Elfini Solver Verification

Preface

This book is the Verification Manual of CATIA ELFINI Solver.

The manual provides a series of mechanical problems to test the accuracy of ELFINI finite element solver.

Each problem has an analytical or reference known solution.

Each problem has an associated CATIA CATPart and CATAnalysis.

Using this Guide

This guide is the Verification Guide of CATIA ELFINI Solver.

The Verification Guide is divided in the following distinctive chapters:

- **Static analysis problems**
  Description of several linear static analysis problems and comparison of results.
- **Modal analysis problems**
  Free vibration analysis of several problems and comparison of results.
- **Buckling analysis problems**
  Computation of critical loads creating structure instability and comparison of results.

Where to Find More Information

Prior to reading this book, we recommend that you read:

- Generative Structural Analysis User's Guide
- Infrastructure User's guide Version 5
- Part Design User's Guide
- Assembly Design User's Guide
- Real Time Rendering User's Guide
- Conventions chapter

Conventions

Certain conventions are used in CATIA, ENOVIA & DELMIA documentation to help you recognize and understand important concepts and specifications.

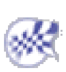
Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted as follows:

This icon...	Identifies...
	estimated time to accomplish a task
	a target of a task
	the prerequisites
	the start of the scenario
	a tip
	a warning
	information
	basic concepts
	methodology
	reference information
	information regarding settings, customization, etc.
	the end of a task
	functionalities that are new or enhanced with this Release.
	allows you to switch back the full-window viewing mode.

Graphic Conventions Indicating the Configuration Required

Graphic conventions indicating the configuration required are denoted as follows:

This icon...	Indicates functions that are...

P1	specific to the P1 configuration
P2	specific to the P2 configuration
P3	specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are denoted as follows:

This icon...	**Gives access to...**
	Site Map
	Split View mode
	What's New?
	Overview
	Getting Started
	Basic Tasks
	User Tasks or the Advanced Tasks
	Workbench Description
	Customizing
	Reference
	Methodology
	Glossary
	Index

Text Conventions

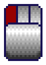
The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File** -> **New** identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

How to Use the Mouse

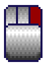
The use of the mouse differs according to the type of action you need to perform.

Use this mouse button...	**Whenever you read...**



- Select (menus, commands, geometry in graphics area, ...)
- Click (icons, dialog box buttons, tabs, selection of a location in the document window, ...)
- Double-click
- Shift-click
- Ctrl-click
- Check (check boxes)
- Drag
- Drag and drop (icons onto objects, objects onto objects)

- Drag
- Move



- Right-click (to select contextual menu)

What's New?

No enhancement in this release.

User Tasks

Static Analysis

This chapter contains static linear analysis problems which illustrate some of the features and capabilities of CATIA-ELFINI.

Static linear analysis consists in finding the deformed shape and the internal strains and stresses of an elastic structure subject to prescribed boundary conditions (displacement and traction types).

This chapter contains the following models and tasks:

Cylindrical Roof Under Its Own Weight

This test lets you check analysis results for a cylindrical roof under its own weight, in the context of a Static Case. You will use 3D shells. This test is also known as the Scordelis-Lo roof.

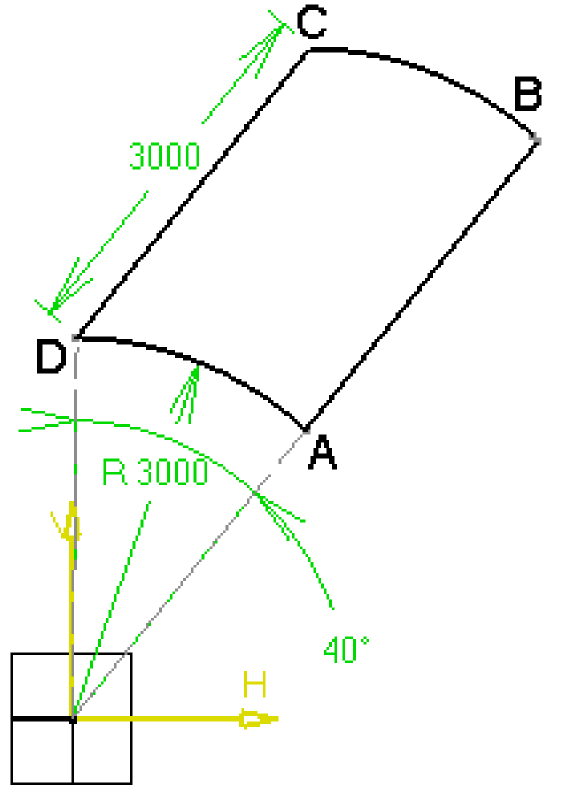
Open Cylindrical_Roof.CATAnalysis and, if needed, Cylindrical_Roof.CATPart from the sample directory.

Before you begin:

1. You defined the number of nodes and elements using **FEM Surface** product, if needed.
2. You defined both geometry and Analysis specifications as shown below.
3. You generated an image called Translational displacement text and computed the CATAnalysis.

Specifications

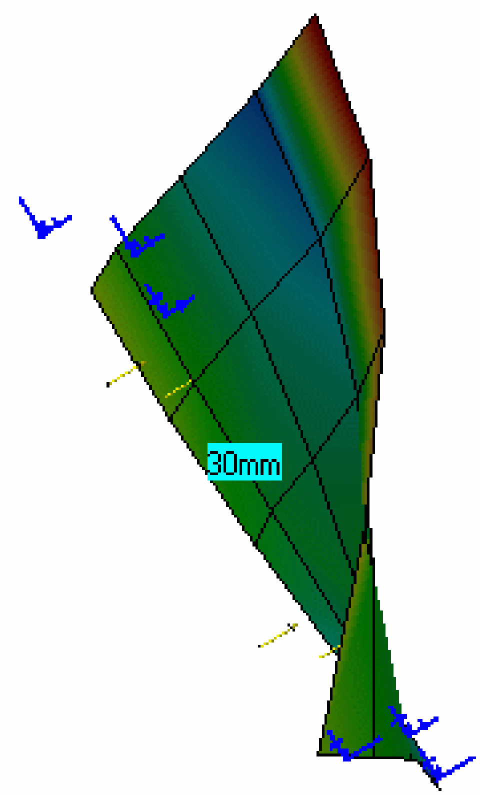
Geometry Specifications	
Length: **L = 3000 mm**	
Radius: **R = 3000 mm**	
Thickness: **th = 30 mm**	

Angle: **Ø = 40 deg**	

Due to symmetry, only a quarter of the roof is modeled (**ABCD**) and the displacement w will be computed.

