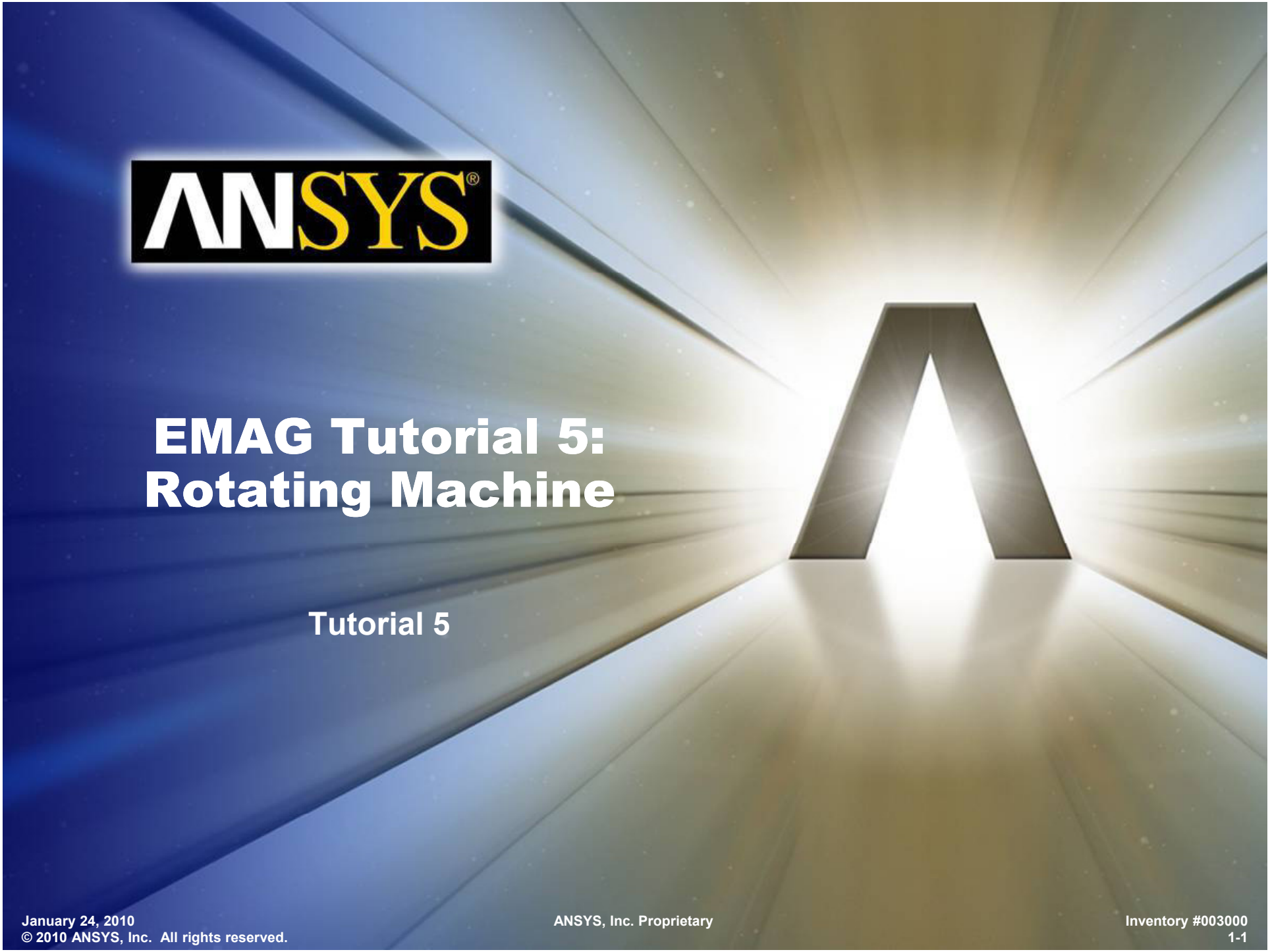
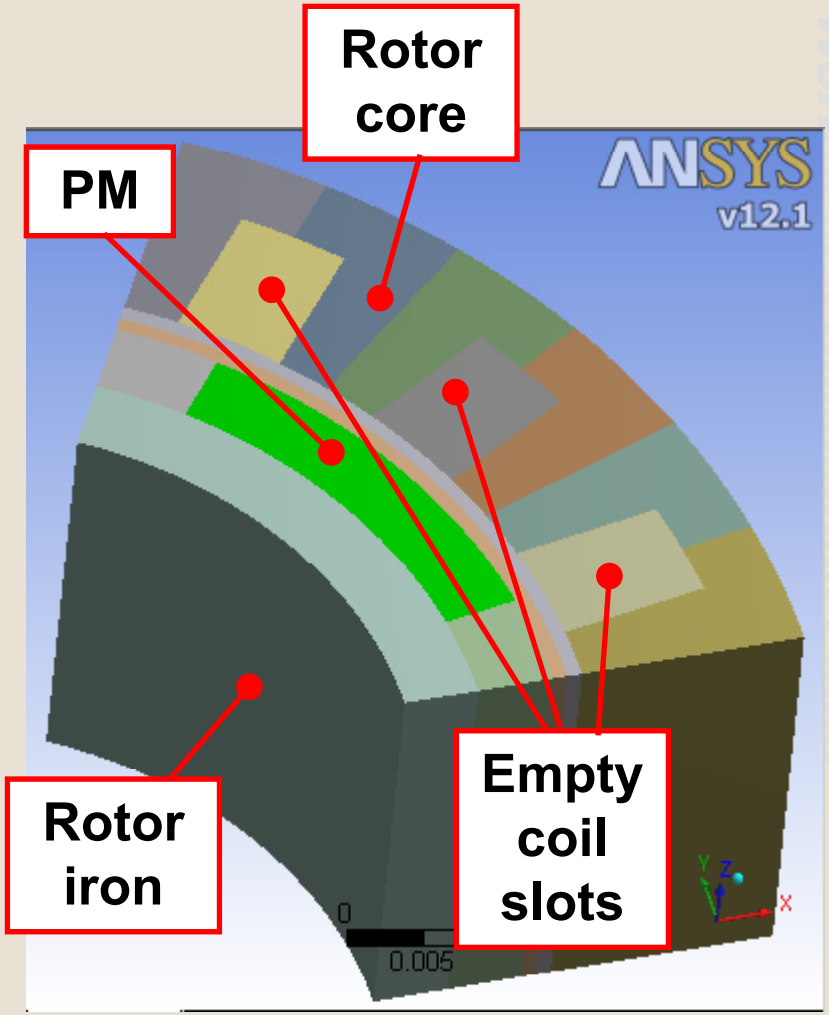


# EMAG Tutorial 5: Rotating Machine

Tutorial 5


© 2010 ANSYS, Inc. All rights reserved.

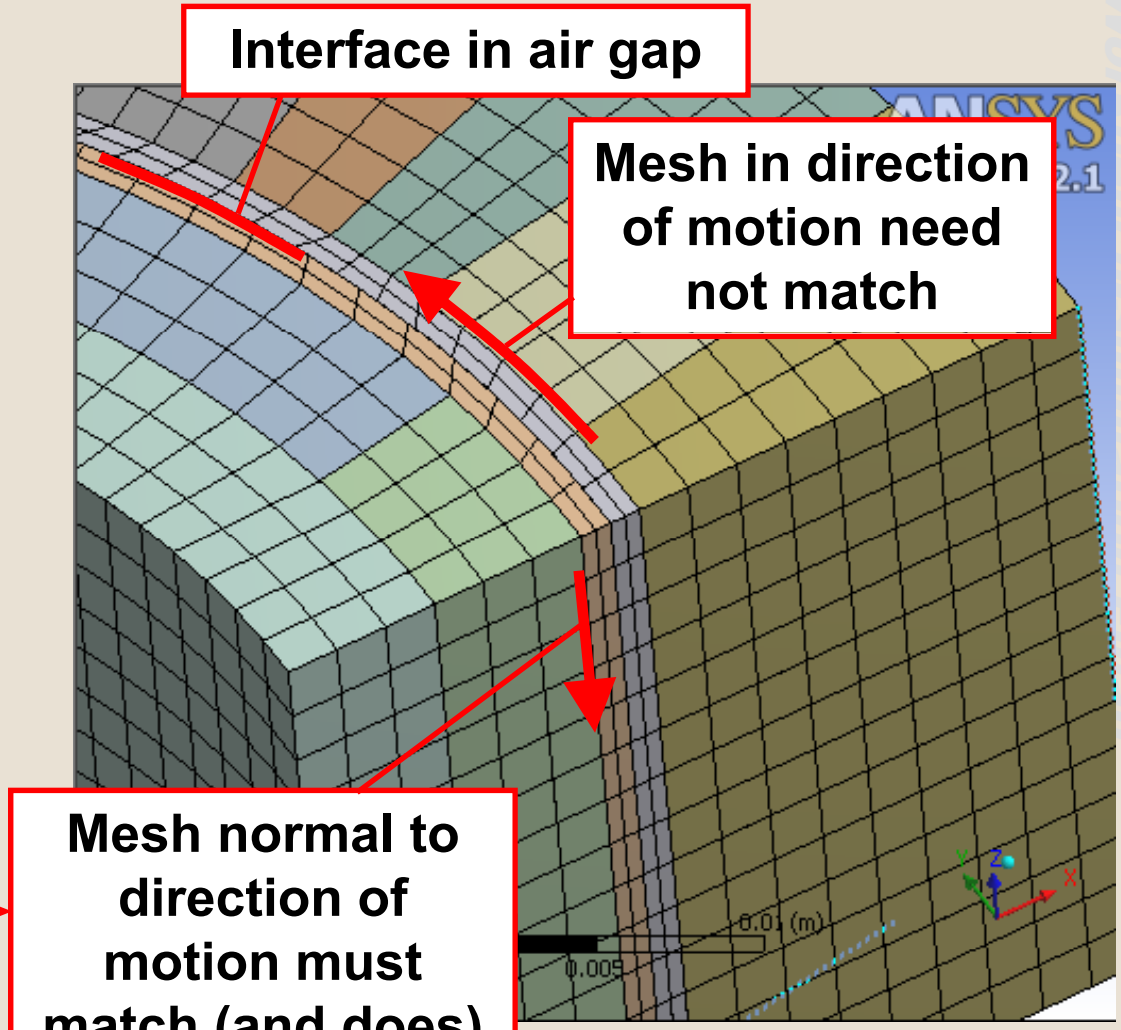
- Objectives
  - Set up a transient analysis of ***one half of the length*** of a ***single periodic sector*** of a rotating machine. ***Odd periodic boundary conditions*** will be used to mimic presence of adjacent periodic sectors.
  - The rotor contains a single permanent magnet which is poled radially outward and attached to a permeable ring.
  - The stator is magnetically permeable structure containing 3 poles. It has no coils in this exercise; a stranded conductor option that can be used to model coils will not be available until version 13 .
  - Define 90º rotation of the rotor which occurs over a one second time interval.
  - The PM will be electrically conductive; eddy currents and Joule heating will be calculated.
  - Net rotor torque will be calculated.





© 2010 ANSYS, Inc. All rights reserved.

- Procedure
  - WB supports only magnetostatic analyses. Transient analyses, especially those including motion effects, will require *command objects*.
  - Rigid body rotation or translational motion can be simulated with an undocumented (even in the traditional ANSYS environment) set of commands ("ARMINT" feature).
  - Insertion of command objects that set up motion effects is largely automated with a special wizard (supplied courtesy of CADFEM).
  - "Ladder mesh" requirement will be necessary to use ARMINT feature.



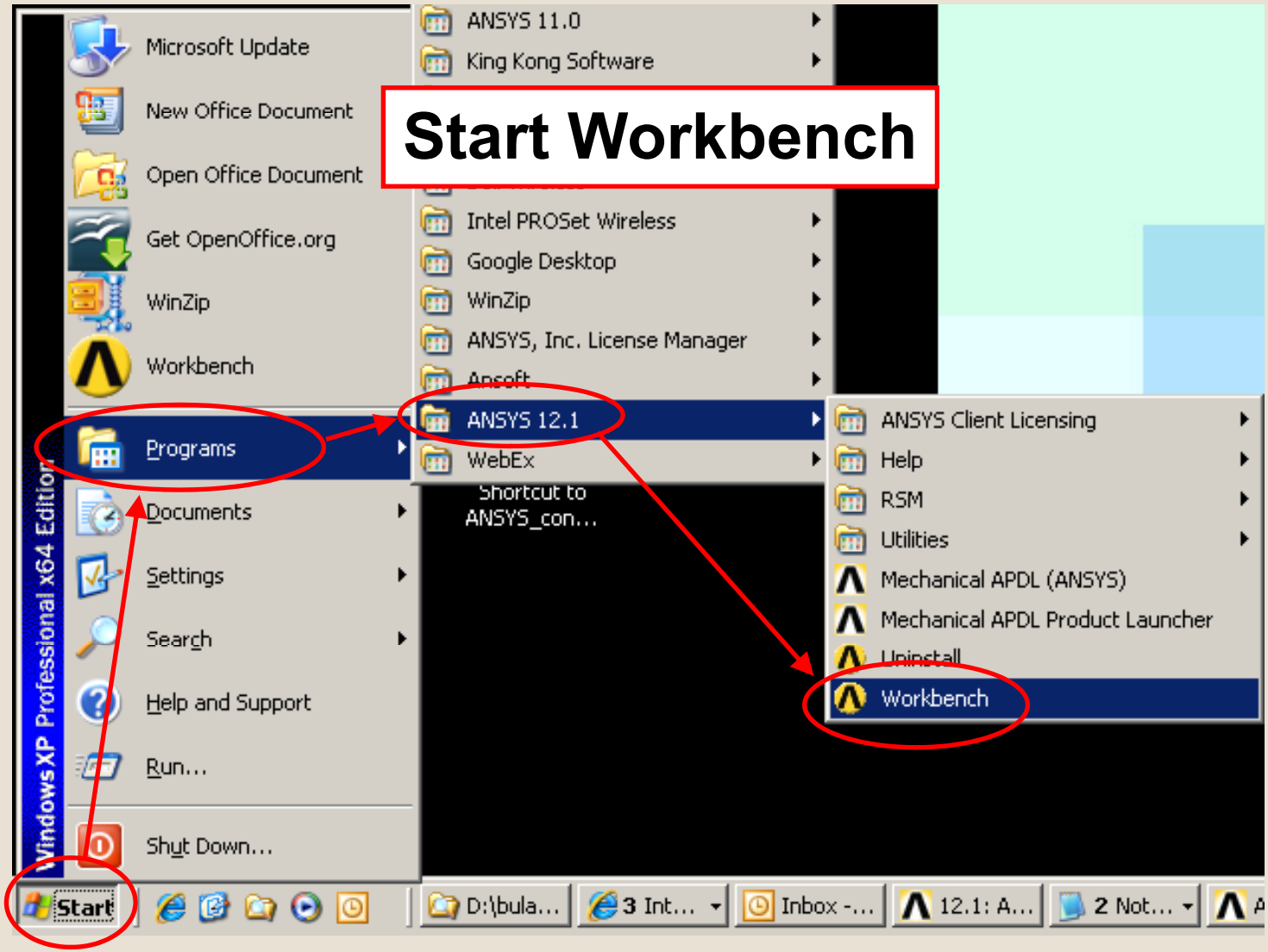


© 2010 ANSYS, Inc. All rights reserved.

# Start Workbench and Read in Geometry

Workbench-Simulation Emag

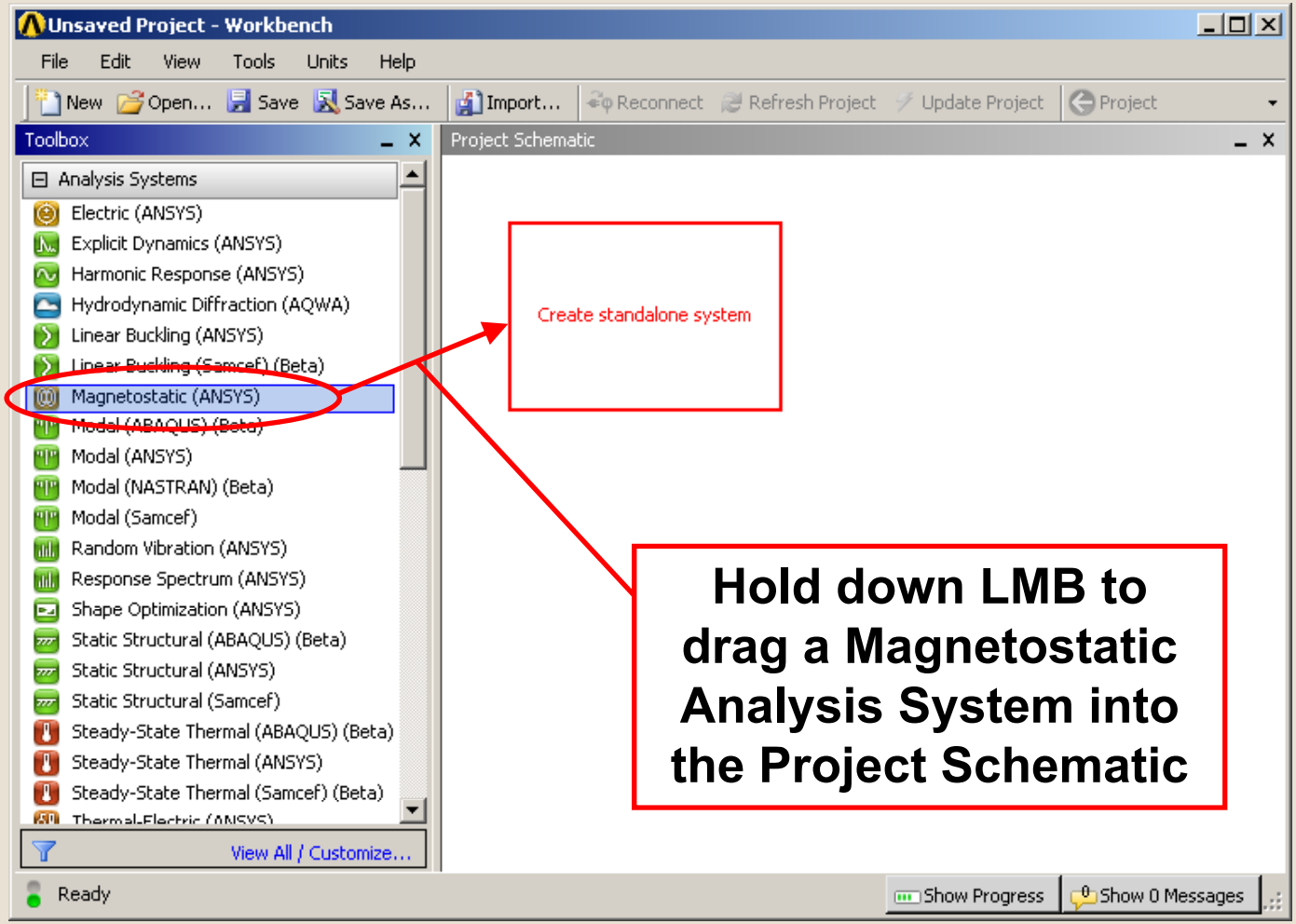

© 2010 ANSYS, Inc. All rights reserved.




Workbench-Simulation Emag

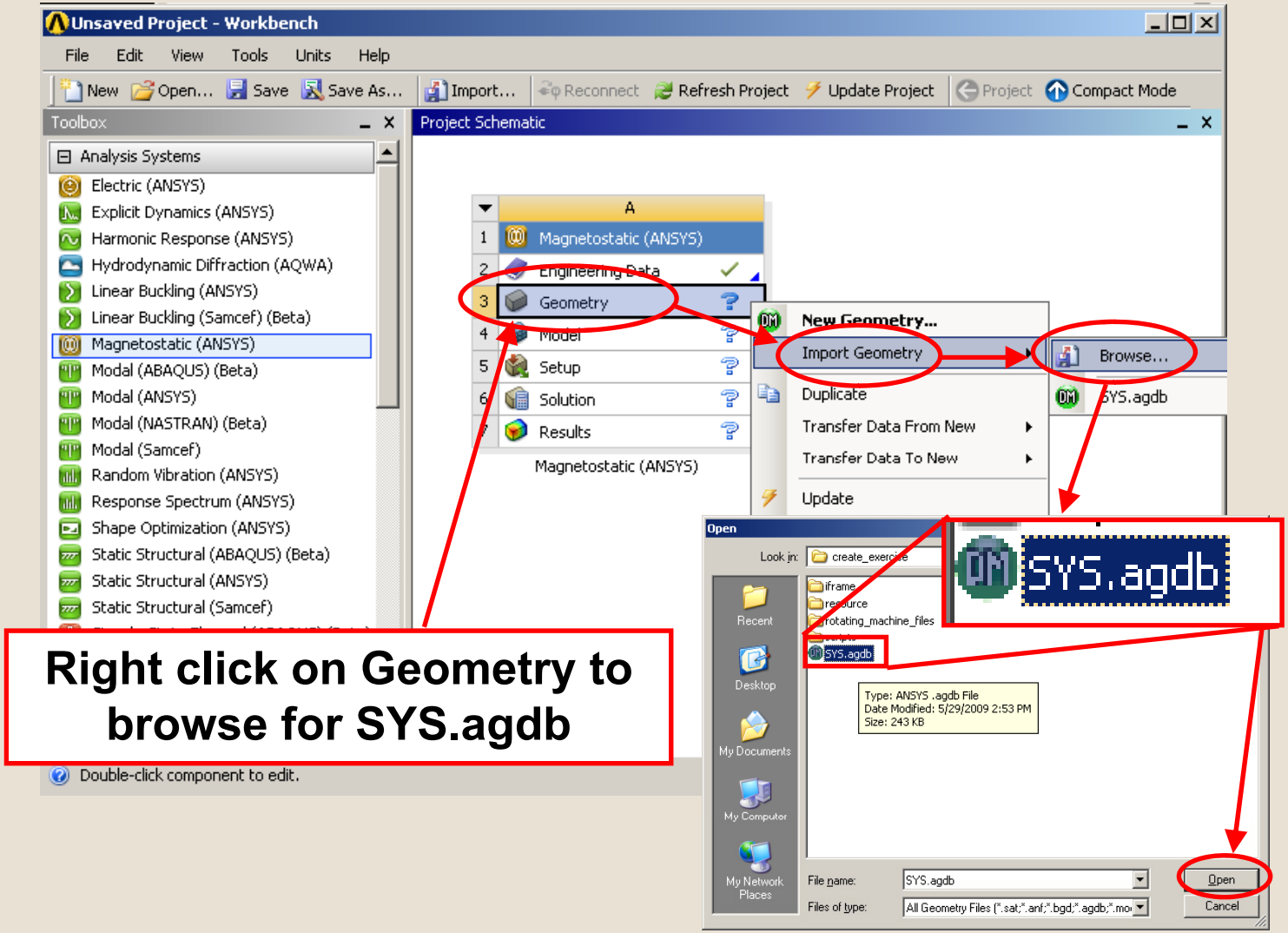

© 2010 ANSYS, Inc. All rights reserved.

# The Project Page Loads…




Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

Workbench-Simulation Emag

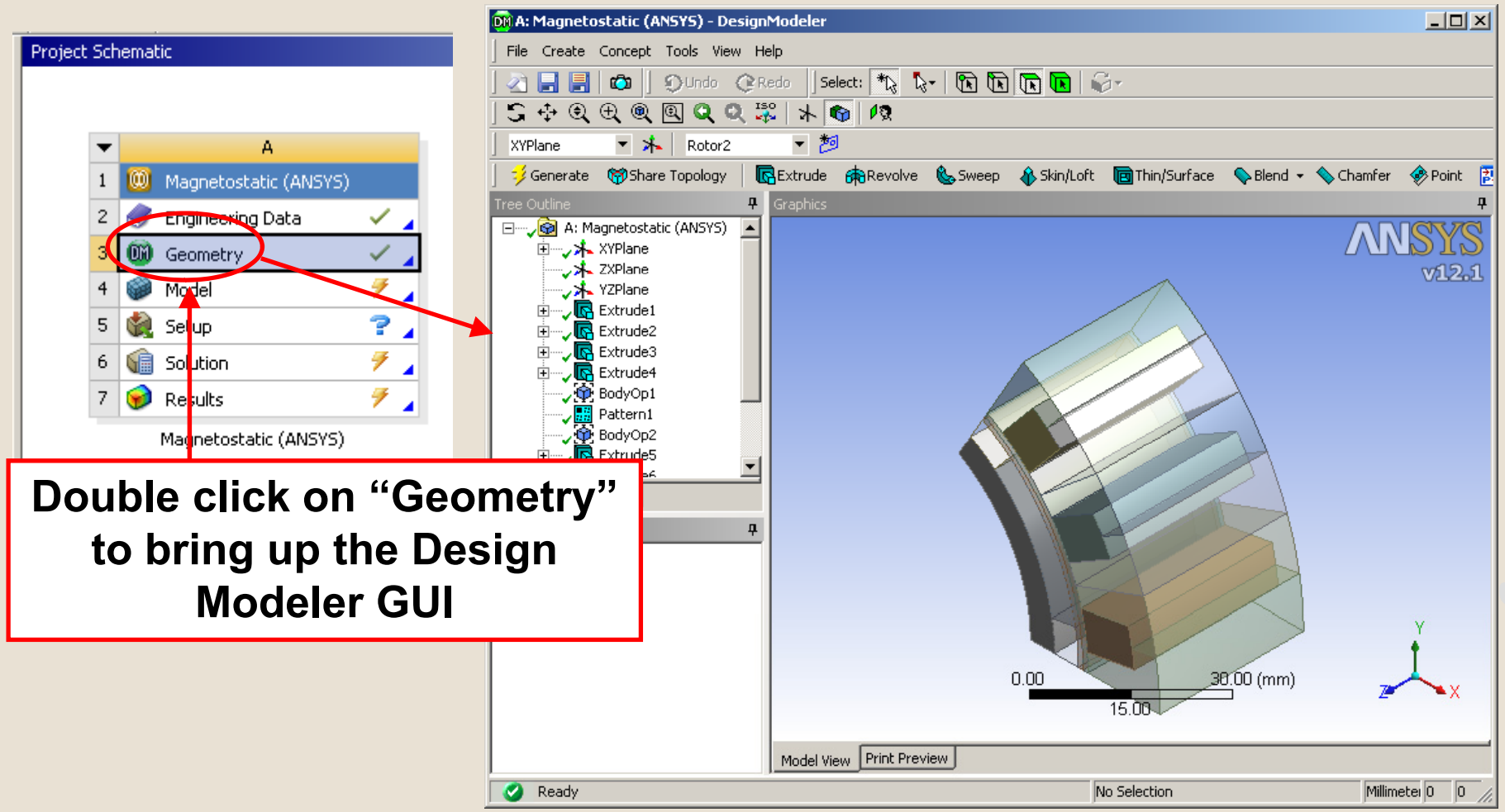
# User-Defined Coordinate Systems for PM Polarization and Periodic Boundary Conditions


© 2010 ANSYS, Inc. All rights reserved.

- We will need to create 2 coordinate systems
  - One coordinate system will be used to define the direction of polarization of the permanent magnet. It will be most convenient to create this coordinate system in DM. Coordinate system "PM_polarization" will have its x axis pointing in the "average radially outward" direction of the permanent magnet. Polarization may only be defined only in the positive or negative x direction, so anytime the direction of polarization is not aligned with the Global x direction, a special properly oriented coordinate system must be created for to define polarization.
  - Another *cylindrical* coordinate system will be needed to specify the odd periodic boundary condition. It will be most convenient to create this coordinate system in Mechanical.

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

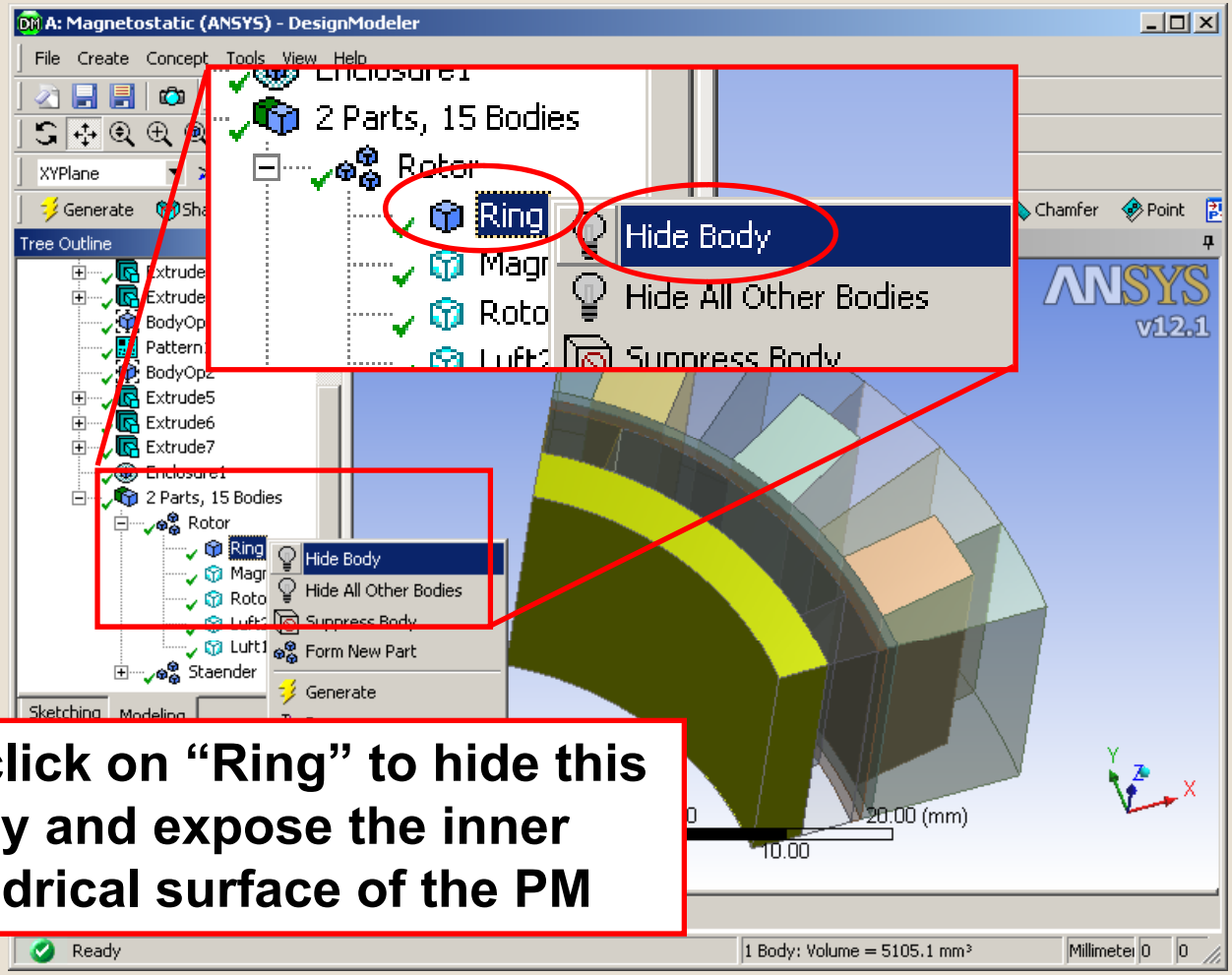



**Double click on "Geometry" to bring up the Design Modeler GUI**

Workbench Simulation Guide


© 2010 ANSYS, Inc. All rights reserved.

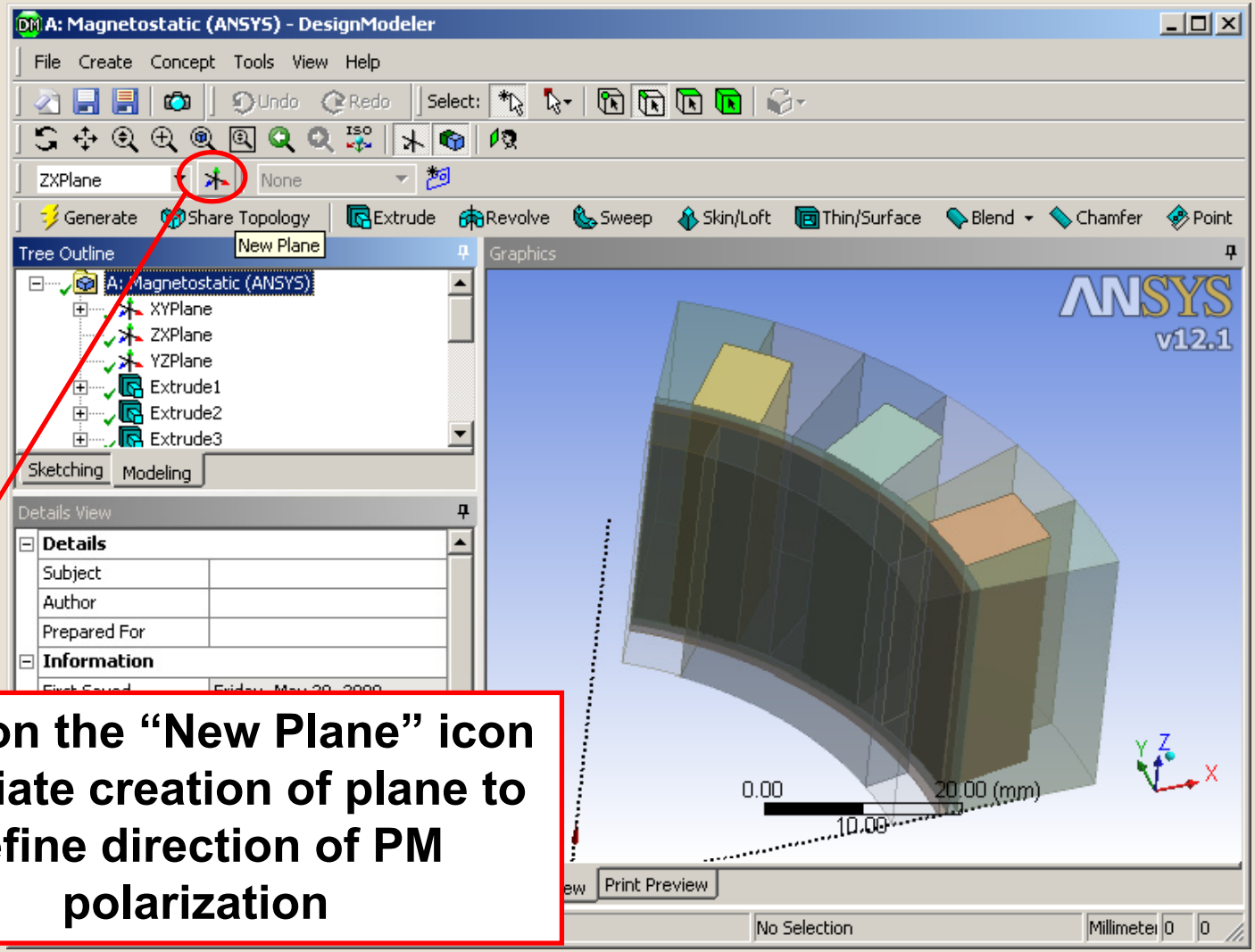
**Begin to define coordinate system used to define PM polarization**




**Right click on "Ring" to hide this body and expose the inner cylindrical surface of the PM**

Workbench-Simulation Emag

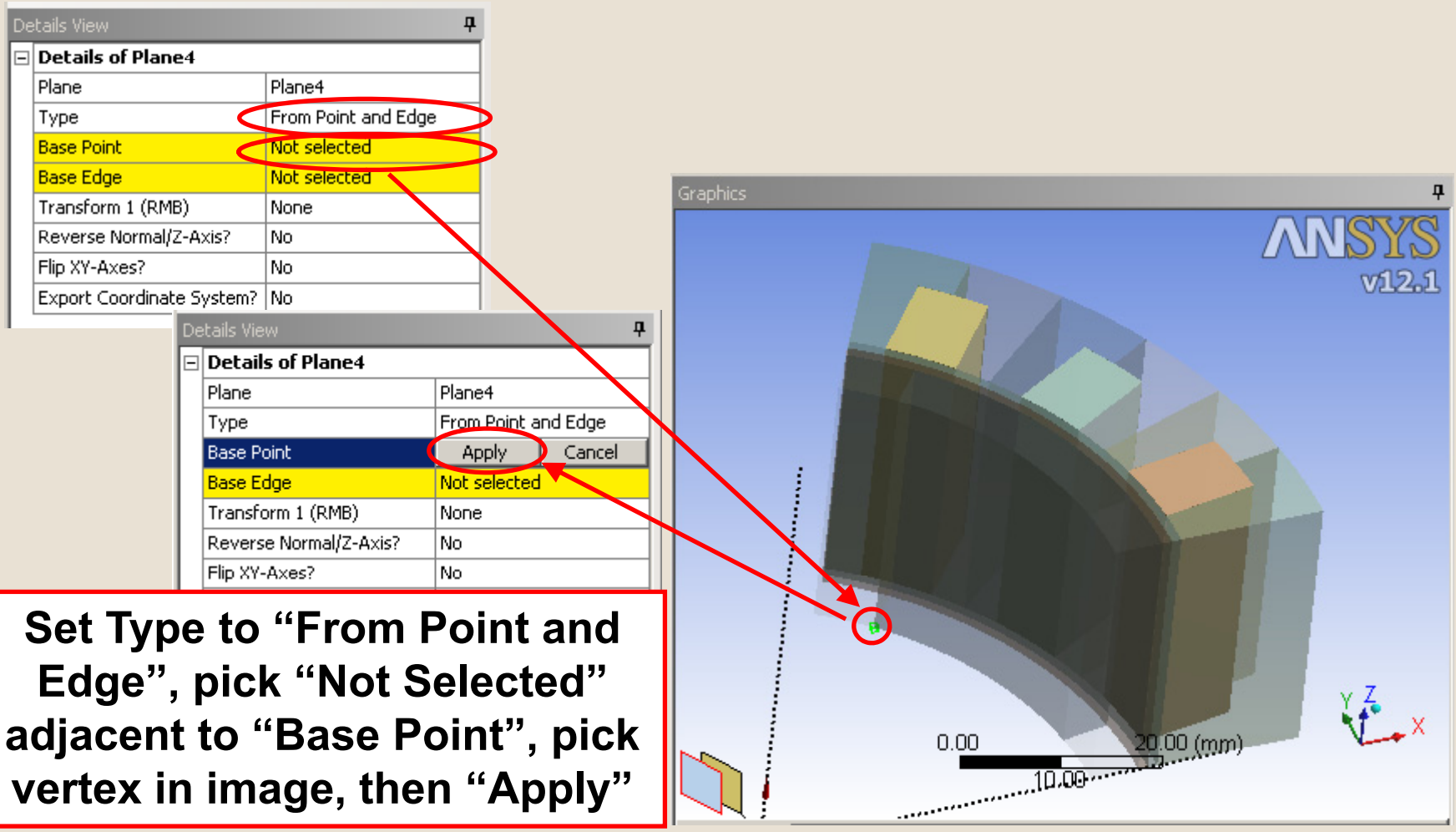

© 2010 ANSYS, Inc. All rights reserved.




