Data stored in this version can be opened in version 13.0.105 (Release2013.0) and higher.

Ticket number(s)	Bug Description	Solution
NWEB-8WB5JY	Export to etabs doesn't work. I checked the help and then searched the project for unsupported elements, but none found. If I missed anything, please tell.	some shapes or elements are not supported for export to Etabs so only supported one are exported/imported to etabs (13.0.70)
IBES-8ZXGBE	Wrong usage of LCS - 2D Parts Please have a look at the wall S1. In this wall the LCS of elements were rotated. They are dosplayed correctly. The 2D sorface load will be linked with the LCS of the wall. This was not changed. The X Axis of LCS of the 2D Part should be displayed on the screen from left to right. But it is wrong displayer (from right to left) The load is inputted correctly in the lce of 2D part which should go to right and also the deformations are correct. But nobody can check this, because the displayed direction of the LCS of the wall is not ident with the direction, which is used for the calculation.	fixed in 13.0.61
CSCT-92GDDK	2D wind - error during load generation Attached problems with latest patch 12.0.1049 for 3D-Windgenerator. We have created attached project (21.11(3)) in 12.0.183; generator runs. Load is not usable because we have a flat roof and strange load f.e. FF69 and FF73 in 3D-Windloadcase 1. If we run generator in 1049 we get attached message about points are not coplanar. If we export it (in 1049) into new .esa-file we get the message that panels are not closed. (Project and message attached, too). PS: In original-project is one double note via structure control.	3D wind problem fixed in WLE 37, 13.0.61 (problem with load generation on LP22 is in another DevTrack - MHEA-975BCX)
NWEB-8ZDJXR	No load from 3D windload generator on the roof.	fixed in WLE 37, 13.0.61

Ticket number(s)	Bug Description	Solution
JBES-956BZW	Issue PNL-calculation (2012.0.1094) See project in attachment -> Concrete with Bi-Linear stress strain diagram: the concrete is initially set on 'Bi-Linear' stress-strain diagram. If you calculate the project (non-linear), and regenerate the document, then you can see in the last table checks for how close you are to the ultimate strain and the average compressive strength (see also image 1) -> Concrete with Parabolic stress strain diagram: now change the non-linear stress-strain diagram to 'parabolic', and then calculate and regenerate the document and look at the same table again (this table comes from the preview from the single check for PNL stress/strain) Now for the second part it does not seem clear what the percentage means. It seems to be calculated from "sigma_parabolic / sigma_bi-linear" (for value for the same strain, but from the two different stress-strain diagrams) => So what is happening which makes us find the 125% ? And why is there no header for that column ?	The problem was solved and tested in version 13.0.80
NWEB-95KDXG	Crash during regeneration of document (not MANGE_STATE, not memory) Open project and calculate it Go to old document Start regeneration ---> Crash	Crashing is caused by the last item of the document (Punching check). The cash iself is fixed in the Deve_09 but it needs also further improvement in punching check --> created new independent bug.
HWRE-95PEFS	Absence model is only working in a linear calculation, not in a 2nd order calculation. Customer said, that this is not written in our manuals. Can you please add this remark?	help for 2013
NDES-95QCRR	In the NF EN 1992-1-1 NA, you can input the value fyk,up (see printscreen). But when changing this parameter in Scia Engineer, we don't see the influence in the calculation of the reinforcement in slabs. Can you explain where this value is taken into account	The problem was solved and tested in version 13.0.80
NWEB-95SDE4	Table with Load combinations is not fully translated	Bug was in CEP_LoadCombi::BuildHeaderImpl. Solved in R_2013
NWEB-95WGYC	Accidental crash at CDD Method: 1) Linear calculation 2) Design reinforcement ULS +SLS for all construction 3) Member design – design for combination msu and all construction. 4) CDD calculation 6 of 10 attempts = crash	According to crash report I fixed the problem but it is not sure that it is enough.

