Data version number: 158
Data stored in this version can be opened in version 13.1.61 (External Release 2013.1) and higher.

Ticket number(s)	Bug Description	Solution
NWEB-8TNF5M	There seem to be a number of results within the Wall Benchmark that are reporting quite different values from the published document for AS2 For example, for the results in the bottom left hand corner of element 5, the benchmark determines 65mm2/m of reinforcement, which is just 33.76/(600/1.15). When we look at the detailed results the calculation seems to be showing that the concrete attains a value of strain even though the concrete is in tension. This only allows the steel to achieve a strain of 17.5E-4 and hence a stress of 350N/mm2. This then results in more steel being required.	Processing of the Ticket revealed, indeed, a systematic bug in the calculation of the steel strain in compression.
RMAA-8U5GHC	The bug at drawing generated free load.	known issue, it is planned to refactor it when it is time....for now PBD (proposal for development)
NWEB-8Y3DVJ	Can you have a look at the attachment please. I have carried out some manual check on SCIA output for SLS checks for 0.2 and 0.1mm cracks SCIA seems to be underestimating RE bar in order to satisfy crackwidths I must admit I am struggling with this at the moment and I would appreciate any input to resolve the differences? We are involved a lot in water retaining structures – so crack widths are important. However I am sure that if it was a problem with SCIA, it would have been highlighted already. Maybe my setup is wrong in SCIA?	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)
RMAA-8YN9XR	Please, you could verify cracks for BS.	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)

JBES-94DEH3	Issue 'design As' (DEVE 2012.004.1725) See project in attachment. If you ask the theoretical reinforcement, and then As,user, then you get the correct result (314 mm²). If I remove the free bar and then ask As,user again, then it is still 314, even though there is no more reinforcement present anymore. The amount of reinforcement still remains, even after recalculating the project... Should it not recalculate the reinforcement if something is changed or deleted in the reinforcement?	The problem was solved. It was tested in version 12.006.194 It was tested in version 13.1.1005
JPOL-95CBS5	2D member ULS design issue. What are the white mesh elements without any error or warrning info in the attached project. If you run the ULS member design on CO1 you will get lots of them (see the picture too). I cannot find errors or warnings about these fields. Also the preview window says nothing but 0 values in all columns. Why? 2. Is it possible to change coordinate system (from GCS to CS of element) in 2D Reinforcement function? When some plates are at an angle to the GCS it is annoying to determine value of that angle and then include it in users reinforcement?	The "white areas" in the NEDIM As portrayal (>Przechwytywanie.jpg>) are regions of vanishing (nul) reinforcement. Thus, there is no program error to claim!(1) Explanation: since there is specified no minimum compression reinforcement in the user's calculation, the regions with overall ("elliptic") compression are assigned no (compression) reinforcement (due to relatively minor inner normal forces);(2) If the minimum COMPRESSION reinforcement is specified and defined, as shown in the attachment file >Setup - minimum compression reinforcement< (AM1=0,2%) the "white areas" disappear, as documented by the attachment file >As1- with min compression reinf<;(3) Hint: there are, however, other undesignable elements/areas, such as "exhausted virtual strut" (i.e. undesignability to m/n) or ""exhausted shear virtual strut" (i.e. undesignability to v), as documented by NEDIM on the exit of the design run - see the attachment file >Non-designabilities<
CSCT-984AZW	The thickness of the stiffener at the haunchend is missed (see pdf; 2nd stiffener).	DrawDescription was missing in drawing functions for these types of stiffeners.

RMAA-98L9N2	Scia reinforcement problem there are 2 beams in the model. One beam has stirrups as group other one as exploded group so as single stirrups. Please make single check for shear force capacity for the beam with group of stirrups , check section ad distance 0,117 m – there is no stirrup. capacity X Next please make single check for shear force capacity of the beam with exploded stirrups at distance 0,117 m – there is stirrup. capacity Y So the question is why stirrup exist in case of single bars and don't exist in case of group? The check has to be the same.	The problem was solved and tested in version 12.006.205. The area of shear reinforcement of REDES stirrups and Free bars stirrups will not be completly same, because different algorithm is used for both type of reinforcement.The problem was tested too in version n13.1.1005
NWEB-98SKBA	In the attached file (8 Mb) the solver crashes each time when running a simple linear calculation. It seems to go through the first few iterations and portions of the solver and then I receive an error that states "FE Solver program closed unexpectedly" "	fixed in 13.1.1027
NWEB-98RKBF	JBE -> HER Click calculation The solver continues to warn me that LP JBE -> HER, 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?,	
NWEB-99B5LW	In the attached project, there is a problem with line loads in load cases WIND X and WIND Y When you do a "Check Data" from the calculation menu, you get an error about a line which does not exist HER: line force on 2D member edge is input on member edge which is marked as cutout. the load should be ignored automatically if possible and warning say that some load is not taken in account because it is input on edge of non existing member	

NWEB-998DTE	In this project there are problems with generating the free loads (they still appear in grey?). I changed some properties but I could not solve this. Am I missing something? Thanks!	This is a feature of load panels.A load panel adds up all the loads acting on the panel and produces linear loads representing the summary load. If the linear loads are to be linked to an input load, one of them has to be selected as the referenced input load (there is a single reference from a generated line load to an input load.) and therefore any other input loads are not linked. There is a proposed improvement which can be done in the next version (data version must be increased.)
JPOL-99JCEX	Crash of solver on attached project during modal analysis., Esa file and crash info attached too.	fixed in 13.1.1027
CSCT-99HHF2	Attached a project. We get a singularity in NC19 and NC20. We have created 2 stability combinations. If we have calculated these 2 stability combinations the solver calculate NC19 and NC20, too. 1) We don´t understand that it calculates after stability!? 2) are the results ok?	The structure is unstable, we fixed the issue when it was possible to calculate after stability analysis
NWEB-99MFRN	Please find the attached model with construction stages For load case LC10, the result does not seem OK In this stage some vertical members are added and self weight is considered but if you look at the normal force in member B1190 you can see some tension and compression but under self weight we would only expect some compression there. See also attached scree shot	
RMAA-99UF44	SOLVER CRASH during IRS analysis The problem with solver. The solver always crash at example in attachments.	
RMAA-9ABHB6	SCIA - 3D wind, After making calculation 3D wind disappear., How to solve it? Please see model in attachment.	follow the procedure which Devi describes in the comment. Then fix the value for -Cpi coefficient for 3D wind load cases for 45 and 90 degree (tested on 2013.1)
NWEB-9AHPMJ	AISC HSS: "nominal wall thickness" instead of "design wall thickness" Question: "nominal wall thickness" in stead of "design wall thickness" used in steel pipe design ? Do we use the wrong thicknesses ?	Article B3.12 of AISC 360-05 or B4.2 of AISC 360-10 was never supported within Scia Engineer.In other words, SEN always takes the design wall thickness equal to the nominal wall thickness or thus assumes that submerged arc welding is used.This article can offcourse be implemented but requires new development.
NWEB-9AMELE	What is Plate/shell nonlinearity?, I tried to find what it does, but no result., What I did found out:, 1. It is linked to concrete, because it only becomes available if you have the concrete material ticked on, 2. It does not give a non-linear FEM property to plates or shells.	

NWEB-9AQB3B	File "$001$064.H0l" can not be opened - crash During CDD calculation an error message (see picture) appears. After them SENG crashes. I think it is related to the FEM Analysis of the cross sections and to the earthquake analysis. The original motel from customer: "2013-08-19 Haus 1-2.esa" Simplified model: "2013-08-19 Haus 1-2 simplified.esa"	The problem was solved. It was tested in version 13.1.1005
JPOL-9AYC88	Wind zones are not generated completely on roof panels in attached model. Please take a look at attached picture where you can see that in some load cases (especially in directions 0°, 90° and 180°), there are missing zones F, G, H and J. Tested in 2013.0.1036	fixed in 13.1.1010
NWEB-9ATJX5	In the attched project, if you display the value of As1+ (reinforcement) graphicaly you get 12.43 cm² But if you ask for the preview with Extreme = Global you get a value of 4.7 cm² in the table (see attached screen shot)	It must be solved as a part of development
NWEB-9AYE8H	See project in attachment. When trying to add punching data on N34 or N40, we receive the message that the position of the column is changed to internal (see print screen). Why do we receiving this message? It is not possible to choose a corner position.	In my opinion, the example is incorrect, because the program during FEM calculation gives me warning on the picture.16.10.2013: The problem was solved. It was tested in version 12.006.194The problem was tested too in version 13.1.1005
NWEB-9B7KPH	In the attached model, the construction stage analysis stops with an error about instability, The supports should be ok	
NWEB-9BAMN2	The customer has noticed that in the Concrete Detailing Provisions for IBC the Min. reinf. factor for beams has to be 200 instead of 100? See image in attachment + p. 52 of the pdf.	I think, the value 200 is correct. see attached picture from the ACI 318 code16/10/2013: The problem was solved. It was tested in version 12.006.194The problem was tested too in version 13.1.1005

NWEB-9BFBUY	When calculating the project in attachment, you will receive an error message about several panels. Changing the mesh settings does not solve the problem (I have changed them into "100mm" and into "Automatic" and both options gave the same error message). How can we solve this? When gerating the load manually on the panels (for example panel LP1) by clicking on "Generate loads" below the properties window, the loads are generated.	It is impossible to fix easily. It needs a specification how it should behave to be more user friendly > PBD. The workaround for user is written in comment.
NWEB-9BFE7Z	Open the project attached. By running the hidden calculation (linear and non linear at once), there are no results in the calculation protocol for the linear calculation, there are only results for the non-linear calculation. If you run the 'normal' calculation (linear and non linear at once), there are results in the calculation protocol for the linear calculation and for the non linear calculation. Why is it not possible to show the calculation protocol for the linear calculatoin, after doing a hidden calculation (lin and non lin)?	fixed in 13.1.1010
CSCT-9BFGJ4	On this walls (in the curve) are no loads from 3D-wind-generator. Why? BT1- W6 BT1- W12 BT1- W14 BT1- W19 BT1- W21 BT1- W23	fixed in 13.1.1010
NWEB-9BMB3G	CDD deformations linear, nonlinear and with creeping are equal	The problem was solved. It was tested in version 13.1.1005
RMAA-9BTHLJ	The problem with error report during TDA calculation Error report: 1Dmacro 199 was not found in phases date. I did the check and everything seems OK.	
RMAA-9BUFU2	The bug at check capacity Where is extreme value at extreme check? More in attachments.	The problem was solved and tested in version 12.006.205 and in version 13.1.1035

ISCS-9BWJ2V	I have made 2 projects as example. A plate supported with a subsoil, and another plate supported with line supports. I have made 2 EC combinations, ULS and SLS. When calculating the project, normally the result classes are created automatically. This is OK for the plate with the line supports, we get 3 classes: 1) all ULS, 2) all SLS, 3) all ULS + SLS. When defining a subsoil, there is only when class made: 1) GEO. Why are these classes not created when having a subsoil as support? TRX: The class GEO should be generated only when funtionality pad founadation check is checked, because it is used only for this check. Other result classes should be created by user only. This work correctly on the Plate-NoSubsoil project (use cleaner first and then calculate).	was tested on the last 2013.1, Geo is created only for functionality pad foundation, another class should be created manually
NWEB-9BWG7B	The same in old document. it is "feature" of Concrete setup It seems that the 1D/2D concrete setup items in the engineering report only give metric units, even when the all the projects units are set to imperial (see attached) file and the Engineering Report.	The problem was solved and tested in version 12.006.205 and in version 13.1.1035
JTRK-9BP954	Issue with 1D concrete design for a bit complicated structure. 1. Take attached project Plaza E-zebro 1np_v4_Trubacek.esa and try to run linear analysis, run 2D member ULS design on selected result class and run 1D member design on selected result class. Crash appeared (also attached 2 reports). Why does this crash happen? 2. On another computer (in Brno, can be seen via remote desktop, IP address 192.168.152.18) no crash happened, but no response is noticed for more than 3,5 hours (afterwared I stopped testing). It is also Win8, 64bit, Scia Engineer version is 13.0.1036. What is the cause of long response? Too many linear combinations in CO1? For just one load case the 1D design runs for 40'' and for one linear combination about 55''. IS the limit 1000 linear combinations limiting? Why such a limit exists? Why the design on one selected member runs smoothly even for the result class? 3. The second project Plaza D_ZD 400 + soilin + piloty 500MNm_130915_v4b_Trubacek collapses also during 1D member design (another crash report attached) For more detailed investigation made by the customer and reasons why he refuses to use SEn and purchased FINE see attached doc files.	see last comment form JLEB, we made some speed improvments.

RMAA-9C2KR8	Why is a punching check at nodes N33, N32, N27? The user reinforcement is not inputted on area punching checks. Tested: 2013.0.2027	The problem was solved and tested in version 13.1.1005
NWEB-9BZ5M6	See project in attachment (created for a dynamic calculation). Under 'masses', you can see a number of surface masses for (for example) MG7 (mass group 7). Run the linear calculation (you can set the number of frequencies to 3 or something to calculate faster). Next you will se that all surface masses are deleted from MG7! How can this be?	Workaround for saved projects is known. The patched version prompts the user to confirm unbinding the mass group from the load case or re-binding to another LC, and deletes references between masses in the MG and loads in the LC if the user confirms the prompt.
NWEB-9C3E87	Open project in attachment Calculate it. You get the following message about the load panels (see image load panels.png) The problem arrived on it's own. The panels worked without any problems for some time, and all of a sudden there is this message. So why does Scia Engineer introduce errors in it's own load panels?	
NWEB-9C4PA6	Solver crash during IRS analysis Do modal analysis for project in attachment., It becomes unresponsive., << It seems the only way I can get the model to run is if I turn off the Reduced Model analysis method for dynamics. Thanks. >>,	
JPOL-9C6JML	Misleading information in the Preview window of steel code check. When I place focus of the mouse cursor on some items in the detailed output of steel code check, there is a description at the bottom of the preview window. However it gives wrong information about Fire Resistance Data or Compression check or other. But it is not related to selected row of the table anyhow. See attached pictures. The information is different for each row and does not correspond to the particular table and the code is not correct, too. Tested in 2013.0.1036	

JPOL-9C99KG	Errors in 32bit and 64 bit solver on user PC running 2011 version. Please see attached pictures and try to give some advice as we tried many usual tricks but no have found solution for the user. He is trying to run Scia Engineer 2011 both with 32 and 64 bit solver but neither gives result. He reinstalled the program but with no success. He turned off antivirus with no success. He tried to run the software from unpacked ace folder that we sent him, but with no success. He reinstalled vcredist packs but with no success. Any other idea? What is the Visual C# Command line compiler? See attached pictures	see comment
RMAA-9CGFT5	If you run TDA calculation, the program displays error report: Sytnax error on line 879, MPOS 261 -2.00000. More in attachments Tested: 2013.0.2030 and 2013.1.14	
NWEB-9CD9V2	Look at the project in attachment. I would expect for the calculation of Lb the formula shown in the picture in attachment. So Lb = 15 + 15 + 3 + 3 + 1/2(14,8 +12.5) = 49,65mm But Scia Engineer takes into account only one plate and is given the value of 34,65 mm (= 49,65 - 15) Can we solve this?	
NWEB-9CAQJE	In attached project run Check of structure data - it deletes a few dupliacted nodes. If you then run Connect members/nodes function, 11 nodes appear again and another Check of structure data wants to delete them again. And this goes in circle. Problematic nodes are numbered N3589-N3599. Tested in 2013.0.2030	
LKGZ-9CGG8K	wrong default setting for geometric nonlinearity	

NWEB-9CPL3W	Error Message caused by free load, There is a simple structure with only one slab and 2 free loads. SENG gives error message about wrong loads during linear calculation. The reason for the error message is, that user defined free load which should be calculated from 3 values of the intensity, but all 3 are equal. So we have a workaround = define constant load. But: 1. This should not cause an error mesage, because mathematically it is no problem to interpolate 3 constant values. 2. If SENG gives an error message, then it should write the loadcase and the load name.	fixed in 13.1.1010; warning about incorrect properties of load + calculation is not possible until distribution points are corrected
JBES-9CQH7A	Difference in internal forces in the results menu and in the document or steel code check. See the document 'Check' in the attached project. (Ned is important) The steel code check is added for a certain results class. Also the internal forces for this class are added. Next I created, using the combination key, a linear combination 'Test'. Also these internal forces are added to the document. > position 0.000 > N=-1386.08 If you check the internal forces in the results menu of Scia Engineer for this combination 'test', than the value of N on the position 0.000 is -988.50. Why is there a difference between these results? What result is correct?	
JPOL-9CRAXT	Crash of Engineering report in 2013.0.2030. Please see attached error report.	Problem solved by the customer by the excepion from the antivirus checking
NWEB-9CQL3Y	Bug report in attachment. Explanation of the customer: The program continually crash. Why?	

