



**SuperGen Marine
Workpackage 2
(T2.3.4)**

**An Appraisal of a Range
of Fluid Modelling Software**

A. P. McCabe

October 2004

**Department of Engineering
Lancaster University
LA1 4YR
U.K.**



CONTENTS

1. Introduction	5
2. Marine Hydrodynamics	7
2.1. Potential Flow	7
2.2. Frequency Domain Solution	8
2.3. Time Domain Solution	9
2.4. Slender Elements – Morison’s Formula and Strip Theory	9
3. Computational Fluid Dynamics (CFD)	11
3.1. Grids and Meshes	11
3.2. Discretized solution Methods	12
3.3. Multiple Fluids and Free Surfaces	12
3.4. Parallel Processing	13
4. Hydrodynamic Software Packages	14
4.1. AQWA	18
4.2. HYDROSTAR	23
4.3. MOSES	26
4.4. NEPTUNE	30
4.5. WADAM	33
4.6. WAMIT	37
4.7. WAVELOAD	41
4.8. TiMIT	45
4.9. Rough Guide	48
5. Computational Fluid Dynamics (CFD) Software Packages	49
5.1. ADINA	53
5.2. CFD++	56
5.3. CFD-ACE++	60
5.4. CFX	64
5.5. EFD.Lab	68
5.6. FLUENT	72

5.7. PHOENICS	76
----------------------	-----------

Appendices

A.1. Hydrodynamic Software Directory	82
A.2. CFD Software Directory	83
A.3. Trademarks	84

1. Introduction

The design of marine energy converters (MECs) eventually requires the calculation of values such as forces and motion amplitudes in order to assess the performance of the device and its suitability for the purpose. The complex nature of the environment within which MECs work, and the consequential complicated mathematical description of it, necessitates the use of computer methods to solve these problems. Many programs have been devised to solve fluid dynamics problems of this nature, some application-specific, others of an extremely general character. Some programs have been produced by research departments mainly for in-house use, others have been developed commercially and are on general sale. There are hundreds of them, so some selection has been unavoidable, though it is hoped that those selected form a representative group. The range of software packages included in this appraisal has been limited, in the main, to commercially available programs, so that it can be assumed that full validation and verification have been carried out. Of course, every rule must have an exception, and, in this case, the exception is the inclusion of TiMIT, a non-commercial time-domain hydrodynamics program produced by the Massachusetts Institute of Technology (MIT). This has been included partly out of academic interest, and partly because of the access MIT give to the theoretical background to the program, which provides much information for a newcomer to the subject.

No attempt has been made to make any sort of comparison between different software packages. To do so would entail, on the one hand, arguments involving both arcane mathematical analyses and equally arcane computational methodologies, and, on the other hand, the assessment of subjective matters such as 'user-friendliness' or 'learning curves'. In addition, the packages vary greatly in terms of price and features offered, such that comparison is rendered virtually meaningless. This appraisal, therefore, tries to present each program using a standardized description so that the asserted capabilities are expressed in a clear and concise manner. A directory is included to provide prospective users with contacts where more detailed information may be obtained.

In marine energy conversion applications, two main categories of software can be used. These are hydrodynamics software and computational fluid dynamics (CFD) software. The former uses mathematical analysis of a much more specialized case than the latter. Hydrodynamics programs are generally used in the investigation of fixed or floating bodies in the presence of gravitationally-driven waves. Section 2 gives a basic description of the derivation of the theory behind marine hydrodynamics, without expounding mathematically. The intention is to introduce the terms for the values produced by the programs, and indicate where they fit into the theory. CFD software packages are designed to solve general equations for unsteady, compressible, viscous flow in one or more fluids and are generally used in the investigation of submerged machinery and structures in the presence of significant flows such as tidal currents. Because of their general nature, these programs are usually differentiated by the computational methods that they use. A basic description of these approaches is given in Section 3, again more as a clarification of terminology than a technical analysis.

Section 4 describes a selection of Hydrodynamic software packages, also giving a brief description of the manufacturer, the development of the software (where known), the types of computer platform on which the software will run, and licensing arrangements where these can be ascertained. Section 5 describes a selection of CFD software

packages, with similar brief descriptions of the manufacturer, software development, computer platform, and licensing arrangements. Prices are quoted in good faith at summer 2004 levels, but will be subject to change at the manufacturers' discretion. The appendices contain directories of contact details for the software packages described in the main text, and a list of trademarks associated with products mentioned in the text.

Readers should note that: The author and publishers of this document accept no responsibility whatsoever for the veracity of claims made by third parties, or for the consequences of omissions or misinterpretations made during the editing process. This appraisal is intended for informational purposes only and no implication of recommendation or disapproval of any product, service or organization is either intended or should be inferred. Prospective purchasers of products mentioned in this document are strongly advised to obtain more detailed information from the relevant manufacturer (who will usually be glad to help) before committing to any transaction.

Andrew McCabe

Department of Engineering
Lancaster University
Lancaster
LA1 4YR
UK

October 2004

2. Marine Hydrodynamics

The analysis of the behaviour of fully or partially submerged bodies needs to encompass the effects introduced by the surrounding fluid. In marine hydrodynamics the general procedure is to calculate the pressures and fluid velocities in the vicinity of the submerged portion of the body under analysis. This body may be permanently fixed, or allowed to float, either free or subject to some constraint such as moorings. From these calculations it is possible to derive both the steady and unsteady forces and moments acting on the body, and, hence, amplitudes of motion and other values of interest.

There are many texts which cover general fluid dynamics, and the derivation from this of marine hydrodynamics by special-case simplifications and assumptions. The general equations for unsteady, compressible, viscous flow in a three-dimensional fluid comprise a continuity equation, three momentum equations, an energy equation and two equations which relate density and static enthalpy, respectively, to temperature and pressure. A major simplification can be made by taking account of the incompressibility of water in normal circumstances. In an incompressible flow, the velocity and pressure fields can be derived from just the continuity and momentum equations. Further simplifications can be made by applying assumptions pertinent to more restricted cases.

2.1. Potential Flow

Potential flow, in which the motion of the fluid is assumed to be inviscid and irrotational, describes the fundamental behaviour of free-surface waves driven by gravity, so is the most commonly-used approach in the analysis of bodies affected by sea waves. In potential flow the momentum equations reduce to a simple relationship between the acceleration and the pressure gradient in a fluid, and the continuity equation becomes the Laplace equation for fluid potential, ϕ ,:

$$\nabla^2 \phi = 0 \tag{1}$$

The solution of equations of fluid motion requires the definition of boundary conditions, relevant to the hydrodynamic problem in hand. Essentially, these delineate the space within which the fluid motion is to be modelled. Typically, boundary conditions are prescribed on the body surface, the free surface, the sea bottom, and at infinity or some practical equivalent thereof. The Laplace equation is linear, so it is possible to superpose elementary solutions to produce solutions for more complicated cases. There are many useful properties associated with potential flows that facilitate the use of boundary-element or panel methods when devising numerical solution techniques. Such methods remain by far the most widely used in commercial software. To derive the fluid potential, the Laplace equation (1) is converted into an integral taken over the body surface. The division of the body surface into a discrete number of panels transforms the overall surface integral into a set of integral equations, one for each panel. Each panel is taken to be a fluid source (or singularity), which has an effect on every other panel source, all contributing to the flow over the surface. The influence any particular panel has over the rest of the panels is governed by a weighting function, known as a Green's Function. The integral equations are discretized into summations for computation.

Special cases exist, however, in which the computational load becomes severe, or where values are required which cannot be calculated in a robust manner using the standard 'low-order' panel method. To avoid these problems, 'higher-order' panel methods were introduced. The majority of higher-order methods involve linear or quadratic approximations of the body geometry and potential on each panel, in the form of first- or second-degree polynomials. In one method B-spline basis functions are used instead of the polynomial functions, to provide a more continuous representation.

The equations of motion are expressed in terms of a six-degree-of-freedom rigid-body model, in which three modes of motion are translational (surge, sway and heave motions), and three are rotational (roll, pitch and yaw). The equations may be solved in either the frequency domain or the time domain. Frequency domain solutions are applicable where the wave excitation is of either simple harmonic form or of the superposition of simple harmonic forms. Additional requirements are that the body motions are of small amplitude and the boundary conditions are linear, or can be made so. Time domain solutions, in the form of simulations, can apply where non-linear effects are too large to be ignored and frequency domain techniques no longer produce viable results.

2.2. Frequency Domain Solution

The linearity conditions which apply for frequency domain solutions to be valid also allow the solutions to be separated into two problems – the diffraction problem and the radiation problem. The diffraction problem addresses the way a body distorts the wave field by its presence. The diffraction potential is often divided into a scattering potential, which represents the disturbance to the incident wave due to the presence of the body, and a potential due to the incident wave itself. The radiation problem deals with the waves generated by a body as it moves in the fluid. Each of the six modes of body motion has an associated radiation potential. Flow and pressure fields in the fluid around the body are derived from the total velocity potential, which is the complex sum of all the diffraction and radiation potentials. Integration of the pressure over the surface of the body gives the hydrodynamic and hydrostatic forces and moments to which the body is subjected. The hydrodynamic forces and moments are separated into two components analogous to the radiation and diffraction potentials, the first expressed in terms of the added mass and damping coefficients, the second in terms of wave exciting forces and moments. These coefficients and forces are then used to derive the motions of the body, in terms of response amplitude operators (RAOs). Existing dynamics, such as those introduced by moorings, etc, may also be included in the calculation of the RAOs. Mean drift forces can be calculated either directly from the pressure integration, or from conservation of momentum in the body.

The integral equations described above tend to become singular as the thickness of a body decreases toward zero. Reducing the panel size will keep the system of linear equations well-conditioned, at the cost of exponentially increasing computation time as the number of panels increases. Alternative, non-singular, forms of Green's integral equations exist for the limit where the thickness vanishes, and these can be used where the thickness of a body or body element can practicably be regarded as zero. The velocity potential is represented by dipoles, singularities which are equivalent to two sources (one on each side of the panel) with no intervening distance. For certain body configurations the integral equations have non-unique solutions at some frequencies, referred to as irregular frequencies. Extended boundary integral equations have been

derived to remove this irregular frequency effect. The separation of the radiation problem into individual components can be extended to cater for the interaction of multiple bodies. The application of suitable boundary conditions and the superposition of reflected incident waves allow the effect of nearby walls to be accommodated.

The basic frequency domain analysis assumes a simple harmonic excitation consisting of a single-frequency sinusoidal wave, called the First-Order Problem. Some software packages also find the frequency domain solutions when the excitation consists of two interacting component waves – the Second-Order Problem. Second-order forces, moments, RAOs, and other coefficients are usually expressed in terms of sum-frequency and difference-frequency components, or as Quadratic Transfer Functions.

2.3. Time Domain Solution

The simplest method used to obtain the time history of responses is the transformation of frequency domain results into the time domain. This method is subject to all the linearity conditions that apply to frequency-domain solutions, so is of dubious value in situations where non-linear behaviour may occur. Non-linearities in the hydrodynamic problem can arise due to a number of causes, such as large amplitude motions leading to significant change in the submerged surface. By far the most important source of non-linearity in marine hydrodynamics is the irregular nature of waves in the sea itself. The free surface behaviour of a real (or realistic) sea state can only be represented by time domain equations in terms of its elevation and pressure, so the solution of the hydrodynamic problem must also develop over time, in the form of a simulation. In general, non-linear hydrodynamic analysis in the time domain is carried out by integration of the pressures over the body surface at each time step in the simulation. This usually entails the use of the three-dimensional panel method with the transient free-surface Green function to obtain the velocity potentials or source strengths distributed over the body surface. Certain linearizations may still be applied, though. The body surface may be taken to be the ‘mean wetted surface’, unchanging during the simulation (the ‘body-linear’ case). Apart from a great saving in computational effort, the use of the ‘mean wetted surface’ may result in an acceptably small loss of accuracy, and is often used in the calculation of the radiation and diffraction forces. For effects such as non-linear restoring forces, however, it is necessary to use the instantaneous wetted surface, computed from the motion responses at each iteration of the simulation (the ‘body-nonlinear’ case). Because the simulation is performing direct (discretized) integrations over time, additional effects, both linear and non-linear, may be included in the equations of motion. These include ‘memory functions’, in the form of convolution integrals of the wave and body motions, to account for effects which persist after motions occur and give a more comprehensive description of the radiation impulse response than that given by the hydrodynamic coefficients alone. Second- and third-order roll-damping terms are also frequently included.

2.4. Slender Elements – Morison’s Formula and Strip Theory

A commonly-used approximation to the wave force on slender elements such as beams and cross-members is Morison’s formula, which assumes that the total wave force is the sum of inertial and viscous forces, or a linear acceleration term and a quadratic velocity term. The inertia term in the equation is identical with the linear force derived by potential theory. Morison’s formula is of particular importance in the analysis of forces on submerged frame structures, such as offshore platforms.

The principal object of marine hydrodynamic research for much of its history has been to investigate the behaviour of ships. It is not unsurprising that a major section of hydrodynamic theory takes advantage of the large disparity between a ship's length and its width (or beam) to effectively reduce the problem from three dimensions to two. The assumption of a small *slenderness parameter* (latitudinal-to-longitudinal dimension ratio) is the basis of Slender-Body or Strip Theory. Several hydrodynamic software packages allow the use of this theory, usually for ship sea-keeping analysis. Care must be taken not to invalidate the basic assumptions if this type of analysis is used.

3. Computational Fluid Dynamics (CFD)

With the potential flow model outlined in the previous section, the assumption of inviscid flow allows the problem to be reduced to a set of surface integrals on the fluid-body interface, or a discretization thereof. With laminar or turbulent flow, however, the entire fluid volume must be modelled to accommodate the variations in flow. The complete solid geometry of the CFD problem then consists of two 'domains', an over-used term which appears later in a different context with a different meaning. The two domains are the solid domain, usually a three-dimensional geometric definition of the body under analysis, and the fluid domain. For problems relating purely to fluid flow, only the fluid domain need be considered. If, however, the problem includes structural analysis of, or deformation to the body under consideration, then both domains must be taken into account.

3.1. Grids and Meshes

The discretization of the three-dimensional model of the fluid domain entails the division of the volume into component cells. A 'grid' of coordinate points (or nodes) is created within the fluid volume, to define the vertices of individual component cells. A 'mesh' is then generated using lines to connect the grid nodes thus forming the edges of the component cells. Grids and their dependent meshes are classified according to the distribution of the nodes through space. 'Structured' grids have a constant pattern of distribution of nodes in each spatial axis. Unless prohibitively small cell sizes are used with such grids there is difficulty in representing even moderately complicated geometries accurately. This is alleviated somewhat by the use of mapping techniques to produce 'body-fitted' or 'body-aligned' meshes that attempt to match the curvature of the domain boundaries, subject to restrictions on the ensuing computational complexity. An alternative method is the 'multi-block' method, which allows the amalgamation of large groups of cells into single blocks. Small cell sizes can then be used to approximate complex surfaces and larger blocks of cells created elsewhere, thus reducing the overall computational burden.

'Unstructured' grids avoid this issue by using arbitrary distributions of nodes in the grid, the only restriction being that the cells occupy the volume completely. Such meshes can consist of two-dimensional elements (triangles or quadrilaterals), or three-dimensional elements (hexahedra, triangular or tetrahedral prism, or pyramids). The advances in computer techniques for the generation and manipulation of meshes has seen a rise in the usage of unstructured grids, with automatic 'fine-tuning' of the mesh density, where required. Not only geometric considerations are taken into account in arranging the mesh distribution. Physical issues may play a part, for example, if a detail of the flow, such as boundary layer behaviour, is under scrutiny, or if the flow consists of more than one fluid, each with different characteristics. The mesh configuration need not be static over the duration of the simulation. Movement of the body, or a portion of it, can be effected by translating, rotating, or deforming all or part of the mesh. For rotating bodies or elements the appropriate axes of reference may be rotated or accelerated.

3.2. Discretized Solution Methods

The general nature of most CFD programs means that, before computation can take place, the physics regime of the problem must be defined. Amongst other things, the model types for the compressibility or otherwise of the fluid, speed of flow, turbulence and viscosity must be chosen and parameters such as boundary and initial conditions set. Whether the CFD solver is based upon the full Navier-Stokes equations, the Reynolds-averaged Navier-Stokes equations (RANSE), or the simplified inviscid-flow Euler equations, the constituent partial derivatives of these equations must be discretized in time to enable a numerical solution to be found. The oldest and simplest method is the 'Finite Difference' approach, which uses the Euler derivative approximation to discretize the partial differential equations. The accuracy of this method can be quite dependent on the uniformity of the grid spacing, so it is not commonly used. 'Spectral' methods, as the name suggests, apply frequency transformations to approximate the governing equations, but are not in widespread use. 'Finite Element' techniques have been in use in computational structural analysis for some time, so it is not unsurprising to find their occasional use in CFD applications. Although more accurate, in a formal sense, than other methods, finite element approaches impose a larger computational burden, which has restricted the use of this method. The most popular technique in use is the 'Finite Volume' method, which computes the values of the conserved variables averaged across the volume of each cell. An advantage of this method is that it can be used with an unstructured mesh, being particularly effective in situations where the mesh deforms to follow interface movements. Various strategies exist to optimize the convergence of the iterative numerical processes, which are particularly useful in cases where the mesh configuration changes during the simulation. 'Relaxation' methods form one class of such strategies, with the aim of rapidly reducing the residual errors inherent in any discretized procedure. Other approaches use different sizes of time-step for different physical domains to avoid unnecessary calculation.

3.3. Multiple Fluids and Free Surfaces

Most CFD solvers can accommodate multiple 'phases' or fluids, and the definition of an interface, or free surface, between them. The movement of the free surface is generally driven in one of two ways. The free surface can be set up in an initially unstable state and the model then predicts how equilibrium is reached. This 'sloshing' model is used to simulate flow between tanks at different levels and incursions of fluid through a breach. Alternatively, the free surface can be driven by the movement of a boundary, such as a wall or immersed object. This 'splashing' model is used to simulate the launching of objects into a fluid. Modelling the behaviour of the free surface is done in several ways. The Volume-of-Fluid (VOL) method assigns an extra identification variable, with an associated transport equation, to each cell. The value of this variable designates to which fluid the cell belongs, and the cell boundaries where the value changes designates the position of the free surface. The VOL method has the drawback of not being able to cope with non-linear effects like breaking waves, which require multiple surface heights at a single location on the horizontal plane. Methods which can cope with these effects include the 'particle-on-surface' or 'marker-cell' technique, which tracks the trajectories of selected cells or cell-centres on the free surface. The drawback with this tactic is that it is computationally expensive. Another line of attack is the 'scalar equation' or 'level-set-function' method, which determines the distance to the free surface for each cell. The free surface is, therefore, where the cell distance is zero, a condition of the equation which may be met by several cell heights. With each of these methods to find the variation in the free surface it is advantageous to refine the mesh

around the surface, but it is usually very difficult to predict where the surface will be at any point in the simulation. In cases where the two fluids have widely differing densities and viscosities (e.g.: water and air), it is possible to create the mesh only for the denser fluid and adapt the mesh configuration for the computed movement of the cells at the surface. This approach, however, is another which cannot handle steep or breaking waves, due to the distortion of the mesh which would occur.

3.4. Parallel Processing

CFD techniques are quite well-disposed to parallel processing, in which a large number of the computational tasks required can be performed simultaneously, thereby reducing the length of time taken to obtain a solution. The most popular strategy for parallelizing the problem is 'domain decomposition', noting that 'domain' in this context can be totally different to the 'domains' mentioned above. The processing domains are sub-divisions of the computational mesh, each being allocated to an available computer processing unit (CPU). These domains may be a single mesh divided into several domains, or several, maybe separate, mesh components included in a single domain. The partition of the computational mesh can be arbitrary, but is usually done on a geometrical basis, by dividing the mesh along an axis of reference, say, or on a physical basis, by dividing the mesh according to common properties. The computation proceeds simultaneously on each CPU with data transfer across domain boundaries (between CPUs) effected by the data communication and synchronization controls within the program. Data exchange may be done on each iteration, or after several iterations, depending on the requirements of the problem solution and governed by the processing management functions of the program. Parallel-processing versions of CFD programs are used for the largest and most complex problems, from reactions at an atomic level to the collision of galaxies.

4. Hydrodynamic Software Packages

Hydrodynamic analysis software packages contain many common features, organized in similar arrangements. A brief overview of the structure of a 'model' package is used here to provide a framework in which the assorted features of the various packages can be presented in a clear and concise manner. The 'model' program contains all the features of all the packages surveyed, although it should not be inferred that the software is necessarily constructed in this way. The 'model' hydrodynamic analysis package consists of three main blocks, as shown in Figure 4.1:

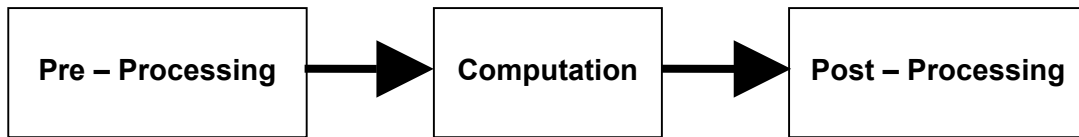


Figure 4.1: Package Structure.

The basic functions of the 'Pre-Processing' block is to prepare the geometric definition of the subject body in a format compatible with the 'Computation' block, which then performs the required calculations. The results are then presented to the user in various forms by the 'Post-Processing' block. A more detailed picture of the available 'Pre-Processing' functions is given in Figure 4.2.

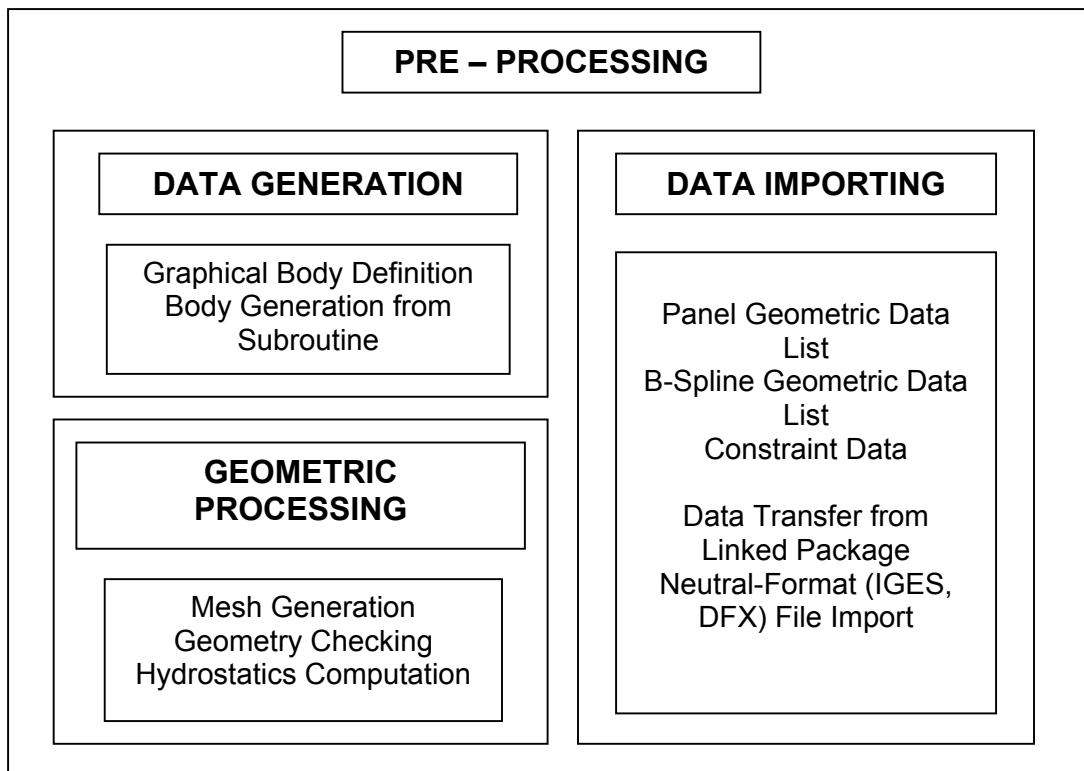


Figure 4.2: The Pre-Processing Block.

Several packages contain an in-built drawing facility which provides a means of defining the body geometry graphically. Alternatively, the geometric data may need to be imported using files of a specific format, generated by a separate program. Usually this separate program is an in-house product, but some packages can interface with other manufacturers' software that provides translation of neutral-format files from general CAD packages. Constraints on the motion of the body, such as those introduced by mooring forces or additional damping, may be input along with the body geometry, so that their effect on body motion is taken into account during computation. A certain amount of processing of the geometric data is then carried out. At a minimum this consists of the calculation of hydrostatic values such as centres of mass and buoyancy and water-plane stiffness. Programs which allow the geometry to be defined or imported in line-drawing form provide mesh-generation and editing facilities to convert the body surface model into panels. This processing may include checks on the geometry definition for slender elements, should alternative procedures for the analysis of these be included, or for possible problem areas such as discontinuities or areas of excessive convexity.

With the geometric data arranged appropriately, it is then passed to the 'Computation' block, so that calculations may proceed. The scope and variety of the available 'Computation' functions are indicated in Figure 4.3. The computation immediately falls into one of two categories, depending on whether frequency-domain or time-domain analysis is being performed. The frequency-domain calculations may also be divided into categories according to the order of the problem, that is, whether the wave input is a single sinusoid, or two interacting sinusoids. The fundamental hydrodynamic analysis program consists of a basic first-order solver which calculates parameters such as added mass, damping, excitation forces, motion amplitudes and drift forces for a single body. These functions are universal to frequency-domain analysis packages. Where programs provide solutions for the second-order problem, these are also for a similar basic case.

In addition to the basic hydrodynamic parameters, most programs derive extra values when solving the first-order problem. Amongst these are solutions for a finite water depth, for multiple bodies, or for a body in proximity to a wall. The distribution of pressure over the panelized surface, implicit in the derivation of other parameters, may be calculated directly. Solutions at a number of frequencies are sometimes calculated to produce response spectra applicable to commonly-used spectra for irregular waves. Alternative methods of calculation may also be included to remove the effects of irregular frequencies, or to accommodate slender elements, forward motion, or drag.

Time-domain calculations also fall into two groups. For large-scale effects the variations in waterline are ignored and calculations are performed over a constant, mean wetted surface. Radiation and diffraction forces are calculated this way, along with the motions and loading of the body, and the fluid velocity and pressure. For some calculations the change in waterline is critical, so the body is rotated and translated appropriately at each time step to produce the instantaneous wetted surface. Various non-linear exciting and restoring forces are calculated using this approach. Additional values are calculated by some packages for effects such as roll-damping and maneuvering, and to take account of the interaction of multiple bodies.

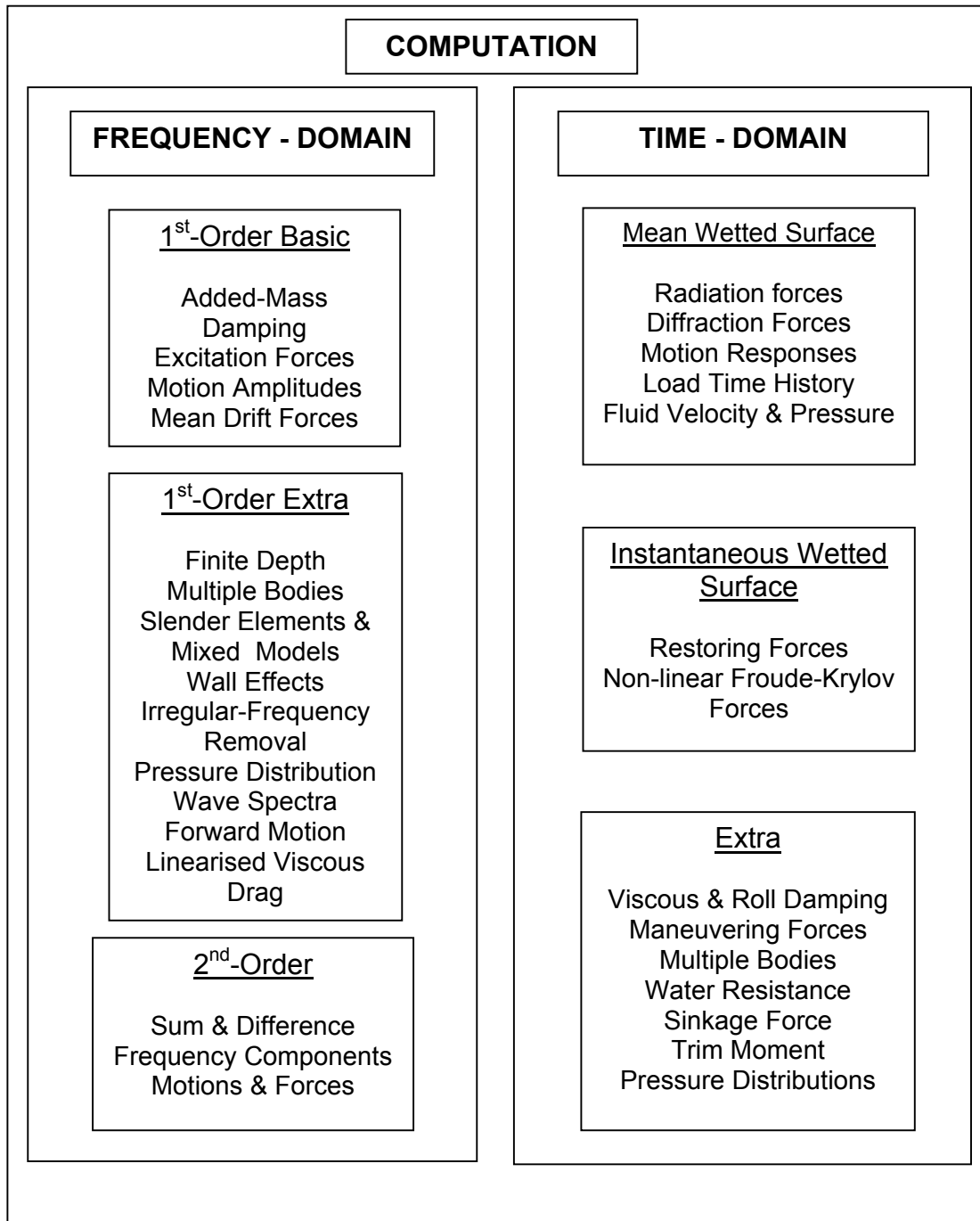


Figure 4.3: The Computation Block.

The results obtained from the calculations are then processed for presentation to the user. These 'Post-Processing' functions are outlined in Figure 4.4. Included here are some basic structural loading calculations that allow the user to estimate stress distributions on the body. Some packages provide for data transfer to specialist structural analysis software, again usually in-house, for a more comprehensive appraisal of the loads involved.

The data output itself, at its most basic, merely entails listing the values in text files. More ambitious programs provide plotting facilities to display the results in graphical form. As well as the standard hydrodynamic parameters, some packages offer the display of free surface elevation and diffracted wave plots. Furthermore, there may be in-built provision for the animation of plots to show body motions or other variations over time. Some packages also have the capacity for statistical analysis of the output data.

