ICCAS 2007, Portsmouth

EFFICIENT GENERATION OF CFD-BASED LOADS FOR THE FEM-ANALYSIS OF SHIP STRUCTURES

H Eisen and **C Cabos**, Germanischer Lloyd AG, Germany

SUMMARY

Strength analysis of ship structures by means of FEM requires realistic loads. The most realistic loads will result from CFD simulations. GL have specified guidelines for a design verification process by means of FEM-analysis with CFD-based loads based on the equivalent design wave approach.

The GL ShipLoad program integrates all algorithms necessary to access and combine the data of the CFD computations and the FEM-model for generating the appropriate CFD-based FEM loads. It also provides a GUI to easily control the above load generation process in a time- and cost-saving manner.

This paper describes the technology for coupling CFD and FEM computations and for selecting the appropriate design waves. One core component for linking CFD and FEM is a repository for the ship hydrodynamic results (SHR). Another important component is an algorithm which maps CFD pressures to FEM nodal loads where CFD and FEM meshes need not coincide. Both the exchange file and the mapping algorithm are independent of the CFD method. Therefore, the currently applied efficient strip method can be replaced by more advanced procedures.

1. INTRODUCTION

Strength analysis of ship structures by means of FEM requires realistic loads. The most realistic loads will result from CFD simulations. GL have specified guidelines for a design verification process by means of FEM-analysis with CFD-based loads based on the equivalent design wave approach [1].

The structural analysis needs to account for external loads (caused by hydrodynamic pressures onto the ship hull) as well as for internal loads (weight and inertia forces caused by the hull structure and cargo). Thus, performing design verification based on CFD loads involves several tasks:

- Generating a FEM mesh matching the structural properties of the whole hull
- Selecting a set of loading conditions to be used for structural analysis
- Adding masses to the finite element model, such as cargo and consumables related to those loading conditions
- Generating a CFD mesh matching the hull's shell
- Selecting appropriate wave situations where loads are to be based on
- Computing related pressures at the hull surface by means of CFD simulation
- Computing the external FEM loads by mapping the pressure loads from the CFD mesh to the FEM mesh
- Balancing the external FEM loads by appropriate internal loads
- Performing the structural analysis based on those FEM loads

Performing most of the above subtasks is fairly state of the art. CFD and FEM codes are in use for a long time. And pre-processors which assist in setting up the related meshes are commercially available.

In contrast, modelling cargo loads in an efficient manner is rather specific to ship design and not addressed by standard tools. There are some FEM codes which provide ready-to-use solutions for applying CFD pressure loads. However, each of those FEM codes only supports some dedicated CFD codes (usually the CFD code from the same software house and not the special potential-based codes preferred for ship design). Furthermore, some solutions imply meshing constraints (such as requiring coinciding FEM and CFD meshes).

Even when software components for all of the above problems are available, performing a CFD-based structural analysis will remain a complex and time consuming task: Programs from different vendors need to be interfaced and executed in a co-ordinated manner. There might also be organizational challenges, too: Experts from the CFD and FEM departments will need to work in close co-operation.

The GL ShipLoad program has been designed to address those problems. It integrates all algorithms necessary to access and combine the data of the CFD computations and the FEM-model for generating the appropriate CFD-based FEM loads. It also provides a GUI to easily control the above load generation process in a time- and cost-saving manner.

This paper focuses on the technology for coupling CFD and FEM computations and for selecting the appropriate design waves. For details on other components, such as the integrated pre-processor for modelling cargo loads and loading conditions, we refer to [2].

One core component for linking CFD and FEM is a repository for the ship hydrodynamic results (SHR). Another important component is an algorithm which maps CFD pressures to FEM nodal loads where CFD and FEM meshes need not coincide. Both the exchange file and the mapping algorithm are independent of the CFD method. Therefore, the currently applied efficient strip method can be replaced by more advanced procedures.

2. LOADS FOR FEM MODELS

2.1 DESIGN BENDING MOMENT APPROACH

Classical rule sets which are based on a beam model simply specify some design loads and require that the ship structure will resist those design loads. E.g. with the IACS design wave bending moment acting, a section modulus is required such that permissible stress is not exceeded.

