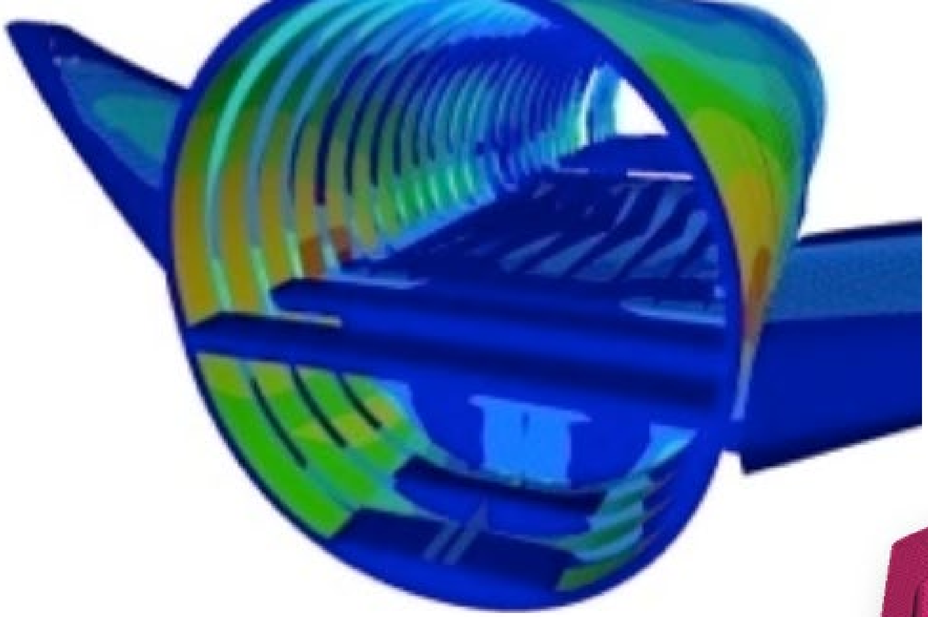
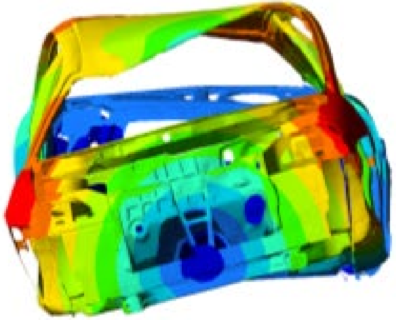
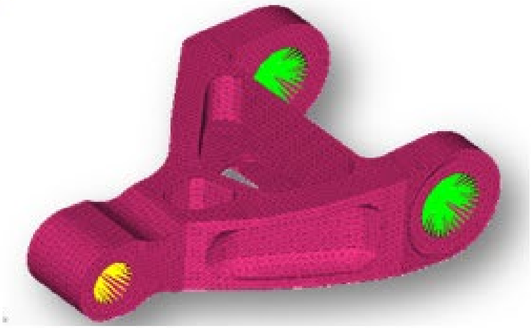






OPTISTRUCT FOR LINEAR ANALYSIS, V2019
CHAPTER 1: INTRODUCTION TO LINEAR ANALYSIS

Altair

AGENDA

ABOUT OPTISTRUCT

Altair **OptiStruct** is an industry proven, modern structural analysis solver for **linear and non-linear structural** problems under static and dynamic loadings. It is the market-leading solution for structural design and optimization.

Based on finite-element and multi-body dynamics technology, and through advanced analysis and optimization algorithms, **OptiStruct** helps designers and engineers rapidly develop innovative, lightweight and structurally efficient designs.

OptiStruct is used by thousands of companies worldwide to analyze and Optimize structures for their strength, durability and NVH (noise, vibration and harshness) characteristics.

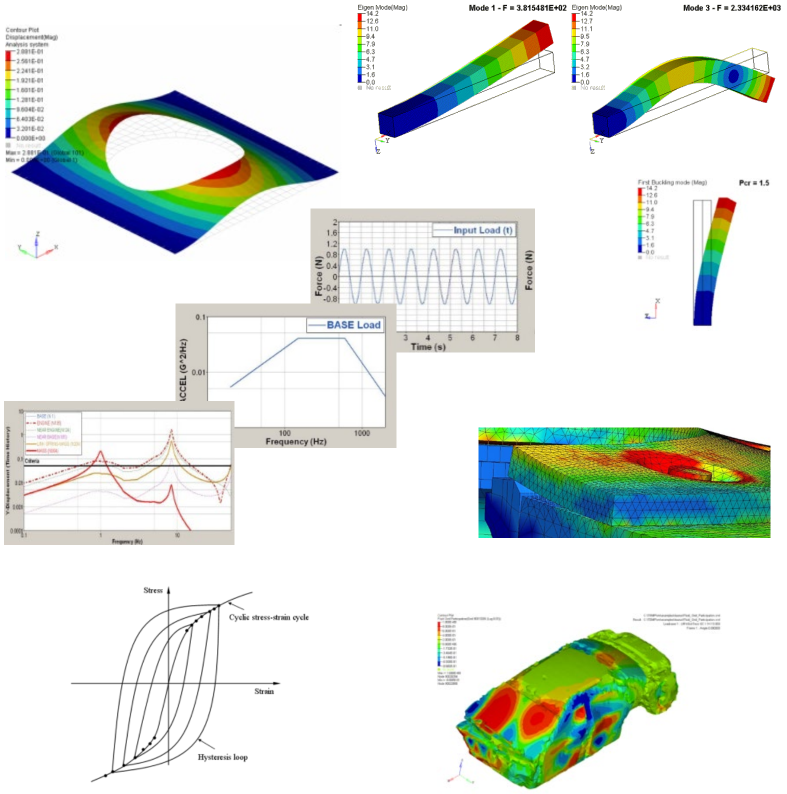
OVERVIEW OF OPTISTRUCT’S LINEAR ANALYSIS CAPABILITIES

