SimuTrain® is ANSYS training on demand!

- 🎓 Basic & Advanced Level Training Modules
- 🎥 ANSYS & SimuTech Training Videos
- 📕 Lectures, Tutorials & Workshops
- 🖥 1 hour Web-based ANSYS Sessions to Practice Hands-on Workshops

Learn more about SimuTrain (/training/simutrain-ansys-training-portal) or get started today by purchasing your subscription (https://secure.2checkout.com/order/checkout.php?PRODS=13329130&QTY=1&CLEAN_CART=all&CARD=2&SHORT_FORM=1).

ADDITIONAL RESOURCES:

ANSYS Software (/ansys-software)

ANSYS Training Courses (/training/ansys-training-courses)

Events & Webinars (/events)

ANSYS Consulting (/consulting-services)

STAY CONNECTED:

Newsletter (https://go.simutechgroup.com/electronic-opt-in)

Blog (/blog)

[in] (https://www.linkedin.com/company/simutech-group/)

(https://www.facebook.com/SimuTechGroup/) ▶ YouTube

(https://www.youtube.com/c/SimuTechGroup)

POST-PROCESSING APDL MODELS INSIDE ANSYS® WORKBENCH V15.0

Posted in Tips & Tricks - Finite Element Analysis (FEA) articles (/tips-and-tricks/fea-articles)
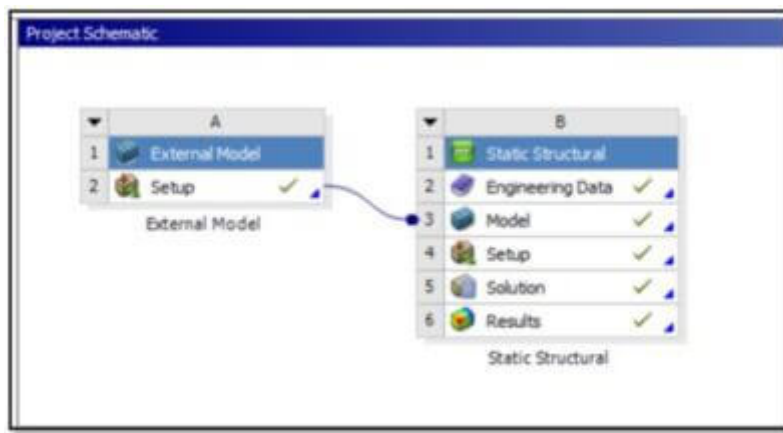
Figure 1: External Model in ANSYS Workbench Mechanical v15.0

ANSYS Workbench Mechanical v15.0 supports "External Model" – a facility for directly importing an ANSYS Mechanical APDL model's CDB mesh file into Workbench for subsequent study or results postprocessing in a Workbench Mechanical schematic. This includes the ability to read a results RST or RTH file that corresponds to the CDB file that has been attached as an External Model. The approach applies to solid and shell models. This article describes an approach with solid elements.

This article reviews steps in creating a CDB file in ANSYS Mechanical APDL, and reading in the CDB file in Workbench, plus importing results from an associated RST results file. This permits the Workbench Mechanical environment's tools to be employed in postprocessing a model that originated in ANSYS Mechanical APDL, as well as generating a report to automate generation of data that might be of interest in the appendix to an engineering report. Good quality modern graphics images of models, animations, results scoping, and other features of the Workbench Mechanical interface may be of interest to users.

Using an ANSYS APDL External Model in Workbench 15.0

Workbench Mechanical at release 15.0 has a new External Model component system that can take a solid or shell finite element mesh from a CDB file, and generate a solid model with a mesh for use in Workbench Mechanical. Users should review the ANSYS v15.0 Help system for information on this External Model feature. It contains information on reading an external model, as well as on rotating and translating the geometry that is imported.
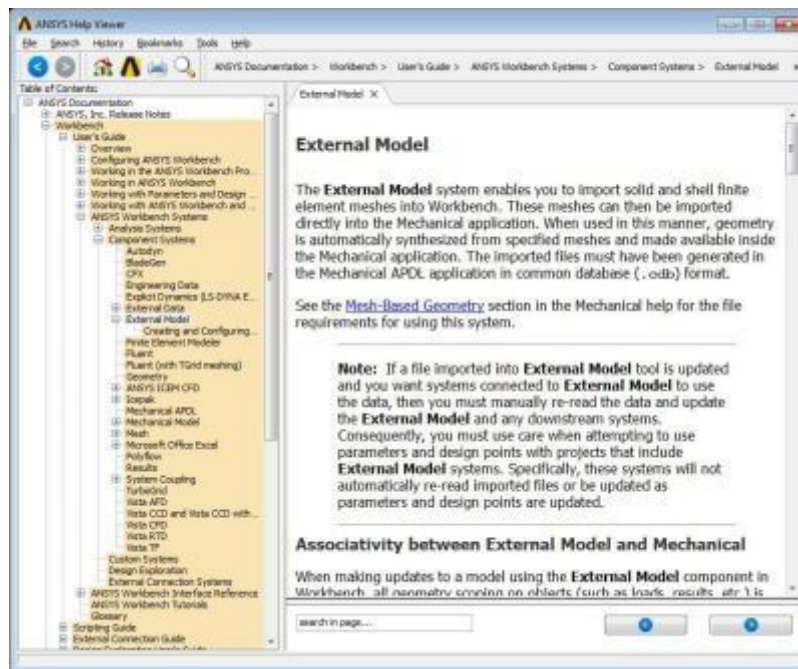
Figure 2: External Model Help Documentation in ANSYS v15.0

The External Model approach for model review is not an officially supported feature of the ANSYS software, so users intended to use this facility must do their own quality control if this feature is employed.