The basic idea is that if the ship is designed to resist an appropriate set of design bending moment loads, the ship will resist all loads expected during its lifetime.

This approach is simple and easily applicable. Despite its simplicity, it is very reliable because the design bending moments are based on experience gathered during a long time. The advantage of experience design loads is that they also account for unknown or unaware load effects. The disadvantage is that experience strongly holds only for designs common so far. For innovative designs, experience based design rules might be less reliable.

As the design bending moment approach is based on a beam model, it does not directly apply to 3D FEM models. Thus, generating FEM loads requires generating a nodal load distribution which causes the design bending moments. But there are many nodal load distributions which cause the same bending moment. Thus, there is no unique approach for generating the corresponding FEM loads. Additional rules for distributing the nodal loads are needed. As an example, we refer to IACS common structural rules for tankers [3] or bulkers [4] which contain such (different) rules.

Rule sets typically specify some basic design loads (such as horizontal bending moment, vertical wave and still water hogging and sagging moments). Additional design loads are usually specified in terms of a combination of the above. The manner of combination might also depend on the ship type and the rule set.

It should also be noted that physical loads result from different phenomena, such as wave pressure and accelerations. Applying nodal loads as above only ensures that the design bending moments are reached. It does not automatically imply that the design moments are generated by a physically realistic combination of pressure and acceleration loads.

2.2 PHYSICAL LOAD APPROACH

In principle, physically realistic loads can be obtained by means of a CFD simulation which takes all major hydrodynamic effects into account. Nodal loads can be computed from the resulting pressures and accelerations.

A time-dependent simulation based on the Navier-Stokes-equation with a free surface will be needed in order to capture all hydrodynamic effects. Such simulations need to be carried out for all wave situations which might occur when operating the ship. Finally, a large amount of FE-results needs to be post-processed. Although possible, this would be a very time-consuming and expensive task.

It should also be noted that computing physically realistic loads like this does not automatically result in design loads which are more reliable. The physical load approach will only account for the modelled physical effects. Thus, making the proper conclusions from the physical load simulations is still subject to experience. In contrast, the experience-based design bending moment approach even accounts for non-physical as well as for unknown physical effects.

2.3 EQUIVALENT DESIGN WAVE APPROACH

The equivalent design wave approach is a compromise between the design bending moment and the physical load approach. The approach assumes that if the ship is designed to resist the loads caused be certain design waves, the ship will resist all loads expected during operation.

The method implemented in GL ShipLoad chooses design waves among a set of harmonic waves. Each wave in the set causes a certain bending moment which is computed by means of a hydrodynamic simulation. The amplitude of the waves is then scaled such that at least one wave of the set causes the design bending moment while no wave will cause larger moments.

FEM loads are computed by combining the pressure loads from the hydrodynamic calculations with the appropriate inertia loads. As a wave of the set causes the design bending moment, the corresponding FEM load case will do so, too. Thus, the approach can also be interpreted as a special method of generating FEM loads consistent with the design moments. Therefore, it inherits the advantage of being backed up by experience. In addition to the classical design bending moment approach, it automatically ensures that loads correspond to a physically possible situation. Finally, the design wave loads will be consistent with the classification society's rules (which are based on the same design moments).

In contrast to the physical load approach, the magnitude of the load depends on the experience-based design moments and not on the physical reliability of the CFD method. Moderate errors caused by the CFD method are negligible because the finally applied loads are calibrated by means of the design bending moments, anyway. Thus, it is sufficient to use a simple but fast (e.g. strip) method instead on an expensive free surface Navier-Stokes-solver.

Rule sets based on the design bending moment approach usually rely on explicitly specified combinations of design loads in order to account for coinciding load situations. In contrast, the design wave approach as supported by GL ShipLoad does not specify the related design wave a-priory. Instead, it analyses the hydrodynamic results in order to find waves which cause critical loads.

3. CONTROLING LOAD CALCULATION

3.1 MODELING

The load calculation control process in GL ShipLoad is designed to rely primarily on the FE model because this is available anyway. The model will be set up by the user's favourite FEM pre-processor.

