

## Sumari

<b>SUMARI</b>	<b>1</b>
<b>ANNEX A: DISSENY DE SISTEMES D'ARRIOSTRAMENT PER ESTRUCTURES PORTICADES</b>	<b>3</b>
A.1. Generalitats .....	3
A.2. Arriostrament vertical.....	4
A.2.1. General .....	4
A.2.2. Pòrtic d'arriostrament .....	5
A.3. Arriostrament de coberta.....	5
<b>ANNEX B: CÀRREGUES</b>	<b>7</b>
<b>ANNEX C: COMBINACIONS DE CÀRREGUES</b>	<b>13</b>
<b>ANNEX D: ELEMENT BEAM 188</b>	<b>19</b>
D.1. BEAM188 Element Description .....	19
D.2. BEAM188 Input Data.....	20
D.2.1. Generalized Beam Cross Sections .....	24
D.2.2. Standard Library Sections.....	24
D.3. BEAM188 Output Data.....	34
D.4. BEAM188 Assumptions and Restrictions.....	39
<b>ANNEX E: COORDENADES PÒRTIC</b>	<b>43</b>
<b>ANNEX F: MODES VINCLAMENT 2<sup>a</sup> ETAPA MÈTODE GENERAL</b>	<b>47</b>
<b>ANNEX G: CÀLCUL D'UNIONS</b>	<b>55</b>
G.1. Càlcul base pilar articulada .....	55
G.2. Càlcul encastament biga – columna .....	56
G.3. Càlcul encastament biga - biga .....	58
<b>ANNEX H: ESTUDI D'IMPACTE AMBIENTAL</b>	<b>63</b>
<b>ANNEX I: PRESSUPOST</b>	<b>65</b>
<b>ANNEX J: PLÀNOL PÒRTIC</b>	<b>67</b>





## ANNEX A: Disseny de sistemes d'arriostrament per estructures porticades

Aquest annex ofereix orientacions sobre el disseny de sistemes d'arriostrament transversal i fora del pla per estructures porticades.

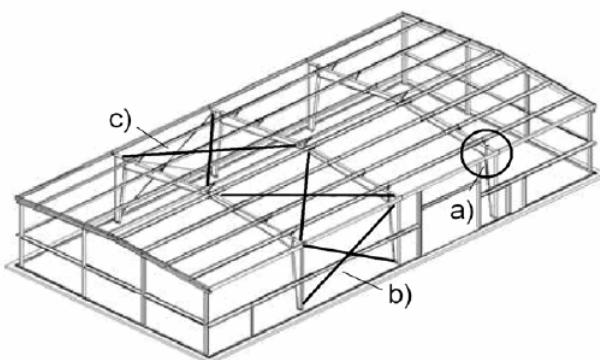
### A.1. Generalitats

L'arriostrament és necessari per resistir càrregues laterals, principalment càrregues de vent i efectes desestabilitzadors de les imperfeccions definides a §5.3 de EN 1993-1-1 [2]. Aquest arriortament ha d'estar correctament posicionat, i tenir una resistència i rigidesa adequades per justificar les suposicions realitzades durant l'anàlisi i les comprovacions dels elements.

Es essencial proveir arriostrament que sigui lo suficientment fort i rígid en tots els punts que es suposa estaran restringits en els càlculs de disseny. Això és especialment important quan l'ala interior del pòrtic està en compressió.

EN 1993-1-1 §5.3 [2] permet que les imperfeccions siguin descrites tant com imperfeccions geomètriques com mitjançant esforços horizontals equivalents.

Els esforços horizontals equivalents que produeixen els esforços a l'arriostrament no augmenten la càrrega total de l'estructura, doncs formen un cas de càrrega auto equilibrant.



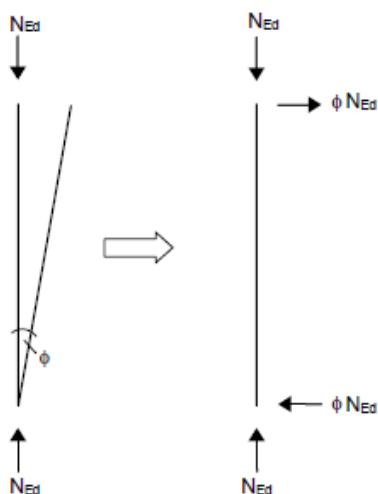
- a) Unió resistent a moment
- b) Arriostrament vertical transversal
- c) Arriostrament horitzontal



## A.2. Arriostrament vertical

### A.2.1. General

És fonamental que els pilars de l'estructura estiguin estabilitzats. En el pla d'un pòrtic, aquesta estabilitat es aportada per l'accio del pòrtic i la unió pilar/biga resistent a moment. S'ha de subministrar fixació perpendicular al pla del pòrtic mitjançant elements addicionals per a mantenir la verticalitat dels pilars, i resistir les càrregues tals com les de vent, les quals són perpendiculars al pòrtic. Es pressuposa que els pilars es construeixen lleugerament fora de la vertical; el mètode més simple per tenir en compte aquest efecte és per mitjà de les forces horizontals equivalents indicades a la següent figura. Aquestes forces es poden produir en qualsevol direcció, però es considera que actuen en una sola direcció cada vegada.



L'ariostrament està dissenyat per resistir càrregues de vent i forces horizontals equivalents. Les forces equivalents es calculen a partir de les imperfeccions per l'anàlisi global de pòrtics indicades a l'apartat §5.3.2 (3) de EN 1993-1-1 [2], i constitueixen aproximadament el 0,5% de les forces verticals que provoquen la compressió axial.

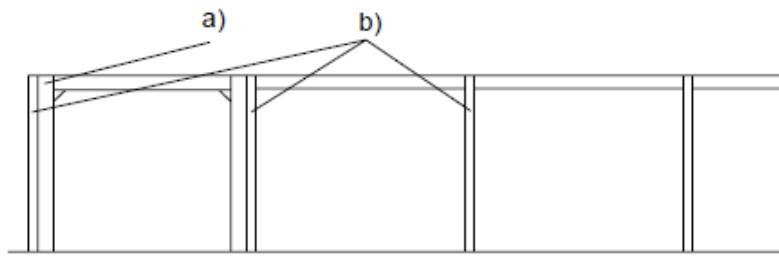
Si els pilars suporten un esforç de tracció net, com en el cas de càrrega ascendent deguda al vent, aquesta càrrega no desestabilitza l'estructura, pel que pot ser menyspreada al calcular les forces equivalents.



### A.2.2. Pòrtic d'arriostrament

El terme “pòrtic d’arriostrament” s’utilitza normalment per descriure un sistema d’arriostrament format per pòrtics enllloc de per creus, tots ells emprats per oferir la subjecció normal a les estructures principals. S’utilitza freqüentment per oferir estabilitat lateral a la part superior dels pilars interns, ja que l’ús d’arriostraments en creu de Sant Andreu tindria com a resultat una limitació inacceptables de la llibertat d’ús. També s’utilitza en murs exteriors , on l’arriostrament en creu podria obstruir finestres, portes i demés.

Els pòrtics d’arriostrament estan dissenyats per resistir les forces horitzontals equivalents totals de tots els pilars, l’estabilitat dels quals, depèn d’aquests pòrtics, juntament amb la càrrega de vent pertinent.



- a) Pòrtics d’arriostrament
- b) Pòrtics principals

Una solució més eficient és disposar alguns pilars de forma que l’eix fort estigui en la direcció longitudinal, de manera que les forces puguin ser suportades per la resistència a la flexió.

### A.3. Arriostrament de coberta

S’ha de col·locar arriostrament en els plans de coberta per donar resistència i rigidesa als punts de subjecció considerats a les verificacions de vinclament dels elements. A més, l’arriostrament ha de resistir qualsevol esforç aplicat perpendicular al pòrtic.

L’arriostrament de coberta, o una acció de diafragma equivalent dins les xapes d’acer de la coberta, és necessari per tal de suportar les forces horitzontals resultants de:

- Forces del vent en l’extrem del pinyó.

- Forces d'estabilitat procedents de pilars que no estan arriostrats pel seu propi sistema d'arriostrament de pla vertical.
- Forces d'estabilitat local de les ales de les bigues i els reforços.

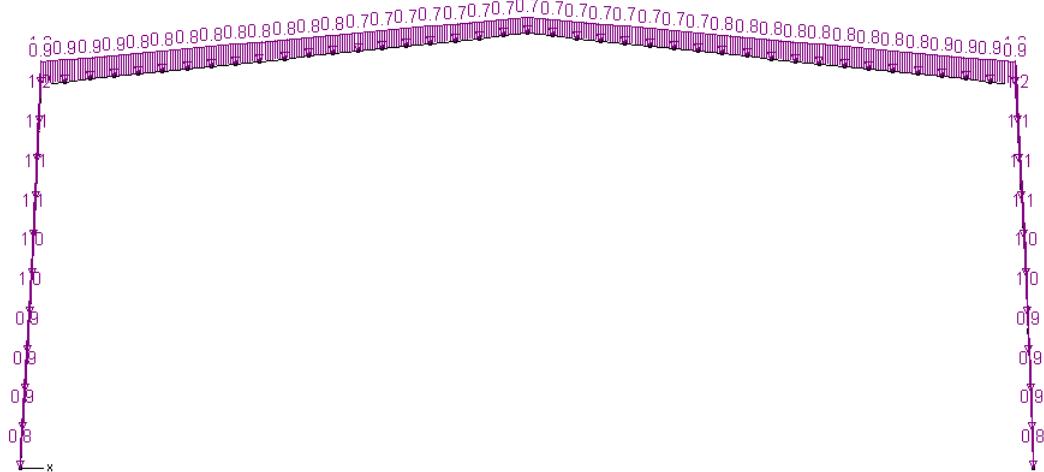
La xapa adequadament fixada ajudarà a resistir càrregues d'arriostrament en pla. No obstant, cal tenir en compte que l'ús de les xapes per a aquest fi, pot estar prohibit en alguns països, doncs la seva eficàcia pot ser difícil de demostrar.



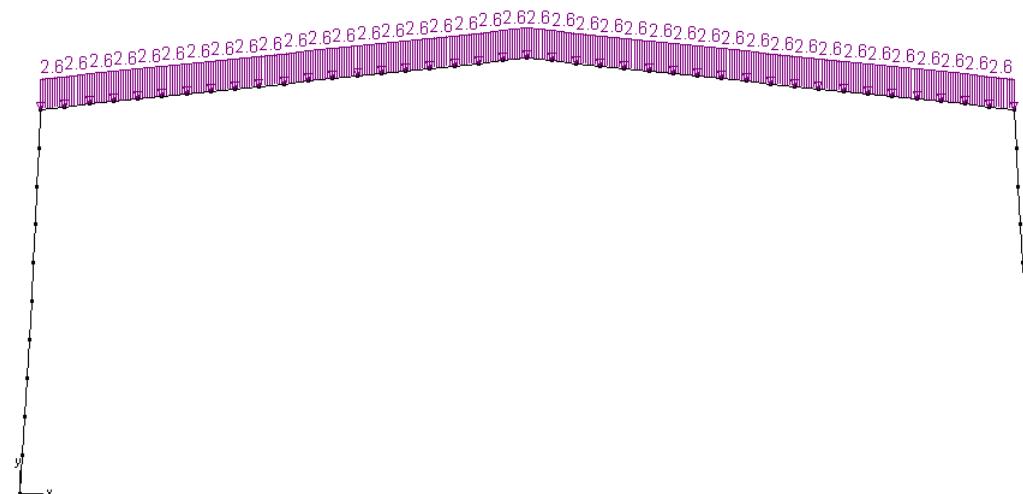
## ANNEX B: Càrregues

A continuació es mostren les accions que actuen sobre l'estructura objecte d'estudi, e intervenen alhora de generar les combinacions de càrregues corresponents.

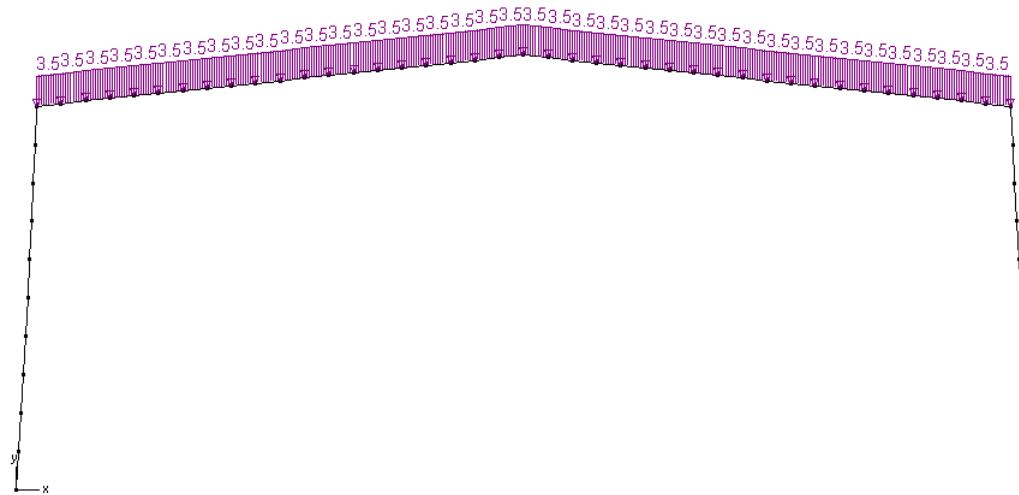
- Pes propi



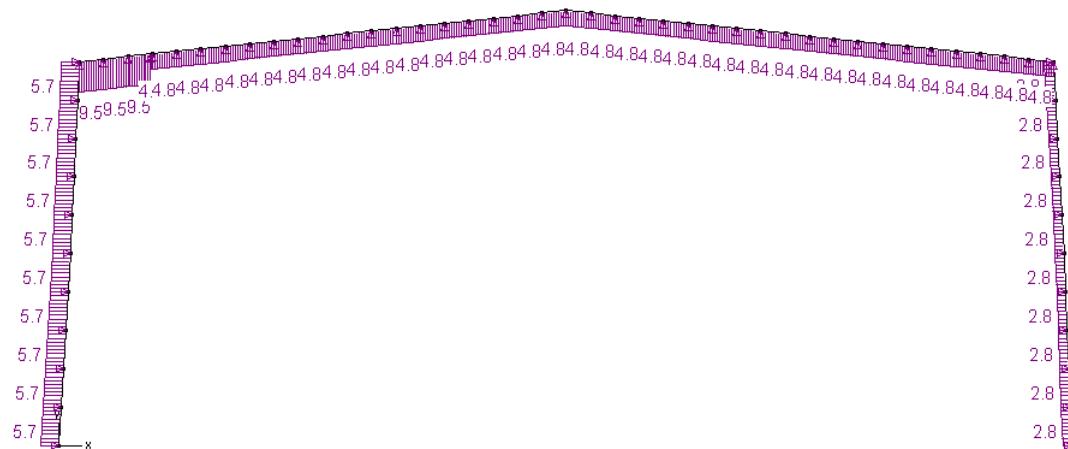
- Càrrega permanent (coberta + corretges)



