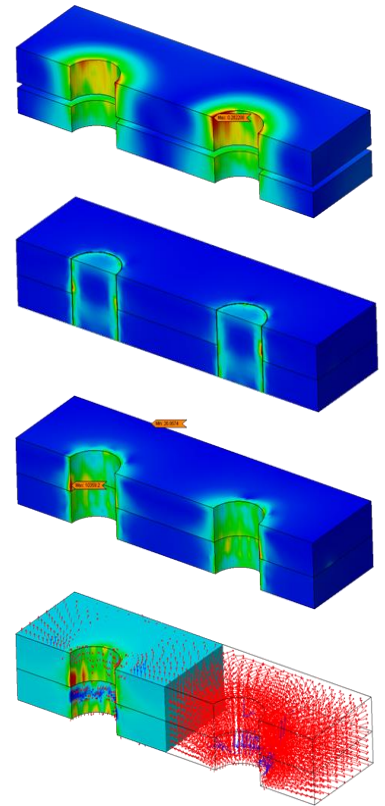


# Introduction to midas FEA NX

---

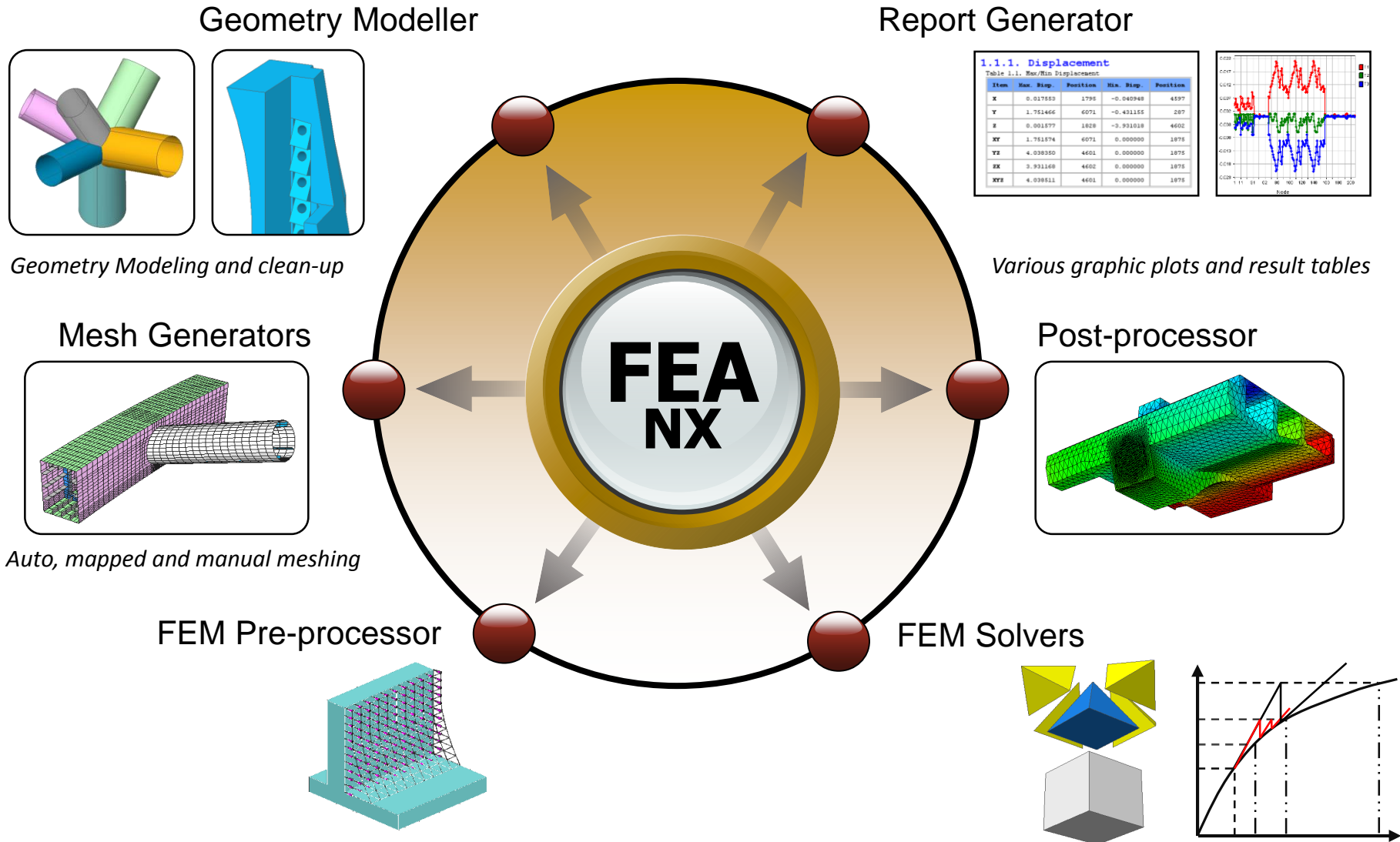
New Paradigm in Advanced Structural Analysis



# Contents

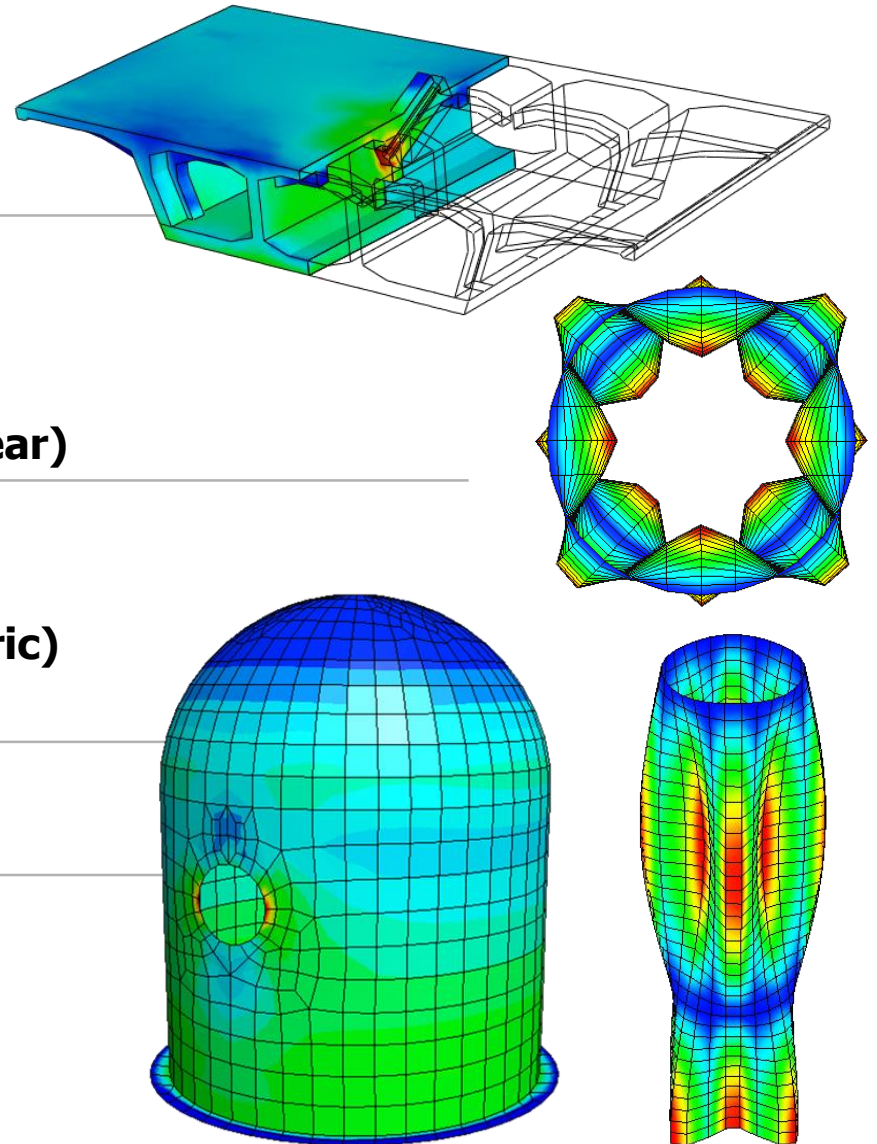
1. Overview
2. Geometry Modeling
3. Mesh Generation
4. Analysis
5. Post-processing

# Configuration



# Analysis Capabilities

- **Static Analysis**
- **Construction Stage Analysis**
- **Reinforcement Analysis**
- **Buckling Analysis**
- **Eigenvalue Analysis**
- **Response Spectrum Analysis**
- **Time History Analysis(Linear/Nonlinear)**
- **Static Contact Analysis**
- **Interface Nonlinearity Analysis**
- **Nonlinear Analysis(Material/Geometric)**
- **Concrete Crack Analysis**
- **Heat of Hydration Analysis**
- **Heat Transfer Analysis**
- **Slope Stability Analysis**
- **Seepage Analysis**
- **Consolidation Analysis**
- **Coupled Analysis(Fully/Semi)**





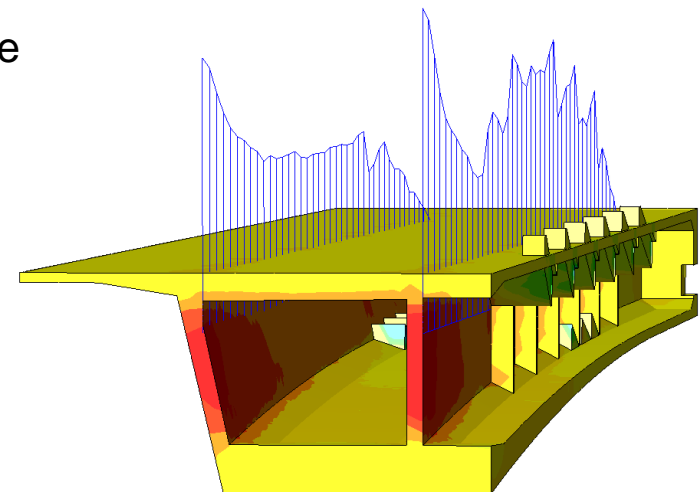
# Applicable Problems

## General Detail Analysis (Linear, Material/Geometry Nonlinear)

- General detail FE analysis (linear static/dynamic analysis of concrete and steel)
- Buckling analysis of steel structure with material and geometric nonlinearity

## Concrete and Reinforcement Nonlinear Analysis

- Detail analysis of composite structure (steel + concrete)
- 3D detail analysis considering steel, concrete and reinforcement simultaneously
- Detail analysis of CFT (Concrete Filled Tube) Columns and analysis of the long-term behaviour (differential settlement)
- Crack initiation and propagation in concrete structure
- Discrete Modeling and analysis of masonry
- Composite Modeling and analysis of wall in shear
- Detail analysis for tendon anchorage



# Applicable Problems

## **Thermo-Elastic Analysis (Heat Transfer, Heat of Hydration)**

- Analysis of heat of hydration (general, special, nonlinear)
- Detail analysis for assessment of shear capacity of pavement (Debonding Failure)
- Analysis of thermal effect due to the asphalt pavement (Guss Asphalt)
- Fire effect on a reinforced concrete slab
- Evaluation of residual stress and integrity of welded part

## **Special Analysis (Contact, Geotechnical Analysis and etc.)**

- Crack and fatigue analysis of the surface of structures
- Soil structure interaction analysis considering soil properties

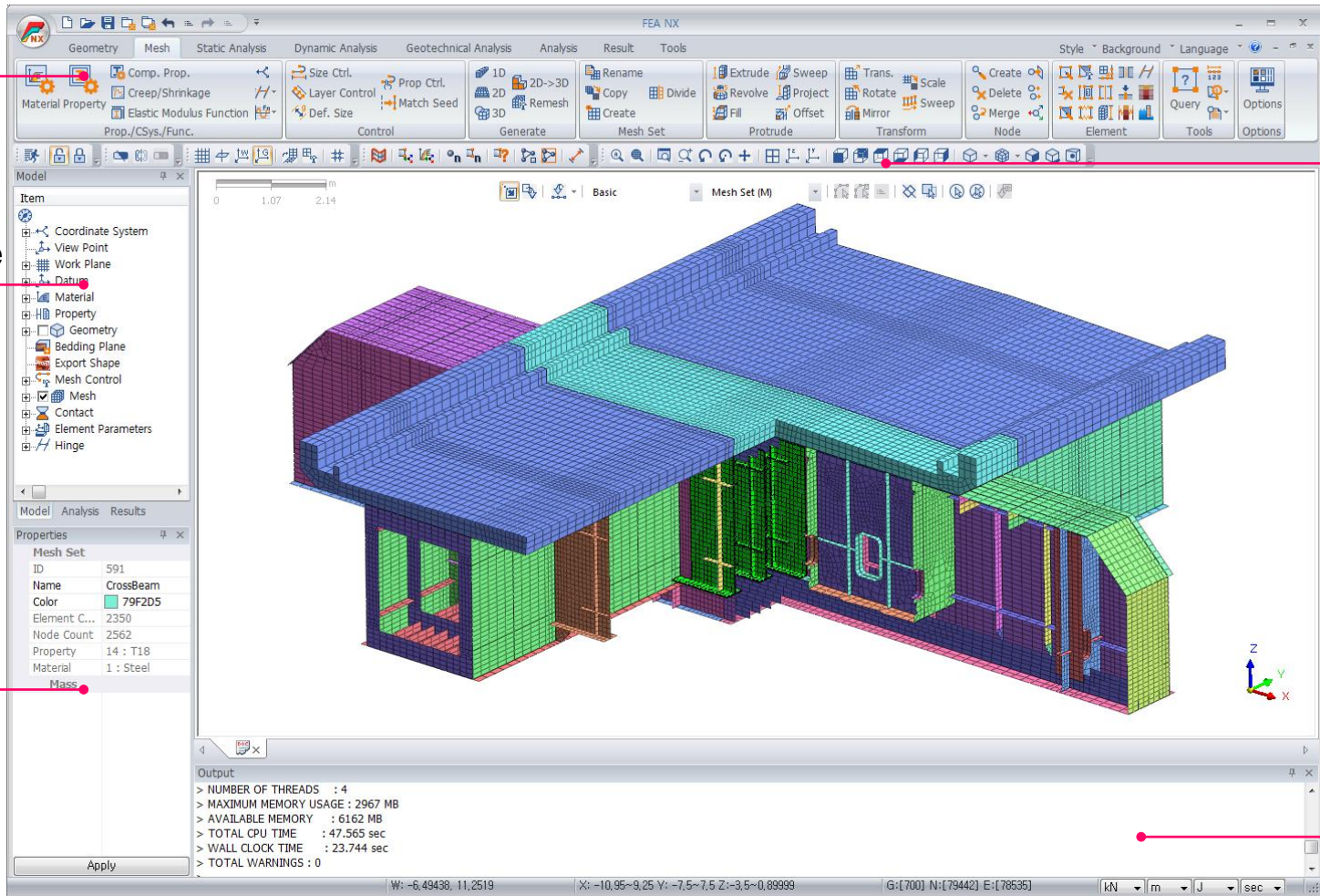
# Framework

Main Menu

Works Tree

Property Window

Tabbed  
Toolbar



Output  
Window

Developed based-on **Task-oriented Design Paradigm**

# 64-bit environment platform

## Next Generation Platform

Complete support for 64-bit

Control large scale model

## Automated Modeling

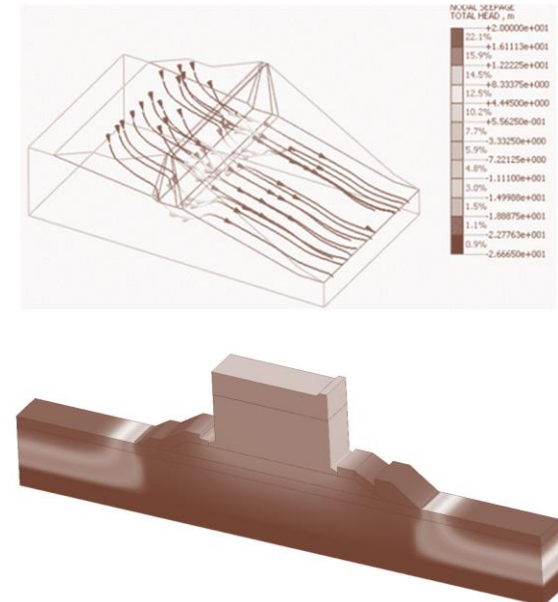
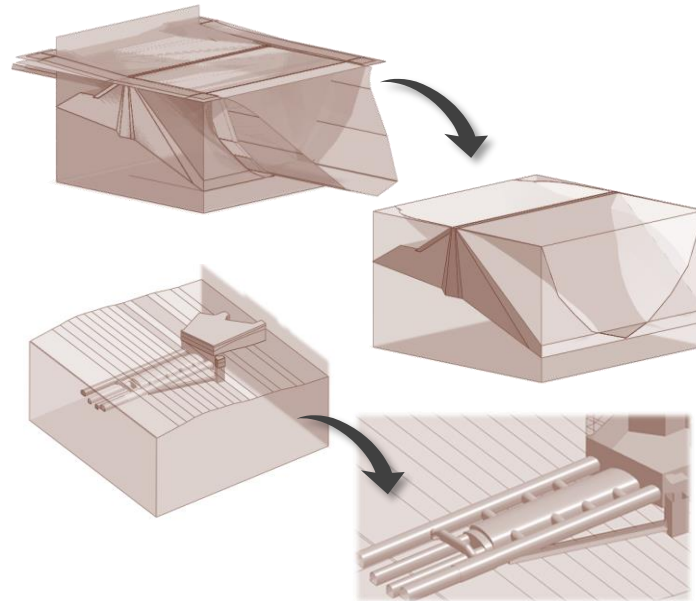
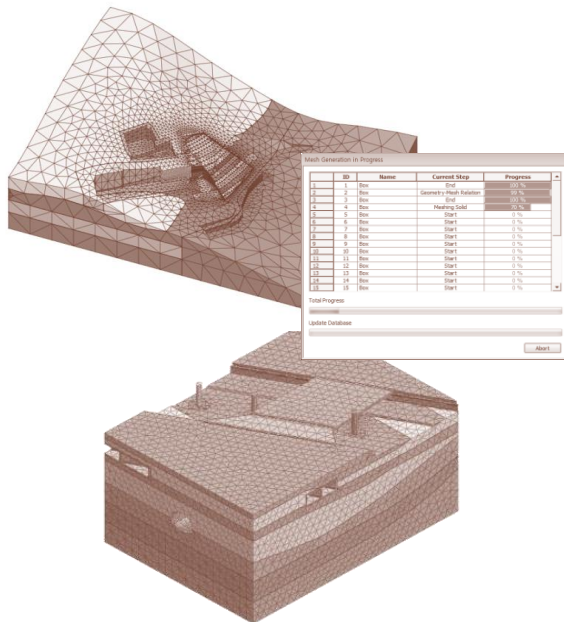
Stabilized geometric operation

Diverse automatic tools

## Universal Solver

New MIDAS solver

Advanced analysis conditions





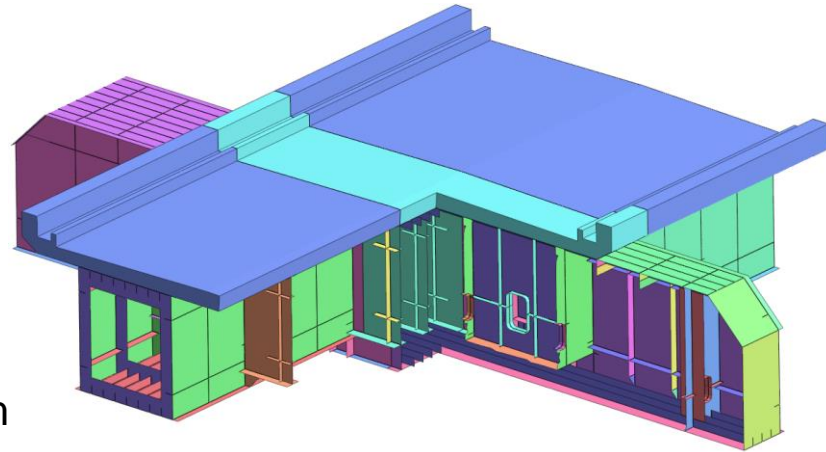
# Contents

1. Overview
- 2. Geometry Modeling**
3. Mesh Generation
4. Analysis
5. Post-processing

# Data Exchange

## ■ Import (Geometry)

- AutoCAD
- Parasolid / ACIS / STEP
- IGES / Pro-E
- CATIA V4 / V5
- SolidWorks / Unigraphics
- Inventor Part / Inventor Assem



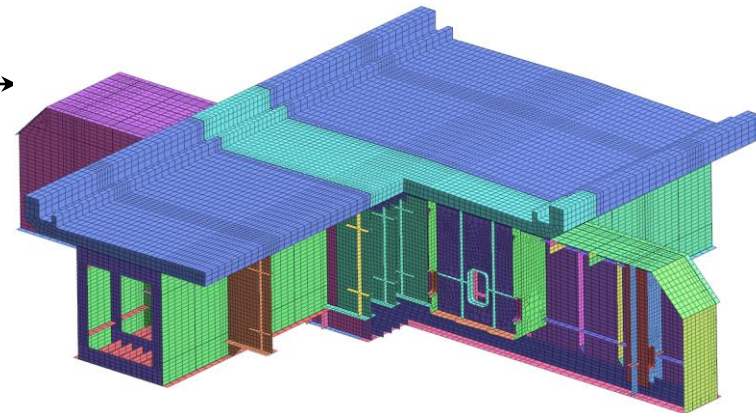
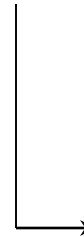
Imported CAD Geometry

## ■ Export (Geometry)

- Parasolid
- STL

### **Standards for CAD Data Exchange**

- *STEP (STandard for the Exchange of Product Model Data)*
- *IGES (Initial Graphics Exchange Specification)*



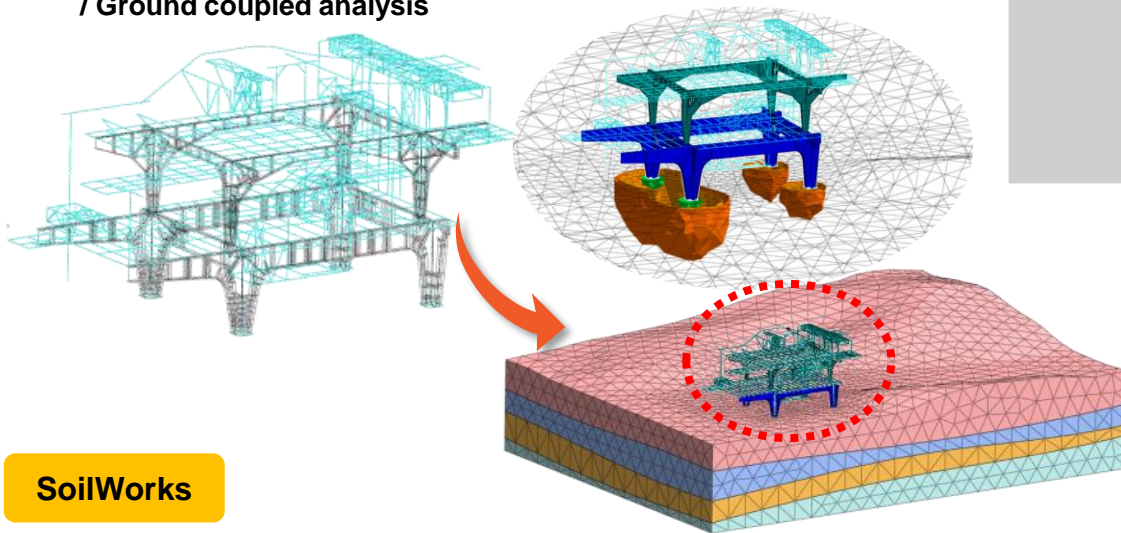
Generated Mesh



# Interface with other MIDAS

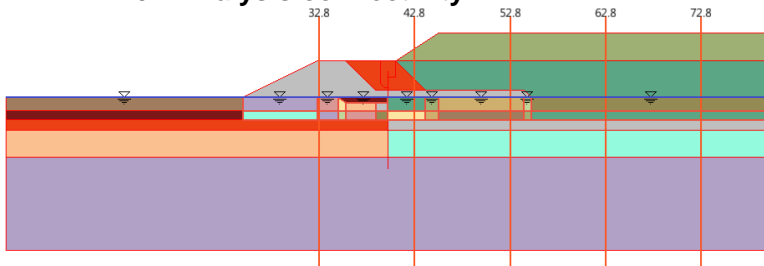
## Gen

Architectural structure foundation  
/ Ground coupled analysis



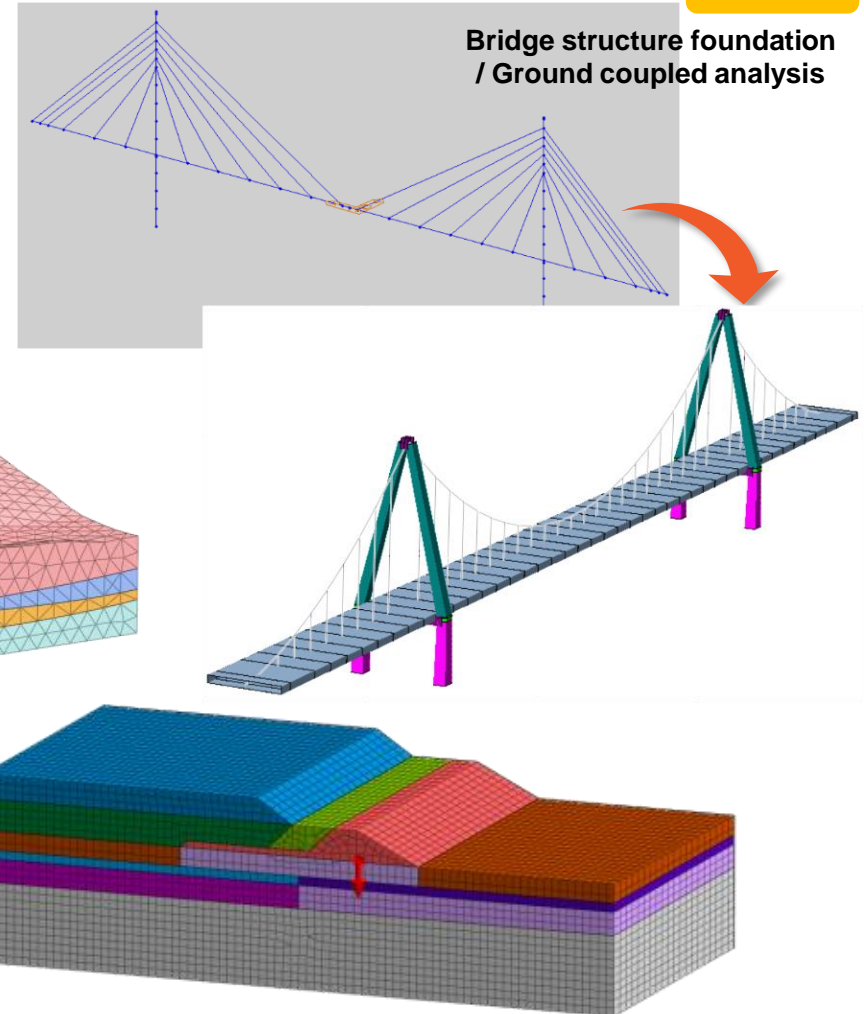
## SoilWorks

2D FE ↔ 3D Analysis connectivity

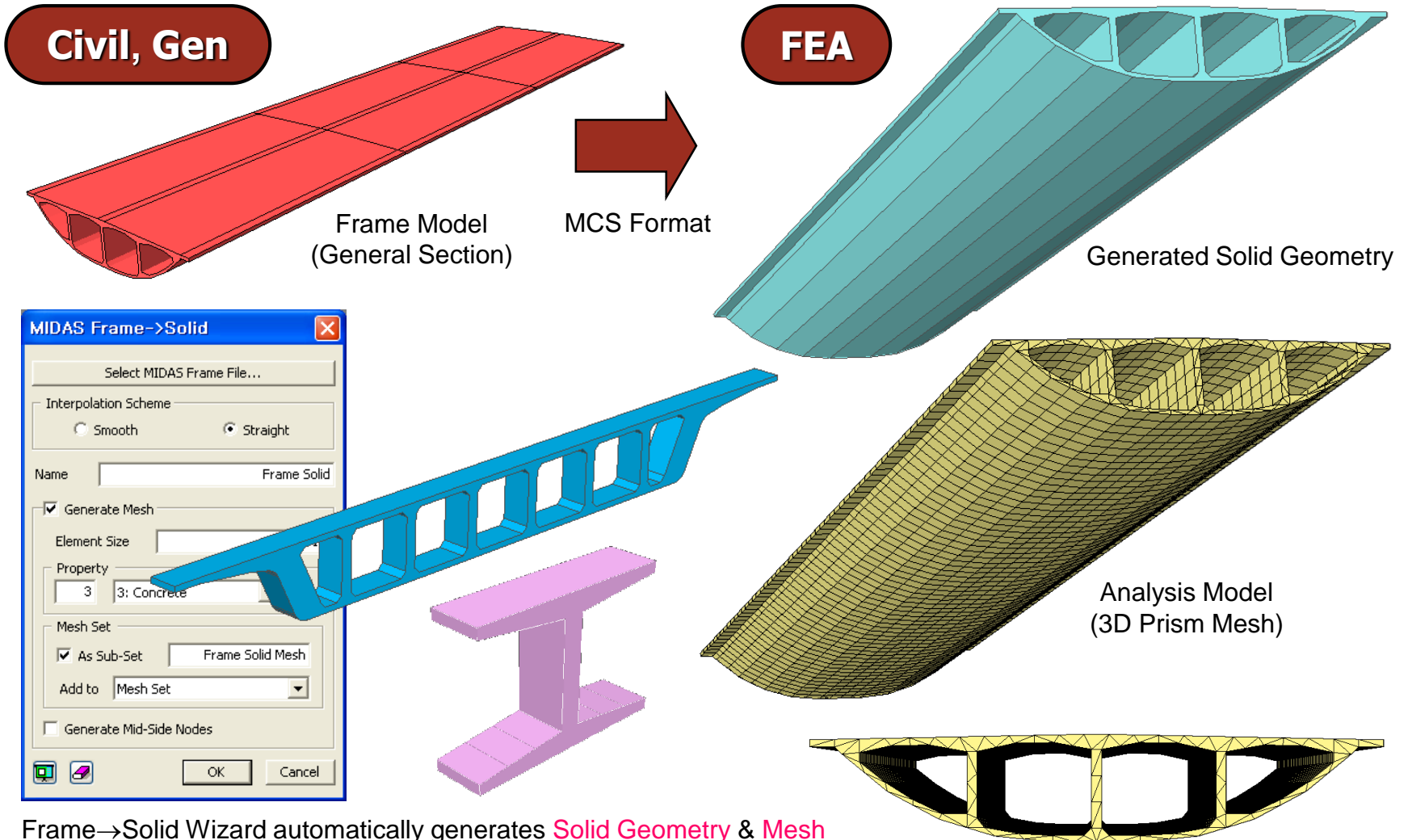


## Civil

Bridge structure foundation  
/ Ground coupled analysis

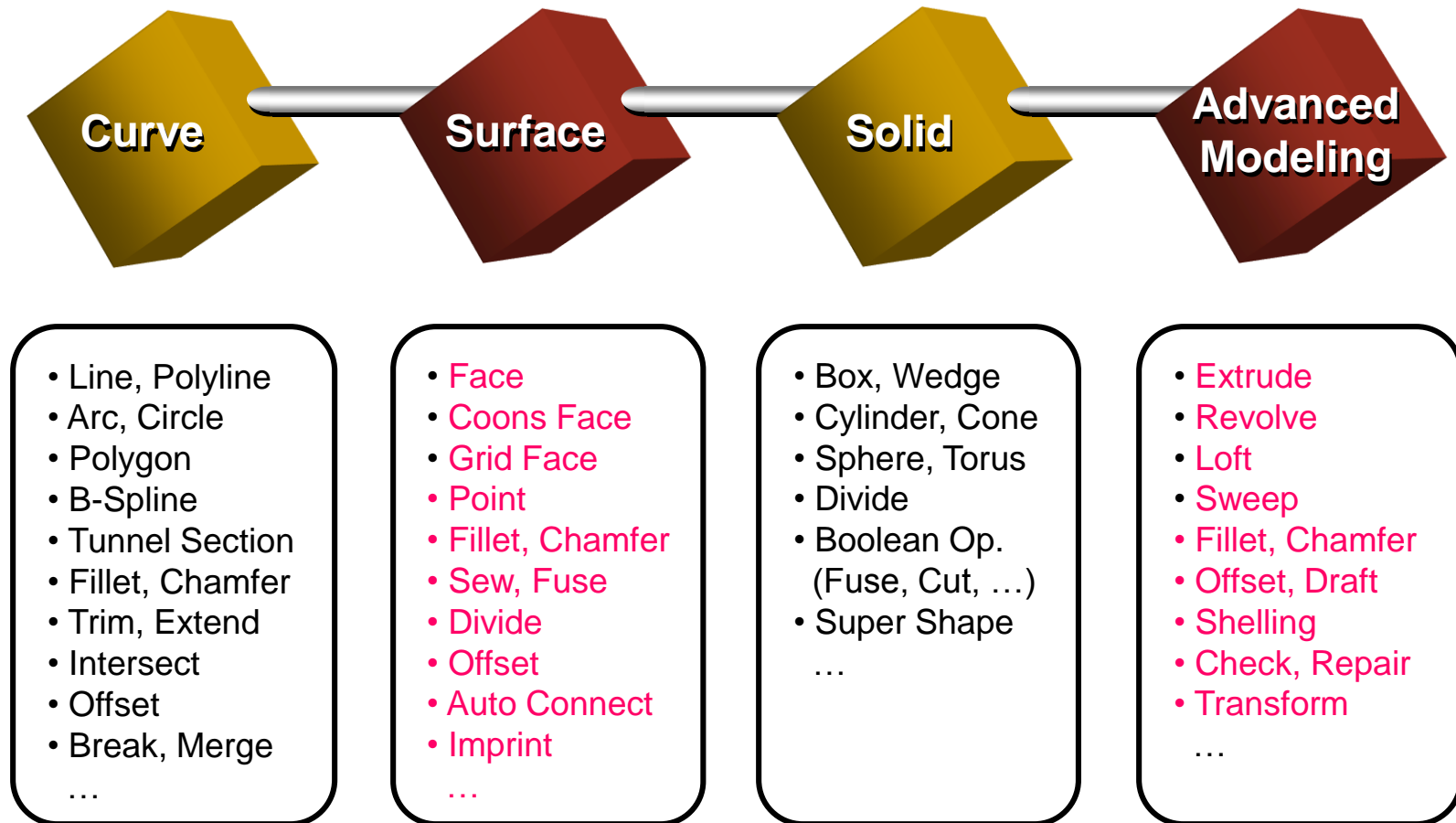


# Frame→Solid Wizard



Frame→Solid Wizard automatically generates **Solid Geometry & Mesh** by importing **Frame Model** (\*.MCS) from **Civil** and **Gen**.

# Geometry Modeling



Advanced Modeling functions support both **top-down** and **bottom-up** approaches in **surface** & **solid** Modeling.

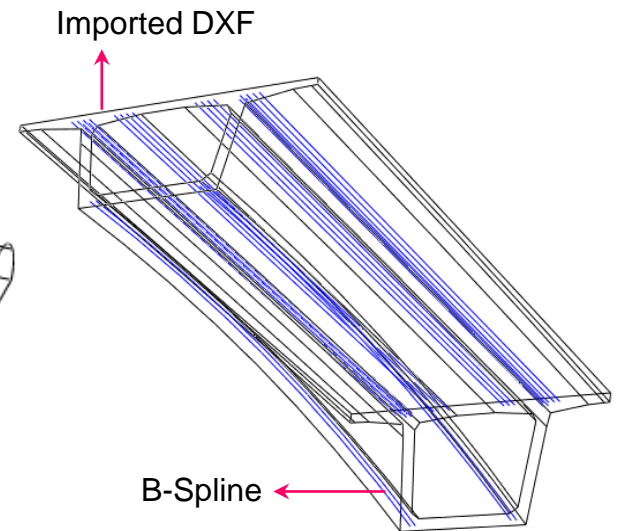
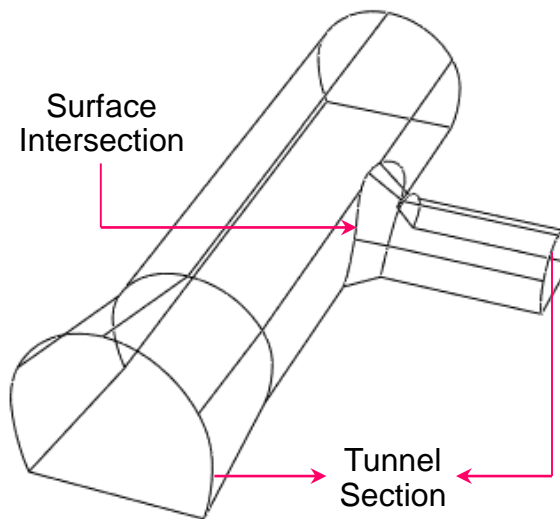
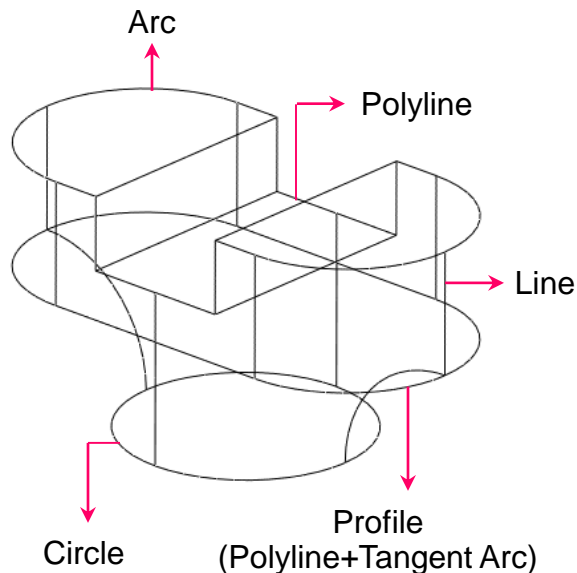
# Curve Modeling

## Generation

- Line
- Arc
- Circle
- Ellipse
- Tunnel
- B-Spline
- Polyline
- Rectangle
- Polygon
- Profile
- Shortest Path Line
- Surface Intersection
- Offset Curve
- Extrude Point

