



CONCHA – Getting Started

TAILSIT, Graz 2017

Contents

1	Introduction	2
2	Minimal example	3
3	Free vs. full version	7
4	Technical background	8
5	User interface	9
6	License	14
7	Contact	14
	List of commands	14

1 Introduction

The Rhino3D plug-in CONCHA provides the numerical simulation of shell structures. Based on the Finite Element Method, CONCHA lets you analyse the mechanical behaviour of complex geometries subject to arbitrary loads without leaving the design environment of Rhino3D. This document covers the installation, a guide through an example simulation with CONCHA and a description of available commands.

1.1 System requirements and installation

The prerequisites for CONCHA are

- Rhino 5 for Windows (64 bit)
- Rhino Service Release 13 or higher
- Microsoft .NET Framework 4.5.1
- Visual C++ Redistributable for Visual Studio 2015 (64 bit)

If above listed requirements are met, you can proceed with the installation of CONCHA by starting *concha.msi*. Simply follow the instructions of the installer. You can check the installation in Rhino3D by typing **PluginManager** in the command line and looking for CONCHA in the list. The release notes and license can be found in the installation folder.

2 Minimal example

In this section, a simple cantilever plate is considered in order to demonstrate the basic workflow with CONCHA. It shall familiarise you with the following commands:

 [ConchaAddSurface](#)

 [ConchaCreateMeshes](#)

 [ConchaSetMaterial](#)

 [ConchaMeshSettingsSurface](#)

 [ConchaSetConstraintsOnEdges](#)

 [ConchaAnalyze](#)

 [ConchaSetLoadsOnEdges](#)

 [ConchaSetView](#)

It is recommended to use a shaded or rendered view in Rhino3D, in particular for the post-processing of results.

Step 1: Define the model

The starting point is the Figure 1: a rectangle of dimensions $10\text{ m} \times 1\text{ m}$. The first step in every simulation with CONCHA is the definition of the analysis model. Using the command [ConchaAddSurface](#), the rectangle is added to the plug-in and can be processed further. For convenience, it is recommended to change the view settings in order to have a clean workspace. Using the dialog from [ConchaSetView](#), you can hide the original geometry and only display the analysis model.

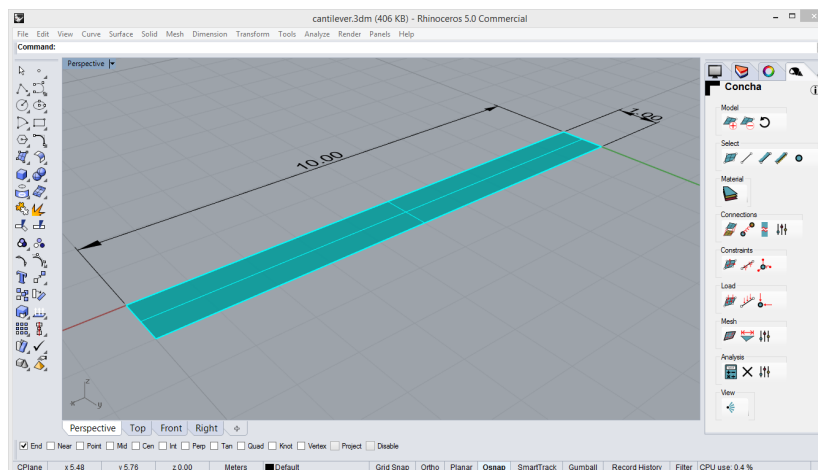


Figure 1: Screenshot of cantilever surface

Step 2: Set the material

Next, the material is defined by selecting the considered surface and using the command **ConchaSetMaterial**. You will get the dialog as in Figure 2. In this example a steel plate with Young's modulus $E = 21\,000 \text{ kN/cm}^2$ and thickness $t = 4 \text{ mm}$ is chosen.

