OPTANT: User Manual

Delft University of Technology
Aerospace Structures & Computational Mechanics

Contents

Chapter 1

The Input File

The input file is a simple text file which contains the full description of a finite element model. It includes the definition of Solution Type, Materials, Sections, Properties, Nodes, Elements, Nodal loads, Pressure loads, Temperature loads, Single point constraints, Multiple point constraints, and Load cases. The definition of each of these entities is encapsulated by a start and end keyword, e.g. `$Materials0` and `$Materials1` for the material definition. The order in which each entity is defined can be chosen arbitrarily, however, the order implicated by this manual is advised.

The general layout of an input file is shown in Fig. 1.1. The exact description of all materials, sections, etc., is presented in the following chapters in which each of the entities is considered in more detail.

```
Any text may be added outside the encapsulations of each entity definition,
   except for the start and end keywords of each entity.

$Solution0
   ... (Definition of the solution type)
$Solution1

$Materials0
   ... (Definition of the materials)
$Materials1

$Sections0
   ... (Definition of the sections)
$Sections1
.
.  (All other entities)
.
```

Figure 1.1: General layout of the input file.

Chapter 2

Solution Type Definition

The solution type definition contains the specification of the type of solution which is requested, i.e. linear static or linear buckling. The first line of the solution type definition has the following format:

	1	2	3	4	5	6
Line 1	solCard	solType				

Parameter	Description
solCard	Solution type card: SOL.
matType	Solution type (10→ linear static; 11→ linear buckling).

The contents of the following line(s) depends on the specified solution type.

Linear Static (SOL=10)

A linear static solution does not require any additional input.

Linear Buckling (SOL=11)

A linear buckling solution requires the specification of the number of modes to be computed. In addition, a prestress Load case may be specified. In that case the prestress load case is solved using a linear static solve and the resulting stresses are used as prestresses for the linear buckling solve.

	1	2	3	4	5	6
Line 2	nmCard	n_{modes}				
* Line 3	psCard	loadCaseID				

* Only specified if prestresses are applied.

Parameter	Description
nmCard	Number of modes card: NMODES.
psCard	Prestress load case card: PRESTRESS.
n_{modes}	Number of buckling modes (integer≥1).
loadCaseID	Identification number of Load case to be used as prestress.

An example of a linear buckling solution type with 20 buckling modes and including prestresses is given in Fig. 2.1.

```
$Solution0
SOL          11
NMODES       20
PRESTRESS    2
$Solution1
```

Figure 2.1: Solution type definition example.

Chapter 3

Material Definitions

The first line of each material definition has the following format:

	1	2	3	4	5	6
Line 1	matID	matType	numLines	matName		

Parameter	Description
matID	Material identification number (integer$\geq$1).
matType	Material type (0$\rightarrow$ isotropic; 1$\rightarrow$ 2D orthotropic; 2$\rightarrow$ 2D anisotropic).
numLines	Number of lines that follow after Line 1.
matName	Material name (string $\leq$ 20 characters).

The contents of the following line(s) depends on the specified material type.

Isotropic Material

An isotropic material (matType = 0) is defined by six material properties:

	1	2	3	4	5	6
Line 2	ρ	E	ν	T_{ref}	α	λ

Parameter	Description
ρ	Material density.
E	Young's modulus.
ν	Poisson's ratio.
T_{ref}	Reference temperature.
α	Coefficient of thermal expansion.
λ	Thermal conductivity.

Note that for the material definitions the number of properties per line may be chosen arbitrarily. This means that an isotropic material may also be specified as:

	1	2	3	4	5	6
Line 2	ρ	E				
Line 3	ν	T_{ref}	α	λ		

The numLines parameter in Line 1 has to be specified accordingly. An example of two isotropic material definitions is given in Fig. 3.1.

```
$Materials0
4   0   1   Aluminium
2.7800000e-6  7.30000000e4  3.3000000e-1  2.73150000e2  2.3100000e-5  2.3700000e-1
3   0   2   Steel
7.7800000e-6  2.10000000e5  3.0000000e-1  2.73150000e2
1.0800000e-5  1.6300000e-2
$Materials1
```

Figure 3.1: Material definition example.

