

RamSeries

**Three-dimensional environment for structural analysis.
Reference manual and tutorials.**



RamSeries

Table of Contents

Chapters	Pag.
Introduction	1
Manual organization	1
Reference Manual	2
Item help	2
Units	2
Visual reference	3
Simulation data	4
Units	4
Gravity	5
Coupling data	5
General data	5
Analysis	5
Dynamic analysis data	7
Modal analysis	8
Spectrum analysis	9
Non-Linear analysis data	9
Automatic inc	10
Line-Search	10
Auto ARC-switch	10
Use Stabilization	10
Incremental analysis data	11
Results	11
Advanced	12
Linear solver	12
Eigen solver	12
Dynamic Output	13
Constraints	13
Constraints and prescribed displacements	13
Elastic constraints	14
Connections	14

Disconnect All	15
Disconnect Group	15
Disconnection examples: beams	16
Disconnection examples: shells	17
Contacts	17
Penalty contacts	18
Contacts Structural	19
Analysis configuration	19
Master-Slave contacts Surf-Surf	19
Master-Slave contacts Surf-Line	22
Mesh definition recommendations	23
Process Information	23
Advanced Contacts data	24
Dynamic conditions	25
Initial Conditions	25
Masses	25
Point mass	25
Beam mass	26
Surface shell mass	27
General properties	27
Global and Local axes for beams	27
Global and Local axes for shells	28
Beams	30
Rectangular Section	30
Generic Section	30
Sections Library	31
Naval Stiffeners	32
Damage Assesment	34
Cables	35
Cable section	35
Shells	35
Isotropic Shell	35

Orthotropic Shell	36
Laminate Shell	37
Stiffened Shell	38
Damage Assesment	39
Membranes	40
Isotropic membrane	40
Orthotropic membrane	41
Solid properties	42
Isotropic Solid	42
Orthotropic Solid	42
Materials	43
Load cases	43
Simple load cases	44
Combined load cases	44
LoadCase Properties	44
Waves data	45
Static loads	45
Point load	45
Self weight load	45
Beam Pressure load	45
Shell Pressure load	46
Shell Boundary Pressure load	48
Solid Surface load	48
Thermal strain load	49
Strain load	49
SeaFEM waves load (static)	49
Tdyn pressure load (static)	50
Tdyn thermal load (static)	50
Tdyn strain (static)	51
Dynamic loads	51
Dynamic Point load	52
Beam pressure load	52

Dynamic shell surface load	53
Dynamic Contour Pressure load	54
SeaFEM waves load (dynamic)	54
Tdyn pressure load (dynamic)	54
Tdyn thermal load (dynamic)	55
Tdyn strain (dynamic)	55
Morison load	56
Results	56
General results	56
Results in beams	57
Results in shells	59
Results in solids	61
Appendixes	63
Appendix 1: Graphics of results depending on the number of nodes	63
Square plate with uniform load	63
Square plate with punctual load	63
Beam analyzed as solid 3D	64
Appendix 2: Theory of beam analysis	65
Basic assumptions	65
Appendix 3: Theory of shell analysis	66
Basic assumptions	66
Appendix 4: Theory of 3D solid analysis	68
Basic assumptions	68
Appendix 5: Theory of dynamic analysis	69
Modal Analysis	69
Direct integration	70
Spectrum analysis	71
Appendix 6: Theory of nonlinear analysis	72
Nonlinear equations systems	73
J2 plasticity	73
Contact-impact	74
Appendix 7: Wave theory	75

Potential flow for waves	75
Appendix 8: Composite materials Theory	78
Composite sheet concept	78
Elastic properties of the composite sheet	78
Laminate shell concept	79
Laminate shell Theory	80
TSAI-WU Failure Criteria	80
Appendix 9: Buckling Analysis in FEA	80
Appendix 10: Fatigue Damage Assesment	83
Procedure	83
S-N curves	85
Appendix 11: Coupled Analyses	85
Coupling data	85
Coupled Analyses with Tdyn	87
Coupled Analyses with SeaFEM	87
Appendix 12: Stiffened Shell model	88
Inertia of the stiffened section	88
Equivalent model formulation	90
Stiffened shell test 1	92
Stiffened shell test 2	96
Stiffened shell test 3	97
Stiffened shell test 4	99
References	102

1 Introduction

RamSeries can perform the structural analysis of either beams, shells, solids, cables and membranes, or of 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.

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 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.

Item help

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

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 $\text{Pa} = \text{N}/\text{m}^2$

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 $\text{N} \cdot \text{m}/\text{m}$
 - Stresses are in $\text{Pa} = \text{N}/\text{m}^2$
- 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 $\text{N}\cdot\text{mm/mm}$
- 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 $\text{Kp}\cdot\text{cm/cm}$
 - 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 $\text{kN}\cdot\text{m/m}$ ($\text{kN}=10^3 \text{ N}$)
 - Stresses are in $\text{Mpa}=10^6 \text{ Pa}$

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

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.

Visual reference

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



Define groups



Data tree view



Define general data



Assign constraints



Assign contacts



Assign properties



Assign loads



Draw constraints and connections



Draw properties



Draw loads



Start calculation



View process info



Cancel calculation process

Simulation data

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

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.

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.

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.

Coupling data

These options are available when *Multiphysics Analysis* or *Coupled Seakeeping-Structural Analysis* are selected:

Simulation data ► **Simulation type**

Please refer to [Coupling data -pag. 85-](#) for further information on Coupled Analyses.

General data

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

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.

Internal triangular element:

- **DKT:** 3-noded Discrete Kirchoff triangular elements will be used.
- **6-noded elem:** If chosen, mesh generated as 3-noded triangle elements will be internally calculated as 6-noded triangles. It gives more precision in the results and needs more computer time. It should be always used except when strengths in the shell plane are dominant. In this last case, both options are feasible.
- **BEST:** Newly implemented rotation-free triangle element.

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

Yatch: Same as Naval, but considering also composite laminates.

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.

Dynamic analysis data

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

Type: There are three possible types of dynamic analysis:

- 1 .Direct Integration
- 2 .Modal Analysis
- 3 .Spectrum 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.

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)

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

Type: Modal analysis

Δt : 0.01 s

Number of steps: 10

Type of modal analysis: Number of modes

Number of modes: 10

OK Cancel

General

Type: Modal analysis

Δt : 0.01 s

Number of steps: 10

Type of modal analysis: Range of modes

Number of modes: 10

Range of modes, min. value (Hz): 0

Range of modes, max. value (Hz): 250

OK Cancel

If chosen, the following option appear:

Type of modal analysis: There are two options:

- **Number of modes:** Indicates the number of modes to be considered in the vibrations analysis.
- **Range of modes:** Natural frequencies between the range given will be calculated and only the number of modes indicated will be printed. If the number of modes is equal than 0, all modes between the range will be calculated and printed.

Spectrum analysis

If chosen, the following options appear:

Number of modes: Indicates the number of modes to be considered in the vibrations analysis.

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.

Non-Linear analysis data

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

(***Note:** **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:

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

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

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**.

***Note:**

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.

Incremental analysis data

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

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.

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 Misses in both, the face

up and the face down of the shell are calculated.

Smooth 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.

Advanced

This section treats advanced options for analysis.

Linear solver

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

- **Automatic:** The program chooses the best solver depending on the size of the problem.
- **Skyline:** This solver is not currently available for choosing from the GUI. It is just used internally, for natural frequency analysis. A Cholesky solver with skyline storage of the matrix is used. This is a direct solver that requires a bigger amount of memory. The advantage is that it always gives a solution if the problem is correctly defined. If there are several load cases, a direct solver can save a lot of computing time.
- **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.

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 **Automatic**, or **Hybrid-Sparse**.

Eigen solver

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.

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. This information RamSeries has the possibility to choose what results are required (Write Displacements, Write Velocities, Write Accelerations, Write Strengths and Write Reactions)

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

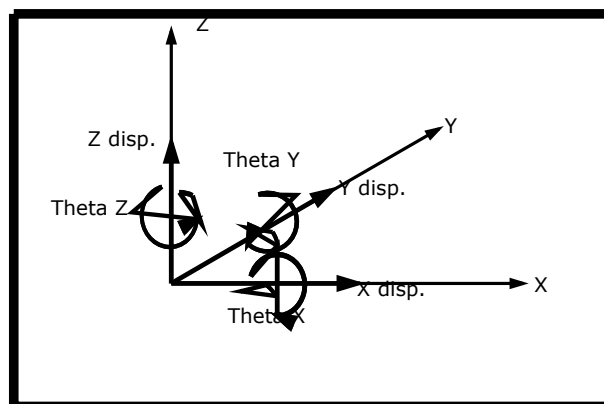
Constraints

Conds. & Init. Data > Constraints

Constraints and prescribed displacements

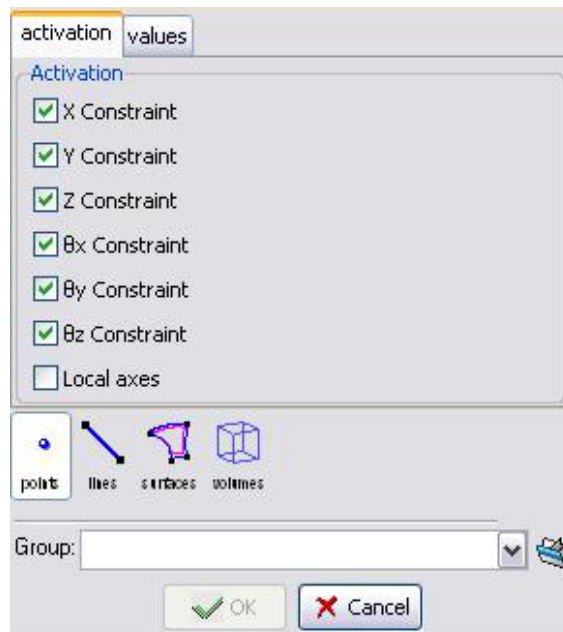
The parts of the structure that have any type of external constraint in their movements, are called constraints. These constraints can be applied for every degree of freedom (X, Y, Z and rotations for beams and shells), and for every node in the model

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.



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.

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.

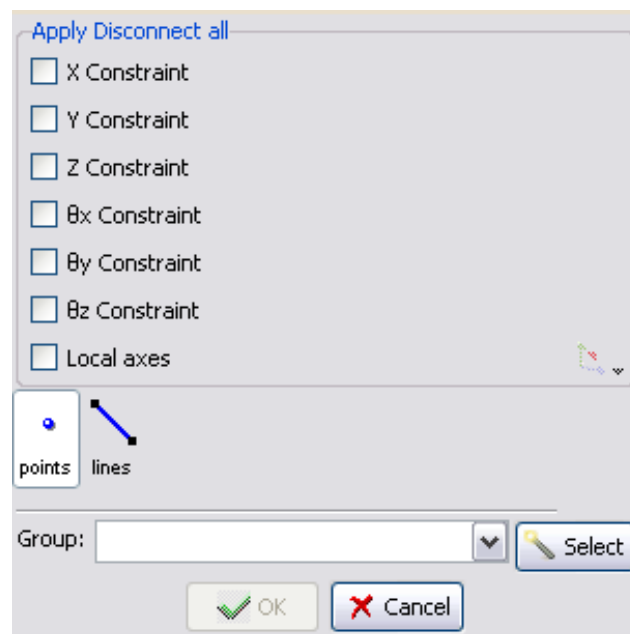
The disconnection of several degrees of freedom can be made in several ways. The most simple ones are those known as **rotules**.

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.

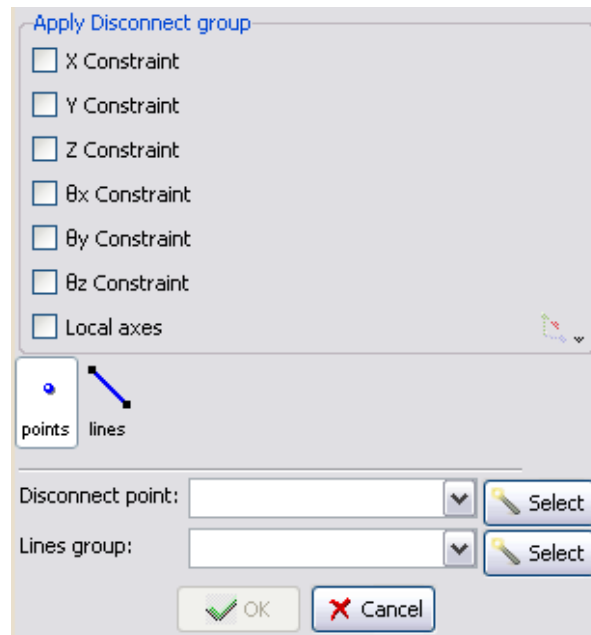
The elements that belong to the group must be marked with condition **Disconnect Group**.

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

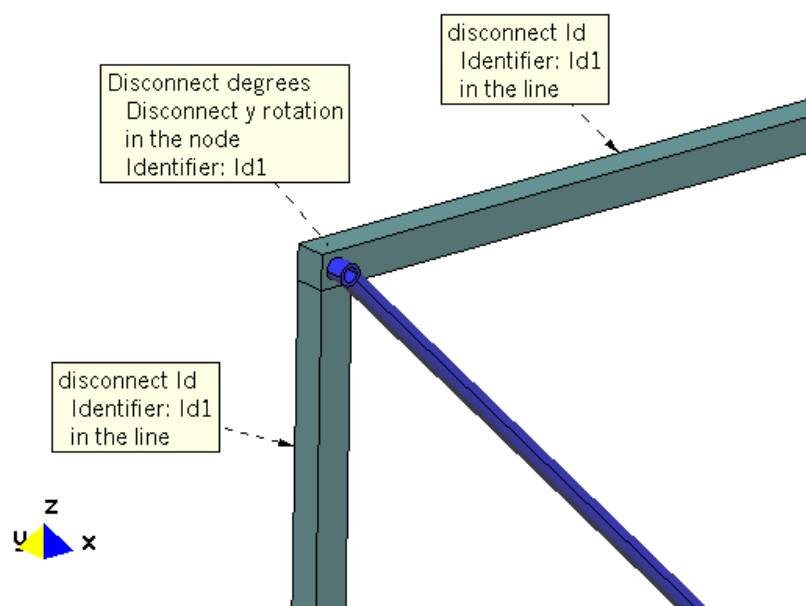


Disconnect Group

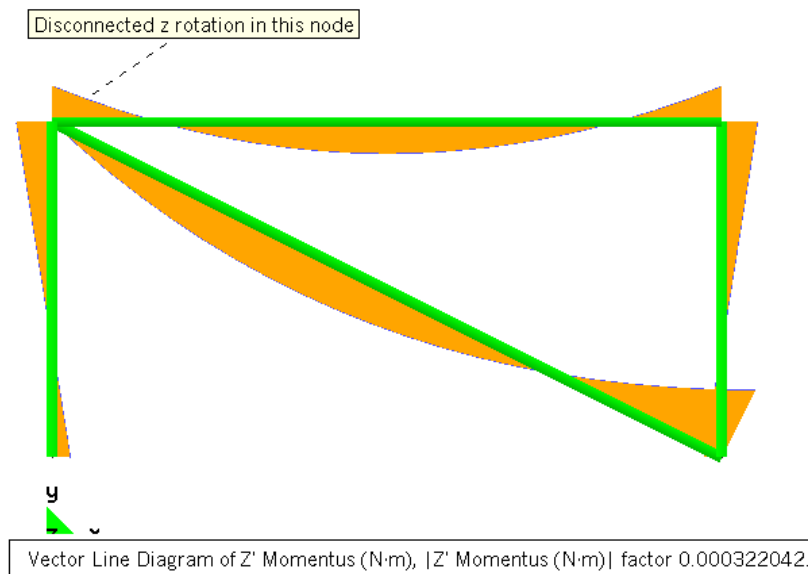
This condition is used in collaboration with the condition **Disconnect All** and is used to mark the elements that belong to a common part with a common **Group**.



Disconnection examples: beams

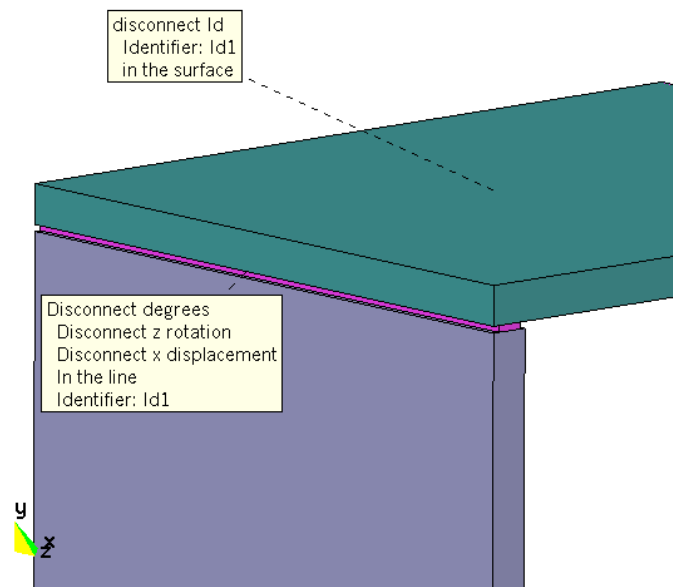


In this example, one part, called Id1, is defined with the two bars that have a rigid connection between them. Degree **y rotation** is disconnected from the rest of the bars. So, the third bar has free rotation related to these two.



