WP1 – DATABASE OF THE NUMERICAL PARAMETRIC STUDIES

This database was set up to collect all the results of the parametric analyses in such a way that all information was stored in a way that is identical for all the partners. This will facilitate a probable use of it in the future.

Each file is an ASCII text file in order to be readable by any software, for instance a spreadsheet program or home-made software in any language.

The information from each numerical test is stored in one line of the file. One line represents one simulation. On this line all parameters representative of the chosen parameters are separated by one (or more) blank character. Use of the TAB key to generate blank was not admitted.

Each tested beam or column has two ends called “min” and “max”. In case of prismatic members, the size of the section is the same at both ends. In tapered members, the depth of the section is different at both ends and the smallest value at end “min”.

In all sections, the temperature is assumed to be uniform in the section.

Each section has a strong axis (called y-y axis) and a weak axis (z-z). A moment around the strong axis produces a variation of the bending stresses on the depth of the section in the direction of the web whereas there is no variation on the width of the section in the direction of flanges. Flexural buckling is in the strong axis in the direction parallel to the web. Flexural buckling in the weak axis and lateral torsional buckling are in the direction parallel to the flanges.

First line:

This line contains some words which are used to indicate the meaning of the numbers contained in the next lines.

Subsequent lines:

Each line contains the parameters describing the simulation. These parameters are written horizontally in the line but for a convenient description in this report they are arranged vertically:

Name [-]:	A chain of characters that gives the name of the section. It could be, for example IPE200 for a hot-rolled cross-section, or 1000x12+200x20 for a welded section. It has been put here for convenience, in order to help users to quickly identify a section type in the database. No blank character is allowed in the name
Type of section:	Only the characters H or W are allowed here. H is for hot-rolled sections and W is for welded sections
$h_{w,min}$ [mm]:	Height of the web at the minimum end. This height is measured between the flanges. For a hot rolled section, this value should be equal to the depth of the section minus twice the thickness of the flange
$h_{w,max}$ [mm]:	Height of the web at the maximum end. Equal to $h_{w,min}$ for prismatic members, different for tapered ones.
t_w [mm]:	Thickness of the web
b [mm]:	Width of the section (width of the flanges)
t_f [mm]:	Thickness of the flanges
r [mm]:	Radius of the root fillet between the flanges and the web. A value of 0 must be used in case of hot-rolled section if the root fillet has not been taken into account in the simulation
L [mm]:	Length of the member between the ends. If this element is a column, this is also the distance between the hinges.
$L_{b,strong}$/L [-]:	Ratio between the buckling length around the strong axis and the length of the member. Normally, this ratio is equal to 1. Possible exceptions might be: if a cantilever column of beam would be simulated, the ratio would be 2, a ratio of 0.5 for a beam would mean a beam on 3 supports, a ratio of 0.0 for a column would mean that it was really desired in the simulation to have buckling around the weak axis without any interaction with strong axis.
$L_{b,weak}$/L [-]:	Ratio between the buckling length around the weak axis and the length of the member. A ratio of 0.0 for a column would mean that buckling has been forced to occur around the strong axis. A ratio of 0.0 for a beam would mean that lateral torsional buckling has been prevented.
k_w:	1 if warping is free at both end, 0.5 if warping is prevented at both ends
f_{yw} [MPa = N/mm²]:	Yield strength of the web

f_{yf} [MPa = N/mm²]:	Yield strength of the flanges
Residual stress indicator:	0 if no residual stresses have been considered in the simulation, 1 if residual stress have been considered
T [°C]:	Steel temperature
N [kN]:	Axial load (can be equal to 0 for a beam)
$M_{min,strong}$ [kN.m]:	Bending moment around the strong axis at the minimum section
$M_{max,strong}$ [kN.m]:	Bending moment around the strong axis at the maximum section
$M_{min,weak}$ [kN.m]:	Bending moment around the weak axis at the minimum section
$M_{max,weak}$ [kN.m]:	Bending moment around the weak axis at the maximum section
Q [kN/m]:	Uniformly distributed load on the strong axis
Z_G [mm]:	Distance between the center of the section and the application of Q (positive on the compressive flange)
Software code:	ABAQUS, ANSYS or SAFIR
Version of the software:	e.g. “2013.b.2”, “v14” or “6.10”
Research centre	Ulg, CTICM, UniAv; TECNALIA or CTU
Degree [-]:	Degree of the shell finite element, =1 for linear elements (4 nodes), =2 for second order (parabolic) elements (8 nodes)
NG [-]:	Number of integration points on the thickness of the shell finite elements
NL [-]:	Number of finite elements on the length of the member
N_h [-]:	Number of finite elements on the height of the web of the member
N_b [-]:	Number of finite elements on the width of the flange
Date of the simulation:	YYYYMMDD