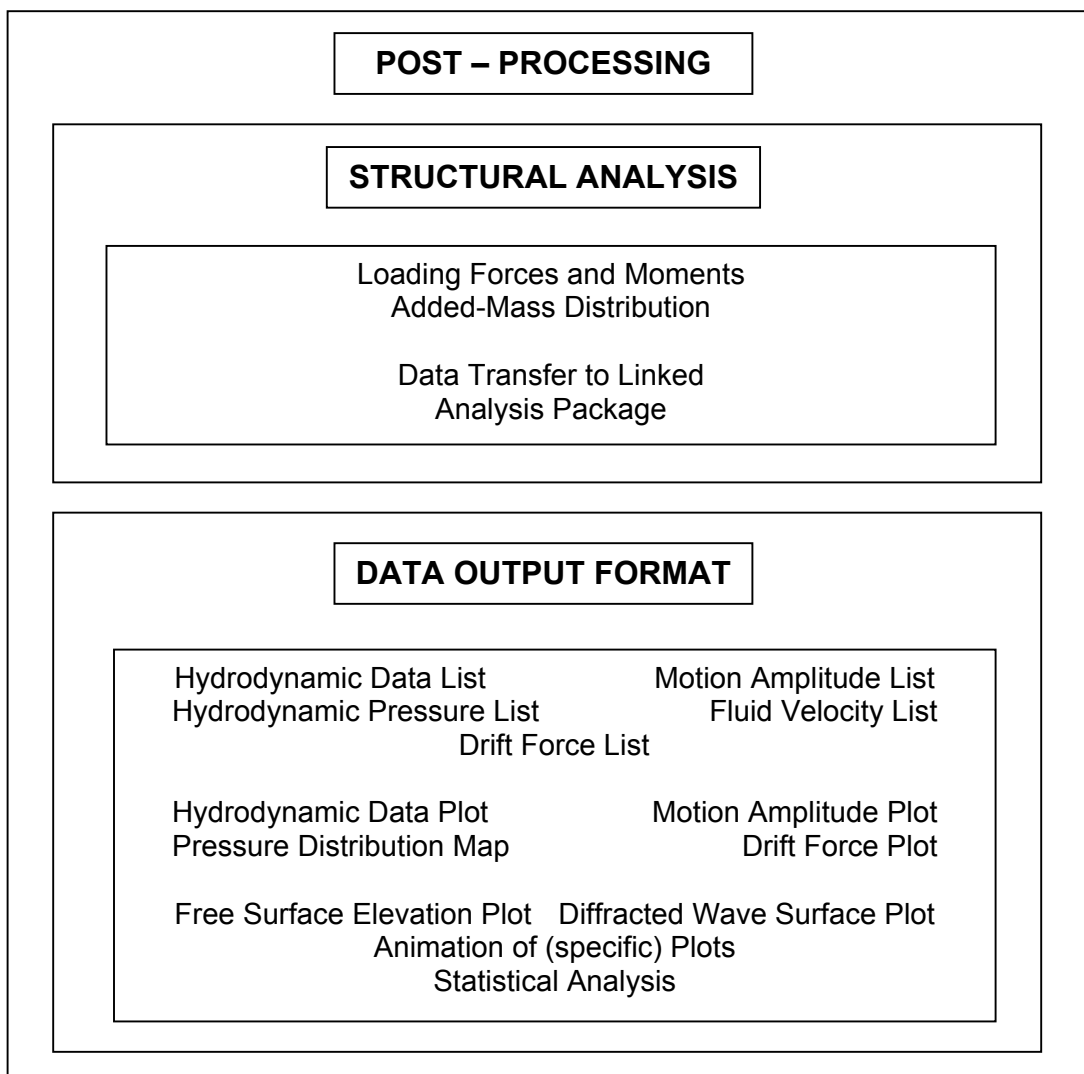


Figure 4.4: The Post-Processing Block.

4.1 AQWA (Century Dynamics Ltd.)

Century Dynamics is an international developer and supplier of engineering and scientific simulation software. Founded in 1985, Century Dynamics operates internationally throughout the Americas, Europe, Asia, Africa and the Middle East, and has offices in Concord, California, Horsham, England and Houston, Texas.

Developed by WS Atkins, AQWA has been a leading analysis system for the assessment of marine and offshore structures for over 20 years. AQWA was the first commercially available integrated system for hydrodynamic analysis, and has been used in many applications, including aerospace, military, naval, and the petrochemical industry.

Machine Requirements: PC computers running Microsoft-WINDOWS 9x/NT or later. The facility to read AQWA results from UNIX workstations is under development.

Licensing Arrangements: AQWA consists of various modules, which may be licensed in different combinations. The full AQWA-Suite can be licensed commercially on an annual basis at UK£12,600 p.a. (excluding coupled cable dynamics), or at UK£16,310 p.a. (including coupled cable dynamics). Paid-up commercial licences are also available for the same packages at UK£31,500 or UK£40,775, respectively. Details of teaching and academic licenses, and the use of the software therewith, are obtainable on request. Support services are provided, although, with some licenses, these incur a fee.

User Interface: The AQWA Graphical Supervisor is an interactive system which encloses the AQWA hydrodynamic analysis suite, runs the main AQWA programs, oversees database editing, data analysis and graph presentation.

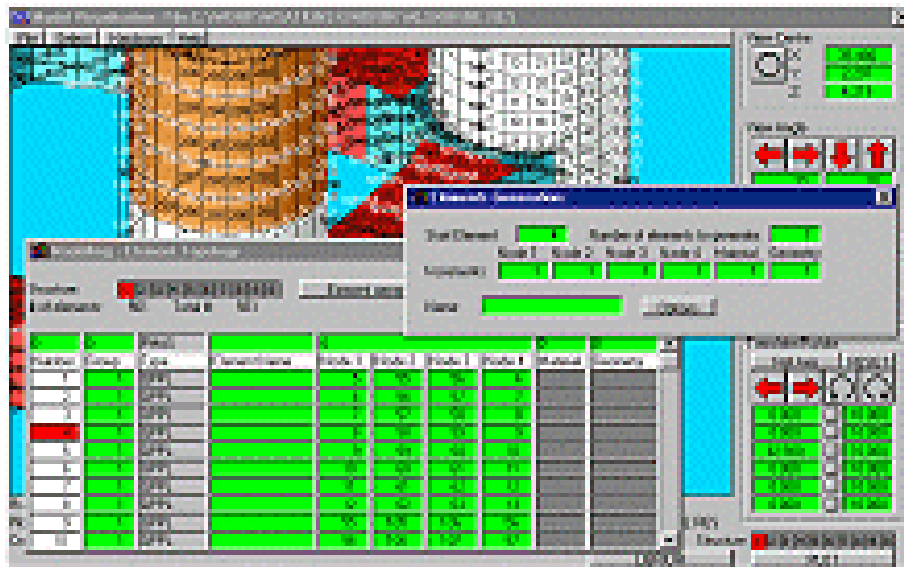


Figure 4.5: The AQWA Graphical Supervisor (copyright Century Dynamics Ltd.).

Noting that the AQWA Graphical Supervisor interface accommodates both data input and output, the pre-processing functions provided by AQWA can be summarized as shown in Figure 4.6. Modelling of the body geometry is graphical in a window-based environment, with automatic mesh generation capability. Mesh modification and manipulation, where refinement is required, is possible. Graphical user interfaces (GUIs) are used for setting global parameters, such as water depth, environment specifications, such as regular wave or wave spectra input, and the dynamics of constraints, such as mooring lines, thrusters, or tethers. Geometric data may be imported into AQWA, but requires the use of a separate translation program, FEMGV (a FEMSYS product), to convert the neutral-format CAD package output into a form that AQWA can use.

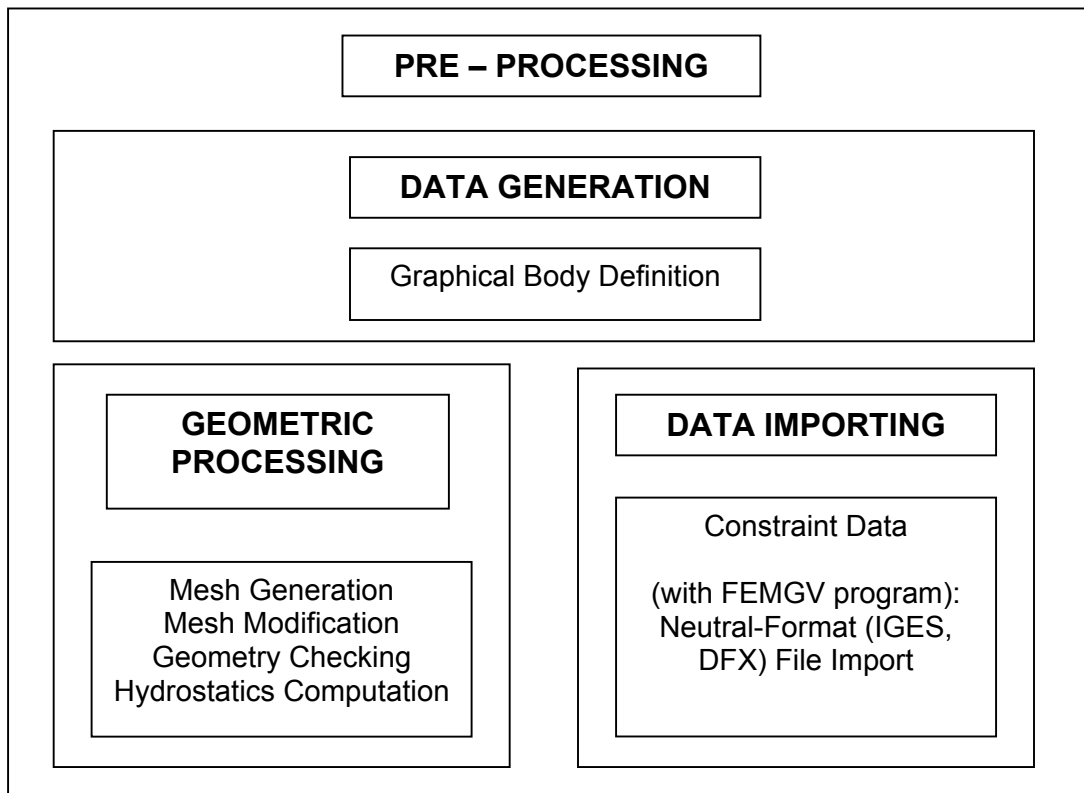


Figure 4.6: AQWA Pre-Processing Function Summary.

The calculations performed by AQWA can be summarized in the form shown in Figure 4.7. The AQWA package consists of several modules covering different areas of the computation. The principal ones are: AQWA-LINE, AQWA-FER, AQWA-NAUT, AQWA-DRIFT and AQWA-LIBRIUM. The AQWA-LINE module performs a classical 3-dimensional diffraction/radiation analysis of wave action around a single floating body to obtain the diffraction force, added mass and radiation damping on the body. Additionally, these values are combined with the body's mechanical mass, viscous damping, and any mooring stiffness, to deduce the body's motions in all six degrees of freedom and the associated steady wave drift forces. The module is also able to take account of hydrodynamic interaction between a floating structure and an adjacent fixed structure, and has been recently upgraded to take account of shallow water effects.

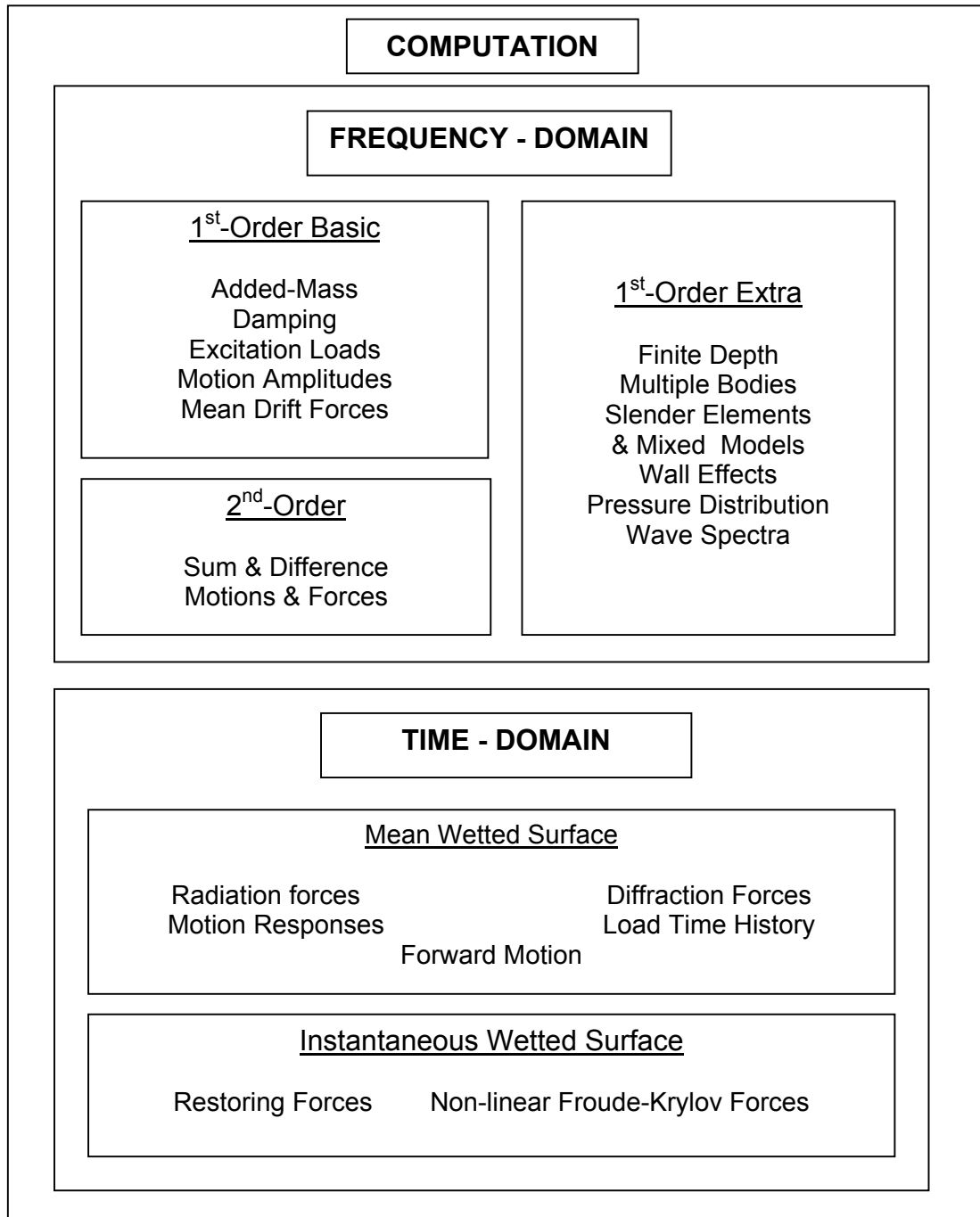


Figure 4.7: The AQWA Computation Summary.

The AQWA-FER module computes the RAOs for the motions and loads at specified points and deduces the linear response spectra to a given sea spectrum. From this the significant and extreme linear response is calculated. The mean value and spectrum of second order forces on the assembly can also be calculated, giving the mean, significant and extreme second order responses. The AQWA-NAUT module computes the motions of a body, or each floating body in an assembly, in a regular wave train, including the effects of mooring lines and articulations. The calculation is done as a time history, so

non-linear mechanical effects can be included. Non-linear hydrodynamic effects related to the varying immersion of the structures is modelled at each time step and Morison loading on tubular elements is also calculated. Current and wind loads can also be modelled from pre-defined empirical coefficients. The AQWA-DRIFT module calculates body motions in an irregular wave train of any given spectrum. Separate bodies can be linked by articulations or mooring lines, which are modelled in a fully non-linear way. Wind and current loads from any direction are included, and all loading calculations take account of the changing headings of the various floating bodies. Wave frequency motions are not assumed to be independent of mooring forces, so that complicated non-linear mooring snatch phenomena can be accurately simulated.

The AQWA-LIBRIUM module calculates the steady drift forces in an irregular wave train of any given spectrum. Wind and current loads from any direction are included, and all loading calculations take account of the changing headings of the various floating bodies. This module also computes the buoyancy forces at any position of the body, or bodies. Additional modules are available. These calculate the motion of and stresses in tethers and risers, the motions of and loads in certain classes of structures, the dynamics of coupled cables, and hydrodynamic pressures, in a form which can then be transferred to the structural analysis package ASAS (a Century Dynamics product).

The AQWA Graphical Supervisor (AGS) interface also accommodates data output, the post-processing functions summarized in Figure 4.8.

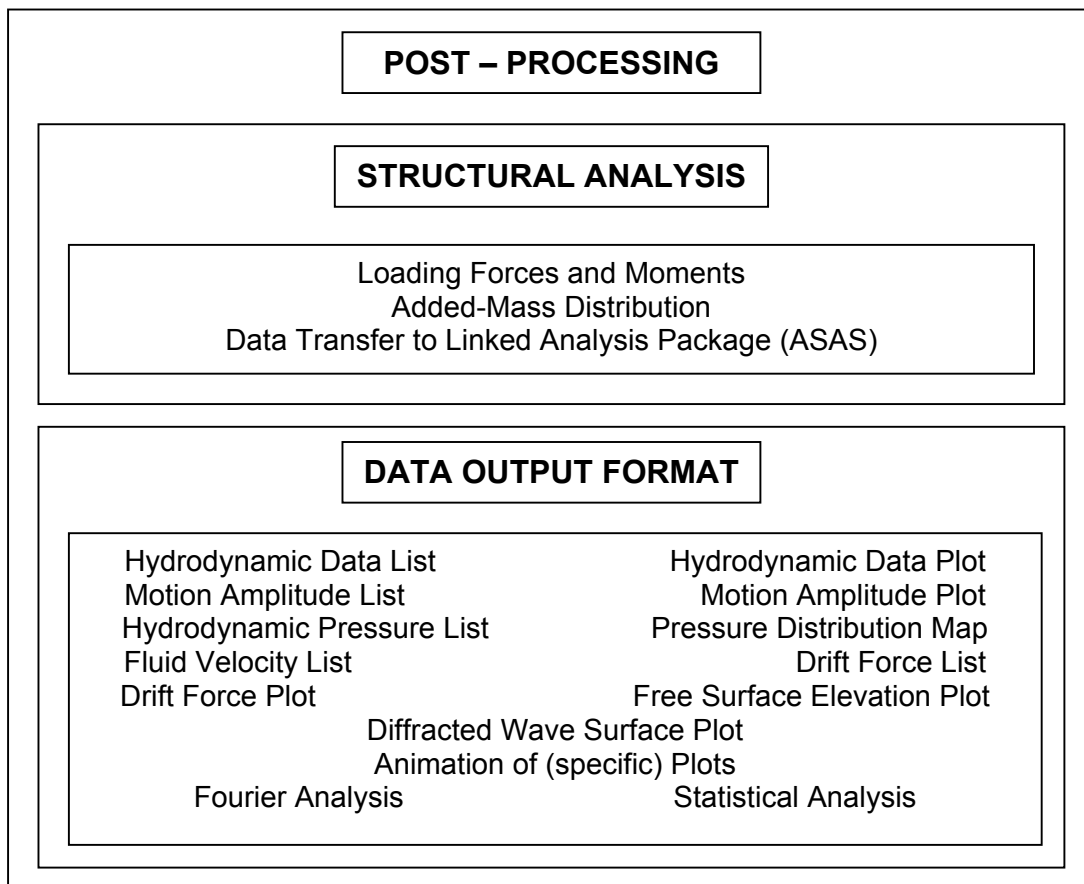


Figure 4.8: AQWA Post-Processing Summary.

The AGS interface also includes facilities for colour graph display, graph merging and zooming, and statistical processing of the data. The incident, diffraction, radiation and hydrostatic components of wave pressure can be plotted over the model surface and displayed as an animation in real time. The diffracted wave surface can be viewed either as a contour plot or superimposed around the model in a plot window, both options can be viewed in real time as an animation. The bending moment and shear forces can be calculated either under static (still water) conditions or with a user-defined sea state.

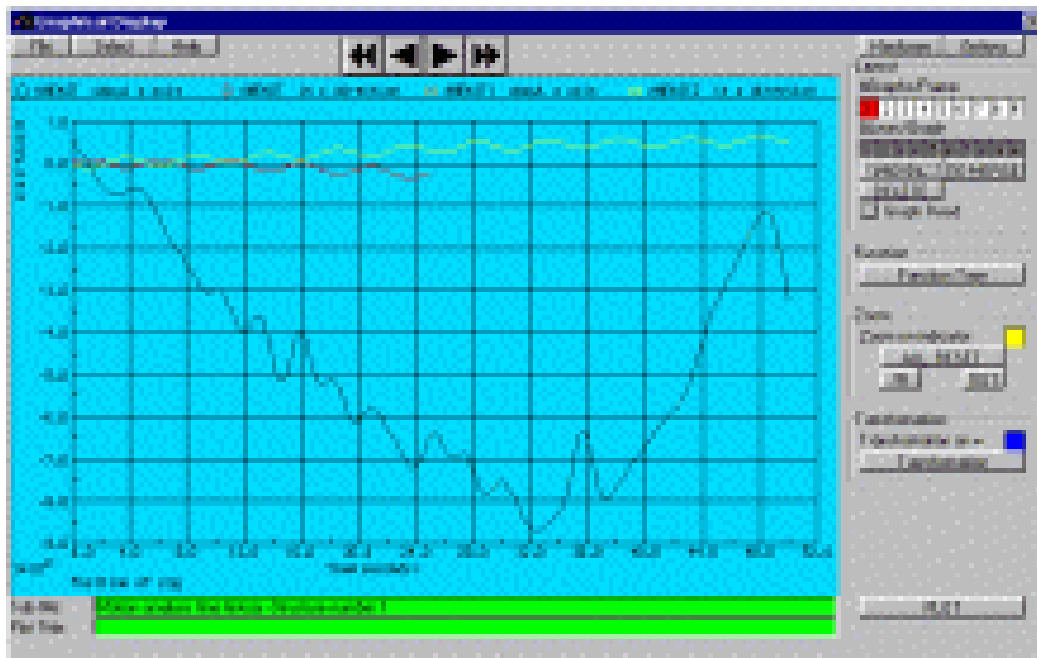


Figure 4.9: AQWA AGS Graph Output (copyright Century Dynamics Ltd.).

4.2 HYDROSTAR (Bureau Veritas)

Bureau Veritas is a service company specializing in quality, health, safety and environment management and social accountability, with a network that covers 140 countries and includes 600 offices and laboratories. It offers an extensive range of technical services and solutions in the fields of certification, conformity assessment, consulting and training. The Bureau Veritas Marine Division offer a group of technical solutions under the name VeriSTAR. VeriSTAR Offshore is made of several components, each specialized to provide solutions to offshore-specific problems and interface with each other in a thoroughly streamlined manner. HYDROSTAR forms part of this suite and is a 3D diffraction and radiation frequency-domain analysis program for 1st- and full 2nd-order wave loads.

Machine Requirements: No manifest specification. From publicity material it appears to be 'windows'-based.

Licensing Arrangements: Licensing details have not been ascertained.

User Interface: A graphical interface is provided by the VISU4D program (a Bureau Veritas product), as shown in Figure 4.10.

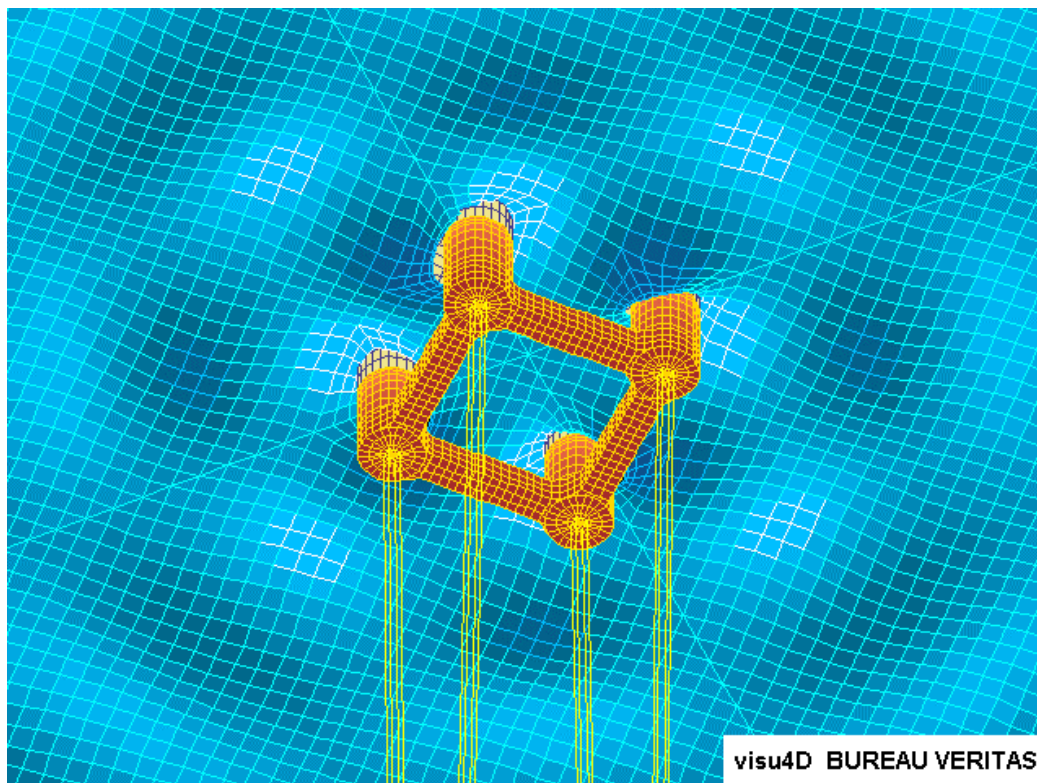


Figure 4.10: Visu4D mesh visualization in HYDROSTAR (copyright Bureau Veritas).

The pre-processing functions provided by HYDROSTAR are summarized in Figure 4.11. Data of the body geometry can be input as line or frame designs, with mesh generation and checking performed by HYDROSTAR, viewed through VISU4D.

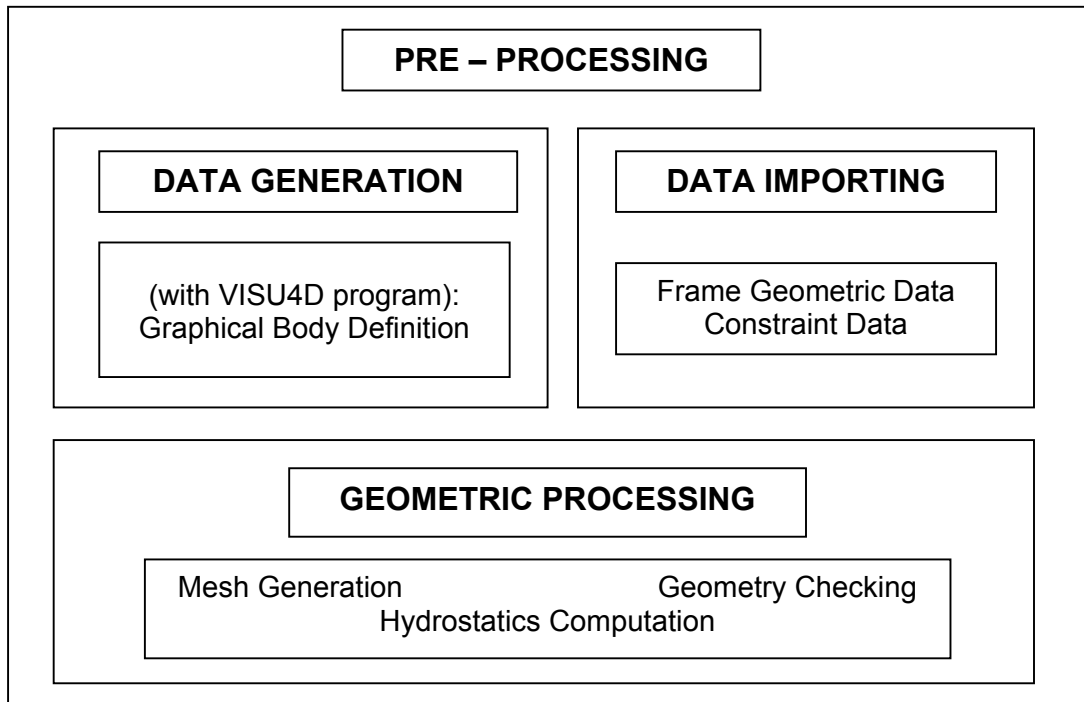


Figure 4.11: HYDROSTAR Pre-Processing Function Summary.

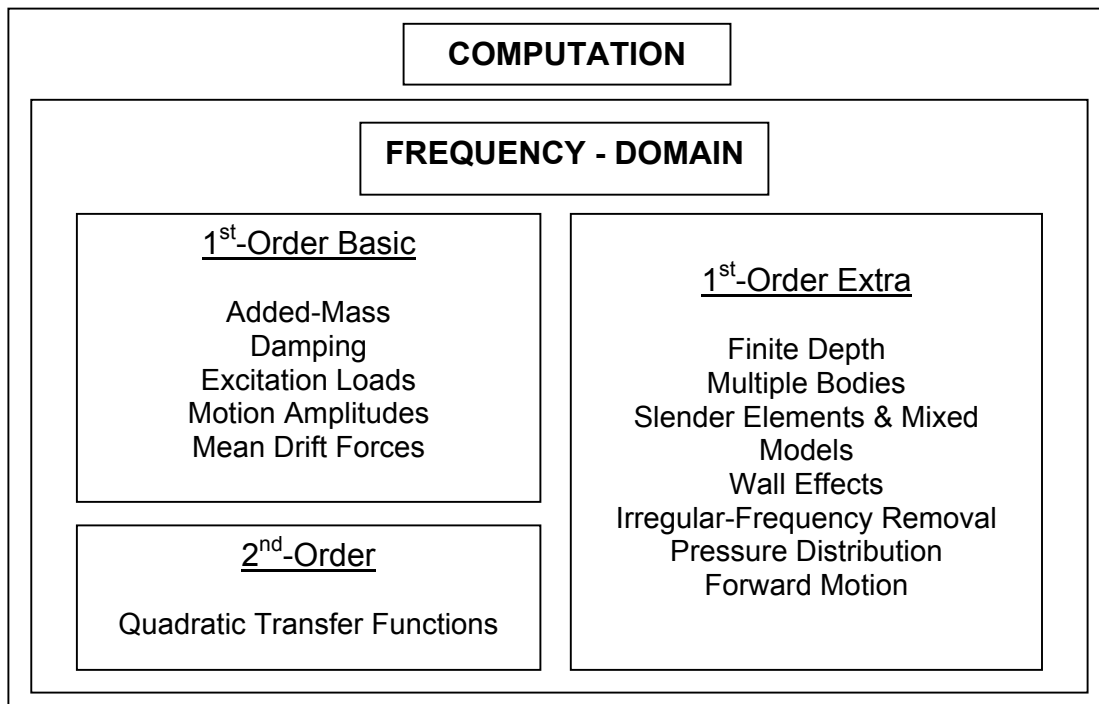


Figure 4.12: The HYDROSTAR Computation Summary.

A summary of the values computed can be seen in Figure 4.12. First-order linear analysis provides hydrodynamic coefficients, response amplitude operators and diffraction and drift forces at the wave frequency. Morison's formula is applied to slender elements, and mixed models are supported. The computation takes account of viscous damping, and the effect of shallow water, and of adjacent walls. Currents and moderate forward speed effects are accommodated by encounter frequency approximation. The hydrodynamic interaction, motions and connecting forces between several bodies can be computed. When required, there is dismissal of irregular frequencies. Second order diffraction-radiation analysis is used to obtain the quadratic transfer functions.

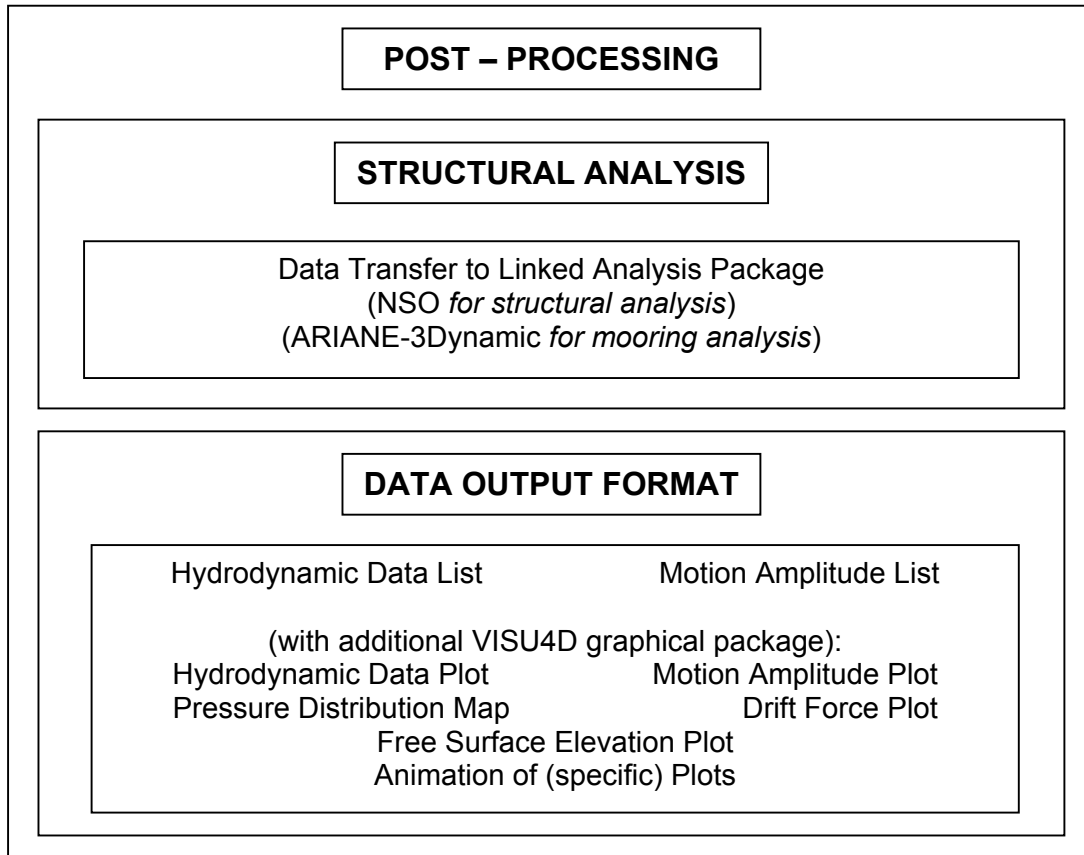


Figure 4.13: The HYDROSTAR Post-Processing Function Summary.

