# Connecting Ansys Workbench with CivilFEM for ANSYS

Workflow

- There are three possibilities or approaches to connect Ansys Workbench with CivilFEM for Ansys:
  1. Create only geometry in Design Modeler (or Space Claim) and then send the geometric model to CivilFEM (Ansys Mechanical APDL). (Approach I).
  2. Create FE Model in WorkBench Mechanical Environment and then send the FE Model (with loads and Boundary conditions if you wish input them in WB instead of CivilFEM) to CivilFEM (Ansys Mechanical APDL) (Approach II).
  3. Export geometry or fem model directly from Workbench Environment to ANSYS Mechanical APDL (Approach III).
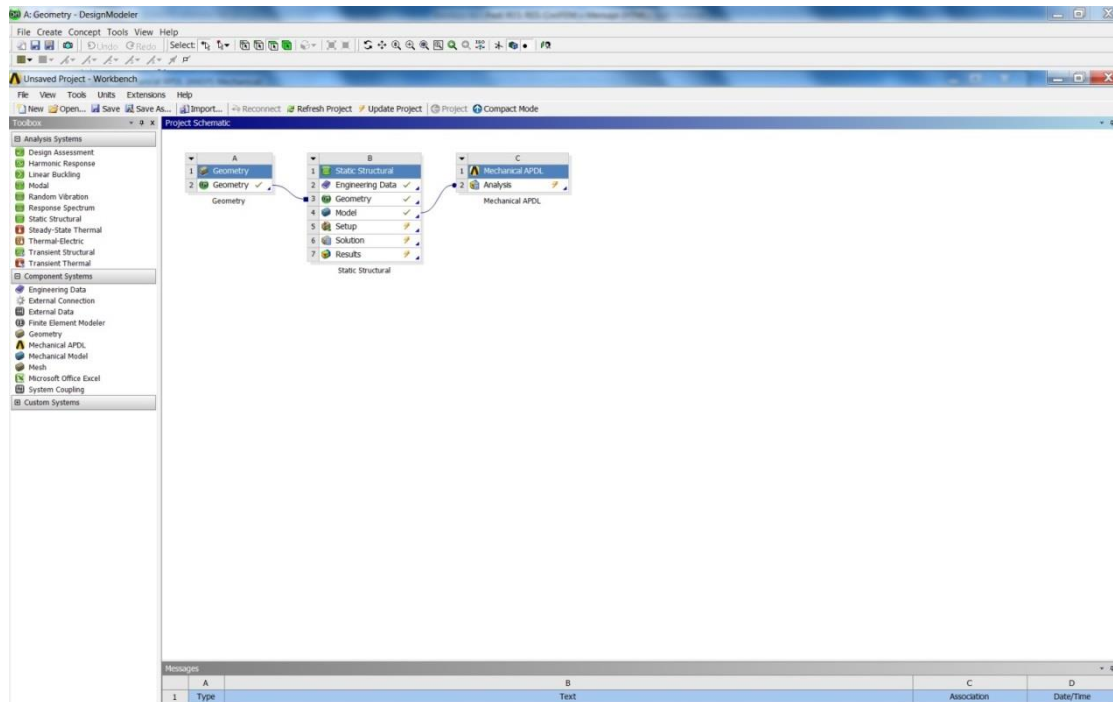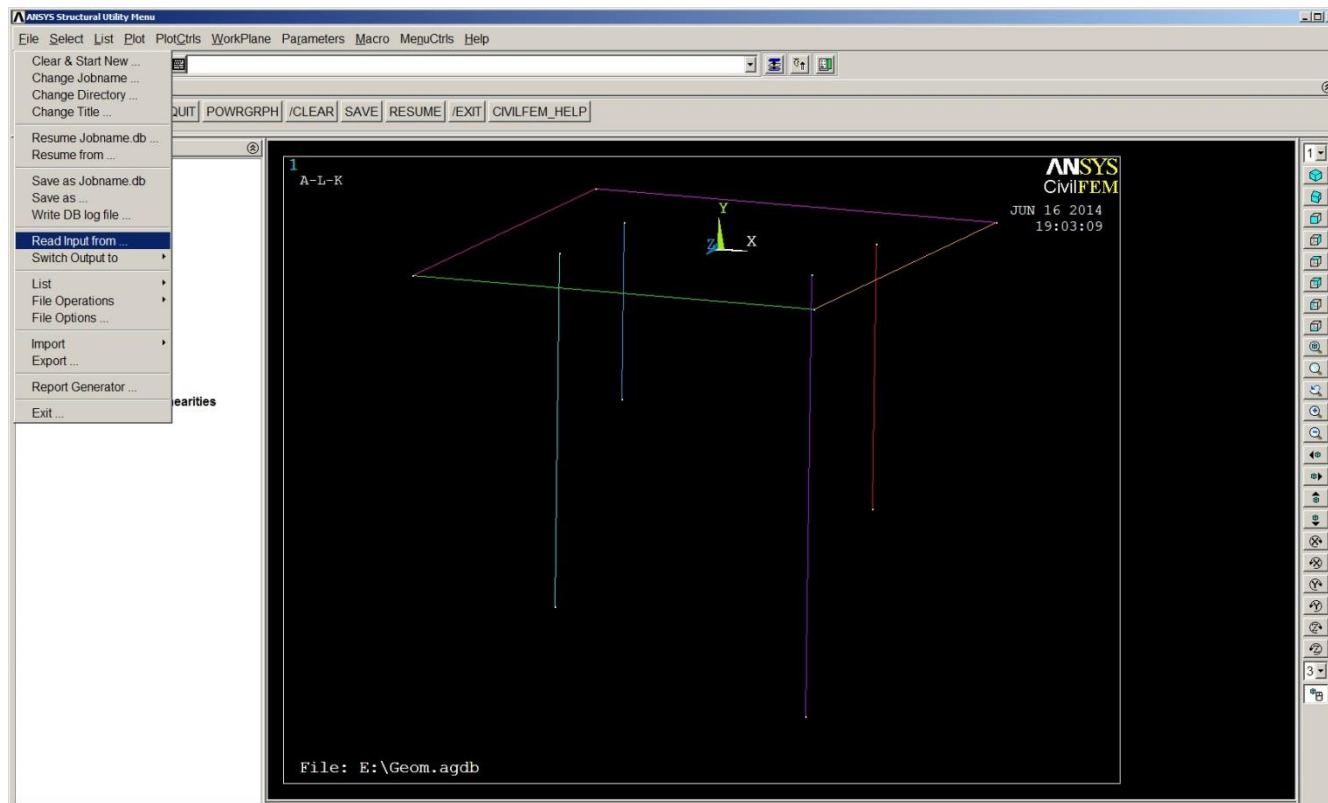
