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

Ticket number(s)	Bug Description	Solution
NWEB-95K6QB	Steel BS: Check is not executed for all members when using selection 'All'	This has been fixed in R 2013.0.1021.The fix requires the installation of a new BS2000 module version 1.2.186.0.
CSCT-94DD2J	What can be a cause of attached message during start and what to do? Costumer has steeltemplates for Allplan (apesa) Dongle 552965 - - - - - - - - - - CS19/3: We have still these messages on w7 after running the batch. On xp it´s ok. Any other idea? DevTrack Status: No Bug To fix it allow to register COM object as administrator. And run a batch: NemetInstallExCom.bat placed in \Allplan\2012.0\ directory (2012.0 is version of Allplan)	should be fixed in next patch (13.0.1004)
NDES-95ZFF7	Make a new project and choose for the Eurocode, NA = Belgium. Go to this national annex to "EN 1991-1-3" and change the psi factors, as shown in the printscreen "snow" in attachment. When saving those settings, the psifactors under EN 1990 should be changed also (this is the case for the Standard NA), but this is not okay for the Belgium NA (see also printscreen Psi factors.jpg)	fixed in 13.0.1009
	Version 13.0.34: In this project, the engineering report can't be opened. See attachment for the project and the error message.	Works fine in SDI branch
NWEB-96MDF4	Error message improvement, Dear Miss Hermanova,, Please run the calculation of the attached project. You will get Message:, "Zero length for the slope determination in the definition of the linear varying rectangular or polygonal surface load in surface No.0 (FE 2D macro)!", Could you please add the number of the surface load in the error message?, Now we have to delete one load after another and try to run calculation until the mnessage disappears and this is time consuming.	fixed in 13.0.1018
NWEB-955CDH	In Rel. 2012 with German interface, the help was linked to a german help file. In 2013, it is linked to an english PDF. Customer says, the timber documentation is not complete in some parts, for example the parts in attached picture are missed.	The mentioned PDF file is available in all (standard) languages in v. 2013 (added to Setup on 8 May 2013)

Ticket number(s)	Bug Description	Solution
NWEB-96VDTN	In the attached project, the category of the terrain for the wind is not correctly shown. In the project data, there is shown category 3, and not category 3b. Also in the document, the same problem. This happens not only with this project, but with every project. This customer even had the case where category 1 was shown. In the French code, there is no category 1. We have checked the results, and they were correct. So it is just a problem with the display of the category of the terrain, while the calculation is done correctly.	fixed in 13.0.1009
JPOL-978C8A	Several crashes on attached project appeared - namely during test of input data and during calculation. I happens with the original file as well as with adjusted mesh setup. Crash reports attached as well. In case of the first crash report I might say there is another bug because the report was created incomplete. Tested in 2013.0.50	set in Mesh setup - Minimal distance between two points to 0,1m.
NWEB-96UJ56	Export Allplan project to SENG through Roundtrip - error message. There is an Allplan 2012 project attached (SIC2012.ZIP). If you export the project you get an error message which is attached as a picture. I found out, that the message is caused by 3 short walls in the left bottom corner (marked in the pictures). After I delete these 3 walls, then it works. But how should the user know, what is the problem? Could the error messag ebe improved, or better, the user should be asked if he wants to export also these parts.	fixed in next patch (13.0.1004)
NWEB-979KB6	ISSUE: export model to ifc -> the top flange of S55 and S59 should be on the same level now. When using the option Brep then this is OK, but when using the option Swept Solid then this is NOT ok.	fixed in next patch (13.0.1004)
NWEB-96NMAZ	Crash Report - do you see why? Could you see why Scia Engineer crashes by closing of the software? Customer sent me the crash report.	Solved in Deve_09 (see also last commen from Martin Mikelecky). Available in next R_patch
NWEB-97C5KY	Wrong topology on an non-existing element. Take a look at attached pictures. I get an error message during mesh generation that there is a problem on S746.	some improvement was done (for small simplified structure) but it does not help for whole project. It is necessary to correct it manually - make lines/edges which have

