FeaTureD User’s Manual

release 0.5

Ton van den Boogaard

January 13, 2006

Contents

Chapter 1

Introduction

This manual describes the use of the demonstration program for the use of the FEATURE toolkit: FEATURED. The FEATURED analysis uses one input file, in which the model is described and the analysis that is performed with the model. First the way in which a numerical model is described in an input file is treated and then the analysis control. The input file consists of several tables, some obligatory and some optional. In general, data that is used in one table must already be defined in a previous table. E.g. before an element can refer to nodes and a material, these must already be known. A table starts with a command that has a '$\star$' as a first character and it is ended by the start of the following table. The input is ended by the keyword '$\star$`end`'.

An input line can be split in two or more physical lines by using a continuation character. In FEATURED this is a backslash '\'. After a backslash you can put comments, since the program continues at the next line after reading a '\'. Blank lines may be included at any stage and are meaningless, except that it ends an input line if the previous line ended with a continuation symbol.

For all input items uppercase or lowercase letters may be used. The number of leading, intermediate and trailing spaces or tabs is insignificant, except for the keywords with a star, where the '$\star$' must be in column 1.

1.1 Keywords and parameters

In this manual keywords and parameters in syntax descriptions are presented with a different typeface. The meaning of the typefaces are given in table 1.1.

Table 1.1: Use of typefaces in this manual

★starword	A special keyword, starting with a star. Must be typed exactly as presented, with the star (an asterisk actually) in column number 1.
⋆starword	The same as the previous, used in running text.
keyword	A keyword. Must be typed exactly is presented.
keyword	The same as the previous, used in running text.
intparm$_i$	An integer parameter. The user must select a value.
realparm$_r$	A real parameter. The user must select a value.
string$_s$	A character string. The user must select a value.
parm	Any parameter, used in running text.
[anything]	An item between square brackets is optional.
a | b | c	A choice between a, b or c.
`example`	An example.

Chapter 2

Model Description

2.1 Introduction

In this chapter the tables are described that are necessary to describe the model, including geometry, material, loads and (other) boundary conditions. The definition of the model if performed with the following tables:

★title	all lines up to the next form the title
★nodes	define the nodal coordinates
★material	define materials
★geometry	define e.g. thicknesses and cross sections
★elements	define element types, material, geometry and connectivity
★suppress	defines (Dirichlet) boundary conditions for values=0
★loadef	creates load set definitions, to be used later

2.2 Title

syntax

★title
text$_s$

text	All lines after the keyword ⋆title are considered to be part of the title of the model, up to the next keyword (starting with a ⋆). Completely blank lines are ignored. The title is used in printed output.

2.3 Nodes

syntax

★**node**
node$_i$ ***x***$_r$ ***y***$_r$ ***z***$_r$
...

---

node the node number.

x,y,z the x-, y- and z-coordinates.

The nodes are defined one at a line by the node number and the coordinates in 3 dimensions.

example

```
*nodes
  1    0.0   0.0   0.0
  2    1.0   0.0   0.0
  3    0.0   1.0   0.0
  4    1.0   1.0   0.0
  5    0.0   2.0   0.0
  6    1.0   2.0   0.0
  7    0.0   3.0   0.0
```

2.4 Material

Each material is defined by its own ⋆material line. The first line following ⋆material gives the type of material. If this line is omitted a linear elastic material is assumed.

syntax

★**material** [***matnr***$_i$]
mattype$_s$

---

matnr Reference number for this material (default 1).

mattype The name of one of the material models.

The material type can be one of the following:

linearelastic

xplinearelastic

nonlinearelastic

vonmises

thermal

thermoelastic

thermoplastic

mass

2.4.1 Linear elastic

syntax

linearelastic
young $\boldsymbol{e_r}$
poisson $\boldsymbol{nu_r}$
[**density** $\boldsymbol{rho_r}$]

syntax

xplinearelastic
young $\boldsymbol{e_r}$
poisson $\boldsymbol{nu_r}$
[**density** $\boldsymbol{rho_r}$]

linearelastic selects the linear elastic material model. This model does not calculate thickness strains in a plane stress situation.

xplinearelastic selects the expanded linear elastic material model. This model also calculates the thickness strain in a plane stress situation.

young selects Young's modulus.

poisson selects Poisson's ratio.

density selects the density of the material. Used in dynamics only.

The material models linearelastic and xplinearelastic are equivalent, except for the fact that in a uniaxial and a plane stress analysis the thickness strains are calculated from the Poisson's ratio.

