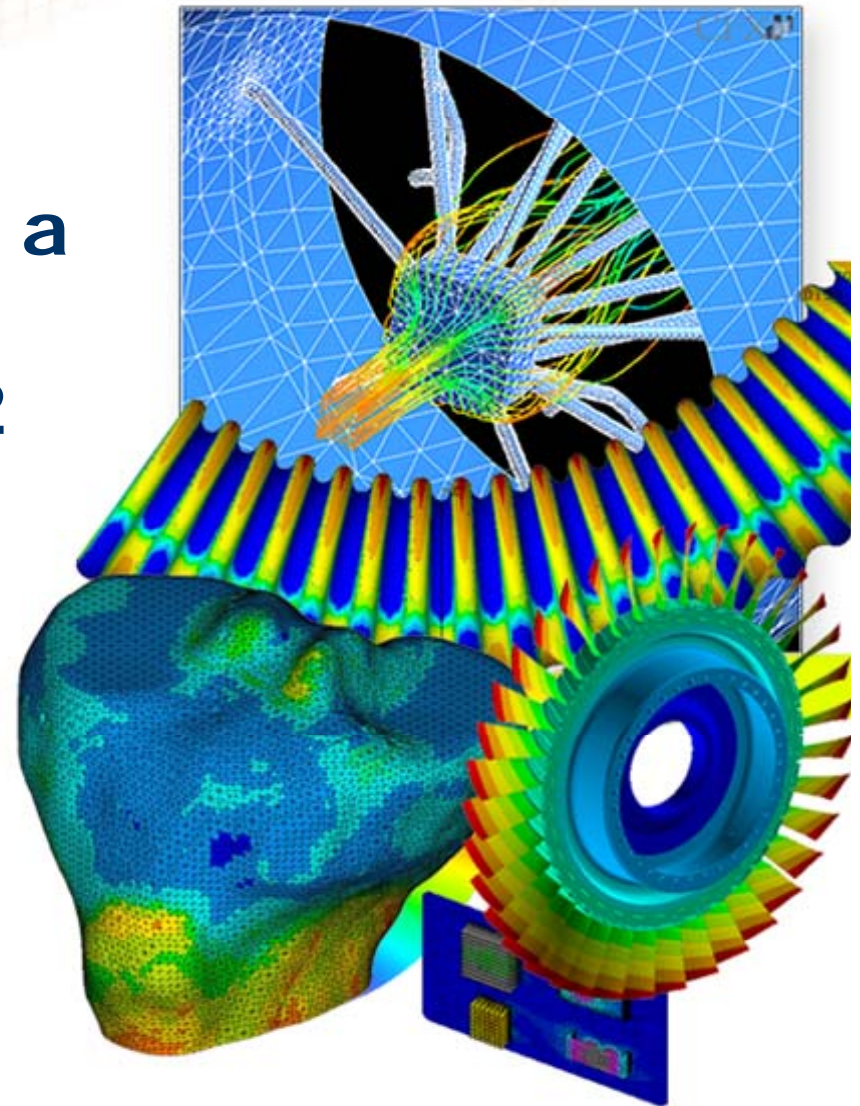


Generating Geometry of a Deformed Shape using ANSYS Workbench v12

Patrick Cunningham
August 2009



Step 1 – Updating Nodal Coordinates



- Updating nodal coordinates to a deformed shape is accomplished in ANSYS using the UPCOORD command.
- This command is not supported by the Workbench Mechanical environment and must be input using a Solution command block as shown here.
- Following the UPCOORD command, CDWRITE is executed from the Preprocessor to write out a node/element file with the updated nodal coordinates.

The screenshot shows the ANSYS Workbench interface. On the left is the Outline tree, and on the right is the Command Editor.