Ticket number(s)	Bug Description	Solution
	However, such an entity doesn't exist in the model. How should I read the messages SEn is giving me? How could I proceed to correct the model and generate mesh successfully? Please use the project calles SCIA - cela konstr (POD). Although the original project can't be analysed neither, I would start with this one, which has got no openings and subregions. Tested in 2012.0.1094, student version.	the same shape (then intersections are generated only there where they are required and not where they would not be)).
NWEB-97PC8X	Scaffolding Coupler: Incorrect internal forces when using Result Class	The Result Class did not properly get results for CN4, leading to an extreme result caused by CN3.This has been solved in R13.0.1007
NWEB-97PHR9	All units in this project has been set to Imperial system (ft) Go to results menu, choose Internal forces result and open the detailed outpu. You can see there that lengths are not taken in feets but in meters	fixed in 13.0.1013
CSCT-97QCWA	If we do a linear calculation we get a message about a 2D-macro with 0-length. Any idea? It seams because of any connection..	problem with 0 length macro is fixed, but the structure is not stable, it needs to be supported properly... (tested on build 13.0.1004, will be in next patch)
NWEB-97SFDB	ISSUE: fire resistance check: the value of the Fire resistance is 180.02 min., but this does not correspond with the critical temperature and the data in the fire curve. This 180.02 min. seems to be some limitvalue, because there are a lot of beams that have this value for Fire Resistance. see S12342 for NLCombi12 in SciaTest3.esa	As an example member I will use S12342 for NLCombi12.This member has a critical temperature of 1009,86°C. To determine the Fire resistance time, this critical temperature is looked up in the temperature-time curve.The curve set for this member however, only goes up until 740 °C. In other words, the critical temperature of 1009,86°C is never attained, so the member is perfectly able to withstand the given fire.Within Scia Engineer, a limit of 3 hours (= 180 min) is applied i.e. the program searches for this critical temperature, each step increasing the time by one second. When the time exceeds 3 hours this searching stops and the time at that point is returned.More specifically, 3 hours = 3 * 60 * 60 = 10800 sec. So if this is exceeded we have a time of 10801 sec which gives 180,02 min as shown on the output.The unity check concerns the ratio of the temperatures, so this is independent on the time.The 3hr limit is mainly used to avoid lengthy iterations + anything above 3hrs is unrealistic anyway (the different Eurocode parts mainly give data up until R180)..In order to make it more clear + to account for very extreme cases like R240 two modifications have been done:1. The limit was increased to 5 hrs or 300 min2. A note is printed in case this limit is reached, to inform the user why the fire resistance in that case was set to 300 min (previously the 180,02 min) and to indicate that thus the member is not subjected to failure by fire.For the user's project there is no difference, the unity check remains unchanged..This change was done in R13.0.1007
NWEB-984M28	Crash report Crash while regenerating the document (see esa file + crash report in attachment)	Problem was caused by the table Setup manager. Slight improvement was done in Deve_09 (available in next R_patch). But it does not solve the problem completely. It is better to use Eng. Report

Ticket number(s)	Bug Description	Solution
NWEB-9869V9	User commands not working for other languages TUM: Copy attached *.usercmd into USER directory !! make sure this is the only *.usercmd there !! Set language to German Defined shortcuts does not work. It seems that usercmd works only for English	Solved in Deve_09. Available in R_2013.0.1016
JBES-98BD78	Engineering report: When customer changes language (for example to dutch), then 'Project', 'Author', etc (in header) is not changed to correct language	Solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98BGKE	Crash during analysis Hit calculation. It crashes.	clear temp data (tools>cleaner) before running the analysis and it will run finetested in SEN 2013.0.114
NWEB-98DJHW	DocX64 - damaged data of Engineering report ("Not available items") + Exception during regeneration of picture	Fixed report is atached in the first comment from 7.6.2013. Potentital sources of simila problems sere improved in the source code in Deve_09. Fix available in R_2013.0.1016
NWEB-98EHBR	Crash report	Problem identified and solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98FBFT	Accidental crash	Solved
NDES-98JFJ2	When openinig a project in Scia Engineer 2013 and going to the engineering report, adding something in the engineering report and close this, I will always (for each proejct) receive a crash of the engineering report. Crash report in attachment. Do you know the reason for this?	Solved
NDES-98JFPS	Open the project in attachment and change the free surface load FF1 into "from - to" (see also printscreen). Now this load is not taken into account on panel LP1 anymore. Can you solve this please.	fixed in 13.0.1019
NWEB-98JKAW	Open the project in attachment and go to the document. It is Not possible to change the name of picture in the document	Solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98JHQV	Open the project in attachment and start the calculation. There are several panels with problems. What is the problems with this panels? Why is the transfer of the load not working? The project has been created in Scia Engineer 2010 and here the panels and transfer of the loads was okay.	fixed in 13.0.1019

