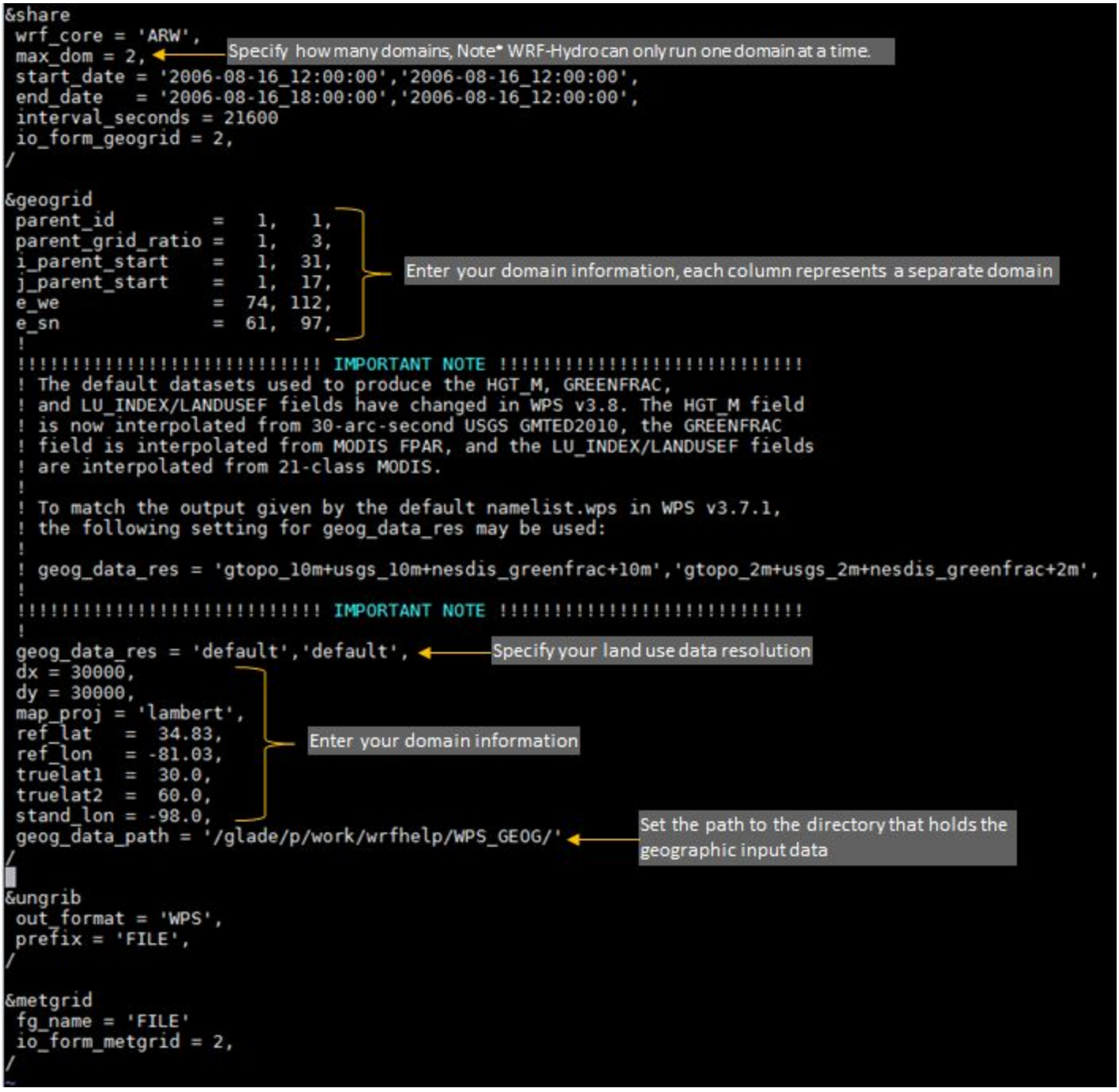
Use the coordinate system that is appropriate for your location: Mercator is appropriate for low-latitude regions. Lambert Conformal Conic is appropriate for mid-latitude regions. Polar Stereographic is appropriate for high-latitude regions.

WRF-Hydro Currently supported GEOGRID coordinate systems:

MAP_PROJ = 1 (Lambert Conformal Conic)
MAP_PROJ = 3 (Mercator)
MAP_PROJ = 6 (Cylindrical Equidistant but NOT w/ rotated pole)
MAP_PROJ = 2 (Polar Stereographic)

Note: the WRF Hydro GIS Pre-processing Toolkit (the next step in the Workflow) cannot handle: Cylindrical Equidistant with a rotated pole.

An example `namelist.wps` file is below.



Variables are defined in the tables below.

Table 1. Namelist.wps: (Fortran) namelist options for the geogrid program taken from WPS documentation. For further explanation see the WPS user guide

PARENT_ID	A list of MAX_DOM integers specifying, for each nest, the domain number of the nest's parent; for the coarsest domain, this variable should be set to 1. Default value is 1.
PARENT_GRID_RATIO	A list of MAX_DOM integers specifying, for each nest, the nesting ratio relative to the domain's parent. No default value.
I_PARENT_START	A list of MAX_DOM integers specifying, for each nest, the x-coordinate of the lower-left corner of the nest in the parent unstaggered grid. For the coarsest domain, a value of 1 should be specified. No default value.
J_PARENT_START	A list of MAX_DOM integers specifying, for each nest, the y-coordinate of the lower-left corner of the nest in the parent unstaggered grid. For the coarsest domain, a value of 1 should be specified. No default value.
S_WE	A list of MAX_DOM integers which should all be set to 1. Default value is 1.
E_WE	A list of MAX_DOM integers specifying, for each nest, the nest's full west-east dimension. For nested domains, e_we must be one greater than an integer multiple of the nest's parent_grid_ratio (i.e., e_we = n*parent_grid_ratio+1 for some positive integer n). No default value.
S_SN	A list of MAX_DOM integers which should all be set to 1. Default value is 1.
E_SN	A list of MAX_DOM integers specifying, for each nest, the nest's full south-north dimension. For nested domains, e_sn must be one greater than an integer multiple of the nest's parent_grid_ratio (i.e., e_sn = n*parent_grid_ratio+1 for some positive integer n). No default value.

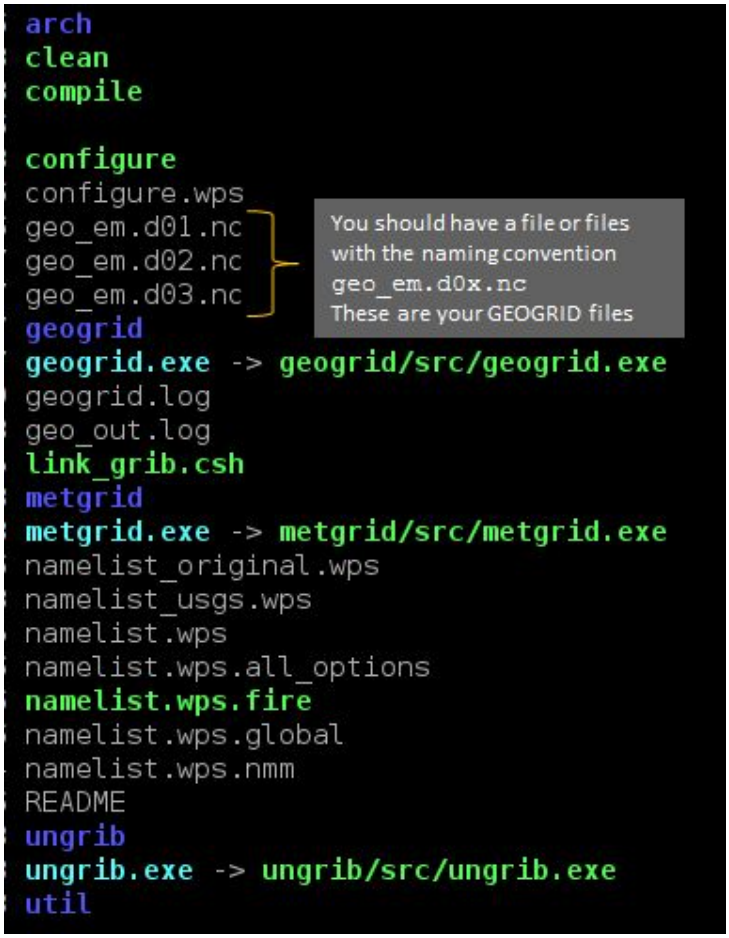
GEOG_DATA_RES	A list of MAX_DOM character strings specifying, for each nest, a corresponding resolution or list of resolutions separated by + symbols of source data to be used when interpolating static terrestrial data to the nest's grid. For each nest, this string should contain a resolution matching a string preceding a colon in a rel_path or abs_path specification (see the description of GEOGRID.TBL options) in the GEOGRID.TBL file for each field. If a resolution in the string does not match any such string in a rel_path or abs_path specification for a field in GEOGRID.TBL, a default resolution of data for that field, if one is specified, will be used. If multiple resolutions match, the first resolution to match a string in a rel_path or abs_path specification in the GEOGRID.TBL file will be used. Default value is 'default'.
DX	A real value specifying the grid distance in the x-direction where the map scale factor is 1. For ARW, the grid distance is in meters for the 'polar', 'lambert', and 'mercator' projection, and in degrees longitude for the 'lat-lon' projection; for NMM, the grid distance is in degrees longitude. Grid distances for nests are determined recursively based on values specified for parent_grid_ratio and parent_id. No default value.
DY	A real value specifying the nominal grid distance in the y-direction where the map scale factor is 1. For ARW, the grid distance is in meters for the 'polar', 'lambert', and 'mercator' projection, and in degrees latitude for the 'lat-lon' projection; for NMM, the grid distance is in degrees latitude. Grid distances for nests are determined recursively based on values specified for parent_grid_ratio and parent_id. No default value.
MAP_PROJ	A character string specifying the projection of the simulation domain. For ARW, accepted projections are 'lambert', 'polar', 'mercator', and 'lat-lon'; for NMM, a projection of 'rotated_ll' must be specified. Default value is 'lambert'.
REF_LAT	A real value specifying the latitude part of a (latitude, longitude) location whose (i,j) location in the simulation domain is known. For ARW, ref_lat gives the latitude of the center-point of the coarse domain by default (i.e., when ref_x and ref_y are not specified). For NMM, ref_lat always gives the latitude to which the origin is rotated. No default value.
REF_LON	A real value specifying the longitude part of a (latitude, longitude) location whose (i, j) location in the simulation domain is known. For ARW, ref_lon gives the longitude of the center-point of the coarse domain by default (i.e., when ref_x and ref_y are not specified). For NMM, ref_lon always gives the longitude to which the origin is rotated. For both ARW and NMM, west longitudes are negative, and the value of ref_lon should be in the range [-180, 180]. No default value.
TRUELAT1	A real value specifying, for ARW, the first true latitude for the Lambert conformal projection, or the only true latitude for the Mercator and polar stereographic projections. For NMM, truelat1 is ignored. No default value.
TRUELAT2	A real value specifying, for ARW, the second true latitude for the Lambert conformal conic projection. For all other projections, truelat2 is ignored. No default value.
STAND_LON	A real value specifying, for ARW, the longitude that is parallel with the y-axis in the Lambert conformal and polar stereographic projections. For the regular latitude-longitude projection, this value gives the rotation about the earth's geographic poles. For NMM, stand_lon is ignored. No default value.
GEOG_DATA_PATH	A character string giving the path, either relative or absolute, to the directory where the geographical data directories may be found. This path is the one to which rel_path specifications in the GEOGRID.TBL file are given in relation to. No default value.

