Data stored in this version can be opened in version 14.0.101 (Release 14.0) and higher.

Ticket number(s)	Bug Description	Solution
NWEB-9EBPL7	Issue: multiple crashes So can you maybe find more information from the crash report?	This bug is already solved. Please install latest version of 2013.1
NWEB-9EBC7T	The problem with translation from Slovakia language to English	Solved in R14.0.22 The translation is now OK.
NWEB-9F7R6U	Crash report of the Engineering Report in attachment. What could be the cause of the crash?	Already fixed in SDI branch. Fix will be available in version 14
NWEB-9FUGY7	See project in attachment: It is a model in environment 'Frame XYZ', however, we only get options for 2D like for example: - properties of supports only for nodes in 2D model - not possible to change the workplane As workaround, you can change the project to 'General XYZ'.	Solved in DEVE 04 build 14.004.40 and verified in R 14.0.65.This was caused when 'cancelling' Member Buckling Data.
NWEB-9G4KP8	Perform a linear calculation in the project in attachment. You will get an error message which is saying that there are no load cases are defined. However, there are load cases and loads in this project? PS: the client says this problem has occured since he has updated to version 13.1.1040.	fixed in 14.0.32
NWEB-9G8J7L	Viewer mode -- service Results can be entered only once	Solved together with "protection lost" problem. Fix will be available in version 14
RMAA-9GBDPK	Crash report - Accidental crash of ER	The problem should be already solved in P_version
NWEB-9GGG6X	Crash report. The crash at ER.	Closing of ER was improved, It should be ok in next release
JPOL-9GQBXU	There is a message when entering Results service in viewer mode that "licence has been lost, please save the file..." This is obviously irrelevant, since there is no licence needed for the viewer mode and the user cannot save the file in the same mode. Could you cancel that message (see attachment)?	Solved in version 14

Ticket number(s)	Bug Description	Solution
NWEB-9GUDHS	Question: See project in attachment. There are 2 joined C-profiles (not of same size, on is 4mm and other 2,75mm thick) What is the best way to model this in the general cross-sections? - In the cross - section 'P', the initial shape is not recognised correctly. - In the cross - section 'P ecart' (0,01mm distance between the 2 C-profiles has been used), the initial shape is recognised better, but the 2D FEM analysis can't be used on the section	1) The initial shape is based on the centerline. For this composed cross-section, the two parts are connected using fictive centerline elements with zero thickness (see CSS TB for more info). For the generation of the initial shape, those fictive centerline elements are not accounted for. As a result, the remaining elements are not at both ends connected and thus correctly receive the type outstand (UO).A correction has been made to the element types in R14 build 14.0.42. The types are now set before the removal of the zero thickness elements, thus all elements expected to be Internal (I) are indeed set as Internal.Note: To account for the above in an existing file, de-activate and re-activate the initial shape. 2) The 2D FEM analysis indicates that two points coincide. As the user indicated, the distance between the two sections is just 0,01 mm while by default the minimum point distance for the 2D FEM mesh is set to 0,1 mm.Different points are thus taken together as being one point, leading to the message. The solution is simple, reduce the minimum point distance for 2D FEM to something smaller than the 0,01 mm of the cross-section dimension, for example 0,001
DPIS-9H5F2N	Problem with the viewer of Scia Engineer. After opening a project with the viewer of Scia Engineer, and going to a certain menu (e.g. results), there is a message that the protection is lost. (message attached). After clikcing on OK, there is not immediatly a problem, you can check the results without any problems. But if you leave the results menu, you can't do any thing any more in the veiwer. You can't go to any menu anymore. The only way is to close Scia Engineer and restart it again. Tested in the viewer of Scia Engineer 2013.1.1040.	This problem is already solved in R_14
JPOL-9HBAV5	Error reports from ER, please see attachment. User says it happens often, when he puts several drawings at once from inbox directly to document.	TUM 21.3.: See comment from LAT==It is crashing because of out-of-memory. Most probably due to big number of repeating pictures. This functionality was improved in version 14. Could you please ask user whether he can provide us his data so we can use it as a testcase for this new improved working with chapters?

