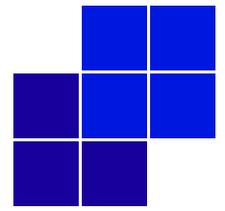


News.St7



Newsletter for Strand7 and Straus7 users

Strand7 is marketed as Straus7 in Italy

Issue 9, July 2013

Welcome to another issue of News.St7. In this issue we highlight some of the new features in the recently released Strand7 R2.4.5. We also report on the very successful launch of the Strand7 Webnotes – a comprehensive online library of resource material available to supported Strand7 users. As usual, we include interesting technical content and *Did You Know?* items. We also profile a new Strand7 API user development for the design of steel members to AS 4100.

We hope you enjoy reading this issue of News.St7, and look forward to hearing your comments. We also welcome suggestions for topics you would like to see in future issues. 

If you have any feedback or suggestions regarding the content of News.St7 or you would like to contribute a case study showing how you use Strand7, please email us at news.st7@strand7.com. If you would like to receive your copy of News.St7 automatically by email, simply send a blank email to newsletter-subscribe@strand7.com. All care is taken to ensure that information in News.St7 is accurate and up to date at the time of publishing. However Strand7 Pty Limited accepts no responsibility for inaccuracies in, or changes to, such information.

In this Issue

• Strand7 Release 2.4.5	1
• Strand7 Webnotes	2
• Analysis of an Exhaust System	3
• Import and Export	4
• Glass Façade Blast Analysis	6
• Strand7 API Developer	7
• Training and Exhibitions	8
• Website Update	8

Strand7 Release 2.4.5

Strand7 R2.4.5 is now available for download from our update site, www.strand7.com/st7update. It enhances R2.4.4 with greater stability, improved nonlinear convergence, and a few subtle user-interface enhancements such as better support for multiple monitor environments.

R2.4.5 also addresses a number of issues in Strand7 R2.4.4, which have been brought to our attention since its release; so please do take advantage of your release entitlement and download this worthwhile update.

Changes in R2.4.5

The following is a summary of the changes. For a complete list, please see the document **What's New in Strand7 R24.pdf**, located in your Strand7 installation folder.

1. New Selection Mode – Selection All Behind

Holding the **Alt + Ctrl** keys when selecting now selects all entities behind the selection box, even if you can't see them. This selection mode is indicated by a solid double line when selecting.

Note that the previously available selection mode, Line Selection, can still be used. By pressing the **Alt** key (without the Ctrl key) when dragging, the selection box becomes a line, which is very useful for selecting entities inclined in the current view.

See Main Tool Bar/Select/Individual Select in the Strand7 Online Help for more information and examples.

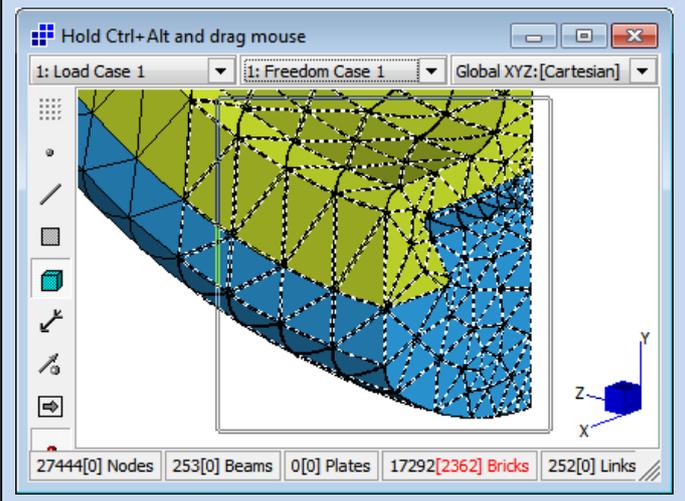
Did You Know?

Selecting All Entities Behind the Selection Box

One new feature in R245 is the ability to select both visible and obscured entities by using **Ctrl+Alt** when dragging a selection box.

Instead of selecting only those entities which are visible from the current view point, all entities behind those shown are also selected.

This alternative selection mode is graphically shown during selection on the screen as a solid double line.



2. New Groups Right-click Options

In the **Groups** dialog, right-clicking on a group gives four options for expanding the sub-groups within: **Expand All**, **Collapse All**, **Expand Selected** and **Collapse Selected**.

3. Brick Mesh Internal Attachment Links

When attaching elements with **Tools/Attach Parts**, the new option **All Faces** for brick attachments allows the attachments to form between elements embedded within a brick mesh and the interior faces of the brick elements, not just the outer (or free) faces.

4. Beam Fibre Strain Increment Sub-Stepping Control

For material nonlinear solutions involving nonlinear fibre stress beam elements, a sub-stepping trigger has been added which can be used to limit the incremental plastic strain a beam can experience in a single load increment or time step. This helps convergence stability in MNL problems.

5. Save Solver Restart Files Only Last Step Option

When saving restart files, a new option **Save only last step (Minimum size)** is now available. This option is intended to allow a restart from the last saved increment only, instead of at every saved increment. This can save a significant amount of disk space in large models.

6. Surface Automesher Enhancements

The **Skip Transitioning** option has been renamed to **Apply Auto Transitioning**. A new control **Consider Nearby Vertices** is added to allow the user to consider or ignore adjacent vertices which are not actually connected to the current face. Thus if there is a gap between two parts, that gap would determine the local mesh size if the **Consider Nearby Vertices** option is set.