NWEB-9CRC8N	Open the project in attachment and run the calculation. Look at the reaction forces afterwards. You will see the results of image 1.jpg. Now zoom in on a support. You will see the results of image2.jpg. You can't read the results anymore. Can you solve this? Changing the font type does not solve the problem.	
HWRE-9CSH93	Deformation from loadcase 3 (unsymetric) do not fit to inputed load (symetric). See pictures in document of attached esa file.	duplicate members (S1-S11, S2- S12), use chak structure data for correcting of structure, then the deformation is symetrical
RMAA-9CSJLF	The problem with dumping at 2D elements, You compare results at example in attachments. It seems that 1D elements run OK, but for 2D elements there are always the same results. , Tested:2013.0.2030 a 2013.1.23	tested in D7 12.007.255 ==> OK
CSCT-9CSHJB	CS->ZH, Please open esa-file "...Tamm2-2012"., In 12.0 it calculates and in 2013.0.2030 it stopps in the 3rd NC. Why?	We did not received any bug concerned membranes for long time. Linear calculation of membranes does not any sense and should not be allowed. As regards the run time errors where a fortran error is dispayed, it was probably run in an old version. In the new version we cannot obtain such errors. Concerning the message about singularity, it can be avoided by introducing more incremets. In this particular case at least 30.
NWEB-9CRL4W	Scia Engineer crashes when starting -> we have received the crash report in attachment. I adviced him to try: 1) erase temp folder + 2) erase registry workspace in user folder. Probably this will solve the crash, or not?	Solved by customer by deleting of reigistry key Workspace
NWEB-9CWKNS	Two of our Frensh clients are complaining about how the orthotropy option "One direction works" The problem is when you give the program a cross section to calculate the rigidity in the x direction and a value of height h to calculate the rigidity in y direction, self weight of the blab is not correctly applied. Scia Engineer does not take into account the self weight of cross section but only self weight of additional height "h". In the attached example, total reaction is equal to 1.23kN which corresponds to 0.05*2.5*9.81 so this only additional concrete stiffness. Any improvement is planned about this ? Or can it be fixed as a bug ?	

LSKI-9CXCGN	There is a problem in name selection when you send a table to the Engineering Report. Open the attached project, go to result menu and select Internal forces. Display internanl forces for "Poteau" named selection. Now if you do Table to Engineering report, the table is sent but not with the correct named selection.	Solved in R_2013.1
RMAA-9CXDDJ	We have bug at combination EN ULS (STR/GEO) set B and equation 6.10. More detail in attachments. Tested: 2013.0.2030 and 2013.1.23	
JPOL-9CRBTF	Issue with soilin calcualtion - no results found for the attached project file. Soilin cannot finish., May the problem be in combination of soilin support with flexible supports under walls/columns? Without these flexible supports SEn can give the solution., May the problem be in connection of the foundation slab with walls (connected by internal edge) that reach lower level than the slab?	We have some recomendation to the model - the mesh should be a bit smoother, there are triangular mesh elements which are always a problematic, the wind loading should not be in combination for the soilin calculation, there should be only longtherm loadings, the soilin will automatically support the edges, but this is not correct when there are underground walls, we recommend to add small springs around plates where underground walls are placed. there are also very stiff supports and the plate is broken over them, those supports may be less stiff, so the calculation will be a bit better, if we look on results, it seems that 3rd iteration could be used as result, so user may set the number of iteration to 3 and accept results from 3rd iteration.
DPIS-9CYGTG	Problem with mesh generating. If you disconnect the members, the mesh can be generated. If you connect the elements, the mesh can't be generated because there is a problem with the form of the 2D elements.	the only soution is to manually fix it as it is described in comment. We need t find any user friendly automatic solution for this cases with precision
NWEB-9CVC8M	The problem at displaying calculation protocol in service results and Engineering report	Solved in R_2013.1.It is necessary to recalculate the project
NWEB-9CYKAX	Accidental crashes on user computers. There is a company which complains about crashes "at least couple times per day (the same problem exists on different computers)". No SEn's crash report is created but they have sent us attached information. They also stated that "this problem exists on different computers, different projects and there is no repeatable path to duplicate this error. Sometimes it takes couple hours to reproduce this error but sometimes it happens couple times per hour." They tried some tricks to bypass this issue but... "We’ve updated our SCIA Engineer software to newest version as well as we’ve emptied TEMP folders and checked antivirus software. The problem still exists." Any idea where could be the problem?	Please try to add more info. - See comment from LAT

NWEB-9D5D3C	See explanation in pdf file	The problem was solved. It was tested in version 12.006.216 and in version 13.1.1035
JPOL-9D6CLK	Issue with export of old document or results in Preview into rtf.	Problem solved
NWEB-9D7L3W	By deleting the content of a load case, all combinations that contain this load case, are also deleted. Tested in Scia Engineer 2013.0.2030 and Scia Engineer 2013.1.23. See printscreen attached.	
NWEB-9D7LGK	In the attached project, if you compare results for Total reaction for Self Weight (2814.77 kN) and from Bill of material (18562.7 kg) these do not match, It seems that there is something wrong in the way the phased cross section is considered (see attached screenshot), HER: big difference is caused by haunches, but when I delete them the values are still not the same...	fixed in 13.1.1010
NWEB-9D8EKN	Client opened his project, printed the document (with a wrong output - see pdf). Then he had saved the file en shutted down Scia Engineer. Now, it is not possible anymore to open the file. We get the error that we can try to delete the temp folder, but this is not a solution. Can you restore the file? Thanks.	Opening of damaged projects improved in R_2013.1
NWEB-9D7B2K	The problem at results recalculated bending moment Mz, recal at B6 Why didn´t the program use second order?	In my opinion, incorrect description is attached, therefore I put it to NO bug
JPOL-9D9C2D	Runtime error when changing selection to Named selection Crash fo SEn in the old document. Open attached project, run analysis and go to the document. Refresh it and select "Deformace na prutu" in the content. When you try to switch selection in properties from All to Named selection, crash (as in attached pdf) appears. Crash report is offered afterwards but when I want to save it, SEn closes and no report is saved. Therefore I cannot enclose it as well. (Workaround - when I delete existing named selection and create new one, crash does not appear.)	Solved in R_2013.1. Problem was caused by empty named selection
NWEB-9D7L3X	Crash report - ER	Crash caused by blocking of one file by some antivirus. Improvement done in R_13.1. User should exclude Scia TEMP folder from antivirus testing to prevent those problems. M.
NWEB-9D8J4U	Crash report from crashing ER.	Problem during closing of Eng report. This problem should be reduced in 2nd patch of R_2013.0

NWEB-9D9KPF	Masses which are not bounded to a loadcase disappear after calculation (tested in 13.1.23) See for example project in attachment: MG1 and MG2 are bounded to a loadcase, when you insert masses in MG3 and MG4 and calculate the project, MG3 and MG4 will be empty.	
JPOL-9DCEHZ	Crash of SEn when exporting empty Preview, before any calculation	Solved in R_2013.1
ISCS-9DCDV2	Customer wants to delete his dampers in his project. But it is not possible to select them all by using the filter by properties or the selection command 'sel dg*'? Can you test it? They have all the same properties.	fixed 13.1.1010
NWEB-9D9NU2	Crash report - user cannot start SEn See crash report in attachment. Scia Engineer crashes when trying to create a new project or trying to open a file created before. This student already reinstalled the program, but still the same problem. I already adviced him to delete temp or delete workspace in registry, but no solution. Any advice?	The crash report does not contain any information which can tell us where is the problem. Let the user to try install 2nd patch of R_2013 (it is a student so ther will be no licencing problems)
JPOL-9D8MD4	Issue with a connection of beam to wall. In attached esa file the	see attachement
RCCA-9DCKHZ	Prefab beam	

NWEB-9D9FCP	Problem with generated loads on load panels. Consider LP11. In Load case CP-do-Chargement permanent dome ouvert, LP11 is correctly loaded by the generated loads. In load case EXP-do-Axploitation dome ouvert, LP11 is not loaded with the generated loads. In boith cases, the original load is the same (except the value). Why is this load panel LP11 not loaded correctly in one load case and correctly in another load case? If you select this load panel, and click on generate loads, than the loads are generated. But after recalculation of the project, the generated loads are deleted. Remark: LP11 is not the nly load panel with this problem Tested in Scia Engineer 2013.0 and 2013.1	fixed in 13.1.55
NWEB-9DCNFJ	Problem with non uniform damping:, In the attached project, the default material damping is 4% and some members have 8% damping but the software calculates a final constant damping of 2%	tested in R 13.1.1002 - please run "check structure" to clean the model first
ISCS-9DCKVY	Where do the values given in the legend of the results of seismic detailed come from? What do they mean? I do not see them in the preview? See print screen.	when selecting "Values > deformed mesh", the program shows the separate components in the preview table and the vector sum in the graphical output:graph output = sqrt(Ax^2 + Ay^2 + Az^2)now, for some reason, there is a discrepancy between the table and the drawing in the screenshot: the table shows accelerations and the drawing shows displacement values. Try to change settings and refresh (this behaviour has been improved recently)
NWEB-9D5NTF	The customer has imported a longitudinal reinforcement template (LReinf_T13) with dimensions 1600x1200 (see print screens). There is also modelled a rib with these dimensions, but when trying to choose the template, there is no possibility because it does not appear in the list, why?	I do not completly understand. Could you send me some detailed information or some video

NWEB-9CZJWF	Look at one of the two models in attachment. The connection in the upperbeam is the standard suggestion of Scia Engineer of the position of the bolts. This does not seems to be correct? How will this be calculated? And is the value e1n correctly taken into account? And the other results?	
ISCS-9DEFE5	The customer has a .dwg which he wants to import in Scia Engineer. Since Scia Engineer 2011, Scia Engineer does not recognize the layers anymore (see the pdf for the layers). He only has one layer "0". The customer has also made 2 movies where he explains the difference between the versions 10.1.556 and 11.0.1102 (and higher). (He suspect it has something to do with the fact that the lines in the block are layer 0 and the blocks have the different layers defined).	In DWG file thjere is lots of nested layers in blocks (block layer AAA, in which are entities in layer BBB). During import block have to be broken into basic elements (line, arc, etc) what means the basic elements with their layers are imported (i.e. BBB)
NWEB-9DDJ4S	Calculation of interaction factor for EC ltb check: kzy for member S1 must be smaller in the opinion of the customer. See attchments.	Please review the user's PDF and comment.a) In the comment the user indicates that the section is Class 1/2. This is not the case, the first page clearly shows that at 0m the section is Class 4. Keep in mind the difference between the classification for the section check and the (worst) classification for the stability check.In this specific case, the section check at 2,002m is done for a Class 1 section while the stability check is done for a class 4 section, the worst class over the member.b) In the comment the user indicates that he expects Table B.1 is used. Table B.1 however is only valid for members not susceptible to torsional deformations. In the LTB check it can be seen that this member is susceptible to torsional deformations since the Chi,LT value is not equal to 1,00. As a result Table B.2 is used.In case the user is certain that this member is not subjected to any LTB (the compressed flange is held in place for example), he could modify the LTB length in such a way that Chi,LT = 1,00. In that case Table B.1 will be used.
NWEB-9D7LGH	Look at the Engineering Report and go to the chapter "Resuls" the picture "BMD". It is not possible to see the picture here. The customer has deleted and input the picture again already 3 times. Any idea how to solve this?	I've added again the Inbox item "BMD", regenerated it and its content has appeared correctly. (Tested in 13.1.46)I advice to wait till all tasks from the queue are processed beforre starting of next modifications
NWEB-9DE6SY	Please see the below comments from one of our users. They were having issues accessing the engineering report because of read/write privileges. Are there other locations or portions of Scia that would require read/write privileges to the Program Files? Please let me know.	As far as I know It is necessary to have Read Write access to "Temporary files" folder, "User setting files" folder and "Project files" folder.Then it is necessary to be able to write into directory with licence file (by default in ProgramData\Scia\ subfolder)

JBES-9DFJRZ	American steel code check -> stress increase factor? User wants to put the stress increase increase factor (amod) equal to 1,00 to calculate a blast. Where can it be changed? And if this is not possible, can we then add a setting for it?	The rule where the user refers to comes from the ASD 9th Edition. This code dates to 1989. The format of the interaction equations as it is today has been modified already in 1986 in the LRFD code.In the latest versions of AISC 360-05 and AISC 360-10 this 'new' format is still used (see screenshot). The AISC codes have harmonized the ASD and LRDF design.In other words, the old format which contains this 'increase factor' is not in the latest versions of the AISC anymore.We will not make anymore changes into the old ASD 1989 code, that does not make much sense ... that code is 24 years old.In version R 2013.1 we support the latest AISC 360-10, the user is advised to use that code instead of the old outdated 1989 code.
DPIS-9DFKMW	There seems to be a difference in sorting supports in table in the Engineering Report (also in the old document). If you create a table of the supports, the sorting is done well (see image). If you create a table of the reactions, the sorting is done differently (see image). Here it seems that the sorting is done by the first number of the support. Let's say you have 4 supports: Sn1, Sn2, Sn10 and Sn11. Than the sorting is done like this: Sn1 > Sn10 > Sn11 > Sn2. Can this be changed?	It works fine in Eng report. See attached picture (13.1.46)
RCCA-9DFMTR	Damaged Eng report data - After opening of ER. error message about URL is displayed --> this can be solved by deleting content of the Inbox - There are some pictures (cca half of them) which cannot be regenerated - somehow damaged?	Pictures are somehow damaged. Part of data is missing. It is necessary to create them again. The source of the problem is not known. May be memory problems which sometimes apeared in version 2013.0. It would be helpfull to know how the problem appeared.