Young's modulus and Poisson's ratio must be specified and for a dynamic analysis or dead weight loading the density must be specified.

example

```
*material 1
 linearelastic
  young   200000.0
  poisson 0.3
  density 7800.0
```

2.4.2 Nonlinear elastic

syntax

nonlinearelastic
young $\boldsymbol{e}_r$
[**density** $\boldsymbol{rho}_r$]

nonlinearelastic selects the nonlinear elastic material model.

young selects the material parameter E.

density selects the density of the material. Used in dynamics only.

The nonlinear elastic material model can only be used in a uniaxial element (truss elements). The implemented model has no physical meaning and is added to test the nonlinear solution algorithms. It represents an elastic stress–strain relation of the form:

$$\sigma = 2E\left(\sqrt{|\varepsilon| + 1} - 1\right)\operatorname{sgn}(\varepsilon)$$

example

```
*material 2
 nonlinearelastic
  young   200000.0
  density 7800.0
```

2.4.3 Von Mises Plasticity

syntax

vonmises
young $\boldsymbol{e}_r$
poisson $\boldsymbol{nu}_r$
nadai $\boldsymbol{c}_r$ $\boldsymbol{n}_r$ $\boldsymbol{s0}_r$
[**density** $\boldsymbol{rho}_r$]

vonmises selects the Von Mises plastic material model.

young selects Young’s modulus.

poisson selects Poisson’s ratio.

nadai selects the parameters for the Nadai hardening curve.

density selects the density of the material. Used in dynamics only.

The Von Mises material model is an elastoplastic model with a Von Mises yield function and Nadai hardening curve. It uses an Euler backward (radial return) integration algorithm.

example

```
*material 1
 vonmises
  young  200000.0
  poisson 0.3
  nadai  50 1.0 5
  density 7800.0
```

The parameters in the Nadai line represent the constant C, the exponent n and the initial yield stress σ_{y0} respectively such that:

$$\sigma_{eq} = C(\varepsilon_{eq} - \varepsilon_0)^n$$

with $\sigma_{y0} = C\varepsilon_0^n$.

2.4.4 Linear thermal

syntax

thermal
lambda ***lam***$_r$
capacity ***c***$_r$
density ***rho***$_r$

thermal selects the linear thermal material model.

lambda selects coefficient of conduction λ.

capacity selects the thermal capacity c.

density selects the density of the material.

The thermal material model is a linear thermal model with a constant conductivity, capacity and density.

example

```
*material 1
 thermal
  lambda  1.0
  capacity 0.5
  density  1.0
```

2.4.5 Thermo elastic

syntax

---

thermoelastic
young $\boldsymbol{e_r}$
poisson $\boldsymbol{nu_r}$
lambda $\boldsymbol{lam_r}$
capacity $\boldsymbol{c_r}$
density $\boldsymbol{rho_r}$
alpha $\boldsymbol{al_r}$

---

thermoelastic selects the linear thermoelastic material model.

young selects Young’s modulus.

poisson selects Poisson’s ratio.

lambda selects coefficient of conduction λ.

capacity selects the thermal capacity c.

density selects the density of the material.

alpha selects the thermal expansion coefficient.

The thermoelastic material model needs the same parameters as the linear elastic material and the thermal material model. Additionally the thermal expansion coefficient must be provided.

example

```
*material 1
 thermoelastic
  young  210000.0
  poisson 0.3
  lambda 10.0
  density 1000.0
  capacity  5.0
  alpha   1.E-5
```

2.4.6 Thermo plastic

syntax

---

thermoplastic
young $\boldsymbol{e_r}$
poisson $\boldsymbol{nu_r}$
nadai $\boldsymbol{c_r}$ $\boldsymbol{n_r}$ $\boldsymbol{s0_r}$

lambda ***lam*** $_r$
capacity ***c*** $_r$
density ***rho*** $_r$
alpha ***al*** $_r$

---

thermoplastic selects the thermoplastic Von Mises material model.

young selects Young's modulus.

poisson selects Poisson's ratio.

nadai selects the parameters for the Nadai hardening curve.

lambda selects coefficient of conduction λ.

capacity selects the thermal capacity c.

density selects the density of the material.

alpha selects the thermal expansion coefficient.

