This project has recived funding from European Union's Horizon 2020 research and innovation programme under grant agreement Nº 723360

Computational Analysis Tools

Julio García-Espinosa

La Ciotat/ 24-26th May 2019

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

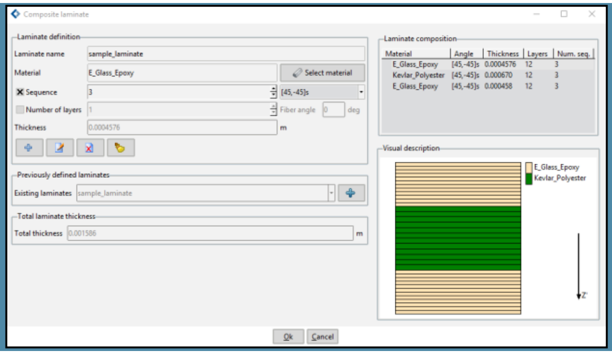
Solution: Implementation of a new constitutive model for FRP materials (basic component)

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).

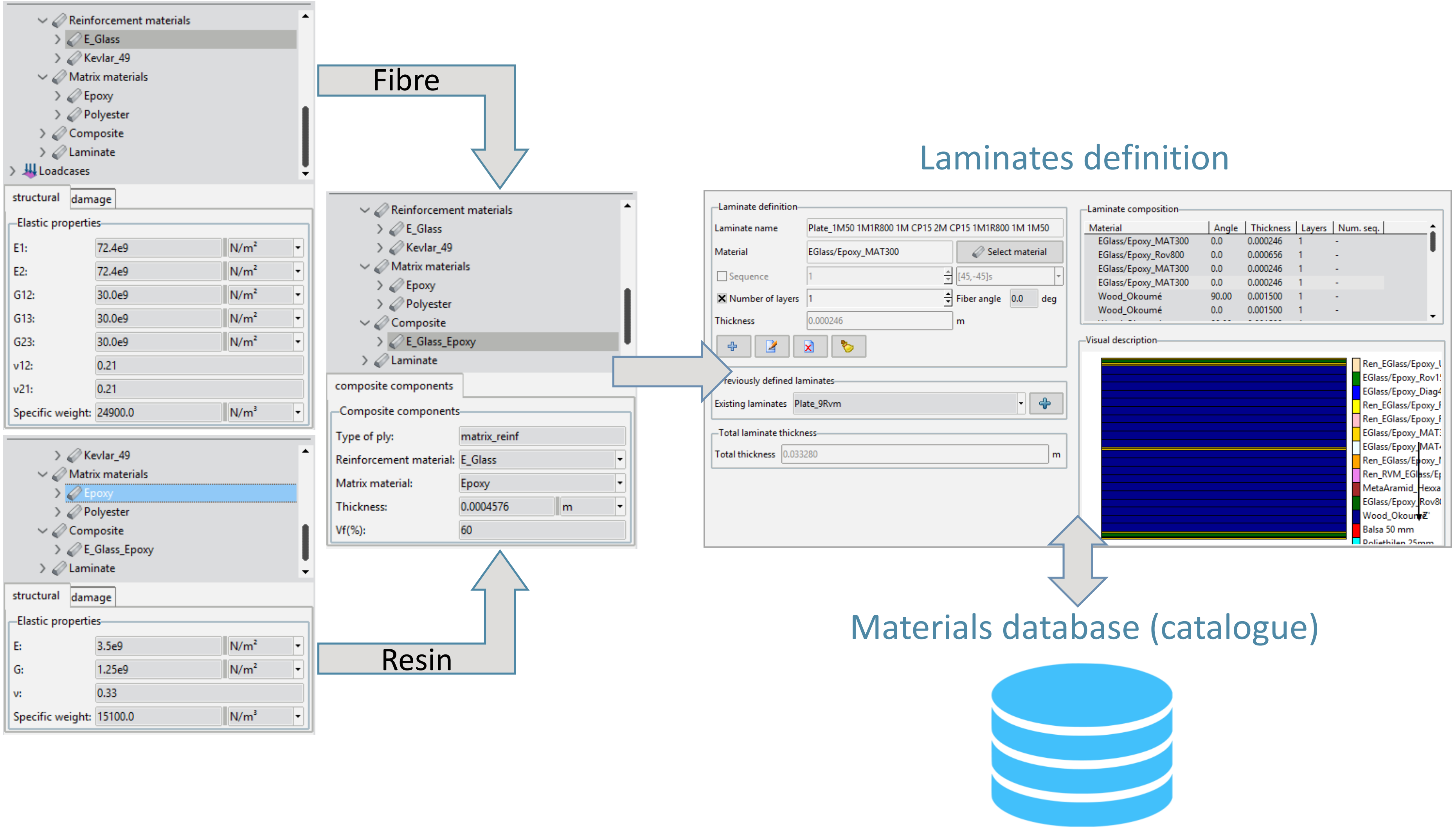
Other characteristics: Usability (easy definition, local axes management, new groups manag. tools, ...).