2D Orthotropic Material

2D Anisotropic Material

3D Anisotropic Material

Chapter 4

Section Definitions

This chapter considers the section definitions

Chapter 5

Property Definitions

The first line of all property definitions follows the same format:

	1	2	3	4	5	6
Line 1	propID	propInput	numLines	propCard		

Parameter	Description
propID	Property identification number (integer$\geq$1).
propInput	Property input option (0$\rightarrow$ from a Section; 1,2$\rightarrow$ depends on Property card).
numLines	Number of lines that follow after Line 1.
propCard	Property card (PBEAM, PSHELL).

Each property requires a different set of input parameters, which are treated in the following sections.

5.1 PBEAM

The PBEAM property allows the user to implement standard elements, such as the Euler-Bernoulli beam, as well as very general elements with a user specified coupling between shear load and normal/bending loads. The property parameters can be specified by one of available propInput options, which are presented in order.

propInput=0

The beam properties may be derived directly from a Section definition. In this case only the section ID is required in the input file:

	1	2	3	4	5	6
Line 2	sectID					

Parameter	Description
sectID	Section identification number.

propInput=1

This option allows the user to specify conventional beam properties, such as the Euler-Bernoulli beam and Timoshenko beam. The required input consists of the material, geometric properties and shear correction factor:

	1	2	3	4	5	6
Line 2	matID	A	$1/k_{shear}$			
Line 3	J_x	I_{yy}	I_{zz}	I_{yz}	J_{xz}	J_{xy}

Parameter	Description
matID	Material identification number.
A	Cross-sectional area.
$1/k_{shear}$	Inverse of the shear correction factor. If $1/k_{shear} = 0$ the Euler-Bernoulli beam is obtained, if $1/k_{shear} = 6/5$ the Timohenko beam is obtained.
J_{xx}	Torsional rigidity of the beam (see axes convention in Fig. 5.1).
J_{xz}, J_{xy}	Torsion-bending coupling rigidity.
I_{yy}, I_{zz}, I_{xy}	Components of inertia tensor (second moment of area), using the axes convention in Fig. 5.1.

NOTE: The input for I_{yz}, J_{xz}, and J_{xy} is currently not used.

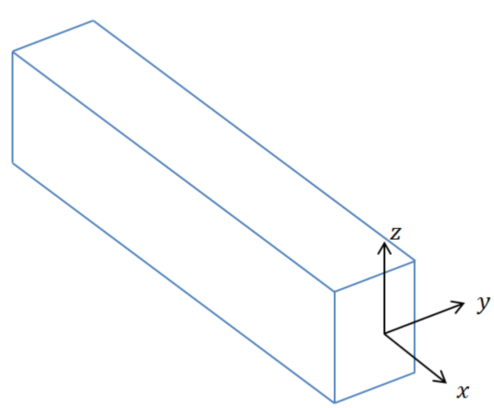

Figure 5.1: Convention of x, y, z-axes system.

An example of a defining a Timoshenko beam property is given in Fig. 5.2.

```
$Properties0
1   1   2   PBEAM
1   7.85398163e1    1.11111111e0
9.81747704e2  4.90873852e2  4.90873852e2  0.00000000e0  0.00000000e0  0.00000000e0
$Properties1
```

Figure 5.2: PBEAM definition of a circular Timoshenko beam with radius $r = 5$ and shear correction factor $k = 5/6$.

propInput=2

The most general beam properties can be specified with input option. The user is enabled to specify the full stiffness tensors for normal/bending, shear, and normal/bending-shear coupling. The required input parameters are given and explained below:

	1	2	3	4	5	6
Line 2	$\tilde{C}_{11}$	$\tilde{C}_{12}$	$\tilde{C}_{13}$	$\tilde{C}_{14}$	$\tilde{C}_{22}$	$\tilde{C}_{23}$
Line 3	$\tilde{C}_{24}$	$\tilde{C}_{33}$	$\tilde{C}_{34}$	$\tilde{C}_{44}$		
Line 4	$\tilde{G}_{11}$	$\tilde{G}_{12}$	$\tilde{G}_{13}$	$\tilde{G}_{14}$	$\tilde{G}_{21}$	$\tilde{G}_{22}$
Line 5	$\tilde{G}_{23}$	$\tilde{G}_{24}$	$\tilde{S}_{11}$	$\tilde{S}_{12}$	$\tilde{S}_{22}$	
* Line 6	$\tilde{c}_1$	$\tilde{c}_2$	$\tilde{c}_3$	$\tilde{c}_4$	$\tilde{g}_1$	$\tilde{g}_2$

* May be omitted if no Temperature loads loads are applied.