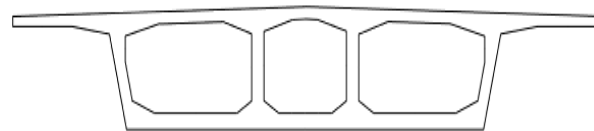
## Modification

- Fillet / Chamfer
- Trim / Extend
- Merge / Break
- Intersect
- Make Wire

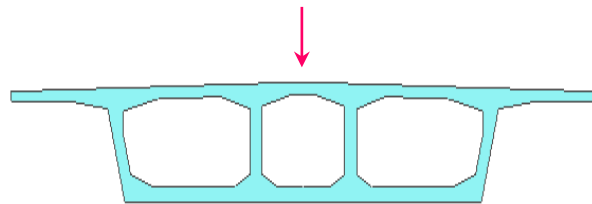




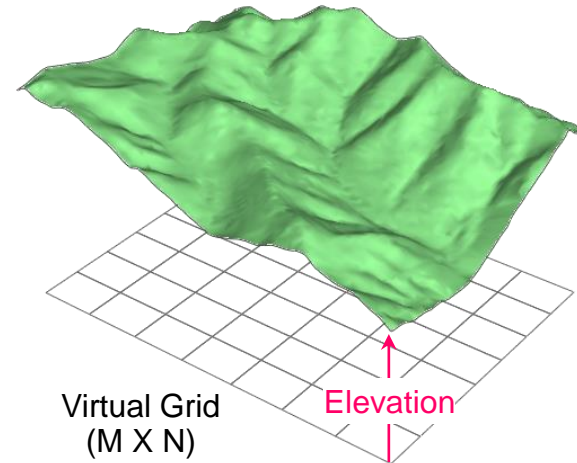
# Surface Modeling



Co-planar Curves

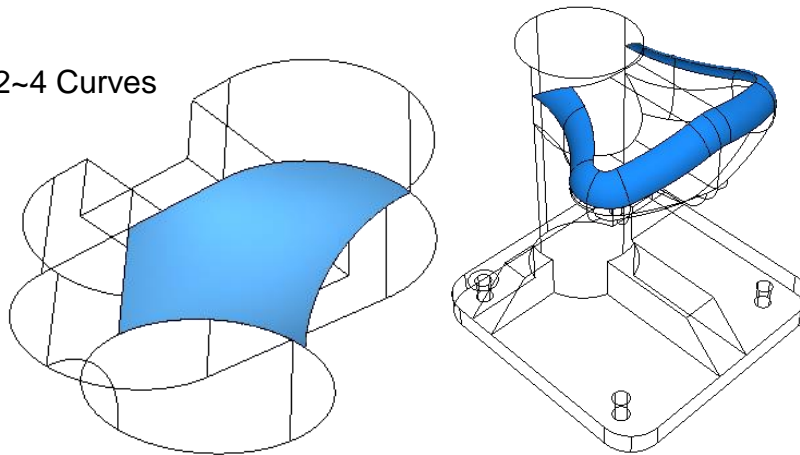


Plane Patch

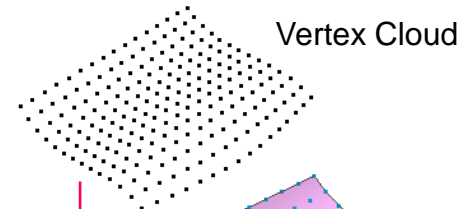


Grid Patch

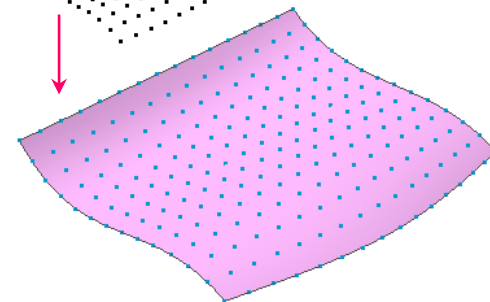
2~4 Curves



Coons Patch

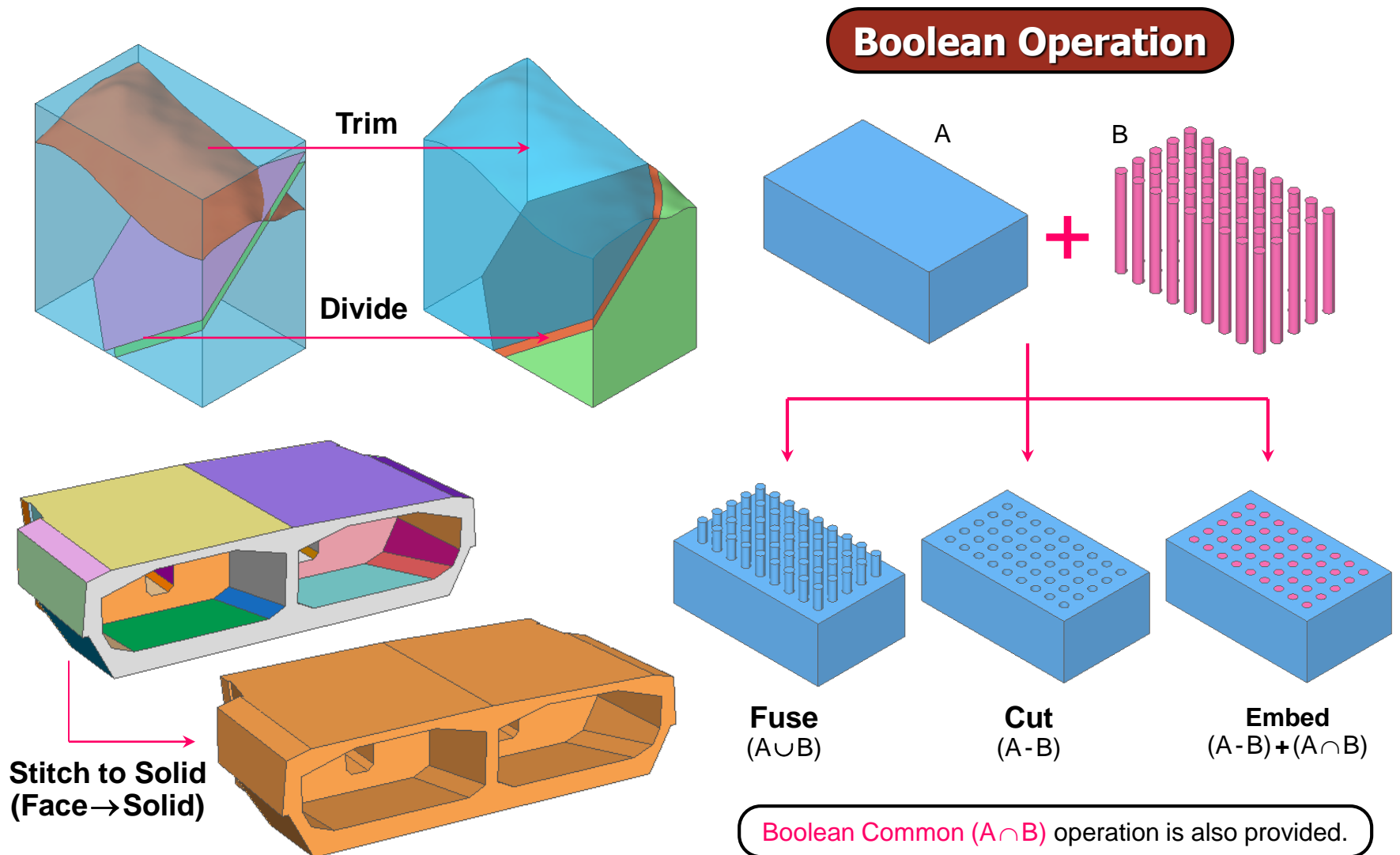


Vertex Cloud



Vertex Patch

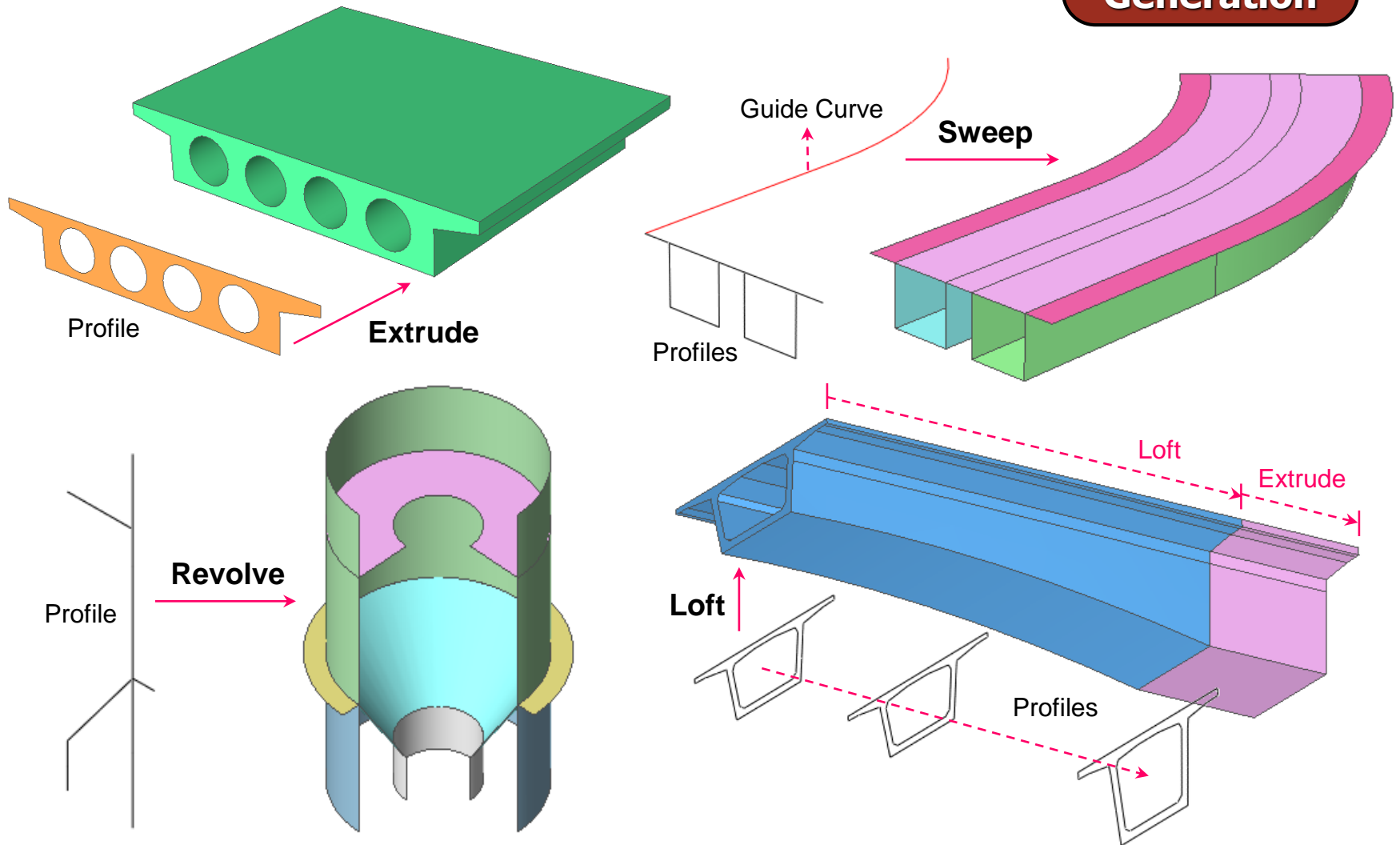
# Solid Modeling



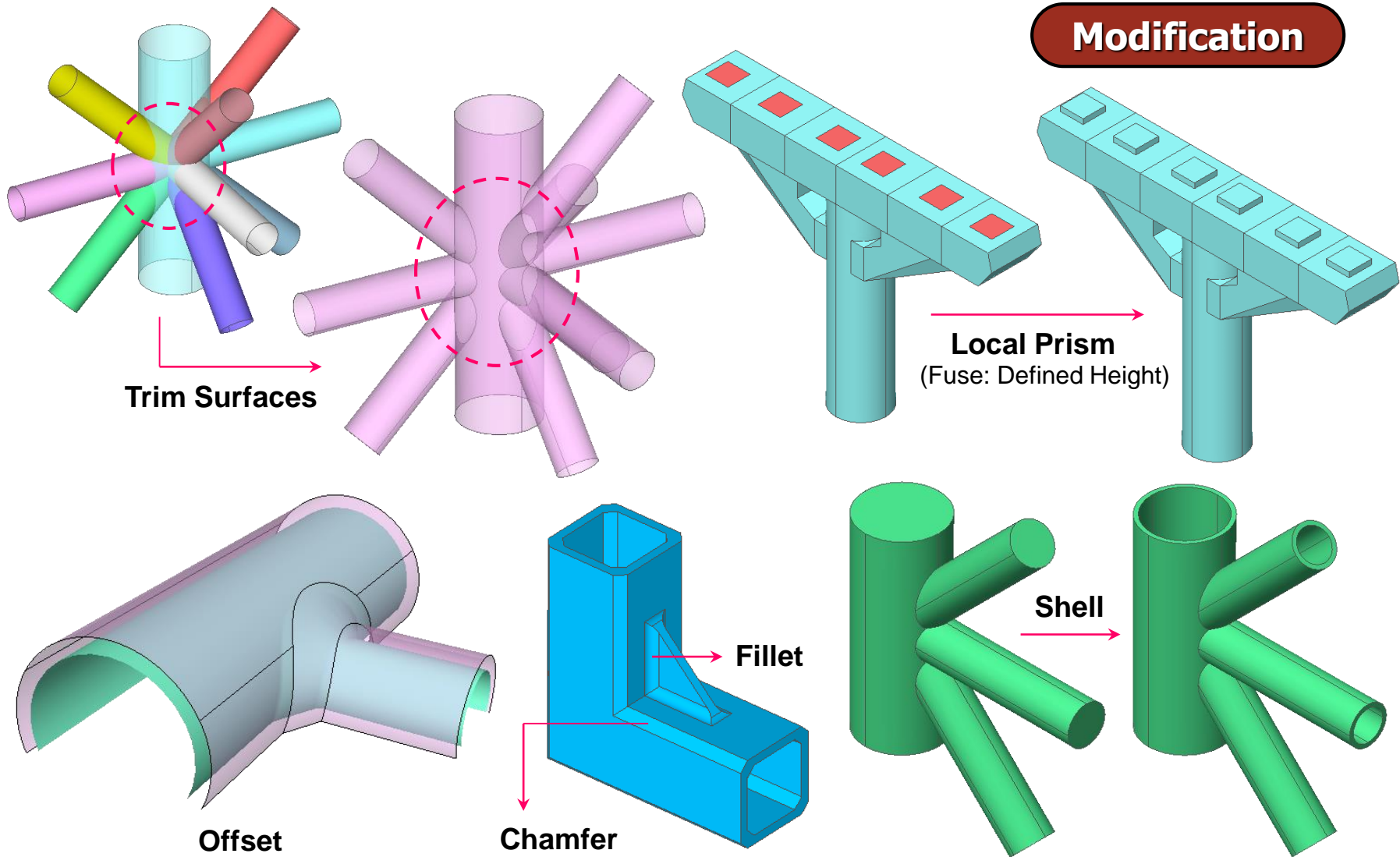


# Advanced Modeling

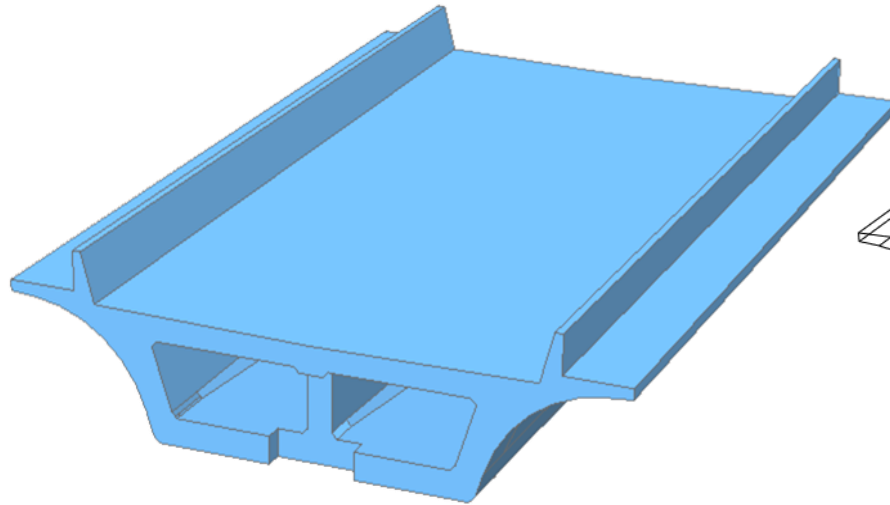
## Generation



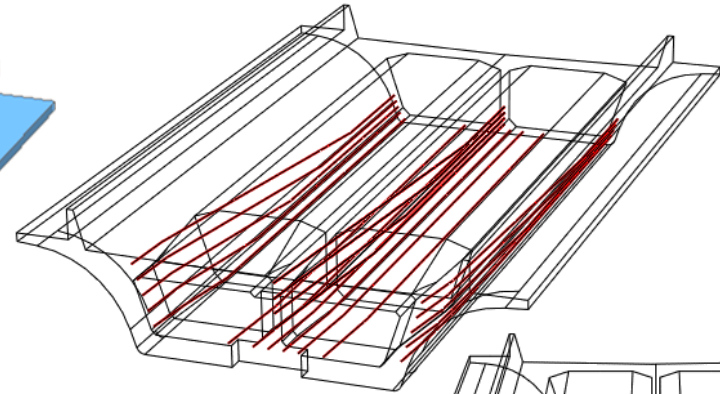
# Advanced Modeling



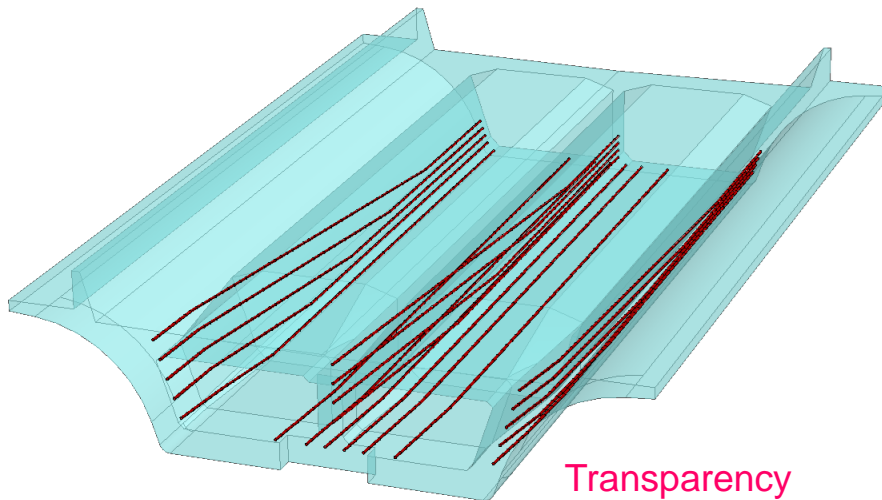
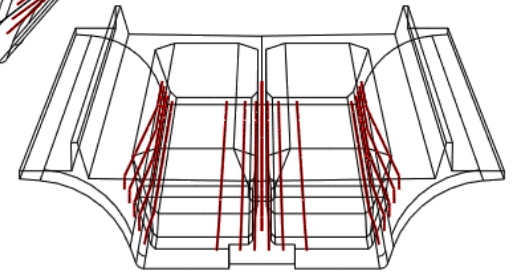
# Graphic Display - Geometry



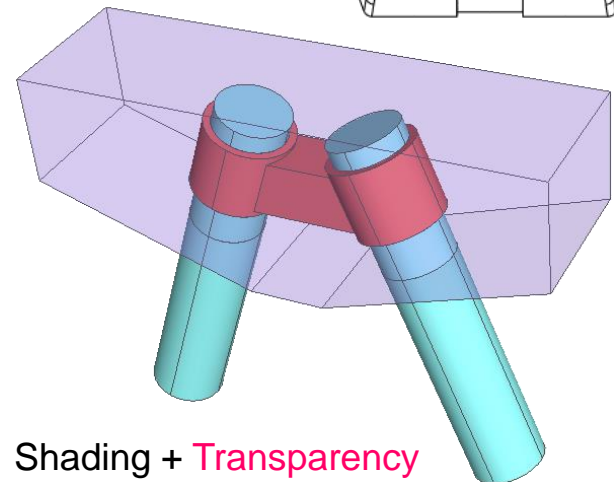
Shading with Edge



Wireframe



Transparency

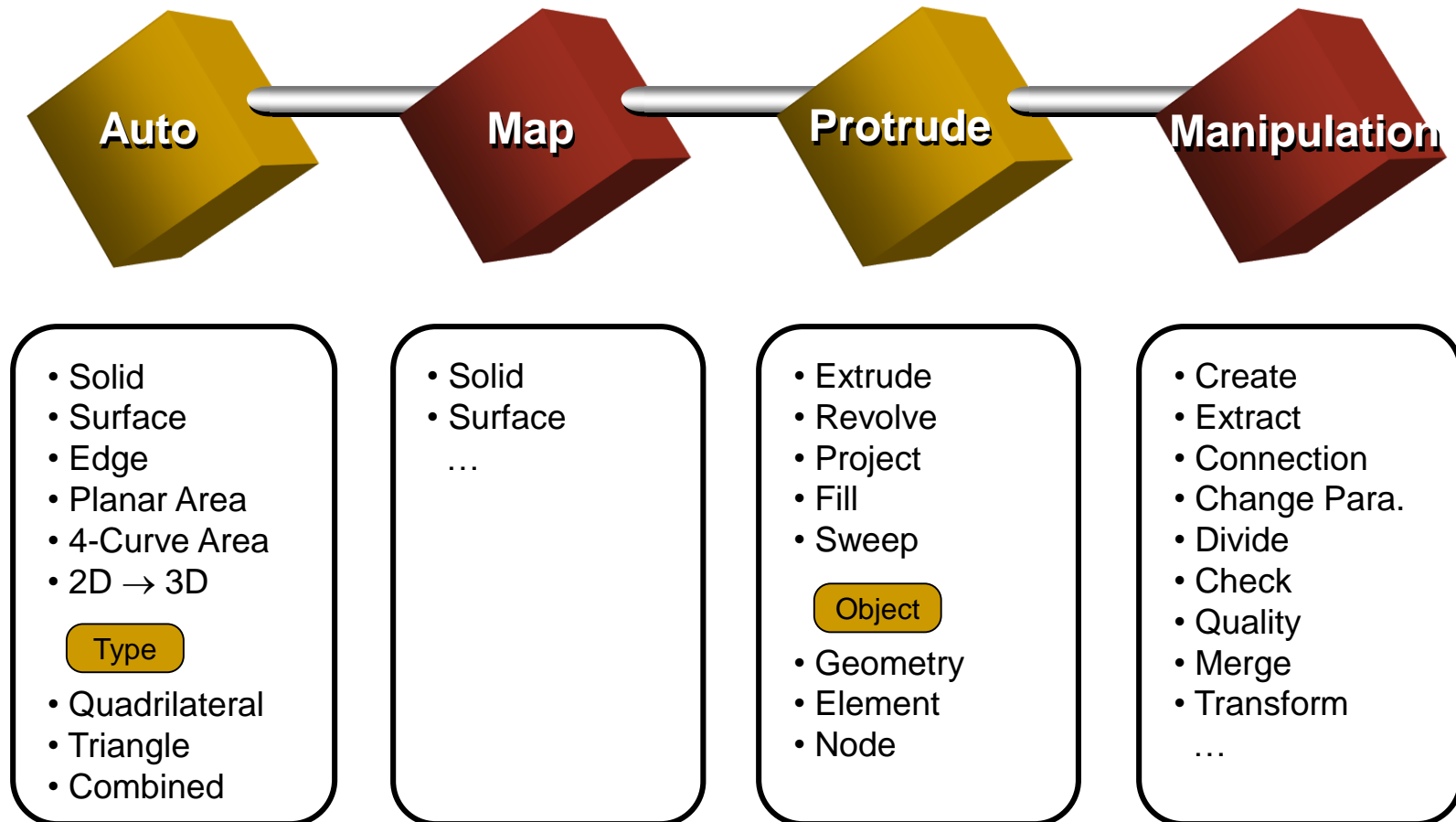


Shading + Transparency

# Contents

1. Overview
2. Geometry Modeling
- 3. Mesh Generation**
4. Analysis
5. Post-processing

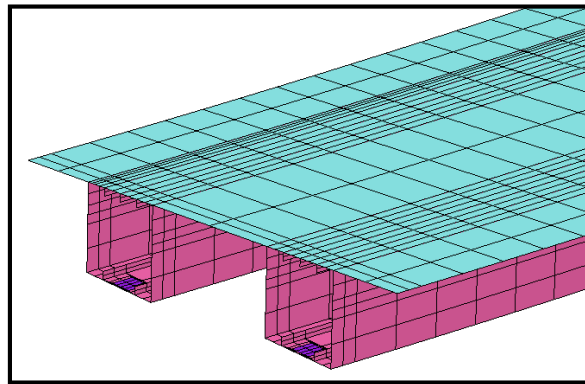
# Mesh Generation



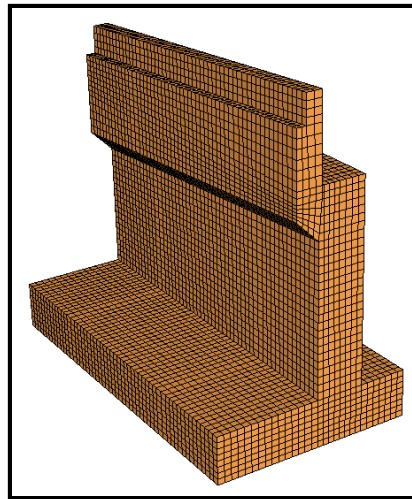
Various of methods for generating **Reinforcements** and **Interface Elements** are provided. (auto & manual)



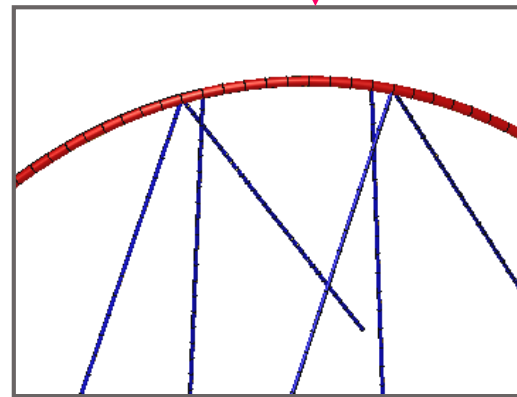
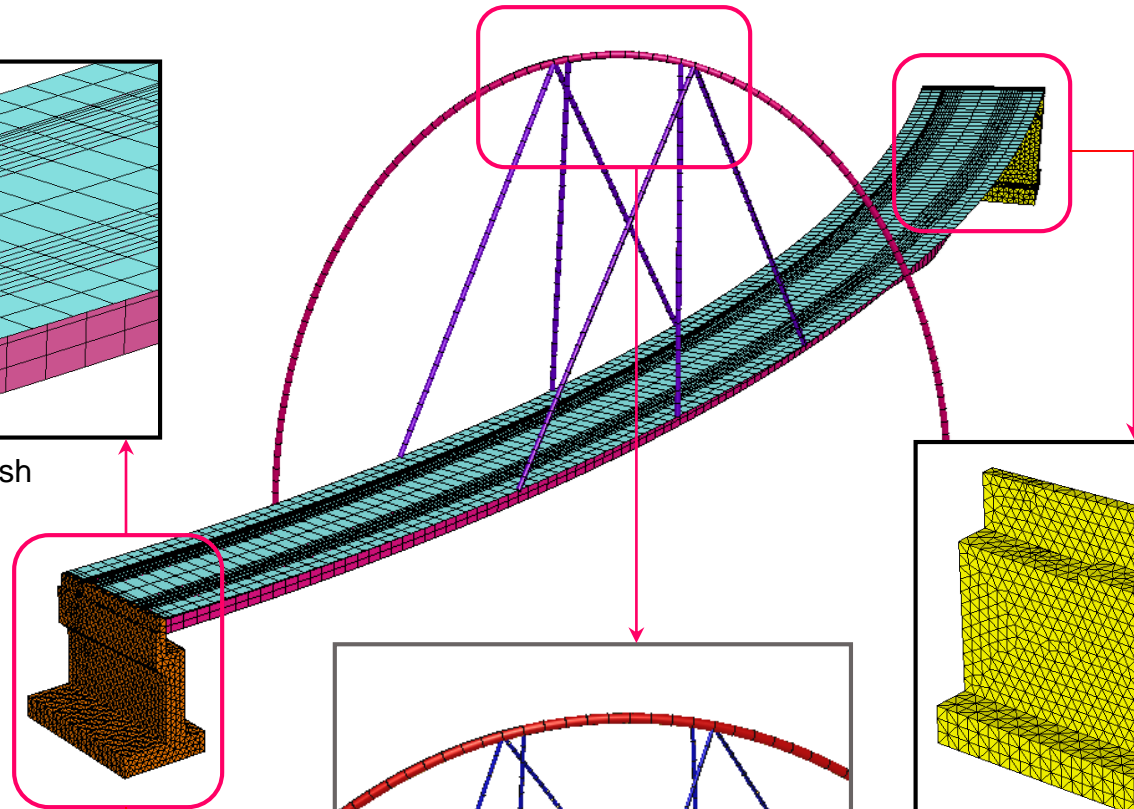
# Mesh Generation



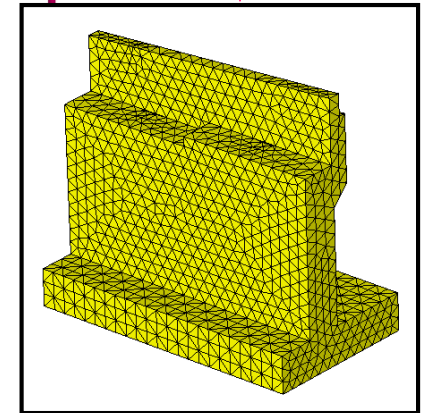
2D Quadrilateral Mesh



3D Hexahedral Mesh



1D Linear Mesh

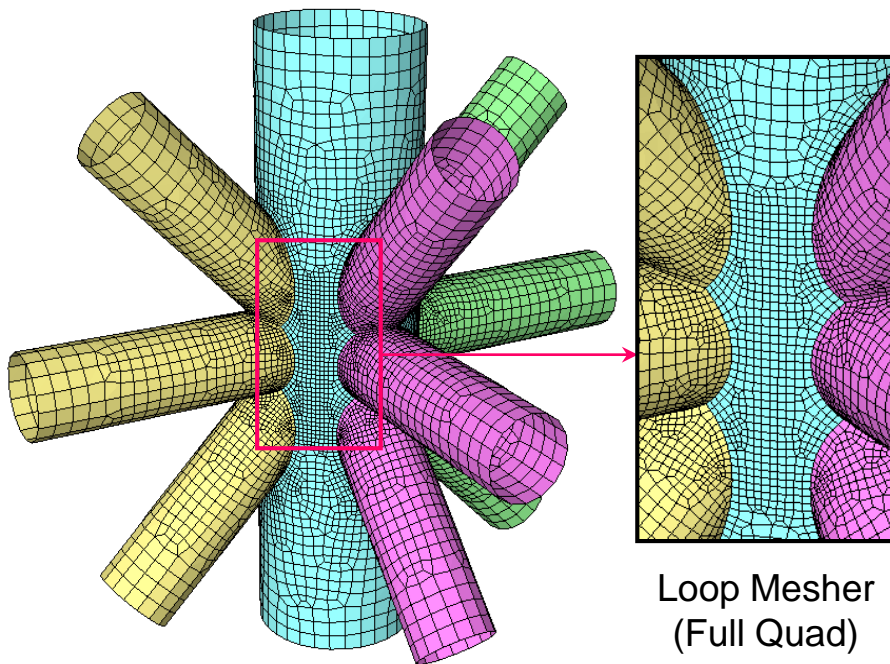


3D Tetrahedral Mesh

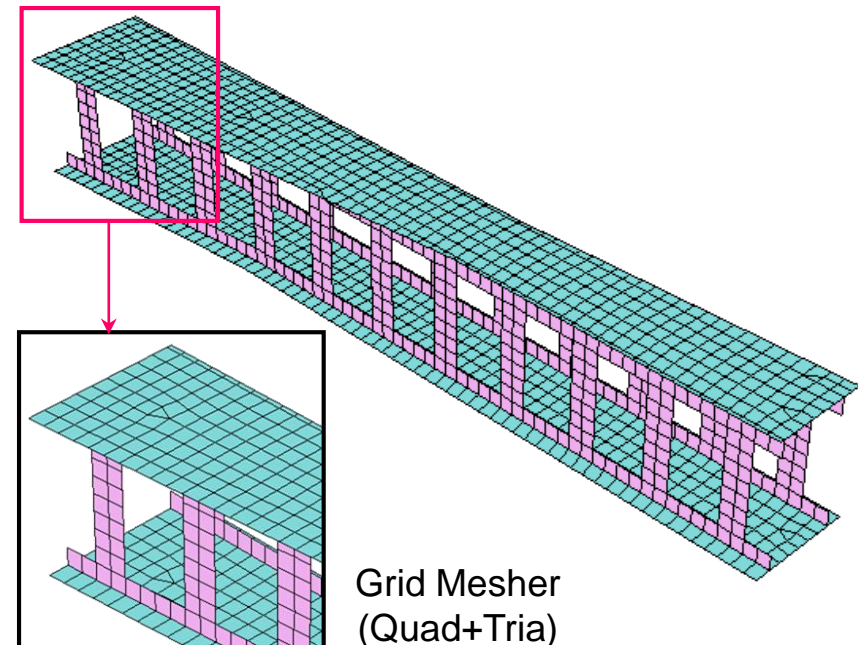


# Automatic Surface Meshing

	Regularity Uniformity	Boundary Sensitive	Orientation Insensitive	Sizing Control ( $< 1/2$ )	Internal Curve/Point
Loop Mesher	○	○	○	○	△
Grid Mesher	○	○	△	×	○
Delaunay Mesher	△	○	○	○	○



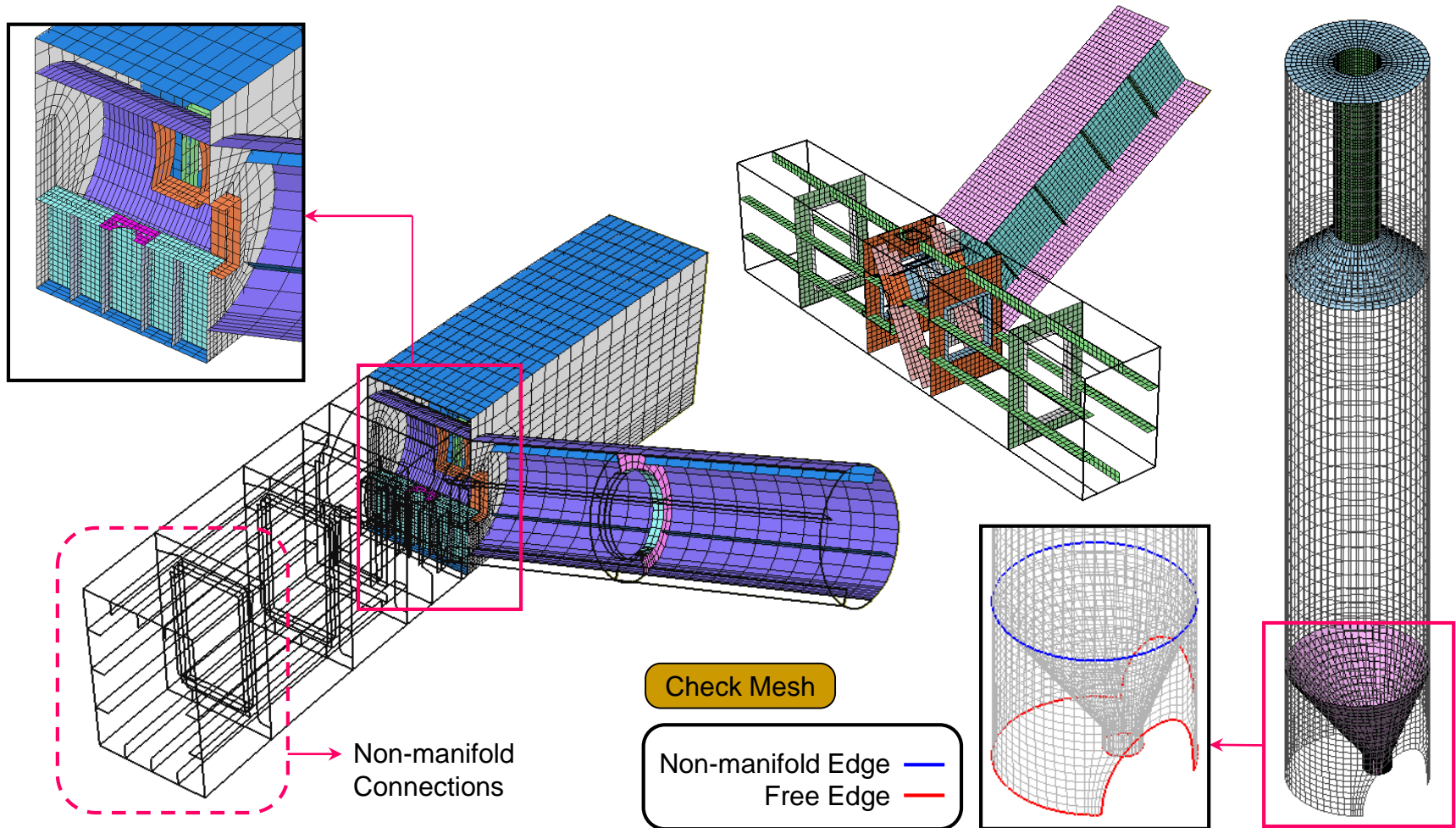
Loop Mesher  
(Full Quad)



Grid Mesher  
(Quad+Tria)

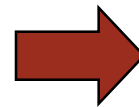
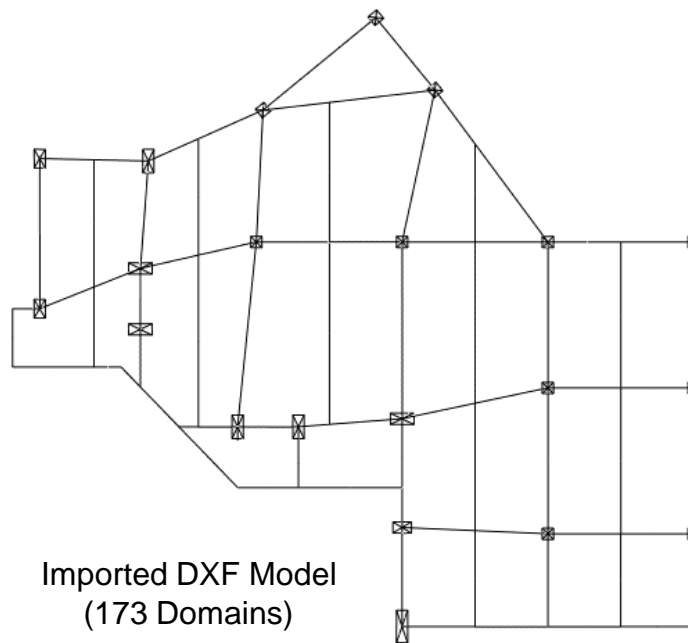
# Automatic Surface Meshing

- FEA provides a number of Modeling and meshing functions for **non-manifold surface** models.

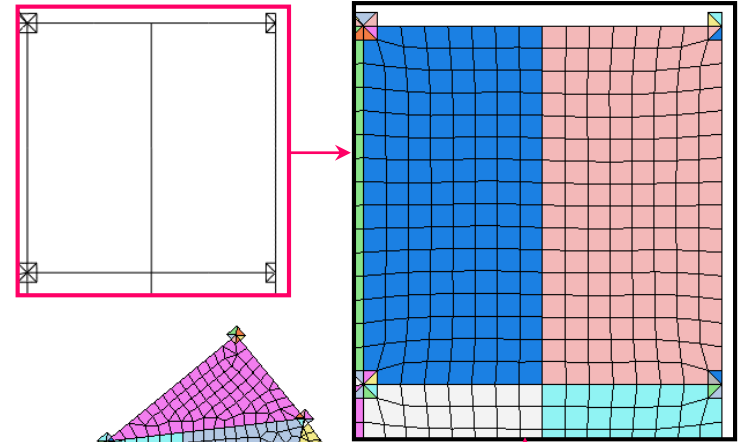
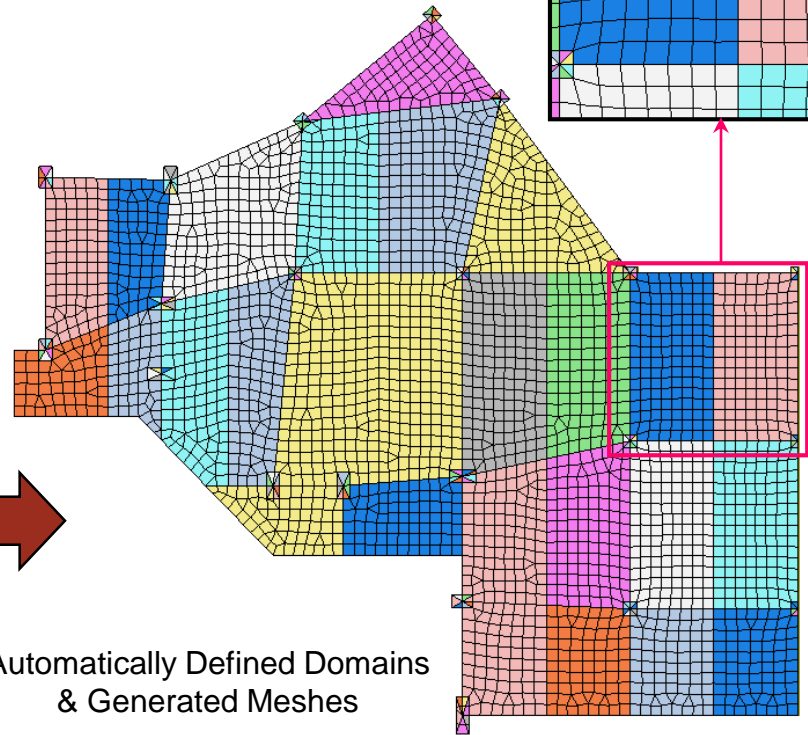


# Automatic Surface Meshing

- FEA provides **automatic defining & meshing function** which defines mesh-able domains from curves (without creating surfaces) and then generates mesh for each domain.
- **Automatic defining & meshing function** is very useful for **complex 2D models** which were originally modelled in **AutoCAD**.



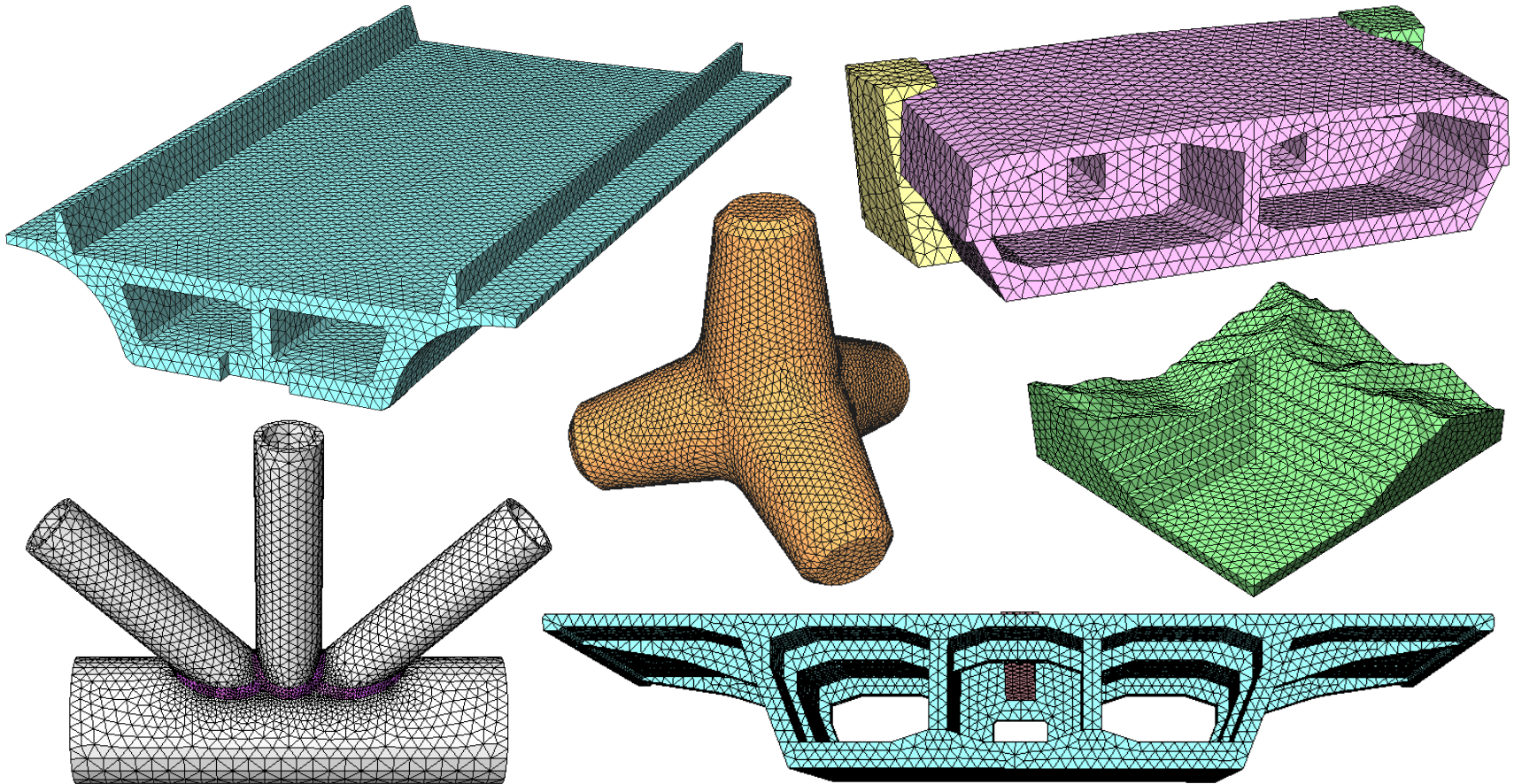
Automatically Defined Domains  
& Generated Meshes





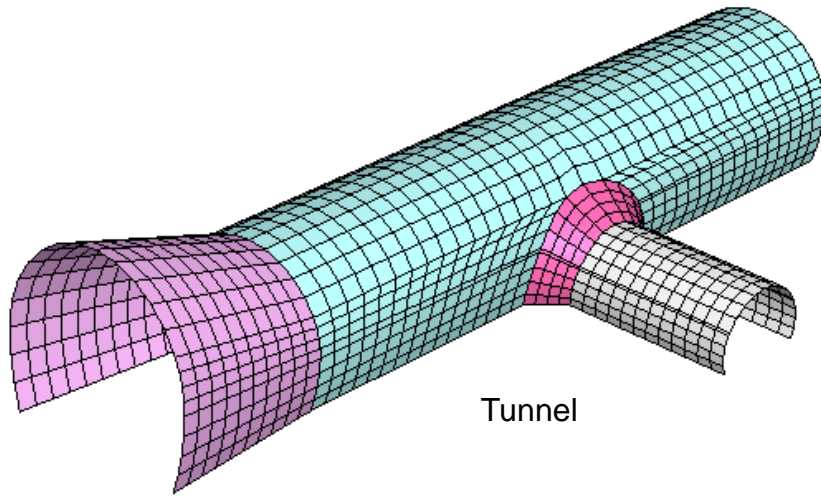
# Automatic Solid Meshing

- FEA's **Tetra Mesher** auto-generates **tetrahedral solid mesh** with variable sizes in smooth transition. (200,000 Tetra's/min)
- FEA's **Tetra Mesher** is capable of including **holes**, **curves** and **points** that are present in/on solids.