7. Automatic Load Path Divisions

This new option is found in the transient solver **Moving Load...** dialog. When set, it overrides the **Divisions** attribute of the load paths, instead making sure that there is at least one division where each vehicle may land at any time step. In other words, the load path divisions are controlled by the time step. This enhances the accuracy of transient moving load solutions by ensuring that enough divisions are taken along a load path.

Load paths default to 20 divisions, with the option to manually change this with **Attributes/Load Path/Divisions**. This is still available, and if the number of user-defined divisions is adequate for a particular solution, the **Automatic Load Path Divisions** option will not make a difference.

8. Seismic Load Cases Consider NSM Dynamic Factors

A new option **Apply Dynamic Factors** in seismic load cases specifies that the dynamic factors applied to the included non-structural mass should be considered in the equivalent static seismic analysis. Previously, you could scale all non-structural mass via the phi parameter. This new option allows NSM in different load cases to be scaled independently.

9. Enhanced Import Functionality

SAP2000 imports now support **SAP2000 TABLE: "AREA LOADS - UNIFORM TO FRAME"** loading. Additionally, a new option, **Merge Duplicate Freedom Cases** is available. Nastran imports now support SPCADD, MPCADD, PCOMPG, BAROR and BEAMOR bulk data entries. Ansys imports now support the *GET command.

10. Updated Material and Section Libraries

Composites libraries for Aramid, Carbon and Glass Fibre have been updated. A new library for Glass has been added. Several South African section libraries have also been updated.

11. New API Functionality

The following new API functions have been added in R2.4.5:

- St7SetPropertyCreepID
- St7GetPropertyCreepID
- St7EnableSaveLastRestartStep
- St7DisableSaveLastRestartStep
- St7ShowSelectionToolBar
- St7HideSelectionToolBar
- St7SetSelectionToolBarPosition
- St7GetSelectionToolBarPosition
- St7DeleteGroup
- St7GetInertiaReliefResults
- St7SetPlateLayers
- St7GetPlateLayers
- St7EnableAutoAssignPathDivisions
- St7DisableAutoAssignPathDivisions
- St7EnableModelStrainUnit
- St7DisableModelStrainUnit
- St7EnableModelRotationUnit
- St7DisableModelRotationUnit
- St7SetQuasiStaticRestartFile
- St7GetQuasiStaticRestartFile
- St7SetNodeHistoryFile
- St7GetNodeHistoryFile

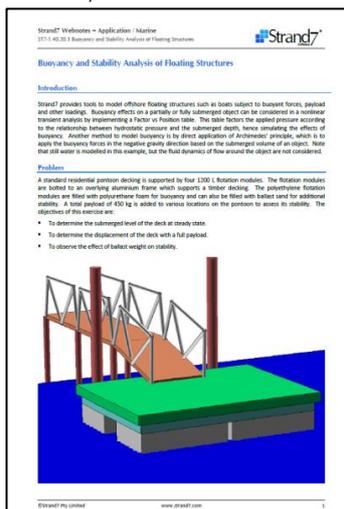
Additionally, API headers and examples for C# and Python have been added to the R2.4.5 distribution. ■■■

Strand7 Webnotes

Strand7 Webnotes is a repository of documents and Strand7 model files, available online to supported Strand7 users. Many topics are covered, with cross-linked content between Webnotes. A Webnote is a PDF document or a ZIP file, if there are associated model files.

Webnotes are found on our Website at www.strand7.com/webnotes. The index of all Webnote descriptions can be browsed without entering a password, and is searchable. Once a Webnote link is selected, the user is asked to enter a username and password to download the content. This username and password is the same as the one you would use to download Strand7 updates, and can be found in your installation CD jacket, by contacting your IT support, or by emailing webnotes@strand7.com.

Each Webnote is prefixed by a unique ID number which indicates the section and sub-section. For example, [ST7-1.10.20.2 Harmonic Response Analysis](#) is the second document in major section **Linear** (10) and



Making finite element analysis easier.

minor section **Dynamics** (20). There are presently nine major sections, each with multiple sub-sections.

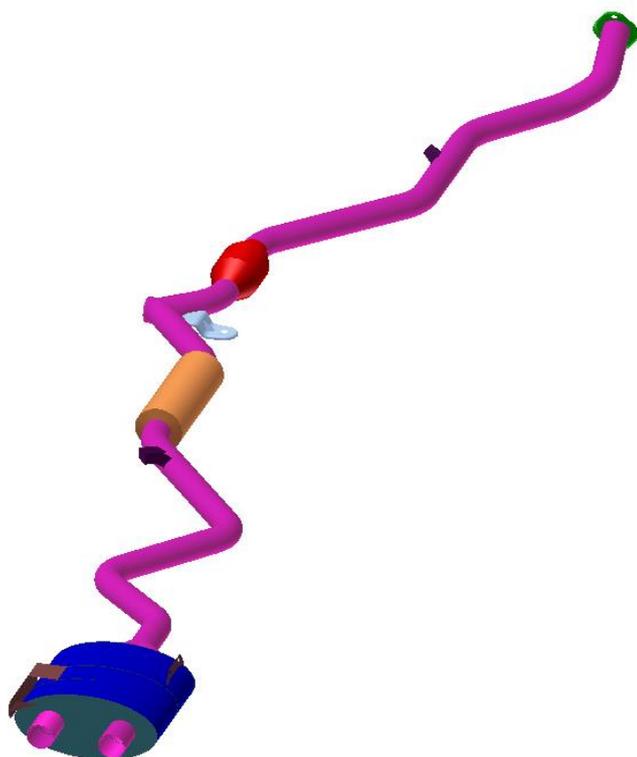
Since our Webnotes launch on 16-May, we have been receiving regular positive feedback about this initiative. We are expanding the Webnotes library with new releases posted regularly, so please check the website for the latest additions, or email us to get on the mailing list for Webnotes announcements.

