Computational Analysis Tools

Julio García-Espinosa

La Ciotat/ 24-26th May 2019
<table>
<thead>
<tr>
<th>ITEM</th>
<th>LEADER</th>
<th>FORESEEN PERIOD</th>
<th>STATUS</th>
<th>PROGRESS (%) up to M24</th>
</tr>
</thead>
<tbody>
<tr>
<td>WP3-DEVELOPMENT AND VALIDATION OF NUMERICAL AND CAE TOOLS</td>
<td>COMPASS</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.1: Development and tuning of a coupled Seakeeping-FEA analysis tool</td>
<td>COMPASS</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.2: Implementation of the Inverse Finite Element Model Updating method</td>
<td>CIMNE</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.3: Development of thermo-mechanical behaviour analysis and collapse assessment tools for laminated composite structures</td>
<td>COMPASS</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.4: Validation and benchmarking for the software developments and applications</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Delayed</td>
<td>60%</td>
</tr>
<tr>
<td>Task 3.5: Demonstration of the developments and implementations</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.6: Graphical user interface integration of the developments and implementations</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Validation and testing</td>
<td>90%</td>
</tr>
<tr>
<td>Task 3.7: Simulation tool version release, documentation and training</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Validation and testing</td>
<td>90%</td>
</tr>
</tbody>
</table>

Register for free at https://www.scipedia.com to download the version without the watermark
Constitutive model for FRP materials

Solution: Implementation of a new constitutive model for FRP materials (basic component)

Objectives: Integrate (within a FEM GUI) an advanced constitutive model for FRP materials based on the Serial/Parallel mixing theory (SP-RoM) & isotropic Kachanov-type damage (including model implementation and validation) & thermo-mechanical model & fatigue assessment model.

Components: GiD, Tdyn-Ramseries, SP-RoM model (new development).

Other characteristics: Usability (easy definition, local axes management, new groups manage. tools, ...).

Register for free at https://www.scipedia.com to download the version without the watermark
Constitutive model for FRP materials

Definition of FRP laminates

Register for free at https://www.scipedia.com to download the version without the watermark
Constitutive model for FRP materials

Definition of structural FEM model

Register for free at https://www.scipedia.com to download the version without the watermark
Wöhler (S-N) experimental curves are defined for parallel and serial directions (matrix and fiber).

Rainflow counting-type algorithm (Miner’s Rule) is used to estimate the damage per ply.

Damage estimate of the composite is evaluated based on the per-ply value.
Solution: Fire simulation & collapse assessment tool

Objectives: Coupled computational analysis solution for fire and smoke propagation (fire dynamics and fire propagation) and collapse assessment of FRP composite structures (thermo-mechanical + pyrolysis + isotropic damage FEM).

Components: CFAST, FDS, GiD, Tdyn-Ramseries / Abaqus, S/P thermomechanical model + isotropic damage (new), 1D-2D pyrolysis model (new).

Other characteristics: Usability (included integrated GUI), Import/Export tools, Practicability (Fire propagation vs Fire dynamics).

Register for free at https://www.scipedia.com to download the version without the watermark.
Fire simulation & collapse assessment tool

FDS GUI (based on GiD-Ramseries)

• New GUI tools allow to define the main namelist groups of FDS (obstacles, vents, reactions, fire events ...) directly in GiD-RamSeries and to run FDS solver.
• Geometrical information is shared with the structural model and can be used for the definition of the FDS model.
• Importation tools (STEP and IGES, including XML’s FORAN data)

CFAST GUI (based on GiD-Ramseries)

• New GUI tools also allow to define compartments, vents, fire events and targets directly in RamSeries and to run the CFAST solver.
• Geometrical information from the structural model can be used for the definition of the CFAST model if necessary.
• Importation tools (STEP and IGES, including XML’s FORAN data)
**Fire simulation & collapse assessment tool**

**FDS / CFAST GUI**
- FDS: Temperature maps over structural components (beams, decks and bulkheads) are calculated.
- CFAST: Two-zones temperature evolution is calculated.
- FDS/CFAST: Furthermore, time evolution of (adiabatic) temperature in a distributed network of control points.