Constitutive model for FRP materials



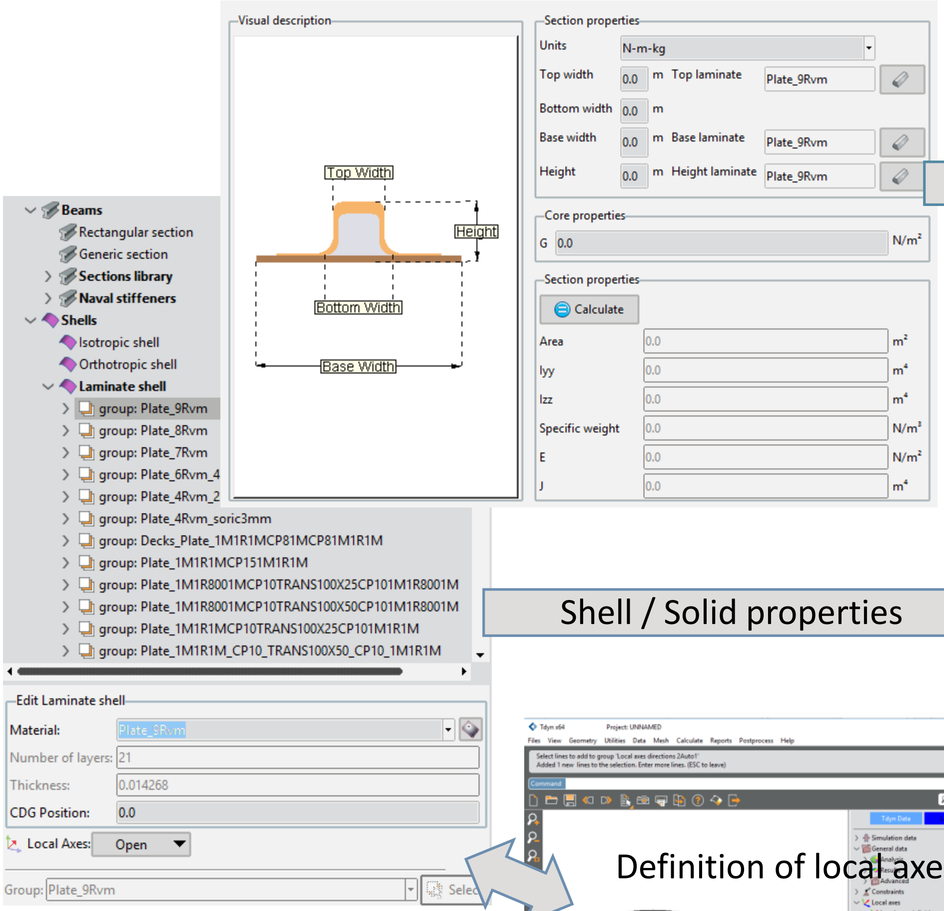
Constitutive model for FRP materials

Definition of FRP laminates



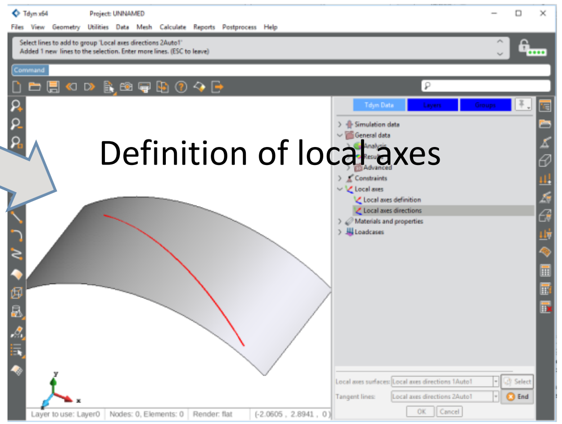
Constitutive model for FRP materials

Definition of structural FEM model

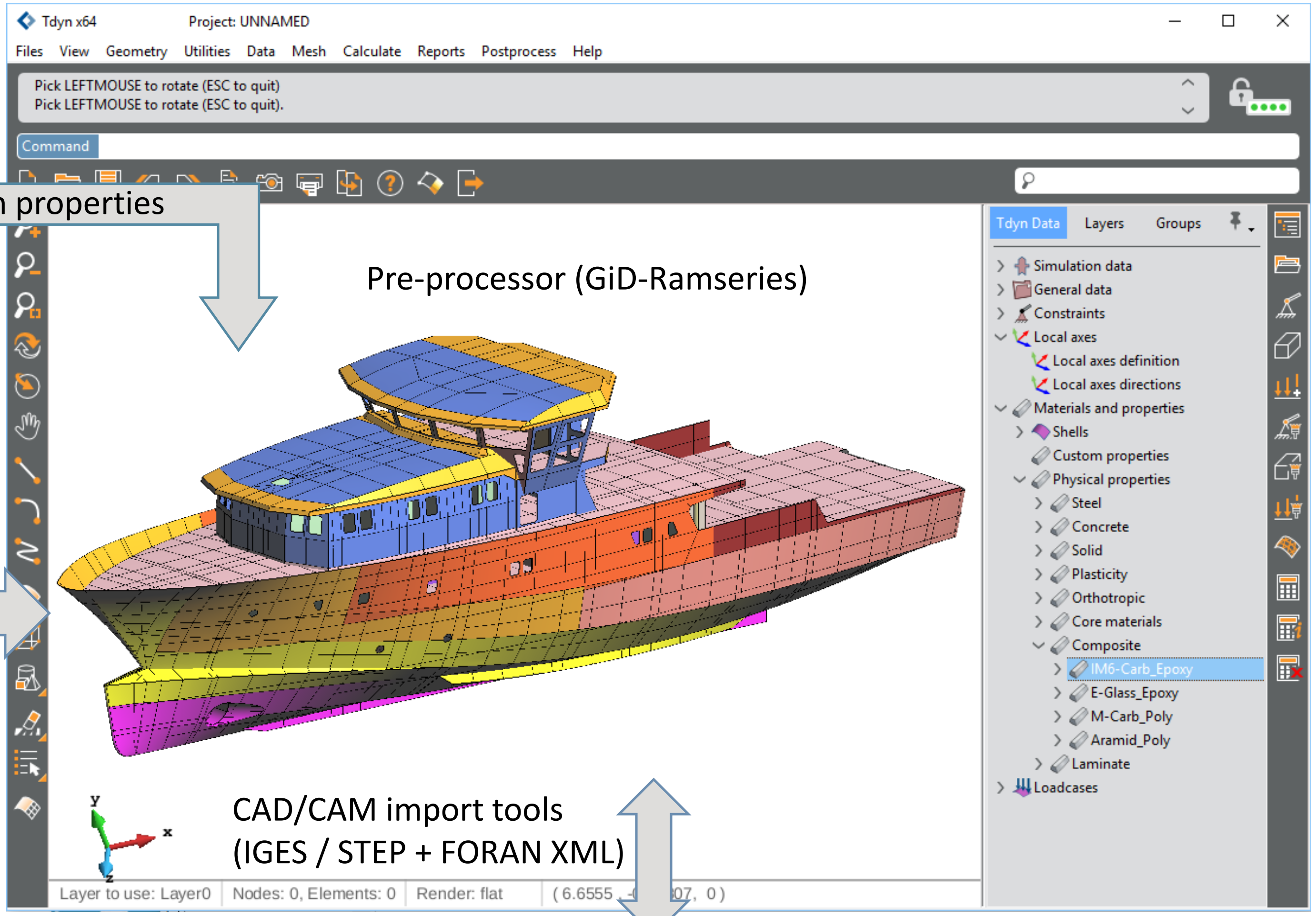


Beam properties

Shell / Solid properties



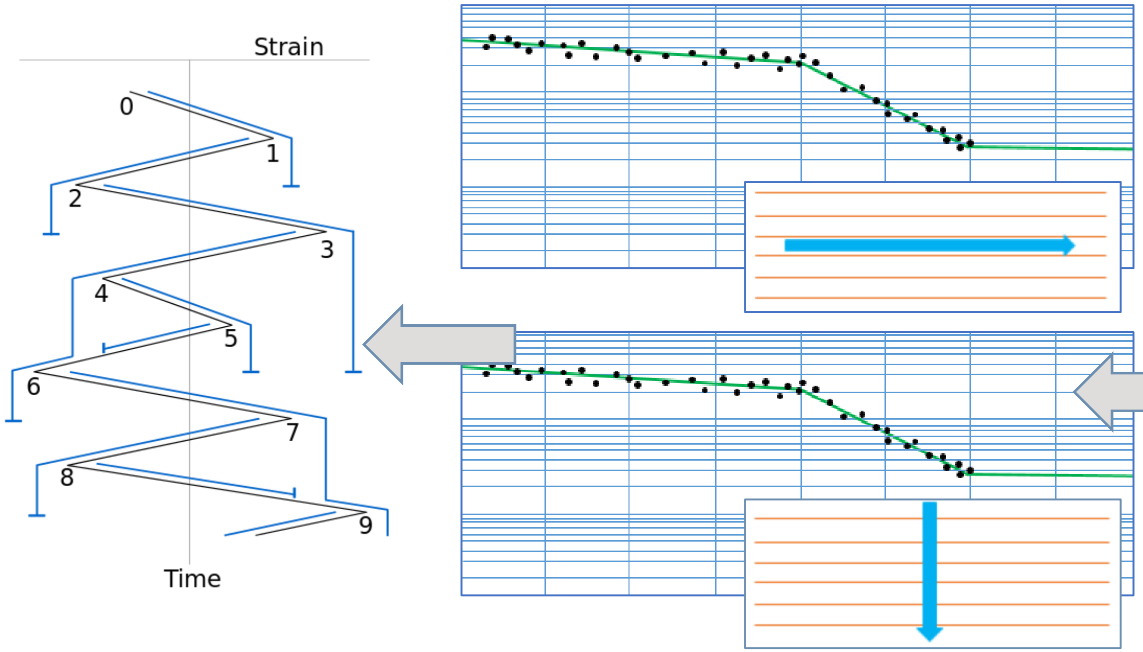
Definition of local axes

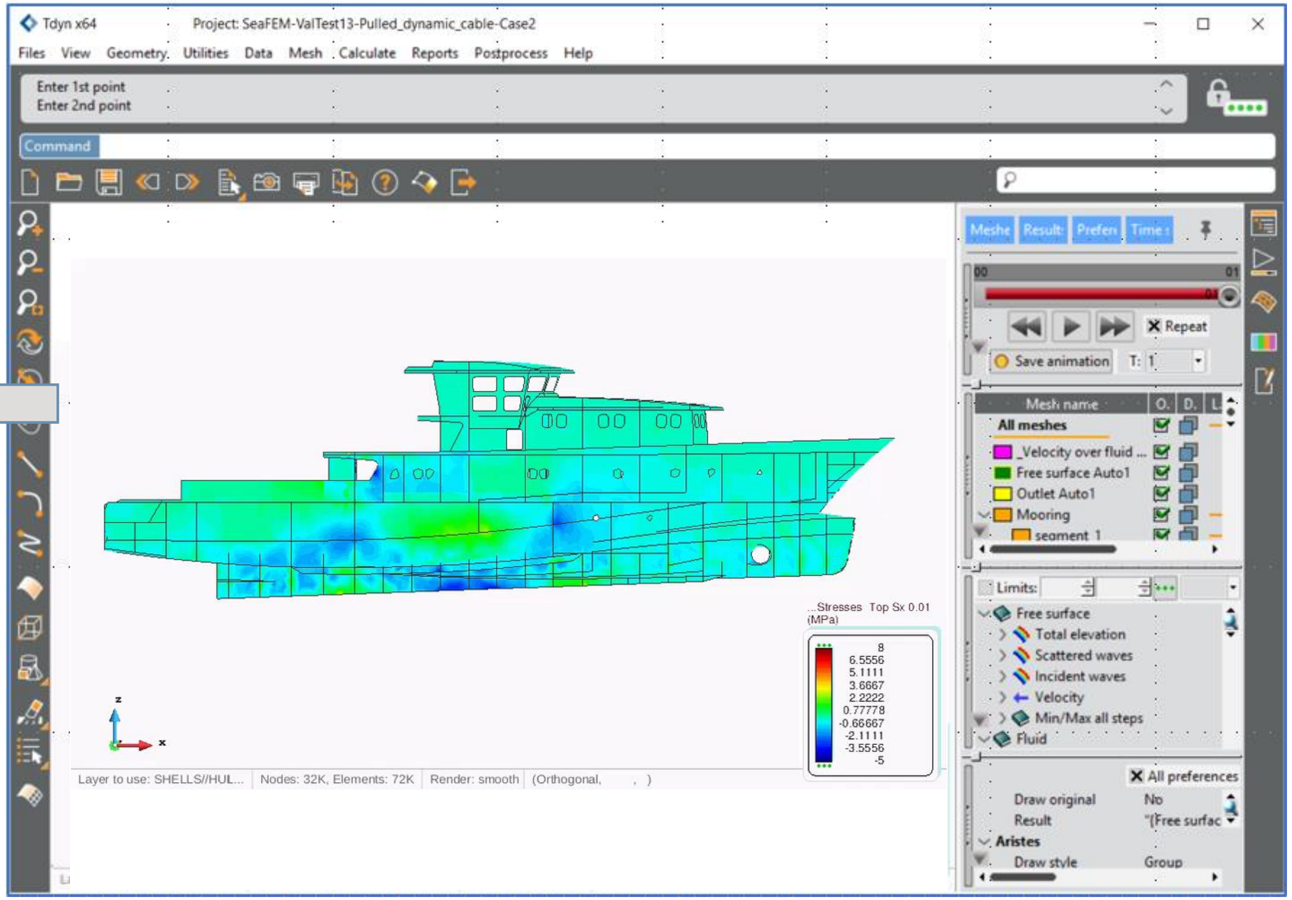


Pre-processor (GiD-Ramseries)

CAD/CAM import tools
(IGES / STEP + FORAN XML)

Fatigue assessment tool





- Wöhler (S-N) experimental curves are defined for parallel an 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.

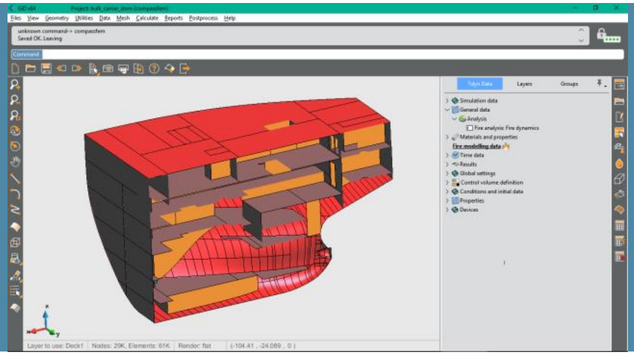
Solution: Fire simulation & collapse assessment tool

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).