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

Contacts

Penalty contacts

In the 3D solids RamSeries can perform an Contact-Impact analysis between bodies.

To model Contact-Impact between bodies using RamSeries, it's necessary to create the geometry of each body and indicate the potential contact areas. To define such contacts, RamSeries has two options:

1-) Use the menu options:

Geometry ► Create ► Contact ► Surface

and

Geometry ► Create ► Contact ► Volume

to create contact between lines and surfaces, respectively. This option allows to automatically generating contact elements that join the nodes of the impact bodies. Nonetheless, it can only be used when the two surfaces or volumes to connect are homogeneous.

2-) Connect explicitly the potential points of contact starting from lines. This option is very useful when it's known beforehand the potential nodes to be in contact.

With the condition Contact can be indicated which are the contact elements between bodies. This condition can be applied to lines, surfaces or volumes.

Penalty indicates the stiffness of the contact element.

Contacts Structural

Information for analyses with contacts among elements.

Analysis configuration

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

- **Analysis:**
 - **Analysis type:** for static analysis **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 analysis with contacts it is recommended to use **Energy Conserving** as **Integration method**.

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.

Apply Master-Slave contacts Surf-Surf

Contact type: Sticking

Assign Master: [dropdown] [Select]

Assign Slave: [dropdown] [Select]

[OK] [Cancel]

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.

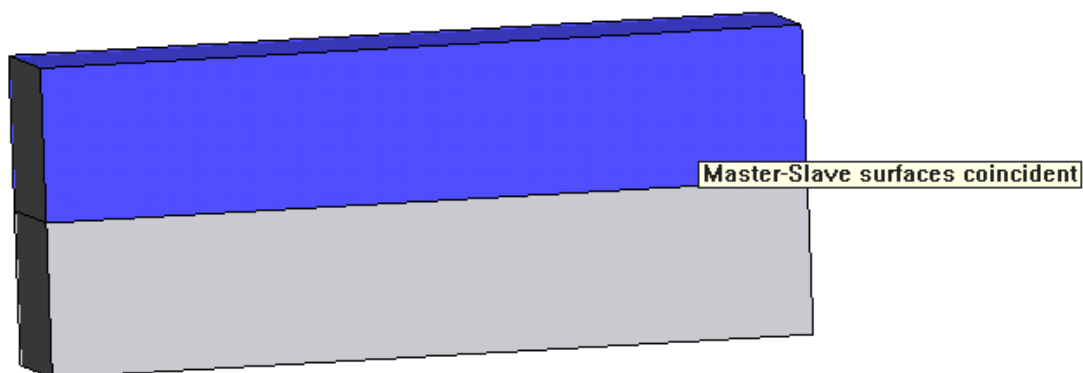
The contact domain must be defined selecting the pair of surfaces where there will be contact. It is possible to choose shell's surfaces or solid's surfaces.

One surface will be assigned as **master** and the other as **slave**.

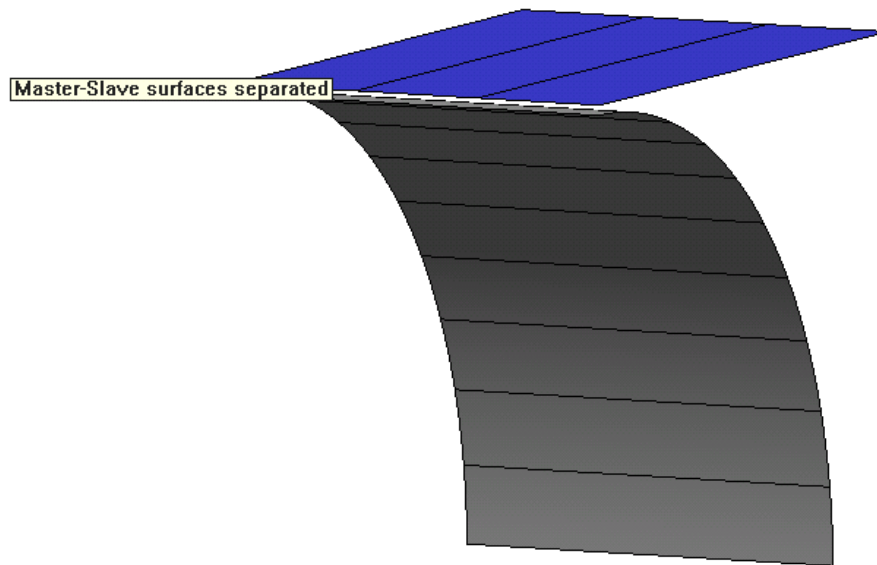
There is not a fix rule about which surface must be **master** and which must be **slave**. However, it is advisable to choose as master surface the more rigid. On the other hand, stresses in slave surface will be more accurate than in master slave, so choosing as slave the surface where a better accuracy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

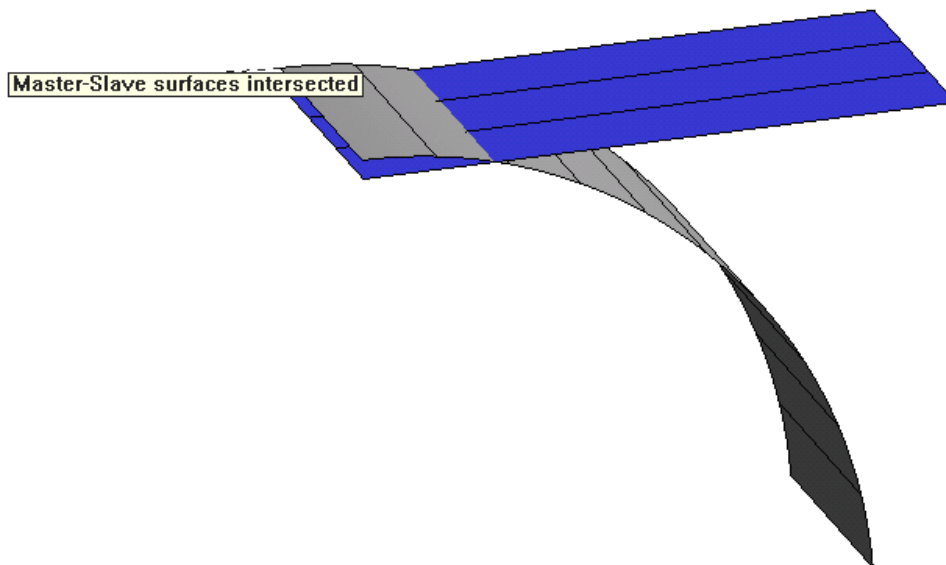
Example of coincident master and slave surfaces.



Example of separated master and slave surfaces:



Example of intersected master and slave surfaces:



In this case, it is necessary to define master surfaces's normals and deactivate **Normals sense swap** (see advanced contacts data).

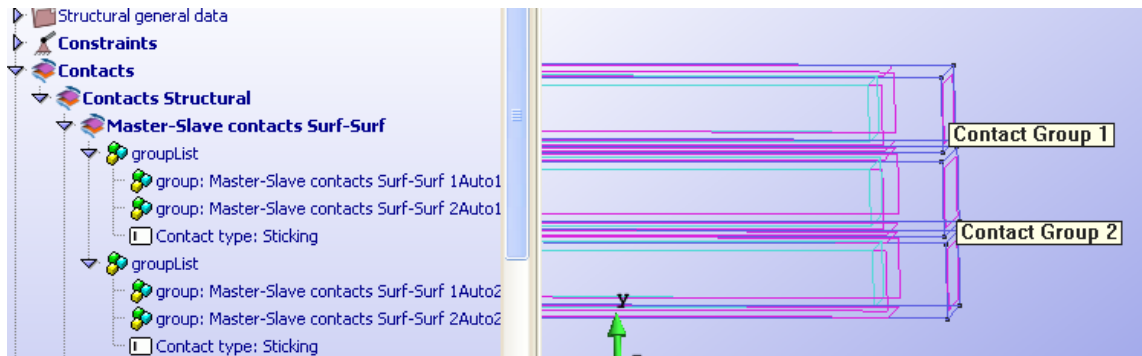
Fix constraint in slave nodes

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.

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.

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.

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:



View process info

Following, an extract of process info is shown:

```

Increment      4

Starting solution process
Total Hybrid-Sparse solver time = 0.021 min. = 1.3 seg.
Current stiffness factor= 2.19883
Total LD factor= 0.4, Inc factor=0.1 and Inc Length= 1.35927
Number of contacts added or changed: 57
Number of active contacts before iterations: 57
Iteration 1: .....Convergence factor=0.745542
Iteration 2: .....Convergence factor=0.0240676
Iteration 3: .....Convergence factor=3.42056e-007
Maximum traction force=0.0391899 in node=5810
Number of disconnected contacts: 3
Iteration 4: .....Convergence factor=0.111426
Iteration 5: .....Convergence factor=0.000574928
Iteration 6: .....Convergence factor=4.55896e-007
Maximum penetration=-5.79217e-008 in node=6166
Number of contacts added or changed: 54
Iteration 7: .....Convergence factor=0.00132538
Iteration 8: .....Convergence factor=0.0240671
Iteration 9: .....Convergence factor=3.10638e-007
Maximum penetration=-4.50313e-013 in node=5848
Number of active contacts after iterations: 54

Highest penetration: -4.50313e-013. Node: 5848
    
```

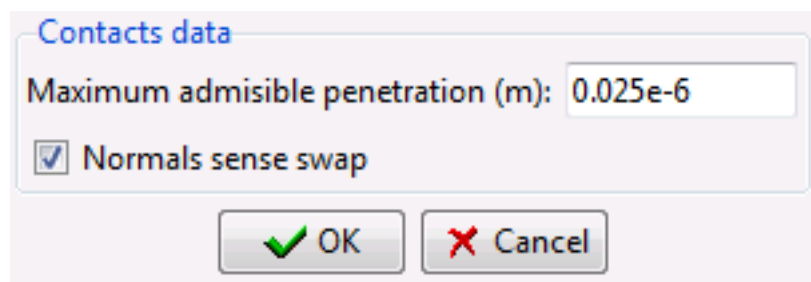
At the begining of convergence bucle, the number of contacts added or changed are shown. A contact changes when the slave node changes its master element or the position of contact.

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. The process info shows the number of contacts disconnected, added or changed, as well as the highest penetration and traction force.

Advanced Contacts data

Advanced contacts data is available in

Data ► Structural general data ► Advanced ► Contacts data



When iteration routine is executed, if there are slave nodes with penetration minor than

maximum admissible, new contacts will not be created.

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

Dynamic conditions

Initial Conditions

In the Linear-Dynamics and Non-Linear Dynamics analysis the Initial Conditions of the structured 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.

Apply Initial conditions

Displacement x: 0 m

Displacement y: 0 m

Displacement z: 0 m

Velocity x: 0.0 m/s

Velocity y: 0.0 m/s

Velocity z: 0.0 m/s

points lines surfaces volumes

Group: Select

OK Cancel

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.

The 'Apply Masses' dialog box contains the following fields and controls:

- Mass x:** 0.0, kg
- Mass y:** 0.0, kg
- Mass z:** 0.0, kg
- M. inercia x:** 0, kg·m²
- M. inercia y:** 0, kg·m²
- M. inercia z:** 0, kg·m²
- Selection Mode:** Three icons for 'points' (a blue dot), 'lines' (a blue line segment), and 'surfaces' (a purple polygon). The 'points' icon is currently selected.
- Group:** A dropdown menu.
- Buttons:** 'Select' (with a key icon), 'OK' (with a green checkmark), and 'Cancel' (with a red X).

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.

The 'Apply Masses' dialog box for beam elements contains the following fields and controls:

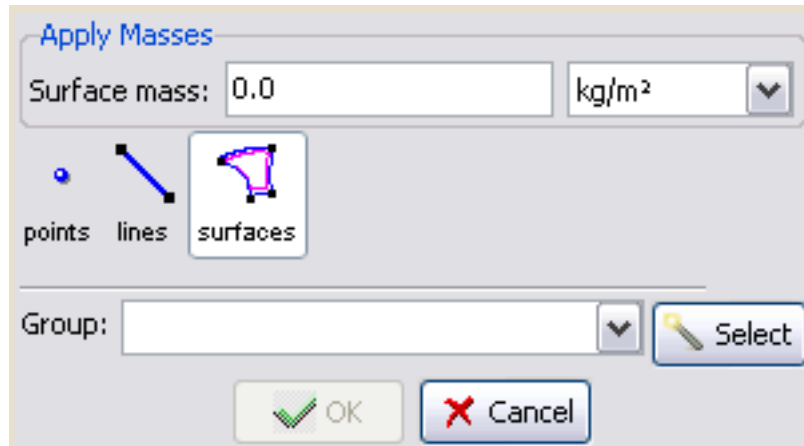
- Trasl mass:** 0, kg/m
- M. inercia x:** 0, kg·m
- M. inercia y:** 0, kg·m
- M. inercia z:** 0, kg·m
- Selection Mode:** Three icons for 'points' (a blue dot), 'lines' (a blue line segment), and 'surfaces' (a purple polygon). The 'lines' icon is currently selected.
- Group:** A dropdown menu.
- Buttons:** 'Select' (with a key icon), 'OK' (with a green checkmark), and 'Cancel' (with a red X).

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 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.



General properties

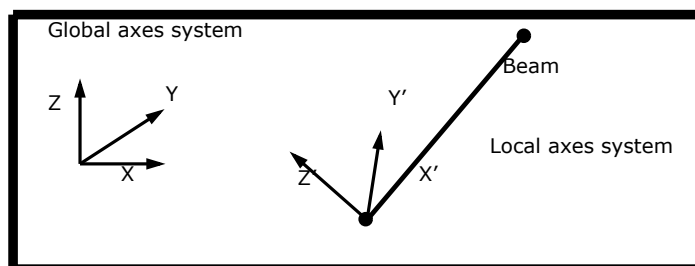
These properties define both the material parameters and the sections, shells, solids, membranes and cables parameters.

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.
- 3 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 axe must have the same direction as the beam. than 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' axe has the direction of the beam.
- If X' axe has the same direction than global Z axe, Y' axe has the same direction than global X. If not, Y' axe is calculated so as to be horizontal (orthogonal to X' and Z).
- Z' axe is the cross product of X' axe and Y' axe. 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' axe in the direction of global X. All the other beams have the Y' axe horizontal and with the Z' axe 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 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.

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

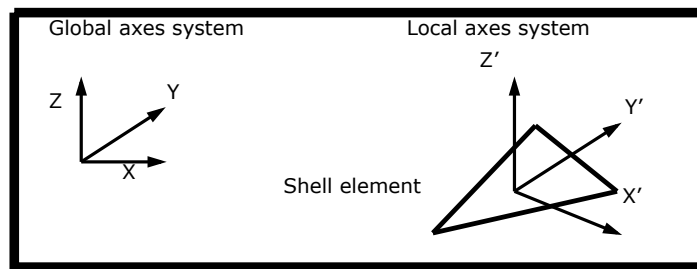
Note 2: The final local axes system for every beam can be visualized in the post-process stage. It is convenient to check the correctness of these systems after calculation is performed.

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.
- 2 Some of the loads (that have the prefix *Local*) are related also to this system.
- 3 Strength results over the shell are referred to this local axes system.
- 4 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:

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** ($U \times N$).
- If not, X' axe will be in the direction of the cross product of **V** and **N** ($V \times N$).
- 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 ► 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.

Beams

Rectangular Section

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

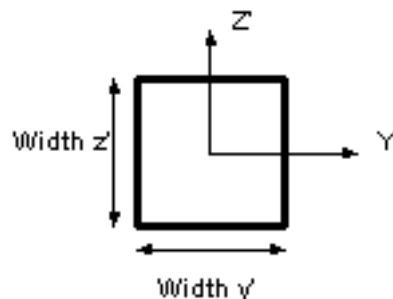
Apply Rectangular section

Width y:	0	m	▼
Width z:	0	m	▼
Material:	User defined ▼		
E:	2.1e11	N/m ²	▼
G:	8.1e10	N/m ²	▼
Specific weight:	0.0	N/m ³	▼
Maximum stress:	0.0	Pa	▼

Local Axes: Open ▼

Group: ▼ Select

OK Cancel



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: $G = E / (2 \cdot (1 + \nu))$ (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 m^2 and for the inertia modulus are m^4 .

The 'Data' dialog box is shown with the following fields and units:

- Area: 0 m^2
- Inertia y: 0 m^4
- Inertia z: 0 m^4
- J: 0 m^4
- Material: User defined
- E: $2.1e11$ N/m^2
- G: $8.1e10$ N/m^2
- Specific weight: 0 N/m^3

Below the fields, there is a 'Local Axes: Open' dropdown, a 'Predefined sections' button, a 'Group:' dropdown, and a 'Select' button. At the bottom are 'OK' and 'Cancel' buttons.

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.