We hope that providing online access to the Webnotes will enhance the skills of our user base, as well as the quality of the Strand7 support experience. We welcome any feedback about this new system. Requests for topics which are not yet covered can be directed to webnotes@strand7.com. Contact info@strand7.com if you are unsure of the support status of your company, or if you wish to purchase support. ■■

Analysis of an Exhaust System

This article presents a summary of the Webnote [ST7-1.40.70.8 Tuning an Exhaust System with Harmonic Analysis](#).

Automotive exhaust systems play a large role in the comfort of a vehicle. How the exhaust system interacts with the engine and car body can be determined using harmonic response analysis. This type of analysis determines the steady-state vibration response of a system to a sinusoidally varying input, such as that from a car engine. To provide maximum sound attenuation, the exhaust system must be "tuned" such that the range of forcing vibration of the car does not cause it to resonate. Additional considerations include avoiding resonance between the floor of the car body and the exhaust system.



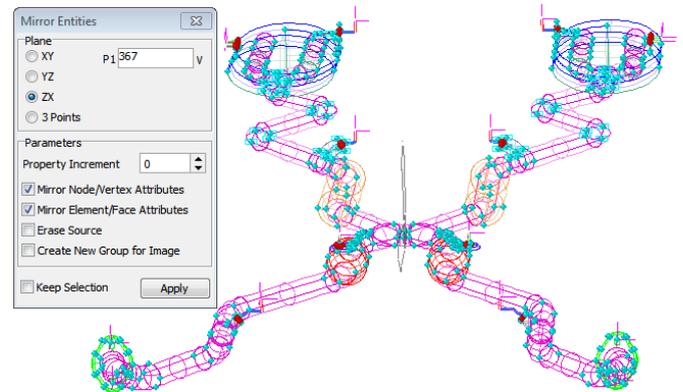
Half symmetric exhaust system CAD geometry.

The example geometry is based on a V8 exhaust system. It has two symmetric, centrally linked exhaust pipes. Each has a catalytic converter, a resonator and a muffler.

The exhaust system is forced by pressure pulses and by direct engine vibration at the front. The acoustic tuning of the exhaust system (i.e. dimensions of the internal cavities etc.) is assumed to have been previously performed to attenuate acoustic energy within the exhaust gas.

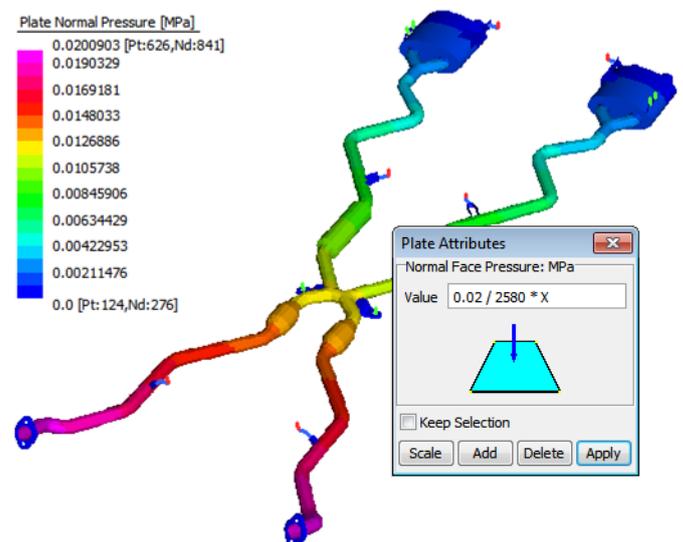
The goal of the tuning procedure is to avoid known car body floor resonance points of 65 Hz, 96-100 Hz, 106 Hz and 127 Hz by at least 3 Hz, and to attenuate direct engine vibration by at least -20 dB from idle to 3500 RPM, and by -10 dB above 3500 RPM as measured at the rubber mounts. Additionally, the peak maximum principal stress should be below the endurance limit of the steel (assumed to be 250 MPa) to avoid fatigue cracking.

Geometry is imported from CAD (using IGES format) and mirrored to form the whole system. This is necessary because vibrational modes which are not symmetric will be present.



Strand7 geometry after import and mirror - note that the mirror operation on the geometry has been performed in Strand7.

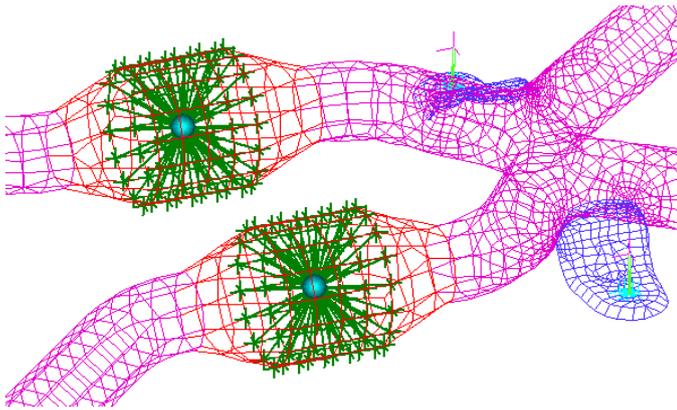
Normal pressure is applied as a function of distance from the engine to model the drop in pressure from the header to the muffler outlet.



