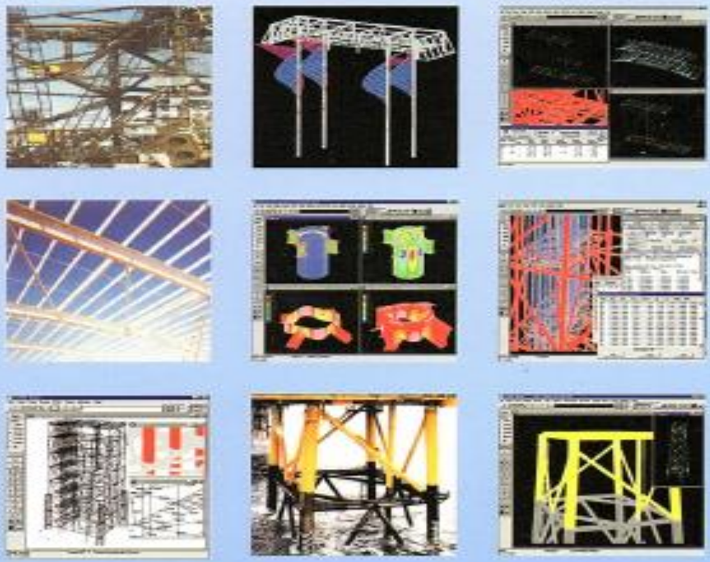
FS2000

Advanced Structural Analysis Software



Verification Examples

Version 1.5

John Moore

03/03/25

Analytical Engineering Services Ltd

Table of Contents

Introduction

The verification document presents FS2000 solutions to a range of problems which have solutions available from another sources. These solutions may be analytical, analysis solutions in the public domain or other software packages.

A comparison with third party solutions is not always that exact. Models may not be that same or have the same degree of refinement when comparing FE solutions. Analytical solutions may differ because of numerical accuracy or the difficulty in imposing the precisely the same idealised conditions. For this reason, no attempt has been made to include % comparisons.

This document is not intended as a tutorial document and the model description etc. have been restricted to single page for each problem regardless of example complexity. However, the solutions may be useful in illustrating the approach to various types of problems.

Although areas of application overlap, an attempt has been made to broadly categorise the solution by section heading.

Section 1 Generally Linear – Can be solved using the Standard 3-D solver.

Section 2 Non-Linear – P-Delta and Large Displacement

Section 3 Non-Linear – Large Displacement (Flexibles)

Section 4 Non-Linear - Elasto-Plastic

Section 5 Dynamic

Section 6 Heat Transfer

Section 7 Specific Applications

Note that 2-D and 3-D Elasto-Plastic solid solutions can only be solved using **DyNoFlex** because the degrees of freedom (DoF) are defined as 2 or 3 accordingly. Plastic shell (6 DoF) solutions can be solved using **3-D Non-Linear**.

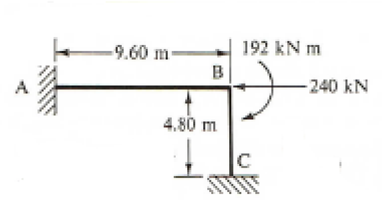
Earlier version of FS2000 may experience convergence issues with some Elasto-Plastic solutions that use solid elements.

Example 1.1 Plane Frame – Beam Elements

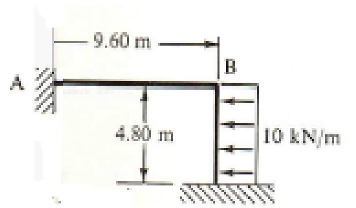
Model: **PlaneFrame1**

The example is a two-element rigid frame with fully fixed supports. Two cases are considered, one with concentrated nodal loads and one with distributed elements loads.

I = 8E-4m^4; A = 2E-3m^2; E = 210GPa



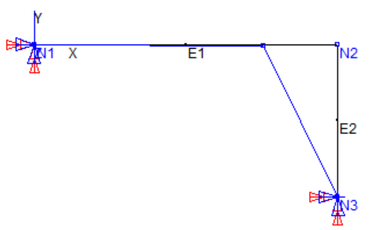
Case 1 Evaluate the nodal displacements.



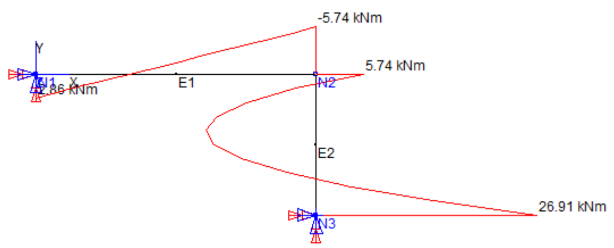
Case 2 Evaluate the moment distribution.

Reference Solution: Structural Analysis, RC Coates, MG Coutie, FG Kong, Second Edition 1980 Page 254-255.

The refence solution values are shown in parentheses.



Case 1
ΔX = -0.4478mm (0.448mm)
ΔY = -.01024mm (0.102)
Δθ = -8.215E-4Rad (8.22E-4)



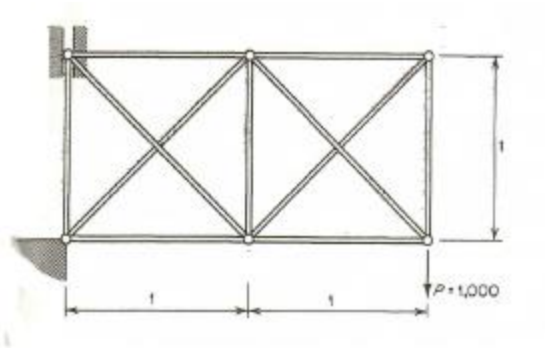
Case 2

Moment at A = 2.86kNm (2.86)
Moment At B = 5.74kNm (5.74
Moment At c = 26.91Nm (26.9

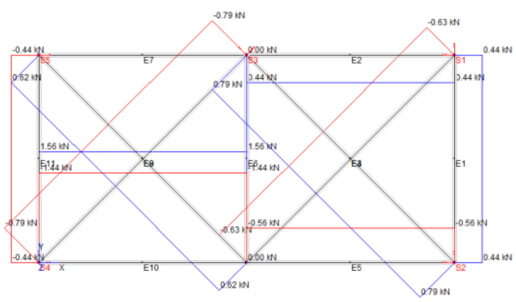
Example 1.2 Simple Truss – Beam and Couple Elements

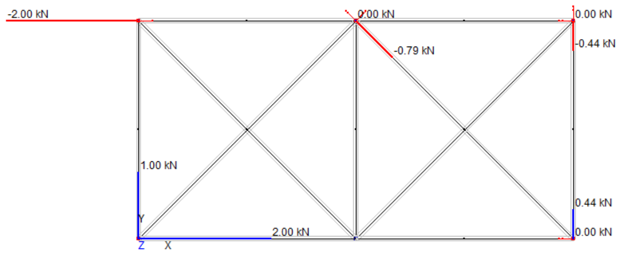
Mode:**PinnedTruss**

This is a simple pinned truss is modelled with rigid beam elements but because of the slenderness and the fact that there are only nodal forces, the frame behaves as pinned structure. Two of the elements are connected to the frame using couple elements. These couple elements have stiffness about 1000 times that of the beam axial stiffness. The couple local axis is referenced to the connected beams i.e. the local x axis are aligned. The model is restrained using node to ground couples using a similar stiffness.



Reference Solution: Theory of Matrix Structural Analysis, J.S. Przemeiniecki, 1985, Page229.





Member axial forces

Couple Forces

The couple forces show the restrained reactions and the forces in E1 and E3 at the connection points.

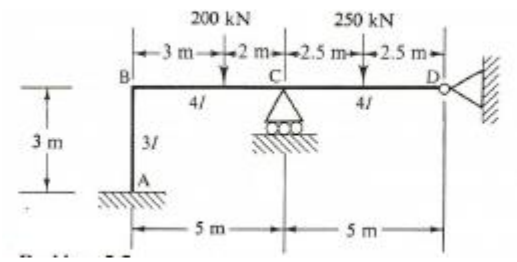
The following table give the normalised axial forces from the reference solution.

1	2	3	4	5	6	7	8	9	10	11
0.442	.442	0.789	-0.625	-0.558	0	1.558	0.625	-0.798	-1.442	-0.442

Example 1.3 Plane Frame Contact – Beam and Couple Elements

Mode:**BeamContact**

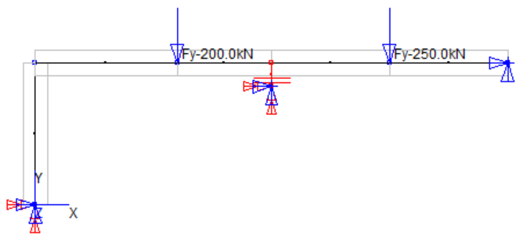
This is a frame arrangement formed with rigid beam elements. The model incorporates a compression only contact element at point C. The objective is to establish the support reaction.



To ensure compatibly with the reference solution the area of the section is defined with high value, the shear is made zero and the I value were varied using the E values.

The solution uses the Standard 3-D Solver with the Contact Option active.

Reference Solution: Structural Analysis, RC Coates, MG Coutie, FG Kong, Second Edition 1980 Page 185.



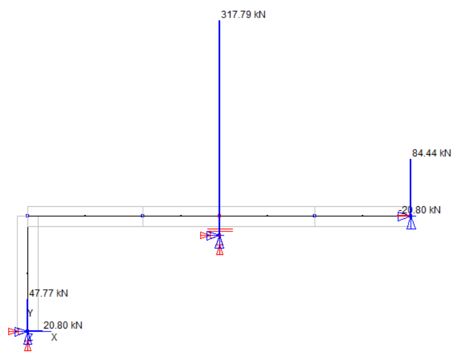
Case 1 Downward Loads.

Case 2 Upward Loads.

Case I Contact Closed

The refence solution values are shown in parentheses.

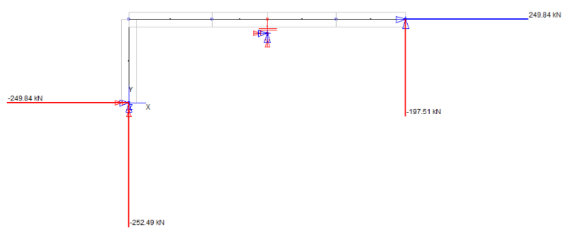
Action	A	B	C
Vertical kN	47.77 (47.8)	317.79 (318)	84.44 (84.4)
Horizontal kN	20.80 (20.8)	0	-20.80 (20.8)
Moment kNm	20.78 (20.8)	0	0



Case 2 Contact Open

No reference solution but solution clearly in equilibrium.

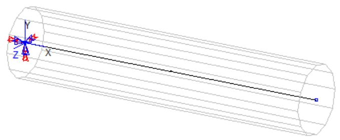
Action	A	B	C
Vertical kN	-252.49	000.00	-197.51
Horizontal kN	-249.84	0	249.84
Moment kNm	-249.65	0	0



Example 1.4 Pressurised Pipe – Pipe Element

Model: **Pipe**

The is an example of a pipe undergoing changes in pressure and temperature. Several different loading scenarios are considered.
Reference Solution: Roark.



Pipe Diameter = 219.1mm; Wall = 8.18mm; D/t =26.78.
E =203GPa; Poiss Ratio = 0.3; Coeff of Thermal Exp = 1.093E-5

Internal Pressure = 200Bar: External Pressure = 20 Bar
Change in Temperature 100C

The pipe is fully fixed at the LHS. The RHS has two conditions, axially free or axially restrained.

Case 1 Pressure only - free
Case 2 Temperature only - free
Case 3 Pressure and temperature – free
Case 4 Pressure only - fixed
Case 5 Temperature only - fixed
Case 6 Pressure and temperature – fixed

In pipe stiffness analysis the pipe ends are always assumed to be end capped and therefore when the pipe is fixed both the end cap pressure load and the wall restrain load contribute the restraining reaction. In this model the effective axial force is also the restraint reaction. The effective axial force is the force that can cause Euler buckling. In FS2000 the pipe axial force is always the effective axial force.

Theory
The evaluated hoop stress the standard output from FS2000 always uses Sh = $\Delta p.Do/2t$. Note that this may be different from that used in piping design codes
Roark presents formula for the axial displacement of thick-walled cylinders subjected to both internal and external pressure. These can be combined and re-arranged the give the following.
Pressure strain is based on $\varepsilon_P = (1 - 2.\mu)(Pi.Ai - Po.Ao) / (EAs) = 2.07E-4$
Thermal Strain $\varepsilon_T = \alpha.\Delta T = 1.093E-3$
Pressure Restraint force = $\varepsilon_P.As.E = 2.281E5$ Thermal Restraint force = $\varepsilon_T.As.E = 1.2049E6$

True Wall Axial Stress = (Effective Axial Force + End Cap Force) / As
End Cap Force = Pi.Ai - Po.Ao = 5.70E5: As = 5.4202E-3

The above agree exactly with the solution output given in the table.

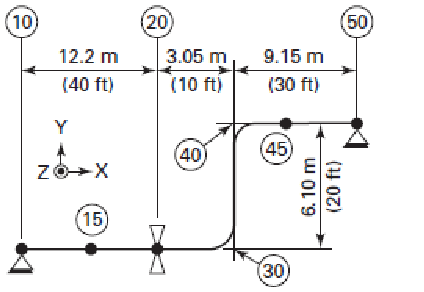
Case No	Hoop Stress MPa	True Wall Axial Stress MPa	Effective Axial Force kN	End Displacement mm
1	241.05	105.21	0	0.207
2	0	0	0	1.093
3	241.6	105.21	0	1.3
4	241.6	63.13	-228.1	0
5	0	-222.32	-1204.95	0
6	241.6	-159.19	-1433.05	0

Example 1.5 Piping Flexibility Analysis – Pipe Elements

Model: **Piping2**

This is a B31.3 example of a piping system subjected to gravitational, internal pressure and thermal expansion. The arrangement has two pipe bends that significantly reduce the expansion load induces in the pipe.

Fig. S301.1 Simple Code Compliant Model



OD 406.4mm (16”); Wall 9.53mm
Bend Radius 609.6mm (1.5D)

Material: ASTM A106Grade B
E = 203.4GP; μ = 0.3; α_T = 1.093E-5; Density 7850 kg/m^3

Bend Flexibility factor = 9.506 (FS2000 evaluated)

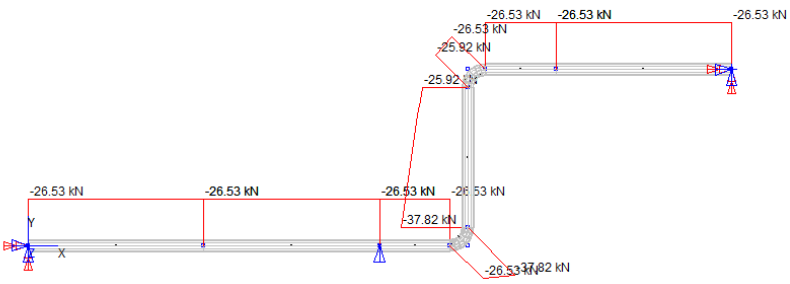
Pipe weight 248.36 kg/m

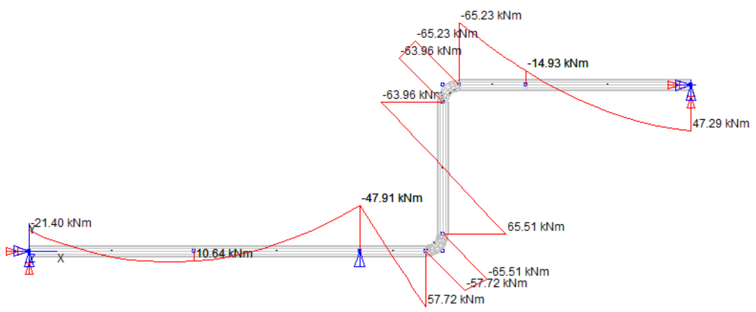
The pipe weight effect contents (SG-1) and insulation coating 127mm thick density 176kg/m^3.

Reference Solution: ASME B31.3 – 2010, Appendix S, Example 1 S301.
The refence solution values are shown in parentheses (averaged from commercial programs).

Note that longitudinal pressure effects are excluded for the expansion case.

Node	Axial Force	Bending Moment	X Deflection	Y Deflection
10	26.53 (26.5)	21.4 (21.52)	0	0
15	26.53 (26.5)	10.64 (10.71)	18.361 (18.3)	-1.304 (-1.3)
20	26.53 (26.5)	47.91 (47.56)	36.698 (36.7)	0
45	26.56 (26.5)	14.93 (14.9)	-18.361 (-18.3)	13.461 (13.5)
50	26.53 (26.5)	47.29 (47.48)	0	0



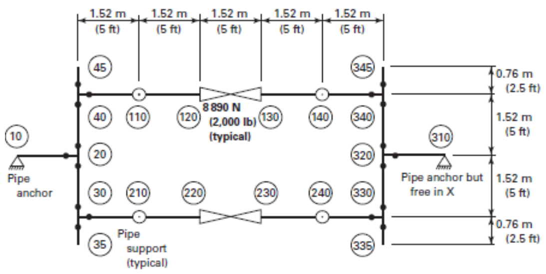


Example 1.6 Piping Flexibility Analysis – Pipe Elements

Model: **Piping3**

This is a B31.3 example of a piping system subjected to internal pressure and thermal expansion. The objective to evaluate the displacement stress/force ranges resulting from two operational conditions.

Fig. S303.1 Moment Reversal Model



Header
OD 609.6mm (24”); Wall 9.53mm
Branch
OD 508mm (20”); Wall 9.53mm
Material: ASTM A53Grade B
E = 203.4GP; μ = 0.3; α_T = 1.093E-5; Density 7850 kg/m^3

Valve Stiffness Factor = 10
Ambient Temp = 4.5C

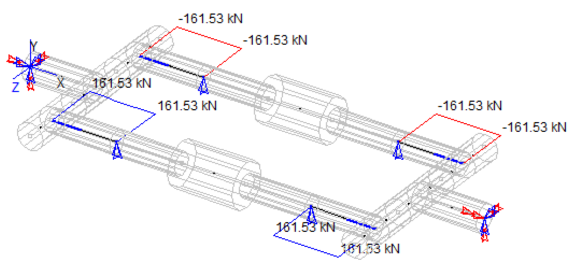
Condition	Headers		West Branch 30-330		East Branch 40-340	
Case 1	17.24 Bar	121 C	17.24 Bar	121 C	0	4.5C
Case 2	17.24 Bar	121 C	0	4.5 C	17.24 Bar	4.5C

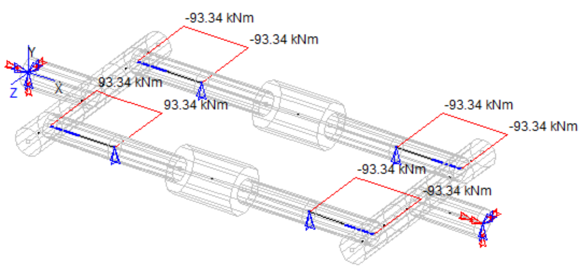
In this example there are only two operating conditions therefore the range can be obtained using a load case combination. Load Case Combination 10 subtracts Case 2 from Case 1 to obtain the range between the two cases.

Reference Solution: ASME B31.3 – 2010, Appendix S, Example 3 S301.
The refence solution values are shown in parentheses (averaged from commercial programs).

Branch - Axial Force Range = 161.54kN (156.97); Moment Range = 93.34kNm (91.8).

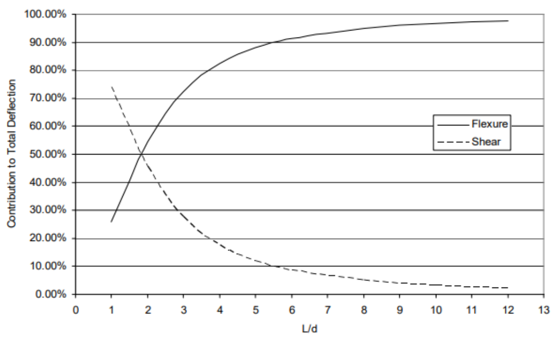
The reference solution has slightly lower value indications are more flexible arrangement. B31.3 does state that a variation can be expected depending on the stiffness parameters used. If a unity value was used for the valve stiffness factor the force and moment would reduce to 151.4kN and 78.16kNm.





Example 1.7 Cantilever Beam – Shear Deflection

Model: **ShearBeam**



In this example a deep cantilevered I beam is subjected to a tip. The resulting deflection is due to flexural deflection and shear deflection.

Unless the beam is deep the contribution from shear is generally small and is often neglected in hand calculations.

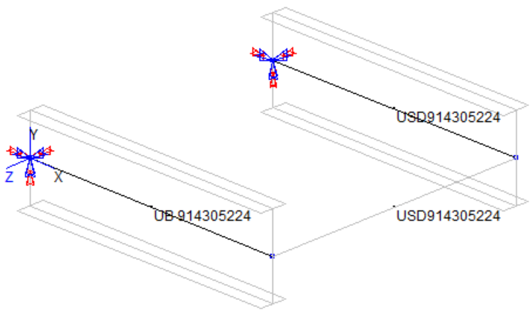
The contribution from shear is a function L/D for a specific section i.e. the slenderness.

The beam formulations in FS2000 include shear stiffness and therefore when comparing solution that don't a difference may be identified. In this example the deflection contributions are identified.

Reference Solution: S. Timoshenko, Strength of Material, Part II, Elementary Theory and Problems, 3rd Edition, D. Van Nostrand Co., Inc., New York, NY, 1955, article 39.

$$\Delta = \frac{PL^3}{3EI} + \frac{PL}{GA_v}$$

The comprises of two beam elements, one has shear deflection active and the other not.



The I beam is a UB914305224.

L = 3m
A_S = 1.447E-2 m^2
G = 78.85GPa
W = 600kN

Deflection due to Shear = W.L/As/G = 1.5776mm

Deflection including shear deflections = 8.583mm

Deflections excluding shear deflections = 7.006mm

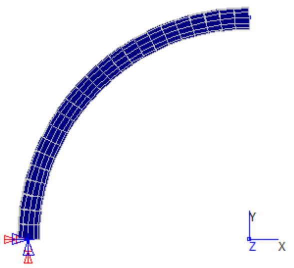
Contribution from shear deflections = 8.583mm - 7.006mm = 1.577mm (1.5776)

In this section this represents 18% difference for a L/D of 3.3.

Example 1.8 Curved Beam – Bend and Beam Elements

Model: **CurvedBeam**

In this example a cantilevered curved pipe beam is subjected to vertical and horizontal tip loading in the plane of curvature and normal to the plane of the curvature respectively.



Bend radius: 1m; Pipe OD; 100mm; Pipe wall:5mm
E = 203.4GPa ; Poisson's Ratio = 0.3;G=78.23GPa

Tip Load = 1 kN. A case with 10 Bar internal pressure is also analysed.
The model contains 3 bends formed from:

- One Type 3 Bend Element
- Four Type 3 Bend Elements
- Four Type 0 Straight Beam Elements

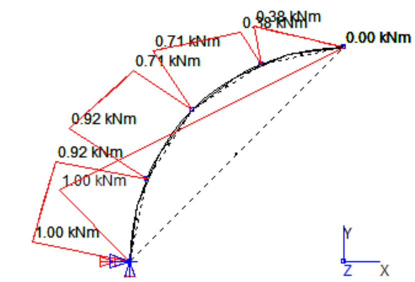
Reference Solution: S. Timoshenko, Strength of Material, Part II, Elementary Theory and Problems, 3rd Edition, D. Van Nostrand Co., Inc., New York, NY, 1955, article 80 & 85. To make the FS2000 solutions compatible with the slender theory solutions the shear displacements are excluded (Shear area =0).

Case 1 In-plane Vertical	1 Bend Elem	4 Bend Elems	4 Beam Elems
Vertical Displacement mm	2.290*(2.290)	2.290	2.218
Horizontal Displacement mm	1.455	1.455	1.445

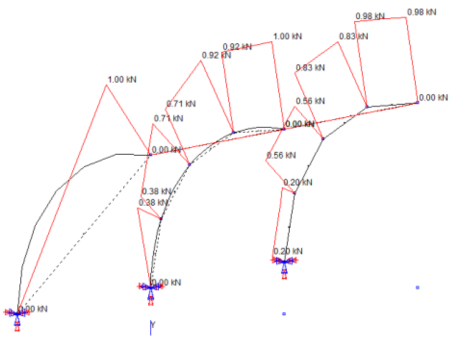
*With shear displacements included 2.304

Case 2 Out of plane Horizontal	1 Bend Elem	4 Bend Elems	4 Beam Elems
Horizontal Displacement mm	3.636 (3.636)	3.636	3.543

To be expected the bend element give identical results. The segmented bend formed by 4 straight elements gives very similar results but slightly stiffer.



The overlaid bending moment plots are identical for the 3 bend configurations.



The shear force plots show how the segmented bend configuration approximates the shear and axial effects. More segments would improve the accuracy.

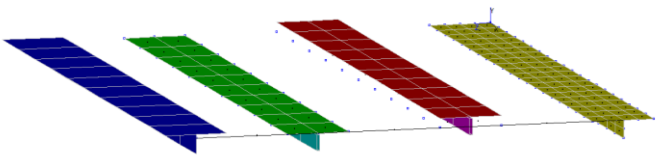
The batch file also incudes the same cases but using a non-linear time history solution.

Example 1.9 Cantilever – Beam & Shell Elements with Offsets

Model: **PlateBeams**

In this example a T section cantilever is modelled using four distinct techniques.

1. Conventional Beam Elements.
2. Shell elements with offset beams.
3. Offset shell element with beams.
4. All shell elements.

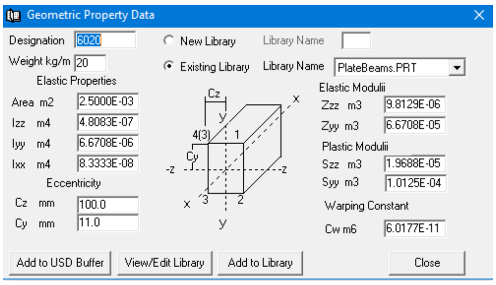


The use of offsets is commonly used to stiffen beam assemblies.

The T section has the following properties: Depth:60mm; Width:200mm; Thickness:10mm: Length:2m
E = 205GPa; Poisson's Ratio = 0.3. The tip load on the cantilever is 1kN.

The beam sections used in the offset configurations is a rectangular section 50mm deep by 10mm wide. This offset by 30mm, the distance between the flange and stem centroids.

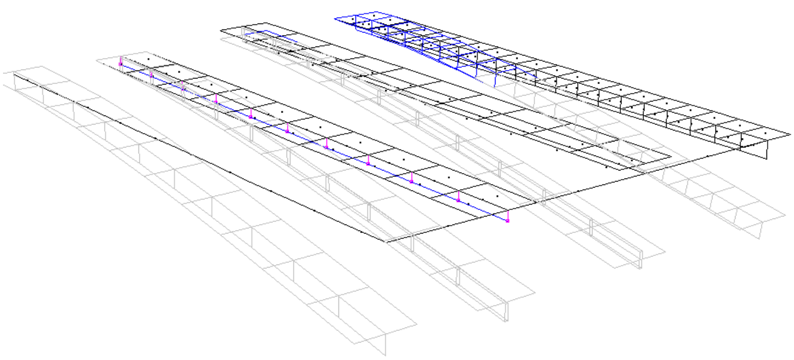
Reference Solution: Engineers Beam Bending.



The Tee section properties were evaluate using FS2000's property generator.

I = 4.808E-7m^4 ; Z = 9.8129E-6m^3; Shear Area = 6E-4m^2

Tip Displacement = Wl^3/3EI = 27.055mm
Bending Stress (stem) = W.l/Z = 203.8MPa
Shear Stress = 1.66MPa



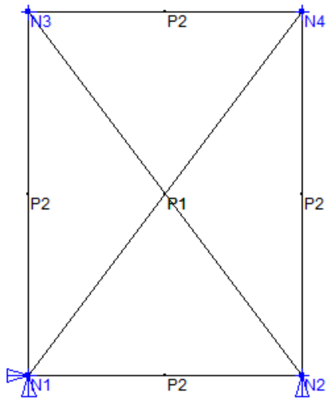
	Theory	Beam	Shell – Offset Beam	Offset Shell - Beam	Shell
Deflection mm	27.06	27.1	27.29	27.29	26.97
Stem Stress MPa	203.8	203.8	231.8*	231.8*	196.8
Flange Stress MPa	45.83	45.76	51.34	51.34	46.68

*Bending + Axial Stress

Example 1.10 Plane Truss – Thermal Expansion – Beam Elements

Model: **PlaneTruss**

The is an example of a simple 2-D truss in which one member is 1mm too short but is forced into place. The solution to the problem is to use thermal strain simulate the member being too short.



The frame is 4m high and 3m wide.

P1 members have a csa = 500mm^2
P2 members have a csa = 1000mm^2

The member between N2 to N3 which is 5m long is 1mm too short.

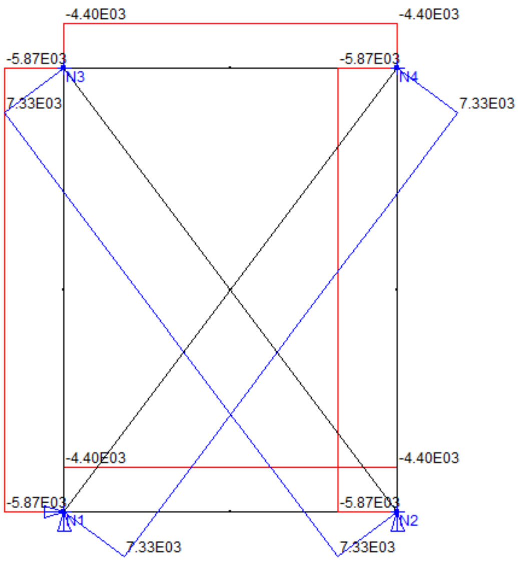
E = 200GPa
Coefficient of Thermal Expansion = 1E-5

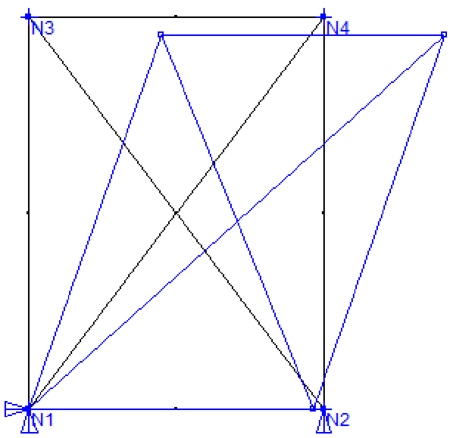
Reference Solution: Structural Analysis, RC Coates, MG Coutie, FG Kong, Second Edition 1980.

The model uses Type0 beam elements. To ensure the frame behaves as a truss the I values have been defined with a very low value (1E-12). More convenient than defining moment releases.

Temperature Difference applied to member = ε / α = 1E-3/5 / 1E-5 = 20C

The forces shown below agree exactly to those quoted in the reference solution.

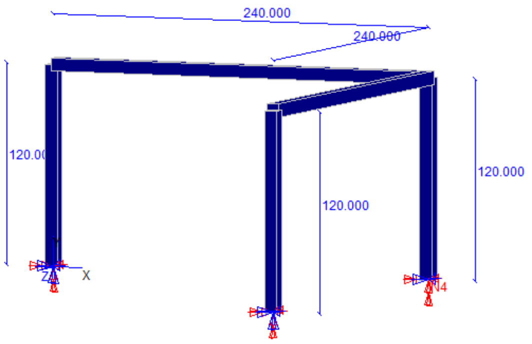




Example 1.11 3-D Portal Frame Settlement – Prescribed Displacement – Beam Elements

Model: **FrameSettlement**

The is an example of a simple 3-D portal in which column member sinks by 0.5 inches. There are no other loads on the frame. The model is US units.



A = 10ins^2; Ix = Iy = 300ins^4; Ix = 10ins^4

E = 29000ksi; μ = 0.3

Mid column settlement (N5) = 0.5”

All degrees of freedom at the column bases are are fixed.

Load Case defines a vertical downward displacement of -.5” at N4. No other loads are applied

The solution used the 3-D Standard Solver. This solver does allow displacements to be define in restrained freedoms. Note that the non-linear solver do not allow this.

