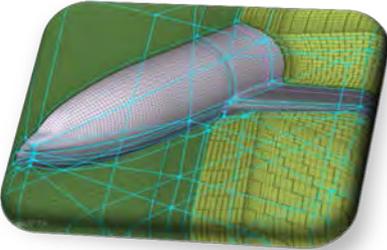
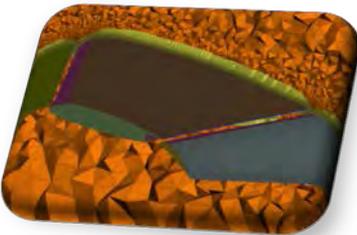
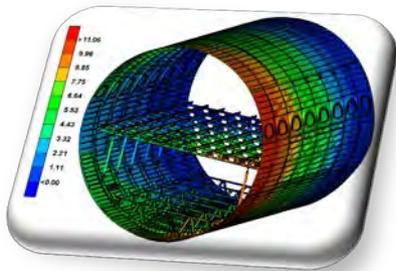
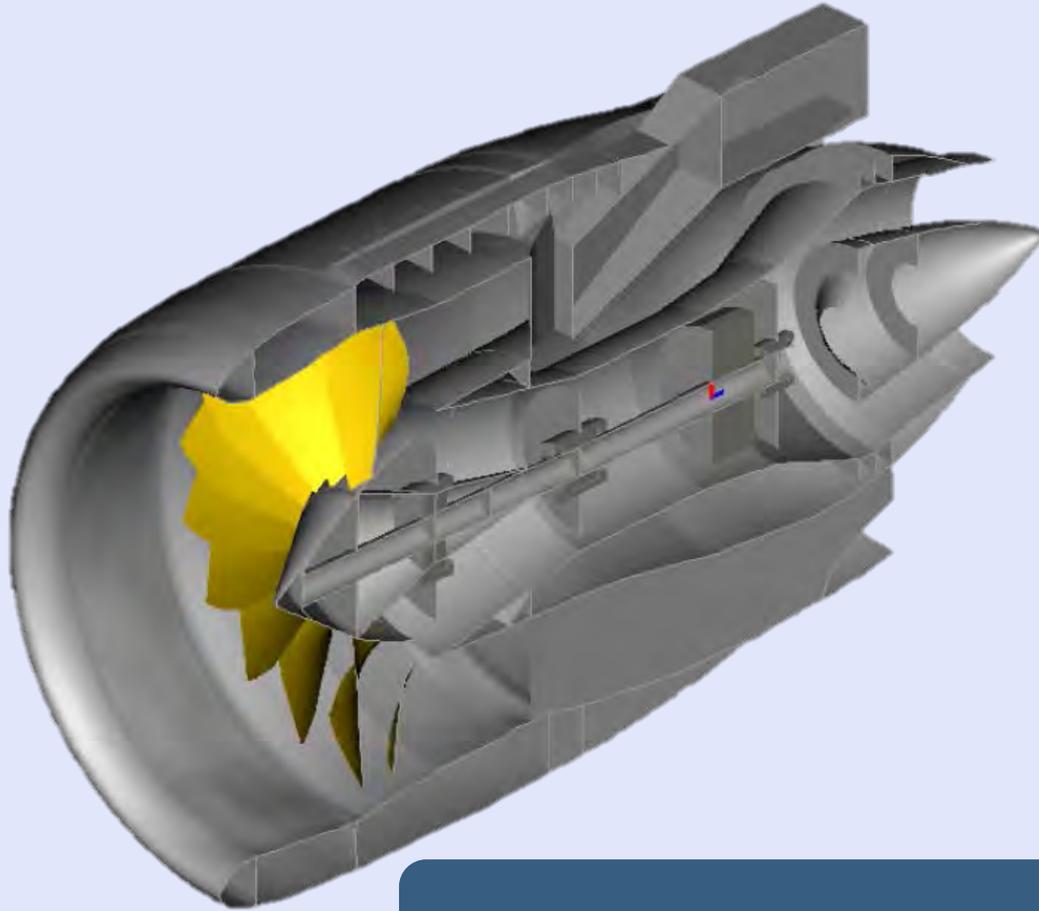


ANSA and META in AeroSpace Computational Mechanics

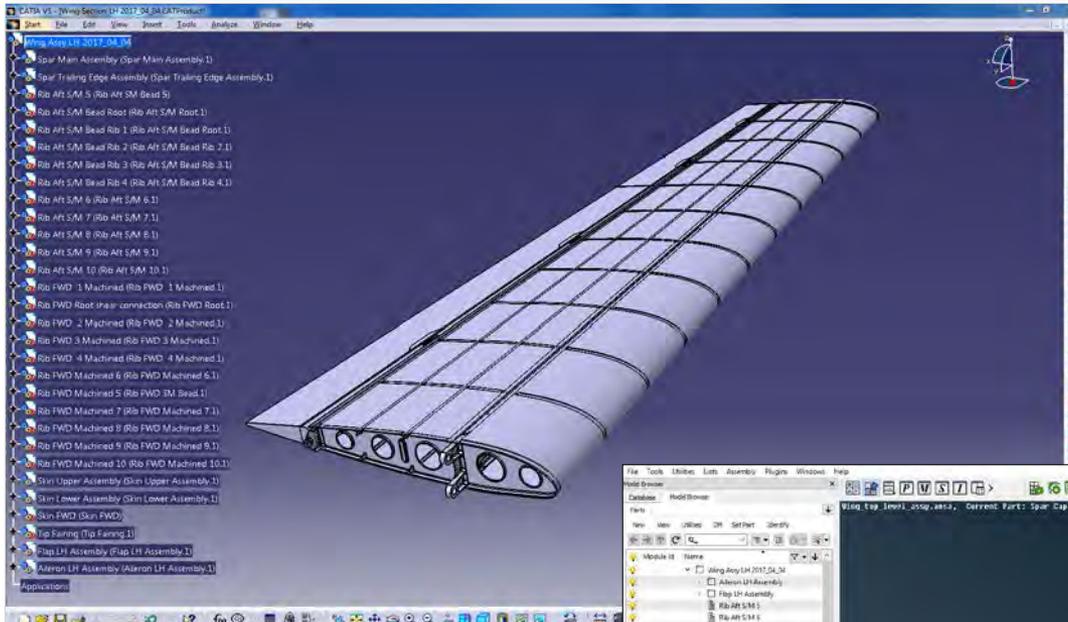




INTERFACES

Interfaces

ANSA input / output

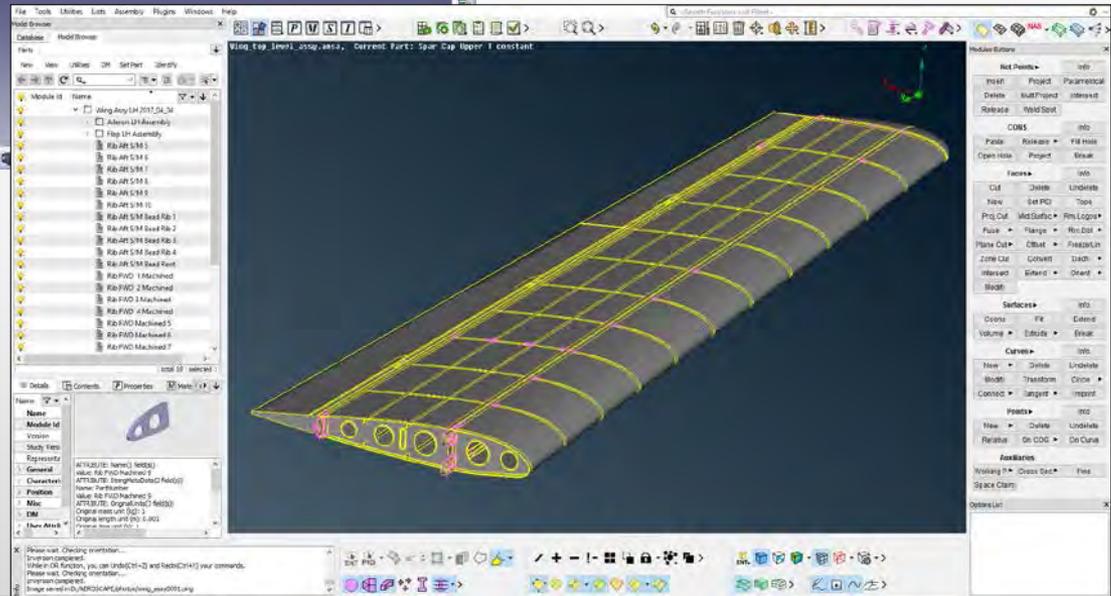


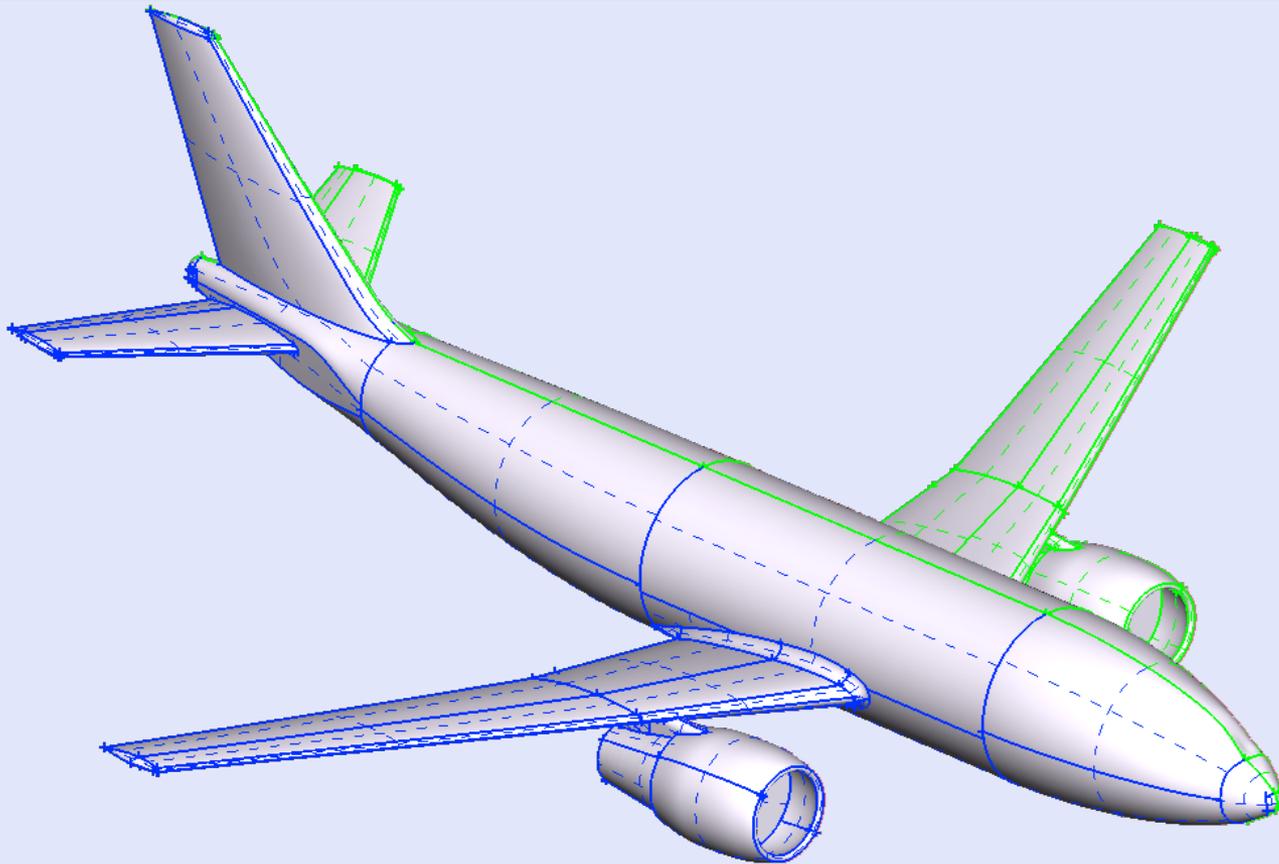
...to

- Ready-to-run solver input files
- CAD files
- Tessellated data
- META Product Tree

From...

- CAD & PDM data
- Keyword files
- Tessellated data
- Other (e.g. Medina)

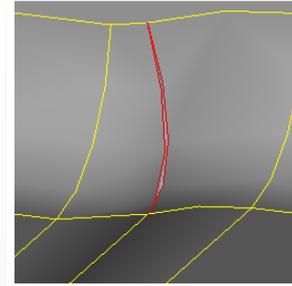
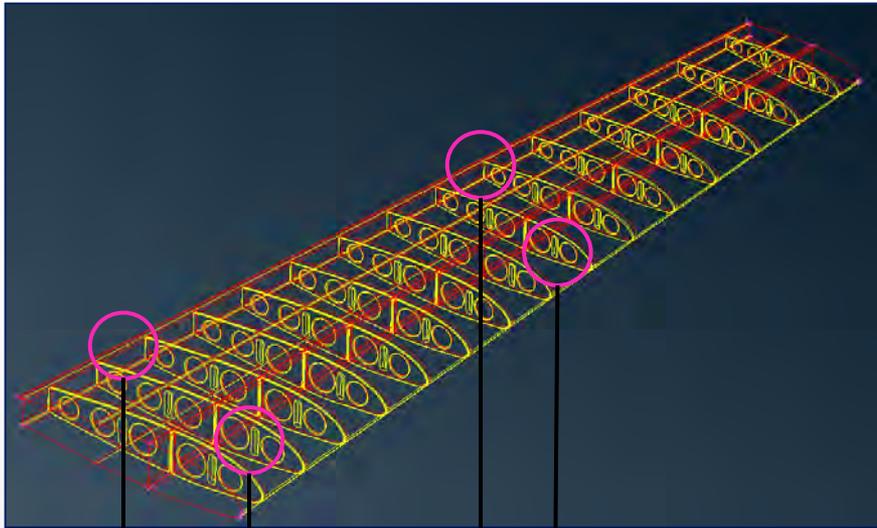




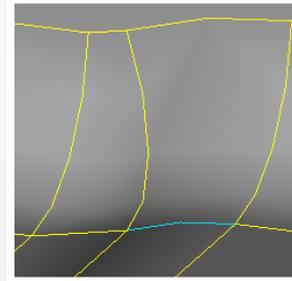
GEOMETRY

Geometry Checks

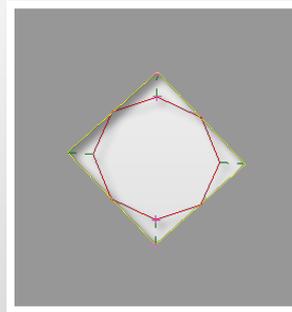
Topological errors identification and fix



Cracks



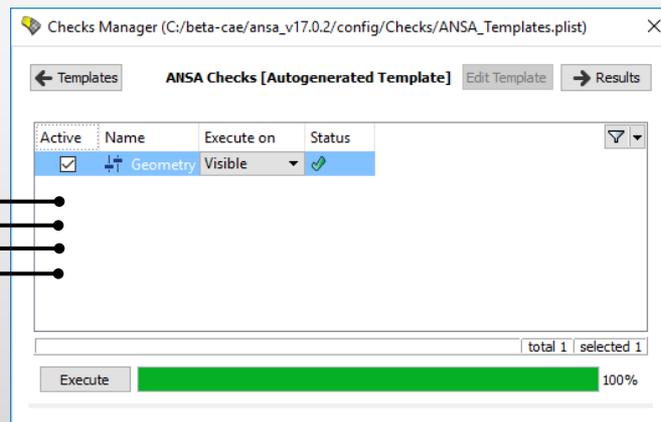
Needle Faces



Distorted Faces

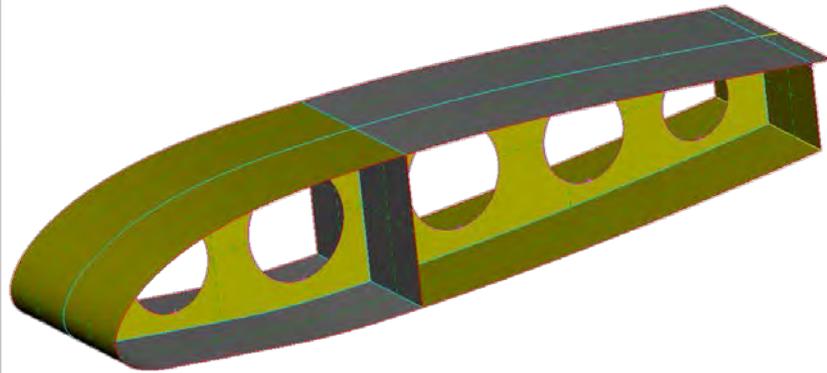
Geometry check for:

- Free Edges
- Cracks
- Overlap Faces
- Needle Faces
- Triple Edges
- Collapsed Edges
- Distorted Faces
- Unmeshed Areas



Geometry

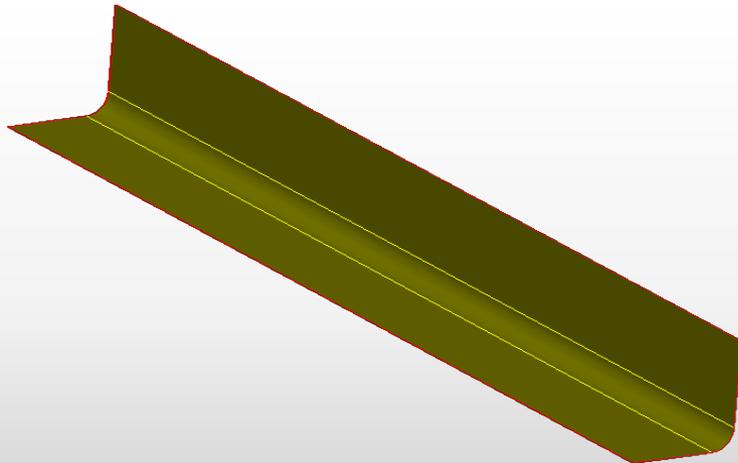
Geometry Creation



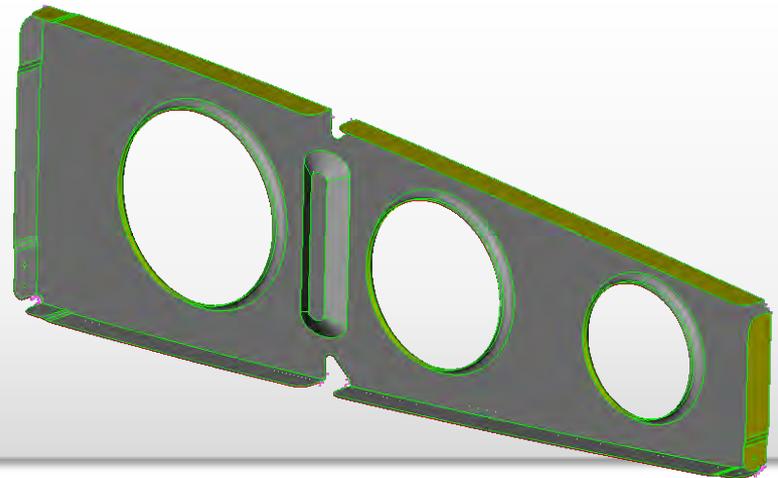
Handling Symmetry



Holes treatment

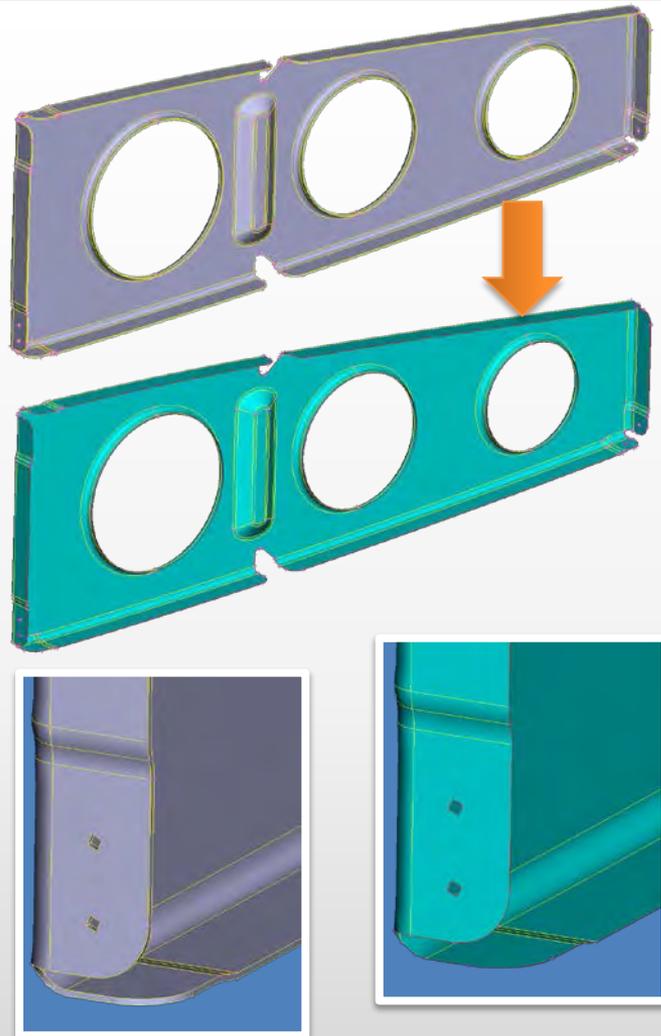


Fillets handling

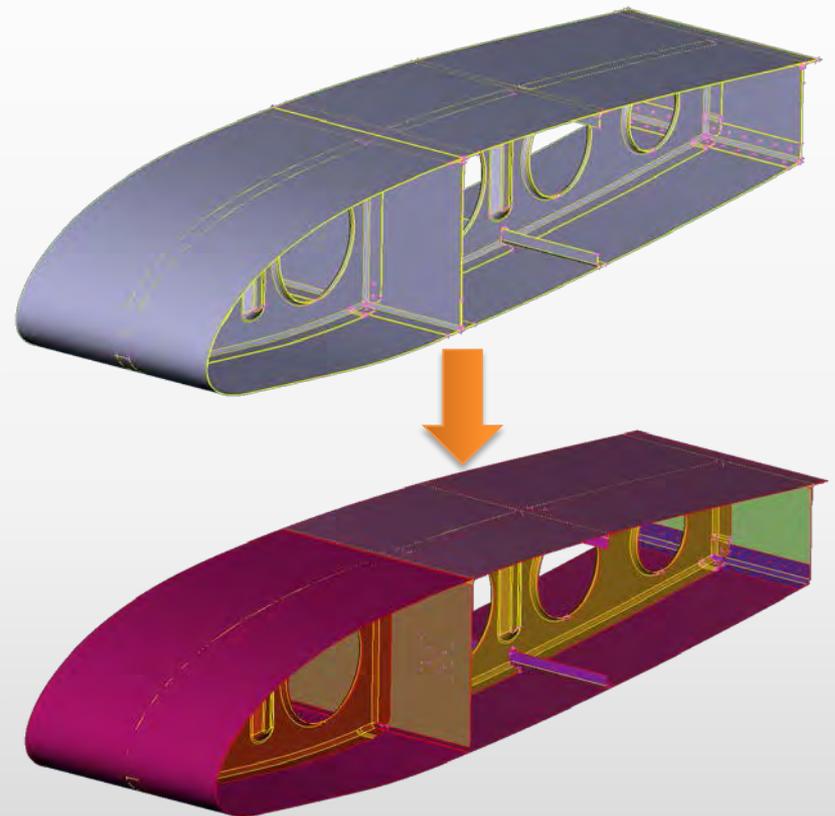


Geometry

Middle skin extraction – Sheet Metal Parts

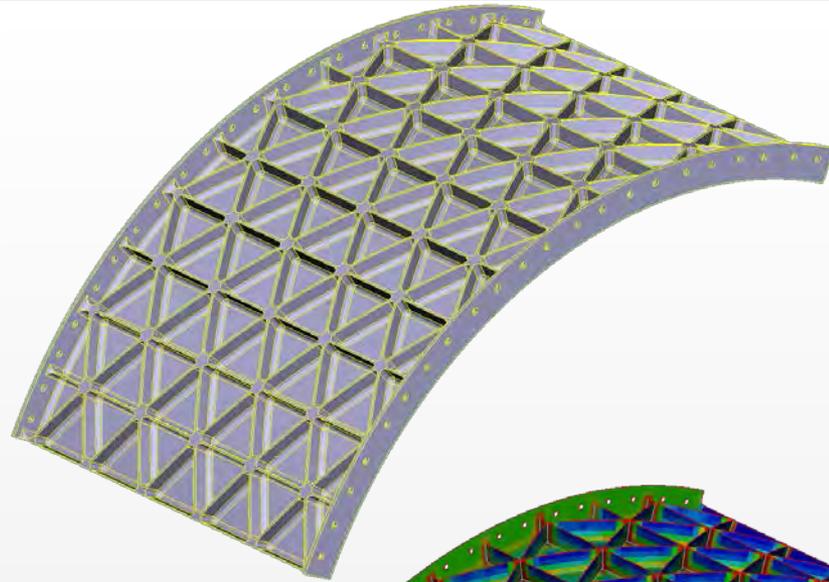


Middle skin extraction – Assemblies

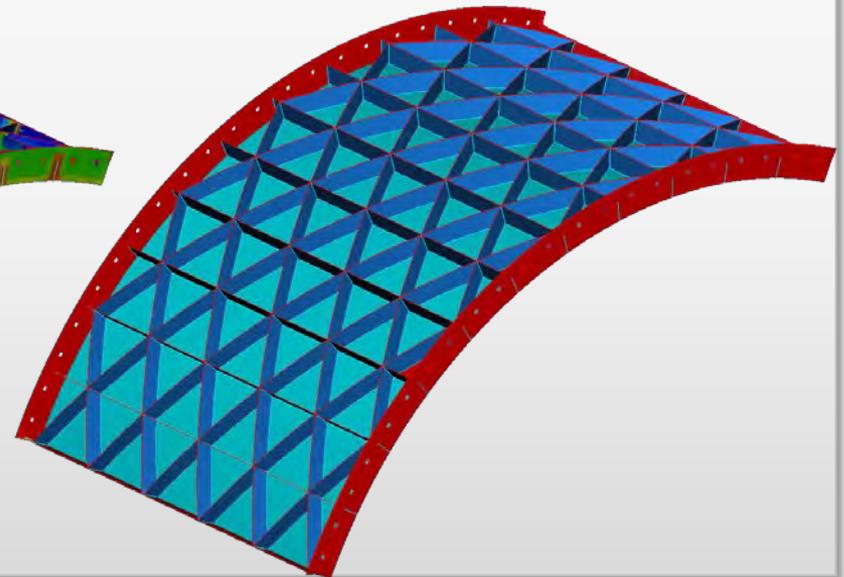
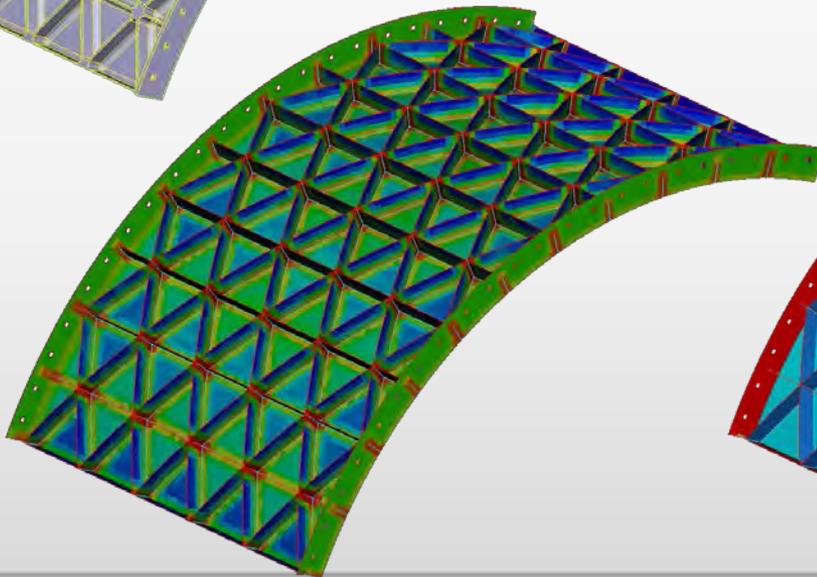


Geometry

Middle skin extraction – Casting Parts

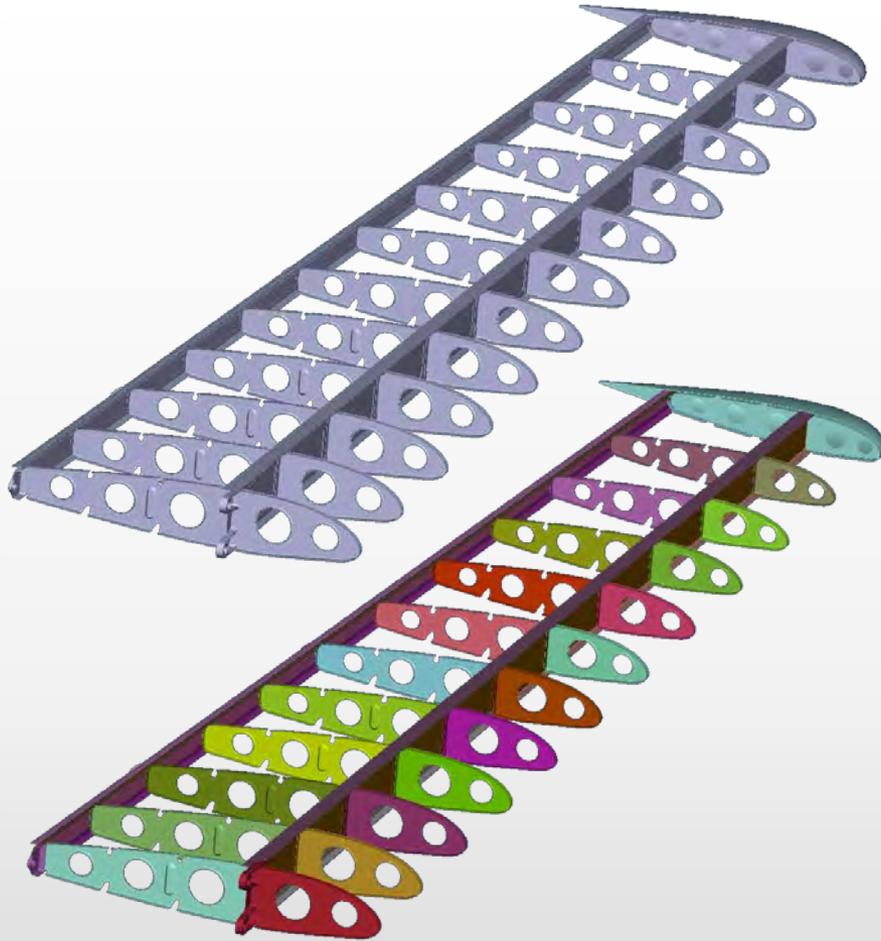


- Fully automated process
- Assign nodal thickness
- Identification of ribs - base
- Assign properties according to element thickness



Geometry

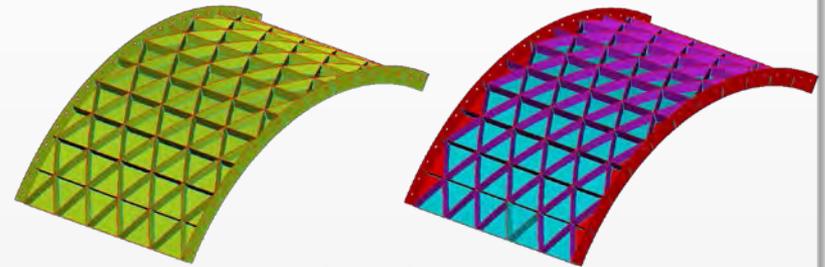
Groups Isolation



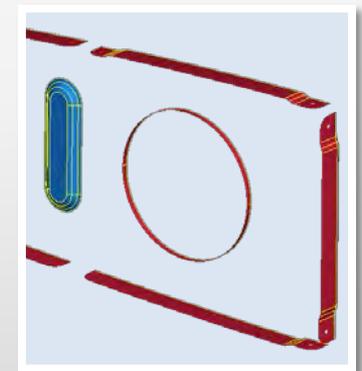
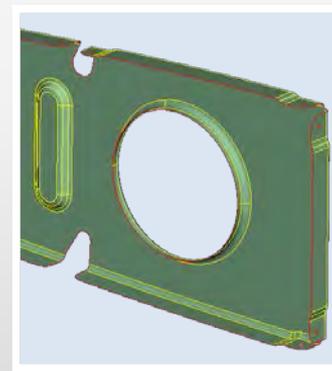
Isolate Connectivity Groups

Isolate

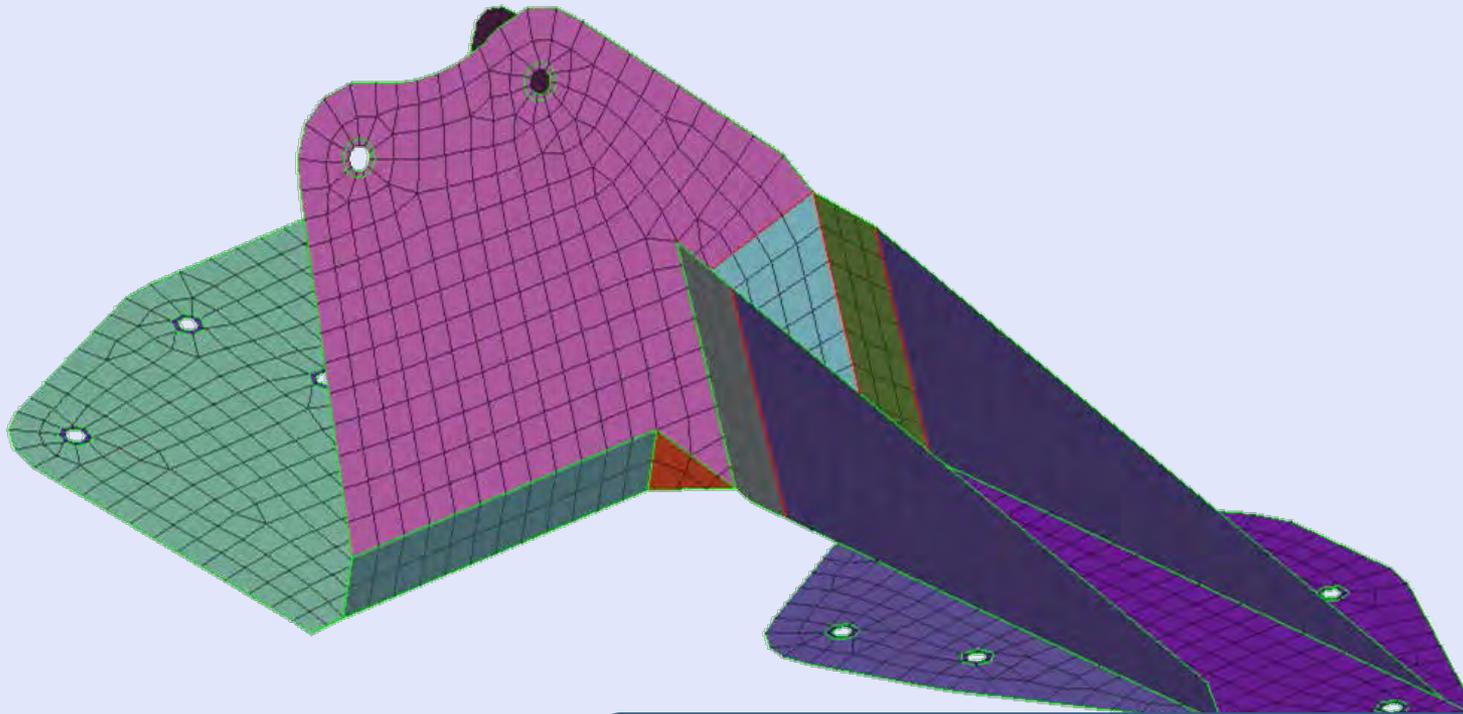
Flanges, Ribs, Radius, Washers, Skin, Exterior, Logos, etc...