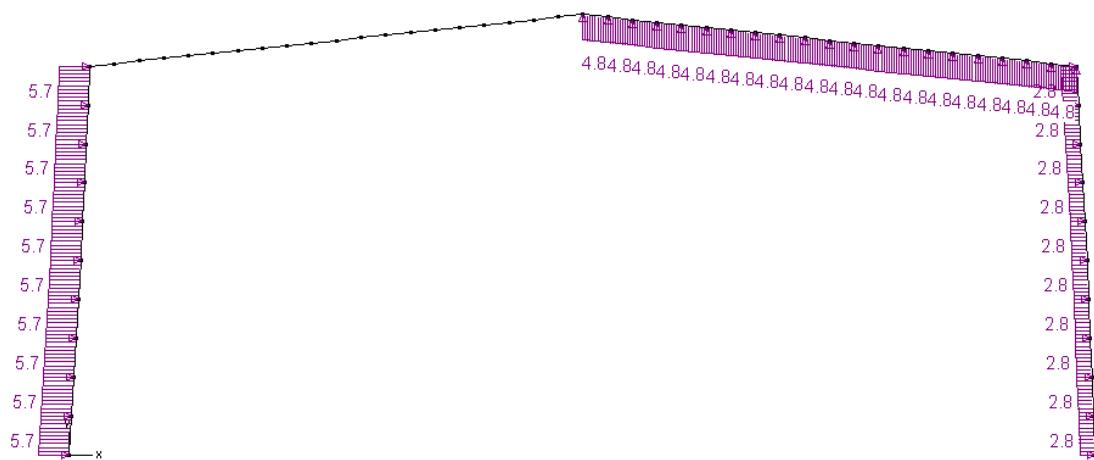
- Càrrega cobertes no transitables



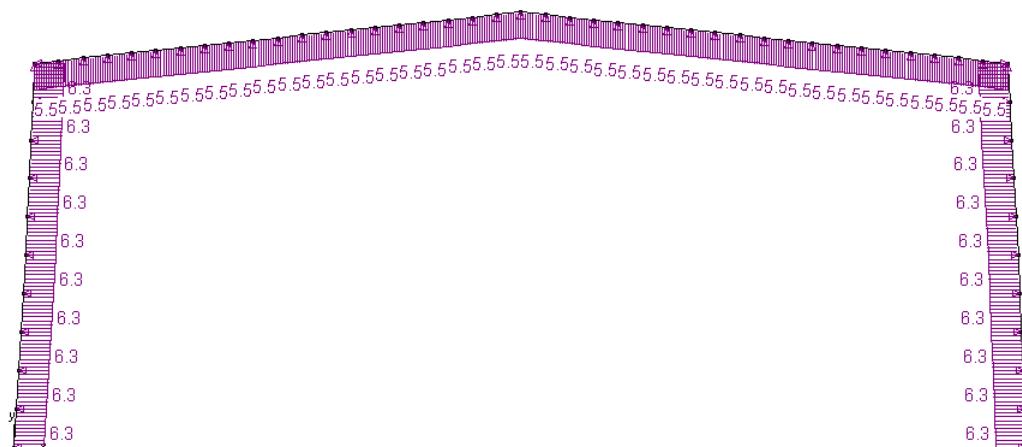
- Vent 1



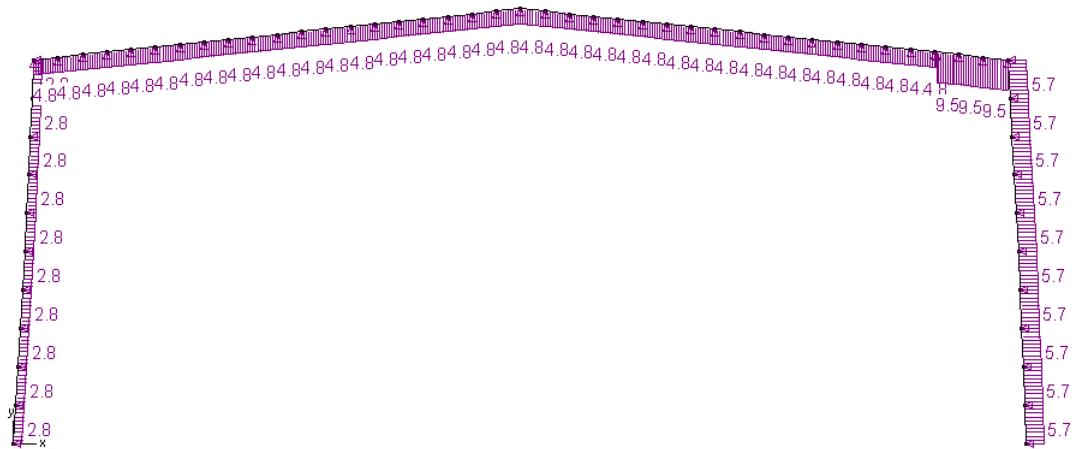
- Vent2



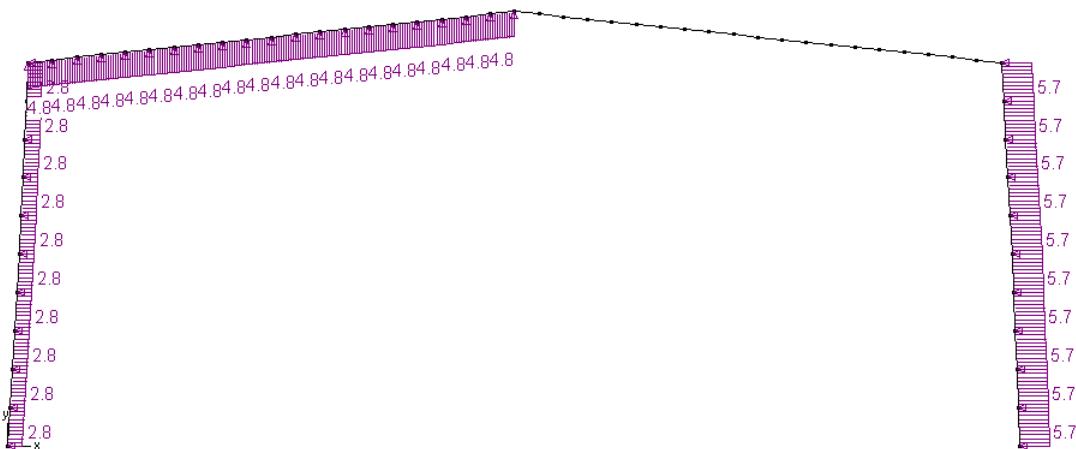
- Vent3



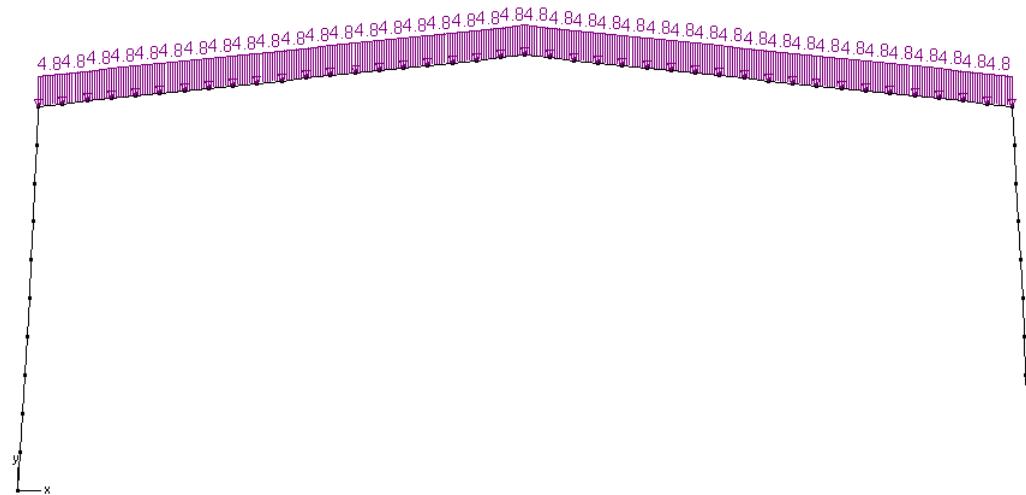
- Vent 12



- Vent 21



- Neu





## ANNEX C: Combinacions de càrregues

Nom combinació	Tipus	Pes propi	Càrrega permanent	Càrrega cobertes no transitables	Vent1	Vent2	Vent3	Neu	Vent12	Vent21
Pes propi	Grup de càrrega	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
Càrrega permanent	Grup de càrrega	0,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
Càrrega cobertes no transitables	Grup de càrrega	0,00	0,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00
Vent1	Grup de càrrega	0,00	0,00	0,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00
Vent2	Grup de càrrega	0,00	0,00	0,00	0,00	1,00 x 1,00	0,00	0,00	0,00	0,00
Vent3	Grup de càrrega	0,00	0,00	0,00	0,00	0,00	1,00 x 1,00	0,00	0,00	0,00
Neu	Grup de càrrega	0,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,00	0,00	0,00
Vent12	Grup de càrrega	0,00	0,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,00	0,00
Vent21	Grup de càrrega	0,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,00
ELU CF 1	ELU CF	1,00 x 1,35	1,00 x 1,35	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 2	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	1,00 x 1,50	0,00	0,00	0,50 x 1,50	0,00	0,00
ELU CF 3	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	1,00 x 1,50	0,00	0,50 x 1,50	0,00	0,00
ELU CF 4	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	1,00 x 1,50	0,50 x 1,50	0,00	0,00
ELU CF 5	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,60 x 1,50	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 6	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	0,50 x 1,50	1,00 x 1,50	0,00
ELU CF 7	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	0,50 x 1,50	0,00	1,00 x 1,50
ELU CF 8	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 9	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,60 x 1,50	0,00	1,00 x 1,50	0,00	0,00
ELU CF 10	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,60 x 1,50	1,00 x 1,50	0,00	0,00
ELU CF 11	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	1,00 x 1,50	0,60 x 1,50	0,00

Nom combinació	Tipus	Pes propi	Càrrega permanent	Càrrega cobertes no transitables	Vent1	Vent2	Vent3	Neu	Vent12	Vent21
ELU CF 12	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,60 x 1,50
ELU CF 13	ELU CF	1,00 x 1,00	1,00 x 1,35	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 14	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	1,00 x 1,50	0,00	0,00	0,50 x 1,50	0,00	0,00
ELU CF 15	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	1,00 x 1,50	0,00	0,50 x 1,50	0,00	0,00
ELU CF 16	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	1,00 x 1,50	0,50 x 1,50	0,00	0,00
ELU CF 17	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,60 x 1,50	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 18	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	0,50 x 1,50	1,00 x 1,50	0,00
ELU CF 19	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	0,50 x 1,50	0,00	1,00 x 1,50
ELU CF 20	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 21	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,60 x 1,50	0,00	1,00 x 1,50	0,00	0,00
ELU CF 22	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,60 x 1,50	1,00 x 1,50	0,00	0,00
ELU CF 23	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	1,00 x 1,50	0,60 x 1,50	0,00
ELU CF 24	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,60 x 1,50
ELU CF 25	ELU CF	1,00 x 1,35	1,00 x 1,00	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 26	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	1,00 x 1,50	0,00	0,00	0,50 x 1,50	0,00	0,00
ELU CF 27	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	1,00 x 1,50	0,00	0,50 x 1,50	0,00	0,00
ELU CF 28	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	1,00 x 1,50	0,50 x 1,50	0,00	0,00
ELU CF 29	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,60 x 1,50	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 30	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	0,50 x 1,50	1,00 x 1,50	0,00
ELU CF 31	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	0,50 x 1,50	0,00	1,00 x 1,50
ELU CF 32	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 33	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,60 x 1,50	0,00	1,00 x 1,50	0,00	0,00
ELU CF 34	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,60 x 1,50	1,00 x 1,50	0,00	0,00
ELU CF 35	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,60 x 1,50	0,00

Nom combinació	Tipus	Pes propi	Càrrega permanent	Càrrega cobertes no transitables	Vent1	Vent2	Vent3	Neu	Vent12	Vent21
ELU CF 36	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,60 x 1,50
ELU CF 37	ELU CF	1,00 x 1,00	1,00 x 1,00	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 38	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	1,00 x 1,50	0,00	0,00	0,50 x 1,50	0,00	0,00
ELU CF 39	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	1,00 x 1,50	0,00	0,50 x 1,50	0,00	0,00
ELU CF 40	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	1,00 x 1,50	0,50 x 1,50	0,00	0,00
ELU CF 41	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,60 x 1,50	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 42	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,50 x 1,50	1,00 x 1,50	0,00
ELU CF 43	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,50 x 1,50	0,00	1,00 x 1,50
ELU CF 44	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
ELU CF 45	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,60 x 1,50	0,00	1,00 x 1,50	0,00	0,00
ELU CF 46	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,60 x 1,50	1,00 x 1,50	0,00	0,00
ELU CF 47	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,60 x 1,50	0,00
ELU CF 48	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,60 x 1,50
ELU CF 49	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 50	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 51	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 52	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00
ELU CF 53	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00
ELU CF 54	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00
ELU CF 55	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00
ELU CF 56	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00
ELU CF 57	ELU CF	1,00 x 1,35	1,00 x 1,35	0,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50
ELU CF 58	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00
ELU CF 59	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00

Nom combinació	Tipus	Pes propi	Càrrega permanent	Càrrega cobertes no transitables	Vent1	Vent2	Vent3	Neu	Vent12	Vent21
ELU CF 60	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00
ELU CF 61	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00
ELU CF 62	ELU CF	1,00 x 1,00	1,00 x 1,35	0,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50
ELU CF 63	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00
ELU CF 64	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00
ELU CF 65	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00
ELU CF 66	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00
ELU CF 67	ELU CF	1,00 x 1,35	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50
ELU CF 68	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00	0,00
ELU CF 69	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00	0,00
ELU CF 70	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	1,00 x 1,50	0,00	0,00	0,00
ELU CF 71	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50	0,00
ELU CF 72	ELU CF	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00	1,00 x 1,50
ELS CR 1	ELS CR	1,00 x 1,00	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00
ELS CR 2	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	1,00 x 1,00	0,00	0,00	0,50 x 1,00	0,00	0,00
ELS CR 3	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	1,00 x 1,00	0,00	0,50 x 1,00	0,00	0,00
ELS CR 4	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	1,00 x 1,00	0,50 x 1,00	0,00	0,00
ELS CR 5	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,60 x 1,00	0,00	0,00	1,00 x 1,00	0,00	0,00
ELS CR 6	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,50 x 1,00	1,00 x 1,00	0,00
ELS CR 7	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,50 x 1,00	0,00	1,00 x 1,00
ELS CR 8	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	0,00	0,00	0,00
ELS CR 9	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,60 x 1,00	0,00	1,00 x 1,00	0,00	0,00
ELS CR 10	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,60 x 1,00	1,00 x 1,00	0,00	0,00
ELS CR 11	ELS CR	1,00 x 1,00	1,00 x 1,00	0,00	0,00	0,00	0,00	1,00 x 1,00	0,60 x 1,00	0,00





## ANNEX D: Element BEAM 188

3-D Linear Finite Strain Beam

MP ME ST <> <> PR <> <> PP ED

### D.1. BEAM188 Element Description

BEAM188 is suitable for analyzing slender to moderately stubby/thick beam structures. This element is based on Timoshenko beam theory. Shear deformation effects are included.