Reference Solution: A STAAD model.

ALL UNITS ARE -- KIP INCH (LOCAL)

MEMBER	LOAD	JT	AXIAL	SHEAR-Y	SHEAR-Z	TORSION	MOM-Y	MOM-Z
1	1	1	0.97	-0.08	0.15	-0.46	-19.22	107.07
		2	-0.97	0.08	-0.15	0.46	0.65	-116.64
2	1	2	0.08	0.97	0.15	0.65	-0.46	116.64
		3	-0.08	-0.97	-0.15	-0.65	-36.66	116.82
3	1	3	-1.95	-0.07	0.07	-0.00	-116.17	-116.17
		4	1.95	0.07	-0.07	0.00	107.18	107.18
4	1	3	0.08	-0.97	-0.15	-0.65	36.66	-116.82
		5	-0.08	0.97	0.15	0.65	0.46	-116.64
5	1	5	0.97	0.15	0.08	0.46	-116.64	-0.65
		6	-0.97	-0.15	-0.08	-0.46	107.07	19.22

Note: kip-ins for moments

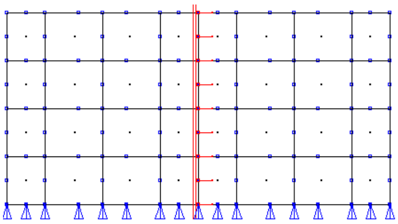
Elem	Node	Fx kip	Fy kip	Fz kip	Mx kip-ft	My kip-ft	Mz kip-ft	Mryz kip-ff
1	1	-0.97	0.08	-0.15	0.04	1.60	-8.92	9.06
	2	-0.97	0.08	-0.15	0.04	0.05	-9.72	9.72
2	2	-0.08	-0.97	-0.15	-0.05	0.04	-9.72	9.72
	3	-0.08	-0.97	-0.15	-0.05	-3.06	9.74	10.20
3	4	1.95	0.07	0.07	0.00	8.93	-8.93	12.63
	3	1.95	0.07	0.07	0.00	9.68	-9.68	13.69
4	3	-0.08	0.97	0.15	0.05	-3.06	9.74	10.20
	5	-0.08	0.97	0.15	0.05	0.04	-9.72	9.72
5	6	-0.97	-0.15	0.08	-0.04	8.92	-1.60	9.06
	5	-0.97	-0.15	0.08	-0.04	9.72	-0.05	9.72

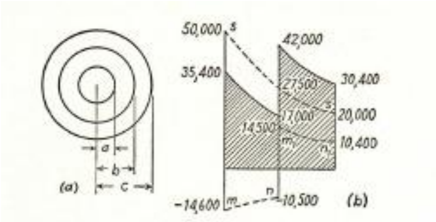
Example 1.12 Stresses in Thick Cylinders – Axisymmetric Elements with Contact

Model: **CylinderContact**

The is an example of a pressurised 2-part composite thick cylinder. A built-up cylinder with a radial interference is subjected to an internal pressure.

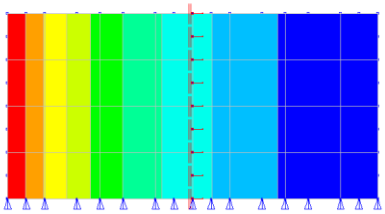
The assembly is modelled using Type40 2-D axisymmetric elements. The interface between the two cylinders uses contact elements – Type 12 Couple elements. Thermal strain was used to create the interference strain.

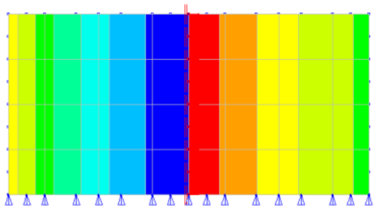




a = 4”
b = 6”
c = 8”
Interference = 0.005”
(137.25F)

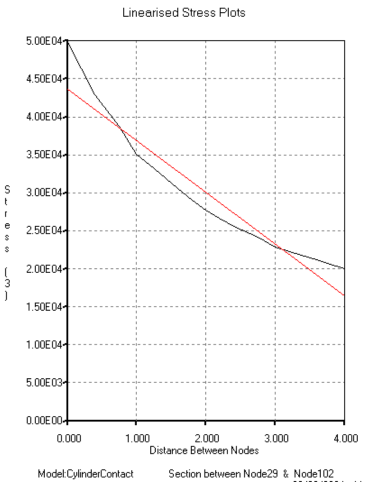
Reference Solution: S. Timoshenko, Strength of Material, Part II, Advanced Theory and Problems, 3rd Edition, D. Van Nostrand Co., Inc., New York, NY, 1956, pg. 211, problem 1 and pg. 213, article 41.

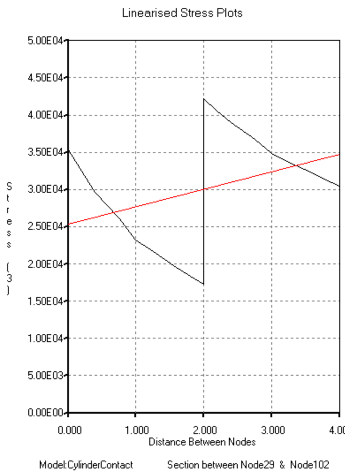




LHS – No radial interference.
Hoop Stress = 49.827ksi

RHS – With radial interference.
Hoop Stress = 42.192ksi





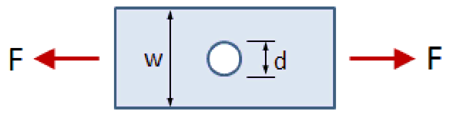
The interference fit decreases the hoop stress from 49.827ksi (50) to 42.192ksi (42).

The slight difference to the reference solution is due in most part to the linear extrapolation of the stresses from the Gauss point to the nodes.

Example 1.13 Tensile Plate with a Hole – 2-D Plane Stress Elements

Model: **Plate_Hole**

The is an example of a flat plate with a hole. The objective is to establish the SCF at the hole and the linearised stresses at the critical section. The model uses 8 Node Type 30 2-D plane stress elements.

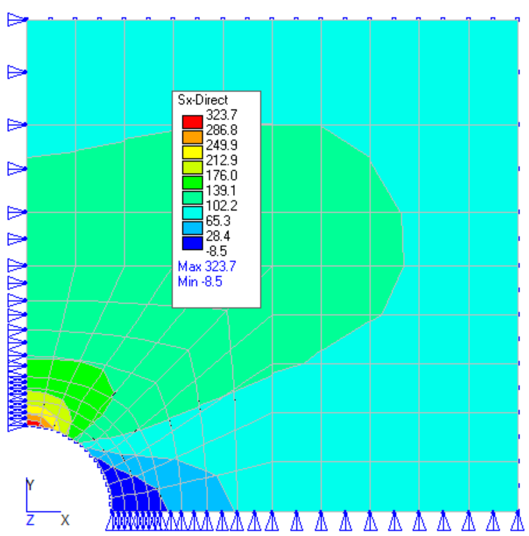


F = 2000kN
w = 400mm
t = 50mm
d = 70mm

Reference Solution: Roark

Nominal Stress = F/w.t = 100MPa

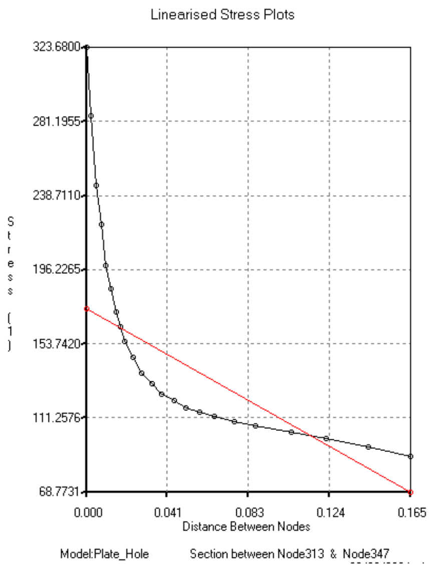
The refence solution values are shown in parentheses.



Maximum Stress = 323.7 MPa (310)

K_t = 3.24 (3.1)

If the size of the plate is increases to 800mm the SCF becomes 3.03 (the exact value for a hole in an infinite plate is 3.0)



LINEARISED STRESSES

Surface 1 at Node 313
Surface 2 at Node 347
Section width 0.165

Stresses at Surface 1

Sx Sy Sz Sxy Syz Szx
Peak 323.680 0.750 0.000 0.070 0.000 0.000
Linear 173.571 34.091 0.000 0.075 0.000 0.000

Derived Stresses (Linearised Components) at Surface 1
Principle Stress S1 = 173.5708
Principle Stress S2 = 34.09086
Principle Stress S3 = 0
Stress Intensity = 173.5708
Von Mises Stress = 159.2854

Mean (Membrane) accross section

Sx Sy Sz Sxy Syz Szx
121.172 13.754 0.000 0.073 0.000 0.000

Derived Stresses at Mean Section
Principle Stress S1 = 121.1719
Principle Stress S2 = 13.75423
Principle Stress S3 = 0
Stress Intensity = 121.1719
Von Mises Stress = 114.9138

Stresses at Surface 2

Sx Sy Sz Sxy Syz Szx
Peak 88.970 -0.410 0.000 -0.010 0.000 0.000
Linear 68.773 -6.582 0.000 0.070 0.000 0.000

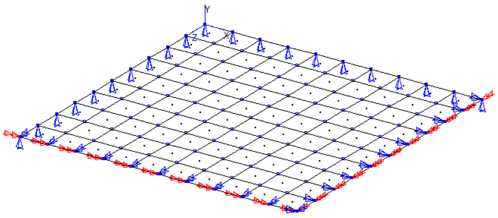
Derived Stresses (Linearised Components) at Surface 2
Principle Stress S1 = 68.77312

Example 1.14 Simply Supported Plate – Shell Elements

Model: **SSPlate**

The is an example of a simply square support flat plate subjected to out of plane uniform Pressure. The objective is to establish the displacement and the maximum bending stress in the plate.

Because the model and the loadings are symmetrical only a ¼ of the plate is modelled. The model uses a 10 x 10 – 4Node element mesh. Two solutions are undertaken, a thin plate solution using Type50 shells (Kichhoff theory) and a thick plate solution Type 52 shell (Mindlin theory). Note the next example uses 3-D elements to model the same plate.

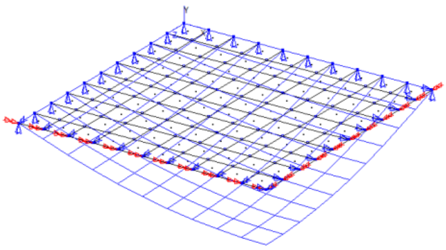


Length = Width = 1m Thickness = 100mm

E = 205GPa; Poiss = 0.3

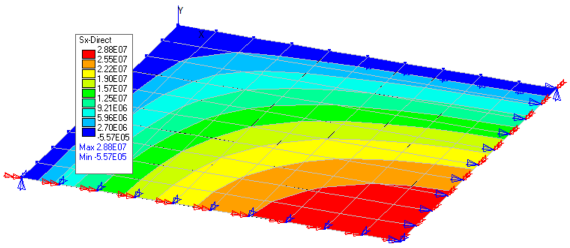
UDL = 1000 kN/m^2

Reference Solution: Roark(thin shell) & ANSYS(thick shell-SHELL43)



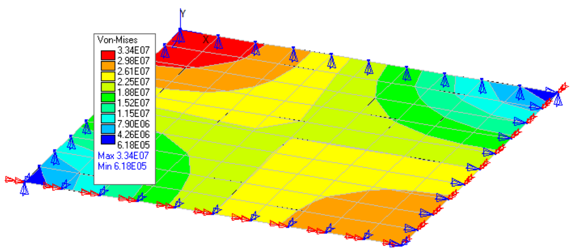
Thin wall solution 0.216 (.2166)

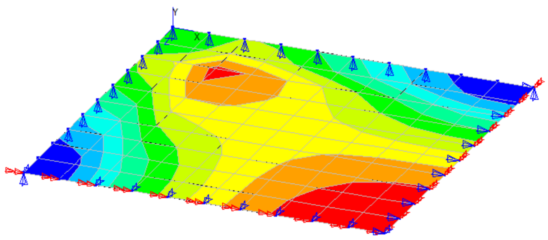
Thick wall solution 0.246mm (0.244)



Thin wall solution 28.8MPa (28.740)

Thick wall solution 30.53MPa (30.39)





Thin plate Von-Mises 33.4MPa (n/a))

Thick plate Von-Mises 30.3(MPa (30.39)

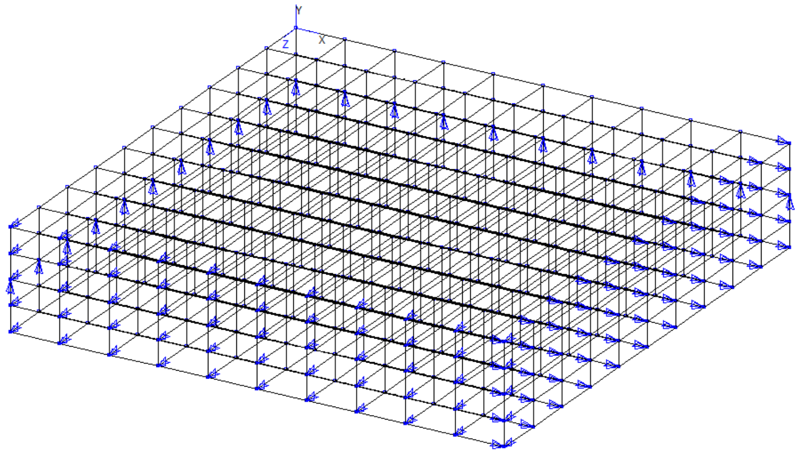
The twisting moment (Sxy Shear) is a maximum at the corner in the Kirchhoff thin plate theory.

Example 1.15 Simply Supported Plate – 3-D Solid (Type70) Elements

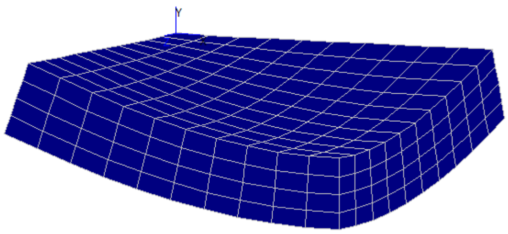
Model: **SSPlateBrick**

The is an example of a simply square support flat plate subjected to out of plane uniform Pressure. The objective is to establish the displacement and the maximum bending stress in the plate.

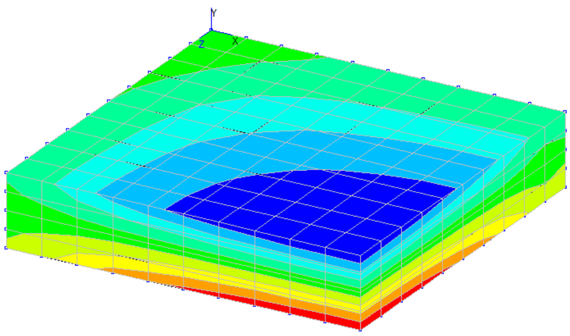
It is the same scenario as that of the previous example which used shell elements. This model was formed by extruding the shell in the y direction to give a depth (thickness) of 100mm. The model has 20 x 20 x 4 mesh of Type 70 elements. As with the previous example ¼ model symmetry utilised.



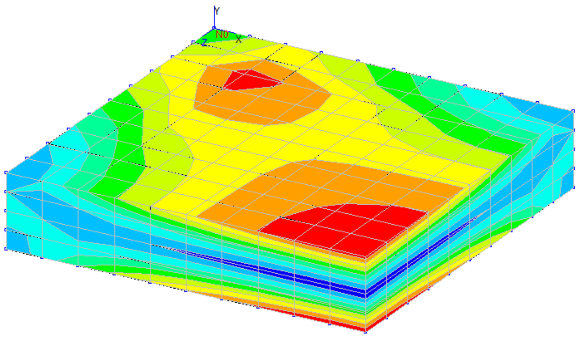
Refence Solution: Previous thick shell example.



Deflection 0.243mm (0.246)



Bending Stress 30.5MPa (30.3)

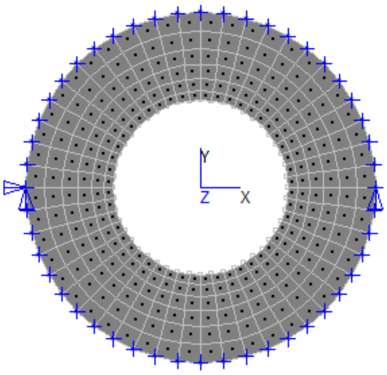


Von-Mises Stress 30.6 MPa (30.3)

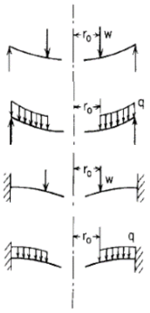
Example 1.16 Flat Ring - Linear Shell Elements

Model: **Flat_Ring**

The is an example of a flat circular ring subjection subjected to out of plane loading. The model uses Type 50 shell elements.
Reference solution: Roark



OD = 2m
ID = 1m
t = 20mm
E = 205GPA
Poisson Ratio = 0.3



Case 10 100 kN applied as a line load at the inner edge. Outer edge pinned.

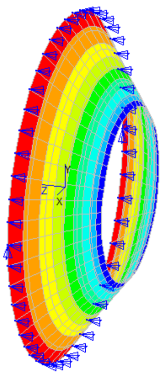
Case 11 100 kN/m2 applied as a UDL. Outer edge pinned.

Case 12 !00 kN applied as a line load at the inner edge. Outer edge fixed.

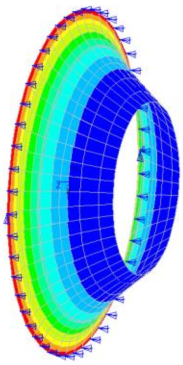
Case 13 100kN applied as a UDL. Outer edge fixed.

The refence solution values are shown in parentheses.

Case	Max Deflection	Inner σt MPa	Outer σr MPa
10	40.89 (40.97)	371.3 (370.3)	0
11	41.43 (41.54)	358.9 (360.6)	0
12	4.911(4.937)	69.7 (67.39)	115.8 (118.3)
13	3.540 (3.529)	41.3 (40.65)	124.0 (120.0)



No Radial Curvature



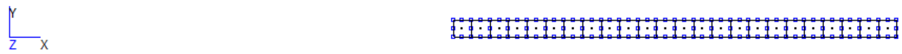
Radial Curvature at outer edge

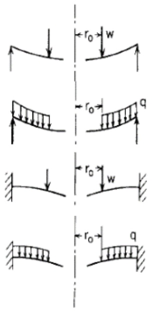
Example 1.17 Flat Ring - Linear Shell Elements

Model: **Flat_Ring-Solid**

The is an example of a flat circular ring subjection subjected to out of plane loading. The model uses 8 Node Type 40 axisymetric solid elements. The flat ring and loading are identical to that of the previous shell element example.
Reference solution: Roark

OD = 2m; ID = 1m; t = 20mm; E = 205GPA; Poisson Ratio = 0.3.





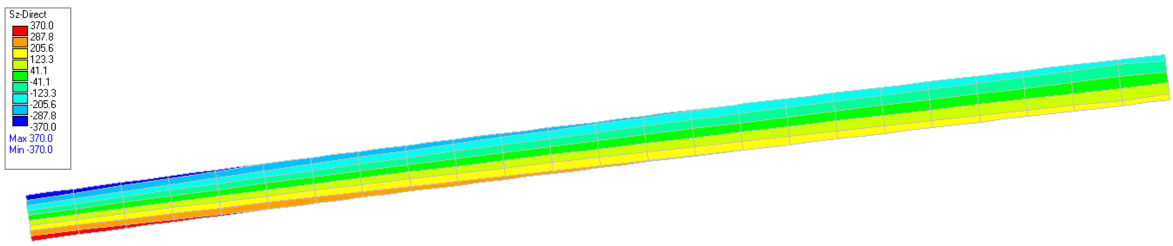
Case 10 100 kN applied as a line load at the inner edge. Outer edge pinned.

Case 11 100 kN/m2 applied as a UDL. Outer edge pinned.

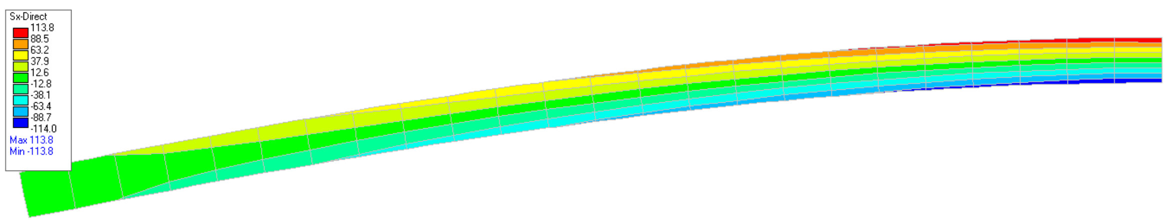
Case 12 !00 kN applied as a line load at the inner edge. Outer edge fixed.

Case 13 100kN applied as a UDL. Outer edge fixed. The refence solution values are shown in parentheses.

Case	Max Deflection	Inner σt MPa	Outer σr MPa
10	40.96 (40.97)	370.0 (370.3)	0
11	41.54(41.54)	360.3 (360.6)	0
12	4.918(4.937)	67.2 (67.39)	113.8 (118.3)
13	3.540 (3.529)	40.5 (40.65)	120.1 (120.0)



Case 10 No significant radial curvature



Case 12 Radial curvature at outer edge

Example 1.18 Torsion of a Square Box Beam - Linear Shell Elements/Beam Elements

Model: **BoxBeam**

The is an example of a thin-walled box beam being subjected to torsional moment. The objective is to establish the shear stress and the angle of twist. The model uses Type 50(0) shell elements and Type0 beam elements

The model has two sections, The RHS is modelled used shell elements and the LHS is modelled using one beam element. The centre of the section is fixed, and the torsion moments are applied at the free ends.

Length = 1m; Width (Height) = 150mm; t =T= 3mm; E = 205GPa; Poisson Ratio = 0.3: G = 78.85GPa.

Applied Torque = 3kNm

Reference solution: Roark

J = 2.T.t.(B-t)**2.(D-T)**2 / (B.t + D.T - T**2 - t**2)

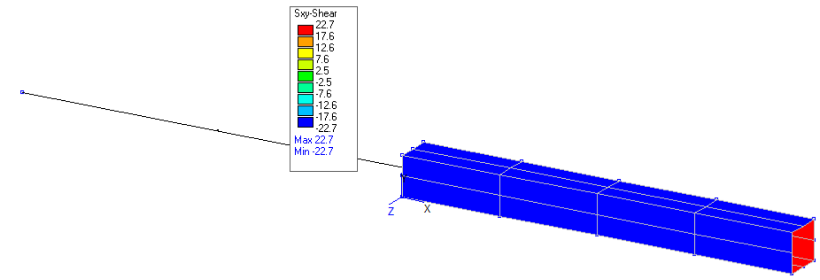
Zt = 2At where t is the smaller of web or flange thickness and

A =the mean enclosed area

Twist = T.L / (J.G) = 0.00599 Rads

Shear Stress = T / Zt = 23.14 MPa

The shell mid plane is at the mean wall i.e. the shell box is 147mm sq.



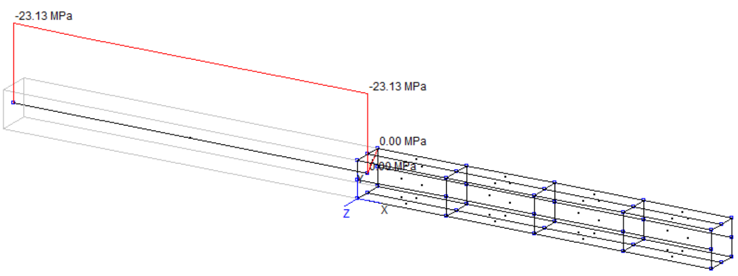
Mid Plane Shear Stress

22.7MPa (23.14)

End Twist

0.00588 Rads (.00599)

The beam torsional properties are based on the reference properties.



Torsional Beam Stress

23.13MPa (23.14)

End Twist

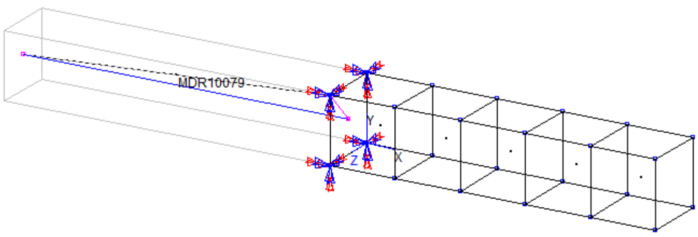
0.00599 Rads ((.00599)

Example 1.19 Bending of a Solid Beam - Linear Solid Hex/Beam Element with Offset

Model: **SolidBeam**

The is an example of a solid square sectioned beam being subjected to an end load shear load. The objective is to establish the stresses stress and the tip deflection. The model uses Type 70 brick elements and Type0 beam elements. The model also includes Beam offsets.

The model has two sections, The RHS is modelled using solid elements and the LHS is modelled using one beam element with the aft node offset. The centre of the section is fixed, and the shear loads are applied at the free ends.



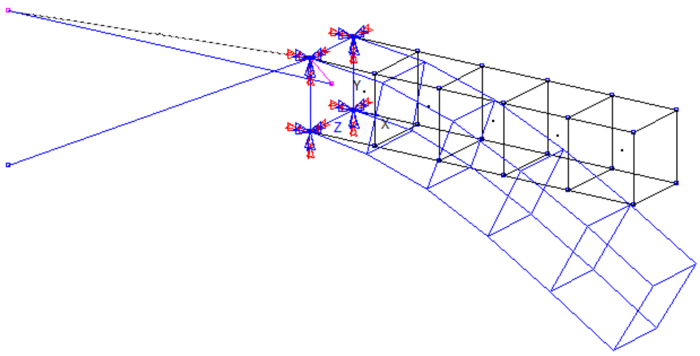
Length = 500mmm
Width (Height) = 100mm
E = 205GPa.

I = BD^3/12 = 8.3333E-6m^4

W = 4kN

Reference solution: Engineers Bending Theory - Deflection =WL^3/3EI

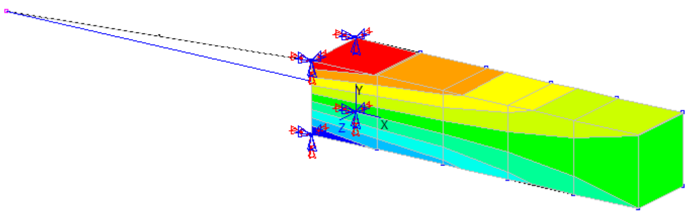
The refence solution values are shown in parentheses.



Deflection Solid Elements
0.971mm (0.976)

Deflection Beam Element
(includes shear deflection)
1.009mm (0.976)

Deflection Beam Element
(excluding shear deflection)
0.976mm (0.976)



Solid Element Stress
Bending(Nodal Ave) 117MPa
Shear 4MPa (W/csa)

Beam Element Stress
Bending 120MPa (120)
Shear 5.3MPa*(W/0.75A)

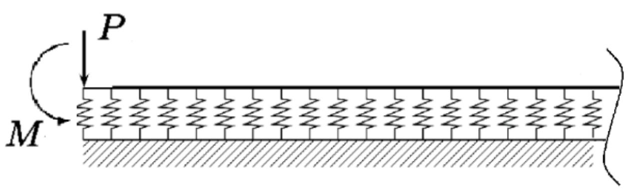
*Be default the shear area for solid rectangular beam sections is taken as 0.75BD.

The mesh density of the solid mesh is far too coarse to capture other that the basic bending which nevertheless is excellent for this type of element when they are square hexagonal.

Example 1.20 Beam on an Elastic Foundation – Linear Beam/Linear Springs

Model: **BeamWinkler**

This is an example of beam supported on an elastic foundation (Winkler Foundation). The objective is to establish the stresses in the beam and the tip deflection. The model uses discrete linear springs to represent the continuous foundation support i.e. a lumped approach.



Semi- infinite beam.
I = 7.2E-3 m^4
E = 21.7 $GN.m^2$
k = Foundation Modulus = 4000kN/m/m

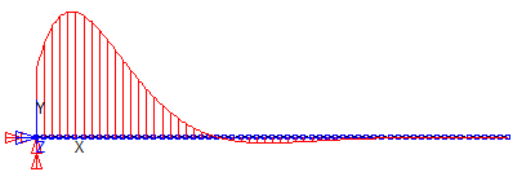
P = 100 kN   M = 100 kNm

Reference Solution: Roark

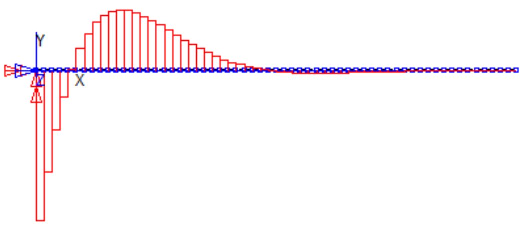
The solution of a beam on an elastic foundation is periodic and the wavelength Lw = 2π/β where $\beta = (k/4EI)^{0.25}$
This is a useful parameter when discretising the beam. A minimum element length of L/24 will provide a reasonable solution. If loading is concentrated as in this example a smaller element length may be more suitable in the vicinity of the loading. The length of the model need not be any longer than 6/β to represent an infinite length foundation.
For this model β = 0.2828 and L = 22.214m
Using these parameters as guidance the element length will be 0.45 (approx. Lw/48) and length of the model 21.6m. The model will be extended to 30m for demonstration purposes.
The stiffness of the foundation spring = 1.8E6 N/m.

The refence solution values are shown in parentheses.

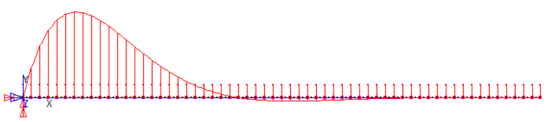
Case	End Displacement mm	Max Shear Force kN	Max Moment kNm
1 P + M	10.08 (18.14)	83.72	184.3
2 P	14.11 (14.14)	87.3 (100) f(ele len)	113.1 (114)
3 M	3.97 (4.00)	18.14 (18.24)	100(100)



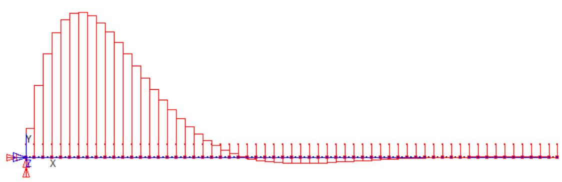
Moment Distribution for Case 1



Shear Distribution for Case 1



Moment Distribution for Case 2 - Shear Only

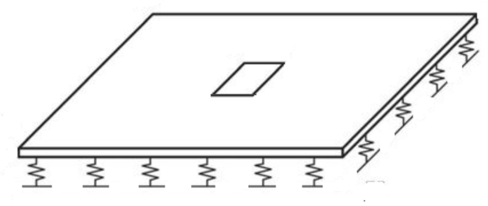


Shear Distribution for Case 3 - Moment Only

Example 1.21 Plate on an Elastic Foundation – Shell Elements

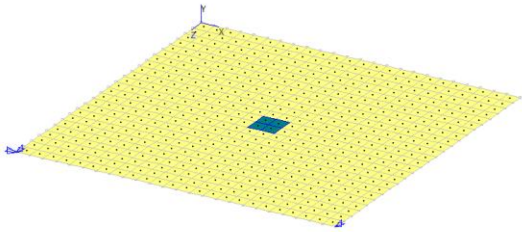
Model: **PlateWinkler**

This is an example of square plate supported on an elastic foundation (Winkler) Foundation. A weightless plate has a load distributed over a small square area in the centre of the plate.



Length = Width = 6m   Thickness = 250mm
E = 25GPa; Poiss = 0.25
Foundation Modulus K = 4000kN/m^3

Central UDL = 2000 kN/m^2
Loaded Area 500mm x 500mm square

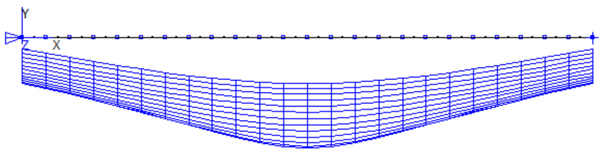


The plate is modelled using a 24 x 24 mesh of Type 50 Shell elements.

