File: docu/description.txt

Description of Software for the

-------------------------------

Spectral Transform Shallow Water Model

--------------------------------------

Version 2.0

-----------

by

Ruediger Jakob

National Center for Atmospheric Research

Boulder, CO 80307-3000

August 1992

Table of Contents

-----------------

Section

1) Introduction

2) Overall Program Flow

3) Compile Time Parameters

4) Run Time Parameters

5) Input/Output Files

6) Compilation Guide

7) Execution Guide

References

Appendix

A) Example NAMELIST Input File

B) Calling Tree

C) File Format Description

D) Table of Reference Solutions

Section 1: Introduction

-----------------------

1.1) Purpose

This note describes the software for a global shallow water model

based on the spectral transform method. The particular

implementation is set up to provide the reference solutions to the

test cases proposed by Williamson et. al. [6].

The actual solutions are published in a Technical

Note by Jakob et. al. [3]. In addition this code can serve

as an educational tool for numerical studies of the shallow water

equations. A detailed description of the spectral transform method,

and a derivation of the equations used in this software, can be

found in the Technical Note by Hack and Jakob [2].

The Fortran source code of this model is about 10,000 lines long and

uses the NCAR graphics library, a Fast Fourier Transform library

routine, and the netCDF library for portability of real data reference

solutions.

1.2) How to get the software

The software is available via anonymous FTP from the machine

ftp.ucar.edu (IP address 128.117.64.4)

in directory

chammp/shallow

Please read the file 'README' in that directory for directions on

how to obtain needed library routines and for latest corrections

and changes. An electronic copy of the software description you are

reading is available in directory 'docu' as file 'description.txt'.

If you have difficulties accessing these files,

please contact the NCAR computer consulting office at

303-497-1278 (email: consult1@ncar.ucar.edu).

Software bugs, along with suggested fixes, should be reported to

the email address

stswm@ncar.ucar.edu

Messages to this address will be forwarded to the authors.

1.3) Relation to NCAR Community Climate Model

The spectral transform algorithm of this model follows closely the

NCAR Community Climate Model's handling of the dynamical part of

the primitive equations [5]. As in the full model, time history is

kept in gridpoint space, semi-implicit time differencing is

standard (with explicit time differencing as an option), and

momentum forcing terms can be included.

1.4) Acknowledgments

This code has evolved from a spectral shallow water model written

by J.J. Hack (NCAR/CGD) to compare numerical schemes designed to

solve the divergent barotropic equations in spherical geometry [1].

The major change to the original code is that time history of the

state information is now carried in gridpoint space as opposed

to spectral space. Analysis and graphics routines have all been

generalized and initialization and analysis code for the test cases

in Williamson et. al. [6] have been included.

Special thanks to Jim Hack and David Williamson who helped in

the design and testing of this code.

The normal mode initialization software used to generate the

initial conditions for test case 7 was obtained from

Andy van Tuyle and Ron Errico. It is not distributed with this code.

I also want to acknowledge the debugging help of the Scientific

Computing Division's consulting group.

This work was funded by the Department of Energy's "Computer Hardware,

Advanced Mathematics, Model Physics" research program as part of the

U.S. Global Change Research Program.

Section 2: Overall Program Flow

-------------------------------

For the default model setup, the control flow in the main program

STSWM is as follows:

The main program calls INPUT to set up arrays for the spectral

transformations and read the NAMELIST model parameters from the

standard input file (Section 4). Then routine INIT is called to

set up the steady state analytic solution for the test case 2

(parameter ICOND).

The routine ANLYTC is then called to copy this

analytic solution into the initial model state.

This initial condition is plotted by the call to PLOTS,

then an initial conservation analysis is done by NRGTCS,

followed by an initial error analysis by ERRANL. The spectral

coefficients of the initial state are computed by SHTRNS and ZD.

If the NAMELIST parameter TAUO is set less than or equal TAUE,

the initial spectral coefficients are written to a file by OUTPTP.

Similarly, if the NAMELIST parameter SPCFRQ is set less than

or equal TAUE, a spectral analysis is done by SPCANL.

