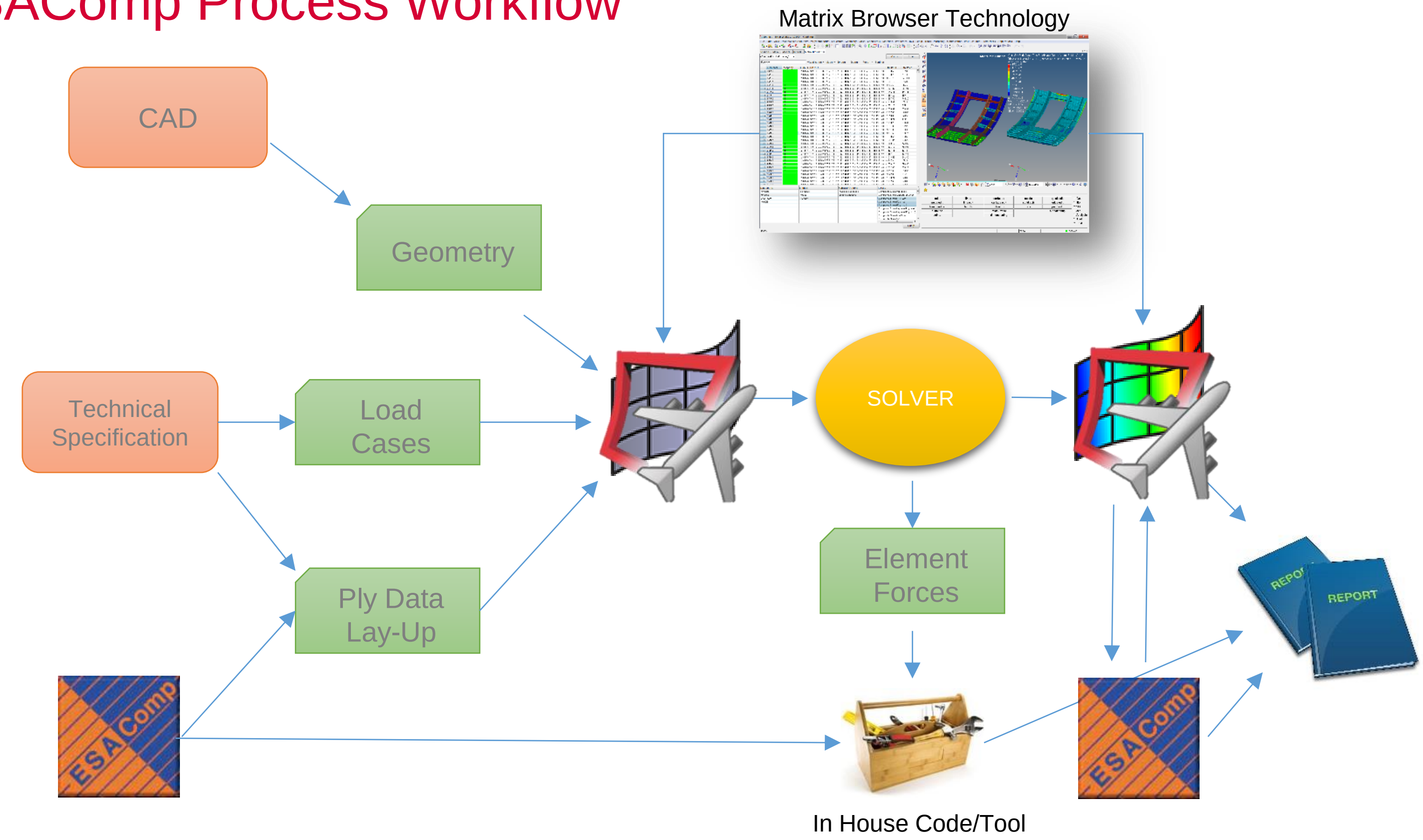
ESAComp – HyperWorks Interface

recommended workflows

André Mönicke, Componeering Inc.
Frank Ehrhart, Altair Engineering GmbH

COMPONEERING Inc.

HW/ESAComp Process Workflow



What are the benefits?

- One integrated work environment
- No additional customization needed to exchange data between software
- Easy-to-use
- Graphical display of ESAComp results within HyperWorks, including reserve factors, calculated failure modes etc.
- Loop over several load cases, allows fast identification of critical load cases, critical elements, critical plies, failure modes
- Exchange of information between HyperWorks and ESAComp by means of standard file formats (CSV)
- Quick reporting functionalities using HyperWorks OTB capabilities

- Note: For now supported on Nastran and OptiStruct solver base, functionality is planned to be expanded to all solver profiles

Reference model and workflow example

Reference Model: Material

Ply : **T800;Epoxy;UD-.425/306/40**

Keywords : Fiber;Carbon;Toray;Matrix;Epoxy;
Modified : Fri Jan 08 12:08:11 2010

Physical nature : **reinf.ply** Mech. behavior : **transv.is.23**
Form of reinf. : unidirectional

t = 0.425 mm	m_A = 603.5 g/m²	V_f = 40%
	rho = 1420 kg/m³	f_1/2 = 100/0%

Engineering constants (transv.is.23)

E_1 = 105 GPa	G_12 = 3.5 GPa	nu_12 = 0.3
E_2 = 7.5 GPa	G_31 = 3.5 GPa	nu_13 = 0.3
E_3 = 7.5 GPa	G_23 = 2.88462 GPa	nu_23 = 0.3

Thermal and moisture expansion coeff. (transv.is.23)

alpha_1 = -0.45 e-6/°C	beta_1 = 0 e-2/w%
alpha_2 = 30 e-6/°C	beta_2 = 0.6 e-2/w%
alpha_3 = 30 e-6/°C	beta_3 = 0.6 e-2/w%

First failure stresses and strains - Nominal (transv.is.23)

X_t / X_eps,t = 1350 MPa / 1.28571 %	X_c / X_eps,c = 1000 MPa / 0.952381 %
Y_t / Y_eps,t = 35 MPa / 0.466667 %	Y_c / Y_eps,c = 160 MPa / 2.13333 %
Z_t / Z_eps,t = 35 MPa / 0.466667 %	Z_c / Z_eps,c = 160 MPa / 2.13333 %
S / S_eps = 55 MPa / 1.57143 %	(12)
R / R_eps = 55 MPa / 1.57143 %	(31)
Q / Q_eps = 34.6154 MPa / 1.2 %	(23)

Ultimate stresses and strains - Nominal (transv.is.23)

X_t / X_eps,t = 1350 MPa / 1.29 %	X_c / X_eps,c = 1000 MPa / 0.97 %
Y_t / Y_eps,t = - MPa / - %	Y_c / Y_eps,c = 160 MPa / 2.2 %
Z_t / Z_eps,t = - MPa / - %	Z_c / Z_eps,c = 160 MPa / 2.2 %
S / S_eps = 55 MPa / 2 %	(12)
R / R_eps = 55 MPa / 2 %	(31)
Q / Q_eps = - MPa / - %	(23)

ESAComp Data Base

Name	Value
Solver Keyword	MAT8
Name	T800_EPOXY_UD_05
ID	1
Color	
Include File	[Master Model]
Defined	☑
Card Image	MAT8
User Comments	Do Not Export
E1	105000.0
E2	7500.0
NU12	0.3
G12	3500.0
G1Z	3500.0
G2Z	2900.0
RHO	1.4e-009
A1	-4.5e-007
A2	3e-005
TREF	300.0
Xt	1350.0
Xc	1000.0
Yt	35.0
Yc	160.0
S	55.0
GE	
F12	

