

RamSeries reference manual

Three-dimensional environment for structural analysis.

<http://www.compassis.com>

info@compassis.com

November 2018

Version: 15.1.0

1. Introduction

RamSeries can perform structural analyses of beams, shells, solids, cables and membranes, or combinations of them, using the finite element method. It assumes linear elasticity in the material and small displacements in the overall structure. The beam, shell and solid hypotheses assumed in the analysis are described in the appendixes 2, 3 and 4. It can also dimension the analyzed concrete beams and shells with the necessary steel based on the EHE Spanish regulation.

Furthermore, material non-linearities (plasticity), and geometrical non-linearities (big displacements) can be analyzed. Non-linear elastic boundary conditions can also be used.

All the analysis modules are embedded inside GiD, the pre and post-processor. Knowledge of GiD is assumed in order to create the geometry, generate the mesh and visualize the results in the post-processing stage. Please, refer to the GiD manual for more details about the use of this program.

It is strongly recommended to fully read the Reference Manual, and study the tutorials before starting to use RamSeries.

1.1. Manual organization

This manual is organized as follows:

- A first chapter including a set of tutorials which try to cover most of the basic program capabilities.
- A second chapter which includes the Reference Manual
- The last chapter consists of different appendixes refer to theory issues concerning RamSeries capabilities and functionality.

2. Global and Local axes for beams

To access local axes definition and assigning interface, go to:

(in the menu)

Data ► Local axes ► Open

Or,

(in the data tree)

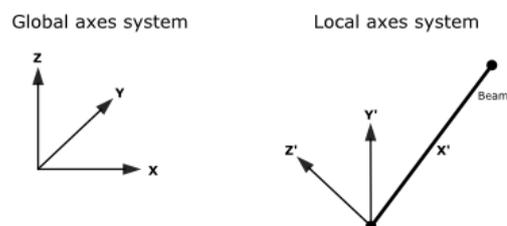
Local axes ► Local axes definition

When creating a model in RamSeries, the geometry of the structure under analysis is referred to a global axes system, labeled XYZ, that is unique for the entire problem. On the other

hand, each beam of the structure must have associated its own local axes system, labeled X'Y'Z'. Such local axes system is used to:

1. Refer section properties like inertia modulus, thickness and height to the local system.
2. Some of the load conditions available in RamSeries can be local or global. When using the local load type, the load is referred to the local axes system. If not, the load is referred to the global system.
3. Strength results over the beams are referred to the local axes system.

In all cases the local axes system must be defined so that the local X' axe is directed along the longitudinal direction of the beam as shown in the figure below.



Global vs. local axes coordinate systems. Note that for local axes in beams, X' axis is directed along the longitudinal axis of the beam.

There exist several ways of defining local axes within RamSeries. The various methods can be summarized as follows:

1. **Default method.** The calculation program assigns a different local axes system to each beam with the following criteria:
 - X' direction is along the longitudinal axis of the beam.
 - If X' axis has the same direction than the global Z axis, then Y' axis takes the direction of the global X axis. If not, Y' axis is calculated so as to be horizontal (orthogonal to X' and Z).
 - Z' axis results from the cross product of X' and Y' axes. It will try to point in the sense of the global Z (dot product of Z and Z' axes will be positive or zero).

Note that the basic idea underneath the default method for local axes construction is that vertical beams have the Y' axis in the direction of the global X. All other beams have the Y' axis horizontal and with Z' axis pointing up.

Automatic method. This is similar to the previous default method, but the local axes system's assignment is not performed by the calculation program at run time but is done automatically by GiD. Hence, the actual orientation can be checked in the graphic user interface with the *Draw Local Axes* option in the GiD Conditions window.

3. **Automatic alt method.** This is similar to the previous automatic method, but an alternative proposal of local axes is actually performed. Typically, the user should assign Automatic local axes and check their actual orientation after the assignment with the *Draw local axes* option. If a different local axes system is desired (normally rotated 90 degrees from the first one) then it is only necessary to assign again the same condition to the entities with the Automatic alt option described here.

4. **User defined method.** The user can create different local axes systems with the following command and using the various methods described above.

Data ► Local axes ► Define

These user defined systems are labeled with an identification name and added to the list of systems ready to be chosen.

Notes on the use of local axes:

- RamSeries tries to correct the local axes system if the local X' axis does not point to the direction of the beam. Nevertheless, it will fail if local X' axis is orthogonal to the direction of the beam. Hence, local axes should be assigned carefully to avoid these particular problems.
- The final local axes system for each beam can be visualized in the post-process stage. It is convenient to check for the correctness of these systems' orientation after the calculation is performed to ensure the actual proper interpretation of the results.

3. Global and Local axes for shells

To access local axes definition and assigning interface, use the following command's sequence from the main menu:

Data ► Local axes ► Open

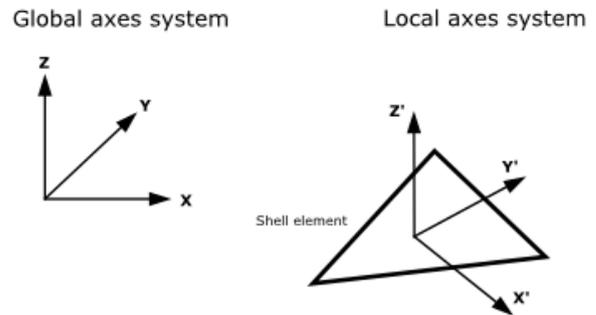
or, use the following option from the data tree:

Local axes ► Local axes definition

When creating a model in RamSeries, the geometry of the structure under analysis is referred to a global axes system, labeled XYZ, that is unique for the entire problem. Nevertheless, as in the case of beams, every shell element of the structure must have associated its own local axes system, labeled X'Y'Z'. Such local axes system is used to:

5. Refer section properties like orthotropy properties to the local system.
6. Some of the load conditions available in RamSeries can be local or global. When using the local load type, the load is referred to the local axes system. If not, the load is referred to the global system.
7. Strength results over the shell are referred to this local axes system.
8. Dimensionalization of steel shells is based on X' and Y' directions.

The main property of this local axes system is that the local Z' axe must take the direction of the normal of the shell element.



As in the case of beams, there exist several ways of defining local axes for shells within RamSeries. They can be summarized as follows:

1. **Default.** The program assigns a different local axes system to the shell with the following criteria:
 - Be **N** the unitary normal of the shell element, **U** the vector (0,1,0) and **V** the vector (0,0,1). Then:
 - Z' axe has the direction and sense of **N**.
 - If $N_x < 1/64$ and $N_y < 1/64$, then X' axe will be in the direction of the cross product of **U** and **N** (**UxN**).
 - If not, X' axe will be in the direction of the cross product of **V** and **N** (**VxN**).
 - Y' axe will be the cross product of Z' axe and X' axe.

Note: Intuitively, this local axes system is calculated so as if element is approximately contained in the plane XY, local X' axe will point towards global X axe. If not, this X' axe is obtained as orthogonal to global Z axe and local Z' axe.

2. **Automatic.** Similar to the previous one but the local axes system is assigned automatically to the shell by GiD. The final orientation can be checked with the *Draw Local Axes* option in the GiD Conditions window.
3. **Automatic alt.** Similar to the previous one but an alternative proposal of local axes is given. Typically, User should assign Automatic local axes and check them, after assigning, with the *Draw local axes* option. If a different local axes system is desired, normally rotated 90 degrees from the first one, then it is only necessary to assign again the same condition to the entities with the **Automatic alt** option selected.
4. **User defined.** User can created different named local axes systems with the GiD command:

Data ► Local axes ► Define

and with the different methods that can be chosen there. The names of the defined local axes will be added to the menu where Local axes are chosen in the conditions window.

Note 1: RamSeries tries to correct the local axes system if the local Z' axe does not point to the direction of the normal of the element. It will fail if local Z' axe is orthogonal to the direction of the normal.

Note 2: The final local axes system for every shell element can be visualized in the post-process stage. It is convenient to check the correctness of these systems after calculation is performed.

4. Reference Manual

The following sections will explain the capabilities of the pre-processing part, the results that can be displayed in the post-processing part as well as more detailed information about several issues.

4.1. Item help

It is possible to obtain help for several items in the toolkit and windows by pressing right mouse button over them.

4.2. Units

The default units system for RamSeries is the International System (IS). They are:

- Time in seconds (s)
- Lengths in meters (m)
- Masses in Kilograms (kg)
- Forces in Newton (N)
- A derived unit is Pascal (Pa), where $Pa=N/m^2$

These units can be changed in several parts of the program. Every window that asks for data for constraints, load or properties has a field to choose the units for that window. The chosen units are only applicable in the data attached to it in the same window.

The units in which are expressed the coordinates of the geometrical model are chosen in the Problem data window:

Data ► Problem data ► Units ► Mesh units

The units in which are expressed the results of the analysis can be chosen in:

Data ► Problem data ► Units ► Result units

Other predefined units that can be chosen are:

- Forces: Kilogram-force (kp)
- Lengths: centimeter (cm), millimeter (mm)

Results units can be expressed as:

- N-m-kg where:
 - Displacements are in m
 - Velocities are in m/s
 - Accelerations are in m/s^2
 - Strengths are in N/m or N·m/m
- Stresses are in $Pa= N/m^2$
- N-mm-kg where:
 - Displacements are in mm
 - Velocities are in mm/s
 - Accelerations are in mm/s^2
 - Strengths are in N/mm or N·mm/mm
- Stresses are in N/mm^2
- Kp-cm-utm where:
 - Displacements are in cm
 - Velocities are in cm/s
 - Accelerations are in cm/s^2
 - Strengths are in Kp/cm or Kp·cm/cm

Stresses are in Kp/cm^2

- KN,m,Mpa where:
- Displacements are in m
- Velocities are in m/s
- Accelerations are in m/s^2
- Strengths are in kN/m or kN·m/m ($kN=10^3 N$)
- Stresses are in $Mpa=10^6 Pa$

Note that the units in this system does not form a compatible system.

The three components of the **gravity** define a vector, which will be normalized by RamSeries, and represents the direction of the gravity if the self-weight is considered. In case *self-weight* load, the gravity components are not used:

Simulation data ► Gravity

Note for advanced users: In order to add new units to the predefined set in *compassfem*, it is necessary to modify two files: *compassfem.cnd* and *compassfem.bas* contained in the directory *compassfem.gid*. This directory is the predefinition of the problem type RamSeries inside GiD. In the file *compassfem.cnd*, it is necessary to search the fields **Units** and add the new unit to all of these fields. In the file *compassfem.bas* it is only necessary to add an entry to the units table that is at the beginning of the file.

4.3. Visual reference

A visual reference for the icons corresponding to the tools and functions described later, is showed in this section.

Structural analysis preprocessor toolbar	
	Show/Hide the pre-processor data tree
	Define general data
	Assign constraints
	Assign properties
	Assign loads
	Draw constraints and connections
	Draw properties
	Draw loads
	Calculate
	View process information
	Cancel calculation process

In Toolbar are available only the most common options shown before, but all of them are in data tree.

4.4. Simulation data

In this case, for RamSeries, the corresponding *Simulation type* is **Structural Analysis**.

Simulation data ► Simulation type

4.4.1. Units

Geometry units are the units in which are represented the coordinates of the geometry and mesh. **General units** are the units used to represent the results of the analysis.

Simulation data ► Units

Geometry units can be: meter (m), centimeter (cm) or millimeter (mm), etc,...

General units can be expressed as:

- N-m-kg where:
 - Displacements are in m
 - Velocities are in m/s
 - Accelerations in m/s²
 - Strengths are in N/m or N-m/m
- Stresses are in Pa= N/m²
 - N-mm-kg where:
 - Displacements are in mm
 - Velocities are in mm/s
 - Accelerations in mm/s²
 - Strengths are in N/mm or N-mm/mm
 - Stresses are in N/mm²
 - Kp-cm where:
 - Displacements are in cm
 - Velocities are in cm/s
 - Accelerations in cm/s²
 - Strengths are in Kp/cm or Kp-cm/cm
 - Stresses are in Kp/cm²
 - KN,m,Mpa where:
 - Displacements are in m
 - Velocities are in m/s
 - Accelerations in m/s²
 - Strengths are in kN/m or kN·m/m (kN=10³ N)
 - Stresses are in Mpa=10⁶ Pa

Note that the units in this last system does not form a compatible system.

4.4.2. Gravity

The three components of the **gravity** define a vector, which will be normalized inside *the program*, and represents the direction of the gravity if the self-weight is considered.

Simulation data ► Gravity

4.4.3. Coupling data

These options are available when *Multiphysics Analysis*, *Thermomechanical Analysis*, *Fluid-Structure Interaction Analysis* or *Coupled Seakeeping-Structural Analysis* are selected:

Simulation data ► Simulation type

Please refer to [Coupling data](#) for further information on Coupled Analyses.

4.5. General data

This is the information needed for the analysis and not related to the geometry.

4.5.1. Incremental analysis data

Num increments: indicates the number of parts that the total load is divided

4.5.2. Analysis

General data ► Analysis

Simulation Dimension: The options are 3D, 2D Plain Stress, or 2D Plane Strain. If the problem to analyze has only elements in one plane (2D) and it can be considered as a plane strain or plane stress problem, select one of these options. In these cases, the input of the data is the same than for the general analysis but the problem to solve is much faster. So, it saves a lot of computing time.

Element type: Available options are Beams, Shells, Solids, cables and membranes. It is possible to set up problems using all of the elements (option "All"), or problems combining Beams & Shells & Solids, or just Beams & Shells combination.

Analysis type: This option select the type of analysis to be perform. RamSeries can perform the following analysis:

- Static analysis
- Incremental loads analysis
- Dynamic analysis

Material constitutive model: Allows to choose set the behaviour of the material whether it is going to be linear or plastic.

- Linear materials
- Plasticity on materials

Geometric constitutive model: Indicates if geometrical non-linearities are going to be considered (i.e. great deformations):

- Linear geometry.
- Non-linear geometry.

Boundary conditions: Allows to use non-linear elastic constraints:

- Linear BC
- Non-linear BC.

Use Laminate/Composite materials: Activates Composite module capabilities and tools, to allow defining composite materials, create laminates and analyse the results layer by layer with advanced and specific fail criterii.

Internal triangular element: This option controls the type of element formulation to be used when working with triangular meshes. RamSeries provides three different formulations for triangular elements whose characteristics can be summarized as follows:

DKT: this is a 3-noded triangular element that mixes the classic plane stress theory together with the Discrete Kirchoff Triangular elements (DKT) theory for plates.

- **6-noded element:** this is a 6-noded element generated internally in RamSeries from 3-noded triangular meshes. It gives more precise results at the expense of more computational time. This element is based on the classical plane stress theory combined with the Reisner-Mindlin plate's theory. This element is actually equivalent to the quadratic triangular element directly generated over the mesh. Nevertheless, it has the advantage that it can be directly combined with linear 2-noded beam elements, which is not allowed by the quadratic triangular element.
- **Drill-Rot:** if this option is chosen, the Drilling-Rotation triangular element will be used. This is a 3-noded triangular element based on the DKT formulation that adds in-plane rotation degrees of freedom. It largely improves the solution so that its use is recommended in most cases.

Note that RamSeries also provides a 4-noded quadrilateral element which implements the classical plane stress theory together with the Reisner-Mindlin plate's theory. In the case the mesh is generated using quadrilateral elements, this element type is automatically used irrespectively of the 'internal triangular element' option.

Coplanarity angle:

Marine tools: Activates some specific Naval Architecture tools, as automatic ship equilibration, static wave loads, stiffened shells, buckling shell analysis, among others.

Damage assessment: Activates fatigue damage assessment tools for lines (welded joints) and shells.

SN curves file: If Damage assessment is activated, it is possible to choose a file containing SN experimental curves.

BeamP-Delta: If this option is set, the beam P-delta method is applied in order to calculate second order effects for columns.

Be careful because one of the effects in a bad designed structure is that it will not be possible to obtain a solution. The recommended way of working is to calculate first in first order, and after with second order. If the strengths increment more than a certain value (20%-30%), a redesign of the structure is advised.

Initial configuration: Activates options for setting-up an initial configuration analysis, which will be performed prior to dynamic analyses.

Linearized prebuckling analysis: this option is available for Static or Incremental Load Analysis. If it is activated, **Buckling data** will appear after **Analysis** and a buckling analysis will be done at the beginning of calculation, before Static or Incremental Analysis. Buckling can be performed only for models with triangles.

4.5.3. Automatic equilibration

4.5.4. Non-Linear analysis data

Requisites: Non-Linear analysis data is only available if at least one non-linear feature (i.e. material, geometric or boundary conditions non-linearity) is previously activated. Non-linear features that actually activate non-linear analysis data are those listed here:

General data ► Analysis ► Material constitutive model ► Plasticity on materials

General data ► Analysis ► Geometric constitutive model ► Non-linear geometry

General data ► Analysis ► Boundary conditions ► Non-linear boundary conditions

General data ► Non-Linear analysis data

If the Analysis Type option is set in the Dynamic, the following data must be reviewed (Note that advanced options only apply to *Incremental* analysis):

Solver Control: indicates the type of load control required for the analysis. RamSeries can perform a Load Control, Displacement Control and Arc-Length control.

Conv. tolerance: represents the convergence value for the Non-Linear analysis.

Iteration type: indicates when the structural stiffness matrix is recalculated. RamSeries can perform a recalculation in each iteration of each load step (Full_Newton-Raphson) or a recalculation in the first iteration of each load step (Modified_Newton-Raphson).

Max iterations: indicates the maximum number of iteration allowed in each load step

Line-Search: indicates if the line-search method is required

Automatic increment: indicates if an automatic incrementation procedure is required

Auto-ARC-switch: indicates if an automatic switch to the Arc-Length Control is required.

Use Stabilization: indicates if extra stabilization steps (increments) need to be performed.

Automatic inc

Num Iteration: indicates the number of iteration desired in each load step.

ΔP max: indicates the maximal load increment allowed.

ΔP min: indicates the minimal load increment allowed.

Line-Search

If the Line-Search option is chosen, the following data must be indicated:

Loops: define the maxim number of Line-Search loops.

Tolerance: define tolerance ratio desired.

Min: define the maximal step-length of Line-Search.

Max: define the minimal step-length of Line-Search.

Max. amplitude: define the maximal amplitude of any step.

Auto ARC-switch

If the Auto ARC-switch option is chosen, the following datum must be indicated

C Stiff: indicate the desired current stiffness parameter for switching.

Use Stabilization

In certain analyses, singular and very bad conditioned (high condition number¹) stiffness matrices may appear. This usually

happens, for example, when dealing with cables, membranes or very thin shells elements.

In general, if the analyses are solved dynamically, there should be no problem achieving convergence, for the damping would compensate the singularity of the stiffness matrix ($M \cdot \Delta x'' + C \cdot \Delta x' + K \cdot \Delta x = F_{ext}$)

Nevertheless, for static non-linear (incremental) analyses, convergence problems may arise when the mentioned type of elements are involved, due to the lack of damping ($K \cdot \Delta x = F_{ext}$). Therefore, a method is implemented in RamSeries so that convergence can be achieved. This is done via adding an stabilization or "artificial damping":

$$(*SFM) \cdot \Delta x' + K \cdot \Delta x = *SFM \cdot \Delta x / \Delta t + K \cdot \Delta x = (*SFM / \Delta t + K) \cdot \Delta x = F_{ext}$$

The artificial damping is given by a diagonal matrix which can be called *Stability Factor Matrix (SFM)*, defined as:

$$SFM_{ij} = *SFM_{ij} / \Delta t = SF \quad (i=j)$$

$$SFM_{ij} = 0.0 \quad (i \neq j)$$

SF is called the **Stabization Factor**.

After finishing the **Num. increments** which the user has inserted, RamSeries can perform extra increments in order to stabilize the analysis and achieve the desired convergence. These extra increments can be inserted in the **Stabilization increment number**.

¹.Condition number (κ) of a matrix (A), considering $\| \cdot \|_2$ norm, is defined as:

$$\kappa(A) = | \lambda_{max}(A) / \lambda_{min}(A) |$$

Where $\lambda_{max}(A)$ and $\lambda_{min}(A)$ are maximal and minimal (by moduli) eigenvalues of A respectively.

4.5.5. Dynamic analysis data

General data ► Dynamic analysis data

If the Analysis Type option is set to Dynamic, the following data must be reviewed:

General data ► Dynamic analysis data ► General

Type: There are three possible types of dynamic analysis:

5. Direct integration
6. Modal analysis
7. Frequency domain analysis.

ΔT: Indicate the step of time to be considered in the dynamic analysis. It is possible to define different ranges of time steps (This options is very useful when different time step are required along the analysis).

Number of steps: Indicate the total number of steps of the dynamic analysis. The total physical time will be the number of

steps times the time step.

General data ► Dynamic analysis data ► Integration data

Integration method indicates the algorithm of temporal integration to be performed in the Dynamic analysis. Options for implicit schemes are: Newmark, Bossak-Newmark, Hilber-Hughes-Taylor and Energy conserving/decaying.

Alpha BN, Alpha HHT, Alpha E-C/D: Parameters for Bossak-Newmark, Hilber-Hughes-Taylor and Energy Conserving/decaying schemes, respectively.

