COSMOSWorks Verification Problems

This document contains verification problems to demonstrate the accuracy of the COMSOSWorks software in comparison to analytical results. Problems are included for linear static, frequency, buckling, thermal, and nonlinear static studies.

LINEAR STATIC ANALYSIS...........................................................................................................4
Simply Supported Rectangular Plate................................................................................................4
Deflection of a Cantilever Beam........................................................................................................5
Tip Displacements of a Circular Beam.................................................................................................6
Cylindrical Shell Roof..........................................................................................................................7
Torsion of a Square Box Beam............................................................................................................8
Effect of Transverse Shear on Maximum Deflection........................................................................9
Reactions and Deflections of a Cantilever Beam ..............................................................................10
Bending of a T Section Beam............................................................................................................11
Bending of a Circular Plate with a Center Hole .............................................................................12
Circular Plate Under a Concentrated Load........................................................................................13
Test of a Pinched Cylinder with Diaphragm....................................................................................14
Clamped Square Plate Under Uniform Loading ...........................................................................15
Analysis of an Elliptic Membrane Under Pressure .......................................................................16
Thermal Stress Analysis of a 2D Structure.......................................................................................17
Clamped Beam Subject to Prescribed Displacement....................................................................18
Bending of a Solid Beam................................................................................................................19
Thermal Stress Analysis of a 3D Structure......................................................................................20
Rotating Solid Disk..........................................................................................................................21
Laterally Loaded Tapered Beam.......................................................................................................22
Bending of a Cantilever Beam..........................................................................................................23
Deformation of a Uniformly Loaded Beam.....................................................................................24
Shear Stress in Hollow Cylinder.....................................................................................................25
Deflection of a Cantilever Under Gravity......................................................................................26
Conical Shaped Vessel Under Centrifugal Load...........................................................................27
Tensile Stress in a Steel Bar.............................................................................................................28
Hoop Stress in Thin-Walled Pressure Vessel..................................................................................29
Pin with Rotational Spring...............................................................................................................30
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>FREQUENCY ANALYSIS</td>
<td>31</td>
</tr>
<tr>
<td>Frequencies of a Triangular Wing</td>
<td></td>
</tr>
<tr>
<td>Frequency Analysis of a Simply Supported Plate</td>
<td>32</td>
</tr>
<tr>
<td>Frequency Analysis of a Clamped Circular Plate</td>
<td>33</td>
</tr>
<tr>
<td>Frequencies of a Cylindrical Thin Shell</td>
<td>34</td>
</tr>
<tr>
<td>Vibration of a Clamped Wedge</td>
<td>35</td>
</tr>
<tr>
<td>Simply Supported Rectangular Plate With Inplane Pressure</td>
<td>36</td>
</tr>
<tr>
<td>Frequency Analysis of a Cantilever Beam</td>
<td>37</td>
</tr>
<tr>
<td>Natural Frequencies of a Two-Mass Spring System</td>
<td>38</td>
</tr>
<tr>
<td>Frequency Analysis of a Simply Supported Beam</td>
<td>39</td>
</tr>
<tr>
<td>Frequency of a Cantilever Beam with Lumped Mass</td>
<td>40</td>
</tr>
<tr>
<td>Lateral Vibration of an Axially Loaded Bar</td>
<td>41</td>
</tr>
<tr>
<td>Natural Frequencies of a Long Bar</td>
<td>42</td>
</tr>
<tr>
<td>Natural Frequencies of a Ring</td>
<td>43</td>
</tr>
<tr>
<td>Frequency Analysis of a Building Frame</td>
<td>44</td>
</tr>
<tr>
<td>BUCKLING ANALYSIS</td>
<td>45</td>
</tr>
<tr>
<td>Buckling of a Frame</td>
<td></td>
</tr>
<tr>
<td>Buckling of a Quarter Ring Under a Radial Force</td>
<td>46</td>
</tr>
<tr>
<td>Buckling Analysis a Cantilever Beam</td>
<td>47</td>
</tr>
<tr>
<td>Buckling of a Bar Under End and Distributed Axial Loads</td>
<td>48</td>
</tr>
<tr>
<td>Buckling of Columns with Variable Cross Sections</td>
<td>49</td>
</tr>
<tr>
<td>THERMAL ANALYSIS</td>
<td>50</td>
</tr>
<tr>
<td>Steady State Heat Conduction in a Plate</td>
<td></td>
</tr>
<tr>
<td>Steady State Heat Flow in an Orthotropic Plate</td>
<td>51</td>
</tr>
<tr>
<td>Transient Heat Conduction in a Long Cylinder</td>
<td>52</td>
</tr>
<tr>
<td>Thermal Analysis with Phase Change</td>
<td>53</td>
</tr>
<tr>
<td>Temperature Distribution in a Slab Exchanging Radiation with Ambient Atmosphere</td>
<td>54</td>
</tr>
<tr>
<td>Heat Conduction Due to Heating Cables</td>
<td>55</td>
</tr>
<tr>
<td>Transient Heat Conduction in a Slab of Constant Thickness</td>
<td>56</td>
</tr>
<tr>
<td>Heat Transfer from Cooling Fin</td>
<td>57</td>
</tr>
<tr>
<td>Heat Conduction with Temperature-Dependent Conductivity</td>
<td>58</td>
</tr>
<tr>
<td>Radiation from a Rod</td>
<td>59</td>
</tr>
<tr>
<td>Surface to Surface Radiation</td>
<td>60</td>
</tr>
</tbody>
</table>
NONLINEAR STATIC ANALYSIS

Simply Supported Rectangular Plate Under Normal Pressure

Large Displacement of a Circular Plate

Large Deflection of a Clamped Square Plate

Thermal Stress - Temperature-Dependent Properties

Contact Between Two Cubes

Deflection of a Nonlinear Elastic Cantilever Beam

Large Displacement of a Cylindrical Shell - Force

Buckling and Post Buckling of a Simply Supported Plate

Large Displacement of a Cylindrical Shell - Pressure

Viscoelastic of a Bar Under Constant Axial Load

Uniformly Loaded Elastoplastic Plate

Large Displacement Analysis of a Cantilever Beam

Bearing Capacity for a Strip Footing

Three-Point Bending of a Nitinol Wire
Linear Static Analysis

Simply Supported Rectangular Plate

Description
Calculate the deflection at the center of a simply supported isotropic plate subjected to:
A. Concentrated load \( F = 400 \text{ lbs} \), and
B. Uniform pressure \( P = 1 \text{ psi} \).
Dimensions of the plate are as follows: \( h = 1 \text{ in} \), \( a = b = 40 \text{ in} \).

File Name: <install_dir>\Examples\Verification\Static_1.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness is 1" - Thin formulation.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = \( 3 \times 10^7 \text{ psi} \), Poisson’s ratio = 0.3.
Modeling Hints: Due to double symmetry in geometry and loads, only a quarter of the plate is modeled.

Results
The deflection at the center of the plate is calculated and compared to analytical solution.

<table>
<thead>
<tr>
<th>Case</th>
<th>X (in)</th>
<th>Y (in)</th>
<th>Deflection at the center (UY), in</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Theory</td>
</tr>
<tr>
<td>Study A</td>
<td>20</td>
<td>20</td>
<td>0.00270230</td>
</tr>
<tr>
<td>Study B</td>
<td>20</td>
<td>20</td>
<td>0.00378327</td>
</tr>
</tbody>
</table>

Reference
Deflection of a Cantilever Beam

Description
A cantilever beam is subjected to a concentrated load \( F = 1 \) lb at the free end. Determine the deflections at the free end and the average shear stress. Dimensions of the cantilever are: \( L = 10'' \), \( h = 1'' \), \( t = 0.1'' \).

File Name: <install_dir>\Examples\Verification\Static_2.SLDprt
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.1 in - thin shell formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = \( 3 \times 10^7 \) psi, Poisson's ratio = 0.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deflection at free edge (UY), inch</td>
<td>0.001333</td>
<td>0.0013413</td>
</tr>
<tr>
<td>Average Shear Stress (TauXY), psi</td>
<td>10.0</td>
<td>9.8</td>
</tr>
</tbody>
</table>

NOTE: To generate the above results, plot the displacement in the Y direction (UY) and the shear stress (TauXY) then use the List Selected tool to list the plotted result on the free edge.
Tip Displacements of a Circular Beam

Description
A circular beam fixed at one end and free at the other end is subjected to a 200 lb force as shown in the figure. Determine the deflections in the X, Y direction. Radius of curvature of the beam = 10". The beam width and thickness are 4" and 1" respectively.

File Name: <install_dir>\Examples\Verification\Static_3.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1 in - Thin formulation.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.

Results
The UX and UY displacements at the free edge are calculated and compared to analytical values:

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>X Deflection at free edge (UX), inch</td>
<td>0.712E-2</td>
<td>0.713E-2</td>
</tr>
<tr>
<td>Y Deflection at free edge (UY), inch</td>
<td>0.714E-2</td>
<td>0.100E-1</td>
</tr>
</tbody>
</table>

Reference
Cylindrical Shell Roof

Description
Determine the vertical deflections across the midspan of a shell roof under its own weight. Dimensions and boundary conditions are shown in the figure below. The radius of curvature of the shell is \( R = 25 \text{ ft} \).

![Diagram of Cylindrical Shell Roof]

File Name: `<install_dir>\Examples\Verification\Static_4.SLDPRT`

Study Type: Static.

Mesh Type: Shell mesh using surfaces.

Shell Parameters: Shell thickness = 0.25 ft - Thin formulation.

Meshing Options: High, Standard, 4 Points, and Smooth surface.

Meshing Parameters: Use the default Global Size.

Material Properties: Modulus of elasticity = \( 3 \times 10^6 \text{ psi} \), Poisson's ratio = 0, Density = 0.2083 lbs/in\(^3\).

Modeling Hints: Due to symmetry, a quarter of the shell is considered for modeling. The shell weight is applied as a gravity loading.

Results
The vertical displacement (UX) at midspan of the free edge is compared to analytical value:

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>-0.3024</td>
<td>-0.3018</td>
</tr>
</tbody>
</table>

Reference
Pawsley, S. F., "The Analysis of Moderately Thick to Thin Shells by the Finite Element Method," Report No. USCEM 70-12, Dept. of Civil Engineering, University of California, I970.
# Torsion of a Square Box Beam

**Description**
Find the shear stress and the angle of twist for the square box beam shown in the figure. The free end is subjected to a 300 lb-in torque. The beam has a length of 1500". The beam cross section is a square with a side length of 150" and a thickness of 3".

![Square Box Beam Diagram]

**File Name:** `<install_dir>\Examples\Verification\Static_5.SLPRT`
**Study Type:** Static.
**Mesh Type:** Shell mesh using surfaces.
**Shell Parameters:** Shell thickness = 3 in - Thin formulation.
**Meshing Options:** High, Standard, 4 Points, and Smooth surface.
**Meshing Parameters:** Use a Global Size of 75 in.
**Material Properties:** Modulus of elasticity = 7.5 psi, Poisson’s ratio = 0.3.