Parameter	Description
$\tilde{C}_{11}$ - $\tilde{C}_{44}$	Components of the symmetric 4×4 tensor $\tilde{\boldsymbol{C}}$ as defined in (5.4).
$\tilde{G}_{11}$ - $\tilde{G}_{24}$	Components of the 2×4 tensor $\tilde{\boldsymbol{G}}$ as defined in (5.5).
$\tilde{S}_{11}$ - $\tilde{S}_{22}$	Components of the symmetric 2×2 tensor $\tilde{\boldsymbol{S}}$ as defined in (5.6).
$\tilde{c}_1$ - $\tilde{c}_4$	Components of the 4×1 vector $\tilde{\boldsymbol{c}}$ as defined in (5.7).
$\tilde{g}_1$ - $\tilde{g}_2$	Components of the 2×1 vector $\tilde{\boldsymbol{g}}$ as defined in (5.8).

To understand the meaning of $\tilde{\boldsymbol{C}}$, $\tilde{\boldsymbol{G}}$, and $\tilde{\boldsymbol{S}}$, we first consider the strain energy Φ in the beam, defined as:

$$\Phi = \frac{1}{2}\boldsymbol{\varepsilon}^T\boldsymbol{C}\boldsymbol{\varepsilon} + \frac{1}{2}\left(\boldsymbol{\varepsilon}^T\boldsymbol{G}\boldsymbol{\gamma} + \boldsymbol{\gamma}^T\boldsymbol{G}\boldsymbol{\varepsilon}\right) + \frac{1}{2}\boldsymbol{\gamma}^T\boldsymbol{C_s}\boldsymbol{\gamma} - \boldsymbol{\varepsilon}^T\boldsymbol{c}\Delta T - \boldsymbol{\gamma}^T\boldsymbol{g}\Delta T \tag{5.1}$$

with ΔT the applied change in temperature and:

$$
\boldsymbol{\varepsilon} = \begin{pmatrix} u' \\ \theta_x' \\ \theta_y' \\ \theta_z' \end{pmatrix} \qquad \boldsymbol{\gamma} = \begin{pmatrix} v' - \theta_z \\ w' + \theta_y \end{pmatrix} \tag{5.2}
$$

$\boldsymbol{C}$, $\boldsymbol{C_s}$, $\boldsymbol{G}$ represent the stiffness tensors for normal/bending, shear, and normal/bending-shear coupling, respectively. Direct application of the strain energy will lead to shear locking of the beam element. A modified Hellinger-Reissner principle is applied, which yields the strain energy shown in (5.3).

$$
\Phi^{**} = \frac{1}{2}\boldsymbol{\varepsilon}^T\tilde{\boldsymbol{C}}\boldsymbol{\varepsilon} + \boldsymbol{V}^T\tilde{\boldsymbol{G}}\boldsymbol{\varepsilon} - \frac{1}{2}\boldsymbol{V}^T\tilde{\boldsymbol{S}}\boldsymbol{V} - \boldsymbol{\varepsilon}^T\tilde{\boldsymbol{c}}\Delta T - \boldsymbol{V}^T\tilde{\boldsymbol{g}}\Delta T \tag{5.3}
$$

with $\boldsymbol{V} = (V_y, V_z)^T$ being the shear force, and:

$$
\tilde{\boldsymbol{C}} = \boldsymbol{C} - \boldsymbol{G}\boldsymbol{S}\boldsymbol{G}^T \tag{5.4}
$$
$$
\tilde{\boldsymbol{G}} = \boldsymbol{S}\boldsymbol{G}^T \tag{5.5}
$$
$$
\tilde{\boldsymbol{S}} = \boldsymbol{C_s}^{-1} \tag{5.6}
$$
$$
\tilde{\boldsymbol{c}} = \boldsymbol{c} - \boldsymbol{G}\boldsymbol{S}\boldsymbol{g}^T \tag{5.7}
$$
$$
\tilde{\boldsymbol{g}} = \boldsymbol{S}\boldsymbol{g} \tag{5.8}
$$