Basic analysis features

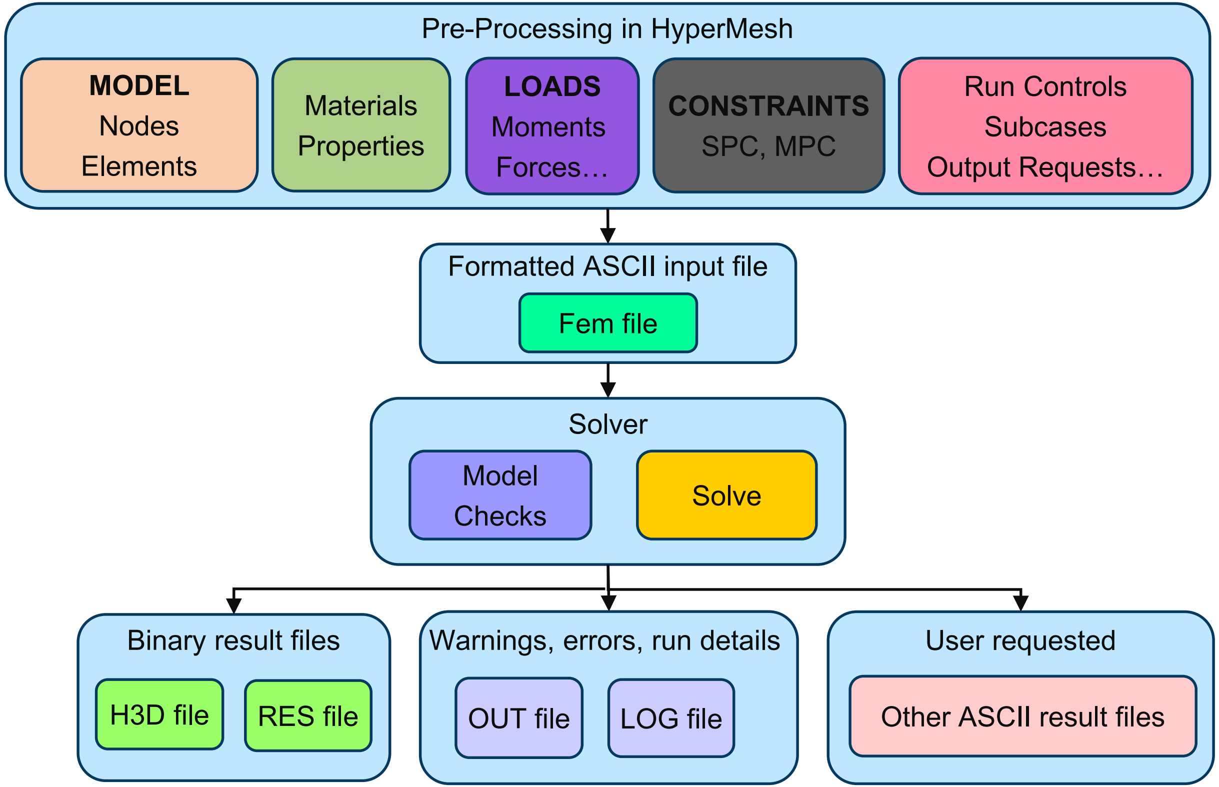
- Linear static, normal modes, linear buckling analysis & thermal-stress steady state analysis

Advanced analysis feature

- (Direct and Model) Frequency response function (FRF) analysis
- Random response analysis
- (Direct and Modal) Transient response analysis
- (Direct and Model) Transient response analysis (Fourier method)
- Non-linear contact analysis
- Acoustic Analysis (Structure and Fluid)
- Fatigue Analysis (σN and εN)



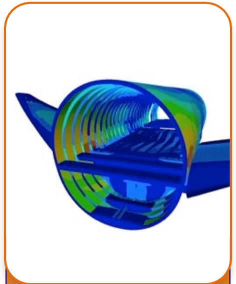
LINEAR ANALYSIS SOLUTION PROCESS



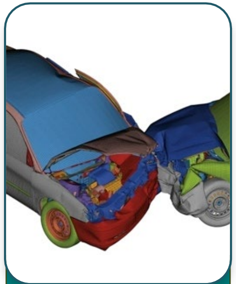
TYPE OF ANALYSIS

WHICH TYPE OF ANALYSIS?

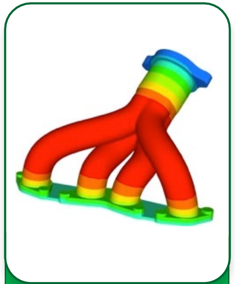
The analysis type usually belongs to one of the following disciplines:



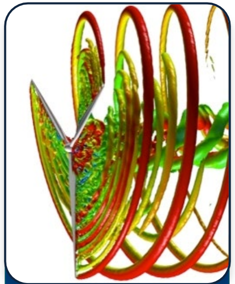
Structural Analysis



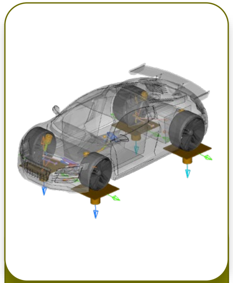
Crash, Safety, Impact & Blast



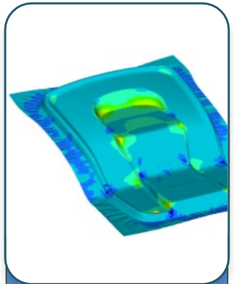
Thermal Analysis



Fluid Dynamics



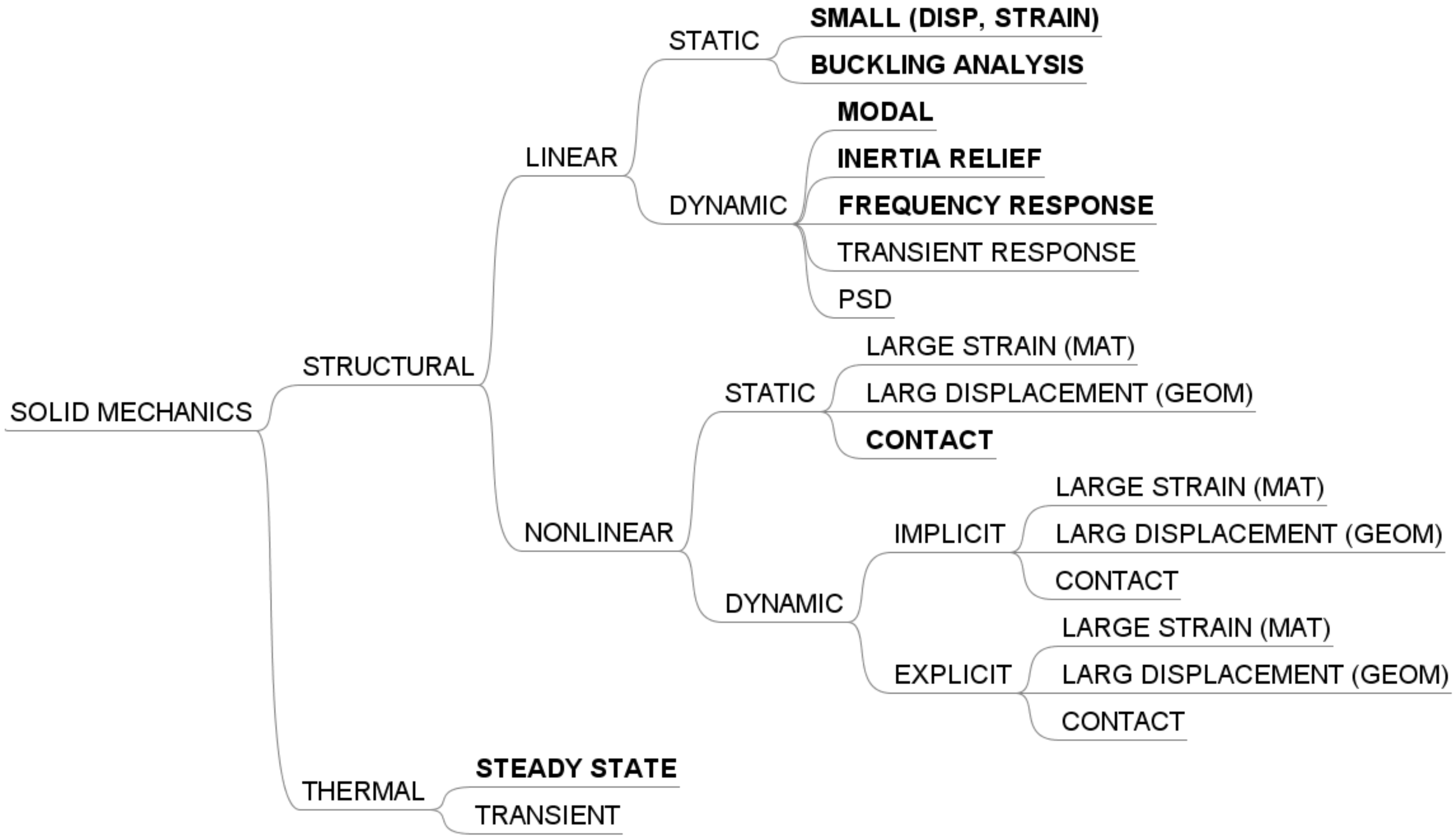
Systems Simulation



Manufacturing Simulation

Electro-Magnetics

WHICH TYPE OF ANALYSIS? – STRUCTURAL?



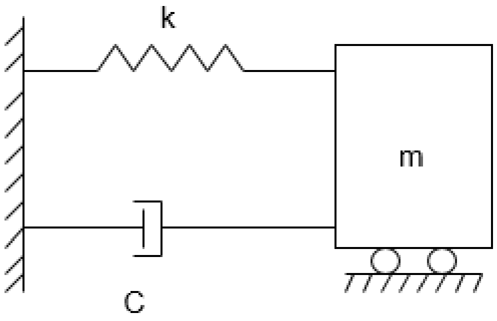
WHICH TYPE OF ANALYSIS? – STRUCTURAL?

Structural analysis, then the next questions are:

- Linear or nonlinear analysis?
- Static or dynamic analysis?

When Structure is subjected to some loading excitation & its responds with the following:

- Static/Dynamic forces due to stiffness?
- Inertia forces due to mass?
- Damping effect?

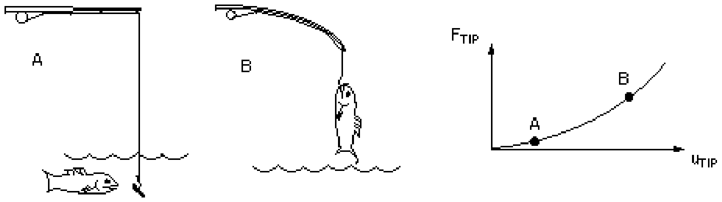


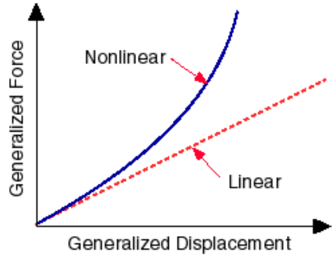
k :spring stiffness (N/m)
c: damping coefficient (N-s/m)
m: mass (kg)

WHICH TYPE OF ANALYSIS? – LINEAR OR NONLINEAR?

A linear analysis assumes that the loading causes negligible changes to the stiffness of the structure.

Typical characteristics are:



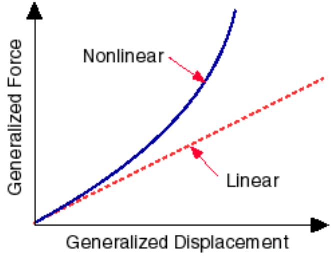


- All deformations and strains are small.
- Structural deformations are proportional to the loads applied. This infers that the loading pattern does not changed due to the deformed shape and no geometric stiffening occurs due to the application of the load.
- All materials behave in a linear elastic fashion. Therefore, the material deforms along the straight line portion of the stress-strain curve (no plasticity or failures occur). Highly localized stress concentrations are usually permitted as long as gross yielding does not take place.
- No abrupt changes in stiffness such as two bodies coming into and out of contact.

WHICH TYPE OF ANALYSIS? – LINEAR OR NONLINEAR?

A nonlinear analysis is needed if the loading causes significant changes in the structure's stiffness.

Typical reasons for stiffness to change significantly are:

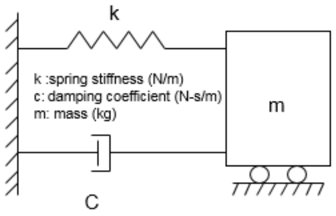


- Stresses that exceed that of the limit of proportionality of the material.
- Major changes in geometry
- Changes in geometry that remain after the process is finished.
- Processes involves buckling, crushing, wrinkling or plastic flows.
- Temperatures exceeding the melting temperature of the material.
- Large strains, finite strains can occur in hyperplastic materials.
- Nonlinear stress-strain laws, some materials have different compressive than tensile strengths.
- Boundary conditions change due to the application of load (contact).
- The direction of load application changes with deformation (follower forces such as pressures).

WHICH TYPE OF ANALYSIS? – STATIC OR DYNAMIC?

Static vs. Dynamic Analysis: For a single degrees of freedom system

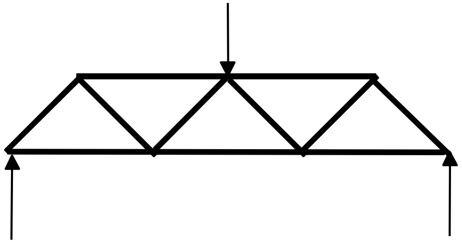
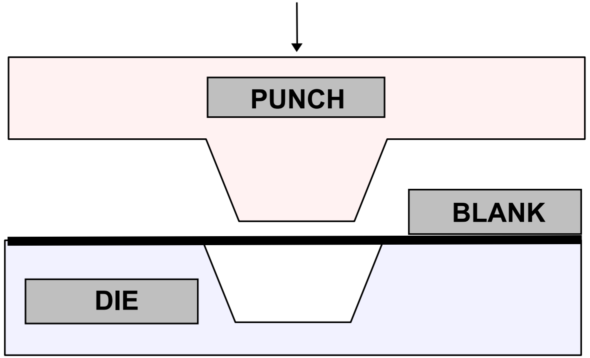
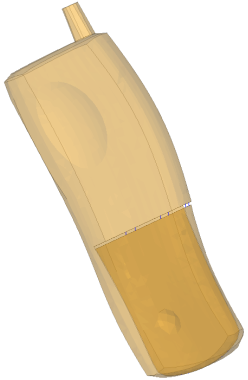
Static	Dynamic
A Static analysis is independent of time. Inertial forces are either ignored or neglected.	A dynamic analysis is dependent of time. Inertial forces are not ignored and damping effects the solution.
Force is Constant	Force is function of time
Velocity is zero	Velocity is function of time
Acceleration is zero	Acceleration is function of time



For example, consider the analysis of a trampoline.