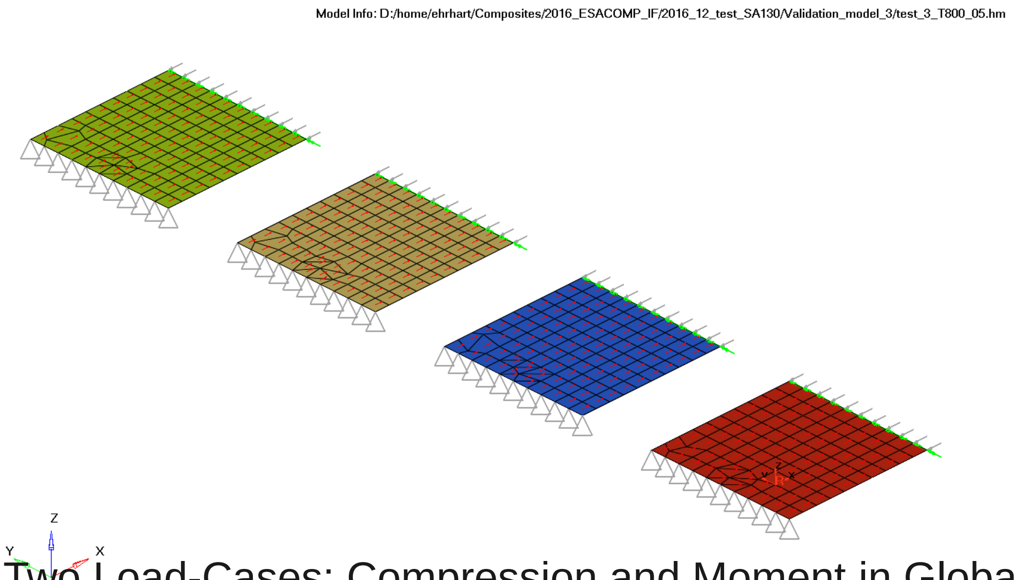
MAT8 definition for OptiStruct (N,mm, model setup)

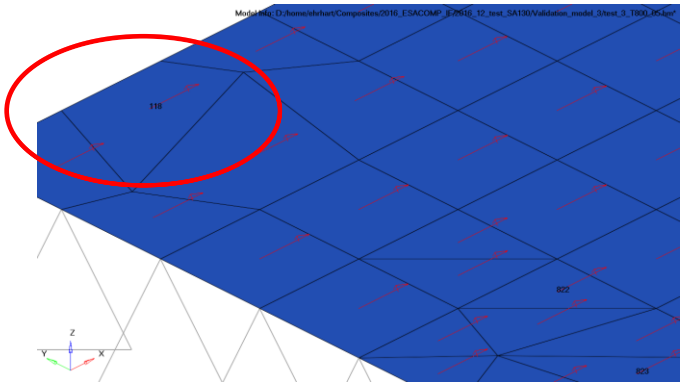
Property definition: 2 layups with 2 different offset options

Symmetric lay-up		Non-symmetric lay-up	
Symmetric material (Z0 = -LAM_thickness/2)	First ply on shell-element Z0 = 0	Symmetric material (Z0 = -LAM_thickness/2)	First ply on shell-element Z0 = 0
0	0	0	0
90	90	90	90
45	45	90	90
-45	-45	-45	-45
-45	-45	90	90
45	45	-45	-45
90	90	45	45
0	0	0	0

Each ply having is thickness of 0.5 mm

Load cases and Element types

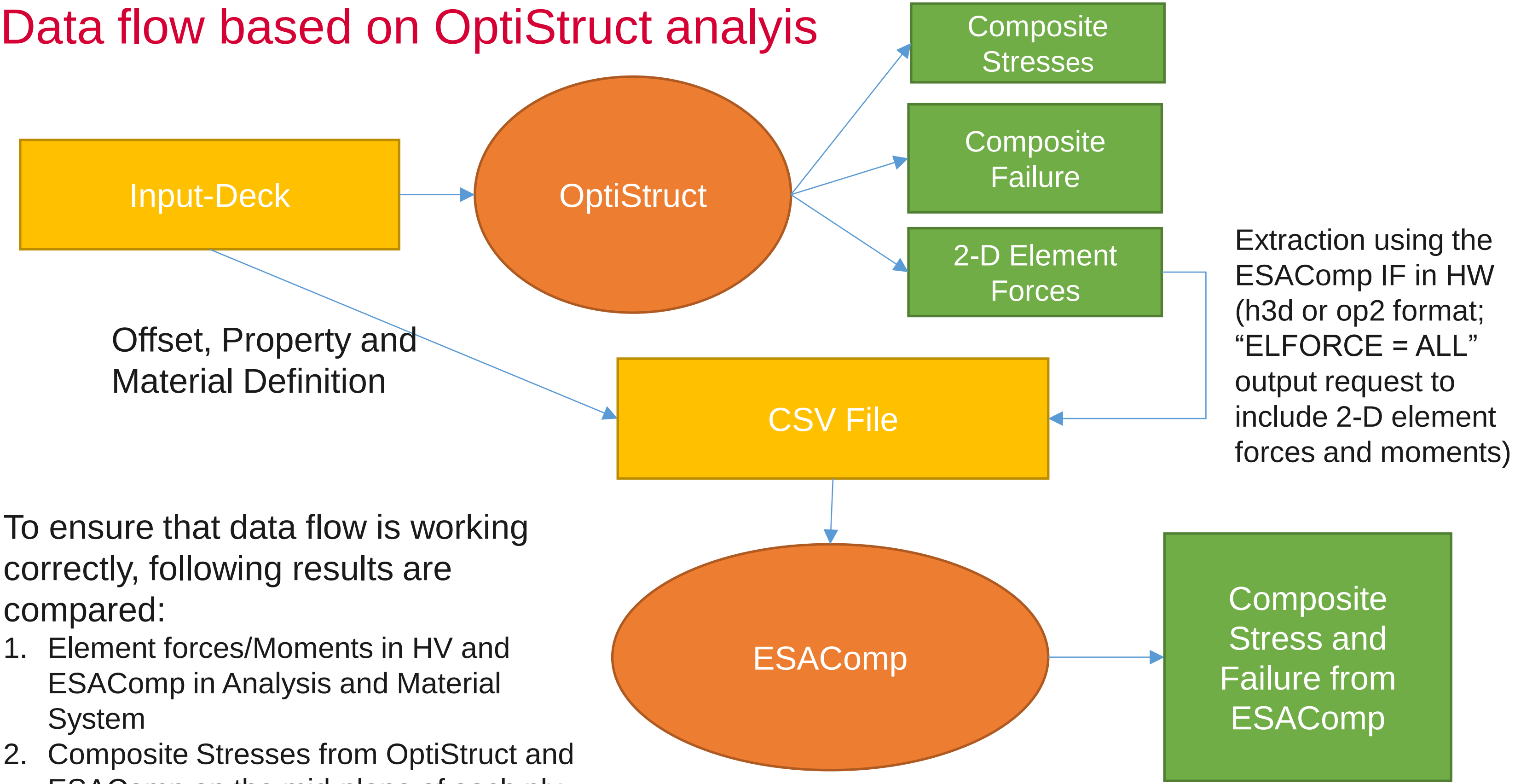




Two Load-Cases: Compression and Moment in Global y direction.

One distorted Quad-Element will be used as Validation element (Element 118, LC Compression, Property PCOMPG_SYM_Z0_0 with Z0 = 0).

