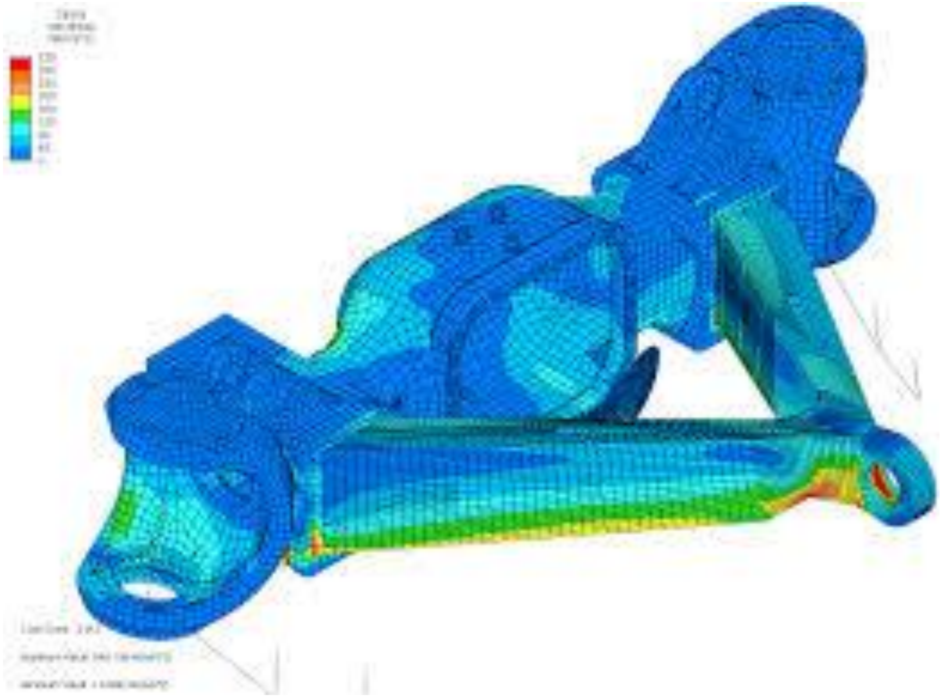
Courtesy Ansys Inc

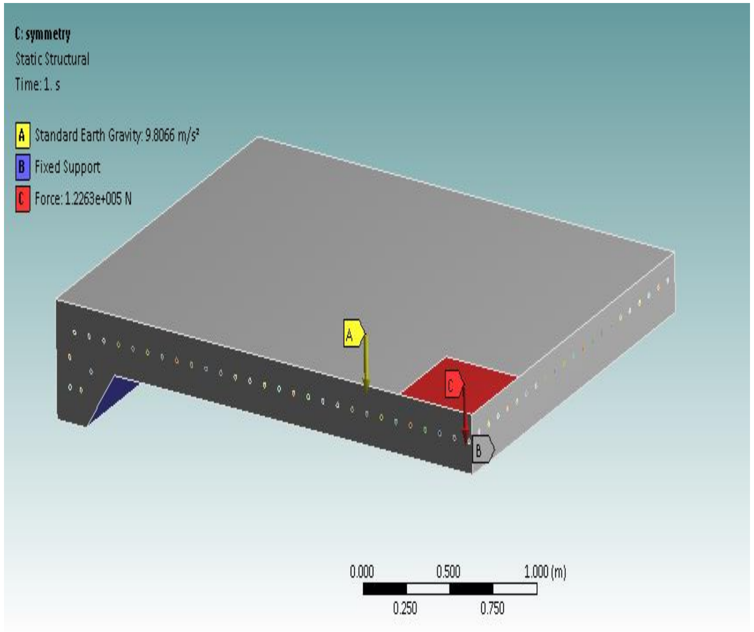
Introduction to Structural Analysis-Ansys WB



© Technology Training & Research Centre

Overview

- In this chapter, performing linear static structural analyses in Simulation will be covered:
  A. Geometry and Elements
  B. Assemblies and Contact Types
  C. Analysis Settings
  D. Environment, including Loads and Supports
  E. Solving Models
  F. Results and Post processing



- The capabilities described in this section are generally applicable to *ANSYS Design Space Entra* licenses and above.
  - Some options discussed in this chapter may require more advanced licenses, but these are noted accordingly.

 © Technology Training & Research Centre

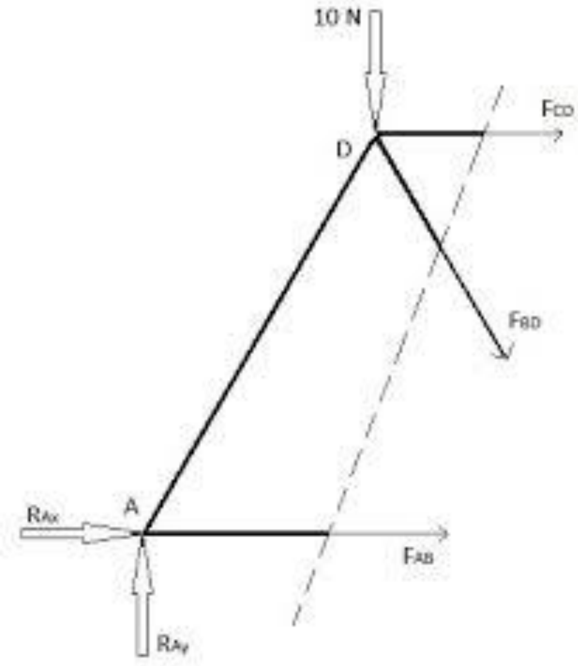
Courtesy Ansys Inc

Basics of Linear Static Analysis

- For a linear static structural analysis, the displacements {x} are solved for in the matrix equation below:

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

  Assumptions:
  - [K] is constant
    - Linear elastic material behavior is assumed
    - Small deflection theory is used
    - Some nonlinear boundary conditions may be included
  - {F} is statically applied
    - No time-varying forces are considered
    - No inertial effects (mass, damping) are included



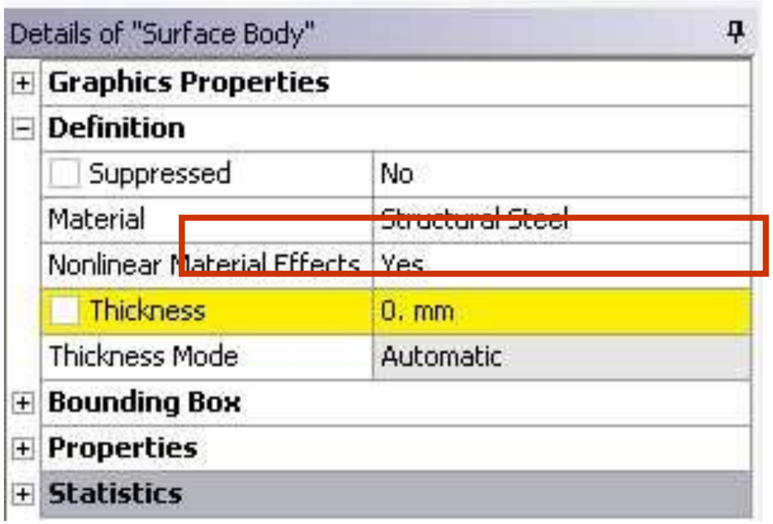
- It is important to remember these assumptions related to *linear static* analysis. *Nonlinear static* and *dynamic* analyses are covered in later chapters.

© Technology Training & Research Centre

A. Geometry

- In structural analyses, all types of bodies supported by Simulation 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 Design Modeler and are imported into Simulation automatically.

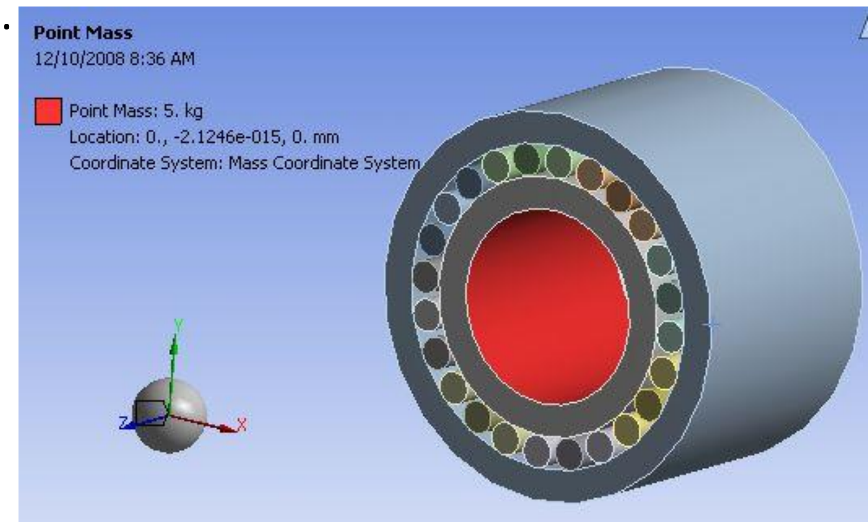
© Technology Training & Research Centre

… Point Mass

- A Point Mass can be added to a model (Geometry branch) to simulate parts of the structure not explicitly modeled:
  - A point mass is associated with surface(s) only.
  - The location can be defined by either:
    - (x, y, z) coordinates in any user-defined Coordinate System.
    - Selecting vertices/edges/surfaces to define location.
  - Point mass is affected by "Acceleration," "Standard Earth Gravity," and "Rotational Velocity". No other loads affect a point mass.
  - The mass is 'connected' to selected surfaces assuming *no stiffness* between them.
  - No rotational inertial terms are present.

Details of "Point Mass"

Scope	
Geometry	1 Edge
X Coordinate	26.13 mm
Y Coordinate	47.27 mm
Z Coordinate	0. mm
Location	Click to Change
Definition	
Mass	0. kg
Mass Moment of Inertia X	0. kg·mm²
Mass Moment of Inertia Y	0. kg·mm²
Mass Moment of Inertia Z	0. kg·mm²
Suppressed	No
Behavior	Deformable