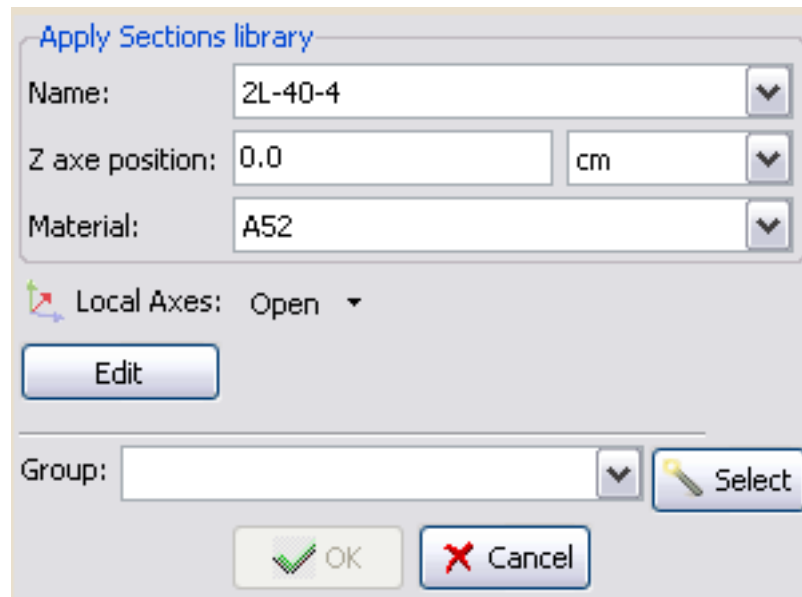
The 'Sections' dialog box is shown with the following sections:

- Sections:** A dropdown menu showing 'Double T'.
- Visual Description:** A diagram of a Double T section.
- Moments of Inertia:** A list of properties: Area, I_{yy} , I_{zz} , I_{yz} , and J, each with a unit dropdown menu.
- Section Properties:** Fields for Top Width, Bottom Width, Height, Wings Thick, and Central Thick, each with a unit dropdown menu. A 'Calculate' button is below these fields.
- Origin of Coordinates:** Fields for Y center and Z center, each with a unit dropdown menu. A 'Centroid' button is below these fields.

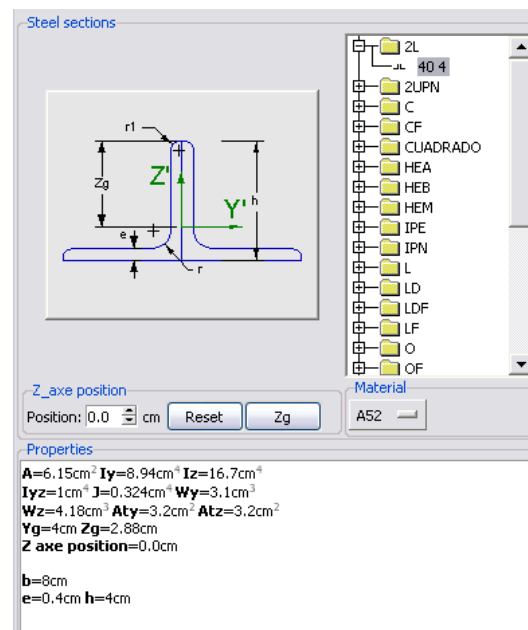
At the bottom are 'OK', 'Save...', and 'Cancel' buttons.

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.

Apply Naval stiffeners

Area: 0.0 m² ▾

G: 0.0 N/m² ▾

E: 0.0 N/m² ▾

Specific weight: 0.0 N/m³ ▾

Top laminate:

Base laminate:

Height laminate:

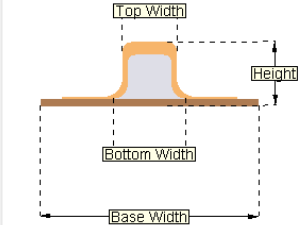
Local Axes: Open ▾

Group:

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

Composite naval stiffeners

Visual description



Section properties

Units: N-m-kg ▾

Top width: 0.0 m Top laminate:

Bottom width: 0.0 m

Base width: 0.0 m Base laminate:

Height: 0.0 m Height laminate:

Core properties

G: 0.0 N/m²

Section properties

Area: 0.0 m²

I_{yy}: 0.0 m⁴

I_{zz}: 0.0 m⁴

Specific weight: 0.0 N/m³

E: 0.0 N/m²

J: 0.0 m⁴

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:

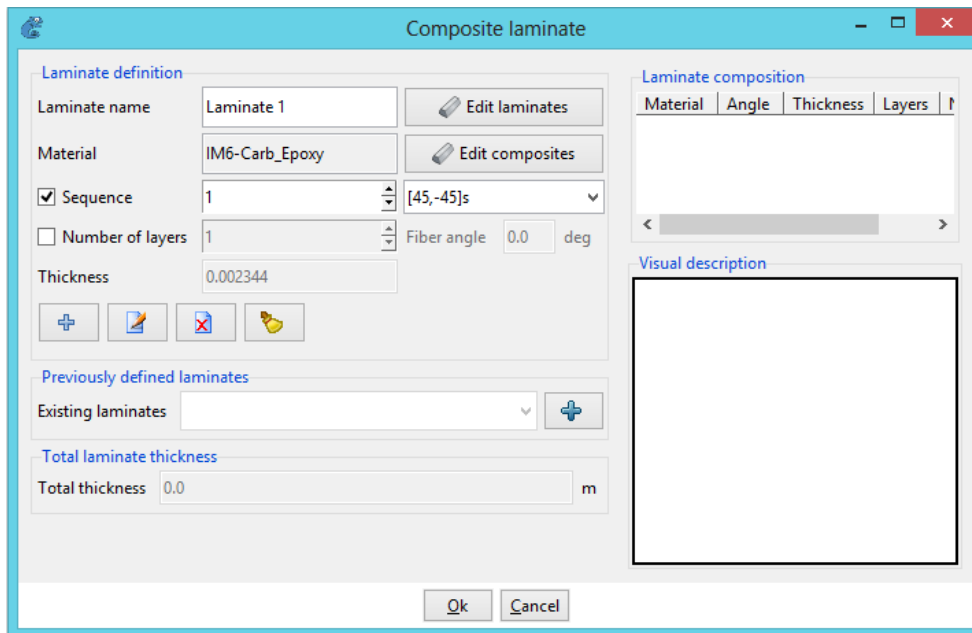
Materials and properties ▶ **Materials** ▶ **Composite**

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 data tree.



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 -pag. 83-](#)).

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.

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.

Apply Welded joints

k:	0.0
m:	4.0
log(a):	15.117
Fatigue limit (MPa):	106.97
Welded connections:	Welded Tubular Joints
Thickness (mm):	25
SCF:	1
Max. stress factor:	1.0
Min. stress factor:	1.0

SN curves

Group:

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.

Cables

Cable section

Data

Area:	0.0	m ²
Material:	User defined	
E:	2.1e11	N/m ²
Specific weight:	0.0	N/m ³
Pre Stressing:	0.0	N/m ²

☒ Is compressive force allowed

 Local Axes: Open

Group:

Shells

Isotropic Shell

Apply Isotropic shell

Thickness: 0 m

Material: User defined

E: 2.1e11 N/m²

v: 0.3

Specific weight: 0 N/m³

Local Axes: Open

Group: [] Select

OK Cancel

Orthotropic Shell

Properties defined in this condition are similar to that of the *Isotropic shell* 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:

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

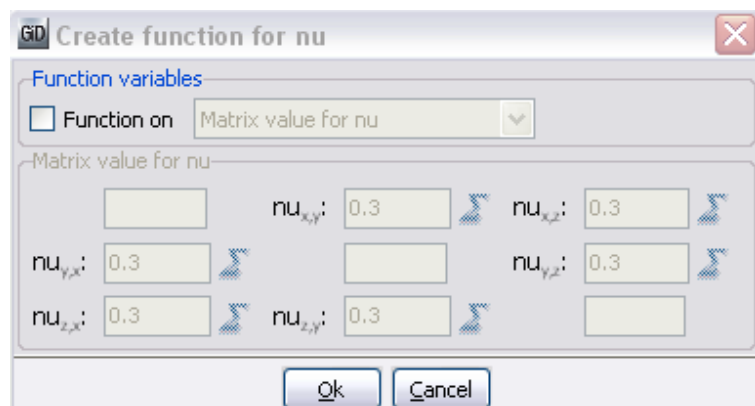
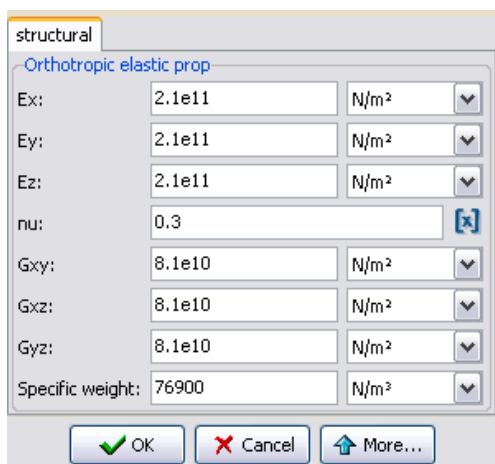
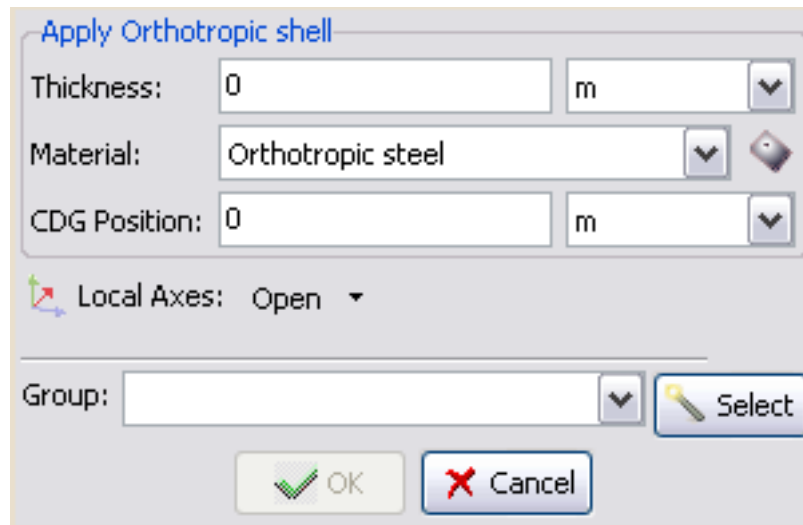
Default units for E and G are N/m^2 and ν is non-dimensional.

Remember that for an isotropic material:

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

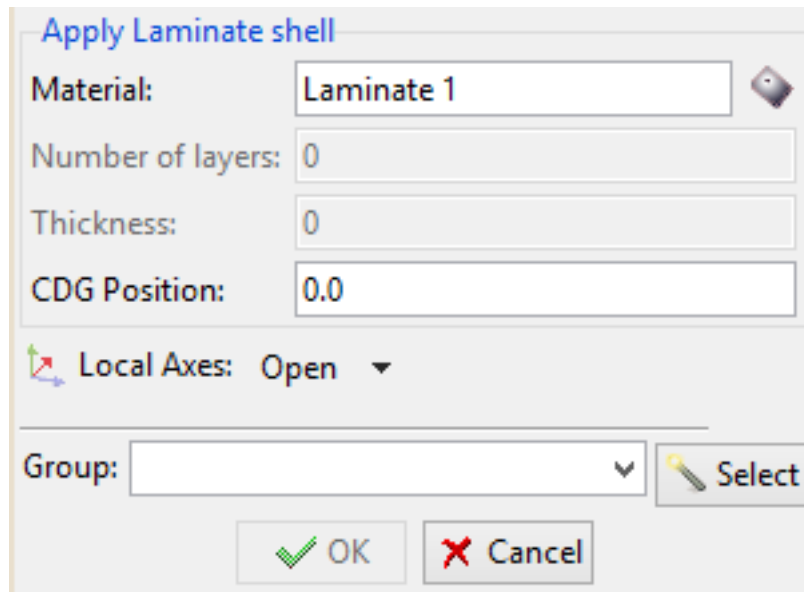
For and orthotropic material:

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



Laminate Shell

As well as for the "Naval stiffeners", it is possible to assign laminated fiber reinforced plastic properties to shells.

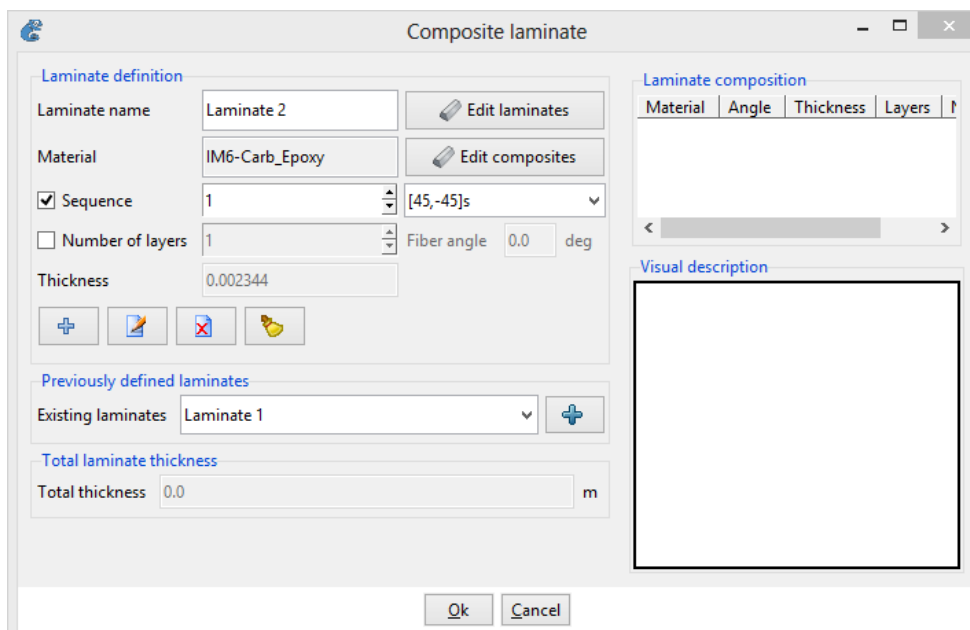


Clicking on the button "**Edit Laminate**", the list of the existing laminates is displayed.

The data tree item

Materials and properties ▶ **Materials** ▶ **Laminate**

allows to define the laminates materials using the Composite laminate window. It should be noted that this window is also available in the **Laminate** contextual menu.



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).

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 -pag. 83-).

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.

By assigning this condition to any surface of the geometry, the user is indicating the program 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

Membranes

Isotropic membrane

Apply Isotropic membrane

Thickness: 0.0 m

Material: User defined

E: 2.1e11 N/m²

ν : 0.3

Specific weight: 0.0 N/m³

Minimum compression stress: 0.0

Iteration Goal: 0.0

Local Axes: Open

Group: Select

OK Cancel

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

Apply Orthotropic membrane

Thickness: 0.0 m

Minimum compression stress: 0.0

Iteration Goal: 0.0

Material: Orthotropic steel

Local Axes: Open

Group: Select

OK Cancel

Solid properties

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 (2.1×10^{11} N/m² for steel) and **v** 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:

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

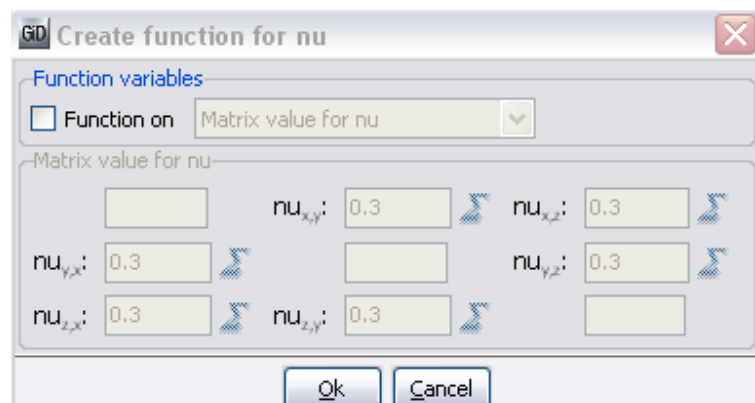
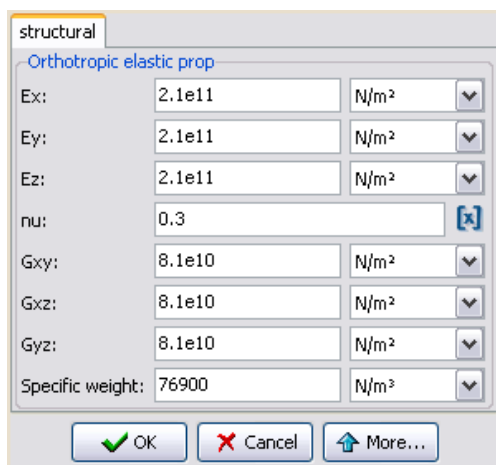
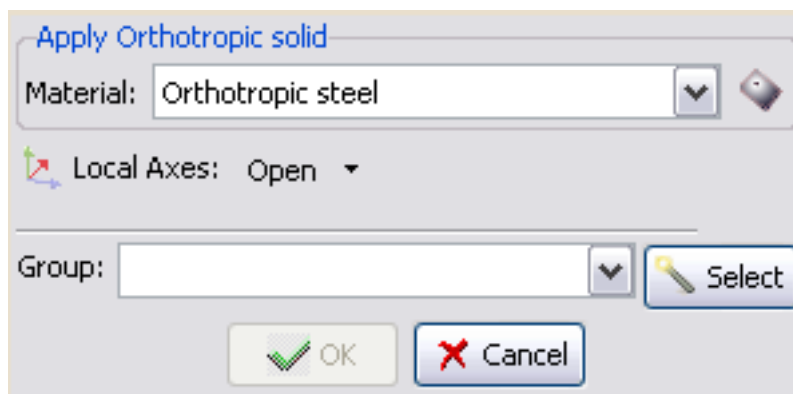
Default units for E and G are N/m² and v is non-dimensional.

