

# ARTEMIS developments at HR Wallingford

Noémie Durand, Sébastien Bourban, Nigel Tozer  
Coasts & Oceans Department  
HR Wallingford  
Wallingford, United Kingdom  
N.Durand@hrwallingford.com

**Abstract**— The ARTEMIS model (Agitation and Refraction with TELEMAC on a Mild Slope) from the TELEMAC suite of solvers solves the elliptic mild slope equation using finite element techniques. Its main field of application is wave disturbance studies inside harbours or small bays, often driven by a regional model. ARTEMIS is a well-established model, used in consultancy studies for over 20 years.

ARTEMIS was originally developed by the Laboratoire National d'Hydraulique et d'Environnement (LNHE of EDF-R&D). In recent years, with the transfer of the model to open source, development efforts have been open to entities other than LNHE. This paper presents developments of the ARTEMIS model, at HR Wallingford, available or soon to be available in future releases of the TELEMAC system.

## I. SPECIFYING BOUNDARY NODE PROPERTIES

This feature was introduced on the opentelemac cue system (<http://cue.opentelemac.org/>) as Feature #391: LECLIM to replace coding in BORH.

### A. What's new?

Boundary conditions in the TELEMAC modelling system are specified via a table of variables known as the CONLIM or .cli file. The CONLIM file holds as many lines as there are boundary nodes in the finite element mesh, with each line defining the boundary node properties, characterised by 13 variables (refer table below).

TABLE 1: BOUNDARY NODE PROPERTIES IN THE TELEMAC SYSTEM.

Data column	Parameter name	Parameter description
1	LIHBOR	Code for Depth
2	LIUBOR	Code for U (velocity or flow)
3	LIVBOR	Code for V (velocity or flow)
4	HBOR	Prescribed value for Depth
5	UBOR	Prescribed value for U (velocity or flow)
6	VBOR	Prescribed value for V (velocity or flow)
7	CHBORD	Friction coefficient
...	...	...
12	NBOR	Global node number in mesh
13	K	Sequential boundary node number
14	BndName	Optional: # Name of the boundary

Historically in ARTEMIS, while the boundary node type LIHBOR is routinely defined in the CONLIM file (e.g. solid, open, incident), other boundary node properties have been coded in subroutine BORH. This is where the incident wave height HB, angle of incident wave attack TETAP, reflection coefficient RP etc. are informed, typically using a do loop on boundary node numbers. This process can be prone to coding errors and feature #391 is meant as a user friendly, simpler alternative.

In this new approach, the user can document all the boundary node properties directly in the CONLIM file: HB in column 4 (in place of HBOR), TETAP in column 5 (in place of UBOR), ALFAP in column 6 (in place of VBOR) and RP in column 7 (in place of CHBORD). The values are then read in automatically by the LECLIM subroutine and available for future use, without invoking subroutine BORH.

This meant minor changes to the following subroutines:

- ARTEMIS,
- CONDIH,
- DECLARATIONS\_ARTEMIS,
- LECLIM, and
- POINT\_ARTEMIS

to allow the CONLIM values to be read in and used. These changes were committed to the repository and are available since release v7p3.

It is noted that the use of the CONLIM file in this way is not compulsory, but is an added functionality. FORTRAN files developed with the BORH subroutine will still work and give the same results.

### B. Validation

The test cases developed for ARTEMIS and available from the repository have all been migrated to using an edited CONLIM file rather than coding in BORH, but for test cases *flam* and *kochin*. Test case *creocean* has both functionalities (while *art\_creocean.cas* works with a CONLIM file, *art\_creocean\_2.cas* works with subroutine BORH).

It was verified that the results of simulations run with a CONLIM file were identical to those of simulations run with subroutine BORH.

### C. How is it triggered?

By default the boundary node properties are read in from the CONLIM file.

HB values are read from the 4<sup>th</sup> column.

TETAP values are read from the 5<sup>th</sup> column.

ALFAP values are read from the 6<sup>th</sup> column.

RP values are read from the 7<sup>th</sup> column.

An example is shown below.

