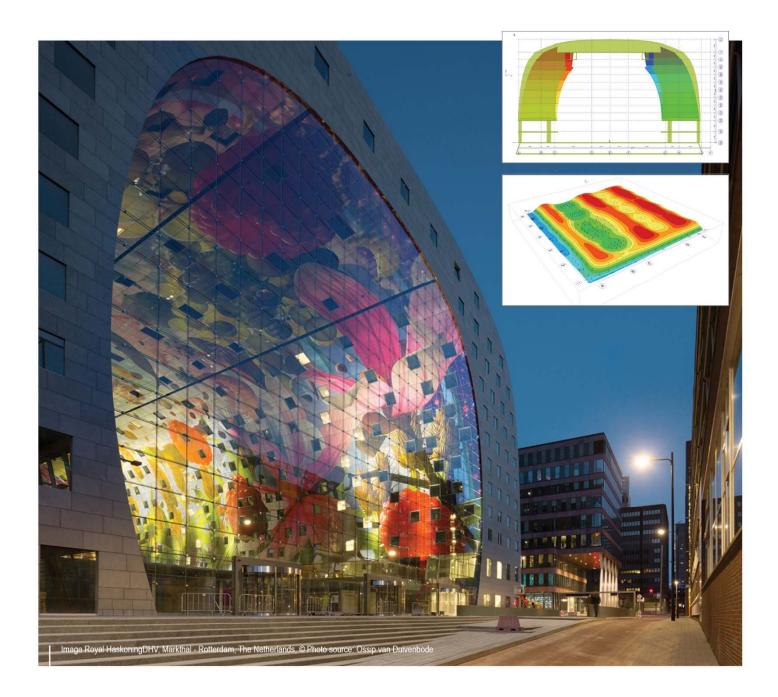
SCIAENGINEER



Open BIM

Integrating the Architectural and Structural Engineering workflows

Belgium Headquarters	France
SCIA nv	SCIA France sarl
Industrieweg 1007	Centre d'Affaires
B-3540 Herk-de-Stad	29, Grand' Rue
Tel: +32 13 55 17 75	FR-59100 Roubaix
E-mail: info@scia.net	Tel.: +33 3.28.33.28.67
Support Phone	Fax: +33 3.28.33.28.69
CAE (SCIA Engineer)	E-mail: france@scia.net
Tel: +32 13 55 09 90	Agence commerciale
CAD (Allplan)	8, Place des vins de france
Tel: +32 13 55 09 80	FR-75012 Paris
Support E-mail:	Tel.: +33 3.28.33.28.67
support@scia.net	Fax: +33 3.28.33.28.69
	E-mail: france@scia.net
Brazil	USA
SCIA do Brasil Software Ltda	SCIA North America
Rua Dr. Luiz Migliano, 1986 - sala 702 , CEP	7150 Riverwood Drive
SP 05711-001 São Paulo	21046 Columbia, MD
Tel.: +55 11 4314-5880	Tel.; +1 443-542-0638
E-mail: brasil@scia.net	Fax.:+1 410-290-8050
	E-mail: usa@scia.net
Netherlands	Switzerland
SCIA Nederland B.V.	SCIA Swiss Office
Wassenaarweg 40	Dürenbergstrasse 24
NL-6843 NW ARNHEM	CH-3212 Gurmels
Tel.:+31 26 320 12 30	Tel.: +41 26 341 74 11
Fax.: +31 26 320 12 39	Fax: +41 26 341 74 13
E-mail: info@scia.net	E-mail: info@scia.ch
Czech Republic	Slovakia
SCIA CZ s.r.o. Praha SCIA CZ s.r.o. Brno	SCIA SK s.r.o.
Evropská 2591/33d Slavickova 827/1a	Murgašova 1298/16
160 00 Praha 6 638 00 Brno	SK-010 01 Žilina
Tel.: +420 226 205 600 Tel.: +420 530 501 570	Tel.: +421 415 003 070
Fax: +420 226 201 673 Fax: +420 226 201 673	Fax: +421 415 003 070
	E-mail: info@scia.sk
E-mail: info.praha@scia.cz E-mail: info.brno@scia.cz	
Austria	Germany
SCIA Datenservice Ges.m.b.H.	SCIA Software GmbH
Dresdnerstrasse 68/2/6/9	Technologie Zentrum Dortmund, Emil-Figge-Strasse 76-80
A-1200 WIEN	D-44227 Dortmund
Tel.: +43 1 7433232-11	Tel.: +49 231/9742586
Fax: +43 1 7433232-20	Fax: +49 231/9742587
E-mail: info@scia.at	E-mail: info@scia.de
Support	
Tel.: +43 1 7433232-12	
E-mail: support@scia.net	

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIAis not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2016 SCIA nv. All rights reserved.

Table of Contents

Table of Contents General Information	
SCIA Engineer Support SCIA Engineer – General Environment Introduction – Open BIM for the Structural Engineer	4
Part 1 – Verify BIM model and import into SCIA Engineer	8
3 rd Party neutral IFC viewer: Import into SCIA Engineer: Part 2 – Extract structural system from model	10
1 BIM Toolbox Part 3 – Align and Connect Model	
1 Align 2 Connect Part 4 – Apply Loading Conditions	25
1 Load panels 2 Floor loading 3 Wind Loading 4 Load Combinations Part 5 – Analysis and Results	28 30 31
1 Finite Element Mesh 2 Static Linear Analysis 3 Results Part 6 – Structural Model Export	33
Part 7 – Drawings	45
Conclusion	
Annex 1: Connection of entities Annex 2: Conventions for the results on 2D members Membrane effects (walls, shells) Annex 3: Results in Mesh elements and Mesh nodes > 4 Locations Annex 5: Overview of the Icons in Windows & Toolbars	50 51 52

GENERAL INFORMATION

SCIA Engineer Support

Via the SCIA Support website: https://www.scia.net/en/support-downloads

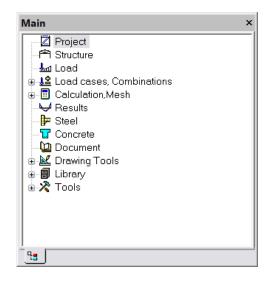
SCIA Engineer – General Environment

Setup > Options Help > Contents > Reference guide File > New > Project data

Project data				×
Basic data	Functionality Loads	Combinations Protection		
100	Data		Material	
1	Name:	Example 1	Concrete	
	Part:	Base Training	Material A992 Timber Other	-
R.	Description:	Frame	Aluminium	
	Author:	Mark Flamer		
25	Date:	05. 07. 2011		
	Structure:		Code National Code:	
	Frame XZ		• IBC •	
	Project Level:	Model:		
Щ¢	Advanced	▼ One ·		
			OK Cano	cel

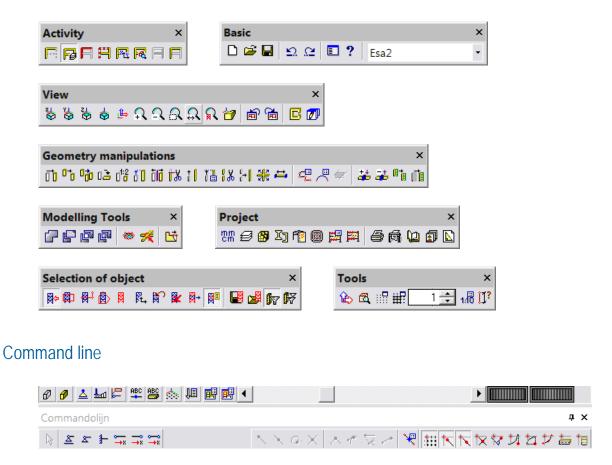
Overview of the Menus

Main Menu & Properties Menu + Actions



Pro	operties	×
Me	ember (1)	🗕 Va V/ /
\square	Name	S1 ^
	Туре	column (100)
	Analysis model	Standard 🔹
	CrossSection	CS1-HEA200 💌 🗉
	Alpha [deg]	0,00
	Member system-lin	centre 💌
	ey [mm]	0
	ez [mm]	0
	LCS	standard 🔹
	LCS Rotation [deg]	0,00
	FEM type	standard 🔹
	Buckling and relativ	Default 🔹
	tions	L = = 1
	uckling data	>>>
	uckling coefficient	>>>
Ti	able edit geometry	>>>

Overview of the Toolbars

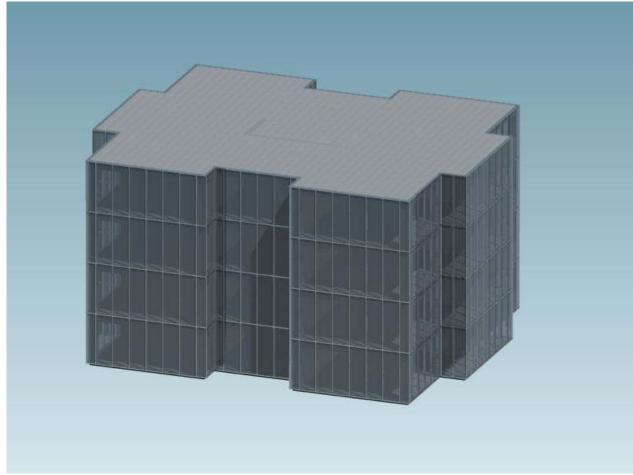


REMARK: If a menu or toolbar has been (accidentally) removed from the project, it can be re-activated via View > Toolbars.

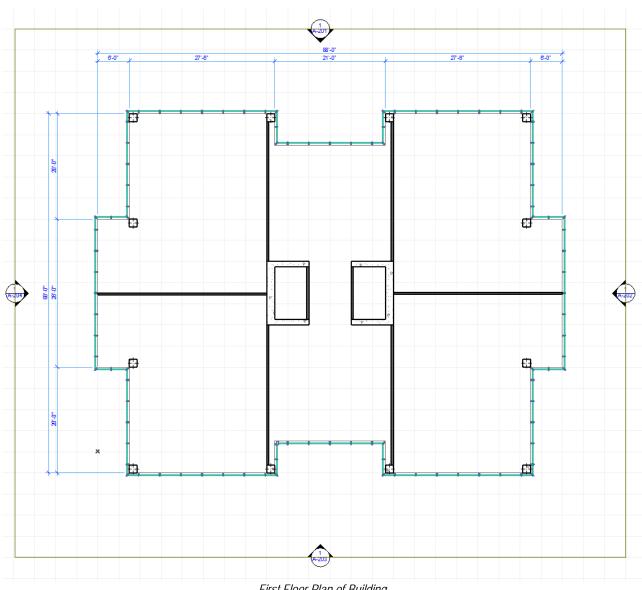
INTRODUCTION – OPEN BIM FOR THE STRUCTURAL ENGINEER

This exercise is designed to walk a student or new user of the BIM process through a small example project highlighting the benefits and pitfalls of model based collaboration using open file formats. The project is a 4 story reinforced concrete building with a non-structural curtain wall cladding system. The students play the part of a structural engineering consultant who has just received an LOD200 BIM model from the project architect in the schematic design stage. We begin the exercise by verifying the model in a third party IFC viewing application. Once verification is complete, we import the model into our structural analysis application where we leverage the efforts of the architect into a simplified finite element model, apply loading conditions and run a simple linear analysis. We will then make a few small changes to the model, export back out as IFC and once again verify in an IFC viewer prior to sending it back to the architect or downstream to detailers or fabrication consultants. We conclude the exercise by extracting a calculation report and some simple drawings from our model.

