

Community Climate System Model

National Center for Atmospheric Research, Boulder, CO

<http://www.cesm.ucar.edu/models>

CVS tag \$Name\$ Build date: July 12, 2004

**Community Sea Ice Model (CSIM) User's Guide
Version 5.0**

Released with CCSM3.0

Julie Schramm
Cecilia Bitz
Bruce Briegleb
Marika Holland
Elizabeth Hunke
Bill Lipscomb
Dick Moritz

Contents

1	Introduction	3
1.1	What's new in CSIM5?	3
2	Quick Start Guide	4
2.1	What is needed to run CSIM?	4
2.2	Downloading Source Code and Input Datasets	5
2.3	Running CSIM Coupled	5
2.4	Running CSIM Uncoupled	6
2.4.1	Multiple Processors with MPI	7
2.4.2	Single Processor with MPI	7
2.4.3	Single Processor without MPI	7
3	The CSIM Scripts	7
3.1	Coupled Model Scripts	8
3.2	Uncoupled Run Script	8
3.2.1	Using the Ocean Mixed Layer Model within CSIM	8
3.2.2	Changing Grid Resolution	8
3.2.3	Changing the Number of Processors	9
3.3	Uncoupled Setup Script	9
3.4	The Build Environment	11
3.4.1	CSIM Preprocessor Flags	11
3.4.2	CSIM Compiler Options	12
4	Namelist Variables	12
4.1	Changing the Length of the Model Run	12
4.2	Changing the timestep	13
4.3	Writing Output	13
4.4	Model Physics	13
4.5	File Names	16
4.6	Ocean Mixed Layer Model	17
4.7	Atmospheric Forcing	18
4.8	Example Namelists	18
4.8.1	Example 1: CCSM Fully Coupled	18
4.8.2	Example 2: Coupled Ice Only Model	19
4.8.3	Example 3: Uncoupled Ice Model	20
4.8.4	Example 4: Uncoupled Ice Model with Atmospheric Forcing	20
4.8.5	Example 5: History File Namelist	21
5	Model Input Datasets	21
5.1	Atmospheric Forcing	22
6	Run Types	22
6.1	Startup Runs	22
6.2	Continue Runs	23
7	Changing the Number of Ice Thickness Categories	23
8	Output Data	24
8.1	Stdout Output	24
8.2	Restart Files	24
8.3	History Files	25
8.3.1	Caveats Regarding Averaged Fields	25
8.3.2	Changing Frequency and Averaging	26

8.3.3	Changing Content	26
9	Troubleshooting	29
9.1	Code does not Compile or Run	29
9.2	Negative Ice Area in Horizontal Remapping	29
9.3	Conservation Error	30
9.4	MPDATA transport unstable	30
9.5	NX does not divide evenly into grid	30
9.6	Enabling the Debugger	30
9.7	Unit 50 Error	31

1 Introduction

This User's Guide accompanies the CCSM3 User's Guide, and is intended for those who would like to run CSIM coupled or uncoupled, on a supported platform, and "out of the box". Users running CSIM fully coupled should first look at the CCSM3 User's Guide. It includes a quick start guide for downloading the CCSM3 source code and input datasets, and information on how to configure, build and run the model. The supported configurations and scripts for building the fully coupled model are also described in the CCSM3 User's Guide. The CSIM User's Guide is intended for users interested in making modifications to the ice model scripts or namelists or running the uncoupled ice model. Users interested in modifying the source code should see the CSIM Code Reference/ Developer's Guide.

CSIM5 is the latest version of the NCAR Community Sea Ice Model. It is the result of a community effort to develop a portable, efficient sea ice model that can be run coupled in a global climate model or uncoupled as a stand-alone ice model. It has been released as the sea ice component of the Community Climate System Model (CCSM), a fully-coupled global climate model that provides simulations of the earth's past, present and future climate states. CSIM5 is supported on high- and low-resolution Greenland Pole grids, which are identical to those used by the Parallel Ocean Program (POP) ocean model. The high resolution version is best suited for simulating present-day and future climate scenarios while the low resolution option is used for paleoclimate simulations and debugging.

An uncoupled version of CSIM is now available. It provides a means of running the sea ice model independent of the other CCSM components. It reads in atmospheric and ocean forcing, which eliminates the need for the flux coupler, and the atmosphere, land and ocean data models. It can be run on a reduced number of processors, or without MPI (Message Passing Interface) for researchers without access to these computer resources.

The physics in the uncoupled ice model are identical to those in the ice model used in the fully coupled system. CSIM is a dynamic-thermodynamic model that includes a subgrid-scale ice thickness distribution (Bitz et al. (2001); Lipscomb (2001)). It uses the energy conserving thermodynamics of Bitz and Lipscomb (1999), has multiple layers in each thickness category, and accounts for the influences of brine pockets within the ice cover. The ice dynamics utilizes the elastic-viscous-plastic (EVP) rheology of Hunke and Dukowicz (1997). Sea ice ridging follows Rothrock (1975) and Thorndike et al. (1975). A slab ocean mixed layer model is included. A Scientific Reference is available that contains more detailed information on the model physics.

An attempt has been made throughout this document to provide the following text convention. Variable names used in the code are **typewritten**. Subroutine names are given in *italic*, and file names are in **boldface**.

1.1 What's new in CSIM5?

CSIM5 is an upgraded version of CSIM4 which was released in October 2002. The model physics are similar to that of CSIM4, but a majority of the code has been rewritten for vectorization and to make the CSIM code similar to that of CICE, the LANL sea ice model. The major changes are:

- A module for a new incremental remapping transport scheme was added called **ice_transport_remap.F**. The MPDATA transport scheme, formerly in **ice_transport.F**, was moved to **ice_transport_mpdata.F**. Open water advection was added to the incremental remapping.
- A bug in **ice_albedo.F** was fixed to avoid negative albedos for thin, bare, melting ice.
- A bug in **ice_ocean.F** was fixed to include **fswthru** in the calculation of sea surface temperature.
- A salt flux calculation was added so the ice reference salinity could be changed to a non-zero value.
- The sea ice momentum equation modified for the free drift regime. The dynamics scheme treats areas with lower ice concentrations more accurately. See Hunke and Dukowicz (2003).
- **ice_coupling.F** has been rewritten to be compatible with the latest version of the CCSM coupler.
- An additional field **Qref** is calculated in *atmo_boundary_layer* and passed to the coupler.

- Each ice thickness category has 4 thickness layers. Previously, the two thinnest categories had two layers.
- A sub-cycling timestep `ndyn_dt` was added to the dynamics to get around a model instability that would manifest itself in MPDATA.
- The snow and ice albedos, used for coupled model tuning, were moved to the namelist to make modification easier.
- It is now possible to run CSIM as an uncoupled model. The module `ice_flux_in.F` has been added to read in forcing data.
- Most modules have been modified to run efficiently on vector platforms. Grid indices `i`, `j` are no longer passed into subroutines. Shorter loops over ice categories and vertical layers have been moved outside the longer loops over `i` and `j`. Directives have been placed before certain loops to enforce vectorization.
- The thermodynamics modules from CSIM4, `ice_tstm.F`, `ice_vthermo.F`, `ice_therm_driver.F`, and `ice_dh.F`, have been replaced by two new modules `ice_therm_vertical.F` and `ice_therm_itd.F`. The calculations in `ice_therm_vertical.F` are done before the `to_coupler` call, and those in `ice_therm_itd.F` are done after this call.
- New modules `ice_exit.F` and `ice_work.F` have been added that contain code for aborting the model and globally accessible work arrays.
- The gx3v4 grid had been replaced by a new gx3v5 grid. The coupled model produced a poor meridional overturning circulation (MOC) with the gx3v4 grid. The new grid has points in different locations and has higher resolution in the North Atlantic than gx3v4. The simulation with gx3v5 gives a better ice thickness distribution and produces a better MOC.
- The prescribed ice model is not supported in this release.
- Most CSIM and CICE modules are very similar, except for the mechanical redistribution modules.

The CSIM source code is based on the LOS Alamos sea ice model CICE model. After a code merger with CICE was carried out to take advantage of the vector-friendly code, the models are very similar. If there are some topics that are not covered in the CSIM documentation, users are encouraged to look at the CICE documentation Hunke and Lipscomb (2004). It is available at <http://climate.lanl.gov/Models/CICE/index.htm>.

2 Quick Start Guide

2.1 What is needed to run CSIM?

A number of target architectures are supported for CCSM including IBM SP, SGI Origin, SGI Altix, Linux, NEC Earth Simulator, Cray X1, and Compaq ES. The resources required to run CSIM coupled in CCSM are listed in the CCSM3 User's Guide at:

<http://www.cesm.ucar.edu/models/ccsm3.0/ccsm>.

Two target architectures are supported for uncoupled CSIM: IBM and SGI. Below is a list of what is required to run CSIM uncoupled:

- Operating System: IBM AIX, or SGI IRIX64
- Tools: gunzip, gnumake
- Compilers: Fortran90, C
- Permanent disk space

- Temporary disk space
- Libraries: MPI, netCDF
- Input Data : 10.86 MB for gx3v5 grid
- CSIM source code: 0.83 MB
- Atmospheric Forcing Data : 2.7 GB for gx3v5 grid

2.2 Downloading Source Code and Input Datasets

The source code to run CSIM fully coupled or uncoupled and the required datasets can be obtained via a web download. The source code, input datasets, and documentation for CCSM are available via the web at:

<http://www.cesm.ucar.edu/models/ccsm3.0>

Instructions for downloading and untarring the CCSM3.0 distribution are in the CCSM3 User's Guide. If you have the source code for CCSM, you also have all the source code to run CSIM uncoupled. If you only need the source code and input files for the uncoupled model, it is available at:

<http://www.cesm.ucar.edu/models/ice-csim5/distribution>

The source code, input data and atmospheric forcing for the uncoupled ice model come in the following form:

- **csim5_code.tar.gz**
- **csim5_inputdata_gx3v5.tar.gz** for the low resolution grid
- **csim5_inputdata_gx1v3.tar.gz** for the high resolution grid
- **csim5_atmforcing_gx3v5.tar.gz**

To uncompress and untar these files, use the following Unix gunzip and tar commands:

```
gunzip -c csim5_code.tar.gz | tar -xf -
gunzip -c csim5_inputdata_gx3v5.tar.gz | tar -xf -

gunzip -c csim5_atmforcing_gx3v5.tar.gz | tar -xf -
```

The atmospheric forcing datasets for uncoupled CSIM5 are available at the same URL, but are not necessary to get the model set up and running.