So, $\tilde{\boldsymbol{C}}$ represents the normal/bending stiffness tensor, $\tilde{\boldsymbol{G}}$ is the normal/bending-shear coupling tensor, and $\tilde{\boldsymbol{S}}$ denotes the shear stiffness tensor. $\tilde{\boldsymbol{c}}$ and $\tilde{\boldsymbol{g}}$ represent the 'stiffness' of the beam with respect to applied temperature loading for normal/bending and shear respectively. As an example, a Timoshenko beam with symmetric cross section is represented by:

$$
\tilde{\boldsymbol{C}} = \begin{bmatrix} E_{xx}A & 0 & 0 & 0 \\ 0 & G_{yz}J_{xx} & 0 & 0 \\ 0 & 0 & E_{xx}I_{yy} & 0 \\ 0 & 0 & 0 & E_{xx}I_{zz} \end{bmatrix} \qquad \tilde{\boldsymbol{G}} = \boldsymbol{0} \qquad \tilde{\boldsymbol{S}} = \begin{bmatrix} \frac{1}{k_{shear}G_{xy}A} & 0 \\ 0 & \frac{1}{k_{shear}G_{xz}A} \end{bmatrix}
$$
$$
\tilde{\boldsymbol{c}} = \begin{pmatrix} \alpha_x E_{xx}A \\ 0 \\ 0 \\ 0 \end{pmatrix} \qquad \tilde{\boldsymbol{g}} = \boldsymbol{0} \tag{5.9}
$$

An example of a general beam property is given in Fig. 5.3. Note that this property definition has not been based on a real beam, but merely serves as an example.

```
$Properties0
1  2   4   PBEAM
1.00000000e2  5.00000000e1  2.50000000e1  1.00000000e1  2.00000000e2  1.50000000e2
1.25000000e2  3.00000000e2  2.50000000e2  4.00000000e2
1.00000000e1  9.00000000e0  8.00000000e0  7.00000000e0  6.00000000e0  5.00000000e0
4.00000000e0  3.00000000e0  5.0000000e-1  2.0000000e-1  8.0000000e-1
$Properties1
```

Figure 5.3: PBEAM definition of a general beam property without $\tilde{\boldsymbol{c}}$ and $\tilde{\boldsymbol{g}}$ specified.

5.2 PSHELL

The PHSELL property is used to specify shell properties using a predefined Section or using the $\boldsymbol{A}$,$\boldsymbol{B}$,$\boldsymbol{D}$ matrices and $\boldsymbol{a}$,$\boldsymbol{b}$ vectors of the shell. The $\boldsymbol{A}$,$\boldsymbol{B}$,$\boldsymbol{D}$ matrices are derived from classical laminate theory, whereas $\boldsymbol{a}$,$\boldsymbol{b}$ relate normal forces and bending moments to applied temperature respectively. The $\boldsymbol{a}$,$\boldsymbol{b}$ vectors are analogous to $\boldsymbol{A}$,$\boldsymbol{B}$ in the sense that $\boldsymbol{b} = \boldsymbol{0}$ for symmetric laminates and the third entry of $\boldsymbol{a}$ equals zero for balanced laminates.

The PSHELL property card is especially aimed at composite materials, but also it also allows the user to specify any other isotropic or anisotropic shell property. The property parameters can be specified by one of available propInput options, which are presented in order.

propInput=0

The shell properties may be derived directly from a Section definition. This section may e.g. define the lay-up of a composite laminate. Only the section ID is required in the input file for this type of input:

	1	2	3	4	5	6
Line 2	sectID					

Parameter	Description
sectID	Section identification number.

propInput=1

This option allows the user to specify the $\boldsymbol{A}$,$\boldsymbol{B}$,$\boldsymbol{D}$ matrices and $\boldsymbol{a}$,$\boldsymbol{b}$ vectors of the shell directly. These entities may characterize a composite laminate, anisotropic shell, isotropic shell, etc. For example, an isotropic shell property would be characterized by $\boldsymbol{A}$ equal to the plane stress stiffness of the material, $\boldsymbol{B} = \boldsymbol{0}$, and $\boldsymbol{D} = \frac{h^3}{12}\boldsymbol{A}$ with h being the thickness of the shell.
The required input format is:

	1	2	3	4	5	6
Line 2	A_{11}	A_{12}	A_{16}	A_{22}	A_{26}	A_{66}
Line 3	B_{11}	B_{12}	B_{16}	B_{22}	B_{26}	B_{66}
Line 4	D_{11}	D_{12}	D_{16}	D_{22}	D_{26}	D_{66}
* Line 5	a_1	a_2	a_6	b_1	b_2	b_6

* May be omitted if no Temperature loads loads are applied.