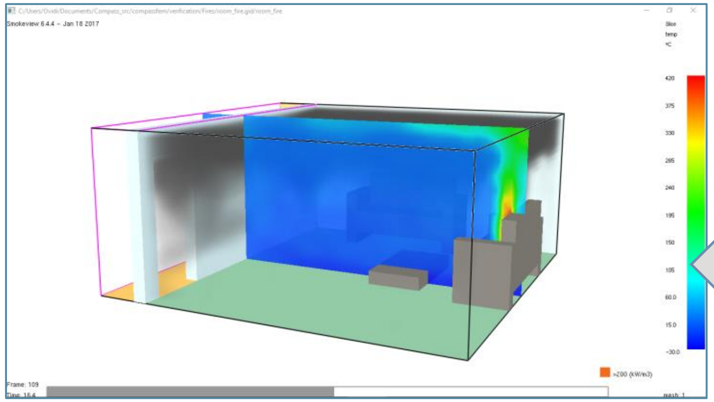
Other characteristics: Usability (included integrated GUI), Import/Export tools, Practicality (Fire propagation vs Fire dynamics).

Fire/smoke propagation & collapse assessment tool



Fire simulation & collapse assessment tool

FDS GUI (based on GiD-Ramseries)



SOME CHARACTERISTICS

- 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)

Fire specification, ship compartments configuration, wall thermal properties...

Fire dynamics analysis (FDS) | Fire propagation analysis (CFAST)

Temperature time evolution curves

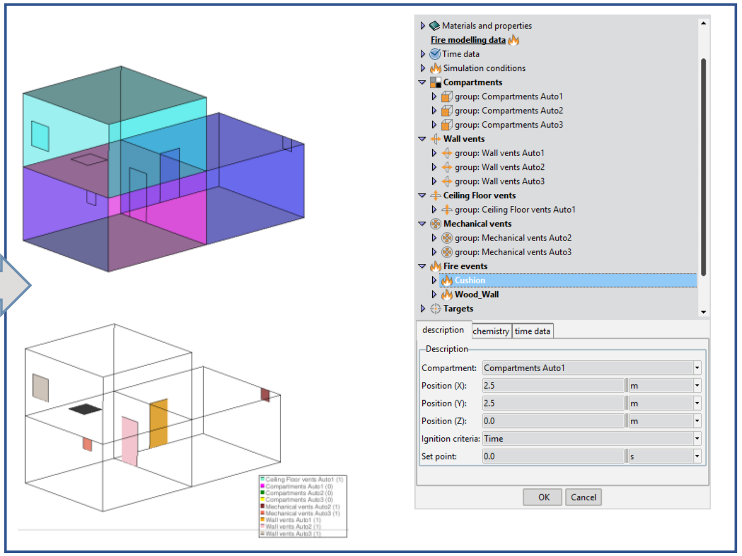
Beams and shells thermal analysis (RamSeries)

Pyrolysis model (through thickness temperature)

Beams and shells structural analysis (RamSeries)

Collapse risk assessment

CFAST GUI (based on GiD-Ramseries)