Data flow based on OptiStruct analyis



To ensure that data flow is working correctly, following results are compared:

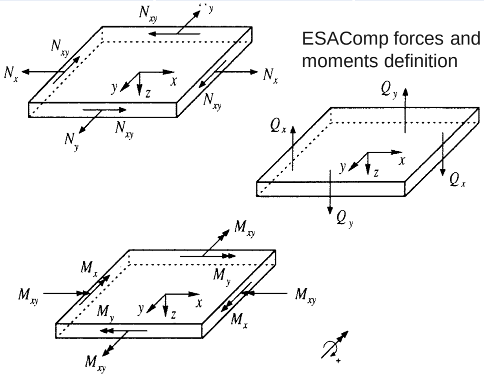
1. Element forces/Moments in HV and ESAComp in Analysis and Material System
2. Composite Stresses from OptiStruct and ESAComp on the mid-plane of each ply

Comparison of 2-D element forces/moments

EL Id	Tensor components in Elemental system							
HW [N,mm] system	XX	YY	XY	Moment XX	Moment YY	Moment XY	Force ZX	Force YZ
118	-71.58	-1086.0	-210.8	68.23	347.9	162.0	7.44	13.83
ESAComp [N,m] system	N-x	N-y	N-xy	M-x	M-v	M-xv	Q-x	Q-v
118	-71583	-1086417	-210821	-68.23	-347.93	-161.98	7443.6	13834.45

The elemental forces in Elemental system are the same in both tools. However, ESAComp uses SI unit system, therefore the Forces per Unit length are scaled with a factor.

Due to the opposite sign convention of the moments in ESAComp (compared to Nastran and OptiStruct), HV displays positive moments, whereas ESAComp shows negative Moments.



ESAComp forces and moments definition

Comparison of 2-D element forces/moments (in ESAComp notation)

EL Id	Tensor Components in Material system							
HW [N,mm] system	XX	YY	XY	Moment XX	Moment YY	Moment XY	Force ZX	Force YZ
118	-1226	-32.47	56.7	**388.4**	**27.1**	**-115.2**	14.77	-5.36
ESAComp [N,m] system	N-x	N-y	N-xy	M-x	M-v	M-xv	Q-x	Q-v
118	-1125533	-32468	56741.95	**1862.62**	**37.22**	**1.69**	14765	-5364

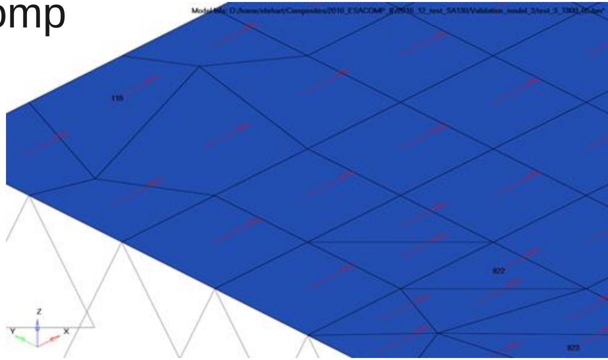
The elemental forces in Material system are the same in both tools. However, ESAComp uses SI unit system, therefore the Forces per Unit length are scaled with a factor.

The different values of the moments are caused by a different transformation:

- HV rotates the element forces and moments from elemental to material system (both are in the same plane here)
- ESAComp uses element forces and moments at mid-plane for post-processing. When Z0 unequal -1/2*thickness of the property, the element forces induce moments. Thus ESAComp rotates the element forces and moments, and additionally compensates the moments for the offset between FE reference plane and mid-plane of the laminate.

Comparison of composite stresses in-plane stresses per ply

EsaComp



Ply	theta	sig_1 MPa	sig_2 MPa	tau_12 MPa
1	0	-1698.4	-15.9	13.5
2	90	235.6	-96.2	-10.6
3	45	-348.8	-47.7	46.1
4	-45	-400.3	-26.5	-36.2
5	-45	-226.1	-16.8	-26.2
6	45	-86.3	-5.4	16.3
7	90	154.9	0.9	4.1
8	0	246.4	14.8	-7.0

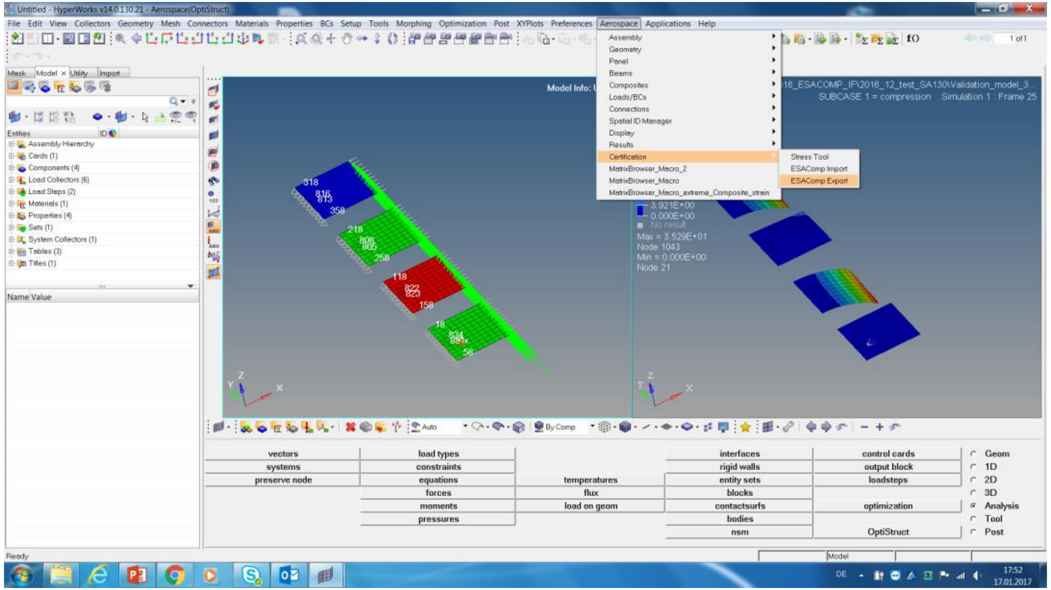
OptiStruct

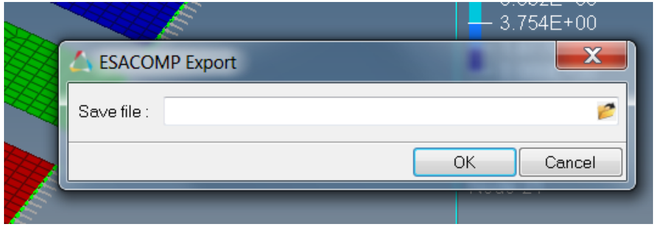
Ply	theta	sig_1 MPa	sig_2 MPa	tau_12 MPa
1	0	-1698.4	-15.9	13.5
2	90	235.6	-96.2	-10.6
3	45	-348.8	-47.7	46.1
4	-45	-400.3	-26.5	-36.2
5	-45	-226.1	-16.8	-26.2
6	45	-86.3	-5.4	16.3
7	90	154.9	0.9	4.1
8	0	246.4	14.8	-7.0

Both tools calculate the same in-plane stresses in each ply. The IF is working correctly in case the models are built according to the reference model.