**Results**

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear stress (τ), psi</td>
<td>0.0021365</td>
<td>0.00213645</td>
</tr>
<tr>
<td>Rotation (θ), radians</td>
<td>0.01541</td>
<td>0.01553</td>
</tr>
</tbody>
</table>

**NOTE:** The angle of rotation θ is calculated as: \( \theta = \sin^{-1}(\text{resultant displacement of a vertex on the free cross section/distance from that vertex to the center of the cross section}) \)

**Reference**
Effect of Transverse Shear on Maximum Deflection

Description
Find the effect of transverse shear on maximum deflection of an isotropic simply supported square plate of side \(a = 24"\) subjected to a constant pressure \(q = 30\) psi. The plate thickness \(H\) varies according to the table shown below (in terms of the ratio \(H/a\)).

File Name: `<install_dir>`\Examples\Verification\Static_6.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: The input shell thickness is 0.001" - Thin/Thick formulations.
Meshing Parameters: Use a Global Size of 1 in.
Material Properties: Modulus of elasticity = \(3 \times 10^7\) psi, Poisson's ratio = 0.3.
Modeling Hints: Due to symmetry, only one quarter of the plate is considered. Thick shell formulation is used. A design scenario with 3 thickness values is defined to automate the processing. To run the design scenarios, right-click the study icon and select Run Design Scenario. To list the maximum resultant displacements, right-click the Design Scenario Results folder and select Show Summary.

Results

<table>
<thead>
<tr>
<th>Thickness (inches)</th>
<th>Thickness ratio (H/a)</th>
<th>Maximum Deflection*</th>
<th>(\beta^{**}) Reissner Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2</td>
<td>0.05</td>
<td>0.008989</td>
<td>0.044936</td>
<td>0.046817</td>
</tr>
<tr>
<td>2.4</td>
<td>0.1</td>
<td>0.0012096</td>
<td>0.046659</td>
<td>0.050399</td>
</tr>
<tr>
<td>3.6</td>
<td>0.15</td>
<td>0.00039104</td>
<td>0.049533</td>
<td>0.054989</td>
</tr>
</tbody>
</table>

* The resultant displacement (not UZ) is used for the maximum deflection.

** The Reissner coefficient \(\beta\) is given by: \(\beta = E \frac{H^3}{q a^4}\), where \(W_{max}\) is the maximum deflection.

Reference
Reactions and Deflections of a Cantilever Beam

Description
Calculate reactions and deflections of a cantilever beam subjected to an 8 lb force acting on the free end of the cantilever as shown in the figure below. The cantilever dimensions are: L = 10" and W = 4".

File Name: Click <install_dir>\Examples\Verification\Static_7.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1 in - Thin formulation.
Meshing Parameters: Use a Global Size of 1 in.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson’s ratio = 0.3.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum deflection at tip, in</td>
<td>2.667e-4</td>
<td>2.570e-4</td>
</tr>
<tr>
<td>Total reaction force, lb</td>
<td>8</td>
<td>8</td>
</tr>
</tbody>
</table>
Bending of a T Section Beam

Description
Calculate the deflections of a cantilever T beam of length L = 2000" subjected to a force of 100 lbs acting on its free end.

File Name: <install_dir>\Examples\Verification\Static_8.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using mid-surfaces.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 25 in.
Material Properties: Modulus of elasticity = 1 X 10^{11} psi, Poisson's ratio = 0.3.

Results
The following analytical solutions are used to calculate displacements and rotations:

\[ \delta \text{ (displacement)} = \frac{PL^3}{3EI}, \quad \phi \text{ (rotation)} = \frac{PL^2}{2EI}. \]

Where P is the value of the applied force, E is the cantilever's modulus of elasticity, and I is the moment of inertia of the beam cross section.

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free end</td>
<td>Y-displacement (in)</td>
<td>-5.546e-6</td>
</tr>
<tr>
<td></td>
<td>X-Rotation (rad)</td>
<td>4.159e-9</td>
</tr>
</tbody>
</table>
Bending of a Circular Plate with a Center Hole

Description
A circular plate with a center hole is fixed along the inner edge. The outer edge of the plate is subjected to bending by a moment $M = 10$ in-lb/in. Determine the maximum deflection and the maximum slope of the plate. The plate thickness is 0.25" and the outer and inner radii of the plate are 30" and 10" respectively. Due to symmetry of the problem, a 10° wedge is modeled. The applied moment is equivalent to applying a moment of 52.359 lb-in per 10° segment.

File Name: <install_dir>\Examples\Verification\Static_9.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.25 in - Thin formulation.
Meshing Parameters: Use a Global Size of 1 in.
Material Properties: Modulus of elasticity = $3 \times 10^7$ psi, Poisson's ratio = 0.3.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum deflection (UZ), inch</td>
<td>0.04906</td>
<td>0.04787</td>
</tr>
<tr>
<td>Maximum rotation, rad</td>
<td>0.0045089</td>
<td>0.0045239</td>
</tr>
</tbody>
</table>

Reference
Circular Plate Under a Concentrated Load

Description
A circular thick plate of radius 5" is subjected to a load of 4 lb at its center. The plate is clamped at its boundary. Determine the transverse displacement along the radius r.

File Name: <install_dir>\Examples\Verification\Static_10.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 2 in - Thin formulation.
Meshing Parameters: Use a Global Size of 1 in.
Material Properties: Modulus of elasticity = 1.09 X 10^6 psi, Poisson's ratio = 0.3.
Modeling Hints: Due to axial symmetry of the model, only a quarter of the plate is modeled.

Results

<table>
<thead>
<tr>
<th>Distance from the plate center, r (in)</th>
<th>Analytical</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>---</td>
<td>5.48E-06</td>
</tr>
<tr>
<td>1.0</td>
<td>3.53748E-06</td>
<td>3.60E-06</td>
</tr>
<tr>
<td>2.0</td>
<td>2.19719E-06</td>
<td>2.22E-06</td>
</tr>
<tr>
<td>3.0</td>
<td>1.14364E-06</td>
<td>1.16E-06</td>
</tr>
<tr>
<td>4.0</td>
<td>3.88628E-07</td>
<td>3.90E-07</td>
</tr>
<tr>
<td>5.0</td>
<td>0.00E+00</td>
<td>0.00E+00</td>
</tr>
</tbody>
</table>

The following formula was used to generate the analytical results:

\[ U(r) = \frac{PR^2 \left[ 1-(r/R)^2 \right]^2 \ln(R/r) - 2(r/R)^2 \ln(r/R)/(KGtR^2) \right] / 16\pi D \]

where:
- \( U(r) \) = the displacement at distance \( r \) from the plate center
- \( R \) = the radius of the plate
- \( P \) = the force value
- \( D = EX t^2/12(1-NUXY) \), where \( EX \) and \( NUXY \) are the modulus of elasticity and the Poisson's ratio of the plate and \( t \) is the thickness of the plate
- \( G = EX /2(1+NUXY) \)
- \( K = 0.8333 \) (shear correction factor)

Reference
Test of a Pinched Cylinder with Diaphragm

Description
A cylindrical shell of thickness 3" is covered at both ends with rigid diaphragms to allow displacement only along its axial direction. At the cylinder mid-span, a load of 1 lb is applied as shown in the figure below. Determine the radial deflection at the point where the load is applied. The radius and length of the cylinder are 300" and 600" respectively.

File Name: <install_dir>\Examples\Verification\Static_11.SLDPR

Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 3" - Thin formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 25 in.
Material Properties: Modulus of elasticity = 3 X 10^6 psi, Poisson's ratio = 0.3.
Modeling Hints: Due to symmetry, only one-eighth of the cylinder is modeled. To simulate the rigid diaphragm, translations in the radial and circumferential directions and rotations about the axial direction are all set to zero.

Results

<table>
<thead>
<tr>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Radial deflection (UX), in</td>
<td>1.8248e-5</td>
</tr>
</tbody>
</table>

Reference
Clamped Square Plate Under Uniform Loading

**Description**
Determine the deflection at the center of a square plate of side 2" and thickness (t). The plate is clamped at its boundaries and subjected to uniform pressure (q). Various span-to-thickness ratios are investigated.

![Diagram of clamped square plate](image)

**File Name:** `<install_dir>\Examples\Verification\Static_12.SLDPR1
**Study Type:** Static.
**Mesh Type:** Shell mesh using surfaces.
**Shell Parameters:** Thin shell formulation.
**Meshing Options:** High, Standard, 4 Points, and Smooth surface.
**Meshing Parameters:** Use a Global Size of 0.25 in.
**Material Properties:** Modulus of elasticity = 1 X 10^7 psi, Poisson's ratio = 0.3.
**Modeling Hints:** Due to symmetry, only one quarter of the plate is modeled. A design scenario was defined to model the cases under investigation. To run the analysis, right-click the study icon and select Run Design Scenario.

### Results

<table>
<thead>
<tr>
<th>Span/Thickness Ratio</th>
<th>Deflection (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Theory</td>
</tr>
<tr>
<td>10 (q = 1.0 psi)</td>
<td>-2.7518E-6</td>
</tr>
<tr>
<td>100 (q = 1.0 psi)</td>
<td>-2.7518E-3</td>
</tr>
<tr>
<td>1000 (q = 0.01 psi)</td>
<td>-2.7518E-2</td>
</tr>
</tbody>
</table>

**Reference**
Analysis of an Elliptic Membrane Under Pressure

Description
Calculate the stresses at point (D) of an elliptic membrane under a uniform outward pressure of magnitude 10 MPa.

File Name: <install_dir>\Examples\Verification\Static_13.SLDPRT

Study Type: Static.

Mesh Type: Shell mesh using surfaces.

Shell Parameters: Shell thickness = 0.1 m - Thin shell formulation.

Meshing Options: High, Standard, 4 Points, and Smooth surface.

Meshing Parameters: Use a Global Size of 0.14 m.

Material Properties: Modulus of elasticity = 210 X 10^3 MPa, Poisson's ratio = 0.3.

Modeling Hints: Due to symmetry, only a quarter of the plate is modeled.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>σ, at point D (MPa)</td>
<td>92.70</td>
<td>92.75</td>
</tr>
</tbody>
</table>

Reference
Thermal Stress Analysis of a 2D Structure

Description
A rectangular plate of dimensions 1"X2"X0.1" is subjected to a uniform temperature rise of 100°F. The plate is restrained as shown in the figure. Determine the maximum Y displacement and the normal stress in the X direction.

File Name: <install_dir>\Examples\Verification\Static_14.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.1 in - Thin shell formulation.
Meshing Parameters: Use a Global Size of 1 in.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.25, Thermal expansion coefficient = 6.5 x 10^-6/°F.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Y - displacement (in)</td>
<td>0.001083</td>
<td>0.001081</td>
</tr>
<tr>
<td>SX - stress (psi)</td>
<td>26000</td>
<td>25935</td>
</tr>
</tbody>
</table>
Clamped Beam Subject to Prescribed Displacement

Description
Determine the end forces of a clamped beam due to a 1 inch settlement at the right end. The length of the beam is 80" and the beam has a square cross section of dimensions 2"X2".