Analysis Specifications	
Young Modulus (material): **E = 30000 MPa**	
Poisson's Ratio (material): **n = 0**	

Load:
Surface Force Density (weight of the roof): **q = 6250 N_m 2** on y

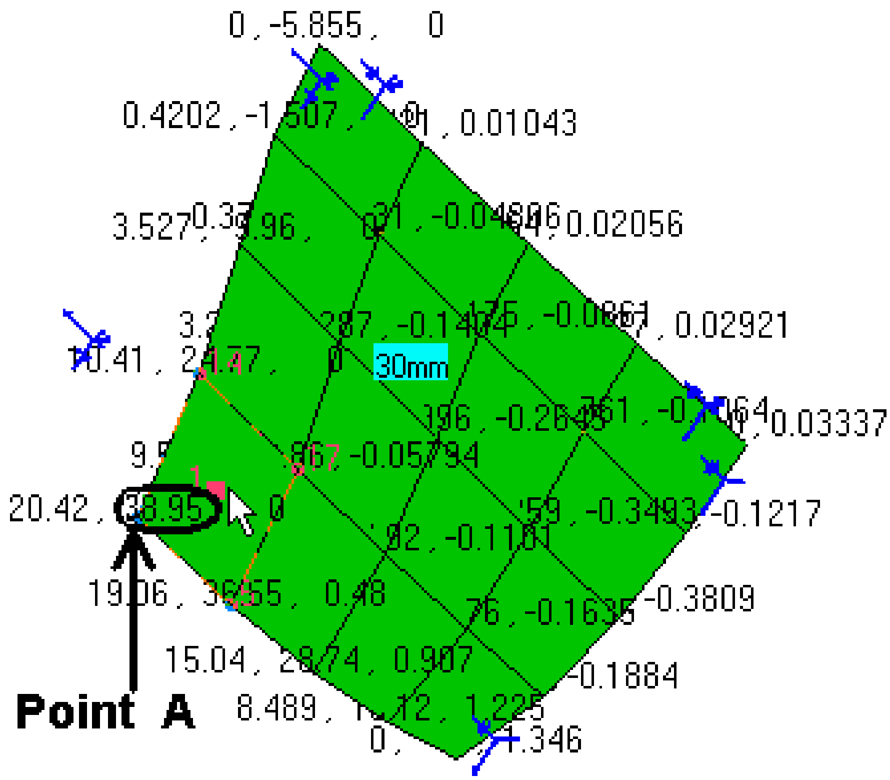


B-C: Translation 1, 2 and Rotation 3 are restrained
C-D: Translation 3, Rotation 1 and 2 are restrained
D-A: Translation 3, Rotation 1 and 2 are restrained

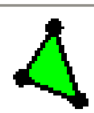
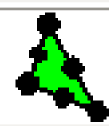
Results

The results correspond to the normalized vertical displacement at **Point A**.

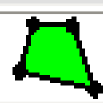
Here we have got an illustration of the 5x5 quadrangle four nodes output results, for example at point A.



1. Triangle Output Results

Number of Nodes	Number of Mesh Elements	Triangle Three Nodes	Triangle Six Nodes	Theoretical Values	
				Shallow Shell	Deep Shell
5x5	32	31.52	33.17	37.03	36.10
9x9	100	35.73	37.5	37.03	36.10
15x15	182	38.16	36.89	37.03	36.10

2. Quadrangle Output Results

Number of Nodes	Number of Mesh Elements	Quadrangle Four Nodes	Theoretical Values	
			Shallow Shell	Deep Shell
5x5	16	39.24	37.03	36.10
9x9	64	37.84	37.03	36.10
15x15	256	36.95	37.03	36.10

Reference:

Computer analysis of Cylindrical Shells. SCORDELIS A.C., LO K.S., Journal of the American Concrete Institute VOL 61 pp 539-561 1969

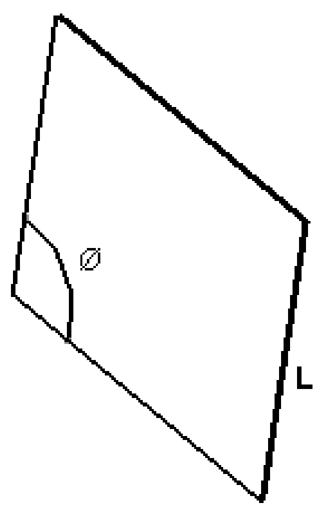
Morley's Problem

The purpose of this test is to evaluate the sensitivity of a plate element to the direction (angular orientation) of the mesh on a plate in bending.
An oblique plate is subjected to a uniform pressure. The quantity of interest is the center point displacement. The analysis case is Static type and the model is 2D Shells type.

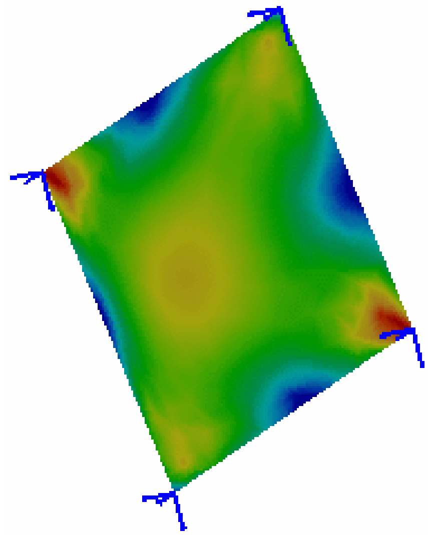
Open Morley02.CATAnalysis (5x5 Nodes) or Morley01.CATAnalysis (11x11 Nodes) from the sample directory, and if needed, Morley01.CATPart.