The post-processing functions carried out by HYDROSTAR are summarized in Figure 4.13. The graphical interface with VISU4D provides visualization of the results of the computations. Charts include plots of the motions of each body mode, free surface elevation, dynamic pressure on the hull, fluid kinematics and internal dynamic forces on structures in waves. The motions can be represented by animation of the floating body with the generated wave trains, along with the effects of diffraction. The results of several analyses at different wave frequencies may be linked together. Additional analysis is facilitated by interfaces with the structural analysis software New Strudl Offshore (NSO) and with the mooring analysis software ARIANE-3Dynamic (*both Bureau Veritas products*).

4.3 MOSES (Ultramarine, Inc.)

Ultramarine, Inc. offer engineering services to the technical community, concentrating on engineering design and analysis, and on developing custom-built computer programs for such purposes. Previously, Ultramarine developed the OSCAR / OTIS[®] system for the design and analysis of floating structures, now replaced by the more general-purpose Multi-Operational Structural Engineering Simulator (MOSES[™]). MOSES is fundamentally different in concept to other hydrodynamics packages, inasmuch that it is a computer language in its own right, rather than a ready-made program in a standard computer language, such as FORTRAN or C.

MOSES is a computer language intended for the modelling, simulation, and analysis of the stresses which occur in marine circumstances. The MOSES language is assembled around a proprietary database manager explicitly created to store and retrieve scientific models and simulation results. MOSES combines both stress analysis and hydrodynamic simulation into a single program, and the subject model is constructed in a particular way so that both types of analysis may be carried out. To calculate the hydrodynamic forces on a system, three hydrodynamic theories are applied. These are Morison's equation, two-dimensional diffraction theory, and three-dimensional diffraction theory. MOSES has been used in many offshore projects and is popular with the petroleum industry.

Machine Requirements: Computers with INTEL Pentium or Celeron or AMD Athlon processor, running WINDOWS or LINUX, or with a SPARC processor, running SOLARIS 2.x.

Licensing Arrangements: Ultramarine software can be obtained on one of three types of lease. A perpetual lease allows the continuing use of the software, and maintenance for twelve months, after the payment of an initial amount. Maintenance includes updates of the program, minor enhancements and user support, and is subject to a fee after twelve months. Secondly, a yearly lease permits the use of the program for a period of one year following each annual payment. Maintenance is included with this lease. Thirdly, the user may obtain a monthly lease, although the minimum lease period is six months. The price for a lease also varies according to the type of machine on which it is installed, the number of users who can execute the program simultaneously, and the capabilities one wants to include. Additional capabilities can be added later, at an incremental cost.

User Interface: There are three optional user interfaces. The default interface is a "window", which provides the user with a display area, a tool bar, a scroll bar, and a command line box. Alternatively, a "terminal" interface may be used, which comprises MOSES running in an existing terminal or console window provided by the operating system. The third option is the "silent" interface, which is a "terminal" interface which only produces output as directed by programmed commands.

The pre-processing functions provided by MOSES are summarized in Figure 4.14.

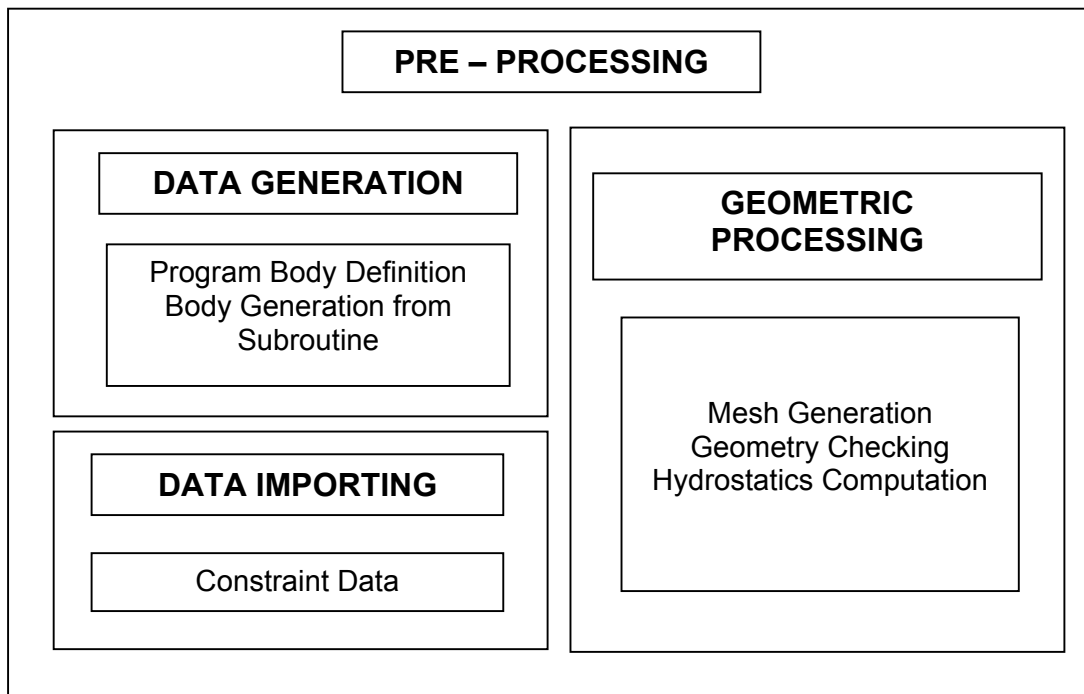


Figure 4.14: MOSES Pre-Processing Function Summary.

The creation of the model of a structure is treated as the amalgamation of one or more 'hulls', or large elements, and tubular and / or plate elements into a single body. Generation options and interactive graphics enable the modelling of unusual shapes. Models can be created from new, from user-defined macros, or from a library of vessels, to which the user may add. The modelling program permits the use of looping options, so that blocks of data can be entered automatically, rather than manually, of routines that execute only when data changes, and of globally-defined variables.

Hydrostatic, hydrodynamic and plate meshes are generated automatically. Coarse meshes may be refined, and the intersection, union and difference of defined polygons used in mesh generation are calculated. Constraints to motion such as mooring line dynamics or sling assemblies may be defined. Mooring lines can have many segments and act like non-linear springs or exhibit force elongation behaviour. Hydrostatic curves of form can be generated for a series of draughts and trim angles, showing, for example, displacement, waterplane area, or the location of centre of buoyancy. Single body equilibrium is found, given the load and ballast of the vessel. The equilibrium of connected multiple bodies automatically includes wind, wave and current forces.

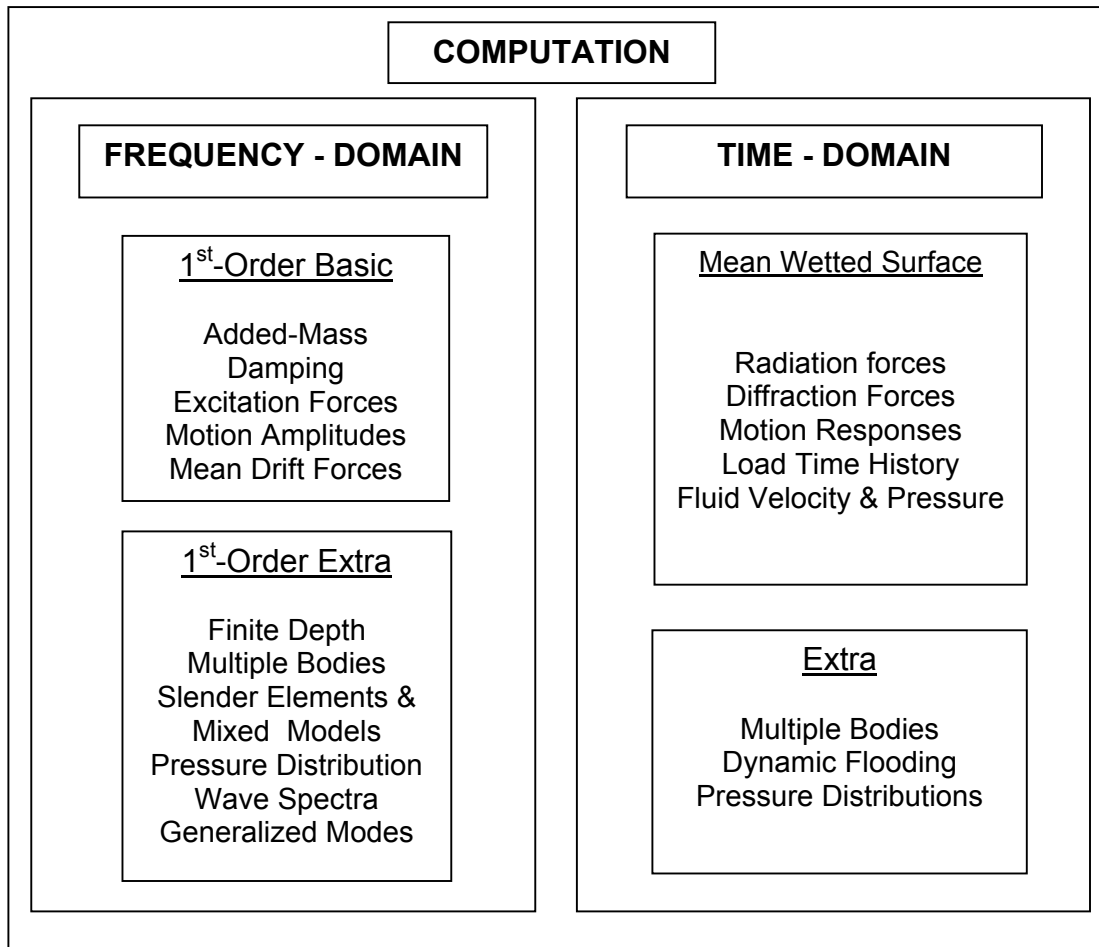


Figure 4.15: The MOSES Computation Summary.

The computations performed by MOSES are summarized in Figure 4.15. Frequency domain analysis is used to ascertain the response of a structure to the excitation of a set of waves of given period and direction. It is used to calculate added mass and damping, the pressures on the body and total forces and moments. Morison's equation is employed in the analysis of an arrangement of plates and tubes used to simulate a semi-submersible, for example. Strip theory is employed for traditional nautical hull shapes. Three-dimensional diffraction theory is used to capture 'bottom effects' and the interaction between multiple bodies. Response amplitude operators (RAOs) are calculated for the motion of any point on the structure, and also for the inertial forces and moments on bodies attached to the structure. Nonlinear, slowly-varying wave drift forces can be included in the frequency domain analysis. Generalized degrees of freedom can be used to consider the effect of deformation on the amount of buoyancy, on the frequency response and on the hydrodynamic interaction between two vessels

Time-history response simulation is performed by transforming frequency domain results into the time domain. The sea state can be composed of irregular waves, current, or wind, in any combination. The motions and interaction of multiple bodies can be analyzed. The simulation may also include the dynamic flooding of tanks, taking into account both value properties and the actual differential head.

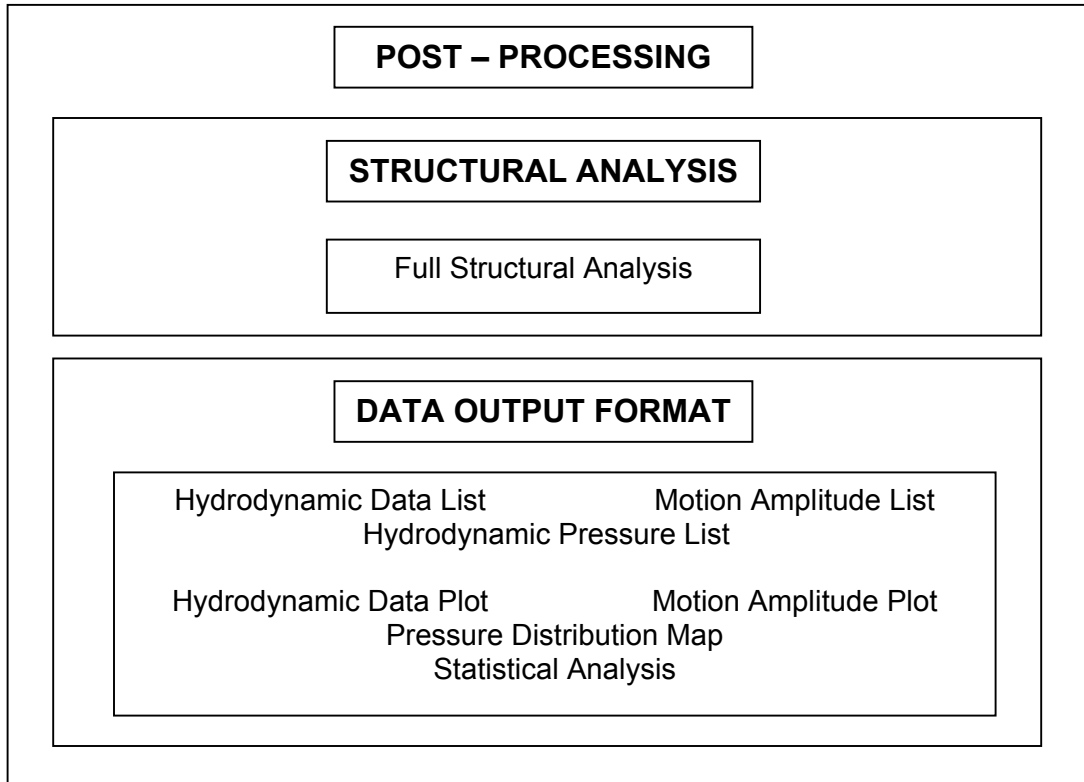


Figure 4.16: The MOSES Post-Processing Function Summary.

The post-processing functions provided by MOSES are summarized in Figure 4.16. The interactive graphics capabilities of MOSES allow the user to generate graphs of results and three-dimensional views of models. The post-processing of results can be customized by the user. Statistics of results can be computed using RAOs, and an assortment of spectra, such as ISSC, JONSWAP, or user-defined.

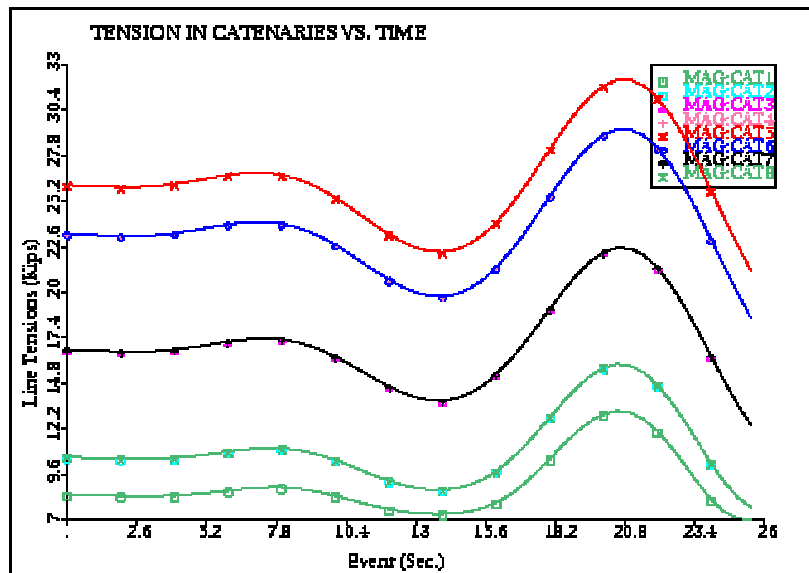


Figure 4.17: A time simulation plot in MOSES (copyright Ultramarine, Inc.).

4.4 NEPTUNE (Zentech, Inc.)

Zentech Inc., is an engineering consulting and software development company with headquarters in Houston, Texas, specializing in providing services to the marine, petroleum, and construction industries. Zentech also offers a variety of software packages which are used by engineering firms, drilling contractors, oil companies, classification societies, and shipyards worldwide.

NEPTUNE is a 3D constant panel diffraction-radiation frequency-domain program designed to analyze structures ranging from simple barges to complex semi-submersibles.

Machine Requirements: IBM compatible PC running Microsoft WINDOWS 95 or later, with maths co-processor and at least 16 Mb of memory.

Licensing Arrangements: NEPTUNE may be obtained in two separate modules or as a whole. The pre- and post-processing modules can be licensed for \$8,000, and the main computation module for \$10,000. The whole package can be licensed for \$15,000. Upgrades and technical support are free for 3 months, after which they are contracted for 15% of the original license fee. Alternatively the program may be leased for \$600 per month for a minimum 3 months.

User Interface: In-built graphical interface for model generation and review of results. An example body definition is shown in Figure 4.18.

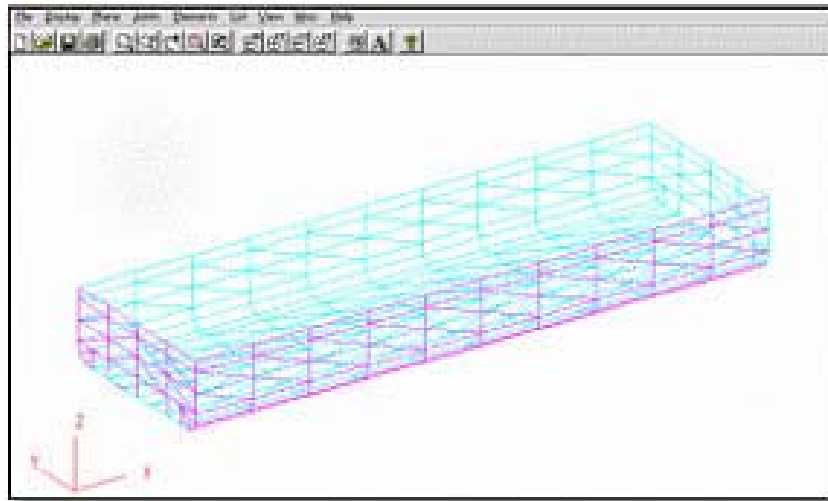


Figure 4.18: Geometry in the NEPTUNE graphical interface (copyright Zentech, Inc.).

The pre-processing functions provided by NEPTUNE are summarized in Figure 4.19. The body geometry is input via the integral drawing facility, with automatic mesh generation and editing. Full use is made of symmetry and mixed models may be defined using panels for larger components and Morison elements for slender components.

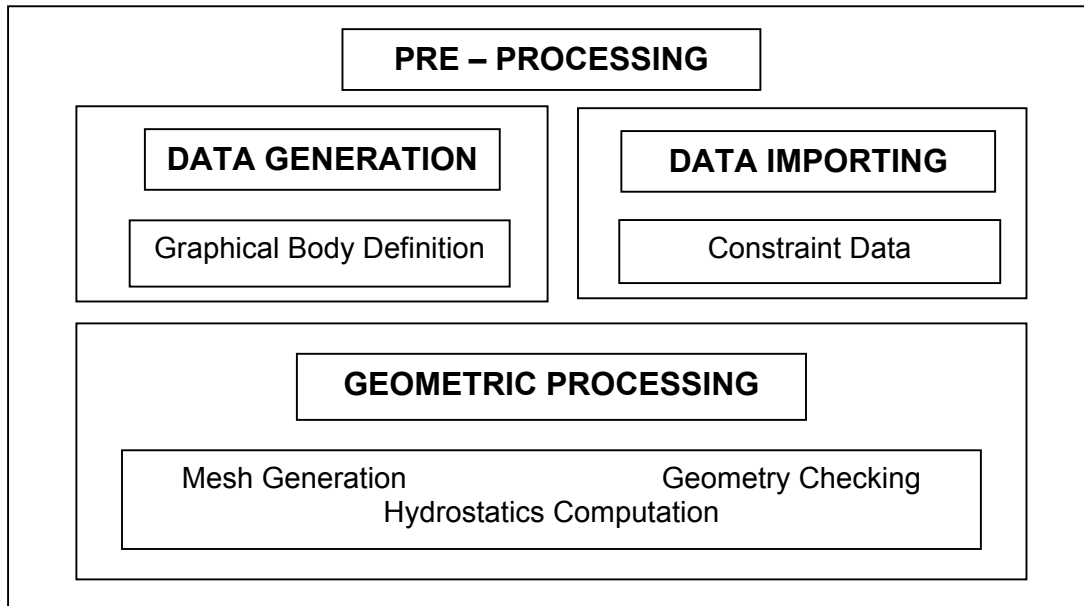


Figure 4.19: NEPTUNE Pre-Processing Function Summary.

A summary of the values computed by the diffraction-radiation analysis is given in Figure 4.20.

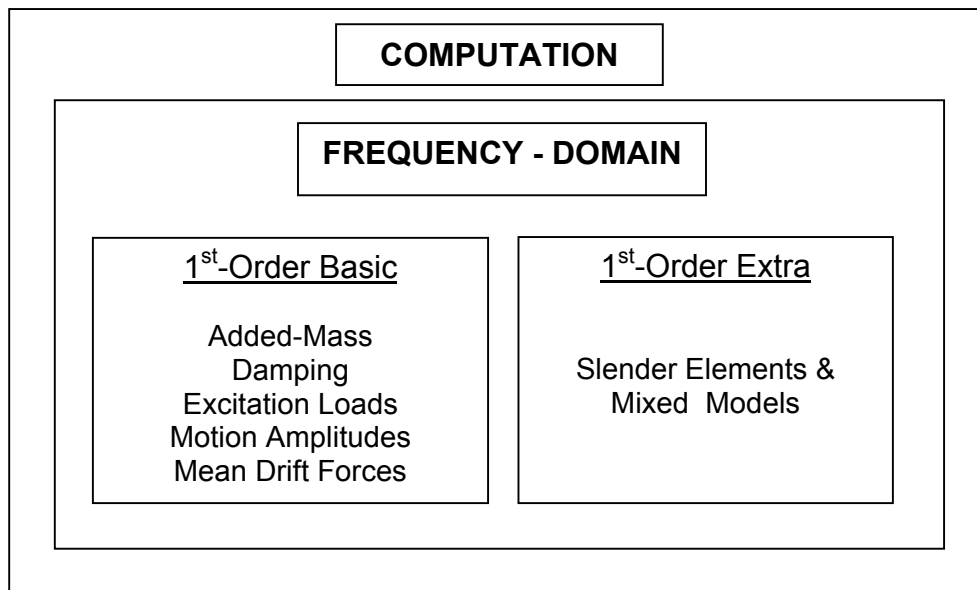


Figure 4.20: The NEPTUNE Computation Summary.

Basic values computed include hydrodynamic coefficients, response amplitude operators (RAOs), fluid pressure, motion, and free surface elevation, wave loads and drift forces. Morison's formula is applied to slender elements, and mixed models are supported. The computation takes account of static loads from wind, current and mean position, and also the effect of constraints such as moorings. Wave inputs may be regular or random.

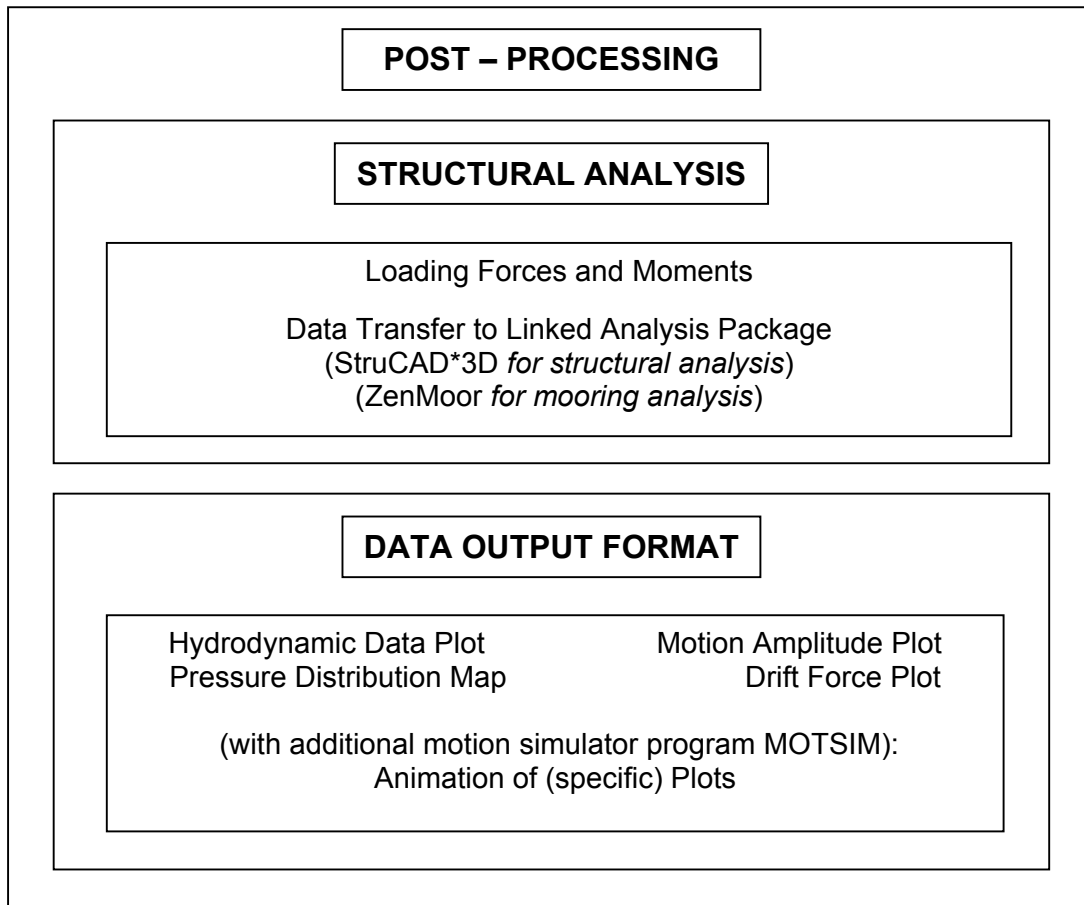


Figure 4.21: The NEPTUNE Post-Processing Function Summary.

The post-processing functions carried out by NEPTUNE are summarized in Figure 4.21. Loading calculations include the load balance and automatic load generation for structural strength and fatigue analysis, in addition to generating transfer functions for the load responses. The loads are mapped to the structural beam and plate elements of the body.

The graphical post-processing of results produces plots of the hydrodynamic coefficients, RAOs and pressure contours. NEPTUNE is fully integrated with the structural analysis program StruCAD*3D (*marketed by AceCAD software*), the mooring analysis program ZenMoor (*a Zentech product*), and the non-linear motion simulator MOTSIM (*an Offshore Structure Analysis (OSA), Inc. product*).

4.5 WADAM (DNV – Det Norske Veritas)

DNV is a leading international provider of services for managing risk, and is one of the world's leading classification societies, helping the maritime industry manage risk in all phases of the ships life, through ship classification, statutory certification, fuel testing and a range of technical, business risk, financial and competency related services. DNV have developed SESAM, a modular system consisting of pre- and post-processors, hydrodynamic and structural analysis programs.

WADAM is a general hydrodynamic analysis program for calculating wave-structure interaction for fixed and floating structures of arbitrary shape. It is based on three-dimensional radiation-diffraction theory employing a panel model; and the linearized Morison equation for beam models. The radiation-diffraction part of WADAM is based on software developed by Massachusetts Institute of Technology.

Machine Requirements: No manifest specification. From publicity material it appears to be 'windows'-based.

Licensing Arrangements: The Wadam program comprises three components, the 'Basic' (1st order module), the optional '2ORD' (2nd order module), and the optional 'NBOD' (multiple bodies module). In addition to these components, separate pre- and post-processing modules are required. The license can be a Commercial License, a Commercial Academic License (programs may be used for funded research with full support and maintenance service), or an Academic License (programs may only be used for teaching, or individual thesis work, with a very limited amount of support).

User Interface: SESAM-compatible graphical user interface, entitled 'Manager', a windows-based program for selecting and executing the appropriate programs, and managing the files involved from modelling to post-processing. Alternatively, WADAM is used as the computational engine within HydroD (a DNV product), which is an interactive tool used for the design of floating structures.

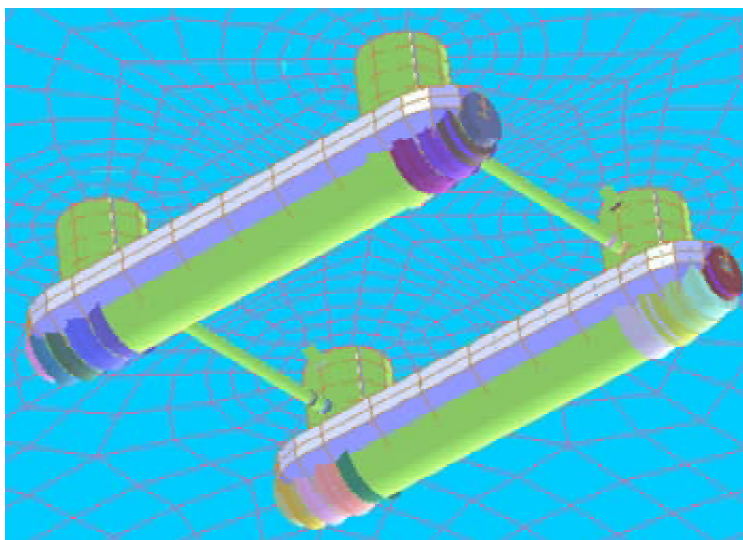


Figure 4.22: A view from the WADAM graphical interface (copyright Det Norske Veritas).

Noting that WADAM forms part of a suite of programs, some of which perform functions which are built into other software packages, the pre-processing functions provided by WADAM are summarized in Figure 4.23.

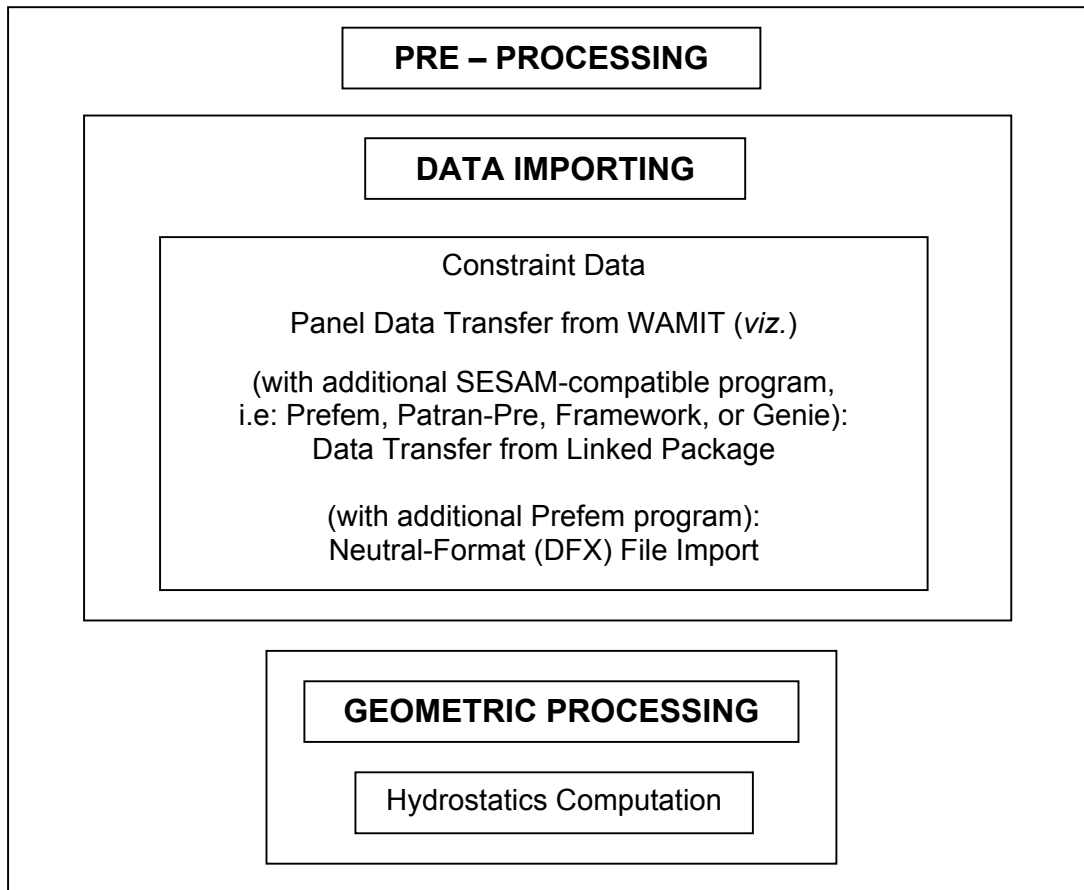


Figure 4.23: WADAM Pre-Processing Function Summary.

Models of significant dimensions are created as panel models in Prefem (*a DNV product*) or in Patran-Pre (*an MSC software product*). It is also possible to import panel models from Wamit (in the *.GDF file format). Slender elements are analyzed using Morison's equation in linearised form requiring a beam model, created in Prefem, Preframe, Genie, or Framework (*all DNV products*). For a structure comprised of both slender and voluminous parts the two methods may be used in combination. By establishing a so-called dual model the advantages of both methods may be utilized. WADAM will then include both radiation and diffraction effects and viscous effects. For shell or solid element analysis WADAM provides algorithms for adapting surface pressure loads of a normally coarse panel mesh to a refined finite-element mesh.