File Name: <install_dir>\Examples\Verification\Static_15.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 2 in - Thick shell formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3 X 10^7 psi.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>End stress (psi)</td>
<td>28125</td>
<td>28357</td>
</tr>
<tr>
<td>End reaction (lb)</td>
<td>937.50</td>
<td>959.48</td>
</tr>
</tbody>
</table>

Reference
Bending of a Solid Beam

Description
A 10" long cantilever beam has a rectangular cross section of 1" width and 2" height. Find the deflection of the free end under the effect of the following loads:
- an end moment of 2000 in-lb, and
- a shear force of 300 lbs.

File Name: <install_dir>\Examples\Verification\Static_16.SLDRT
Study Type: Static.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 2 in.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.

Results

<table>
<thead>
<tr>
<th>Y-displacement at the free end (UY), in</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>End moment (Moment Study)</td>
<td>-0.005</td>
<td>-0.005007</td>
</tr>
<tr>
<td>Shear force (Force Study)</td>
<td>0.005</td>
<td>0.005089</td>
</tr>
</tbody>
</table>

Reference
Thermal Stress Analysis of a 3D Structure

Description
Determine the maximum displacements of a solid rectangular block of dimensions 1" X 1" X 2" due to a uniform temperature rise of 100 °F. Three orthogonal faces of the block are restrained in their normal directions.

File Name: `<install_dir>\Examples\Verification\Static_17.SLDPRT
Study Type: Static.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 1 in.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.25, Thermal expansion coefficient = 6.5 x 10^-6/°F.

Results

<table>
<thead>
<tr>
<th>Z-translation (UZ), in</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>At Mid plane</td>
<td>6.5e-4 (1<em>100</em>6.5e-6)</td>
<td>6.5e-4</td>
</tr>
<tr>
<td>At Free End</td>
<td>1.3e-3 (2<em>100</em>6.5e-6)</td>
<td>1.3e-3</td>
</tr>
</tbody>
</table>
# Rotating Solid Disk

## Description

A solid disk of radius 9" and thickness 1" rotates about its center with angular velocity $\omega = 25$ rad/sec. Determine the stress distribution in the disk.

![Diagram of a rotating solid disk](image)

## File Name:

`:\<install_dir>\Examples\Verification\Static_18.SLDPRT`

## Study Type:

Static.

## Mesh Type:

Solid mesh.

## Meshing Options:

High, Standard, 4 Points, and Smooth surface.

## Meshing Parameters:

Use a Global Size of 0.25 in.

## Material Properties:

Modulus of elasticity $= 3 \times 10^7$ psi, Poisson's ratio $= 0.3$, Density $= 7.7244$ lb/in$^3$.

## Modeling Hints:

Due to symmetry, only a 45° wedge is analyzed.

## Results

<table>
<thead>
<tr>
<th>Location ($r = 0.5&quot;$)</th>
<th>Radial stress ($SX$), psi</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>416.37</td>
<td>404.3</td>
</tr>
<tr>
<td></td>
<td>Tangential stress ($SY$), psi</td>
<td>416.91</td>
<td>415.2</td>
</tr>
<tr>
<td>Location ($r = 8.5&quot;$)</td>
<td>Radial stress ($SX$), psi</td>
<td>45.12</td>
<td>44.76</td>
</tr>
<tr>
<td></td>
<td>Tangential stress ($SY$), psi</td>
<td>203.16</td>
<td>202.7</td>
</tr>
</tbody>
</table>

**Tip:** To obtain the above results, define the SX and SY plots using Axis1 as a reference then use the Probe tool to probe these plots at the desired locations ($X = 0.5"$ and $X = 8.5"$).

## Reference

Laterally Loaded Tapered Beam

Description
A cantilever beam of width 2" and length 50" has a depth which tapers uniformly from 3" at the tip to 9" at the wall. The cantilever beam is loaded by a 4000 lb force at the tip. Find the maximum bending stress at the mid-span of the cantilever.

File Name: <install_dir> \Examples\Verification\Static_19.SLDPR.T).
Study Type: Static.
Mesh Type: Solid mesh/shell mesh using mid-surface.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.

Results

<table>
<thead>
<tr>
<th>Bending stress at midspan (SX), psi</th>
<th>Solid mesh</th>
<th>Shell mesh using mid-surfaces</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>8333</td>
<td>8333</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>8373</td>
<td>8380</td>
</tr>
</tbody>
</table>

Reference


Bending of a Cantilever Beam

Description
Calculate the maximum deflection and maximum rotation ($\theta$) of a cantilever beam loaded by a shear force of magnitude 1 lb acting on the free end of the cantilever. The length of the cantilever is 10" and the dimensions of its cross section are 1"X1". The cantilever beam is modeled as two identical cantilevers connected at their common interface with a Bonded contact condition.

File Name: <install_dir>\Examples\Verification\Static_20.SLDASM
Study Type: Static.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 1 X 10^6 psi, Poisson's ratio = 0.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement at the free end (UX), in</td>
<td>-0.004</td>
<td>-0.004</td>
</tr>
<tr>
<td>Rotation at the free end, rad</td>
<td>-0.0006</td>
<td>-0.0006</td>
</tr>
</tbody>
</table>

The following relation was used to calculate the end rotations: $\theta = 3 \frac{\delta}{(2L)}$, where $\delta$ is the deflection at the free end.
Deformation of a Uniformly Loaded Beam

Description
Determine the maximum displacement in Y-direction of a uniformly loaded beam with a fixed support at one end and a simple support at the other end. The length of the beam is 20" and the beam section is a square of side 1".

File Name: <install_dir>\Examples\Verification\Static_22.SLDPRT
Study Type: Static.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.28.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Displacement in the Y-direction (UY), inch</td>
<td>1.729e-3</td>
<td>1.755e-3</td>
</tr>
</tbody>
</table>

Reference
Shear Stress in Hollow Cylinder

Description
Determine the shear stress in a hollow concentric cylinder fixed at one end and subjected to a torque of 10 lb-in at the other end. The inner and outer radii of the cylinder are 1" and 2" respectively and the length of the cylinder is 12".

File Name: <install_dir>\Examples\Verification\Static_23.SLDPRT
Study Type: Static.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Alloy Steel.

<table>
<thead>
<tr>
<th>Results</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Shear Stress (TYZ), psi</td>
<td>2.353</td>
<td>2.377</td>
</tr>
</tbody>
</table>

Reference
Deflection of a Cantilever Under Gravity

**Description**
Determine the maximum displacement in Y-direction of a cantilever, fixed at one end, under its own weight. The length of the cantilever is 20" and its section is a square of side 1".

![Cantilever Diagram]

File Name: `<install_dir>`\Examples\Verification\Static_24.SLDPRT  
**Study Type:** Static.  
**Mesh Type:** Solid mesh.  
**Meshing Options:** High, Standard, 4 Points, and Smooth surface.  
**Meshing Parameters:** Use the default Global Size.  
**Material Properties:** Modulus of elasticity = $3 \times 10^7$ psi, Poisson’s ratio = 0.28, Density = 0.2782 lb/in$^3$.

**Results**

<table>
<thead>
<tr>
<th>Maximum Displacement in the Y-direction (UY), inch</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>2.2197e-3</td>
<td>2.221e-3</td>
</tr>
</tbody>
</table>

**Reference**
Conical Shaped Vessel Under Centrifugal Load

Description
Determine the hoop stress in a thin walled conical shaped vessel subjected to centrifugal load due to angular velocity of 5 rad/sec.

File Name: <install_dir>\Examples\Verification\Static_25.SLDPRT
Study Type: Static.
Mesh Type: Shell mesh.
Shell Parameters: Shell thickness = 0.001" - Thin formulation.
Meshing Parameters: Use the default Global Size.
Material Properties: Alloy Steel.
Modeling Hints: Due to symmetry, a quarter of the vessel is modeled.

Results

<table>
<thead>
<tr>
<th>Hoop Stress (SY), psi</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>6.979e-4</td>
<td>7.121e-4</td>
</tr>
</tbody>
</table>

Reference
Tensile Stress in a Steel Bar

Description
Determine the Maximum tensile stress in a steel bar (shaped like a truncated cone with radii 1.5” and 0.5” and a height of 24”) rigidly held at both ends and subjected to a temperature drop of $\Delta T = 50 \text{ F}$.

File Name: `<install_dir>\Examples\Verification\Static_26.SLDPRT`
Study Type: Static.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.3, Thermal expansion coefficient = 6.5 X 10^-6/F.
Modeling Hints: Use the FFEPlus or the Direct sparse solver with the Use soft springs to stabilize model flag on.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Tensile Stress (SY), psi</td>
<td>2.9265e4</td>
<td>3.0719e4</td>
</tr>
</tbody>
</table>

Reference
Hoop Stress in Thin-Walled Pressure Vessel

Description
Determine the hoop stress in a thin-walled pressure vessel under uniform radial pressure of magnitude 100 psi. The radius and thickness of the pressure vessel are 1" and 0.01" respectively.

File Name: <install_dir>\Examples\Verification\Static_28.SLDPR
Study Type: Static.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.01 in - Thin formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Alloy Steel.
Modeling Hints: Due to symmetry, only a quarter of the model is considered.

Results

<table>
<thead>
<tr>
<th>Hoop Stress (SY), psi</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>10005</td>
<td>9959</td>
</tr>
</tbody>
</table>

Reference
Pin with Rotational Spring

Description
The joint shown in the figure provides a 1000 lb.in/radian resistance to relative rotation. The rotation of the moving part is verified against the theoretical value.

File Name: <install_dir>\ Examples\ Verification\ Static_28.sldasm
Study Type: Static.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Default with global contact set to Touching Faces: Free.
Modeling Hints: Define a pin between the two cylindrical holes with a specified rotational stiffness of K = 1,000 lb.in/radian.

Results
- View the UY displacement with reference to the axis of the cylindrical faces. The UY value shown in the table corresponds to the average value of UY on the cylindrical face of the moving arm.
- UY (theoretical) = (Moment/K)* Radius = (5*2/1000)*0.5= 0.005 in

<table>
<thead>
<tr>
<th>UY Tangential Displacement (in)</th>
<th>COSMOSWorks</th>
<th>Theory</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.0027e-3</td>
<td>5.0000e-3</td>
<td>5.0000e-3</td>
</tr>
</tbody>
</table>
Frequency Analysis

Frequencies of a Triangular Wing

Description
Calculate the natural frequencies of a right-angle isosceles triangle wing. The equal sides of the triangle are 6" in long and the thickness is 0.034". One of the equal sides of the triangle is fixed.

File Name: <install_dir> \Examples\Verification\Frequency_1.SLDPRT
Study Type: Frequency.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.034 in - Thin formulation.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 6.5 x 10^6 psi, Poisson's ratio = 0.3541, Density = 0.06411 lb/in^3.

