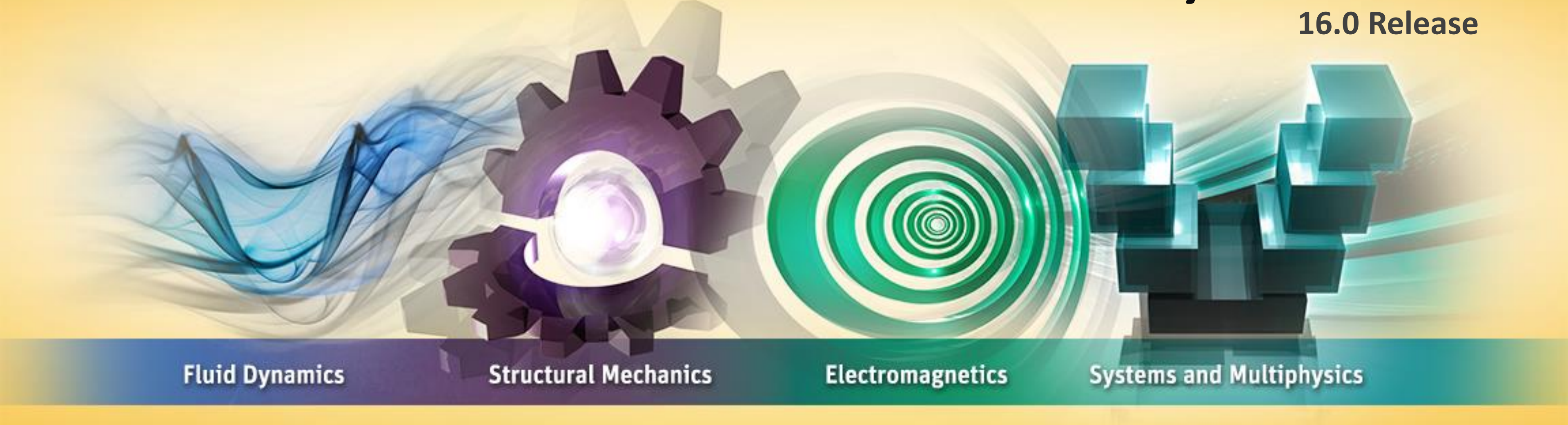
Lecture 7
Static Structural Analysis

16.0 Release



Introduction to ANSYS Mechanical

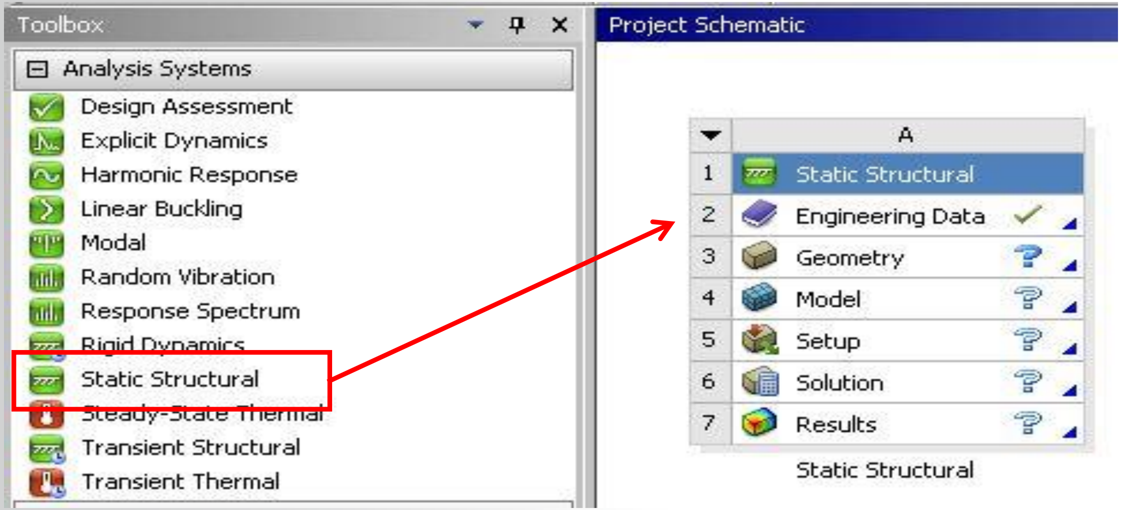
Chapter Overview

In this chapter, performing linear static structural analyses in Mechanical will be covered:

A. Basics of Linear Static Analysis
B. Geometry
C. Material Properties
D. Contact
E. Analysis Settings
F. Loads
G. Supports
H. Load and Support Display
I. Contact vs Supports
J. Solving Models
K. Workshop 7.1, Pump Assembly With Contact
L. Results and Postprocessing
M. Linear vs Non Linear
N. Workshop 7.2, Using Beam Connections
O. Appendix

A. Basics of Linear Static Analysis

The schematic setup for a linear static structural analysis is shown here.



. . . Basics of Linear Static Analysis

For a linear static structural analysis, the global displacement vector {x} is solved for in the matrix equation below:

$$[K]\{x\} = \{F\}$$

Assumptions made for linear static structural analysis are:

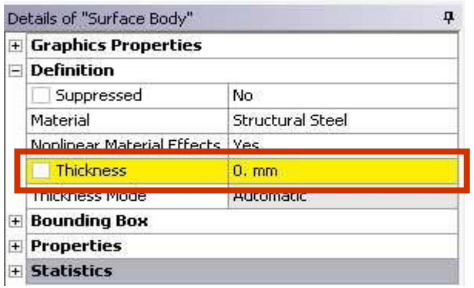
- [K] , which is the global stiffness matrix, is constant
  - Linear elastic material behavior is assumed
  - Small deflection theory is used
- {F} , which is the global load vector, is statically applied
  - No time-varying forces are considered
  - No damping effects

It is important to remember these assumptions related to *linear static* analysis. *Nonlinear static* and *dynamic* analyses are covered in other training courses.

B. Geometry

In structural analyses, all types of bodies supported by Mechanical may be used.

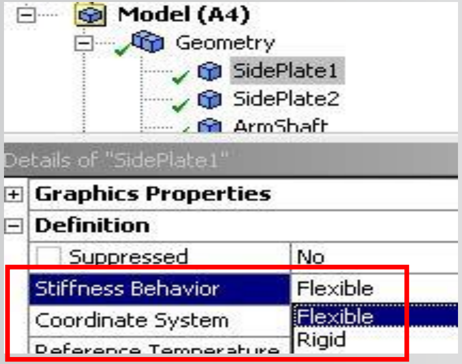
For *surface bodies*, thickness must be supplied in the “Details” view of the “Geometry” branch.

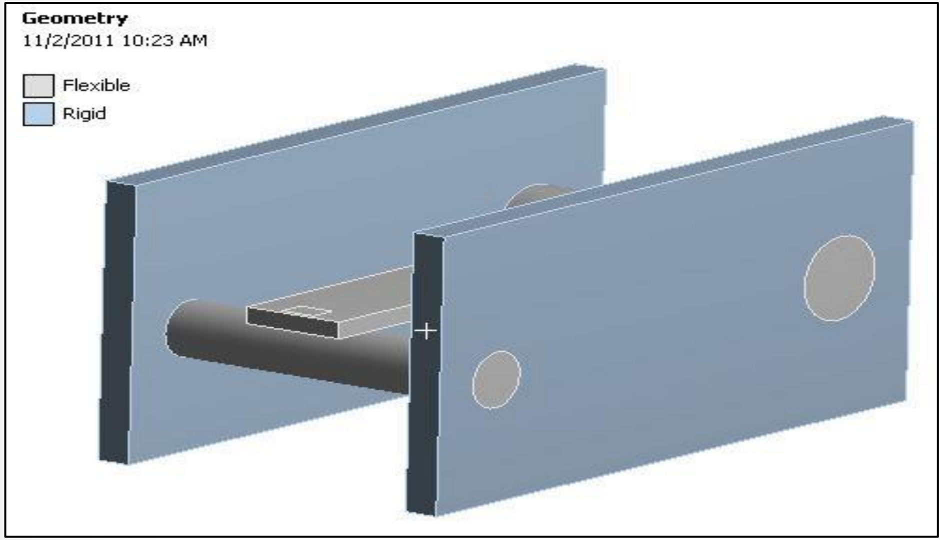


The cross-section and orientation of *line bodies* are defined within DesignModeler and are imported into Mechanical automatically.

. . . Geometry

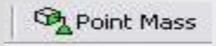
- **Mechanical allows a part's stiffness behavior to be defined as rigid/flexible.**
  - **A rigid body is not meshed with traditional finite elements. Rather it is represented using a single mass element and is thus very efficient in terms of solution times.**
  - **Parts in an assembly that are included only to transfer loads can be designated as rigid to reduce solution times and model sizes.**





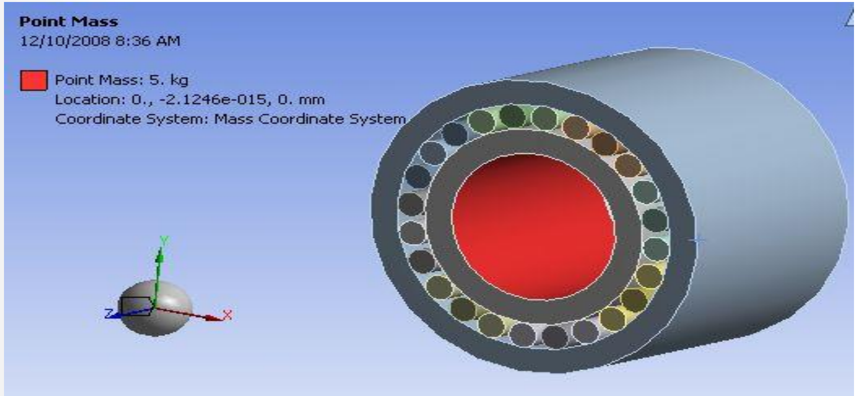
. . . Point Mass

A Point Mass can be added to a model's Geometry branch to simulate parts of the structure not explicitly modeled:



- **Point mass is affected by "Acceleration," "Standard Earth Gravity," and "Rotational Velocity". No other loads affect a point mass.**

Scope	
Scoping Method	Geometry Selection
Applied By	Remote Attachment Direct Attachment Remote Attachment
Geometry	
Coordinate System	
X Coordinate	1.5e-002 m
Y Coordinate	1.5e-002 m
Z Coordinate	3.e-002 m
Location	Click to Change
Definition	
Mass	1. kg
Mass Moment of Inertia X	0. kg·m²
Mass Moment of Inertia Y	0. kg·m²
Mass Moment of Inertia Z	0. kg·m²
Suppressed	No
Behavior	Deformable
Pinball Region	All

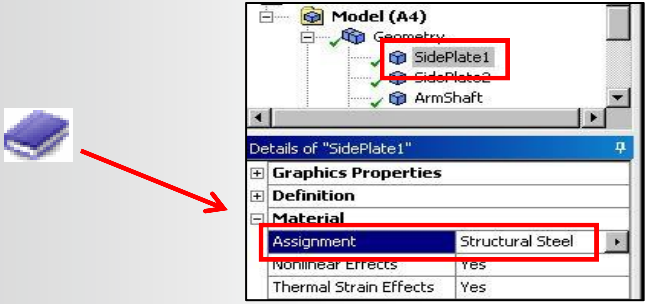


C. Material Properties

Young's Modulus and *Poisson's Ratio* are always required for linear static structural analyses:

- *Density* is required if any inertial loads are present.
- *Thermal expansion coefficient* is required if a temperature load is applied.
- *Stress Limits* are needed if a Stress Tool result is present.
- *Fatigue Properties* are needed if Fatigue Tool result is present.
  - Requires *Fatigue Module* add-on license.