**Click on the "New Plane" icon to initiate creation of plane to define direction of PM polarization**

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




**Set Type to "From Point and Edge", pick "Not Selected" adjacent to "Base Point", pick vertex in image, then "Apply"**

Workbench Simulation Emag

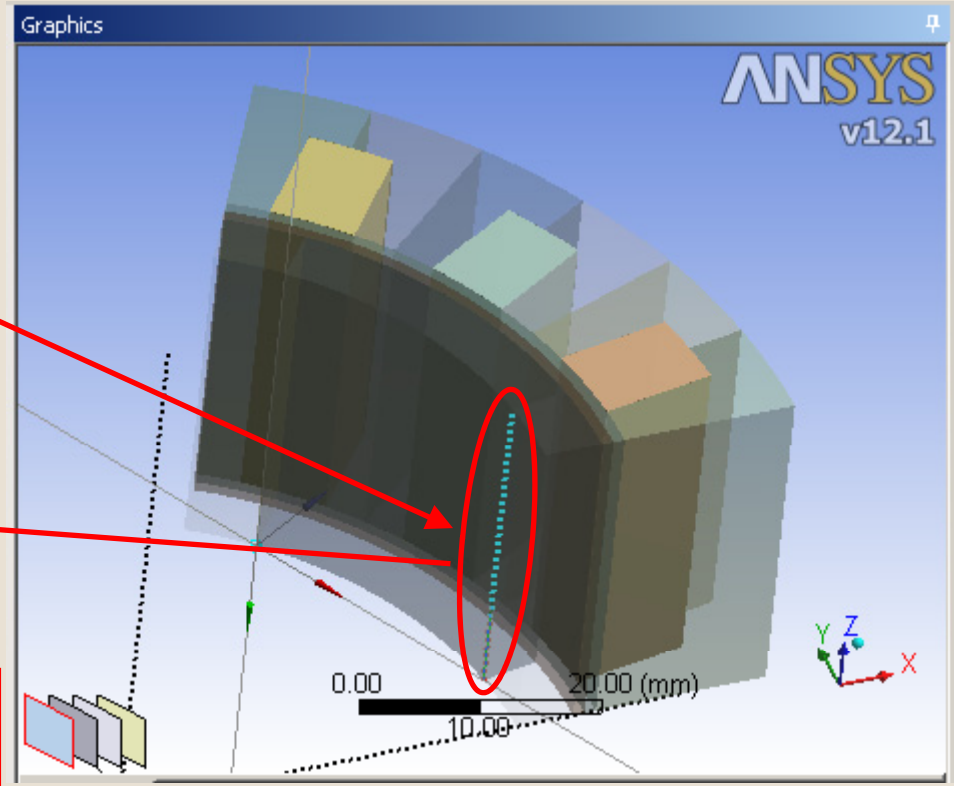

© 2010 ANSYS, Inc. All rights reserved.

Details View

| Details of Plane4 | |
|---|---|
| Plane | Plane4 |
| Type | From Point and Edge |
| Base Point | Vertex |
| Base Edge | Not selected |
| Transform 1 (RMB) | None |
| Reverse Normal/Z-Axis? | No |
| Flip XY-Axes? | No |
| Export Coordinate System? | No |

Details View

| Details of Plane4 | |
|---|---|
| Plane | Plane4 |
| Type | From Point and Edge |
| Base Point | Vertex |
| Base Edge | Apply Cancel |
| Transform 1 (RMB) | None |
| Reverse Normal/Z-Axis? | No |
| Flip XY-Axes? | No |
| Export Coordinate System? | No |

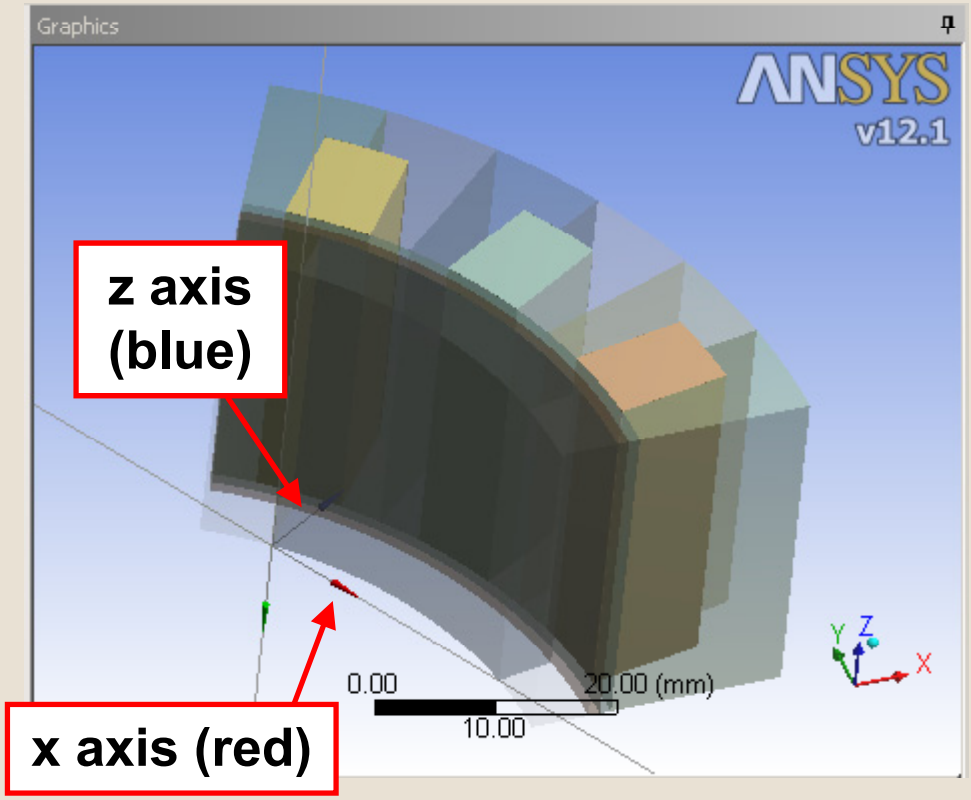



**Pick "Not Selected" adjacent to "Base Edge", pick edge in image, then "Apply"**

Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

Note that the x direction (red axis) points between the PM vertices. Since the polarization can only act in the x direction, we want to rotate the coordinate system so that the x axis points in what is presently the z direction (blue axis).

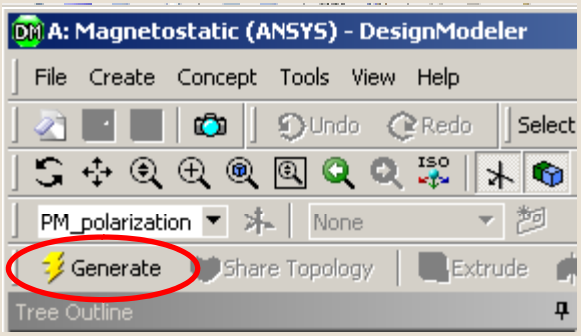



Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

Details View

| Details of PM_polarization | |
|---|---|
| Plane | PM_polarization |
| Type | From Point and Edge |
| Base Point | Vertex |
| Base Edge | 3D Edge |
| Transform 1 (RMB) | Rotate about Y |
| FD1, Value 1 | -90 ° |
| Transform 2 (RMB) | None |
| Reverse Normal/Z-Axis? | No |
| Flip XY-Axes? | No |
| Export Coordinate System? | Yes |



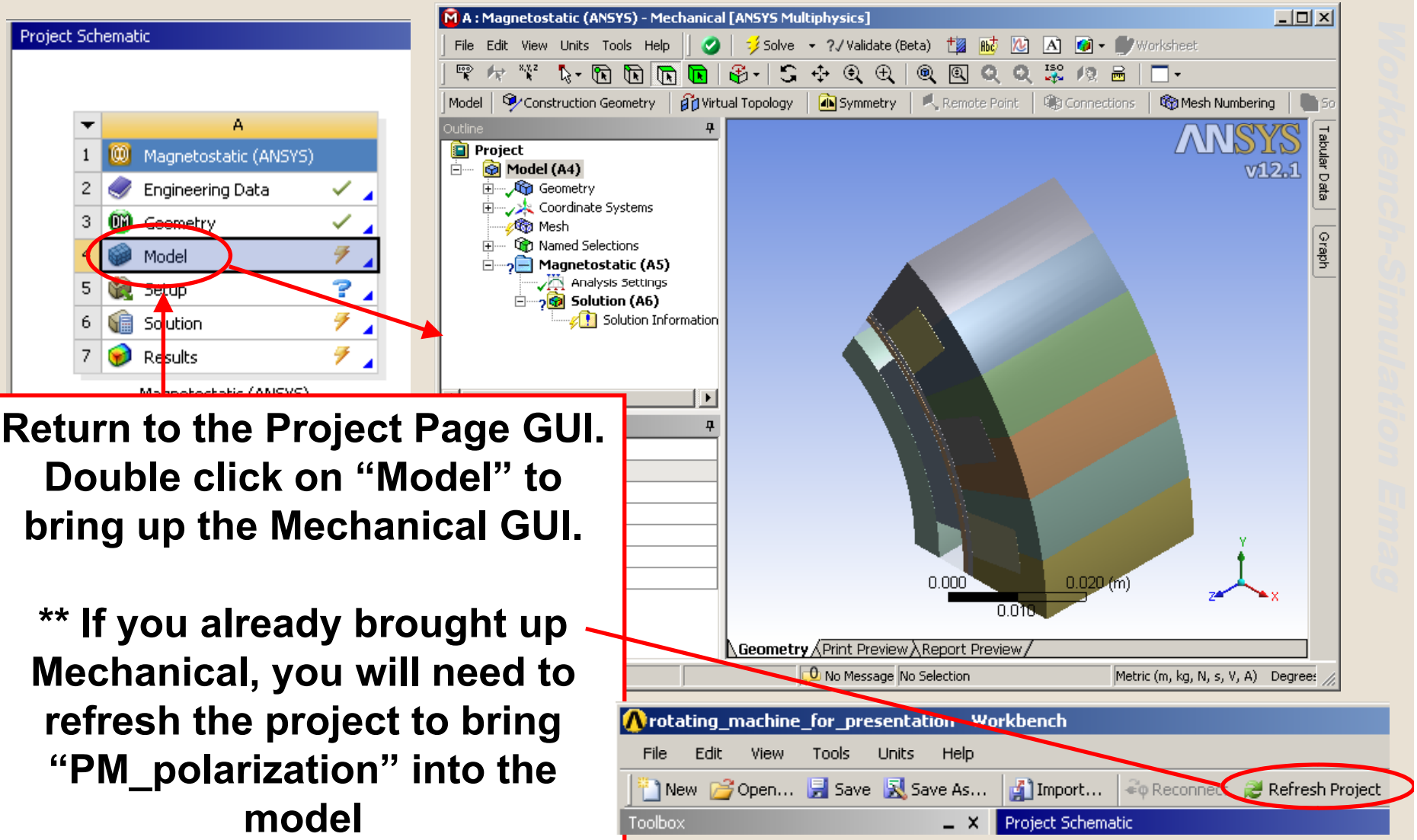

**To accomplish this rotation, set "Transform 1 (RMB)" to "Rotate about Y" and specify a -90º rotation. Change the name of the coordinate system to "PM_polarization".**

**IMPORTANT!**
**Next, set "Export Coordinate System?" to "Yes"**

**Now click "Generate"**

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




**Return to the Project Page GUI. Double click on "Model" to bring up the Mechanical GUI.**

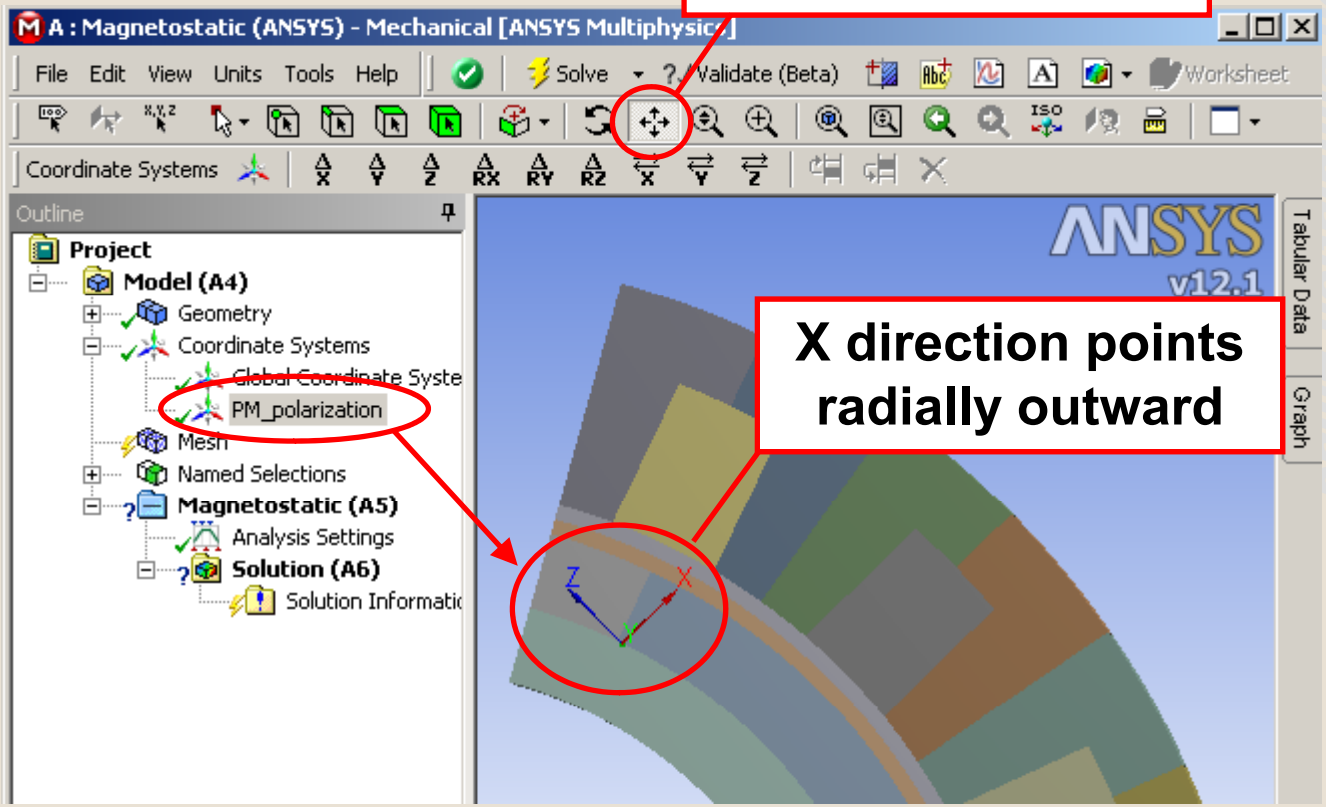
**** If you already brought up Mechanical, you will need to refresh the project to bring "PM_polarization" into the model**

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

Ensure that the "PM_polarization" coordinate system has been successfully imported into Mechanical

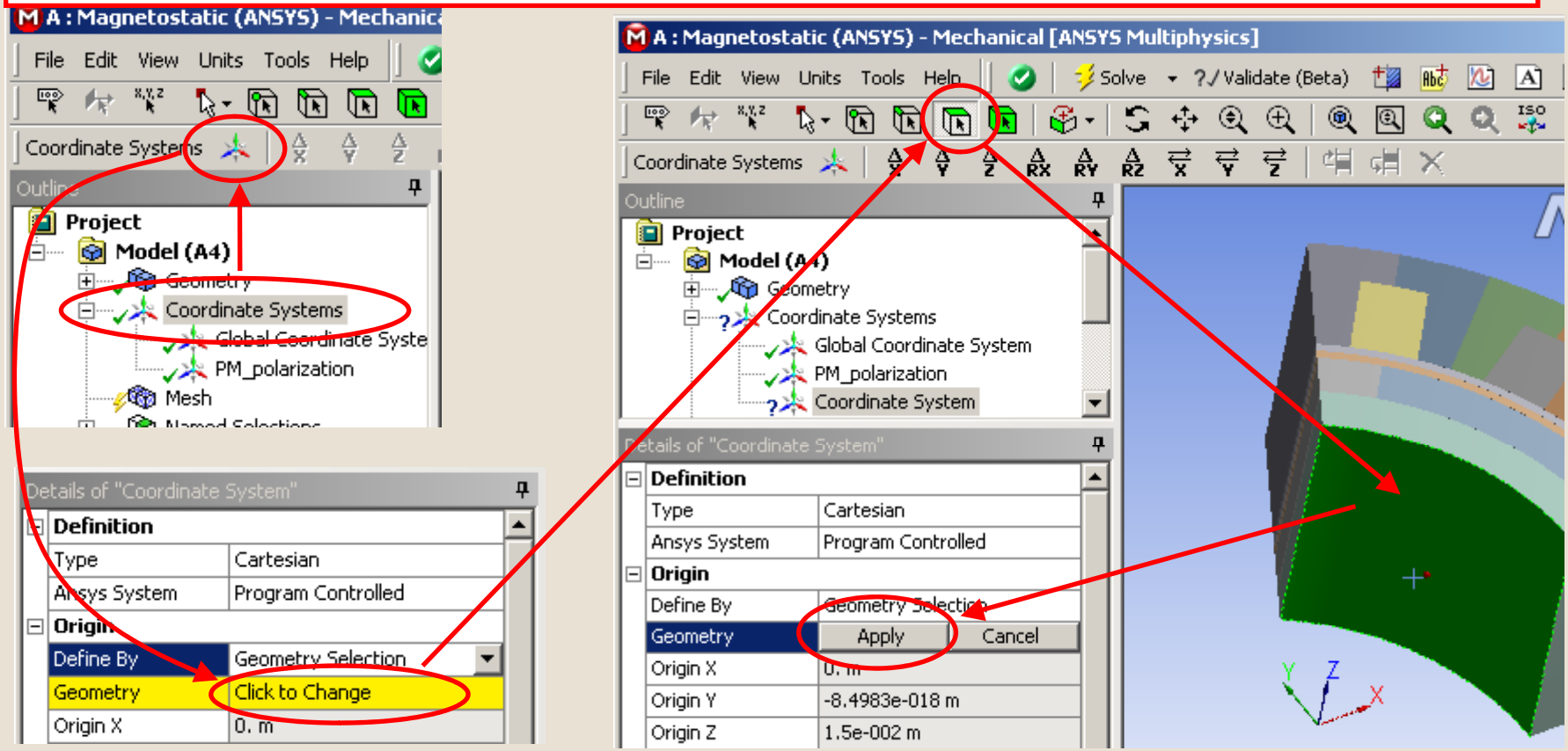
Convenient option for manipulating viewing direction





© 2010 ANSYS, Inc. All rights reserved.

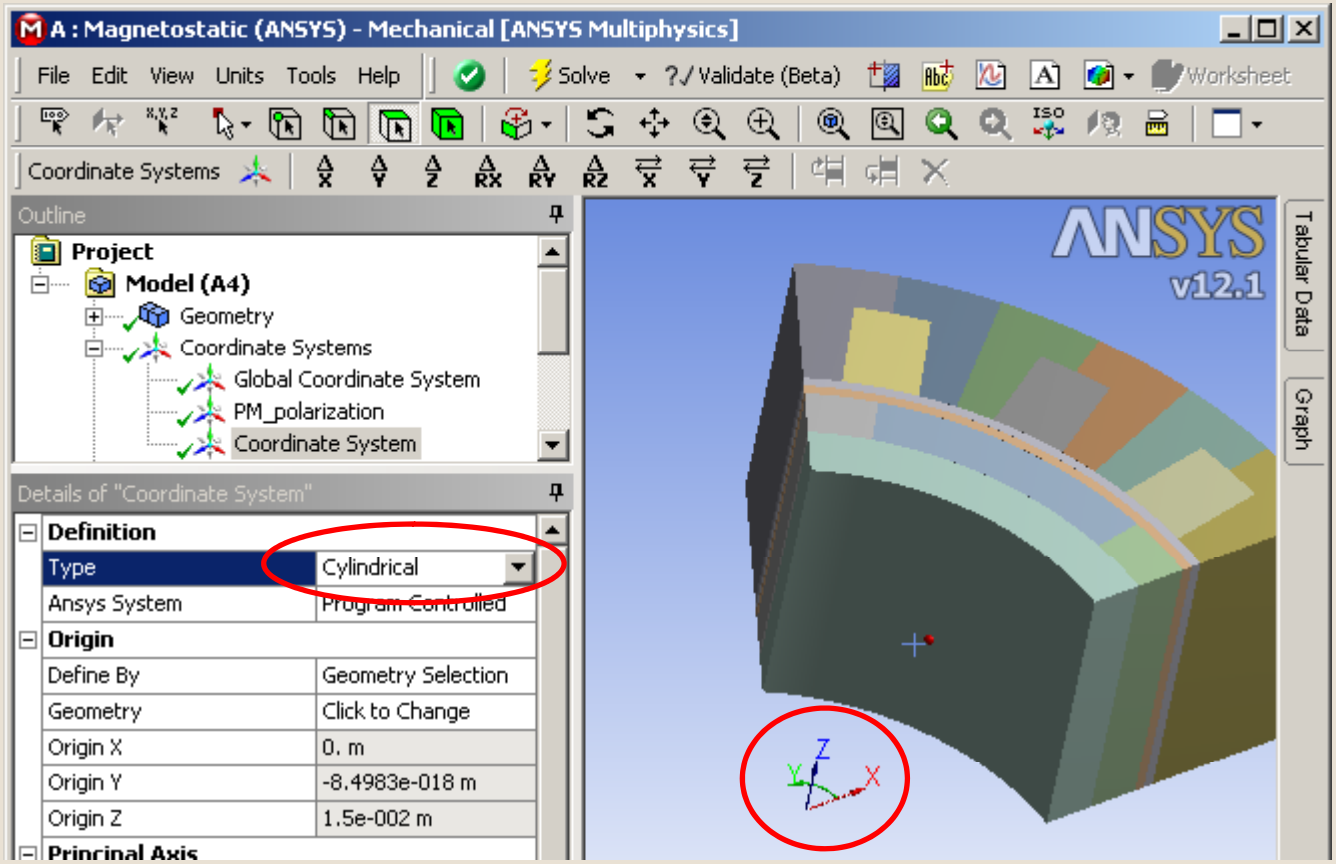
**To create a cylindrical coordinate system for the periodic condition, choose "Coordinate Systems" in the tree and click on the new coordinate system icon. Pick the inner cylindrical surface or the rotor iron as shown, then click Apply.**





© 2010 ANSYS, Inc. All rights reserved.

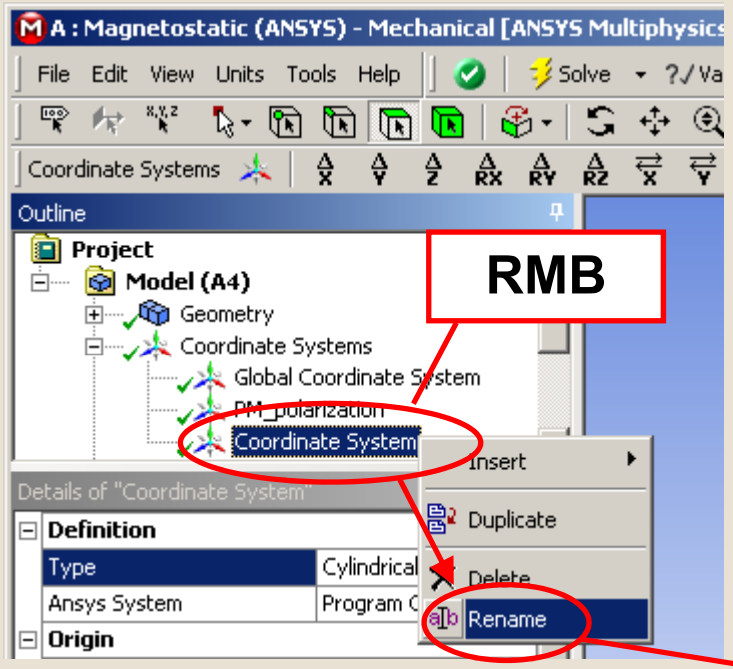
**The coordinate system will automatically be located at the center of curvature of the cylindrical surface. Set "Type" to "Cylindrical".**

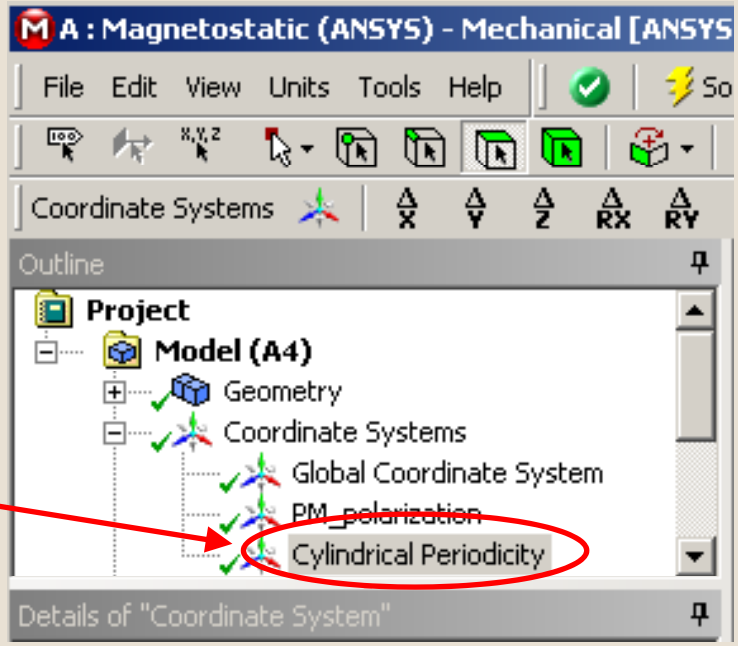



Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

# Rename the coordinate system to “Cylindrical Periodicity”




Workbench-Simulation Emag

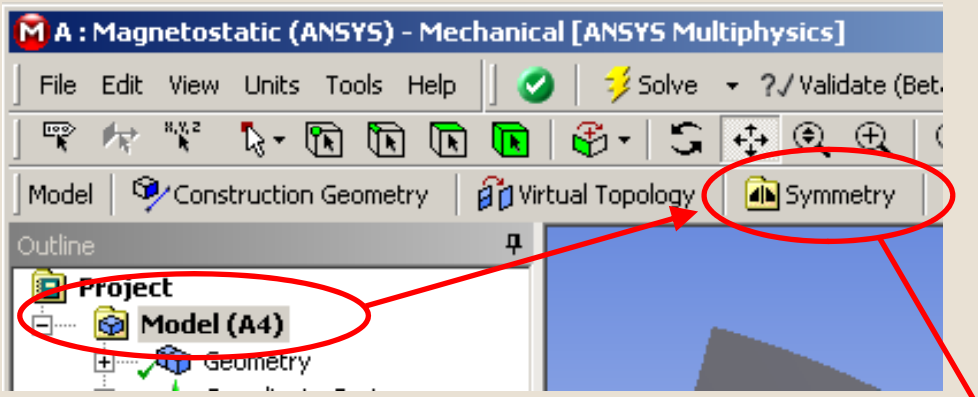

© 2010 ANSYS, Inc. All rights reserved.

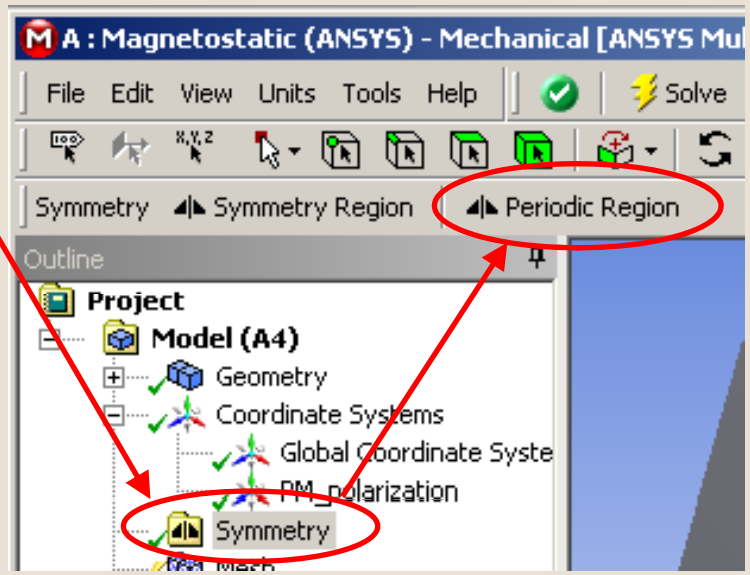
# Create Periodic Boundary Conditions

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

**Insert a symmetry condition object. Specify a periodic region symmetry condition.**

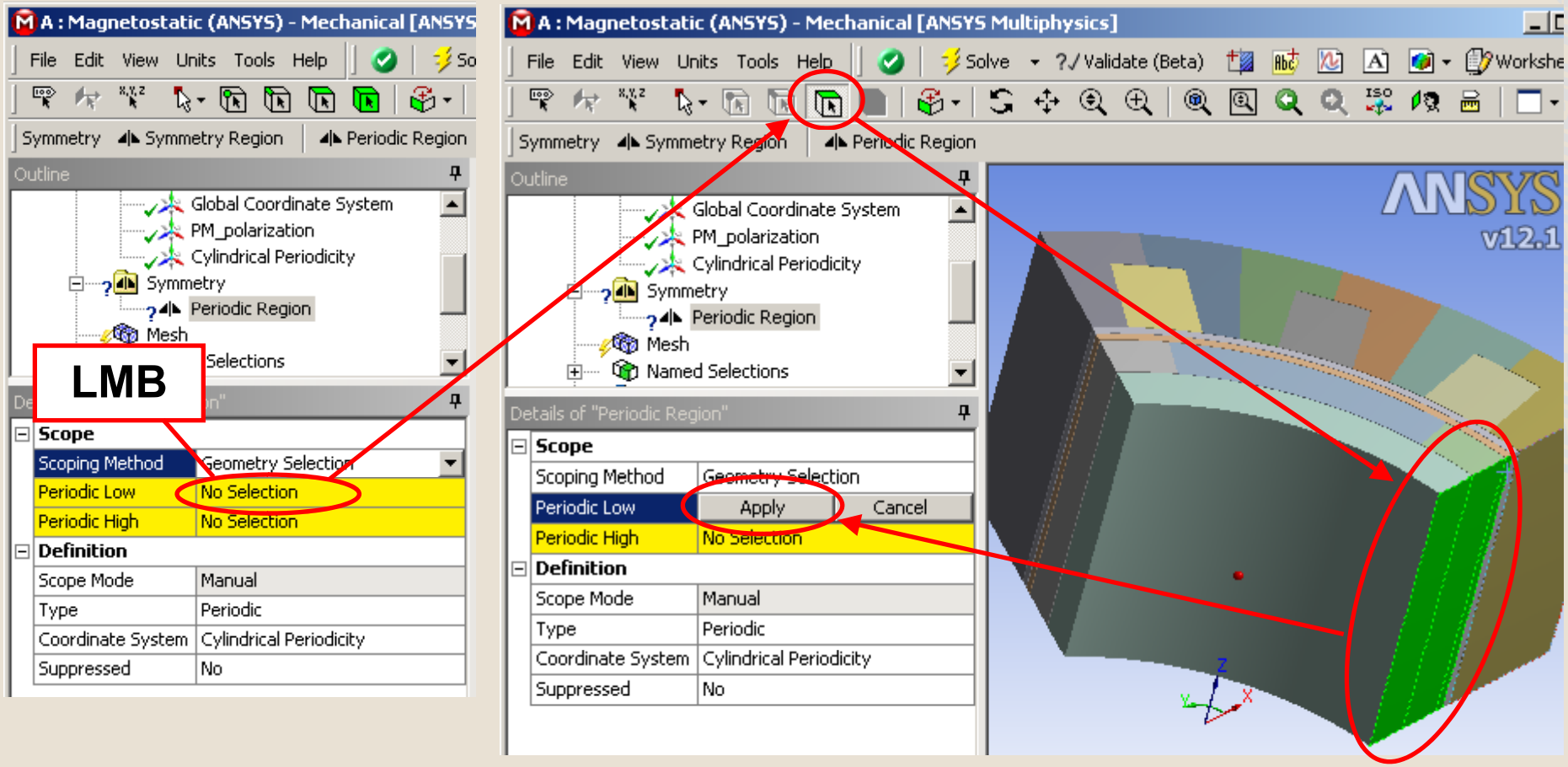




© 2010 ANSYS, Inc. All rights reserved.

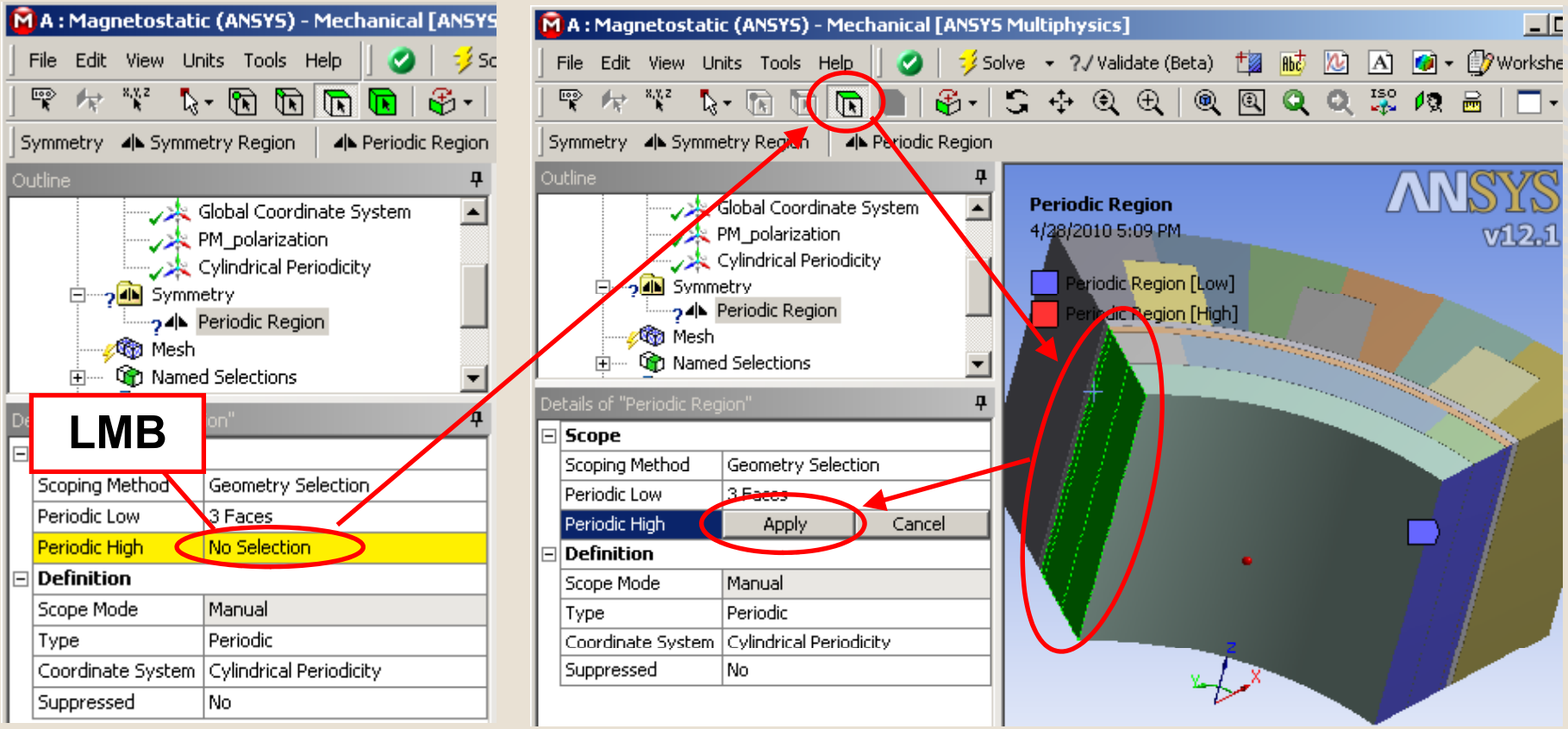
**For the "Periodic Low" surfaces, choose the rotor side periodic surfaces shown. Use Ctrl-LMB to select multiple surfaces.**





© 2010 ANSYS, Inc. All rights reserved.