Results

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Draft mesh</td>
</tr>
<tr>
<td>1</td>
<td>55.9</td>
<td>55.910</td>
</tr>
<tr>
<td>2</td>
<td>210.9</td>
<td>211.20</td>
</tr>
<tr>
<td>3</td>
<td>293.5</td>
<td>292.81</td>
</tr>
</tbody>
</table>

Reference
Frequency Analysis of a Simply Supported Plate

Description
Calculate the first natural frequency of a simply supported rectangular plate of dimensions 40"X40"X1".

File Name: <install_dir>\Examples\Verification\Frequency_2.SLDPRT
Study Type: Frequency.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1 in - Thin formulation.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.3, Density = 115.866 lb/in^3.
Modeling Hints: The first mode is symmetrical and a quarter of the model could have been used.

Results
<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Draft mesh</td>
</tr>
<tr>
<td>1</td>
<td>5.940</td>
<td>5.912</td>
</tr>
<tr>
<td></td>
<td></td>
<td>High quality mesh</td>
</tr>
<tr>
<td>1</td>
<td></td>
<td>5.903</td>
</tr>
</tbody>
</table>

Reference
Leissa, A.W. "Vibration of Plates," NASA, sp-160, p. 44.
Frequency Analysis of a Clamped Circular Plate

Description
Obtain the first three natural frequencies of a 40"-radius, 1" thick circular plate clamped at its edge.

Three shell studies are created as follows:

<table>
<thead>
<tr>
<th>Study name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Full</td>
<td>Full model</td>
</tr>
<tr>
<td>Quarter</td>
<td>Quarter model</td>
</tr>
<tr>
<td>Full_Quarter</td>
<td>Detached full and quarter models</td>
</tr>
</tbody>
</table>

This verification problem illustrates the limitation of using symmetry in frequency analysis as unsymmetrical modes are not detected. The Full_Quarter study demonstrates the ability of the solvers to solve detached models simultaneously.

File Name: `<install_dir>\Examples\Verification\Frequency_3.SLDPRT`

Study Type: Frequency.

Mesh Type: Shell mesh using surfaces.

Shell Parameters: Shell thickness = 1 in - Thick formulation.

Meshing Options: High, Standard, 4 Points, and Smooth surface.

Meshing Parameters: Use the default Global Size.

Material Properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.3, Density = 0.282072 lb/in^3.

Modeling Hints: Due to the double-symmetry, only a quarter of the plate is modeled.

Results

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>COSMOSWorks (Hz)</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Full Model</td>
<td>Quarter</td>
<td>Full and Quarter Models</td>
</tr>
<tr>
<td>1</td>
<td>62.199</td>
<td>62.226</td>
<td>62.224</td>
</tr>
<tr>
<td>2</td>
<td>129.210</td>
<td>211.850</td>
<td>62.226</td>
</tr>
<tr>
<td>3</td>
<td>129.23</td>
<td>241.52</td>
<td>129.33</td>
</tr>
<tr>
<td>4</td>
<td>211.52</td>
<td>421.73</td>
<td>129.33</td>
</tr>
<tr>
<td>5</td>
<td>211.54</td>
<td>511.47</td>
<td>211.85</td>
</tr>
<tr>
<td>6</td>
<td>241.08</td>
<td>538.70</td>
<td>211.85</td>
</tr>
<tr>
<td>7</td>
<td>308.71</td>
<td>688.57</td>
<td>211.87</td>
</tr>
<tr>
<td>8</td>
<td>308.77</td>
<td>843.16</td>
<td>241.50</td>
</tr>
<tr>
<td>9</td>
<td>367.61</td>
<td>924.58</td>
<td>241.52</td>
</tr>
<tr>
<td>10</td>
<td>367.70</td>
<td>950.81</td>
<td>309.46</td>
</tr>
</tbody>
</table>

Animate the modes to see that the quarter study does not detect unsymmetric modes like the 129.33 Hz mode. Also animate all the modes in the Full_Quarter study to see the sequence of mode extraction. Results agree closely with the reference.

Reference
Frequencies of a Cylindrical Thin Shell

Description
Determine the first three natural frequencies of the cylindrical shell shown in the figure. The model dimensions are: L = 6 in, R = 3 in, and t = 0.01 in. Consider symmetric modes only.

File Name: `<install_dir>\Examples\Verification\Frequency_4.SLDPRT
Study Type: Frequency.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.01 in - Thin formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 30,000,000 psi, Poisson's ratio = 0.3, Density = 0.282072 lb/in³.
Modeling Hints: Since symmetric modes are considered, 1/8 of the cylinder is modeled.

Results

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>552</td>
<td>538</td>
</tr>
<tr>
<td>2</td>
<td>736</td>
<td>727</td>
</tr>
<tr>
<td>3</td>
<td>783</td>
<td>768</td>
</tr>
</tbody>
</table>

Reference
Vibration of a Clamped Wedge

Description
Determine the fundamental frequency of the lateral vibrations of a cantilever wedge-shaped plate. The plate has the dimensions shown in the figure. The dimensions of the wedge are as follows: \( t = 1 \text{ in}, b = 2 \text{ in}, L = 16 \text{ in}. \)

File Name: `<install_dir>\Examples\Verification\Frequency_5.SLDPRT`

**Study Type**: Frequency.

**Mesh Type**: Shell mesh using surfaces.

**Shell Parameters**: Shell thickness = 1 in - Thin formulation.

**Meshing Options**: High, Standard, 4 Points, and Smooth surface.

**Meshing Parameters**: Use a Global Size of 0.75”.

**Material Properties**: Modulus of elasticity = 3 X 10^7 psi, Density = 0.2812 lb/in^3.

**Results**
The first in-plane natural frequency is calculated by: \( f_1 = \left( \frac{5.315 \; b}{2 \pi L^2} \right) \left( \frac{E}{3 \rho} \right)^{1/2}. \)
The first and second natural frequencies are calculated using Ritz approximate method by: \( f_1 = \left( \frac{5.319 \; b}{2 \pi L^2} \right) \left( \frac{E}{3 \rho} \right)^{1/2} \) and \( f_2 = \left( \frac{17.301 \; b}{2 \pi L^2} \right) \left( \frac{E}{3 \rho} \right)^{1/2}. \) Where \( E \) is the modulus of elasticity and \( \rho \) is the density.

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>774.55</td>
<td>778.36</td>
</tr>
<tr>
<td>2</td>
<td>2521.3</td>
<td>2264.6</td>
</tr>
</tbody>
</table>

**Reference**
Simply Supported Rectangular Plate With Inplane Pressure

Description
Obtain the fundamental frequency of a 40"X40"X1" simply supported square plate subjected to an in-plane pressure of 33.89 psi as shown in the figure.

File Name: <install_dir>\Examples\Verification\Frequency_6.SLDPRT
Study Type: Frequency.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1 in - Thin formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 7.5".
Material Properties: Modulus of elasticity = 3 X 10^4 psi, Poisson's ratio = 0.3, Density = 0.1158 lb/in^3.
Modeling Hints: The Direct sparse solver is used with the Use in-plane effect flag turned on.

Results

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.20</td>
<td>4.19</td>
</tr>
</tbody>
</table>

The study WithoutLoad shows the fundamental frequency to be 6.095 Hz when the compressive load is not considered. Compressive loads lower the natural frequencies.

Reference
Frequency Analysis of a Cantilever Beam

Description

Find the first two natural frequencies of a cantilever beam in the X-Y plane. The cantilever is 6" long and has a rectangular cross section of 0.1" by 0.2".

File Name: <install_dir>\Examples\Verification\Frequency_7.SLDPRT
Study Type: Frequency.
Mesh Type: Solid (Study 1) - Shell mesh using mid-surface (Study 2).
Meshing Options: High/Draft (Study 1 & Study 2), Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 1 X 10^7 psi, Poisson’s ratio = 0.3, Density = 0.0946239 lb/in^3.

Results

<table>
<thead>
<tr>
<th></th>
<th>1st Mode (Hz)</th>
<th>2nd Mode (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>181.17</td>
<td>1136.29</td>
</tr>
<tr>
<td>COSMOSWorks (Solid mesh)</td>
<td>High</td>
<td>184.10</td>
</tr>
<tr>
<td></td>
<td>Draft</td>
<td>202.77</td>
</tr>
<tr>
<td>COSMOSWorks (Shell mesh)</td>
<td>High</td>
<td>181.23</td>
</tr>
<tr>
<td></td>
<td>Draft</td>
<td>184.77</td>
</tr>
</tbody>
</table>

The following formulas were used to calculate the natural frequencies of the beam:

\[ f_1 = (1.875)^2 \left( \frac{EI}{ML^3} \right)\frac{1}{2}\pi \] and \[ f_2 = (4.694)^2 \left( \frac{EI}{ML^3} \right)\frac{1}{2}\pi. \]

Where:

E: Modulus of elasticity, I: Moment of inertia of the section of the beam, M: Mass of the beam, and L: the length of the beam.
Natural Frequencies of a Two-Mass Spring System

Description
Determine the normal modes and natural frequencies of the spring-mass system shown below.

![Diagram of a two-mass spring system]

File Name: <install_dir>\Examples\Verification\Frequency_8.SLDASM
Study Type: Frequency.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 1 in.
Spring and Mass Properties: \( m_2 = 2m_1 = 1 \text{ lb-sec}^2/\text{in}, k_2 = k_1 = 200 \text{ lb/in}, k_c = 4k_1 = 800 \text{ lb/in}. \)
Modeling Hints: A spring connectors is used to simulate the spring between the two masses. Elastic supports are used to simulate the springs between the masses and the walls. Small cubes are used to model the masses.

Results

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.581</td>
<td>2.606</td>
</tr>
<tr>
<td>2</td>
<td>8.326</td>
<td>8.335</td>
</tr>
</tbody>
</table>

Reference
Frequency Analysis of a Simply Supported Beam

Description
Determine the natural frequencies of a simply supported 80" long beam with 2"X2" square cross-section. Consider modes in the XY plane only.

File Name: <install_dir>\Examples\Verification\Frequency_9.SLDPRT
Study Type: Frequency.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 2 in - Thick formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 30,000,000 psi, Poisson's ratio = 0.3, Density = 0.282072 lb/in³.

Results

<table>
<thead>
<tr>
<th>Frequency No.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>28.78</td>
<td>28.68</td>
</tr>
<tr>
<td>2</td>
<td>115.12</td>
<td>113.80</td>
</tr>
<tr>
<td>3</td>
<td>259.00</td>
<td>251.79</td>
</tr>
</tbody>
</table>

Reference
Frequency of a Cantilever Beam with Lumped Mass

Description
A 10" long cantilever beam has a square cross-section of 0.25" X 0.25". A mass of 10 lbs is attached to the free end of the cantilever as shown in the figure. Determine the natural frequency of the system if the mass is displaced slightly and released.