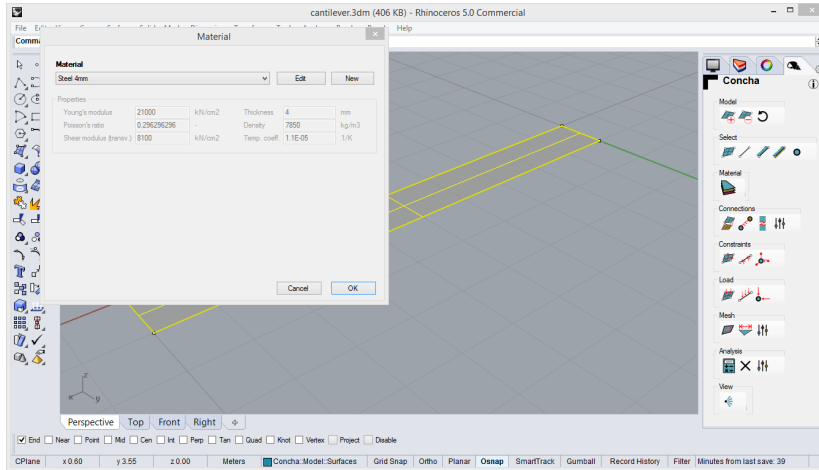


Figure 2: Screenshot of material selection

Step 3: Set constraints and loads

In the following, constraints are defined. For the given example, all displacement components and the rotation of one of the smaller edges of the model surface are set to zero (clamped boundary). To this end, first the edge is selected and then the dialog of the command **ConchaSetConstraintsOnEdges** allows to choose the constraints. Similarly, selecting the opposite edge and using the command **ConchaSetLoadsOnEdges** allows to define loads on that edge. Here, a vertical load of $p = 0.01 \text{ kN/m}$ is entered. In the view settings, **ConchaSetView**, you can adjust the visibility of the model part in order to get the image Figure 3.

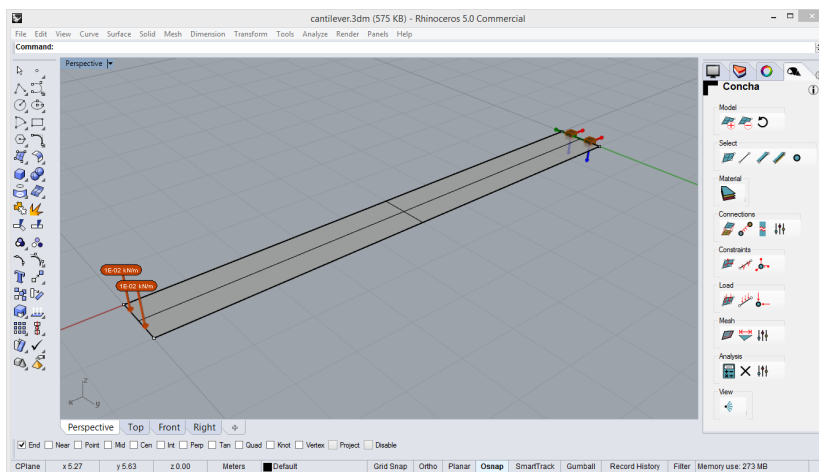


Figure 3: Clamped edge and loaded edge

Step 4: Create a mesh

Now, the problem is fully described and the analysis can be started. As a preliminary, the Finite Element (FE) mesh is needed. Therefore, the command [ConchaCreateMeshes](#) gives directly a mesh for the model geometry. If a different mesh is requested, the command [ConchaMeshSettingsSurface](#) allows to fine-tune the meshing algorithm per surface. Here, a maximal mesh size of 0.4 is used and the mesh as in Figure 4 is obtained.