Gamma, Beta: Parameters for the implicit schemes of temporal integration.

Matrix Storage: Indicates the type of storage for the Mass and Damping matrices. There are two possibilities: Lumped and Consistent. A lumped matrix storage means that the masses are concentrated in the nodes and then the mass matrix are reduced to vector. A consistent matrix storage means that the masses are consider consistently with the finite element approach and then the mass matrix is storage completely.

Initial Conditions: indicate which type of initial conditions are required. The initial condition can defined manually for the users (User_Defined) or can be assigned like initial conditions the values obtained in the static linear elastic analysis performed in the Combined Load Case 1 (Comb_Load_1)

General data ► Dynamic analysis data ► Damping data

Damping Type: Indicates the way that the damping is considered. There are two possibilities: Modal Damping and Rayleigh Damping. In the modal damping option the damping is taking in account in the equations of motions of each mode without compute any Damping Matrix, with this option is necessary to input the damping ratio. In the Rayleigh damping option is computed a Damping Matrix proportional to the stiffness and mass Matrices, with this option is necessary to input the coefficients Alpha_M and Alpha_K.

Alpha M: represents the coefficient of the Mass Matrix in the Rayleigh Damping

Alpha K: represents coefficient of the Damping Matrix in the Rayleigh Damping

Modal analysis

General data ► Dynamic analysis data ► General ► Type: Modal analysis

Only calculate natural freqs: If it is set, only natural frequencies will be calculated, otherwise, a dynamic analysis will be done after natural frequencies calculation is finished. Only it is possible to performance a dynamic analysis if there are loads applied and model is completely constrained. Therefore, this option allows to calculate natural frequencies of models without loads or not completely constrained.

If chosen, the following option appear:

Type of modal analysis:

- **Number of modes:** Indicates the number of modes to be considered in the vibration analysis.
- **Range of modes:** Natural frequencies between the given range will be calculated. If **Compute all modes** is set, all modes between the range given will be calculated, otherwise, the first number of modes between the range will be considered.

Freq. domain analysis

General data ▶ Dynamic analysis data ▶ General ▶ Type: Freq. domain analysis

If chosen, the following options appear:

Type of modal analysis:

- **Number of modes:** Indicates the number of modes to be considered in the vibration analysis.
- **Range of modes:** Natural frequencies between the given range will be calculated. If **Compute all modes** is set, all modes between the range given will be calculated, otherwise, the first number of modes between the range will be considered.

General data ▶ Dynamic analysis data ▶ Spectrum data

Spectrum Analysis type: Indicates the type of Spectrum Analysis to perform. There are two possibilities:

- **Seismic Codes:** It is possible to perform the spectrum analysis established in the NCSE-94 Spanish regulation.
- **User defined spectrum:** The user has the possibility to input his own spectrum of accelerations.

4.5.6. Buckling data

Only if **Linearized prebuckling analysis** is set, **Buckling data** is available.

- **Num. of buckling modes:** Number of buckling modes to be output.
- **Imperfections factor:** amplitude of geometric imperfection related to the first buckling mode. If Imperfections factor=0 then, structure will be considered ideal.

4.5.7. Results

General data ▶ Results ▶ Beams

Granularity means the number of subdivisions that will have every beam to represent the results. More subdivisions give more quality in the results visualization and more disk space. This option does not modify the precision of the result, only its visualization.

Output Maximums When there are more than one load case, it is possible to output a special load case containing the maximums for beams. Option **Automatic**, outputs it only if there are steel sections. Option **Always**, output it if there are more than one load case.

General data ▶ Results ▶ Beams and shells

Shell smooth results: If chosen, RamSeries will smooth the strength results where possible. The results of the calculation are strengths in the interior of every element that are discontinuous from one element to another. Smoothing means to approximate other values of the strengths so as they are continuous from one element to another. This can only be made if the geometry is smooth by itself from one element to another.

Output beams and shell stresses If chosen, the stresses and Von Mises in both, the face up and the face down of the shell are calculated. Von Mises results for beams will be also available in the postprocess.

General data ▶ Results ▶ Solids

Yield criterion Allows the user to choose the stress criterion to be visualized in the postprocess, for solids: Von Mises or Rankine.

Smooth results per material (option only available for solids). When mixing two materials very different, like steel and concrete, the stresses are not continuous between materials. Then, it is necessary to set this option in order to see the stresses jump in the two materials boundary. If active, stresses will be discontinuous between materials. Internally, nodes will be duplicated.

General data ▶ Results ▶ Composite sec. factor

If selected, different Failure Criteria for Composites will be output.

4.5.8. Advanced

This section treats advanced options for analysis.

Linear solver

General data ▶ Advanced ▶ Structural solver ▶ Equations solver

The option Solver type enables the user to choose the equation solver. The options are:

- **Hybrid-Sparse:** A direct solver with sparse storage. The most advisable option, for it takes the same memory as the sparse solver (same matrix storage), and it is much quicker than the direct solver used in the Skyline option.
- **Sparse:** A conjugate-gradients solver with sparse storage is used. This is an iterative solver that requires much less memory than a direct one. It may not converge in some cases. If this option is enabled, some parameters can be adjusted:

-Solver tolerance: When two successive iterations do not differ by more than the specified tolerance, the solver will halt.

-Solver tolerance minimum: If the solver arrives at its maximum iteration number, this tolerance will decide if results are accepted.

-Max. iterations: maximum number of iterations permitted.

- **Stabilized bi-conjugated gradients**

Parallel or sequential mode: For Hybrid-Sparse solver it is possible to use parallelization options, so solver operations can be performed in various processors.

Number of CPU's: Number of processors that the Hybrid-Sparse solver will be allowed to use.

The most advisable and suitable option is to choose **Hybrid-Sparse**.

Eigen solver

Eigensolver max iterations: The eigensolver used in the analysis of frequencies and modes of vibrations is the subspace iterative method. The option Eigensolver max iterations indicates the maximum number of iteration to be performed by the eigensolver.

Eigensolver min stages: if a model has a lot of nodes and many

modes must be calculated, the analysis may be interrupted because of insufficient memory. In these cases, natural frequencies calculation can be split into more than one stage.

Dynamic Output

A dynamic analysis generates a large amount of results which are passed on to the post-processing. In order to reduce the amount of data, RamSeries offers the possibility to choose which results are required. RamSeries have the possibility of choose what results are required:

- Write Displacements
- Write Rotations
- Write Velocities
- Write Accelerations
- Write Strengths
- Write Reactions
- Global parameters: Total energy, Kinetic energy, Potential energy, Elastic strain energy, Linear and Angular momentum.

Using the option Output Step, the user can specify how many interim results should be included in the post-process file.

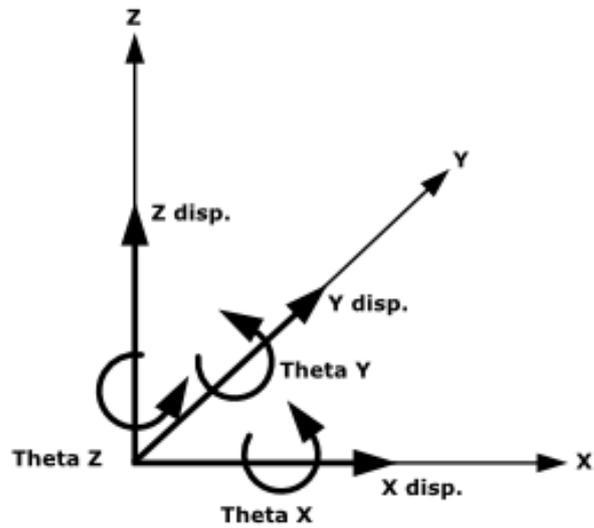
4.6. Constraints

Conds. & Init. Data > Constraints

4.6.1. Constraints and prescribed displacements

The parts of the structure that has any type of external constraint in their movements, are called constraints. These constraints can be applied for every degree of freedom (X, Y, Z and rotations for beams and shells), and for every node in the model

In this condition, the local axes have no relationship with the beam local axes defined in the properties section. The GLOBAL option means to prescribe related to the global axes of the problem. Local axes are used to prescribe the displacement or rotation in a direction not coincident with any of the global axes. The *values part* of the condition is used to prescribe a fixed amount of displacement or rotation. Default units are meters for the X, Y and Z displacements and radians for the prescribed rotations. X Constraint, Y Constraint and Z Constraint mean the displacements along the axes. Theta x Constraint, theta y constraints and theta z constraints mean the rotations around the axes. Signs are as follows (right hand rule):



This condition can be applied to either points, lines or in the solid analysis, to surfaces.

Note: 3D solids have only three degrees of freedom: displacements in X, Y and Z.



4.6.2. Elastic constraints

The elastic constraints are similar to the constraints but instead of prescribing the displacement or rotation of a point, an elastic spring is attached to that node for each prescribed degree of freedom. The first three constraints: X-constraint, Y-constraint and Z-constraint are the prescriptions for the three displacements. If any is set, a value must be given that represents the stiffness of that spring. The last three constraints: theta-X-constraint, theta-Y-constraint, theta-Z-constraint, are prescriptions for the three rotations

This condition can be applied to either:

- Points: Units for the stiffness in IS are: Newton/meter and Newton · meter/radians
- Beam lines: Units for the stiffness in IS are: Newton/meter² and Newton·m/(radians·m)
- Surface shells: Units for the stiffness in IS are: Newton/meter³ and Newton/(radians·m)
- Surfaces that are contour of volumes: Units for the stiffness in IS are: Newton/meter³ and Newton·m/(radians·m²)

It is possible to use a combination of normal and elastic constraints for the same point. The only condition is that every degree of freedom must have prescribed only a displacement or an elastic movement.

This constraint can be used in the analysis of foundations and interactions with the ground and terrain.

4.6.3. Connections

Conds. & Init. Data > Connections

As defined in last section, constraints are the restrictions applied externally to the model in order to avoid some movements or prescribe some of these movements. We define connections as a way to change the relative movements between different parts of the model.

By default, all the elements and parts of the model are attached together as completely rigid. This condition is used to disconnect some degrees in one or several nodes between different parts of the structure. In this way, it is easy to define Rotules, that permit free rotation between several parts of the structure.

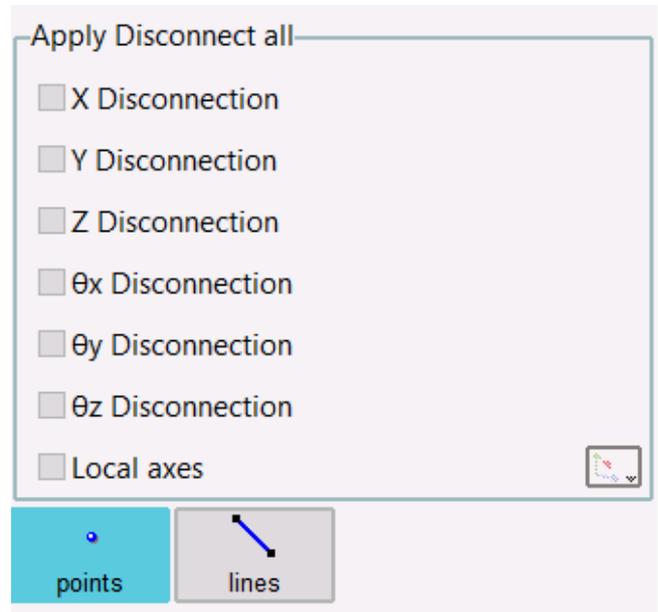
The disconnection of several degrees of freedom can be made in several ways. The most simple ones are those known as **rotules**.

Disconnect All

This condition defines a group, identified by a name, that work together as a part. This group will have the marked degrees disconnected of the rest of beams or shell elements that share the same node.

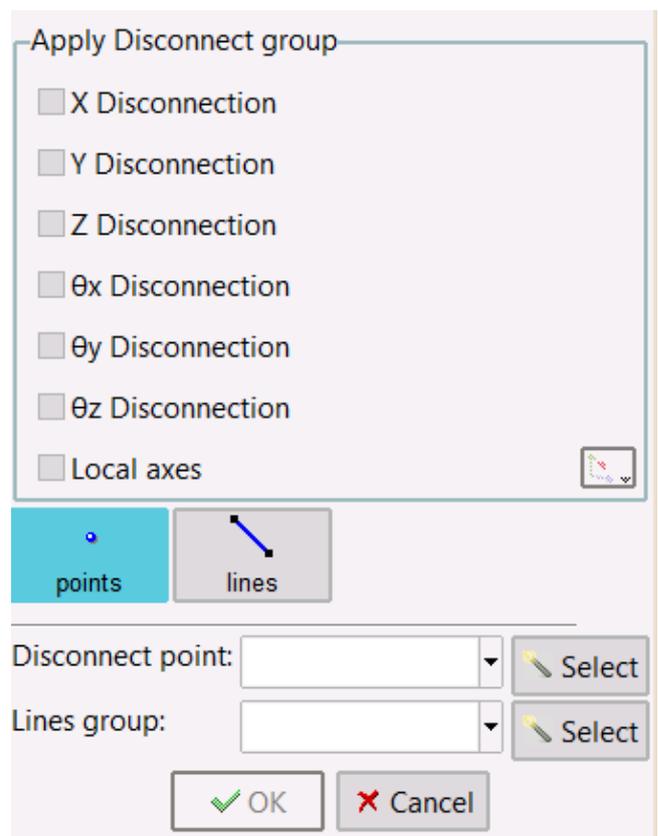
The elements that belong to the group must be marked with condition **Disconnect Group**.

If local axes are defined, the degrees disconnected are related to that local axes. If not, they are related to the global axes.

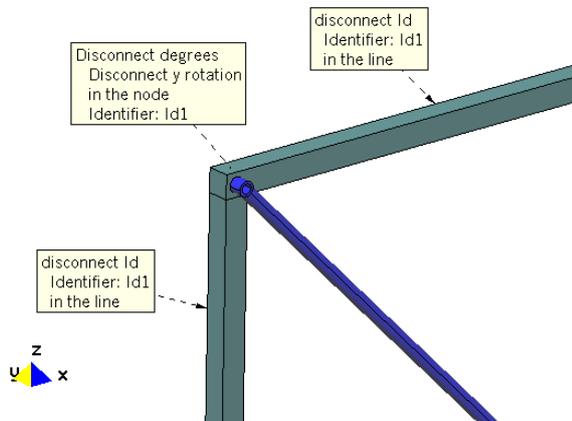


Disconnect Group

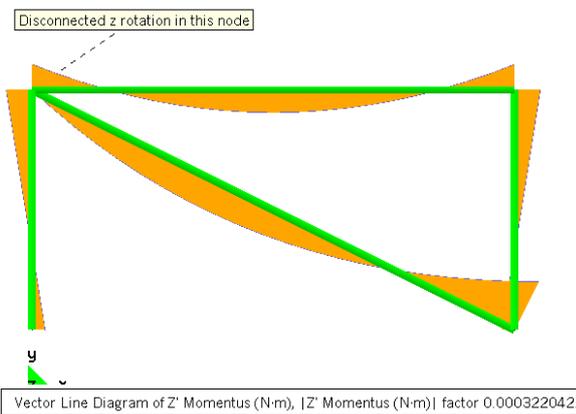
This condition is used in collaboration with the condition **Disconnect All** and is used to mark the elements that belong to a common part with a common **Group**.



Disconnection examples: beams

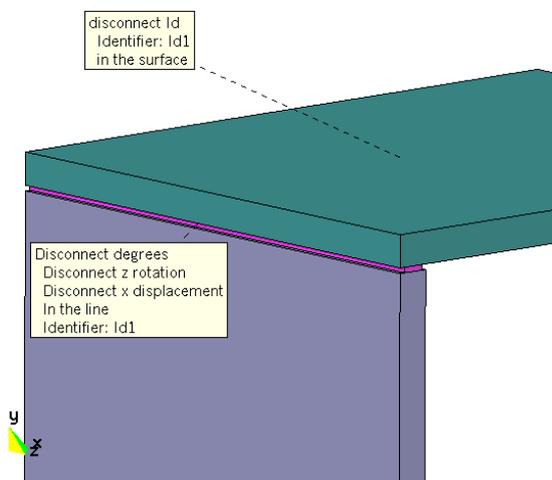


In this example, one part, called Id1, is defined with the two bars that have a rigid connection between them. Degree **y rotation** is disconnected from the rest of the bars. So, the third bar has free rotation related to these two.



Typical results in momentum for that node.

Disconnection examples: shells



In this example, one part, called Id1, is defined with all the elements that belong to one of the two surfaces. Degree **z rotation** and **x displacement** are disconnected for all the

elements that belong to one of the surfaces and contain one node or 2 on the connection line. So, the elements of the other surface have free rotation related to the elements of this surface

Virtual connections

It creates a rigid connection among a point and other geometric entities as points, lines or surfaces.

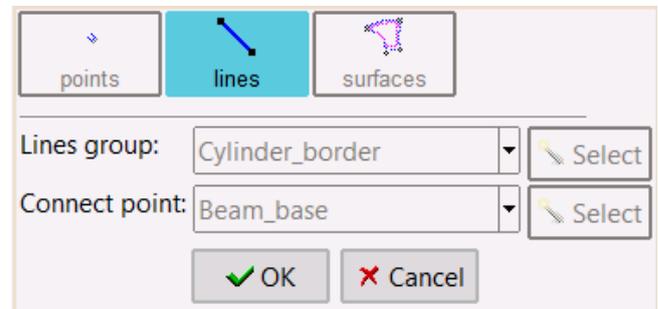
The following option must be selected to apply a virtual connection:

Constraints ▶ **Connections** ▶ **Virtual connections**

This option is available only if beams or shells are selected in

General data ▶ **Analysis** ▶ **Element types**

To define the connection, it is necessary to enter a point in Connect point and a group of points, lines or surfaces depending on the pressed button.



4.7. Contacts

Requisits: structural contacts are only available once either "Dynamics" or "Incremental loads" analysis options is selected

General data ▶ **Analysis** ▶ **Analysis type** ▶ **Dynamics Analysis**

General data ▶ **Analysis** ▶ **Analysis type** ▶ **Incremental Loads Analysis**

Ramseries has two type of contacts.

- **Master-Slave contacts Surf-Surf:** it is a contact between nodes and elements on different surfaces that can contact during the analysis. It is necessary to provide separately the 'Master' and the 'Slave' surfaces.
- **Master-Slave contacts Surf-Line:** it is a contact between nodes and elements on different surfaces and line entities that can get in contact during the analysis. It is necessary to provide separately the entities (surfaces and lines) pertaining to either the 'Master' and the 'Slave' groups.

In both cases, the type of contact can be sticking or frictionless.

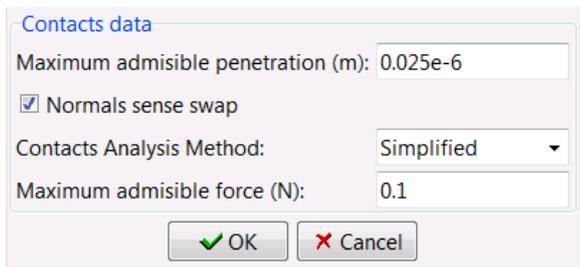
4.7.1. Master-Slave contacts

Information for analyses with contacts among elements.

Advanced contacts data

Advanced contacts data is available in

Data ▶ **Structural general data** ▶ **Advanced** ▶ **Contacts data**



Maximum admissible penetration

When iteration routine is executed, if there are slave nodes with penetration minor than maximum admissible, new contacts will not be created.

Normals sense swap

If it is set, Ramseries will swap master surfaces's normals to point to slave nodes.

Contacts Analysis Method

There are two methods:

- **Full:** This option is the most efficient when bodies are hard. If they are soft, analysis may not converge.
- **Simplified:** This option improves the convergence, but needs more calculation time.

Maximum admissible force

When iteration routine is executed, if there are contacts with traction force in the slave node minor than maximum admissible, these contacts will not be disconnected.

Analysis configuration

A contact analysis is non-linear. Below is show to setup a static or dynamic analyses with contacts.

- **Analysis:**
 - **Analysis type:** for static analyses **Incremental Loads Analysis** must be selected and for dynamics **Dynamic Analysis**.

It is necessary to activate one type of non-linear calculation **Plasticity Materials, Non-linear geometry of Non-Linear boundary conditions**.

- **Non-Linear analysis data:**
 - Solver control: **Arc-length control** is not available for analyses with contacts.
 - Advanced parameters of Non-Linear analysis data as **Line-Search, Automatic increment, Auto ARC switch** are not available for analyses with contacts.

For dynamic analyses with contacts it is recommended to use **Energy Conserving** as **Integration method**.

Master-Slave contacts Surf-Line

This option is used to perform a Contact Analysis among a surface and a line (Shell - Beam, Solid - Beam).

Contact options are shown in the interface if a type of non-linear analysis, dynamic or static, is selected.



Two types of contacts are available:

- **Sticking contact:** tangential relative motion between surface and beam is not allowed.
- **Frictionless contact:** tangential relative motion between surface and beam is allowed without friction.

The contact domain must be defined selecting the surface and the beam where there will be contact. It is possible to choose shell's surfaces or solid's surfaces.

The surface must be assigned as **master** and the beam must be assigned as **slave**.