1. You generated an image called Translational displacement component and computed the CATAnalysis.
2. You defined geometry and Analysis specifications as shown here:

Specifications

Geometry Specifications	
Values	Illustration
Length: **L = 1 mm**	
Thickness: **th = 0.001 mm**	
Angle: **Ø = 80, 60, 40, 30**	

Analysis Specifications

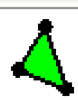
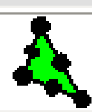
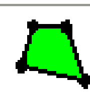
Values	Illustration
Young Modulus (material): **E = 3. 10 7 MPa**	
Poisson's Ratio (material): **n = 0.3**	
Boundary conditions: simply supported on all edges	
Pressure load: 1 MPa on the plate	

Results

The results for various values of the Ø angle corresponding to various mesh directions are given in the following tables.

Normalized vertical displacement at the center are listed: computed vertical displacement divided by the analytical result.

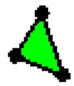
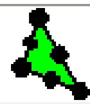
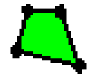
See Morley02.CATAnalysis

1. Output Results (6x6 Nodes)				
Angle	Triangle three nodes 	Triangle six nodes 	Quadrangle four nodes 	Theoretical value
80 deg	1.375 10-3	1.416 10-3	1.1293 10-3	1.409 10-3
60 deg	0.9066 10-3	0.9134 10-3	0.9029 10-3	0.9318 10-3

40 deg	0.3251 10-3	0.3327 10-3	0.2788 10-3	0.3487 10-3
30 deg	0.1402 10-3	0.1482 10-3	0.1273 10-3	0.1485 10-3

See Morley01.CATAnalysis

2. Output Results (11x11 Nodes)

Angle	Triangle three nodes	Triangle six nodes	Quadrangle four nodes	Theoretical value
				
80 deg	1.414 10-3	1.426 10-3	1.398 10-3	1.409 10-3
60 deg	0.9095 10-3	0.9392 10-3	0.9266 10-3	0.9318 10-3
40 deg	0.3425 10-3	0.3492 10-3	0.3449 10-3	0.3487 10-3
30 deg	0.1483 10-3	0.1484 10-3	0.1481 10-3	0.1485 10-3

Reference:

- *MORLEY L.S.D., Skew plates and Structures, Pergamon Oxford 1963*
- *RAZZAQUE A., Program for triangular bending elements with derivative smoothing, IJNME Vol 6, pp 333-343*, 1973

Twisted Beam

This test lets you check analysis results for a twisted beam, in the context of a Static Case. You can use either 3D shells or 3D solids.

Open, from the sample directory:

- Twisted_beam.CATAnalysis (volume geometry)
- Twisted_beam_surface.CATAnalysis (surface geometry)

and if needed,

- Twisted_beam_surface.CATPart
- Twisted_beam_volume.CATPart

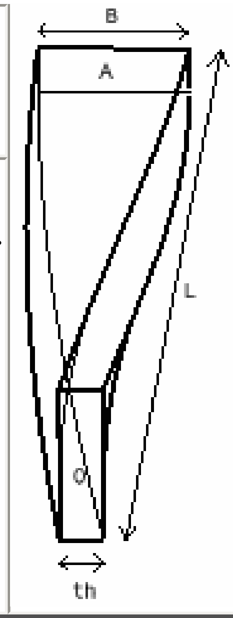
Before you begin:

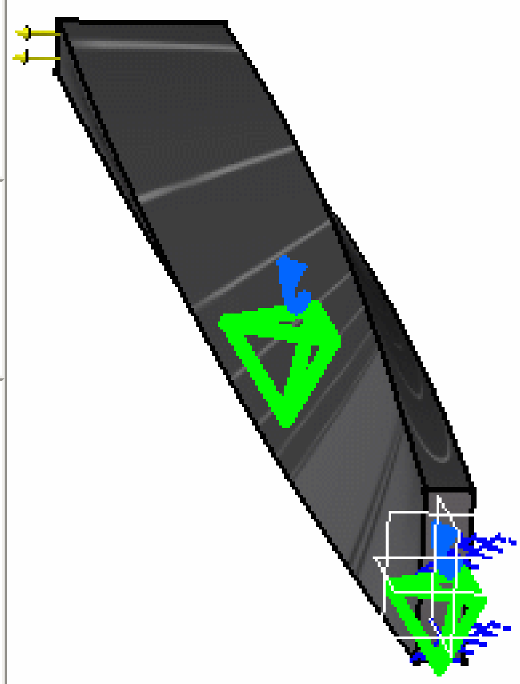
1. You defined the number of nodes and elements using **FEM Surface** product, if needed.
2. You defined both geometry and Analysis specifications as shown below.
3. You generated an image called Translational displacement text and computed the CATAnalysis.

Specifications

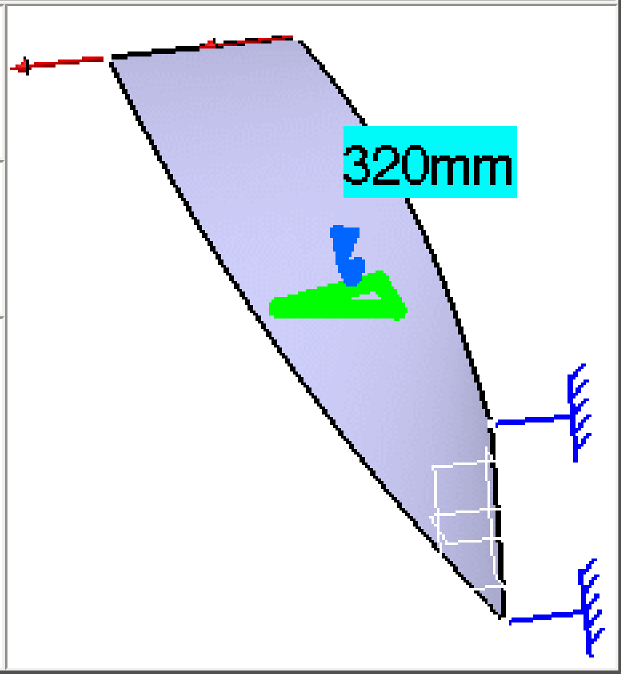
Volume Geometry

Geometry Specifications	
Length: **L = 1 mm**	
Width: **B = 1.1 m**	

Thickness: **th = 0.32 m**	
90 degrees rotation between **O** and **A**.	