Isolate Ribs



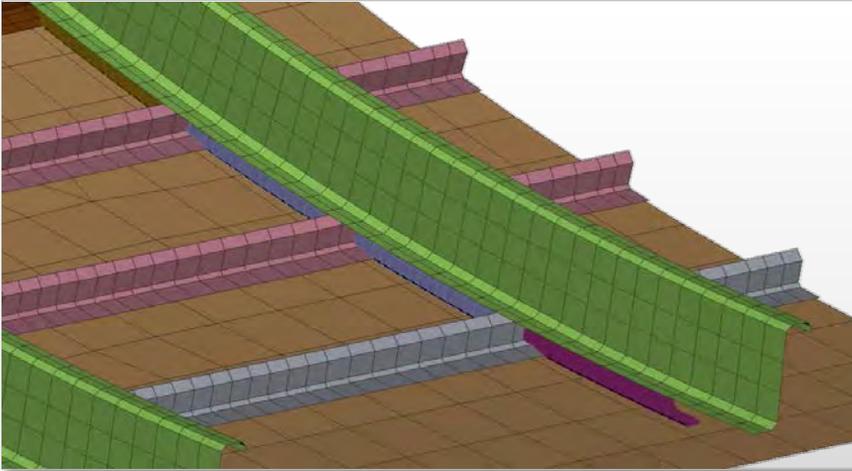
Isolate Flanges and Logos



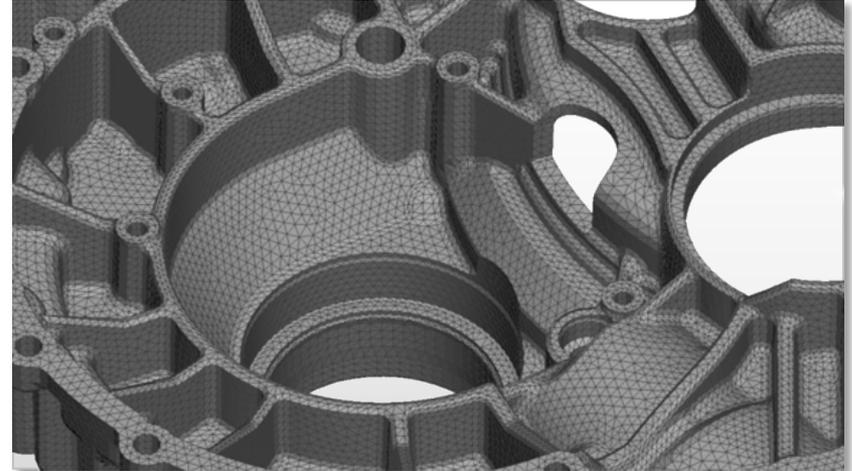
SHELL MESH

Shell Meshing Types

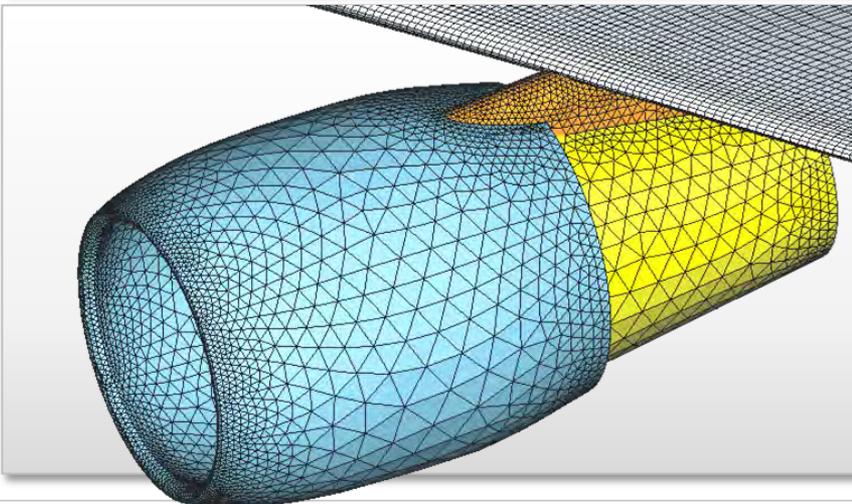
Surface Meshing



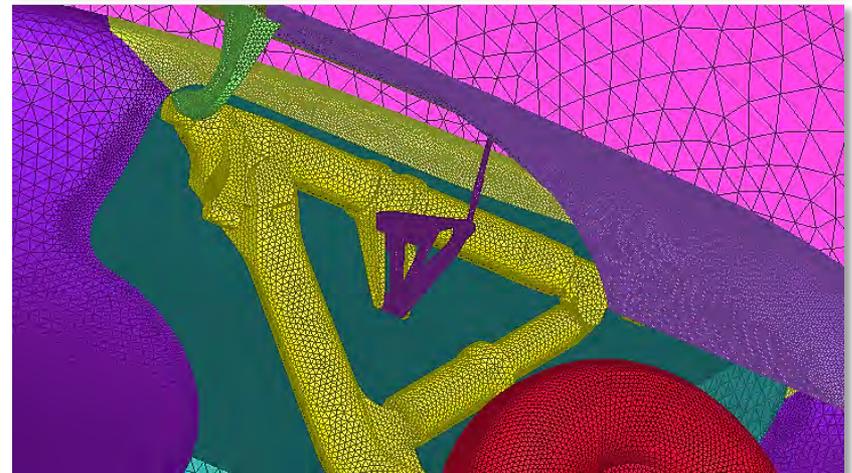
Structural Mesh for Solid components



CFD Meshing

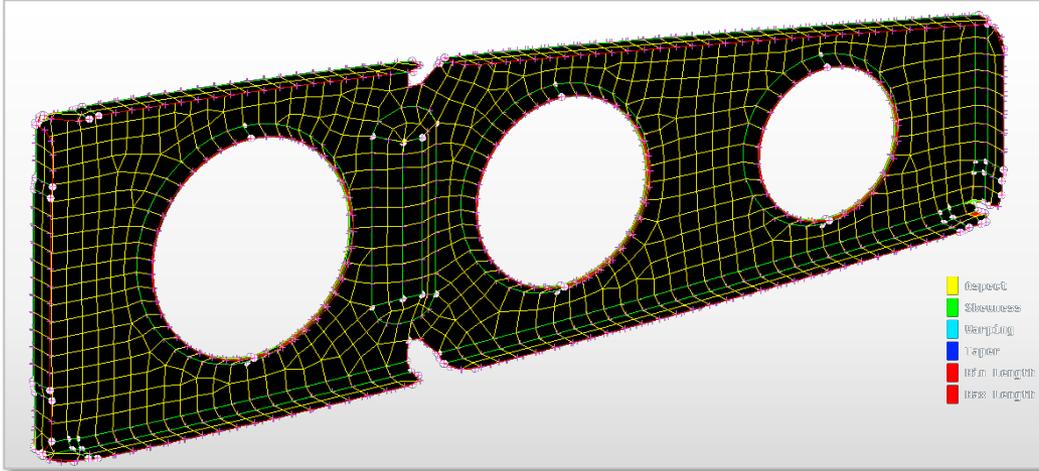


Wrapping

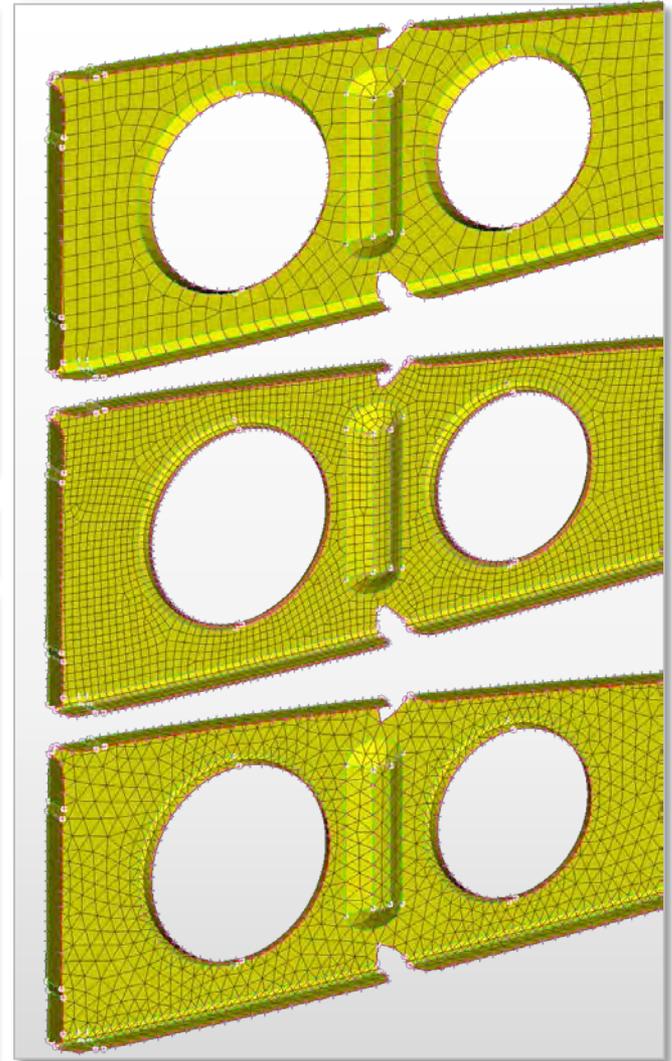


Shell Meshing

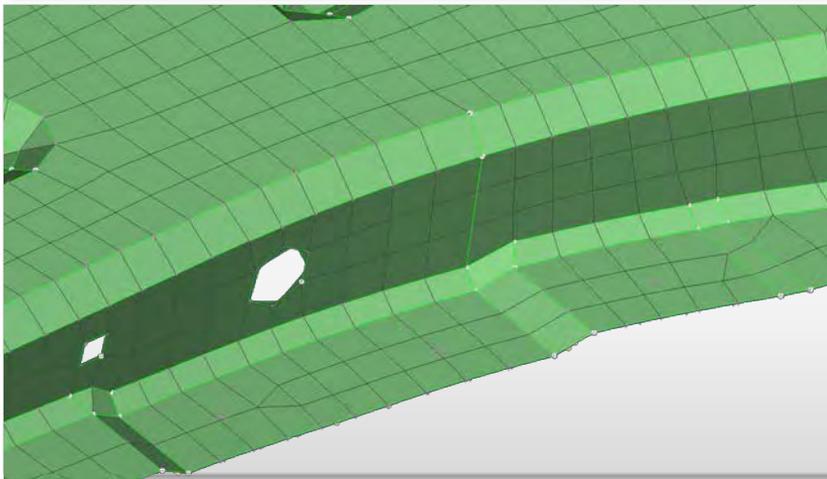
Quality Improvement



Reconstruction

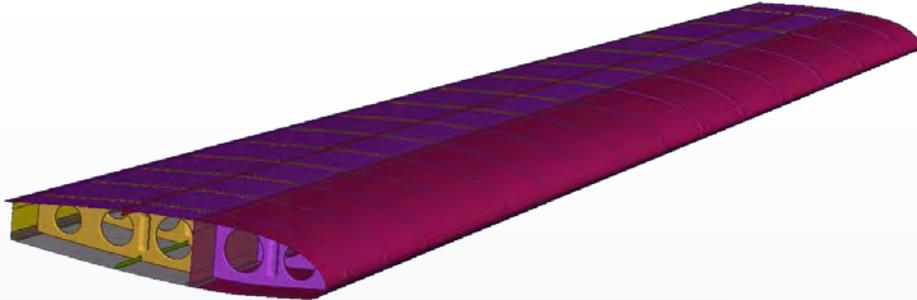


Meshing Features Simplify



Shell Meshing

Batch Meshing

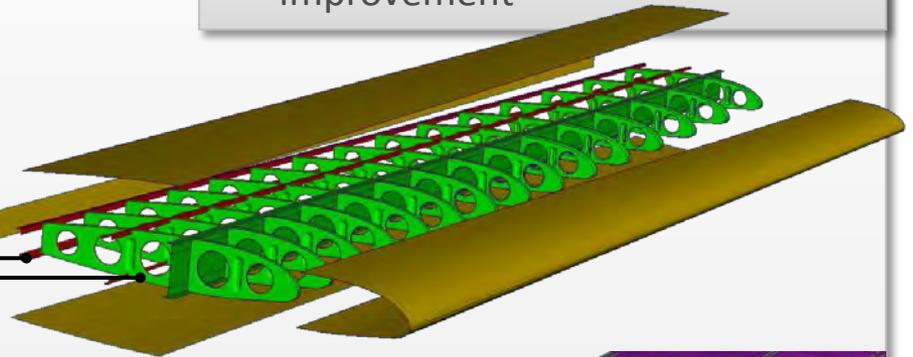


- Definition of meshing parameters and quality criteria
- Features treatment and model simplification
- Automatic meshing and quality improvement

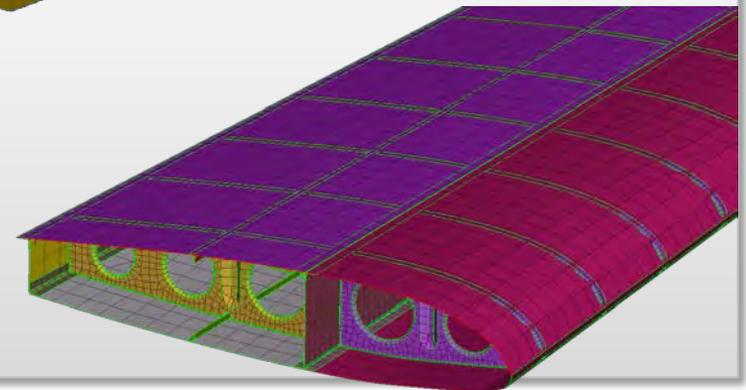
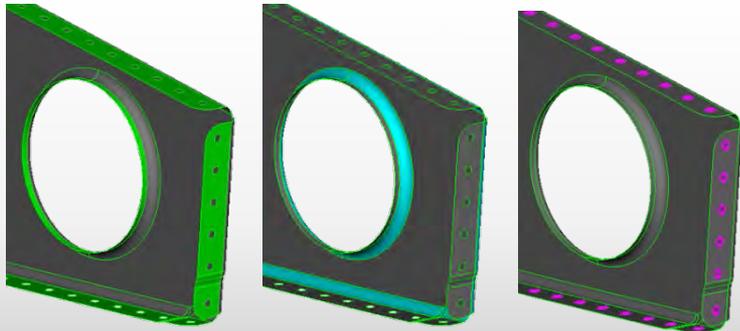
Batch Mesh Manager

New Read Scenario Autoload Run Update status

Name	Contents	Color	Mesh Parameters	Quality Criteria	Status
✓ Meshing_Scenario_1	17				Completed
✓ Session_3	4	Yellow	50mm	Untitled	Completed
✓ Session_2	4	Red	15mm	Untitled	Completed
✓ Default_Session	9	Green	15mm	Untitled	Completed

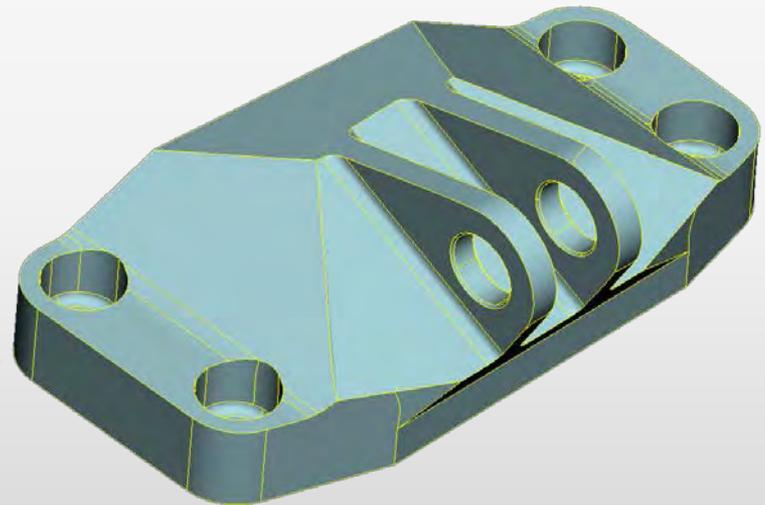
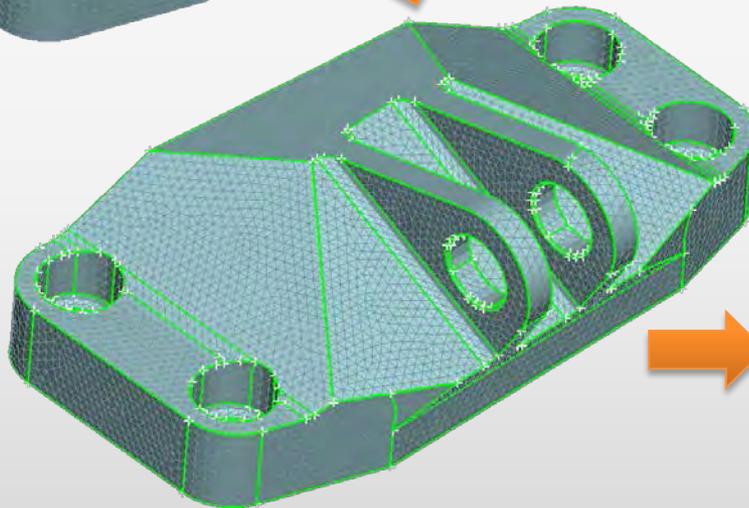
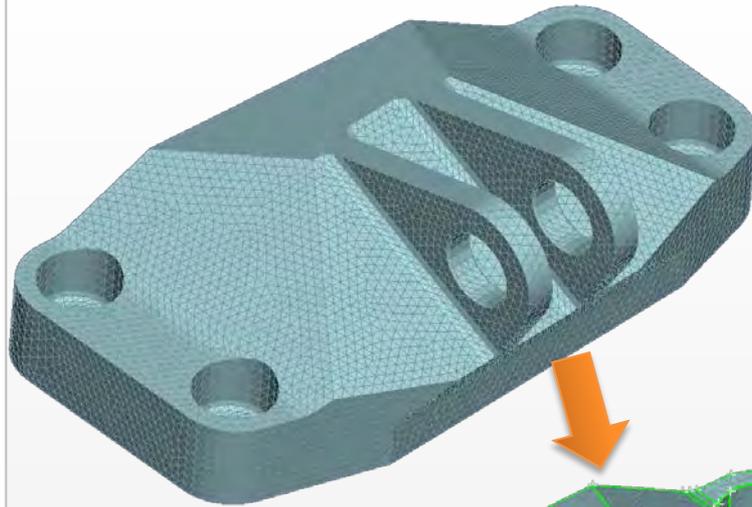


Feature treatment



Shell Meshing

FE to Geometry



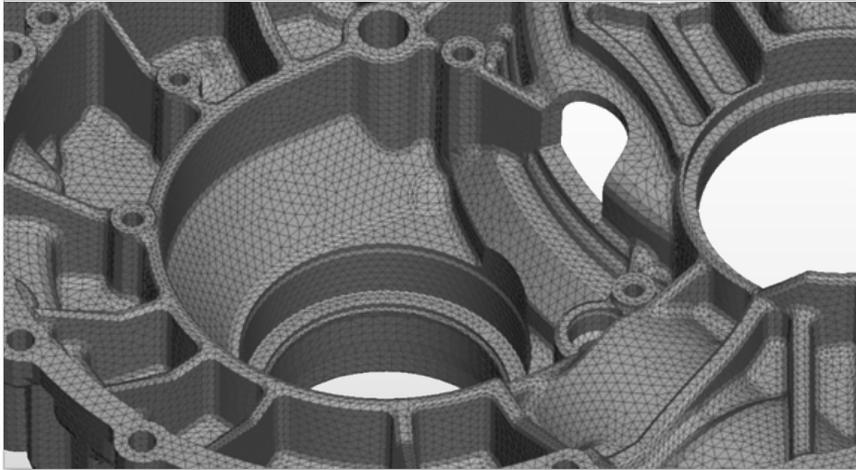
- Automatic definition of geometry from the FE Model
- Faces creation in respect of the model features



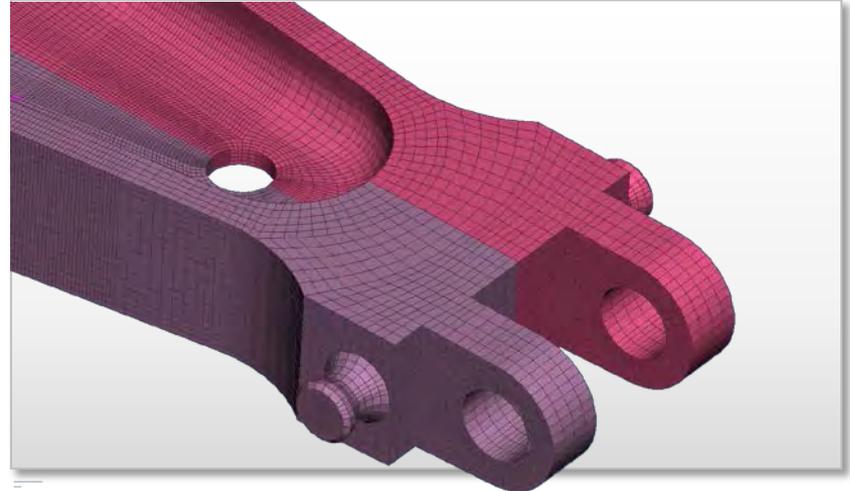
VOLUME MESH

Volume Meshing

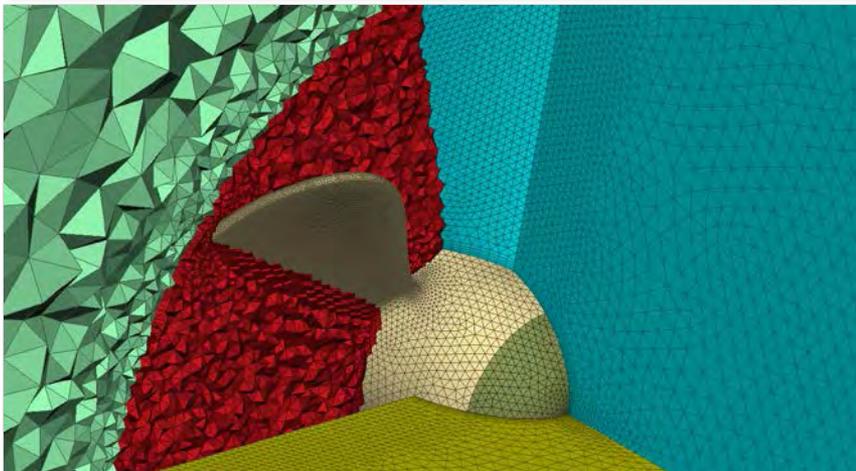
Unstructured Mesh: Tetra



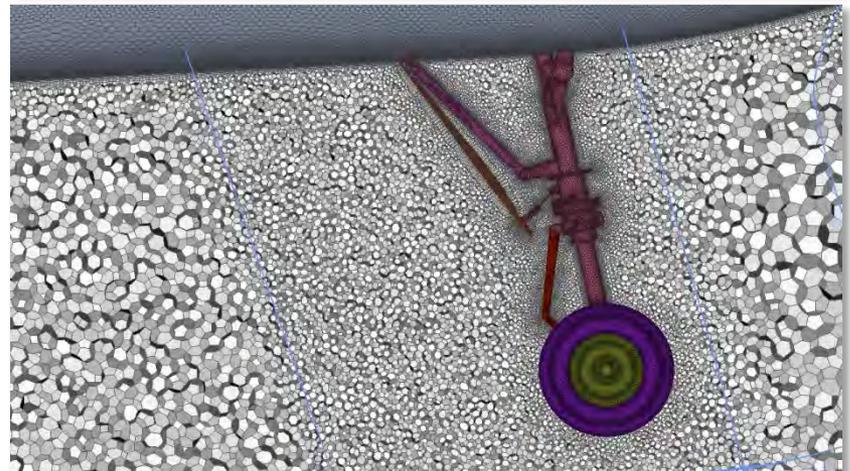
Structured Mesh: Hexa



CFD Mesh

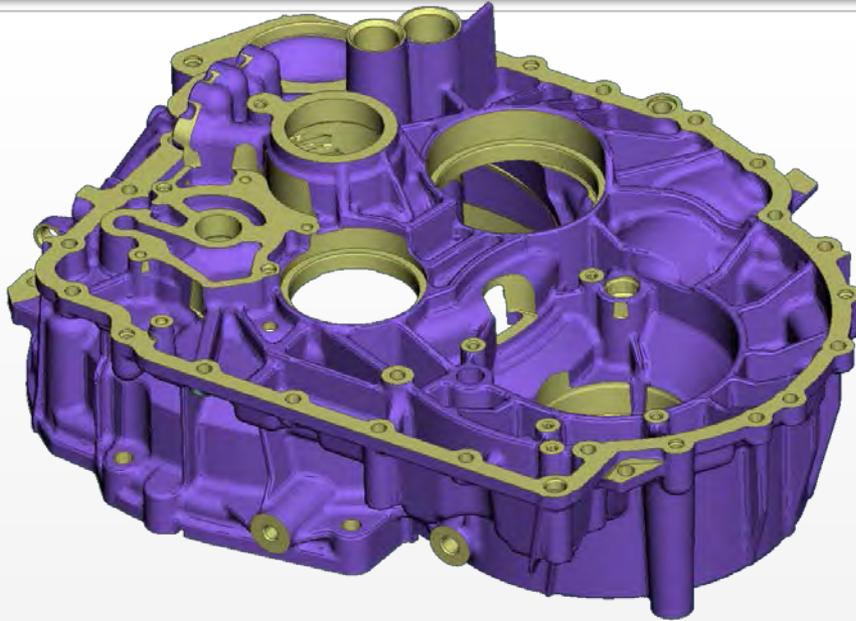


Polyhedral Mesh

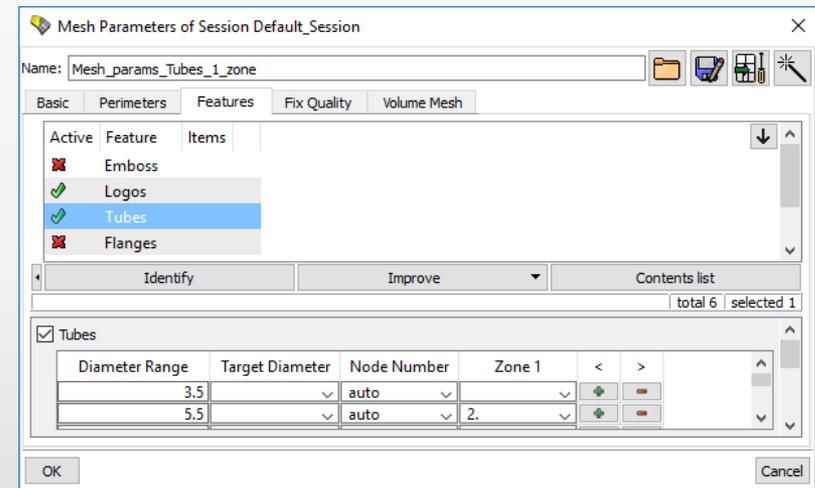


Volume Meshing

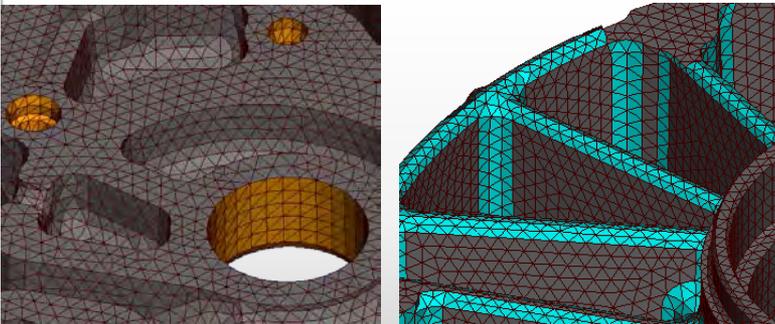
Batch Meshing



- Automatic defeaturing
- Mesh quality control
- Feature treatment
 - Tubes
 - Fillets
 - Sharp Edges