JPOL-9DDGB7	Questions about dynamics and nonlinearity in SEn. Please answer questions below. 1. How do we calculate nonlinear analysis with plastic hinges in steel? What is the logic behind (some iterative increase of load up to the My,pl may be)? Is there any theoretical background about this topic? 2. When using IRS in modal analysis what is the stiffness between reduced nodes? Mass is obviously reduced to these nodes (one per floor) but what about stiffness? Is it also reduced? 3. Can we calcualte Karman vibration on chimneys with arbitrary profile? I know that the structure has to be divided into segments - the question is if all segments has to have identical cross-section or if it may be diffenet at the top and at the bottom of chimney. However, each segment would be prismatic. ---	
JPOL-9DGBZD	Crash of SEn when importing XML file. I cannot import attached XML file created in 2013.0.2030 version. Basically I wanted to transfer model data (materials, cross-sections, nodes, beams, slabs...) from 2013 to 2012 version. I created XML file and I tried to import it to 2012.0 (and afterwards to 2013.0.2030 too) but with no succes. I also tried to run Update XML into an existing (but empty) project - again with crash. The bahaviour is not always the same and differs in time and in used version. However, all crash reports and error messages (that appear without crash report) are attached. Sametimes I also have to click 5 or 12 or 20 times on some timber material in the library before it crashes. Please find a way how to import attached XML file (or find some error in the XML - by the way, it was created from project most215Opory2 výp.esa).	crash is caused by incorrect table of steel connection, dialog of materials is fixed in 13.1.1010
NWEB-9DGCZH	Crash report of Engineering Report	Source of the crash is still not clear but we focused our investigation on usecases with printing to PDF.
JBES-9DGD3D	In the project in attachment, you can activate 'anchorage' for the longitudinal practical reinforcement. Let's say I pick the following settings: - Location = both - Type = A - Permanant Code Check = YES If you would enter 'L1[mm]=1mm', then the program would augment it to 340mm. How does Scia Engineer calculate this 340mm?	The value is calculated according to formula 8.3 in EN 1992-1-1, see attached document form SE in last comment

JPOL-9DFKLN	Resultant position of seismic loads. Please explain or confirm presumptions of seismic analysis below. Imagine a simple cantilever loaded by horizontal linear force. Since the columns is fixed, this load gives us horizontal reaction and moment reaction in the support. From these values I can easily calculate resultant of the force applied - its value corresponds to horizontal reaction and position along the cantilever is simply Rx/My (for uniform load in the middle obviously). Now - how does it behacve in case the load case is seismic? I made a test case with 5 cantilevers. Self weight is eliminated and mass is applied in five different ways (see the picture and attached project). When the load is concentrated (in upper node, in the middle or next to the support) position of the resultant is (practically) at the mass position. It can be calculated from Ry and My reactions for LC2. However the position for uniformly distributed mass is not in the middle but at 59% of the length from bottom. Can you explain why? Tested in 2013.0.2030	the resultant force is not in the middle, because it depends not only on the distribution of masses but also on the ditribution of accelerations, and the acceleration is not uniform along the cantilever.the position of the resultant moves upwards because accelerations are higher towards the tip of the cantilever.Simple case: assuming the acceleration varies linearly along the cantilever (more complex in reality), the resultant would be exactly at 2/3 of the length instead of in the middle.
JBES-9DGDCB	Open project in attachment. Do a single check in for the CAPACITY CHECK, and send the output to the engineering report (chose 'insert and close' = directly into the report, not to the inbox). Problem = it won't show properly in the engineering report (2013.1.23)	Sovled in R_2013.1
NDES-9DGF9S	CSS: Wrong Initial shape for FC 123, 124, 125 Create a new profile in Scia Engineer and choose in the cold formed library for a "SADEFSEP 350x4.0D10" and look at the initial shape of this profile (see also printscreen in attachment). Can we solve this?	Fixed in R2013.1.63.The respective Sigma sections again have a correct initial shape.

JBES-9DGESZ	How can we display the As used for punching check in the preview (for the swiss code, SIA)? Example: 1. Open Esa_EC, and look at the preview for the punching check. You will see 4 tables, with in the second table the amount of reinforcement he used for the punching check. 2. Open Esa_SIA, and look at the preview for the punching check. Here you can only see 1 which does not contain the amount of longitudinal reinforcement which was used in the punching check (if you would go to single check, then this information is there, so why can't we add a table for it?) --> If you would adjust the table in the table composer, then there are columns for As1-, As1+, ... in the list, but the longitudinal reinforcement used in the punching check won't show in the table. So, why not, since the information should be loaded and available for the preview!	
JBES-9DGJ6N	Look at the project in attachment. The internal forces on beam for 'Load case = M2', 'Integration strip = yes', 'prestressing = yes'. You will see that My is different for the plates as for the variable beam. So in fact the same post-tensioned beam gives completely different results dependant on the sizes of the plates. Although all of them have the same size. So what is causing these differences, and can we implement a solution so that it will not be necessary to chop the plates up into very tiny pieces?	
NWEB-9CQDEW	Tekla Interface	without particular project we could not find the reason of the problem, either we could not say if it is incorrect setting on the pc or problem in plugin > no bug
NWEB-9DGM9B	Accidental crash of SEn when entering mesh setup. No crash on ym PC. Tested in 2013.0.2030	see comment

JPOL-9DGL64	Self weight of members with cross-section inputed by user using the new css editor is not calculated. Please see attached (calculated) project, go to results and check bill of material. Only 2 out of 10 css are mentioned (standard circular sections). When I change the rest of 8 css to some standard library profile, they are also listed. Compare attached pictures. Tested in 2013.1.23	Solved in R2013.1.1008The self-weight for library sections is always calculated based on the area defined in the Profile Library.In this specific case, the user did not define any values for the area in the Profile Editor and thus the self weight was zero. This was in fact a not covered use case since all sections in the SEN Library at least have an area defined.From now on also for library sections the area from the cross-section manager will be used. Note that in this case it requires an update of the cross-sections..In addition a test was added to the Profile Editor: For any section type it is now required that at least the area 'A' and the inertia 'Iy' are filled in. Both are required for the AutoDesign sorting.
NWEB-9D96SW	Question: If there is some lag with the license server which causes you not to be able to open another design form right after you close one. Then how could you reduce that lag?	It sounds like slow traffic on the network. When check-in / check-out of the licnece there is sent many of very small packets. May be they are delayed by the network
JPOL-9DKH7U	Openings O700 and O744 are not visible, why? Please open attached project and check slabs S532 and S2304. There are openings that are not graphically represented. Only nodes in cornes can be seen. However, these openings can be selected and edited. Project was created in 2011 version but tested also in 2013.1.23	Openings are not directly in the main slab plane, it causes the invisible openings. If they are moved into main slab plane, they are displayd correctly, I propose to use align function for it.
NWEB-9DLDTT	Issue: Open project in attachment. For the construction stages, you can not remove a 2D element in a certain stage. Is this normal?	
NWEB-9DLELT	Set solver setting back so it calculates all non-linear combinations at once. Start calculation and you will get a solver crash...	not reproductible - proceeding as described, no crash. Tested on R13.1.58
ISCS-9DLF3C	See project and print screens in attachment. There is a load panel LP1 which is giving different results when using: - Selection of entities: all -> this looks ok, the outer beams are loaded with 1,80 and the inner 3,60. - Selection of entities: user selection -> we have selected all the beams, so I would expect the same result, but this is not the case (see red rectangular in print screens)	fixed in 13.1.1029

JPOL-9DLJEC	Wrong CZ spelling in the dialog that appears just before new project is opened. Please correct "krouticích" and "charakteristic", as highlighted on picture below.	
NWEB-9DMKA7	Crash report attached	4 possible sources of this problem were identified and solved
LSKI-9DMLT2	Free loads are not generated in the attached model	fix the incorrect geometry of load panel Rdc_1 - see picture in comment
NWEB-9DKE76	Open the project in attachment and first go to "Tools -> Cleaner" and clean everything under "General" (otherwise I don't get results at all) Now calculate the project and go to "Concrete -> 2D member -> Member check - crack control" (see also printscreen in attachment) => Scia Engineer does not find any reinforcement in the subregion. Why not? There is user reinforcement here.	The problem was solved and tested in version 12.006.216 and in version 13.1.1035
RMAA-9DNJQ3	I tried to explain user that his verification isn´t correct, but without success.	According to the theory of 1st order elasticity, the axial force in a 1D memberdepends solely on its axial deformation. Rotational degrees of freedom act strictlyonly on the bending, shear and torsion of the member.The assumption that transverse displacement influence the axial force is typical of the 3rd order theory, also known as "large displacement theory".To take it into account, enable geometric nonlinearity in the project settings and select Newton-Raphson in the solver settings.All links in layer "level 1c" are declared as "axial force only" AND have hingeson one end for all transverse DoF. This is unconsistent: those DoF do not exist inaxial force only members. Use either hinges on those members or set them as axial force only, but never both.Also note, that "press only" members are automatically considered as "axial forceonly" as doing otherwise might lead to unconsistent behaviour of those members(imagine a member that is disabled because of a tension and still carry some bending,this corresponds to a stress state that is unrealistic).Regardless of the unsufficient connections of the links in level 1b and 1c,the structure is unstable when running a geometric nonlinear analysis.Please keep in mind, that the in-plane rotation DoF is NOT connected in 2D finite elements. As a result, the inner ring can rotate freely around the GCS Z-axis.This appears in the geometrical nonlinear analysis as the "tricks" used in the linear analysis cannot be applied to compensate that issue.A solution for that could be adding a 2nd (outer) ring as a thin 1D member along the upper edge of the shell to stabilize the rotational DoF along that edge.Tip: for faster analysis, use the direct solver instead of the iterative one.The iterative solver is more accurate on large systems, but it is hardly relevanton 64bit platforms.

NDES-9DPK5Y	Small remark: Open a project in Scia Engineer and change the width of the properties window and the main tree (see printscreen1.jpg). Click on the icon on the top right (as shown in printscreen 2) and maximize the screen back afterwards. => The width has been changed again to the first (default) width of Scia Engineer. Exactly the same when closing and reopening the program. In the previous versions, the width has been maintained always. Can we arrange this again?	tested on 13.1.55 - the change is saved
NWEB-9DPCZ2	Look at the results for the combination Combi1. I would expect the highest values for the reinforcement As2- as indicated in the red rectangular in my printscreen, based on the loads and on the stresses in the plate. But at the edge of the load, the necessary reinforcement is higher and I don't understand why. Can you explain this?	
NDES-9DSCW4	Printing to pdf, is not okay for landscape pages. Open the Engineering Report of the project in attachment. Choose to PRINT (so not export) this to pdf creator => The landscape page will be exported as a portret page to pdf.	We support only one paper format when printing. You can use export to PDF and then print from Acrobat reader which can hanle printing of documents with multiple formats.Print to PDF is completely different thing than export to PDF
NWEB-9DKKA7	Open the project "Example9 Concrete Frame.esa" and look at the results for the "Concrete -> 1D member -> Member design - Design". When looking at the Single check, I do see some results in the tab "Loads", but no results at all in the other tabs, like for "Stress 3D" no visualisation will be shown. Also, when switching the combo box from “selected section” to “extreme section” the extreme values and locations are not updated when switching the combo box between values (My-, Vz-, N, etc.).	1/ the results in tab-sheets Stres3D and others are not available for member B4, because area aof additional reinforcement is zero2/The problem with "extreme section" was solved in version 13.1.1035.

NWEB-9DPL38	Please find attached esa file and look at edge columns. They have got the same buckling systems but one of them has got additional nodes. When I check member SR02 everything is correct. When I try to use check for member SR05 there's an information that "C section parameters for catilevers" which is not true. The same problem was discovered by the user aprox. one year ago and fixed by scia (he says). This example shows that problem still exist. Tested in 2013.0.61	The decision of a Cantilever is caused here by the geometry.Member SR05 goes from N30 to N48 while the buckling system is fixed in N17 and N33.The cantilever algorithm detects that the system length of this span is 3,85 m and this matches also the length of SR05. It then checks the begin/end of the member and those nodes are free thus leading to the cantilever decision.Any other member in the project like SR02 does not have this kind of double node input, leading to the expected C-factors.Especially since the cross-section of SR04, SR05, SR06 are equal there doesn't seem to be a reason why there are double nodes. It thus seems like this is an input error: nodes N33 and N47 should coincide and nodes N17 and N30 should coincide. Then the expected result will be obtained since then the geometry correctly matches the same case as for SR02.In a future refactoring of the LTB check we will review the algorithm to see if this very special case can be accounted for but at this time it seems like an input error.
NWEB-9DQ6RB	Crash report POD: Error report from SEn when working with the document. Please see attached report and esa file. User says that cleaning temp file, restarting the machine, using different printer (for example xps printer) or engineering report does not fix it. He has the same problem several times. No problem on my PC with 2013.1.60 version.	There are not available all necessary data in the crash report, but it looks like crash in the graphic card. Try to investigate which graphic card user has (ATI usualy makes problems), try to switch window display mode to software emulation (very slow displaying), try to decrease graphic card acceleration and so on, try to use differnt version of graphic card driver
NWEB-9DQG5L	Scaling of model data and results doesn't work when you try to input exact value. It works only when using arrow buttons (up and down). Any value that you want to type appears in command line:-) Please see attached picture demonstration.	problem solved. Fix will be available in next R_patch
JBES-9DSKDP	Go to Engeering report and set the first page to 2 Also add a header/footer. You will see that it says for the first page 2, but the maximum pages has not changed. Example: for an Engineering Report with 2 pages, the first page will be 2/2, and the second page will be 3/2. So the maximum page number must be corrected.	The numbering of pages needs to be more flexible. The more flexible logic must be implemented

NWEB-9DNBU4	Issue with performance of Scia Engineer on different hardware. The user argues that they cannot use SEn 2013 on their rather old (4 years) computers (Win XP, CPU Intel Core 2 Duo E8400 3,00 GHz, 4GB DDR2). On newer hardware configuration it is better, they say, however still not as good as it used to be. Especially slow is analysis, stress displaying and steel code check. Could you explain it somehow? Is there any particular reason for this? Read user's email in CZ below for better understanding of the problem or see attachment with PC configuration.	It seems that problem is with the slow disk access. Try to check the state of HDD on the old Win XP computer (full / fragmented / ...). And both systems make sure that TEMP folder is excluded from the antivirus checking
SROR-9DTCDV	speed up structural optimization (external process using esa_xml)	I do not have ultimate solution, but definitely we can provide some small hints.1) Each opening of a esa file from older version makes a data server content upgrade -> slowdown. That does not need to happen every iteration.So we propose to resave the template in the actual version which is used for the iterations.2) I have noticed that XML contains a really large amount of cross-sections, which are from several reasons slow to update. If the ammount of cross-sections contained in the XML can be decreaset the speed will increase proportionally to that.3) We measured some significan peformance issue in cross-section update due to material filter, I hope we can improve that with a help of steel team. I would ask BRO to have a look at it, maybe we can find some solution.
SROR-9DTCDV	speed up structural optimization (external process using esa_xml)	
NWEB-9DSCLX	At the end of document output of steel base plates, there is a big space for stiffness diagrams who are not there. See attachments.	
JBES-9DTDH4	Issue: instability non linear calculation -> project has been stripped down to file in attachment -> non-linear calculation does not pass and there is also a peak in fi & u after some increments (and also at time of instability message) The problem is strongly linked to the 2nd order effect, but we can not identify what we must change to make it stable.	the linear stability analysis gives a critical load factor of 0.52 for the first combination, which indicates that the structure is clearly too weak to sustain the applied loads.nonlinear stability analysis gives a similar value (0.50) that confirms this results. Looking at the structure modelization, I think that the problems come from the connection of the horizontal bracing to the columns. In reality, the bracing is not connected to the columns, but rather to the lower flange of the beams. The modelization does not reflect this at all. The bracing is connected to the columns, which have a very weak cross-section.The upper part of the columns should be stiffened to represent correctly the real structure. After testing that (using the css of the big beams for the upper part of the columns), it improves slightly the behaviour of the structure,but it is not good enough - by far. IMHO those columns are far too weak. The relative slenderness of the HEB100 columns is around 970 !!! (B1121 and B2263)The structure needs either bracing or stronger columns.

