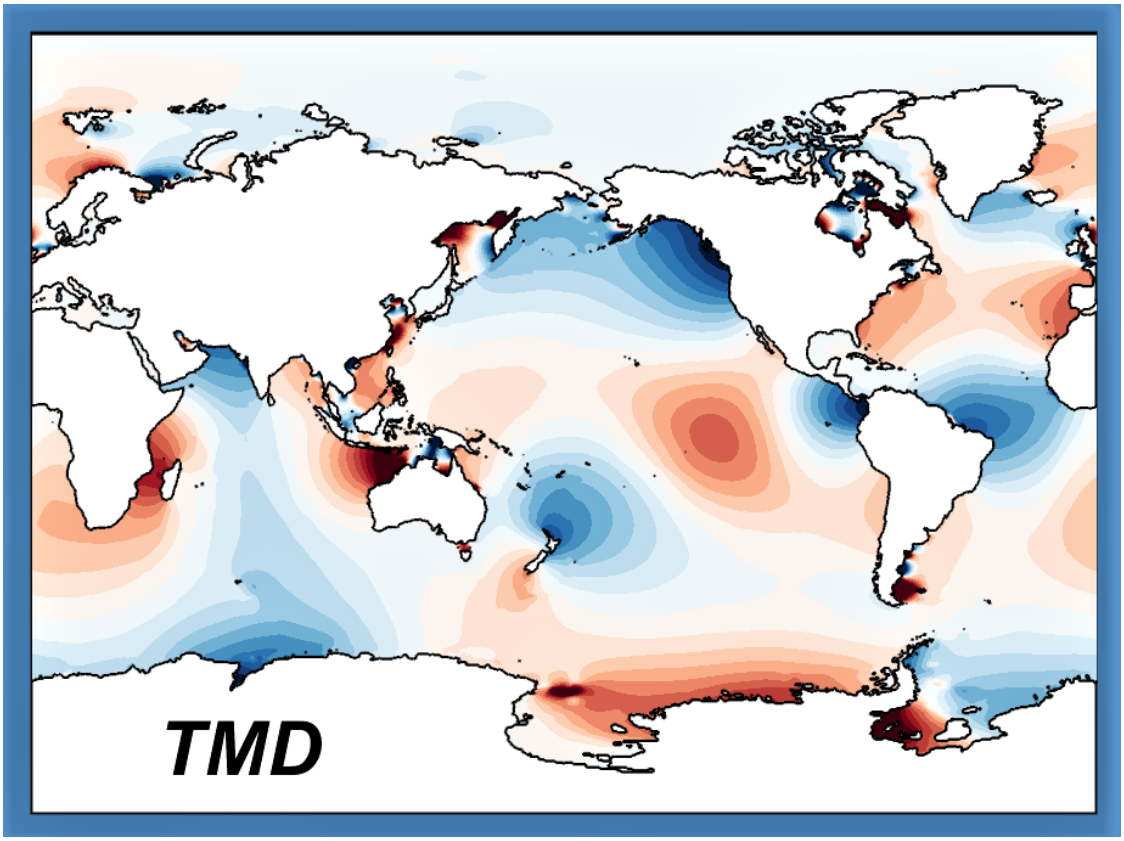
Tide Model Driver (TMD) Manual: Version 2.5

Laurie Padman
Earth & Space Research
padman@esr.org

Version 2.5: October 14, 2015

Modified: 7 April 2020 (LP), 17 Jan 2022 (SH & LP)



User Manual for Tide Model Driver (TMD): version 2.5

Report problems to ***padman@esr.org*** and ***showard@esr.org***.

TMD is a MATLAB toolbox for accessing the harmonic constituents for tide models created by Oregon State University (OSU) and Earth and Space Research (ESR), and for making predictions of tide height and currents. TMD includes two components: (1) a set of scripts for accessing tidal fields and making predictions; and (2) a graphical user interface (GUI)[1] for quickly browsing tide fields, zooming in on regions of interest, and selecting points and time ranges for predictions of specific variables. Each component is described below.

TMD was written by Lana Erofeeva at OSU and is maintained by Laurie Padman and Susan Howard at ESR.