- As shown earlier material properties are assigned in the part details in Mechanical. The user can chose from the dropdown list materials available to this project



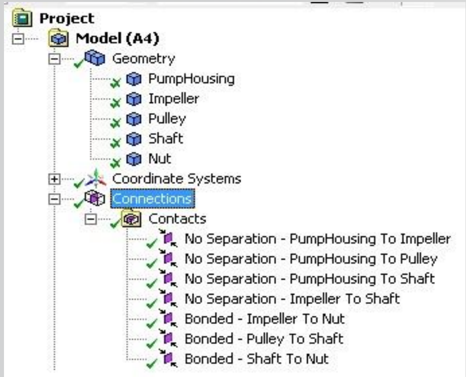
D. Contact

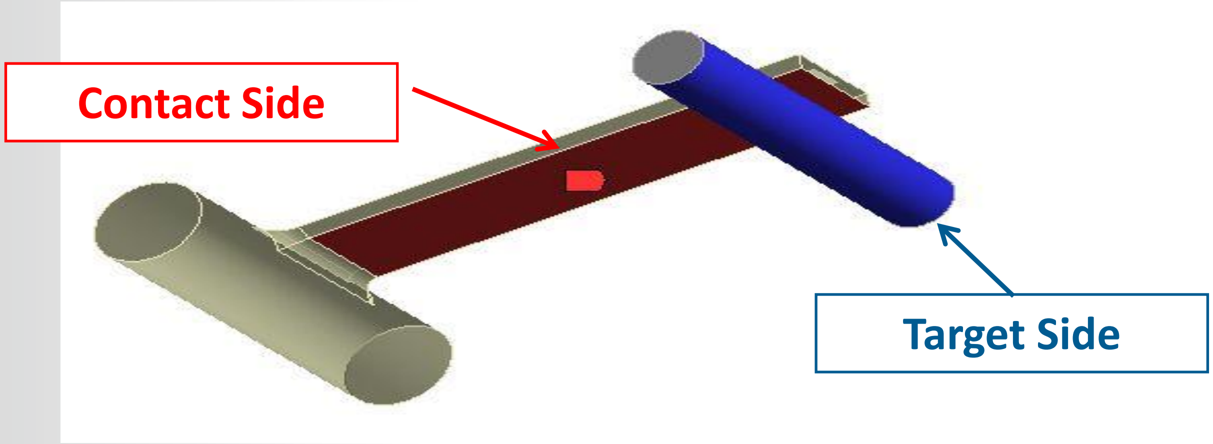
Surface contact elements in Mechanical can be visualized as a “skin” covering the surfaces of the parts in an assembly.

It is these elements that define the behavior when parts are in contact (e.g. friction, bonding, heat transfer, etc.).

In Mechanical contact pairs are color coded as shown here.

Note, contact is covered in more detail in connection lecture (L05_connections).

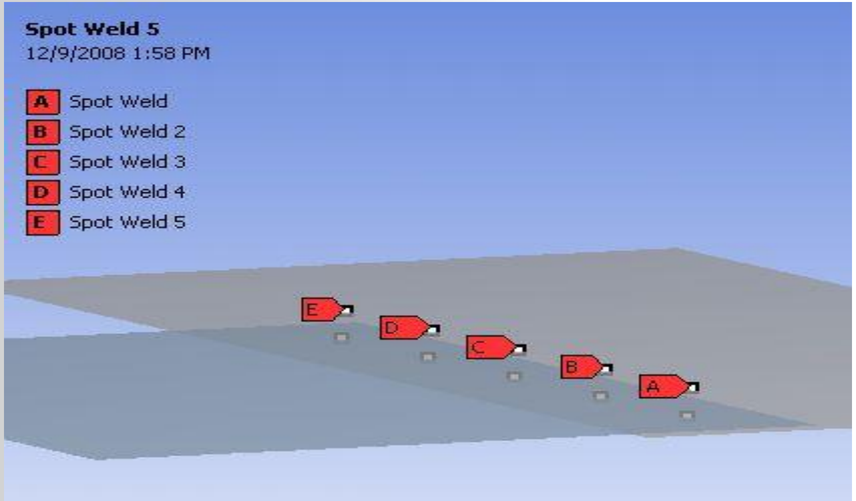




… Contact - Spot Weld

Spot welds provide a means of connecting shell assemblies at discrete points:

- **Spotweld definition is done in the CAD software. Currently, only DesignModeler and Unigraphics define supported spot weld definitions.**



E. Analysis Settings

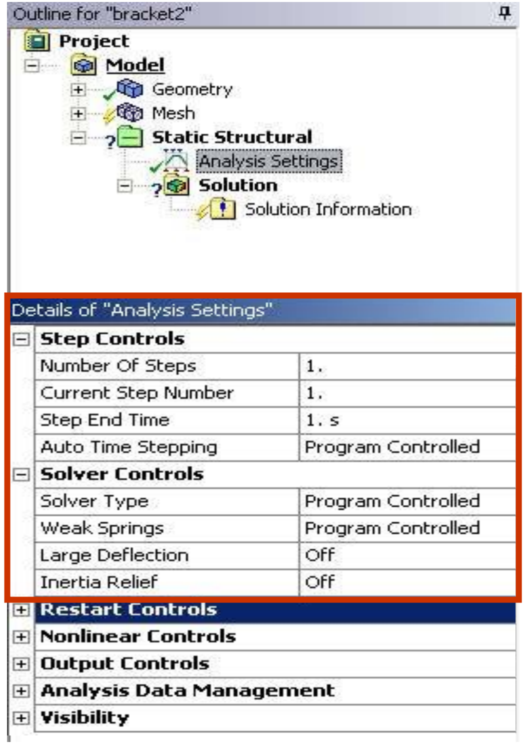
The “Analysis Settings” details provide general control over the solution process:

Step Controls:

- Specify the number of steps in an analysis and an end “time” for each step.

Solver Controls:

- Two solvers available (default program chosen):
  - Direct solver (Sparse solver in MAPDL).
  - Iterative solver (PCG solver in MAPDL).
- Weak springs:
  - Mechanical tries to anticipate under-constrained models.



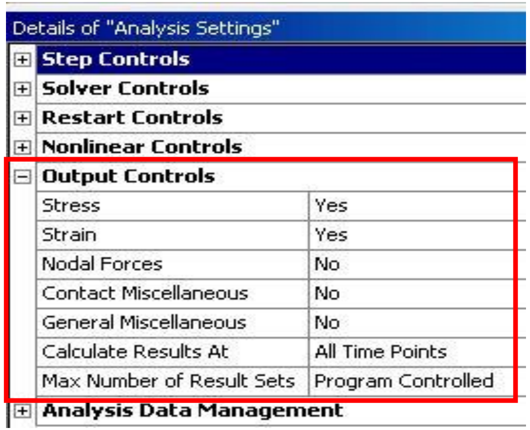
. . . Analysis Settings

The “Output Controls” section of the analysis settings configures what items are to be written to the results file (defaults shown).

Output controls are intended to allow users to write efficient results files containing only the desired information thereby limiting file sizes.

The most general results quantities are written by default.

<u>Be sure to review the documentation before starting an analysis to make sure the desired results will be written.</u>



Note: the default configuration for output controls can be changed in “Tools > Options > Analysis Settings and Solution”.

F. Loads

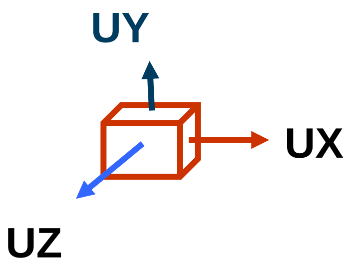
Loads and supports respond in terms of the degrees of freedom (DOF) available for the elements used.

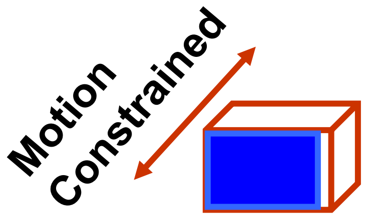
With solid geometry the DOF are X, Y and Z translations (for shells and beams we add rotational DOF rotX, rotY and rotZ).

Boundary conditions, regardless of actual names, are always defined in terms of these DOF.

Boundary conditions can be scoped to geometry items or to nodes (depending on load type).

Example: a “Frictionless Support” applied to the face of the block shown would indicate that the Z degree of freedom is no longer free (all other DOF are free).





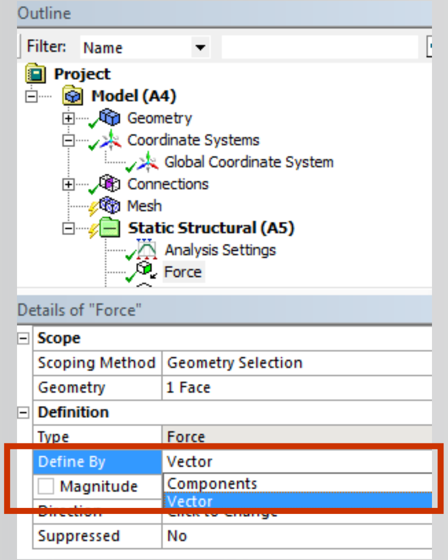
Frictionless surface

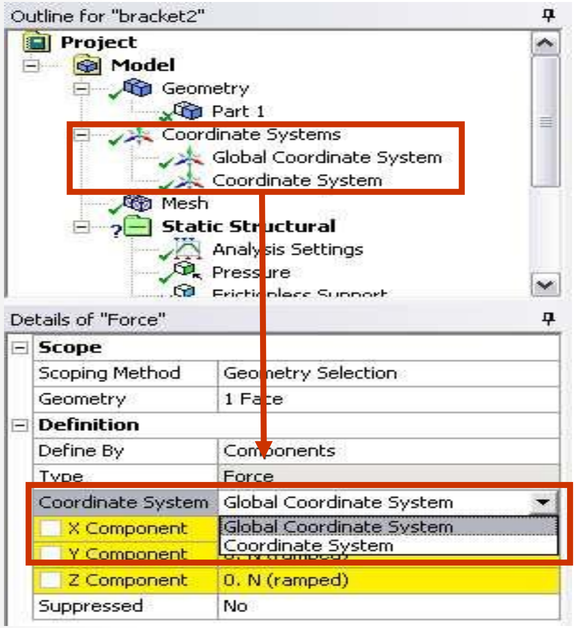
. . . Loads

Most of loads and supports can be defined by components of by vector.