Ticket number(s)	Bug Description	Solution
JBES-9HDF3M	Project in attachment is a prestressed beam with a cross section consisting of 2 phases. What client indicates, is the result when you would have only the first phase. - Point of gravity with the pressure layer in the section (so only fase 1) would be 388mm from the bottom - Point of gravity of the complete cross section according to Scia Engineer would be 564mm from the bottom side. This last one deviates only 11mm from hand calculation, so it's OK. Now here comes the problem: --> To check the results, there have been 2 tendon set into the first field, which are at 70mm from the bottom side. --> The normal force corresponding to these 2 tendons are also given correctly by Scia Engineer: 2 · 93 mm2 · 1440 N/mm2 = 267,8 kN --> The moment that is to be expected should be (388 mm – 70 mm)/1000 · 267,8 kN = 85,1604 kNm according to simple manual calculation. But Scia Engineer gives (564 mm – 70 mm)/1000 · 267,8 kN = 132,2932 kNm… (so it looks like we take the wrong center of gravity) (tested in 2103.1.1040 and in 14.0.32 --> both give 133 kNm for load case "P1 - Veld 1 voorspanning" while it should be 85 kNm)	
NWEB-88TH7Y	message: "Array size exceeded" caused by rigid links	solved in deve2 (11.002.61)
NWEB-8F8CS6	We don´t get linear results for attached project. It seams that it calculates (>1h) but Scia don´t write results. Attached my message: 2010.1.269, vista 64bit, 3GB Costumer has: 2010.1.400, Windows 7 64 Bit, 12 GB Final project raises from 3mb -> 3,6 GB(!) > 200 Loadcases	2011: the program computes the message : "not enough ram memory"
NWEB-8W99JV	ISSUE: how to change the color of the load symbor of load panels ?	fixed in R14 (14.0.81)

Ticket number(s)	Bug Description	Solution
RMAA-929FVG	Please, you could verify cracks for BS. Details in attachments. Compare BS8110-2 and Hand Calculations	EH, 29.11.2013(1) A bug in the NEDIM BS Crack Proof algorithm was localized in the formula of acr (BS 8110/2,Par.3.8.3)(2) The results Ar1- of the erroneous and correct Crack Proof calculation are presented in the attachment: files >Ar1- before NEDIM V13,1,0,0.jpg< and >Ar1- under NEDIM V13,1,0,0.jpg<(3) The corrected NEDIM Version is V13,1,0,0.(C) There occured the overall enquiry, how the NEDIM Crack Proof works. Here is my principal statement:(a) A typical hand calculation starts on a set of parameters such as {As,prov;m/n, fy,fc,h,d,c etc.} and yields the corresponding crack width Wcal;(b) Unlike the hand calculation, NEDIM (using the given physical and geometric quantities as well as the calculated As,ULS (maybe modified by min reinf requirements)) starts an iterative (stable, "monotonously damped", asymptotic) process which stops when the crack width required, e.g. Wreq=0.20 (like in this example), is attained with required accuracy (0.5% in NEDIM);(c) During the iterative process, the starting reinforcement amount As,ULS is (if necessary) modified, i.e. augmented, to meet the Norm requirements for limiting the crack width to the given value (e.g. W=0.20 mm). The result reinforcement As,ULS+SLS is the "crack reinforcement".CONCLUSION:(A) The delayed procession of this Ticket had its contractual reasons!(B) Thanks for your thoroughful documentation of the program bug!Dr. Eduard Hobst
CSCT-98FK7Y	In attached file we get the toplogy-error at the 2 plates at the column. If we refine the mesh as in this ticket CSCT-98JCXM it works. Why (so strictly)?	switch on elastic mesh in mesh setup, detailled explanation is in comment from Zeiner, tested on 14.0.53
JPOL-99N9H4	esa.exe is stuck in processes and I cannot run it again unless I close in manually in task manager. I reported this issue some time ago (when I used Windows 7 PC) but was not able to give adequate information about the problem. A few days ago I got completely new computer (with Windows 8) and I can declare the problem is replicable. Originally there was "typical" installation of R-version Scia Engineer 2013 and everything was OK. But I need to have more languages installed (more than default EN and CZ), therefore I run setup again and added more languages (namely SK, DE, PL, GR and RU). From that time I cannot run SEn for the second time a day because esa.exe is still running in processes even if I close the application. Really annoying.....	Solved
JPOL-9BUJLB	Web Help issue. Current web help as well as reference guide doesn't contain description of properties of 2D/1D upgrade in case I select integration strip. It describes only properties of the dialogue in case I choose prefab slab made of ribs. See pictures attached and compare the dialogue and cut from the help. Please add description of parameters of the other dialogue (like switch orientation, ...) Tested in 2013.0.1036 and help.nemetschek-scia.com on 24.9.2013	See attached file,See:http://help.nemetschek-scia.com/13.1/en/#rb/results/upgrade_from_2d_to_1d_project.htm