The main loop of the program begins at label 500, where NSTEP

is incremented for each time step, and TAU contains the model

time in hours. Subroutine STEP is responsible for managing the

circular index into the three time levels that are kept by the

model. STEP in turn calls COMP1 which advances the model state

one time step. If requested by the NAMELIST parameters

EQYFRQ, SPCFRQ, ERRFRQ, GPHFRQ or TAUO, analysis and output

subroutines are called at the specified model time intervals.

NAMELIST parameter EQYFRQ specifies the time intervals between

conservation analysis in routine NRGTCS, parameter SPCFRQ the time

interval between spectral analysis in routine SPCANL, parameter

ERRFRQ the time interval between error analysis in routine ERRANL,

parameter GPHFRQ the time interval between 2D plots of the model

state in routine plots, and parameter TAUO the time interval

between output of spectral coefficients in routine OUTSPC.

This main loop is continued until the model time TAU is larger

than the model duration (parameter TAUE) of 120 hours.

Finally, the time series of the conservation analysis and

error analysis are plotted with calls to NRGTCS and ERRANL.

The overall control flow of the program can be derived from the

calling tree in Appendix B of this description.

Section 3: Compile Time Parameters

----------------------------------

3.1) Changing spectral resolution

Changes in the spectral resolution of the model require

a recompilation. The truncation parameters are defined

and explained in the INCLUDE file 'PARAMS.i'. A selection

of common triangular truncation choices is given in comment

lines (Figure 1). To change truncation, uncomment the new

truncation, comment the current truncation, and recompile.

Figure 1. Common spectral truncation choices in file 'params.i'

C

C COMMON TRIANGULAR TRUNCATION CASES (MM=NN=KK):

C T-42 (default), T-63, T-106

C

PARAMETER (MM=42, NN=42, KK=42, NLAT=64, NLON=128)

C PARAMETER (MM=63, NN=63, KK=63, NLAT=96, NLON=192)

C PARAMETER (MM=106, NN=106, KK=106, NLAT=160, NLON=320)

C PARAMETER (MM=126, NN=126, KK=126, NLAT=192, NLON=384)

C PARAMETER (MM=213, NN=213, KK=213, NLAT=320, NLON=640)

C

The model requires NLAT to be even to take advantage

of the hemispherical symmetry of the spherical basis functions.

Also, NLON must have a factorization with only 2,3 and 5 as

prime factors to take advantage of the efficient ECMWF

Fast Fourier Transform library.

As described by [2], the unaliased transform of quadratic

terms requires

NLON .GE. 3\*MM+1

and

NLAT .GE. (3\*KK+1)/2

for triangular truncation.

For high resolution runs where the analytic solutions are obtained

from a NetCDF file, the following two parameters must be changed

to prevent spectral truncation:

RTRUNC: truncation limit for output of spectral coefficient during

a high resolution reference run (Default: 42).

MAXH: truncation limit for input of spectral coefficients from

reference high resolution solutions (Default: 42).

3.2) Array size limits

For extreme parameter choices, some arrays may be dimensioned too

small. In this case, the model will terminate with an error message

that mentions the array size parameter that must be changed.

In file PARAMS.i:

NGRPHS: maximum number of data points for conservation analysis plots

(quotient TAUE/EQYFRQ) and error analysis plots (quotient

TAUE/ERRFRQ). (Default: 1100)

In file CPCNRC.f:

LRWK,LIWK: real and integer workspace size for NCAR graphics. See

documentation for NCAR Graphics, Version 3.00 for a

discussion of these array sizes. (Defaults: LRWK=4000,LIWK=40000)

Section 4: Run Time Parameters

------------------------------

The selection of most model parameters is done at run time from

an input file that contains NAMELIST variables. The only other

runtime parameter is the experiment number which is written to

standard output and shown on all graphical output. This experiment

number can be used to uniquely identify model runs. The four-digit

experiment number is defined by the Unix environment variable

EXPERIMENT, e.g. in the Bourne shell:

export EXPERIMENT

EXPERIMENT=0001

which allows its use by operating system commands for

filename identification.

There are four types of input parameters in the NAMELIST input file:

physical model parameters, experiment definition parameters,

plotting parameters, and filename parameters.

A sample copy of such an input file can be found in Appendix A of

this description.

1) Physical Model Parameters

ALPHA is the rotation angle \alpha as used in test cases

1, 2 and 3 of the test set paper [6]. (Default: PI/4.0 radian)

A, OMEGA, and GRAV are the radius, rotational velocity and

gravitational acceleration of the earth, initialized to the

standard values.