The thermoplastic material model needs the same parameters as the Von Mises material and the thermal material model. Additionally the thermal expansion coefficient must be provided. The thermal expansion is the only thermal influence on the mechanical behavior.

example

```
*material 1
 thermoplastic
  young  200000.0
  poisson 0.3
  nadai   50 1.0 5
  lambda  2.0
  density 1.0
  capacity 0.5
  alpha   1.E-5
```

2.4.7 Mass

syntax

mass *m*

---

mass the mass of an item, usually a 'point mass element'.

For point masses, the only relevant parameter is the total mass. This is specified as a (pseudo) material parameter.

example

```
*material 2
 mass 1.0
```

2.5 Geometry

syntax

★geometry [***geomnr***$_i$]
crosse ***a***$_r$
thick ***t***$_r$
iz ***iz***$_r$

---

crosse The cross sectional area. Used for truss elements.

thick The thickness of an element. Used for plane stress elements.

iz The area moment of inertia for bending along the element z-axis (in the x-y plane).

Each geometry is defined by its own ⋆geometry line. The number after ⋆geometry represents the geometry number (default geomnr=1). The ⋆geometry line is followed by lines that can define the thickness (for plane stress elements) or the cross sectional area (for truss elements) a presented in the following example:

example

```
*geometry  1
  crosse 45.0
*geometry  2
  thick 0.3
```

2.6 Elements

In FEATURED a number of elements are included. If not otherwise specified all connectivities are defined counterclockwise for 2D elements and for 3D elements counterclockwise on the base plane and then on ever increasing planes. Two alternative quadratic triangular and quadratic elements are provided where first the vertex nodes are numbered as e.g. specified by the Patran and GiD pre-/postprocessors.

The degrees of freedom in the model are generated, based on the elements that are attached to the nodes.

syntax

★elements
type$_s$ ***matnr***$_i$ [***geomnr***$_i$]
elmnr$_i$ ***node1***$_i$ ... ***noden***$_i$
...

---

type element type as given in the next sections.

matnr the material number that defines the material data for this element set.

geomnr the geometry number that defines the geometrical data for this element set.

elmnr the element number.

node1 ... noden a sequence of node numbers connected to the element.

A number of element lines may be repeated and can be followed by another element type.

example

```
*elements
 quad4pss 1  3
  1   1  2  4  3
  2   3  4  6  5
  3   5  6  8  7
  4   7  8 10  9
  5   9 10 12 11

 tri3pss 1  3
  6  11 12 13
  7  13 12 14
```

If the geometry-nr is not necessary for the particular element type, it can be omitted.

The elements are based on geometrically linear theory (small displacements and small deformations) unless stated otherwise.

2.6.1 Trusses

Trusses can only withstand forces in the direction of the truss itself i.e. there is no bending included. In FEATURED only one truss element is included.

truss

A straight truss element with two nodes. Geometrically linear behavior is assumed. The element has two degrees of freedom, one displacement in each node with the direction equal to the line from node 1 to node 2. A reference to a geometry item (geomnr) must specify the initial cross sectional area of the element. The material of the truss element can only be the linearelastic or the nonlinearelastic model.

2.6.2 Beams

Beams can withstand forces in the direction of the beam itself and also bending. In FEATURED only a 2-dimensional linear beam element is included.

beam2d

A straight Bernoulli beam element with two nodes. Geometrically linear behavior is assumed. The element has six degrees of freedom, two displacements and one rotation in each node. A reference to a geometry item (geomnr) must specify the initial cross sectional area and the moment of inertia of the element. The material of the beam element can only be the linearelastic model.

2.6.3 Linear triangular elements

Linear triangles all share the same (bilinear) interpolation functions. They have three nodes, numbered counter clockwise. Numerical integration is performed with 1 integration point in the center of the element.

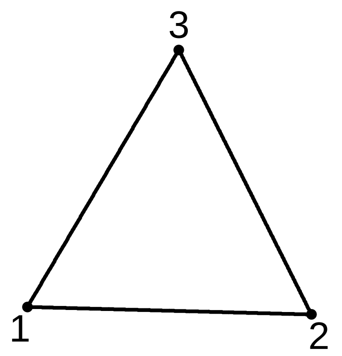


Figure 2.1: Linear triangular element

tri3psn

This element has 6 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain).

tri3pss

This element has 6 degrees of freedom, one displacement in x- and one in y-direction in every node. The stress in thickness direction is zero (plane stress). A reference to a geometry item (geomnr) must specify the initial thickness of the element.

tri3axi

This element has 6 degrees of freedom, one displacement in x- and one in y-direction in every node. The displacements are considered to be axi-symmetric. The y-axis is the axis of symmetry and the x-axis represents the radial axis (which should be positive).