This tutorial is designed to take between 4 and 6 hours to complete. A folder with all the necessary files has been provided with this document. In this folder is a subfolder titled "Step By Step Files". Each of these files has all the previous steps completed. If you get stuck and are short on time, skip ahead to the next section using one of these files.



Rendering of Architectural Model



First Floor Plan of Building

PART 1 – VERIFY BIM MODEL AND IMPORT INTO SCIA ENGINEER

3rd Party neutral IFC viewer:

- Tekla BIMsight is a free IFC viewing and mark-up program that can be freely downloaded from the Tekla website.
 - http://www.teklabimsight.com/downloads.jsp

* TEKLA [®] potential ³		Login
Home What is Tekla BIMsight Help Cent	er	Download Tekla BIMsight
Accurate,	d Tekla BIMsight BIM-based construction project collabo nication and easy, effective clash mana	pration and
	oit Windows 🛛 🖓 64bit Windo	DWS
	Start your free download	
By download	ding you agree to Tekla <u>license agreements</u> and <u>terms ar</u>	nd conditions.
-	do not have Microsoft .NET 4.0 Framework on your c uring Tekla BIMsight installation. This may take seve	
1 Download	² Install	3 Register
Download the Tekla BIMsight installer. This will take a few minutes. Click Run to install Tekla BIMsight right after downloading.	If installation does not start immediately, double-click the Tekla BIMsight installer and let the software do the rest. This will only take a few minutes.	Use the form in the application to register your copy of the software and access all community features in the application and on the website.

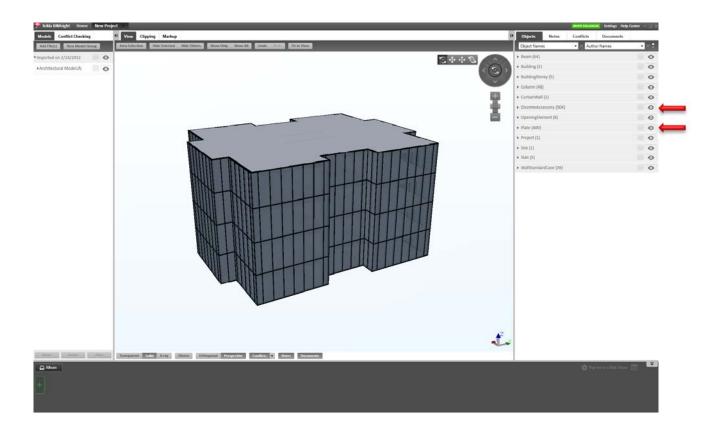
- Because IFC is an open, vendor neutral file format it is always a good idea to validate any models received from another party
 prior to incorporating them into your engineering process. In this exercise we will use Tekla BIMsight for this purpose.
 - Download, install and open BIMsight.



- Click the "New Project" button.
- A new project panel will appear. Click the "View" button to open the new project you have created.

🚰 Tekla BIMsight Home	
Projects	
New Project Open	
Author GRAPHSOFT\mflame Modified 2/14/2012 12:38:40 !	
View Save as a Package	J Welcome
New Project 2/14/2012 12:38:40 PM	Tekla BIMsight demo project v1.4

- Navigate to the file "Architectural Model.ifc" provided in the project folder.
- After a few moments the model should appear.
- Orbit the model by holding the left mouse button and dragging in the model window.
- Make the cladding objects invisible by clicking the eye buttons on the right side of the window.
- Verify that the model has transfered correctly.

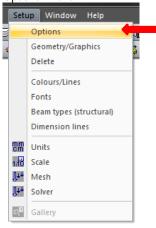


- You may now close BIM sight. The project file will be saved automatically.

Import into SCIA Engineer:

1 Download and Install

- The free student version of SCIA Engineer is available at www.scia-campus.com.
- A new installation of SCIA Engineer may have a simplified UI by default. To complete this exercise, we need access to the complete set of commands. Open the "Setup Options" menu.



• In the "Options" dialogue, verify that "Current style of toolbars" is set to "Full toolbars"

ptions					23
Environment	Templates	Directories	Other	Protection	
Window set	tings				
Show so	rollbars in vie	w			
Rendering				Enable (OpenGL - ha	ardware) 🔹
Hidden lines				Invisible	·
Display s	urfaces inters	ections			
Line pattern	length			3 🔹	
Select pen v	vidth [pixel]			2 🔹	
Command s	ettings				
Right mo	use button cl	ick generate:	s End of	function.	
Skins					
Select skin				Microsoft® Office 20	007 •
Select style	(Office 2007)	only)		Microsoft® Office 20	007 Obsidian black 🔹
Maximal no.	of grouping p	roperties			500
Maximum no	. of entities fo	r default sele	ction 'All		100
🔽 Display p	ictures below	properties			Save commands to file
🔽 Use verti	cal splitter in	properties			Save commands to me
🔽 Display g	lobal coordin	ates in status	bar		
Move docke	ed windows a	nd toolbars to	o initial po	ositions (after restart)	Reset GUI
Current style	of toolbars				Full toolbars
a 🖻				ОК	Cancel Help

2 Import IFC

When importing an IFC file, SCIA Engineer creates a new file as opposed to placing the IFC into an existing file or template. For this example, the instructor has already done some preparation in a template file to speed up the exercise, so it will be necessary to import the IFC model and then transfer it into the template.

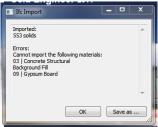
Open SCIA Engineer. If prompted to select a file to open, click cancel so SCIA Engineer starts with no file currently
open. Your screen should look like this. Note: SCIA Engineer offers different UI "skins" so the coloring may be
different (light blue).



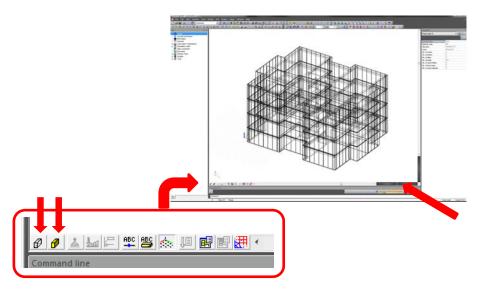
- In the "File" menu select "Import" "IFC 2x3". Navigate to the IFC file "Architectural Model.ifc" provided in the project folder.
- You will be prompted with some options regarding the method of import. Select "import as general solids" as shown below and leave the rest as default. SCIA Engineer allows you to import an IFC file as either native analytical members (beams, columns, plates, etc...) or as "General Solids". General solids are neutral geometry that have no effect on analysis, but can be used as a "Reference Model". Because this model came from an architect, chances are it contains a lot of geometry which is unnecessary or unwanted in our analysis. Things like door knobs and light fixtures. Importing as general solids allows us to ignore this additional geometry and offers us more control.



When this process has completed successfully, you should see the following dialog. Click "OK" to complete the import.

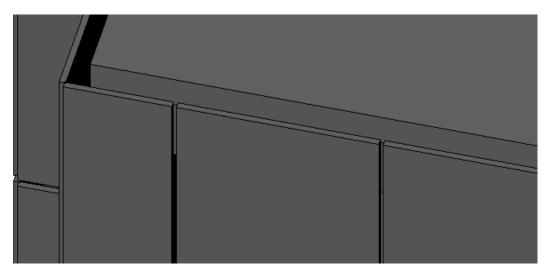


• The view can be set to render filled and volumetric with the following 2 buttons. This tool bar is for quick adjustment of view and visibility settings.



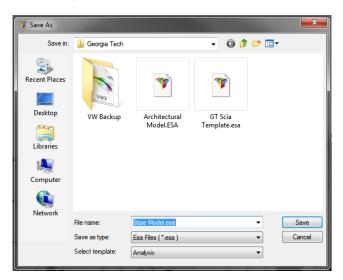
• Zoom with the mouse wheel or the 3 navigation bars on the lower right corner of the window.

- Orbit by holding down the "Ctrl" button and the right mouse button while moving the mouse. Or, by using the 3 navigation bars.
- The strength of an open file format like IFC is the ability to exchange BIM models between software packages
 regardless of what their native file format is. This offers freedom in the choice of software instead of being locked in to
 using software from a specific vendor. The drawback is that each software vendor reads and writes the IFC format a
 little differently and inevitably discrepancies arise. This situation is getting better as more users demand consistent
 IFC model exchange from the software companies.
- You will notice an example of this in the imported model. The window mullions were not imported into SCIA Engineer. This discrepancy is of no significance for our structural model or analysis but, shows the value in validating the model in a neutral viewing application prior to import.

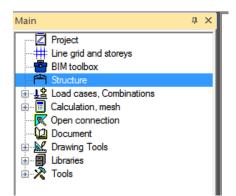


3 Transfer Model to Provided Template File

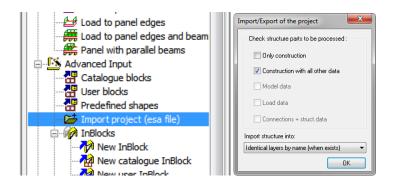
• In the "File" menu select "Save As". Navigate to the project folder and name the file "Base Model" as shown below.



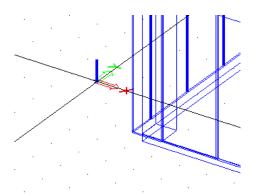
- Close the file.
- Open the file "GT SCIA Template.esa" using the "File" menu or the 🖻 button on the tool bar.
- Double click the "Structure" service in the "Service Tree" as shown below. The Service Tree is the large UI panel on the left side
 of the SCIA Engineer window in this tutorial. The majority of the tools in SCIA Engineer are presented in the various "Services"
 available in this "Tree".



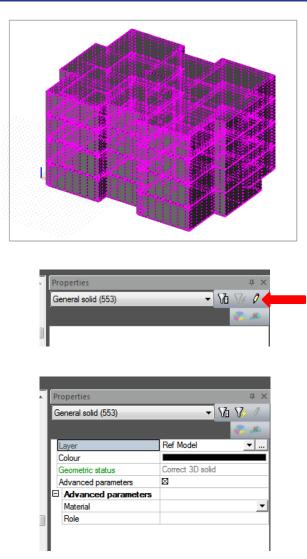
• Double click "Import Project (esa file)" and navigate to the "Base Model.esa" file in the project folder we saved in an earlier step. Click open and accept the default "Construction with all other data" in the window presented.



• You will notice your cursor dragging the imported model in the 3D workspace. Click at the origin to place the model. Click "Esc" to end the command.



Zoom out so you can see the entire model. Left click, hold and drag a window to select the entire model. With the model
selected click the pencil button in the "Properties" palette which is on the right side of the SCIA Engineer window throughout this
tutorial. This will refresh the properties display due to the large number of objects selected. Like most CAD or modelling
programs, SCIA Engineer has the concept of layers as a way of separating and organizing complex files. Change the layer for
the selected objects to "Ref Model" in the properties palette. As our structural model develops, this will allow us to always refer
back to the original geometry and turn it off when convenient.



