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

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NWEB-95XB3K** | I don't know if you are the responsible person.<br>When importing the .stp file in Esa Prima Win, I will get the printscreen in attachment "EPW.jpg". Here 2 cross-sections are orientated wrongly in comparison with other files (see pdf).<br>But when importing this in Scia Engineer 2013, I don't see any beams at all. Can you please take a look at this? | import ok in 2013 (13.0.22) - see picture in comment |
| **NWEB-976C8V** | Customer crashes with many projects in Document. I tried it, but on my PC it works well. | Antivirus |
| **RMAA-97QGG5** | Crash report - Accidental crash<br>Accidental crash | possible cause of crash identified and fixed (Deve_06). it will be available in nexr R_patch |
| **NWEB-97SCMB** | Excel Check: Multiple messages from Excel during Check<br>ISSUES: Custom check<br>1- Detailled check of a Class -> you get several (+-8) messages with the question if you want to save your excel -> this makes this check not user-friendly and time-consuming to use<br>2- Crash when exporting (to html, rtf, csv,...) detailled output, after pressing the export button<br>(3- The value of the Global extreme on the screen, does not correspond with the numerical output) | 1) The multiple messages during check execution (Detailed output) were caused by newer versions of MS Excel. A modification has been done in DEVE 11 (merged to R 13.0.2009) to avoid these messages from being displayed. @ 2) When I make a correct otx (nice evertical table, not a huge horizontal table which overflows the page width in an extreme way) then the .rtf export works without issues.@ 3) The check is executed for both a class and a ULS combination together, so this can be related to the unexpected difference numerical >< graphical. In any case, I have tested the custom check on B73 and the Detailed output + Graphical output matched. |
| **JPOL-97WB3Z** | Explain the attached picture of a member check, please.<br>The maximum value on the screen as well as in Preview is 0,84. However, SEn says in the Preview table it doesn't satisfy and also the warning in the graph (E735) signifies it is not OK. So what is the result? Satisfy or not?<br>Project attached. It is an overall check of members B93 and B110 for combination CO1.<br>Tested in 2013.0.84 | I tested in version 13.0.95 and the results in numerical and graphical output are the sameVAL: 24/09/2013: The problem was solved and tested in version 13.0.2020. |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **JBES-97YCH3** | Issue As,torsion:<br>Look at project in attachment.<br>Calculate, then go to concrete -> Theoretical reinforcement, refresh (for Combination = ULS, Value = As)<br>In the preview, you will see that for As,t he finds 97mm² for the top side, and 131mm² for the bottom side. This is because he uses different combinations for both sides. The program takes a combination where he looks at My and N and then decides on the combination to calculate As, while he should look at the different As that are calculated and he should take the maximum afterwards.<br>If you check more in detail, you will see that for the top side, he used 1,00x (load case with torsion) and for the bottom side 1,35x (load case with torsion)<br>Can this be changed so that the theoretical reinforcement takes max(As) instead of max(forces) to determine As,max in a section? | The problem was solved and tested in version 13.0.2020 |
| **NWEB-984PNJ** | Problem with parametrization of polygonal CSS<br>Look at the project in attachment, one parameter (integer) has been defined.<br>No go to the cross-section dialogue and change the value "n" into this integer (see also image [3]". But when choosing for this parameter, the value will be input but the parameter will not be assigned to this value. Can you take a look at this? | Solved in Deve_11. Fix available in next R_patch |
| **RMAA-985JWX** | Regular crash at punching Select view flag on picture in attachments. | The problem was solved and tested in Deve_06 (version 133) and in version 2013 (1034) |
| **NWEB-98DE74** | Issue installation.<br>customer just installed Scia Engineer 2013.<br>When he launces a predifined template, then Scia Engineer crashes (see dump files in attachment). Any advice? | Predefined shapes files were converted to 2013 |
| **NWEB-98DGXB** | Crack check for arbitrary beam | The problem was solved and tested in version Deve_06 (12.006.161) and too in version 13.0.2020 |