It is necessary that master surfaces's normals point to slave lines for detecting contacts correctly. If master surface and slave line are coincident or separated by some distance, and **Normals sense swap** option is activated (see advanced contacts data), Ramseries swaps master surfaces's normals.

Contact's groups

It is possible to define different pairs of master-slaves.

Master-Slave contacts Surf-Surf

This option is used to perform a Contact Analysis among two surfaces (Shell - Shell, Solid - Solid or Shell - Solid).

Contact options are shown in the interface if a type of non-linear analysis, dynamic or static, is selected.



Two types of contacts are available:

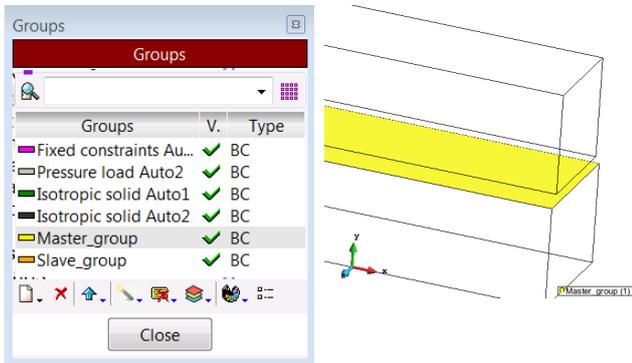
- **Sticking contact:** tangential relative motion between surfaces is not allowed.
- **Frictionless contact:** tangential relative motion between surfaces is allowed without friction.

The contact domain must be defined selecting the pair of surfaces where there will be contact. It is possible to choose shell's surfaces or solid's surfaces.

One surface will be assigned as **master** and the other as **slave**. It is recommended to give different names to the master group and slave group.



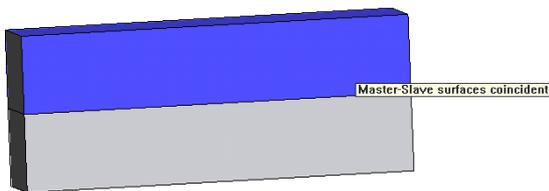
Groups window can be used to draw master and slave groups.



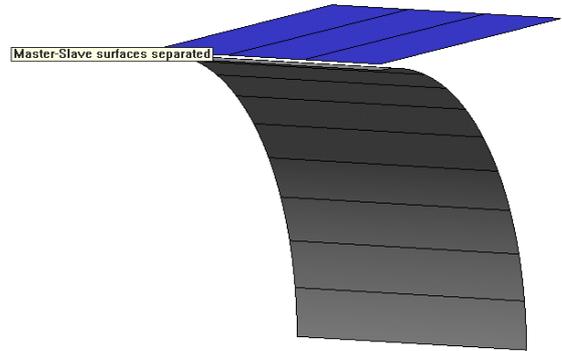
There is not a fix rule about which surface must be **master** and which must be **slave**. However, it is advisable to choose as master surface the more rigid. On the other hand, stresses in slave surface will be more accurate than in master slave, so choosing as slave the surface where a better accuracy is requested, is a good option.

It is necessary that master surfaces's normals point to slave surfaces for detecting contacts correctly. If master and slave surfaces are coincident or separated by some distance, and **Normals sense swap** option is activated (see advanced contacts data), Ramseries swaps master surface's normals.

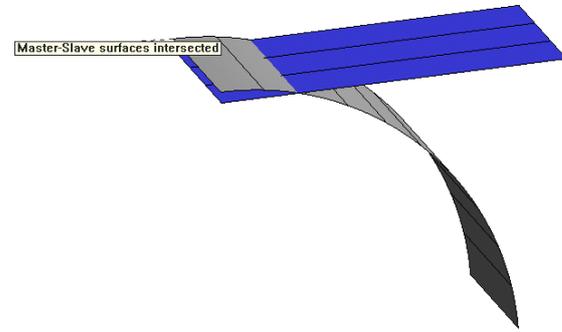
Example of coincident master and slave surfaces.



Example of separated master and slave surfaces:



Example of intersected master and slave surfaces:



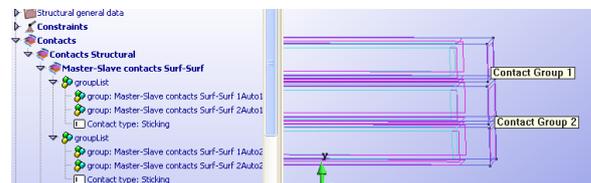
This case is not allowed because slave nodes penetrate into master elements.

Fix constraint in slave nodes

If a slave node contacts with a master element but this slave node has defined a fix constraint, the contact will not be created. Therefore, it is advisable to define elastic constraints instead of fix constraints in slave nodes. If the stiffness value of the elastic constraint is 10^3 times greater than the material's stiffness, it will work than a fix constraint.

Contact's groups

It is possible to define different pairs of master-slaves.



However, all groups must be the same type of contact, sticking or frictionless.

Mesh definition recommendations

For improving stress accuracy in contacts elements, it is recommended to use smaller elements in slave surface than master surface, because doing so, every master element will have more than slave node in contact.

Moreover, master elements normals must be coherent. They can be swapped manually using tools in

Utilities ► Swap normals

If analysis is run and master elements normals are not coherent, a warning message appears and contacts may not be detected correctly.

Process information

It is possible to know valuable information about contacts during calculation process.

For viewing process info, the icon shown below must be selected:

Calculate ► View process info...



View process info

Following, an extract of process info is shown:

```
Increment 10

Starting solution process
Total Hybrid-Sparse solver time = 0.014 min. = 0.83 seg.
Current stiffness factor= 0.777783
Total LD factor= 1, Inc factor=0.1 and Inc Length= 1.22816
Number of contacts added: 12
Number of contacts edited: 5
Number of contacts removed: 1
Number of active contacts before iterations: 33
Iteration 1: .... Load convergence factor=3.04516
Iteration 2: .... Load convergence factor=0.00313476
Iteration 3: .... Load convergence factor=6.85433e-007
Maximum traction force=1.20575e+006 in node=525
Number of disconnected contacts: 3
Iteration 4: .... Load convergence factor=0.257102
Iteration 5: .... Load convergence factor=7.48152e-005
Iteration 6: .... Load convergence factor=6.11272e-009
Maximum traction force=443600 in node=424
Number of disconnected contacts: 1
Iteration 7: .... Load convergence factor=0.070735
Iteration 8: .... Load convergence factor=3.5208e-005
Iteration 9: .... Load convergence factor=1.47341e-009
Maximum traction force=146419 in node=448
Number of disconnected contacts: 1
Iteration 10: .... Load convergence factor=0.0225059
Iteration 11: .... Load convergence factor=6.00846e-006
Number of active contacts after iterations: 28

Highest penetration: -8.89911e-016. Node: 513
```

At the beginning of convergence bucle, the number of contacts added, edited or removed are shown. A contact is edited when the slave node changes its master element. A contact is removed when is not inside any master element, and a contact is disconnected when traction force is positive.

When **Convergence factor** is minor than **convergence tolerance** (See Non-Linear Analysis Data), all contacts with traction force are disconnected and all contacts are added when slave nodes penetrates a master element. Only if the highest penetration is higher than **Maximum admissible penetration** (See Advanced contacts data), new contacts will be created. The process info shows the number of contacts disconnected or added, as well as the highest penetration and traction force.

4.8. Dynamic conditions

4.8.1. Initial Conditions

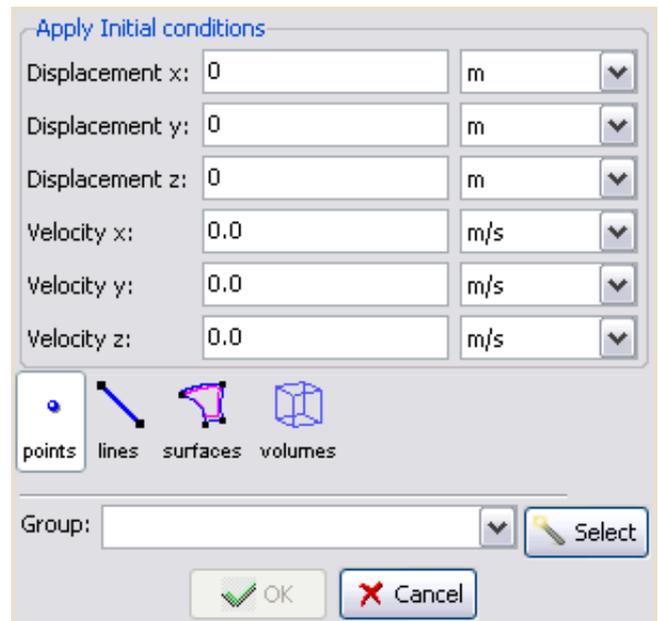
In the Linear-Dynamics and Non-Linear Dynamics analysis the Initial Conditions of the structure must be defined. These initial conditions can be applied for every degree of freedom (X, Y, Z and rotations for beams and shells), and for every node in the model

Default units are meters for the X, Y and Z displacements and radians for the prescribed rotations. X Constraint, Y Constraint and Z Constraint mean the displacements along the axes. Theta

x Constraint, theta y constraints and theta z constraints mean the rotations around the axes.

This condition can be applied to either points, lines or in the solid analysis, to surfaces.

Note: 3D solids have only three degrees of freedom: displacements in X, Y and Z.

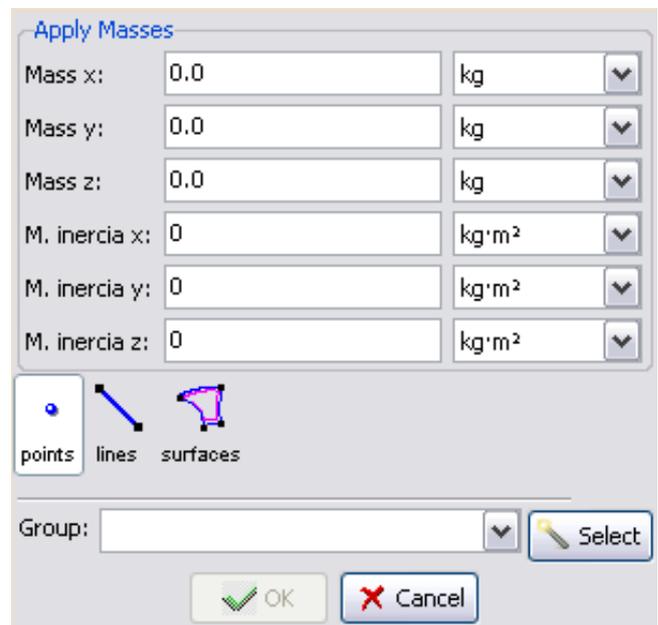


4.8.2. Masses

When performing a dynamic analysis, the mass of the structure has to be specified.

Point mass

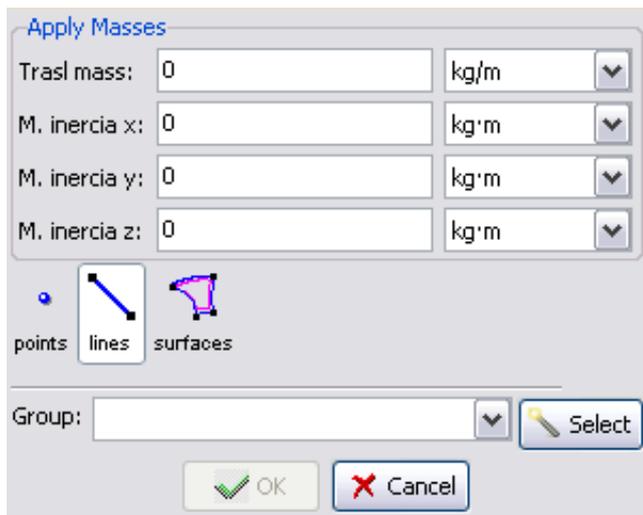
This is a mass applied to one point of the structure. Default units are kilogram for the translation mass components and kilogram by square meter for the rotation mass components.



Beam mass

A beam mass is specified when considering the distributed mass of a beam.

Default units are kilogram/meter for translation mass and kilogram.meter for rotational mass.

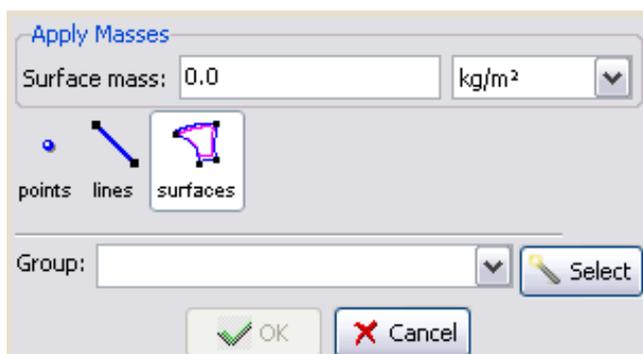


Note: if this condition is applied over a line which is contour of a shell but has not got beam properties assigned, it will automatically be considered as a "Face-shell mass".

Surface shell mass

This is a mass applied to a shell element of the structure.

Default units are kilogram/ square meters.



4.9. Materials and properties

These properties define both the material parameters and the sections, shells, solids, membranes and cables parameters.

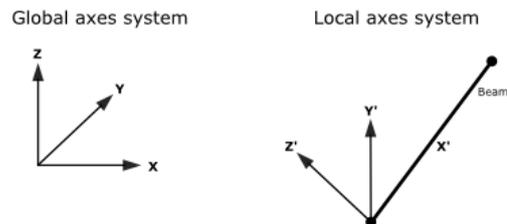
4.9.1. Global and Local axes for beams

The model is created in reference to a global axes system XYZ that is unique for the entire problem. However, every beam must have its own local axes system X'Y'Z' in order to:

1. Refer section properties like *Inertia modulus* or *thickness* and *height* to this system.
2. Some of the loads (the Local loads that have the prefix *Local*) are specified in relation to this coordinate system. related also to this system.

The resulting forces in the beams refer to this system. Strength results over the beams are referred to this local axes system.

The main property of this system is that the local X' axis must have the same direction as the beam.



The ways for defining local axes systems are:

1. **Default.** The program assigns a different local axes system to every beam with the following criteria:
 - X' axis has the direction of the beam.
 - If X' axis has the same direction than global Z axis, Y' axis has the same direction than global X. If not, Y' axis is calculated so as to be horizontal (orthogonal to X' and Z).
 - Z' axis is the cross product of X' axis and Y' axis. It will try to point to the same sense than global Z (dot product of Z and Z' axes will be positive or zero).

Note: The intuitive idea is that vertical beams have the Y' axis in the direction of global X. All the other beams have the Y' axis horizontal and with the Z' axis pointing up.
2. **Automatic.** Similar to the previous one but the local axes system is assigned automatically to the beam by GiD. The final orientation can be checked with the *Draw Local Axes* option in the GiD Conditions window.
3. **Automatic alt.** Similar to the previous one but an alternative proposal of local axes is given. Typically, User should assign Automatic local axes and check them, after assigning, with the *Draw local axes* option. If a different local axes system is desired, normally rotated 90 degrees from the first one, then it is only necessary to assign again the same condition to the entities with the **Automatic alt** option selected.
4. **User defined.** User can create different named local axes systems with the GiD command:

Data ► Local axes ► Open

and with the different methods that can be chosen there. The names of the defined local axes will be added to the menu where Local axes are chosen.

Note 1: RamSeries tries to correct the local axes system if the local X' axis does not point to the direction of the beam. It will fail if local X' axis is orthogonal to the direction of the beam.

Note 2: The final local axes system for every beam can be visualized in the post-process stage. It is convenient to check the correctness of these systems after calculation is performed.

4.9.2. Global and Local axes for shells

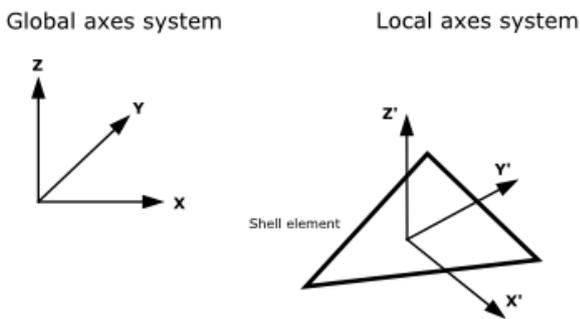
The model is created related to a global axes system XYZ that is unique for the entire problem. But every shell element must have its own local axes system X'Y'Z' in order to:

1. Refer section properties like orthotropy properties to this

system.

- Some of the loads (that have the prefix *Local*) are related also to this system.
- Strength results over the shell are referred to this local axes system.
- Dimension of steel in shells is based on X' and Y' directions.

The main property of this local axes system is that the local Z' axe must have the same direction than the normal of the element.



The ways for defining local axes systems are:

- Default.** The program assigns a different local axes system to the shell with the following criteria:
 - Be **N** the unitary normal of the shell element, **U** the vector (0,1,0) and **V** the vector (0,0,1). Then:
 - Z' axe has the direction and sense of **N**.
 - If $N_x < 1/64$ and $N_y < 1/64$, then X' axe will be in the direction of the cross product of **U** and **N** (**UxN**).
 - If not, X' axe will be in the direction of the cross product of **V** and **N** (**VxN**).
 - Y' axe will be the cross product of Z' axe and X' axe.

Note: Intuitively, this local axes system is calculated so as if element is approximately contained in the plane XY, local X' axe will point towards global X axe. If not, this X' axe is obtained as orthogonal to global Z axe and local Z' axe.
- Automatic.** Similar to the previous one but the local axes system is assigned automatically to the shell by GiD. The final orientation can be checked with the *Draw Local Axes* option in the GiD Conditions window.
- Automatic alt.** Similar to the previous one but an alternative proposal of local axes is given. Typically, User should assign Automatic local axes and check them, after assigning, with the *Draw local axes* option. If a different local axes system is desired, normally rotated 90 degrees from the first one, then it is only necessary to assign again the same condition to the entities with the **Automatic alt** option selected.
- User defined.** User can created different named local axes systems with the GiD command:

Data ► Local axes ► Open

and with the different methods that can be chosen there. The names of the defined local axes will be added to the menu where Local axes are chosen in the conditions window.

Note 1: RamSeries tries to correct the local axes system if the local Z' axe does not point to the direction of the normal of the

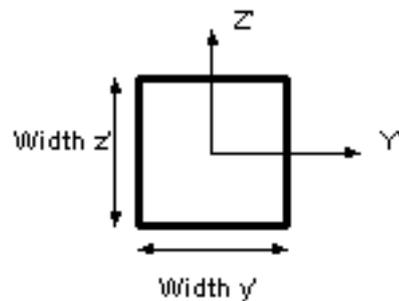
element. It will fail if local Z' axe is orthogonal to the direction of the normal.

Note 2: The final local axes system for every shell element can be visualized in the post-process stage. It is convenient to check the correctness of these systems after calculation is performed.

4.9.3. Beams

Rectangular Section

This condition is assigned to beams with its transversal section rectangular.



The **Width y** is the width of the rectangular section following the Y' direction. **Width z** is the width in the Z' direction. G is the torsion modulus (8.1e10 N/m² for steel) and E is the Young modulus (2.1e11 N/m² for steel).

Note: Remember that for an isotropic material: $G = \frac{E}{2 \cdot (1 + \nu)}$ (for steel $\nu=0.3$)

- The Specific Weight (different from density). The weight is considered in the direction of the gravity, which will be defined later. Its default units are N/m³.

Once the values are filled in, the condition must be assigned to the beams.

Generic Section

The definition of the properties for a general beam section is very similar to that of the rectangular section. Instead of giving the width and height of the section, the area (A), the Torque modulus (J) and the inertia modulus for the Y' (Inertia y) and Z' axes (Inertia z) of the beam are given. Default units for the area are m² and for the inertia modulus are m⁴.

It is also possible to automatically calculate the properties of predefined sections (*Rectangular Solid, Trapezoidal Solid, Circular Solid, SemiCircular Solid, Tube, Double T*), inserting the desired dimensions in the window which pops-up after pressing the "Predefined Sections" button.

Sections Library

There is a wide library available in RamSeries for defining standard steel sections.

If the button **Edit** is pressed, the sections library window is displayed, and it is possible to choose the desired section and check the geometrical properties.

Naval Stiffeners

In RamSeries, it is possible to assign properties of fiber reinforced plastic stiffeners (very common in naval motorcraft or sailing plastic hulls) to beams.

After pressing the button "Edit", the following window will show up:

It will be possible to define the section of the core, and its mechanical properties in the following way,

After defining the properties of all the compounds to be used:

Materials and properties ► Materials ► Composite

it is necessary to introduce the different laminates, which will be used. This can be done in the Composite naval stiffeners window (clicking on the button called "Laminates creator"), or in the

Materials and properties ► Materials ► Laminate

item in the date tree.

Damage Assesment

If Naval (or yacht) analysis is activated, and for Dynamic Analyses, it is possible to perform a Damage Assesment based on the FEA stress results (see Appendix 10: Fatigue Damage Assesment).

It is necessary to activate it in:

General data ► Analysis ► Damage assesment

Furthermore, predefined S-N curves may be inserted from a file, choosing

General data ► Analysis ► SN curves file

Ask Compass IS for a sample file regarding this issue.

Beams Damage Assesment:

By assigning this condition to any line of the geometry, the user is indicating the program that it is a welded connection, and that it must be verified for fatigue damage.