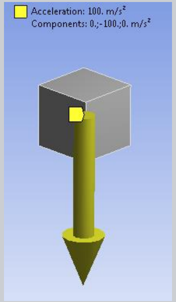
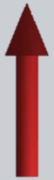
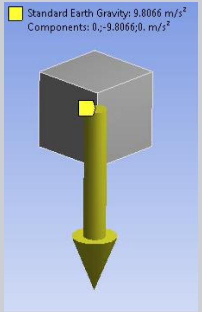
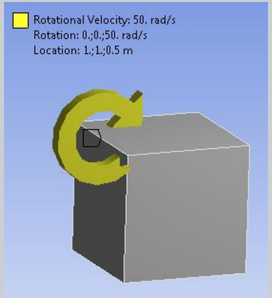
Loads and supports having a direction component can be defined in global or local coordinate systems:

- In the Details view, change “Define By” to “Components”. Then, select the appropriate CS from the pull-down menu.



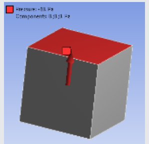
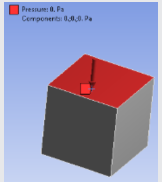
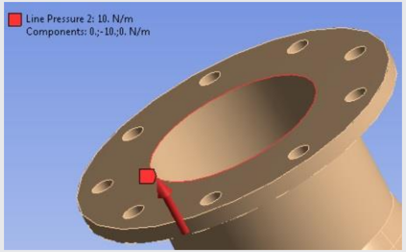


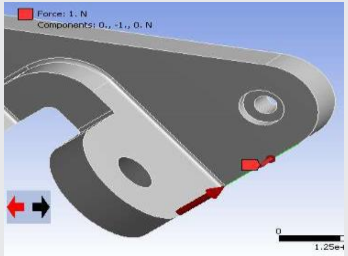
. . . Loads

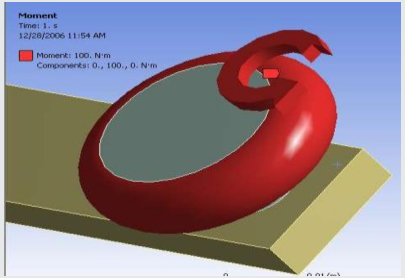
	Acceleration (all bodies)	Standard Earth Gravity (all bodies)	Rotational Velocity (selected bodies)
Units	length/time^2	length/time^2	Radian/second or RPM
Defined by	Components Vector	A Direction Local or global coordinate system	Components Vector
Notes	 Resulting forces: 	 Resulting forces: 	

. . . Loads

	Pressure	Line Pressure	Force	Moment
Scoping				
Units	Force/area	Force/length	Mass*length/time²	Force*length
Definition	Components Vector Normal to	Vector Component direction Tangential (along line)	Components Vector	Components Vector
	Positive value into surface, negative value acts out of surface.		If multiple entities are selected, the loads are evenly distributed	

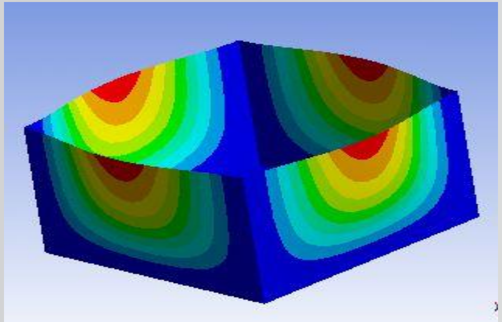




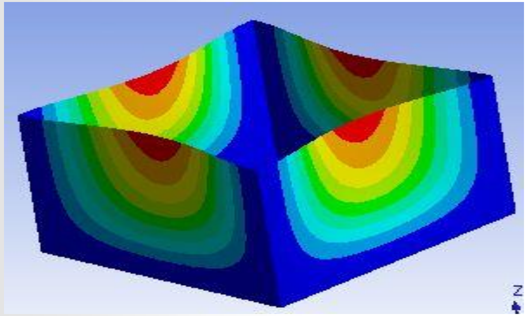
. . . Loads

Hydrostatic Pressure:

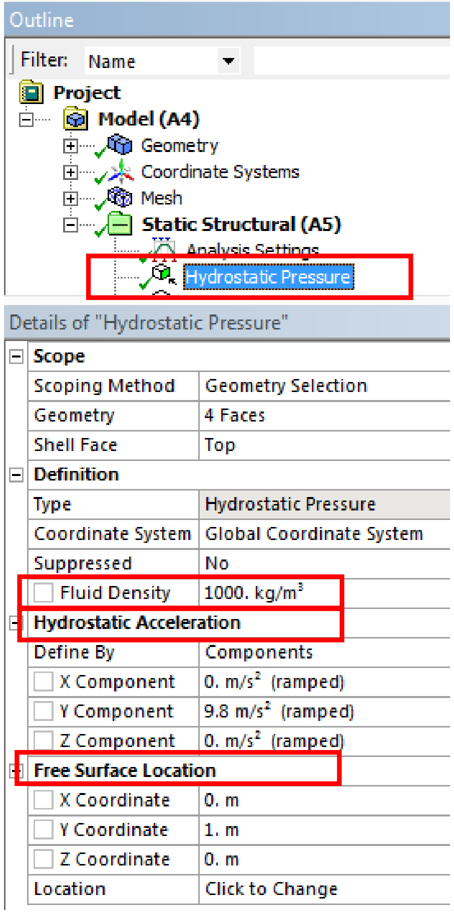
- Applies a linearly varying load to a surface (solid or shell) to simulate fluid force acting on the structure.
- Fluid may be Internal (contained fluid) or external (submerged body).



Internal



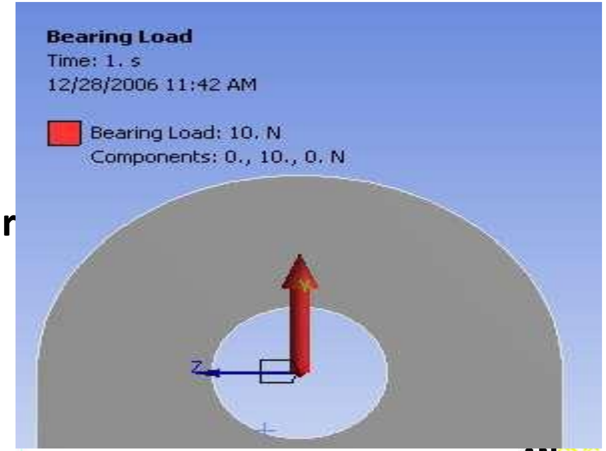
External

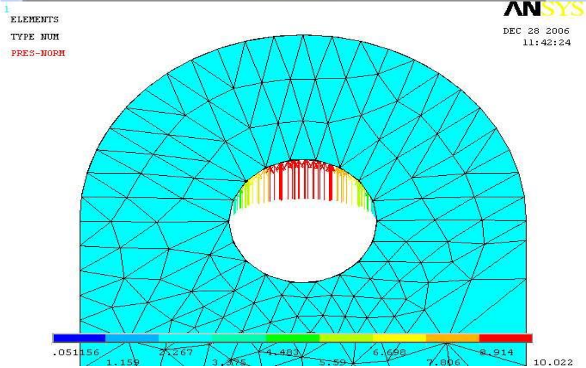


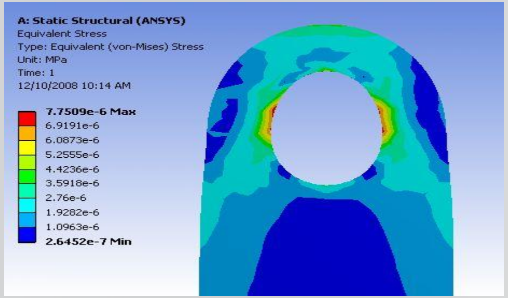
. . . Loads

Bearing Load (force):

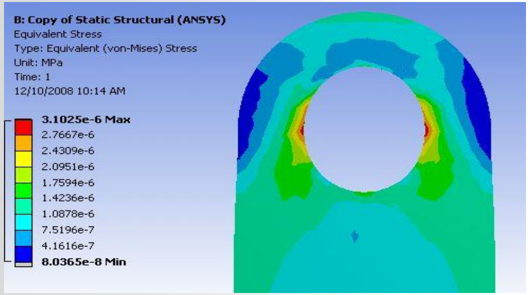
- **Forces are distributed in compression over the projected area:**
  - No axial components.
  - Use only one bearing load per cylindrical surface.
    - If the cylindrical surface is split, select both halves of cylinder when applying the load.
- **Bearing loads can be defined via vector or component method.**







Bearing Load



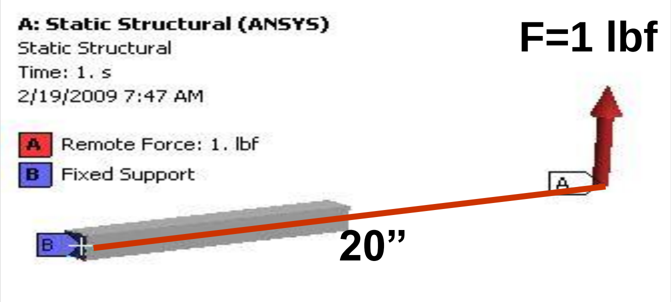
Force Load

. . . Loads

Remote Force Loading :

- The user supplies the origin of the force (geometry or coordinates). Mechanical automatically creates a remote point at the force location.
- Can be defined using vector or component method.
- Applies an equivalent force and moment on the surface.

- Example: 10 inch beam with a 1 lbf remote force scoped to the end of the beam. Remote force is located 20 inches from the fixed support.