1	2	3	4	5	6	7
LIHBOR	LIUBOR	LIVBOR	HB	TETAP	ALFAP	RP
1	5	5	1.4142135	0.00	0.000	0.000
2	5	5	0.000	90.00	0.000	1.000
2	5	5	0.000	90.00	0.000	1.000

Figure 1: Example CONLIM file (excerpt from test case stive85).

The values read from the CONLIM file can be updated at a later stage through the subroutine `BORH`. This is usually the case when the user does not document all the boundary node properties in the CONLIM file (historical approach).

It is noted that, since release v6p2, the use of the `LECLIM` subroutine has been uniformed across all modules of the TELEMAC system. In `LECLIM` are performed, in particular, some general consistency checks geared towards the flow modules. The implication is that the TETAP and ALFAP values are reset to 0 when the corresponding codes LIUBOR and LIVBOR are other than KENT (5) or KENTU (6).

It is, therefore, essential to give LIUBOR and LIVBOR (2<sup>nd</sup> and 3<sup>rd</sup> columns) values of either 5 or 6 (see Figure 1) such that the values of TETAP and ALFAP (5<sup>th</sup> and 6<sup>th</sup> columns) are taken into account correctly.

For simple applications, the CONLIM file can be defined directly in software such as Blue Kenue. For more complex applications, or even to facilitate change of boundary node properties, the user may benefit from an external CONLIM file generator (e.g. using QGIS). Such a tool has not been developed at HR Wallingford to date.

## II. PRESCRIBING THE PHASE AT THE INCIDENT BOUNDARY

This feature was introduced on <http://cue.opentelemac.org/> as Feature #392: `PHBOR` to compute phasing automatically, starting with the angle of wave impact.

### A. What's new?

Prior to release v6p2, the phase of the incident boundary was computed internally, without user intervention. This has changed with release v6p2, from which point the user was expected to document the phase ALFAP for incident boundary node in particular in subroutine `BORH`. This has sometimes been perceived as a hindrance, and feature #392 is meant to revert to a more user friendly approach.

One of the reasons for the change in release v6p2 was compliance with parallel computations.

Subroutine `PHBOR` was modified in this development to host the automatic phase calculation as it did prior to release v6p2, with a few changes:

- Allowance is made for parallel computations,
- The first step is the identification of the “node of attack”: the incident boundary node first hit by waves from the specified direction (Figure 2),
- The phase of all other incident boundary nodes is then iteratively computed from that node to the next.

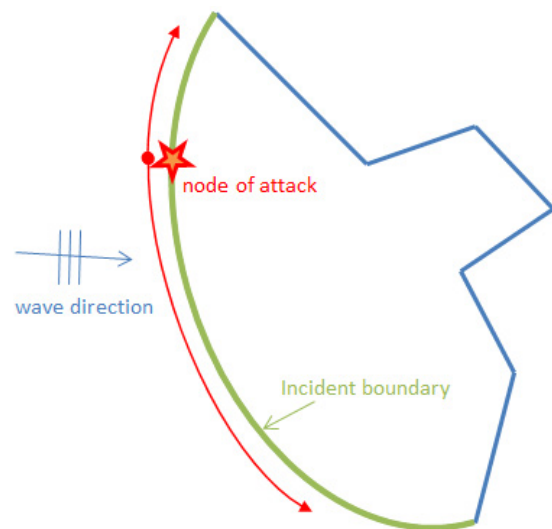


Figure 2: Definition of “node of attack” for phase computation.

These changes were committed to the repository and are available since release v7p3.

### B. Validation

The test cases developed for ARTEMIS and available from the repository have all been repeated and validated with the modifications.

### C. How is it triggered?

Nothing is required from the user to activate the automatic phase calculation. The phase for the incident boundary non longer needs specifying in subroutine `BORH`, unless specifically required for particular applications.

## III. GENERATING FREE SURFACE ANIMATIONS

This feature was introduced on <http://cue.opentelemac.org/> as Feature #185: Generate the files necessary to produce an animation of the free surface.

### A. What's new?

The ability to animate the free surface elevation over the model domain, as predicted by ARTEMIS, is thought to be useful in helping to explain and illustrate the physical processes of wave transformation and disturbance, in particular to a non-scientific audience.

An example is shown below in Figure 3.