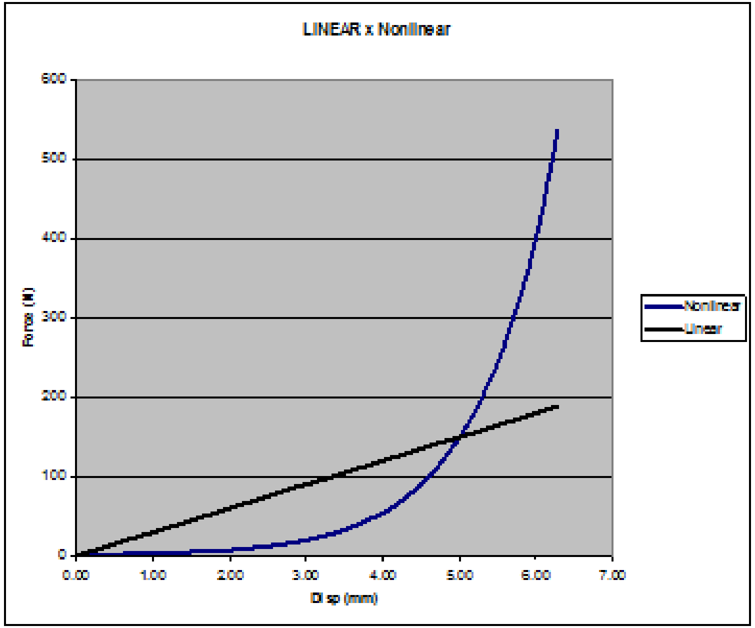
- If the boy is standing still, it might be sufficient to do a static analysis.
- But if the boy is jumping up and down, you will need to do a dynamic analysis.

WHICH TYPE OF ANALYSIS? – STATIC OR DYNAMIC?

STATIC	“QUASI” STATIC	DYNAMIC
		
Structural problems	Metal forming	Impact problems
S F = 0	**S F » 0**	**S F = ma**

GUIDELINE QUESTIONS FOR RUNNING A LINEAR ANALYSIS

- What are the loads on this system or component?
- Are the loads static or dynamic?
- Which is the expected load path?
- What are the design criteria? (Stress, displacement, Strain, Life, etc.…)
- Where do we expect the high stresses and what limit can I accept?
- Is it a linear or non-linear problem?
- How can I verify the FEA results?

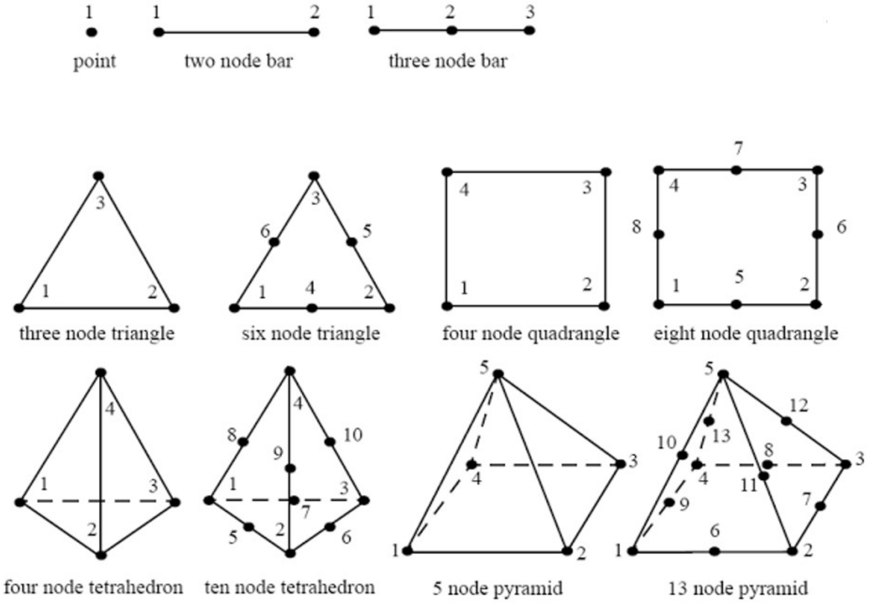


TYPES OF ELEMENTS AND MATERIALS

INTRODUCTION: 1D, 2D & 3D ELEMENTS

In OptiStruct , there are a selection of element types available to the user. They are usually categorized as 1D, 2D & 3D types.

- 1D or Line Elements
- 2D or Area Elements
- 3D or Volume Elements
- Special Elements



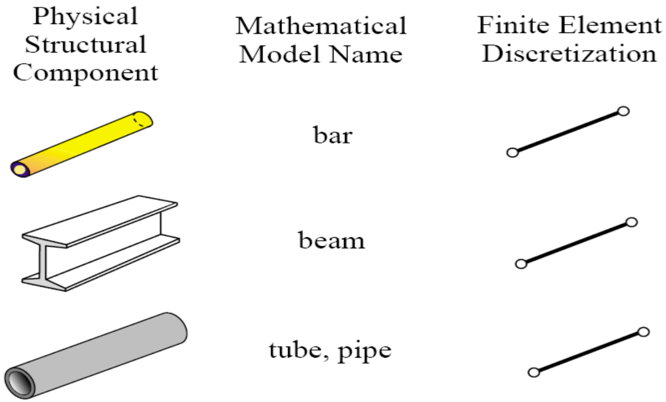
TYPES OF 1D OR LINE ELEMENTS

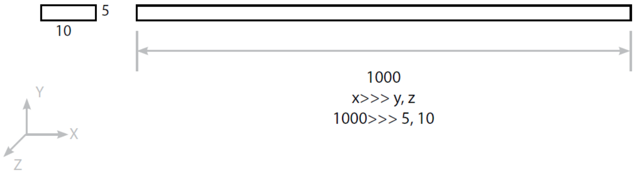
List of few 1D elements

Spring	CBUSH - a general spring-damper element that supports forces, moments, and displacements along the axis of the element.
Beam	CBEAM - a general beam element that supports all types of action listed above.
Rods	CROD - a simple, axial bar element that supports only axial forces and torsional moments.
Gap	CGAP - a gap element that supports axial and friction forces. It does not have to be placed between grid points. It can also connect surface patches.
Weld	CWELD - a simple, axial bar element that supports forces, moments, and torsional moments. It does not have to be placed between grid points. It can also connect surface patches

TYPES OF 1D OR LINE ELEMENTS

One of the dimension is very large in comparison to the other two:





- Element shape – Line
- The remaining two dimensions, the cross sectional area as property.
- Element type – Rod, bar, beam, & bush etc.
- Practical applications – Long shafts, beams, pin joint, connection elements, etc..

