ELEMENT IRREDUCIBLE TYPE H8P0 command

Synopsis

The ELEMENT IRREDUCIBLE H8P0 command is used to describe all irreducible 8-node tri-linear hexahedral continuum elements that are to be used in mechanical analyses.

Syntax

The following syntax is used to describe a typical H8P0 irreducible hexahedral continuum element:

```
ELEment IRReducible TYPe H8P0 NODes #:#:#
   (MATerial #) (INItial #) (INTcode #)
   (CONstruction #) (EXCavation #)
   (1_Additional #) (1_Increment #)
   (2_Additional #) (2_Increment #)
   (3_Additional #) (3_Increment #)
   (DONT_PRINT_Results)
   (DONT_PRINT_STRAins) (DONT_PRINT_STREsses)
   (PRINT_PRIN_STRAins) (PRINT_PRIN_STREsses)
   (PRINT_VOLumetric_strain)
```
Explanatory Notes

• The H8P0 is an irreducible, tri-linear, isoparametric hexahedral continuum element [2]. The element
  – Contains eight (8) vertex nodes.
  – Has three (3) displacements degrees of freedom at each node.
  – Possesses a total of twenty-four (24) displacement degrees of freedom.

• The numbering order of NODES associated with H8P0 elements, which must be specified sequentially from 1 to 8, is shown in Figure 1.

NOTE: Presently APES does not possess the ability to generate H8P0 elements. It is assumed that the analyst will thus use some stand-alone pre-processing software to accomplish this task. The resulting element and node data will then be translated to the format expected by APES.

![Figure 1: Node Numbering Associated with a Typical Irreducible 8-Node Hexahedral Trilinear Continuum Element](image)

• The MATERIAL keyword is used to specify the number of the material idealization associated with the element. The default values for the MATERIAL number is one (1).

• The INITIAL keyword is used to specify the initial state number associated with the element. The default value for the INITIAL is zero (0).

• The value specified in conjunction with the INTCODE keyword describes the order of numerical integration scheme to be used in developing the element equations for the element.
The “commonly” used numerical integration rule for H8P0 elements corresponds to a 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) for the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) for the secondary dependent variables (i.e., strains and stresses). This is the default condition and requires no input using the INTCODE keyword. If a quadrature order different from the default condition is desired, the following integer values are associated with this keyword:

**INTCODE = 11:** A 1-point Gauss-Legendre quadrature scheme (degree of precision equal to 1) is used to compute both the primary dependent variables (i.e., nodal displacements) and for the secondary dependent variables (i.e., strains and stresses).

This serves to uniformly under integrate the element. The disadvantage of uniform reduced integration is that the rank of the element stiffness matrix $K^{(e)}$ may be reduced, resulting in the singularity or near singularity of the global $K$ [2]. The associated element instabilities manifest themselves in the form of mechanisms or hourglass modes [1]. Consequently, the specification of INTCODE = 11 should only be used to investigate the possible pathologies associated with uniform reduced integration and not in standard analyses.

**INTCODE = 21:** A 2 by 2 by 2 Gauss-Legendre quadrature scheme (degree of precision equal to 3) is used to compute the primary dependent variables (i.e., nodal displacements) and a 1-point Gauss-Legendre scheme (degree of precision equal to 1) is used to compute the secondary dependent variables (i.e., strains and stresses). This is equivalent to the aforementioned default setting.

- The incremental CONSTRUCTION and EXCAVATION numbers represent the time increment in which the material in this element(s) is added to or removed from the model. A CONSTRUCTION number equal to zero corresponds to a material in existence at the beginning of the analysis. Since this is the default condition, no input is required in such a case. The condition of no excavation is likewise the default.

- If the body being analyzed can be divided into a layer (or layers) of elements, and if the characteristics of the element (i.e., the MATERIAL, the INITIAL state, the incremental CONSTRUCTION and EXCAVATION numbers, the THICKNESS, and the INTCODE) are the same for several elements within a layer, and if the nodes are numbered in a consistent fashion, then an element data generation option can be employed. To generate a sequence of elements within a single layer, node numbers are specified only for the first element, together with appropriate values for 1_ADDITIONAL and 1_INCREMENT. If several layers of elements have the same attributes, the above generation option can be carried one step further by making use of the 2_ADDITIONAL, and 2_INCREMENT keywords, and possibly the 3_ADDITIONAL, and 3_INCREMENT keywords.

- The purpose of the PRINT commands is to eliminate unnecessary output generated by APES. More precisely, if the time history of strains and/or stresses is desired only for a select few elements, this option greatly speeds program output and facilitates inspection of
results by the user. Information associated with the elements specified in this section will
be printed for every solution (time) step. If generation is performed using this **ELEMENT IRREDUCIBLE** command, then all the elements generated will be affected in a like manner
by the above print control commands.

- Specification of the keyword **DONT_PRINT_Results** indicates that the analyst does not
desire to see output of secondary dependent variables (i.e., strains and stresses) for this ele-
ment.

- Specification of the **DONT_PRINT_STRAINS** keyword indicates that element strains are
not to be printed. Under the *default* condition both strains are printed.

- Specification of the keyword **DONT_PRINT_STRESSES** indicates that stresses are not to
be printed. Under the *default* condition stresses are printed.

- The **PRINT_PRIN_STRAINS** keyword indicates that principal strains are to be computed
and printed for the element. Under the *default* condition these quantities are not computed
and printed.

- The **PRINT_PRIN_STRESSES** keyword indicates that principal stresses are to be com-
puted and printed for the element. Under the *default* condition these quantities are not
computed and printed.