tri3therm

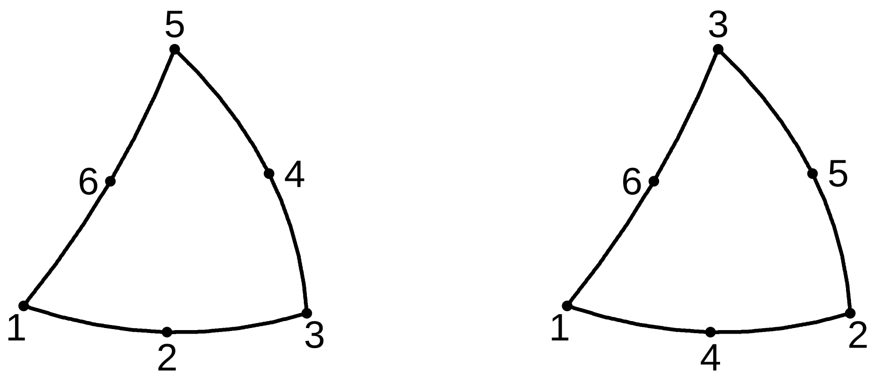
A thermal 3-node planar element. This element has 3 degrees of freedom, one temperature in every node. The flow in thickness direction is zero.

tri3psntherm

A thermo-mechanical plane strain element, combining tri3therm and tri3psn. For both interpolations a linear function is used, which is questionable from the computational point of vue. It is used to demonstrate the hierarchical derivation of a thermo-mechanical element from a mechanical and a thermal element.

2.6.4 Quadratic triangular elements

Quadratic triangles all share the same (biquadratic) interpolation functions. They have six nodes. Numerical integration is performed with 3 integration points.



(a) standard numbering (b) alternative numbering

Figure 2.2: Quadratic triangular elements

tri6psn

This element has 12 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain). The nodes are numbered according to figure 2.2a.

tri6pss

This element has 12 degrees of freedom, one displacement in x- and one in y-direction in every node. The stress in thickness direction is zero (plane stress). The nodes are numbered according to figure 2.2a. A reference to a geometry item (geomnr) must specify the initial thickness of the element.

tri6axi

This element has 12 degrees of freedom, one displacement in x- and one in y-direction in every node. The displacements are considered to be axi-symmetric. The y-axis is the axis of symmetry and the x-axis represents the radial axis (which should be positive). The nodes are numbered according to figure 2.2a.

tri6psna

This element has 12 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain). The nodes are numbered according to figure 2.2b (Patran convention).

tri6pssa

This element has 12 degrees of freedom, one displacement in x- and one in y-direction in every node. The stress in thickness direction is zero (plane stress). The nodes are numbered according to figure 2.2b (Patran convention). A reference to a geometry item (geomnr) must specify the initial thickness of the element.

tri6axia

This element has 12 degrees of freedom, one displacement in x- and one in y-direction in every node. The displacements are considered to be axi-symmetric. The y-axis is the axis of symmetry and the x-axis represents the radial axis (which should be positive). The nodes are numbered according to figure 2.2b (Patran convention).

2.6.5 Linear quadrilateral elements

Linear quadrilaterals all share the same interpolation functions (bilinear with one quadratic coupling term). They have four nodes, numbered counter clockwise. Numerical integration is performed with a 2×2 Gauss integration scheme.

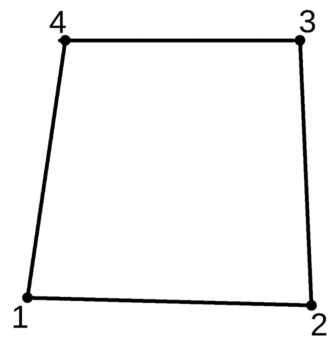


Figure 2.3: Linear quadrilateral element

quad4psn

This element has 8 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain).

quad4pss

This element has 8 degrees of freedom, one displacement in x- and one in y-direction in every node. The stress in thickness direction is zero (plane stress). A reference to a geometry item (geomnr) must specify the initial thickness of the element.

quad4axi

This element has 8 degrees of freedom, one displacement in x- and one in y-direction in every node. The displacements are considered to be axi-symmetric. The y-axis is the axis of symmetry and the x-axis represents the radial axis (which should be positive).