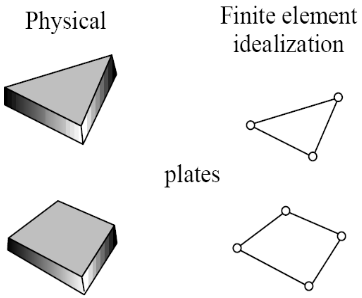
TYPES OF 2D OR AREA ELEMENTS

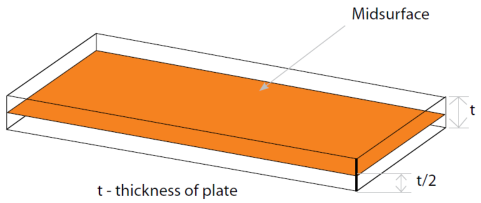
List of few 2D elements

Membrane	CTRIA3, CTRIA6, CQUAD4, CQUAD8
Plane Stress	CTRIA3, CTRIA6, CQUAD4, CQUAD8
Plane Strain	CTRIA3, CTRIA6, CQUAD4, CQUAD8
Shear Stress	CSHEAR - These elements support shear stress in their interior and extensional forces between their adjacent grid points. These may also be used to model thin buckled plates. These elements are used in situations where the bending stiffness and axial membrane stiffness of a plate is negligible

WHEN TO USE 2D OR AREA ELEMENTS

2-D elements are used when two of the dimensions are very large in comparison to the third dimension.





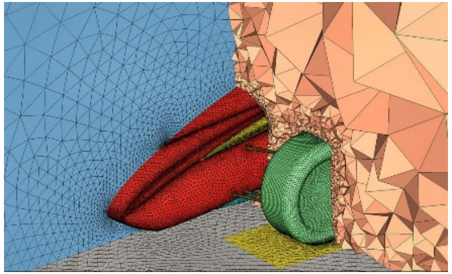
- Element shape – Quad, tria
- Remaining dimension i.e. thickness as element property
- Element type – Thin shell, plate, membrane,
- Practical applications – Sheet metal parts, plastic components like instrument panel etc.

TYPES OF 3D OR VOLUME ELEMENTS

The three-dimensional solid elements are used to model thick plates, solid structures. In general, structures in which the lateral dimensions are of the same order of magnitude as the longitudinal dimensions can support the use of three-dimensional solid elements in modeling.

The elements in this category are the `CHEXA`, `CPENTA`, `CPYRA`, and `CTETRA`.

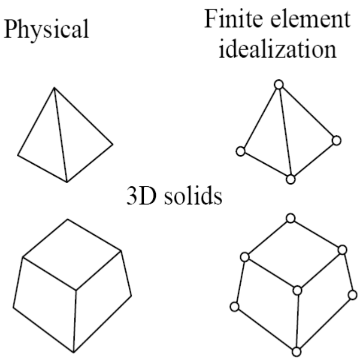
The property definition associated with these elements is called `PSOLID`.

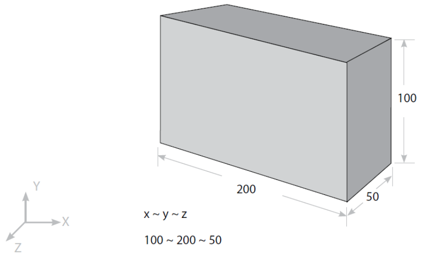


Practical applications – Casting, Molding, or Extruded components etc.

WHEN TO USE 3D OR VOLUME ELEMENTS

3-D elements should be used when all dimensions are comparable.





- Element shape – Tetra, penta, hex, pyramid
- Element type – Solid
- Practical applications – Gear box, engine block, crankshaft, etc.

SPECIAL ELEMENTS

OptiStruct has a range of special elements for modelling boundary conditions and connections. The main ones are outlined here:

- Gap elements for contact modelling. Gap elements are used to model point-to-point contact conditions. The elements have a large stiffness when active and a small (but non-zero) stiffness when in an inactive state. The element becomes active when it comes within a specified proximity of a specified object in the analysis.. The active state can usually accommodate both compression and tension.
- Spring elements for linear/nonlinear support or semi-rigid connection modelling. They are generally used for elastic support or for a specified connection behavior.
- Rigid elements are generally used to model rigid parts of a structures without having to assign the computationally expensive usual elastic elements. They are useful in dynamic analysis to account for distributed mass and inertia. They are also regularly used in kinematic type simulations away from any areas of interest in the model.
- Mass elements are used to define concentrated mass in a model. They are useful in parts of the model where there is geometry missing or there is more material than there should be (due to the oversimplification at the feature suppression stage, e.g. filled-in holes).

SPECIAL ELEMENTS

- Contact elements are similar to gap elements, they are used to define a contact between two surfaces. On the creation of the elements, a set of slave nodes and master nodes are defined. Contact occurs if either group of nodes attempts to penetrate the other. Friction properties can usually be applied to the element.
- Connector elements are used to couple two independent meshes at an interface. They are particularly useful when two parts of the model were created independently of each other. There are variants for coupling surfaces to surfaces, beams to surfaces and shell edges to surfaces.
- A SUPERELEMENT is a reduced representation of the behavior and performance of a structure or a part of a structure. DMIG is the acronym for Direct Matrix Input at Grid points. This matrix is defined by a single header entry and one or more column entries. The DMIG Bulk Data Entry can be used to directly define matrices to be included in the model (such as stiffness and mass matrices). DMIG entries can be selected in the case control section using K2GG, M2GG, B2GG entries and so on for inclusion in the model
- GENEL – General Finite Element. Defines the stiffness or flexibility of a general element connected to an arbitrary number of grids.

VALIDATION OF ELEMENTS WITH AN I-BEAM BENDING MODEL

This validation document will compare the analytical results of a simply supported I-beam undergoing bending to those obtained from OptiStruct.

Details: The I-Beam in consideration has a uniform cross section and is assigned a homogeneous, isotropic, linear elastic material (steel). The I-Beam is loaded at its center of gravity, under the action of one single point of force.

Click on the document to open

I-Beam Bending
Validation of 1D, 2D, and 3D OptiStruct FE models to analytical results

Introduction

Oil and Gas industry uses steel made I-Beams for many offshore and onshore application. For this paper, we are using common profile dimensions for those I-Beams [1]

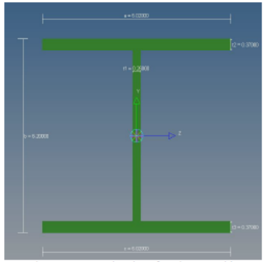


Figure 1: Cross section view of steel I-Beam with dimensions used.

Figure 3: I-Beam with vertical load at middle

Theoretical Calculations

The I-Beam has a uniform cross section with linear elastic isotropic homogenous material, Steel [2] The I-Beam is not subjected to gravity, and has no pre-strains present.

	Young Modulus	Density
Steel	30,000,000	0.3

Figure 2: Material properties used for the I-Beam

In our comparisons, we only consider one scenario, the I-beam has a vertical point load at the center of gravity of the I-Beam.

