

PLE-micro-CAD version 3.07

The new version 3.07.08 contains many adaptations and extensions of functionality resulting in new applications coming available. The new developed modules are:

- W-module , containing functions to determine wave and current loadings on the pipeline (in design function 4.1)
- T1-module , performing loading and stress checking in compliance with the Belgian Law (and ASME B31.8), for gas pipelines.
- T8-module , performing loading and stress checking in compliance with ASME B31.8 (gas pipelines)
- O-module , containing the option to specify table supports, i.e. the support is working only in case of vertical movement downward, resulting in a vertical upward reaction and horizontal friction reactions. Movement upward can take place freely,
- V-module , containing the option to specify anisotropic material properties (for instance glass fibre reinforced plastics)
- Q-module , containing facilities taking into account:
 - the influence of the bend angle on the bend behaviour
 - the non-linear nature of the (bend) ovalisation
- M-module , performing calculations based on non-linear elastoplastic material behaviour. Several stress-strain relations are available.

With regard to version 3.06 the following modifications have been made.

See also USER MANUALS Vol. 1 and 2, being made available in a complete new version.

SN008.0 General functions

- When copying all input tables from another database (option I on Main Menu Copy) the graphic definition screens (multigraphs) are copied too.
- The switch for the pipeline Standard to follow has been moved to a place behind function 1 and contains the following options –if the related modules are present:-
 - GENERAL
 - NEN3650
 - ASME B31.8
 - BELGIAN LAWThe switch is locked after processing Design function 3.1, because in this function code-dependent tests are performed, in this case checks on minimum wall thickness.

SN008.1 Table functions

- Apart from the MAXIMA and MINIMA functions for result tables an EXTREMA function has been created producing the extreme values of the table columns without regard to the sign.
- In all result tables with a first NODE or ELEM column a selection of element groups can be made, if these have been specified in the new table GROUPS (see SN008.2). The numerical table functions like MAXIMA, etc. are applicable then for the selected element group only. Concerning the graphical functions this is valid for S-GRAPH only.

SN008.2 Function 2: Compose pipeline configuration

- Specifying branches occurs in this function now: table CONNECT has been moved from function 3.3 to function 2.
- A new facility is the specification of a connection as a Tee in compliance with ASME B31.8. In table CONNECT reference can be made to the new tables TEECONF containing the configuration data of the Tee(s) and TEESPEC in function 3.1 containing the type, material and cross-sectional dimensions of the Tee(s). The configuration data are collected in result table T-CDATA.
- In the new table GROUPS pipeline elements may be divided in groups (with a name), for instance straight pipe elements and bend elements. Maxima and minima and single graphs can be obtained for each of the specified element groups. An element may be assigned to 4 different groups as shown in result table GRELEM. Result table AVGRPS contains some group statistics.

SN008.3 Function 3.1: Compose pipe data model

- The name of table MATS has been changed to ISTROP (isotropic material) and 2 columns have been added with respect to possible plastic material behaviour (only in M-module).
- The properties of anisotropic material are specified in the new table ANTROP being active only if the V-module is present.
- Table WALL has been extended with the option to specify an absolute manufacturing tolerance for the wall thickness.
- In case of NEN3650 calculations the wall thicknesses entered are checked against the minimum required values as given in table 5 of NEN3650.
- In the new table TEESPEC material and cross-sectional data of Tees are specified. All data provided are collected in result table T-SDATA.
- Table DEADW has been moved from function 4.2 to this function due to the fact that the deadweight data may be required to process function 4.1 (generation of wave/ current forces). The element values are contained in result table PIPEDIM, but remain also in table LOADATA in function 4.2.

SN008.4 Function 3.3: Compose model boundary conditions

- Table CONNECT has been moved to function 2.
- The name of table ELSPRL has been changed to SUPLOC (Support Location).
- Non-linear vertical spring supports with friction in the horizontal plane can be specified in the new table TFSPRS (Table FricSion Spring Specification) in module O.

SN008.5 Function 4.1: Generate wave and current loads

- Wave and/ or current loads on the pipeline are generated by means of the Morison equation requiring the hydrodynamic drag, inertia and lift coefficients C_d , D_m , C_l . Wave and current characteristics are required, various wave theories are available. The influence of nearby constructions, marine growth, wave slamming and pipe vibrations due to vortex shedding can be taken into account.

SN008.6 Function 4.2: Compose pipeline loadings

- Table DEADW has been moved to function 3.1, but the generated values remain in result table LOADATA.