- The keyword **PRINT_VOLUMETRIC_STRAIN** causes the volumetric strain to be com-
puted and printed for the element. In addition, the ratio of the absolute value of the volumetric
strain to the absolute value of the minimum non-zero normal strain in the element is printed.
That is,

\[
\frac{|\varepsilon_{\text{vol}}|}{\min (\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33})} ; \quad \min (\varepsilon_{11}, \varepsilon_{22}, \varepsilon_{33}) \neq 0
\]

This ratio is instructive in the assessment of mixed and mixed/penalty elements used to
simulate material response in the incompressible limit. As such, this keyword would likely *not*
be used in conjunction with the **H8P0** element. Under the *default* condition the volumetric
strain and the aforementioned ratio are *not* computed and printed.
Example of Command Usage

Element Performance in Simple “Patch” Test

In order to verify the implementation of the \textbf{H8P0} elements, the simple “patch” of elements shown in Figure 2 is analyzed. The geometry of the patch is taken from the set of test problems proposed by MacNeal and Harder [3]. Further details pertaining to the so-called engineering patch test are given in Appendix D of [2].

The solution domain is a unit cube. Following MacNeal and Harder [3], the material is idealized as being isotropic linear elastic. The elastic modulus is equal to \(1.0 \times 10^6\) and a Poisson’s ratio equal to 0.25 is assumed. A minimum number of nodal specifications are made so as to constrain displacement of the body and prevent rigid body translations and rotations. A tensile stress of 1000.0 is applied in the \(x\)-direction via suitable nodal forces. In a similar manner, a compressive stress of 2000.0 is applied in the \(y\)-direction. Finally, a tensile stress of 3000.0 is applied in the \(z\)-direction.

\begin{figure}[h]
\centering
\includegraphics[width=0.6\textwidth]{patch.png}
\caption{Simple “Patch” of Trilinear (\textbf{H8P0}) Hexahedral Elements}
\end{figure}

The input data associated with this problem is given below.
analysis title "patch test 'C' for hexahedral mesh after MacNeal & Harder (1985)"
analysis title "the applied stress generates the following stress state throughout the patch:
analysis title "\( \sigma_{11} = 1000.0 \) (tensile stress)"
analysis title "\( \sigma_{22} = 2000.0 \) (compressive stress)"
analysis title "\( \sigma_{33} = 3000.0 \) (tensile stress)"

! echo func off
echo grav off
echo ini off
echo warn off
!
anal act analyze
anal type mechanical
anal description linear
anal idealization three-dimensional
anal temporal transient
!
dim max material isotropic elastic 1
dim max nodes 16
dim max h8p0 8
!
fin sett
!
mat elastic isotropic number 1 desc "sample isotropic material" &
    modulus 1.0e+06 Poissons 0.25
!
nodes line number 1 x1 0.249 x2 0.342 x3 0.192
nodes line number 2 x1 0.826 x2 0.288 x3 0.288
nodes line number 3 x1 0.850 x2 0.649 x3 0.263
nodes line number 4 x1 0.273 x2 0.750 x3 0.230
nodes line number 5 x1 0.320 x2 0.186 x3 0.643
nodes line number 6 x1 0.677 x2 0.305 x3 0.683
nodes line number 7 x1 0.788 x2 0.693 x3 0.644
nodes line number 8 x1 0.165 x2 0.745 x3 0.702
nodes line number 9
nodes line number 10 x1 1.000
nodes line number 11 x1 1.000 x2 1.000
nodes line number 12 x2 1.000
nodes line number 13 x3 1.000
nodes line number 14 x1 1.000 x3 1.000
nodes line number 15 x1 1.000 x2 1.000 x3 1.000
nodes line number 16 x2 1.000 x3 1.000
!
element irreducible type "h8p0" nodes 15 16 13 14 7 8 5 6 mat 1
element irreducible type "h8p0" nodes 6 14 15 7 2 10 11 3 mat 1
element irreducible type "h8p0" nodes 1 2 3 4 9 10 11 12 mat 1
element irreducible type "h8p0" nodes 16 8 7 15 12 4 3 11 mat 1
element irreducible type "h8p0" nodes 8 16 13 5 4 12 9 1 mat 1
element irreducible type "h8p0" nodes 8 5 6 7 4 1 2 3 mat 1

! spe conc mec nod 9 1_dis 2_dis 3_dis
spe conc mec nod 10 1_force 1_val 250.0 1_his 0 2_dis 3_dis
spe conc mec nod 11 1_force 1_val 250.0 1_his 0 2_force 2_val -500.0 2_his 0 3_dis
spe conc mec nod 12 1_dis 2_force 2_val -500.0 2_his 0 3_dis
spe conc mec nod 13 1_dis 2_dis 3_force 3_val 750.0 3_his 0
spe conc mec nod 14 1_force 1_val 250.0 1_his 0 2_dis & 3_force 3_val 750.0 3_his 0
spe conc mec nod 15 1_force 1_val 250.0 1_his 0 2_force 2_val -500.0 2_his 0 & 3_force 3_val 750.0 3_his 0
spe conc mec nod 16 1_dis 2_force 2_val -500.0 2_his 0 3_force 3_dis

! finish data
!
solution time final 1.0 increments 1 output 1:10:1
!
finished loading

The results shown below are obtained using the above data in conjunction with the APES computer program. For clarity, the “header” that is printed at the top of the file is omitted from this file.
patch test 'C' for hexahedral mesh after MacNeal & Harder (1985)
the applied stress generates the following stress state throughout the patch:
sig_11 = 1000.0 (tensile stress)
sig_22 = 2000.0 (compressive stress)
sig_33 = 3000.0 (tensile stress)

======================================================================
| DYNAMIC STORAGE ALLOCATION |
======================================================================

Largest NODE number which can be used in the mesh = 16
Max. no. of ISOTROPIC, LINEAR ELASTIC materials = 1
Max. no. of 8-node hexahedral (H8PO) elements = 8