File Name: <install_dir>\Examples\Verification\Frequency_10.SLDASM
Study Type: Frequency.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 30,000,000 psi, Poisson's ratio = 0, Density = 0.282072 lb/in^3.
Modeling Hints: A small block of artificially high density is used to model the mass.

Results
Two studies are used: study Nodensity does not consider the mass of the beam (density = 0). Study Density considers the mass of the beam.

<table>
<thead>
<tr>
<th></th>
<th>Frequency, Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>5.355</td>
</tr>
<tr>
<td>COSMOSWorks (Nodensity)</td>
<td>5.351</td>
</tr>
<tr>
<td>COSMOSWorks (Density)</td>
<td>5.341</td>
</tr>
</tbody>
</table>

Reference
Lateral Vibration of an Axially Loaded Bar

Description
Determine the first three natural frequencies of lateral vibration of a simply supported beam. The plate has a rectangular cross-section 2"X2" and its length is 80". An axial force of 40,000 lbs is applied at the roller support.

File Name: <install_dir>\Examples\Verification\Frequency_12.SLDPRT
Study Type: Frequency.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 2 in.
Material properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.3, Density = 0.2812683 lb/in^3.
Modeling Hints: Use the Sparse solver with the Use In-plane Effect flag on.

Results

<table>
<thead>
<tr>
<th></th>
<th>f_1 (Hz)</th>
<th>f_2 (Hz)</th>
<th>f_3 (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>17.055</td>
<td>105.32</td>
<td>249.39</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>17.068</td>
<td>104.68</td>
<td>245.19</td>
</tr>
</tbody>
</table>

The study NoLoadEffect shows the fundamental frequency to be 28.729 Hz when the load is not considered. Compressive loads lower the natural frequencies.

Reference
Natural Frequencies of a Long Bar

Description
Calculate the first three natural frequencies of a long bar fixed at one end. The bar is 50" long and has a square cross section of side 0.9".

File Name: \<install_dir>\Examples\Verification\Frequency_13.SLDPRT

Study Type: Frequency.

Mesh Type: Solid mesh.

Meshing Options: High, Standard, 4 Points, and Smooth surface.

Meshing Parameters: Use the default Global Size.

Material properties: Modulus of elasticity = 3 X 10^7 psi, Poisson's ratio = 0.3, Density = 0.2835 lb/in^3.

Results

<table>
<thead>
<tr>
<th></th>
<th>f1 (Hz)</th>
<th>f2 (Hz)</th>
<th>f3 (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>11.79</td>
<td>74.47</td>
<td>208.54</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>11.77</td>
<td>73.62</td>
<td>205.62</td>
</tr>
</tbody>
</table>

Reference
Natural Frequencies of a Ring

Description
Calculate the natural frequencies of an Alloy Steel ring with an outer diameter of 40", inner diameter of 36", and a thickness of 2 inches.

File Name: <install_dir>\ Examples\ Verification\ Frequency_14.sldprt
Study Type: Frequency.
Mesh Type: Solid mesh.
Mesh Parameters: Default Global Size and Tolerance.
Number of modes: The number of modes is set to 6 in the properties of the study.
Restraints: A planar face is constrained in its normal direction.

Results:
NOTE: The first 3 modes are rigid body modes (0 frequency).

<table>
<thead>
<tr>
<th>Mode shape no.</th>
<th>Theory (Hz)</th>
<th>COSMOSWorks (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>139.87</td>
<td>141.11</td>
</tr>
<tr>
<td>5</td>
<td>139.87</td>
<td>141.13</td>
</tr>
<tr>
<td>6</td>
<td>392.15</td>
<td>395.76</td>
</tr>
<tr>
<td>7</td>
<td>392.15</td>
<td>395.85</td>
</tr>
<tr>
<td>8</td>
<td>742.79</td>
<td>750.01</td>
</tr>
<tr>
<td>9</td>
<td>742.79</td>
<td>750.16</td>
</tr>
</tbody>
</table>

Reference
Frequency Analysis of a Building Frame

Description
A 3-story building frame is shown in the figure. The mass of the frame is lumped in the girders, with values as shown, and the columns are assumed to be weightless. Also, the girders are assumed to be rigid, so that the columns in each story act as simple lateral springs with stiffness values as indicated. It is desired to calculate the resonance frequencies of the frame.

\[
\begin{align*}
\text{Mass} &= 3.8604\times10^5 \text{ lbs} \\
K &= 6\times10^5 \text{ (lb/in)/in}^2 \\
\text{Mass} &= 5.7906\times10^5 \text{ lbs} \\
K &= 1.2\times10^6 \text{ (lb/ft)/in}^2 \\
\text{Mass} &= 7.7208\times10^5 \text{ lbs} \\
K &= 1.0\times10^6 \text{ (lb/ft)/in}^2
\end{align*}
\]

The problem is idealized as the mass-spring system shown below:

File Name: <install_dir>\ Examples\ Verification\ Frequency_15.sldasm
Study Type: Frequency.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a global element size that corresponds to the leftmost position on the mesh slider. A larger element size can be used since only rigid body modes are of interest.
Modeling Hints: Artificial material densities are used to account for proper mass values (mass = volume*density). The Spring and Elastic Support connectors are used to define column stiffness values.

Results

<table>
<thead>
<tr>
<th>Mode</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>First Mode (rad/sec)</td>
<td>14.5</td>
<td>14.4</td>
</tr>
<tr>
<td>Second Mode (rad/sec)</td>
<td>31.1</td>
<td>31.2</td>
</tr>
<tr>
<td>Third Mode (rad/sec)</td>
<td>46.1</td>
<td>46.6</td>
</tr>
</tbody>
</table>

Reference
Buckling Analysis

Buckling of a Frame

Description
Two parallel sides of a 100" square frame are loaded axially as shown. The frame has a uniform square cross section of 1"X1". Find the symmetrical buckling load in the plane of the frame.

File Name: <install_dir>\Examples\Verification\Buckling_1.SLDPRT

Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1" - Thin formulation.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 1 X 10^7 psi.
Modeling Hint: A quarter of the frame is modeled due to double symmetry and since only the symmetrical modes are considered.

Results:

<table>
<thead>
<tr>
<th></th>
<th>Buckling Load (lbs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>1372</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td></td>
</tr>
<tr>
<td>High</td>
<td>1381</td>
</tr>
<tr>
<td>Draft</td>
<td>1421</td>
</tr>
</tbody>
</table>

The theoretical value is calculated from: 

\[ P_c = 16.47EI / L^2 = 1372.45 \]

where:

\[ P_c \] = critical (buckling) force, \( E \) = Modulus of elasticity, \( I \) = Moment of inertia of the cross section, \( L \) = Length of the frame side.

The critical load from COSMOSWorks is calculated as:

\[ P_c \ (COSMOSWorks) = (Applied \ force) \times (Buckling \ load \ factor) \]

Reference:
Buckling of a Quarter Ring Under a Radial Force

Description
A quarter ring has a radius of 5", a width of 1", and a thickness of 0.1". The quarter ring is supported symmetrically. Find the buckling load applied radially as shown.

File Name: <install_dir>\Examples\Verification\Buckling_2.SLDPRT

Mesh Type: Shell mesh using surfaces.

Shell Parameters: Shell thickness = 0.1" - Thin formulation.


Meshing Parameters: Use the default Global Size.

Material Properties: Modulus of elasticity = 1 X 10^7 psi.

Results:

<table>
<thead>
<tr>
<th></th>
<th>Buckling Load (lbs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>26.66</td>
</tr>
<tr>
<td>COSMOSWorks (Shell mesh)</td>
<td></td>
</tr>
<tr>
<td>High</td>
<td>26.66</td>
</tr>
<tr>
<td>Draft</td>
<td>26.67</td>
</tr>
</tbody>
</table>

The theoretical value is calculated from Donnell approximation as: \( P_{cr} = \frac{4EI}{R^3} = 26.66 \text{ lb/in}, \) where: 

- \( P_{cr} \) = critical (buckling) force,
- \( E \) = Modulus of elasticity,
- \( I \) = Moment of inertia of the cross section of the shell,
- \( R \) = Radius.

The critical load from COSMOSWorks is calculated as:

\[ P_{cr} \text{ (COSMOSWorks)} = (\text{Applied force}) \times (\text{Buckling load factor}) \]

Reference:
Buckling Analysis a Cantilever Beam

Description
A cantilever beam of length 50" and a square cross section of side 1" is fixed from one end and subjected to a compressive normal force of magnitude 1 lb. Calculate the buckling load factor.

File Name: <install_dir>\Examples\Verification\Buckling_3.SLDPRT
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3X10^7 psi, Poisson's ratio = 0.

Results

<table>
<thead>
<tr>
<th></th>
<th>Buckling Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>2467.4</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>2466.7</td>
</tr>
</tbody>
</table>

Reference
Buckling of a Bar Under End and Distributed Axial Loads

Description
A 20"-uniform bar of 1"X1" square cross section, made of Alloy Steel, is fixed at one end and subjected to an axial load of magnitude \( P = 50 \) lbs at the other free end. Another nonuniform axial load that varies linearly from a value of \( p = 10 \) lbs at the fixed end to a zero value at a distance \( a = 10" \) from the fixed end is also applied. Assuming that \( P \) does not cause buckling on its own, calculate the value of the product \((p*a)'\) that will cause buckling.

\[
(p*a)' = K \pi^2 \frac{E}{I/L},
\]

where \((p*a)'\) is the theoretical critical value of the product \((p*a)\), \(K\) is an empirical factor that depends on the ratio \( a/L \) and \( P/(p*a) \). The value of \(K\) is looked up from table 34 of the reference cited below as 0.49. \(E, I,\) and \(L\) are the Elasticity modulus of Alloy Steel, the moment of inertia of the cross section of the bar, and the length of the bar respectively.

<table>
<thead>
<tr>
<th>COSMOSWorks</th>
<th>Analytical Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Buckling value ((p*a)), lbs</td>
<td>31202</td>
</tr>
</tbody>
</table>

Reference
Buckling of Columns with Variable Cross Sections

Description
A long column consists of two portions each with a constant cross section. Both cross sections are squares with sides 1" and 0.795" with the thicker portion in the bottom. The thicker portion is fixed at its base and a force of 100 lbs is applied to the free end of the thinner portion. The lengths of the thicker and thinner portions are 60" and 40" respectively. Calculate the buckling load of the column.

File Name: <install_dir>\Examples\Verification\Buckling_5.SLDPRT
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Modulus of elasticity = 3X10^7 psi, Poisson's ratio = 0.3.

Results

<table>
<thead>
<tr>
<th>Critical (Buckling) Load</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
</tr>
<tr>
<td>COSMOSWorks</td>
</tr>
</tbody>
</table>

The buckling load is calculated as: \( P_c = \frac{2.12 \ E I_z}{L^2} \), where:

\( P_c \) = critical (buckling) force, \( E \) = Modulus of elasticity, \( I_z \) = Moment of inertia of the cross section of the thicker portion of the column, \( L \) = total length of the column.

