# Using AQWA-WAVE to apply hydrodynamic forces to ANSYS

It is often required to apply the hydrodynamic pressures that are calculated within AQWA to a finite element model. This is achieved by using AQWA-WAVE to extract the forces and create the loading in a series of pressure loads compatible with input into ANSYS. It can also be used to create an analysis file for ASAS.

The process to create the loading is as follows:

1. Create ANSYS model.

2. Use ANStoAQWA macro to create hydrodynamic model file.

3. Run AQWA-LINE to generate loading.

4. Use ANStoASAS macro to create structural model file.

5. Run AQWA-WAVE to generate structural loading and to transform into ANSYS loading.

6. Use /INPUT to import loading to ANSYS model.
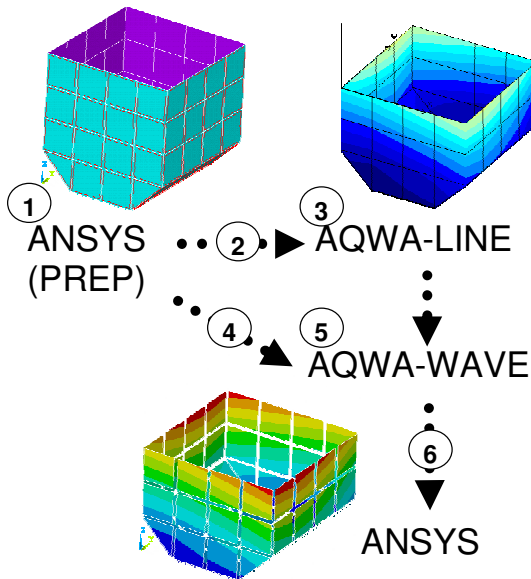
In this case the ANSYS model is just a representation of a ship hull; it is kept simple as the point is to illustrate the process which can then be applied to more complex models.

The model that we are using is as below, each panel is a 5m x 5m square and the water line is located 15m from the bottom of the model. As conventional with offshore structures, the vertical axis is in the Z direction (positive up).
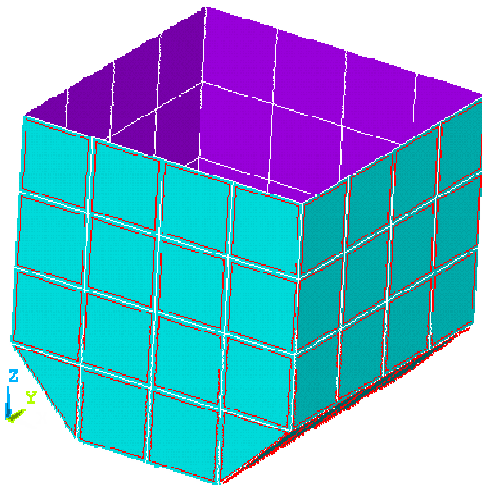
Two main requirements of the AQWA-WAVE model are:

- External faces are those for which element numbering is clockwise.

- Wetted faces are identified as those will pressure loading applied as loadcase 1000.

In this case the hydrodynamic and the structural model are identical, however, this is not a requirement, it is quite often that the mesh density for the hydrodynamic model is course compared with the structural model.

Creating and analysing the hydrodynamic model

With version 11 and later versions of ANSYS there are two APDL based macros supplied to create both AQWA and ASAS models. The AQWA macro can be invoked by calling ANStoAQWA, following which the below dialog is displayed.
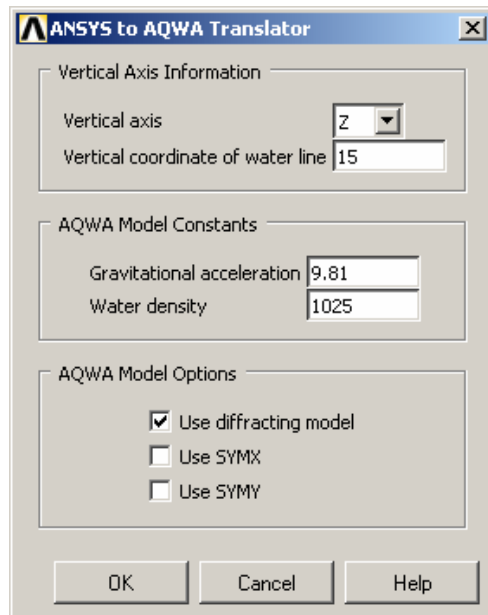
In this dialog the vertical axis (i.e. the one perpendicular to still water surface) and the height (or vertical coordinate) of the water line are specified.

To account for unit differences you are also able to specify G and the water density.

In this case we have not employed symmetry, but we are using a diffracting model.

Once happy with these choices, selecting OK will create an file that can be run by AQWA, some further changes do need to be made to the file as not all parameters can currently be defined in ANSYS.

The AQWA model that is generated needs a couple of minor changes to make it suitable for use with AQWA-WAVE, defaults exist for items that currently can not be defined in ANSYS (wave directions etc.) so the file is created for data check purposes.