Remember that for an isotropic material:

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

For and orthotropic material:

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



Materials

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

Properties ► Materials

The properties of these materials can also be edited.

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

Load cases

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.

Combined load cases

Combined load cases are defined in the load cases window

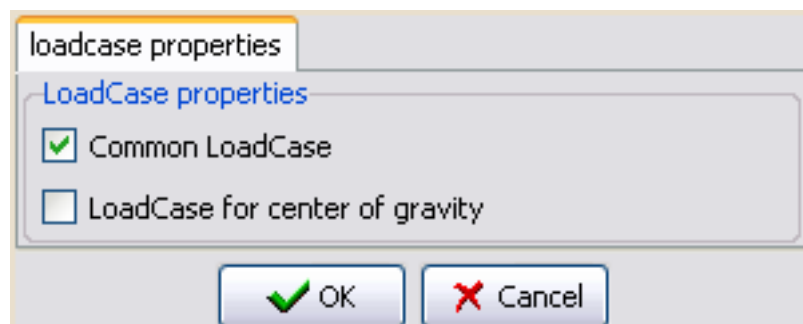
One default combined load case is already existent. To create more, press right mouse button over the existent combined load case name. A menu appears that offers several options.

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

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).

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.

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.

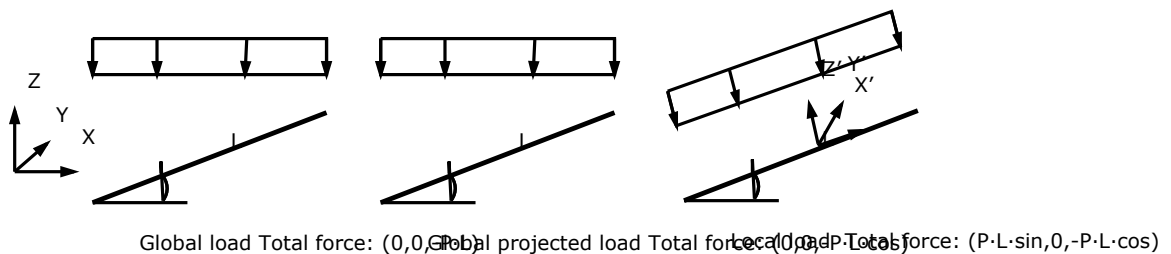
This condition can be applied to either lines for beams, surfaces for shells and volumes for the solid analysis.

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. A load of $(0,0,-P)$ N/m in either case would be (for a beam of length L):



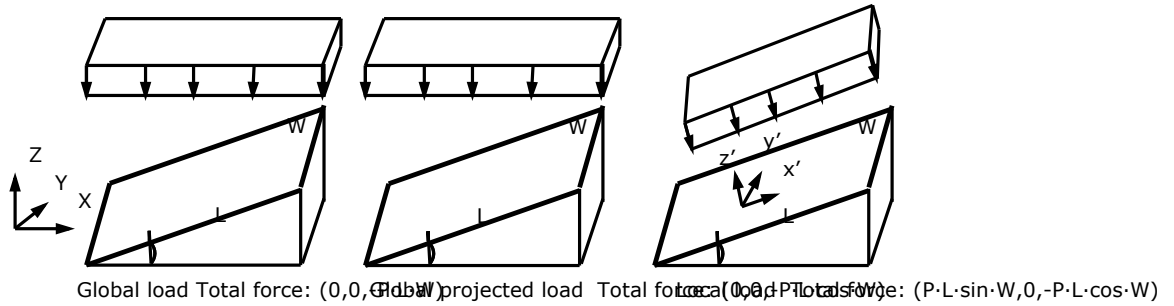
In the Local beam load there is an additional field that is the local torque momentum.

Shell Pressure load

There are five types of surface loads for shells:

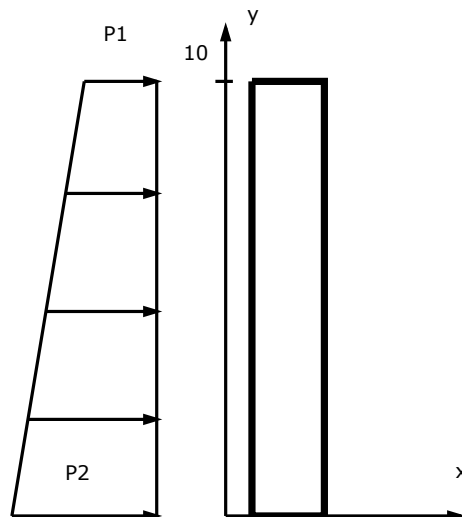
- 1 Global shell load
- 2 Global projected shell load
- 3 Local shell load
- 4 Triangular load
- 5 Hydrostatic load

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. A load of $(0,0,-P)$ N/m² in either case would be (for a shell of length L and width W):

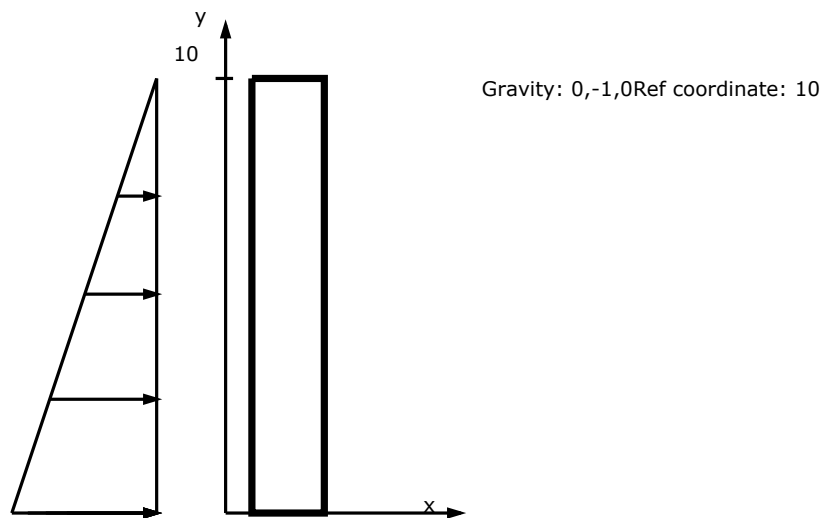


Triangular load is like a Global shell load but with a triangular variation in its values. It is defined by two points, given by its coordinates, and pressure values associated to each one of these points. The pressure assigned to the elements that project between the points is a linear interpolation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

Example:



Point 1: 0,10,0 Pressure 1: P1,0,0 Point 2: 0,0,0 Pressure 2: P2,0,0



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/m^3 .

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

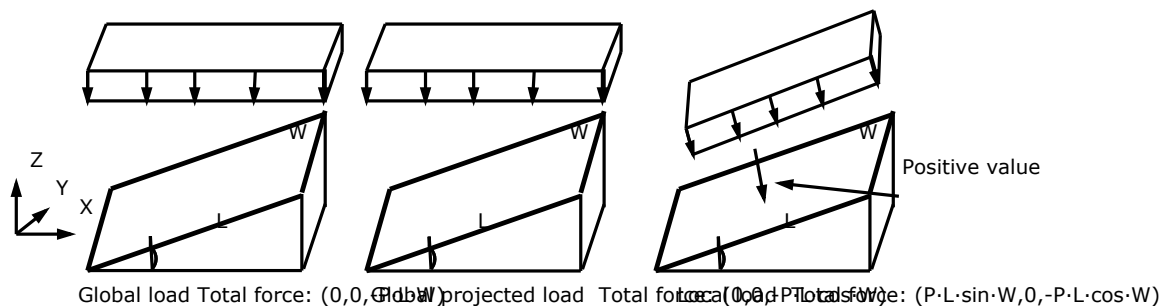
Solid Surface load

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. A load of (0,0,-P) N/m² in either case would be (for a solid contour surface of length L and width W):

Note: In the local load the value would be +P



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

Thermal strain load

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: $\varepsilon = \alpha \cdot \Delta T$

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: $\varepsilon = \alpha \cdot \Delta U$

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.

Apply Seakeeping waves load

Time step: 0.0 s

Input file: [file icon]

Edit

Group: [dropdown] Select

OK Cancel

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.

Apply Tdyn pressure load

Time step: 0.0 s

Input file: [file icon]

Edit

Group: [dropdown] Select

OK Cancel

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.

Apply Tdyn thermal load

α : 0.000012 1/ Δ K

T₀: 0.0 K

Time step: 0.0 s

Input file:

Edit

Group: Select

OK Cancel

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.

Apply Tdyn strain

Time step: 0.0 s

Input file:

Edit

Group: Select

OK Cancel

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.

Create function for Factor

Function variables

☐ Function on geometry

☒ Function on time

Interpolation function at t

Factor=f1(t)

Interpolation function at t

data entry

☒ Single value ☐ Linear ramp ☐ Equation

t units: s

t: 0.0 Factor: 0.0

Add at end Add before Clear all

Ok Apply Delete Cancel

Dynamic Point load

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. A load of $(0,0,-P(t))$ N/m would take on the following form for each case respectively (considering a beam of length L): in either case would be (for a beam of length L):

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.

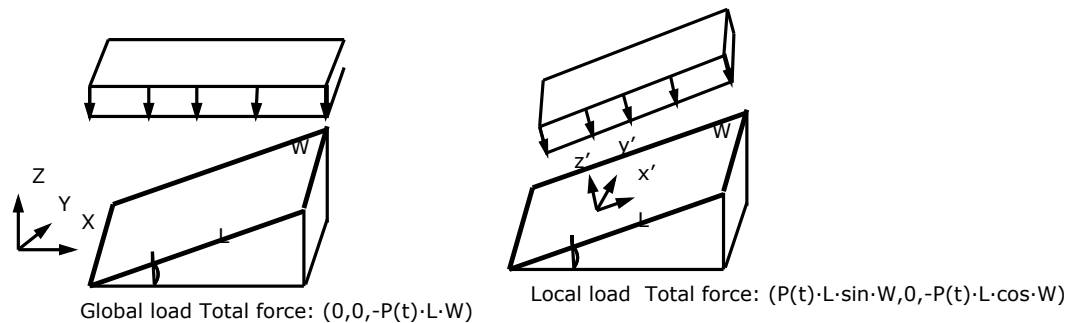
Dynamic shell surface load

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. A load of $(0,0,-P(t))$ N/m² would take on the following form for each case respectively (considering a shell of length L and width W) in either case would be (for a shell of length L and width W):

Y'



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:

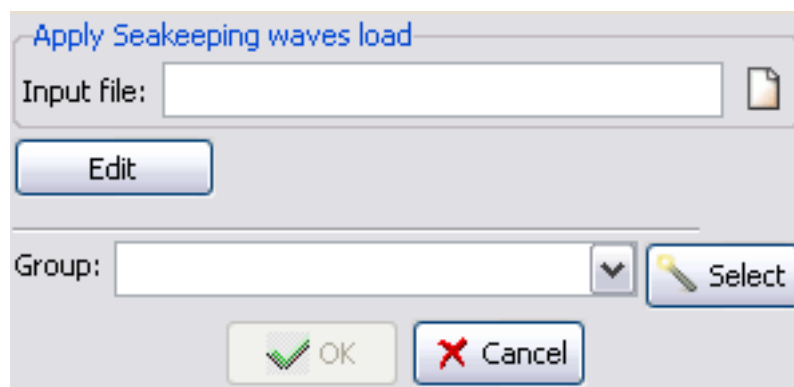
Data ► Local axes ► Surfaces ► Draw

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.

Apply Tdyn strain

Input file: 

Group: 


Morison load

Apply Morison loads

Axis Dir. X:

Axis Dir. Y:

Axis Dir. Z:


Cyl. Diam.: 

☐ Coeffs. Definition

Cd:

Cm:

Surface Roughness:

Group: 

Results

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

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
Velocities	Vel_X, Vel_Y, Vel_Z, velocities	m/s	Absolute velocities in global axes. Only available in dynamic analysis.
Accelerations	Accel_X, Aceel_Y, Accel_Z, accelerations	m/s^2	Absolute accelerations in global axes. Only available in dynamic analysis
Vibrations modes	One for every mode	m	Absolute displacements in global axes. Only available in dynamic analysis
Reactions	X,Y,Z	N	They are nodal vectors
Momentus reactions	Mx,My,Mz	N·m	They are nodal vectors. Not applicable for solids

Results in beams

Local axes and strengths for beams.

Name		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	$M_{y'}, M_{z'}$	$N \cdot m$	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	$N \cdot m$	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:

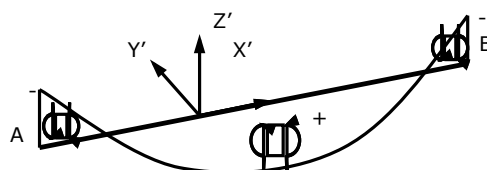


Diagram is drawn in the plane $X'Z'$ and in the side of the beam where the traction is. Positive values of the momentum mean that traction is in the $-Z'$ side (in the negative side of Z').

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

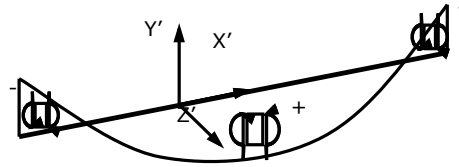
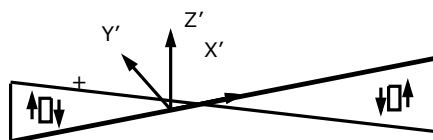


Diagram is drawn in the plane X'Y' and in the side of the beam where the traction is. Positive values of the momentum mean that traction is in the -Y' side (in the negative side of Y').

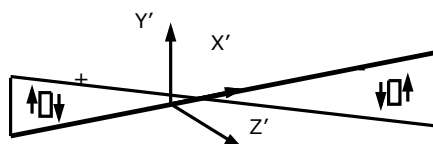
Shear in Z' is the shear in the X'Z' plane. Sign criteria is:

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.

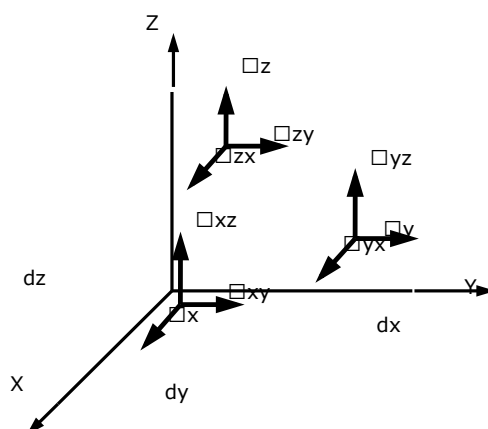
Results in shells

Local axes and strengths for shells

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

Axial force	$N_{x'}, N_{y'}, N_{x'y'}$	N/m	Axial force in all the shell thickness per unit width of the shell
Momentum	$M_{x'}, M_{y'}, M_{x'y'}$	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:

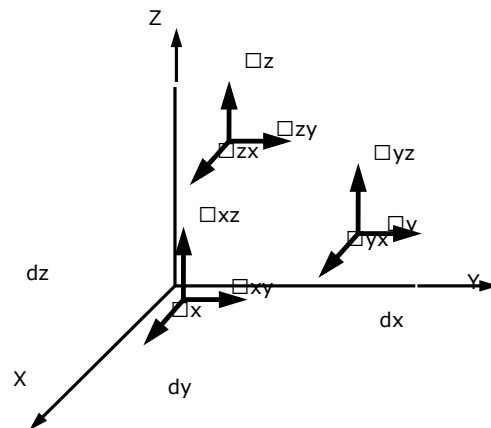


Results in solids

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:



3 Appendixes

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

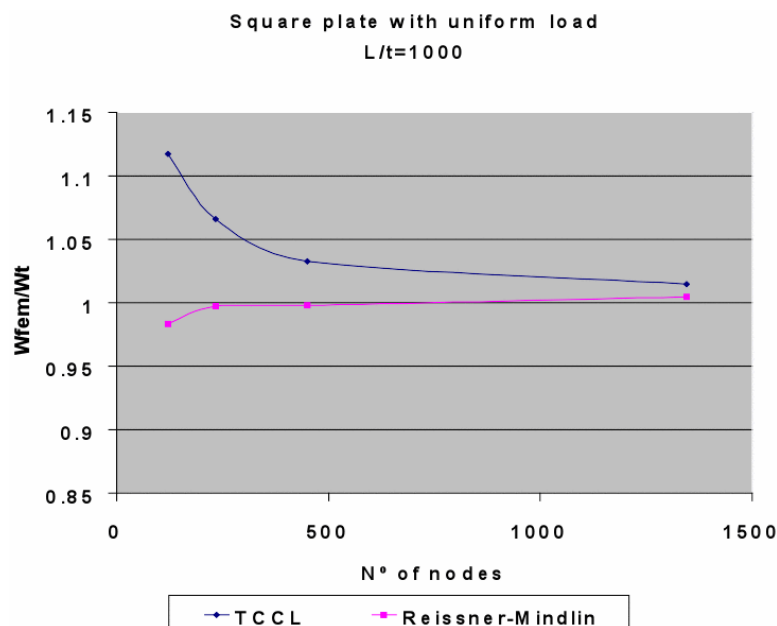
These graphics compares the analytical solution of the classical problem of a thin square plate with different types of loads. The variable compared is the displacement of the center of the plate. W_t is the theoretical value and W_{fem} is the result of the finite element analysis for different meshes.