The critical load from COSMOSWorks is calculated as:

\( P_c \) (COSMOSWorks) = (Applied force)*(Buckling load factor)

Reference
Thermal Analysis

Steady State Heat Conduction in a Plate

Description
The temperature at one of the edges of a square plate 4mX4mX0.1m is maintained at 100 °C. The temperature of the other 3 edges is maintained at 0 °C. Find the temperature at the center of the plate.

File Name: <install_dir> \ Examples\ Verification\Thermal\ Thermal_1.SLDPRT
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.1 m - Thin shell formulation.
Meshing Options: High/draft (2 studies), Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use the default Global Size.
Material Properties: Thermal conductivity = 43 w/(m.°C).
Modeling Hints: One half of the plate is modeled. The plate is split into two surfaces so that the center can be easily identified.

Results:

<table>
<thead>
<tr>
<th>Theory</th>
<th>Temperature at Center (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>COSMOSWorks</td>
<td>High 25</td>
</tr>
<tr>
<td>Draft</td>
<td>25</td>
</tr>
</tbody>
</table>

Reference:
Steady State Heat Flow in an Orthotropic Plate

Description
A 1mX2mX0.1m rectangular plate is generating heat at a rate of \( Q = 100 \text{ w/m}^3 \). Two adjacent edges are insulated and the two other edges are dissipating heat to the atmosphere at 0 °C. The plate has orthotropic properties. Determine the steady state temperature distribution in the plate.

File Name: <install_dir> \ Examples\ Verification\ Thermal\ Thermal_2.SLDPRT
Study Type: Steady state thermal analysis.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.1 m - Thin shell formulation.
Meshing Parameters: Use the default Global Size.
Material Properties and Other Inputs: Thermal conductivity in X = \( K_X = 10 \text{ w/(m}.\text{°C)} \), Thermal conductivity in Y = \( K_Y = 20 \text{ w/(m}.\text{°C)} \). Convection coefficient along the long edge = \( h_1 = 10 \text{ w/m}^2.\text{°C} \). Convection coefficient along the short edge = \( h_2 = 1 = 20 \text{ w/m}^2.\text{°C} \).
Modeling Hint: One half of the plate is modeled. Insulated conditions are automatic when no other condition is applied.

Results
The graph of the temperature variation in the X direction along the bottom edge of the model is shown in the figure below. The graph is in good agreement with reference results.

TIP: Plot the temperatures, then right-click on the plot, choose List Selected, select the bottom edge, click Update, and then click Plot to generate the graph.

Reference
Transient Heat Conduction in a Long Cylinder

Description
A long aluminum cylinder, 50 mm in diameter is initially at 200 °C, is suddenly exposed to a convection environment at 70 °C and a convection coefficient of 525 W/m²°C. Calculate the temperature at a radius of 12.5 mm, one minute after the cylinder is exposed to the environment.

File Name: <install_dir> \ Examples\ Verification\ Thermal\ Thermal_3.SLDPRT
Study Type: Transient thermal analysis.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 10 mm - Thin shell formulation.
Meshing Parameters: Use the default Global Size.
Material properties: Thermal conductivity = 215 W/m°C, Density = 2700 Kg/m³, Specific heat = 936.8 J / Kg°C.
Modeling Hint: A small wedge away from the ends of the cylinder is used. The effect of convection of the circular end faces of the cylinder is ignored. The model is insensitive to thickness as the heat dissipation per unit length of the cylinder is constant. The problem is solved using solid and shell elements. Split lines are used so that the results along the 12.5 mm radius are easily evaluated.

Results

<table>
<thead>
<tr>
<th>Theory</th>
<th>Temperature at radius 0.0125 m</th>
<th>Error %</th>
</tr>
</thead>
<tbody>
<tr>
<td>COSMOSWorks Shell</td>
<td>119.49 (average)</td>
<td>1%</td>
</tr>
<tr>
<td>COSMOSWorks Solid</td>
<td>119.49 (average)</td>
<td>1%</td>
</tr>
</tbody>
</table>

The listed temperature is the average temperature at a radius of 12.5 mm at 60 seconds.

Reference
Thermal Analysis with Phase Change

Description
A uniform infinite slab of liquid is considered to be initially at zero degree temperature. Suddenly, the temperature of the surface \( x = 0 \) is reduced to -45 F and maintained constant. Calculate the temperature distribution in the slab and the time variation (response) at \( x = 1" \).

File Name: <install_dir>\Examples\Verification\Thermal_4.SLDPRT

Study Type: Transient thermal analysis.

Mesh Type: Shell mesh using surfaces.

Shell Parameters: Shell thickness = 1" - Thin formulation.

Meshing Options: High, Standard, 4 Points, and Smooth surface.

Meshing Parameters: Use a Global Size of 0.1".

Material Properties: Density = 1 lb/in\(^3\), Thermal conductivity = 1.08 BTU/(in-sec-F). The specific heat of the liquid (C) has the following temperature variation:

Modeling Hints: A zero temperature is initially applied to the model. The study properties are set to Transient with a Total time of 2 sec and 0.1 sec Time increment.

Results: The following response at \( x = 1" \) is found to be in good agreement with the results reported in the reference mentioned below.

Reference
Temperature Distribution in a Slab Exchanging Radiation with Ambient Atmosphere

Description
A slab of dimensions 2mX1mX1m has the following boundary conditions: heat flux of 10.4 W/m² on the face AD, convection on the face AB with heat transfer coefficient of 1.2 W/(m²·K), and a radiation boundary condition on the face BC with emissivity of 0.5. The ambient temperature for convection and radiation is 100 °C. Calculate the steady state temperature at point R (1.6m,0.5m).

File Name: <install_dir>\Examples\Verification\Thermal_5.SLDPRT
Study Type: Steady state thermal analysis.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1 m - Thin formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 0.1 m.
Material Properties: Thermal conductivity = 1 W/(m·K).

<table>
<thead>
<tr>
<th>Results</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature at point R, °C</td>
<td>101.69</td>
<td>101.69</td>
</tr>
</tbody>
</table>

Reference
R. Siegel and J. R. Howell, "Thermal Radiation Heat Transfer,".
Heat Conduction Due to Heating Cables

Description
A series of heating cables have been placed in a conducting medium. The medium has conductivities of 
$K_X = 10 \text{ W/(m.°K)}$ and $K_Y = 15 \text{ W/(m.°K)}$. The lower surface is bounded by an insulating medium. Assuming 
that each cable is a point source of 250 W, determine the temperature distribution in the medium.

File Name: <install_dir>\Examples\Verification\Thermal_6.SLDPRT

Study Type: Steady state thermal analysis.

Mesh Type: Shell mesh using surfaces.

Shell Parameters: Shell thickness = 1 m - Thin shell formulation.

Meshing Options: High, Standard, 4 Points, and Smooth surface.

Meshing Parameters: Use a Global Size of 0.5 m.

Material Properties: Thermal conductivity in the x direction $K_X = 10 \text{ W/(m.K)}$, Thermal conductivity in the 
y direction $K_Y = 15 \text{ W/(m.K)}$.

Modeling Hints: Since the cables are uniformly distributed throughout the medium, the problem can be 
simplified by analyzing only the section shown in the figure. Because of symmetry, the two vertical sides 
are insulated.

Results

<table>
<thead>
<tr>
<th>Temperature at cable location, °C</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>299.1</td>
<td>298.8</td>
<td></td>
</tr>
</tbody>
</table>

Reference
Transient Heat Conduction in a Slab of Constant Thickness

Description
A large plate of thickness 62.8 cm is initially at a temperature of 50°C. Suddenly, both of its faces are raised to and held at 550°C. Determine:

1. The temperature at a plane 15.7 cm from the left surface, 5 sec after the sudden change in surface temperature.
2. Instantaneous heat flux at the left surface at the end of 5 seconds.

File Name: <install_dir>\Examples\Verification\Thermal_7.SLDPRT
Study Type: Transient thermal analysis.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1 cm - Thin formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 3.9 cm.
Material Properties: Density = 23.2 Kg/m$^3$, Thermal conductivity = 46.4 W/(m.K), Specific heat = 1000 J/(Kg.K).
Modeling Hints: Since the other dimensions of the plate are infinitely large, conduction occurs through the plate thickness, i.e., along the x axis. For the transient study properties, the total solution time is set to 5 sec and the time increment is set to 0.05 sec.

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature at x = 15.7 cm, °C</td>
<td>183.9</td>
<td>183.4</td>
</tr>
<tr>
<td>Heat flux, W/m$^2$</td>
<td>130880</td>
<td>132180</td>
</tr>
</tbody>
</table>

Reference
Heat Transfer from Cooling Fin

Description
A cooling fin of square cross section of area 0.0069 ft$^2$, and length 0.667 ft extends from a wall maintained at temperature 100°F. The surface convection coefficient between the fin and the surrounding air is 1.929e-6 BTU/(s.in$^2$.F), the air temperature is 0°F and the tip of the fin is insulated. Determine the temperature of the fin tip.

File Name: <install_dir>\Examples\Verification\Thermal_8.SLPRT
Study Type: Steady state thermal analysis.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 0.4".
Material Properties: Thermal conductivity = 0.000579 BTU/(s.in.F).

Results

<table>
<thead>
<tr>
<th>Temperature of the fin tip, °F</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>68.594</td>
<td>68.581</td>
</tr>
</tbody>
</table>

Reference
Heat Conduction with Temperature-Dependent Conductivity

Description
Determine the temperature distribution in a slab of dimensions (2mx0.1mx1m) which is insulated on one face, and subjected to a constant temperature of 100°C on the other face. Assume a constant internal heat generation in the slab of 1e5 W/m³ and a linear variation of thermal conductivity.

File Name: <install_dir>\ Examples\ Verification\ Thermal\ Thermal_9.SLDPR
Study Type: Steady state thermal analysis.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1 m - Thin shell formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 0.026 m.
Material Properties: Thermal conductivity has the following temperature dependency: K(T) = 50(1+2T) W/(m.°C).

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature at the right end, °C</td>
<td>118.24</td>
<td>118.25</td>
</tr>
<tr>
<td>Temperature at the middle of the slab, °C</td>
<td>113.96</td>
<td>113.96</td>
</tr>
</tbody>
</table>

Reference
Radiation from a Rod

Description
Determine the temperature distribution in a rod of dimensions 10m×1m×1m in which one end has a fixed temperature of 1000°C and the other end is radiating heat into an ambient atmosphere of temperature 10°C. Use a value of 1 for both radiation view factor and emissivity of the radiating surface.

File Name: <install_dir>\Examples\Verification\Thermal_10.SLDPRT
Study Type: Steady state thermal analysis.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 0.4 m.
Material Properties: Thermal conductivity = 45 W/(m.°C).

