Data stored in this version can be opened in version 13.1.61 (Release2013.1) and higher.

Ticket number(s)	Bug Description	Solution
NWEB-8VQDZJ	Our Italian customer Mr. Predicatori found some spelling mistakes in the Italian Language version of Scia Engineer 2012. You will find them in the attached PDF. Hopefully, they can be corrected.	Italian translation has been reviewed and corrected by Mr Mayer (former translator) and Mr Antonini (recent translator).
NWEB-94GKJX	I am not sure who is responsible for the translations, therefore I send it to you. Please if you are not the right person, then reassign it. Attached you will find some remarks about translation of SENG to IT. My Remark additionaly: The steel checks are completely in English if IT is selected as language.	Italian translation has been reviewed and corrected by Mr Mayer (former translator) and Mr Antonini (recent translator).
HWRE-95PK2T	Translation correction for steel code check output: "b/t" is an DIN18800 expression and must be "c/t". See attachment.	27.03.2013Correction done. Generally, where there is "Width-to-Thickness" in the Englisch original, "Breite/Dicke" was inserted instead of "b/t" in order to avoid discrepances with user for special NormsEH
CSCT-965HHT	We have some structure-troubles every day, so we want to discuss a general plan to create such a structure in scia. Please have a look at it. 1) biggest problem is to delete defined cuts. It´s not possible to select the gray line to delete. Costumer has to restart the project to delete the cuts. Do you know the problem? 2) We have many toplogy-errors because of cuts. Is the structure to complex for Scia or is there any sequence to define structures like that. (F. e. the mesh works, but if we define rips at working elements we have new topology-errors and have to create it new. 3) Problems with intersections f.e. if you open the structure you see an opening because of any of the three intersections. Or if we do astructure control we get error on S104... What do you think? Is the strucure to complex or do we have to create it on another way?	in 13.0.50 - no problem occours, it seems to be fixed in 2013 version
NWEB-94GKJX	Italian Translation incomplete	Italian translation has been reviewed and corrected by Mr Mayer (former translator) and Mr Antonini (recent translator).
NWEB-975J5N	When using the 3D wind generator there is an error given in the project of the customer (Overzicht_stab_van_Hende.esa) that LP32 is not coplanar., When I delete this load panel and I draw a new one, the error is gone, but now I get the message that the structure is open (IS-Overzicht_stab_van_Hende.esa)., I do not understand why we are getting this message, could you help me?,	fixed in 13.1.47 + WLE44

Ticket number(s)	Bug Description	Solution
JBES-979JQN	Issue about 'shear effect control' (see image concrete setup) If you take the esa-file in attachment, and calculate it with the 'shear effect control' deactivated, then you get the results in the image "dwarskrachteffect_SR2.jpg" where there are no peak stresses. But then when I activate the shear effect control, then you get these peaks (when you check the 'design internal forces' in the concrete menu, or when you check the theoretical reinforcement). But the maximal moment should never be increased by this shear effect, because the shear effect comes from the shear force, and the shear force is linearly dependant on the derivative of the moment, and the derivative of the moment is zero at maximums or minimums. So why does the design moment m1 go from +/-100 -> 220 kNm/m when you change this option? And why does the design As1- go from +/-1000 -> 1800 mm²/m when you change this option?	the topic "Shear Effect" continues to be our "evergreen", and in a special sense it is a satisfaction to me, after I have had spent many, many time in trying to understand its real meaning, generalizing the 1D idea to the much more complicated 2D case and developing the functionning NEDIM algorithm, which has been in use under SEN since 2000. On the other side (of the positive feeling of "satisfaction") there is a feeling of being tired to explain things again and again. You know, the 2D matter is very time and energy consuming - thousands of values that have to be checked and understand when following a DeveTrack Ticket with his hints about suspected" defects of the NEDIM Shear Effect algorithm. Well, I do it this way:(0) Please, understand that I cannot start numerical invertigations to respond exactly to your (partially vague) hints! I would need, as ever, a particular element/node with the values there encountered and direct hints as to why you mean they are wrong;(1) To your statements formulated in the "Description" field above I can, however, take position:(a) When the bending moment function attains its extreme (max/min), the corresponding shear force vanishes (V->0). Well, but this is true in a 1D case. (b) In 2D, the state of stress is characterized by the tensors (mx, my, mxy); the shear forces are derivatives, as known from the Plates theory, but the matter is, of course, more complicated. In the standard Mindlin-type FEM model, the shear forces are, by definition, independent functions, i.e. there is no (direct) differential connection between the moment and shear quantities at all! So what... (2) The best reference to the Shear Effect theory and algorithmic background you'll find in the reference [HOBST], attached below. As a fact (I believe that this is a fact), no othe software has this Shear effect algorithm implemented! CONCLUSION: I declare the Status="No Bug". However, you can reopen it, i.e. reset to "OPEN" and supply me with concrete data - element/node, Load Case or LC Combination (please, not Class, this is too "much" for testing) and your hand calculation...
HWRE-984J73	Wir haben bei uns den Namen für das neue Dokument abgestimmt	"Altes Dokument" dürfte von so manchem Kunden doch missverstanden werden. Mein Gegenvorschlag: in Anlagebild >DesignForms.PGN< ("New attachment") finde ich den Begriff "Dokument im alten Stil", was als Equivalent zu "Old style document" gemeint ist. Was ist damit? Ist es das "Alte Dokument" oder eine dritte Dokumentart? Bitte um Erwägung bzw. Erklärung und abschließenden Vorschlag!(2) Gemacht - danke für den Hinweis!10.06.2013Final solution for "Engineering document":(1) Instead of "Ingenieruprotokoll", the translation "Ingenieurdokument" ist in use;(2) The term "Altes Dokument" is not implemented at all. There is the only appearence of "Dokument im alten Stil" in the Vocab file (see >DesignForms.PNG attached<, yet this is the direct translation of "Old style document";(3) The appearance of "Protokoll" is almost everywhere replaced by "Document", consequently12.11.2013Final solution, as aggreed upon by the German team, including the Bug Owner: "Ingenieurprotokoll" --> "Berechnungsprotokoll".EH