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

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.

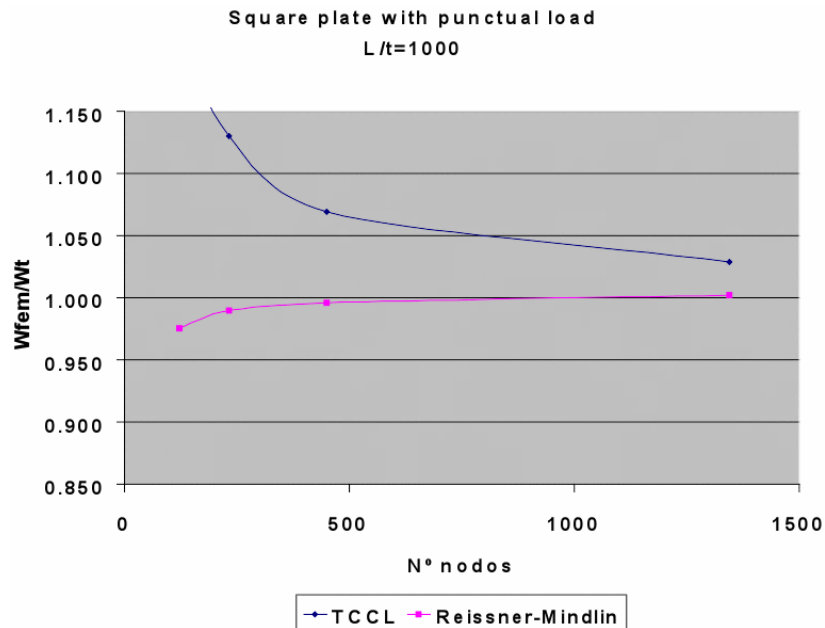


Note: The element called Reissner-Mindlin is the 6-noded triangle implemented in RamSeries.

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.

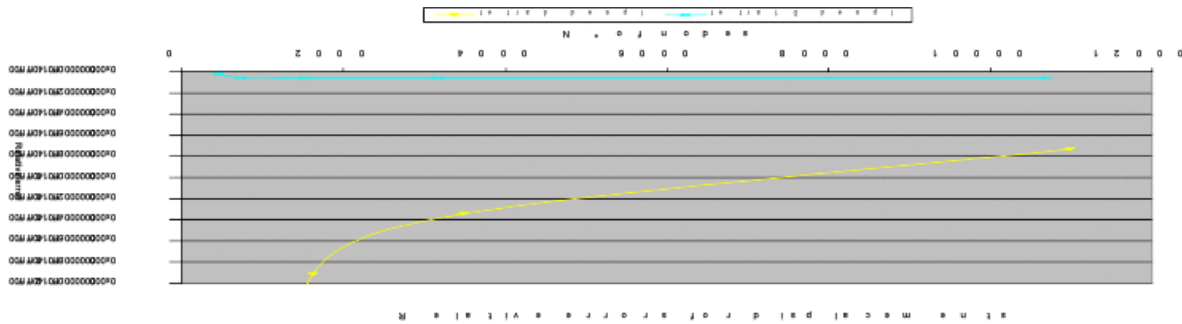
The ratio span/thickness is 1000.



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

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.

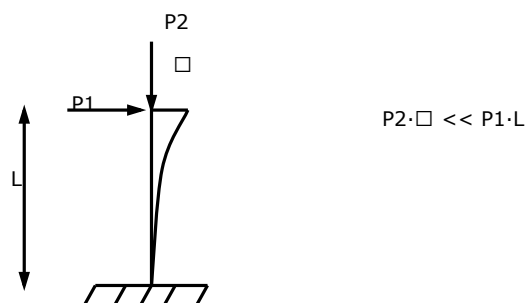


Appendix 2: Theory of beam analysis

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.

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

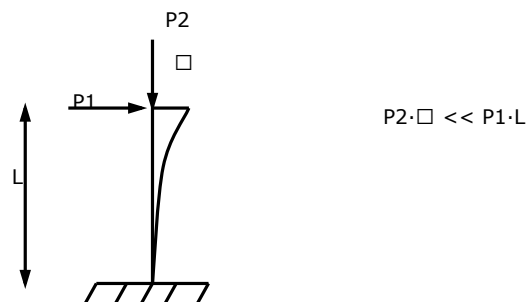
$$M_T = \frac{GJ}{L} \Delta$$

Appendix 3: Theory of shell analysis

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

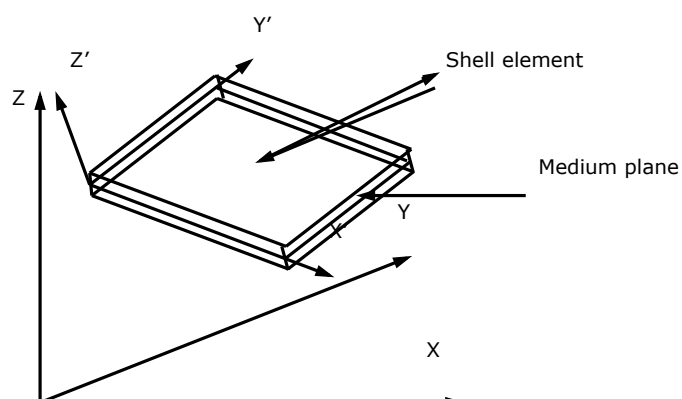
Basic assumptions

The basic assumptions that are assumed by RamSeries for the shell 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 considered for shells



The analysis of shells is based on simplifying the representation of a real solid, with one of its dimensions much smaller than the other two, with its medium plane.

The Reissner-Mindlin hypothesis:

- 5 All the points belonging to one normal to the medium plane have the same vertical displacement (in z' sense).
- 6 Normal stress σ_z is negligible.
- 7 The points that before the deformation belonged to one normal to the medium plane, after deformation continue belonging to one straight line, not being this, necessarily, orthogonal to the deformed medium plane.

From these assumptions it is possible to extract the following equations:

$$\begin{bmatrix} z_{x'} \\ z_{y'} \\ \gamma_{x'y'} \\ \dots \\ \gamma_{x'z'} \\ \gamma_{y'z'} \end{bmatrix} = \begin{bmatrix} \frac{\partial u'_0}{\partial x'} \\ \frac{\partial v'_0}{\partial y'} \\ \frac{\partial u'_0}{\partial y'} + \frac{\partial v'_0}{\partial x'} \\ \dots \\ 0 \\ 0 \end{bmatrix} + \begin{bmatrix} -z' \frac{\partial \theta_{x'}}{\partial x'} \\ -z' \frac{\partial \theta_{y'}}{\partial y'} \\ -z' \left(\frac{\partial \theta_{x'}}{\partial y'} + \frac{\partial \theta_{y'}}{\partial x'} \right) \\ \dots \\ \frac{\partial w'_0}{\partial x'} - \theta_{x'} \\ \frac{\partial w'_0}{\partial y'} - \theta_{y'} \end{bmatrix}$$

where

$$u'_0, v'_0, w'_0, \theta_{x'}, \theta_{y'}$$

are the 5 degrees of freedom that, adding the 6th degree θ_x that comes from joining the shells in the space, give the total 6 degrees of freedom that every node has.

$$\begin{bmatrix} \sigma_{x'} \\ \sigma_{y'} \\ \tau_{x'y'} \\ \dots \\ \tau_{x'z'} \\ \tau_{y'z'} \end{bmatrix} = \begin{bmatrix} D'_{11} & \dots & 0 \\ \dots & \dots & \dots \\ 0 & \dots & D'_{33} \end{bmatrix} \begin{bmatrix} z_{x'} \\ z_{y'} \\ \gamma_{x'y'} \\ \dots \\ \gamma_{x'z'} \\ \gamma_{y'z'} \end{bmatrix}$$

$$D' = \frac{1}{1 - \nu_{x'y'}\nu_{y'x'}} \begin{bmatrix} E_{x'} & \nu_{x'y'}E_{x'} & 0 \\ \nu_{y'x'}E_{y'} & E_{y'} & 0 \\ 0 & 0 & (1 - \nu_{x'y'}\nu_{y'x'})G_{x'y'} \end{bmatrix}$$

$$D' = \begin{bmatrix} \alpha G_{x'x'} & 0 \\ 0 & \alpha G_{y'y'} \end{bmatrix}$$

where α is the coefficient to correct the transversal tangential work.

From here it is possible to define the strengths on the shell, which are defined as:

$$\begin{bmatrix} N_{x'} \\ N_{y'} \\ N_{x'y'} \\ \dots \\ M_{x'} \\ M_{y'} \\ M_{x'y'} \\ \dots \\ Q_{x'} \\ Q_{y'} \end{bmatrix} = \int_{-\frac{t}{2}}^{\frac{t}{2}} \begin{bmatrix} \sigma_{x'} \\ \sigma_{y'} \\ \tau_{x'y'} \\ \dots \\ z'\sigma_{x'} \\ z'\sigma_{y'} \\ z'\tau_{x'y'} \\ \dots \\ \tau_{x'x'} \\ \tau_{y'x'} \end{bmatrix} dz$$

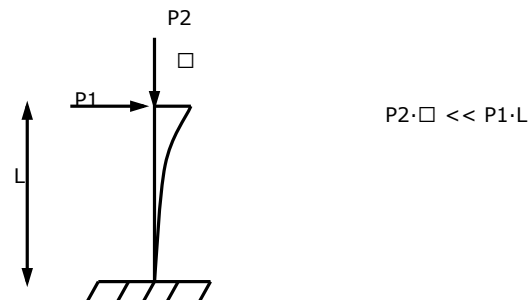
where t is the thickness of the shell.

Appendix 4: Theory of 3D solid analysis

This section gives a brief summary of the theory of the analysis of solids. Its purpose is to explain **what** RamSeries does and **not how** the program does it.

Basic assumptions

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



1 Small displacements

2 Lineal elasticity of the materials

3 Superposition of loads (Consequence of point 2)

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{bmatrix} = \mathbf{D} \begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{bmatrix}$$

The basic equation that these hypothesis define is:

$$\mathbf{D} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{(1-\nu)} & \frac{\nu}{(1-\nu)} & 0 & 0 & 0 \\ & 1 & \frac{\nu}{(1-\nu)} & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 \\ & & & \frac{1-2\nu}{2(1-\nu)} & 0 & 0 \\ & & & & \frac{1-2\nu}{2(1-\nu)} & 0 \\ & & & & & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$$

Symmetric

Appendix 5: Theory of dynamic analysis

This section gives a brief summary of the theory of dynamic analysis of structures.

Modal Analysis

In dynamic analysis, the system of equations corresponding to a structure with n degrees of freedom is:

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{C}\dot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{P}(t)$$

The corresponding free vibrations not dampened by the structural model are described by the following system of equations:

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{0}$$

which corresponds to n eigenfrequencies and n eigenvectors. These are the solutions to the following homogeneous system of equations:

$$(-\omega^2 \mathbf{M} + \mathbf{K})\mathbf{D} = \mathbf{0}$$

These eigenvectors are called modal forms and are orthogonal to the mass matrix and the stiffness matrix.

Forming a complete base

$$\mathbf{D} = [\varphi_1 \dots \varphi_2 \dots \varphi_n]$$

, can be written

$$\mathbf{D} = \sum_{i=1}^n \varphi_i y_i(t)$$

where $y_i(t)$ is a scalar function of time, called generalized coordinate.

Using the previous expressions and taking the orthogonality properties into account, we can transform one system with n degrees of freedom into a system of n equations with one degree of freedom each:

$$\mathbf{M}_j^* \ddot{y}_j(t) + \mathbf{C}_j^* \dot{y}_j(t) + \mathbf{K}_j^* y_j(t) = \varphi_j^T \mathbf{P}(t)$$

where :

$$\mathbf{M}_j^* = \varphi_j^T \mathbf{M} \varphi_j$$

$$\mathbf{C}_j^* = \varphi_j^T \mathbf{C} \varphi_j$$

$$\mathbf{K}_j^* = \varphi_j^T \mathbf{K} \varphi_j$$

Direct integration

The direct integration methods seek the history of the dynamic response in time. The response is not obtained in continuous form but rather for a predetermined series of points in time t_i .

One of the most popular direct integration methods is the Newmark method. Starting-point is the governing equation of a structure with various degrees of freedom:

$$\mathbf{M}\ddot{\mathbf{D}} + \mathbf{C}\dot{\mathbf{D}} + \mathbf{K}\mathbf{D} = \mathbf{P}(t)$$

For time $t=t_i$, this equation takes on the following form:

$$\mathbf{M}\ddot{\mathbf{D}}_{i+1} + \mathbf{C}\dot{\mathbf{D}}_{i+1} + \mathbf{K}\mathbf{D}_{i+1} = \mathbf{P}_{i+1}$$

The velocity and the acceleration are expressed as follows:

$$\begin{aligned}\dot{\mathbf{D}}_{i+1} &= \frac{1}{\beta \Delta t^2} [\mathbf{D}_{i+1} - \mathbf{D}_{i+1} - \mathbf{D}_i \Delta t] - \left(\frac{1}{2\beta} - 1 \right) \dot{\mathbf{D}}_i \\ \ddot{\mathbf{D}}_{i+1} &= \frac{\gamma}{\beta \Delta t} [\mathbf{D}_{i+1} - \mathbf{D}_i] + \left(1 - \frac{\gamma}{\beta} \right) \ddot{\mathbf{D}}_i + \left(1 - \frac{\gamma}{2\beta} \right) \Delta t \ddot{\mathbf{D}}_i\end{aligned}$$

Substituting these expressions into the equation of motion, we obtain

$$\mathbf{\tilde{K}} \mathbf{D}_{i+1} = \mathbf{\tilde{P}}_{i+1}$$

where,

$$\begin{aligned}\mathbf{\tilde{K}} &= \mathbf{K} + \frac{1}{\beta \Delta t^2} \mathbf{M} + \frac{\gamma}{\beta \Delta t} \mathbf{C} \\ \mathbf{\tilde{P}}_{i+1} &= \mathbf{P}_{i+1} + \mathbf{M} \left[\frac{1}{\beta \Delta t^2} \mathbf{D}_i + \frac{1}{\beta \Delta t} \dot{\mathbf{D}}_i + \left(\frac{1}{2\beta} - 1 \right) \ddot{\mathbf{D}}_i \right] \\ &\quad + \mathbf{C} \left[\frac{\gamma}{\beta \Delta t} \mathbf{D}_i + \left(\frac{\gamma}{\beta} - 1 \right) \dot{\mathbf{D}}_i + \left(\frac{\gamma}{2\beta} - 1 \right) \Delta t \ddot{\mathbf{D}}_i \right]\end{aligned}$$

Usually the initial conditions are that the structure possesses neither displacement nor velocity.

Spectrum analysis

Spectrum analysis is used when the dynamic action causes seismic movement at the base of the foundation. In this case, the decoupled motion equation for each mode is expressed as:

$$\mathbf{M}_j \ddot{y}_j(t) + \mathbf{C}_j \dot{y}_j(t) + \mathbf{K}_j y_j(t) = - \frac{\boldsymbol{\varphi}_j^T \mathbf{M} \mathbf{J}}{\boldsymbol{\varphi}_j^T \mathbf{M} \boldsymbol{\varphi}_j} \mathbf{a}(t)$$

where $\mathbf{a}(t)$ is the seismic acceleration.