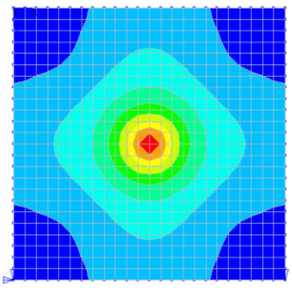
Loadin was applied as an element face pressure on 4 central elements.

Reference Solution: An ANSYS model using SHELL 63 elements which are similar to Type 50 (Kirchhoff Plate theory) was used. The ANSYS SHELL 63 elements were used because this element has a foundation stiffness capability.

The refence solution values are shown in parentheses.

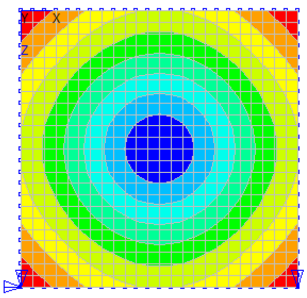


Maximum Deflection 6.269mm (6.264)



Maximum Von-Mises Equivalent Stress = 12.04 MPa (11.8)

[8 Node Type 51 model 12.4MPa]



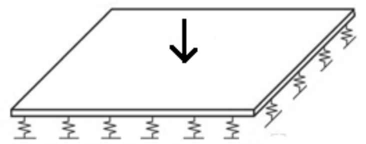
Foundation Bearing Pressure Max = 25.1kN/m2 (n/a)
The reference solution software does not evaluate bearing stress but it can be inferred from the y displacements (K*6.264) i.e. 25.056 kN/m2

[8 Node Type 51 model 6.386 (includes shear displacement)]

Example 1.22 Plate on an Elastic Foundation – Shell Elements

Model: **PlateWinkler2**

This is an example of infinite plate supported on an elastic foundation (Winkler) Foundation. A weightless plate has a point load in the centre of the plate.



Length = Width = 300ins   Thickness = iins

E = 29ksi; Poiss = 0.3

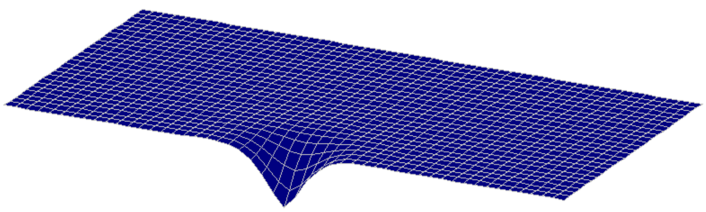
Foundation Modulus K = 800kips/ft^3

Central Point Load = 50Kips

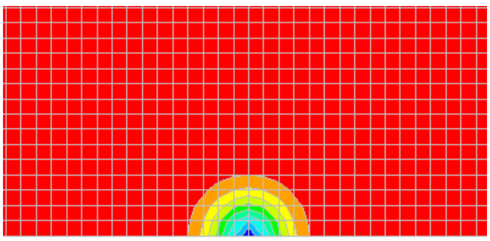
The plate is modelled using a 50 x 25 mesh of Type 50 Shell elements. Because of symmetry only half of the plate is modelled.

Reference Solution: A SAP2000 solution using the same mesh density.

Table 6-15. Center Displacement for Thin Plate Elements				
Model and Modulus	**Output Parameter**	**SAP2000**	**Independent**	**Percent Difference**
50 × 50 mesh k = 800 k/ft3	U_z at center of plate (in)	−0.1827	−0.1782	2.53



Deflection at centre of plate = 0.179ins



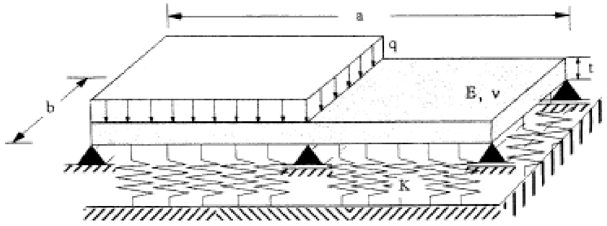
Max Foundation Bearing Pressure = 82.9 psi (n/a)

$800/12^3*0.179 = 82.87$

Example 1.23 Plate on a Tensionless Elastic Foundation – Shell Elements

Model: **PlateWinklerContact**

This is an example of plate supported on a tensionless elastic foundation (Winkler Foundation. The model is somewhat academic having very soft foundation. The foundation stiffness has very little influence on the plate displacement it is effectively a monitor for the shell displacement.



a = 10; b = 0.2; t = 0.4; E = 1E6; v = 0

K = 7.168

q = 1

The model uses 40 Type52 shell element and the foundation modulus K is defined as tensionless.

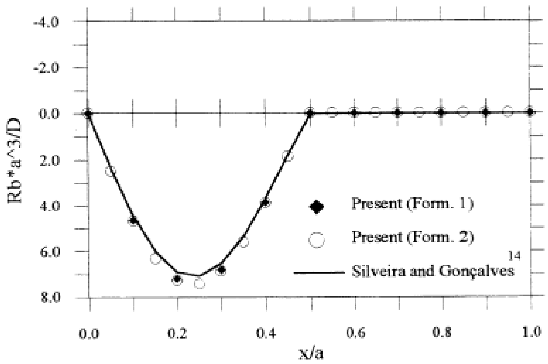
Reference Solution: "Behaviour of plates under contact constraints imposed by elastic foundations", R.A.M. Silviera, A.R.D. Silva<" and P.B. Gon§alves.

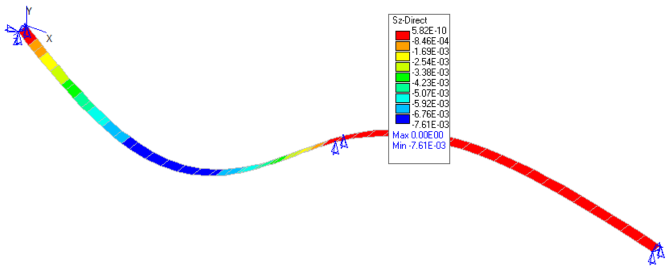
Case 1 is a tensionless Winkler foundation solution.

The maximum foundation contact stress = σ = -7.61E-3. Max deflection = 1.03E-3 (1.03E-3*7.168 = 7.6E-3).

The foundation contact stress is zero at the RHS.

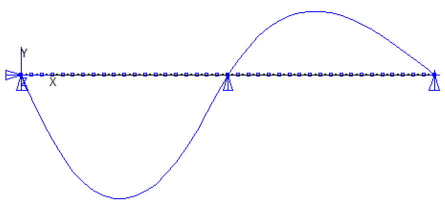
$R*b*a^3/D = σ*0.25*.2*ba^3/D = 0.761/D$ (Note that D is not defined other than being an elastic parameter).

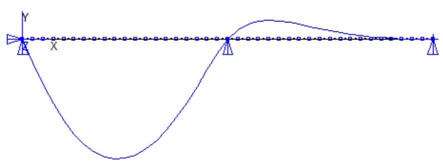




Case 2 is a normal Winkler foundation and indicates a tensile foundation stress of 3.3E-3. (0.461E-3*7.168=3.)

If the modulus is increase to 7.168E3 it has a significant effect on the displacements and the displacements are very different – see below.





Tensionless Winker Foundation RHS-Liftoff

Std Winkler Foundation

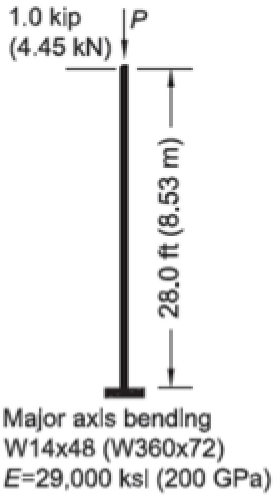
Example 2.1 AISC – P-Delta Analysis

Model: **P-Delta_2**

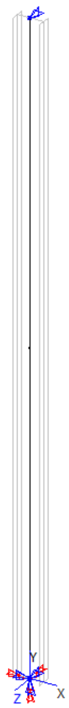
P-Δ Effects

Specifications for Structural Steel Buildings, ANSI/AISC 360-10, published by the American Institute of Steel Construction gives some benchmark problems to assess the effectiveness of software to account for the P-Δ and P-δ second order effects.

P-Δ Effects are the effects of loads acting on the displaced locations of joints in a structure.



Axial Force, P (kips)	0	100	150	200
M_{base} (kip-in.)	336 [336]	470 [469]	601 [598]	856 [848]
Δ_{tip} (in.)	0.907 [0.901]	1.34 [1.33]	1.77 [1.75]	2.60 [2.56]



The model is created in US units. It has one element. Because the loading produces only single curvature sway no mid span nodes are required.

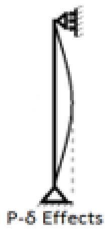
The model has 4 load cases to match the load cases in the example. The model is run using the 3-D Standard Solver with the P-Delta option active.

The results obtained compare almost exactly with those in the AISC table given above. Note that the Engineers Unit option is Kip-ft (Switch off to see kip-ins)

Axial Force	0	100	150	200
Moment	336	470.2	600.9	853.08
Defln	0.907	1.342	1.766	2.585

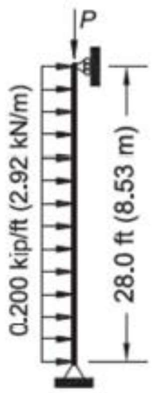
Example 2.2 AISC – P-Delta Analysis

Model: **P-Delta_1**



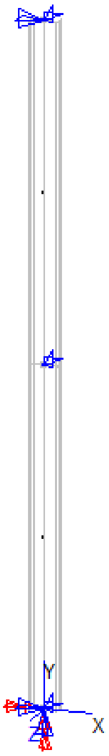
Specifications for Structural Steel Buildings, ANSI/AISC 360-10, published by the American Institute of Steel Construction gives some benchmark problems to assess the effectiveness of software to account for the P-Δ and P-δ second order effects.

The P-δ Effects are the effect of loads along the deflected shape of a member between joints. This is a local member effect.



Major axis bending
W14x48 (W360x72)
E=29,000 ksi (200 GPa)

Axial Force, P (kips)	0	150	300	450
M_{mid} (kip-in.)	235 [235]	270 [269]	316 [313]	380 [375]
Δ_{mid} (in.)	0.202 [0.197]	0.230 [0.224]	0.269 [0.261]	0.322 [0.311]



The model is created in US units. It has 2 elements. Because the loading produces single curvature bending between supports only one mid-span load is required.

The model has 4 load cases to match the load cases in the example. The model is run using the 3-D Standard Solver with the P-Delta option active.

The results obtained compare almost exactly with those in the AISC table given above. Note that the Engineers Unit option is Kip-ft (Switch off to see kip-ins)

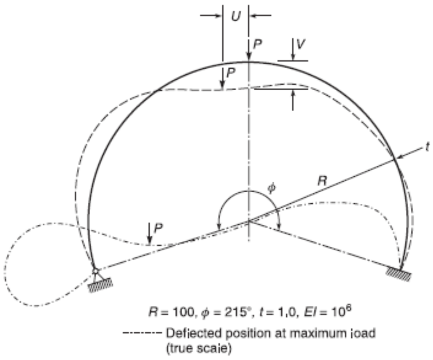
Axial Force	0	150	300	450
Moment	235.2	269.8	315.6	379.7
Defln	0.201	0.230	0.268	0.321

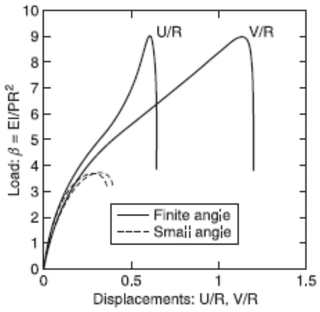
Example 2.3 Beams-Large Displacement

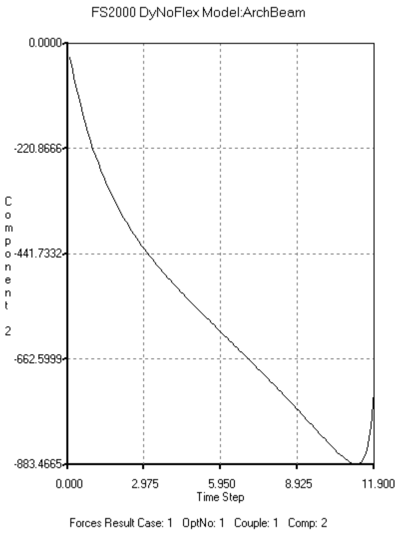
Model: ArchBeam

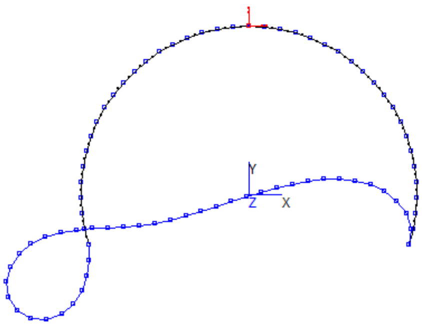
The is an example of a circular arch subjected to a concentrated load. The model uses 40 Type 6 beams to represent the arch and uses a DyNoFlex solution. The loading is applied using prescribed displacements. To enable the load to monitored the load is applied through a Type7 couple.

The reference solution: O C Zienkiewicz, "The Finite Element Method, Volume 2 Solid Mechanics, Page 372.

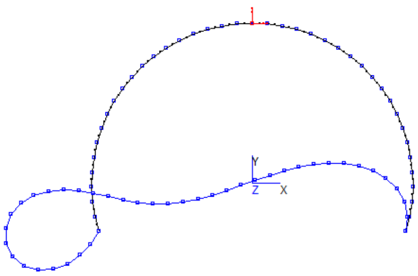








Case 1 Displacement (1.19) just beyond snap through(P=731).



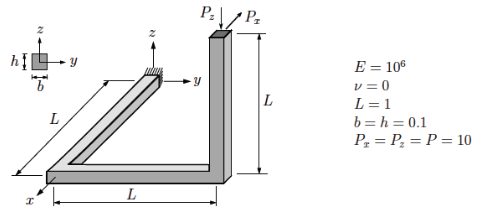
Case 2 Displacement (1.13) at Maximum Load (P=8.84)

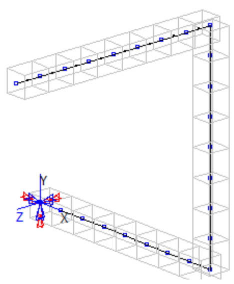
Example 2.4 Beams-Large Displacement

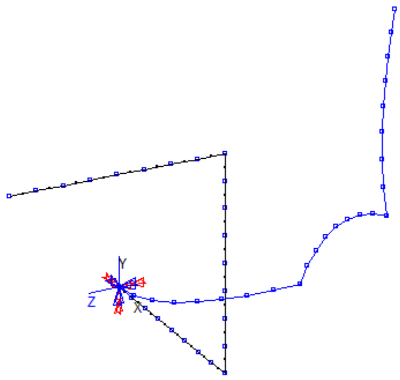
Model: SpaceBeam

This example is a 3 leg right angles cantilever. Two nodal loads applied at the tip. The model uses Type 6 beams and a FS-DyNoFlex solution.

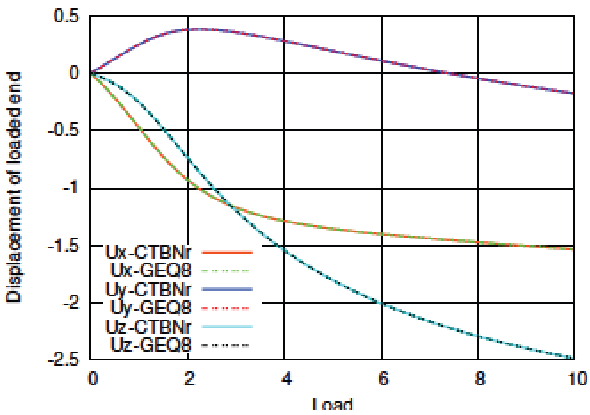
The reference solution is from:
National Conference on Computational Mechanics MekIT'17 B. Skallerud and H I Andersson (Eds)
A COMPARATIVE STUDY OF BEAM ELEMENT FORMULATIONS FOR NONLINEAR ANALYSIS: COROTATIONAL VS GEOMETRICALLY EXACT FORMULATIONS

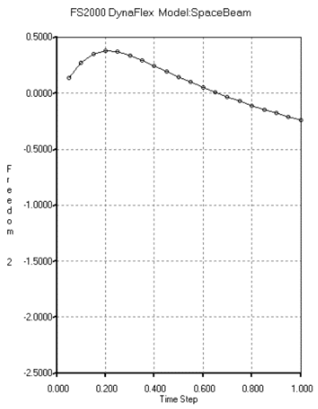


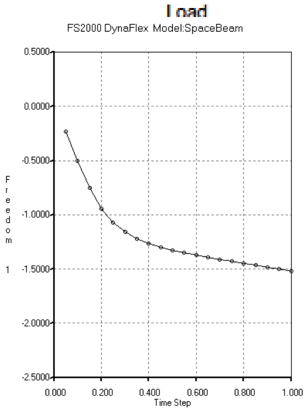


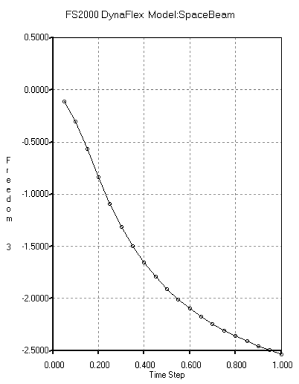


Result Case 1 True scale deflection









Example 2.5 Shell Element 1st & 2nd Order Solutions

Model: Plate_In-Plane_Loaded

This example is a rectangular simple supported plate subjected to normal and in-plane loading. Three solutions are obtained. A linear, small displacement P-Delta and a large displacement (updated geometry).

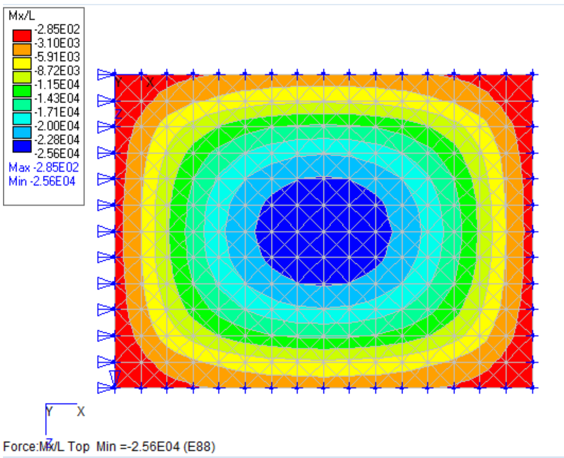
The model uses Type 52 shell elements and a 3-D Standard solution with and without the P-Delta option active. The Type 50 or 53 could have been used but the Type 52 Mindlin plate was used to match the reference solution. Case 3 is large displacement solution using FS-DyNoFlex – considered the most accurate solution method.

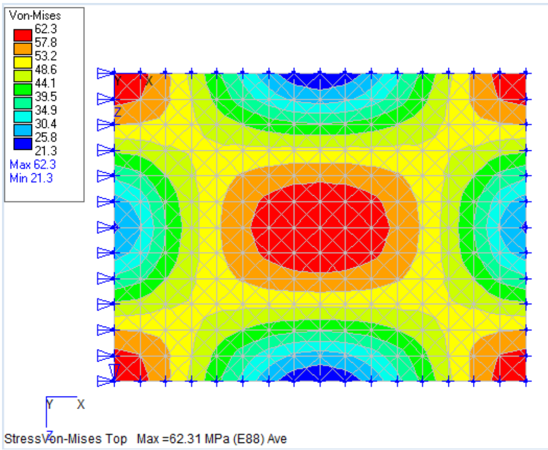
A SS support 8m x 6m x 50mm is subjected to an edge load of 1000kN/m and a normal face load of 10kN/m2. Plate material properties are: E=210GPa and PoissRatio=0.3

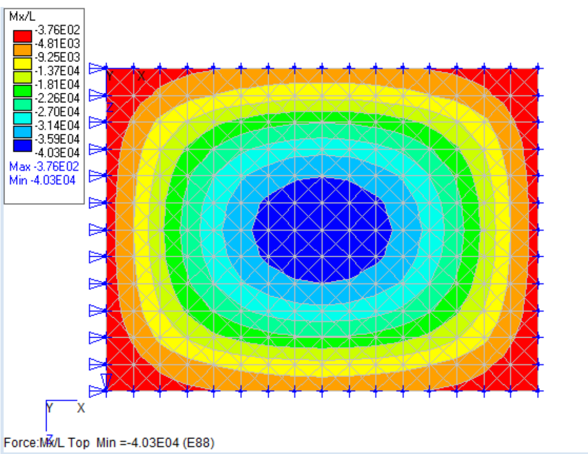
A reference solution is quoted from: StruSoft Verification Examples- shown in ( ). ANSYS solution used for Large displacement.

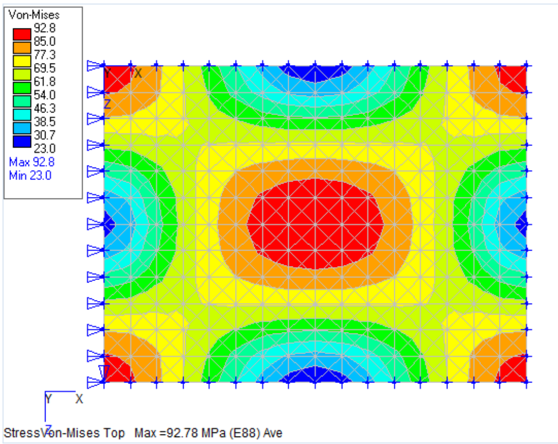
	Deflection	Mx	My	Mxy	VonMises
	Mm	kNm/m	kNm/m	kNm/m	MPa
Case1 – Linear	35.7(35.38)	25.6(26.62)	18.0(18.05)	13.9(13.68)	62.31
Case 2- P-Delta	55.2(54.69)	40.3(40.30)	28.4(28.50)	20.9(20.53)	92.78
Case 3 LargeDisp	46.1(*46.1)	32.3	22.3	18.8	81.2 (*82.2)

*ANSYS solution using Shell 181









Example 2.6 Portal Frame Buckling - Beam Element- Eigen Buckling

Model: **Portal_P, Portal_1, and Portal_2**

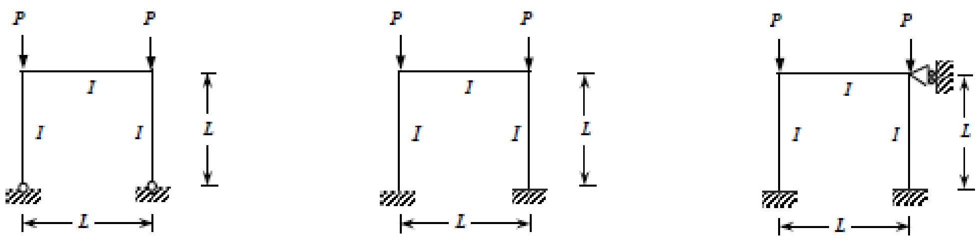
The model uses Type 0 beam elements. The 3-D Standard solution with the P-Delta option active produces the necessary result case for the Eigen buckling solution.

The model represents a 20mmSHS portal frame with L=1m. It is subjected to vertical point loads equal to the column Euler buckling load.

The model illustrates the importance of mid span node when undertaking 2nd order solutions.

The pinned frame bucking mode has only single curvature bending for the first mode i.e. sway frame. accordingly mid-span nodes are not required to capture the P-Δ effects.

The other frames are non-sway and have double curvature bending and mid-span nodes are required to capture the P-δ effects.



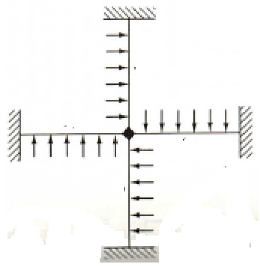
$$P_{cr} = 1.82\frac{EI}{L^2} \qquad \rho = \frac{P_{cr}}{\frac{\pi^2 EI}{L^2}} = 0.1844$$

$$P_{cr} = 7.34\frac{EI}{L^2} \qquad \rho = \frac{P_{cr}}{\frac{\pi^2 EI}{L^2}} = 0.7437$$

$$P_{cr} = 25.2\frac{EI}{L^2} \qquad \rho = \frac{P_{cr}}{\frac{\pi^2 EI}{L^2}} = 2.55329$$

Model	Portal_P	Portal_1	Portal_2
No mid span nodes	0.185	0.752	3.02
1 Mid span node	0.184	0.748	2.59
2 Mid span nodes	0.184	0.746	2.55

Example 2.6 Thermal Buckling- Beam Elements Eigen Buckling



Model: **ThermalBuckling**

In this example the cruciform frame is subjected to an increase in temperature. At what temperature does the buckle.

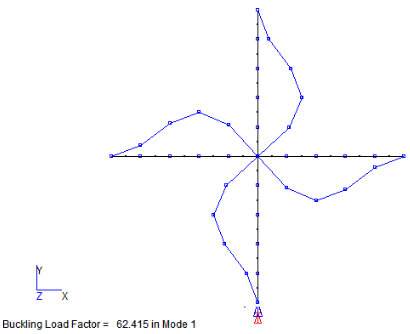
The properties of each member are:
A = 50mm^2; I = 220mm^4; L = 250mm; E = 70GPa

The model uses Type6 beam elements, but Type 0 could be used for the Eigen solution.

The 3-D Standard solution with the P-Delta option active produces the necessary result case for the Eigen buckling solution.

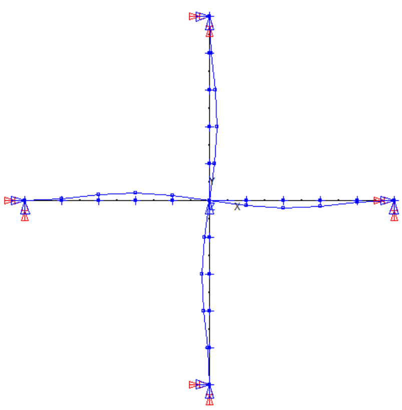
Reference Solution: Structural Analysis, RC Coates, MG Coutie, FG Kong, Second Edition 1980 Page 332.
The reference solution stated the buckling temperature to be 62C.

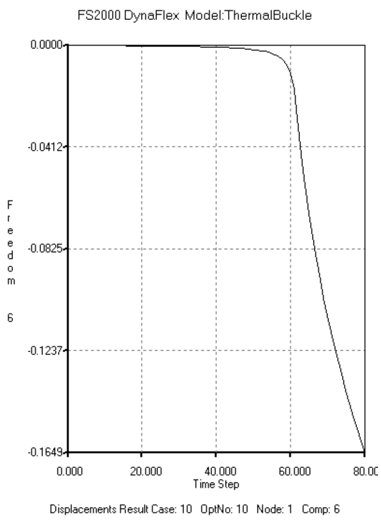
Load Case 1 has a temperature of 1C applied to all elements using property definition. This is solved using the 3-D Standard Solver with the P-Delta option active. The Eigen buckling solution give the following solution.



The resulting load factor indicates an Eigen buckling temperature of 62.415C.

A DyNoFlex large displacements solution as also undertaken. A small arbitrary UDL was initially applied to give a small displacement. The temperature was then ramped up to 80C. The following true scale deflection plot and time history plot of the centre rotation were obtained. Clearly showing a sharp increase in the vicinity of 62C.





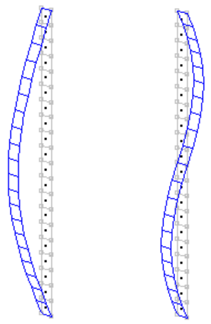
Example 2.7 Shell element- Euler column buckling.

Model: **ShellColumn**

This example is a pinned column subjected to an axial compressive load. Two solutions are used. One is linear Eigen Buckling solution and the other is a Large Displacement solution using FS-DyNoFlex. The model uses Type 53 shell elements and a 3-D Standard solution with and without the P-Delta option active.

The column is 5m load and has a 100mm x 10mm rectangular cross sections. The is subjected to a compressive load of 10kN and a 1% disturbing mid-span lateral load. Plate material properties are E=203GPa and PoissRatio=0.3.

Reference Solution: The theoretical Euler buckling load Pe = $\pi^2 EI/L^2$ = 1.35 kN.
Case 1 Linear Buckling Modes

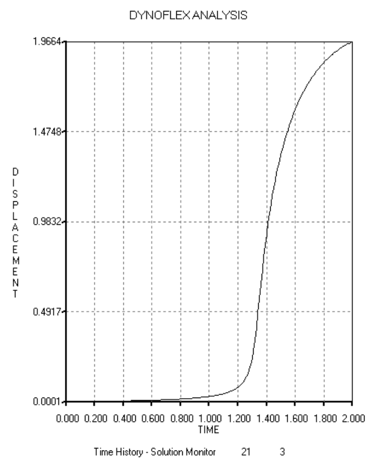


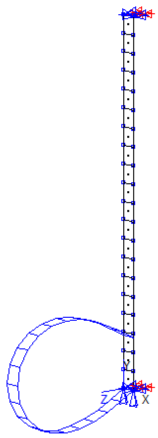
The Eigen solution gave the first 4 buckling mode.

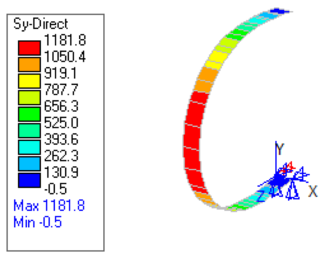
Mode	Load Factor
1	1.34956
2	5.40684
3	12.1973
4	21.7626

Case 2 Large Displacement - DyNoFlex Solution produced the following for a load factor of 2.
Case 3 Using prescribed displacement (top) to laterally deform the column to state .

The plot shows the lateral displacement. Cases 2 & 3 gives similar results.







Mid Span Bending Stress = 6*1E3*0.1*0.1966/0.1/0.01^2 = 1180 MPa

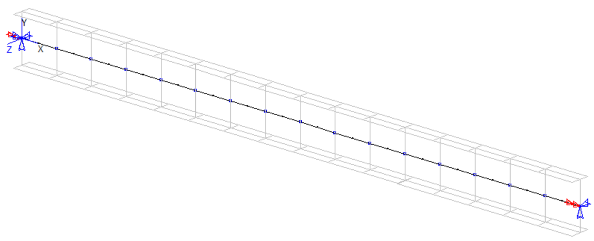
A model called **BeamColumn** uses Type6 beam elements for same configuration.

Example 2.8 AISC – Beam Twist Flexural Buckling

Model: **ASIC_Appendix_1_CA11**

This benchmark example from AISC 360-22 demonstrates that twisting of beams under bending and axial action can be included. In addition to small displacement P-δ effects, a large displacement (geometry updating) is also required.

The model uses Type 6 beam elements and a DyNoFlex solution with the P-Delta and Large Disp options active.



To capture the twisting effects several mid-span nodes are required.

For purely flexure P-δ effects only 2 or 3 are required for double curvature bending and 1 for single curvature bending.

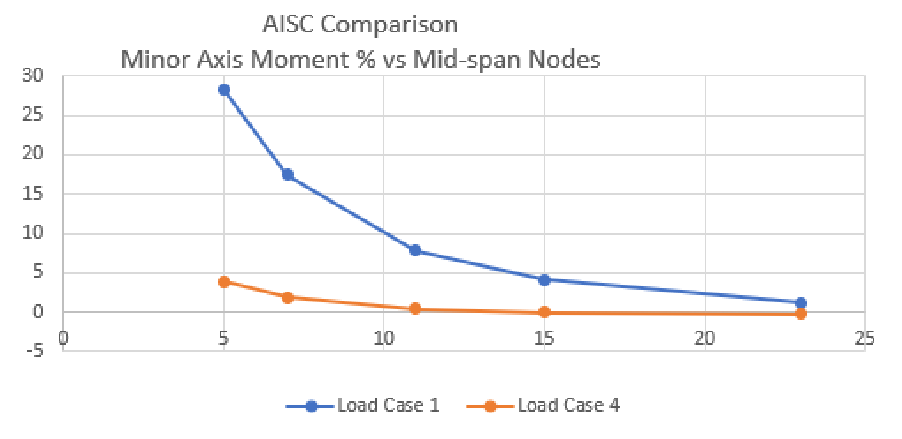
The table below shows the comparisons between the AISC results for their (b) configuration (Cw=0) and the FS2000 results.