Outline Tree:

- Project
 - Model (B4)
 - Geometry
 - Coordinate Systems
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Displacement
 - Frictionless Support
 - Pressure
 - Solution (B6)
 - Solution Information
 - Total Deformation
 - Commands (ANSYS)

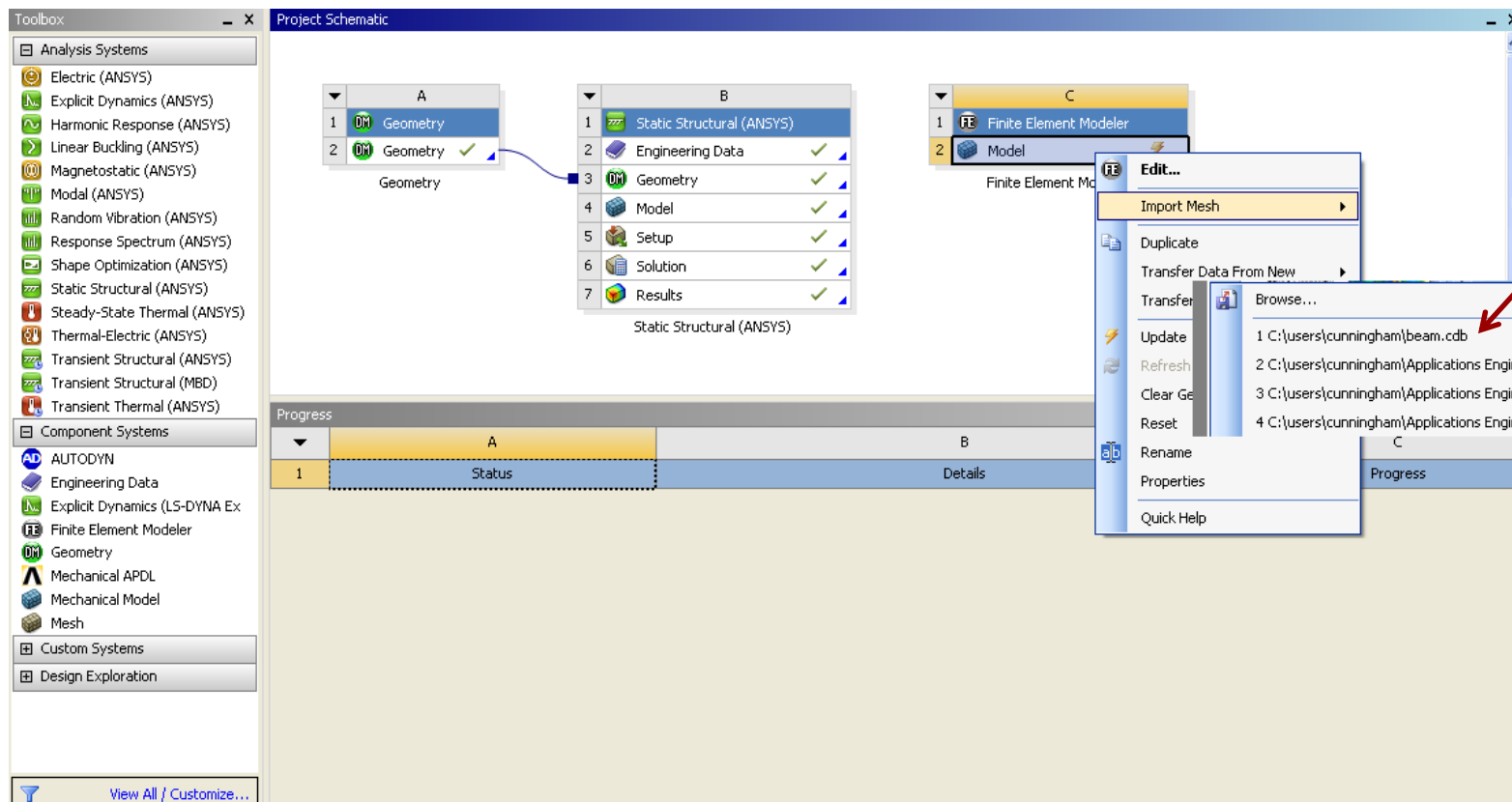
Command Editor:

```
! Commands inserted into this file will be executed immediately after the Ansys /POST1 command.  
  
! Active UNIT system in Workbench when this object was created: Metric (mm, dat, N, s, mV, mA)  
  
set,last  
upcoord,1,on  
fini  
/prep7  
cdwrite,db,c:\users\cunningham\beam,cdb  
fini  
/post1
```