OptiStruct

OptiStruct is an implicit Finite Element Analysis solver with embedded optimization technologies, developed by Altair. This software is commonly used for structural analysis with linear and nonlinear problems under static and/or dynamic loadings across many industries, including Oil and Gas.

Finite Element Analysis: Model Building

Models were built using beam (1D) shell (2D) and solid (3D) discretization schemes. A second shell model was built with a coarser mesh to accommodate for a traditional Aspect Ratio (AR) of 1:5 and a thickness to length ration of 1:10. All the Finite Element models were solved using OptiStruct.

Mesh	No. Elems	No. Nodes	AR
1D: CBEAM	20	21	N/A
2D: CQUAD4 Fine Mesh	13392	13801	1.03
2D: CQUAD4 Coarse Mesh	1128	1238	2.04
3D: CHEXA 4 elem across section	220800	279465	4.7

VALIDATION OF ELEMENTS WITH AN I-BEAM BENDING MODEL

Observation about 1D – Beams (for this particular problem)

- Use beams when you want to see overall deflections in along, slender beam or truss-type structure.
- Although the beam elements give the closest results to the equations in this simple case, beam elements do not show local stress concentrations where the beams are attached. The cross section of beam elements does not deform under loading.
- You must carefully examine the beam stress results. You may display stresses at different recovery points or display stress contours on the cross section of an individual beam element.
- Although beam elements take longer to define since you must define the cross section properties, the solve time is the fastest with this element type.

Objective	Modeling Time	Solving Time	Quality of Results
Best	Quicker	Fastest	Best

VALIDATION OF ELEMENTS WITH AN I-BEAM BENDING MODEL

Observation about 2D – Shells (for this particular problem)

- Use shell elements for thin-walled parts. Use these elements if you have a general 2D model to analyze.
- With thin-shell elements, be careful to enter the correct physical property to define the thickness. This element assumes that the stress varies linearly from top to bottom. These elements give a good display of stresses along the plane of the elements, but display stresses on both top and bottom surfaces to get the complete picture.
- A thin-shell model requires more elements than beams to get good results, but still solves relatively fast.

Objective	Modeling Time	Solving Time	Quality of Results
Better	Quickest	Faster	Better

VALIDATION OF ELEMENTS WITH AN I-BEAM BENDING MODEL

Observation about 3D – Solids (for this particular problem)

- Use solid elements when you cannot take advantage of the 1D/2D element technique and you need to analyze the structure lots of fine details (Fillets, Holes, Chamfers).
- Solid elements give a complete picture of the 3D stress through the part, but use more degrees of freedom (DOF), so take longer to solve. Models may become very complex and take a very long time to solve if you have small features to accurately model

Objective	Modeling Time	Solving Time	Quality of Results
Good	Quick	Fast	Good

NUMBER OF DOFS ELEMENTWISE

Dimensional	Element	Description	DOFs*
One Dimensional	Bush	CBUSH - a general spring-damper element that supports forces, moments, and displacements along the axis of the element.	6
	Beam	CBEAM - a general beam element that supports all types of action listed above.	6
	Rods	CROD - a simple, axial bar element that supports only axial forces and torsional moments.	6
	Gap	CGAP - a gap element that supports axial and friction forces. It does not have to be placed between grid points. It can also connect surface patches.	1
	Weld	CWELD - a simple, axial bar element that supports forces, moments, and torsional moments. It does not have to be placed between grid points. It can also connect surface patches	96
Two Dimensional	PSHELL	CTRIA3, CTRIA6, CQUAD4, CQUAD8	6
	Plane Strain Plane Stress Membrane	CTRIA3, CTRIA6, CQUAD4, CQUAD8	3
	Shear Stress	CSHEAR - These elements support shear stress in their interior and extensional forces between their adjacent grid points. These may also be used to model thin buckled plates. These elements are used in situations where the bending stiffness and axial membrane stiffness of a plate is negligible	3
Three Dimensional	CHEXA CPENTA CPYRA CTETRA		3
Zero Dimensional	Spring	CELAS - scalar spring element	1

* Number of DOFs on each node

MATERIAL CLASSIFICATIONS

Isotropic	Orthotropic	Anisotropic	Laminates
Material having identical values of a property in all directions	Material having different values of a property in three directions	Material's properties such as Young's Modulus, change with direction along the object	Two or more layers of fibrous composite materials bonded together.
Properties independent of direction or axes	Different Properties along three axes	21 Independent constants	The individual layers generally are orthotropic or transversely isotropic with the laminate then exhibiting anisotropic
Metals	Wood, Concreate, Rolled metals	All real life materials are anisotropic, & we simplify them into Isotropic or Orthotropic	Fibre-reinforced polymers, carbon-fiber-reinforced polymer, glass-reinforced plastic.

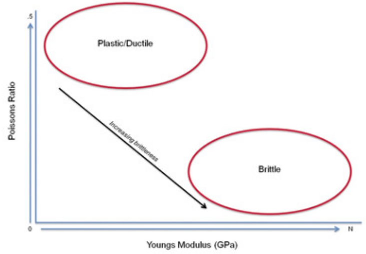
MATERIAL CLASSIFICATIONS

	Isotropic	Orthotropic	Anisotropic	Laminates
	Material having identical values of a property in all directions	Material having different values of a property in three directions	Material's properties such as Young's Modulus, change with direction along the object	Two or more layers of fibrous composite materials bonded together.
Structural	MAT1	MAT8	MAT2 MAT9 MAT9ORT	MAT1 MAT2 MAT4 MAT8
Thermal	MATT1 MAT4	MATT8	MAT5 MATT2 MATT9	MAT5

EXAMPLE MAT1

Defines the material properties for linear, temperature-independent, and isotropic materials.

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS						

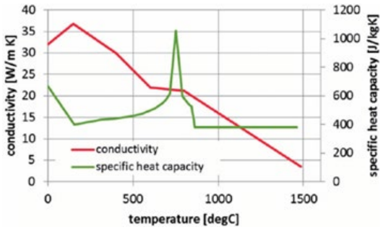


- 'E' Modulus of elasticity is slope of normal stress-strain curve in linear elastic domain. It is defined as normal stress / normal strain.
- 'G' Modulus of rigidity is slope of shear stress and strain curve in linear elastic domain. It is defined as shear stress/shear strain.
- 'NU' Poisson's ratio is defined as ratio of lateral strain to longitudinal strain. It is unit less entity.
- 'RHO' Mass density or more precisely, the volumetric mass density, of a substance is its mass per unit volume.

EXAMPLE MAT4

Defines constant thermal material properties for conductivity, density, and heat generation.

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)
MAT4	MID	K	CP	RHO	H		HGEN



- ‘K‘ Thermal conductivity (is thermal resistivity) is the property of a material to conduct heat.
  It is usually expressed in kelvin-meters per watt
- Heat capacity or thermal capacity is a measurable physical quantity equal to the ratio of the heat added to (or removed from) an object to the resulting temperature change.
  The unit of heat capacity is joule per kelvin.
- ‘RHO’ Mass density or more precisely, the volumetric mass density, of a substance is its mass per unit volume.

