SECURE WITH STEEL
Esch, November 5, 2015

New features in SAFIR® 2016

Jean-Marc Franssen
Thomas Gernay
1) 3D brick finite elements
2) Local buckling in steel beam finite elements
3) Negative terms on the main diagonal
4) New laws for reinforcing steel
5) New PRNSTRAIN command
6) Spring finite element
7) Orientation of the re-bars in shell finite elements
8) LOCAFI fires
9) New DIAMOND
1) 3D brick finite elements
Currently: brick linear elements with 6 or 8 nodes

Future: tetrahedra
Thermal analysis

Materials: Concrete – Steel – Wood
– Gypsum – Aluminum – User

3D thermal calculation
Steel-concrete joint
31 502 nodes
25 411 solid elements
Mechanical analysis

Capabilities
- Materials: Concrete and steel
- Fully multiaxial stress states
- Ambient and elevated temperature

Assumptions and limitations
- Static calculations
- Large displacements not taken into account
- No spalling in concrete
- Smeared crack model for concrete
Mechanical analysis

**Concrete model**
- Plastic-damage model
- Drucker Prager – Rankine
- Eurocode
- Crack closure effect
- Explicit transient creep

**Steel model**
- Plastic model
- Von Mises
- Eurocode


Mechanical analysis

Example: shear punching in flat slab
Ambient temperature (EPFL 2006)
Mechanical analysis

Examples of possible applications:

Concrete:
- Shear punching
- Prestressed hollow core slabs
- Concrete masses (e.g. in nuclear applications)

Steel:
- Flame straightening of heavy sections

Concrete & Steel
- Joints

Main limitation
- No contact element
2) Local buckling in beam finite elements
Steel sections made of slender plates may deform locally (local buckling)

=> They cannot be modelled with beam finite elements
Why can’t they?

Diamond 2012.a.0 for SAFIR

FILE: New
NODES: 4251
BEAMS: 0
TRUSSES: 0
SHELLS: 4136
SOILS: 0
SOLIDS: 0

SHELLS PLOT
IMPOSED DOF PLOT
POINT LOADS PLOT
Structure Not Displaced selected

Flanges: 250 x 12
Web: 500 x 4
Membrane forces
  In the web

Perpendicular displacement
  In the web
Membrane forces
In the web

Perpendicular displacement
In the web
Effective width: $b_{eff}$

$$b_{eff} = f(\sigma_{max}) \quad \sigma_{max} = f(b_{eff})$$

If used in F.E, it must be introduced \textit{at the element level}
Effective stress: $\sigma_{\text{eff}}$

$$\sigma_{\text{eff}} = f(\varepsilon) \bigg|_{\text{support conditions, slenderness, temperature}}$$

If used in F.E, it can be introduced at the material level
New proposal: Effective stress

Effective width

Effective stress

\[ f_y \]

\[ \sigma_{\text{actual}} \]

\[ \sigma^* \]

\[ b_{\text{eff}}/2 \]

\[ b_{\text{eff}}/2 \]

\[ a \]

\[ b \]
How do we determine $\sigma_{\text{eff}} = f(\varepsilon)$?

By a series of numerical push-over tests made for different:
- Support conditions
- Plate slendernes
- Temperatures
Various support conditions

Flange
Simply supported on 3 sides

Web
Simply supported on 4 sides
The stress-strain relationship is the one of Eurocode 3.

For each combination of Temperature – Slenderness - support condition, the values of $f_{p,\text{eff}}$ and $f_{y,\text{eff}}$ in compression are modified in the material law ($E$ and $\varepsilon_u$ remain unchanged, as well as all parameters in tension).
The user gives in the input file $f_y$, b/t and the support condition for the web and the flange.

In the members, every point of integration follows its own $\sigma_{eff}-\varepsilon$ relationship.
Stress strain relationship in tension => compression
Simulations with shell finite elements
Material model used in beam finite elements
Stress strain relationship in compression => tension
VALIDATIONS AGAINST SHELL F.E.

Simply supported beam with UDL
L = 6 meters
Section: H = 300x4, B = 150x4

T = 20°C
VALIDATIONS AGAINST SHELL F.E.

Simply supported beam with UDL
L = 6 meters
Section: H = 300x4, B = 150x4

$T = 500^\circ C$
VALIDATIONS AGAINST EXPERIMENTAL TESTS

Simply supported column N = 122 kN \( \text{ecc}_{\text{weak}} = 5 \text{ mm} \)

H = 2.7 meters

Section: H = 450x4, B = 150x5
VALIDATIONS AGAINST EXPERIMENTAL TESTS

Simply supported column  \( N = 204 \text{ kN} \)  \( \text{ecc}_{\text{weak}} = 4-13 \text{ mm} \)

\( H = 2.7 \text{ meters} \)

Section: \( H = 450 \times 4, \ B = 150 \times 5 \)
VALIDATIONS AGAINST EXPERIMENTAL TESTS

Simply supported column

N = 231 kN  \( \text{ecc}_{\text{strong}} = 71 \, \text{mm} \)

H = 2.7 meters

Section: H = 360x4, B = 150x5
CONCLUSIONS

✓ The proposed technique is extremely CPU efficient compared to shell F.E.