Analysis Specifications on Volume Geometry	
Young Modulus (material): **E = 29 MPa**	
Poisson's Ratio (material): **n = 0.22**	
Two loads (Surface force density) are applied on the beam section at **A**. Surface Force vector: **Pz = 1 N OR Py = 1 N**	

Surface Geometry

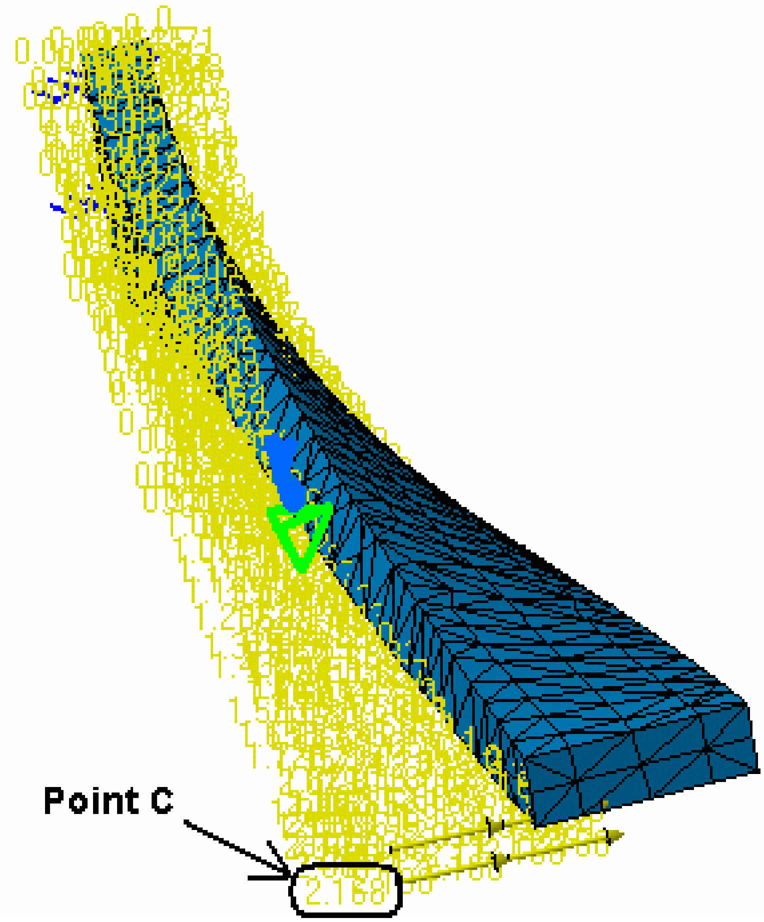
Analysis Specifications on Surface Geometry	
Young Modulus (material): **E = 29 MPa**	
Poisson's Ratio (material): **n = 0.22**	
Two loads (Distributed force) are applied on the beam section at **A**. Surface Force vector: **Pz = 1 N OR Py = 1 N**	

Results

The table below presents the results for meshes consisting of **3x13x2** nodes for 3D elements.

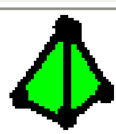
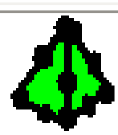
The results correspond to the vertical displacement at **Point C**.

Here we have got an illustration of the 13x3 quadrangle four nodes output results with Py=1, for example at **Point C**.



Volume Geometry

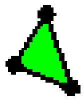
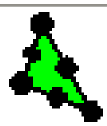
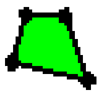
See Twisted_beam.CATAnalysis

Output Results (Mesh size = 320mm)					
Type of Mesh Element	Number of Elements	Force Vector		Theoretical Solutions	
		Pz=1	Py=1	uz	uy
	1749	0.7878	2.168	**1.75 mm**	**5.42 mm**
	1749	1.751	5.42	**1.75 mm**	**5.42 mm**

Surface Geometry

See Twisted_beam_surface.CATAnalysis

Output Results (Thickness = 320mm)

Type of Mesh Element	Number of Elements	Force Vector		Theoretical Solutions	
		Pz=1	Py=1	**uz for Pz=1**	**uy for Py=1**
	58	1.93	6.676	**1.75 mm**	**5.42 mm**
	64	1.798	5.578	**1.75 mm**	**5.42 mm**
	34	1.72	5.42	**1.75 mm**	**5.42 mm**

Reference: *MAC NEAL,R.,HARDER,R.L., A proposed standard set of problems to test finite element accuracy, Finite Element Analysis Design, Vol. 1, P.3-20, 1985.*

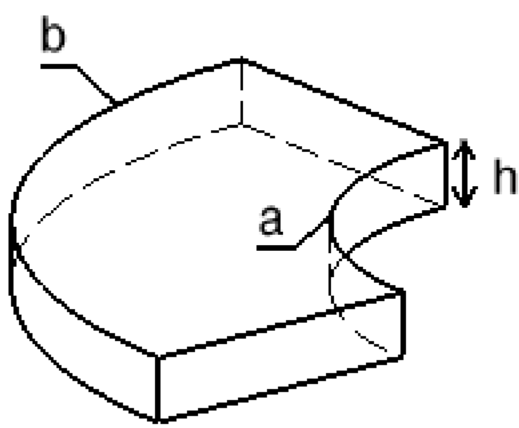
Thick Cylinder Under Internal Pressure

This test lets you check analysis results for a thick cylinder under internal pressure, in the context of a Static Case. You will use a 3D model.