For both coupled and uncoupled models, the source code should be extracted from the tar file in a small, permanent disk, such as your home directory or a cross-mounted file system. If possible, the data input files should also be extracted on a large, permanent cross-mounted disk. These files are copied to the temporary disk during the build stage.

2.3 Running CSIM Coupled

The scripts for running CSIM coupled are documented in the CCSM3 User's Guide:

<http://www.cesm.ucar.edu/models/ccsm3.0/ccsm>.

There are several configurations that may be of interest to ice modelers. The B configuration is the fully coupled model with active atmosphere, ice, land and ocean components communicating through the flux coupler. This configuration will result in the most realistic ice simulations.

The D configuration consists of CSIM, coupled to the data atmosphere `datm6` (<http://www.cesm.ucar.edu/models/ccsm3.0/datm6>), data ocean `docn6` (<http://www.cesm.ucar.edu/models/ccsm3.0/docn6>), and data land `dlnd6` (<http://www.cesm.ucar.edu/models/ccsm3.0/dlnd6>), components. This configuration runs quickly and is used for testing the software engineering aspects of the model. This configuration will not result in the best sea ice simulation, since `docn6` does not allow ice growth in open ocean regions or leads, and the data read in by `datm6` is from previous atmospheric model simulations.

The M configuration will result more realistic sea ice simulations than the D configuration. This setup replaces `datm6` in the D configuration with the `latm6` (<http://www.cesm.ucar.edu/models/ccsm3.0/latm6>), data atmosphere model and the ocean mixed layer model within the ice model. The ocean mixed layer is a simple slab model which computes an ocean surface temperature and allows for ice formation due to ocean supercooling. More information on the formulation of this model can be found in the Scientific Document. `latm6` runs on the T62 grid, and one year of NCEP forcing is included with this release.

2.4 Running CSIM Uncoupled

It is assumed that the user has downloaded the source code and input data from the web page described in Section 2.2. This section is intended to get uncoupled CSIM running "out of the box" with a minimal amount of information. More information on modifying the scripts is given in Section 3.1. The default configuration is a 10 day, startup run on the `gx3v5` grid, using 8 processors and the message passing interface (MPI). The debugging option is turned on, history files are written out daily, and restart files are written out every 5 days. The debugging option should be turned off, and the output frequency should be decreased before starting any production runs. If your system does not have 8 available processors or MPI, see Section 2.4.3 on how to run the model on a single processor without MPI.

Running this configuration will verify that the library and compiler options are properly set in the **Macros.<OS>** file, all the input data is in the correct place and the environment variables are set correctly before any further changes are made to the scripts or the source code. This will also provide benchmark output.

NOTE: If you are running this model a machine other than an IBM running AIX or an SGI running IRIX, you may need to make an equivalent **Macros.<OS>** file with the paths and settings modified for your system.

Before you start, modifications will be needed in the run script `csim_run` to set the directories for the source code, input data, and executables. The following is a list of the environment variables that will need to be changed by the user:

```
setenv CSIMDIR /home/$LOGNAME/csim5           # directory of scripts
setenv CSIMDATA /fs/cgd/csm/inputdata/ice/csim4 # dir for input data
setenv CASE test.me                          # Case name
setenv EXEROOT /ptmp/$LOGNAME/$CASE          # run model here
setenv SHRCODE $CSIMDIR                      # dir for share code
```

`$CSIMDIR` is the top directory of the source code, where the scripts are located. `$CSIMDATA` is the directory where the input data sets are located. `$CASE` is a string with a case name for the model run and should be kept short since it is used in path and file names. `$EXEROOT` is typically a large temporary disk where the executable files, and input data sets will reside during execution. Information output by the model will also be written to `$EXEROOT`.

The location of `$SHRCODE` will depend on where the source code was downloaded. If it was obtained with CCSM3.0 distribution, the `$SHRCODE` directory will be under `ccsm3/models/csm_share`. If you only have the source code for CSIM, the share code will be in the same directory as the rest of the ice model source code.

			*for uncoupled *
(Makefile macros	(resolution-dependent	(F90 source	*ice model *
for uncoupled	input files)	code)	* * * * * * * *
ice model)			

3.1 Coupled Model Scripts

The CCSM3 scripts have been significantly upgraded from CCSM2 and are based on a completely different design philosophy. The new scripts will generate a set of "resolved scripts" for a specific configuration determined by the user. The configuration includes components, resolution, run type, and machine. The run and setup scripts that were previously in the `/scripts` directory for CCSM2 are now generated automatically. See the CCSM3 User's Guide for information on how to use the new scripts. The file that contains the ice model namelist is now located in `ccsm3/scripts/$CASE/Buildnml_Prestage`. The script containing the environment variables used for building the executable file for the ice model is in `ccsm3/scripts/$CASE/Buildexe`. The contents of the ice model namelist are described in section 4.

3.2 Uncoupled Run Script

The run script for the uncoupled model is called `csim_run` and is located in `/ccsm3/models/ice/csim4/src`. Its purpose is to coordinate setting the batch system options, the environment variables, executing the CSIM setup script, setting up the stdout file, and submitting the model to run.

At the top of the run script, the settings for the IBM SP and the SGI Origin 2000 batch queue environments are set. These commands are machine and site dependent. Following this, the variables for the run environment are defined. These variables are listed in Table 1.

`$CASE` is a character string that identifies a particular model run. It can be up to 16 characters long, but it is best kept short, since it is used as part of the restart, history, and initial filenames. `$CASESTR` is a longer string that describes a model case.

`$RUNTYPE` is a character string that specifies the state in which the model is to begin a run. `startup` and `continue` are the supported run types. A startup run can be initialized by reading input from a file or from initial conditions set within the ice model. This option is controlled by the environment variable `$RESTART` in the setup script (see Section 6.1). Continue runs are described in Section 6.2.

`$NCAT` is an integer that sets the number of ice thickness categories. The default value is 5 categories. If you are considering changing `$NCAT` to values other than 3 or 10, read Section 7. This is an involved process that deserves its own section.

3.2.1 Using the Ocean Mixed Layer Model within CSIM

`$OCEANMIXED_ICE` is a logical variable, if `.true.`, is used to implement the slab ocean mixed layer model in the ice model. It is a simple model that is forced using output from a POP ocean simulation. More details on the physics of the ocean mixed layer model can be found in the Physics Documentation. It can be run with the `gx3v5` or the `gx1v3` grid. To use the mixed layer model, set

```
setenv \ $OCEANMIXED\_ICE .true.
```

in `csim_run`.

3.2.2 Changing Grid Resolution

`$GRID` is a character string used to specify the horizontal grid. Presently, two resolutions are supported for the ice model: `gx3v5` and `gx1v3`. In both of these grids, the North Pole has been displaced into Greenland. `gx3v5` is the coarser grid, with longitudinal resolution of 3.6 degrees. The latitudinal resolution varies, with a resolution of approximately 0.9 degrees near the equator. `gx1v3` is the finer resolution grid, with a longitudinal resolution of approximately one degree. Its latitudinal resolution is also variable, with a resolution of approximately 0.3 degrees near the equator.

3.2.3 Changing the Number of Processors

`$NX` and `$NY` are the number of processors used by the ice model for internal parallelization. Currently, `$NX` and `$NY` MUST divide evenly into the grid dimensions. There are checks for this in the setup script and in the ice source code; the model will stop if these criteria are not met. `$NLAT` and `$NLON` are used for this purpose. For load balancing purposes, `$NY` should be ≤ 2 . If it is greater than this, the processors assigned subdomains near the equator will not be doing much work.

For the `gx3v5` grid, the ice model is typically run on 8 tasks, with `NX=4`, `NY=2`. Running the ice model with `NX=8` and `NY=1` tasks on the `gx3v5` grid will result in an error, since 8 does not divide evenly into the 100 longitude points. When this happens, the model will stop with an error message written to the log file.

If you are submitting the model to a batch queue with the number of processors modified from the default, you will also have to modify the batch queue environment information at the top of the script. The default setting for the IBM is:

```
# @ total_tasks = 8
# @ node = 1
```

These two lines request a total of eight MPI processes, on one 8-way node. For the NCAR SGI, the default setting is:

```
# QSUB -l mpp_p=8          # request 8 processors
```

and for other SGI's it may be

```
# BSUB -n 8 .
```

3.3 Uncoupled Setup Script

The purpose of the setup script, `csim.setup.csh`, is to build an executable version of the ice model, document what source code and data files are being used in the `ice.log.$LID` file, and gather or create any necessary input data files. `$LID` is a time stamp set in the run script. The environment variables set locally in the ice setup script are listed in Table 2.

NOTE: The variables shown in Table 2 will rarely have to be modified by the user, since they depend on variables set in the run script. The most common changes made in the script file will be to the namelist discussed in Section 4.