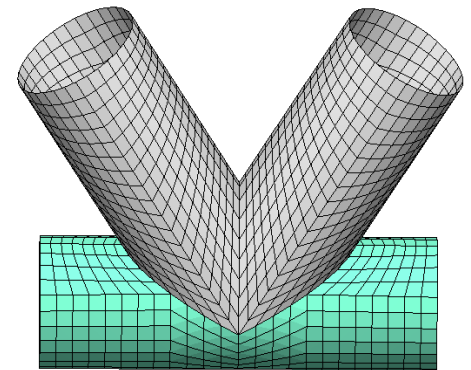


# Mapped Mesh Generation

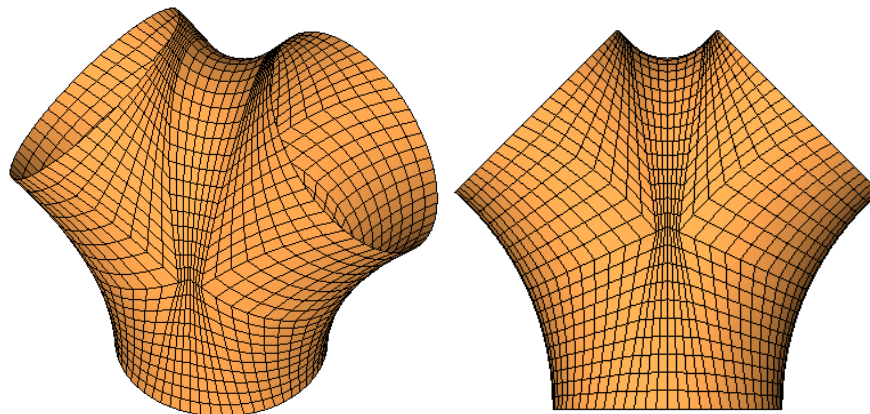
- FEA's **Map Mesher** generates **structured (regular & orthogonal) mesh** both on surfaces and in solids.



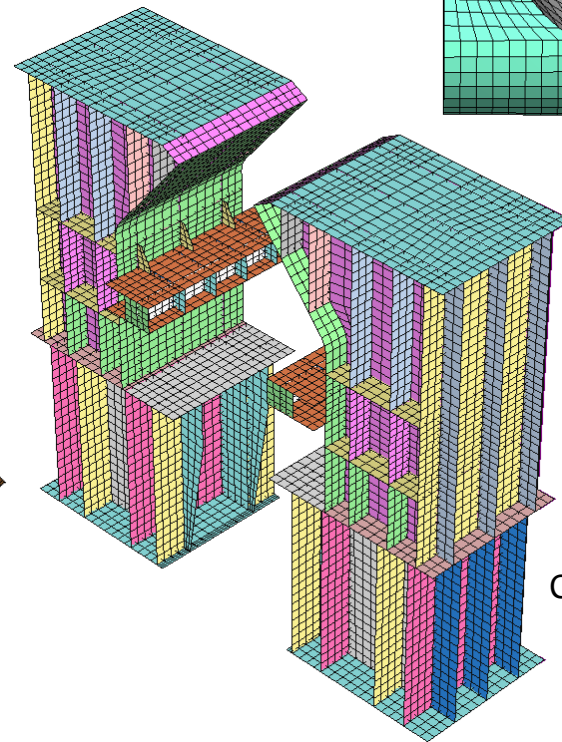
Tunnel



Pipe Junction



Connection Part

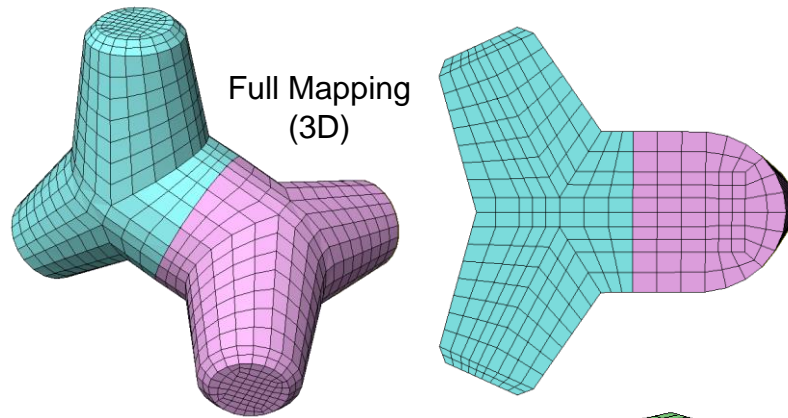


Cargo Carrier

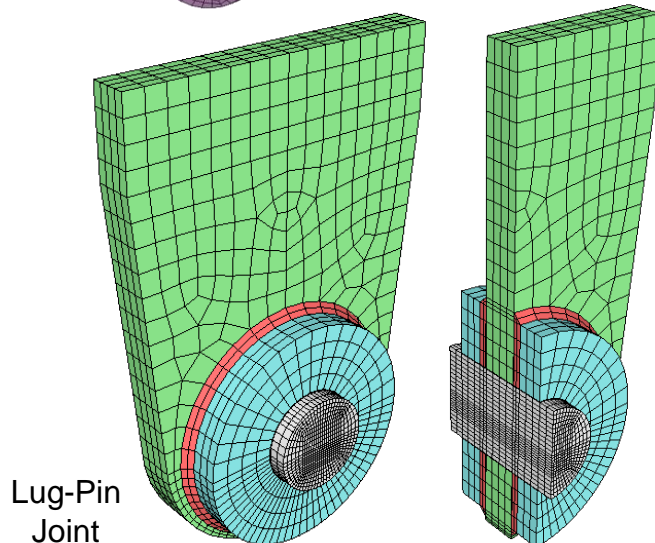
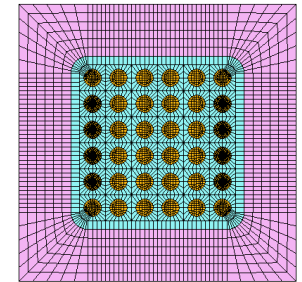
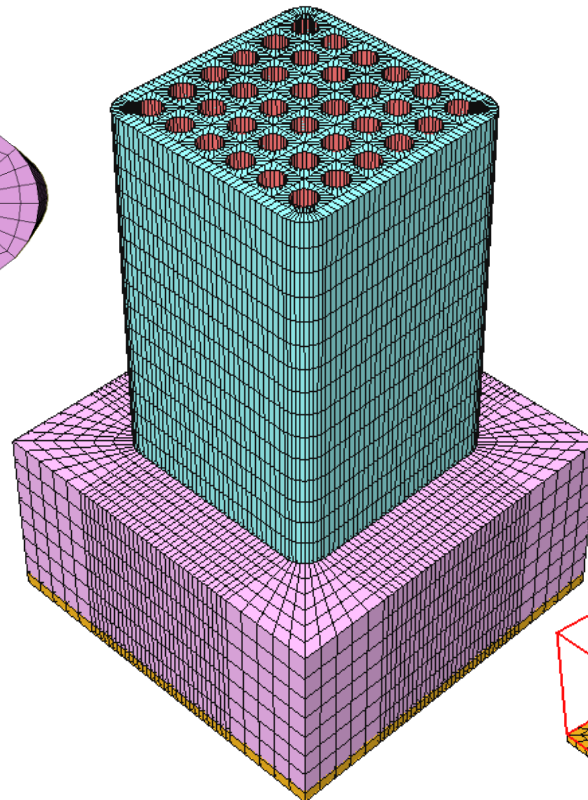


# Mapped Solid Meshing

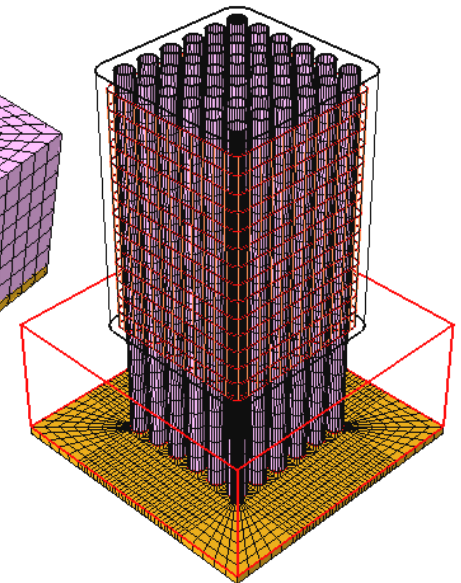
- FEA's **Solid Map Mesher** generates hexa and/or penta mesh in simple solids by **full mapping** or **combination (auto+map)**.



Full Mapping  
(3D)



Lug-Pin  
Joint

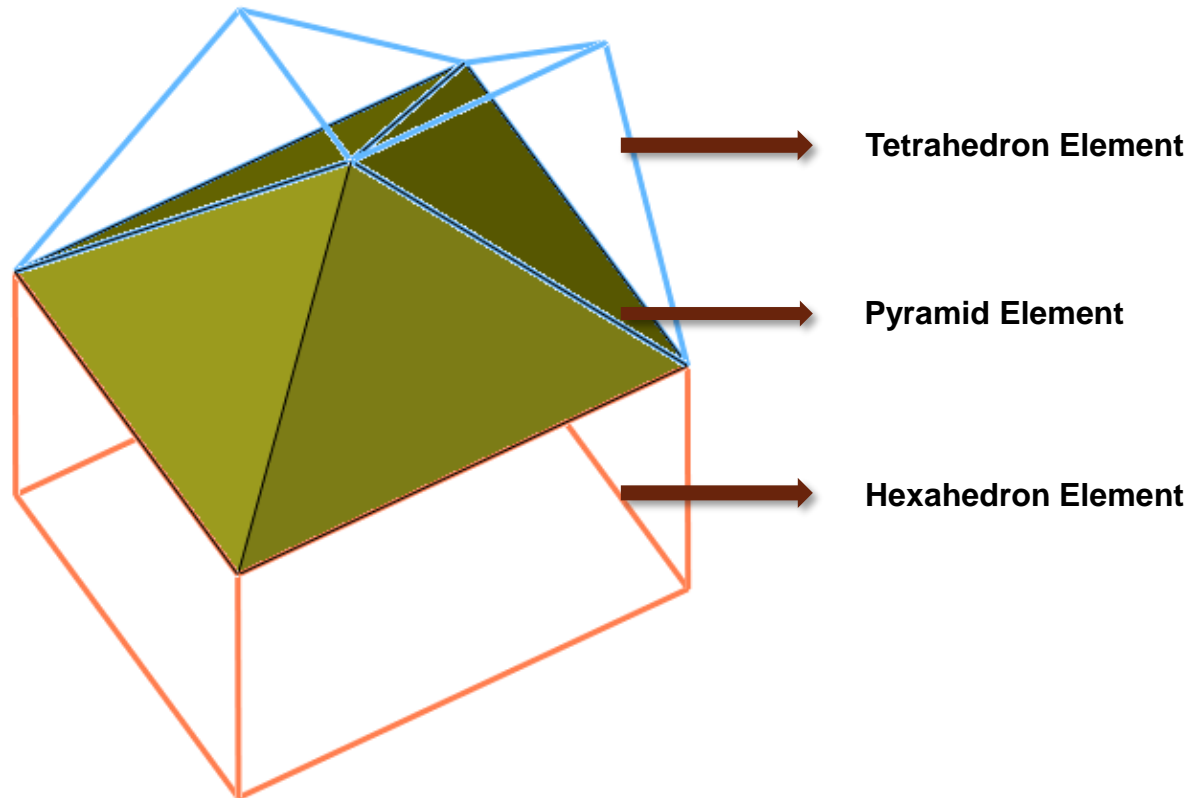
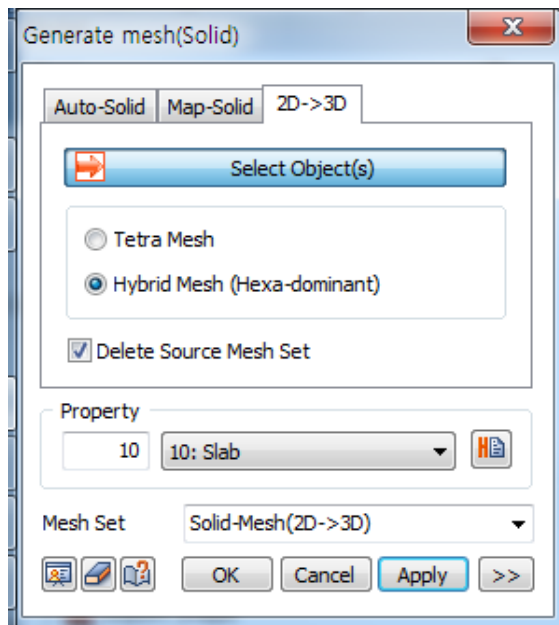


Pier (Pipe + Reinforcement Steel + Concrete)



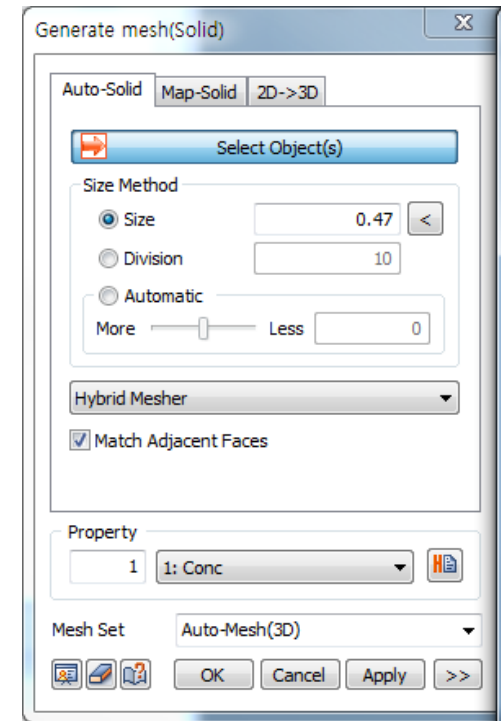
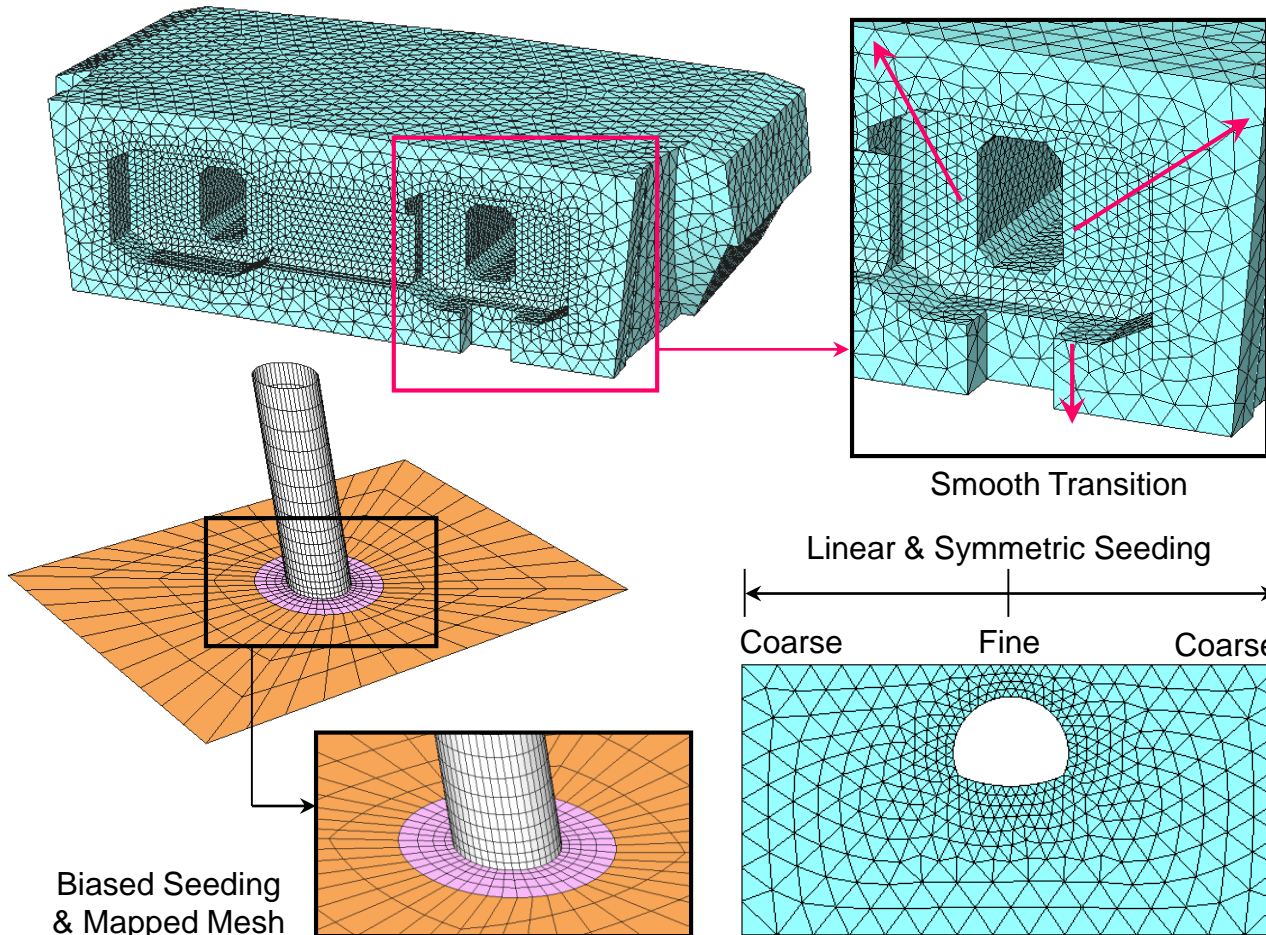
# Hexa Dominant Mesh

- Transitions from a tetrahedron element to a hexahedron element through a pyramid element

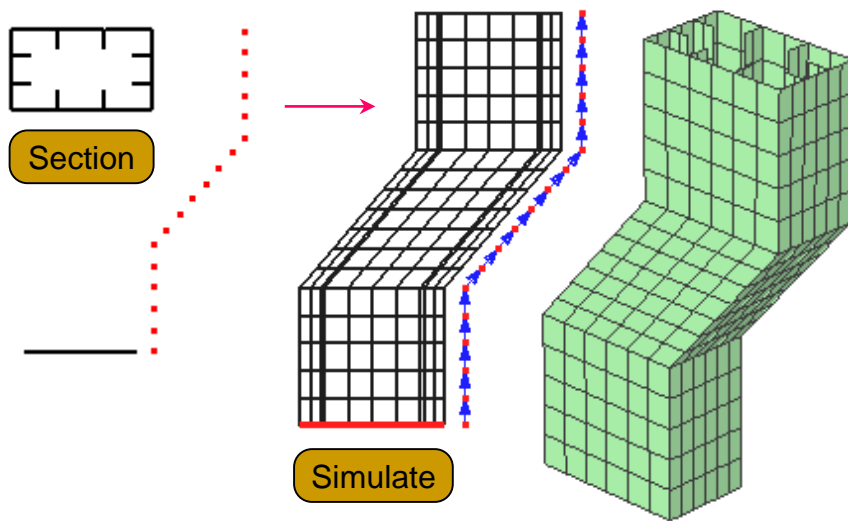
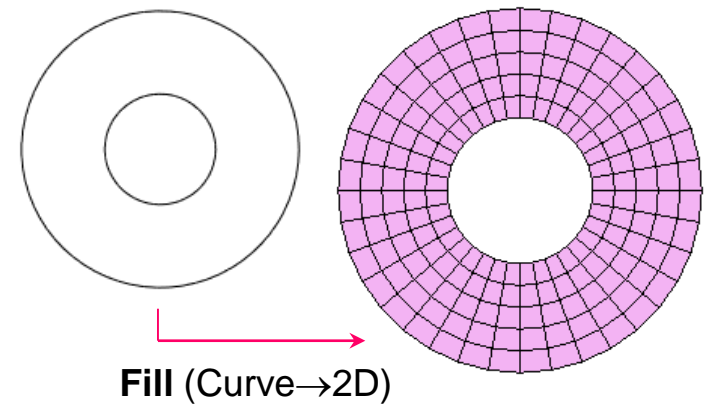
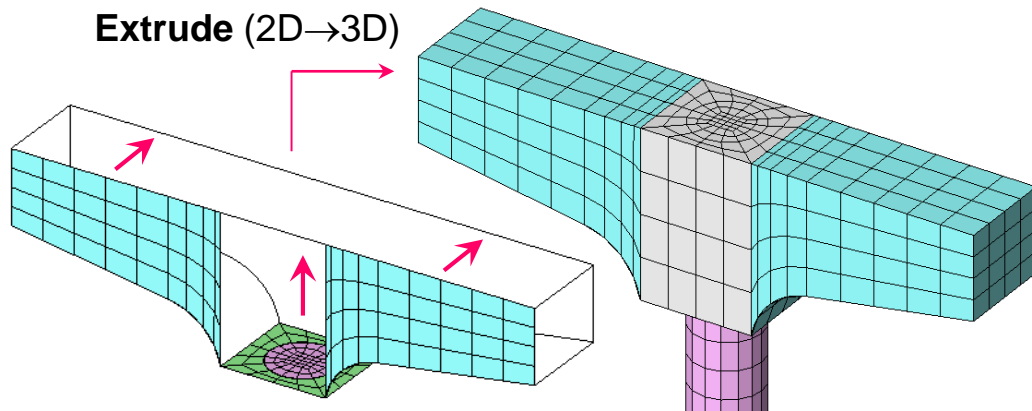


# Size Control

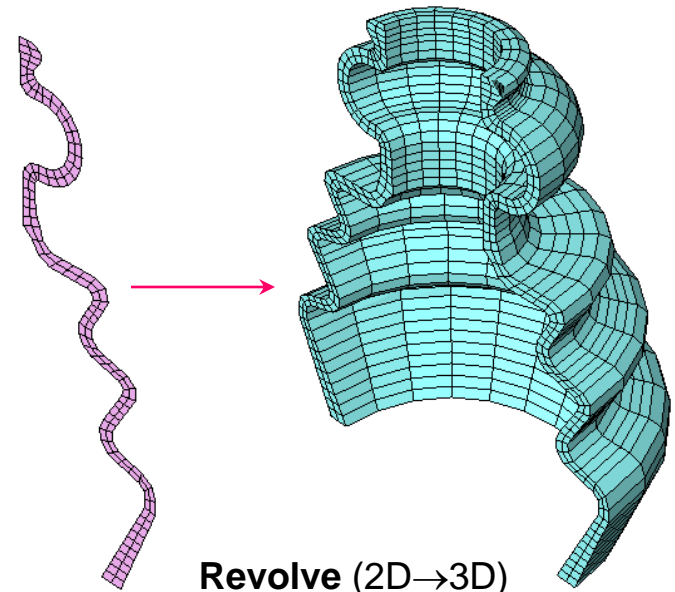
- FEA provides various size control methods and **adaptive seeding function** based on **user-specified mesh size** and **geometric characteristics**.



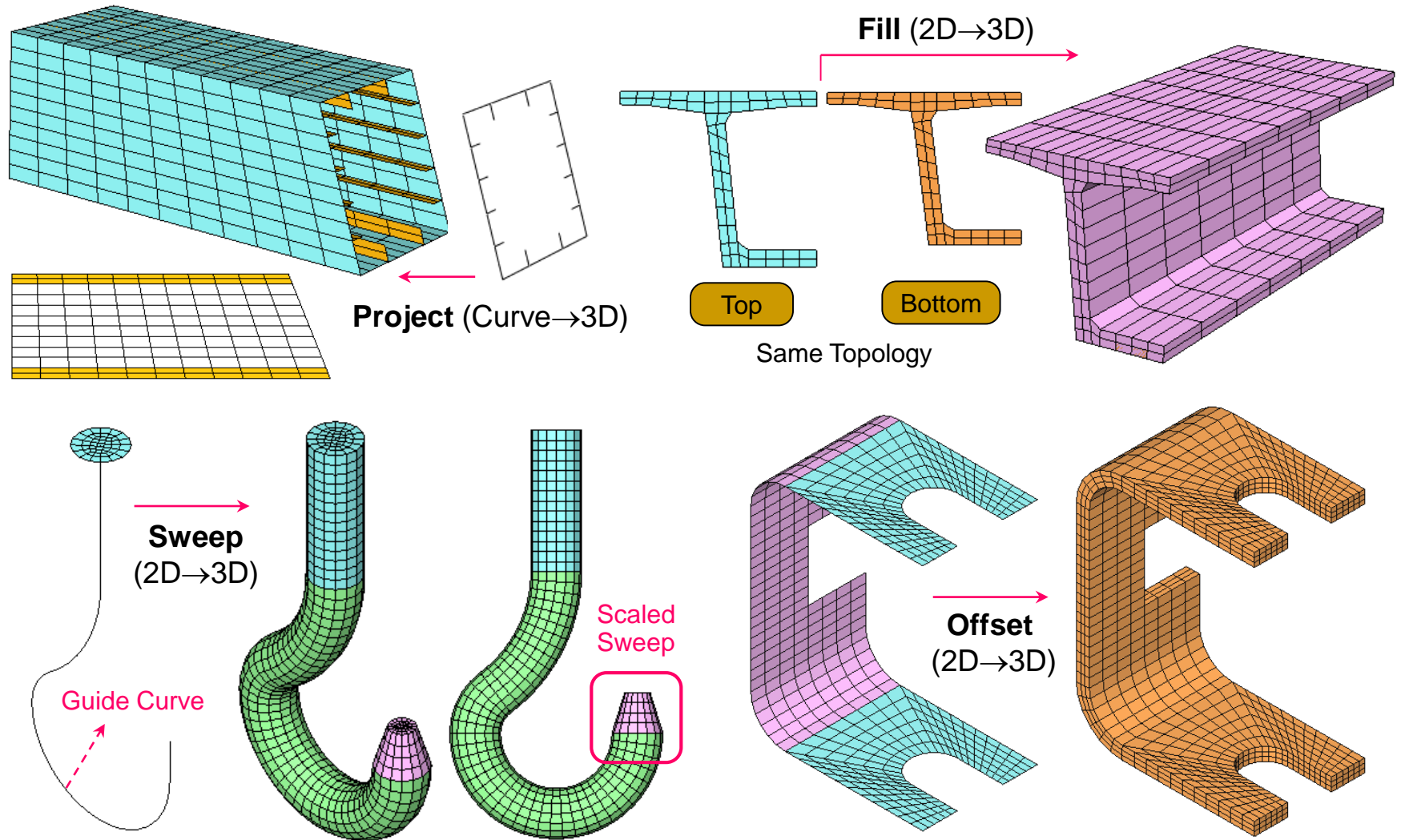
# Mesh Protrusion



**Extrude** thru Node Sequence (Curve→2D)



# Mesh Protrusion





# Element Libraries

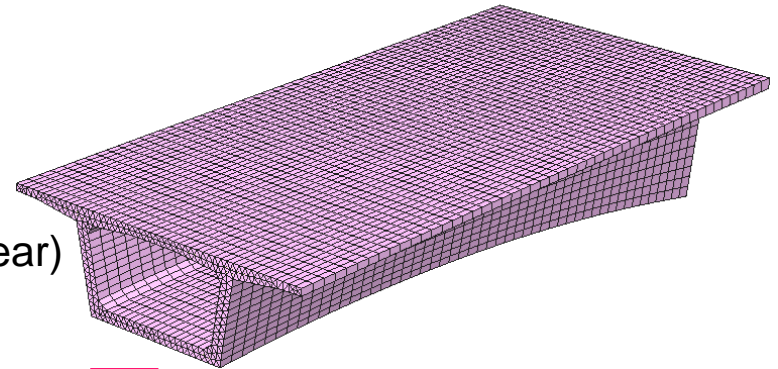
Category	Elements		Order
<b>Structural</b>	<b>1D</b>	Truss	1 <sup>st</sup> , 2 <sup>nd</sup>
		Beam	1 <sup>st</sup> , 2 <sup>nd</sup>
	<b>2D</b>	Plane Stress (Quad / Tria)	1 <sup>st</sup> , 2 <sup>nd</sup>
		Plane Strain (Quad / Tria)	1 <sup>st</sup> , 2 <sup>nd</sup>
		Axisymmetry (Quad / Tria)	1 <sup>st</sup> , 2 <sup>nd</sup>
		Plate (Quad / Tria)	1 <sup>st</sup> , 2 <sup>nd</sup>
		Shell (Quad / Tria)	1 <sup>st</sup> , 2 <sup>nd</sup>
	<b>3D</b>	Hexa / Penta / Tetra	1 <sup>st</sup> , 2 <sup>nd</sup>
<b>Nonstructural</b>	<b>Connection</b>	Elastic Link	-
		Rigid Link	-
	<b>Nodal Mass</b>	Point	-
		Matrix	-
	<b>Interface</b>	3D Point	-
		2D	1 <sup>st</sup> , 2 <sup>nd</sup>
		3D (Quad / Tria)	1 <sup>st</sup> , 2 <sup>nd</sup>
<b>Reinforcement</b>	Embedded Truss / Beam		1 <sup>st</sup> , 2 <sup>nd</sup>
	Tendon		1 <sup>st</sup>
<b>Heat Transfer</b>	1D, 2D, 3D, Cooling Pipe, Thermal Link		1 <sup>st</sup> , 2 <sup>nd</sup>



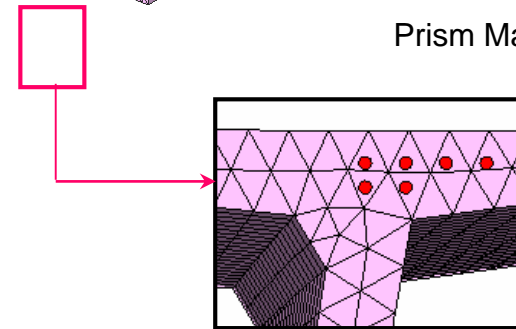
# Reinforcement Elements

## ■ Modeling Method

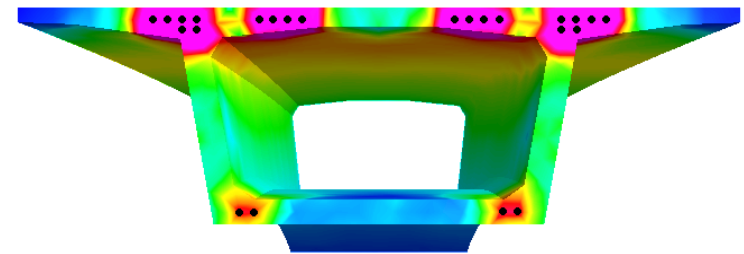
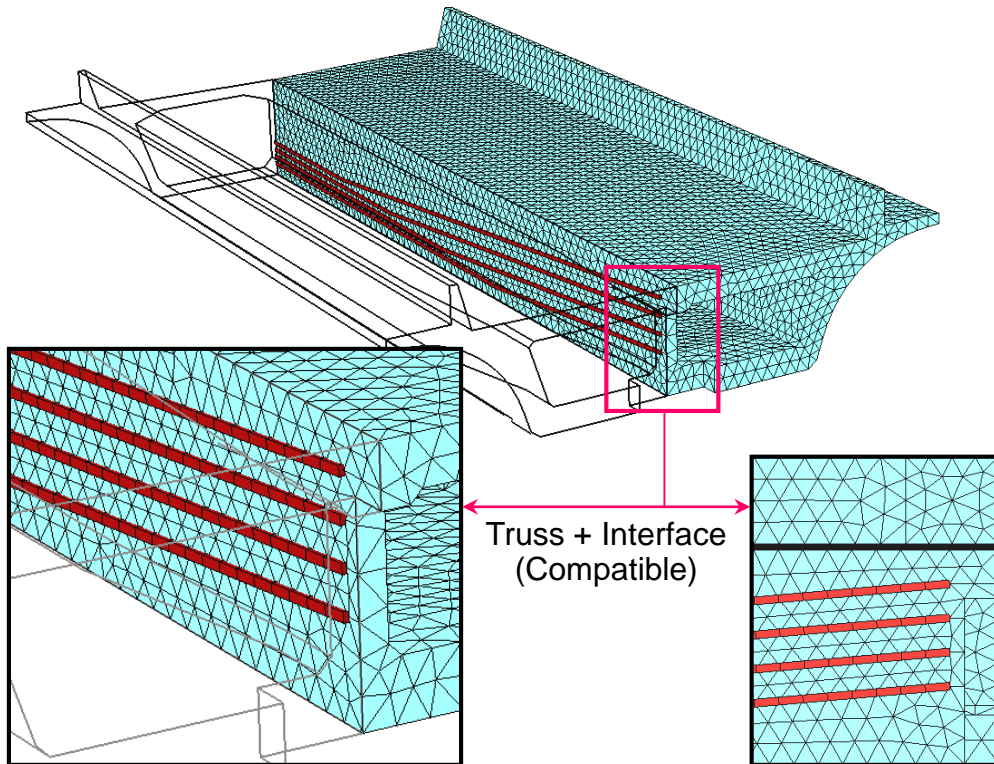
- Embedded Element (Incompatible Mesh)
- Truss (Compatible Mesh) + Interface (Slip, Friction, Cracking, Nonlinear)



Prism Map-Mesh



Embedded Element (Incompatible)

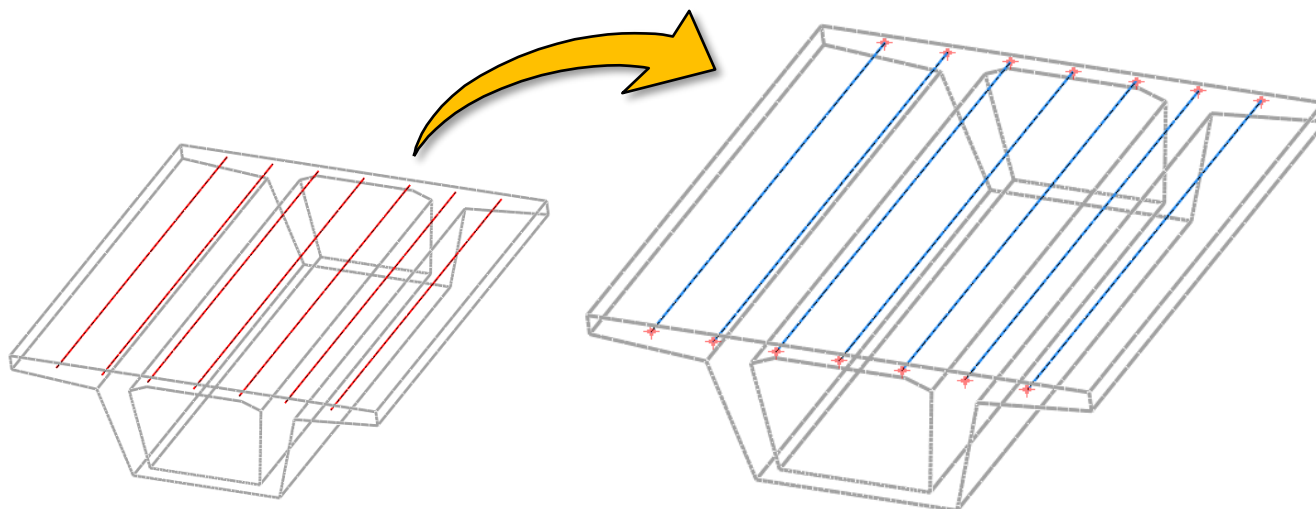


Analysis Result

# Embedded Interface Element

## ■ Embedded Interface Element

- Auto-generation of **embedded line interface element** from the line element embedded in a solid
- Auto-generation of **embedded point interface elements** at each end of the embedded line interface element.