In addition to the FEM-model the process requires a CFD model. For the currently integrated strip method, the model can be set up directly by means of the user's favourite tool. In addition, GL ShipLoad contains a module which can compute a strip model from the FE model. Thus, the user can perform design wave calculations without the need of setting up a CFD model explicitly.

The third class of input data consists of mass distributions which are needed by GL ShipLoad in order to properly account for weights and inertia loads. GL ShipLoad provides a sophisticated pre-processor for this task because there are no standard tools available. This pre-processor allows the user to define different type of basic mass items on a high level (e.g. tanks, container bays) as well as on a low level (e.g. FEM loads on nodes located within a certain area). Finally, the user can define several loading conditions in terms of those mass items.

The first step is to set up and import the FE model because every automated modelling task supported by GL ShipLoad is based on the FE geometry.

The next step is to set up the basic mass items which represent the structure, equipment, cargo, or consumables. The complete mass distribution for each loading condition is finally defined in terms of those mass items and related load factors. A hydrostatic equilibrium calculation based on these loading conditions and the FE geometry is finally performed in order to compute static loads.

3.2 SELECTING DESIGN WAVES

In contrast to modelling, which is usually performed once in the beginning of the load calculation process, selecting the appropriate design waves is an iterative process:

- A full set of waves is specified by parameters
- A hydrodynamic calculation related to these waves is performed
- The resulting pressure loads are combined with the appropriate inertia loads in order to obtain the total load acting on the ship structure.
- The total loads are analysed
- Based on this load analysis, certain design waves might be selected
- Wave parameters or selection criteria may be modified and the calculation process is repeated.

The user can control most of these steps himself. However, for certain standard tasks (such as calibrating the wave amplitudes and selecting the design waves according to GL's procedure) automatic control modules are also provided.

The internal design of GL ShipLoad aims to decouple the hydrodynamic calculation from the load selection control as much as possible. The CFD code deposits its results in a hydrodynamic result database which is independent of the CFD method. This will allow replacing the currently built-in strip solver with other CFD codes easily.

As the equivalent design wave approach aims at being compatible with the design bending moment approach, load analysis is based on section load distributions.

An external section load distribution is computed from the pressures which are read from the result database. The internal section load distribution is computed from the mass distribution related to the loading condition. This is done by computing the six inertia force distributions resulting from unit accelerations with respect to the ships rigid body modes. An appropriate linear combination of these six force distributions is added to the external force distribution such that the total force is zero.

Plots of the section load distributions can be inspected directly by the user. Selecting critical design waves automatically is supported by a search feature. The user specifies a functional in terms of the section load distribution which should be maximised or minimised. For example, it is possible to automatically find the wave which causes the largest value of horizontal plus vertical bending moment.

3.3 GENERATING FEM LOADS

After a set of critical design waves has been selected, FEM loads related to these waves can be generated. The pressure loads related to the selected waves are read from the hydrodynamic result database. GL ShipLoad contains a sophisticated module which analyses the geometry of the CFD mesh and the FEM mesh and maps the CFD pressure to FEM nodal loads. The resulting external FEM loads are balanced against internal loads such that the total load onto the FEM model is zero.

4. THE SHIP HYDRODYNAMIC RESULT REPOSITORY (SHR)

4.1 PURPOSE AND GENERAL IDEAS

The SHR data model and file format aims at providing a common interface from CFD codes to other programs. The CFD codes will store their results in a neutral data file. Other programs for ship structural analysis and design will read the CFD results from that neutral data file.

It is restricted to data needed by ship design tools which want to apply CFD based loads. It is not intended to be a neutral hydrodynamic model file in any way (would be difficult with all those rather different hydrodynamic methods in use).

The focus is on storing the results of the hydrodynamic calculations. These are usually pressures, but other result types – such as fluid potentials – are also possible.

The file also stores some basic information about the CFD model geometry in a CFD-method-independent manner. This is restricted to data which is needed by structural analysis codes in order to apply pressure-induced loads. It is generally not possible to reproduce the CFD model geometry from the information present in the SHR file.

Pressure results are stored only for locations where the fluid domain boundary faces the ship structure. Results related to other locations are not needed for computing structural loads (and some CFD methods don't compute them, anyway).