Workflow: Step by step export to ESAComp

Step 1: Export from HW to ESAComp



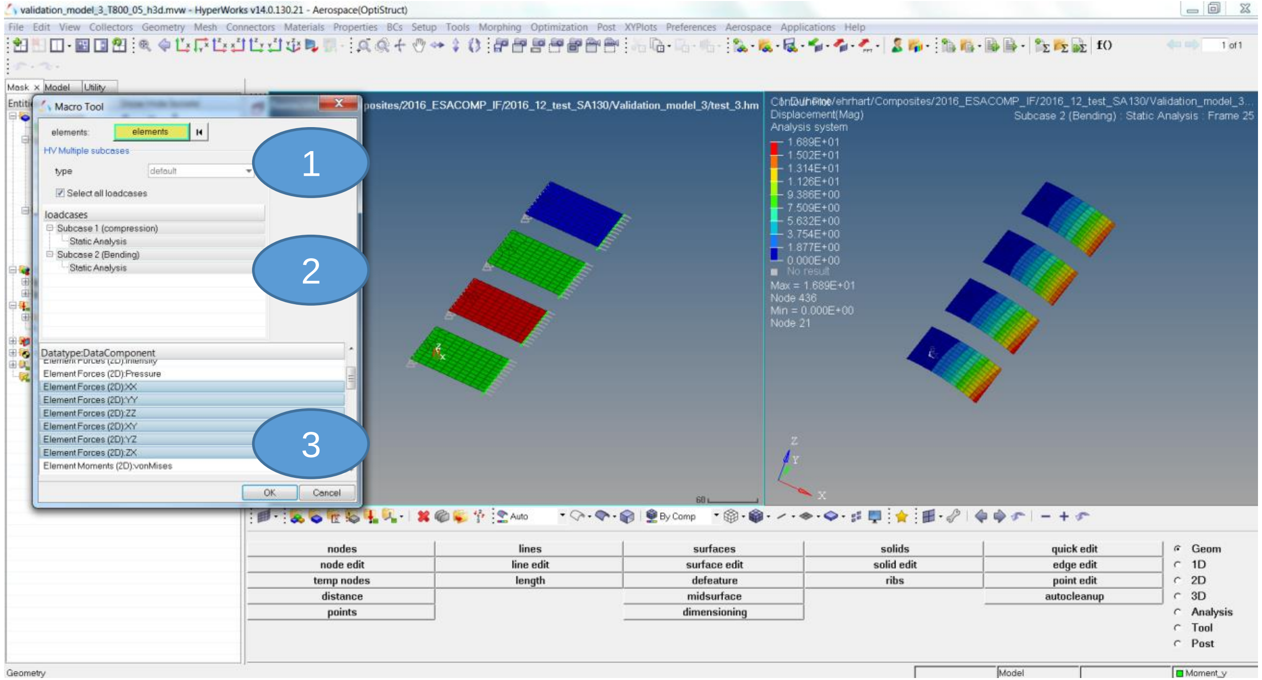


1. Start ESAComp Export via Aerospace => Certification =>ESAComp Export (Aerospace toolbar is available in Engineering Solutions => Aerospace profile)

2.Define out put csv file

Step 2: Select Load Cases elements and element forces

1. Select elements
2. Select load cases
3. Select element forces and moments
4. Hit Ok and the CSV file is created



3. CSV File for ESAComp input

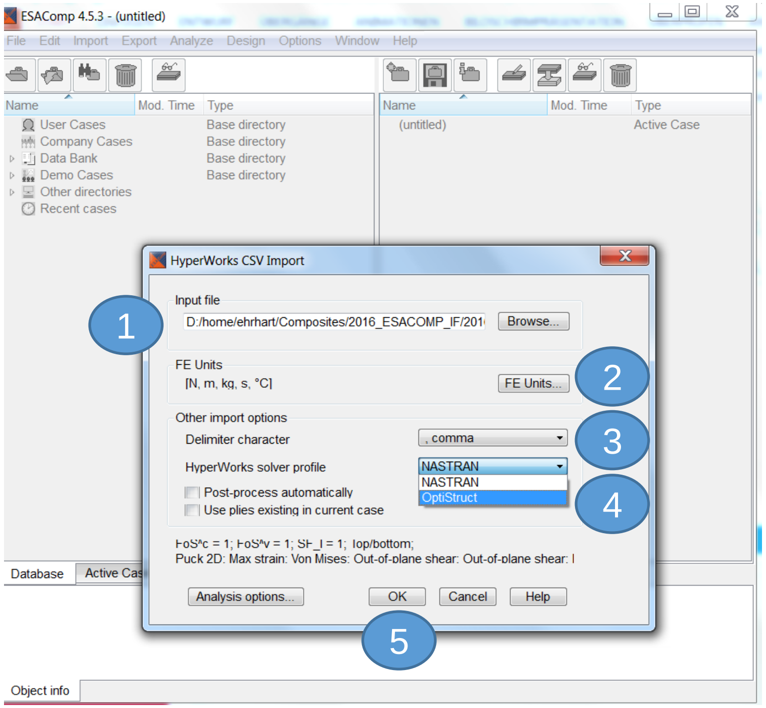
materials	mat_name	MAT8	MAT8_E1	MAT8_E2	MAT8_NU12	MAT8_G12	MAT8_RHO	MAT8_A1	MAT8_A2	MAT8_G1Z	MAT8_G2Z	MAT8_Xt	MAT8_Xc	MAT8_Yt	MAT8_Yc
1	T800_EPOXY_	0	105000	7500	0.3	3500	1.40E-09	-4.50E-07	3.00E-05	3500	2900	1350	1000	35	160
properties	prop_name	cardimage	cardimagetyp	PCOMPG	PCOMP_CON	Number_of_P	PCOMP_SB	PCOMP_FT	PCOMP_TREF	PCOMPG_GPI	PCOMP_MID	PCOMP_T	PCOMP_THET	PCOMP_SOU	PCOMP_Z0
1	PCOMPG_SYN	PCOMPG	2D	0	1	8	0		0	1 2 3 4 5 6 7 8	1 1 1 1 1 1 1 1	0.5 0.5 0.5 0.5	0 90 45 -45 -4	YES YES YES Y	-2
2	PCOMPG_SYN	PCOMPG	2D	0	1	8	0		0	1 2 3 4 5 6 7 8	1 1 1 1 1 1 1 1	0.5 0.5 0.5 0.5	0 90 45 -45 -4	YES YES YES Y	0
3	PCOMPG_NO	PCOMPG	2D	0	1	8	0		0	1 2 3 4 5 6 7 8	1 1 1 1 1 1 1 1	0.5 0.5 0.5 0.5	0 90 90 -45 90	YES YES YES Y	-2
4	PCOMPG_NO	PCOMPG	2D	0	1	8	0		0	1 2 3 4 5 6 7 8	1 1 1 1 1 1 1 1	0.5 0.5 0.5 0.5	0 90 90 -45 90	YES YES YES Y	0
elements	property	subcaseid	Element Force	Element Mom	Element Force	Element Mom	Element Force	Element Force	Element Mom	Element Mom	Element Force	Element Mom	Element Force	Element Mom	THETA
18	1	1	-1059.37158	0	0	0	-94.1213837	0	0	0	279.209167	0	0	0	108.148
58	1	1	-1111.55237	0	0	0	-12.4581108	0	0	0	38.9657631	0	0	0	90
831	1	1	-1046.36499	0	0	0	-287.848236	0	0	0	-136.263458	0	0	0	78.3861
834	1	1	-1056.8479	0	0	0	-195.828018	0	0	0	-116.862534	0	0	0	-95.8003
118	2	1	-1086.41736	161.982224	7.44365025	0	-71.5839157	13.8344545	347.927643	0	-210.821533	68.2315216	0	0	81.6827
158	2	1	-1111.47229	8.01672363	-5.13194084	0	-12.7700672	14.9737024	-149.373398	0	39.7924461	3.05524564	0	0	90
822	2	1	-1061.11377	96.5195389	-1.88315046	0	-175.565125	4.65622711	-76.7484512	0	-24.4352894	220.352264	0	0	-90
823	2	1	-1048.89868	-86.7677078	0.22668755	0	-291.619751	-6.32165194	-151.892685	0	-140.627182	353.248993	0	0	78.2014
218	3	1	-1130.30981	96.9369965	-1.33578301	0	-11.7969131	-17.8537464	-370.421326	0	172.720551	6.54020691	0	0	100.418
258	3	1	-1129.45349	-116.706284	7.52396917	0	-11.7052984	-19.2115841	113.383919	0	31.5541744	41.9008942	0	0	90
805	3	1	-987.659485	-69.7955627	-3.58686829	0	-252.124908	4.90143204	119.828957	0	253.367188	226.792664	0	0	105.732
808	3	1	-988.39679	-78.8209915	1.21787381	0	-189.190582	-3.20157099	51.0718193	0	-241.236816	129.492416	0	0	-104.579
318	4	1	-1111.55615	-4.54870701	-0.08503389	0	-8.78311634	5.34243631	-110.603752	0	151.304443	-10.1473207	0	0	98.2274
358	4	1	-1114.09839	-44.6571999	-0.97512907	0	-3.45273805	5.20308065	221.598434	0	7.93914461	9.7947197	0	0	90
813	4	1	-1055.62183	-24.2679691	-0.55147672	0	-139.944687	-3.32266951	-32.8759651	0	147.883667	207.345184	0	0	98.5891
816	4	1	-1051.05908	-65.3229523	1.74405456	0	-101.057617	5.06850863	-112.303009	0	-131.045868	122.243462	0	0	-97.4435
elements	property	subcaseid	Element Force	Element Mom	Element Force	Element Mom	Element Force	Element Force	Element Mom	Element Mom	Element Force	Element Mom	Element Force	Element Mom	THETA
18	1	2	0	273.576416	-0.28329849	0	0	8.27237034	-832.536011	0	0	-92.3065948	0	0	108.148
58	1	2	0	42.3194427	-2.92646742	0	0	7.40331936	-1185.50195	0	0	-9.86583614	0	0	90
831	1	2	0	-181.289749	0.10052246	0	0	-3.05629611	-1128.93738	0	0	-113.045593	0	0	78.3861
834	1	2	0	-80.7169952	-0.56493944	0	0	1.86763692	-1093.59375	0	0	-71.3156586	0	0	-95.8003
118	2	2	0.16182362	-129.221161	4.01862717	0	-0.60683453	6.88806915	-912.706665	0	-0.32243282	-34.0587158	0	0	81.6827