`$HSTDIR`, `$RSTDIR` and `$INIDIR` are the directories in `$EXEROOT` where the history, restart, and initial condition files are output, respectively. The ice model input templates are located in `$ICESRC`. These templates are fortran modules that contain information on grid dimensions, number of ice thickness categories and vertical resolution in the ice categories. The grid is determined in the run script and the resolution `$RES` is set in `csim.setup.csh`: 100x116 for the `gx3v5` grid and 320x384 for `gx1v3`. Depending on `$RES` and the number of ice thickness categories `$NCAT` set in `csim_run`, the appropriate input template `ice_model_size.${RES}x${NCAT}` will be copied into the directory where the ice model is being built and renamed `ice_model_size.F`. Files exist for the following configurations:

```
$CSIMDIR/input_templates/ice_model_size.F.100x116x1
$CSIMDIR/input_templates/ice_model_size.F.100x116x3
$CSIMDIR/input_templates/ice_model_size.F.100x116x5
$CSIMDIR/input_templates/ice_model_size.F.100x116x10
$CSIMDIR/input_templates/ice_model_size.F.320x384x1
$CSIMDIR/input_templates/ice_model_size.F.320x384x3
$CSIMDIR/input_templates/ice_model_size.F.320x384x5
$CSIMDIR/input_templates/ice_model_size.F.320x384x10
```

Table 1: Environment Variables Set in the Run Script (**csim_run**)

Variable	Description
CASE	case name
CASESTR	short descriptive text string, used in history files
OCEANMIXED_ICE	logical variable used to implement ocean mixed layer model
ICE_GRID	ice model grid (gx3v5 or gx1v3)
RUNTYPE	run type (startup or continue)
NCAT	number of thickness categories in the ice thickness distribution
NX	number of processors assigned in the x direction, used for MPI grid decomposition
NY	number of processors assigned in the y direction, used for MPI grid decomposition
BINTYPE	Set to MPI for internal parallelization, set to single for non-MPI runs
CSIMDIR	source code base directory
SHRCODE	share code directory
CSIMDATA	input data base directory
CBLD	makefile and Macros directory
EXEROOT	Run model, mv data, output put here
LID	timestamp for file ID string
OBJDIR	ice model code is built here

NOTE: Files exist only for certain numbers of ice thickness categories (1, 3, 5, and 10). If you need a number of categories other than these, the model will not run as is. See Section 7 for information on how to change the number of ice thickness categories.

The variable `$OML_ICE_SST_INIT` is used if `$OCEANMIXED_ICE` is set to `.true.` in the run script and determines the initial sea surface temperature. If the run is a startup run, this variable is set to true in the ice setup script, and the January 1 value of the sea surface temperature is read from the POP forcing file. Thereafter, the value of `$OML_ICE_SST_INIT` is set to `.false.`, and sea surface temperature and the freeze/melt potential is read from a restart file.

The ice model contains two namelists. The variables for both lists are set in `csim.setup.csh` and are written to the file `ice.in` in `$EXEROOT` when the setup script is executed. Changes to the namelists must be made in the run or setup script, not in the `ice.in` file. The ice model reads his file at runtime. Therefore,

Table 2: Environment Variables Set in the Ice Setup Script

Symbol	Description
HSTDIR	directory where history files are written
RSTDIR	directory where restart files are written
INIDIR	directory where initial condition files are written
ICESRC	directory where ice model input templates are located
RES	grid dimensions used to select model resolution
NLAT	number of latitudes in grid resolution
NLON	number of longitudes in grid resolution
OML_ICE_SST_INIT	logical variable, if true initialize ocean mixed layer temperature from within ice model
RESTART	logical variable used to initialize model from a restart file
RSTFILE	name of restart file

the namelist will be updated even if the ice model is not recompiled.

One namelist is called `icefields_nml` and is defined in `ice_history.F`. It contains a list of logical variables that correspond to ice fields that will be written to the history file. By default, all these variables are set to `.true.`, so leaving the namelist blank will result in all fields being written to the history file. The available fields are listed in Table 11. Changing the content of the history files via the namelist is discussed in section 8.3.3.

The other namelist is called `ice_nml` and is defined in `ice_init.F`. It contains variables that control the physics used in the model. They are listed in Tables 3-8. Some of the variables in the namelist are determined from environment variables set in the scripts. Variables that are commonly changed directly in the namelist are the timestep `dt`, the length of the model run `npt`, and the number of subcycles per timestep in the ice dynamics `ndte`.

3.4 The Build Environment

The coupled and uncoupled ice models use the same Makefile and make environment. These files are located in the `ccsm3/models/bld` directory for the coupled model and in `ccsm3/models/ice/csim4/src/bld` for the uncoupled model. These directories contain the following files:

- **Macros.AIX** contains build settings specific to AIX (IBM SP3).
- **Macros.IRIX64** contains build settings specific to IRIX64 (SGI Origin).
- **makdep.c** evaluates the code dependencies for the source code
- **Makefile** is a generic gnumakefile

There is a **Macros.<OS>** file for each supported platform. These files contain machine dependent preprocessor, compiler and library information for building the model. The **Macros.<OS>** files for the uncoupled model have been simplified, since most of the libraries used by the coupled model are not used by the uncoupled ice model. If you are running the model on a platform other than those tested, you will need to create a new **Macros.<OS>** file and modify the paths and settings for your system. In some cases, CSIM has a set of options that are different from the default values at the top of the file. These are after the line `ifdef $(MODEL),csim` in the **Macros.<OS>** files and are described below.

3.4.1 CSIM Preprocessor Flags

Preprocessor flags are activated in the form `-Doption` in the **Macros.<OS>** files. The flags specific to the ice model are

```
CPPDEFS := $(CPPDEFS) -Dcoupled -DNPROC_X=$(NX) -DNPROC_Y=$(NY) -D_MPI
```

The option `-Dcoupled` is set to activate the coupling interface. This will include the source code in `ice_coupling.F`, for example. For uncoupled runs, it has been removed. If a coupler other than the CCSM coupler is used, there is a flag called `-Dfcd_coupled` that will keep the fluxes from being divided by the ice area. In coupled runs, the CCSM coupler multiplies the fluxes by the ice area, so they are divided by the ice area in CSIM to get the correct fluxes.

The options `-DNPROC_X=$(NX)` and `-DNPROC_Y=$(NY)` set the number of processors used in each grid direction. These values are set automatically in the scripts for the coupled model, and in `csim_run` by the user for uncoupled runs. `NX` and `NY` must divide evenly into the grid, and are used only for MPI grid decomposition. If `NX` or `NY` do not divide evenly into the grid, the model setup will exit from the setup script and print an error message to the `ice.log*` (standard out) file.

The flag `-D_MPI` sets up the message passing interface. This must be set for runs using a parallel environment. To get a better idea of what code is included or excluded at compile time, grep for `ifdef` and `ifndef` in the source code or look at the `*.f` files in the `/obj` directory.

3.4.2 CSIM Compiler Options

The name of the Fortran compiler is set by the variable `FC` in the `Macros.<OS>` files. The default name of the compiler is `f90` on the SGI and `mpxlf90_r` on the IBM SP. CSIM uses the following compiler options on the SGI platform:

```
FFLAGS      := -c -64 -mips4 -O2 -r8 -i4 -show -extend_source
```

On the IBM, the following compiler options are set in `Macros.AIX`:

```
FFLAGS      := -c -O2 -qstrict -Q -qmaxmem=-1 -qrealsize=8 \
              -qarch=auto -qtune=auto
```

4 Namelist Variables

CSIM uses the same namelists for both the coupled and uncoupled models. This section describes the namelist variables in the namelist `ice_nml`, which determine time management, output frequency, model physics, filenames, and options for the mixed layer ocean model. The ice namelists for the coupled model are now located in `ccsm3/scripts/$CASE/Buildnml.Prestage`. Modifications to the uncoupled namelist can be made in `ccsm3/models/ice/csim4/src/csim.setup.csh`.

A script reads the input namelist at runtime, and writes the namelist information to the file `ice.in` in the directory where the model executable is located. Therefore, the namelist will be updated even if the ice model is not recompiled. The default values of `ice_nml` are set in `ice_init.F`. If they are not set in the namelist in the script, they will assume the default values listed in Tables 3-8, which list all available namelist parameters. The default values shown here are for the coupled model, which is set up for a production run. Several of the variables have different values for the uncoupled, which is set up for a 10 day test run with more frequent output. Only a few of these variables are required to be set in the namelist; these values are noted in the paragraphs below. An example of the default namelist is shown in Section 4.8.1.

Table 3: Namelist Variables for Time Management

Variable	Type	Default Value	Description
<code>runid</code>	Character	'unknown_runid'	Text used in netCDF global attributes
<code>runtype</code>	Character	'unknown_runtype'	Determines if BASEDATE is received from coupler or restart file
<code>istep0</code>	Integer	0	Step counter, number of steps taken in previous integration
<code>npt</code>	Integer	99999	Total number of timesteps in a model run, model stops when <code>istep</code> exceeds <code>npt</code> (not used in coupled runs)
<code>dt</code>	Double	3600.	Timestep in seconds
<code>ndyn_dt</code>	Integer	1	Times to loop through ice dynamics
<code>ndte</code>	Integer	120	Number of subcycles per timestep in ice dynamics

The time management namelist options are shown in Table 3. `runid` is a character string that contains descriptive information gathered from the run script. This information is written to the global attributes in the history files. `runtype` is determined from the value of `$RUNTYPE` set in the run script. The options for this are discussed in section 6. `istep0` is the number of steps taken in a previous integration and is written to the restart file.