ELEMENTS AND MATERIAL COMPATIBILITY

		MAT1	MAT2	MAT3	MAT4	MAT5	MAT8	MAT9	MAT9ORT	MAT10	MATF#	MATHE	MATHF	MATPE1	MATS1	MATT#	MATUSR	MGASK
0D/1D Elements	NA	CMASS,CVISC, CDAMP, CGAP, CFAST,CBUSH, CELAS, CONM, RBE																
	CSEAM	✓						✓			✓					✓		
	CMASS#																	
	CWELD	✓									✓					✓		
	CBAR	✓			✓						✓					✓		
	CBEAM	✓			✓						✓					✓		
	CROD	✓			✓						✓					✓		
	CTUBE	✓									✓					✓		
2D Elements	CTRIA#	✓	✓		✓	✓	✓				✓		✓		✓*	✓		
	CQUAD#	✓	✓		✓	✓	✓				✓		✓		✓*	✓		
	CTAXI	✓		✓							✓							
	CTRIAX6	✓		✓							✓							
3D Elements	CHEXA#	✓			✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	✓	
	CTETRA#	✓			✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	✓	
	CPYRA#	✓			✓	✓		✓	✓	✓	✓			✓	✓	✓	✓	
	CPENTA#	✓			✓	✓		✓	✓	✓	✓	✓		✓	✓	✓	✓	
	CGASK#	✓																✓

MATERIAL AND ELEMENT PROPERTY COMPATIBILITY

	0D/1D Element Properties							2D Element Properties					3D Element Properties			
	NA	PSEAM	PWELD	PBAR	PBEAM	PROD	PTUBE	PSHELL	PCOMP	PCOMPG	PCOMPP	PAXI	PSOLID	PLSOLID	PCOMPLS	PGASK
MAT1		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓
MAT2								✓	✓	✓	✓					
MAT3												✓				
MAT4				✓	✓	✓		✓	✓	✓	✓		✓			
MAT5								✓	✓	✓	✓		✓			
MAT8	PELAS,							✓	✓	✓	✓					
MAT9	PBUSH,	✓											✓			
MAT9ORT	PFAST,												✓			
MAT10	PGAP,												✓			
MATF#	PDAMP,	✓	✓	✓	✓	✓	✓	✓				✓	✓			
MATHE	PVISC,													✓		
MATHF	PMASS							✓								
MATPE1													✓			
MATS1								✓					✓			
MATT#		✓	✓	✓	✓	✓	✓	✓					✓			
MATUSR													✓			
MGASK																✓

TYPE OF LOADS & BOUNDARY CONDITIONS

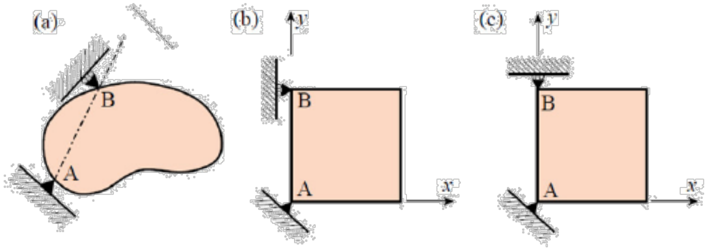
DEGREES OF FREEDOM ACROSS ALL DISCIPLINE

Discipline	Degree of Freedom	Label
Structural OptiStruct	Translations	UX, UY, UZ
	Rotations	ROTX, ROTY, ROTZ
Thermal OptiStruct	Temperature	TEMP
Acoustics OptiStruct	Velocities	VX, VY, VZ
	Pressure	PRES
Fluid (CFD) AcuSolve	Velocities	VX, VY, VZ
	Pressure	PRES
	Turbulent Kinetic Energy	ENKE
	Turbulent Dissipation Rate	ENDS

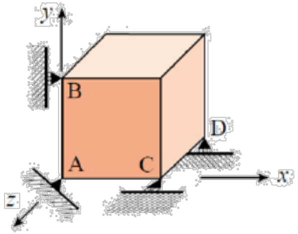
ILLUSTRATION: DEGREES OF FREEDOM OR BOUNDARY CONDITION

Constraints are used to prevent structures from having rigid body motion

Supporting Two Dimensional Bodies



Supporting Three Dimensional Bodies

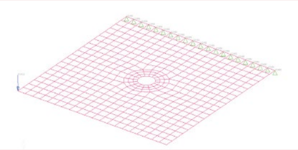
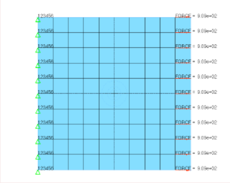
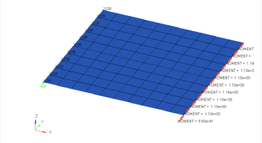


- If a structure is not constrained properly, an applied loads will cause infinite displacements
- Regardless of loading condition, 2D structure must be constrained against two translations (in this e.g. *x* and *y*) and one rotation (in this e.g. about *z*)
- Minimum no. of constraints for 2d structure is three.
- For 3d structure the minimum no. of degrees of freedom to be constrained is 6.
- More than one way to specify the minimum constraints

FORCE TYPE LOAD ACROSS ALL DISCIPLINE

Discipline	Force	Label
Structural OptiStruct	Forces	FX, FY, FZ
	Moments	MX, MY, MZ
Thermal OptiStruct	Heat Flow Rate	HEAT
Fluid (CFD) AcuSolve	Fluid Flow Rate	FLOW

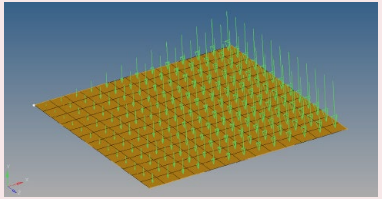
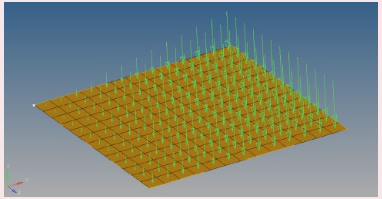
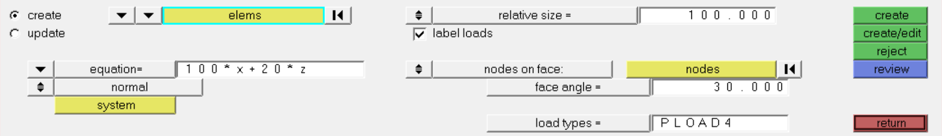
ILLUSTRATION: FORCES/MOMENT