Parameter	Description
A_{11}-A_{66}	Components of the $\boldsymbol{A}$ matrix of the shell.
B_{11}-B_{66}	Components of the $\boldsymbol{B}$ matrix of the shell.
D_{11}-D_{66}	Components of the $\boldsymbol{D}$ matrix of the shell.
a_1-a_6	Components of the $\boldsymbol{a}$ vector of the shell.
b_1-b_6	Components of the $\boldsymbol{b}$ vector of the shell.

An example of the definition of a balanced and symmetric composite laminate is given in Fig. 5.4.

```
$Properties0
1   1   3   PSHELL
8.39620000e4  2.65680000e4  0.00000000e0  8.39620000e4  0.00000000e0  2.86970000e4
0.00000000e0  0.00000000e0  0.00000000e0  0.00000000e0  0.00000000e0  0.00000000e0
1.31580000e4  8.23500000e3  0.00000000e0  1.31580000e4  0.00000000e0  8.64700000e3
2.54220000e0  2.54220000e0  0.00000000e0  0.00000000e0  0.00000000e0  0.00000000e0
$Properties1
```

Figure 5.4: PSHELL definition of a composite laminate with ply properties $E_x = E_y = 68900\text{MPa}$, $\nu_{xy} = \nu_{yx} = 0.05$, $G_{xy} = 4850\text{MPa}$, $\alpha_x = \alpha_y = 2.3 \cdot 10^{-5}\text{K}^{-1}$, $t_{ply} = 0.1905\text{mm}$. The lay-up of the laminate is $\theta = [\pm 45, 0, 90]_s$.

Chapter 6

Node Definitions

Each node is defined by its identification number and its location. The total number of nodes should be explicitly mentioned in the input file. The format is as follows:

	1	2	3	4	5	6
Line 1	numNodes n					
Line 2	nodeID	X	Y	Z		
$\vdots$	$\vdots$	$\vdots$	$\vdots$	$\vdots$		
Line $n+1$	nodeID	X	Y	Z		

Parameter	Description
numNodes	Total number of nodes.
nodeID	Node identification number (integer$\geq$1).
X, Y, Z	Nodal location in global coordinate system.

An example of the definition of a rectangular grid of 3×3 nodes is given in Fig. 6.1.

```
$Nodes0
9
1      0.0        0.0        0.0
2      50.0       0.0        0.0
3      100.0      0.0        0.0
4      0.0        10.0       0.0
5      50.0       10.0       0.0
6      100.0      10.0       0.0
7      0.0        20.0       0.0
8      50.0       20.0       0.0
9      100.0      20.0       0.0
$Nodes1
```

Figure 6.1: Nodes definition for a rectangular grid of 3×3 nodes.

Chapter 7

Element Definitions

The definition of elements is slightly different from commercial packages as MSC Nastran or Abaqus. The reason is that in our case multiple Properties can be defined per element, and the total number of elements per element type has to be stated explicitly in the input file. The general layout of the element definition is given in Fig. 7.1.

```
$Elements0
$ElementType
CBEAM   2

   ... (Definition of 2 CBEAM elements)

$ElementType
CTRIA   5

   ... (Definition of 5 CTRIA elements)

$Elements1
```

Figure 7.1: General layout of element definitions in the input file.

It is shown in Fig. 7.1 that the definition of a set of elements with a specific element type needs to be preceded by the `$ElementType` keyword. The line following this keyword must have the following format:

	1	2	3	4	5	6
Line 1	elemCard	numElem				

Parameter	Description
elemCard	Element card (CBEAM, CTRIA).
numElem	The total number of elements defined in this ElementType section.

Each element type requires a different specification in the input file. These element types are considered in the following sections of this chapter.

7.1 CBEAM

The CBEAM element is a two-noded 1D beam element and its properties are defined using the PBEAM property card. A CBEAM element is defined by means of two lines in the input file. The first line specifies the geometric properties and the second the beam properties of the element:

	1	2	3	4	5	6	7
Line 1	elemCard	elemID	$n1$	$n2$	z_X	z_Y	z_Z
Line 2	numProp	$p1$	$(p2)$	$(p3)$	$(p4)$	$(p5)$	$(p6)$

Parameter	Description
elemCard	Element card: CBEAM.
elemID	Element identification number (integer$\geq$1).
$n1$, $n2$	Identification numbers of first and second Nodes respectively.
z_X, z_Y, z_Z	Orientation of beam's z-axis (see Fig. 5.1) in the global coordinates (X, Y, Z). NOTE: this orientation has to be different from the orientation of the beam element.
numProp	Number of properties for this element.
$p1, p2, ..$	Identification numbers of the PBEAM properties for this element. The number of specified properties should be equal to the value of numProp.