© Technology Training & Research Centre

Courtesy Ansys Inc

… Material Properties

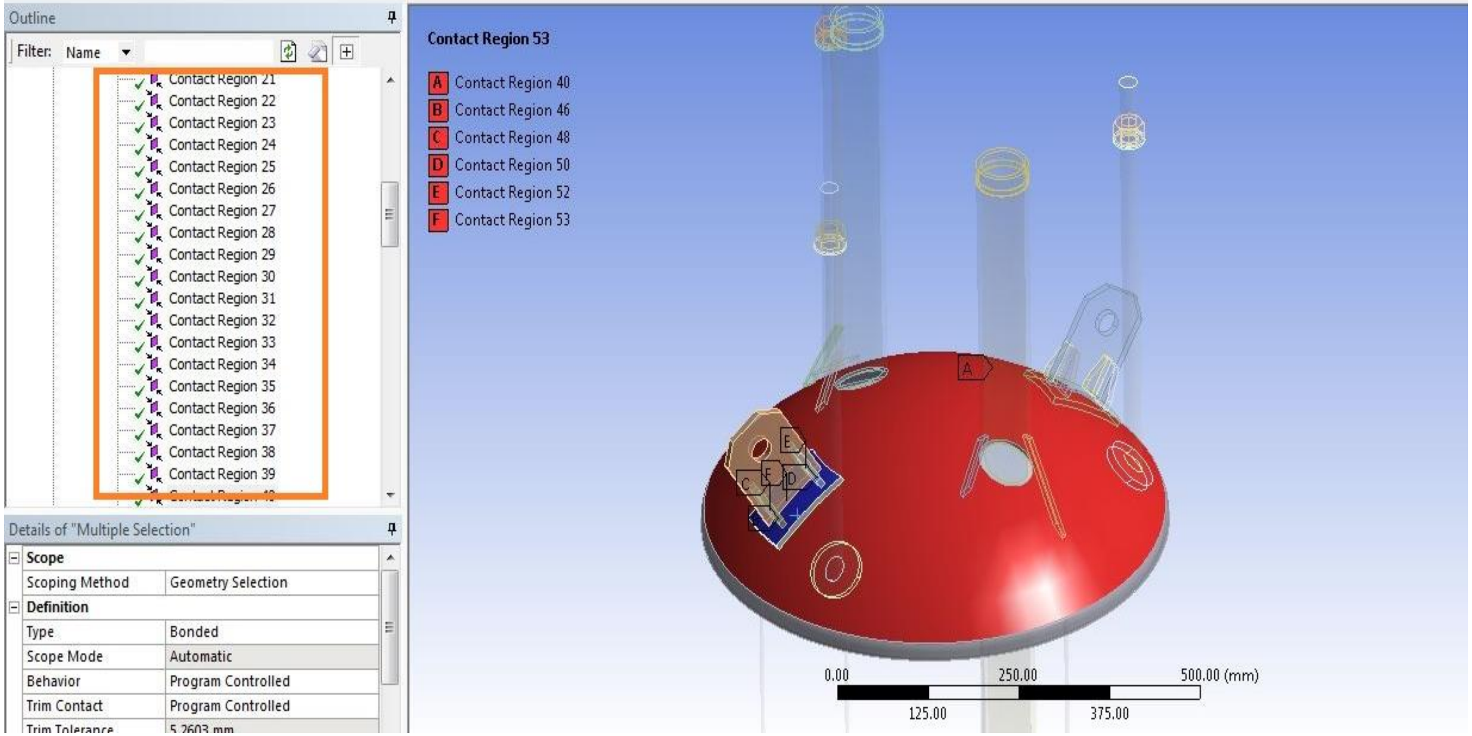
- *Young's Modulus* and *Poisson's Ratio* are required for linear static structural analyses:
  - Material input is handled in the "Engineering Data" application.
  - *Mass density* is required if any inertial loads are present.
  - *Thermal expansion coefficient* is required if a uniform temperature load is applied.
  - Thermal conductivity is NOT required for uniform temperature conditions.
  - *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.

© Technology Training & Research Centre

Courtesy Ansys Inc

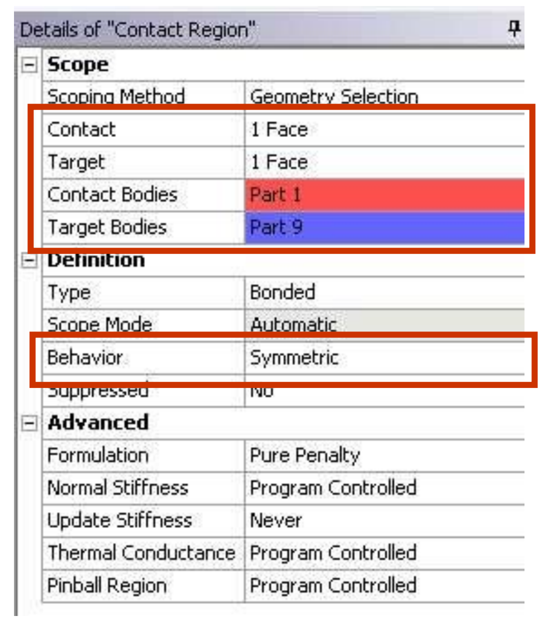
B. Assemblies – Solid Body Contact

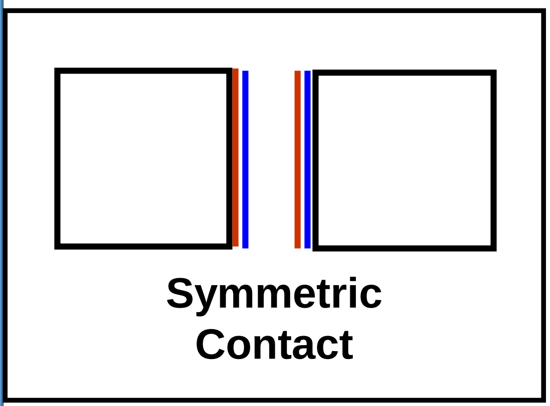
- When importing assemblies of solid parts, contact regions are automatically created between the solid bodies.
  - Contact allows non-matching meshes at boundaries between solid parts
  - Tolerance controls under "Contact" branch allows the user to specify distance of auto contact detection via slider bar

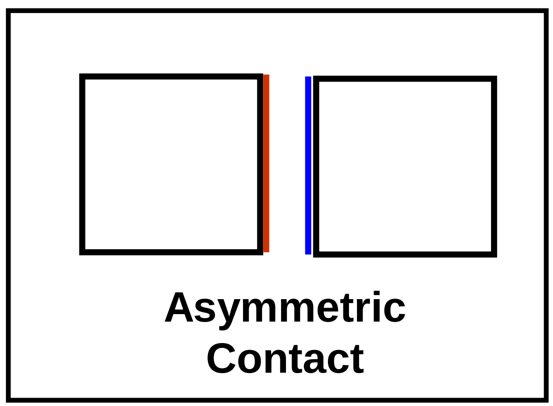


… Assemblies – Solid Body Contact

- In Simulation, the concept of *contact* and *target* surfaces are used for each contact region:
  - One side of a contact region is referred to as a contact surface, the other side is referred to as a target surface.
  - The contact surfaces are restricted from penetrating through the target surface.
    - When one side is designated the contact and the other side the target, this is called *asymmetric contact*.
    - If both sides are made to be contact & target this is called *symmetric contact.*
    - By default, Simulation uses *symmetric contact* for solid assemblies.
    - For *ANSYS Professional* licenses and above, the user may change to asymmetric contact, as desired.







© Technology Training & Research Centre

… Assemblies – Solid Body Contact

- Five contact types are available:

Contact Type	Iterations	Normal Behavior (Separation)	Tangential Behavior (Sliding)
Bonded	1	No Gaps	No Sliding
No Separation	1	No Gaps	Sliding Allowed
Frictionless	Multiple	Gaps Allowed	Sliding Allowed
Rough	Multiple	Gaps Allowed	No Sliding
Frictional	Multiple	Gaps Allowed	Sliding Allowed

  - *Bonded* and *No Separation* contact are linear and require only 1 iteration.
  - *Frictionless, Rough and Frictional* contact are nonlinear and require multiple iterations.

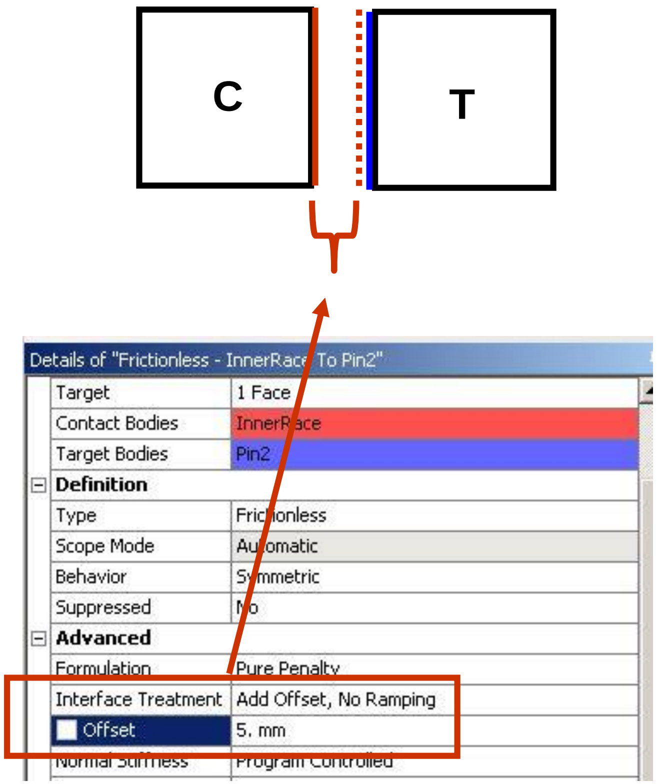
- Nonlinear contact types allow an "interface treatment" option:
  - "Add Offset": input zero or non-zero value for initial adjustment
  - "Adjusted to Touch": ANSYS closes any gap to a just touching position (*ANSYS Professional* and above)

© Technology Training & Research Centre

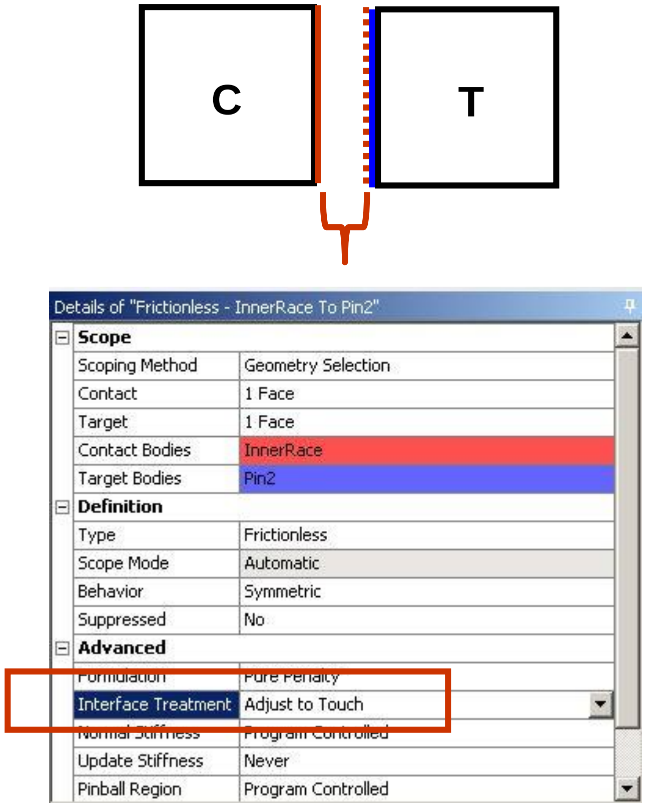
Courtesy Ansys Inc

… Assemblies – Solid Body Contact

- Interface treatment options:



Add offset: contact surface is numerically offset a given amount in positive or negative direction (offset can be ramped on).