To use this feature, a CDB file that contains a supported mesh type must be generated in ANSYS Mechanical APDL. The CDWRITE command of ANSYS is used in /PREP7 to generate the CDB file. Arguments are set to generate the mesh, without the geometry data, for example:

CDWRITE, *Option, Fname, Ext, --, Fnamei, Exti, Fmat*
Writes geometry and load database items to a file

The following example generates a mesh file only (no associated geometry):

CDWRITE,DB,myfilename,CDB

The following example model contains SOLID185 8-node brick solid structural elements subject to pressure loading and to constraint on node movements. It could be solved in ANSYS Mechanical APDL, in which case an RST structural results file would be generated in the working directory:
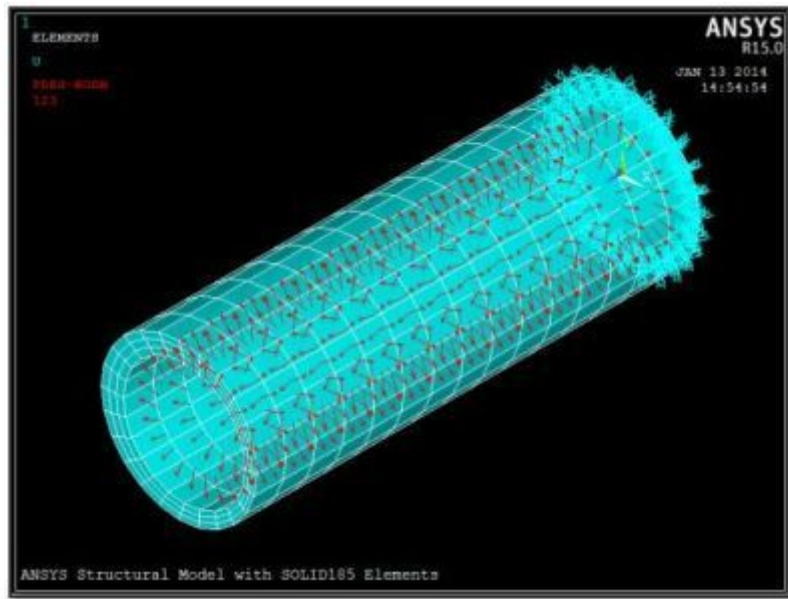
Figure 3: ANSYS Mechanical APDL Model with SOLID185 Elements and Loads

In the ANSYS Mechanical APDL interface, the following menu item will generate a CDB file for the above mesh:
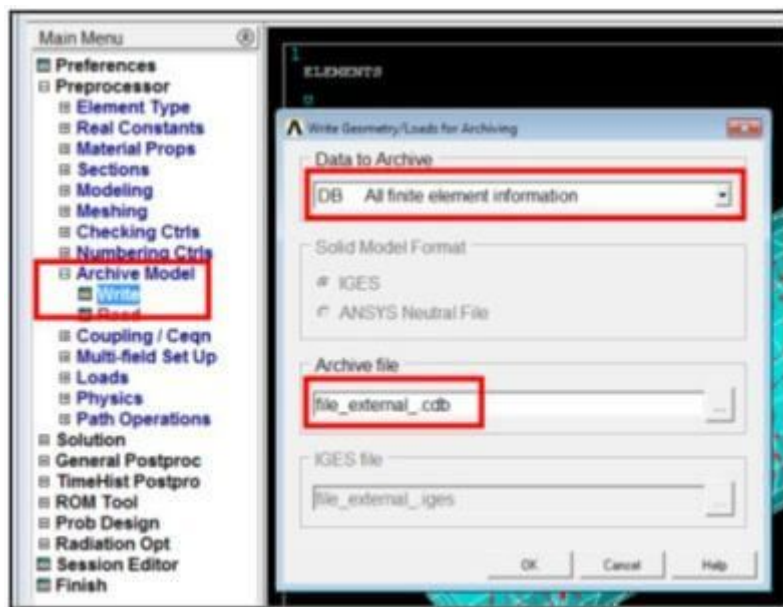

Figure 4: Creating a CDB File of the Mesh via the ANSYS Mechanical APDL Menu System

Workbench v15.0 is capable of reading the CDB file of the mesh, fitting a geometry to the mesh, and sending the resulting model of geometry and mesh into Workbench Mechanical for study in that environment, making the various tools of Workbench Mechanical available for further studies on models that originate in the APDL interface.

For simplicity, start by setting Units to those employed in the CDB file—the user must know this in advance:
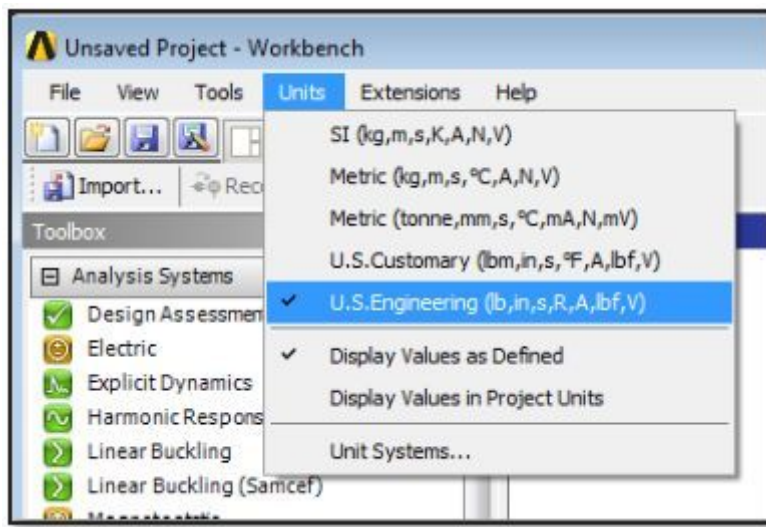
Figure 5: Set Units before Starting

In the Workbench project page, drag an External Model component system into the Project Schematic.
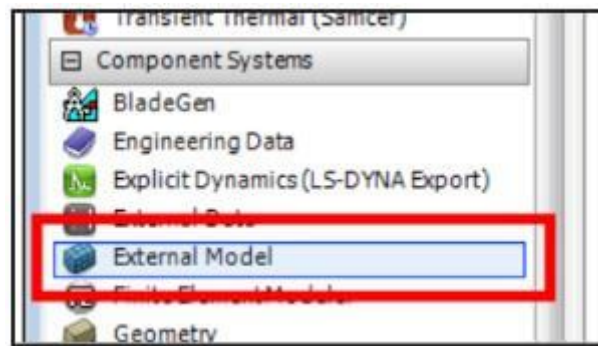