In addition to the results of CFD computations the file formats supports storing of the load attributes (such as wave parameters) related to the results. Thus, the load parameters can be accessed by subsequent programs in order to select certain results, to perform evaluations which depend on certain load parameters explicitly or to interpolate between results.

The data model is based on the following principles:
- principal results are pressure fields acting on a two-dimensional surface
- pressure results are present for several load cases
- load attributes (e.g. wave parameters) can be stored separately.
- For each stored load case result, a reference to related load attributes is maintained.

4.2 GEOMETRY

Pressure results are stored for a discrete set of pressure field points located at the hull's shell. Only some very basic geometric information related to those pressure result points are maintained by two spatial attributes:
- The location $\vec{x}_i$ of each pressure result point is stored by means of its cartesian coordinates.
- The direction and size of the pressure area which is discretized by point i is stored by means of an area vector or $\vec{a}_i$.

This basic information is necessary and sufficient for
- locating close finite elements and nodes,
- computing pressure-induced forces $\vec{f}_i = p_i \vec{a}_i$.

No particular semantics with respect to the CFD model is implied by this. Depending on the CFD method, the values might be obtained from different model items, e.g.:
- Panel method: location = panel's centre of gravity, area vector = normal vector of panel.
- Strip method: location = collocation point of cross section curve, area vector parallel to cross section plane, length determined from distance to neighbour strips and neighbour collocation points.
- Finite volume method: area vector = normal vector of those cell faces forming the fluid domain boundary, location = centre of gravity of these faces.

The computation of pressure-induced forces as well as search strategies for locating close finite element items does not depend on the original semantics of these spatial attributes. Thus, the results stored in an SHR file can be used independently from the CFD program which has written the file.

Although the above spatial attributes are sufficient for processing the pressure results we might be interested in more specific information about the area which is discretized by a pressure result point. With most CFD models, such areas are (or can be approximated by) polygons. As an optional third spatial attribute, the SHR data model supports storing the vertices of these polygons.

For panel or finite volume methods, which already define the model in terms of vertices, these can be directly taken from the fluid mesh. For strip methods, a reasonable choice might to compute four vertices by shifting $\vec{x}_i$ half the way towards neighbour nodes and neighbour strips.

Again, further computations do not depend on the original semantics of the polygon vertices. Programs processing the SHR results remain independent of the CFD method.

4.3 LOAD PARAMETERS

Load results will relate to certain load parameters. There are several classes of load parameters, such as

- wave parameters (e.g. wave vector, amplitude)
- environmental parameters (e.g. water density, depth)
- operational parameters (e.g. speed, centre of gravity, mass)
- …

Programs accessing the CFD results will frequently also need access to the load parameters related to the results. Therefore, the SHR data model also supports storing load attributes.

Unfortunately, the load attributes which characterize a certain load might depend on the hydrodynamic method. For example, loads related to time step from a time domain simulation will be characterised by a \`time´-attribute. For a CFD program operating in frequency domain, a \`time´-attribute won’t make sense but a \`frequency´-attribute will be appropriate.

For this reason, the SHR data model keeps load attribute data and CFD result data as separated as possible. Applications might access load attributes without reading CFD results. Or they might access the CFD results without reading the load attributes. The relationship between the results and the corresponding load attributes is also stored separately. This allows adding new types of load attributes at will without breaking existing applications. Nevertheless, any application interested in the new load attribute type will be able to access it.

4.4 PRESSURE RESULTS

A pressure results relates to a discretized pressure field and a load case. Thus, a logical array is used for storing such result. The first dimension extends along the spatial attributes axis, the second along a load case axis.

In addition to the pressure results proper, a load case identifier is stored along the load case axis whenever a pressure result is written. The same load case is also used to mark load attribute data related to the just written results. This will allow reading applications to match CFD results and related load attributes by means of the load case identifier.

It might sometimes be useful to store different result components which related to the same pressure field and load data. E.g. it is sometimes usefully to hydrodynamic and distinctly from the quasi-static pressure. This is supported by means of a third dimension related to the result component. The physical semantics of the values related to each result component might be remembered by storing some result type identifiers as component attributes.

Some CFD methods yield complex-valued results. Internally, the real and imaginary part might are treated as a fourth dimension.