Methods of applying Force	**Representation**	**Description**
Concentrated Force: Force applied at a Point or Single Node.		Concentrated loads impose High Stress Gradients Are they realistic Line Loads or Surface Loads to be more realistic
Concentrated Force: Equal force applied to all the nodes on the edge		Equal force applied to all the nodes on the edge Red hotspots on the corners of the plate Forces applied on the corner nodes act only on ½ of element edge
Bending Moment: applied to all the nodes on the edge		Moment is represented by double arrow, direction of the moment given by the right hand rule Magnitude of the moment at corner nodes is ½ that of the inside nodes

SURFACE LOAD TYPE ACROSS ALL DISCIPLINE

Discipline	Surface Load	Label
Structural OptiStruct	Pressure	PLOAD
Thermal OptiStruct	Convection	CONV
	Heat Flux	HFLUX
Fluid (CFD) AcuSolve	Fluid-Structure Interface	FSI
	Impedance	IMPD

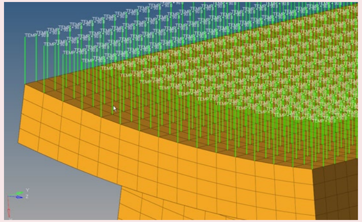
ILLUSTRATION: SURFACE LOAD

Methods of applying Force	Representation	Description
Pressure on Surfaces		Applied to the centroid of the element Direction of pressure is computed with the right hand rule using the grid point sequence specified on the element (on which the pressure is being applied) card Magnitude and direction computed automatically by OptiStruct using the shape functions of the element
Pressure on Surfaces with 'Variable' magnitude or with Equation		Pressure is a function of the *x* and *z* coordinates of the centroid of elements 

BODY LOAD TYPE ACROSS ALL DISCIPLINE

Discipline	Body Load	Label
Structural OptiStruct	Temperature	TEMP
Thermal OptiStruct	Heat Generation Rate	HGEN
Fluid (CFD) AcuSolve	Heat Generation Rate	HGEN
	Force Density	FORC

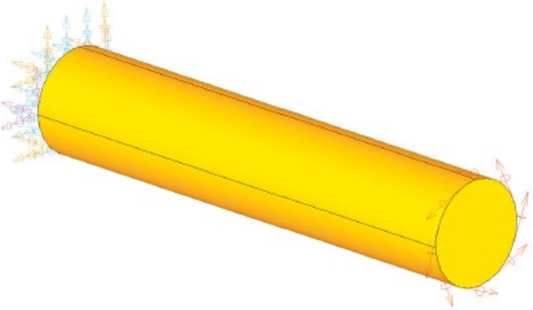
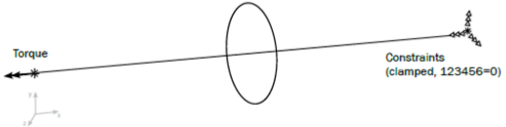
ILLUSTRATION: BODY LOAD

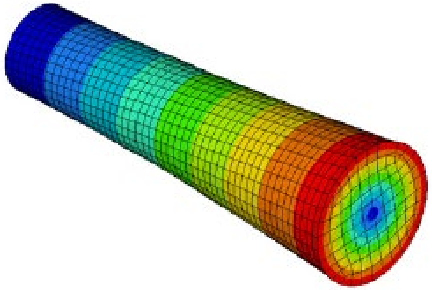
Methods of applying Force	Representation	Description
Temperature applied on nodes		Temperature is a body loads in the Structural analysis, which applied on the nodes.

ADDITIONAL LOAD TECHNIQUES

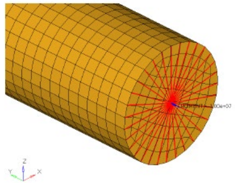
Torque

- bending moment applied parallel to the axis of a shaft

- Based on the right-hand rule: Point the thumb of your right hand towards the arrow direction. The direction of four fingers indicate the direction of torque.
- Applying Torque/Moments to Solid Elements, they do not have rotational stiffness. Common mistake to apply torques and moments to the nodes of the solid elements directly
- Correct way is to use (RBE2 or) RBE3 element to apply torque/moment

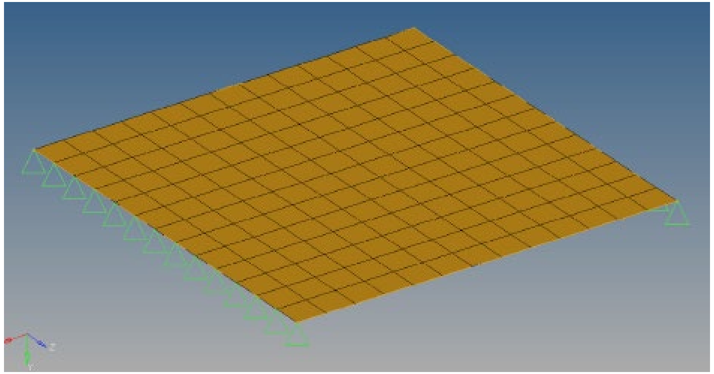


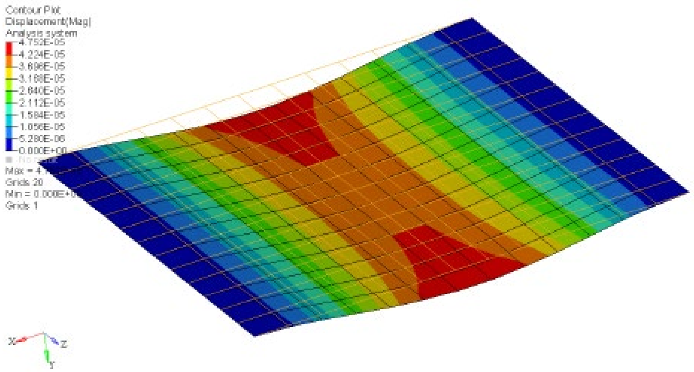
ADDITIONAL LOAD TECHNIQUES

Gravity Load

- Specify direction of gravity
- Make sure the value of acceleration due to gravity is consistent with the unit system being used

Name	Value
Solver Keyword	GRAV
Name	loadcol
ID	1
Color	
Include	[Master Model]
Card Image	GRAV
User Comments	Hide In Menu/Export
CID	<Unspecified>
G	9810.0
N1	0.0
N2	1.0
N3	0.0





TYPE OF LOAD & BOUNDARY CONDITION

Subcase Type	Bulk Load Cards
LINEAR STATIC	GRAV, RFORCE, FORCE, FORCE1, SPC, SPCD, PLOAD4, PLOAD, PLOAD1, PLOAD2, MOMENT, MOMENT1, ACCEL/ACCEL1/ACCEL2, DEFORM, PTFORCE, PTADJST
MODES	EIGRA, EIGRL SPC, MPC
BUCKLING*	GRAV, RFORCE, EIGRL, FORCE, FORCE1, SPC, SPCD, PLOAD4, PLOAD, PLOAD1, PLOAD2, MOMENT, MOMENT1, ACCEL/ACCEL1/ACCEL2, DEFORM, PTFORCE, PTADJST
THERMAL	SPC, TEMP, QBDY1, QVOL

* Include all the linear and modal analysis cards

QUESTIONS & ANSWERS