BEAM188 is a linear (2-node) or a quadratic beam element in 3-D. BEAM188 has six or seven degrees of freedom at each node, with the number of degrees of freedom depending on the value of KEYOPT(1). When KEYOPT(1) = 0 (the default), six degrees of freedom occur at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. When KEYOPT(1) = 1, a seventh degree of freedom (warping magnitude) is also considered. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications.

BEAM188 includes stress stiffness terms, by default, in any analysis with NLGEOM,ON. The provided stress stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling or collapse studies with arc length methods).

BEAM188 can be used with any beam cross-section defined via SECTYPE, SECDATA, SECOFFSET, SECWRITE, and SECREAD. The cross-section associated with the beam may be linearly tapered.

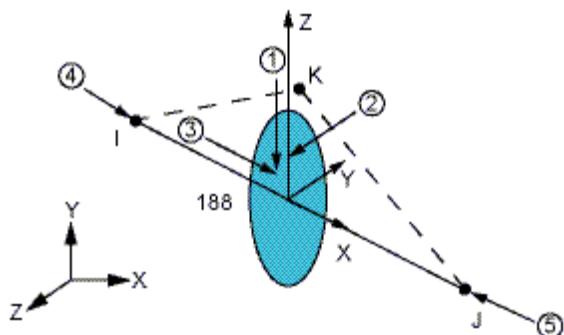
Elasticity, creep, and plasticity models are supported (irrespective of cross-section subtype). A cross-section associated with this element type can be a built-up section referencing more than one material.

BEAM188 ignores any real constant data beginning with Release 6.0. See the SECCONTROLS command for defining the transverse shear stiffness, and added mass.

For BEAM188, the element coordinate system (/PSYMB,ESYS) is not relevant.



Figure 188.1 BEAM188 Geometry



## D.2. BEAM188 Input Data

The geometry, node locations, and coordinate system for this element are shown in Figure 188.1: "BEAM188 Geometry". BEAM188 is defined by nodes I and J in the global coordinate system.

Node K is a preferred way to define the orientation of the element. For information about orientation nodes and beam meshing, see Generating a Beam Mesh With Orientation Nodes in the *ANSYS Modeling and Meshing Guide*. See the LMESH and LATT command descriptions for details on generating the K node automatically.

BEAM188 may also be defined without the orientation node. In this case, the element x-axis is oriented from node I (end 1) toward node J (end 2). For the two-node option, the default orientation of the element y-axis is automatically calculated to be parallel to the global X-Y plane. For the case where the element is parallel to the global Z-axis (or within a 0.01 percent slope of it), the element y-axis is oriented parallel to the global Y-axis (as shown). For user control of the element orientation about the element x-axis, use the third node option. If both are defined, the third node option takes precedence. The third node (K), if used, defines a plane (with I and J) containing the element x and z-axes (as shown). If this element is used in a large deflection analysis, it should be noted that the location of the third node (K) is used only to *initially* orient the element.

The beam elements are one-dimensional line elements in space. The cross-section details are provided separately using the SECTYPE and SECDATA commands (see Beam Analysis and Cross Sections in the *ANSYS Structural Analysis Guide* for details). A section is associated with the beam elements by specifying the section ID number



(SECNUM). A section number is an independent element attribute. In addition to a constant cross-section, you can also define a tapered cross-section by using the TAPER option on the SECTYPE command (see Defining a Tapered Beam).

The beam elements are based on Timoshenko beam theory, which is a first order shear deformation theory: transverse shear strain is constant through the cross-section; that is, cross-sections remain plane and undistorted after deformation. BEAM188 is a first order Timoshenko beam element which uses one point of integration along the length with default KEYOPT(3) setting. Therefore, when SMISC quantities are requested at nodes I and J, the centroidal values are reported for *both* end nodes. With KEYOPT(3) set to 2, two points of integration are used resulting in linear variation along the length.

BEAM188/BEAM189 elements can be used for slender or stout beams. Due to the limitations of first order shear deformation theory, only moderately "thick" beams may be analyzed. The slenderness ratio of a beam structure ( $GAL^2/(EI)$ ) may be used in judging the applicability of the element, where:

G

Shear modulus

A

Area of the cross section

L

Length of the member

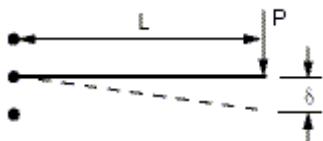
EI

Flexural rigidity

It is important to note that this ratio should be calculated using some global distance measures, and not based on individual element dimensions. The following graphic provides an estimate of transverse shear deformation in a cantilever beam subjected to a tip load. Although the results cannot be extrapolated to any other application, the example serves well as a general guideline. We recommend that the slenderness ratio should be greater than 30.



Figure 188.2 Transverse Shear Deformation Estimation



Slenderness Ratio ( $GAL^2/(EI)$ )	$\delta$ Timoshenko / $\delta$ Euler-Bernoulli
25	1.120
50	1.060
100	1.030
1000	1.003

These elements support an elastic relationship between transverse shear forces and transverse shear strains. You can override default values of transverse shear stiffnesses using the SECCONTROLS command.

The St. Venant warping functions for torsional behavior are determined in the undeformed state, and are used to define shear strain even after yielding. ANSYS does not provide options to recalculate in deformed configuration the torsional shear distribution on cross-sections during the analysis and possible partial plastic yielding of cross-sections. As such, large inelastic deformation due to torsional loading should be treated and verified with caution. Under such circumstances, alternative modeling using solid or shell elements is recommended.

BEAM188/BEAM189 elements support “restrained warping” analysis by making available a seventh degree of freedom at each beam node. By default, BEAM188 elements assume that the warping of a cross-section is small enough that it may be neglected (KEYOPT(1) = 0). You can activate the warping degree of freedom by using KEYOPT(1) = 1. With the warping degree of freedom activated, each node has seven degrees of freedom: UX, UY, UZ, ROTX, ROTY, ROTZ, and WARP. With KEYOPT(1) = 1, bimoment and bicurvature are output.

In practice, when two elements with “restrained warping” come together at a sharp angle, you need to couple the displacements and rotations, but leave the out-of-plane warping decoupled. This is normally accomplished by having two nodes at a physical location and using appropriate constraints. This process is made easier (or automated) by the



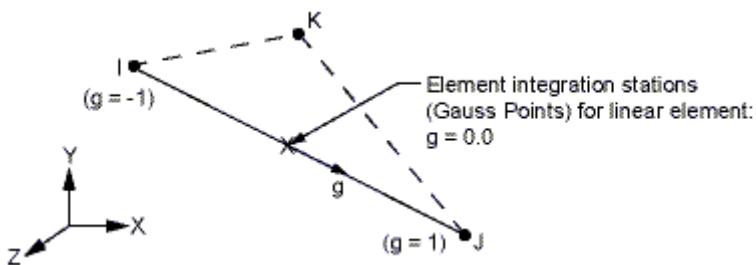
ENDRELEASE command, which decouples the out-of plane warping for any adjacent elements with cross-sections intersecting at an angle greater than 20 degrees.

BEAM188 allows change in cross-sectional inertia properties as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved after deformation. The default is suitable for elastoplastic applications. By using KEYOPT(2), you can choose to keep the cross-section constant or rigid. Scaling is not an option for nonlinear general beam sections (SECTYPE,,GENB).

Element output is available at element integration stations and at section integration points.

Integration stations (Gauss points) along the length of the beam are shown in Figure 188.3: "BEAM188 Element Integration Stations".

Figure 188.3 BEAM188 Element Integration Stations



The section strains and forces (including bending moments) may be obtained at these integration stations. The element supports output options to extrapolate such quantities to the nodes of the element.

BEAM188/BEAM189 can be associated with either of these cross section types:

Generalized beam cross sections (SECTYPE,,GENB), where the relationships of generalized stresses to generalized strains are input directly.

Standard library section types or user meshes which define the geometry of the beam cross section (SECTYPE,,BEAM). The material of the beam is defined either as an element attribute (MAT), or as part of section buildup (for multi-material cross sections).



### D.2.1. Generalized Beam Cross Sections

When using nonlinear general beam sections, neither the geometric properties nor the material is explicitly specified. *Generalized stress* implies the axial force, bending moments, torque, and transverse shear forces. Similarly, *generalized strain* implies the axial strain, bending curvatures, twisting curvature, and transverse shear strains. (For more information, see Using Nonlinear General Beam Sections.) This is an abstract method for representing cross section behavior; therefore, input often consists of experimental data or the results of other analyses.

The BEAM188/BEAM189 elements, in general, support an elastic relationship between transverse shear forces and transverse shear strains. You can override default values of transverse shear stiffnesses via the SECCONTROLS command.

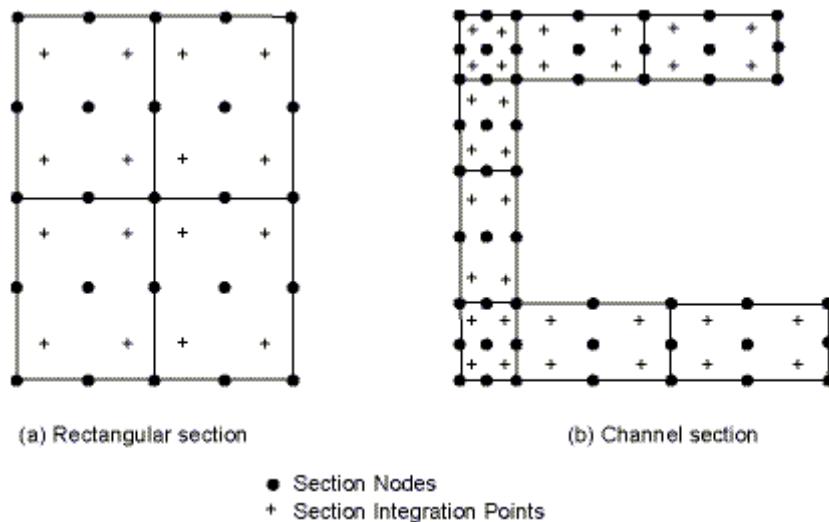
When the beam element is associated with a generalized beam (SECTYPE,,GENB) cross section type, the relationship of transverse shear force to the transverse shear strain can be nonlinear elastic or plastic, an especially useful capability when flexible spot welds are modeled. In such a case, the SECCONTROLS command does not apply.

### D.2.2. Standard Library Sections

BEAM188/BEAM189 are provided with section-relevant quantities (area of integration, position, Poisson function, function derivatives, etc.) automatically at a number of section points using SECTYPE and SECDATA. Each section is assumed to be an assembly of a predetermined number of 9-node cells. The following graphic illustrates models using the rectangular section subtype and the channel section subtype. Each cross-section cell has 4 integration points and each may be associated with an independent material type.



Figure 188.4 Cross-Section Cells



BEAM188/BEAM189 provide options for output at the section integration points and/or section nodes. You can request output only on the exterior boundary of the cross-section. (PRSSOL prints the section nodal and section integration point results. Stresses and strains are printed at section nodes, and plastic strains, plastic work, and creep strains are printed at section integration points.)

When the material associated with the elements has inelastic behavior or when the temperature varies across the section, constitutive calculations are performed at the section integration points. For more common elastic applications, the element uses precalculated properties of the section at the element integration points. However, the stresses and strains are calculated in the output pass at the section integration points.

If the section is assigned the subtype ASEC, only the generalized stresses and strains (axial force, bending moments, transverse shears, curvatures, and shear strains) are available for output. 3-D contour plots and deformed shapes are not available. The ASEC subtype can be displayed only as a thin rectangle to verify beam orientation.

BEAM188/BEAM189 allow for the analysis of built-up beams, (i.e., those fabricated of two or more pieces of material joined together to form a single, solid beam). The pieces are assumed to be perfectly bonded together. Therefore, the beam behaves as a single member.



The multi-material cross-section capability is applicable only where the assumptions of a beam behavior (Timoshenko or Bernoulli-Euler beam theory) holds.

In other words, what is supported is a simple extension of a conventional Timoshenko beam theory. It may be used in applications such as:

bimetallic strips

beams with metallic reinforcement

sensors where layers of a different material has been deposited

BEAM188/BEAM189 do not account for coupling of bending and twisting at the section stiffness level. The transverse shears are also treated in an uncoupled manner. This may have a significant effect on layered composite and sandwich beams if the layup is unbalanced.

BEAM188/BEAM189 do not use higher order theories to account for variation in distribution of shear stresses. Use ANSYS solid elements if such effects must be considered.

Always validate the application of BEAM188/BEAM189 for particular applications, either with experiments or other numerical analysis. Use the restrained warping option with built-up sections after due verification.

For the mass matrix and evaluation of consistent load vectors, a higher order integration rule than that used for stiffness matrix is employed. The elements support both consistent and lumped mass matrices. Use LUMPM,ON to activate lumped mass matrix. Consistent mass matrix is used by default. An added mass per unit length may be input with the ADDMAS section controls. See "BEAM188 Input Summary".

Forces are applied at the nodes (which also define the element x-axis). If the centroidal axis is not colinear with the element x-axis, applied axial forces will cause bending. Applied shear forces will cause torsional strains and moment if the centroid and shear center of the cross-section are different. The nodes should therefore be located at the desired points where you want to apply the forces. Use the *OFFSETY* and *OFFSETZ* arguments of the SECOFFSET command appropriately. By default, ANSYS uses the centroid as the reference axis for the beam elements.



Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on Figure 188.1: "BEAM188 Geometry". Positive normal pressures act into the element. Lateral pressures are input as force per unit length. End "pressures" are input as forces.

When KEYOPT(3) = 0 (default), BEAM188 is based on linear polynomials, unlike other Hermitian polynomial-based elements (for example, BEAM4). Refinement of the mesh is recommended in general.

When KEYOPT(3) = 2, ANSYS adds an internal node in the interpolation scheme, effectively making this a Timoshenko beam element based on quadratic shape functions. This option is highly recommended unless this element is used as a stiffener and you must maintain compatibility with a first order shell element. Linearly varying bending moments are represented exactly. The quadratic option is similar to BEAM189, with the following differences:

The initial geometry is always a straight line with BEAM188 with or without the quadratic option.

You cannot access the internal node; and thus boundary conditions/loading cannot be specified on those nodes.

Offsets in specification of distributed loads are not allowed. Non-nodal concentrated forces are not supported. Use the quadratic option (KEYOPT(3) = 2) when the element is associated with tapered cross-sections.