This equation can also be resolved using the response spectra. In this case, only the maximum response of the structure is obtained. If a maximum acceleration of

$$|\mathbf{a}(t)|_{\max} = S$$

is assumed for the previous equation, it is evident that the maximum response acceleration of the system would amount to

$$\left| \ddot{y}_j(t) \right|_{\max} = \frac{\varphi_j^T \mathbf{M} \mathbf{J}}{\varphi_j^T \mathbf{M} \varphi_j} (\ddot{S}_s)_j$$

As a consequence, the maximum displacement is:

$$\left| y_j(t) \right|_{\max} = \frac{\varphi_j^T \mathbf{M} \mathbf{J}}{\varphi_j^T \mathbf{M} \varphi_j} \frac{(\ddot{S}_s)_j}{\omega_j^2}$$

Using this answer, we can calculate the maximum displacements in all of the nodes for mode j :

$$\mathbf{D}_{\max}^j = \begin{bmatrix} D_1^j \\ D_2^j \\ \vdots \\ D_n^j \end{bmatrix} = \varphi_j \left| y_j(t) \right|_{\max} = \varphi_j \frac{\varphi_j^T \mathbf{M} \mathbf{J}}{\varphi_j^T \mathbf{M} \varphi_j} \frac{(\ddot{S}_s)_j}{\omega_j^2} = \mathbf{A}_j \frac{(\ddot{S}_s)_j}{\omega_j^2}$$

where \mathbf{A}_j is the vector of the modal participation coefficients corresponding to mode j of the vibration. Supposing that for each degree of freedom the maximum response does not occur at the same instant in each mode, the maximum response of the structure will not be equal to the sum of the maxim corresponding to each mode:

$$\mathbf{D}_{\max} \neq \sum_{j=1}^q \mathbf{D}_{\max}^j$$

Different formulas for finding the value of \mathbf{D}_{\max} through \mathbf{D}_{\max}^j have been proposed. The most simple and at the same time most used is that which establishes that the response is equal to the square root of the sum of the squares of the modal responses. .

$$\mathbf{D}_{\max} = \sqrt{\sum_{j=1}^q (\mathbf{D}_{\max}^j)^2}$$

As for the stresses, reactions, and in general any response \mathbf{R} which is to be determined, we get analogously:

$$\mathbf{R}_{\max} = \sqrt{\sum_{j=1}^q (\mathbf{R}_{\max}^j)^2}$$

Appendix 6: Theory of nonlinear analysis

This section gives a brief summary of the theory of nonlinear analysis of solid and structures.

Several aspects related to the resolution of a nonlinear equations system are commented. Typical nonlinear constitutive models for metals like J2 plasticity are briefly explained. Some aspects of contact-impact problem are commented.

Nonlinear equations systems

The strategies of resolution of a nonlinear equations system are related with the decomposition of it in several linear systems each of them are solved with the known methods.

A very known strategy of resolution of nonlinear equations systems is the Newton-Raphson methods. In this method an incremental-iterative scheme is utilised. The total load is decomposed in several increments. For each increment an iterative procedure is performed until the convergence criteria is obtained.

There are several methods to control the following of the load-displacement curve of the structure. Each one of these methods control different parameters like load, displacement or arc-length increments.

Some advances procedures like line search, automatic incrementation or automatic arc-length switch in order to improve the speed of calculations can be applied.

Deeper explanations can be found in Bathe (1996), Crisfield (1991).

J2 plasticity

The J2 plasticity are based in the following considerations:

i-) Additive decomposition of the strain tensor. One assumes that the strain tensor $\boldsymbol{\epsilon}$ is decomposed into an elastic and plastic part, denoted by $\boldsymbol{\epsilon}^e$ and $\boldsymbol{\epsilon}^p$ respectively, according to the relationship.

$$\boldsymbol{\epsilon} = \boldsymbol{\epsilon}^e + \boldsymbol{\epsilon}^p$$

ii-) Elastic stress response. The stress tensor $\boldsymbol{\sigma}$ is related to the elastic strain by means of a stored-energy function W according to the relationship $\boldsymbol{\sigma} = \partial W / \partial \boldsymbol{\epsilon}^e$

For linearized elasticity, W is a quadratic form in the elastic strain, i.e. $W = 1/2 \boldsymbol{\epsilon}^e : \mathbf{C} : \boldsymbol{\epsilon}^e$, where \mathbf{C} is the tensor of elastic moduli which is assumed constant. Then the stress tensor is written $\boldsymbol{\sigma} = \mathbf{C} : (\boldsymbol{\epsilon} - \boldsymbol{\epsilon}^p)$.

iii-) Yield condition. We define a function $f(\boldsymbol{\sigma}, \mathbf{q})$ called yield criterion where \mathbf{q} is a vector of internal variables. The admissible states $\{ \boldsymbol{\sigma}, \mathbf{q} \}$ are constraint by $f(\boldsymbol{\sigma}, \mathbf{q}) \leq 0$.

A choice of internal variables which is typically of metal plasticity is $\mathbf{q} = \{ \xi, \boldsymbol{\beta} \}$. Here, ξ is the equivalent plastic strain that defines isotropic hardening of the Von Mises yield surface, and $\boldsymbol{\beta}$ defines the center of the Von Mises yield surface in stress deviator space. The resulting J2-plasticity model has the following yield condition:

$$\boldsymbol{\eta} = \text{dev} [\boldsymbol{\sigma}] - \boldsymbol{\beta}, \text{tr}[\boldsymbol{\beta}] = 0$$

$$f(\boldsymbol{\sigma}, \mathbf{q}) = \sqrt{\boldsymbol{\eta} \cdot \boldsymbol{\eta}} - \sqrt{2/3} K(\xi)$$

iv-) Flow rule and hardening law. The flow rule and hardening law for a J2-plasticity model are:

$$d\boldsymbol{\varepsilon}^p = \gamma \frac{\boldsymbol{\eta}}{\sqrt{\boldsymbol{\eta} \cdot \boldsymbol{\eta}}}$$

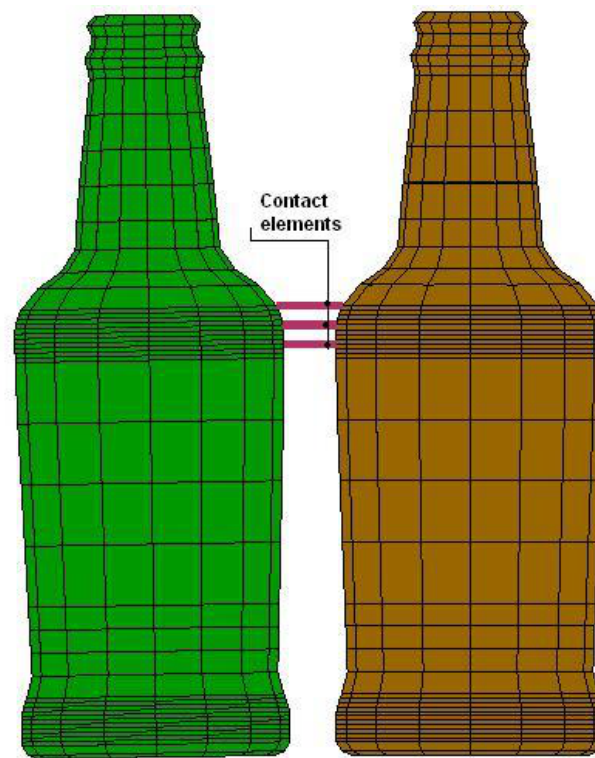
$$d\boldsymbol{\beta} = \gamma \frac{2}{3} H(\xi) \frac{\boldsymbol{\eta}}{\sqrt{\boldsymbol{\eta} \cdot \boldsymbol{\eta}}}$$

$$d\xi = \gamma \sqrt{2/3}$$

Further explanations can be found in Simo and Hughes (1997)

Contact-impact

The contact-impact problem is a typical nonlinear analysis of boundary conditions. There are two general approaches to solve the problem: the penalty method and the augmented Lagrangian method. In RamSeries program a simple variant the penalty method is utilised. The idea is to simulate the contact by the use of contact element of very high stiff (penalty) that began to be active when any small penetration is achieved.

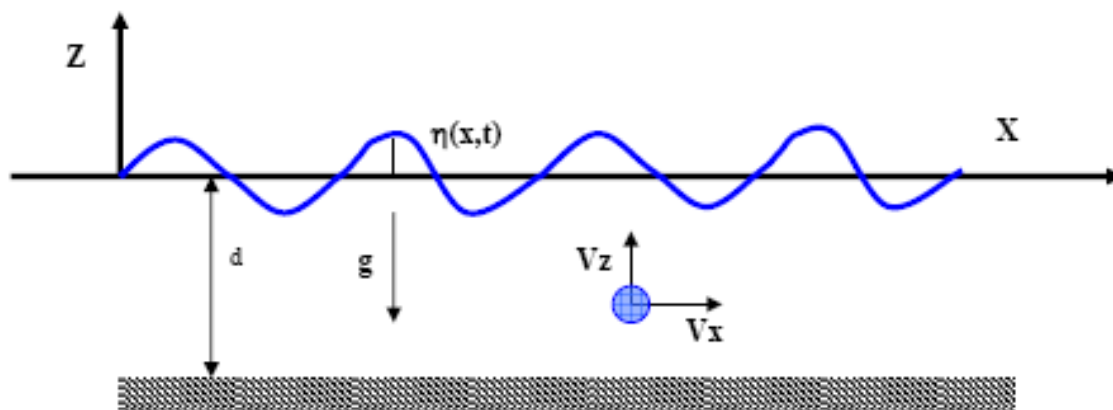


Appendix 7: Wave theory

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.

Potential flow for waves

A linearized model for the waves can be used for calculating the 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:

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

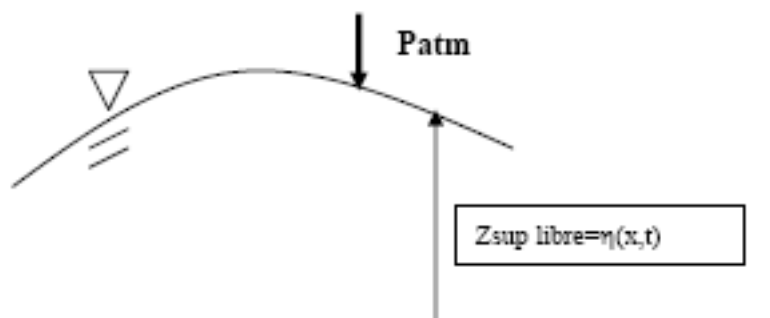
Balance of momentum:

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

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:

$$\frac{\partial V_x}{\partial t} + \frac{1}{2} \frac{\partial V^2}{\partial x} = -\frac{1}{\rho} \frac{\partial p}{\partial x}$$

$$\frac{\partial V_z}{\partial t} + \frac{1}{2} \frac{\partial V^2}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} - g$$

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 ($z=-d$):

$$\frac{\partial \phi}{\partial z} \Big|_{(z=-d)} = 0$$

- Free surface ($z=0$):

$$\frac{\partial \eta}{\partial t} = \frac{\partial \phi}{\partial z}$$

$$\frac{\partial \phi}{\partial t} + g\eta = 0$$

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

$$\phi = \frac{Hg}{2\omega} \frac{\cosh\left(kd \cdot \left(1 + \frac{z}{d}\right)\right)}{\cosh(k \cdot d)} \cdot \sin(k \cdot x - \omega \cdot t)$$

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).

Appendix 8: Composite materials Theory

Composite sheet concept

In Composite material built structures, the basic unit of the whole laminate is the sheet. The three most frequent composite sheets are:

- **Unidirectional sheet:** All fibers are oriented in the same direction. *Transversal isotropy* is considered for this kind of sheet (same properties in all directions contained in a plane orthogonal to the fiber's plane).
- **Roving sheet:** All fibers oriented in two orthogonal directions. The longitudinal fibers are crisscrossed with the transversal ones. 50% of the fibers can be oriented in each direction (equilibrated roving laminate), or there can be more fibers in one direction than in the other (non-equilibrated roving laminate). *Orthotropy* is considered for this kind of sheet.
- **Mat sheet:** It is made of small cut fibers, oriented at random. Isotropy is considered (same properties in all directions) in the sheet plane.

Elastic properties of the composite sheet

$$E_1 = V_f \cdot E_f + (1 - V_f) \cdot E_m$$

$$E_2 = \frac{(1 - V_m^2) \cdot (1 + 0.85 \cdot V_f^2)}{(1 - V_f)^{1.25} + (E_m/E_f) \cdot V_f / (1 - v_m^2)}$$

$$G_{12} = (G_m \cdot \frac{1 + 0.6 \cdot V_f^{0.5}}{(1 - V_f)^{1.25} + V_f / (1 - v_m^2)})$$

$$v_{12} = V_f \cdot v_f + (1 - V_f) \cdot v_m$$

$$E_1=E_2=V_f \cdot \left(E_f \cdot \frac{1}{2} + E_m \cdot \frac{3}{2} \right) + E_m$$

$$G_{12} = \frac{E_m}{\frac{4 \cdot V_f + 1}{3}}$$

$$\nu_{12} = \frac{G_{12}}{E_1}$$

$$E_{ur} = \frac{C_e \cdot E_L + (1 - C_e) \cdot E_T}{\lambda}$$

$$E_{tr} = \frac{C_e \cdot E_T + (1 - C_e) \cdot E_L}{\lambda}$$

$$\nu_{ur} = \frac{\nu_{LT}}{C_e + (1 - C_e) \cdot E_L / E_T}$$

$$G_{ur} = G_{LT}$$

Con:

$$\lambda = 1 - \nu_{LT} \cdot \nu_{TL}$$

$$C_e = \frac{(N_f \cdot T)_{urdimbre}}{(N_f \cdot T)_{urdimbre} + (N_f \cdot T)_{trama}}$$

N

$f \rightarrow$ Number of threads per centimetre

T

\rightarrow Thread title (in Tex)

$$E_1=E_2=V_f \cdot \left(E_f \cdot \frac{16}{45} + E_m \cdot 2 \right) + E_m \cdot \frac{8}{9}$$

$$G_{12} = V_f \cdot \left(E_f \cdot \frac{2}{15} + E_m \cdot \frac{3}{4} \right) + E_m \cdot \frac{1}{3}$$

$$\nu_{12} = \frac{1}{3}$$

Laminate shell concept

The basic configuration of a composite material structure is the sheet. As, in general, the sheet thickness is very low (0.1 - 1.5 mm), the used resistant configurations consist of sequential sheet association, called laminates.

A composite material laminate can work in plane stress or in flexing. For both, the *laminate shell theory* will be applied.

Laminate shell Theory

The classic laminate shell theory uses the plane deformation components of the laminate: $\varepsilon_1, \varepsilon_2$ y ε_6 and the curvatures κ_1, κ_2 y κ_6 . It does not consider inter-laminar effects in the laminate: not the normal inter-laminar deformation ε_3 , neither the inter-laminar shear deformation components ε_4 and ε_5 .

Thus, the application of this theory reduces to:

- Structures with small deformations.
- Lineal deformation along the laminate thickness.
- Thin plates. Considering as thin plate, the one that verifies: $L/h > 10$. Where "h" is the total thickness of the shell, and "L" its length.

Formulas and a detailed description of this theory can be found at *Reference 7*.

TSAI-WU Failure Criteria

The Tsai-Wu failure criterion is very similar to the Hoffman criteria, but has different value for the coefficient associated with the interaction of the direct stresses. In the Hoffman criterion it is the same as the coefficient associated with the direct longitudinal stresses. In the Tsai-Wu criterion, F_{12} is a term that should be determined from a test which induces a combined state of stress. Due to the difficulty of configuring such test, some references suggest setting the value of the term to zero. Tsai recommends an specific relationship with other coefficients:

$$F_{12} = -0.5 \cdot (F_{11} \cdot F_{22})^{0.5}$$

For the Tsai-Wu criterion:

$$FI = (F_{11} \cdot \sigma_1^2 + F_{22} \cdot \sigma_2^2 + 2 \cdot F_{12} \cdot (\sigma_1 \cdot \sigma_2) + F_{66} \cdot \sigma_{12}^2) + F_1 \cdot \sigma_1 + F_2 \cdot \sigma_2$$

Appendix 9: Buckling Analysis in FEA

This appendix provides information about the method implemented in RamSeries for obtaining the buckling stress and edge ratios for elementary plate panels (**EPP**) from a finite element calculation. This method is called **"Displacement Method"**.

This method is taken from **Germanischer Lloyd Rules, Chapter 7, Appendix 2 - "Displacement based buckling assessment in Finite Element Method", (Common Structural Rules for Bulk Carriers)**

For no interpolation will be performed in RamSeries, the nodes of the buckling panel should be chosen from the FE mesh nodes. Therefore, the displacements will be transferred directly.

- **Transformation in local system:**

The transformation of the nodal displacements from the global FE-system into the local system of the buckling panel is performed by:

$$(u) = [\lambda] \cdot (u_g)$$

Where:

(u) : Local displacement vector

(u_g) : Global displacement vector

$[\lambda]$: Transformation matrix (2x3), of direction cosines of angles formed between the two sets of axes.