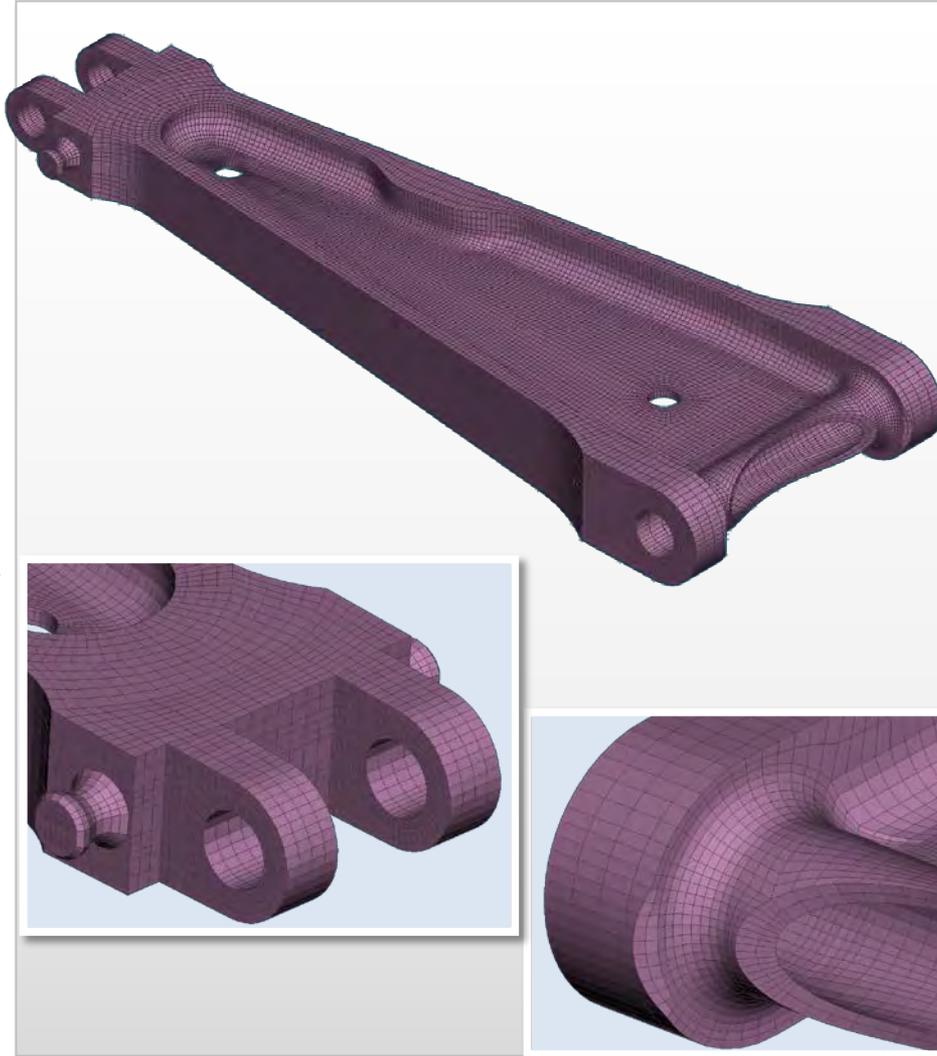


Feature treatment

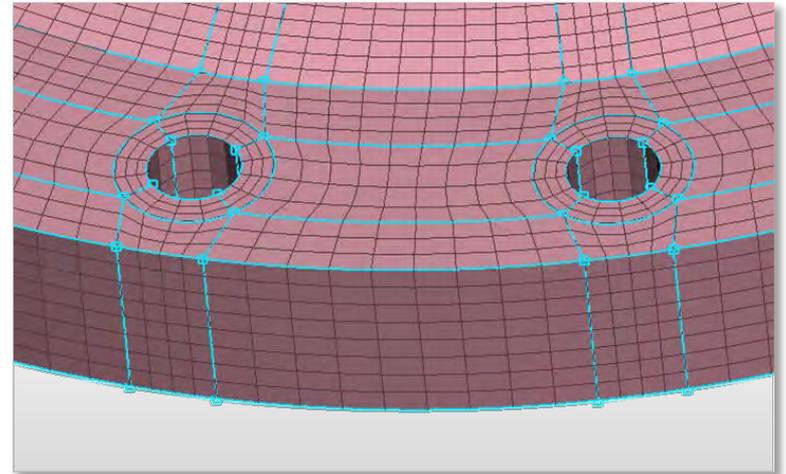


Hexa Block Tool

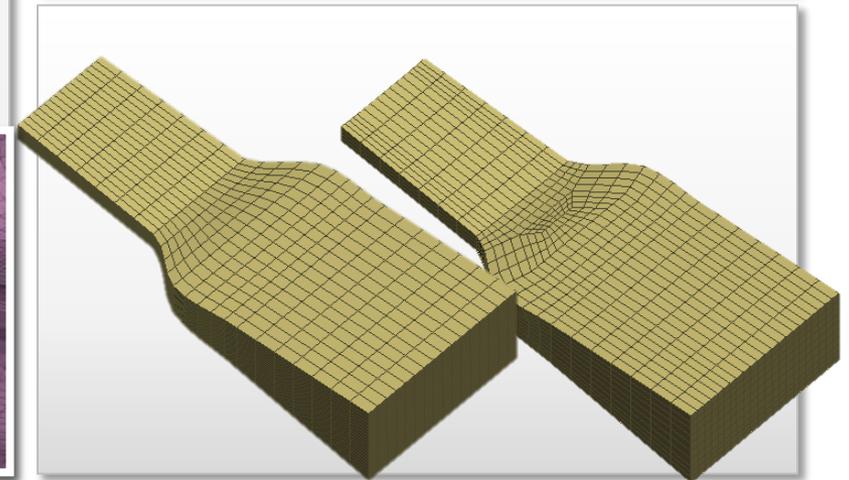
Structural models

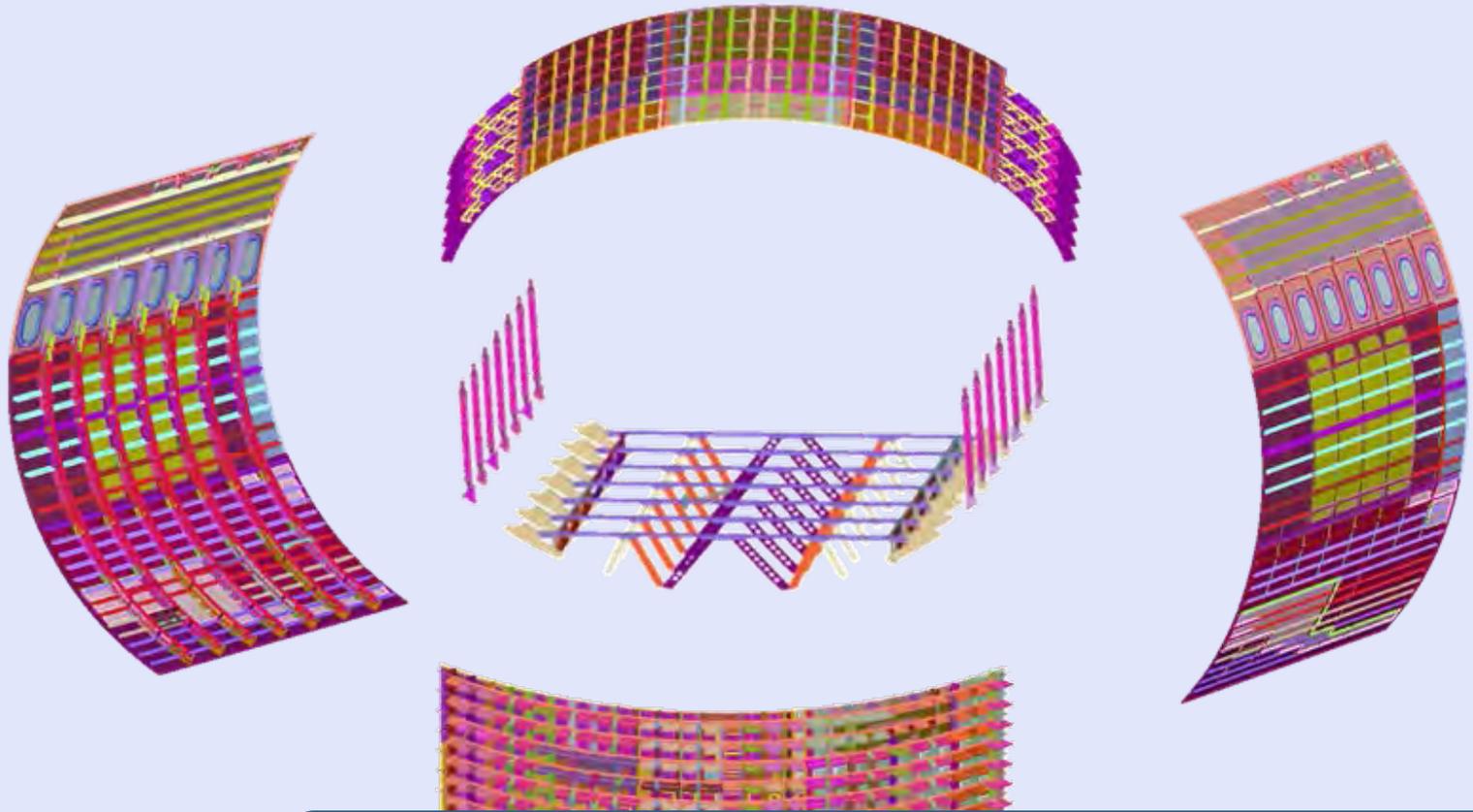


Element zones generation



C/L type structure generation

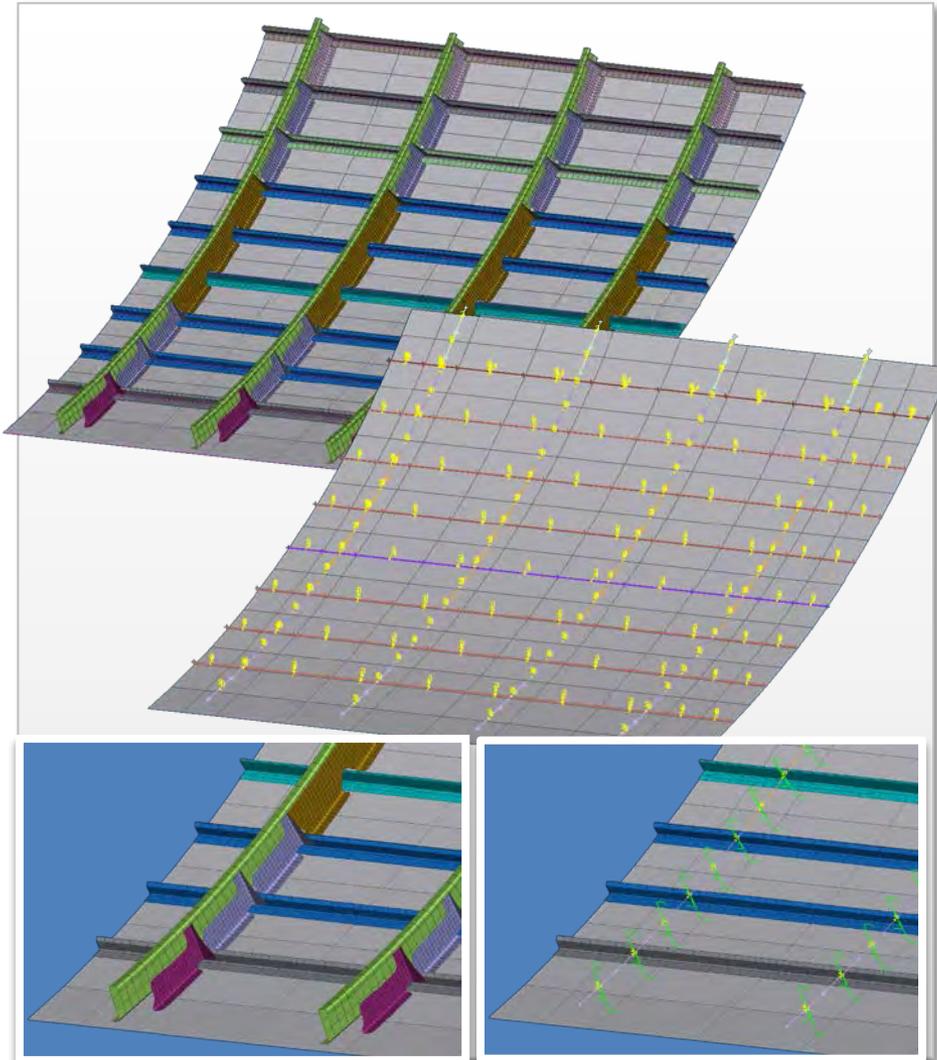




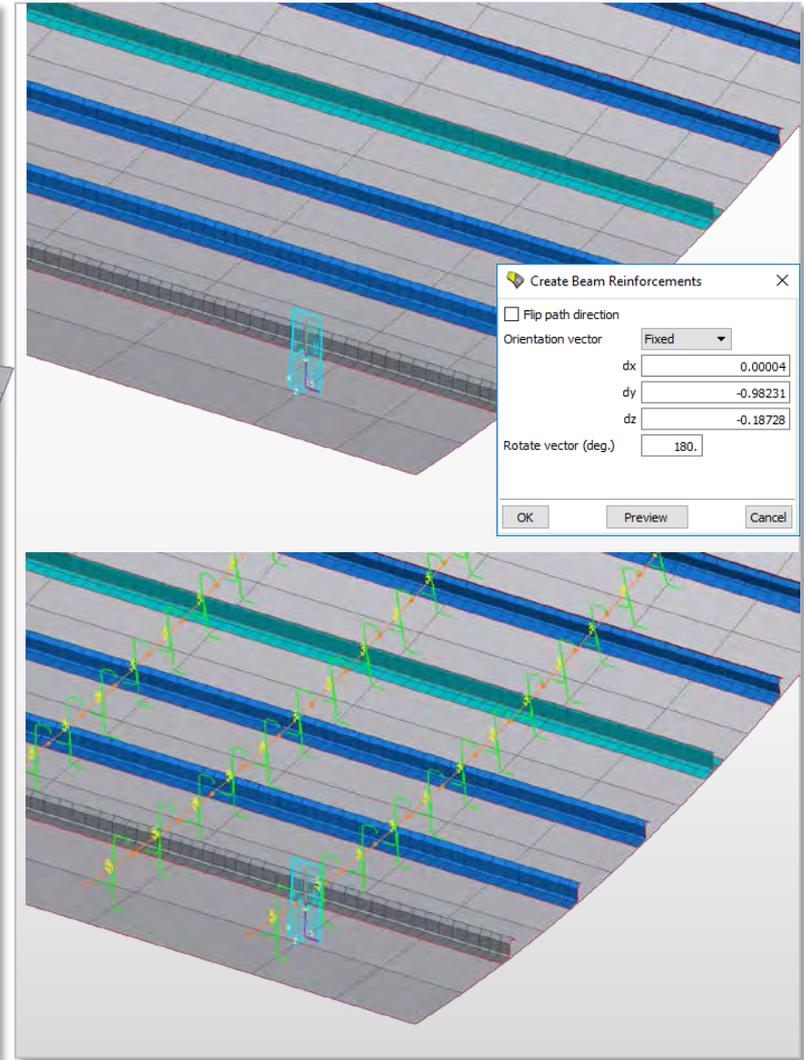
ANALYSIS TOOLS

Analysis Tools

Shell to Beams Stiffeners replacement

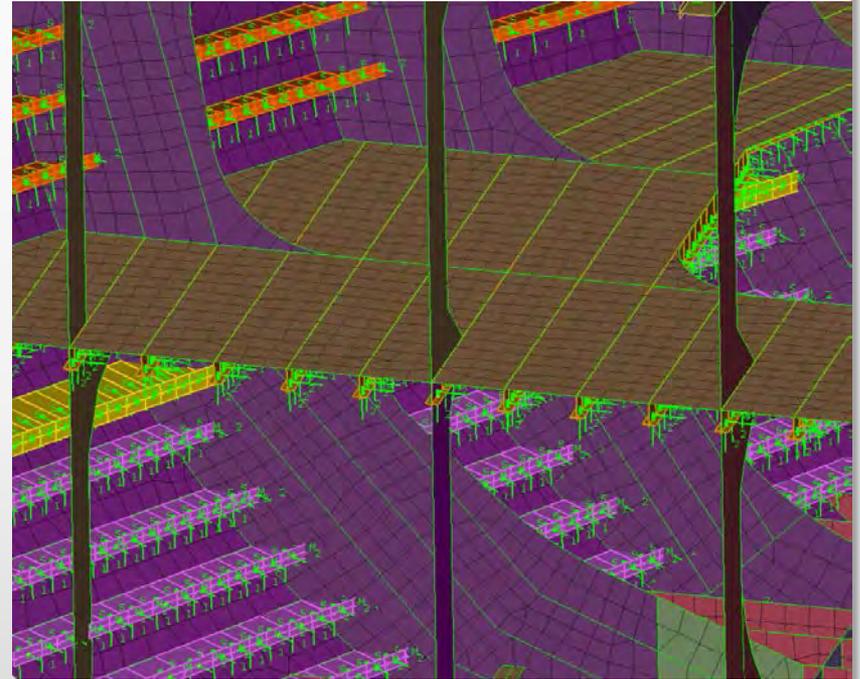
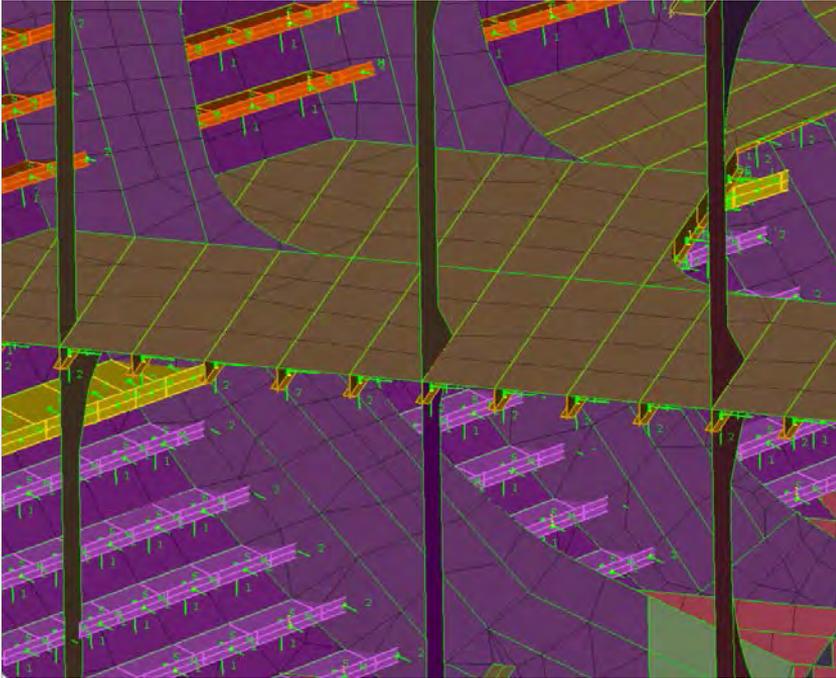


Beam Stiffeners definition



Analysis Tools

Reconstruction of shell – beam panels



Analysis Tools

Directional re-numbering

Directional Renumbering

Entities

Type: Shells

Entities IDs: 1, 32, 33, 34, 35, 36

General Info

Coordinate System: 1

Initial ID: 100000

Primary Axis

Tolerance: 0.5

Step: 1

Axis: X Y Z

Order: Ascending Descending

Secondary Axis

Tolerance: 0.5

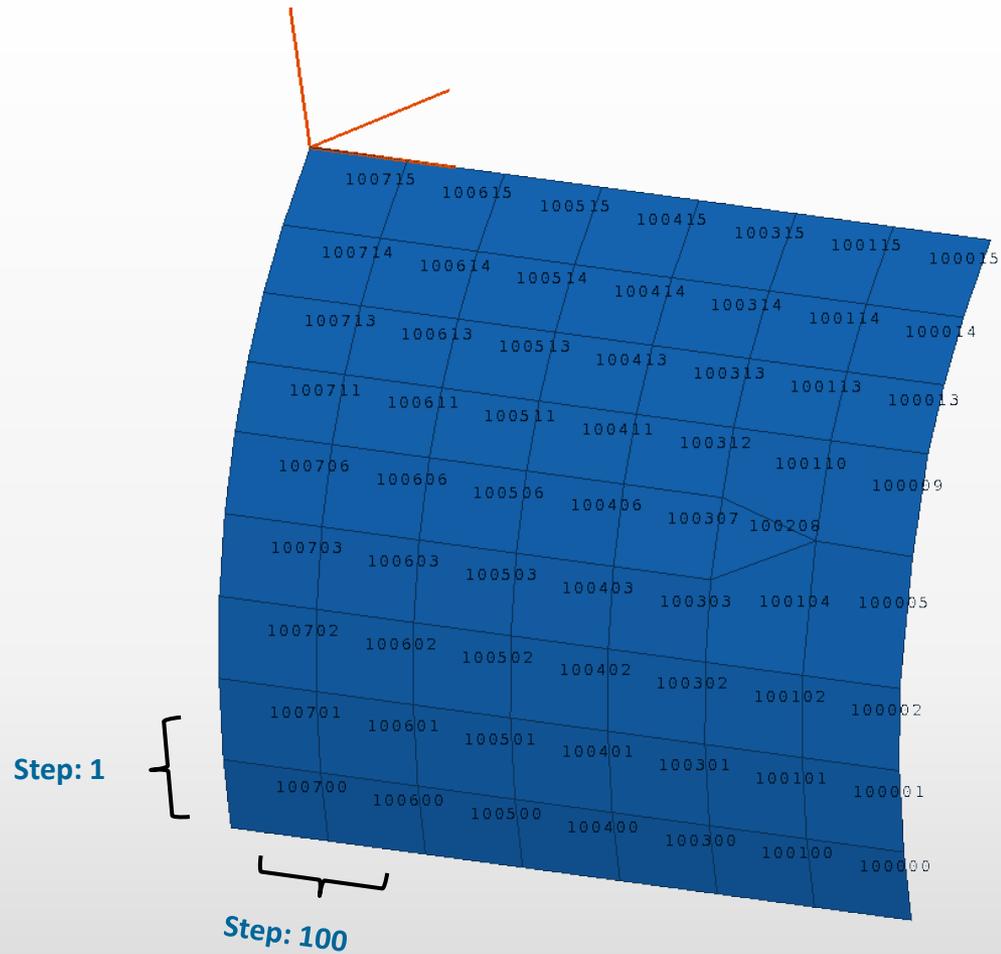
Step: 100

Axis: X Y Z

Order: Ascending Descending

Freeze IDs

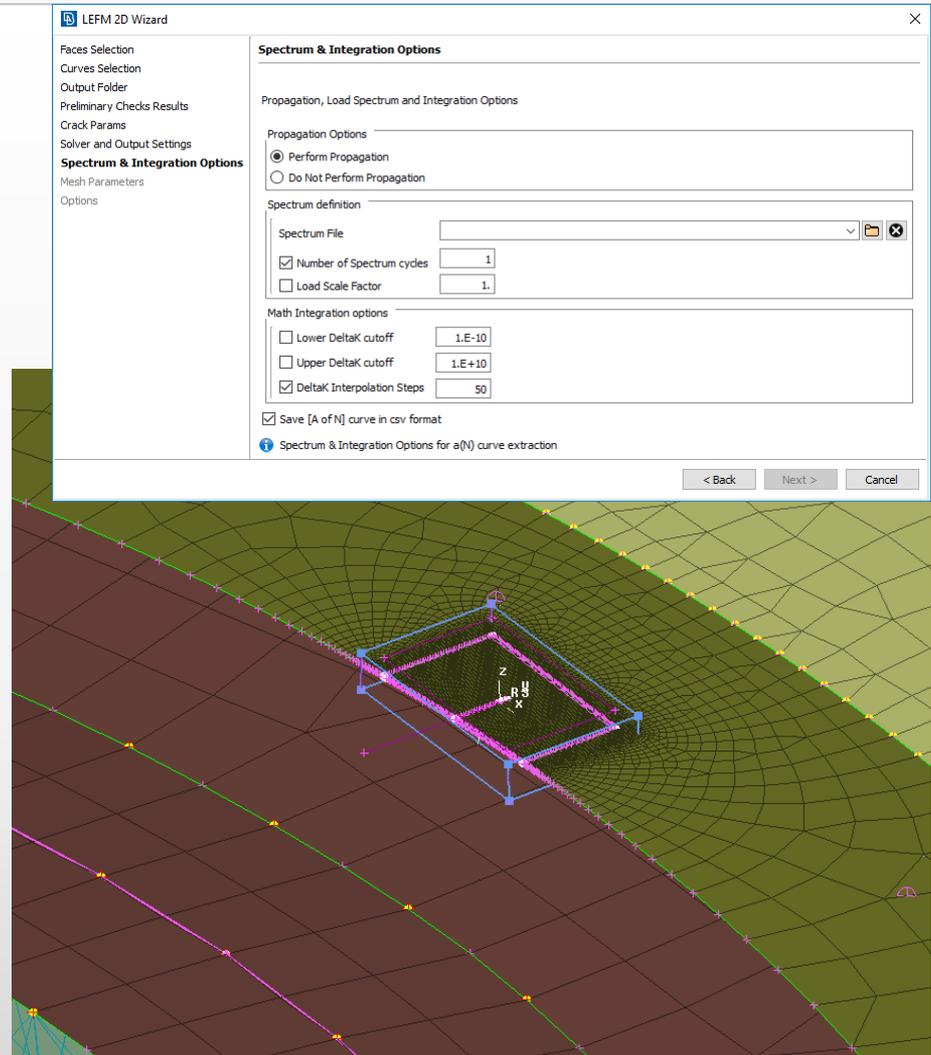
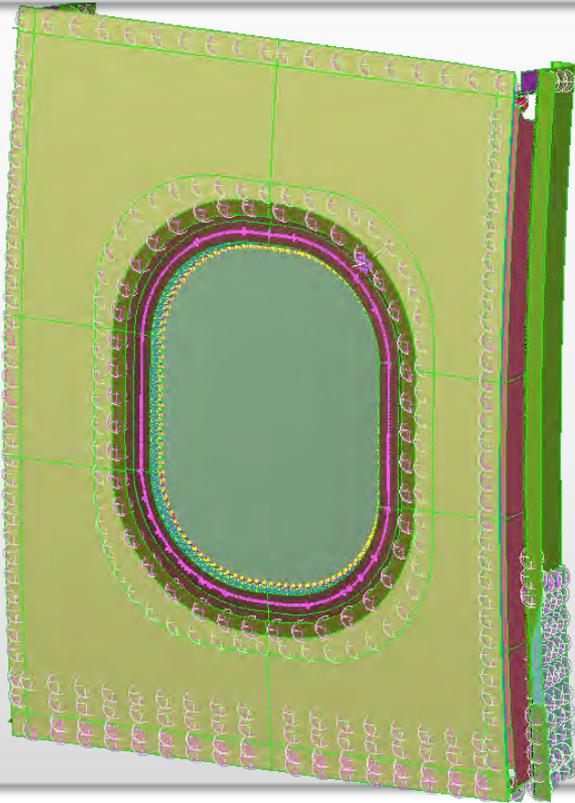
OK Cancel



Analysis Tools

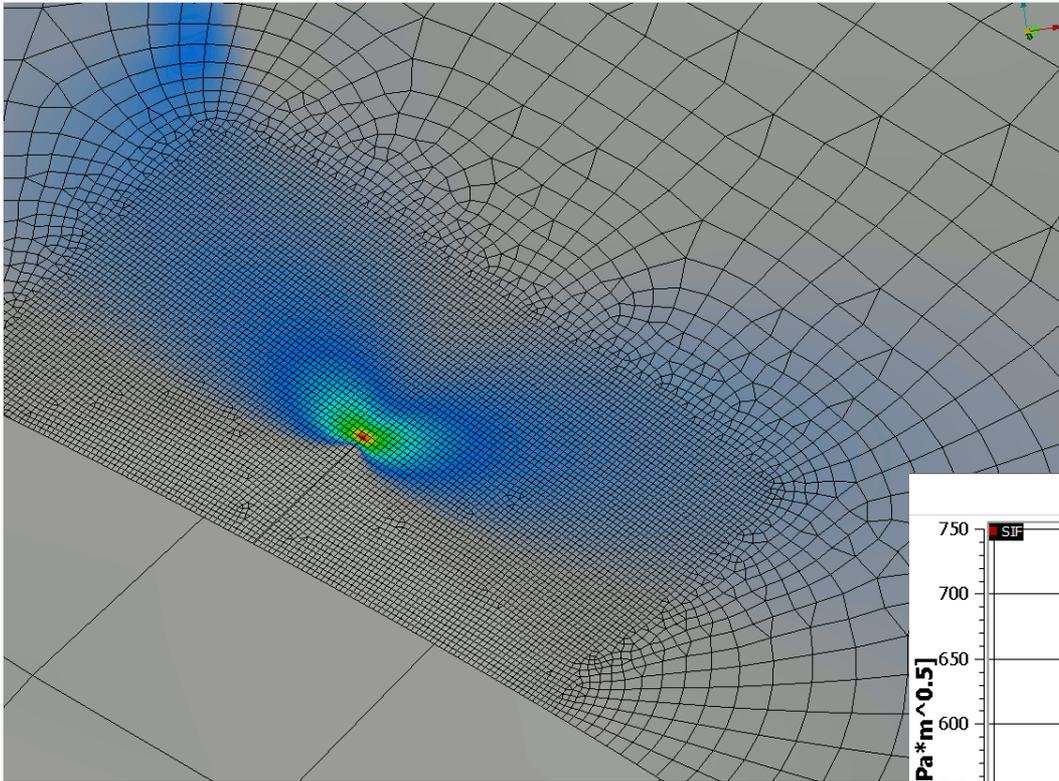
Crack Propagation Tool

- Easy set up through wizard
- EPILISYS / NASTRAN solver selection
- Read in cyclic loads (spectrum file)
- Works on multiple cracks



Analysis Tools

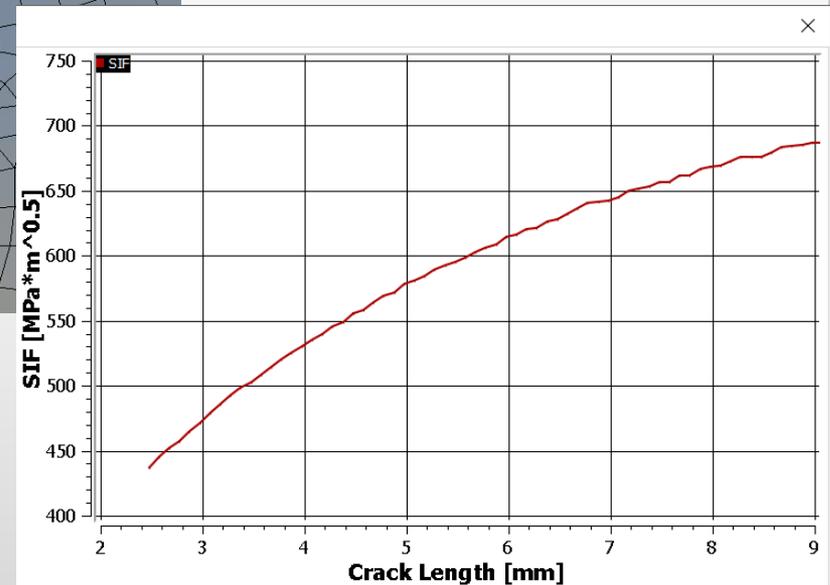
Crack Propagation Tool



- Automatic re-meshing around crack tip area
- Determines crack growth direction
- Paris Law integration
- Stress Intensity Factor Plot

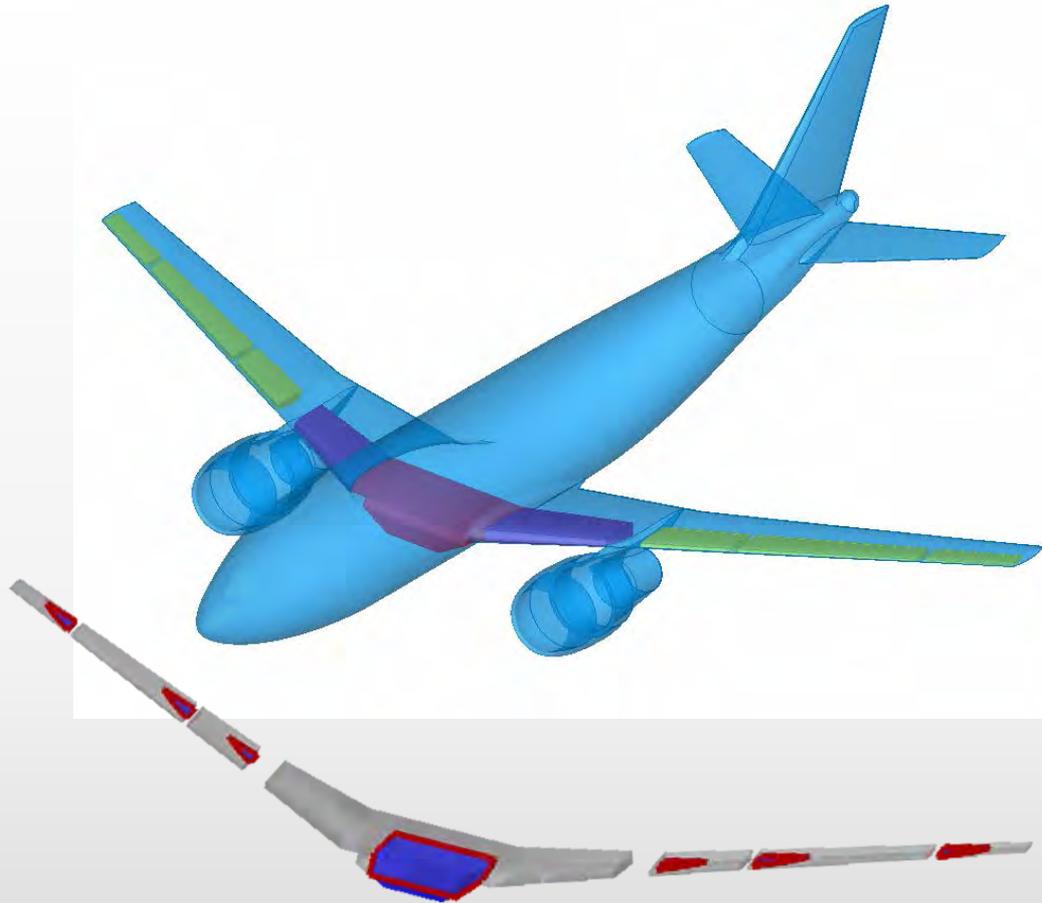
Termination conditions

- Max crack length reached
- Inactive crack status
- Unstable crack
- Spectrum completely applied

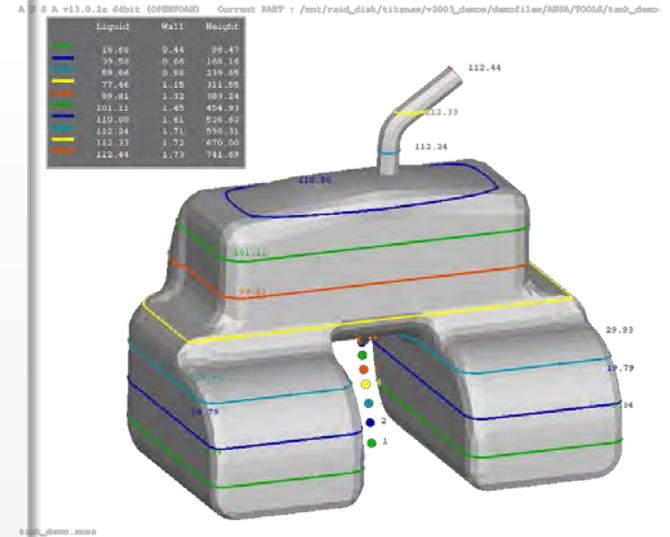


Analysis Tools

Tank Tool



Unused liquid traps



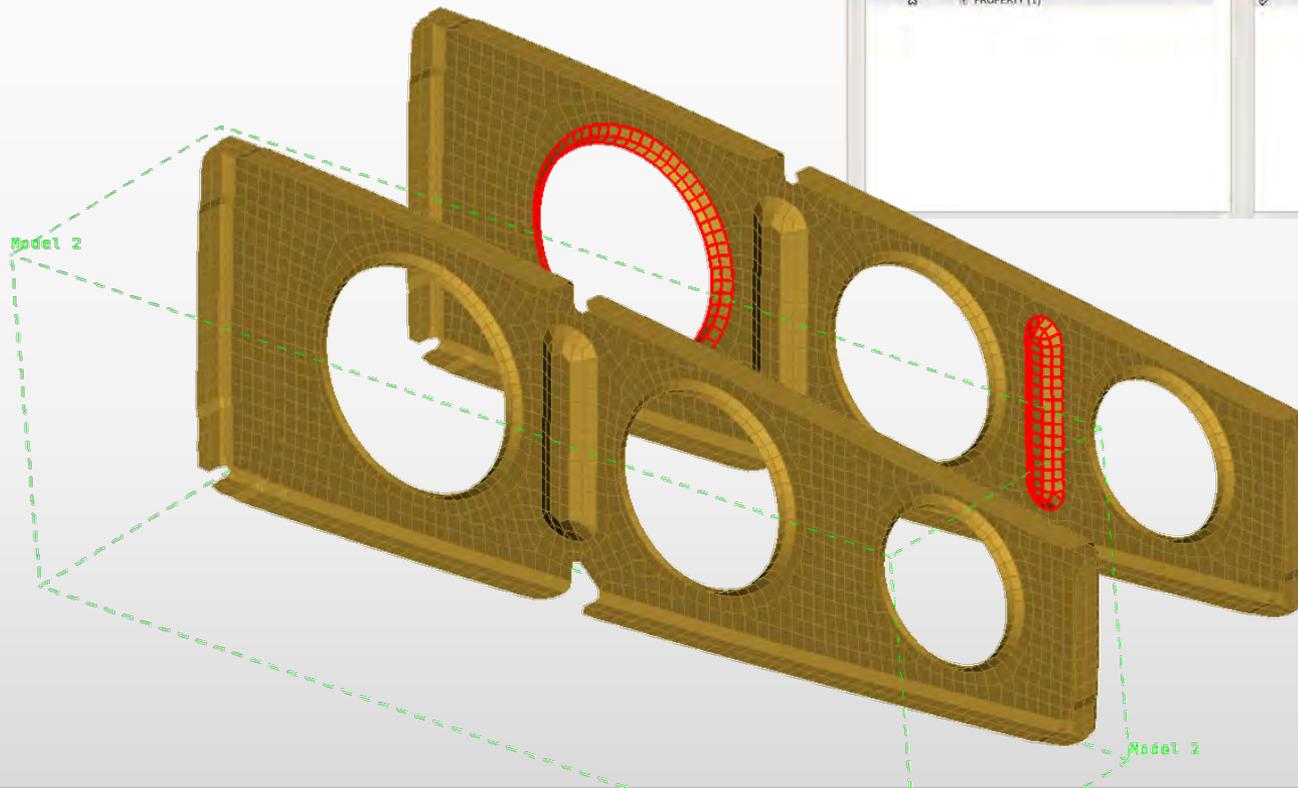
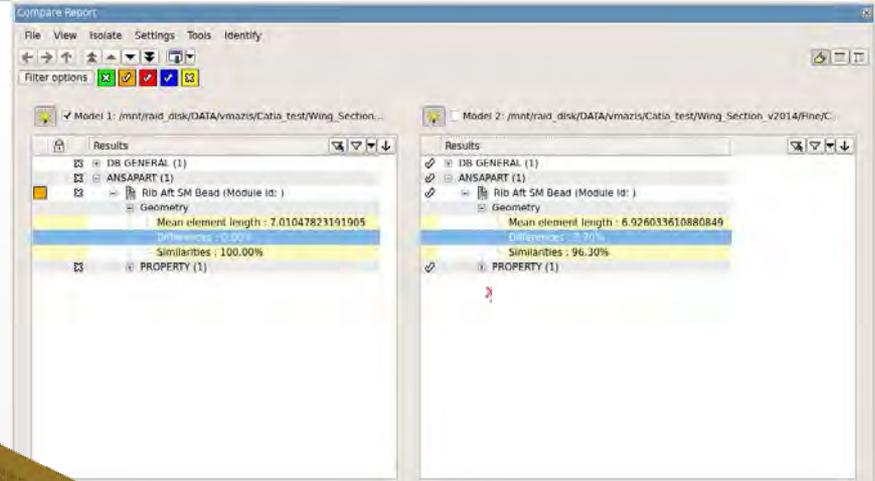
Calculation of:

- liquid levels
- liquid volume
- unused trapped liquid
- center of gravity (COG) for various tank configurations and positions

Analysis Tools

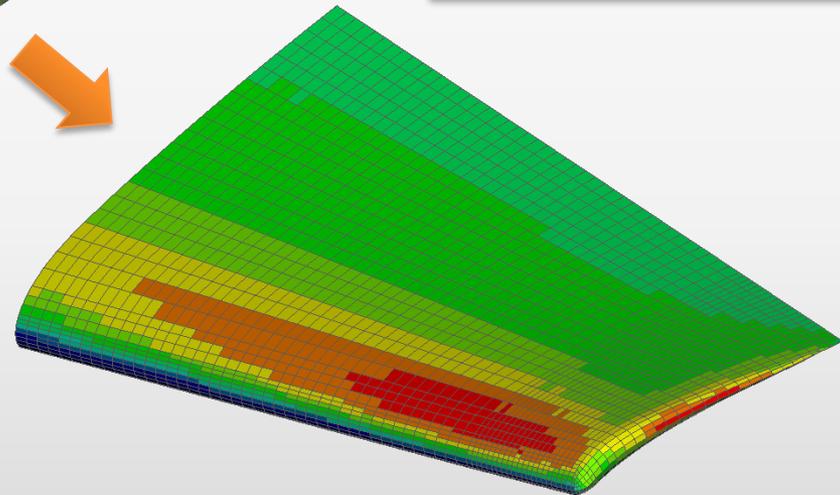
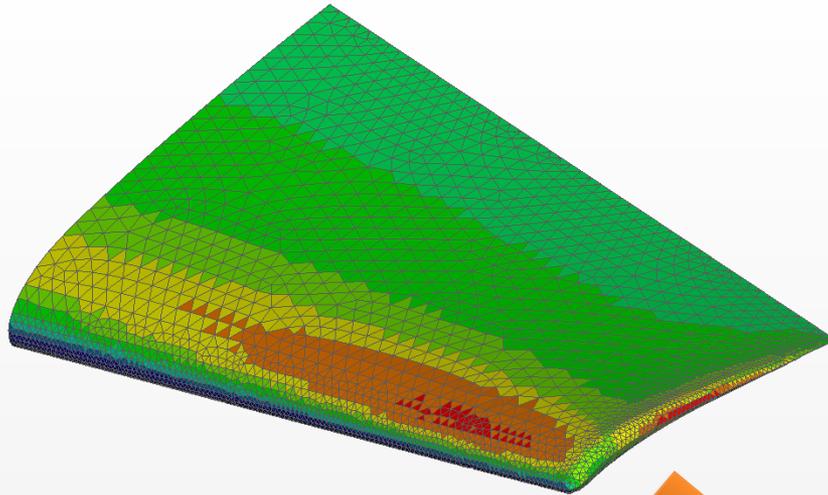
Compare Tool