quad4psstherm

A thermo-mechanical plane stress element, combining quad4pss and a linear quadrilateral thermal element. For both interpolations linear(+) functions are used, which is questionable from the computational point of vue. It is used to demonstrate the hierarchical derivation of a thermo-mechanical element from a mechanical and a thermal element. A reference to a geometry item (geomnr) must specify the initial thickness of the element.

quad4psntherm

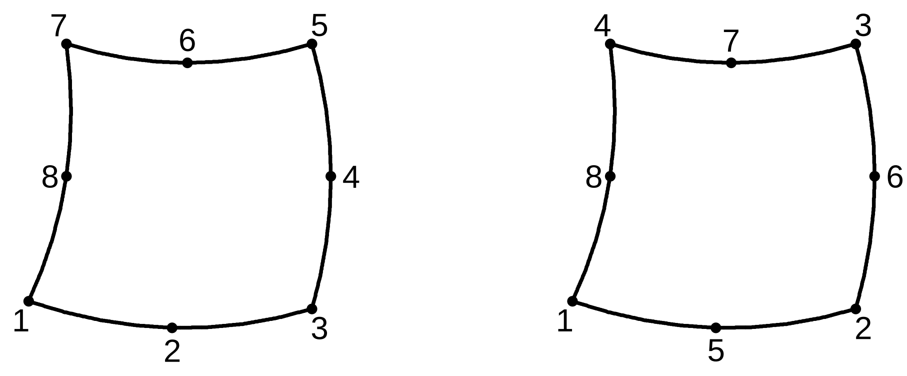
A thermo-mechanical plane strain element, combining quad4psn and a linear quadrilateral thermal element. For both interpolations linear(+) functions are used, which is questionable from the computational point of vue. It is used to demonstrate the hierarchical derivation of a thermo-mechanical element from a mechanical and a thermal element.

tlquad4psn

This element has 8 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain). The stress–strain relation follows a total Lagrange analysis (2nd Piola–Kirchhoff stress and Green Lagrange strain). This makes the element suitable for large displacement analysis, limited to small deformations.

2.6.6 Quadratic quadrilateral elements

Quadratic quadrilaterals all share the same interpolation functions (biquadratic with two cubic coupling term). They have eight nodes. Numerical integration is performed with a 2×2 Gauss integration scheme.



(a) standard numbering

(b) alternative numbering

Figure 2.4: Quadratic quadrilateral (serendipity) elements

quad8psn

This element has 16 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain). The nodes are numbered according to figure 2.4a.

quad8pss

This element has 16 degrees of freedom, one displacement in x- and one in y-direction in every node. The stress in thickness direction is zero (plane stress). The nodes are numbered according to figure 2.4a. A reference to a geometry item (geomnr) must specify the initial thickness of the element.

quad8axi

This element has 16 degrees of freedom, one displacement in x- and one in y-direction in every node. The displacements are considered to be axi-symmetric. The y-axis is the axis of symmetry and the x-axis represents the radial axis (which should be positive). The nodes are numbered according to figure 2.4a.

quad8psna

This element has 16 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain). The nodes are numbered according to figure 2.4b (Patran convention).

quad8pssa

This element has 16 degrees of freedom, one displacement in x- and one in y-direction in every node. The stress in thickness direction is zero (plane stress). The nodes are numbered according to figure 2.4b (Patran convention). A reference to a geometry item (geomnr) must specify the initial thickness of the element.

quad8axia

This element has 16 degrees of freedom, one displacement in x- and one in y-direction in every node. The displacements are considered to be axi-symmetric. The y-axis is the axis of symmetry and the x-axis represents the radial axis (which should be positive). The nodes are numbered according to figure 2.4b (Patran convention).

tlquad8psn

This element has 16 degrees of freedom, one displacement in x- and one in y-direction in every node. The strain in thickness direction is zero (plane strain). The nodes are numbered according to figure 2.4a. The stress–strain relation follow the total Lagrange analysis (2nd Piola–Kirchhoff stress and Green Lagrange strain). This makes the element suitable for large displacement analysis, limited to small deformations.

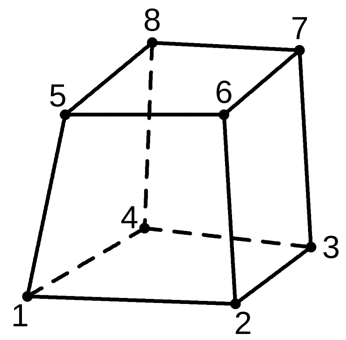


Figure 2.5: Linear hexahedral element

2.6.7 Hexagonal elements

hex8

This is an 8-node hexahedral element with 24 degrees of freedom, one displacement in x-direction, one in y-direction and one in z-direction for every node. The interpolation function is trilinear and has 3 additional quadratic (coupling terms) and one cubic coupling term. The stress–strain relation if fully 3-dimensional (6 independent stress measures). In figure 2.5 the node connectivity is presented. Numerical integration is performed with a $2 \times 2 \times 2$ Gauss integration scheme.