Ticket number(s)	Bug Description	Solution
HWRE-98DD5S	Table "Loadcases" don´t fits on page of ER, see attachments. Please shorten somehow the "Vorherrschender Lastfall" in the translations.	"Vorherrschender Lastfal" -> "Vorherrsch. LF" geändertEH
JPOL-99EHRK	Issue with wind zone coefficients (Cpe) in 3D WLE. Attached project contains roof with angle 5,7°. For wind directions 0° and 180° there are zones F, G, H, I and J. The table 7.4a in EN 1991-1-4 specifies values of Cpe for these zones and these are applied correctly. The code also says, that there should be taken into account 4 combinations of maximum values in zones F, G, H with maximum values in zones I, J for roofs with angle between -5° and +45°(notation 1 under table 7.4a of the code). But there is also a sentence that there should't be mixed positive and negative values on one side of the roof. However, this is what we can see in SEn, for example LC 3DWind1 (see also picture attached). Why? Tested in 2013.0.122	Positive and negative values are not mixed if it is possible. In case of 5deg there is Cpe for I zone only negative so it is used always. It is correct according CADS who produce WLE which is used in Scia Eng.
NWEB-99QHRG	copying of the last loadcase (BG30....) takes a very long time. Exporting, cleaning, etc. does not help. What is wrong here ? Is the project file corrupted ?	fixed in 13.1.36
CSCT-99UFRK	Übersetzung: Anbei ein Bild. Als Befehl steht zur Bildübergabe an zweiter Stelle "Bewegliches Bild in Ingenieurdokument". Als Vorschlag für "Bewegliches" hätte ich "Generierbares" oder "Aktualisierbares", aber vielleicht fällt Dir auch was besseres ein... PS: Ingenieurdokument heißt schon Berechnungsprotokoll im nächsten Patch, oder?	01.11.2013(1) Ingenieurdokument ist bereits überall durch "Berechnungsprotokoll" - darauf hat sich das Gremium nach Beratungen, die sehr ernst genommen wurden, geeinigt;(2) Bewegtes Bild -> "Aktualisierbares Bild. Danke für den Hinweis!EH
RMAA-99QF4U	The deformation is deleted in service concrete if you select all layers and switch on check box Current used activity. Tested: 2013.0.122	The problem was solved and tested in version 13.1.28
NWEB-99WE6Y	Problem with 3D Wind Generator., After you try to run this tool, following error message is displayed:, “Exception while calling GenerateBuildingInfo: Index was out of range., Its value must be non-negative and less than the size of the collection., Parameter name: index.” ,	fixed in 13.1.47 + WLE44
NWEB-99Y5LB	I have attached a model in which I am receiving errors when attempting to run , the 3D wind load generator. I modeled the plates first to make sure they were , planer elements and when running the meshing and test of input data I receive , no errors for non-planer elements. Ultimately, I am concerned about Scia , Engineer’s lack of tolerance. The issue may be that there are some , nodes/planes which have a uz = 0.00000036 when selecting a specific roof as , working plane. This seems a bit insane to think that the software does not , have a tolerance and requires perfection. It is also possible that I am , missing something and there is a way to solve these issues. Any help in this , situation would be great. The file is for a client that is using the Eurocode , and he hired me as a modeling consultant on the project. Thank you. "	see attached docummment from Devi and check the structure; (divide LP137, LP144 and tick off 3D wind for Member2d#302174); tested in 13.1.47 + WLE 44