Material

Property

Element forces per element and load case

4. Import to ESAComp



1. Choose csv-file
2. Define unit system of the fem-model
3. Choose delimiter character (comma or semicolon)
4. Choose source of origin
   OptiStruct or Nastran
5. Hit OK

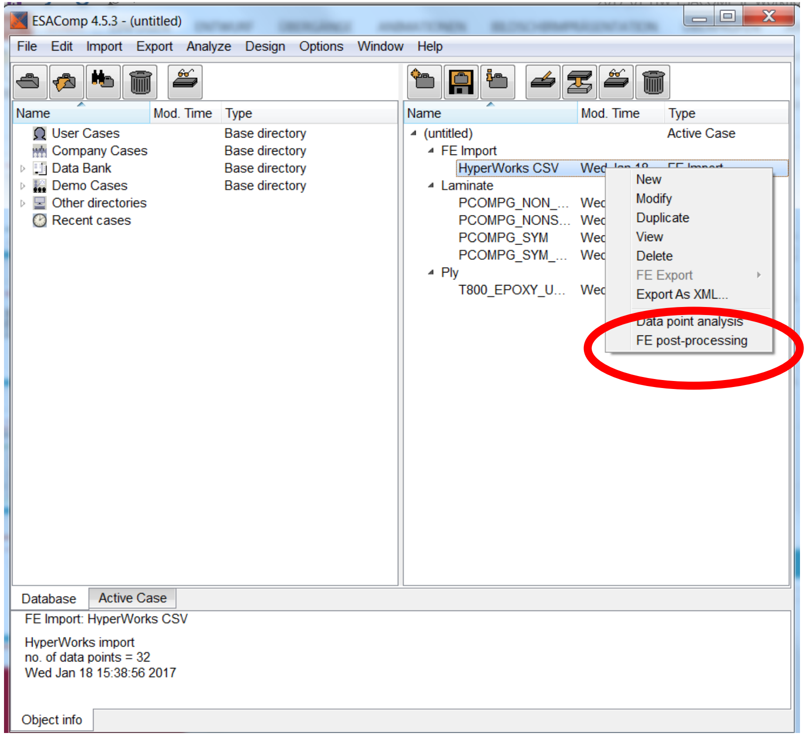
Options:

a) Post process automatically:
   - post-processing is done immediately after import. „...results.csv“ and „...layer_results.csv“ are created in the folder from which HyperWorks data was imported)

b) Use plies existing in current case:
   - if ESAComp has materials with the same ID in the current case file, it will use instead the ones from HyperWorks import
   - Edit => Extension variables allows the user to modify the FE_Mat_ID for each ply

c) Analysis options:
   - Allow to set various failure criteria for the different types of plies

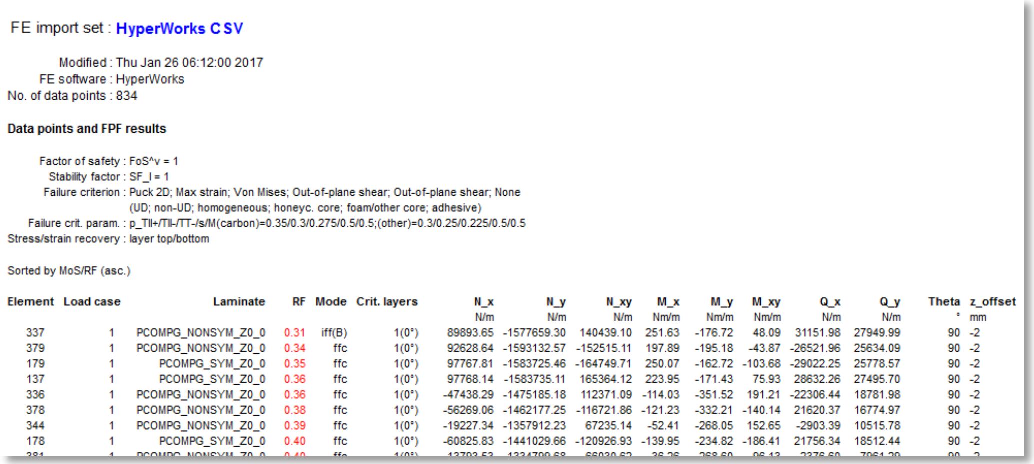
5. Analyse the laminate in ESAComp



Run FE post analysis either automatically after import or later using the FE post-processing feature.

Two files are created after automatic post-processing. The one named „...layer_results.csv" is needed in HyperWorks.

Results can be ordered and filtered already to check most crtitical elements.