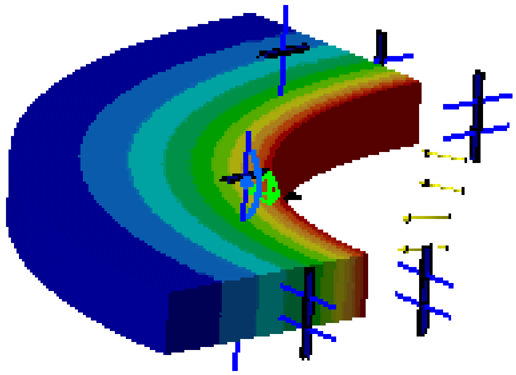
Open Thick_cylinder_02.CATAnalysis from the sample directory.

1. You generated an image called Translational displacement magnitude, Stress principal tensor component (nodal value), with a cylindrical axis system and computed the CATAnalysis.
2. You defined both the geometry and the Analysis specifications as shown here:

Specifications

Geometry Specifications	
Values	Illustration
*inner radius: **a = 0.1 m***	
*outer radius: **b = 0.2 m***	
height: ***H=20mm***	

The structure and the load being cylindrically symmetrical, only one quarter of the cylinder is modeled.

Analysis Specifications	
Values	Illustration
Young Modulus (material): **E = 20. 105 MPa**	
Poisson's Ratio (material): **v = 0.3**	
Pressure (Load: uniform radial pressure): **P=60 MPa**	

The analytical solution of this problem is:

- For both components of the stress tensor within the part cylindrical axis

$$\sigma_{rr} = p \frac{a^2}{(b^2 - a^2)} \left[1 - \frac{b^2}{r^2}\right]$$

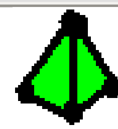
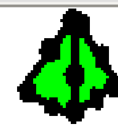
$$\sigma_{\theta\theta} = p \frac{a^2}{(b^2 - a^2)} \left[1 + \frac{b^2}{r^2}\right]$$

- For the component of a displacement using the same axis

$$u_r = \frac{p}{E} \frac{a^2}{b^2 - a^2} r \left[(1 - \nu) + (1 + \nu)\frac{b^2}{r^2}\right]$$

Results

Output Results

Position in Space	Image Type	Reference Value	TE4 Elements 	TE10 Elements 
Inner quarter	Stress principal tensor component (with C3 component filter)	-60 MPa	-56.8 MPa	-60 MPa
	Stress principal tensor component (with C1 component filter)	100 MPa	107 MPa	100 MPa
	Translational displacement vector	59 10-6 m	57.2 10-6 m	57.2 10-6 m
Outside quarter	Stress principal tensor component (with C3 component filter)	0 MPa	-0.045 MPa	0.067 MPa
	Stress principal tensor component (with C1 component filter)	40 MPa	39 MPa	40 MPa
	Translational displacement vector	40 10-6 m	36.2 10-6 m	36.4 10-6 m

Reference:
New developments in the finite element analysis of shells. LINDBERG G.M., OLSON M.D., COWPER G.R. - Q. Bull div. Mech. Eng. and Nat. Aeronautical Establishment, National Research Council of Canada.

Space Structure on Elastic Support

This test lets you check Space Structure on Elastic Support, in the context of a Static Case. You will use 3D beams.

This test proposed by SFM is used to validate the following attribute: discrete elastic coupling.

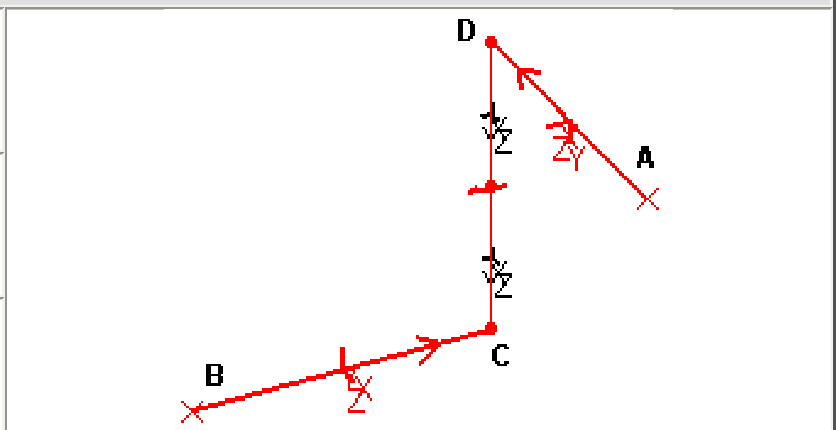
Open SpaceStructure.CATAnalysis and, if needed, SpaceStructure.CATProduct from the sample directory.

The structure drawn used four BEAMV elements with releases and two SPRING elements.

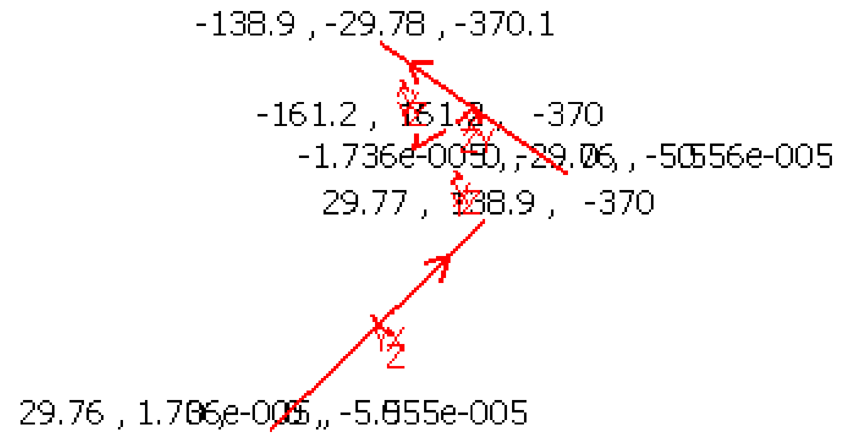
Before you begin:

1. You defined both geometry and Analysis specifications as shown below.
2. You generated two images called Translational displacement text and Point Moment. You then computed the CATAnalysis.