HDC is the horizontal diffusion coefficient K\_{4}; for a reference

see Section 2.i of [5]. For nonzero values, potential enstrophy

is no longer conserved. (Default: 0.0 m^4/s)

2) Experiment Definition Parameters

DT is the time step in seconds; the model uses a leapfrog time step

of 2\*DT. (Default: 2400.0 seconds)

EGYFRQ is the time interval in hours between conservation analysis

of the model state. The following invariants are checked: total mass,

total energy, relative vorticity, divergence, and

potential enstrophy. In addition a time series for a midlatitude

point is generated for gravity wave analysis. These are measured as

defined by the test set paper. To disable this analysis, set higher

than TAUE. (Default: 3.0 hours)

ERRFRQ is the time interval in hours between error analysis of the

model state. The L\_{1}, L\_{2} and L\_{\infty} errors of height and

vector velocity are computed as described in the test set paper.

In addition, relative errors of height minimum, maximum, average and

variance are computed. For the test cases for that no analytic solution

is known, a reference solution is generated from spectral coefficients

in a file. To disable this analysis, set higher than TAUE.

(Default: 24.0 hours)

SPCFRQ is the time interval in hours between spectral analysis

of the kinetic energy and geopotential of the model state. To disable

this analysis, set higher than TAUE. (Default: 999.0 hours)

TAUE is the length in hours of the experiment. (Default: 120.0 hours)

TAUO is the time interval in hours between output of the model

state as spectral coefficients to a file, to be used as a reference

solution. To disable spectral output, set higher than TAUE.

(Default: 999.0 hours)

GPHFRQ is the time interval in hours between 2D plots of the

model state, analytic solution and error. To disable these 2D plots,

set higher than TAUE. (Default: 24.0 hours)

AFC is the Asselin filter coefficient that can be used to prevent

modal splitting between even and odd time steps caused by the

leapfrog time stepping procedure. (Default: 0.0)

SITS is a logical flag that selects between semi-implicit (.TRUE.)

and explicit (.FALSE.) time stepping. Test case 1 (advection) must be

run with explicit time stepping. (Default: .TRUE.)

FORCED is a logical flag that enables external forcing if .TRUE.;

For case 4 forcing must be enabled. (Default: .FALSE.)

MOMEMT is a logical flag the selects between momentum forcing

(.TRUE.) and vorticity-divergence forcing (.FALSE.) for test case 4.

(Default: .FALSE.)

ICOND selects the test case to be executed; Currently, cases 1

through 7 have been defined, corresponding to the test cases in

Williamson et al. [6]. (Default: 2)

3) Plotting Parameters

LGPHS is a logical flag that enables plotting; when set to .FALSE.,

all graphical output is disabled. (Default: .TRUE.)

LCONT is a logical flag that adds continental outlines to the

2D plots when .TRUE. (Default: .TRUE.)

LOP, LCP, and LPSP select the map projection: exactly one of them

must be .TRUE.; LOP selects orthographic projection, LCP cylindrical

projection, and LPSP north/south polar stereographic projection.

(Defaults: LOP=.TRUE.,LCP=.FALSE.,LPSP=.FALSE.)

LG enables contour plots for the height field.

(Default: .TRUE.)

LU enables contour plots of the eastward (u) wind component.

(Default: .FALSE.)

LV enables contour plots of the northward (v) wind component.

(Default: .FALSE.)

LZ enables contour plots of the absolute vorticity.

(Default: .FALSE.)

LD enables contour plots of the divergence.

(Default: .FALSE.)

LVV enables vector plots of the wind field.

(Default: .TRUE.)

LVVG enables an overlay of height contours and wind vectors.

(Default: .FALSE.)

POLAT,POLNG,POROT defines the center latitude, center longitude

and rotation angle for plots.

(Defaults: POLAT=0.0,POLNG=0.0,POROT=0.0 degrees)

4) Filename Parameters

FNIN is the filename of the file containing the spectral

coefficients to be read in as a reference solution (see Section 5.2).

Default filename is 'VDGDATA.cdf'.

FNOUT is the filename of the file into which the spectral

coefficients of this model run are to be written (see Section 5.6).

Default filename is 'REFDATA.cdf'.