| **NDES-98NK96** | Everytime when calculating the project, I receive the printscreen from attachment and the solver will crash. | The solver crash is caused by incorrect data in the cross-sections. The cross-sections in question are CS188, CS236 and CS245.For these thinwalled cross-sections, the centerline calculation is not able to return calculation results. Closer examination shows that the issue is in the input.For example, CS188 in the lower left corner has two Points P3 and P5 with different coordinates: -573,124 and -573,123 mmThese nodes thus differ by 1e-3 mm which leads to a segment with zero length between them. A segment with zero length is impossible to calculate in the centerline theory..=> Conclusion:a) In case of such incorrect input, from now on the centerline calculation is not run anymore. The standard (part 1 - see TB) calculation is run leading to filled properties for these sections so the solver does not crash etc. (merged to R 13.0.2009)b) The root of the issue is in the input: the user should review the three mentioned sections and make the needed corrections to have a correct geometry definition of those sections. |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NWEB-98NBEX** | Open the project, select the load and change its value, not from the Loads menu<br>You can see that the result menu do not disappear and you still can display results even after modification. | fixed in 13.0.2003 |
| **NWEB-98KGKC** | Crash report from Engineering report | Problem of crash identified and fixed. Fix will be available in next R_patch |
| **NDES-98FJ2T** | AutoDesign: Difference between Advanced AutoDesign and single parameter<br>18/06 PVT: Please view the file in attachment.<br>1. Go to the Steel Service and perform AutoDesign for section Traverses<br>2. Select "Advanced Autodesign" and set 'AutoDesign' for the parameter of the I-section and for the parameter 'thb'. For 'thb' set also the use of list 'Toles'<br>The point is that 2 parameters are selected for design<br>3. When pressing 'Search for optimal', nothing changes, the section remains unchanged.<br>4. Now modify the Parameter combo from "Advanced Autodesign" to 'thb' and make the same settings as before for 'thb'<br>=> Now 'thb' changed from 20 to 3<br>=> Why didn't this change happen in "Advanced AutoDesign" also? | This example has been debugged, the following information was obtained:Lets start with simple autodesign of the I-section part. Just by pressing Down / Up button we can see the results of the check for different steps (please note that sort is according to height):IPE450A - 1.269IPE450 - 1.020IPE400 - 0.747IPE500A - 0.912IPE500 - 0.755IPE500O - 0.569IPE550A - 0.683=> It can be seen that, due to the sorting by height, the Autodesign 'jumps' i.e. it goes from high to low to high etc..Now lets look at the check result for thb (using list Toles):For all thb values the unity check value is 0.755 i.e. the unity check does not change..In this section, we start with IPE500 + 20mm, for that case the unity check is 0.755Advanced AutoDesign changes all parameters one by one and evaluates the influence to the unity check.In this algorithm is however the following condition:We only continue in case the start_check - actual_check < 0 or in other words, the actual_check has to be bigger than the start_check.This condition was specifically implemented in the Advanced AutoDesign and is used only in case the UC is smaller that 1..The logic is the following: when the check is bigger than 1,00 the Advanced AutoDesign modifies the parameters with the biggest influence to get below 1,00.When below 1,00 the algorithm tries to get as close to 1,00 as possible i.e. it moves back up. It thus does not allow parameter changes that would lead to an even lower check..The standard AutoDesign does not have this feature. |
| **NWEB-98RKPU** | A research in "Stahlbau Teil 1: Grundlagen" von Kindmann/Krüger shows that the plastic interaction formulas for double symmetric I sections, given in chapter 6.2.9.1 (N-My \| N-Mz \| N-My-Mz), are on the unsafe side. Other rules are described for saver checks. See scans; did we hear about it? | There are several publications with new research regarding the formulas used in Eurocode. Beside this one regarding the interaction equations there is for example new research in the classification limits, the behaviour of Class 3 sections, new plastic interaction equations for angle sections, new rules for class 4 (buckling) effects of tube sections etc.It is expected that each of these researches will eventually find its way into a correction sheet or amendment of the Eurocode. At present we add these researches into our PBD database, in a future release they might be included, but this is not directly planned at the moment.Note that the software is fully up to date according to the latest version of the Eurocode. |