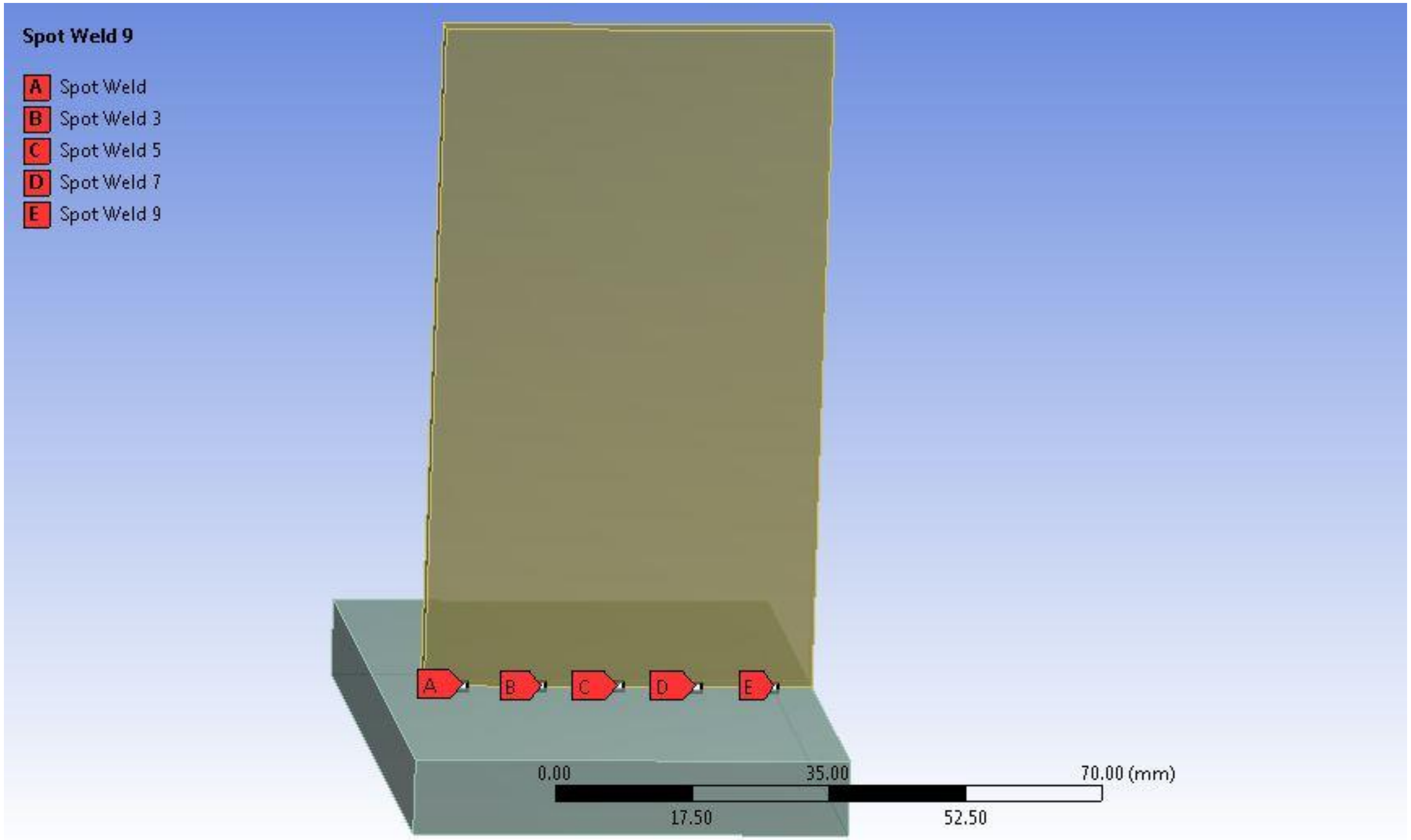
Adjusted to touch: offsets contact surface to provide initial contact with target regardless of actual gap/penetration.

© Technology Training & Research Centre

Courtesy Ansys Inc

… Assemblies – Spot Weld

- Spot welds provide a means of connecting shell assemblies at discrete points:
  - Spot-weld definition is done in the CAD software. Currently, only Design Modeler and Unigraphics define supported spot weld definitions.



© Technology Training & Research Centre

… Assemblies – Contact Summary

- Summary of contact types and options available in Simulation:

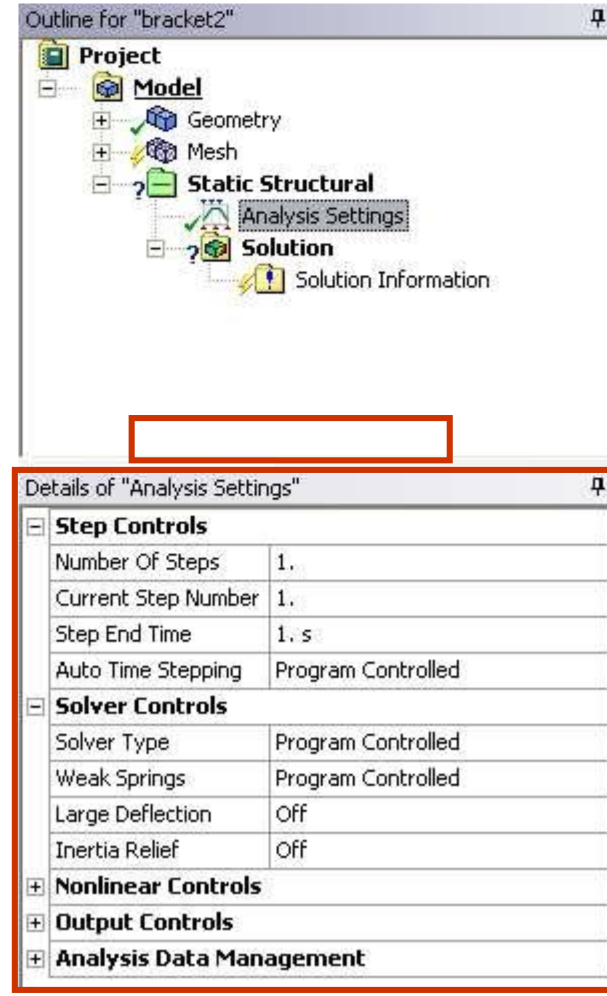
Contact Geometry	Solid Body Face (Scope = Contact)	Solid Body Edge (Scope = Contact)	Surface Body Face (Scope = Contact)	Surface Body Edge (Scope = Contact)
Solid Body Face	All types	Bonded, No Separation	Bonded, No Separation	Bonded only
	All formulations	All formulations	All formulations	MPC formulation
(Scope = Target)	Symmetry respected	Asymmetric only	Symmetry respected	Asymmetric only
Solid Body Edge	Not supported for solving[1]	Bonded, No Separation	Not supported for solving[1]	Bonded only
		All formulations		MPC formulation
(Scope = Target)		Asymmetric only		Asymmetric only
Surface Body Face	Bonded, No Separation	Bonded, No Separation	Bonded, No Separation	Bonded only
(Scope = Target)	All formulations	All formulations	All formulations	Augmented Lagrange, Pure Penalty, and MPC formulation
	Symmetry respected	Asymmetric only	Symmetry respected	Asymmetric only
Surface Body Edge	Not supported for solving[1]	Bonded only	Not supported for solving[1]	Bonded only
(Scope = Target)		MPC formulation		Augmented Lagrange, Pure Penalty, and MPC formulation
		Asymmetric only		Asymmetric only

1 – For Face/Edge contact, faces must always be designated as targets and edges must always be designated as contacts

© Technology Training & Research Centre

C. Analysis Settings

- The "Analysis Settings" details provide general control over the solution process:
- Step Controls:
  - Manual and auto time stepping controls.
  - Specify the number of steps in an analysis and an end "time" for each step.
  - "Time" is a tracking mechanism in static analyses (discussed later).
- Solver Controls: Two solvers available (default program chosen):
  - Direct solver (Sparse solver in ANSYS).
  - Iterative solver (PCG solver in ANSYS).
  - Weak springs: Simulation tries to anticipate under-constrained models.



© Technology Training & Research Centre

Courtesy Ansys Inc

. . . Analysis Settings – Analysis Data Management

- Analysis Data Management:
  - Solver Files Directory shows location where associated analysis files will be saved.
  - Future Analysis: indicates whether a down stream analysis (e.g. pre-stressed modal) will use the solution. This is set automatically when coupled analyses are configured in the project schematic.
  - Scratch Solver Files Directory: temporary directory used during solution.
  - Save ANSYS db.
  - Delete Unneeded Files: may choose to save all files for future use in Mechanical APDL.
  - Solver Units: Active System or manual.
  - Solver Unit System: if the above setting is "manual", you may choose 1 of 8 possible solver unit systems to insure consistency when data is shared with Mechanical APDL (does not affect results/load displays in the GUI).

Details of "Analysis Settings"

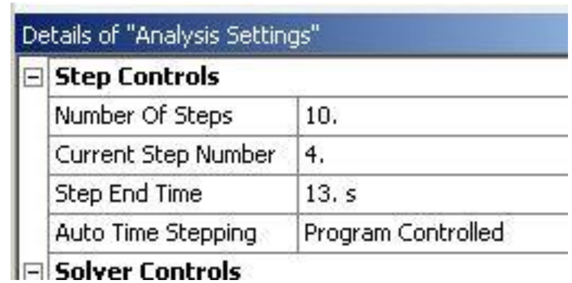
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Nonlinear Controls	
Output Controls	
Analysis Data Management	
Solver Files Directory	D:\Solver_Temp\WB_PGHX6
Future Analysis	Prestressed analysis
Scratch Solver Files Directory	
Save ANSYS db	Yes
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	nmm

© Technology Training & Research Centre

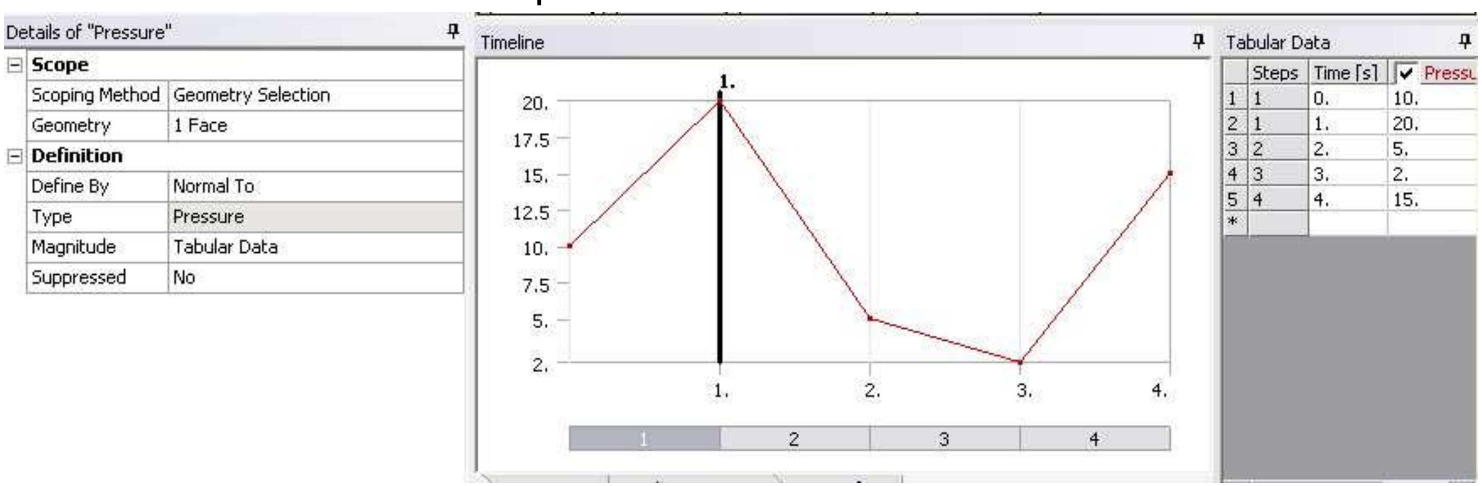
. . . Analysis Settings – Step Controls

Courtesy Ansys Inc

- Step Controls:
  - Multiple steps allow a series of static analyses to be set up and solved sequentially.
  - For a static analysis, the end time can be used as a counter/tracker to identify the load steps and sub steps.
  - Results can be viewed step by step.
  - Load values for each step can be entered in the “Tabular Data” section provided.