2.6.8 Mass elements

Mass elements have no stiffness, but contribute only mass to the model. This can be usefull in e.g. a dynamic analysis. Since the vector and matrix contributions for 2D and 3D situations differ, separate elements are provided. Mass elements connect to only one node.

mass2d

A 2-dimensional version of the mass element. The connected (pseudo)material must provide the mass of the point.

mass3d

A 3-dimensional version of the mass element. The connected (pseudo)material must provide the mass of the point.

example

```
*elements
 mass3d 2
  1    8
  2   18
 hex8   1
 ...
```

This example generates two 3-dimensional mass elements, connected to nodes 8 and 18, with a mass as described in material number 2.

2.7 Boundary conditions

The boundary conditions contain so called Dirichlet and Neumann boundary conditions with values of zero or different. In the implementation of FEATURED, a zero boundary force or thermal flux are the natural boundary conditions, therefore these do not have to be specified explicitly.

Degrees of freedom that will allways have a value of zero (fixed supports) are eliminated from the system and are therefore treated different from degrees of freedom that have a non-zero prescribed value. The latter is seen as a real load and is described in the load definition part.

2.7.1 Suppressed degrees of freedom

syntax

★suppress
dof$_s$ ***node***$_i$ ...

---

dof Specifies the degree of freedom to be suppressed. This can be UX, UY, UZ, RX, RY, RZ or T for the displacements in x-, y- and z-direction, rotation along the x-, y- and z-direction and the temperature respectively.

node A number of nodes to which this condition applies.

For example

example

```
*suppress
 UY 1 2
 UX 1
```

suppresses the displacements in y-direction of node 1 and 2 and the displacement in x-direction for node 1.

2.7.2 Load definitions

syntax

★loadef
set ***nr***$_i$
prescribed
...
loads
...

pressure
...

---

set Defines a reference number *nr* for this load set. The load set can be used later by refering to this number. The sizes of the loads in the definition of the load set define the unit load increments for this set. When the load set is applied these sizes will be multiplied by a given factor.

prescribed Defines prescribed values for the degrees of freedom (displacements and temperatures).

loads Defines nodal loads (nodal forces and nodal thermal fluxes).

pressure Defines pressures on element sides (2D) or faces (3D).

With the load definition block, load sets are defined that can be incremented later. A number of sets can be defined, although only one set can be incremented yet in the current version.

Prescribed degrees of freedom

syntax

---

prescribed
dof$_s$ ***size***$_r$ ***node***$_i$ ...
...

---

dof Specifies the degree of freedom to be prescribed. This can be UX, UY, UZ, RX, RY, RZ or T for the displacements in x-, y- and z-direction, rotation along the x-, y- and z-direction and the temperature respectively.

size The size of the displacement, rotation or temperature increment. This determines the values for this load set, that can be scaled when the load set is used.

node A number of nodes to which this load applies.

For example

example

```
*loadef
 set 1
  prescribed
   T 100.0   21 22
```

increments the temperature of nodes 21 and 22 by 100.0 if load set 1 is incremented by 1.0.

Nodal loads

syntax

---

loads
dof $_s$ ***size*** $_r$ ***node*** $_i$ ...
...

---

dof Specifies the degree of freedom to be loaded. This can be FX, FY, FZ or Q for the forces in x-, y- and z-direction z-direction, moments along x-, y- and z-direction and the thermal flux respectively.

size The size of the load, moment or flux increment. This determines the values for this load set, that can be scaled when the load set is used.

node A number of nodes to which this load applies.

For example

example

```
*loadef
 set 1
  loads
   FY  1000.0 21
   FY -1000.0 22
```

increments the nodal force in y-direction by 1000 for node 21 and by -1000 for node 22 if load set 1 is incremented by 1.0 .