6. After editing, close the `namelist.wps`.

7. In the WPS directory, run the `geogrid.exe`. This will create your domain.

The command is **`./geogrid.exe`**

After running `geogrid.exe` your WPS directory should contain the following files and directories:



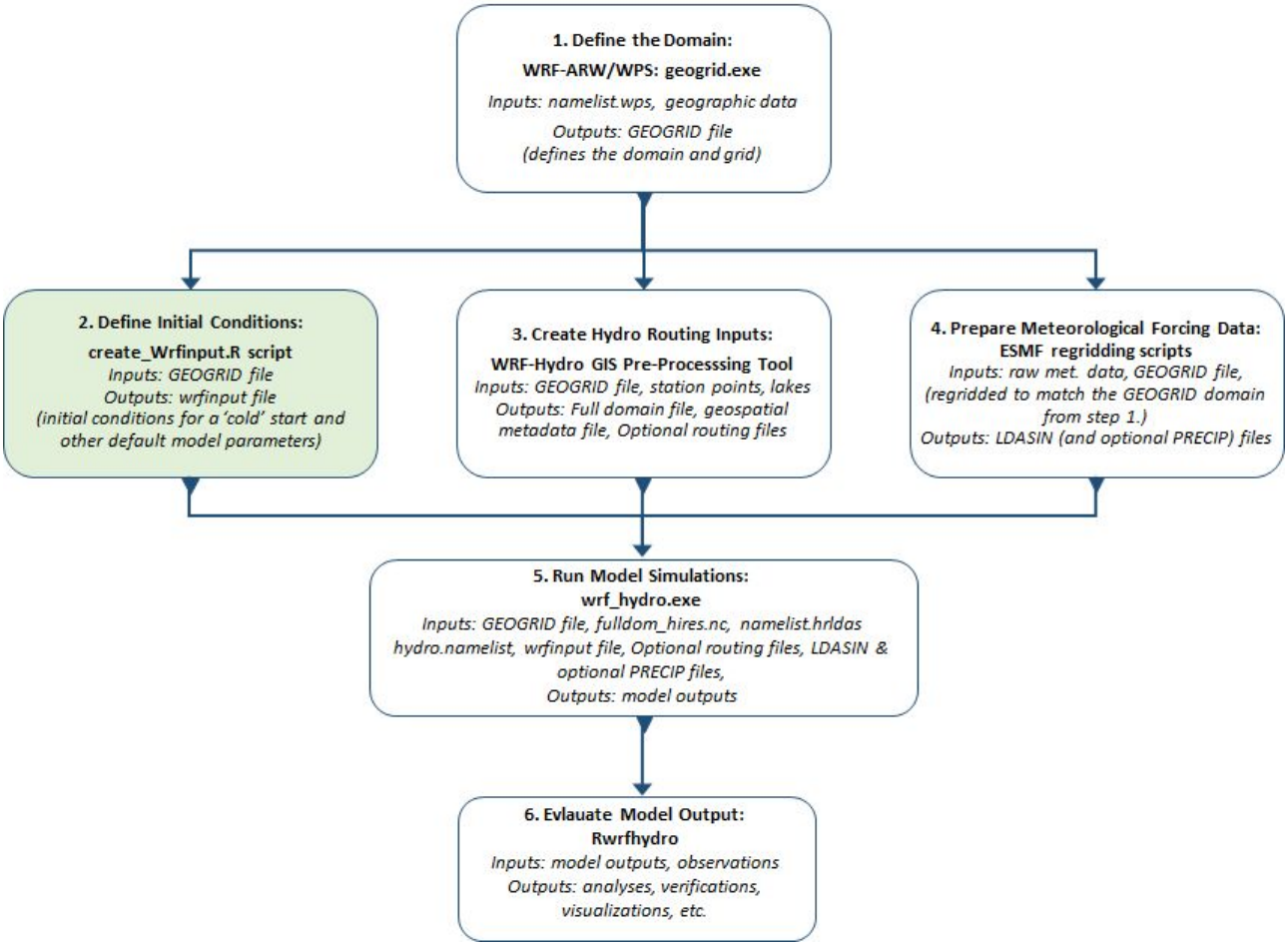
One geogrid netCDF file with the naming convention `geo_em.d0x.nc` should have been created for each domain you specified (i.e. if you set `max_dom = 3` in the `namelist.wps` and filled it out accordingly you should get 3 geogrid files) . The geogrid file contains coordinate system information contained in global attributes and static 2D and 3D gridded variables. It contains the size, spatial location, horizontal resolution of the grid cells, as well as other projection parameters that define the coordinate system. The geogrid file will be utilized by the *WRF-Hydro GIS Pre-processing Toolkit*, the *create_Wrfinput.R script*, the *ESMF regridding scripts* and *WRF-Hydro*.

Note: Each following step in the workflow, and WRF-Hydro can only utilize one GEOGRID file at a time.

See Chapter 3 in the WPS user guide for WPS output fields global attributes and fields for a description of the `geo_em.d0x.nc` "geogrid" files.