The time and load value are displayed in the graphics window



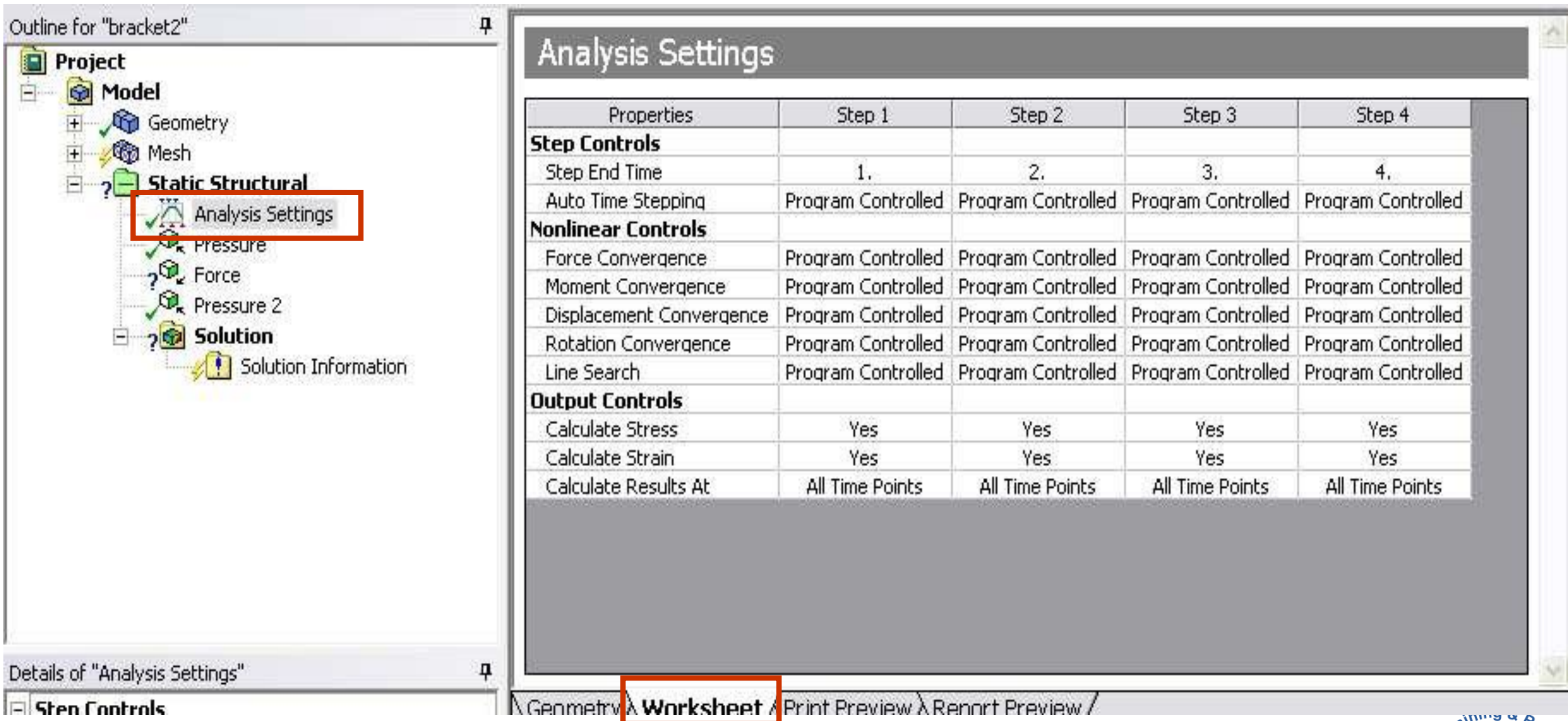
© Technology Training & Research Centre

TTRC

Courtesy Ansys Inc

. . . Multiple Steps

- A summary of all the different steps can be viewed by highlighting “Analysis Type” and then selecting the “Worksheet” tab.

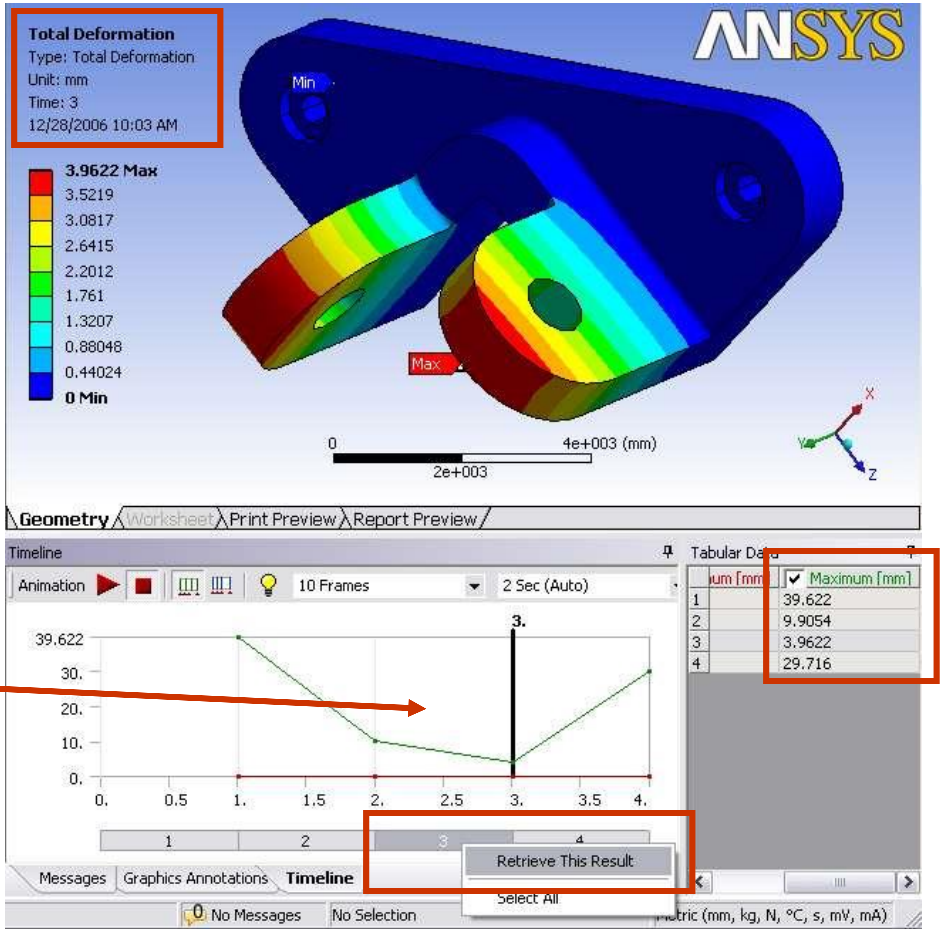


Properties	Step 1	Step 2	Step 3	Step 4
Step Controls				
Step End Time	1.	2.	3.	4.
Auto Time Stepping	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Nonlinear Controls				
Force Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Moment Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Displacement Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Rotation Convergence	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Line Search	Program Controlled	Program Controlled	Program Controlled	Program Controlled
Output Controls				
Calculate Stress	Yes	Yes	Yes	Yes
Calculate Strain	Yes	Yes	Yes	Yes
Calculate Results At	All Time Points	All Time Points	All Time Points	All Time Points

© Technology Training & Research Centre

. . . Multiple Steps

- Results for each individual step can be viewed after the solution by selecting the desired step and RMB >"Retrieve This Result".

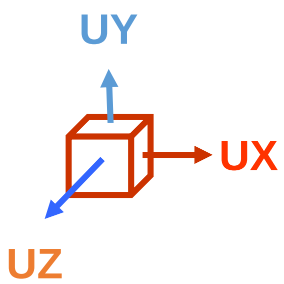


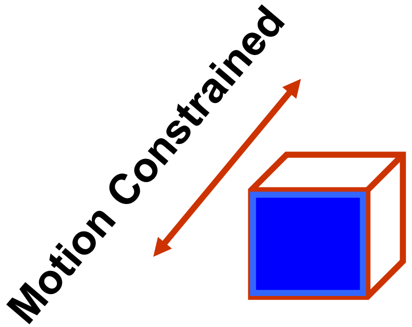
Select desired step and RMB to retrieve result

© Technology Training & Research Centre

D. Loads and Supports

- Loads and supports are thought of in terms of the degrees of freedom (DOF) available for the elements used.
- In solids the DOF are x, y and z translations (for shells we add rotational DOF rotx, roty and rotz).
- Supports, regardless of actual names, are always defined in terms of DOF.
- For example a "Frictionless Support" applied to the Z surface of the block shown would indicate that the Z degree of freedom is no longer free (all other DOF are free).





Frictionless surface

© Technology Training & Research Centre

Courtesy Ansys Inc

. . . Loads and Supports

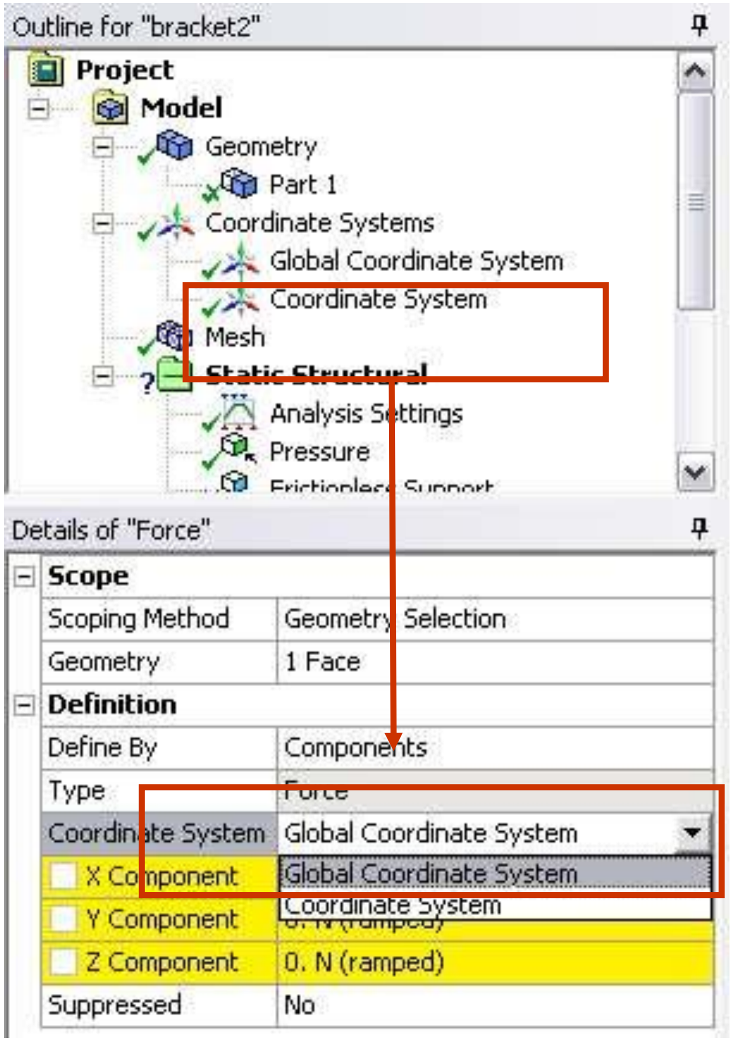
- Load types:
  - Inertial loads:
    - These loads act on the entire system.
    - Density is required for mass calculations.
    - These are only loads which act on defined Point Masses.
  - Structural Loads:
    - Forces or moments acting on parts of the system.
  - Structural Supports:
    - Constraints that prevent movement on certain regions.
  - Thermal Loads:
    - The thermal loads which result in a temperature field causing thermal expansion/contraction in the model.