1D Truss or Beam Element

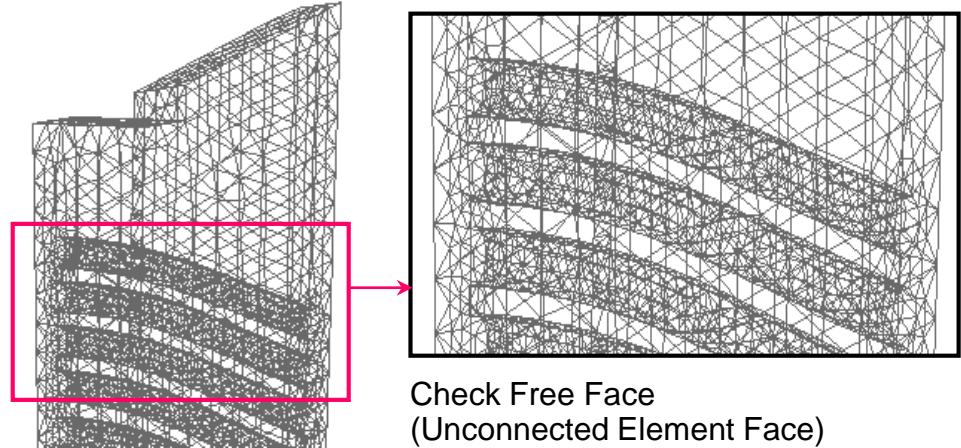


1D Truss or Beam Element  
+  
Embedded Interface Element

# Check & Quality Assurance

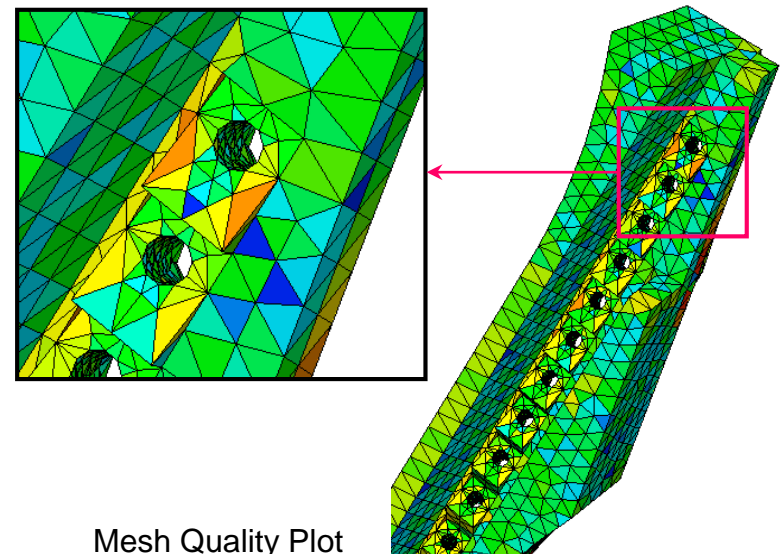
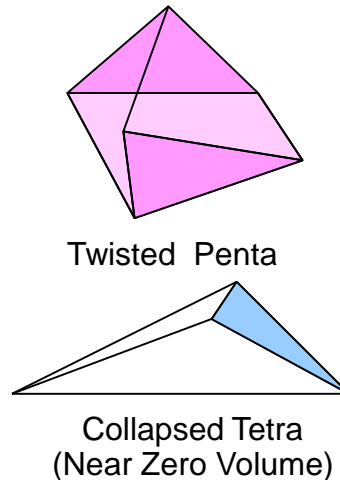
## ■ Check & Verify

- Feature Edges
- Free Faces
- Non-manifold Edges
- Clamped Element
- Overlapped Element(2D)
- Penetrated Element(2D)



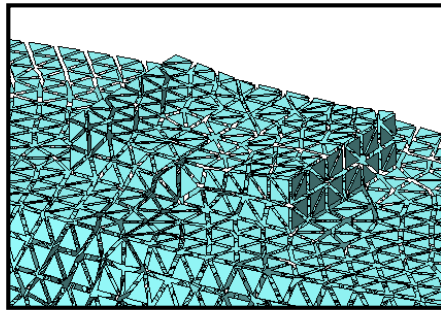
## ■ Quality Assurance

- Aspect Ratio
- Skew Angle
- Taper
- Warpage
- Jacobian Ratio
- Twist
- Length

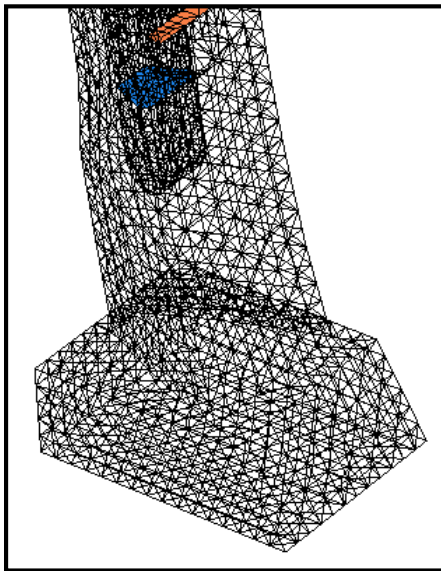




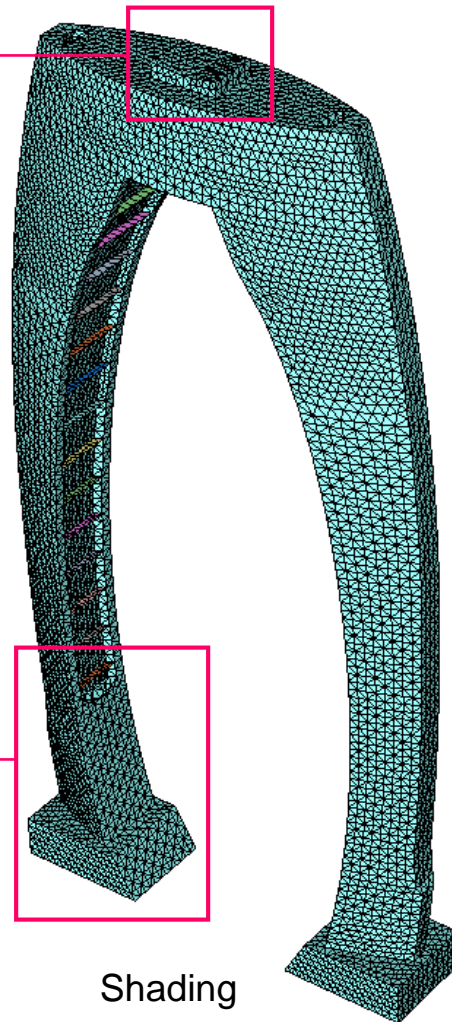
# Graphic Display - Mesh



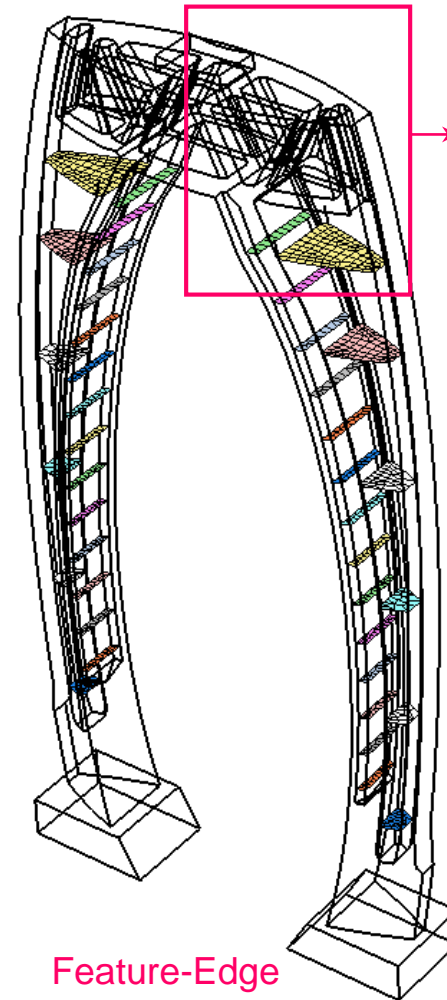
Shrink



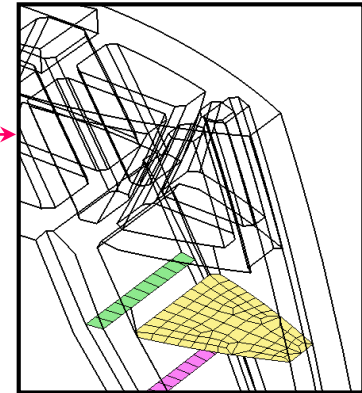
Wireframe (Free-Face)



Shading

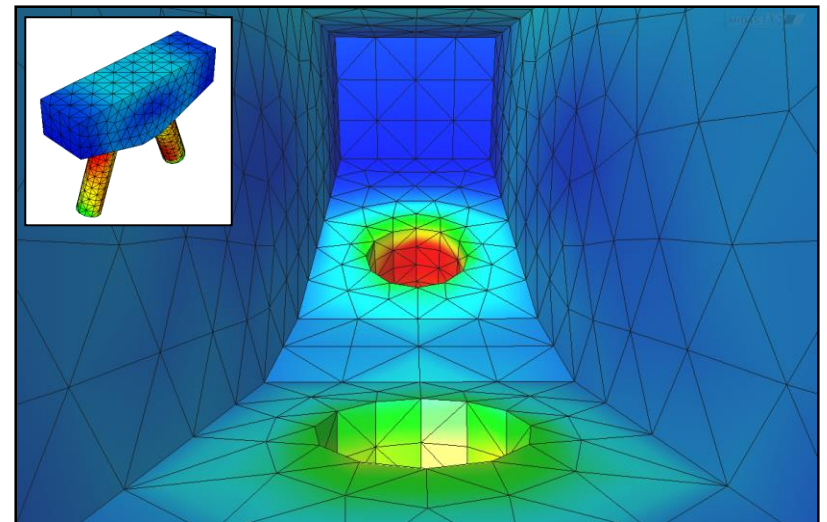
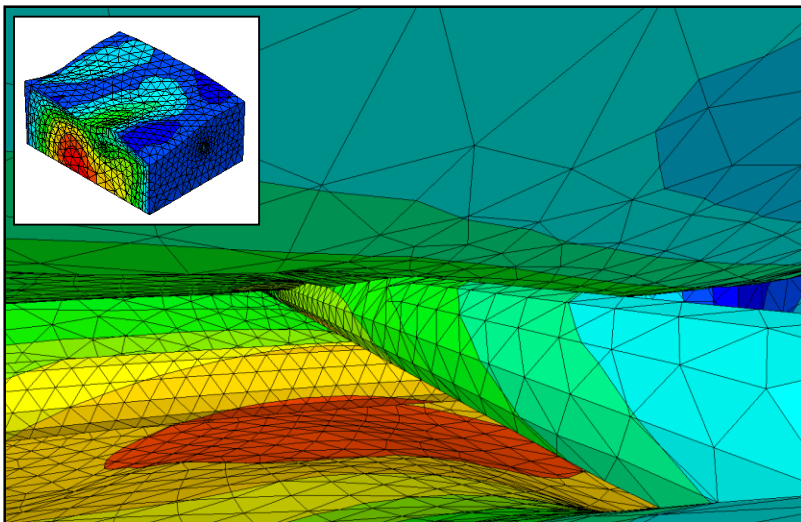
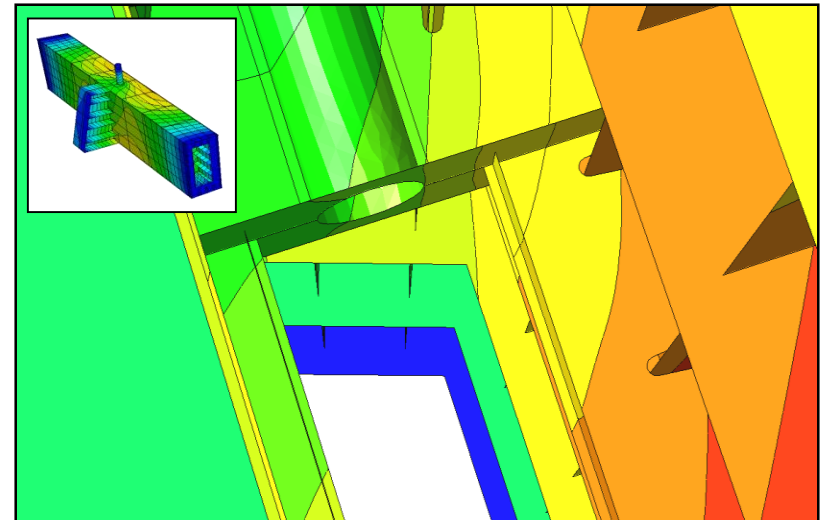
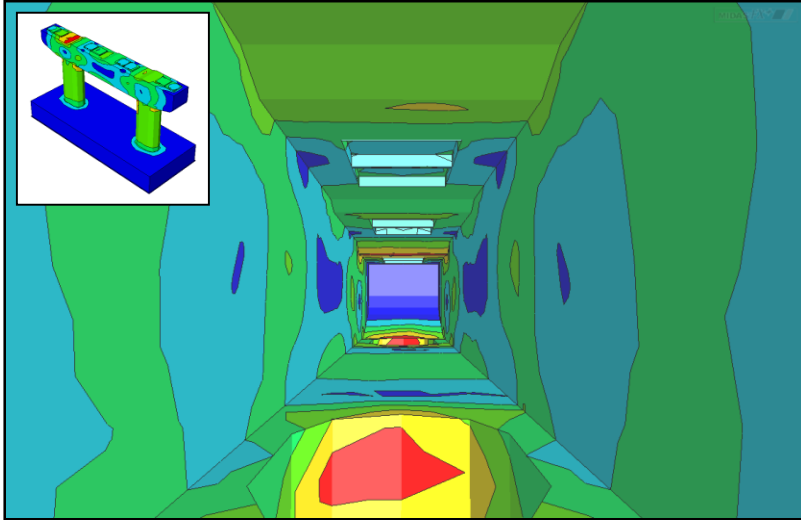


Feature-Edge



Feature-Edge

# Flying View

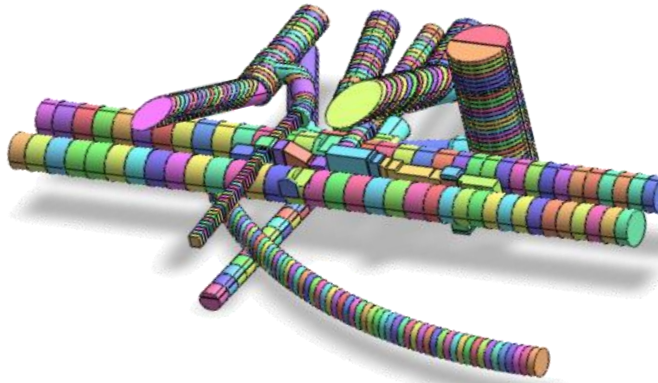




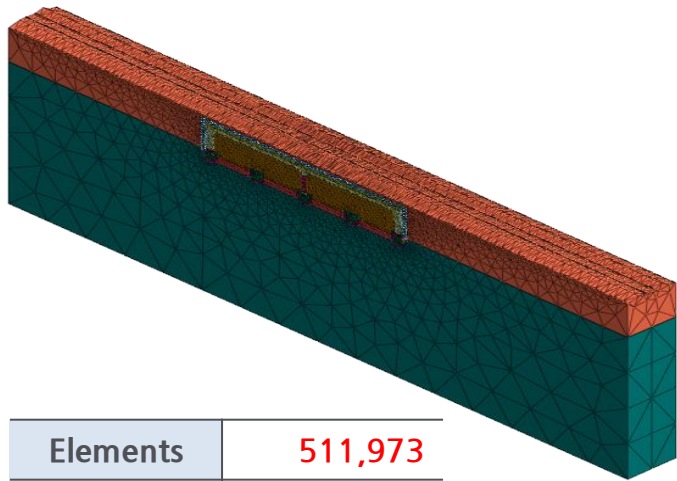
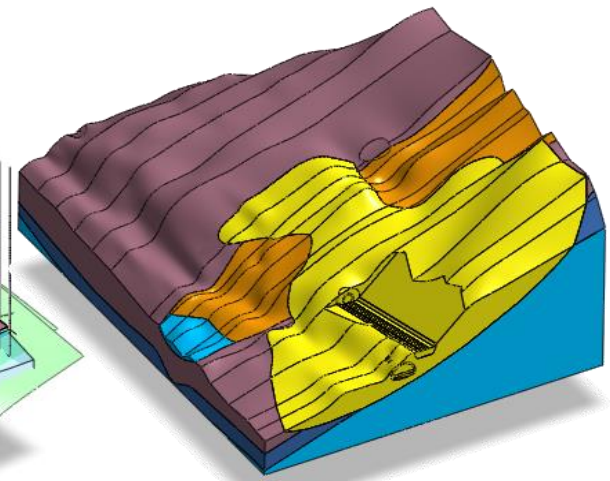
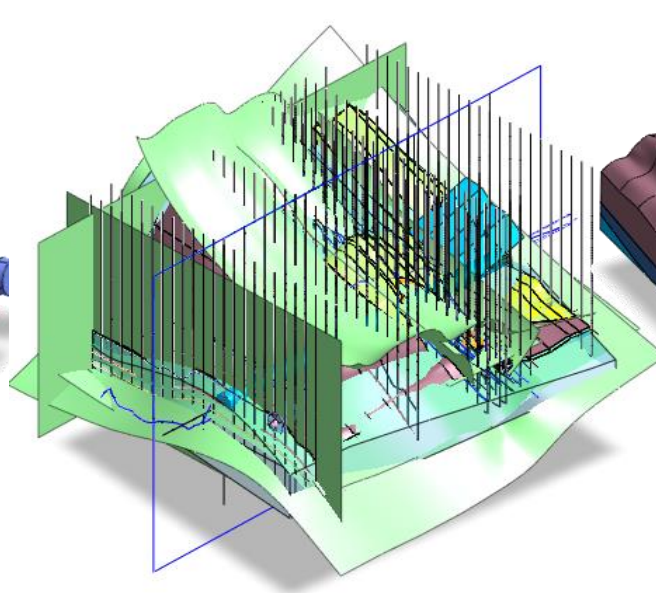
# Contents

1. Overview
2. Geometry Modeling
3. Mesh Generation
- 4. Analysis**
5. Post-processing

# Support and operation for large-scale models



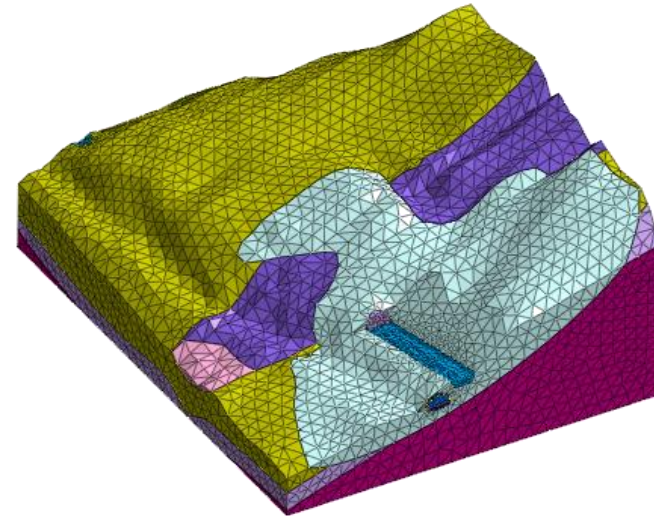
Solids	526
--------	-----



Elements	511,973
----------	---------

Nodes	80,903
-------	--------

Solids	3,250
Elements	394,100
Nodes	75,112
Mesh sets	1,004
No. of stages	700



New framework for 64-bit and improvement in functions through new graphics engine

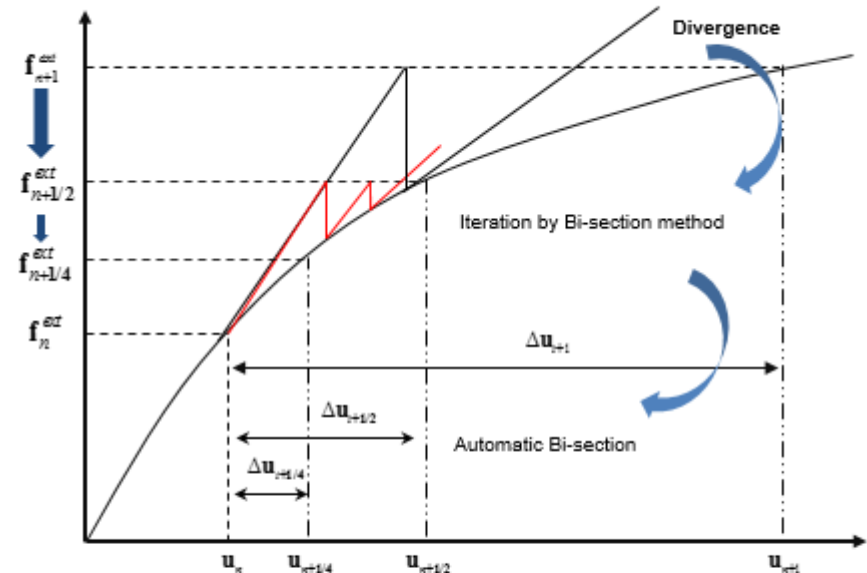
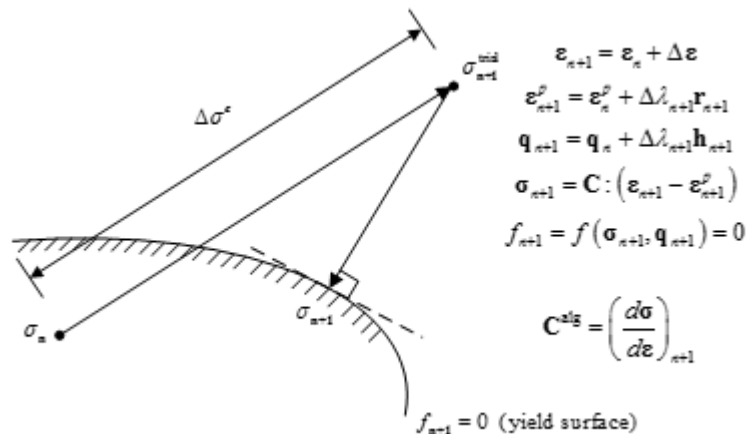
# Support and operation for large-scale models

## Bi-section method

- Automatic load step
- Stable stress convergence regardless of the magnitude of applied load

## Consistent tangent matrix

- Faster convergence speed of nonlinear analysis



# Linear Static Analysis

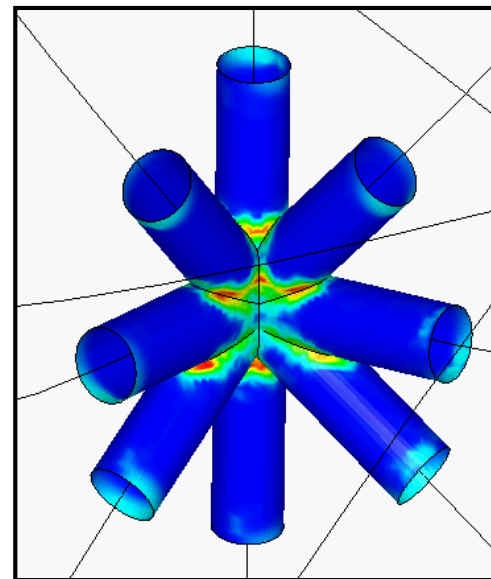
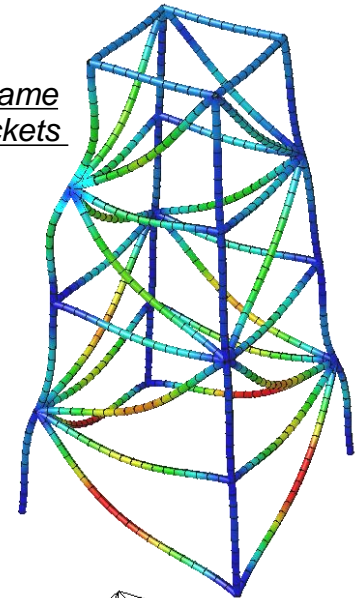
## ■ Linear Static Analysis

- Multiple Load Cases
- Result Combination and Transformation

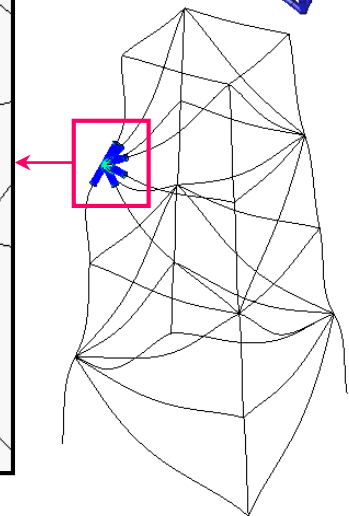
## ■ Equation Solvers

- Direct Solvers
  - Multi-frontal Sparse Gaussian Solver
  - Dense
  - AMG

*Offshore Platform / Steel Frame  
Composed of Cylindrical Jackets  
(Plate + Frame)*



*Stress Distribution of Jacket*

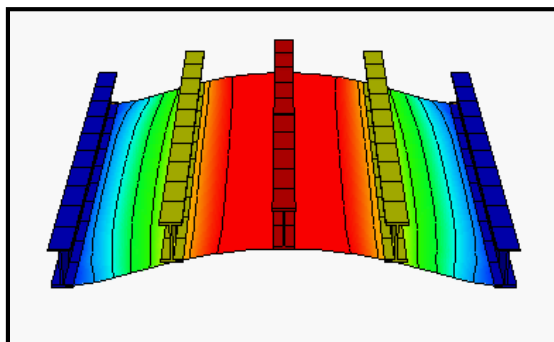




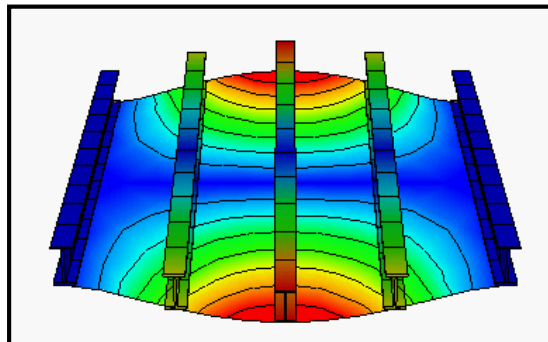
# Eigenvalue Analysis

## ■ Modal Analysis

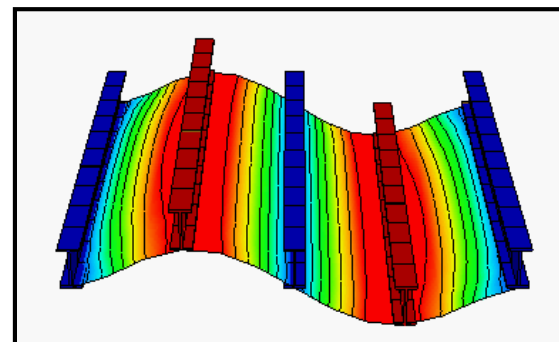
- Lanczos Method



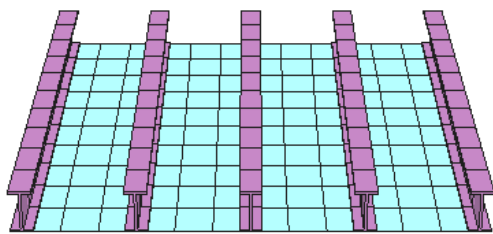
1<sup>st</sup> Mode (64.58 Hz)



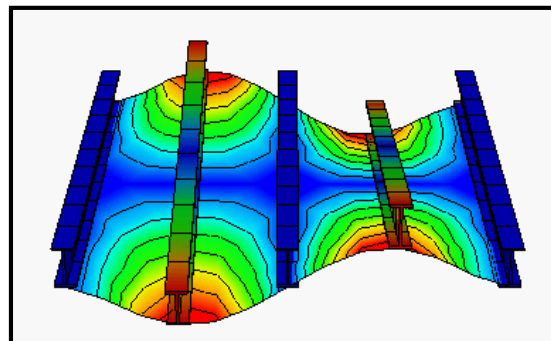
2<sup>nd</sup> Mode (106.05 Hz)



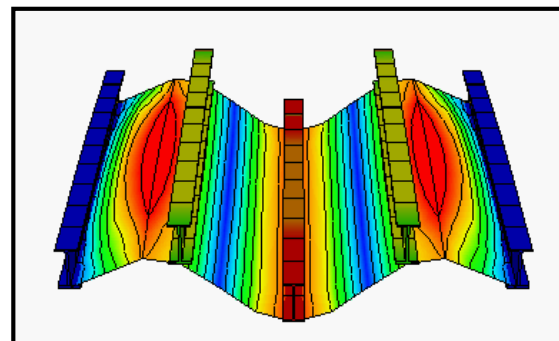
3<sup>rd</sup> Mode (208.96 Hz)



Simply Supported Stiffened Plate  
(Plate + Beam)



4<sup>th</sup> Mode (270.00 Hz)



5<sup>th</sup> Mode (440.58 Hz)

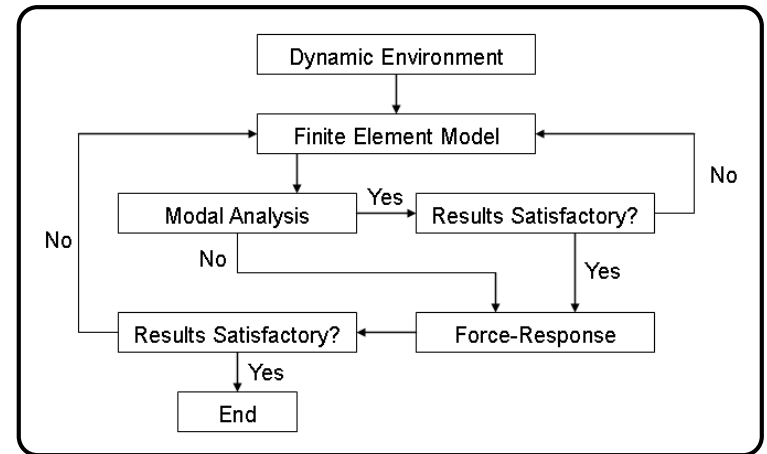
## ■ Linear Buckling Analysis

- Critical Buckling Modes
- Buckling Modes
- Load Combination

# Dynamic Analysis

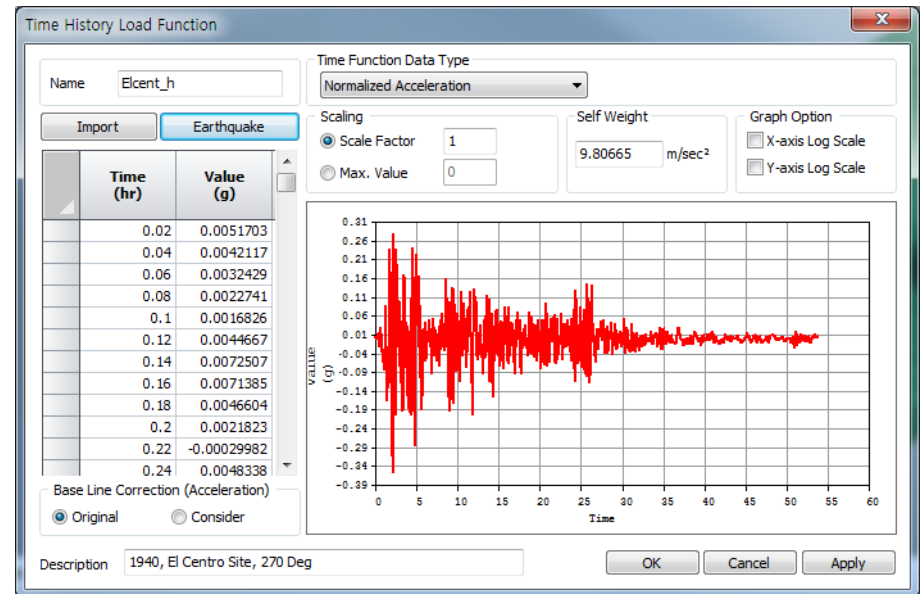
## ■ Transient Response Analysis

- Direct Transient Response
- Modal Transient Response
- Time Forcing Function DB  
(54 Earthquake Acceleration Records)



## ■ Spectrum Response Analysis

- SRSS, CQC, ABS, NRL, TENP
- Design Spectrum DB



# Material Nonlinearity Analysis

## ■ Material Models

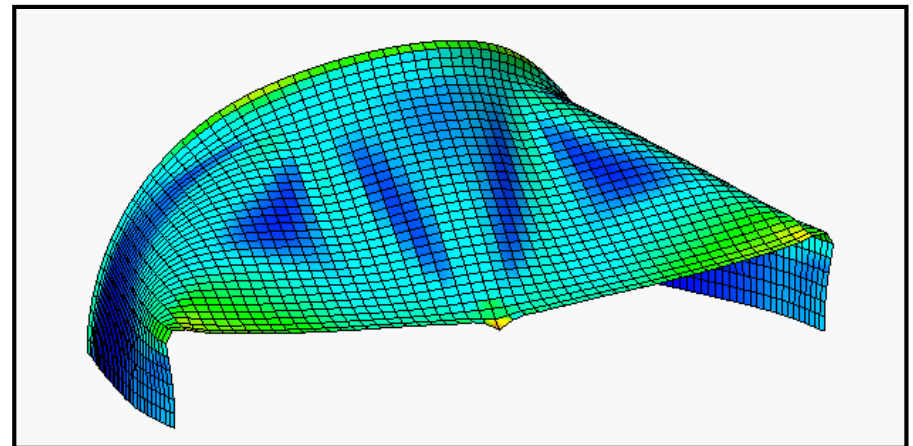
- von Mises
- Tresca
- Mohr-Coulomb
- Drucker-Prager
- Rankine
- Hoek Brown
- Hyperbolic
- Strain Softening
- Cam Clay
- Modified Cam Clay
- Jardine
- D-min
- Soft Soil
- Concrete Smeared Crack
- Masonry
- Hardening Soil
- User-Supplied Material

## ■ Nonlinear Behaviours

- Hardening (Isotropic)
- Softening

## ■ Iteration Methods

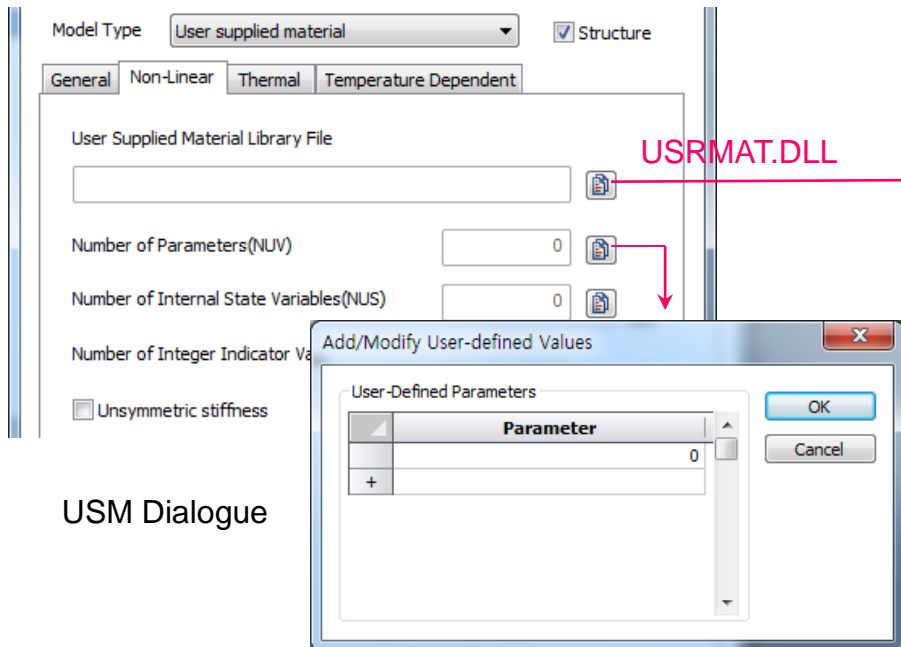
- Full Newton-Raphson (*Auto Load Step*)
- Modified Newton-Raphson
- Arc-Length Method
- Initial Stiffness
- Quasi-Newton(Secant)



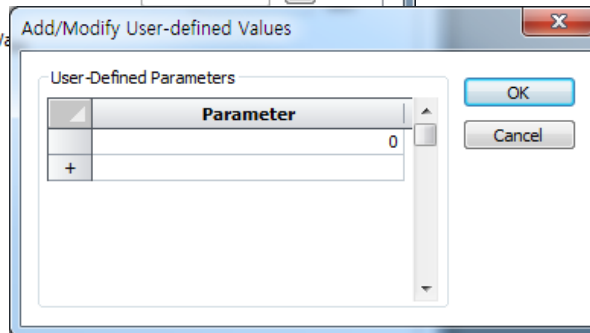
*Pinched Cylinder (Plate) – von Mises  
Material & Geometry Nonlinear Analysis*

# User-Supplied Materials

- In FEA NX, users can use their own defined material models via **Fortran-coded library** file.
- FEA NX's user-supplied material model supports **nonlinear elastic** and **elasto-plastic** behaviours.
- User-supplied material can be used seamlessly with all elements which allow material nonlinear behaviours.



USM Dialogue



User-defined Parameters Input Dialogue

```

*****
!      USER SUPPLIED MATERIAL SUBROUTINE
*****
SUBROUTINE USRMAT( EPS0, DEPS, NS, IELEMEN, INTPT, COORD, &
                  SE, ITER, USRVAL, NUV, USRSTA, NUS, IUSRIND, &
                  NUI, SIG, STIFF, ID)

IMPLICIT NONE

!DEC$ ATTRIBUTES DLLEXPORT::USRMAT

INTEGER, INTENT(IN)      :: NS
INTEGER, INTENT(IN)      :: IELEMEN
INTEGER, INTENT(IN)      :: INTPT
INTEGER, INTENT(IN)      :: ITER
INTEGER, INTENT(IN)      :: ID
INTEGER, INTENT(IN)      :: NUV
INTEGER, INTENT(IN)      :: NUS
INTEGER, INTENT(IN)      :: NUI
REAL*8, INTENT(IN)       :: EPS0(NS)
REAL*8, INTENT(IN)       :: DEPS(NS)
REAL*8, INTENT(IN)       :: COORD(3)
REAL*8, INTENT(IN)       :: SE(NS, NS)
REAL*8, INTENT(INOUT)    :: SIG(NS)
REAL*8, INTENT(INOUT)    :: STIFF(NS, NS)
REAL*8, INTENT(INOUT)    :: USRSTA(NUS)
REAL*8, INTENT(IN)       :: USRVAL(NUV)
INTEGER, INTENT(INOUT)   :: IUSRIND(NUI)

INTEGER :: I, J
REAL*8  :: EMOD, EPS(NS)

SIG(1:NS) = 0.0D0
EMOD      = USRVAL(1)
EPS(1:NS) = EPS0(1:NS) + DEPS(1:NS)

-----
TOTAL STRESS

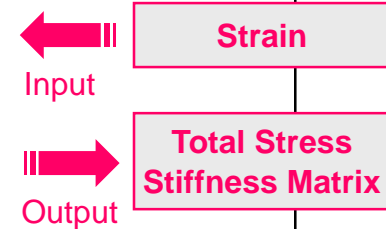
SIG(1) = EMOD * EPS(1) + 1000.0D0 * EMOD * EPS(1)**2
SIG(2) = EMOD * EPS(2) + 1000.0D0 * EMOD * EPS(2)**2
SIG(3) = EMOD * EPS(3) + 1000.0D0 * EMOD * EPS(3)**2
SIG(4) = (EMOD * EPS(4)) / 2.0D0
SIG(5) = (EMOD * EPS(5)) / 2.0D0
SIG(6) = (EMOD * EPS(6)) / 2.0D0

-----
MATERIAL STIFFNESS MATRIX

STIFF(1,1) = EMOD + 2000.0D0 * EMOD * EPS(1)
STIFF(2,2) = EMOD + 2000.0D0 * EMOD * EPS(2)
STIFF(3,3) = EMOD + 2000.0D0 * EMOD * EPS(3)
STIFF(4,4) = 0.5D0 * EMOD
STIFF(5,5) = 0.5D0 * EMOD
STIFF(6,6) = 0.5D0 * EMOD

RETURN

END
*****
    
```



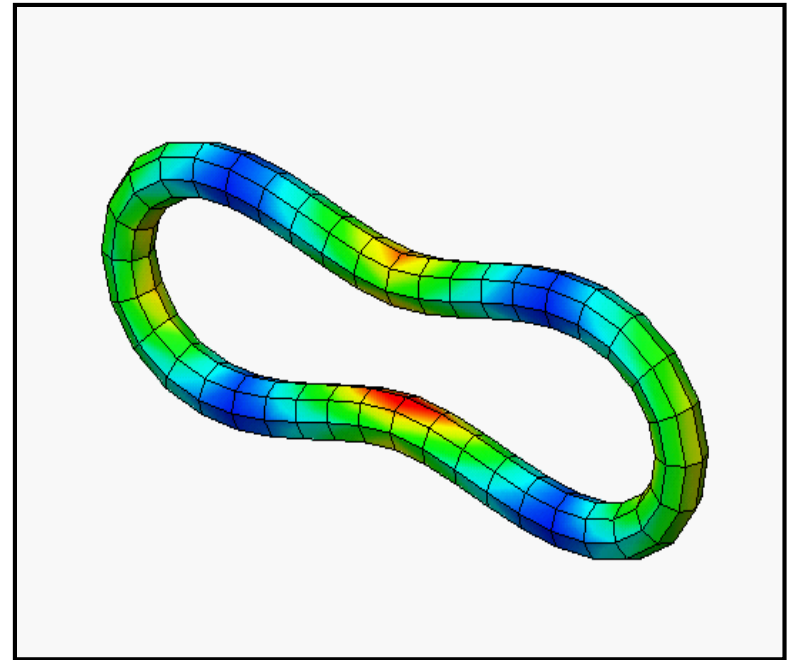
<Ex> Nonlinear Elastic Material for Solid Element



# Geometry Nonlinearity Analysis

## ■ Iteration Methods

- Full Newton-Raphson (*Auto Load Step*)
- Modified Newton-Raphson
- Arc-Length Method
- Initial Stiffness
- Quasi-Newton(Secant)

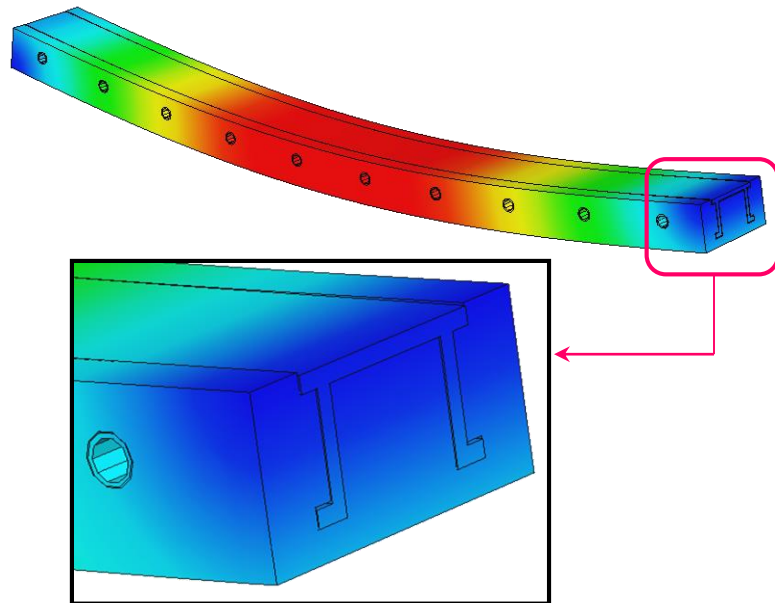


Ring (Solid) – Total Lagrangian

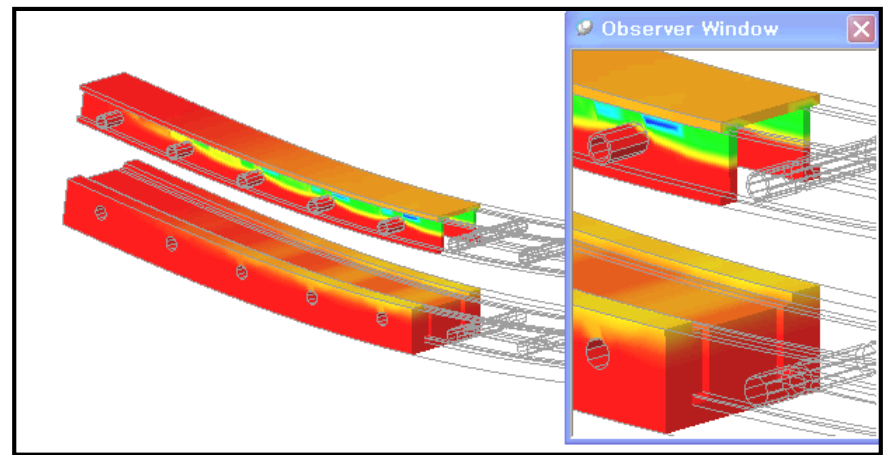
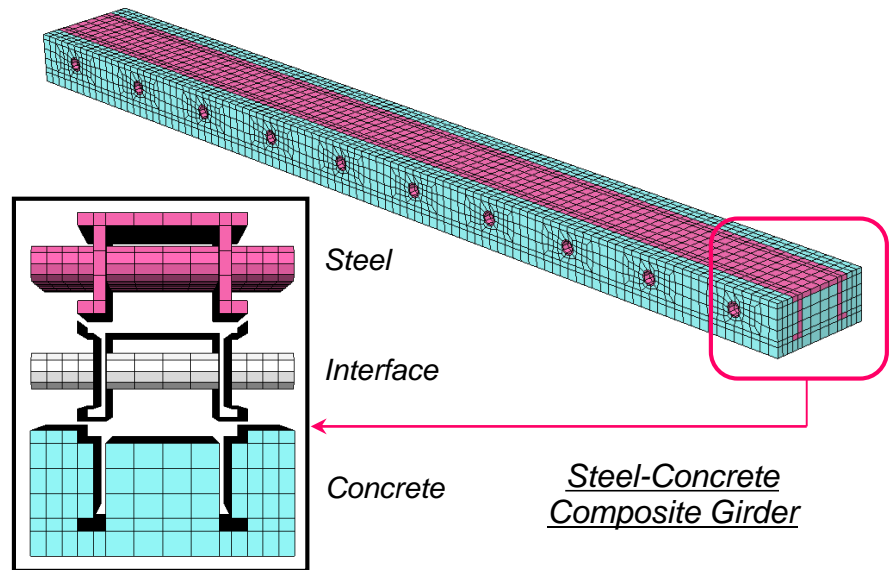
# Interface Nonlinearity Analysis

## ■ Interface Models

- Coulomb Friction
- Discrete Cracking
- Bond-Slip
- Nonlinear Elastic
- Combined (Cracking-Shearing-Crushing)