======================================================================
= GENERAL ANALYSIS INFORMATION =
======================================================================

--> MECHANICAL analysis shall be performed
--> Fluid flow is NOT accounted for in the analysis
--> Thermal effects are NOT accounted for in analysis

--> THREE-DIMENSIONAL solution domain assumed
--> Nodal coordinates will NOT be updated
--> solver type used: SKYLINE

--> storage type: SYMMETRIC

--> "Isoparametric" scheme used for native mesh generation (if applicable)

======================================================================
= INTEGRATION OPTIONS =
======================================================================

In approximating time derivatives, the value of "THETA" = 6.667E-01
= NONLINEAR ANALYSIS INFORMATION =

--> LINEAR analysis

= MATERIAL IDEALIZATIONS =

--> Material number:  1

---------------
type : isotropic linear elastic
info. : sample isotropic material

Modulus of Elasticity = 1.000E+06
Poisson’s ratio = 2.500E-01

Elastic bulk modulus of the solid phase = 0.000E+00
Material density of the solid phase = 0.000E+00
Combined bulk modulus for solid/fluid = 0.000E+00

= NODAL COORDINATES =

node :  1  x1 = 2.490E-01  x2 = 3.420E-01  x3 = 1.920E-01
node :  2  x1 = 8.260E-01  x2 = 2.880E-01  x3 = 2.880E-01
node :  3  x1 = 8.500E-01  x2 = 6.490E-01  x3 = 2.630E-01
node :  4  x1 = 2.730E-01  x2 = 7.500E-01  x3 = 2.300E-01
node :  5  x1 = 3.200E-01  x2 = 1.860E-01  x3 = 6.430E-01
node :  6  x1 = 6.770E-01  x2 = 3.050E-01  x3 = 6.830E-01
node :  7  x1 = 7.880E-01  x2 = 6.930E-01  x3 = 6.440E-01
node :  8  x1 = 1.650E-01  x2 = 7.450E-01  x3 = 7.020E-01
node :  9  x1 = 0.000E+00  x2 = 0.000E+00  x3 = 0.000E+00
node : 10  x1 = 1.000E+00  x2 = 0.000E+00  x3 = 0.000E+00
node : 11  x1 = 1.000E+00  x2 = 1.000E+00  x3 = 0.000E+00
node : 12  x1 = 0.000E+00  x2 = 1.000E+00  x3 = 0.000E+00
node : 13  x1 = 0.000E+00  x2 = 0.000E+00  x3 = 1.000E+00

9 V. N. Kaliakin
node : 14 x1 = 1.000E+00 x2 = 0.000E+00 x3 = 1.000E+00
node : 15 x1 = 1.000E+00 x2 = 1.000E+00 x3 = 1.000E+00
node : 16 x1 = 0.000E+00 x2 = 1.000E+00 x3 = 1.000E+00

======================================================================
= ELEMENT INFORMATION =
======================================================================

--> number: 1 (type: H8P0 ) (kind: IRREDUCIBLE )
nodes: 15 16 13 14 7 8 5 6
integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

--> number: 2 (type: H8P0 ) (kind: IRREDUCIBLE )
nodes: 6 14 15 7 2 10 11 3
integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

--> number: 3 (type: H8P0 ) (kind: IRREDUCIBLE )
nodes: 1 2 3 4 9 10 11 12
integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

--> number: 4 (type: H8P0 ) (kind: IRREDUCIBLE )
nodes: 16 8 7 15 12 4 3 11
integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

--> number: 5 (type: H8P0 ) (kind: IRREDUCIBLE )
nodes: 8 16 13 5 4 12 9 1
integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

--> number: 6 (type: H8P0 ) (kind: IRREDUCIBLE )
nodes:
1 5 6 2 9 13 14 10
integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

--> number: 7 (type: H8P0 ) (kind: IRREDUCIBLE )
nodes:
8 5 6 7 4 1 2 3
integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

======================================================================

= N O D E   P O I N T   S P E C I F I C A T I O N S =
======================================================================

Node Number ( c o o r d i n a t e s ) s p e c i f i c a t i o n:
------- -----------------------------

9 : ( x1 = 0.000E+00, x2 = 0.000E+00, x3 = 0.000E+00 )
displacement-1 = 0.000E+00 ; history no. = -2
displacement-2 = 0.000E+00 ; history no. = -2
displacement-3 = 0.000E+00 ; history no. = -2

10 : ( x1 = 1.000E+00, x2 = 0.000E+00, x3 = 0.000E+00 )
force-1 = 2.500E+02 ; history no. = 0
displacement-2 = 0.000E+00 ; history no. = -2
displacement-3 = 0.000E+00 ; history no. = -2

11 : ( x1 = 1.000E+00, x2 = 1.000E+00, x3 = 0.000E+00 )
force-1 = 2.500E+02 ; history no. = 0
force-2 = -5.000E+02 ; history no. = 0
displacement-3 = 0.000E+00 ; history no. = -2
12: ( x1 = 0.000E+00, x2 = 1.000E+00, x3 = 0.000E+00 )
  displacement-1 = 0.000E+00 ; history no. = -2
  force-2 = -5.000E+02 ; history no. = 0
  displacement-3 = 0.000E+00 ; history no. = -2

13: ( x1 = 0.000E+00, x2 = 0.000E+00, x3 = 1.000E+00 )
  displacement-1 = 0.000E+00 ; history no. = -2
  displacement-2 = 0.000E+00 ; history no. = -2
  force-3 = 7.500E+02 ; history no. = 0

14: ( x1 = 1.000E+00, x2 = 0.000E+00, x3 = 1.000E+00 )
  force-1 = 2.500E+02 ; history no. = 0
  displacement-2 = 0.000E+00 ; history no. = -2
  force-3 = 7.500E+02 ; history no. = 0