© Technology Training & Research Centre

Courtesy Ansys Inc

… Directional Loads

- 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.



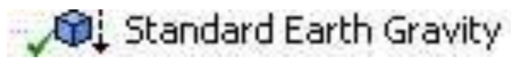
Load	Supports Coordinate Systems
Acceleration	No
Standard Earth Gravity	Yes
Rotational Velocity	Yes
Force	Yes
Remote Force	Location of Origin Only
Bearing Load	Yes
Moment	Yes
Given Displacement	Yes

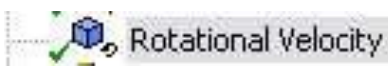
© Technology Training & Research Centre

Courtesy Ansys Inc

… Acceleration & Gravity

- Acceleration: 

  - Acts on entire model in length/time^2 units.
  - Acceleration can be defined by Components or Vector.
  - Body will move in the opposite direction of the applied acceleration.
- Standard Earth Gravity: 

  - Value applied coincides with selected unit system.
  - Standard Earth Gravity direction is defined along one of three global or local coordinate system axes.
  - Body will move in the same direction of the applied gravity.
- Rotational velocity: 

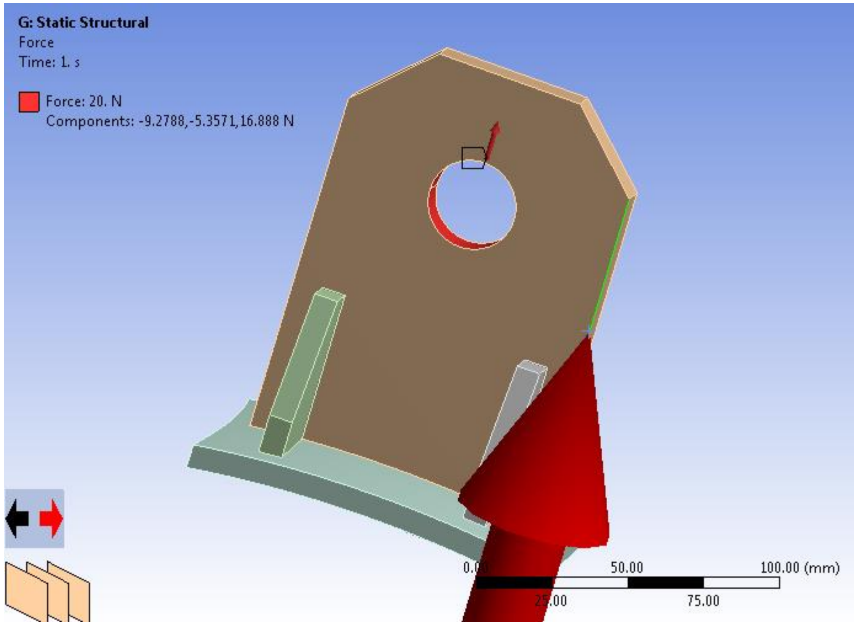
  - Entire model rotates about an axis at a given rate.
  - Define by vector or component method.
  - Input can be in radians per second (default) or RPM.

© Technology Training & Research Centre

Courtesy Ansys Inc

… Forces and Pressures

- Pressure loading: Pressure
  - Applied to surfaces, acts normal to the surface.
  - Positive value into surface, negative value acts out of surface.
  - Units of pressure are in force per area.



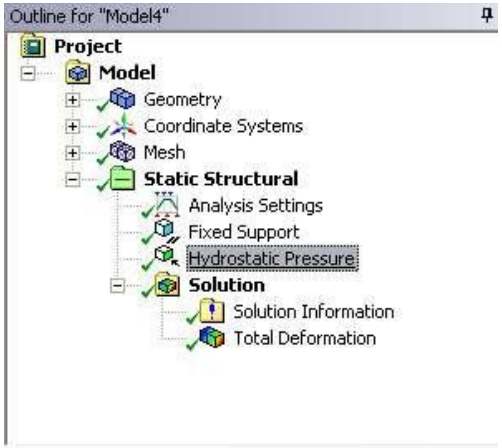
- Force loading: Force
  - Forces can be applied on vertices, edges, or surfaces.
  - The force will be *evenly distributed on all entities*. Units are mass*length/time^2.
  - Force can be defined via vector or component methods.

© Technology Training & Research Centre

Courtesy Ansys Inc

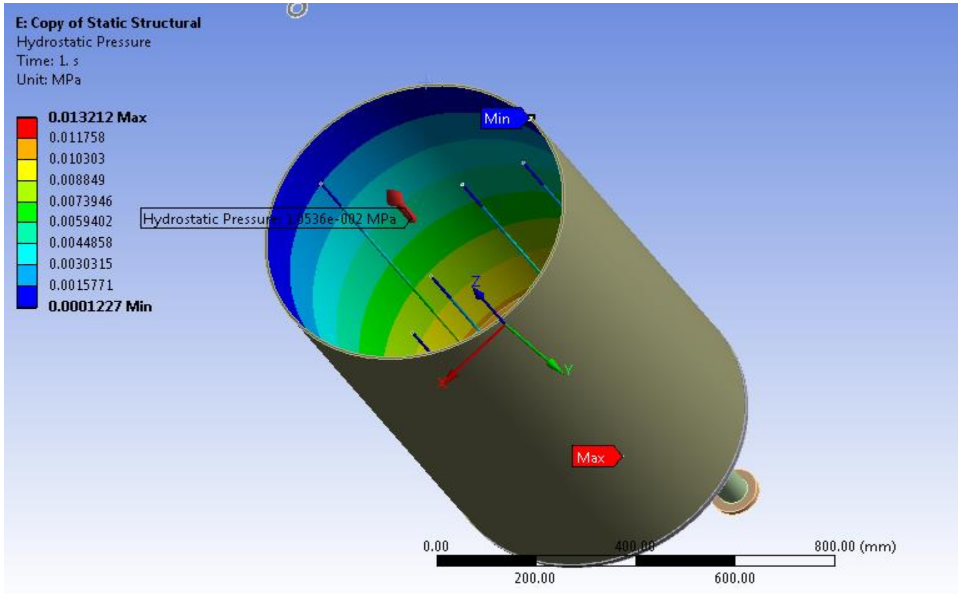
… Hydrostatic Pressure

- Hydrostatic Pressure Hydrostatic Pressure
  - Applies a linearly varying load to a surface (solid or shell) to mimic fluid force acting on the structure.
  - Fluid may be contained or external.
    - User specifies:
      - Magnitude and direction of acceleration.
      - Fluid Density.
      - Coordinate system representing the free surface of the fluid.
      - For Shells, a Top/Bottom face option is provided.



Details of "Hydrostatic Pressure"

Scope	
Scoping Method	Geometry Selection
Geometry	5 Faces
Shell Face	Top
Hydrostatic Acceleration	
Define By	Vector
Magnitude	9.8 m/s²
Direction	Click to Change
Definition	
Type	Hydrostatic Pressure
Suppressed	No
Fluid Density	1000. kg/m³
Free Surface Location	
Coordinate System	Coordinate System
X Coordinate	0. m
Y Coordinate	0. m
Z Coordinate	0. m
Location	Click to Change

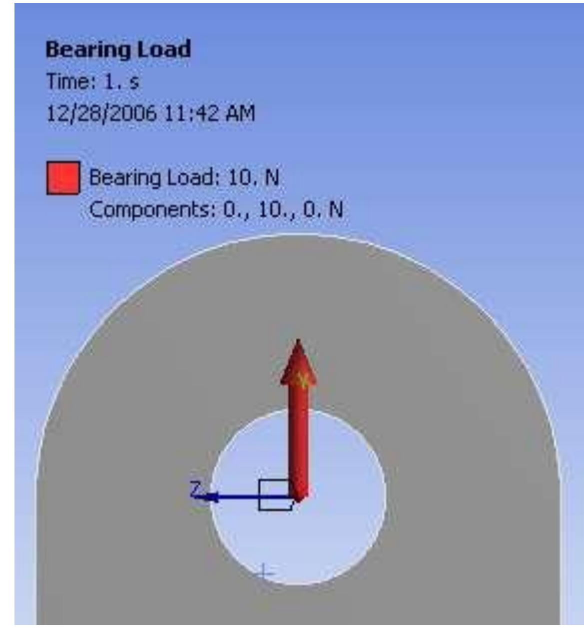


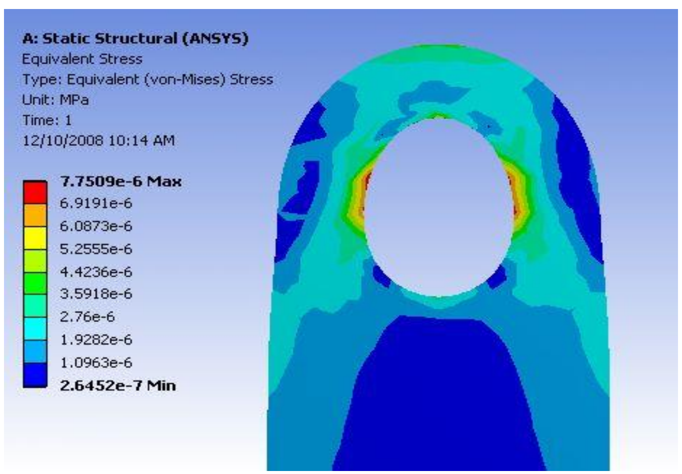
© Technology Training & Research Centre

Courtesy Ansys Inc

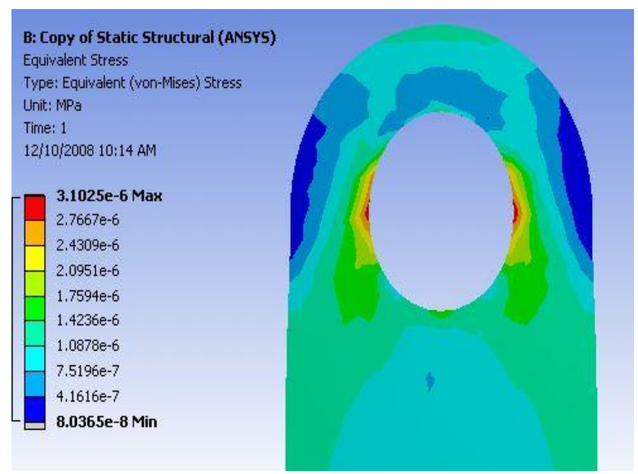
… Bearing Load

- Bearing Load (force): Bearing Load
  - Force component distributed on compressive side using projected area.
    - Axial components are not allowed.
    - Use only one bearing load per cylindrical surface.
      - If the cylindrical surface is split be sure to select both halves of cylindrical surface when applying this load.
  - Bearing load can be defined via vector or component method.