Temperatures may be input as element body loads at three locations at each end node of the beam. At each end, the element temperatures are input at the element x-axis ( $T(0,0)$ ), at one unit from the x-axis in the element y-direction ( $T(1,0)$ ), and at one unit from the x-axis in the element z-direction ( $T(0,1)$ ). The first coordinate temperature  $T(0,0)$  defaults to TUNIF. If all temperatures after the first are unspecified, they default to the first. If all temperatures at node I are input, and all temperatures at node J are unspecified, the node J temperatures default to the corresponding node I temperatures. For any other input pattern, unspecified temperatures default to TUNIF.

You can apply an initial stress state to this element through the ISTRESS or ISFILE command. For more information, see Initial Stress Loading in the *ANSYS Basic Analysis*



*Guide.* Alternately, you can set KEYOPT(10) = 1 to read initial stresses from the user subroutine USTRESS. For details on user subroutines, see the *Guide to ANSYS User Programmable Features*.

The effects of pressure load stiffness are automatically included for this element. If an unsymmetric matrix is needed for pressure load stiffness effects, use NROPT,UNSYM.

A summary of the element input is given in "BEAM188 Input Summary".

### BEAM188 Input Summary

#### Nodes

I, J, K (K, the orientation node, is optional but recommended)

#### Degrees of Freedom

UX, UY, UZ, ROTX, ROTY, ROTZ if KEYOPT(1) = 0

UX, UY, UZ, ROTX, ROTY, ROTZ, WARP if KEYOPT(1) = 1

#### Section Controls

TXZ, TXY, ADDMAS (See SECCONTROLS)

(TXZ and TXY default to A\*GXZ and A\*GXY, respectively, where A = cross-sectional area)

#### Material Properties

EX, (PRXY or NUXY), ALPX, DENS, GXY, GYZ, GXZ, DAMP

#### Surface Loads

##### Pressure --

face 1 (I-J) (-z normal direction),

face 2 (I-J) (-y normal direction),



face 3 (I-J) (+x tangential direction),

face 4 (J) (+x axial direction),

face 5 (I) (-x direction).

(use a negative value for loading in the opposite direction)

I and J denote the end nodes.

## Body Loads

### Temperatures --

T(0,0), T(1,0), T(0,1) at each end node

## Special Features

Plasticity

Viscoelasticity

Viscoplasticity

Creep

Stress stiffening

Large deflection

Large strain

Initial stress import

Birth and death (requires KEYOPT(11) = 1)

Automatic selection of element technology



Supports the following types of data tables associated with the TB command:  
BISO, MISO, NLISO, BKIN, MKIN, KINH, CHABOCHE, HILL, RATE, CREEP,  
PRONY, SHIFT, PLASTIC, and USER.

Generalized cross section (nonlinear elastic, elasto-plastic, temperature-dependent)

#### Note

See the *ANSYS, Inc. Theory Reference* for details of the material models.

See Automatic Selection of Element Technologies and ETCONTROL for more information on selection of element technologies.

#### KEYOPT(1)

Warping degree of freedom:

0 --

Default; six DOF, unrestrained warping

1 --

Seven DOF (including warping). Bimoment and bicurvature are output.

#### KEYOPT(2)

Cross-section scaling, applies only if NLGEOM,ON has been invoked:

0 --

Default; cross-section is scaled as a function of axial stretch

1 --

Section is assumed to be rigid (classical beam theory)

#### KEYOPT(3)



Interpolation scheme:

0 --

Default; linear polynomial. Mesh refinement is recommended.

2 --

Quadratic shape functions (effectively a Timoshenko beam element); uses an internal node (inaccessible to users) to enhance element accuracy, allowing exact representation of linearly varying bending moments

KEYOPT(4)

Shear stress output:

0 --

Default; output only torsion-related shear stresses

1 --

Output only flexure-related transverse shear stresses

2 --

Output a combined state of the previous two types.

KEYOPT(6)

Output control at element integration point:

0 --

Default; output section forces, strains, and bending moments

1 --

Same as KEYOPT(6) = 0 plus current section area

2 --



Same as KEYOPT(6) = 1 plus element basis directions (X,Y,Z)

3 --

Output section forces/moment and strains/curvatures extrapolated to element nodes

Note

KEYOPT(6) through KEYOPT(9) are active only when OUTPR,ESOL is active. When KEYOPT(6), (7), (8), and (9) are active, the strains reported in the element output are total strains. "Total" implies the inclusion of thermal strains. When the material associated with the element has plasticity, plastic strain and plastic work are also provided. Alternatively, use PRSSOL in /POST1.

KEYOPT(7)

Output control at section integration point (not available when section subtype = ASEC):

0 --

Default; none

1 --

Maximum and minimum stresses/strains

2 --

Same as KEYOPT(7) = 1 plus stresses and strains at each section point

KEYOPT(8)

Output control at section nodes (not available when section subtype = ASEC):

0 --

Default; none

1 --

Maximum and minimum stresses/strains



2 --

Same as KEYOPT(8) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --

Same as KEYOPT(8) = 1 plus stresses and strains at each section node

KEYOPT(9)

Output control for extrapolated values at element nodes and section nodes (not available when section subtype = ASEC):

0 --

Default; none

1 --

Maximum and minimum stresses/strains

2 --

Same as KEYOPT(9) = 1 plus stresses and strains along the exterior boundary of the cross-section

3 --

Same as KEYOPT(9) = 1 plus stresses and strains at all section nodes

KEYOPT(10)

User-defined initial stresses:

0 --

No user subroutine to provide initial stresses (default)

1 --



Read initial stress data from user subroutine USTRESS

Note

See the *Guide to ANSYS User Programmable Features* for user written subroutines.

KEYOPT(11)

Set section properties:

0 --

Automatically determine if pre-integrated section properties can be used (default)

1 --

Use numerical integration of section (required for birth/death functionality)

KEYOPT(12)

Tapered section treatment:

0 --

Linear tapered section analysis; cross section properties are evaluated at each Gauss point (default). This is more accurate, but computationally intense.

1 --

Average cross section analysis; for elements with tapered sections, cross section properties are evaluated at the centroid only. This is an approximation of the order of the mesh size; however, it is faster.

## D.3. BEAM188 Output Data

The solution output associated with these elements is in two forms:

Nodal displacements and reactions included in the overall nodal solution

Additional element output as described in Table 188.1: "BEAM188 Element Output Definitions"



Where necessary, ANSYS recommends KEYOPT(8) = 2 and KEYOPT(9) = 2. See the *ANSYS Basic Analysis Guide* for ways to view results.

To view 3-D deformed shapes for BEAM188, issue an OUTRES,MISC or OUTRES,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, you must expand the modes with element results calculation active (via the MXPAND command's *Elcalc* = YES option).

### Linearized Stress

It is customary in beam design to employ components of axial stress that contribute to axial loads and bending in each direction separately. Therefore, BEAM188 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions:

SDIR is the stress component due to axial load.

SDIR =  $FX/A$ , where FX is the axial load (SMISC quantities 1 and 14) and A is the area of the cross section.

SBYT and SBYB are bending stress components.

$$SBYT = -M_Z * y_{max} / I_{zz}$$

$$SBYB = -M_Z * y_{min} / I_{zz}$$

$$SBZT = M_Y * z_{max} / I_{yy}$$

$$SBZB = M_Y * z_{min} / I_{yy}$$

where MY, MZ are bending moments (SMISC quantities 2,15,3,16). Coordinates  $y_{max}$ ,  $y_{min}$ ,  $z_{max}$ , and  $z_{min}$  are the maximum and minimum y, z coordinates in the cross section measured from the centroid. Values Iyy and Izzy are moments of inertia of the cross section. Except for the ASEC type of beam cross section, ANSYS uses the maximum and minimum cross section dimensions. For the ASEC type of cross section, the maximum and minimum in each of Y and Z direction is assumed to be +0.5 to -0.5, respectively.

Corresponding definitions for the component strains are:



EPELDIR = EX

EPELBYT = -KZ \*  $y_{\max}$

EPELBYB = -KZ \*  $y_{\min}$

EPELBZT = KY \*  $z_{\max}$

EPELBZB = KY \*  $z_{\min}$

where EX, KY, and KZ are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22).

The reported stresses are strictly valid only for elastic behavior of members. BEAM188 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses may at best be regarded as linearized approximations and should be interpreted with caution.

The Element Output Definitions table uses the following notation:

A colon (:) in the Name column indicates the item can be accessed by the Component Name method [ETABLE, ESOL]. The O column indicates the availability of the items in the file Jobname.OUT. The R column indicates the availability of the items in the results file.

In either the O or R columns, Y indicates that the item is *always* available, a number refers to a table footnote that describes when the item is *conditionally* available, and a - indicates that the item is *not* available.

Table 188.1 BEAM188 Element Output Definitions

Name	Definition	O	R
EL	Element number	Y	Y
NODES	Element connectivity	Y	Y
MAT	Material number	Y	Y
C.G.:X, Y, Z	Center of gravity	Y	Y
AREA	Area of cross-section	1	Y
SF:Y, Z	Section shear forces	1	Y



Name	Definition	O	R
SE:Y, Z	Section shear strains	1	Y
S:XX, XZ, XY	Section point stresses	2	Y
E:XX, XZ, XY	Section point strains	2	Y
MX	Torsional moment	Y	Y
KX	Torsional strain	Y	Y
KY, KZ	Curvature	Y	Y
EX	Axial strain	Y	Y
FX	Axial force	Y	Y
MY, MZ	Bending moments	Y	Y
BM	Bimoment	3	3
BK	Bicurvature	3	3
SDIR	Axial direct stress	-	1
SBYT	Bending stress on the element +Y side of the beam	-	1
SBYB	Bending stress on the element -Y side of the beam	-	1
SBZT	Bending stress on the element +Z side of the beam	-	1
SBZB	Bending stress on the element -Z side of the beam	-	1
EPELDIR	Axial strain at the end	-	1
EPELBYT	Bending strain on the element +Y side of the beam.	-	1
EPELBYB	Bending strain on the element -Y side of the beam.	-	1
EPELBZT	Bending strain on the element +Z side of the beam.	-	1
EPELBZB	Bending strain on the element -Z side of the beam.	-	1
TEMP	Temperatures T0, T1(1,0), T2(0,1)	-	1

**Note**

More output is described via the PRSSOL command in /POST1.

See KEYOPT(6) description.

See KEYOPT(7), KEYOPT(8), KEYOPT(9) descriptions.

See KEYOPT(1) description.

Table 188.2: "BEAM188 Item and Sequence Numbers" lists output available through ETABLE using the Sequence Number method. See Creating an Element Table in the *ANSYS Basic Analysis Guide* and The Item and Sequence Number Table in this manual for more information. Table 188.2: "BEAM188 Item and Sequence Numbers" uses the following notation:

Name



output quantity as defined in the Table 188.1: "BEAM188 Element Output Definitions"

Item

predetermined Item label for ETABLE

I,J

sequence number for data at nodes I and J

Table 188.2 BEAM188 Item and Sequence Numbers

Output Quantity Name	ETABLE and ESOL Command Input		
	Item	I	J
FX	SMISC	1	14
MY	SMISC	2	15
MZ	SMISC	3	16
MX	SMISC	4	17
SFZ	SMISC	5	18
SFY	SMISC	6	19
EX	SMISC	7	20
KY	SMISC	8	21
KZ	SMISC	9	22
KX	SMISC	10	23
SEZ	SMISC	11	24
SEY	SMISC	12	25
Area	SMISC	13	26
BM	SMISC	27	29
BK	SMISC	28	30
SDIR	SMISC	31	36
SBYT	SMISC	32	37
SBYB	SMISC	33	38
SBZT	SMISC	34	39
SBZB	SMISC	35	40
EPELDIR	SMISC	41	46
EPELBYT	SMISC	42	47
EPELBYB	SMISC	43	48
EPELBZT	SMISC	44	49
EPELBZB	SMISC	45	50
TEMP	SMISC	51-53	54-56



### Transverse Shear Stress Output

BEAM188/BEAM189 formulation is based on three stress components:

one axial

two shear stress components

The shear stresses are caused by torsional and transverse loads. BEAM188/BEAM189 are based on first order shear deformation theory, also popularly known as Timoshenko Beam theory. The transverse shear strain is constant for the cross section, and hence the shear energy is based on a transverse shear force. This shear force is redistributed by predetermined shear stress distribution coefficients across the beam cross-section, and made available for output purposes. By default, ANSYS will only output the shear stresses caused by torsional loading. KEYOPT(4) of BEAM188/BEAM189 may be used to activate output of shear stresses caused by flexure or transverse loading.

The accuracy of transverse shear distribution is directly proportional to the mesh density of cross-section modeling (for determination of warping, shear center and other section geometric properties). The traction free state at the edges of cross-section, is met only in a well-refined model of the cross-section.

By default, ANSYS uses a mesh density (for cross-section model) that provides accurate results for torsional rigidity, warping rigidity, inertia properties, and shear center determination. The default mesh employed is also appropriate for nonlinear material calculations. However, more refined cross-section models may be necessary if the shear stress distribution due to transverse loads must be captured very accurately. Note that increasing cross-section mesh size, does not imply larger computational cost if the associated material is linear. SECTYPE and SECDATA command descriptions allow specification of cross-section mesh density.

The transverse shear distribution calculation neglects the effects of Poisson's ratio. The Poisson's ratio affects the shear correction factor and shear stress distribution slightly.

## D.4. BEAM188 Assumptions and Restrictions

The beam must not have zero length.



By default (KEYOPT(1) = 0), the effect of warping restraint is assumed to be negligible.

Cross-section failure or folding is not accounted for.

Rotational degrees of freedom are not included in the lumped mass matrix if offsets are present.

It is a common practice in civil engineering to model the frame members of a typical multi-storyed structure using a single element for each member. Because of cubic interpolation of lateral displacement, BEAM4 and BEAM44 are well-suited for such an approach. However, if BEAM188 is used in that type of application, be sure to use several elements for each frame member. BEAM188 includes the effects of transverse shear.

This element works best with the full Newton-Raphson solution scheme (that is, the default choice in solution control). For nonlinear problems that are dominated by large rotations, we recommend that you do *not* use PRED,ON.

Note that only moderately "thick" beams may be analyzed. See "BEAM188 Input Data" for more information.

When a cross-section has multiple materials and you issue the /ESHAPE command (which displays elements with shapes determined from the real constants or section definition) to produce contour plots of stresses (and other quantities), the element averages the stresses across material boundaries. To limit this behavior, use small cross-section cells around the material boundaries. There are no input options to bypass this behavior.

For this element, the /ESHAPE command supports visualization of stresses, but not of plastic strains.

Stress stiffening is always included in geometrically nonlinear analyses (NLGEOM,ON). It is ignored in geometrically linear analyses (NLGEOM,OFF) when specified by SSTIF,ON. Prestress effects can be activated by the PSTRES command.

When the element is associated with nonlinear general beam sections (SECTYPE,,GENB), additional restrictions apply. For more information, see Considerations for Employing Nonlinear General Beam Sections.



### BEAM188 Product Restrictions

When used in the product(s) listed below, the stated product-specific restrictions apply to this element in addition to the general assumptions and restrictions given in the previous section.

ANSYS Professional.

The only special features allowed are stress stiffening and large deflections.