15: ( x1 = 1.000E+00, x2 = 1.000E+00, x3 = 1.000E+00 )
  force-1 = 2.500E+02 ; history no. = 0
  force-2 = -5.000E+02 ; history no. = 0
  force-3 = 7.500E+02 ; history no. = 0

16: ( x1 = 0.000E+00, x2 = 1.000E+00, x3 = 1.000E+00 )
  displacement-1 = 0.000E+00 ; history no. = -2
  force-2 = -5.000E+02 ; history no. = 0
  force-3 = 7.500E+02 ; history no. = 0

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
end of mathematical model data
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

At time 1.000E+00 (step no. 1): NO iteration was required

======================================================================
= E L E M E N T S T R A I N S & S T R E S S E S =
======================================================================

--> element 1 ( type = H8P0 ):

.................................
\( \begin{align*}
@ (x_1 = 4.938E-01, x_2 = 4.911E-01, x_3 = 8.340E-01): \\
\text{eps}_{11} &= 7.500E-04 \ ; \ \text{eps}_{22} = -3.000E-03 \ ; \ \text{eps}_{33} = 3.250E-03 \\
\text{gam}_{12} &= -9.659E-12 \ ; \ \text{gam}_{13} = 4.586E-12 \ ; \ \text{gam}_{23} = -1.113E-12 \\
\text{sig}_{11} &= 1.000E+03 \ ; \ \text{sig}_{22} = -2.000E+03 \ ; \ \text{sig}_{33} = 3.000E+03 \\
\text{sig}_{12} &= -3.864E-06 \ ; \ \text{sig}_{13} = 1.834E-06 \ ; \ \text{sig}_{23} = -4.451E-07 \\
\end{align*} \)

--> element 2 ( type = H8P0 ):

\( \begin{align*}
@ (x_1 = 8.926E-01, x_2 = 4.919E-01, x_3 = 4.848E-01): \\
\text{eps}_{11} &= 7.500E-04 \ ; \ \text{eps}_{22} = -3.000E-03 \ ; \ \text{eps}_{33} = 3.250E-03 \\
\text{gam}_{12} &= 2.012E-12 \ ; \ \text{gam}_{13} = -5.212E-12 \ ; \ \text{gam}_{23} = -2.342E-12 \\
\text{sig}_{11} &= 1.000E+03 \ ; \ \text{sig}_{22} = -2.000E+03 \ ; \ \text{sig}_{33} = 3.000E+03 \\
\text{sig}_{12} &= 8.048E-07 \ ; \ \text{sig}_{13} = -2.085E-06 \ ; \ \text{sig}_{23} = -9.368E-07 \\
\end{align*} \)

--> element 3 ( type = H8P0 ):

\( \begin{align*}
@ (x_1 = 5.248E-01, x_2 = 5.036E-01, x_3 = 1.216E-01): \\
\text{eps}_{11} &= 7.500E-04 \ ; \ \text{eps}_{22} = -3.000E-03 \ ; \ \text{eps}_{33} = 3.250E-03 \\
\text{gam}_{12} &= 4.631E-12 \ ; \ \text{gam}_{13} = -4.284E-12 \ ; \ \text{gam}_{23} = -3.809E-12 \\
\text{sig}_{11} &= 1.000E+03 \ ; \ \text{sig}_{22} = -2.000E+03 \ ; \ \text{sig}_{33} = 3.000E+03 \\
\text{sig}_{12} &= 1.853E-06 \ ; \ \text{sig}_{13} = -1.714E-06 \ ; \ \text{sig}_{23} = -1.523E-06 \\
\end{align*} \)

--> element 4 ( type = H8P0 ):

\( \begin{align*}
@ (x_1 = 5.095E-01, x_2 = 8.546E-01, x_3 = 4.799E-01): \\
\text{eps}_{11} &= 7.500E-04 \ ; \ \text{eps}_{22} = -3.000E-03 \ ; \ \text{eps}_{33} = 3.250E-03 \\
\text{gam}_{12} &= -1.136E-11 \ ; \ \text{gam}_{13} = -4.588E-12 \ ; \ \text{gam}_{23} = -1.155E-11 \\
\text{sig}_{11} &= 1.000E+03 \ ; \ \text{sig}_{22} = -2.000E+03 \ ; \ \text{sig}_{33} = 3.000E+03 \\
\text{sig}_{12} &= -4.543E-06 \ ; \ \text{sig}_{13} = -1.835E-06 \ ; \ \text{sig}_{23} = -4.622E-06 \\
\end{align*} \)

--> element 5 ( type = H8P0 ):

\( \begin{align*}
\end{align*} \)
\( \oplus (x_1 = 1.259E-01, x_2 = 5.029E-01, x_3 = 4.709E-01): \)
\[
\begin{align*}
\varepsilon_{11} &= 7.500E-04 ; \varepsilon_{22} = -3.000E-03 ; \varepsilon_{33} = 3.250E-03 \\
\gamma_{12} &= -2.672E-11 ; \gamma_{13} = 4.130E-11 ; \gamma_{23} = 1.186E-11 \\
\sigma_{11} &= 1.000E+03 ; \sigma_{22} = -2.000E+03 ; \sigma_{33} = 3.000E+03 \\
\sigma_{12} &= -1.069E-05 ; \sigma_{13} = 1.652E-05 ; \sigma_{23} = 4.742E-06
\end{align*}
\]

--> element 6 ( type = H8P0 ): .................................................................