Step 2– Importing the Mesh into FE Modeler

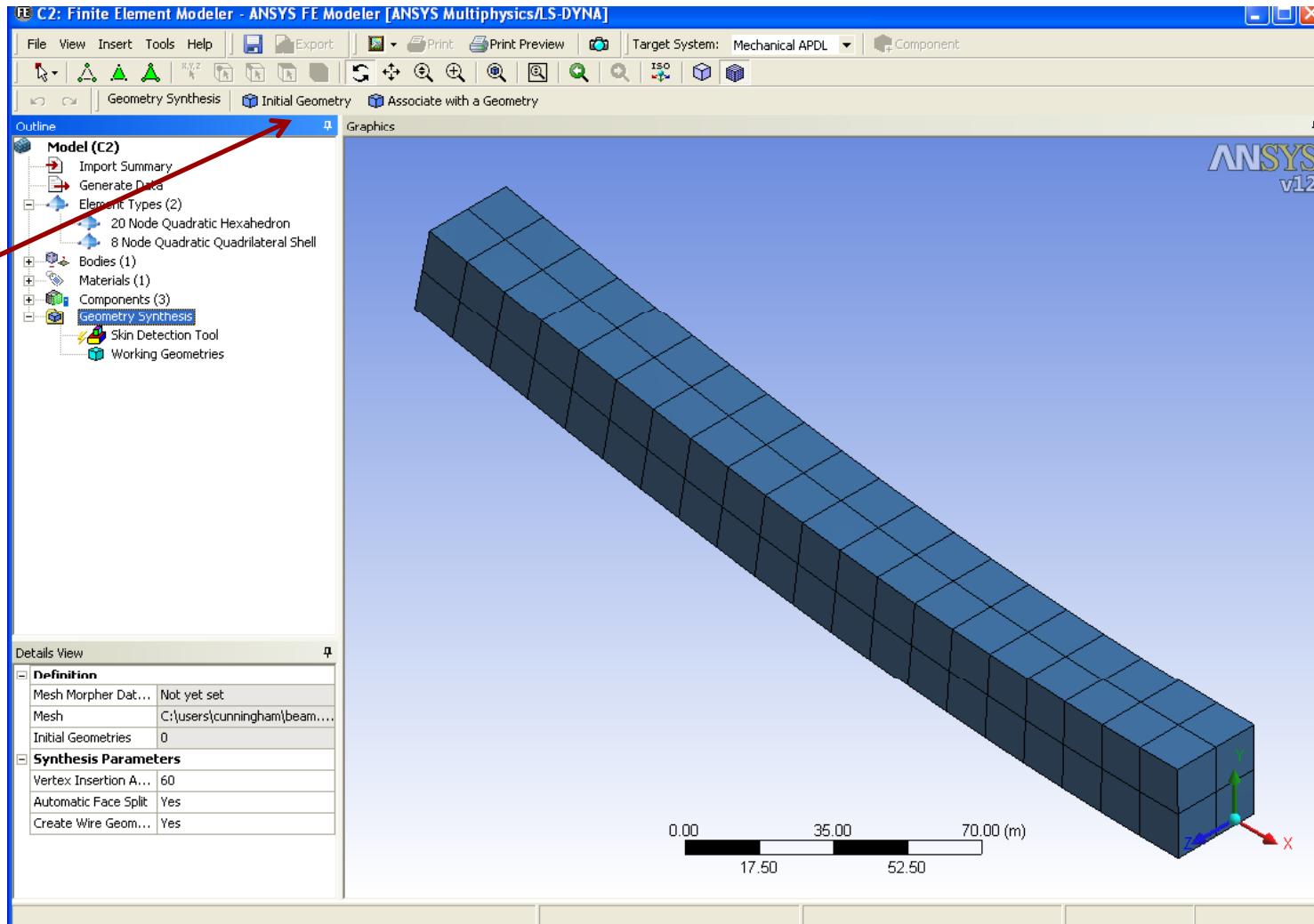


- Drag and drop FE Modeler from the Component Systems menu onto the Project Schematic (as a standalone system).
- RMB click on the Model object and select “Import Mesh”
- Browse to the cdb file generated in the Mechanical window and open.