If predefined S-N curves have been inserted through a valid file as mentioned above, they will appear automatically by clicking

on the [SN curves] button.

4.9.4. Cables

Cable section

4.9.5. Shells

Isotropic Shell

The definition of an elastic shell requires the specification of material properties and the actual thickness of the shell. In the isotropic case, the specification of material properties simply consists on providing the Young's modulus, the Poisson's coefficient and the specific weight of the material. This can be done by either providing user defined values (as seen in the figure below) or selecting a pre-defined material from those available in the drop-down list. In addition, the actual thickness of the shell must be provided since this is not an intrinsic material's parameter but a physical property of the structural element to which the shell is going to be applied.

Note that for an isotropic material the shear modulus is already

given once the Young's modulus and the Poisson coefficient have been specified:

$$G = \frac{E}{2(1 + \nu)}$$

Once all the required properties have been introduced, the shell's definition can be assigned to the desired group of surface geometrical entities in the CAD model.

Orthotropic Shell

Similarly to the isotropic case, orthotropic shell's definition requires the user to provide the actual thickness of the shell and the properties of the constituent material. In the orthotropic case, material properties are specified by selecting a pre-defined orthotropic material from the drop-down list. The actual thickness of the shell must be provided since this is not an intrinsic material's property but a geometrical parameter of the structural element to which the shell is going to be applied. In addition, the orthotropy axes must be defined using the *Local axes* option in the dialog box shown below. If local axes are not specified, the orthotropy axes are set by default to coincide with the global axes.

New orthotropic materials can be created in the following section of the data tree:

Materials and properties ► Physical properties ► Orthotropic

The properties required to define an orthotropic material are shown in the dialog box in the figure below. These consist on the longitudinal and transverse Young's modulus (E_x, E_y), the major Poisson's ratio (ν_{xy}) and the shear modulus (G_{xy}, G_{xz}, G_{yz}). Note that all these properties are referred to the orthotropy axes of the material. The orientation of these material axes with respect to the global axes is given by the definition of the shell's local axes.

Note that to maintain the elasticity hypothesis and due to the fact that the stress and strain tensors are symmetric, the following relation must be fulfilled:

$$E_x \cdot \nu_{yx} = E_y \cdot \nu_{xy}$$

Hence, only the major Poisson's ratio needs to be specified since the minor Poisson's ratio is internally evaluated in accordance to the relation above. Note that the transverse Young's modulus E_z has no effect if this material is used to define an orthotropic shell. E_z is only taken into account when the orthotropic material is assigned to a solid. The same argument applies to Poisson's coefficients different from the major Poisson's ratio ν_{xy} . Additional Poisson coefficients can be defined in the form of a 3x3 matrix (see the image below) but have only effect when the corresponding orthotropic material is assigned to a generic 3D solid.

Default units for E and G are N/m^2 and ν is non-dimensional. Also note that for an orthotropic material the following relation should be satisfied.

$$\frac{1}{G_{xy}} = \frac{1 + \nu_{yx}}{E_x} + \frac{1 + \nu_{xy}}{E_y}$$

Once all the required properties have been introduced, the shell's definition can be assigned to the desired group of surface geometrical entities in the CAD model.

Laminate Shell

As well as for the case of "Naval stiffeners", it is possible to use in RamSeries laminated fiber reinforced polymer shells. To this aim, the user must simply select a pre-defined laminate material from the drop-down list and apply it to the desired group of surface geometrical entities in the CAD model.

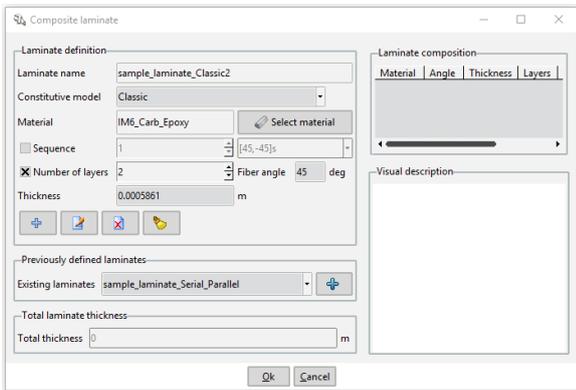
By clicking on the "Edit Laminate" button (at the right side of the material's entry), a list of the available laminates is displayed in a separate window (as seen in the figure below) thus allowing the selection and the interactive edition of existing laminates, and even the creation of new ones.

As stated above, during the definition of a laminated shell group a previously defined laminate material must be selected. RamSeries graphic user interface provides few sample

laminates but new ones can be created in the following section of the data tree:

Materials and properties ► Physical properties ► Laminate

When creating a new laminate the 'Composite laminate' window is used (see the figure below):



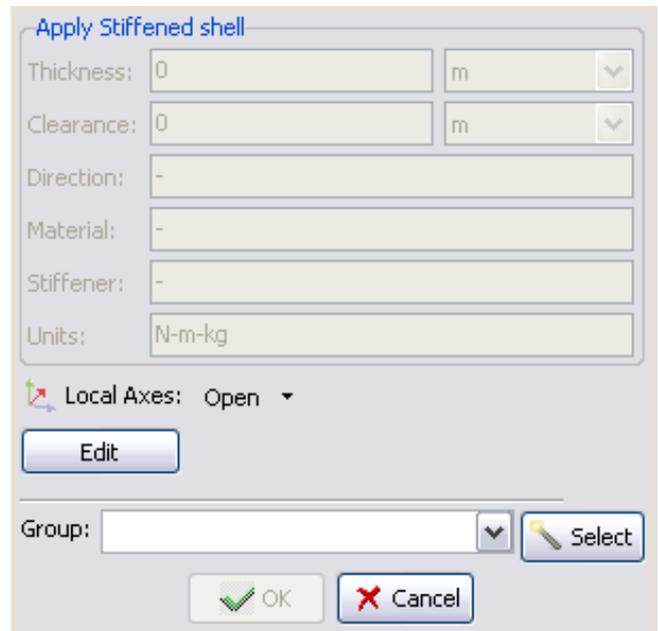
The steps that must be followed to define a laminate can be summarized as follows:

- The first step is to provide an identifier name for the new laminate
- Next, the laminate constitutive model must be selected. This corresponds to the actual formulation RamSeries will use internally for the calculation of the corresponding laminate. Currently, three different laminate constitutive models are provided:
 - **Classic:** it corresponds to the classical lamination theory (CLT) described in the RamSeries theory manual. When the 'Classic' constitutive model is selected, only orthotropic materials can be selected to be used in the definition of the various plies of the laminate.
 - **Heterogeneous:** it corresponds to the heterogeneous rule of mixtures (RoM) formulation described also in the RamSeries theory manual. When the 'Heterogeneous' constitutive model is selected, only materials defined as composite layers can be selected to be used in the definition of the various plies of the laminate.
 - **Serial-Parallel:** it corresponds to the Serial-Parallel rule of mixtures (SP-RoM) formulation also described in the RamSeries theory manual. As in the case of the 'Heterogeneous' constitutive model, when the 'Serial-Parallel' model is selected, only materials defined as composite layers can be selected to be used in the definition of the various plies of the laminate.
- Next, the laminate stacking sequence must be specified by selecting the material, the ply thickness and the fibre orientation. Layers can be introduced individually or as a predefined sequence. In any case, the thickness of each individual layer must be provided. When using a predefined sequence, all the layers within the sequence will have the same thickness. On the other hand, the orientation of each layer within the sequence is given by the standard sequence orientation notation. Finally, pre-defined existing laminates can be used as building blocks of more complex laminates. The total thickness of the laminate is computed and visualized for the sake of guidance. A graphic representation of the laminate is drawn as well.

Stiffened Shell

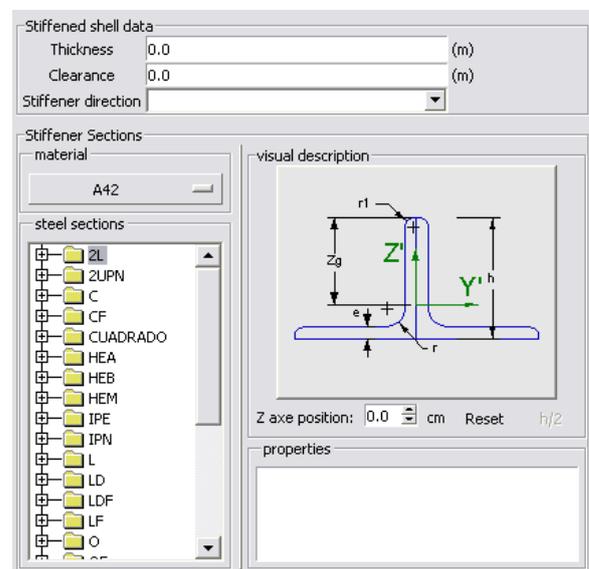
It is possible in RamSeries to define a shell which will have the

properties of a reinforced plate with stiffeners welded onto it.



For defining the properties of this plate, the data necessary is: the plate **Thickness**, the distance between stiffeners (**Clearance**), the **Stiffener Direction** (please take special care here with the local axes of the shell), and the type of stiffener section to use.

Also, it is possible to choose the steel type (A37, A42 or A52).



Plasticity Shell

Damage Assessment

If Naval (or yacht) analysis is activated, and for Dynamic Analyses, it is possible to perform a Damage Assessment based on the FEA stress results (see [Appendix 10: Fatigue Damage Assessment](#)).

It is necessary to activate it in

General data ► Analysis ► Damage assessment

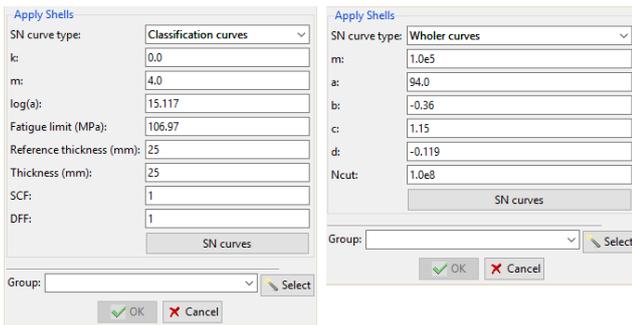
Furthermore, predefined S-N curves may be inserted from a file, choosing

General data ► Analysis ► SN curves file

Ask Compass IS for a sample file regarding this issue.

Shells Damage Assesment:

By assigning this condition to any surface of the geometry, the user is indicating the program that it must be verified for fatigue damage. The user can choose between the Standard Classification SN curves, or the Wholer curves.



If predefined S-N curves have been inserted through a valid file as mentioned above, they will appear automatically by clicking on the **[SN curves]** button.

4.9.6. Membranes

Isotropic membrane

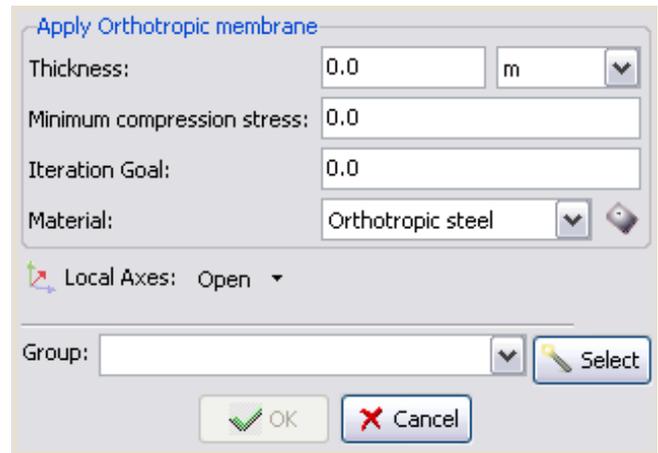


Minimum compression stress: a membrane does not resist compression loads, but it is wrinkled. When compression stress is higher than this limit, wrinkling algorithm, that manages the wrinkling process, is activated.

Iteration goal: it is the limit of iterations where the wrinkling

algorithm is executed. If iteration goal is 0 membrane will be considered as taut and wrinkling algorithm will not be executed. On the other hand, if iteration goal is higher than the maximum of iterations, wrinkling algorithm will be executed in every iteration.

Orthotropic membrane



4.9.7. Solids

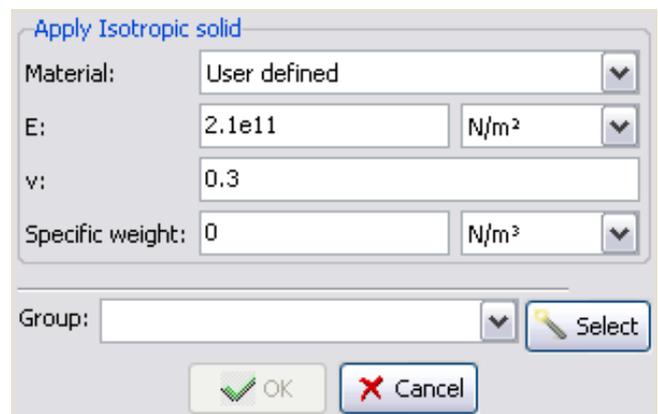
Isotropic Solid

This condition is assigned to solid volumes.

Properties to enter are:

- **Material.** To be chosen from library, or "User defined".
- **E** is the Young modulus (2.1e11 N/m² for steel) and **ν** is the Poisson coefficient with value 0.3 for steel (it is non-dimensional).
- The **Specific Weight** is the self-weight of the solid (different from density). The weight is considered in the direction of the gravity, which will be defined later. Its default units are N/m³.

Once the values are filled in, the condition must be assigned to the volume that defines the solid.



Orthotropic Solid

Properties defined in this condition are similar to that of the *Isotropic solid* condition. The difference is that the material is orthotropic. The orthotropy axes are the ones defined in the *Local axes* section. The properties to enter are the Young

Modulus (E_x, E_y), the Poisson coefficients (ν_{xy}, ν_{yx}) and the Shear modulus (G_{xy}, G_{xz}, G_{yz}). They are all referred to the local axes X and Y'.

Note: Remember that to maintain the elasticity hypothesis it is necessary to accomplish that:

$$E_x \cdot \nu_{xy} = E_y \cdot \nu_{yx}$$

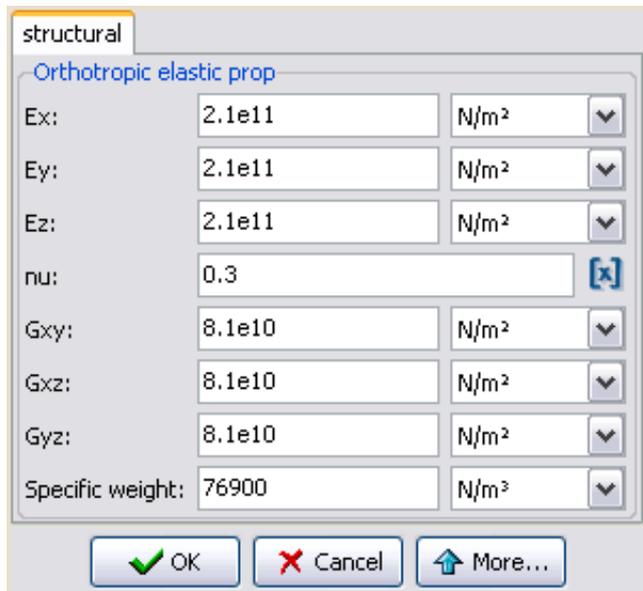
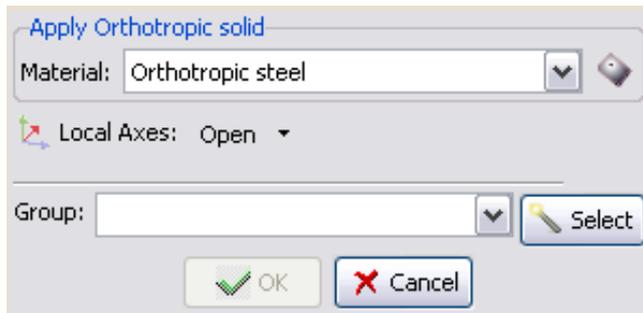
Default units for E and G are N/m² and ν is non-dimensional.

Remember that for an isotropic material:

$$G_{x'y'} = G_{x'z'} = G_{y'z'} = \frac{E}{2 \cdot (1 + \nu)}$$

For an orthotropic material:

$$\frac{1}{G_{x'y'}} = \frac{1 + \nu_{y'x'}}{E_{x'}} + \frac{1 + \nu_{x'y'}}{E_{y'}}$$



Plasticity solid

Laminate solid

4.9.8. Custom properties

4.9.9. Physical properties

There is a wide library of materials available to choose in RamSeries

Properties ► Materials

The properties of these materials can also be edited and new materials can be created.

These materials are the ones which can be chosen in the different Beams, Cables, Shells, Membranes and Solids Properties windows.

Material properties are grouped into various tabs according to their use and relevance for the various constitutive models available in RamSeries:

Structural:

- Elastic properties
 - E: Young modulus
 - G: Torsion modulus
 - ν : Poisson coefficient
 - Specific weight: weight per unit volume of the material
 - Maximum stress: maximum acceptable stress for the material
- Plasticity J2
- Tensile yield stress: initial yield limit for the onset of plasticity.
- Isotropic hardening: for isotropic hardening, the shape of the yield surface does not change during the deformation but its size increases with plastic deformation in accordance with the hardening law. Hence, the type of hardening law to be used must be specified for isotropic hardening. The available type are:
 - Linear: linear (i.e. constant) hardening law. The constant ratio between the stress increment and the strain increment within the plastic regime must be provided.
 - Exponential: a power law is used to characterize the hardening of the material. A saturation stress must be provided together with the power law exponent.
- Kinematic hardening: for kinematic hardening, neither the shape nor the size of the yield surface change during the

deformation. But the yield surface is translated in the stress space. It allows for modelling non-symmetric tension/compression behaviors such for instance the Baushinger effect in metals.

Damage: the parameters in this section concern the non-linear isotropic damage model in RamSeries.

- Sy_t: yield stress of the material
- Damage law: type of damage law to be used. It can be linear or exponential
- Linear
 - H/S Ratio: hardening/softening ratio
- Exponential
 - Gf: Fracture energy
 - S_sat: Saturation stress
- Damage norm: damage norm to be used.
- Symmetric Traction/Compression
- Traction
 - C/T Ratio: Traction/compression ratio
- Non-symmetric Traction/Compression

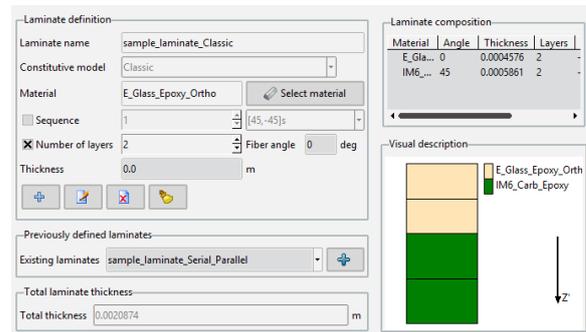
Allowable stresses: the parameters in this section are used to evaluate failure criteria when the material under consideration is used in the definition of laminated composite materials.

- Sc1: allowable compression stress in direction 1
- Sc2: allowable compression stress in direction 2
- St1: allowable tension stress in direction 1
- St2: allowable tension stress in direction 2
- T12: allowable shear stress (12)
- T23: allowable shear stress (23)
- T13: allowable shear stress (13)

Composite components: this set of parameters is only available for the definition of composite layers to be further used in the definition of laminated composite materials.

- Fibre/Matrix
- Reinforcement material: reinforcement material to be used in the current composite ply
- Matrix material: matrix material to be used in the current composite ply
- Vf(%): volume content of fiber of the current composite ply
- Monocomponent/core
- Ply material: monocomponent material to be used in the definition of a core ply

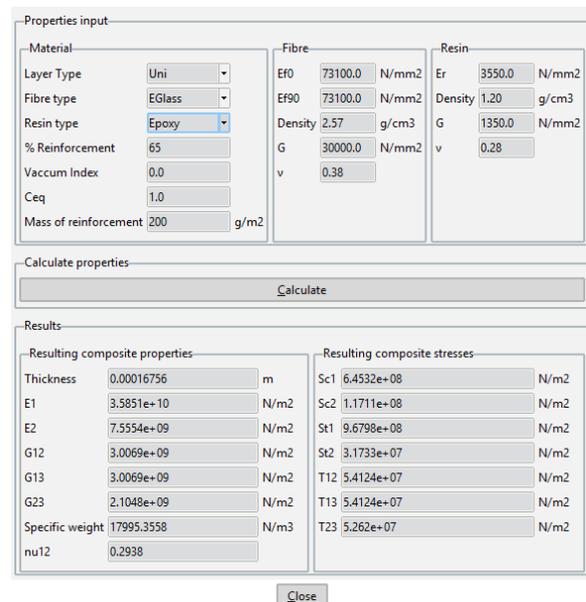
Laminates definition: in this section, laminated composites can be defined based on composite layers and/or orthotropic materials previously created within the materials' library of RamSeries. To this aim, the laminate's definition window shown in the following figure is provided to ease the creation of new laminated systems.



4.9.10. Composite materials' properties

A composite materials 'calculator' is provided in RamSeries to make easier the definition of composite material properties.