Figure 6: Drag External Model into the Project Schematic

Then, drag an appropriate analysis system into the Project Schematic, such as the Static Structural. Next, drag the Setup cell of the External Model to the Model branch of the Static Structural system:

Figure 7: *Drag* the Setup Cell of the External Model to the Model Cell of the Static Structural System

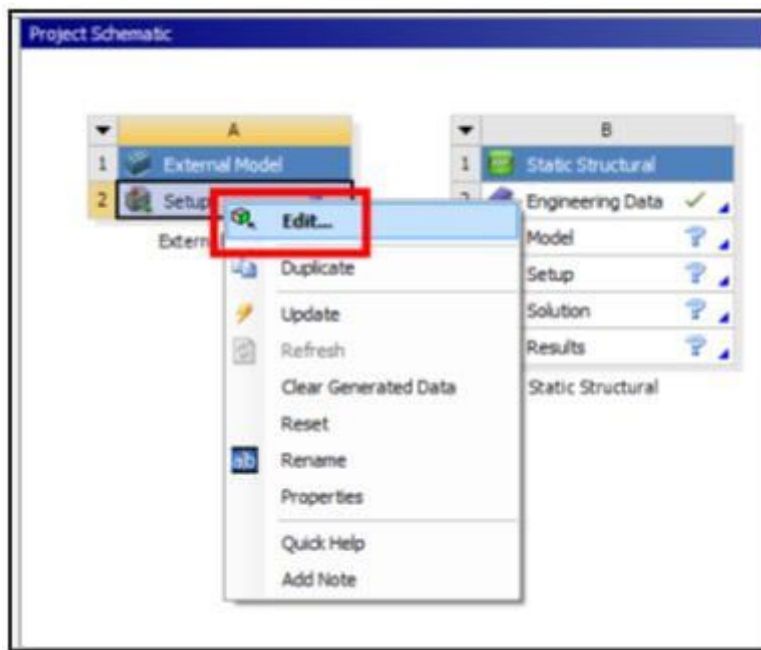Right-click the Setup cell of the External Model system, and choose Edit:

Figure 8: Edit Setup for the External Model Schematic

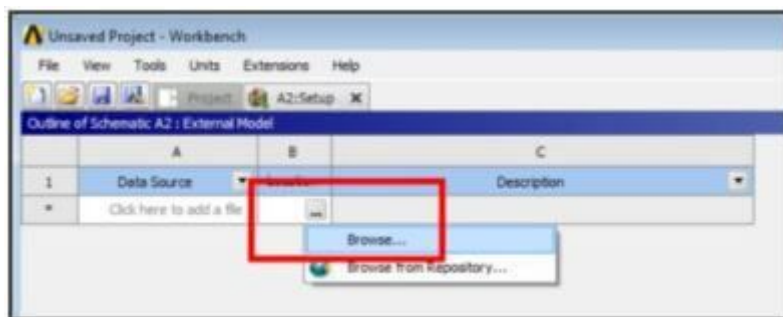A Data Source window opens; Browse to the CDB file of interest:

Figure 9: Browse to the CDB File to be Imported

Find the CDB file for import, click it, and click Open. Now, close the window by clicking the X:
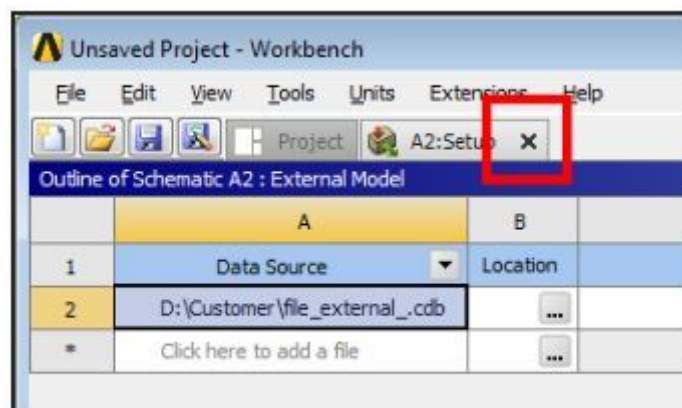

Figure 10: Exit this Window

Click the linked Model cell in the Static Structural system, right click, and choose Properties:
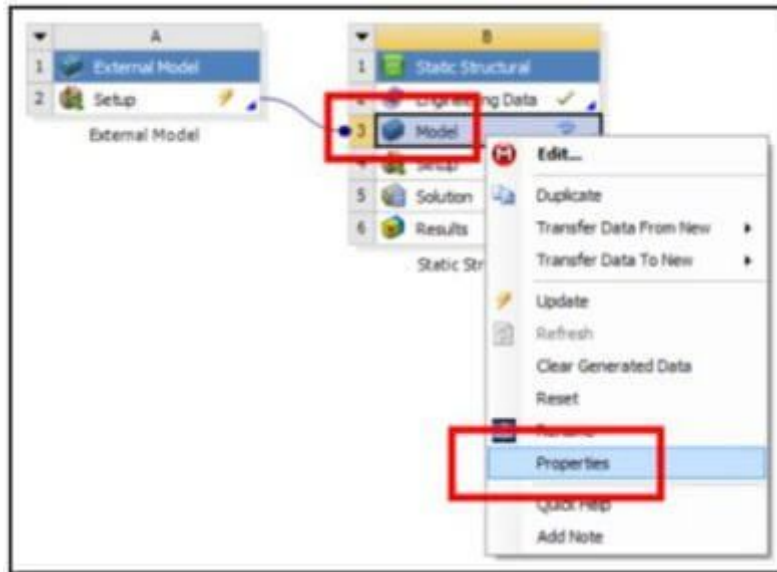