*Deformation (Discontinuity btwn Steel & Concrete)*



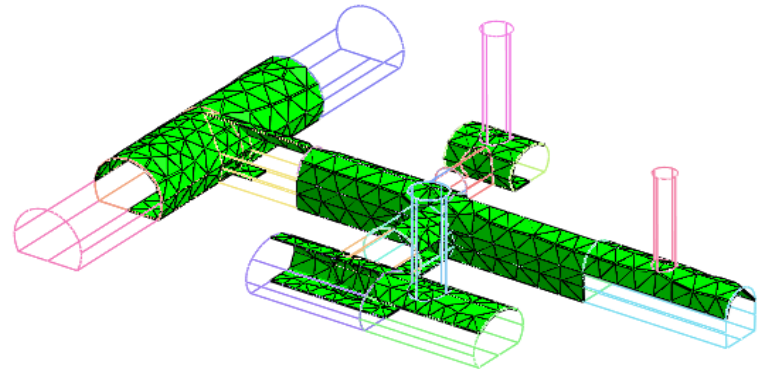
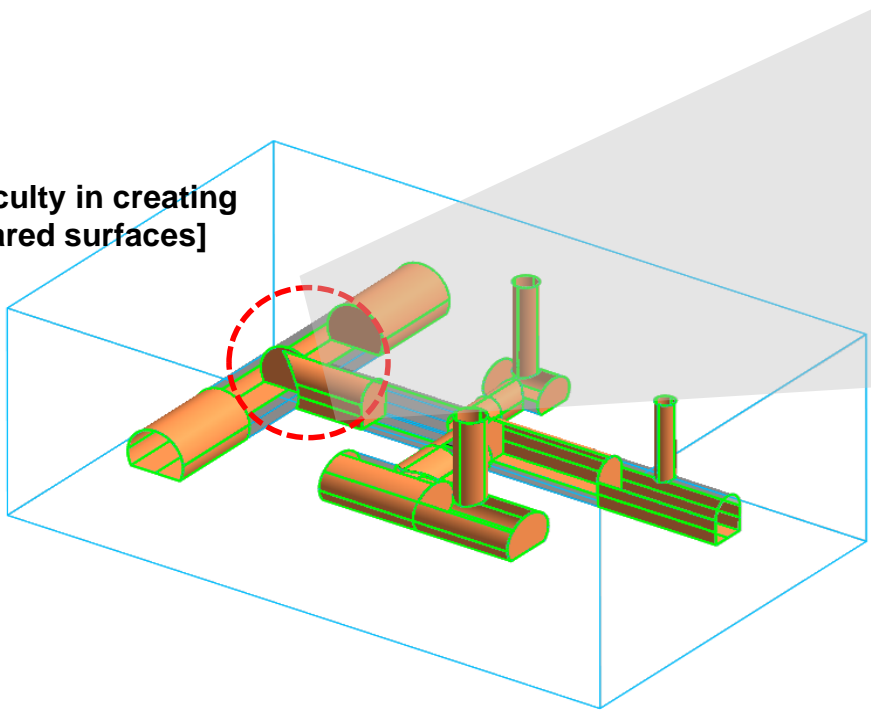
*Principal Stress (Virtually Transformed & Clipped View)*

# Auto Contact

## ■ Contact Type

- Welded
- General
- Bi-directional Sliding Contact
- Rough
- Breaking-Weld

[Difficulty in creating shared surfaces]



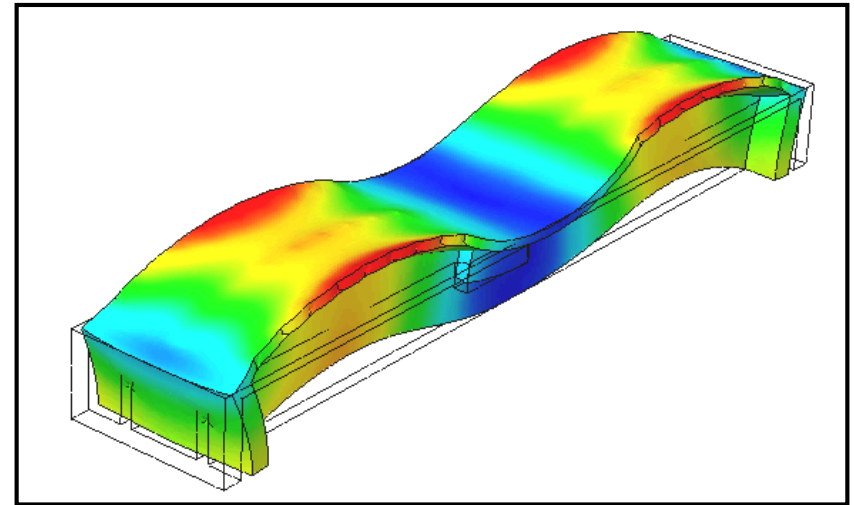
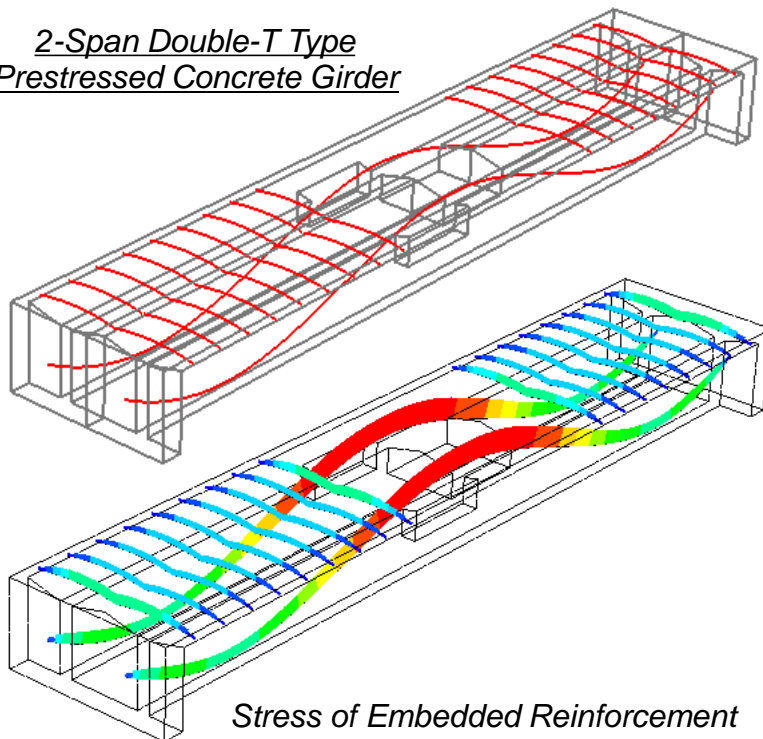
[Automatic search for free faces and endowment of conditions for rigid contact]

# Reinforcement Analysis

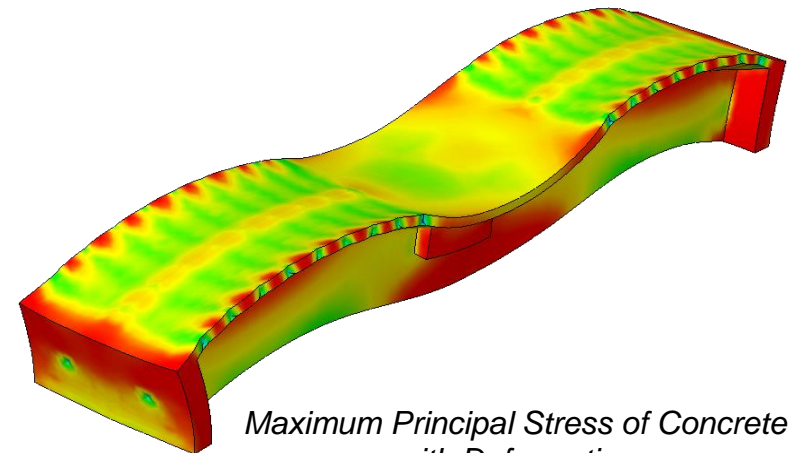
## ■ Reinforcement

- Embedded Truss/Beam
- Truss + Interface (Slip/Friction)

*2-Span Double-T Type  
Prestressed Concrete Girder*



*Deformation*



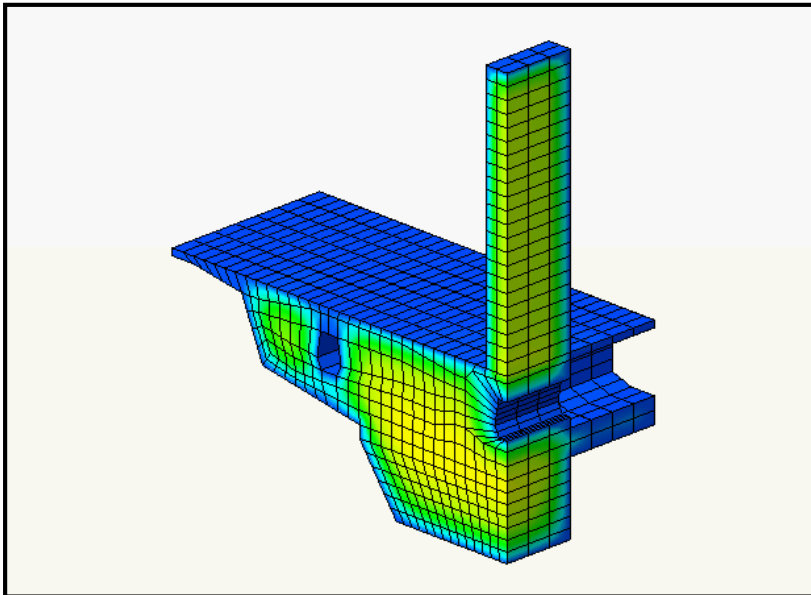
*Maximum Principal Stress of Concrete  
with Deformation*



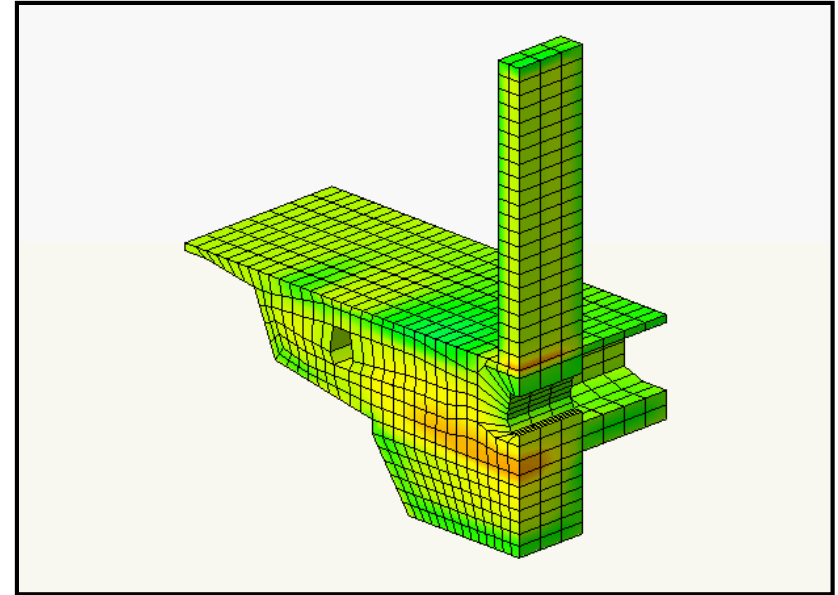
# Heat Transfer and Heat of Hydration Analysis

## ■ Heat Transfer

- Steady-State & Transient
- Conduction, Convection,
- Cooling Pipe
- Heat Flux
- Heat Flow
- Temperature Gradient Display

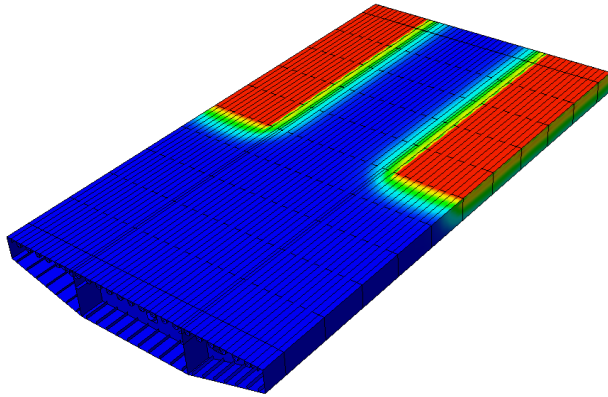


Pier Table (Construction Stage) - Temperature

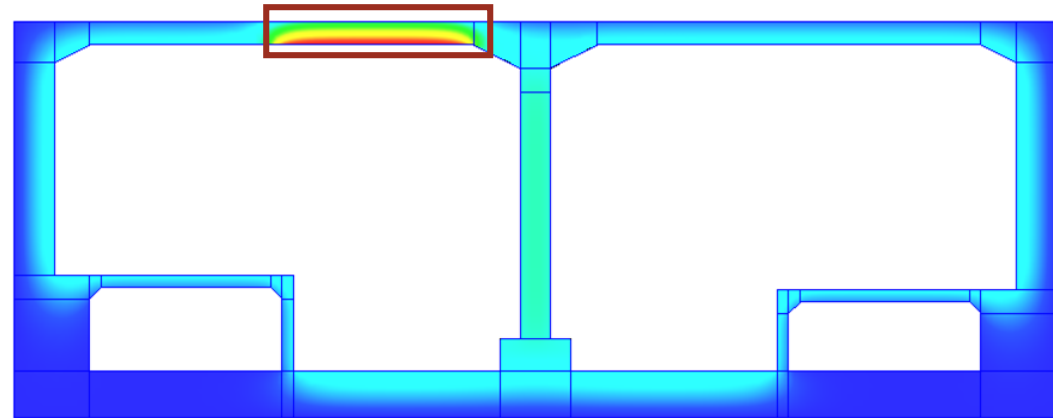


Pier Table (Construction Stage) - Stress

# Heat Transfer and Heat of Hydration Analysis

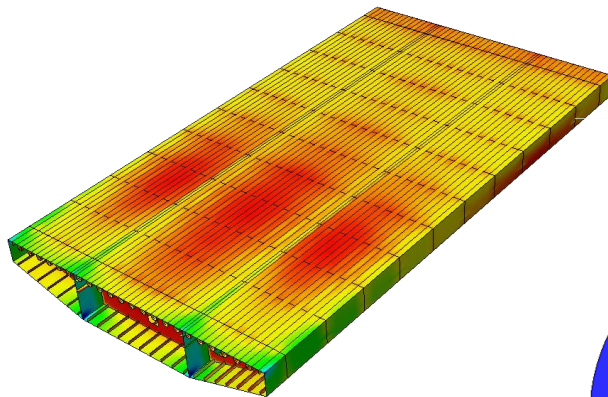


Temperature at Top Plate

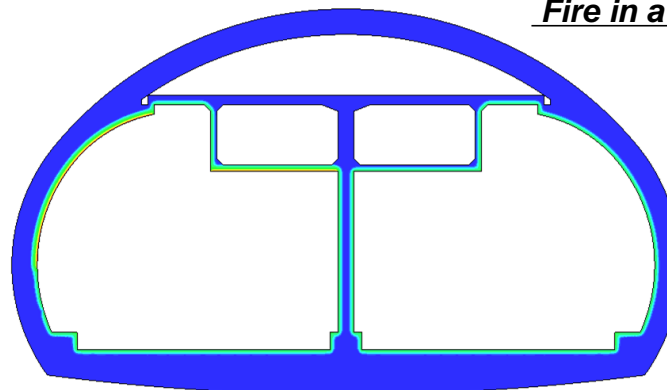


Temperature Distribution

Fire in a Subway Structure

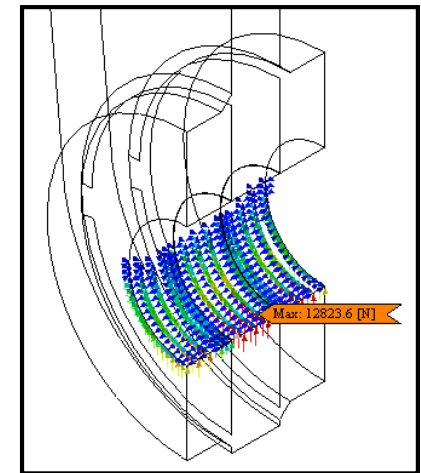
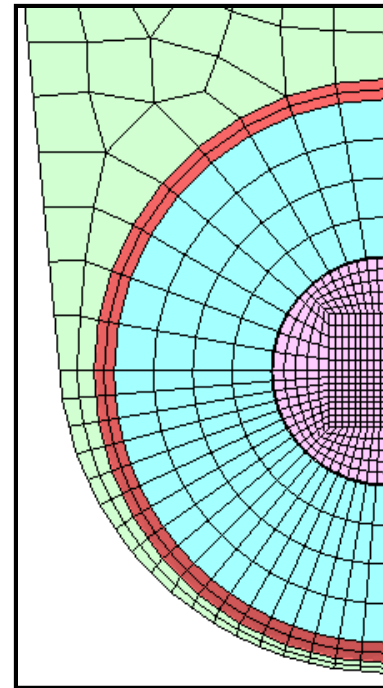
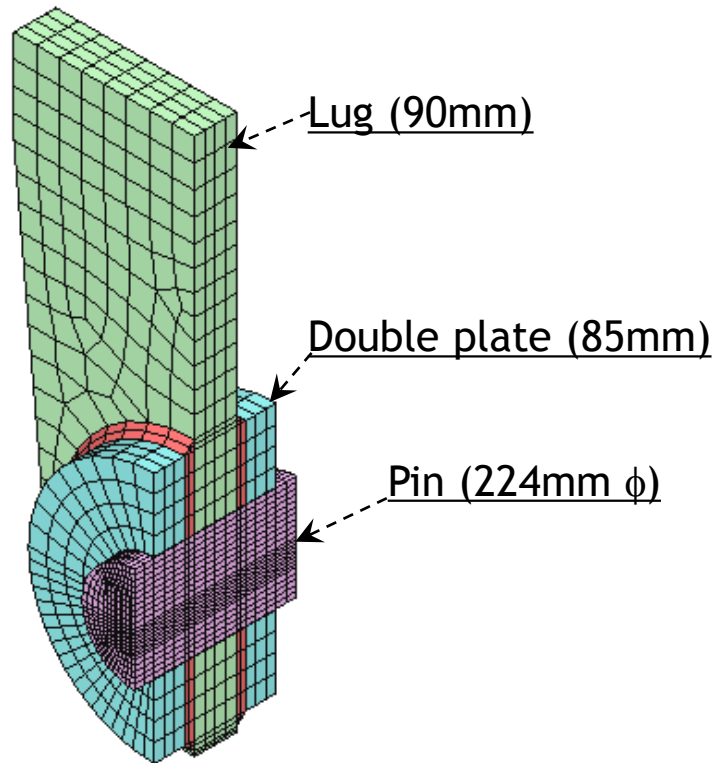


Guss Asphalt Pavement

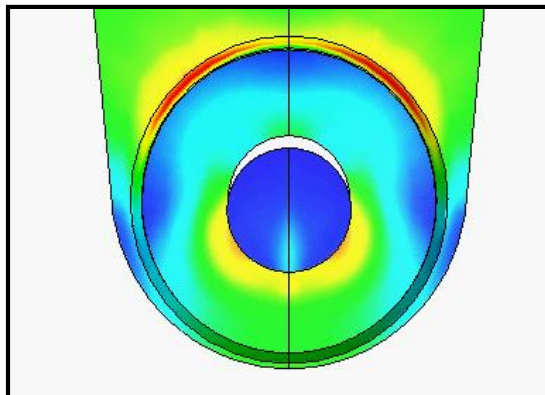


Thermal Stress

# Static Contact Analysis

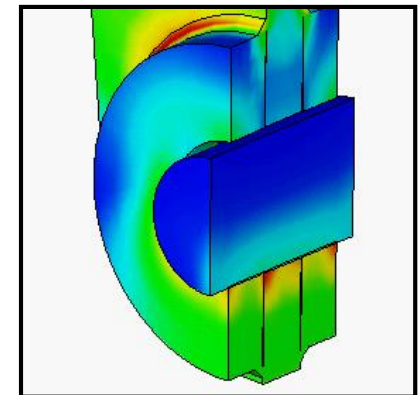


*Contact Force*



*Hanger of Arch Bridge (Lug-Pin Joint)*

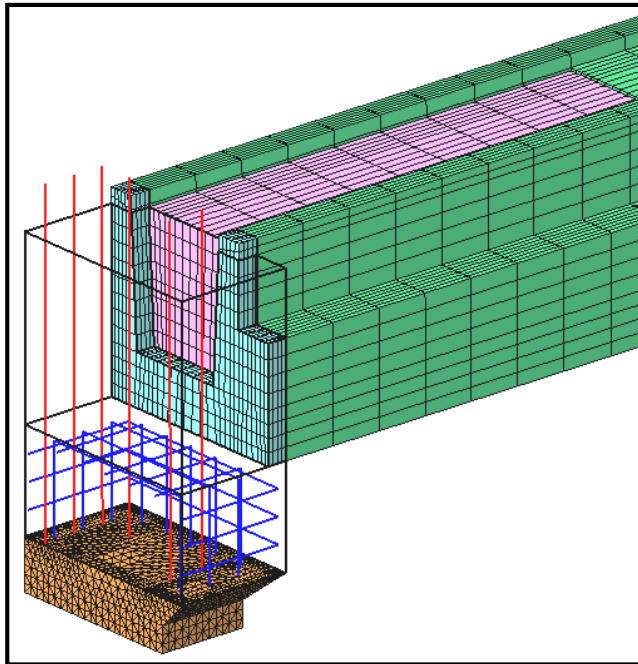
*Deformation and Stress Distribution by Contact*



# Cracking Analysis (1)

## ■ Cracking Models

- Smeared
- Crack Index

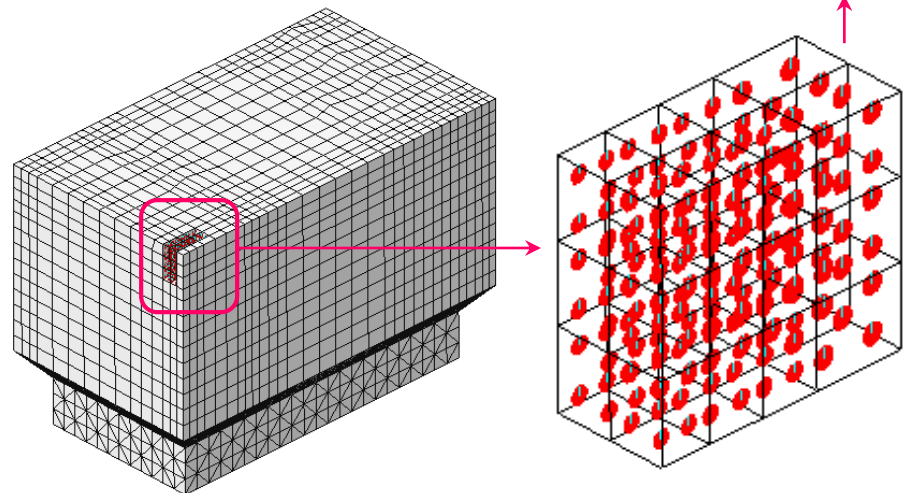
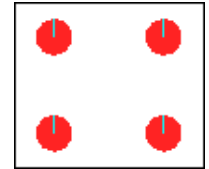


*Steel Reinforced Concrete Bracket*

## ■ Results

- Crack Pattern (Crack Stress/Strain)
- Element Status
  - Cracking: Partially/Fully Open, Closed, Not Yet
  - Plasticity: Previously Plastic, Elastic, Plastic, Critical
  - Contact: No Contact, Slip, Stick

- Symbols at Gauss Points
- Disc Normal: Opening Direction
- Disc Color : Magnitude
- Line : Shearing Direction

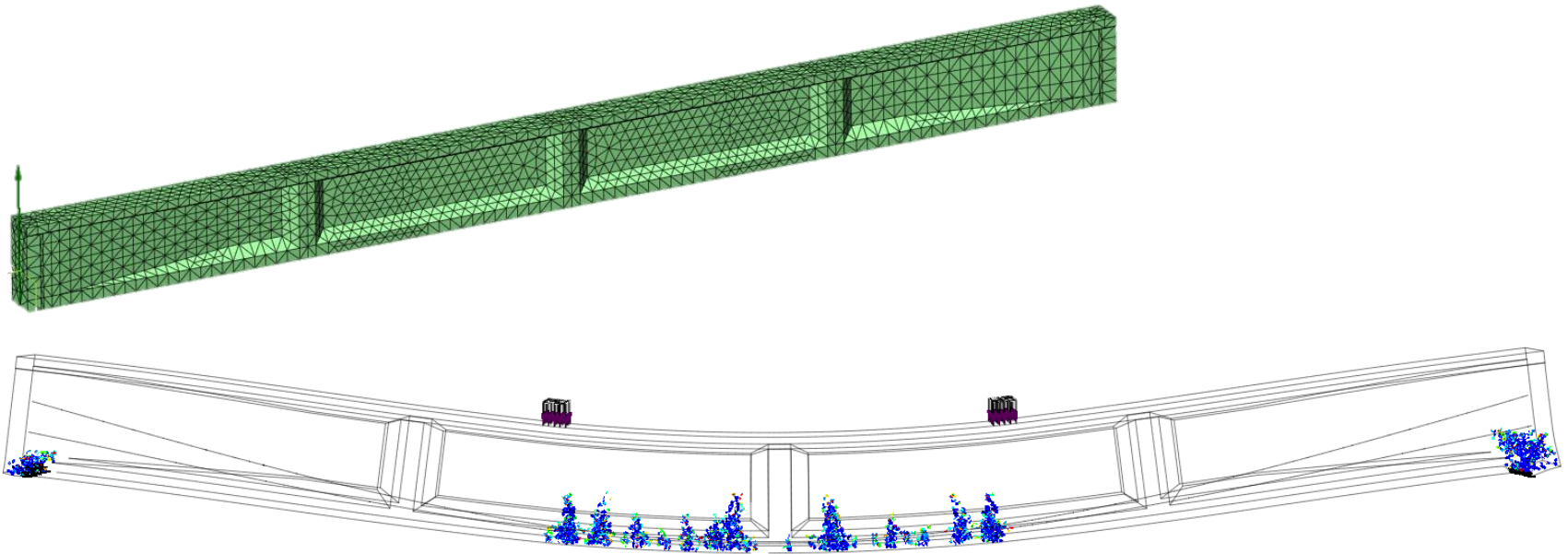


*Crack Pattern (Disc Plot)*



# Cracking Analysis (2)

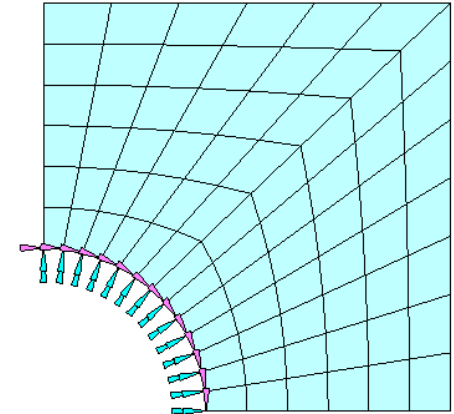
- **Crack width calculation in the nonlinear analysis of reinforced concrete elements**
  - In concrete Smeard Crack model, the crack width can be determined as the product of the crack-band width ( $h$ ) and the difference of crack direction strain and crack direction stress divided by the original Young's modulus.



# Load & B.C.

## ■ Loadings

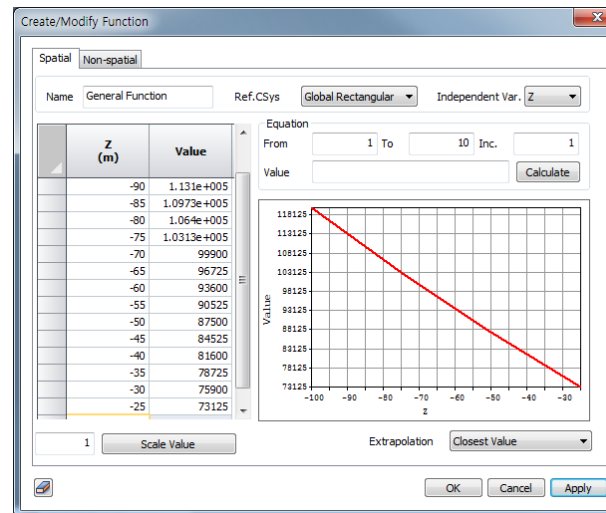
- Self Weight
- Force / Moment
- Displacement
- Pressure
- Beam Load
- Prestress
- Temperature
- Initial Equilibrium Force
- Heat Flux
- Pipe Cooling
- Time Forcing Function
- Time Varying Load
- Ground Acceleration
- Response Spectrum Function



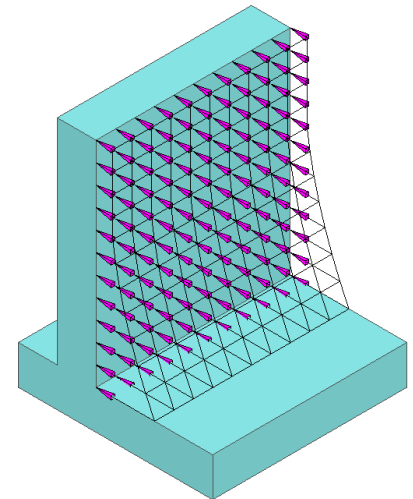
Constraint based-on CSys.

## ■ Boundary Conditions

- Constraint
- Constraint Equation
- Contact Conditions
- Convection

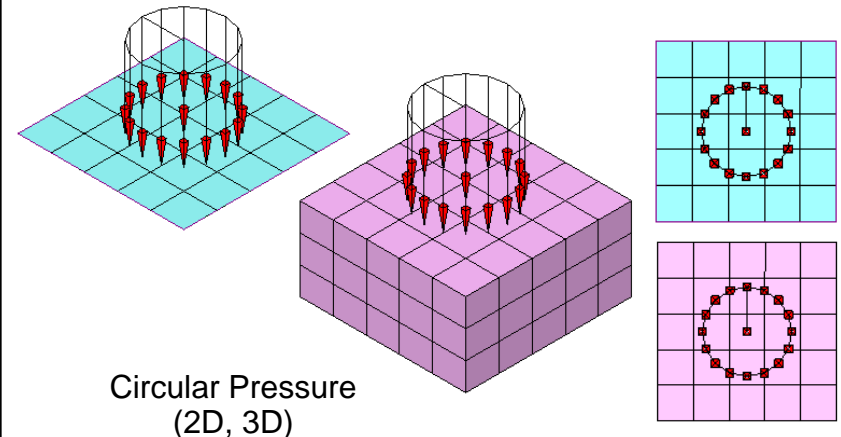
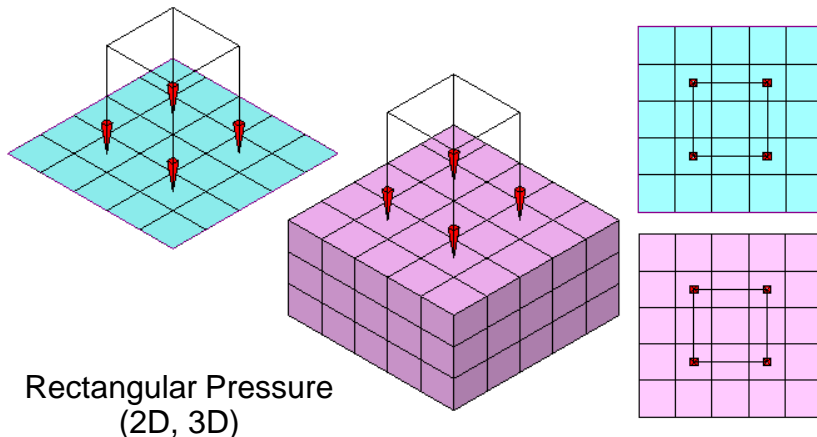
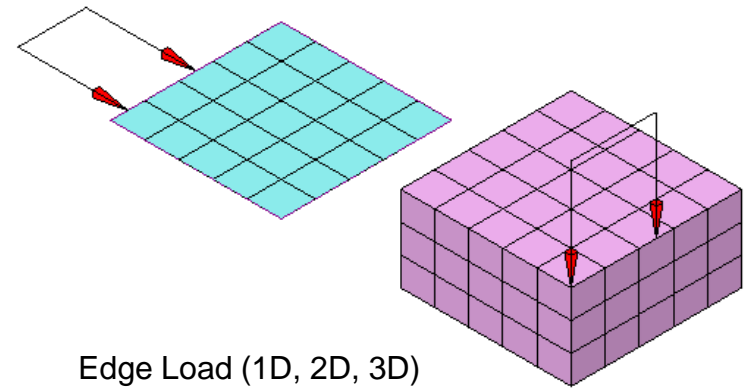
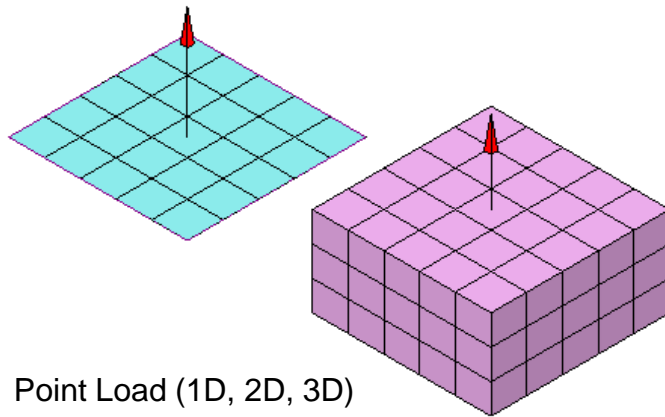


Spatially Varying Pressure (Function Applied)

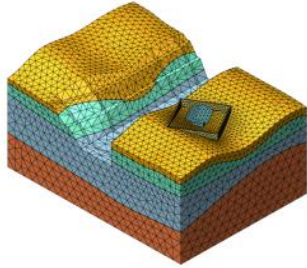


# Arbitrary Loading

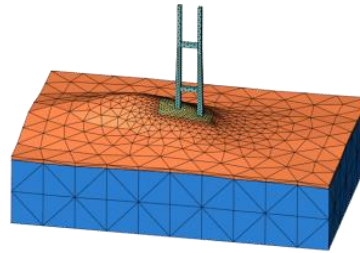
- FEA provides arbitrary loading function which can be applied to arbitrary locations/areas regardless of node and/or element connection.



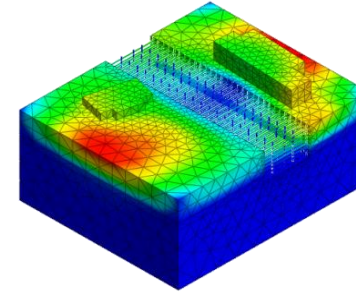
# Geotechnical analysis



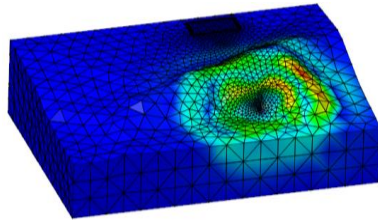
- Strength Reduction Method (SRM)
- Strength Analysis Method (SAM)
- Construction stages Slope stability (SRM/SAM)



- Linear static analysis
- Nonlinear static analysis



- Stress (drained/undrained) analysis
- Seepage analysis for each stage
- **Stress-seepage- slope coupled**
- Consolidation analysis for each stage
- **Fully coupled stress & seepage**



- Eigenvalue/Reaction Spectrum analysis
- Linear Time History (mode/direct methods)
- **Nonlinear Time History analysis**
- 1D/2D Equivalency Linear analysis
- **Nonlinear time history + SRM Coupled**

Slope  
Stability  
Analysis

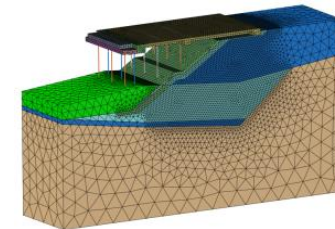
Static  
Analysis

Construction  
Stage  
Analysis

**FEA  
NX**

Dynamic  
Analysis

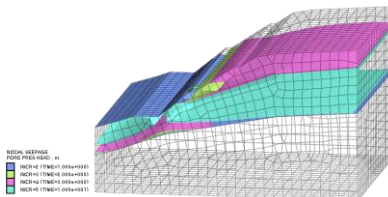
Consoli-  
dation  
Analysis



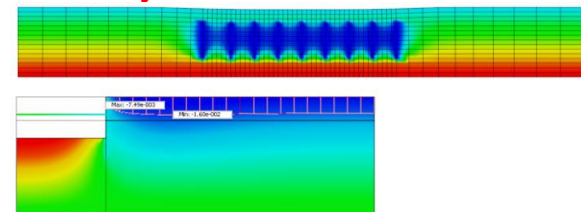
Seepage  
Analysis

Stress-  
seepage  
fully  
coupled

- Consolidation Analysis
- **Stress-seepage fully coupled analysis**



- Steady state seepage analysis
- Transient seepage analysis





# Contents

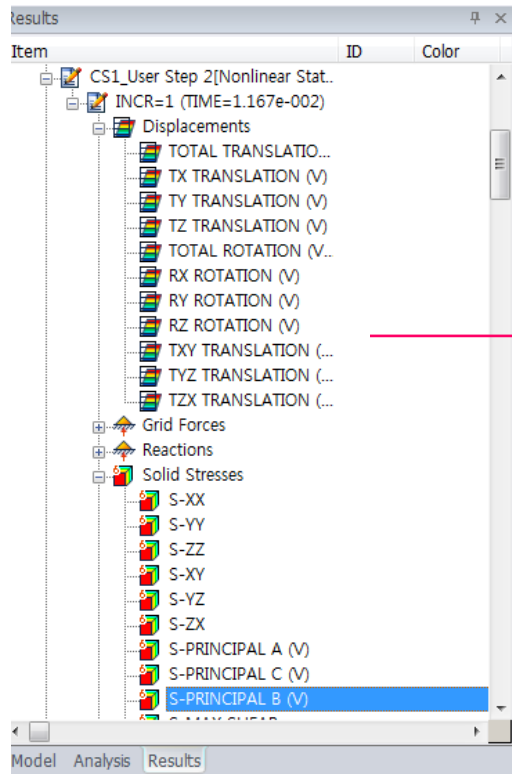
1. Overview
2. Geometry Modeling
3. Mesh Generation
4. Analysis
- 5. Post-processing**

# Post-processing

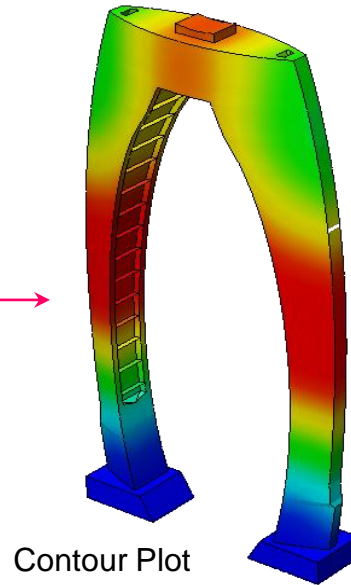
## ■ Complete Support for Visualisation and Interpretation

- Flexible User-control on Legends, Colors, Fonts, Magnification, etc.
- Multiple Plots, Graphs and Tables in Multiple Windows
- Deformed Shape Combined with Undeformed Shape (including Mode Shape)
- Local Plots defined by Geometrical Topology or User-selection
- Contour Plots and Animations (AVI)
- Iso-value Lines (2D) and Surfaces (3D)
- Clipping Planes and Slice Lines/Planes
- Partitioned Plots
- History Plots in Various Graphs and Animations (AVI)
- Result Values in MS-Excel compatible Tables
- Result Probe and Extraction
- Result Extraction for Construction Stage Analysis and Time History Analysis
- Screen-shots in JPG, BMP, PNG, GIF Picture Formats

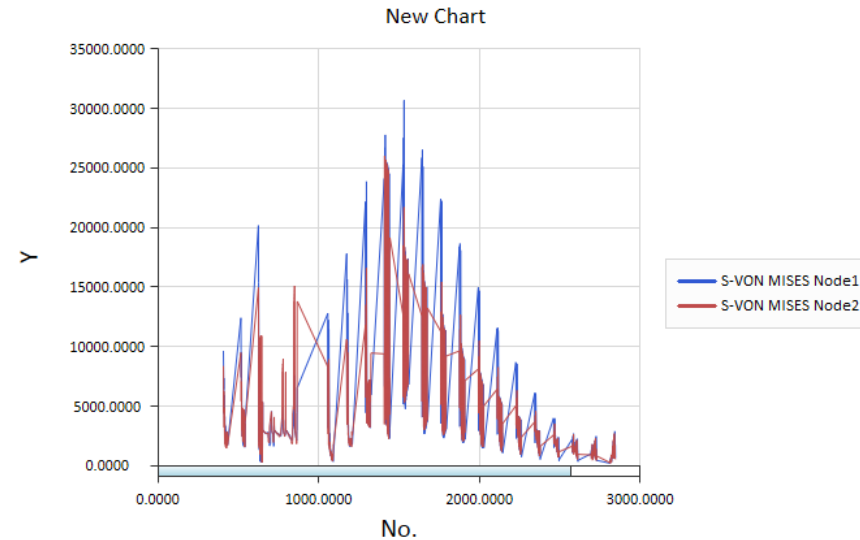
# Post-processing



Works Tree



Contour Plot



Result Graph

Node	TOTAL TRANSLATION (...)	TX TRANSLATION (...)	TY TRANSLATION (...)	TZ TRANSLATION (...)
100	2.291e-005	-1.420e-006	0.000e+000	-2.287e-005
104	3.552e-005	1.456e-005	0.000e+000	-3.239e-005
108	3.130e-005	9.075e-006	0.000e+000	-2.995e-005
109	2.677e-005	3.633e-006	0.000e+000	-2.652e-005
120	4.873e-005	3.112e-006	0.000e+000	-4.863e-005
121	4.855e-005	5.408e-006	0.000e+000	-4.825e-005
122	4.828e-005	8.470e-006	0.000e+000	-4.753e-005
123	4.783e-005	1.240e-005	0.000e+000	-4.619e-005
124	4.728e-005	1.644e-005	0.000e+000	-4.433e-005
125	4.366e-005	1.761e-005	0.000e+000	-3.995e-005
126	4.065e-005	1.826e-005	0.000e+000	-3.632e-005
151	2.698e-005	1.935e-005	0.000e+000	-1.880e-005
152	2.858e-005	2.002e-005	0.000e+000	-2.040e-005
153	3.097e-005	1.802e-005	0.000e+000	-2.519e-005
154	4.496e-005	3.167e-006	0.000e+000	-4.485e-005
155	4.349e-005	3.034e-006	0.000e+000	-4.339e-005
156	4.114e-005	2.779e-006	0.000e+000	-4.104e-005
157	4.079e-005	4.733e-006	0.000e+000	-4.051e-005
158	4.016e-005	7.325e-006	0.000e+000	-3.949e-005

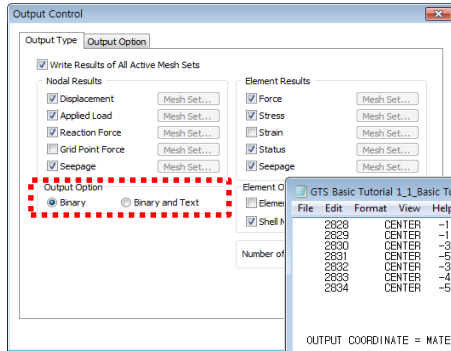
Result Table

MS-Excel

	A	B	C	D
1	1	0,004394	0,000727	-0,004332
2	2	0,004824	-0,002044	-0,004367
3	3	0,006663	-0,00499	-0,004402
4	4	0,003325	0,000601	-0,003268
5	5	0,003477	-0,001071	-0,003307
6	6	0,004475	-0,00295	-0,003352
7	7	0,002358	0,000509	-0,002299