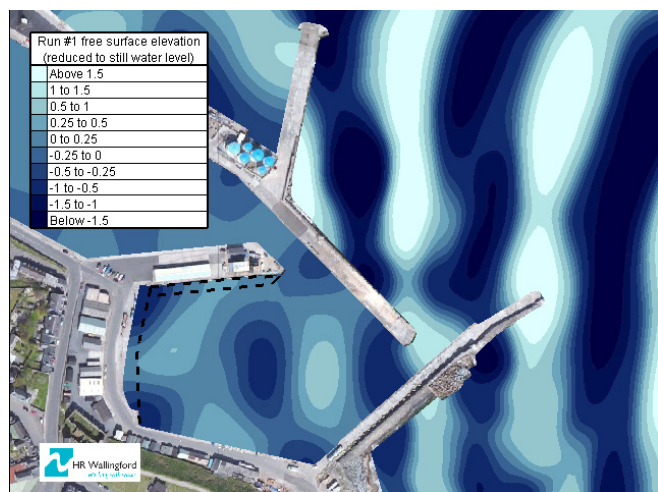


Figure 3: Example of an animation.

Standard output from random wave runs in ARTEMIS does not allow animations to be generated, which can currently only be achieved by running each component separately and outputting the amplitude and phase for each single component run.

New keywords were added that allow a more automatic approach, and generate an amplitude/phase file covering the model area for an ARTEMIS multi-component run. A post-processing tool was developed that will run, if requested, after the ARTEMIS simulation is complete. This will compute the free surface elevation at every point in the model area from the phase and amplitude predicted for each period/direction component run in ARTEMIS and for a user-specified range of times and time increment. A free surface elevation file in SELAFIN format is generated from these.

This post-processing development meant minor changes to the following subroutines and python scripts:

- ARTEMIS,
- DECLARATIONS\_ARTEMIS,
- LECDON\_ARTEMIS,
- POINT\_ARTEMIS, and
- runcode.py,

as well as the introduction of subroutine BIEF\_ANIMP.

These changes were committed to the repository and are available since release v7p3.

### B. Validation

Two test cases out of those developed for ARTEMIS and available from the repository have been converted to produce time histories of free surface elevation. These are *bj78* and *creocean*.

Time histories were generated at a number of locations throughout the model areas and Fast Fourier Transform of the resulting time histories were performed to estimate the associated significant wave height ( $H_s$ ) and mean wave period ( $T_{m02}$ ). These were successfully compared against

the wave height parameters predicted by ARTEMIS directly to within less than 0.5%.

### C. How is it triggered?

New keywords were introduced in the ARTEMIS steering file that output the amplitude and phase for each run component and covering the model area:

- FREE SURFACE ANIMATION (yes/no)
- AMPLITUDE AND PHASE FILE'

A post-processing tool was added to the artemis.py script that will generate a time-varying free surface file, for a user-specified range of times and time increment, should the following keywords be documented. It is noteworthy that the size of the resulting file will quickly become prohibitive so careful consideration should be given to these values.

- FREE SURFACE FILE
- FIRST TIME IN THE FREE SURFACE FILE
- TIME STEP
- NUMBER OF TIME STEPS

## IV. NESTING ARTEMIS IN A REGIONAL MODEL

This feature was introduced on <http://cue.opentelemac.org/> as Feature #205: TOMAWAC / ARTEMIS coupling.

### A. What's new?

Current practice in the application of the ARTEMIS model consists in specifying representative uniform conditions along the ARTEMIS model boundary. If a regional model is used to inform these conditions (functionality available since release v7p0), a single representative point is used for sharing the information between the regional (e.g. TOMAWAC) and the ARTEMIS models, such that variations along the ARTEMIS model boundary are not allowed.

It is recommended that the incident model boundary sit in reasonably constant water depth. However, there are circumstances when this is not practical and when, as a consequence, allowing variations along the incident model boundary may be beneficial. This is the case, for example when the modelling includes a navigation channel, which modifies the refraction wave patterns.

The nesting approach developed at HR Wallingford aims at reducing loss of information between the regional spectral wave model and the local wave disturbance model, with a more accurate representation of boundary conditions. This allows in particular spatial variations.