Ticket number(s)	Bug Description	Solution
NWEB-9BXGY8	Go to the document of the project in attachment. The values for the green loads of chapter 2.2.1. are printed double and are not clear anymore (see also printscreen in attachment). Can you please take a look at this?	Solved in R_14
LSKI-9CCFWE	See attached pdf for explanation	please check the comment and let us know
JPOL-9CJCQH	Incorect result properties in picture	Solved in R_14
LKGZ-9DEE48	It takes about 10 seconds to enter a service menu from main menu tree, for example the Structure menu. - temp folder was cleared - new installation was done - the projects are small (< 100 members) Any other idea?	Problem solved. Wrong USB3.0 driver
NWEB-9DPKA9	Open the project in attachment and look at the results for Vz (see also the printscreen Vz.jpg in attachment) and choose to send this picture to the document. In the document you will not see the values for Vz anymore (see printscreen Document.jpg)	Solved in R_14
NWEB-9DPL3C	Question: When loooking at the table of the member design ULS + SLS, we see the values As,add1 and As,add2. Why are they displayed in this table? It is confusing what they exactly mean because the explanation differs when changing the value of reinforcement in the properties, but the name (As,add -> what normal means additional reinforcement) stays the same? See also print screens in attachment.	Solved in Deve_04_DocX64_SDI
NWEB-9DSNTU	The cross section is slightly unsimmetrical then it give a high value of shear stress. Is this correct ?	Solved in DEVE 11 Build 12.011.374 and merged to R 14.0.22In this specific case, the angle of the principal axis of the cross-section exceeded 90° which lead to incorrect shear stress results in the fibres.After this fix, the shear stresses in B55 and B61 are nearly the same.
NWEB-9DTEL3	Run Modal analysis Go to service Results, select Calculation protocol and in its properties set to "Eigen frequencies" Send the table to ER --> nothing is displayed Problems: - Selection is set to list, but nothing is selected - In Scia Engineer the result does not have property Selection - Once the selection is set to All, the output is generated two times in Eng report	Solved in R_14
RCCA-9DZGGN	Shear reinforcement : See doc in attachment.	Lukas could you add to project for EN in SDFThe problem was solved and template updated
NWEB-9EHJ59	Problem with floating Command line on two screens	Solved in R_14

Ticket number(s)	Bug Description	Solution
LSKI-9EJM3U	Open the engineering report and try to regenerate => it crashes Isn't it possible to use indented tables for Detailed storey results ?	Solved in version 14
NWEB-9F3QSC	Problem with 3D wind generation. The client has approximated to cruved part using multiple plane panels but the structure is closed.	fixed in WLE 58 (14.0.87)
NWEB-9F49H2	Translation is not working correctly in the Engineering Report (and old style document). By changing the language of the Engineering report, there are some elements in the steel code checked, that are not translated. See printscreens attached. Tested in Scia Engineer 13.1.64	Solved in R14.0.22 The translation is now OK.
NWEB-9F6BFF	Steel Cold Formed: Division by zero in combined stability check Perform a linear analysis and then execute the Steel Check for the member B286 for load case LC1 self weight. As shown on the screenshot in attachment, the combined check according to formula (6.36) is undetermined. Looking into the table it can be seen that Nb,Rd and Mb,Rd are zero which is the cause of the division by zero. They seem to be zero because neither a buckling check nor a LTB check has to be done. In such a case those values should be taken just as NRd and My,Rd. In fact, it's the Chi values which are 1,00 so Nb,Rd = 1,00 * NRd and Mb,Rd = 1,00 * My,Rd Compare this for example to the standard method where NRk and My,Rk remain unchanged here. Note: In this specific example the buckling check is not even printed since there is no compression force. In that case this interaction check shouldn't even be executed since there is just one component.	Solved in DEVE 11 build 12.011.373 and merged to R 14.022In this specific case there was no compressive axial force (so no buckling check nor resistance) and a low LTB slenderness (so no LTB check nor resistance).As a result, the alternative interaction lead to a division by a zero resistance. This has now been corrected.