| **JBES-98SJ37** | Engineering report:<br>no point, line or surface loads visible in the engineering report !! | Is possible in 2013.0.112 |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NWEB-98RBUL** | convergence/instability issue in nonlinear analysis with hinges and cables | I'd suggest to set those hinges with uy = flexible, giving them some small stiffness value, e.g. 0.01 MN/m and see what happens.Please note, that if the program gives large displacement using that trick, it probably means, that there are out-of-balance forces in those "internal supports",which could be caused either by some modelization inaccuracy or some structural problem.In case it is the latter, it would mean that the real structure is indeed mobilizing some friction in those elements to ensure equilibrium...(and honestly, looking at the amount of rust on the photos, I doubt that those supports can still slip freely ;-) |
| **NWEB-98SG6Q** | Picture from gallery to old document - inpossible to change name | Solved in patch |
| **HWRE-98SD9F** | CSS: Initial shape from thinwalled representation does not account for left/right alignment | Modified in DEVE 11: The initial shape now accounts for the alignment of the thinwalled representation.Note: the initial shape must be de-activated and re-activated for existing cross-sections. Merged to R 13.0.2009 |
| **NWEB-98UE95** | problem with reinforcement in a rib<br><br>in attached project, after running function Check capacity of concrete beam/rib (B61), Scia displays following message more attachments:<br><br>But check calculations are performed and we can receive results. The proper reinforced template was applied – for T-cross section.<br><br>Could you please tell me why this message is displayed? | The problem was solved and tested in version 13.0.2020 |
| **NWEB-992J5H** | the Report is damaged - see Not available items<br>I tried to Copy-Paste correct items to new Report<br>There I got Crash when trying to switch to Named selection - see attached crash report | Solved in Deve_09. Fix will be available in next R_patch |
| **NWEB-994GKZ** | SOLVER CRASH at the end of modal superposition | fixed in R 2013.0.2014 |
| **RCCA-994RZ2** | Problem with the mesh | fixed in 13.0.2018 |
| **JBES-994CXB** | Issue protection / productivity toolbox (esa.06) | Problem solved. Availability of Table input avaluated after the change of the protection |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **LKGZ-94WHAA** | Attached a picture (and project). There is a problem with the pciture of diagonal-connection. | View have no sense in this connection type and isn't created anymore. |
| **NWEB-998N3T** | Open the project then go to Concrete Menu<br>Under 1D memeber, open Concrete Setup menu<br>Scia Enngineer crashes each time you do this. | The crashes is solved, I suggest to switch the unit to metric system and then back to imperial system for recalculation some values in Concrete setup |
| **NWEB-98UGK8** | Steel: Discrepancy in graphical and detailed output | Solved in DEVE 11, merged to R 13.0.2009.The difference was caused by calculation of C-factors for LTB combined with LTB restraints. The results in the Detailed output, graphical output and Single check are now all the same. |
| **RMAA-999D3E** | Unwanted copying of combinations happens each time I copy load case and confirm to copy also dependant items.<br>Try to open arbitrary project, create a few load cases and combinations. If you copy LC and want to copy also dependant items (you are asked by a message box), all combinations that contain that load case are copied. This was not the case in 2012 version. Please correct this behaviour.<br>Tested in 2013.0.122 | fixed in 13.0.2018 |
| **NWEB-998ELF** | Mesh generation error | fixed in 13.0.2018, 2nd patch of 2013 |
| **NWEB-99J5LL** | Steel: Moments in Simple Construction<br>12/07 PVT: Please view the file in attachment.<br>The check is executed for column member B751 for LC3.<br>At the top of the column an additional moment Madd,z is generated of 3 kNm<br>.<br>The shear force at the end of beam B750 is 30 kN<br>Column B751 is an SHS (formcode 2) with B = H = 200 mm<br>The eccentricity is thus: e = 100 + 200/2 = 200 mm<br>=> Expected additional moment: 30 kN * 0,2 m = 6 kNm<br>The question is thus why SEN shows 3 kNm<br>. | Solved in DEVE 11, merged to R 13.0.2009.An incorrect eccentricity was used. Note that the correct value is not 6 kNm as the user expects but 5,6 kNm. The shear force of 30 kN is at the end of the beam while the force to be used is those at position 200mm in the beam i.e. 28 kN => the additional moment is 0,2m * 28 kN = 5,6 kNm. |