4.5 IMPLEMENTATION BY MEANS OF HDF5

HDF5 [5] is a storage scheme which was designed and implemented by NCSA for storing large amounts of scientific numeric data. It organizes the data like a file system:

- There are datasets (which correspond to files).
- The datasets are structured like multi-dimensional arrays.
- The data is stored compactly in a binary manner.
- The data storage is portable. It can be written and read using different operating systems and hardware architectures.
- There are HDF5 groups (corresponding to directories) which contain datasets or other HDF5 groups. This allows organizing the data in a hierarchical manner.
- The open source license does not restrict commercial use.

Due to these features, the SHR repository can be implemented easily and efficiently on top of HDF5. Pressure result arrays are directly implemented as HDF5 datasets. Random access to selected load cases is supported by HDF5 methods. Table-oriented data can be easily represented by a HDF5 group containing datasets corresponding to the table columns.

5. MAPPING PRESSURES TO FEM-LOADS

5.1 GENERAL CONSIDERATIONS

In order to keep generality and flexibility, an algorithm for mapping CFD pressure results to FEM loads should neither depend on the FEM program nor on the CFD method. As the FE model and the CFD model will differ, the finite elements subject to the pressure loads will not coincide with a related CFD model item. Even worse, there will usually be gaps between both mesh surfaces.

There are some finite element programs which allow defining loads in terms of pressures onto element surfaces. However, this can be misguiding. Although intuitive at first, we already run into interpretation problems when the meshes don't match: How should pressures be interpolated then? If fluid elements and finite elements are not aligned in parallel the direction of related pressure forces will be different. Thus, forces are hardly conserved by a naïve pressure mapping approach.

We should be aware that the real problem is not to transfer pressures from the fluid mesh to the FEM mesh but to transfer the forces resulting from the CFD simulation to forces onto the structural model.
In contrast, mapping forces can be conceptually very simple. For each pressure result, we just need to select some FE nodes and distribute the pressure force onto those nodes such that the sum of the output forces equals the input force. Thus, transferring forces instead of pressures is therefore more robust and conserving the total force acting on the model can be achieved easily. And it keeps independence because it is not restricted to FEM codes supporting pressure loads.

In principle, mapping forces like this consists of the following:

- For each pressure result p_i (acting at location x_i), select some FEM nodes (located at $\vec{x}_{i,n}$)
- Compute the pressure force $\vec{f}_i = p_i \vec{a}_i$
- Distribute that pressure force among the selected nodes such that
  - force is conserved:
    $$\vec{f}_i = \sum_n \vec{f}_{i,n} \qquad (1)$$
  - moment is conserved
    $$\sum_n \vec{f}_{i,n} \times \left(\vec{x}_{i,n} - \vec{x}_i\right) = 0 \qquad (2)$$

It is generally possible to choose appropriate weights

$$\vec{f}_{i,n} = w_{i,n} \vec{f}_i \qquad (3)$$

such that the above conditions are fulfilled.

This general approach is very flexible about the selection of FEM nodes where the mapped forces shall be acting. As long as each pressure result is distributed onto the nodes in a manner conserving force and moments of a local pressure result, total force and moments will be preserved.

In practice, we are also interested in the local quality of the resulting force distribution. Therefore, we need a more specific and advanced algorithm which determines appropriate target nodes and weights.

5.2 FINDING MATCHING FINITE ELEMENTS

For reasonable local mapping quality, only the FEM nodes in the neighbourhood of a pressure result point shall be subject to the related pressure forces. Thus, the first step of a concrete mapping procedure consists of an algorithm which determines an appropriate set of `neighbourhood nodes´.

For each pressure result location, the algorithm applied in GL ShipLoad first selects a unique finite element which is considered closest to the pressure result location. Then the nodes of that closest element are added to the neighbour node set.

Only those elements which constitute the hull's shell are considered in the search. The same element set has already been needed in a previous (hydrostatic) computation step. Therefore, it has already been defined by the time when the hydrodynamic results shall be mapped.