JPOL-9F6HCX	Mistakes in translations. See attached pictures and correct please: - word "kontreolu" on the first one - the whole English expresion "number of..." on the second picture Tested in 2013.1.64	konreolu = fixednumber of ... = input as a separate bug - see the link in attachments

Ticket number(s)	Bug Description	Solution
JPOL-9F8JE3	Misleading description of cross-section properties. Axis systems in cross-section property dialog is not correct. The value cZUCS is described as coordinate in Z-direction of Input axis system. However the input axis system has no Z, is it only XY (see picture attached). It should be local Z-direction or input Y-direction. Similarly in the table of named items coordinates (see another picture). Here we see coordinates in local acis system, hoewver displayed is input axis system only. Either don't show it or rename it so that it correcponds to the picture next to the table. Tested in 2013.1.64	a) UCS axis labels: Solved in DEVE 11 build 12.011.373 The UCS labels have been modified to "Y" and "Z" and merged to R 14.0.22b) Named fibre coordinates: The coordinates were shown in the principal system, this will be modified to the UCS system, see bug PVTO-9F8H3G
JBES-9F8HLY	Solver crash due to zero system length Issue:, Do batch calculation (linear + non-linear), --> model crashes at non linear calculation (version 2013.1.64)., --> tested on my pc and same crash, But if you throw away all buckling data, then it runs correctly....	Solved in R14 Build 14.0.42- The crash in the non-linear analysis was caused by incorrect imperfection loads- The incorrect imperfection loads were caused by a zero system length- The zero system length was caused by the fact that on some members, for example S1218, the user inputted a node and a cross-link on exactly the same position. Both of these entities are accounted for in the buckling system and since they are on the same position the system length is zero.In the buckling system, such input errors are automatically discarded, however when using member buckling data the actual system, as inputted, was used, leading to the zero length and incorrect imperfection loads derived from it.The member buckling data has been revised and now also ignored such zero length parts.
JBES-9FFKRE	If I try to import this dwg file in to Scia Engineer, then it crashes.	The problem is in the library which is used for the DWG import. There is a loop and the iteration cannot finish. SEN has no report from the library which will show us where is the problem, so we cannot fix it and even we cannot show some error message. In this case, the exploding of all blocks in the DWG helps, you may use attached file from Pavel Lokvenc.
LKGZ-9FCGTC	Check add data: Linking symbols to model scale We cannot adapt the size of alu-weld-labels (Screenshot)	ALU LTB restraints, ALU transverse welds, Steel local transverse forces data etc are now properly linked to the Model data scale. Fixed in DEVE 11 and merged to R 14.0.22
NWEB-9FRN39	Crash reports of the Engineering Report in attachment.	Crash started at droping of dragged item. We changed the code to prevent this crash but please ask user to send us project & procedure how to invitate this crash.This look as not implement usecase (drop into/between/after) item.
NWEB-9FRNG7	Engineering Report Imperial Unit Issue See attached file and screenshots. Scia Engineer lists a value for reinforcement in in2/ft in the preview dialogue and in the Engineering Report. However, the value in the Engineering Report is not correct. The value in the Engineering Report that is being used is actually in2/m even though the table lists in2/ft as the units.	Solved in R_14
CSCT-9FVCTW	Please view the file in attachment. The check for member B16590 works for the Graphical & Brief output but crashes when using the Detailed output or Single check. Investigation shows that the issue is caused by the name of the material: "S235 min 15%" When I remove the "%" symbol everything works.	Solved in DEVE 11 build 12.011.376 and merged to R 14.022The issue was caused by the usage of the " % " symbol in the name of the material.