Data ► Composite materials' properties



The properties of the composite's components (fibre and resin) can be introduced separately in the calculator together with the volume fraction properties of the material. Based on these information and following the formulation in [16], the equivalent composite properties are evaluated, as well as the composite strengths to be used in the evaluation of the Tsai-Wu failure criteria.

By closing the calculator's window, a new 'composite layer' orthotropic material is automatically created as well as the corresponding FRP reinforcement (Fiber) and FRP matrix (Resin) components.

4.10. Load cases

4.10.1. Simple load cases

One load case is a group of one or more loads assigned to entities. When a new model is defined in RamSeries, a default load case, called *Load case 1*, is defined. It is possible to rename this load case by double-clicking on the Load case label; it is also possible to define new ones by using the right-click on one of the load cases labels in the tree and choosing the 'Copy' option, or right-click on the 'Loadcases' label and 'Create new Loadcase'.

When a load is assigned to one or several entities, this load is inserted inside the load case that is currently active.

If combined load cases are not defined, RamSeries calculates just one analysis that is equivalent to that of all the loads belonging to one load case.

4.10.2. Combined load cases

Combined load cases are defined in the load cases window

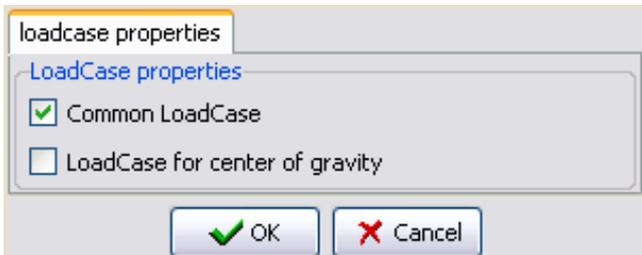
One default combined load case is already existent. To create more, press right mouse button over the existent combined load case name. A menu appears that offers several options.

One factor must be entered for every simple load case. This factor will multiply the load in order to create the combined load case. In the post-processing part, after the analysis, there will be one different result for every combined load case. Enter value order to deactivate that load case.

The field ELU does not modify the strengths result of the analysis. It is only considered in the concrete section dimension. Its meaning is:

- If it is activated, the combined load case is used for calculating the section to collapse. Typically, magnifying factors are bigger than 1.0
- If it is not activated, the combined load case is used for calculating the section in service. Typically, magnifying factors are 1.0

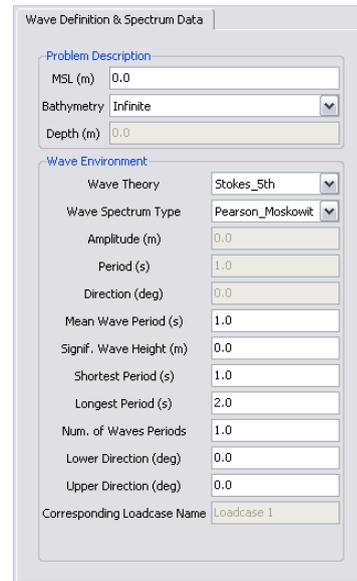
4.10.3. LoadCase Properties



This options are available by double-clicking on each Loadcase name, and indicate whether the load case is a "normal" one, used for applying loads, or if by contrary, will only be used for calculating the center of gravity, weight and other properties of the structure (inertia, total weight and radii of gyration).

4.10.4. Waves data

When Naval (or Yacht) option is activated, it is possible to define wave loads (Morison). Furthermore, it is possible to define a different set of wave characteristics for each Loadcase.



4.10.5. Static loads

When a static load is assigned to entities, it is automatically inserted in the active load case. See the load cases section for details.

Point load

This is a load applied to one point of the structure. Default units are Newtons for the force components and Newton-meter for the momentum components. Sign of the components is equal to that defined for the constraints. 3D solids loads have only three components.

Self weight load

If this condition is applied to a beam, shell element or solid element, the load due to its self-weight is applied, based on the specific weight and other parameters described in the properties.

This condition can be applied to either lines for beams, surfaces for shells and volumes for the solid analysis.

Beam Pressure load

There are three types of pressure loads for beams:

1. Global beam load
2. Global projected beam load
3. Local beam load

In all the cases the pressure applied is given in Newton/meter in default units. In the global load, the load is given related to the global axes. The global projected load is given also in global axes but the length considered of the beam is orthogonal to the load. Local load is related to the local axes defined in the properties section.

In the Local beam load there is an additional field that is the local torque momentum.

Shell Pressure load

There are five types of surface loads for shells:

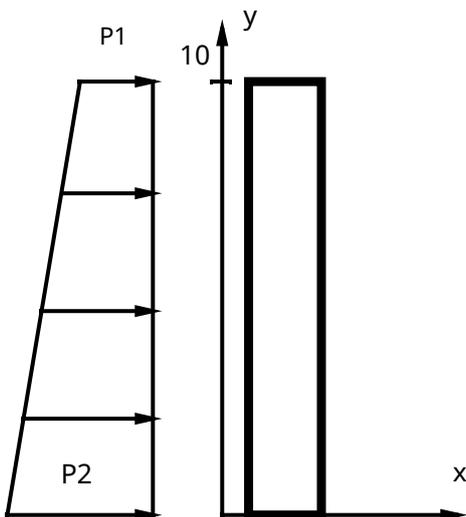
1. Global shell load
2. Global projected shell load

- Local shell load
- 4. Triangular load
- 5. Hydrostatic load

In all the cases the pressure applied is given in Newton/meter² in default units. In the **global load**, the load is given related to the global axes. The **global projected load** is given also in global axes but the area considered of the shell is orthogonal to the load. Local load is related to the local axes defined in the properties section.

Triangular load is like a Global shell load but with a triangular variation in its values. It is defined by two points, given by its coordinates, and pressure values associated to each one of these points. The pressure assigned to the elements that project between the points is a linear interpolation between the two values. The elements that project outside have a pressure value of zero.

Example:



Point 1: 0,10,0 Pressure

Hydrostatic load is defined related to the gravity direction entered in the problem data section. The reference coordinate is related to that direction. Self-weight water is given, in default units as N/m³.

Note that the **direction** of the acting pressure will be the normal to the surface on which it is applied, but the the **sense** will be the **opposite** to that of this normal to the surface.

Shell Boundary Pressure load

With this condition it is possible to apply a pressure to a line that is a boundary of a shell surface. The pressure is given in default units in N/m. If the local axes field is set to **Global**, the pressure vector is given related to the global axes. Option **Automatic** permits to define an automatic local axes system that is different for every element. Pressure vector will be related to these axes. This last option is useful to assign a local pressure to the boundary of the shell. Check:

Data ▶ Local axes ▶ Surfaces ▶ Draw

to see the automatic local axes defined.

Triangular and hydrostatic face load are defined equal to the ones in shell surface loads.

Solid Surface load

There are five types of surface loads for solids:

1. Global pressure load
2. Global projected pressure load
3. Local pressure load
4. Triangular load
5. Hydrostatic load

In all the cases the pressure applied is given in Newton/meter² in default units. In the global load, the load is given related to the global axes. The global projected load is given also in global axes but the area considered of the contour of the solid is orthogonal to the load. Local load has only one component and it is the normal pressure to the contour surface. Its value is positive when the pressure points inwards the volume.

Triangular and hydrostatic loads are defined equal to the ones in shell surface loads.

Thermal strain load

Gravity: 0,-1,0 Ref coord
It is possible to define for every beam, shell or solid, a load due to changes in temperature. The values to enter are:

- **Alpha** (α): The constant expressed in 1/°C (Celsius degrees)
- **Delta T** (ΔT): Temperature increment in Celsius degrees.

The deformation added to the beam, shell or solid, is: $\epsilon = \alpha \cdot \Delta T$

Strain load

It is possible to define for every beam, shell or solid, a load due to linear changes in certain magnitude. The values to enter are:

- **Alpha** (α): The constant expressed in 1/ ΔU
- **Delta U** (ΔU): Magnitude increment.

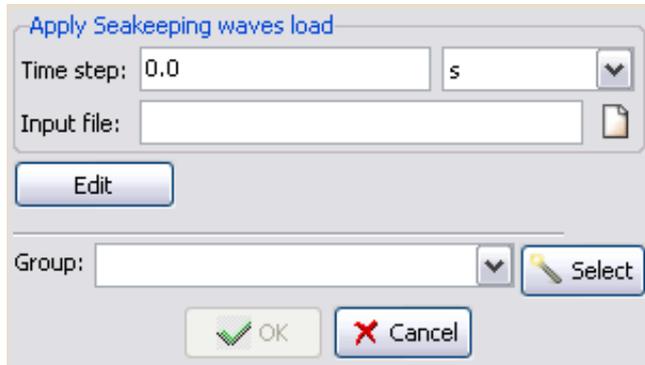
The deformation added to the beam, shell or solid is: $\epsilon = \alpha \cdot \Delta U$

SeaFEM waves load (static)

Allows to use as a pressure load the result of a previously

calculated SeaFEM analysis (**Input file**), at a certain **Time step**.

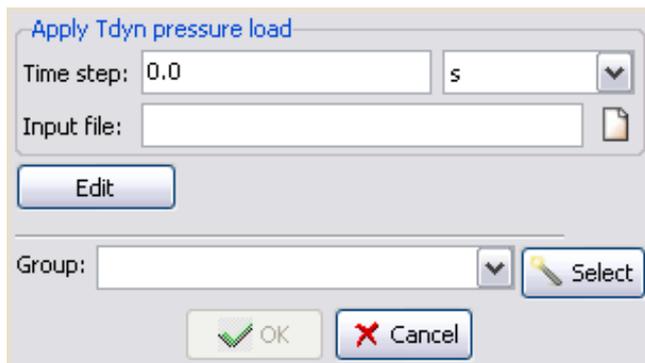
This load applies over shell elements.



Tdyn pressure load (static)

Allows to use as a pressure load the result of a previously calculated Tdyn analysis (**Input file**), at a certain **Time step**.

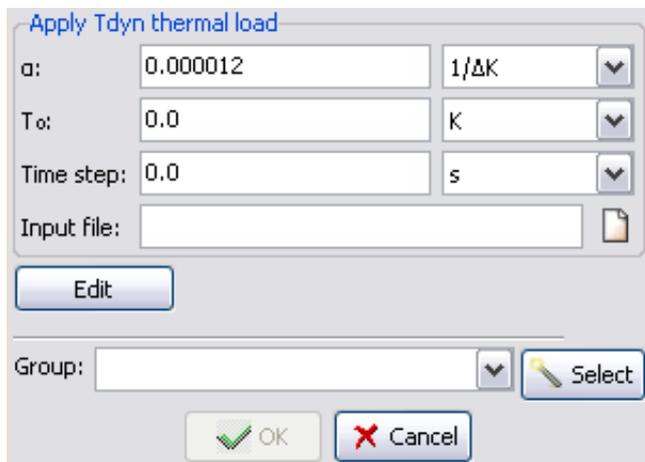
This load applies over shell elements.



Tdyn thermal load (static)

Allows to use the result of a previously calculated Tdyn heat transfer analysis (**Input file**), at a certain **Time step**, to create a thermo-mechanical strain.

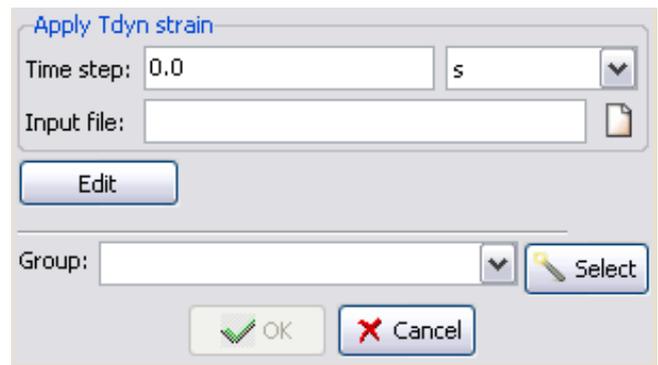
This load applies over solid elements.



Tdyn strain (static)

Allows to use the result of a previously calculated Tdyn URSOLVER analysis (**Input file**), at a certain **Time step**, to create a mechanical strain.

This load applies over solid elements.

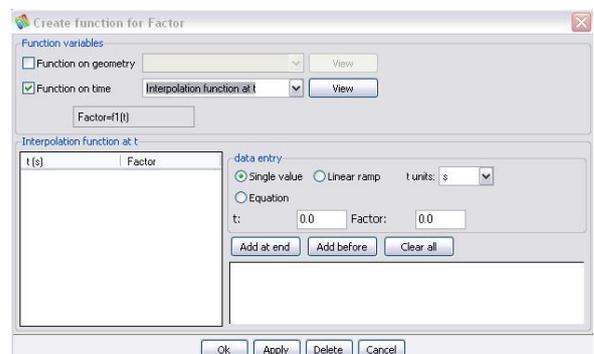
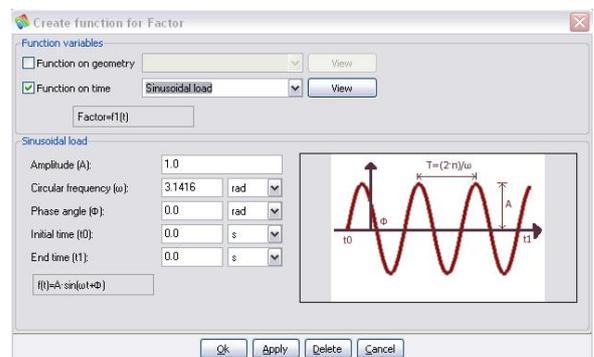


4.10.6. Dynamic loads

When a dynamic load is assigned to an entity (point., line or area), it is automatically added to the active load case inserted in the active load case. See the load cases section for details. Each type of dynamic load has loads have the following options:

Amplitudes: Indicate the amplitude of the dynamic loads.

Parameters: Define the time variation of the dynamic loads variation in time of the dynamic loads.



The time variation can be defined using harmonic variations (Sine/Cosine load) or by specifying discrete values in form of a table (Table).

Dynamic Point load

This is a load applied to one point of the structure. The default unit is Newton. Default units are Newtons for the force components and Newton-meter for the momentum components. The signs of the components follow the rule established for the constraints. Sign of the components is equal to that defined for the constraints.

Beam pressure load

There are two types of pressure loads for beams:

1. Dynamic global beam load
2. Dynamic local beam load

In all the cases the applied pressure is by default specified in Newton/meter. pressure applied is given in Newton/meter in default units. The global load is specified in global coordinates. In the global load, the load is given related to the global axes. The local load is specified in the local coordinates defined in the properties section. Local load is related to the local axes defined in the properties section.

The dynamic local beam load has an additional field for the local moment of torque. In the Dynamic local beam load there is an additional field that is the local torque momentum.

Dynamic shell surface load

There are five types of surface loads for shells:

1. Dynamic global shell load
2. Dynamic local shell load

In all cases the applied pressure is by default specified in given in Newton/meter² in default units. The global load is specified in global coordinates. In the global load, the load is given related to the global axes. The local load is specified in the local coordinates defined in the properties section. Local load is related to the local axes defined in the properties section.

Dynamic Contour Pressure load

With this condition it is possible to apply a dynamical pressure to a line that forms a boundary of a shell surface. The pressure is by default given in N/m. If the local axes field is set to **Global**, the pressure vector is specified according to the global axes. Option **Automatic** permits the definition of an automatic local axes system that is different for every element. The pressure vector will refer to these axes. This last option is useful to assign a local pressure to the boundary of the shell. See:

Data ► Local axes ► Surfaces ► Draw

to see how the automatic local axes are defined.

SeaFEM waves load (dynamic)

Allows to use as a time variable pressure load the result of a previously calculated SeaFEM analysis (**Input file**).

This load applies over shell elements.



Tdyn pressure load (dynamic)

Allows to use as a time variable pressure load the result of a previously calculated Tdyn analysis (**Input file**).

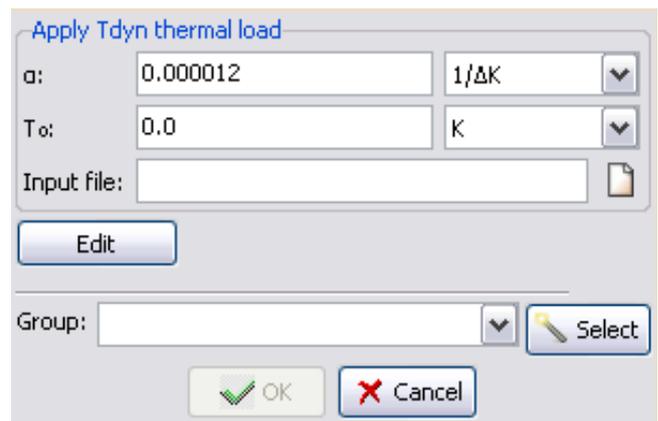
This load applies over shell elements.



Tdyn thermal load (dynamic)

Allows to use the result of a previously calculated Tdyn time variable heat transfer analysis (**Input file**), to create a thermo-mechanical strain.

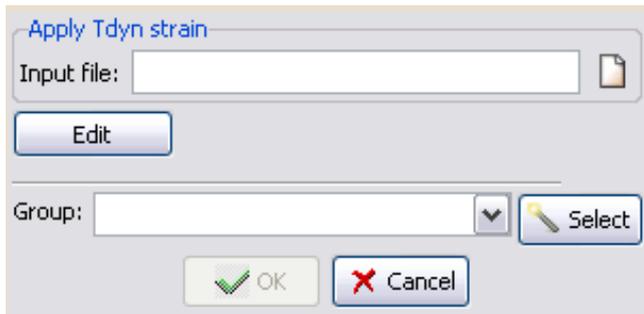
This load applies over solid elements.



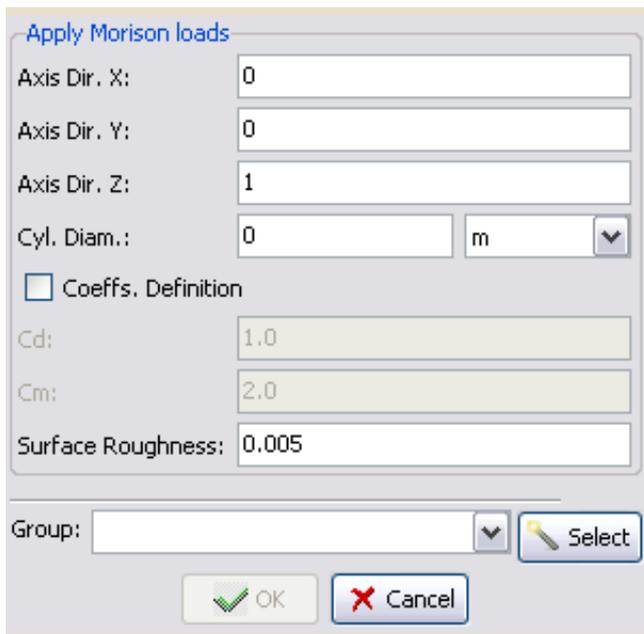
Tdyn strain (dynamic)

Allows to use the result of a previously calculated Tdyn time variable URSOLVER analysis (**Input file**), to create a mechanical strain.

This load applies over solid elements.



Morison load



4.11. Results

A reference for all the results that can be displayed is described in this section.

4.11.1. General results

They are applicable to beams, shells and solids.

		Default units	Remarks
Loadcases	One for every comb. loadcase	NONE	All results are displayed for the active loadcase
Displacements	Disp_X, Disp_Y, Disp_Z, displacements	m	Absolute displacements in global axes
Rotations	Rot_X, Rot_Y, Rot_Z, Rot	rad	Absolute rotations in global axes.
Velocities	Vel_X, Vel_Y, Vel_Z, velocities	m/s	Absolute velocities in global axes. Only available in dynamic analysis.

Accelerations	Accel_X, Accel_Y, Accel_Z, accelerations	m/s ²	Absolute accelerations in global axes. Only available in dynamic analysis
Vibration modes	One for every mode	m	Absolute displacements in global axes. Only available in dynamic analysis
Vibration modes rotations	One for every mode	rad	Absolute rotations in global axes. Only available in dynamic analysis if Rotations is set in Dynamic Output .
Reactions	X,Y,Z	N	They are nodal vectors
Momentum reactions	Mx,My,Mz	N·m	They are nodal vectors. Not applicable for solids
Kinetic energy		N·m	It is a scalar result saved in node 1. Only available in dynamic analysis if Global parameters is set in Dynamic Output .
Potential energy		N·m	It is a scalar result saved in node 1. Only available in dynamic analysis if Global parameters is set in Dynamic Output .
Elastic strain energy		N·m	It is a scalar result saved in node 1. Only available in dynamic analysis if Global parameters is set in Dynamic Output .
Total energy		N·m	It is a scalar result saved in node 1. Only available in dynamic analysis if Global parameters is set in Dynamic Output .

Linear momentum	$p_x, p_y, p_z, \text{Linear_Momentum} $	N·s	It is a vector result saved in node 1. Only available in dynamic analysis if Global parameters is set in Dynamic Output .
Angular momentum	$L_x, L_y, L_z, \text{Angular_Momentum} $	N·m·s	It is a vector result saved in node 1. Only available in dynamic analysis if Global parameters is set in Dynamic Output .

4.11.2. Results in beams

Local axes and strengths for beams.