Specifications

Geometry Specifications	
Length: **L = 2000 mm**	
$I_x / 2 = I_y = I_z = 10^{-6}\ m^4$	
$A = 10^{-3}\ m^2$	

You will find here below Analysis specifications:

<table>
<tr><th colspan="2">Analysis Specifications</th></tr>
<tr><td>Young Modulus (material): **E = 210 GPa**</td><td rowspan="3">
</td></tr>
<tr><td>Poisson's Ratio (material): **n =** 0.333333</td></tr>
<tr><td>Load: traction at D **Fz = -10000 N**</td></tr>
<tr><td colspan="2">Boundary conditions:
• displacement type:
○ at A: 4 springs (1 in translation, 3 in rotation, one of which is infinitely rigid)
$\tau_y = 0$, $K_y = 52500$ N/m , $K_{\tau x} = K_{\tau y} = 52500$ N x m/rad
○ at B: 4 springs (1 in translation, 3 in rotation, one of which is infinitely rigid)
$\tau_y = 0$, $K_y = 52500$ N/m , $K_{\tau x} = K_{\tau y} = 52500$ N x m/rad
○ at H: ball joint (articulation with all three rotations reseted)
• traction type:
○ at D: $F_Z = -10000$N</td></tr>
</table>

Results

The results correspond to the quantity at each point and according to X, Y, or Z.

Beam Output Results			
Point	**Quantity**	**Reference**	**Result**
A	Mx (N m)	-8437.5	-8438.14
A	My (N m)	-1562.5	-1562.3
A	Mz (N m)	3125.0	3124.6
B	Mx (N m)	1562.5	1562.5
B	My (N m)	8437.5	8437.05
B	Mz (N m)	3125.0	3125.0
A	*v* (mm)	-29.76	-29.76
A	θx (deg)	9.208	9.209
D	*w* (mm)	370.04	370.07

Reference:
Guide de validation des progiciels de calcul de structures, SSLL 04/89, pp.26-27, AFNOR technique 1990, SFM 10 Avenue Hoche 75008 PARIS

Modal Analysis

This chapter contains modal analysis problems. A modal analysis consists in finding the natural vibrations frequencies and mode shapes of an elastic structure.

This chapter contains the following models and tasks:

Free Vibrations of a Compressor Blade

This test lets you check analysis free vibrations results for a compressor blade in the context of a Modal (or Frequency) Case. You will use 3D shells.

Open Compressor_Blade01.CATAnalysis, and if needed, Compressor_Blade_01.CATPart from the sample directory.

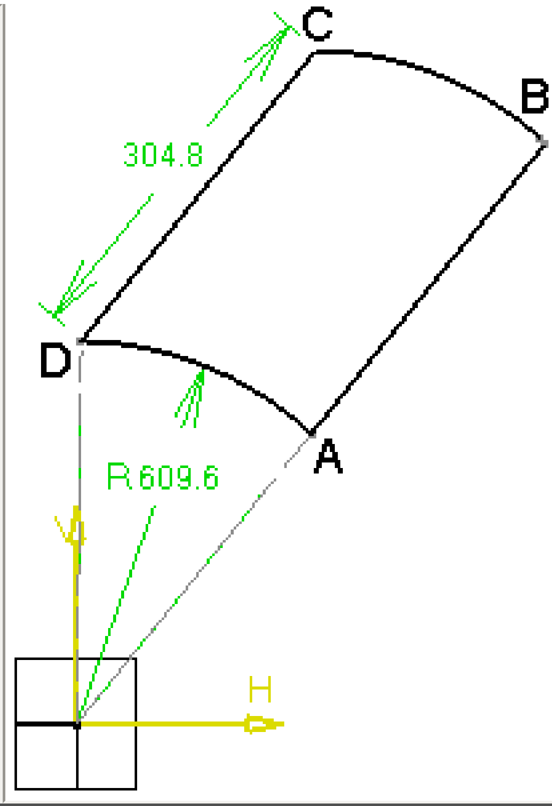
Before you begin:

1. You defined the number of nodes and elements using **FEM Surface** product, if needed.
2. You defined both geometry and Analysis specifications as shown below.
3. You generated an image called Deformed mesh and computed the CATAnalysis.

Specifications

Geometry Specifications	
Length: **L = 304.8 mm**	
Radius: **R = 609.6mm**	
Thickness: **th = 3.048 mm**	

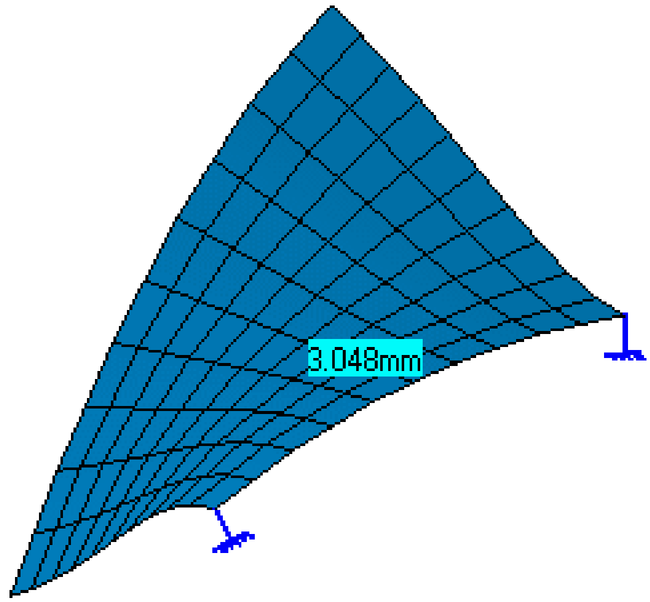
Angle: **A = 28.6479 deg (or 0.5 rad)**



The simplified compressor blade model natural frequencies have been determined experimentally by Olson and Lindberg in 1971.