Section 5: Input/Output Files

-----------------------------

This section describes the up to six different files that are used

by the shallow water model. For an example on how these files are

used in a Unix environment, see Section 7.

1) Namelist Input File

The NAMELIST input file is extensively described in Section 4.

This input file is expected from the standard input unit.

For test cases 1 through 7 of Williamson et al. [6],

Namelist input files are provided in the directories

'test1' through 'test7'.

2) Input File with Spectral Coefficients

For test cases 5,6, and 7 of Williamson et al. [6] the analytic

solution is defined by spectral coefficients that have been

previously computed.

Appendix D of this description contains a table of such reference

solutions, which are located in subdirectory /netcdf.

The files are in netCDF format [4], which is described in Appendix C.

The default input filename is 'VDGDATA.cdf'. This filename must

be changed to point to the desired reference solution, using the

parameter FNIN of the NAMELIST input file (see Section 4.4).

3) Standard Output

The following information is written to standard output (Unit 6).

First, a copy of the NAMELIST input file with the run time parameters.

Next, the spectral truncation parameters as defined by the compile

time parameters in the INCLUDE file 'PARAMS.i'.

If requested by the NAMELIST parameters EGYFRQ, ERRFRQ, or SPCFRQ

the model state is analyzed at the specified time intervals,

including the initial model state, and the results are written

to standard output.

To disable this output, set the NAMELIST parameters higher than the

model length parameter TAUE.

The output from these analysis routines should be self-descriptive

and can be used for postprocessing.

4) Error Output

Certain conditions cause the shallow water model to terminate

with a fatal error. In most cases an error message is written to

the error output unit 0 before the STOP command.

Also, warning messages and comments are written to this file,

for example if the time step excced the CFL limit initially,

or if spectral coefficients were truncated during I/O to/from

NetCDF files.

5) NCAR Graphics Metafile

NCAR graphics produces a computer graphics metafile called 'gmeta'

by default. The name of this output file can be changed in a UNIX

environment by redefining the environment variable NCARG\_GKS\_OUTPUT:

export NCARG\_GKS\_OUTPUT

NCARG\_GKS\_OUTPUT=filename

Graphics output can be disabled by setting the NAMELIST variable

LGPHS to .FALSE. (see Section 4.3).

6) Output File with Spectral Coefficients

The state of the model can be output as spectral coefficients to

a file, which can later serve as a reference solution spectral

input file (see 2). The file format is netCDF format [4] and described

in Appendix C of this description. The default filename is 'REFDATA.cdf'.

Again, the filename can be changed using the parameter FNOUT of the

NAMELIST input file (see Section 4.4).

Section 6: Compilation Guide

----------------------------

This section contains some guidelines for compilation of the

model code in directory 'src'. This directory contains about 50

Fortran files and 8 Include files. The filenames are the same as

the subroutine or function defined in them.

1) Portability

With two exceptions, the model code is written in Fortran 77:

INCLUDE files and NAMELIST input parameters. The code has been

compiled and executed on Sun and IBM workstations and Cray

supercomputers.

INCLUDE files have been used for PARAMETER definitions, global

variable definitions in COMMON blocks and netCDF declarations.

Each INCLUDE file is itself commented, so the following list is

just an overview of the contents:

Filename Content

----------------------------------------------------------------------

params.i All global parameter definitions (see Section 3); this

file must be included before other INCLUDE files since

it contains needed array sizes.

complt.i Plotting parameters for routine PLOTS

consts.i Experiment definition and physical parameters

finit.i Initial conditions for test cases

case4.i Parameters for test case 4

tdvars.i Model state variables vorticity, divergence, geopotential,

wind field and time level index

trnsfm.i Spherical basis functions for transform grid<->spectral

wrkspc.i Workspace (mainly for FFTs and graphics output)

netcdf.inc Variable declarations for netCDF routines; this file

contains variables in lower case letters and variables

names that are longer than 6 characters.

Except for the netCDF INCLUDE file, these files are assumed to

reside in the same directory as the Fortran files. The netCDF

INCLUDE file is assumed to be located in directory '/usr/local/include'.

If this is not the case, modify the variable CDFINC in the makefile.

NetCDF routines are called from the routines INPTP, OUTPTP, PRNT

and INFLD.