Figure 11: Adjust Properties for the Geometry to be Fitted

In the Properties window that opens on the right, choose the Length Unit as in the CDB file, tweak the Tolerance Angle if needed to aid in fitting geometry faces to the mesh in a complex model, and indicate whether the analysis is 3D or 2D. Nodal Components will be included by default and might be desired to use as Named Selections:
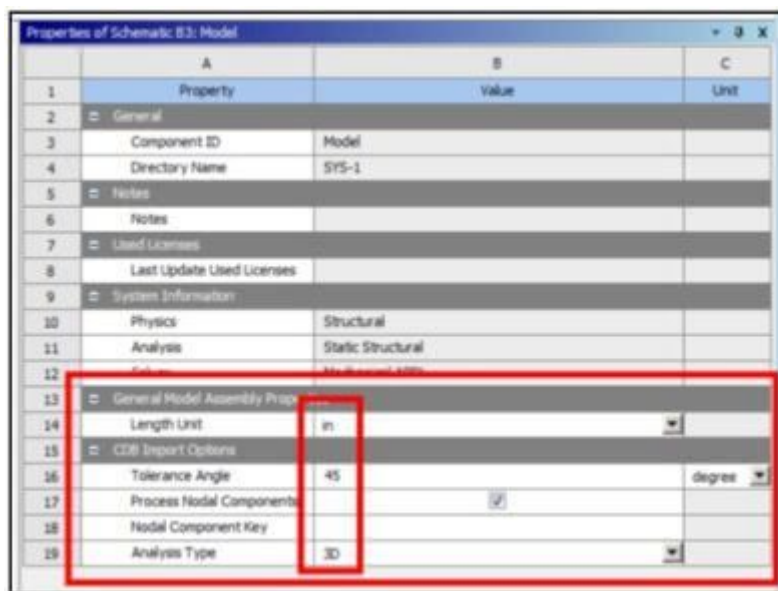

Figure 12: Model Properties and CDB Import Options

Note the "Process Nodal Components" entry above. Nodal components in the APDL model (see the CM command) become Named Selections in Workbench Mechanical, corresponding to faces in 3D. To postprocess results on these components, ensure the check mark is set as in the figure above.

Now, the External Model can be updated by clicking Setup, right-clicking, and choosing Update.
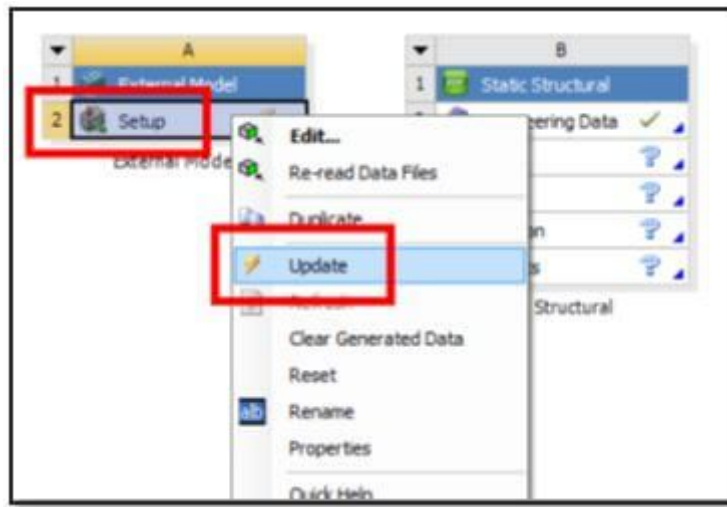
Figure 13: Updating the Setup Cell

Geometry will have been fitted to the mesh, and the Static Structural system can be opened to commence work:
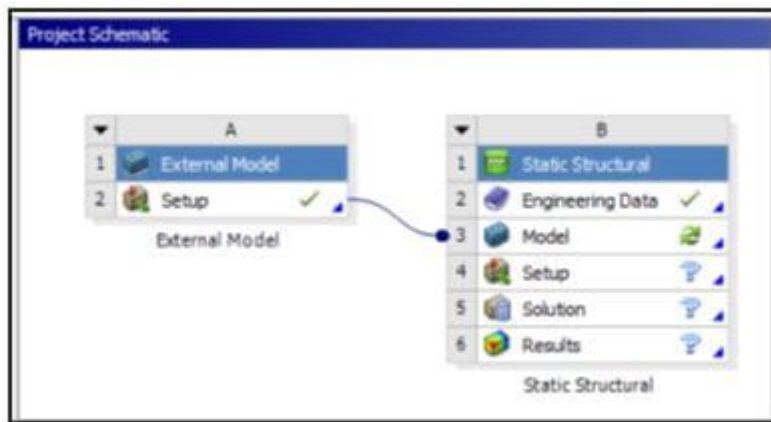

Figure 14: Ready to Open Workbench Mechanical for the Resulting Model

Right-click Model in the Static Structural system, and choose Edit:
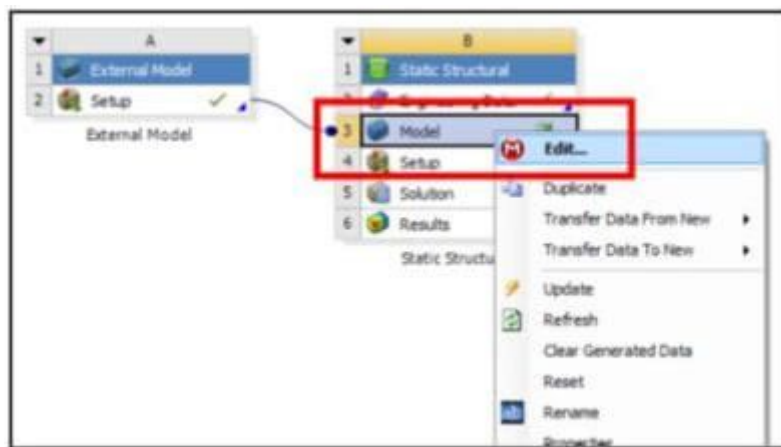