FE import set : **HyperWorks CSV**

Modified : Thu Jan 26 06:12:00 2017
FE software : HyperWorks
No. of data points : 834

Data points and FPF results

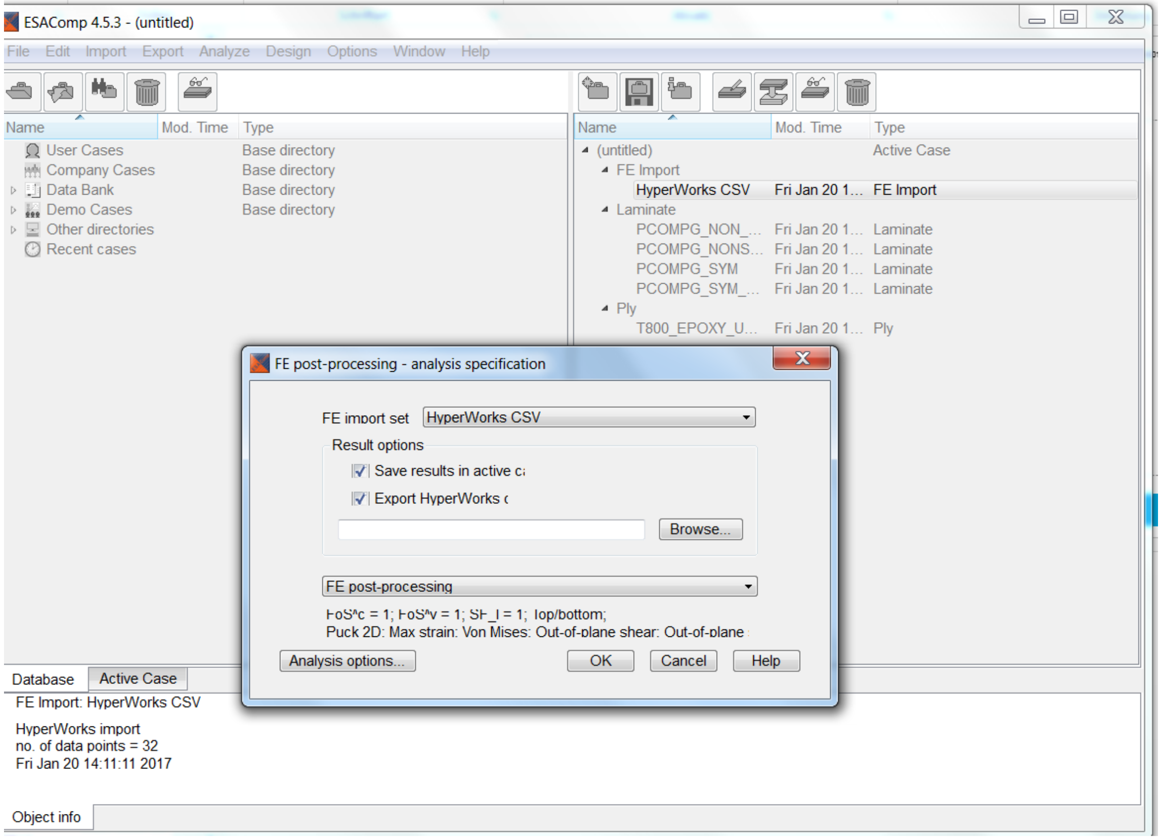
Factor of safety : FoS^v = 1
Stability factor : SF_I = 1
Failure criterion : Puck 2D; Max strain; Von Mises; Out-of-plane shear; Out-of-plane shear; None (UD; non-UD; homogeneous; honeyc. core; foam/other core; adhesive)
Failure crit. param. : p_TII+/TII-/TT-/s/M(carbon)=0.35/0.3/0.275/0.5/0.5;(other)=0.3/0.25/0.225/0.5/0.5
Stress/strain recovery : layer top/bottom

Sorted by MoS/RF (asc.)

Element	Load case	Laminate	RF	Mode	Crit. layers	N_x	N_y	N_xy	M_x	M_y	M_xy	Q_x	Q_y	Theta	z_offset
						N/m	N/m	N/m	Nm/m	Nm/m	Nm/m	N/m	N/m	°	mm
337	1	PCOMPG_NONSYM_Z0_0	0.31	iff(B)	1(0°)	89893.65	-1577659.30	140439.10	251.63	-176.72	48.09	31151.98	27949.99	90	-2
379	1	PCOMPG_NONSYM_Z0_0	0.34	ffc	1(0°)	92628.64	-1593132.57	-152515.11	197.89	-195.18	-43.87	-26521.96	25634.09	90	-2
179	1	PCOMPG_SYM_Z0_0	0.35	ffc	1(0°)	97767.81	-1583725.46	-164749.71	250.07	-162.72	-103.68	-29022.25	25778.57	90	-2
137	1	PCOMPG_SYM_Z0_0	0.36	ffc	1(0°)	97768.14	-1583735.11	165364.12	223.95	-171.43	75.93	28632.26	27495.70	90	-2
336	1	PCOMPG_NONSYM_Z0_0	0.36	ffc	1(0°)	-47438.29	-1475185.18	112371.09	-114.03	-351.52	191.21	-22306.44	18781.98	90	-2
378	1	PCOMPG_NONSYM_Z0_0	0.38	ffc	1(0°)	-56269.06	-1462177.25	-116721.86	-121.23	-332.21	-140.14	21620.37	16774.97	90	-2
344	1	PCOMPG_NONSYM_Z0_0	0.39	ffc	1(0°)	-19227.34	-1357912.23	67235.14	-52.41	-268.05	152.65	-2903.39	10515.78	90	-2
178	1	PCOMPG_SYM_Z0_0	0.40	ffc	1(0°)	-60825.83	-1441029.66	-120926.93	-139.95	-234.82	-186.41	21756.34	18512.44	90	-2

Workflow: Step by Step import of ESAComp Results to HW

Step 1: Run Analysis in ESAComp and export results



If automatic post-processing was skipped, run FE post-processing feature.

Two files are created. The one named „...layer_results.csv” is needed in HyperWorks.

2. Created outputs are (in two csv-files)

Here the more detailed „...layer_results.csv“ is shown.

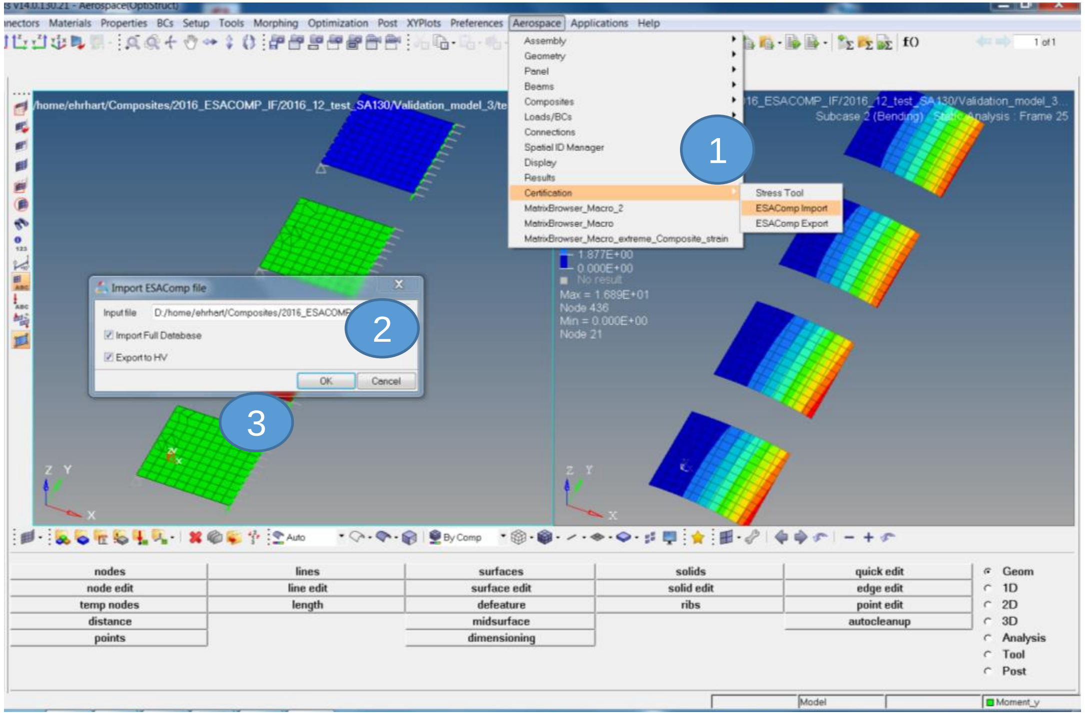
elements	load case	RF	FailureMode	FailedLayer
18	1	1.27641	1c	(0°)
18	2	2.25666	2t	7(90°)
58	1	1.28848	1c	(0°)
58	2	1.74755	2t	7(90°)
118	2	2.24416	2t	7(90°)
118	1	0.625422	1c	1(0°)
158	1	0.533882	1c	1(0°)