Details of "Moment Reaction"

Options	
Results	
X Axis	20. lbf·in
Y Axis	3.488e-009 lbf·in
Z Axis	-2.1246e-007 lbf·in
Total	20. lbf·in

Moment Reaction

Details about remote points are covered in L06_Remote BC

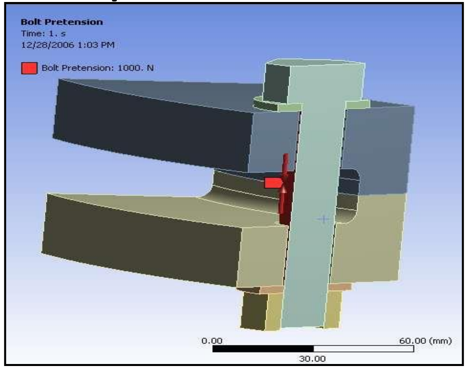
. . . Loads

Bolt Pretension:

- Applies a pretension load to a <u>solid cylindrical section</u> or <u>beam</u> using:
  - Pretension load (force)
  - OR
  - Adjustment (length)
- For body loading a local coordinate system is required (preload in z direction).
  - Face selection assumes axial direction of cylindrical surface.
- For multistep analyses additional options are available (covered later).

Details of "Bolt Pretension"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Bolt Pretension
Suppressed	No
Define By	Load
Preload	1000. N

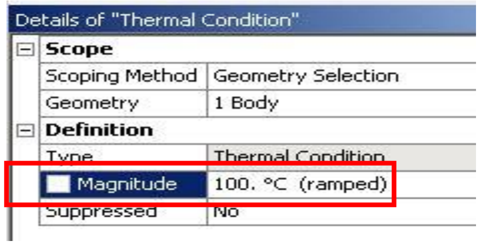


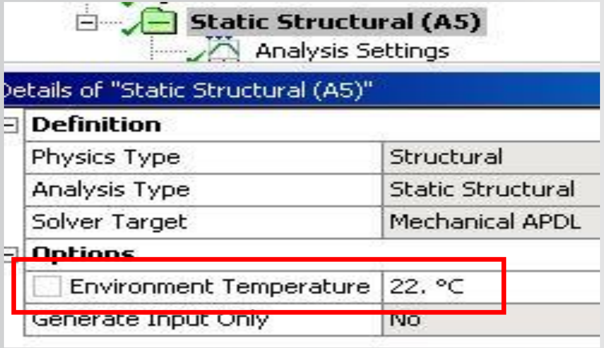
. . . Loads

Thermal Condition:

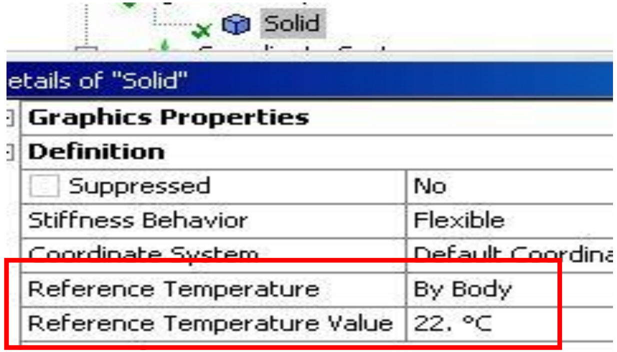
$$\varepsilon_{th}^{x} = \varepsilon_{th}^{y} = \varepsilon_{th}^{z} = \alpha\left(T - T_{ref}\right)$$

- Applies a uniform temperature in a structural analysis.
- A reference temperature must be provided (can apply to all bodies or individuals).





Reference temperature in Environment (e.g. Static Structural), applies to all bodies



Reference temperature can be applied to individual bodies

G. Supports

Fixed Support :

- Constraints all degrees of the selected entity:
  - Solid bodies: constrains x, y, and z.
  - Surface and line bodies: constrains x, y, z, rotx, roty and rotz.

Displacement:

- Allows for imposed translational displacement in x, y, and z (in user-defined Coordinate System).
- Entering “0” means that the direction is *constrained,* leaving the direction blank means the direction is free.

Details of "Displacement"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Define By	Components
Type	Displacement
X Component	0. mm (ramped)
Y Component	Free
Z Component	Free
Suppressed	No

Elastic Support :

- Applies “flexible” frictionless support to a face.
- Foundation stiffness is the pressure required to produce unit normal deflection of the foundation.

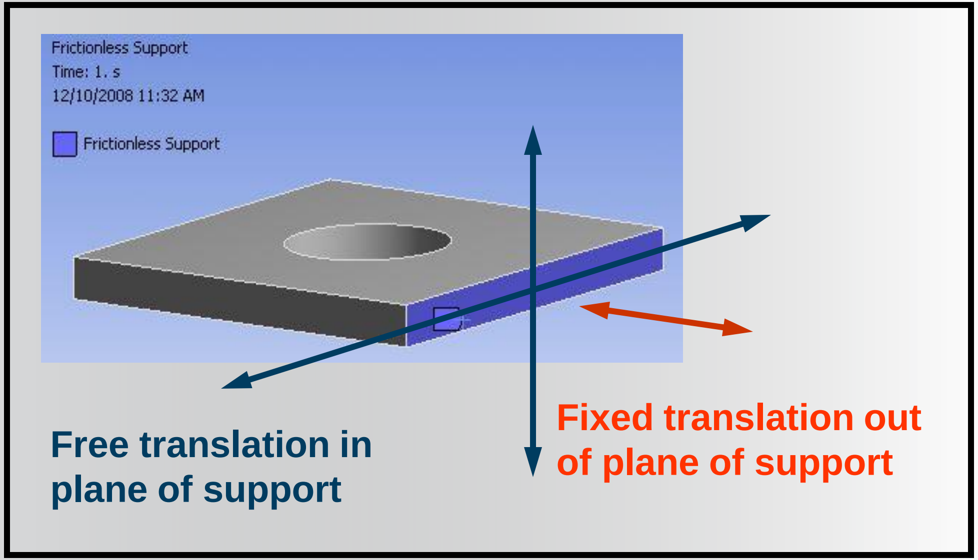
Details of "Elastic Support"

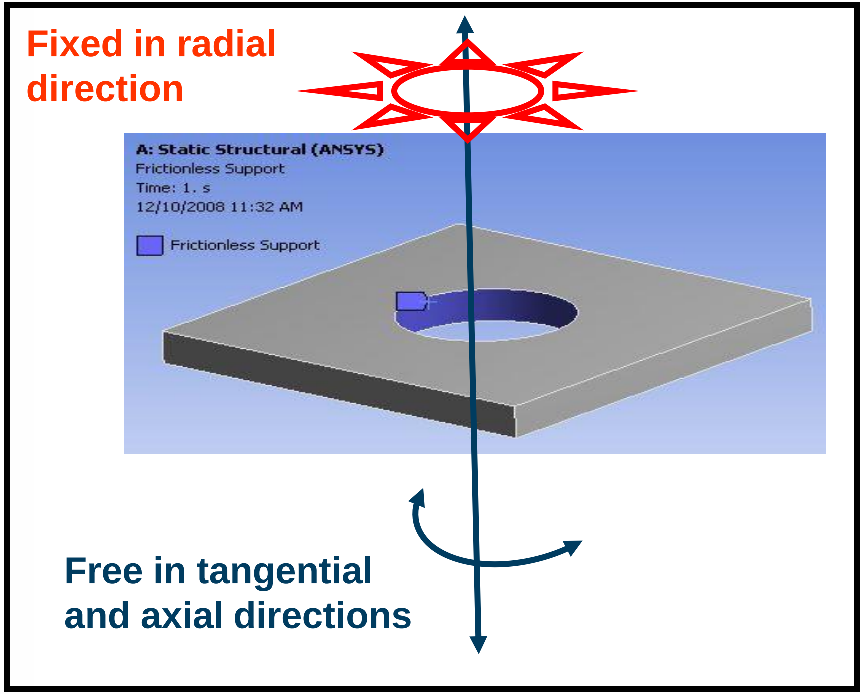
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Elastic Support
Suppressed	No
Foundation Stiffness	1. N/mm³

... Supports

Frictionless Support:

- Applies constraints (fixes) in the normal direction on surfaces.
- For solid bodies, this support can be used to apply a structural 'symmetry' boundary condition.
- Examples . . .





… Supports

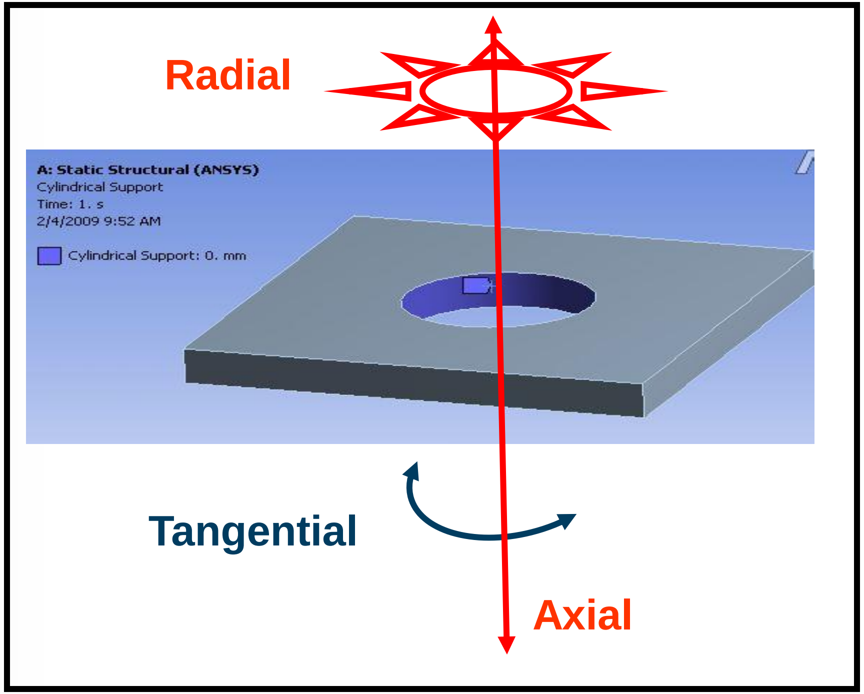
Cylindrical Support:

- **Provides individual controls (fixed/free) for axial, radial, or tangential constraints.**
- **Applied on <u>cylindrical surfaces only</u>.**

Details of "Cylindrical Support"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Cylindrical Support
Radial	Fixed
Axial	Fixed
Tangential	Free
Suppressed	No