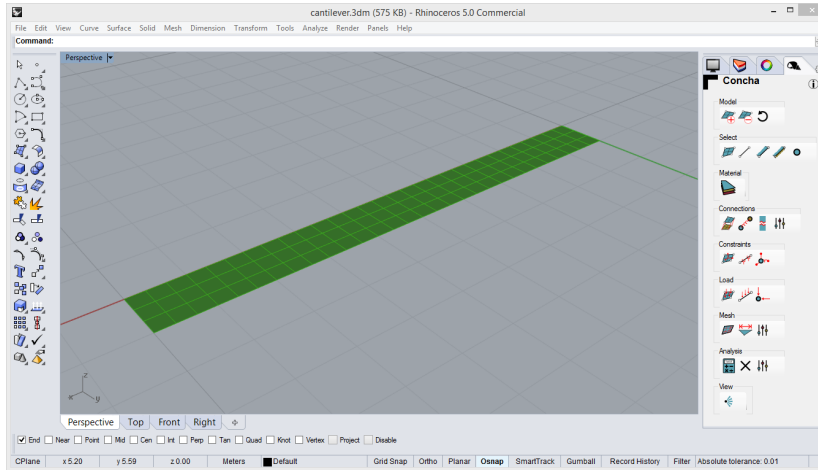


Figure 4: Finite Element mesh

Step 5: Run the analysis

Using the command [ConchaAnalyze](#), the FE calculus is started and automatically gives the deformed shape of the surface. By adjusting the view, you get Figure 5. For a more quantitative analysis, the view settings allow to display other quantities such as the individual displacement components or equivalent stresses, see Section 5.9 and Figure 6 for the z-displacement component.

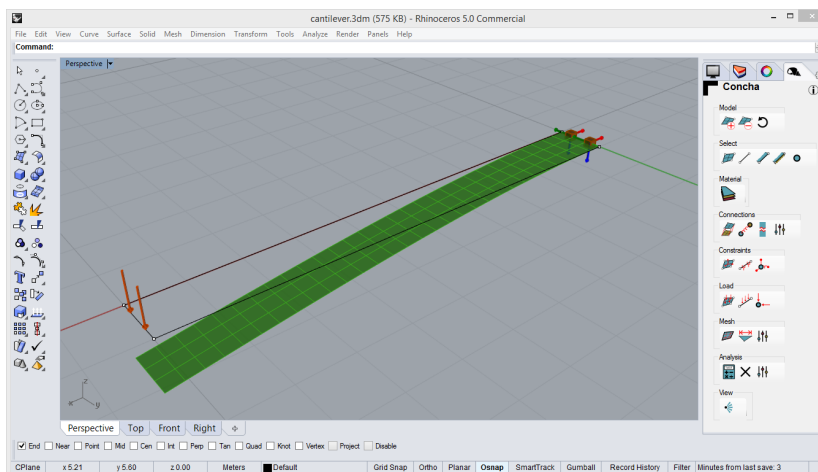


Figure 5: Deformed cantilever plate

Step 6: Post-processing

Using a standard engineering formula for beam deflection, one gets

$$u_{z,\max} = \frac{P\ell^3}{3EI} \quad (1)$$

for the maximal vertical displacement of the corresponding Bernoulli beam model. Here, the values are $P = p \cdot 1 \text{ m} = 10 \text{ N}$, $\ell = 10 \text{ m}$, $E = 2.1 \cdot 10^{11} \text{ N/m}^2$, and $I = \frac{bt^3}{12} = \frac{16}{3} \cdot 10^{-9} \text{ m}^4$. Putting all these values together, one gets approximately $u_{z,\max} = 2.976 \text{ m}$. A close look at Figure 6 reveals a slightly smaller value of 2.951 m. Owing to the difference in models, beam vs. shell, there is a good agreement between these results.