Analysis Specifications	
Young Modulus (material): **E = 2.069e+01N_m2**	
Poisson's Ratio (material): **n = 0.3**	

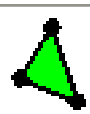
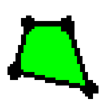
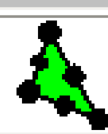
r = 7857.2 Kg/m 3



Restraint:
C-B straight edge is free
A-D straight edge is clamped

Results

The table below presents the results for a mesh consisting of **11x11** nodes. Relative errors are listed:

11x11	Triangle three nodes	Quadrangle four nodes	Triangle six nodes	Reference Values (Hz)
				
Mode 1	85.9397	84.8614	85.4351	85.6

Mode 2	140.157	137.392	138.269	134.5
Mode 3	254.157	250.458	246.831	259.
Mode 4	342.408	340.47	342.597	351.
Mode 5	389.679	382.544	387.101	395.
Mode 6	556.975	737.185	531.215	531.

Reference:

- *OLSON,M.D.,LINDBERG,G.M., Vibration analysis of cantilevered curved plates using a new cylindrical shell finite element, 2nd Conf. Matrix Methods in Structural Mechanics, WPAFB, Ohio 1968*
- *OLSON,M.D.,LINDBERG,G.M., Dynamic analysis of shallow shells with a doubly curved triangular finite element, JSV, Vol. 19, No 3, pp 299-318, 1971*

Free Thin Square Plate

This test lets you check analysis results for a free thin square plate, in the context of a Free Frequency Case. You will use 2D shells.
This test proposed by NAFEMS is used to validate the following attributes:

- Rigid body modes (3 modes)
- Repeated eigen values
- Kinematically incomplete suppressions

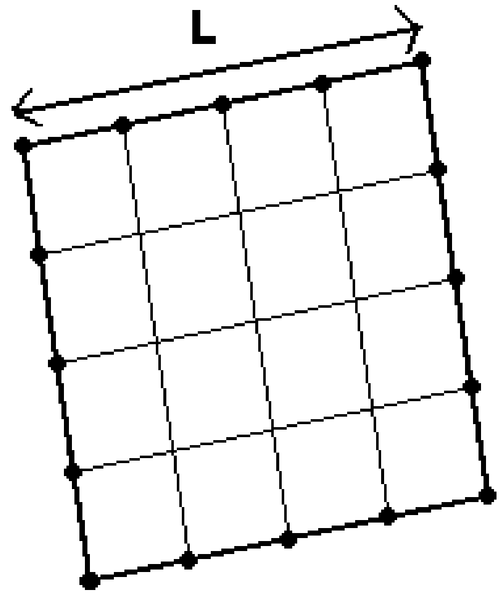
Open Thin_Square_Plate.CATAnalysis, and, if needed, Thin_Square_Plate.CATPart from the sample directory.

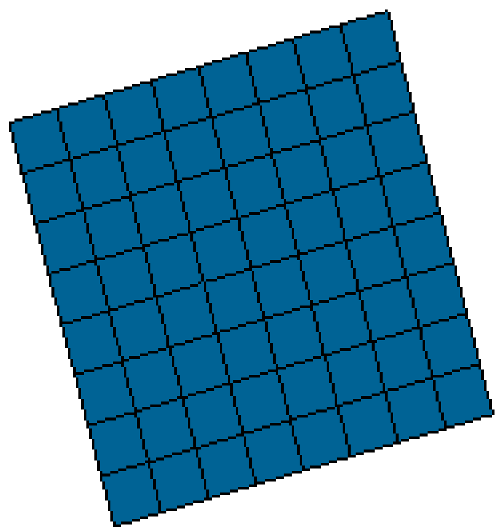
Before you begin:

1. You defined the number of nodes and elements using **FEM Surface** product, if needed.
2. You defined both geometry and Analysis specifications as shown below.
3. You generated an image called Deformed Mesh and computed the CATAnalysis.

Specifications

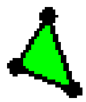
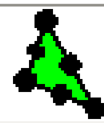
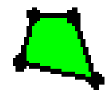
Geometry Specifications	
Length: **L = 10000mm**	
Thickness: **th = 50mm**	

Density: þ = **8000kg/m 3**	

Analysis Specifications	
Young Modulus (material): **200 GPa**	
Poisson's Ratio (material): **n = 0.3**	
The structure is free, hence the boundary conditions are: **u = v = qz = 0**	

Results

The generated meshes are as shown here:

mode	Ref.	Triangle three nodes	Triangle six nodes	Quadrangle four nodes
				
7	1.622	1.617	1.600	1.62432
8	2.360	2.389	2.358	2.38901
9	2.922	2.979	2.918	2.97946
10	4.233	4.253	4.171	4.25237
11	4.233	4.280	4.174	4.25237
12	7.416	7.786	7.338	7.79341

Reference:

BENCHMARK newsletter, April 1989, p.17, NAFEMS - Glasgow

Plane Vibration of Simply Supported Double Cross

This test lets you check the plane vibration of a simply supported double cross in the context of a Modal (or Frequency) Case. You will use 2D beams.This test proposed by NAFEMS is used to validate the following attributes: bending-extension coupling and multiple and close eigenvalues.

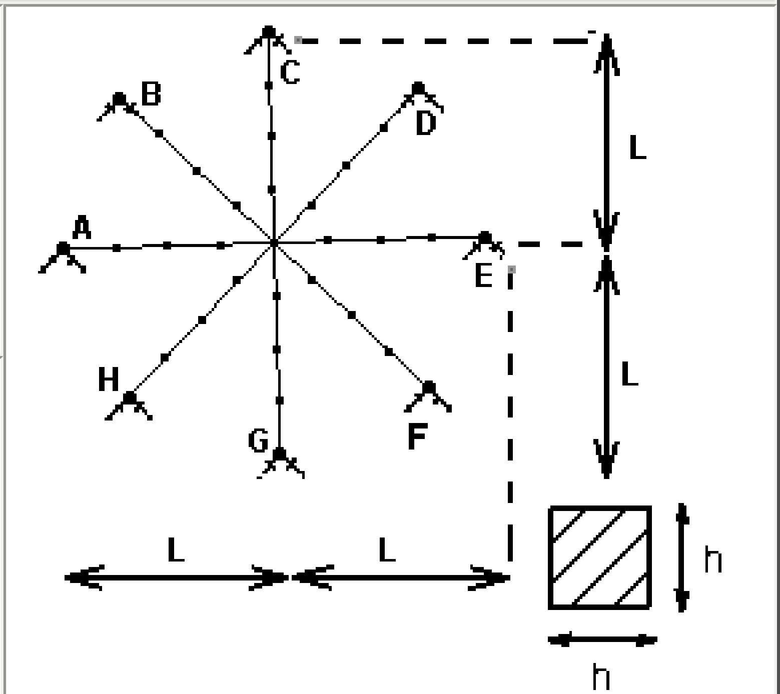
Open DoubleCross.CATAnalysis and if needed, DoubleCross.CATPart.