Pressure load applied with variation along the length. In Strand7 it is easy to apply attributes that vary with spatial position - simply enter the required equation in any of the attribute dialogs ("0.02/2580*X" has been used above to apply pressure that varies with X).

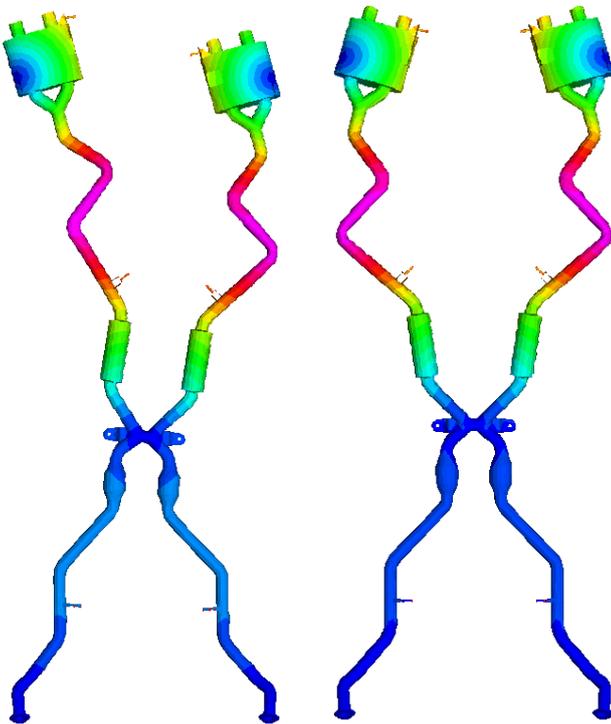
Making finite element analysis easier.

Catalytic converters are included as lumped masses connected to the housings with interpolated multi-point links.



Catalytic converter mass added using multi-point links. The advantage of these links over rigid links is that they distribute the mass over the connected nodes without affecting the structural stiffness. See Main Tool Bar/Create/Link/Multi-Point Link in the Strand7 Online Help for a simple example that compares the difference between these and the rigid links.

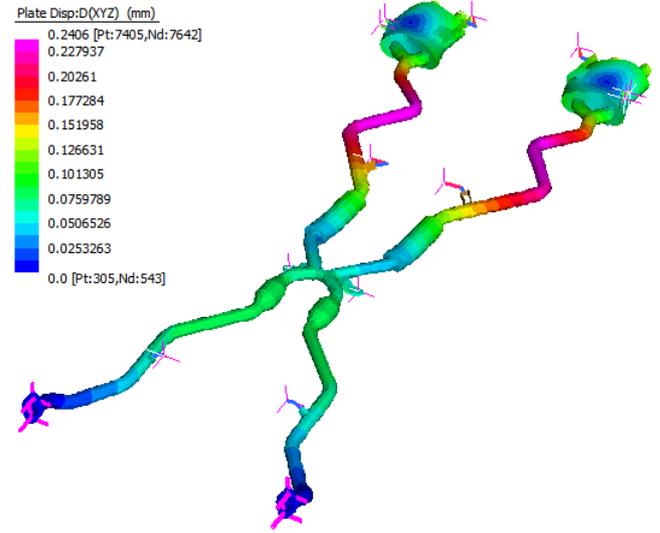
Natural frequency results are obtained to determine the natural vibrational response of the structure.



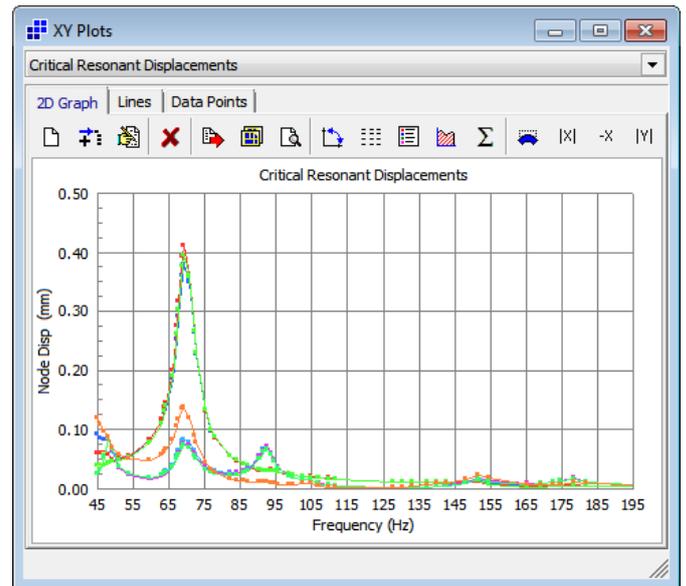
Two natural frequency mode shapes of the system.

The natural frequency results are used to produce a Harmonic Response analysis. This gives peak results including displacement, velocity, acceleration and stress across the operational range of the structure.

This output is graphed and compared to the frequencies which are to be avoided.



Harmonic response of the system to exhaust pulses.

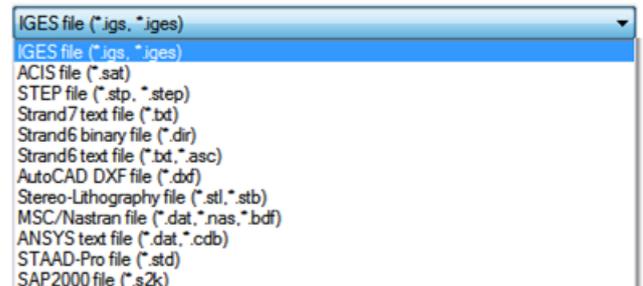


Harmonic response displacement vs frequency results.