Ticket number(s)	Bug Description	Solution
NWEB-966KQB	when defining rib, and setting the collaborating width = "Default", then we take 20x the plate thickness (this value comes from the solver setup) BUT when you do Single Check in Concrete Design, then we only take 10x plate thickness into account (and not 20x)	The problem was solved and tested in version 13.0.80
JPOL-969BWF	Aluminium: Do not show 'Edit Initial Shape' for Formcode 3 How to deal with HAZ data on thinwalled tubes correctly? The user is having issues with attached project that contains 3 tubes in css library. However, no HAZ data inputted in the initial shape dialogue seem to have an effect on reduced shape or reduced css properties. What is the meaning of parameters Ybeg/Yend and Zbeg/Yend in initial shape setting? Is it important to have these nodes in the middle of the tube thickness or at the edge? What is the meaning of the red area in the graphics of HAZ data on tube css?	See also the screenshot in attachmenta) For each cross-section an initial shape is generated which divides the cross-section into parts. Each part has its own designation (I, UO, SO) which is used in the classification and later on in the effective section calculation.The Ybeg/Zbeg and Yend/Zend coordinates show the coordinates of the initial shape parts.b) When defining a longitudinal weld the red zone shows the location of the weld, see also the screenshot of the typical I-section in attachment.c) A CHS however is something special, this is a unique shape which is not actually divided in straight parts.Within Scia Engineer a single straight part is generated but this is not actually used. The unique shape of the CHS is used instead.As such the definition of the weld, which is defined on the single straight part, is not defined on the unique shape and thus has no influence. The definition of a longitudinal weld was never intended for a CHS, also the editing of the initial shape has no meaning for a CHS. Therefore, this will be modified in Scia Engineer, the 'Edit Initial Shape' option will not be visible anymore for CHS sections.This has been modified in DEVE 11 and will be merged to R 2013.0.87
NWEB-96ECPM	FE-Calculation 64 - The maximum allowable data field size has been exceeded Hello, could you please explain the meaning of the mentioned message and give indication about what can be done by the user to fix it ?, or is it a sign that some variable size is too strict and should be increased ?	capacity problem fixed in the solver problem in the modelization: nearly mechanism in the "rails" cross-section. Mesh refinement not necessary: change 1D member subdivision from 10 to 1 (sufficient for this model).
RMAA-96GF7L	crash report - accidental crash accidental crash	Problem identified and solved in R_2013
JBES-96LD4X	Question about 'order in code'.

In the image in attachment, you can see the 'order in code'. Where does the number exactly stand for? Is it possible to modify this number in such a way, that the material is not taken into account for checks? (for example when using dummy bars) | Order in code is internally used as description of the material type in many places. So it should not be editable. I changed it to NOEDIT flag. |
| NWEB-96LC8Y | Fire resistance check EC3: member S2 is classified as 1 -> ok. But member S7 (same cross section, same combination NC121, same internal forces) is classified on another position as 4. See attached project and pdf. | For S7 the internal forces used in the classification were taken from the start of the buckling system. At that position there is pure compression, leading to the defavourable classification limits. The reading of correct internal forces has been corrected in DEVE 11 and merged to R13.0.74. |

Ticket number(s)	Bug Description	Solution
NWEB-96SAAH	Generation of 3D wind terminates with an error in the attached file.	fixed in WLE 37, 13.0.61
JPOL-96SCUD	Problem with crashing SEn, all the times but especially during the analysis and document regeneration and pdf print.	Problem in Nedim dialogue. Fix available in R_2013
NWEB-96SKCQ	3D Wind Load Generator Incorrect Loading	fixed in 13.0.61
NWEB-96THRW	Crash when accessing unamed material - possible extension of Check Structure Data When opening the document of this project, there is a crash of Scia Engineer when 'Materials' is inserted (abnormal program termination - see print screen). Also, when I try to export the project there is a fatal error (also about the materials).	I have removed the crash. And it seems that the material is wrongly assigned reinforcement steel (e.g. B 500). With the new build you can "change element" (fourth icon) in the Material library for real one. If you have some info how was this situation caused, we could fix the primary usecase. Was it some "National Code" switch?
HWRE-96VDT8	Material table don´t fit on page of engineering report, see picture.	Solved in R_2013. It is necessary to re-read the table layout by selecting "Default" Table template in its properties
NWEB-96V8PW	Subject - Question EC3 german annex Why you have a gamma m0 of 1.1 for the German EC3? Normally it is 1.0. And why you have a gamma m1 of 1.0 in connection . Can I change the gamma in German annex by hand??	Please review the Theoretical Background regarding National Annexes (see attachments)a) For EN 1993-1-1 the German NA is using a specific routine.By default the following are used:Gamma M0 = 1,0Gamma M1 = 1,1Gamma M2 = 1,25.However in case the Steel Check is executed for a Non-linear 2nd order combination the following are used:Gamma M0 = 1,1Gamma M1 = 1,1Gamma M2 = 1,25.And in case the Steel Check is executed for an Accidental combination the following are used:Gamma M0 = 1,0Gamma M1 = 1,0Gamma M2 = 1,15.b) For EN 1993-1-3 the German National Annex defines the safety factors as follows:Gamma M0 = 1,1Gamma M1 = 1,1Gamma M2 = 1,25.c) For EN 1993-1-8 (Connections) the German National Annex refers to EN 1993-1-1 Within version R 2012 the safety factors for EN 1993-1-8 were defined in the National Annex Setup and could be manually changed. This led to the difference between the safety factors as defined in the DIN NA to EN 1993-1-1.Therefore, in R 2013 this has been modified: for the DIN NA to EN 1993-1-8 the same method for the safety factors is used as defined in the DIN NA to EN 1993-1-1.
HWRE-972CJ4	1) Open attached ESA file (national code DIN) with version 2013.0.50 2) Switch to EC EN 3) No national annex for Germany available 2)	Solved in DEVE 11, and merged to R13.0.74
HWRE-973DZS	Table of steel code check for thin walled cross sections don´t fits on page, see screen shot and esa file.	The table has been modified in DEVE 11 and will be merged to R 2013.0.87The units are now shown on a separate line so the table fits. In next steps of the steel refactoring actual Greek symbols will be used which will reduce the size even more.
NWEB-978B2S	Crash during regeneration of doc The document of this project (DOC - standaard) crashes when regenerating.	Workaround found. Cause of problem identified and fixed in Deve_09. Fix available in R_2013

