

# Course in ANSYS

## Import + meshing issues

# Course Outline

Introduction

Lesson 1. Modeling reviewed – Boolean's

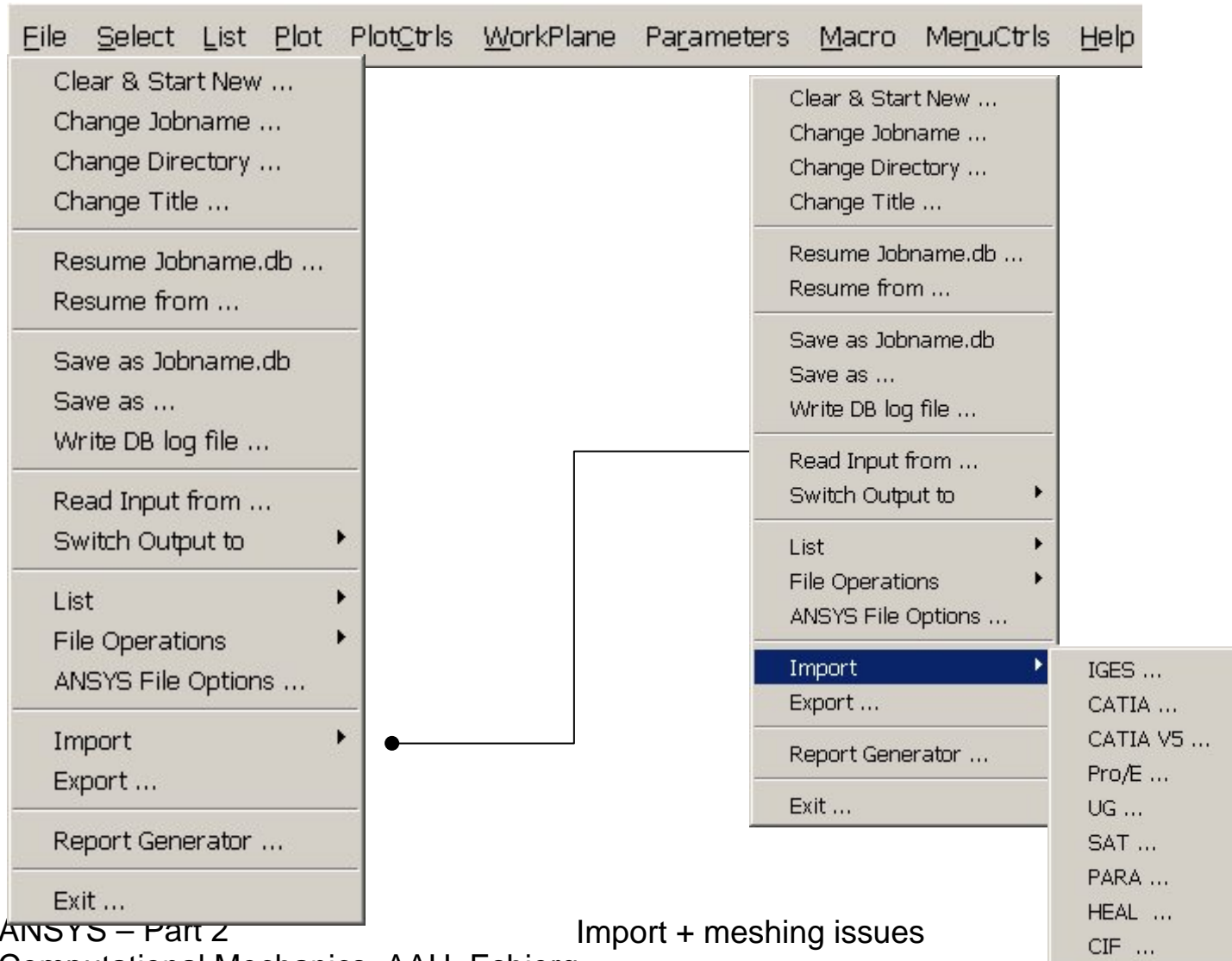
Lesson 2. Boolean's + meshing issues

Lesson 3. Operate + meshing issues

**Lesson 4. Import + meshing issues**

Lesson 5. Meshing – advanced topics

# ANSYS - Import



# ANSYS – Connection products/Import

CAD Package	File Type	Preferred ANSYS Connection Connection Product
CATIA 4.x and lower	.model or .dlv	CATIA
CATIA 5.x	.CATPart or .CATProduct	CATIA Version 5
Pro/ENGINEER	.prt	Pro/ENGINEER
Unigraphics	.prt	Unigraphics
Parasolid	.x_t or .xmt_txt	Parasolid
Solid Edge	.x_t or .xmt_txt	Parasolid
SolidWorks	.x_t	Parasolid
Unigraphics	.x_t or .xmt_txt	Parasolid
AutoCAD	.sat	SAT
Mechanical Desktop	.sat	SAT
SAT	.sat	SAT
Solid Designer	.sat	SAT

# Import - Considerations

## **While Building the Model in the CAD System:**

- Observe ANSYS solid modeling procedures with regard to planning, symmetry, and the amount of detail needed for a finite element analysis. For example, for axisymmetric models, the ANSYS program requires that the global Y axis be the axis of rotation.

# Import - Considerations

## **While Building the Model in the CAD System:**

- Avoid creating closed curves (that is, a line that starts and ends at the same point and closed surfaces (such as a surface that starts and ends at the same edge). ANSYS can't store closed curves or closed surfaces (it requires at least two keypoints). If a closed curve, closed surface, or "trimmed" closed surface - defined by IGES entities 120 *and* 144 or 128 *and* 144 - is encountered while reading an IGES file, ANSYS will attempt to split it into two or more entities.
- As much as possible, write to the IGES file by data that the ANSYS program supports.

# Import - Considerations

## **While Writing the IGES File From the CAD Program:**

- Transfer only the portion of the geometry required for the analysis. A finite element analysis may not need as much detail as a CAD model requires.
- For trimmed surface transfer, include global XYZ data along with UV data in the IGES file.