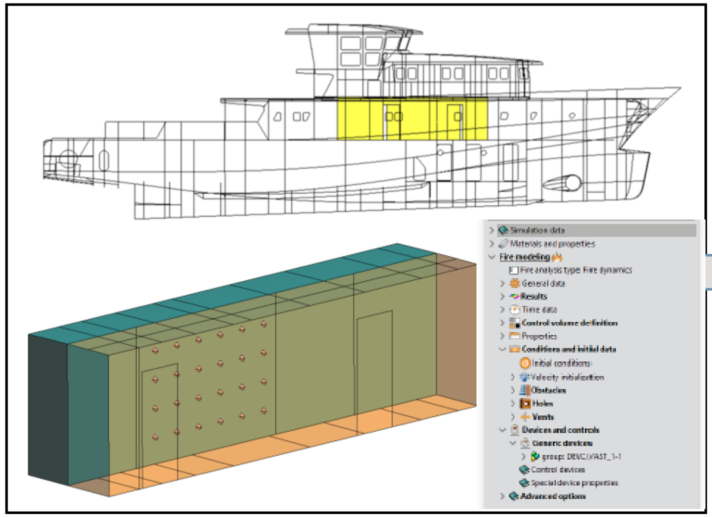


SOME CHARACTERISTICS

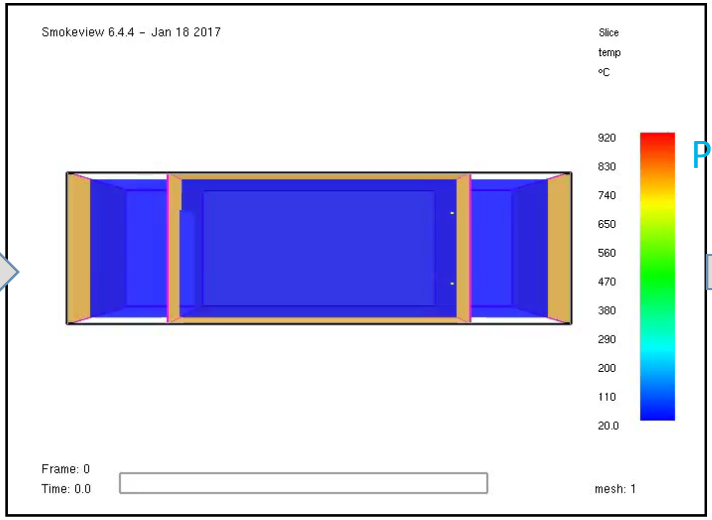
- New GUI tools also allow to define compartments, vents, fire events and targets directly in RamSeries and to run the CFAST solver.
- Geometrical information from the structural model can be used for the definition of the CFAST model if necessary..
- Importation tools (STEP and IGES, including XML's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

Fire simulation & collapse assessment tool

FDS / CFAST GUI

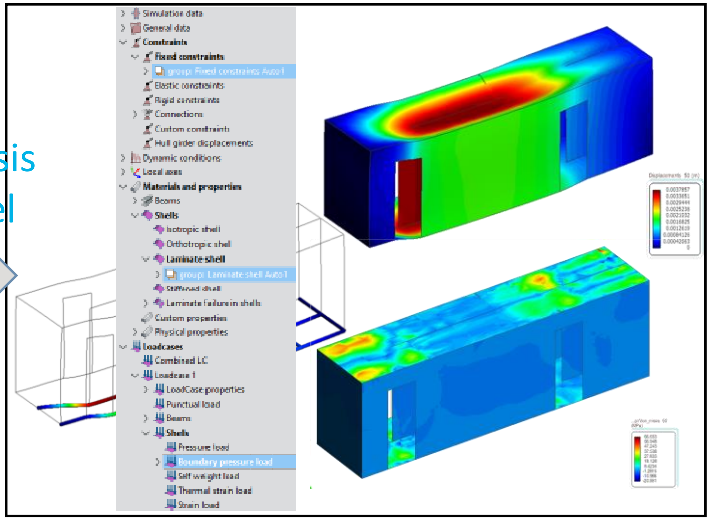


Fire Dynamics



Pyrolysis model

Thermo-mechanical analysis



SOME CHARACTERISTICS

- 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

- 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).

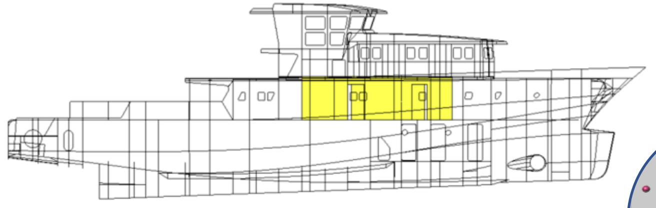
SOME CHARACTERISTICS

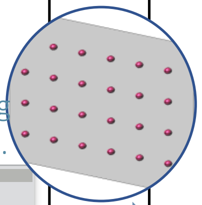
- Displacements, strains and stresses are calculated on structural components using a thermo-mechanical composites constitutive model.
- An isotropic damage model is used to assess the collapse risk of the structure.

Fire simulation & collapse assessment tool

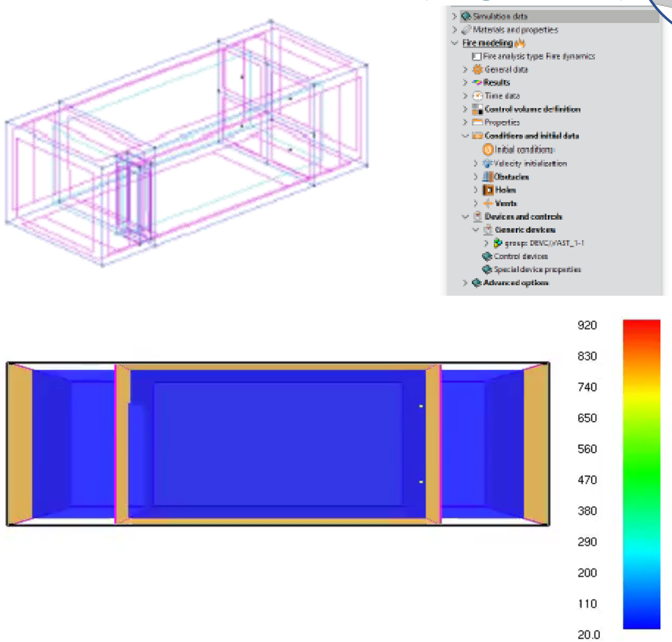
Example of application: Fire-resisting division test

Fire Dynamics



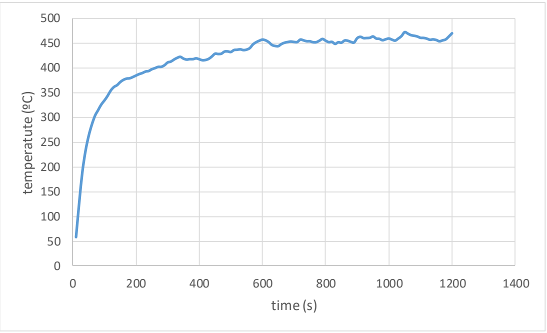


The compartment delimited by the fire-resisting division of interest is modelled (integrated GUI).



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.

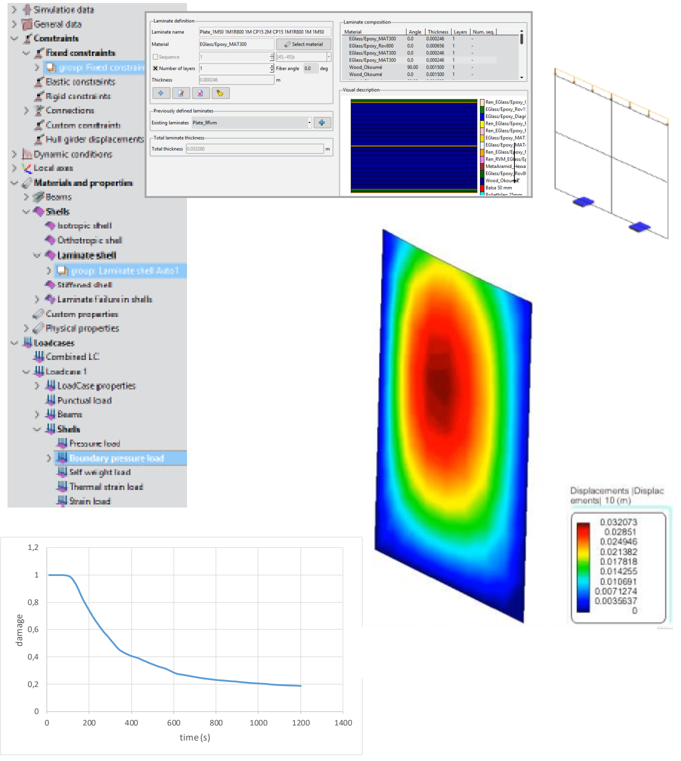
$$\rho C_p \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right) - \frac{\partial\left(w_g h_g\right)}{\partial x} + \dot{m}_{s\to g}\left(Q_p + h_c\right)$$

$$\rho C_p \frac{\partial T}{\partial t} = \nabla\cdot\left(k_T \nabla T\right) - \rho_g C_{pg} \mathbf{v}_g \cdot \nabla T + \dot{m}_{s\to g}\left(Q_p + h_c - h_g\right)$$

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)

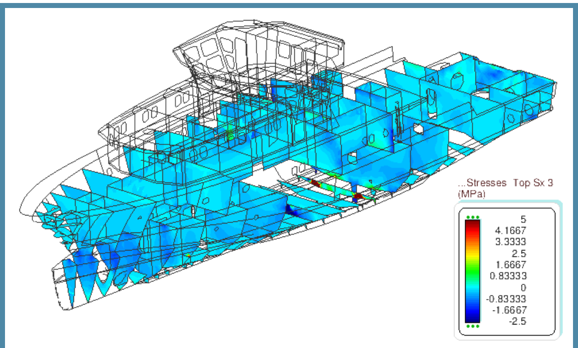
Solution: Full 3D hydroelastic (& non-linear) solver.