Minimum RF per Element
Failure mode
Failed layer
(Critical layer ID and orientation)

elements	load case	RF	FailureMode	CriticalLayerIı	CriticalLayerII	CriticalLayerC	Ply_1 RF Failu	Ply_2 RF Failu	Ply_3 RF Failu	Ply_4 RF Failu	Ply_5 RF Failu	Ply_6 RF Failu	Ply_7 RF Failu	Ply_8 RF Failur
18	1	1.27641	1c	(0°)			1 1.27641 1c	2 3.63847 2c	3 2.45975 s	4 2.16962 s	5 2.16962 s	6 2.45975 s	7 3.63847 2c	8 1.27641 1c
18	2	2.25666	2t	7(90°)	7	90	1 2.36832 1c	2 9.49514 2c	3 11.5383 s	4 24.3701 1c	5 16.4498 2t/	6 4.68799 2t	7 2.25666 2t	8 2.6103 1t
58	1	1.28848	1c	(0°)			1 1.28848 1c	2 3.67602 2c	3 2.24327 s	4 2.43434 s	5 2.43434 s	6 2.24327 s	7 3.67602 2c	8 1.28848 1c
58	2	1.74755	2t	7(90°)	7	90	1 1.81466 1c	2 7.5416 2c	3 7.22867 s	4 24.0144 s	5 11.7636 2t	6 4.10433 2t/	7 1.74755 2t	8 2.06244 1t
118	2	2.24416	2t	7(90°)	7	90	1 2.39143 1c	2 9.56146 2c	3 11.6683 s	4 24.2889 1c	5 16.2389 2t/	6 4.61023 2t	7 2.24416 2t	8 2.55998 1t

and RF of each ply for each load-case

3. Import Results to HW



1. Start ESAComp import
2. Select csv-file
   1. choose options
3. Hit ok

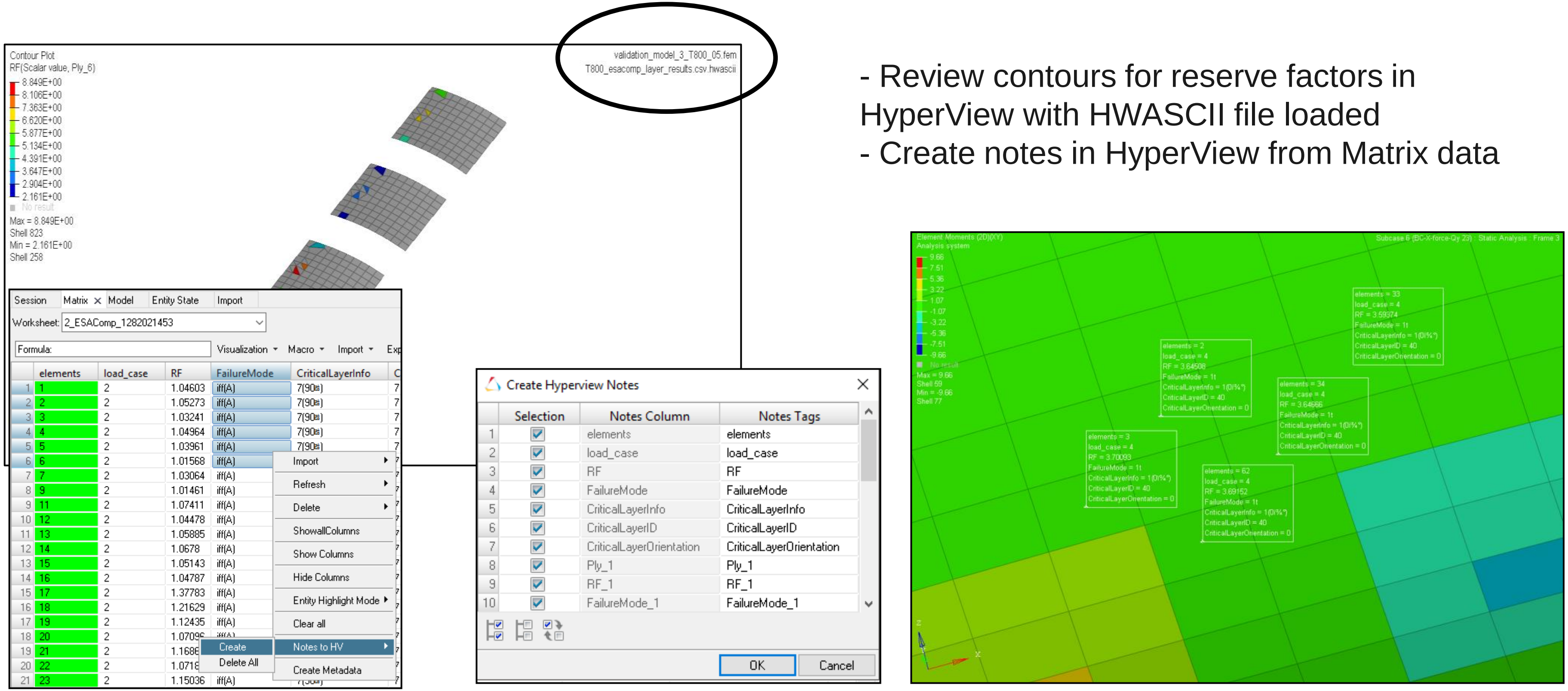
Options:

a) Import full database
   - When selected all ply reserve factors and failure types are read by the matrix browser additionally to the laminate results. Might be slow for high number of elements.

b) Export to HV
   - The import always creates as HWASCII file that can be read in HyperView. When selected the HWASCII file automatically becomes the result file in HyperView.