*** NODAL DISPLACEMENTS ***

Node	RC	Tx	Ty	Tz	Rx	Ry	Rz
		ins	ins	ins	Rad	Rad	Rad
8-G1	1	-0.009	-0.686	0.932	0.10499	0.00000	0.00000
			-0.694	0.967	0.1078		
8-G1	2	-0.028	-0.520	0.927	0.07830	0.00000	0.00000
			-0.524	0.951	0.0790		
8-G1	3	-0.040	-0.340	0.818	0.04747	0.00000	0.00000
			-0.342	0.833	0.04710		
8-G1	4	-0.061	-0.200	1.378	0.03551	0.00000	0.00000
			-0.201	1.397	0.03580		

*** ELEMENT FORCES AND MOMENTS ***

Elem	Node	RC	Fx	Fy	Fz	Mx	My	Mz
			kip	kip	kip	kip-ft	kip-ft	kip-ft
1	8-G1	1	0.00	2.49	-0.26	-0.35	20.70	198.90
							21.5	198.80
1	8-G1	2	-75.00	1.91	-0.28	-0.25	19.03	152.22
							19.5	152.17
1	8-G1	3	-125.00	1.31	-0.24	-0.15	15.79	102.93
							16.0	102.92
1	8-G1	4	-175.00	0.69	-0.41	-0.11	25.46	52.07
							25.75	52.00



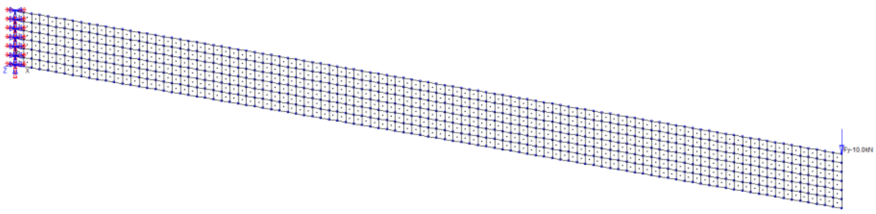
This shows the importance of mid-span nodes for this type of solution (torsional-lateral) especially when axial loading dominates.

Example 2.9 Plate Cantilever Flexural Buckling

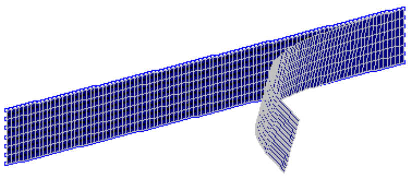
Model: **PlateCant_Buckle**

The example uses Type 53 shell elements to model a plate cantilever. The cantilever becomes unstable due to lateral-torsional bucking. The cantilever is a steel plate 10m long with a depth and thickness of 438mm and 40mm respectively. A point load of 10 kN is applied at the tip.

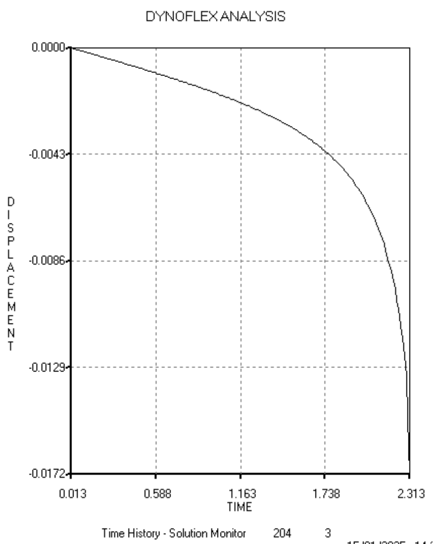
Reference Solution: Roark. The this predicts a critical tip load of 23.3kN for a load applied at the top of the tip.



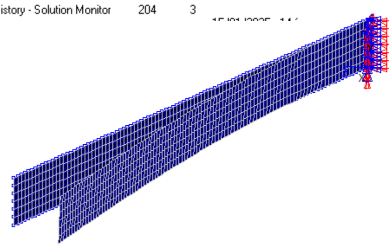
Two solutions have been undertaken. A linear Eigen buckling solution and a non-linear DyNoFlex large displacement solution. The DyNoFlex load case included a 1% lateral disturbing load.

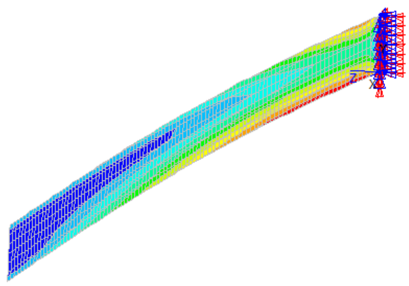


The Eigen solution predicted a load factor of 2.413 for the first buckling mode. The compare to 2.33 from the reference solution.



This plot shows the lateral displacement at the point of load application. The maximum load factor is 2.313. This clearly show that the onset of instability is within the region predicted by the reference.

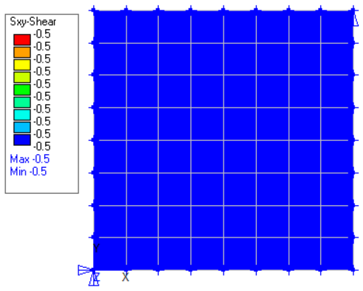




Example 2.10 Plate Buckling due to pure Shear

Model: **Shear_Plate**

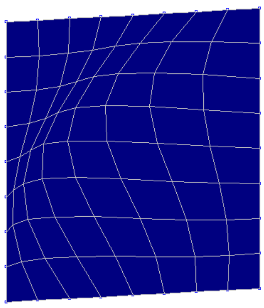
The example uses Type 53 shell elements to model a SS shear panel under the action of pure shear. The panel becomes unstable due to shear bucking. The panel is a steel plate 1m square plate with a thickness of 2mm. The plate is subjected to a constant shear load applied as a uniform edge load 1kN/m.



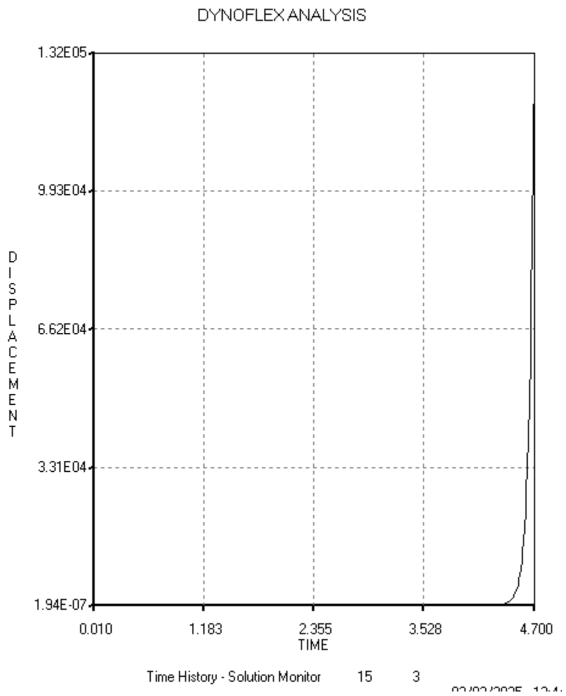
Reference Solution: Roark. The this predicts a critical shear stress of 2.384MPa. (Load Factor = 4.768)

The 1 kN/m edge load produce a uniform shear stress of 0.5MPa

Two buckling solutions have been undertaken. A linear Eigen buckling solution and a non-linear DyNoFlex large displacement solution. The DyNoFlex load case included a lateral disturbing load which produces a centre deflection of 2.27mm.

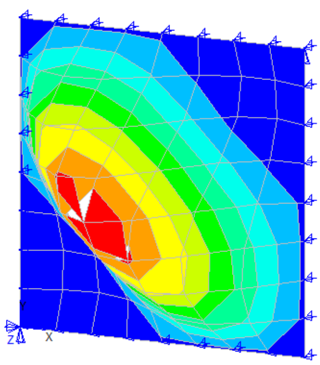


The Eigen solution predicted a first mode buckling stress of 5.523*0.5 = 2.766MPa (2.384).



The DyNoFlex solution indicated a buckling limit if just prior to the reference solution's 4.768.

The buckling mode was similar the Eigen solution.

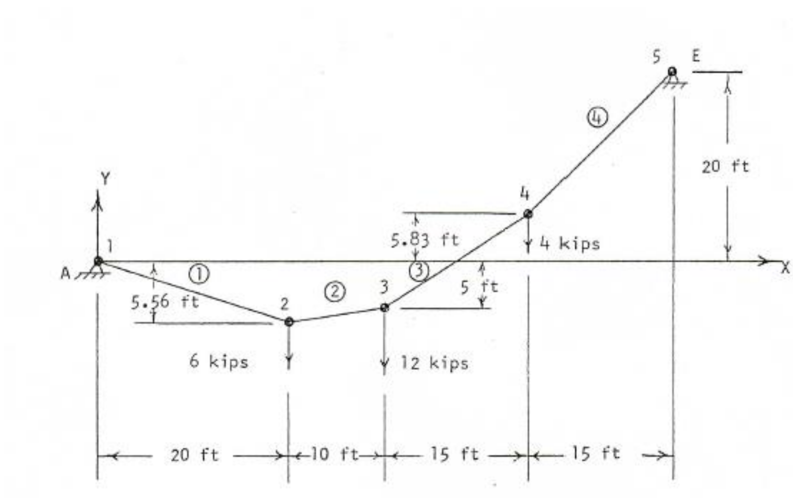


Example 3.1 Cable Supporting Hanging Loads – P-Delta – Large Displacement

Model: **CableLoads**

This model represents a cable carrying three vertical loads. The solution will show that in the initial position the cable is in equilibrium.
Reference Solution: Vector Mech for Engineers, Beer and Johnson, Page 260, Prob 7.8 (ANSY exp).

Result Case 1 is a P-Delta using the 3-D Standard Solver
Results Case 3 is a P-Delta using the DyNoFlex Solver (initial strain required for first iteration).



Both solutions give the following.

Reactions

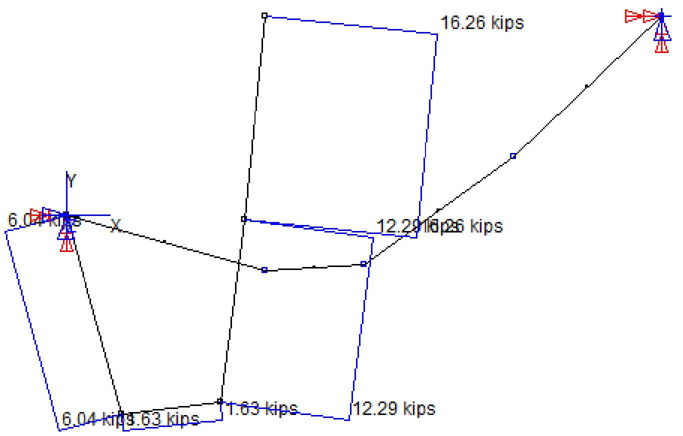
Va = 5 kips   Ve = 17 kips
Ha = -17.99   He = 17.99 kips

Tensions

T1 = 18.67 kips
T4 = 24.75 kips

Results Cases 10 and 11 are P-Delta + Large Displacement (Geometry Updating) using a DyNoFlex static time history solution.

In Result Case 10 the support at E is move 40 ft towards A.



Reactions

Va = 5.82 kips   Ve = 16.18 kips
Ha = -1.62   He = 1.62 kips

Tensions

T1 = 6.04 kips
T4 = 16.26 kips

In Result Case 11 the support E is moves 40 ft towards A and then back 40 ft to its original position. This results in the same loading as given above for Case 1 or 3.

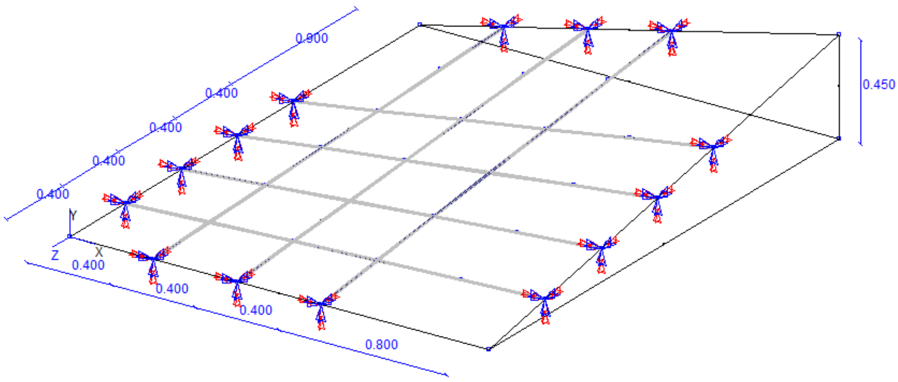
Example 3.2 Cable Net Supporting Hanging Loads – P-Delta – Large Displacement

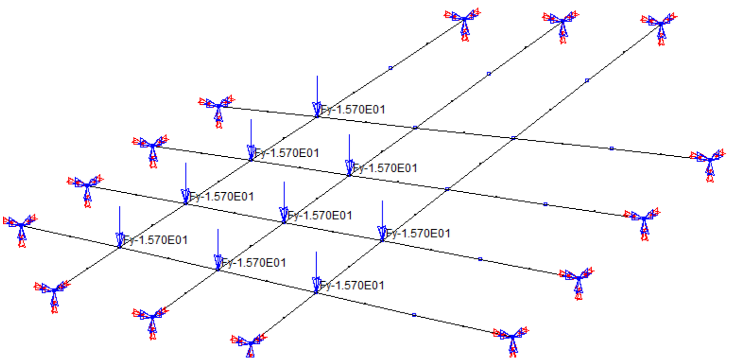
Model: **ParabolicNet**

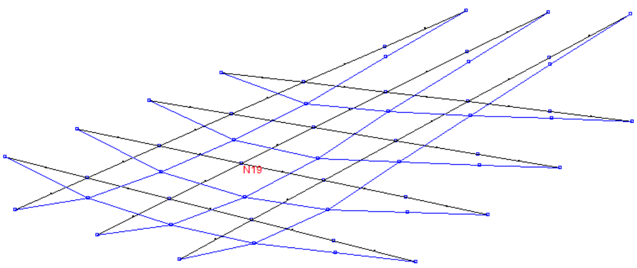
This model represents a pre-tensioned cable net subjected to a series of 15.7N concentrated loads and establishes the displacements due to these loads. The cable net is pre-tensioned to 200N. The cable gravitational load is 195 N/m. E = 128.3 KN/m2, csa = 0.785 mm2. The preload and the concentrated nodal forces dominate. This enables an accurate solution to be obtained using very few spar elements.
Reference Solution: Nonlinear Analysis of Cable Structures under General Loadings, Adab, Shooshtari et al, Finite Elements in Analysis and Design 73 (2103) 11-19.

The DyNoFlex solution used P-Delta and Large Displacement options (initial strain required for first iteration).

Result Case is the Wg + Preload
Result Case 2 is Wg + Preload + Concentrated Loads
Result Case 10 is a post-processes combination case: Case 2 – Case 1 to give the displacement due to the point loads.







Vertical Displacement at Node 19 = 33.34mm

Reference Solution.

Experiment 33.6mm
Analytical* 33.8, 34.00, 34.16 and 33.94

*Different investigators were quoted

Example 3.3 Beams-Large Displacement-Tension Stiffening

Model: **TrapezeWire**

This example is a tensioned cable subjected to a concentrated mid-span load. The LHS is fixed and the RHS simply supported

Length==15m; E Value =90GN ;Diameter= 10mm ;Coeff of Thermal Exp =1.1E-5 .
Tension= 10kN
Load in centre span 850N.

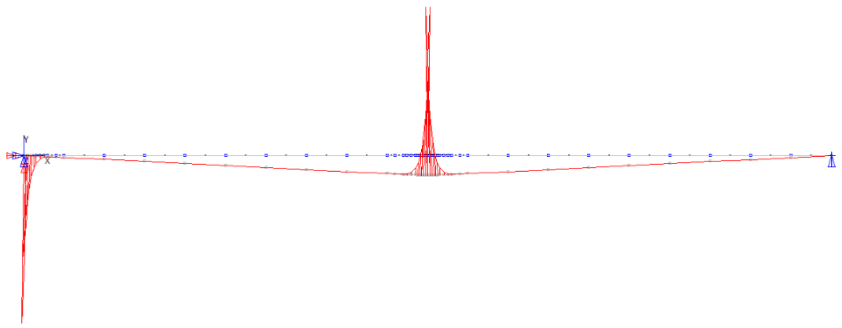
The objective to establish the bending stress at the point of load application. The cable proportions are such that the load is supported by cable tension with bending stiffness being virtually zero. However, at the point of application load there will be local bending, and this could be significant with respect to cyclic loading (fatigue).

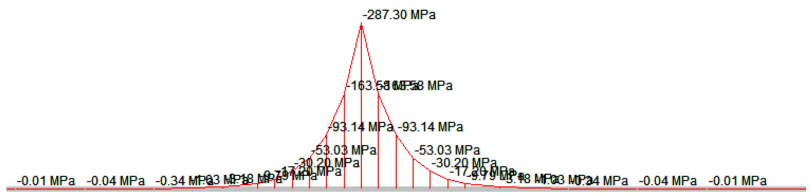
The model used Tyep16 beams (P-δ effects based on stability functions). This model the require mesh refinement in the vicinity of the load and LHS support (mesh sensitivity checks).

Case 3 used the 3-D Standard with P-Delta active and cable tension by force definition.
Case 7 used the 3-D Standard with P-Delta active and cable tension by thermal strain definition.
Case 100 used the 3-D Non-linear with P-Delta active and cable tension by force definition.

Reference Solution. The bending stress in a fixed ended cable at an angle ϑ and with tension F, can be evaluated using the following expression (French Stay Cable Standard).

Stress = ϑ.r(E.F/I)**0.5 = 287.73MPa (ϑ obtained from simple statics or model ϑ = W/2F)



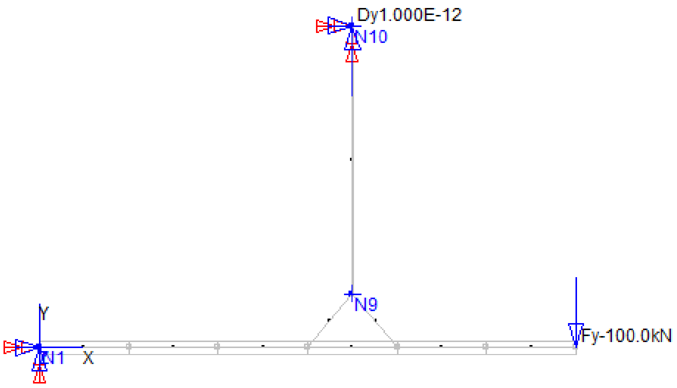


Bending Stress at load point = 287.3 MPa

Case 7 uses DyNoFlex (Large Disp +PD) Cable tension due to deflected shape. Tension less, deflection higher and bending stress higher (Bending Stress 311MPa) – considered more exact.

Example 3.4 Beam Lift using a Pulley Element – Dynamic - P-Delta – Large Displacement.

Model: **Pulley_BeamLift**

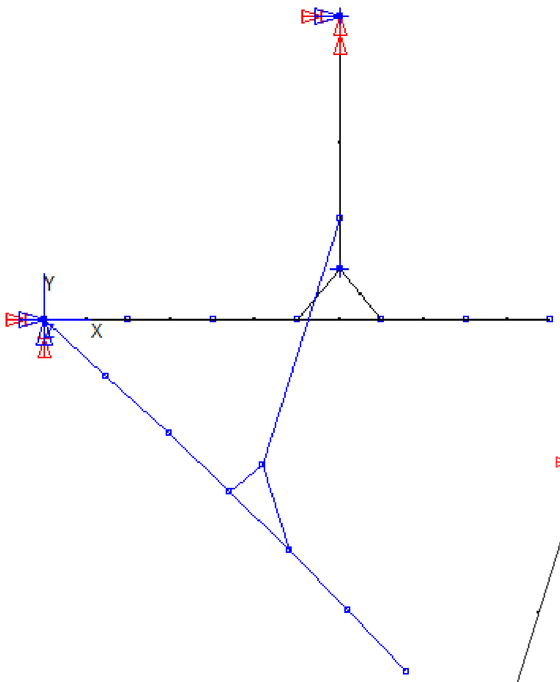


This solution traces the displacement of a 10m I beam, pinned at one (N1) end and supported near mid span using an arrangement with a running pulley.
The pulley is located at N9.
The only load is a 100 kN applied at the free end of the beam.
A vertical prescribed displacement applied at N10 moves the moves the pulley vertically downward by 10m.

Reference Solution: Validated using Statics (Linear Static solution based on final configuration)).

A pulley by its nature is a mechanism and requires a dynamic solution to establish an equilibrium state. Accordingly, the solution uses a DyNoFlex dynamic time history solution.

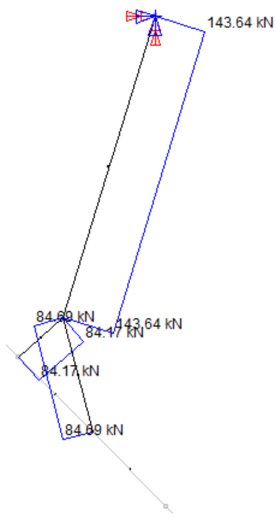
Result Case 2 is the initial state when the 100 kN is applied and the N10 is in its initial position.
Result Case 100 is the final state after N10 is moved vertically downward 10m.



This final state displacement plot shows the relative movement of the pulley.

The initial and final tensions in the pulley bridle are 106.63kN and 84.33 kN.

The initial and final tensions in the pulley lifting cable are 172.15kN and 143.71 kN.



The deformed geometry from the final state was used to create a linear model (Pully_Beam_Check.MOD). This gave the following tensions.

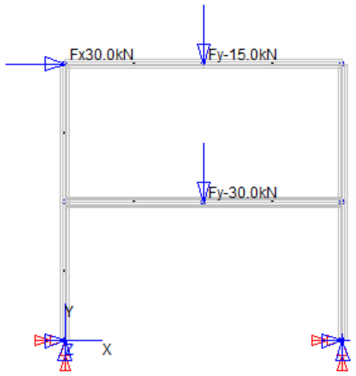
Pulley lifting cable tension 143.64kN.
Average pulley bridle tension 84.43 kN.

Example 4.1 Plastic Collapse of a Two Storey Frame – Frame Plasticity

Model: **PL_Frame_1**

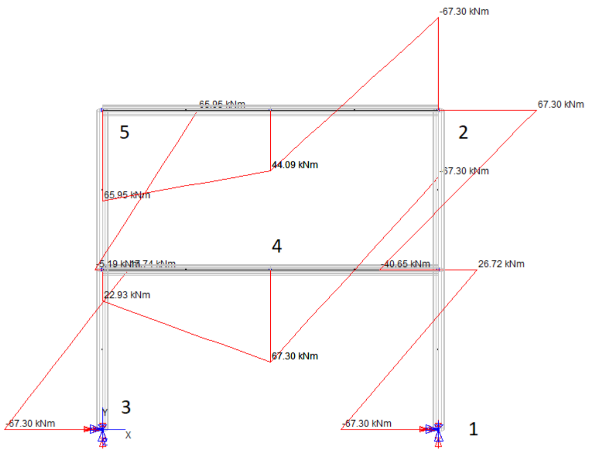
The plastic collapse of this is based on the theory of "Perfect Plasticity". With perfect plasticity the frame member nodal joints behave elastically at moment load levels below the plastic moment limit. Above this limit the member moment remains constant at that limit and the excess moment is either distributed to connected elements, if possible or the collapse occurs. In FS2000 this called Frame Plasticity.

This model is a based on Example 14.7-2 from the book Structural Analysis (2nd Ed), Coates, Coutie & Kong. In the book a load factor of 2 is found to produce plastic collapse of the structure.



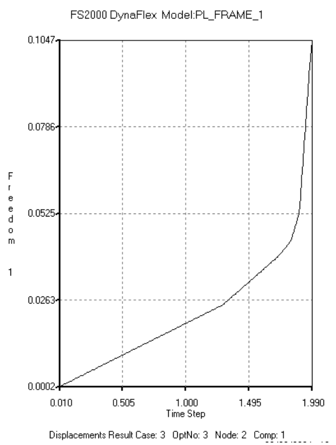
Plastic Moment Limit = 67.29 kNm

The model has a batch file that will run 8 Combination Cases using the **3-D Non-Linear Solver** with LF = 1 to 2(1.99).



The sequence of the formation of the plastic hinges is indicated. When the 5th hinge is formed the frame will collapse.

Note that plastic interaction is not active. If it were, the presence of axial load would reduce the moment capacity. In this example Hinge 1 would reduce to 66, only very slightly.



This displacement plot obtained from a DyNoFlex solution shows the horizontal displacement of the frame as the loading is stepped up to the collapse loads.

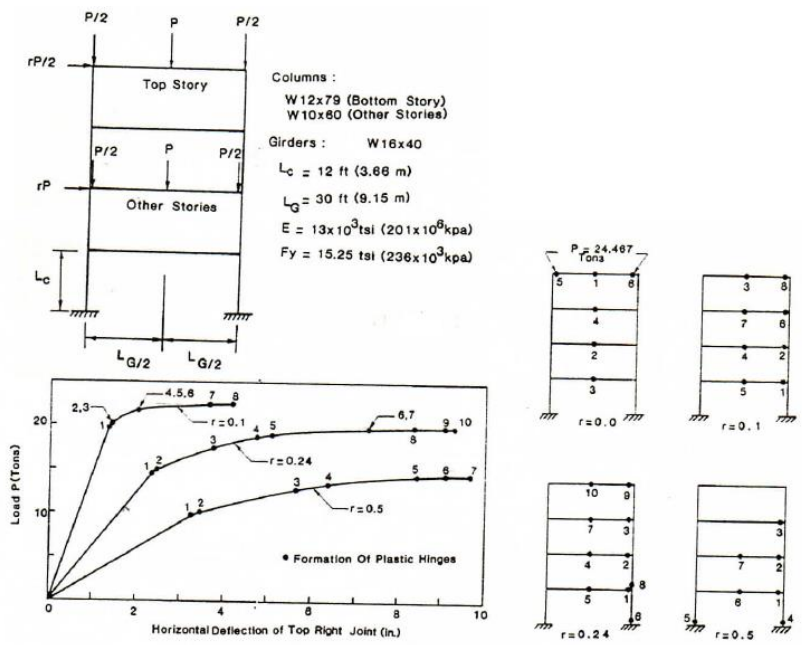
4 Hinges are shown and when the 5 is formed a mechanism is formed and the solution fails.

Example 4.2 Plastic Collapse of a Four Storey Frame – Elastic-Plastic – Large Displacement

Model: **PL_Frame_2**

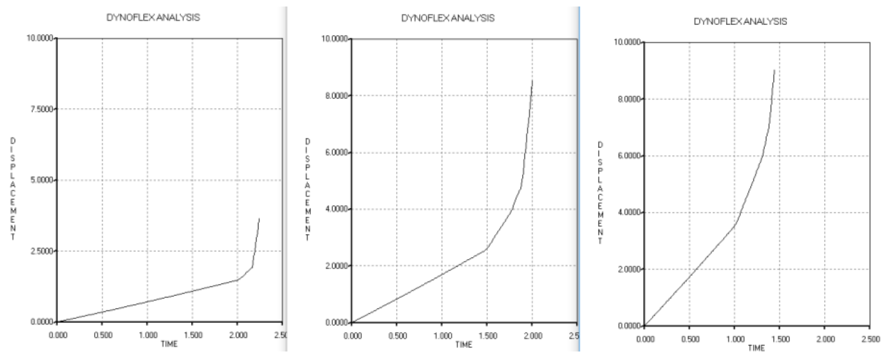
The plastic collapse of this is based on the theory of “Perfect Plasticity” combined with a large displacement solution.

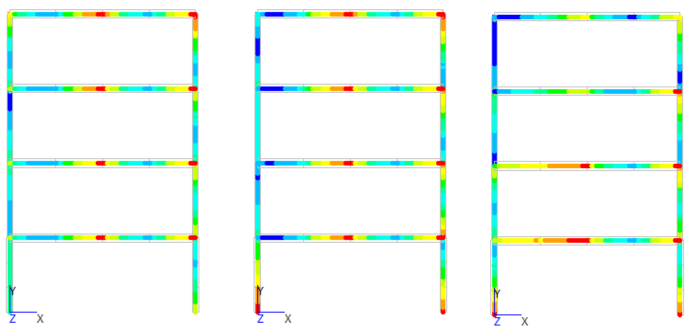
Reference Solution: Large Deformation Analysis of Elastic-Plastic Frames, Aslam Kassimali, Journal of Structural Engineering, Vol 109 , No 8, 1983, ASCE.



FS2000 Limiting Conditions

Case r	Forces	Lat-Deflection
	Tons	ins
1-0	24.3	n/a
10-0.1	22.4	3.66
11-0.24	20(20.1)	8.54(9.26)
12-0.5	14.4	9.05



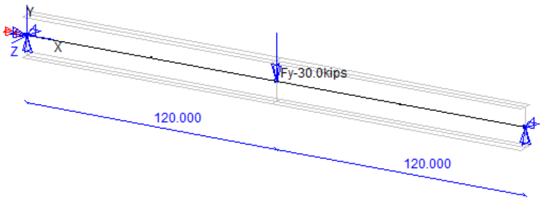


r = 0.1   r = 0.24   r = 0.5

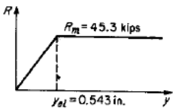
Von-Mises plots indicate the location of the plastic hinges.

Example 4.3 Dynamic Response of Plastic Beam – Dynamic - Frame Plasticity

Model: **DynPlastBeam**

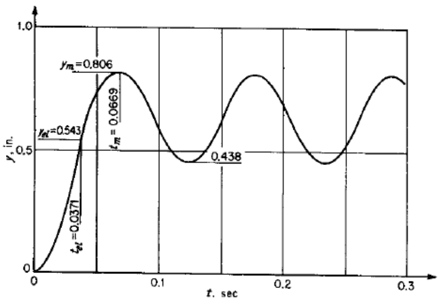


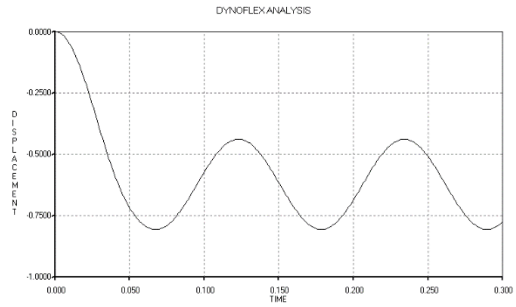
A simply supported undamped beam is subjected to a suddenly applied load of 30 kips.
The beam mass is represented by a concentrated weight of 10 kips located to the load point.



Beam Properties: E=30E3ksi; YST = 30ksi; I = 854.5ins^4 ; Depth=18ins;Z_P = 90.6ins^3.
The onset of perfect plasticity occurs a deflection of 0.5432ins due to a load of 45.3kips.
Reference Solution: J. M. Biggs, *Introduction to Structural Dynamics*, McGraw-Hill Book Co., Inc., New York, NY, 1964, pg. 69, article 2.7.