Now let's make our reference model transparent so that we can begin to create a structural system within it. Right click with the
mouse anywhere in the empty space of the 3D window, and select "Set view parameters for all". In the "Modelling/Drawing" tab, set
the rendering to "transparent"

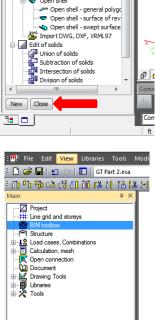
] Che	ck / Uncheck group	Lock position
	Structure	EB Labels
M	Modelling/Drawing 🛛 🔗 Attribute	s 🌌 Misc. 🔍 View
l Ch	eck / Uncheck all	
	ieneral Solid	
_	endering	transparent
	olour	colour by layers
	lash check info	
_	isplay conflicts	✓
	eneral structural shape	<u></u>
_	isplay vertexes	
	lane sections	P
D	isplay section/plan view	
	endering	wired -
S	ection plane	
F	ront plane	
В	ack plane	
D	isplay drawing	
3 P	lane sections - labels	
D	isplay label	✓
Ν	ame	
T,	уре	
	escription	✓
	ame of assigned drawing style	
	ype of plane	▼
	abel	
	isplay labels	▼
D	isplay vertex	✓
	ΠΚ	Cancel

• Save your file and continue on to part 2.

PART 2 – EXTRACT STRUCTURAL SYSTEM FROM MODEL

1 BIM Toolbox

- One of the strengths of SCIA Engineer is its ability to leverage model data from many sources by letting you convert general solids into native objects with analytical properties.
- Open either the file you saved at the end of Part 1 or the file provided in the "Step by Step Files" folder.
- Navigate to the "BIM Toolbox" by double clicking in the Service Tree. If you are already in another "Service" as a result of the last chapter, click "Close" at the bottom of the Service Tree Palette.

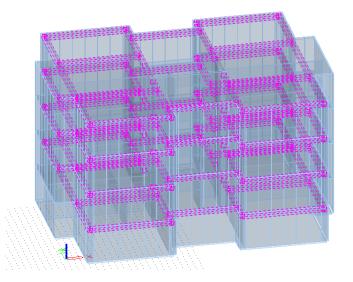


• Select a beam element with the mouse in the 3D model.



• Highlight the "Role" property field in the properties palette and then click the Funnel/Lightning Bolt button at the top of the palette as shown below. This is a quick way to select all elements with a matching property in SCIA Engineer. In this case all general solids with the role of "beam". This "role" property is an additional data field in the IFC file.

F	Properties		φ×
I	General solid (64)	▼ 10	7/ 🔶
			🌮 🔉
Г	Layer	Ref Model	▼
L	Colour		
L	Geometric status	Correct 3D solid	
L	Advanced parameters		
E	Advanced parameters		
L	Material		-
L	Role	beam	



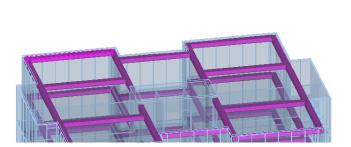
 You should now have 64 general solids selected. Convert them into native SCIA Engineer analytical 1D members by double clicking the "General solid into beam/column" tool in the Service Tree. SCIA Engineer provides options for extraction of multiple cross sections along the length of tapered or non-prismatic members, but for this simple example the defaults are adequate.



You should notice that the beams are no longer transparent and are now a "pink" color. With the beams still selected, change
the layer to "Structure" in the properties pallette. This will help us to keep our project organized.

0.000
0.000
3.281
0.00
standard 🔹
Default 💌
Structure 💌
68.000
Line

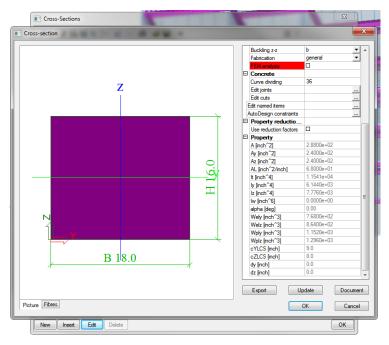
• Select one of the beams and in the properties palette click the small elipsis button to the right of the "Cross Section" property. This will take you to the cross section database where we can verify the section properties.



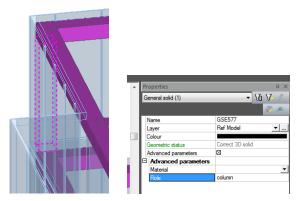
Properties	ф X
Member (1)	- Va V/ /
Name	81
Туре	beam (80)
Analysis model	Standard 💌
CrossSection	CS1 - Rectangle (16.0; 💌
Alpha [deg]	0.00
Member system-line at	centre 💌
ey [inch]	0.0
ez [inch]	0.0
LCS	z by vector 💌
X [ft]	0.000
Y [ft]	0.000
Z [ft]	3.281
LCS Rotation [deg]	0.00
FEM type	standard 💌
Buckling and relative lengths	Default 💌
Layer	Ref Model 🔻

🚯 CSI - Rectangle (I	Name	
		CS1 Rectangle
	Type Detailed	16.0: 18.0
	E Parameters	16.0; 18.0
	Material	C4000
	H [nch]	16.0
	B [nch]	18.0
	E General	16.0
	Draw colour	Normal colour
	Colour	Normal Colour
	Fibre text zoom	1.0
	Properties editable	
	Bucking edtable	8
	Bucking yy	b
	Buckling z-z	b
	Fabrication	general
	FEM analysis	
	E Concrete	
	Curve dividing	36
	Edit joints	
	Edt cuts	
	Edt named items	
	AutoDesion constraints	

• Select "CS1 – Rectangle" in the database list and click "Edit".



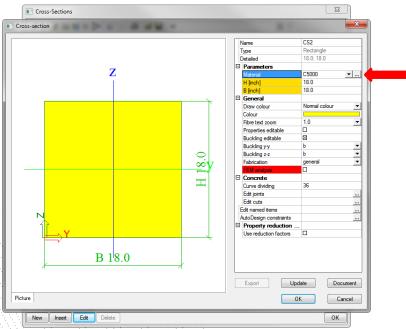
- Notice the cross sectional properties (Area, Moment of Inertia, etc. ...)
- Click "Cancel" then "OK" to exit back to the work space.
- De-select any members by clicking "Esc" on the keyboard.
- Select all general solids with the role "column" using the same process as described above for beams.



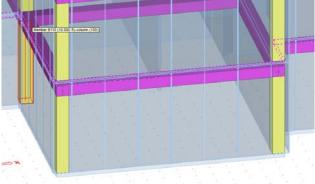
• Use "General solid into beam/column" to convert 48 members and set the layer to "Structure" as described above.



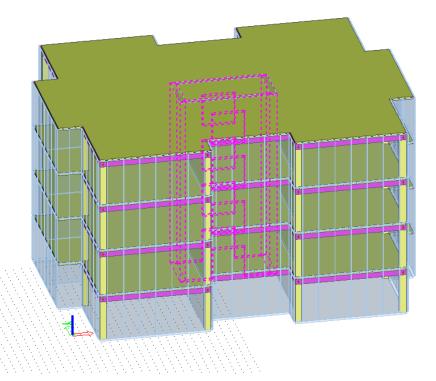
• Enter the Cross Section database using the method described above or using this 🖆 📭 🖬 📩 button in the tool bar. Set the material to C5000 (5000psi concrete).



- Repeat the process above selecting all general solids with the role "slab".
- Holding "Shift", left mouse click on the ground floor slab to de-select it. This would be a slab on grade in this project and we do
 not want to include it in our analysis.



• Convert the 4 selected solids into slabs using "General solid into plate/wall" accepting all defaults. Your model should now look like this. Set the layer of the slabs to "Structure".



- Manually select the 4 solids representing the core walls as shown above and convert into walls using "General solid into plate/wall". Set the layer to "Structure".
- We are done with the reference model for now so we will turn off the visibility of the layer it is in. SCIA Engineer has several methods of organizing the model for visibility. These are called "Activities" and are manipulated through the "Activities" toolbar shown below. Select "Activity by Layers" and turn off the layer "Ref Model". This will make it easier to work on our analysis model and see results.

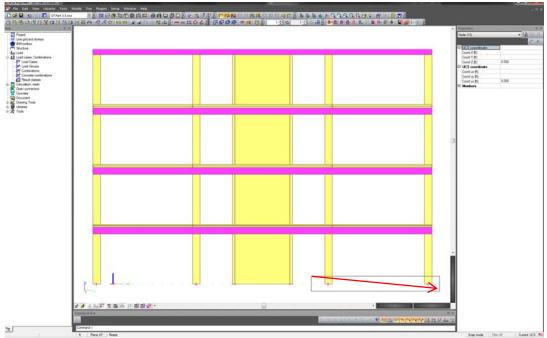
	Layer activi	ty		×
Settings_2 Ref Model BK no Rid Model no Structure B yes Olad no Layer1 no All layers active All layers mactive	🖉 🕸 🗶 I	1 k k k 🔤 🗠 🗠	Al	• 7
Settings_2	One layer acti	vity		
Structure & yes Oad no Layer1 no All layers inactive Al layers inactive				
Olad Ino Layer1 Ino All layers active Ind				
Layer1 D no Al layers active Al layers inactive				
Al layers inactive Al layers inactive				
Al layers inactive				
			All layers inactive	
New Inset Edt Delete Cos				
RRR R R R R R R R R R R R R R R R R R				
	1			
Los units under the second sec			l	
		ert Edit Delete		Close

🗄 🖕 🖕 🖕 🗛 🕰 🕰 🎧 😭 🖓 📄 🛗 🔂 🔊 📄

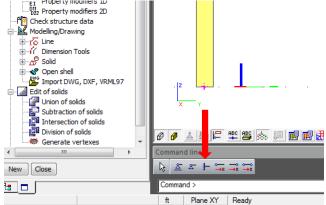
• Set the 3D window's view to be from the X direction using the following button on the "View" toolbar.

-

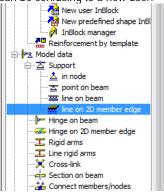
• Click-Drag twice to select all the nodes at the base of the columns as shown below. The top of the properties palette should indicate that 12 nodes are selected.



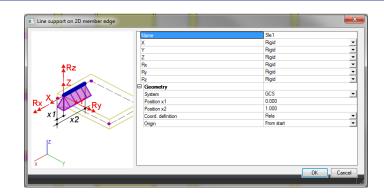
 Double click "Structure" in the Service Tree to gain access to the structure tools. With the 12 nodes still selected, click the "Fixed Support" Quick input button to add support conditions to the base of the model.



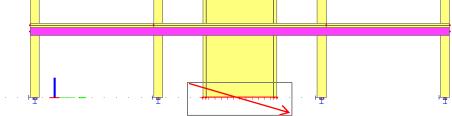
- Each Service in SCIA Engineer provides a set of "Quick Input" buttons just above the command line. These allow easy access to commonly needed commands. All of these commands and more are also available in the service tree and menus.
- Double click "line on 2D member edge" in the service tree. Pay close attention to the hierarchy of the tree in the image below. If
 one of these sub-sections is closed, finding a tool can be confusing to a new user.



Accept the defaults in the dialogue by clicking "OK".



• Click-Drag to select the base of all the core walls as shown below. This will apply a fixed support condition at the base of the walls. When you are done selecting the base of the walls, click the "Esc" button to end the command.

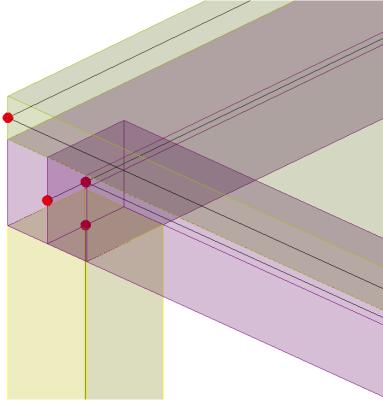


• Save your file and continue on to Part 3.