- Explore differences
- Capture and re-use



Analysis Tools

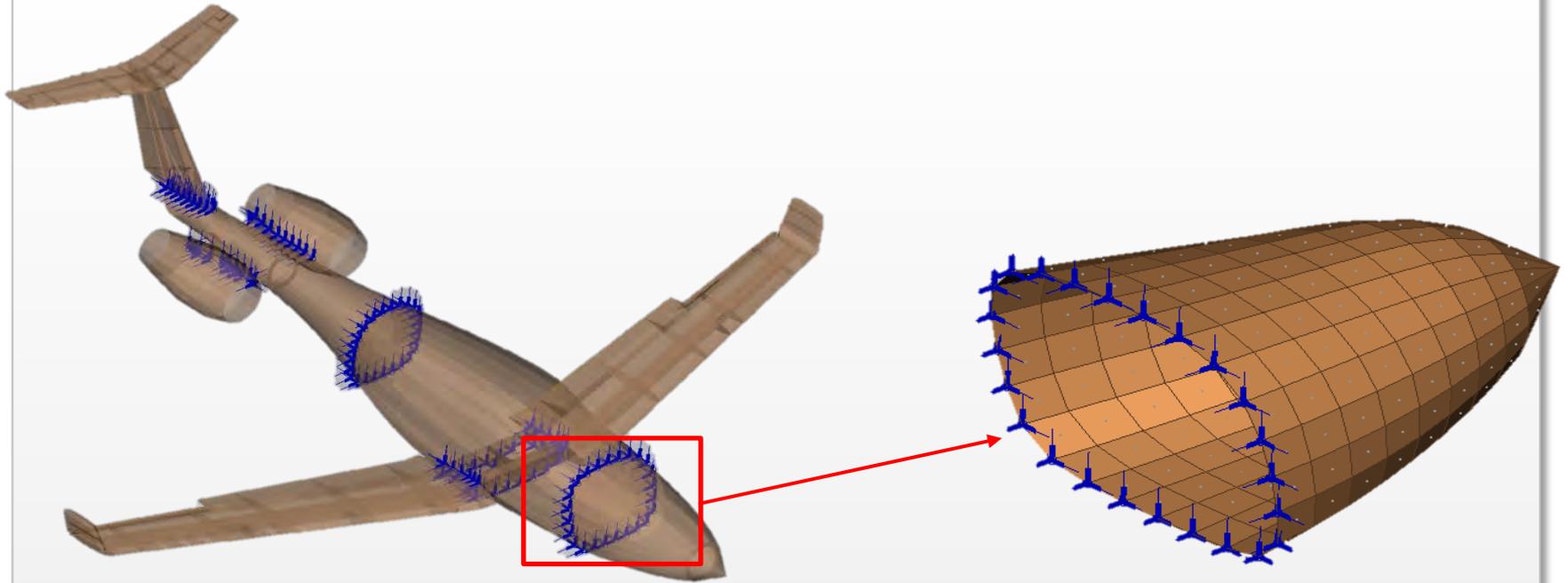
Results Mapping



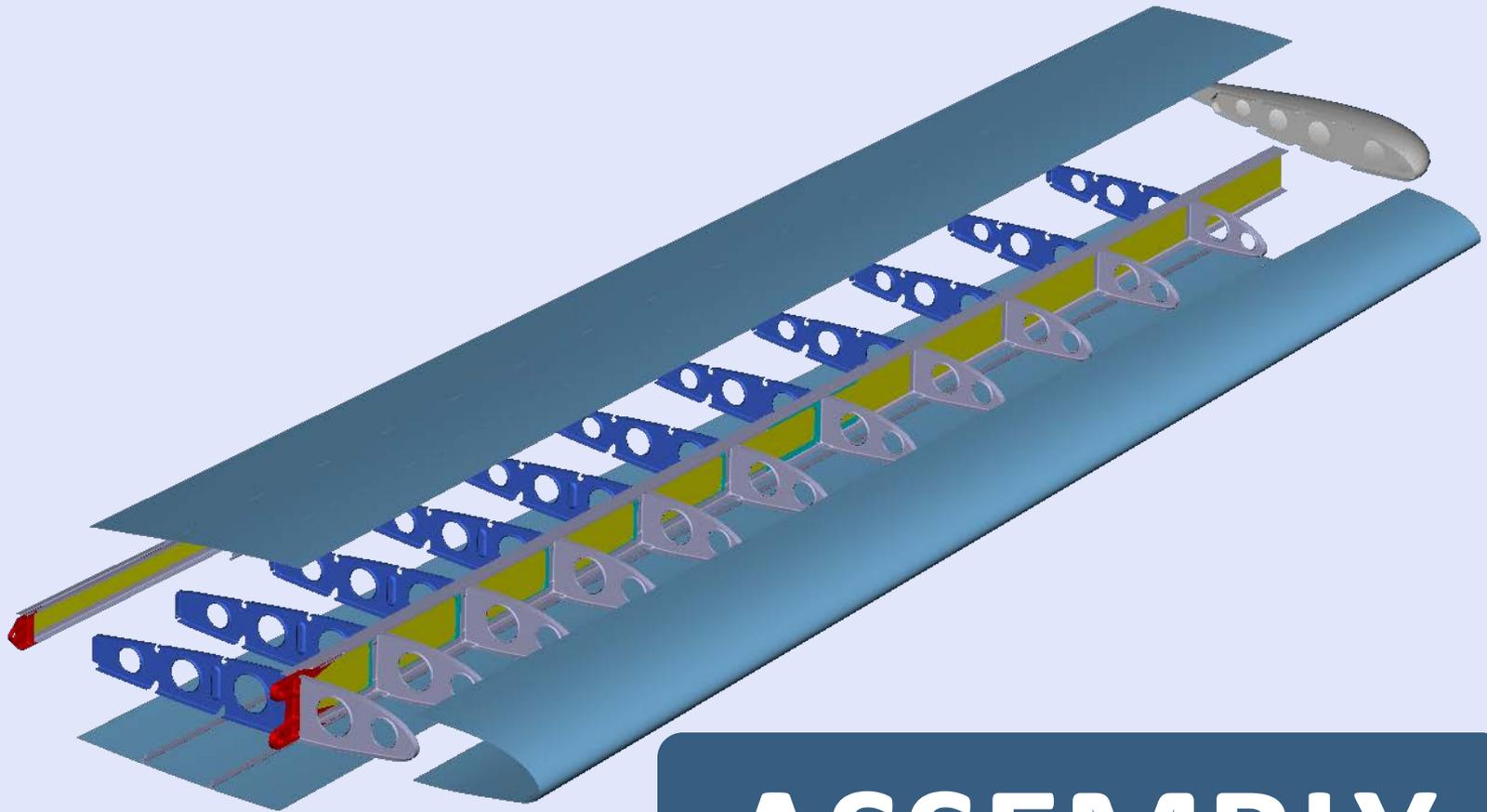
- Nodal thickness
- Pressure
- Initial Stress
- Initial temperature
- Equivalent Plastic Strain
- Strain
- Material Orientation

Analysis Tools

NASTRAN External Superelements



Easy definition of partitioned superelements (EXTRN or SEBULK) through boundary points (SECONCT)



ASSEMBLY

Managed data types

Library Data

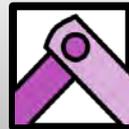
Meshing
Settings



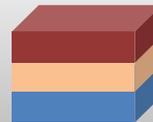
Connection
Templates



Connector
library



Materials



Header
templates



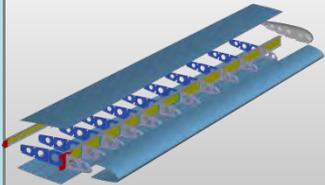
Process
templates



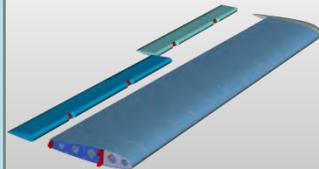
DM

Model & Simulation Data

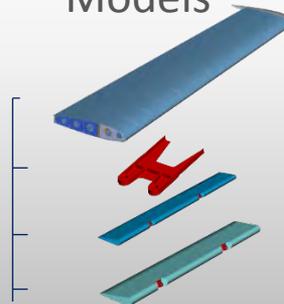
Parts &
Connections



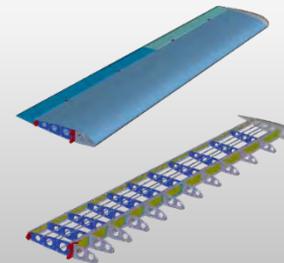
Sub-systems



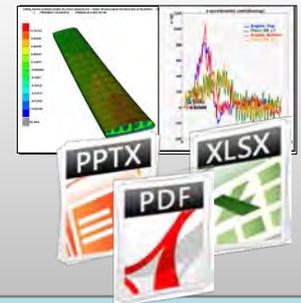
Simulation
Models



Simulation
Runs

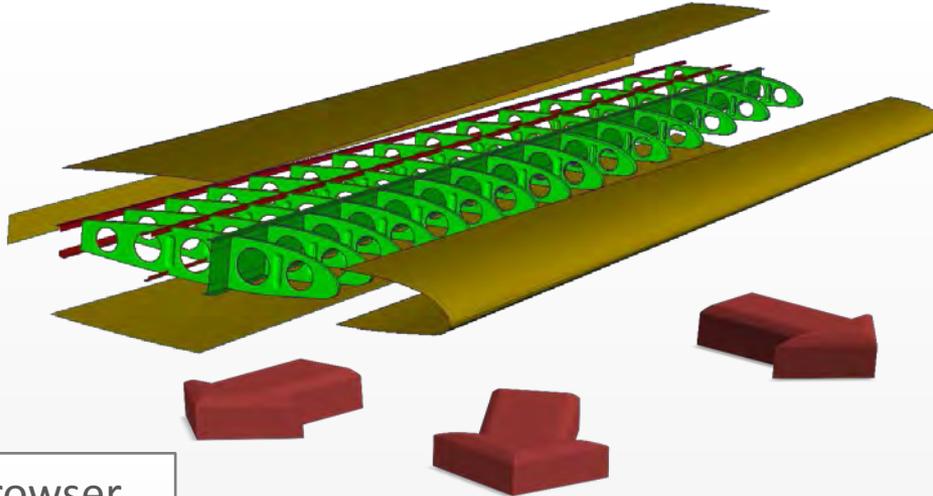


Key-results &
reports



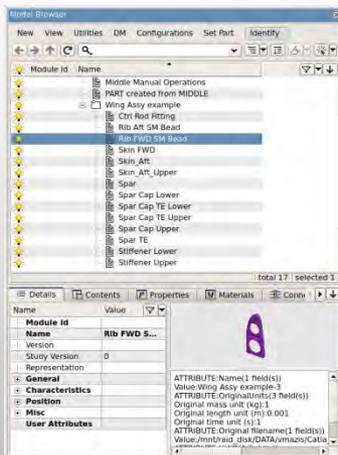
Assembly

Lists and Tools



- Multi-Instantiated Parts generation
- Mesh Representations
- Version Control

Model Browser



Material, Property, SETs Lists

Id	Name	T	MID1	MID	_type
27	PartBody PSOLID			1	PSOLID
28	PartBody@SKIN	1.2	1	1	PSHELL
29	PartBody@SKIN	1.	1	1	PSHELL
30	PartBody@SKIN	1.	1	1	PSHELL
31	PartBody@SKIN	0.8	1	1	PSHELL
32	PartBody@SKIN	0.8	1	1	PSHELL
33	PartBody@SKIN	1.	1	1	PSHELL
34	PartBody@SKIN	0.8	1	1	PSHELL
35	PartBody@SKIN	0.8	1	1	PSHELL
36	PartBody@SKIN	1.	1	1	PSHELL
37	PartBody@SKIN	0.8	1	1	PSHELL
38	PartBody@SKIN	0.8	1	1	PSHELL
39	PSHELL Property created f...	1.2	1	1	PSHELL
40	PSHELL Property created f...	1.2	1	1	PSHELL
41	PartBody@SKIN	2.	1	1	PSHELL
42	SPOT WELD 3.2 mm PFA...				PFFAST
43	SPOT WELD 4.0 mm PFA...				PFFAST

Database Browser

Name	Number
ANSAPART	2
> B.C. SET	2
<input type="checkbox"/> > BC	4
<input checked="" type="checkbox"/> > CONNECTION	76
<input checked="" type="checkbox"/> > COORD	1
<input type="checkbox"/> > CROSS	14
EDGE	
<input checked="" type="checkbox"/> > ELEMENT	14818
<input checked="" type="checkbox"/> > GEOMETRY	6101
<input type="checkbox"/> > GRID	15088
> MATERIAL	5
<input checked="" type="checkbox"/> > MEASUREMENT	2
> METHOD	1
> Nastran Header	1
> PROPERTY	3
SET	1
SOLIDFACET	

Assembly

Part Management

The screenshot displays the Model Browser software interface. The left pane shows a tree view of simulation runs under the 'floor-vibration_trimmed-body.nas' folder. A callout box labeled 'Simulation Run Tree' points to this tree. The right pane shows a 'References' table with columns for various simulation types and checkboxes for their inclusion. A callout box labeled 'Simulation Run Configuration' points to this table. Below the panes is a 'Details' table for the selected simulation run.

Name	Value
Name	front-left-door_taurus-demo_15w24_nvnh_nvnh-fe_001_0
Module Id	front-left-door
Study Version	0
Representation	nvnh-fe
File Type	ANSA
Project	taurus-demo
Release	15w24

- Parts
- Configurations
- Subsystems
- Simulation Models
- LoadCases
- Simulation Runs
- Side Panel

Assembly

Includes Manager

Includes Manager

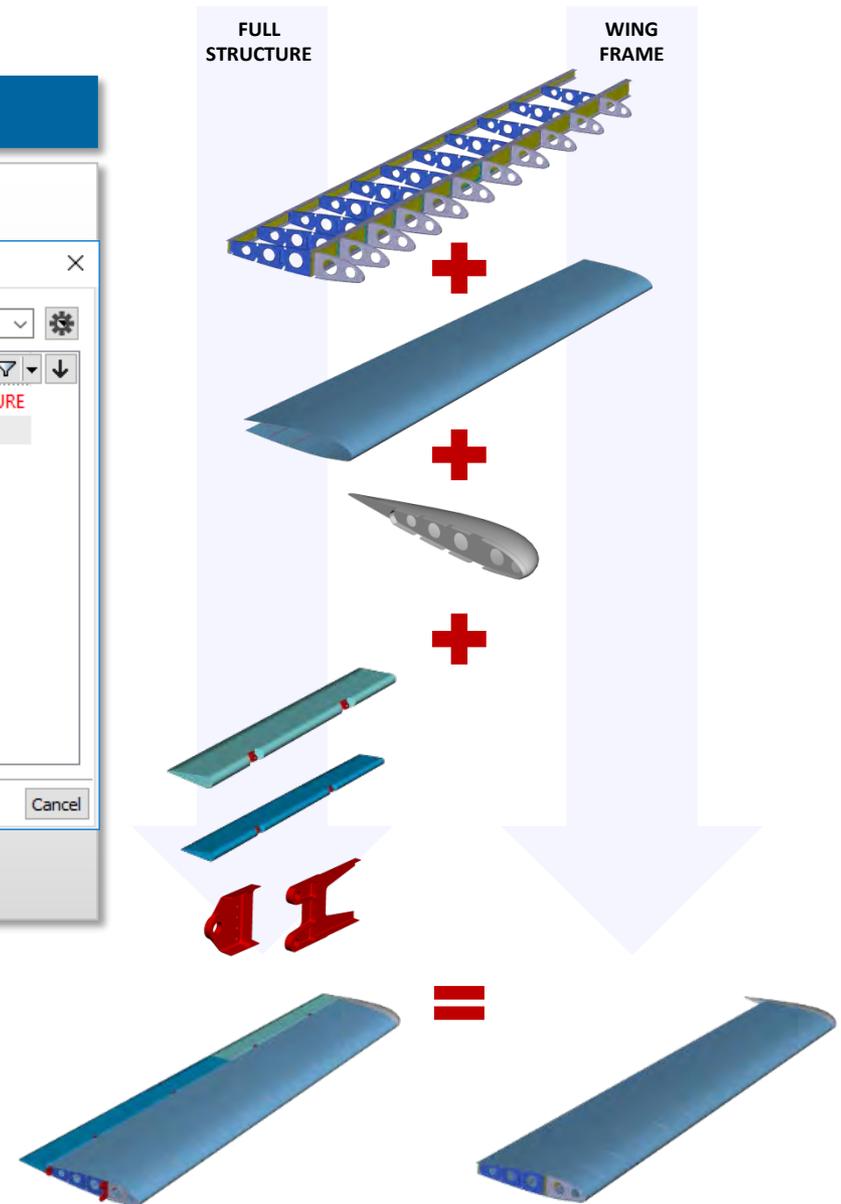
Search:

Id	Name	FULL STRUCTURE	WING FRAME
1	SKIN_UPPER	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
2	SKIN_LOWER	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
3	RIBS_AFT_S/M	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
4	HINCHES	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
5	AILERON_LH	<input checked="" type="checkbox"/>	<input type="checkbox"/>
6	FLAP_LH	<input checked="" type="checkbox"/>	<input type="checkbox"/>
7	SPAR_MAIN_ASSY	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
8	SPAR_TRAILING_EDGE_ASSY	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
9	TIP_FAIRING	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
10	RIBS_FWD_MACHINED	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
11	SPAR_REINFORCEMENT	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
12	FITTING_TE	<input checked="" type="checkbox"/>	<input type="checkbox"/>

Search:

Id	Name
1	FULL STRUCTURE
2	WING FRAME

Buttons: OK, Load Files, Apply, Clear, New Configuration, Log File, Cancel



Assembly

Connection Manager

The screenshot displays the Connection Manager software interface. The main window is titled "Connection Manager" and has a menu bar with "View", "Functions", and "Options". The "Spotweld Points" tab is active, showing a list of connections with columns for Connectivity, ANSA Id, Name, and Status. A purple callout box labeled "Connectivity" points to the Connectivity column. Below this list is a table of connection entities with columns for ID, D, P1, P2, P3, P4, Status, Error Class, Name, Comment, custom, and TID. A purple callout box labeled "Connection Entities" points to this table. To the right, the "FE Rep. Settings" panel is visible, showing various configuration options for the FE representation type, which is set to "CFAST". A purple callout box labeled "FE Representation Type" points to the FE Rep. Type field. The bottom right of the interface shows a 3D model of a mechanical assembly with a purple mesh overlaying the components.

- Point, Line, Surface Connections
- Numerous FE Representations

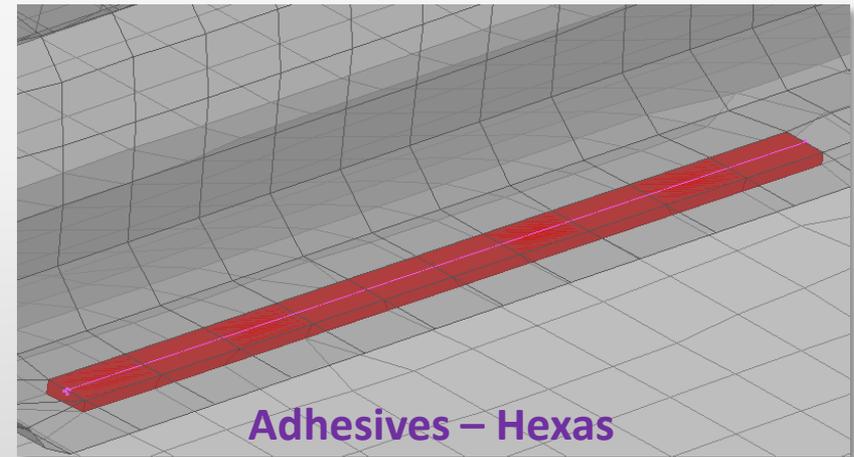
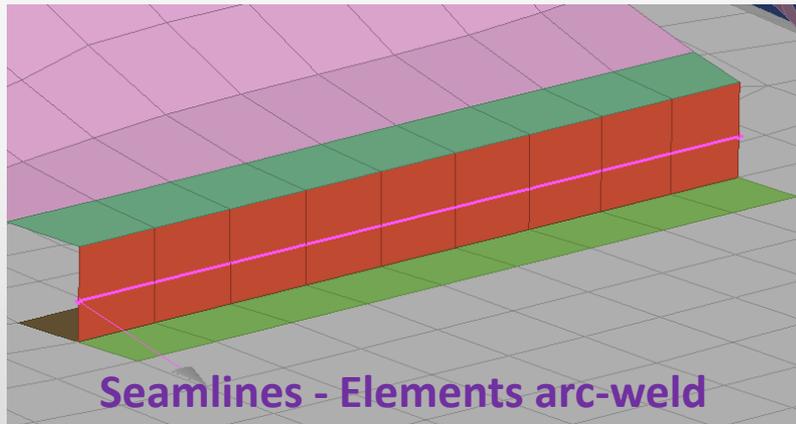
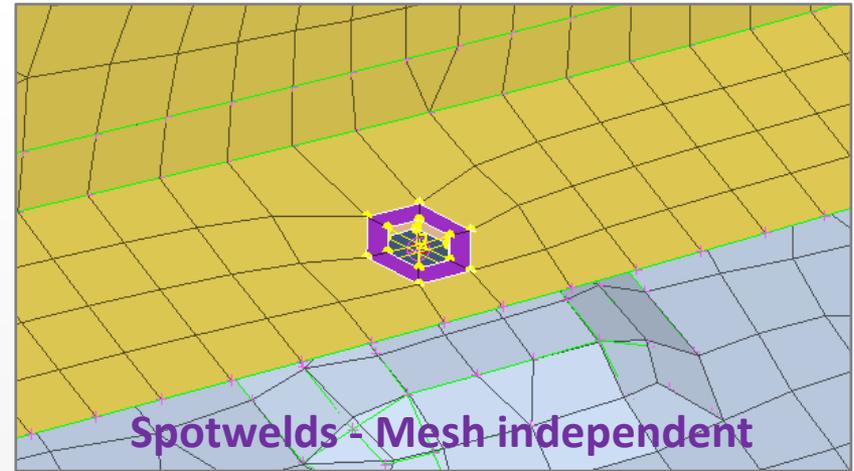
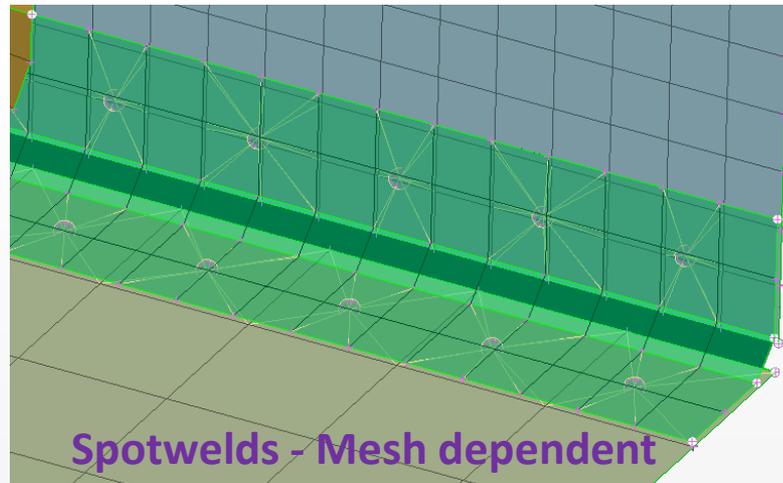
Connectivity	ANSA Id	Name	Status
-5		Rib FWD SM Bead	
-6		Skin FWD	
-2		PART created from MIDDLE	
-16		Spar	
-4		Rib Aft SM Bead	
-11		Skin_Aft	

ID	D	P1	P2	P3	P4	Status	Error Class	Name	Comment	custom	TID
100077	3.2	-11	-2			Ok	CFAST				
100082	3.2	-11	-2			Ok	CFAST				
100083	3.2	-11	-2			Ok	CFAST				
100091	3.2	-11	-2			Ok	CFAST				
100093	3.2	-11	-2			Ok	CFAST				
100100	3.2	-16	-2			Ok	CFAST				
100107	3.2	-16	-2			Ok	CFAST				
100110	3.2	-16	-2			Ok	CFAST				
100112	3.2	-16	-2			Ok	CFAST				

TID	FE Rep Type
	CFAST
General	
Search Dist	
Use Thickness to Diameter Map	<input checked="" type="checkbox"/>
Body	
PFAST ID	
Weld Type	ELEM
Create Piercing Points GA,GB	<input type="checkbox"/>
Flange Positioning	
Dist From Perim	
Force CFAST Flange Criteria	<input checked="" type="checkbox"/>
Max Flange Angle	
Interactions	
Cut off Adhesives	<input checked="" type="checkbox"/>

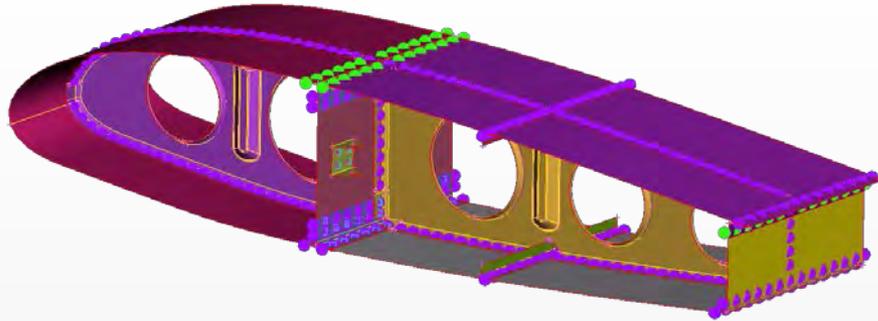
Connection Manager

Weld points and lines FE-representations

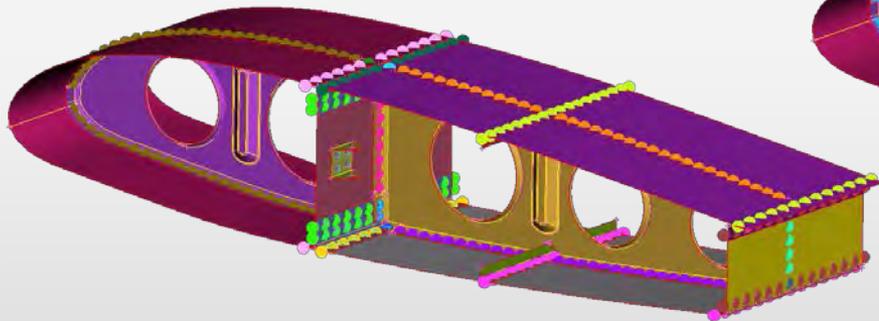


Connection Manager

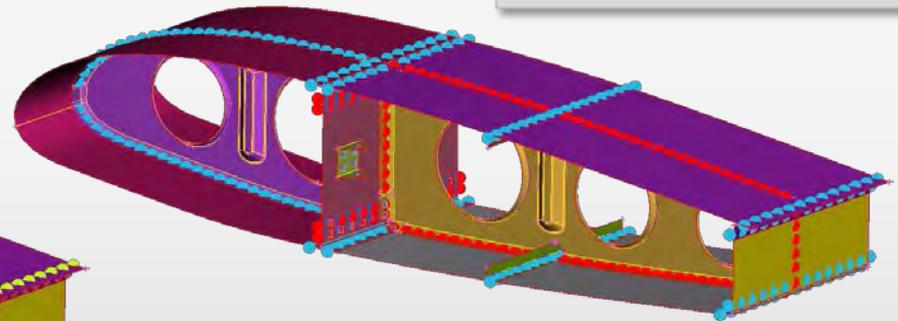
Color coding



...by diameter



...by number of parts



...by status

Color Coding by:

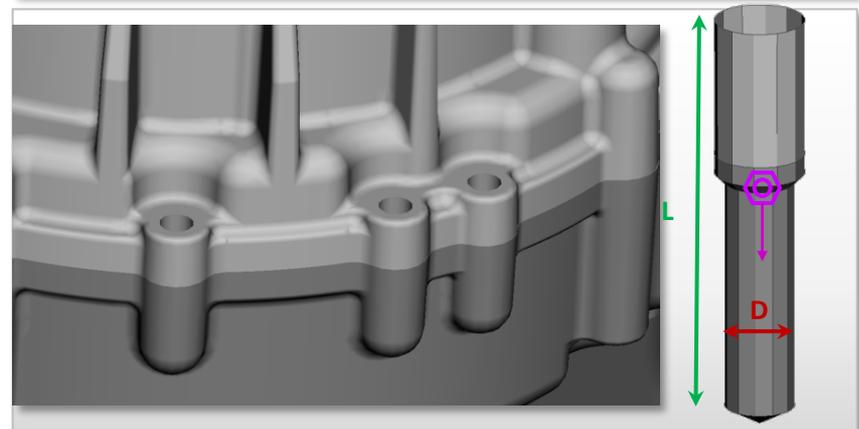
- Number of parts
- Connectivity
- Type
- Status
- FE-Representation
- Diameter
- Template

Connection Manager: Bolts

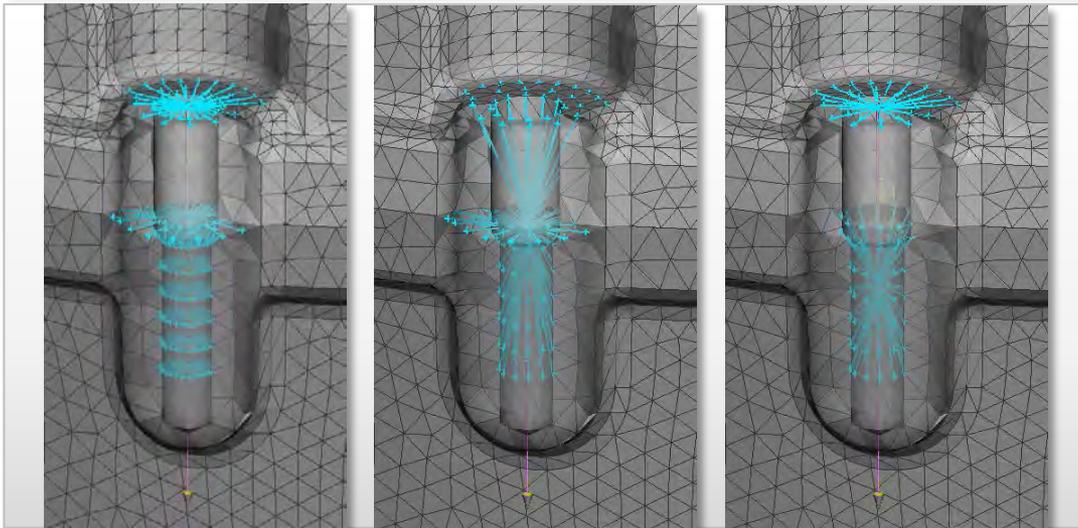
From bolt geometry...



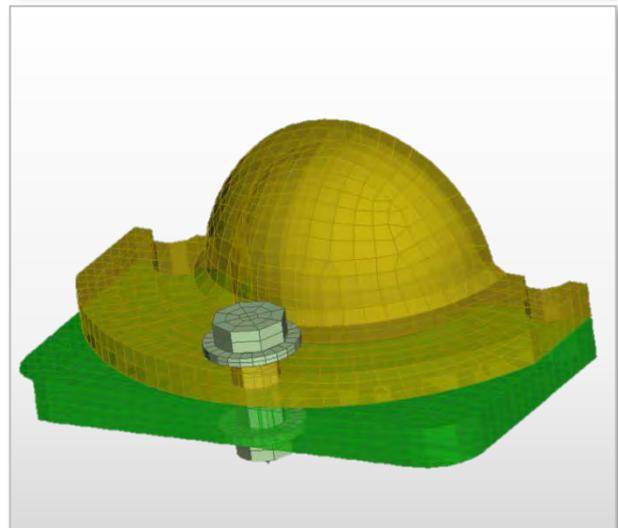
From tubes...