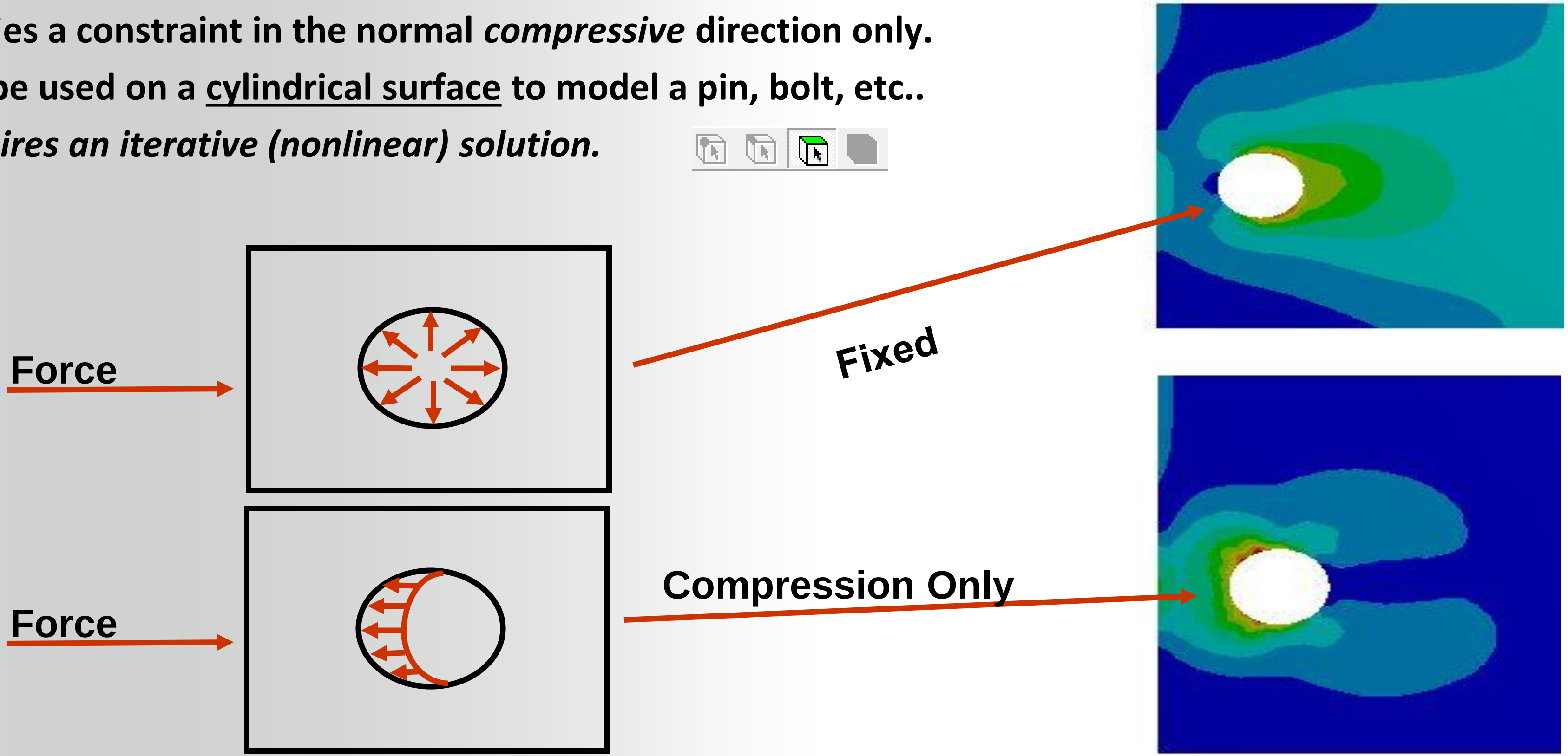
Example . . .



... Supports

Compression Only Support :

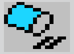
- **Applies a constraint in the normal *compressive* direction only.**
- **Can be used on a <u>cylindrical surface</u> to model a pin, bolt, etc..**
- ***Requires an iterative (nonlinear) solution.***



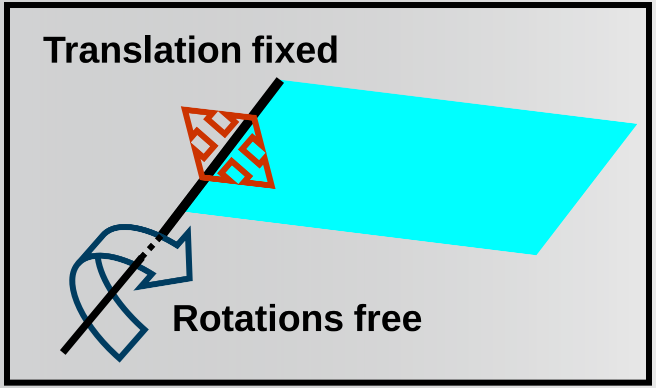
… Supports

 **Simply Supported:**

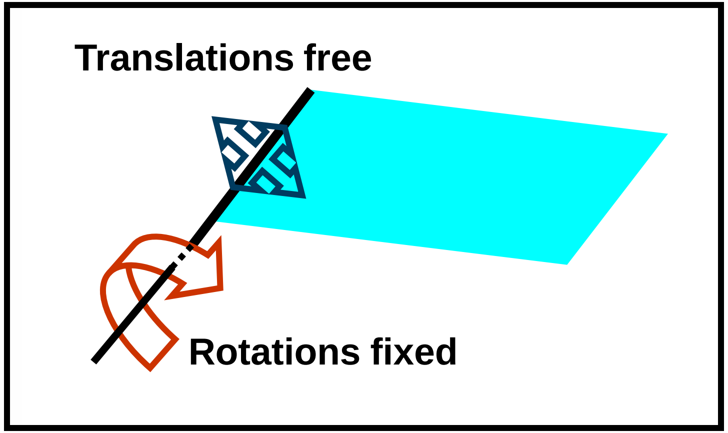
- Can be applied on edge or vertex of surface or line bodies
- Prevents all translations but all rotations are free

 **Fixed Rotation:**

- Can be applied on surface, edge, or vertex of surface or line bodies
- Constrains rotations but translations are free



Simply Supported Edge

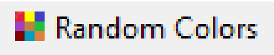


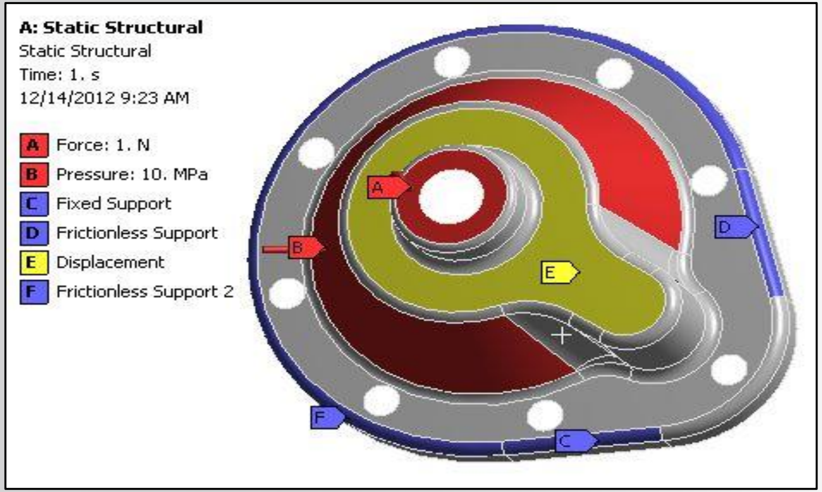
Fixed Rotation Edge

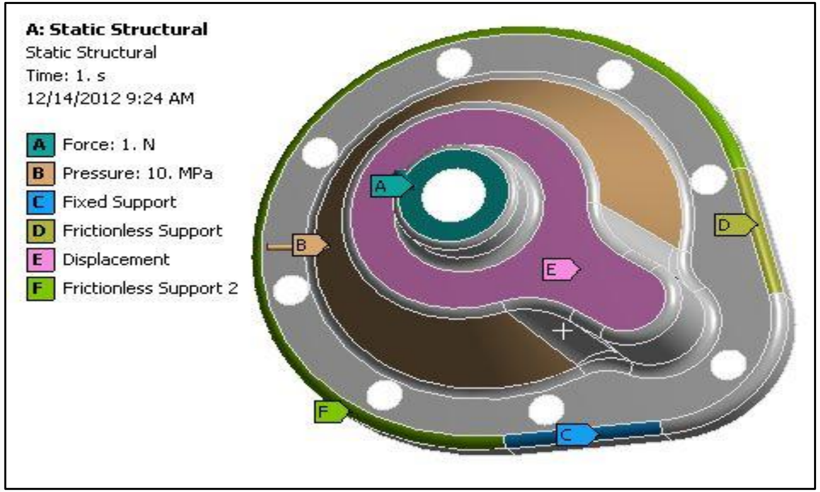
H. Load and Support Display

Two display options are available for boundary conditions:

- **By default, boundary conditions are displayed using a color scheme relating to the class of the condition such as loads, supports, displacements, etc. (figure on left).**
- **Users can toggle on "Random Colors" to assign each boundary condition a unique color (figure on right).**



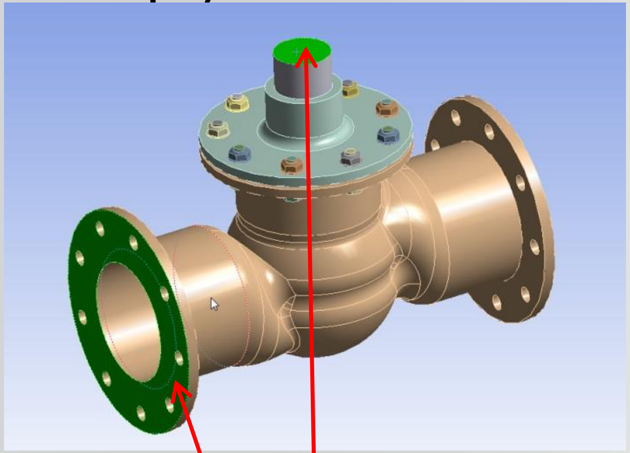


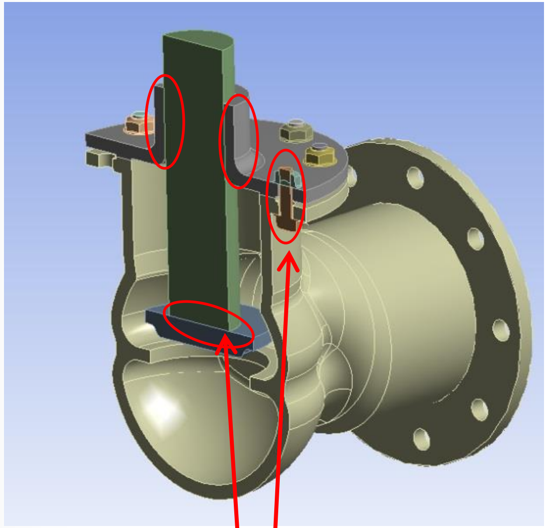


I. Supports vs Contacts

Contacts are used to define the interaction between <u>two represented bodies</u> on an assembly model.