Ticket number(s)	Bug Description	Solution
NWEB-9B7HQH	Open project in attachment Calculate it Go to 'Concrete -> 2D member -> Member design -> internal forces ULS' (class is already set to 'Alle UGT' and selection is already set to current) In the middle of the plate, n2- can not be bigger than 1400 kN/m. Now calculate the theoretical reinforcement ULS for the same settings From the theoretical background, we would expect the reinforcement to be 1400000/435 = 3218mm²/m. But instead, Scia Engineer gives 7200 mm²/m. So what are we doing wrong? And how is it really calculated?	The explanation of the large difference between the NEDIM and the "handmade" reinforcement values comes, again, from the "evergreen" notion of SHEAR EFFECT. In the user's setup of the example ESA file, the (interal) NEDIM parameter ISEF was used by its default value ISEF=1,i.e. "Shear effect active". Thus in regions, i.e. the elements and nodes of, with high values of the shear force vector vdim, the effect of shear stress upon the longitudinal reinforcement courses, that act as the 2nd-course shear reinforcement, complementary to the "vertical stirrups", may become decisive, i.e. the amount of the longitudinal shear reinforcement may exceed that of the "normal" m/n design !!!!! This is a notoriously known fact which can easily be shown on particular examples.(2) For the purpose of the demonstration ad (1) I use the inner node 3 of the finite element 21 and analyse it under the LCC Class "ULS". In order to reduce the amount of data I restrict the demonstration to the reinforcement course As2+:(a) the "normal" m/n design yields the value As2+ = 87,8 [cm²/m]. This corresponds to the 11th LCC (of 16) and its 6th single Load Case (i.e. min. mxy);(b) the corresponding shear force is V,TRd = 7759.3 [kN/m] (very high!). Since the shear strut angle is set 45°, the secondary (i.e. longitudinal) shear force to be resisted by the upper/lower net reinforcement is F,cd = V,TRd = 7759.3 [kN/m] as well;(3) The NEDIM Shear Force algorithm distributes the longitudinal shear force F,cd to all 4 reinforcement courses following a unique algorithm, as described in ref. [HOBST], see attached below. From this transformation, in the following 2nd design step the final net reinforcement is calculated:As2+,sef = 167.2 [cm²/m] as final result. Note: all numerical values were taken from the s.c. "Test Strategy" run of NEDIM, itest=13 and itest=32.(4) A rough numerical control follows from the direct formula delta As´,tot = F,cd/fyd = 7759.3/43.5 = 178.5 [cm²/m]. However, this is just a comparative value - F,cd has to be distributed according to the algorithm as described in [HOBST];(5) The attachments on "SEN run ISEF=0" and "SEN run ISEF=1" demonstrate the typical differences of the ISEF control;(6) Important hint: The Shear Effect is to be considered in regions which are designed to "genuine" Shear force, as stipulated by the Norm. In regions with PUNCHING, that are designed according to the rules for Punching, the Shear Effect has no impact on! The punching design follows other rules.CONCLUSION: Status set "NO BUG" - please approve !!Dr. Eduard Hobst
NWEB-9BAEKS	strange time history settings in calculation protocol	tested in D4 12.004.217 ==> OK
JBES-9BDAZ4	Try to import the .r2s (Revit to Scia) file in Scia Engineer. Scia Engineer crashes every time. (the .r2s comes from the Revit file in attachment, but without the columns, because there is not yet a correct material attached to these columns, but the 22 framing members are in the .r2s file, and should be imported in Scia Engineer)	there is a message about bad reading property - D04