Note that the CBEAM element matrices are computed based on principal of minimum potential energy, using the strain energy function given in (5.3). An example definition of five CBEAM elements is shown in Fig. 7.2. The beam elements have been assigned different properties in this example.

```
$Elements0
$ElementType
CBEAM  5
CBEAM  10    1     2     0.0        0.0        1.0
2      1     1
CBEAM  11    2     3     0.0        0.0        1.0
2      1     2
CBEAM  12    3     4     0.0        0.0        1.0
3      1     2     1
CBEAM  13    4     5     0.0        0.0        1.0
3      2     2     1
CBEAM  14    5     6     0.0        0.0        1.0
2      2     2
$Elements1
```

Figure 7.2: CBEAM element definition for five elements with different properties.

7.2 CTRIA

The CTRIA element is a three-noded 2D element and its properties are defined using e.g. the PSHELL property card. Two lines of input data are used to define a CTRIA element. These lines contain the geometric properties and shell properties of the element:

	1	2	3	4	5	6	7	8
Line 1	elemCard	elemID	$n1$	$n2$	$n3$	$\theta_{0,X}$	$\theta_{0,Y}$	$\theta_{0,Z}$
Line 2	numProp	$p1$	$(p2)$	$(p3)$	$(p4)$	$(p5)$	$(p6)$	$(p7)$

Parameter	Description
elemCard	Element card: CTRIA.
elemID	Element identification number (integer$\geq$1).
$n1, n2, n3$	Identification numbers of first, second, and third Nodes respectively.
$\theta_{0,X}, \theta_{0,Y}, \theta_{0,Z}$	Orientation of the principal material direction (e.g. the 0^o ply in a composite laminate) in the global coordinates (X, Y, Z). This orientation is projected onto the element plane, therefore, it cannot be perpendicular to this element plane.
numProp	Number of properties for this element.
$p1, p2, ..$	Identification numbers of the PSHELL properties for this element. The number of specified properties should be equal to the value of numProp.

Note that the element matrices of the CTRIA element are based on the BCIZ0 element as defined in Felippa [1]. The definition of eight CTRIA elements with different element properties is shown in Fig. 7.3 as an example.

```
$Elements0
$ElementType
CTRIA  8
CTRIA  10    1     2     4     1.0          1.0          0.0
2     1     2
CTRIA  11    2     4     5     1.0          1.0          0.0
2     1     2
CTRIA  12    2     3     5     1.0          1.0          0.0
2     1     2
CTRIA  13    3     5     6     1.0          1.0          0.0
3     1     2     2
CTRIA  14    4     5     7     1.0          1.0          0.0
3     1     2     2
CTRIA  15    5     7     8     1.0          1.0          0.0
3     1     2     2
CTRIA  16    5     6     8     1.0          1.0          0.0
3     1     2     2
CTRIA  17    6     8     9     1.0          1.0          0.0
3     1     2     2
$Elements1
```

Figure 7.3: CTRIA element definition for eight elements with different properties.

Chapter 8

LOAD Definitions

The LOAD card, representing a nodal load, is discussed in this chapter. Each nodal load is attributed to a load set. This set of LOADs can be applied, or 'activated', as a whole using the Load case definition. A single LOAD is defined in one line of the input file:

	1	2	3	4	5	6	7	8	9
Line 1	loadCard	loadSetID	nodeID	f_X	f_Y	f_Z	m_X	m_Y	m_Z

Parameter	Description
loadCard	Load card: LOAD.
loadSetID	Load set identification number (integer≥1). If this loadSetID is activated in a Load case, then all LOADs with this loadSetID are applied for that load case.
nodeID	Node identification number of the node at which this LOAD is applied.
f_X, f_Y, f_Z	Applied nodal force in global coordinate system (X, Y, Z).
m_X, m_Y, m_Z	Applied nodal moment in global coordinate system (X, Y, Z).

An example of the definition of four nodal loads divided over two load sets is shown in Fig. 8.1.

```
$LOAD0
LOAD   2     9     10.0       10.0       10.0       0.0        0.0        0.0
LOAD   3     3     -10.0      0.0        0.0        0.0        0.0        0.0
LOAD   3     6     -10.0      0.0        0.0        0.0        0.0        0.0
LOAD   3     9     -20.0      0.0        0.0        0.0        0.0        5.0
$LOAD1
```

Figure 8.1: LOAD definition of four nodal loads divided over two load sets.