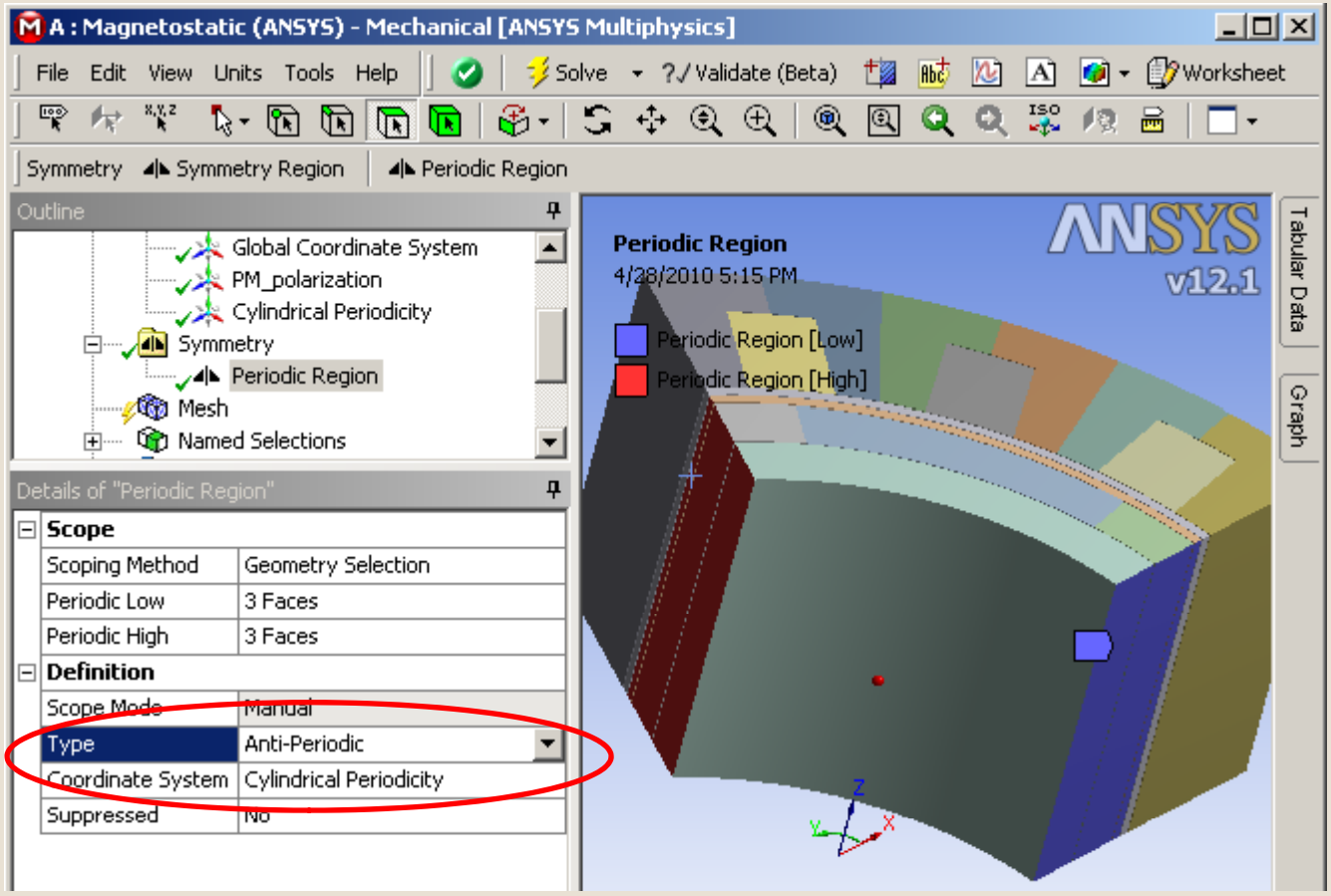
**For the "Periodic High" surfaces, choose the rotor side periodic surfaces shown. Use Ctrl-LMB to select multiple surfaces.**





© 2010 ANSYS, Inc. All rights reserved.

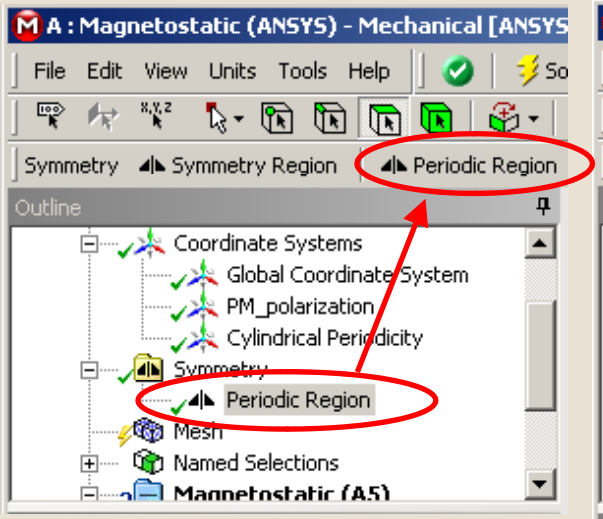
## Set “Type” to “Anti-Periodic” and “Coordinate System” to “Cylindrical Periodicity”

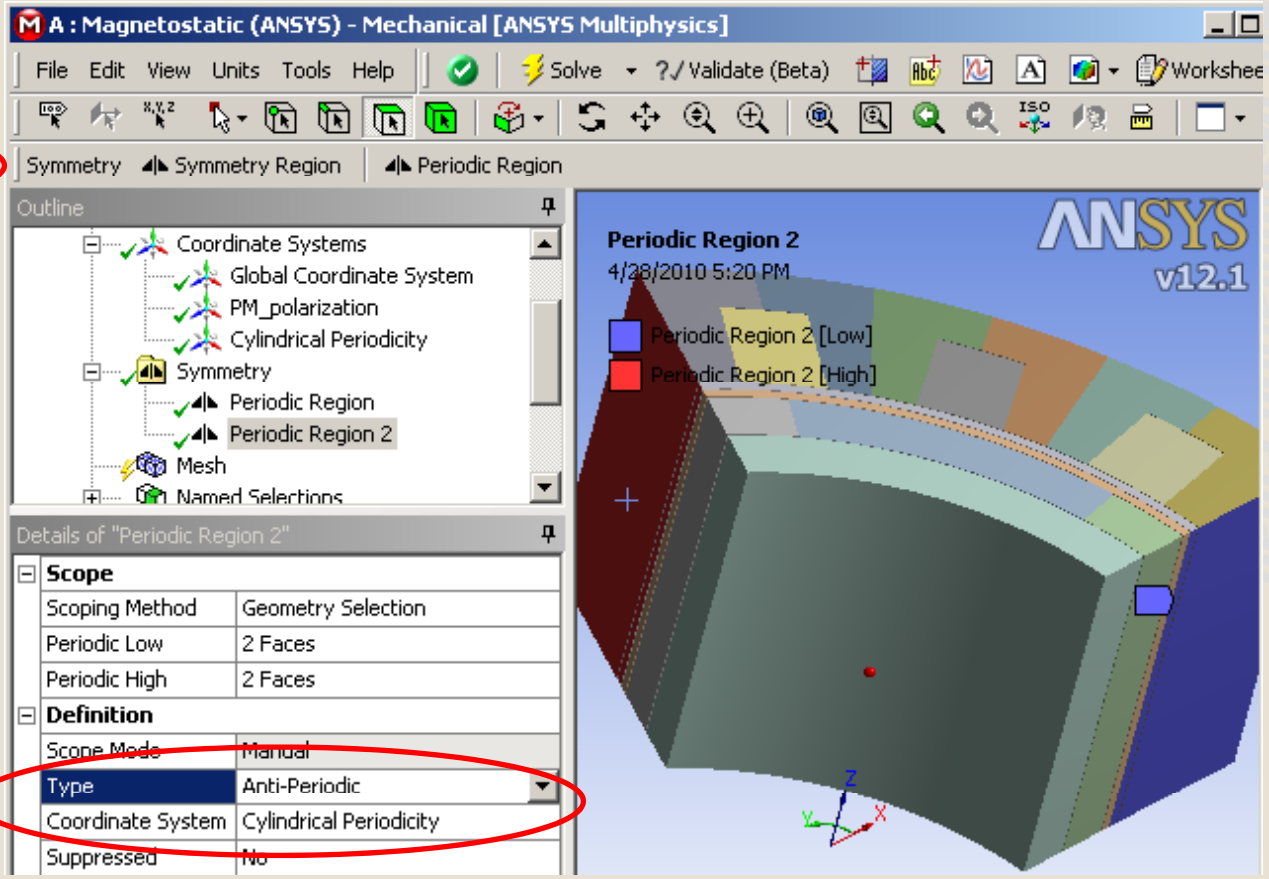



Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

**Insert another Periodic Region. Repeat the procedure to create periodic boundaries straddling the stator geometry.**





© 2010 ANSYS, Inc. All rights reserved.

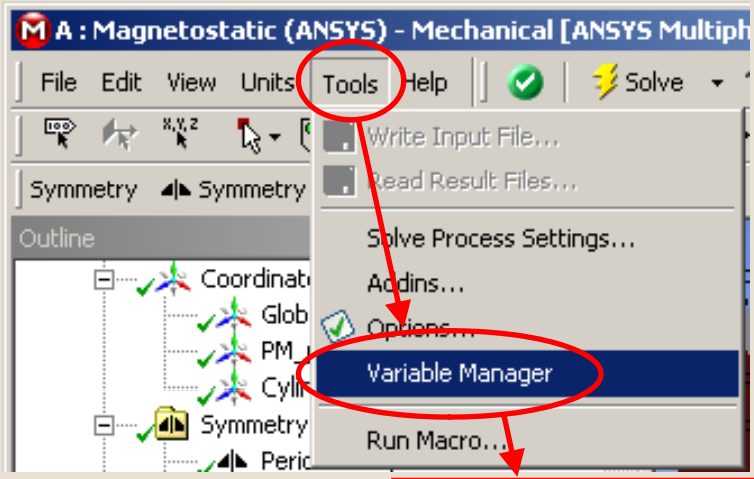
Workbench-Simulation Emag

# Magnetic Interface Wizard to Automate Set up of ARMINT Feature

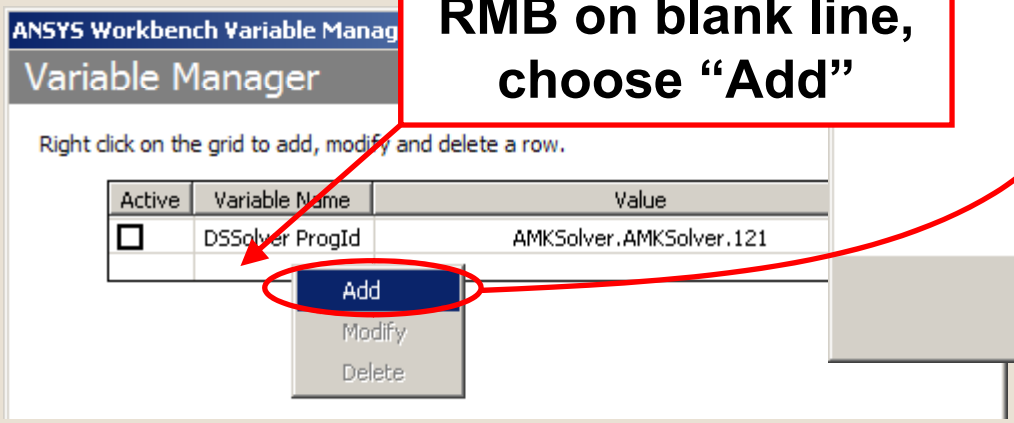
1) Set Key to Use New SOLID236 Electromagnetic Elements
2) Load the Magnetic Interface Wizard
3) Create the Mesh
4) Run the Magnetic Interface Wizard to set up Magnetic Interface Interpolation in Air Gap enabling motion effects

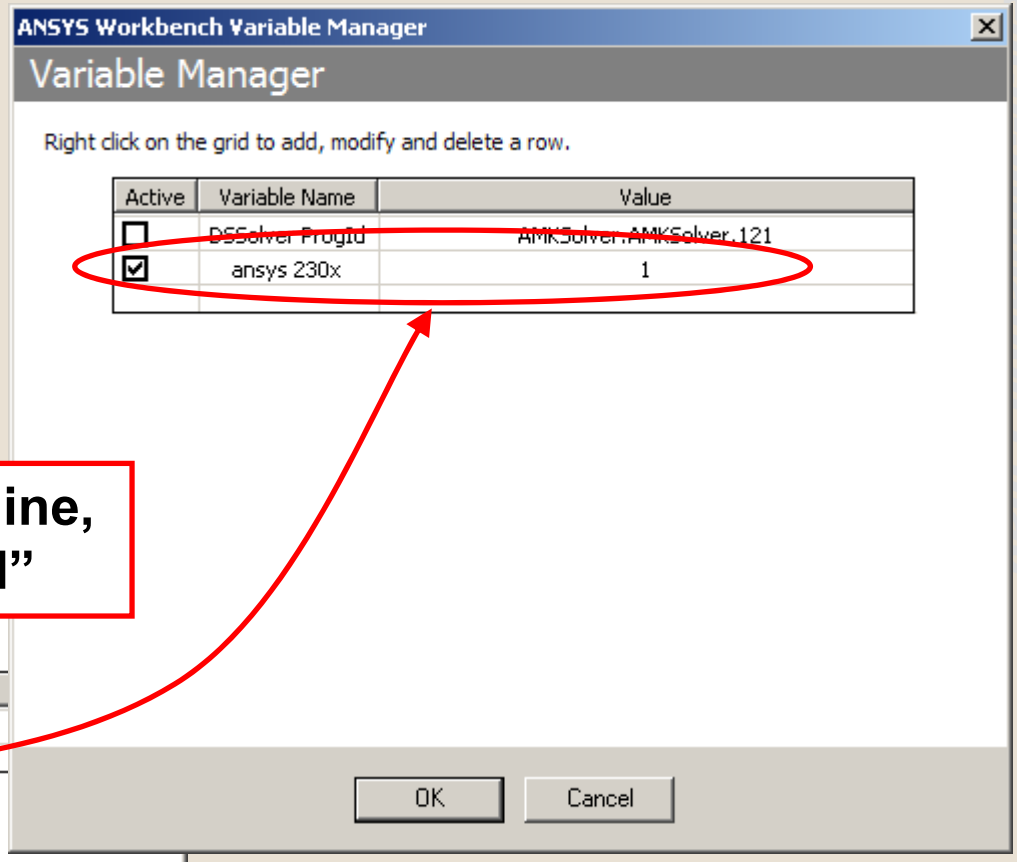

© 2010 ANSYS, Inc. All rights reserved.

**Define Variable Name "ansys 230x" in the variable manager, set "Value" to 1, and check the "Active" box.**




**RMB on blank line, choose "Add"**

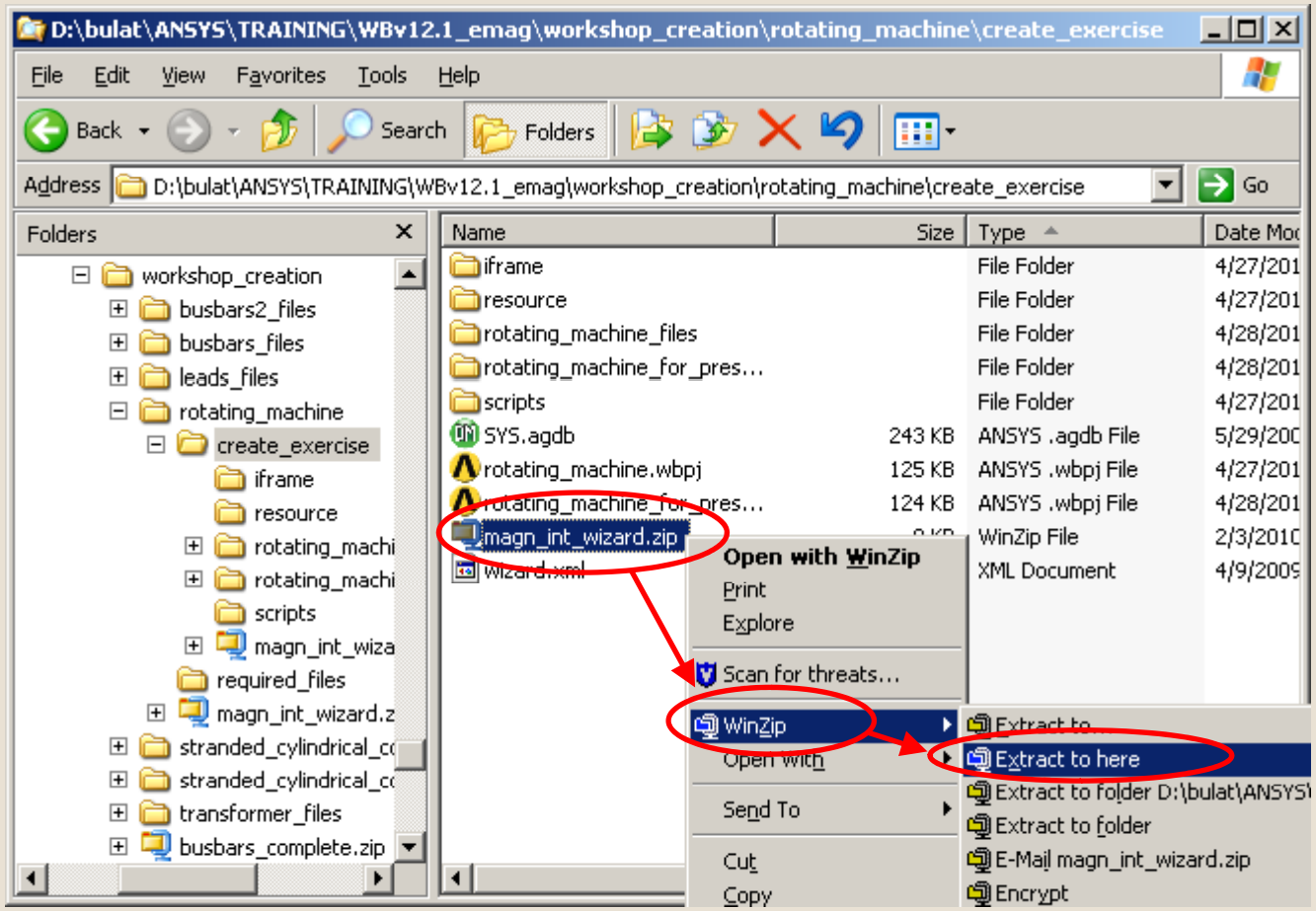




© 2010 ANSYS, Inc. All rights reserved.

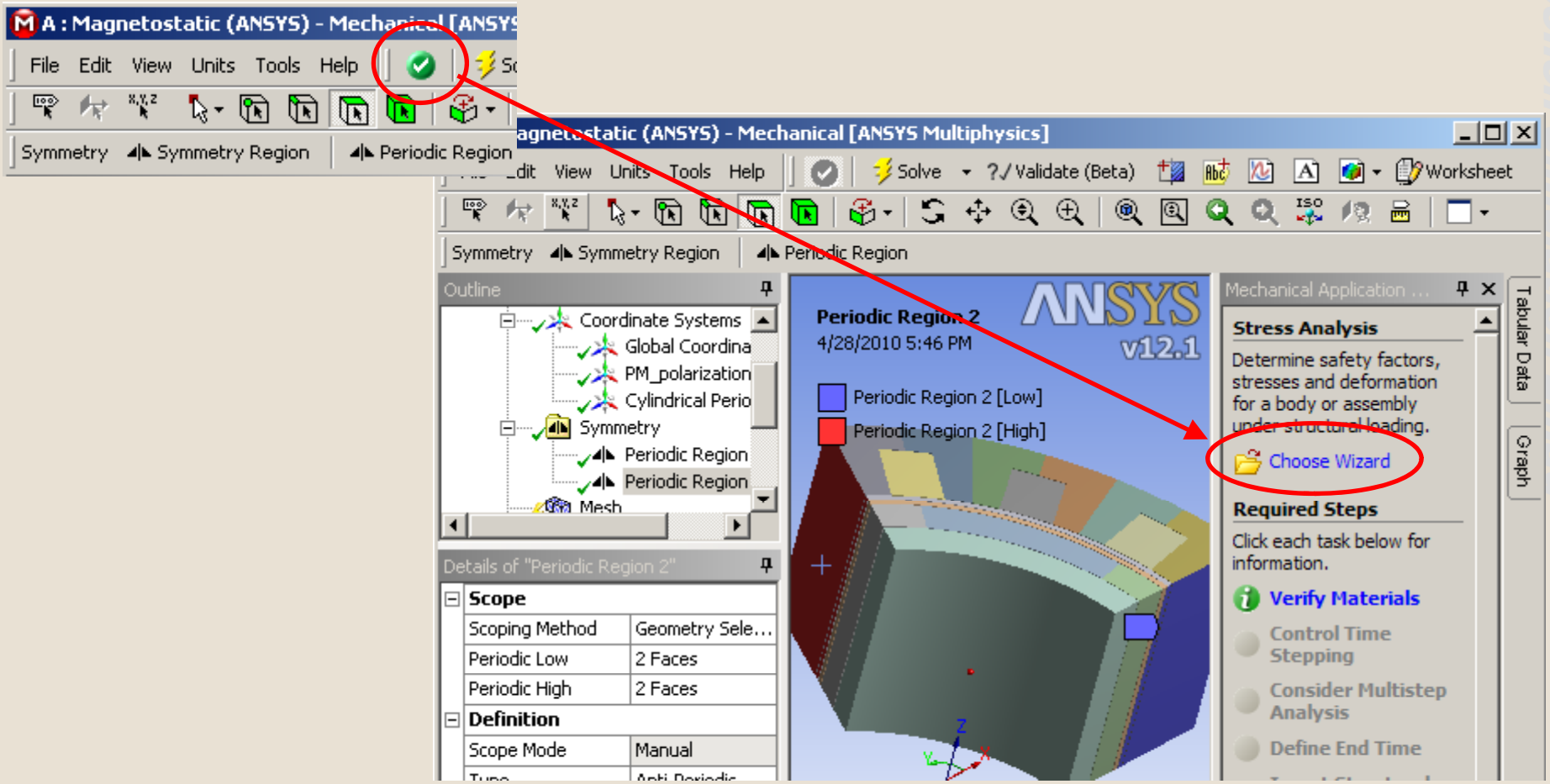
**Use Windows Explorer to unzip the contents of magn_int_wizard.zip to the directory in which your project is being created/saved.**





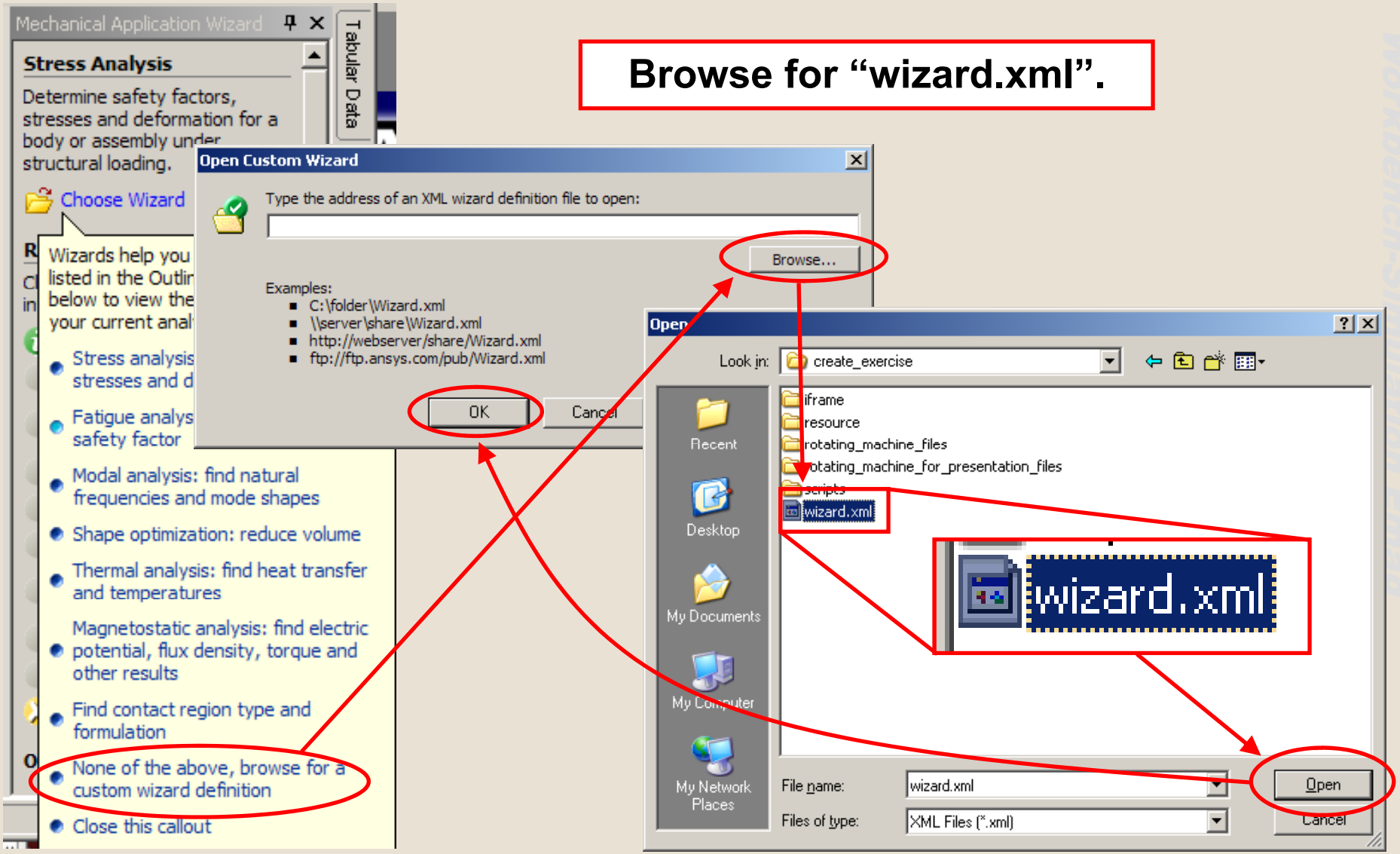
© 2010 ANSYS, Inc. All rights reserved.

**Bring up the Mechanical Application Wizard and click on "Choose Wizard".**





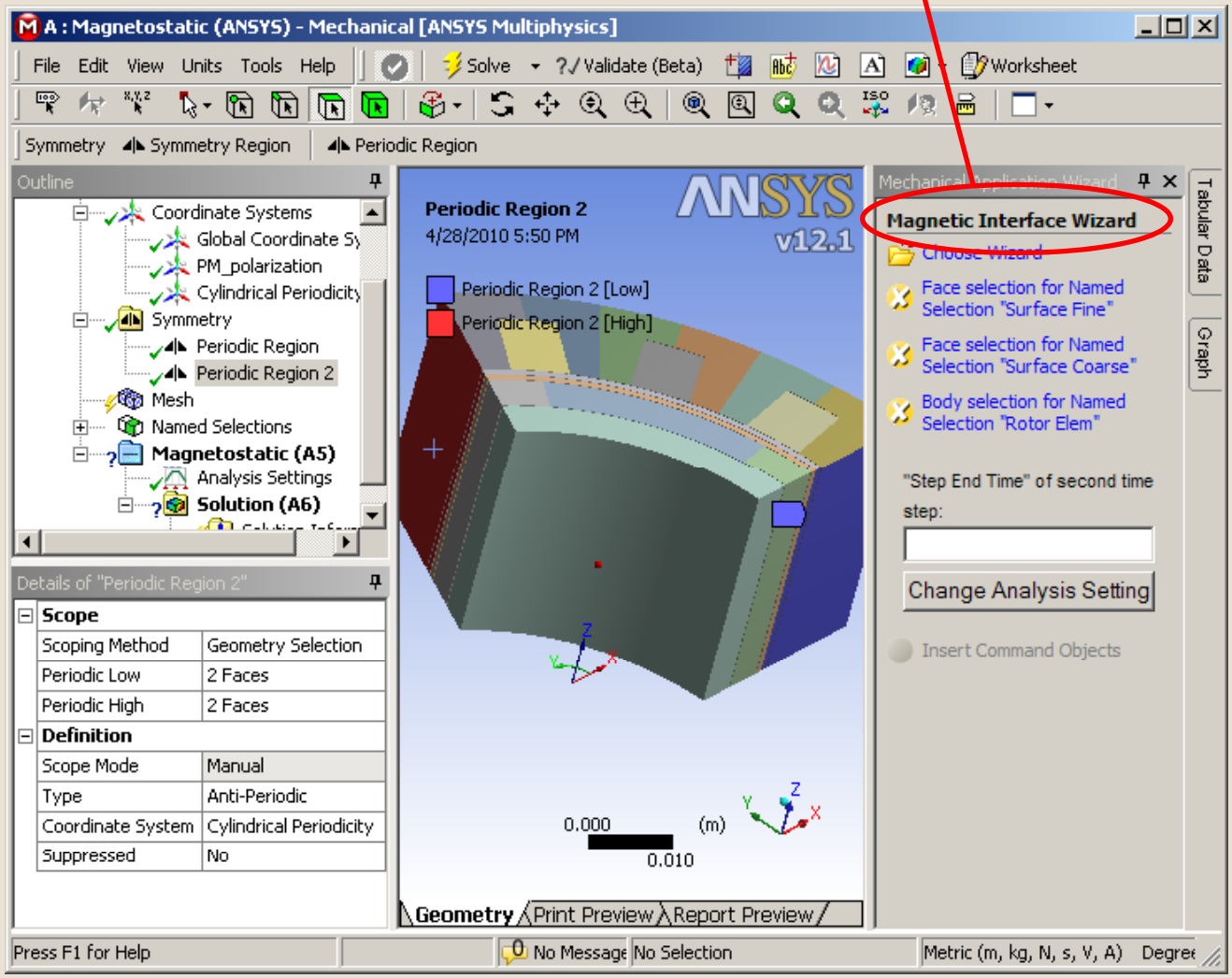
© 2010 ANSYS, Inc. All rights reserved.





© 2010 ANSYS, Inc. All rights reserved.

## Magnetic Interface Wizard Comes up



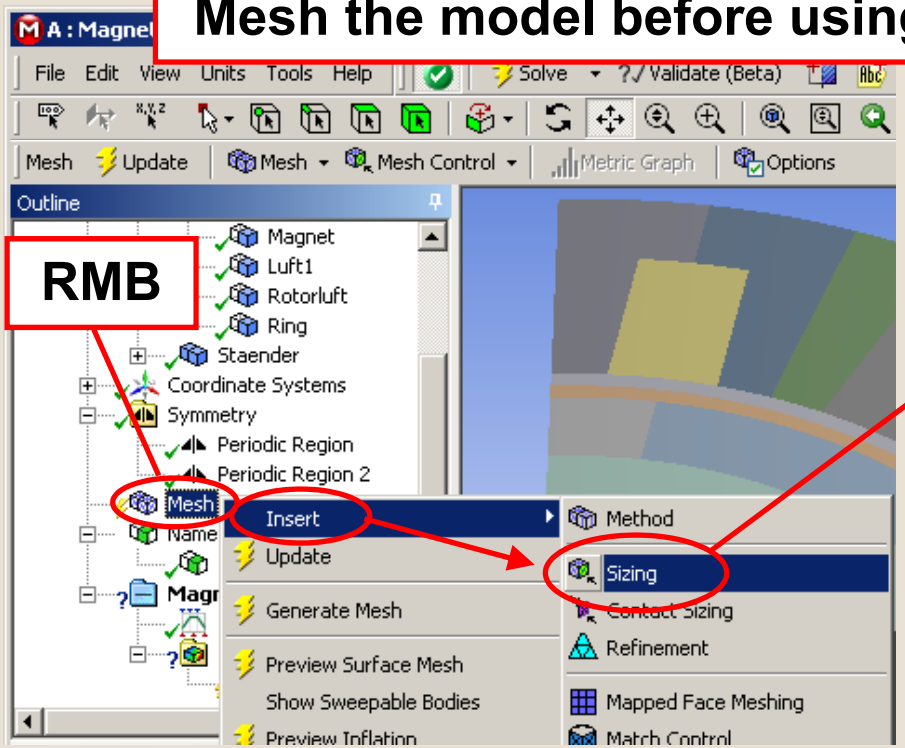


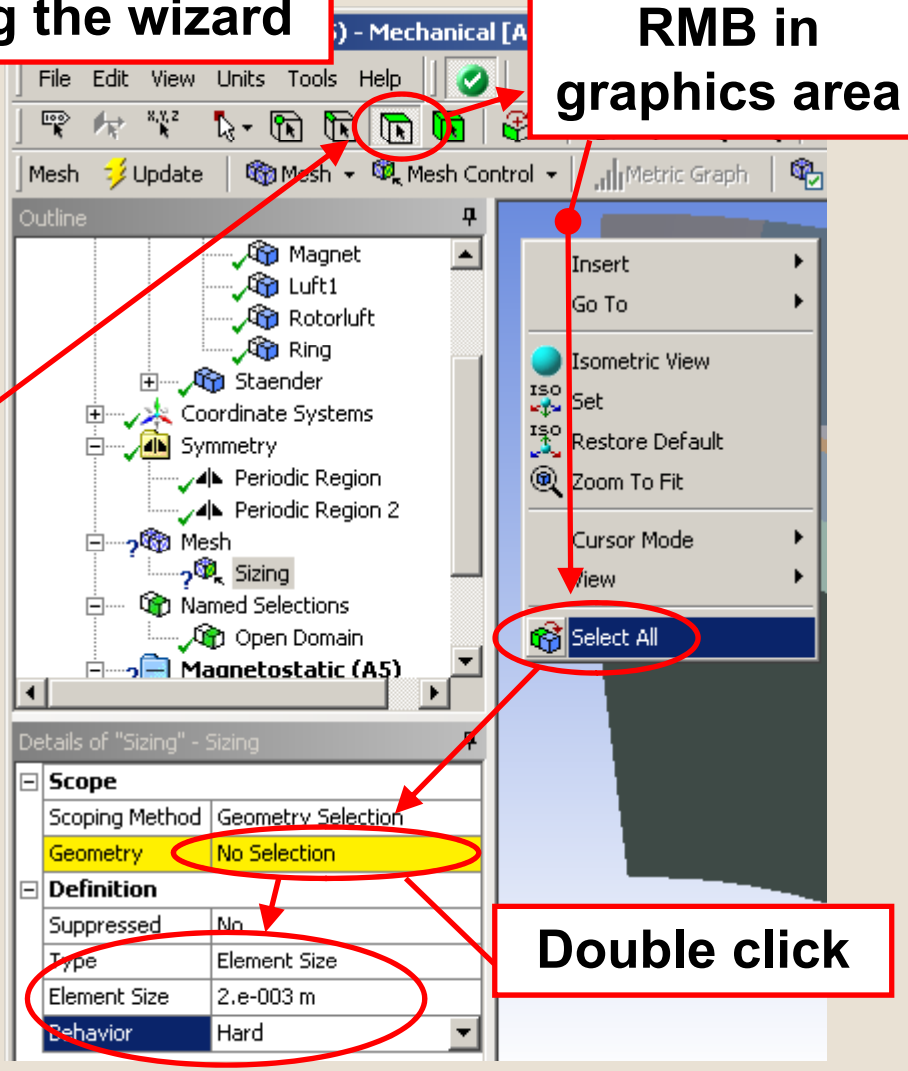
© 2010 ANSYS, Inc. All rights reserved.