Ticket number(s)	Bug Description	Solution
NWEB-9FVJ55	See project attached. Customer wants to test membrane plates, but there is no difference in bending moments between a normal plate and membrane plate?	fixed in 14.0.32
NWEB-9FVGXM	Crash report - 2D member geometry invalid Accidental crash: Crash while adjusting model. The crash occurred while I was trying to deal with an error. When starting a calculation, the error message “2D member ... geometry is invalid”. I’ve been moving things around, replacing others, and the message stills occurs. The crash occurred after I’d been fiddling with it for a while. Any suggestions on what might be causing the message?	The source of crash was in DocSMN.dll. It has been already solved in 12/2013
NWEB-9FYJWQ	Crash report	Problem solved in Deve_04. Fix will be available in version 14
RMAA-9G2GAU	The file Esa.exe stays to hang in processes Windows after closing Scia Engineer. It is always, if user switch on ER. It is OK without ER. Operation system: Windows 7 profesinal 64 bit. SK , update full, service pack1 Intel Core i5 procesor Version Scia Engineer 2013.1.64 There is antivirus Nod32, but there are exceptions on folders (SCIA, TEM , USER) We tried to work as administrator = the same problem. We tried to uninstall Scia Engineer and to clean register, afterwards new installation. It was without success. You can see in attachments comment by user.	Problem solved in R_14
NWEB-9G3BEW	crash out of memroy during drawing of surfaces - not a big project The crash is easily reproducible: open project Switch ON surfaces --> crash The project des not look so big to crash during drawing	The user has defined an unfortunatelly high precision factor for geometry. We recommend more realistic calues in attachements. That works just fine.
DPIS-9G7F86	definition of Romanian seismic response spectrum The coeff acceleration for Bucuresti (Romania), is now 0,24. According to a this customer, this value should be changed from 0,24 to 0,32. Can you have a look into it? And do the changes if needed?	I suppose that we are talking about the update of the Romanian design codes from last year.This is planned for implementation. Hopefully, we will have it in SEN 14, but it is not 100% sure.For the time being, users have to use manual definition of the spectrum parameters by selecting "other" city.

Ticket number(s)	Bug Description	Solution
RMAA-9G7JLU	The problem with value Cmy The value Cmy should be 0,9 according to user for check. Setting check is in attachments. It was tested in version 2013.1.1040.	Modified in DEVE 11 build 12.011.376 and merged to R 14.022This issue was specifically in the Cold-Formed check and has been modified.
JPOL-9G7JR4	User cannot export a project into ESA IN format, please see attached screen shots and error report.	Solved in Deve_04. Fix will be available in version 14
JBES-9G8D8J	Open project in attachment. Open the 'Template Dialogue'. 1. Change L to 30 2. Change n to 12 --> this last should be possible because the boundaries of n are determined by the formulas: (L-6)/2 ... (L-3.6)/2 --> so for L = 25, the boundaries of n are 9,5 & 10,7 --> and for L = 30, the boundaries should be 12 & 13,2 for n. So it seems like there is a small issue here.	New button "Apply" was added into the Tempalte dialogue. It enables to update vaues of all formulea used in parameters and also update the range defined by formula.
NWEB-9G4A9Y	Member Buckling Data: 'Cancel' causes project change from 'General XYZ' to 'Beam'	Solved in DEVE 04 build 14.004.40
RMAA-9GBH45	The problem at ER with table by member stress More detail in attachements It was tested: 2013.1.1040	Solved in version 14
NWEB-9G7FRJ	CSS: Unexpected Shape Type and Initial shape 1) A General CSS was inputted using a thin-walled section from the LIB and a thin-walled line. The Section is correctly set as thin-walled and a centerline is generated. When pressing update however the section switches to thick-walled => Why does the section switch to thick-walled? . 2) The centerline shows that almost all elements are 'Internal' which is correct. There are just two outstands. However when looking at the Initial shape, all those internal elements are set as UO.	Modified in R14 build 14.0.42a) The cross-section switched to 'thick-walled' since not all parts of the general section are created out of library sections. This has been corrected.b) The initial shape is based on the centerline. In this cross-section, fictive centerline elements with zero thickness are used to connect both shapes (see CSS TB).Since those elements have zero thickness they are not accounted for in the initial shape. That is why several initial shape parts are set as outstands (UO) since they are indeed not connected on both sides.A correction has been made to the element types. The types are now set before the removal of the zero thickness elements, leading to all elements of the QRO as being Internal (I)Note: To account for the above in an existing file, de-activate and re-activate the initial shape.