A NAMELIST file is used for input of the run time parameters to the

model in routine INPUT. For a description, see Section 4.

Another non-portable feature is the use of the GETENV Unix library

function to access environment variables. The model uses this function

in subroutine INPUT to access the experiment number. If this

environment variable is undefined, experiment number 0000 is chosen.

2) Library Subroutines

Please read the file 'README' for directions on how to obtain

the library routines used by the model. Only the FFT routine is

essential for model execution. It is possible to run the model

without netCDF software, if no input/output of spectral coefficients

is needed. Test cases 1-4 should work without this feature. It is

also possible to run the model without NCAR graphics software

(Set NAMELIST parameter LGPHS = .FALSE.). In any case, the analysis

routines write their values in ASCII form to standard output for

postprocessing. The integration routine D01AHE is only used by test

cases 3 and 4.

3) Makefile

Directory 'src' also contains a file 'makefile' for compilation of the

code in UNIX environments. Currently, the makefile is set up for Sun

Workstations, IBM RS 6000 and Cray Y-MP. Use the

make -e MACHINE=sun

make -e MACHINE=ibm

make -e MACHINE=cray

commands respectively. Alternatively, you can change the 'MACHINE'

variable in the makefile.

The 'make' command will compile each source file separately, link

them with the required libraries and create an executable file 'stswm'.

The current setup is to use the NCAR graphics and NetCDF libraries,

but replace the NAG integration routine and ECMWF FFT library with

the provided source code. If you have access to the NAG and ECMWF

libraries, change the makefile variables D01AHE and FFT accordingly.

If you do not have access to the NCAR graphics and netCDF libraries,

change the makefile variables GRAPHICS and NETCDF accordingly.

The makefile also contains the dependencies on the INCLUDE files so

that changes in the INCLUDE files automatically cause a recompilation

of the affected object modules.

On other machines than Sun, IBM or Cray please pay attention

that the NCAR graphics library routines AGPWRT, CPCNRC, CPMPXY and VELVCT

are replaced by the modified routines when invoking the

linker/loader.

Section 7: Execution Guide

--------------------------

1) Execution in a UNIX Environment

Section 5 describes which I/O files are used in this model.

For test cases 1 to 4 of Williamson et al. [6], only the NAMELIST

input file is required from the standard input unit:

stswm < exp.data

Because of the voluminous output, it is recommended that standard

output be redirected to a file:

stswm < exp.data > stdout

For test cases 5 to 7 of Williamson et al. [6], the analytic solution

is computed from spectral coefficients in a netCDF file.

The filename for this file is defined by the NAMELIST parameter FNIN.

The default filename is 'VDGDATA.cdf'. See Appendix D for a table of

references solutions that are provided in directories 'test5' through

'test7'. It is also possible to write the computed spectral coefficients

to a netCDF file for later analysis or comparison. The filename

for this file is defined by the NAMELIST parameter FNOUT. The

default filename is 'REFDATA.cdf'.

The environment variables NCARG\_GKS\_OUTPUT (Section 5.5) and

EXPERIMENT (Section 4) can be set to change the default filename

'gmeta' for NCAR graphics output and to change the default

experiment number '0000'.

The directories 'test1' through 'test7' contain README files,

example NAMELIST input files and Sun and Cray

job scripts. For test purposes, each directory contains output from

execution on a Sun workstation. It is recommended that tests 1

through 7 are run in sequence and results compared for general consistency.

2) Approximate Memory Requirements

On a Cray Y-MP/8, the model code uses about 350 Kwords = 2.8 MByte

A T-42 model run has a maximum memory usage of 800 Kwords = 6.4 MByte

3) Approximate Run Time

On a single processor Cray Y-MP/8, a 5 day model run without any

analysis or graphics output at T-42 resolution takes about

10 sec, executing at a computational rate of about 100 MFLOPS.

References

----------

1) Browning, G.L., J.J. Hack and P.N. Swarztrauber, A comparison of

three numerical methods for solving differential equations on

the sphere, Monthly Weather Review, 117:1058-1075, 1989.

2) Hack, J.J. and R. Jakob, Description of a global

shallow water model based on the spectral transform method,

NCAR Technical Note TN-343+STR, January 1992.

3) Jakob, R., J.J. Hack and D.L. Williamson, Reference solutions to

shallow water test set using the spectral transform method,

NCAR Technical Note, in Preparation.