# Import - Considerations

## While Writing the IGES File From the CAD Program:

- If the model to be analyzed is very large, use the CAD program's selection capabilities to create several IGES files, each containing a portion of the model. The ANSYS program will use the next available entity number as each file is read. You can then use the PREP7 merge feature (NUMMRG command or menu path **Main Menu> Preprocessor> Numbering Ctrl> Merge Items**) to merge coincident entities.
- Write the IGES file in ASCII format, with 80 characters per record.
- For the Pro/ENGINEER program, use these additional guidelines:
  - Set the Config.pro option "iges\_out\_trim\_xyz" to "yes."
  - Set the accuracy to 1E-6 and regenerate the model.



# Import - Considerations

## **While Reading the IGES File into ANSYS:**

- Pay attention to the messages issued by the ANSYS program. Warning messages give details such as IGES entities not transferred and the corresponding ANSYS entity numbers.
- If any IGES entities were not transferred, reconstruct them using ANSYS solid modeling commands. The SMOOTH IGES filter is capable of reading in any rational B-spline curve entity (type 126), or rational B-spline surface entity (type 128) with a degree less than or equal to 20. Attempts to read in B-spline curve or surface entities of degree higher than 20 may result in error messages.

# Import - Considerations

## While Reading the IGES File into ANSYS:

- Duplicate lines and keypoints are possible when transferring a model in from an IGES file. This often happens with CAD models due to the tolerances and practices that they were created with. You sometimes need to "clean up" these solid models with ANSYS commands that merge duplicate entities together (NUMMRG command or menu path **Main Menu> Preprocessor> Numbering Ctrls> Merge Items**).
- Merging is done automatically when an IGES file is read into ANSYS [IGESIN] in AUX15. Default tolerances are used to determine if keypoints should be merged together into a single keypoint. Sometimes the default tolerances are not adequate and must be adjusted.