Name	Module	Default units	Remarks
Beam displacements	Module	m	Absolute displacements in global axes
Beam velocities	Module	m/s	Absolute velocities in global axes. Only available in dynamic analysis
Beam accelerations	Module	m/s ²	Absolute accelerations in global axes. Only available in dynamic analysis
Axial force	N	N	Axial force in the beam
Momentum	M_y, M_z	N·m	Momentum in the local axes of the beam
Shear	Q_y, Q_z	N	Shear strength in the local axes of the beam
Torsor	T	N·m	Torsor momentum in the beam
Beam local axes			Strengths are expressed in these axes
Steel resistance factor		Non-dimensional	A factor, calculated only for profiles of the steel library, that means security if $RF < 1$ and non-security for $RF > 1$

Sign criteria for Y' momentum is:

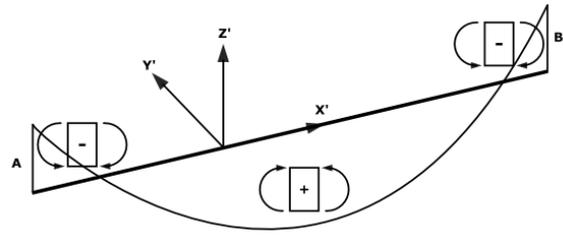


Diagram is drawn in the plane $X'Z'$ and in the side of the beam where the traction is. Positive values of the momentum mean that traction is in the $-Z'$ side (in the negative side of Z').

Z' momentum is the momentum that rotates around the Z' axe. Sign criteria for Z' momentum is:

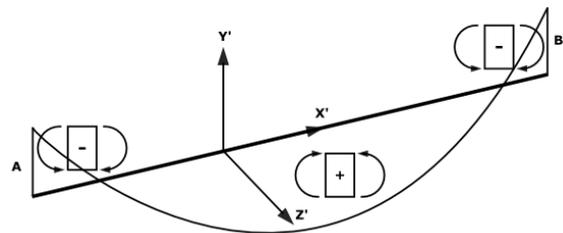
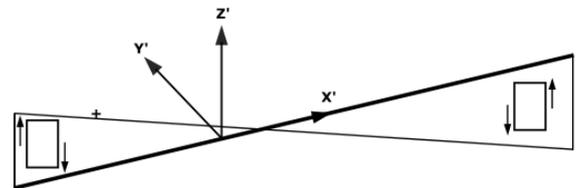


Diagram is drawn in the plane $X'Y'$ and in the side of the beam where the traction is. Positive values of the momentum mean that traction is in the $-Y'$ side (in the negative side of Y').

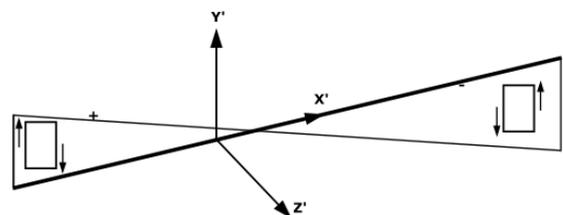
Shear in Z' is the shear in the $X'Z'$ plane. Sign criteria is:

View results -> Beam diagrams -> Z' shear



Shear in Y' is the shear in the $X'Y'$ plane. Sign criteria is:

View results -> Beam diagrams -> Y' shear



If more than one load case is selected, there is one special load case, called **Min Max**, where some special diagrams for beams are displayed. They show the maximums and minimums, for all the load cases, for all the strengths. The result **steel resistance factor** is also inside this special load case.

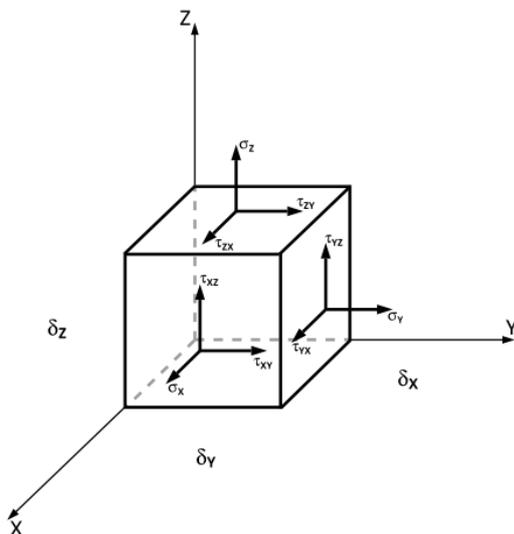
4.11.3. Results in shells

Local axes and strengths for shells

Name	Default units	Remarks
------	---------------	---------

Axial force	N_x, N_y, N_{xy}	N/m	Axial force in all the shell thickness per unit width of the shell
Momentum	M_x, M_y, M_{xy}	N·m/m	Momentum per unit width of the shell
Shear	Q_x, Q_y	N/m	Shear strength per unit width of the shell
Main axial force	N_{11}, N_{22}	N/m	Axial force in principal axes
Main momentum	M_{11}, M_{22}	N·m/m	Momentum in principal momentum axes
Shear in main	Q_{m1}, Q_{m2}	N/m	Shear in the principal momentum axes
Strengths	Top and Bottom and Von Misses	N/m ²	Stresses in the top face and bottom face for the shell. Sign criteria is the same than for solids. To display them it is necessary to activate one option in PD
Local axes			Strengths are expressed in these axes

The sign criteria for these strengths is:



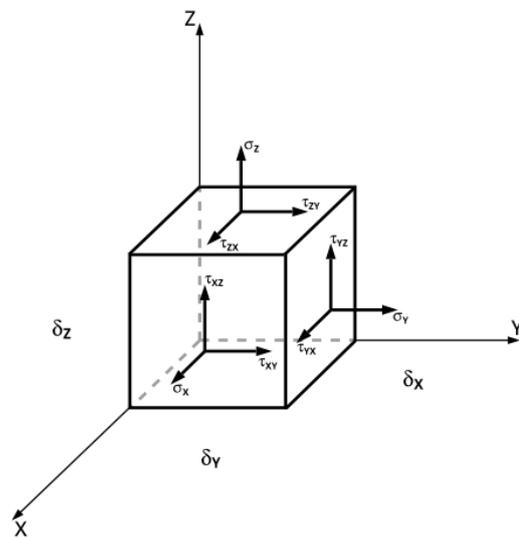
4.11.4. Results in solids

Strengths for 3D solids.

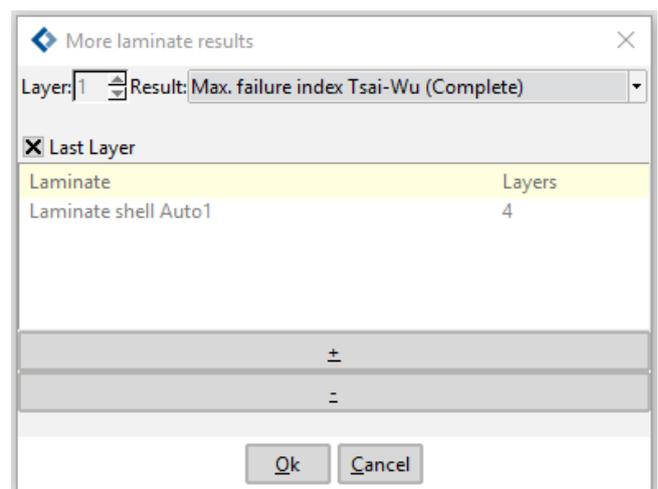
Name	Default units	Remarks
------	---------------	---------

Normal strengths	S_x, S_y, S_z	N/m ²	Also called $\sigma_x, \sigma_y, \sigma_z$ are the stresses for every global axe
Tangential strengths	T_{xy}, T_{xz}, T_{yz}	N/m ²	Also called $\tau_{xy}, \tau_{xz}, \tau_{yz}$
Main stresses	S_i, S_{ii}, S_{iii}	N/m ²	Main stresses expressed in the main axes
Von Misses		N/m ²	Scalar value for every node

The sign criteria for these strengths is:



4.12. Postprocessing tools



- Max. failure index Tsai-Wu (Complete) - Failure index taken at bottom and top of the laminate
- Security coef. Tsai-Wu (Complete) -
- Security coef. Tsai-Wu (Simple) - Tsai-Wu security factor calculated at the mid-plane of the lamina
- Security coef. LaRC04_m1 - Matrix tensile failure (Mode 1)
- Security coef. LaRC04_m2 - Matrix compressive failure

(Mode 2)

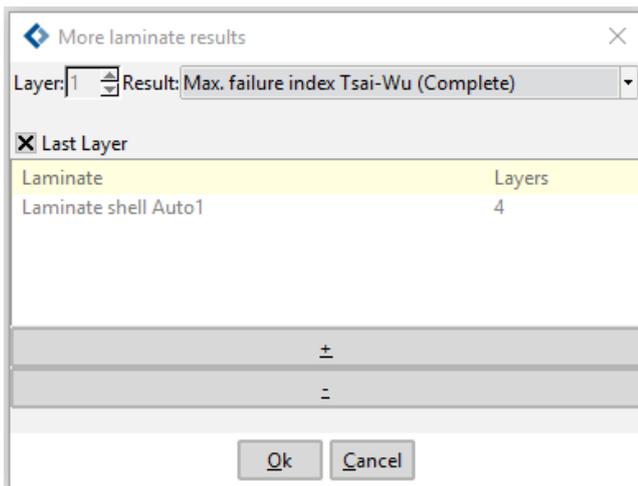
- Security coef. LaRC04_m3 - Fibre tensile failure (Mode 3)
- Security coef. LaRC04_m4 - Fibre compressive failure. Kink-band formation. (Mode 4)
- Security coef. LaRC04_m5 - Matrix failure under biaxial compression (Mode 5)
- Security coef. LaRC04_m6 - Combined matrix and fibre failure (Mode 6)
- Layer Stresses
- Layer Strains
- Layer Stress Top
- Layer Stress Base
- Layer Shear Stresses
- Layer Shear Strains

Of the many failure criteria available, RamSeries implements the Tsai Wu. This corresponds to a first ply failure analysis for which the sequence of load application is not significant. First ply failure criteria can be used to assess the global strength of a laminate and is the most common criteria used in linear analysis. Within this context, the structural integrity is usually reported in terms of a reserve factor (RF) or margin of safety. This reserve factor can be obtained layer per layer by invoking the calculation of the 'Security coef. Tsai-Wu (Complete)' and/or 'Security coef. Tsai-Wu (Simple)' results in the post-process tool. Alternatively, a failure index (FI) can be obtained by invoking the calculation of the 'Max. failure index Tsai-Wu (Complete)' result. Note that a FI value larger than one indicates failure. On the contrary, failure is predicted by a RF value smaller than unity.

4.12.1. Laminate results

After calculation, laminate results can be analyzed layer per layer in the post-process.

Potprocess ► Laminates



Within this window, any layer of the laminate can be selected together with one of the results available in the drop-down menu. The actual results available read as follows:

- Max. failure index Tsai-Wu (Complete) - Failure index taken at bottom and top of the laminate
- Security coef. Tsai-Wu (Complete) -
- Security coef. Tsai-Wu (Simple) - Tsai-Wu security factor calculated at the mid-plane of the lamina

Security coef. LaRC04_m1 - Reserve factor for matrix tensile failure mode

- Security coef. LaRC04_m2 - Reserve factor for matrix compressive failure mode
- Security coef. LaRC04_m3 - Reserve factor for fibre tensile failure mode
- Security coef. LaRC04_m4 - Reserve factor for fibre compressive failure mode (kink-band formation)
- Security coef. LaRC04_m5 - Reserve factor for matrix failure under biaxial compression mode
- Security coef. LaRC04_m6 - Reserve factor for combined matrix and fibre failure mode
- Layer Stresses
- Layer Strains
- Layer Stress Top
- Layer Stress Base
- Layer Shear Stresses
- Layer Shear Strains

Of the many failure criteria available, RamSeries implements the Tsai Wu. This corresponds to a first ply failure analysis for which the sequence of load application is not significant. First ply failure criteria can be used to assess the global strength of a laminate and is the most common criteria used in linear analysis. Within this context, the structural integrity is usually reported in terms of a reserve factor (RF) or margin of safety. This reserve factor can be obtained layer per layer by invoking the calculation of the 'Security coef. Tsai-Wu (Complete)' and/or 'Security coef. Tsai-Wu (Simple)' results in the post-process tool. Alternatively, a failure index (FI) can be obtained by invoking the calculation of the 'Max. failure index Tsai-Wu (Complete)' result. Note that a FI value larger than one indicates failure. On the contrary, failure is predicted by a RF value smaller than unity.

Alternatively, the LaRC04 (Langley Research Center) failure criteria is also implemented in RamSeries [17]. This is actually a set of failure criteria based on physical models for each failure mode. The corresponding security factor can be obtained by selecting the 'Security coef. LaRC04' result of the failure mode under analysis.

5. Appendixes

5.1. Appendix 1: Graphics of results depending on the number of nodes

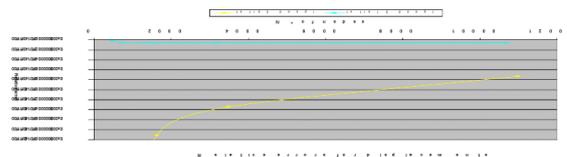
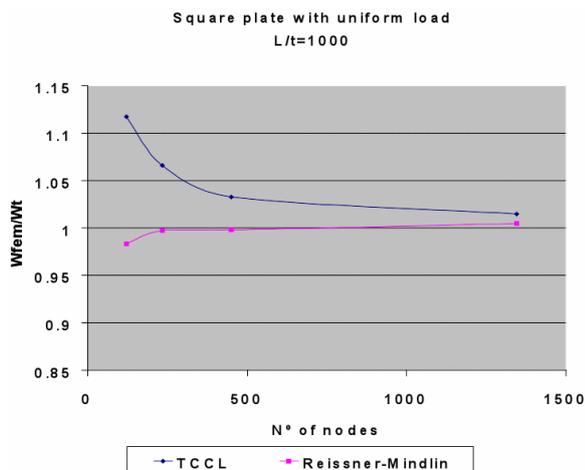
These graphics compares the analytical solution of the classical problem of a thin square plate with different types of loads. The variable compared is the displacement of the center of the plate. W_t is the theoretical value and W_{fem} is the result of the finite element analysis for different meshes.

The 6-noded *Reissner-Mindlin* element is the type of element used in RamSeries. The *TCCL* is another element type used as a test.

5.1.1. Square plate with uniform load

The load is uniformly distributed all over the plate. As there is symmetry, only one fourth of the plate is analyzed.

The ratio span/thickness is 1000.

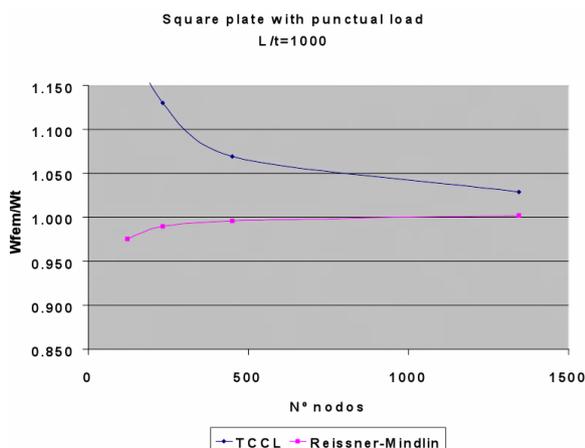


Note: The element called Reissner-Mindlin is the 6-noded triangle implemented in RamSeries.

5.1.2. Square plate with punctual load

The load is punctual in the center of the plate. As there is symmetry, only one fourth of the plate is analyzed.

The ratio span/thickness is 1000.



Note: The element called Reissner-Mindlin is the 6-noded triangle implemented in RamSeries.

5.1.3. Beam analyzed as solid 3D

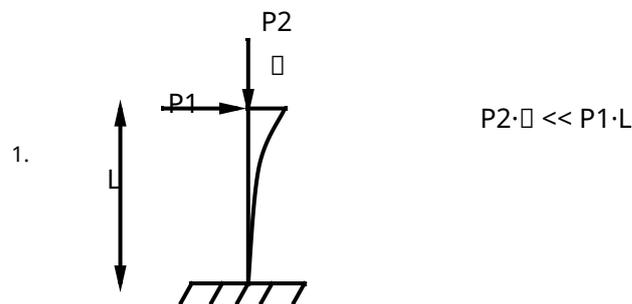
The maximum displacement of the solid is compared with the solution of the beam. Take into account that the 3D solution may be more precise than the simplified beam solution.

5.2. Appendix 2: Theory of beam analysis

In this part a brief theory of the analysis of the beams is given. Its purpose is to explain **what** RamSeries does and **not how** the program does it.

5.2.1. Basic assumptions

The basic assumptions that are assumed by RamSeries for the beam analysis are the following:



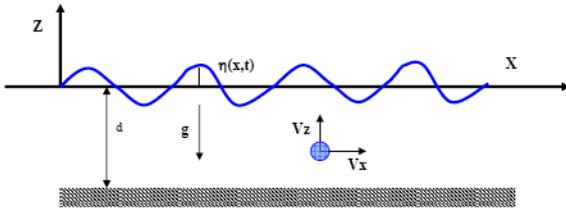
1. Small displacements
2. Lineal elasticity of the materials
3. Superposition of loads (Consequence of point 2)
4. Shear deformation is not considered for beams
5. Simple torsion $M_T = \frac{GJ}{L} \Delta$

5.3. Appendix 7: Wave theory

In this part a brief theory of the analysis of wave pressures on shells is given. Its purpose is to explain **what** RamSeries does and **not how** the program does it.

5.3.1. Potential flow for waves

A linearized model for the waves can be used for calculating de total force induced by the potential flow, as the integral of the pressure field.



Equation of Continuity:

$$\frac{D\rho}{Dt} + \rho \cdot \nabla \cdot \vec{V} = 0$$

As the fluid considered is a liquid, it can be assumed $\rho = \text{constant}$, and thus:

$$\nabla \cdot \vec{V} = 0 \quad \longrightarrow \quad \frac{\partial V_x}{\partial x} + \frac{\partial V_z}{\partial z} = 0$$

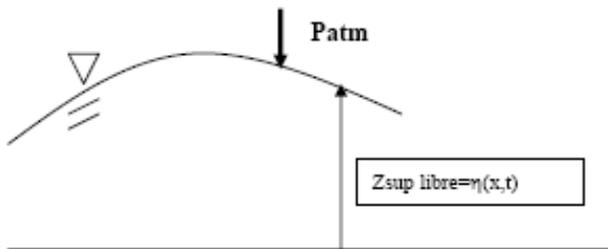
Balance of momentum:

$$\rho \cdot \frac{D\vec{V}}{Dt} = \frac{\partial(\rho \cdot \vec{V})}{\partial t} + \nabla(\rho \cdot \vec{V} \cdot \vec{V}) = \rho \cdot \vec{f}_m - \nabla p + \nabla \vec{T}$$

Assuming $\rho = \text{constant}$, and neglecting viscous effects (rotationally movement), the previous equations result as:

$$\begin{aligned} \rho \frac{DV_x}{Dt} &= -\frac{\partial p}{\partial x} \\ \rho \frac{DV_z}{Dt} &= -\rho g - \frac{\partial p}{\partial z} \end{aligned}$$

For solving these equations the boundary conditions to consider are:



Velocity normal to the surface equal to zero, and tangent velocity non-zero:

$$\frac{D\eta(x,t)}{Dt} = V_z \Rightarrow \frac{\partial \eta}{\partial t} + V_x \frac{\partial \eta}{\partial x} = V_z$$

Calling:

$$V = \sqrt{V_x^2 + V_z^2}$$

The momentum equations result:

$$\begin{aligned} \frac{\partial V_x}{\partial t} + \frac{1}{2} \frac{\partial V^2}{\partial x} &= -\frac{1}{\rho} \frac{\partial p}{\partial x} \\ \frac{\partial V_z}{\partial t} + \frac{1}{2} \frac{\partial V^2}{\partial z} &= -\frac{1}{\rho} \frac{\partial p}{\partial z} - g \end{aligned}$$

A solution deriving from a potential ϕ is taken. It must satisfy Laplace equation

$$V_x = \frac{\partial \phi}{\partial x}; V_z = \frac{\partial \phi}{\partial z}$$

$$\nabla^2 \phi = 0$$

Momentum equation now reduces to Bernoulli equation, taken a constant C as integration constant.

$$\frac{p}{\rho} = c - gz - \frac{1}{2} V^2 - \frac{\partial \phi}{\partial t}$$

For integration, imposing the boundary conditions, the problem simplifies:

- Interior:

$$\nabla^2 \phi = 0$$

$$\frac{p}{\rho} = \frac{p_{atm}}{\rho} - gz - \frac{\partial \phi}{\partial t}$$

- Bottom ($z=d$):

$$\left. \frac{\partial \phi}{\partial z} \right|_{(z=d)} = 0$$

- Free surface ($z=0$):

$$\begin{aligned} \frac{\partial \eta}{\partial t} &= \frac{\partial \phi}{\partial z} \\ \frac{\partial \phi}{\partial t} + g\eta &= 0 \end{aligned}$$

A solution for the Laplace problem of the type $\phi = f(z) \cdot g(x)$ can be found:

$$\phi = \frac{Hg}{2\omega} \frac{\cosh\left(kd \cdot \left(l + \frac{z}{d}\right)\right)}{\cosh(k \cdot d)} \cdot \sin(k \cdot x - \omega \cdot t)$$

With:

- $K = L/2\pi$; wave number

ω ; angular frequency

$$\omega^2 = K.g.tanh(K.d)$$

In our particular case, the expression that seems to suit better for the pressures created by the wave over the ship's hull is:

$$P(z, x) = P_{atm} - \rho g z - \frac{1}{2} \rho A^2 \omega^2 e^{2Kz} + \rho g A e^{Kz} \cos(K(x - x_0))$$

"z" coordinates will be negative in this case (i.e. the reference system origin is the wave surface, with the Z axis pointing downwards).