Bearing Load



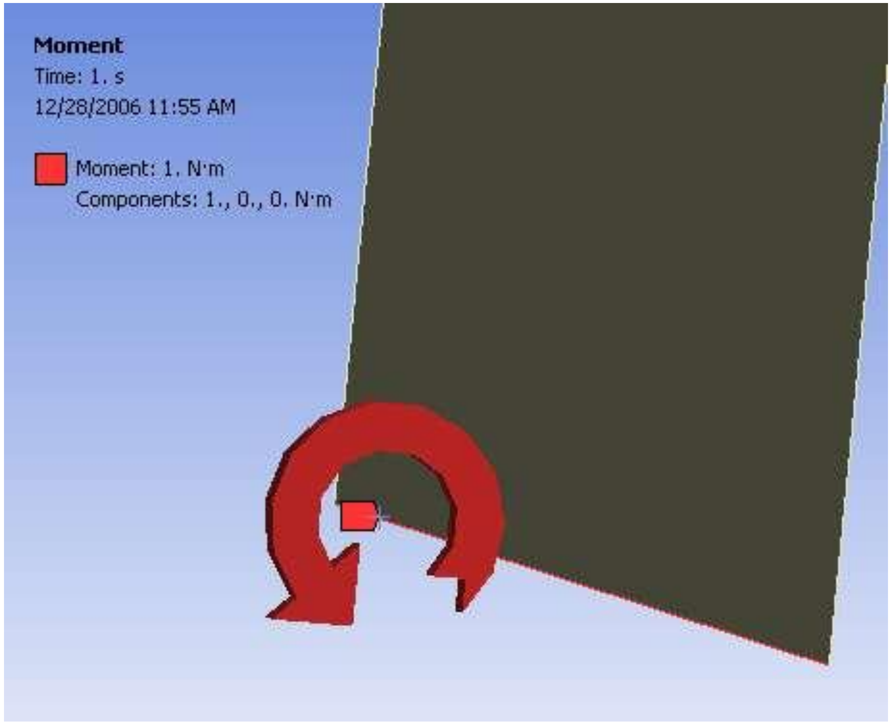
Force Load

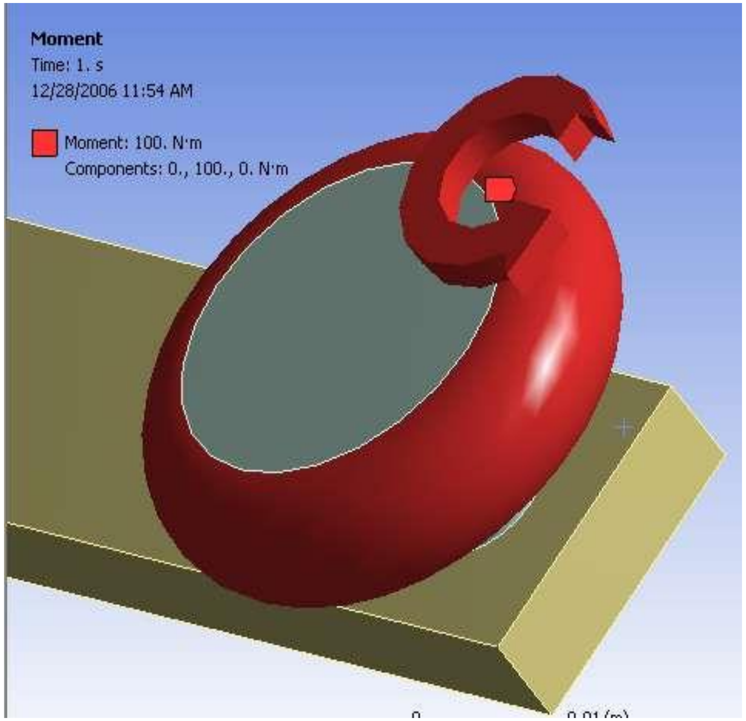
Technology Training & Research Centre
TTRC

Courtesy Ansys Inc

… Moment Load

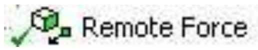
- Moment Loading : Moment
  - For solid bodies moments can be applied on a surface only.
  - If multiple surfaces are selected, the moment load is evenly distributed.
  - Vector or component method can be employed using the right hand rule.
  - For surface bodies a moment can be applied to a vertex, edge or surface.
  - Units of moment are in Force*length.





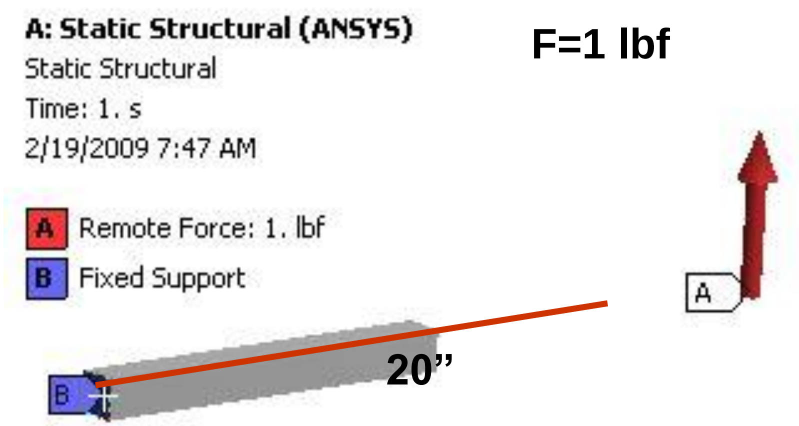
© Technology Training & Research Centre

… Remote Load

- Remote Force Loading : 

  - Applies an offset force on a surface or edge of a body.
  - The user supplies the origin of the force (geometry or coordinates).
  - 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

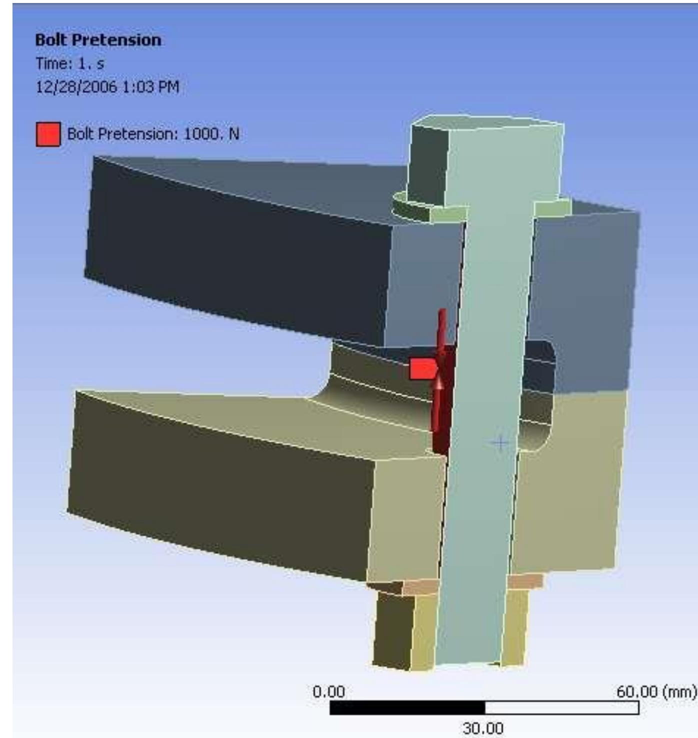
Moment Reaction

© Technology Training & Research Centre

Courtesy Ansys Inc

. . . Bolt Pretension

- Bolt Pretension: Bolt Pretension
  - Applies a pretension load to a cylindrical section using:
    - Pretension load (force) OR
    - Adjustment (length)
  - For body loading a local coordinate system is required (preload in z direction).
  - Automatic two load step solution:
    - LS1: pretension load, boundary conditions and contact conditions are applied.
    - LS2: relative motion of the pretension section is fixed and external loads are applied.



Details of "Bolt Pretension"

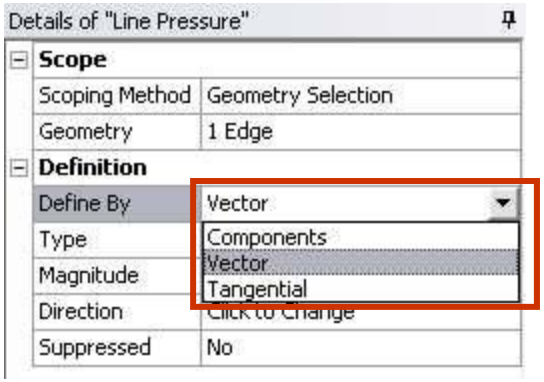
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Bolt Pretension
Suppressed	No
Define By	Load
Preload	1000. N

© Technology Training & Research Centre

. . . Line Pressure

- Line Pressure loading :

  - Applies a distributed force on one edge only for 3-D simulations, using force density loading.
  - Units are in force/length.
  - Can be defined by :
    - Magnitude and Vector
    - Magnitude and component direction (global or local coordinate systems)
    - Magnitude and tangential



© Technology Training & Research Centre

Courtesy Ansys Inc

… Supports

- Fixed Support :
  - Constraints all degrees of freedom on vertex, edge, or surface
    - Solid bodies: constrains x, y, and z
    - Surface and line bodies: constrains x, y, z, rotx, roty and rotz

Fixed Support

Displacement

- Given Displacement :
  - Applies known displacement on vertex, edge, or surface
  - 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.

Elastic Support

- Elastic Support :
  - Allows faces/edges to deform according to a spring behavior.
  - Foundation stiffness is the pressure required to produce unit normal deflection of the foundation

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

Details of "Elastic Support"

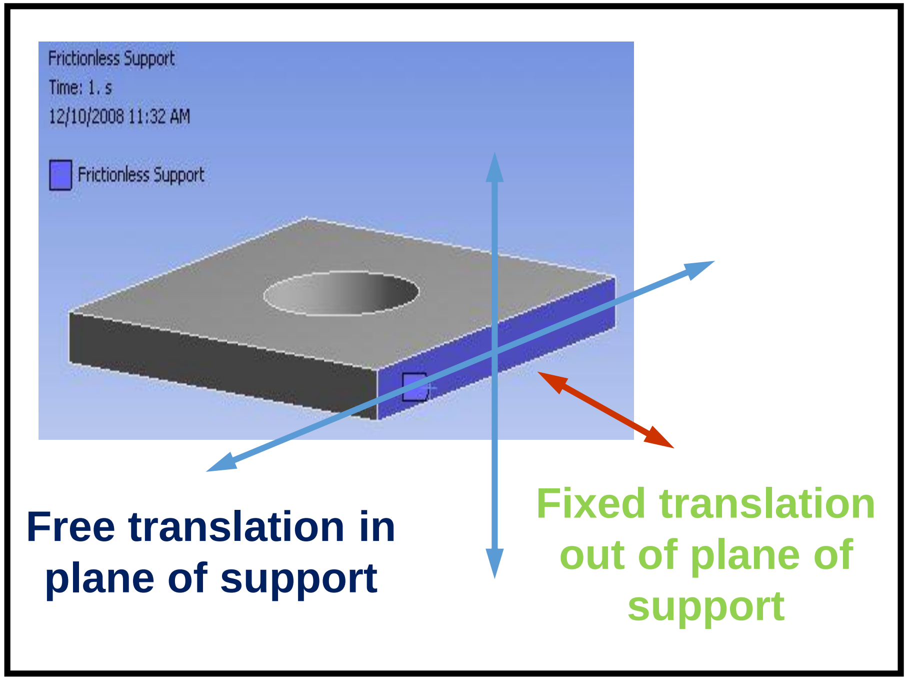
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Elastic Support
Suppressed	No
Foundation Stiffness	1. N/mm³