...to numerous realization patterns

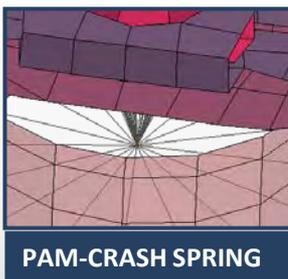
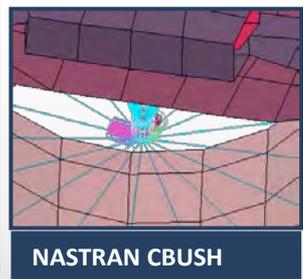
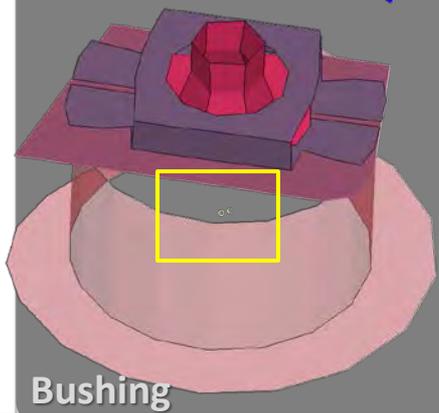
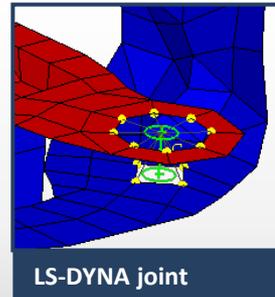
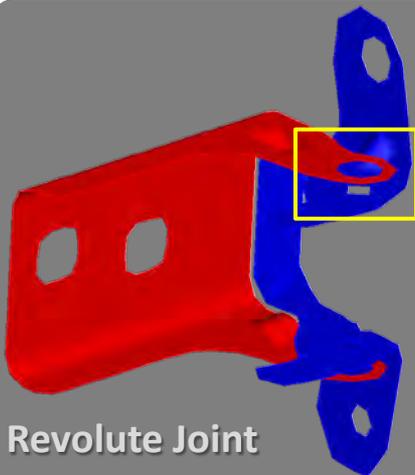


Solid bolt representation



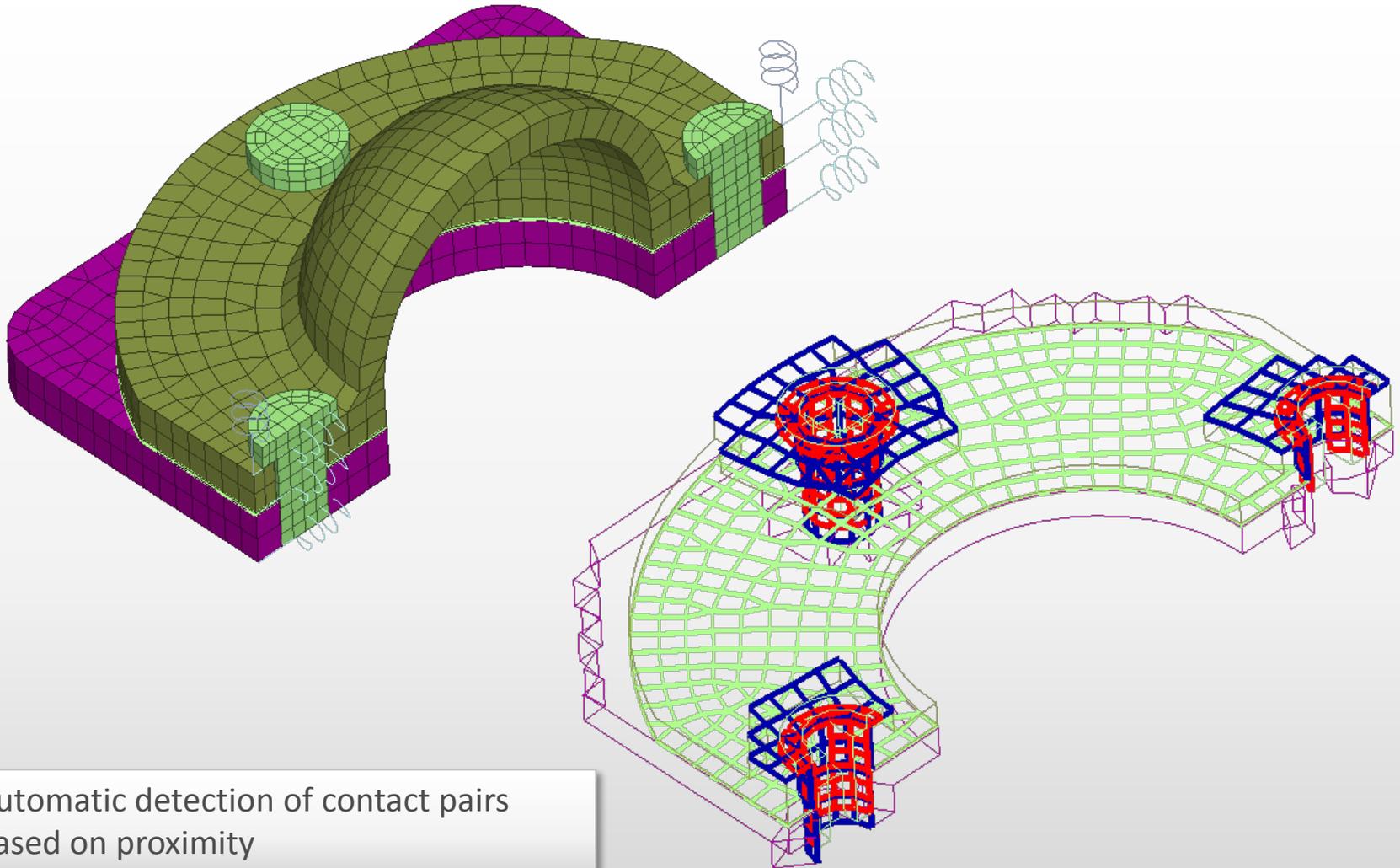
Connector Entities

Library-based and fully customizable result

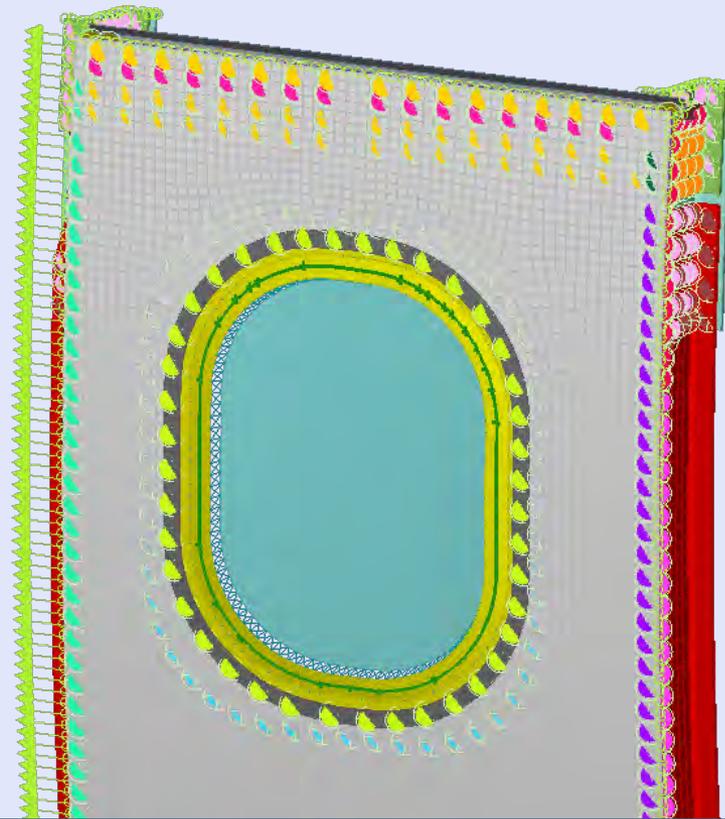


Contacts

Automatic contact definition



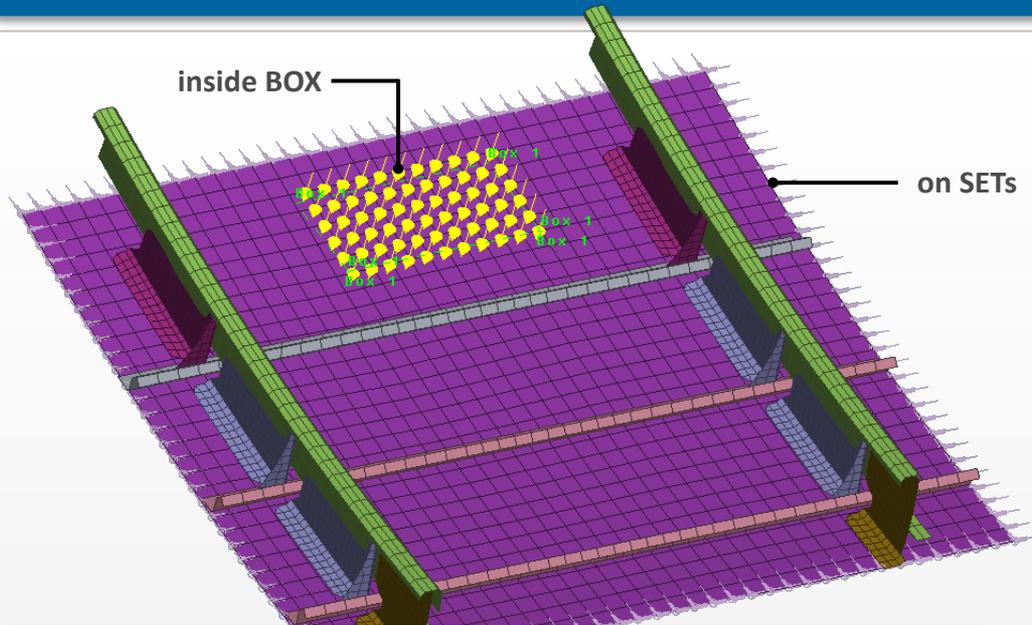
Automatic detection of contact pairs
based on proximity



ANALYSIS SET-UP

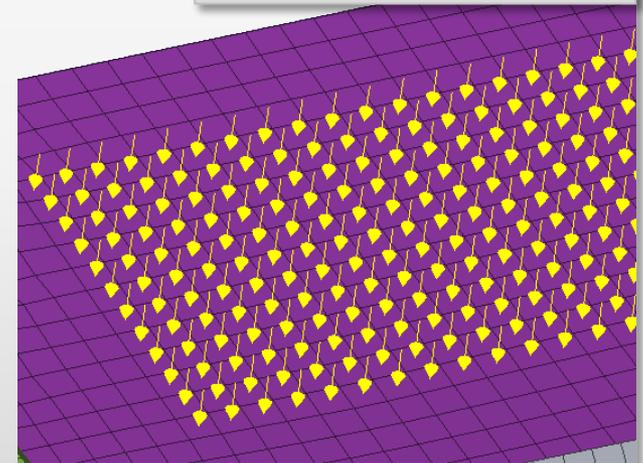
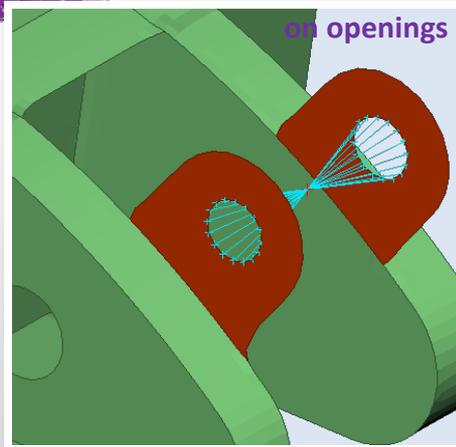
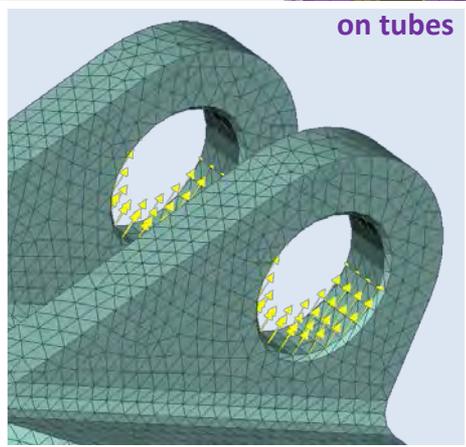
Analysis Set-up

Loads and Constraints



...applied on

- SETs
- tubes
- around openings
- entities inside BOX
- entities by Name
- entities by ID range
- overlapping areas



mesh independent application !!

Analysis set-up

Material Database

- Loaded automatically during start-up
- Can load one MatDB per deck
- Materials update from MatDB:
 - By material name
 - By material id

Material Mapping



Automatic transfer of curves and tables between materials !!

Analysis set-up

Model validation through build in Checks

Hundreds of model checks

General and solver checks

Automatic fix functions

Custom checks through script

Check templates

Color coding of results

Type	Entity	ID	Description
[-] Duplicate			
[-] PLINKS			
[-] Mass Scale Ratio			
[-] Initial Velocity			
[-] Incomplete			
[-] Funct:End Time			
[-] Curves:Axes			
[-] Dependency Pamcrash			
[+] Check Contact 409999			
[-] Undefined Properties			
[+] Undefined Materials			
[-] Range			
[-] Multiple Boundary Conditions			
[+] Massless			
[+] Free:Whole DB			
[-] Warning	NODE	313	Node is Free and doesn't belong to any element
[-] Warning	NODE	312	Node is Free and doesn't belong to any element

Show only: Show Hide All

total 1167 selected 0

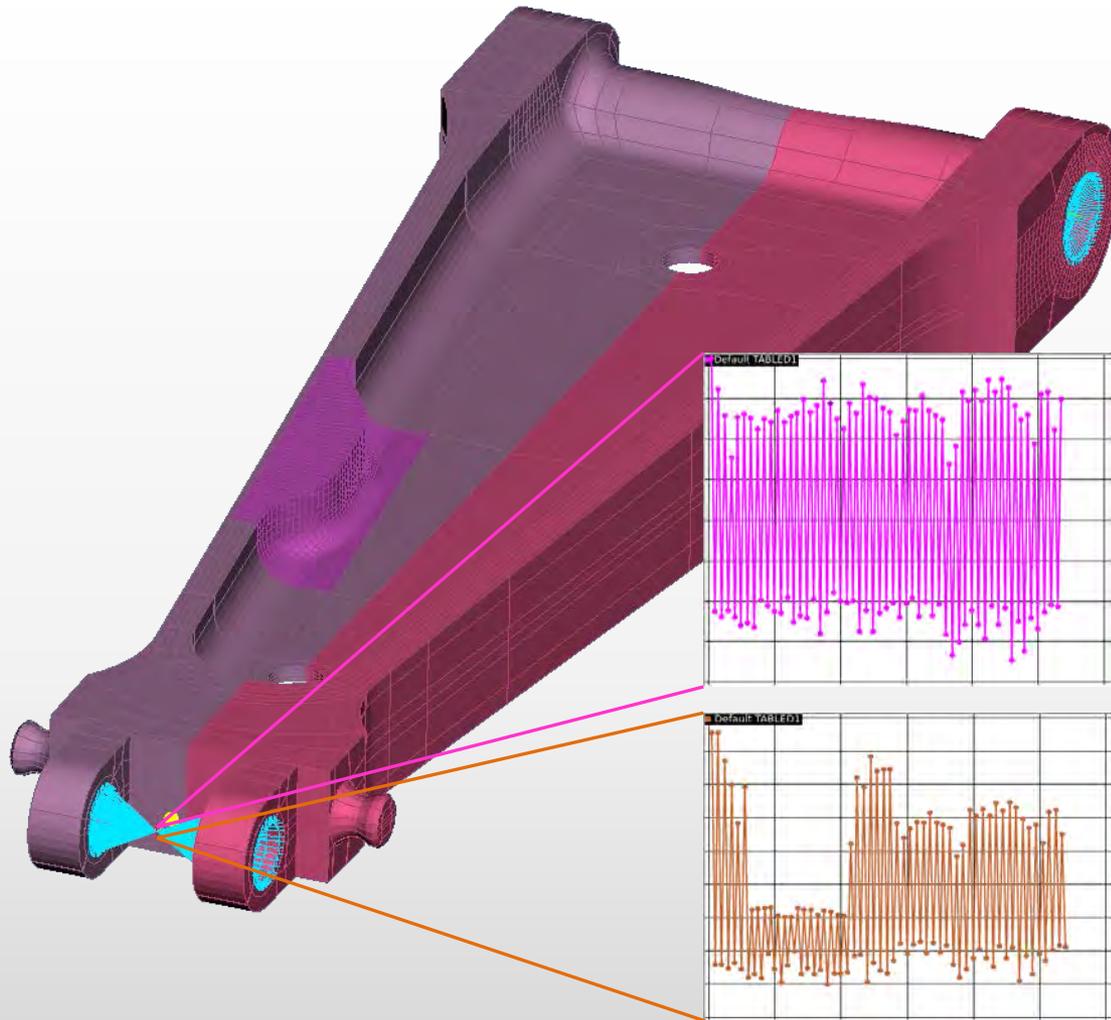
 **Errors**

 **Warnings**

 **OK**

Analysis Set-up

NASTRAN Embedded Fatigue

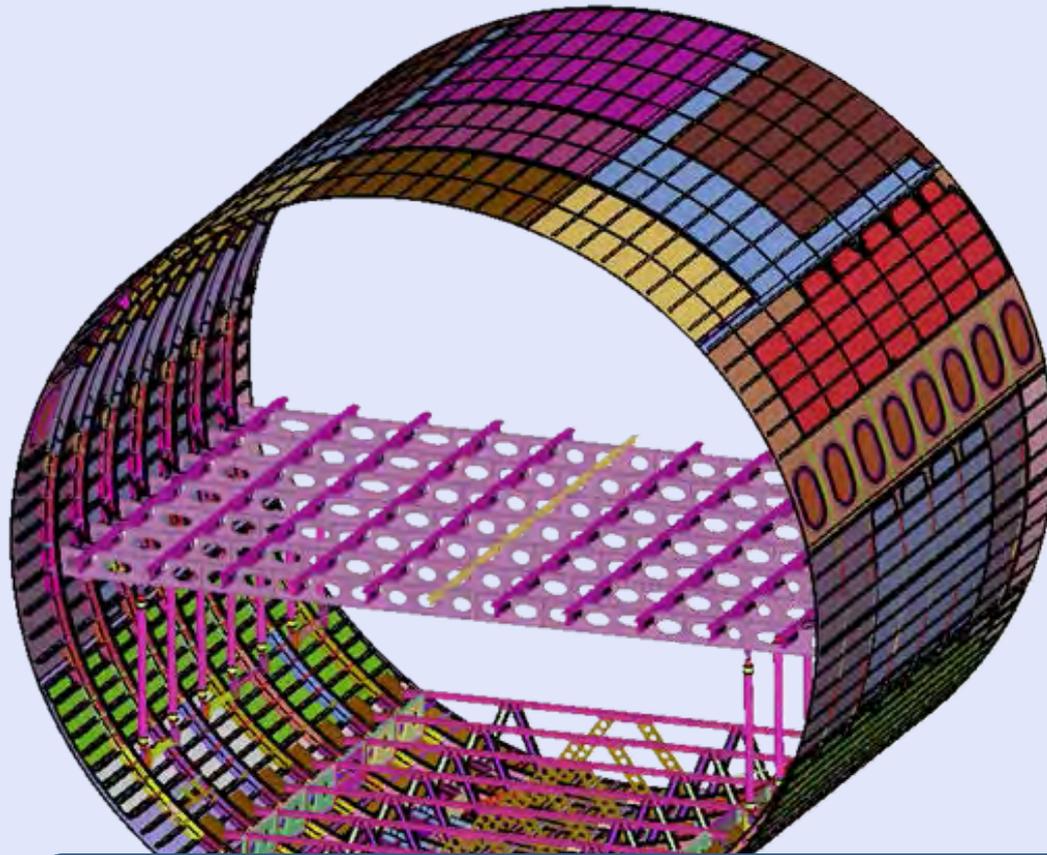


Support of all fatigue analysis types:

- S-N Analysis
- E-N Analysis
- Multiple Loading
- Modal Transient Analysis
- Design Optimization

NEF supported keywords:

- FATIGUE
- DTI, UNITS
- FTGDEF
- FTGPARAM
- FTGEVNT
- MATFTG
- PFTG
- TABLFTG
- UDNAME



PROCESS AUTOMATION

Process Automation

Python Scripting Language



Python Scripting Capabilities

Entities Handling

Create

Delete

Modify

Acquire Info

Custom GUI

Line edits

Buttons

List boxes

Check buttons

Radio groups

Tables

Task Manager

ASCII files I/O

Read lists

Write XML files

Create lists

Read XML files

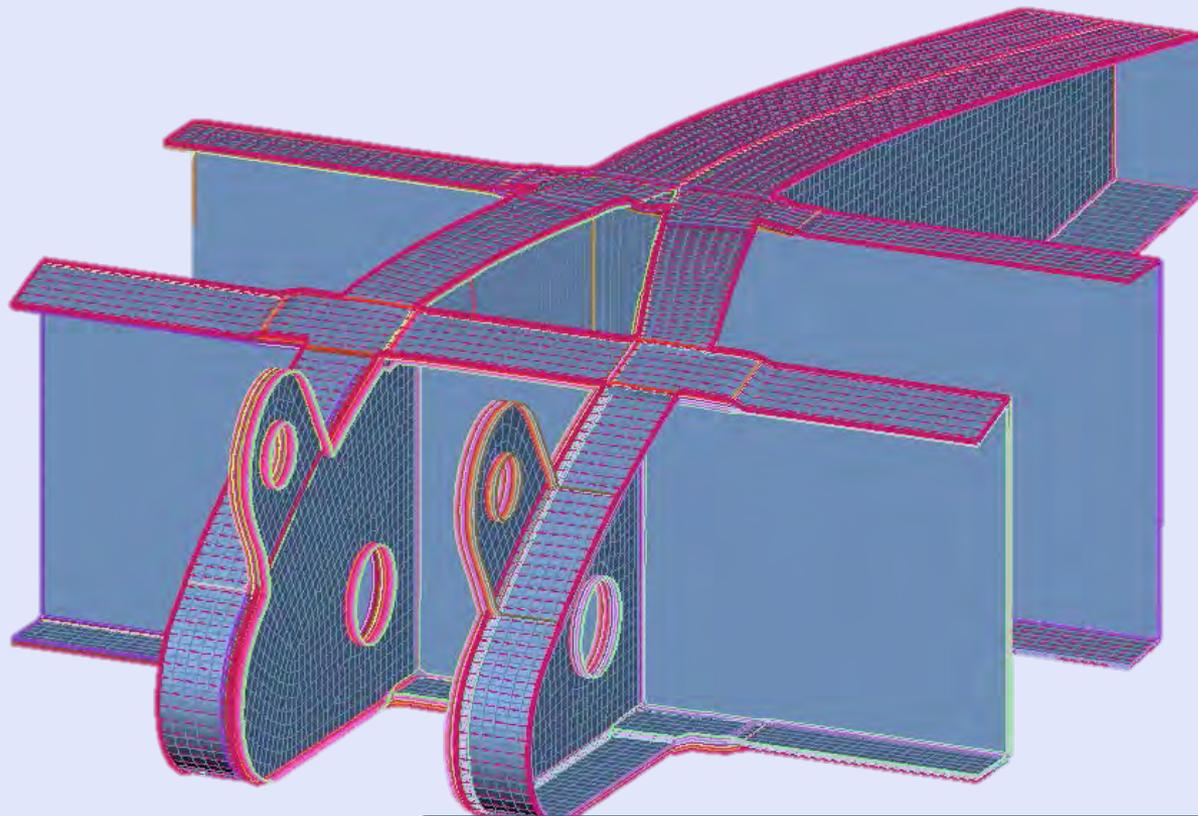
ANSA core functionality in batch

CAD files translation

Geometry checks

Batch Mesh

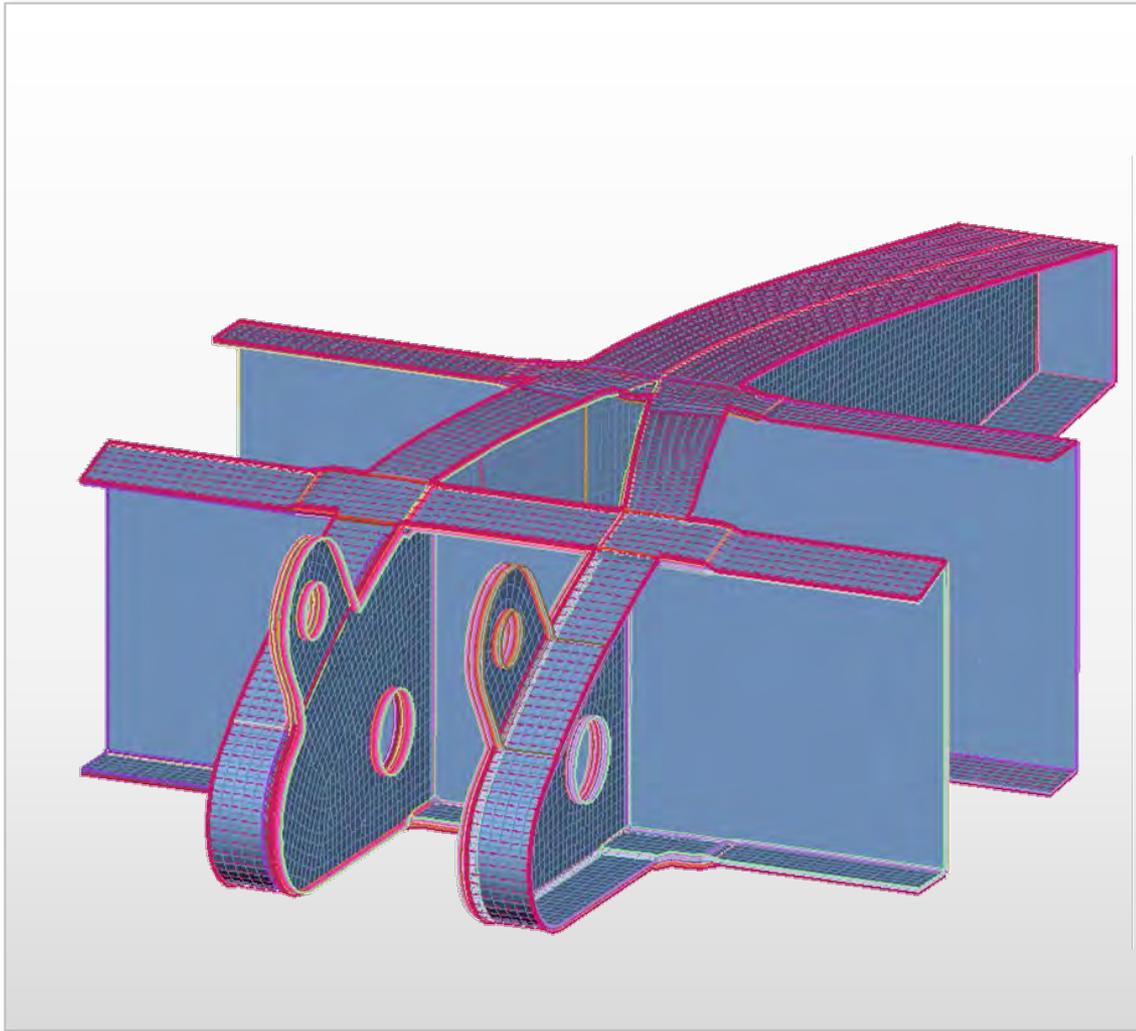
Middle skin extraction



COMPOSITES

Composites

Composites modeling



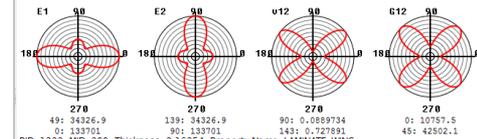
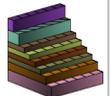
Effective Engineering Properties:
 E1: 133701, E2: 56938.6, N1: 0.208924, N2: 0.0889734, G: 10757.5

ABD matrix:
 237046.75037 21090.862969 -56.2394123 | 215599.34446 13245.172318 -26.9949173
 21090.862969 21090.862969 -56.2394123 | 13245.172318 90427.936983 -26.9949173
 -56.2394123 -56.2394123 18718.040818 | -26.9949173 -26.9949173 11285.839368

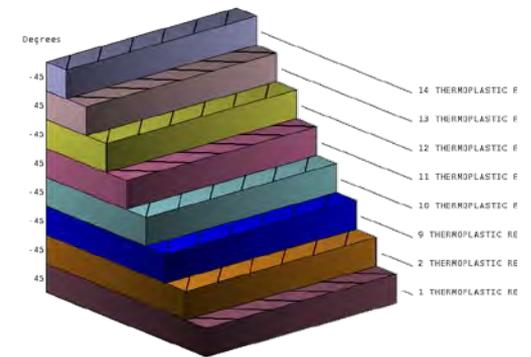
 215599.34446 13245.172318 -26.9949173 | 236610.80718 11739.555779 -13.43746965
 13245.172318 90427.936983 -26.9949173 | 11739.555779 116470.81542 -13.43746965
 -26.9949173 -26.9949173 11285.839368 | -13.43746965 -13.43746965 9056.8427551

Laminate PART:
 Total thickness: 1.74
 Reference plane: Bottom surface
 Symmetry: None

Layer 1:	thick: .31999999285,	angle:	0.,	material:	309 Default MAT8
Layer 2:	thick: .31999999285,	angle:	45,	material:	309 Default MAT8
Layer 3:	thick: 0.1400000006,	angle:	0.,	material:	318 Default MAT8
Layer 4:	thick: 0.1400000006,	angle:	0.,	material:	318 Default MAT8
Layer 5:	thick: 0.1400000006,	angle:	0.,	material:	318 Default MAT8
Layer 6:	thick: 0.1400000006,	angle:	0.,	material:	318 Default MAT8
Layer 7:	thick: .21999999881,	angle:	0.,	material:	320 Default MAT8
Layer 8:	thick: .31999999285,	angle:	0.,	material:	320 Default MAT8

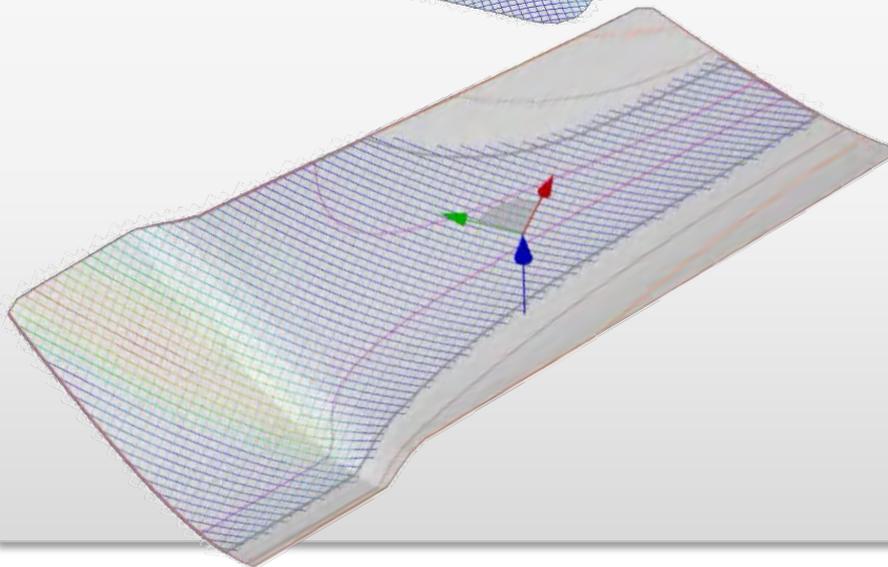
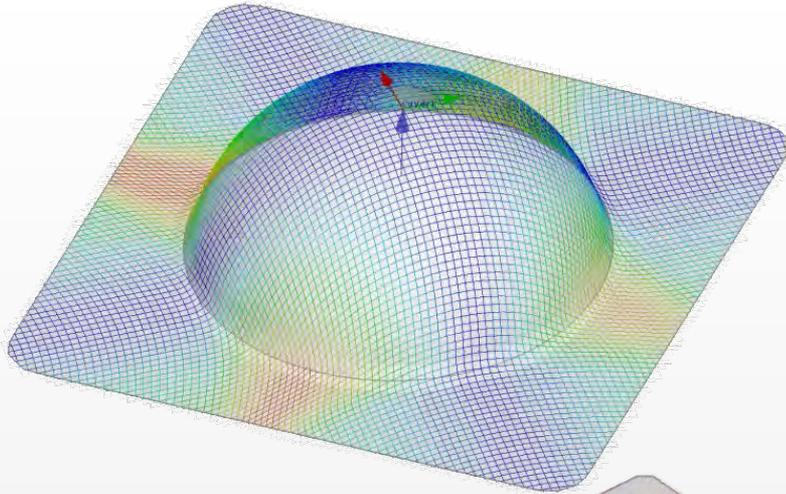


PID: 1233, MID: 309, Thickness: 2.16354, PropertyName: LAMINATE_WING
 MODULE ID: PART NAME: front_wing_v3



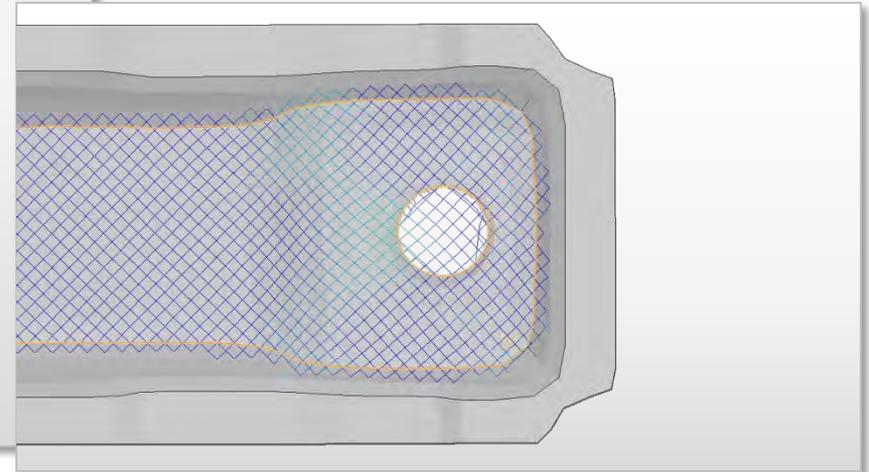
Composites

Draping Tool



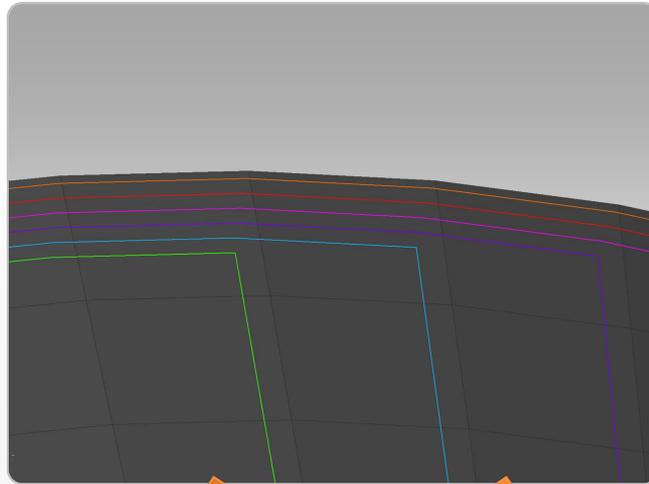
- Inherit Fishnet algorithm
- Based on Rosettes with start and stack directions
- Calculate thickness and orientation of warp and weft woven
- Define initial material orientation

Disregard holes / beams option

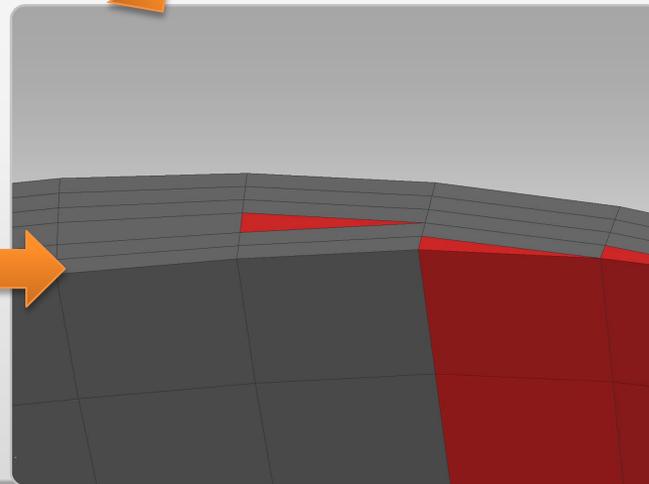
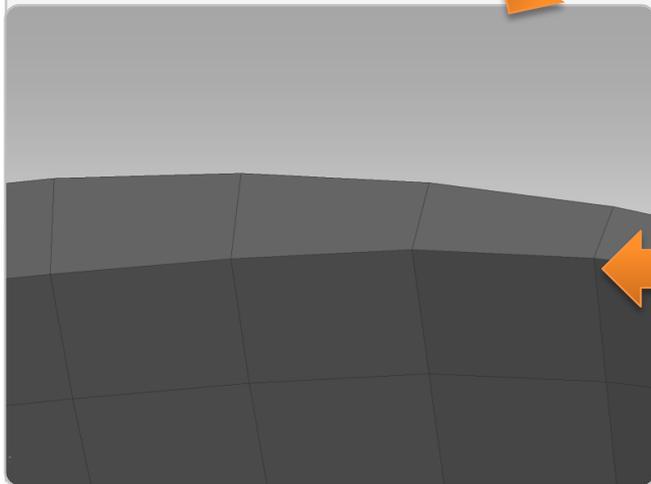