Objectives: To implement a coupled time-domain radiation/diffraction seakeeping analysis solver (2nd 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 hydro-elasticity solver

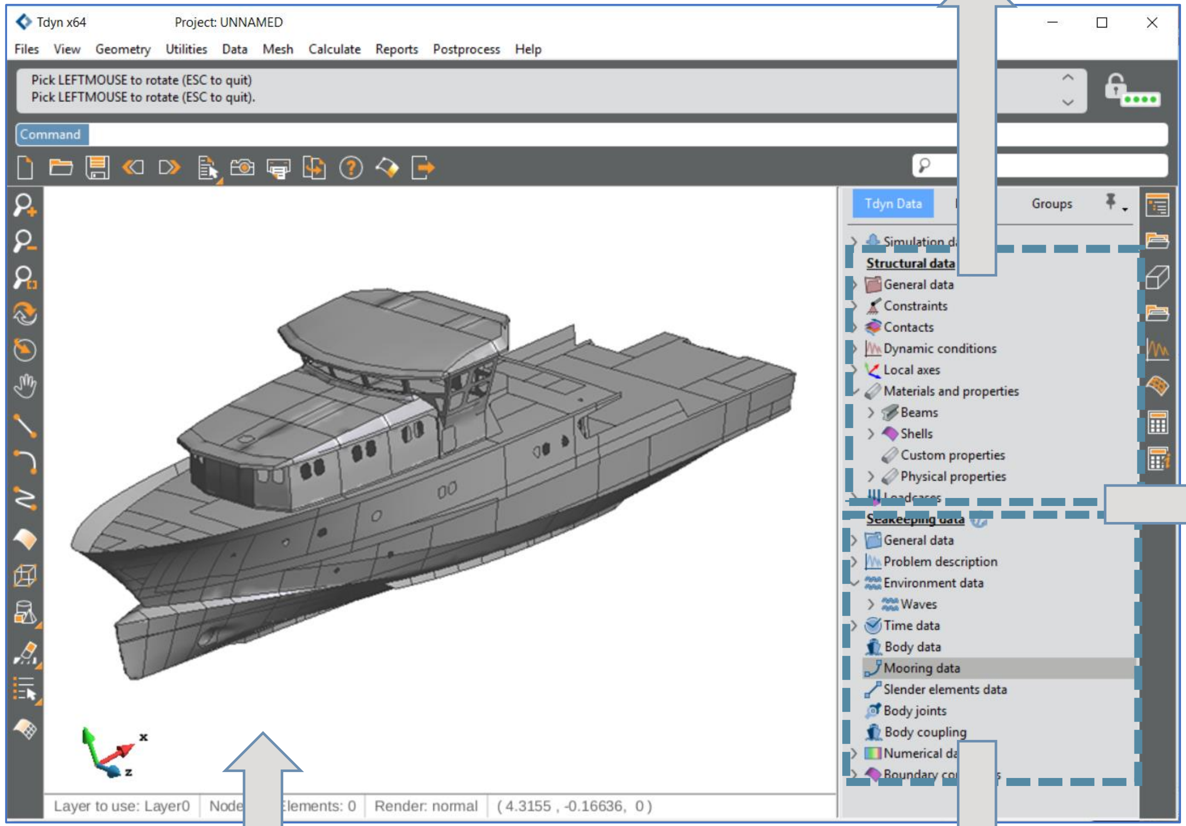


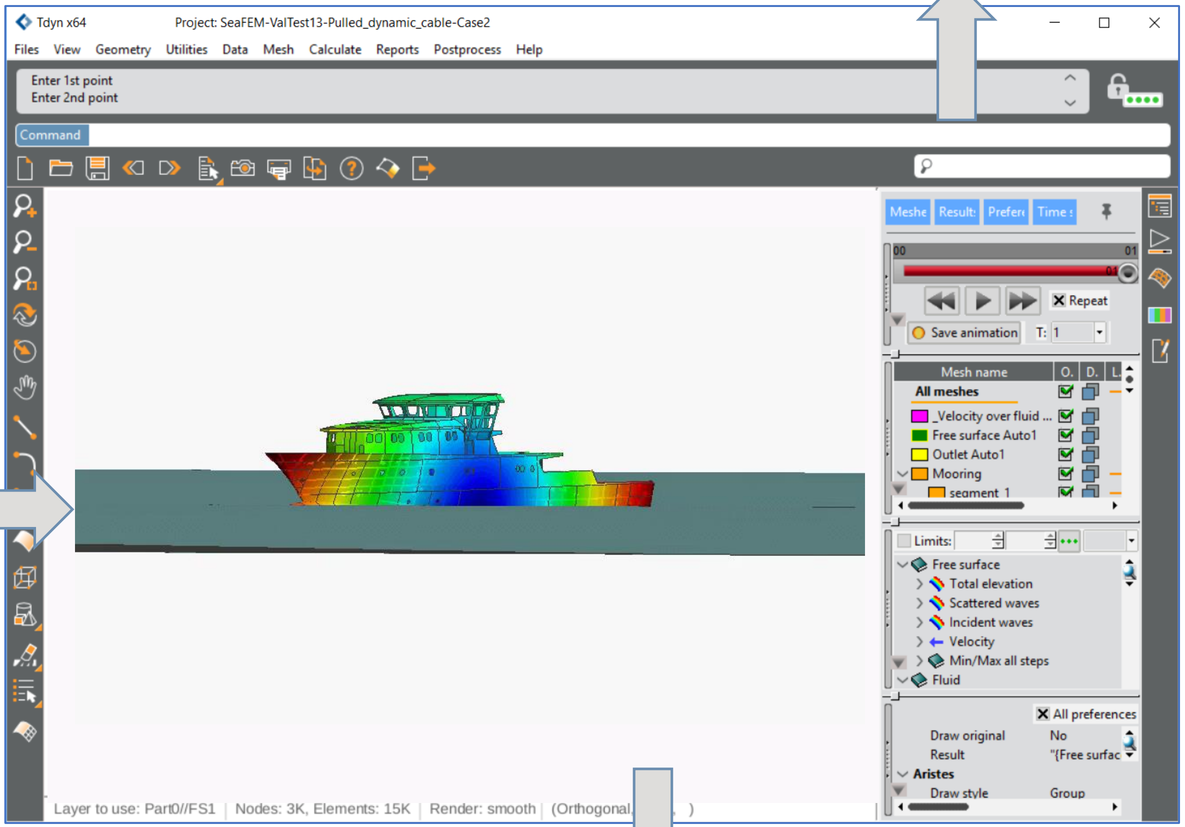
Full 3D hydroelastic solver

La Ciotat / 24-26th May 2019

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

Hydrolelastic solver
- Monolithic coupling





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

FEM seakeeping solver data
- Time domain
- 1st and 2nd order

Validation
Validation to be done with the data gathered onboard Zim Luanda container ship (carried out by TSI)

Hull girder model + 3D FEM analysis

Solution:

- Hull girder model (basic component for fatigue assessment and health structural monitoring tools) + 1D to 3D FEM interface.