Supports are used to define the interaction between <u>a body and the environment</u> (ground for example)





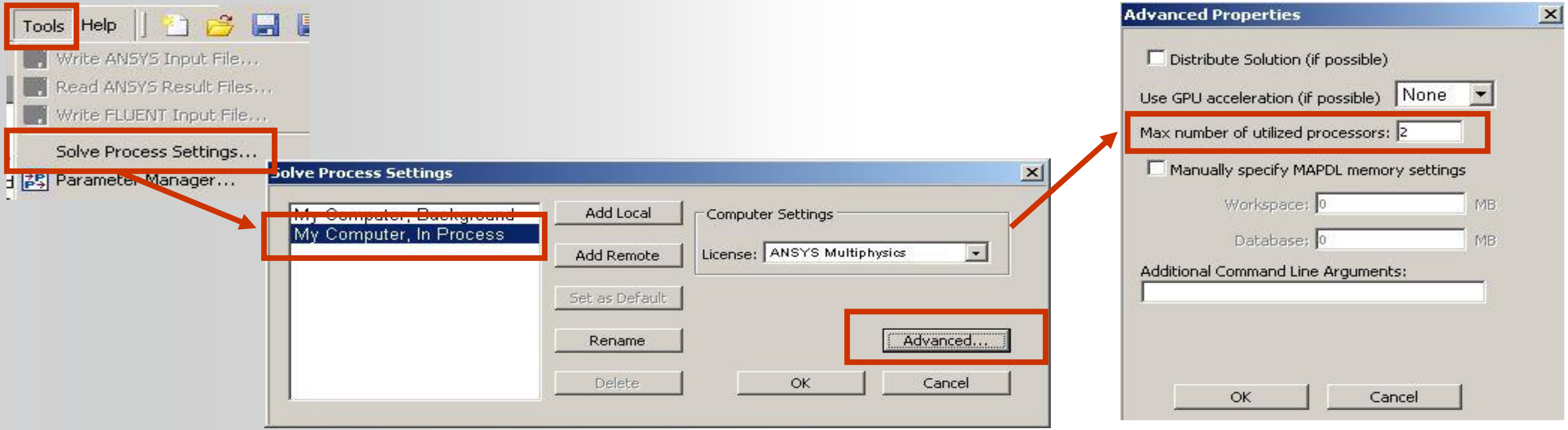
Supports will define interaction with the environment

Contact will be defined at bodies' interfaces

J. Solving the Model

To solve the model click on the “Solve” button on the Standard Toolbar.

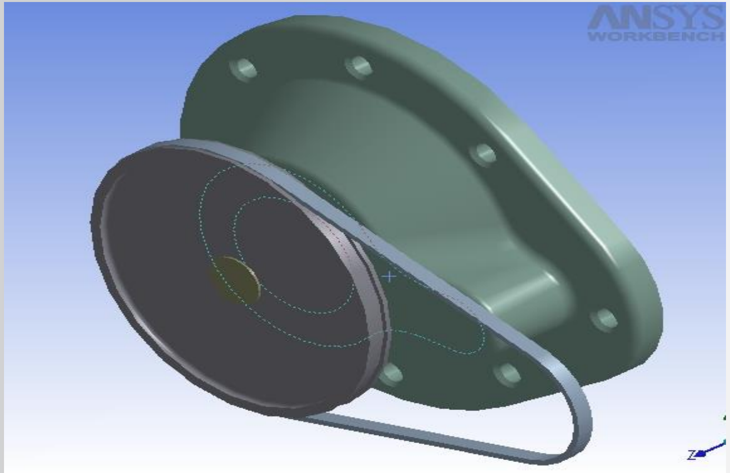
- Two processors used if present (default).
- To set the number of processors use, “Tools > Solve Process Settings”.

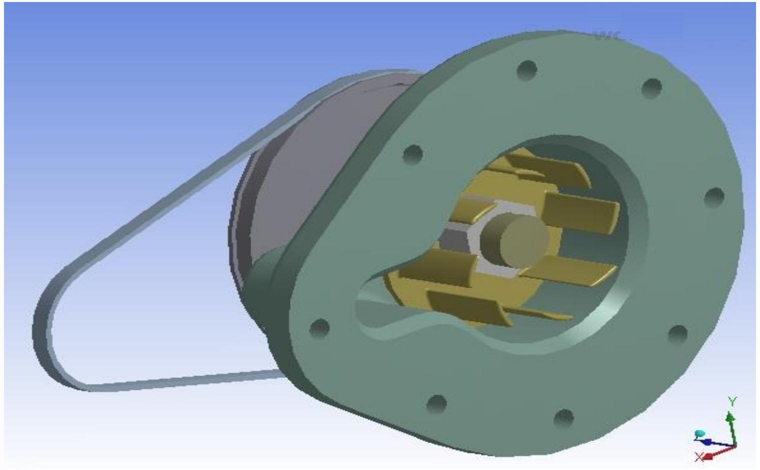


Note : By default, Mechanical use two processors. In order to run faster your models, it is possible to define more CPUs to use for a solve. This feature is available using a specific license (HPC).

K. Workshop 7.1

- **Workshop 7.1 – Structural Analysis With Contact**
- **Goal:**
  - **A five part assembly representing an impeller type pump is analyzed with a 100N load on the belt.**





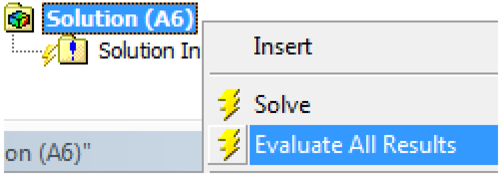
L. Results and Postprocessing

Numerous structural results are available:

- Directional and total deformation.
- Components, principal, or invariants of stresses and strains.
- Contact output.
- Reaction forces.
- More . . . .

In Mechanical, results may be requested before or after solving.

- If you solve a model then request results afterwards, click on the “Solve” button, and the results will be retrieved (the results file is re-read).
- You can also right click the Solution branch or a new result item and “Evaluate All Results”.
- A new solution is not required.

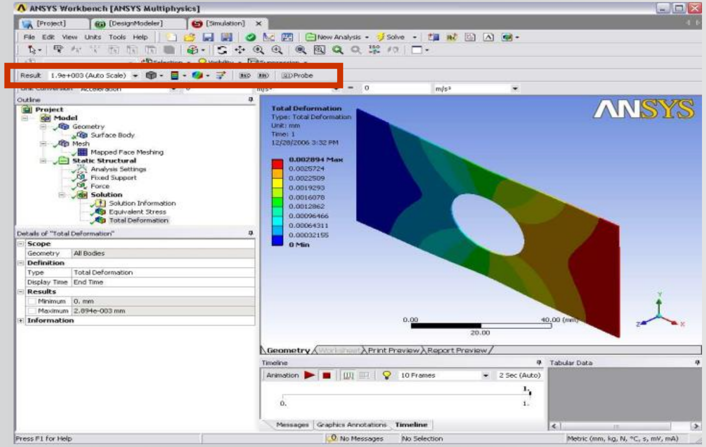


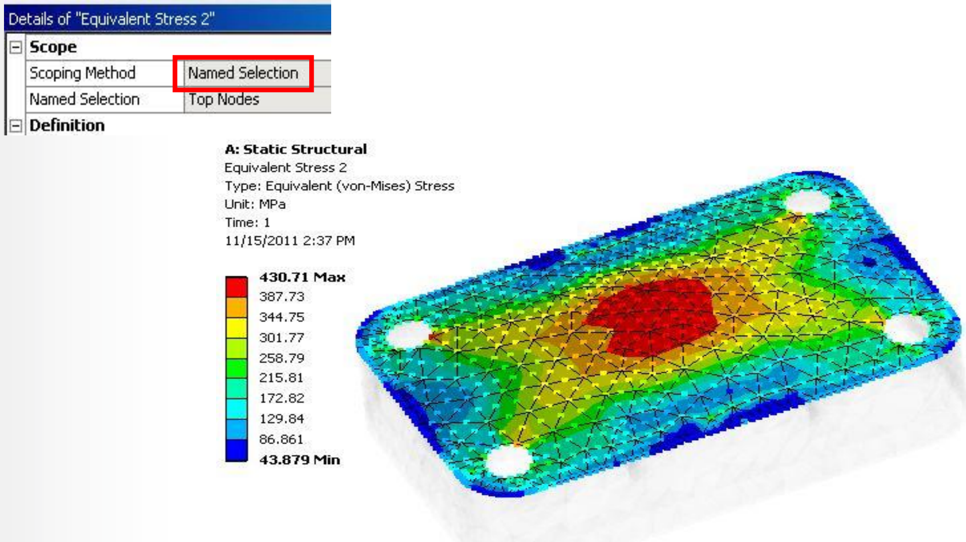
. . . Results and Postprocessing

Contour and vector plots are usually shown on the deformed geometry.

Use the Context Toolbar to change settings.

Results can be scoped to various geometry and FE entities as well as named selections. (note these controls are covered in a later chapter).