| **NWEB-99JE73** | Open the project in attachment and draw a new column (in the menu structure).<br>It is not possible to snap on the intersections of the lines of the linegrid. (the snap setting to line grid is on). You can only snap on all endpoints of the lines of the linegrid. | fixed in 13.0.2018 |
| **NWEB-99H9GN** | Engineering report can't translate all texts, see attached rtf fi<br><br>1. result property Name is generated according to GUI language. This property should not be in the OTX (TLX) of the result table (I think that default OTX does not contain property Name)<br>2. part of header of user defined check is not traslated --> I think it is for Steel team | Problem with Excel check solved in Deve_11. Fix will be available in next R_patchChapter 19 - Internal forces. OTX of this result table contains property Name which is according to the GUI language. The default OTXs does not contain this property. It was probably changed by the customer |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NWEB-99JCM4** | Two bolted diagonals are made in the attached project. When clicking on 'results' of a bolted diagonal we receive an error a few times: 'Enter a number' and next a table with not all of the results (see screen shot).<br>As I have already discussed with PVT, this has something to do with the German National Annex.<br>Can you take a look at this? | temporary set default in the case of zero gammas |
| **NWEB-99PJWY** | The external and internal coefficients for a pitched roof (5 degrees) do not seem correct | Provided values were correct. The axact angle of the roof is 4,99875 deg (see attached screenshot) which is less than 5 deg. In such case the design code sais that coeficients for flat roof should be used. To make it less tricky we aplied rounding of the angle to two decimal places. So newly the limit for flat roof is 4,95 deg. Fix available in next R_patch |
| **NWEB-99RFD6** | Engineering report x translation between Czech and English language.<br>Version 2013.0.122 | Formcode - 23/07 PVT: Defined as separate bug PVTO-99VHKQ"Německá DIN-EN NA" solved by TUM in Deve_09Psi factors - 23/07 PVT: Defined as separate bug PVTO-99VHKVAno/Ne - created separate bug for LOKLoad panels and Calculation protocol - solved by CAD-Solver Team in Deve_23. Fix will be available in next R_patch |
| **NWEB-99MN2V** | Engineering Report doesn´t display all item in calculation protocol.<br>problem is not in TLX: The content of calculation protocol is one IP file block. It must be something special in the source code. | Solved in Deve_09. Fix will be available in next R_patch |
| **GVAN-99VABV** | Either the drawing of the rendered geometry seems wrong, or the calculation is wrong --- the excentricity that is drawn does not correspond with the excentricity that is calculated.<br>See word document for a detailled description | fixed in 13.0.2018, the input of cut-outs does not reverse the member system any more, please defined for E14 member system at top as other slabs have |
| **NWEB-99VH3T** | Open the project "12-147 EEM-model lokaal assenstelsel.esa" from attachment and make first the whole modell visible with the activities. Afterwards click on "connect member nodes". And after this, click on "Check structure data". Click after the check on "OK" to correct the double nodes and afterwards on "continue" to delete the invalid positions and parts<br>=> And Scia Engineer will crash! | fixed in 13.0.2023 |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NWEB-99JE72** | For the file in attachment the decomposition of the torsional moment is examined.<br>The geometry concerns a cantilever, warping fixed at one end and free at the other ... with a local torsional moment at the middle<br>.<br>The model solution in SEN however is based on having the local torsional moment at the free end ... which thus leads to nonsense Mxp & Mxs values.<br>The Excel file "Warping_PVT" shows that, when the torsional moment is at the end, we give the correct result while when it's not at the end we have wrong results.<br>.<br>In the Excel I have also given a possible solution:<br>In the case of a cantilever with a local torsion moment, the length L used in the calculation should be the distance at which the torsion moment is applied.<br>Beyond that distance there is no torsional moment and thus all variables are zero.<br>But we need to investigate this under the debugger to see what happens at the moment. I suggest we look at it together. | This in fact concerned a missing functionality.As indicated in the diagrams shown in the Theoretical background, for a cantilever, the local torsional load is expected to always act at the free end.The example presented by the user, with a loading in the middle of the beam, is of a more academic nature and was not covered by the decomposition algorithm.We have now extended the algorithm for cantilevers to support 'n' local torsional moments along any position of the cantilever.This development was done in DEVE 11, merged to R 13.0.2009. |