Exchanging Geometry and Model Data

Strand7 imports and exports a variety of formats to facilitate the sharing of data and the building of models based on CAD geometry.

This functionality is found in **File/Import** and **File/Export**. The file type can be selected at the bottom of the dialogs, as shown below:



Strand7 can import and export the following file types:

Making finite element analysis easier.

1. IGES (*.igs, *.iges)

IGES files hold 2D and 3D CAD geometry. It is typically exported from CAD and imported into Strand7 to use with the automesh. You can also export Strand7 geometry into IGES format after modifying (or generating) it in Strand7. The IGES format supported by Strand7 does not hold finite element information – only CAD geometry. Available for import and export.

2. ACIS (*.sat)

ACIS files are roughly equivalent to IGES files. Strand7 supports most of the ACIS types, with a few exceptions such as rolling ball blends. Available for import only.

3. STEP (*.stp, *.step)

STEP is also equivalent to IGES, but is a newer ISO standards specification. If IGES isn't importing your geometry well, try STEP or ACIS. STEP can also be used to export geometry from Strand7 into CAD. Available for import and export.

4. Strand7 Text File (*.txt)

The Strand7 text file format is a Strand7 model in readable ASCII (text) format. This can be used to pass newer Strand7 models into older versions of Strand7 (e.g. opening an R2.4.x model in R2.3.x). It may also be useful if you want to manually modify specific parts of the model using a text editor. This format only holds element information, so no CAD geometry is transferred using this format. Available for import and export.

5. Strand6 Binary File and Text File *.dir, *.txt, *.asc

These import types can be used to translate Strand6 models into Strand7 format. Strand6 was the precursor to Strand7, and was DOS based. Available for import only.

6. AutoCAD DXF (*.dxf)

The DXF format is used by Strand7 to import simple geometry which is represented by plate and beam elements. This format is not appropriate for importing geometry for automeshing but can be very useful for transferring wireframe models such as beam frameworks. Exporting a model in this format will produce a file which AutoCAD can read as a 3D model. If you plan to automesh, don't use this format – use IGES, ACIS or STEP. Available for import and export.

7. Stereo-Lithography (*.stl, *.stb)

This file format is a triangulated surface representation of an object. The triangles (represented by Tri3 elements in Strand7) are generally not constrained by quality concerns if sourced from a typical CAD package, and so may have very small angles, which makes them undesirable from a finite element perspective. This format has been used to exchange data from biomedical applications, such as MRI scanned solids. Available for import and export.

8. MSC/Nastran (*.nas, *.dat, *.bdf)

This is the text-file model format for NASTRAN, a finite element code which has been around for a very long time. The format is correspondingly widely used, and can be used to exchange model information with many FEA packages. Available for import and export.

9. ANSYS (*.dat, *.cdb)

This is the text-file model format for ANSYS, another finite element code. Use this format to exchange models with ANSYS, or other programs which can import ANSYS models. The import functionality includes the option to import IGES entities embedded within the ANSYS file. Available for import and export.

10. STAAD-Pro (*.std)

This is the text-file model format for STAAD-Pro, a structural analysis based finite element and design program. Use this format to import STAAD-Pro models. Available for import only.

11. SAP2000 (*.s2k)

This is the text-file model format for SAP2000, a structural design and analysis program similar to STAAD-Pro. Use this format to import SAP2000 models. Available for import only. 

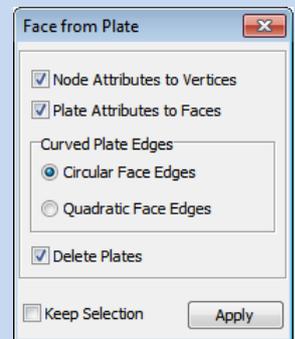
Did You Know?

Geometry Creation in Strand7

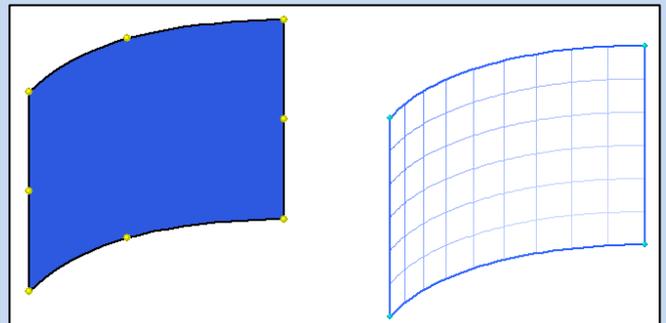
As an alternative to importing CAD geometry for automeshing, you can create geometry in Strand7. This is done by creating plate elements, and using **Tools/Geometry Tools/Face from Plate** to convert them into geometry faces.

Curved plates (i.e. Quad8, Quad9 and Tri6) can be used to create curved geometry faces.

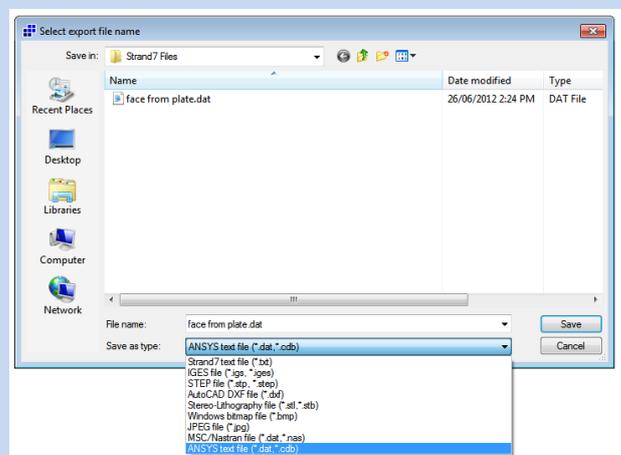
Set the option **Circular Face Edges** to create geometry faces on underlying cylindrical geometry. Otherwise, the quadratic shape of the original plate is retained for the geometry and the face becomes a b-spline.