TMD works with Arctic and Antarctic models available from ESR's web page (https://www.esr.org/research/polar-tide-models/), and global and regional models provided by OSU (https://www.tpxo.net/) if they are served in 'bin' format with all constituents packed into a single file.

Coordinate systems: Tide models that work with TMD are provided in two coordinate systems: (1) uniform in latitude and longitude; and (2) uniform in polar stereographic ("PS") *x* and *y* distances (in km). The PS models are for polar applications. In these models, the file named 'Model_*' contains a fourth line that defines the script used to convert between (*x*,*y*) and (*lat*,*lon*). In most scripts, e.g., tmd_tide_pred.m (see below), the user still specifies latitude and longitude, not (*x*,*y*). However, when extracting an entire grid (e.g., tmd_get_bathy.m (see below)), the returned coordinates with be vectors of *x* and *y*. If you wish to view this grid in latitude and longitude, you will need to convert the (*x*,*y*) grid using the script specified in the Model_* file. When you do this, first turn the vectors into 2D grids using meshgrid:

```
[x2d,y2d]=meshgrid(x,y);
```

1. TMD Scripts

The TMD package includes MATLAB functions that can be used to access the model directly, for example where the user wishes to run several predictions in batch mode. Please let us know if the explanations of usage are not clear, or another script would be useful (email Laurie Padman at padman@esr.org). To the experienced user, the scripts are probably much more useful than the GUI (see section 2, below).

[1] The GUI is not maintained: its performance was sensitive to Operating System and MATLAB version. It may work, but otherwise use scripts.

Hint: Add the location of each model you want to the MATLAB default path (use the 'File' 'Set Path' option on the command bar). This allows you just to specify the model name rather than the whole path in the calls to the script functions. We add the model directory to the end of the current 'pathdef'. However, it probably does not matter since you are unlikely to have similarly named files on your system.

For recent versions of MATLAB, script names are case-sensitive. All script names in the TMD toolbox are entirely lower-case. The available functions are as follows. `Courier script refers to printout of "help *.m" commands.`

tmd_exerciser.m

This script runs through a few of the TMD scripts, demonstrating model calls and basic plotting.

tmd_get_bathy.m

This function retrieves *x*, *y* and water depth (which is replaced by 'water column thickness' under ice shelves) for the specified model.

```
>> help tmd_get_bathy
================================================================================
  tmd_get_bathy.m

  Gets map of bathymetry (water column thickness under ice shelves) for
    specified model.

  Written by:   Laurie Padman (ESR): padman@esr.org
                August 18, 2004

  Sample call:
               [long,latg,H]=tmd_get_bathy('Model_ISPOL');

 ================================================================================
  TMD release 2.02: 21 July 2010
```

Note that, for models coded on a polar stereographic coordinate system (Arctic and Antarctic regional models), the sample call would be

```
[x,y,H]=tmd_get_bathy(Model_name);
```

and *x* and *y* are vectors in units of km.

tmd_extract_hc.m

This function is designed to extract the tidal harmonic constants for a specified location. Usage is explained below:

```
>> help tmd_extract_hc

Function to extract tidal harmonic constants out of a tidal model
```

```
  for given locations
  USAGE
  [amp,Gph,Depth,conList]=tmd_extract_HC(Model,lat,lon,type,Cid);

  PARAMETERS
  Input:
  Model - control file name for a tidal model, consisting of lines
          <elevation file name>
          <transport file name>
          <grid file name>
          <function to convert lat,lon to x,y>
  4th line is given only for models on cartesian grid (in km)
  All model files should be provided in OTIS format
              lat(L),lon(L) or lat(N,M), lon(N,M) - coordinates in degrees;
              type - char*1 - one of
                     'z' - elvation (m)
                     'u','v' - velocities (cm/s)
                     'U','V' - transports (m^2/s);
  Cid - indices of constituents to include (<=nc); if given
              then included constituents are: ConList(Cid,:),
              if Cid=[] (or not given),
              ALL model constituents included

  Output:
             amp(nc0,L) or amp(nc0,N,M) - amplitude
             Gph(nc0,L) or Gph(nc0,N,M) - Greenwich phase (o)
             Depth(L) or Depth(N,M)  - model depth at lat,lon
             conList(nc,4) - constituent list
             if Cid==[], L=nc, else L=length(Cid); end

  Sample call:

[amp,Gph,Depth,conList]=tmd_extract_HC('DATA/Model_Ross_prior',lat,lon,'z');

  Dependencies:  h_in,u_in,grd_in,XY,rd_con,BLinterp,checkTypeName

  TMD release 2.02: 21 July 2010
```