Ticket number(s)	Bug Description	Solution
JPOL-98LBJ2	Exception in Engineering report. As I "want to help you and send you the report about unexpected exception" I attach the project where I wanted to input 3 live pictures to Engineering report. I stored them in inbox and when I tried to put them into the report content, following reports appeared. No crash.	Thnaks for "want to help you and send you the report about unexpected exception". Problem was identified and solved in Deve_09
JPOL-98LBYJ	Live pictures in Engineering report remain marked after another picture's regeneration. Pictures in the attached project can't up-to-date completely. If I regenerate one, the red exclamation mark disappears. When I do the same for another picture it appears again by the first picture. It's not affecting the output but it is very annyoing for the user. Attached is the error report too. Tested in 2013.0.112 and 2013.0.1001	Not valid pictures with results is "feature" (see comment from LAT). Source of crash identified and fixed
JBES-98LB9J	Engineering report 'mechanische uitvoer' (2013.0.105) If we want to see the internal forces for 2D elementen (UGT combination) for maximal forces, then it sets it back to minimum.	Solved in D_09. Will be available in next R_patch
RMAA-98FG9R	The regular crash of Scia Engineer during check of steel. Tested: 2013.0.112	The crash was caused by the combination of the 2D FEM method together with a thinwalled section. In this case the centerline was discarded.This has been fixed in R 2013.0.1015.Important note: it is required to 'Update' the cross-sections in this project, using the 'Update All' button in the Cross-section Manager.
RMAA-98LCZJ	It is not possible change name at pictures in document. More information in attachments. Tested: 2013.0.112	Solved in Deve_09. Available in next R_patch
NWEB-98LCM2	ISSUE: the descriptions of the load cases (when doing a copy of loads from one LC to another) are wrong. Here we see 'earthquake', but these are wind-loads ?? (see jpg)	fixed in 12.0.1026
NWEB-98LAA3	Crash during results - storeys - detailed results for the combination GZT-EF Attached you will find one crash report and esa file. Problem description: During the refreshing of storey results (detailed results) for the combination GZT-EF SENG crashes. I found following things: 1. the loadcases (Earthquake) have werong linket response spectrum 2. in one loadcase the CQC analysis was selected, but without damping. This all was because of another crash. All these things were fixed, I also exported the structure to another ESA file, deleted masses, earthquake loadcases, etc,	tested in 13.0.1035 ==> OK

Ticket number(s)	Bug Description	Solution
	I also changed the storeys, so that they start at 0,0 but all without success. SENG still crashes. The changes are stored in file "...changed.esa" Try to display storey results there for combination"E"	
JBES-98LGYG	See project in attachment 1. change cross section for a certain element 2. do lineair calculation -> not all free loads generate loads (which makes results wrong, too little deformations (result of 32mm in this project is too little), too little internal forces, ...)	it seems to be fixed in 13.0.1026 (1st patch of R2013)
NWEB-98JJHV	Results:, How are design magnitudes made for classes?, In the project in attachment, you get mxD- (max) = 695 kNm/m., In the excel file, you can see the results., How is it found, because I can't find this moment again by myself (see row 300 in the Excel file), (see also image in ticket)	Results for classes are the extreme value of all members of the class. mxD are not calculated from max mx and max mxy, but it is the extreme value of mxD- values for class members.
RMAA-98LJEV	Accidental crash	Problem identified and solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98MGBN	In attachment 2 crash reports and the project. Crashes in the Engineering Reports	Problem most probably solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98JELC	The problem with document and gallery picture.	Solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98MJJG	Issue load panels (2013.0.112): There seems to be some issue with the load panels (see image). Press calculation and see the message. Why are some beams invalid?	fixed in 13.0.1030
NWEB-98JJWD	Pond Load: Iteration stops at 0mm	a) The solver messages are not related to water accumulation but to the load panels within this project. Please check this with the CAD/Solver Team if needed.b) Water accumulation: The reason for the 0mm deflection is as follows:- The user has defined an envelope combination, for example CO1- The user defines a Pond load and since an envelope combination exists the setting "Use other permanent load" is available.- The user activates this setting and selects CO1- Then the user deleted combination CO1As a result, the "Use other permanent load" is not displayed anymore since there is no more envelope combination, however this setting was still activated.In other words, the water accumulation is refering to a combination which the user deleted. No combination also means no loading and thus zero deformations.In version R2013.0.1026 this setting was modified: in the event that a user deletes a combination which he/she set for water accumulation, the "Use other permanent load" is hidden but also de-activated.That way the standard self weight will be used and correct results are obtained.