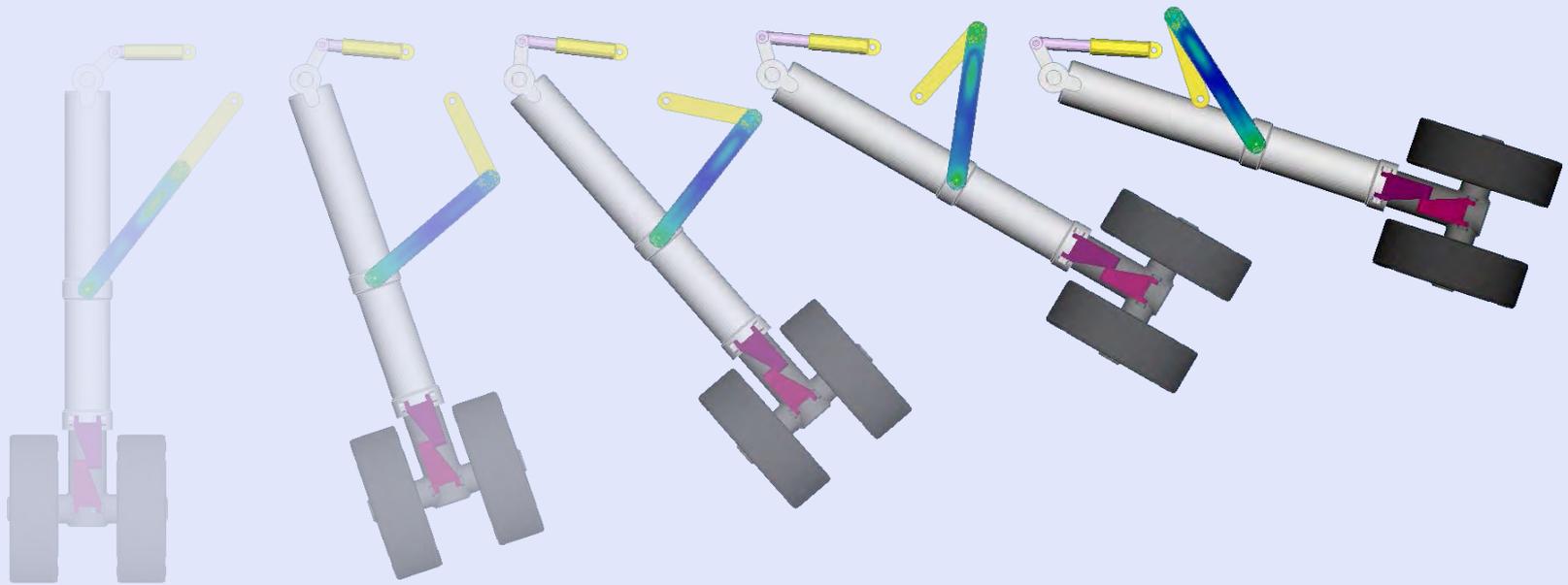
Composites

Solid Composites modeling



- **Volumize** Composite shell elements
- **Single** element modeling (modify layer stackup)
- **Per-ply** element modeling (fill ply drop-offs)
- **Bidirectional** single to per-ply conversion





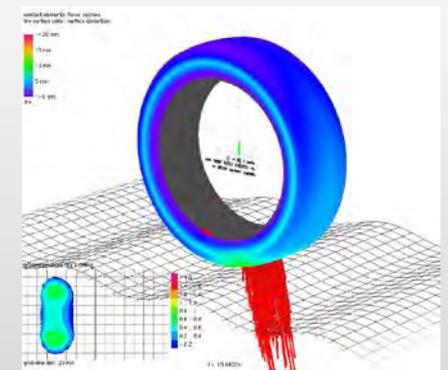
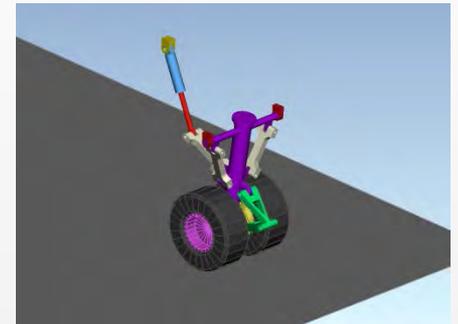
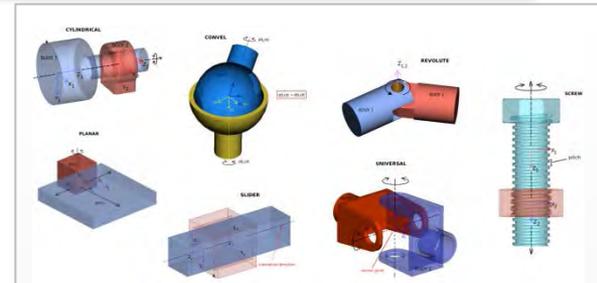
KINETICS

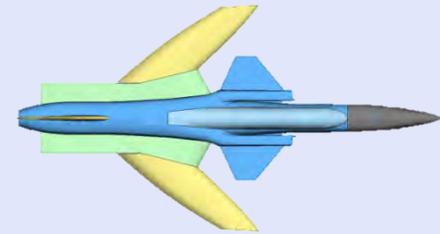
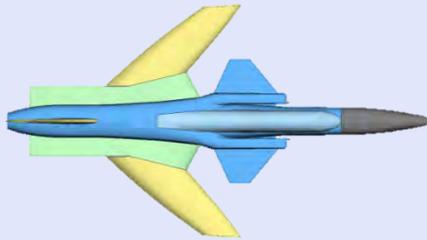
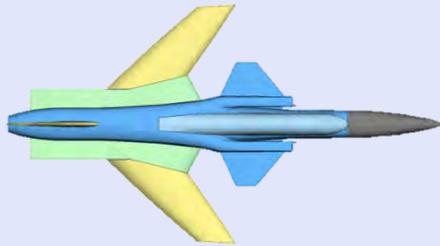
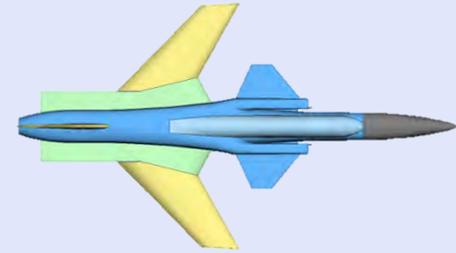
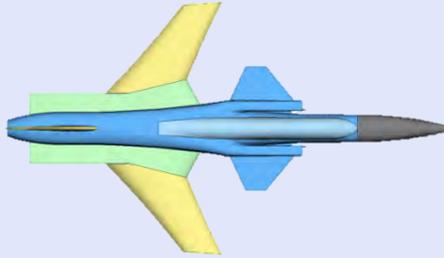
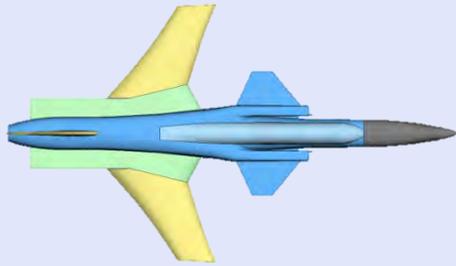
Kinetics

Multi Body Dynamics Solver



- KINETICS is an integrated MultiBody Dynamics software within ANSA that incorporates an in-house solver
- MBD is used to study the kinematic or dynamic behavior of mechanical systems that undergo large displacements
- MBD applications can be found in many areas of engineering like automotive, machining, industrial, robotics, biomechanics and more
- Dedicated menu for KINETICS

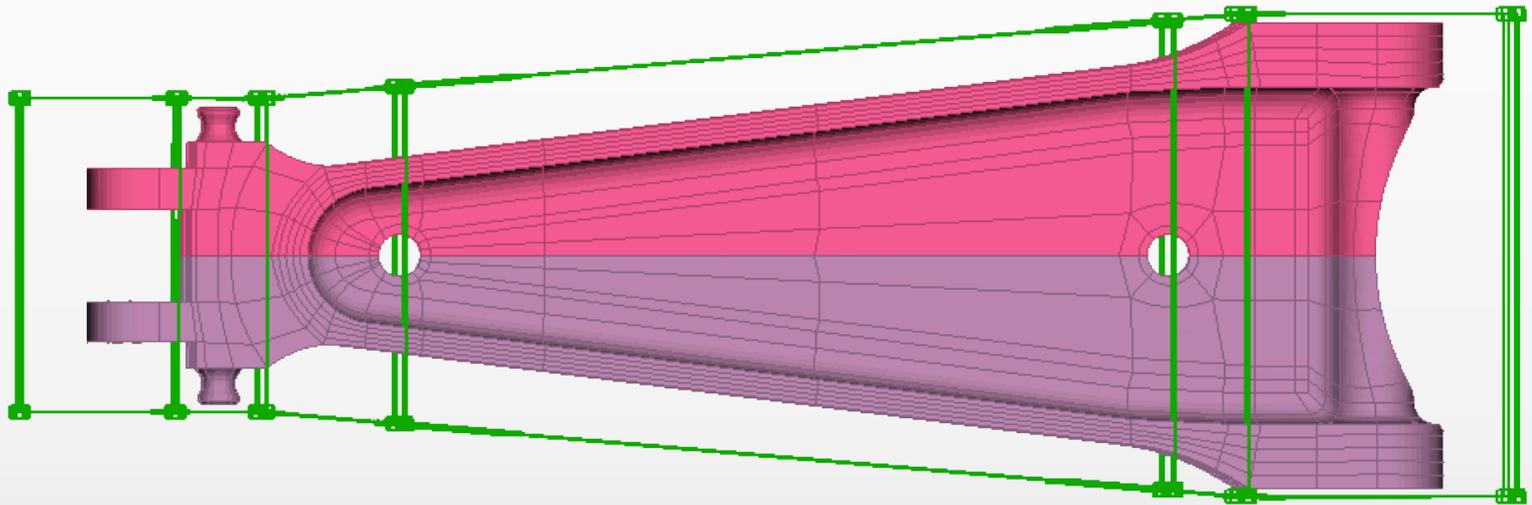




MORPHING - OPTIMIZATION

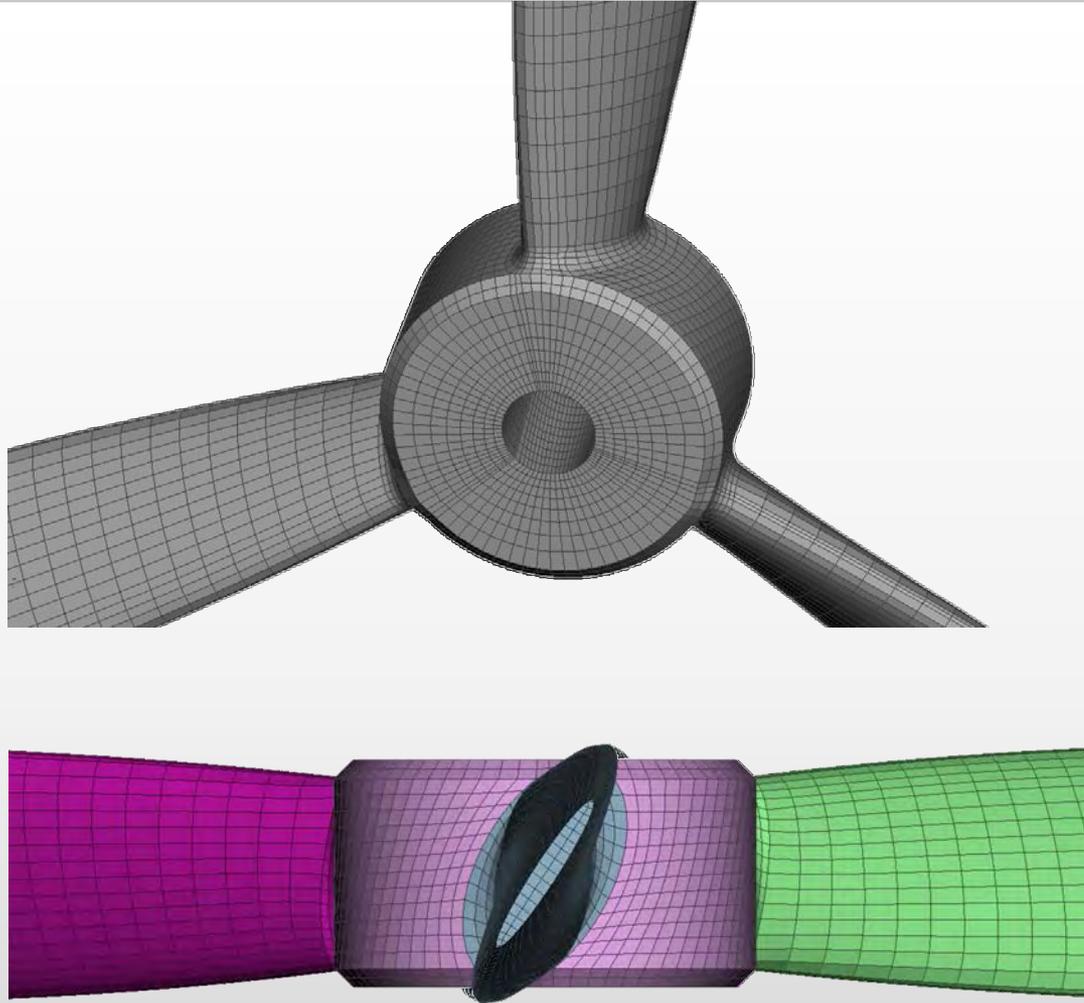
Morphing

Box Morphing



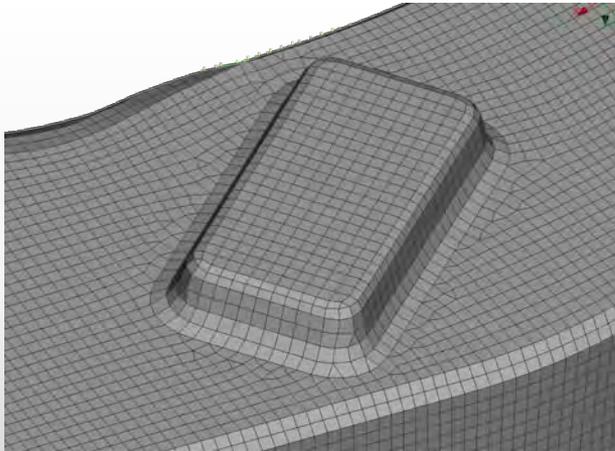
Morphing

Direct Morphing

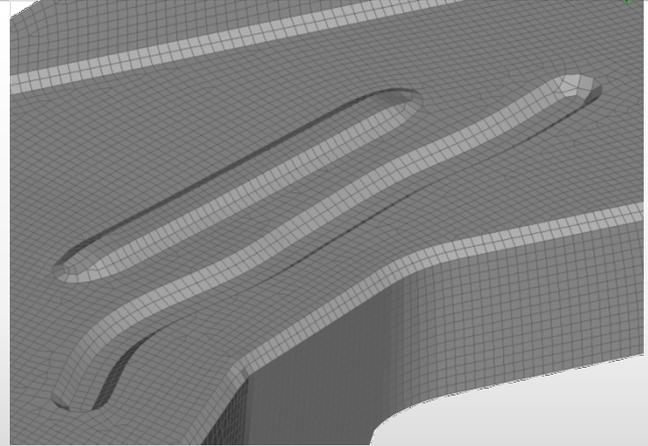


Features Creation

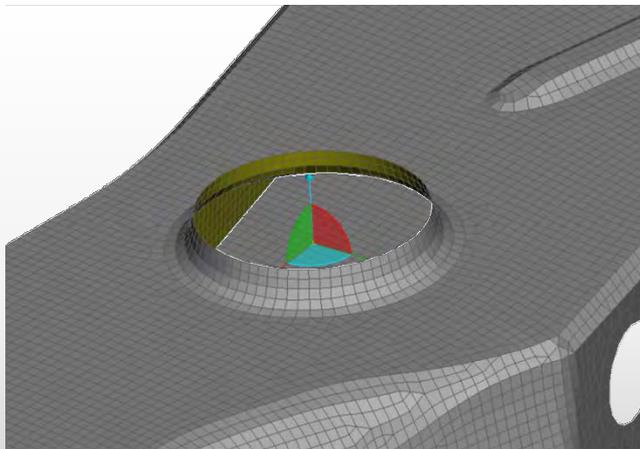
Rectangular Rounded Stamp



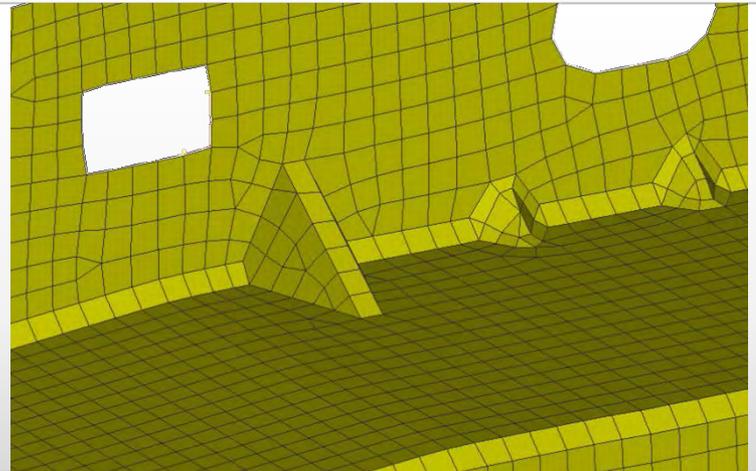
Curved & Rounded Flat Bead



Circular flange opening

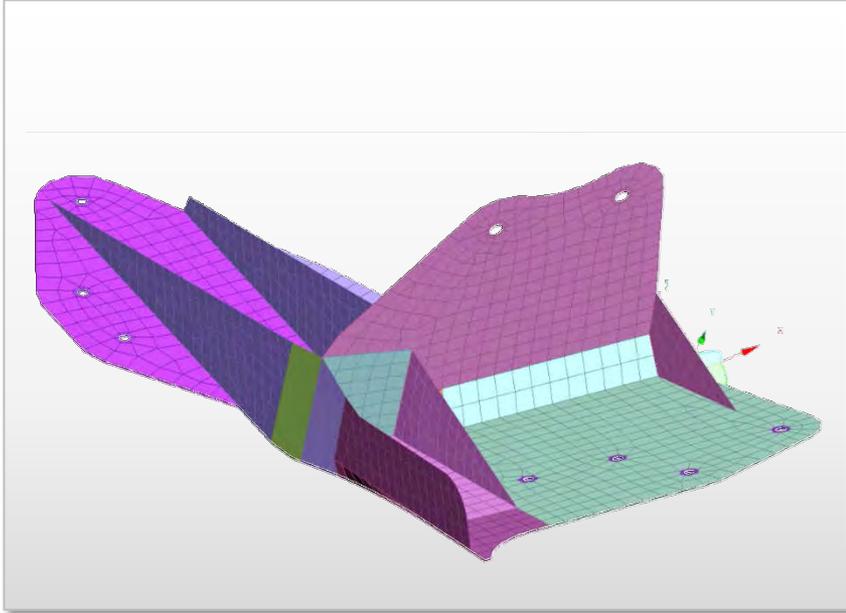


Gussets

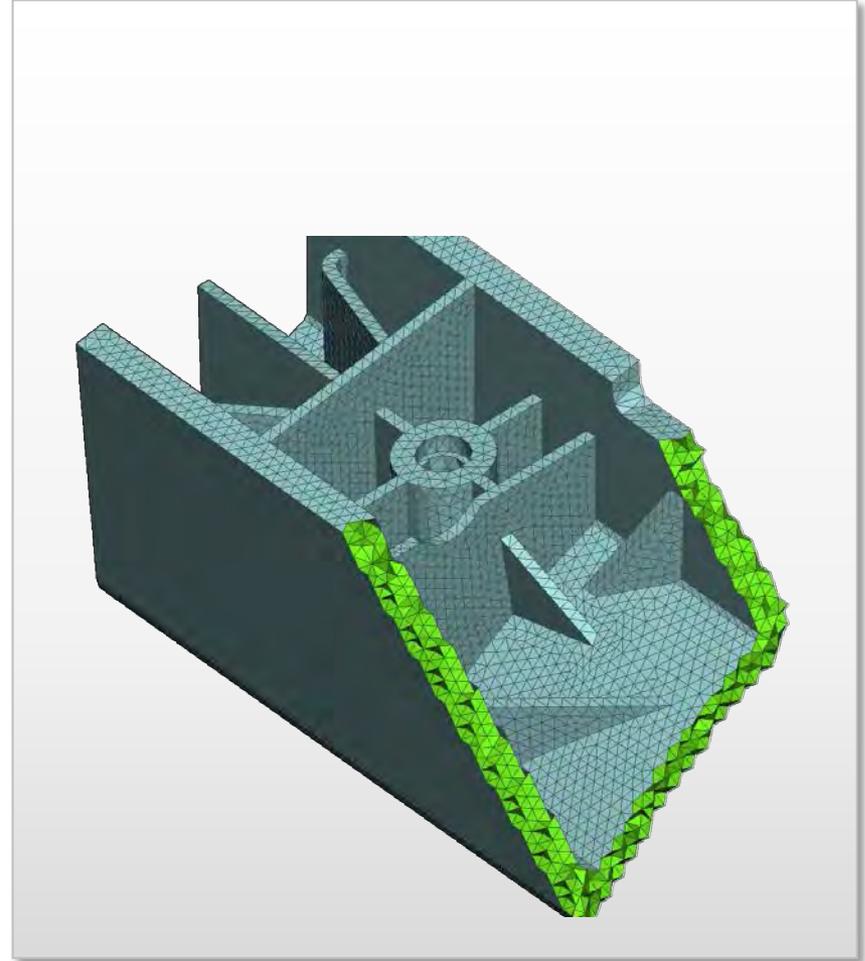


Features Creation

Shell Rib

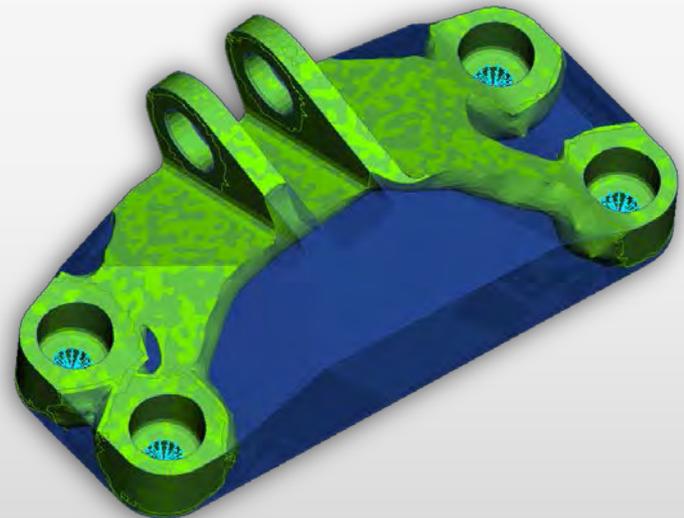
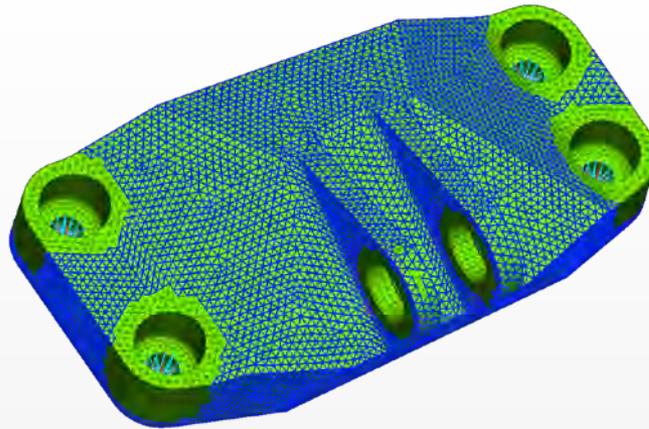
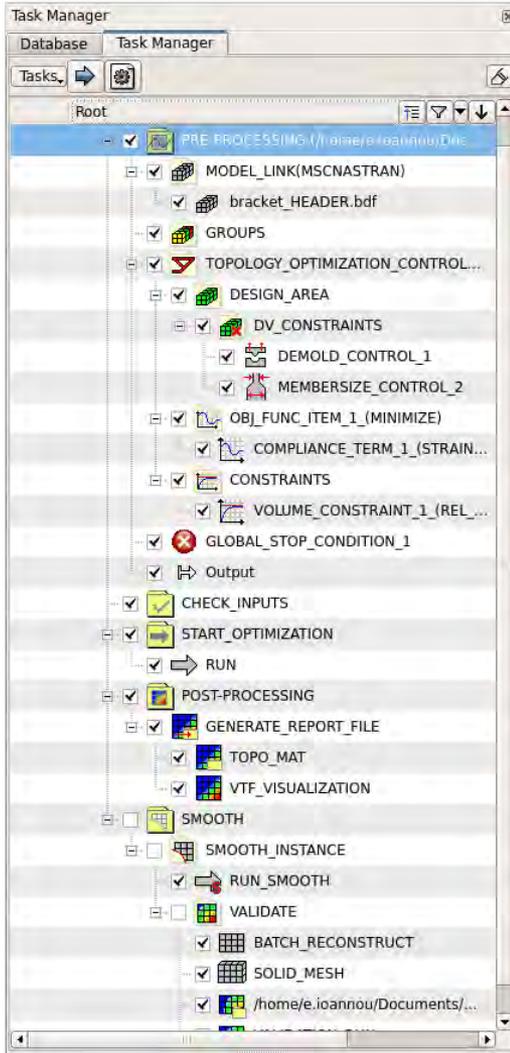


Solid Rib



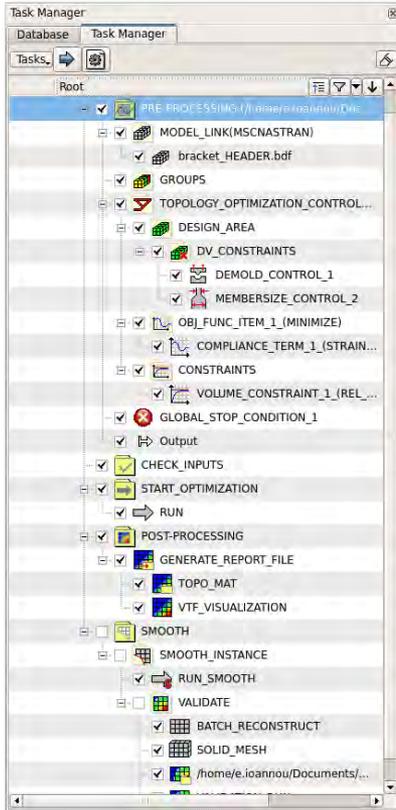
Optimization

Topology Optimization: TOSCA



Optimization

Topology Optimization: NASTRAN SOL 200



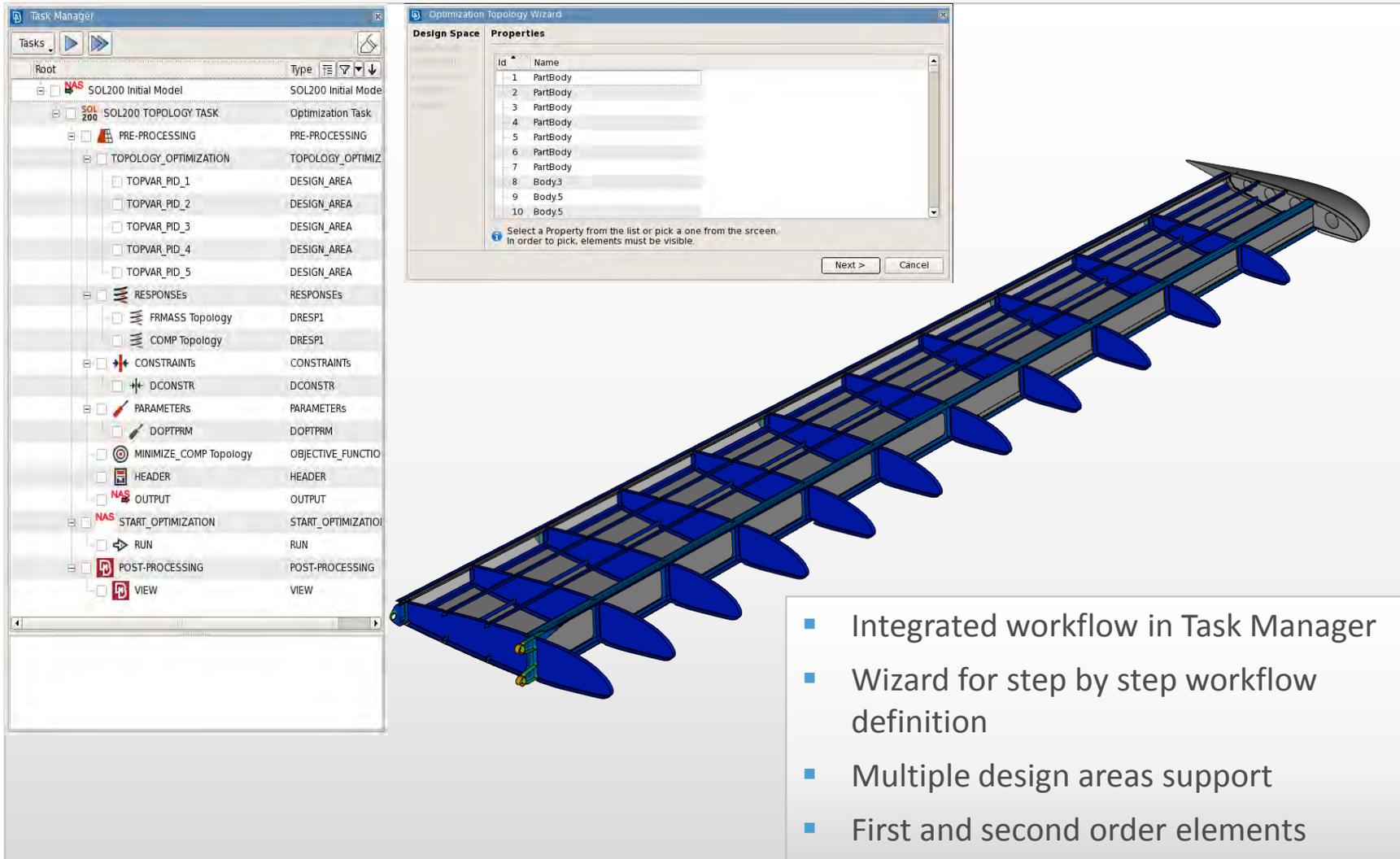
0:Actuator_1.nas : ORIGINAL STATE
1:Door_only_Extd_4.nas : ORIGINAL STATE
2:Goose_Neck_w_Beam_9_200_Subcases_0_all_des.nas : Scalar: Stresses,Signed Von Mises,Max of Top Bottom,Centroid : : Cycle 12 : SUBCASE 1 ::SUBCASE 1



Mass Reduction 60%

Optimization

Topology Optimization: EPILYSIS SOL 200



The image displays a software interface for topology optimization. On the left, the 'Task Manager' window shows a hierarchical tree of tasks for 'SOL200 Initial Model'. The 'SOL200 TOPOLOGY TASK' is expanded, showing sub-tasks like 'PRE-PROCESSING', 'TOPOLOGY_OPTIMIZATION', 'RESPONSES', 'CONSTRAINTS', 'PARAMETERS', 'MINIMIZE_COMP Topology', 'HEADER', 'OUTPUT', 'START_OPTIMIZATION', 'RUN', 'POST-PROCESSING', and 'VIEW'. The 'TOPOLOGY_OPTIMIZATION' task is further detailed with 'TOPVAR_PID_1' through 'TOPVAR_PID_5' (all 'DESIGN_AREA' type) and 'RESPONSES' (FRMASS Topology and COMP Topology, both 'DRESP1').

In the center, the 'Optimization Topology Wizard' dialog box is open. It has two tabs: 'Design Space' and 'Properties'. The 'Properties' tab is active, showing a table with 10 rows:

Id	Name
1	PartBody
2	PartBody
3	PartBody
4	PartBody
5	PartBody
6	PartBody
7	PartBody
8	Body3
9	Body5
10	Body5

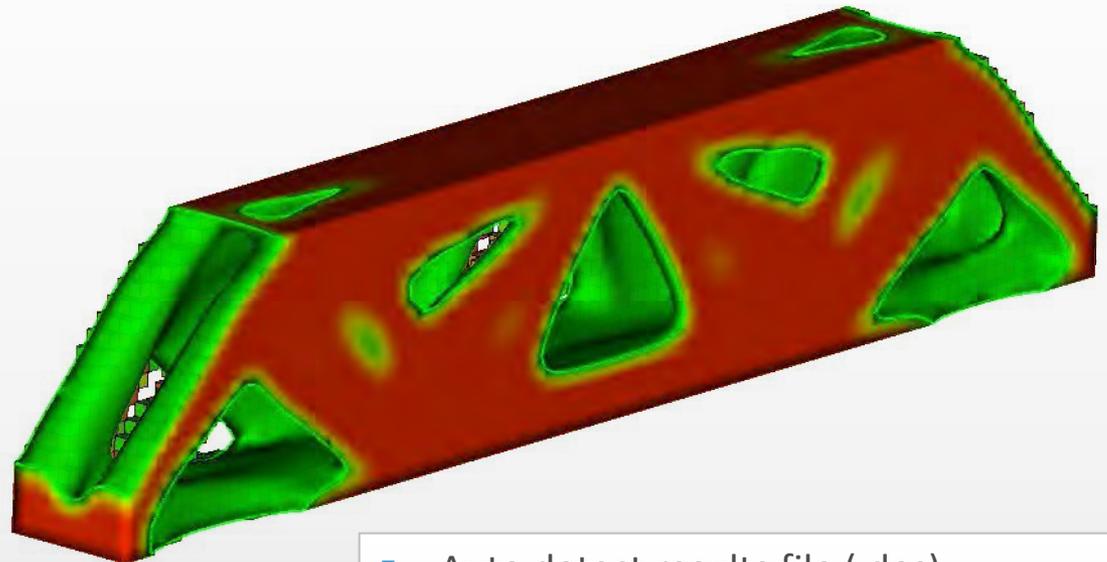
Below the table, there is a note: "Select a Property from the list or pick a one from the screen. In order to pick, elements must be visible." and buttons for 'Next >' and 'Cancel'.

On the right, a 3D model of a wing structure is shown, with several blue rectangular design areas highlighted on its surface.