## ANNEX E: Coordenades pòrtic

Punt	COORDENADES							
	Generatriu		Imperfeccions Globals		Imperfeccions Locals		Generatriu + Imperfeccions	
	x (Absolutes)	y (Absolutes)	x (Relatives)	y (Relatives)	x (Relatives)	y (Relatives)	x (Absolutes)	y (Absolutes)
1	0	0	0	0	0	0	0	0
2	21,673595	426,52912	1,1898905	0	5,344796249	0	28,20828175	426,52912
3	43,34719	853,05824	2,379781	0	10,55798588	0	56,28495688	853,05824
4	65,020785	1279,58736	3,5696715	0	15,51120285	0	84,10165935	1279,58736
5	86,69438	1706,11648	4,759562	0	20,08248253	0	111,5364245	1706,11648
6	108,367975	2132,6456	5,9494525	0	24,15926484	0	138,4766923	2132,6456
7	130,04157	2559,17472	7,139343	0	27,64116588	0	164,8220789	2559,17472
8	151,715165	2985,70384	8,3292335	0	30,44244967	0	190,4868482	2985,70384
9	173,38876	3412,23296	9,519124	0	32,49413932	0	215,4020233	3412,23296
10	195,062355	3838,76208	10,7090145	0	33,74571541	0	239,5170849	3838,76208
11	216,73595	4265,2912	11,898905	0	34,16636	0	262,801215	4265,2912
12	238,409545	4691,82032	13,0887955	0	33,74571541	0	285,2440559	4691,82032
13	260,08314	5118,34944	14,278686	0	32,49413932	0	306,8559653	5118,34944
14	281,756735	5544,87856	15,4685765	0	30,44244967	0	327,6677612	5544,87856
15	303,43033	5971,40768	16,658467	0	27,64116588	0	347,7299629	5971,40768
16	325,103925	6397,9368	17,8483575	0	24,15926484	0	367,1115473	6397,9368
17	346,77752	6824,46592	19,038248	0	20,08248253	0	385,8982505	6824,46592
18	368,451115	7250,99504	20,2281385	0	15,51120285	0	404,1904564	7250,99504
19	390,12471	7677,52416	21,418029	0	10,55798588	0	422,1007249	7677,52416
20	411,798305	8104,05328	22,6079195	0	5,344796249	0	439,7510207	8104,05328
21	433,4719	8530,5824	23,79781	0	0	0	457,26971	8530,5824

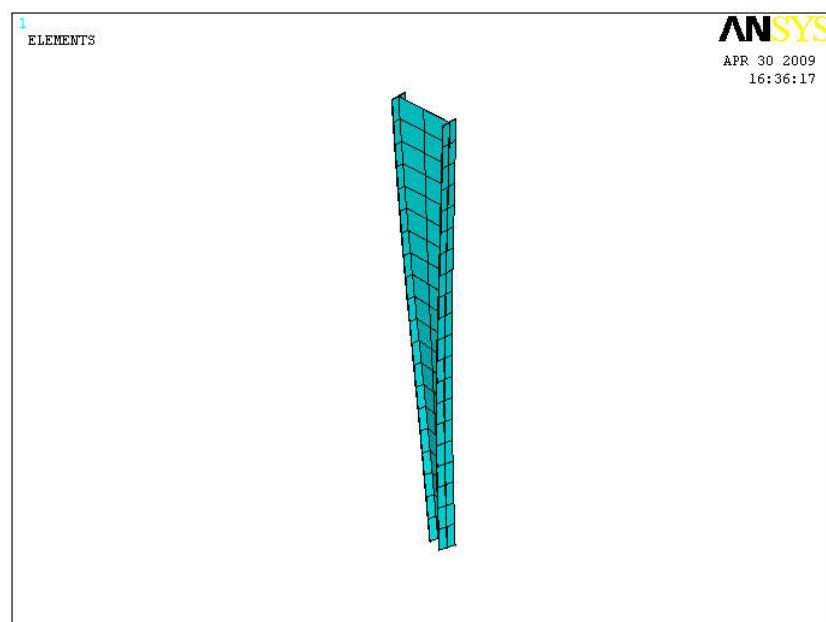
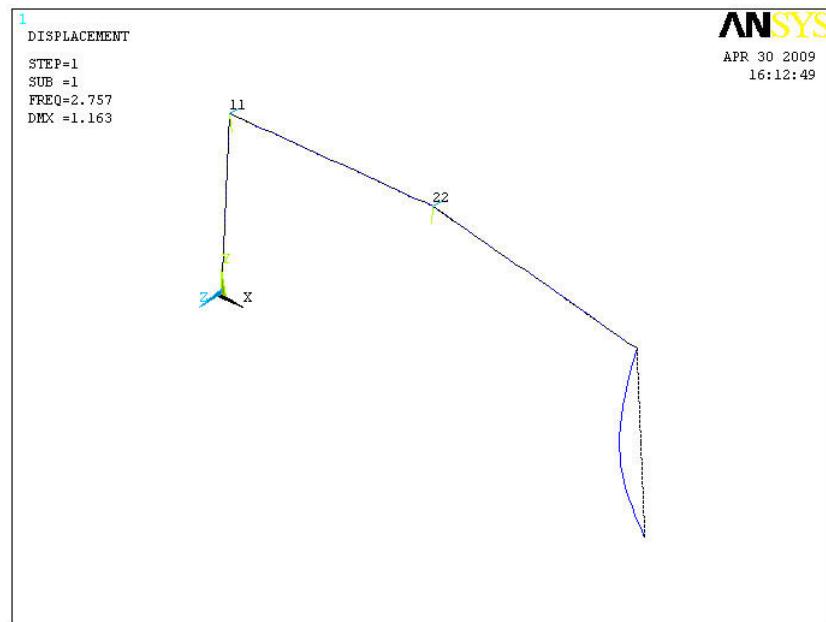
Punt	COORDENADES							
	Generatriu		Imperfeccions Globals		Imperfeccions Locals		Generatriu + Imperfeccions	
	x (Absolutes)	y (Absolutes)	x (Relatives)	y (Relatives)	x (Relatives)	y (Relatives)	x (Absolutes)	y (Absolutes)
22	974,81767	8587,15389	23,79781	0	0	0	998,61548	8587,15389
23	1516,16344	8643,72538	23,79781	0	0	0	1539,96125	8643,72538
24	2057,50921	8700,29687	23,79781	0	0	0	2081,30702	8700,29687
25	2598,85498	8756,86836	23,79781	0	0	0	2622,65279	8756,86836
26	3140,20075	8813,43985	23,79781	0	0	0	3163,99856	8813,43985
27	3681,54652	8870,01134	23,79781	0	0	0	3705,34433	8870,01134
28	4222,89229	8926,58283	23,79781	0	0	0	4246,6901	8926,58283
29	4764,23806	8983,15432	23,79781	0	0	0	4788,03587	8983,15432
30	5305,58383	9039,72581	23,79781	0	0	0	5329,38164	9039,72581
31	5846,9296	9096,2973	23,79781	0	0	0	5870,72741	9096,2973
32	6388,27537	9152,86879	23,79781	0	0	0	6412,07318	9152,86879
33	6929,62114	9209,44028	23,79781	0	0	0	6953,41895	9209,44028
34	7470,96691	9266,01177	23,79781	0	0	0	7494,76472	9266,01177
35	8012,31268	9322,58326	23,79781	0	0	0	8036,11049	9322,58326
36	8553,65845	9379,15475	23,79781	0	0	0	8577,45626	9379,15475
37	9095,00422	9435,72624	23,79781	0	0	0	9118,80203	9435,72624
38	9636,34999	9492,29773	23,79781	0	0	0	9660,1478	9492,29773
39	10177,69576	9548,86922	23,79781	0	0	0	10201,49357	9548,86922
40	10719,04153	9605,44071	23,79781	0	0	0	10742,83934	9605,44071
41	11260,3873	9662,0122	23,79781	0	0	0	11284,18511	9662,0122
42	11801,73307	9605,44071	23,79781	0	0	0	11825,53088	9605,44071
43	12343,07884	9548,86922	23,79781	0	0	0	12366,87665	9548,86922

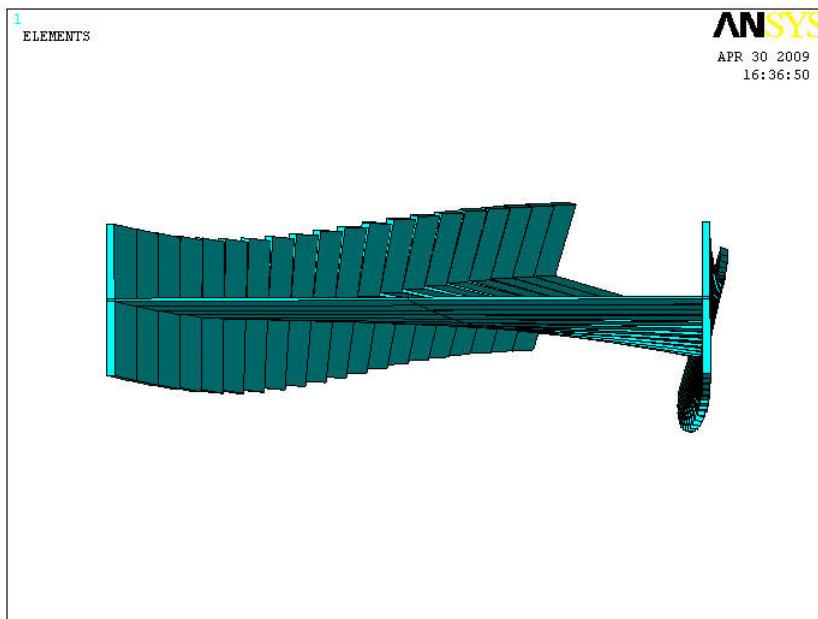
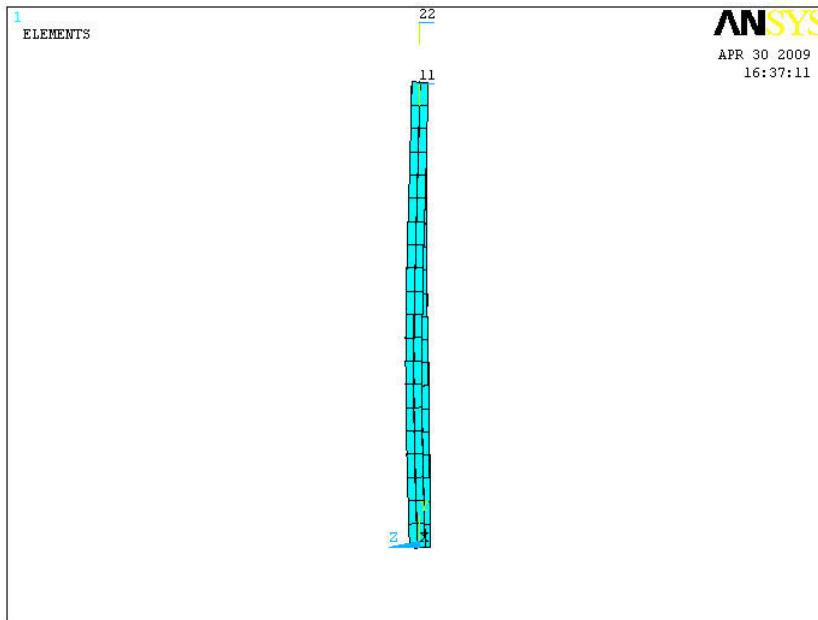
Punt	COORDENADES							
	Generatriu		Imperfeccions Globals		Imperfeccions Locals		Generatriu + Imperfeccions	
	x (Absolutes)	y (Absolutes)	x (Relatives)	y (Relatives)	x (Relatives)	y (Relatives)	x (Absolutes)	y (Absolutes)
44	12884,42461	9492,29773	23,79781	0	0	0	12908,22242	9492,29773
45	13425,77038	9435,72624	23,79781	0	0	0	13449,56819	9435,72624
46	13967,11615	9379,15475	23,79781	0	0	0	13990,91396	9379,15475
47	14508,46192	9322,58326	23,79781	0	0	0	14532,25973	9322,58326
48	15049,80769	9266,01177	23,79781	0	0	0	15073,6055	9266,01177
49	15591,15346	9209,44028	23,79781	0	0	0	15614,95127	9209,44028
50	16132,49923	9152,86879	23,79781	0	0	0	16156,29704	9152,86879
51	16673,845	9096,2973	23,79781	0	0	0	16697,64281	9096,2973
52	17215,19077	9039,72581	23,79781	0	0	0	17238,98858	9039,72581
53	17756,53654	8983,15432	23,79781	0	0	0	17780,33435	8983,15432
54	18297,88231	8926,58283	23,79781	0	0	0	18321,68012	8926,58283
55	18839,22808	8870,01134	23,79781	0	0	0	18863,02589	8870,01134
56	19380,57385	8813,43985	23,79781	0	0	0	19404,37166	8813,43985
57	19921,91962	8756,86836	23,79781	0	0	0	19945,71743	8756,86836
58	20463,26539	8700,29687	23,79781	0	0	0	20487,0632	8700,29687
59	21004,61116	8643,72538	23,79781	0	0	0	21028,40897	8643,72538
60	21545,95693	8587,15389	23,79781	0	0	0	21569,75474	8587,15389
61	22087,3027	8530,5824	23,79781	0	0	0	22111,10051	8530,5824
62	22108,9763	8104,05328	22,6079195	0	5,344796249	0	22136,92901	8104,05328
63	22130,64989	7677,52416	21,418029	0	10,55798588	0	22162,6259	7677,52416
64	22152,32349	7250,99504	20,2281385	0	15,51120285	0	22188,06283	7250,99504
65	22173,99708	6824,46592	19,038248	0	20,08248253	0	22213,11781	6824,46592

Punt	COORDENADES							
	Generatriu		Imperfeccions Globals		Imperfeccions Locals		Generatriu + Imperfeccions	
	x (Absolutes)	y (Absolutes)	x (Relatives)	y (Relatives)	x (Relatives)	y (Relatives)	x (Absolutes)	y (Absolutes)
66	22195,67068	6397,9368	17,8483575	0	24,15926484	0	22237,6783	6397,9368
67	22217,34427	5971,40768	16,658467	0	27,64116588	0	22261,6439	5971,40768
68	22239,01787	5544,87856	15,4685765	0	30,44244967	0	22284,92889	5544,87856
69	22260,69146	5118,34944	14,278686	0	32,49413932	0	22307,46429	5118,34944
70	22282,36506	4691,82032	13,0887955	0	33,74571541	0	22329,19957	4691,82032
71	22304,03865	4265,2912	11,898905	0	34,16636	0	22350,10392	4265,2912
72	22325,71225	3838,76208	10,7090145	0	33,74571541	0	22370,16697	3838,76208
73	22347,38584	3412,23296	9,519124	0	32,49413932	0	22389,3991	3412,23296
74	22369,05944	2985,70384	8,3292335	0	30,44244967	0	22407,83112	2985,70384
75	22390,73303	2559,17472	7,139343	0	27,64116588	0	22425,51354	2559,17472
76	22412,40663	2132,6456	5,9494525	0	24,15926484	0	22442,51534	2132,6456
77	22434,08022	1706,11648	4,759562	0	20,08248253	0	22458,92226	1706,11648
78	22455,75382	1279,58736	3,5696715	0	15,51120285	0	22474,83469	1279,58736
79	22477,42741	853,05824	2,379781	0	10,55798588	0	22490,36518	853,05824
80	22499,10101	426,52912	1,1898905	0	5,344796249	0	22505,63569	426,52912
81	22520,7746	0	0	0	0	0	22520,7746	0