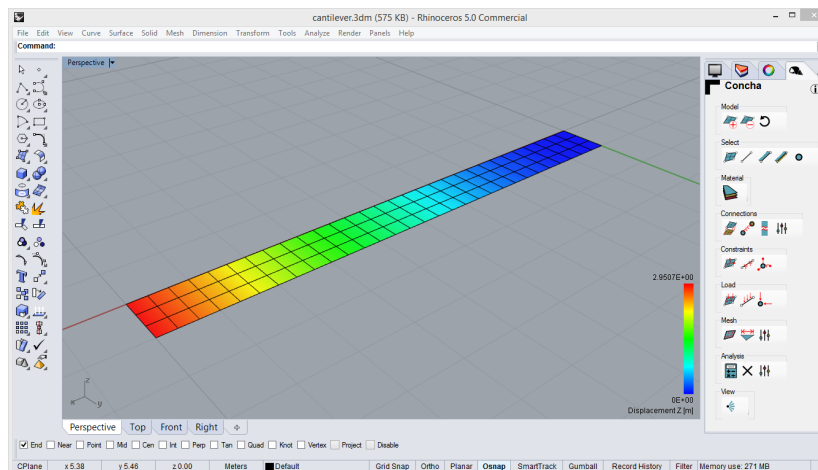


Figure 6: Cantilever coloured by z-displacement

3 Free vs. full version

The version of CONCHA available at <http://www.food4rhino.com/app/concha> will be referred to as the *free* version because it comes without any cost. Consequentially, there is a *full* version with an augmented set of features. Table 1 gives a quick overview of these two versions of CONCHA.

Feature	CONCHA free	CONCHA full
maximal number of elements	2000	unlimited
constraints and connections	rigid	rigid and springs
surface loads	forces	forces, weight and temperature
analysis type	linear	linear and large deformations

Table 1: Comparison of CONCHA versions

In the remaining part of this document, feature descriptions that are dependent on the version (free or full), are visually marked **by using red text** . If you interested in a full version of CONCHA or in any feature not supported by the free version, please contact us:

TAILSIT GmbH
 Nikolaiplatz 4
 8010 Graz, Austria

concha@tailsit.com
 +43 676 974 6416
www.tailsit.com

4 Technical background

CONCHA is a stand-alone plug-in for the structural analysis of shell structures within the CAD environment of Rhino3D. Design surfaces in Rhino3D can be registered as mechanical shells for this plug-in. Furthermore, the user defines supports, inter-object connections, and loads by means of the provided GUI extensions. After creating the Finite Element (FE) mesh, the plug-in's analysis capability can be used to compute the deformed state of the objects under considerations. To this end, the initial and the deformed geometry of the objects are represented by the location of the shell's mid-surface and rotation angles of virtual cross sections. Using uppercase letters for the initial geometry and lowercase for the deformed state, these two configurations are given as follows

$$\mathbf{X}(\boldsymbol{\xi}) = \check{\mathbf{X}}(\xi_1, \xi_2) + \xi_3 \mathbf{D}(\xi_1, \xi_2) \quad \text{and} \quad \mathbf{x}(\boldsymbol{\xi}) = \check{\mathbf{x}}(\xi_1, \xi_2) + \xi_3 \mathbf{d}(\xi_1, \xi_2) \quad (2)$$

with the in-plane parameters ξ_1 and ξ_2 in-plane and the thickness coordinates ξ_3 . Moreover, $\check{\mathbf{X}}$ and $\check{\mathbf{x}}$ denote the locations of the initial and final mid-surface. \mathbf{D} is a unit vector normal to the tangent plane at $\check{\mathbf{X}}$ which is mapped to \mathbf{d} due to the deformation. With the configurations (2) at hand, the state of deformation can be measured by means of the Green-Lagrange strain tensor

$$\mathbf{E} = E_{ij} \mathbf{G}^i \otimes \mathbf{G}^j, \quad \text{with} \quad E_{ij} = \frac{1}{2} \left[\mathbf{g}_i \cdot \mathbf{g}_j - \mathbf{G}_i \cdot \mathbf{G}_j \right]. \quad (3)$$

Here, the co-variant basis vectors $\mathbf{G}_i = \partial_i \mathbf{X}$ and $\mathbf{g}_i = \partial_i \mathbf{x}$ are used, respectively, and the contra-variant counterparts \mathbf{G}^i such that $\mathbf{G}^i \cdot \mathbf{G}_j = \delta_j^i$. The second Piola-Kirchhoff stress tensor $\mathbf{S} = S^{ij} \mathbf{G}_i \otimes \mathbf{G}_j$ is related to the strains by means of a *linear* relation

$$S^{ij} = C^{ijkl} E_{kl} \quad (4)$$

with the fourth-order elasticity tensor C^{ijkl} . Finally, the balance of linear momentum leads to the weak form of the equilibrium equations

$$\delta W^{\text{int}} - \delta W^{\text{ext}} = 0, \quad \text{with} \quad \delta W^{\text{int}} = \int_{\Omega} \mathbf{S} : \delta \mathbf{E} \, d\Omega. \quad (5)$$

Here, the integration is carried out over the undeformed geometry Ω and the integrand contains the first variation of the Green-Lagrange strains, $\delta \mathbf{E}$. Equation (5) is the non-linear equation whose Finite Element solution is provided by CONCHA. For the discretisation of this equation, MITC9 and MITC6 shell elements¹ are used for quadrilaterals and triangles, respectively.