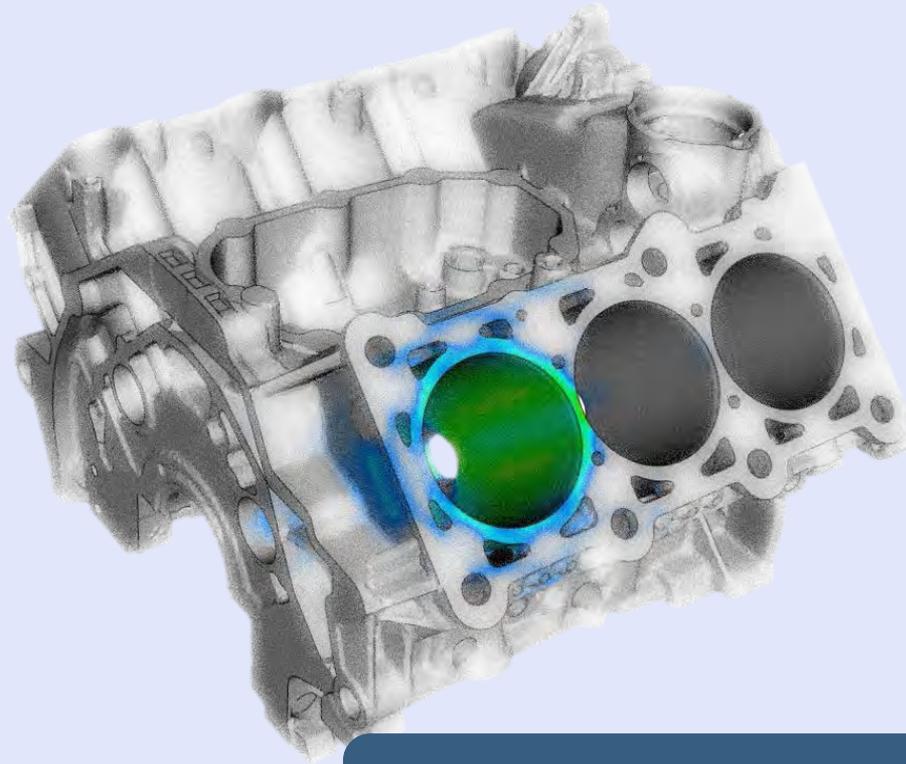
- Integrated workflow in Task Manager
- Wizard for step by step workflow definition
- Multiple design areas support
- First and second order elements

Optimization

Topology Optimization: META Tool



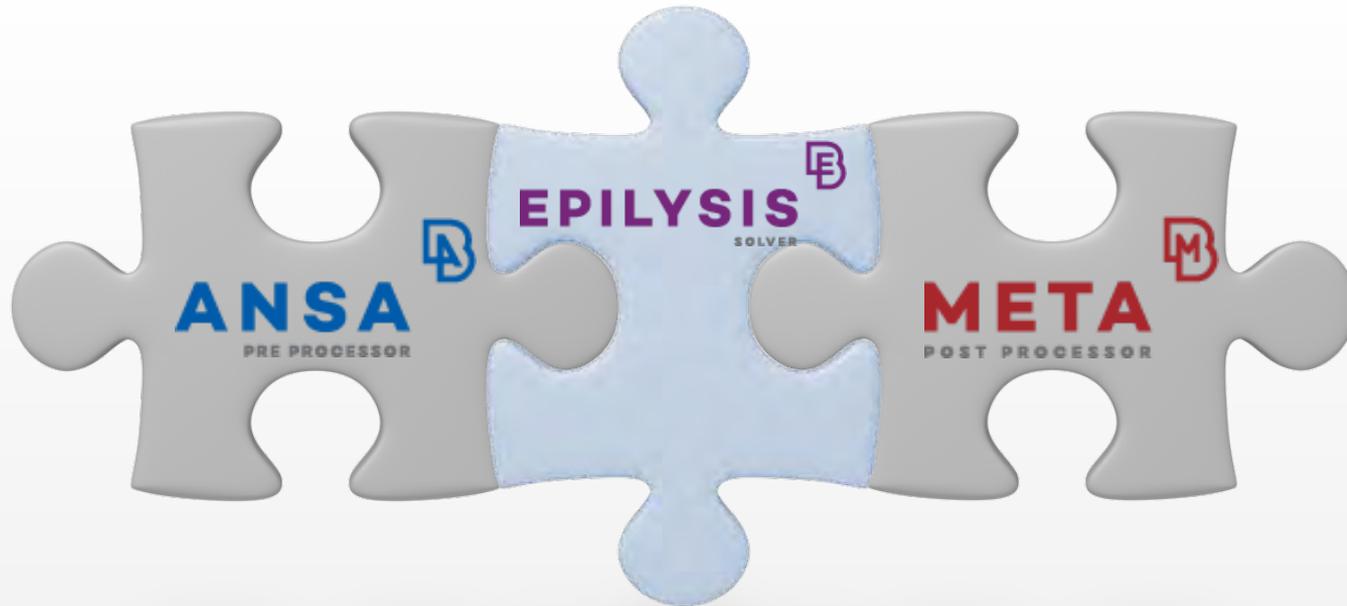
- Auto detect results file (.des)
- Several View Modes
- Creates smoothed surface from the optimized area
- Exports the surface in neutral format



EPILYSIS

EPILYSIS

Solver for FE analysis



- General-purpose FE Analysis Software
- One Solver for many engineering problems
- Standalone & integrated in ANSA

Επίλυσις

[epi-li-sis]

noun 1. solution

EPILYSIS

EPILYSIS Characteristics

Programming Language

- C++
- Double Precision

Quality

- Benchmarks accurate (NAFEMS, SFM and other – More than 3500 tests)
- Robustness
- Repeatability

Performance

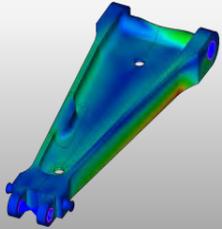
- Shared Memory Parallel Processing (SMP)
- Automatic utilization of system resources without time consuming tuning
- Multiple high-performance direct sparse linear solvers

Easy to Use

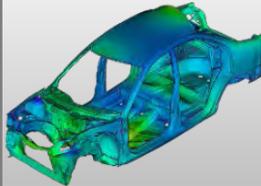
EPILYSIS

Linear analyses

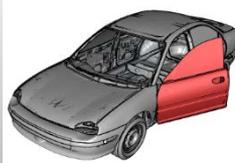
Static



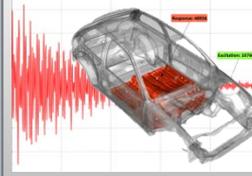
Real Eigenvalue



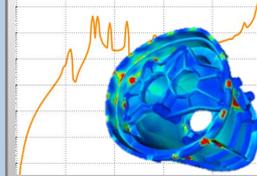
Direct Transient Response



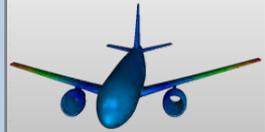
Direct Frequency Response



Modal Transient Response

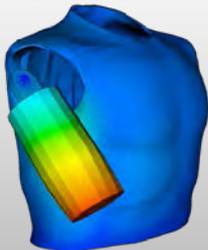


Modal Frequency Response



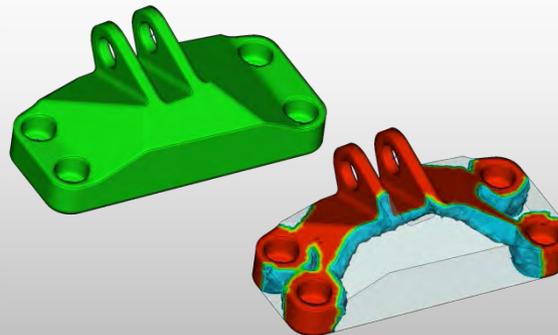
Non - linear analyses

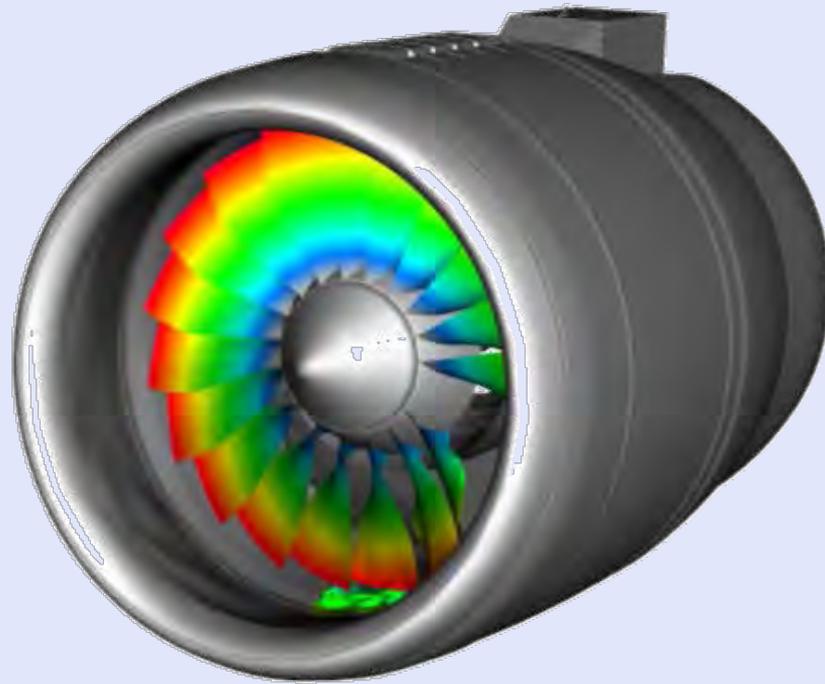
Static: linear elements / materials -non linear contacts



Optimization

Topology optimization

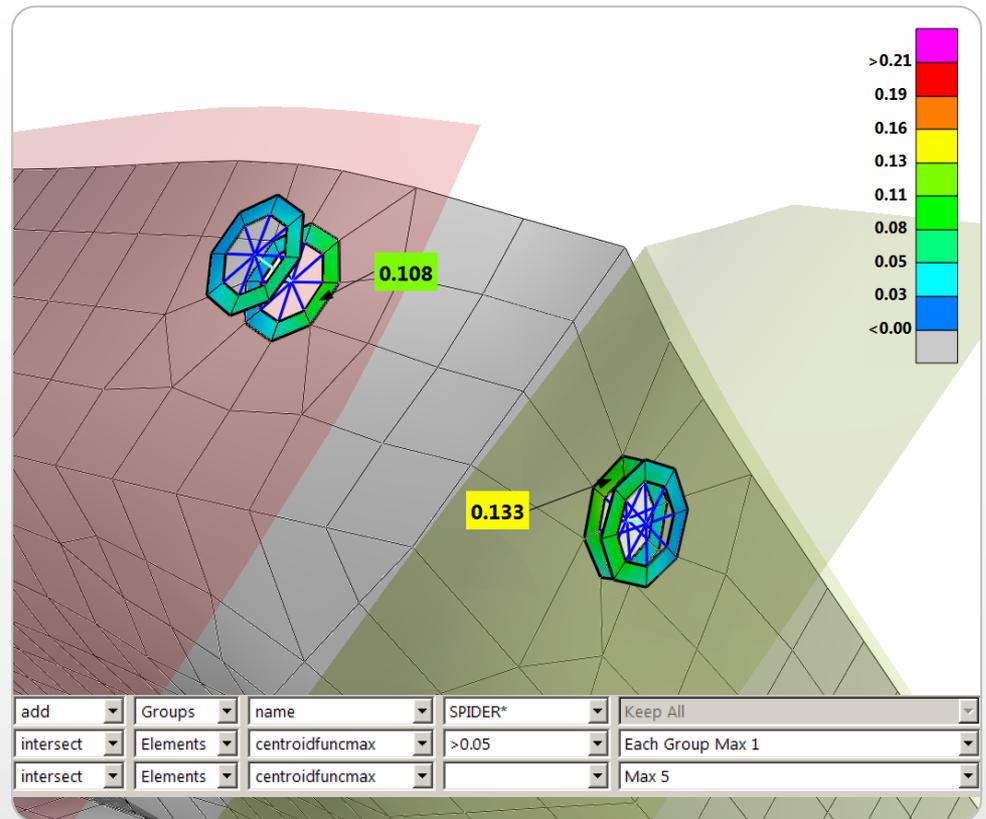
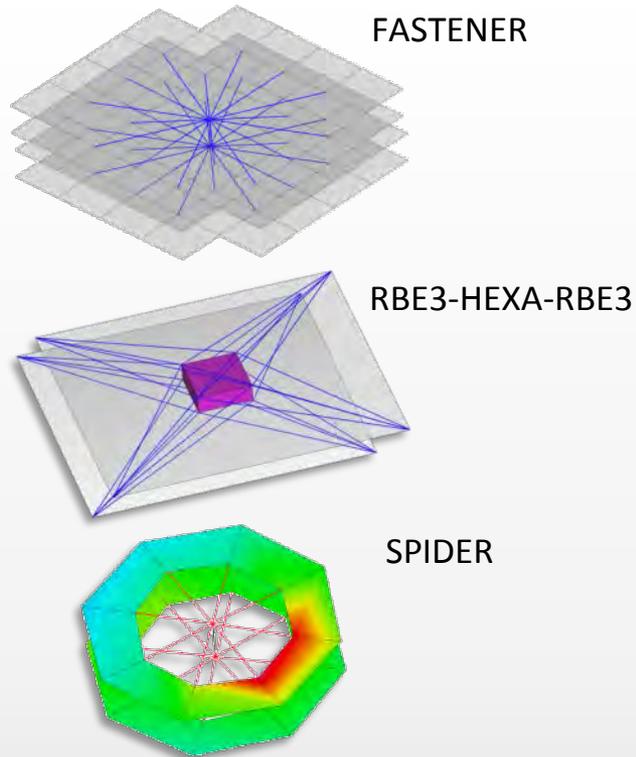




META TOOLS

META Tools

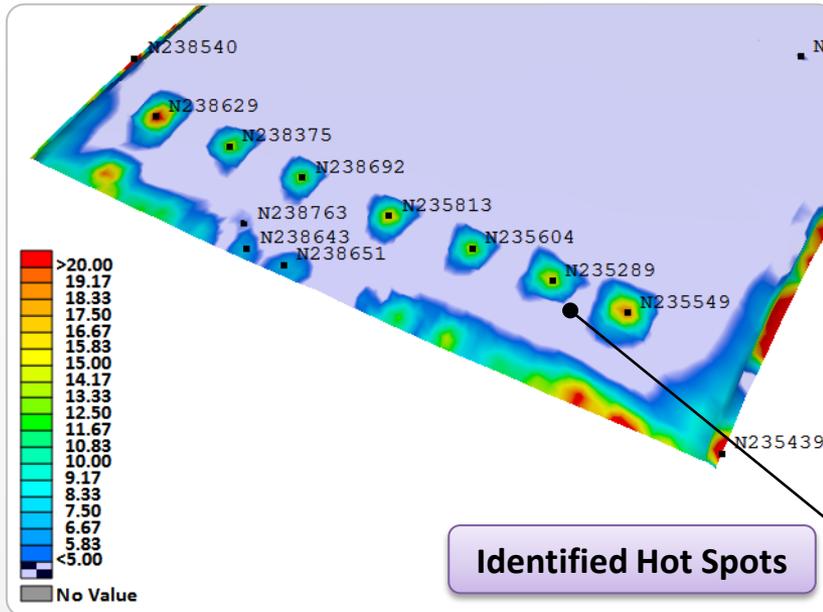
Connections management & spotweld analysis



- Connection information read from ANSA comments
- Connections grouped by type
- Identification of the critical element per connection with Advanced Filter

META Tools

Hot Spots identification



Current State:Stresses,Von Mises,Max of Top Bottom : SUBCASE 1 ::TORSION: SUBCASE :

Id :: C0	Max :: C1	Mid :: C9	Type :: C10
238692	15.3112	1218	NODE
240526	14.8633	1218	NODE
237836	14.535	1218	NODE
240621	13.0989	1218	NODE
240529	11.2032	1218	NODE
240588	9.8942	1218	NODE
240640	7.93867	1218	NODE
238651	7.47679	1218	NODE
238643	7.43315	1218	NODE
237472	6.03507	1218	NODE
236696	5.75861	1218	NODE
238763	5.17451	1218	NODE
Sum (...)	117.722		

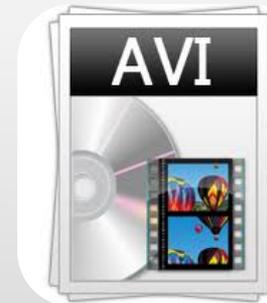
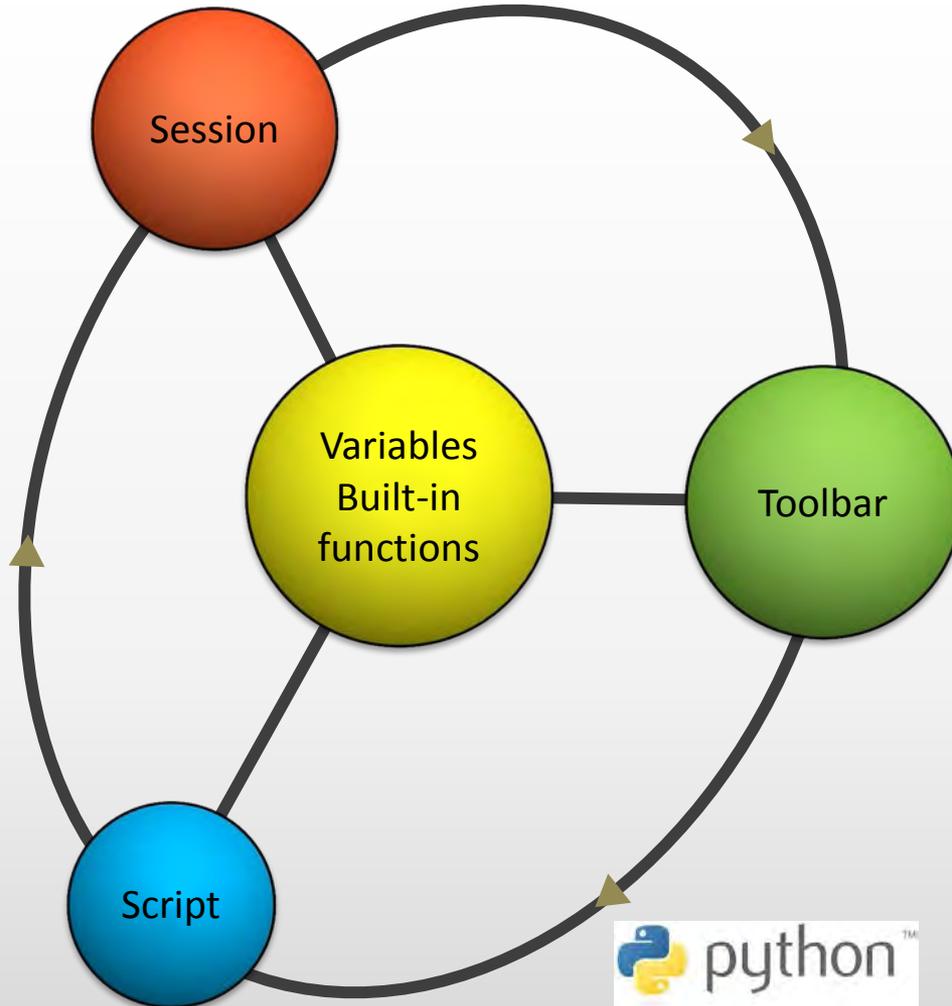
Hot Spots in Statistics Table

All Invert Visible Pick Filtering:

Model 0 Function All All Entities

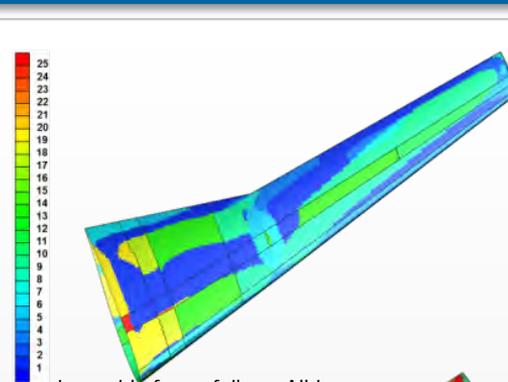
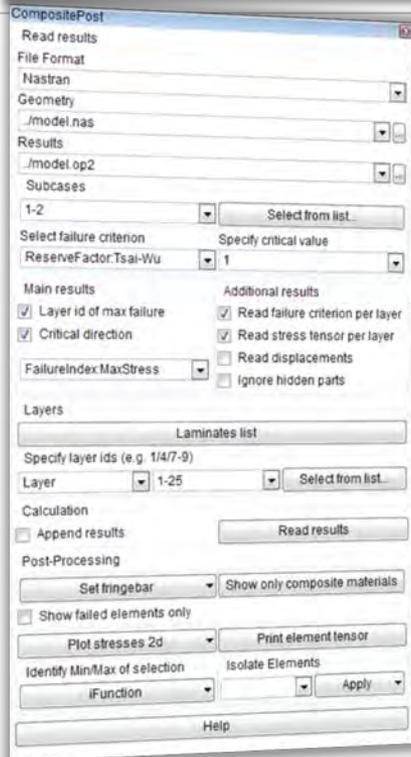
META Tools

Automation tools – Reporting

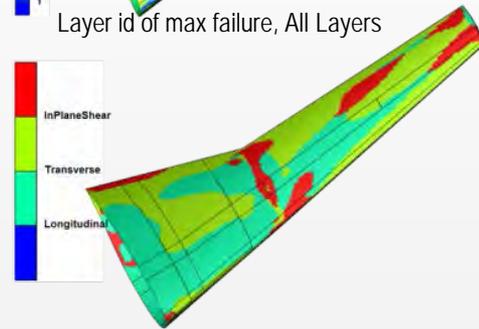


META Tools

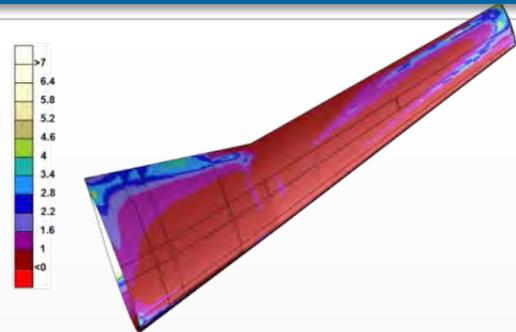
Composite Post toolbar



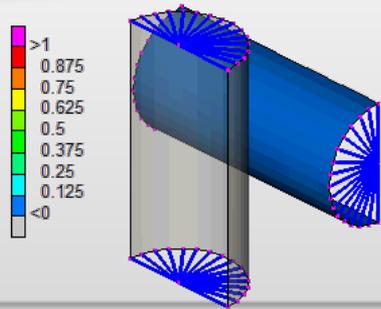
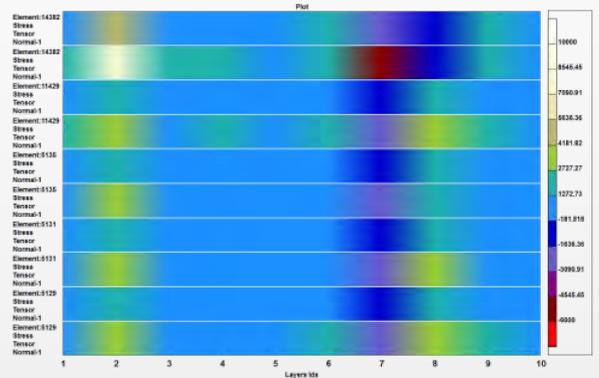
Layer id of max failure, All Layers



Critical direction, FailureIndex:MaxStress, Layer:1



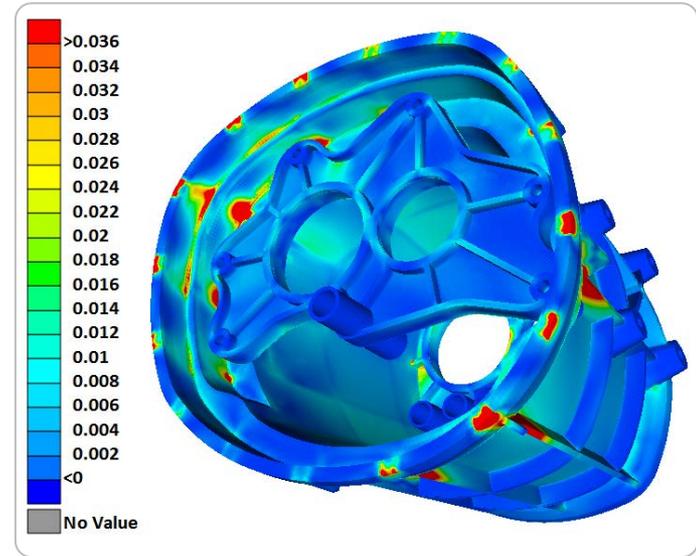
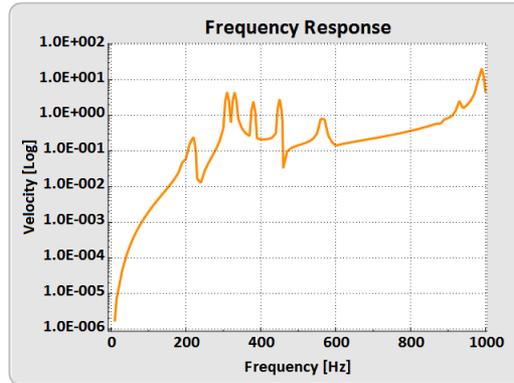
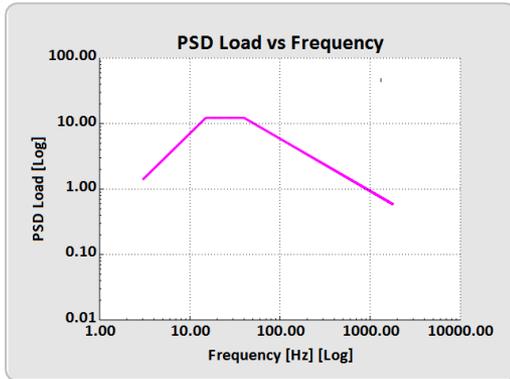
CompositeStresses, ReserveFactor:Tsai-Wu, MinofAllLayers



- Calculation of Failure criteria - Reserve Factor - Effort (Tsai-Wu, Puck, ...)
- Identification of Layer id with max failure
- Identification of Critical direction
- Show failed elements only
- Curves plot of stress tensor results through thickness
- Re-evaluation of failure criteria by modifying materials' properties

META Tools

Random Response



Random Response loadcase
(including Coupled Excitations)

Frequency Response analysis

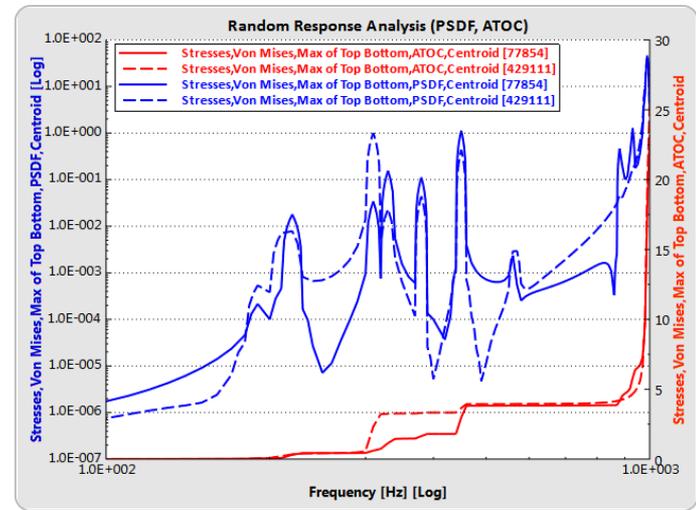
Correlation Matrix of Subcases

$$\begin{bmatrix} 1 & \dots & 0,003 \\ \vdots & \ddots & \vdots \\ 0,003 & \dots & 1 \end{bmatrix}$$

Calculations for Nastran:

- Von Mises (PSDF, ATOC, NO, RMS) and other...

Stresses, Von Mises, Max of Top Bottom, PSDF, Centroid @ 340 Hz

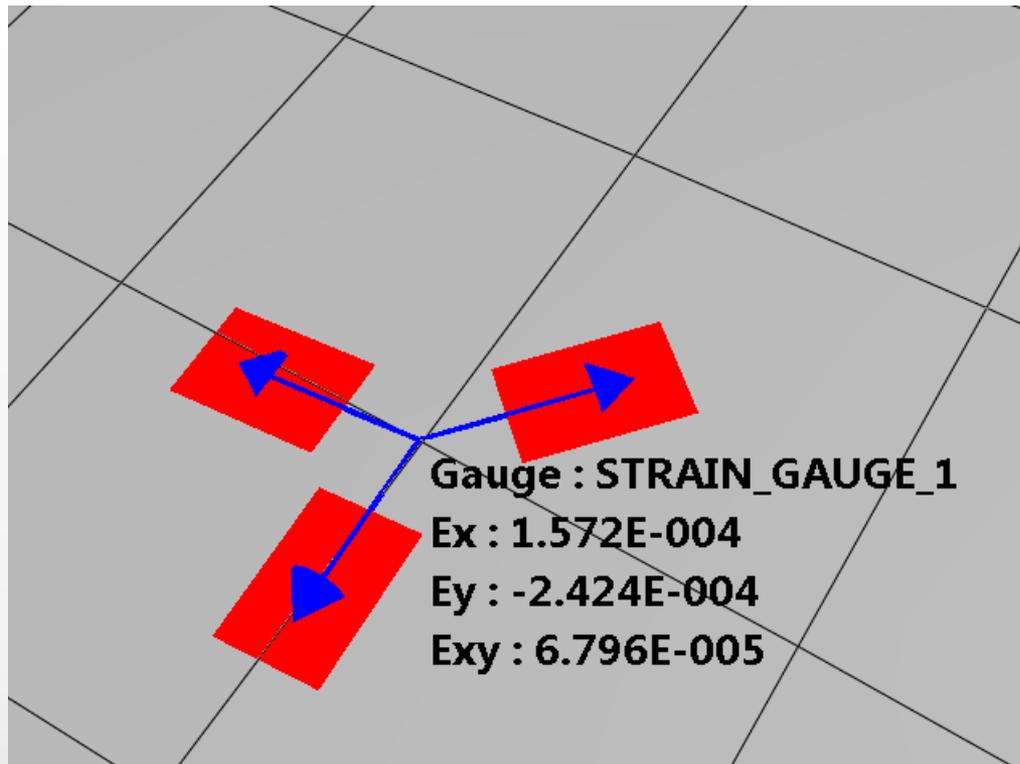


Random Response results

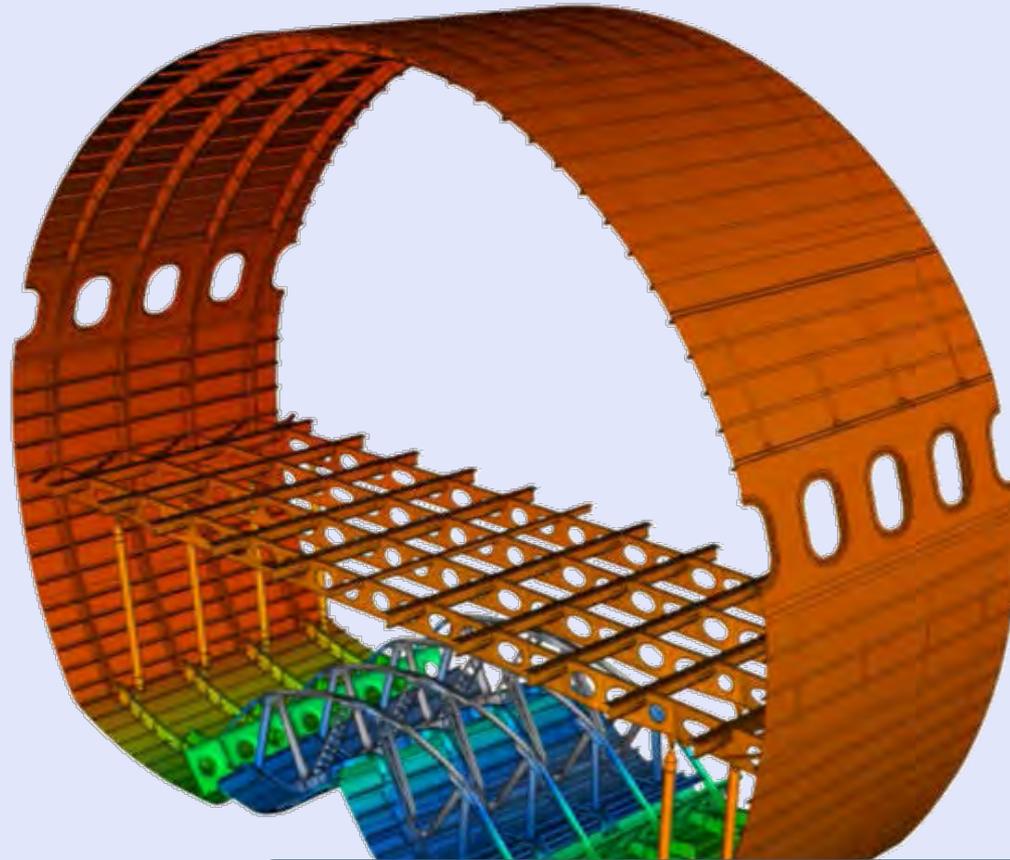


META Tools

Strain Gauges

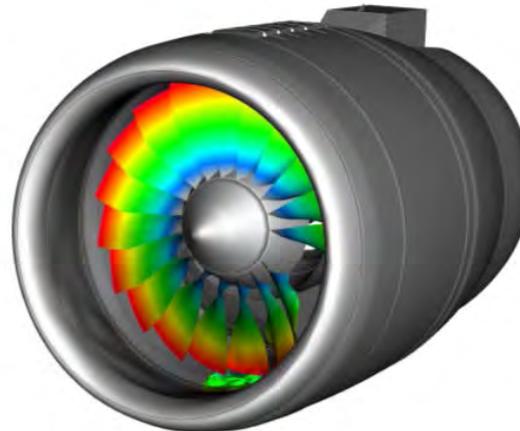
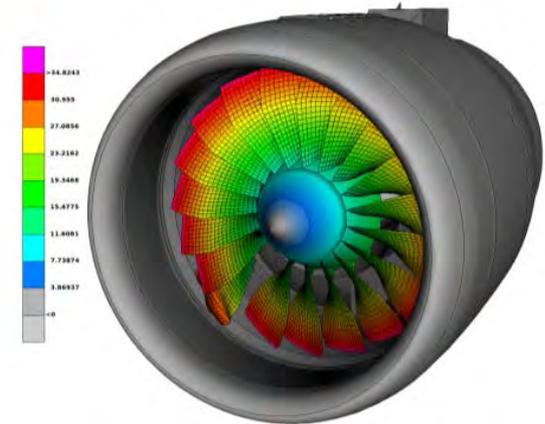
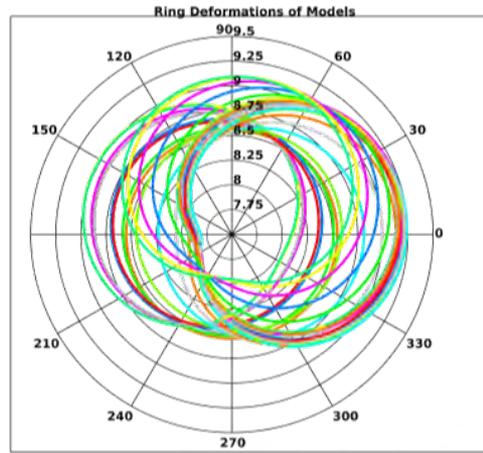


- Tool to Create/Edit/View strain gauges
- Calculate Strain results
- Correlate physical tests with FE calculations



CASE STUDIES

Case Study - Fan Blade Out and Rotor Unbalance Simulation



Fan Blade Out (FBO):

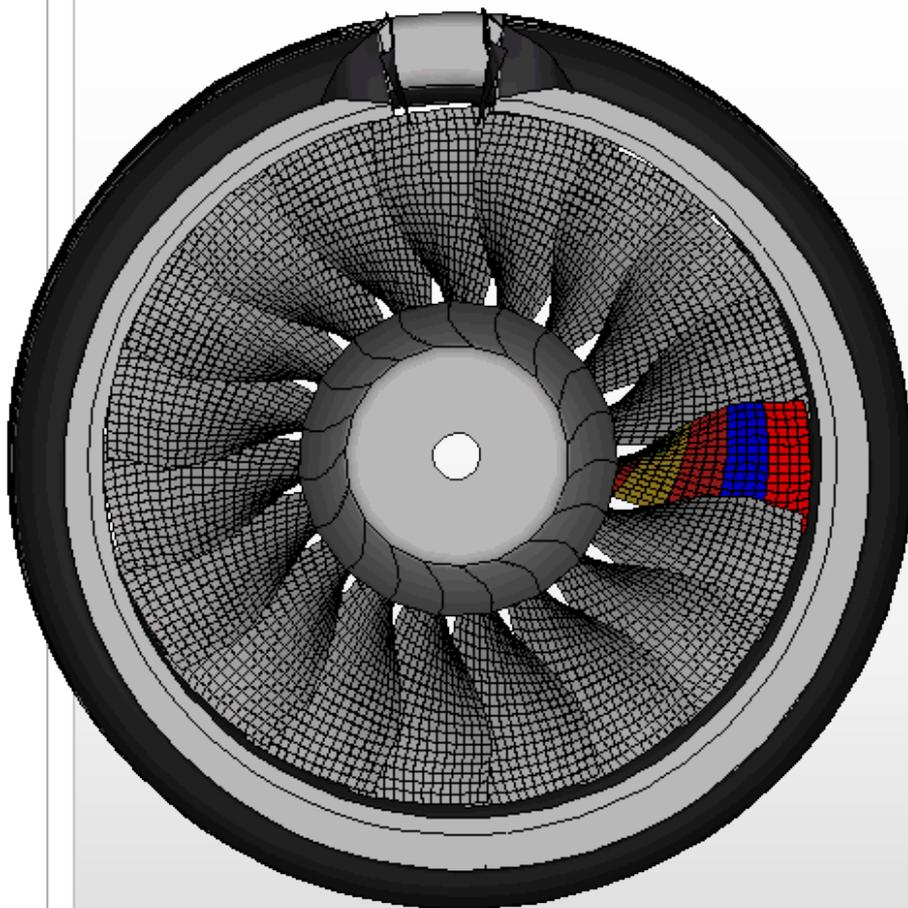
MD NASTRAN 2010 Transient Explicit Dynamic Analysis - 64000 Elements