JBES-9DTJXE	Issue: If you open the project in attachment, you can see a load panel and generated loads. The free loads on this load panel are 7 kN/m^2 (or more) over a span of more than 14m. That should give line loads of at least 7 * 14 / 2 = 49kN/m one each side. The generated line loads are 13 to 20 kN/m.... So that's quite some bit too small...	fixed in 13.1.1029
NWEB-9DTG64	Legend for 2D results is not correct in 13.0.2030 and 13.1. See attach	Solved in next R_patch
NWEB-9DUCLX	When closing the project in attachment, the customer receives the crash report from attachment. Any idea why?	Cannot reproduce the crash. Please be more specific about the manipulations.Old version: project last saved in SEN 11.0.1223 - not supported anymore. Try using a more recent version.
NWEB-9DNJ4Q	Look at the project "ND_woning test.esa". In this project I have cleaned up everything that is not important for the problem. When calculating the project, the load on panel LP2 will be generated. But I would expect a linear load of -1,9 at two sides of the panel. This is correct at one side, but the other side is loaded with 2 high loads (see also printscreen "linear.jpg"). Can you please take a look at this?	On the other side there is many short beams which are not exactly in the plane of load panel. If select all beams nodes and the panel nodes and put the z value - 2,83, then it is generated correctly. It is a precision issue. I recoment to change the panel type to Panel load to edges and beams. There is possible to define the tolerances and load is generated.
NWEB-9DTKAN	Solver doesnt start	
NWEB-9DRFCK	Results of linear combination containing one load case doesn't correspond to result of that load case. Please open attached esa file and run analysis. In results (as well as on attached pictures) you see strange difference in sign of results for load case EY-seismic loda (both positive and negative) and for linear combination EY-COMBINATION (only negative). Why? I noticed that the load case has got ticked "predominant mode" possibility. However, even when it is not ticket, results for simple load case are completely positive and for linear combination are completely negative. Why?	The seismic load case is SIGNED (predominant mode) ==> the results returned by the individual load case are signed. No Mystery. Fundamentally, a seismic load case is an ENVELOPE with min and max resutls. NEVER use a seismic load case in a linear combination. This is wrong. The program will return either the min or max result, depending on previous settings, because linear combinations do not allow selecting min or max results. ALWAYS use a seismic load case in an ENVELOPE combination. The will allow the program to compute correctly the full envelope, as well as alternate the sign of individual internal forces components for checks/design (concrete, steel...).And doing so will allow the user to choose between min and max results for the output of all results.
IBES-9DUKMW	Shear reinforcement 1D part	The problem was solved and tested in version 12.ô006.216. Now there is only small change after loading on Non-annex in concrete solver, because that concrete cover from 2.5 cm to 3.5 cm is changed and it is OKThe problem was tested in version 13.1.1035.

NWEB-9DTDT2	User claims that filter for css doesn't work i 2013.1 like it used to be in 2013 version. Please see attached picture or short video. Tested in 2013.1.61	Fixed in the latest Setup, a backslash was missing in the registry path
NWEB-9DUJ4J	unstable structure - info about the result?	see comment
NWEB-9DUHBL	open the project in attachment and go to "Steel -> Connections -> Frame bolted/welded strong axis" and created a new connection between the two beams S122 en S200. => Scia Engineer will crash	
NWEB-9DUKP5	open the project in attachment and go to "Steel -> Connections -> Frame bolted/welded strong axis" and create a new connection between column S13 and beam S15. So select those two beams and deselect the beams perpendicular on the connection and click on escape to input the connection => Scia Engineer will crash (see also printscreen in attachment)	
LKGZ-9DUKCT	Not possible to start SE 2013 via remote desktop. According the client, this was possible with his 2010 installation. Are there some changes in the protection?	It was disabled to use standalone protection under the remote desktop. Users who wants to use Remote desktop needs to have Floating licence
NWEB-9DUJ4M	In the attached project, there is a small problem with load panels. There is one load panel, loaded members > selection by user. If you use this option, and select all the members, it is possible to select all the members. If you again click on 'border/member selection', only some parts of the members are shown as selected. However, all the members are loaded correctly (generated loads). So this is just a small problem, but can be confusing for the customer.	incorrect technical design - the same select function is for 1 D and 2D members. For 2D members is necessary to select part by part (part = edge). As a result of using the same select function only first part of beam with more spans is highlighted. This is a feature which I also complain for. Unfortunately it would need a huge development/refactoring -> PBD. For displaying of selected element there is an option in View parameter setting (Display linked members), which display correctly all parts of beams.
ISCS-9DVLCH	Question: Do you maybe know when the reference guide in Scia Engineer will be updated? Because we have customers who are following the help, but it is not always correct anymore. Also, there are now more options, which are not yet explained in the help (and it would be a lot easier if we can refer to the refecerence guide in the help).	
NWEB-9DUCZF	Crash report in attachment. The crash happened when closing the file.	Several possible sources of this problem were identified and solved

NWEB-9DNG69	Question or Issue: Go to the solver setting, and activate the 'Reduced model' and do the modal calculation. How can we solve the message that appears during calculation: 'IRS modal reduction: Detect of nonassociated r-node' ? (2013.1.61)	That error message occurs when no mass is attached to a reduction point, which is not allowed in IRS analysis.Either remove the reduction point (i.e. remove the corresponding storey)or make sure that non-zero mass is attached to it. This can occur, for instance, when 1D members are not subdivided and some storey does not contain any slab nor beam (typically the base level of a frame structure).It can happen in such case, that all the 1D elements of the bottom storey are assigned to the 2nd reduction point, leaving nothing attached to the 1st one.In that case, subdividing the 1D members usually fixes the problem. In very simple structures, with only 1 column axis (typical in simple models used for testing purposes), rotational mass components are not provided by the structural elements and must be fed using manually input nodal masses.
NWEB-9DWCMN	Since 2013.1, the icons of print data and print pictures are grey, when the old style document is not checked in functionalities. So, it is also not possible to send data or pictures to the Engineering Report. This is not practical when customer uses the Engineering Report. Is this a bug?	Solved in next R_patch
NWEB-9DTDEK	The problem with column B136 a My recal by combination SGN. The condition is if M0e,z < M01,z => Mz recal = M01,z does not run at beam B136.	Fixed
NWEB-9DWEZ8	Clipboard problem in ER The engineering report crashes each time the cliet tries to open the report Version used : 2013.1.61	From the crash report it seems that the problem is caused by the very big data in the Clipboard. The problem should be gone after restart of computer or after cleaning of the ClipboardThe improvement of working with the big clipboard data is done in next R_patch
RMAA-9DZGX2	I sent reaction by user on ticket attechements.	Consider a horizontal plate subjected to in-plane deformations. Due to numerical compatibility issues, it is not possible to define the rotation around the vertical axis in a particular point.Because of shear deformations, that rotation varies, depending on the direction considered to measure the rotation (imagine we are looking at therotation around the Z-axis of a small segment placed in the X-direction, then do the same with a small segment placed in the Y-direction: the measured rotationwill not be the same, because of the shear deformation of the plate).Connecting the in-plane rotation to some unique degree of freedom in a point would imply that all possible segments in that points have the same value of rotation around the Z-axis, hence the shear deformation in that point would be zero, and so would be the shear stress.This is known as shear-locking. That's why finite elements usually do not connect the rotation aroung their normal axis and it is not possible to transmit in-plane rotation to a plate just in one point. Instead, a segment must be explicitly modelized to transmit the rotation to the plateas a couple of translations. This is precisely what I suggested to do by adding a beam alont the circumference of the shell. The bending stiffness of thatbeam should be sufficient to transmit the rotation to the plate as described.

NWEB-9DZCML	The client sometimes gets the attached error when he calculates the project.	fixed in 13.1.1027
NWEB-9DNG68	I have been looking at the system and SCIA does not appear to compute an increase in tensile stress with an increase in load on the member/truss. We have created a simplistic physical model and as the load on the truss is increased and it begins to deflect there is more tension in the PT cables.	
NWEB-9DSLUQ	Importing into Scia Engineer 2013 I wonder if you could help me, I have imported a model into Scia Engineer 2013 using the file format IFC 2x3 from Autodesk Revit, I have attached the saved model after my import. Please could you give me some advice as to how it is best to connect the members up within Scia, as you can see all the members do not intersect and therefore creates abnormalities when the analysis is run, Is this something that can be solved using Scia Engineer whilst keeping the correct positioning and setting out of elements, i.e. a tool that can connect members together? or does an analytical model need to be set within Autodesk Revit first before importing into Scia? And therefore when its brought into Scia all members are correctly connected.	use BIM toolbox for creating analytic model - see tutorial in attachment. TRX: The structure imported by IFC cannot be prepared better in Revit, you have to use the Align tool which is devleped by the CAD team. Im escalating this issue to CAD team PDE.

NWEB-9DTCZH	See project in attachment. When using the wind generator on a construction with a sloping roof or a construction with a sloping roof and a part flat roof, we get different pressure coefficients for the sloping roofs. Why? What indicates that these coefficients would be different?	Sloping roof with part of flat roof is identified as a multipitch roof, not duopitch. this causes the diffent coeficients
NWEB-9DZE7M	In the modal analysis we get the warning message "Number of the nonzero numbers in the mass vector should be at least twice as bigas it is". Normally, a solution is to increase the number of tiles of 1D element (or decrease the number of eigenfrequencies) but this is not a solution. What are we doing wrong?	the only used material in that structure has a zero density, which means that the only mass in the analysis is the nodal mass input in the middle of the structure.Because of that and the fact that all rotational components of the nodal mass are zero, this system has only 3 eigenmodes, no more, no less.Asking for 5 eigenmodes is physically impossible with that configuration. Additionnally, there are limitation on the number of eigenvalues that can be calculated, depending on the selected method.With Lanczos and the subspace iteration, it is possible to obtain only half of the possible eigenmodes, this to ensure convergence and good quality of the results. Hence it is possible to obtain ONLY 1 eigenmode with that dataset.With the ICG iteration, more values can be obtained, but it is significantly slower and, in this particular case, becomes unstable for higher order modes.The Lanczos method does not allow computing only 1 mode in this case, making it impossible to use with that input without changing it. To simplify it all, I'd recommend to assign a small - but NON ZERO - density to the material of the structure (e.g. 1 kg/m3). This will allow for more eigenmodes and make the entire calculation much smoother.Use however subspace iteration rather than Lanczos in this case and do not request more than 3 eigenmodes, as the higher order values are irrelevant anyway.
RCCA-9DWR6R	Issue:As 2D member	I compare the results for 1D and 2d member and the results are similar, see attached pictures
RCCA-9DWQYE	Issue: Check response	
RCCA-9DZGNA	Issue: Asw column	
RCCA-9DWQSM	Question: In some calculation cases with wind, the column will have envelope result where there will be values in both directions for My, for instance (see picture in attachment) How will SDF handle with this situation? Because now I don’t have any answer from SDF when I use a class with this situation.	Couly you send me example in SEN to this problem
ISCS-9E3E8M	When printing a picture, you can choose in the printer setup for 'portrait' or 'landscape'. When choosing landscape, the format stays defined as portrait.	the limitation is added to the webhelp, the paper is defined by the template, so portrait has no effect

NWEB-9E2MND	inconsistent behaviour in nonlinear analysis IS > SRo, See original project in attachment., 1) When calculating this model nonlinear, there are no problems., When changing some hinges (see also printscreens) we get an instability in the nonlinear calculation (calculation stops at nonlinear combination 2)., The customer is convinced the structure is stable. Why does Scia Engineer says it is unstable?, 2) So, we have made a stability combination and run the stability calculation., Afterwards, the nonlinear calculation gives no problem anymore. Why do we not get a singularity this time?, See adapted project in attachment "Project Nghia stabiliteit"., 3) When looking at the results, in only tension members, we now get pressure., So the customer does not trust the results anymore., What is happening here? ,	
RMAA-9E3G25	The different value at check of capacity B1, combination CO1 Beam check = 0,78 Single check =0,92	The problem was solved. It was tested in version 12.006.216 and in version 13.1.1035
RMAA-9E3C3D	Tested: 2013.0.2030 and 2013.1.61 Error report during - Incorect line load Line no.2605 Error report during calculation : Incorrect line load Line no.2605 I found that problem is somewhere at masses. If I create only masses by self weight so the calculation runs without problem. HER: incorrect export of mass - see *_upr1.esa. Line 1 is 6,15m but the mass is exported ABS on line 1 with position 11 and 14, what is incorrect. The mass position is according to whole upper edge, which is not line 1, line 1 is only part of it.	fixed in 13.1.1029

DPIS-9DVLVF	Issue: model in attachment can not pass the non-linear calculation - changed all elements with 'type' = column to HEB500 - put 'tension only to all elements which have 'normal forces only' --> project still becomes instable... (see columns_HEB500...) and see image What troubles me more, are these peaks which I can not explain (even not at low loads)	the instabilities come from bending of the beams in the horizontal plane.This is irrealistic, as there is most certainly some light floor on the platform, holding all the beams and avoiding any in-plane blucking of the beams.Please note, that in such structures the safety margin is usually quite low and modelizing the connections as perfect hinges is too conservative. Using some small stiffness value on all hinges of the platform helps avoiding unrealistic local buckling (e.g. 0.05 MNm/rad) (instead of just "free" rotations). Also note:- 20 subdivisions on 1D members is useless on such a structure. 4 is good enough and makes the analysis much faster- use Newton-Raphson instead of modified Newton-Raphson, that also improves the convergence in this case
NWEB-9DWKQJ	See project in attachment. Problem with generating the loads from LP27: - on member S750 everything is like expected, there is a load of 0,55 kN/m - on member S476 it looks like this load is divided by 2 -> we see 2 times the load of 0,28 kN/m Why is this going wrong? We get this result when using the user selection. When selection is set to "all", we have the load of 0,55 kN/m.	fixed in 13.1.1029
HWRE-9E3LLB	Line loads into engineering report and regenerate document takes long time (about 1:40 minutes on my computer). See attached project.	Time of regeneration is comparable with old document. There is 9070 line loads. It means 309 pages of report. On my computer it took 1:05 min
JPOL-9E4AX5	Error reports from Engineering report. See attached zip files. Used version is 2013.1.61	Crash reports does not contain any relevant information about cause of the crash. It starts somewhere in the Microsoft dlls. Is the problem somehow reporoducible?
IBES-9E4JFK	Problem with letters and numbers in ER exported into PDF	Solved in next R_patch. Quality of numbers can be influenced by the DPI setting (see attached pictures)
NWEB-9E56RC	Crash in IRS analysis	
NWEB-9DZJHQ	In the properties of concrete, you have an option 'silica fume'. What is the difference in results when checking this option? Customer expects a difference (of 1MPa) in stresses but we do not see any? See print screen.	