Before you begin:

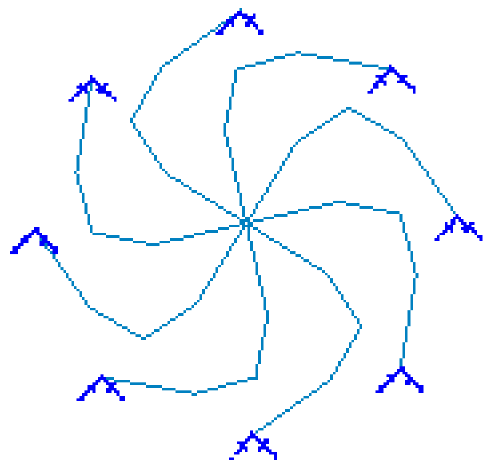
1. You defined both geometry and Analysis specifications as shown below.
2. You generated an image called Deformed mesh and computed the CATAnalysis.

Specifications

The recommended FE model uses four beam elements per arm.

<table>
<tr><th colspan="2">Geometry Specifications</th></tr>
<tr><td>Length: L = 5000 mm</td><td rowspan="2">
</td></tr>
<tr><td>Height: h = 125 mm</td></tr>
</table>

The Analysis specifications are the following:

Analysis Specifications	
Young Modulus (material): **E = 200 GPa**	
Poisson's Ratio (material): **n = 0.3**	
Density: **8000 Kg / m^3**	

The structure is planar (all out-of plane motion is blocked). The eight ends of the double cross are simply supported, hence the boundary conditions are: **u=v=0 at A, B, ..., H**.

The output consists of the 16 lowest natural frequencies of this structure.

Results

The table below presents the results (frequencies in Hz).

Mode	Exact	BEAM	% Error
1	11.336	11.336	0 %
2-3	17.687	17.687	0 %
4-8	17.715	17.715	0 %

9	45.477	45.477	0 %
10-11	57.364	57.364	0 %
12-16	57.683	57.683	0 %

Reference:

BENCHMARK newsletter, April 1989, p.14, NAFEMS - Glasgow

Buckling Analysis

This chapter contains buckling analysis problems. A buckling analysis consists in finding the buckling mode shapes and the buckling critical factors corresponding to a specified load case applied to an elastic structure.

This chapter contains the following model and task:

Buckling of a Straight Beam (Out-of-Plane Buckling)

Buckling of a Straight Beam (Out-of-Plane Buckling)

This test lets you check analysis results for the buckling of a straight beam (Out-of-Plane Buckling). In other words, you will study the quality of distorded membrane and shells elements in buckling analysis. You will use 2D shells.

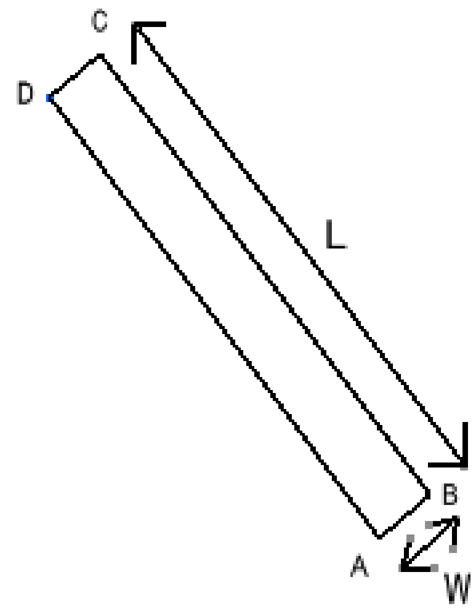
Open Straight_Beam.CATAnalysis, and, if needed, Straight_Beam.CATPart from the sample directory.

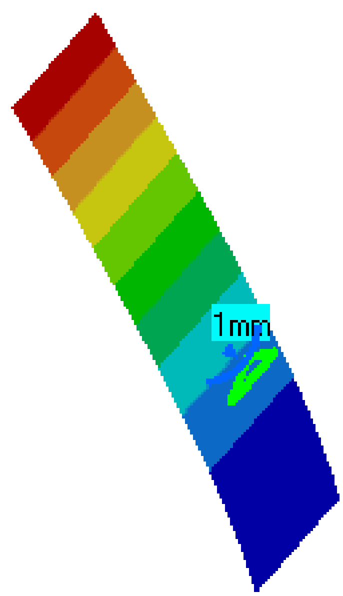
Before you begin:

1. You defined the number of nodes and elements using **FEM Surface** product, if needed.
2. You defined both geometry and Analysis specifications as shown below.
3. You generated an image called Translational displacement text and computed the CATAnalysis.

Specifications

Geometry Specifications	
Length: **L = 100 mm**	
width: **W = 12 mm**	

Thickness: **th = 1 mm**	

Analysis Specifications	
Young Modulus (material): **E = 1e+09N_m**	
Poisson's Ratio (material): **n=0**	
Restraint: A-B straight edge is clamped	
Load: Lineic force **q=-83.333N_m** applied on CD **(along Y)**	
The beam is free to move in the x (transverse) direction.	

Results

The normalized results for regular meshes are presented in the tables below: the computed buckling factor is divided by the analytical buckling factor.

Output Results		
Type of Mesh		**Regular Mesh**
	Triangle three nodes	0.2472
	Triangle six nodes	0.2467
	Quadrangle four nodes	0.2467
Theoretical value		0.2467

The critical load is given by column buckling theory:

$$N_{cr} = \frac{1}{4}\frac{\pi^2 EI}{L^2}$$

In the present case this gives:

Nc = **0.2467** *N*

Reference:

Beam theory

Index