4.1 Changing the Length of the Model Run

The length of an uncoupled model run is controlled by the variable `npt`. It is the total number of time steps taken in an integration. The value of `npt` is not used in a coupled run, since the point at which integration

Table 4: Maximum values for ice model timestep dt

Grid	$\min(\Delta x, \Delta y)$	$\max \Delta t$
gx3v5	28845.9 m	4.0 hr
gx1v3	8558.2 m	1.2 hr

is stopped is determined by the coupler. The length of a coupled run should be set in the

4.2 Changing the timestep

dt is the timestep in seconds for the ice model thermodynamics. The thermodynamics component is stable but not necessarily accurate for any value of the timestep. The value chosen for dt depends on the stability of the transport and the grid resolution. A conservative estimate of dt for the transport using the MPDATA advection scheme is

$$\Delta t < \frac{\min(\Delta x, \Delta y)}{4\max(u, v)}. \quad (1)$$

Maximum values for dt for the two standard CCSM POP grids, assuming $\max(u, v) = 0.5m/s$, are shown in Table 4. The default timestep for CSIM is one hour.

The calculation of the ice velocities is subcycled ndte times per timestep so that the elastic waves are damped before the next timestep. The subcycling timestep is calculated as $\mathit{dte} = \mathit{dt}/\mathit{ndte}$ and must be sufficiently smaller than the damping timescale T , which needs to be sufficiently shorter than dt

$$\mathit{dte} < T < \mathit{dt} \quad (2)$$

This relationship is discussed in Hunke (2001); also see Hunke and Lipscomb (2002), section 4.4. The best ratio for $[\mathit{dte} : T : \mathit{dt}]$ is $[1 : 40 : 120]$. Typical combinations of dt and ndte are (3600., 120), (7200., 240) (10800., 120).

Occasionally, ice velocities are calculated that are larger than what is assumed when the model timestep is chosen. This causes a CFL violation in the transport scheme. A namelist option was added ($\mathit{ndyn_dt}$) to subcycle the dynamics to get through these instabilities that arise during long integrations. The default value for this variable is one, and is typically increased to two when the ice model reaches an instability. The value in the namelist should be returned to one by the user when the model integrates past that point.

4.3 Writing Output

The namelist variables that control the frequency of the model diagnostics, netCDF history files, and binary restart files are shown in Table 5. By default, diagnostics are written out once every 24 timesteps to the ascii file **ice.log.\$LID** (see section 8.1). $\$LID$ is a time stamp that is set in the main script.

$\mathit{histfreq}$ controls the output frequency of the netCDF history files; writing monthly averages is the default. The content of the history files is described in section 8.3. The value of $\mathit{hist_avg}$ determines if instantaneous or averaged variables are written at the frequency set by $\mathit{histfreq}$. If $\mathit{histfreq}$ is set to '1' for instantaneous output, $\mathit{hist_avg}$ is set to **.false.** within the source code to avoid conflicts. $\mathit{dumpfreq}$ and $\mathit{dumpfreq_n}$ control the output frequency of the binary restart files; writing one restart file per year is the default.

If $\mathit{print_points}$ is **.true.**, diagnostic data is printed out for two grid points, one near the north pole and one near the Weddell Sea. The points are set at the top of **ice_diagnostics.F**. This option can be helpful for debugging.

4.4 Model Physics

The namelist variables for the ice model physics are listed in Table 6. **restart** is almost always true since most run types begin by reading in a binary restart file. See section 6 for a description of the run types and

Table 5: Namelist Variables for Writing Output

Variable	Type	Default	Description
<code>diagfreq</code>	Integer	24	Frequency of diagnostics written (min, max, hemispheric sums) to standard output 24 => writes once every 24 timesteps 1 => diagnostics written each timestep 0 => no diagnostics written
<code>histfreq</code>	Character	'm'	Frequency of output written to history file 'D' or 'd' writes daily data 'W' or 'w' writes weekly data 'M' or 'm' writes monthly data 'Y' or 'y' writes yearly data '1' writes every timestep '0' no history data is written
<code>hist_avg</code>	Logical	.true.	If true, averaged history information is written out at a frequency determined by <code>histfreq</code> . If false, instantaneous values rather than time-averages are written.
<code>dumpfreq</code>	Character	'y'	Frequency restart data is written to file 'D' or 'd' writes restart every <code>dumpfreq_n</code> days 'M' or 'm' writes restart every <code>dumpfreq_n</code> months 'Y' or 'y' writes restart every <code>dumpfreq_n</code> years '0' no restart data is written
<code>dumpfreq_n</code>	Integer	1	Frequency restart data is written to file
<code>print_points</code>	Logical	.false.	print diagnostic data for two grid points

Table 6: Namelist Variables for Model Physics

Variable Name	Type	Default Value	Description
<code>restart</code>	Logical	<code>.false.</code>	If true, model is initialized using a restart file, if false, model is initialized using initial conditions in ice_init.F .
<code>kcolumn</code>	Integer	0	Column model flag. 0 = off 1 = column model (not tested or supported)
<code>kitd</code>	Integer	1	Determines ITD conversion 0 = delta scheme 1 = linear remapping
<code>kdyn</code>	Integer	1	Determines ice dynamics 0 = No ice dynamics 1 = Elastic viscous plastic dynamics
<code>kstrength</code>	Integer	1	Determines pressure formulation 0 = Hibler (1979) parameterization 1 = Rothrock (1975) parameterization
<code>evp_damping</code>	Logical	<code>.false.</code>	If true, use damping procedure in evp dynamics (not supported).
<code>snow_into_ocn</code>	Logical	<code>.true.</code>	If true, snow on ridged ice falls into ocean.
<code>advection</code>	Character	<code>'remap'</code>	Determines horizontal advection scheme. <code>'remap'</code> = incremental remapping <code>'mpdata2'</code> = second order advection <code>'upwind'</code> = first order advection
<code>grid_type</code>	Character	<code>'displaced_pole'</code>	Determines grid type. <code>'displaced_pole'</code> or <code>'rectangular'</code> (not supported)
<code>albicev</code>	Double	0.73	Visible ice albedo
<code>albice_i</code>	Double	0.33	Near-infrared ice albedo
<code>albsnowv</code>	Double	0.96	Visible snow albedo
<code>albsnow_i</code>	Double	0.68	Near-infrared snow albedo
<code>no_ice_ic</code>	Logical	<code>.false.</code>	Initializes ice model with no ice

about using restart files and internally generated model data as initial conditions. `kcolumn` is a flag that will run the model as a single column if is set to 1. This option has not been thoroughly tested and is not supported.

`kitd` determines the scheme used to redistribute sea ice within the ice thickness distribution (ITD) as the ice grows and melts. The linear remapping scheme is the default and approximates the thickness distribution in each category as a linear function (Lipscomb (2001)). The delta function method represents $g(h)$ in each category as a delta function (Bitz et al. (2001)). This method can leave some categories mostly empty at any given time and cause jumps in the properties of $g(h)$.

`kdyn` determines the ice dynamics used in the model. The default is the elastic-viscous-plastic (EVP) dynamics Hunke and Dukowicz (1997). If `kdyn` is set to 0, the ice dynamics is inactive. In this case, ice velocities are not computed and ice is not transported. Since the initial ice velocities are read in from the restart file, the maximum and minimum velocities written to the log file will be non-zero in this case, but they are not used in any calculations.

The value of `kstrength` determines which formulation is used to calculate the strength of the pack ice. The Hibler (1979) calculation depends on mean ice thickness and open water fraction. The calculation of Rothrock (1975) is based on energetics and should not be used if the ice that participates in ridging is not well resolved.

`evp_damping` is used to control the damping of elastic waves in the ice dynamics. It is typically set to `.true.` for high-resolution simulations where the elastic waves are not sufficiently damped out in a small timestep without a significant amount of subcycling. This procedure works by reducing the effective ice strength that's used by the dynamics and is not a supported option.

The value of `snow_into_ocean` determines what happens to the snow on ice that is ridged. The default value is `.true.`, so the snow cover on ice that undergoes ridging is put into the ocean. If this variable is `.false.`, the snow on ice undergoing ridging remains on the ice.

`advection` determines the horizontal transport scheme used. The default scheme is the incremental remapping method (Lipscomb and Hunke (2004)). This method is less diffusive and is computationally efficient for large numbers of categories or tracers. The MPDATA scheme is also available. It is second order accurate, and more computationally expensive than remapping. The upwind scheme is only first order accurate.

For coupled runs, both supported grids (`gx3v5` and `gx1v3`) are `'displaced_pole'`. The `'rectangular'` option for a regular grid with constant latitude and longitude spacing is not supported.

The values of the snow and ice albedos are now set in the namelist. The ice albedos are those for ice thicker than `ahmax`, which is currently set at 0.5 m. This thickness is a parameter that can be changed in `ice_albedo.F`. The snow albedos are for cold snow. `no_ice_ic` provides an option to initialize the ice model with no ice cover.

4.5 File Names

The namelist parameters listed in Table 7 are for initial condition, restart, and history file and directory information. During execution, the ice model reads grid and land mask information from the files `grid_file` and `kmt_file` that should be located in the executable directory. There are commands in the scripts that copy these files from the input data directory, rename them from `global_$ICE_GRID.grid` and `global_$ICE_GRID.kmt` to the default filenames shown in Table 7.

The namelist variable `pointer_file` is set to the name of the pointer file containing the restart file name that will be read when model execution begins. The pointer file resides in the scripts directory and is created initially by the ice setup script but is overwritten every time a new restart file is created. It will contain the name of the latest restart file. The default filename `ice.restart.file` shown in Table 7 will not work unless some modifications are made to the ice setup script and a file is created with this name and contains the name of a valid restart file; this variable must be set in the namelist. More information on restart pointer files can be found in section 8.2.

`incond_dir`, `restart_dir` and `history_dir` are the directories where the initial condition file, the restart files and the history files will be written, respectively. These values are set at the top of the setup script and have been modified from the default values to meet the requirements of the CCSM filenames convention.