An additional feature of CONCHA is that the meshes of adjacent surface patches need not be conforming. In other words, the FE nodes along the edge that joins the two surfaces are at different geometric locations for the two sides. CONCHA uses a technique based on a penalty method that can handle this case. The benefit of this approach is that there is a larger degree of freedom in the design of FE mesh because the constraining condition of a matching node locations along the interface is relaxed.

¹see, e.g., K-J Bathe, *Finite Element Procedures*, Prentice Hall, 2007


5 User interface


The plug-in CONCHA is controlled via a panel which can be activated by checking *Concha* in the panel properties usually located on the right of the Rhino3D window. The top-down design of the panel follows a typical workflow for the simulation with CONCHA. Additionally, a toolbar is provided and located in the installation directory.


In the following sections the buttons and commands that CONCHA provides are briefly explained.

5.1 Analysis model

In this section, you can add surfaces and polysurfaces to the analysis model. These surfaces are registered and *copied* in the document and every subsequent action of the plug-in CONCHA operates on these copies. Geometry objects that are generated throughout the analysis, such as constraint symbols or load arrows, can be identified by the prefix *Concha* in their names.


 **ConchaAddSurface** Add a surface to the analysis model.
This command adds boundary representations to the model which can be surfaces, polysurfaces or selected faces of polysurfaces and extrusions. After selecting the objects, CONCHA generates duplicates which become the analysis surfaces in the document. These duplicates are managed in CONCHA's own layer tree. The edges and vertices of the analysis surfaces are determined by the topology of the original boundary representation.


 **ConchaRemoveSurface** Remove surface from the analysis model.
Allows to remove CONCHA-generated analysis surfaces from the model. Make sure to pick those by selecting surfaces with the prefix *Concha*. The connected edges and vertices will automatically be removed too.




 **ConchaReset** Reset the analysis model.
Using this command, all objects generated by the plug-in are removed and the analysis is completely aborted.

5.2 Selection of items

You can always select single or multiple geometry objects by the normal Rhino3D selection mechanisms and choosing those generated by CONCHA. The following commands help you select directly all objects of a specific kind.


 **ConchaSelectSurfaces** Select all surfaces.
With this command you automatically select *all* surfaces that are part of the analysis model. This means all surfaces that have been created with [ConchaAddSurface](#) are selected here.

 **ConchaSelectEdges** Select all edges.
Select all edges of the model geometry. Note that these edges are created when [ConchaAddSurface](#) is used to add surfaces to the model.

-  **ConchaSelectFreeEdges** Select free edges only.
Edges that are adjacent to only one surface are labelled as *free*. This command selects all such edges.
-  **ConchaSelectInternalEdges** Select internal edges with face angle.
All edges that connect (at least) two surfaces are labelled as *internal*. This command automatically selects all internal edges whose adjacent faces do not lie in the same plane.
-  **ConchaSelectVertices** Select all vertices.
Every two adjacent edges that are created by the plug-in are joined by a *vertex*. This command selects all of them.




5.3 Material definition

Different materials types, single and multiple layers, are supported by CONCHA. Moreover, custom materials can be added to the analysis. **The free version of CONCHA only allows a limited material selection and custom materials can only be composed of one layer. Moreover, in the full version you can edit an external CSV-formatted material database which defines the materials available to CONCHA.** By default, the material of every surface is set to plain steel of 4 mm.

-  **ConchaSetMaterial** Define material behaviour of a surface.
Opens the material dialog for selected surfaces. Then you can choose from a list of predefined materials or define a custom material for which the relevant material parameters have to be entered.