Ticket number(s)	Bug Description	Solution
CSCT-98NKFF	I have done the steelcheck by class. I cannot see the important combinations in short output.	1. Please note that the user can freely modify TLX files in the Engineering Report and can thus add/remove any columns from output tables.2. In R 13.0.1028 the 'Case' column was added to the default TLX.
NWEB-98NDSK	Open the project in attachment and look at the deformations of the plate under "Results -> 2D members -> Deformation of nodes" (see also printscreen). I would expect a symmetric result around the c-axis, but this is not the case.	see the attached comment
NWEB-98KKBH	Looks like there is problem in 2D wind generation in the attached project. The direction of wind is wrong, see attached pdf file.	Fixed in next R_patch.
CSCT-98RKRX	Project crashes in 2012 and 2013 if want to see the labels (value) of the load (loadcase "Füllung"). Any idea?	fixed in 13.0.1030
NWEB-98SGK8	Ak vlozim obrazok do stareho dokumentu (verzia 2013), tak sa nazov obrazku vygeneruje automaticky. Aj ked ho pri vkladani do dokumentu zmenim, tah sa do nazvu nepouzije moj text ale ten vygenerovany. Dodatocne sa to tiez neda zmenit.	Solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98RKBF	Click calculation - The solver continues to warn me that LP9 and LP10 have an invalid selection of supporting members and that I should perform the Update Selection command. After reading this, I select each of the LP's and under Actions select "Update Beam/Edge Selection". When I try and run the analysis again, I receive the same warning. Where does it come from & how to solve this?	LP fixed in 13.0.1030
JBES-98TEUC	Click calculation The solver continues to warn me that LP9 and LP10 have an invalid selection of supporting members and that I should perform the Update Selection command. If I just continue with the analysis, the solver crashes and will not continue for both the linear and modal analysis or even just linear.	fixed in 13.0.1030
JPOL-98RGLP	Steel Cold Formed EC-EN: Issue in Sway Setting for Cm coefficients in combined stability check	a) Buckling curve: see also the Advanced Training Steel as well as the Theoretical background. This has already been explain by mail to Lindab:- For the EN 1993-1-1 check the standard curves are used. In case the shape is not defined in Table 6.2 the curve from the cross-section is used.- For the EN 1993-1-3 check, the code specifies 'Other cross-sections' in Table 6.3 as curve c, so for any undefined shape curve c is used.- In Version R2013.1 the overruling in the cross-section will be possible for any section, even those defined within Tables 6.2 and 6.3.b) The Cm modification for Method 2 when using 'Sway' was properly functioning for EN 1993-1-1 but not for EN 1993-1-3. This has been corrected in R 13.0.1028.
NWEB-98TGXP	Mystic problem in new ingenieering-report. There is the filter missed in material-table and if we click on the selection Scia	The property is corrupted. It is necessary to delete this table and insert it again. The "Exception" was replaced by the message box

Ticket number(s)	Bug Description	Solution
	Engineer crashes.	
NWEB-98UCZH	It is not possible to change the title of the pictures in the document	Solved
HWRE-98VDC6	Old Document: 1) Insert a load picture into document => not possible to change name of picture in document	Fix available in R_2013.0.1016
KVEE-98JEAG	POD: Set of error reports from Engineering report, see attachments. Any chance this user will be ever able to use Egnineering report...?	most problems from crash reports were already solved in Deve_09 (will be in next regural R_patch - not in the Hotfix). Other problems must be solved independently with project (see comments from LAT)
NWEB-98VHQQ	Document: If you change the name of the header of a picture (in it's properties) in the document, then it won't change in the document itself (see images) (User also complains that Engineering report becomes unstable for larger projects, but I don't experience any problems with it myself, so we can only fix the old document for this customer)	Solved in D_09. Fix available in R_2013.0.1016
RMAA-98YA6W	Accidental crash	Problem identified and solved in Deve_09. Fix available in R_2013.0.1016
NWEB-98ZANS	Accidental crash	Crash already fixed. Fix will be available in next R_patch
NWEB-98ZHCL	Accidental crash	Fixed
RMAA-98Z9S2	Accidental crash at Engineering report.	Already fixed. You can try to send to the user the EsaEngineeringReport.exe
NWEB-98ZJJ6	Accidental crash	Already fixed. Will be available in next R_patch
NWEB-992FSF	Open project in attachment. Go to document, and try to change the name of the header of the picture. You will see that it is not possible (as shown in the images) We get many complaints about this on support, can you please fix this (even though it is still in the old document) ?	Already solved. Will be available in next R_patch
NWEB-993EZM	We have bug at displaying value of load at free line load. There is 1000x bigger value than properties. Detail in attachements. Tested: 2013.0.122	fixed in 13.0.1030
NWEB-992KQ7	Look at the crash reports in attachments. Crashes when generating the Engineering report. Do you know how this can happen?	Already fixed. Will eb available in next R_patch
JBES-997FRL	REQUEST OF DESIGN FORMS HELP IN PRINT FRIENDLY VERSION	added to attachement, it will be on web soon