Note that, for polar stereographic models, the user still specifies “lat, lon”. The fourth line of a polar stereographic Model file (‘Model_*’) defines the script for converting between (*lat*,*lon*) and (*x*,*y*).

tmd_tide_pred.m

tmd_tide_pred.m makes predictions of a specified tidal variable (height, u, v, U, V) at a specified location. The user can also specify which tidal harmonics to include. Predictions are made with nodal corrections included.

>> help tmd_tide_pred

```
%% Predict tidal time series in a given locations at given times
 %%% using tidal model from a file
  USAGE:
```

```
  [TS,ConList]=tmd_tide_pred(Model,SDtime,lat,lon,ptype,Cid);

  PARAMETERS
  Input:
  Model - control file name for a tidal model, consisting of lines
          <elevation file name>
          <transport file name>
          <grid file name>
          <function to convert lat,lon to x,y>
  4th line is given only for models on cartesian grid (in km)
  All model files should be provided in OTIS format
  SDtime  - vector of times expressed in serial days:
              see 'help datenum' in matlab

  lat,lon  - coordinates in degrees;
  Depending on size of SDtime,lat,lon 3 functional modes possible:

            "Time series": SDtime(N,1),lon(1,1),lat(1,1)
            "Drift Track": SDtime(N,1),lon(N,1),lat(N,1)
            "Map"        : SDtime(1,1),lon(N,M),lat(N,M)

  ptype - char*1 - one of
             'z' - elevation (m)
             'u','v' - velocities (cm/s)
             'U','V' - transports (m^2/s);
  Cid - indices of consituents to include (<=nc); if given
              then included constituents are: conList(Cid,:),
              NO minor constituents inferred;
              if Cid=[] (or not given), ALL model constituents
              included, minor constituents inferred if possible

  Ouput:      TS(N) or TS(N,M) - predicted time series or map
              conList(nc,4) - list of ALL model
                              constituents (char*4)

  Dependencies: 'Fxy_ll',h_in,u_in,grd_in,XY,rd_con,BLinterp, tmd_extract_HC
                harp1,constit,nodal,checkTypeName

  Sample calls:

  SDtime=[floor(datenum(now)):1/24:floor(datenum(now))+14];
  [z,conList]=tmd_tide_pred('DATA/Model_Ross_prior',SDtime,-73,186,'z');
  ConList([5,6])=
  k1
  o1
  [z1,conList]=tmd_tide_pred('DATA/Model_Ross_prior',SDtime,-73, 186, 'z',
[5,6]);

   TMD release 2.02: 21 July 2010
```

Users should note the three “modes” for prediction: “time series”, “track” and “map”. The general rule when writing a script that uses tmd_tide_pred.m is to minimize the total number of calls to this script: most time is spent loading grids rather than carrying out calculations.

tmd_ellipse.m

tmd_ellipse.m calculates the tidal ellipse parameters (major and minor semi-axes, ellipse phase, and ellipse inclination) for a specified tidal harmonic at a specified location. This is useful, for example, for comparing the model to tidal ellipse analyses of current meter data.

```
>> help tmd_ellipse

Calculate tidal ellipse parameters at given locations using a model

  USAGE
  [umajor,uminor,uphase,uincl]=tmd_ellipse(Model,lat,lon,constit);

  PARAMETERS

  INPUT
  Model - control file name for a tidal model, consisting of lines
          <elevation file name>
          <transport file name>
          <grid file name>
          <function to convert lat,lon to x,y>
  4th line is given only for models on cartesian grid (in km)
  All model files should be provided in OTIS format

  lat(L),lon(L) - coordinates (degrees) -> outputs 1D arrays
  OR lat(n,m), LON(n,m) - could be 2D arrays -> then outputs are 2D arrays

  constit - constituent name, char length <=4

  OUTPUT
  umajor,uminor,uphase,uincl - tidal ellipse parameters (cm/s,o) in
                               lat,lon

  Dependencies: u_in,grd_in,XY,rd_con,BLinterp,TideEl,checkTypeName

  Sample call:
  [umaj,umin,uph,uinc]=tmd_ellipse('DATA/Model_Ross_prior',-73,186,'k1');

  TMD release 2.02: 21 July 2010
```