DPIS-9GHJ8H	In the Engineering report, in the table of the cross section, the values for Wply and Wplz are ranked in a strange way. Normally, it is always first the y value, and next the z value. See image attached for the values with the strange ranking of the values.	Solved in version 14

Ticket number(s)	Bug Description	Solution
DPIS-9GJLXE	BS Steel Check: Check returns just one section In the attached project, the steel code check (BS), can only be done in one section on beam B199., For all the other sections, there is no result., For beam B200, the situation is even stranger. Several sections with results, and several sections without results., See printscreens attached., Tested in Scia Engineer 2013.1.1040., The check according to the Eurocode can be done properly without any problems.	Solved in R14 build 14.0.42When checking one member which is in a buckling system with other members, all critical combinations over the whole buckling system need to be mapped to the CADS Steel Designer.In this specific project there was an issue in the mapping, thus the CADS Steel Designer returned results for a combination on the buckling system which was not listed as critical for this member.Since the combination did not match, no result was shown in those sections.
NWEB-9GMDTM	ISSUE: crash and error report - crash happened when closing Scia Engineer	This problem was already fixed in newer version of 13.1
NWEB-9GNB4W	Crash report - POD: An exception during work in Engineering report. See attached info POD: An exception during work in Engineering report. See attached information from the user who can, however, continue with ER.	Similar problem was already solved. The fix will be available in R_2014
JPOL-9GNDFW	User complaints that stirrup diameter is not visible in the properties window when there are more than just one regions - therefore more possible diameters. It is true that when there is more than just one region of stirrups where each region has got different stirrup diameter, tha value of diameter and cover disappear from properties. This is not logical and doesn't follow the system of Scia Engineer. Anywhere else in the program, when there are different values of a particular property, the item remains in properties and the value field is empty. Thus the user knows it is not identical value. But here is simply hides the whole property. Any chance we can harmonize this behaviour with the software logic? See attached pictures for clearness. Tested in 2013.1.1040	The behaviour of property Diamater , where more diamater is defined is correctly and follow the system of SCIA Enginnier, becuase - the item Diameterr is appeared and it is empty if the two same object are selected (for example two stirrups zonesd with diofferent diamater) - the Diameter is not appared, if in one object, the different diamater of stirrups are defined (more part or more zone is defined)The system is used in whole SEN system. It follows, the property Diamater can not be appeared, because one object contains the different properties (diameter of stirrups).
NWEB-9GPL3T	Steel: Diaphragm data not editable and division by zero Why isn't it possible to change the data values for library diaphragms (see attached screen shot1) ? Then in the detailed output for steel wee get some weird symbols (see screen shot2)	a) The Non-editing of the library was intentional and was done several years ago during the Composite Development. It has been made editable again in DEVE 06 and merged to R14.0.65b) The division by zero due to zero stiffness has been fixed in DEVE 11 and merged to R 14.022
NWEB-9GVDFH	See project in attachement. Try to edit the cross section in the cross sections editor. The customer wants to have rounded corners, so he uses the command "Polyline edit" > "Create fillet in polyline". This has worked on one corner, as you can see, but when trying it with another we receive a crash. Could you also test it? How can we solve it?	fixed in 14.0.32

Ticket number(s)	Bug Description	Solution
NWEB-9GVHCB	Free surface load is wrongly generated, selection doesn't work. Please see attached project test.esa: - Local case "pokus1" consist of a free surface load that covers all three spans of the structure but should be generated to side spans only. This is correctly displayed by orange loads. However 2D data viewer shows that all 3 spans are loaded (see picture) Please see attached project test2.esa: - load case "pokus2" has got two independant free surface forces that are assigned to side spans and selection is set to the corresponding slabs. However the result is zero (see picture) -load case "pokus3" has got only one free surface load but 2D data viewer shows that nearly all slabs are loaded. Even more - some finite elements at the edge are not loaded while neighbouring are. See picture enclosed again. Tested in 2013.1.1040	LC1 fixed in 14.0.32; LC is correct (one load has empty selection), LC3 - when the load is not on line of mesh then the load is displayed by FZ (it is a feature)
NWEB-9GXMMR	The attached project can't be opened > crash report. Crash report is also attached.	Solved in version 14
NWEB-9H4HEU	bug in FTD pre-processor	bug fixed in VBA macros - please use ftd_1.3.xlsIMPORTANT: in this specific case, due to the large volume of generated data, the old Excel file format .xls cannot handle it. (the number of lines for dynamic load functions exceeds 65535).The Excel file MUST be first saved as .xlsm (Excel 2007 and above), closed and re-opened to be able to generate the xml file.Please note, that it has never been tested with such a large amount of data.The import in Scia Engineer and the dynamic analysis might take quite a long time.