\( \oplus (x_1 = 5.090E-01, x_2 = 1.401E-01, x_3 = 4.758E-01): \)
\[
\begin{align*}
\varepsilon_{11} &= 7.500E-04 ; \varepsilon_{22} = -3.000E-03 ; \varepsilon_{33} = 3.250E-03 \\
\gamma_{12} &= -4.682E-12 ; \gamma_{13} = -2.211E-12 ; \gamma_{23} = -1.147E-11 \\
\sigma_{11} &= 1.000E+03 ; \sigma_{22} = -2.000E+03 ; \sigma_{33} = 3.000E+03 \\
\sigma_{12} &= -1.873E-06 ; \sigma_{13} = -8.844E-07 ; \sigma_{23} = -4.589E-06
\end{align*}
\]

--> element 7 ( type = H8P0 ): .................................................................

\( \oplus (x_1 = 5.185E-01, x_2 = 4.948E-01, x_3 = 4.556E-01): \)
\[
\begin{align*}
\varepsilon_{11} &= 7.500E-04 ; \varepsilon_{22} = -3.000E-03 ; \varepsilon_{33} = 3.250E-03 \\
\gamma_{12} &= 7.708E-12 ; \gamma_{13} = -1.229E-11 ; \gamma_{23} = 1.833E-11 \\
\sigma_{11} &= 1.000E+03 ; \sigma_{22} = -2.000E+03 ; \sigma_{33} = 3.000E+03 \\
\sigma_{12} &= 3.083E-06 ; \sigma_{13} = -4.916E-06 ; \sigma_{23} = 7.332E-06
\end{align*}
\]

At time 1.000E+00 (step no. 1):

======================================================================
= N O D A L Q U A N T I T I E S =
======================================================================

node: 1 ( x1 = 2.490E-01, x2 = 3.420E-01, x3 = 1.920E-01 )
\[ u_1 = 1.868E-04, u_2 = -1.026E-03, u_3 = 6.240E-04 \]

node: 2 ( x1 = 8.260E-01, x2 = 2.880E-01, x3 = 2.880E-01 )
\[ u_1 = 6.195E-04, u_2 = -8.640E-04, u_3 = 9.360E-04 \]
<table>
<thead>
<tr>
<th>Node</th>
<th>x1</th>
<th>x2</th>
<th>x3</th>
<th>u_1</th>
<th>u_2</th>
<th>u_3</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>8.500E-01</td>
<td>6.490E-01</td>
<td>2.630E-01</td>
<td>6.375E-04</td>
<td>-1.947E-03</td>
<td>8.548E-04</td>
</tr>
<tr>
<td>4</td>
<td>2.730E-01</td>
<td>7.500E-01</td>
<td>2.300E-01</td>
<td>2.048E-04</td>
<td>-2.250E-03</td>
<td>7.475E-04</td>
</tr>
<tr>
<td>5</td>
<td>3.200E-01</td>
<td>1.860E-01</td>
<td>6.430E-01</td>
<td>2.400E-04</td>
<td>-5.580E-04</td>
<td>2.090E-03</td>
</tr>
<tr>
<td>8</td>
<td>1.650E-01</td>
<td>7.450E-01</td>
<td>7.020E-01</td>
<td>1.238E-04</td>
<td>-2.235E-03</td>
<td>2.282E-03</td>
</tr>
<tr>
<td>9</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>2.320E-23</td>
<td>-4.582E-23</td>
<td>3.281E-23</td>
</tr>
<tr>
<td>10</td>
<td>1.000E+00</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>7.500E-04</td>
<td>-3.939E-23</td>
<td>5.976E-23</td>
</tr>
<tr>
<td>11</td>
<td>1.000E+00</td>
<td>1.000E+00</td>
<td>0.000E+00</td>
<td>7.500E-04</td>
<td>-3.000E-03</td>
<td>6.265E-23</td>
</tr>
<tr>
<td>12</td>
<td>0.000E+00</td>
<td>1.000E+00</td>
<td>0.000E+00</td>
<td>2.156E-23</td>
<td>-3.000E-03</td>
<td>2.978E-23</td>
</tr>
<tr>
<td>13</td>
<td>0.000E+00</td>
<td>0.000E+00</td>
<td>1.000E+00</td>
<td>2.092E-23</td>
<td>-4.131E-23</td>
<td>3.250E-03</td>
</tr>
<tr>
<td>14</td>
<td>1.000E+00</td>
<td>0.000E+00</td>
<td>1.000E+00</td>
<td>7.500E-04</td>
<td>-4.487E-23</td>
<td>3.250E-03</td>
</tr>
<tr>
<td>15</td>
<td>1.000E+00</td>
<td>1.000E+00</td>
<td>1.000E+00</td>
<td>7.500E-04</td>
<td>-3.000E-03</td>
<td>3.250E-03</td>
</tr>
<tr>
<td>16</td>
<td>0.000E+00</td>
<td>1.000E+00</td>
<td>1.000E+00</td>
<td>2.146E-23</td>
<td>-3.000E-03</td>
<td>3.250E-03</td>
</tr>
</tbody>
</table>

apes -> end of analysis . . . . . .
Simulation of Three-Dimensional Beam Bending

A three-dimensional simulation of beam bending is investigated in this sample analysis. The specific problem considered is the bending of a cantilever beam loaded by a concentrated force applied at its free end in the manner shown in Figure 3.

![Figure 3: Physical Problem Analyzed: Cantilever Beam Subjected to Concentrated Load Applied at Free End](image)

The material is idealized as being isotropic linear elastic. The elastic modulus is equal to 10,000, and a Poisson’s ratio equal to 0.20 is assumed. Other than the applied load, the numbers appearing in Figure 3 represent dimensions. The finite element mesh used in the analysis is shown in Figure 4a.