8. Transfer the `geo_em.d0x.nc` file to your **DOMAIN** directory.

Create your model initial conditions



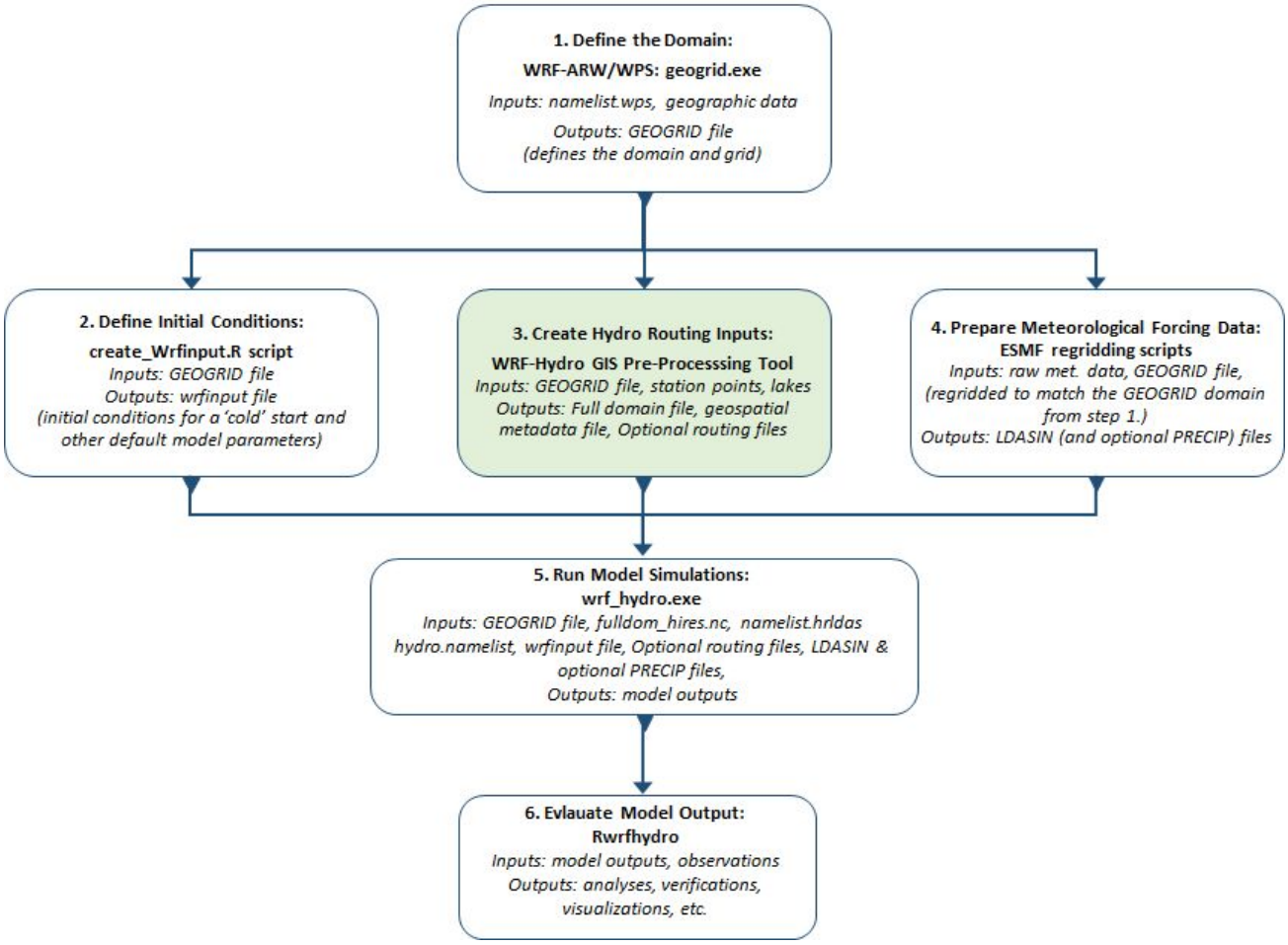
There are two ways to create initial conditions for your model run. One is to utilize the WRF program `real.exe`. This requires you to build and run WRF. For information on `real.exe` see the WRF documentation. However, if using the Noah-MP land surface model (LSM) you can also utilize the `create_WRFinput.R` script to create a very basic WRF-Hydro initialization file from a WPS GEOGRID file and a set of user-specified conditions. It produces fields of spatially uniform initial model states of soil moisture, soil temperature, soil liquid water content and skin temperature among a few other variables necessary for model execution. This output file is used as a “cold start” for longterm model spin.

Requirements: R, NCO, and the NetCDF4 R package

Outputs: `wrfinput_d0x.nc` this file provides the initial conditions for a “cold” start and other default model parameters. It is a 2d file containing initialization information for the Noah-MP LSM (e.g., starting soil moisture and snow states); Note: some (but not all) of these variables are replaced when doing a "warm" start from a restart file.

1. Download the create_WRFinput.R script from https://ral.ucar.edu/projects/wrf_hydro/pre-processing-tools
2. Update script to point to the location of your geogrid file `geo_em.d0x.nc`.
4. Update script to specify the desired soil category value to be used in case of conflicts between soil water and land cover water cells.
5. Update script to specify the desired month to use for leaf area index (LAI) initialization.
6. Add the `wrfinput_d0x.nc` file to your **DOMAIN** directory.

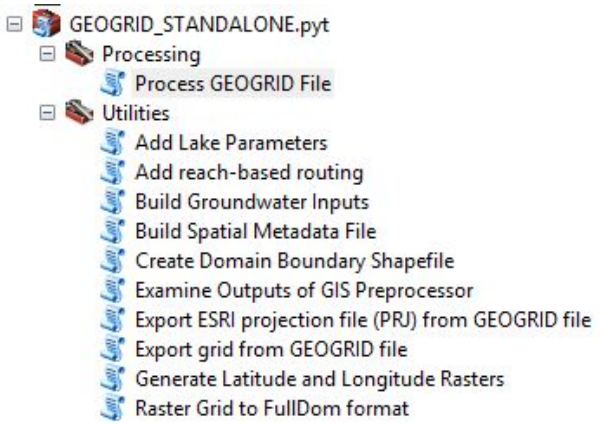
Create your hydrologic routing input files using the WRF-Hydro GIS Pre-processing Toolkit

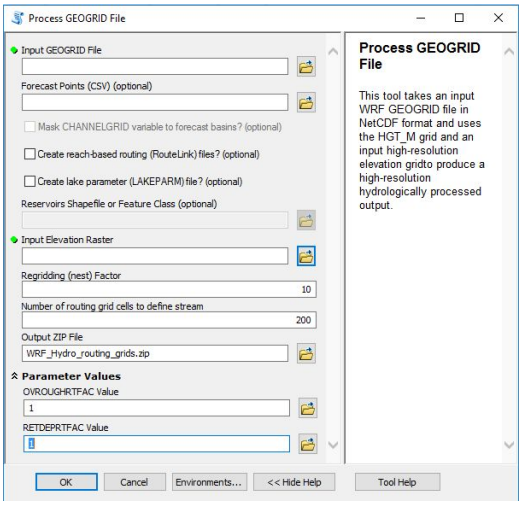


1. Download the WRF-Hydro GIS Pre-processing Toolkit and follow instructions in the documentation for setup.

The WRF-Hydro GIS Pre-processing Toolkit is a set of tools written in Python using ArcGIS python API (arcpy).
This tool will create the data layers for terrestrial overland flow, subsurface flow and channel routing processes required by WRF Hydro. It outputs a 'routing stack', as set of netCDF format higher-resolution grids and other data; the base set of terrain data used to build implementations of WRF-Hydro. The WRF-Hydro GIS Pre-processing Tool utilizes ArcGIS for Desktop version 10.3.1 and Spatial Analyst Extension.

2. Enter a geogrid netCDF file (`geo_em.d0x.nc`), your DEM, and your optional station forecast points and optional lake polygons into the WRF-Hydro GIS Pre-processing Toolkit to create your full domain files, and optional geospatial metadata file, parameter and routing files. See the WRF-Hydro GIS Preprocessing Toolkit documentation for full guidance. Below is an illustration of the WRF-Hydro GIS Preprocessing Toolkit programs and GUI interface.