JPOL-9HCCJE	Functionality "Old style document" is active every time I open a project even though I have deactivated it before I saved the project. This functionality cannot be deactivated once it was active, even though there is no document in the project. Therefore there is always unwanted item "Document" in the main tree. Try to disable this functionality, save the project and see that if you open it again, it is ticked. Tested in 2013.1.1040	Solved in R_14
NWEB-9HECNH	Attached project and picture. One cut is missed between at S218 to S219.	probably fixed in R14 (tested in 14.0.40)
JPOL-9HEE55	Crash of SEn regularly appears on the following project. User cannot continue working on this project. Please investigate from attached crash reports (from user and also from my PC) the reason. I only opened the project, run connect members and nodes function and SEn crashed. The same happened for linear analysis. Tested in 2013.1.1040	fixed in R14, problem was caused by genex (connection and mesh generation)
NWEB-9HCJ6H	When clicking on "connect member nodes" I will receive the message from attachment. When deleting all lines in the project, still same message. Do you know where this comes from?	fixed in R14 (14.0.32) with a new mesh

Ticket number(s)	Bug Description	Solution
NWEB-9HJG8N	When connecting member/nodes an error message is shown (see printscreen). What is might be the failure reason?	fixed in R14 14.0.68 (13.123.67)
NWEB-9HMBG2	DocX64 - Issue: materials in the engineering report. By inserting the materials in the engineering report, there is a problem by selecting a list of materials. Why can't the table be generated for the following list? (see attachement)	Solved in R_14
NWEB-9HRCZG	Project as been loaded in into Scia Engineer 2013.1.1040, and now gives this error message in the linear calculation (also gives this message in Scia Engineer 14.0.32). Calculation still continues, but where did message come from?	Fixed in R14 (tested on 14.0.46)
JPOL-9HXG8W	An error message appears during connection of members. Please see attached picture and esa file. The information that something doesn't exist is not useful for the user and it seems that he cannot repair the model himself. Therefore please correct the program so that similar errors doesn't appear and the model can be connected automatically.	fixed in R14 (14.0.68) 13.123.67
NWEB-9HTKAP	Look at the project in attachment and click on "Connect member nodes". You will receive the message of figure "image002". Can you please explain the problem here?	fixed in 14.0.68 (R14)
JPOL-9JVELY	Unstable structure for a bit inclined columns. Take a look at attached structure. When all supporting beams (B60-66) are strictly vertical, there is no problem. However, when I change Y coordinate of the bottom nodes (N91, N79, N89) to 10,59 m (that makes those columns imperceptibly inclined) there is a singularity and calculation cannot be finished. But when I change the supports of these three nodes to fixed, there is no problem again. And if you compare results on columns when there are hinges and fixed support they are almost identical (with the exception of Vy). Conclusion: - vertical columns supported by hinges - OK - vertical columns supported by fixed supports - OK - inclined columns supported by hinges - SINGULARITY - inclined columns supported by fixed siupports - OK Results are nearly identical for all cases. Could you explain this? Tested in 2013.0.1048	fixed in R14
NWEB-9GYKBX	Open project in attachment 'wdc675_0_10scia.esa'. Calculate project, go to steel code check, ask unity check for class RC3 for all beams with section 'ES-XEZ - Euro Upright ...' The preview does not have a good lay-out (as you can also see in the pdf). Tested in 2013.1.1040. How to get a good lay-out?	Problem is caused y the sorting of the table. Go to table composer and switch the sorting off

Ticket number(s)	Bug Description	Solution
NWEB-9GYKBX	Open project in attachment 'wdc675_0_10scia.esa'. Calculate project, go to steel code check, ask unity check for class RC3 for all beams with section 'ES-XEZ - Euro Upright ...' The preview does not have a good lay-out (as you can also see in the pdf). Tested in 2013.1.1040. How to get a good lay-out? (work-around: send table to engineering report, the ER displays it on a good way)	Problem is caused y the sorting of the table. Go to table composer and switch the sorting off.Problem is inj old document which is not bugfixed any more.