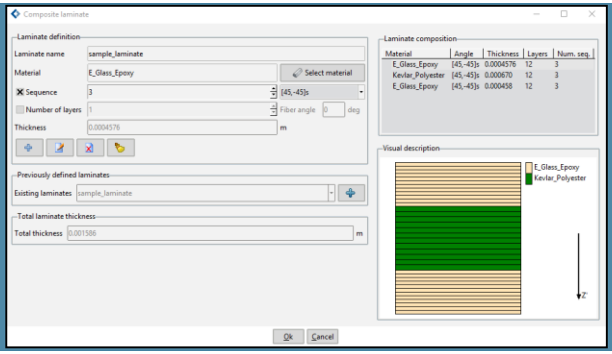
Objectives:

- 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.

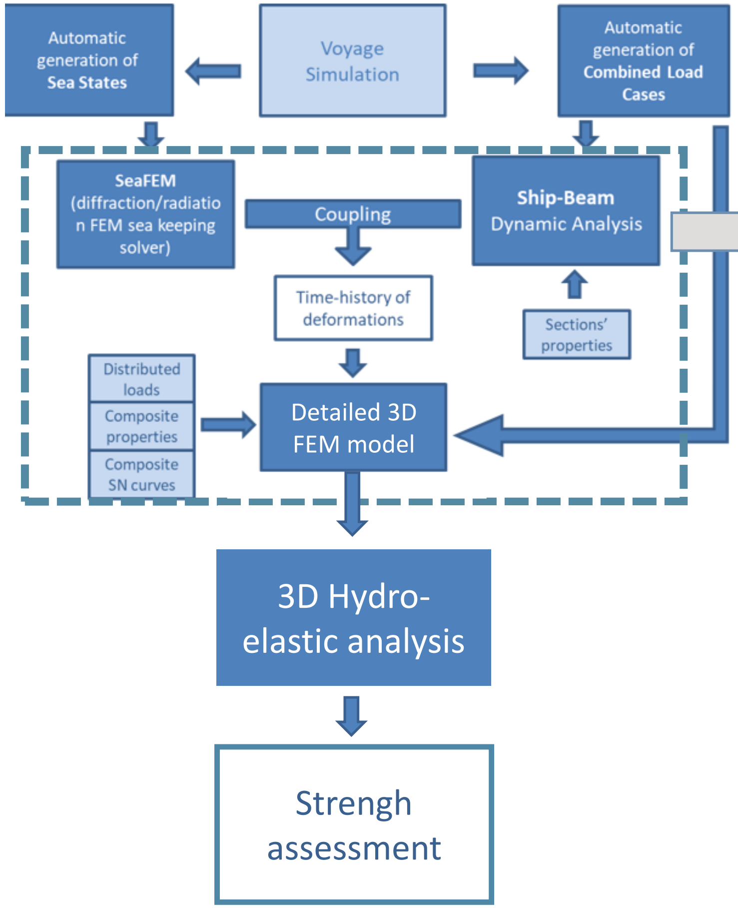
Other characteristics: Hull girder to 3D FEM model interface, Reduced computational cost, Usability (new GUI), Practicality.