The input data associated with this problem is given below. Note that since all nodal coordinates have been specified via the straight line (NODES LINE NUMBER) option, that the GENERATE commands need not (and for efficiency, should not) be specified.

```
analysis title "cantilever beam involving four H8P0 elements."
analysis title "First (coarse) mesh."

!anal act analyze
anal type mechanical
anal description linear
anal idealization three-dimensional
anal temporal transient
!
echo init off
echo grav off
echo trans off
echo warnings off
!
```

V. N. Kaliakin
Figure 4: Finite Element Models Used in Analyses: a) Coarse Mesh, b) Refined Mesh

dim max material isotropic elastic 1
dim max nodes 20
dim max h8p0 4
!
fin sett!
!
mat elastic isotropic number 1 desc "sample isotropic material" &
    modulus 10000.0 poissons 0.20
!
    nodes line number  1  x2 -1.5  x3  1.0
    nodes line number 13 x1 12.0  x2 -1.5  x3  1.0 incr 6
    nodes line number  2  x2 -1.5  x3 -1.0
    nodes line number 14 x1 12.0  x2 -1.5  x3 -1.0 incr 6
    nodes line number  3  x3  1.0
    nodes line number 15 x1 12.0  x3  1.0 incr 6
nodes line number 4 x3 -1.0
nodes line number 16 x1 12.0 x3 -1.0 incr 6
nodes line number 5 x2 1.5 x3 1.0
nodes line number 17 x1 12.0 x2 1.5 x3 1.0 incr 6
nodes line number 6 x2 1.5 x3 -1.0
nodes line number 18 x1 12.0 x2 1.5 x3 -1.0 incr 6
!
element irreducible type "h8p0" nodes 1 7 8 2 3 9 10 4 mat 1 &
   1_add 1 1_incr 6 2_add 1 2_incr 2
!
! note use of repeated nodal ranges in 2nd spec. conc. mec. command
!
spe conc mec nod 1:6:1 1_dis 2_dis 3_dis
spe conc mec nod 13:14:1, 17:18:1 2_forc 2_value -1.25 2_hist 0
spe conc mec nod 15:16:1 2_forc 2_value -2.50 2_hist 0
!
finish data
!
solution time final 1.0 increments 1 output 1:10:1
!
finished loading

The results shown below are obtained using the above data in conjunction with the APES computer program. For clarity, the “header” that is printed at the top of the file is omitted from this file.
cantilever beam involving four H8P0 elements. First (coarse) mesh.

======================================================================
|               D Y N A M I C   S T O R A G E   A L L O C A T I O N           |
======================================================================

Largest NODE number which can used in the mesh = 20
Max. no. of ISOTROPIC, LINEAR ELASTIC materials = 1
Max. no. of 8-node hexahedral (H8P0) elements = 4

======================================================================
=   G E N E R A L   A N A L Y S I S   I N F O R M A T I O N   =
======================================================================

--> MECHANICAL analysis shall be performed
--> Fluid flow is NOT accounted for in the analysis
--> Thermal effects are NOT accounted for in analysis

--> THREE-DIMENSIONAL solution domain assumed

--> Nodal coordinates will NOT be updated

--> solver type used: SKYLINE
--> storage type: SYMMETRIC

--> "Isoparametric" mesh generation scheme used

======================================================================
=           I N T E G R A T I O N   O P T I O N S           =
======================================================================

In approximating time derivatives, the value of "THETA" = 6.667E-01
= NONLINEAR ANALYSIS INFORMATION =

--> LINEAR analysis

= HISTORY FUNCTION INFORMATION =

<<< NONE >>>

= MATERIAL IDEALIZATIONS =

--> Material number: 1

---

type : isotropic linear elastic
info. : sample isotropic material

Modulus of Elasticity = 1.000E+04
Poisson’s ratio = 2.000E-01

Elastic bulk modulus of the solid phase = 0.000E+00
Material density of the solid phase = 0.000E+00
Combined bulk modulus for solid/fluid = 0.000E+00

= NODAL COORDINATES =

node :  1  x1 =  0.000E+00  x2 = -1.500E+00  x3 =  1.000E+00
node :  2  x1 =  0.000E+00  x2 = -1.500E+00  x3 = -1.000E+00
node :  3  x1 =  0.000E+00  x2 =  0.000E+00  x3 =  1.000E+00
node :  4  x1 =  0.000E+00  x2 =  0.000E+00  x3 = -1.000E+00
node :  5  x1 =  0.000E+00  x2 =  1.500E+00  x3 =  1.000E+00
node :  6  x1 =  0.000E+00  x2 =  1.500E+00  x3 =  1.000E+00
node :  7  x1 =  6.000E+00  x2 = -1.500E+00  x3 =  1.000E+00
node :  8  x1 =  6.000E+00  x2 = -1.500E+00  x3 =  1.000E+00
node :  9  x1 =  6.000E+00  x2 =  0.000E+00  x3 =  1.000E+00
node : 10  x1 =  6.000E+00  x2 =  0.000E+00  x3 =  1.000E+00
node : 11  x1 =  6.000E+00  x2 =  1.500E+00  x3 =  1.000E+00
node : 12  x1 =  6.000E+00  x2 =  1.500E+00  x3 =  1.000E+00
node : 13  x1 =  1.200E+01  x2 = -1.500E+00  x3 =  1.000E+00
node : 14  x1 =  1.200E+01  x2 = -1.500E+00  x3 =  1.000E+00
node : 15  x1 =  1.200E+01  x2 =  0.000E+00  x3 =  1.000E+00
node : 16  x1 =  1.200E+01  x2 =  0.000E+00  x3 =  1.000E+00
node : 17  x1 =  1.200E+01  x2 =  1.500E+00  x3 =  1.000E+00
node : 18  x1 =  1.200E+01  x2 =  1.500E+00  x3 =  1.000E+00

======================================================================
= ELEMENT INFORMATION =
======================================================================

--> number:  1  (type: H8P0 )  (kind: IRREDUCIBLE )
    nodes:
    1  7  8  2  3  9 10  4
    integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
    integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
    material no. =  1
    material type: isotropic linear elastic