NWEB-9DVLVA	Steel: Crash of check when using 1D Member openings with 2D FEM	The member in question concerns a cold-formed section. The cold-formed check is only valid for uniform members, so for arbitrary members automatically the standard check is used.In this specific case, the 1D member openings were not properly detected as arbitrary, thus the cold-formed check was run leading to the crash (the sections within the openings are thick-walled, have no initial shape, no warping data etc).This was fixed in R13.1.1023. Members with openings are now correctly detected as arbitrary so the cold-formed check is not run and the standard check is run instead.
NWEB-9DSJHP	The customer has noticed that in the Concrete Detailing Provisions for IBC the Min. reinf. factor for beams has to be different. See image in attachment. The default should be x = 3,0 according to ACI 318-08 10.5.1	The second valus (value 200) is corrected in new version 2013.1. The problem with value x =3.0 is caused by rounding off, because the formulas are implemented according to metric formatIf we want to have the presice coefficient according to metric and imperialy system, it means that two codes for concrete has to be implementd for IBC code. I put this problem to PBD
NWEB-9DUKP7	Look at model in attachment. For member S1 you will see a moment Mz and a moment Mx generated by the load case BG3. These should not be there, since the line load and support are supposed to be in the center of gravity of the composite cross section.	
NWEB-9DNFD8	comparison RSM vs time history	A dynamic load function is a combination of 2 functions. In this case, Function1 is the seismic force, Function2 is zero.Here the composed function is calculated as the product of Function1 and Function2, hence all dynamic actions are identically zero.Please change the composition rule: use "sum" instead of "multiply" Please compact project before sending it to support: delete results, remove any document unless related to the issue, delete dynamic time history results load cases, possibly export to a new project.In this case, the cleaned up project data weighs only 226 KB, i.e. roughly 2'000 times less than one piece of the received data.
NWEB-9E3DT3	poutre mixte dans scia engineer	
NWEB-9E4MNC	Can you tell me how the rotational stiffness Rx and Ry of a column support is calculated?, Thanks in advance.,	The rotational stiffnesses Rx, Ry of the support means the moments Mx, My which cause rotation Fix, Fiy by 1 radian.
JPOL-9E5FNB	Secondary effects on prestressed structure. Please check attached esa file and see internal forces on beams B10092-4 (current selection) from LC8-	
JPOL-9E4LGA	Snow and wind load generators on a bit complicated timber roof.	First issue - frames cannot be in one plane. You can move frames or copy snow load on those framesSecond issues solved in comming R_patch

JBES-9E8C3L	Issue: bad strings for French language -> see word file (if needed re-assign to person who can change the strings)	strings that could be adapted have been, but several of the mentioned texts are used in multiple contexts and cannot be modified without causing inconsistencies."Unity check" remained untouched, as it is practically a named feature of SEN
JBES-9E8E2U	Issue: the Reliatbility Class coefficients are not correct for the Belgium annex - look at NBN-EN-1990, page 5, table A1.2 (B) - if I take one value as example: gamma_Q calculated by Scia Engineer for RC3 would be 1,1 * 1,5 = 1,65 while the table in the national annex says the correct answer is 1,8. So we should change the coefficients to properly match the national annex	please review the TB for EN 1990 (see attachment), it shows the coefficients used in SEN.Specifically for a Gamma,Q for RC3 we indeed use 1,20 (Set B combination) leading to 1,2 * 1,5 = 1,8. So if you get 1,65 it means that you either have a Set C combination or are not using the default coefficients..Note that within SEN we use the standard implementation for KFI i.e. as a multiplication factor.The Belgian NA is doing something more here, they are using different coefficients for G and Q and also different depending on the class.For example, for G they use 0,88888 1,00 and 1,11111 while for Q they use 0,86666 1,00 and 1,2Since in SEN we use the standard logic of one KFI we set 0,9 1,00 and 1,2 as defaults as shown in the TB.
JBES-9E8ELR	Issue deflection: Different deflection for 'Beam + internal edge' vs 'Rib' -> look at plate E4. same cross sections underneath, but one is 'beam + internal edge' and the other is a 'rib'. Then I do the CDD calculation with only symmetrical loads, and with the same reinforcment in both elements, but the final deformation is not symmetrical for plate E4. So where does this go wrong, or what is it that we must take into account when calculating a plate with ribs in CDD?	The CDD deformation are not same, because - internal forces with influence of the rib for the rib and beam are not same, see picture 1 - the area of required reinforcement is not same too, see picture 2
JBES-9E8FLR	Look at the detailled preview for the theoretical reinforcement ULS+SLS. There seems to be a wrong string in the preview (see image)	
NWEB-9E5JJ9	CDD calculation is not applicable with IBC Code but "Concrete combinations" menu is still available when you chose IBC code in a project. That menu should be hidden for IBC users.	The problem was tested in version 13.1.1035

JPOL-9E8J7A	Altering stiffness of steel connections. Please see attached esa file and explaint changes in connection stiffness. 1. Caluculate the project and see properties of hinges at the bottom of columns (also picture one), one of them has 0,00 stiffness the other one 2,12 MNm/rad 2. When you delete automatically created hinges, untick "update stiffness" option in the two adjacent connections (Conn4 and Conn6) adn calculate again, both hinges has got non-zero stiuffness (picture two) 3. If you then tick "update stiffness" again and re-calcualte again, the two hinges are now both non-zero, with different stiffness from the original state (picture 3) Why? Tested in 2013.1.61	The zero value occurs whan you update the stiffness of a connection without first determining its stiffness.As simple illustration: - Calculate the project without hinges and without the 'update stiffness' activated.- Now refresh Conn4 so it's stiffness is calculated and then activate 'update stiffness' for this Conn4. This action has purged the results. - Now select Conn6, don't refresh and activate 'update stiffness'. Since there are no more results and Conn6 was not calculated there is no stiffness and thus a hinge with zero value will be generated.- From now on you will get zero moments at Conn6 due to the fact that there is a 'real' hinge there i.e. zero stiffness.The correct procedure would thus be to, after calculation, first refresh both connections, then select both and in the property window activate 'Update stiffness' so that it's activated for both in one go.Note: This is something specific for baseplates since a baseplate stiffness depends on the actual MEd value and can therefore become zero.
NWEB-9E6DF8	Question about redistribution of bending moments	The method for redistribution according to EN 1992-1-1, 5.5.4 can not be used because conditions for this method si not fulfilled, see attached picture. Only user input or moment resistance method can be used in this case
JBES-9E9BXL	Can we add a numeration to indicate the 'version' of the document? For example: if I make a document, and I have to change it a bit later on, that I can indicate that the document has been revised 1 time? --> So.... to be complete, you should be able to add a table or something which indicates which changes has been made in each revision and the date of each revision. (like you can see in our ticket system for the history of each ticket)	This must be investigated and developed. Honeslty I do not expect soon solution unless there is pressure from more customers.Curently user can use e.g. formatted text for manual revision controll or copy-paste his own table e.g. from Excel
GVAN-9E9DAJ	Question: Engineering report table of named selections + indented tables -> does not work table of named selections + indented pictures -> does not work What is the reason ?	Reason is simple: It has not been yet developed :-)ESR with this demand already exists but there has not been time yet to develope it
JBES-9E9ENF	For PNL calculation, we take into account the cracked cross-section, but we don't take into account creep. If we could add creep to PNL, then it could be used to calculate concrete walls as combination of beams (you can check a project with PNL in the project in attachment) If this is not possible or already planned for one of the next releases, can you put it to PBD then?	The creep for PNL calculation can be taken into account only by mutiplying the strain in stress-strain diagram according to chapter 5.8.6(4) in EN 1992-1-1

NWEB-9E3E86	Error Message for aliminium check In this project we have ALU members on a 'Structural only' layer. When executing the check a warning message is given (which is OK) but the string of the message refers to a load case which leads to confusion for the user. .	Solved in R 13.01.1023: The warning message has been modified.
JBES-9E9CAE	What possibilities or examples do we have concerning 15% redistribution of moments? (corresponding to part 5.5 from Eurocode 1992-1-1 -> see pdf in attachment) If this is not possible or not already planned for one of the next releases, can you put it to PBD then?	The possibility of redistribution according to EN 1992-1-1 is described in the attached manual
JBES-9E9DV2	For shear reinforcement, we now support types C and D, but can we also forsee that shear reinforcement can be placed as in the image in attachment (you can check a project with 1D-member shear reinforcement in the project in attachment) If this is not possible or already planned for one of the next releases, can you put it to PBD then?	Installed new version -> new stirrup anchorage already possible
JBES-9E9DZT	For a CDD calculation, it is not possible to take into account which non-linearities are activated. Can we implement a non-linear CDD calculation (so using non-linear calculation in the intermediate steps) (you can check a project prepared for CDD in attachment) If this is not possible or already planned for one of the next releases, can you put it to PBD then?	The new CDD calculation with new solver link is planned, but i do not know in which version.
JBES-9E9GTT	Question: Can we determine non-linear creep according to 3.1.4 (4) for columns? (you can see this article in attachment) Because I couldn't really find the formula we use in the theoretical background. All I could find was: "Creep may be taken into account according to according to EN 1992-1-1, clause 5.8.6"	The creep in SEN is calculated only according to annex B1 in EN 1992-1-1. If the user wants to use non-linear creep coef. or calculation of creep coefficientt according to another formulas, he has to calculate manually and input as user value in concrete setup or concrete member data
JBES-9E9H6V	Factor for size can not be inputted manually anymore (see image)	both scales works on my comp - 13.1.1007
JPOL-9E9ATE	Accidetnal crash of ER, please see attached error report	Similar problems were already solved. Fix will be available in next R_patch
JBES-9E9HKG	How is the average in nodes calculated for 'location = average in node' ? (what theory or formula is used?)	

JBES-9E9J7G	Is there an option to let the size of the gallery editor (where you edit the pictures yourself) always be full screen? (this used to be automatically full screen in older version, but not anymore in newer version)	This problem is causes by some problems with rights. The size of the window is saved to the registers, but some firewalls or antiviruses may restrict this, then this fucntionality is not available. Please check the firewall and antivirus settings. The problem is not on the Scia side.
JBES-9E9GTT	Question: Can we determine non-linear creep according to 3.1.4 (4) for columns? (you can see this article in attachment) Because I couldn't really find the formula we use in the theoretical background. All I could find was: "Creep may be taken into account according to according to EN 1992-1-1, clause 5.8.6"	The creep coefficinet in SEN is calculated only according to annex B in EN 1992-1-1. It the user wants to use non-linear creep coefficient , he has to calculate manually and input as user value in concrete setup or concrete member data
NWEB-9E9EZ8	I cannot open this .esa file in 2013.1.64 !! However, I can open it in 2013.0.2030. What is wrong here ?	Problem solved in comming R_patch
NWEB-9E8DDW	Open project 'test.esa' Chose 'Test of input data' in calculation for project in attachment -> error message (see image in attachment) about the load panels (tested in 2013.1.61) How can we solve this error, or provide a workaround?	fixed in 13.1.1027
NWEB-9E9D2Q	Remarks about timber code check. I keep the german text from checking instance in the attachment.	Festigkeitswerte: As indicated in the TB: EN 1995-1-1 is refering to the following product standards:- EN 338 for structural timber- EN 1194 for Glulam.The material data within Scia Engineer has thus been taken from the latest versions of those codes and is fully up to date. There is no reference to a DIN EN 14080.Note that this library is completely open i.e; if the user wishes to use any other material characteristics according the publications he mentioned he can freely do so and even save this in a db4 for future usage..2) Nachweis des Holzes auf Druck quer zur Faser: a) kc,90: The user did not supply any project so I cannot comment on his actual case. However, please review EN 1995-1-1 and also the TB: the Eurocode (Amandment A1) is quite strict on its conditions.As an example, let's use the 1,75 value: this is only valid in case l1 >= 2*h and when Glulam timber with l ≤ 400 mm is used. So if the user's project doesn't match those conditions then the 1,75 cannot be used.b) Effektive Auflagerlänge: This is a clear input error. Using member data the user can input 30mm at the ends and 60 mm in the middle. In case he did not input member data but just used the standard Setup value then indeed 30 mm is used everywhere. I again refer to the manual and TB.c) The option to not execute this check is on our list for future improvements."Solange ich SCIA benutze ist er in verschiedenen Versionen verschieden falsch." On first sight all of the above items are related to incorrect inputs, so far I do not see any issue in the calculation of Scia Engineer...
NWEB-9E9E87	Open project 'test' -> select connection. I can place maximally 8 bolt rows in Assembl [N2] (on the B157 side). How can I add a 9th (without going to a smaller diametre)?	The # of possible bolt rows depends on the wrench diameter set for the bolts. Decrease this and you can fit more rows however be careful since that most likely means they cannot be tightened in reality since you can't fit the wrench.

JPOL-9EABLE	Unity check of B4 comes from nowhere. In attached project there is a column B4 which has got unity check of 0,85. However this value cannot be found anywhere in the detailed check (see attachment too). Neither re-calculation, nor usage of cleaner helps. Only when I export the file to new project I see new value of UC = 0,28, which is, on the contrary, lower than maximum UC in the current detailed output. So what is the maximum unity check? Where doses this value come from? Tested in 2013.1.64	As discussed: please review the PDF file, the check of the batten shows a UC of 0.85 for the normal stress.
JBES-9EABLG	Is there a way to ask the forces in a cross-link? If not, can we develop something so we can see these forces?	
NWEB-9E8B2X	Loading cpt data from DINO doesn't work well, use of map is not working correctly. The client has updated his license with the module Pile design. He wants to import from Dino a CPT profile, but the browser displays an error message and the image with the map is locked. We cannot move to the left, right… Please see attached video. The same problem appears on my PC with SEn 2013.1.64	it seems that it was fixed on the server (tested 14.1.2013)
JBES-9E9FE9	Proposal: Can we make it possible to put a load on a plate that has to be calculated by a checker patern, to let Scia Engineer search for the enveloppe forces (due to the checker pattern calculation) automatically? If this is not possible or not already planned for one of the next releases, can you put it to PBD then?	not possible > PBD
NWEB-9E8J4M	Accidental crash of SEn during analysis. Please see attached crash report, no other info is provided by the user.	There is nothing specific in crash report, the crash is accidental and it is not possible to find a way how to fix it.
NWEB-9EADEK	Additional info to bug with offsets Info: additional information to devtrack ticket GVA13-3519E9LRW	Solved in comming R_patch

JPOL-9EAEWM	Timber: No check for specific NL combinations	Fixed in R 13.1.1035The issue was caused by the indexes of the NL combinations. Most likely the user has deleted some combinations and then added new ones, causing the indexes to have a non-logical order. The Results API was modified so it can handle such ordering of combination ID's.
NWEB-9E9G6U	This prospect can not start a new project in Scia Engineer. Whenever he tries, Scia Engineer crashes. The error report is attached.	Problem caused by crash in MSXML. Try to reisntall it directly from MS web page http://www.microsoft.com/en-us/download/details.aspx?id=15697. Vlada will investigate whether it is possible to make the setup of it more robust.
NWEB-9DZLGT	Look at engineering report, p16 (chapter 30). The steel code check gives the value 4,34. But the steel code check passes completely! So why does it give the value 4,34?	Looking in the correspondence I see Nele already formulated the correct answer:"Voor de controle op stabiliteit hier worden er 2 eisen gesteld- Controle op slankheid (wordt altijd uitgevoerd en is hier bepalend-- Controle op weerstand (hoeft niet uitgevoerd te worden als de normaalkracht of slankheid klein is)".Please review the user's PDF once more (see also the screenshot).As Nele indicated, when using an LTA buckling system there are in fact two separate checks which are done:1. A slenderness check2. A buckling checkIn this example the normal force is such that the buckling check doesn't have to be done, so that check is skipped. However the slenderness check is still being done since this has nothing to do with the buckling check.With a slenderness of 521.22 and a limit of 120 this leads to a UC of 4,34.This value is also clearly printed on the table (see screenshot), so the result is correct. The buckling check is ignored, not the slenderness check.
NWEB-9E2C93	One of our clients is annoyed by the amount of digits that he sees when he looks at for example coordinates (image). Can we set it by default that the cursor is completely at the left side when you tick in a box? Because he complains that you can only see that last digit, so setting the default location to right, would solve this	See the explanation in the comment.SHORT:Cursor at the end is windows standard. Violating that would make much more users annoyed because it would be inconsistent.
NWEB-9EAHSH	Issue:, The model in attachment would give big déformations when you calculate it., But you can see the problem already after generating the mesh., - Generate the mesh, - Look at free edges element D763 (see also image) (look at layer 'Cerce'), - There are free edges where the elements should be connected., So these free edges will also result in the two plates not being connected in the results, which gives bad results., Why does the mesh generate a free edge where it must be connected?	The structure is modelled impreciously, the only solution is to correct it manually (see description in comment). I put it as PBD - it is not a bug to fix but maybe we could implement in the future some function for aligning it in a way which is expected