4) NetCDF User's Guide, Version 2.0, NCAR Technical Note TN-334+IA,

November 1991.

5) Williamson, D.J., J.T. Kiehl, V. Ramanathan, R.E. Dickinson and

J.J. Hack, Description of NCAR community climate model (CCM1),

NCAR Technical Note TN-285+STR, June 1987.

6) Williamson, D.L., J.B. Drake, J.J. Hack, R. Jakob and P.S. Swarztrauber,

A standard test set for numerical approximations to the shallow

water equations in spherical geometry, Journal of Computational Physics,

102:211-224, 1992.

Appendix A: Example NAMELIST Input File

---------------------------------------

Input file for spectral shallow water model STSWM

(read as NAMELIST by routine INPUT from standard input unit)

Physical model parameters

E$PHYVAR

ALPHA = 0.78539816,; rotation angle (radian)

A = 6.37122E6, ; radius of earth (meter)

OMEGA = 7.292E-5, ; angular velocity of earth (radian/second)

GRAV = 9.80616, ; gravitational acceleration (meter/second^2)

HDC = 0.0E00, ; horizontal diffusion coefficient (meter^4/second)

$END

Experiment definition parameters

E$EXPDEF

DT = 2400.0, ; time step (second) of simulation

EGYFRQ = 3.0, ; time interval (hour) of conservation analysis

ERRFRQ = 24.0, ; time interval (hour) of L-error analysis

SPCFRQ = 999.0, ; time interval (hour) of spectral analysis

TAUE = 120.0, ; model termination time (hour)

TAUO = 999.0, ; time interval (hour) of spectral output to file

GPHFRQ = 24.0, ; time interval (hour) of plots

AFC = 0.0, ; Asselin filter coefficient

SITS = .TRUE., ; Flag for semi-implicit time stepping

FORCED = .FALSE., ; Flag for external forcing (case 4)

MOMENT = .FALSE., ; Flag for momentum / vorticity-divergence forcing

ICOND = 2, ; Test case #

$END

Plotting parameters

E$PLTDEF

LGPHS = .TRUE., ; Flag to enable plots

LCONT = .TRUE., ; Flag for continental outlines

LOP = .TRUE., ; Flag for orthographic projection

LCP = .FALSE., ; Flag for cylindrical projection

LPSP = .FALSE., ; Flag for polar stereographic projection

LG = .TRUE., ; Flag for height field plot

LU = .FALSE., ; Flag for u-wind field plot

LV = .FALSE., ; Flag for v-wind field plot

LZ = .FALSE., ; Flag for vorticity plot

LD = .FALSE., ; Flag for divergence plot

LVV = .FALSE., ; Flag for vector wind field plot

LVVG = .FALSE., ; Flag for height&vector wind plot

POLAT = 0.0, ; Center latitude for plots (degree)

POLNG = 0.0, ; Center longitude for plots (degree)

POROT = 0.0, ; Rotation angle for plots (degree)

$END

Input/Output filenames

E$FNAMES

FNIN = 'VDGDATA.cdf', ; Filename for input of spectral coefficients

FNOUT = 'REFDATA.cdf', ; Filename for output of spectral coefficients

$END

Appendix B: Calling Tree

------------------------

Following is the commented calling tree for the main program 'STSWM':

(some leaf routines have been deleted to improve readability)

STSWM-+ \* main program

|

+-INPUT-+-EPSLON \* determine machine epsilon

| |

| +-(GETENV) \* read EXPERIMENT environment variable

| |

| +-GLATS--ORDLEG \* compute Gaussian latitudes and weights

| |

| +-CEPS \* compute coefficient arrays for assoc. Leg. functions

| |

| +-CALP \* compute associated Legendre functions

| |

| +-(SET99) \* setup for Fast Fourier Transform

|

+-(OPNGKS) \* setup for NCAR graphics library

|

+-ORTHO \* orthogonality/orthonormality check of basis

|

+-INIT-+ \* setup for analytic solutions for test cases (steady part)

| |

| +-(D01AHE) \* NAG integration routine for balanced geopotential

| |

| +-ROTATE \* rotational coordinate transformation

| |

| +-ZD (4)-+ \* compute vorticity and divergence spectral coeff.