Blade & Rotor (Titanium): Elastic-Plastic Material

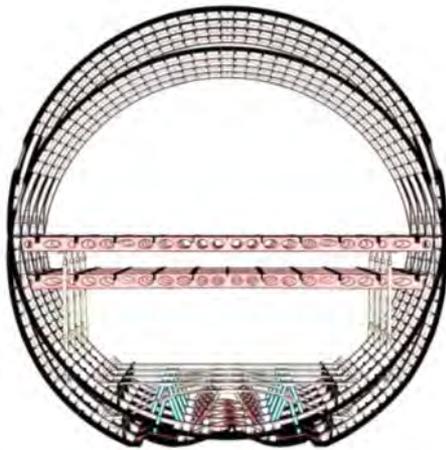
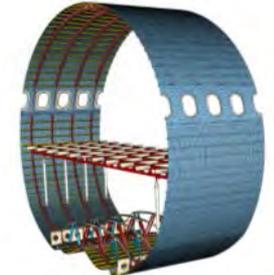
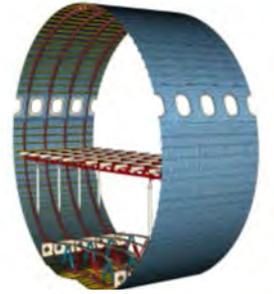
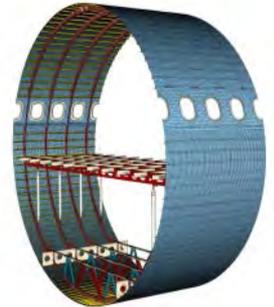
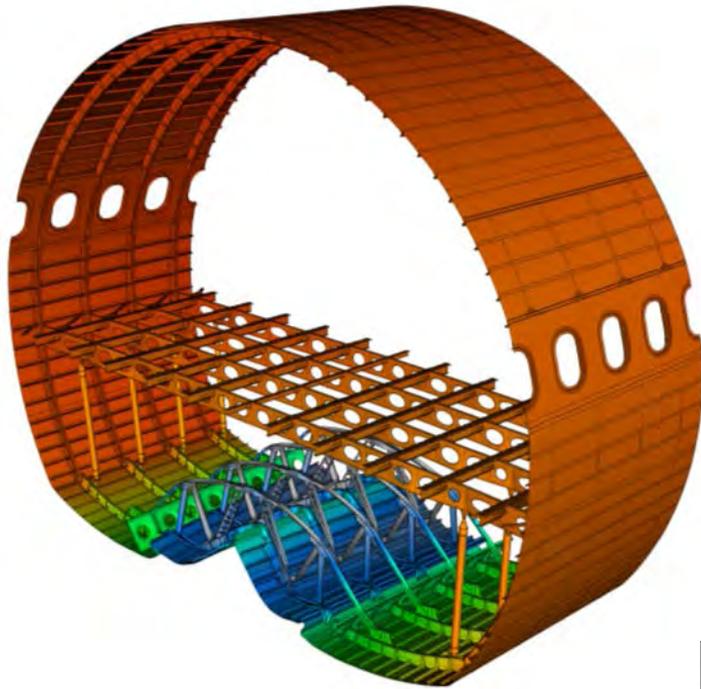
Case: Elastic-Plastic material

Case Study - Fan Blade Out and Rotor Unbalance Simulation

Results - Video Synchronization

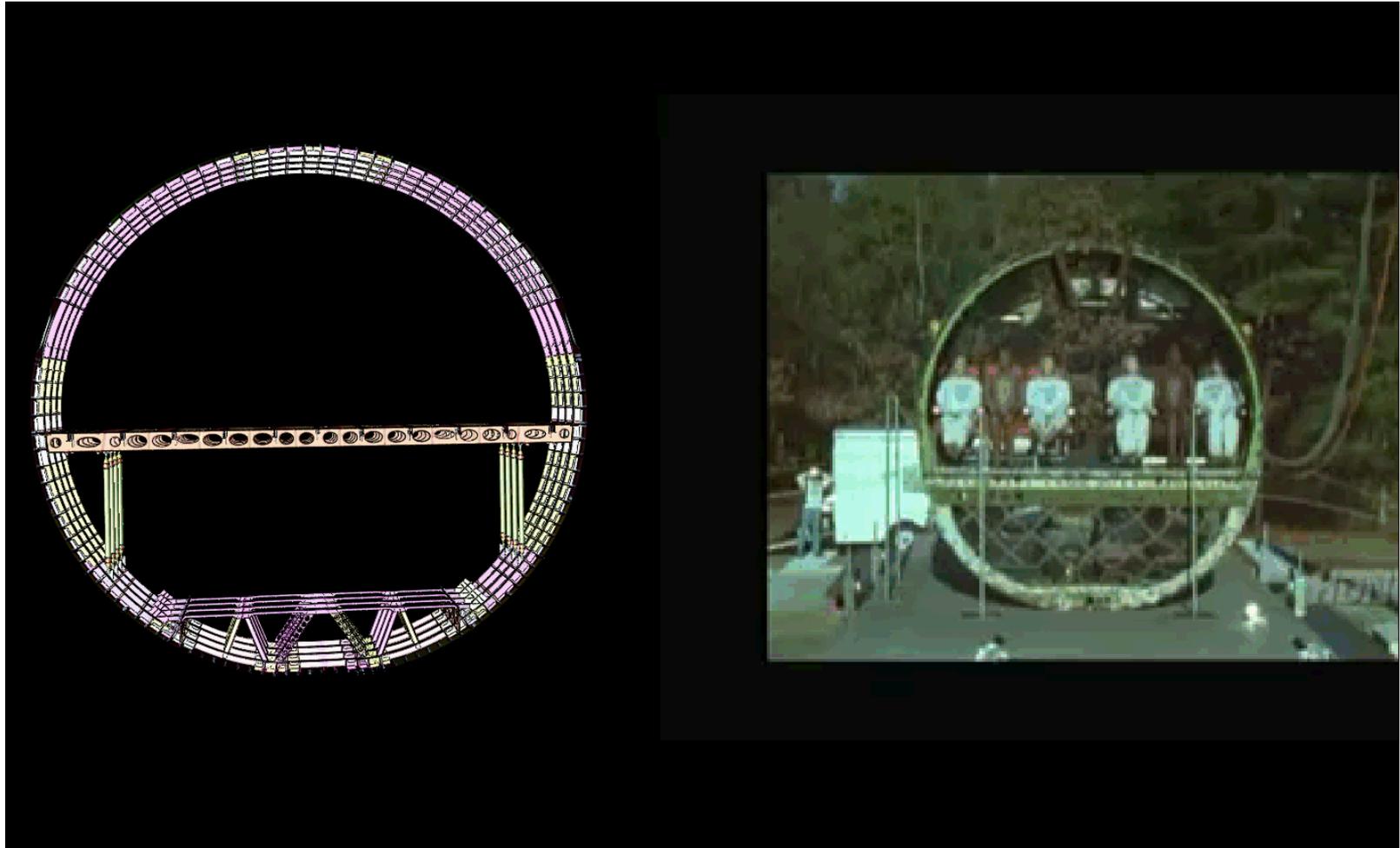


Case Study - Fuselage central section vertical drop test



Case Study - Fuselage central section vertical drop test

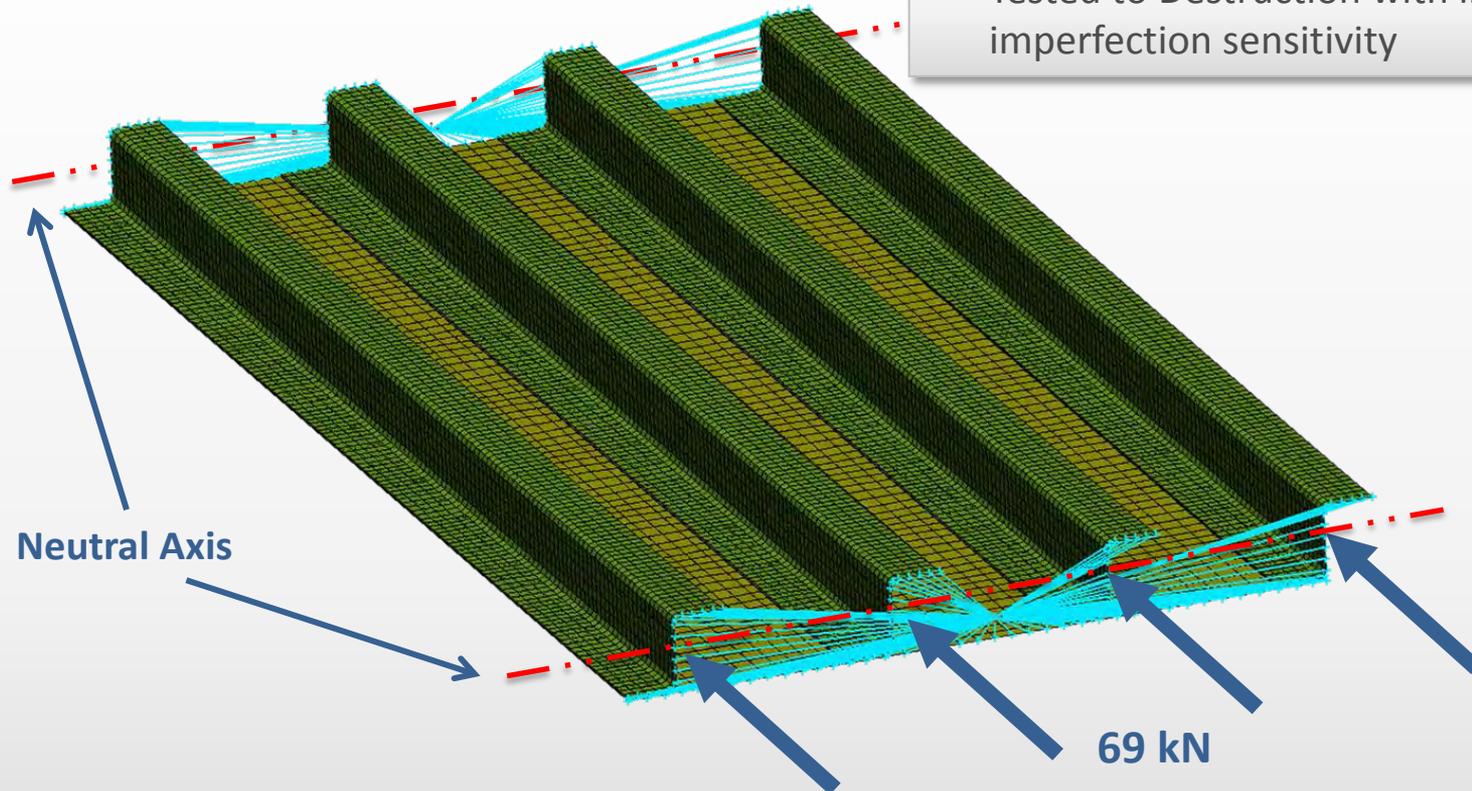
Results - Video Synchronization



Case Studies - Buckling

Post Buckling performance of a stiffened panel

- Design for Post Buckling performance
- Tested to Destruction with initial imperfection sensitivity

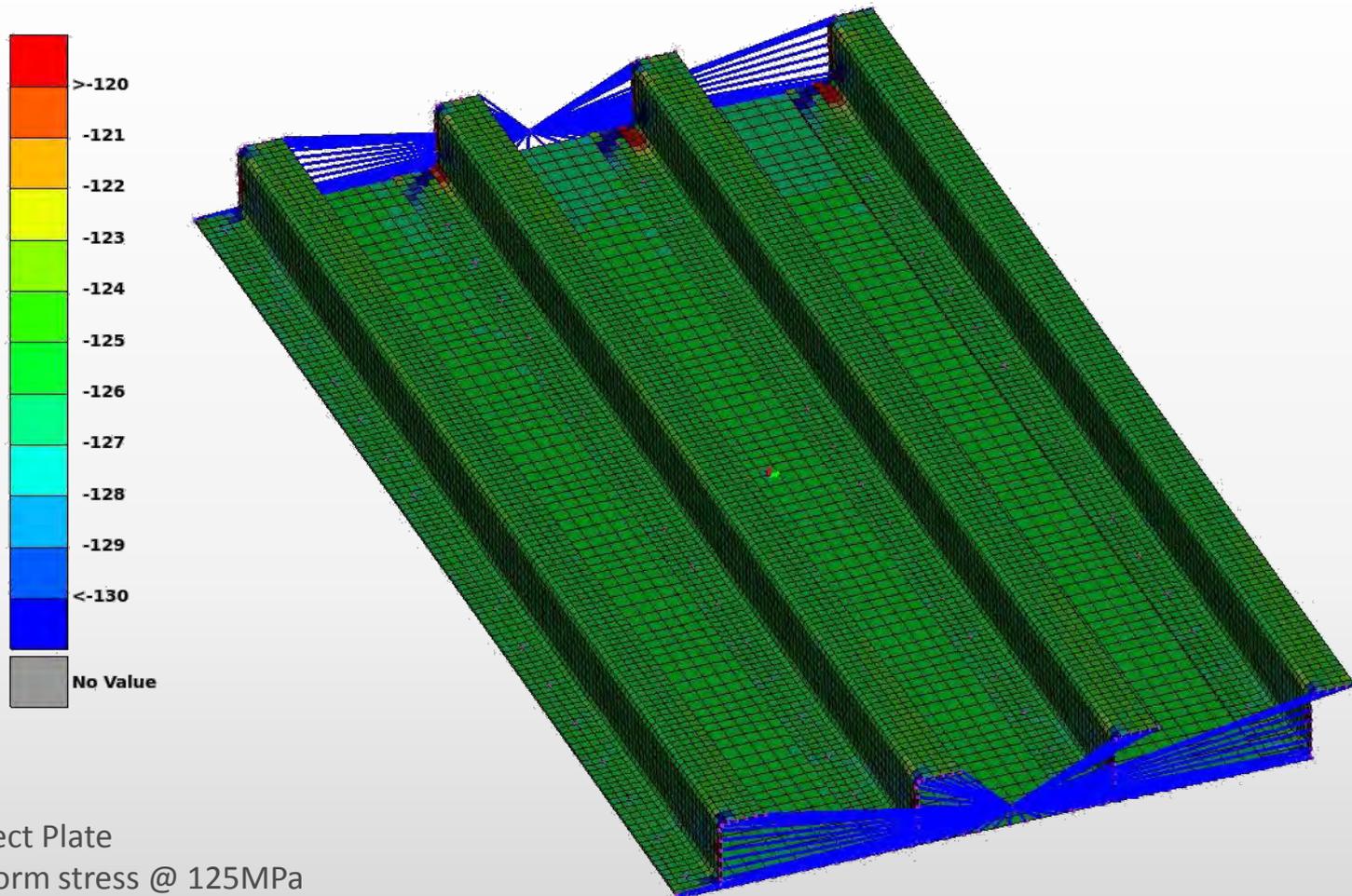


Source: Lillico, M. et al, 2000, "Optimum Design and testing of a postbuckled stiffened panel". 41st AIAA conference .

Case Studies - Buckling

Linear static results in META

0:buckl_Force_tied_RBE2_improved_2_4@COG-6.nas : Scalar: Stresses,Signed Von Mises,Max of Top Bottom : : Scale Factor 1.000E+01 : SUBCASE 1
CASE SUBCASE 1

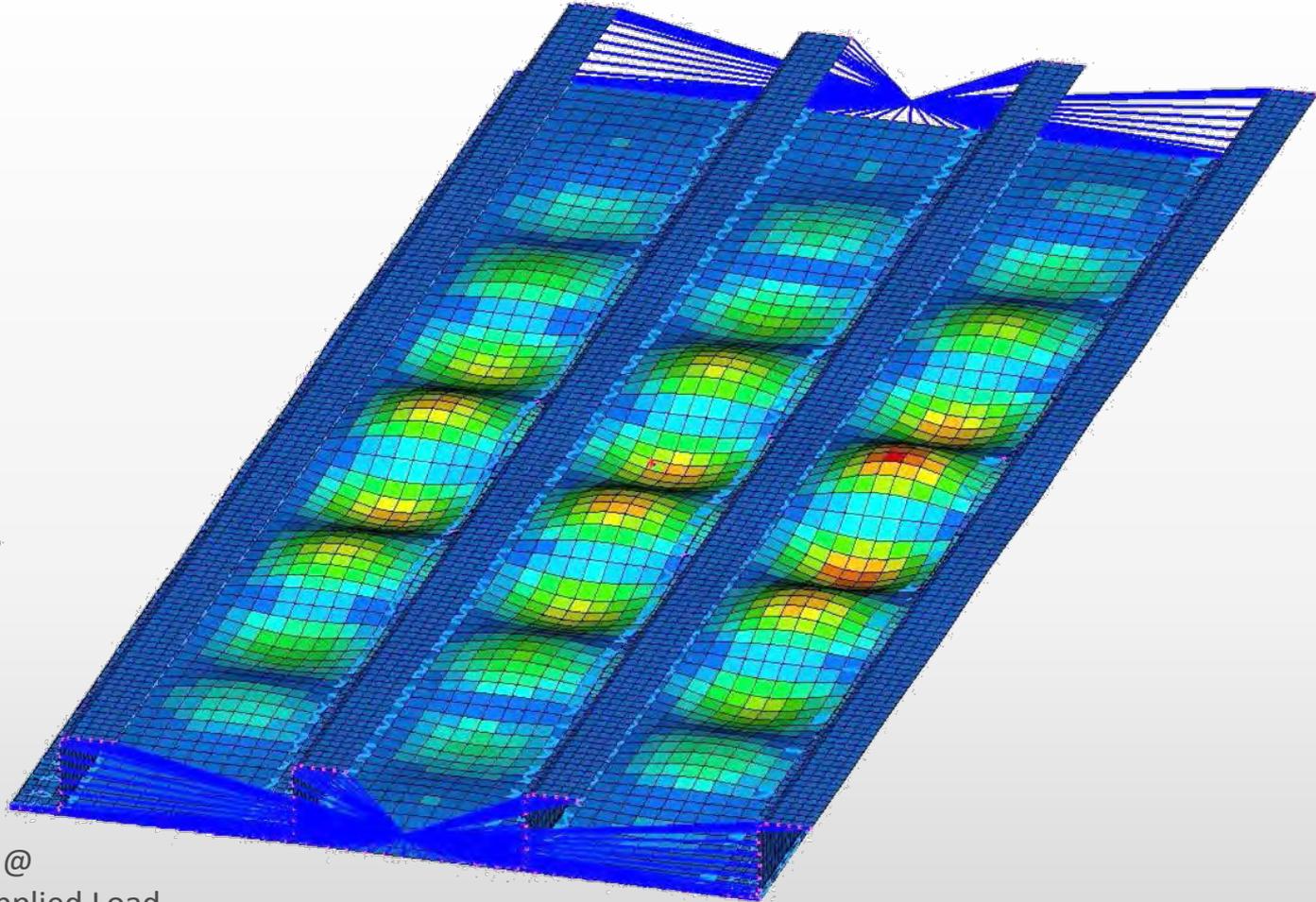
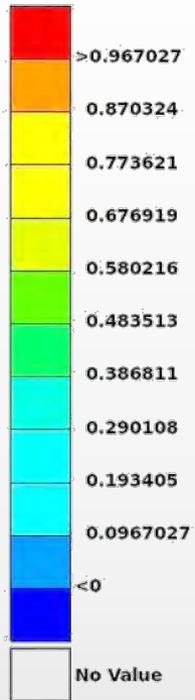


- Perfect Plate
- Uniform stress @ 125MPa

Case Studies - Buckling

Buckling results in META

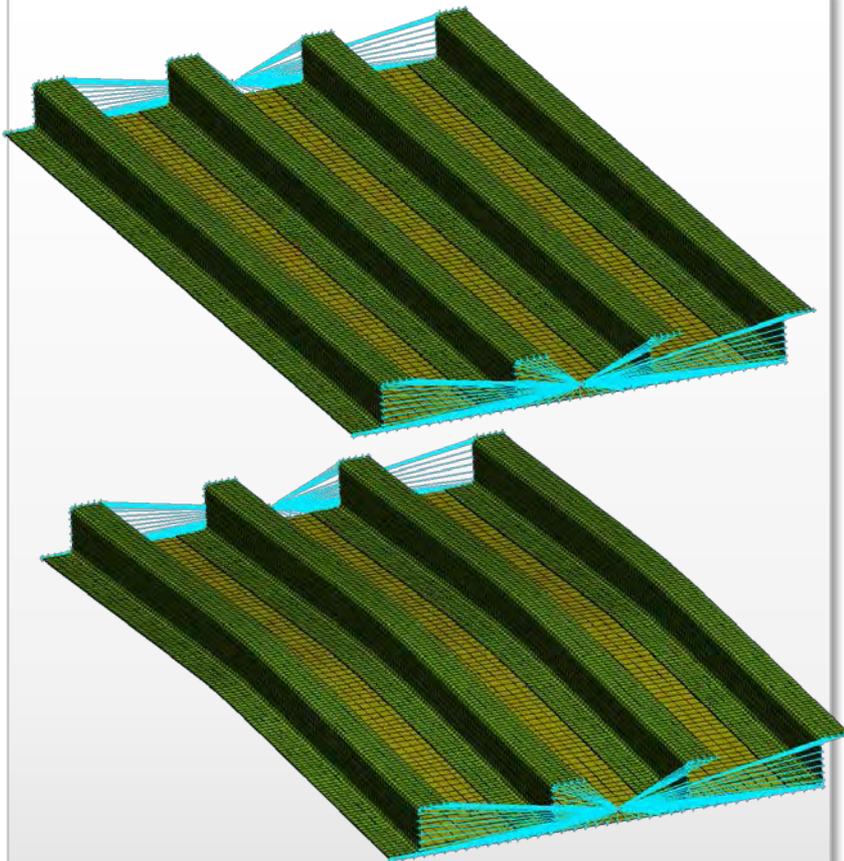
0:buckl_Force_tied_RBE2_improved_2_5@COG-6.nas : Scalar: Eigenvectors,Translational,Magnitude#1 : : Scale Factor 1.000E+01 : SUBCASE 2 ::MODE 1
,FREQUENCY 1.396500E-01 ,EIGENVALUE 7.699131E-01 ::BUCKLING CASE SUBCASE 2



- 1st Mode @
- 77% of Applied Load

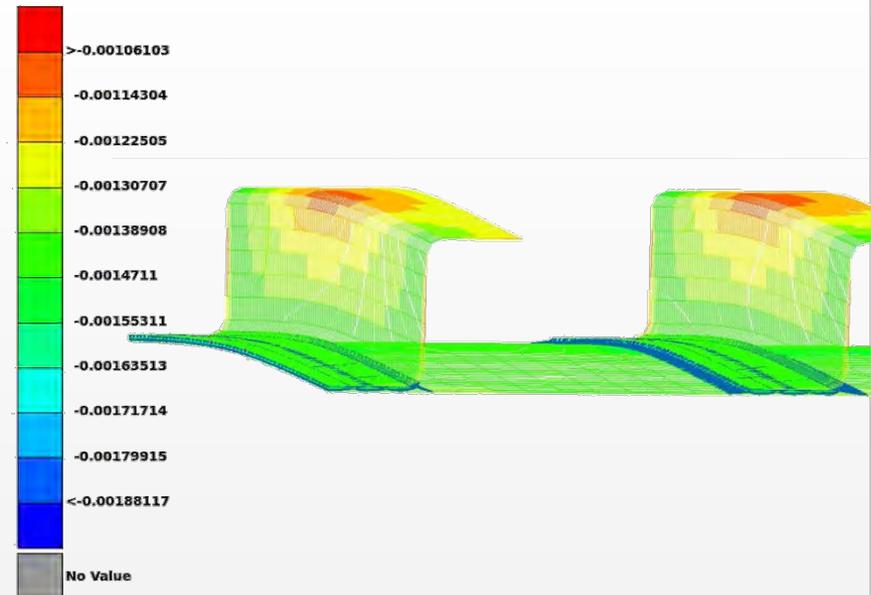
Case Studies - Buckling

Defining eccentricity using Morph



0.2% Eccentricity created using the Morphing Tool which is required for initial imperfection

Eccentricity - Results

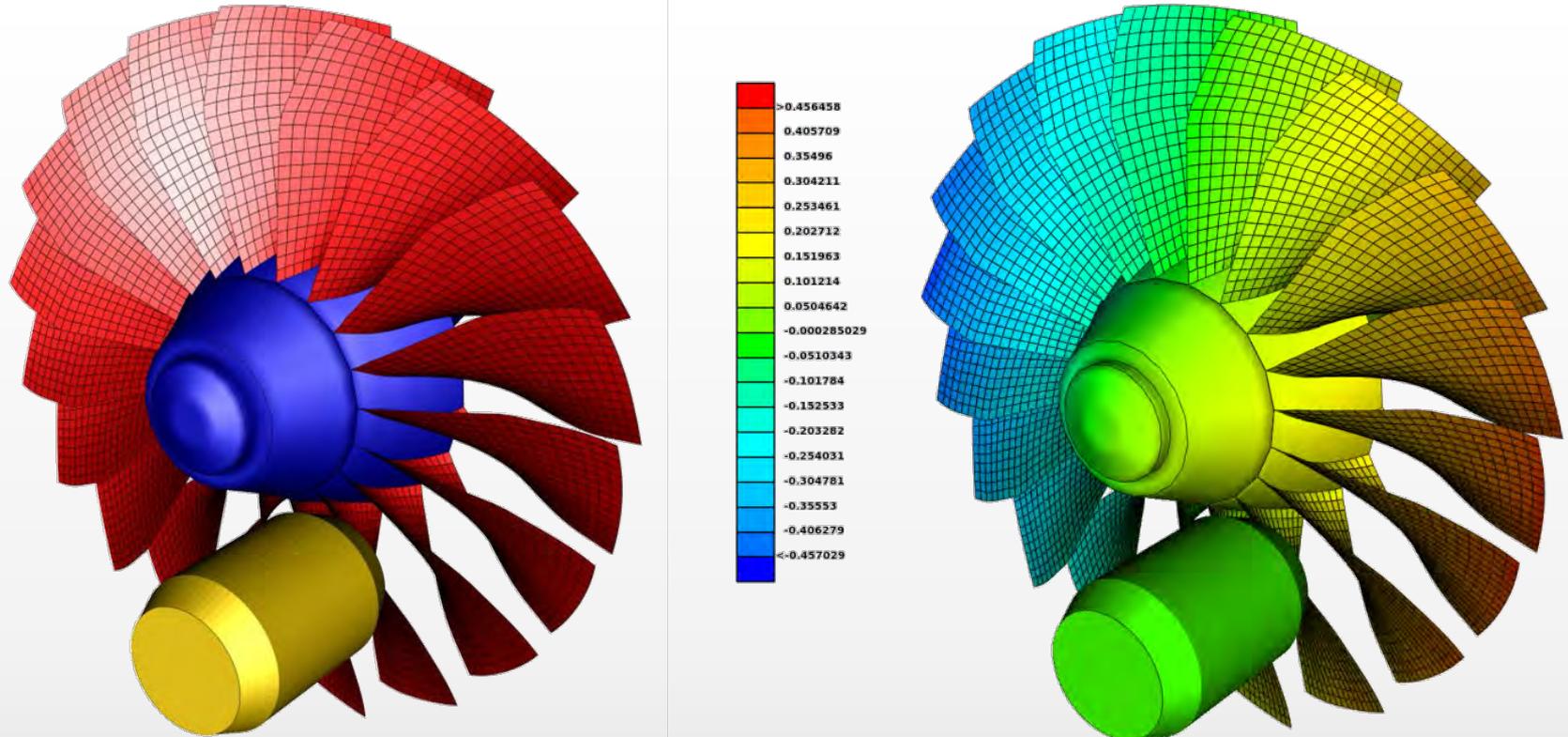


Experiments:

- Experiment 1 : - length/500 = -1.078mm (sagging) EigVal: + 5.6 %
- Experiment 2 : Perfect Panel
- **Experiment 3 : + length/500 = 1.078mm (hogging) EigVal: - 5.9 %**

Case Studies - Bird Impact Simulation

FE model and results in META



Bird Strike:

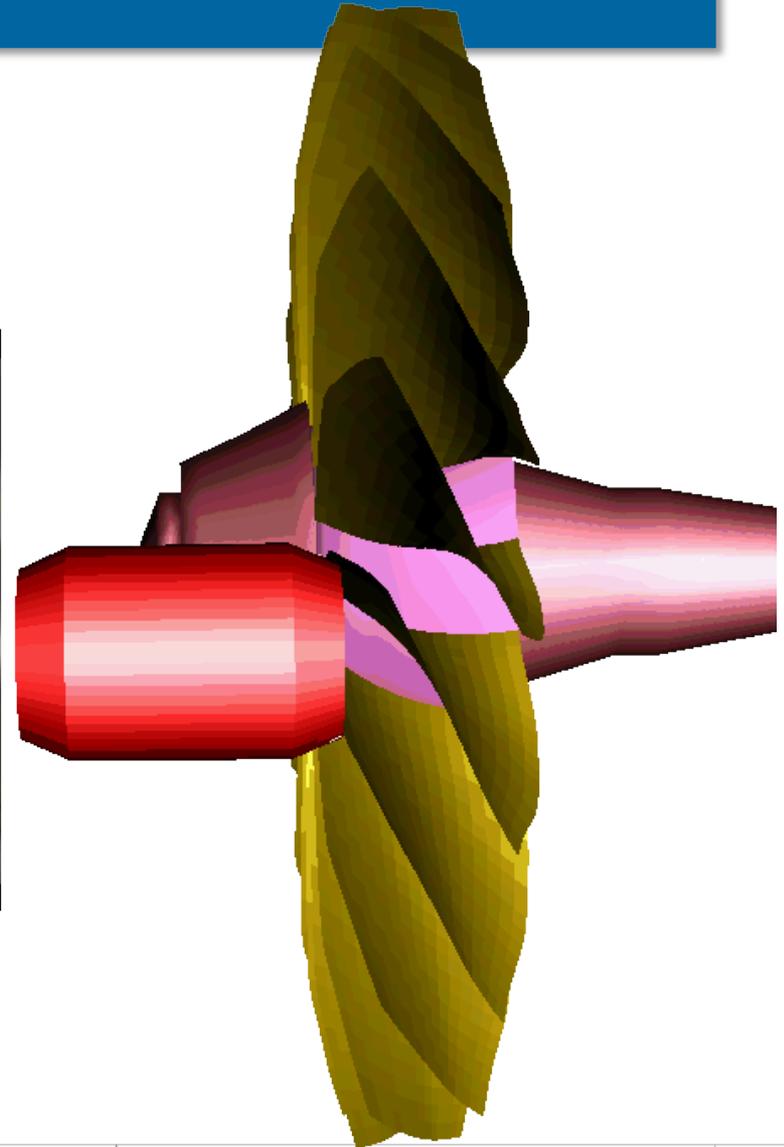
MD NASTRAN 2010 Transient Explicit Dynamic Analysis - 19000 Elements

Blades: Piecewise linear plastic material (MATD024)

Bird: Elastic-Plastic hydrodynamic Material (MATD010) – 10752 Solid Elements

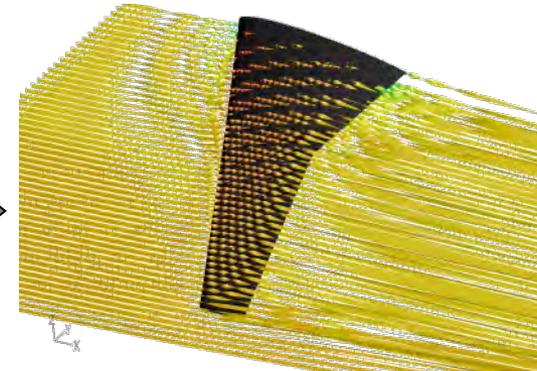
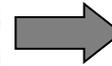
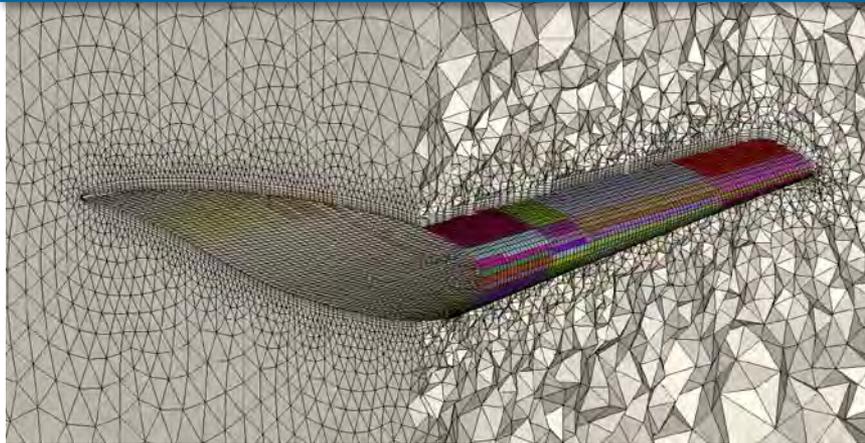
Case Studies - Bird Impact Simulation

Video synchronization

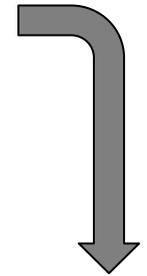


Case Study – Results mapping

Mapping of External Aero results as B.C. for Static Analysis



External Aerodynamics analysis in **Fluent**



External Aerodynamics Pre-Processing in **ANSA**

Wing Deformation in **META**

NASTRAN SOL 101

This block shows the mapping of pressure field results to the wing composite model in ANSA. It features a 3D model of the wing with a color gradient representing the pressure field. A vertical color scale on the left ranges from -224.171 (blue) to 2453.31 (red). A blue box labeled 'Wing Deformation in META' is positioned below the wing model. A blue box labeled 'NASTRAN SOL 101' is positioned to the right of the wing model. A grey arrow points from the wing model to the NASTRAN SOL 101 box. A smaller inset image shows a cross-section of the wing with a mesh.

Mapping of Pressure Field to Wing Composite Model in **ANSA**

This block shows the mapping of pressure field results to the wing composite model in ANSA. It features a 3D model of the wing with a color gradient representing the pressure field. A vertical color scale on the right ranges from 0.1 (blue) to 0.58922 (red). A blue box labeled 'Mapping of Pressure Field to Wing Composite Model in ANSA' is positioned below the wing model. A grey arrow points from the wing model to the NASTRAN SOL 101 box.

Thank you

Stay connected

website

www.beta-cae.com

support

ansa@beta-cae.com

social media