NWEB-9EAKAS	Open project in attachment. Do the calculationg & go to engineering report. Try to regenerate 'Descente de charges Courette'. - With customer it did not work in 2013.0 (= only the title appears, but no table) - With customer it also did not work in 2013.1.64 - With me it works in 2013.1.61. So it's a problem on the customers pc. But what can we do to try and resolve the problem on his pc?	On my computer it works also fine. Please try to:1. Select ther table, press Edit on the ribbon and add the subtable (Descente de charges)2. try to run the linear calculation. I'm not sure whether the table output is available also after nonlinear calculation3. Try the same table in old document. If the table does not work there, then the problem is in the Foundation table itself
NWEB-9E8M98	Free plane load covering more load panels is not generated correctly. Please open attached project and see any of the "test**...." load case. It contains free plane loads applied to load panels (layer is hidden), usualy 4 or 6 of them at once. However, this load is not fully transfered to all beams through load panels - only a few of the load panels generate some linear loads. Therefore the resultant never equals to the original plane load. Why? All load panels have got loaded edges (and simple plane load is generated correctly), but it seems that free plane load can be recaltulated by limited number of panels only (let's say 2).	fixed in 13.1.1029NAMProjection of free load on surface with imprecise geometry (within tolerance in geo setup) works now
JPOL-9EBF7G	Graphical representation of a beam is changed when I click on Refresh action button. In attached project I switched view to structural model and started to input steel connection into node N4. When I defined steel connection and some components (end plate and bolts to be precise) I wanted to recalculate this connection. I hit Refresh button which shone red and the beam B3 lost its rendered appearance. See also picture attached. Why do this happen? It happens for any other frame connection, also in other nodes. Tested in 2013.1.64	

NWEB-9E2KP6	Question: The non-linear behavior of a cable is created by how straight the cable is pulled. But the E-modulus is also subjected to a non-linear behavior. So we recieved a question wether or not it is possible to model this non-linear behavior of the E-modulus of the cable?	The nonlinear behaviour of a cable is primarily due to its geometrical nonlinear behaviour. This is handled by performing a 3rd order analysis (geometric nonlinear using Newton-Raphson).Regarding the relaxation of the cable: this is not strictly talking nonlinear behaviour. It is a time effect, which make the cable slacken progressively over time. It can indeed be seen as an apparent reduction of stiffness.As I understand, the user would like to have the general relaxation function taken into account as a nonlinear stiffness function, which defines the final relaxation value as a function of the initial tension. We do not take that into account as a material property as such, but it might be an idea to take it into account as a nonlinear hinge at one end of the cable. An equivalent force-displacement function could be defined and implemented in the model in that way.I would recommend not to input that nonlinear hinge directly on the cable, but to rather insert a short 1D member at one end of the cable and to apply the nonlinear hinge to that small link. The nonlinear function would be something like this (to be checked):deltaL = L0 * deltaP(P) / Ep / Ap wheredeltaL is the displacement, i.e. the variation of length of the cableL0 is the initial length of the cable (certainly good enough to take it as the distance between the end nodes)deltaP(P) is the relaxation function, giving the loss of tension for a given initial tension (defined in the codes)Ep is the nominal E-modulus of cable steelAp is the cross-section area of the cable
NWEB-9EB6VQ	Crash of SEn during print of old document to pdf using PDFCreator. Tested on user's as well as my PC, both crash reports attached together with the file and pdf file that has been created (incomplete).	It crashes because of out-of-memory. I suggest to split the document into smaller parts. Especially to divide the part with pictures. Check the memory consuption during th eprinting. Once the process of esa.exe consumes more than 1,2 GB it will most probably crash because out-of-memory
NWEB-9E5LGU	We can't import the dwg in attachment into Scia Engineer. It is saved as a AutoCad 2010 version, and it's tested in Scia Engineer 2013.1.61. Do you know where the problem lies?	The problem is in the DWG file, the DirectDWG library is frozen and cannot finnish the import. The developer sent the short list what to do in this case and added a message to SEN to report this issue next time.1) Load the DWG file in the AutoCADU2) Select All (Ctrl-A)3) explode all objects4) SaveAs5) the adapted file can be imported The example files in version 2010 and 2007 are attached in the comment
IBES-9EBRBD	Storey results - average value for wall	
NWEB-9ECFD4	Issue: rib connected to wrong plate ID The customer has followed the following steps to achieve the problem: - first he had 1 plate, E169 - he split this plate in 2 plates. So E169 -> E171 + E172. Problem: S28 was a rib from E169, and is now completly lost, because it still has the property that it should be linked to E169, but it only sees plate E171 against himself. So 'Check of Structure' will indicate that there is a problem with this rib, but can't we provide an automatic check, so that user would get a message or assignment question to assign rib S28 to the correct plate?	

RMAA-9ECBD3	SCIA - prestress concrete (TDA) I need to calculate prestressed massive slab and have several questions: 1) Is it taken into account cracked or not section it TDA analysis? Like in CDD nonlinear analysis (not allow to make CDD analysis with TDA analysis)? 2) What will be end deflection? Total-creep? Please see in attachment calculation (SCIA 2013.1) 3) Is it correctly added all input data? Will be appreciated for fast answer. Thank you	
JBES-9ECJRV	Issue: See Engineering Report of project in attachment. I've added 'Begin Ry', 'Begin Rz', ... But they don't show in the final table. How can I make them visible?	Those properties are not used in any of member in the model. They are related to structural shape.So steps to see those properties are: - switch ON Structural shape in the functionality setting - select at least one member and set its Structural model mode to Manual (see attached picture).Another possibility is to start Editor of table layout and switch OFF property "Clear empty cells" (in this case you will see those columns but copletely empty)
RCCA-9ECP2P	Question: The client wants to model a spiral with a spline or a poliline, but there is always a strange rotation around de axis x o the element 1D (see screenshot1 and screenshot2). How can the client avoid it? ie, he wants to have the local exis z always aligned with the global exis z.	It is not possible to model a spiral with the only beam, it have to consists from more beams or, you can use the spiral ramp from predefined shapes which is modelled with shell.

NWEB-9ECEYM	I do not think that the option described in attachment already exists but maybe this can be considered for future development.	This kind of functionality is available as part of the LTA development: SBS loads.SBS (Secondary Buckling System) items are in fact the bracing members. Within the definition of the SBS load case it is possible to define the % of the normal force resistance, for example 2,5%After the calculation, Scia Engineer will automatically run the Steel check on the members supported by this bracing to determine their buckling resistance Nb,Rd. Then it will take the percentage (2,5%) of that Nb,Rd and insert that as NEd into the bracing members.This is the force for which the bracings need to be designed.Of course this kind of functionality could be extended for beams baced on the Mb,Rd instead of Nb,Rd but thjat would be new development (PBD).
NWEB-9EFJ57	" I would like to report a problem with dialog box Drawing setup in Properties of Results for 2D Elements. When I want to display results on slabs and try to modify drawing setup, there is a problem with labeled isolines: Advanced settings button and Surface with isolines box overlaps - they are in the same place and only one of it can be seen in the same time. Please look on attached screenshot - there are versions of the same dialog window – in first one Advanced settings button is on top, on second Surface with isolines box is on top. "	
NWEB-9EFBFG	Steel code check EN 1993-1-1, member B870, NC215 (or other linear combination): Buckling check is done with A=212 cm^2 if only elastic check is activited (picture). If plastic check is enabled, buckling check is done with Aeff=202 cm^2. We can not find the reason, why.	The member in this case has a class 4 cross-section, so by default the buckling check uses the Aeff of 202 cm^2.In case 'elastic check only' is enabled within Scia Engineer the class is overruled to class 3 and a full elastic check is done, thus using A = 212 cm^2. This is how the 'elastic check only' functionality has functioned since the time of implementation for different codes.For R 14 this functionality has been modified. It has been renamed to 'Elastic verification' and functions in a way the user expects: the elastic verification is executed (using the typical Von Mises interaction) and the class remains 4 in case it's 4. So class 1-2 sections will be checked elastically as class 3 while class 3 and 4 sections keep their class.

NWEB-9EFLG9	Scaffolding: Note is incorrectly shown for a class	Fixed in R 13.1.1023The note in the Scaffolding check is now correctly hidden in case the check is executed for a non-linear combination contained within a Class.
NWEB-9E8EL3	Results for acceleration under displacement of nodes for Eigenmodes and Seismic detailed for a seismic load case are very different. Can you explain difference between those two results ? See Screen Shots in the attached .doc file	in this case, "Accélération aux noeuds" gives the raw values directly from the modal analysis. These values are not linked to a seismic load case.In "Sismique détaillé", all results have been multiplied by the participation factors, which depend on the selected response spectrum.You can find the participation factors in the linear calculation protocol, in the table of the seismic load case (column "G").See attached screenshots: -225.5 (seismic detailed) = -23.9 (acceleration of nodes) * 9.4421 (from calc protocol)
JPOL-9ECHFG	Regular crash of the old document. Please see attached crash report and related esa file. Please investigate the source of problems. No error appeared on my PC.	Source of problem is not fully clear. May be some problem with memory. Similar problem was solved in Deve_04 and fix will be available in next R_patch
RMAA-9EGKCR	Member buckling data: Crash when performing NEL without STB	Fixed in R 13.1.1035In this specific case the non-linear analysis is aborted in case no stability results are available.
RMAA-9EGKS8	The regular crashes of Scia Engineer 2013.1.64 The problem at bill of reinforcement 1. Set renumbered global 2. Click on actions button Renumber 3. Click on button OK	The problem was tested in version 13.1.1035.
NWEB-9EGDT3	Look at the detailed unity check for beam B53. You get the warning underlined in the attached Screen Shot. Two remarks : 1- I have calculated b/t ratios for parts 1, 2, 4 and 5 and they all are less than 50 so I do not understand why this warning is displayed. 2- Shouldn't it be the second line from table 5.1 that has to be used ? in this case b/t<60 and c/t<50.	1) Let's use part 4 as an example: Following your screenshot the length b is ABS(-26) + 96 = 122 mm With a thickness of 2 this gives a b/t ratio of 61 and this exceeds the limit of 50.2) For this I refer to the TB: The section in this case concerns a General Cross-section. "For general sections, the geometrical proportions are checked for elements I, UO and SO using their respective part lengths.Flanges including RI stiffeners are thus considered part by part and not as one whole flange. "

DPIS-9EGKAV	There is a small problem with steel connentions. With a rigid connection, strong axis, there is a problem when you introduice a haunch. If you tick on the option Haunch, then there is automaticly placed a stiffener in the beam at the end of the haunch. You can't delete this stiffener. If you tick off the haunch and tick it on again, the stiffener is deleted, but you can't add the stiffener anymore. PS: This problem does not always occurs. It is possible that the first time it works fine, but the second time, there is a problem. A small test file is attached (tested in Scia Engineer 2013.1.61 and 2013.1.64).	
JTRK-9EHAZ8	Crash of SEn when closing document. When the user closes old document window, SEn reduces the model window (originally maximised) and any next click anywhere on the application cause crash. This happens for different projects on the same PC. Please see attached crash report. However, it doesn't seem to happen on another PC too.	Seems to similar to problem with memory during opening of old document which is solved in Deve_04. It will be available in next R_patch
NWEB-9E9C95	Look at My of (variable height) beams in project and image 'support'. The results for My, as well as the uz displays weird jumps near the supports. This doesn't seem right.	Apparent inconsistencies occur because of the particular shape of the cross-section and the fact that it varies along the beam, causing the PRINCIPAL AXES of the cross-section to ROTATE along the beam.Please keep in mind that haunches are discretized as a series of prismatic segments.Actually, looking closer, the discontinuities occur at EVERY change of css, but it is especially visible at the ends because of the shape of the css.Therefore the reference axes change for each prismatic segment used to define the haunch, causing apparent discontinuities in the results. Displaying the internal forces and beam displacements using "LCS" instead of "Principal" shows smooth results.
NWEB-9ECDEM	This client always gets the attached error first time he calculates a project. It only happens once then it does not show anymore. He tried uninstall the program and install again. He also deleted the temp folder but he always gets that error. Do you have any idea why does this error show ?	it works on my PC without problem, try to use the patch of 2013.1 when it is ready

NWEB-9EHJ56	Crash of SEn when opening the application. Please see attached error report. The user managed to run it just once – he tried to change position of toolbars and program crashed. Now SEn can not be started.	It realy crashes in the calculation of positions of toolbars. deleting of registry entry HKEY_CURRENT_USER\Software\SCIA\ESA\13.1\Workspace should relay help
NWEB-9EHJHP	The attached ifc file can't be imported in Scia Engineer 2013.1 (and not in Scia Engineer 2013.0). It gives an error message (also attached). Deleting the temp folder does not solve the problem. This ifc file can be imported in Scia Engineer 2012. So waht is changed from Scia Engineer 2012 to 2013? Why can't this file be imported in Scia Engineer 2013.1?	fixed in 13.1.1029
ISCS-9EFKGK	So as you can see in the images, we don't have the option to 'align' the labels in the middle of the elements, like indicated on page 14 & 14 of the manual ‘GA drawings – Plane section entity – General arangement drawings – 2D window' (which was made in Scia Engineer 2010) So the question: how do we align the labels in the middle of the elements? (client really wants to get this done before christmas, so put priority to high)	the autolabels are always placed to the end of the beam. This was planned as a new development, but its not added to the projects to 2014 or 2015 as far as I know. Labels may be moved manualy, then fozen, so it wont be replaced during refresh.
NWEB-9EHKAQ	Attached, there is a zip file with an stp file in it. This stp file can't be imported in Scia Engineer. By importing it with the option 'stepsteel cad', there is a message that the file is succesfully imported, but the model is Scia Engineer remains empty. No members are imported. By importing it with the option 'stepsteel', there is a message that the path to the profile library is to large (more than 35 characters). If the this problem is solved, by putting the prof lib on a shorter path, then it seems that the import of the stp file is starting. But after some time, there is an error log file with a lot of errors. Can you have a look why this file can't be imported in Scia Engineer?	Tested in R 2013.1.1011See also screenshots in attachment.I did not get any error regarding the profile library (possibly the client tried the import on an older version).In any case, after import a large error list is given. This list is also given in the file ...temp\EPW\epw\scan.logLooking at the list each time the error indicates there is an unknown instance.When I look into the normal format of an .stp file we expect to see entities like NODE, ELEMENT, MEMBER, CROSS_SECTION, MATERIAL etc.In this .stp file there are however totally different entities like CARTESIAN_POINT, DIRECTION, AXIS2_PLACEMENT_3D etcIn other words, this is a completely different file sheme, it doesn't contain any of the required entities and can thus not be imported.The user should check the origin of the stp file and make sure it contains the correct data.