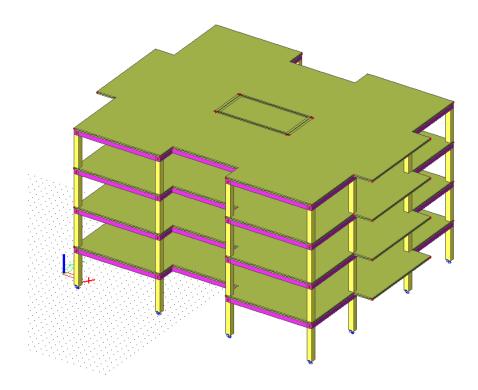
PART 3 – ALIGN AND CONNECT MODEL

1 Align

A model created by an architect or other consultant most likely was not assembled with structural analysis in mind. In a finite
element model, objects such as beams, columns and slabs are represented by their centerline or neutral axis and a stiffness
matrix. Any other, more complex representation is for visualization purposes only. Our conversion from a volumetric
architectural model to a simplified finite element model requires 1 more step. We need to make slight adjustments to the
locations of the members and nodes to achieve transfer of forces and compatibility of deformation. This situation is illustrated in
the image below.



- Fortunately, SCIA Engineer's BIM toolbox has a system to help with this process.
- Click the Clic



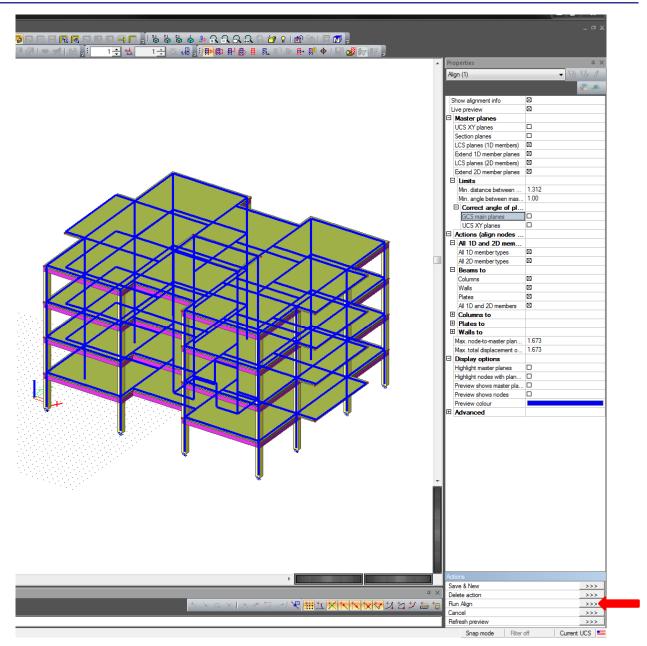
• Open the BIM toolbox in the Service Tree and double click "Align".

🍞 S	cia Eng	gineer	- [GT Pa	art 4.esa : 1]	
: 👧	File	Edit	View	Libraries	Tools	Мо
: D	🖻 🖌	- <u>-</u>	2 🖂	🔳 GT P	art 4.esa	
: d	0° <mark>0</mark> 0	jo 🕞	0 ^{1:2} č () tilð tilk t	1 18	%
BIM	toolbo	x			џ >	
		Gene Part Beam ign Align Conn	ral solid of a gen h/slab int hect men	into beam/c into plate/w eral solid to to general so nbers/nodes ure data check	vall a membe olid	

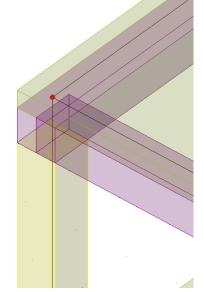
• When prompted to "Proceed with all entities" click "Yes".



• The Properties Palette will now change and display preferences for the align command which we are about to perform. Set the preferences exactly as shown in the image below. You should see the bold blue preview of the aligned structure also.

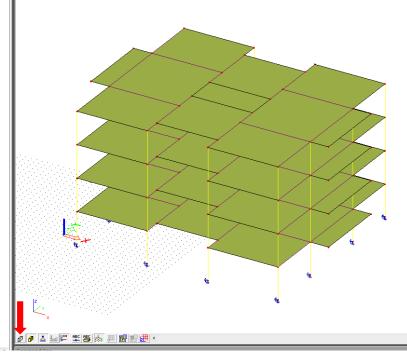


• Click the "Run Align" button in the "Actions" at bottom right corner of the SCIA Engineer window. The centrelines of the objects have now been aligned. Notice the difference in the transparent image below. The 2 beams, column and slab all join in the corner at a single node.



2 Connect

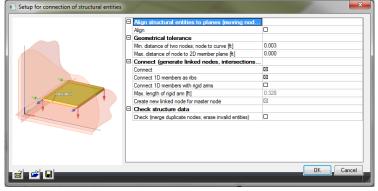
• Turn off the volumetric display of the model using the button in the quick view settings tool bar at the bottom left of the work space.



• Before we proceed with loading and analysis, we need to tell SCIA Engineer to connect our model. With the entire model deselected, click the "Connect members/nodes" button in the "Geometry Manipulations" Toolbar.

👯 🕅 File	Edit Vie	w Libraries	Tools	Modify Tree	Plugins	Setup	Window	Help	
៍ 🗅 🖻	🖬 I 💁 S	2 🔳 GT P	art 4.esa		📜 : 17 🖉	i 👩 Ij	F🖻 🎯 🖡	🛱 🛤	a 🖗 ն
: 🔂 👘	<mark>]</mark>]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]	10 tið t% i	t 1 TB 13	{ 눈 桨 ឝ	내운 곳 4	🗲 📭	oo 💒 ,	- 1 1 (ሰ 🐴 🧀 🚽
571 L I									

- Click "Yes" to proceed with all entities.
- Make sure the "Connect 1D members as ribs" box is checked and click "OK". This will connect our floor and roof beams to the
 floor slab. This is how a monolithically poured, concrete slab and beam system would behave (shear transfer). Otherwise our
 slab would behave like a plate sitting on top of but not connect to, our beams which would be incorrect.

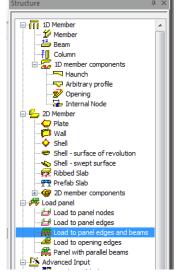


• Save your model and continue on to Part 4.

PART 4 – APPLY LOADING CONDITIONS

1 Load panels

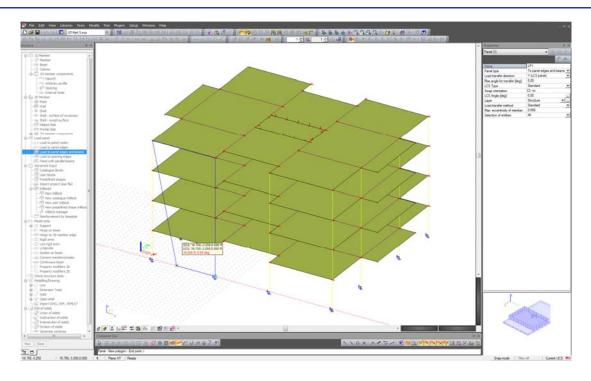
- The curtain wall cladding in this project will be designed by an outside consultant, so we will use a simplification of these elements in our analysis. SCIA Engineer provides a type of 2D object known as a "Load Panel" for this purpose. Load Panels distribute surface loads such as wind to other members, but do not influence the stiffness of the structure.
- Enter the "Structure" service from the service tree and double click "Load to panel edges and beams".



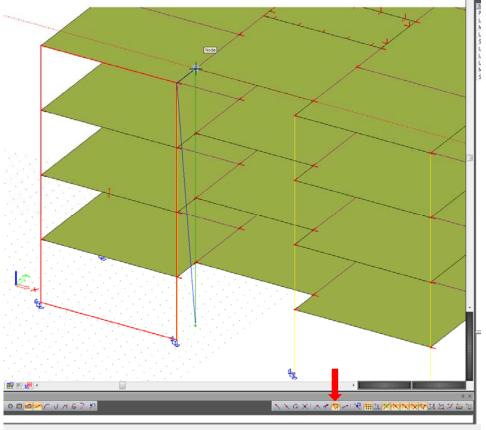
• Verify the defaults are as shown below.

Z Panel type Load transfer direction Max.angle for transfer [deg] LCS Type Swap orientation	To panel edges and beams Y (LCS panel) 5.00
Max.angle for transfer [deg] LCS Type	
LCS Type	5.00
Swap orientation	Standard
Swap one itation	🗆 no
LCS Angle [deg]	0.00
Layer	Structure 🔽 .
Load transfer method	Standard
Max. eccentricity of members [ft]	0.656
Selection of entities	All

• You will now be in a SCIA Engineer "Sketching Mode". Cover the exterior of the building with panels by clicking on the 4 corners of each planar panel as shown below. Pay close attention to "snapping" at each node. End each panel input (4 clicks) by clicking the right mouse button.



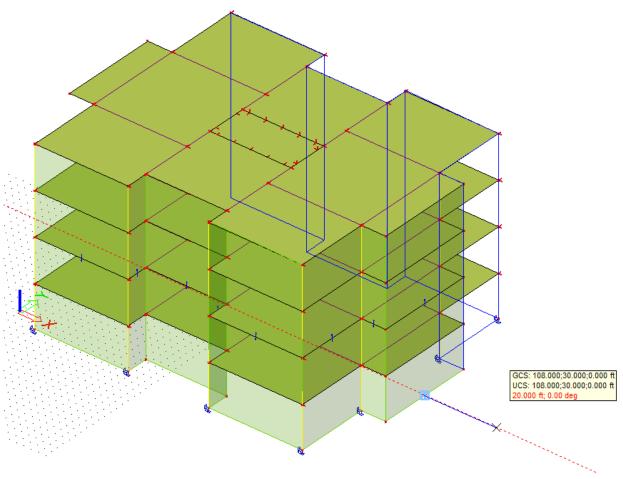
• Use the "Move to working plane" mode on the "Snapping Toolbar" to project down to the ground level from the roof nodes where necessary. This is shown in the image below.



• The "Mirror" tool is also useful in streamlining this process. Input half the panels and then click the "Mirror" button on the "Geometry manipulations" toolbar.

📆 File Edit Vie	w Libraries	Fools Modify	Tree Plugins	Setup Wi	ndow Help	
i 🗅 🚅 🔚 I 💁 🤇	🗵 🔳 GT Par	t 5.esa	- 📜 : BM 🗲	3 👩 Ij f <mark>e</mark>	🞯 🚔 🎇 e	5 🖻 🛄 🗗
: 📬 📬 👘 🚵 (†2	10 čló t% t l	1월 [※ 문] 위	🖁 🛱 I 🤹 🥂	¢7 00 00	👪 👪 👫 (ከ 🐴 🔬 🛒
Structure		д Х				

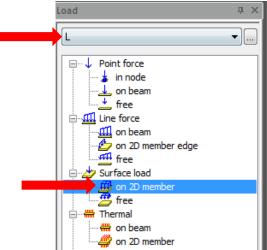
• Click to define the axis of reflection (the midpoint snapping mode will help with this and is enabled by default). A second click completes the action.