A summary of the values computed by the first-order radiation-diffraction and / or Morison's equation analyses is given in Figure 4.24. The computation involves the calculation of global response variables, including hydrodynamic added mass and damping, wave excitation forces and moments, rigid body motions and constant drift forces and moments. Wave elevation and fluid kinematics are determined at specified points and the roll damping coefficient is derived using strip theory.

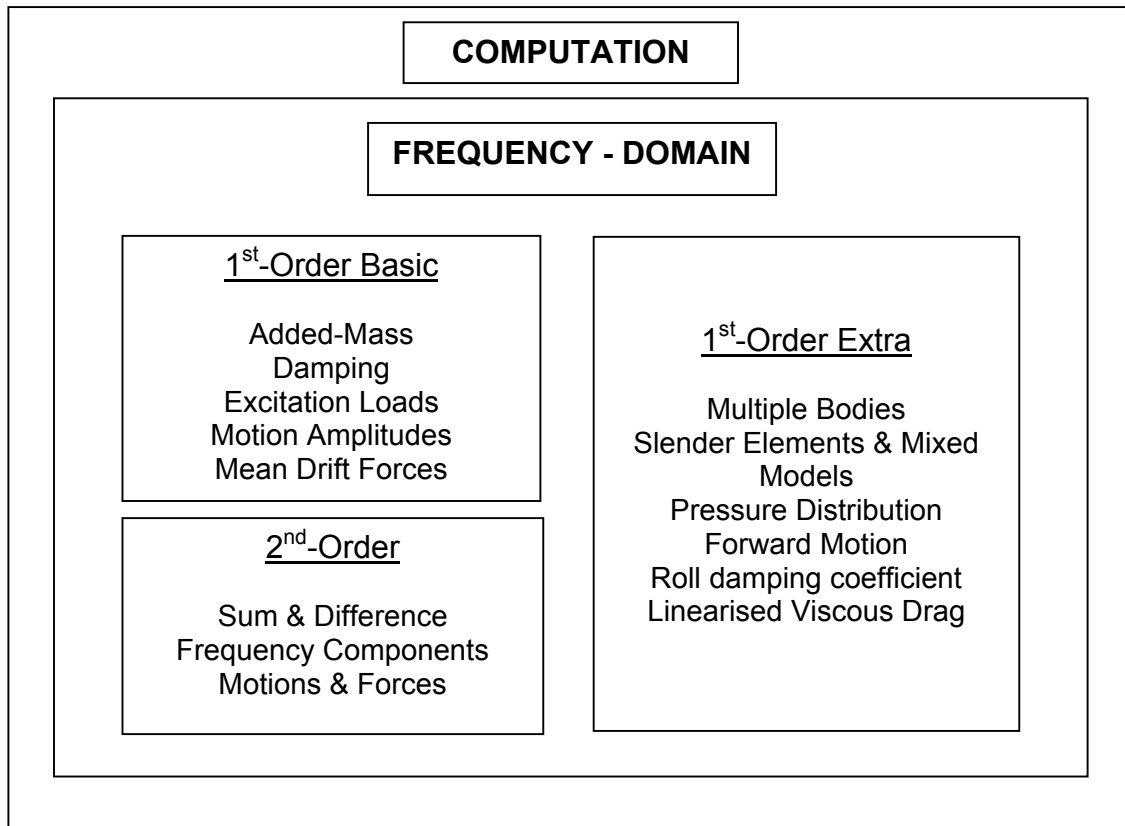


Figure 4.24: The WADAM Computation Summary.

WADAM uses two linearization methods for viscous drag. The first, equivalent linearization is appropriate for structural parts with typical dimensions greater than about one-fifth of the wavelength. The second, stochastic linearization predicts viscous forces more accurately and is therefore suitable for more slender structural parts. Second order sum and difference frequency forces and motions are calculated for bichromatic and bi-directional wave inputs.

The post-processing functions provided by WADAM, or associated software, are summarized in Figure 4.25. WADAM creates a hydrodynamic results interface file and a loads interface file. The hydrodynamic results interface file may be used when performing statistical post-processing, for example in Postresp (a DNV product), an interactive postprocessor for statistical processing and presentation of responses in frequency and time domains, which can be started from the HydroD user interface. This interface file may also be used directly as input to DeepC (a DNV product) for coupled vessel motion analysis in the time domain. The WADAM print file, which is directly accessible from HydroD or can be exported to Microsoft Excel, contains the information on hydrostatic data, inertia properties and analysis control parameters. Animations may be performed by Xtract (a DNV product), a general purpose model and results visualization program.

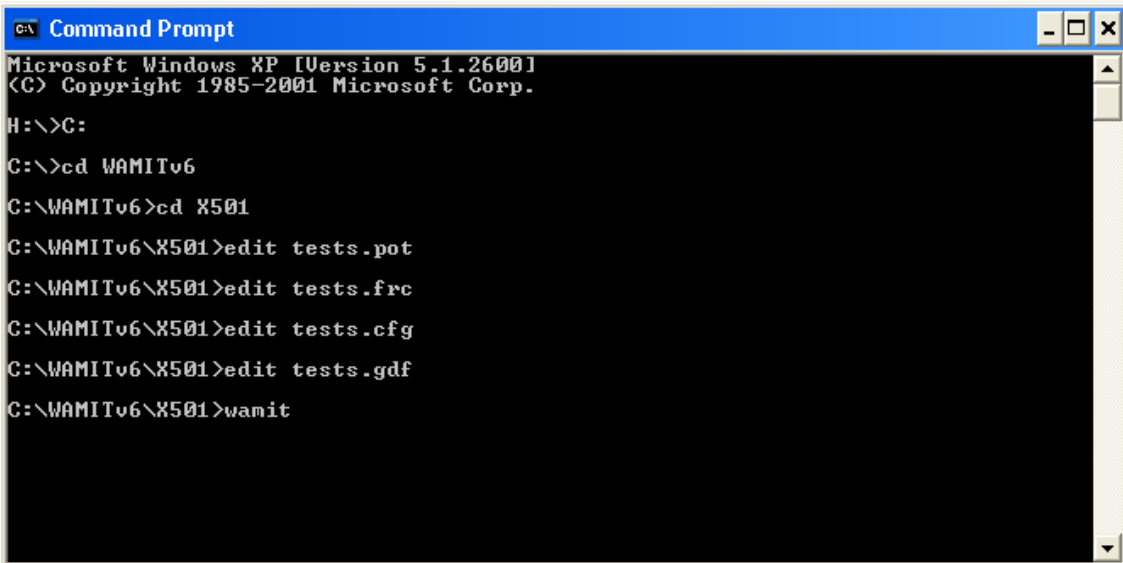
4.6 WAMIT (WAMIT Inc.)

WAMIT is a computer program based on the panel method for analyzing hydrodynamic interactions with floating or submerged bodies, in the presence of ocean waves. Version 1 was announced in 1987 at the Massachusetts Institute of Technology (MIT). Now at version 6.1, the latest in a progression of more powerful and versatile programs, WAMIT, includes a higher-order method of solution, which uses B-splines to describe both the body geometry and the velocity potential on the body surface.

Machine Requirements: Both the standard Version 6.1 and the extended version 6.1S, which includes the second-order analyzer, are supplied in Fortran-90/95 source code. Versions 6.1PC and 6.1SPC, the PC-executable versions, are intended for use on Pentium PCs, and can also be used on some older PC systems. To permit modifications of the generalized modes pre-processor and higher-order geometry; a Fortran compiler is required.

Licensing Arrangements: WAMIT, Inc. provide an assortment of license agreements for both version 6.1 and version 6.1S. The source code may be obtained on a permanent site license. PC versions are available on a permanent site license or a permanent machine license, or on temporary machine licenses, renewable monthly or annually. Version 6.1PC is also available to educational establishments on a discounted annual lease, with restriction on use to teaching and research. Indicative costs are approximately: U.S.\$ 1,000 per annum for the educational license and U.S.\$10,000 for the permanent machine license.

User Interface: Version 6.1PC, running on MS WINDOWS, uses the Command Prompt (see Figure 4.26) for communication with the WAMIT software.



```
C:\ Command Prompt
Microsoft Windows XP [Version 5.1.2600]
(C) Copyright 1985-2001 Microsoft Corp.
H:\>C:
C:\>cd WAMITv6
C:\WAMITv6>cd X501
C:\WAMITv6\X501>edit tests.pot
C:\WAMITv6\X501>edit tests.frc
C:\WAMITv6\X501>edit tests.cfg
C:\WAMITv6\X501>edit tests.gdf
C:\WAMITv6\X501>wamit
```

Figure 4.26: The WAMIT PC interface (screen capture).

The pre-processing functions provided by WAMIT are summarized in Figure 4.27. WAMIT employs one of two solution methods, a conventional low-order panel method or a higher-order solution method based on B-spline representations, according to user choice. The three-dimensional body geometry may be arbitrary and is specified in several ways. With the conventional low-order panel method the body geometry is defined in a special format text file, which the user must supply. With the higher-order solution method several options exist. These include defining the body as a set of conventional flat rectangular 'patches', which are then sub-divided into panels in WAMIT, or defining the body using B-spline representations. Both options require the user to supply special format text files.

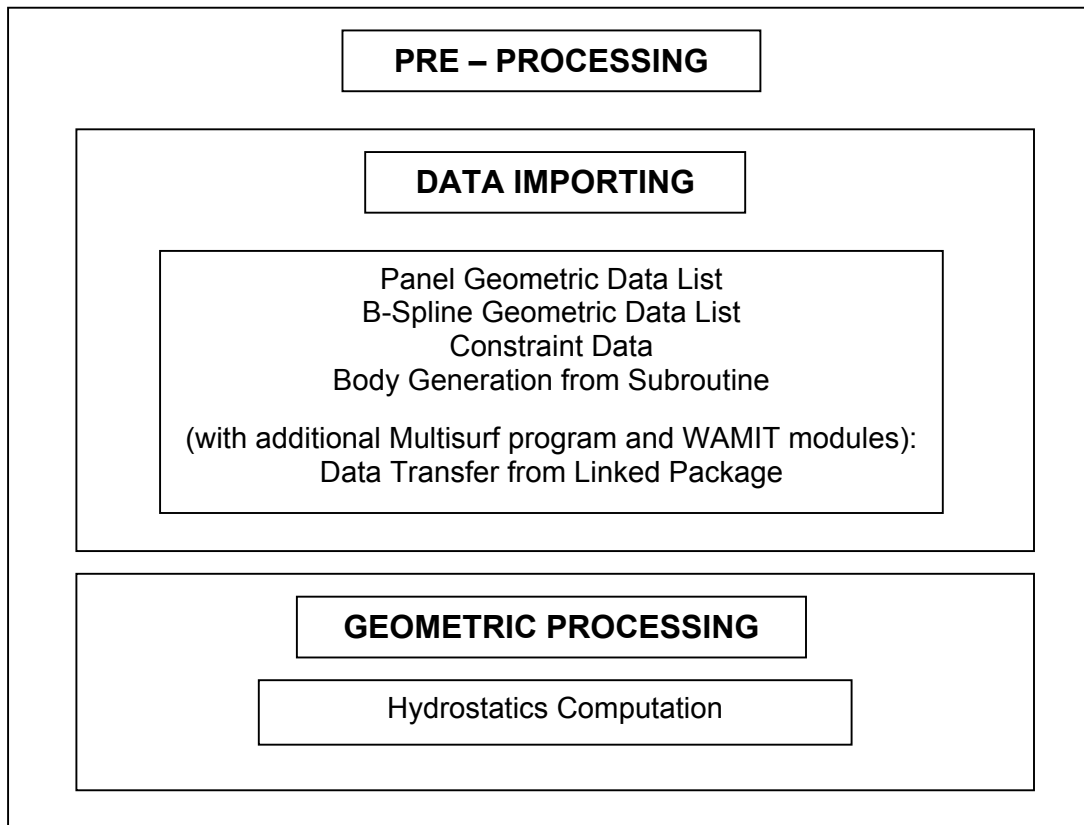


Figure 4.27: WAMIT Pre-Processing Function Summary.

Alternatively geometric models may be imported from the CAD program MultiSurf (*an AeroHydro, Inc. product*), a feature which requires the additional license of interface modules RG2WAMIT and RGKernel. As a final option, the user can make an exact definition of the body shape using subroutines, which entail the use of a Fortran compiler. The exact representation of a number of generic structures are embodied in dynamic link library (DLL) files, requiring only the input of relevant dimensions. The DLL file library can be extended to include other types of structures. Each input format offers optional specification of linearized external force or moment constraints. Constraints and other exterior dynamics may be configured in the input files. Additional input files set the parameters for program execution.

A summary is given in Figure 4.28 of the values computed. The first-order values computed include the added-mass and damping coefficients, the body motions in waves, and the pressures and fluid velocities on the body. Exciting forces and moments may be calculated from the diffraction pressure or using the Haskind relations which express the exciting forces and moments in a form that is independent of the diffraction potential. Drift forces and moments can be evaluated either by momentum conservation or by pressure integration. Additionally, the pressure, fluid velocity and free-surface elevation may be calculated at specified points on the body, referred to as 'field' points. In the low-order method, some or all of the body panels can be defined to represent thin plates or appendages of small thickness, such as damper plates or strakes. Modes may be stipulated as either free or fixed. For fixed modes, the conventional response amplitude operators are replaced by the corresponding wave loads without additional post-processing.

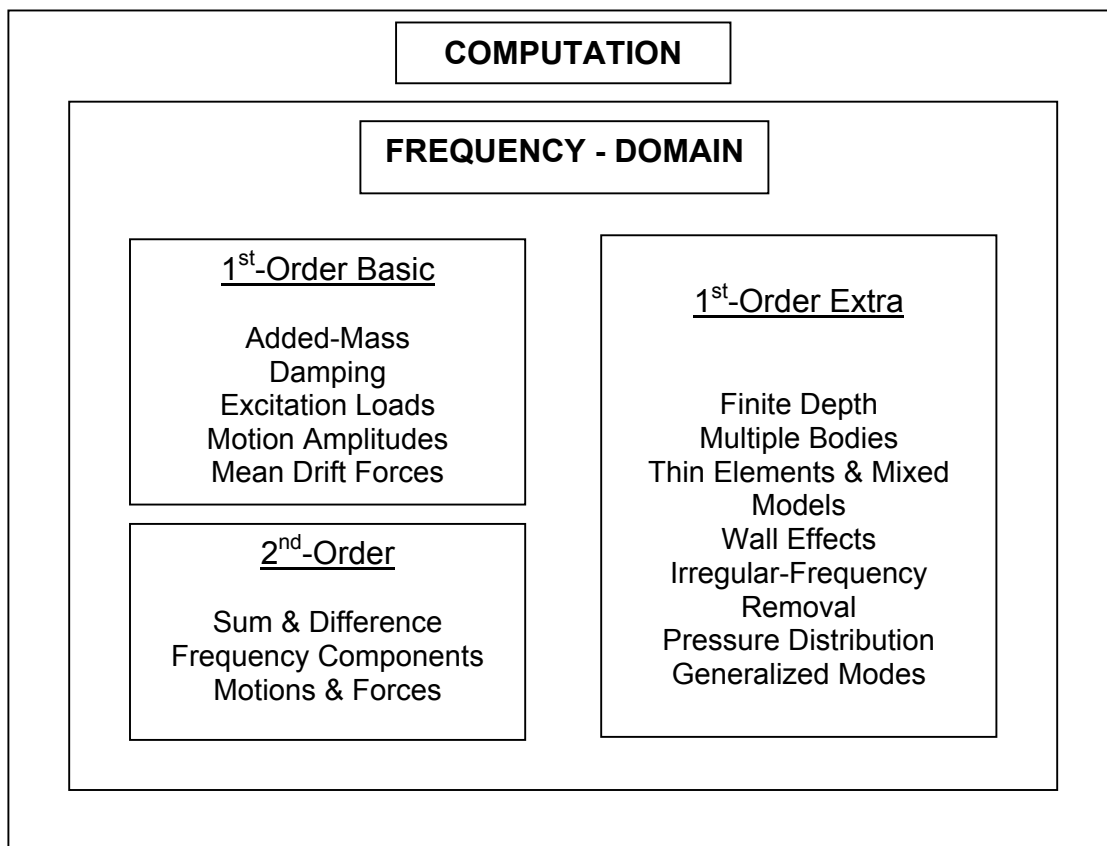


Figure 4.28: The WAMIT Computation Summary.

Options include the removal of irregular-frequency effects, the analysis of multiple, freely floating, constrained or fixed interacting bodies, and the hydro-elastic analysis of flexible structures using generalized modes. For generalized modes the mode shapes can be defined in a separate DLL file using a Fortran compiler. The computation allows the specification of finite water depth.

WAMIT Version 6.1S contains an extension for the complete second-order nonlinear analysis. This extension does not support generalized modes, combined fixed and free modes, adjacent walls or zero-thickness structures. The calculated values include the sum- and difference-frequency components of the second order forces and moments, the second-order hydrodynamic pressure on the body surface and in the fluid domain, the second-order wave elevation on the free surface, and the second-order response amplitude operator. The second-order forces and moments can be evaluated both by direct and indirect approaches. The second-order hydrodynamic pressure can be computed at user-specified points in addition to the second-order quadratic pressure force along the waterline. All of the second-order quantities are evaluated in the presence of bichromatic and bidirectional waves and the solution can be evaluated either by the low order method or by the higher-order method based on the B-spline representation of the second-order solution. An option exists for automatic free surface discretization which simplifies the use of the second-order extension, particularly for the analysis of multiple – body interactions.

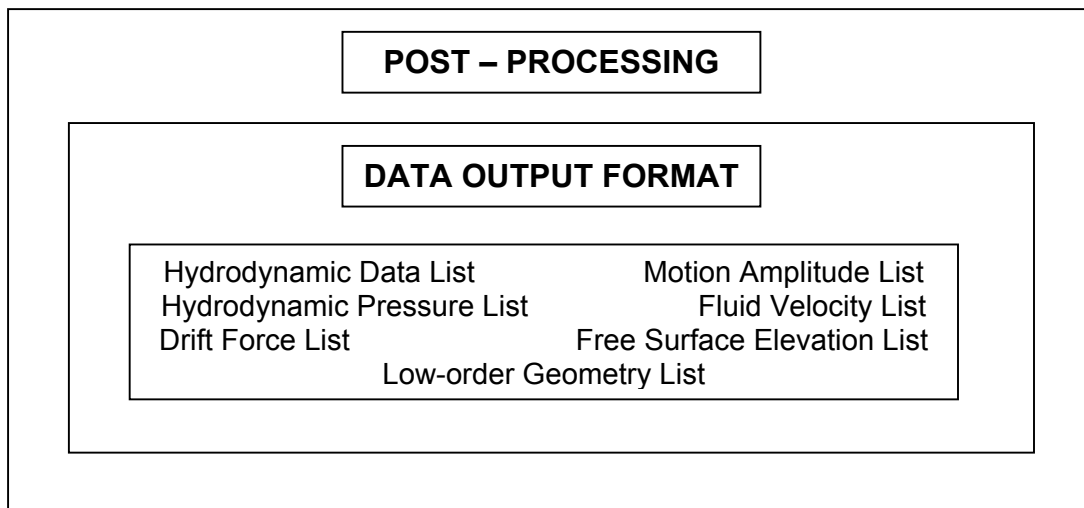


Figure 4.29: WAMIT Post-Processing Function Summary.

The post-processing functions provided by WAMIT, are summarized in Figure 4.29. Data is output in the form of text files, and further post-processing must be done externally. No specific post-processing software is favoured. To facilitate the three-dimensional plotting of higher-order geometry, this is converted to a low-order panel form.

4.7 WAVELOAD (Martec Ltd.)

Formed in 1973, Martec Limited is a privately owned company with its headquarters in Halifax, Nova Scotia, Canada. Martec is an engineering firm which specializes in the development and application of mathematical modelling and engineering analysis software. Martec provides consulting services, engineering software and contract research for a wide variety of industries including Marine, Offshore, Aerospace and Defence.

WaveLoad is a hydrodynamic analysis tool for ships and offshore structures. It consists of two main parts, FD-WaveLoad, which computes frequency domain solutions, and TD-WaveLoad which computes time domain solutions. WaveLoad hydrodynamic codes are also available as the Diffraction modules of SACS, Engineering Dynamics Incorporated's (EDI) integrated Structural Analysis Computer System.

Machine Requirements: PC systems with Intel Pentium processor, MS WINDOWS 2000 or XP, minimum 512MB of RAM, and minimum 30MB of available hard-drive space.

Licensing Arrangements: WaveLoad may be purchased for commercial use on a perpetual license, priced at US\$8,500 for FD-WaveLoad and US\$15,000 for TD-WaveLoad. Several lease options are also offered. Purchase of the software includes 90 days of free software maintenance, including updates and technical support. Academic versions of both FD-WaveLoad and TD-WaveLoad are available for an annual license fee of US\$1,000 and US\$1,500, respectively. This license includes upgrades and limited technical support, and is intended only for educational uses.

User Interface: In-built graphical interface for model generation and review of results. An example display is shown in Figure 4.30.

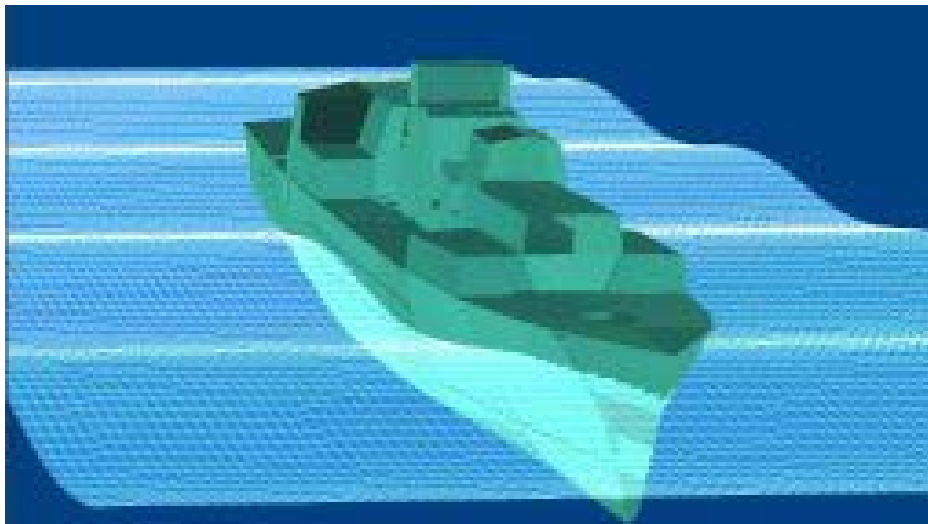


Figure 4.30: A view from the WaveLoad graphical interface (copyright Martec Ltd.).

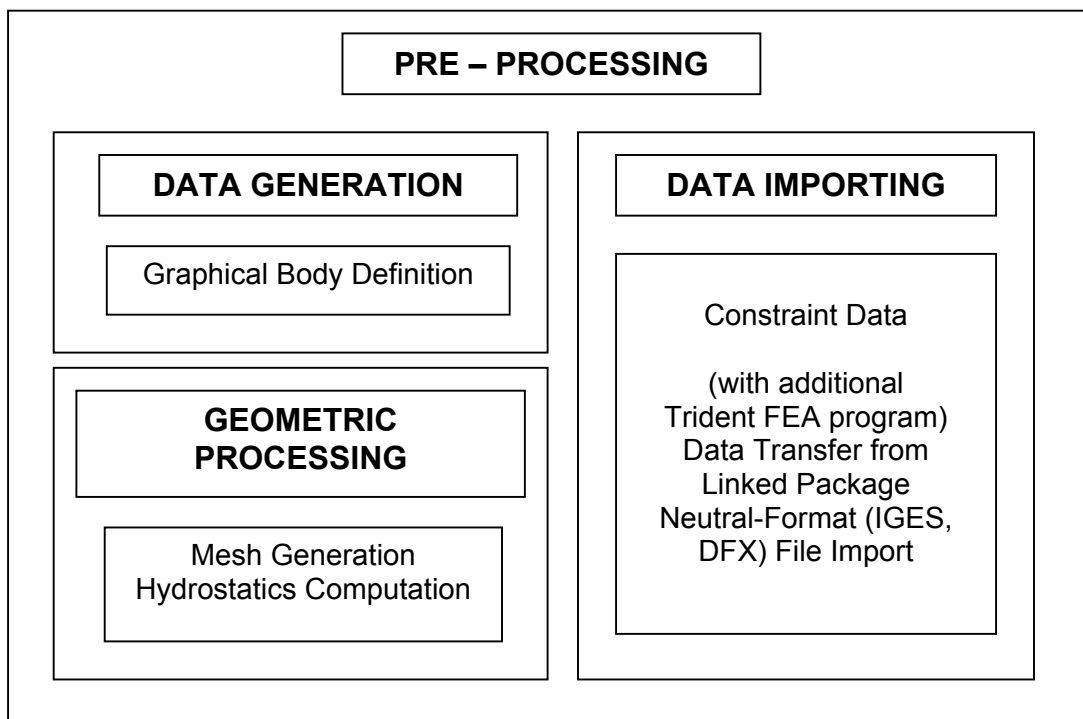


Figure 4.31: WaveLoad Pre-Processing Function Summary.

The pre-processing functions provided by WaveLoad are summarized in Figure 4.31. WaveLoad has been designed to model single- or multiple-hulled ships and floating or fixed offshore platforms. The input of body geometry is simplified by the provision of automatic panel generation, and complete models, or parts thereof may be imported from the Trident FEA structural analysis program (*a Martec product*). Since Trident FEA allows the import of neutral-format files from other CAD and FEA software, then this provides a route for the import of neutral-format files into WaveLoad. The presence of mooring lines, or similar constraints, may also be taken into account.

The FD-WaveLoad module uses a panel-based three-dimensional, zero-speed Green function with forward speed correction to perform hydrodynamic analysis in the frequency domain. A brief summary of the values computed is given in Figure 4.32. These include added-mass, damping coefficients, exciting forces, and response amplitude operators. Motions are calculated in all six degrees of freedom in both regular and irregular waves. In the latter case, motions are computed as spectral components, each corresponding to a component in the input wave spectra, of which several standard versions are implemented. The analysis allows the derivation of hydrodynamic pressure distributions, and the radiated, diffracted and incident wave components thereof. Similarly, it is possible to take account of the variation in the hydrostatic pressure due to the motion of the body. Analysis of the hydrodynamic interaction of multiple bodies may be performed, including the computation of the interference wave height in the intervening surface between the bodies. Computation also takes account of forward-speed, and incorporates the calculation of added resistance and viscous roll-damping coefficients. The depth of water may be varied from infinite to shallow.

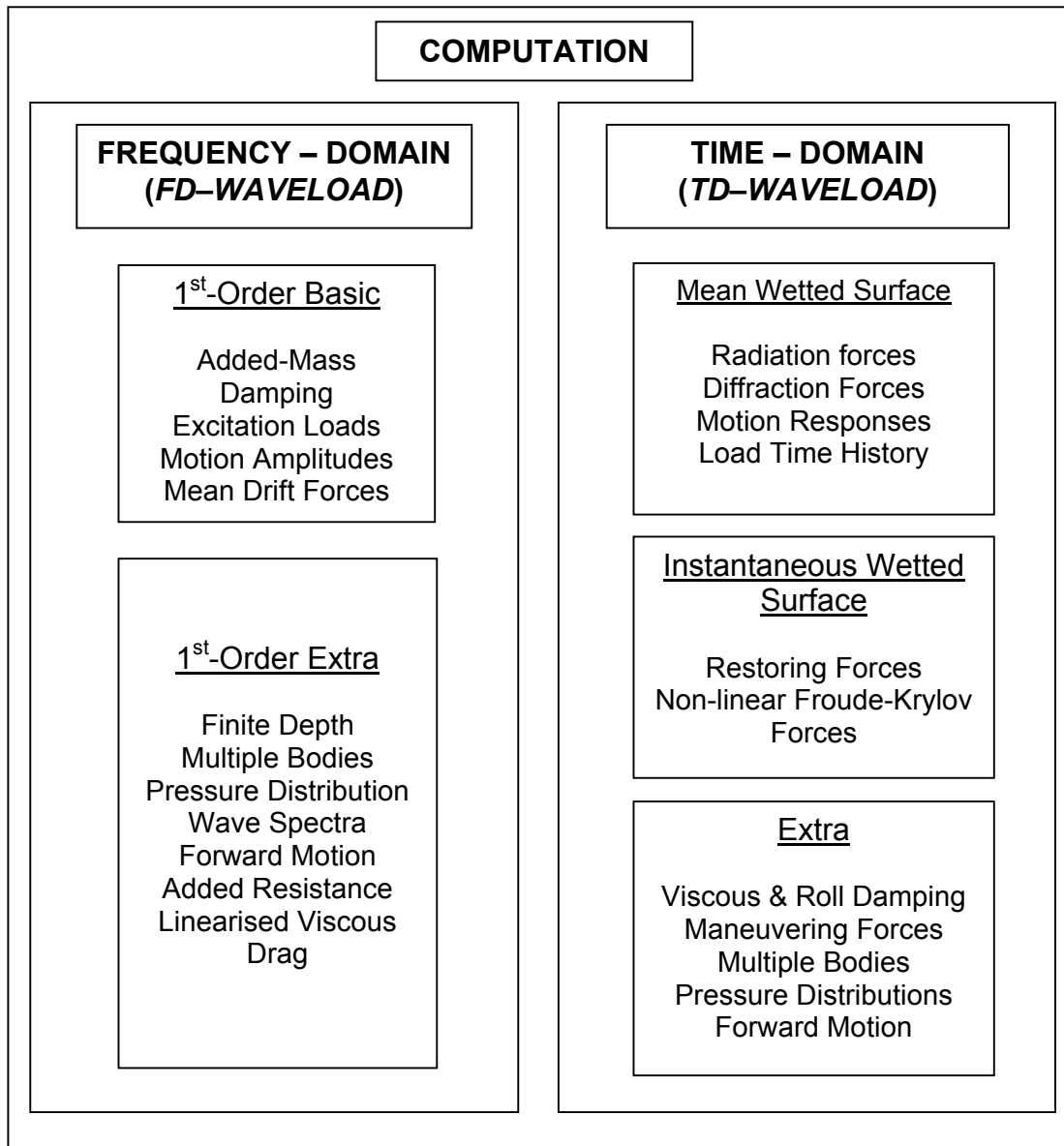


Figure 4.32: The WaveLoad Computation Summary.

The TD-WaveLoad module performs non-linear hydrodynamic analysis in the time domain using the three-dimensional panel method with the time-domain Green function. The radiation and diffraction forces are computed on the mean wetted surface. The non-linear Froude-Krylov forces and restoring forces are computed on the instantaneous wetted surface. Other non-linear forces such as viscous damping forces, impact forces and maneuvering forces are also taken into account. Incident waves may be regular or irregular. Body motions are derived as a time series of displacements, velocities and accelerations. Hydrodynamic pressure distributions are calculated over the wetted surface, and forward motion may be included. The interaction between multiples bodies may also be modelled.

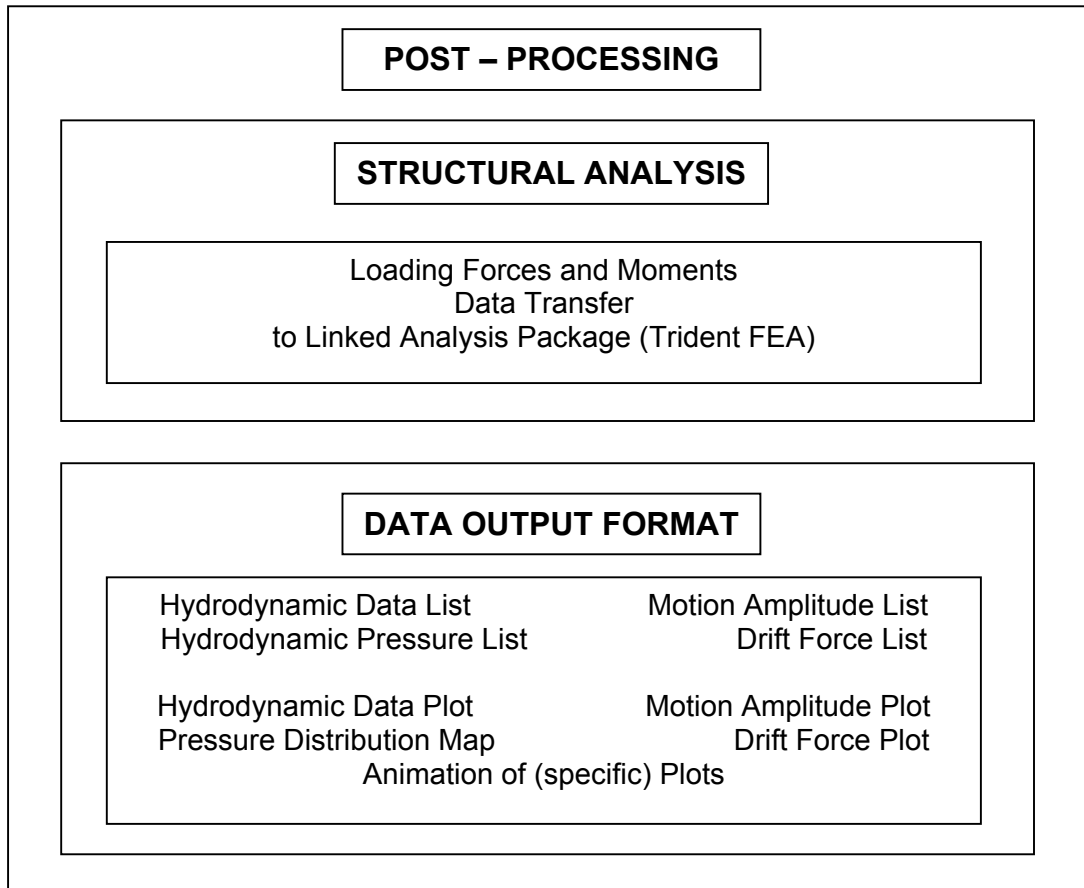


Figure 4.33: WaveLoad Post-Processing Function Summary.