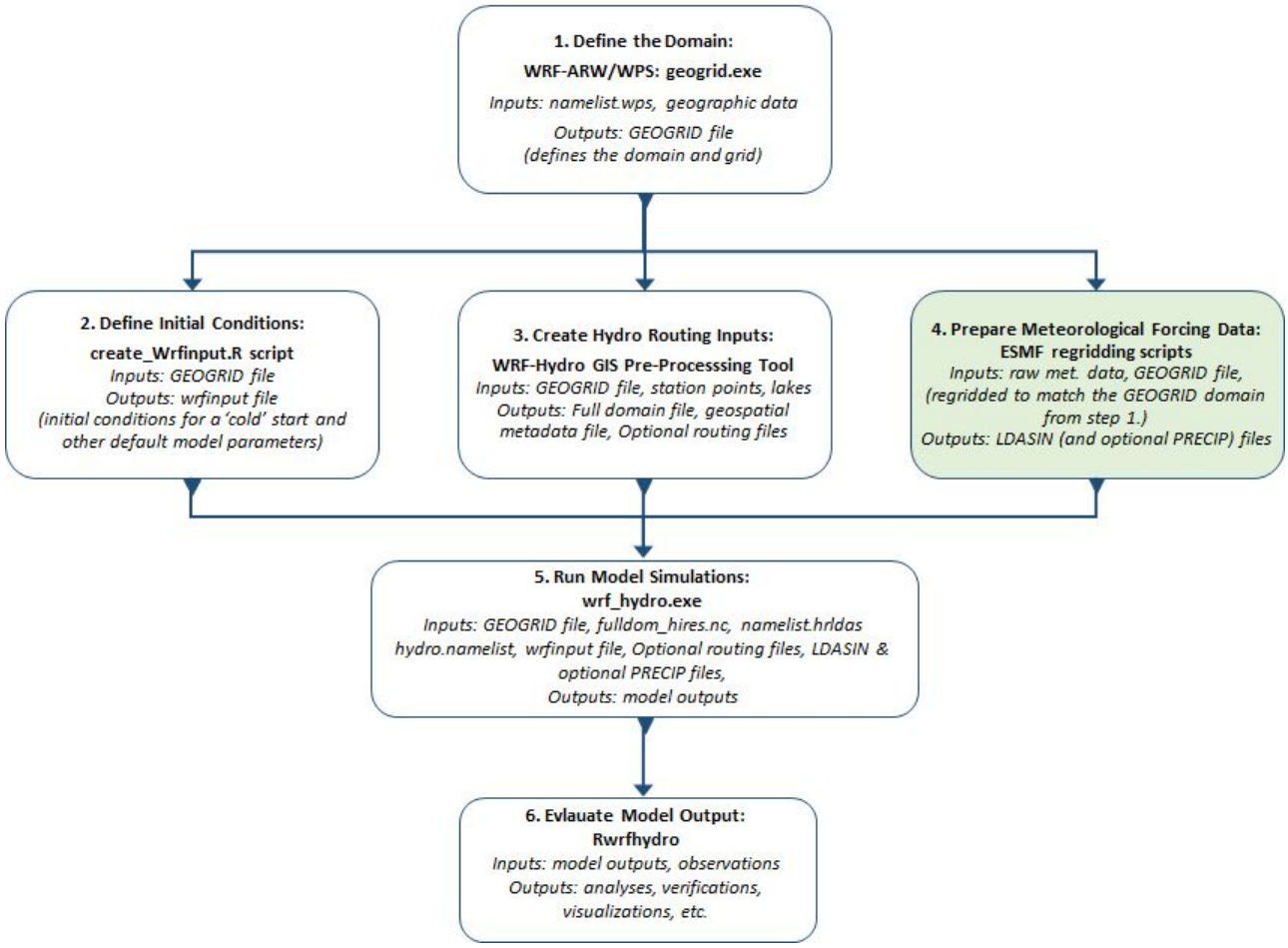
Required Input	
geo_em.d0x.nc	Geogrid file generated from the geogrid.exe program in the namelist.wps from the WPS (step 1 in the workflow)
DEM	User provided digital elevation model/ data in raster format with units in meters
Optional Input	
Forecast points	User provided .csv file that includes FID, latitude, longitude, and station name
Lake polygons	User provided polygon feature class or .shp file
Required Output	
Fulldom_hires.nc	(aka "routing stack") 2d file containing the high-resolution routing base grids (e.g., topography, flow direction, channels) as well as a few routing-specific parameters (LKSATFAC, OVROUGHRTFAC, RETDEPRTFAC)
Optional output	
Route_Link.nc	1d file containing the channel reach parameters (ComID, gage ID, bottom width, slope,roughness, order, etc.) **This file is required for reach based routing and user defined mapping in WRF-Hydro.**
Gw_basins_geogrid.txt Or GWBASINS.nc	an ASCII raster of the 'basn_msk' grid, but regridded to the GEOGRID file resolution. This file is only created if a CSV gage location file is provided. Required for ground water in WRF-Hydro. **Required when the baseflow bucket model is turned on and user defined mapping is off.**
GWBUCKPARM.TBL or GWBUCKPARM.nc	1d file containing the groundwater bucket model parameters for each catchment. **Required when the baseflow bucket model is turned on.**
LAKEPARM.TBL or LAKEPARM.nc	1d file containing the lake model parameters for each catchment. The table will contain a record for each lake feature in the FullDom 'LAKEGRID' variable, and contain derived and default parameters for each lake.R. **Required with lake and reservoir routing is turned on.**

It is important to note that the geogrid file (`geo_em.d0X.nc`), which is created by the WPS Geogrid program, does not contain projection information in a standard format that GIS applications such as ArcGIS can read. If you want to properly view variables in your GEOGRID file with ArcGIS, you can use the "Export grid from GEOGRID file" tool in the Utilities toolset of the WRF-Hydro ArcGIS Pre-processing tools. That tool will properly read and display any of the variables that are on the mass-grid (stagger = "M") in your GEOGRID file.

More information and frequently asked questions regarding the WRF-Hydro GIS Pre-processing Tool can be found in the tool documentation and online at https://ral.ucar.edu/projects/wrf_hydro.

3. Add the output files from the WRF-Hydro GIS Pre-Processing Toolkit to your **DOMAIN** directory.

Gather and regrid forcing data



1. Gather data. It is up to you to choose what type of data you would like to use to force the model. WRF-Hydro accepts hourly or minute time step data. Below are two types of commonly used data sources:
NLDAS – North American Land Data Assimilation System
GFS – Global Forecast System
The required input variables are listed in the table below:

Variable name	Description	Units
SWDOWN	Incoming shortwave radiation	W/m^2
LWDOWN	Incoming longwave radiation	W/m^2
Q2D	Specific humidity	kg/kg
T2D	Air temperature	K
PSFC	Surface pressure	Pa
U2D	Near surface wind in the u-component	m/s
V2D	Near surface wind in the v-component	m/s
RAINRATE	Precipitation rate	mm/s or $kg/m^2/s$

2. Regrid the data to match the WRF-Hydro geogrid domain grid. This can be done in a variety of ways, however you can utilize the ESMF regridding scripts provided from https://ral.ucar.edu/projects/wrf_hydro/regridding-scripts . The ESMF regridding scripts require NCL 6.1 or later.

Each output file should include only one time step. The output needs to be netCDF format with following file naming convention: `YYYYMMDDHH.LDASIN_DOMAINx.nc` where YYYY= Year, MM = Month, DD = Day, HH = Hour, LDASIN_DOMAIN = Input file naming convention for the land surface model, X = Domain number.

Optional precipitation data should be netCDF format with the following file naming convention: `YYYMMDDHHMM.PRECIP_FORCING.nc` where where YYYY= Year, MM = Month, DD = Day, HH = Hour, MM = Minutes, PRECIP_FORCING = Input file naming convention for precipitation data.

3. Add these files to your **FORCING** directory
The forcing type namelist option will be described and chosen in the `namelist.hrldas` file associated with the Land Surface Model (LSM). The `namelist.hrldas` file is provided with the WRF-Hydro model code.

Directories and their contents

At this point you will want to make sure the following directories contain the respective files.

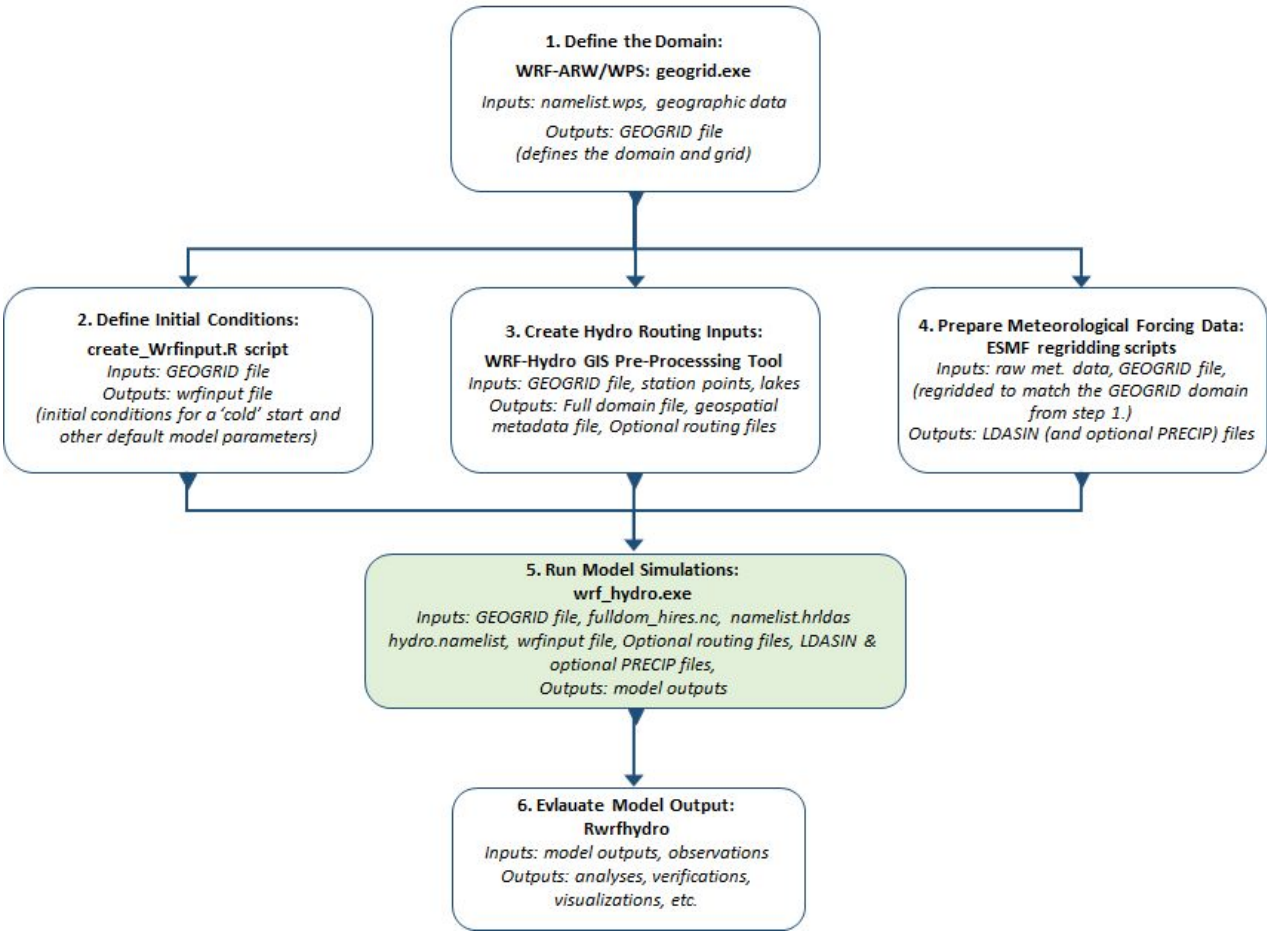
DOMAIN

Filename	Required
geo_em.d0x.nc	Yes
wrfinput_d0x.nc	Yes
Fulldom_hires.nc	Yes
Route_Link.nc	When reach based routing is used (including user defined mapping)
GWBASINS.nc	When the baseflow bucket model is turned on and user defined mapping is off
GWBUCKPARM.nc	When the baseflow bucket model is turned on
LAKEPARM.nc	When lake and reservoir routing is turned on