Figure 15: Open Workbench Mechanical

The Workbench Mechanical window is exposed. Elements from the CDB file are present, as is the geometry that was fitted to the elements. The materials used are not those of the CDB file—they are taken from the Engineering Data branch of the Static Structural system, and defaults will be assigned. Loads are not read from the CDB file, either. Note that the Static Structural system in the figure below is not ready to be solved —neither loads nor boundary conditions are present.
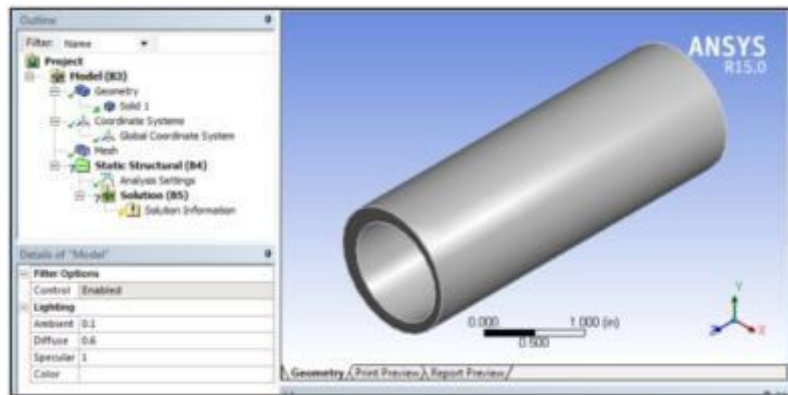

Figure 16: Meshed Model Imported into Workbench Mechanical—Not Ready to Solve

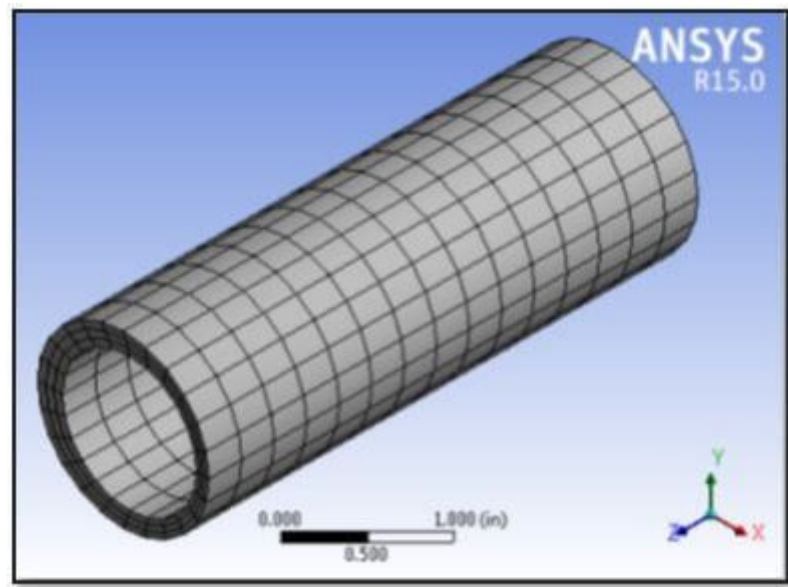Here is the mesh that came from the CDB file in this example:

Figure 17: Mesh on which Geometry Was Built in Workbench

As stated neither loads nor materials are imported from the CDB model. At this point, a user could assign Engineering Data materials to geometric entities, place loads and boundary conditions on the model, and solve the FEA model. Alternatively, the CDB model's RST results file can be imported, which will be illustrated next.

The model indicates that it is not ready to be solved because of the absence of loads and boundary conditions:
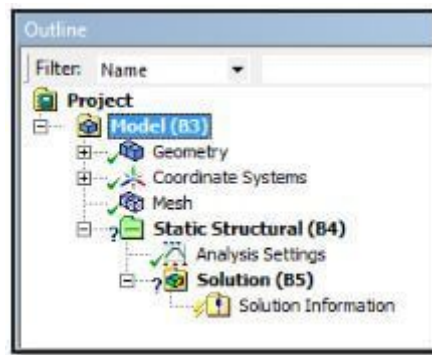
Figure 18: The Environment Is Not Ready to Solve

Place an empty APDL Commands Object in the Static Structural environment. Its presence lets Workbench Mechanical assume loading exists, so that a user can load results from an RST file. This is seen in the figure below:
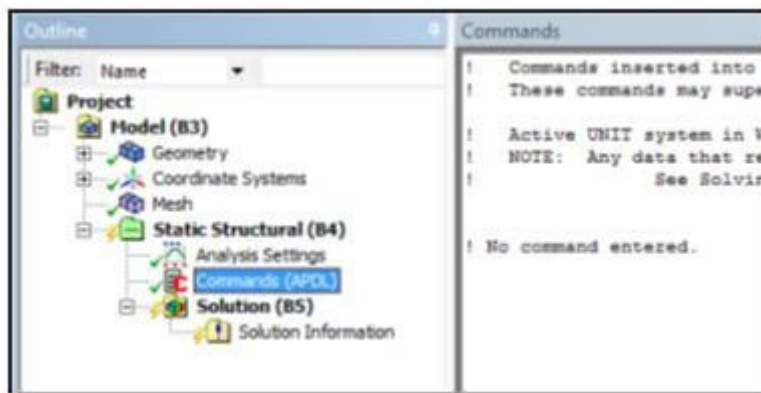