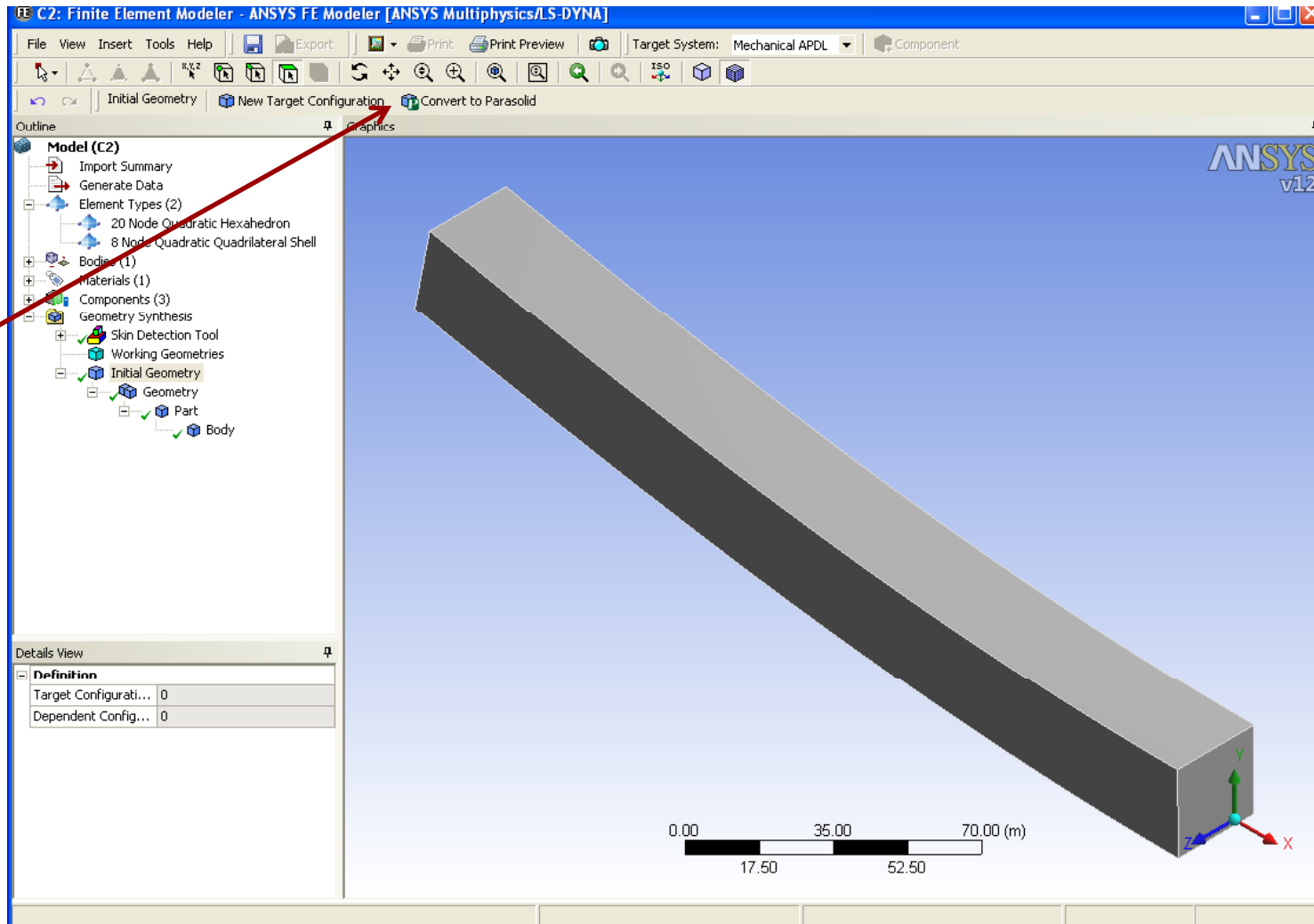
Step 3 – Creating Deformed Geometry

- Select “Initial Geometry” from the Geometry Synthesis folder.