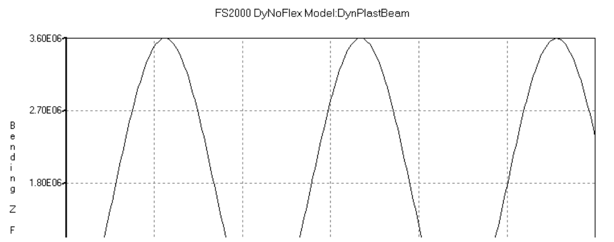
Note that this model cannot be run under load control because the model becomes a mechanism when the plastic hinge is formed (kx=0). In a dynamic solution the mass provides stability (ma + kx = 0)

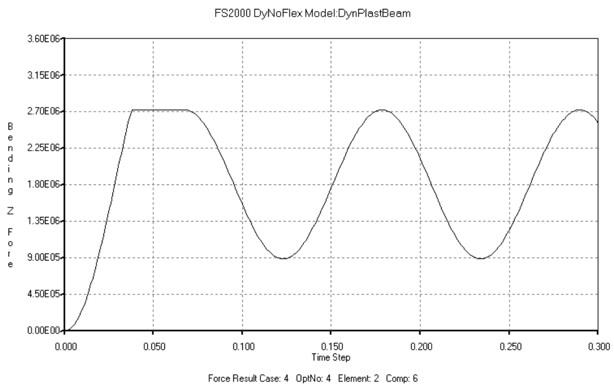




Displacements

At t =0 .371	u = 0.534 (0.543)	Reference solution shown in parentheses.
At t = 0.669	u = 0.808 (0.806)	
At t = 0.122	u = 0.439 (0.338)	





Moments

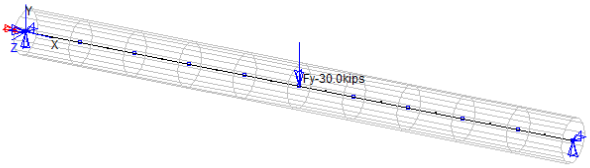
Linear Solution M_{MAX} = 3600 kip-ins

DAF = 2 for a suddenly applied load on an elastic beam, M_{STAT} 1800 kip-ins

Plastic Solution M_{MAX} = 2718 kip-ins (M_P)

Example 4.4 Dynamis Response of Plastic Pipe – Dynamic – Strain Plasticity

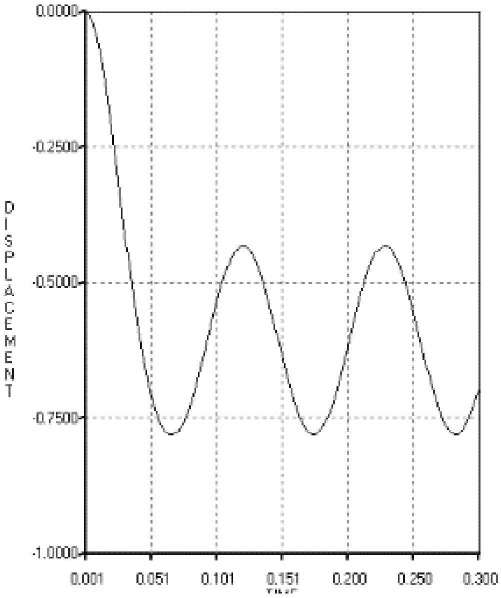
Model: **DynPlastPipe**



This is essentially the same as the previous beam example, but the plasticity is based on a defined bi-linear stress-strain curve.

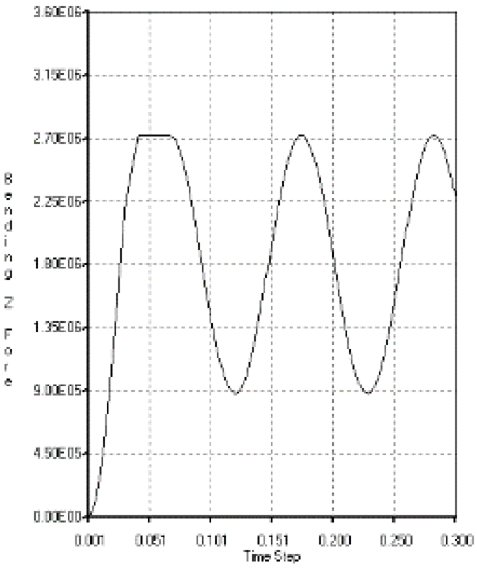
A simply supported undamped beam is subjected to a suddenly applied load of 30 kips. The beam mass is represented by a concentrated weight of 10 kips located to the load point.

Beam Properties: E=40.2E3ksi; YST = 30ksi; I = 630.7ins^4; OD=18ins; t=.289ins; Z_P = 90.6ins^3. Note the E value was adjusted to give the same EI value as the previous beam example.
The onset of perfect plasticity occurs a deflection of 0.5432ins due to a load of 45.3kips. This equates a strain of 0.0745% at 30ksi. A Von-mises yield function is assumed.



At t = 0.66 u = 0.7807 (0.806). This slightly less than the frame plasticity solution but can be expected because the frame solution assumes a concentrated hinge point.

Reference solution shown in parentheses.
At t =0 .371 u = 0.521 (0.543)
At t = 0.669 u = 0.781 (0.806)
At t = 0.122 u = 0.4392(0.338



Plastic Solution M_{MAX} = 2718 kip-ins (M_P)

The spread of plasticity across the section as the load reached the plasticity limit can also be observed in the moment plot.

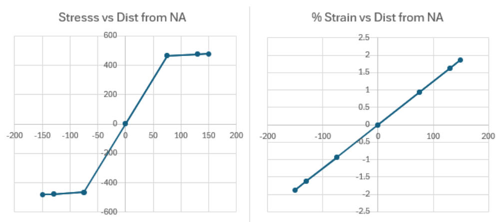
Example 4.5 Pipe Cantilever Bending –Strain Plasticity

Model: **PlastPipeMoment**

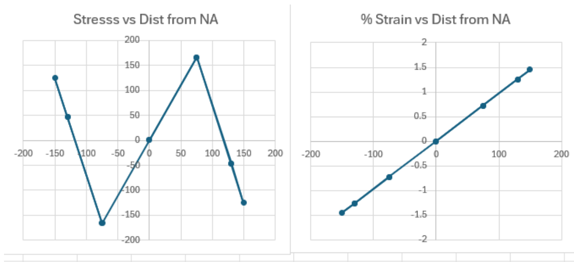
The model represents a cantilever, 22m long, subjected to a vertical load at the tip (Type6(7) beam). The load on the tip is gradually increased until the section becomes fully plastic, just above Mp. The loading is then removed leaving the cantilever permanently deformed and a residual stress state. Material strain hardening is present to prevent a mechanism being formed. A DyNoFlex (Material + Large Displacement) solution employed. Note that the plots values are indicative of the shape not his solution.

Beam Properties: E=207GPa; YST = 448MPa; OD=323.9mm; t=24.3mm; Z_P = 2.186E-3m^3; M_P=979.3kNm.Stress-Strain Curve: Ramberg-Osgood AlphaR=1.31 N=25.61. A Von-mises yield function is used.

Condition 1 F_{EQ} = 1.1Mp Tip Vertical Load = 44.51kN based 22m offset
Section Plastic at all stress points. Max Stress 484.38MPa; Max Axial Strain 2.3108%. Plastic Axial Strain 2.076% Deflections Y=4.972m X=0.631 (Elastic 2.955m at Mp).
M = 1046.47kNm [(22-.631)*1.1*44.51kN=1046.25kNm]



Condition 2 F = 0 and permanently deformed.
Max Residual Stress 167.3MPa; Outer Residual Stress 126.5MPa; Outer Residual Axial Strain 2.015%.
Moment= 0 Tip Y Deflection 1.864m (Elastic 0m)



FS2000's Moment-Curvature Utility can be used to evaluate stress strain histories (evaluation based only on static equilibrium using defined curvature). The results obtained below show excellent agreement with those from the above DyNoFlex solutions.

Condition 1 - Indicates a Stress of 485.57MPa for a strain of 2.31%. This occurs at a curvature of 0.1544 (strain = C.r_M =0.1544*0.1498=2.31%). The moment is 1047.6kNm.

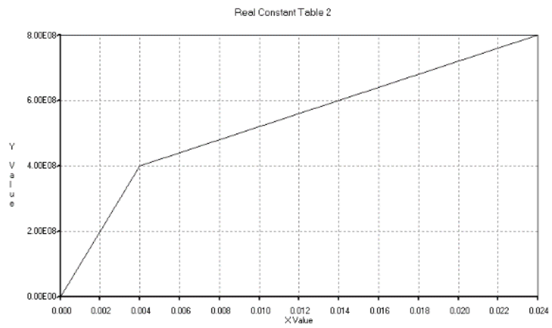
Condition 2 - A -ve curvature change of .01544 results in near zero moment 18kNm. This predicts an outer stress of 137 MPa and a corresponding strain of 2.012%.

The curvature at the two conditions can also be estimated from nodal displacements using the ETABLE routine which uses a quadratic curve fit interpolation near the support. A plot of ETable 3 for the two cases indicates reasonable agreement.

Example 4.6 Elasto-Plastic Analysis of an axially load bar.

Model: **PlasticRod**

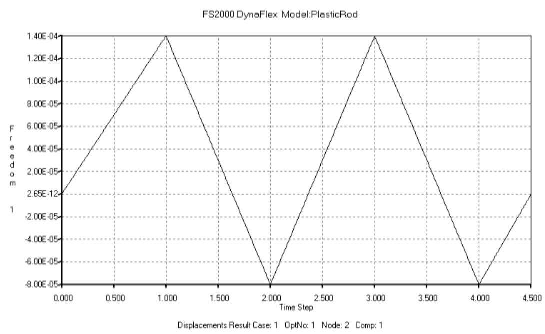
The model shows the non-linear material response of a bar subjected to a cyclic end load. The model uses a single Type 15 spar element. The loading in the element is due to prescribed end displacements.
Reference Solution: Trivial – Loading traces stress-strain curve.

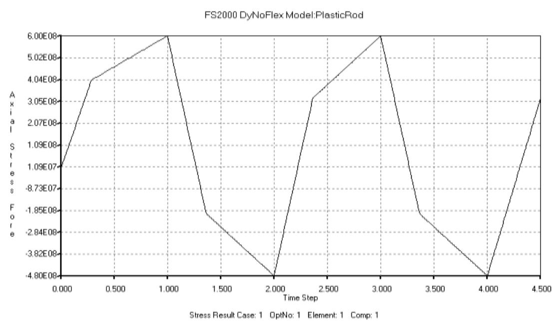


Geometric Properties: Length 10mm, CSA = 1mm^2
Material Properties: Bi-linear stress/strain curve. E = 1E11 N/m2, Ep = E/5, Yield = 400MPa

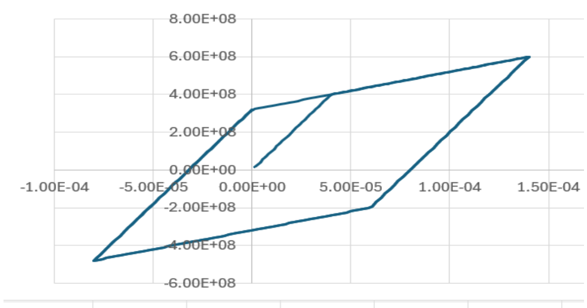
The model has two solutions one uses a kinematic memory model (GeomType22) and one uses an isotropic memory model (GeomType24).

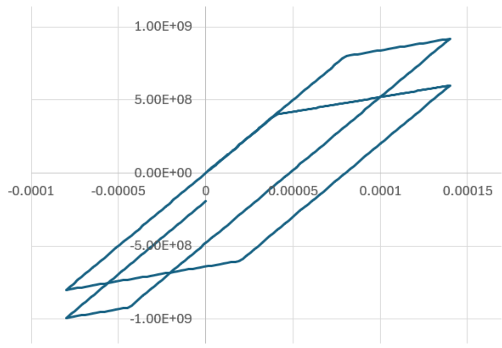
The solution uses a DyNoFlex time history solution to produce the following incremental plastic response curves.





End Displacement History





Axial Stress History (Kinematic)

Kinematic Response

Isotropic Response

The following were obtained using 3-D Nonlinear at specific load points.

Point	1	2	3	4	5
Strain (x10E5)	1.4	-0.8	1.4	-0.8	0
Stress (Kin) MPa	600	-480	600	-480	320
Stress (Iso) Mpa	600	-800	920	-992	-192

Example 4.7 Elasto-Plastic Analysis of Pressurised Pipe.

Model: **PipePressureAxial**

This model demonstrates axial and hoop interactions during axial load cycling above the VM plastic limits. Reference Solution: See next example – 3-D solid element subject to same loading.

The model comprises of a single pipe element Type6(7). For each loading history the pressure is held constant (+ve or -ve), and the axial load is cycled. The pipe (400mmOD, 5mmWall 10m long) material is Yield=448MPa, E=207GPa, Ep=0.56GPa. The cyclic material model of a Type6(7) element is a bi-linear kinematic memory model.

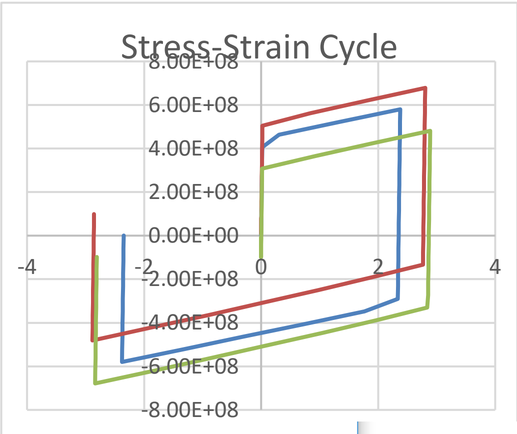
A pressure of 51.25Bar induces a hoop stress of 205MPa and an end cap stress of 98.67MPa. An axial force of 3.598MN induces an axial stress of 580MPa.

The batch file will run one complete reversed cycle for each of the loading (Time steps 1 to5).

The table below list the conditions at the end of Time Step 2.

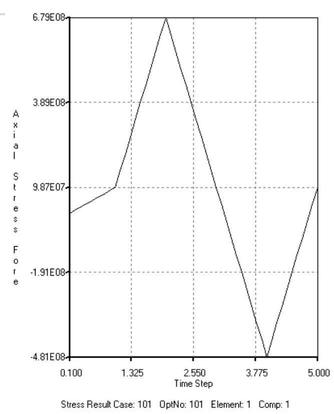
Case No	Hoop Stress MPa (52.25Bar)	Axial Stress MPa (Load=3.598MN)	Tot Wall Axial Strain %	Acc Plastic Strain %	True Wall Axial Stress MPa
101	+205	+580	+27.298	28.12	+678.4
102	0	+580	+23.772*	23.492	+580.0
103	+205	-580	-28.947	29.91	-481.04
104	-205	+580	+28.947	29.91	+481.04
105	-205	-580	-27.298	28.122	-678.4

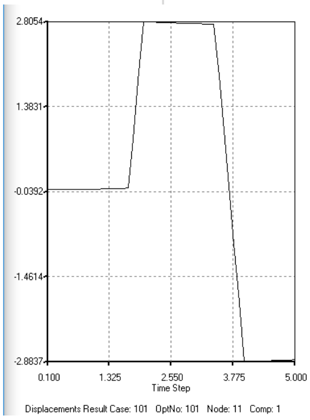
* Eff Plastic Strain = TotStrain – EffStress/E = 23.772 - 580/207E3*100 = 23.492%



This plot shows the true wall axial stress-strain cycle for Cases 101, 102 and 104.

For Case 102 The permanent plastic axial displacement is 2.351(23.49%). The corresponding hoop and radial strain would each be 0.5*23.349% = 11.75%.





This shows the time history for Case 101

At the end of the cycle:
Axial stress = hoop end cap = 98.73(98.8)MPa
Displacement = 2.859m
Acc Plastic Strain = 58.00%
Total Strain=28.87%

Example 4.8 Elasto-Plastic Analysis of 3-D Solid

Model: **PipePressure3D**

This model demonstrates 3-D plastic interactions of a Type 70 Solid element. The model a single 1m square brick element. The model is loading using the same loading as the previous plastic pipe example.

The X axis represents the pipe axial direction.	Pressure Endcap Stress = 98.67MPa
	Applied Axial Stress = 580Mpa
The Y axis represents the pipe hoop direct.	Pressure Hoop Stress = 205MPa
The Z axis represents the radial pipe direction	Zero – Plane Strain Condition

The model comprises of a single square brick element Type70. For each loading history the pressure is held constant (+ve or -ve), and the axial load is cycled. The material is 448MPa, E=207GPa, Ep=0.56GPa. The cyclic material model of a Type70 element is an isotropic memory model.

The table below list the conditions at the end of Time Step 2. Note that this is an isotropic memory model therefore cycling beyond this point will produce a different response to that of a kinematic pipe especially at these high strain levels with this tangent modulus.

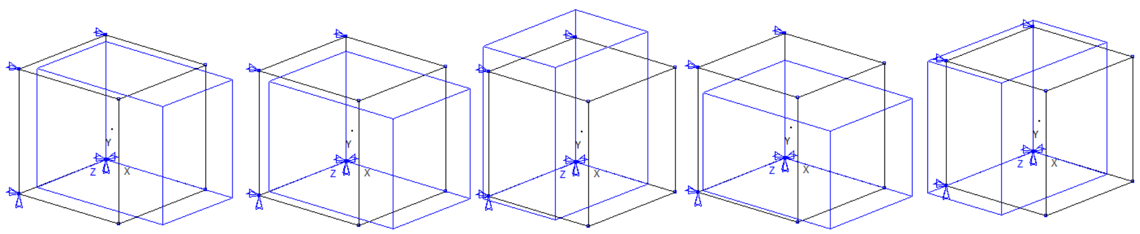
Case No	Hoop Stress MPa (52.25Bar)	Axial Stress MPa (Load=5.598MN)	Axial Strain %	Acc Plastic Strain %	True Wall Axial Stress MPa
101	+205	+580	+26.6	27.86	+678.7
102	0	+580	+23.77*	23.49	+580
103	+205	-580	-27.56	29.00	-481.3
104	-205	+580	+27.56	29.00	+481.3
105	-205	-580	-26.6	27.86	-678.7

* Eff Plastic Strain = TotStrain – EffStress/E = 23.77 - 580/207E3*100 = 23.49%

The above strains are obtained form the ST files produced in batch by ETABLE

Shown below are the displacement plots for different load conditions at Time Step 2. The displacements are predominately due to plastic flow. For Case 102 ΔX=23.77, ΔY=11.83 & ΔZ=11.83. This corresponds to an almost overall effective Poisson ratio of 0.5 which is to be expected for plastic flow. If Case 102 is cycled to Time Step 5, the load is completely removed, only the plastic strains remain and the deflections are ΔX=23.49, ΔY=11.74 & ΔZ=11.74.

101 102 103 104 105



The Res Plastic Strain in the above table compare favourably with those for the Type6(7) pipe considering the pipe plastic formulation has only one independent variable (x) whereas the solid element has three independent variables (x, y & z).

Example 4.9 Plastic Collapse of a Suspension Structure – Elastic-Plastic-Large Displacement

Model: **SuspensionStruct**

The plastic collapse of this is based on the theory of "Perfect Plasticity" for beam action and non-linear cable plasticity defined by a stress-strain curve. The paper did not specify the beam yield strength but an assumed value of 35 ksi gave comparable results for plastic hinge formation.
Reference Solution: Inelastic Stiffened Suspension Space Structures, Journal of the Structural Division, Proc ASCE, Vol 96 No ST6,1970.

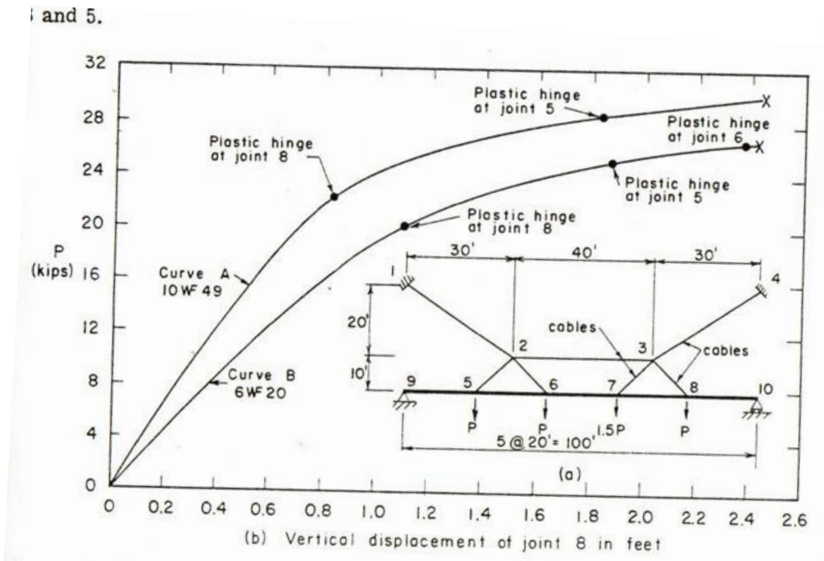
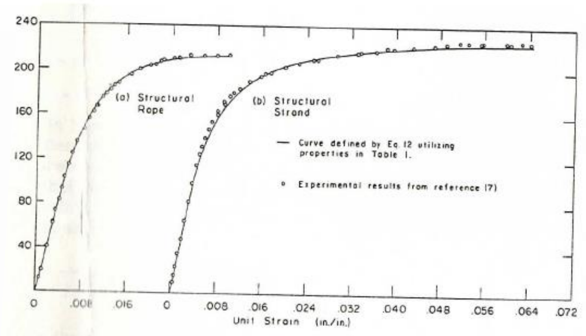
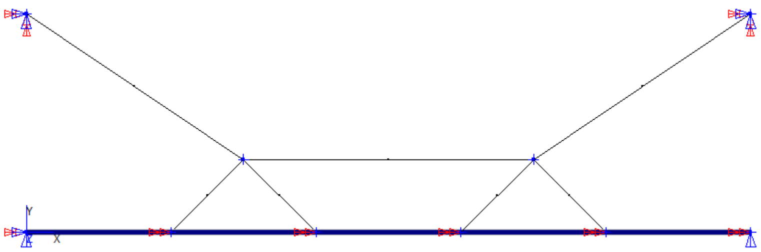


FIG. 10.—RESULTS OF INELASTIC ANALYSIS OF PLANE STIFFENED SUSPENSION STRUCTURE

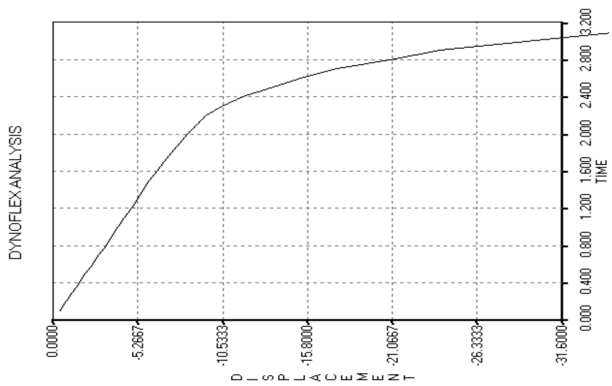


IG. 5.—STRESS-STRAIN CURVE OF STRUCTURAL: (a) ROPE; (b) STRAND

Structural Rope UTS 214 ksi



Case	Defln Ft	Rope Stress ksi	P kips
1	2.47	211	30
2	2.60	213	27



Deflection Plot for Case 1 (Curve A above)

First Hinge for Case 1 at P= 22.5 kips

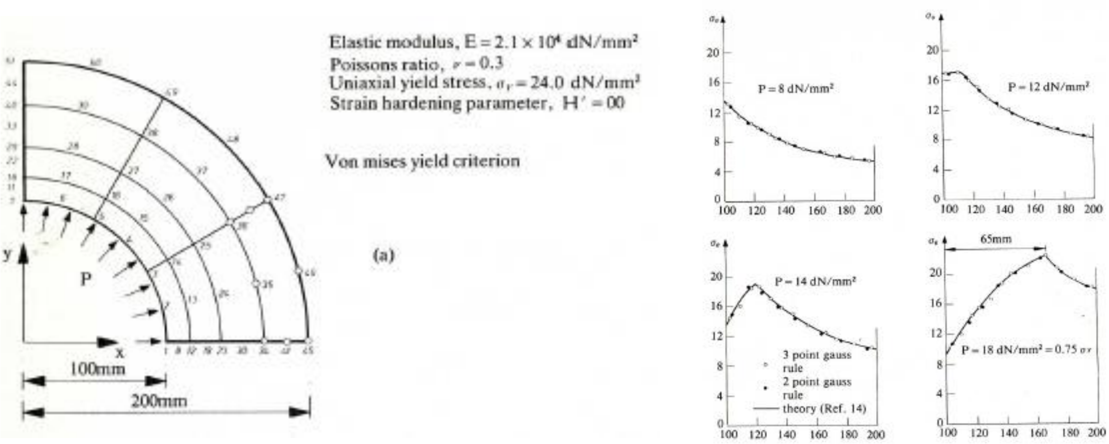
First Hinge for Case 2 at P=20.0 kips

Example 4.10 Elasto-Plastic Analysis of a Thick Cylinder Under Internal Pressure

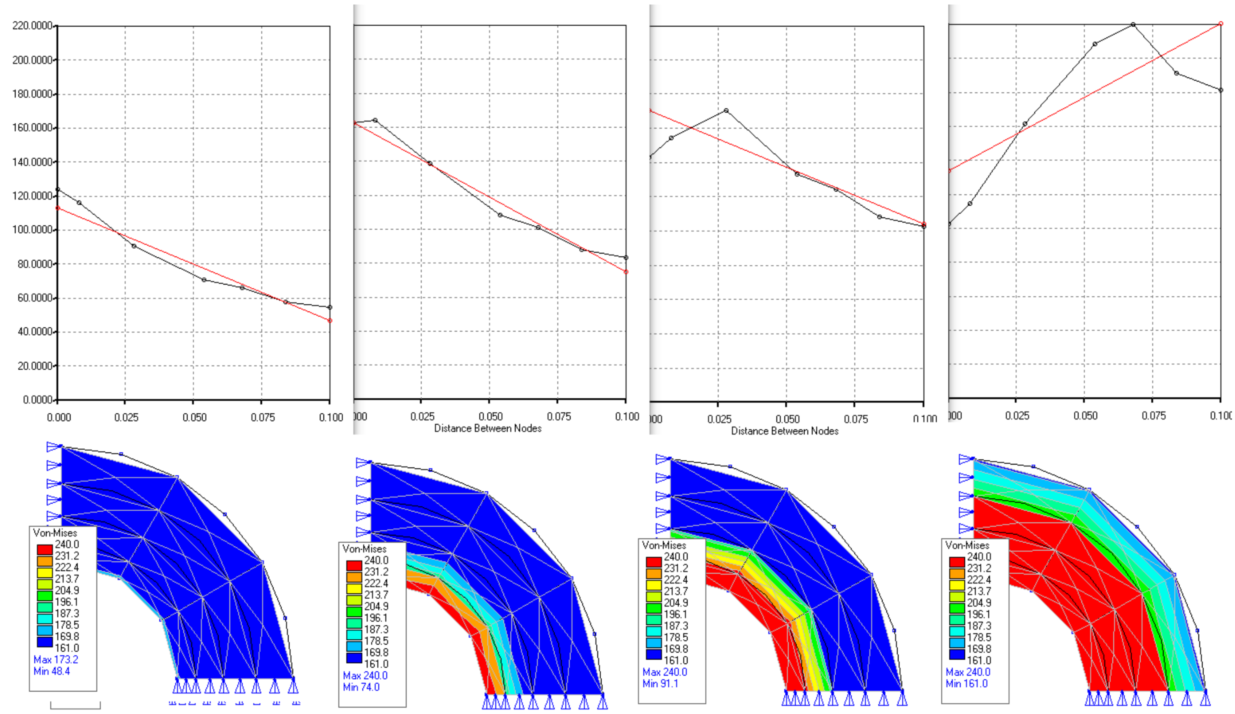
Model: **ThickCylinder**

An infinitely long thick cylinder of internal and external radii 100 mm and 200 mm respectively is subject to an increasing internal pressure. Twelve Type 30-8 node 2-D plane strain elements are used. The mesh is identical to the reference solution (considered bit coarse). He next example uses Type 40 2-D Axisymmetric elements with a more refined mesh.
Reference Solution: Owen, D.R.J., Hinton, E. Finite Elements in Plasticity: Theory and Practice Publisher.Pineridge Press Ltd. Swansea, U.K. 1980. ISBN 0-906674-05-2

The plots below show the hoop stress as the pressure increases.



P = 80 MPa P=120MPa P = 140MPa P = 180MPa

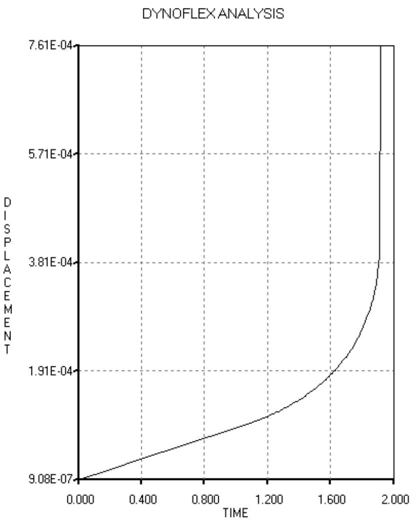


The cylinder becomes fully plastic at 192 MPa.

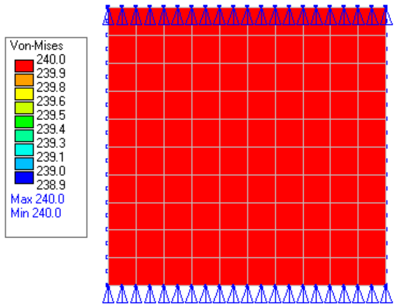
Example 4.11 Elasto-Plastic Analysis of a Thick Cylinder Under Internal Pressure

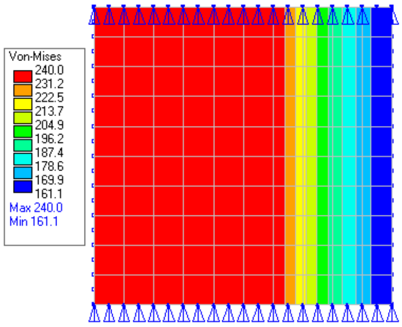
Model: **ThickCylinderAxy**

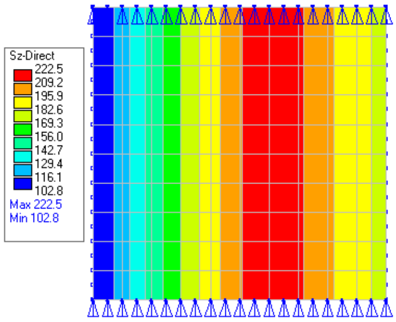
This is the same as the previous example. An infinitely long thick cylinder of internal and external radii 100 mm and 200 mm respectively is subject to an increasing internal pressure. Type 40-8 node 2-D Axisymmetric elements are used. The mesh is more refined than the previous model.



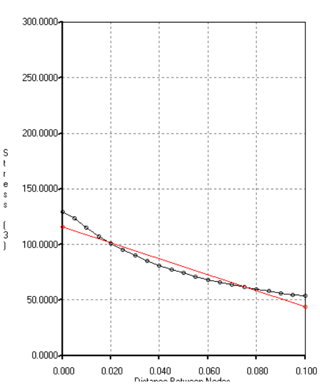
The bi-linear stress-strain curve has no strain hardening and collapse occurs at 192MPa. The Von-mises stress is at yield across the whole of the section.



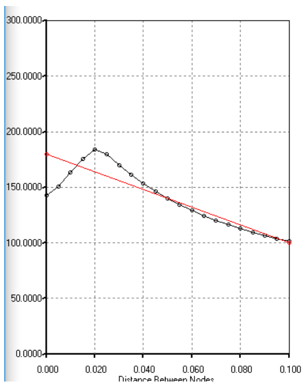




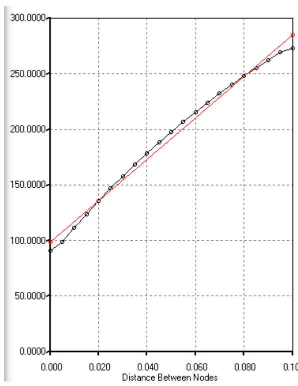
Von-Mises and Hoop stress at 180MPa – Case 4



P = 80MPa



p =140 MPa



P = 192MPa (collapse)

Example 4.12 Thermal-Elasto-Plastic Analysis of Pressurised Pipe.

Model: **ThermalPlasticPipeBeam**

This model demonstrates thermal axial expansion and hoop stress interactions in a pipe (400mmOD, 5mmWall 10m long) for loadings above the VM plastic limits. The pipe has an initial pressure of 75Bar and is 124.6C above ambient which results in the VM stress being at yield. The temperature is then increased from ambient to level that produce stress levels above the VM plastic limit.

The model comprises of a single pipe element Type6(7) and represents a section of pipe between fully fixed anchors. Pipe material properties: Yield=448MPa; E=207GPa; Ep=0.56GPa; α =1.17E-5.