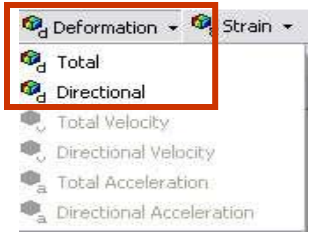


. . . Results and Postprocessing

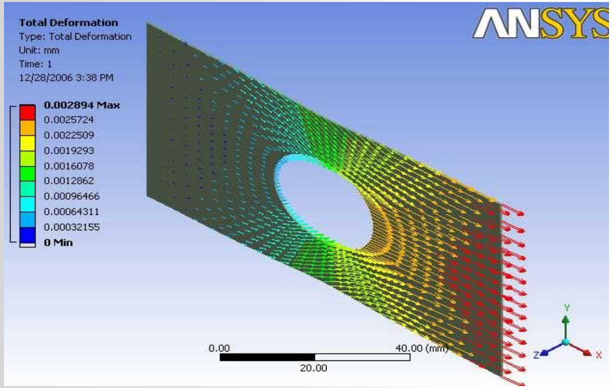
The deformation of the model can be plotted:

- **Total deformation is a scalar quantity:**

$$U_{total} = \sqrt{U_x^2 + U_y^2 + U_z^2}$$



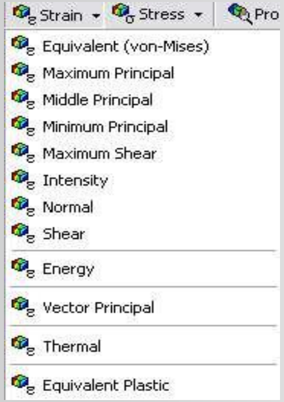
- **The x, y, and z components of deformation can be requested under "Directional", in global or local coordinates.**
- **Vector plots of deformation are available (see below).**

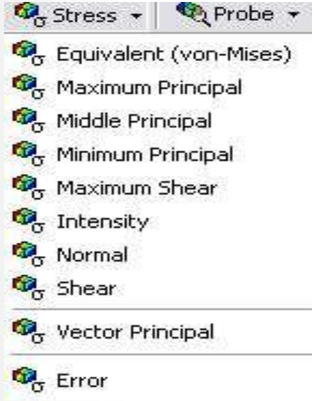


. . . Results and Postprocessing

Stresses and strains:

- Stresses and (elastic) strains have six components (x, y, z, xy, yz, xz) while thermal strains have three components (x, y, z)
- For stresses and strains, components can be requested under “Normal” (x, y, z) and “Shear” (xy, yz, xz). For thermal strains, (x, y, z) components are under “Thermal.”
- Principal stresses are always arranged such that s1 > s2 > s3
- Intensity is defined as the largest of the absolute values
  - s1 - s2, s2 - s3 or s3 - s1



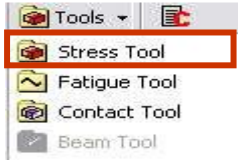


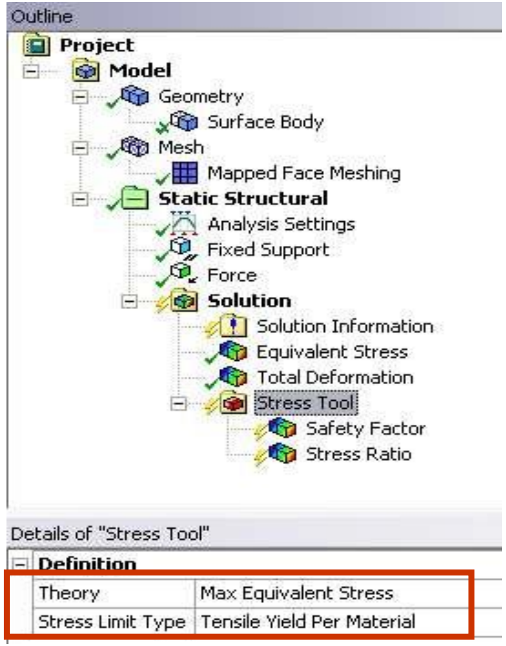
. . . Results and Postprocessing

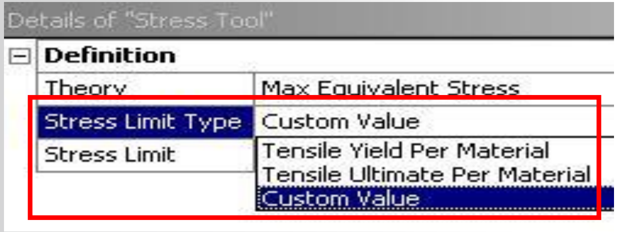
Stress Tool:

Calculates safety factors based on several material failure theories (4):

- **Ductile Theories:**
  - **Maximum Equivalent Stress**
  - **Maximum Shear Stress**
- **Brittle Theories:**
  - **Mohr-Coulomb Stress**
  - **Maximum Tensile Stress**
- **Safety factor, safety margin and stress ratio can be plotted.**
- **User specified failure criteria can be entered.**







ANSYS

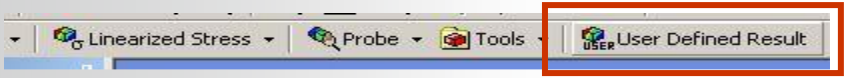
. . . Results and Postprocessing

In addition to the standard result items one can insert "user defined" results.

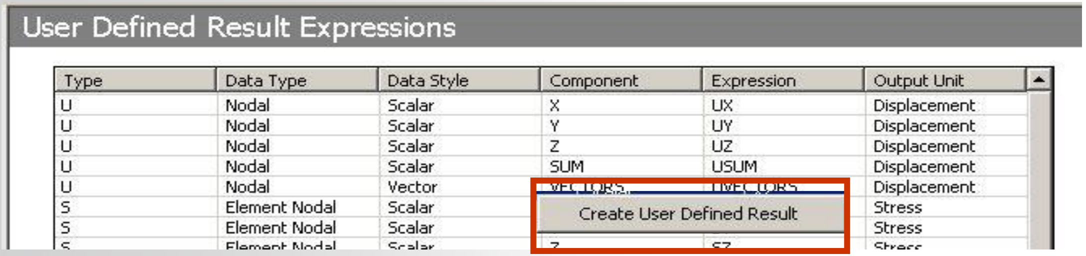
These results can include mathematical expressions and can be combinations of multiple result items.

Define in 2 ways:

- **Select "User Defined Result" from the solution context menu**



- **OR - From the Solution Worksheet highlight result > RMB > Create User Defined Result.**

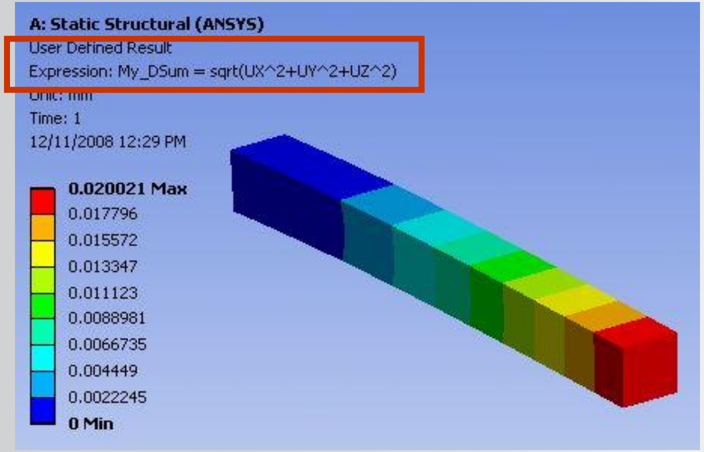


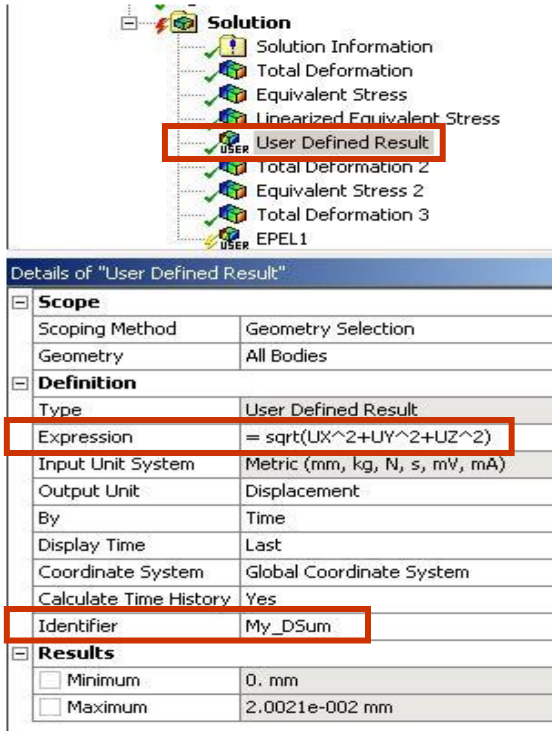
. . . Results and Postprocessing

Details allow an expression using various basic math operations as well as square root, absolute value, exponent, etc..

User defined results can be labeled with a user “Identifier”.

Result legend contains identifier and expression.

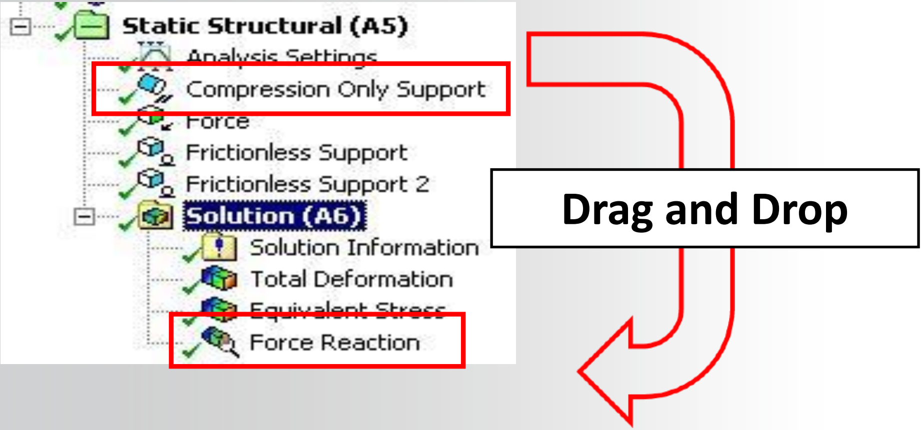




. . . Results and Postprocessing

Reaction forces at constraints or contacts can be obtained using a “Reaction Probe”.

Probes can be inserted manually (like other results) or constraints/contacts can be drag and dropped onto the Solution branch as a shortcut.



Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Compression Only Support
Orientation	Global Coordinate System
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	-100. N
Y Axis	-7.7846 N
Z Axis	-1.6597e-015 N
Total	100.3 N
Maximum Value Over Time	
X Axis	-100. N
Y Axis	-7.7846 N
Z Axis	-1.6597e-015 N
Total	100.3 N
Minimum Value Over Time	
X Axis	-100. N
Y Axis	-7.7846 N
Z Axis	-1.6597e-015 N
Total	100.3 N

. . . Results and Postprocessing

After a static structural solve, you have to check results. A check consists of verify the static equilibrium:

$$F_{applied} = F_{reaction}$$

Reaction probes have to be used to calculate static equilibrium.

Simple example : $F_{applied}$: Acceleration : Fy=m*a => Fy=6.08 N

TYPE	MASS
1	0.619953E-03

Details of "Acceleration"

Scope	
Geometry	All Bodies
Definition	
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. mm/s² (ramped)
Y Component	9810. mm/s² (ramped)
Z Component	0. mm/s² (ramped)
Suppressed	No

$F_{reaction}$:

- Static Structural (B5)
  - Analysis Settings
  - Acceleration
  - Fixed Support
  - Solution (B6)
    - Solution Information
    - Force Reaction

Details of "Force Reaction"

Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Fixed Support
Orientation	Global Coordinate Syst
Suppressed	No
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	2.9915e-014 N
Y Axis	6.0817 N
Z Axis	1.1562e-013 N
Total	6.0817 N

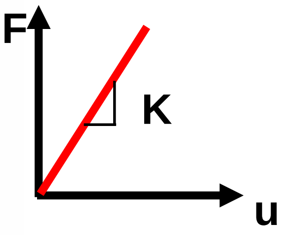
Drag and Drop

M. Linear vs Non Linear solve

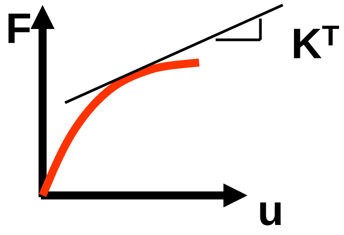
In a linear analysis, the matrix equation [K]{x}={F} is solved in one iteration. That means the model stiffness does not change during solve : [K] is constant.