This functionality can be used to patch holes in pre-existing geometry, or to build geometry to automesh in Strand7, independently of CAD software.



These faces can then be automeshed, or exported to CAD using **File/Export** and selecting IGES or STEP formats.

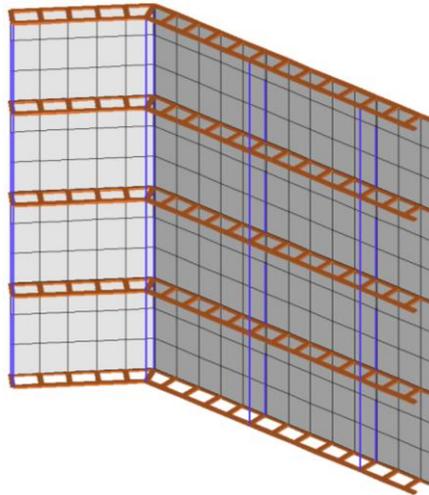


Note that it is also possible to convert planar polygons into faces by using **Tools/Geometry Tools/Face from Beam Polygon**.

Blast Analysis of a Glass Façade

This article presents a summary of the Webnote [ST7-1.40.35.1 Glass Façade Blast Loading](#).

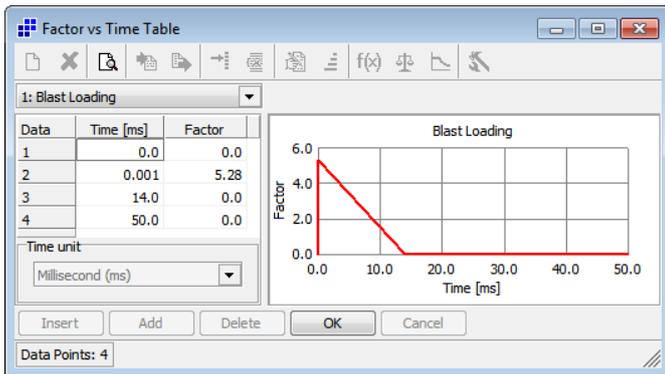
Glass Façade construction is subject to blast loading considerations. These unique structures come in a variety of custom configurations suited to the particular needs of each installation. Due to the variability of each design, Finite Element Analysis can be a very powerful and cost-effective tool in the qualification of these structures.



Four load cases represent the loads on the structure:

1. Self-weight (1g load)
2. Blast Pressure (6.894 kPa)
3. Cable Pre-Tensioning (20 kN)
4. Wind Load (0.478 kPa)

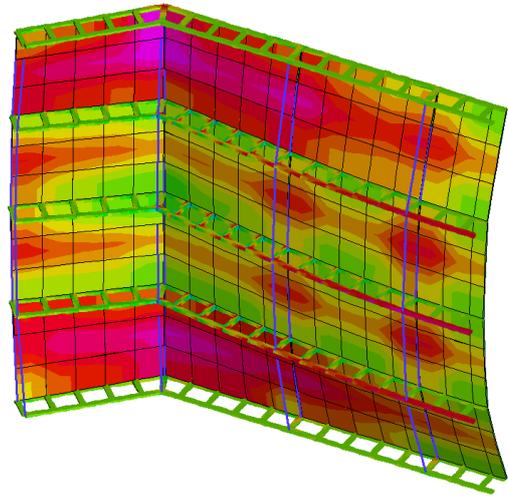
A load table is defined to scale the blast pressure through time in the nonlinear transient dynamic analysis. The actual blast pressure is factored by 5.28, and decreased to zero over a period of 14 ms.



Blast pressure factor vs time table.

The nonlinear static solver is run before the blast load is applied. The results for the Self-Weight + Wind case (pre-blast static condition) are shown below for the glass and horizontal frames. Deflection has been exaggerated for clarity.

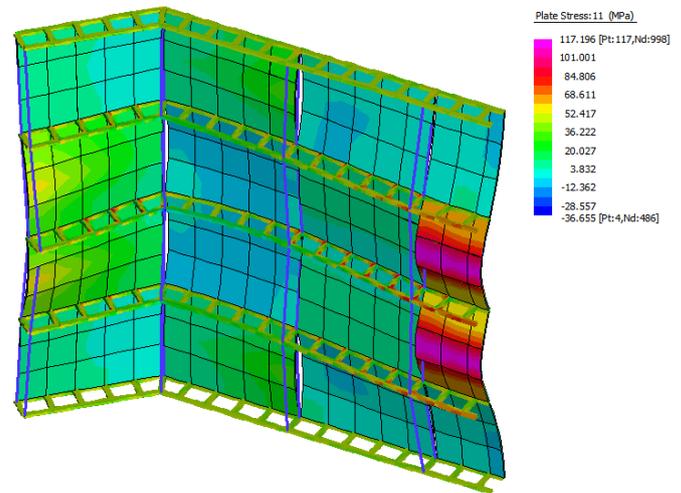
The nonlinear transient dynamic solver requires the definition of loads to apply, the tables that factor the loads, and a time stepping scheme. The time stepping depends on the nature of the problem, and in this case we will choose to take 100 x 1 ms steps, for a total simulation time of 100 ms. Additionally, we must specify any initial static conditions for the façade.