| | |

| | +-(FFT991) \* Fast Fourier transform

| |

| +-SHTRNS (5)--(FFT991) \* inverse Legendre and Fourier

| transform to grid values

|

+-ANLYTC (6)-+ \* compute analytic solution at given time

| |

| +-ROTATE \* rotational coordinate transformation

| |

| +-INPTP \* read reference solution from file

| | |

| | +-INFLD \* read spectral coefficients

| |

| +-EVAL-+ \* evaluation at arbitrary point on sphere

| |

| +-DCEPS \* coefficients for assoc. Leg. functions

| |

| +-DFTFAX \* compute Fourier basis for

| | arbitrary longitude

| |

| +-DCALP \* compute assoc. Leg. functions for

| | arbitrary latitude

| |

| +-DTRNS--DFT991 \* evaluate vorticity,

| | divergence, geopotential

| |

| +-DUV--DFT991 \* evaluate u,v wind field

|

+-PLOTS (7)-+ \* plots of computed, analytic or error fields

| |

| +-CPCNRC (1) \* contour line plots

| |

| +-VELVCT \* vector plots for wind vector

|

+-NRGTCS-+ \* Conservation analysis

| |

| +-(EZXY) \* time series plot

|

+-ERRANL-+ \* L\_1,L\_2,L\_\infty error analysis

| |

| +-(EZXY) \* time series plot

| |

| +-ANLYTC see 6 \* get analytic solution

| |

| +-PLOTS see 7 \* 2D plots of analytic solution and error

|

+-ZD see 4 \* compute vorticity-divergence spectral coeff.

| for initial kinetic energy spectrum plots

|

+-SHTRNS see 5 \* compute geopotential spectral coeff.

| for initial height spectrum plots

|

+-OUTPTP-+ \* write model spectral state to file

| |

| +-PRNT \* write one spectral field to file

|

+-SPCANL-+ \* spectral analysis of kinetic energy and geopotential

| |

| +-(EZXY) \* wavenumber-kinetic/potential energy plots

|

+-STEP-+ \* advances model one time step (handles 3 time-level buffer)

| | and smooth start

| |

| +-COMP1-+ \* selects time-stepping algorithm + spectral

| | diffusion

| |

| +-FTRNIM-+ \* semi-implicit time stepping;

| | | compute nonlinear products

| | |

| | +-FORCE (8) \* compute forcing terms (case 4)

| | |

| | +-(FFT991) \* FFT of nonlinear products

| | |

| | +-FTRNVE \* vorticity equation

| | |

| | +-FTRNDI \* divergence equation (semi-impl.)

| | |

| | +-FTRNPI \* geopotential equation (semi-impl.)

| |

| +-ADVECT-+ \* advection equation;

| | | compute nonlinear products

| | |

| | +-FORCE see 8 \* optional forcing

| | |

| | +-(FFT991) \* FFT of nonlinear products

| | |

| | +-FTRNPE \* geopotential equation (expl.)

| | |

| | +-SHTRNS see 5 \* inverse transform geopotential

| |

| +-FTRNEX-+ \* explicit time stepping;

| | | compute nonlinear products

| | |

| | +-FORCE see 8 \* compute forcing terms (case 4)

| | |

| | +-(FFT991) \* FFT of nonlinear products

| | |

| | +-FTRNVE \* vorticity equation

| | |

| | +-FTRNDE \* divergence equation (expl.)

| | |

| | +-FTRNPE \* geopotential equation (expl.)

| |

| +-UV--(FFT991) \* inverse spherical transform to

| | get u,v winds

| |

| +-SHTRNS see 5 \* inverse spherical transforms of

| height, vorticity, divergence

|

+-(CLSGKS) \* close NCAR graphics file

The following subroutines and functions are indirectly called:

AGPWRT \* used for high quality character labels IN EZXY

CPMPXY \* irregular grid remapping function used in CPCNRC and VELVCT

CPMVXY \* irregular grid remapping function used in VELVCT

DRAWCL \* used to draw contour lines in CPCNRC

FU--US \* function to be integrated by routine D01AHE for test

case 3 in routine INIT

FUNC2--BUBFNC \* function to be integrated by routine D01AHE for test

case 4 in routine INIT

SHADER \* draws diagonal lines for negative values in CPCNRC

Appendix C: File Format Description

-----------------------------------

The spectral coefficients are stored in netCDF [4] format for

wide portability. Please pay attention that netCDF files are binary

files when copying or using FTP.