Theory: Below yield (Roark). Above yield, the same pipe is modelled in the next example using solid elements.
Hoop Stress due to pressure Sh = Δp.Do/2t =300 MPa.
Axial Stress due to pressure Sap = μ Δp.2r^2/(R^2-r^2) = 86.64 MPa
Von-Mises stress due to pressure = SQRT(Sh^2 + Sa^2 – Sa.Sh) = 267.4 MPa
Axial Stress due to thermal expansion = Sat = α.ΔT.E = -301.8MPa
True wall axial stress = Sa = Sap + Sat = -211.8 MPa
Von-Mises stress due to pressure & expansion = SQRT(Sh^2 + Sa^2 – Sa.Sh) = 448.1 MPa

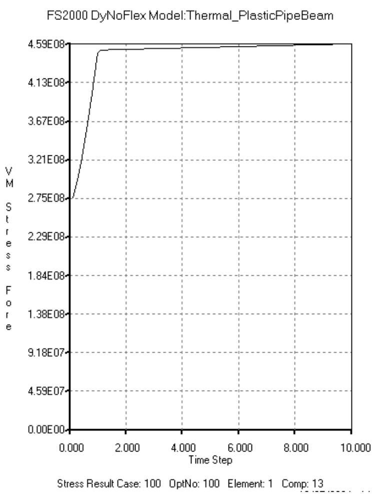
At a temperature of 124.6C the pipe wall Von-Mises stress equals the material yield limit.

Cases 1 to 3 are 3-D Standard linear solutions. Case 100 is a DyNoFlex (C100) solution. Pressure is applied and then the temperature is ramped up to a value of 10 time the temperature that produces yield i.e. 1246C (mechanical properties are constant). Case 10 is a DyNoFlex (L10) that ramps up only the temperature.

It should be noted that hoop strain in Type 6 beam elements is not an independent variable and accordingly it should never have a value greater than yield when undertaking plasticity solutions.

The table below shows exact agreement with the above theoretical values for load levels below yield.

Case No	Press Bar	Temp C	Hoop Stress MPa	Axial Stress MPa	VM Stress MPa	Acc Plast Strain
1	75	0	300	86.63M	267.4	0
2	0	124.6	0	-301.8	301.8	0
3	75	124.6	300	- 215.1	448.1	0
100	75	1246	300	-228.76	459	1.56%
10	0	1246	0	-455.0	455.0	1.238%
101	100	1246	400	-108.1	464	1.979%



This plot shows the VM stress as a function of temperature.
At t=0 the stress is that due to pressure alone 267.4MPa.
At t=1 the point where the temperature is 1246C the VM stress reaches 448MPa, the yield limit. Further increase in temperature produces plastic flow. The VM stress increases slightly due to strain hardening. If there were no strain hardening the VM stress would remain constant.

* This hoop stress is evaluate using the a hoop stress based on Δp.(Do-t)/2t (hoop Stress option in GUI) which is also used in DyNoFlex.

Example 4.13 Thermal-Elasto-Plastic Analysis of Pressurised Pipe.

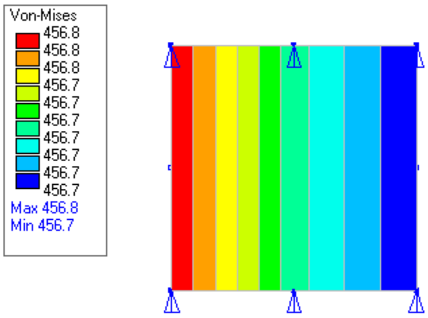
Model: **ThermalPlasticPipe**

This model demonstrates thermal axial expansion and hoop stress interactions in a thin wall pipe (400mmOD, 5mmWall 10m long) for loadings above the VM plastic limits. This is the same as the previous example. The pipe has an initial pressure of 75Bar and is 124.6C above ambient which results in the VM stress being at yield. The temperature is then increased from ambient to level that produce stress levels above the VM plastic limit.

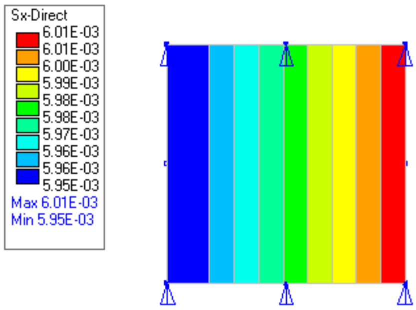
The model comprises of a single Type 40, 8 Node Axisymmetric 2-D solid element and represents a section of pipe between fully fixed anchors. Pipe material properties: Yield=448MPa; E=207GPa; Ep=0.56GPa; α =1.17E-5.

At a temperature of 124.6C the pipe wall Von-Mises stress equals the material yield limit (Case 3).

Cases 1 to 3 are 3-D Standard linear solutions. Case 100 is a DyNoFlex (C100) solution. Pressure is applied and then the temperature is ramped up to a value of 10 time the temperature that produces yield i.e. 1246C (mechanical properties are constant). Case 10 and 11 are DyNoFlex cases that ramps up only the temperature and pressure alone.



Case 100 Von-Mises Stress



Case 100 Radial Deflection

Case No	Press Bar	Temp C	Hoop Stress MPa	Axial Stress MPa	VM Stress MPa	Acc Plast Strain	Defln Radial
1	75	0	288.8-296.2	86.6	256.6-269.3	0	.257
2	0	124.6	0	301.8	301.8	0	.379
3	75	124.6	288.7-296.2	215	437.9-445.5	0	0.633
100	75	1246	291.7-293.3	232.5-234.5	456.7-456.8	1.552-1.568	6.013
10	0	1246	0	455	455	1.238	4.286
11	180	0	699.7-704.3	352-359	613-618	29.4-30.3	52.39
101	100	1246	390-391	113-114.7	459	2-2.02	7.16

Reference Solution ANSYS (STIFF82 Axy)

Case No	Press Bar	Temp C	Hoop Stress MPa	Axial Stress MPa	VM Stress MPa	Acc Plast Strain	Defln
3	75	124.6	289-296	215	438-445	0	0.633
100	75	1246	292-293	233-235	457	1.566-1.583	6.059
10	0	1246	0	455	455	1.238	4.286
11	180	0	700-704	342-350	613-618	29.4-30.3	52.37
101	100	1246	390	113-115	459	2-2.03	7.2

Example 4.14 Thermal-Elasto-Plastic Analysis of Pressurised Pipe.

Model: **ThermalPlasticPipePS**

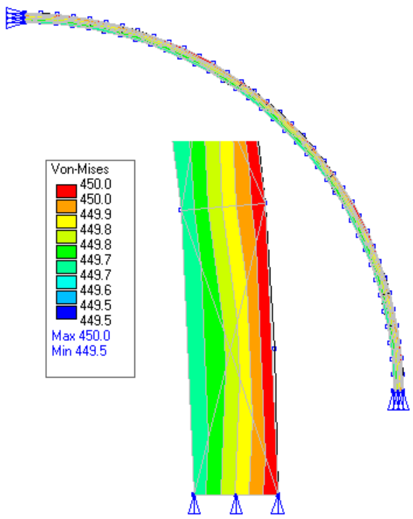
This model demonstrates thermal axial expansion and hoop stress interactions in a thin wall pipe (400mmOD, 5mmWall 10m long) for loadings above the VM plastic limits. This is the same as the previous example. The pipe has an initial pressure of 75Bar and is 124.6C above ambient which results in the VM stress being at yield. The temperature is then increased from ambient to level that produce stress levels above the VM plastic limit.

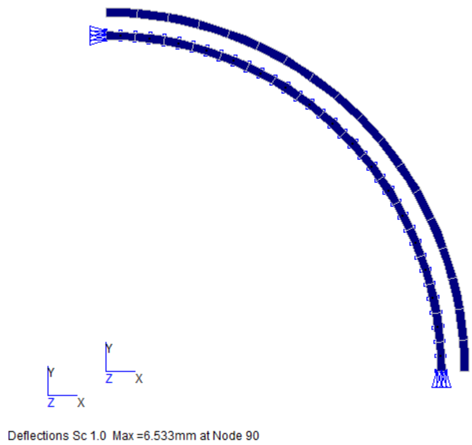
The model comprises of Type 30, 8 Node Plain Strain 2-D solid element and represents a section of pipe between fully fixed anchors. ¼ symmetry is assumed. A single element width represents the wall thickness.

Pipe material properties: Yield=448MPa; E=207GPa; Ep=0.56GPa; α =1.17E-5.

At a temperature of 124.6C the pipe wall Von-Mises stress equals the material yield limit (Case 3).

The results obtained are almost identical to the single axisymmetric model.





Case 100 Von-Mises Stress

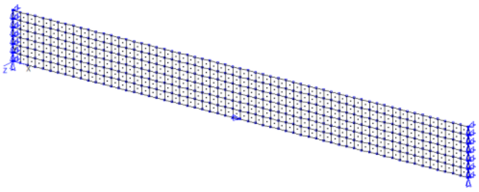
Case 100 Radial Deflection

Case No	Press Bar	Temp C	Hoop Stress MPa	Axial Stress MPa	VM Stress MPa	Acc Plast Strain	Defln Radial
1	75	0	288.6-296.6	86.6	256.3-269.9	0	0.257
2	0	124.6	0	301.8	301.8	0	0.379
3	75	124.6	288.6-296.6	215.1	437.7-445.9	0	0.633
100	75	1246	291.9-292.8	224.2-226	449.5-450	1.476%	6.53
10	0	1246	0	-455.0	455.0	1.235%	5.16
11	180		699.2-704.1	352.2-359.9	612.5-618.3	29.44-30.32	52.417

Example 4.15 Elasto-Plastic Collapse of a Square Beam-Shell Elements

Model: **PlasticBeam1 (PlasticBeam2)**

This model evaluates the elasto-plastic collapse of a simply supported solid square section beam subjected a UDL. The beam span is 1m and the section width/depth is 100mm. The total load on the beam is a UDL. The model uses Type 52 4-Node Shell Elements This represents a very thick shell aspect ratio. Model **PlasticBeam2** also models the same beam using Type 30 2-D Plane stress elements for the in-plane loading Cases 1 & 2.



Ideal plasticity is assumed.

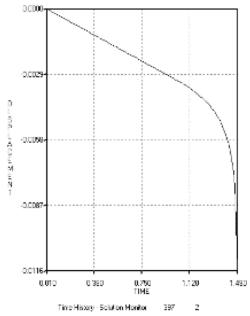
Beam Properties: E=205GPa; Poiss=0.3; YST = 300MPa.

Reference Solution: Basic Beam Theory.

I = $bd^3/12$ = 8.333E-6 m^4; Elast Modulus=1.6667E-4 m^3; Plastic Mod=2.5E-4 m^3

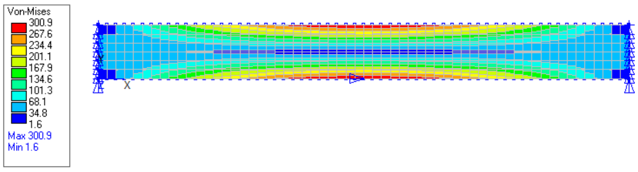
Yield Moment=50 kNm; Yield Load=8M/L = 400kN

Plastic Moment = 75 kNm; Plastic Load = 600kN.



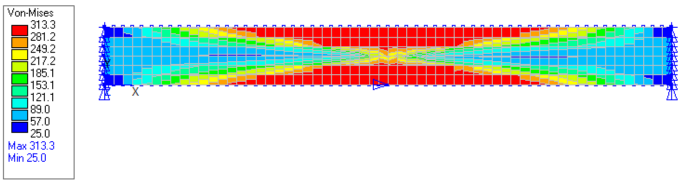
The plots below are for un-averaged stresses.

Case 1 In-Plane Linear Elastic Load Factor = 1.0



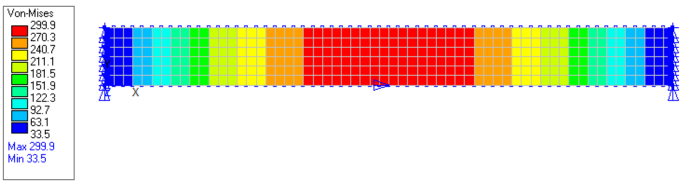
Max Elastic stress at yield

Case 2 In-Plane Plastic Load Factor = 1.49



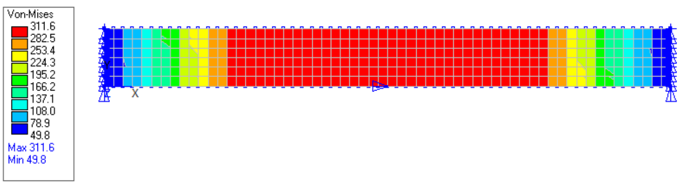
Plasticity spreading through beam depth

Case 3 Out of Plane Linear Elastic Load Factor = 1.0



Top Surface: Max Elastic stress at yield

Case4 Out of Plane Plastic Load Factor = 1.5



Top Surface: Plasticity spreading along beam

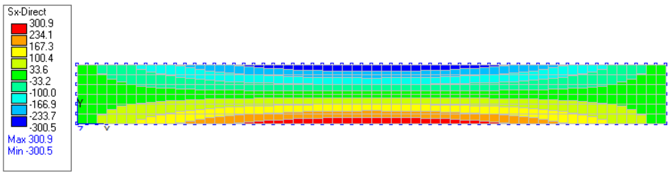
Example 4.16 Elasto-Plastic Collapse of a Square Beam-2D & 3D Solids

Model: **PlasticBeam2 & PlasticBeam3**

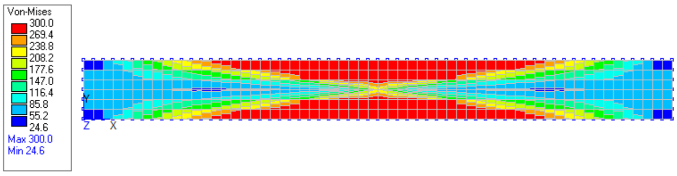
This model evaluates the elasto-plastic collapse of a simply supported solid square section beam subjected a UDL. The beam span is 1m and the section width/depth is 100mm. The total load on the beam is a UDL.
PlasticBeam2 uses Type 30 2-D Plane stress elements.
PlasticBeam3 uses Type 70 3-D Hex elements. The aspect ratio of the hex elements is suited only for stress variation in the X direction (vertical load) for a non-linear plastic solution. A linear solution in the lateral direction (Case 3) does however give the same results as that for the vertical direction.

This is the same as the previous example. Yield Load Limit=400kN. Plastic Load Limit(LF=1.5) = 600kN

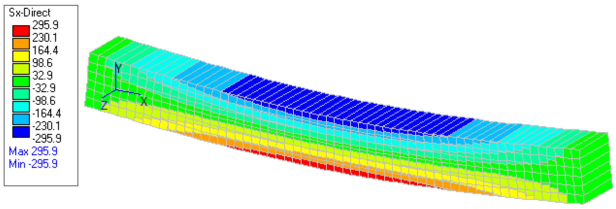
Both models have two load cases each applied the vertical Y direction. Case 1 is linear elastic to the yield limit and Case 2 is at the plastic limit.



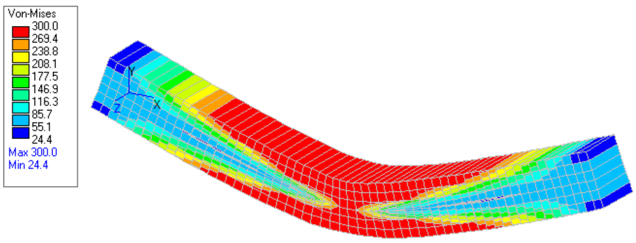
Case 1 Max X stress at 400kN = 300.9MPa



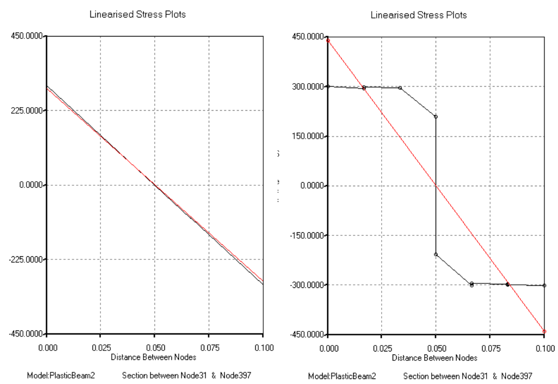
Case2 Von-Mises at 600kN = 300MPa



Case 1Max X stress at 400kN = 209.9MPa



Case 2 Von-Mises at 600kN = 300MPa



X Direction Stress from the 2-D model at mid span.

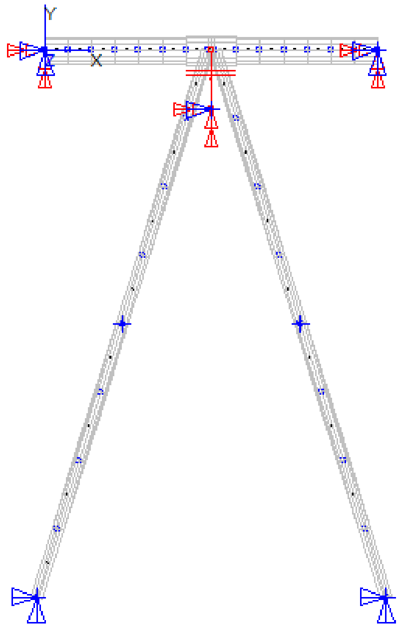
Case 1 Linear Variation 300MPa Yield at Top & Bottom

Case 2 Stresses at Yield thru section at Plastic Limit

Example 4.17 Elasto-Plastic Collapse of a K-Braced Frame

Model: **KBracedSTST**

This model assesses the elasto-plastic collapse of a K-Braced frame. The model is based on a test frame that was loaded to its ultimate capacity. (T. Moan et al, “Collapse Behaviour of Trusswork Steel Platforms”, Behaviour of Offshore Structure, 1985). Full material details are not available in the paper but sufficient are given to create a similar model and undertake a solution with surprisingly good correlation.

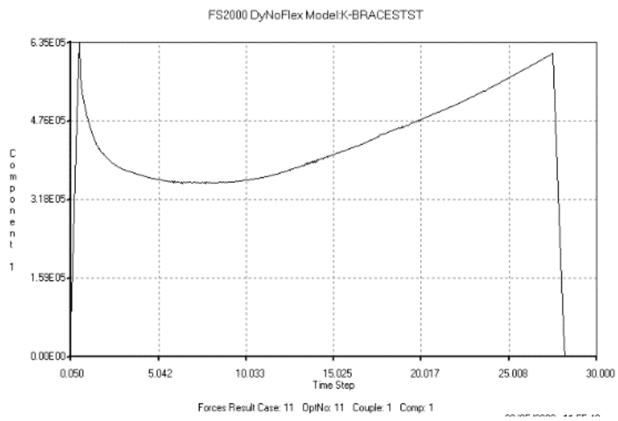


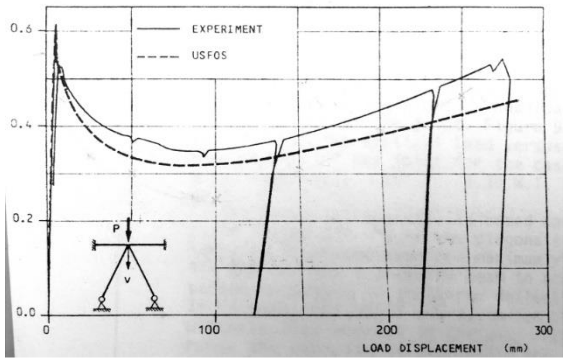
The beam elements used are Type 6(7). The Geom Type 7 is a bi-linear stress-strain material model. The stress-strain data is based on E=205GPa, YST=345 MPa and UTS is 490MPa at 30% strain.

This type of failure frame produces a negative effective stiffness matrix (mechanism formed) and cannot be solved by load control. The loading is applied as a prescribed displacement, applied through a load monitoring tension only couple.

The predicted collapse of 636 kN load compared well with the reported results.

An Eigen buckling solution is also undertaken. This predicted a buckling load of 590 kN (0.987 LF).



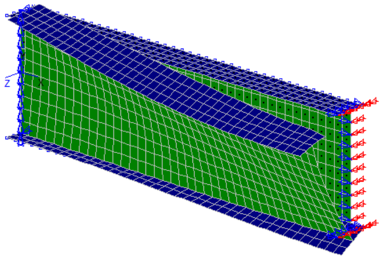


Note that the FS2000 plot is Force vs Time Step and not Force vs Displacement (slope sign different on reversal)

Example 4.18 Elasto-Plastic Collapse of a Deep I Beam

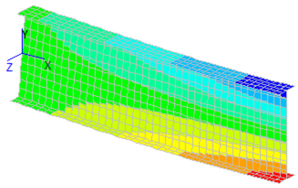
Model: **Plate_I_Beam**

This model investigates the elasto-plastic collapse of a simply supported symmetric I Beam subjected to a concentrated mid span load (598.5kN). The beam span is 4m.

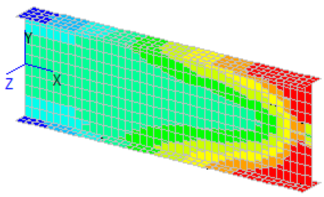


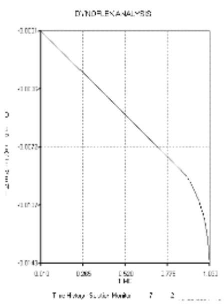
Depth=600mm: Width=180mm
Flange=10mm: Web=8mm
E=210GPa: Poiss=0.3; Yield=345MPa;Et=3E8
Von-Mises yield criteria
The beam model uses Type 52 shell elements. Half model symmetry.
The base case loading is the section plastic moment i.e. LF=1.0 gives the Plastic Moment = 598.5 kNm. Elastic moment = 509.9 kNm (SF=1.174).

Reference Solution: LT Beam & Basic Plastic Beam Theory.

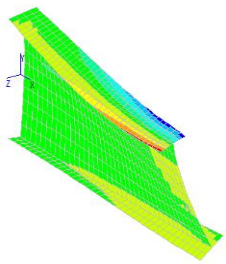


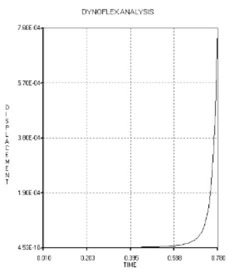
Case 1 Linear Elastic (Linear Solver)
Maximum Flange Stress=444.3MPa (Implies a plastic shape factor of 1.29)
Deflection=10.08mm



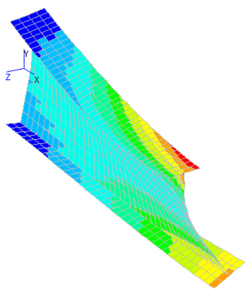


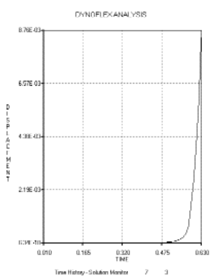
Case 2 Plastic
Solution fails at LF=1.02 due to plastic yielding.
Max Von-Mises = 345.1 MPa -Spread across flanges outer web section. Perfects plasticity make full section plasticity difficult to achieve.
Max Deflection (top flange) =12.9 mm Vertical





Case 3 Elastic Large Displacement
Excessive lateral deflection at LF=0.78 due to elastic lateral buckling.
Max Von-Mises=345MPa
<u>Max Deflection:7.809mm Vertical:0.76mm Lateral</u>
LT Beam gives Mc=436.93kNm (LF=0.73)



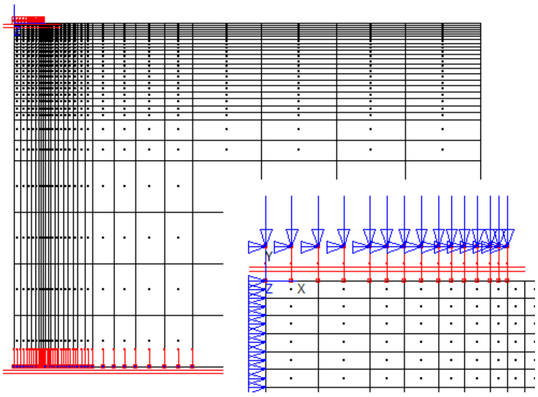


Case 4 Plastic Large Displacement
Solution Fails at LF=0.62 due to plastic flange bending.
Maximum Von-Mises = 346MPa
Deflection (Top flange): 6.3mm Vertical   7.4mm Lateral

Example 4.19 Elasto-Plastic Soil Foundation

Model: **SoilBearing**

This model evaluates the vertical bearing capacity factors (Nc) of smooth strip foundation footing using a Mohr-Coulomb plasticity model. The model uses 2-D 8Node plane strain elements. To avoid numerical instabilities the solution employs a displacement-controlled approach.



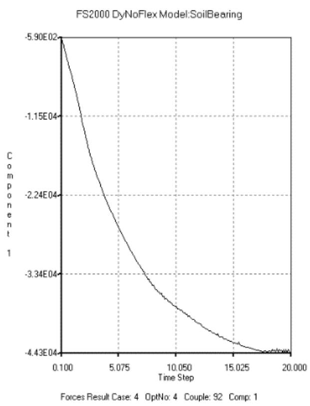
The bearing load is applied using prescribed displacements (Case 4). These are applied through Type 12 contact elements. The vertical restraint also uses node to ground contact elements. These contact elements are used solely for convenient load monitoring purposes. q used below is the couple reaction summation.

E = 2E5 kN/m^2; Poiss =0.3 Cohesion= 20 kN/m^2
To evaluate Nc the spoil is assumed to be weightless.
Foundation width w = 1.4

Reference Solution: "Computation of vertical bearing factors N_C of strip footing by FEM" Phuor Ty et al 2019 IOP Conf. Ser.: Mater. Sci. Eng. 527 012017.

This plot of load contact is for a friction angle of 15, it shows that the bearing pressure due to the prescribed displacement has reached a maximum. Nc=q/c.w = 310E3/1.4/20E3 = 11.07 q=reaction summation

The displacement vector plot shows the typical orientation of the slip planes.



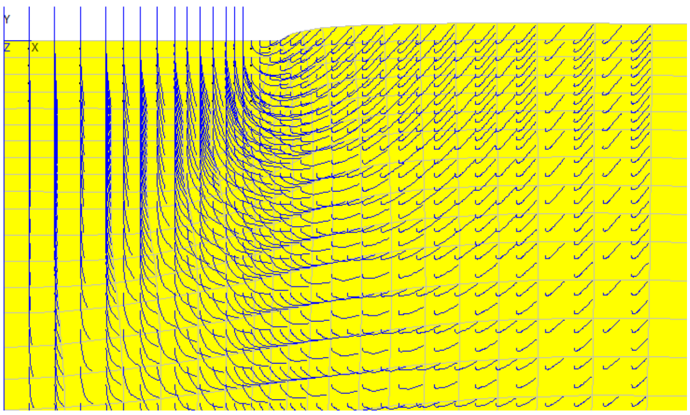


Table from reference solution.

Authors	Methods	Soil friction angle ϕ (degree)								
		5	10	15	20	25	30	35	40	45
Prandtl, 1921 [4]	Slip line method	6.49	8.35	10.98	14.83	20.72	30.14	46.12	75.31	133.88
Terzaghi, 1943 [1]	Limit equilibruim	7.34	9.61	12.86	17.69	25.13	37.16	57.75	95.66	172.28
Soubra, 1999 [3]	Upper bound analysis	6.5	8.36	10.99	14.86	20.77	30.24	46.33	75.77	134.99
Present analysis	Finite element	6.57 (6.73)	8.43 (8.52)	11.03 (11.10)	14.64 (14.69)	20.39 (20.32)	29.44 (29.14)	43.64 (44.28)	68.67 (67.91)	103.73 (107.59)

FS2000 evaluated N_C factors.

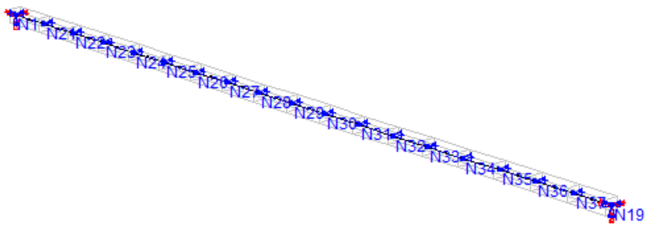
Frictionϕ	N_C
5	6.55
15	11.07
25	20.91
35	46.54
45	109.31

Example 5.1 Natural Frequencies of a Beam – Beam Elements

Model: **CantileverFreq**

This example evaluates the first three natural frequencies of a solid rectangular steel beam due to self-weight.

Two support conditions are considered. Cantilevered and fully fixed at both ends.



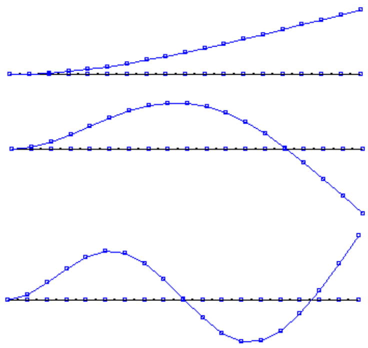
50mm Square section

I = 5.208E-7m^4
A = 2.5E-3m^2
E= 205GPa
L = 3m
Density = 7860kg/m^3

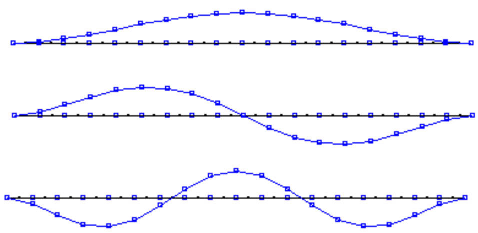
Reference Solution: "Vibration Theory and Applications", W.T.Thomson, Prentice-Hall Inc, 1965, page 275.

The refence solution values are shown in parentheses.

Condition	1 Mode Rad/s	2nd Mode Rad/s	3rd Mode Rad/s
Cantilevered	28.79 (28.8)	180.23 (183)	503.77 (505)
Fully Fixed	182.88 (183)	505.95 (505)	983.04 (991)



Cantilever Mode Shapes

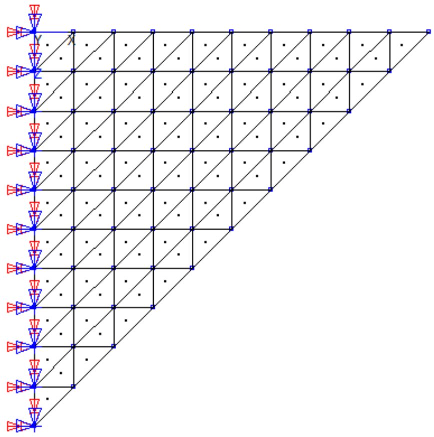


Fully fixed Mode Shapes

Example 5.2 Triangular Wing Eigen Values – Shell Elements

Model: **Triangular Wing**

The example evaluates the natural frequencies of a triangular wing.



The length and width are 6 ins. Thickness = 0.034 ins.

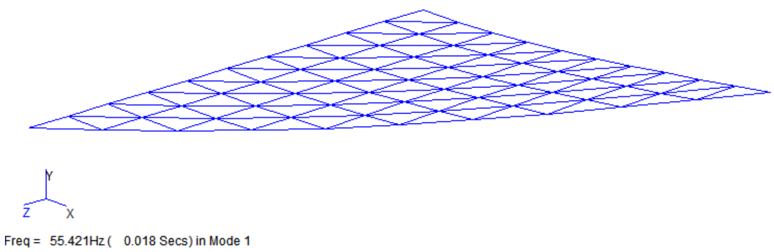
E =6.5E3 ksi; ν = 0.3541; ρ = 0.166E-3 $lb.sec^2/ins^4$

The model uses Type 50 3-Node shell elements.

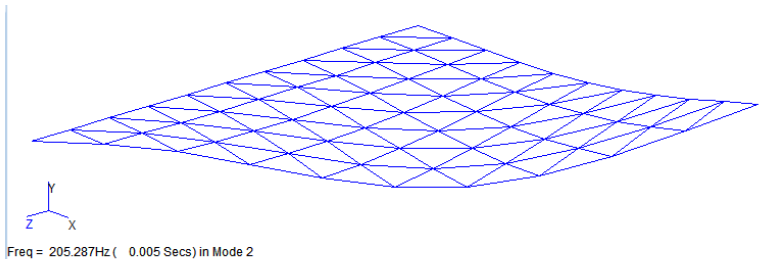
The mass case is self-generated.

Reference Solution: “ASME Pressure vessel and Piping 1972 Computer Programs Verification” ed IS Tuba and WB Wright, ASME Publication I-24, Problem 2.