--> number:  2  (type: H8P0 )  (kind: IRREDUCIBLE )
    nodes:
    7 13 14  8  9 15 16 10
    integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
    integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
    material no. =  1
    material type: isotropic linear elastic

--> number:  3  (type: H8P0 )  (kind: IRREDUCIBLE )
    nodes:
    3  9 10  4  5 11 12  6
    integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
    integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
    material no. =  1
    material type: isotropic linear elastic

--> number:  4  (type: H8P0 )  (kind: IRREDUCIBLE )
nodes:
9 15 16 10 11 17 18 12

integration rule for primary variables: 2 x 2 x 2 Gauss-Legendre
integration rule for secondary variables: 1 x 1 x 1 Gauss-Legendre
material no. = 1
material type: isotropic linear elastic

= NODE POINT SPECIFICATIONS =

<table>
<thead>
<tr>
<th>Number</th>
<th>(coordinates)</th>
<th>specification:</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 : ( x1 = 0.000E+00, x2 = -1.500E+00, x3 = 1.000E+00 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-1 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-2 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-3 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2 : ( x1 = 0.000E+00, x2 = -1.500E+00, x3 = -1.000E+00 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-1 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-2 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-3 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3 : ( x1 = 0.000E+00, x2 = 0.000E+00, x3 = 1.000E+00 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-1 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-2 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-3 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4 : ( x1 = 0.000E+00, x2 = 0.000E+00, x3 = -1.000E+00 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-1 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-2 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-3 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5 : ( x1 = 0.000E+00, x2 = 1.500E+00, x3 = 1.000E+00 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-1 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-2 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-3 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6 : ( x1 = 0.000E+00, x2 = 1.500E+00, x3 = -1.000E+00 )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-1 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
<tr>
<td>displacement-2 = 0.000E+00 ; history no. = -2</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

V. N. Kaliakin
displacement-3 = 0.000E+00 ; history no. = -2

13 : ( x1 = 1.200E+01, x2 = -1.500E+00, x3 = 1.000E+00 )
  force-1 = 0.000E+00 ; history no. = -2
  force-2 = -1.250E+00 ; history no. = 0
  force-3 = 0.000E+00 ; history no. = -2

14 : ( x1 = 1.200E+01, x2 = -1.500E+00, x3 = -1.000E+00 )
  force-1 = 0.000E+00 ; history no. = -2
  force-2 = -1.250E+00 ; history no. = 0
  force-3 = 0.000E+00 ; history no. = -2

15 : ( x1 = 1.200E+01, x2 = 0.000E+00, x3 = 1.000E+00 )
  force-1 = 0.000E+00 ; history no. = -2
  force-2 = -2.500E+00 ; history no. = 0
  force-3 = 0.000E+00 ; history no. = -2

16 : ( x1 = 1.200E+01, x2 = 0.000E+00, x3 = -1.000E+00 )
  force-1 = 0.000E+00 ; history no. = -2
  force-2 = -2.500E+00 ; history no. = 0
  force-3 = 0.000E+00 ; history no. = -2

17 : ( x1 = 1.200E+01, x2 = 1.500E+00, x3 = 1.000E+00 )
  force-1 = 0.000E+00 ; history no. = -2
  force-2 = -1.250E+00 ; history no. = 0
  force-3 = 0.000E+00 ; history no. = -2

18 : ( x1 = 1.200E+01, x2 = 1.500E+00, x3 = -1.000E+00 )
  force-1 = 0.000E+00 ; history no. = -2
  force-2 = -1.250E+00 ; history no. = 0
  force-3 = 0.000E+00 ; history no. = -2

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
end of mathematical model data
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

At time 1.000E+00 (step no. 1) NO iteration was required
= ELEMENT STRAINS & STRESSES =
======================================================================

--> element  1 ( type = H8P0 ):

@ (x1 = 3.000E+00, x2 = -7.500E-01, x3 = 0.000E+00):
eps_11 = -5.492E-04 ; eps_22 = 5.615E-05 ; eps_33 = 4.949E-05
gam_12 = -4.000E-04 ; gam_13 = 7.689E-19 ; gam_23 = -2.300E-18
sig_11 = -5.808E+00 ; sig_22 = -7.641E-01 ; sig_33 = -8.196E-01
sig_12 = -1.667E+00 ; sig_13 = 3.204E-15 ; sig_23 = -9.584E-15

--> element  2 ( type = H8P0 ):

@ (x1 = 9.000E+00, x2 = -7.500E-01, x3 = 0.000E+00):
eps_11 = -5.492E-04 ; eps_22 = 5.615E-05 ; eps_33 = 4.949E-05
gam_12 = -4.000E-04 ; gam_13 = 7.689E-19 ; gam_23 = -2.300E-18
sig_11 = -5.808E+00 ; sig_22 = -7.641E-01 ; sig_33 = -8.196E-01
sig_12 = -1.667E+00 ; sig_13 = 3.204E-15 ; sig_23 = -9.584E-15

--> element  3 ( type = H8P0 ):

@ (x1 = 3.000E+00, x2 = 7.500E-01, x3 = 0.000E+00):
eps_11 = 5.492E-04 ; eps_22 = -5.615E-05 ; eps_33 = -4.949E-05
gam_12 = -4.000E-04 ; gam_13 = 7.689E-19 ; gam_23 = -2.300E-18
sig_11 = 5.808E+00 ; sig_22 = 7.641E-01 ; sig_33 = 8.196E-01
sig_12 = -1.667E+00 ; sig_13 = 4.407E-14 ; sig_23 = -4.413E-15

--> element  4 ( type = H8P0 ):