NetCDF uses a netCDF description language (CDL)

to define the file format. Below is a copy of a CDL description

of the format used in this code, generated by using the

'ncdump -h' command:

netcdf VDGDATA {

dimensions:

complex = 2 ;

m-wave+1 = 107 ;

n-wave+1 = 107 ;

timestep = UNLIMITED ; // (1 currently)

variables:

long testcase ;

float rot\_angle ;

rot\_angle:units = "radian" ;

float time(timestep) ;

time:units = "hour" ;

long mm ;

long nn ;

long kk ;

float vorticity(timestep, n-wave+1, m-wave+1, complex) ;

vorticity:units = "second-1" ;

float divergence(timestep, n-wave+1, m-wave+1, complex) ;

divergence:units = "second-1" ;

float geopotential(timestep, n-wave+1, m-wave+1, complex) ;

geopotential:units = "meter2 second-2" ;

}

Appendix D: Table of Reference Solutions

----------------------------------------

The following table lists the reference solutions that are provided with

the software. The files are in netCDF format (see Appendix C) and therefore

must be accessed in binary mode when using FTP.

It is important that the following NAMELIST input parameters

match the ones that were chosen for the reference run:

ICOND = ... Test case numbers must be identical

ERRFRQ = ... Must be a multiple of output interval TAUO=24.0 hours

of the reference run

ALPHA = 0.0 Rotation angle must be identical

The following parameter should also be checked:

MAXH = 106 Spectral truncation for input coefficients should match

spectral truncation T-106 of reference runs. If MAXH is

smaller, the data will be spectrally truncated.

The initial conditions for the observed heigth and wind fields

were obtained from ECMWF analysis, using Kevin Trenberth's CCM

Processor dataset, which is available on a T-42 grid. However,

the mean height of the fluid was changed to 10 km.

For the high resolution reference solutions of test case 7

the initial conditions were obtained via nonlinear normal

mode initialization. In all files only the spectral coefficients

corresponding to T-106 trunctaion have been stored, although the

model runs were done at higher resolution.

The following files are located in subdirectory /netcdf:

Case | Filename | Description

-----+-------------+---------------------------------------------------------

5 | REF0114.cdf | Zonal flow over an isolated mountain: Reference Solution

| | Model Truncation: T-213

| | Time step: 360 s

| | Diffusion: 8.0E12 m^4/s

| | Length of run: 360 hours (15 days)

| | Output interval: 24 hours

-----+-------------+---------------------------------------------------------

6 | REF0092.cdf | Rossby-Haurwitz Wave: Reference Solution

| | Model Truncation: T-213

| | Time step: 180 s

| | Diffusion: 8.0E12 m^4/s

| | Length of run: 336 hours (14 days)

| | Output interval: 24 hours

-----+-------------+---------------------------------------------------------

7 | INI0077.cdf | December 21, 1978: Initial Data

| | (before Normal Mode Initialization)

-----+-------------+---------------------------------------------------------

7 | REF0077.cdf | December 21, 1978: Reference Solution

| | Model Truncation: T-213

| | Time step: 360 s

| | Diffusion: 8.0E12 m^4/s

| | Length of run: 120 hours (5 days)

| | Output interval: 24 hours

-----+-------------+---------------------------------------------------------

7 | INI0087.cdf | January 9, 1979: Initial Data

| | (before Normal Mode Initialization)

-----+-------------+---------------------------------------------------------

7 | REF0087.cdf | January 9, 1979: Reference Solution

| | Model Truncation: T-213

| | Time step: 360 s

| | Diffusion: 8.0E12 m^4/s

| | Length of run: 120 hours (5 days)

| | Output interval: 24 hours

-----+-------------+---------------------------------------------------------

7 | INI0088.cdf | January 16, 1979: Initial Data

| | (before Normal Mode Initialization)

-----+-------------+---------------------------------------------------------

7 | REF0088.cdf | January 16, 1979: Reference Solution

| | Model Truncation: T-213

| | Time step: 360 s

| | Diffusion: 8.0E12 m^4/s

| | Length of run: 120 hours (5 days)

| | Output interval: 24 hours

-----------------------------------------------------------------------------