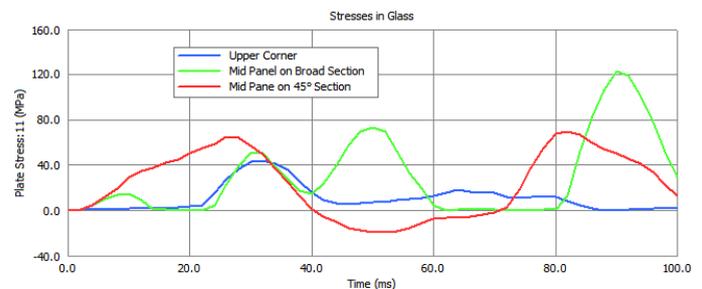
Pre-blast static condition.

Small 1 ms time steps are taken to adequately resolve the blast load, which occurs over the first 14 ms of the analysis. Only every second calculated time step is saved during this phase.

The deformed structure at 90 ms demonstrates the whipping action of the glass as it reverses direction due to loading from the supporting horizontal frames.

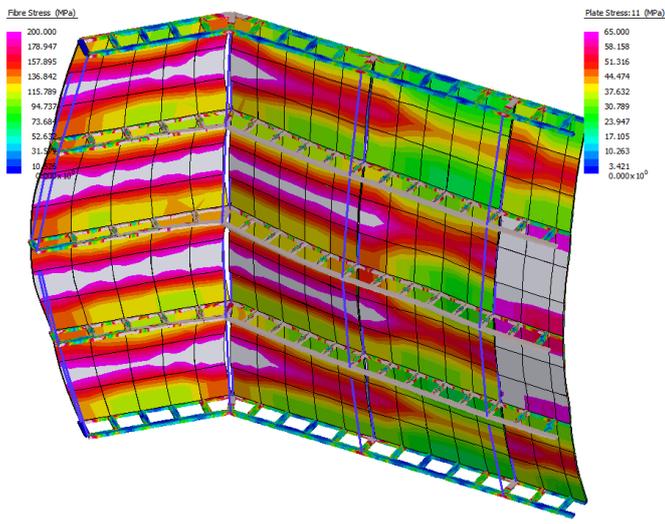


Deformed shape and stress at 90 ms.



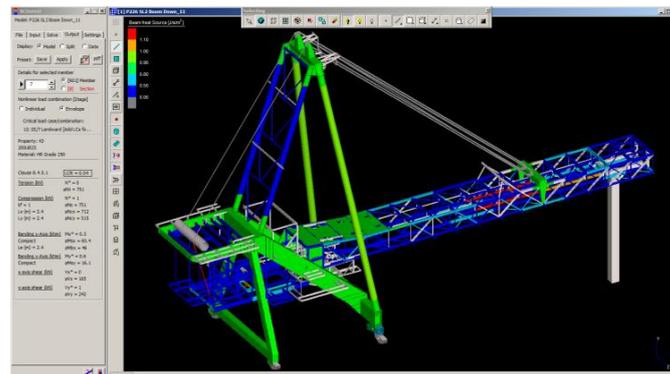
Graph of glass stress time history.

An envelope of the glass stresses through time is easily created (using **Results/Envelopes**), and shows the regions of the façade which exceed the 65 MPa limit (shown in grey in the figure below).



Envelope of maximum blast stresses through time.

BCDsteel has been in use in a number of consulting offices throughout Australia since 2010. Earlier this year, with the release of Strand7 R2.4.5, BCDsteel became commercially available. It is capable of analysing all types of steel structures from simple building frames to complex models of machines like the shiploader shown below.



BCDsteel uses the API to display a Strand7 model window very similar to Strand7, including most of the Strand7 tools used for displaying a model. A separate panel on the left of the display contains steel design tools. This panel changes to define members, select load cases/combinations to analyse, and for displaying results.

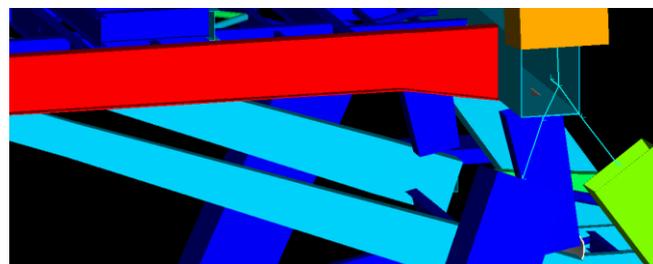
The program is simple to use - it is laid out in a logical order from input through to results. Most controls display a Windows hint when the mouse pointer rests on the control, with F1 displaying a context sensitive Help file for additional information.

BCDsteel analyses steel members that have been modelled using beam elements. However, models like the shiploader above typically also include string groups, links, plates and brick elements, often have multiple freedom cases, and sometimes staged analyses.

In general, each steel design "member" includes a number of beam elements in a row, with different restraints at the nodes along the member. The program will automatically define all members, restraints and effective lengths in a model. This is based on the geometry of the structure taking into account node restraints, beam end release, orientation/alignment, etc. However, it is important for the user to check that the members are correctly defined. Tools are provided to modify members, restraints and/or effective lengths where necessary.

Some features are listed here, some of which are possibly unique to BCDsteel:

- Members may be tapered, and may even have a totally different section type along their length.