Ticket number(s)	Bug Description	Solution
NWEB-975HR6	Non linear stability terminates with an error Hello,, in SEN 12.0.1094, the nonlinear stability terminates with an error (see screenshot)., in SEN 13.0.68, it works fine., Can you tell where that difference of behaviour comes from ?, And is there a way of defining the solver settings in SEN 12.0.1094 to get the correct results ?	The structure is experiencing a behaviour similar to "smooth snapthrough" (normally typical of shallow arches): the entire structure went below the supports and is hanging from the supports. Obviously, at that point, no instability can occur and the iteration keeps running. Apparently, the snapthrough phenomenon itself happened too smoothly to be detected as an instability.The physical interpretation of this could be, that there is actually no elastic instability occuring at all. The reasonable thing to do would be to increase progressively the loading and watch the STRESSES. Plastification is definitely going to occur before any elastic instability phenomenon.The process has been improved in SEN 2013, where the critical load is found. There is unfortunately no way of getting it in SEN 2012 at this time.
NWEB-979QST	make section cut on X=0 plane -> Scia Engineer does not respond any more	There was calculation of intersection with some cca. 2000 very complex general solid objects (AUTOMATIC ALOCATION). I have increased some hashtables and that improved the performance. Now the AUTOMATIC ALOCATION is calculated in less than 7 seconds.
NWEB-97AFRD	Look at the file + xls in attachment. In the excel file a manual calculation of the buckling factor is given (following the method in Scia Engineer). But this results in a different value than in Scia Engineer itself. I don't find any wrong results in the excel file. Can you please take a look at this?	In this specific example, when reading internal forces for the buckling algorithm the forces were read in the first section at 2,3m instead of at the second section. This caused the difference in the final result.The reading of internal forces has been modified, the result now matches the manual calculation.This has been fixed in DEVE 11 and will be merged to R 2013.0.87.
HWRE-97CFU3	Steel LTB German NA - Alternative Case LTB check of rolled section according DIN EN 1993-1-1, chapter 6.3.2.3, equitation 6.57, alphaLT can not be reduced as in chapter 6.3.2.2 in the meaning of the client. In NDP of 6.3.2.2(2) remark 1 the reduction is only done for equitation 6.56.	10/05 PVT: Modified in DEVE 11, will be merged to R2013.0.87From now on, the modified Alpha,LT* values will only be used in the 'General case', not the 'Rolled and equivalent welded case'.
LSKI-97QCLE	In the attached project I made an IRS calculation I do not understand the result under Summary results > Story Data If I consider the first level which is FL1 and given the symmetry of the members, I would expect that the coordinates of the mass center would be : 4 ; 4 ; 1 but Scia Engineer returns 4 ; 4 ; 1.182	this depends on the mesh. Actually, during reduction, each NODE of the original mesh is reduced to the closest R-node.With 10 subdivisions, there is a mesh node exactly at mid-height of each column, which could be reduced either to FL1 or to FL2 R-node, that's a matter of numerical sensitivity...Moreover, the bottom node of each column is taken out of the process, because it is blocked by a support.If all mid-nodes would be reduced to FL2, the mass center of FL1 would be at 4*(0.05+0.4 / 2) = 1.00mIf all mid-nodes would be reduced to FL1, the mass center of FL1 would be at 4*(0.05+0.5 / 2) = 1.20mNow, probably because of some numerical small number issue, some nodes are reduced to FL1 and some to FL2. I can't explain in detail the obtained value, but it is between the min and max described here.Moreover, if you increase the number of subdivisions, the position of the mass center of FL1 tends to 1.00m.
HWRE-97SK7N	1) Result of e,tot,z is in ER -749 mm, but in old docuent -687 mm. See attached project. 2) 1D concrete design don´t fits on page of ER.	1) Property Type of Check is different in old document and ER. Once it is the same the value of e,tot,z is equal2) Template fo check was modified

Ticket number(s)	Bug Description	Solution
JPOL-975AVD	Wind zones can't be edited. Each time I select "manual design" in the wind zones editor and click on "divide" button I cannot save my setting because two dialogues are closed in the same time when I confirm the "dialogue" (strange name of the dialogue, but this is the way how it is called) by OK button. Picture attached.	Solved in 13.0.93
GVAN-97ZH8R	I am using DEALER protection, and when using the old document I cannot export to HTML, EXCEL, .. any more	Solved in 13.0.102