A non linear solve allow stiffness changes and uses an iterative process to solve the problem.
In a static structural analysis, ANSYS runs a non linear solve automatically when the model contains :

- **Non linear material laws : Plasticity, Creep, Gasket, Viscoelasticity …**
- **Non linear contact : Frictionless, Rough, Frictional**
- **Large deflection turned « ON»**
- **Compression only support**
- **Joints**
- **Bolt pretension**
- **Compression only or tension only Spring**



LINEAR



NON LINEAR

N. Workshop 7.2

- **Workshop 7.2 – Using Beam Connections**
  - **In the flange model shown we will use Mechanical’s beam connection feature to simulate bolted fasteners in the model.**

O. APPENDIX

- Analysis Settings : Restart controls
- Nodal Load and Supports
- Results and Postprocessing

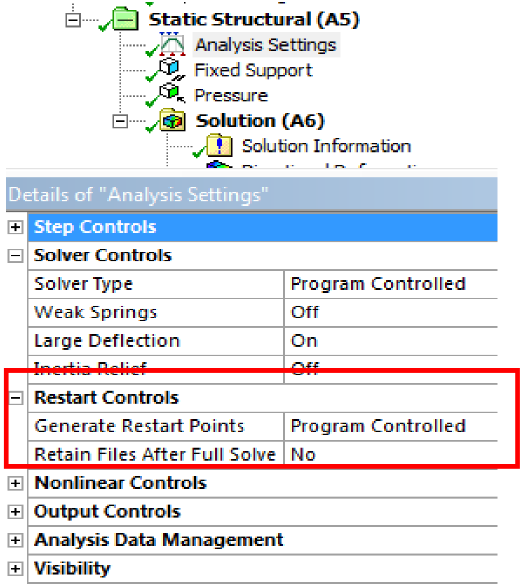
Analysis Settings : Restart controls

The “Restart Controls” section of the analysis settings configures how ANSYS produces restart points throughout a solve.

This is very handy for nonlinear analysis as a converged solution is not guaranteed, and the user has to occasionally adjust the solver setting from a particular intermediate stage of the solve.

For linear static analysis, by default no restart points are created.

This is covered in more detail in the Introduction to Structural Nonlinearity course.



Nodal Loads and Supports

Certain loads and supports can be applied directly to the nodes.

Nodal Loads:

- **Must be applied to nodal named selections.**
- **Load directions depend on the orientation of the node.**

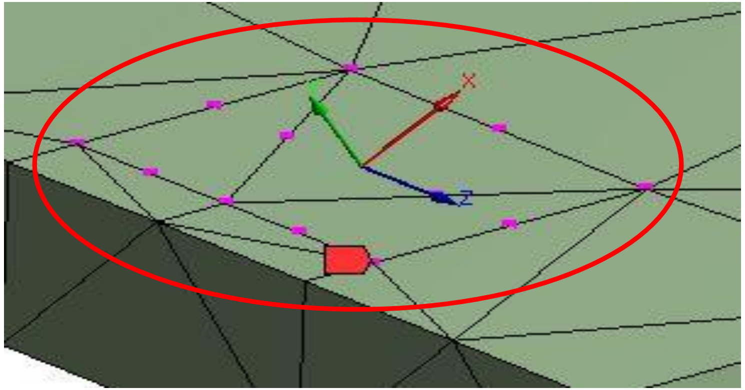
Note : for example it could be useful in case of restart when a load must be added after a full solve

Details of "Nodal Force"

Scope	
Scoping Method	Named Selection
Named Selection	Load Nodes
Definition	
Type	Force
Coordinate System	Nodal Coordinate System
X Component	0. N (ramped)
Y Component	10. N (ramped)
Z Component	0. N (ramped)
Divide Load by Nodes	Yes
Suppressed	No

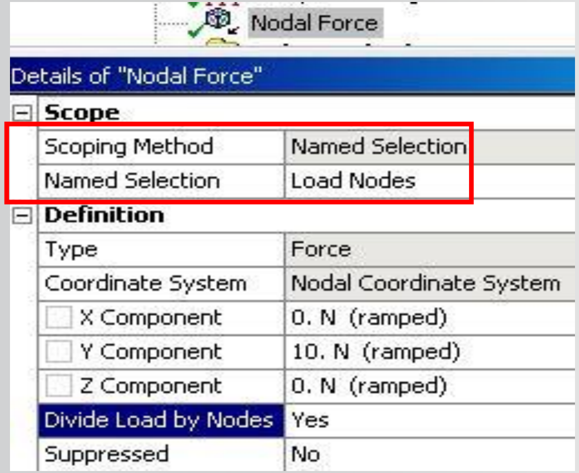
Each node has an associated local "nodal" coordinate system. By default these systems are aligned with the global Cartesian system but can be reoriented into a local system for loading purposes.

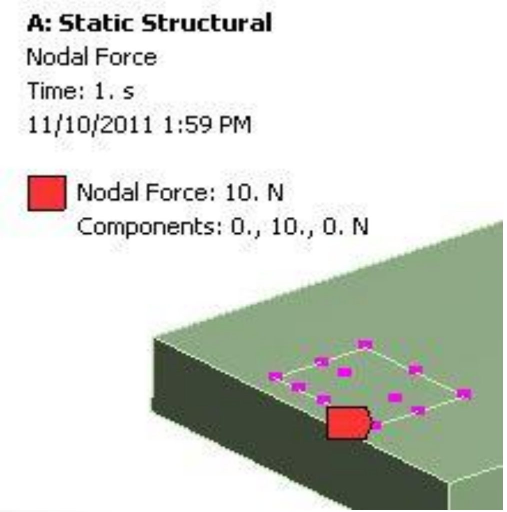
<u>**It is with respect to these nodal coordinate systems that the X, Y, Z components of FE loads are defined.**</u>



. . . Nodal Loads and Supports

To apply a nodal load, after choosing the type of load from the "Direct FE" menu the load details allow choice of named selection and magnitude.



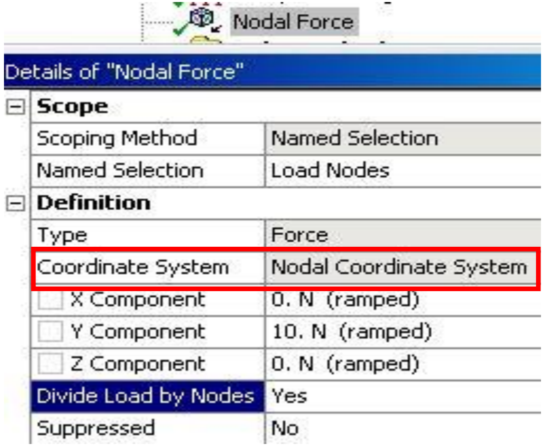


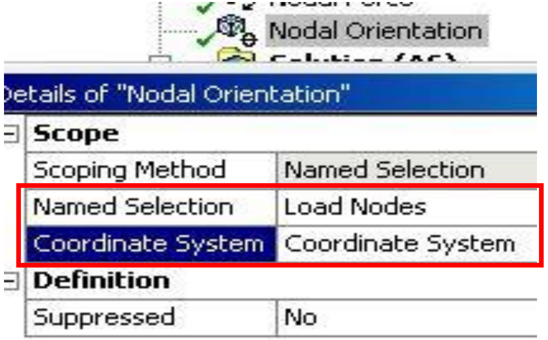
. . . Nodal Loads and Supports

To change a nodal load direction the node's orientation must be changed using a local coordinate system.

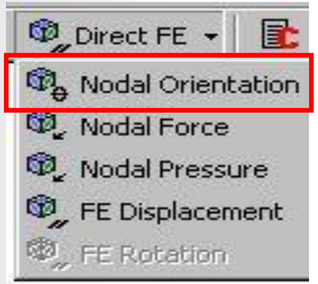
Choose "Nodal Orientation" from the Direct FE menu:

- Pick the nodal named selection in the details from a drop down list.
- Pick the coordinate system to reference from a drop down list.





Note: a "Nodal Orientation" branch will be placed in the tree's environment branch.



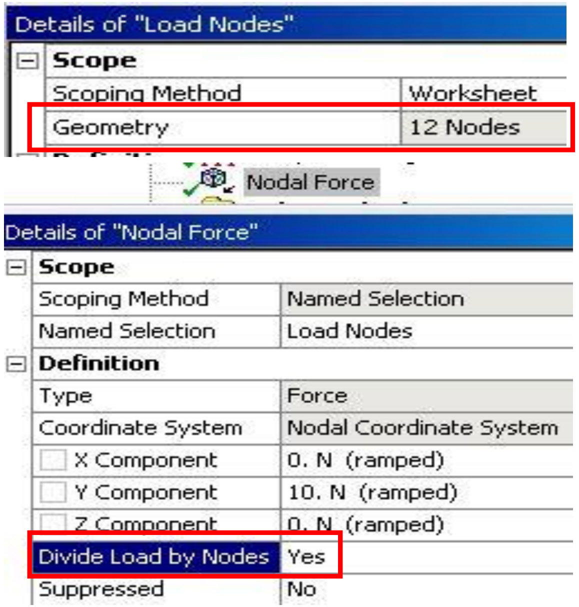
. . . Nodal Loads and Supports

Notes on nodal boundary conditions:

- FE Rotation applies only when surface or line geometry is present (rotational DOF).

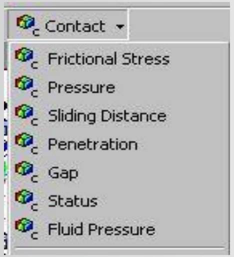
Since Direct FE loads are often applied to multiple nodes there is a control for how the load is distributed.

- **Divide Load by Nodes (default):**
  - **Yes: divides the magnitude by the number of nodes and applies equal loads to each (F/Num to each node).**
  - **No: applies the full load magnitude to each node (F applied to each node).**
- **Using the example at right choosing "No" results in a total load of (10 N * 12 Nodes = <u>120 N</u>).**



Results and Postprocessing

Contact results are requested via a "Contact Tool" under the Solution branch.



Contact regions can be selected in the graphics window or using a Worksheet.