Chapter 9

PLOAD Definitions

The PLOAD card is used to represent pressure loads on an element. Each pressure load is attributed to a pressure load set. This set of LOADs can be applied, or 'activated', as a whole using the Load case definition. A single PLOAD is defined in one or more lines of the input file:

	1	2	3	4	5	6	7	8	9	10	11
Line 1	pload-Card	pload-SetID	p	eID_1	eID_2	eID_3	eID_4	eID_5	eID_6	eID_7	eID_8
Line 2				eID_9	eID_{10}	...	...	...	...	...	...
⋮			...	...	...	...	...	...	...	...	...

Parameter	Description
ploadCard	Pressure load card: PLOAD.
ploadSetID	Pressure load set identification number (integer≥1). If this ploadSetID is activated in a Load case, then all PLOADs with this ploadSetID are applied for that load case.
p	Applied pressure.
eID_i	Element identification numbers of the elements at which this PLOAD is applied.

An example of the definition of one pressure load is presented in Fig. 9.1.

```
$PLOAD0
PLOAD   2       4.5     17      18      19      20      21      22      23      24
                        25      26      27      28      29      30      31      32
                        33      34      35      36      37      38
$PLOAD1
```

Figure 9.1: PLOAD definition of one pressure load on multiple elements.

Chapter 10

TEMP Definitions

The TEMP card can be applied to specify a temperature load by means of a ΔT. Each temperature load is attributed to a temperature load set. This set of TEMPs can be applied, or 'activated', as a whole using the Load case definition. A single TEMP is defined in one line of the input file:

	1	2	3	4	5	6	7	8	9
Line 1	tempCard	tempSetID	eID	ΔT_1	(ΔT_2)	(ΔT_3)	(ΔT_4)	(ΔT_5)	(ΔT_6)

Parameter	Description
tempCard	Temperature load card: TEMP.
tempSetID	Temperature load set identification number (integer$\geq$1). If this tempSetID is activated in a Load case, then all TEMPs with this tempSetID are applied for that load case.
eID	Element identification number of the element at which this TEMP is applied.
ΔT_i	Applied temperatures. NOTE: at least one ΔT value is required. If multiple values are specified, these are interpolated over the element.

An example of the definition of five temperature loads in the same load set is shown in Fig. 10.1.

```
$TEMP0
TEMP    2      1       0.        1.4726
TEMP    2      2      1.4726    2.94505
TEMP    2      3      2.94505   4.41723
TEMP    2      4      4.41723   5.88897
TEMP    2      5      5.88897   7.36015
$TEMP1
```

Figure 10.1: TEMP definition of five temperature loads in the same load set.

Chapter 11

SPC Definitions

Single point constraints (SPCs) are used to prescribe a fixed value to one or more degrees of freedom (DOF). These constraints are used to apply clamp or joint constraints, but also to apply a prescribed non-zero displacement. Each SPC is attributed to an SPC set, which can be applied, or 'activated', as a whole using the Load case definition. An SPC is defined in one line of the input file:

	1	2	3	4	5	6
Line 1	spcCard	spcSetID	nodeID	DOF	value	

Parameter	Description
spcCard	SPC card: SPC.
spcSetID	SPC set identification number (integer≥1). If this spcSetID is activated in a Load case, then all SPCs with this spcSetID are applied for that load case.
nodeID	Node identification number of the node at which this SPC is applied.
DOF	Nodal degrees of freedom (DOF) to which this SPC applies. The prescribed DOF should be specified as a string of (combinations of) the values 1-6, which correspond to $u_X, u_Y, u_Z, \theta_X, \theta_Y, \theta_Z$ respectively.
value	The prescribed value for the constrained DOF.

An example of the definition of three SPCs divided over two sets is shown in Fig. 11.1. The SPCs in set 2 describe a clamp constraint at node 1 and prescribed displacement equal to 5.0 in all directions at node 9. SPC set 3 only represents a clamp constraint at node 1.

```
$SPC0
SPC    2      1      123456 0.0
SPC    2      9      123    5.0
SPC    3      1      123456 0.0
$SPC1
```

Figure 11.1: SPC definition of three SPCs divided over two SPC sets.