The approach can be summarised in the following simple steps:

- The spatial output from the regional (e.g. TOMAWAC) model is used to inform the spatial variations in significant wave height along the ARTEMIS incident boundary.

- The spectral output from the regional model (more than one point allowed) is used to determine the NPALE directional components independently at each of the incident boundary nodes.
- The spectral output is also used, at a user-designated location along the ARTEMIS incident boundary, to select the NPALE period components that will be run in the model.

A number of existing subroutines were amended to allow the new nesting functionality. These are:

- ARTEMIS,
- DEALL\_ARTEMIS,
- DECLARATIONS\_ARTEMIS,
- INTERFACE\_ARTEMIS,
- LECDON\_ARTEMIS,
- POINT\_ARTEMIS,
- SPD,
- SPE, and
- TWCALE.

Some subroutines were also added, in keeping with the TELEMAC coding standards and use of the HERMES module to access SELAFIN files. These are:

- ALLSPEC,
- FASP\_SP,
- GET\_DATA\_TIMESTEP,
- GET\_TOMSPEC\_DIMENSIONS,
- GET\_TOMSPEC\_VALUES,
- GET\_TOMSPEC\_VALUE1,
- GET\_TOMSPEC\_VALUE2,
- LISSAGE,
- STIRLING,
- STWC1,
- STWC2,
- TWCALE2,
- TWCCLOSEST,
- XY\_TOMAWAC.

It is proposed to make these developments available to the TELEMAC community but more testing is required before then. The following is an initial example that illustrates how the spatially varying information is passed from the regional model to the ARTEMIS model.

### B. Validation

An old ARTEMIS test case: *beach* was reinstated to validate the developments. In this test case, the TOMAWAC model is run on a uniformly sloping beach. The ARTEMIS model is nested before the top of the beach, in varying water depths (and wave field). Waves approach from the South-West and so, in ARTEMIS, the westernmost and southernmost boundaries are defined as incident, while the easternmost and northernmost boundaries are defined as open. `AUTOMATIC TETAP CALCULATION` is turned on in these tests.

Figures 4 to 6 show the results of the TOMAWAC (background) and ARTEMIS simulations under different configurations:

- Figure 4: currently available nesting approach, using a single representative point to share information between the regional and ARTEMIS models;

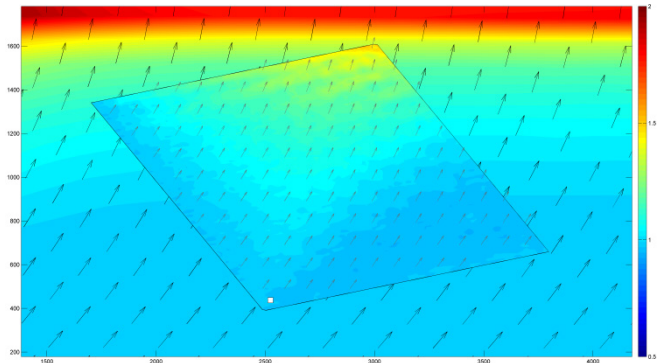


Figure 4: *beach* test case  
background: global model wave field (Hs and direction);  
foreground: ARTEMIS wave field with current nesting approach;  
white square: nominated spectrum.

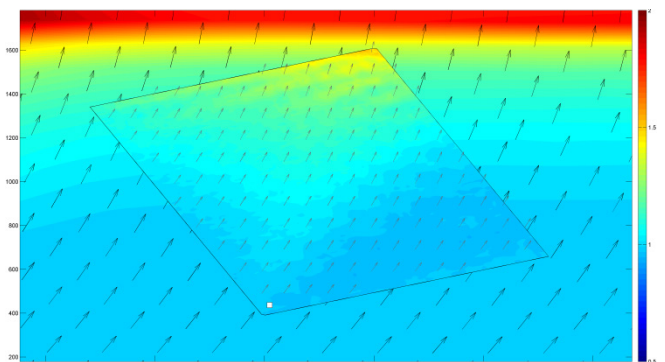


Figure 5: *beach* test case  
background: global model wave field (Hs and direction);  
foreground: ARTEMIS wave field with new nesting approach;  
white square: nominated spectrum.