Mesh the model before using the wizard

RMB in graphics area

RMB



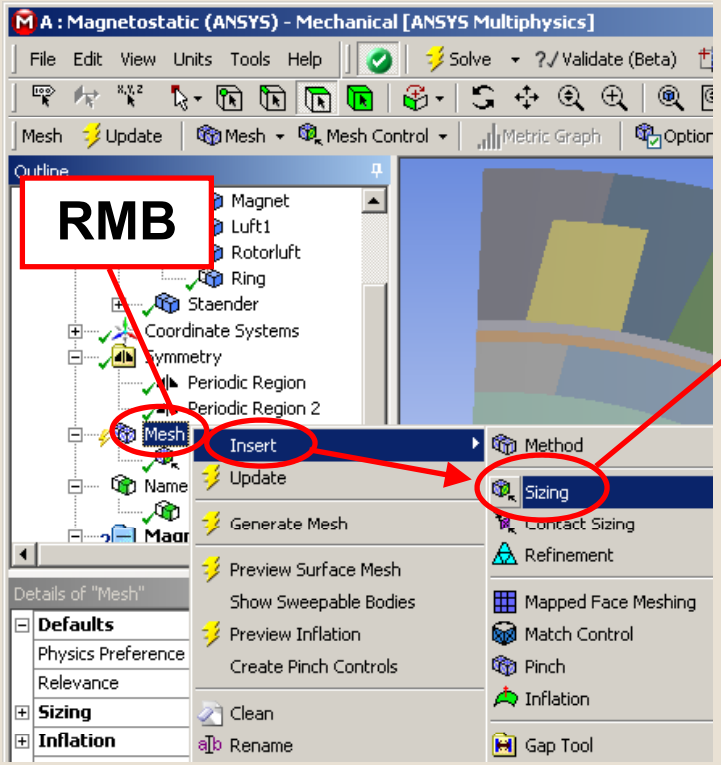

Set element size on all surfaces to 0.002 m. Set behavior to “Hard”.

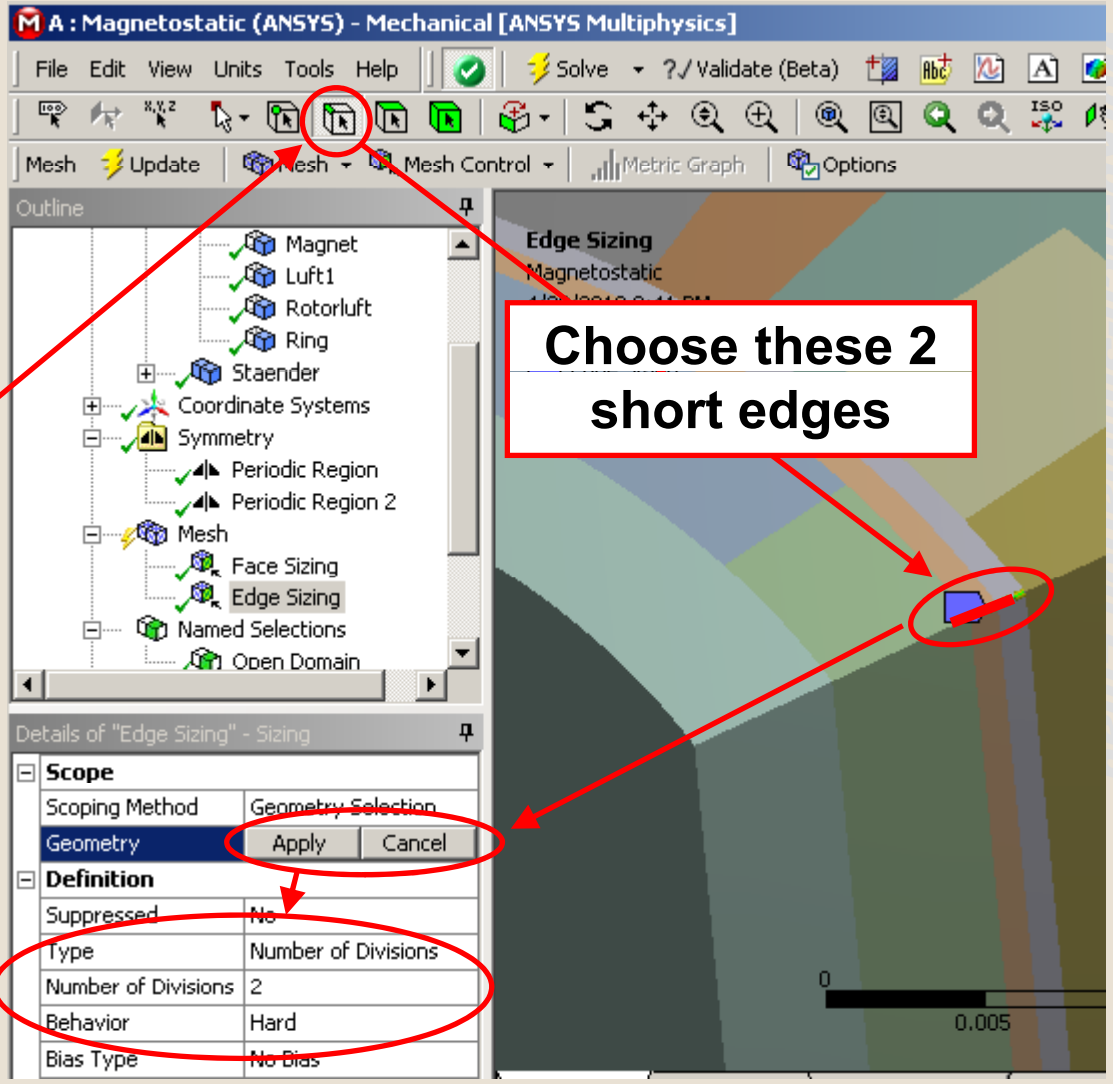
Double click


© 2010 ANSYS, Inc. All rights reserved.

# Electromagnetic Analysis

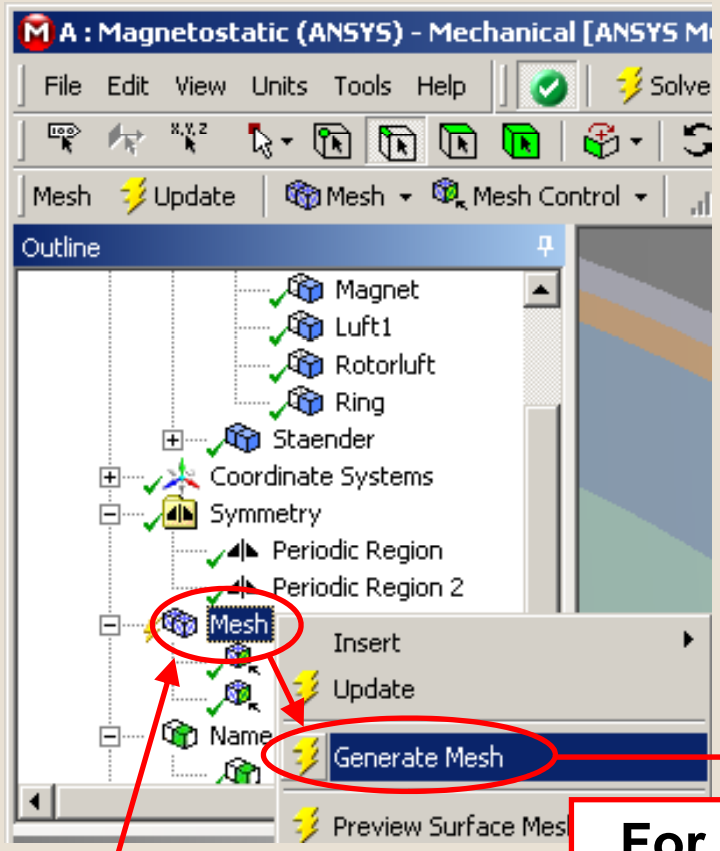




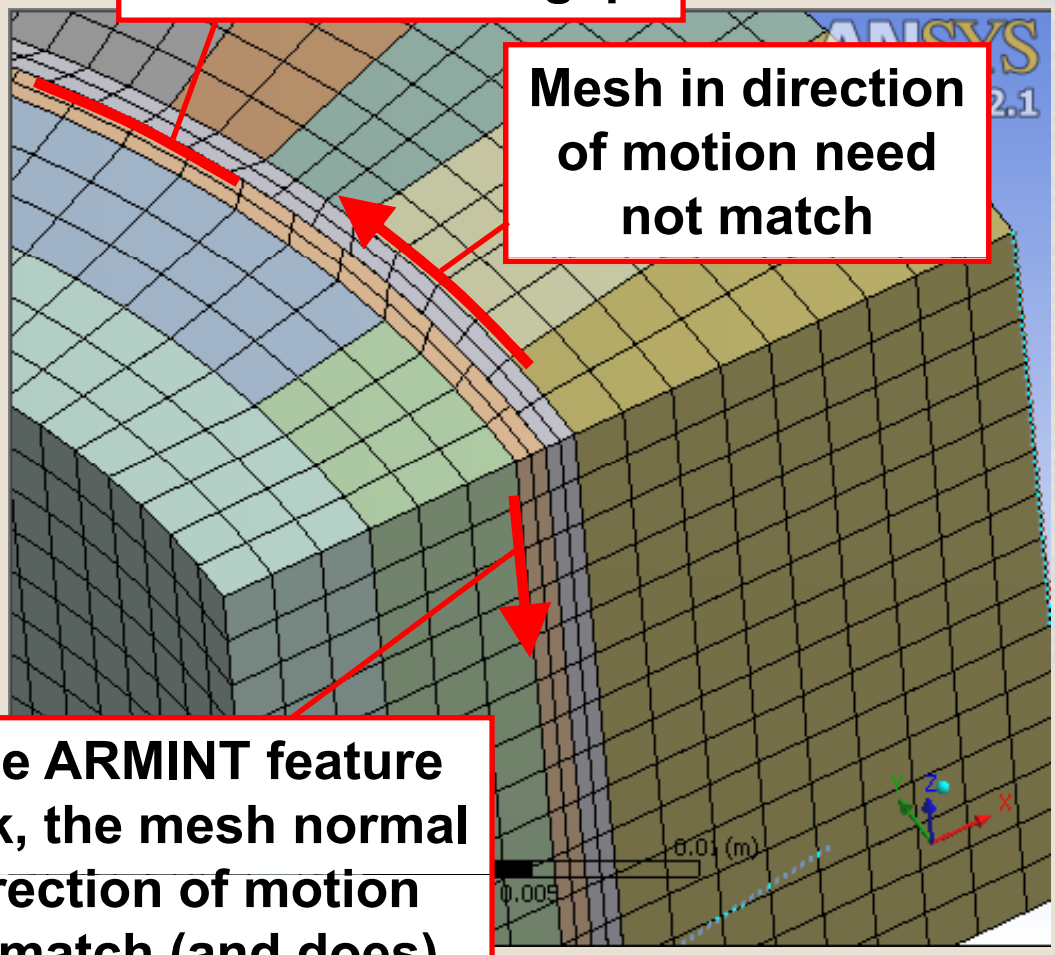




**Set number of divisions on 2 short edges along air gap to 2. Set behavior to "Hard".**


© 2010 ANSYS, Inc. All rights reserved.

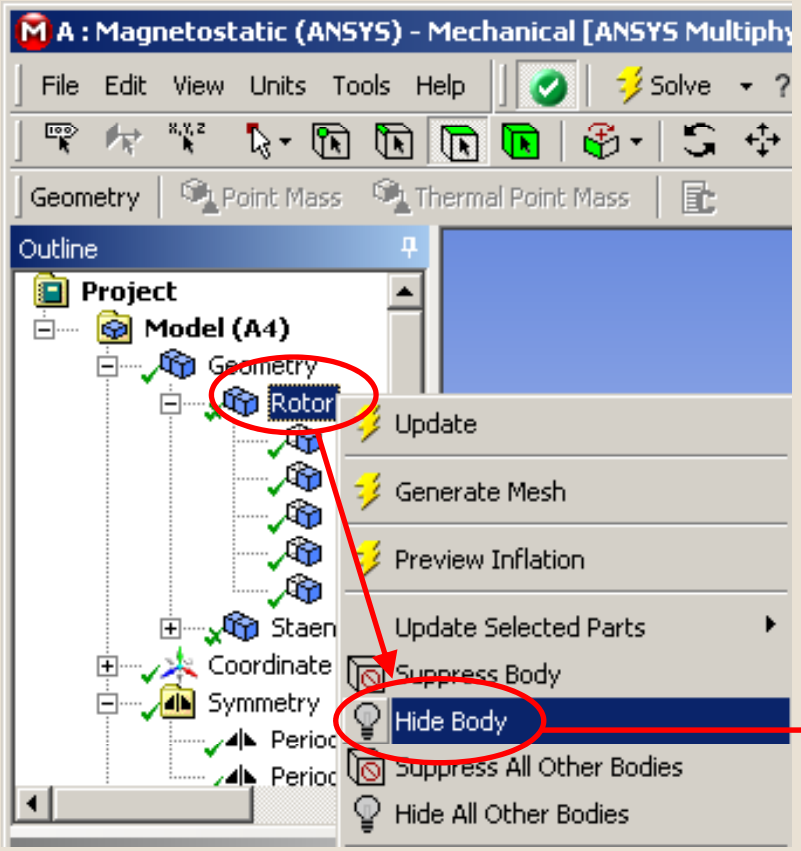



Generate the mesh

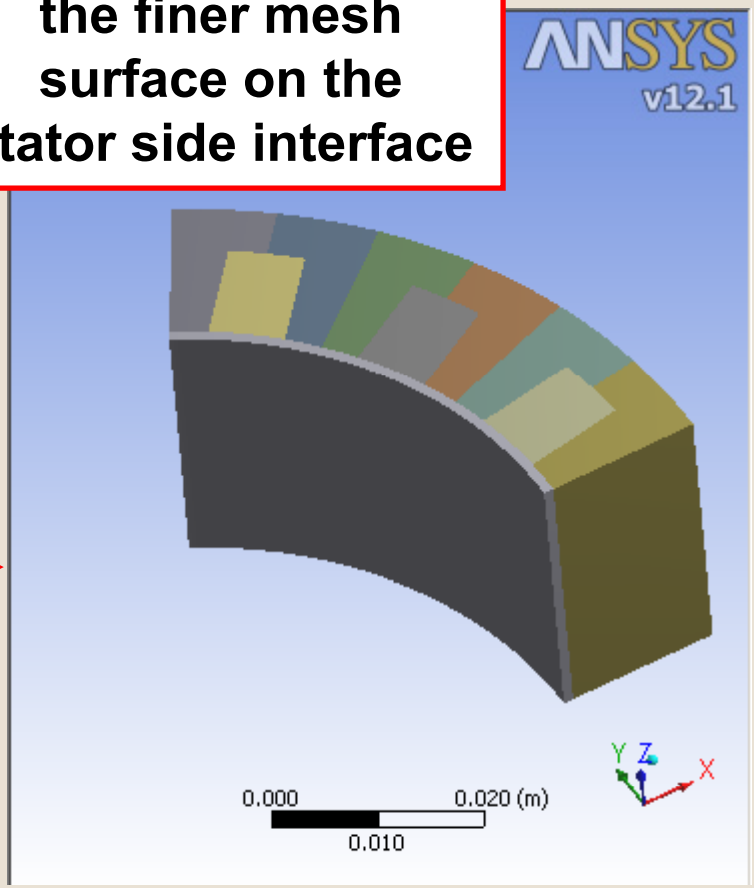



For the ARMINT feature to work, the mesh normal to direction of motion must match (and does)


© 2010 ANSYS, Inc. All rights reserved.



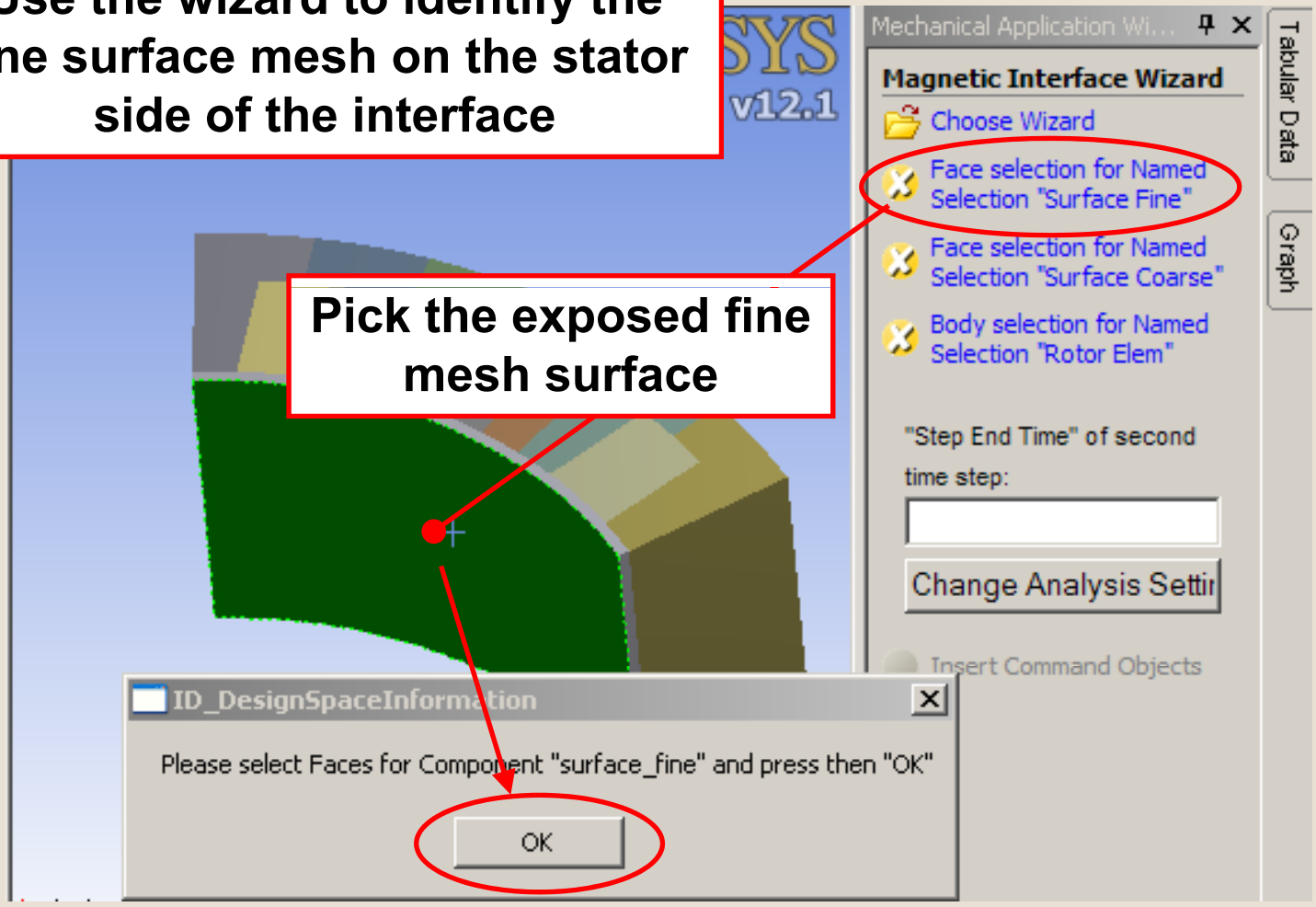

**Hide the rotor bodies to expose the finer mesh surface on the stator side interface**




Workbench Simulation Emag

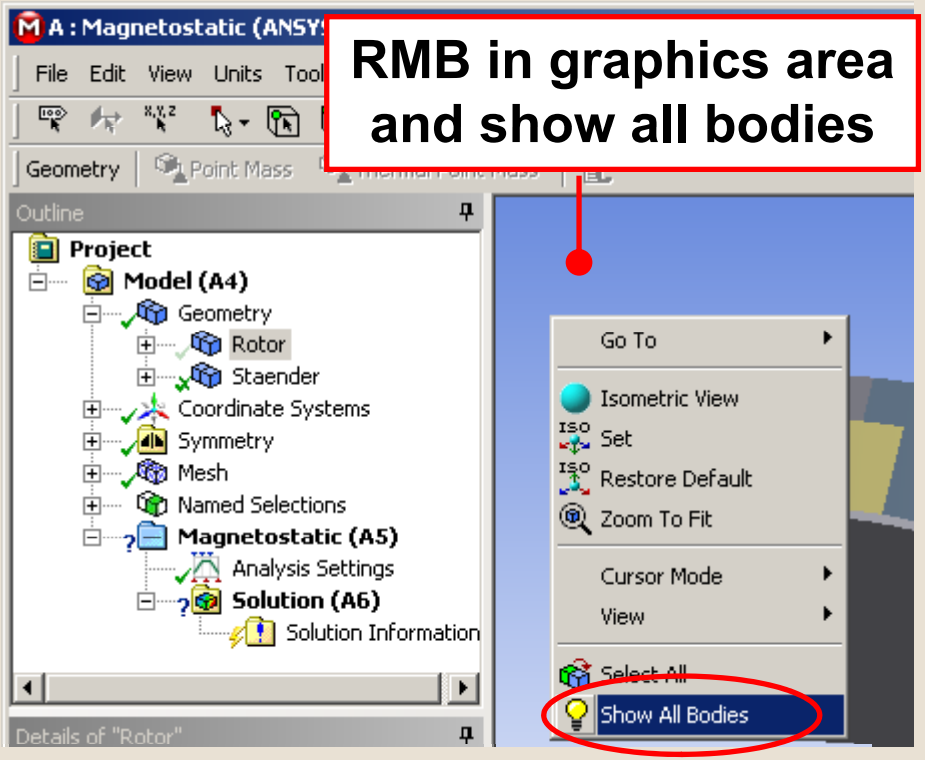

© 2010 ANSYS, Inc. All rights reserved.

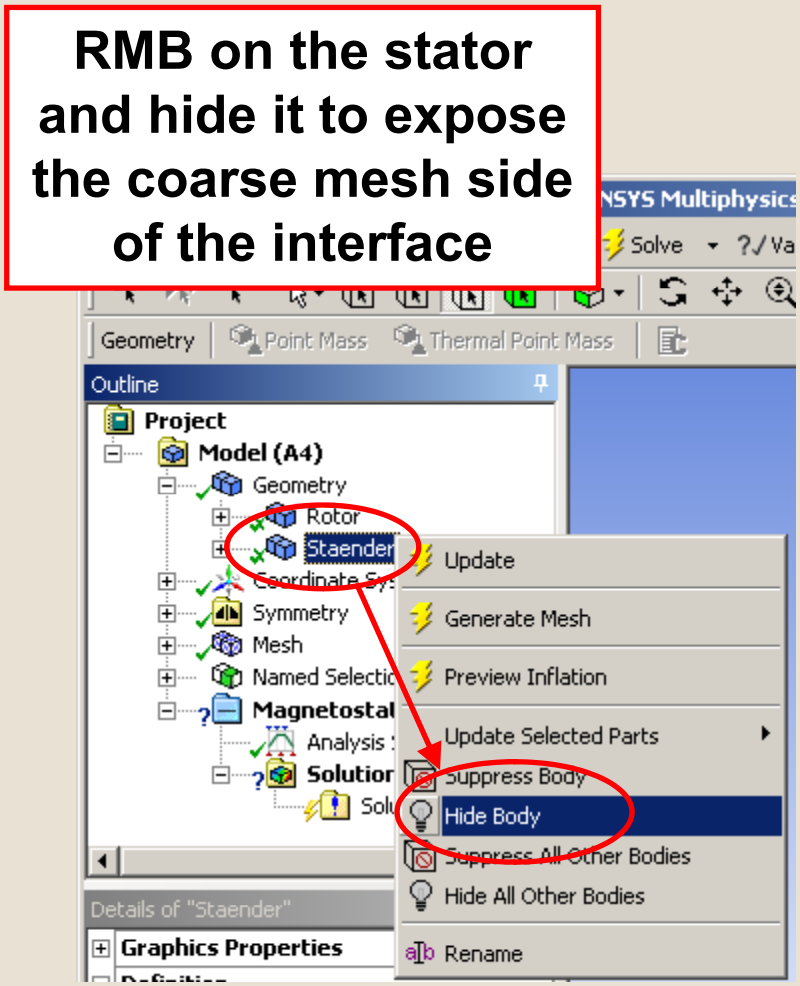
Use the wizard to identify the fine surface mesh on the stator side of the interface





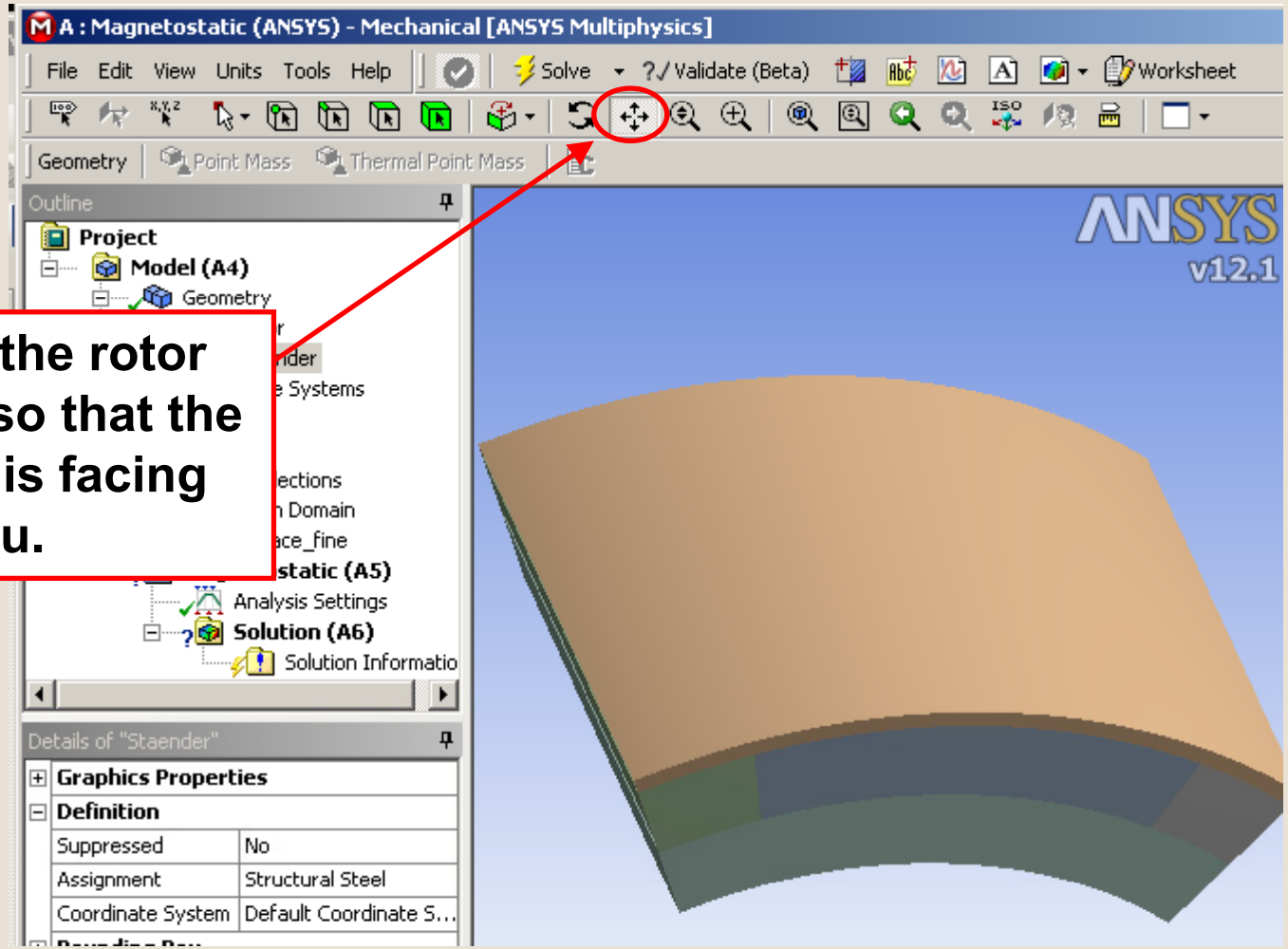
© 2010 ANSYS, Inc. All rights reserved.




Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.



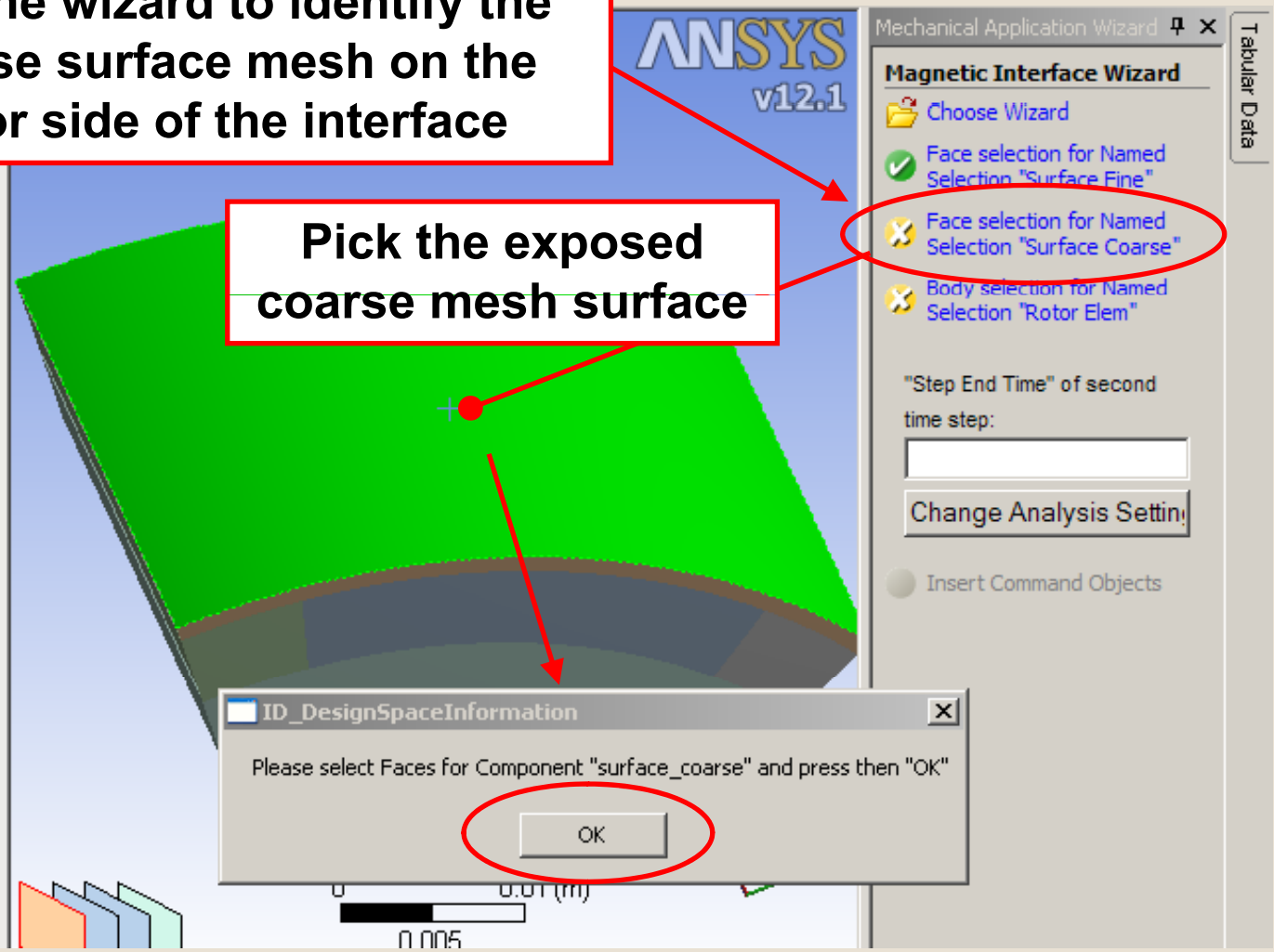

**Reorient the rotor assembly so that the interface is facing you.**

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

Use the wizard to identify the coarse surface mesh on the rotor side of the interface

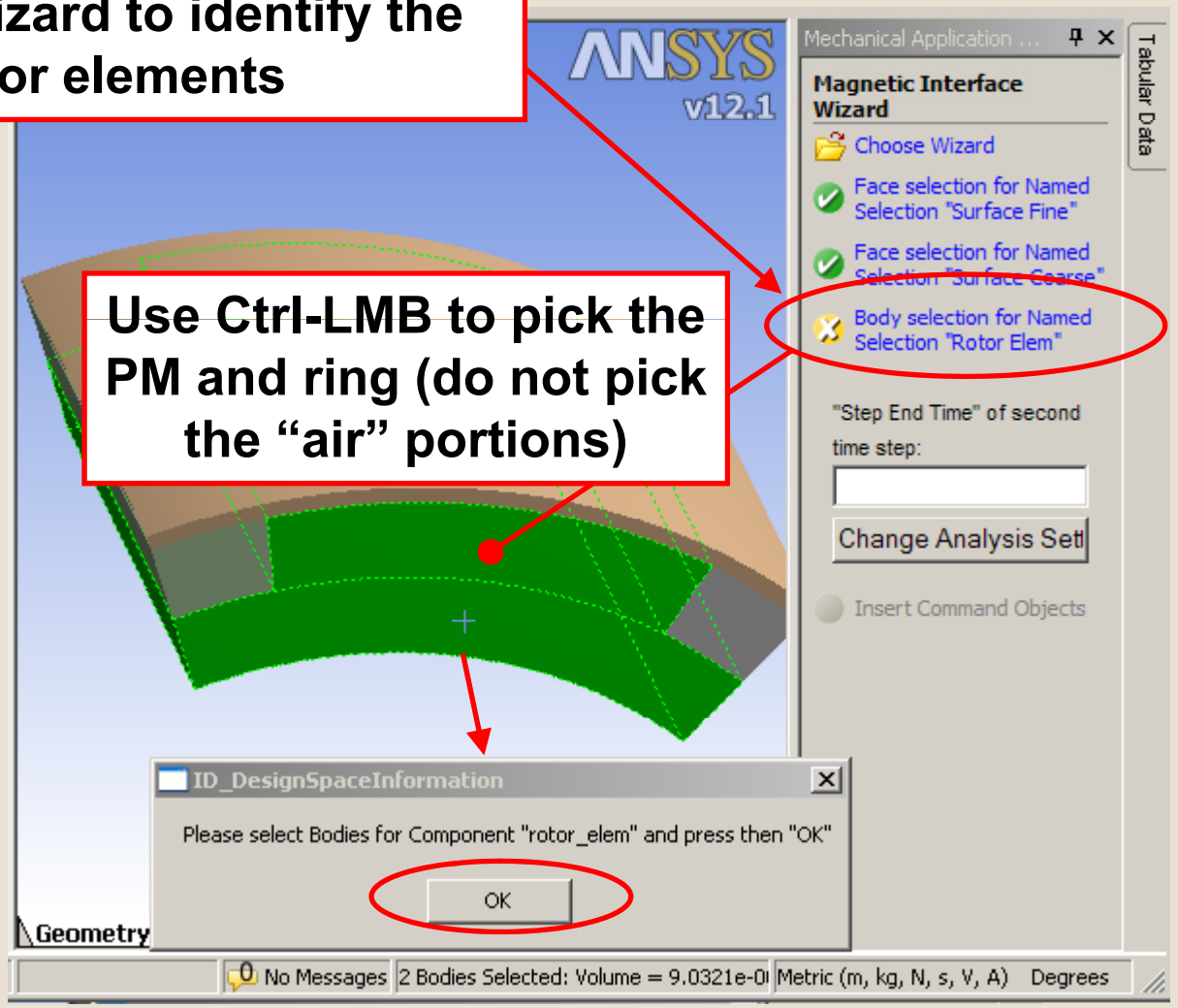
Pick the exposed coarse mesh surface




Workbench-Simulation Emag

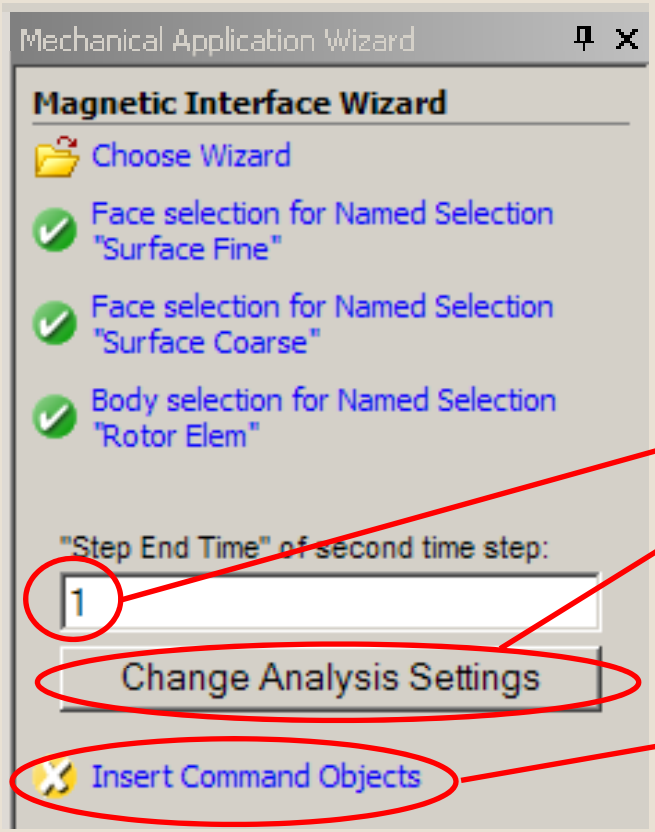

© 2010 ANSYS, Inc. All rights reserved.