To change the file so that it is suitable the following modifications are required to the preliminary data area

OPTIONS REST END   OPTIONS REST **LDOP GOON** END
RESTART 1 2   RESTART 1 **3**

Once these changes have been made, the file can be run with AQWA-LINE to calculate the hydrodynamic forces, producing the below dialog, once the analysis is complete it can be checked that no errors occurred and the dialog closed.
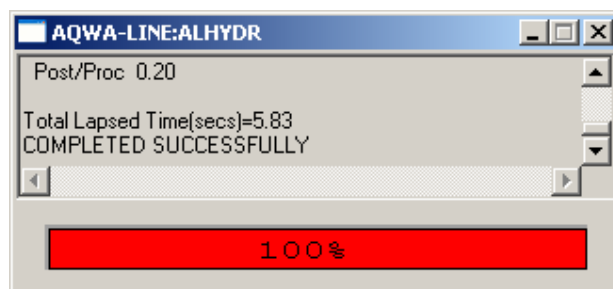
The AQWA Graphical Supervisor can be used to inspect the analysis results in greater detail.

Using AQWA-WAVE to form pressure loads for ANSYS

AQWA-WAVE's purpose is to convert the pressures that are applied to the hydrodynamic model and to map these onto the structural model. Only those elements that are defined as external and as wetted will attract load, as stated previously, the nodes and elements do not need to match exactly as the forces will be interpolated to establish the load values.

An ASAS model is required as the structural model input into AQWA-WAVE and, if ASAS is selected as the output format, this forms the bases of the ASAS compatible analysis file. In this case we are using ANSYS as our analysis package, so an option is applied to create the loading in that format.

The ASAS file is simply created using the APDL macro ANStoASAS, no further intervention is required before AQWA-WAVE can be run.

A fully illustrated AQWA-WAVE file is detailed at the end of this document; however, in brief the data file includes sections for the following:

Preliminary data	Specification of Hydrodynamic and structure files
FELM data	Enables differences between the hydrodynamic and structural to be addressed, e.g. units, origin position, axes. Also includes option to specify load format (ASAS/ANSYS).
ASGN data	Drag load information for Morrison elements. Not yet applicable for ANSYS load output
CURR data	Sea current data referred to by the load data
LOAD data	Which sea states (wave direction, height, etc.) to have loadcases created for, these can be combined with current forces as defined in CURR

When AQWA-WAVE is run, it will create separate files for each load case, these files can then be imported into the ANSYS file using the /INPUT option.

Plotting the pressure contours in ANSYS is shown to the right; the same wave case plotted in AQWA is shown below for reference.
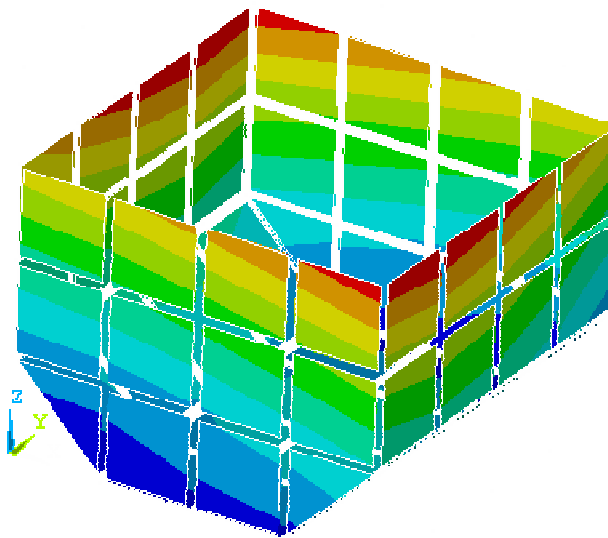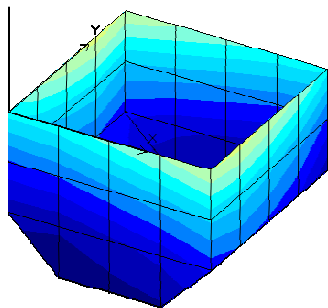
ANSYS commands format:

A summary of the commands that are used to produce the hydrodynamics and structural models, run AQWA-WAVE and then import the loading into ANSYS are below. These are included in the ANSYS.dat file, which also includes the structure definition in order that it can be read into ANSYS and run automatically as an example. This file should be distributed with this document.

```
! Create the hydrodynamic file for aqwa....
ANSTOAQWA,'alhydr','Z',9.81,1025,15.0,1,0,0

! Run the file using aqwa,
!  Presently the file needs to have a .dat extension to be run
/SYS, rename alhydr.aqwa alhydr.dat

!  Modify the AQWA data, for restart stage 3, LDOP & GOON options
/SYS, notepad alhydr.dat
!  Run AQWA with the modified data
/SYS, c:\aqwa\v57a\aqwa57a alhydr.dat          This path may require updating

! Create the structure in the ASAS format for AQWA-WAVE to read
ANSTOASAS,'structure',0

! Run AQWA-WAVE with the default input file, this uses the structure written by ANSTOASAS
and the AQWA file.
/SYS, c:\aqwa\v57a\aqwawave aqwawave.dat          This path may require updating

/SOLU

! Import the generated loading into ANSYS, also invokes solve.
/INPUT, structure_aqld60, dat
```

AQWA-WAVE commands format:

A summary of the commands that are contained in AQWAWAVE.dat referred to above to run with AQWA-WAVE are below. A fully commented version should be distributed with this document.

```
SYSTEM DATA AREA 50000
JOB NEW LINE
PROJECT ANSY                    Selection of the
                                structural model
EXTENSION DAT
END

structure.asas                  Selection of the
END                             hydrodynamic model
AQWAID alhydr

FELM
FEPG ANSY                       Selection of ANSYS as
END                             output format

LOAD
CASE 0 5 5 1.3 0                Selection of sea state
LCOF 59                         (loadcase) to obtain from
END                             hydrodynamic model.
STOP
```