# Post-processing

## [Summary of analysis results]



[Print analysis results]

GTS Basic Tutorial 1.1\_Basic Tutorial 1 - Notepad

```

2828  CENTER  -1.82219E+004  -2.68514E+004  -4.27594E+003  0.00000E+000  0.00000E+000
2829  CENTER  -1.29398E+004  -2.62462E+004  -2.28204E+003  0.00000E+000  0.00000E+000
2830  CENTER  -3.78229E+004  -8.09912E+004  -7.36542E+003  0.00000E+000  0.00000E+000
2831  CENTER  -5.23835E+004  -1.34788E+005  -1.11247E+004  0.00000E+000  0.00000E+000
2832  CENTER  -3.04203E+004  -9.27267E+004  -7.78062E+003  0.00000E+000  0.00000E+000
2833  CENTER  -4.04327E+004  -1.17115E+005  -8.94961E+003  0.00000E+000  0.00000E+000
2834  CENTER  -5.36525E+004  -1.51760E+005  -1.37335E+004  0.00000E+000  0.00000E+000

STRESSES IN GEO PLANE STRAIN ELEMENTS

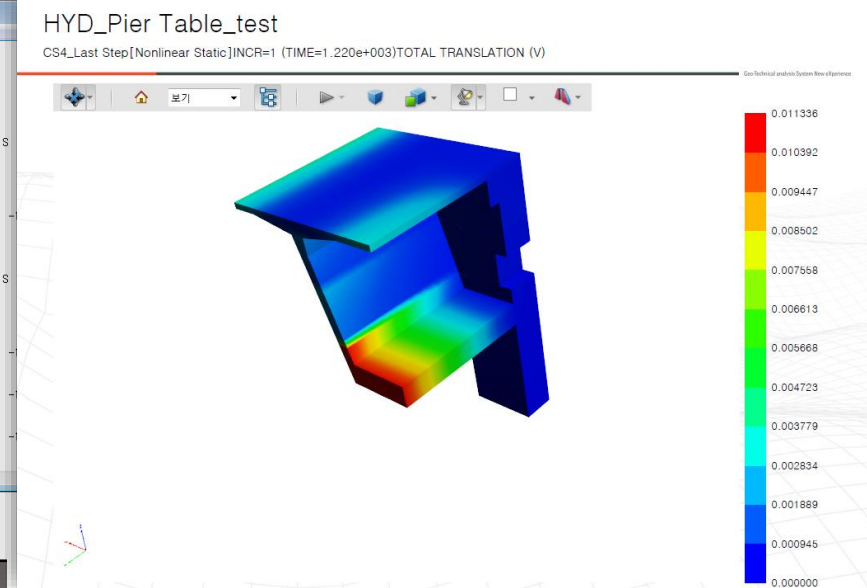
OUTPUT COORDINATE = MATERIAL X-AXIS = X NORMAL = Z
ELEMENT  GRID  STRESSES IN OUTPUT COORDINATE SYSTEM  PORE STRESS  PRINCIPAL
ID        ID    TOTAL NORMAL EFFECTIVE NORMAL SHEAR-XY  TOTAL    EXCESSIVE  ANGLE
2370  CENTER  XX  -2.89348E+006  -2.89348E+006  -9.74779E+005  0.00000E+000  0.00000E+000  -52.6208
      VY  -2.36220E+006  -2.36220E+006
      ZZ  -2.77024E+006  -2.77024E+006

STRESSES IN GEO PLANE STRAIN ELEMENTS

OUTPUT COORDINATE = MATERIAL X-AXIS = X NORMAL = Z
ELEMENT  GRID  STRESSES IN OUTPUT COORDINATE SYSTEM  PORE STRESS  PRINCIPAL
ID        ID    TOTAL NORMAL EFFECTIVE NORMAL SHEAR-XY  TOTAL    EXCESSIVE  ANGLE
1        CENTER  XX  -2.55110E+006  -2.55110E+006  -2.43281E+005  0.00000E+000  0.00000E+000  -72.8483
      VY  -1.83792E+006  -1.83792E+006
      ZZ  -2.74401E+006  -2.74401E+006
2        CENTER  XX  -2.52865E+006  -2.52865E+006  -2.40461E+005  0.00000E+000  0.00000E+000  -72.7843
      VY  -1.82711E+006  -1.82711E+006
      ZZ  -2.72826E+006  -2.72826E+006
3        CENTER  XX  -2.57366E+006  -2.57366E+006  -2.45886E+005  0.00000E+000  0.00000E+000  -72.9252
      VY  -1.84687E+006  -1.84687E+006
      ZZ  -2.75966E+006  -2.75966E+006
    
```

## [Decibel reference levels (ISO R 1683)]

Quantity		Definition	Ref. level
Amplitude	Acc. Level	$L_a = 20 \log_{10} (a/a_0) \text{ dB}$	$a_0 = 10^{-6} \text{ m/s}^2$
	Vel. Level	$L_v = 20 \log_{10} (v/v_0) \text{ dB}$	$v_0 = 10^{-9} \text{ m/s}$
	Dis. level	$L_d = 20 \log_{10} (d/d_0) \text{ dB}$	$d_0 = 10^{-12} \text{ m}$

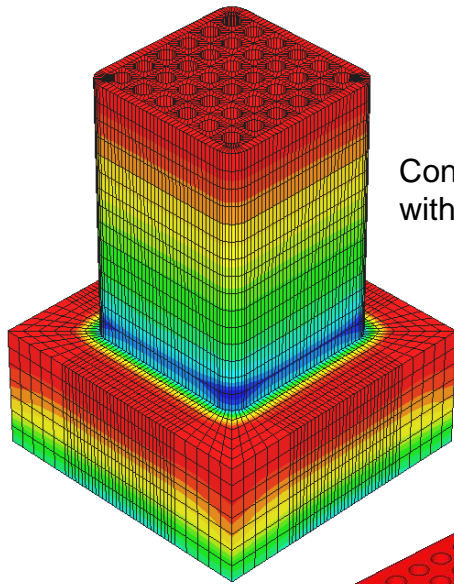


[3D PDF]

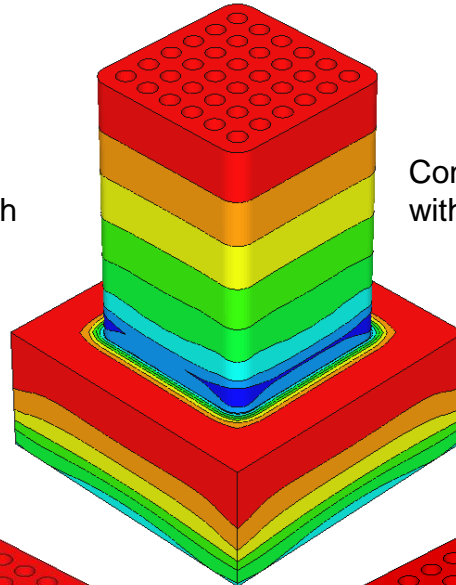
Printable PDF file that includes an adjustable 3D view and a decibel unit transformation feature for measuring noises



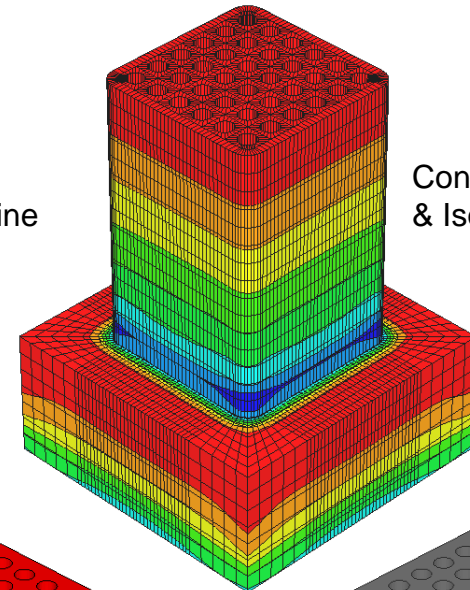
# Contour Plot Type



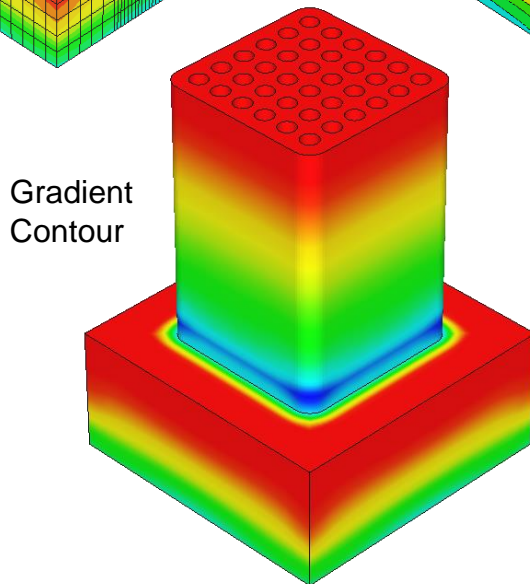
Contour  
with Mesh



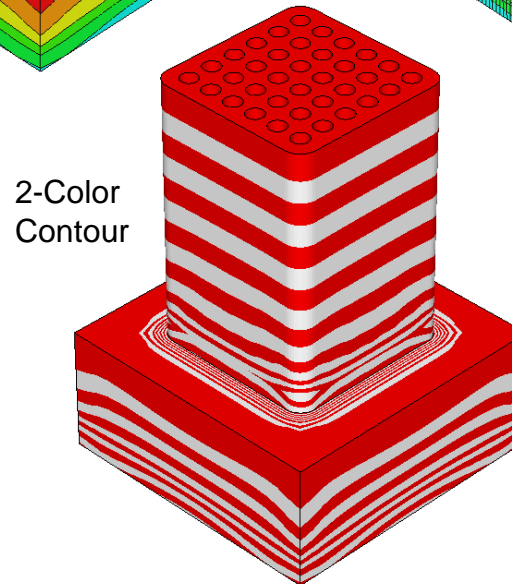
Contour  
with Iso-line



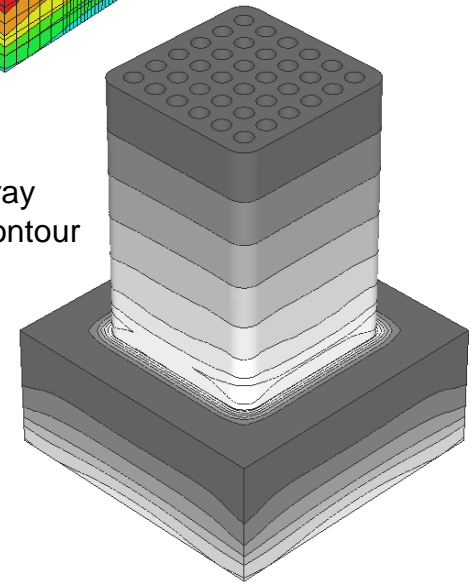
Contour with Mesh  
& Iso-line



Gradient  
Contour

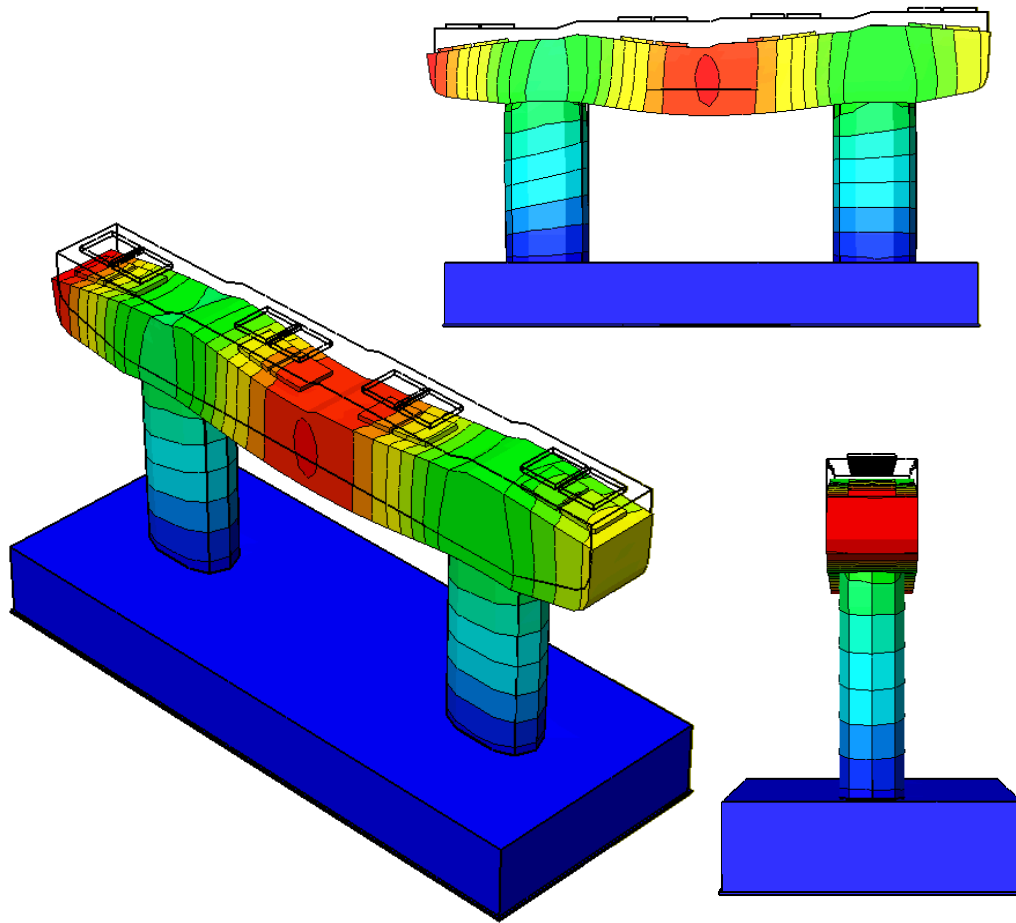


2-Color  
Contour

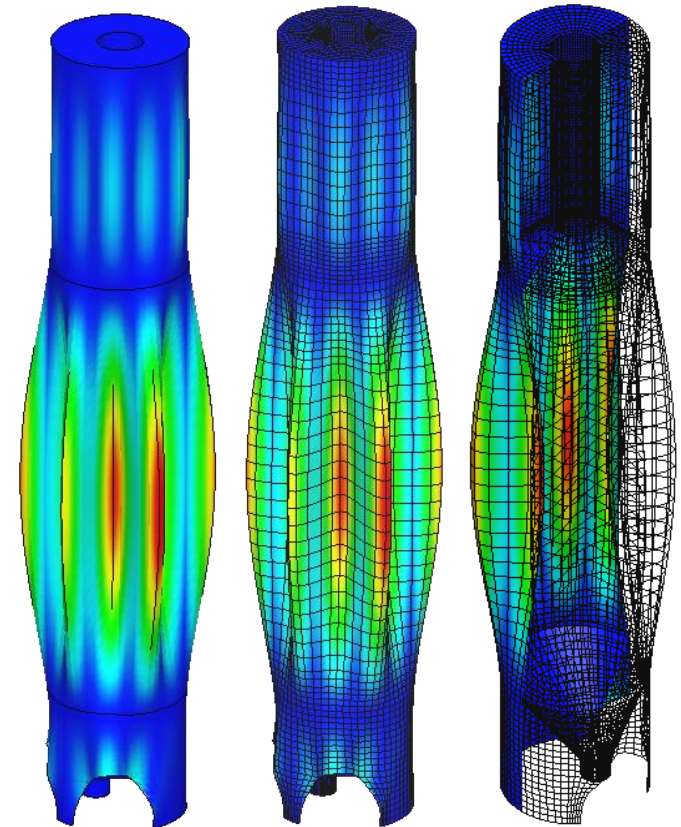


Gray  
Contour

# Deformed Shape

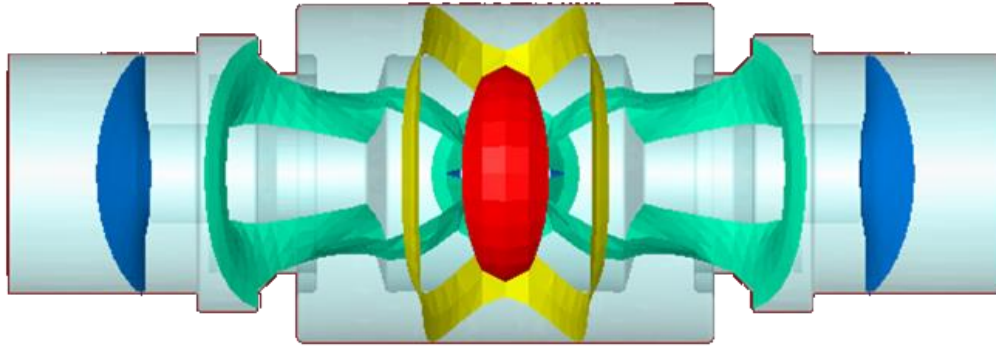


Deformed Contour with Original Shape  
(Static Analysis)

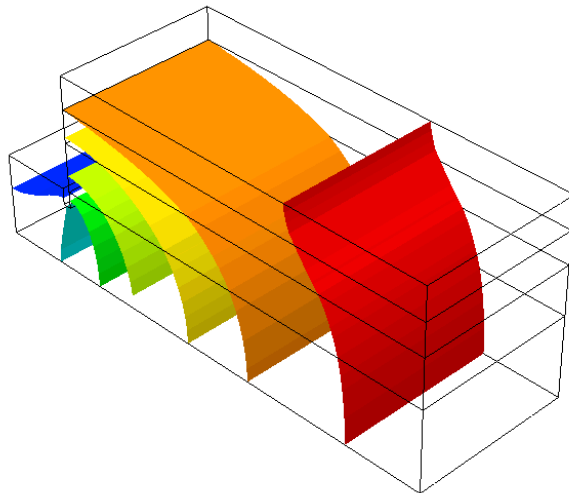
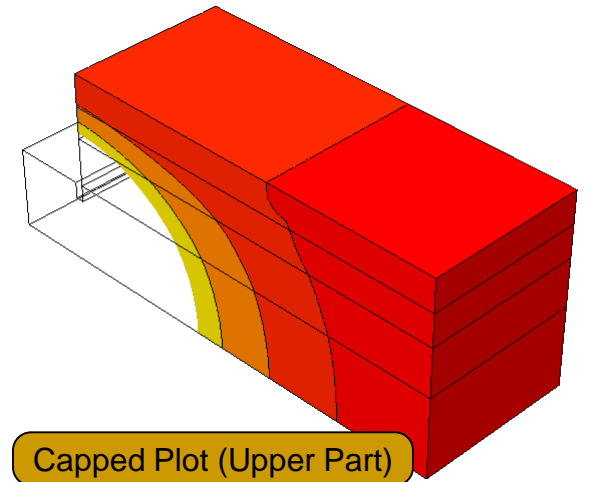
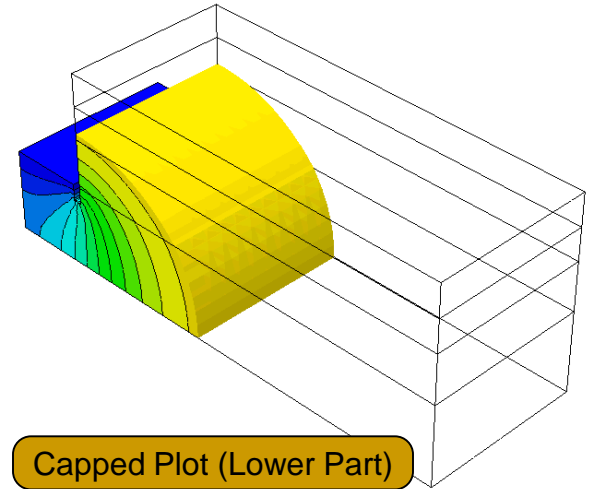


Mode Shapes  
(Stability Analysis)

# Iso-surface Plot

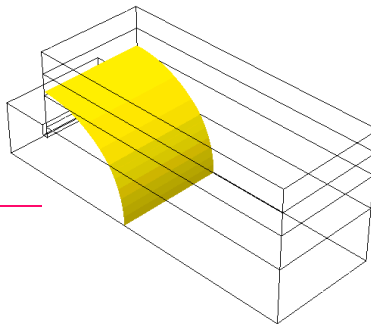


Iso-Surface in Transparent Solid Geometry



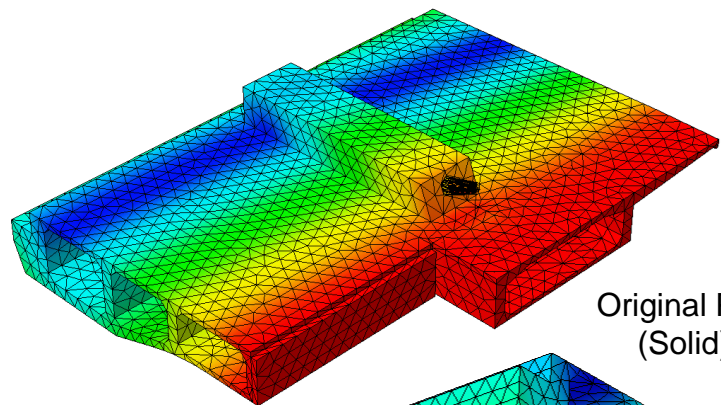
Multiple Iso-surfaces

Base Iso-surface

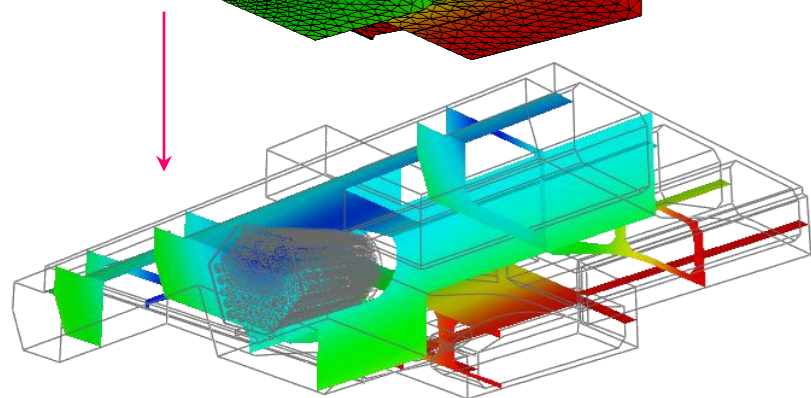
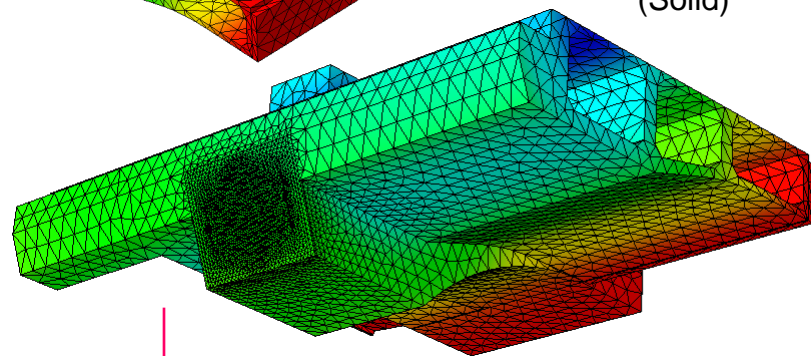




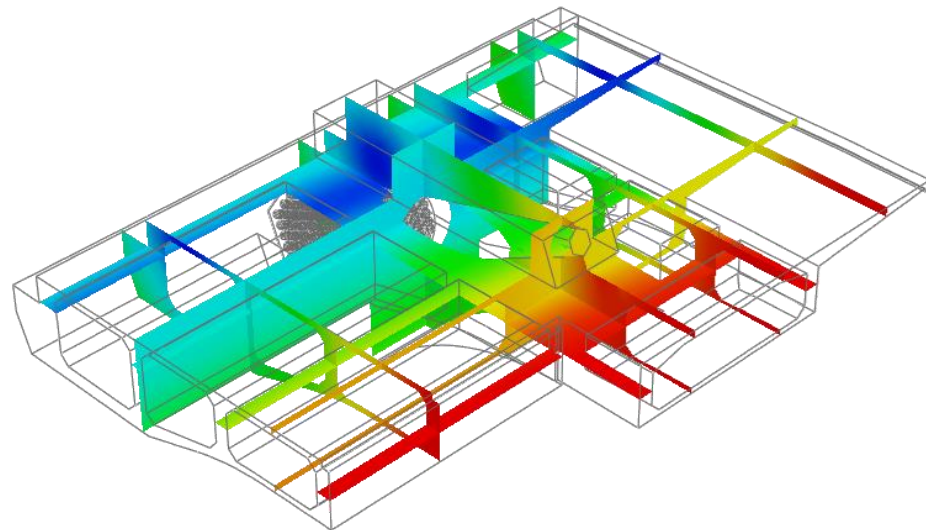
# Slice Plot



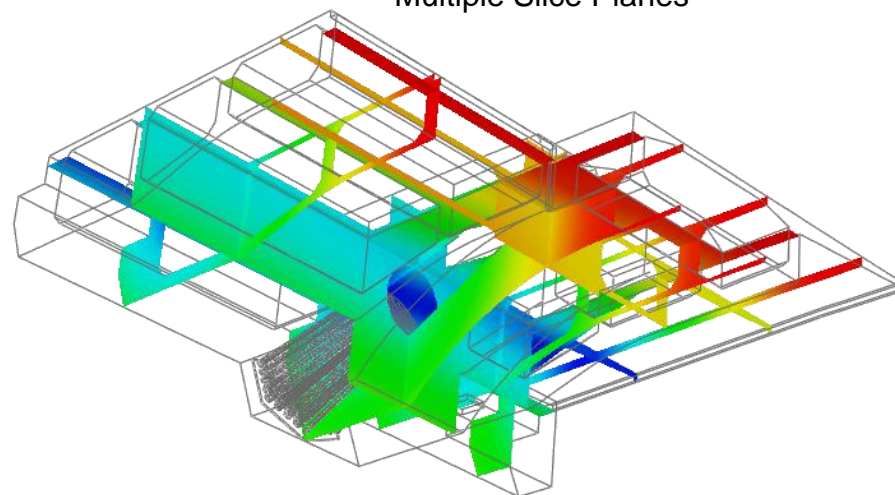
Original Plot  
(Solid)



Slice Plot at Arbitrary Plane

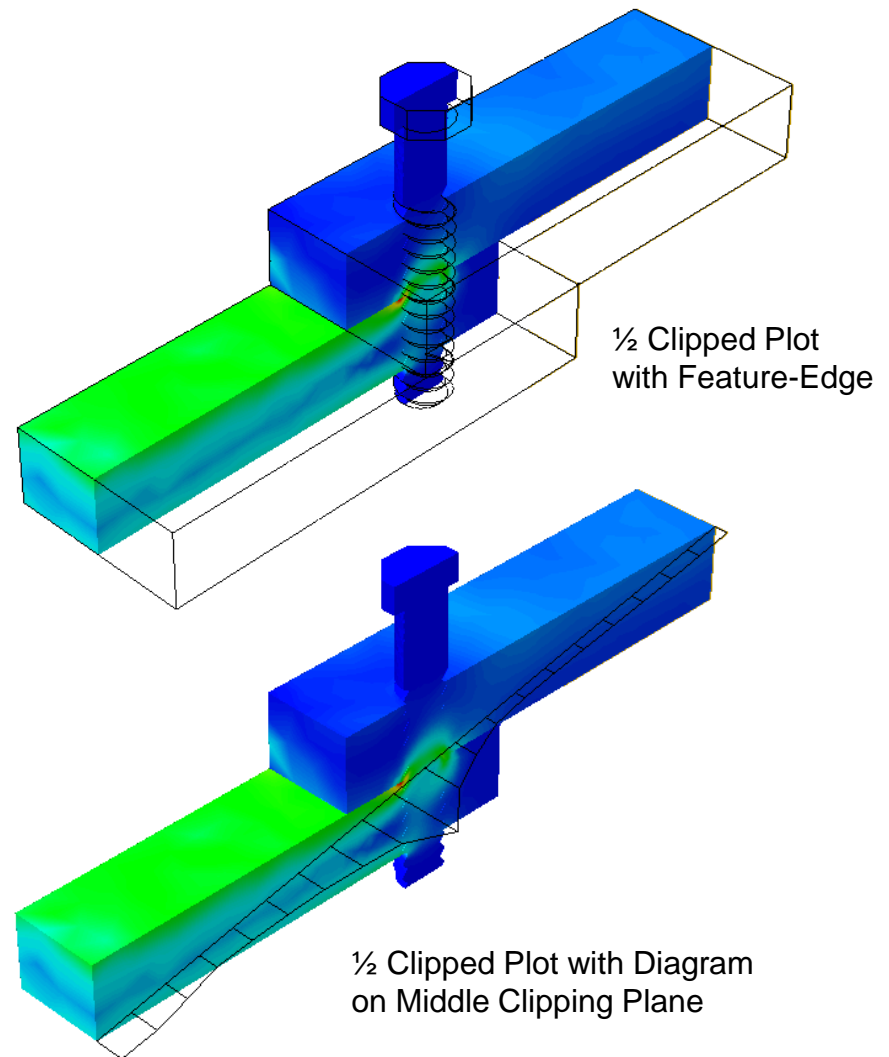
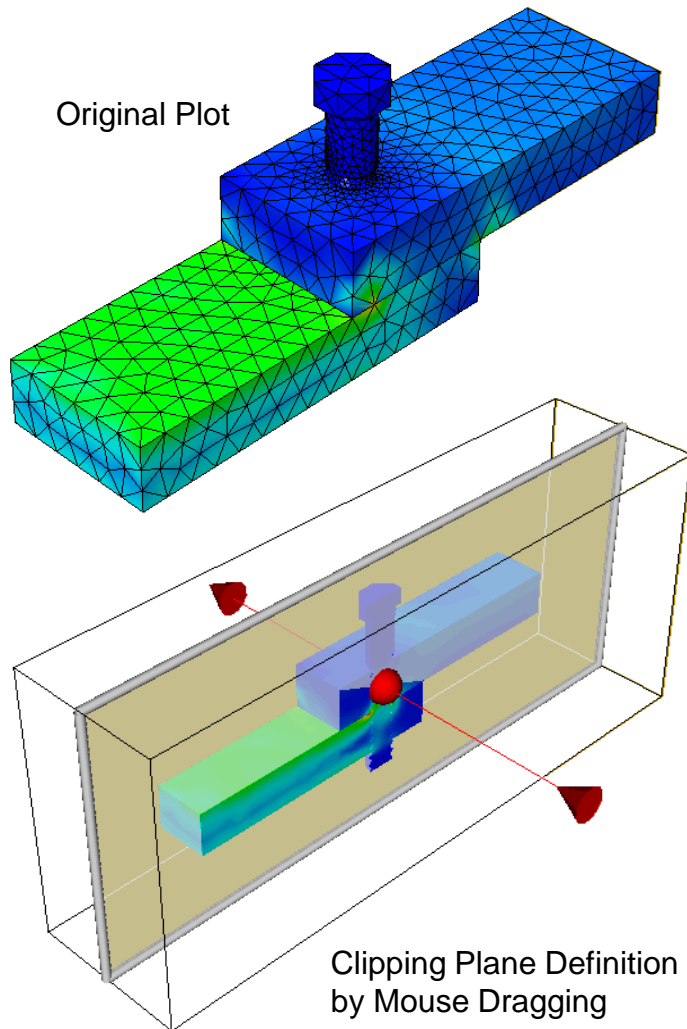


Multiple Slice Planes



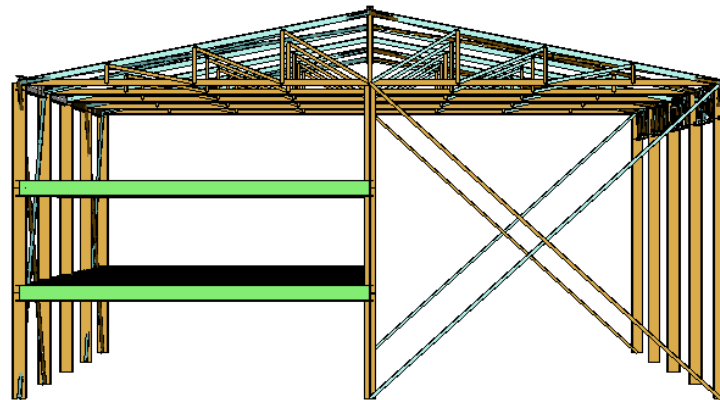
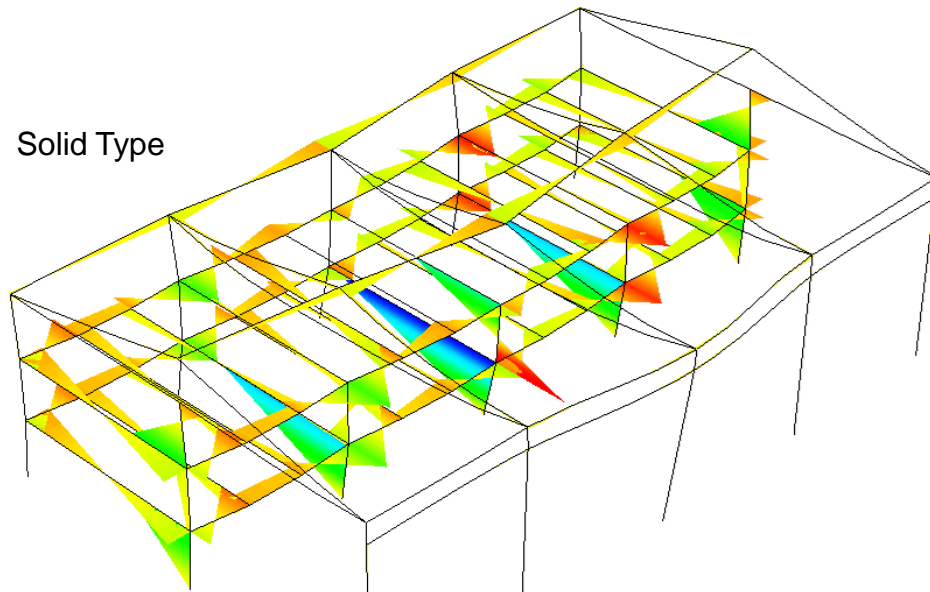


# Clipping Plot



# Diagram Plot

Solid Type



Section Plot of Frames

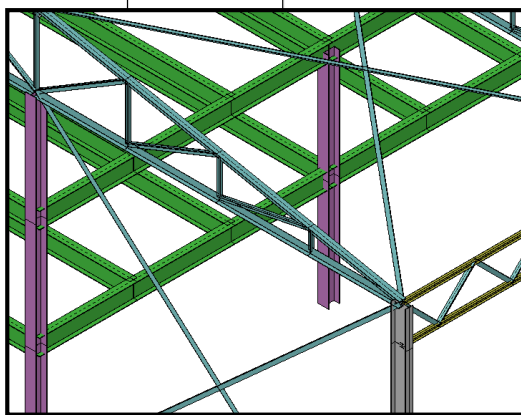
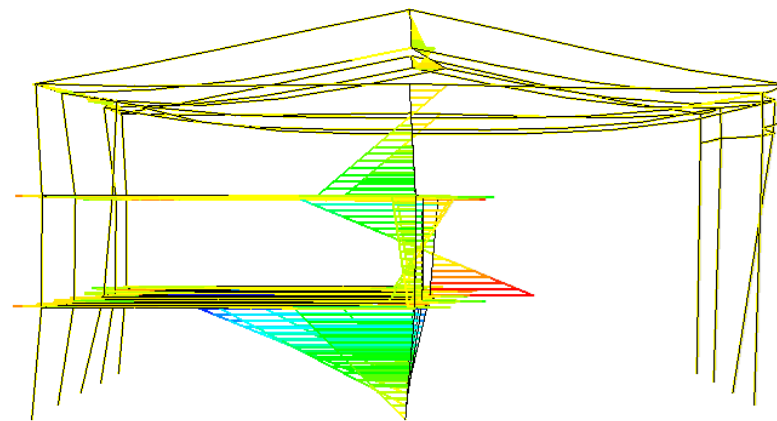


Diagram Plot with Deformed Shape

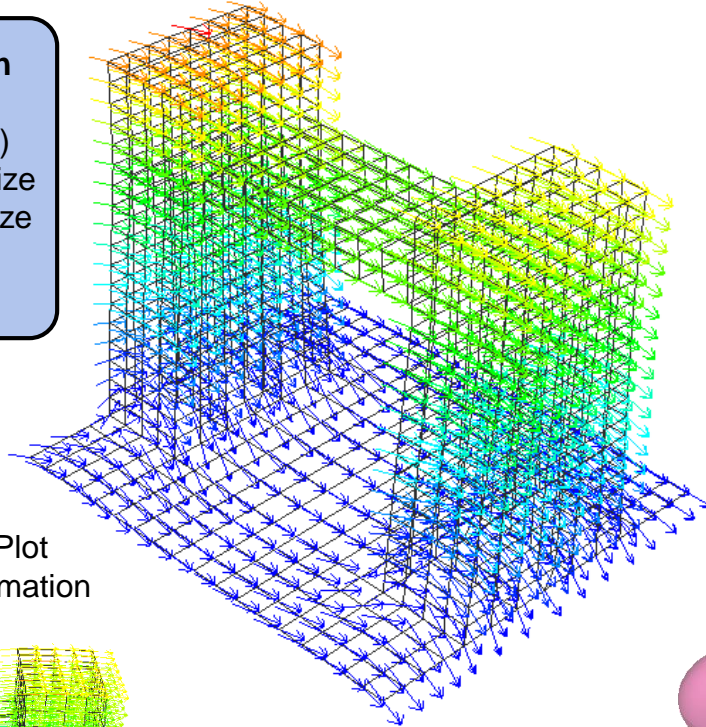


Line Type

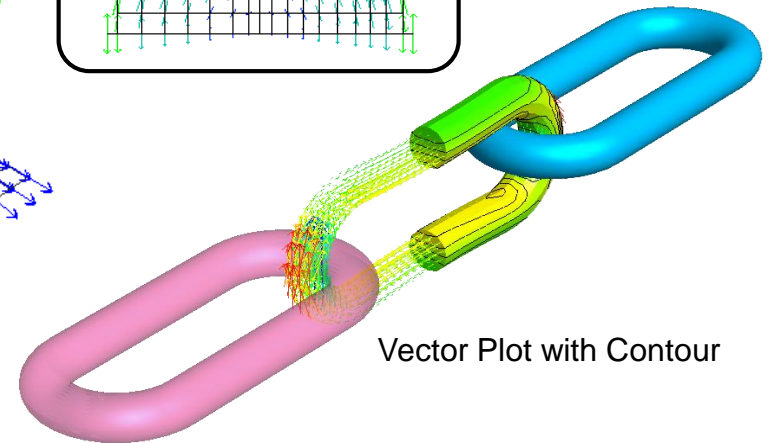
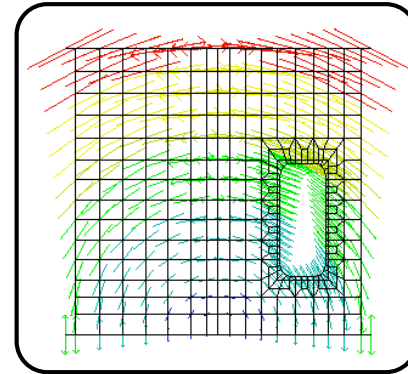
# Vector Plot

## Vector Plot Option

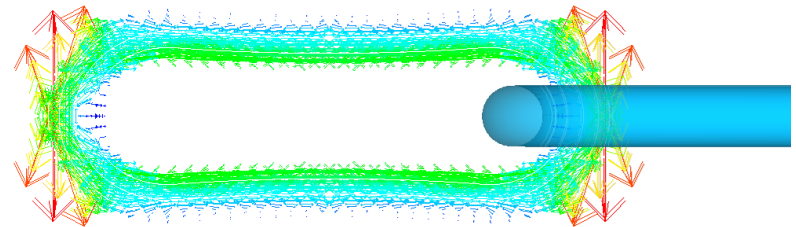
- Head Type  
(Both, One, None)
- Constant Head Size
- Constant Body Size
- Color  
(Contour, Mono)



Vector Plot  
with Deformation



Vector Plot with Contour



Vector Plot with Transparent Geometry



# Result Extraction

**Extract Results**

Output Data

Analysis Set: **NLS**

Result Type: **Plane Stress Forces**

Results: **MEMBRANE FORCE XX**

Step: Results

☒ Nonlinear Static:INCR=1 (LOAD=0.100):MEI  
☒ Nonlinear Static:INCR=2 (LOAD=0.200):MEI  
☒ Nonlinear Static:INCR=3 (LOAD=0.300):MEI  
☒ Nonlinear Static:INCR=4 (LOAD=0.400):MEI  
☒ Nonlinear Static:INCR=5 (LOAD=0.500):MEI  
☒ Nonlinear Static:INCR=6 (LOAD=0.600):MEI  
☒ Nonlinear Static:INCR=7 (LOAD=0.700):MEI  
☒ Nonlinear Static:INCR=10 (LOAD=1.000):MEI

Select All Unselect All

Order

☒ Step ☐ Node/Element

Object

☒ Node ☐ Element

Nodal Results Extraction

☒ User Defined

Select Object

Sort **X** **Y** **Z** ☐ Ascending

☐ Maximum ☐ Minimum ☐ Abs. Max

☐ Only Show Node/Element

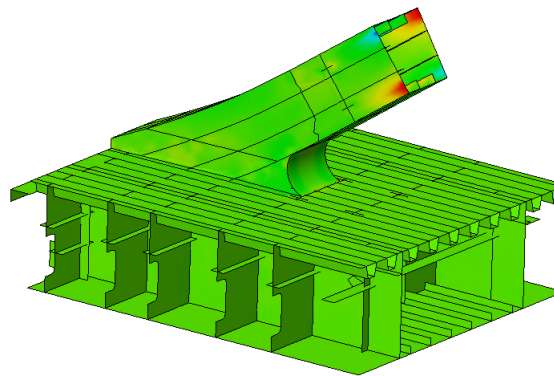
Extraction Position in Element

Table Close



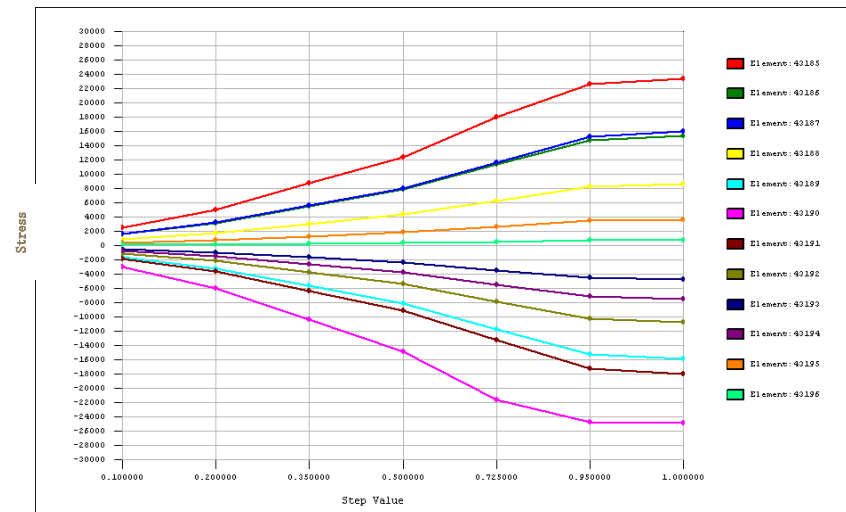
No	Step	Step Value	Element:71 MEMBRANE FORCE XX Center (N/mm)	Element:173 MEMBRANE FORCE XX Center (N/mm)	Element:174 MEMBRANE FORCE XX Center (N/mm)	Element:175 MEMBRANE FORCE XX Center (N/mm)	Element:176 MEMBRANE FORCE XX Center (N/mm)
1	Nonlinear Static:INCR=1 (LOAD=0.100)	1.000000e-001	1.05056e+001	-4.951021e+001	-2.050049e+001	7.315966e+000	3.910260e+001
2	Nonlinear Static:INCR=2 (LOAD=0.200)	2.000000e-001	2.101312e+001	-9.902041e+001	-4.100098e+001	1.463193e+001	7.820520e+001
3	Nonlinear Static:INCR=3 (LOAD=0.300)	3.000000e-001	3.151968e+001	-1.485306e+002	-6.150147e+001	2.194790e+001	1.173078e+002
4	Nonlinear Static:INCR=4 (LOAD=0.400)	4.000000e-001	4.202623e+001	-1.980408e+002	-8.200196e+001	2.926386e+001	1.564104e+002
5	Nonlinear Static:INCR=5 (LOAD=0.500)	5.000000e-001	5.253279e+001	-2.475510e+002	-1.025024e+002	3.657983e+001	1.955130e+002
6	Nonlinear Static:INCR=6 (LOAD=0.600)	6.000000e-001	6.303935e+001	-2.970612e+002	-1.230029e+002	4.389580e+001	2.346156e+002
7	Nonlinear Static:INCR=7 (LOAD=0.700)	7.000000e-001	7.354591e+001	-3.465715e+002	-1.435034e+002	5.121176e+001	2.737182e+002
8	Nonlinear Static:INCR=8 (LOAD=0.800)	8.000000e-001	8.405247e+001	-3.960817e+002	-1.640039e+002	5.852773e+001	3.128208e+002
9	Nonlinear Static:INCR=9 (LOAD=0.900)	9.000000e-001	9.455903e+001	-4.455919e+002	-1.845044e+002	6.584370e+001	3.519234e+002
10	Nonlinear Static:INCR=10 (LOAD=1.000)	1.000000e+000	1.05056e+002	-4.951021e+002	-2.050049e+002	7.315966e+001	3.910260e+002

MS-Excel compatible Table



Nonlinear Analysis

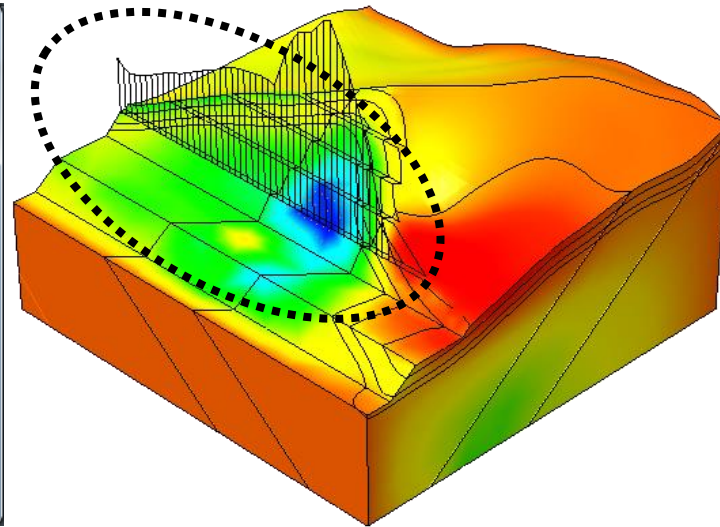
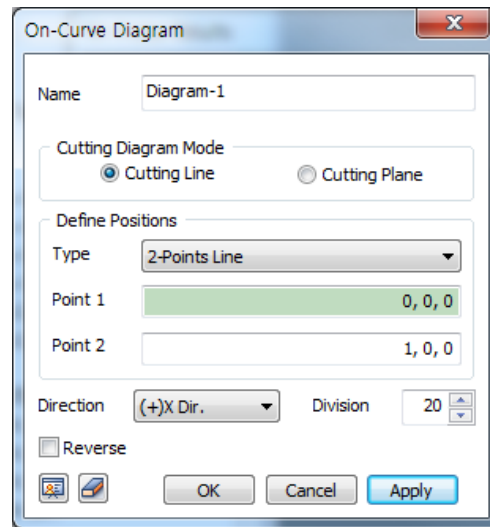
Graph (Stress vs. Time step)



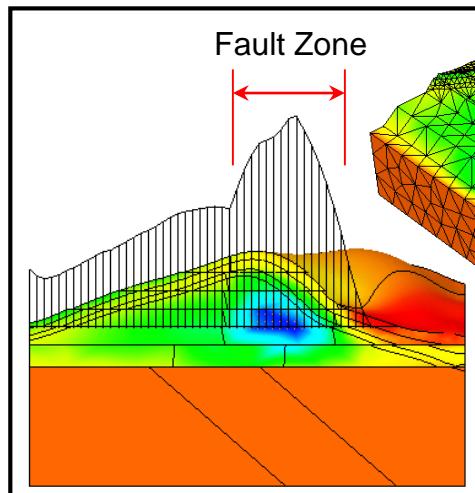
- Results can be extracted based on:
  - Analysis Case
  - Analysis Variables
  - Step (Nonlinear / Construction Stage Analysis)
  - Node / Element (GRC Sys. GCC Sys.)