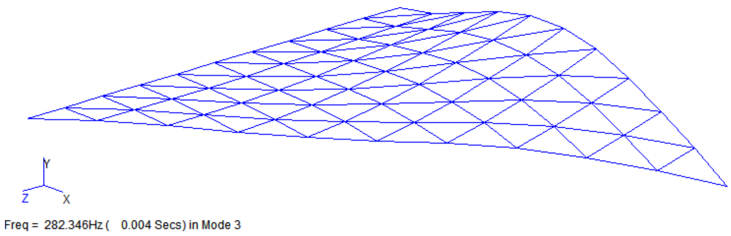
The refence solution values are shown in parentheses are from a COSMOS examples using quads for the same 6.problem.



First Mode 55.42 Hz (55.4)



Second Mode 205.29 Hz (205.3)



Third Mode 282.35 Hz (282.3)

Example 5.3 Period of a Pendulum – Dynamic – Large Displacement.

Model: **Pendulum_Dyn_LD**

The model comprises of a single massless Type 6 beam element. The beam is hinged at the top and has concentrated mass at the free end. The starting position of the beam is at 90 degrees. A time history solution starts with mass being released from the 9 0'clock position. The time history runs for about 12 cycles.

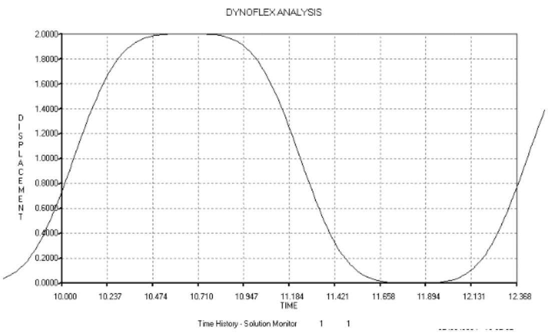
The natural period of a swinging pendulum is a function of both length and amplitude.

$$T_0 = 2\pi\sqrt{\frac{\ell}{g}}$$

For small amplitudes << 1 Degree this formular is often used.
For a 1m long pendulum with amplitude of 90 degrees the small displacement solution is 2.006s

$$\frac{4T_0}{\left(1+\sqrt{\cos\frac{\theta_0}{2}}\right)^2}.$$

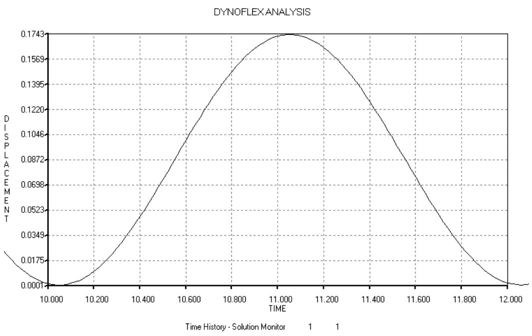
For larger amplitudes this formula may be used.
For a 1m long pendulum with amplitude of 90 degrees the large displacement solution is 2.368s.



The time displacement shows a period very closely in the region 2.368s.

The periodic motion is not sinusoidal.

If the initial position is changed from 90 degrees to 5 degrees, the is following obtained.



The time displacement shows a period very closely in the region of 2s and the motion appears sinusoidal.

Eigen Frequency Solution

Case 2 (Gravity applied in the X direction) is a small displacement Eigen solution. This gives:

Mode	Freq(Rad/s)	Freq(Hz)	Period(s)
1	3.132092	0.4984879	2.006067
2	14.14158	2.250702	0.4443058

First Mode:Exact agreement with the small disp. analytical formula.

Example 5.4 Harmonic (Modal) Response of Two Mass Spring System

Model: **HarmonicResp1**

This model determines the dynamic response amplitudes of two masses when excited by a harmonic force.

m1=m2=0.5 lb-sec^2/ins
k1=k2=k3=200 lb/ins
F1= 200 lbs

The model uses unit length beam elements to model the springs (A=1; E=200).

The solution is obtained by first determining the natural frequencies and then undertaking a modal response solution.

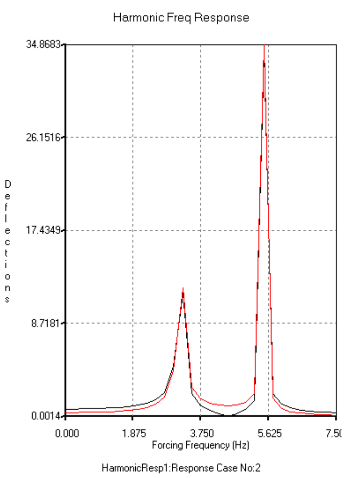
Reference Solution: "Vibration Theory and Applications", W.T.Thomson, Prentice-Hall Inc, 1965, Exp 6.6-1,page 178. The refence solution values are shown in parentheses.

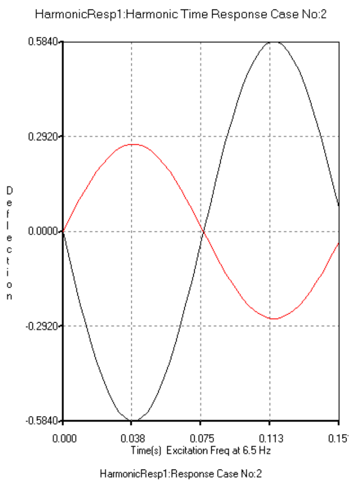
Frequency Solution 1st mode = 3.183Hz (3.183) 2nd Mode = 5.5132 Hz(5.513)

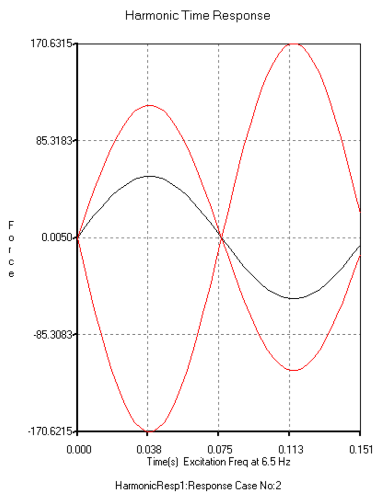
Response at specific frequencies.
The values are shown in parentheses are from an ANSYS verification example.

Freq Hz	X1	Phase	X2	Phase
1.5	0.8227 (0.8227)	0	0.4627 (0.47274)	0
4	0.5115 (0.51145)	180	1.215 (1.2153)	180
6.5	0.5851 (0.58512)	180	0.2697(0.26965	0

The response is obtained for a frequency range of 0 to 7.5 Hz as shown below. Also shown below are the displacement response at a frequency of 6.5 Hz and the forces in the springs at 6.5 Hz.







The model solution also included an results case (FCASE) at 0.1168s for the 6.5Hz excitation.

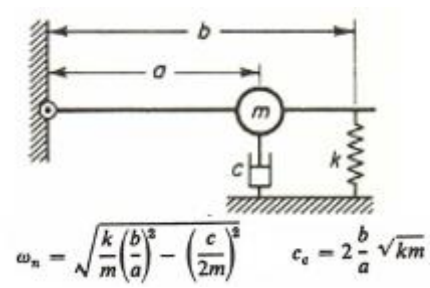
DyNoFlex Time History Solution

A solution was also undertaken using a DyNoFlex time history solution (in Batch). This gave similar results but did include a small contribution from the initial transient solution during the ramping up of the sinusoidal force.

Example 5 5 Transient Response of Viscous Damped System

Model: **DampedVibration**

The model evaluates the natural frequencies and response of a damped system with varying levels of damping.



a = 3m; b = 4m
k = 40kN/m
m = 1 Tonnes

The beam is massless and is relatively stiff to behave as a rigid bar.

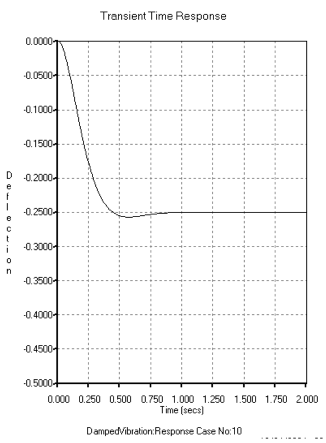
The model has two load cases: Case 1 is the mass case; Case 2 is a 10kN load applies at the free end.

Reference Solution:“Vibration Theory and Applications”, W.T.Thomson, Prentice-Hall Inc, 1965, Exp 13,page 49. From the refence solution so following table can be constructed.

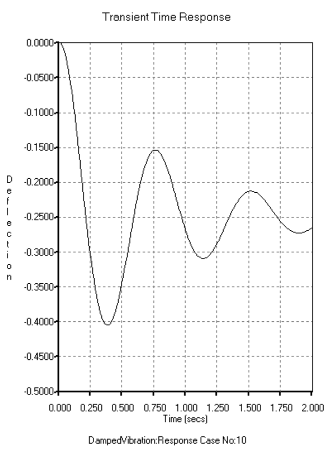
Damping % Crit	75	50	25	15	10	5	0
Cc kN-s/m	12.65	8.433	4.216	2.530	1.687	0.843	-
Nat Freq Rad/s	5.578	7.303	8.165	8.337	8.390	8.422	8.433
Nat Freq s	1.126	0.860	0.695	0.754	0.749	0.746	0.745

Case 1 Eigen frequency solution - First mode = 8.4306 Rad/s (8.433)
Case 2 A static solution gives results in a deflection of 250mm at the load point (10E3/40E3).
Case 10 Dynamic Solution Load Case 2 suddenly applied – Modal Response.
Case 11 Dynamic Solution Load Case 2 suddenly applied – Incremental Time History (DyNoFlex)

Case 10 75% Critical Damping



Case 10 15% Critical Damping



The model uses Δt=0.01, time step

Peak for 75% Damping occurs at t=0.57s y=256.958mm (0.563s)

Peak for 15% Damping occurs at t =0.39s y=404.955mm (0.377s)

The theoretical reduction in the natural period due to increased damping agrees well with that from the modal response solution.

Case 11 Produced almost the same response.

The incremental solution uses a Δt=0.01, time step

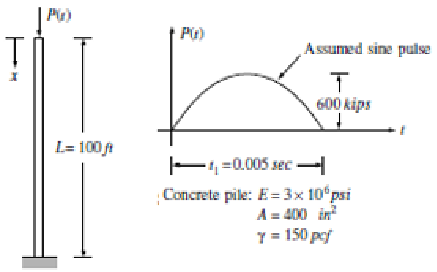
Peak for 75% Damping occurs at t=0.57s y=257.1mm (0.563s)

Peak for 15% Damping occurs at t=0.38s y=404.9mm (0.377s)

Example 5.6 Pile Driving Impact – Wave Propagation - Modal Response

Model: **PileImpact**

This is a model of a concrete pile subjected to a hammer blow. The hammer blow is represented by a half sine impulse. The model shows axial wave propagation as the stress wave moves down the pile.



Wave propagation velocity = Vw = $(E/\rho)^{0.5}$ = (3E6*386/8.681E-2)
= 115.5E3 ins/s

Length of half sine impulse = 115.5E3*0.005 = 577.5 ins.

51 elements in the pile will give just over 24 elements in half wave, enough to capture the response.

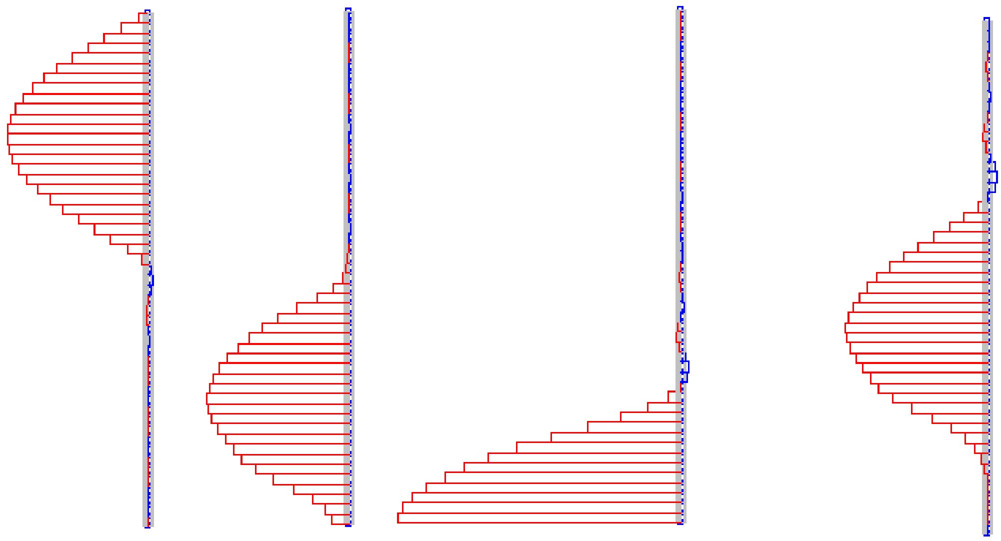
The pile nodes are only free in the y direction and the tip is fixed.

Reference Solution: Clough & Penzien, Dynamics of Structures, McGraw-Hill 1975, Example E19-5.

The IMPULSE command is used to apply a nodal load in the Y direction at the top of the pile.

```
IMPULSE,4,0.005,0,20,2,-6.000E05,2
```

The time take for the pulse to move from the head to the tip = 1200/Vw = 0.01038s. Because the tip is fixed, the pulse will then start being reflected from the tip and become a maximum at t = 0.1038+577.5/115E3 = 0.129s. The tip is rigid therefore the maximum will be double the wave load in the pile to 1200 kips. The reflected wave will travel back up the pile with no loss of energy.



T = 0.005s	t = 0.01038s	t = 0.0129s	t= 0.0170
Wave fully in pile	Wave just reaching tip	Wave reflecting	Wave travelling back up
599.96 kips	601.9 kips	1197.69 kips	602.55 kips

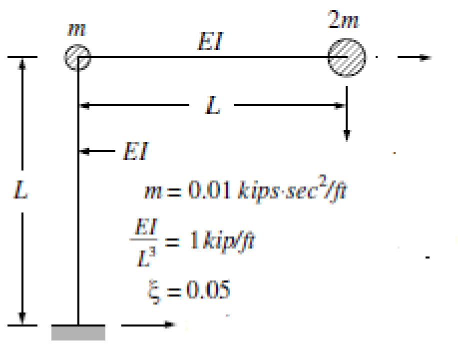
It is interesting to note that when the wave reaches the head which is free, the wave will be reflected as a tensile stress (Case 13). Similarly, if the tip support was very soft the wave would be reflected as a tensile stress.

A DyNoFlex incremental solution (Case 20, t=0.0129) produces the same results at the above.

Example 5.7 Seismic Response (Modal Spectrum) of a Three Storey Frame.

Model: SeismicResp_Cant

The model undertakes the response spectrum analysis of tower type structure with an offset. The solution uses an Eigen solver and the Seismic Response module.



E = 30E6 ksi

I = 38.4 ins^4

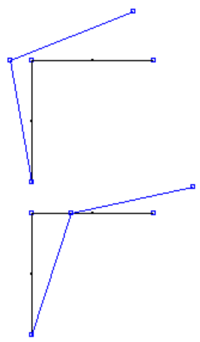
L = 240 ins

m = 322 Lbs/g

Reference Solution: Clough & Penzien, Dynamics of Structures, McGraw-Hill 1975, Example E26-5.

The refence solution values are shown in parentheses.

The spectral accelerations corresponding the two modes are : 8.89 and 42.15 ft/s^2 respectively (obtained from the response spectra from the reference for 5% damping).

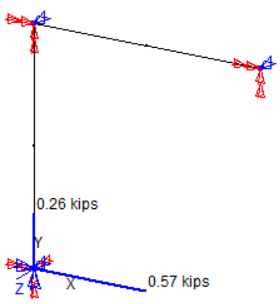


```
Response Spectrum Curve
Period (Secs)        Value
      0.372        289.8000
      1.143        106.6800

Response Spectrum Curve - Modal Points
Curve Factor  1
Mode  Period (Secs)     Disp(ins)    Velocity(ins/s)  Accel(ins/s2)
 1        1.144        3.5361       19.4225      106.6800
 2        0.373        1.0193       17.1815      289.6075

Total Mass (excl mass on rest.) 2.500E+00  Load Case Mass 2.500E+00
Excitation X Direction  1
Mode     Freq   Freq     Period     Modal     Cumulative       Damping
No       Rad/s  Hertz    Seconds   Eff Mass    Eff Mass   %    Ratio

 1       5.49    0.87     1.1439  5.468E-01  5.468E-01  21.87  0.05
 2      16.86    2.68     0.3728  1.953E+00  2.500E+00 100.00  0.05
```



The SRSS reactions are:

Vertical 0.26 kips (0.26)

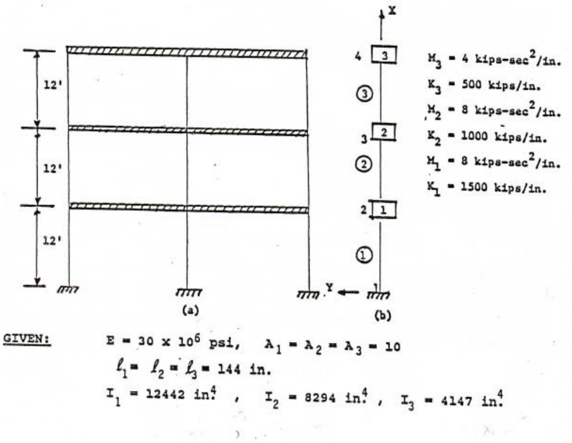
Horizontal 0.57 kips (0.57)

Example 5.8 Seismic Response (Modal Spectrum) of a Three Storey Frame.

Model: **SeismicResp_Frame**

The model undertakes the response spectrum analysis of three storey frame represented by an equivalent three beam structure. The solution uses an Eigen solver and the Seismic Response module.

Reference Sol'n: Biggs J.M, "Introduction to Structural Dynamics! McGraw-Hill Book Co.,1964, page 266-269. The refence solution values are shown in parentheses (slide rule accuracy).



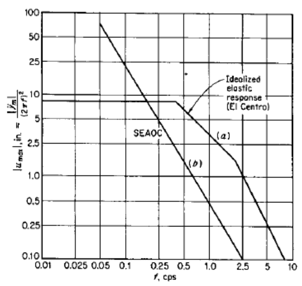


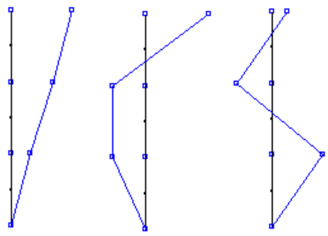
FIGURE 6.12 Idealized response spectrum for El Centro earthquake, May 18, 1940, N-S component and SEAOC recommendation.

```
Response Spectrum Curve
Period (Secs)        Value
       0.123         0.0100
       0.500         1.6500
       2.515         8.3000
     100.000         8.3000

Response Spectrum Curve - Modal Points
Curve Factor  1
Mode   Period (Secs)     Disp(ins)     Velocity(ins/s)   Accel(ins/s2)
 1          1.003        3.3110         20.7380         129.8885
 2          0.459        1.4712         20.1441         275.8252
 3          0.315        0.8439         16.8410         336.0773

Total Mass (excl mass on rest.) 2.000E+04  Load Case Mass 2.000E+04
Excitation X Direction   1
Mode       Freq    Freq     Period      Modal     Cumulative          Damping
No         Rad/s   Hertz    Seconds    Eff Mass    Eff Mass    %      Ratio

 1         6.26    1.00     1.0032  1.683E+04  1.683E+04  84.17   0.00
 2        13.69    2.18     0.4589  2.000E+03  1.883E+04  94.17   0.00
 3        19.96    3.18     0.3149  1.166E+03  2.000E+04 100.00   0.00
```



Modal Frequencies

Mode 1 0.997 (1.00) Hz

Mode 2 2.179 (2.18)Hz

Mode 3 3.176 (3.18) Hz

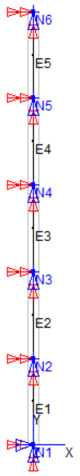
Displacements (ins)		
Node	ABS	RMS
2	2.087 (2.01)	1.526 (1.5)
3	3.734(4.09)	3.213 (3.24)
4	5.462 (6.78)	4.701(5.03)

Shear Force (kips)		
Element	ABS	RMS
1	3130 (3020)	2288 (2250)
2	2170(2080)	1784 (1740)
3	1411 (1345)	923(895)

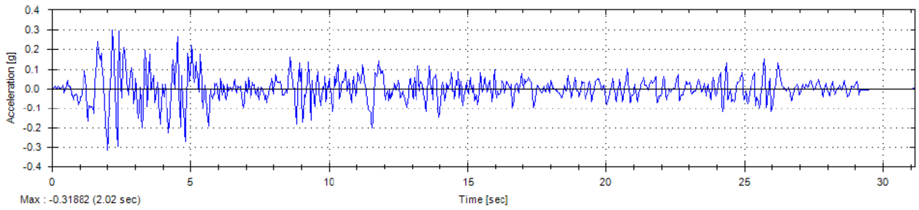
Node 4 ABS acceleration 352 (338) ins/s^2 Node 4 RMS acceleration 231 (225) ins/s^2

Example 5.9 Seismic Response (Modal Time History) of a 5 Storey Frame

Model: **Earthquake**



The model undertakes a time history model response analysis of a five-storey frame represented by an equivalent five beam structure. The model is subjected to ground accelerations define by a g acceleration record (ElCentro N-S).

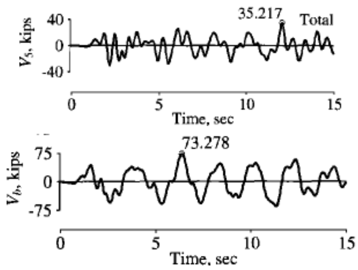


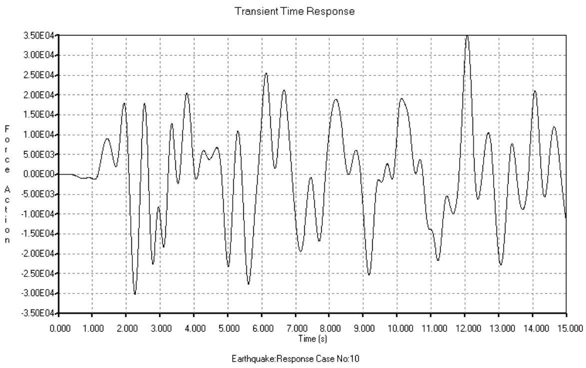
Floor Mass = 100 kips/g at all floors.
Floor Stiffness = 31.54 kips/ins
Column rotation at each floor is zero. Column height = 144ins
E = 29.5E3 ksi; Equiv I = 125.2 ins^4; Damping = 5%

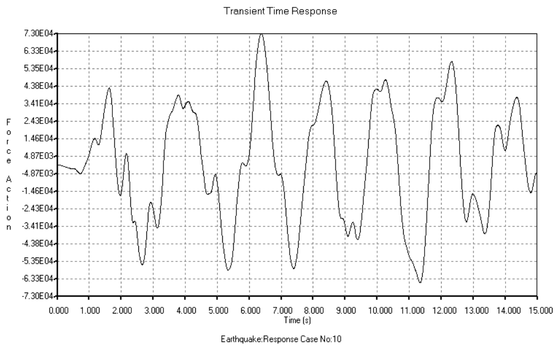
Reference Solution: Chopra A. K., "Dynamic of Structures, Theory and Application to Earthquake Engineering", Prentice-Hall, 1995.

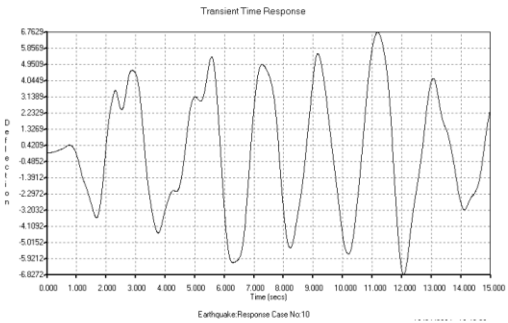
Natural Modes (Seconds): 2, 0.685, 0.435, 0.338 and 0.297.

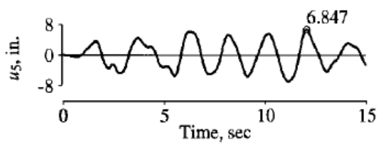
Maximum Top Storey Shear = 35 kips (35.217)
Maximum Base Shear = 73 kips (73.278)
Maximum Top Storey Displacment 6.827 (6.847)











Example 5.10 Beams-Dynamic Large Displacement

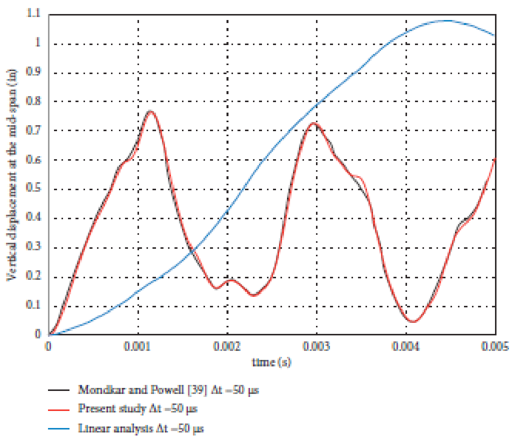
Model: FixedBeam

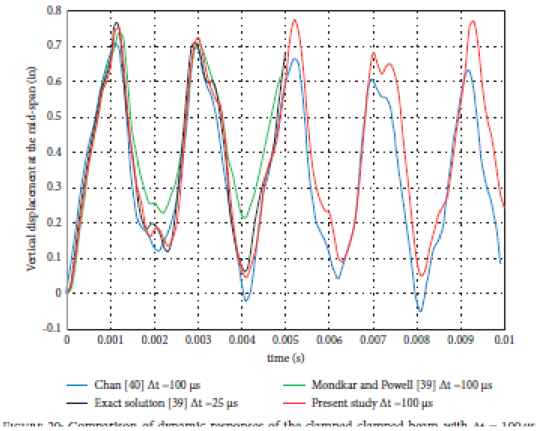
This example is a built-in beam subjected to a concentrated load being suddenly applied at centre span. The model uses 10 Type 6 beams and a FS-DyNoFlex solution.

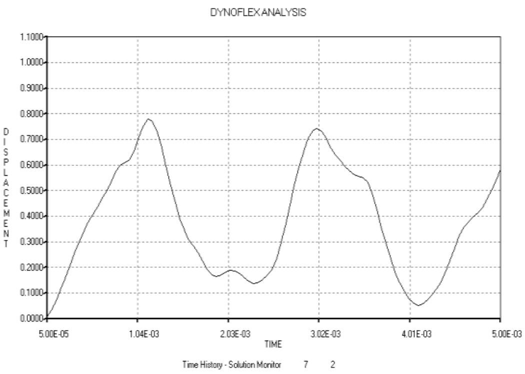
The reference solution is from:

Shock and Vibration Volume 2023, Article ID 6675678, 30 pages.

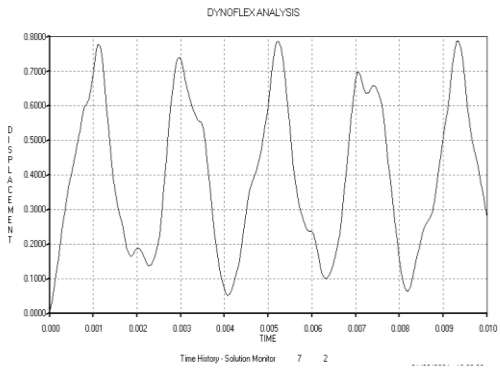
Corotational Finite Element Dynamic Analysis of Space Frames with Geometrically Nonlinear Behaviour Based on Tait–Bryan Angles



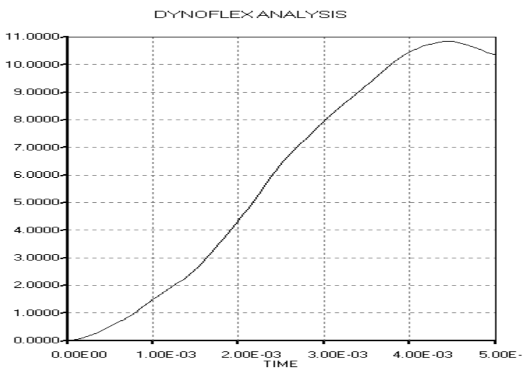




Result Case 1



Result Case 3

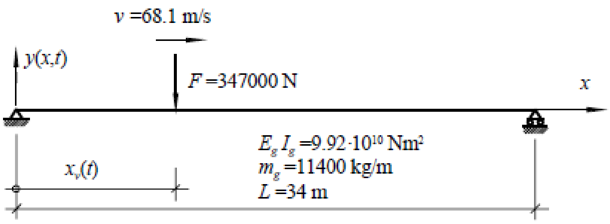


Result Case 3 (Linear – No tension stiffening)

Example 5.11 Response of a Moving Load on a SS Beam

Model: **Beam-MovingLoad**

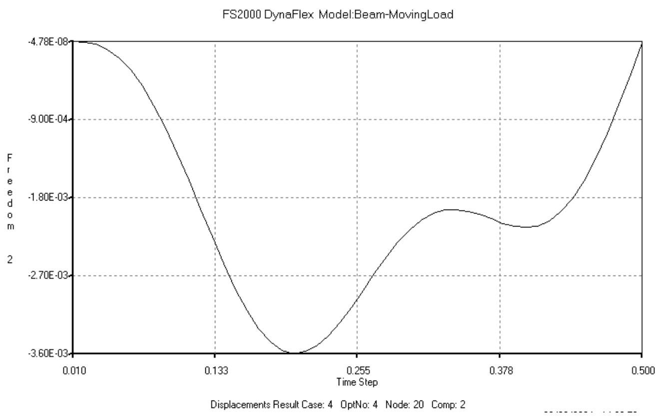
The example evaluates the response of a concentrated moving load as it passes along the span of ss beam. history solution.



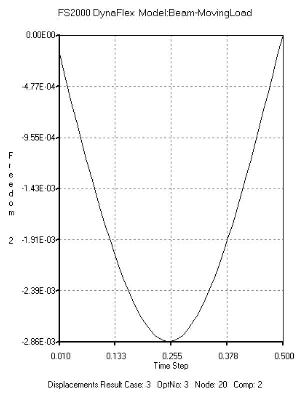
The SS beam was modelled using 34-1m long Type 6 beam elements.

The solution used a DyNoFlex linear time history and the Moving Load Generator.

Reference Solution: Response Of Cable-Stayed and Suspension Bridges to Moving Vehicles, TRITA-BKN, Bulletin 44, 1998

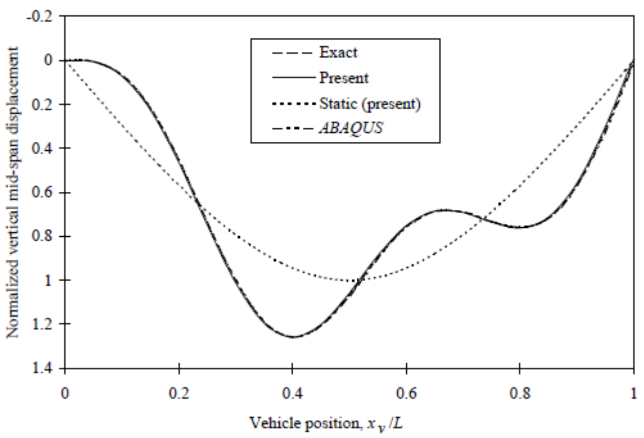


Dynamic



Static

DAF = 1.256

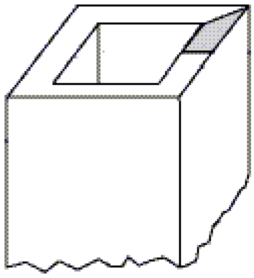


DAF plot from reference solution.

Example 6.1 Heat Conduction across a Chimney

Mode: **Chimney**

The is a model evaluates the temperature distribution across the chimney due to conduction and convection.

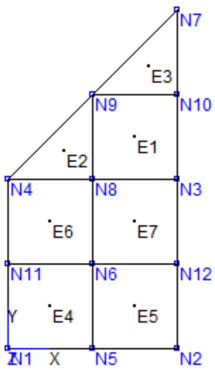


The chimney is 4ft square and 1ft thick.