FORCING

Filename	Required (multiple files)
YYYYMMDDHH.LDASIN_DOMAINx.nc	Yes
YYYMMDDHHMM.PRECIP_FORCING.nc	If running with optional precipitation

Run WRF-Hydro with your customized geographic inputs, idealized forcing, and the land surface model (LSM) only



The idealized forcing type namelist option uses a preset diurnal cycle of temperature, radiation, etc. along with a single uniformly distributed rainfall event for the first model timestep with a rate of 1 inch/ hour.This will drop an inch of precipitation evenly across the landscape in the first time step and nothing after that. Running WRF-Hydro with idealized forcing is an easy way to test your model to see if the water is being routed properly and that your configuration is stable. More information is available in the *WRF-Hydro V5 Technical Description*. In this step you will also configure your namelists to run the land surface model only. It is important to test the model with this configuration before adding in physics and routing options. This step will indicate that the model runs with your customized geographic input and that the land surface model is running for your region of interest.

Run: Simulation setup and execution

The steps outlined below are offered as practical guidance on setting up and organizing a WRF-Hydro implementation.

1. ***Ensure a proper Run/ directory is setup:*** You should have already done this in the previous steps if you were following this guide. However, if not, information is repeated here.
   On code compile, a `wrf_hydro_nwm/trunk/NDHMS/Run` directory will be created that includes a model executable file (`wrf_hydro.exe`), example namelists (`namelist.hrldas`, `hydro.namelist`), and example parameter files (e.g., `SOILPARM.TBL, MPTABLE.TBL, GENPARM.TBL, HYDRO.TBL, CHANPARM.TBL` - the list will vary depending on the LSM selected). For descriptions of the parameter tables see the *WRF-Hydro V5 Technical Description*. You can use this directory as your model Run directory or copy to a new location.

2. ***Double check that your DOMAIN directory has all the needed input files.***

3. ***Edit the namelists.*** In the `Run` directory, open and edit `namelist.hrldas` and `hydro.namelist` to set up a simulation to your specifications. See the *WRF-Hydro V5 Technical Description* for detailed information of the model physics options, Appendix 5 for a description of the `hydro.namelist`. See Appendix 4 for a description of the Noah-MP `namelist.hrldas`.

   Below we have included some annotated sample namelists with the configuration that you would want to set to run with idealized forcing (FORCTYP = 4) and the LSM only. Note: The namelists below relate to the Noah-MP land surface model.

WRF-Hydro namelist examples for running in standalone mode with the LSM only

Annotations are indicated by <BLUE TEXT>

hydro.namelist

```
&HYDRO_nlist
!!!! ---------------------- SYSTEM COUPLING ---------------------- !!!!

! Specify what is being coupled:  1=HRLDAS (offline Noah-LSM), 2=WRF, 3=NASA/LIS, 4=CLM
sys_cpl = 1

!!!! ------------------- MODEL INPUT DATA FILES ------------------- !!!!

! Specify land surface model gridded input data file (e.g.: "geo_em.d01.nc")
GEO_STATIC_FLNM = "./DOMAIN/<YOUR GEOGRID FILE>"

! Specify the high-resolution routing terrain input data file (e.g.: "Fulldom_hires.nc")
GEO_FINEGRID_FLNM = "./DOMAIN/<YOUR FULLDOMAIN FILE>"

! Specify the spatial hydro parameters file (e.g.: "hydro2dtbl.nc")
! If you specify a filename and the file does not exist, it will be created for you.
HYDROTBL_F = "./DOMAIN/hydro2dtbl.nc"

! Specify spatial metadata file for land surface grid. (e.g.: "GEOGRID_LDASOUT_Spatial_Metadata.nc")
LAND_SPATIAL_META_FLNM = "./DOMAIN/<YOUR GEOSPATIAL METADATA FILE>"

! Specify the name of the restart file if starting from restart...comment out with '!' if not...
!RESTART_FILE  = 'HYDRO_RST.2013-09-12_04:00_DOMAIN3' <COMMENT THIS OUT>

!!!! --------------------- MODEL SETUP OPTIONS --------------------- !!!!

! Specify the domain or nest number identifier...(integer)
IGRID = <YOUR DOMAIN ID FROM THE GEOGRID FILE>

! Specify the restart file write frequency...(minutes)
! A value of -9999 will output restarts on the first day of the month only.
rst_dt = -9999 <SET TO -9999 FOR MONTHLY>

! Reset the LSM soil states from the high-res routing restart file (1=overwrite, 0=no overwrite)
```

```
! NOTE: Only turn this option on if overland or subsurface routing is active!
rst_typ = 0 <SET TO 0 FOR LSM ONLY>

! Restart file format control
rst_bi_in = 0       !0: use netcdf input restart file (default)
                    !1: use parallel io for reading multiple restart files, 1 per core
rst_bi_out = 0      !0: use netcdf output restart file (default)
                    !1: use parallel io for outputting multiple restart files, 1 per core

! Restart switch to set restart accumulation variables to 0 (0=no reset, 1=yes reset to 0.0)
RSTRT_SWC = 0

! Specify baseflow/bucket model initialization...(0=cold start from table, 1=restart file)
GW_RESTART = 0 <SET TO 0 SO THE MODEL DOES NOT LOOK FOR A RESTART FILE>

!!!! ---------- MODEL OUTPUT CONTROL ---------- !!!!

! Specify the output file write frequency...(minutes)
out_dt = 60 <SET TO HOURLY>

! Specify the number of output times to be contained within each output history file...(integer)
!   SET = 1 WHEN RUNNING CHANNEL ROUTING ONLY/CALIBRATION SIMS!!!
!   SET = 1 WHEN RUNNING COUPLED TO WRF!!!
SPLIT_OUTPUT_COUNT = 1

! Specify the minimum stream order to output to netcdf point file...(integer)
! Note: lower value of stream order produces more output.
order_to_write = 1

! Flag to turn on/off new I/O routines: 1 = with scale/offset/compression,
! 2 = with scale/offset/NO compression, 3 = compression only, 4 = no scale/offset/compression (default)
io_form_outputs = 4 <SET TO DEFAULT FOR SIMPLICITY>

! Realtime run configuration option:
! 0=all (default), 1=analysis, 2=short-range, 3=medium-range, 4=long-range, 5=retrospective,
! 6=diagnostic (includes all of 1-4 outputs combined)
io_config_outputs = 0 <SET TO DEFAULT FOR FULL OUTPUTS>

! Option to write output files at time 0 (restart cold start time): 0=no, 1=yes (default)
t0OutputFlag = 1

! Options to output channel & bucket influxes. Only active for UDMP_OPT=1.
! Nonzero choice requires that out_dt above matches NOAH_TIMESTEP in namelist.hrldas.
! 0=None (default), 1=channel influxes (qSfcLatRunoff, qBucket)
! 2=channel+bucket fluxes    (qSfcLatRunoff, qBucket, qBtmVertRunoff_toBucket)
! 3=channel accumulations    (accSfcLatRunoff, accBucket) *** NOT TESTED ***
output_channelBucket_influx = 0

! Output netcdf file control <TURN OFF ALL HYDRO OUTPUTS>
CHRTOUT_DOMAIN = 0          ! Netcdf point timeseries output at all channel points (1d)
                        !     0 = no output, 1 = output
```

```
CHANOBS_DOMAIN = 0          ! Netcdf point timeseries at forecast points or gage points (defined in
Routelink)
                        !     0 = no output, 1 = output at forecast points or gage points.
CHRTOUT_GRID = 0            ! Netcdf grid of channel streamflow values (2d)
                        !     0 = no output, 1 = output
                        !     NOTE: Not available with reach-based routing
LSMOUT_DOMAIN = 0           ! Netcdf grid of variables passed between LSM and routing components (2d)
                        !     0 = no output, 1 = output
                        !     NOTE: No scale_factor/add_offset available
RTOUT_DOMAIN = 0            ! Netcdf grid of terrain routing variables on routing grid (2d)
                        !     0 = no output, 1 = output
output_gw = 0               ! Netcdf GW output
                        !     0 = no output, 1 = output
outlake  = 0                ! Netcdf grid of lake values (1d)
                        !     0 = no output, 1 = output
frxst_pts_out = 0           ! ASCII text file of forecast points or gage points (defined in Routelink)
                        !     0 = no output, 1 = output

!!!! ---------- PHYSICS OPTIONS AND RELATED SETTINGS ---------- !!!!

! Specify the number of soil layers (integer) and the depth of the bottom of each layer... (meters)
! Notes: In Version 1 of WRF-Hydro these must be the same as in the namelist.input file.
!      Future versions will permit this to be different.
NSOIL=4
ZSOIL8(1) = -0.10
ZSOIL8(2) = -0.40
ZSOIL8(3) = -1.00
ZSOIL8(4) = -2.00

! Specify the grid spacing of the terrain routing grid...(meters)
DXRT = <YOUR GRID SPACING>

! Specify the integer multiple between the land model grid and the terrain routing grid...(integer)
AGGFACTRT = <MULTIPLE BETWEEN LAND AND TERRAIN GRIDS>

! Specify the channel routing model timestep...(seconds)
DTRT_CH = 10

! Specify the terrain routing model timestep...(seconds)
DTRT_TER = 10

! Switch to activate subsurface routing...(0=no, 1=yes)
SUBRTSWCRT = 0 <TURN OFF SUBSURFACE ROUTING>

! Switch to activate surface overland flow routing...(0=no, 1=yes)
OVRTSWCRT = 0 <TURN OFF SURFACE ROUTING>

! Specify overland flow routing option: 1=Seepest Descent (D8) 2=CASC2D (not active)
! NOTE: Currently subsurface flow is only steepest descent
rt_option = 1

! Switch to activate channel routing...(0=no, 1=yes)
```

```
CHANRTSWCRT = 0 <TURN OFF CHANNEL ROUTING>

! Specify channel routing option: 1=Muskingam-reach, 2=Musk.-Cunge-reach, 3=Diff.Wave-gridded
channel_option = 3

! Specify the reach file for reach-based routing options (e.g.: "Route_Link.nc")
!route_link_f = "./DOMAIN/Route_Link.nc" <COMMENT OUT ROUTELINK>

! Specify the lake parameter file (e.g.: "LAKEPARM.nc" for netcdf or "LAKEPARM.TBL" for text).
! Note REQUIRED if lakes are on.
!route_lake_f = "./DOMAIN/LAKEPARM.nc" <COMMENT OUT LAKEPARM>

! Switch to activate baseflow bucket model...(0=none, 1=exp. bucket, 2=pass-through)
GWBASESWCRT = 0 <TURN OFF GROUNDWATER>

! Groundwater/baseflow 2d mask specified on land surface model grid (e.g.: "GWBASINS.nc" for netcdf
! or "gw_basns.txt" for ascii). Note: Only required if baseflow  model is active (1 or 2) and UDMP_OPT=0.
gwbasmskfil = "./DOMAIN/GWBASINS.nc" <COMMENT OUT GWBASINS>

! Groundwater bucket parameter file (e.g.: "GWBUCKPARM.nc" for netcdf or "GWBUCKPARM.TBL" for text)
GWBUCKPARM_file = "./DOMAIN/GWBUCKPARM.nc" <COMMENT OUT GWBUCKPARM>

! User defined mapping, such NHDPlus: 0=no (default), 1=yes
UDMP_OPT = 0 <TURN OFF UDMP>

! If on, specify the user-defined mapping file (e.g.: "spatialweights.nc")
!udmap_file = "./DOMAIN/spatialweights.nc" <COMMENT OUT SPATIAL WEIGHTS>
/
```