NWEB-9JMGZ2	We can't calculate this project. It crashes. Also crashes when trying to export the model to a new esa file! (tested calculation & export in 2013.1.1048 & 14.0.32)	see comment (workaround)
NWEB-9DGHQ6	Open the model in attachment and go to "Tools -> Cleaner" and clean everything under "General". Calculate the model again and go to "Concrete -> 2D member -> Member check - Crack control" and look at the preview for the results as shown in the printscreen "Crack.jpg". When looking at mesh element 169 (in the middle of the plate) you will see that you dont have any m1 or m2 here (see printscreen "Results.jpg"). But this is not correct. We do have a value for m1 and m2 here!	see bug JBE14-1019HYC32
NWEB-9EFHQN	If I generate the mesh of the project in attachment, then there are complete parts missing (elements E12 & E13, see also image). What causes this wrong mesh generation (since there are no cut-outs in these parts)?	fixed in R14
JPOL-9F4CEP	Strange error message appears before the analysis in case free plane load with constant value but linear distribution is defined. See attached pictures and esa file that proves end of FEM analysis due to wrong input of load. However the load is quite simple. But if I change the distribution from "Direction Y" to "Uniform", analysis is ended successfully.	fix in R14; remove temporary files and generated load by Cleaner
RMAA-9FDEBE	The problem with topology in slab S1368 The strange is if I delete all mesh on surface or there set size 1 m, afterwards the mesh is OK.	fixed in R14 (13.123.77)
JPOL-9GYEXH	Calculation of maximal transverse distance of stirrups. 1. In attached project there is a column B93 where the unity check of detailing provisions is lower than 1,0 with stirrup distance less than 180mm. The question is why? From EC EN 9.2.2(6) I can see that s_l,max is 0,75*d(1+cotg alpha) where alpha is 90° and d is 325,1mm - resulting in s_l,max=244mm. (see picture too) 2. Beam B92 has got stirrups of 4 cuts, however it seems that SEn takes into account each stirrup separately for analysis of maximal transverse distance. The distance calulated is 211mm, which corresponds geometry of single stirrup, not both of them (see picture too).	1/ value slc,max for column and for Slovak NA is calculated according to picture below and the value 1810 mm is correct (15*min. diamater = 15*12)2/ send for analysis SE
NWEB-9C4PA6	When calculating, the error in image keeps appearing. (calculation will still continue, but are results then still ok?)	fixed in 14.0.96 (13.123.77), save in R14 and then calculation works

Ticket number(s)	Bug Description	Solution
JPOL-9HUHL7	Hidden calculation gives different results than standard calculation!!! Please open attached project and compare standard and hidden calculation. You can repeate them as you like, it is not dependant on the order of analyses. Results on beam S429 are shown on attached pictures. This is very serious bug! Explain in the shortest possible time which result is correct and fix this.	fixed in R14 (14.0.96)
NWEB-9G8JL4	Wrong recalculated internal forces (concrete) in ribs Look at project in attachment (tested in 2013.1.1040 & 14.0.32).	Internal forces from plate are integrated into ribs in hidden calculation as well
NWEB-9J5FSA	The regular crash of SEN during linear calculation I found that problem is somewhere at load cases by 3D wind, after deleting these LCS, the calculation runs without problem.	fixed in R14 (14.096)
NWEB-9J7FEG	Crash report	Problem identified and solved in v14
GVAN-9JEDWZ	cannot show the Internal forces - an error appears in the Preview window.	Shoul be fixed in 14.0.98, changed installation folder for OpenCheck files
RMAA-9JFEQY	The problem at linear calculation 1. Run linear calculation 2. Scia Engineer displays error report = Maximum field size exceeded! Node/Line/Macro No.2657 Note: I found that problem is somewhere at cross-link setting type coupler. If I set type fixied, the calculation is without problem. More in attachments.... It was tested: Scia Engineer 2013.1.1048 and 14.0.72 ------------- Could you give a msg which told the user what the problem is? I think the structure is unstable due to CL177 and CL178...	fixed in R14
JPOL-9JLGPG	Accidental crash of SEn when changing the view.	Problem identified and solved in v14
JPOL-9JNANB	Table of materials in Russian ER doesn't fit on a page while other languages work fine. Please see attached picture and correct the default table layout. I guess it is in wrapping of some columns which is not ticked as default. The user can correct it himself, of course, but this should be OK as default. Tested in 2014.0.76	Solved in R_14. It is necessary to reload Derfault template of the table
NWEB-9HSCMK	Join nodes elements - Macroelement 2147483647 not found Dear	fixed in R14, the issue with mesh generation is described in comment from Zeiner, the issue is with S914 and S 1718