The post-processing functions provided by WaveLoad are summarized in Figure 4.33. The computation of sea loads includes shear forces, bending and torsion moments on any cross-section. Results can be transferred directly into Trident FEA (see above) for additional, more complex structural analysis.

Hydrodynamics coefficients may be reviewed and displayed as two-dimensional or three-dimensional contour plots. Body motions and sea loads are displayed as result or time-history curves. Animation of body motions and pressure distributions is also possible.

4.8 TiMIT (Massachusetts Institute of Technology)

Unlike its sister program, WAMIT, the time-domain analysis program TiMIT, which was also developed at the Massachusetts Institute of Technology (MIT), has not been developed commercially, yet. TiMIT has been cited in several academic papers, a selection of which (Clauss, G.F., 2003; Qiu, et al., 2003; Clauss, et al., 2004; Lee and Newman, 2004) is given at the end of this section. Inquiries about the use of this program should be directed to the Research Laboratory of Electronics, MIT.

TiMIT performs panel-based linear hydrodynamic analysis of stationary or moving bodies, with the assumption that the fluid is ideal and in irrotational motion. The program handbook (Korsmeyer, F.T., Bingham, H.B., and Newman, J.N. *TiMIT – A panel method for transient wave-body interactions*, Research Laboratory of Electronics, MIT, 1999.) can be found by accessing the WAMIT bibliography website at www.wamit.com/biblio.htm, then following the link thereon.

Machine Requirements: A Unix environment with Fortran-90/95 compiler is assumed, although other operating systems, such as MS-DOS or VMS can be used with appropriate (user-provided) macros or command files.

User Interface: Command-line editing.

Not unsurprisingly, the pre-processing functions provided by TiMIT are fairly basic, these being summarized in Figure 4.34.

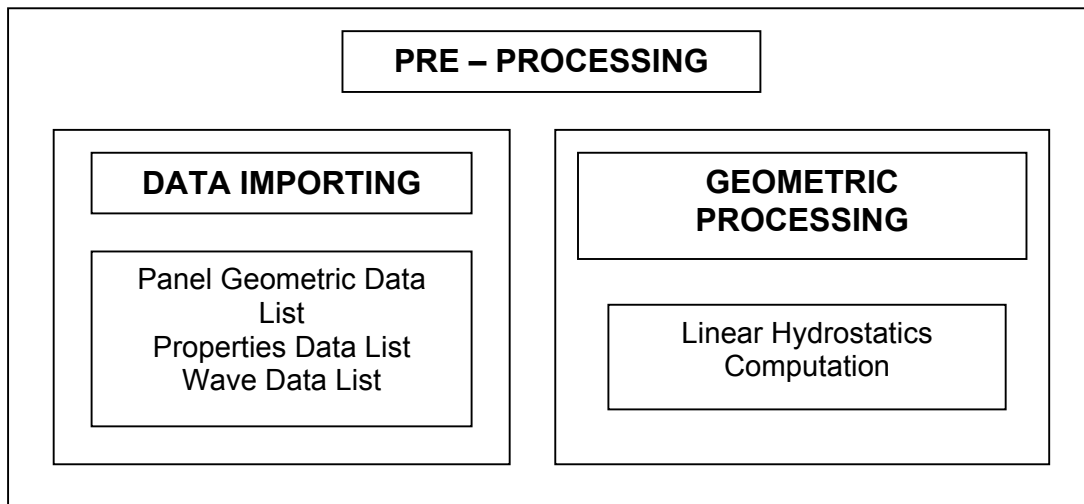


Figure 4.34: TiMIT Pre-Processing Function Summary.

Input files for the body geometry and properties, and for program control are similar in format and principle to those used with WAMIT. In addition there are files for the definition of the underlying fluid flow and incident wave elevation time-history. Also there are input files specifying the entire geometry (including freeboard above the free surface) for computations requiring the instantaneous wetted-surface.

A concise summary of the values computed by TiMIT is given in Figure 4.35.

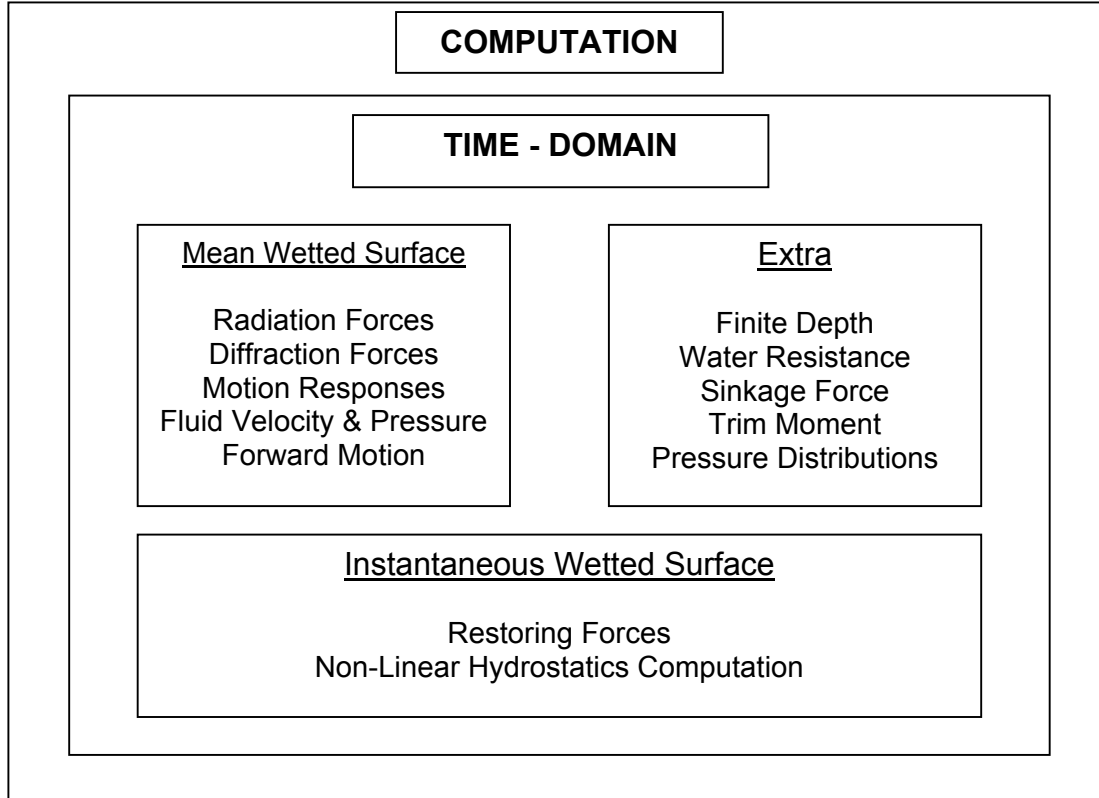


Figure 4.35: The TiMIT Computation Summary.

In TiMIT, Green's theorem and the transient free-surface Green function are used to find either the velocity potential or the source strength distribution on the body surface in order to linearize the general hydrodynamic problem of a moving body on, or near, the free surface of a fluid. Continuing effects of the radiation force and the entirety of the diffraction force are both represented by 'memory' functions, in the form of impulse-response functions. The body may be fully or partially submerged, and be subject to a steady translation in addition to the small motions about its mean position. Fluid depth may be infinite or finite, the latter situation requiring a longer time record for accuracy.

The solutions of the transient radiation and diffraction problems are derived given a user-defined wave profile input of arbitrary frequency content and a single angle of incidence. The results are used to obtain the steady forces and moments, the time histories of body responses in all six degrees of freedom, and the fluid velocity and pressure at selected points. Also computed are the transient first-order calm water resistance, sinkage force, and trim moment. The body motions, and fluid velocities and pressures at selected points may also be calculated including non-linear hydrostatic restoring forces. These are derived by subjecting the extended body geometry to the appropriate rotations and translations. The impulse response functions may also be Fourier transformed to get frequency-domain quantities such as hydrodynamic coefficients, response amplitude operators of body motions and fluid velocity or pressure, and second-order mean forces and moments.

The post-processing functions provided by TiMIT are also utilitarian, and are summarized in Figure 4.36.

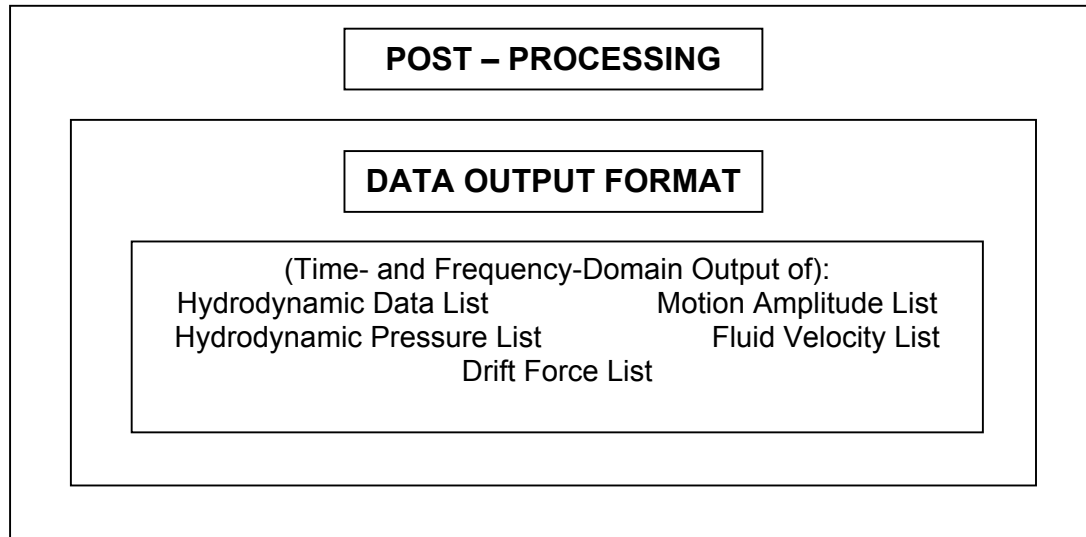


Figure 4.36: TiMIT Post-Processing Function Summary.

Output data is listed in ASCII-text files. Any further post-processing must be done externally, with no specific post-processing software being favoured.

References:

Clauss, G.F., 2003, *Genesis of design wave groups in extreme seas for the evaluation of wavestructure interaction*, Twenty-Fourth Symposium on Naval Hydrodynamics, National Academies Press. (books.nap.edu/books/NI000511/html/231.html).

Clauss, G.F., Stutz, K., Schmittner, C., 2004, *Rogue wave impact on offshore structures*, Offshore Technology Conference, 3 – 6 May 2004, Houston, Texas, U.S.A.

Lee, C.-H., Newman, J.N., 2004, *Computation of wave effects using the panel method*, in *Numerical models in fluid-structure interaction*, Preprint, Editor S. Chakrabarti, WIT Press, Southampton, 2004. (www.wamit.com/wit2003.pdf)

Qiu, W., Peng, H. and Hsiung, C.C., 2003, *A Panel-Free Method for Time-Domain analysis*, Twenty-Fourth Symposium on Naval Hydrodynamics, National Academies Press. (books.nap.edu/books/NI000511/html/963.html).

4.9 A Rough Guide

Table 3.1: Rough Guide to Hydrodynamic Software Packages												
	Data Input			Frequency-Domain			Time - Domain			Data Output		
	Graphical User Interface	Mesh Generation / Editing	Model Importing	First - Order Basic	First - Order Extra	Second - Order	Mean Wetted Surface	Instantaneous wetted surface	Extra	Structural Analysis	Data Listing	Data Plotting
AQWA :	√	√	√	√	√	√	√	√		√	√	√
HYDROSTAR :	√	√		√	√	√				√	√	√
NEPTUNE :	√	√		√	√					√	√	√
WADAM :	√	√†	√†	√	√	√				√	√	√
WAMIT :			√††	√	√	√					√	
WAVELOAD :	√	√	√	√	√		√	√	√	√	√	√
TIMIT :			√††				√	√	√		√	

√† - via extra software modules

√†† - via user-generated text files

5. Computational Fluid Dynamics (CFD) Software Packages

In a similar manner to hydrodynamic analysis software packages, CFD packages also possess many common features. Again a 'model' package is used as a framework to present the various features of the packages in a clear and concise way. As in the case of the 'model' hydrodynamic analysis package (shown in Figure 3.1), the 'model' CFD package consists of three main blocks: the 'Pre-Processing', 'Computation', and 'Post-Processing' blocks. A summary of the types of 'Pre-Processing' functions that are available is given in Figure 5.1.

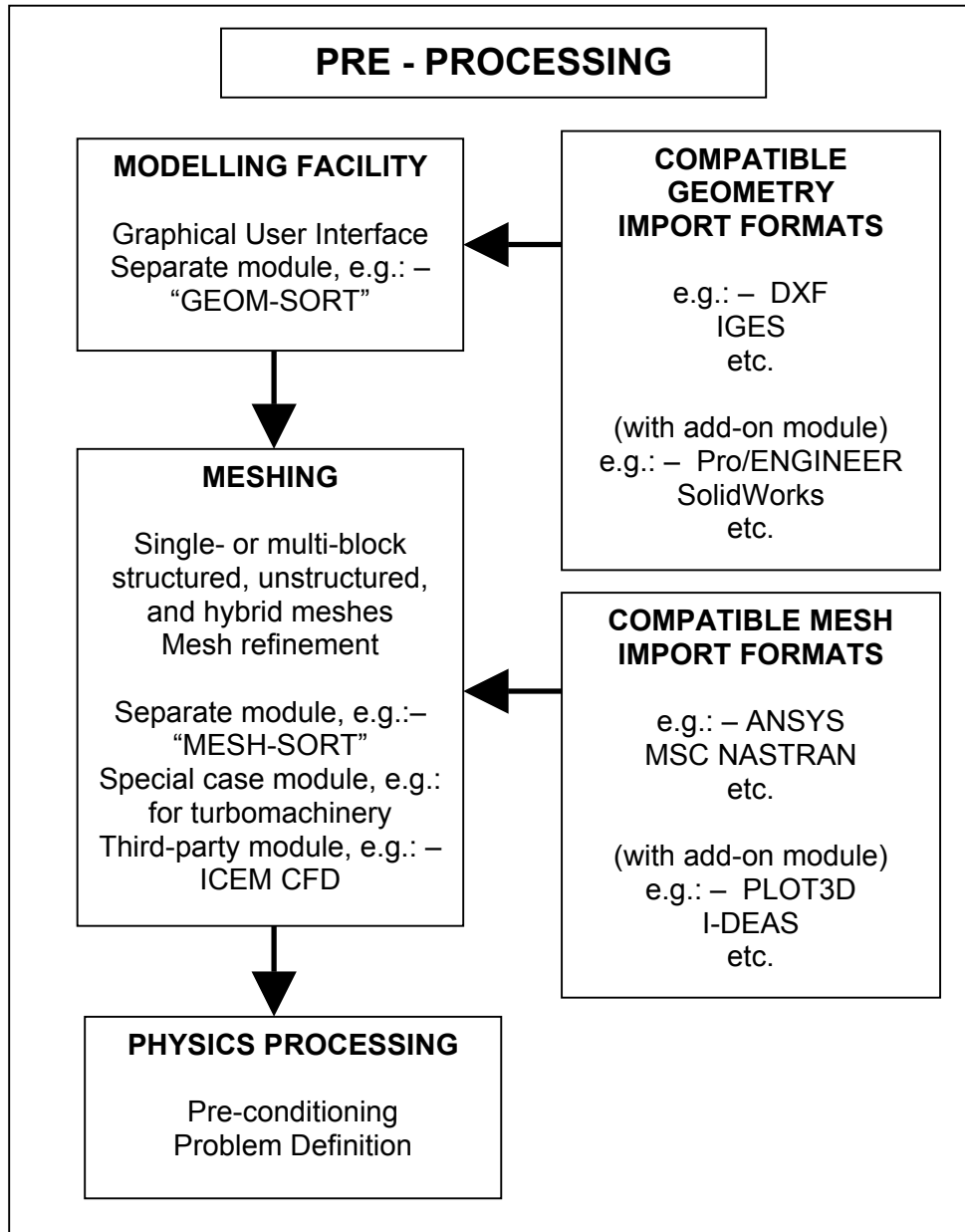


Figure 5.1: The Pre – Processing Block.

The geometry of the body to be analyzed may be defined using a built-in modelling facility, in the user interface itself, or using a separate in-house module, given the name “GEOM-SORT” in Figure 5.1. Alternatively, the body geometry may be imported from a third-party CAD package as a file of compatible format, although this may require the use of an add-on module for format translation. The geometry is then converted into a two-dimensional surface mesh or three-dimensional volume mesh, according to the analysis required. The mesh arrangement may be single- or multi-block, in a structured, unstructured, hybrid, or other more esoteric form. Again, there may be an in-built facility for automatically generating the mesh, and for refining the distribution of nodes across the mesh where necessary. The meshing process may be provided by a separate module for all cases, given the name “MESH-SORT” in Figure 5.1, or just for special cases such as turbomachinery. Some packages allow completed meshes to be imported directly from other manufacturers’ software in a particular format, or permit this with the intermediation of an add-on module. The final part of pre-processing defines the physics of the problem to be solved, as different approaches may be taken to different applications. The process involves such actions as the setting of boundary conditions, selection of turbulence model type, or the prescription of parameters within which the problem is defined.

With the geometry suitably transformed and the problem definition established, the data is then passed to the ‘Computation’ block, so that calculations may proceed. The range and diversity of the available ‘Computation’ functions are indicated in Figure 5.2. The general nature of the CFD approach allows its application to many physical problems. With the best will in the world it is hard to envisage a purpose for using a plasma physics modelling program in the design of a marine energy converter. However, one should not preclude the utilization of exotic power take-off arrangements, if that be the case, so mention is made of the capability of performing analyses of which the relevance is not immediately apparent. It is indicated in the text whether these extra capabilities are an integral part of the package or available as additional modules.

The physical models which are regarded as immediately relevant to marine energy converter analysis are those which relate to fluid dynamics, fluid-structure interaction, and the structural analysis of stationary and rotating bodies. Most fluid dynamics modellers cover far more cases than should be required, including both steady-state and transient, inviscid, laminar and turbulent flow in both compressible and incompressible fluids. Most provide the modelling of multiple-phase systems, that is, systems of more than one fluid, often incorporating and free surfaces between the fluids. The movement of this free surface, however, can only be driven by pouring or ‘sloshing’ effects, or by penetration by another object. Additional physical models cover a broad range, including heat transfer, chemical reactions, spray and particle behaviour, electromagnetics and electrostatics, flow through porous media, and the aforementioned plasma physics.

The discretization of the body surface or volume into boundary or finite element structures, respectively, allows the parallelization of the CFD process, which some packages accommodate, either on multiple-processor machines or on networked workstations or PCs. Specific provision must, in this case, be made for the parallel processing management, and the communication between the linked processors.

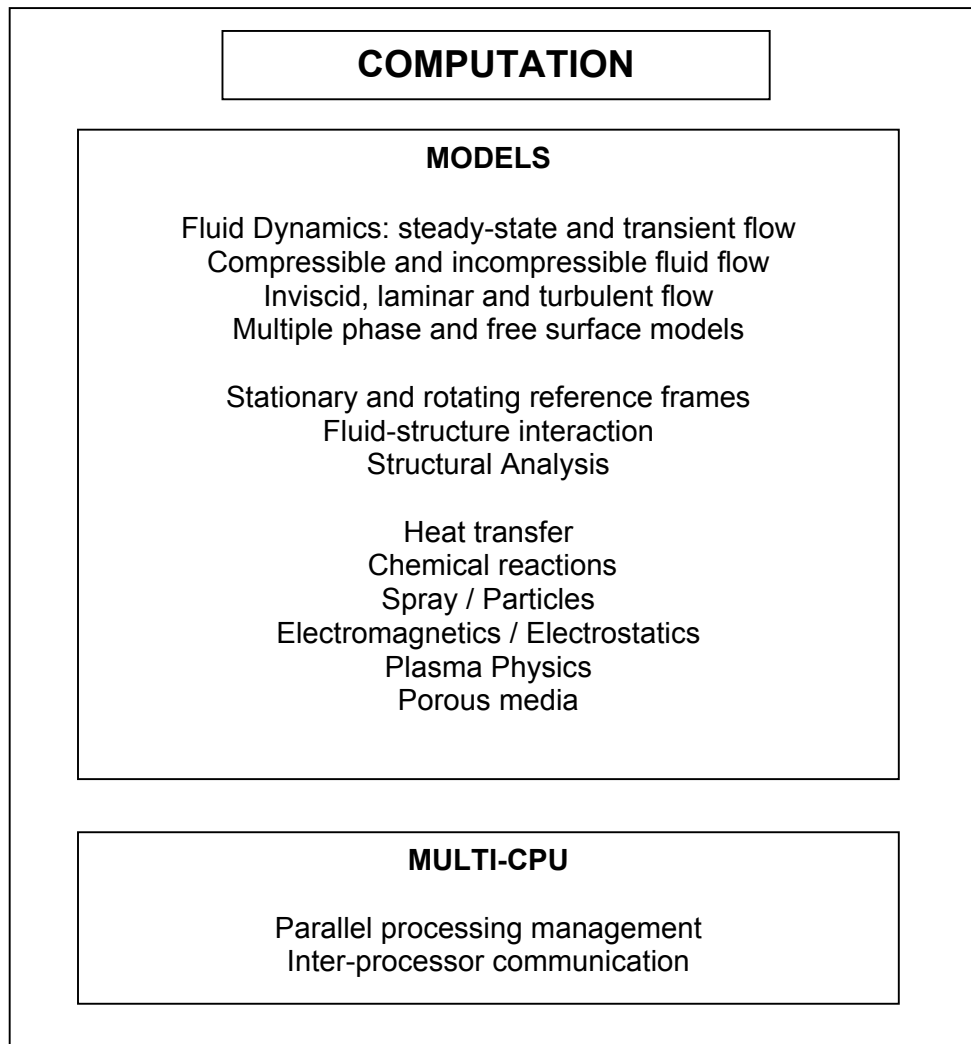


Figure 5.2: The Computation Block.

The results obtained from the CFD calculations are then processed for presentation to the user, using 'Post-Processing' functions such as those outlined in Figure 5.3. The basic numerical data can be listed in text files, plotted on graphs, or accessed on-screen by 'virtual probes', an interactive arrangement for obtaining data at a specified point in the body geometry. The dynamic nature of most CFD problems usually requires the solutions to be developed as time histories of flows, forces, pressures and temperatures, and, depending upon the type of flow model employed, particle trajectories. Visual data output, therefore, may be produced as images at a fixed time point, or as animations of the entire time history. The data, images and animations may be produced in formats that are compatible with other software, such as '.gif' picture files or '.avi' movie files, and so can be exported to third-party software for further use.

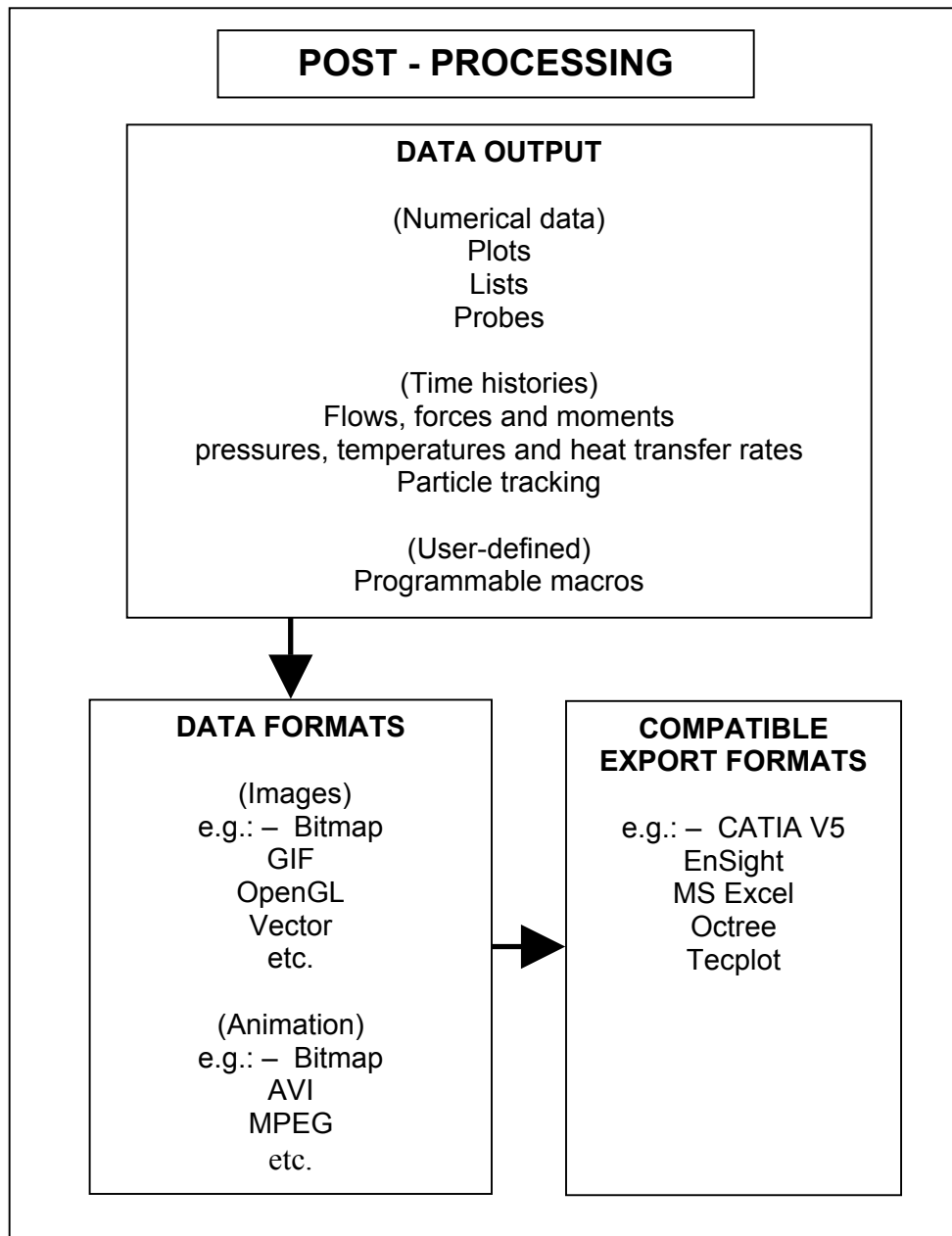


Figure 5.3: The Post – Processing Block.

5.1 ADINA (ADINA R & D, Inc.)

ADINA R & D, Inc. was founded in 1986 to further the development of the ADINA software system for the analysis of solids, structures, fluids and fluid flow with structural interactions. ADINA consists of modules which combine to form a one-system program for comprehensive finite element analyses of structures, fluids, and fluid flows with structural interactions.

Machine Requirements: Intel Pentium PCs running WINDOWS 98 / Me / NT / 2000 / XP, with 64MB RAM (128MB or more is recommended), 215MB disk space, CD-ROM drive and graphics display with 4MB video card. Unix or Linux operating system with 64MB RAM (128MB or more is recommended), about 250MB disk space, CD-ROM drive and graphics display.

Licensing Arrangements: The maximum cost for a two-year license to use ADINA is US\$2,400. The educational license is offered at a substantial discount. ADINA may also be obtained in reduced form, called the '900 nodes version', for one year usage on any number of PCs. After a year, a replacement CD can be requested for a nominal price. The '900 nodes version' is priced at US\$140 in U.S. and Canada, US\$160 in Europe, and US\$180 elsewhere.

User Interface: The ADINA-AUI program module is an interactive graphical user interface covering for all the modelling, pre- and post-processing activities for all the ADINA modules.

The 'Pre-Processing' functions are performed in ADINA by the ADINA-AUI and ADINA-M modules and are summarized in Figure 5.4. The model geometry can be created directly in the ADINA-AUI module, can be combined with geometry imported from third-party CAD software, or imported as a whole. The ADINA Modeler (ADINA-M) module is a separate add-on module to the user interface that allows the user to perform solid modelling. The ADINA-M module uses a Parasolid-based system, so solid geometries from any Parasolid-based CAD system can be amended and imported directly into ADINA-AUI via the ADINA-M module.

Mesh generation is automatic with control over the distribution of element sizes. With minimal geometries the meshing can be mapped on to the geometry. Physical properties may be assigned directly on to the geometry of the body, and may be done anisotropically.

The range of 'Computation' functions are indicated in Figure 5.5. The separate ADINA modules are given along with a general description of their functions. The ADINA-F program module uses an arbitrary Lagrangian-Eulerian formulation with the full Navier-Stokes or Euler equations to carry out finite element and / or finite volume discretization schemes, taking the most efficient approach applicable to the solution. The flows can be incompressible or compressible, laminar or turbulent. Both steady-state and transient analyses are possible. Multiple-fluid flows which contain free surfaces can be handled.

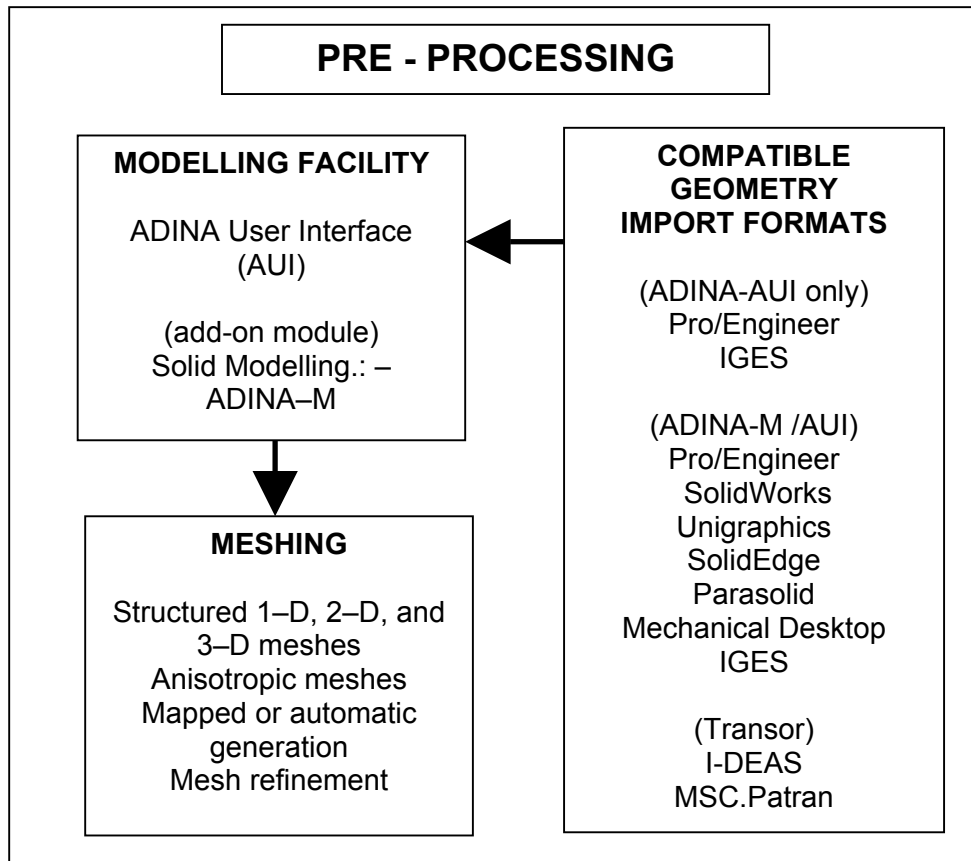


Figure 5.4: The ADINA Pre-Processing Block.