✓ It reasonably well accounts for local buckling arising from longitudinal stresses

✓ Sometimes too severe
3) Negative terms on the main diagonal

Since version 2014.a.3, a time step is considered as converged even if some negative terms are found on the main diagonal of the stiffness matrix. With the previous versions, such an occurrence would lead to either a return to the previous time step or to a stop of the run if the minimum value of the time step was reached, with some runs stopping with no obvious physical reason.
4) New laws for reinforcing steel

☑ Hot rolled or cold formed bars can be chosen (see Table 3.2a of EN 1992-1-2)

☑ Class A, B or C can be chosen (see Figure 3.3 of EN 1992-1-2)

In previous versions, Cold worked, class B or C was used
5) New PRNSTRAIN command

The command

PRNSTRAIN  0.05

will print a message in the output file when the stress-related strain in a bar of a shell finite element exceeds 5%
6) Spring finite element

This element:
- is attached to one single node (the other virtual extremity of the element is the foundation);
- is characterized by a direction (in 2D or in 3D);
- has a particular « Displacement–Force » behaviour (both in the defined direction)
Input parameters are:

NSPR: Number of the element.
NNODE: Node where this element is attached.
CX: Cosinus of the angle between X axis and this element.
CY: Cosinus of the angle between Y axis and this element.
CZ: Cosinus of the angle between Z axis and this element.
Fₜₜsup: Superior limit of the load.
Fₜₜinf: Inferior limit of the load.
K: Stiffness of the element for elastic loading or unloading.
A: Area of influence (all forces are multiplied by A).
Dᵢ: Displacement in the configuration of reference (time t = 0).
Fᵢ: Force in the configuration of reference (time t = 0).
Application example
Cut and cover tunnel (Guide d’application from « CETU »)
2D model with beam F.E. and spring F.E.

- No restrain
- Trapezoidal loads on the walls replaced by spring F.E.
Deformed shape and soil pressures at $t = 60$ s

Note: Diamond can plot the loads $F$ or the pressure $F/A$ in the spring elements
Deformed shape and soil pressures at $t = 3\,720\,s$
(just before plastic hinges appear)
Deformed shape and soil pressures at $t = 3800$ s
(just after plastic hinges appear)
Deformed shape and soil pressures at $t = 5.135$ s (just before failure)
Deformed shape and soil pressures after failure
3D model
7) Orientation of the re-bars in shell finite elements

For each bar layer, there are two methods to give the orientation of the bars in the plane of the element.
**Method 1:** with respect to the local system of coordinates of each element.

1 card.
- "ANGLE"
- **angle** Angle in degrees between the local x axis and the layer of rebars, see Figure in which the bars of the layer are represented by dotted lines. This angle cannot be smaller than $-180^\circ$.
This method is not appropriate in unstructured meshes.
Method 2: with respect to the global system of coordinates of the structure.

1 card.
- "NORMAL"
- \( N_1 \)
- \( N_2 \)
- \( N_3 \)

\( <N_1, N_2, N_3> \) is a vector in the global system of coordinates of the structure. The norm of the vector does not have to be 1.

This vector is used to define the position of the bar layers in the shell elements with respect to the global system of coordinates according to the following technique, see Figure.

The bars have the orientation of the line which is the intersection between the shell element and a plane that is perpendicular to the normal.

If the norm of the vector is 0, then the orientation of this bar layer is perpendicular, in each element, to the previous bar layer (not possible for bar layer 1).
Figure 3: bars in a plate. Use method 2 if the mesh is unstructured.

Figure 4: bars in a hyperbolic paraboloid. Method 2.

Figure 5: bars in a dome. Method 2.
ACKNOWLEDGEMENT
DEVELOPMENT OF THE SECOND METHOD HAS BEEN SUPPORTED BY « HOLMES FIRE »
8) LOCAFI fires

See presentation by François Hanus.
8) New DIAMOND

Completely translated in C++

Many subroutines modified

New features (limited)

More easily adaptable (format of output numbers)

Can read and treat bigger files

Opens faster
Output file in XML format (you can use or develop your own viewer)

<N>
<P1> 0.15000E+00</P1>
<P2> 0.15000E+00</P2>
</N>
</NODES>
<FIX>
</FIX>
<SOLIDS format="I6">
<S>
<N> 1</N>
<N> 2</N>
<N> 14</N>
<N> 13</N>
<MS> 1</MS>
</S>
</S>
Additional new features?

1) Make your wish list.

2) Be patient because

3) debugging must come first!
Distribution policy

✓ Free for licences bought in 2015

✓ Demo versions: free

✓ Academic licences: 200 Euros (-20% for SWS members)

✓ Commercial licences: 1 000 Euros (-20% for SWS members)
Thank you