namelist.hrldas

```
&NOAHLSM_OFFLINE

 HRLDAS_SETUP_FILE = "./DOMAIN/<YOUR WRFINPUT FILE>"
 INDIR = "./FORCING"
 !SPATIAL_FILENAME = "./DOMAIN/soil_properties.nc" <COMMENT OUT SPATIAL PARAMS>
 OUTDIR = "./"

 START_YEAR  = 2013
 START_MONTH = 09
 START_DAY   = 12
 START_HOUR  = 04
 START_MIN   = 00

 !RESTART_FILENAME_REQUESTED = "RESTART.2013091204_DOMAIN1" <COMMENT OUT
RESTART>

 ! Specification of simulation length in days OR hours <SET RUN LENGTH>
 KDAY = 1
 ! KHOUR = 8

 ! Physics options (see the documentation for details)
```

```
 DYNAMIC_VEG_OPTION                = 4
 CANOPY_STOMATAL_RESISTANCE_OPTION = 1
 BTR_OPTION                        = 1
 RUNOFF_OPTION                     = 3
 SURFACE_DRAG_OPTION               = 1
 FROZEN_SOIL_OPTION                = 1
 SUPERCOOLED_WATER_OPTION          = 1
 RADIATIVE_TRANSFER_OPTION         = 3
 SNOW_ALBEDO_OPTION                = 2
 PCP_PARTITION_OPTION              = 1
 TBOT_OPTION                       = 2
 TEMP_TIME_SCHEME_OPTION           = 3
 GLACIER_OPTION                    = 2
 SURFACE_RESISTANCE_OPTION         = 4

 ! Timesteps in units of seconds
 FORCING_TIMESTEP = 3600  <SET TO HOURLY>
 NOAH_TIMESTEP    = 3600     <SET TO HOURLY>
 OUTPUT_TIMESTEP  = 3600   <SET TO HOURLY>

 ! Land surface model restart file write frequency
 RESTART_FREQUENCY_HOURS = -9999 <SET TO MONTHLY>

 ! Split output after split_output_count output times.
 SPLIT_OUTPUT_COUNT = 1

 ! Soil layer specification
 NSOIL=4
 soil_thick_input(1) = 0.10
 soil_thick_input(2) = 0.30
 soil_thick_input(3) = 0.60
 soil_thick_input(4) = 1.00

 ! Forcing data measurement height for winds, temp, humidity
 ZLVL = 10.0

 ! Restart file format options
 rst_bi_in = 0      !0: use netcdf input restart file
                    !1: use parallel io for reading multiple restart files (1 per core)
 rst_bi_out = 0     !0: use netcdf output restart file
                    !1: use parallel io for outputting multiple restart files (1 per core)
/

&WRF_HYDRO_OFFLINE

 ! Specification of forcing data:  1=HRLDAS-hr format, 2=HRLDAS-min format, 3=WRF,
 !                                 4=Idealized, 5=Ideal w/ spec. precip,
 !                                 6=HRLDAS-hr format w/ spec. precip,
 !                                 7=WRF w/ spec. precip
 FORC_TYP = 4 <SET TO IDEALIZED>
/
```

4. Run the WRF-Hydro executable `wrf_hydro.exe` with idealized forcing and the LSM only.

> ***Execute the model.*** For parallel runs the command may differ according the specifications of the parallel-processing software on individual machines but a common execution command may look like `mpiexec -n # wrf_hydro.exe`, where **#** is the number of processors to be used.

5. Check that the model ran and that it produced output files.
If run successfully, output files as specified in the namelists will be generated as a series of netCDF files (and optional ASCII files) with associated time and date information in the filenames. See section 6 of the *WRF-Hydro V5 Technical Description* and the supplementary document, *WRF-Hydro V5 Model Output Variables* (https://ral.ucar.edu/projects/wrf_hydro) for information and descriptions of the output files and variables.

6. Create a new directory for your output files and move your output files to that directory. Clean out the directory for the next run.

Run WRF-Hydro again with custom geographic inputs & custom forcing data with the land surface model only

In this step you will change the forcing type namelist option to match your input forcing data type and run the model again with only the land surface model. This will indicated that the model is working for your customized geographic input, the land surface model is running for your region of interest and that the model is accepting your customized forcing data.

Follow the instructions for *'Run: Simulation set up and execution'* from the previous sections, but this time in the namelists change the forcing type option to match your forcing data and change the forcing timestep to match your forcing data.

Run WRF-Hydro with custom geographic inputs, custom forcing data, & minimal routing physics

In this step you will invoke or 'turn on' the `hydro.namelist` and sequentially add routing components to test out physics options and routing configurations. This will indicated that the model is working for your customized geographic input, the land surface model is running for your region of interest, the model is accepting your customized forcing data, and physics/routing options are working. Be sure to activate only those options for which you have the required data. For information on physics and routing options refer to the *WRF-Hydro V5 Technical Description*. **Note**: The routing timestep 'DTRT' must be set in accordance with the routing grid spacing in order to satisfy Courant constraints (see the *WRF-Hydro V5 Technical Description* section 3.5 for a discussion on Courant constraints).