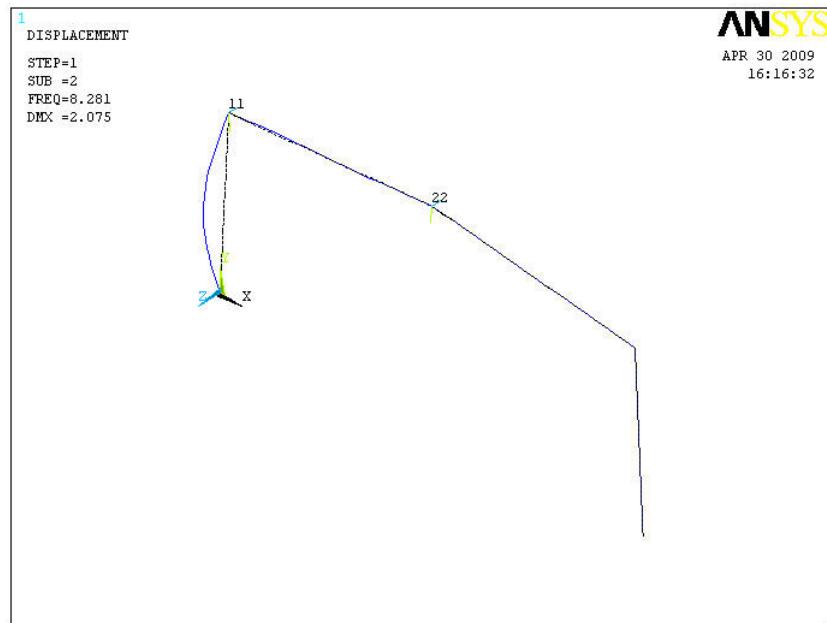
## ANNEX F: Modes vinclament 2<sup>a</sup> etapa Mètode General

- Primer mode (FREQ=2,757):

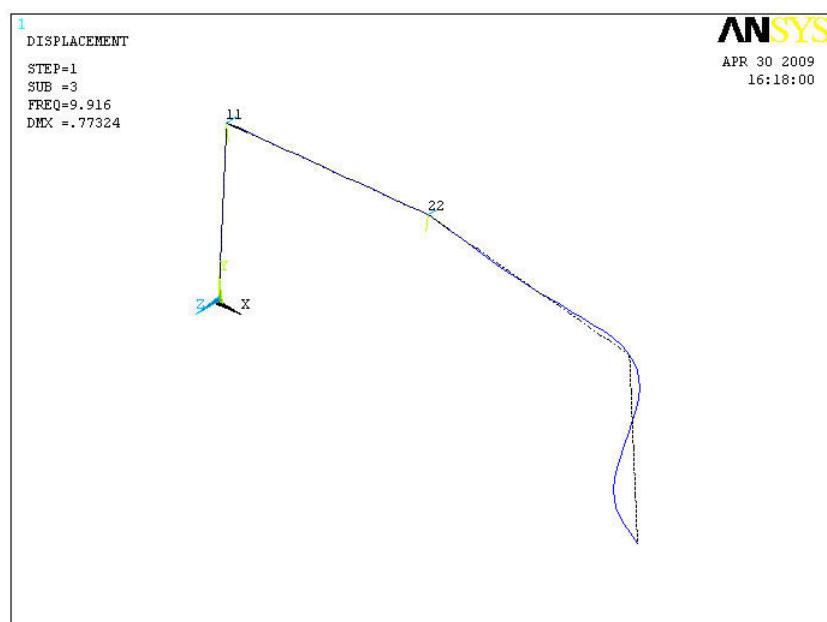


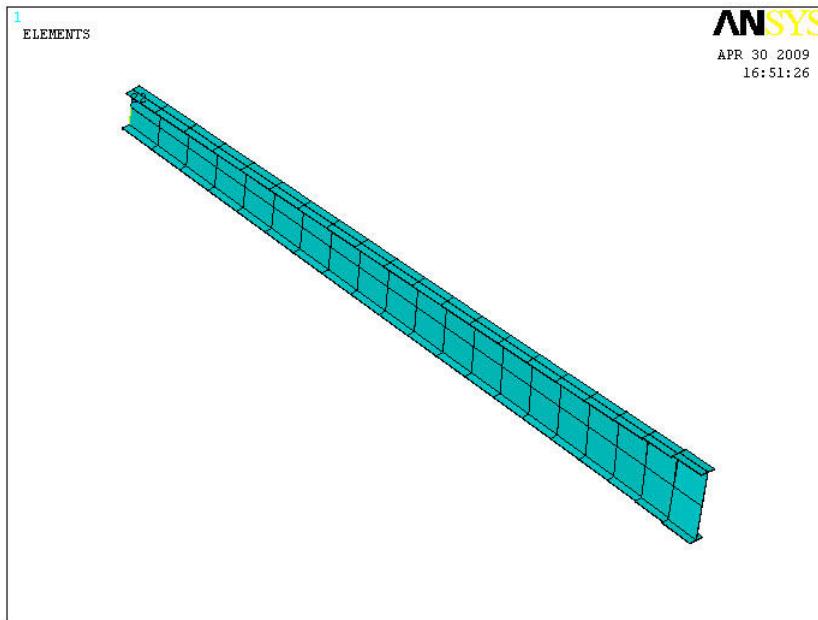


- Segon mode (FREQ=8,281):

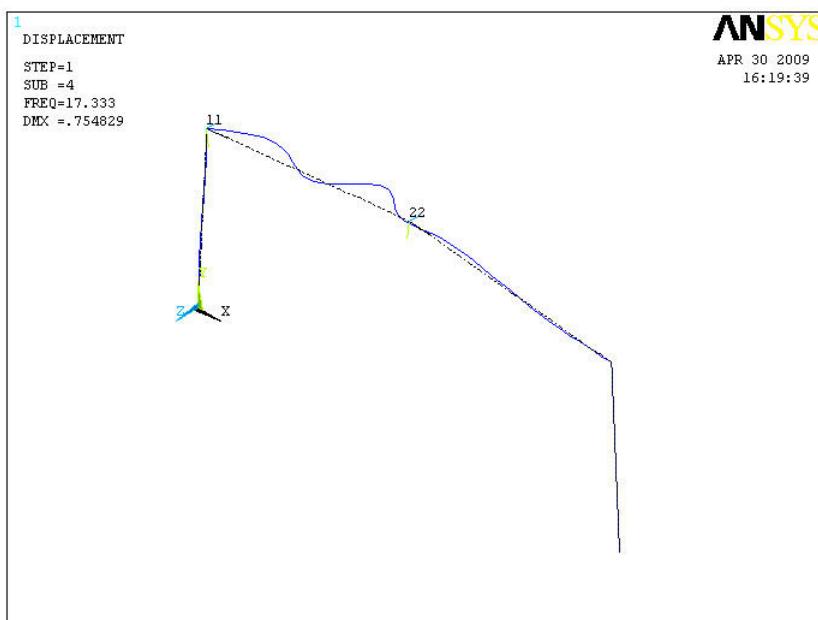


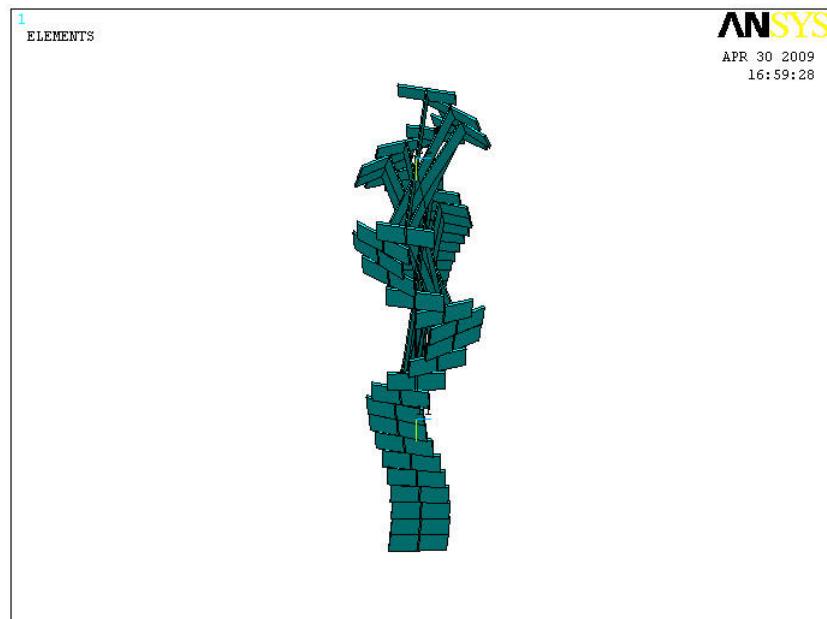
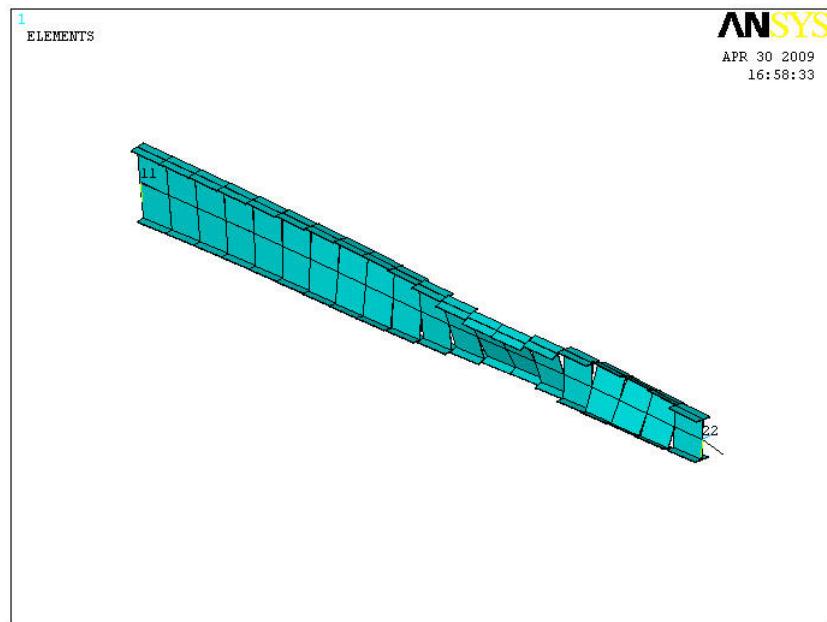
- Tercer mode (FREQ=9,916):



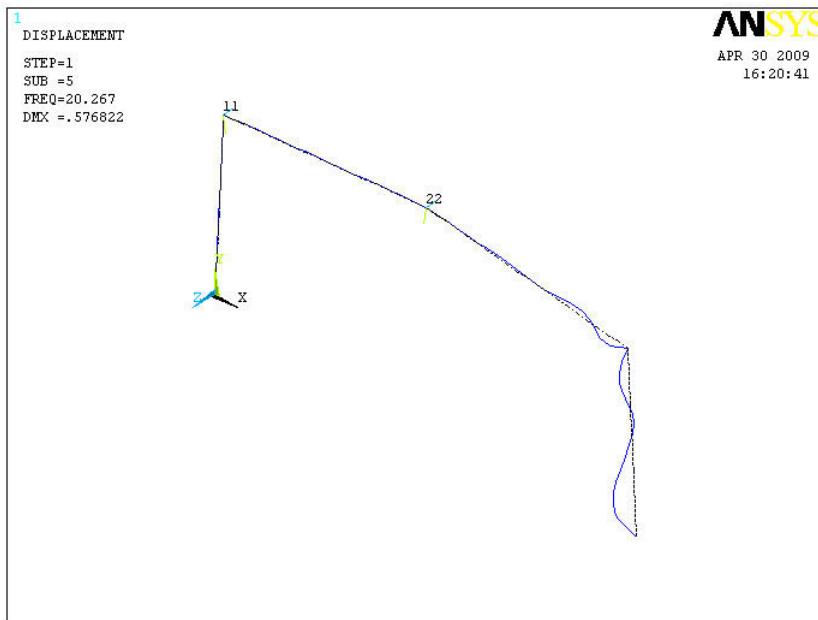


- Quart mode (FREQ=17,333):

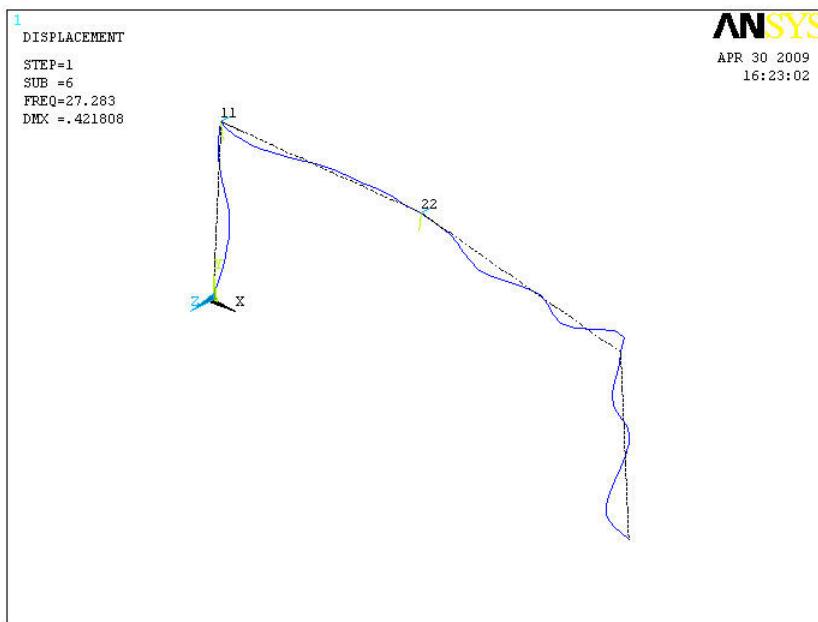




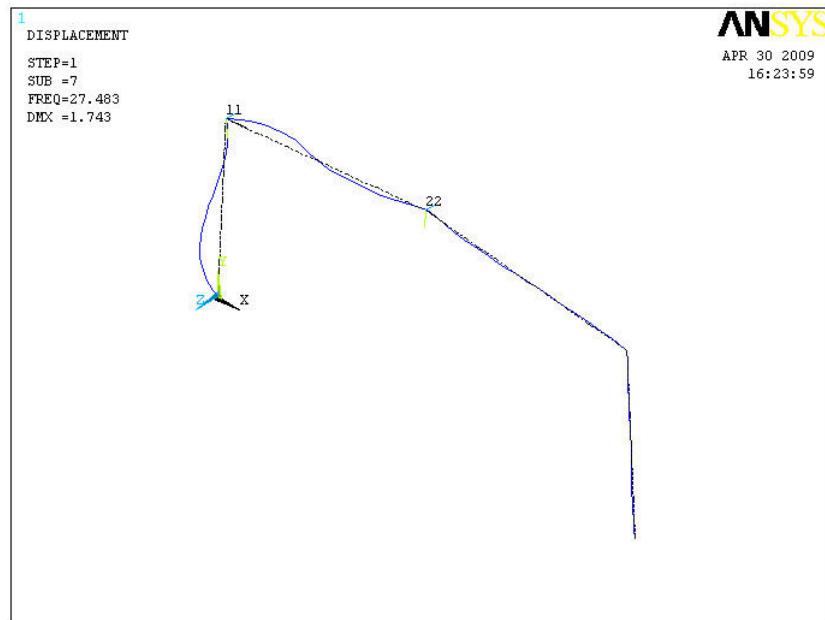
- Cinquè mode (FREQ=20,267):