| **RMAA-99VERZ** | Error report: Panels do not transfer loads. Do you know how to fix it? More error report in attachments.<br>Tested:2013.0.1030 | fixed in 13.0.2003 |
| **JBES-99WDNK** | Go to gallery, choose 'New by wizard', and generate reinforcement schemes for beam S156<br>(this issue is also mentioned in another ticket, but I am making a new ticket, because this part should be considered as 'bug' (while the other part was PBD))<br><br>For S156, there is no indication for the shear reinforcement. For S100, there is an indication for the shear reinforcement. Why is it different, and how to generate it for beam S156 ? | The generated pictures are based on View parameters, there is no indication in the model, so it cannot be in the picture also.How was it created? why is there no indication in the model? This is the problem, not the wizard.VAL 23/09/2013: The problem was solved nad tested in version 13.0.2020 |
| **NWEB-99VKBC** | copy of load case with dependent entities<br>It seems to be a problem when copy a load case.<br>If we confirm that we want to copy the dependent entities, the program copy the distribution and value of the load we've just copied and generates a new combination. Why the program generates a new combination without the new load case? It is possible to be implemented in the existing combination, or the new generated combination should contain all the loads? | fixed in 13.0.2018duplicate bug JPO13-19699GBSK |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NWEB-99WGJM** | I can not select plates to it when using the action button 'update 2D member selection'. | fixed in 13.0.2023 |
| **NWEB-99YJW7** | Load panels:<br>Open project in attachment.<br>Do the linear calculation.<br>There is a problem concerning the generation of loads (see also image)<br>Tested in Scia Engineer 2013.0.1033 | Remove curved LP10 and create it newly as a flat panel, for LP8 select members.<br>Then in R version 13.0.2003 it works |
| **CSCT-9A4BX6** | We need looong time to open the project and if it´s open we need hours to copy the openings in the plate. Any idea? | fixed in 13.0.2003, it takes cca 3-4min |
| **NDES-9A3KR2** | Look at the loadcase "New TR30 - TR1/LP129.000 m" and look at the free loads "FL350" and "FL349". In the properties of this load, the validity has been put on "+Z" but still an orange generated load is visible below those loads. But this is only a problem in the visualisation of the generated loads. The generated loads GFF1299 and GFF1297 are not taken into account in the calculation. | Please remove the generated load and generate it again. |
| **NWEB-9ABFDR** | Crash report - Accidental crash Version: 2013.0.1036 | Source of problem identified and fixed in Deve_09 Fix will be available in next R_patch |
| **RMAA-9AC9ZX** | Some problem at free load at this esa file.<br>1) Select free load FF3 in LC Q_20G<br>2) Use action button Update 2D members selection<br>3) Try to add this load FF3 to plate S103<br>It was tested: 2013.0.1036 | fixed in 13.0.2023 |
| **NWEB-9A4J5X** | We have discussed it, but it´s still wrong in 13.0.1036.<br>If I do a check short check or by class (group B and accidential) we don´t get the correct 104% unity check. | It is solved when exporting the project |
| **NWEB-9AHEYN** | Look at the load case "ventAB" of the project in attachment and at the description of "Error 1" in the wordfile. | fixed in 13.0.2018, remove the the generated load and mesh and run generation again |
| **NWEB-9A7A9F** | Accidental crash<br>I attached a few errors received on closing Engineering report and SCIA. | Solved in Deve_09. Fix will be available in 2nd patch of R_2013 |
| **RMAA-9AKJQ3** | We have some problem at SIN beam check and national annex DIN. | Solved in DEVE 11, merged to R 13.0.2009..DIN uses a special algorithm to determine the safety factors (based on the type of combination and type of analysis). This algorithm is now properly used also in the SIN beam checks. |
| **NWEB-9AK7HX** | DocX64 - Caption of a table cannot be edited in advance | Solved in Deve_09. Fix will be available in next R_patch |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **RMAA-9AQJMP** | The check of timber does not run on this structure.<br>Tested:2013.0.2001 | Solved in DEVE 11, merged to R 13.0.2009..The issue was caused by a change in the buckling system. |