Figure 19: An APDL Commands Object Lets a User Continue

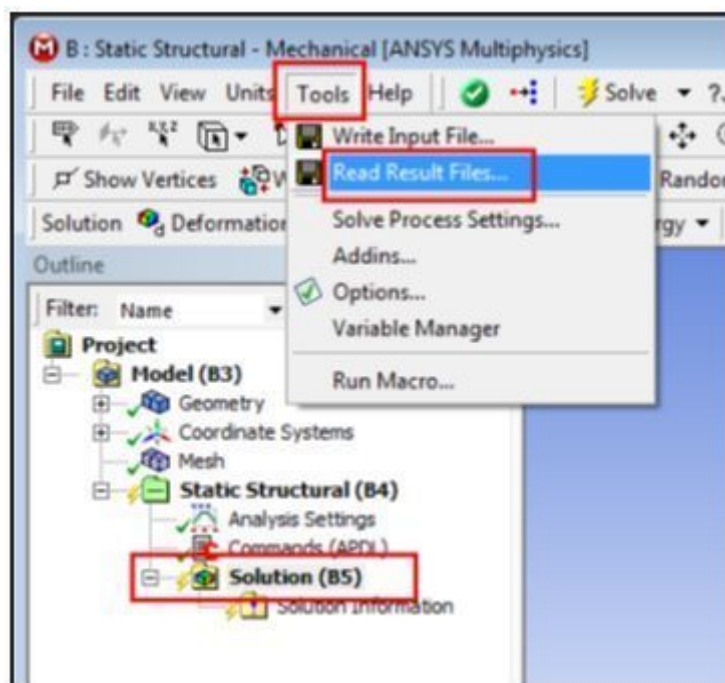Highlight the Solution branch, and go to Tools > Read Results Files… to access the RST file:

Figure 20: Access the RST File of the CDB Model

The RST file is selected, usually from the working directory in which the CDB file was found. Note that the .ERR error file for the model must exist there. If none exists, a dummy file with the correct job name can be created there, using the .ERR extension.
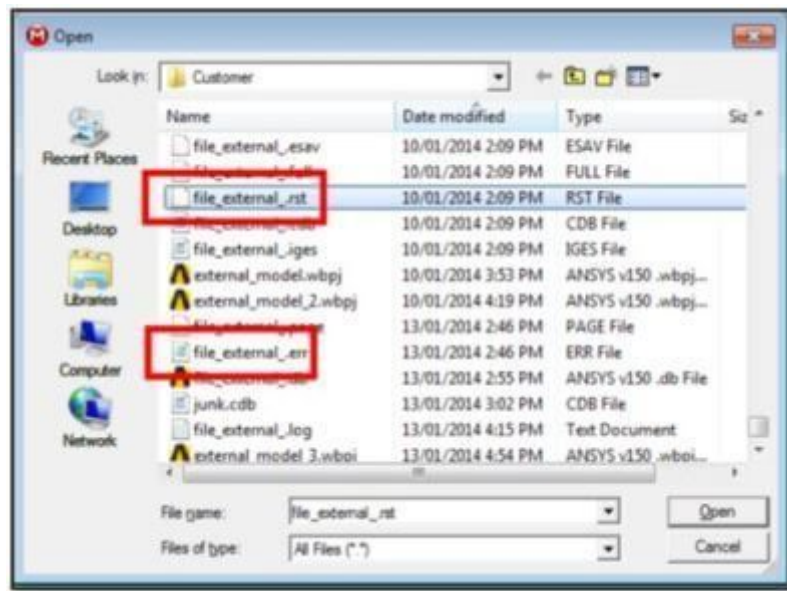

Figure 21: Select the RST File for the CDB Model

The above window closes when the Open button is clicked. This is followed by a dialog box requesting that the user indicate the Units employed... the user must ensure that the correct choice is made, and click OK:
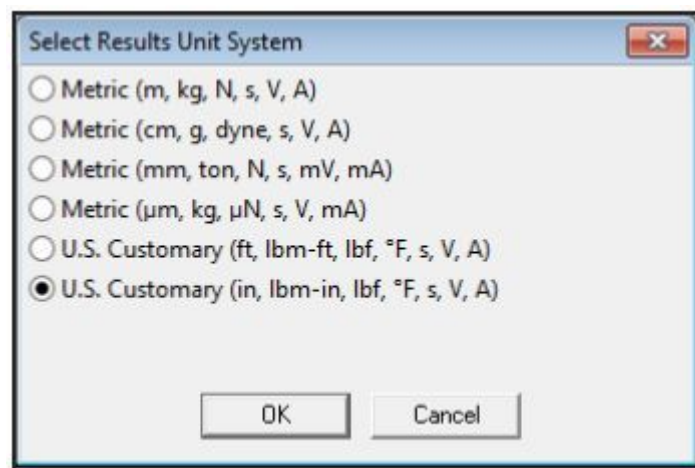

Figure 22: Indicate Units in the CDB File and RST File

With this done, a user can begin to request results below the Solution branch.

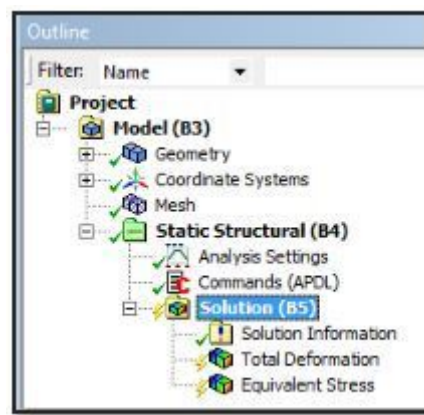Branches for results such as Deformation or Stress might be inserted:

Figure 23: Create Branches for Results of Interest.
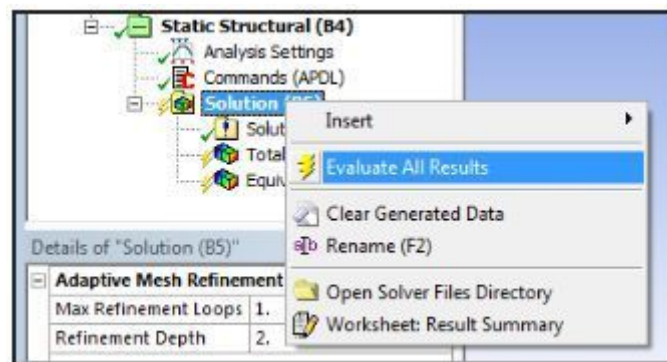
A request to Evaluate All Results is made next:

Figure 24: Evaluate Results

Users can employ the various tools of Workbench Mechanical to review results from the RST file.
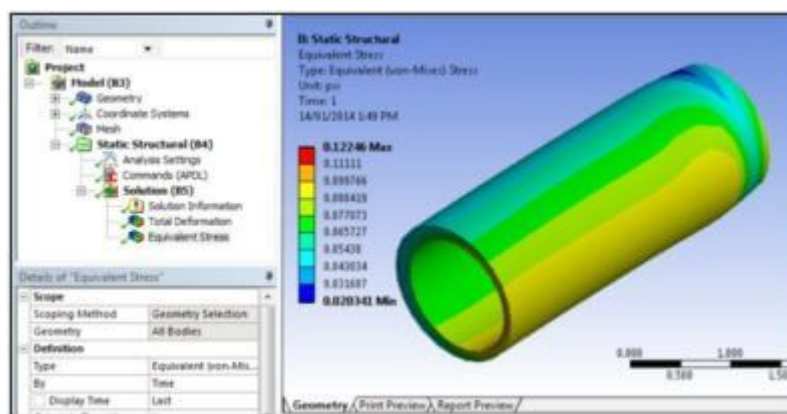
Figure 25: Results Imported from the RST File

If desired, a user can work directly from an RST file, without a CDB or database DB file. Start ANSYS Mechanical APDL, go to /POST1, and use the File command to indicate an RST results file of interest. Use the SET command to read a result of interest—this works for RST files that, by default in APDL, contain element and node data for the model. Go to /PREP7 and use CDWRITE to write a CDB mesh file. Workbench

can now read the CDB file, fit geometry to the mesh, and read the RST file (the .ERR file must exist—create a blank jobname.err file if necessary matching the CDB file). Workbench Mechanical can now postprocess a model for which only an old RST file exists. (RST files generated by Workbench Mechanical will not contain this mesh data, for size efficiency.) As seen in Figure 25 above, this approach can display degree of freedom data (displacement in a structure). It can display stress and strain if they exist in the RST file. Probes cannot be created for reaction forces and forces across contacts since no such objects exist in the Outline—APDL Commands Objects would review such things.

During model review, Construction Geometry can be used to create Paths and Surfaces to be used in postprocessing objects. Figure 26 below illustrates stress on a surface object.
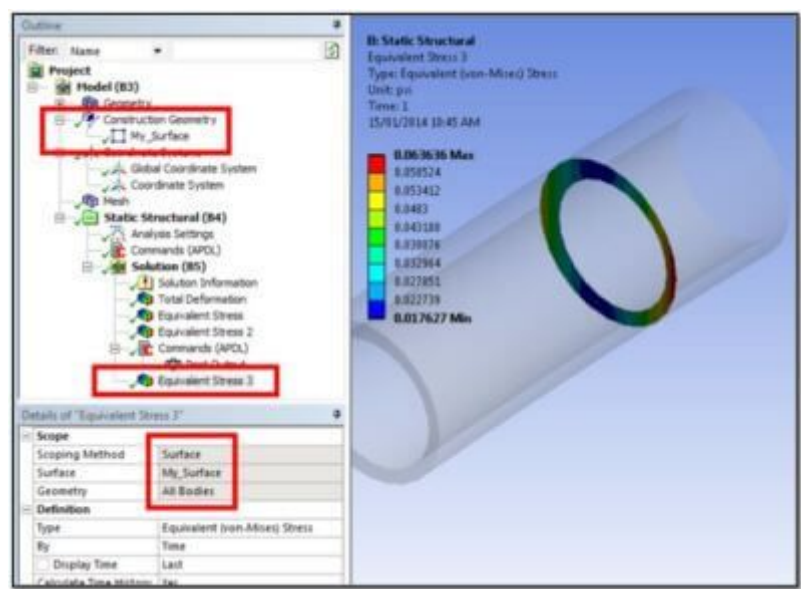

Figure 26: Result on a Surface

A Path can be created by various techniques, and results or linearized results shown on the path, for example:
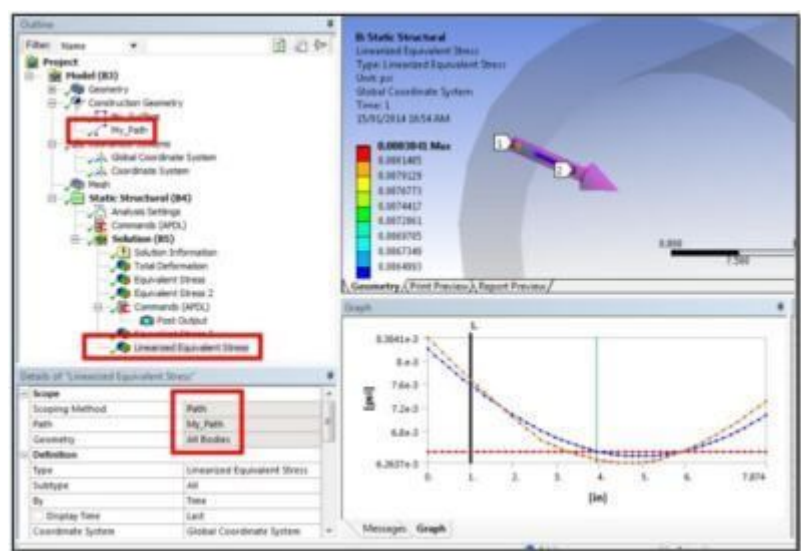