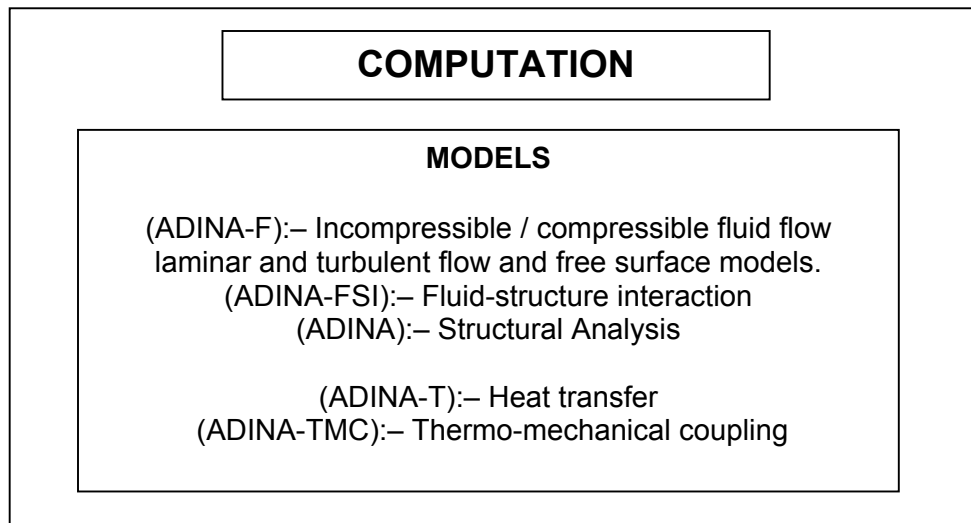


Figure 5.5: The ADINA Computation Block.

The ADINA-FSI program module is designed for the analysis of fluid flows coupled with structural interactions, or multi-physics problems. As such the module is an amalgamation of the ADINA structural analysis module and the ADINA-F flow module into a single module. Different meshes can be used for the structural and the fluid flow models. The ADINA program module itself provides for the linear, or nonlinear, stress analysis of solid bodies and structures undergoing both static and dynamic loading. The program can analyze a range of solids and structures, in materials such as metals, soils and rocks, plastics, rubber, fabrics, wood, ceramics or concrete.

The ADINA-T program module is used to solve heat transfer problems in solids and structures. The ADINA-TMC program module can be used for fully-coupled thermo-mechanical analysis of problems such as heat generation due to plastic deformation, or heat generation due to friction on surfaces in contact.

The ADINA-AUI user interface also performs the 'Post-Processing' functions, which are shown in outline in Figure 5.6. Variables may be plotted on graphs or listed on-screen or to file. Graphics may be output in vector or bitmap formats, as band or contour plots, and as vector or tensor plots. Results may be animated by the creation of AVI movies.

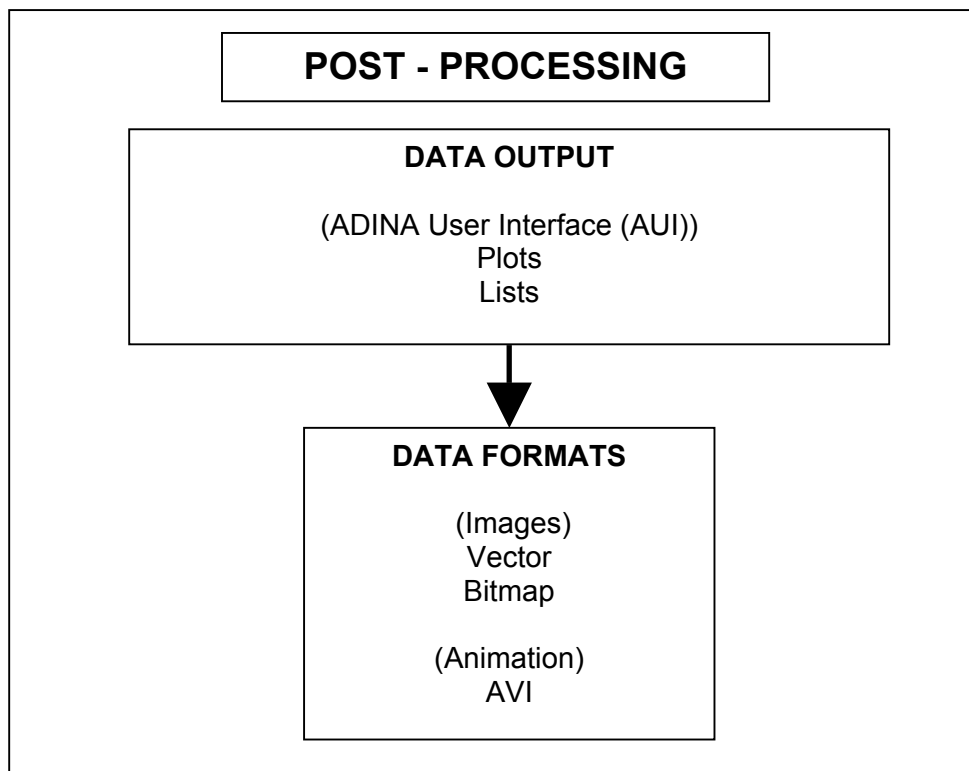


Figure 5.6: The ADINA Post-Processing Block.

5.2 CFD++ (Metacomp Technologies, Inc.)

Metacomp Technologies, Incorporated, was founded in 1994 to conduct research and development, software development, distribution and support, and consultation services. CFD++ is the seventh generation of CFD software developed by Metacomp, and is a software suite built to undertake engineering and scientific applications in aerospace, automotive, marine, biomedical, manufacturing, electronics, chemical, power, construction, cooling systems and household products.

Machine Requirements: CFD++ packages are available to run on a variety of machines including PCs running Windows NT, PCs running UNIX operating systems, such as Linux or Solaris, stand-alone or networked workstations (e.g.: Sun, SGI, IBM, HP, DEC, etc.), and multiple CPU computers (e.g.: IBM SP2/SP3, SGI Power Challenge, SGI/Cray Origin 2000, Cray T3D/T3E, etc.).

Licensing Arrangements: CFD++ software licensing is based on the level of functionality offered and is backed by training and support. Prices range from US\$8000 – US\$12000. It is available in terms of executables, libraries and examples compiled for the various computer platforms. CFD++ can be obtained in one of three packages, (compressible perfect gas, compressible multi-species real gases, and incompressible fluid), each available in three editions, (professional, educational institution, and student). The Educational Institution Edition is only for teaching and research use and licensing arrangements are flexible, typically working on an annually-renewable basis, but also over longer or shorter license periods. Substantial discounts are offered to academic institutions on a case-by-case basis depending on usage, i.e. teaching or paid research project. The Student Edition includes almost full functionality but with reasonable restrictions on problem size. Pricing will depend on the choice of package, choice of edition and the number of CPUs to be used. Inviscid flow versions can be made available at a discount. Combined packages are also available.

User Interface: The in-built graphical user interface performs all the pre- and post-processing activities. The 'Pre-Processing' functions are summarized in Figure 5.7. Geometry can be imported and converted from plot-3D, NASTRAN- compatible, and other miscellaneous formats.

Meshes can consist of one-dimensional elements (lines), two-dimensional elements (triangles or quadrilaterals), or three-dimensional elements (hexahedra, triangular or tetrahedral prism, or pyramids). The mesh topology is selected at generation and can be single- or multi-block, structured or unstructured. Relative motion between blocks can be taken into account. The problem variables can be pre-conditioned for incompressible and low-speed compressible fluid flows, and boundary conditions set accordingly, via the user interface.

CFD++ employs unified physics, mesh and computation techniques, using Reynolds-averaged Navier-Stokes (RANS) equations, large eddy simulation (LES) models or hybrid LES / RANS models. Discretization of the equations is achieved using multi-dimensional or least-squares interpolation schemes and slope limiting to reduce numerical oscillations. A variety of solution methods are used, including explicit, implicit and multi-grid relaxation schemes, and dual time-stepping.

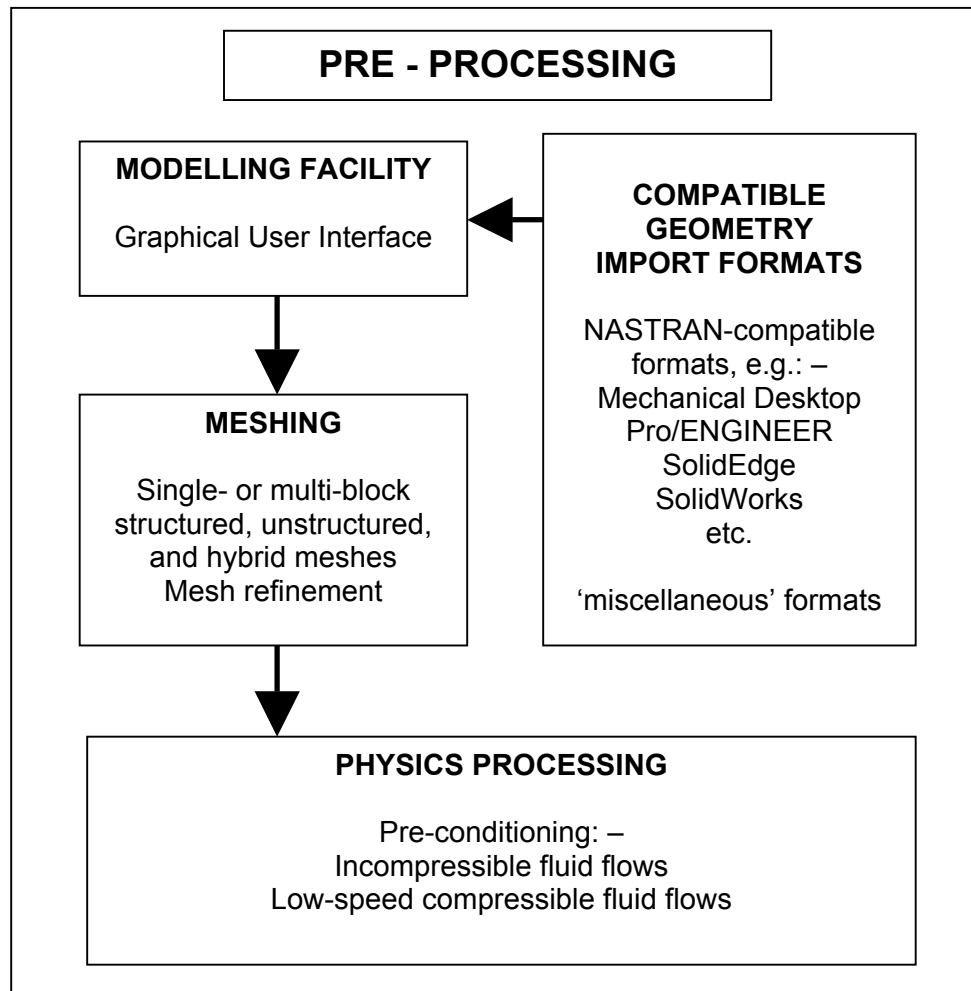


Figure 5.7: The CFD++ Pre-Processing Block.

The different 'Computation' functions available with CFD++ are shown in Figure 5.8. Incompressible flows are modelled after preconditioning of the upstream flow equations and multi-phase flows can be modelled using the Eulerian, or finite volume distribution, method. Turbulence models are independent of any parameter of the topology, with pointwise 1-, 2-, and 3-equation linear, and 2-, and 3-equation non-linear models available for use.

Modules are also produced to perform thermodynamics and chemical reaction modelling. The thermodynamics program handles multi-species perfect gases in compressible and incompressible flows and conjugate heat transfer. The reacting flow chemistry program handles finite and infinite rate chemistry in single and multi-species reacting flows. The flows may also be high speed reacting or non-reacting flows or low speed compressible reacting or non-reacting flows with suitable preconditioning. There is also the capability of modelling electrostatic systems.

The parallel-processing version of CFD++ uses domain decomposition of the mesh, using MPI (Message-Passing Interface standard)-based inter-domain communication.

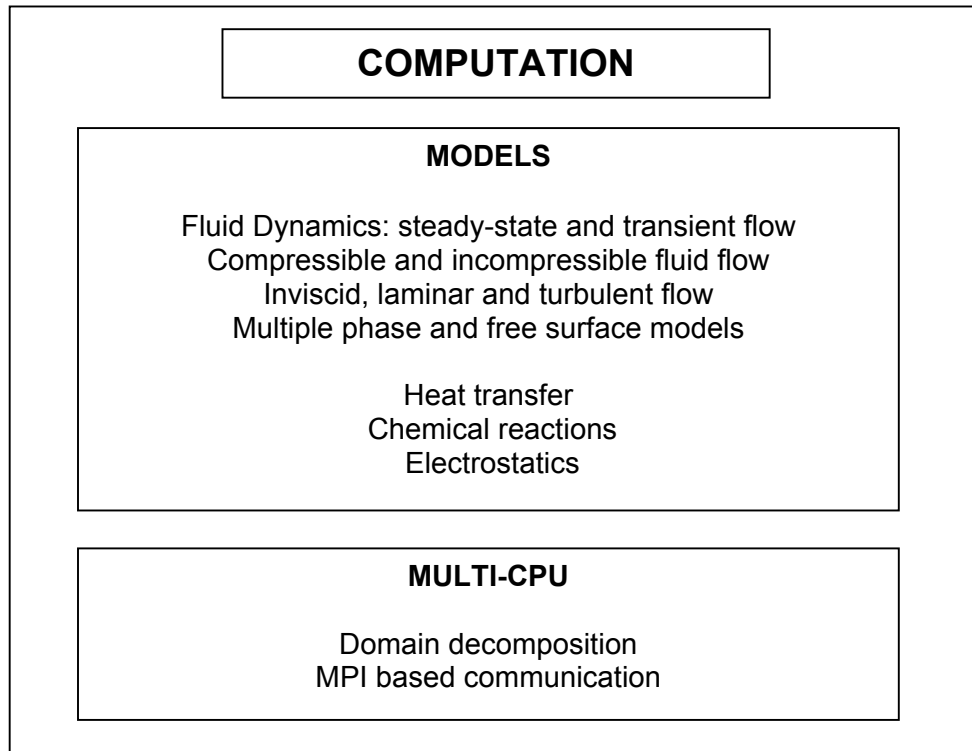


Figure 5.8: The CFD++ Computation Block.

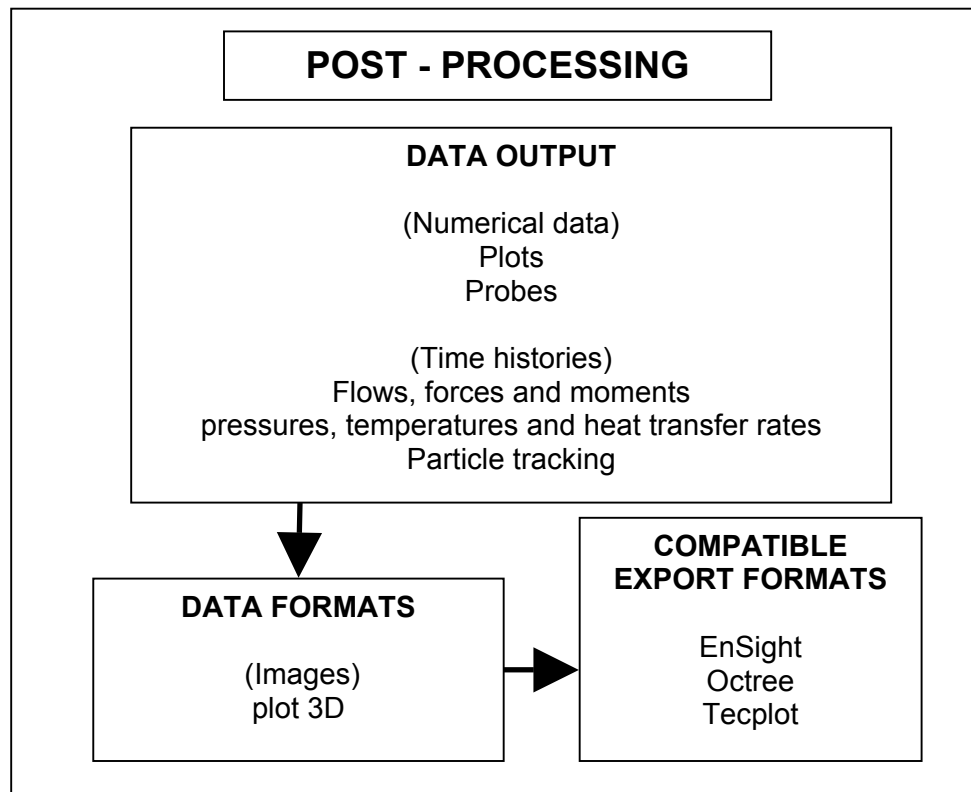


Figure 5.9: The CFD++ Post-Processing Block.

Some of the 'Post-Processing' functions available in CFD++ are shown in Figure 5.9. The graphical user interface allows the user to convert structured grids back into plot 3D format, and also contains a number of solution plotting and information probe devices. Solutions that are plotted include several time histories. One is the time history of flows, forces and moments summed over each section relating to a particular boundary condition. Another is the time history of pressure at each boundary element face at selected points or taken over all the elements. Similarly, the time history of temperature and the heat transfer rate at selected points can be produced. Particle trajectories can be tracked and displayed. Results may also be exported in other software formats, such as Tecplot, EnSight and Octree.

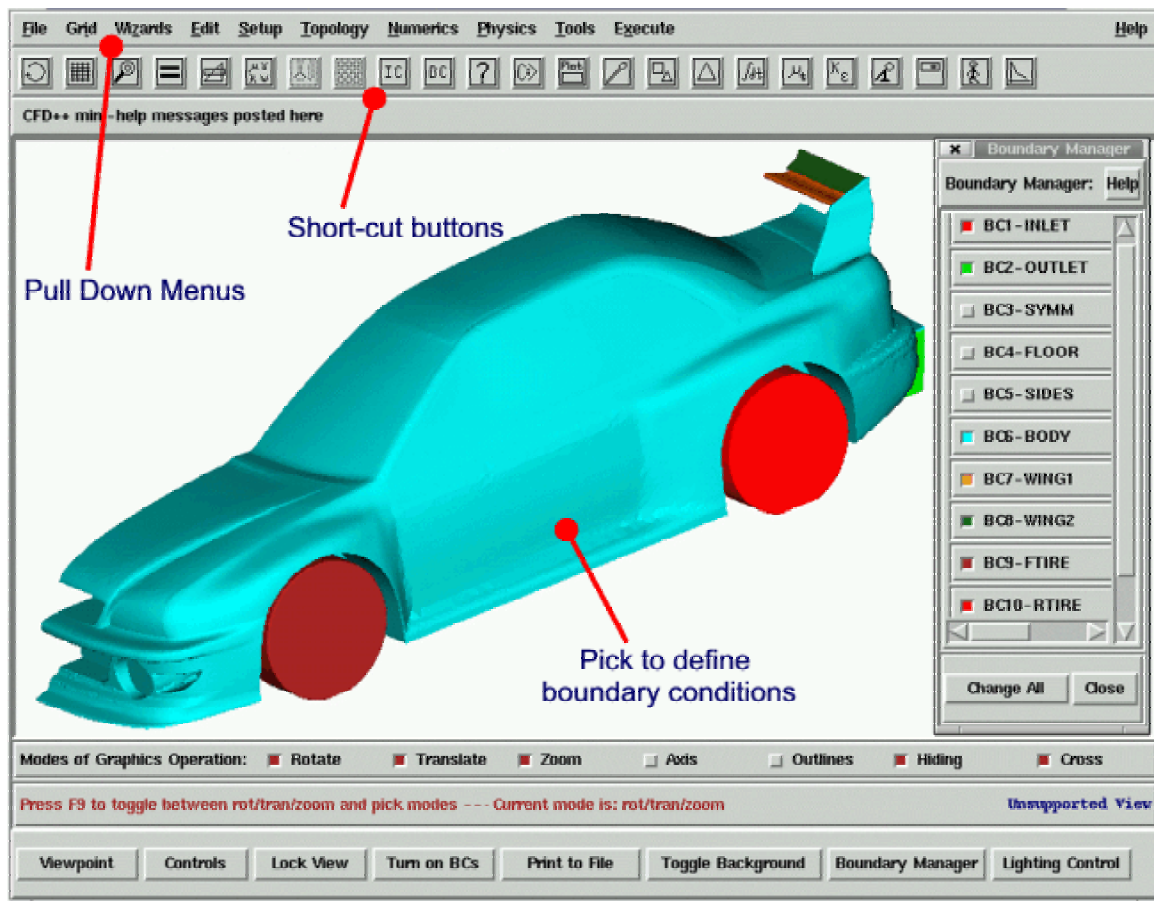


Figure 5.10: The CFD++ graphical user interface (copyright Metacomp Technologies, Inc.).

5.3 CFD-ACE+ (CFDRC)

CFD Research Corporation (CFDRC) was inaugurated in 1987. The company carries out research and development projects funded by private industry and U.S. Government agencies, develops and markets integrated software products and provides engineering analysis and design services. CFDRC software products and engineering services are used by over 600 organizations worldwide, in the Semiconductor, Biotechnology, Fuel Cells, MEMS, Plasma, Combustion, Propulsion, Materials, Defense, Aerospace, Automotive, Chemical, Electronics, Power Generation, and Environmental industries.

CFD-ACE+ is a CFD and multi-physics software package which makes possible coupled simulations of fluid, thermal, chemical, biological, electrical and mechanical phenomena. It supports all grid technologies and most of the commonly-used CAD, CAE and EDA data formats. It is designed for parallel computing on high performance workstations and PC clusters, but is also claimed to work on all computer hardware / software system combinations.

Machine Requirements: PCs with Intel or AMD Athlon processors running WINDOWS 2000 / XP or Linux Redhat V7.2 (or later versions). Machines running UNIX-based operating systems such as SGI IRIX 6.5, SunOS 5.8, Solaris 8, HP UX11.0, (or later versions of the aforementioned). For other platforms or different versions of quoted platforms, the prospective user should consult the CFDRC support engineers.

Licensing Arrangements: CFD-ACE+ is modular and is offered as a basic package of flow, heat transfer and turbulence modules, with optional extra modules as required. Prices have not been ascertained.

User Interface: The general-purpose geometry and grid generation system, CFD-GEOM, is used for geometry creation and manipulation, CAD import, and mesh generation (see Figure 5.11). Numerical results are post-processed using the interactive graphics program, CFD-VIEW.

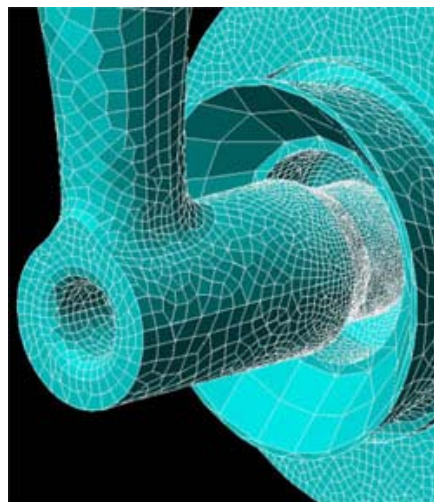


Figure 5.11: An example of CFD-GEOM mesh generation (copyright CFD Research Corp.).

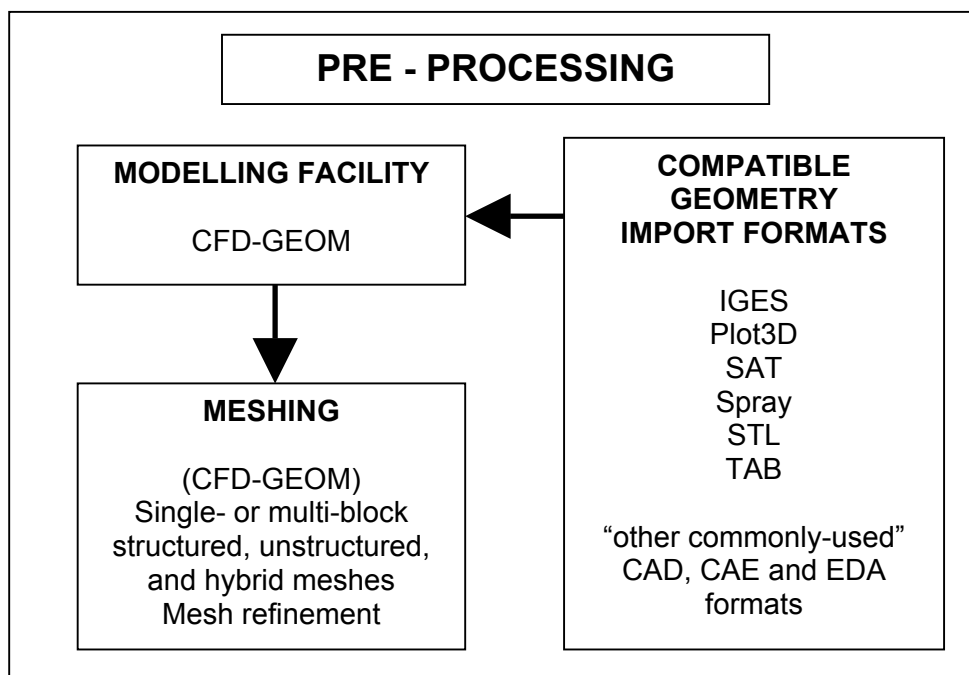


Figure 5.12: The CFD-ACE+ Pre-Processing Block.

CFDRC packages use a general-purpose geometry and grid generation program module, CFD-GEOM (*a CFDRC product*) to provide the ‘Pre-Processing’ functions shown in Figure 5.12. The program module allows geometry creation, manipulation, and importation from a number of formats. Mesh generation is also performed, producing multi-block structured, unstructured, hybrid, and body-aligned polyhedral meshes from the body geometry, with control over local mesh density and overall grid quality. CFD-GEOM also supports the open-source scripting language, Python, which is popular with many LINUX programmers.

A summary of the various ‘Computation’ functions provided by CFD-ACE+ are shown in Figure 5.13. The CFD-ACE+ Flow Module allows the user to model most fluid systems, using a pressure-based finite-volume-method discretization of the Navier-Stokes equations. The module supports arbitrary mesh interfaces and moving, deforming and rotating meshes. Flows may be both internal and external, and at any speed, with Newtonian or non-Newtonian viscosity. Two-phase flows are modelled using several techniques including the Eulerian two-fluid model, (two fluids as continuous, interspersed media), and the Volume-of-Fluid (VOF) method, (two fluids as continuous but separated phases). The Free Surface module uses the VOF method to simulate the hydrodynamics and heat transfer for combinations of two, non-mixing, fluids, including the effects of surface tension and gravity, and can also be used together with the modules for stress, strain, and fluid structure interaction. Turbulent flow models include Reynolds Averaged Navier Stokes (RANS) models as well as Large Eddy Simulation (LES) models. The Stress module provides the capability of finite element structural analysis, either stand-alone for pure structural analysis, or coupled with the other modules for multidisciplinary analyses. In such cases modules can be used along with the Grid Deformation module, which performs grid deformation in a feedback loop with other modules. Parallel processing on multiprocessor machines and groups of workstations is MPI-based.

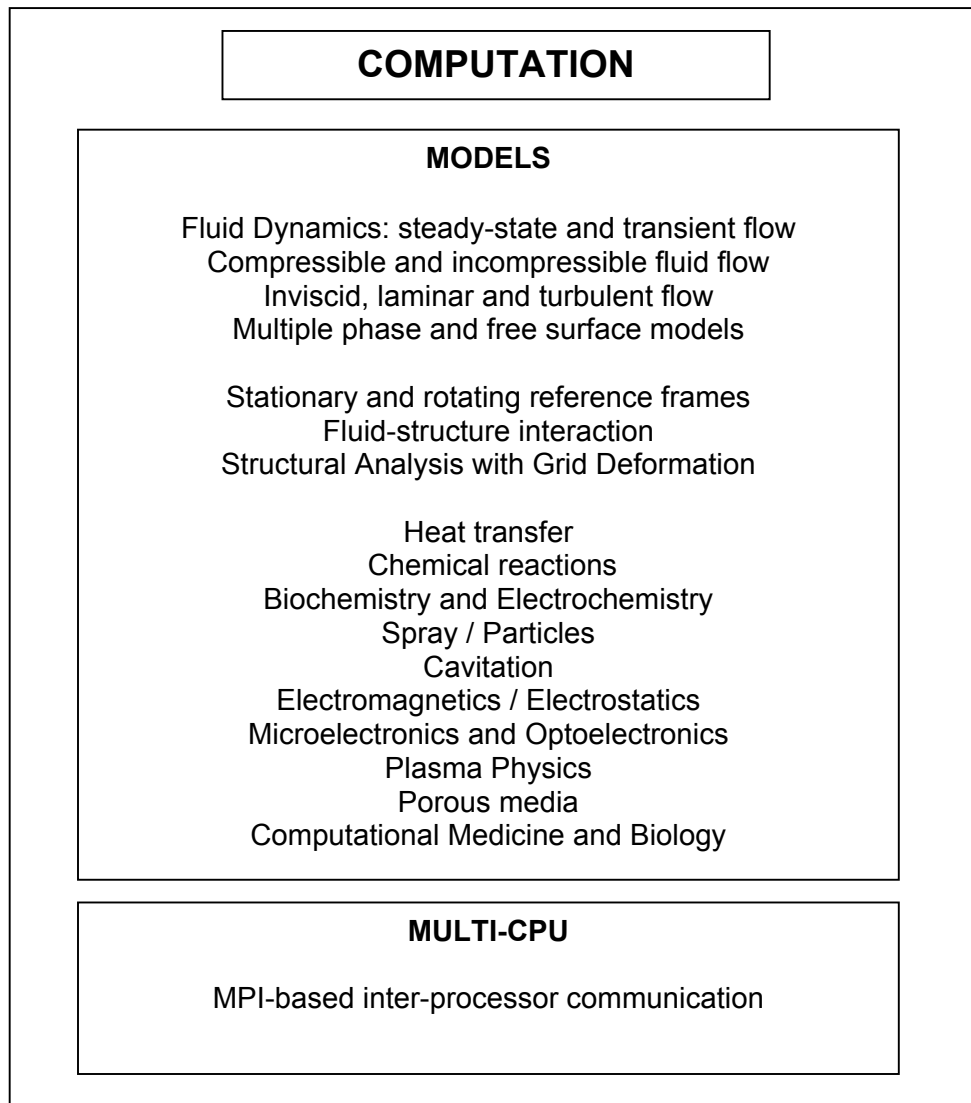


Figure 5.13: The CFD-ACE+ Computation Block.

The 'Post-Processing' functions for CFD-ACE+ are performed by the program module CFD-VIEW (a CFDR product), and are summarized in Figure 5.14. CFD-VIEW is an interactive graphics program for processing numerical results from CFD and other engineering disciplines. The program provides an object-oriented interface combined with a CFD function calculator, which contains both pre-programmed and user-defined functions. Data may be plotted or obtained for a single point using interactive data probes. Some data, such as particle trajectories, may be animated and recorded as movies. The program also provides facilities for annotation and presentation editing. CFD-VIEW is integrated with other CFDR software using the in-house CFD-DTF format. Data may be output in several formats, including bitmap images, enhanced postscript files, or MPEG video format. Like CFD-GEOM, CFD-VIEW supports the open-source scripting language Python. An example screen shot is shown in Figure 5.15.