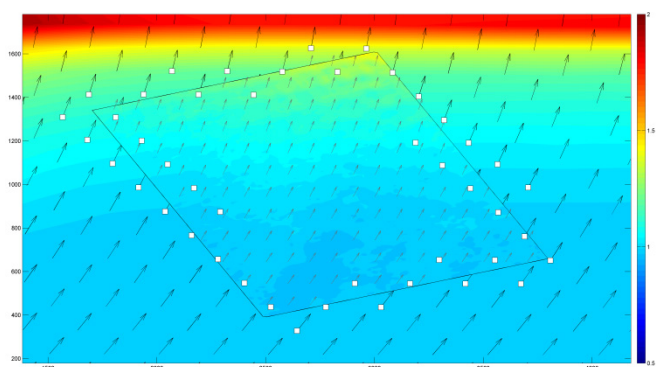


Figure 6: *beach* test case  
background: global model wave field (Hs and direction);  
foreground: ARTEMIS wave field with new nesting approach;  
white square: 45 input spectra.

- Figure 5: new nesting approach, using the same single representative point. Notice how the wave heights at the westernmost boundary of the ARTEMIS model are a better match to the wave heights predicted by TOMAWAC. This is due to the use of the TOMAWAC spatial output to inform the spatial variations in significant wave height along the ARTEMIS incident boundary;
- Figure 6: new nesting approach, making use of more spectral data points from TOMAWAC to improve the representation of the incident boundary. Notice how the wave directions at the westernmost boundary of the ARTEMIS model are now a better match to the wave directions predicted by TOMAWAC.

### C. How is it triggered?

New keywords are introduced or modified in the ARTEMIS steering file that trigger the proposed nesting approach:

- `COUPLING WITH TOMAWAC OUTER MODEL`
- `TOMAWAC OUTER SPECTRAL FILE`
- `TOMAWAC OUTER RESULT FILE`
- `COORDINATES OF THE REFERENCE F SPECTRUM`

It is noted that the currently available nesting method (using a single representative spectrum) is preserved as `COUPLING WITH TOMAWAC OUTER MODEL = 1`. `COUPLING WITH TOMAWAC OUTER MODEL = 2` activates the new nesting approach. `COUPLING WITH TOMAWAC OUTER MODEL = 0` is the default, meaning no nesting is required.

The `TOMAWAC OUTER SPECTRAL FILE` is the `.spe` file from a TOMAWAC run. This is a time-varying file and the user is required to specify `INSTANT FOR READING TOMAWAC SPECTRUM`, time from which the ARTEMIS boundary conditions need developing.

The `TOMAWAC OUTER RESULT FILE` is the spatial file of integrated parameters from a TOMAWAC run. This is also a time-varying file and the time specified in `INSTANT FOR READING TOMAWAC SPECTRUM` will be used to extract boundary conditions. However, currently, and for efficiency purposes, the development relies on the boundary conditions interpolated from the regional model being specified in a binary file (`BINARY DATA FILE 1`) created by `convertToBND.py` in the python script folder.

```
convertToBND.py [input1: ARTcliFile] [input2:
ARTgeoFile] [input3: TOMslfFile] [output: ARTbndFile]
```

The `TOMAWAC OUTER RESULT FILE` is currently only directly used to access the coordinates of the spectral data points in the `TOMAWAC OUTER SPECTRAL FILE`. A process more transparent to the user may be considered in time.

The `COORDINATES OF THE REFERENCE F SPECTRUM` are required if using `COUPLING WITH TOMAWAC OUTER MODEL = 2`. This will select the closest data point, within those present in the `TOMAWAC OUTER SPECTRAL FILE`, from which to compute the frequency spectrum. This spectrum is taken to be representative of the incident

boundary conditions and is the basis for the discretisation into NPAL period components in the ARTEMIS run.

This development is subject to further testing before it can be shared with the community, and as such is not part of the general ARTEMIS release at the moment.

### CONCLUSIONS

This paper presents developments of the ARTEMIS model, at HR Wallingford. Some of these developments are already available to the TELEMAC community since release v7p3. Others (namely nesting ARTEMIS in regional models) are still ongoing and subject to further testing before they become available in future releases of the TELEMAC system.

Comments and suggestions on the developments presented in this paper are welcome.