Ticket number(s)	Bug Description	Solution
NDES-9BDQ6B	Look at the wordfile for the problem description. Add the units for stiffness	Units was added - 13.1.28. The deformation could not be the same as the input data are not the same (see comment). Not possible to compare orthotropy with two height and "standard" stiffness. You also see the D coefficients are not the same as for slab E2 are for each height separated.
JPOL-9BNCB8	Regular crash according to user: It is regular crash on user´s computer, when you go to service concrete and run member design/design.	Problem solved in R_13.1
NWEB-9BNHR9	Crash report - Exception in the Eng report	The problem should be solved in 2nd patch of R_2013
FLFA-9BNRWV	Crash report - Engineering report	Problem was in InBox items. Test of its content was improved for R_13.1
JPOL-9BPAR5	Crash report of Engineering report, see attached esa file and correspond	Problem identified and solved in R_13.1
NWEB-9BQDF3	Crash report - Crash before of calculation	Problem identified and solved in R_13.1
DPIS-9BVDYN	Problem with displaying of pictures - Opening of the report takes 30 minutes - Once it is opened and Fast picture preview is switched OFF, the preview of report is completely stucked after some small scrolling - Unable to save presentation data to cache	Displaying of pictures was significantly improved in version 2013.1. The rendering is now done using the graphic card which reduced time necessary for displaying of rendered pictures
NWEB-9C2PN6	Customer crashes during adding of new loadcases. But on my PC it don't crash. Can you see from the crash report, what is the reason?	The problem already solved in R_13.1
JPOL-9C4AZF	Feedback from the training - resulting in bugs or at least proposals: 1. Colours of background of concrete setup items are not visible when using beamer. This should be changed to more clear graphics (not necessary just some light shade of colour) . We (Radim and I) are even not able to agree what kind of colours are there - blue and green? gray and green? some dirty white? 2. I was trying to input stirrup distance (in the action button dialogie) in milimeters because I didn't notice that meters are required. The message that required is value between 0,000 and 100,000 convinced me that my margins are correct but I was not sure, why exactly 220 cannot be inputted. This results into two problems: Why is the input in meters when diameter, cover and other parameters of stirrups are in milimeters? Why is the limit for stirrup distance absurd 100m? Tested in 2013.0.1036	1.Standard color settings are in Setup, Options, Enviromnment, Skins, Select skin available. 2.Units [m] are ok in this case. These values are in range of value of the beam length. 3.Limits - have been changed to max. 1 meter.

Ticket number(s)	Bug Description	Solution
JPOL-9CKCAG	Soilin cannot finish with defined number of iterations in attached project., Is the reason that user combines soilin support and flexible supports in node? Without flexible supports it ends after third iteration., If so, how to proceed? Change soilin support to surface support of type "individual" with some defined flexibility?, Tested in 2013.0.2030	when you display the generated mesh, there are lot of members overlapping, so the geometry is not correct..
ISCS-9CKD7Q	526 MB consumed by LineLoadOnLines See project in attachment. This project has a great size, but why? The project data are not that many, and also, after deleting combinations it still has a great size. When I try to do some things in the project, it is very difficult, because it crashes. For example, a crash report in attachment when I tried to export it.	See the comment from Lenka:Problem can be solved by changes in mesh settings. Option "Definition of mesh element size for panels" has to be changed from "Automatic" to "Manual" with "Average size of panel element" left set to 1.0. I've tested it in simplified project (attached in this comment) with only one load case and after this change amout of line loads was reduced from about 29000 to about 5500 (equally has changed size of *.69 file).
JPOL-9CLGFX	Please explain the following load generation. On attached picture I highlighted load FF85 (in load case LC24). However, there are many generated loads far from this one with reference to FF85. Why? Next - if you take a look at the mentinoed load case LC24 there are a few grey free loads (e.g. FF91 or FF94). Why are they grey? Is the problem that geomtery system is set to GCS (and not LCS of entitites) and the load panel is inclined? Since it is snow load it has to be in global Z direction. Tested in 2013.0.2030	Remove the calculated incorrect load and run generation again. Then the load is correctly generated. Maybe there was some issue which is now fixed as I am not able to create the incorrect data on the last patch. The gray load is results of Accurate method where only to one load can be linked all generated one - this issue could be fixed in 2014
NWEB-9CJE7R	siroke tabulky	Solved in R_13.1
NWEB-9CLGXD	Fonts colours and heights are not fully applied to all cells in old document tables. See attached pictures and esa file too (althought it is not the source file of pictures) and see that advance setting of fonts is not applied to all rows or columns. Is there any chance to repair it and reach unified appearance of these tables? Tested in 2013.0.2030	Let's try to increase size of fonts marked on attached picture
NWEB-9CQB37	Load Panel in attached project "Wustec_scia1.esa": with setting "Exact (FEM)", no line loads are generated. With setting "Standard", the line loads are generated. (Load case "verkehr").	it works in last patch - tested on 13.0.2030
NWEB-9CPLGF	Look at the wordfile for the problem description.	There are mixed two settings: i) Path to the folder containing licence files (*.LIC + *.e2c). This folder is common for all versions of Scia Engineer and by default is something like this "C:\Documents and Settings\All Users\SCIA\Licence\". This is what are you changing on your screen shots.ii) Path to the folder with selection of modules (*.cms). This folder (with selection of modules) is different for each version and it does not have any explicit setting. It is always in the USER\Protection\Sets.For this customer just check, whether he has different location of USER directory for each version of Scia Engineer