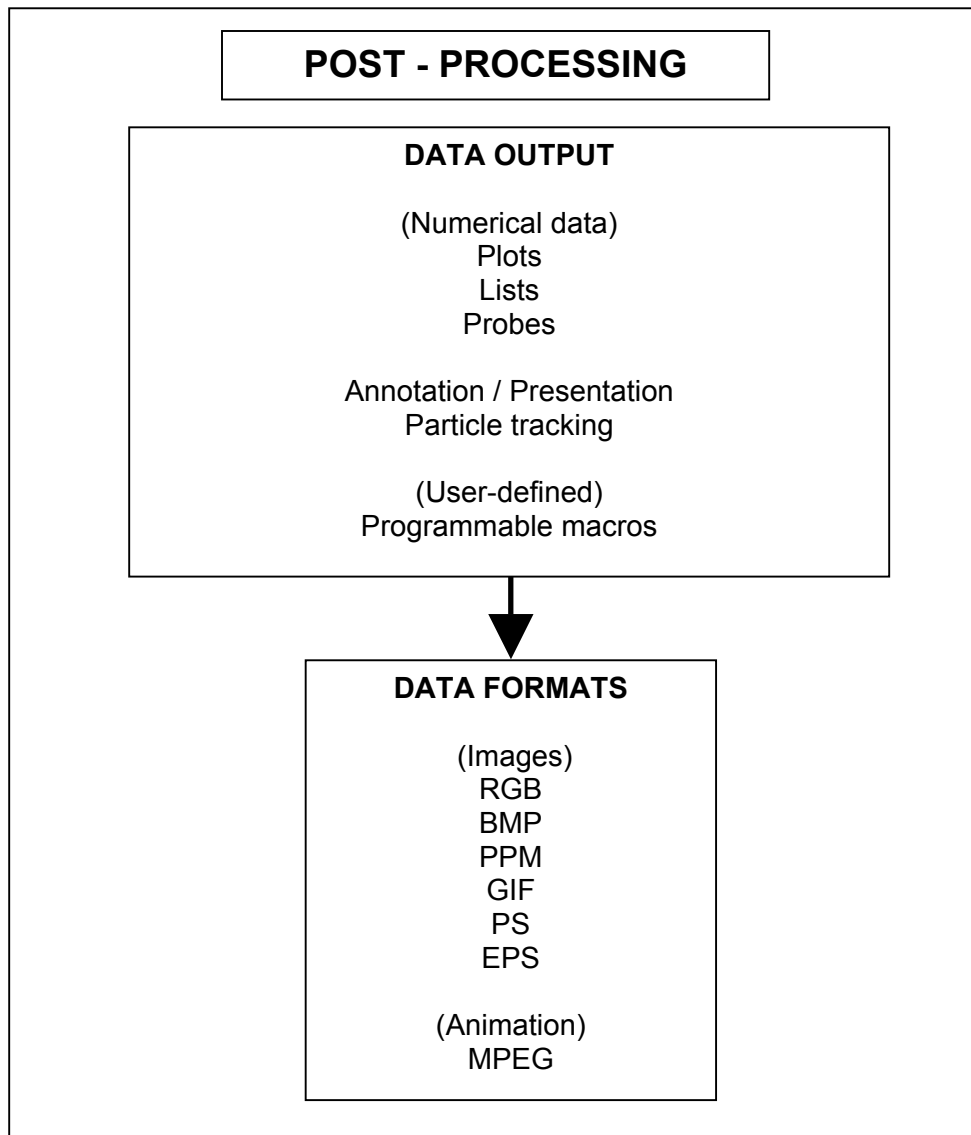


Figure 5.14: The CFD-ACE+ Post-Processing Block.

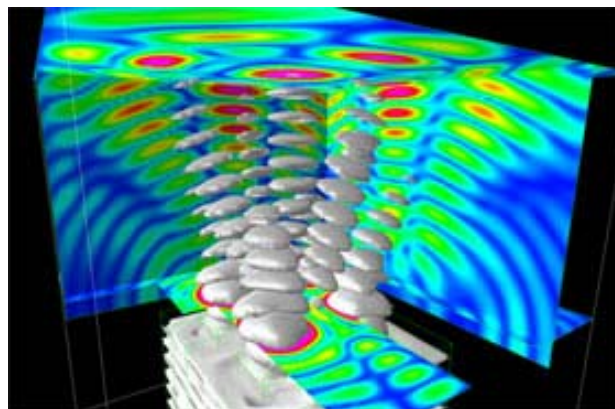


Figure 5.15: An example of CFD-VIEW results output (copyright CFD Research Corp.).

5.4 CFX (ANSYS, Inc.)

ANSYS, Incorporated was founded in 1970, and designs, develops, markets and supports engineering simulation solutions, from mechanical analysis to computational fluid dynamics.

CFX-5 is the latest CFD software from ANSYS, and incorporates coupled multi-grid linear solver technology, meshing flexibility and parallel efficiency in an open architecture. CFX has its origins in the 1970's when AEA Technology, PLC developed CFD for in-house use, the first commercial copy of this predecessor to CFX being leased to ABB in 1987. In 1997 CFX took over ASC Ltd., a Canadian CFD company, and, in 2003, was taken over themselves by the U.S. firm, ANSYS Inc.

Machine Requirements: Specified for PCs running WINDOWS 2000 / XP or Linux Redhat 9.0, also known to run on Redhat 8.0, Redhat Enterprise, Linux 2.1/3.0, Athlon systems, and Opteron systems using 32-bit executables. Also Intel Itanium II systems running Redhat Enterprise, Linux 2.1, or HPUX-11i v2 (11.23). Unix-based systems supported include Sun Microsystems Solaris 8 (also known to run on Solaris 9), SGI IRIX 6.5, Hewlett-Packard HPUX-11i v1 (11.11), and IBM AIX 5.1.

Licensing Arrangements: CFX is priced by the package. The same packages that are available to commercial customers, consisting of pre- and post-processors, along with CFX-4, CFX-5, or CFX-TASCflow, are on offer to the academic community for teaching and/or research purposes at substantial discounts, with flexible licensing options. The licensing costs of CFX are different for serial and parallel usage, with the cost of licensing additional CPU's for a parallel system being less than 10% base price.

User Interface: Geometry can be created from scratch using ANSYS DesignModeler or imported from all major CAD packages. Mesh generation is performed using separate ANSYS or other applicable software. Post-processing is performed by the graphical interface CFX-Post.

The 'Pre-Processing' functions (shown, in summary, in Figure 5.16) are performed by several program modules. The DesignModeler for CFD Applications module (*an ANSYS, Inc. product*) provides geometry creation and meshing functions to add on to CFX-5, and allows the user to amend an imported model. It forms a shared source of geometry not only for CFD, but for other analysis packages produced by ANSYS. Geometry may be imported, or exported in alternative formats such as Parasolid and ACIS. The CFX-Mesh program module has the capability of automatic surface and volume meshing. The program incorporates Advancing Front with Inflation (AFI) meshing technology to construct meshes that model important features precisely. The volume meshing procedure is fully parallelized to accelerate the creation of large meshes. The procedure automatically fine-tunes the mesh spacing for the curvature of surfaces and areas where separate surfaces are in close proximity. The module also gives the user the choice of creating prismatic elements normal to walls by extruding the triangular surface mesh. ICEM CFD (*an ANSYS, Inc. product*) can also be used to produce structured or unstructured meshes, independent of the CAD configuration, for use in CFX-5. ICEM CFD employs object-oriented unstructured meshing technology and generates surface and interior meshes using the Octree approach, automatically applying successive refinements until all mesh density conditions are fulfilled.

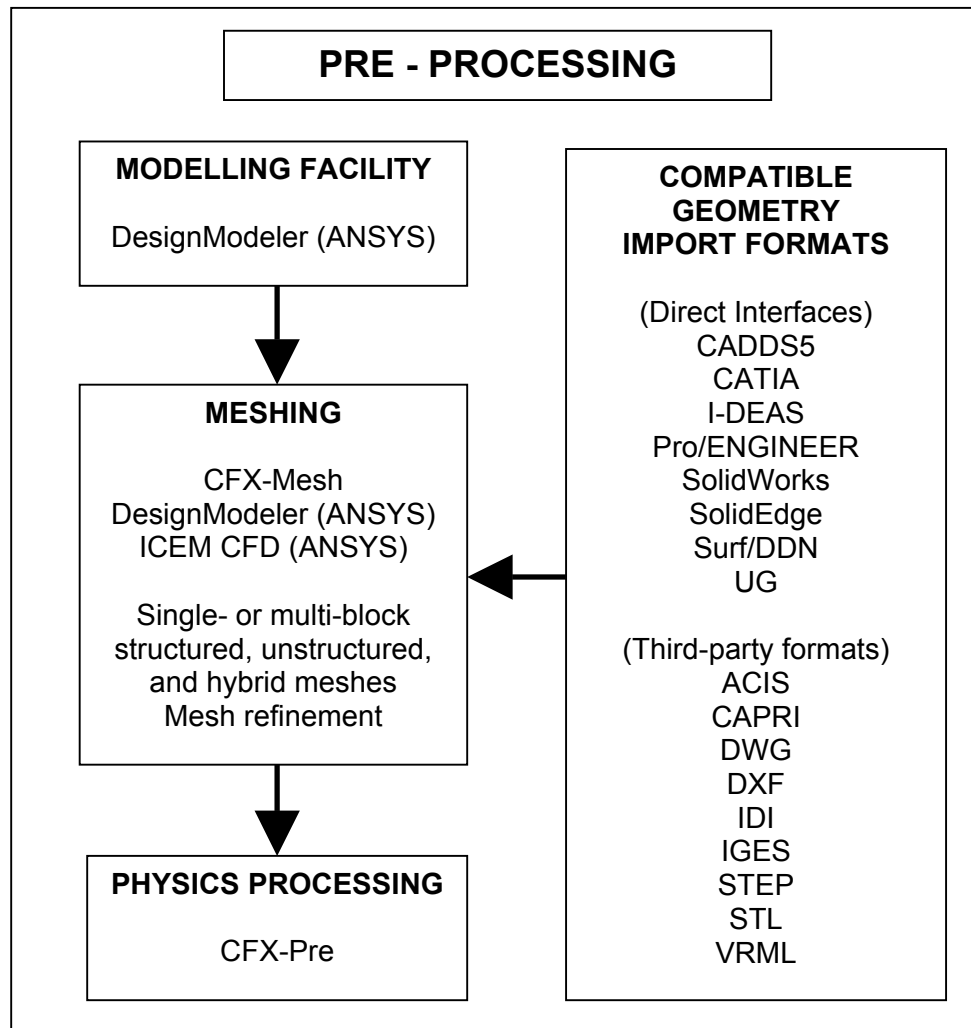


Figure 5.16: The CFX Pre-Processing Block.

The CFX-Pre program module is an interface to permit the definition of the required CFD problem. Problem 'domains' may be defined with a single mesh divided into several domains, or with numerous meshes defined as a single domain, dependent upon the physics of the problem. In addition to the general mode of operation, there are also modes specific to certain applications, such as multi-stage turbomachinery analyses.

CFX-5 uses a coupled algebraic multi-grid Navier-Stokes equations solver, with an adaptive numeric scheme which modifies the discretization locally to approximate second-order problems as closely as possible, whilst guaranteeing that the solution is bounded. The solver does so with efficient utilization of the computer memory. Figure 5.17 shows some of the 'Computation' functions provided by CFX-5. CFX-5 incorporates more than 16 different turbulence models. The behaviour of multiple phases is derived simultaneously using the coupled solver, which supports both Eulerian-Eulerian and Eulerian-Lagrangian multiphase models. Free-surface flow phenomena such as sloshing or filling can be simulated. The program can also take account of surface tension effects. The program supplies a general framework for inter-phase mass transfer as well as models for cavitation, condensation, evaporation and boiling.

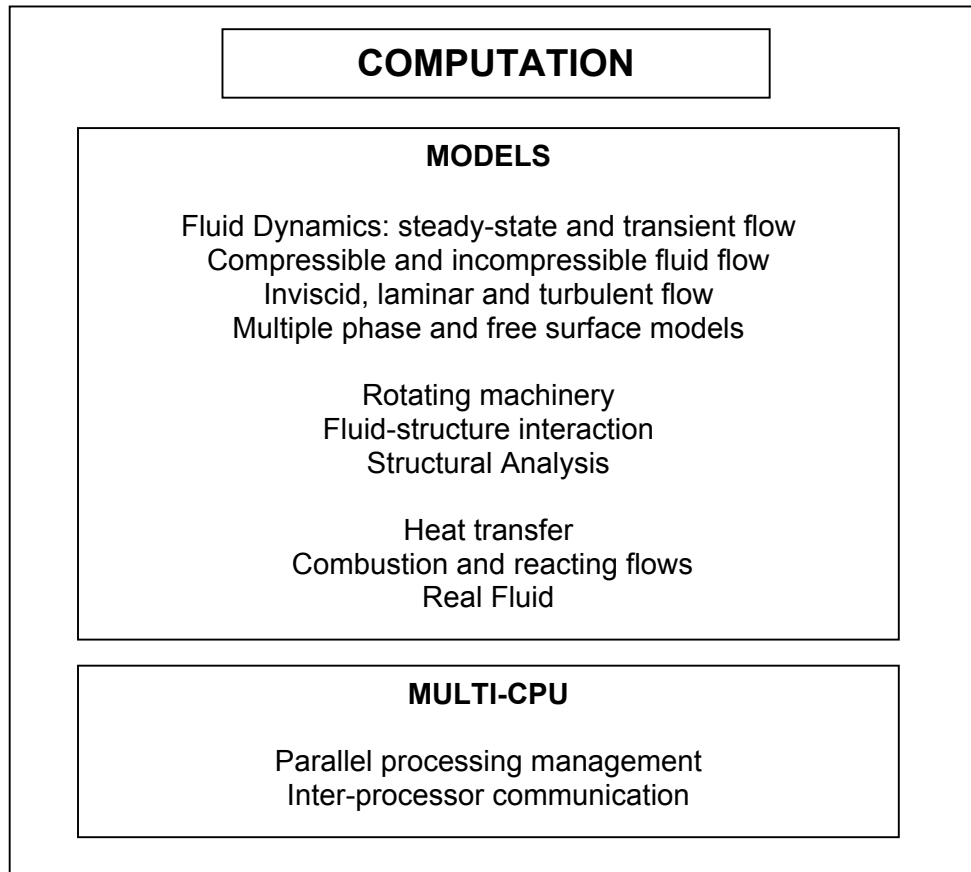


Figure 5.17: The CFX Computation Block.

Fluid-structure interaction simulations can be carried out. In the case of large-scale solid deformations or motions, CFX-5 can be inter-connected dynamically with ANSYS stress analysis and dynamics programs, to give a multi-physics capability. CFX software has a strong history in the analysis of rotating machinery such as pumps, compressors, turbines, propellers, etc. The program contains numerical models for multiple frames of reference and for change of pitch, and steady and transient interaction between rotor and stator. Specialized pre- and post-processing facilities are provided for turbomachinery analysis.

Additionally, CFX-5 may be used to solve heat transfer problems, including conjugate heat transfer and other thermal and combustion problems, for instance, fire propagation, burners and furnaces, and solar heating. The program includes a general structure for single- and multiple-phase reactors, so that the combustion of a range of solid, liquid and gaseous fuels may be modelled, along with multi-fuel or multi-oxidant combustion, radiative heat transfer, prediction of nitrous oxide emissions, and explosions and deflagrations.

The solver has been designed to be fully parallel to distribute the CFD calculation among several processors on any combination of single- or multiple-CPU or networked UNIX workstations and Windows NT machines, including mixed UNIX and Windows NT clusters.

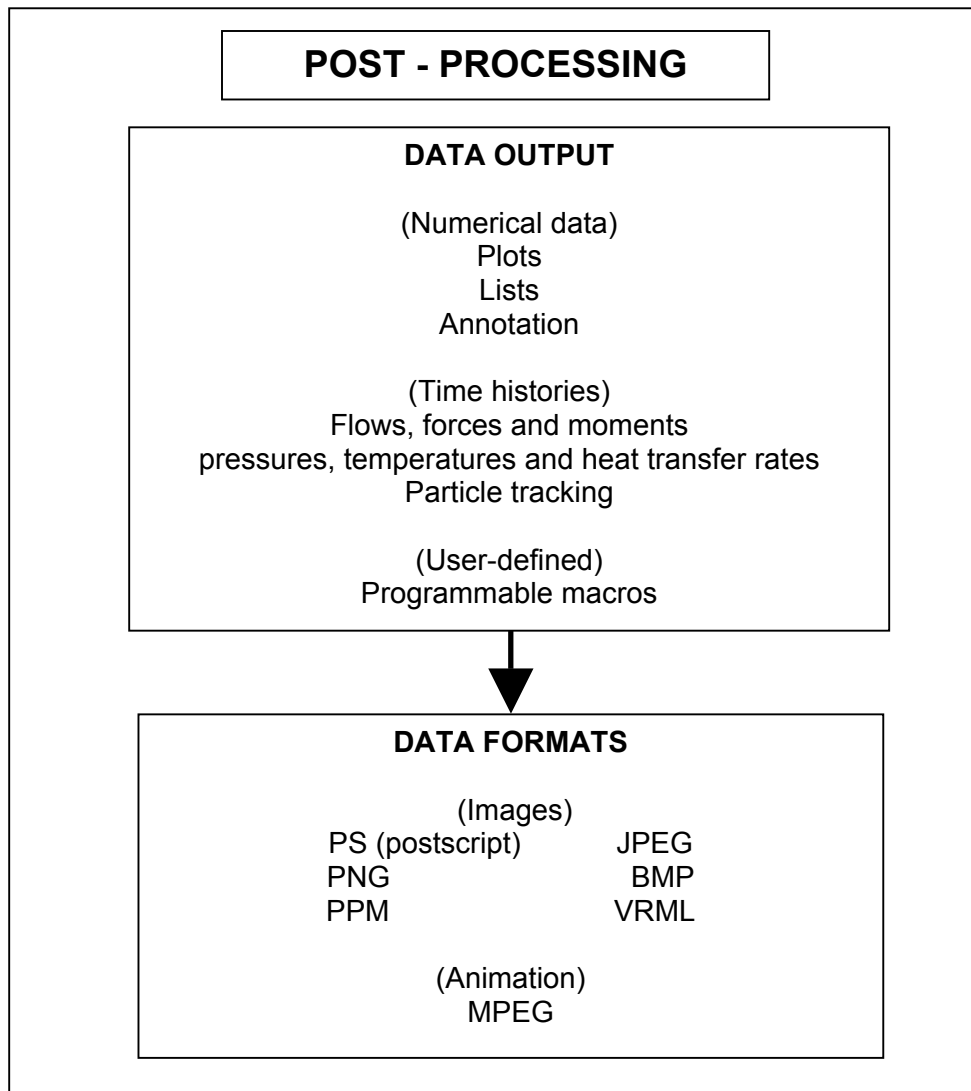


Figure 5.18: The CFX Post-Processing Block.

The CFX-Post program module performs the 'Post-Processing' functions for CFX-5, a synopsis of which is shown in Figure 5.18. The CFX-Post post-processor produces plots (on any region of the body) in vector form or as XY graphs with control over rendering. Contours, fringes, and 'streaklines' may also be plotted. The user can make data comparisons over multiple results and annotate plots with legends and two- or three-dimensional text labels. Results can be animated with automatic MPEG-format file creation. The program also performs quantitative calculations such as geometric evaluations, force and torque calculations, and values for user-defined variables. Data can be exported in several standard image formats. CFX-Post allows programmable post-processing including loops, logic, and subroutines (or macros). There is also special provision for turbomachinery post-processing, with 'turbo-specific' plots and performance macros.

5.5 EFD.Lab (NIKA GmbH.)

NIKA is a privately-held company established in 1999, with world headquarters in Germany. The company develops software applications to model fluid flow and the effects of its contact with solid materials.

While based on the same exact mathematical foundation as other CFD programs, EFD.Lab programs are intended to suit the design engineer, rather than the fluid dynamics specialist. EFD.Lab is, therefore, goal-oriented with a heavy emphasis on CAD integration. EFD.Lab is used by clients in the aerospace, automotive, electronics and military industries.

Machine Requirements: Intel Pentium 4 or AMD Athlon based PC running Microsoft Windows XP Professional, or Windows 2000 with 512MB RAM minimum, (although more is recommended) and at least 700MB free hard disk space for installation, with more required for simulation models. Useful adjuncts include a mouse or other pointing device, a CD-ROM drive, Microsoft Office XP, or Microsoft Office 2000, and Internet Explorer 5.0 or later.

Licensing Arrangements: Licensing arrangements and prices have not been determined. EFD.Lab supports flexible license sharing through LAN or WAN networks.

User Interface: EFD.Lab has a graphical user interface, as shown in Figure 5.19, giving the option of either creating the model using a parametric solid modeler, or importing geometry from other CAD packages.

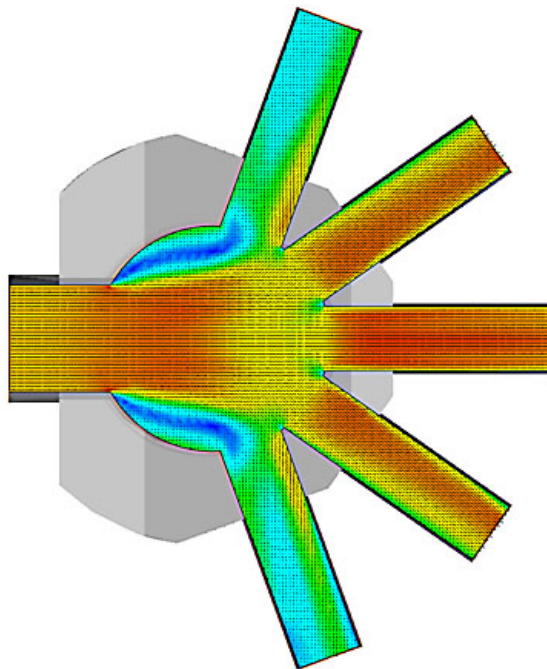


Figure 5.19: An example of EFD-Lab results output (copyright NIKA GmbH.).

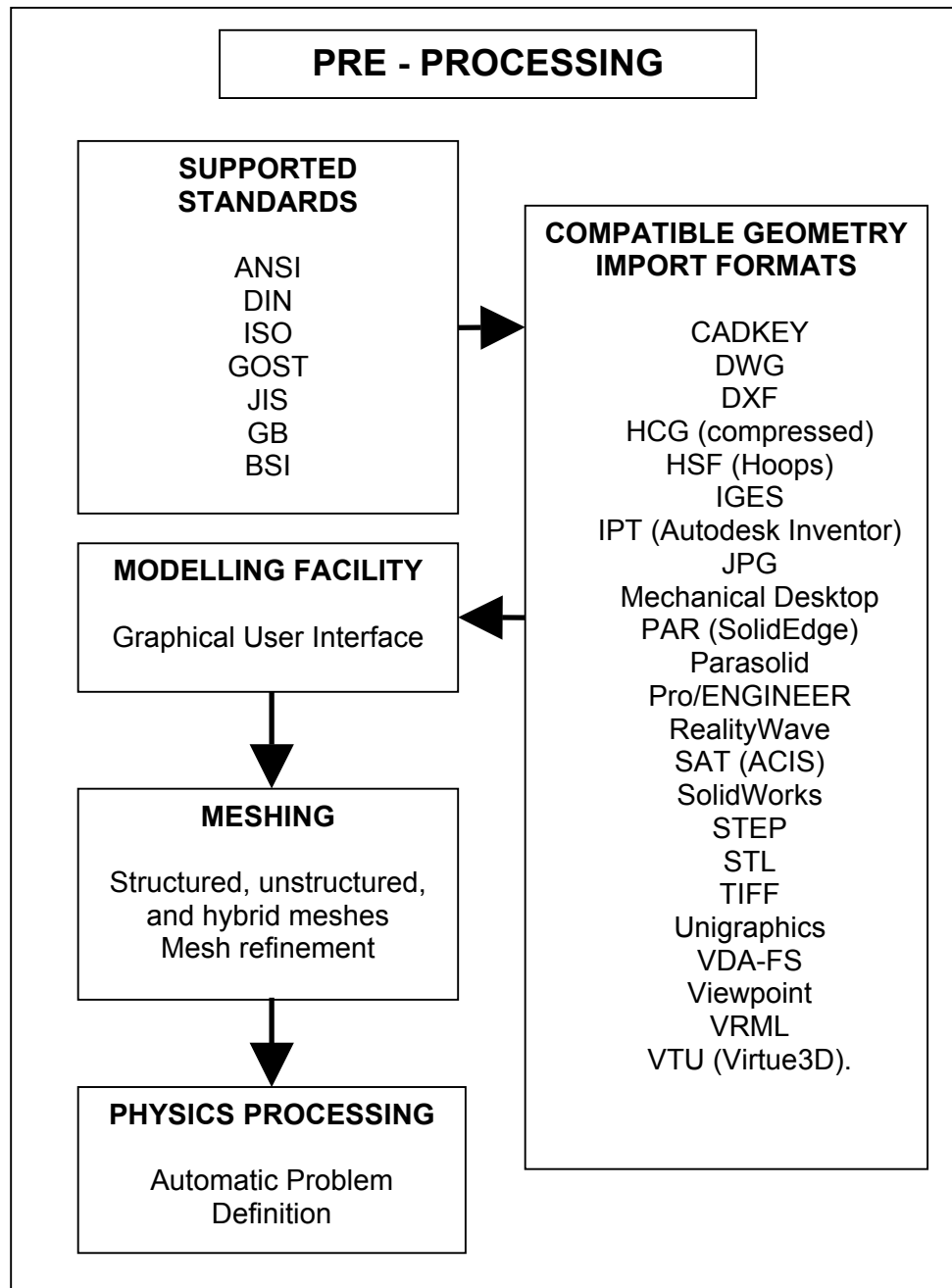


Figure 5.20: The EFD.Lab Pre-Processing Block.

The EFD.Lab 'Pre-Processing' functions are summarized in Figure 5.20. EFD.Lab includes facilities for solid part modelling, which is fully associative and parameterized, and assembly modelling. The modeller supports multiple body design and the creation of three-dimensional models from existing two-dimensional data. Use may also be made of the FeatureManager (*a SolidWorks Corp. product*) dynamic design tree for functions such as re-ordering, etc. Alternatively, the body geometry may be imported from a third-party CAD package. EFD.Lab provides translators for a plethora of third-party formats.

Mesh generation can be automatic or manually controlled, with direct meshing of the solid model geometry and no requirement for a separate model of the fluid domain. Mesh refinement or coarsening, depending on the physics of the problem, can be achieved automatically or manually with the capability of local mesh settings for optimized meshes. The program uses the definition of engineering simulation goals as the basic commands for the analysis control procedure, by intelligent determination of the computational domain for both internal and external flows. There is also provision for verifying input data and partial solution data to avoid geometric errors.

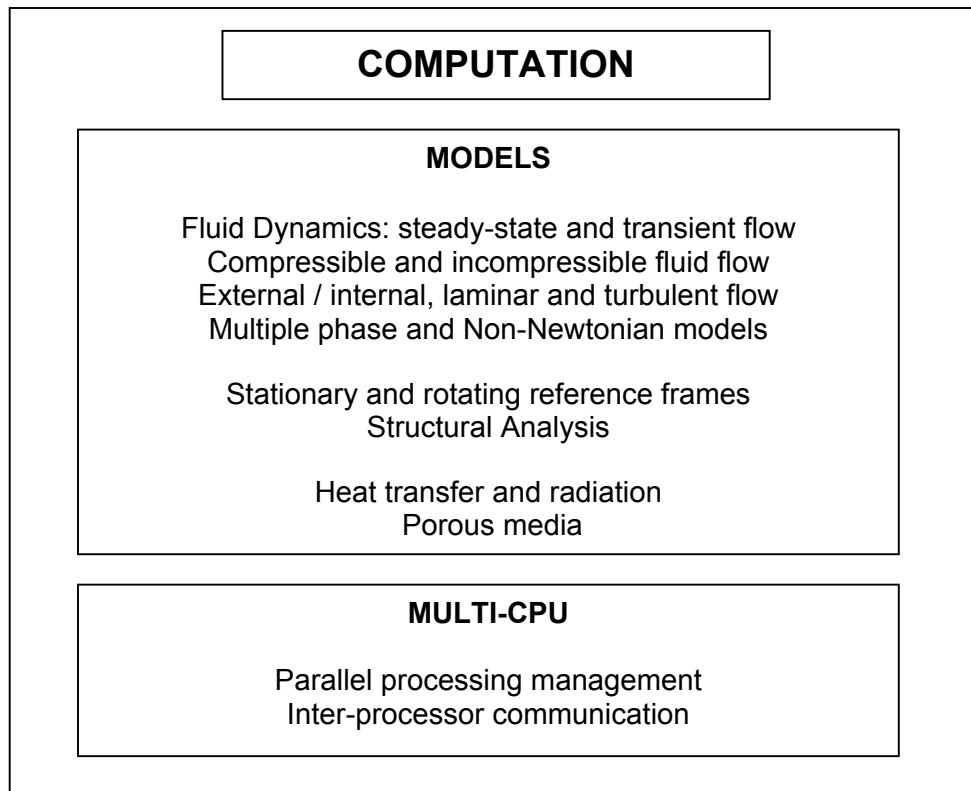


Figure 5.21: The EFD.Lab Computation Block.

The 'Computation' functions performed by EFD.Lab are shown in Figure 5.21. The physical modelling capabilities include compressible or incompressible liquid viscous flows and gas viscous flows in all speed regimes. The flows may be external or internal and the choice between laminar or turbulent solutions (with transition between the two) is made automatically. Non-Newtonian liquids and multiple-fluid flow may be modelled, subject to forced, free or mixed convection. Rotating parts are accommodated by the use of a rotating reference frame and the model includes centrifugal and Coriolis forces.

EFD.Lab provides limited linear static analysis capabilities, such as stress or displacement derivation, itself, but offers compatibility with the finite-element structural analysis package COSMOSWorks (*a SolidWorks Corp. product*). Also modelled is conjugated heat transfer in fluids and solids, along with conduction, convection and radiation. Fluid flows in porous media may also be modelled. EFD.Lab supports multi-processor PCs for parallel processing

A synopsis of the EFD.Lab 'Post-Processing' functions is depicted in Figure 5.22. Results such as time-dependent fluid flow, heat and mass transfer may be plotted or listed or displayed directly either on the solid model, or, optionally, on the computational mesh. EFD.Lab offers full support of OpenGL graphics. Virtual probes indicate local steady-state and transient results at specified points. Flow trajectories may be plotted as traces in three-dimensional space. Particle tracking is possible, with additional data including density, size, and initial conditions. Both steady-state and transient results can be animated. Some solutions may be output in numerical form to MS Excel or ASCII file. Selected plot types may be exported in VRML format. There is compatibility with Microsoft Office for the export of geometry configurations and the evaluation of analysis results, such as text reports, which can be generated in MS Word for Windows.

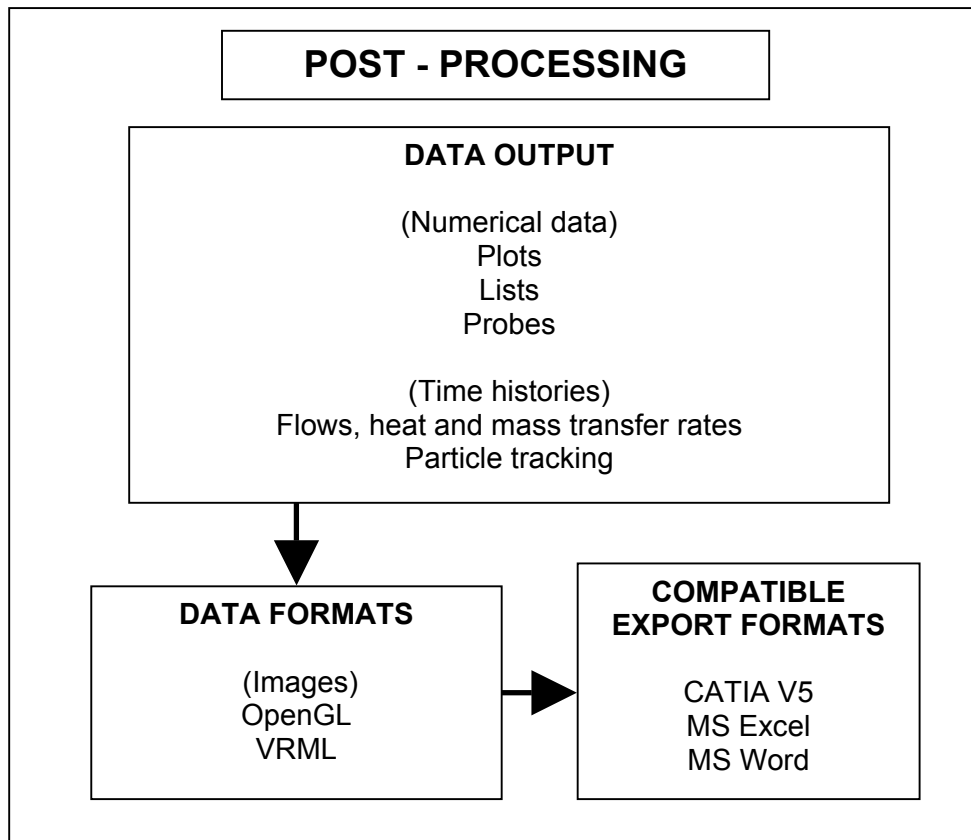


Figure 5.22: The EFD.Lab Post-Processing Block.

5.6 FLUENT (Fluent, Inc.)

The first version of the FLUENT code was launched in October 1983 by a New Hampshire company called Creare Inc. The Fluent group at Creare became a separate company in 1988. In August, 1995, Fluent Inc. was acquired by Aavid Thermal Technologies, Inc., a company specializing in the thermal management of electronic systems. In May, 1996, Fluent acquired Fluid Dynamics International, the developer of the FIDAP software code and this was followed in 1997 by the acquisition of Polyflow S.A., the developer of the POLYFLOW software code for CFD analysis of laminar flows. In January, 2000, Aavid Thermal Technologies was merged with a newly formed entity owned by Willis Stein & Partners, a equity investment firm.