Table 7: Namelist Variables for File Names

Variable	Type	Default Value	Description
<code>grid_file</code>	Character	'data.domain.grid'	Input filename containing grid information.
<code>kmt_file</code>	Character	'data.domain.kmt'	Input filename containing land mask information.
<code>pointer_file</code>	Character	'ice.restart_file'	Pointer file that contains the name of the restart file.
<code>incond_dir</code>	Character	' '	Directory where netCDF initial condition file is output.
<code>restart_dir</code>	Character	' '	Directory where restart files are output.
<code>history_dir</code>	Character	' '	Directory where history files are output.
<code>incond_file</code>	Character	'incond'	Root name of netCDF output initial condition file.
<code>dump_file</code>	Character	'iced'	Prefix for output file containing restart information.
<code>history_file</code>	Character	'iceh'	Root name of history files.

Table 8: Namelist Variables for Ocean Mixed Layer Model

Variable Name	Type	Default Value	Description
<code>oceanmixed_ice</code>	Logical	.false.	If true, run model with ocean mixed layer model.
<code>oceanmixed_ice_file</code>	Character	'oceanmixed_ice.nc'	Name of file with ocean mixed layer data.
<code>oceanmixed_ice_sst_init</code>	Logical	.false.	If true, Jan 1 sst is read from forcing file.
<code>prntdiag_oceanmixed</code>	Logical	.false.	If true, print ocean mixed layer diagnostics.

This allows each type of output file to be written to a separate directory. If the default values are used, all of the output files will be written to the executable directory.

`incond_file`, `dump_file` and `history_file` are the root filenames for the initial condition file, the restart files and the history files, respectively. These strings have been determined by the requirements of the CCSM filename convention, so the default values listed in Table 7 are not the same as those shown in the namelist in Section 4.8.1. See sections 8.2 and 8.3 for an explanation of how the rest of the filename is created.

4.6 Ocean Mixed Layer Model

An ocean mixed layer model has been incorporated into the ice model since the CCSM data ocean component does not allow frazil ice growth to occur. This is due to the minimum ocean mixed layer temperature being fixed at the freezing point. It is a simple slab ocean mixed layer model that is forced using output from a POP ocean simulation. More details on the physics of the ocean mixed layer model can be found in the Physics Documentation. This option can be run with the `gx3v5` or the `gx1v3` grid.

The namelist variables for the ocean mixed layer model within the ice model are shown in Table 8. To use the slab ocean model, `$OCEANMIXED_ICE` must be set to `.true.` in the namelist. There are commands in the scripts that will copy the grid dependent forcing file from the input data directory to the executable directory and rename it `oceanmixed_ice.nc`. This is generally not the best ocean forcing, but can be used as a template for creating an ocean forcing file appropriate for the application.

The variable `$OML_ICE_SST_INIT` determines the initial sea surface temperature. For an initial or startup run, this variable should be set to true, and the January 1 value of the sea surface temperature will be read

Table 9: Namelist Variables for Atmospheric Forcing

Variable Name	Type	Default Value	Description
<code>atm_data_dir</code>	Path		Directory for atmospheric forcing data
<code>fyear_init</code>	Integer	First year of atmospheric forcing data	
<code>ocn_data_dir</code>	Path		Directory for oceanic forcing data
<code>ycycle</code>	Integer		Number of years in forcing data cycle
<code>year_init</code>	Integer	1	Initial year, if not using restart file

from the POP forcing file. For continuation runs, the value of `$OML_ICE_SST_INIT` should be set to false, and sea surface temperature and the freeze/melt potential will be read from a restart file. This variable will be automatically set in the scripts depending on the run type. When the slab ocean mixed layer within the ice model is used, the data that is received from the coupler from the ocean component (docn or POP) is overwritten by the values calculated by the ocean mixed layer. Therefore, it is not appropriate to use the ocean mixed layer option coupled to an active ocean model. Also, using the ice model with the slab ocean mixed layer turned on, coupled to an active atmosphere and a data ocean model will require changes to the coupler, since the ocean values calculated in the ice model will not be sent to the coupler and received by the atmosphere component.

4.7 Atmospheric Forcing

CSIM5 can be run uncoupled using atmospheric data from 1997, available from <http://www.cesm.ucar.edu/models/ice-cs>

These data files are on the low resolution grid and were created for testing the ice model. They will not produce the best sea ice simulation. Module `ice_flux.in.F` can be modified to change the forcing data. Namelist options for using the atmospheric forcing data are shown in Table 9. `atm_data_dir` is the root directory where the forcing resides. `fyear_init` and `year_init` should be set to 1997 and `ycycle` should be set to 1 for the forcing provided. `ocn_data_dir` should not be set since no ocean forcing is provided. Subroutines in `ice_flux.in.F` are available for reading in ocean sea surface temperature and salinity data and may be used for initializing and/or restoring the mixed layer properties; these subroutines currently are commented out in `ice_flux.in.F`. A sample namelist using these options is shown in section 4.8.4. A brief discussion of the atmospheric forcing files is given in section 5.1 as well as changes that should be made to `ice.F` to use this forcing.

4.8 Example Namelists

This section shows several examples of namelists from the coupled and uncoupled ice models. These examples are taken directly from `csim.buildnml_prestage.csh` for the coupled model and from `csim.setup.csh` for the uncoupled model. Most of the variables in the namelist are determined from environment variables set elsewhere in the scripts. Since the namelists from the coupled model are "resolved" by the scripts, meaning that the values of most of the shell script variables are put directly into the namelist, examples are shown for the most commonly used configurations. Variables that are commonly changed directly in the namelist are the timestep `dt` and the number of subcycles per timestep in the ice dynamics `ndte`.

4.8.1 Example 1: CCSM Fully Coupled

The following example is the namelist used for CCSM fully coupled, or the B configuration. The variables that are still set to shell script variables have been set at the top of `csim.buildnml_prestage.csh` or in other scripts. A completely resolved version of the namelist will be written to `ice_in` in the executable directory.

```

cat << EOF >! ice_in
&ice_nml
  runid          = 'TER.01a.T85_gx1v3.B.bluesky.105608'
, runtype       = '$runtype'
, istep0        = 0
, dt            = 3600.0
, ndte         = 120
, ndyn_dt       = 1
, npt           = 99999
, diagfreq      = 24
, histfreq      = 'm'
, dumpfreq      = 'y'
, dumpfreq_n    = 1
, hist_avg      = .true.
, restart       = $restart
, print_points  = .false.
, kitd          = 1
, kdyn          = 1
, kstrength     = 1
, evp_damping   = .false.
, snow_into_ocn = .true.
, advection     = 'remap'
, grid_type     = 'displaced_pole'
, grid_file     = 'data.domain.grid'
, kmt_file      = 'data.domain.kmt'
, incond_dir    = '$runinidir/'
, incond_file   = '$CASE.csim.i.'
, restart_dir   = '$runrstdir/'
, dump_file     = '$CASE.csim.r.'
, history_dir   = '$runhstdir/'
, history_file  = '$CASE.csim.h'
, albicev       = 0.73
, albice_i      = 0.33
, albsnowv      = 0.96
, albsnow_i     = 0.68
, no_ice_ic     = $no_ice_ic
, oceanmixed_ice = .false.
, oceanmixed_ice_file = 'oceanmixed_ice.nc'
, oceanmixed_ice_sst_init = .false.
, prntdiag_oceanmixed = .false.
, pointer_file  = 'rpointer.ice'
/

```

4.8.2 Example 2: Coupled Ice Only Model

This example is the M configuration. It is CSIM with the latm data atmosphere model, data land model, and the ocean mixed layer model within the ice model all communicating through the coupler. The following modifications will be made to the namelist when the resolved scripts are created for the M configuration. See the CCSM3 User's Guide (<http://www.cesm.ucar.edu/models/ccsm3.0/ccsm>) on how to create scripts for the M configuration.

```

  runid          = 'TER.01a.T62_gx1v3.M.bluesky.095234'
, oceanmixed_ice = .true.
, oceanmixed_ice_sst_init = $oml_ice_sst_init

```

4.8.3 Example 3: Uncoupled Ice Model

This example is the namelist from the uncoupled ice model that resides in the file `csim.setup.csh`. `npt` has been modified, since it determines the length of the uncoupled run. The snow and ice albedos used by CCSM are not set in the name list. The default values set in `ice_init.F` are used.

```
cat << EOF >! ice_in
&ice_nml
  runid          = '$CASE $CASESTR'
  , runtime      = '$RUNTYPE'
  , istep0       = 0
  , dt           = 3600.0
  , ndyn_dt      = 1
  , ndte         = 120
  , npt         = 240
  , diagfreq    = 24
  , histfreq    = 'm'
  , dumpfreq    = 'd'
  , dumpfreq_n  = 5
  , hist_avg    = .true.
  , restart     = $RESTART
  , print_points = .false.
  , kitd        = 1
  , kdyn        = 1
  , kstrength   = 1
  , evp_damping = .false.
  , snow_into_ocn = .true.
  , advection   = 'remap'
  , grid_type   = 'displaced_pole'
  , grid_file   = 'data.domain.grid'
  , kmt_file    = 'data.domain.kmt'
  , incond_dir  = '$INIDIR/'
  , incond_file = '$CASE.csim.i.'
  , restart_dir = '$RSTDIR/'
  , dump_file   = '$CASE.csim.r.'
  , history_dir = '$HSTDIR/'
  , history_file = '$CASE.csim.h'
  , pointer_file = '$CSIMDIR/rpointer.ice'
  , oceanmixed_ice = $OCEANMIXED_ICE
  , oceanmixed_ice_file = 'oceanmixed_ice.nc'
  , oceanmixed_ice_sst_init = $OML_ICE_SST_INIT
  , prntdiag_oceanmixed = .false.
/
```