• Verify that all the walls of your building are clad with load panels.

2 Floor loading

- Enter the "Load" service through the service tree.
- Load cases and combinations have already been prepared and provided in the template file you started with. Set the active load case to "L" (Live) as shown below.



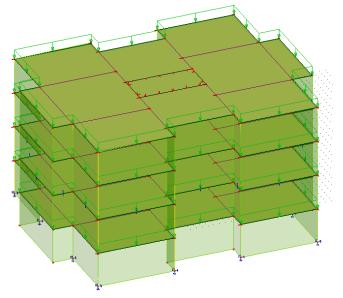
• Double click the "Surface load – on 2D member" button in the list of load types and set the options as shown below.

Surface force	- And -	X
	Name	SF1
	Direction	Ζ 🗸
	Туре	Force 💌
	Value [kip/ft^2]	-0.04
-P	Geometry	
	System	GCS
	Location	Length 🗾
A REAL PROPERTY AND A REAL		
x		
		OK Cancel

• Enable the "Visibility selection mode" in the "Selections" toolbar. This will make selection of 2D elements easier. This mode can be turned on and off anytime.

i 📴 🛱	₽1 (8)	₿	R. 8	<u>ጉ</u> 🖹	₿→	₿	\$	1	-
							T		

• Pick the 4 slabs to apply the -40psf live loading. Complete the action with the "Esc" button on the keyboard.



• Set the active load case to "Dsup". This load case represents superimposed dead load for additional partitions, mechanical components, etc... The self-weight of the structural members is already included in the load case "Dself". Repeat the process above applying a 15psf load to the 5 slabs.

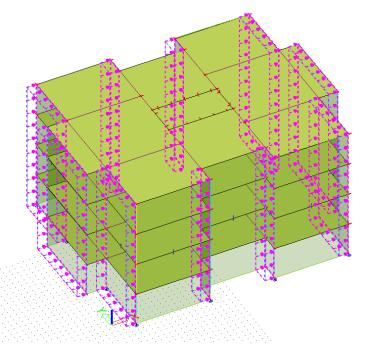
Surface force		X
	Name	SF5
	Direction	Ζ 🔹
	Туре	Force 🔹
	Value [kip./ft^2]	-0.015
	Geometry	
-P	System	GCS 🗸
	Location	Length 👻
x y		
		OK Cancel

3 Wind Loading

• Set the active load case to "Wx". Double click "Surface load – on 2D member" again this time applying the settings shown below. Instead of specifying a force as we did for the live and dead loads we are going to set the "Type" of load to "Wind". This will pull the surface pressures, which vary over the height of the structure from a pre-defined wind curve.

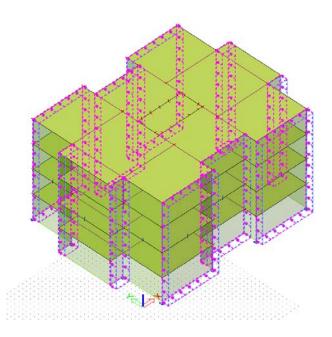
	Name	SF9	
	Direction	x	-
	Туре	Wind	-
	Coeff	1.000	_
• •	Geometry		
-P	System	GCS	+
	Location	Length	-
and the	-		

• Apply this loading to all the load panels which are perpendicular to the wind direction (X). It may be easier to complete this action with the "Visibility selection mode" disabled, due to the panels being transparent.



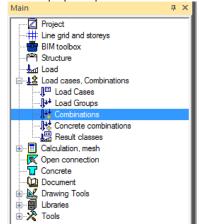
• Set the active load case to "Wy" and repeat the process for the Y direction.

Surface force		
	Name	SF21
	Direction	Y
	Туре	Wind
	Coeff	1.000
-P	Geometry	
-P	System	GCS 🗨
	Location	Length 💌
x y		
		OK Cancel



4 Load Combinations

• Click "Close" at the bottom of the service tree palette to return to the main tree. Now enter the "Combinations" service and review the load combinations that have been prepared prior to this exercise.



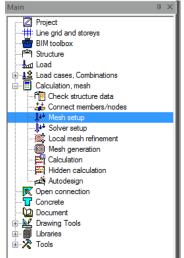
Combinations			x
🚚 🤮 🗶 📸 🔽 🗠	Input combinations	٣	
ULS ULS ULS 1.4D ULS3-1.2D + 1.6L ULS3-1.2D + 1.6Wx ULS3-1.2D + 1.6Wx ULS3-1.2D + 1.6Wx ULS3-1.2D+0.5L+1.6Wx ULS12-0.9D+1.6Wx ULS12-0.9D+1.6Wy SLS SLS1-D+L	Image Name Description Type Contents of comb Dsef [-] Dsup [-] L [-]	SLS1 D+L Linear - ultimate	· · · · · · · · · · · · · · · · · · ·
New Insert Edit Del	ete		Close

- ULS combinations are strength (factored) load combinations and envelopes.
- SLS combinations are serviceability (un-factored) load combinations and envelopes.
- Save your file and proceed to Part 5.

PART 5 – ANALYSIS AND RESULTS

1 Finite Element Mesh

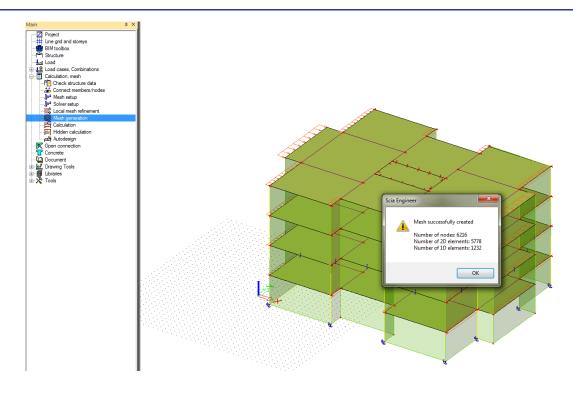
- The finite element mesh in SCIA Engineer is automatically generated prior to running the solver. This is a significant
 advantage over the more manual meshing process encountered in other software. The choice of mesh size and
 refinement can have an impact on the validity of results when using the finite element method, therefore it is always
 important to visually verify the mesh before accepting any results.
- Enter the "Mesh Settings" from the service tree.



• Set the number of 1D mesh elements to 5 and the 2D mesh size to 2ft as shown below.

14	Name	
Ξ	Mesh	
	Minimal distance between two points [m]	0.001
	Average number of tiles of 1D element	5
	Average size of 2D element/curved element [ft]	2.000
	Definition of mesh element size for panels	Automatic
	Average size of panel element [ft]	3.281
-	1D elements	
	Minimal length of beam element [ft]	0.328
	Maximal length of beam element [ft]	328.084
	Average size of cables, tendons, elements on subsoil, n	3.281
	Generation of nodes in connections of beam elements	
	Generation of nodes under concentrated loads on beam	
	Generation of eccentric elements on members with varia	
	Division on haunches and arbitrary members	5
	Division for 2D-1D upgrade	50
	Apply the nodal refinement	Only 2D members
-	2D elements	
	To generate predefined mesh	
	To smooth the border of predefined mesh	
	Maximal out of plane angle of a quadrilateral [rad]	0.0
	Predefined mesh ratio	1.5
- 6	Hanging nodes for prestressing	

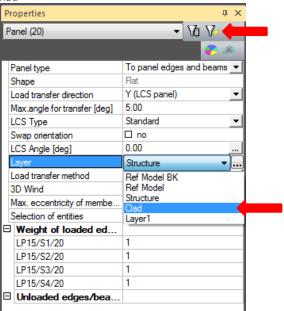
- Click "OK" to apply the settings.
- Click "Yes" if you receive a warning about changing the settings.
- Double click "Mesh generation" in the service tree. You should be presented with a dialog displaying the results of the meshing process.



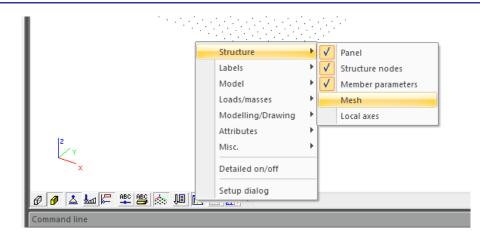
- Click "OK" to accept.
- Turn off the visibility of loads in the guick view settings tool bar.

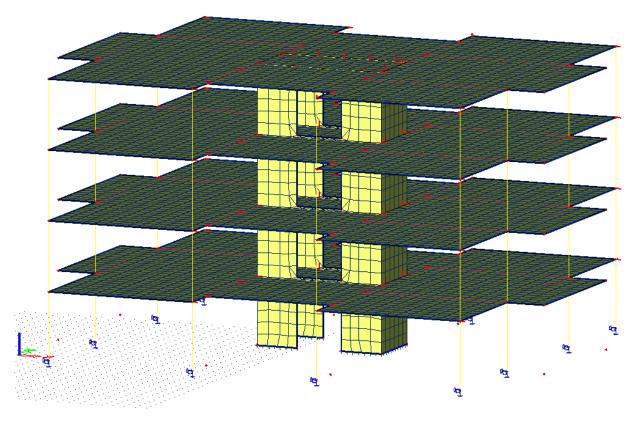
0 1 🛓	- • 🖽 🔤 🔍 🔕 🕮 •
Command line	
N	

- Select all the load panels using the "Select Elements by Property" button and a matching property.
- Set the layer of the load panels to "Clad"



• Enable the mesh visibility in the quick visibility settings.





• Verify that the generated mesh is acceptable.

2 Static Linear Analysis

- Double click "Calculation" in the service tree
- Select "Linear Calculation" and click "OK"

FE analysis	1112	×	
	Single analysis Batch analysis		
	Linear calculation		
	Nonlinear calculation		
	🔿 Modal analysis		
	🔿 Linear stability		
	Concrete - Code Dependent Deflections (CDD)		
	 Construction stage analysis 		
	🔿 Nonlinear stage analysis		
	🔿 Nonlinear stability		
	Test of input data		
	Number of load cases: 5		
	Solver setup Mesh setup		
	OK Cancel		

• During the calculation process you should see a solver status window.

🖌 FEM solver - 64 bit.					
Calculation					
Input data processing					
Plane element stiffness matrices					
Bar element stiffness matrices					
Equations assembling: righthan	1 side				
Equations solving: lefthand side					
Equations solving: rephand side					
Internal forces evaluation	Internal forces evaluation				
static, linear 39 %					
Project	SCIA Engineer 2011				
Model	3D Shell				
Number of plane elements	5778				
Number of beam elements	1232				
Number of nodes	6216				
Number of equations Cases solved	37296				
Requested disk space, MB	207.7				
Pause / Break					
Pause / t	rause / biedk				

3 Results

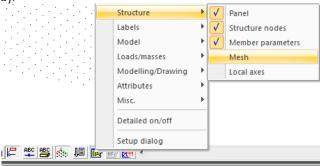
• When the solver is complete you will see that a new service "Results" has appeared in the tree.