Element loads

syntax

pressure
element*i* ***face****i* ***size****r*
...

element	Specifies the element to which the pressure is applied.
face	Specifies the edge (2D) or face (3D) to which the pressure is applied (Patran convention) see figure 2.6 and tables 2.1 to 2.3.
size	Defines the (unit) size of the pressure increment. This determines the values for this load set, that can be scaled when the load set is used.

example

```
*loadef
set 1
pressure
  1  2  150.0
  2  2  150.0
  3  2  150.0
  4  2  150.0
  5  2  150.0
```

Each time that load set nr. 1 is incremented by 1.0, this load increments the pressure on face 2 of elements 1 to 5 with 150.

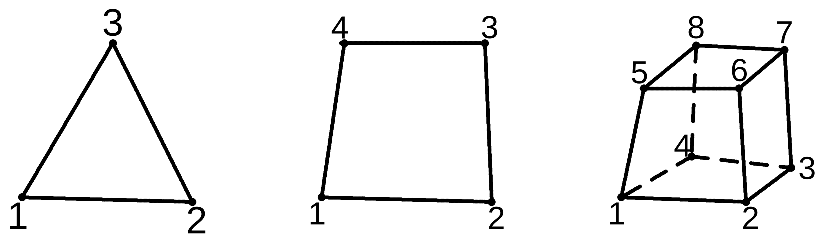


Figure 2.6: Basic element shapes for edge/face definitions

Table 2.1: Edge definition triangles

Edge ID	Node 1	Node 2
1	1	2
2	2	3
3	3	1

Table 2.2: Edge definition quadrilaterals

Edge ID	Node 1	Node 2
1	1	2
2	2	3
3	3	4
4	4	1

Table 2.3: Face definition hexahedrons

Face ID	Node 1	Node 2	Node 3	Node 4
1	1	2	6	5
2	3	4	8	7
3	1	2	3	4
4	2	3	7	6
5	5	6	7	8
6	1	5	8	4

Chapter 3

Analysis Control

Three types of analyses are included in FEATURED. Linear static analysis, eigenfrequency analysis and nonlinear static and dynamic transient analysis. A linear analysis can also be performed by using only one increment and using linear material and geometrically linear elements. Note that pressure loading on elements allways results in nonlinear behavior (displacement dependent loads).

3.1 Linear static analysis

syntax

★linear

A linear static analysis is performed by using the keyword ⋆linear. This is also the default analysis type if no analysis keyword is used.

3.2 Eigenfrequency analysis

The eigenfrequency analysis is performed with the keyword ⋆eigen. This keyword requires a model with stiffness and mass.

syntax

★eigen
frequencies n_i
shift s_r

---

frequencies	n is the number of eigenfrequencies and eigenmodes that will be determined (default: 5).
shift	s is the eigenfrequency shift in Hz (default: 0). A positive shift can be used to determine the eigenfrequencies and modes of an unsupported structure. A negative shift can be used to determine the eigenfrequencies that are close to minus the shift value.

The undamped eigenvalue problem $(\mathbf{K} - \omega_i^2 \mathbf{M})\mathbf{v}_i = \mathbf{0}$ is solved by a subspace iteration method. The subspace dimension is equal to the minimum of $2n$ and $n+8$ and the starting vectors consist of the diagonal terms of $\mathbf{M}$ and subsequently unit vectors with a component +1 on the row with the smallest ratio of stiffness and mass: K_{ii}/M_{ii}. The eigenfrequencies are reported in Hz: $f_i = \omega_i/2\pi$.

If a shift is used an effective stiffness matrix $\mathbf{K}_{\text{eff}} = \mathbf{K} + \mu\mathbf{M}$ is used, with $\mu = 2\pi s^2$ if $s \geq 0$ and $\mu = -2\pi s^2$ if $s < 0$.

3.3 Nonlinear analysis

syntax

★nonlinear [**static** | **dynamic**]

static — inertia forces are ignored, a (quasi-)static analysis is performed. (default)

dynamic — inertia forces are included, a transient dynamic analysis is performed. In this case also structural damping can be included in the analysis.

The nonlinear analysis is initialized with the keyword ⋆nonlinear. If this keyword is omitted a default analysis with one increment will be performed, using load set nr. 1 with a unit load as the applied load. This can be used e.g. for a linear analysis.

A truly nonlinear analysis will use at least an incremental approach (section 3.3.1) and possibly an iterative algorithm (section 3.3.2). For a dynamic analysis some additional data is needed (section 3.3.3).