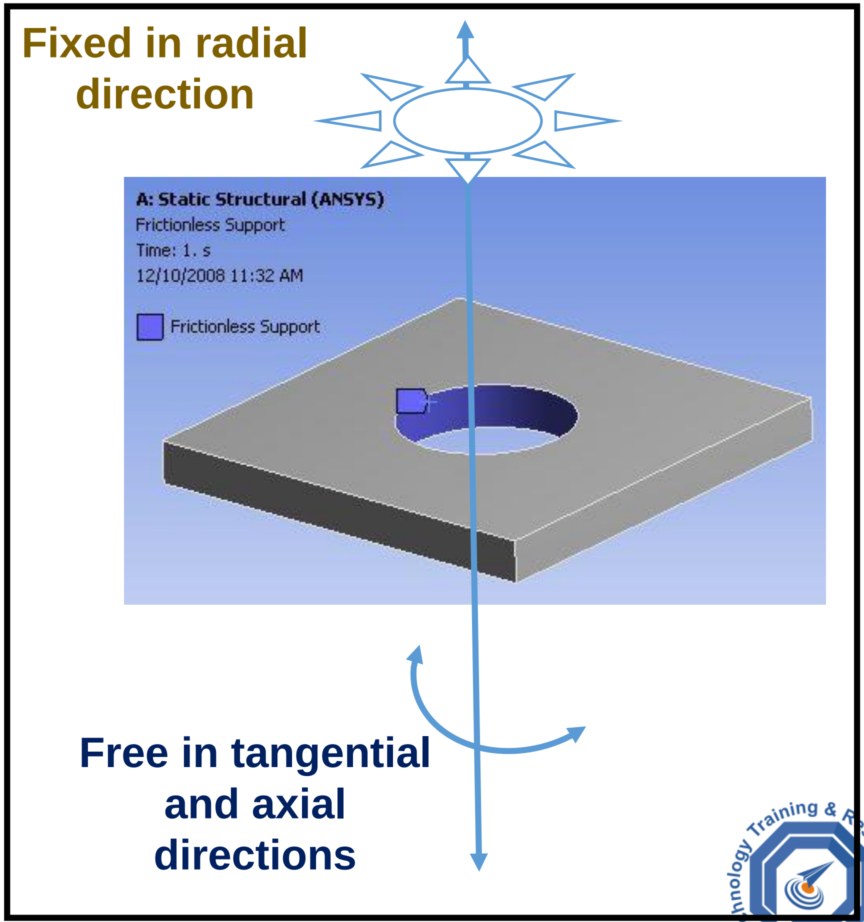
© Technology Training & Research Centre

... Supports

Courtesy Ansys Inc

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





© Technology Training & Research Centre

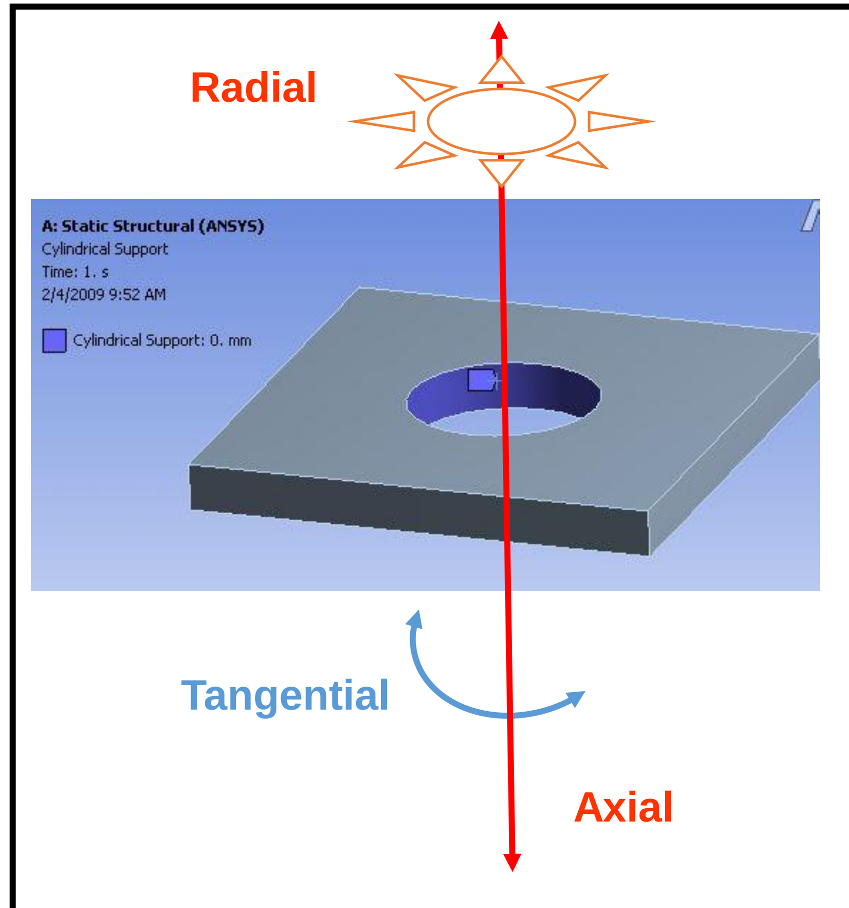
... Supports

- Cylindrical Support: Cylindrical Support
  - Provides individual control for axial, radial, or tangential constraints.
  - Applied on cylindrical surfaces.

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 . . .



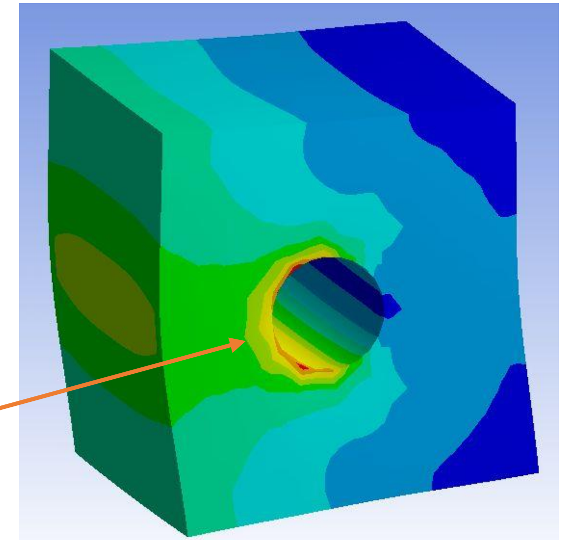
© Technology Training & Research Centre

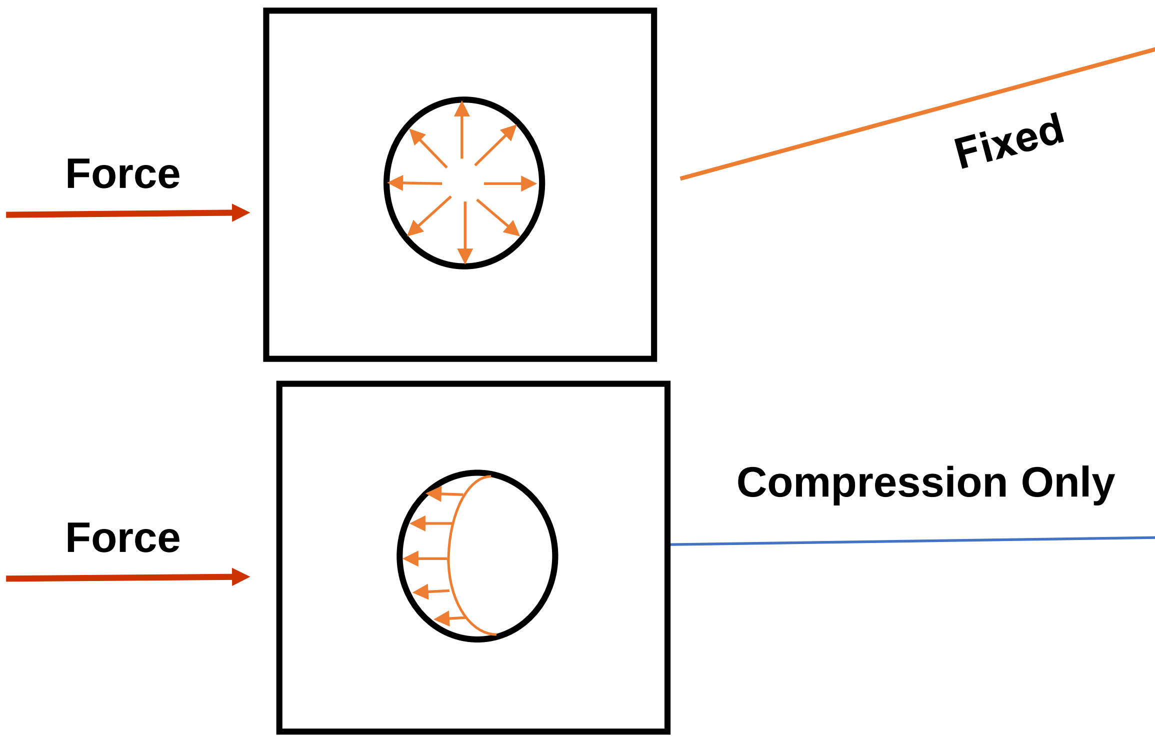
Courtesy Ansys Inc

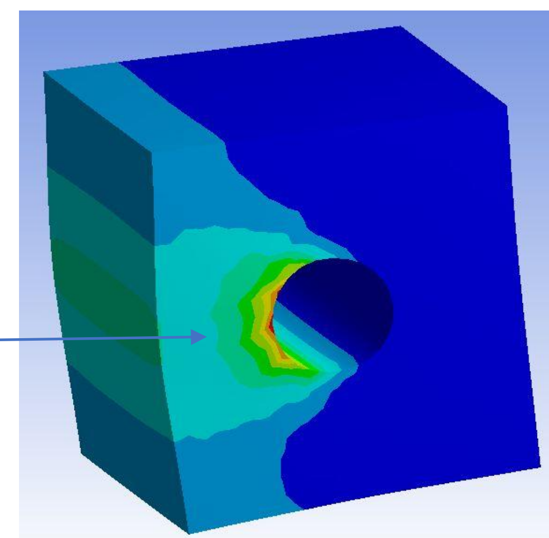
… Supports (Solid Bodies)

- Compression Only Support :