- Enter this "Results" service, select "Reactions" and set the following properties in the properties palette. •
- Click "Refresh" in the "Actions" at the bottom right of the SCIA Engineer window. • д×

∲ ²) Deformed Structure ⊟ <u>△</u> Supports	Properties	д >
Reactions	Reactions (1)	- 🖬 🏹 🖉
Trensity The sultant of reactions Foundation table Yodal space support resultant	_neactions (1)	
Beams My Internal forces on beam	Name	Reactions
	Selection	Al
Relative deformation	Type of loads	Combinations
Shear stress	Combinations	ULS
Connection Forces	Filter	No
2D Members	Values	Rz
 Member 2D - Internal Forces Member 2D- Stresses 	Extreme	Combinations ULS No Rz Node
	Drawing setup 1D	
Integration strip	Rotated supports	
드 🛱		387 14

- These are the critical support reactions for the ULS envelope. •
- Turn off the mesh display. •

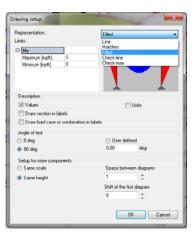
Results



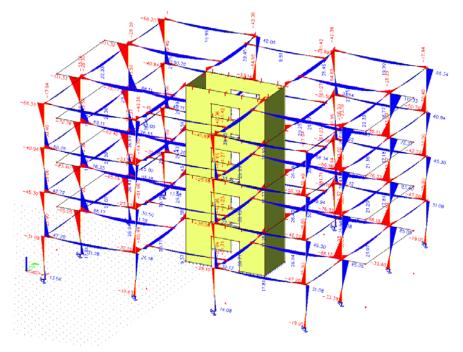
Select "Internal Forces on Beam" in the service tree. Set the properties as shown below. •

ults 🕂 🛪	Properties		φ×
······································	Internal forces on member	(1) 🔹 🚺	V/ /
E→▲ Supports			8 🔊
Resultant of reactions Foundation table	Name	Internal forces on me	mber
Nodal space support resultant	Selection	All	-
Beams	Type of loads	Combinations	-
·····································	Combinations	ULS	-
Member Stress	Filter	No	-
	Rib / Integration strip		
Connection Forces	Prefab slab beam		
Displacement of nodes	Values	My	-
Member 2D - Internal Forces	System	Principal	•
	Extreme	Member	-
Averaging strip	Drawing setup 1D		
	Section	All	-
Galculation protocol			

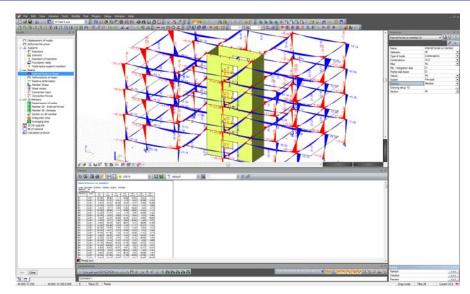
- Click the ellipsis button next to "Drawing setup 1D" in the properties palette.
- Set the "Representation" to "Filled"



• Refresh the results display. You should see the strength level Y axis moment envelopes for the beams and columns.



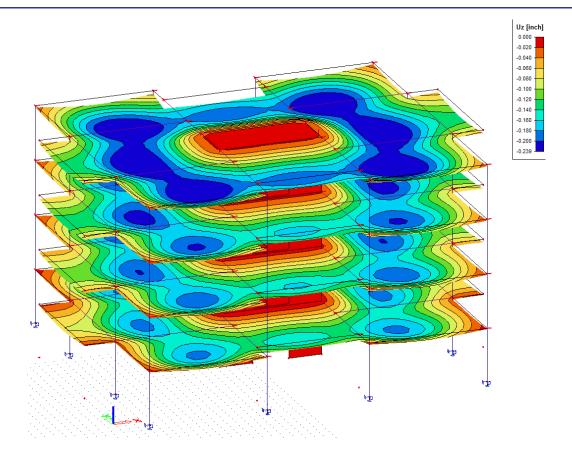
• Click "Preview" in the "Actions" at the bottom right corner of the SCIA Engineer window.



- The "Preview" window allows you to display results and other data in tabular form. In the current table the critical results for each 1D member are displayed. Close the preview window by clicking the "X" button at the top right of the table.
- Select "Displacement of nodes" in the tree and set the following properties in the properties palette.

esults 4 ×	Properties	џ ×
Displacement of nodes Deformed Structure Supports	Displacement of no	
Reactions Intensity Reactions Foundation table Nodal space support resultant Beams Member Stress Shear stress Connection input Shear stress Shear stress Member 2D - Internal Forces Member 2D - Stresses Member 2D - Stresses Averaging strip Averaging strip Averaging strip Shear tail Calculation protocol	Name Selection Type of loads Combinations Filter Structure Standard Section Edge Values Extreme Drawing setup 2D	Displacement of nodes All Combinations SLS1 - D+L No Deformed Uz Global

• Refresh the display. This is the vertical deformation under service level loads. This is for elastic behaviour only. In a real project we would need to account for cracking, creep and other non-linear and time dependant effects.



• Right click in the 3D window and select "Picture to Gallery".

\sim	
<u></u>	Set view parameters for all
C	Set view parameters for selected
× ×	Cursor snap setting
2	Print/ Preview table
	Table to document
1	Print picture
	Picture to document
	Picture to gallery
<u> </u>	Save picture to file
	Copy picture to clipboard
9	Wired model in view manipulations
	Advanced graphic setup
[]]?	Coordinates info
1	Picture wizard
	n

• Set the "Display Mode" to "Standard" and click "OK". This will save this view of the model in our "Picture Gallery". We will use this in Part 6.

Gallery picture		
Name	Picture 1	
Scale 1 :	100	
Picture width [mm]	297	
Picture height [mm]	210	
Display mode	Standard	•
Load units in regen.	8	
lext scale factor	1	
harset of texts	ANSI (USA, UK, Europe)	•
ine pattern length	3	
Display GCS icon	To picture.comer	•
Performance		
Settings		>>>

- As you can see there are many ways to display the analysis results in SCIA Engineer.
- Save your file and proceed to Part 6.

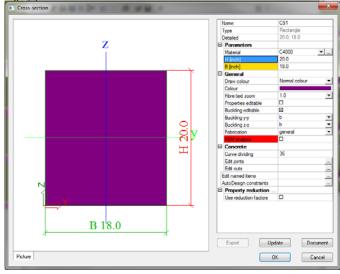
PART 6 - STRUCTURAL MODEL EXPORT

One of the features that differentiate SCIA Engineer from traditional Finite Element Analysis software is that it maintains 2 versions of the model in parallel. SCIA Engineer has the traditional "Analysis Model" which consists of bars, nodes, shells etc. ... This model is what you are actually analysing, but is not very useful for communicating design intent or for exchange with another consultant. The second model representation is the "Structural Model" which is dimensionally accurate and can be used to extract drawings, generate renderings and be exported for collaboration.

It would be very unlikely that the member sizes defined by the architect would be correct from an engineering point of view. In this exercise we have determined that the beam sizes are inadequate and we need to propose a change and communicate this back to the architect through a revised model.

	: 👧	File	Edit	View	Libraries	Tools	Modify	Tree	Plugins	Setup	Window	Help
					🔳 GT Pa						1 👘 🔘 🖡	
Open the Cross-Sections Database.	÷ 📬	0† <u>0</u> 0	ita 👌	C ¹² Č (l t <mark>ið t</mark> ä t	1 78 8	※는 왕	i 🛱	ᅊᄸ	\$ 1	• 🚥 🚢 .	 1

- Select "CS1" and click "Edit"
- Change the "H" (height) parameter to 20in.



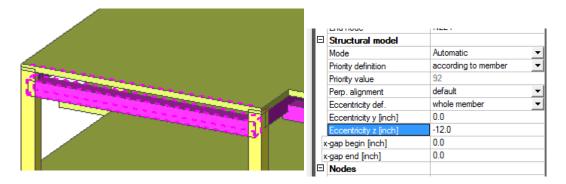
• Exit the cross section database and set the rendering mode to volumetric and filled.



 Select all 1D members with the property "Structural model – Mode" set to "General. If executed correctly you should have 112 members selected.

Pr	operties	џ ×	
M	ember (112)	- Vi V/	
		💞 🗶	
1	Analysis model	Standard 💌	
(CrossSection	▼	
	FEM type	standard 💌	
	Buckling and relative lengths	Default 🗨	
	ayer	Structure 💌	
Ξ	Geometry		
	Shape	Line	
	Beg. node		
	End node		
Ξ	Structural model		_
	Mode	General 🗾 🚽	
	Locked geometry	🗆 no	-
Ξ	Nodes		

- Set "Structural model Mode" to "Automatic". These modes determine how much flexibility the user has in manually adjusting the "Structural Model" in relation to the "Analytical Model". In our case we want SCIA Engineer to take care of most of this automatically.
- Select all the beams which have the "Structural model Eccentricity z" set to -12in. Set this parameter to 0in.



• Click the "Generate structural model" button to refresh the view.



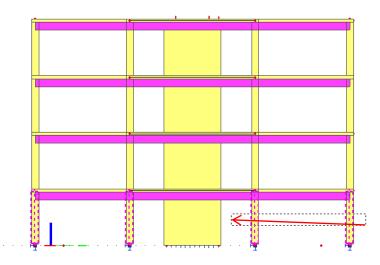
• Select all the 1D members with "Type" set to "column".

Properties	д ×
Member (1)	
	💞 🍂
Name	B73
Туре	column (100)
Analysis model	Standard 🔹
CrossSection	CS2 - Rectangle (18.0; 💌
Alpha [deg]	0.00
Member system-line at	centre 💌
	0.0

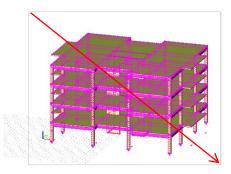
• Set the "Structural model – Mode" to "Manual" and set the "begin x-offset" and "end x-offset" parameters to 4in.

	Structural model	
	Mode	Manual 🗨
	Priority definition	according to member
	Priority value	100
	Perp. alignment	default 💽
	Eccentricity def.	whole member
	Eccentricity y [inch]	0.0
	Eccentricity z [inch]	0.0
E	End-cuts	
	begin x-offset [inch]	4.0
	begin Rz [deg]	0.00
	begin Ry [deg]	0.00
	end x-offset [inch]	4.0
	end Rz [deg]	0.00
	end Ry [deg]	0.00
Ξ	Nodes	

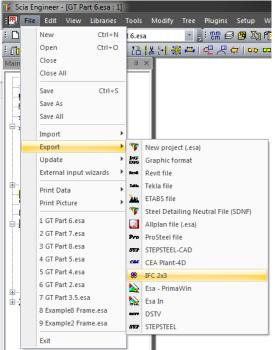
• Set your view to be from the X direction . Drag a selection box across the first floor columns from right to left. When you select from right to left all objects crossed by the selection box are selected. When you select from left to right only objects completely inside of the selection box are selected.



- Set the "begin x-offset" property to 0in.
- Refresh the structural model.
- Drag a selection box around the entire model from left to right. When we export our revised model to IFC, we don't need to include the reference model layers. An easy way to filter the geometry to export is through selection.



• Select "File – Export – IFC 2X3" from the menu.



- Select a suitable name and location for the exported file.
- When prompted with options for the geometrical representation, set the parameters as shown below. Different
 software packages expect geometry to be represented using different data structures. "SweptSolid" is a simple
 extrusion with a profile and path. Most software can support this geometry type and will convert it into a native object
 of some type. "Brep" is short for boundary representation. This data structure supports more detail than swept solids,
 but typically does not get converted into a native object type in the target application. There are benefits and
 shortcomings to each method depending on the context.