4.8.4 Example 4: Uncoupled Ice Model with Atmospheric Forcing

To run the uncoupled ice model with atmospheric forcing, the following options should be added to the above namelist:

```
  , ycycle      = 1
  , year_init   = 1997
  , fyear_init  = 1997
  , atm_data_dir = '/ptmp/$LOGNAME/csim_forcing/atm/gx3v5/'
```

4.8.5 Example 5: History File Namelist

The second namelist controls what variables are written to the history file. By default, all files are written to the history file. Variables that are not output are set in the namelist `icefields_nml`. Some of the following fields are not written to the history file since they can be retrieved from the ocean history files. The melt and freeze onset fields are not used, since the information they contain may not be correct if the model is restarted mid-year. The ice areas and volumes for categories six through ten are not used, since the default thickness distribution consists of five ice categories.

```
&icefields_nml
  f_sst      = .false.
, f_sss      = .false.
, f_uocn     = .false.
, f_vocn     = .false.
, f_frzmlt  = .false.
, f_strtltx  = .false.
, f_strtltly = .false.
, f_mlt_onset = .false.
, f_frz_onset = .false.
, f_aice6    = .false.
, f_aice7    = .false.
, f_aice8    = .false.
, f_aice9    = .false.
, f_aice10   = .false.
, f_vice1    = .false.
, f_vice2    = .false.
, f_vice3    = .false.
, f_vice4    = .false.
, f_vice5    = .false.
, f_vice6    = .false.
, f_vice7    = .false.
, f_vice8    = .false.
, f_vice9    = .false.
, f_vice10   = .false.
/
```

5 Model Input Datasets

Both the coupled and uncoupled CSIM require a minimum of three files to run:

- `global_${ICE_GRID}.grid` is a binary file containing grid information and is renamed `data.domain.grid` when it is copied to the executable directory.
- `global_${ICE_GRID}.kmt` is a binary file containing land mask information and is renamed `data.domain.kmt`.
- `iced.0001-01-01.${ICE_GRID}.20lay` or `iced.0001-01-01.gx3v5.040213` are binary files containing initial condition information for the gx1v3 and gx3v5 grids, respectively. The thickness distribution in this restart file contains 5 categories, each with 4 layers.

Depending on the grid selected in the scripts, the appropriate `global*` and `iced*` files will be copied and renamed in the executable directory. Currently, only gx3v5 and gx1v3 grids are supported for the ice and ocean models.

An additional data file is necessary to use the ocean mixed layer within the ice model, depending on the specified grid:

- **pop_frc_gx1v3_010815.nc** contains monthly averaged ocean forcing from POP output.
- **pop_frc_gx3v5_040212.nc** same as above, but for the gx3v5 grid.

This file is renamed **oceanmixed_ice.nc** when it is copied into the executable directory.

5.1 Atmospheric Forcing

The uncoupled ice model will run without atmospheric forcing. It will use the fluxes set in subroutine *init_flux*. For atmospheric forcing, the following datasets are available on the gx3v5 grid, and can be read in using the module **ice_flux_in.F**. The directory where the data is located will have to be set in this module, and not in the scripts. The files are as follows:

```
/wherever/you/put/it/atm/gx3v5/ISCCPM/MONTHLY/RADFLX/cldf.1997.dat
/wherever/you/put/it/atm/gx3v5/ISCCPM/MONTHLY/RADFLX/swdn.1997.dat
/wherever/you/put/it/atm/gx3v5/MXA/MONTHLY/PRECIP/prec.1997.dat
/wherever/you/put/it/atm/gx3v5/NCEP/4XDAILY/STATES/dn10.1997.dat
/wherever/you/put/it/atm/gx3v5/NCEP/4XDAILY/STATES/q_10.1997.dat
/wherever/you/put/it/atm/gx3v5/NCEP/4XDAILY/STATES/t_10.1997.dat
/wherever/you/put/it/atm/gx3v5/NCEP/4XDAILY/STATES/u_10.1997.dat
/wherever/you/put/it/atm/gx3v5/NCEP/4XDAILY/STATES/v_10.1997.dat
```

These files are in direct access binary files, and the source is evident from the path names. **cldf.1997.dat** and **swdn.1997.dat** contain the monthly averaged cloud fraction and downwelling shortwave. **prec.1997.dat** is the monthly averaged precipitation in mm/month. The remaining files are the atmospheric density, specific humidity, air temperature, and wind fields. Note that these datasets are meant for testing the model and are not the best observational data for research. Users are advised not to publish results based on these datasets. To use this forcing, the following lines in **ice.F** need to be uncommented:

```
call init_getflux
call getflux
```

6 Run Types

The run types available for the coupled model are described in the CCSM User's Guide. There are two run types available for the uncoupled ice model and are described in this section.

6.1 Startup Runs

If \$RUNTYPE is set to **startup**, the model will read in the restart file that resides in \$CSIMDATA called **iced.0001-01-01.\$ICE_GRID.20lay**. The conditions in this file are for the gx1v3 grid and are from an equilibrium run using modified NCEP forcing. The setup script will create a pointer file named **rpointer.ice** with the name of the initial restart file in it. Startup runs can also be initialized using data created within the ice model, as described in the next section.

Using Initial Conditions from within CSIM

Initial conditions can be calculated within the ice model in a subroutine called *init_state* in **ice_init.F**. Here, the ocean surface is initialized at the freezing point everywhere north of 40 degrees and south of -40 degrees latitude, allowing ice to form everywhere in these regions. While running the ice model with a data ocean will melt any extra ice during the first year of integration, is not recommended that these initial conditions be used when running the ice model coupled to an active ocean model. The advantage of using this input is that it is not grid or land mask dependent. To use these initial conditions, set \$RESTART in **csim.setup.csh**:

```
set RESTART = .false.
```

Initializing the model using a restart file from an equilibrium run will result in a more physically reasonable scenario than the initial conditions set within CSIM. The drawbacks are that the data is binary, difficult to edit, and is date and grid dependent. A restart file will be used as initial conditions if

```
set RESTART = .true.
```

in `csim.setup.csh`.

6.2 Continue Runs

A continue run is an exact continuation of a previous run. This means that the run will produce a bit-for-bit answer as if the existing run had not stopped. The input data file is determined by the filename set in the restart pointer file (see section 8.2). It is assumed that `$CASE` has not changed. For a continue run, the only change required in the run script is to set:

```
RUNTYPE      continue
```

The date will continue from the previous run, since it is read in from the restart file.

7 Changing the Number of Ice Thickness Categories

The number of ice thickness categories affects ice model input files in three places:

- `$NCAT` in the run script
- The input template files `ice_model_size.F.$RESx$NCAT` in `$CSIMDIR/input_templates`
- The initial condition (restart) file in the input file directory

The number of ice thickness categories is set in `ccsm3/scripts/$CASE/Buildexe/csim.buildexe.csh` (coupled) or `csim_run` (uncoupled) using the variable called `$NCAT`. The default value is 5 categories. `$NCAT` is used to determine which module `ice_model_size.F.$RESx$NCAT` is copied to `/obj` before compilation where it is renamed `ice_model_size.F.` `$RES` is the resolution of the grid, 100x116 (gx3v5) and 320x384 (gx1v3) for low and high resolution grids, respectively. The input templates are located in `$CSIMDIR/input_templates` for 1, 3, 5 and 10 thickness categories on the default grid sizes of 100x116 and 320x384.

NOTE: To use one ice thickness category, the following changes will need to be made in the namelist:

```
, kitd          = 0
, kstrength     = 0
```

With these settings, the model will use the delta scheme instead of linear remapping and a strength parameterization based on open water area and mean ice thickness.

To create an input template with a number of categories other than the above values, copy an existing template that has the desired grid. Change `ncat` to the appropriate number of categories.

The information in the initial restart file is dependent on the number of ice thickness categories and the total number of layers in the ice distribution. An initial condition file exists only for the default case of 5 ice thickness categories, with four layers in each category. To create an initial condition file for a different number of categories or layers, these steps should be followed:

- Set `$NCAT` to the desired number of categories in `csim_run` (uncoupled) or `ccsm3/scripts/$CASE/Buildexe/csim.buildexe.csh` (coupled).

- Set the namelist variable `dumpfreq = 'm'` in `csim.setup.csh` (uncoupled) or `ccsm3/scripts/$CASE/Buildnml_Prestage/csim.buildnml_prestage.csh` (coupled) to print out restart files monthly.
- Set the namelist variable `restart = .false.` in `csim.setup.csh` (uncoupled) or `ccsm3/scripts/$CASE/Buildnml_Prestage/csim.buildnml_prestage.csh` (coupled) to use the initial conditions within the ice model.
- Create an input template (or use an existing one) with the desired number of categories.
- Run the model to equilibrium.
- The last restart file can be used as an initial condition file.
- Change the name of the last restart file to `iced.0001-01-01.$GRID`.
- Copy the file into the input data directory or directly into the `/rest` directory under the executable directory.

Note that the date printed inside the binary restart file will not be the same as 0001-01-01. For coupled runs, `$BASEDATE` will be the starting date and the date inside the file will not be used. For uncoupled runs, the starting date may have to be set in the code or changed in the restart file.

8 Output Data

The ice model produces three types of output data. A file containing ASCII text, also known as a log file, is created for each run that contains information about how the run was set up and how it progressed. A series of binary restart files necessary to continue the run are created. A series of netCDF history files containing gridded instantaneous or time-averaged output are also generated during a run. These are described below.