| **DPIS-9AQKBL** | Free point and line forces in the plane of a load panel (DEVE).<br><br>In this project there is a horizontal load panel. (Z=7,200).<br>I have inputted a free point force in the direction Y according to the GCS, and a free line force in with the same properties.<br>After the calculation, no loads have been generated, and the original loads are colored gray.<br>After changing the geometry system from GCS into Member LCS of Load LCS, the problem is solved.<br>Why is it not possible if the geometry system is set to GCS? For this load panel, the Y direction of the GCS is identical to the Y direction of the load panel. | 13.0.2018, load is generated also for Member LCS |
| **NWEB-9ARB3Z** | When generating the mesh of the project in the zipfile, you will get a message about a topology error., The problem is situated in the element E11. I have simplified the modell (see project test.esa) and same problem in this simple project. Can you please take a look and explain how to solve this? | remove the temporary results data by cleaner before mesh generation; fixed in 13.0.2028 |
| **NWEB-9ATDEX** | Crash report - ER | Source of problem identified and fixed in Deve_09. Fix will be available in next R_patch |
| **RMAA-9AUCWV** | The strange behaviour of 2D free load.<br>The Scia Engineer doesn´t display nothing in selection of plate at 2D free load even though it is made up selection. Method in attachments.<br>Tested:2013.0.1036 | fixed in 13.0.2023 |
| **JPOL-9B2CHY** | Senseless reinforcement arrangement in arbitrary beam with opening.<br>Take a look at attached beam with arbitrary profile and opening (quite common in practice). It cannot be reinforcement by longitudinal bars because it is created in weird positioins. See attached picture where I show the arrangement under different begin and end positioin setting. See also the picture with whole beam and its reinforcing that cross each other and goes out of the solid part into openings.<br>1. Please correct this behaviour<br>2. What is the way how to input reinforcement into such a beam? Is it possible? It works fine only without openings. | The problem was solved and tested in version 13.0.2020, see attached picture |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NWEB-9B2CA3** | Free surface load doesn't work good.<br>Please open attached esa file and check FF10. Although the file was saved with generated load, it is grey. Therefore not generated.<br>When I use action button "generate load" nothing happens.<br>When I use action button "update plate selection" nothing happens. No message in command line appears, no action is proceed.<br>I can only change the selection to "all" and then the load is generated. But when I switch back to "selected" no selection can be made and no load is generated.<br>Tested inn 2013.0.1036 | the load has empty selection, if you select a slab then it is correctly generated in 13.0.2028 (the issue with selection is fixed in this build) |
| **JPOL-9AX9JZ** | Crash reports - A set of Engineering report errors. | First two problems were already solved and will be available in 2nd patch of R_2013. Third problem was caused by the out of memory. Memory consumption was also improved. |
| **NWEB-9B3L4Q** | Crash report - Engineering report crash. | problem already solved. It will be available in next R_patch |
| **NWEB-9B9GYN** | Crash report | Problem identified nad fixed. Fix will be available in 2nd patch of R_2013 |
| **NWEB-9BAJHD** | Another accidental crash of Engineering report. | This problem was already solved and fix will be available in 2nd patch of R_2013 |
| **CSCT-9B9HCT** | Attached two pictures. Why is there a limit to 10mm in the settings? There is a distance between header and tables, we want to minimise more. | Solved in 2nd patch of R_2013 |
| **JPOL-9BEAA4** | "Přizpůsobit zkratky" button in Setup - Options is not translated into Czech. It is displayed in English. See attached picture.<br>Tested in 2013.0.1036 | Czech version was updated on 12 September 2013 |
| **NWEB-9BDDTC** | Another crash report of Engineering report. See attachment. | Crash caused by memory overwriting. Some problem causing memory overwriting vere fixed. Fixes will be available in 2nd patch of R_2013. |
| **JBES-9BGACB** | Open project in attachment.<br>Go to gallery, and create images with the image wizard.<br>Generate images according to the 3D linegrid. Make sure you set the text scale factor on 1,9, and the font size on 2mm.<br><br>After generating the images, click on the first generated image.<br>You will see that the 'S' from 'Snede A' is out of the border.<br><br>The current workaround is to edit the picture manually each time, but for 100+ images, this is a big time loss. | It seems that it is just on this job, I checked other in R2013, and it was generated OK, also on Pmain, it behaves fine. |