@ (x1 = 9.000E+00, x2 = 7.500E-01, x3 = 0.000E+00):
eps_11 = 5.492E-04 ; eps_22 = -5.615E-05 ; eps_33 = -4.949E-05
gam_12 = -4.000E-04 ; gam_13 = 7.689E-19 ; gam_23 = -2.300E-18
sig_11 = 5.808E+00 ; sig_22 = 7.641E-01 ; sig_33 = 8.196E-01
sig_12 = -1.667E+00 ; sig_13 = 4.407E-14 ; sig_23 = -4.413E-15
sig_{12} = -1.667E+00 ; sig_{13} = 1.285E-14 ; sig_{23} = 6.430E-15

======================================================================
= NODAL QUANTITIES =
======================================================================

node:  1 ( x1 = 0.000E+00, x2 = -1.500E+00, x3 = 1.000E+00 )

node:  2 ( x1 = 0.000E+00, x2 = -1.500E+00, x3 = -1.000E+00 )
      u_1 = -3.123E-23, u_2 = -3.542E-24, u_3 = -3.248E-24

node:  3 ( x1 = 0.000E+00, x2 = 0.000E+00, x3 = 1.000E+00 )
      u_1 = -4.697E-37, u_2 = 2.983E-24, u_3 = 4.047E-38

node:  4 ( x1 = 0.000E+00, x2 = 0.000E+00, x3 = -1.000E+00 )
      u_1 = 5.673E-37, u_2 = 2.983E-24, u_3 = 8.188E-38

node:  5 ( x1 = 0.000E+00, x2 = 1.500E+00, x3 = 1.000E+00 )
      u_1 = 3.123E-23, u_2 = -3.542E-24, u_3 = -3.248E-24

node:  6 ( x1 = 0.000E+00, x2 = 1.500E+00, x3 = -1.000E+00 )
      u_1 = 3.123E-23, u_2 = -3.542E-24, u_3 = 3.248E-24

node:  7 ( x1 = 6.000E+00, x2 = -1.500E+00, x3 = 1.000E+00 )
      u_1 = -6.590E-03, u_2 = -1.566E-02, u_3 = 1.979E-04

node:  8 ( x1 = 6.000E+00, x2 = -1.500E+00, x3 = -1.000E+00 )
      u_1 = -6.590E-03, u_2 = -1.566E-02, u_3 = -1.979E-04

node:  9 ( x1 = 6.000E+00, x2 = 0.000E+00, x3 = 1.000E+00 )
      u_1 = -4.536E-17, u_2 = -1.550E-02, u_3 = 2.024E-16

node: 10 ( x1 = 6.000E+00, x2 = 0.000E+00, x3 = -1.000E+00 )
      u_1 = 6.648E-17, u_2 = -1.550E-02, u_3 = 2.035E-16

node: 11 ( x1 = 6.000E+00, x2 = 1.500E+00, x3 = 1.000E+00 )
      u_1 = 6.590E-03, u_2 = -1.566E-02, u_3 = -1.979E-04

node: 12 ( x1 = 6.000E+00, x2 = 1.500E+00, x3 = -1.000E+00 )
      u_1 = 6.590E-03, u_2 = -1.566E-02, u_3 = 1.979E-04

node: 13 ( x1 = 1.200E+01, x2 = -1.500E+00, x3 = 1.000E+00 )
\begin{align*}
  &\begin{array}{lll}
    u_1 &= -8.863E-03, & u_2 = -4.889E-02, & u_3 = 5.667E-06 \\
  \end{array} \\
  \text{node: } & 14 ( x_1 = 1.200E+01, \ x_2 = -1.500E+00, \ x_3 = -1.000E+00 ) \\
  &\begin{array}{lll}
    u_1 &= -8.863E-03, & u_2 = -4.889E-02, & u_3 = -5.667E-06 \\
  \end{array} \\
  \text{node: } & 15 ( x_1 = 1.200E+01, \ x_2 = 0.000E+00, \ x_3 = 1.000E+00 ) \\
  &\begin{array}{lll}
    u_1 &= -6.022E-17, & u_2 = -4.888E-02, & u_3 = 6.231E-16 \\
  \end{array} \\
  \text{node: } & 16 ( x_1 = 1.200E+01, \ x_2 = 0.000E+00, \ x_3 = -1.000E+00 ) \\
  &\begin{array}{lll}
    u_1 &= 8.995E-17, & u_2 = -4.888E-02, & u_3 = 6.199E-16 \\
  \end{array} \\
  \text{node: } & 17 ( x_1 = 1.200E+01, \ x_2 = 1.500E+00, \ x_3 = 1.000E+00 ) \\
  &\begin{array}{lll}
    u_1 &= 8.863E-03, & u_2 = -4.889E-02, & u_3 = -5.667E-06 \\
  \end{array} \\
  \text{node: } & 18 ( x_1 = 1.200E+01, \ x_2 = 1.500E+00, \ x_3 = -1.000E+00 ) \\
  &\begin{array}{lll}
    u_1 &= 8.863E-03, & u_2 = -4.889E-02, & u_3 = 5.667E-06 \\
  \end{array}
\end{align*}

To gain some insight into the results obtained, note that the Bernoulli-Euler solution for the transverse displacement of the free end \( (v_{\text{max}}) \) is \( PL^3/3EI \). Noting that the moment of inertia for bending about the \( x_3 \) axis is equal to \( I = (2)(3)^3/12 = 4.50 \), it follows that \( v_{\text{max}} \) is equal to \(-0.1280\). Now since the span to depth ratio \( (4:1) \) is not overly large, it follows that the contribution of shear deformations to \( v_{\text{max}} \) cannot be neglected. As such, the tip deflection of \(-0.04889\) (nodes 13 to 18) indicates that, owing to the rather coarse mesh of tri-linear elements, the beam is too stiff. To remedy this situation, the refined mesh shown in Figure 4b should next be analyzed.
Bibliography