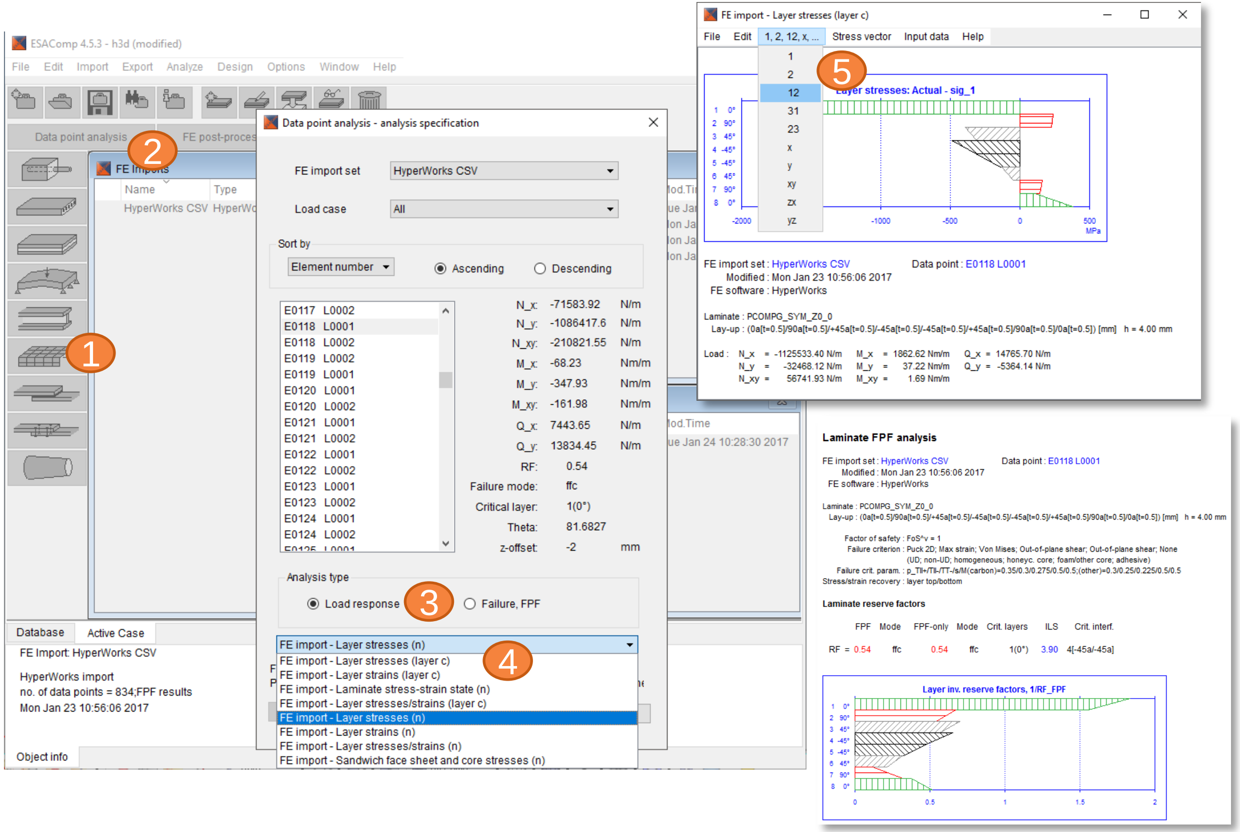
4. Review ESA-Comp Results in HV

- Review contours for reserve factors in HyperView with HWASCII file loaded
- Create notes in HyperView from Matrix data



5. Check details in ESAComp

- Knowing locations of interes identified in HyperView create through the thickness plots for stresses, strains and reserve factors (inverse reserve factors and margin of safety respectively)
- Change set of failure criteria for post-processing whole set or single elements for comparison



Known Issues

HyperWorks 14.130 and ESAComp 4.6

- Which Software-Versions should be used:
  - **HW 14.130** including a Software Patch provided by Altair and **ESAComp 4.6**
- Why should exactly these versions (or newer) be used ?
  - There are known Issues with older versions of HyperWorks and ESAComp, which lead to wrong results calculated in ESAComp

HyperWorks 14.130 and ESAComp 4.6

- **Recommended workflow for bigger size models**
  - Use HyperView for first level post-processing and identify critical areas
    - HyperWorks 14.130 will allow the evaluation of PUCK 2D inside HyperView
    - OptiStruct 2017 calculates PUCK 2D directly
  - Export the hotspots to ESAComp for detailed analysis
  - Import the Hotspot results back to HyperWorks for further post-processing and report-creation
- **Why do we recommend this workflow?**
  - The current implementation via Matrix-Browser and Excel-Format as transfer, is limited due to performance to roughly **1.000 to 10.000 elements** maximum
  - Using more elements, there is no smooth user-experience and each step can be time consuming => No interactice workflow anymore
  - There is an ESAComp batch-mode , which solves some performance issues using more elements
- **Please cross-check first, that ESAComp in batch and manual deliver the same results.**

The batch currently has two versions:
1. -hwb used to execute the post processing
"C:\Program Files (x86)\ESAComp 4.6\ESAComp46.exe" -hwb HW-export.csv
 (default FE import units and analysis settings need to be modified in advance)

2. -hw used to import the file and open ESAComp in GUI mode
"C:\Program Files (x86)\ESAComp 4.6\ESAComp46.exe" -hwb HW-export.csv

COMPONEERING Inc.

HyperWorks 14.130 and ESAComp 4.6: Known Issues

Element/Property offset:

In case the property/Element Offset is not defined in the model, ESAComp can interpret the element/property offset z0 wrongly, as the HyperWorks export currently contains z0=0 in this case rather than -1/2*thickness, which is the default Nastran or OptiStruct revert to when z0 is undefined.

Solution:

Define the element/property offset without a blank in HyperMesh (for example using Matrix Browser), before exporting to ESAComp

MAT1 used as ply-material

In case MAT1 is used as ply material, the material properties are not written to the csv and ESAComp crashes during the analysis

Solution:

Redefine as MAT8 in HyperMesh before import and change to isotropic material manual un ESAComp

Out-of-plane allowables:

Out of plane allowables for Core Strength analysis (stress or strain) are not defined in HyperMesh and need to added manually in ESAComp. Alternatively, existing ESAComp data can be used.

HyperWorks 14.130 and ESAComp 4.6

Planned fixes:

- Correct transfer of element and property offsets

Ideas for further improvement, (based on customer feedback):

- Support of isotropic materials
- Increasing performance:
  - Further integration of ESAComp into HyperWorks possible, which would avoid the different ASCII formats and opens the IF to full-size models
- Adding more solvers and properties
- . . . .

More Information

ESACamp

www.esacomp.com

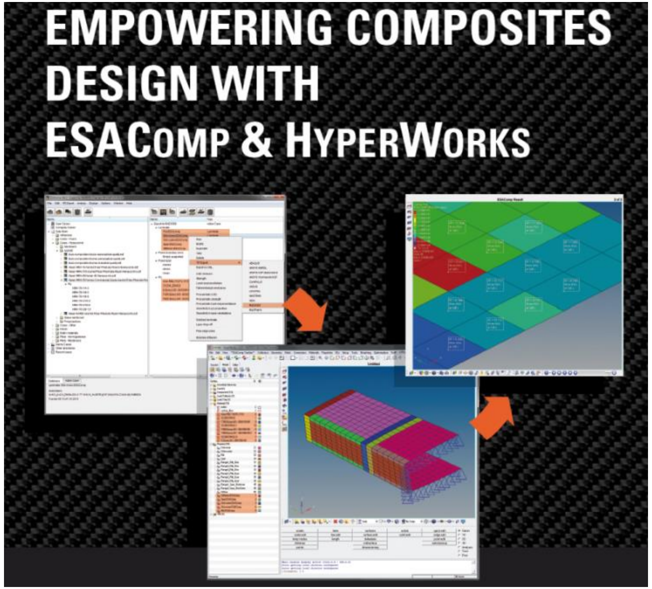
esacomp@componeering.com

+358 9 4342 1550 (English)

+49 34324 18339 (German/English)

HyperWorks

www.altairhyperworks.com



COMPONEERING Inc.

Altair