Use the wizard to identify the rotor elements




Workbench-Simulation Emag

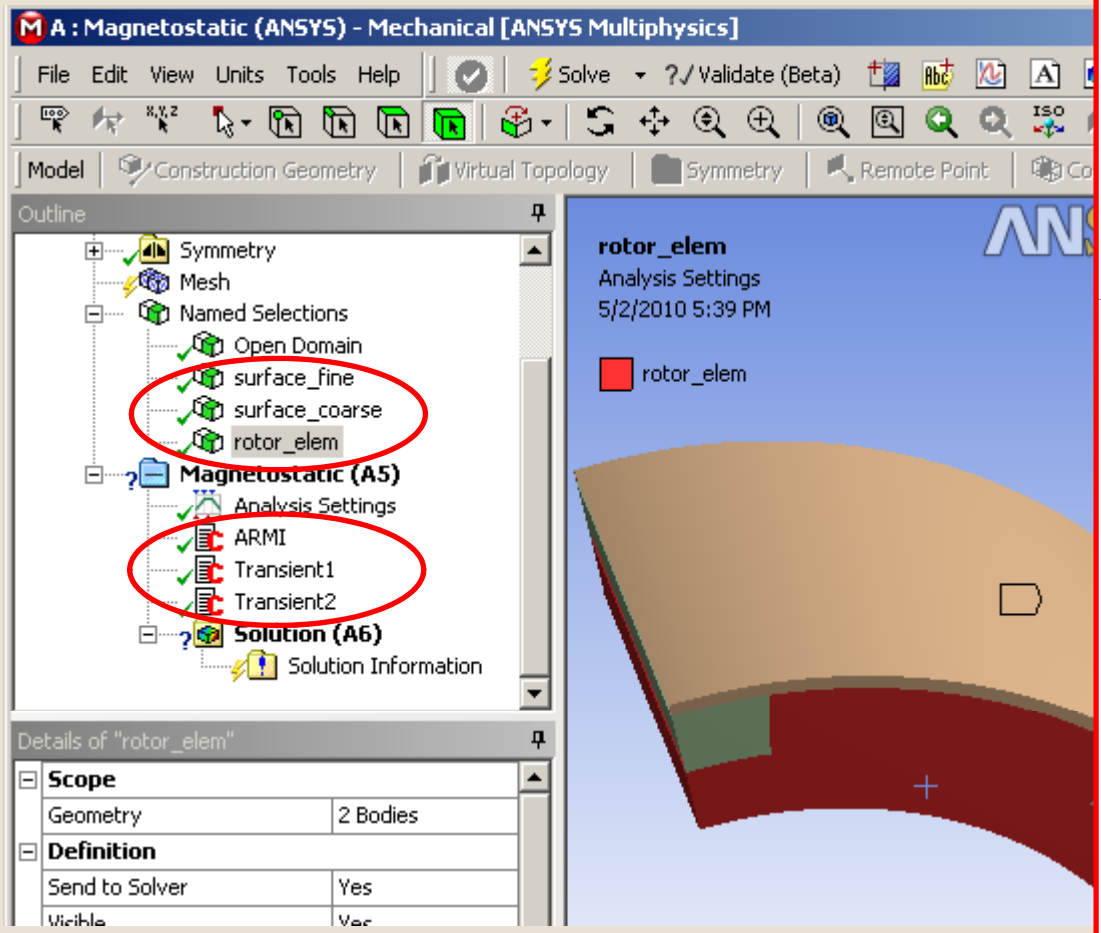

© 2010 ANSYS, Inc. All rights reserved.




**Enter "Step End Time" = 1 second, click "Change Analysis Settings" and "Insert Command Objects"**

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




**Note Named Selections created by the wizard:**

1) surface_fine
2) surface_coarse
3) rotor_elem

**Note the command objects created by the wizard:**

1) ARMI
2) Transient1
3) Transient2

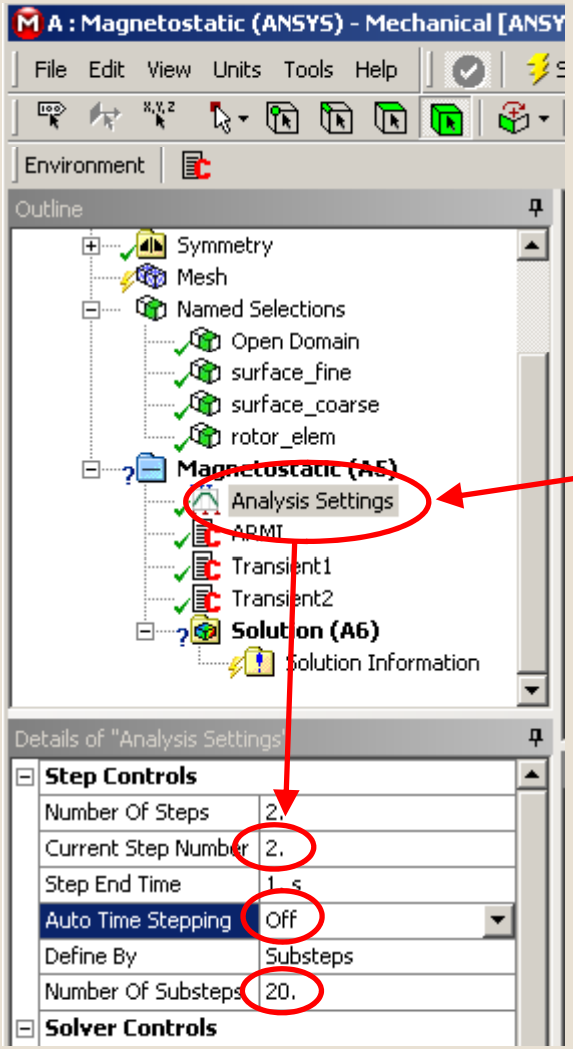
Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

# Set Analysis Options (integration time step size)

Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




**Click on "Analysis Settings", enter "2" for "Current Step Number", set "Auto Time Stepping" to "Off", and set "Number Of Substeps" to 20.**

Workbench-Simulation Emag

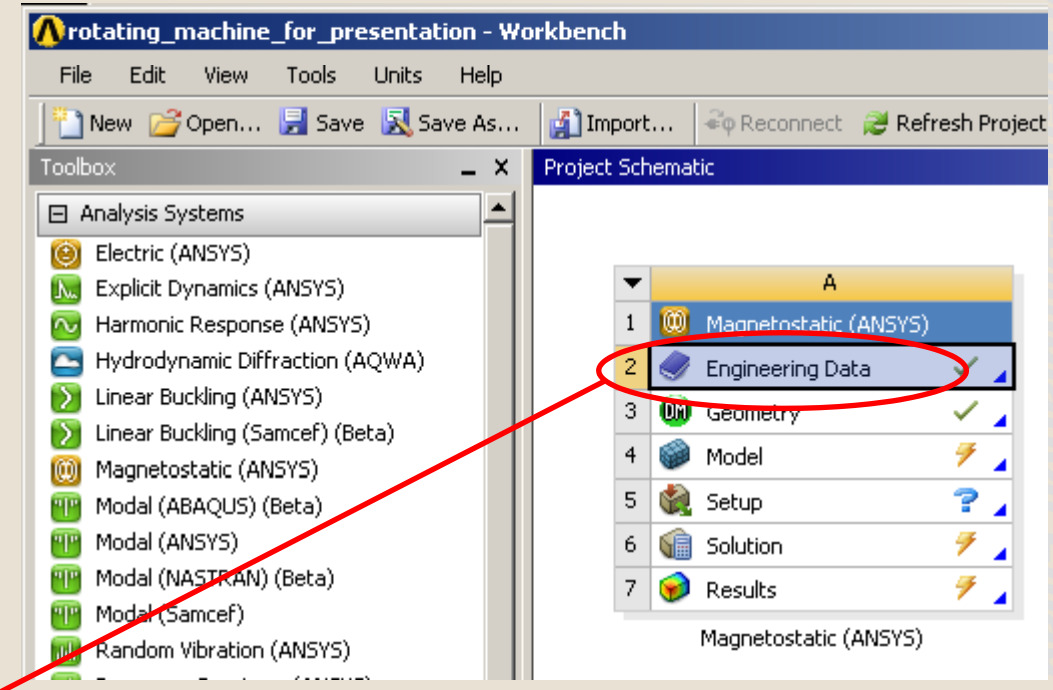

© 2010 ANSYS, Inc. All rights reserved.

# Import and Modify PM Material Properties

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

- Recall we want to calculate eddy currents in the permanent magnet
  - Must add Neodymium PM material to list of materials included in project
  - Must modify Neodymium PM material to include electric resistivity
  - Must assign the modified Neodymium PM material to the PM body
- Begin by returning to the Project Page and double clicking on “Engineering Data”

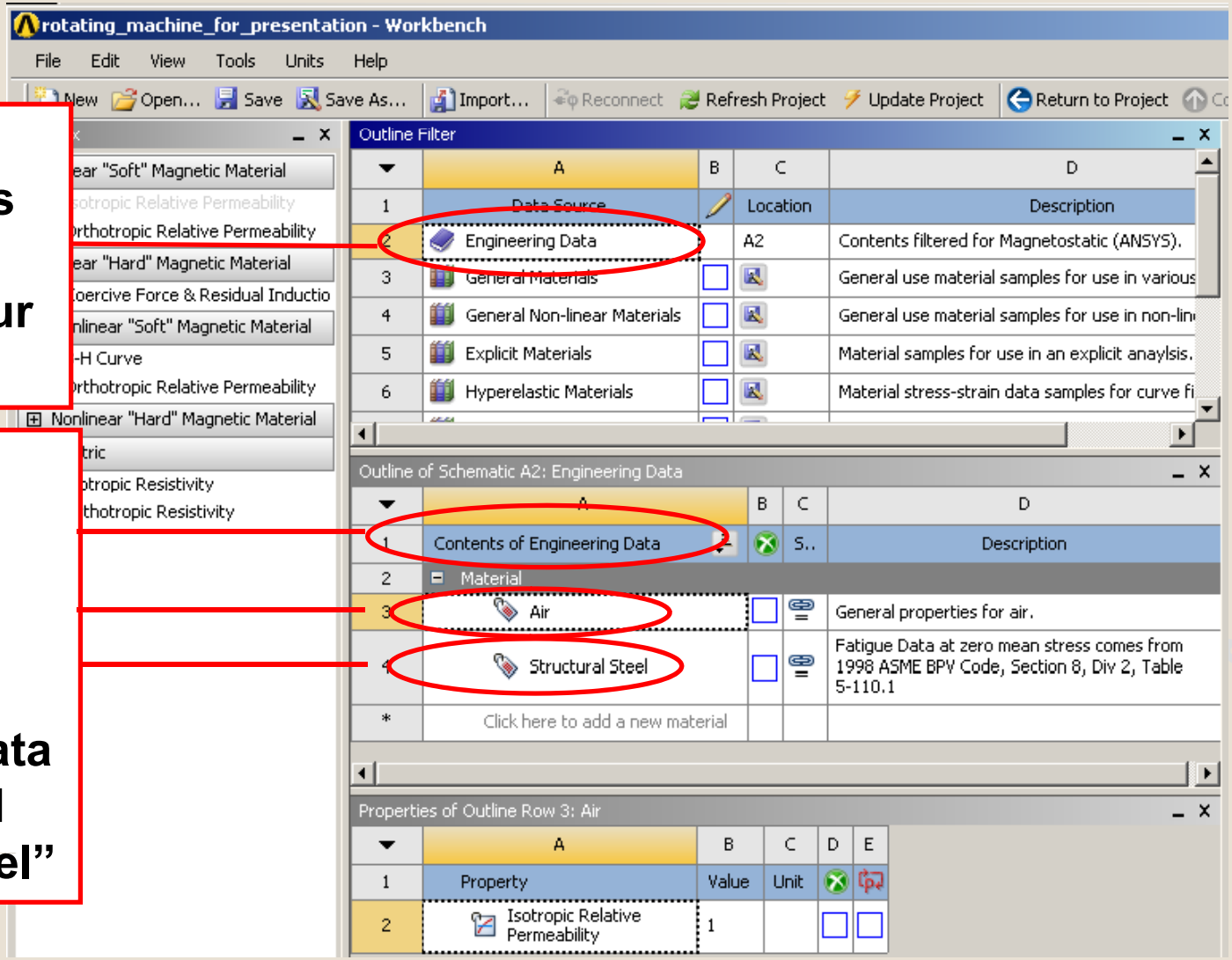




© 2010 ANSYS, Inc. All rights reserved.

"Engineering Data" contains materials available in your model

The only 2 material properties currently available in Engineering Data are "Air" and "Structural Steel"





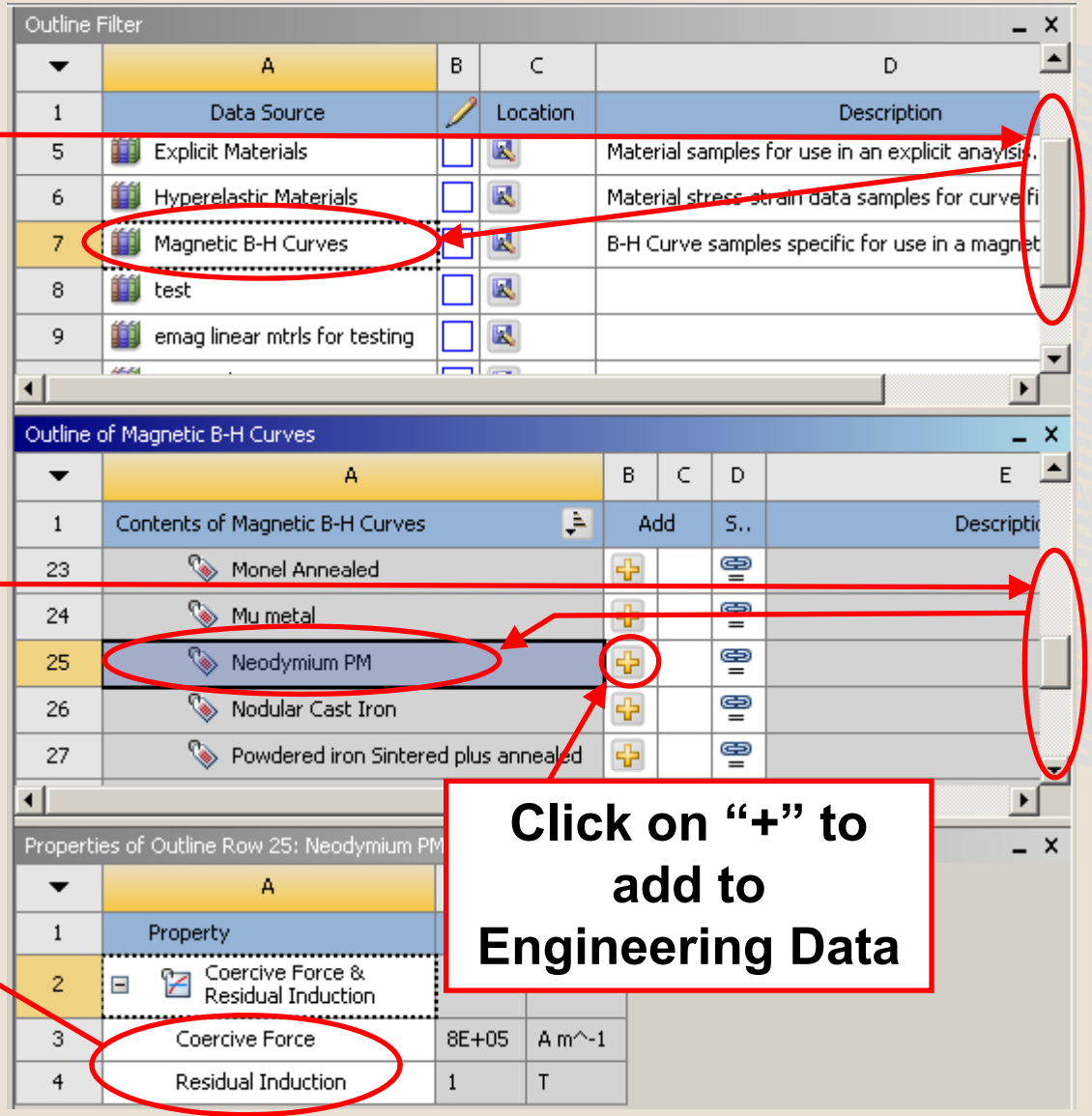
© 2010 ANSYS, Inc. All rights reserved.

Scroll down to "Magnetic B-H Curves"

Scroll down to "Neodymium PM"

Note only two material properties for "Neodymium PM" are defined (Coercive Force and Residual Induction)

Click on "+" to add to Engineering Data



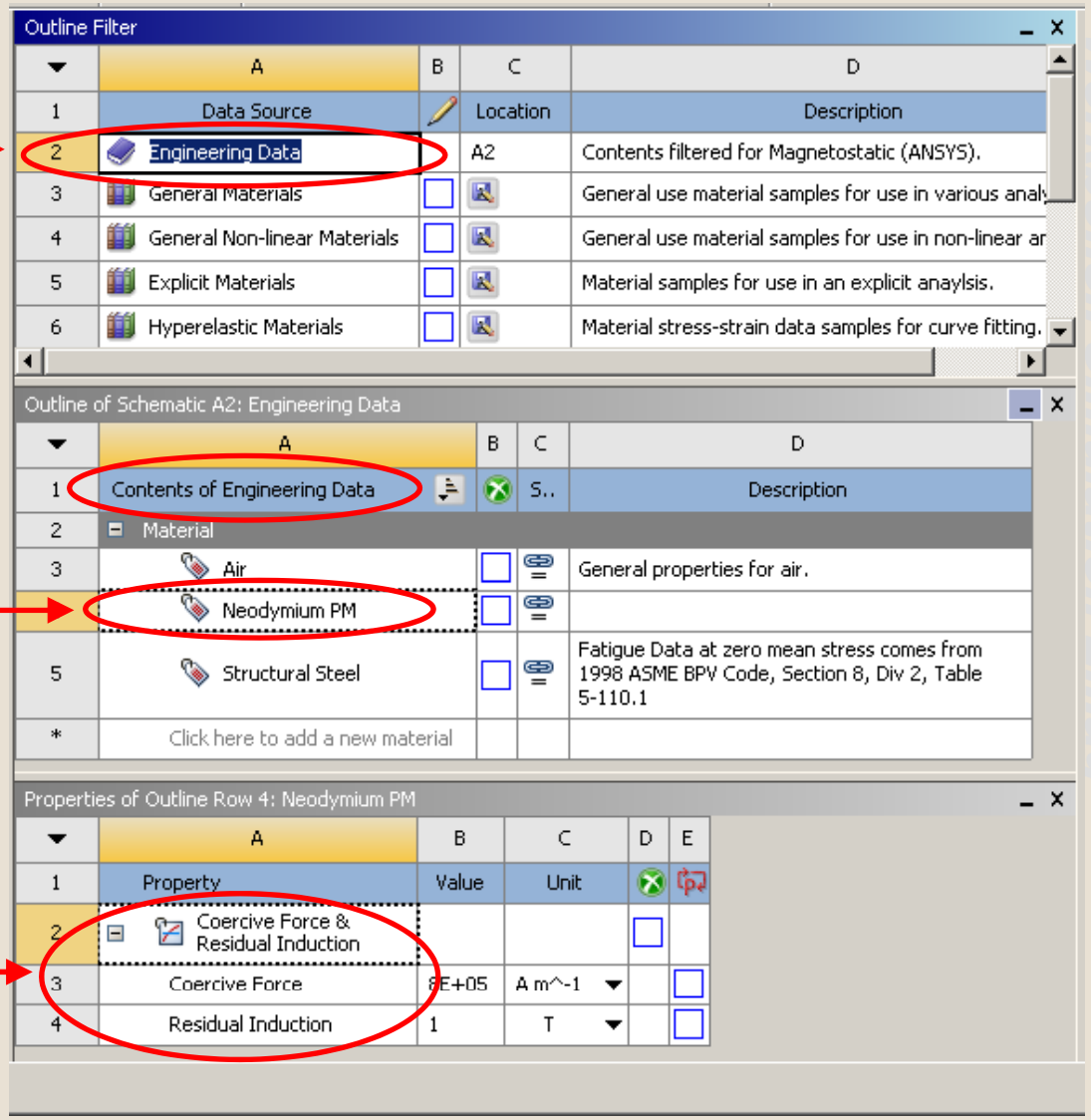


© 2010 ANSYS, Inc. All rights reserved.

Click on Engineering Data

Note that "Neodymium PM" has been added to the contents of Engineering Data; it is now available in your model

We still need to add electrical conductivity to the PM material so we can calculate induced eddy currents

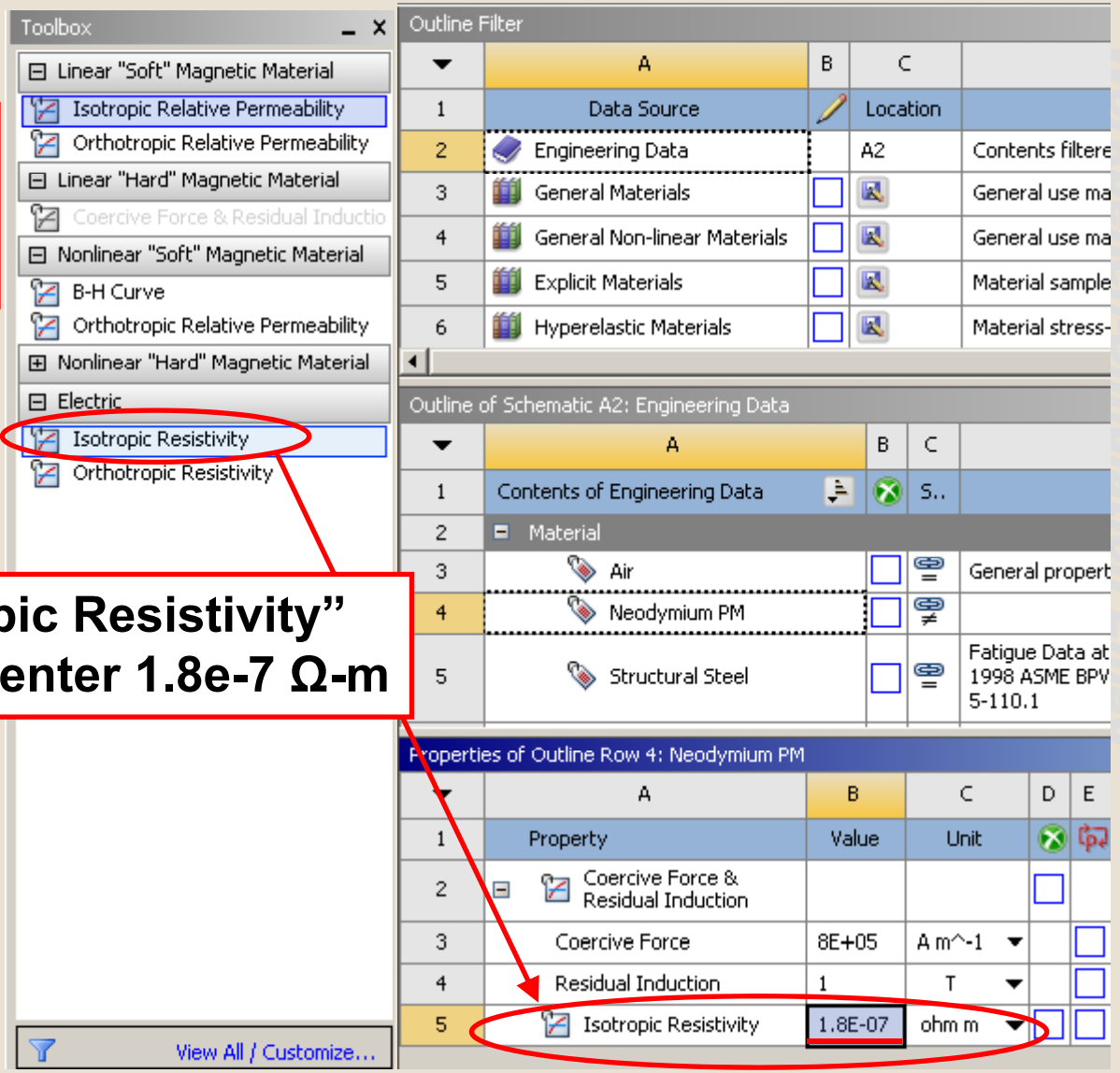




© 2010 ANSYS, Inc. All rights reserved.

Double click to add electrical conductivity to the PM material

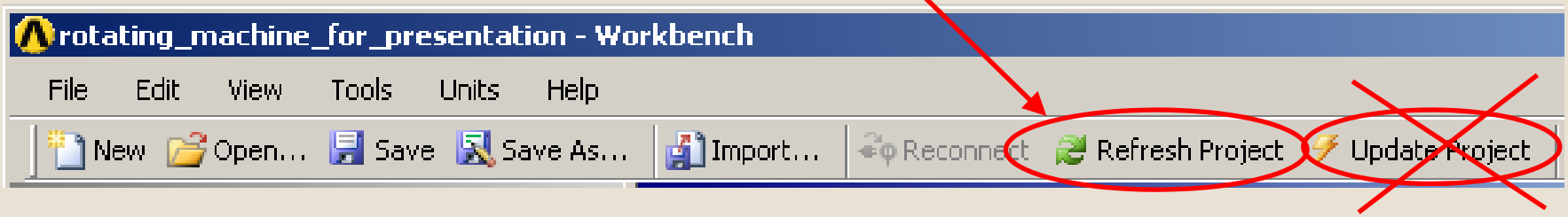
"Isotropic Resistivity" appears, enter 1.8e-7 Ω-m





© 2010 ANSYS, Inc. All rights reserved.

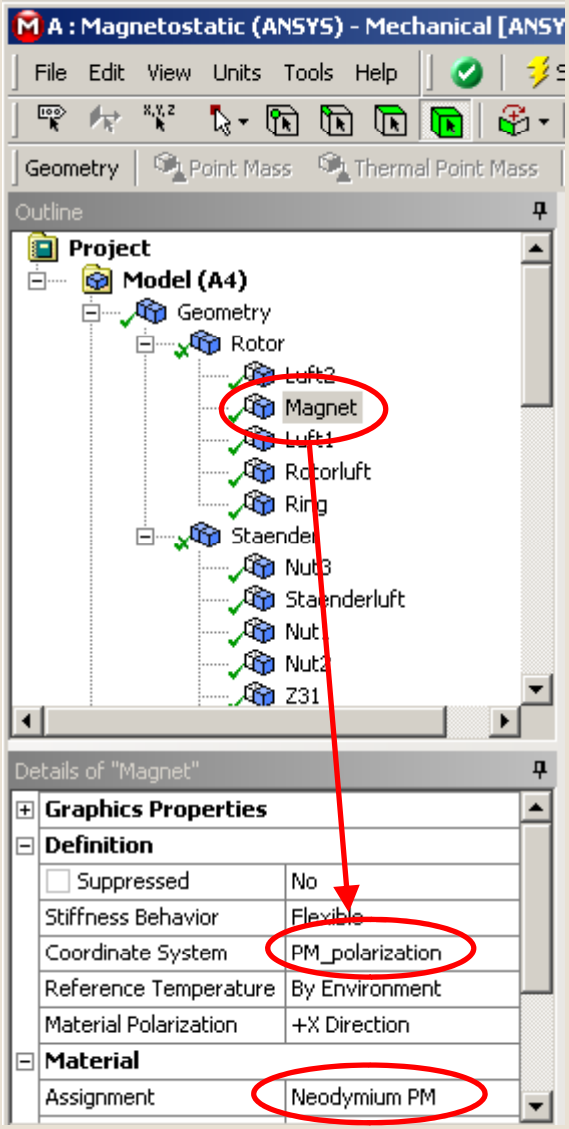
**The new PM material property now exists in your database, but Mechanical is not aware of this. Refresh the project to read new PM data into Mechanical. DO NOT choose Update Project! Doing so may initiate the solution – we are not yet ready for this.**




Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

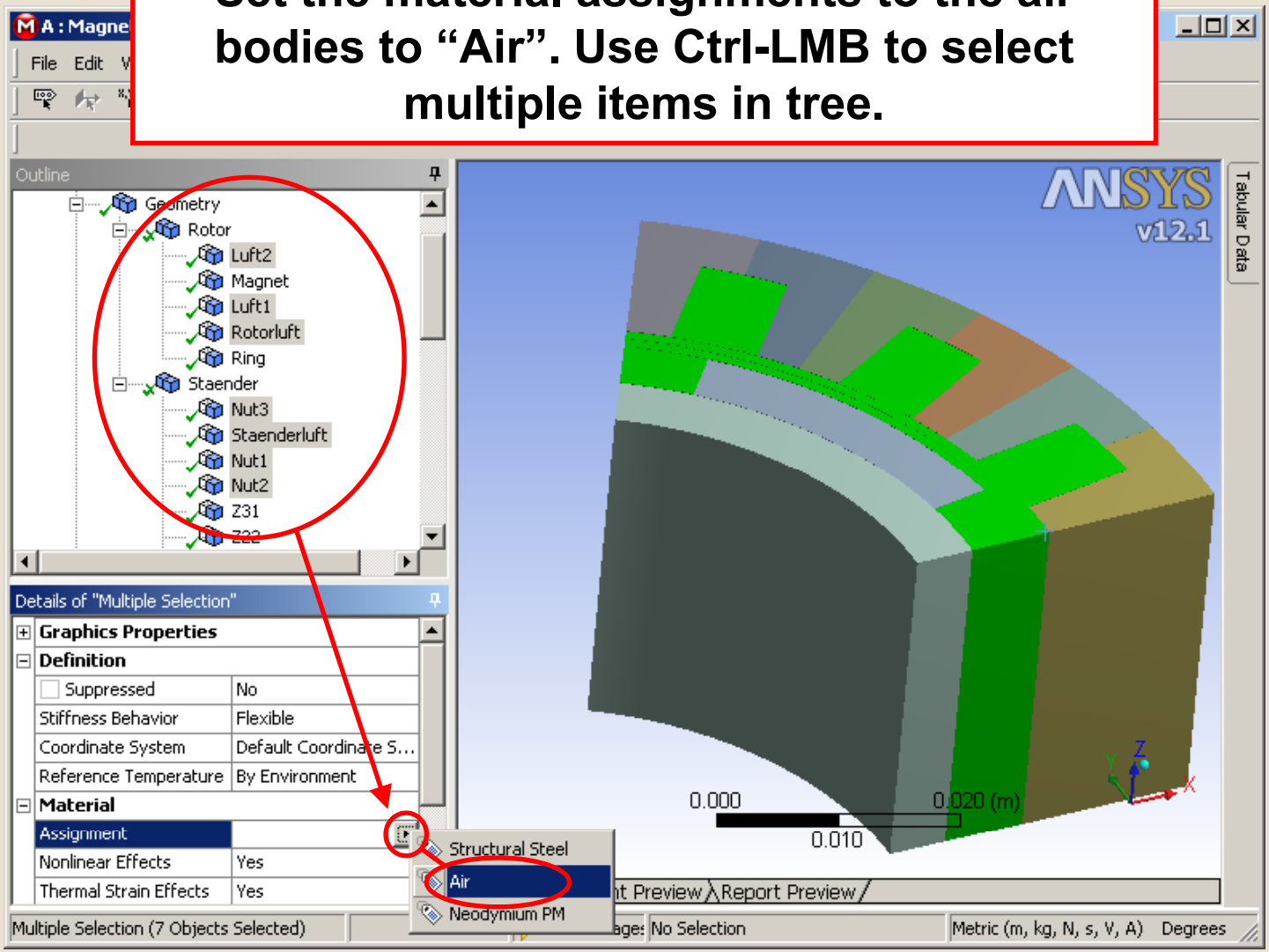
**Return to the Mechanical GUI. Set the Magnet Coordinate System to "PM_polarization" and its Material Assignment to "Neodymium PM"**




Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

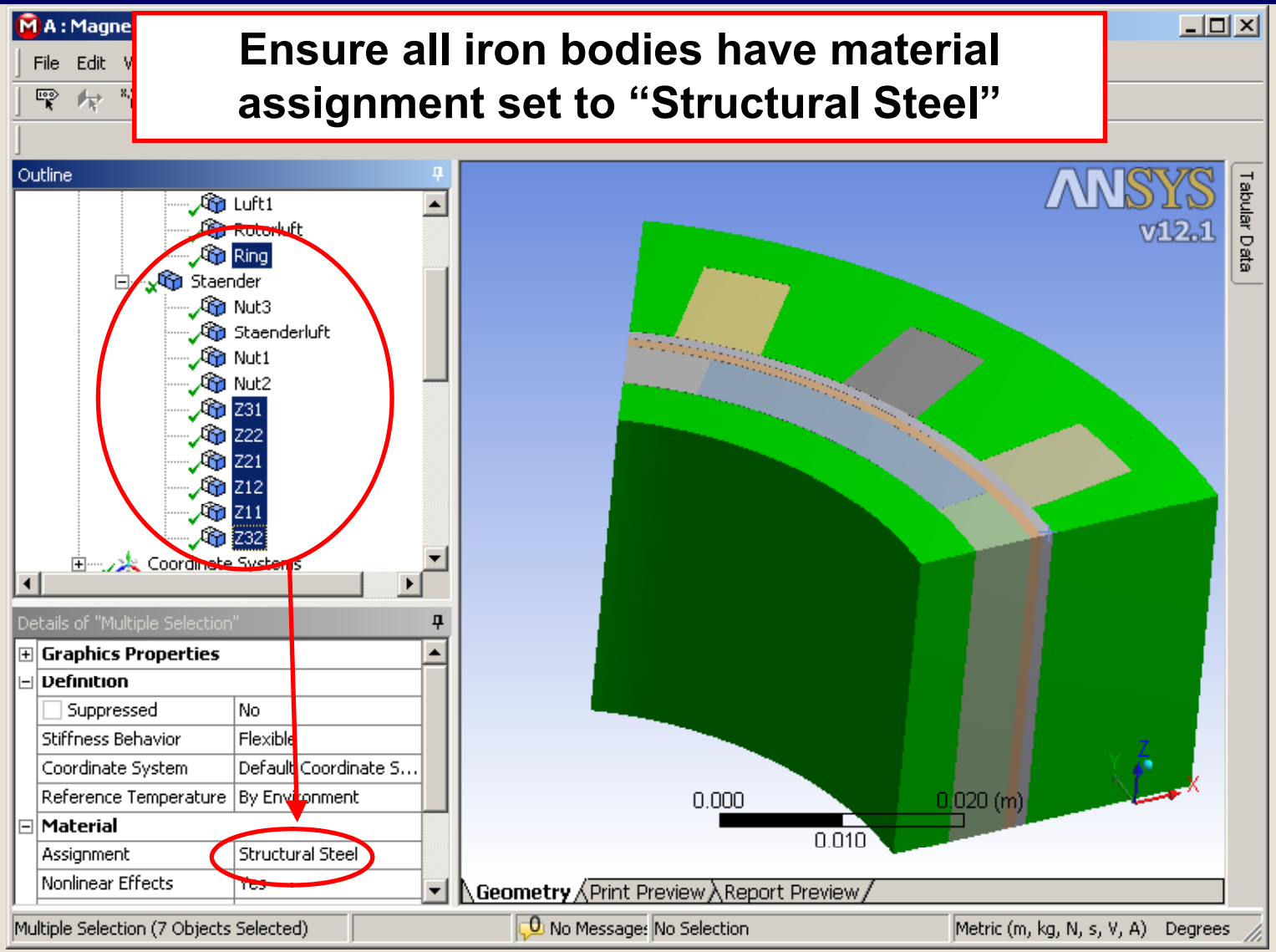
**Set the material assignments to the air bodies to “Air”. Use Ctrl-LMB to select multiple items in tree.**




Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

**Ensure all iron bodies have material assignment set to "Structural Steel"**



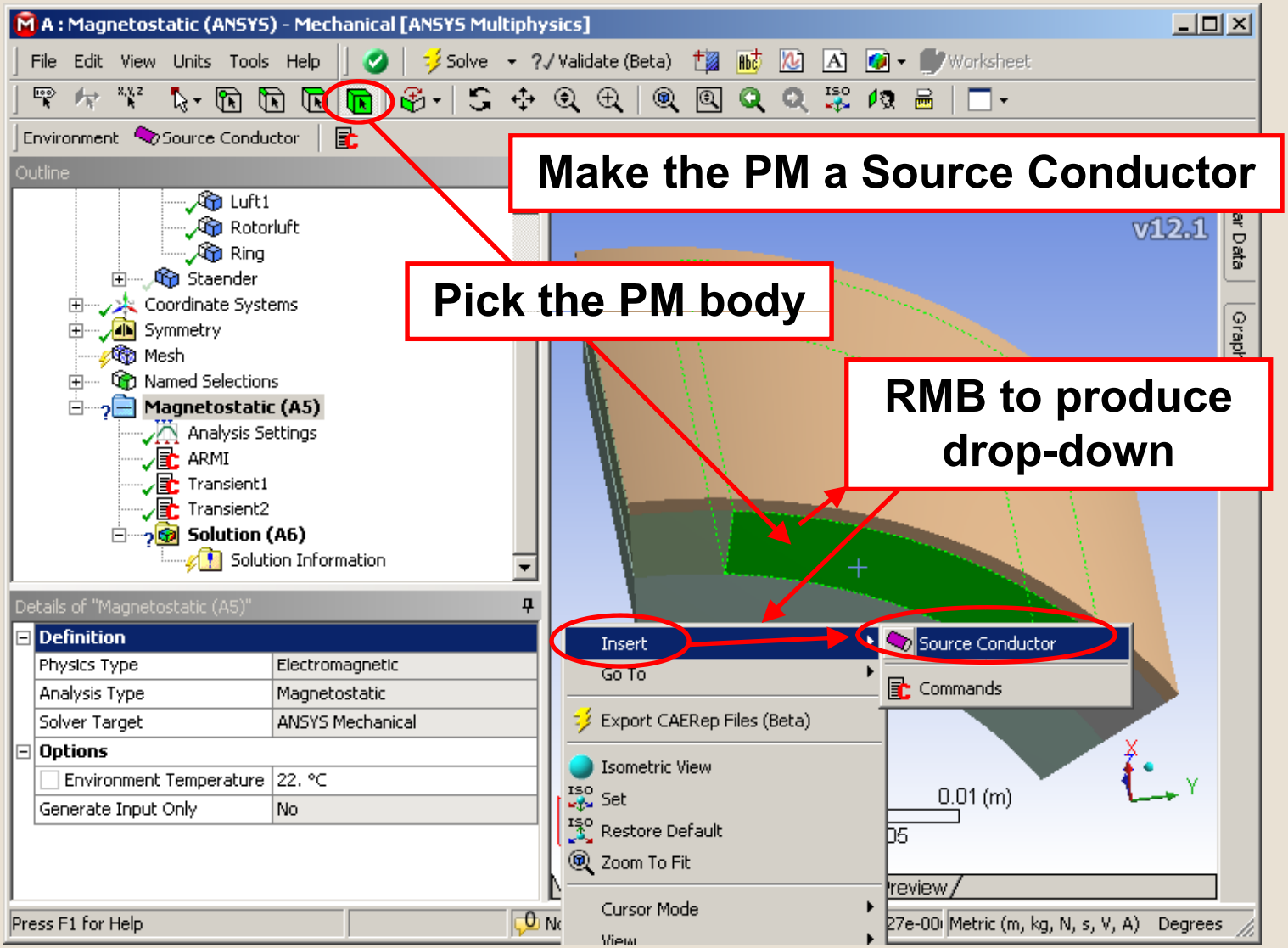

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

# Applying boundary conditions

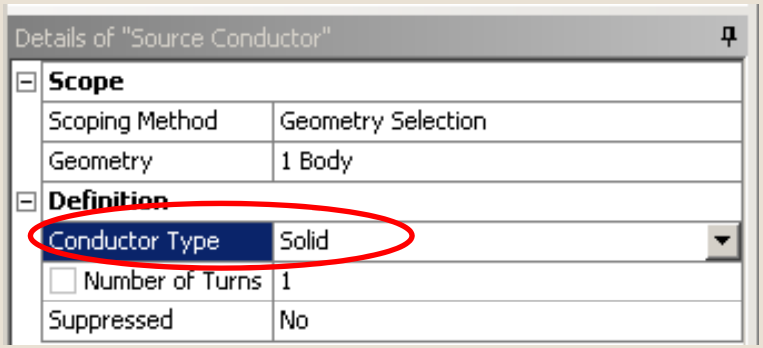
Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.