Strand7 API Developer Profile

The Strand7 API (Application Programming Interface) offers Strand7 users the ability to interact with Strand7 through a computer program instead of the Strand7 Graphical User Interface (GUI). The computer program can be a compiled program created from a traditional programming language such as C, Fortran, Pascal, etc., or an interpreted program within environments such as Matlab, Python, Excel (with VBA), amongst others.

The Strand7 API can be used for a range of tasks such as automating repetitive operations, building Strand7 models in a parametric way, extracting results in customised formats, performing automated design checks, or for developing complete applications where FEA forms part of the process.

Many Strand7 users have discovered the benefits of the Strand7 API in recent years for their day-to-day analysis tasks, and some users have also developed commercial applications based on the Strand7 API.

One such development is the program marketed as BCDsteel, which offers steel design capabilities of Strand7 models to AS4100. BCDsteel has recently become commercially available. The following article has been written by Paul Cain, the developer of BCDsteel. As BCDsteel is not a product developed by Strand7 Pty Ltd, anyone interested in the product should visit the official product website www.BCDsteel.com.

If you have developed a Strand7 API product and you would like Strand7 users to know about it, please send an email and a short article to news.st7@strand7.com and we will consider publishing it in a future edition of News.St7.

Steel Design to AS 4100

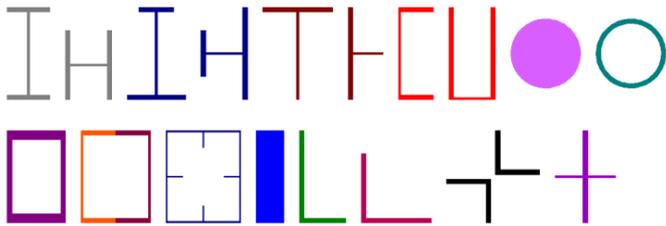
by Paul Cain, developer of BCDsteel.

BCDsteel has been developed independently of Strand7 and is a good example of the professional quality software that can be created using the Strand7 API.

Making finite element analysis easier.

- Section capacities are calculated for slender sections by omitting the ineffective width from each "flat plate element". These are calculated for bending about both the major and minor principal axes plus axial compression.
- Star equal angle sections used for bracing are analysed to Clause 6.4.3.
- Equal angle web members in trusses can be analysed in accordance with Clause 8.4.6. This can result in significantly higher capacity than Clause 8.4.5.

The following figure shows the range of section types presently supported. Fabricated RHS can have up to 3 longitudinal stiffeners per face.



The program uses the Strand7 API to get axial forces, bending moments and shear forces from the Strand7 model. The analyses generally use the highest tier equations in AS 4100, with some exceptions to simplify user input. The program will analyse linear and nonlinear results. The members are coloured to indicate the Load to Capacity Ratio, which must be < 1.00 to comply with AS 4100.

BCDsteel has been verified where possible by comparison with the Design Capacity Tables published by the Australian Steel Institute and with Limsteel. Many aspects such as slender section capacity, the equal angle solvers, flexural torsional buckling of struts, and tapered member capacity have had to be verified against data generated specifically for this purpose.

A solver for the Eurocode EC3 is presently under development. The section capacity calculator has been completed and this also includes Class 4 (slender) sections.

To run BCDsteel, Strand7 users will need to have a copy of the Strand7 API module. Further details including pricing and trial software are available at www.BCDsteel.com.

Strand7 Training

The Strand7 training calendar is regularly updated and published at www.strand7.com, so please check our site periodically. If you are on our mailing list you will also be kept informed by email whenever new courses are scheduled. If you are not on our mailing list but would like to be, please email info@strand7.com.

At the time of publishing this newsletter, the following courses were scheduled to be presented in our office in Sydney:

Date	Course Title
20-23 August 2013	Introducing Strand7
26 August 2013	Automeshing with Strand7
27-28 August 2013	Structural Analysis with Strand7
29 August 2013	Nonlinear Analysis with Strand7
30 August 2013	Dynamic Analysis with Strand7

To register, please go to www.strand7.com and follow the links.

We are also planning to run Strand7 courses in the UK later in the year. If you are interested in attending, please contact us via info@strand7.co.uk

Exhibitions

Between now and the end of 2013, we will be exhibiting Strand7 at the following events:

Date	Exhibition	Location
18-21 Sep 2013	NCSEA Booth 14	Atlanta, GA, USA
25-26 Sep 2013	TCT Live Stand F36	Birmingham, UK
2-3 Oct 2013	Engineering Design Show Stand C75	Coventry, UK
6-8 Nov 2013	SNAME Booth 314	Bellevue, WA, USA

Links to these exhibitions are on our website. If you are able to attend, please come to our booth and say hello - we would welcome the opportunity to meet you.

Website Update

Some updates to the Strand7 website have recently been implemented, coinciding with the Webnotes launch. Each page now has a unique URL, which allows you to bookmark locations within the page, and share specific links.

Additionally, a custom Google search has been installed for quick searching of all Strand7 content.

Head Office

Strand7 Pty Ltd

Suite 1, Level 5
65 York Street
Sydney NSW 2000
Australia

Tel +61 2 9264 2977
Fax +61 2 9264 2066
Email info@strand7.com

UK Office

Strand7 UK Ltd

Suite 3, 1st Floor
The Stables
6 Church Street, St. Neots
Cambridgeshire PE19 2BU
United Kingdom

Tel +44 1480 211 011
Fax +44 1480 211 020
Email info@strand7.co.uk