8.1 Stdout Output

Diagnostics from the ice model are written to an ASCII file that contains information from the compilation, a record of the input parameters, and how hemispherically averaged, maximum and minimum values are evolving with the integration. Certain error conditions detected within the ice setup script or the ice model will also appear in this file. Upon the completion of the simulation, some timing information will appear at the bottom of the file. The file name is of the form `ice.log.$LID`, where `$LID` is a timestamp for the file ID. It resides in the executable directory. The frequency of the diagnostics is determined by the namelist parameter `diagfreq`.

8.2 Restart Files

Restart files contain all of the initial condition information necessary to restart from a previous simulation. These files are in a standard IEEE 64 bit binary format. A restart file is not necessary for an initial run, but is highly recommended. The initial conditions that are internal to the ice model produce an unrealistic ice cover that an uncoupled ice model will correct in several years. The initial conditions from a restart file are created from an equilibrium solution, and provide more realistic information that is necessary if coupling to an active ocean model. The frequency at which restart files are created is controlled by the namelist parameter `dumpfreq`. The names of these files are preceded by the namelist parameter `dump_file` and, by default are written out yearly to the `/rest` directory under the executable directory. To change the directory where these files are located, modify the variable `$RSTDIR` at the top of the setup script. The names of the restart files follow the CCSM Output Filename Requirements. The form of the restart file names are as follows:

`$CASE.csim.r.yyyy-mm-dd-sssss`

For example, the file `$CASE.csim.r.0002-01-01-00000` would be written out at the end of year 1, month 12. A file containing the name of a restart file is called a restart pointer file. This filename information allows the model simulation to continue from the correct point in time, and hence the correct restart file.

Restart Pointer Files

A pointer file is an ascii file named `rpointer.ice` that contains the path and filename of the latest restart file. The model uses this information to find a restart file from which initialization data is read. The pointer files are written to and then read from the executable directory. For `startup` runs, a pointer is created by the ice setup script. Whenever a restart file is written, the existing restart pointer file is overwritten. The namelist variable `pointer_file` contains the name of the pointer file. Pointer files seldom need editing. The contents are usually maintained by the setup script and the component model.

8.3 History Files

History files contain gridded data values written at specified times during a model run. By default, the history files will be written to the directory `history_dir` defined in the namelist. The netCDF file names are prepended by the character string given by `history_file` in the `ice_nml` namelist. This character string has been set according to CCSM Output Filename Requirements. If `history_file` is not set in the namelist, the default character string `'iceh'` is used. The user can specify the frequency at which the data are written. Options are also available to record averaged or instantaneous data. The form of the history file names are as follows:

Yearly averaged: `$CASE.csim.h.yyyy.nc`

Monthly averaged: `$CASE.csim.h.yyyy-mm.nc`

Weekly averaged: `$CASE.csim.hw.yyyy-mm-dd-sssss.nc`

Daily averaged: `$CASE.csim.h.yyyy-mm-dd.nc`

Instantaneous (`histfreq = 'y', 'm', 'w' or 'd'`): `$CASE.csim.h.yyyy-mm-dd-sssss.nc`

Instantaneous (written every dt, `histfreq = 1`): `$CASE.csim.h.yyyy-mm-dd-sssss.nc`

`$CASE` is set in the main setup script. All history files are written to `/hist` in the executable directory. Changes to the frequency and averaging will affect all output fields. The best description of the history data comes from the file itself using the netCDF command `ncdump -h filename.nc`. Variables containing grid information are written to every file and are listed in Table 10. In addition to the history files, a netCDF file containing a snapshot of the initial ice state is created at the start of each run. The file name is `$CASE.csim.i.yyyy-mm-dd-sssss.nc` and is written to `/init` in the executable directory.

8.3.1 Caveats Regarding Averaged Fields

In computing the monthly averages for output to the history files, most arrays are zeroed out before being filled with data. These zeros are included in the monthly averages where there is no ice. For some fields, this is not a problem, for example, ice thickness and ice area. For other fields, this will result in values that are not representative of the field when ice is present. Some of the fields affected are:

- Flat, Fsens - latent and sensible heat fluxes
- albsni - snow/ice broadband albedo
- evap - evaporative water flux
- Fhnet - ice/ocn net heat flux
- Fswabs - snow/ice/ocn absorbed solar flux
- strairx, strairy - zonal and meridional atm/ice stress
- strcorx, strcory - zonal and meridional coriolis stress

For some fields, a non-zero value is set where there is no ice. For example, `Tsfc` has the freezing point averaged in, and `Flwout` has σT_f^4 averaged in. At lower latitudes, these values can be erroneous.

To aid in the interpretation of the fields, a field called `ice_present` is written to the history file. It contains information on the fraction of the time-averaging interval when any ice was present in the grid cell during the time-averaging interval in the history file. This will give an idea of how many zeros were included in the average.

The second caveat results from the coupler multiplying fluxes it receives from the ice model by the ice area. Before sending fluxes to the coupler, they are divided by the ice area in the ice model. These are the fluxes that are written to the history files, they are not what affects the ice, ocean or atmosphere, nor are they useful for calculating budgets. The division by the ice area also creates large values of the fluxes at the ice edge. The affected fields are:

- Flat, `Fsens` - latent and sensible heat fluxes
- `Flwout` - outgoing longwave
- `evap` - evaporative water flux
- `Fresh` - ice/ocn fresh water flux
- `Fhnet` - ice/ocn net heat flux
- `Fswabs` - snow/ice/ocn absorbed solar flux

When applicable, two of the above fields will be written to the history file: the value of the field that is sent to the coupler (divided by ice area) and a value of the flux that has been multiplied by ice area (what affects the ice). Fluxes multiplied by ice area will have the suffix `_aice` appended to the variable names in the history files. Fluxes sent to the coupler will have "sent to coupler" appended to the long_name. Fields of rainfall and snowfall multiplied by ice area are written to the history file, since the values are valid everywhere and represent the precipitation rate on the ice cover.

8.3.2 Changing Frequency and Averaging

The frequency at which data are written to a history file as well as the interval over which the time average is to be performed is controlled by the namelist variable `histfreq`. Data averaging is invoked by the namelist variable `hist_avg`. The averages are constructed by accumulating the running sums of all variables in memory at each timestep. The options for both of these variables are described in Table 5. If `hist_avg` is true, and `histfreq` is set to monthly, for example, monthly averaged data is written out on the last day of the month.

8.3.3 Changing Content

The second namelist in the setup script controls what variables are written to the history file. To remove a field from this list, add the name of the logical variable associated with that field to the `&icefields_nml` namelist in `csim.buildnml_prestage.csh` (coupled) or `csim.setup.csh` (uncoupled) and assign it a value of `.false..` For example, to remove ice thickness and snow cover from the history file, add

```
&icefields_nml
  f_hi   = .false.
  , f_hs = .false.
/
```

to the namelist.

Table 11: Standard Fields Available for Output to History File

Table 10: Time and Grid Information Written to History File

Field	Description	Units
time	model time	days
time_bounds	boundaries for time-averaging interval	days
TLON	T grid center longitude	degrees
TLAT	T grid center latitude	degrees
ULON	U grid center longitude	degrees
ULAT	U grid center latitude	degrees
tmask	ocean grid mask (0=land, 1=ocean)	
tarea	T grid cell area	m ²
uarea	U grid cell area	m ²
dxt	T cell width through middle	m
dyt	T cell height through middle	m
dxu	U cell width through middle	m
dyu	U cell height through middle	m
HTN	T cell width North side	m
HTE	T cell width East side	m
ANGLET	angle grid makes with latitude line on T grid	radians
ANGLE	angle grid makes with latitude line on U grid	radians
ice_present	fraction of time-averaging interval that any ice is present	

Logical Variable	Description	Units
f_hi	grid box mean ice thickness	m
f_hs	grid box mean snow thickness	m
f_Tsfc	snow/ice surface temperature	C
f_aice	ice concentration (aggregate)	%
f_aice1	ice concentration (category 1)	%
f_aice2	ice concentration (category 2)	%
f_aice3	ice concentration (category 3)	%
f_aice4	ice concentration (category 4)	%
f_aice5	ice concentration (category 5)	%
f_aice6	ice concentration (category 6)	%
f_aice7	ice concentration (category 7)	%
f_aice8	ice concentration (category 8)	%
f_aice9	ice concentration (category 9)	%
f_aice10	ice concentration (category 10)	%
f_vice1	ice volume (category 1)	m
f_vice2	ice volume (category 2)	m
f_vice3	ice volume (category 3)	m
f_vice4	ice volume (category 4)	m
f_vice5	ice volume (category 5)	m
f_vice6	ice volume (category 6)	m
f_vice7	ice volume (category 7)	m
f_vice8	ice volume (category 8)	m
f_vice9	ice volume (category 9)	m
f_vice10	ice volume (category 10)	m
f_uvel	zonal ice velocity	cm s ⁻¹