FLUENT is in widespread use in many applications and is based around an unstructured, finite volume based solver for many types of flow and speed regimes. In addition to FLUENT, the company also provides three other general-purpose products. FloWizard is a general-purpose CFD product for designers. FIDAP and POLYFLOW are also used in a wide range of fields, with emphasis in the materials processing industries.

Machine Requirements: Current platform availability is obtainable on request (when one is registered with the Fluent Online Support system). A minimum of 256MB RAM is recommended. Disk space required for the software is 35MB on a machine running WINDOWS, and 50-75MB on a machine running UNIX or LINUX. An additional 50MB will be needed if the documentation and online help is downloaded to the hard drive. Fluent Inc. products support various graphics display drivers, the principal ones being Professional Open GL graphics cards on WINDOWS-based machines, and the X11 windows driver on UNIX systems.

Licensing Arrangements: Available on request, (when one is registered with the Fluent Online Support system). Annually-renewable university department or site licenses are available at over 90% discount from standard commercial software rates.

User Interface: Three preprocessing tools are available for use with FLUENT. Additionally, third-party preprocessing tools can also be used. Full post-processing capabilities are included, but third-party post-processors may also be used.

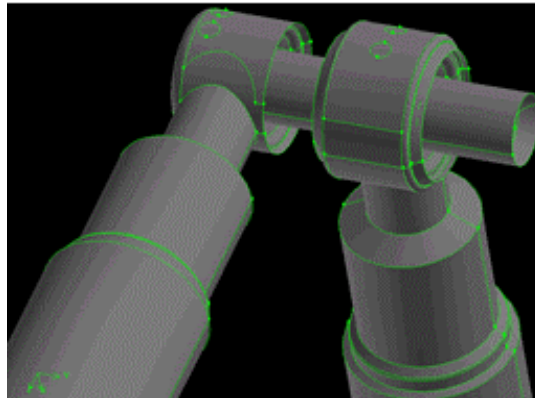


Figure 5.23: An example of fluid domain analysis in GAMBIT (copyright FLUENT, Inc.).

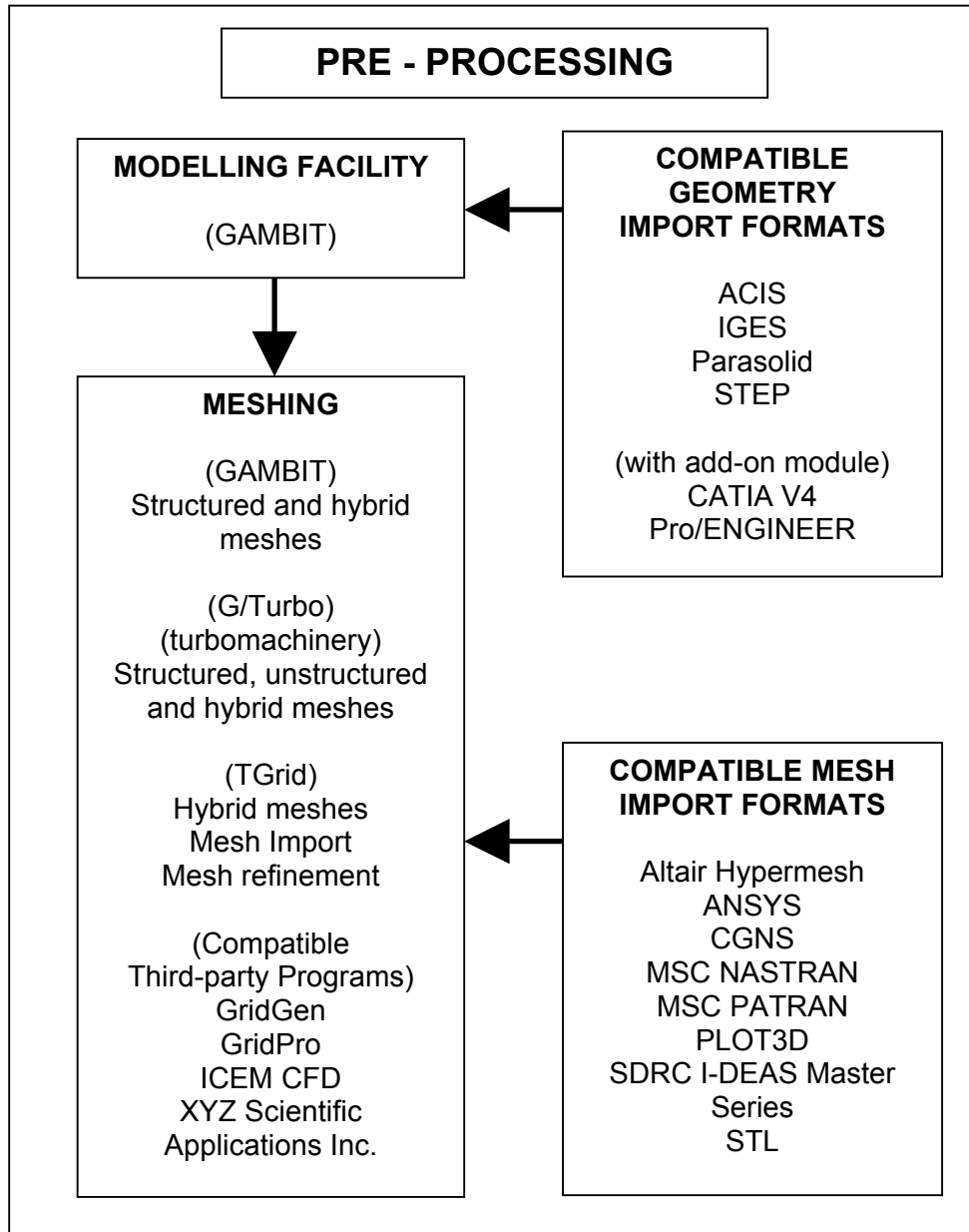


Figure 5.24: The FLUENT Pre–Processing Block.

Figure 5.24 gives a summary of the FLUENT ‘Pre–Processing’ functions. For geometry modelling and meshing, FLUENT, Inc., produces GAMBIT, a single interface for both activities, which can also import geometry from many CAD/CAE software platforms, some requiring extra, add-on translators. The solid-modelling-based geometry includes means to extract the fluid domain from imported geometry, with further breakdown achieved using Boolean operations. There are also semi-automatic geometry repair mechanisms to identify and retouch defects such as holes and overlapping faces. Facilities for logging activities allow the user to edit and rerun sessions. The meshing capabilities provided with GAMBIT permits geometry decomposition for structured

hexahedral meshing or performs hexahedral meshing automatically, giving the user control over element clustering. Triangular surface meshes and tetrahedral volume meshes can be created along with hybrid meshes. G/Turbo (a FLUENT, Inc., product) is designed for the creation of FLUENT-format meshes for turbomachinery applications. In addition to being application-specific, G/Turbo offers unstructured meshing. TGrid (a FLUENT, Inc., product) is designed for hybrid volume and two-dimensional mesh generation. TGrid also facilitates the importation of surface and volume meshes from third-party mesh generation programs and includes means for enhancing the quality of imported meshes and for assembling meshes from several components. Some third-party pre-processors can write mesh files directly in the correct format for FLUENT.

The 'Computation' functions that can be carried out in FLUENT are shown in Figure 5.25. Steady-state and transient, Newtonian or non-Newtonian, flows can be modelled in the low subsonic, transonic, supersonic, and hypersonic speed regimes. The flows can be inviscid, laminar, and turbulent and be multi-phase with a free surface. The model reference frames can be stationary (inertial), rotating or accelerating (non-inertial), with dynamic mesh reconfiguration to enable the modelling of flows around moving bodies. Multiple reference frames and sliding meshes are added options in this context.

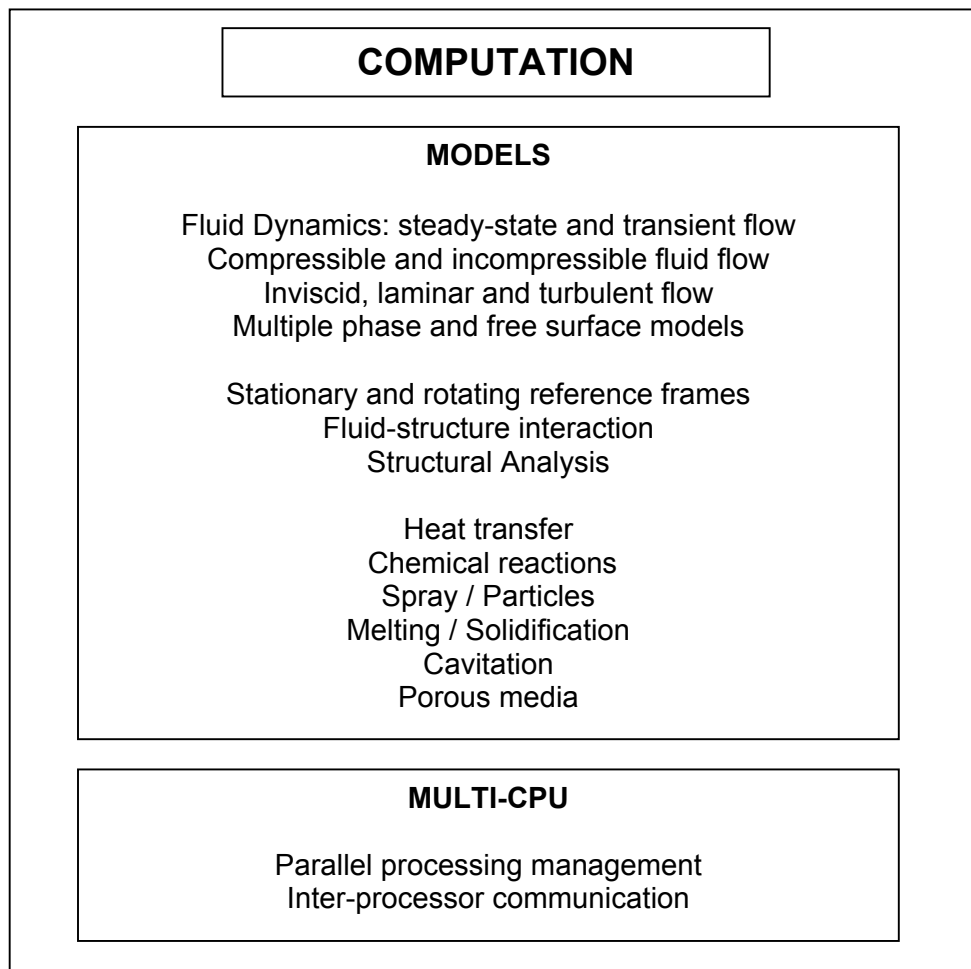


Figure 5.25: The FLUENT Computation Block.

Additionally FLUENT models heat transfer with forced, natural, or mixed convection, with conjugate heat transfer, and radiation. Chemical species transport and reaction can be modelled, including combustion and surface reaction. The trajectories of particles, droplets, and bubbles in a flow can be tracked. FLUENT can model phase change in melting or solidification problems. The flow models may also include cavitation or porous media. FLUENT supports multi-processor systems for parallel processing.

FLUENT, Inc., also manufacture FloWizard, an automated flow simulation package for design and process engineers with a rudimentary comprehension of fluid dynamics. FloWizard is designed for ease of use and high productivity.

FLUENT assert that all their software includes full post-processing capabilities, which can be broadly customized by the use of user-defined functions. Specific details, however, are not publicized. FLUENT software is compatible with a range of third-party post-processors, such as the following:

- Acuitiv (A Division of Fuel Tech, Inc.)
- AVS (Advanced Visual Systems Inc.)
- EnSight and EnSight Gold (MSC Software)
- FAST (Flow Analysis Software Toolkit, NASA Advanced Supercomputing Division)
- Fieldview (Intelligent Light)
- IBM Open Visualization Data Explorer
- Tecplot (Tecplot, Inc.)

5.7 PHOENICS (CHAM, Ltd.)

Concentration, Heat and Momentum Limited (CHAM) was founded in 1974 as an engineering consultancy and software company, operating world-wide from headquarters in Wimbledon, England. CHAM provides flow-simulation services, utilizing CFD techniques, in a range of situations from commissioned consultancy, through remote-access computing, to the sale or lease of software and its user support.

PHOENICS was the first general-purpose CFD code to be made available commercially, and was launched by CHAM in 1981. PHOENICS is designed to cope with multi-phase turbulent fluid flow, solid-stress and thermal interactions simultaneously.

Machine Requirements: PHOENICS is supplied as 'WinDF PHOENICS' for any PC using the Windows NT or Windows 95 operating system. In this case, PHOENICS employs the Digital Fortran compiler with the Open-GL graphics library. Windows-95/98 (but not Windows-NT) users can make use of the Salford Compiler and DBOS memory manager in the 'WinSD PHOENICS' version. PHOENICS, its add-ons and utilities can be supplied for all popular UNIX work-stations, details available on request. PHOENICS has been "parallelized" in a generic manner, to run efficiently on a wide range of parallel-architecture computers and PC clusters, using both the LINUX and WINDOWS-NT operating systems.

Licensing Arrangements: A set of rules, of somewhat Byzantine complexity, governing the relationship of different set-up configurations and platforms to a base price, which is not quoted, can be found on the PHOENICS website. CHAM is making a 'test-drive' version of PHOENICS accessible via the Internet.

User Interface: The WinDF PHOENICS version Virtual-Reality user interface is 'Windows-like'. The WinSD VR interface is not as flexible as the WinDF version, since the Open-GL format cannot be used, owing to limitations of the Salford/DBOS system.

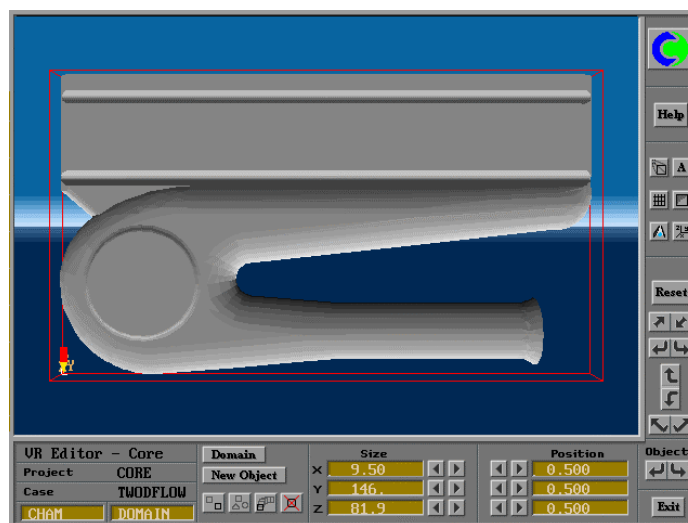


Figure 5.26: A CAD file imported into the PHOENICS-VR Editor (copyright CHAM, Ltd.).

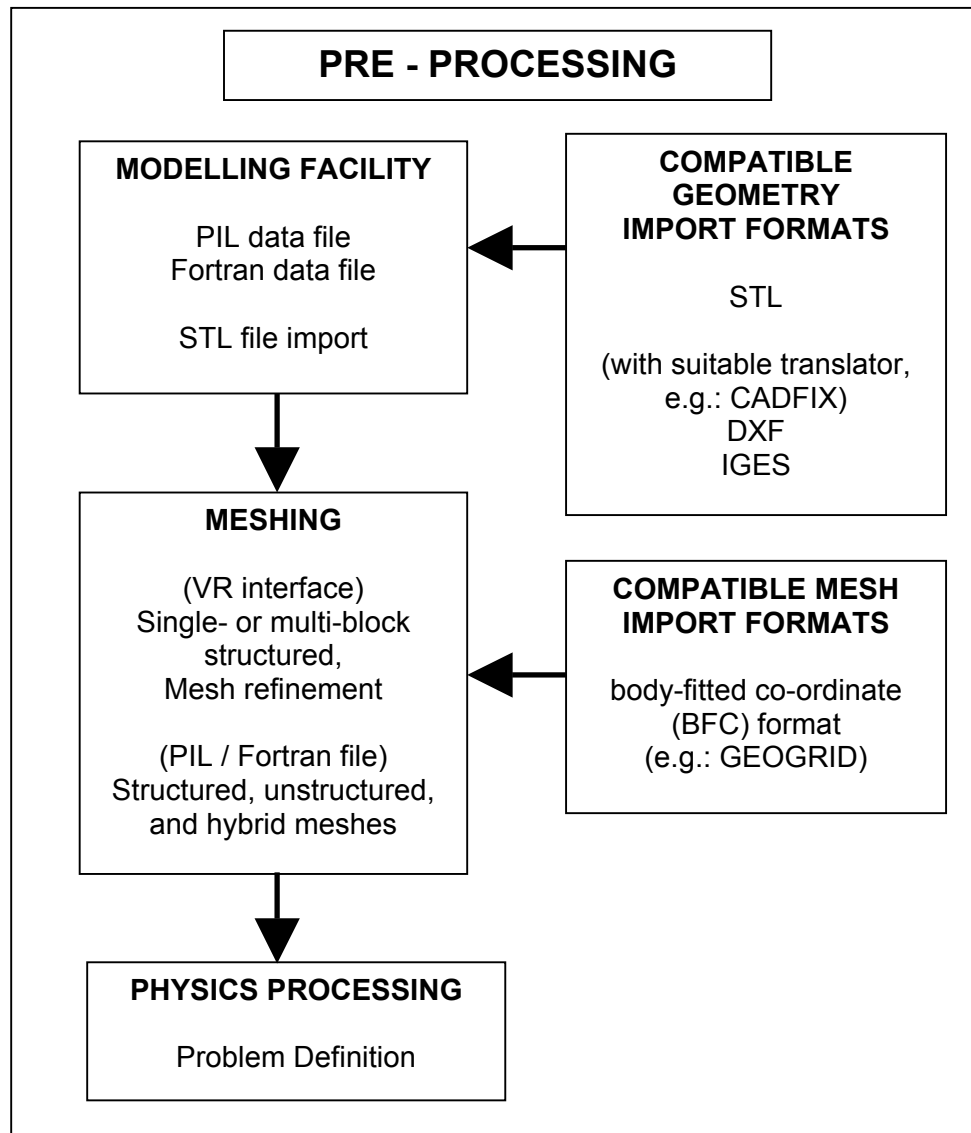


Figure 5.27: The PHOENICS Pre-Processing Block.

The PHOENICS 'Pre-Processing' functions are summarized in Figure 5.27. The graphical 'Virtual-Reality' user interface handles geometric data input and meshing. The geometric data may be input as a command file, written in the PHOENICS Input Language, PIL, describing the discretized surface panels (called 'facets' in PHOENICS). Users are assisted by libraries of input files, which can form the basis of new flow simulations, and which can be extended by the user. In a similar manner, data may also be input using Fortran coding. Solid-object geometry can be created in a CAD package and imported into the PHOENICS-VR interface, possibly with the use of an intermediary program. The 'Virtual-Reality' interface includes a converter which generates the multi-block structured surface mesh from the imported geometry. Alternatively, the surface mesh itself may be imported, in body-fitted co-ordinate (BFC) form from a suitable mesh generation package. Problem-defining values such as material properties, boundary and initial conditions are input into the VR-interface by the use of menus.

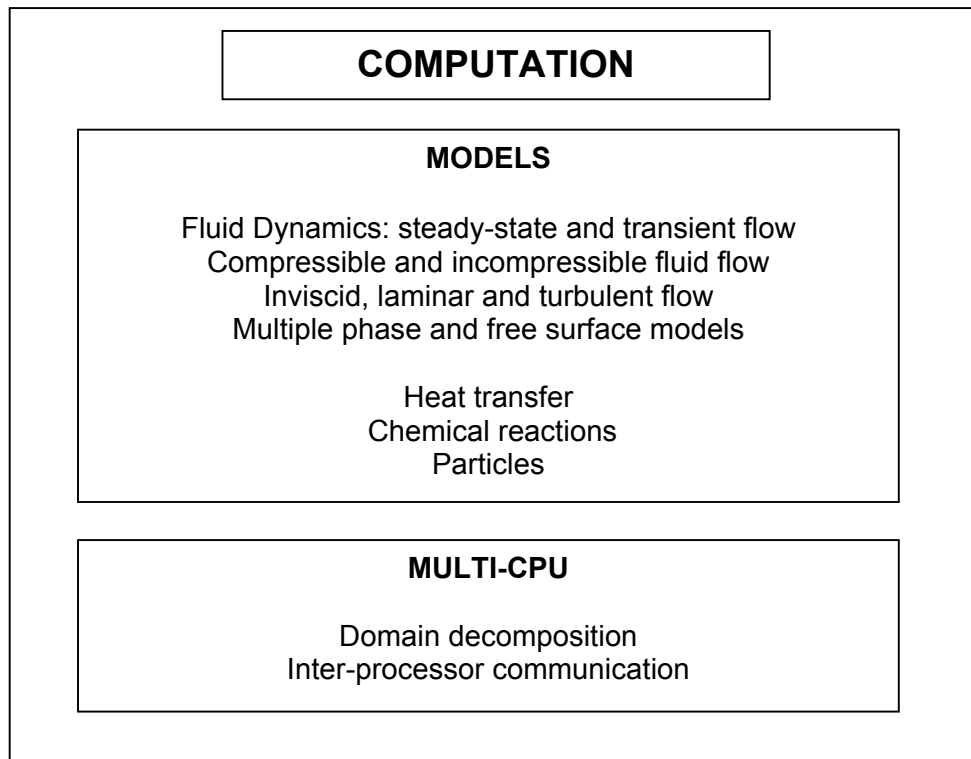


Figure 5.28: The PHOENICS Computation Block.

Figure 5.28 shows the variety of 'Computation' functions that PHOENICS carries out. PHOENICS simulates four separate types of multi-phase flow. Two are models of interpenetrating continua, of two or more fluids. The third is of two non-interpenetrating continua, separated by a free surface, which is tracked by one of three methods. The three tracking methods are: the height-of-fluid method; the scalar-equation method; and the particle-on-surface method. The first method cannot deal with breaking waves, since the free-surface height at the pertinent horizontal position becomes multi-valued in this case. The fourth multi-phase model is a particulate phase for which the particle trajectories are computed as they move through a continuous fluid by solving the Lagrangian equations of motion. PHOENICS contains an embarrassment of turbulence models, two of which are unique to PHOENICS.

Additionally, PHOENICS has several procedures for computing heat radiation and for simulating chemical-reaction processes, especially those involving combustion. There are also several special-purpose versions of PHOENICS dedicated to processes such as: blast-furnaces, explosions and fires in off-shore oil platforms, the flow of air, heat and smoke inside buildings and other enclosures, the flow of heat and air in electronic equipment, and oil spills in rivers. The parallelization strategy used in PHOENICS is 'domain-decomposition', achieved by dividing the entire three-dimensional grid into the same number of sub-grids as processors available for use, with the transfer of information from one sub-grid to the next performed within the innermost iteration loops of the solution algorithm.

An outline of the 'Post-Processing' functions available with PHOENICS is given in Figure 5.29. Numerical results are saved to a text file, which, when commanded appropriately, contains either a small or a large amount of information. The VR-interface is also used for inspecting the simulation results, in the guise of the 'VR Viewer', in which objects can be viewed as wire-frames, and data can be plotted as streamlines, velocity vectors on planes, temperature contours on a plane and surfaces of equal temperature. Successive planes can be plotted and animated. There is also the interactive graphics program, PHOTON (PHOENICS OuTput optiON), which can output the mesh grids, or mesh outlines, over the whole or parts of the domain; vector fields, contour plots, streamlines, particle tracks, and text annotation. Results may also be exported to a number of third-party packages.

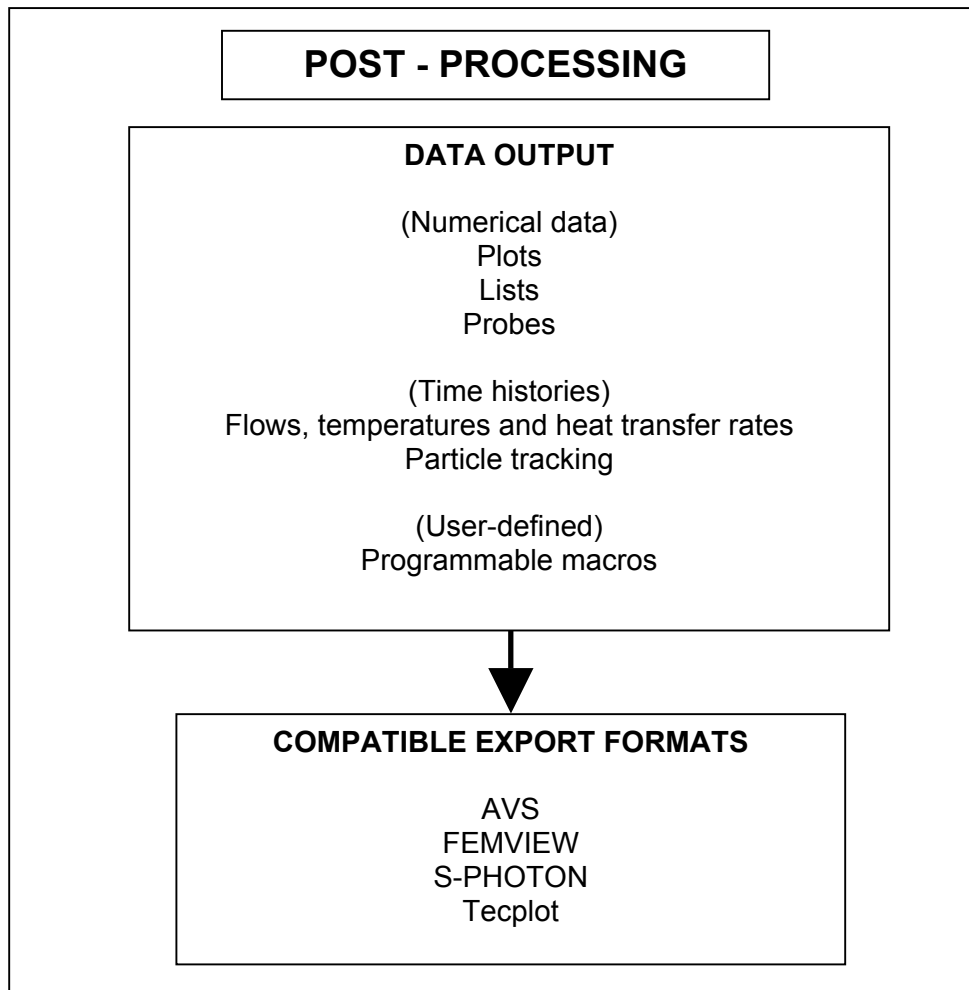


Figure 5.29: The PHOENICS Post-Processing Block.

APPENDICES

A.1 Hydrodynamics Software Directory

	Address	Tel. / Fax. / E-mail / Internet
AQWA	Century Dynamics Ltd. Dynamics House Hurst Road Horsham West Sussex RH12 2DT ENGLAND	Tel: +44 (0) 1403 270066 Fax: +44 (0) 1403 270099 E-mail: all@centdyn.demon.co.uk Internet: http://www.centurydynamics.co.uk
HYDROSTAR	Bureau Veritas 17 bis place des Reflets 92077 Paris La Defense Cedex FRANCE	Tel: +33 1 42 91 33 60 Fax: +33 1 42 91 33 95 E-mail: veristarinfo@bureauveritas.com Internet: http://www.veristar.com/veristar_offshore/hydrostar.html
MOSES	3100 S. Gessner Suite 325 Houston Texas 77063 USA	Tel : +1 713-975-8146 Fax: +1 713-975-8179 E-mail: support@ultramarine.com Internet: http://www.ultramarine.com/g_info/moses/moses.htm
NEPTUNE	Zentech, Inc. 8582 Katy Freeway Suite 205 Houston Texas 77024 USA	Tel: +1 713-984-9171 Fax: +1 713-984-9175 E-mail: info@zentech.co.uk Internet: http://www.zentech.co.uk/neptune.htm
WADAM	DNV has several offices in U.K., go to website at: www.dnv.com , follow 'find us' link and click on 'United Kingdom'	Tel: / Fax: <i>see left</i> E-mail: requests via electronic form on website Internet: http://www2.dnv.com/software/Products/SESAM/ProductPages/Wadam/Wadam.htm
WAMIT	WAMIT, Inc. 822 Boylston Street Suite 202 Chestnut Hill Massachusetts 02467-2504 USA	Tel: +1 617-739-4488 Fax: +1 617-739-4499 Email: info@wamit.com Internet: http://www.wamit.com
WAVELOAD	Martec Limited 1888 Brunswick Street Suite 400 Halifax, Nova Scotia CANADA B3J 3J8	Tel: +1 902-425-5101 Fax: +1 902-421-1923 Email: info@martec.com Internet: http://www.martec.com/Software/software_wave.html

A.2 CFD Software Directory

	Address	Tel. / Fax. / E-mail / Internet
ADINA	ADINA R & D, Inc. 71 Elton Avenue Watertown, Massachusetts 02472, USA	Tel: +1 (617) 926-5199 Fax: +1 (617) 926-0238 E-mail: info@adina.com Internet: http://www.adina.com/
	Product Development Services Ltd. 58 West Street Warwick UK CV34 6AW	Tel: +44 (0)1926 490035 Fax: +44 (0)1926 490095
CFD++	Metacomp Technologies, Inc. 28632-B Roadside Drive, Suite 255 Agoura Hills, California 91301, USA	Tel: +1 (818) 735-4880 Fax: +1 (818) 735-4881 E-mail: info@metacomptech.com Internet: http:// www.metacomptech.com/
CFD-ACE+	CFD Research Corporation 215 Wynn Drive Huntsville, Alabama 35805, USA	Tel: +1 (256) 726-4800 Fax: +1 (256) 726-4806 E-mail: cfdinfo@esi-group-na.com; webcontact@cfdr.com Internet: http://www.cfdr.com/ products/ace/
CFX	CFX UK Gemini Building Fermi Avenue Harwell International Business Centre Didcot, Oxfordshire, UK OX11 0QR	Tel: +44 (0)1235 448018 Fax: +44 (0)1235 448001 E-mail: cfx-info-uk@ansys.com Internet: http://www-waterloo.ansys.com/ cfx/products/cfx-5/
EFD.Lab	NIKA GmbH Eiserne Hand 19 60318 Frankfurt am Main Germany	Tel: +49.69.130253 0 Fax: +49.69.130253 53 Email: info@nika.biz Internet: http://www.nika.biz/index2.htm
FLUENT	Fluent Europe Ltd Sheffield Business Park Europa Link Sheffield, UK S9 1XU	Tel: +44 (0)1142 818888 Fax: +44 (0)1142 818818 Email: info@fluent.co.uk Internet: http://www.fluent.com/index.htm
PHOENICS	CHAM Ltd Bakery House, 40 High Street, Wimbledon Village, London UK SW19 5AU	Tel: +44 (0)1819 477651 Fax: +44 (0)1818 793497 E-Mail: sales@cham.co.uk Internet: http://www.simuserve.com/phoenics/ d_polis/d_info/phover.htm

A.3 Trademarks

The following product names, mentioned in the text, are trademarks:

AceCAD	Genie	Plot3D
ACIS	GEOGRID	PNG
Acuitiv	GIF	POLYFLOW
ADINA	GridGen	Postresp
AIX	GridPro	PPM
Altair	HCG	Prefem
AMD Athlon	HPUX	Preframe
ANSYS	HSF	Pro/ENGINEER
AQWA	HydroD	PS
ARIANE 3D	HydroSTAR	RealityWave
ASAS	IBM	Redhat
AVI	ICEM CFD	RGB
AVS	I-DEAS	SACS
BMP	IDI	SAT
CADDS5	IGES	Sestra
CADFIX	INTEL	SOLARIS
CADKEY	IPT	SolidEdge
CAPRI	IRIX	SolidWorks
CATIA	Itanium	SPARC
Celeron	JPEG	S-PHOTON
CFD++	JPG	Spray
CFD-DTF	LINUX	STEP
CFD-GEOM	Mechanical Desktop	STL
CFD-VIEW	MOSES	StruCAD*3D
CFX	MOTSIM	SunOS
CFX-TASCflow	MPEG	Surf/DDN
CGNS	MS Excel	TAB
COSMOSWorks	MS Office	Tecplot
DeepC	MS WINDOWS	TGrid
DesignModeler	MS Word	TIFF
DWG	MSC NASTRAN	Trident FEA
DXF	MSC PATRAN	UG
EFD.Lab	MultiSurf	Unigraphics
EnSight	NEPTUNE	UNIX
EPS	NSO	VDA-FS
FAST	Octree	VeriSTAR
FeatureManager	OpenGL	Viewpoint
FEMGV	Opteron	VISU4D
FEMVIEW	OSCAR	VRML
FIDAP	OTIS	VTU
Fieldview	PAR	WADAM
FloWizard	Parasolid	WAMIT
FLUENT	Patran-Pre	WaveLoad
Framework	Pentium	Xtract
G/Turbo	PHOENICS	XYZ S.A.
GAMBIT	PHOTON	ZenMoor