5.4. Appendix 10: Fatigue Damage Assessment

The main goal of the fatigue oriented design is to assure that the structure under analysis will have enough resistance when dealing with fatigue, i.e, it has an acceptable *fatigue life*.

The **resistance to fatigue** is usually given in terms of **S-N graphs**, which depend on the different materials. These graphs show the number of cycles (N) corresponding to a certain stress range (S).

The design fatigue life for structural components is based on an specified **service lifetime**. If this service lifetime is not specified, a 20 year lifetime will be assumed.

For more detailed description of the method, and further information on fatigue damage assessment in ships, please review Reference [1].

5.4.1. Procedure

The contour fill obtained, will be an indication of the fatigue failure risk zones. The failure criteria is given by the Normative design requirements. This criteria is:

$$D_D \leq 1$$

According to Method 1 of the followed Normative (**Reference [11]**), the accumulated fatigue damage level, is:

$$D_D = DDF \cdot D_C$$

Being *DDF* a design factor, given by the following chart:

Location	Accessibility for inspection and repair of initial fatigue and coating damages (8)	Corrosion protection	Corrosion allowance (10)	S-N curve	DDF
Atmospheric zone	Yes	Coating (1)	No	In air	1.0
	No	Coating (1)	No	In air	2.0
Splash zone (3)	Yes	Coating (2) (3)	Yes (6)	Combination of curves marked "air" and "free corrosion" (9)	3.0
	No	Coating (2) (3)	Yes (6)		3.0
Submerged zone	Yes	Cathodic protection and optional coating (2) (4)	No	In seawater	2.0
	No	Cathodic protection and optional coating (2) (4)	Yes (7)		3.0
Scour zone	Yes	Cathodic protection, coating near free surfaces and above free surfaces (12)	Yes (11)		3.0
	No	Cathodic protection, coating near free surfaces and above free surfaces (12)	Yes (11)		2.0
Below seabed	Yes	None	No		3.0
	No	None	No		2.0
Closed compartments with seawater	Yes	Cathodic protection, coating near free surfaces and above free surfaces (12)	Yes (11)		3.0
	No	Cathodic protection, coating near free surfaces and above free surfaces (12)	Yes (11)		3.0

1) Coating for structures above the splash zone shall be a high quality multilayer coating in accordance with corrosivity category CSM in ISO 12944.
2) Coating for structures in the splash zone and below the splash zone shall be taken as for (1) and shall furthermore be qualified for compatibility with cathodic protection systems. Selection and qualification of coating systems shall include consideration of all conditions relevant for necessary repair after installation.
3) Coating shall be selected with due consideration of loads from impacts from service vessels and floating ice.
4) Below the splash zone coating is considered optional. Coating can provide a reliable corrosion protection and can be designed such as to reduce marine growth. However, coating can be damaged during inspection and maintenance sessions where marine growth is removed.
5) Splash zone definition according to Sec. 11 B200.
6) The corrosion allowance in the splash zone shall be selected in accordance with the corrosion rate for the structural steel in seawater and in accordance with the planned inspection and repair strategy. In the North Sea the corrosion allowance for coated primary steel structures without planned coating repair in a 20-year design life is 6 mm. A corrosion allowance of minimum 2 mm is recommended for replaceable secondary structures.
7) In the scour zone the cathodic protection might not be fully effective and anaerobic corrosion might occur. For typical North Sea conditions it is recommended to design with a corrosion allowance of 3 mm in the scour zone.
8) If the designer considers the steel surface accessible for inspection and repair of initial fatigue damage and coating this must be documented through qualified procedures for these activities. See also Sec. 11 and Sec. 13.
9) The basic S-N curve for unprotected steel in the splash zone is the curve marked "free corrosion". The basic S-N curve for coated steel is the curve marked "in air". It is acceptable to carry out fatigue life calculations in the splash zone based on accumulated damage for steel considering the probable coating conditions throughout the design life - i.e. once damaged and repaired. The coating conditions shall refer to an inspection and repair plan as specified in Sec. 13. For coating systems with a specified coating life of 15 years and without any qualified coating repair procedure, it is acceptable to use the "in seawater" S-N curve as a representative fatigue curve throughout a service life of 20 years.
10) The corrosion allowance shall be considered in all limit state analyses. Fatigue calculations can be based on a steel wall thickness equal to the thickness that corresponds to half the allowance over the full service life.
11) The corrosion allowance for closed compartments with seawater shall be established from experience data on a case to case basis.
12) Biocides and scavengers can reduce corrosion in closed compartments.

D_C is the characteristic accumulated damage which is given by the following expression:

$$D_C = \sum_{i=1}^I \left(\frac{n_{C,i}}{N_{C,i}} \right) \cdot \left(\frac{t_{tot}}{t_{sim}} \right)$$

Where;

- I : Number of stress ranges levels σ_i = $SCF \cdot \Delta\sigma_{nominal}$. For obtaining the stress ranges and their corresponding cycle counts ($n_{C,i}$), the **Rainflow Counting Method** is applied to the stresses time history of each combined load case or non-linear run. The number of levels is defined by the user.
- $n_{C,i}$: Number of load cycles in level "i".
- $N_{C,i}$: Number of load cycles to failure, for each stress range $\Delta\sigma_i$, given for the corresponding S-N curve.
- t_{tot} : Expected total lifetime
- t_{sim} : Actual simulation time

5.4.2. S-N curves

For structural steel, the S-N curve used is the following:

$$\log_{10}(N) = \log_{10}a - m \cdot \log_{10}(\Delta\sigma \cdot \left(\frac{t}{t_{ref}} \right)^k)$$

Where:

- N = Number of cycles to failure, for a given

stresses range ($\Delta\sigma = SCF \cdot \Delta\sigma_{nominal}$).

- **SCF**= Stress Concentration Factor
- **t** = Thickness through which a crack will most likely grow. $t = t_{ref}$ is used for $t < t_{ref}$.
- **t_{ref}** = Reference thickness: $t_{ref} = 32 \text{ mm}$ for tubular joints, $t_{ref} = 25 \text{ mm}$ for welded connections other than tubular joints.
- **m, k, log₁₀a** = given by the following table:

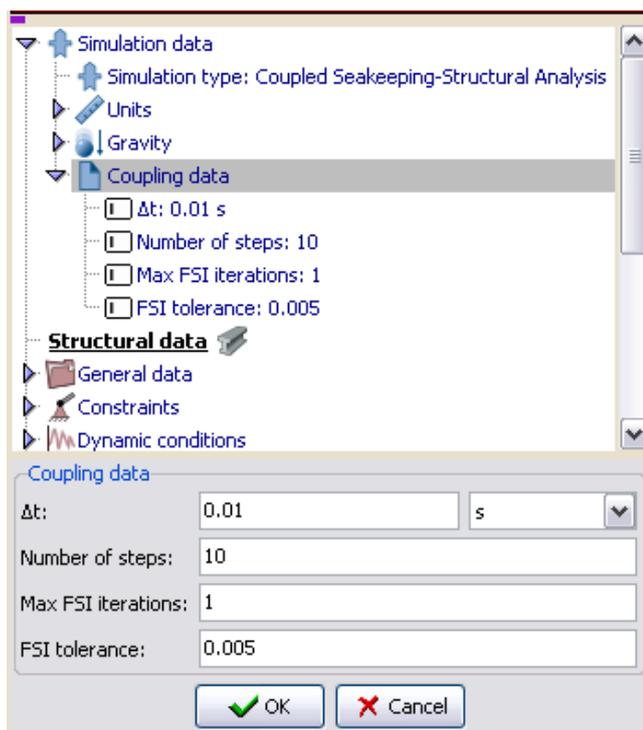
Structural detail	Environment										
	In air			In seawater with corrosion protection			Free corrosion				
	loga	m	Range of validity	k	loga	m	Range of validity	k	loga	m	k
Weld in tubular joint	12.164	3	$N < 10^7$	0.25	11.764	3	$N < 10^6$	0.25	11.687	3	0.25
	15.606	5	$N > 10^7$	0.25	15.606	5	$N > 10^6$	0.25			
Butt weld and tubular girth weld, weld toe (1)	12.164	3	$N < 10^7$	0.20	11.764	3	$N < 10^6$	0.20	11.687	3	0.20
	15.606	5	$N > 10^7$	0.20	15.606	5	$N > 10^6$	0.20			
Butt weld and tubular girth weld, weld root (1) (2)	11.855	3	$N < 10^7$	0.25	11.455	3	$N < 10^6$	0.25	11.378	3	0.25
	15.091	5	$N > 10^7$	0.25	15.091	5	$N > 10^6$	0.25			
Non-load carrying welded attachments of length L, in main stress direction	12.010	3	$N < 10^6$ $L < 50 \text{ mm}$	0.20	11.610	3	$N < 10^6$ $L < 50 \text{ mm}$	0.20	11.533	3	0.20
	15.350	5	$N > 10^7$ $L < 50 \text{ mm}$	0.20	15.350	5	$N > 10^6$ $L < 50 \text{ mm}$	0.20			
	11.855	3	$N < 10^6$ $50 \text{ mm} < L < 120 \text{ mm}$	0.25	11.455	3	$N < 10^6$ $50 \text{ mm} < L < 120 \text{ mm}$	0.25	11.378	3	0.25
	15.091	5	$N > 10^7$ $50 \text{ mm} < L < 120 \text{ mm}$	0.25	15.091	5	$N > 10^6$ $50 \text{ mm} < L < 120 \text{ mm}$	0.25			
	11.699	3	$N < 10^6$ $120 \text{ mm} < L < 300 \text{ mm}$	0.25	11.299	3	$N < 10^6$ $120 \text{ mm} < L < 300 \text{ mm}$	0.25	11.222	3	0.25
	14.832	5	$N > 10^7$ $120 \text{ mm} < L < 300 \text{ mm}$	0.25	14.832	5	$N > 10^6$ $120 \text{ mm} < L < 300 \text{ mm}$	0.25			
	11.546	3	$N < 10^6$ $L > 300 \text{ mm}$	0.25	11.146	3	$N < 10^6$ $L > 300 \text{ mm}$	0.25	11.068	3	0.25
	14.576	5	$N > 10^7$ $L > 300 \text{ mm}$	0.25	14.576	5	$N > 10^6$ $L > 300 \text{ mm}$	0.25			

1) For girth welds welded from both sides, the S-N curves for the weld toe apply at both sides. For girth welds welded from one side only, the S-N curves for the weld toe position apply to the side from which the weld has been welded up, and the S-N curves for the weld root apply to the other side.
2) Transverse butt weld on a temporary or permanent backing strip without fillet welds.

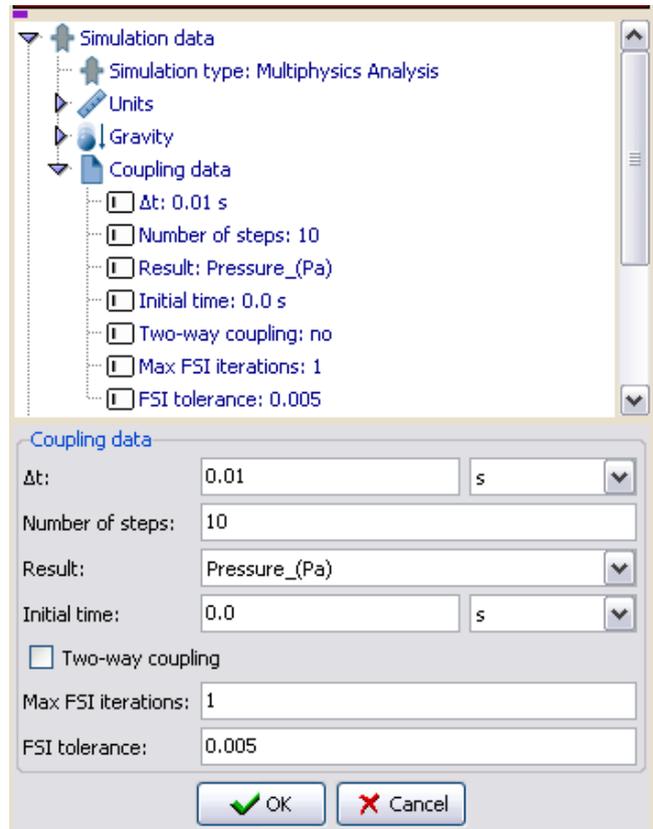
5.5. Appendix 11: Coupled Analyses

5.5.1. Coupling data

In this section, analysis data necessary for performing a coupled simulation (either with SeaFEM or with Tdyn) can be inserted.



Coupling data (Seakeeping/structural)



Coupling data (Multiphysics Analysis)

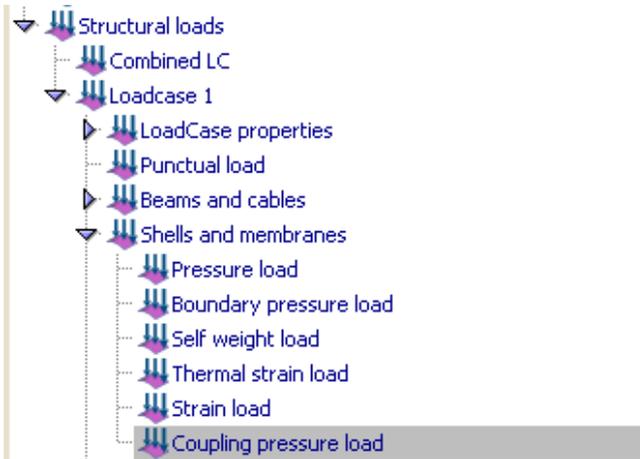
- **Δt**: Is the **time step** that will be used for the simulation. Please note that this time step will be used for both structural and multiphysics (or seakeeping) analysis.
- **Number of steps**: Total number of steps for the simulation. The total physical time of the analysis will be the number of steps times the time step.
- **Result** (only for Multiphysics coupled analysis): Allows to choose which result from Tdyn will be transferred to RamSeries for the coupling. Available results are:
 - Pressure (Pa)
 - Traction (Pa)
 - Variable1 (U)
- **Initial time** (only for Multiphysics coupled analysis): Time from which the coupling will start. For example, Tdyn may run alone until a certain time (initial time), and then start to couple with RamSeries.
- **Two-way coupling** (only for Multiphysics coupled analysis): Indicates if mesh deformation is sent to Tdyn or not.
- **Max FSI iterations**: Maximum number of equilibrium iterations with Tdyn (or SeaFEM) within one time step.
- **FSI tolerance**: Relative tolerance of displacement increment between two iterations for structural part convergence of coupled equilibrium with Tdyn (or SeaFEM).

5.5.2. Coupled Analyses with Tdyn

When performing coupled analyses with Tdyn, it is necessary to assign an specific condition which will transfer the load from the fluid to the structure.

This can be done assigning the following load:

Structural loads ▶ Loadcase # ▶ Shells/Solids ▶ Coupling pressure load



5.5.3. Coupled Analyses with SeaFEM

When performing coupled analyses with SeaFEM, it is necessary to assign a specific condition which will transfer the load from the fluid to the structure.

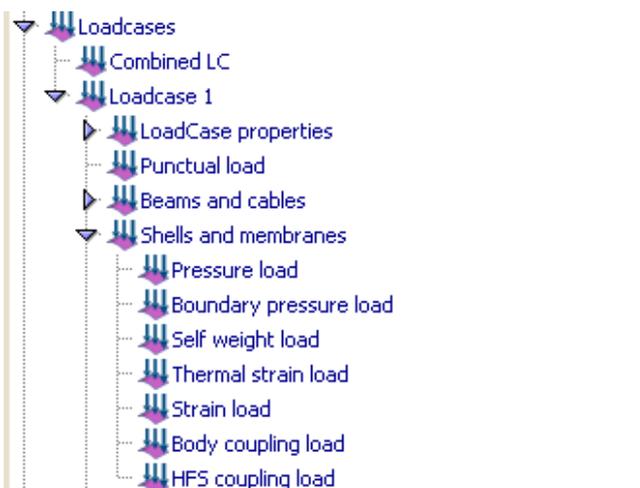
This can be done assigning the following loads:

- For body coupling (RamSeries solves the rigid body movement dynamics)

Structural loads ▶ Loadcase # ▶ Shells/Solids ▶ Body coupling load

- For HFS coupling (SeaFEM solves the rigid body movement dynamics)

Structural loads ▶ Loadcase # ▶ Shells/Solids ▶ HFS coupling load



5.6. Appendix 12: Stiffened Shell model

This model allows to use shell which is equivalent to a shell with beam elements, acting as stiffeners.

5.6.1. Inertia of the stiffened section

The following mechanical properties of the system can be obtained from the stiffener and the attached shell data:

Inertia of the system respect to an axis passing at mid-plane of the shell:

$$I_{A0} = A_{sh} \cdot z_{sh}^2 + A_{st} \cdot z_{st}^2 + I_{Gstiff} + I_{Gshell}$$

- $A_{sh} = c \cdot h$: Transversal area of the shell.
- A_{st} : Transversal area of the stiffener.
- $z_{sh} = h/2$
- h : Shell thickness.
- h_{Gst} : height of the stiffener center of gravity respecto to its bottom.
- $z_{st} = h_{Gst} + h/2$: height of the stiffener center of gravity respect to the mid-plane of the shell.
- I_{Gstiff} : Inertia of the stiffener respect to its center of gravity.
- $I_{Gshell} = 1/12 \cdot c \cdot h^3$: Inertia of the shell respect to its center of gravity.
- c : distance between stiffeners.

Inertia of the system respect to the neutral axis of the system:

$$I_{NA} = I_{A0} - (A_{sh} + A_{st}) \cdot \delta^2$$

- $\delta = (A_{sh} \cdot z_{sh} + A_{st} \cdot z_{st}) / (A_{sh} + A_{st})$: Height of the neutral axis.

Inertia of the stiffener respect to the mid-plane of the shell:

$$I_{MSstiff} = I_{Gstiff} + A_{st} \cdot (h_{Gst} + h/2)^2$$

Inertia of the stiffener respect to neutral axis of the system:

$$I_{ENstiff} = I_{Gstiff} + A_{st} \cdot (h_{Gst} + h/2 - \delta)^2$$

Inertia of the shell respect to neutral axis of the system:

$$I_{ENshell} = I_{Gshell} + A_{sh} \cdot \delta^2$$

An equivalent shell thickness will be calculated. This condition to accomplish is that the inertia remains the same:

$$I_{EQ} = I_{NA}$$

$$\bullet I_{EQ} = 1/12 \cdot c \cdot h_{EQ}^3$$

Therefore:

$$h_{EQ} = (12 \cdot I_{NA} / c)^{(1/3)}$$

5.6.2. Equivalent model formulation

The resulting shell must be equivalent regarding weight, flexural rigidity (inertia) and membrane stiffness (area). These three effects are explained in the following lines.