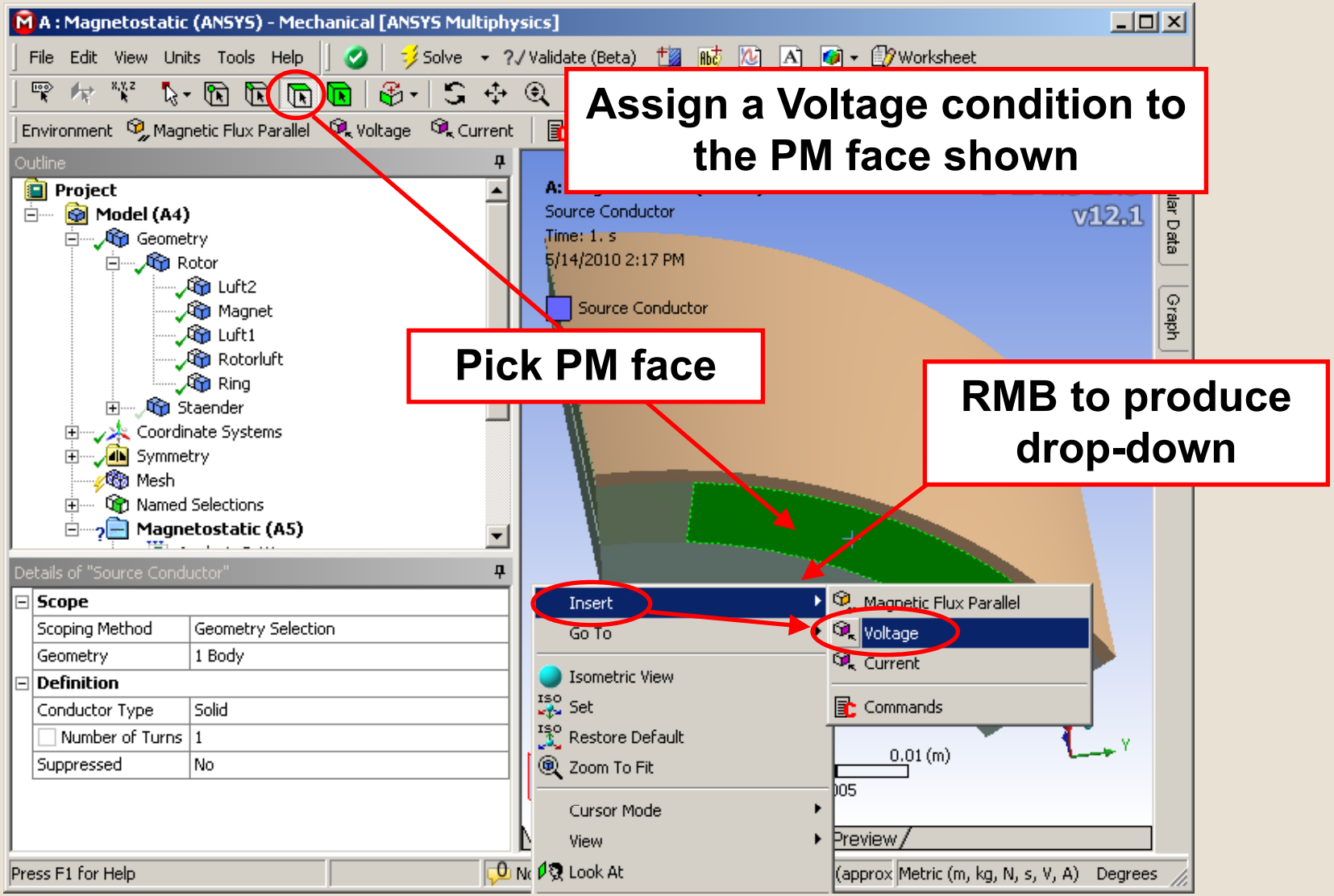
© 2010 ANSYS, Inc. All rights reserved.




**Set the Source Conductor Type to "Solid"**

Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

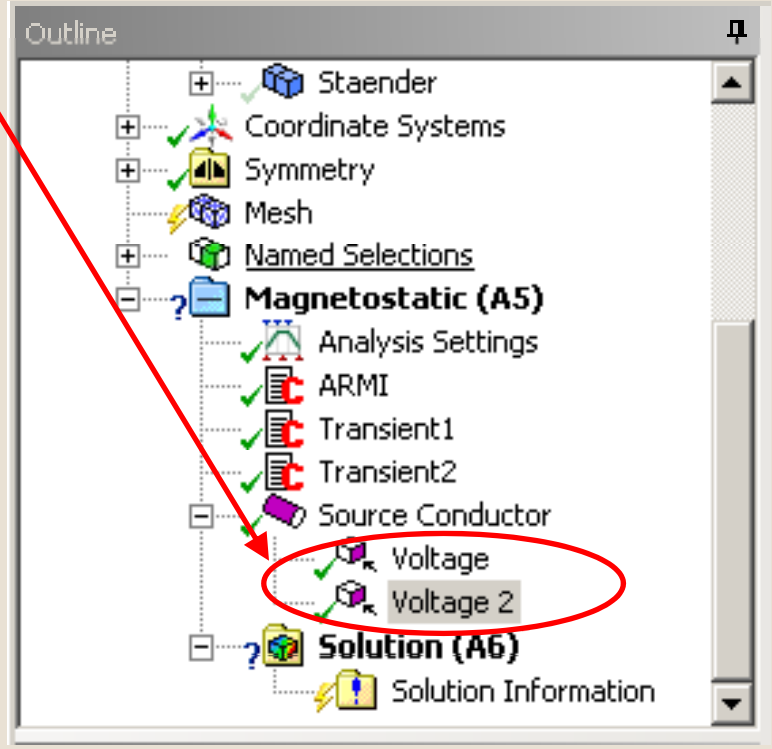




© 2010 ANSYS, Inc. All rights reserved.

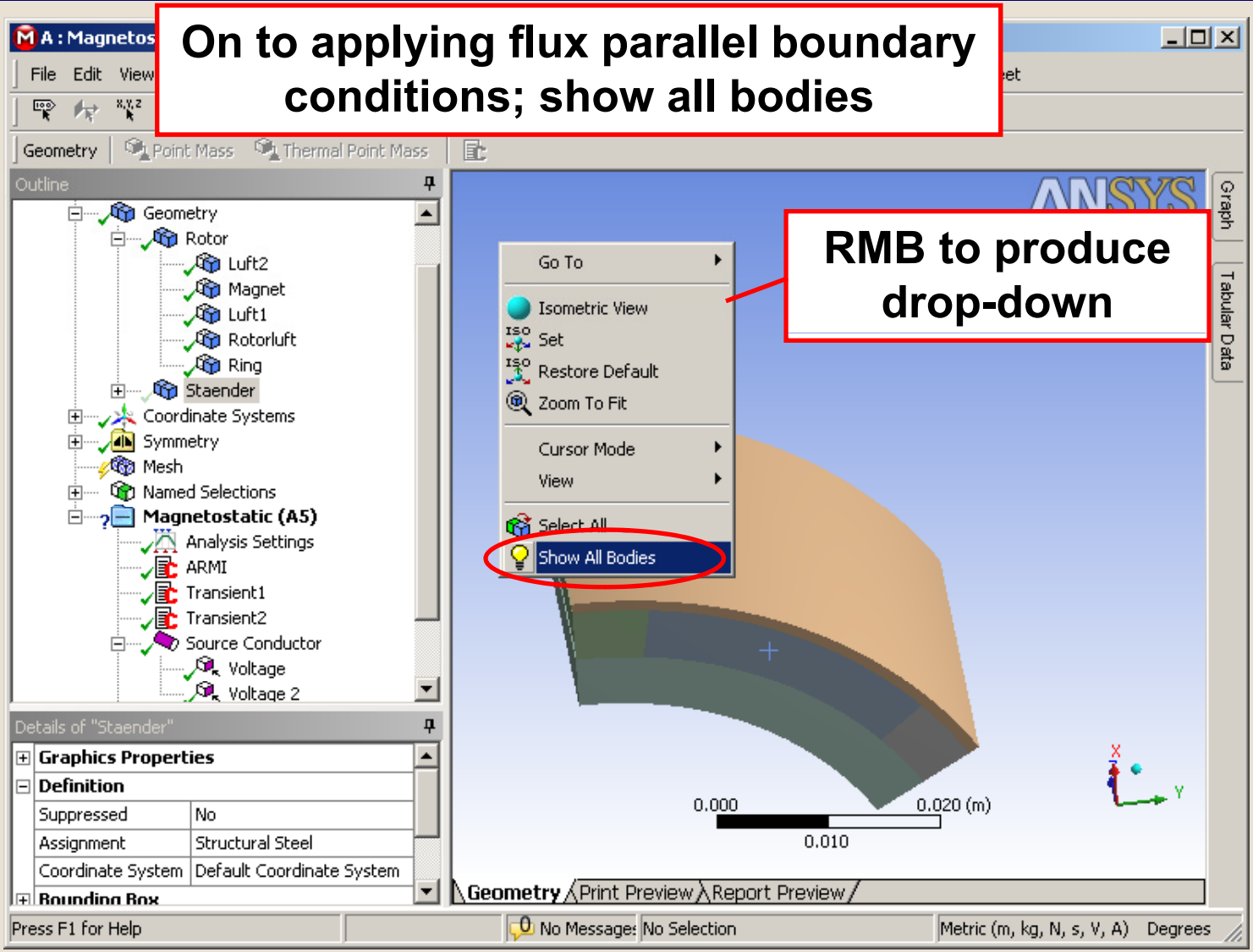
- Repeat the procedure on the previous slide to apply a second voltage condition ***on the same PM face***
  - Note: Mechanical is guiding you to define a source conductor; it therefore expects either two voltage conditions *or* one voltage condition and an impressed current. But by applying two voltage conditions on the same face, the conductor will behave like a passive conductor with an electric symmetry plane. Eddy currents will be induced in the conductive PM by time varying fields, current will flow normal to the VOLT=0 surface.

Should see two voltage conditions in the tree when done



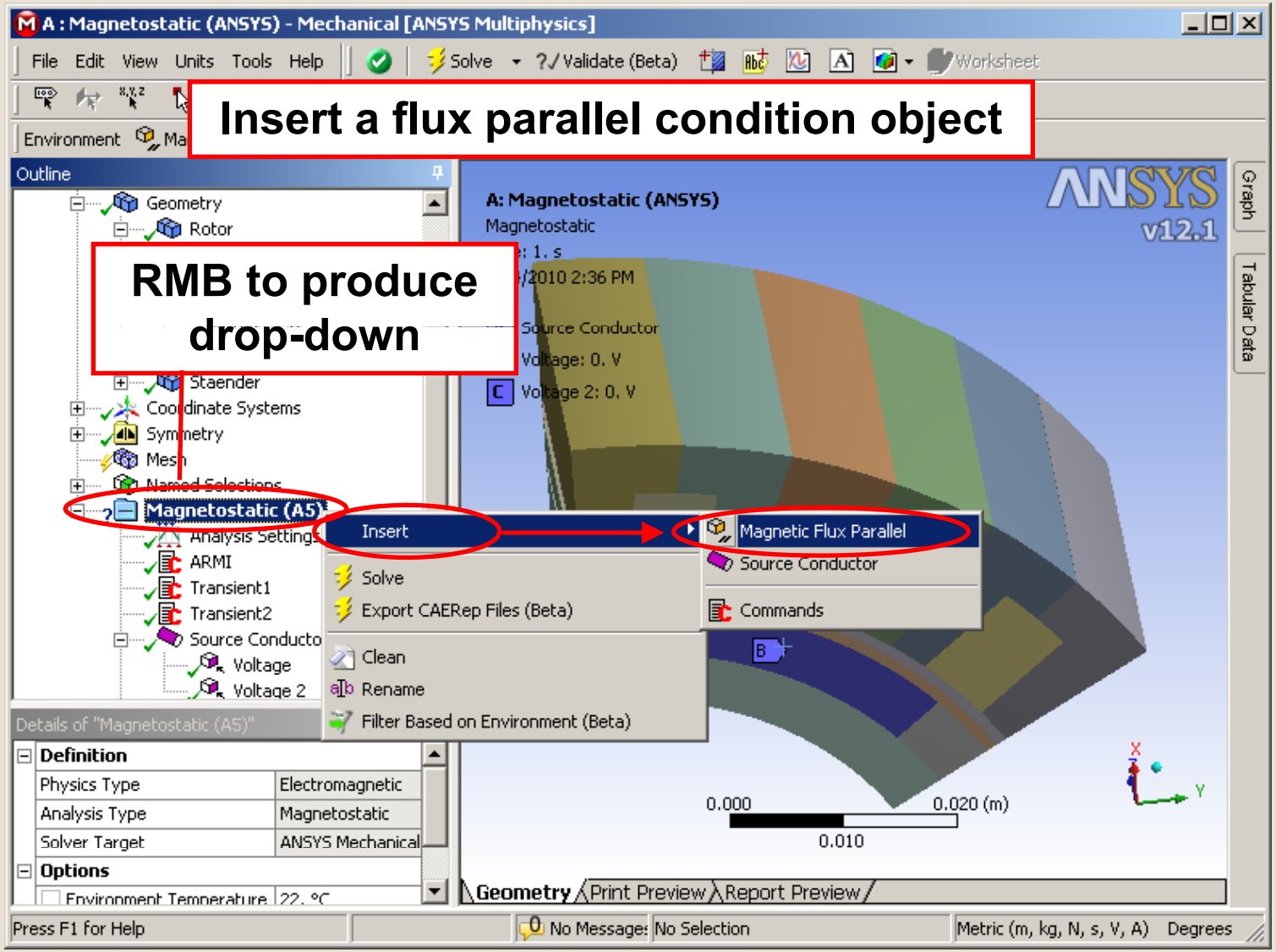

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.



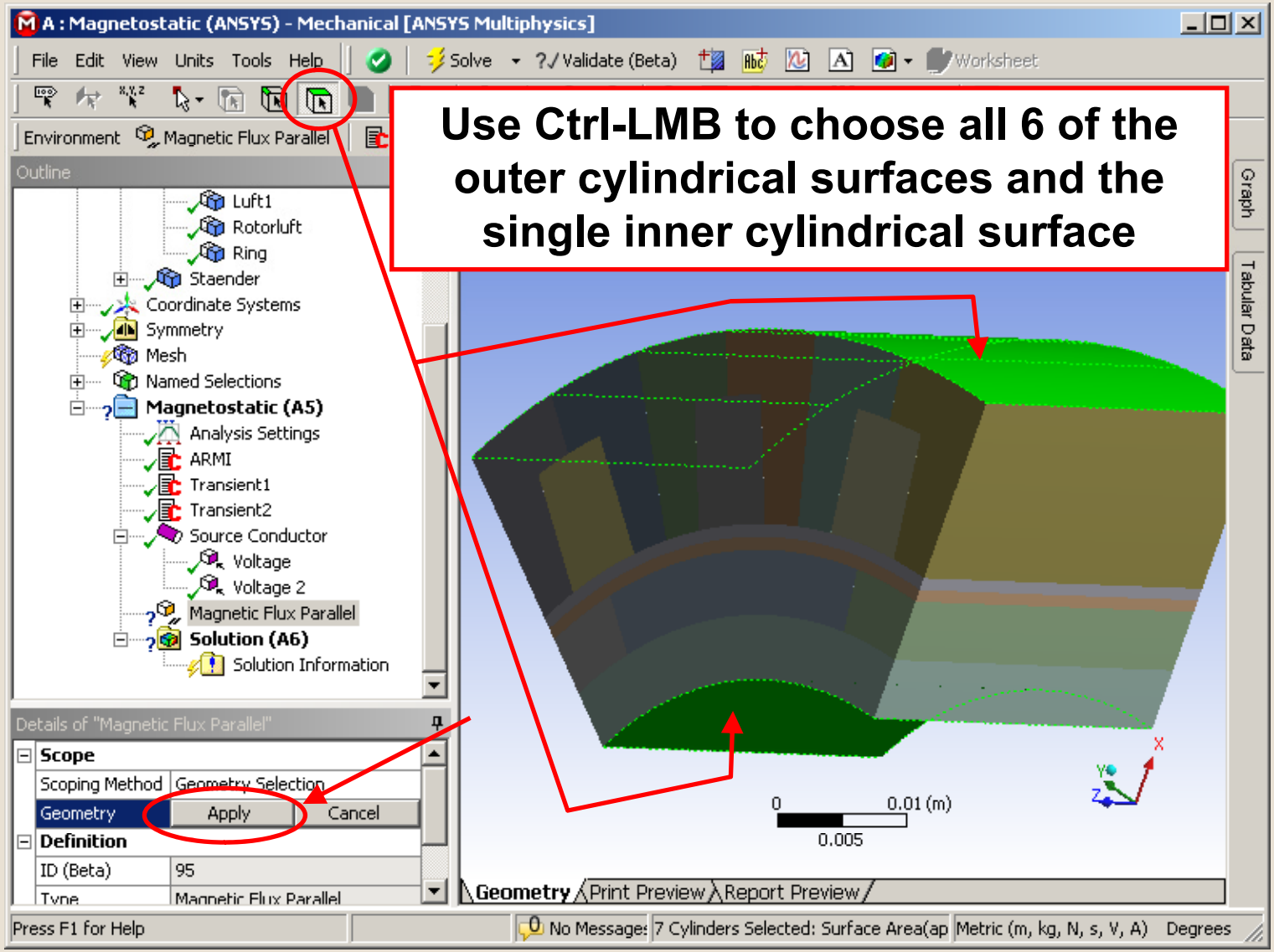

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




Workbench-Simulation Emag

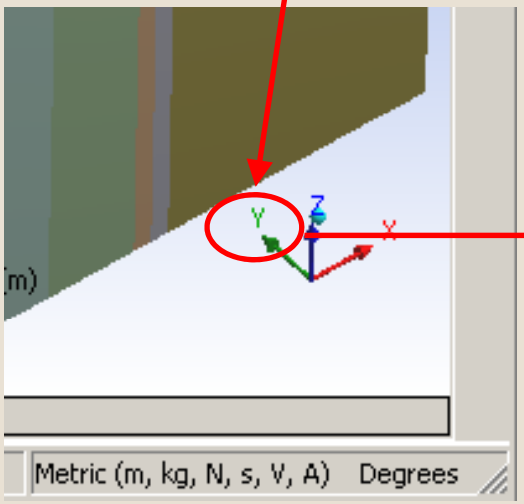

© 2010 ANSYS, Inc. All rights reserved.

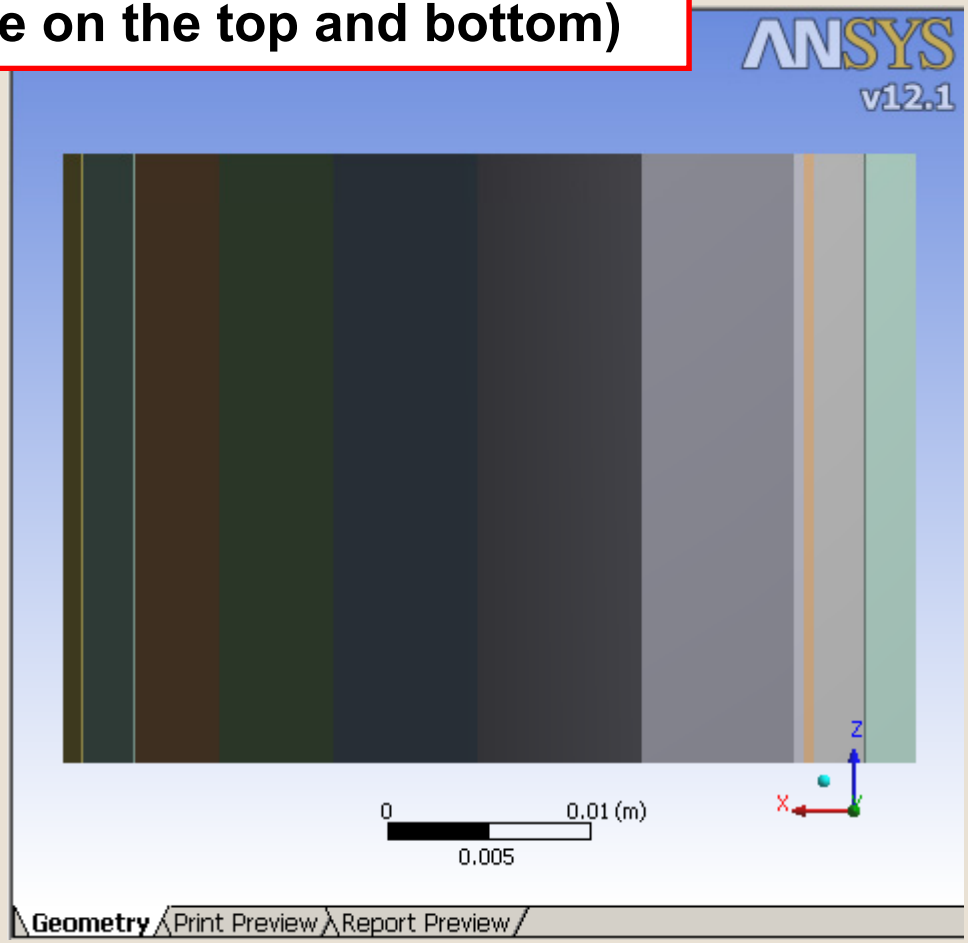



Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

To most conveniently apply flux parallel boundaries to the end surfaces, click on the Y axis to obtain the view shown (the end surfaces are on the top and bottom)

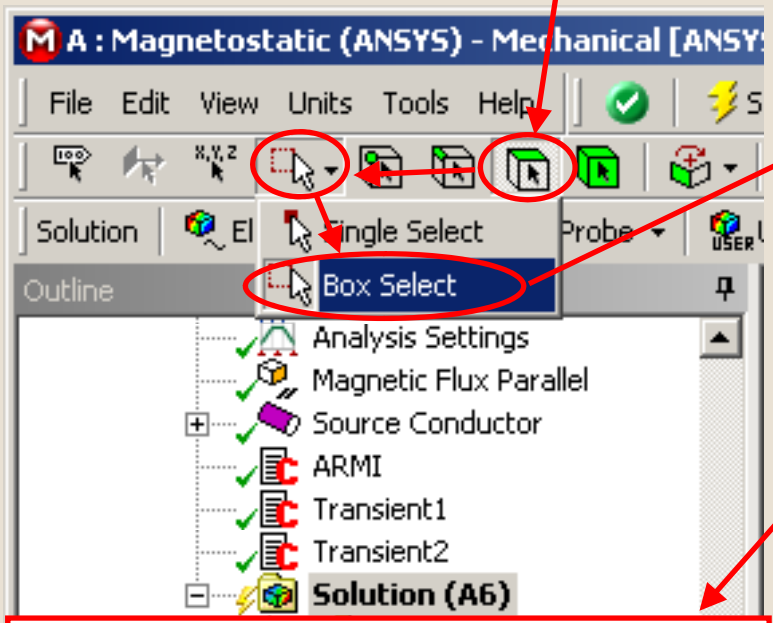


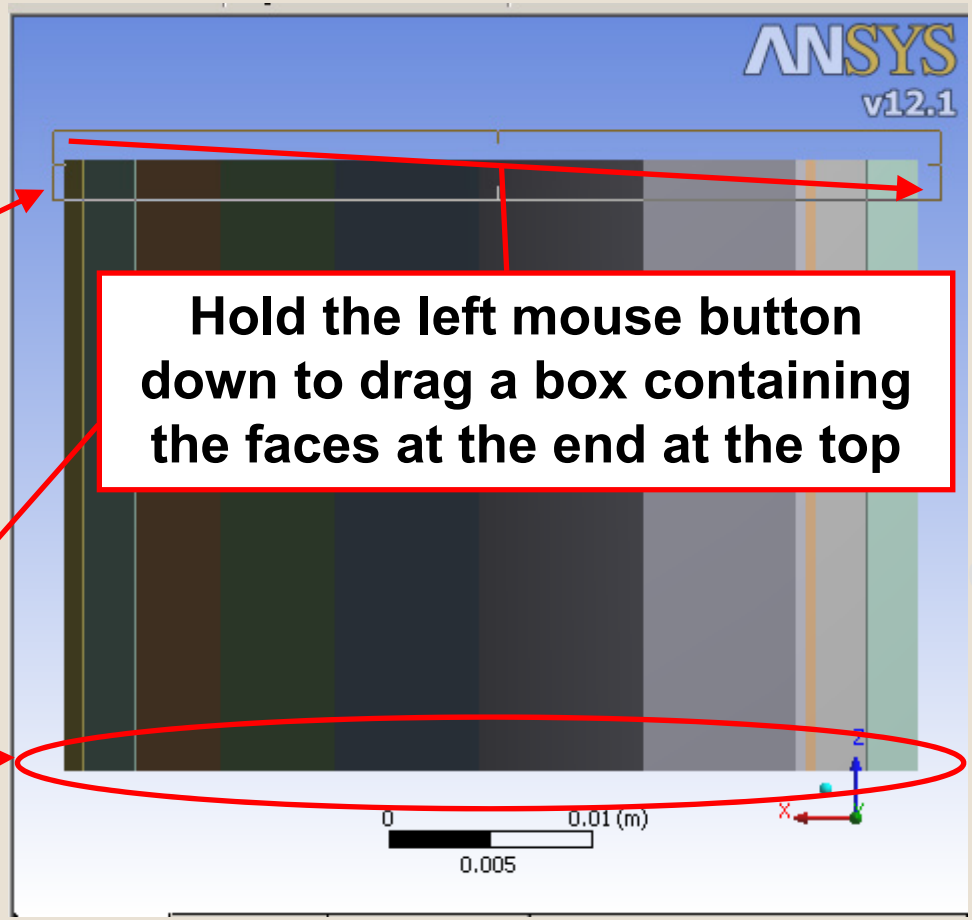

Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

Then choose object selection to “Face” and set the pick mode to Box Select




Use Ctrl-LMB to add the faces at the end on the bottom to the selected set

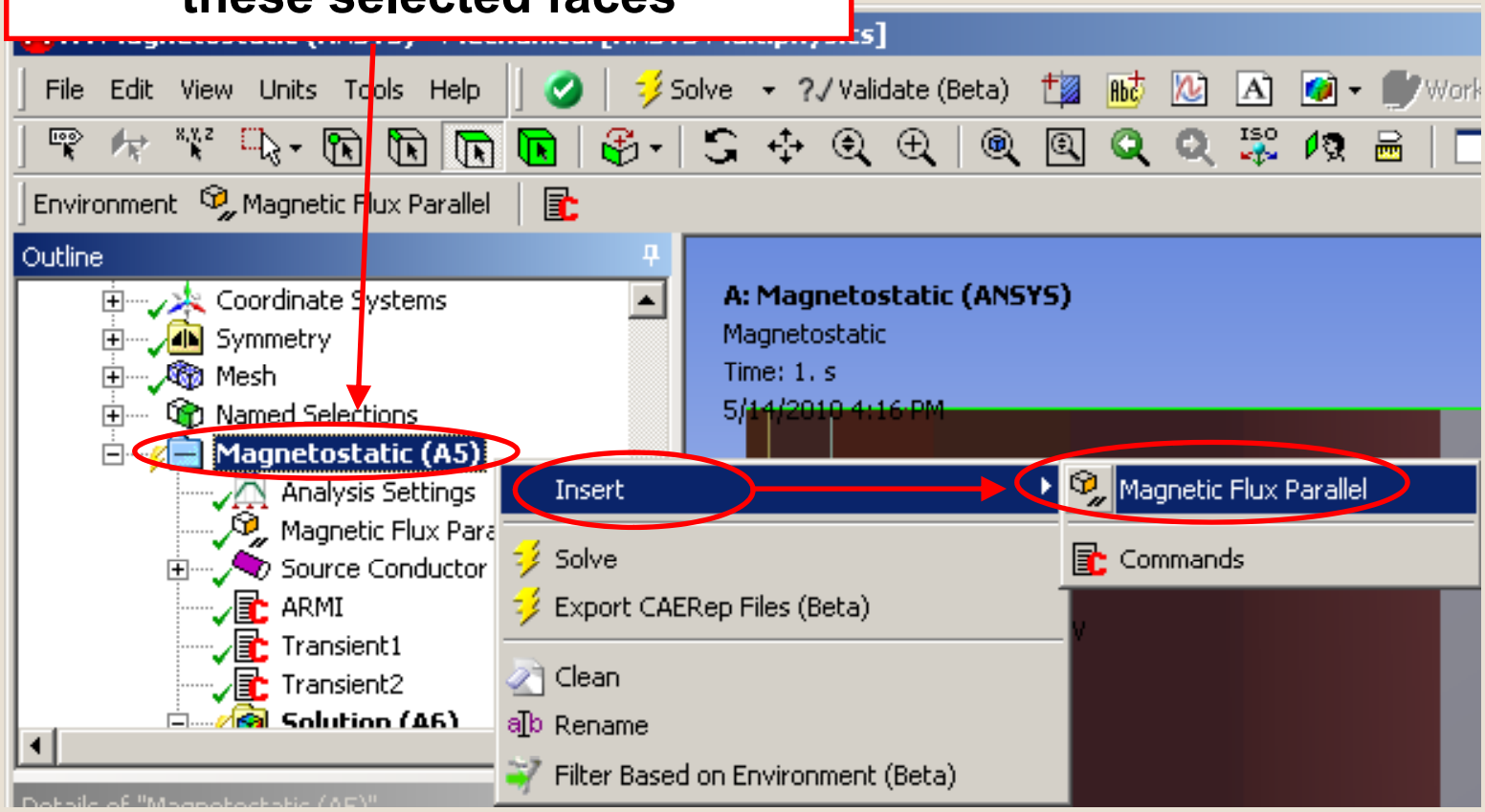



Hold the left mouse button down to drag a box containing the faces at the end at the top

Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

**Then LMB on Magnetostatic and insert a flux parallel condition on these selected faces**



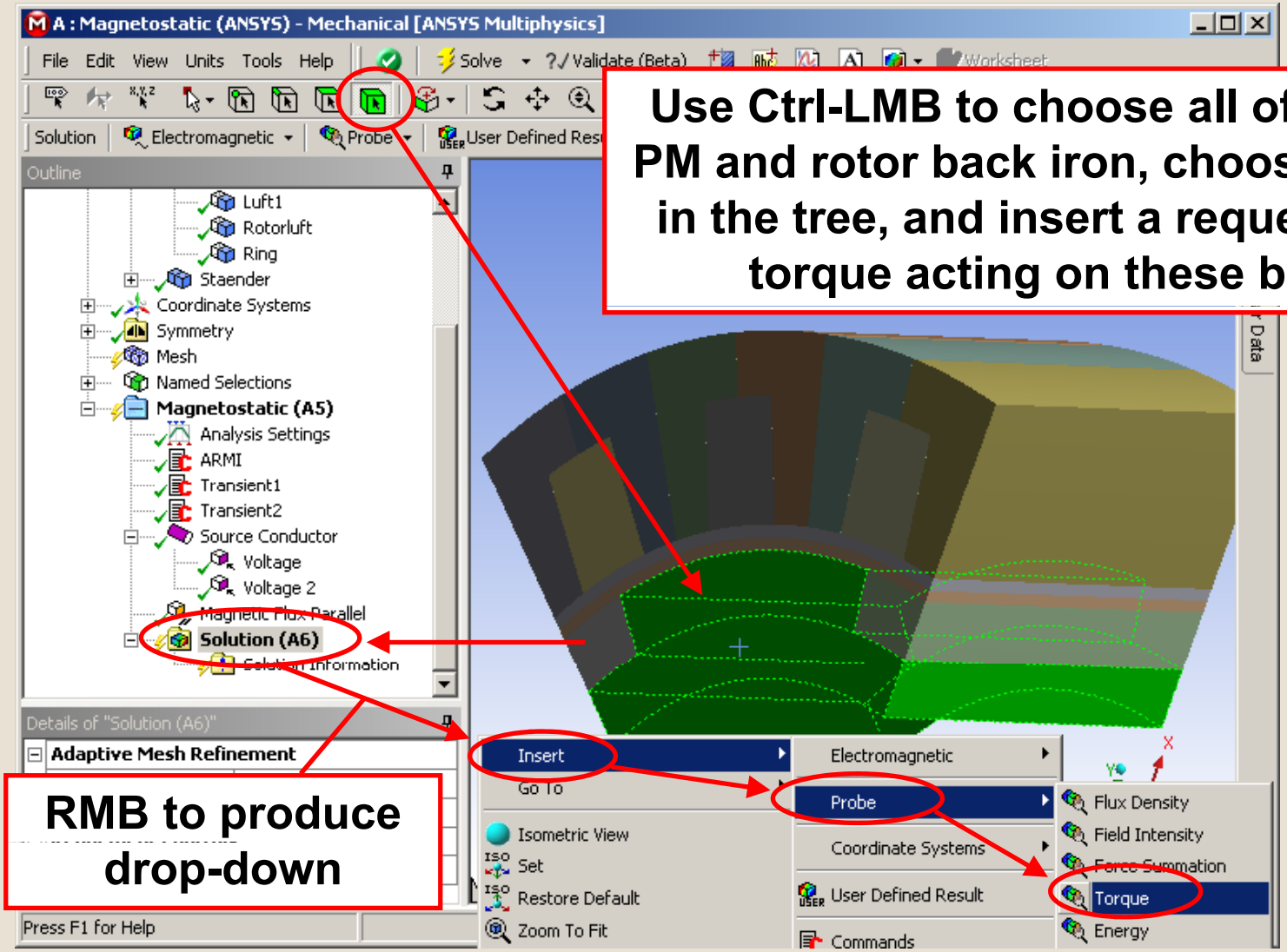


© 2010 ANSYS, Inc. All rights reserved.