SN008.7 Function 5: Determine pipeline behaviour

- For as Tee specified connections stress intensification factors are calculated in compliance with appendix E of ASME B31.8 and reported in the new result table TEEFAC. These factors are accounted for automatically when calculating the stresses in design functions 6.2 - 6.5.
- The new table DISCOOR contains the global coordinates of the pipeline nodes in the displaced position, so the sum of the values in tables NODES and DISPLAC.
- The new table W-LOAD reports the wave/current loads generated in function 4.1 and multiplied by the load factor from table LOCASE.
- Design function 5 has a (new) sub-screen which appears after clicking the ADVANCD function key. This sub-screen shows the following analysis options (Standard/Advanced):
 - phase : Initial/ Continue/ Hysteresis/ Predis (F-module)
 - bend angle: Infinite/ Limited (Q-module)
 - geometry : Linear/ Non-linear (NL-modules)
 - section : Non-ovl/ Oval (Q-module)
 - material : Linear/ Non-linear (M-module)
 See also module specifications.
- The new table DEFORM reports cross-sectional deformations if the options OVAL and/or MAT NON-LIN are active.

SN008.8 Function 6.1: Compose cross-section data model

- In the new table SOILSUP horizontal soil support pressure can be specified as a real soil pressure. This table is an alternative for or an addition to table LAMBDA. Table CROSDAT contains the soil support values per element.
- The 'ADDCROS' mechanism may be switched off by means of the new ADDCROS function key, resulting in an empty table ADDCROS.
- The way of redistribution has been changed and adapted to the new approach to non-vertical and non-horizontal directed soil reactions. See functions 6.2 - 6.5.
- The lay-out of tables W-REDIS and T-REDIS have been changed and made equal to the lay-out of the new table B-REDIS.
- In case of options OVAL and/or MAT NON-LIN (resulting in a locked REDIS ON key) the ovalisation of the cross-sections over the bend(s) is redistributed and the results are reported in table B-REDIS.
- **A warning is given if SOILNB, TOPLOAD, LAMBDA in and/ or SOILSUP has been specified at the location of SLACK elements (e.g. in a bore hole or jacketpipe); however the soil and traffic pressures are still applied at the elements in question. Especially the combination with the ADDCROSS option can result in large additional bending moments. In most cases this will be a wrong modelling of the actual situation.**

SN008.9 *Function 6.2 - 6.5: Perform cross-section calculations*

- The stress weighing factors specified in table SWEIGH are generated per element and reported in the new result table CSWEIGH.
- Up till now the soil reaction, making an arbitrary angle with the local (horizontal) y-axis of the cross-section, was resolved into a horizontal and a vertical component. The resulting deformations of both components were superposed. See chapter 6.7 of the Theoretical Manual and the various Codes using the method of Spangler. However, on closer investigation it turns out that the horizontal component behaves as a soil support pressure counteracting the ovalisation due to the vertical component.

In the new version 3.07 a new approach has been adopted. The soil reaction is not resolved anymore and ovalisation occurs in the direction of the reaction. Soil overburden and topload (traffic) remain acting in a vertical direction and possibly specified horizontal soil support pressures through tables LAMBDA and/or SOILSUP remain horizontal. The (combined) soil support pressures are reported in column SUP-TOTAL in table CSLOAD.

The supporting angle β is assumed as follows:

At a reaction angle $225^\circ \leq \phi_{LAT} \leq 315^\circ$: $\beta = \text{SUPANG}$

At a reaction angle $135^\circ < \phi_{LAT} < 225^\circ$ or

$315^\circ < \phi_{LAT} < 45^\circ$: $\beta = 120^\circ$

At a reaction angle $45^\circ \leq \phi_{LAT} \leq 135^\circ$: $\beta = 180^\circ$

This new approach causes different wallbending stresses (SFORA, SFIRA, etc.) in the cross-sections in question, with the largest differences for elements loaded by the soil under 45° with the horizontal and vertical direction. As a result, when in program version 3.07 an old design database is called, the question appears: CONVERT DESIGN TO LATEST VERSION (3.07)? and the message: Conversion will SET-BACK function 6. The same conversion and set-back is done when copying a complete old design into a new one under version 3.07 (main menu copy). For, the previously calculated stress results do not fit in the new program.

- Tables RDPLMAX and RDISPLC obtained a more clear content.
- The plastic stress components have been removed from the existing (elastic) stress result tables and inserted in stress tables on a special design function screen appearing in case of active MAT NON-LIN option.
- After processing design function 6.2/6.3/6.4/6.5 a RESTART key appears on the location of the RESET/STOP key. By means of this key elements may be added to the result tables. The RESTART key replaces the previous function of the SETBACK (6.1 and 6.2). The 6.1 and 6.2 SETBACK functions now have the same result as the setback of the other functions, namely emptying the result tables.
- The NEN3650 module(T0) contains 4 extra result tables, TSTRMAX, MSTRMAX, TSTRESS, MSTRESS in view of the check on alternating yielding.
- A definition of the TRESCA stress within PLE-micro-CAD has been attached (was lacking in the Theoretical Manual).