Functions on the following page access specific tidal harmonic coefficients, either amplitude and phase, or ellipse parameters.

tmd_get_coeff.m

tmd_get_coeff.m extracts amplitude and phase grids from the model, for the specified data type ('h', 'u', or 'v') and specified tidal harmonic. This is useful, for example, for batch files to print out maps of individual harmonics for the entire domain, or a specified limited region of interest.

>> help tmd_get_coeff

```
  function to extract amplitude and phase grids from
  a model ModName (OTIS format) calculated on bathymetry grid
  Gridname

  usage:
  [x,y,amp,phase]=get_coeff(Model,type,cons);
  PARAMETERS

  INPUT
  Model - control file name for a tidal model, consisting of lines
          <elevation file name>
          <transport file name>
          <grid file name>
          <function to convert lat,lon to x,y>
  4th line is given only for models on cartesian grid (in km)
  All model files should be provided in OTIS format
  type - one of 'z','u','v' (velocities),'U','V' (transports)
  cons - tidal constituent given as char*

  output:
  amp - amplituide (m, m^2/s or cm/s for z, U/V, u/v type)
  phase - phase degrees GMT
  x,y - grid coordinates

  sample call:
  [x,y,amp,phase]=get_coeff('DATA/Model_Ross_prior','z','k1');
```

tmd_get_ellipse.m

tmd_get_ellipse.m extracts current ellipse parameters major and minor axes, ellipse orientation, and ellipse phase, for the specified harmonic. As with get_coeff.m, this is useful for for batch files to print out maps of individual harmonics (e.g., major axis) for the entire domain, or a specified limited region of interest.

```
>> help tmd_get_ellipse
  function to extract tidal ellipse grids from a model

  usage:
  [x,y,umaj,umin,uphase,uincl]=get_ellipse(Model,cons);

  Model - control file name for a tidal model, consisting of lines
          <elevation file name>
          <transport file name>
          <grid file name>
          <function to convert lat,lon to x,y>
  4th line is given only for models on cartesian grid (in km)
  All model files should be provided in OTIS format
  cons - tidal constituent given as char*

  output:
  umaj,umin - major and minor ellipse axis (cm/s)
  uphase, uincl - ellipse phase and inclination degrees GMT
  x,y - grid coordinates

  sample call:
  [x,y,umaj,umin,uphase,uincl]=get_ellipse('DATA/Model_Ross_prior','k1');
```

tmd_mk_submodel.m

tmd_mk_submodel.m creates a new model set (Model*, UV*, h*, grid*) for a specified subdomain. The primary use of this script is to make a smaller model that uses less computer resources and so runs more quickly. The sample call in the helpfile, below, takes a small area of the AOTIM5 model to provide a small model for a specific experiment.

```
>> help tmd_mk_submodel
  function to make a submodel from a model ModName (TMD format)
  calculated on bathymetry grid Gridname

  usage:
  ()=tmd_mk_submodel(Name_old,Name_new,limits);

  PARAMETERS

  INPUT
  Name_old - root in "DATA/Model_root" control file for EXISTING
             tidal model. File Model_* consists of lines:
          <elevation file name>
          <transport file name>
          <grid file name>
          <function to convert lat,lon to x,y>
  4th line is given only for models on cartesian grid (in km)
  All model files should be provided in TMD format

  Name_new - root in "DATA/Model_root" control file for SUBMODEL of
             tidal model. The submodel is defined by
  limits - [lon1,lon2,lat1,lat2] OR [x1 x2 y1 y2] for a model in km;
           might be slightly CHANGED to adjust to original model grid

  OUTPUT:

  in TMD/DATA
  Model_<Name_new> - control file of 3 or 4 lines
                     (as in old)
        h.<Name_new> - elevation file
        UV.<Name_new> - transports file
        grid_<Name_new> - grid file

  sample call:

  tmd_mk_submodel('AOTIM5','AOTIM5_subdomain_test',[-1000 1000 -1000 1000]);
```

2. TMD GUI

Note: *The GUI script has not been maintained as carefully as the scripts in Section 1 and may not work on some combinations of Operating System and MATLAB version. We strongly recommend that users develop their own scripts using calls to TMD scripts described in previous pages.*

The GUI is accessed in the following manner:

1. Open the MATLAB dialog box.
2. Change directory to the location where the GUI code resides.
3. Run "TMD" (***Case-sensitive***).