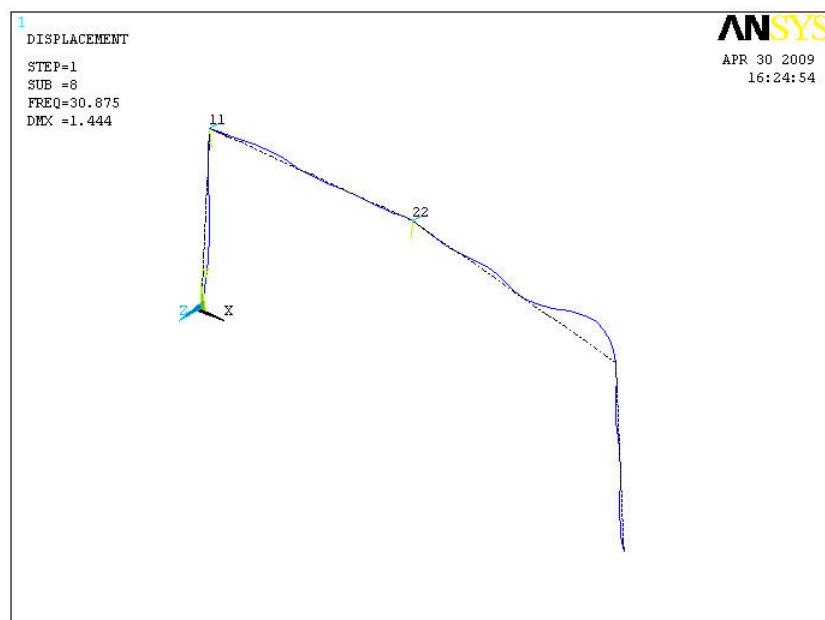
- Sisè mode (FREQ=27,283):



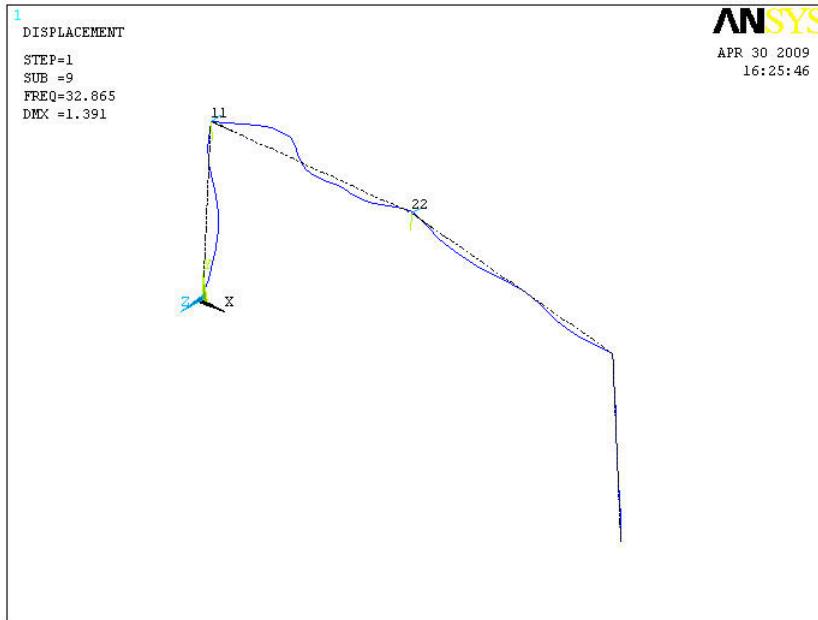
- Setè mode (FREQ=27,483):



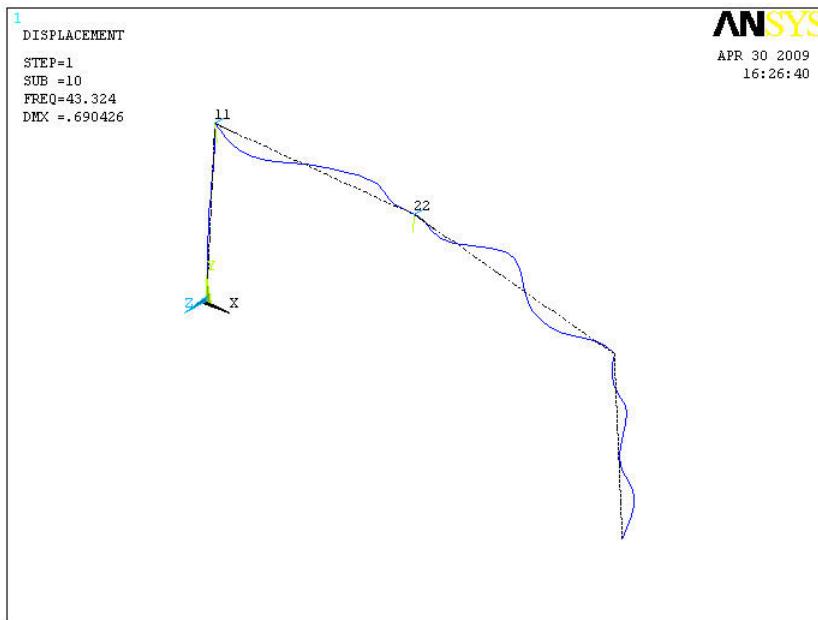
- Vuitè mode (FREQ=30,875):



- Novè mode (FREQ=32,865):



- Desè mode (FREQ=43,324):

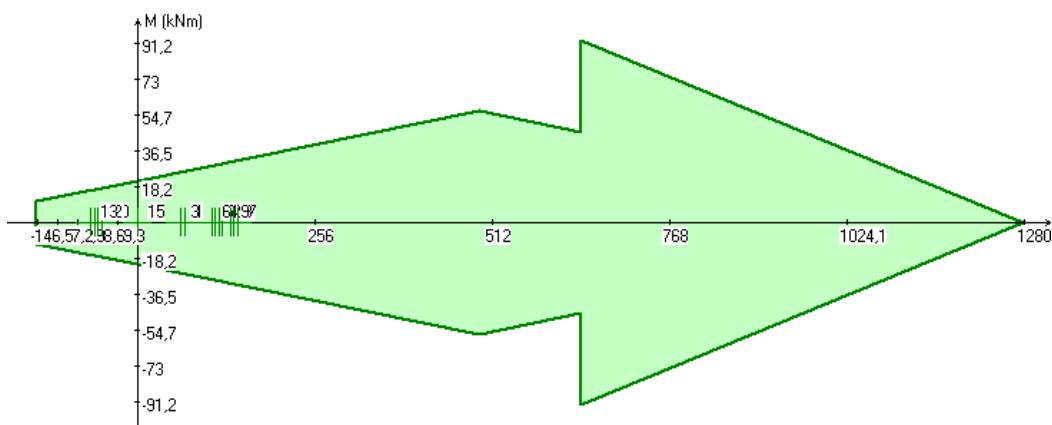


## ANNEX G: Càcul d'unions

El present annex conté el càcul de les diferents unions del pòrtic, totes elles calculades a través del programa PowerConnect, seguint la norma EN 1993-1-8:2005.

### G.1. Càcul base pilar articulada

#### Momento y esfuerzo normal



#### Lista de combinaciones

- |                       |                       |                       |
|-----------------------|-----------------------|-----------------------|
| 1) ELU CF 7 (1) : V   | 2) ELU CF 12 (1) : V  | 3) ELU CF 19 (1) : V  |
| 4) ELU CF 24 (1) : V  | 5) ELU CF 36 (1) : V  | 6) ELU CF 48 (1) : V  |
| 7) ELU CF 49 (1) : V  | 8) ELU CF 50 (1) : V  | 9) ELU CF 51 (1) : V  |
| 10) ELU CF 53 (1) : V | 11) ELU CF 58 (1) : V | 12) ELU CF 63 (1) : V |
| 13) ELU CF 68 (1) : V | 14) M+ : V            | 15) M- : V            |

#### Momento en soldaduras

Máximo momento positivo permitido por las soldaduras = 204,3 kNm  $\geq$  Momento aplicado (MSd) = 0 kNm

La combinación crítica es: - ELU CF 63 (1) -

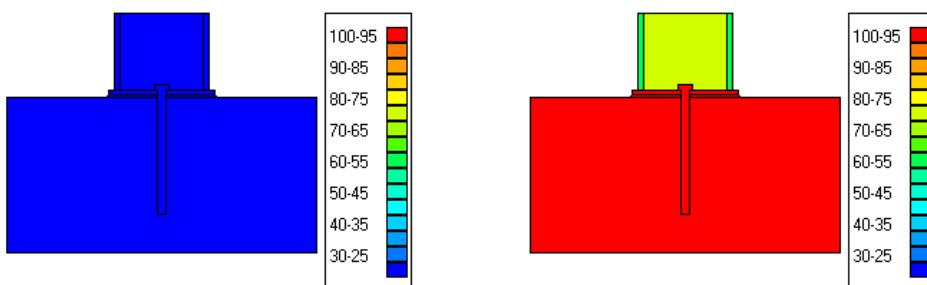
Máximo momento negativo permitido por las soldaduras = -210,4 kNm  $\leq$  Momento aplicado (MSd) = 0 kNm

La combinación crítica es: - M- -



### Gráfico con el ratio de utilización para todas las combinaciones

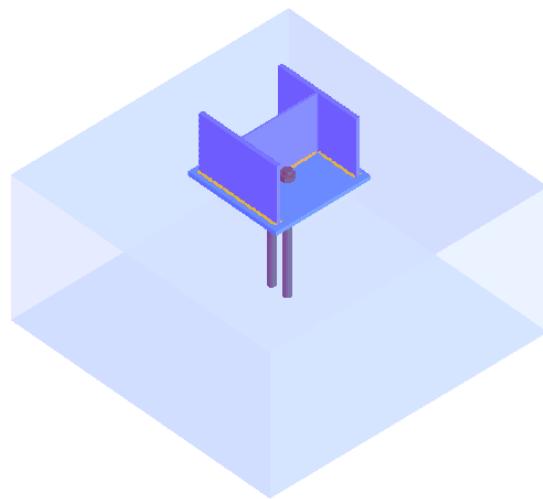
o de utilización considerando el momento y el esfuerzo axial apliGráfico de utilización para el máximo momento y esfuerzo axial resistente [elemento más débil]



### Cortante

Cortante máximo ( $VR_d$ ) = 90,8 kN  $\geq$  Cortante aplicado ( $VS_d$ ) = 78,8 kN

La combinación crítica es: - ELU CF 68 (1) -



## G.2. Càlcul encastrement biga – columna

### Conexión derecha

#### Momento

Máximo momento positivo ( $MR_d+$ ) = 805,7 kNm  $\geq$  Momento aplicado ( $MS_d$ ) = 499,1 kNm

La combinación crítica es: - ELU CF 12 (10,21) -

Máximo momento negativo ( $MR_d-$ ) = -418,2 kNm  $\leq$  Momento aplicado ( $MS_d$ ) = -367,3 kNm



La combinación crítica es: - ELU CF 68 (10,21) -

Máximo momento positivo permitido por las soldaduras = 672,1 kNm ≥ Momento aplicado (MSd) = 499,1 kNm

La combinación crítica es: - ELU CF 12 (10,21) -

Máximo momento negativo permitido por las soldaduras = -670,9 kNm ≤ Momento aplicado (MSd) = -367,3 kNm

La combinación crítica es: - ELU CF 68 (10,21) -

#### Gráfico con el ratio de utilización para todas las combinaciones



#### Esfuerzo normal

Máxima tracción en la viga (TRd) = 1368,2 kN ≥ Tracción aplicada (TSd) = 61,4 kN

La combinación crítica es: - ELU CF 70 (10,21) -

Máxima compresión en la viga (CRd) = 1483,1 kN ≥ Compresión aplicada (CSd) = 53,8 kN

La combinación crítica es: - ELU CF 49 (10,21) -

#### Momento con esfuerzo normal

Nombre de la combinación	MSd	MRd	NSd	NRd	MSd/MRd + NSd/NRd	< 1
ELU CF 9 (10,21)	222,39	672,12	49,59	1483,14	0,36	V
ELU CF 12 (10,21)	499,13	672,12	49,51	1483,14	0,78	V
ELU CF 24 (10,21)	489,49	672,12	48,25	1483,14	0,76	V
ELU CF 36 (10,21)	464,39	672,12	45,31	1483,14	0,72	V
ELU CF 48 (10,21)	454,75	672,12	44,05	1483,14	0,71	V



ELU CF 49 (10,21)	442,65	672,12	53,80	1483,14	0,69	V
ELU CF 50 (10,21)	433,01	672,12	52,54	1483,14	0,68	V
ELU CF 51 (10,21)	407,91	672,12	49,61	1483,14	0,64	V
ELU CF 55 (10,21)	-130,22	418,20	-55,90	1368,20	0,35	V
ELU CF 58 (10,21)	-332,60	418,20	-2,97	1368,20	0,80	V
ELU CF 60 (10,21)	-139,87	418,20	-57,16	1368,20	0,38	V
ELU CF 63 (10,21)	-357,70	418,20	-5,90	1368,20	0,86	V
ELU CF 65 (10,21)	-164,96	418,20	-60,09	1368,20	0,44	V
ELU CF 68 (10,21)	-367,34	418,20	-7,16	1368,20	0,88	V
ELU CF 70 (10,21)	-174,60	418,20	-61,35	1368,20	0,46	V

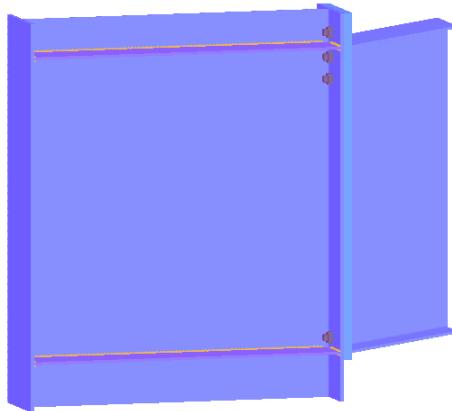
### Cortante

Cortante máximo ( $VR_d$ ) = 1174,5 kN  $\geq$  Cortante aplicado ( $VS_d$ ) = 130,8 kN