Ticket number(s)	Bug Description	Solution
NWEB-9CRCM4	Steel: Crash for empty NL Combinations	The crash is caused by the fact that there are several Non-linear combinations which are empty. The Steel check tries to get results for those but can't, leading to the crash.Empty combinations cannot be inputted, so most likely this occured when load cases were deleted. That way they are removed from the combinations, leaving empty combinations behind.The solution is thus the following: remove the empty Non-linear combinations (9-31) and then the check will work without issues.In version 12.011.338 the crash itself is fixed, verified in R13.1.45. Now in case of an empty combination the check returns an empty result i.e. the check is not run.
NWEB-9CQE7T	Issue in calculation of Vwp,Rd in attached example. Please see connection Pol.2 in N5 and compare the result in SEn (chapter 2.1.1., 683,49 kN) with my hand calculation of Vwp,Rd: Av,c = 5380 - 2*150*11 + (7+2*15)*11 = 2487 mm2 (for IPE300, column B3) Vwp,Rd = 0,9*355*2487/sqrt(3)/1,0 = 458,76 kN Where is the difference?	The calculation you have made using formula (6.7) gives Vwp,Rd for an unstiffened column web.In this project however the user also applied a diagonal stiffener. This implies that the component 'web panel in shear' is not limiting.To make it 'not limiting' its resistance is taken equal to another component of the shear/compression zone, namely the beam flange and web in compression.As the output shows, this is where the 683 kN comes from..As a minor remark, the Avc you calculated manually is not fully correct: make sure to use sufficient digits for the CSS dimensions. The flange thickness is not 11mm but 10,7 mm and the web thickness is 7,1 mm instead of 7 mm.Shown in ticket: Av,c = 5380 - 2*150*11 + (7+2*15)*11 = 2487 mm2 Exact: Av,c = 5380 - 2*150*10,7 + (7,1+2*15)*10,7 = 2566,97 mm2
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-9CT9QU	The problem with text at pictures (display mode: wired) during export to word from documents. Tested: 2013.0.2030 and 2013.1.23	I have not succceeded in exporting of picture properly. It is possible to export similar picture to RTF properly from ER (2013.1)
NWEB-9CTFRV	see document -> what is this strange error ? did it get corrupted ?	It realy seems that data were currupted. It cannot be restored. Some items were indented under the Table fo content?
NDES-9CTGJN	Tested in Scia Engineer 2013.1 Open the model in attachment and go to "Steel -> Connections -> Connections setup" and activate here the option "Use last bolt only for shear capacity" (see also printscreen) Click on okay, wait a few seconds and Scia Engineer will crash	nexis part alredy fixed during JPOA-9CHHK6
NWEB-9CTC8N	Connections: "An exception occured during calculation" It is not possible to look at the results for the connections, for example connections Conn5.3 and Conn5.4	Solved in DEVE 11 build 12.11.338The check is run without this exception message appearing. Verified in R13.1.45
NWEB-9CW9GK	A client is asking if Scia Engineer could handle a project with 60 000 2D elements. Is there a limit to the number of members in Scia Engineer ?	There is no limit of number of elemets. With a really big project can be (and probably will be) a problem with not enough memory. It does not depend only on number of element but also on number of add data, mesh size etc.

Ticket number(s)	Bug Description	Solution
NWEB-9CTJ5E	Crash Engineering Report: see crash report and warnings in attachment. What causes the crash?	Crash caused by blocking of file in TEMP folder by antivirus or some other program. Exclude the TEMP folder from antivirus checking (or by protection by the other software) or upgrade to the 2nd patch of R_2013. There is used safer deleting of file.
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-9CWFCN	When calculating this project linear, you will get a singularity (see project: ND_Netconstructie kroonluchter 2 (met 3 steunpunten op onderring).esa). But I don't find why. Do you have an idea how to solve this?	very nice structure :-)is that a candidate for the next user contest?this is a practically "pure" cable structure, so to say. I would not dare analyzing that linear. Not if I want to obtain meaningful results. However, if you really want to do it, change the FE setting from "Axial force only" to "standard" on all 1D members and run the analysis.You will see that the deformed shape is unrealistic, involving important transverse displacements in some parts of the structure. Those transverse displacement can be balanced only by bending in linear analysis, which is not the strong point of that type of structure.In nonlinear analysis, the transverse effects will be balanced by the 2nd order effects of the axial forces (not possible in linear analysis),giving a totally different distribution of displacements.
NWEB-9CWJ4S	Attached a docx with two pictures. If we have edit the buckling-values in row 1 (f.e. kyz from 1 to 1,2) and then select row 3 the kyz-value is transferred into row3. If we deselect row 3 the value jumps back to 1,0. Other values are fixed f.e. value k. In my opinion -> dangerous.	This is in fact intended behaviour which is related to the spans in the buckling system.According to the first picture you have a buckling span from point 1 to point 4. By activating point 3 you get a span from 1 to 3 so point 3 takes over the coefficients from 1.
JPOL-9CXAQZ	Crash report - Crash of SEn during work with document. Please see attached error	The crash was caused by memory overwriting during opening of old document. We were not able to identify which entity caused the memory overwriting.
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.
NWEB-9CYDSM	Problem with crash of SEn 2013.0 Please see attached error message and issue description. Is it caused by a graphic card? Should the user change rendeting setting?	The message was generated by the graphic card driver. I would recommend to try different version of the card driver (newer or older). Or try to change the setting of hardware acceleration in the setting of graphic card