5.4 Constraints

In this section, the displacements (and rotations) of geometry objects can be controlled by applying specific constraints. **In the free version, displacements and rotations can only be constrained by rigid supports. The feature of applying springs with custom stiffness values is only available in the full version.**

-  **ConchaSetConstraintsOnSurfaces** Apply constraints to surfaces.
For all selected surfaces the global displacement components (*x*-, *y*- or *z*-direction) can be set to zero.
-  **ConchaSetConstraintsOnEdges** Apply constraints to edges.
Global displacement components of selected edges and the rotation around the edge axis can be set to zero.
-  **ConchaSetConstraintsOnVertices** Apply constraints to vertices.
The global displacement components of the selected vertices can be set to zero.

5.5 Connections

CONCHA allows to connect the displacements of different geometry objects with the same dimension, that is vertices with vertices, edges with edges and so on. In these connections,

chosen displacement components are forced to be equal. In the free version of CONCHA, these connections are rigid. The full version also supports the use of springs with custom stiffness that connect specific displacements. Note that surface edges at the same location (that is the edges that join adjacent surfaces) are connected by default. Their connection status can be set by [ConchaSetConnectionProperties](#).

**ConchaSetMasterSurface**

Connect surfaces.

Surfaces that have an identical shape, size and orientation, but are located at different positions in the document can be connected.

**ConchaSetMasterVertex**

Connect vertices.

Any two vertices can be connected with this dialog.

**ConchaRemoveMaster**

Remove a connection.

Previously established connections can be undone with this command.

**ConchaSetConnectionProperties**

Set the properties of a connection.

Any connection established by [ConchaSetMasterSurface](#), [ConchaSetMasterVertex](#) or by default for edges can be modified with this command.

5.6 Loads

In this section, external loads on different geometry objects can be specified. Loads are always constant along the object on which they apply. Whereas the free version only allows for external forces to be applied, the full version also supports thermal loads by specification of a temperature change and the selection of the material weight as an additionally considered load.

**ConchaSetLoadsOnSurfaces**

Apply loads on surfaces.

For the selected surfaces, force loads in global direction components or normal to the surface can be specified.

**ConchaSetLoadsOnEdges**

Apply loads along edges.

For the selected edges, force loads in global direction components can be specified.

**ConchaSetLoadsOnVertices**

Apply loads on vertices.

For the selected vertices, force loads in global direction components can be specified.

5.7 Mesh

Here, the FE analysis mesh will be specified. For the subsequent analysis, all the surfaces of the analysis model are decomposed into quadrilaterals or triangles, whereas the former are preferred. Note that the mesh is carried out surface-wise and need not be conforming across the edges of adjacent surfaces. CONCHA will glue these meshes together without requiring a mesh conformity! Note that the free version of CONCHA allows for a maximum of 2000 elements. The full version does not pose this restriction.

**ConchaCreateMeshes**

Create mesh.

Creates the FE analysis meshes for surfaces added by [ConchaAddSurface](#). Any existing mesh is discarded.

**ConchaMeshSettingsSurface**

Set mesh properties for surfaces.

For all selected surfaces, the mesh properties can be specified with this command. In particular, you can set the maximal mesh size, the maximal aspect ratio and a minimal number of elements per surface. Using auto-meshing, global parameters for meshing as defined by [ConchaMeshSettingsGlobal](#) are used.

**ConchaMeshSettingsGlobal**

Global mesh parameters.

Choose if CONCHA's built-in meshing heuristic is used or define a global mesh size.

5.8 Analysis

The model is defined by a set of surfaces, constraints and connections, the material parameters and the applied loads. Once the FE analysis mesh is created, the calculation can be carried out. For the free version of CONCHA, only a linear analysis can be carried out. The full version supports arbitrarily large deformations by means of a Newton method with a consistent tangent stiffness matrix. Furthermore, in the full version the analysis generates data files in the VTU format for more enhanced post-processing features.

**ConchaAnalyze**

Invoke the FE analysis.