- **Calculation of buckling stresses and edge stress ratios:**

The displacements, derived at the corners of the elementary plate panel, are to be considered as input from which the stresses at certain stress-points are derived. In the 4-node buckling panel these points are identical. The locations and numbering convention is taken from Figure 1:

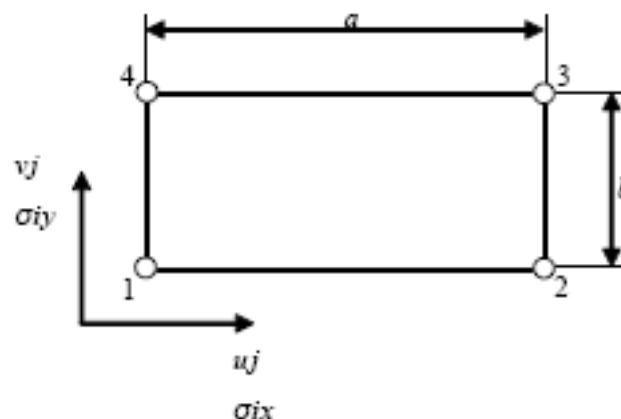


Figure 1: 4-node buckling panel

The derived stresses at EPP can be directly used as input for the buckling assessment according to Germanischer Lloyd Rules Ch 6, Sec 3.

- **4-node buckling panel:**

Stress displacement relationship for a 4-node buckling panel (compressive stresses are positive). From the displacements of the EPP corner nodes can be obtained using:

$$\begin{pmatrix} \sigma_{1x}^* \\ \sigma_{1y}^* \\ \tau_1 \\ \sigma_{2x}^* \\ \sigma_{2y}^* \\ \tau_2 \\ \sigma_{3x}^* \\ \sigma_{3y}^* \\ \tau_3 \\ \sigma_{4x}^* \\ \sigma_{4y}^* \\ \tau_4 \end{pmatrix} = -C \cdot \begin{pmatrix} -2/a & -2\nu/b & 2/a & 0 & 0 & 0 & 0 & 2\nu/b \\ -2\nu/a & -2/b & 2\nu/a & 0 & 0 & 0 & 0 & 2/b \\ -m/b & -m/a & 0 & m/a & 0 & 0 & m/b & 0 \\ -2/a & 0 & 2/a & -2\nu/b & 0 & 2\nu/b & 0 & 0 \\ -2\nu/a & 0 & 2\nu/a & -2/b & 0 & 2/b & 0 & 0 \\ 0 & -m/a & -m/b & m/a & m/b & 0 & 0 & 0 \\ 0 & 0 & 0 & -2\nu/b & 2/a & 2\nu/b & -2/a & 0 \\ 0 & 0 & 0 & -2/b & 2\nu/a & 2/b & -2\nu/a & 0 \\ 0 & 0 & -m/b & 0 & m/b & m/a & 0 & -m/a \\ 0 & -2\nu/b & 0 & 0 & 2/a & 0 & -2/a & 2\nu/b \\ 0 & -2/b & 0 & 0 & 2\nu/a & 0 & -2\nu/a & 2/b \\ -m/b & 0 & 0 & 0 & 0 & m/a & m/b & -m/a \end{pmatrix} \cdot \begin{pmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{pmatrix}$$

Where:

- a : Length of the longer plate panel side
- b : Length of the shorter plate panel side
- x : Direction parallel to a , taken as the longitudinal direction
- y : Direction parallel to b , taken as the transverse direction
- C : Coefficient taken equal to:

$$C = \frac{E}{2(1-\nu^2)}$$
- ν : Poisson ratio
- m : Coefficient taken equal to:

$$m = 1 - \nu$$

$(\sigma_{1x}^*, \sigma_{1y}^*, \tau_1, \dots, \sigma_{4x}^*, \sigma_{4y}^*, \tau_4)^T = (\sigma^*)$: Element stress vector

$(u_1, v_1, \dots, u_4, v_4)^T = (u)$: Local node displacement vector

If both σ_x^* and σ_y^* are compressive stresses, then σ_x and σ_y must be obtained as follows:

$$\begin{aligned} \sigma_x &= (\sigma_x^* - 0.3\sigma_y^*) / 0.91 \\ \sigma_y &= (\sigma_y^* - 0.3\sigma_x^*) / 0.91 \end{aligned}$$

Where compressive stress fulfils the condition $\sigma_y^* < 0.3\sigma_x^*$, then $\sigma_y = 0$ and $\sigma_x = \sigma_x^*$

Where compressive stress fulfils the condition $\sigma_x^* < 0.3 \cdot \sigma_y^*$, then $\sigma_x = 0$ and $\sigma_y = \sigma_y^*$

This leads to the following stress vector:

$$(\sigma) = (\sigma_{1x}, \sigma_{1y}, \tau_1, \dots, \sigma_{4x}, \sigma_{4y}, \tau_4)^T$$

Finally, the relevant buckling stresses ratios are obtained by:

- LC 1: Longitudinal compression

$$\sigma_l = \max\left(\frac{\sigma_{1x} + \sigma_{4x}}{2}, \frac{\sigma_{2x} + \sigma_{3x}}{2}\right)$$

$$\Delta\sigma_l = \frac{1}{2}(-\sigma_{1x} + \sigma_{4x} - \sigma_{2x} + \sigma_{3x})$$

$$\sigma_x = \sigma_l + 0.5|\Delta\sigma_l|$$

$$\psi_x = 1 - |\Delta\sigma_l|/\sigma_x$$

- LC 2: Transverse compression

$$\sigma_t = 0.25 \sum_{i=1}^4 \sigma_{iy}$$

$$\Delta\sigma_t = \frac{1}{2}(-\sigma_{1y} - \sigma_{4y} + \sigma_{2y} + \sigma_{3y})$$

$$\sigma_y = \sigma_t + 0.5|\Delta\sigma_t|$$

$$\psi_y = 1 - |\Delta\sigma_t|/\sigma_y$$

- LC 3: Shear

$$\tau = 0.25 \sum_{i=1}^4 |\tau_i|$$

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.

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
	Yes				2.0
Splash zone (5)	No	Coating (2) (3)	Yes (6)	Combination of curves marked "air" and "free corrosion" (9)	3.0
	Yes				2.0
Submerged zone	No	Cathodic protection and optional coating (2) (4)	No		3.0
Scour zone	No		Yes (7)		3.0
Below seabed	No	None	No	In seawater	3.0
	Yes				2.0
Closed compartments with seawater	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 C5M 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 - intact, 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:

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

Where;

- **I** = Number of stress ranges blocks ($\sigma_i = SCF \cdot \Delta \sigma_{nominal}$). This blocks have been associated with the combined load cases, specifically identified for being used in fatigue assesment. The maximum stress is calculated in each mesh node, for each block, and it is used as $\Delta \sigma_i$.
- **$n_{C,i}$** = Number of load cycles in block "i". This number is inserted by the user, and there is one for each combined load case.
- **$N_{C,i}$** = Number of load cycles to failure, for each stress range $\Delta \sigma_i$, given for the corresponding S-N curve.

The stress range blocks are combined loadcases designed by the user for being specifically used in the fatigue assesment.

S-N curves

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

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

Where:

- **N** = Number of cycles to failure, for a given stresses range ($\Delta\sigma = \mathbf{SCF} \cdot \Delta\sigma_{nominal}$).
- **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 followin table:

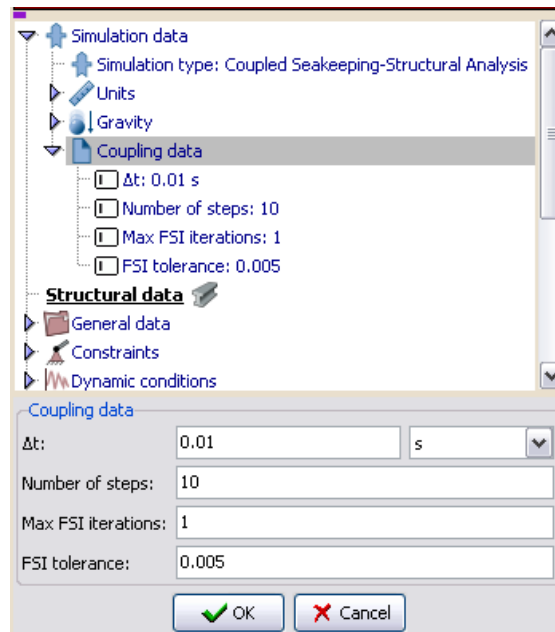
Table J1 S-N curves for most frequently used structural details											
Structural detail	Environment										
	In air				In seawater with corrosion protection				Free corrosion		
	loga	m	Range of validity	k	loga	m	Range of validity	k	loga	m	k
Weld in tubular joint	12.164	3	$N < 10^7$	0.25	11.764	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	$N < 10^7$ $L < 50 \text{ mm}$	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	$N < 10^7$ $50 \text{ mm} < L < 120 \text{ mm}$	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	$N < 10^7$ $120 \text{ mm} < L < 300 \text{ mm}$	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	$N < 10^7$ $L > 300 \text{ mm}$	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.

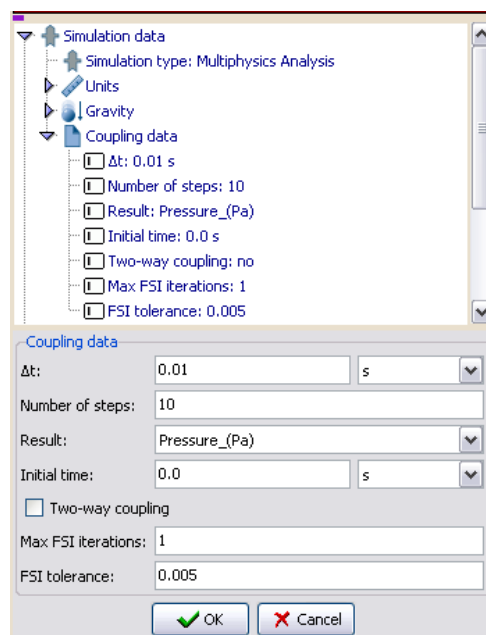
Appendix 11: Coupled Analyses

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)

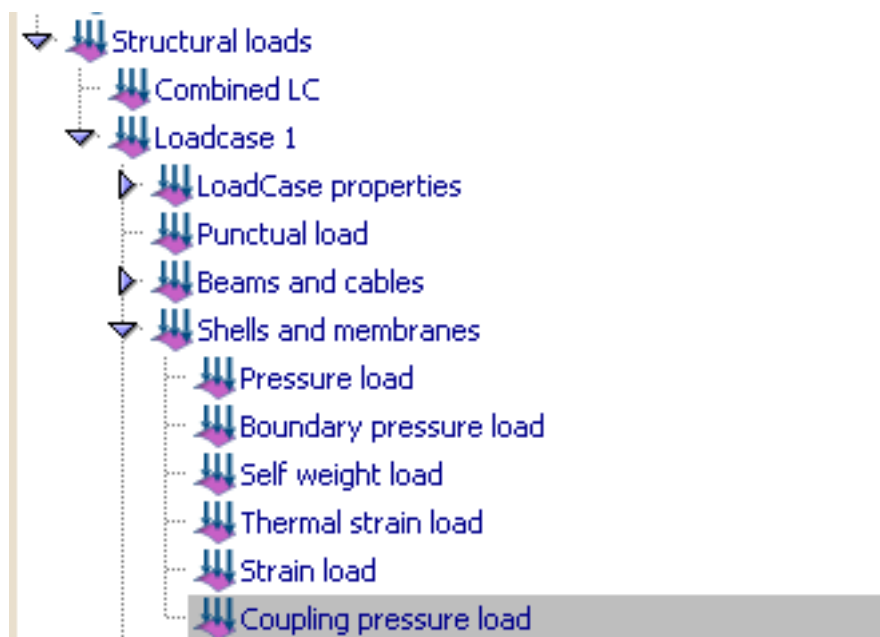
- **Initial time** (only for Multiphysics coupled analysis): Time from which the coupling will start. For example, Tdyn may run alone until a certain time (initial time), and then start to couple with RamSeries.
- **Two-way coupling** (only for Multiphysics coupled analysis): Indicates if mesh deformation 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).

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



Coupled Analyses with SeaFEM

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

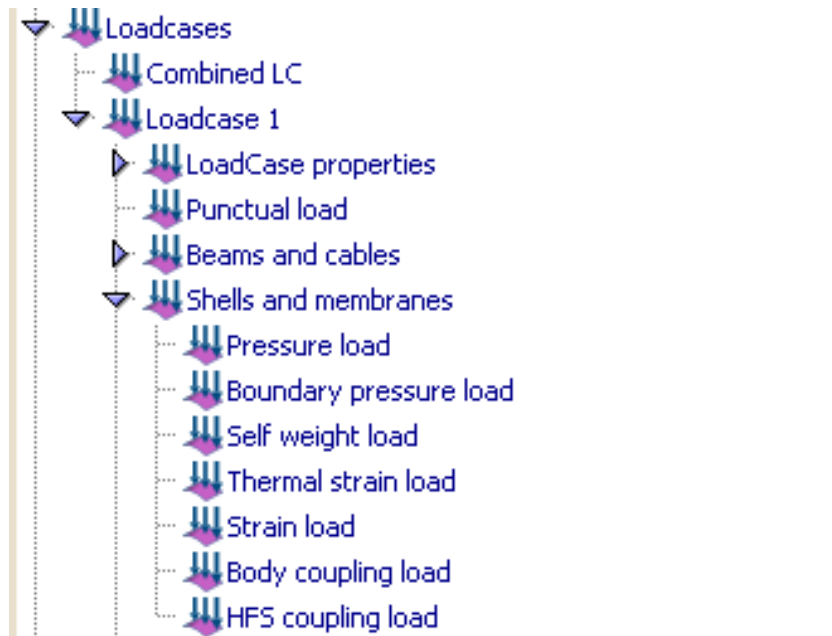
This can be done assigning the following loads:

- 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**



Appendix 12: Stiffened Shell model

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

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:

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

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

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:

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

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:

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

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:

$$\mathbf{D}_f = \beta \cdot \begin{bmatrix} \mathbf{E} & \nu \mathbf{E} & \mathbf{0} \\ \nu \mathbf{E} & \mathbf{E} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \varphi \mathbf{G} \end{bmatrix}$$

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

$$\beta = 1/\varphi$$

$$\mathbf{D}_c = \begin{bmatrix} \alpha \mathbf{G} & \mathbf{0} \\ \mathbf{0} & \alpha \mathbf{G} \end{bmatrix}$$

The stiffness matrix of the system (\mathbf{D}_{shell}^*) is obtained as follows, building it with the matrices \mathbf{D}_m^* , \mathbf{D}_{mf}^* , \mathbf{D}_f^* and \mathbf{D}_c^* :

$$\mathbf{D}^*_{shell} = \begin{bmatrix} \mathbf{D}_m^* & \mathbf{D}_{mf}^* & \mathbf{0} \\ \mathbf{D}_{mf}^* & \mathbf{D}_f^* & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{D}_c^* \end{bmatrix}$$

- $\mathbf{D}_m^* = \int_{-h/2}^{+h/2} \mathbf{D}_f \cdot dz$;
- $\mathbf{D}_f^* = \int_{-h/2}^{+h/2} z^2 \cdot \mathbf{D}_f \cdot dz$;
- $\mathbf{D}_{mf}^* = \int_{-h/2}^{+h/2} z \cdot \mathbf{D}_f \cdot dz$;
- $\mathbf{D}_c^* = \int_{-h/2}^{+h/2} \mathbf{D}_c \cdot dz$

After integration:

$$\mathbf{D}_m^* = \begin{bmatrix} \mathbf{D}_{xm} & \mathbf{D}_{xym} & 0 \\ \mathbf{D}_{xym} & \mathbf{D}_{ym} & 0 \\ 0 & 0 & \mathbf{D}_{zm} \end{bmatrix}$$

With:

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

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

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

$$\mathbf{D}_{zm} = h \cdot G$$

$$\mathbf{D}_f^* = \begin{bmatrix} \mathbf{D}_{xf} & \mathbf{D}_{xyf} & 0 \\ \mathbf{D}_{xyf} & \mathbf{D}_{yf} & 0 \\ 0 & 0 & \mathbf{D}_{zf} \end{bmatrix}$$

With:

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

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

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

$$\mathbf{D}_{zf} = h^3 \cdot G / 12$$

$$\mathbf{D}_{mf}^* = [0]$$

(assumed for simplicity)

$$\mathbf{D}_c^* = h \cdot \alpha \cdot \begin{bmatrix} \mathbf{G} & 0 \\ 0 & \mathbf{G} \end{bmatrix}$$

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:

$$\mathbf{D}_{beam}^* = \begin{bmatrix} EA & 0 & 0 & 0 & 0 & 0 \\ 0 & EI_{y'} & 0 & 0 & 0 & 0 \\ 0 & 0 & EI_{z'} & 0 & 0 & 0 \\ 0 & 0 & 0 & GJ & 0 & 0 \\ 0 & 0 & 0 & 0 & \alpha_{y'} GA & 0 \\ 0 & 0 & 0 & 0 & 0 & \alpha_{z'} GA \end{bmatrix}$$

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

$$\mathbf{D}^*_{\text{Beam - Shell}} = \begin{bmatrix} \mathbf{D}^*_{m-BS} & \mathbf{D}^*_{mf-BS} & \mathbf{0} \\ \mathbf{D}^*_{mf-BS} & \mathbf{D}^*_{f-BS} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{D}^*_{c-BS} \end{bmatrix}$$