Ticket number(s)	Bug Description	Solution
ISCS-9CYCLU	It is not possible to have different view parameters for elements in the picture gallery? For example, if we want to see 1D elements behind a 2D element (see small test project in attachment). First step: drawing the 1D elements rendered with edges and send this picture to the picture gallery. Second step: adjust the model and draw a transparent 2D element to see the underlying members. When regenerating the pictures in the gallery, they will have the same view parameters. ========== Testen: een deel van de structuur tekenen (gerenderd met lijnen) > figuren in de afbeeldinggalerij zetten, vervolgens in het model een plaat overtekenen, MAAR transparant zetten zodat we de onderliggende staven nog kunnen zien > bij het hergenereren in de afbeeldinggalerij, wordt de plaat wel getekend, maar gerenderd, dus kunnen we de onderliggende delen niet meer zien (laad activiteit staat aangevinkt).	the last used VF is always used for the picture when it is refresh, if user add some new element, then it is displayed according to VF which was used during creating the picture, if user want something totally different, the new picture is needed
NWEB-9CZE78	see galery -> monodrawing -> side view of connection -> label of the beam : it says Iw (300;12;300;12;...) Compare this with the profile in the project: here we have an Iw of 800mm high. The label of the beam is not correct ?	Tested in 13.1.30. When creating a new mono-drawing or updating the existing one, the incorrect information is removed from the drawing. The label thus indicates "B2[Iw]" without any incorrect dimensions. This is done due to the fact that an arbitrary member is used (the CSS changes over the beam length so no 'one' CSS can be displayed).When using a uniform Iw section (see screenshot) the full data of this section is shown with the correct 800 mm height.So in summary, specifically for arbitrary members the incorrect information was removed from the label.
NWEB-9CYLGF	Open the attached project then display detailed output for EC check on beam B5419 for example. This beam has been defined as a CHSCF 610.0/30.0 cross section But if you look at the preview, the check is done on another cross section CHSCF 457.0/25.0 This happens for many beams not only that one	Tested in R 2013.1.30I'm getting correct results for B5419 as shown on the screenshot in attachment. Possibly on the user's machine there is an issue with the temporary files for the profile library.For R2013.0: When SEN is closed, empty the user/prof folder and also empty the c:\temp Then start a new project and use 'tools > convert profile library' to recreate the cache files.Then re-open this project, calculate it and execute the check.