Follow the instructions for *'Run: Simulation set up and execution'* from the previous sections, but this time in the namelists change the forcing type option to match your forcing data and change the forcing timestep to match your forcing data. Begin with setting a small number of namelist options and date settings etc. Run WRF-Hydro turning on the following options one at a time.
1. Sfc/subsfc
2. GW/baseflow
3. Channel flow
4. Reservoirs

Once you finalize your configuration you can then move on to a full model simulation.
For reference below we have included sample annotated nameslists with physics options added.

WRF-Hydro namelist examples for full physics

Annotations are indicated by <BLUE TEXT>

hydro.namelist

```
&HYDRO_nlist
!!!! ---------------------- SYSTEM COUPLING ---------------------- !!!!

! Specify what is being coupled:  1=HRLDAS (offline Noah-LSM), 2=WRF, 3=NASA/LIS, 4=CLM
sys_cpl = 1

!!!! ------------------- MODEL INPUT DATA FILES ------------------- !!!!

! Specify land surface model gridded input data file (e.g.: "geo_em.d01.nc")
GEO_STATIC_FLNM = "./DOMAIN/<YOUR GEOGRID FILE>"

! Specify the high-resolution routing terrain input data file (e.g.: "Fulldom_hires.nc")
GEO_FINEGRID_FLNM = "./DOMAIN/<YOUR FULLDOMAIN FILE>"

! Specify the spatial hydro parameters file (e.g.: "hydro2dtbl.nc")
! If you specify a filename and the file does not exist, it will be created for you.
HYDROTBL_F = "./DOMAIN/hydro2dtbl.nc"

! Specify spatial metadata file for land surface grid. (e.g.: "GEOGRID_LDASOUT_Spatial_Metadata.nc")
LAND_SPATIAL_META_FLNM = "./DOMAIN/<YOUR GEOSPATIAL METADATA FILE>"

! Specify the name of the restart file if starting from restart...comment out with '!' if not...
!RESTART_FILE  = 'HYDRO_RST.2013-09-12_04:00_DOMAIN3' <COMMENT THIS OUT>

!!!! ---------------------- MODEL SETUP OPTIONS ---------------------- !!!!

! Specify the domain or nest number identifier...(integer)
IGRID = <YOUR DOMAIN ID FROM THE GEOGRID FILE>

! Specify the restart file write frequency...(minutes)
! A value of -9999 will output restarts on the first day of the month only.
rst_dt = -9999 <SET TO -9999 FOR MONTHLY>

! Reset the LSM soil states from the high-res routing restart file (1=overwrite, 0=no overwrite)
! NOTE: Only turn this option on if overland or subsurface routing is active!
rst_typ = 1 <SET TO 1 WHEN TERRAIN ROUTING IS ON>

! Restart file format control
rst_bi_in = 0       !0: use netcdf input restart file (default)
                    !1: use parallel io for reading multiple restart files, 1 per core
rst_bi_out = 0      !0: use netcdf output restart file (default)
                    !1: use parallel io for outputting multiple restart files, 1 per core
```

```
! Restart switch to set restart accumulation variables to 0 (0=no reset, 1=yes reset to 0.0)
RSTRT_SWC = 0

! Specify baseflow/bucket model initialization...(0=cold start from table, 1=restart file)
GW_RESTART = 0 <SET TO 0 SO THE MODEL DOES NOT LOOK FOR A RESTART FILE>

!!!! ---------- MODEL OUTPUT CONTROL ---------- !!!!

! Specify the output file write frequency...(minutes)
out_dt = 60 <SET TO HOURLY>

! Specify the number of output times to be contained within each output history file...(integer)
!   SET = 1 WHEN RUNNING CHANNEL ROUTING ONLY/CALIBRATION SIMS!!!
!   SET = 1 WHEN RUNNING COUPLED TO WRF!!!
SPLIT_OUTPUT_COUNT = 1

! Specify the minimum stream order to output to netcdf point file...(integer)
! Note: lower value of stream order produces more output.
order_to_write = 1 <ADJUST AS DESIRED>

! Flag to turn on/off new I/O routines: 1 = with scale/offset/compression,
! 2 = with scale/offset/NO compression, 3 = compression only, 4 = no scale/offset/compression (default)
io_form_outputs = 4 <SET TO DEFAULT FOR SIMPLICITY>

! Realtime run configuration option:
! 0=all (default), 1=analysis, 2=short-range, 3=medium-range, 4=long-range, 5=retrospective,
! 6=diagnostic (includes all of 1-4 outputs combined)
io_config_outputs = 0 <SET TO DEFAULT FOR FULL OUTPUTS>

! Option to write output files at time 0 (restart cold start time): 0=no, 1=yes (default)
t0OutputFlag = 1

! Options to output channel & bucket influxes. Only active for UDMP_OPT=1.
! Nonzero choice requires that out_dt above matches NOAH_TIMESTEP in namelist.hrldas.
! 0=None (default), 1=channel influxes (qSfcLatRunoff, qBucket)
! 2=channel+bucket fluxes    (qSfcLatRunoff, qBucket, qBtmVertRunoff_toBucket)
! 3=channel accumulations    (accSfcLatRunoff, accBucket) *** NOT TESTED ***
output_channelBucket_influx = 0

! Output netcdf file control <TURN ON RELEVANT OUTPUTS AS YOU TEST THOSE OPTIONS - REFER TO
COMMENTS AND TECH DESCRIPTION FOR MORE INFO>
CHRTOUT_DOMAIN = 0          ! Netcdf point timeseries output at all channel points (1d)
                        !     0 = no output, 1 = output
CHANOBS_DOMAIN = 1          ! Netcdf point timeseries at forecast points or gage points (defined in Routelink)
                        !     0 = no output, 1 = output at forecast points or gage points.
CHRTOUT_GRID = 1            ! Netcdf grid of channel streamflow values (2d)
                        !     0 = no output, 1 = output
                        !     NOTE: Not available with reach-based routing
LSMOUT_DOMAIN = 1           ! Netcdf grid of variables passed between LSM and routing components (2d)
                        !     0 = no output, 1 = output
                        !     NOTE: No scale_factor/add_offset available
RTOUT_DOMAIN = 1            ! Netcdf grid of terrain routing variables on routing grid (2d)
```

```
                    !     0 = no output, 1 = output
output_gw = 1             ! Netcdf GW output
                    !     0 = no output, 1 = output
outlake  = 1              ! Netcdf grid of lake values (1d)
                    !     0 = no output, 1 = output
frxst_pts_out = 0         ! ASCII text file of forecast points or gage points (defined in Routelink)
                    !     0 = no output, 1 = output

!!!! ---------- PHYSICS OPTIONS AND RELATED SETTINGS ---------- !!!!

! Specify the number of soil layers (integer) and the depth of the bottom of each layer... (meters)
! Notes: In Version 1 of WRF-Hydro these must be the same as in the namelist.input file.
!      Future versions will permit this to be different.
NSOIL=4
ZSOIL8(1) = -0.10
ZSOIL8(2) = -0.40
ZSOIL8(3) = -1.00
ZSOIL8(4) = -2.00

! Specify the grid spacing of the terrain routing grid...(meters)
DXRT = <YOUR GRID SPACING>

! Specify the integer multiple between the land model grid and the terrain routing grid...(integer)
AGGFACTRT = <MULTIPLE BETWEEN LAND AND TERRAIN GRIDS>

! Specify the channel routing model timestep...(seconds)
DTRT_CH = 10 <UPDATE WHEN YOU TURN ON CHANNEL ROUTING>

! Specify the terrain routing model timestep...(seconds)
DTRT_TER = 10 <UPDATE WHEN YOU TURN ON TERRAIN ROUTING>

! Switch to activate subsurface routing...(0=no, 1=yes)
SUBRTSWCRT = 1 <SET TO 1 TO TURN ON SUBSURFACE ROUTING>

! Switch to activate surface overland flow routing...(0=no, 1=yes)
OVRTSWCRT = 1 <SET TO 1 TO TURN ON SURFACE ROUTING>

! Specify overland flow routing option: 1=Seepest Descent (D8) 2=CASC2D (not active)
! NOTE: Currently subsurface flow is only steepest descent
rt_option = 1

! Switch to activate channel routing...(0=no, 1=yes)
CHANRTSWCRT = 1 <SET TO 1 TO TURN ON CHANNEL ROUTING>

! Specify channel routing option: 1=Muskingam-reach, 2=Musk.-Cunge-reach, 3=Diff.Wave-gridded
channel_option = 3 <SET TO THE APPROPRIATE CHANNEL ROUTING OPTION>

! Specify the reach file for reach-based routing options (e.g.: "Route_Link.nc")
!route_link_f = "./DOMAIN/Route_Link.nc" <UNCOMMENT AND UPDATE IF USING REACH-BASED ROUTING ONLY>

! Specify the lake parameter file (e.g.: "LAKEPARM.nc" for netcdf or "LAKEPARM.TBL" for text).
! Note REQUIRED if lakes are on.
```