**Some Characteristics**

**Fire Dynamics**
- Transfer temperature (heat flux) information from control points to the structural solver.
- The structural solver includes a pyrolysis model for composites (1D -through thickness-model for shell elements and a 2D model for beam elements), which calculated temperature distribution (per layer).

**Thermo-mechanical analysis**
- Displacements, strains and stresses are calculated on structural components using a thermo-mechanical composites constitutive model.
- An isotropic damage model is used to assess the collapse risk of the structure.

**SOME CHARACTERISTICS**

Register for free at https://www.scipedia.com to download the version without the watermark.
Fire simulation & collapse assessment tool

Example of application: Fire-resisting division test

Fire Dynamics

Pyrolisis model

The heat fluxes/adiabatic temperature are calculated in a set of control points on the bulkhead/Deck.

An 1D/2D model calculates the temperature evolution through the panel/stiffener thickness and the pyrolysis of the polymer matrix.

\[
\rho C_p \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) - \rho_e C_{p,e} V_e \cdot \nabla T + \dot{m}_{\text{loss}} \left( Q_p + h_L \right)
\]

\[
\rho C_T \frac{\partial T}{\partial t} = \nabla \cdot (k_T \nabla T) - \rho_C v_C V_T + \dot{m}_{\text{loss}} \left( Q_T + h_L - h_G \right)
\]

Characterization of materials properties is based on experimental tests (carried out by VTT)

Thermo-mechanical analysis

The CAE structural model of the fire-resisting division is defined (integrated GUI).

Validation will be based on experimental tests (to be carried out by VTT and RINA)

Register for free at https://www.scipedia.com to download the version without the watermark
Solution: Full 3D hydroelastic (& non-linear) solver.

Objectives: To implement a coupled time-domain radiation/diffraction seakeeping analysis solver (2\textsuperscript{nd} order) and dynamic FEM structural solver based on the SP-RoM constitutive model.

Components: GiD, Tdyn-Ramseries, Tdyn-SeaFEM, SP-RoM model (new), monolithic algorithm (new).

Other characteristics: Integrated GUI, Monolithic coupling, Import/Export tools.
Full 3D hydroelastic solver

FEM structural solver data
• Linear / Non-linear dynamic solver
• SP RoM constitutive model

Hydroelastic solver
• Monolithic coupling

FEM seakeeping solver data
• Time domain
• 1\textsuperscript{st} and 2\textsuperscript{nd} order

Importation tools
STEP and IGES, including XML’s FORAN data

Validation
Validation to be done with the data gathered onboard Zim Luanda container ship (carried out by TSI)
Solution:
• Hull girder model (basic component for fatigue assessment and health structural monitoring tools) + 1D to 3D FEM interface.

Objectives:
• To implement a time-domain coupled hull girder – seakeeping analysis tool (linear/non-linear – 1st order/2nd order) and a 1D to 3D FEM interface.

Components: GiD, Tdyn-SeaFEM, Tdyn-Ramseries, fatigue damage model (new), hydro-elastic model.

Other characteristics: Hull girder to 3D FEM model interface, Reduced computational cost, Usability (new GUI), Practicality.
Hull girder model

Definition of FRP laminates

- Automatic generation of Sea States
  - Voyage Simulation
- Automatic generation of Combined Load Cases
  - Ship-Beam Dynamic Analysis
- Detailed 3D FEM model
  - Distributed loads
  - Composite properties
  - Fatigue S-N curves
- 3D Hydro-elastic analysis
- Strength assessment

Hull girder model definition

- 1D to 3D interface
  - Transfer of (time-domain) seakeeping wave loads
  - Hull girder stresses / displacements are imposed at the boundaries of the 3D section under analysis
  - 3D FEM model offers local displacements and stresses using SP-RoM model

Coupled hull girder – seakeeping analysis

La Ciotat / 24-26th May 2019
Fatigue damage is estimated using a rainflow counting-type algorithm (Miner’s Rule) based on the SP RoM model.
Inverse Finite Element Updating Method

La Ciotat / 24-26th May 2019

Solution:

- Inverse Finite Element Updating Method (basic component).
- Structural health monitoring solution.
- Non-destructive testing tools.