JBES-9EJDFT	Why do I get insufficient eigenvalues from results? I am asking for 4, and there should be around 15000-18000 degrees of freedom available. ==	There seems to be too many hinges, causing a lot of local eigenmodes and convergence issues in the Lanczos modal solution.Please use subspace iteration instead (see solver settings) for the calculation of eigenmodes and display the eigenshapes.That will help locating & fixing the issues.
NWEB-9EJ6UE	Crash of 64bit Solver, Solver crashes during the calculation of internal forces after NL calculation., If I rename the 64bit solver and the 32bit solver is used, then it is ok, without crash.	
NWEB-9EBFCE	Issue: Error message (already during test of input data): "Length of element No. 16189 (macro Staaf_LL_UPN27) is zero!" - Check of structure does not solve or find anything - Connect member/nodes does not find anything either - There is a dummy material. If I lower it's E-modulus from 2,1 *10^8 to 2,1*10^5 MPa, then it is also not solved. - Putting the mesh of the load panel to 'automatic' does not solve anything either How can we fix this? Is there a work-around? And how is this caused?	Set in Mesh setup > Minimal distance between two points to 0,0001m or join the probelmatic nodes as e.g. K175 and P797 (distance between them is 0,002m)
NWEB-9ECCM3	I can't change (add or remove) columns in the calculation protocol in the engineering report.	This table cannot be edited. It's layout is hardcoded. If the user needs to fit it on the paper width, use "Style" to use smaller font or he can change the paper orientation to Landscape (item Page format)
NWEB-9EJFS5	There is a problem with the project attached. By running the mesh generation, there is a runtime error and ESA.exe needs to be stopped. How can this be solved?	see comment

RCCA-9EJQDR	A client found a strange result in Scia Engineer. When in the model in attachment, he does the batch analysis the result for the linear (see combination pp+vx1) and non linear calculation (see combination NC8 - pp+serv+vx) are the same for my, see picture 1. But when he clears the results and does only the non linear calculation, the values fo my are different from the batch analysis, see picture 2. Why is there this difference between the results when I have the linear calculation finished and not finished?	
JBES-9EKBC5	Question about Soilin:, How do we treat pre-consolidated ground?, Since we have to input one E-modulus per layer, it does not seem to be able to handle this correctly., The relation between strain and stress per layer underneath an excavated layer, is as shown in the figure., The inclination is equal to the E-modulus., So if you excavate some ground, and then start to apply stress on it again, then it will first appear very stiff (because it follows the first red line), but after the stress you put on it becomes bigger than your initial stress (that was given by the ground you excavated), then it should follow an other E-modulus., So how do we calculate these deformations exactly when using Soilin to look for plate deformation on an excavated hole?, And do we have a kind of benchmark to show this?	from Buček: if there is excavation , the system will use EC or CSN, the m coefficient is used and the soil thickness is bigger because sigma_z minus m, so the settlement is smaller and it match with the assumption that module E is bigger.Or user can use bigger E module manually in the geol. profile, then it just must be considered that this E module is then interpolated to the surrounding soil, so it is good to place some "common" borehole next to this, where E module should be normal value.
JBES-9EKBHK	Issue: can't see X,Y, Z coordinates in preview of 'Concrete -> 2D member -> Member design -> Member design ULS+SLS' - Open project in attachment - Do linear calculation & go to concrete menu - Go to 'Concrete -> 2D member -> Member design -> Member design ULS+SLS' - Refresh & look at preview for 'OUTPUT = DETAILLED' (make sure the Items XYZ are added in the table composer of the preview) Why can't I see the X Y Z coordinates of the calculated values?	
NWEB-9EJLMP	See crash report in attachment. The student cannot open Scia Engineer. I already adviced him to delete temp and delete the workspace in the registry, but no solution.	The problem is caused by the ATI graphic card driver. We can advice to modify the grahic card setting (reduce acceleration) or to try to use different version of the driver (newer or older). The last chance is to switch rendering to Sw emulation of OpenGL (see picture) - but this will cause very slow displaying of structures

NWEB-9EGCLX	Why do I get the error 898 when I do for example the response check for the UGT combination with redistributed moments activated (for member S7)? What does this 'Check of redistribution of bending moments doesn't satisfy' mean? (it is only in the first & last section) And what do we have to conclude from this message?	The error 898 is appeared because check was not done, because redistributed moment was not calculated (some conditions for using method according to EN 1992-1-1 were not fullfilled), see attached poicture
JPOL-9EKBTJ	Small mistake in the Czech translation, see picture attached.	
LSKI-9EKJZA	The user had one big plate which he devided into 3 small ones. He used Modify > Divide surface. Now the loads are not taken into account because the program seems not to recognise them any more. Look for example at internal edge ES122, it is drawn on member D68 but if you look at its properties it shows that ES122 is on D66.	
NWEB-9EHGXN		
NWEB-9EJN3E	It seems that a HEA340 profile is checked for torsional buckling in the stability check. According to NEN-EN 1993-1-1 (6.3.1.4(5) NB), this check is not needed. Can you have a look at this problem? (I don't have this national annex).	In general doubly-symmetric I-sections are not susceptible to torsional buckling, however it is possible that, due to the difference in support conditions and geometrical properties torsional buckling is more limiting than flexural buckling.Therefore, within Scia Engineer the following occurs for doubly-symmetric I-sections:- In case torsional buckling has a lower unity check than flexural buckling this check is omitted and thus not shown on the output.- In case torsional buckling has a higher unity check than flexural buckling it means this check is limiting. Then it is verified and shown on the output.The user's output is a fine example which shows that torsional buckling cannot be neglected for all doubly-symmetric I-sections, it must be evaluated on a case by case basis.For more background information reference is made to "Steel Structures: Design using FEM, Kindmann R., Kraus M.,Ernst & Sohn, 2011" in which, amongst other subjects, the torsional buckling of doubly-symmetric I-sections is also discussed.

NWEB-9ENKPV	Explanation can be found in attached document	It seems this db4 file dates from an older version if Scia Engineer. In R2013.0 it can be seen that the initial shape does not correctly follow the centerline.More specifically, the two web elements have one centerline element (since they are touching) and since both sections have a different thickness the so called fictive elements need to be added to connect the centerline parts.In the sections stored within the db4 file the initial shape does not follow any of those centerline definitions. In addition (see screenshot) the centerline is shifted. Possibly those sections were saved in a version which had separate centerline elements, but for that I would need to know in which version this section was defined.In R2013.1 the initial shape correctly follows the centerline. As can be seen on the screenshots, all elements are accounted for.Due to the fact that the stored data is outdated (doesn't match the centerline), this is discarded in R2013.1. Overal the best advise to the user is to redefine those sections. If they really which to calculate with two separate webs the sections should be inputted with a spacing between them. For more information regarding centerline generation reference is made to the TB for the Cross-section analysis.
NWEB-9EKCZZ	An error is shown when you try to make construction stages analysis Already tried to export to a new project > does not solve the issue	
NWEB-9EVDFV	Error message during opening of Design Form Template	already solved, this should be solved by new update - new C2T and change in LN
LSKI-9EWGES	French Translation Steel Check	
NWEB-9EYE7K	ISSUE: value for Surface A seems to be wrong for CFCHS 355.6x10 seems to be wrong. It gives 0.007.. m², but it should be around 0.010..m²	The area of the CFCHS355.6X10 section was found to be incorrect in the original XLS file supplied to Scia. It has been corrected in DEVE 11 and will be merged to R 13.01.1023
CSCT-9A4BX6	We need long time to open the project and if it´s open we need hours to copy the openings in the plate. Any idea?	It could not be quicker as there are too small circle openings and Scia Engineer is still 32bit. When in 1 or 2 years we migrate to 64bit platform then it will be quciker. Primary Scia Engineer is software for structure analysis of buildings - circular opening with diameter 15mm are not typical compoments for building analysis.
RMAA-9EZH3M	Wrong translation at timber in Czech language - more in attachments. It was tested: 2013.1.1018	

JBES-9F4CB2	In attachment a project where the functionality 'pressure only in 2D elements' is used. Problem: the forces that pass through the wall, towards the beams are not correct. (tested in 2013.1.64). To give the desired result,, plates E6 E12 and E9 have been modified with property modifiers to give the correct moments in the beams over the opening. So why are the moments in the beams that low when we are only using (pressure only in 2D elements)? (you can see the problem the most clear in beams S21& S17)	
NWEB-9F4FD4	See attached project and drawing (pdf file). Correct result for reactions is what we get for the right beams. For these beams the correct loading is introduced manually (this loading is calculated manually from the applied surface load). Now if we look at the results which we get using the panels + free surface loads these are only correct in case of (3). Case (1) and (2) are not correct. It seems that there is something wrong either with the panel or with the free surface load. Regards,	Results for 1 and 2 are correct, the input is differentthen in the beams and in the case of 3. Case of 3 is solved with FEM - results are as expected. In case 1 and 2 you get different loading as there is uniform distribution on beam - all beams has the same weight (it causes different input then the user thinks). Standard method works following : sum of load / total length of beams. Total lenghth of beam = length of beam * weight.
FLFA-9F4TE7	a client asked if it's possible to change the scale of gcs icon in images sent to Engineering Report. He wants it to be bigger.	Hi Fernanda. It is not posible at the moment. I add it as a separate requiement for future development. M.
NWEB-9F4KAR	Try to add a connection to the column shoe > The program crashes. No problem occurs when the variable height property is removed. I tested this in an older version and it works fine with variable height cross sections so it is a problem of 2013.1.	
NWEB-9F3HBM	Message "Der Trägerquerschntit ist nichtvon Typ "I""	07/01 PVT: In this project stiffeners are defined on non-I-sections.Stiffeners are intended to modify the shear buckling field of I-sectiosn and can therefore only be inputted on I-sections.In this project there are stiffeners on several other cross-section types and this causes the error.Most likely this is a (very) old project in which there was not yet a test on the input of stiffeners.The easiest way to fix this is to use 'Tools' > 'Cleaner' and clean up all stiffeners.Note: This project also contaisn other issues, so after cleaning up the stiffeners it is advised to run 'Check Structure'.

NWEB-9F5KC2	See project in attachment -> error topology, but why? I already stripped down this model (see IS_...... .esa) but I am still receiving this error. The strange part is, that when I delete one beam (does not matter which one!) there is no problem anymore.	It is an issue of input data. The distance of internal node and edge of slab is little then the set mesh elements. If you set in Mesh setup - average size of 2D members to 0,2m, then it works. Unfortunetely it is a feature which we could not change as the mesh is done of 3rd party.
FLFA-9F5VJ6	Mr. Theóphilo modelled a concrete tank with property modifiers for 2D elements. According to him, the results are strange. When comparing calculation with and without reduced stiffness he encountered inconsistency in the results. And other results are tending to infinite values. Why is this happening? Is it the model? I thought this could be an input problem, so I'm asking for your help. If not, could you please send it to the right PED? The model still with 100% of axial and bending stiffiness is attached.	http://help.nemetschek-scia.com/13.1/en/#rb/modeldata/property_modifiers.htm?Highlight=property%20modifiers - By default is installed 100% (which means “no correction”) --> If I set e.g. 0 then I get totally different results. For 100 it is the same as no modifier is set.
NWEB-9EGHCT	The cross section in the attached project can not be calculated using FEM Analysis.	The cross-section was inputted as a thin-walled section however the centerlines are not intersecting (see screenshot 1). As a result, the centerline is not connected and thus the normal 1D FEM analysis cannot be run.This is the reason why the 2D FEM option is shown in red.- For 2D FEM analysis a mesh is generated on the outer polygons of the thin-walled elements. However, as shown on the second and third screenshot, there are nodes which fall within the internal part of other elements, elements which are not connected (no connected nodes) etc. This is why no mesh can be generated which leads to the 'Genex' error..=> The solution is quite simple: the user should make sure the centerlines are connected as shown on the final screenshot. When the centerline is properly connected the 1D FEM analysis will be used leading to a fast and accurate torsional analysis.

NWEB-9F5RNM	Would you please have a look at the below for me? I think it is a bug as I can’t understand nor have I heard of 0.225% min tension reinf. For “overtensed” concrete. Do not even know what “overtensed” means to be fair so I checked prestressed but the min reinforcement is still wrong. Can we please check as a matter of urgency.	Dear colleagues, thank you for this inquiry. This is my response:[A] Explanation to "OVERTENSED":(1) The term "overtensed" is, as a fact, not known in the British engineering terminology. I am sorry to confess that it originated, many years ago, in my head. I just translated the (legal) German term "überzogen" litterally into English, thus "overtensionned", as its original spelling was. In the course of time, somebody (not me, I am quite sure) made (a) "overreinforced"; (b) "overtensed" (as here above); (c) maybe other mutations out of it;(2) I remember quite well that, some years ago, there came an inquiry from UK as to the meaning of this, for the British colleagues, unintelligible term. We discussed it then internally: in the result, I confessed, ruefully, my guilt and - I rewrote "overtensionned" everywhere I could find it in my own texts to "under full tension" or "under pure tension" - I am no more sure what term I used really (maybe, not consequently, alternately both);(3) However, I suspect now that the colleagues, participating then in our "terminological discussion" did not do the same in the English SEN interface or in other SCIA English texts, resp. But it was surely their responsibility, not mine, since I had been responsible for the German interface (no more now).Conclusion: Please, let me and Jirka Porada know where you have found the incriminated illegal term "overtensed" (as well as its alleged mutations). We would then do our best to replace it or let replace it on every its occurence by the right term "under pure/full tension".[B] Explanation of the percentage 0.225%.(1) Well, upon the explanation [A], the case might be clear now, is it not?(2) If not (yet), please, have a look at the attachment file >min tension reinf Code_pure tension.jpg< of my provenience (below): 0.225% =0.45%*0.5 - it is half the minimum percentage in <cross-sections under pure tension> per upper/lower face - in Shells, of course!Conclusion: In the sense of [B] I reset the Ticket Status to "NO BUG" since there is no bug in the 2D reinforcement module NEDIM.Dr. Eduard Hobst, Nürnberg
NWEB-9F7D26	inblock-issue. We get an error when updating the inblock-definition. The only way to continue, is to assign a steel section to the columns and then delete the concrete cross-section CS1. After that, we can update the inblock-definition. What is wrong here ?	In-block are not supported any more according to Geert and Cyril. > no bug
NWEB-9F7BTW	Error in Tekla plug-in, during export from Tekla to Scia. Please see attached picture and cfg file in question. How should the user proceed?	there is no path to Esa.exe

FLFA-9F7UPQ	I attached the model with reduced stiffness. A more detailed question from the client is that when reducing stiffness of subregion R5 the results are okay, but when reducing stiffness of R6, R7, R8 and R9, the client got: - smaller deformation compared to the model without modified properties - rotation with "inifite" values and asymetrical results - distorted deformed structures. Do you know why this is happening?	In last version (13.1.1036 - patch of 2013.1) I see expected results, no asymetric results and infinite values...could you check it is the patch when it is released?
RMAA-9F7KTG	Small problem at PGNL Wrong error report during nonlinear calculation and automatic input coefficient by creep into stress/strain diagram It was tested 2013.1.1024	
NWEB-9F7JW7	We ger runtime-error if we want to set a connection to an arbitrary-profile. Attached project from costumer (Pos.4...) Scia crashes if we want to connect beams at N3. I have attached another project (Voute...). I have testet it in 2013.0 and it works. Crash in 13.1.	
NWEB-9F8JHS	Different value of unity check for graphical and numerical representation.	I tested it in the latest build of R2013.1When I don't change anything I also get the differences as you indicated. However if I just run the calculation and then check, without changing anything, I get 0,64 both graphically and numerically.It thus seems that the issue is purely result related i.e. there are old results stored in the project. Just recalculating will fix this..(For info: The new Results API is from now on being used in Timber, that is the reason why suddenly some unexpected behaviour is noticed or why this combination strategy is visible. Let us know should you spot any other issues, we will fix them asap. In the near future, the old results system will be completely replaced by the new Results API for all checks.)