# Import - Considerations

## While Writing an IGES File from ANSYS:

- Set the system of units [/UNITS] before writing the IGES file. This information is captured on the IGES file and is read by many programs that read IGES files. (You cannot access the /UNITS command directly in the GUI.)
- Select all lower level solid modeling entities before writing the file (ALLSEL,BELOW,ALL or menu path **Utility Menu> Select> Everything Below**).
- If you wish to write out a portion of your model, select only those entities (areas) to be written and all corresponding lower level entities (lines and keypoints). Then unselect any higher level entities (volumes) before writing the file.

# Import - .SAT file

## Import using the ANSYS GUI

- Select **File> Import> SAT**. The **ANSYS Connection for SAT** dialog box appears, as shown below.
- Select the desired SAT file. In addition to the file name, location and type options, modify the following values as necessary.
  - **Allow Defeaturing** - Select this button to store the model in the solid database format so that it can be defeatured after import. The default is to store the model in neutral database format, which restricts defeaturing after import.
  - **Geometry Type** - Import the file as
    - **Solids Only** - imported as ANSYS volumes (default).
    - **SurfacesOnly** - Imported as ANSYS areas.
    - **Wireframe Only** - Imported as ANSYS lines.
    - **All** - Import all entities; use this when the file contains multiple entity types.
- Click **OK**.

## Note

- If your imported model contains multiple volumes, Boolean operators are available to further process your geometry.

# Import - .SAT file

## Problems Specific to ANSYS Connection for SAT

- Listed below are several problems you may encounter when running ANSYS Connection for SAT. For further troubleshooting information, see [Appendix B](#).
- If a part fails to convert into ANSYS, examine filename.sat\_log and filename.ans\_log. Both files are located in the working directory. The filename.sat\_log file contains warnings and errors generated during the conversion that relate to problems in the file itself. The filename.ans\_log file contains warnings and errors that relate to building the model in ANSYS.
- If a model is sensitive to tolerance, shapes may not be created properly, which can lead to poorly transferred areas. If you find this problem with a model, reimport it using the following steps:
- Import the file using the default format (neutral).
- Enter the following command in the ANSYS Input window:
  - XANF,filename.aaa,-tolerance\_1.0e-6 where filename.aaa is the name of the file you are working with

# Import - .SAT file

**The following problem may occur on UNIX systems:** The software may not come up if the required shared library (ansys\_inc/ansys81/lib/<platform>/libac4.so) is not present or if the path names listed in this file are incorrect. Check for proper installation.

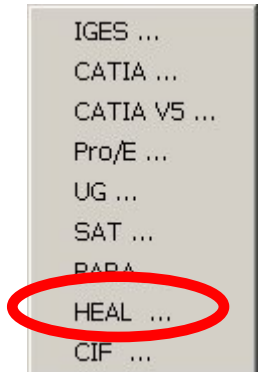
**The following problems may occur on Windows systems:** A sys error may be displayed if the executable is in the wrong directory or if the required shared libraries are missing. Check for proper installation.

- \*\*\*Warning: ANSYS environment variables: ANSYS81\_DIR and/or ANSYS81\_SYSDIR are not set.
- \*\*\*ERROR: ANSYS Connection for SAT directory structure is not set. Unable to find the Connection executable.
- Either the ANSYS81\_DIR or the ANSYS81\_SYSDIR variable needs to be set.

# Import - Healing

## The Healing Module

- This section examines how to “heal” a model that you have imported into ANSYS via either the ANSYS GUI or the [~HEAL](#) command.
- Healing is the process of preparing CAD data to be read into ANSYS. You can use healing with existing ANSYS Connection products, as well as with other imports supported by the ANSYS Automatic Geometry Healing Module (AGHM). If you have a license for an ANSYS connection and an AGHM import license for the same type of data, the AGHM option will be used to read the data. In either case, an .anf file is created.
- Healing via the AGHM is a module available through CADfix.



## Note

- The images shown in this section may vary slightly compared to what displays on your system. However the functionality is the same.