Hull girder model

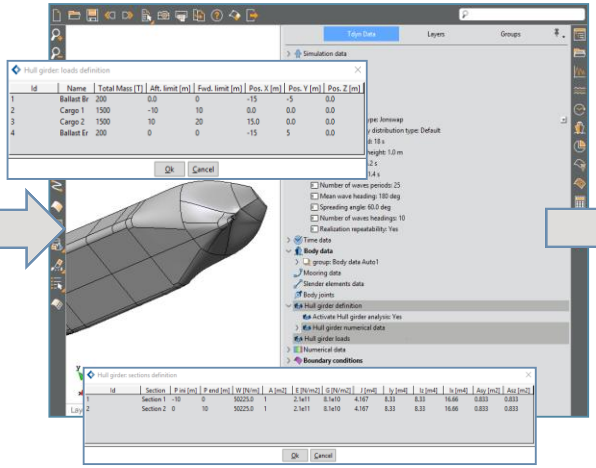


Hull girder model

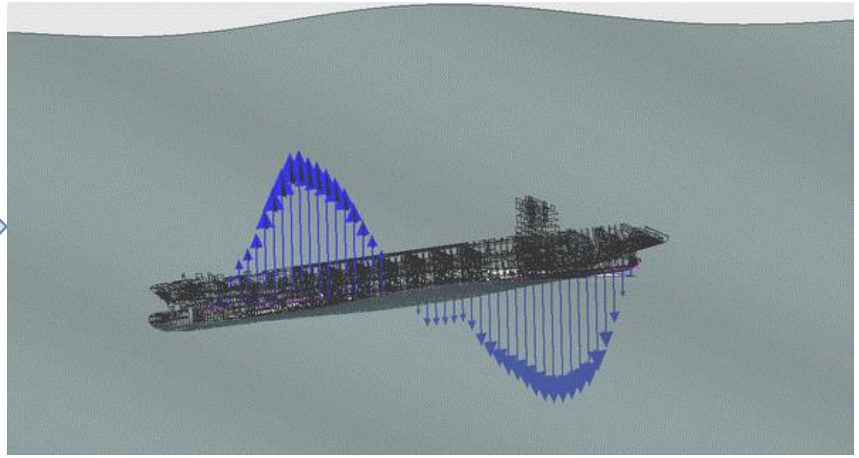
Definition of FRP laminates



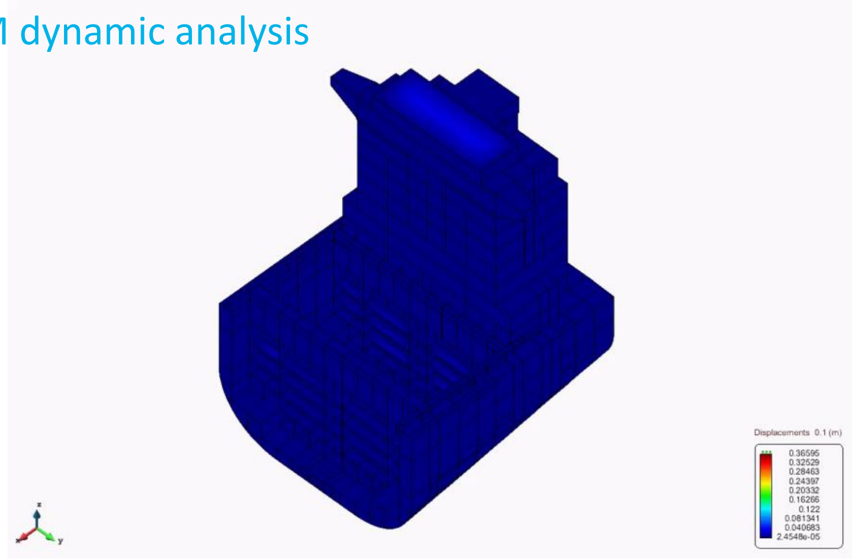
Hull girder model definition



Coupled hull girder – seakeeping analysis



3D FEM dynamic analysis

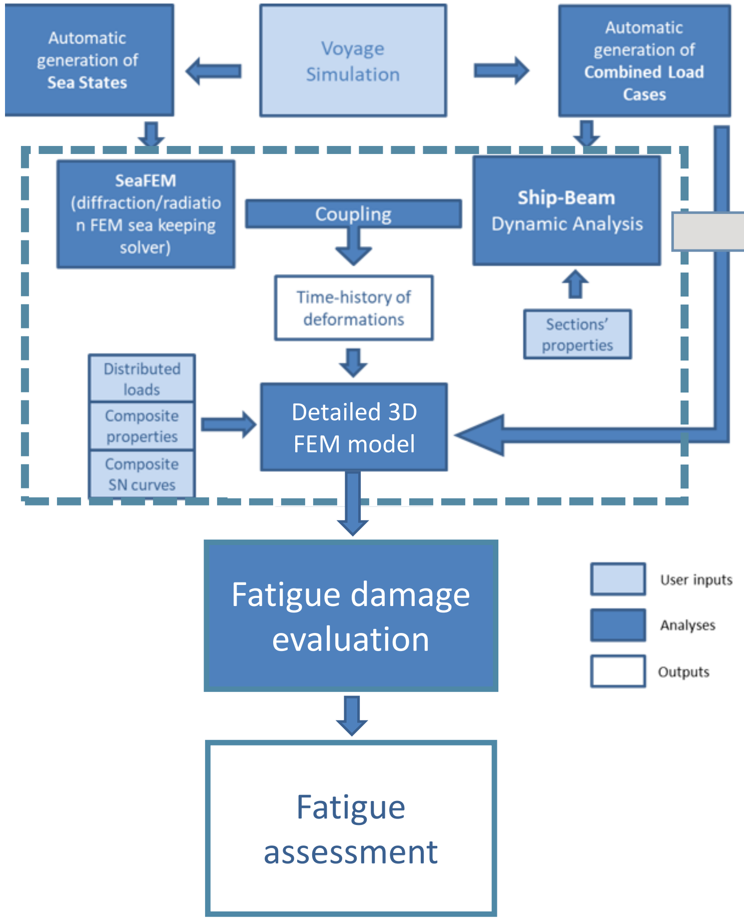


1D to 3D interface

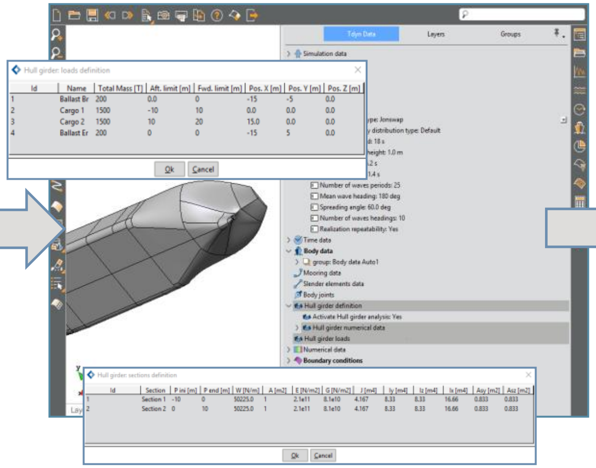
- Transfer of (time-domain) seakeeping wave loads
- Hull girder stresses / displacements are imposed at the boundaries of the 3D section under analysis
- 3D FEM model offers local displacements and stresses using SP-RoM model

Hull girder model

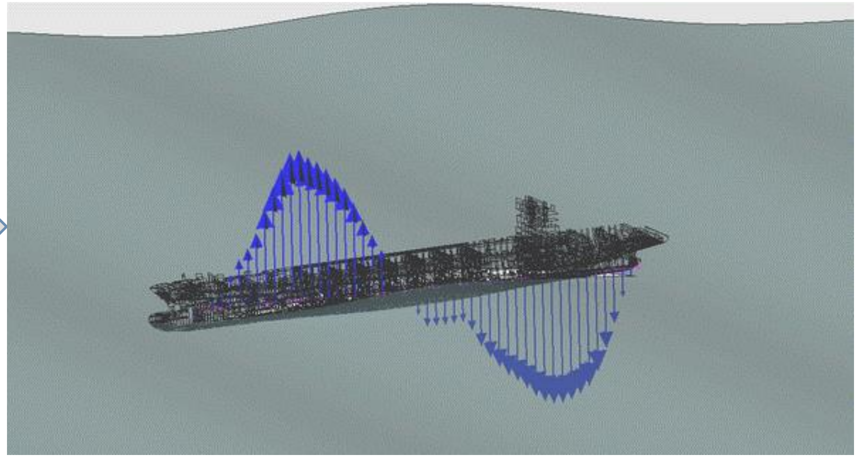
Definition of FRP laminates



Hull girder model definition

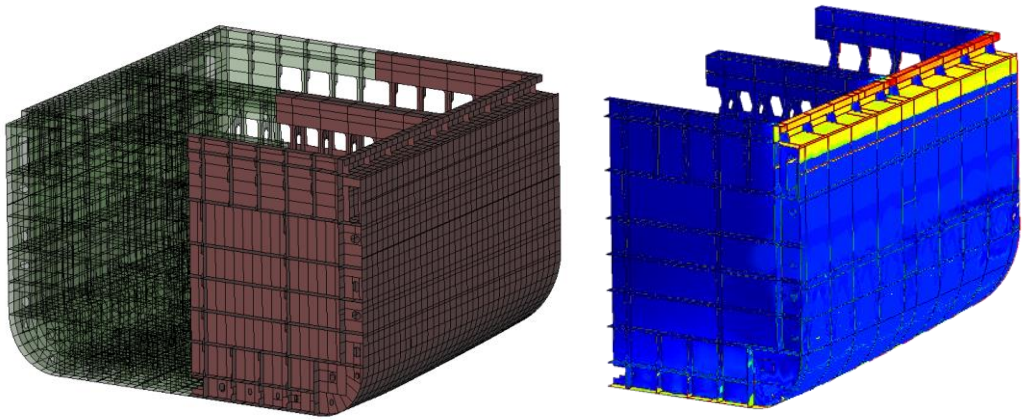


Coupled hull girder – seakeeping analysis



1D to 3D interface

3D FEM fatigue analysis



- Fatigue damage is estimated using a rainflow counting-type algorithm (Miner's Rule) based on the SP RoM model.

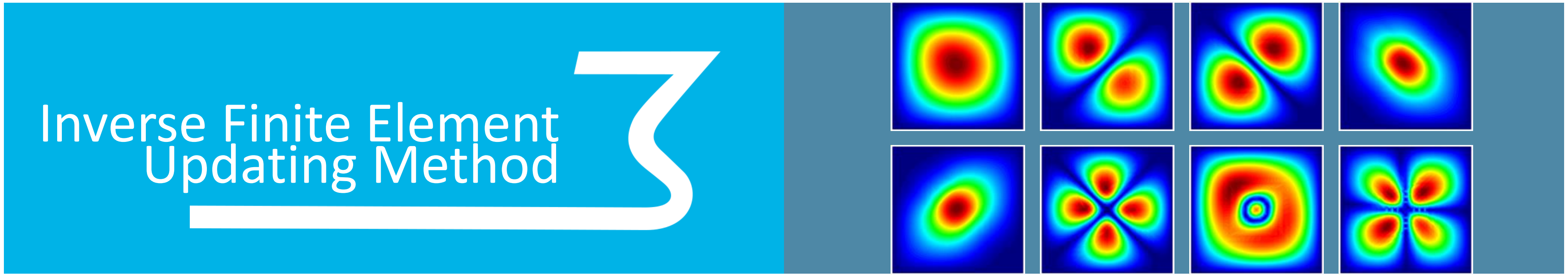
Solution:

- 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).



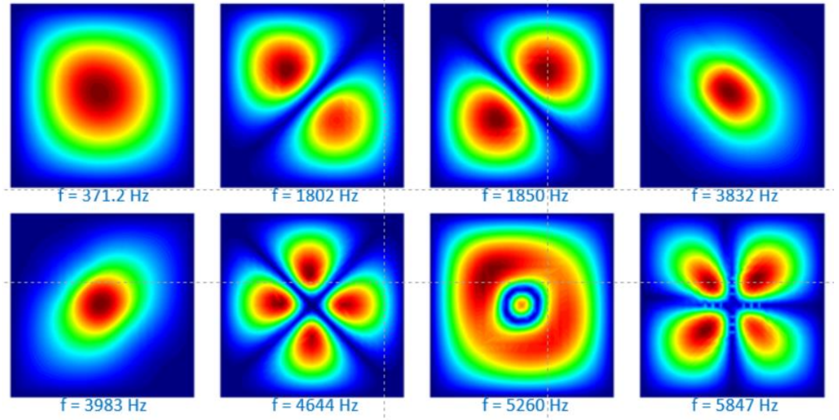
Problem statement

- 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:

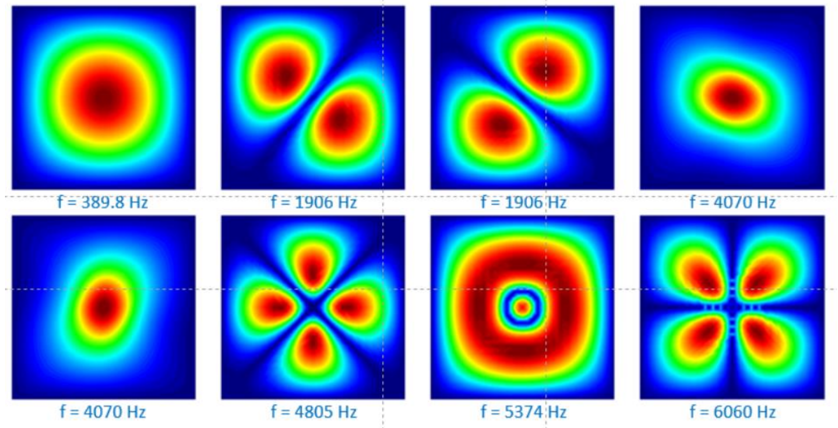
$$f_{obj} = w_i \cdot (f_{ref,i} - f_i)$$

- 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)

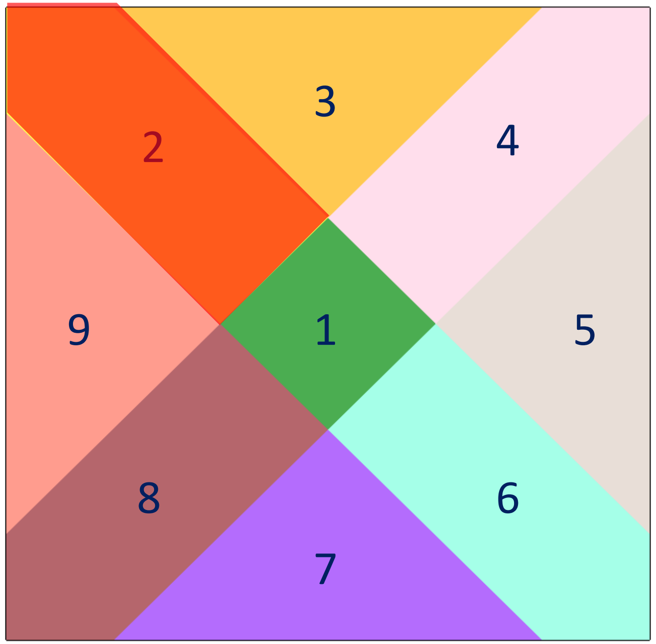
Normal modes / frequencies (zone 2 damaged)



Normal modes / frequencies (undamaged)



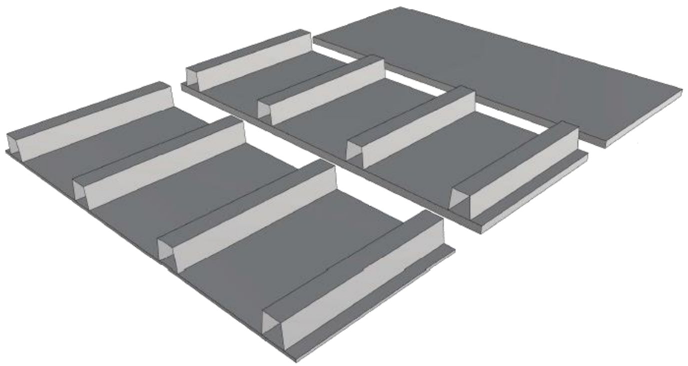
Squared shell panel divided into nine zones

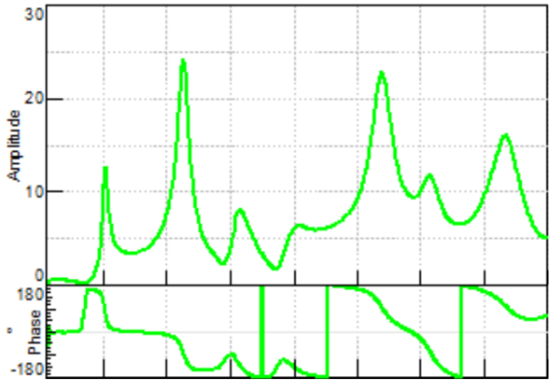


The compartment conaining with the k.

Example of application: Quality assessment of a composite panel

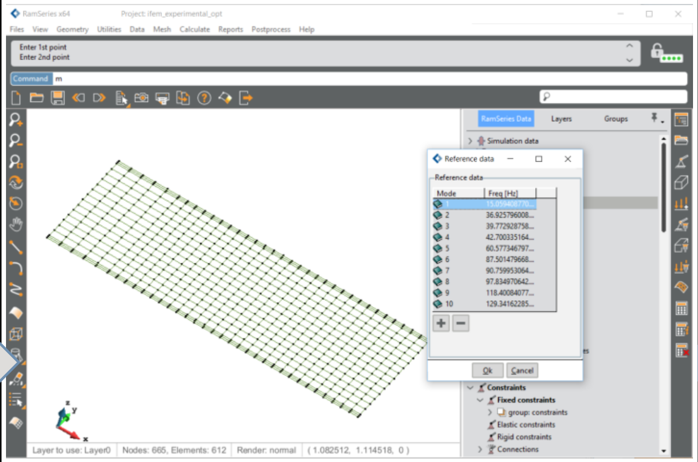
Modal testing

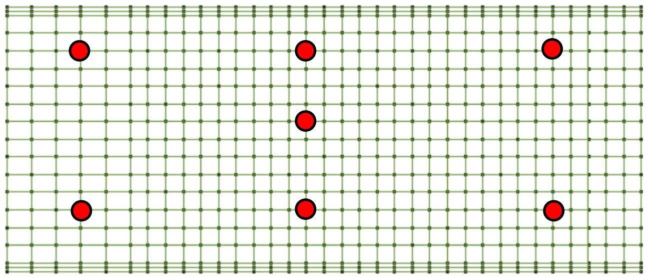




The natural frequencies of the structure are measured.

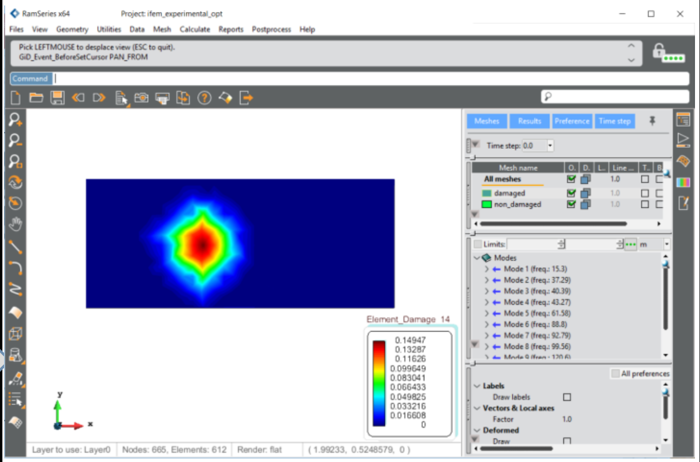
iFEM CAE model

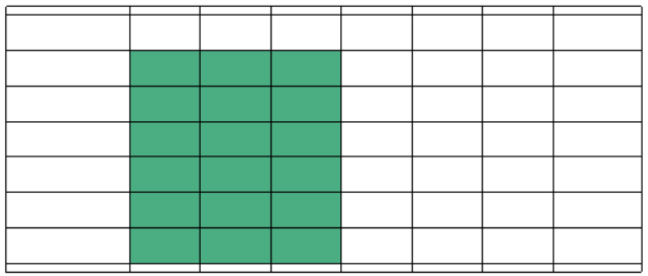




The iFEM model is created: FEM panel model + reference modal frequencies + a set of control points, definind the area of interest.

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

Computational Analysis Tools

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

- 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.

This project has recived funding from European Union's Horizon 2020 research and innovation programme under grant agreement Nº 723360

THANK YOU

www.fibreship.eu