The following is the output shown in the dialog box:

```
>> TMD
Welcome to TMD: Tidal Model Driver!

TMD FILE NAME/FORMAT CONVENTION (MUST follow!):
1. Model and grid files should be in OTIS binary format,
   see http://www.oce.orst.edu/po/research/tide/inv_doc.html
2. Elevation file name should start from 'h'.
3. Transport file name should start from 'UV'.
4. Bathymetry grid file name should start from 'g'.
5. If grid is uniform in km string 'km' should be found
   either in model file names or in grid file name.
5. Default model files location is subdir DATA. For each model
   a control file 'DATA/Model_*' should be given. The file
   MUST contain 3 lines:
         <Elevation file name>
         <Transport file name>
         <Bathymetry grid file name>
   If grid is uniform in km the NAME of function converting
   lat,lon to x,y and back should be given in 4-th line, for example:
         'xy_ll' for Arctic or 'xy_ll_S' for Antarctic
```

4. You will be asked to select a model from a drop-down menu. If there is a ../DATA subdirectory in your current workspace, TMD will look there for TMD-compatible tide models. If you are anywhere else, TMD will display the following message:

```
Subdirectory DATA not found. You must navigate to
location of Model files from current directory.
```

This is straightforward, just like browsing with Windows Explorer.

You now get the following dialog from the MATLAB command window, and a GUI (see next page) appears with bathymetry displayed for the chosen model.

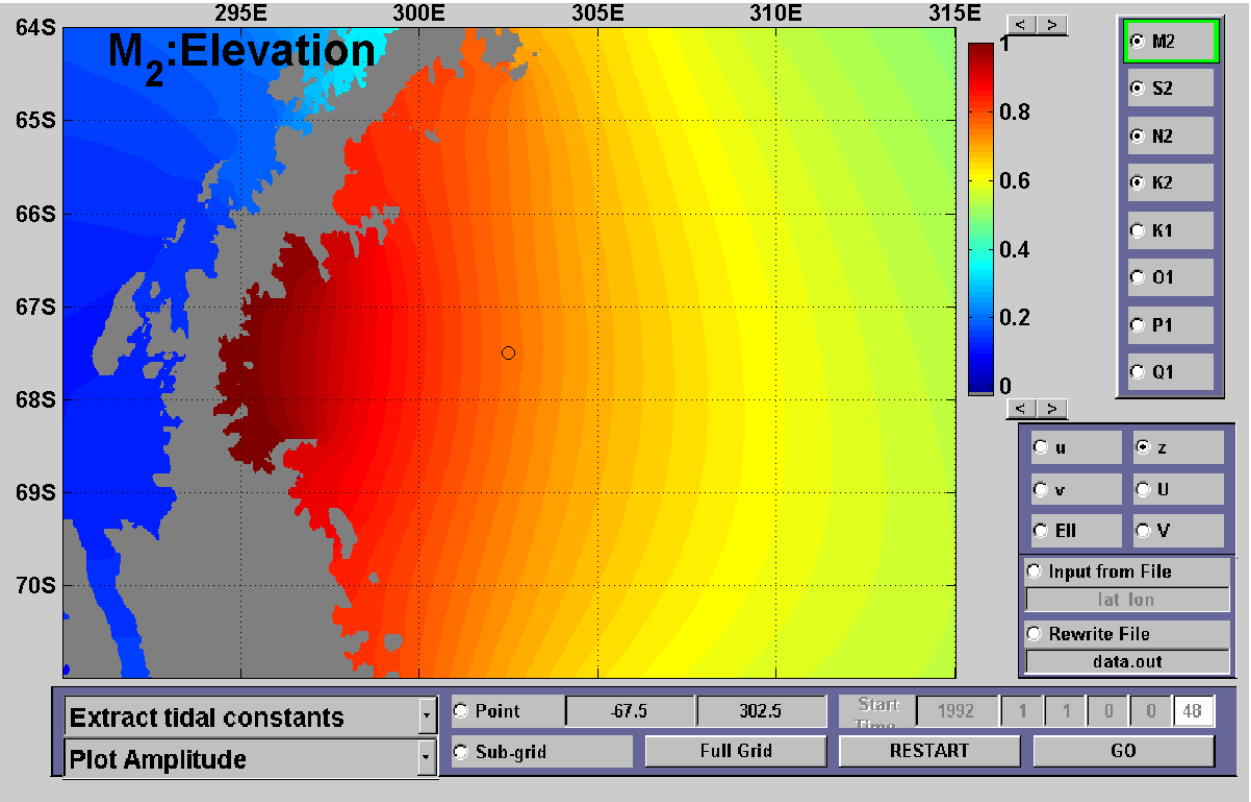
```
The model is on uniform grid in lat,lon
```

```
Loading TMD (Tidal Model Driver)...done

See button tips for HELP.
Type 'help  extract_HC','help tide_pred', 'help ellipse',
Type 'help  get_coeff','help get_ellipse',
if you wish to use the scripts instead of GUI.
Model and files are in D:\Tide_models\TMD\DATA\AntPen\h.Larsen
 and D:\Tide_models\TMD\DATA\AntPen\UV.Larsen.
Bathymetry grid file is in D:\Tide_models\TMD\DATA\AntPen\grid_Larsen.
Input file examples are in: lat_lon, lat_lon_1, lat_lon_2

Programmed by: Lana Erofeeva, 2003
>>
```