continued on next page

continued from previous page

f_vvel	meridional ice velocity	cm s ⁻¹
f_fswdn	downwelling solar flux	W m ⁻²
f_flwdn	downwelling longwave flux	W m ⁻²
f_snow	snow fall rate received from coupler	cm day ⁻¹
f_snow_ai	snow fall rate on ice cover	cm day ⁻¹
f_rain	rain fall rate received from coupler	cm day ⁻¹
f_rain_ai	rain fall rate on ice cover	cm day ⁻¹
f_sst	sea surface temperature	C
f_sss	sea surface salinity	g kg ⁻¹
f_uocn	zonal ocean current	cm s ⁻¹
f_vocn	meridional ocean current	cm s ⁻¹
f_frzmlt	freeze/melt potential	W m ⁻²
f_fswabs	absorbed solar flux sent to coupler	W m ⁻²
f_fswabs_ai	absorbed solar flux in snow/ocn/ice	W m ⁻²
f_albsni	snow/ice broad band albedo	%
f_aldvr	visible direct albedo	%
f_aldvi	near-infrared direct albedo	%
f_flat	latent heat flux sent to coupler	W m ⁻²
f_flat_ai	ice/atm latent heat flux	W m ⁻²
f_fsens	sensible heat flux sent to coupler	W m ⁻²
f_fsens_ai	ice/atm sensible heat flux	W m ⁻²
f_flwout	outgoing longwave flux sent to coupler	W m ⁻²
f_flwout_ai	ice/atm outgoing longwave flux	W m ⁻²
f_evap	evaporative water flux sent to coupler	cm day ⁻¹
f_evap_ai	ice/atm evaporative water flux	cm day ⁻¹
f_Tref	2 m reference temperature	C
f_Qref	2 m reference specific humidity	g/kg
f_congel	basal ice growth	cm day ⁻¹
f_frazil	frazil ice growth	cm day ⁻¹
f_snoice	snow-ice formation	cm day ⁻¹
f_meltb	basal ice melt	cm day ⁻¹
f_meltt	surface ice melt	cm day ⁻¹
f_meltl	lateral ice melt	cm day ⁻¹
f_fresh	ice/ocn fresh water flux sent to coupler	cm day ⁻¹
f_fresh_ai	ice/ocn fresh water flux	cm day ⁻¹
f_fsalt	ice to ocn salt flux sent to coupler	kg m ⁻² day ⁻¹
f_fsalt_ai	ice to ocn salt flux	kg m ⁻² day ⁻¹
f_fhnet	ice/ocn net heat flux sent to coupler	W m ⁻²
f_fhnet_ai	ice/ocn net heat flux	W m ⁻²
f_fswthru	SW transmitted through ice to ocean sent to coupler	W m ⁻²
f_fswthru_ai	SW transmitted through ice to ocean	W m ⁻²
f_strairx	zonal atm/ice stress	N m ⁻²
f_strairy	meridional atm/ice stress	N m ⁻²
f_strltlx	zonal sea surface tilt	m m ⁻¹
f_strltly	meridional sea surface tilt	m m ⁻¹
f_strcorx	zonal coriolis stress	N m ⁻²
f_strcory	meridional coriolis stress	N m ⁻²
f_strocnx	zonal ocean/ice stress	N m ⁻²

continued on next page

continued from previous page

<code>f_strocny</code>	meridional ocean/ice stress	N m^{-2}
<code>f_strintx</code>	zonal internal ice stress	N m^{-2}
<code>f_strinty</code>	meridional internal ice stress	N m^{-2}
<code>f_strength</code>	compressive ice strength	N m^{-1}
<code>f_divu</code>	velocity divergence	$\% \text{ day}^{-1}$
<code>f_shear</code>	strain rate	$\% \text{ day}^{-1}$
<code>f_opening</code>	lead opening rate	$\% \text{ day}^{-1}$
<code>f_sig1</code>	normalized principal stress component	
<code>f_sig2</code>	normalized principal stress component	
<code>f_daidtt</code>	area tendency due to thermodynamics	$\% \text{ day}^{-1}$
<code>f_daidtd</code>	area tendency due to dynamics	$\% \text{ day}^{-1}$
<code>f_dvidtt</code>	ice volume tendency due to thermo.	cm day^{-1}
<code>f_dvidtd</code>	ice volume tendency due to dynamics	cm day^{-1}
<code>f_mlt_onset</code>	melt onset date	
<code>f_frz_onset</code>	freeze onset date	

9 Troubleshooting

9.1 Code does not Compile or Run

Check the `ice.log.*` file in the executable directory, or the standard output and error files for information. Also, try the following:

- Delete the executable directory and rebuild the model.
- Make sure that there is a `Macros.<OS>` file for your platform. Modify the directory paths for the libraries.
- Make sure all paths and file names are set correctly in the scripts.
- If changes were made to the `ice_model_size.F` file in the source code directory, they will be overwritten by the file in `input_templates`.

9.2 Negative Ice Area in Horizontal Remapping

This error is written from `ice_transport_remap.F` when the ice model is checking for negative ice areas. If it happens well into a model integration, it can be indicative of a CFL violation. The output looks like:

```
60: New area < 0, istep = 119588
60: (my_task,i,j,n) = 4 21 380 1
60: Old area = 0.960675000975677174E-05
60: New area = -0.161808948357841311E-06
60: Net flux = -0.976855895811461324E-05
60:(shr_sys_abort) ERROR: remap transport: negative area
60:(shr_sys_abort) WARNING: calling shr_mpi_abort() and stopping
60:(shr_mpi_abort):remap transport: negative area 0
```

The dynamics timestep should be reduced to integrate past this problem. Set

```
, ndyn_dt      = 2
```

in the namelist and restart the model. When the job completes set the value back to 1.

9.3 Conservation Error

This error is written from `ice_itd.F` when the ice model is checking that initial and final values of a conserved field are equal to within a small value. The output looks like:

```
Conservation error: vice, add_new_ice
11 : 14 185
Initial value = 1362442.600400560
Final value = 1362442.600400561
Difference = 2.328306436538696D-10
(shr_sys_abort) ERROR: ice: Conservation error
(shr_sys_abort) WARNING: calling shr_mpi_abort() and stopping
(shr_mpi_abort):ice: Conservation error 0
```

Non-conservation can occur if the ice model is receiving very bad forcing, and is not able to deal with it. This has occurred after a CFL violation in the ocean. The timestep in the ocean may be decreased to get around the problem.

9.4 MPDATA transport unstable

The ice model dynamics is subcycled `ndte` times per timestep so that the elastic waves are damped before the next timestep. When the value of `ndte` is too small, an instability occurs, the model execution stops, and an error message similar to this example is written out to the log file:

```
mpdata phi = -0.122707896960329384E-05 n = 5
(shr_sys_abort) ERROR: (mpdata) transport unstable
(shr_sys_abort) WARNING: calling mpi_abort() and stopping
```

To get around this problem, increase the value of `ndte` in the namelist. Reasonable values of this parameter are discussed in section 4.2. The model run can be restarted from the last set of restart files that were written before the execution halted.

9.5 NX does not divide evenly into grid

If you modify the number of tasks used by the ice model, the model may stop with this error written to the log file:

```
'ERROR: NX must divide evenly into grid,100,8'
```

The number of MPI processors used by the ice model must divide evenly into the grid dimensions. For example, running the ice model with 8 tasks on the `gx3v5` grid will result in an error, since 8 does not divide evenly into the 100 longitude points. To fix this error, change the value of `$NTASKS` for the uncoupled ice model in the main script. In this case, a value of 4 would work, and the task geometry would also have to be changed.

9.6 Enabling the Debugger

This section explains how to set some compiler options for debugging. For the coupled model, set `DEBUG` to `TRUE` in the `env_run` script. For the uncoupled ice model, set `setenv DEBUG TRUE` in `csim.setup.csh`. Before running the model, be sure to delete the object files so that the source code will be recompiled. If a core file is created, it will be in the executable directory. Use `dbx` to look at the core file. Useful information may also appear in the standard error and output files.

9.7 Unit 50 Error

Unit 50 is the restart file. If the path or file name in the **rpointer.ice** file is incorrect, or the restart file does not exist in the **/rest** directory below the executable directory, the model will stop with this message:

```
8:1525-014 The I/O operation on unit 50 cannot be completed because an errno value
of 2 (A file or directory in the path name does not exist.) was received
while opening the file. The program will stop.
```

Verify that the path and file name in the rpointer are correct and that the restart file exists in the correct location.

References

- Bitz, C. M., M. Holland, M. Eby and A. J. Weaver, 2001: Simulating the ice-thickness distribution in a coupled climate model. *J. Geophys. Res.*, **106**, 2441–2463.
- Bitz, C. M. and W. H. Lipscomb, 1999: An energy-conserving thermodynamic model of sea ice. *J. Geophys. Res.*, **104**, 15,669–15,677.
- Hibler, W. D., 1979: A dynamic thermodynamic sea ice model. *J. Phys. Oceanogr.*, **9**, 815–846.
- Hunke, E. C., 2001: Viscous-plastic sea ice dynamics with the evp model: Linearization issues. *J. Comp. Phys.*, **170**, 18–38.
- Hunke, E. C. and J. K. Dukowicz, 1997: An elastic-viscous-plastic model for sea ice dynamics. *J. Phys. Oceanogr.*, **27**, 1849–1867.
- Hunke, E. C. and J. K. Dukowicz, 2003: *The sea ice momentum equation in the free drift regime*. Los Alamos National Laboratory, Tech. Rep. LA-UR-03-2219.
- Hunke, E. C. and W. H. Lipscomb, 2002: *CICE: the Los Alamos sea ice model, documentation and software User's Manual*. T-3 Fluid Dynamics Group, Los Alamos National Laboratory, Tech. Rep. LACC-98-16 v.3.
- Hunke, E. C. and W. H. Lipscomb, 2004: *CICE: the Los Alamos Sea Ice Model, Documentation and Software User's Manual*. T-3 Fluid Dynamics Group, Los Alamos National Laboratory, Tech. Rep. LA-CC-98-16 v.3.1.
- Lipscomb, W. H., 2001: Remapping the thickness distribution in sea ice models. *J. Geophys. Res.*, **106**, 13,989–14,000.
- Lipscomb, W. H. and E. C. Hunke, 2004: Modeling sea ice transport using incremental remapping. *Mon. Wea. Rev.*, **in press**.
- Rothrock, D. A., 1975: The energetics of the plastic deformation of pack ice by ridging. *J. Geophys. Res.*, **80**, 4514–4519.
- Thorndike, A. S., D. S. Rothrock, G. A. Maykut and R. Colony, 1975: The thickness distribution of sea ice. *J. Geophys. Res.*, **80**, 4501–4513.