Chapter 12

MPC Definitions

Multiple point constraints (MPCs) are used to relate multiple DOF using a linear equation. One MPC consists of a single slave DOF and one or more master DOF. The slave DOF is given as a function of the master DOF, as follows:

$$u_S + C_0 = \sum_i w_{M,i} u_{M,i} \tag{12.1}$$

in which, u_S is the slave DOF, C_0 is a constant, and $w_{M,i}$ are the weights for the master DOF $u_{M,i}$. The MPC definition requires the specification of all parameter in (12.1). Each MPC is attributed to an MPC set, which can be applied, or 'activated', as a whole using the Load case definition. The format for defining an MPC is given below:

	1	2	3	4	5	6	7	8
Line 1	mpcCard	mpcSetID	nID$_S$	DOF$_S$		C_0		
Line 2			nID$_{M,1}$	DOF$_{M,1}$	$w_{M,1}$	nID$_{M,2}$	DOF$_{M,2}$	$w_{M,2}$
Line 3			nID$_{M,3}$	DOF$_{M,3}$	$w_{M,3}$	$\cdots$	$\cdots$	$\cdots$
$\vdots$			$\cdots$	$\cdots$	$\cdots$	$\cdots$	$\cdots$	$\cdots$

Parameter	Description
mpcCard	MPC card: MPC.
mpcSetID	MPC set identification number (integer$\geq$1). If this mpcSetID is activated in a Load case, then all MPCs with this mpcSetID are applied for that load case.
nID_S	Node identification number of the slave node. Together with DOF_S, this uniquely determines the slave DOF u_S in (12.1).
DOF_S	Slave DOF within the node (integer$\in [1, 6]$). Together with nID_S, this uniquely determines the slave DOF u_S in (12.1).
C_0	Coefficient C_0 in (12.1).
$\mathrm{nID}_{M,i}$	Node identification number of the i^{th} master node. Together with $\mathrm{DOF}_{M,i}$, this uniquely determines the i^{th} master DOF $u_{M,i}$ in (12.1).
$\mathrm{DOF}_{M,i}$	Master DOF within the node (integer$\in [1, 6]$). Together with $\mathrm{nID}_{M,i}$, this uniquely determines the i^{th} master DOF $u_{M,i}$ in (12.1).
$w_{M,i}$	Master DOF within the node (integer$\in [1, 6]$). Together with $\mathrm{nID}_{M,i}$, this uniquely determines the i^{th} master DOF $u_{M,i}$ in (12.1).

An example of the definition of two MPCs in one set is shown in Fig. 12.1. The first MPC relates the X-displacement at node 6 to the X-displacement at two other nodes. The second MPC relates the rotation around the Y-axis at node 6 to the same rotation at three other nodes.

```
$MPC0
MPC     4       6       1                       0.0
                3       1       1.0             5       1       1.0
MPC     4       6       5                       1.0
                3       5       1.5             4       5       1.5
                5       5       1.5
$MPC1
```

Figure 12.1: MPC definition of two MPCs.

NOTE: It is not allowed to relate a DOF at one node to another DOF at the same node. This is caused by the FEI package from Trilinos, which does not support this.

Chapter 13

Load Case Definitions

A load case is the combination of an SPC set, MPC set, Load set, and/or Temperature set, together with the meshed model. So, a load case defines a full finite element model, which will be solved for the unknown degrees of freedom. If multiple load case have been defined, they will be solved in successive order. The format for specifying a load case in the input file is shown below:

	1	2	3	4	5	6
Line 1	loadCaseID	spcSetID	mpcSetID	loadSetID	tempSetID	

Parameter	Description
loadCaseID	Load case identification number (integer$\geq$1).
spcSetID	SPC set identification number (no SPC set $\rightarrow$ 0).
mpcSetID	MPC set identification number (no MPC set $\rightarrow$ 0).
loadSetID	Load set identification number (no Load set $\rightarrow$ 0).
tempSetID	Temperature set identification number (no Temperature set $\rightarrow$ 0).

An example of the definition of two load cases is shown in Fig. 13.1. The first load case only contains SPCs and LOADs, whereas the second load case also consists of MPCs.

```
$LoadCases0
1       1       0       1       0
2       1       1       2       0
$LoadCases1
```

Figure 13.1: Load case definition of two load cases.

Bibliography

[1] Felippa, C. A. Computational mechanics for the twenty-first century. Civil-Comp press, Edinburgh, UK, UK, 2000, ch. Recent Advances in Finite Element Templates, pp. 71–98.