Workbench Simulation Emag

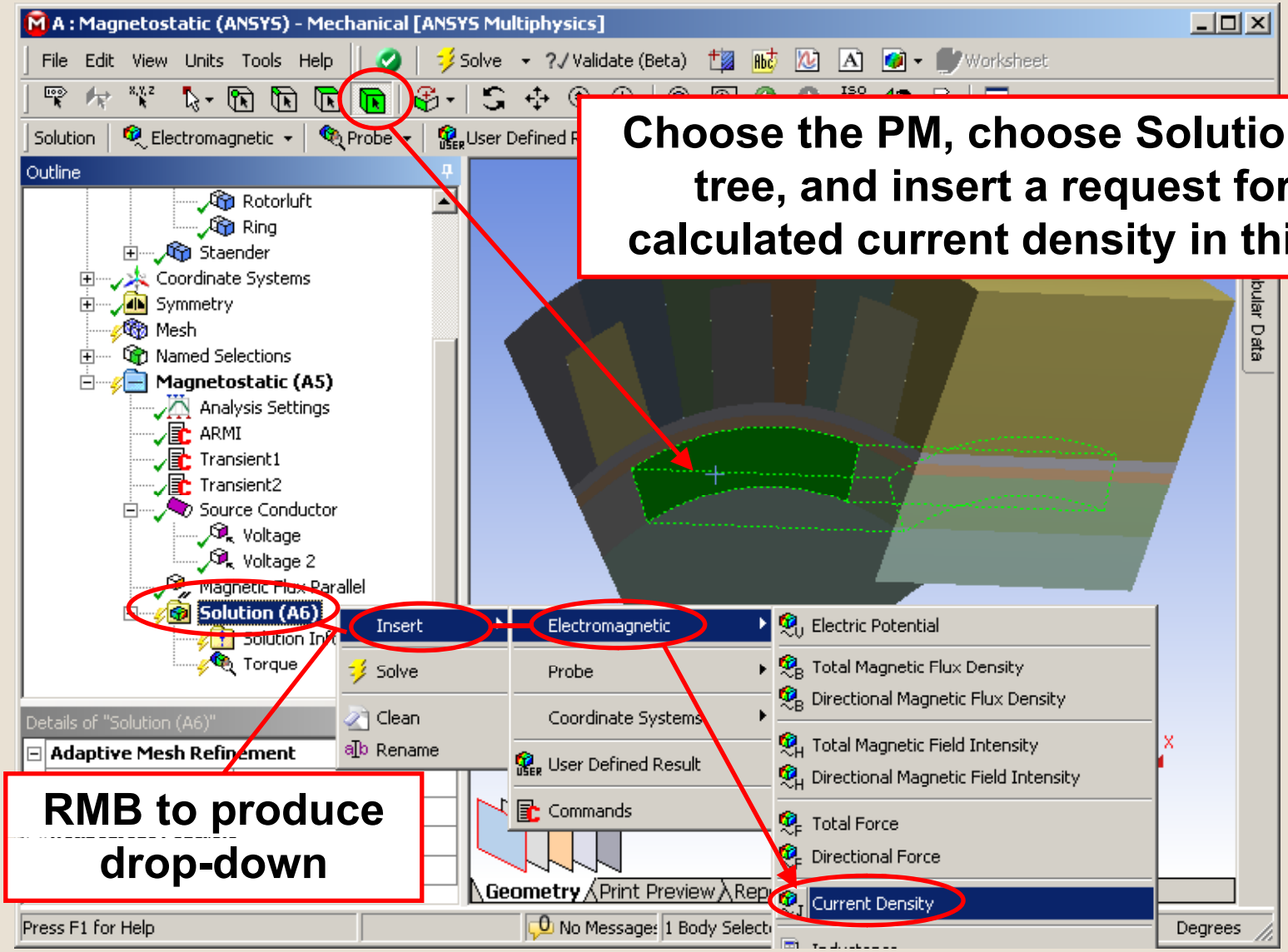
**Requests post processing results plots may be done either now or later after the solution is calculated – we will add them now**


© 2010 ANSYS, Inc. All rights reserved.



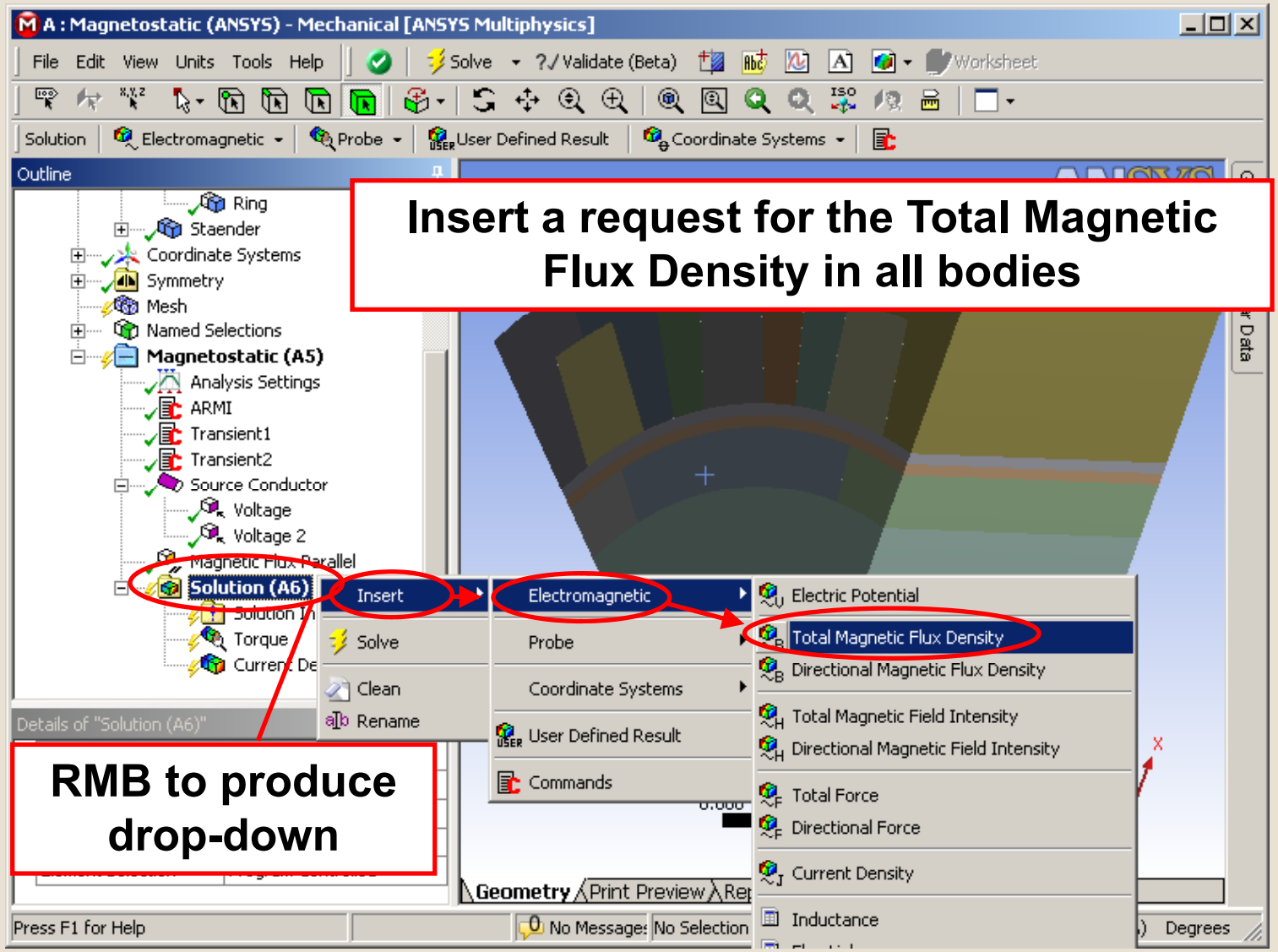


© 2010 ANSYS, Inc. All rights reserved.



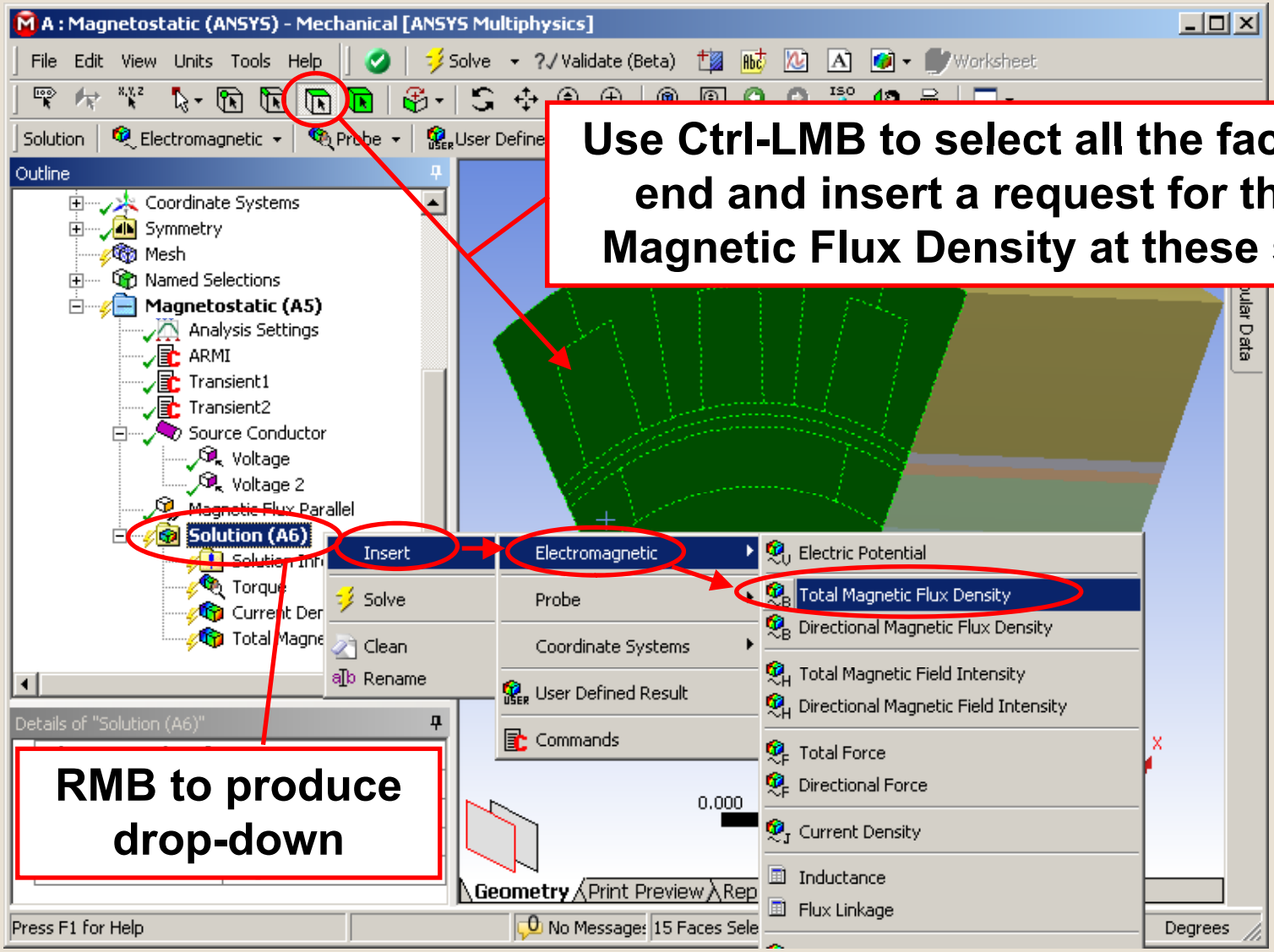


© 2010 ANSYS, Inc. All rights reserved.




Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




Workbench-Simulation Emag

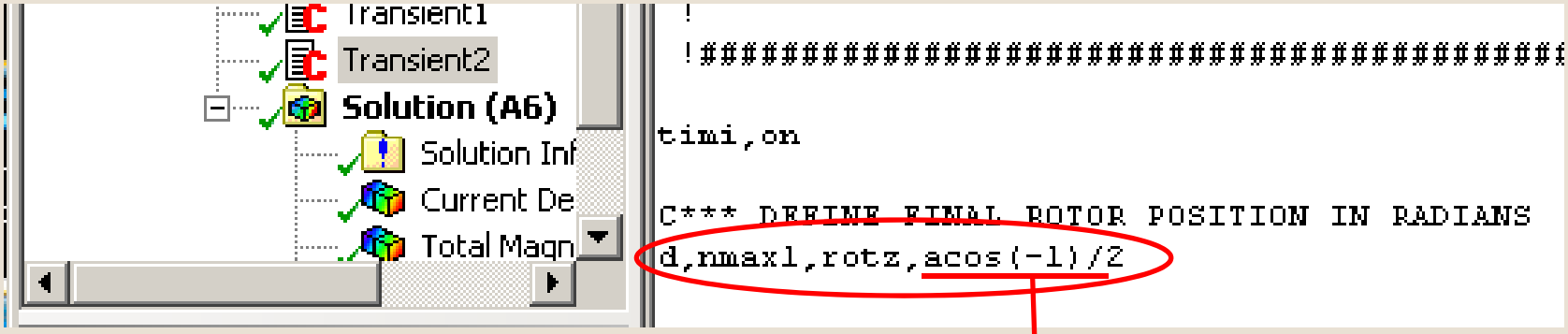

© 2010 ANSYS, Inc. All rights reserved.

# Edit Command Object “Transient2” to add final rotor position in radians at end of transient simulation

Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

**Add line to define final rotor position *in radians* on node number nmax1**




**π/2 = 90°**

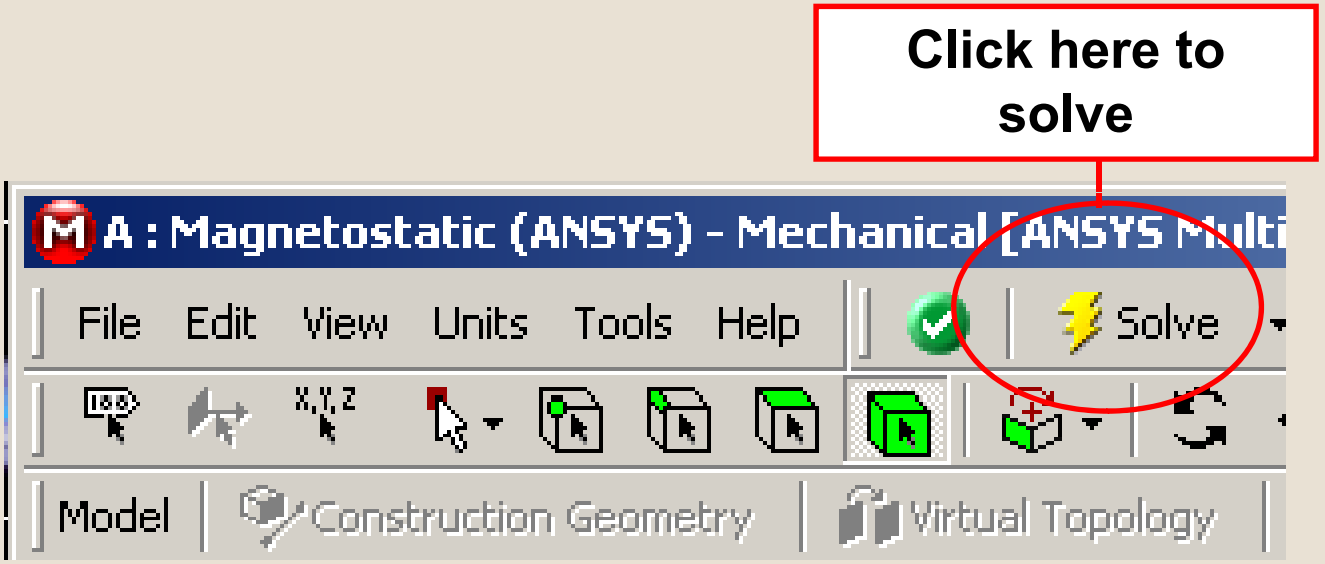
Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

# Solve (takes ~2-3 minutes)

Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.



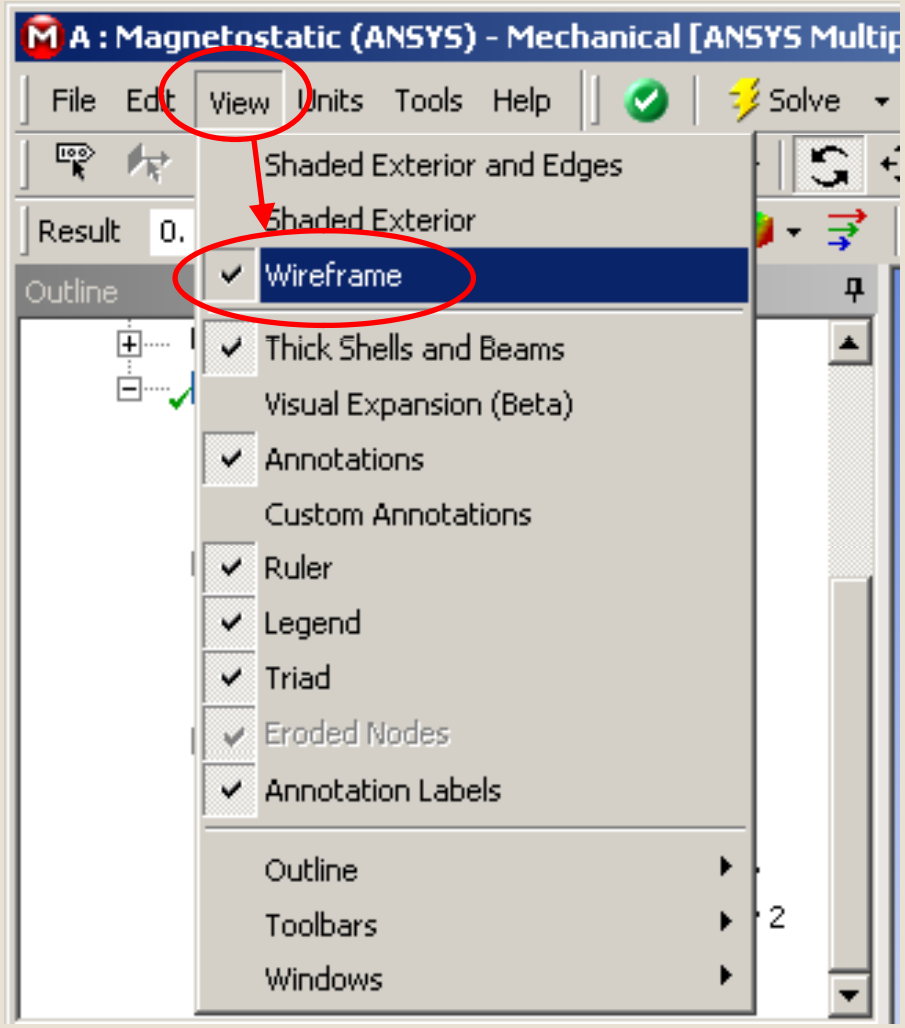

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

# Post Processing

Workbench Simulation Emag

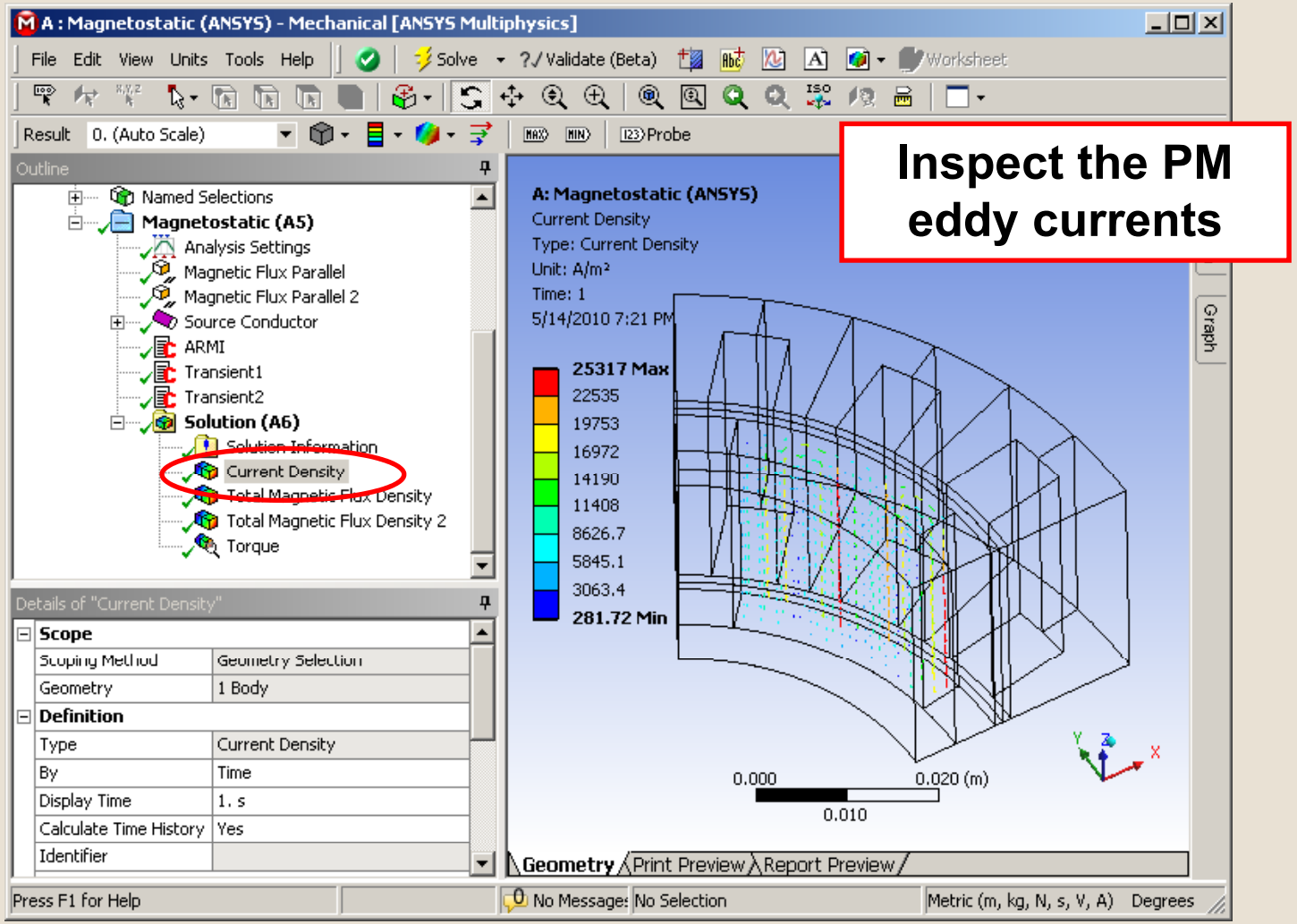

© 2010 ANSYS, Inc. All rights reserved.




**Set the Wireframe view mode**

Workbench Simulation Emag

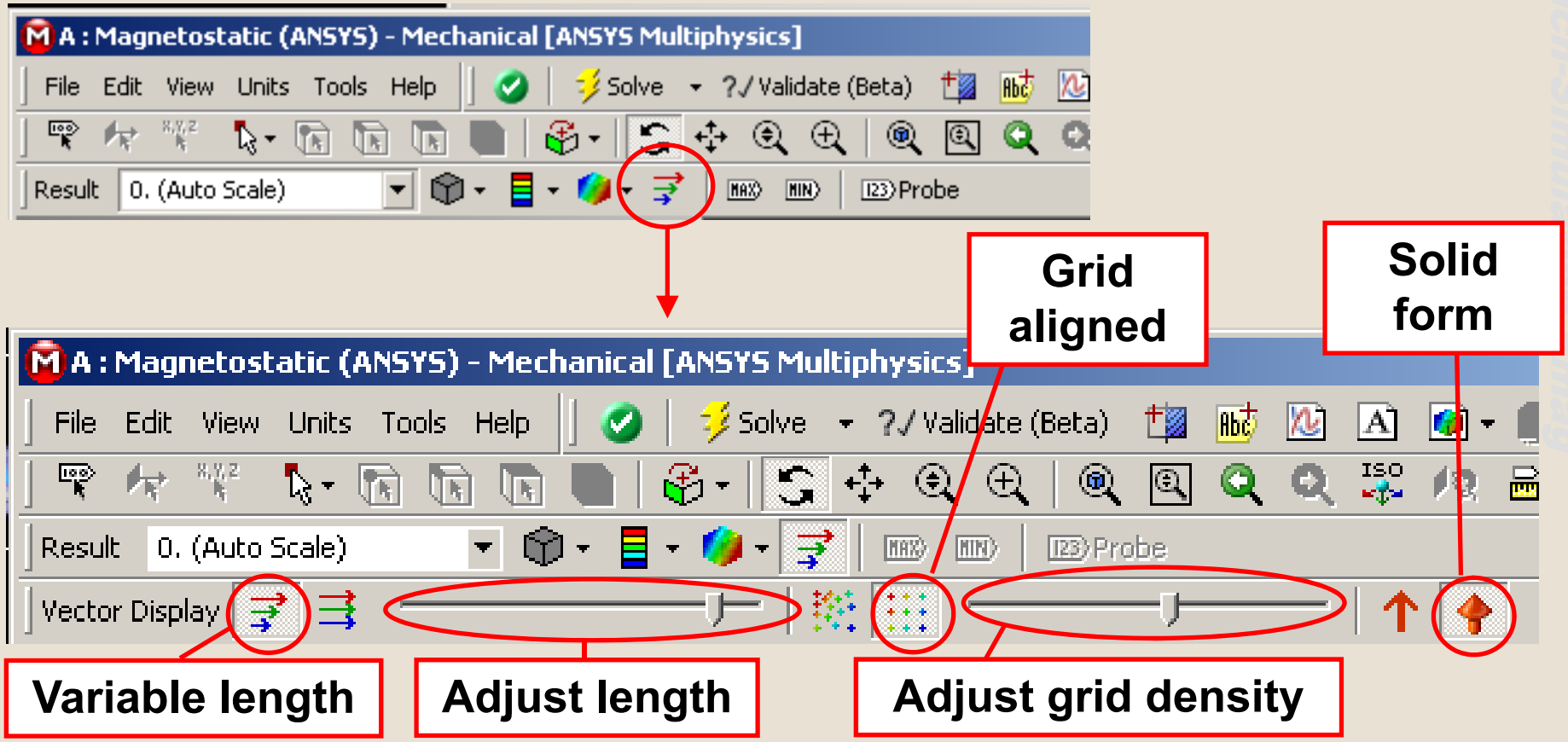

© 2010 ANSYS, Inc. All rights reserved.




Workbench-Simulation Emag

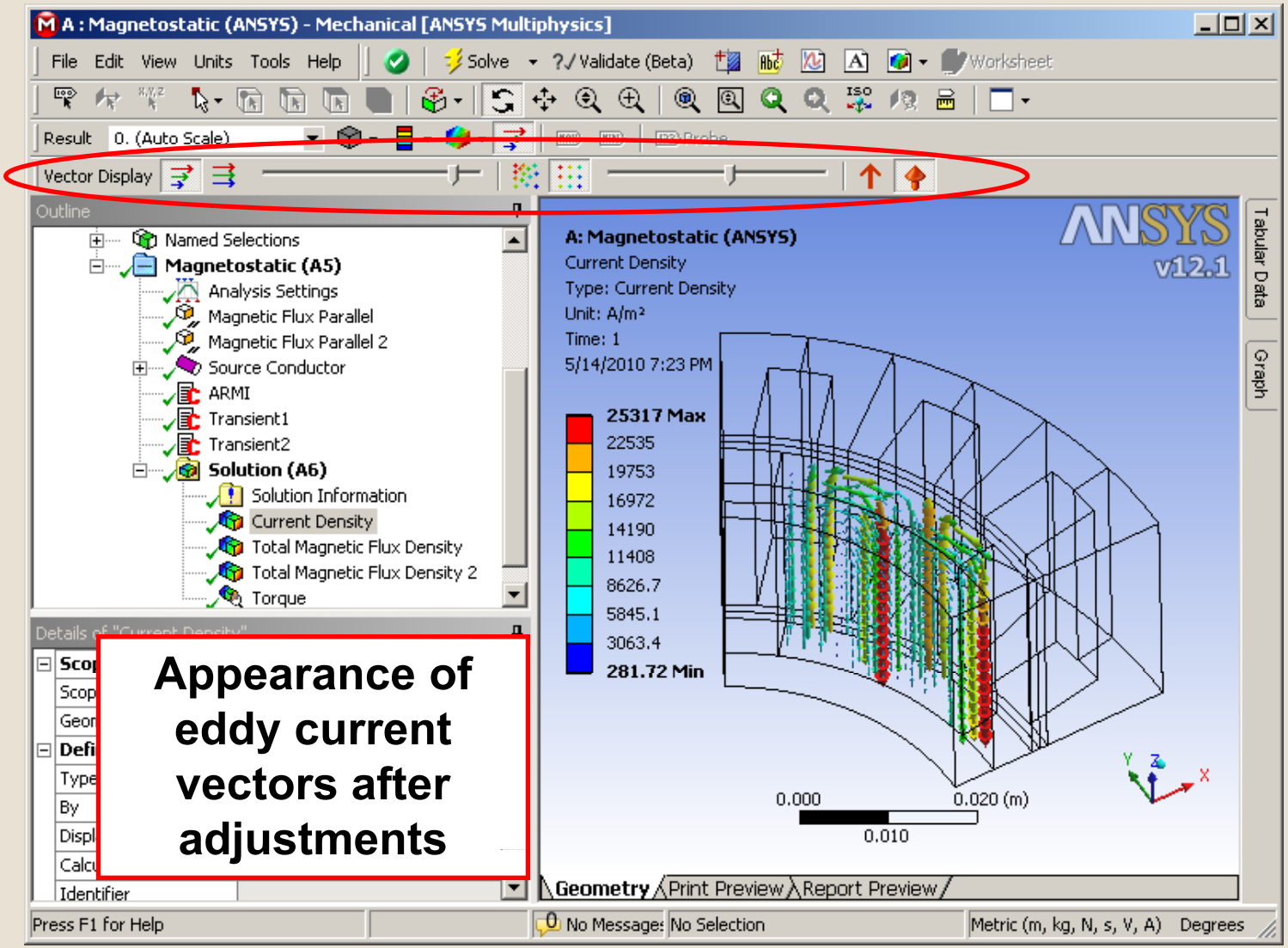

© 2010 ANSYS, Inc. All rights reserved.

**Default vector plots often need to be enhanced for clarity. Click on the vector graphics button to bring up additional controls.**



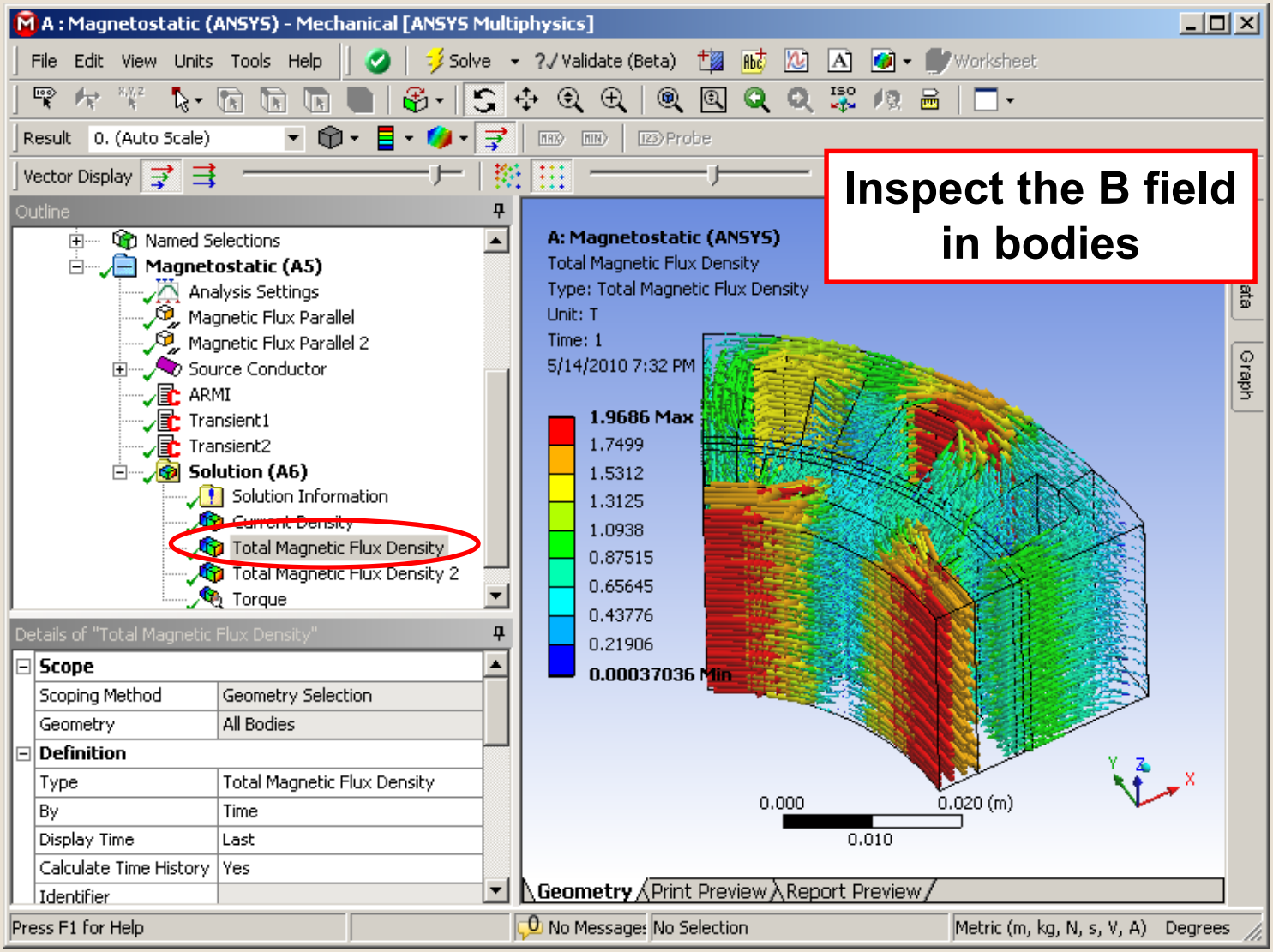

Workbench-Simulation-Level


© 2010 ANSYS, Inc. All rights reserved.



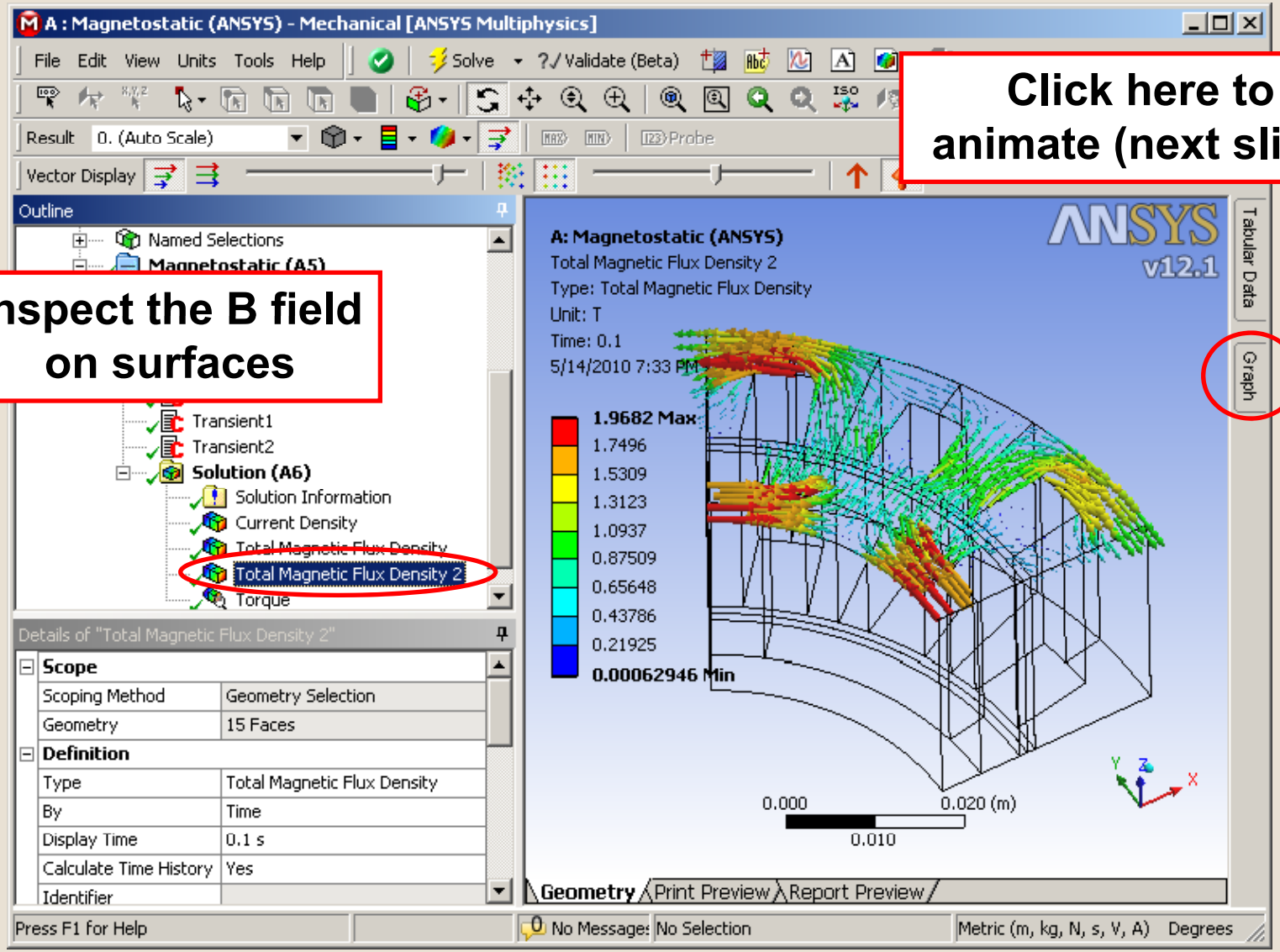

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.