Results

<table>
<thead>
<tr>
<th></th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average temperature of the radiating end, °C</td>
<td>235.77</td>
<td>234.34</td>
</tr>
<tr>
<td>Total heat entering the rod, W</td>
<td>3439</td>
<td>3445.5</td>
</tr>
<tr>
<td>Total heat leaving the rod, W</td>
<td>3439</td>
<td>3445.5</td>
</tr>
</tbody>
</table>

The analytical solution for this problem consists of solving the 1D heat transfer differential equation with the boundary conditions: at x = 0, T = 1000°C and at x = 10 m, T = T_{10}. Where T_{10} is the temperature at the radiating face. Then a heat balance equation at the radiating face is written such that the rate of heat conduction at that face equals the rate of heat lost to the environment by radiation. The resulting fourth order algebraic equation in T_{10} is solved using the Newton-Raphson method. The heat entering and leaving the rod are calculated from the gradients calculated at both ends of the rod.
Surface to Surface Radiation

Description
The thermal energy exchange through radiation between two spherical faces is compared with theoretical results. The two spheres are hollow with fixed temperatures at the inner faces. The two faces are assumed to radiate as black bodies (emissivity = 1.0). Radiation to the atmosphere is not considered (closed system).

File Name: <install_dir>\Examples\Verification\Thermal_11.sldasm
Study Type: Steady state thermal analysis.
Mesh Type: Solid mesh.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Default element size and tolerance. Identical local mesh controls are defined on the outer faces of the two spheres.

Results
- The theory calculates the Radiation View Factor (RVF) of each sphere to the other. The RVF from the smaller sphere to the larger one is calculated as 0.0459. The theoretical thermal energy (E) received by the colder sphere, which is equal to the thermal energy radiated by the hotter sphere, is calculated from:
  \[ E = F_{\text{small} \rightarrow \text{large}} \times \text{Area of small sphere} \times \text{Emissivity} \times (T_{\text{small sphere}}^4 - T_{\text{large sphere}}^4) = 5203.6 \text{ Watts} \]
- Use the List Selected tool to list the resultant heat flux on each of the outer faces of the two spheres. The two values should be equal in magnitude but opposite in sign.

<table>
<thead>
<tr>
<th>Thermal heat exchange at steady state (Watts)</th>
<th>COSMOSWorks</th>
<th>Analytical Solution</th>
</tr>
</thead>
<tbody>
<tr>
<td>For the large sphere = +5138.2 Watts</td>
<td>5203.6 Watts</td>
<td></td>
</tr>
<tr>
<td>For the small sphere = -5051.4 Watts</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Nonlinear Static Analysis

Simply Supported Rectangular Plate Under Normal Pressure

Description
Calculate the large deformation response of a simply supported square plate of side 2" and thickness 0.12" subjected to uniform pressure of magnitude 0.0625 psi.

File Name: <install_dir>\Examples\Verification\Nonlinear_Static_1.sldprt
Study Type: Nonlinear.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.12" - thin shell formulation.
Meshing Parameters: Use the default Global Size.
Material Model: Linear elastic isotropic.
Material Properties: Modulus of elasticity = 1 x 10^7 psi, Poisson's ratio = 0.3162.
Modeling Hints: Due to symmetry, only a quarter of the plate is modeled. Draft quality shell mesh is used.
Restraints and Loads: All edges are simply supported. The pressure load is applied as follows:
Results
The load parameter versus the deflection ratio at the center of the plate is in good agreement with analytical results.

NOTE: The results of the nonlinear analysis have been processed by a graphing software to generate the above graph.

Reference
Large Displacement of a Circular Plate

**Description**
Investigate the large displacement behavior of a simply supported circular plate of radius 100 cm and thickness 1 cm with radially movable edges. The plate is subjected to a normal pressure of magnitude $400 \text{ N/cm}^2$.

![Diagram of circular plate with boundary conditions and pressure application](image)

**File Name:** `<install_dir>\ Examples\ Verification\ Nonlinear_Static_2.sldprt
**Study Type:** Nonlinear.
**Mesh Type:** Shell mesh using surfaces.
**Shell Parameters:** Shell thickness = 1 cm - thin shell formulation.
**Meshing Options:** Draft, Standard, 4 Points, and Smooth surface.
**Meshing Parameters:** Use the default Global Size.
**Material Model:** Linear elastic isotropic.
**Material Properties:** Modulus of elasticity = $2 \times 10^8 \text{ N/cm}^2$, Poisson's ratio = 0.25.
**Modeling Hints:** Due to axial symmetry, only a wedge of the plate is modeled.
**Restraints and Loads:** The circular edge of the plate is simply supported. Symmetry boundary conditions are applied to the straight edges. Pressure is gradually applied normal to the plate as shown:

![Graph of pressure vs. time](image)
Results

<table>
<thead>
<tr>
<th>Deflection at the plate center (cm)</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>7.88</td>
<td>7.905</td>
</tr>
</tbody>
</table>

The deflection at the center of the plate as a function of pseudo time is shown below:

The time history of the displacement at the center of the plate is in good agreement with analytical results.

Reference

Large Deflection of a Clamped Square Plate

Description
Calculate the displacement of a clamped square plate of side 2 m and thickness 0.01 m under a normal pressure $q = 2 \times 10^4 \text{ N/m}^2$.

File Name: <install_dir>\ Examples\ Verification\ Nonlinear_Static_3.sldprt

Study Type: Nonlinear.

Mesh Type: Shell mesh using surfaces.

Shell Parameters: Shell thickness = 0.01 m - thin shell formulation.

Meshing Options: High, Standard, 4 Points, and Smooth surface.

Meshing Parameters: Use the default Global Size.

Material Model: Linear elastic isotropic.

Material Properties: Modulus of elasticity = $10.92 \times 10^{10} \text{ N/m}^2$, Poisson’s ratio = 0.3.

Modeling Hints: Due to double symmetry, only a quarter of the plate is modeled.

Restraints and Loads: The two outer edges are fixed. Symmetry restraints are applied to the other two edges. Pressure is linearly incremented to its full value of $q = 2 \times 10^4 \text{ N/m}^2$.
### Results

<table>
<thead>
<tr>
<th>Deflection at the plate center (m)</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.01594</td>
<td>0.0163</td>
</tr>
</tbody>
</table>

The following graph shows the deflection at the center of the plate versus time:

The results from the above curve are in agreement with analytical results.

### Reference

Thermal Stress - Temperature-Dependent Properties

Description
A plate is made of two different materials as shown in the figure. The plate thickness is 0.1". Determine the thermal stresses due to temperature changes of 100 °F and 200 °F. The full model is shown below:

The following figure depicts the loads and restraints applied to the half model.

File Name: <install_dir>\ Examples\ Verification\ Nonlinear_Static_4.sldasm
Study Type: Nonlinear.
Mesh Type: Solid mesh.
Meshing Options: High, standard, 4 Points, and Smooth surface.
Meshing Parameters: Use a Global Size of 0.5".
Material Model: Linear elastic isotropic material with temperature dependency.
Material Properties: Poisson's ratio = 0, Coefficient of thermal expansion = $1 \times 10^{-5}$ °F. Modulus of elasticity for part 1 of the plate is $3 \times 10^7$ psi. The modulus of elasticity of part 2 is temperature-dependent as follows:
Restraints and Loads: Due to symmetry, only one half of the plate is modeled. The right face of the plate is simply supported and the back faces are restrained in the Z-direction. The lower right edge is restrained in the Y-direction.

Two uniform changes in temperature (\( \Delta T_1 = 100 \, ^\circ F \) and \( \Delta T_2 = 200 \, ^\circ F \)) have been applied, one at a time, to both parts of the model. Each temperature rise followed the time curve shown below:

![Temperature vs. Time Graph]

<table>
<thead>
<tr>
<th>Results</th>
<th>( \Delta T_1 = 100 , ^\circ F )</th>
<th>( \Delta T_2 = 200 , ^\circ F )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum Stresses in X-direction (SX), psi</td>
<td>-30,000</td>
<td>-48,000</td>
</tr>
<tr>
<td>Theory</td>
<td>-30,000</td>
<td>-48,000</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>-30,000</td>
<td>-48,006</td>
</tr>
</tbody>
</table>
Contact Between Two Cubes

Description
A cube is pushed against another cube as shown by $4 \times 10^6$ psi. Calculate the maximum displacement.

File Name: `<install_dir>\ Examples\ Verification\ Nonlinear_Static_5.sldasm`

Study Type: Nonlinear.

Mesh Type: Solid mesh.


Meshing Parameters: Use default Global Size.

Material Model: Linear elastic isotropic.

Material Properties: Modulus of elasticity = $3 \times 10^7$ psi, Poisson's ratio = 0.

Restrains and Loads: The back face of the larger cube is fixed. Right and bottom faces of the two cubes are prevented from moving in their normal directions. A pressure is applied incrementally on the free face of the smaller cube according to the following time curve:

Results

NOTE: Use a deformation scale factor of 1.0 to plot results on the deformed shape. Animate the results to see how the two cubes move together after contact.

The linear theoretical elastic maximum displacement is calculated as $U = \frac{PL_1}{A_1E_1} + \frac{PL_2}{A_2E_2}$, where the subscripts refer to cubes 1 and 2. $L$ is the length, $A$ is the area ($A=L^2$) and $E$ is the Modulus of elasticity.

<table>
<thead>
<tr>
<th>Maximum Displacement (UZ, inch)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Linear Theory</td>
</tr>
<tr>
<td>COSMOSWorks (Node to node)</td>
</tr>
<tr>
<td>COSMOSWorks (Node to surface)</td>
</tr>
</tbody>
</table>
Deflection of a Nonlinear Elastic Cantilever Beam

Description
Determine the deflection of a cantilever beam under a torque applied to its free end. The beam has a length of 100" and a rectangular cross section of dimensions 2"X1".

The material is nonlinear elastic (defined by a stress-strain curve).

<table>
<thead>
<tr>
<th>Stress (psi)</th>
<th>Strain</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3E3</td>
<td>0.001</td>
</tr>
<tr>
<td>60.003E6</td>
<td>2.001</td>
</tr>
</tbody>
</table>

Material Properties: Poisson’s ratio = 0.

Restraints and Loads: The bar is fixed at one end and a torque is applied at the other end as follows:
### Results

<table>
<thead>
<tr>
<th>Deflection at free end (inch)</th>
<th>Linear elasticity range (Torque = 2000 lb-in)</th>
<th>Nonlinear elasticity range (Torque = 7000 lb-in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>-4.9998</td>
<td>-7.5156</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>-4.9748</td>
<td>-7.4939</td>
</tr>
</tbody>
</table>

The theoretical solution is based on thin beam theory.
Large Displacement of a Cylindrical Shell - Force

Description
A shallow cylindrical shell is subjected to concentrated load at its center. The curved edges are free and the straight edges are hinged. Due to symmetry, only a quarter of the plate (shown below) is modeled. Determine the deflection.

File Name: <install_dir>\ Examples\ Verification\ Nonlinear_Static_7.sldprt
Study Type: Nonlinear.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 12.7 mm - thick shell formulation.
Meshing Parameters: Use a Global Size of 0.8".
Material Model: Linear elastic isotropic.
Material Properties: Modulus of elasticity = 3102.75 N/mm², Poisson's ratio = 0.3.
Restraints and Loads: The shell is fixed along its straight edges and symmetry conditions are applied to the edges of symmetry. A force is applied at the center as shown:
Results

The force versus central deflection curve shown below is in agreement with analytical results.