| **NWEB-9BGHR8** | Arbitrary: No message is shown for extreme CSS dimensions | see comments for detailed explanation. Extreme dimensions of CSS are not supported. The check for inserting it during first input was added also for arbitrary beams in the same way as it works for haunches. For editing the dimensions for input data is not implemented for both arbitrary beam and haunch. |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **JPOL-9BLFHP** | Wrong graphics and displaying of model and texts.<br>Attached project has been imported from epw file and resaved as esa file.<br>Afterwards it has been also exported to new esa file. Resulting project looks fine in general but has got really weird graphics of model and texts when looking closer.<br>See attached pictures.<br>Tested in 2013.0.1036 | move the structure close to origin - see explanation in comment for detailed |
| **NWEB-9BLLGR** | It is not possible to open the files in attachment. Why not? | The esa archive is corrupted (see comment). A message of it will be added in next version. |
| **NWEB-9BLKPP** | Crash report - calculation of stresses (memory ??) | Problem identified and fixed. Fix will be available in 2nd patch of R_2013 |
| **NWEB-9BMANF** | Crash report | Problem identified and fixed. Fix will be available in 2nd patch of R_2013 |
| **NWEB-9BNJJU** | Crash report | Crash was caused by problem with evaluation of events from GUI. It was alerady fixed. Fix will be available in 2nd patch of R_2013 |
| **NWEB-9BTD2R** | Error message at opening of the engineering report then complete crash | Files with Engineering report were damaged. Possible source of problems is Undo in Scia Engineer. It was improved and improvemnt will be available in 2nd patch of R_2013 to prevent similar problems. Also crash does not apear any more. But this report is irreversibly destroyed and cannot be used. |
| **NWEB-9BXBFZ** | Crash report - working with pictures in ER<br>The problem with crashes of Scia Engineerig during using live pictures in Engineering report. | The problem was identified already earlier and it is fixed. Fix will be available in 2nd patch of R_2013 |
| **NWEB-9C2PN4** | convergence of linear analysis with subsoil<br>Instability and numerical error after solver<br>Please run linear calculation of the attached example. You will get a message about increasing of accuracy by iterative solver and a message, that sum of loads and reactions is not equal for LF2.<br>The Reason of the message is the excentricity of the slab (there is only one slab in model).<br>After I remove the excentricity and place the slab on the middle line, then it works. But I think, also with the excentricity it should be calculated, because there are enough supports in X and Y Direction with the elastic subsoil. | BUG FIXED:known convergence problem (subsoil) - fixed in 2nd patch of SEN 2013.0 USER ERROR:IMPORTANT - about rotation around Z axis:In-plane rotations are NOT supported by finite elements.If an in-plane rotation must be connected or an in-plane moment must be applied, either apply a couple of forces (with a lever arm) or add e.g. a rib representing the contact surface of the load and apply the moment on the rib.It is also possible to add some internal lines in the plate and connect them to the central nodeusing line rigid arms. |
| **NWEB-9BPJHL** | Loads are not generated on panels during calculation in the attached project | tested on 13.0.2028 - removed the culculation and mesh temporary files by Cleaner, then it is generated correctly. The LP2 has empty selection therefore no load is generated what is correct |
| **NWEB-9C4GYA** | Can you check the attached crash report? | The problem is in size of the file. Some data inside the project are bigger than 4 GB which is the current limit. This problem will be solved in next version (2013.1). |

| Ticket number(s) | Bug Description | Solution |
|---|---|---|
| **NDES-9C4JA7** | Open the project test.esa<br>Look at the combinations and the non linear combinations: there are 3 linear combinations and 12 non linear combinations<br>Go to the load cases and copy a load case (for example load case Wind +Y)<br>Look again at the combinations: now you will have 6 linear combinations and 18 non linear combinations!!<br>This is not project dependant, but happens always when copying a load case!! | it is already solved, 13.0.2028 (2nd patch of 2013) |
| **JPOL-9C5J5T** | Crash report - Accidental crash of Engineering report.<br>POD: Accidental crash of Engineering report.<br>Please see attached error report. | Problem identifies and already fixed. Fix will be available in 2nd patch of R_2013 |