Perform an FE analysis of the defined CONCHA model. That status of the analysis is reported in the top left corner of the Rhino3D view. If the model has insufficient constraints, the user will be informed accordingly. If successful, the analysis generates post-processing data that can be viewed as described for the [ConchaSetView](#) command.

**ConchaAbortAnalysis**

Abort the analysis.

**ConchaSetAnalysisProperties**

Define properties of the FE analysis.

For the free version of CONCHA, this dialog does not offer much. Here, properties for the nonlinear solution algorithm can be set and file paths for VTU and log-files are specified.

5.9 View

This part gives control over which objects are visible and which data are presented for the post-processing. CONCHA generates geometry objects throughout the analysis, which are at the same location as the CAD model used as the starting point of the analysis. For a cleaner interaction with the plug-in you can turn on and off the visibility of the *Geometry* (all objects present in the document before CONCHA was invoked) and of the *Model* (all objects add through [ConchaAddSurface](#) and created through subsequent commands). Moreover, for the model part the visibility of items such as constraint symbols or load vectors can be controlled.

**ConchaSetView**

Set view properties.

The dialog invoked by this command has two tabs:

Model The model tab has the following commands

Show unused geometry Control the visibility of geometry objects that are *not* part of the analysis model.

Show registered geometry Control the visibility of geometry objects that have been registered with [ConchaAddSurface](#)

Show mesh Control the visibility of the FEA mesh

Show model Control the visibility of the CONCHA model. These are all objects that have been created by the plug-in: duplicates of the registered surfaces, edges, vertices, load and constraint symbols.

Results The results tab controls the visibility and appearance of results for displacements, stresses and reaction forces. The user can control the colours, the range and the visibility of a colour bar. In addition, nodal values can be displayed at the model. In detail, the user can control

Displacements: Choose between a warped figure with custom scaling parameter or a colour representation of individual components or the magnitude

Stresses: equivalent stresses of von-Mises type can be shown for the top, middle or bottom layers of each surface. This ordering is associated with the surface normal which points from bottom to top layer. Not that this layers are independent of the layer-structure of the material.

Reaction forces: the forces at the model supports are displayed by either a surface colouring of as arrows located at the nodes of the FE mesh.

5.10 Information

CONCHA has been produced by TAILSIT GmbH, Graz, Austria.

ConchaAbout

Information about CONCHA.

This command opens an information window with contact information, version number of the plug-in and reference to the license.

6 License

The license for the free version of CONCHA can be downloaded from <http://www.tailsit.com/concha-license.rtf>.

7 Contact

For interest in extra features, purchase of a full version, or any comments or requests concerning CONCHA, please contact:

TAILSIT GmbH
Nikolaipplatz 4
8010 Graz, Austria

concha@tailsit.com
+43 676 974 6416
www.tailsit.com

List of commands

ConchaAbortAnalysis, [12](#)
ConchaAbout, [13](#)
ConchaAddSurface, [9](#)
ConchaAnalyze, [12](#)
ConchaCreateMeshes, [11](#)
ConchaMeshSettingsGlobal, [12](#)
ConchaMeshSettingsSurface, [12](#)
ConchaRemoveMaster, [11](#)
ConchaRemoveSurface, [9](#)
ConchaReset, [9](#)
ConchaSelectEdges, [9](#)
ConchaSelectFreeEdges, [10](#)
ConchaSelectInternalEdges, [10](#)
ConchaSelectSurfaces, [9](#)

ConchaSelectVertices, [10](#)
ConchaSetAnalysisProperties, [12](#)
ConchaSetConnectionProperties, [11](#)
ConchaSetConstraintsOnEdges, [10](#)
ConchaSetConstraintsOnSurfaces, [10](#)
ConchaSetConstraintsOnVertices, [10](#)
ConchaSetLoadsOnEdges, [11](#)
ConchaSetLoadsOnSurfaces, [11](#)
ConchaSetLoadsOnVertices, [11](#)
ConchaSetMasterSurface, [11](#)
ConchaSetMasterVertex, [11](#)
ConchaSetMaterial, [10](#)
ConchaSetView, [12](#)