NOTE: The results of the nonlinear analysis have been processed by a graphing software to generate the above graph.

Reference

Hornigmoen, G., "Finite Element Instability Analysis of Free-Form Shells," Report No. 77-1, the Norwegian Institute of Technology, the University of Trondheim, Norway (1977).
Buckling and Post Buckling of a Simply Supported Plate

**Description**
A simply supported isotropic square plate of side 40" and thickness 1" is subjected to in-plane uniform pressure $p$ on two opposite sides. Find the buckling load and investigate the post buckling behavior.

**File Name:** `<install_dir>\ Examples\ Verification\ Nonlinear_Static_8.sldprt`

**Study Type:** Nonlinear.

**Mesh Type:** Shell mesh using surfaces.

**Shell Parameters:** Shell thickness = 1 in - thick shell formulation.

**Meshing Options:** Draft, Standard, 4 Points, and Smooth surface.

**Meshing Parameters:** Use a Global Size of 2.5 in.

**Material Model:** Linear elastic isotropic.

**Material Properties:** Modulus of elasticity = $3 \times 10^4$ psi, Poisson's ratio = 0.3.

**Modeling Hints:** Due to double symmetry, only a quarter of the plate is modeled. The post buckling behavior is studied by applying a transverse force ($F_x$) at the center of the plate.

**Restraints and Loads:** The transverse force and the in-plane pressure are applied as shown in the figures below. In-plane rotations are set to zero and symmetry restraints are applied.

![Graphs showing buckling and post buckling behavior](image-url)
The figure below depicts the in-plane pressure versus the central deflection. The post buckling behavior matches the analytical solution.

NOTE: The above graph was not generated directly in COSMOSWorks. Results have been processed to generate the graph externally.

Reference
Large Displacement of a Cylindrical Shell - Pressure

Description
Determine the response of a cylindrical shell under normal uniform pressure $P$. All edges are fixed. The shell has the following dimensions: $L = 254$ mm, $\theta = 0.1$ rad, $R = 2540$ mm.

File Name: `<install_dir>\ Examples\ Verification\ Nonlinear_Static_9.sldprt
Study Type: Nonlinear.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 3.175 mm - thin shell formulation.
Meshing Parameters: Use a Global Size of 0.6".
Material Model: Linear elastic isotropic.
Material Properties: Modulus of elasticity = 3102.75 N/mm$^2$, Poisson's ratio = 0.3.
Restraints and Loads: The outer edges of the plate are fixed and symmetry conditions are applied to the edges of symmetry. The pressure is applied incrementally as follows:
Results
The pressure versus central deflection curve matches the analytical results.

NOTE: The above graph was not generated directly in COSMOSWorks. Results have been processed to generate the graph externally.

Reference
Viscoelastic of a Bar Under Constant Axial Load

Description
A bar made of a viscoelastic material is fixed at one end and a constant axial pressure is suddenly applied at the other end. Compare the axial strain to analytical results. The bar dimensions are as follows: \( L = 10" \), \( B = 3.14159" \), \( H = 0.25" \).

File Name: <install_dir>\ Examples\ Verification\ Nonlinear_Static_10.sldprt
Study Type: Nonlinear.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.25" - thick shell formulation.
Meshing Parameters: Use a Global Size of 5".
Material Model: Viscoelastic material model.
Material Properties: Modulus of elasticity = 10,000 psi, \((NUXY)_0 = 0.4833\), Shear relaxation modulus = 0.9010, Time value for the shear relaxation modulus = 0.9899 sec.
Restraints and Loads: Restraints are illustrated on the figure above. The pressure is applied suddenly as follows:

![Pressure vs. Time Graph](image)
Results

The following tables summarize the results for the instantaneous and long term behaviors:

<table>
<thead>
<tr>
<th></th>
<th>Instantaneous Behavior (t = 0.001 sec)</th>
<th>Long Term Behavior (t = 50 sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>(NUXY)$_0$ = 0.4833</td>
<td>(NUXY) = 0.4983</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>(NUXY)$_0$ = - EPSY/EPSX = 0.4833</td>
<td>(NUXY) = - EPSY/EPSX = 0.4983</td>
</tr>
</tbody>
</table>

The time history of the axial strain from COSMOSWorks is in good agreement with the analytical solution:

NOTE: The above graph was not generated directly in COSMOSWorks. Results have been processed to generate the graph externally.

Reference

Uniformly Loaded Elastoplastic Plate

Description
A square elastoplastic plate of side 1" and thickness 0.001" is subjected to non-proportional stressing. Use kinematic hardening to study its behavior.

File Name: <install_dir>\ Examples\ Verification\ Nonlinear_Static_11.sldprt
Study Type: Nonlinear.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 0.001" - thick shell formulation.
Material Model: Plasticity - von Mises.
Material Properties: Modulus of elasticity = $3 \times 10^7$ psi, Poisson's ratio = 0.3, SIGYLD = $1 \times 10^4$ psi, Tangent modulus of elasticity = $5 \times 10^6$ psi, Hardening factor = 1 (full kinematic hardening).
Restraints and Loads: Two edges are simply supported and all rotations are set to zero. The plate is subjected to uniform pressures $\sigma_x$ and $\sigma_y$ as illustrated in the following table:

<table>
<thead>
<tr>
<th>Loading Step</th>
<th>$\sigma_x$ (psi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>30,000</td>
</tr>
<tr>
<td>2</td>
<td>32,500</td>
</tr>
<tr>
<td>3</td>
<td>35,000</td>
</tr>
<tr>
<td>4</td>
<td>37,500</td>
</tr>
<tr>
<td>5</td>
<td>40,000</td>
</tr>
<tr>
<td>6</td>
<td>40,500</td>
</tr>
<tr>
<td>7</td>
<td>40,000</td>
</tr>
<tr>
<td>7 through 84</td>
<td>$(\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y)^{1/2} = 40,000$</td>
</tr>
</tbody>
</table>

Note that the plate is loaded into the plastic range in uniaxial tension ($\sigma_y$ = 40,000 psi). The load history is described by the equation in the table and are shown in the following figures:
The following plot shows load input $\sigma_y$ versus $\sigma_x$.

**Results**

The following plot shows $\sigma_y$ versus $\sigma_x$ plot as calculated by the program:
The $\sigma_x$ versus $\varepsilon_x$ plot, shown below, is in agreement with the analytical results.

NOTE: The above graph was not generated directly in COSMOSWorks. Results have been processed to generate the graph externally.

Reference
Large Displacement Analysis of a Cantilever Beam

Description
Investigate large displacements of a cantilever beam shown in the figure below. The cantilever beam has a length of 10" and a square cross section of side 1".

File Name: <install_dir>\ Examples\ Verification\ Nonlinear_Static_12.sldprt
Study Type: Nonlinear.
Mesh Type: Shell mesh using surfaces.
Shell Parameters: Shell thickness = 1" - Thin shell formulation.
Meshing Options: High, Standard, 4 Points, and Smooth surface.
Meshing Parameters: Use 0.3 in Global Size.
Material Model: Linear elastic isotropic.
Material Properties: Modulus of elasticity = 12000 psi, Poisson's ratio = 0.2, Density = 0.2782 lb/in³.
Restraints and Loads: A fixed restraint is applied to one end of the cantilever and a pressure of magnitude 1 lb is applied incrementally (as shown in the figure) to the top and bottom edges. Translation in the normal direction and all rotations are set to zero on the side face of the cantilever.
## Results

<table>
<thead>
<tr>
<th>Displacement at the free edge (UY), inch</th>
<th>Theory</th>
<th>COSMOSWorks</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>7.1697</td>
<td>7.1402</td>
</tr>
</tbody>
</table>

The tip vertical displacement history is shown below:

![Nonlinear Response](image)

Reference

Bearing Capacity for a Strip Footing

Description
Find the maximum vertical displacement of a strip footing of width 120" sitting at the ground surface subjected to a uniform pressure. The elastic properties of the soil are given below (in the material properties section).

File Name: <install_dir>\ Examples\ Verification\ Nonlinear_Static_13.sldprt
Study Type: Nonlinear.
Mesh Type: Solid mesh.
Meshing Parameters: Use the default Global Size.
Material Model: Drucker-Prager elastoplastic material model.
Material Properties: Elastic modulus = 3E4 psi, Poisson's ratio = 0.3, Cohesion strength = 1 psi, Friction angle = 27.27°, Density = 0.069337 lb/in³.
Modeling Hints: Due to symmetry, half of the soil is modeled. Because the bearing capacity of the foundations depends on the self-weight of the soil, an acceleration of gravity in the Y-direction is applied to simulate this effect. When the applied pressure load approaches the limit load (bearing capacity of the foundations), the soil bulging takes place adjacent to the footing.
Restraints and Loads: Symmetry is applied at the face of symmetry of the model. Gravitational acceleration is applied to the model in the -Y direction. The pressure is applied incrementally according to the following time curve:

Results

<table>
<thead>
<tr>
<th></th>
<th>Displacement in the Y-direction (UY), in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Theory</td>
<td>0.5992</td>
</tr>
<tr>
<td>COSMOSWorks</td>
<td>0.6005</td>
</tr>
</tbody>
</table>

Reference
Three-Point Bending of a Nitinol Wire

Description
A three-point bending test is performed on a Nitinol wire of circular cross section with diameter $d = 1.49$ mm. The wire is 20 mm long and it is simply supported at both ends.

a) Obtain the graph of the applied force versus deflection for the mid-span section of the wire.
b) Verify that increasing the ultimate plastic strain for the material results in a closer match with experimental results.

Displacement control is used to solve this problem. The node for displacement control is selected to be same as the node where the force is applied. This node is displaced 5.2 mm in the direction of the force and then brought back to zero.

File Name: $<install_dir>\Examples\Verification\Nonlinear_Static_14.sldprt$

Study Type: Nonlinear.
Mesh Type: Solid mesh.

Meshing Parameters: Use element size of 0.3 mm.

Material Model: Nitinol. Three studies are used (Sample_1, Sample_2, and Sample_3) with different Ultimate plastic strain measure (Tension) of 0.092 mm/mm, 0.15 mm/mm, and 0.092 mm/mm. Sample_3 differs from Sample_1 by using Exponential flow rule. Refer to the material definition in each study for details.
Results
The load factor versus UX displacement curve at the point of force application is shown for the three studies. Curves from study Sample_1 and Sample_2 show close agreement with the reference. The figure from study Sample_3 shows closer results to experimental data presented in the same reference.

Load factor (vertical) versus UX Displacement in mm (horizontal) for study Sample_1.

Load factor (vertical) versus UX Displacement in mm (horizontal) for study Sample_2.

Load factor (vertical) versus UX Displacement in mm (horizontal) for study Sample_3.

Reference