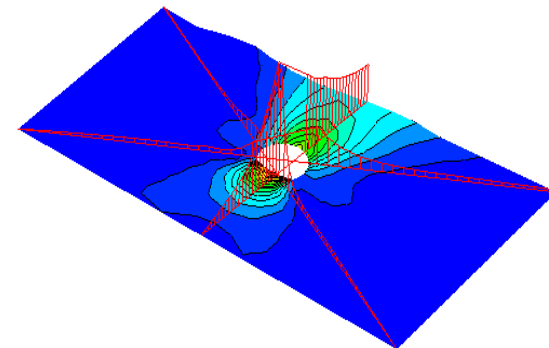
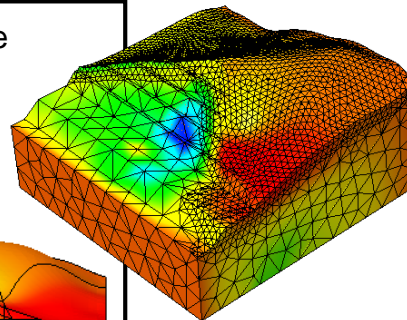
# On-Curve Diagrams



3D On-Curve Graphs on Contour Plot

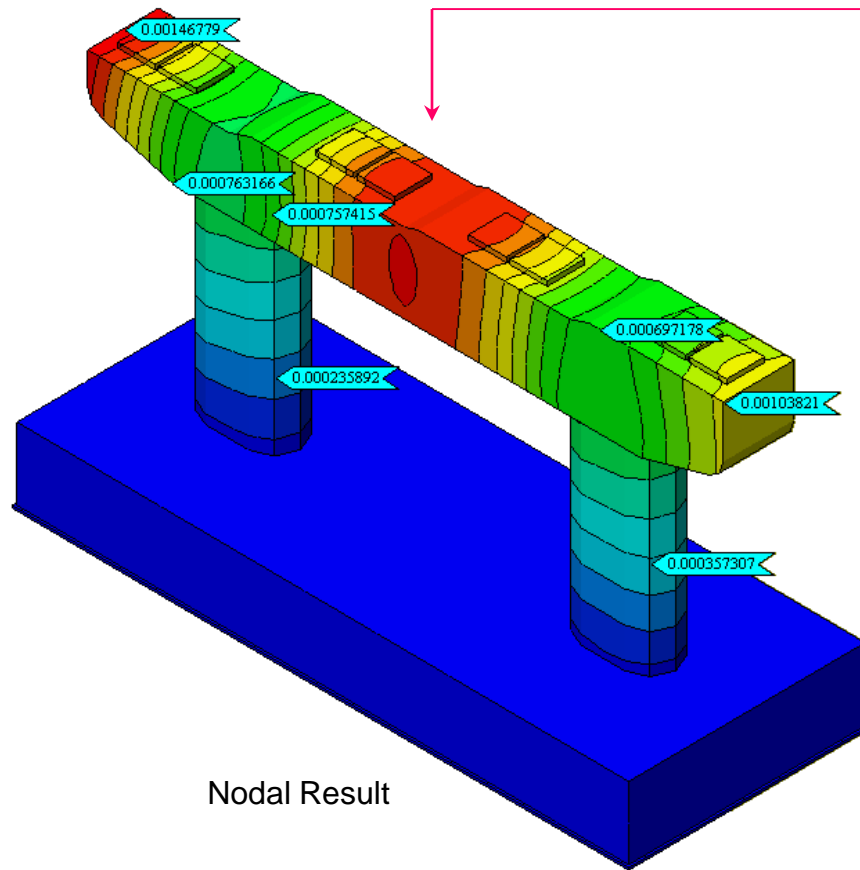


Front View



2D On-Curve Graphs on Contour Plot

# Probe & Result Tag



Nodal Result

Probe Results

Entity Type: ☒ Node ☐ Element

Color: Tag Color  Text Color

Value: ☐ Exponential  Decimal Point

Tag Type:

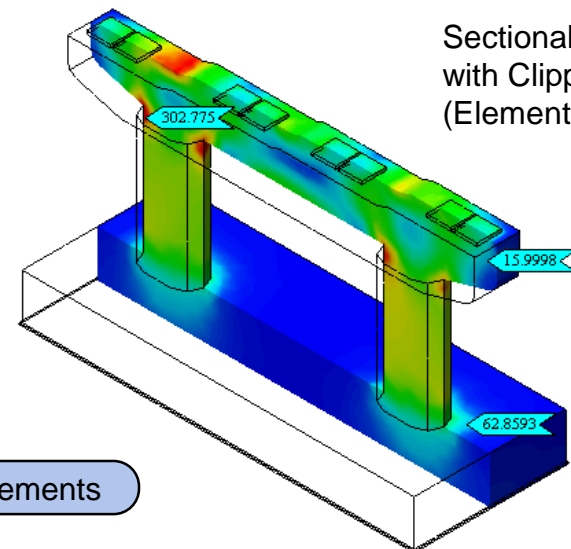
Results

Show	Type	ID	Value
<input checked="" type="checkbox"/>	Node	1292	0.4874
<input checked="" type="checkbox"/>	Node	1367	0.1782
<input checked="" type="checkbox"/>	Node	1157	0.0701
<input checked="" type="checkbox"/>	Node	1203	0.1978
<input checked="" type="checkbox"/>	Node	1166	0.1001
<input checked="" type="checkbox"/>	Node	1210	0.2804
<input checked="" type="checkbox"/>	Node	1254	0.4811
<input checked="" type="checkbox"/>	Node	1339	0.5668
<input checked="" type="checkbox"/>	Node	1424	0.0827
<input checked="" type="checkbox"/>	Node	1296	0.5807

Max Min Abs Max Clear All

☐ Min/Max Value of Each Part

Close

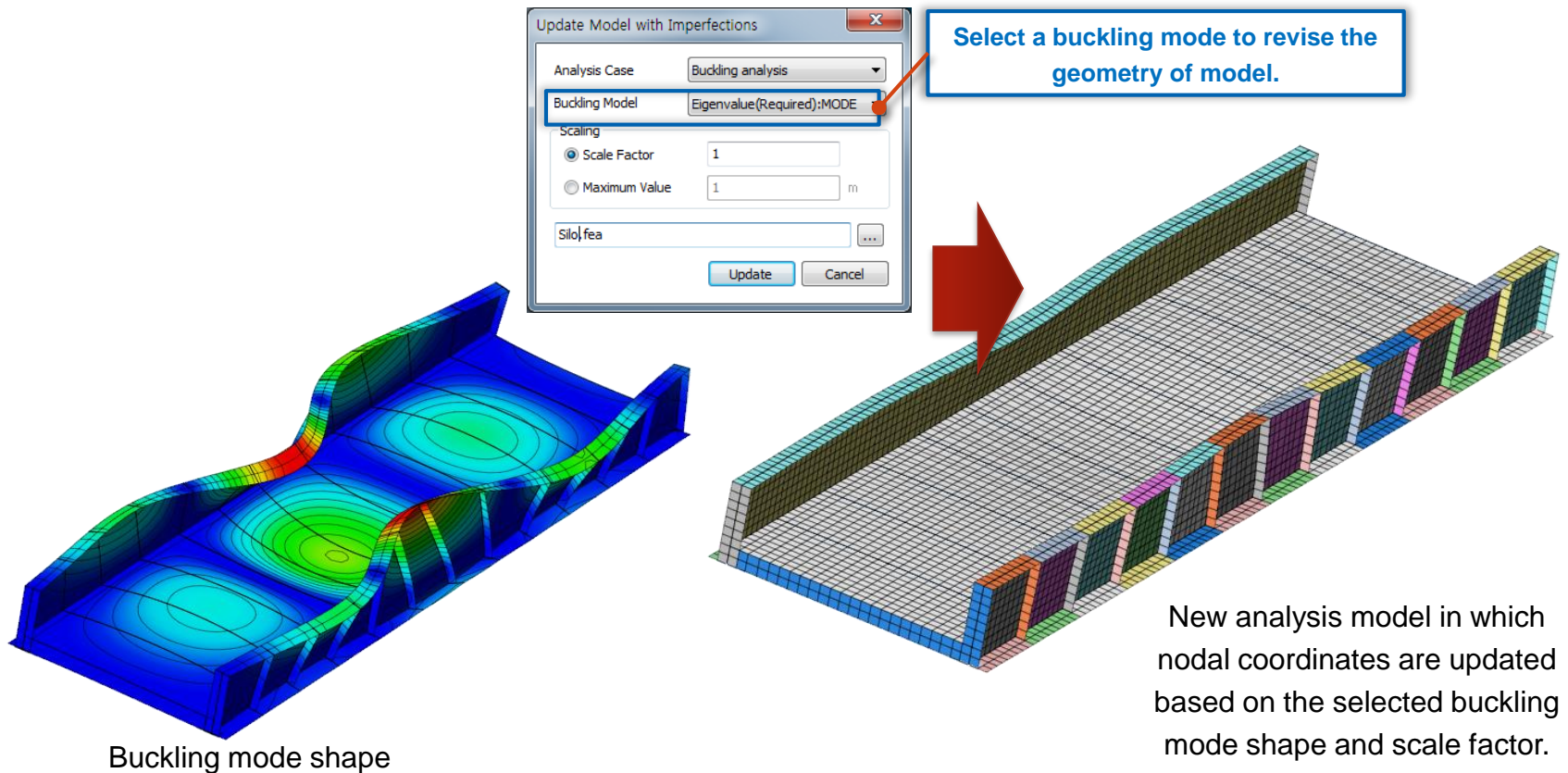


Sectional Result  
with Clipped Plot  
(Element Result)

Probe & Add Result Tags at Specified Nodes/Elements

# Update geometry using buckling mode shape

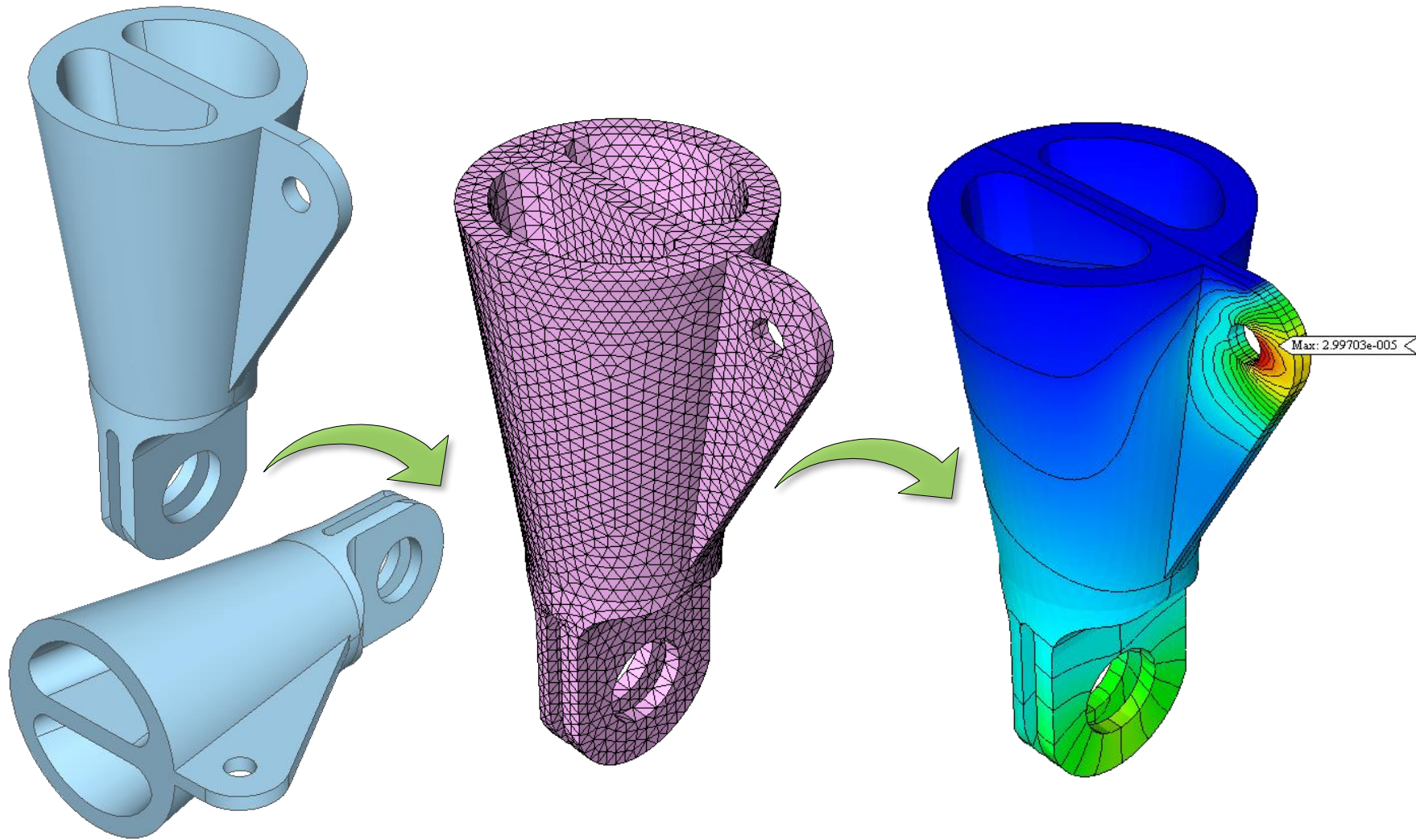
- Create a new model file in which the geometry of analysis model is revised based on the mode shape of linear buckling analysis for geometric and material nonlinear analysis to find a buckling load considering geometric imperfection.



•Geometric imperfection can easily be included in the analysis model. The buckling behavior of the model can be further investigated by performing geometric and material nonlinear analysis.



# Applications



**Solid Geometry Modeling**

**Auto-generated Tetra Mesh**

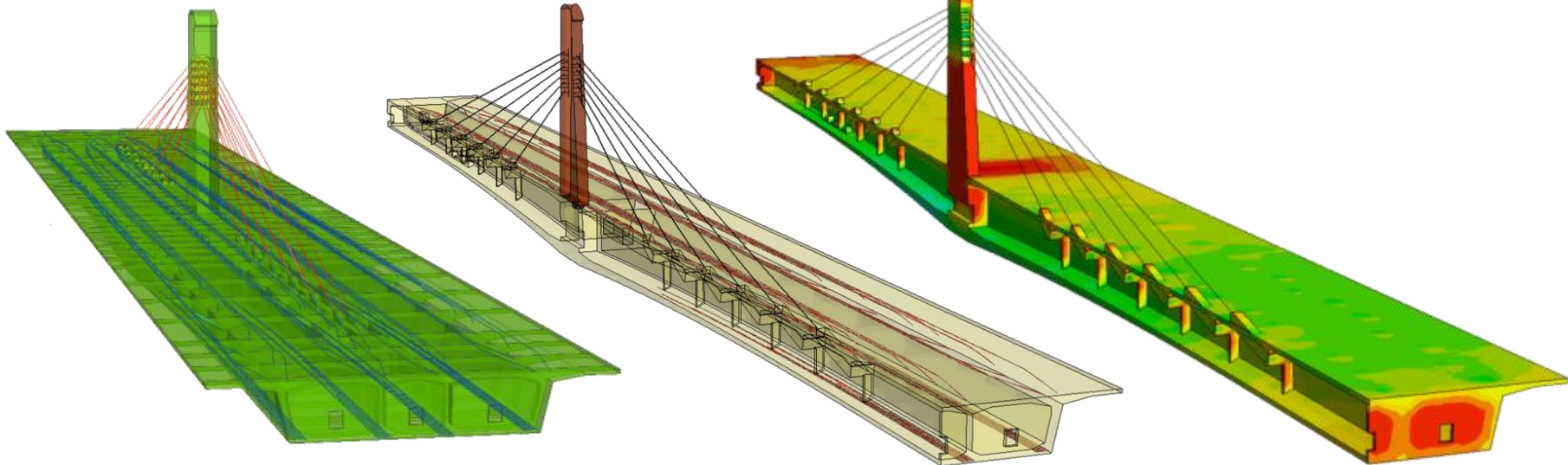
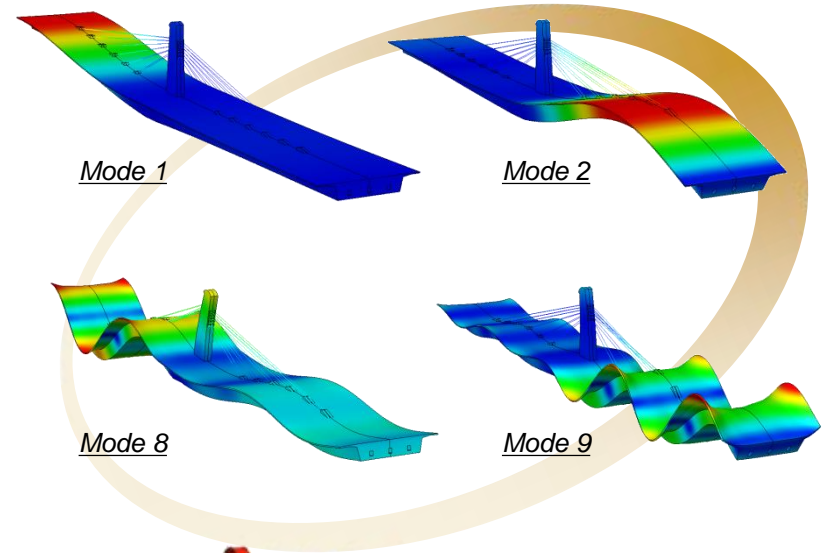
**Post-processing & Result Evaluation**



# Applications



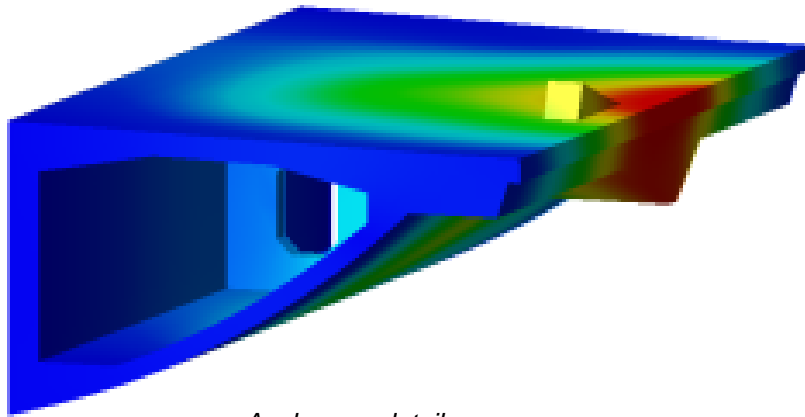
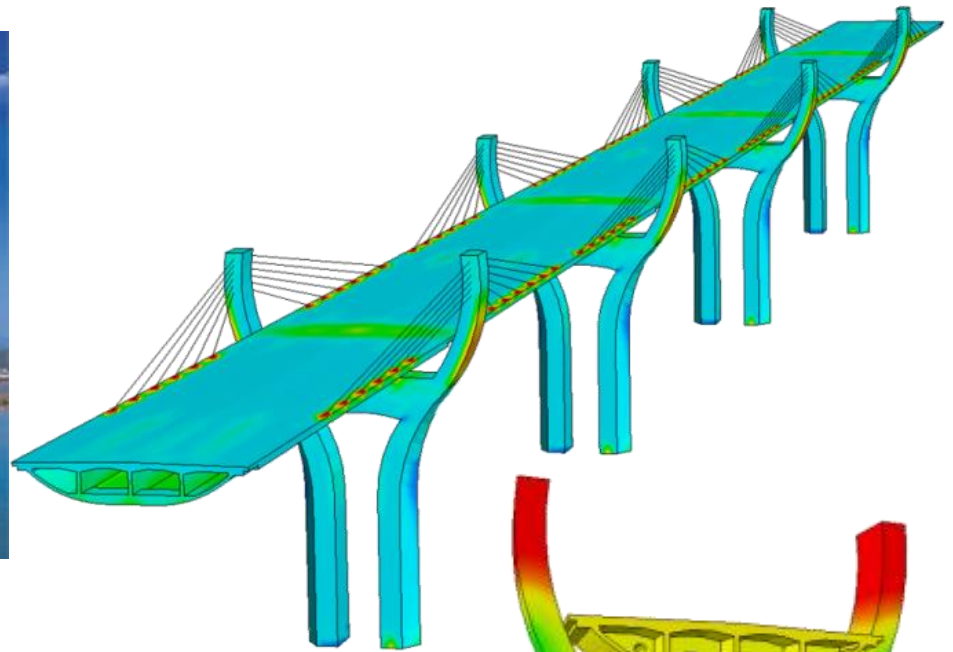
*Woon-Nam Extradosed Bridge – Single Pylon*



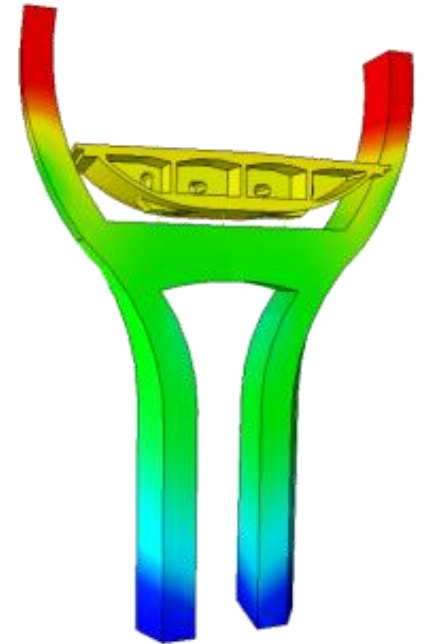
# Applications



*Geumga Extradosed Bridge*



*Anchorage detail*



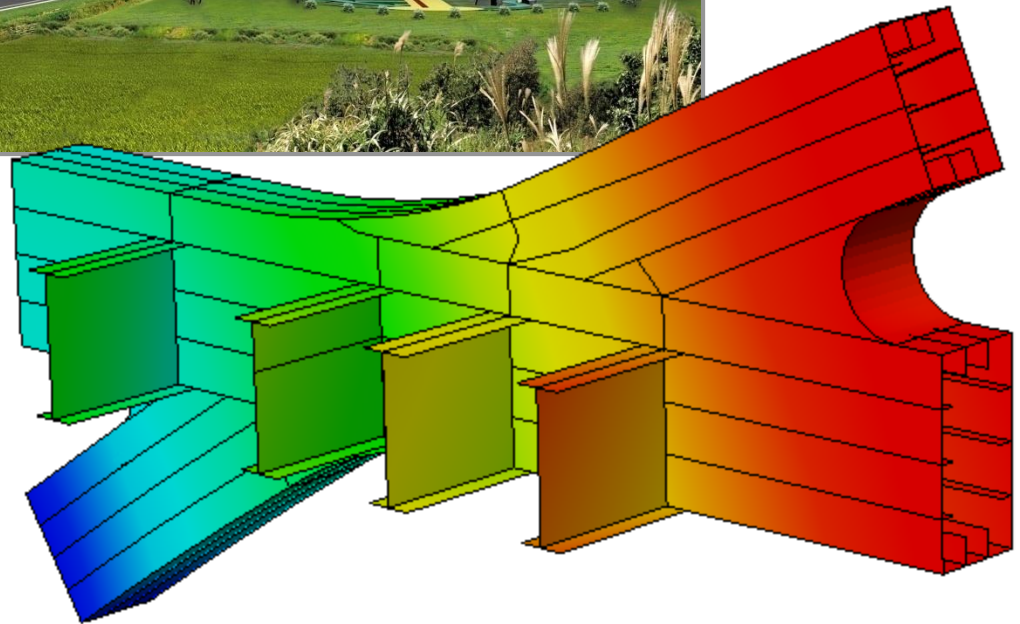
*Double Pylon*



# Applications

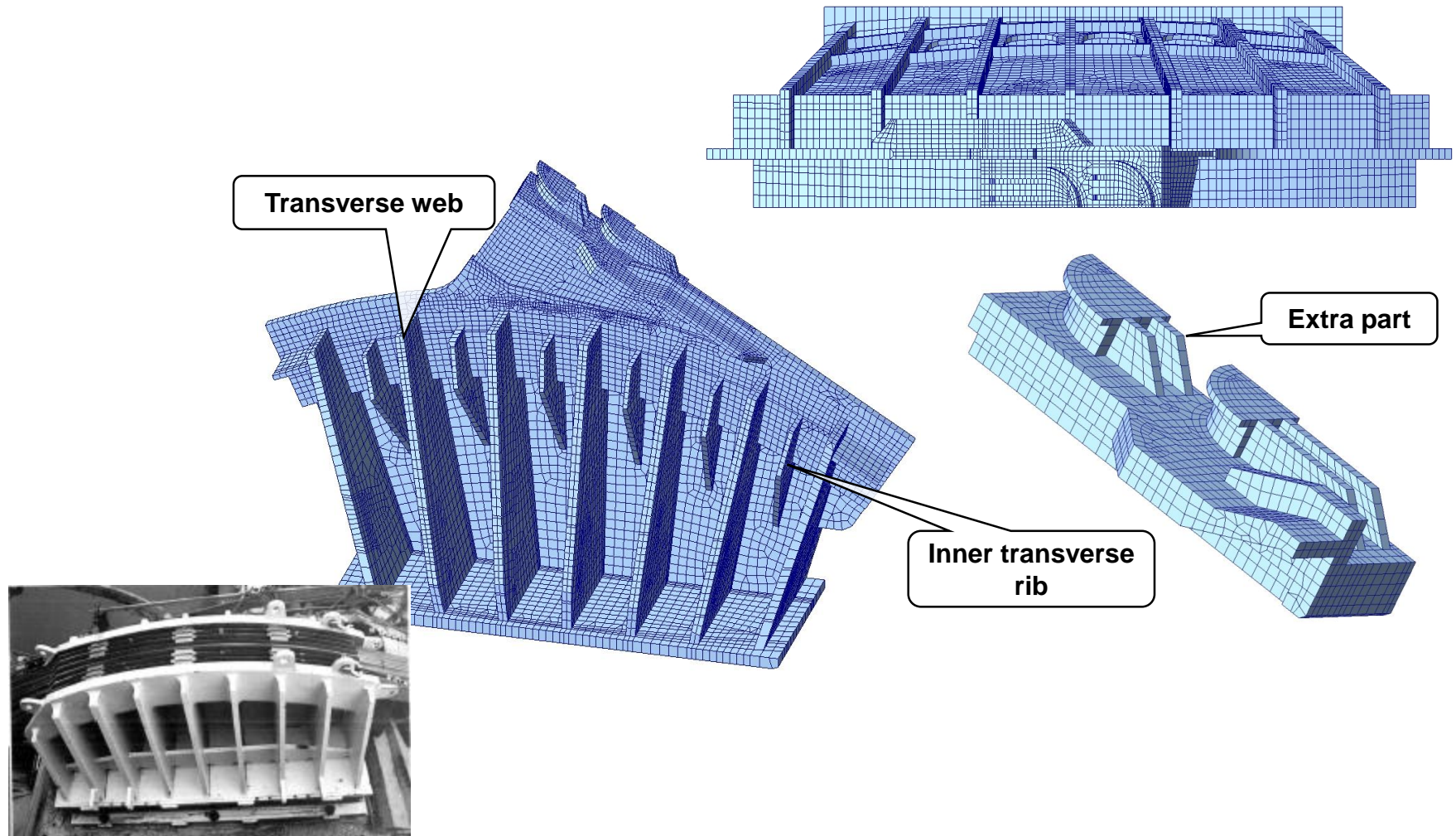


*JECHON-DODAM RAILWAY  
Steel Arch Bridge*



# Applications

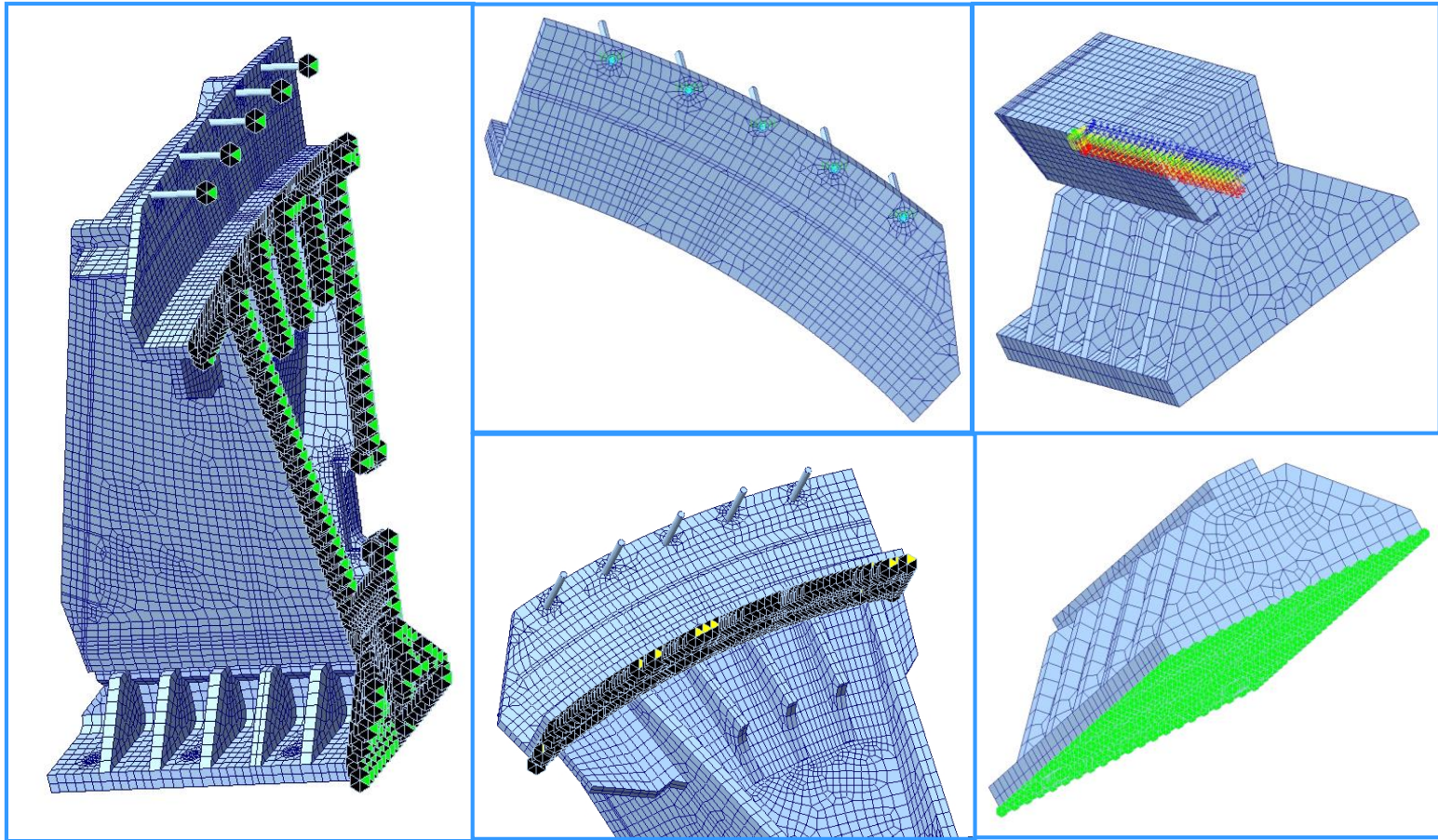
## Solid elements Modeling of Pylon Cable Saddle (Leesoonshin Suspension Br.)





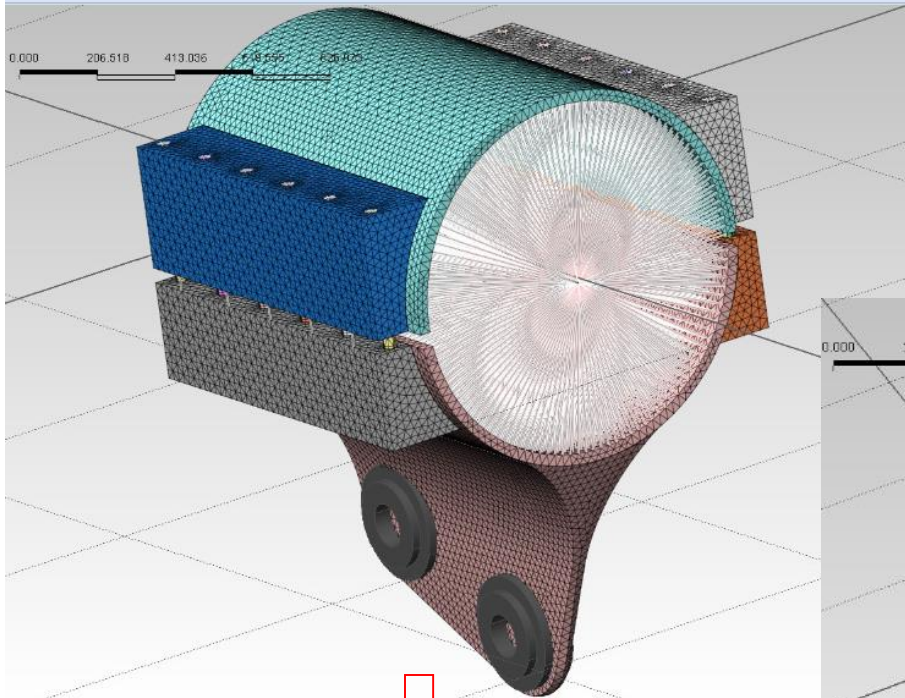
# Applications

Solid elements Modeling of Splay Saddle (New Millennium Suspension Br.)