3.3.1 Incremental analysis

syntax

★nonlinear
increment load ***set***$_i$ **size** ***s***$_r$ **steps** ***nr***$_i$
timestep ***size***$_r$
save all | **last** | **step** ***n***$_i$ ...
restart ***filename***$_s$

increment — *set* is the incremented load set, *s* is the sum of all increments and *nr* is the number of increments to be taken.

timestep — *size* is the time increment for every analysis increment. This line is not needed if time plays no role in the analysis.

save — This keyword is used to save the data of an analysis to make it possible to restart from the saved step. It is followed by one of the keywords step, all or last to indicate which steps are to be saved. In case of step this word is followed by one or more step numbers that have to be saved. If all is selected then all steps are saved and if last is selected the last step is saved.

The data is stored on a file with the basename of the inputfile, extended with the number of the saved step and the extension 'bin'.

restart The keyword restart is followed by a filename that is used to initialize the model data. A subsequent analysis restarts from the saved data onwards.

An example of the input for an incremental analysis is:

example

```
*nonlinear
 restart restart004.bin
 increment load 1 size 0.5 steps 200
 timestep 0.005
 save step 9
```

3.3.2 Iterative analysis

The following keywords are only applicable if an incremental-iterative algorithm is used. This excludes the explicit time integration scheme.

syntax

full | **modified**
unbalance $\boldsymbol{eps}_r$
maxiter $\boldsymbol{mi}_i$
maxresize $\boldsymbol{mr}_i$

full This keyword selects the full Newton–Raphson iterative procedure. The stiffness matrix is set up at every iteration. This is the default behavior for the static analysis and dynamic Newmark scheme.

modified This keyword selects the modified Newton–Raphson iterative procedure. The stiffness matrix is set up only at the start of every increment. This usually leads to more iterations before the same accuracy is reached, compared to a full Newton–Raphson scheme, but every iteration will cost less CPU-time.

unbalance This keyword selects the ratio of the norm of the unbalance force vector to the norm of the load increment vector at which the iterations stop (default $1 \cdot 10^{-4}$).

maxiter This keyword selects the maximum number of iterations *mi* (default 10).

maxresize This keyword is followed by the maximum number of re-sizes that may be performed. If convergence is not achieved the increment size will be halved (and again and again) for a maximum of *mr* times. By default the number of resizes is zero.

example

```
*nonlinear
 modified
 increment load 1 size 1.0 steps 20
 unbalance 1.e-3
 maxiter   20
 maxresize 2
```

3.3.3 Dynamic analysis

syntax

★nonlinear dynamic
newmark [***alpha***$_r$ ***delta***$_r$]
explicit
rayleigh ***alpha***$_r$ ***beta***$_r$

newmark Selects the Newmark time integration scheme. The parameters *alpha* and *delta* represent the Newmark parameters α and δ. By default these values are 0.25 and 0.5 respectively. The Newmark scheme is the default for a dynamic analysis.

explicit This selects the explicit central difference scheme. For this scheme a critical time step exists which must not be exceeded.

rayleigh If a dynamic analysis is performed, then the structural damping can be modelled by so-called Rayleigh damping. This requires two parameters α and β, to be specified as *alpha* and *beta*. The damping matrix is determined as $\mathbf{C} = \alpha\mathbf{M} + \beta\mathbf{K}$. By default no damping is included.

If a Newmark integration scheme is chosen, the input parameters for both the incremental analysis and the iterative analysis can be selected. If an explicit integration scheme is chosen, no iterations take place and only the parameters for the incremental analysis can be chosen.

example

```
*nonlinear dynamic
 newmark
 full
 timestep 0.01
 increment load 2 size 1.0 steps 100
 unbalance 1.e-3
 maxiter   5
 rayleigh 200.0 0.0
```

3.4 Post processing

The selection of postprocessing data must be selected after the selection of the nonlinear control process.

syntax

★post
displacement
equivalent
temperature
stress
all
last
step $\boldsymbol{n}_i$ ...

---

displacement selects the total displacements for post processing

equivalent selects the equivalent plastic strains for post processing

temperature selects the temperature for post processing

stress selects the (Cauchy) stresses for post processing

all selects all steps for post processing

last selects the last step for post processing

step selects specified steps n for post processing

Two different types of items are selected. First the step number for which post processing data is to be saved and second the type of data that is to be saved.

The postprocessing data is written in one file according to the post processing format, required by the program GiD (from CIMNE, Barcelona, `http://gid.cimne.upc.es`).

The data is stored in the nodal points. If element data is selected, then the integration point values of one element are averaged and for each node the average contribution of the connected elements is used as the nodal data.

example

```
*post
 displacement
 equivalent plastic strain
 stress
 step 1 3
 last
```

Index