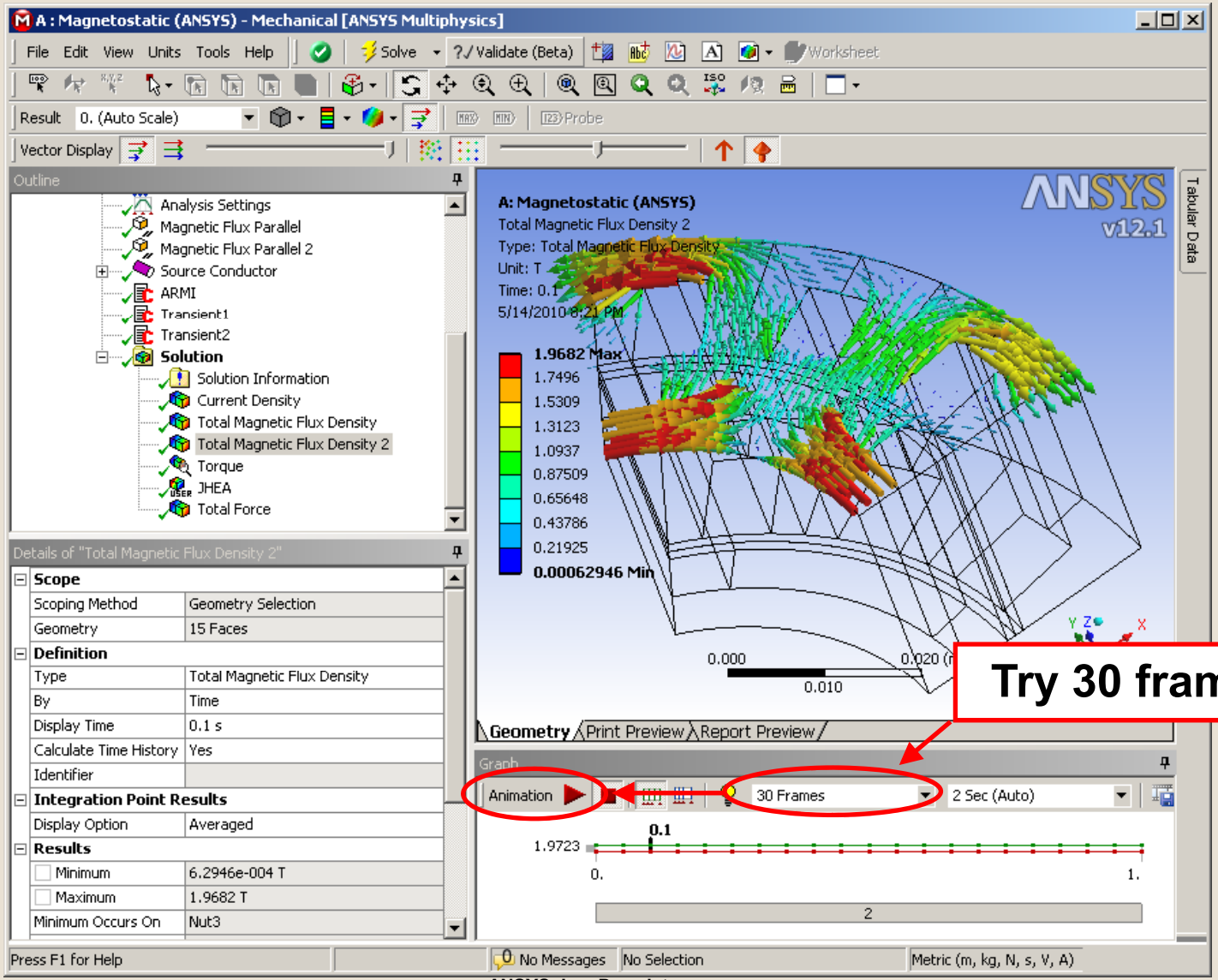


© 2010 ANSYS, Inc. All rights reserved.



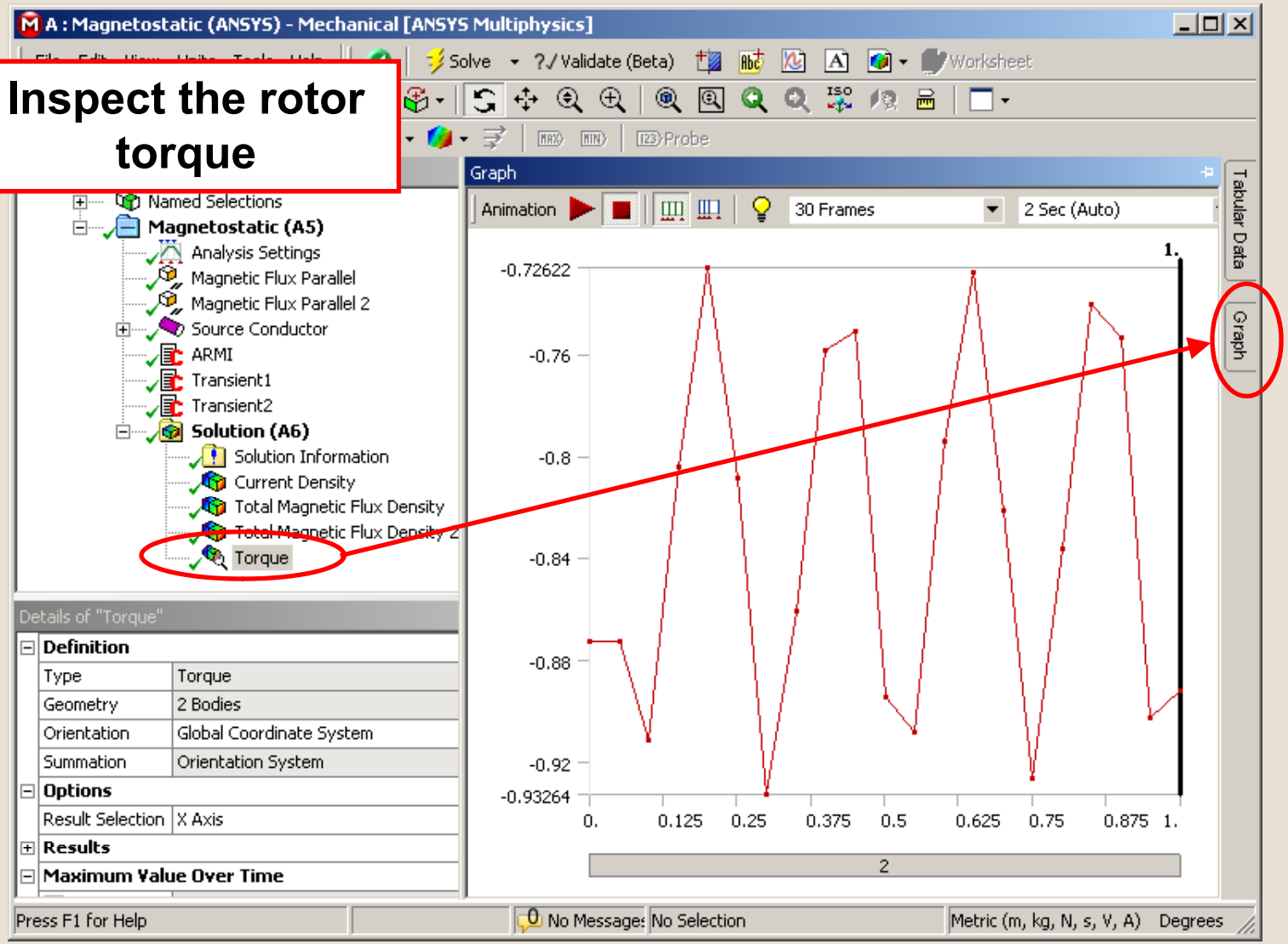


© 2010 ANSYS, Inc. All rights reserved.




Workbench Simulation Emag

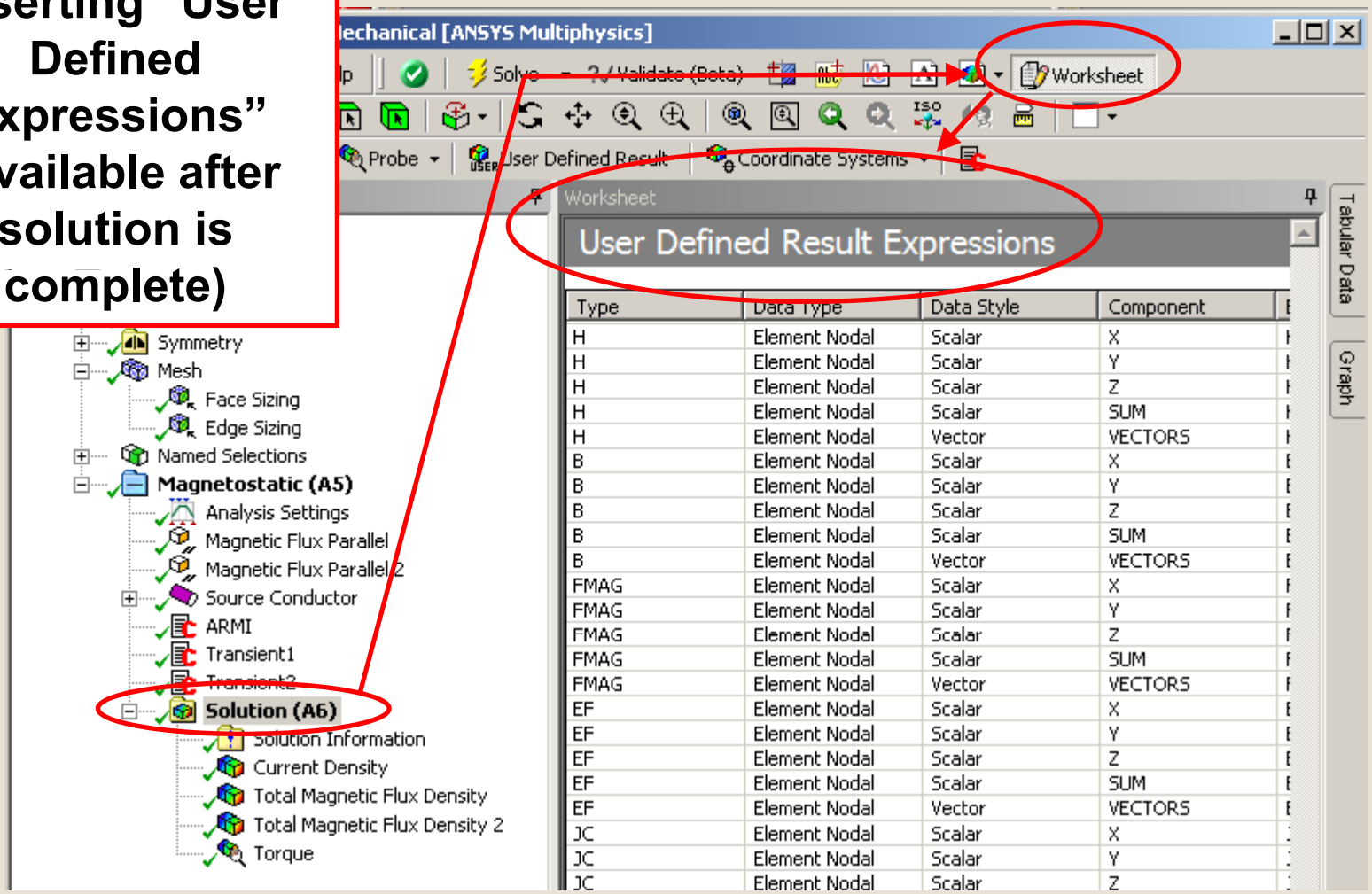

© 2010 ANSYS, Inc. All rights reserved.




Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.

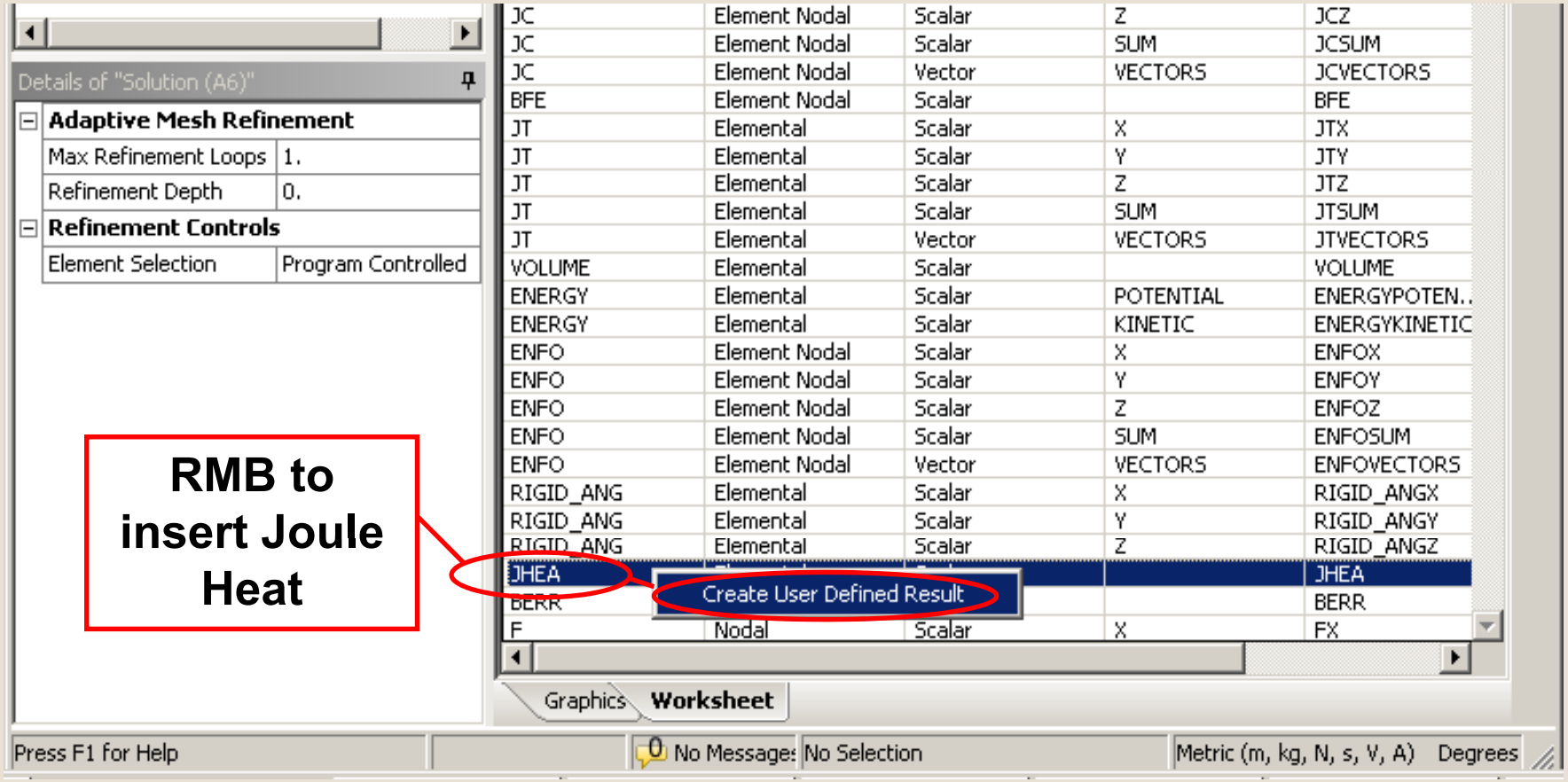
**Inserting "User Defined Expressions" (available after solution is complete)**




| Type | Data Type | Data Style | Component |
|---|---|---|---|
| H | Element Nodal | Scalar | X |
| H | Element Nodal | Scalar | Y |
| H | Element Nodal | Scalar | Z |
| H | Element Nodal | Scalar | SUM |
| H | Element Nodal | Vector | VECTORS |
| B | Element Nodal | Scalar | X |
| B | Element Nodal | Scalar | Y |
| B | Element Nodal | Scalar | Z |
| B | Element Nodal | Scalar | SUM |
| B | Element Nodal | Vector | VECTORS |
| FMAG | Element Nodal | Scalar | X |
| FMAG | Element Nodal | Scalar | Y |
| FMAG | Element Nodal | Scalar | Z |
| FMAG | Element Nodal | Scalar | SUM |
| FMAG | Element Nodal | Vector | VECTORS |
| EF | Element Nodal | Scalar | X |
| EF | Element Nodal | Scalar | Y |
| EF | Element Nodal | Scalar | Z |
| EF | Element Nodal | Scalar | SUM |
| EF | Element Nodal | Vector | VECTORS |
| JC | Element Nodal | Scalar | X |
| JC | Element Nodal | Scalar | Y |
| JC | Element Nodal | Scalar | Z |

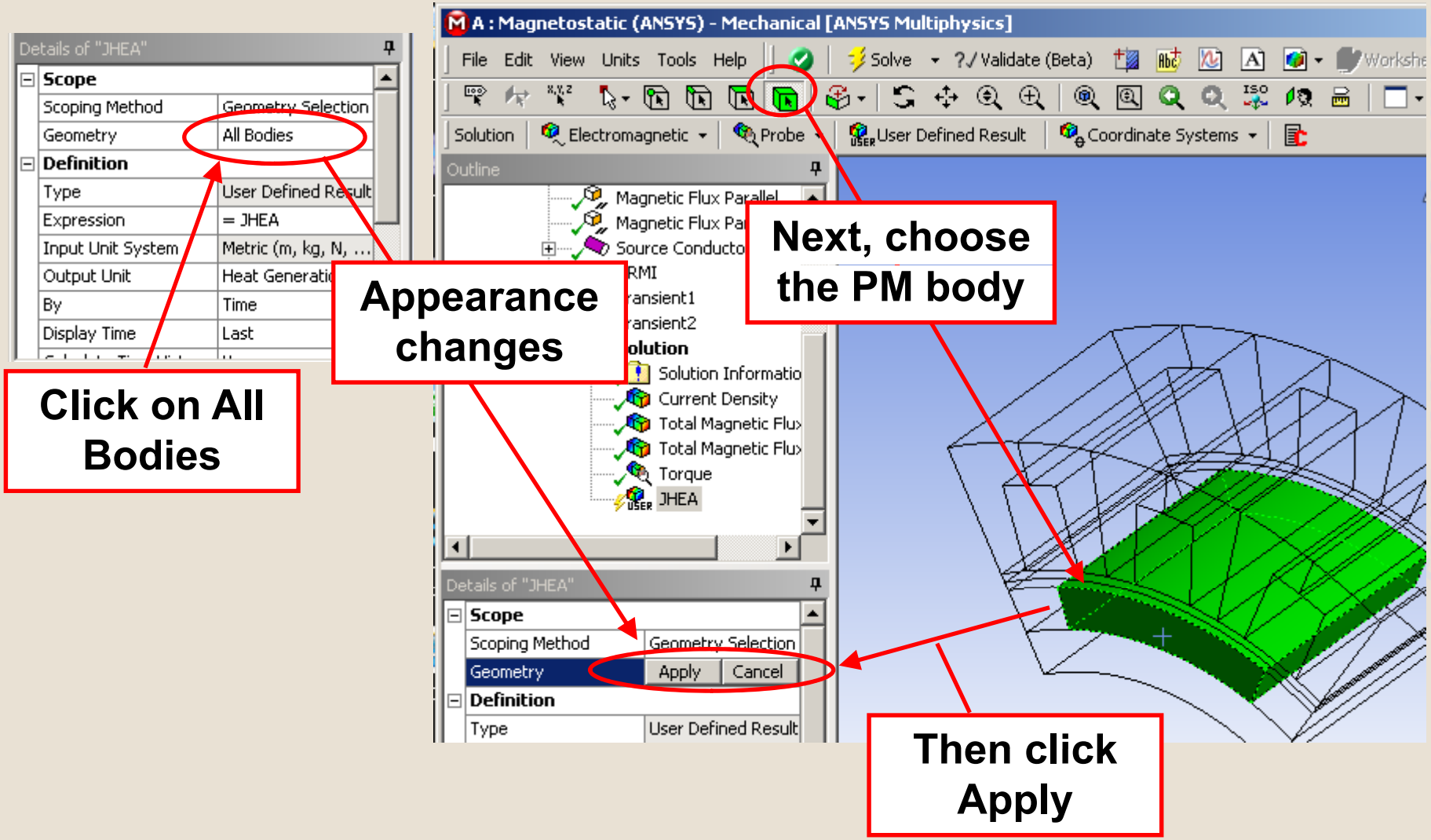
Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




Workbench-Simulation Emag

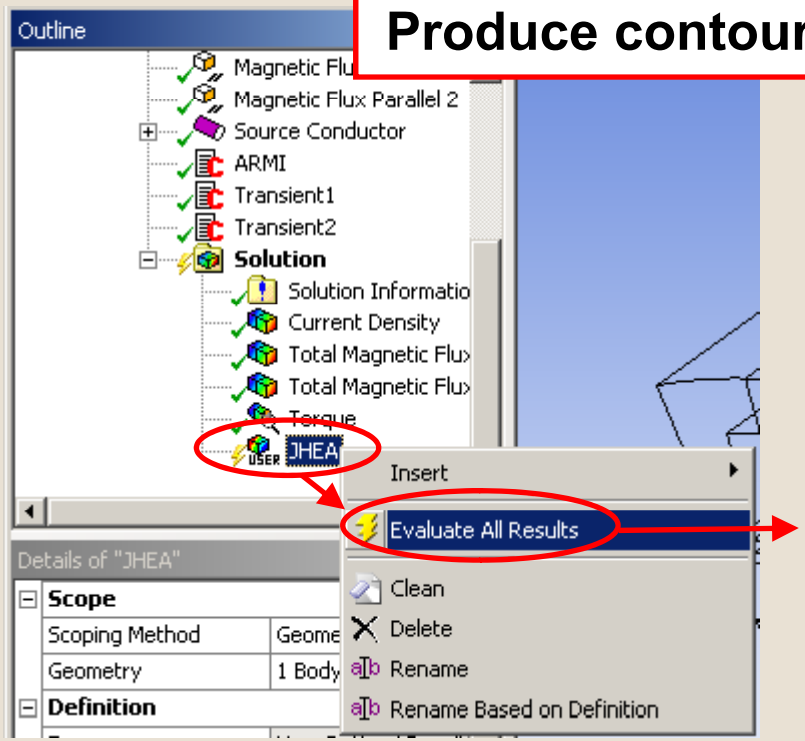

© 2010 ANSYS, Inc. All rights reserved.

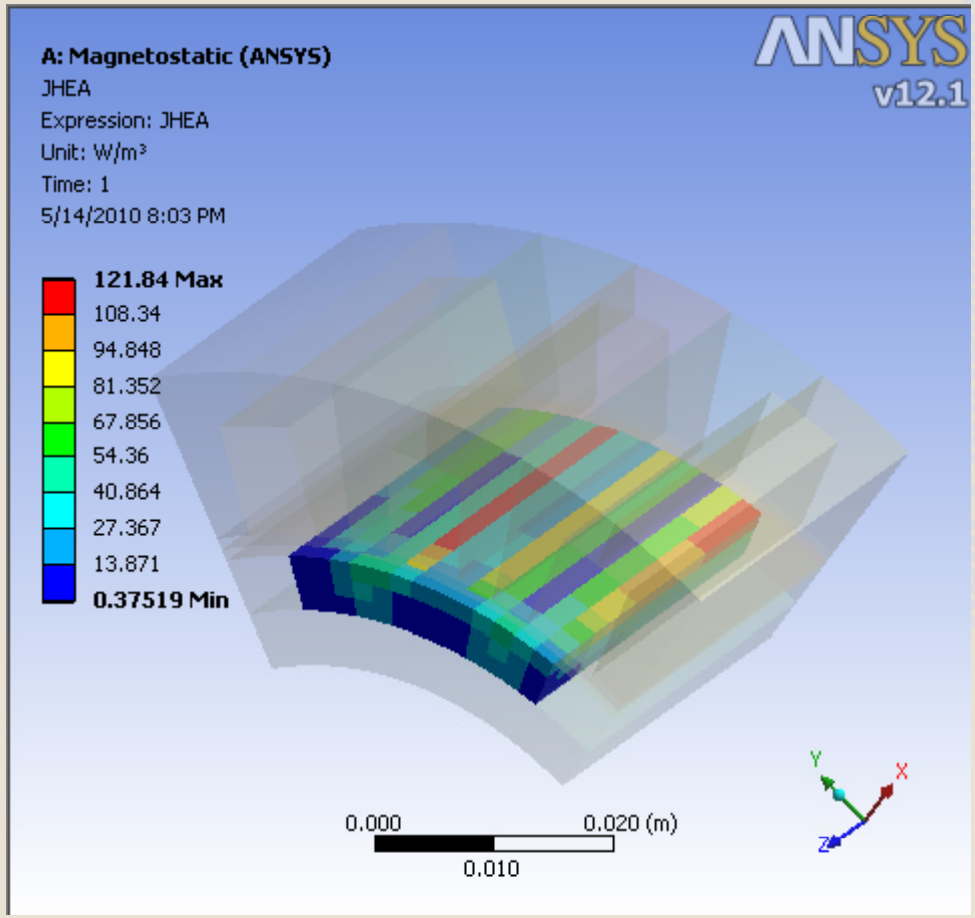




© 2010 ANSYS, Inc. All rights reserved.

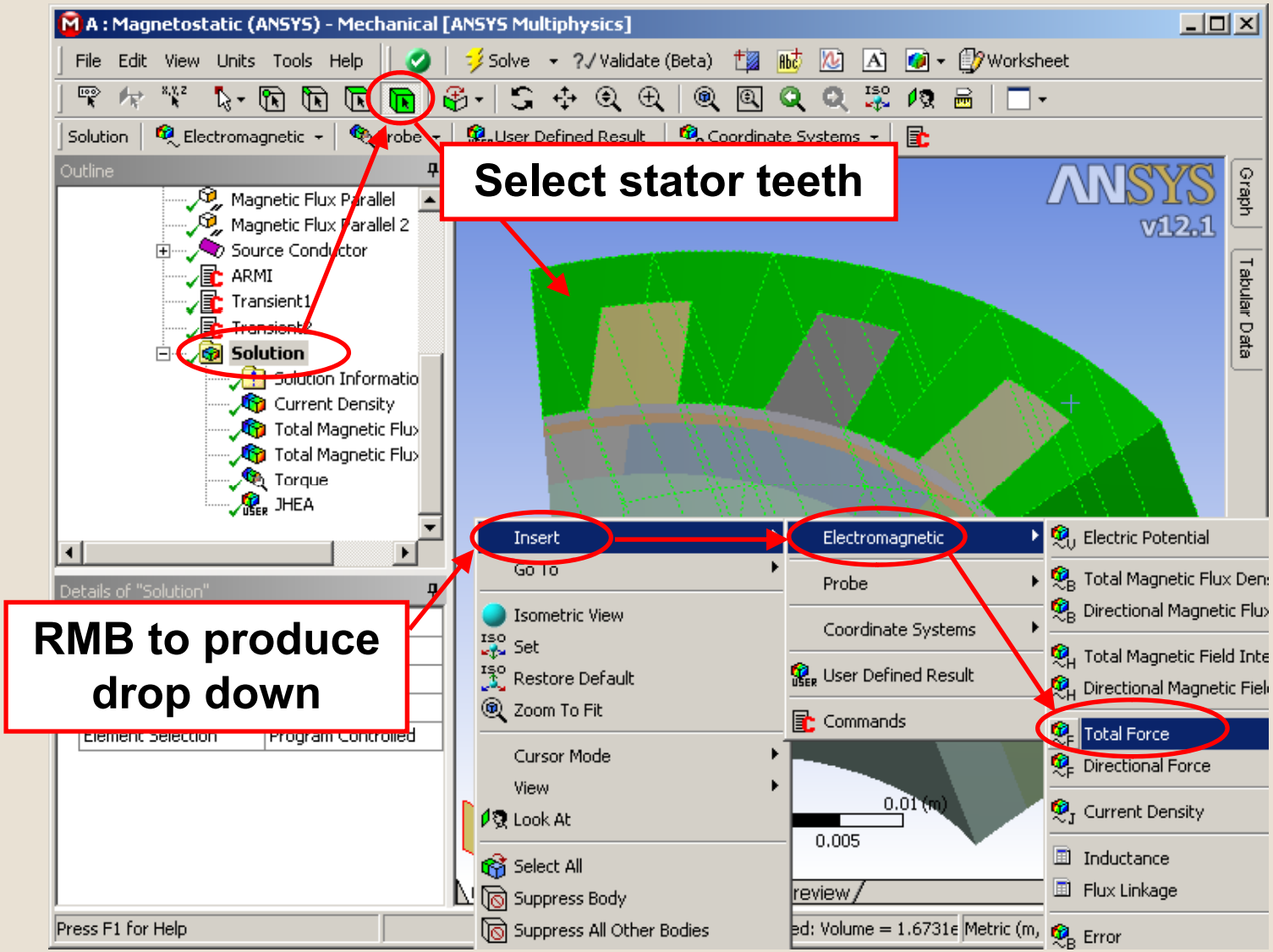
## Produce contour plot of Joule Heat



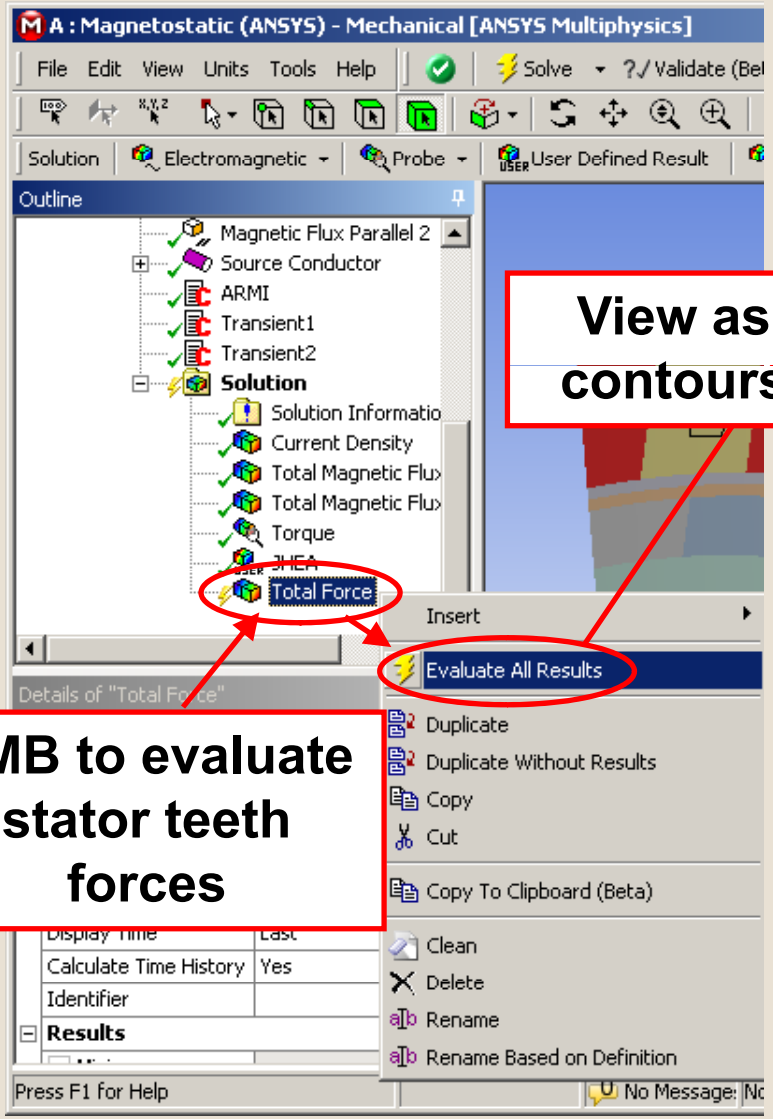


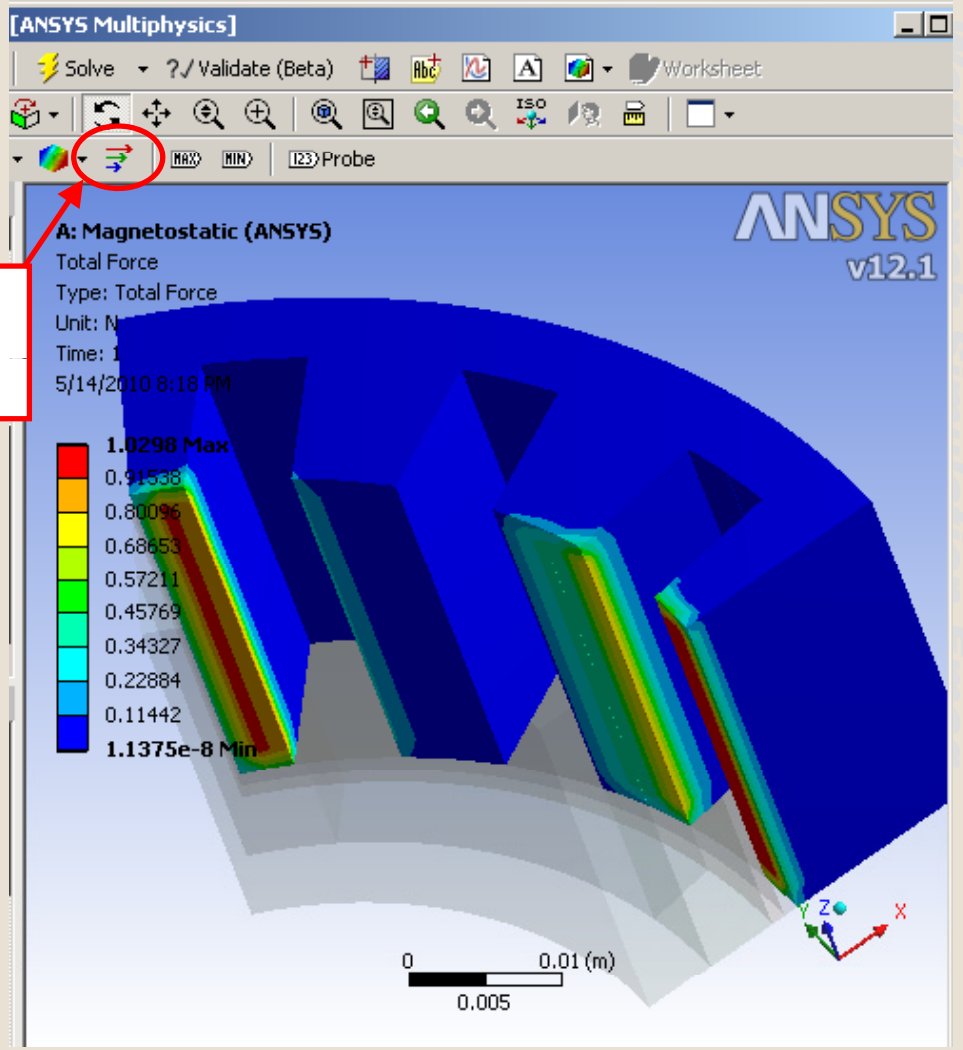
© 2010 ANSYS, Inc. All rights reserved.




Workbench Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.




**View as contours**

**RMB to evaluate stator teeth forces**


© 2010 ANSYS, Inc. All rights reserved.

# Electromagnetic Analysis




*Training Manual*

Workbench-Simulation Emag


© 2010 ANSYS, Inc. All rights reserved.