A pressure result at location $\vec{x}$ acting in direction of a surface area vector $\vec{a}$ defines a line in 3D space

$$\vec{L}(t) = \vec{x} + t\vec{a}\,. \qquad (4)$$

The elements of the shell which are intersected by that line are determined. If there are several intersected elements then the element with the smallest distance from $\vec{x}$ (with respect to direction $\vec{a}$) is selected.

In practice, the intersection test is most easily performed after projecting the element vertices and the result location into a two-dimensional plane which is perpendicular to $\vec{a}$. The intersection test is then equivalent to a point-in-polygon test (projection of $\vec{x}$ is inside the polygon which is formed by the projected element edges).

5.3 MAPPING TO SINGLE ELEMENTS

GL ShipLoad supports mapping pressure forces to the nodes of triangular or quadrilateral elements. All nodes will receive forces with the same direction (parallel to $\vec{a}$). Thus, the problem reduces to determining some nodal weights according to (3).

GL ShipLoad determines the initial nodal weights from a form function formulation related to the element. The nodes of the surface elements are projected onto the plane normal to the area vector $\vec{a}$ (`fluid panel plane´). The projected nodes x_i will define a polygon (triangle or quadrilateral). The projection of the pressure result location x will be inside the polygon because the line (4) intersects the originating element.

Computing the weights by means of form functions automatically ensures that only one node is loaded if that node coincides with the pressure result location.

5.4 HANDLING OVERLAPPING ELEMENTS

Using the node selection procedure described so far, only the nodes of a single element will receive forces related to a particular pressure result point. This usually yields reasonable force distributions if the fluid mesh is finer than the FEM mesh. This is frequently the case when applying the physical load approach where rather fine fluid meshes are common.

In contrast, for the equivalent design wave approach rather coarse fluid models are sufficient. With GL ShipLoad's built-in strip solver, adjacent strips typically span several finite elements. If pressure forces are only distributed on the nodes of the closest element, some nodes will be subject to large concentrated pressure forces. In turn, some nodes (of elements which are not intersected by any line (4)) will receive no pressure force at all. This is not critical for global strength analysis. But it might become a concern when local pressure-induced bending effects shall be investigated.

Fixing this requires a larger set of nodes subject to the pressure forces. Having found these nodes, we determine appropriate nodal weights (3) for the enlarged node set.

Determining above nodes requires a more specific geometric description of the area which relates to each fluid result location. That information is also read from the SHR file which needs to contain the result areas vertices.

GL ShipLoad determines the node set by identifying those finite elements which overlap with the fluid result region. Then the nodes of these elements are selected. The overlapping elements are determined recursively. It starts with an initial element which is determined as described in 5.2. Its adjacent elements are considered. Adjacent overlapping elements are added to the element list. Likewise, the adjacent neighbours of the newly added elements are processed recursively.

The above algorithm depends on a test which determines whether a finite element and the fluid result area do overlap. The term ‘overlap’ does not directly make sense in a three-dimensional context with non-matching grids because fluid result area and finite elements might be separated by a certain distance. Therefore, a specific overlap test (and other computations) will be performed after projecting the fluid result area and the finite elements into the single reference plane.

After the nodes are selected, the weights (3) need to be determined. For each fluid result the weights will be computed and accumulated element-wise:

Loop over all fluid result points (at location $\vec{x}_i$, with surface area vector $\vec{a}_i$):

- compute the projected fluid result area polygon P_i
- Determine the overlapping finite elements e_j and their nodes $v_{i,n}$
- Reset all nodal weights $w_{i,n} := 0$
- Loop over all overlapping elements e_j :
    - Compute the projected element polygon E_{ij}
    - Compute the intersection polygon $P_i \cap E_{ij}$
    - Compute the intersection polygon’s centre of gravity $\vec{x}_{ij}$
    - Compute local mapping weights w_k for the nodes of the current overlapping element e_j by means of the same method as in 5.3, but using the intersections centre of gravity $\vec{x}_{ij}$ instead of $\vec{x}_i$ as pressure result location.
    - Compute the area a_{ij} of the intersection polygon
    - Update the nodal weights $w_{i,n(k)} := w_{i,n(k)} + (a_{ij} / |\vec{a}_i|) w_k$

5.5 FIXING CONSERVATION PROPERTIES

The above algorithm will usually result in a nodal force distribution which visually matches the originating pressure force distribution fairly well. However, total force and moment on the FE-model will usually slightly deviate from the originating CFD result. There are several reasons:

- The form function method applied to single elements does not automatically preserve the moments as in (2)
- Fluid mesh and FEM mesh don’t match exactly. At locations close to model edges of the FE model, there might be some elements which are only partially covered by fluid elements or some fluid areas which are only partially covered by finite elements. At those locations, some fraction of the force might be lost and some spurious moments will be generated.
- There might be some locations where projection of the finite elements results in de-generated polygons which might cause some computations to fail.