JPOL-9FBBSN	Project attached contains a pair of walls supporting a slab. Connection between these horizontal and vertical elements is defined by hinge on 2D member edge so that only uz deformation is restraint. The aim is that walls support the slab in vertical direction only but the slab can slide on it. However, horizontal load give me unexpected results (see picture attached). I would suppose that linear analysis cannot be performed, sinde the slab can "slide away". Why does it not happen? Second case - if you turn off the activity there is also a structure of a box (walls with upper slab). The slab is supported in a similar way as above. Why is the deformed shape from horizontal load deformed, not rectangular, as SEn shows (for LC2) - see another picture attached. Tested in 2013.1.64	
CSCT-9FBJGL	Attached a project with 2 columns. We have set a manuel imperfection in buckling-system and we think that the moment-line has to be same. Why is for B2 a doublecurve (picture)?	This concerns standard behaviour of the imperfection algorithm. - After the first run of the non-linear analysis the deflection in the members is determined. The sign of the deflection in the middle of the buckling system is evaluated and the bow imperfection is applied in such a case that it works in the direction of this deflection.- In case however the deflection in the middle of the buckling span is zero an alternating bow imperfection is applied.In this project, both buckling systems have a zero deflection in the middle.On B4 there is just one span, so it's impossible to apply an alternating imperfection there.For the system of B1 & B2 there are two spans in the buckling system so the alternation can be applied.Using a neglegible small H load the dleflection in the middle can be made non-zero so both systems give the same result.
NWEB-9FBJHS	Open the Engineering Report from the project in attachment, you will get the error messages in attachment (and 3 new pictures are added to the report). The customer said this happened when inserting the 3rd picture, with the first 2 pictures there was no problem. However, deleting the pictures did not help either.	The problem is with exhausting of memory during working with pictures. Scia Eng runs at the memory limit 2.4GB after calculation and display 2D results esa. When then sending picture into ER it causes problem because memory is so fragmented athat the size of picture does not fit in it. I assign the bug to ESR repated to the optimizations.
NWEB-9F7EYK	In version 13.1.64 I could not find the stiffener component for haunch Look at the two attached sreenshots from 13.0 and 13.1	
CSCT-9FCE7S	The splitting of the css-table in new document is confused (picture).	See the picture with explanation of "table-flow ". I think it is quite logic
CSCT-9FCEB8	We cannot type an own value into the scale-buttons for add-datas and results.	it works on 13.1.1030. I see no reason why it should not work on 13.1.64
CSCT-9FCF8H	We need the "step-back arrow button" in different windows, f.e. in pictures-edit of the gallery etc...	I make it as an ESR for some future improvement project. Currently you can use Undo in Eng report to revert changes done in the report picture
CSCT-9FCF9Q	There are some picture-properties not present, if we send the picture into new reports, f.e. the rotation. We have to swicth it afterwards via picture properties.	I attach this demand to project with Eng report improvements

CSCT-9FCGSQ	Why do we have "additional moments" f.e. for L-profile B251? I have deleted rigids, hinges but still the moments...	Because this is a Class 4 section ...Additional moments are caused by the normal force due to a shift in the center of gravity of the effective shape.
	mapping table for Tekla link (Tekla2Scia) can not be changed nor esxtended the used mapping table for the Tekla-2-Scia link has a dbType = "master" this means you can not extend the content without rendering it corrupt and unusable. As soon as so change it and try to use it, an error is produced during export about a missing TKP.cfg file For a user it is indispensable to be able to work with other css than purely the ones we provide.... because we did not put all profiles in the list I have - some time ago - extended the default list with additional families (I added e.g. the HEAxxx, HEBxxx, HEMxxx whereas the default only had the HExxxA, HExxxB, HExxxM etc These profiles were added in the mapping file for Scia -2-Tekla but not in the mapping tbale for Tekla-2-Scia create one single and extensive mapping table for both Scia-2-Tekla and Tekla-2-Scia which is up to date and can be edited by the user	when you edit mapping table Tekla to Scia you have to change "master" to "user" in both css and materrial map. table (even if you do change only in one of them). It is how to work since begining...
RCCA-9FCJTP	Issue: My recalc	I put this problem as No bug, because the similar problem is already in Devtrack database, see bug RCC13-3619EKM34
RMAA-9FDFJH	The problem at Engineering report We cannot display colour in tab of materials in Egineering report. It was tested: 2013.1.64	Linked to the project
NWEB-9FECM9	combination key table in engineering report	Dobry denzde je problem v tom, ze rozhodujici jsou nelinearmi kombinace, ktere se do tabulky Combikey nepridavaji. Pokud ve vlastnostech vysledkove tabulky zvolite nejakou obalkovou kombinaci, tak se rozhodujici kombinace objevi v tabulce Combikey i v pripade ze je vysledkova tabulka zanorena. Je ale potreba dbat na to, aby tabulka Combikey byla vygenerovana az po generaci vysledkovych tabulek.Nove posudky jsou jiz delany tak, ze tabulka Combikey je soucasti primo te vysledkove tabulky.MT

NWEB-9FBP8D	Problem with BS steel check. In the attached project, the BS steel check can not be performed. Problems: a) node 26, a linked node very far from the structure b) the coordinates of the nodes are very inaccurate. After resolving these problems, the BS steel check is still not working. I have deleted the complete structure, and introduice a simple frame (2 column, 1 beam). Also for this simple frame, the BS steel check can't be performed. The project is created using an old project template. So propably, this is causing the problem. Can the problem be solved in this project, or does the client have to model the structure again in a new project?	As discussed, the issue seems to be specifically in this project file. When creating another file etc everything in the BS check works. Also when exporting the structure of the current project and re-importing into a different (new) file it works.It is advised that the user replaced his current update, it seems the issue is specifically in this template (which probably contains some older data).
NWEB-9FCDFA	grey icons for print picture if old document is off.	This problem is already solved. Fix will eb available in comming R_patch
NWEB-9FEL43	Free surface trapezoidal load is wrongly displayed. See attached picture where trapezoidal FF2 is generated into more GFF loads - but those on walls perpendicular to the original load have got also wrong mirrored copy into negative values. Walls parallel to the load are correct. Also results are correct. It is simply the graphics that makes issue.	I see the correct generated load in proper direction. On 2D elements which are penpendicular to load the generated load is drawn always on whole member - there is not possible to do projection easily
HWRE-9FJF7T	Detailed connection drawings: only thickness of steel stiffners are printed in gallery, but material quality is also needed.	
HWRE-9FJFBJ	1) For stiffners of steel connections, steel quality and thickness is depending on temperature and has to be selected according to EN 1993-1-10, table 2.1 2) Z-Value Z,Ed has to be calculated according to EN 1993-1-10, formular (3.2) and table 3.2	

NWEB-9F86UK	This time we found something connected to load panels If you look at the project in attachment (almost the same as in ticket NWEB-9F3QSC = devTrack LSK14-69F4CAX), and try to do 'test of input data' then you get an error about inputted load PG9. How can I fix the 'test of input data' ?	Plane load generator is old and not bugfixed any more since we have load panel. To put two load generators on one place is not logical. The recommendation is to remove all plane load generators with surface load on panel.
ISCS-9FJHJN	Question: 2nd order calculation (see project in attachment): The second order effect is taking into account in all checks for ULS if: 1) the option Use buckling data in Concrete solver or in concrete member data is ON 2) and the calculated slenderness is bigger than the limit slenderness So I would assume that both e2,z and e2,y would be calculated, but only e2,z is calculated. ----> The column is only loaded by the moment in one direction and therefore buckling of the column will be only in the direction of this moment?? I tested this and, with also a moment in the other direction, I also get an e2,y. ----> In an old manual this project was also used, and there we have both e2,z and e2,y (however we have only a moment in one direction). Also the following sentence is written in this manual: "Note that Biaxial bending is the calculation type even though no Mz is applied to the column. This is because of the 2nd order moment that causes a moment Mz to be taken into account for design." So now I am confused. Has M2,z has to be taken into account or not? Is there more information about it or is it somewhere written in the Eurocode?	1/he second order effect is taking into account in all checks for ULS if:1) the option Use buckling data in Concrete solver or in concrete member data is ON2) and the calculated slenderness is bigger than the limit slendernessYes, these conditions have to be fullfilled2/So I would assume that both e2,z and e2,y would be calculated, but only e2,z is calculated.The basic value of eccentricities are calculated in both directions, but then the values are recalculated according to values of slenderness and values of bending moment in both direction, see attached pdf fileIt follows, that both bending moments are taking into account, but final vaklues of eccentricities dpends on values of slenderness and values of bending moment in both direction
NWEB-9FJLG2	Licence is activated. Why such text in attachment?	Some other licence server is runnign on the clients computer. It is necessary to uninstall or at least switch OFF all older Scia Licence servers.

LSKI-9FKCTQ	Please find questions below from Egis about the Fast train module	Il y a semble-t-il un problème de communication.Il n'y a pas de développement en cours concernant ce point.Le seul développement qui a été effectué l'a été sous forme d'un applicatif dans Excelservant de pré-processeur pour la saisie des charges mobiles dynamiques.Le calcul dynamique à proprement parler devait faire l'objet d'une validation par M. Luc Vandemoertle, mais cette partie est restée en suspens depuis pas mal de temps.Nous proposons cette solution sous Excel aux clients qui ont une licence pour le module dynamique TGV dans EPW et les autres ont la possibilité d'acquérir ce mêmepré-processeur pour une somme forfaitaire. Je sais également qu'au moins un BE a développéson propre préprocesseur (nous leur avons fourni des informations techniques surl'interfaçage, mais aucun outil).
NWEB-9FJMMK	Engineering Report: after refreshing of whole document, some items remains with red exclamation mark. How can the customer be sure, that all items are up to date?	Improvement of this "not ideal" validity status is already planned. I attach this bug to the project.
RMAA-9FKKGD	Is it possible to rortate text in Gallery picture editor?	Added as an ESR for future development
JPOL-9FKKN5	How to display As,req results on a picture in ER in black&white? The basic question is - how to display results of reinforcement design on plates by black numbers (not coloured)? (See picture below) If I switch the result type to "numbers" they are coloured. This is not wanted in general by users. One possibility is to change colours of the whole document into black&white but this is suitable for other pictures, usually. So, is there any possibility to set this for one particular picture only? The only solution I have found is to open colours setting of the current picture and change all rows (except background) in the tab colours&lines to black (picture attached). However, when just one row has got different colour from black, it turns to colours again. Never mind which row I change (third picture attached). Please explain. Tested inn 2013.1.64	It is very easy. Just switch the Eng report to Black and white mode just before the picture and then back to coloured mode after the picture using report item Style (see attached pictures and changed project).
RMAA-9FLBAV	How does Scia handle notional horizontal loads recommended say by BS 8110 code. Is there a way to use existing functionality to handle this?	

HWRE-9FLE95	1) In DIN EN 1993-1-3, 6.2.4 ltb and bending check should be done with buckling curve b for class 4 sections. 2) No A,eff should be used. See PDF from "Stahlbaukalender 2013"	1) In the cold-formed check we indeed excecute the LTB check using curve b. Make sure that the section is cold formed and has an initial shape, so it's checked by the EN 1993-1-3 code check. In attachment a sample output of a C-section showing curve b.2) According to the Eurocode as well as several background reference documents, Aeff should be used in the buckling check. In attachment I included two buckling examples from the ECCS Guide "Design of Cold-Formed Steel Structures, ECCS Eurocode Design Manual, Ernst & Sohn, 2012" where the use of Aeff in both the slenderness and the final verification can clearly be seen.
LSKI-9FMM8U	Please find explanation in the attached pdf file	Shear reinforcement for column is calculated only according to detailing provisions, therefore value Vrdc, VRdmax... are zero. Shear reinforcement will be calculated from shear force in new concrete checks and design (some new checks will be available in version 2014)
NWEB-9FPCZU	There is a simple structure with 3 Beams. One is without "Trapezblech" and 2 with. I see, that something will be calculated for the Trapezblech, but it have no influence on the unity check. Why is it so?	When looking at the Detailed output for B2 it can be seen that the data for the Trapezblech is accounted for however the additional stiffness it brings is zero.The output shows us that the C100 value is zero which means there is no value found.Looking at the input the user made it can be seen that this result is correct. The user is using bolts at spacing br, on a positive Trapezblech layout but he indicated the bolts are put through the top flange of the Trapezblech. As the table shows, there is no C100 value for this.The main issue here I think is that top flange, in most cases Trapezblech are fastened through the bottom flange. When you change that, a solution for C100 is found and you see the influence on the check, instantly lowering the UC.
LKGZ-9FJK3Q	If the functionality of old document is switched OFF, we cannot use the buttons to transfer datas into new engineering report. The buttons are inactice (picture) PS: and still the mystic that if we restart the project the functionality switches automatically ON.	Problem with buttons is already solved in R_2013.1.1035. You can also use right mouse button click to send data to Eng report. (The "mystic" still remains)
NWEB-9FRL44	Since the installation of Scia Engineer on a small business server 2011, the server consumes almost all of its physical memory. After a few days, the server is running completely down, so that they have to reboot this server almost every week. Do you know this problem that the service uses unnecessarily many resources with this kind of server?	In the past version of licence server there was a memory leak. Therefore it is possible that the licence server consumes after some time whole memory. Try to upgrade to the latest available version of Scia Licene server (or at least to version 2.2.3). There the problem should be fixed. If you are already using the latest version of Scia licence server, please let me know which proccess on the clients server does consume the whole memory.

ISCS-9FSCEC	See project in attachment. - Run NLCombi71 in 2013.0 --> Max. number of iterations was reached. - Run NLCombi71 in 2013.1 -> Calculates without a problem. Why ? When looking at the contact stresses in 2013.1, we still see tension? It seems that the result of 2013.0 is more logic, because we have only a horizontal force in NLCombi71 so there is tension, but therefore he cannot reach an equilibrium?	Remove temporary solver files by Cleaner. In 2013.1 is not reach the max. iteration because there was incresed precise and warning about singularity is shown. The structure is unstable with km deformations.
NWEB-9FSFRC	I have quite a simpel project from a customer, but it crashes during calculation. Can we find the cause (so that the problem can't occur again)? (the customer already has work-around: exporting project to new esa file)	incorrect data > no bug, run Check structure data before calculation - tested in 13.1.1035
NWEB-9FRLV7	DXF import - problematic solid Please find attached DXF file from one of our clients. There is a complex solid in it. Client said, that some time ago he managed to import it to Scia and convert to slab, but now he receives error message. In my opinion this solid is too complex to convert, but client asked to send it to you for consultancy. Could you please look into it? And if it is possible to convert it to slab, please write me how to do it? Thank you in advance for your help.	this particular example could not be ever imported as or converted into a slab, open polysurface mesh was never supported for it. The only solution is to break the solid into lines in the original application
NWEB-9FSGJC	Eng. Doc.: how to sort short version of steel code check table (attached project) on names of "css".?	You need to edit proper subtable and there switch the sorting according to desired values - see attached pictures. Currently the selection of propert subtable is quite complicated but in version 14 it will be much simplier.

NWEB-9FLDFK	I have exploded the ULS combination to linear combinations and they have already been exploded. Same thing for the SLS combination. There is a load case cat. H and a load case Snow in the combinations -> Problem: Cat. H & Snow should not be togehter in ULS (see image)	28/01 PVT: Pleaxse review the inputs ... when looking in the load case Manager you can see that the BG4 Cat H loadcase is assigned to the load group Snow ...
GVAN-9FTEKE	protection checked out 2x Customer did an investigation and he found this: - Scia Engineer runs, so has 1 license checked out - then there was a short interruption in the VPN connection -- the VPN connection was interrupted during 12 seconds (according to the LOG file) - Scia Engineer was still working, but after this VPN interruption Scia Engineer checked out again 1 license So, now 2 licenses were in use, but only 1 instance of Scia Engineer was running.	Have a look at attached PDF. Go to chapter 13 (Managing optin file) and see commands TIMEOUT and TIMEOUTALL. I hope this can help to the customer
NWEB-9FSFCU	A vertical area load applied to a load panel is producing forces in both horizontal directions (LF394,LF401 shown below) as well as the main vertical line load. Why is this the case, is it due to the fact that the panel is inclined. It appears to affect some but not all members.	Input load acts in LCS of panel but the generated load on beams is always generated in GCS. As a result of recalculating of load in LCS into GCS the forces have to be in all 3 directions.

NWEB-9B2GZY	SETUPLANG=1036 PROTECTIONTYPE=2 LICSERVERADDRESS=27000@Lic-H-Nemetschek-01 FORMATSYSTEM=1 NATIONALCODE={CCEB9512-0B25-11d6-AA9A-0050FC1D5C09NATIONALCODE={CCEB9512-0B25-11d6-AA9A-0050FC1D5C09} ADDLOCAL=ProgramFiles,HelpFiles,StructuralEditionLibs,CompositeColumn,CompositeBeam,BS2000,WindLoadEngine} ADDLOCAL=ProgramFiles,HelpFiles,LangFRB,LangDEU,LangITA,StructuralEditionLibs,CompositeColumn,CompositeBeam,BS2000,WindLoadEngine After installation, Scia Engineer always crashes at start-up. And the protection setup can't find the server license automatically after installation. He is also not able to generate the 'iss'-file with the option /r as indicated in the documentation (see pdf in attachment). He also installed all vcredist packages (see image in attachment) What causes this problem, and how can the command line installation be executed perfectly?	fixed in 2013.1, registry entries are not created if installed under SYSTEM account