Objectives:

- To implement an Inverse Finite Element (iFEM) Updating Method based on the SP-RoM + explicit Kachanov-type damage model.
- To develop a Structural Health Monitoring system for large-length FRP-based ships.
- To develop a non-destructive testing (inspection) for structural elements.

Components: RMOP optimization platform, GiD, Tdyn-Ramseries, SP-RoM model (new), monitoring system (new).
Problem statement

- Damage zones: FEM model topologically / heuristically divided into zones
- Design variables: Average damage on the different zones
- Objective functions: weighted differences between measured and calculated frequencies:
  \[ f_{\text{obj}} = w_i \cdot (f_{\text{ref},i} - f_i) \]
- Strategy: Minimization of the objective functions by means of Newton-based methods
- FEM solver: SP-RoM + explicit Kachanov-type damage model
- The iFEM model can be also applied to estimate damage maps based on different objective functions (such as displacements in control points)
Example of application: Quality assessment of a composite panel

Modal testing

The compartment containing with the k.

The natural frequencies of the structure are measured.

iFEM CAE model

The iFEM model is created: FEM panel model + reference modal frequencies + a set of control points, defining the area of interest.

iFEM analysis / Quality assessment

The model offers as a result an estimation of the damaged area of the panel.
Inverse Finite Element Updating Method

Structural Health Monitoring (SHM)
- Consist on:
  - A modal monitoring (testing) system
  - A model of the structure (global hull girder and local detailed models)
  - A processing unit (iFEM model)
- The collected data is used to feed the iFEM model, which estimates the damage map on the structure for the local detailed models
- The system is conceived to support decision making on maintenance plans

Non-destructive testing tools
- Consist on:
  - A portable modal monitoring (testing) system, including impact hammer.
  - A local model of the structural element (i.e. bulkhead)
  - A processing unit (iFEM model)
- The collected data is used to feed the iFEM model, which identifies possible defects in the structural element
- The tool is conceived for quality control and inspection on structural elements

La Ciotat / 24-26th May 2019
<table>
<thead>
<tr>
<th>ITEM</th>
<th>LEADER</th>
<th>FORESEEN PERIOD</th>
<th>STATUS</th>
<th>PROGRESS (%) up to M24</th>
</tr>
</thead>
<tbody>
<tr>
<td>WP3-DEVELOPMENT AND VALIDATION OF NUMERICAL AND CAE TOOLS</td>
<td>COMPASS</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.1: Development and tuning of a coupled Seakeeping-FEA analysis tool</td>
<td>COMPASS</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.2: Implementation of the Inverse Finite Element Model Updating method</td>
<td>CIMNE</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.3: Development of thermo-mechanical behaviour analysis and collapse assessment tools for laminated composite structures</td>
<td>COMPASS</td>
<td>M1-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.4: Validation and benchmarking for the software developments and applications</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Delayed</td>
<td>60%</td>
</tr>
<tr>
<td>Task 3.5: Demonstration of the developments and implementations</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Finished</td>
<td>100%</td>
</tr>
<tr>
<td>Task 3.6: Graphical user interface integration of the developments and implementations</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Validation and testing</td>
<td>90%</td>
</tr>
<tr>
<td>Task 3.7: Simulation tool version release, documentation and training</td>
<td>COMPASS</td>
<td>M13-M24</td>
<td>Validation and testing</td>
<td>90%</td>
</tr>
</tbody>
</table>

- Beta version of the different computational solutions already available.
- First training course scheduled.
- Final versions (validated tools) to be delivered by September 2019.
- Validation based on a three-tier approach: small, medium and large scale experiments.
- Demonstration on the three targeted vessels by January 2020.