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





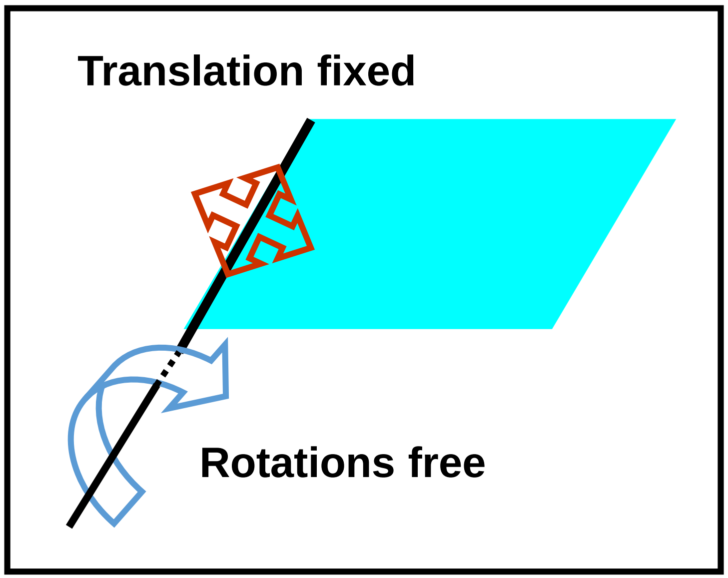


© Technology Training & Research Centre

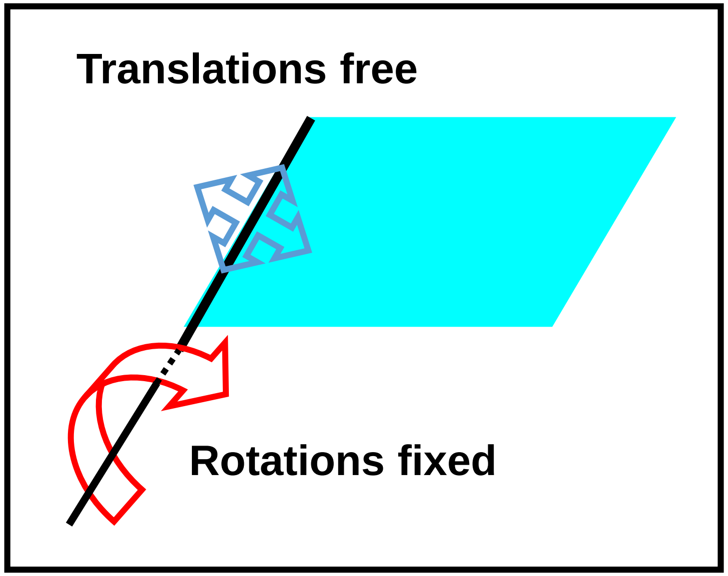
Courtesy Ansys Inc

… Supports (Line/Surface Bodies)

- Simply Supported : Simply Supported
  - Can be applied on edge or vertex of surface or line bodies
  - Prevents all translations but all rotations are free
- Fixed Rotation : Fixed Rotation
  - Can be applied on surface, edge, or vertex of surface or line bodies
  - Constrains rotations but translations are free



Simply Supported Edge

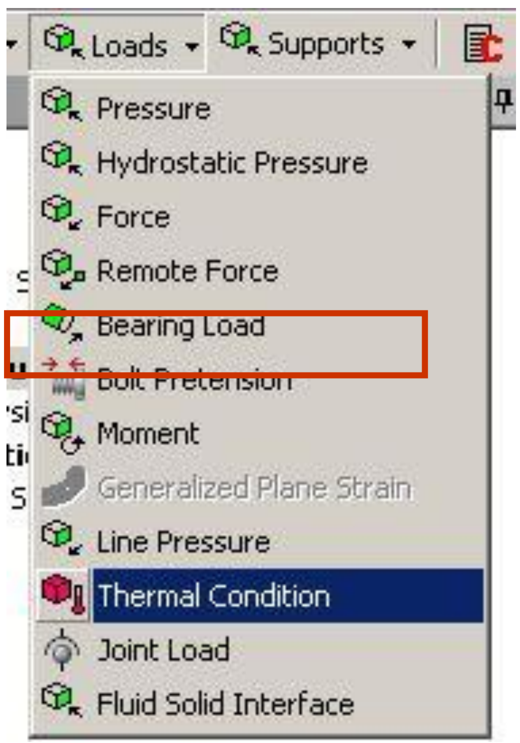


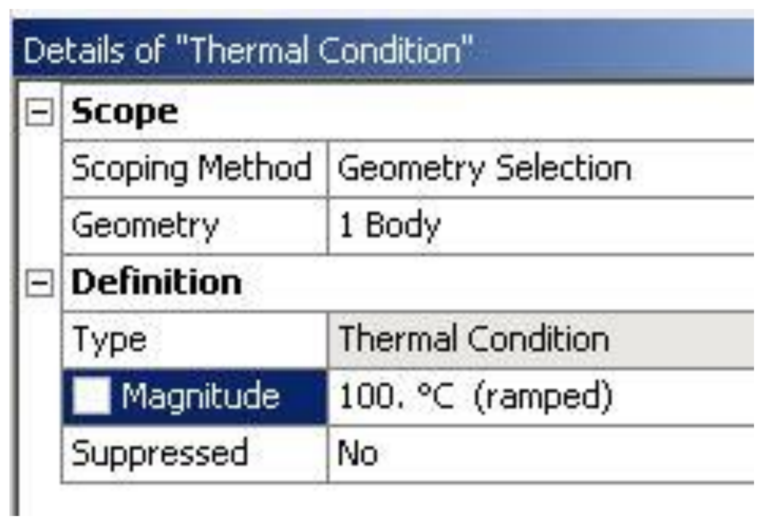
Fixed Rotation Edge

© Technology Training & Research Centre

… Thermal Loading

- Thermal condition :
  - Applies a uniform temperature in a structural analysis.
  - Appears under “Loads” in structural analysis.
  - A reference temperature must be provided (see next slide).





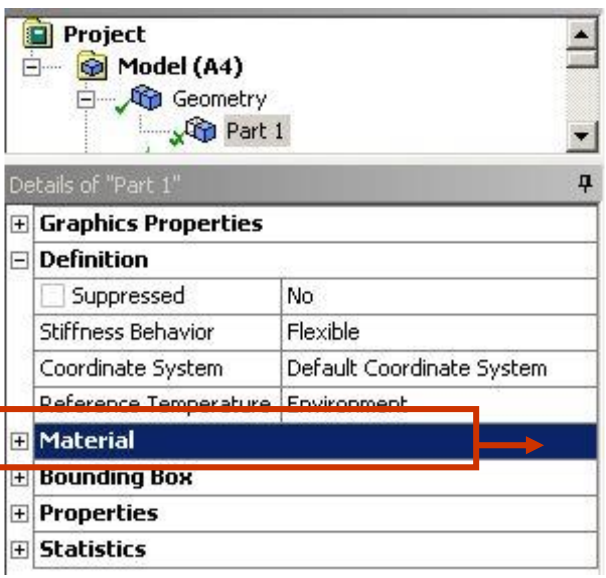
© Technology Training & Research Centre

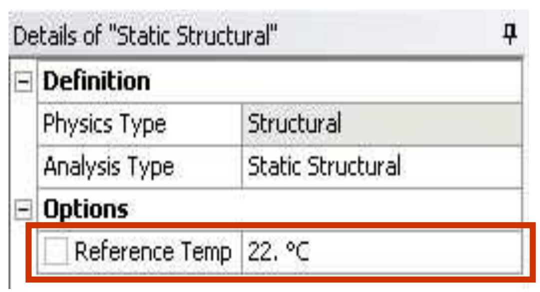
… Thermal Loading

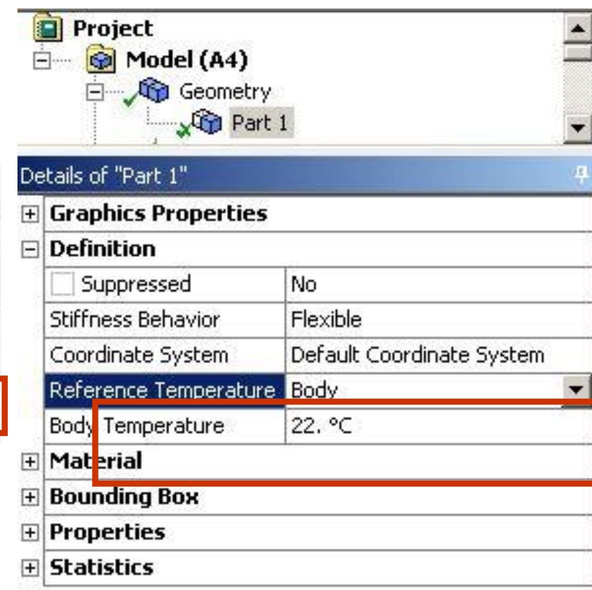
- A temperature differential can cause thermal expansion or contraction in a structure:
  - Thermal strains ($e_{th)}$ are calculated as follows:

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

  - a = thermal expansion coefficient (CTE material property).
  - T_{ref} = reference temperature (thermal strains are zero).
  - T = applied temperature (see previous slide).
  - Reference temperature is defined in the environment branch (global) or as a property of individual bodies.



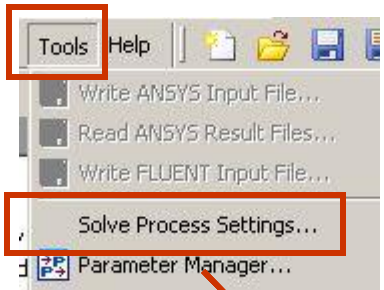


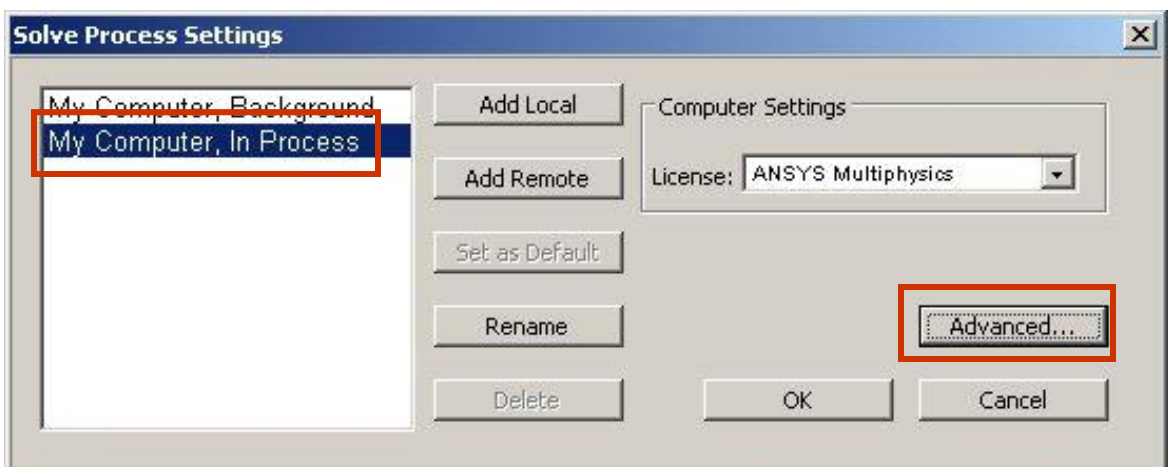


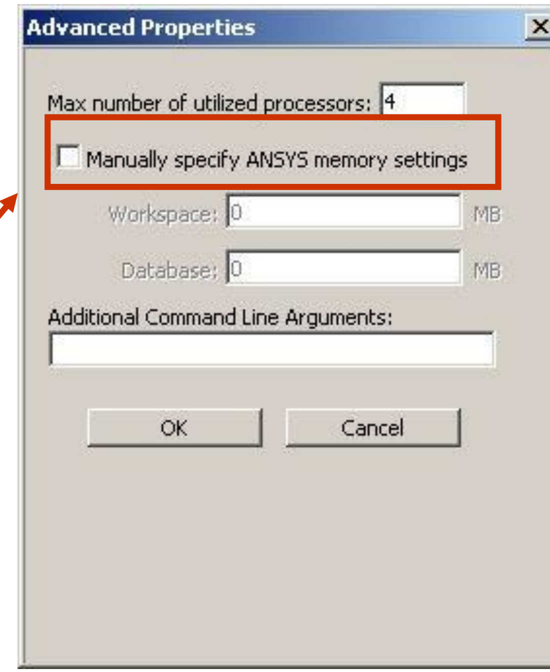
© Technology Training & Research Centre

… 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 use, “Tools > Solve Process Settings”.







© Technology Training & Research Centre

End of Lecture 1

THANK YOU!!

© Technology Training & Research Centre