Ticket number(s)	Bug Description	Solution
NWEB-994KBG	Engineering report, stability issue: Open project in attachment - send 'printscreen to engineering report' & live image to engineering report several times - load these images in the engineering report several times - move all images to different places in the navigator - then regenerate all -> gives 1 out of 2 to 3 times a crash (see images) (image 'error1' says: Method_xxx gives an exception ! ')	Already solved. Fix will be available in next R_patch
NWEB-992JJN	Detailed storey results - CRASH open attached project - run linear analysis - goto detailed storey results - settings acc. screenshot "Detailed storey result - Combination Seismic - Resulting forces - sorry it is closed.PNG" - refresh ==> CRASH	tested in 13.0.1035 ==> OK
NWEB-998C8V	Opent the project attached. Open the Engineering report. When you have a look at the pictures of the deformation, there remains an exclamation mark. There are 2 pictures of deformations. After regenerating the one with the red exclamation mark, the mark will disappear, but there will be a mark on the second picture. The client also keep having crashes when regenerating the complete Engineering report. He is using the latest patch (122). We tested this, but we don't have such crashes. Anyway, you can find a crash report from the client in attachment. What can be the reason that the red exclamation mark will not disappear on both the deformation pictures? And why does the client keep having crashes after regenerating the Engineering report?	Already solved. Fix will be available in next R_patch
GVAN-99AJKP	ISSUE: (2013) CRASH during calculation when a Damping Group is used (damping acc. to material)., Calculation works when Damping group is not used in the Combi of Mass Groups.	tested in 13.0.1035 ==> OK