Ticket number(s)	Bug Description	Solution
NDES-9CYBPS	The attached project shows a problem for welded connection in the weak axis. Using the result button, all cheks are put to 0.	This has been solved already in R 2013.1The check in fact is not zero but quite high due to the fact that EN 1993-1-8 requires an M+N check. For this geometry (weak axis welded connection), there is no Nj,Rd, leading to an infinite check.In R2013.1.30 the following occurs:- When N < Npl,Rd,beam there is no need for a M+N check, Nj,Rd is then not determined.- When N > Npl,Rd,beam in essence N+M needs to be checked but there is no formula for Nj,Rd so the N+M check is set to 999.Note: The fact that the check values so 0,00 is in fact correct, for LC1 the moment is 0,01 kNm compared to a resistance of 7,26 kN. The shear force is 0,25 kN compared to a resistance of 190,17 kN so the unity checks are very very small leading to the print out of 0,00.
NWEB-9CZEYR	See questions in attached pdf	Response to the two questions posed in the attachment file >Questions.pdf<:(0) The Crack Proof acc. to SIA 262:2003 follows the stipulations in Chapter 4.4 "Vérification de l'aptitude au service". The user's enquiries concern the Paragraph 4.4.2.3.4: "Les buts recherchés, les actions et les exigences figurent dans le tableau 16. Les contraintes admissibles se trouvent sur la figure 31."(1) The 1st question is, as a fact, oriented on the Crack Proof acc. to EN 1992-1-1:2004. The answer is: The Crack Proof procedure acc. to SIA 262 is quite different from the Eurocode! The Crack control under SIA 262 follows the "Tableau 16" and the "figure 31";(2) The "Load case processing attribute" control presents the choice of the 2nd, 3rd and 4th row of the Table 16, SIA 262, however, in the order (3,4,2). The 1st row, concerning the Brittle fracture" control (Défaillance fragile) is respected automatically during the design process.Important note: the 2D reinforcement design module NEDIM, running under SEN, expects the choice out of the Table 16 for each LC Class individually, i.e. generally different controls for all LC Classes which start simultaneously. However, the SEN control (as presented in >Questions.pdf<) disables the free choice from Table 16 for different LC Classes: only one such choice is possible for all LC Classes participating in one design run!
LSKI-9D7HU9	The value of TED, torsion on the beam printed in the detailed output for the ULS check does not match the value given for internal forces in beam. See attached pdf wich screen shots.	Please review the pdf document you have sent.The maximal torsional moment at 0m is given for combination ELU/1The Steel check however is executed for combination ELU/4=> This is thus a different combination and that is why there is a different moment .

Ticket number(s)	Bug Description	Solution
JBES-9D8EZP	If you use dampers in all elements (see example 12 on page 134 of the pdf in attachment) -> then does the CQC method still have any effect to calculate the correlations? The text that raised the question, is the following: << In the original example, a Damping Spectrum with a constant damping ratio of 2% was used. Due to the inputted dampers, the calculated Composite Modal Damping Ratios of 2,64% and 3,30% are now used. Using equation (4.13) the Damping Coefficients can be calculated: 76 As was the case in the original example, the damping ratios are lower then the default 5% used in the acceleration spectrum, they will have a negative effect thus augmenting the response of the structure. Since the calculated damping ratios are higher then the original 2%, the response will be less when compared to the original example. Second, the calculated Composite Modal Damping Ratios will be used for the calculation of the Modal Cross Correlation Coefficients of the CQC-method. This will be illustrated in a manual calculation.>>	strange formulation... not quite sure that I understand the question. However: When using non uniform damping (i.e. material damping or dampers),the program calculates a damping spectrum which overrides any manually defined damping spectrum.That is, if a damping spectrum was defined manually wtih the CQC superposition settings, that spectrum is replaced by the computed one. Then, regardless of the way that damping spectrum was defined (manually or computed),it affects the seismic analysis in two aspects:- the acceleration response spectrum is adjusted by the damping factors which are calculated from the damping factor, for each eigenmode- the CQC correlation factors are calculated from the damping spectrum (when CQC superposition is used)
NWEB-9D8FCS	Crash report - Engineering Report in attachment. ===	The crash happened during closing the Engineering report but it was caused by some earlier action. It is not clear what caused the problem.
NWEB-9CQPMP	esa.05 and Engineering Report	Dobry den. Pro spusteni editoru Header/footer, Editoru Stylu a Editoru tabulek je potreba mit ESA.06. ESA.05 neni pro Eng report potreba. Viz prilozeny obrazek.492-Doc_X64_ADVANCED je prave modul pro spousteni techto editoru.
NWEB-9CXJ4T	Instability in non-linear calculation (problem comes from cables) The project is unstable in the non-linear calculation. The problem comes from B5503 (it is a cable element) and will probably also appear in all the other cables. I have added the investigation which I have done in the attachment in the ticket in c.support. Could you tell me what is wrong with the cables?	this cannot possible work, because of the hinges applied to eccentric beams.- the GluLam beams are eccentered and hinged at their ends- the cables are eccentered (-1m), thus attempting to apply an eccentric force at the hinged end of a beam(so to say, the beam if on the left-hand side of the hinge and the cable is attached on the right-hand side of the hinge)==> local mechanism for each eccentric cable==> avoid applying eccentricity to cable members or, more generally, to any axial-force-only memberHowever, be very careful when applying hinges at the connection node of an axial-force-only member (like a cable)Suggestion: those cables are quite long and their bending stiffness is very low, so it might be an idea to avoidusing cable members altogether and just applying an initial stress to those members instead.