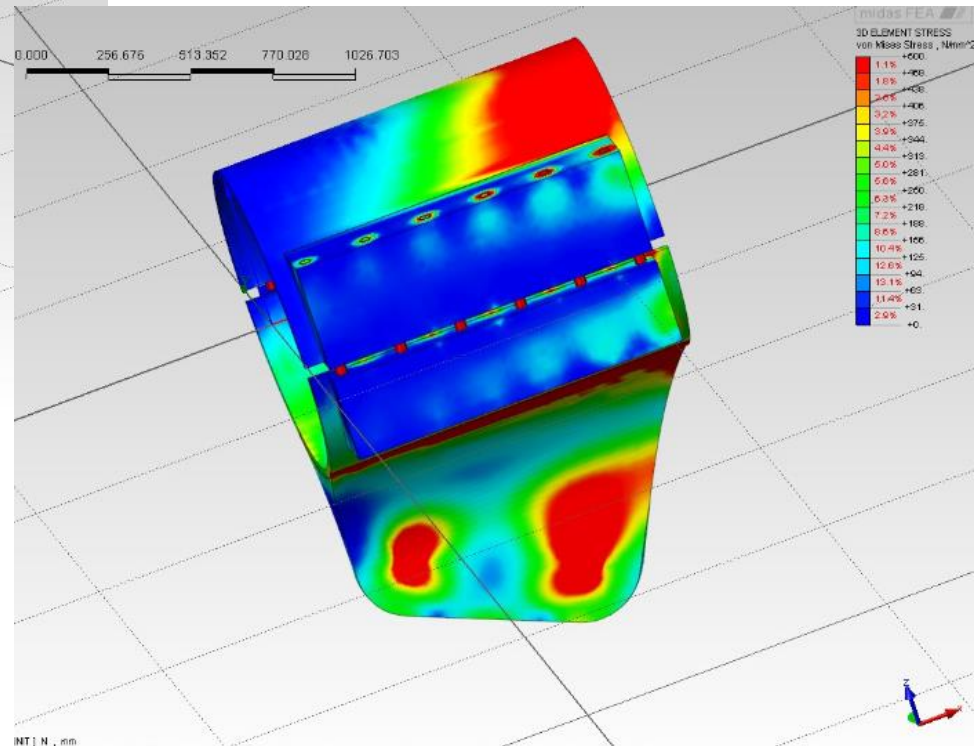
# Applications

## Solid elements Modeling of Hanger Clamp (Bonny River Suspension Br.)



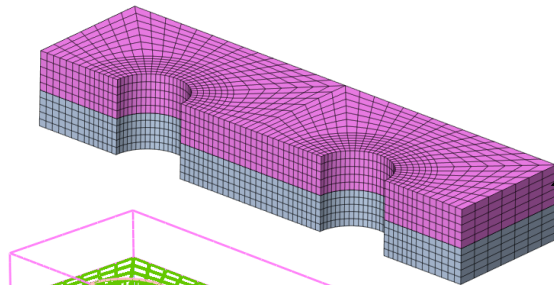
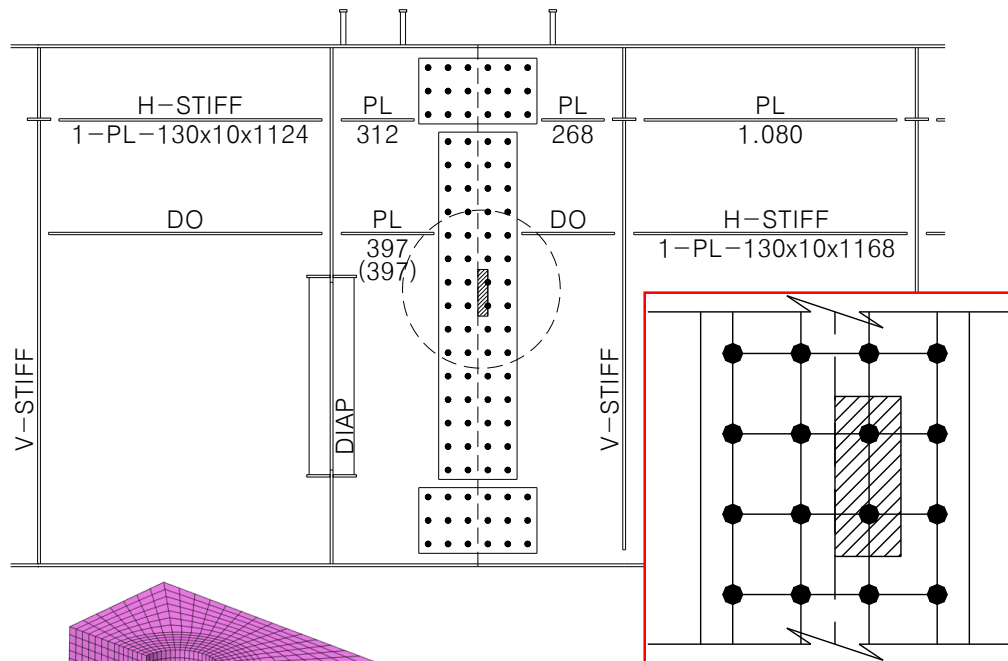
FE Model

Stress Contour

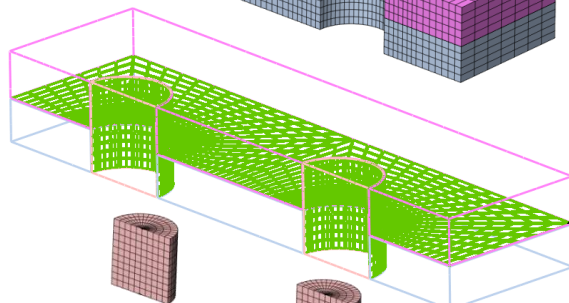




# Applications *Contact Analysis*



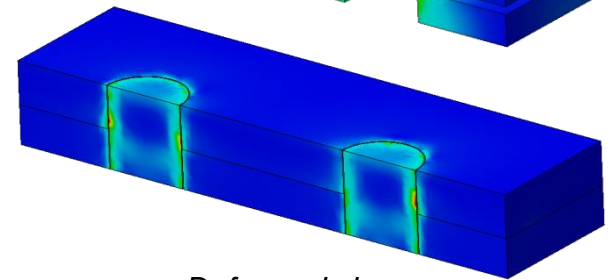
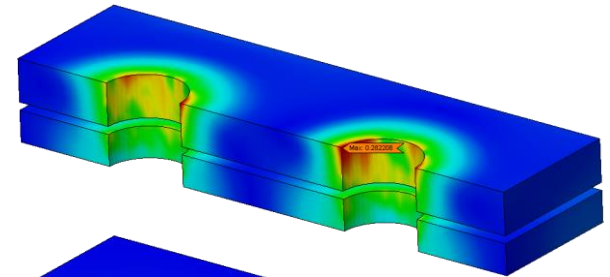
Gusset and web plates



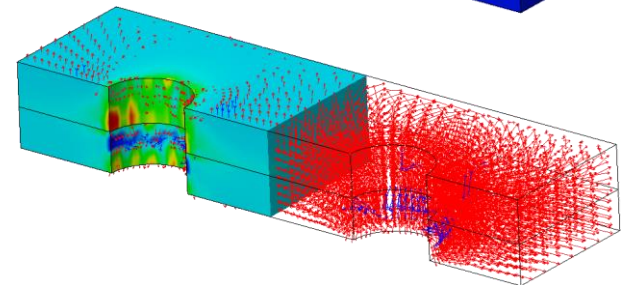
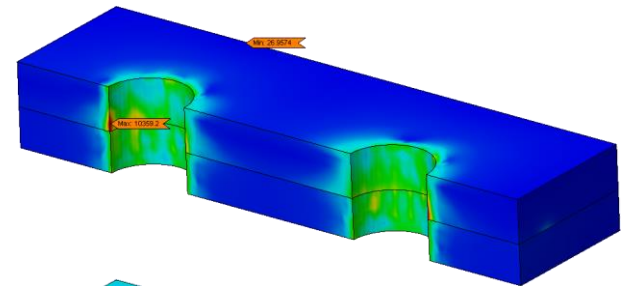
Contact faces



Pins



Deformed shapes



Stress in contact faces

# Improvements in FEA NX

---



# Extended platform optimised for 64-bit environment

## FEA

- 32-bit pre/post handicapped for big models
- 10 year old GUI platform (old graphics)
  - don't provide Preview for LBC
  - model is too big and analysis is not converging

- low computing power in geometric operations for large models and assemblies (Boolean, Division, etc.)
  - I cannot divide solids

- limitation to addition/improvement for analysis features of the solver (DIANA Solver)
  - I can do that with Plaxis

## FEA NX

- Complete support for 64 bits (Pre/Post, Solver)
- Graphics engine supporting large models
- Unified solver with GPU computing

- Adoption of Parasolid Kernel (General kernel used by 3D CAD)
- Reliable geometric computation performance/speed
- Compatible with 3D CAD geometry

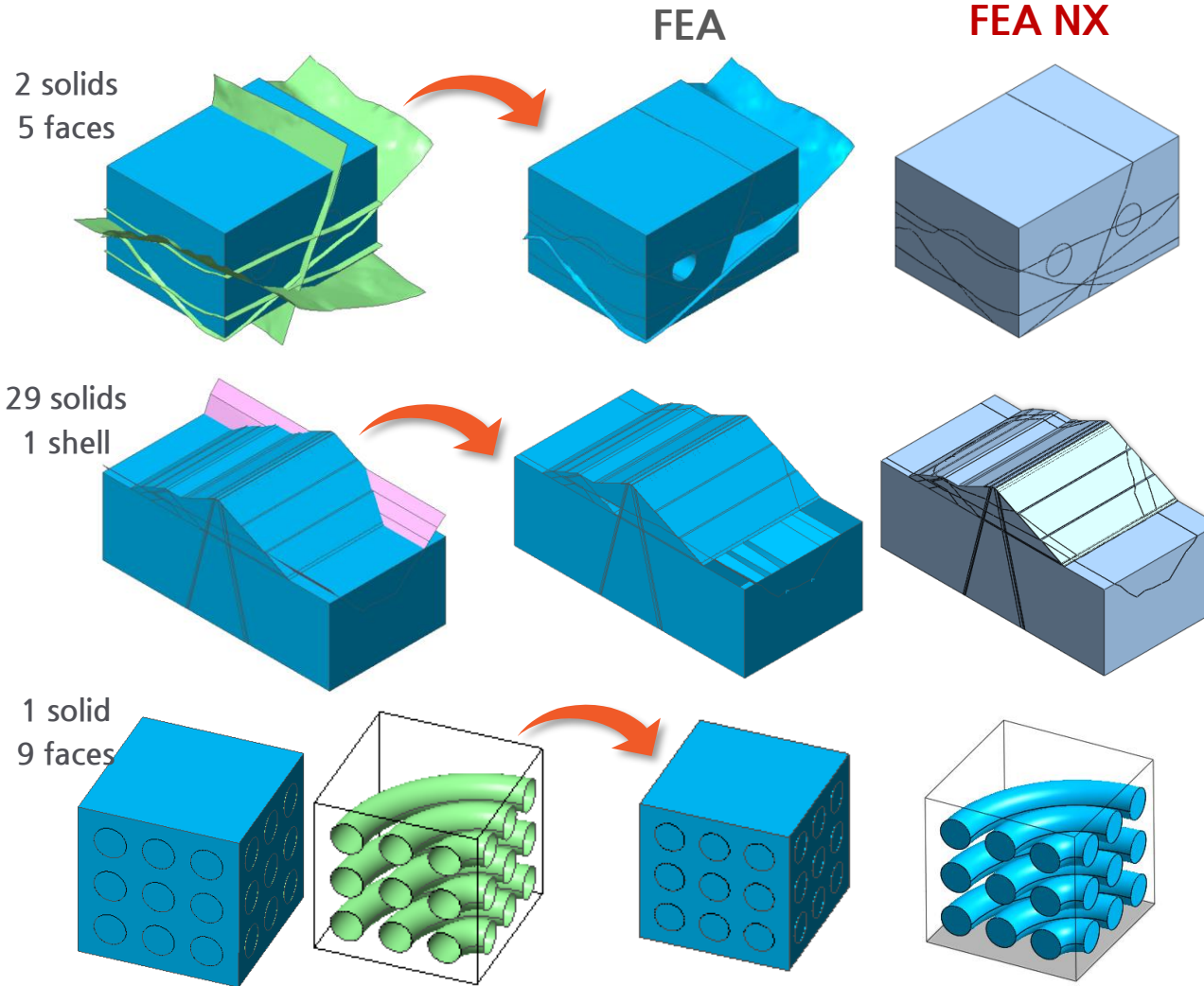
- New Solver fully developed by MIDAS
- Use of advanced features used in mechanical area (Contacts)
- New platform for deploying advanced analysis and features



# Major Improvements in Pre/Post Process



# Improvements in Boolean operations



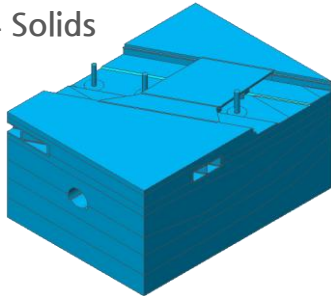
Version	Divide Results
FEA	Failed (39sec) Generation of 5 solids
FEA NX	<b>Completed (22sec)</b>

Version	Divide Results
FEA	Failed (20sec) Generation of broken solids
FEA NX	<b>Completed (5sec)</b>

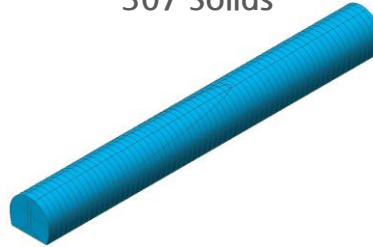
Version	Divide Results
FEA	Completed (1min 29sec)
FEA NX	<b>Completed (less than 1sec)</b>

# Faster auto-meshing using multi-thread parallel

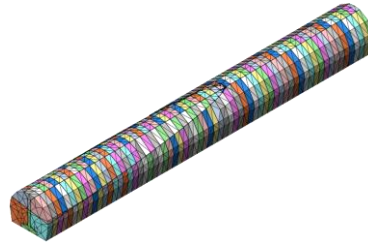
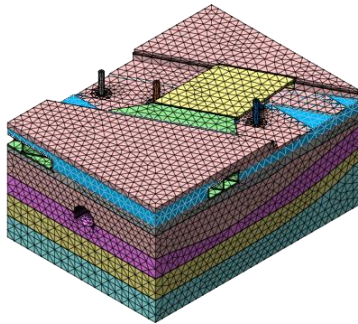
64 Solids



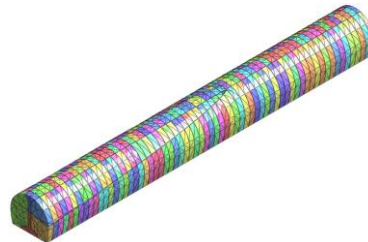
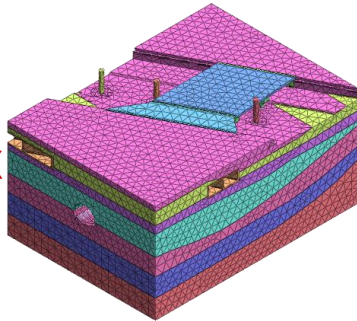
307 Solids



FEA



FEA NX



Mesh Generation in Progress

	ID	Name	Current Step	Progress
1	1	Box	End	100 %
2	2	Box	Geometry-Mesh Relation	99 %
3	3	Box	End	100 %
4	4	Box	Meshing Solid	70 %
5	5	Box	Start	0 %
6	6	Box	Start	0 %
7	7	Box	Start	0 %
8	8	Box	Start	0 %
9	9	Box	Start	0 %
10	10	Box	Start	0 %
11	11	Box	Start	0 %
12	12	Box	Start	0 %
13	13	Box	Start	0 %
14	14	Box	Start	0 %
15	15	Box	Start	0 %

Total Progress

Update Database

Abort

Support multi-thread  
during mesh generation

Version	64 Solids	307 Solids
FEA	326.5 sec (Completed 100%)	58.4 sec (2 solids failed)
FEA NX	21.7 sec (Completed 100%)	9.1 sec (Completed 100%)

90%



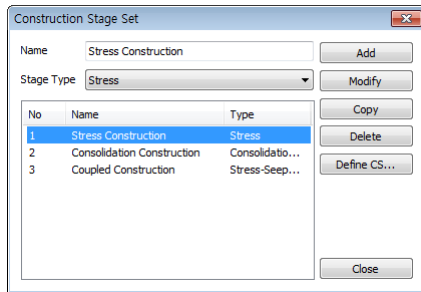
reduced

Tetrahedral auto-meshing generation time is **reduced by over 80% (Multi-Thread)**



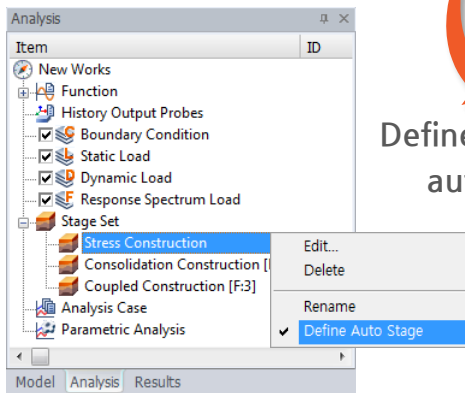
# Construction stage sets & auto-generation of

- Definition/Analysis of **multiple construction stage sets** (can be defined for each analysis type)
- **Graphical auto-definition of construction stages** by Show/Hide status in the work window

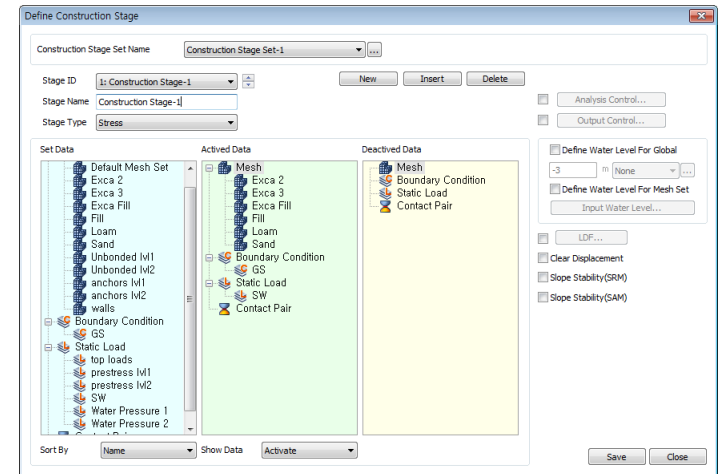
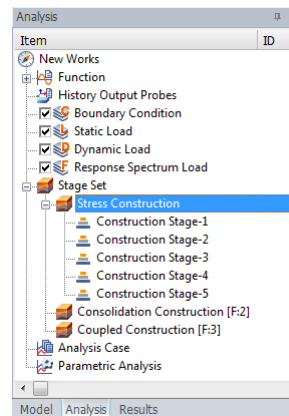


Stress  
Seepage  
Stress-Seepage-Slope  
Consolidation  
Fully Coupled Stress Seepage

Define construction stage sets by Stage Type  
(Sub-stages can be copied)



Define 'Auto Stage' for auto-generation

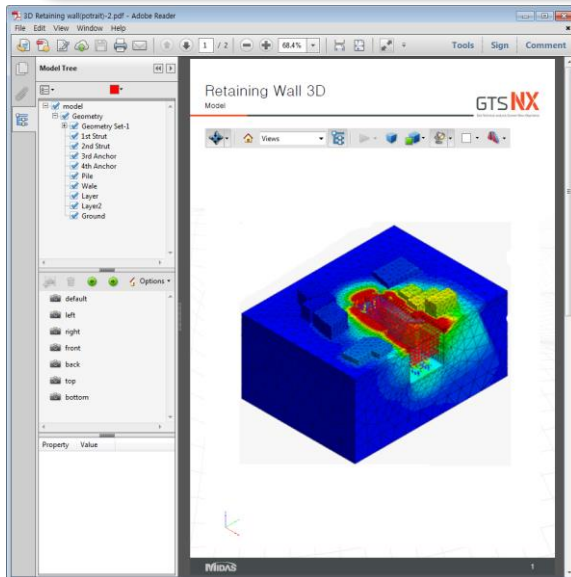


Check auto-generation of construction stages

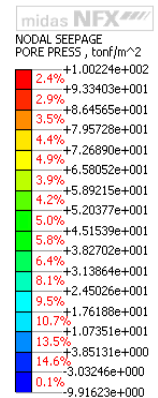
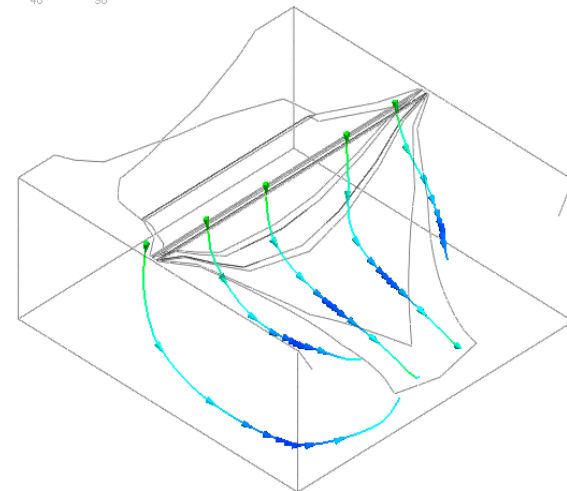
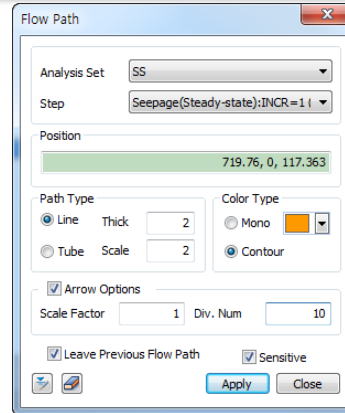
Manage model file, Avoid dual-tasks and Prevent confusion and analysis errors

# Additional post-processing features and performance

- Improve the overall speed of processing operations (stress contour of a practical test model:  
**20.4 sec → reduced to 1.1 sec**)
  - **3D dynamic PDF** report in which 3D model view can be manipulated
    - Real-time 3D flow path for seepage analysis (animation)
  - Element Contour Plot (simultaneous display of results from different element types)
- Improvement in results computation (combination of nodal/element results from different stages/analysis cases, conversion of dynamic analysis results to DB, etc.)



3D dynamic PDF

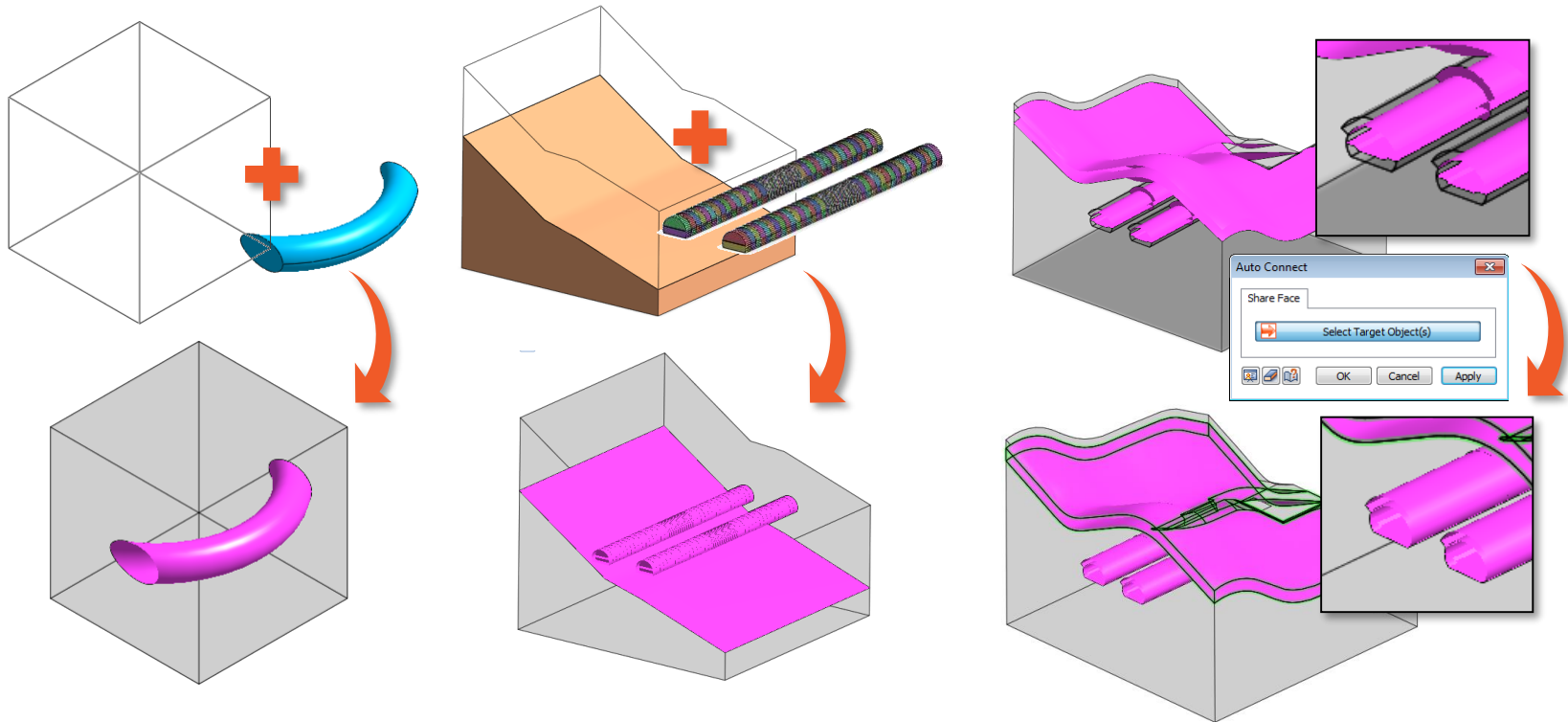


[ DATA ] SS, Seepage(Steady-state), INCR=1 (LOAD=1.000), [ UNIT ] tonf, m

3D flow path animation

# Addition of modelling features

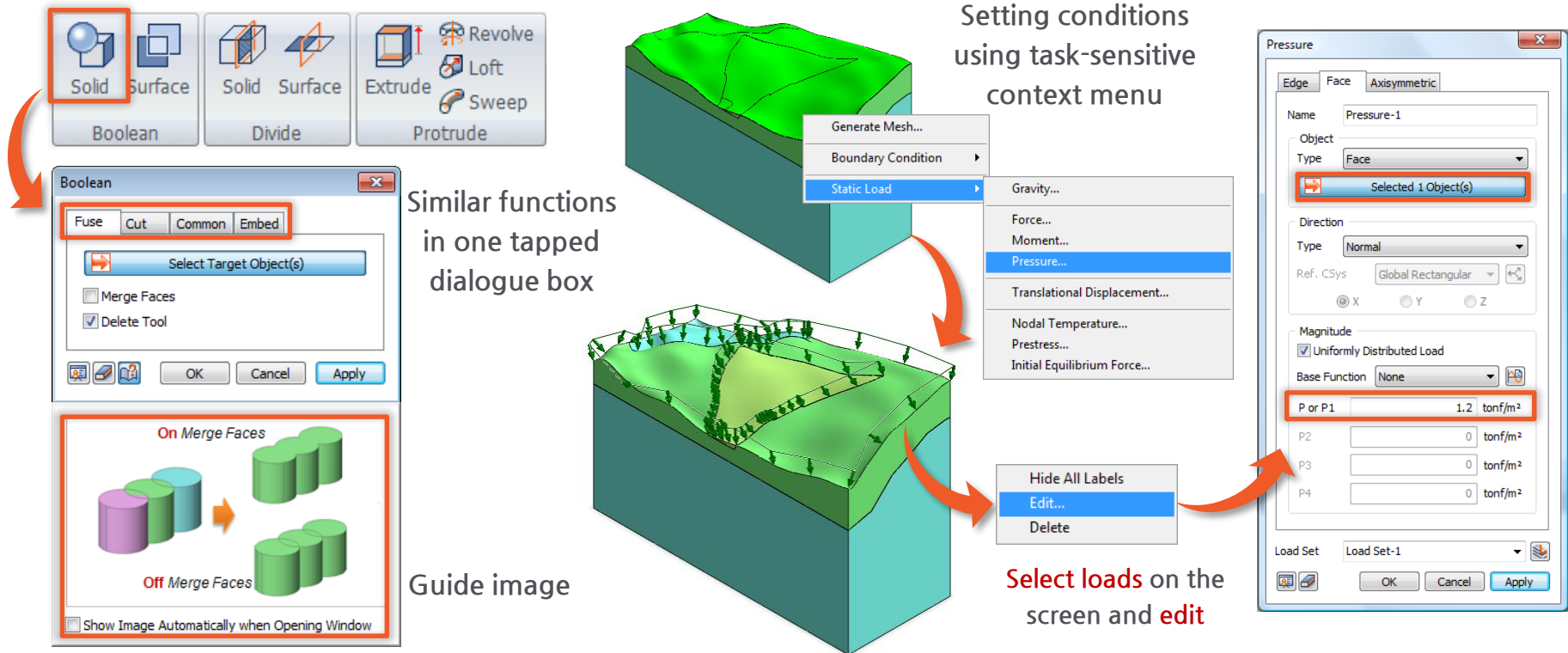
- Auto-generation of **co-face** between adjacent solids
- **Imprint automatically** in faces by selecting edges penetrating solids



Easy enough for beginners to avoid modeling errors from auto-correction

# GUI designed for ease of use for beginners

- Simplified menu structure and efficient use of **Ribbon menu/Tapped dialogue box**
  - Intuitive functions/options usage with **instructional guide images**
- Mouse motions minimised with **enhanced context menu** in the work window





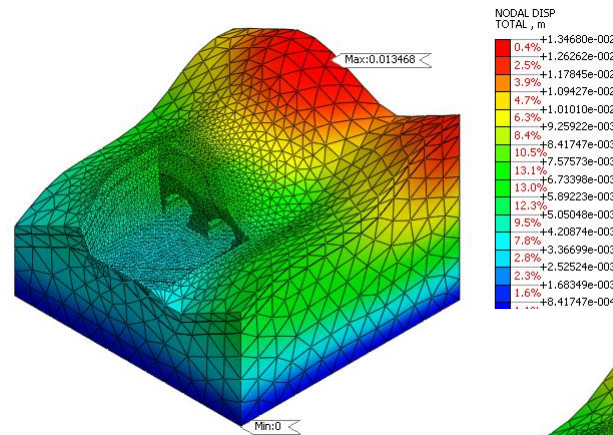
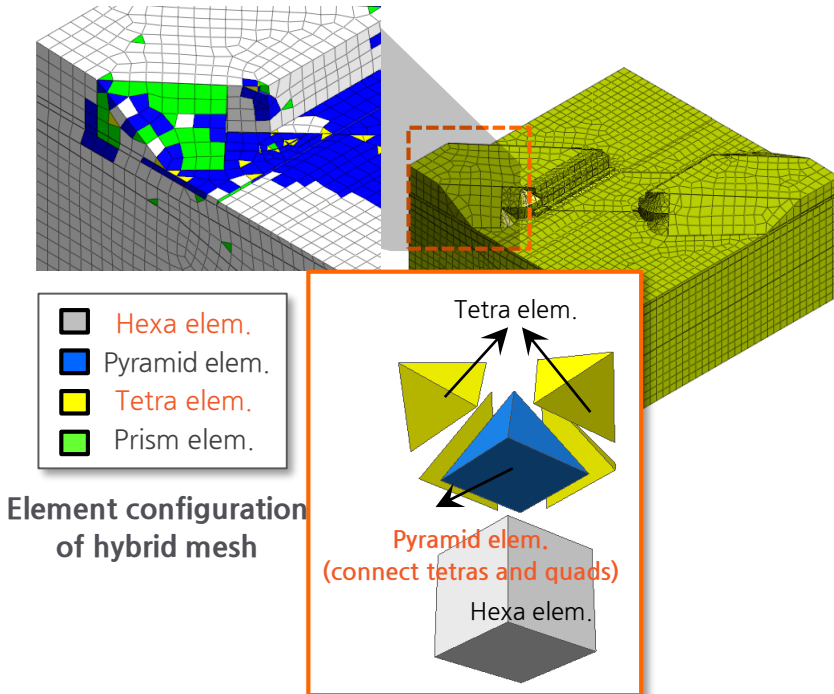


# Major Improvements in Analysis Features



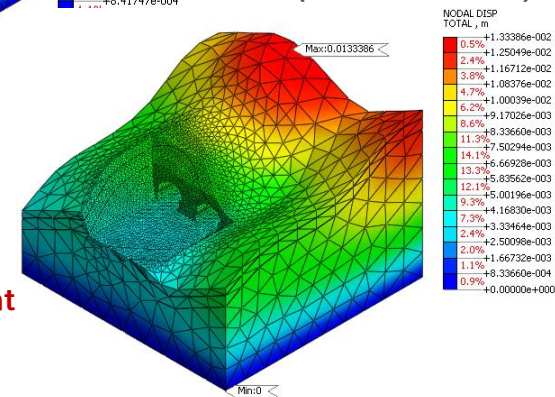
# High quality mesh (hybrid mesh) and bonded contacts

- Addition of **Hybrid mesh** for securing accurate results
- Bonded contacts: Auto-searching adjacent elements and auto-processing **bonded contacts** (for 3D analysis), Continuity - Compatibility Guaranteed



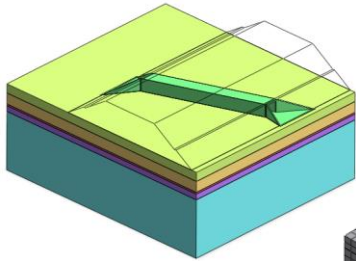
Final construction stage  
0.97% difference in the  
maximum total displacement

Coincident nodes model  
(without contacts)



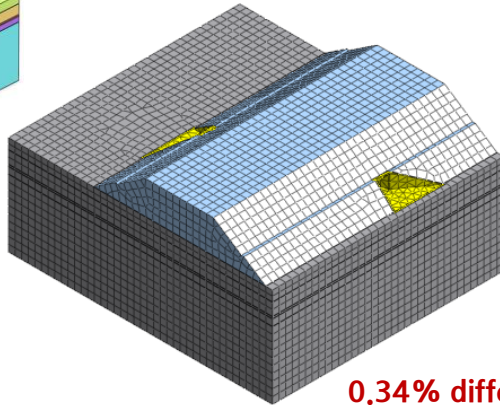
Hexa elem. for accuracy at the boundaries, partial tetra elem. inside the model

# Hybrid mesh, bonded contact results comparison

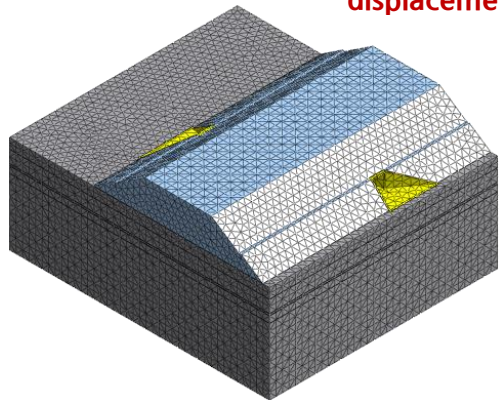


Hybrid elements + contacts

Elements	34,853
Nodes	32,684

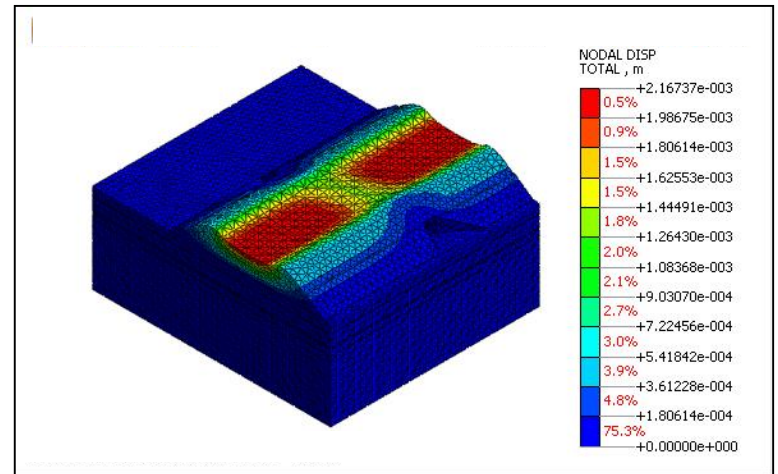
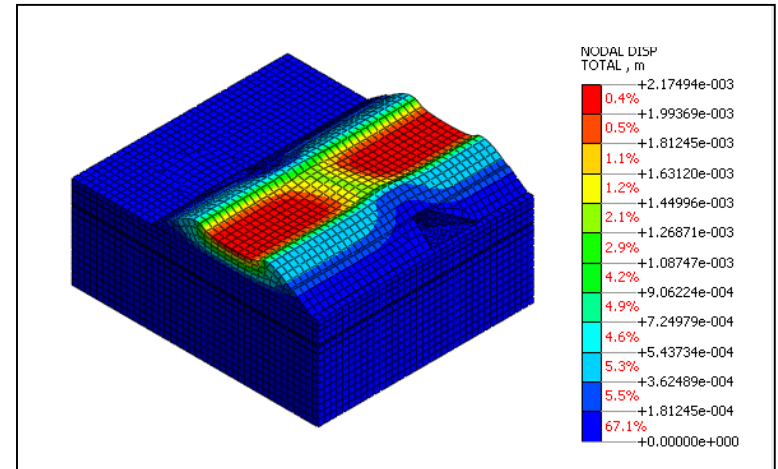


0.34% difference in displacement results



Coincident nodes

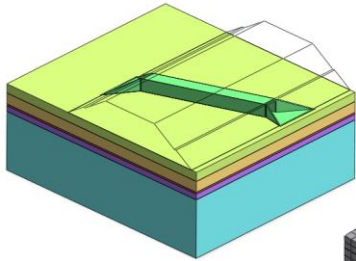
Elements	177,142
Nodes	32,476



Hybrid mesh + Contact vs. Coincident Nodes

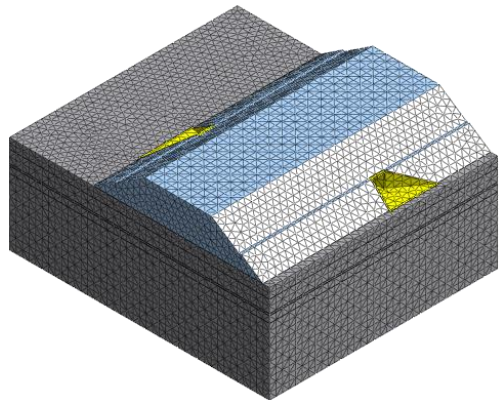
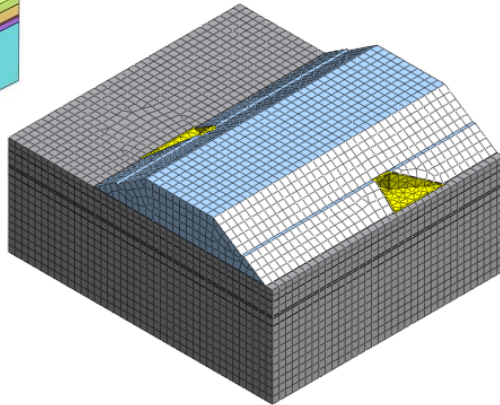


# Hybrid mesh, bonded contact results comparison



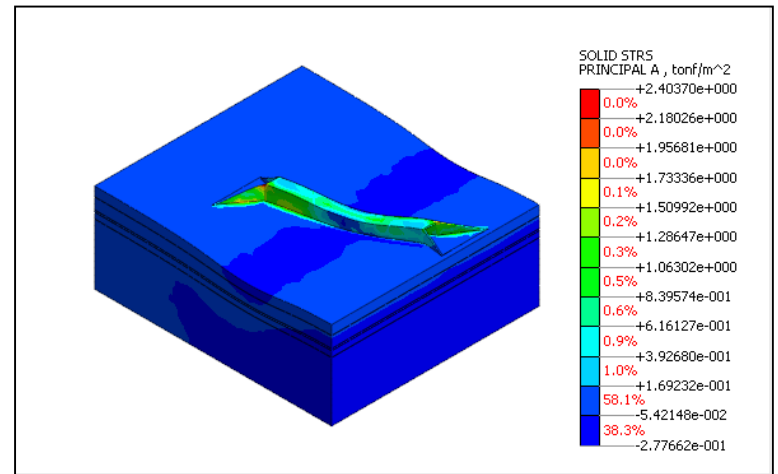
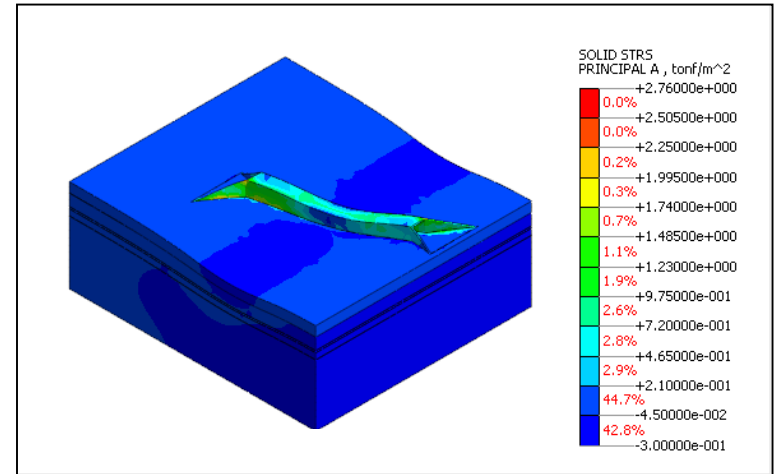
Hybrid elements + contacts

Elements	34,853
Nodes	32,684



Coincident nodes

Elements	177,142
Nodes	32,476

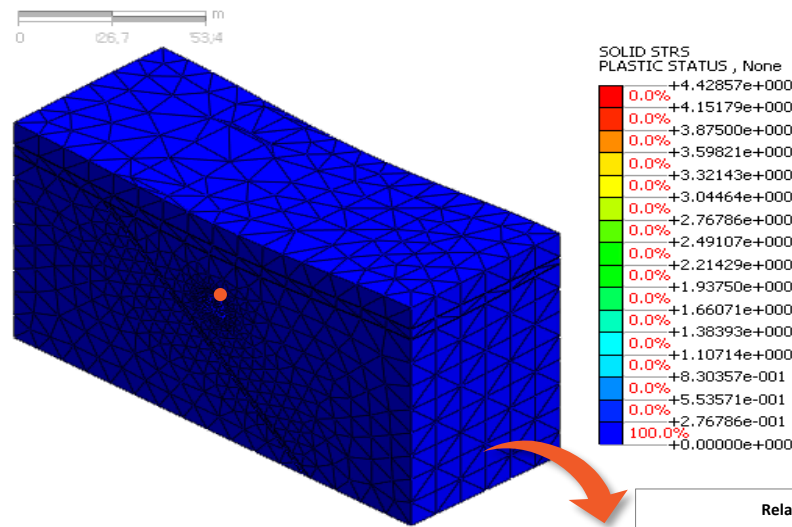
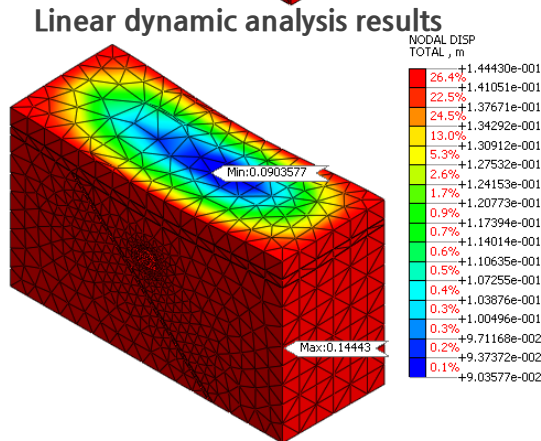
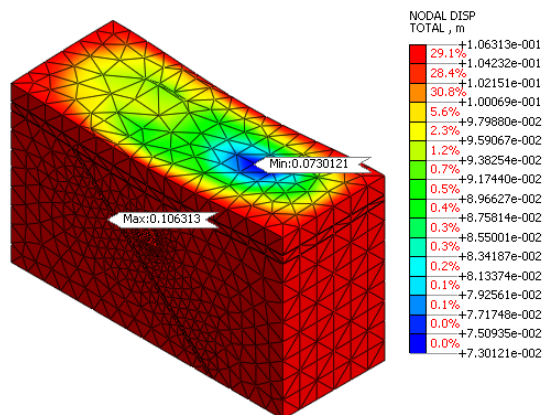


Hybrid mesh + Contact vs. Coincident Nodes



# Comparison of linear-nonlinear time history dynamic

- Areas using Nonlinear Dynamic Analysis: Soil-Structure Interaction Behaviour, Evaluation of embankment stability under dynamic loading, Crash loads, Seismic loads due to structural response assessment, Liquefaction evaluation (user-defined model can be implemented)



[ DATA ] NT, Nonlinear Time History, INCR=1 (TIME=0.1000), [ UNIT ] kN, m

Linear dynamic analysis plastic stress results

Soil deformation is mostly plastic  
→ Nonlinear analysis renders more realistic results