Being:

$$\mathbf{D}^*_{m-BS} = \begin{bmatrix} \mathbf{D}_{xm-BS} & \mathbf{D}_{xym-BS} & \mathbf{0} \\ \mathbf{D}_{xym-BS} & \mathbf{D}_{ym-BS} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{D}_{zm-BS} \end{bmatrix}$$

With:

$$\mathbf{D}_{xm-BS} = h \cdot E / (1 - \nu^2) + E \cdot A_{st} / c$$

$$\mathbf{D}_{ym-BS} = h \cdot E / (1 - \nu^2)$$

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

$$\mathbf{D}_{zm-BS} = h \cdot G$$

$$\mathbf{D}^*_{f-BS} = \begin{bmatrix} \mathbf{D}_{xf-BS} & \mathbf{D}_{xyf-BS} & \mathbf{0} \\ \mathbf{D}_{xyf-BS} & \mathbf{D}_{yf-BS} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{D}_{zf-BS} \end{bmatrix}$$

With:

$$\mathbf{D}_{xf-BS} = I_{ENshell} \cdot E / (c \cdot (1 - \nu^2)) + E \cdot I_{y'} / c$$

$$\mathbf{D}_{yf-BS} = h^3 \cdot E / (12 \cdot (1 - \nu^2))$$

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

$$\mathbf{D}_{zf-BS} = h^3 \cdot G / 12$$

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

$$\mathbf{D}^*_{mf-BS} = [\mathbf{0}]$$

$$\mathbf{D}^*_{c-BS} = h \cdot \alpha \cdot \begin{bmatrix} \mathbf{G} & \mathbf{0} \\ \mathbf{0} & \mathbf{G} \end{bmatrix} + A_{st} \cdot \frac{\alpha}{c} \cdot \begin{bmatrix} \mathbf{G} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} \end{bmatrix}$$

Stiffened shell test 1

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

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

- $a = 2 \text{ m}$
- $b = 4 \text{ m}$

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

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

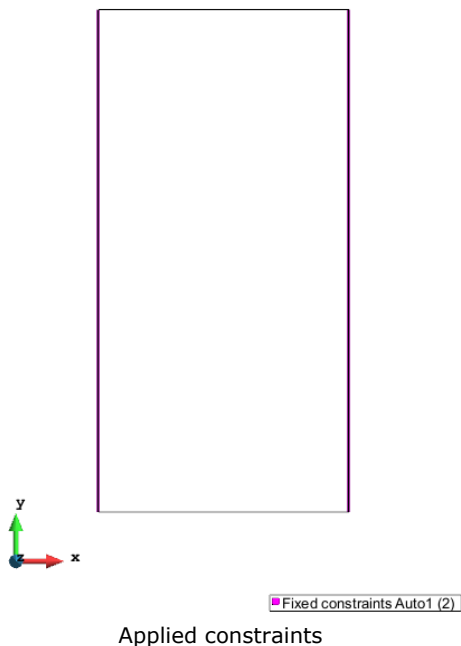
The **equivalent stiffened shell** model used ([Inertia of the stiffened section -pag. 88-](#) and [Equivalent model formulation -pag. 90-](#)) result in the following shell properties:

- **$h_{eq} = 0.1129 \text{ m}$**
- **$\rho_{eq} = 1559.532 \text{ N/m}^2$**

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

- $E = 2.1 \times 10^{11} \text{ N/m}^2$
- $\rho = 76900 \text{ N/m}^3$
- $\nu = 0.3$

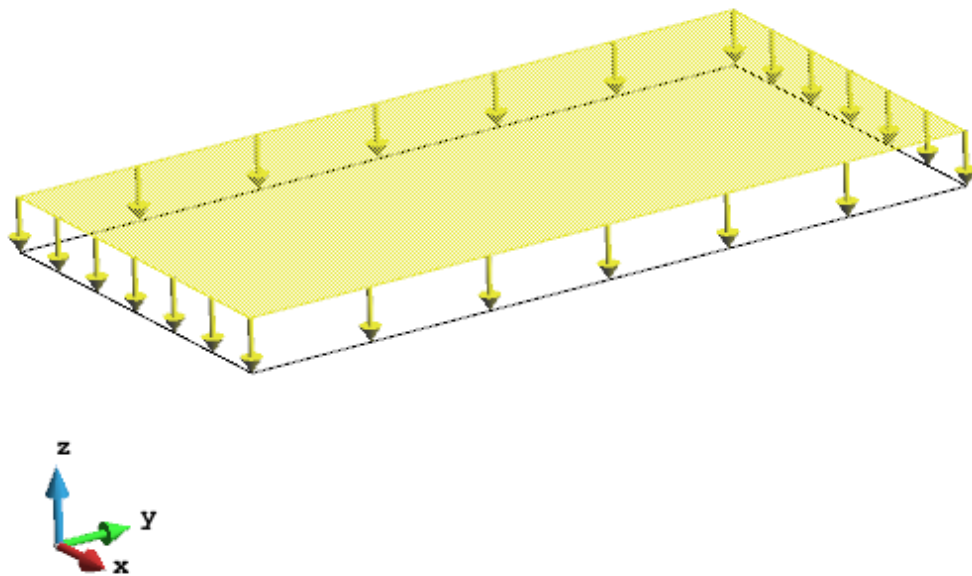
The rectangular shell is **simply supported** in both "**b**" sides:



The model has been loaded with the following values:

- 1 . $q_1 = q_{0z}$
- 2 . $q_2 = 2 \cdot q_{0z}$
- 3 . $q_3 = 3 \cdot q_{0z}$
- 4 . $q_4 = 4 \cdot q_{0z}$

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)):

$$\delta_{z\text{Max}} = \alpha \cdot \frac{q \cdot a^4}{D}$$

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

$a = 2 \text{ m}$

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

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

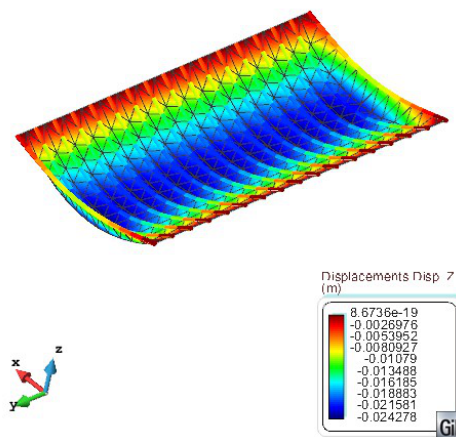
Results comparing with theory

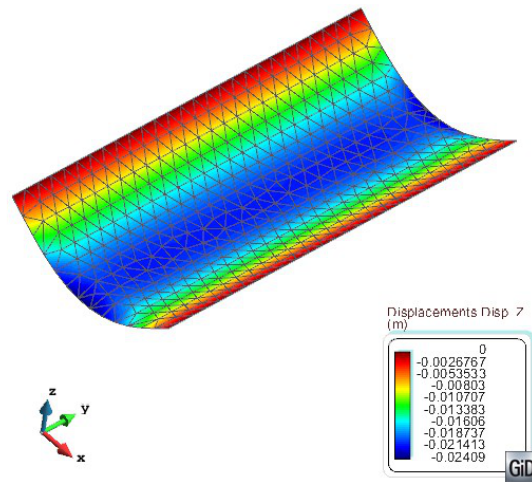
Another test, involving the same model for stiffened shell of the previous example, has been carried out. In this case, the stiffened shell deflection is compared to the deflection of the equivalent reinforced shell (with the stiffeners modelled as beams).

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 q₄). A deformation factor of 20 has been applied to both visualizations.





Stiffened shell test 2

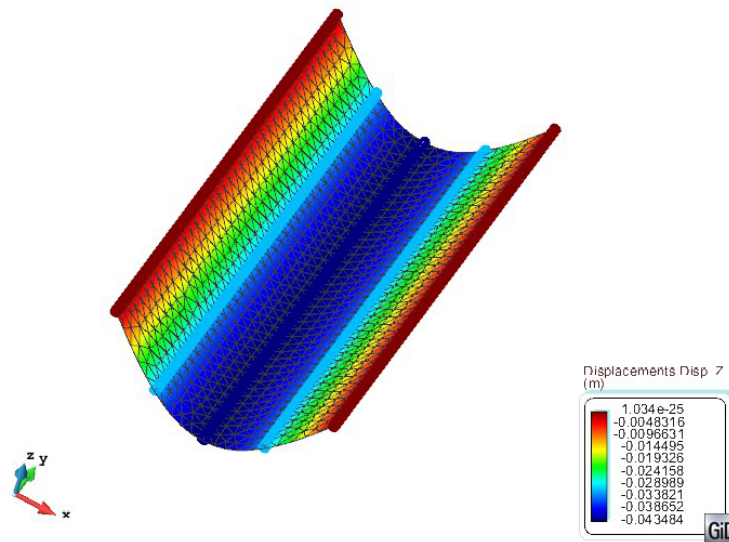
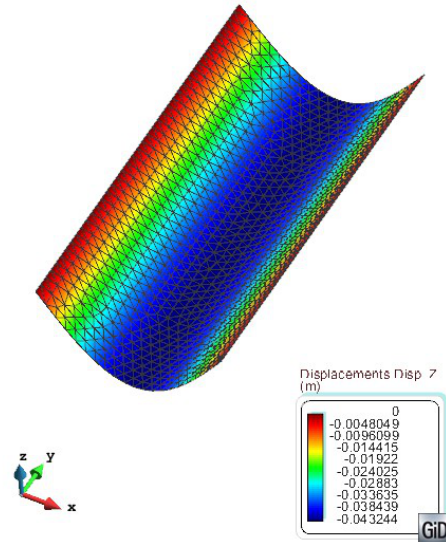
This test case is similar to the previous one. The only difference is the direction of the stiffeners; in this case they are oriented in the Y axis (ortogonal to the direction of the stiffeners of previous test model).

The loads in this case have also been reduced: $q_0 = -1.0e3 \text{ N/m}^2$

The next table shows the results:

q (N/m ²)	d_z (m) StiffS_Ram	d_z (m) Sh+Stiff	E (%)
-1.00E+03	-0.010811	-0.010871	0.55
-2.00E+03	-0.021622	-0.021742	0.55
-3.00E+03	-0.032433	-0.032613	0.55
-4.00E+03	-0.043244	-0.043484	0.55

Results comparing with RamSeries beams&shells model



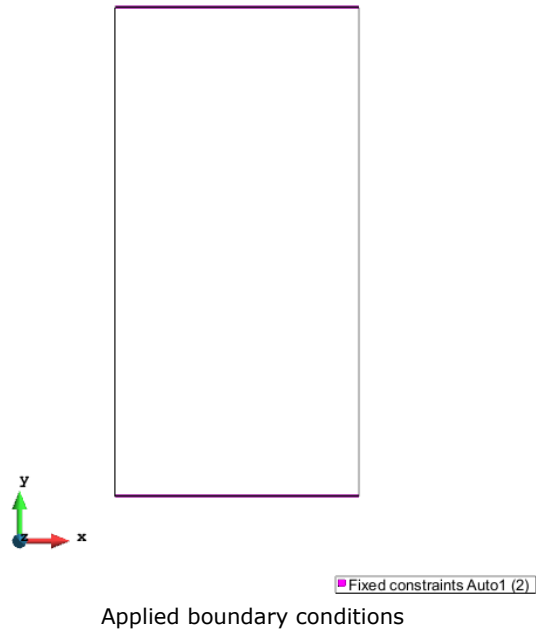
Stiffened shell test 3

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.



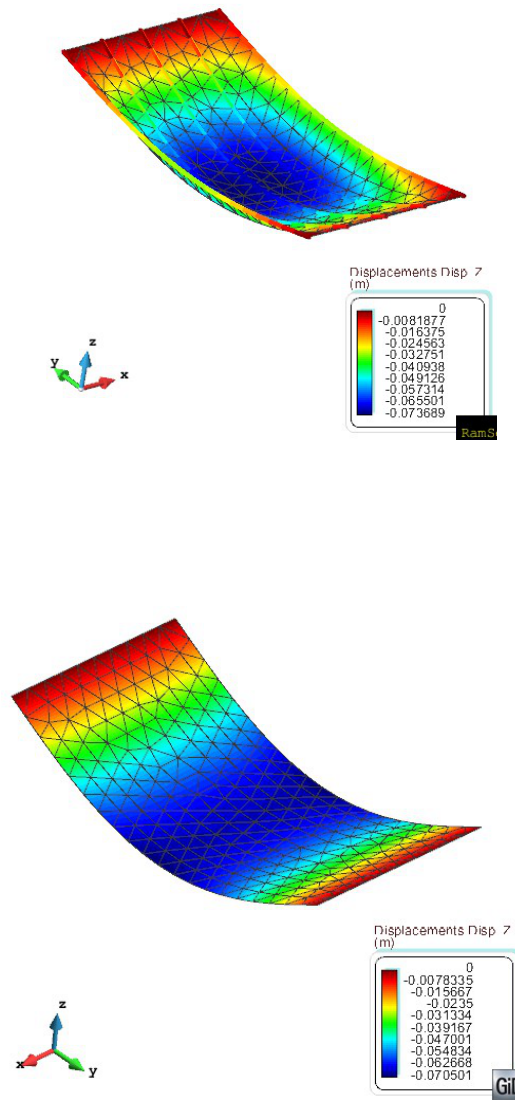
The load in this case is: $q_{0z} = -1.0e5 \text{ N/m}^2$

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_4). A deformation factor of 10 has been applied to both visualizations:



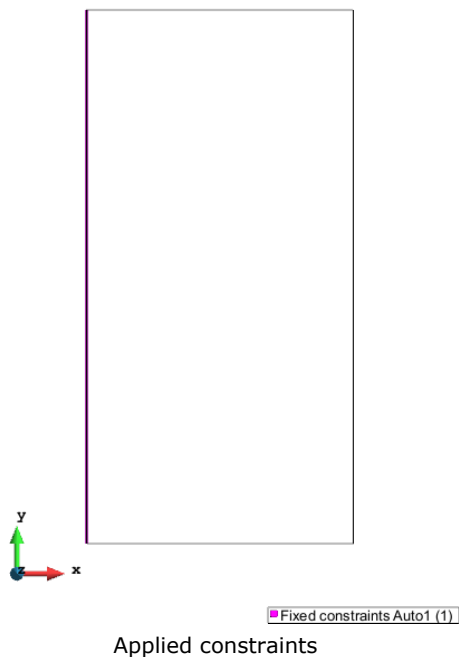
Stiffened shell test 4

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

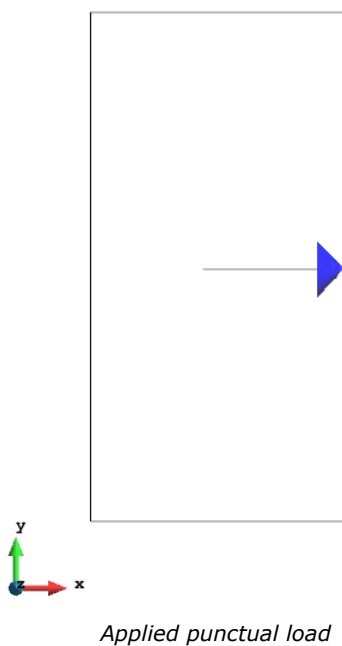
In this case, the stiffeners are oriented in the X axis (as in the first test model).

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): $q_0 = 1.0e8 \text{ N}$

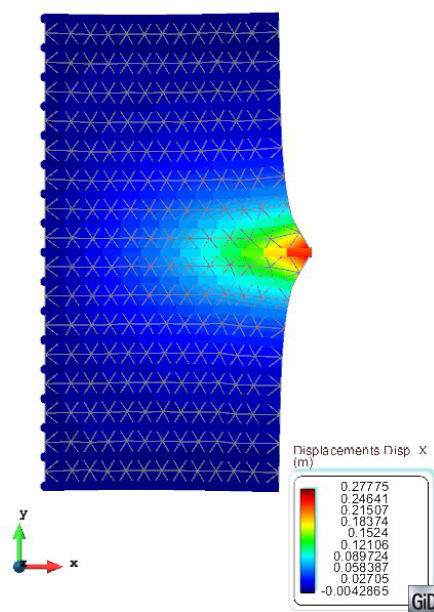
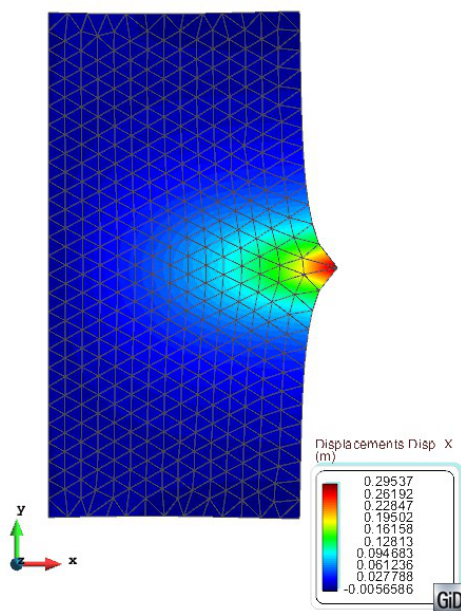


The next table shows the results:

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:



4 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*, 5ª 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*.