Ticket number(s)	Bug Description	Solution
NWEB-9D8G6C	In the Engineering report, is it possible to delete the Nemetschek Scia logo which is printed in each page at the bottom right ? Will the option which enables the user to put 2 or 3 columns of the same table in one page be developped for the Engineering report ? When ?	Scia logo cannot be deleted. It can be just moved to some other corner of the paperNarrow tables can be splitted into more columns in R_13.1 (see attached picture)
NWEB-9D8KAW	See crash report of the Engineering Report in attachment.	The same problem as previous one. Crash during closing Eng report but the source of problem is in some earlier action. Similar problems should be reduced in 2nd patch of R_2013
HWRE-9D9F3E	I want to calculate construction stages with 2D elements, one part is on elastic foundation (see attachment) for a demonstration. In phase 1, only the walls on line support are there. In phase 2, the base plate is sheded. I get a messeage, that stage calculation is not possible together with elastic 2D foundation. Is there any way to get results for this example?	Hi Helmut,yes, that is not allowed, because the handling of surface supports had some conflicts with construction stages.Two possilities:1- use a grid of point supports instead, e.g. every 50cm - that would then be compatible with construction stages2- use absences instead of construction stages A remark, though: if the base plate is cast AFTER the walls, it usually means that it is actually NOT a foundation plate and it will carry only its own weight and loads applied directly on it... which means, that it could be modelizedand analyzed separately from the rest of the building. Well, ok, that would be the "standard" case. Now, it could always happen that some strange things are built ;-)
NWEB-9D8J4U	Error report from crashing ER. See attachment.	Problem during closing of Eng report. This problem should be reduced in 2nd patch of R_2013.0
NWEB-9D9NU2	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)
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

Ticket number(s)	Bug Description	Solution
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-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. Thanks. " " I think a general question we should ask is, what should the configuration be if the normal Scia users only has read/execute privilege in c:\program files directory. We know Engineering reports has issues, what other directory location should we address, and how would we customize them for mass installs? "	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)

Ticket number(s)	Bug Description	Solution
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)
JPOL-9DDLA6	Combination MSP1 in attached project cannot be deleted in reasonsble amount of time. Please open attached project and go to combinations. Select MSP1 and try to delete it. First a dialogue appears that you are going to delete 262 145 entities, which is really suspicious. Why so many entities? If you confirm that you want to delete them, SEn frozes. I waited about half an hour and nothing happens. May b,e more time would bring the desired results but is it really necessary to wait such a long time to delete one combination? (Workaround is to change this combination n to linear one (when it contains only 2 entities) and delete it. Tested in 2013.0.2030	The slowness is indeed caused by the fact that there are > 200.000 backreferences to this combination. You can visualise them in 'Content of combinations'.This can be fixed automatically by SEN when using Check Structure. Note that this takes a while to run.Most likely the user created/calculated these combinations first with different settings. I noticed several load group relations are 'exclusive', most likely initially he has used 'standard' leading to the > 200.000 combinations.After running Check Structure these old references are removed and the behaviour is 'fast' again.
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.

Ticket number(s)	Bug Description	Solution
JPOL-9DLC4V	Surface support in pressure only - how can I model it? Take a look at attached example project where I have modelled two slabs loaded similarly and supported by surface support of type individual with nonlinear function assigned to C1z. However they differ in nonlinear function. One slab has got fixed positive end of the function (so that it cannot lift up) and the second one has got free positive end (to demonstrate support in pressure only). Results on second one are suspocious since they are not symmetrical. Also, calculation does not end with standard message but when maximal number of iterations has been reached. Can I believe such results? How can I model surface support in pressure only (not using soilin)? Please see attached project and pictures. Tested in 2013.1.23	The way this has been modelized is a bunch of singularities. It is bound to have big convergence issues.A few hints:- never modelize surface supports with "infinite" stiffness; the jump from 0 to -0.5 MPa is a big numerical difficulty for the analysis. Insert a small displacement there (e.g. 1mm)- a strictly horizontal branch is also a difficult one to handle; rather apply a small slope there (e.g. -0.51 MPa at the end of the "plastic" branch)- the particular configuration of the structure (applied load higher than the resistance of the subsoil, rigidly blocked surface for upwards displacement in the first slab...) causes a structural response that might be surprising ;-)- always give a non-zero value for C1Z; that value will be used for the first iteration of the analysis; leaving it = 0 will make the convergence even more difficult; try to input there a realistic value, for instance the slope of the first segment of the NL function.- when using NL functions with subsoil, always ZERO the components C2X and C2Y. Those are NOT nonlinear and, since they are directly connected with the same displacements as C1Z, will heavily polute the behaviour of the support.- Newton-Raphson is not necessary for this type of analysis. Just disable geometric nonlinearity. Loading increments are also not necessary here. This is not heavy nonlinearity. With the settings above, the analysis converges fine and the results are clean