Ifc Export Options
Shape representation of 1D members: SweptSolid SweptSolid, not using parametric profiles Brep
Shape representation of arbitrary and haunched 1D members: Use 'SectionedSpine' representation
Shape representation of 2D members: SweptSolid Storeys
Export layers as storeys

 Open the Tekla BIMsight project you created in Part 1. Click "Add File" and select the file you just exported from SCIA Engineer.



Explore the different options for comparing the 2 models.

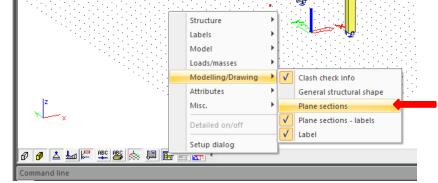
📌 Tekla Bildhäght Hame New Project	annet sinstematic Sections - D in
Mudels Conflict Checking 41 View Clipping Markup	Dijects Notes Conflicts Dir (*
Antibest New Multi Comp Alex Merchin. Marchennet Marchanet. Stor Only . New Alex	Model Names • M Author Names • M
* imported on 2/14/2012	S R R S A A Athlectral Mole. O
#Architectural Model.If:	+ Structural Model.ds (
• reported on 2/18/2012	· ·
	DHO
	£.
D Mass	0 Part for a scalar from [2]

• Save your SCIA Engineer file and proceed to Part 7.

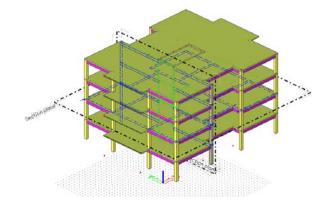
PART 7 – DRAWINGS

One of the immediate benefits of constructing a BIM model as opposed to traditional CAD drawings is coordinated documentation. In the past when a change was made on a 2D drawing, any other related drawings or layout pages also had to be manually reviewed and revised. This is a major source of potential errors. When documents are derived from a 3D BIM model, much of this manual coordination can be automated. SCIA Engineer offers a system for extracting drawing views from your model. These drawings are generated based on styles that the user can predefine to company standards and can be placed on layout pages for printing.

• Enable the display of "Plane sections" from the quick view settings.



 You should see 2 section planes. 1 plan view and 1 section view. These drawing sections were predefined in the template file to save time during this exercise.

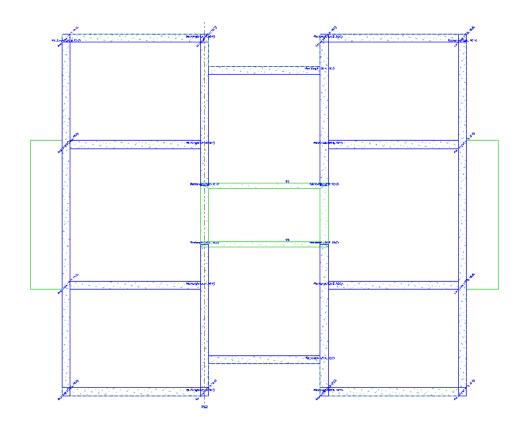


• Enter the drawing manger from the Service Tree. In the "Drawing Manger" mark the checkbox next to PS1 and click "OK".

Main	ά×	
Project		
BIM toolbox		
-F ^{ara} l Structure		
- Load		
- 👫 Concrete combinations		
Result classes		
E Calculation, mesh		
Results		
Open connection		
Concrete		
Document		
🖮 🕍 Drawing Tools		
Picture gallery		
Paperspace gallery		
- 🚧 Section		
🕬 General section		
🗄 📲 Libraries		
🗄 💸 Tools		

🖸 🖂 🚺								
Plane section	Section type	Description	Drawing view	Date modified		Name	PS1	
PS1	Plan view		4	2012 02 14 10:55:20		Description		
PS2	Section	1	Ē.	2012 02 14 11:00:12	- 0	Projection settings		
r de	Sector			2012 OF 14 TEODER		Section type	Plan view	
						Section drawing style	Conc_P	·
						Default scale 1:	50	
						Drawing size [inch]	25 x 21	
						Backward view	no	
						Front plane offset [ft]	1.000	
					110	Back plane offset [ft]	1.000	
						Model	Structural model	Ŧ
						UCS rotation [deg]	0.00	
						Autogeneration pr		
						Layer for labels	Clad	•
						Layer for 2D dimensions	Clad	·
						Geometry		
						Section plane offset [t]	0.000	
						Offset of PSE polyline [It]	3.281	
						General drawing d		
						Title	C no	
					A	tions		
					R	efresh		>>
					B	eset		>>

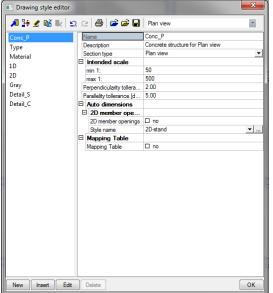
 You should now be in the SCIA Engineer 2D Drawing environment. The drawing generated is based on a set of criteria and associated line types, hatches, labels etc...



• Click the ellipsis button next to "Section drawing style".

Properties	0.07	4
Plane section (1)	<u> </u>	
	6	×
Name	PS1	
Description		
Projection settings		
Section type	Plan view	
Section drawing style	Conc_P	·
Default scale 1:	50	
Drawing size [inch]	25 x 21	
Backward view	🗆 no	
Front plane offset [ft]	1.000	
Back plane offset [ft]	1.000	
Model	Structural model	-
UCS rotation [deg]	0.00	
Autogeneration prefe		

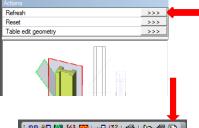
• In the "Drawing style editor" select the "Conc_P" style and click "Edit".



• This is the drawing "style" definition. The general idea is that members and objects that meet criteria in the list on the left are drawn using attributes chosen on the right. Additional "rules" can be added to the list and they are organised by priority from top to bottom. Feel free to experiment with the results of adjusting the rules.

olours & lines Auto	label			<u>s</u>	🎜 24 📾 I	k 🔺 🔍 👘					
Member role	Material type	Material name	Position	Direction	Visibility	Part of element	Draw	Colour	Style	Width Type	Preview
All 1D Members	Concrete		Section	AI		Contours		1		0.2 [mm] Metric	
All 1D Members	Concrete		Back section	AI	Visible	Contours	~	/		0.2 [mm] Metric	
All 1D Members	Concrete		Front section	AI	Visible	Contours	~	/		0.2 [mm] Metric	
All 1D Members	Concrete		Section	AI		Hatched section		/	Concrete	0.1 [mm] Metric	
All 2D Members	Concrete		Section	AI		Contours	-	/		0.2 [mm] Metric	
All 2D Members	Concrete		Back section	AI	Visible	Contours	-	/		0.2 [mm] Metric	
All 2D Members	Concrete		Front section	AI	Visible	Contours		/		0.2 [mm] Metric	
Wall	Concrete		Section	All		Hatched section	-	/	Concrete	0.1 [mm] Metric	
Plane section entity							7	1		0.2 [mm] Metric	
2D linegrid							7	<u>/</u>		0.1 [mm] Metric	
Storeys							5	/		0.2 [mm] Metric	
2D opening - edge	All		Section	All						0.2 [mm] Metric	
2D opening	All		Section	AI			2	/		0.2 [mm] Metric	
Steel connections	All		All	AI	Visible		~	1		0.1 [mm] Metric	
Foundation	All		All	AI	Visible	Contours	1	/		0.2 [mm] Metric	
All structure	All		All	Al	Visible	Contours	7	1		0.2 [mm] Metric	
All structure	All		All	All	All	Contours				0.2 [mm] Metric	
					ок с	ancel Hel	p				

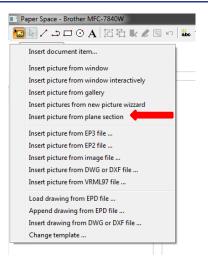
• Return to the drawing environment and click "Refresh" in the "Actions" to refresh the drawing.



- Click "Paperspace gallery" in the tool bars.
- Add a new page to the gallery. Name it something reasonable.

Paperspace gallery	• • • × • •
ا ا ا ا ا ا ا ا ا ا ا ا ا ا ا ا ا ا ا	
1	
Create a new picture.	
New Edit Delete	Close

- SCIA Engineer provides some default title blocks and you are also able to construct or add your own by importing .dwg or .dxf files.
- Select "Insert picture from plane section" from the drawing menu.



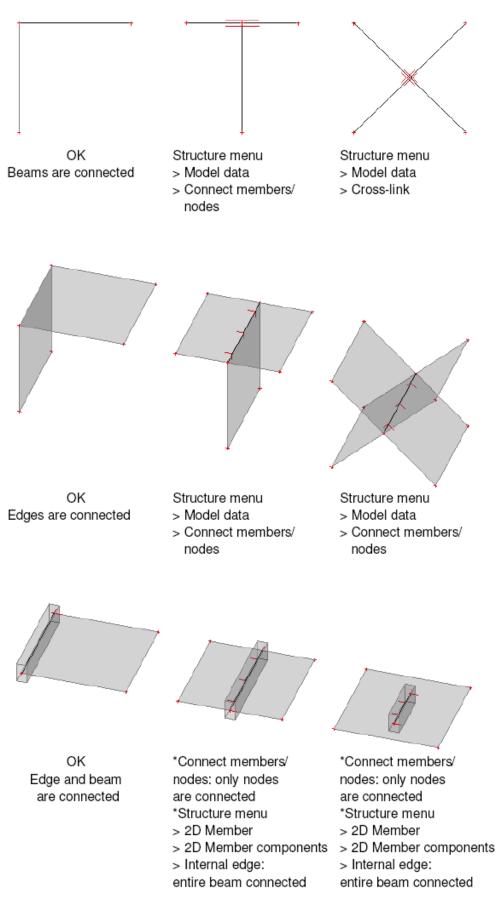
- Choose a drawing from the drawing manager and place it on the layout.
- There are many other items that can be placed on the layout pages including 3D views from the picture gallery, tables from the calculation reports and drawings from other CAD programs.

Conclusion

In this exercise we explored how members of the design team can share BIM models from various sources and authoring tools using open file formats like IFC. We also walked through the steps necessary to utilize this data in performing typical analysis and preparation of construction documentation. I hope this exercise has been interesting and informative. Please send any questions or comments to the author.