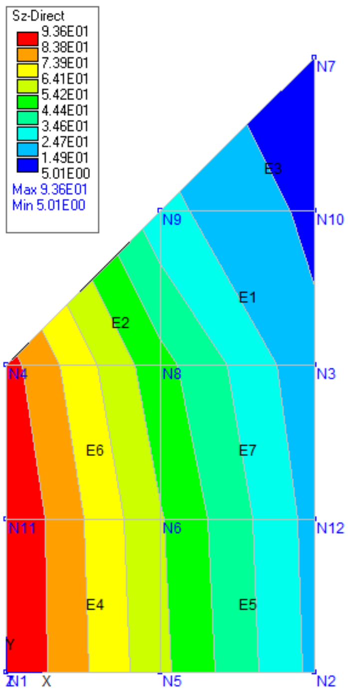
Flue Temperature = 100F; Extremal Temperature 0 F.

Thermal Conductivity 1 Btu/hr-ft-F
Internal Convection Coefficient = 12 Btu/hr-ft^2-F
External Convection Coefficient = 3 Btu/hr-ft^2-F

Reference Solution: ANSYS verification model VM100



Taking advantage of geometry and load symmetry only an 1/8 section is modelled.



Identical contour display as the reference solution.

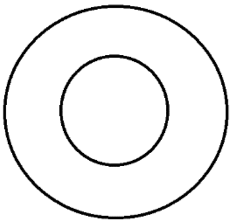
Maximum wall Temp = 93.6F (93.6)

Minimum Wall Temp = 5.01F (5)

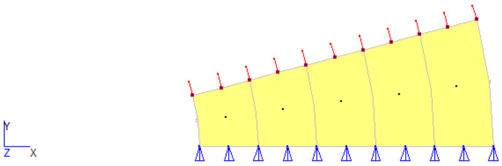
Example 6.2 Cylinder Heat Conduction and Thermal Stresses

Model: **CylinderThermal**

A thick cylinder has defined internal and external wall temperatures. The objective is to establish a stress distribution due to thermal stresses. A heat transfer solution establishes the temperature distribution, and a stress solution then establishes the stress distribution.



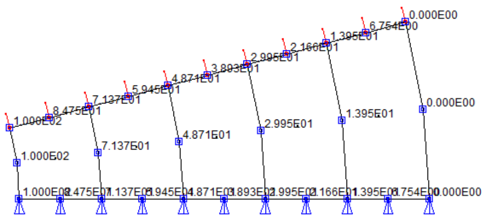
OD = 500mm
ID = 200mm
E =205GPa; v = 0.3; Thermal Expansion Coefficient = 1E-5
Inside wall temperature = 100 C
Outside wall temperature = 0 C
Thermal conductivity = any non zero value (wall temperatures defined).

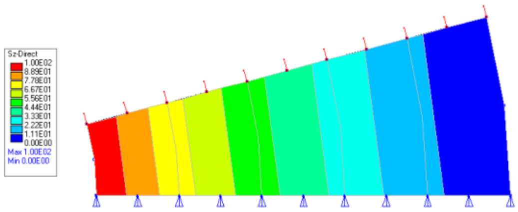


The model uses Type30 plane strain elements. Only a 15 degree segment is modelled. Ground couple referenced to a cylindical coordinate system are used to provide tangential restraint.

Reference Solution: S. Timoshenko, Strength of Material, Part II, Advanced Theory and Problems, 3rd Edition, D. Van Nostrand Co., Inc., New York, NY, 1956, pg. 232,article 44.

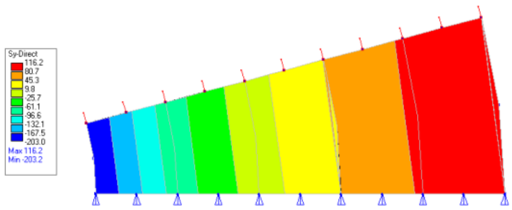
Case 1 - Temperature Distribution from the Heat Transfer Solution.





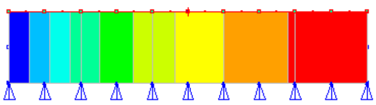
Case 2 - Thermal Stress Distribution from the Stress Solution

The refence solution values are shown in parentheses.



Inner Hoop = 203MPa (207)

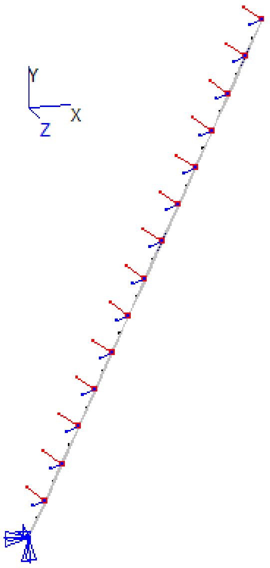
Outer Hoop = 116 MPa (114)



Hoop stress contors from a model that used Type 40 axisymmetric elements for a solution to the same problem (Ver-Example FEExp11).

Example 7.1 Hydrodynamic Wave Loading on a Marine Riser

Model:**WaveLoad2**

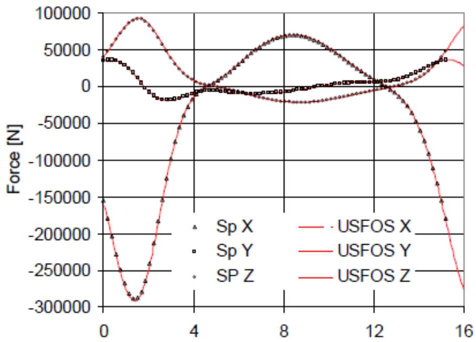


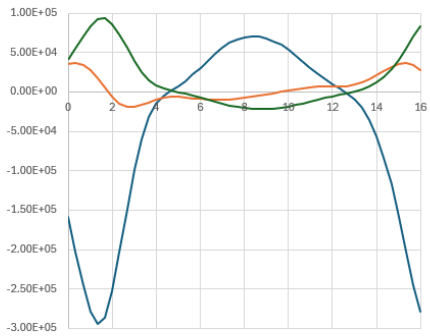
This an example of an inclined riser subjected to hydrodynamic wave loading. Stokes 5[th] Order wave theory is used to evaluate wave motions. This model is based on the overall proportions of that in the reference solution.
The reference solution only evaluated the wave loading and divides the riser into 100 elements and uses 10 subdivisions per element for wave load evaluation. To be more realistic the riser is divided into 14 elements. This gives an element length of about 7m which is a typical of a span length for a riser of this diameter.

The riser is guided at each node using ground couple elements aligned to the riser axis. A deadweight support is provided just above STW.

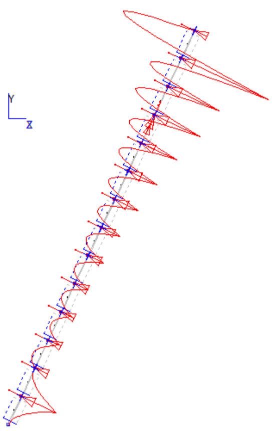
Riser Base: x=0; y=-70; z=0.
Riser Top: : x=30; y=20; z=30.
OD = 200mm; Cd = 1; Cm = 2
STW at y=0; Depth = 70m
H = 30m; T = 15s.
Buoyancy effected are neglected.

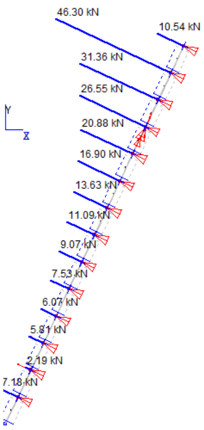
Reference Solution: USFOS, Theory Description of use Verification (no structural solution).
The following show a comparison of loading due to drag only.





ABS Drag Only : Max X=292.7kN (291.5) Max Y=37kN (36.48) Max Z=93.11kN (92.9)
ABS Inertia Only: Max X=8.6kN (8.89) Max Y=2.844kN (3.12) Max Z=3.093kN (3.44)





Major axis Bending Stress and Guide Reactions

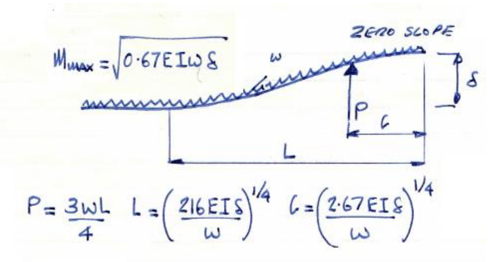
Structural Global Reaction summaries.
RC 10 Drag only.
RC 11 Inertia only.
RC 12 Drag & Inertia.
Note that Case 10 and 11 are at different phases.

RC	Fx kN	Fy kN	Fz kN
10	-292.72	90.65	20.76
11	8.59	-2.50	-1.09
12	-292.85	91.72	17.68

Example 7.2 The Lifting of a Pipeline – Foundation Contact

Model: **PipelineLift**

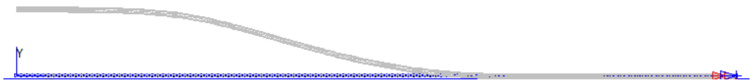
This an example of a pipeline lift in which a single point lift is used to raise the end of a pipe a specific height and maintain a zero slope at the end. The model use Type 7 beam elements. These elements are supported on a distributed foundation stiffness which provides surface contact restraint.
Reference Solution: The following equations can be derived from Macaulay's beam method.



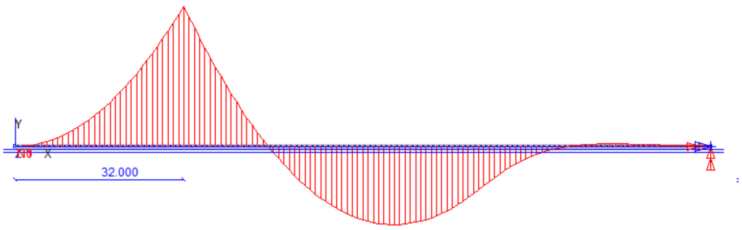
For this example:
W = 555.86kg/m
E =203.4GPa
I = 7.014E-3m^4
δ = 1.5
This gives: L=95.9m; l = 31.98m; M_{MAX} = 2799kNm; P = 393kN.

The model properties represent a 36"OD 1"Wall being lifted in air when supported on a sand foundation. The model used a DyNoFlex time history solution. The weight is applied and then the pipe is raised. Could be done in one step.

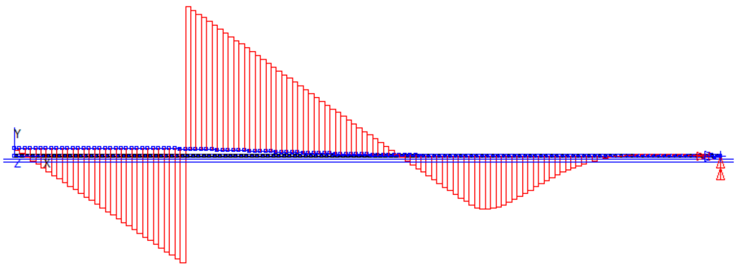
The solution results are summarised below. These results are considered more accurate than the Macaulay approach because the Macaulay assumes unrealistic boundary conditions at touchdown (zero slope, zero moment and zero displacement).



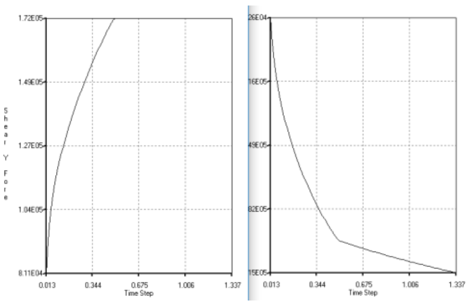
δ = 1507mm
θ = 0.0012 rads



M_{MAX} = 2797kNm



P = 387kN



Shear at the LHS and RHS of the lift point. Once the LHS is lifted clear the LHS shear force remains constant.

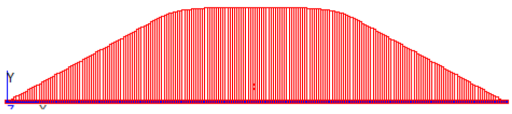
Example 7.3 Thermal Expansion of Buried Pipeline

Model: **PipelineExpansion**

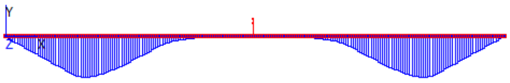
This is an example of thermal expansion and thermal cycling of a buries pipeline. The model represents a 10km concrete coated pipeline (512mm OD x 11mm wt) buried to a depth of 1m. The operational thermal differential of 65C is applied. The soil stiffness characteristics are based on ASCE's "Guideline for the Design of Buried Steel Pipe". The evaluation of the soil springs used FS200's Pipeline Properties utility.

The model uses a Type8 beam element. Type 7 non-linear couples are used to provide an axial kinematic bi-linear soil spring. The loading is applied in two stages. Gravity then thermal. Note that the gravity case is not actually required the expansion case – resistance is from the soil spring.
Reference Solution: Sample Calculation.

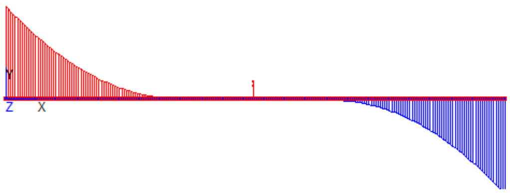
The apparent anchor for is given by : L = α.ΔT.E.A / (μ.w) μ.w = Resistance = 6.6kN/m for this buried pipe.
For ΔT = 65 the L = 379.2m. The locked in force is F = α.ΔT.E.A = 2503kN. This assumes rigid friction.



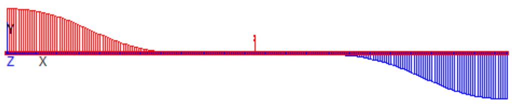
Force at 65C - 2499 kN Case 100
Because of frictional mobilisation no distinct anchor.
379m is within the mid curve portion.



Residual Force at 0C - 1064 kN (max at E35) Case 101

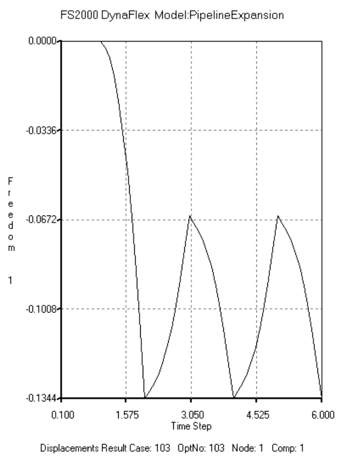


Initial End displacement at 65C - 134.4mm Case 100

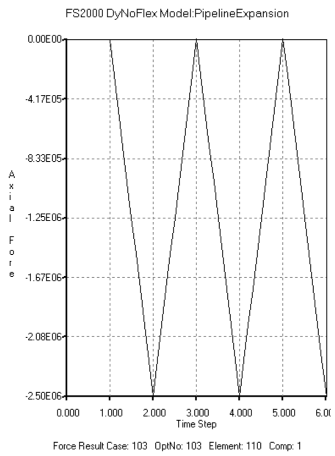


Residual End displacement - 65.7mm Case 101

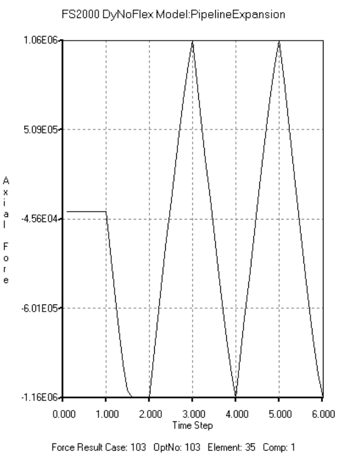
End



Displacement



Mid Force



Force at E35

Example 7.4 Lateral Bucking of a Pipeline– Large Displacement - Contact

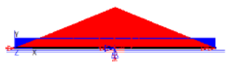
Model: **Pipeline_Buckling**

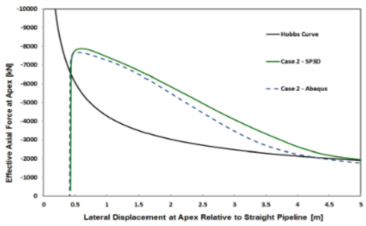
This is an example of pipeline buckling (elastic-plastic) due to axial compression due to thermal and pressure effects. The model represents a 10km pipeline (767.4mm OD x 33.7mm wt) resting on the seabed. The pipeline has an out of straightness of 500mm at the mid-section, defined as an undeformed shape. The operational temperature and pressure differential of 60C and 30MPa is applied.

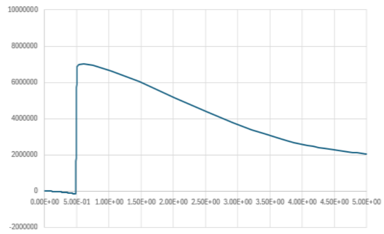
Reference Solution: AN INTEGRATED NUMERICAL APPROACH TO DESIGN OFFSHORE PIPELINES SUSCEPTIBLE TO LATERAL BUCKLING, OMAE2015-42119. No information on the initial lateral shape only a magnitude of 0.5m given.

The local mesh density in the FS2000 model was twice that of that in the reference solution at the prop location. No details of the material model used were available, but the deflection and curvature plots would indicate a plastic solution was undertaken. FS2000 used a commonly used Ramsberg-Osgood relationship for plasticity and a VM failure criteria.

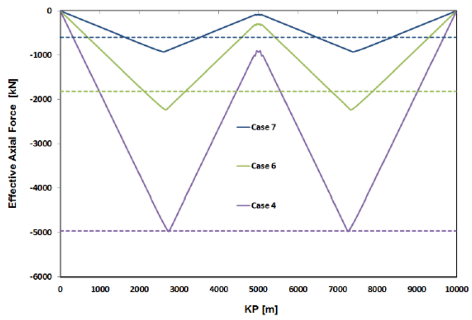
Case 101 Effective axial force of 9284 kN (compares well with Case 1 from the reference plots)

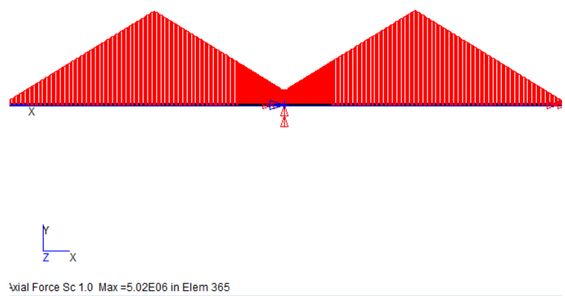




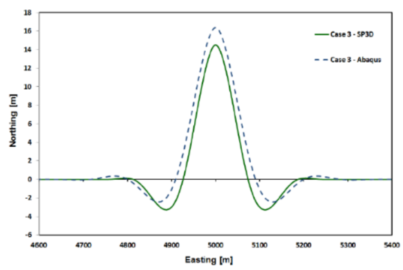


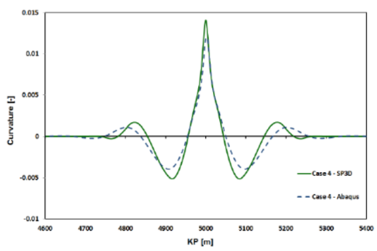
Case 102 Apex buckling load 7040kN. The axial force vs lateral displacement plot shows similar trends.





Case 104 Show similar post buckles axial load distribution 5038kN max and 826kN at buckle.





Case 103 Max Lat Disp 15.6 m.

Case 104 Curvature 0.0166

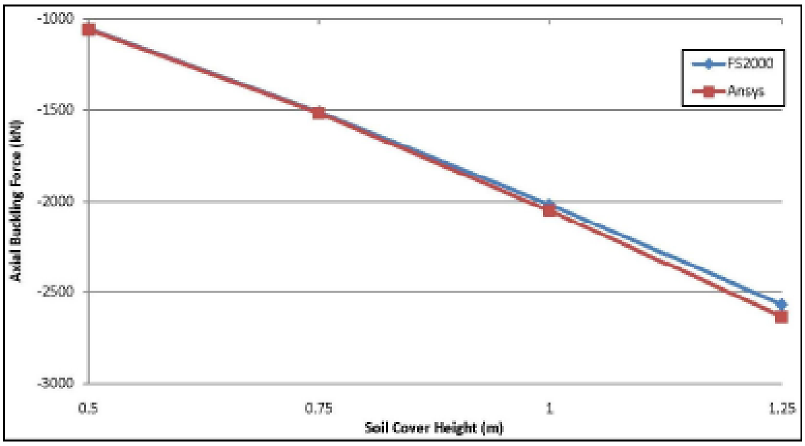
The distributions obtained are very similar. It should be noted that these results are very close considering that an elastic solution produces very different results. Effective plastic strains of 0.615% were obtained from this solution case. The plasticity is evident from the peaked appearance at the location of the maximum.

Example 7.5 Upheaval Bucking of a Pipeline – Large Displacement - Contact

Model: **PipelineUHB**

This is an example of pipeline buckling (UHB) due to axial compression due to thermal and pressure effects. The model represents an 8" buried pipeline with initial vertical upward out straightness. The objective of the model is to assess the susceptibility to Euler buckling for defined cover heights.
Reference Solution: Third party verification of FS2000 and ANSYS for this type of problem.

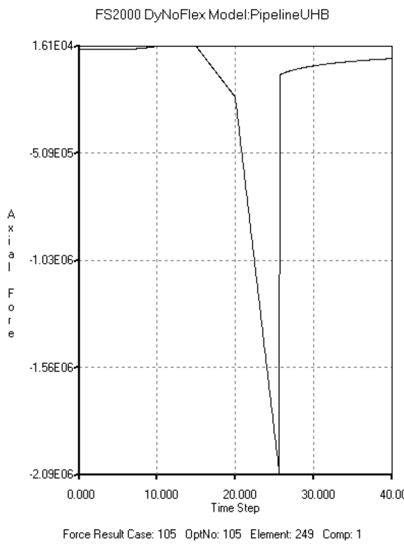
Figure 5.11 – Results Comparison – 0.5m Imperfection



The model was generated using FS2000's Pipeline Properties Utility (not exactly the same FS2000 model used by the third party.

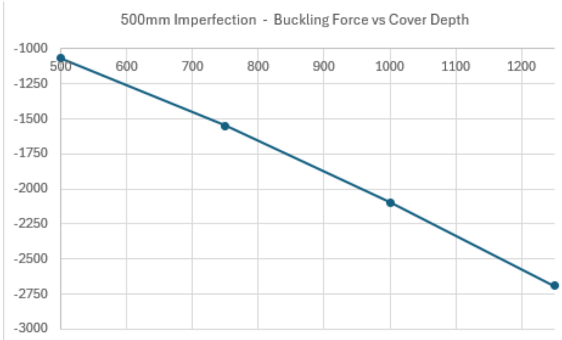
For comparison a 500mm Imperfection with 1000mm Top Cover case was used. This is Case 105 at the top of the Batch File.

A static DyNoFlex time history solution is used to obtain the buckling load. The solution starts with an initial straight pipe, imposes the imperfection, applies a top cover and ramps up the pressure and temperature.



This axial plot indicates that buckling occurs when the axial reaches 2090kN. This compares favourably with that shown in the above verification reference.

The plot below shows the variation with other cover heights, again compares favourably the reference solution.

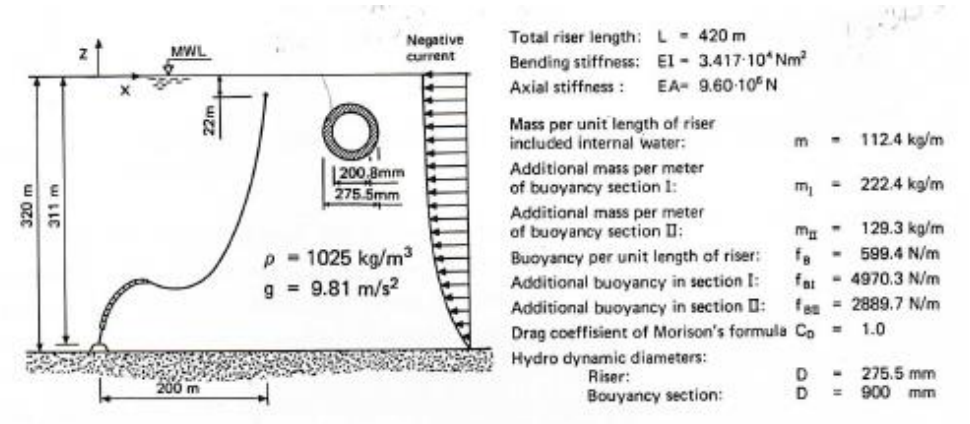


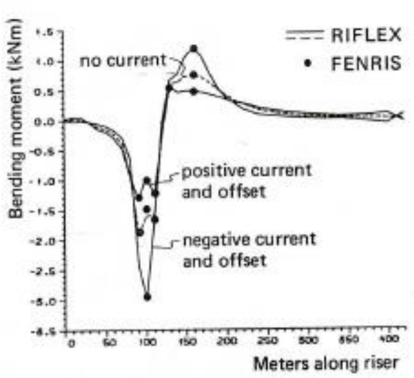
Example 7.6 Static Analysis of a Steep Wave Riser Configuration – DyNoFlex/FS-Wave

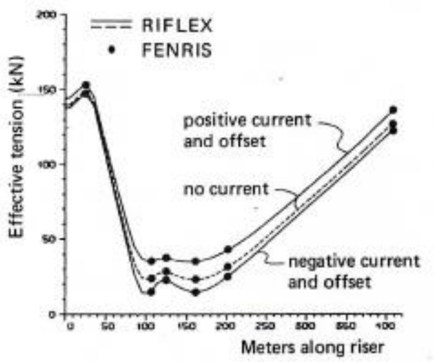
Model: **SteepWaveRiser**

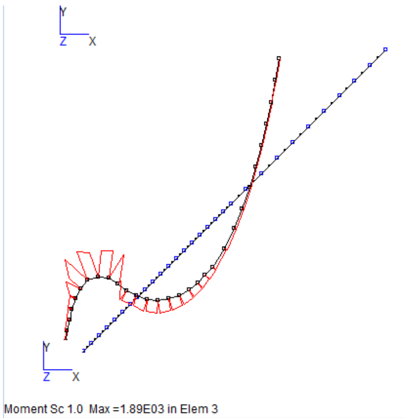
This example evaluates the shape and loading in a riser. The model uses Type 16(8) moment curvature beams (suited for tension dominated flexible structures). Three cases: still water, positive current and negative current. Top offsets of 25.7m applied with currents. DyNoFlex time history solution. Same mesh density as described in the reference solution was used (considered a bit coarse).

Reference Soln: Efficient Method for Analysis of Flexible Risers, BOSS(Behaviour of Offshore Structures), Carl M. Larsen et al.









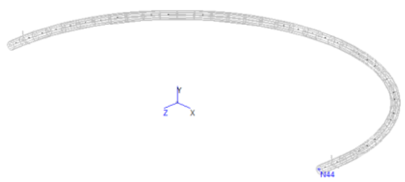
Moments	FS2000	Ref
Mid	1.89	1.9
+veCurr	1.33	1.3
-veCurr	3.04	2.45

Top Tens	FS2000	Ref
Mid	1.28	1.28
+veCurr	1.37	1.37
-veCurr	1.24	1.22

Example 7.7 Dynamic Slugging Flow in a Horizontal Pipe Loop

Model: **Pipe_Bend_Flow**

This is a model of 180-degree horizontal pipe loop. The pipe has a two-phase slugging flow regime.



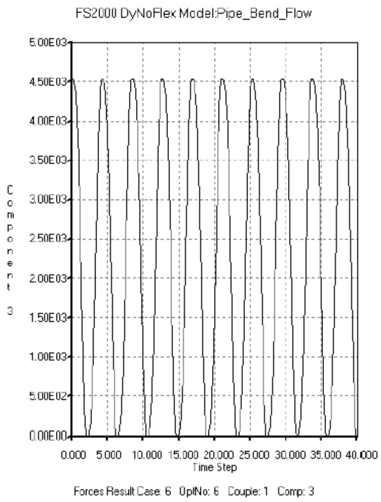
Liquid Phase 30m long 42 kg/m
Vapour Phase 4 5m long 8 kg/m
Velocity 10 m/s

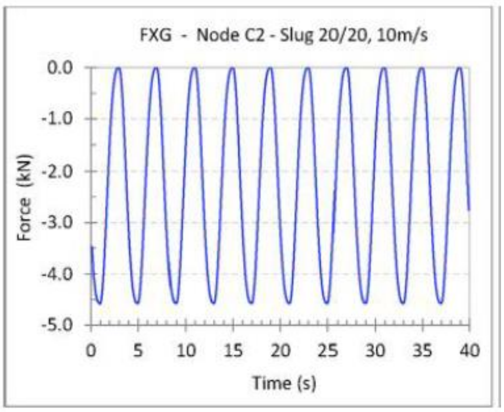
OD=280mm t=20mm Radius=6.366m

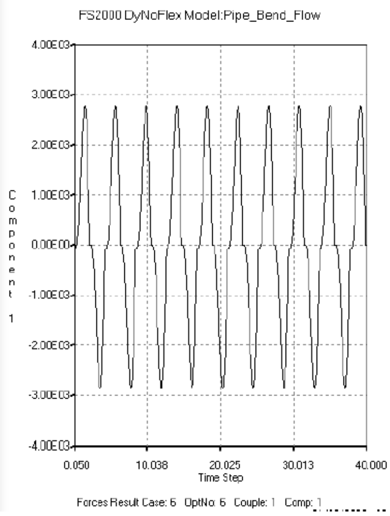
Reference Solution: "Slug flow induced oscillations on subsea petroleum pipelines", Sergio N. Bordalo , Celso K. Morooka, Journal of Petroleum Science and Engineering(2018).

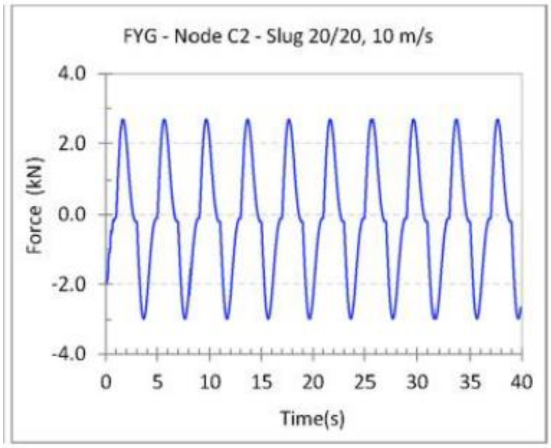
The bend is formed by segmented straight Type 6 pipe elements. The dynamic loading is generated using FS2000's moving load generator. This generator enables gravitational and inertial loads to be evaluated from a train of distributed or concentrated moving loads. A DyNoFlex dynamic time history solution is employed to obtain the response.

The plots below show a comparison of the horizonal (x & z) reactions at the supports for the inertia loads.









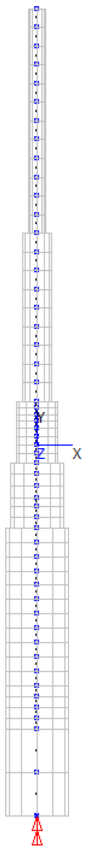
Example 7.8 Heave Frequency of a Floating Column

Model: HyWind

The model is an extract from a larger model which was used to simulate the movement of a 3-line moored column (wind turbine) under wave action. It has been simplified to evaluate only the heave frequency.

The solution is obtained using a DyNoFlex dynamic time history solution. The hydrodynamic data is defined using FS-Wave (only Stillwater used).

Reference Solution: Basic Theory - Heave Period = $2\pi(m/k)^{0.5}$



The diameter of the column varies from 14.4m to 4.1m.
The mass of the structure is 13,333 tonnes (steel weight and distributed ballast).

The total buoyancy is 13,068 tonnes when in its initial position. This when the N28 is at STW i.e. the Y origin.

The model has no restraints, and equilibrium will only exist when the column buoyancy force = gravitational force.

The added mass coefficient is set to zero for purely axial movement. Rayleigh Damping Coefficients provide 10% damping at 25s to 30s periods.

The diameter at the interface is 9.5m.
Floating stiffness $k = \rho gA = 1027*9.81*\pi*D^2/4 = 714$kN/m

Heave frequency = $2\pi(m/k)^{0.5} = 27.16$s

The initial transient from the application of the loads indicates final displacement of about 3.64m i.e. as it sinks to its mean SWL. The period of oscillation in the plot is in the region of 27.15s