Figure 27: Linearized Result on a Path

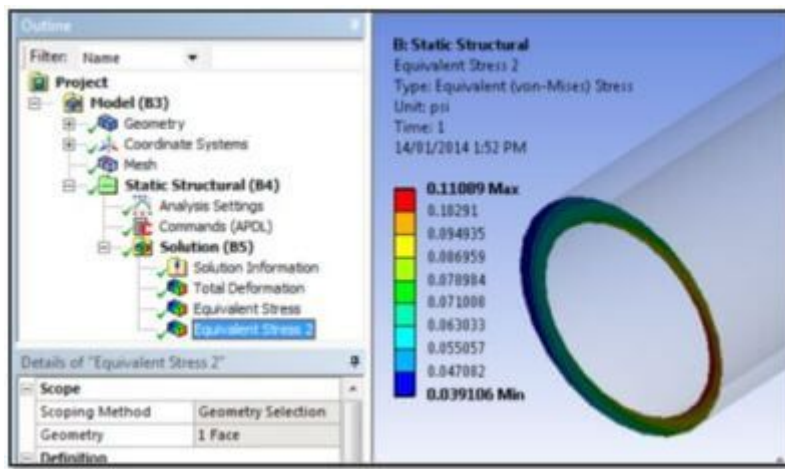Results can also be presented as scoped onto selected geometry:

Figure 28: An Example: Scoped Results Can Be Examined

An imported CDB file could contain contacts. If the RST file contains contact results, these can be inspected in the Workbench Mechanical model. The following figure shows two bodies in contact. A contact pair is created as a result of geometric proximity of parts—this does not reflect what contacts existed in the CDB file. In this example, the contact pair has been given the same "Type" as that in the CDB file.

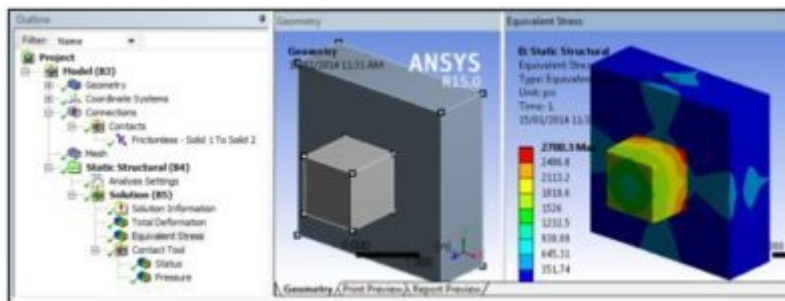They have been squeezed together in the ANSYS Mechanical APDL model:

Figure 29: Bodies in Contact as Imported from CDB and RST Files

A Contact Tool can be created for a contact pair, and if the "contact" surface of the pair in the CDB file contains a result, the contact result can be reviewed in Workbench Mechanical:
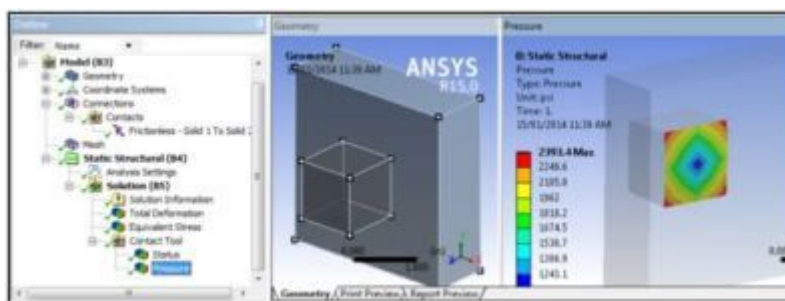

Figure 30: Contact Result as Imported from CDB and RST Files

Conclusions

Workbench Mechanical v15.0 can use External Model to read an ANSYS Mechanical APDL model's CDB file for a mesh (as written by the CDWRITE command) for typical solid and shell elements. Geometry can be fitted to the mesh, and Workbench Mechanical can receive the meshed model. An associated results file (RST file for structures) can be read into Workbench Mechanical, so that the tools of Workbench Mechanical can be used for model review.

Generating a Report within Workbench Mechanical can produce extensive model information of potential interest, which would be tedious to generate manually.

Many creative applications for this External Model capability can be developed, in addition to merely using Workbench Mechanical to review the results of ANSYS Mechanical APDL models. Note that inside ANSYS Mechanical APDL, the UPGEOM and UPCOORD commands can be used to move nodes of an unstressed model into the positions calculated by a SOLVE. A factor can be applied to the movements, using values other than a default 1.0. If a negative factor of -1.0 is used, nodes are moved into a position such that when loaded, a structure moves into the position of the original geometry (in a linear analysis). This might be used, for example, to find how to shape a structure so that it moves into a desired geometric shape after loading. Workbench Mechanical and FE Modeler could be used to find the geometry that fits that pre-deformed mesh. A more complex procedure could be used if large deflection effects were relevant for accuracy.

This External Model approach permits the tools of Workbench Mechanical to be used to examine the results of old ANSYS APDL models with the various advantages of the Workbench interface.

Looking for more ANSYS Tips & Tricks?

Check out 300+ more videos and articles on our SimuTrain® Portal.

GET SIMUTRAIN NOW! (/TRAINING/SIMUTRAIN-ANSYS-TRAINING-PORTAL)

Industry Solutions

Aerospace (/fea-cfd-aerospace)

Automotive (/fea-cfd-automotive)

Building Construction & HVAC (/fea-cfd-building-construction)

Electronics (/fea-cfd-electronics)

Healthcare (/fea-cfd-healthcare)

...Continued

Industrial Products (/fea-cfd-industrial-products)

Oil & Gas (/oil-gas)

Oil & Gas (/oil-gas)

Plastic and Rubber (/fea-cfd-plastic-rubber)

Power Generation (/fea-cfd-power-generation)

Water & Waste (/fea-cfd-water-waste-management)

ANSYS Programs

ANSYS Startup (/ansys-startup)

ANSYS Student (/ansys-student)

Advantage Magazine (https://www.ansys.com/About-ANSYS/Advantage-Magazine)

Dimensions Magazine (https://www.ansys.com/about-ansys/dimensions-magazine)

Company

Case Studies (/case-studies)

Customer Testimonials (/customer-testimonials)

Search Our Website (/search-our-website)

Privacy Policy (/privacy-policy)

Site Map (/site-map)