La combinación crítica es: - ELU CF 49 (10,21) -

Máximo cortante permitido en alma de pilar = 942 kN  $\geq$  Cortante aplicado a alma de pilar = 429,6 kN

La combinación crítica es: - ELU CF 12 (10,21) -



### **G.3. Càlcul encastament biga - biga**

#### Conexión izquierda

#### Momento

Máximo momento positivo ( $MR_{d+}$ ) = 398 kNm  $\geq$  Momento aplicado ( $MS_d$ ) = 181,2 kNm

La combinación crítica es: - ELU CF 49 (40,41) -



Máximo momento negativo ( $MR_d$ ) = -398 kNm ≤ Momento aplicado ( $MS_d$ ) = -66,5 kNm

La combinación crítica es: - ELU CF 71 (40,41) -

Máximo momento positivo permitido por las soldaduras = 360 kNm ≥ Momento aplicado ( $MS_d$ ) = 181,2 kNm

La combinación crítica es: - ELU CF 49 (40,41) -

Máximo momento negativo permitido por las soldaduras = -358,6 kNm ≤ Momento aplicado ( $MS_d$ ) = -66,5 kNm

La combinación crítica es: - ELU CF 71 (40,41) -

#### Gráfico con el ratio de utilización para todas las combinaciones

Gráfico de utilización considerando los momentos aplicados

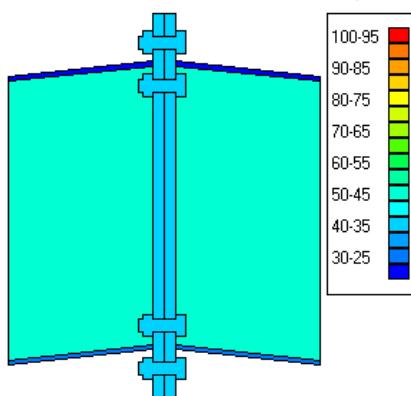
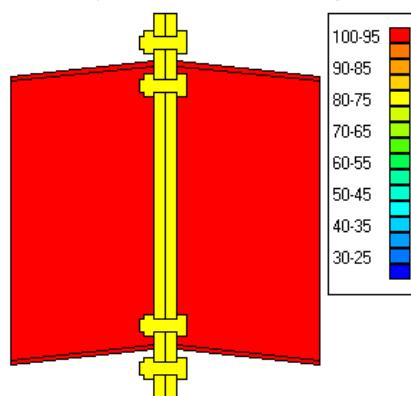


Gráfico de utilización para el máximo momento resistente [elemento más débil]



#### Esfuerzo normal

Máxima tracción en la viga ( $TR_d$ ) = 1891,9 kN ≥ Tracción aplicada ( $TS_d$ ) = 62,2 kN

La combinación crítica es: - ELU CF 70 (40,41) -

Máxima compresión en la viga ( $CR_d$ ) = 1432,2 kN ≥ Compresión aplicada ( $CS_d$ ) = 57,8 kN

La combinación crítica es: - ELU CF 49 (40,41) -

#### Momento con esfuerzo normal

Nombre de la combinación	$MS_d$	$MR_d$	$NS_d$	$NR_d$	$MS_d/MR_d + NS_d/NR_d$	< 1
ELU CF 3 (40,41)	62,73	359,96	23,29	1432,22	0,19	V
ELU CF 7 (40,41)	62,70	359,96	40,13	1432,22	0,20	V
ELU CF 9 (40,41)	143,55	359,96	47,66	1432,22	0,43	V
ELU CF 12 (40,41)	143,54	359,96	57,77	1432,22	0,44	V
ELU CF 15 (40,41)	58,92	359,96	21,94	1432,22	0,18	V



ELU CF 19 (40,41)	58,90	359,96	38,78	1432,22	0,19	V
ELU CF 21 (40,41)	139,75	359,96	46,31	1432,22	0,42	V
ELU CF 24 (40,41)	139,73	359,96	56,42	1432,22	0,43	V
ELU CF 27 (40,41)	48,46	359,96	18,78	1432,22	0,15	V
ELU CF 31 (40,41)	48,43	359,96	35,63	1432,22	0,16	V
ELU CF 39 (40,41)	44,66	359,96	17,43	1432,22	0,14	V
ELU CF 43 (40,41)	44,63	359,96	34,28	1432,22	0,15	V
ELU CF 49 (40,41)	181,18	359,96	57,81	1432,22	0,54	V
ELU CF 50 (40,41)	177,38	359,96	56,46	1432,22	0,53	V
ELU CF 51 (40,41)	166,91	359,96	53,30	1432,22	0,50	V
ELU CF 52 (40,41)	163,11	359,96	51,95	1432,22	0,49	V
ELU CF 54 (40,41)	6,98	359,96	5,68	1432,22	0,02	V
ELU CF 55 (40,41)	-25,25	352,17	-56,35	1891,87	0,10	V
ELU CF 57 (40,41)	6,96	359,96	22,52	1432,22	0,04	V
ELU CF 59 (40,41)	3,18	359,96	4,33	1432,22	0,01	V
ELU CF 60 (40,41)	-29,05	351,98	-57,70	1891,87	0,11	V
ELU CF 62 (40,41)	3,15	359,96	21,17	1432,22	0,02	V
ELU CF 63 (40,41)	-62,38	358,78	-12,65	1891,87	0,18	V
ELU CF 64 (40,41)	-7,28	359,96	1,18	1432,22	0,02	V
ELU CF 65 (40,41)	-39,52	351,54	-60,86	1891,87	0,14	V
ELU CF 66 (40,41)	-62,73	358,80	-4,24	1891,87	0,18	V
ELU CF 67 (40,41)	-7,31	359,96	18,02	1432,22	0,03	V
ELU CF 68 (40,41)	-66,19	358,59	-14,00	1891,87	0,19	V
ELU CF 69 (40,41)	-11,09	359,96	-0,17	1891,87	0,03	V
ELU CF 70 (40,41)	-43,32	351,36	-62,21	1891,87	0,16	V
ELU CF 71 (40,41)	-66,53	358,61	-5,59	1891,87	0,19	V
ELU CF 72 (40,41)	-11,11	359,96	16,67	1432,22	0,04	V

### Cortante

Cortante máximo (VRd) = 1069 kN ≥ Cortante aplicado (VSd) = 46,7 kN

La combinación crítica es: - ELU CF 3 (40,41) -

### Conexión derecha

#### Momento

Máximo momento positivo (MRd+) = 398 kNm ≥ Momento aplicado (MSd) = 181,2 kNm

La combinación crítica es: - ELU CF 49 (40,41) -



Máximo momento negativo ( $MR_d$ ) = -398 kNm ≤ Momento aplicado ( $MS_d$ ) = -66,5 kNm

La combinación crítica es: - ELU CF 71 (40,41) -

Máximo momento positivo permitido por las soldaduras = 360 kNm ≥ Momento aplicado ( $MS_d$ ) = 181,2 kNm

La combinación crítica es: - ELU CF 49 (40,41) -

Máximo momento negativo permitido por las soldaduras = -358,6 kNm ≤ Momento aplicado ( $MS_d$ ) = -66,5 kNm

La combinación crítica es: - ELU CF 71 (40,41) -

#### Esfuerzo normal

Máxima tracción en la viga ( $TR_d$ ) = 1891,9 kN ≥ Tracción aplicada ( $TS_d$ ) = 62,2 kN

La combinación crítica es: - ELU CF 70 (40,41) -

Máxima compresión en la viga ( $CR_d$ ) = 1432,2 kN ≥ Compresión aplicada ( $CS_d$ ) = 57,9 kN

La combinación crítica es: - ELU CF 49 (40,41) -

#### Momento con esfuerzo normal

Nombre de la combinación	$MS_d$	$MR_d$	$NS_d$	$NR_d$	$MS_d/MR_d + NS_d/NR_d$	< 1
ELU CF 3 (40,41)	62,73	359,96	40,17	1432,22	0,20	V
ELU CF 7 (40,41)	62,70	359,96	23,37	1432,22	0,19	V
ELU CF 9 (40,41)	143,55	359,96	57,86	1432,22	0,44	V
ELU CF 12 (40,41)	143,54	359,96	47,78	1432,22	0,43	V
ELU CF 15 (40,41)	58,92	359,96	38,82	1432,22	0,19	V
ELU CF 19 (40,41)	58,90	359,96	22,01	1432,22	0,18	V
ELU CF 21 (40,41)	139,75	359,96	56,51	1432,22	0,43	V
ELU CF 24 (40,41)	139,73	359,96	46,42	1432,22	0,42	V
ELU CF 27 (40,41)	48,46	359,96	35,66	1432,22	0,16	V
ELU CF 31 (40,41)	48,43	359,96	18,85	1432,22	0,15	V
ELU CF 39 (40,41)	44,66	359,96	34,30	1432,22	0,15	V
ELU CF 43 (40,41)	44,63	359,96	17,50	1432,22	0,14	V
ELU CF 49 (40,41)	181,18	359,96	57,93	1432,22	0,54	V
ELU CF 50 (40,41)	177,38	359,96	56,58	1432,22	0,53	V
ELU CF 51 (40,41)	166,91	359,96	53,42	1432,22	0,50	V
ELU CF 52 (40,41)	163,11	359,96	52,06	1432,22	0,49	V

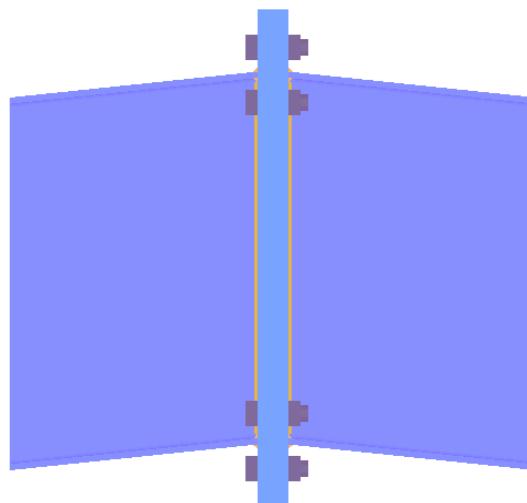


ELU CF 54 (40,41)	6,98	359,96	22,53	1432,22	0,04	V
ELU CF 55 (40,41)	-25,25	352,17	-56,37	1891,87	0,10	V
ELU CF 57 (40,41)	6,96	359,96	5,72	1432,22	0,02	V
ELU CF 59 (40,41)	3,18	359,96	21,17	1432,22	0,02	V
ELU CF 60 (40,41)	-29,05	351,98	-57,72	1891,87	0,11	V
ELU CF 62 (40,41)	3,15	359,96	4,37	1432,22	0,01	V
ELU CF 63 (40,41)	-62,38	358,78	-4,43	1891,87	0,18	V
ELU CF 64 (40,41)	-7,28	359,96	18,02	1432,22	0,03	V
ELU CF 65 (40,41)	-39,52	351,54	-60,88	1891,87	0,14	V
ELU CF 66 (40,41)	-62,73	358,80	-12,51	1891,87	0,18	V
ELU CF 67 (40,41)	-7,31	359,96	1,21	1432,22	0,02	V
ELU CF 68 (40,41)	-66,19	358,59	-5,79	1891,87	0,19	V
ELU CF 69 (40,41)	-11,09	359,96	16,66	1432,22	0,04	V
ELU CF 70 (40,41)	-43,32	351,36	-62,24	1891,87	0,16	V
ELU CF 71 (40,41)	-66,53	358,61	-13,87	1891,87	0,19	V
ELU CF 72 (40,41)	-11,11	359,96	-0,15	1891,87	0,03	V

### Cortante

Cortante máximo (VRd) = 1069 kN ≥ Cortante aplicado (VSd) = 46,4 kN

La combinación crítica es: - ELU CF 7 (40,41) -



## ANNEX H: Estudi d'impacte ambiental

El present Projecte Final de Carrera està dedicat a l'anàlisi d'un pòrtic d'acer de secció variable. El impacte ambiental d'aquest tipus d'element estructural és mínim, doncs l'estructura metàl·lica, o el que és el mateix, l'acer, és un material, del qual s'ha més que demostrat la seva gran capacitat de reciclatge. A continuació es llisten algunes de les característiques que converteixen l'estructura metàl·lica en l'estructura sostenible per excel·lència:

- L'acer té una alta relació resistència - pes, fet que el converteix en un material molt eficient.
- L'acer és 100% recicitable, motiu pel qual minimitza l'esgotament de recursos naturals i el impacte ambiental.
- L'acer té una llarga vida útil que permet amortitzar el impacte ambiental degut a la seva fase de producció.
- L'acer emprat a la construcció es produeix en processos industrials amb consums de recursos, aigua i energia menors que altres materials estructurals.
- Les estructures d'acer poden ser fàcilment adaptades a les noves exigències funcionals durant el cicle de vida de l'edifici.
- L'estructura d'acer, amb poc o ningun manteniment, posseeix una extraordinària durabilitat.
- L'acer és 100% recicitable i pot ser reciclat infinitament sense pèrdua de qualitat.
- A la majoria dels sectors, les taxes de reciclatge d'acer estan entre el 80 i el 100%.
- Els residus durant la construcció es redueixen al mínim i la majoria d'ells són reciclables.
- Creació de nous tipus d'acer a partir d'acer reciclat redueix les emissions de CO<sub>2</sub> (l'any 2006, es va evitar la emissió de al voltant de 894 milions de tones de CO<sub>2</sub>).

Cal destacar també que, tal i com ja s'ha comentat a la memòria, el fet que les bigues d'acer d'inèrcia variable siguin elements estructurals amb seccions longitudinals òptimes, converteix a aquest tipus de secció en un element estructural encara més sostenible.





## ANNEX I: Pressupost

En aquest capítol s'analitzen les despeses del present projecte. El cost total del projecte ve donat per el temps emprat a la realització del present estudi.

El treball de realització del projecte implica per a cada estudiant una dedicació orientativa de  $2,25 \times NC \times 10$  hores, en què NC és el nombre de crèdits que atorga al PFC el pla d'estudis de la titulació. Tenint en compte el nombre de crèdits total assignat al projecte:

$$NC = 24 \text{ crèdits} \Rightarrow 2,25 \cdot 24 \cdot 10 = 540 \text{ hores}$$

El cost final del projecte depèn de les hores realitzades per les diferents persones assignades al projecte. A continuació es citen els diferents tipus de professionals necessaris per a la seva consecució, una estimació de les hores ha dedicar, i el sou estimat per a cada tipus de treballador. Així doncs:

Rol	Hores	Cost/Hora	Import
Enginyer junior	400 h	30 €/h	12.000 €
Becari	140 h	10 €/h	1.400 €
<b>13.400 EUROS</b>			





## ANNEX J: Plànol pòrtic





PFC Disseny i càlcul estructural d'una estructura metàl·lica destinada a poliesportiu

ESCALA:  
1:100

TÍTOL PLÀNDOL:  
PÒRTIC

Alex Montaner Ramoneda