ANNEXES

Annex 1: Connection of entities

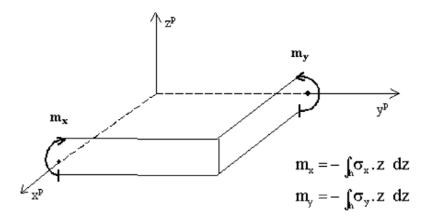


Annex 2: Conventions for the results on 2D members

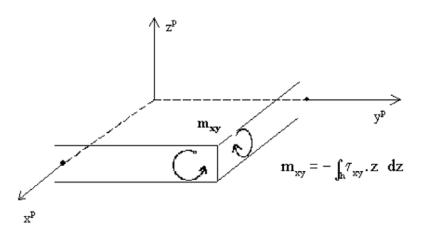
1 Basic magnitudes = Characteristic values

Bending (plates, shells)

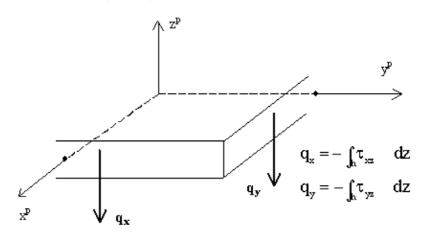
• Bending moments mx, my



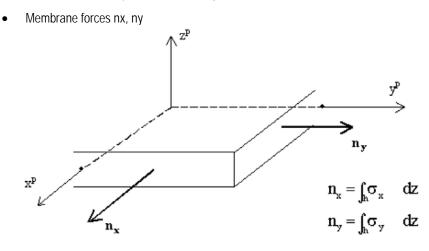
• Torsion moment mxy



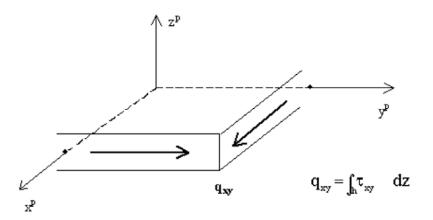
• Shear forces qx, qy (=vx, vy)



Membrane effects (walls, shells)



• Shear forces qxy (=nxy)



2 Principal magnitudes

The principal magnitudes give the results according to the axes of the directions of the largest stresses (principal directions). These directions are defined with the help of the circle of Mohr.

3 Dimensional magnitudes = Design values

To derive the dimensional magnitudes from the basic magnitudes, formulas from the Eurocode are used.

See also Help > Contents > Reference guide for these formulas.

Annex 3: Results in Mesh elements and Mesh nodes > 4 Locations

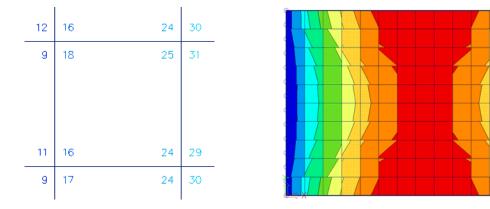
During a calculation in SCIA Engineer, the node deformations and the reactions are calculated exactly (by means of the displacement method). The stresses and internal forces are derived from these magnitudes by means of the assumed basic functions, and are therefore in the Finite Elements Method always less accurate.

The Finite Elements Mesh in SCIA Engineer exists of linear 3- and/or 4-angular elements. Per mesh element 3 or 4 results are calculated, one in each node. When asking the results on 2D members, the option 'Location' in the Properties window gives the possibility to display these results in 4 ways.

1 In nodes, no average

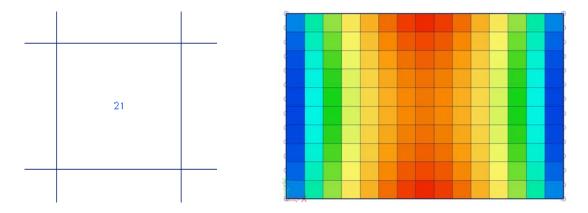
All of the values of the results are taken into account, there is no averaging. In each node are therefore the 4 values of the adjacent mesh elements shown. If these 4 results differ a lot from each other, it is an indication that the chosen mesh size is too large.

This display of results therefore gives a good idea of the discretisation error in the calculation model.



2 In centres

Per finite element, the mean value of the results in the nodes of that element is calculated. Since there is only 1 result per element, the display of isobands becomes a mosaic. The course over a section is a curve with a constant step per mesh element.



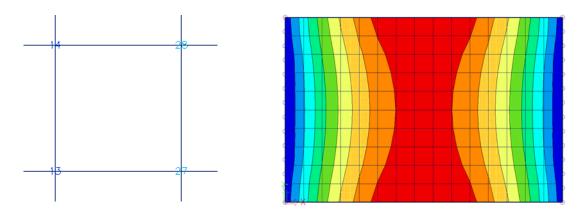
3 In nodes, average

The values of the results of adjacent finite elements are averaged in the common node. Because of this, the graphical display is a smooth course of isobands.

In certain cases, it is not permissible to average the values of the results in the common node:

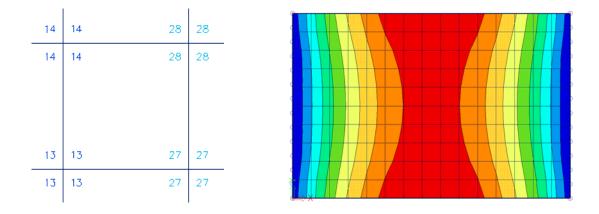
- At the transition between 2D members (plates, walls, shells) with different local axes.

- If a result is really discontinuous, like the shear force at the place of a line support in a plate. The peaks will disappear completely by the averaging of positive and negative shear forces.



4 In nodes, average on macro

The values of the results are averaged per node *only* over mesh elements which belong to the same 2D member and which have the same directions of their local axes. This resolves the problems mentioned at the option 'In nodes, average'.



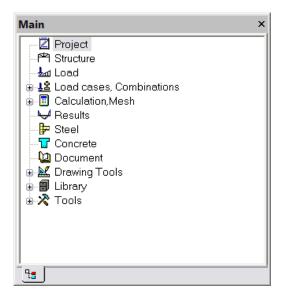
Accuracy of the results

If the results according to the 4 locations differ a lot, then the results are inaccurate and the mesh has to be refined. A basic rule for a good size of the mesh elements, is to take 1 to 2 times the thickness of the plate.

Annex 5: Overview of the Icons in Windows & Toolbars

1 Main Window

In the Main window one can find the links to the most used menus and actions. Some of those links are only activated when they can be effectively used in the project: e.g. the link to the Results menu is only shown after a calculation has been performed, and the Steel and/or Concrete menus are visible depending on the materials applied in the project.



2 Properties Window

The Properties window gives information about selected objects and/or actions. Moreover it is possible to adjust the properties of each object directly via this menu. When several kinds of objects or actions are selected at the same time, it is possible to switch between their properties by means of the little arrow behind the object name. The number between brackets behind the object name represents the number of objects of this kind that is selected at this moment. If the number is larger than 1, only the corresponding properties are shown.

Pro	operties	×
М	ember (1)	💌 Vi V/
Γ	Name	S1 🔨
	Туре	column (100) 🔹 🔽
	Analysis model	Standard 🔹
	CrossSection	CS1-HEA200 💌 📲
	Alpha	0 🔹
	Member system-line at	centre 🔹
	ez [mm]	0
	LCS	standard 🔹
	FEM type	standard 🔹
	Buckling and relative le	Default 💌
	Layer	Layer1 🔹
	Geometry	
	Length [m]	4,000
A	tions	
В	uckling data	>>>
Ľ	able edit geometry	_>>>



Select elements by more properties

Select elements by property

3 Menu Bar

E.	<u>F</u> ile	<u>E</u> dit	<u>V</u> iew	<u>L</u> ibraries	<u>T</u> ools	<u>M</u> odify	T <u>r</u> ee	<u>S</u> etup	<u>W</u> indow	<u>H</u> elp
----	--------------	--------------	--------------	-------------------	---------------	----------------	---------------	---------------	----------------	--------------

These 'written' menus group all actions by subject. A large number of these actions is also available in the Main menu and/or as icon in the toolbars.

4 Standard Toolbars



The 'Activity' toolbar provides options for the visibility / invisibility of specific parts of the structure, which increase the ease of working and the readability of the project.

Activity toggle
Activity by layers
Activity by selection (Selected members On)
Activity by selection (Selected members Off)
Activity by working plane
Activity by clipping box
Invert current activity
Draw inactive members
Basic 👻
🗅 🗃 🖬 💁 🕰 🗉 📍 Esa1

The 'Basic' toolbar contains a number of primary actions with regard to the current project and allows to modify the basic settings of the program (Setup Options).

×

New (Ctrl+N)
Open (Ctrl+O)
Save (Ctrl+S)
Lundo
Redo
E Setup
About SCIA Engineer
Esa1 Name of the opened *.esa file

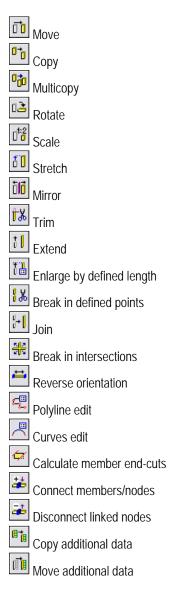


The 'View' toolbar allows for the execution of a whole lot of simple view manipulations.

\$ View in direction X
\$ View in direction Y
\$ View in direction Z
\$ View in direction AXO

<u>i</u> ∰⇒	View perpendicular to working plane
Ç,	Zoom in
2	Zoom out
R	Zoom by cut out
₽	Zoom all
	Zoom selection
1	Perspective view
	Undo view change
٦	Redo view change
C	Generate structural model
2	Regenerate view
Ge	ometry manipulations ×
Ō	哈哈哈哈份的放打 猫 探 汗来 辛 🖷 🥢 🐲 苯 饰 航

In the 'Geometry manipulations' toolbar one can find manipulations with basic entities (nodes, 1D members, 2D members), as well as with additional data.



Modelling Tools × Image: Im
Union of solids
Subtraction of solids
Intersection of solids
Division of solids
Generate vertexes
Clash check of solids
Move vertexes/points
Duringt

The 'Project' toolbar collects various actions, from the definition of databases (layers, materials, cross-sections) for the project, to several options for the output.

Units
B Layers
Materials
Cross-sections
Check structure data
Mesh generation
Calculation
Hidden calculation
Print data
Print picture
Picture gallery
Paperspace gallery
Selection of object

 Selection of object
 ×

 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №
 №

The 'Selection of object' toolbar contains different possibilities to select a specific part of the structure. A selection can also be saved and loaded again later on.

Selection by mouse
 Selection by cut out
 Selection by intersecting line
 Selection by polygonal cut out
 Select all

Relection by working plane

100
Previous selection
Cancel selection
Selection mode toggle (Select or Deselect)
Single selection mode toggle (All found or First found)
Save selection
Load selection
Filter for selection on/off
Filter by service tree on/off
Tools ×
Setup UCS
Clipping box
Clipping box
Clipping box Dot grid setting

nd for the input and graphical display of a structure. Ir

5 Command line Toolbars

Coordinates info

On the Command line itself, a number of commands for the operation of the program can be inputted. Also, during running actions, instructions on the next steps are shown. Apart from that, quite a number of toolbars can be found here; some of them are only available during a certain action or in a

specific menu.

0 0 🕹 🚣 🚝 📽 🦉 🐟 💵 📑 🗲				
Commandolijn				τ×
	$ X \times a \times $	< # 😓 🖂 💘 🛛		ta ե
 Show/Hide surfaces Render geometry Show/Hide supports Show/Hide loads Show/Hide other model data 			-514	
Show/Hide labels of nodes Show/Hide labels of members Show/Hide dot grid				
Set load case for display Fast adjustment of view parameters or	n whole model			

Fast adjustment of view parameters on selection
Cursor snap settings
🔨 🔪 🐼 🦟 🐨 🛹 💘 Change insertion point, available during input of geometry
Q ● ■ ■ ∠ C J × & 7 × Definition of new form, available during input of geometry
🖆 🕿 🗲 茾 🚎 🛱 Fast input of supports & hinges, available in Structure menu
ل الله الله الله الله الله الله الله ال
🚬 💯 👑 ど 🆻 🖌 🕴 🛣 Fast display of results, available in Results menu
Adjust Units of length
Plane XY Change Active working plane
Snap mode Adjust Cursor snap settings
Current UC: Adjust UCS (=User Co-ordinates System)
Change Active code