Unless there are serious model mismatches, the deviations will be small and acceptable for applications like the equivalent design wave approach. However, comparing total FEM with CFD loads is a good error check only if the mapping algorithm preserves the total force. Some other applications, like time domain simulations, might even be sensitive to spurious forces and moments induced by non-conservative force mapping.

As a pragmatic measure, the mapping weights $w_{i,n}$ are finally modified by adding an appropriate linear combination of $w_{i,n}$, $\xi_{i,n} w_{i,n}$ and $\zeta_{i,n} w_{i,n}$ such that the conservation equations (1) and (2) are fulfilled ($\xi_{i,n}$ and $\xi_{i,n}$ are the local coordinates of the nodes in the projection plane).

6. CONCLUSIONS

Applying the previously described technologies, ship designers are able to perform structural FEM-analysis with CFD-based loads, routinely. The additional overhead related to fluid-structure coupling is negligible.

The major computational resources are consumed by the CFD simulation. Only moderate resources are needed when applying the equivalent design wave approach with a strip method.

Most human working time is needed for modelling the mass distributions related to the loading conditions. This task is already supported by GL ShipLoad in an efficient manner.

The neutral SHR file and the program library which maps pressures to nodal forces has also been successfully used outside the context of GL ShipLoad – in particular for generating FEM loads from finite volume CFD simulations. This confirms that independence of the CFD methods is achieved in practice.

7. ACKNOWLEDGEMENTS

Some ideas which are applied in the pressure-to-nodal-force mapping have been inspired by the joint European research project ESPRIT 20161 `CISPAR´ (Open Interface for Coupling of Industrial Simulation codes on Parallel Systems) which was funded by the European Commission.

An initial design and implementation of the SHR data model was carried out while participating in the joint German research programme `WIPS´ (Wettbewerbsvorteile durch Informationstechnisch unterstützte Produktsimulation im Schiffbau – Competitive Ship Technology through IT-based Product Simulation) which was funded by the German Federal Ministry of Education and Research (BMBF).

8. REFERENCES

1. Germanischer Lloyd AG, 'GL-Guideline V, Part 1, Chapter 2, Section 2 Global Strength Analysis (Edition 2007)', *Germanischer Lloyd AG, Hamburg, in `Rules and Guidelines 2007´ (publishing in progress)*, 2007

2. Christian Cabos, Henner Eisen, Matthias Krömer, 'GL.ShipLoad: An Integrated Load Generation Tool for FE-Analysis', *Proceedings of the COMPIT 2006, Delft*, 2006

3. IACS members, 'Common Structural Rules for Double Hull Oil Tankers, Edition January 2006', *Published by Germanischer Lloyd AG, Hamburg, in `Rules and Guidelines 2007´*, 2007

4. IACS members, 'Common Structural Rules for Bulk Carriers, Edition January 2006', *Published by Germanischer Lloyd AG, Hamburg, in `Rules and Guidelines 2007´*, 2007

5. The HDF Group, 'HDF5 Home Page', http://hdf.ncsa.uiuc.edu/products/hdf5/index.html, August 15th 2007

9. AUTHORS' BIOGRAPHIES

Henner Eisen studied mathematics and physics and is working as a senior project engineer in the CAE development department of Germanischer Lloyd. He is performing research and development in data modelling and computational methods, in particular related to fluid-structure-interaction and structural fatigue simulation.

Christian Cabos studied mathematics and physics and did his PhD on methods for computing ship vibration. He works for Germanischer Lloyd since 1989 and is responsible for the department CAE development since 2002. He is performing research in data modelling and computational methods in maritime engineering.