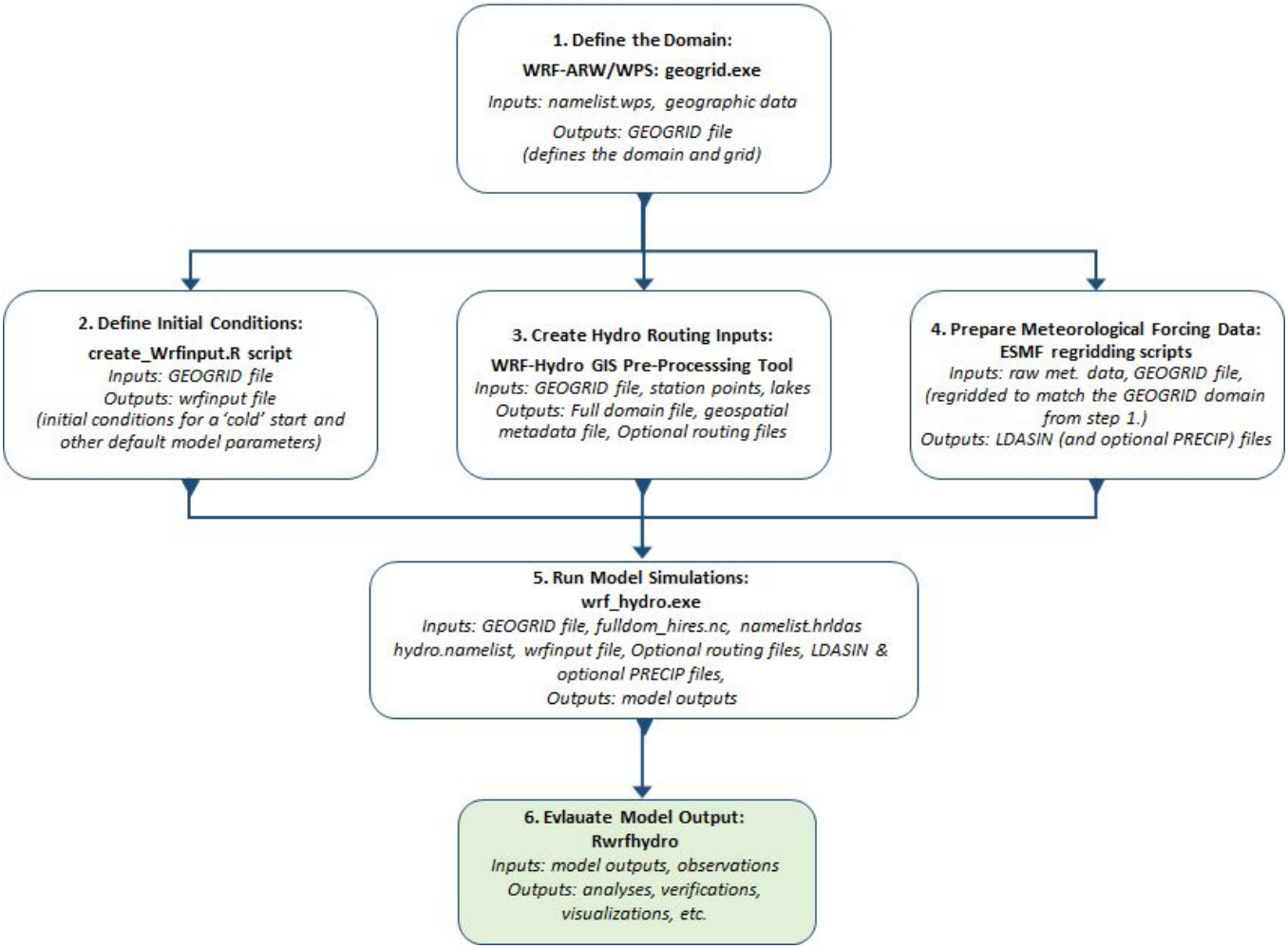
```
route_lake_f = "./DOMAIN/LAKEPARM.nc" <UNCOMMENT AND UPDATE FOR LAKES>

! Switch to activate baseflow bucket model...(0=none, 1=exp. bucket, 2=pass-through)
GWBASESWCRT = 1 <SET TO 1 FOR GROUNDWATER>

! Groundwater/baseflow 2d mask specified on land surface model grid (e.g.: "GWBASINS.nc" for netcdf
! or "gw_basns.txt" for ascii). Note: Only required if baseflow  model is active (1 or 2) and UDMP_OPT=0.
gwbasmskfil = "./DOMAIN/GWBASINS.nc" <UNCOMMENT AND UPDATE FOR GROUNDWATER>

! Groundwater bucket parameter file (e.g.: "GWBUCKPARM.nc" for netcdf or "GWBUCKPARM.TBL" for text)
GWBUCKPARM_file = "./DOMAIN/GWBUCKPARM.nc" <UNCOMMENT AND UPDATE FOR GROUNDWATER>

! User defined mapping, such NHDPlus: 0=no (default), 1=yes
UDMP_OPT = 0

! If on, specify the user-defined mapping file (e.g.: "spatialweights.nc")
!udmap_file = "./DOMAIN/spatialweights.nc"
/
```

namelist.hrldas

```
&NOAHLSM_OFFLINE

 HRLDAS_SETUP_FILE = "./DOMAIN/<YOUR WRFINPUT FILE>"
 INDIR = "./FORCING"
 !SPATIAL_FILENAME = "./DOMAIN/soil_properties.nc" <COMMENT OUT SPATIAL PARAMS>
 OUTDIR = "./"

 START_YEAR  = 2013
 START_MONTH = 09
 START_DAY   = 12
 START_HOUR  = 04
 START_MIN   = 00

 !RESTART_FILENAME_REQUESTED = "RESTART.2013091204_DOMAIN1" <COMMENT OUT RESTART>

 ! Specification of simulation length in days OR hours <SET RUN LENGTH>
 KDAY = 1
 ! KHOUR = 8

 ! Physics options (see the documentation for details) <UPDATE AS DESIRED>
 DYNAMIC_VEG_OPTION                = 4
 CANOPY_STOMATAL_RESISTANCE_OPTION = 1
 BTR_OPTION                        = 1
 RUNOFF_OPTION                     = 3
 SURFACE_DRAG_OPTION               = 1
 FROZEN_SOIL_OPTION                = 1
 SUPERCOOLED_WATER_OPTION          = 1
 RADIATIVE_TRANSFER_OPTION         = 3
 SNOW_ALBEDO_OPTION                = 2
 PCP_PARTITION_OPTION              = 1
 TBOT_OPTION                       = 2
```

```
TEMP_TIME_SCHEME_OPTION          = 3
GLACIER_OPTION                   = 2
SURFACE_RESISTANCE_OPTION        = 4

! Timesteps in units of seconds
FORCING_TIMESTEP = 3600 <UPDATE TO YOUR FORCING DATA TIMESTEP>
NOAH_TIMESTEP    = 3600 <UPDATE TO YOUR DESIRED TIMESTEP>
OUTPUT_TIMESTEP  = 3600 <UPDATE TO YOUR DESIRED TIMESTEP>

! Land surface model restart file write frequency
RESTART_FREQUENCY_HOURS = -9999 <SET TO MONTHLY>

! Split output after split_output_count output times.
SPLIT_OUTPUT_COUNT = 1

! Soil layer specification
NSOIL=4
soil_thick_input(1) = 0.10
soil_thick_input(2) = 0.30
soil_thick_input(3) = 0.60
soil_thick_input(4) = 1.00

! Forcing data measurement height for winds, temp, humidity
ZLVL = 10.0

! Restart file format options
rst_bi_in = 0      !0: use netcdf input restart file
                   !1: use parallel io for reading multiple restart files (1 per core)
rst_bi_out = 0     !0: use netcdf output restart file
                   !1: use parallel io for outputting multiple restart files (1 per core)
/

&WRF_HYDRO_OFFLINE

! Specification of forcing data:  1=HRLDAS-hr format, 2=HRLDAS-min format, 3=WRF,
!                                 4=Idealized, 5=Ideal w/ spec. precip,
!                                 6=HRLDAS-hr format w/ spec. precip,
!                                 7=WRF w/ spec. precip
 FORC_TYP = <SET TO THE TYPE THAT RELATES TO YOUR FORCING DATA>
/
```

Evaluation



There are many ways to look at and evaluate model output. We provide Rwrfhydro a community-contributed toolbox for managing, analyzing, and visualizing WRF-Hydro input and output files in R. It can be accessed from https://github.com/NCAR/rwrfhydro. There are a number of vignettes and examples that walk through different evaluation scenarios.

Considerations for full model simulation

Now that the model has run successfully and you have defined the namelist configuration for your geographical location and research question you can begin to run full model simulations.

Things to consider when using the model for real scientific full model simulations:

- Input data sources (e.g., land cover, soil type, channel network)
- Alternative atmospheric forcing data sources
- Choose physics and routing options appropriate for your region and science questions
- Spin-up (how long to spin-up, repeat forcings, etc.)
    - Your model spin up time is dependent on your research question, your geographical region, and namelist configurations.
- Parameter estimation and calibration