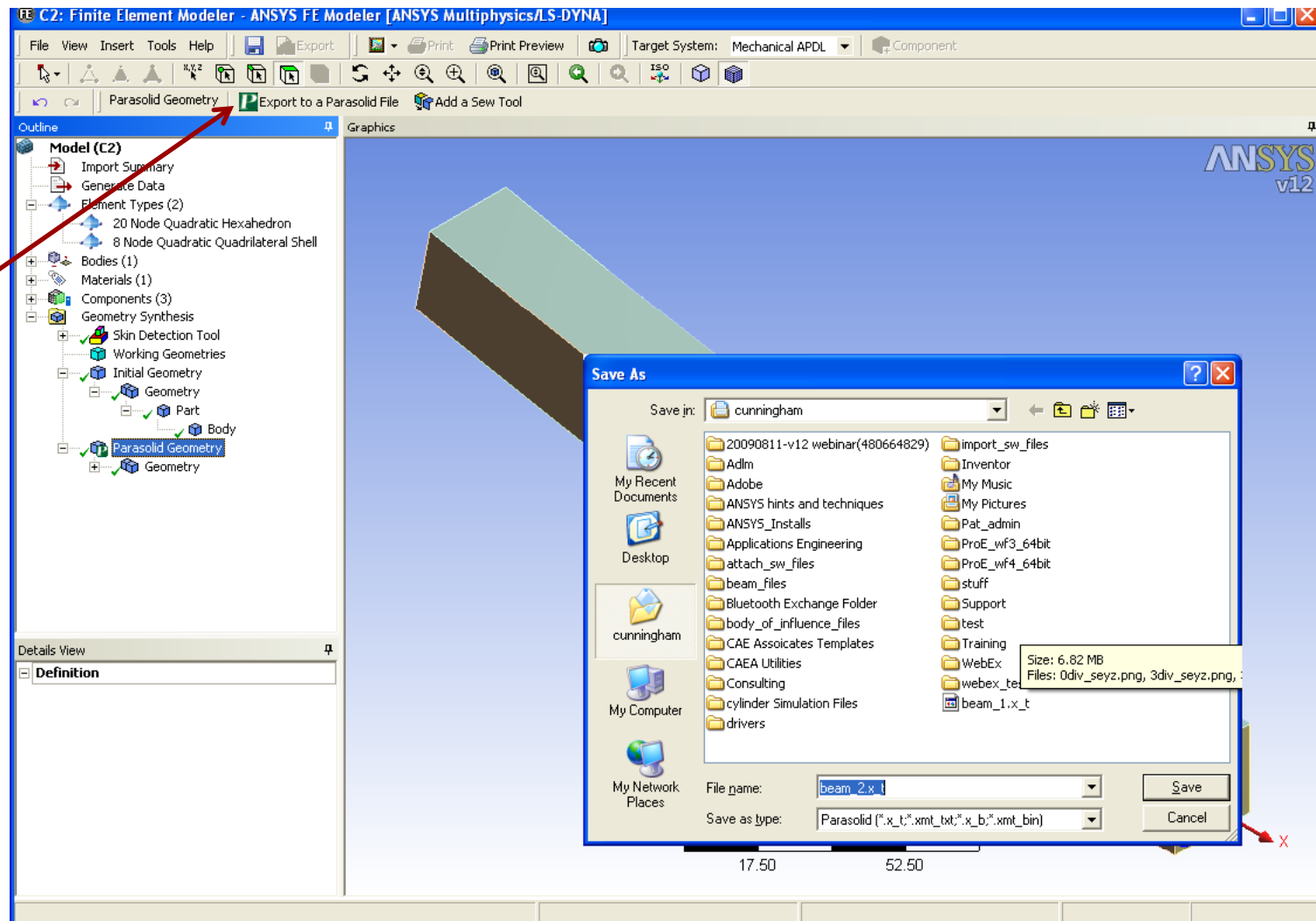
Convert the Geometry

- Select “Convert to Parasolid” from the Initial Geometry toolbar.



Export the Parasolid File

- Select “Export to Parasolid File” and specify filename and location.



Import the Parasolid File

- The parasolid file can then be imported into another geometry application such as Solidworks.