The equivalent shell must have the same **weight** as the one with the system stiffeners-shell, therefore:

$$P_{EQ} = \rho \cdot h_{EQ} \cdot (A_{sh} + A_{st}) / A_{EQ} \text{ [N/m}^2\text{]}$$

Regarding section **inertia** and **area**, the following is considered:

The corresponding shell stiffness matrices, following the Reissner-Mindlin formulation for shells, for an isotropic material are:

$$f = \beta \cdot [[EvE0], [vEE0], [00\phi G]]$$

$$\phi = (1 - \nu^2)$$

$$\beta = 1/\phi$$

$$c = [[\alpha G0], [0\alpha G]]$$

The stiffness matrix of the system (D_{shell}^*) is obtained as follows, building it with the matrices D_m^* , D_{mf}^* , D_f^* and D_c^* :

$$*_{shell} = [[m*mf*0], [mf*f*0], [00c*]]$$

- $D_m^* = \int_{-h/2}^{+h/2} D_f \cdot dz;$
- $D_f^* = \int_{-h/2}^{+h/2} z^2 \cdot D_f \cdot dz;$
- $D_{mf}^* = \int_{-h/2}^{+h/2} z \cdot D_f \cdot dz;$
- $D_c^* = \int_{-h/2}^{+h/2} D_c \cdot dz$

After integration:

$$m_* = [[xmzym0],[xymym0],[00zm]]$$

With:

$$D_{xm} = h \cdot E / (1 - \nu^2)$$

$$D_{ym} = h \cdot E / (1 - \nu^2)$$

$$D_{xym} = h \cdot \nu \cdot E / (1 - \nu^2)$$

$$D_{zm} = h \cdot G$$

$$f_* = [[xfxyf0],[xyfyf0],[00zf]]$$

With:

$$D_{xf} = I_{ENshell} \cdot E / (c \cdot (1 - \nu^2))$$

$$D_{yf} = h^3 \cdot E / (12 \cdot (1 - \nu^2))$$

$$D_{xyf} = h^3 \cdot E \cdot \nu / (12 \cdot (1 - \nu^2))$$

$$D_{zf} = h^3 \cdot G / 12$$

$mf_* = [0]$ (assumed for simplicity)

$$c_* = h \cdot \alpha \cdot [[G0],[0G]]$$

With:

$$\alpha = 5/6$$

This is to be combined correspondingly with the stiffness of a beam, considering its material properties homogeneously distributed over the section:

$$D^*_{beam} = [[EA00000],[0y'0000],[00z'000],[000G]00],[0000y'GA0],[00000z'GA]]$$

Therefore, finally (assuming X axis is the stiffeners direction):

$$*_{Beam-Shell} = [[m - BS_{mf} - BS_0], [mf - BS_f - BS_0], [00c - BS_*]]$$

Being:

$$m - BS_* = [[x_m - BS_{xym} - BS_0], [xym - BS_{ym} - BS_0], [00z_m - BS]]$$

With:

$$D_{x_m-BS} = h \cdot E / (1 - \nu^2) + E \cdot A_{st} / c$$

$$D_{y_m-BS} = h \cdot E / (1 - \nu^2)$$

$$D_{xym-BS} = h \cdot \nu \cdot E / (1 - \nu^2)$$

$$D_{z_m-BS} = h \cdot G$$

$$f - BS_* = [[x_f - BS_{xyf} - BS_0], [xyf - BS_{yf} - BS_0], [00z_f - BS]]$$

With:

$$D_{x_f-BS} = I_{ENshell} \cdot E / (c \cdot (1 - \nu^2)) + E \cdot I_y / c$$

$$D_{y_f-BS} = h^3 \cdot E / (12 \cdot (1 - \nu^2))$$

$$D_{xyf-BS} = h^3 \cdot E \cdot \nu / (12 \cdot (1 - \nu^2))$$

$$D_{z_f-BS} = h^3 \cdot G / 12$$

$I_y = I_{ENstiff}$: Inertia of the stiffener referred to the neutral axis.

$$mf - BS_* = [0]$$

$$c - BS_* = h \cdot a \cdot [[G0], [OG]] + st \cdot \cdot \cdot [[G0], [00]]$$

5.6.3. Stiffened shell test 1

A simple example to test stiffened shells has been set up.

The **geometry** consists of a rectangle of dimensions:

- a = 2 m
- b = 4 m

It is loaded with its **self-weight**, and the material is defined as an stiffened shell with the following characteristics:

- Shell thickness: **h = 0.01 m**
- Distance between stiffeners: **c = 0.25 m**
- Stiffeners in the direction of side "a".
- Stiffener: **HP-200-10**

The **equivalent stiffened shell** model used (Inertia of the stiffened section and Equivalent model formulation) result in the following shell properties:

- **$h_{eq} = 0.1129$ m**
- **$\rho_{eq} = 1559.532$ N/m²**

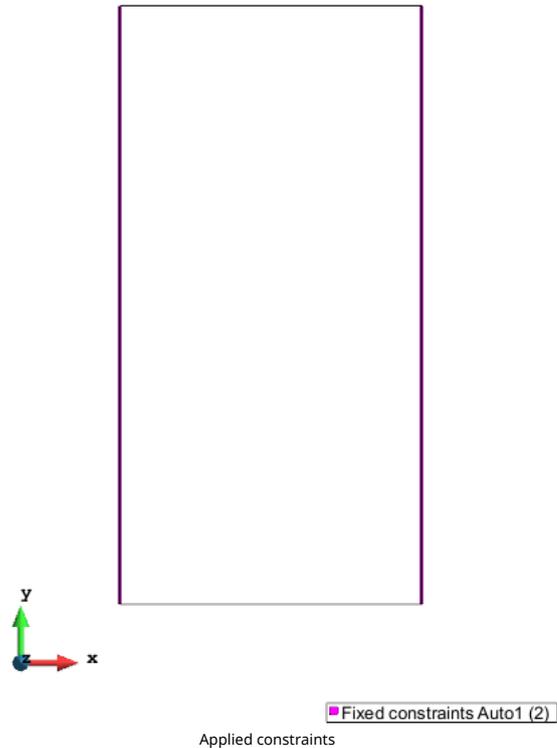
The **material** assigned to the shell is steel:

$$E = 2.1e11 \text{ N/m}^2$$

$$\rho = 76900 \text{ N/m}^3$$

$$\nu = 0.3$$

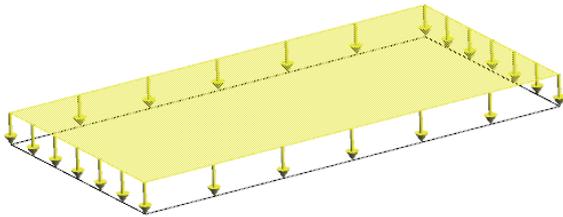
The rectangular shell is **simply supported** in both "b" sides:



The model has been loaded with the following values:

- 6. $q_1 = q_{0z}$
- 7. $q_2 = 2 \cdot q_{0z}$
- 8. $q_3 = 3 \cdot q_{0z}$
- 9. $q_4 = 4 \cdot q_{0z}$

With: $q_{0z} = -1.0e6 \text{ N/m}^2$



Applied distributed vertical pressure

Mesh used (both models with triangular elements; the bemas model also includes linear elements):

440 nodes (aprox. in both models)

The results for the **maximum vertical deflection** are showed in the next table, together with the results for the analytical solution for this problem (extracted from the book "Teoría de placas y láminas" (S. Timoshenko, Ed. Urmo 1970)):

$$z_{Max} = \alpha \cdot \frac{q \cdot 4}{D}$$

Being:

$\alpha=0.01013$ (Factor for simply supported shells under uniform load, for $b/a=2$)

q: q_0, q_1, q_2, q_3, q_4

$a = 2 \text{ m}$

$$D = \frac{E \cdot h^3}{12 \cdot (1 - \nu^2)} = 27711937.9 \text{ N} \cdot \text{m}$$

q (N/m2)	d_z (m) StiffS_Ram	d_z (m) Theory	E (%)
-1.00E+06	-0.0060225	-0.005849	2.97
-2.00E+06	-0.012045	-0.011697	2.97
-3.00E+06	-0.018068	-0.017546	2.97
-4.00E+06	-0.02409	-0.023395	2.97

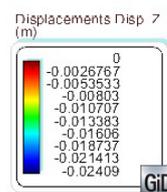
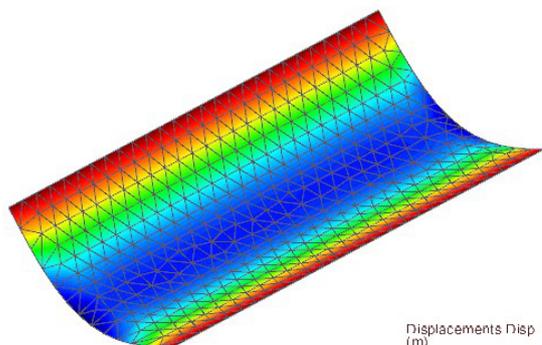
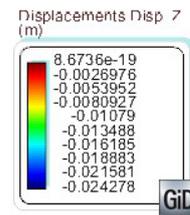
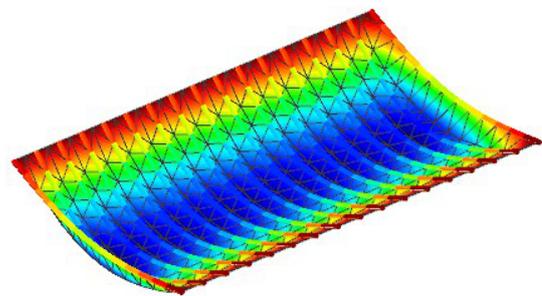
Results comparing with theory

Another test, involving the same model for stiffened shell of the previous example, has been carried out. In this case, the stiffened shell deflection is compared to the deflection of the equivalent reinforced shell (with the stiffeners modelled as beams).

q (N/m2)	d_z (m) StiffS_Ram	d_z (m) Sh+Stiff	E (%)
-1.00E+06	-0.0060225	-0.006070	0.78
-2.00E+06	-0.012045	-0.012139	0.77
-3.00E+06	-0.018068	-0.018209	0.77
-4.00E+06	-0.02409	-0.024278	0.77

Results comparing with RamSeries beams&shells model

The following images show the deformation of both models (both for q_4). A deformation factor of 20 has been applied to both visualizations.



5.6.4. Stiffened shell test 2

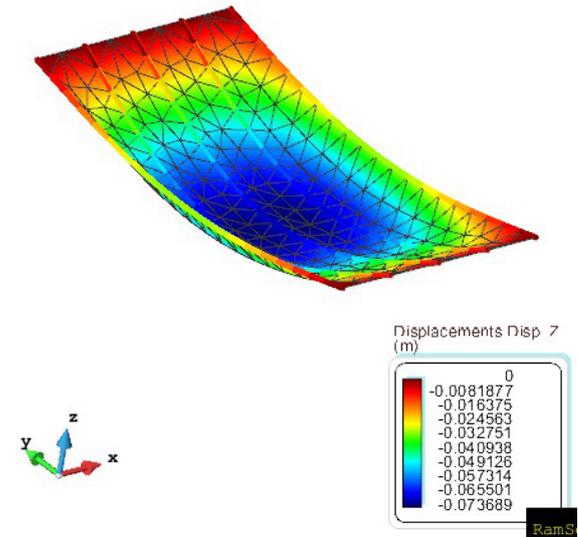
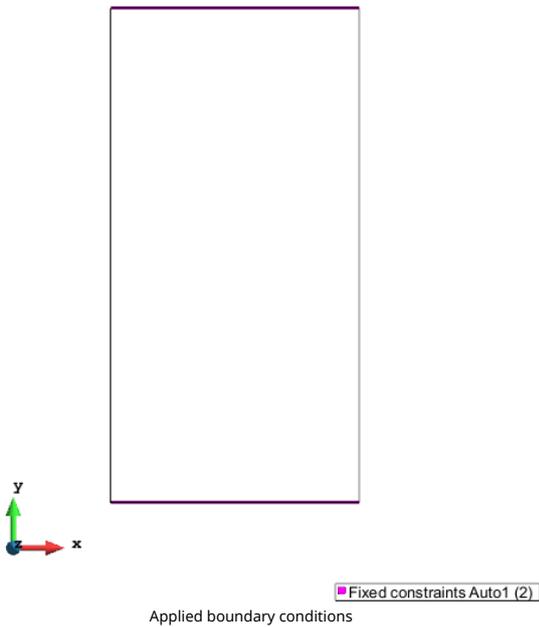
5.6.5. Stiffened shell test 3

This test case is similar to the previous one.

In this case, the stiffeners are oriented in the Y axis (ortogonal to the direction of the stiffeners of first test model).

Distance between stiffeners: **c = 0.5 m**

The boundary conditions have changed, so now, the short edge is the one restrained in the Z direction. Long edges are free.



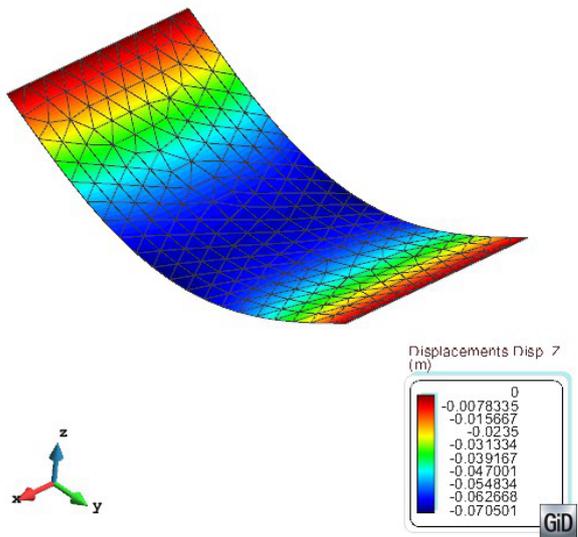
The load in this case is: $q_{0z} = -1.0e5 \text{ N/m}^2$

The next table shows the results:

q (N/m2)	d_z (m) StiffS_Ram	d_z (m) Sh+Stiff	E (%)
-1.00E+03	-0.017625	-0.018422	4.33
-2.00E+03	-0.035251	-0.036845	4.33
-3.00E+03	-0.052876	-0.055267	4.33
-4.00E+03	-0.070501	-0.073689	4.33

Results comparing with RamSeries beams&shells model

The following images show the deformation of both models (both for q_d). A deformation factor of 10 has been applied to both visualizations:



5.6.6. Stiffened shell test 4

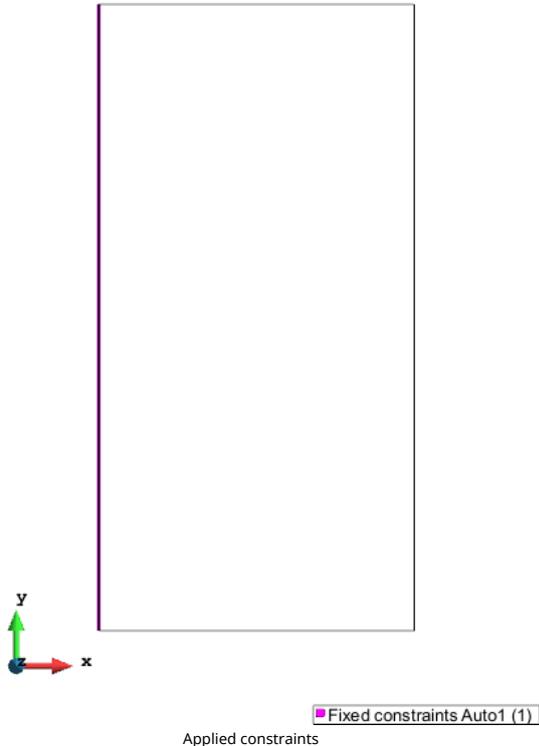
This case is arranged for testing membrane behaviour of the stiffened shell.

In this case, the stiffeners are oriented in the X axis (as in the first test model).

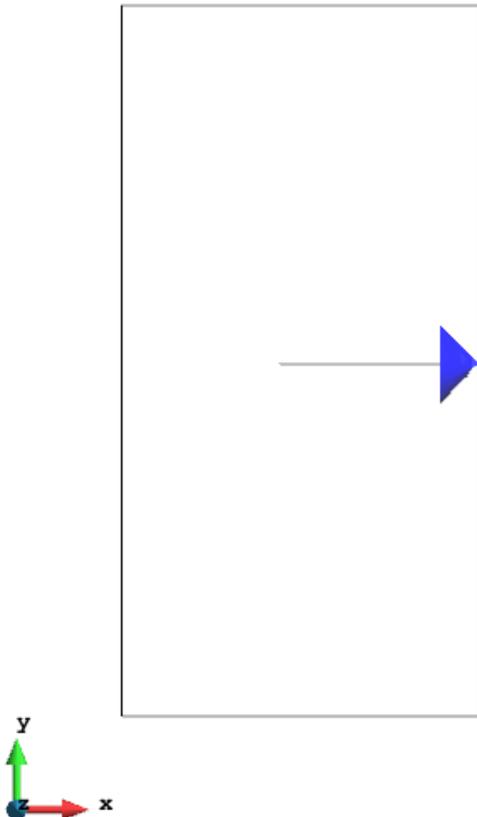
Distance between stiffeners: **c = 0.25 m**

The boundary conditions have changed, so now, the long left

edge is completely restrained. All the other edges are free.



The load in this case is (at point [2,2,0], in the X axis): $q_0 = 1.0e8 \text{ N}$



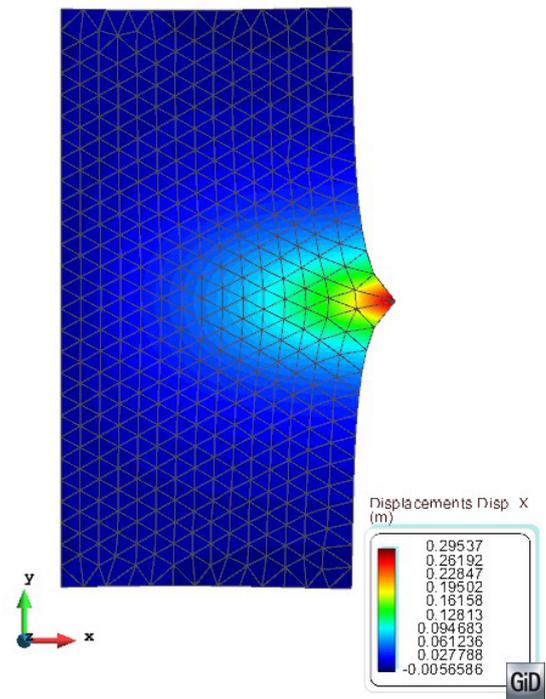
Applied punctual load

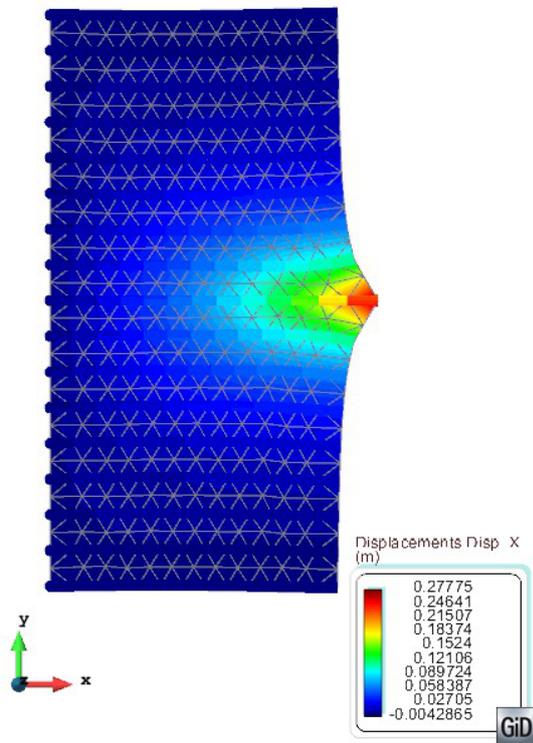
The next table shows the results:

Fx (N)	d_x (m) StiffS_Ram	d_x (m) Sh+Stiff	E (%)
1.00E+08	0.073841	0.069436	6.34
2.00E+08	0.14768	0.138870	6.34
3.00E+08	0.22152	0.208310	6.34
4.00E+08	0.29537	0.277750	6.34

Results comparing with RamSeries beams&shells model

The following images show the deformation of both models (both for q_4). A deformation factor of 1 has been applied to both visualizations:





13. Felippa, C. A. *A study of optimal membrane triangles with drilling freedoms*, Report CU-CAS-03-02, Center for aerospace structures, Colorado 2003.

14. P. Hopkins, "Benchmarks for Membrane and bending analysis of Laminated Shells. Part 1. Stiffness matrix and thermal characteristics.," NAFEMS, 2005.

15. P. Hopkins, "Benchmarks for Membrane and bending analysis of Laminated Shells. Part 2. Strength Analysis.," NAFEMS, 2005.

16. "Hull in composite materials and plywood, material approval, design principles, construction and survey," Bureau Veritas, Feb. 2017.

17. S. T. Pinho, C. G. Dávila, P. P. Camanho, L. Iannucci, and P. Robinson, "Failure models and criteria for FRP under in-plane or 3D stress states including shear non-linearity," National Aeronautics and Space Administration (NASA), Technical memorandum TM-2005-213530, 2005.

6. References

1. Barbat, A. H., Miquel, J. *Estructuras sometidas a acciones sísmica*, 2nd Ed., CIMNE 1994
2. Bathe, K.J. *Finite Element Procedures*, Prentice Hall, New Jersey, USA, 1996.
3. Clough, R. W., Penzien, J. *Dynamic of Structures*, McGraw-Hill, Inc. New York, 1975
4. Comisión permanente del Hormigón, *Instrucción de hormigón EHE*, 5^a Ed., Ministerio de Fomento 1999
5. Crisfield, M.A. "Non-linear finite element analysis of solids and structures", John Wiley & Sons, 1991.
6. Oñate, E. *Cálculo de estructuras por el método de los elementos finitos*, 2nd Ed., CIMNE 1995.
7. MacNeal, R.H., and Harder, R.L, *A proposed standard set of problems to test finite element accuracy*, Finite elements and Design, Vol. 1, pp. 3-20, 1985.
8. Zienkiewicz, O.C., and Taylor, R.L. *The finite element method*, 4th Ed., Mc Graw Hill, Vol. I, 1989, Vol. II, 1991.
9. Miravete, A., *Materiales Compuestos I*, Ed. Antonio Miravete, 2000
10. S. T. Pinho, C. G. Dávila, P.P. Camanho, L. Iannuzzi, P. Robinson. *Failure Models and Criteria for FRP under In-Plane or Three-Dimensional Stress States Including Shear Non-Linearity*. NASA 2005
11. DNV, Recommended practice. *Fatigue Design of Offshore Metal structural*.
12. Compass IS, *Ship fatigue damage assessment (FDA) with RamSeries*.