Ticket number(s)	Bug Description	Solution
NWEB-99B5M7	Multiple new-document-crashes during refresh. Any idea?	One crash was caused by graphic card ATI. Try to change th esetting fo graphic card or try do not use Anialiasing in picturesOther crash was caused by GUI library - it was fixed in Deve_09 (available in next R_patch)Remaining crashes were caused by problems which were already solved before
NWEB-998DTF	Too many releases for 1D element no.736	tested in 13.0.1035 ==> OK
JPOL-98RJA5	Crash report - ER Feed back to Engineering report with error report. Except some general notes to the new development (see below in Czech) there is an error report attached.	Problem already fixed. Fix will be available in next R_patch
JPOL-99EHSR	Engineering report - big consumption of memory Tried to open it in Deve_04_DocX64_SDI (12.004.1875) --> After opening of ER it crashed Tried to open in R_2013.0.1021 --> After opening of ER it crashed too	First identified problem was too big consumption of memory during opening of ER - solved in Deve_09Second identified problem was damaging of report data when the hard drive was utilized (swapping of memory, creating of TLX cache, ...) - This problem is also already investigated and solved. (fixes included in R_patch)
NWEB-99FDTC	Crash of Engineering report. See attached error report.	Problem already solved. Fix will be available in next R_patch
NWEB-99GELE	In the attached project, Beam S4 (B4) has a very high steel unity check. Reason: Mpl = 0.	Solved in R 13.0.1028For this section the Wpl property from the cross-section could not be correctly located, leading to the unexpected result. In the actual project make sure to 'Update All' cross-sections
NWEB-99HBFR	Look at the printscreen "Loads" in attachment. In this project (also in attachment) in load case BG3 a line load of -288kN/m has been input. This will be shown on the screen as "288000" and not as "288" (so in the properties window you will see 288 kN/m, but in the graphical window it will be shown in N/m). This is very confusing. Can we solve this?	fixed in 13.0.1030
NWEB-999DEW	Look at the following two websites: 1/ http://www.fgg.uni-lj.si/kmk/esdep/master/wg07/l0810.htm#SEC_4 => Figure 6 (bottom right) 2/ http://www.ae.msstate.edu/~masoud/Teaching/SA2/chA14.6_text.html => Last picture (top left figure) As you can see, you will have a triangular shear force in the flange. When making a simple example in Scia Engineer 2012, you will see this triangular shear force (project "2012 Jourawsky.esa" and Image1.png"), but when doing exactly the same in Scia Engineer 2013, you will not see this triangular form (project "2013 Jourawsky.esa" and Image2.png"). Because of this Stef Pillaert's thinks the results were better in Scia Engineer 2012. His students always made a manual calculation of the stresses and in Scia Engineer 2012 the manual calculation and the results in Scia Engineer corresponds very well. This is not the case anymore with Scia Engineer 2013. Can you please take a look at this?	There is in fact a difference between the general "shear stress" obtained through the Jouravsky formula and the "shear flow" which can be found in thin-walled sections.According to Jouravsky, a cut is made perpendicular to the direction being examined and the thickness at that cut is used in the calculation of the shear stress. As a result, for vertical shear, all fibres located at the bottom position of the top flange of an I-section have an equal stress value. Following Jouravsky, this vertical stress is constant in a horizontal cut. I refer to an example shown on one of the websites the user mentionned which illustrates this principle: http://www.ae.msstate.edu/~masoud/Teaching/exp/A14.3_4_ex1.html EN 1993-1-1 article 6.2.6(4) and formula (6.20) also indicate that this is the proposed way to calculate shear stresses..On the other hand, for thin-walled sections, there is the concept of the shear flow. This is the concept shown on the pictures provided by the user. For such thin-walled sections the stress can be determined from the flow by multiplying with the actual element thickness. It's very interesting to note that both approaches (the thin-walled shear flow approach and the 'thick-walled' Jouravsky approach) give quite different results. The main reason is found in the fact that

Ticket number(s)	Bug Description	Solution
		Jouravsky shows only one component while the flow shows both..Since our new 1D FEM calculation for thin-walled elements directly gives us the shear stress based on the shear flow (which can be visualised in the cross-section tabs Sheary and Shearz) we have made the following modification:- For thick-walled sections there is no change, the shear stress is calculated using Jouravsky - For thin-walled sections we now determine the unit fibre stresses from the shear flow in the centerline.The Theoretical Background for the Cross-section calculation provides detailed information on how the shear stress on the centerline is calculated. This document has also been updated to provide information of the fibre mapping i.e. how results from the centerline analysis are mapped into the fibre positions..The usage of the actual signs for the shear and torsion stresses is something which is foreseen for a future modification.
NWEB-99JJ5A	In attachment crash reports of a customer, some of the engineering report, some of Scia Engineer itself. Can you please take a look at this and look if you can find a solution?	Problems reported in crash reports have already been solved. Fixes will be available in next R_patch
POL-99ME5V	Crash report - ER POD: Errors with ER. See attached ea file, error report and error message. It appears when regeneration. Tested in 2013.0.122	Problem already solved. Fix will be available in next R_patch
JPOL-99MGA2	Error in ER version 2013 for user check item. User check is not available in the new ER. Nothing than the header of the check appears. See attached pictures and comparison with the table in Preview. Tested in 2013.0.122	Solved in Deve_09. Available in next R_patch
NWEB-99NHCA	Crash report from Engineering Report.	Problem already solved. Fix will be available in next R_patch
GVAN-99PFYU	ISSUE: (document) when using Scia Engineer in dutch, and when making a new header then some texts are in dutch and some are in english. see screencopy	Problem already solved. Fix will be available in next R_patch
NWEB-99QB33	Look at the first picture in the Engineering report. This is a picture inputted from the gallery. But in the gallery the picture is complete, but in the engineering report the top and bottom of this picture is cut off. Can you please take a look at this?	The size of the picture specified in the picture gallery was ignored in ER. The problem was solved fixed.
NWEB-99RANL	Crash Report	Problem already solved. Fix will be available in next R_patch
NWEB-99YCM7	The problem with function check for update. Why is version 2013.0.112 there? Next problem is description in attachments.	Fixed, version 122 is now on the web