5. You can now resize the graphics window so that everything in the screen is clear. Window resizing is monitored, so that for future use of TMD for the same model, window dimensions will be the same as the resized window.
6. Hovering the cursor over specific buttons will give you help for that choice. Note that in the "constituent selection" panel (upper right), you can select multiple constituents. To clear-all/select-all, right-click on the blue-gray frame. If you have multiple constituents selected, the plotted constituent is outlined in green. Right-click on the constituent you wish to view. The green frame will highlight the new selection.
7. To select a point, click the "point" button, then use the cross-hairs to choose the point. Or, enter the point coordinates manually. The selected point will be marked on the plot.
8. To select variables, choose from u (E velocity component), v (N velocity component), U or V (transports: Hu and Hv), z (sea surface height (relative to the seabed)), or Ell (current ellipse properties).
9. If "Extract tidal constants" is selected, the tidal harmonic coefficients for the selected variable and selected constituents will be written to a file. You may either "Append" or "Rewrite" the file. The default file name for ASCII data is 'data.out', although you can change this. A MATLAB file is also written with the same prefix name but with 'mat' suffix.
10. If "Predict tide" is selected, a tidal prediction of the requested variable (u, v, z, U or V, but not 'Ell') for the specified point will be made, and the output saved in the specified data file. Also, a new graphics window will be opened, showing the plot of the predicted variable. The starting time, and length of predicted record, are specified in the lower left above the "Restart" and "Go" buttons.

If anything in the TMD GUI is unclear from either this User Manual or the information for specific buttons, please let us know. Email suggestions or comments to Laurie Padman (padman@esr.org) or Susan Howard (showard@esr.org).

Example TMD GUI display



This screen snapshot shows the TMD GUI for the "Larsen Ice Shelf" submodel (see "mk_submodel.m" documentation, above) extracted from the "AntPen" high-resolution Peninsula-area tide model.

All semidiurnal constituents (M2, S2, N2, K2) have been selected. M2 is displayed (constituent name outlined by green box). M2 amplitude is selected for display (see "Plot amplitude" selection (lower left)). Lower right options include 'u' (E/W) velocity, 'v' (N/S) velocity, U and V (depth-integrated transports; =hu, hv), 'z' surface height, and 'Ell' ellipse properties (major/minor axis, ellipse orientation, ellipse phase).

You can zoom on a subgrid by selecting 'subgrid' (central bottom) or select a specific point using 'Point'. Prediction for a specific point is for the displayed variable, using all selected constituents (not just the one shown highlighted with the green box). Predictions are output to a specified file (click on "rewrite file" dialog box) and are plotted in a new figure window.