(I) Import Geometry from ANSYS Workbench to CivilFEM

- 1. Create only Geometry in Design Modeler or Space Claim (or import from the CAD)
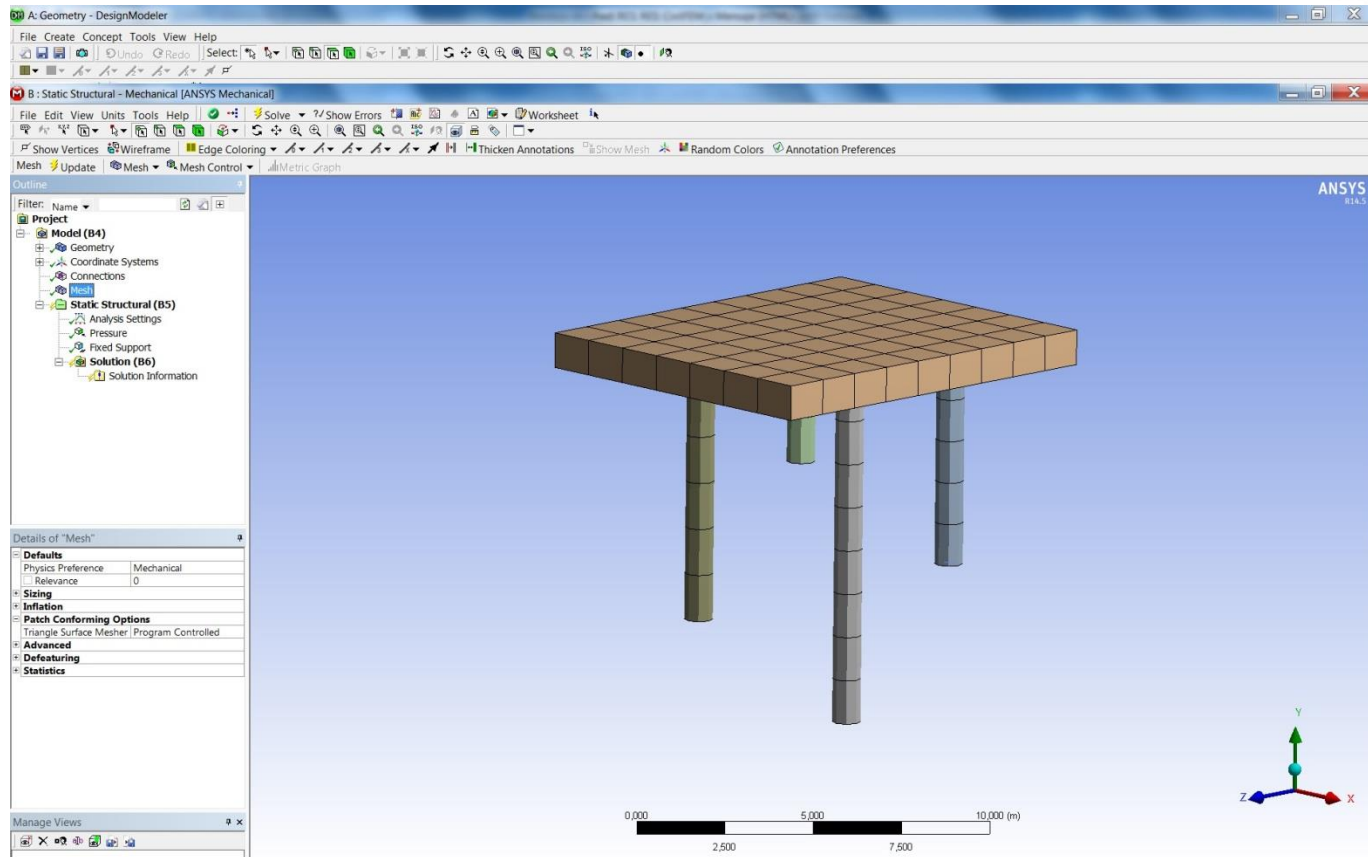- 2. Export Geometry as .anf file
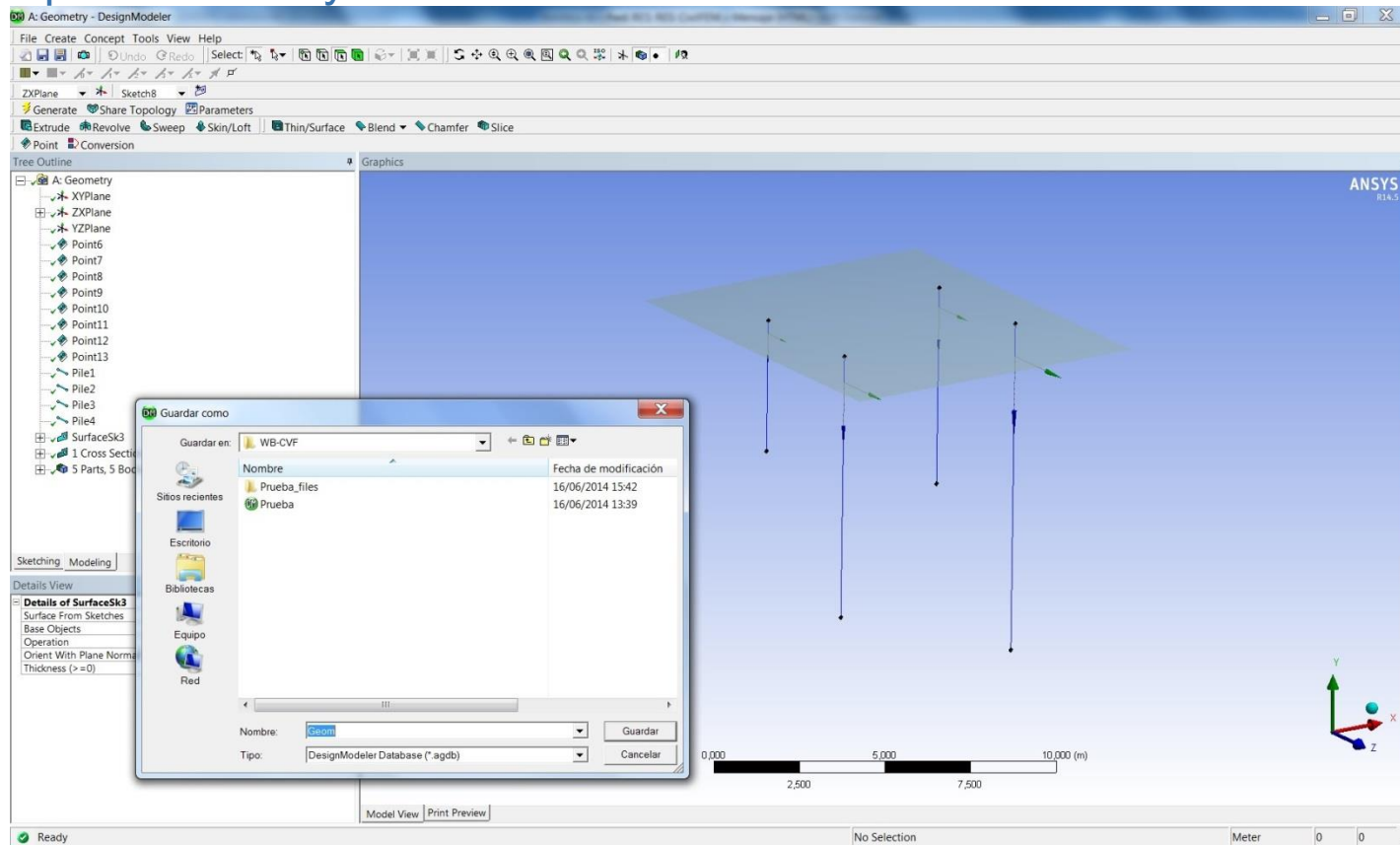
(I) Import Geometry from ANSYS Workbench to CivilFEM

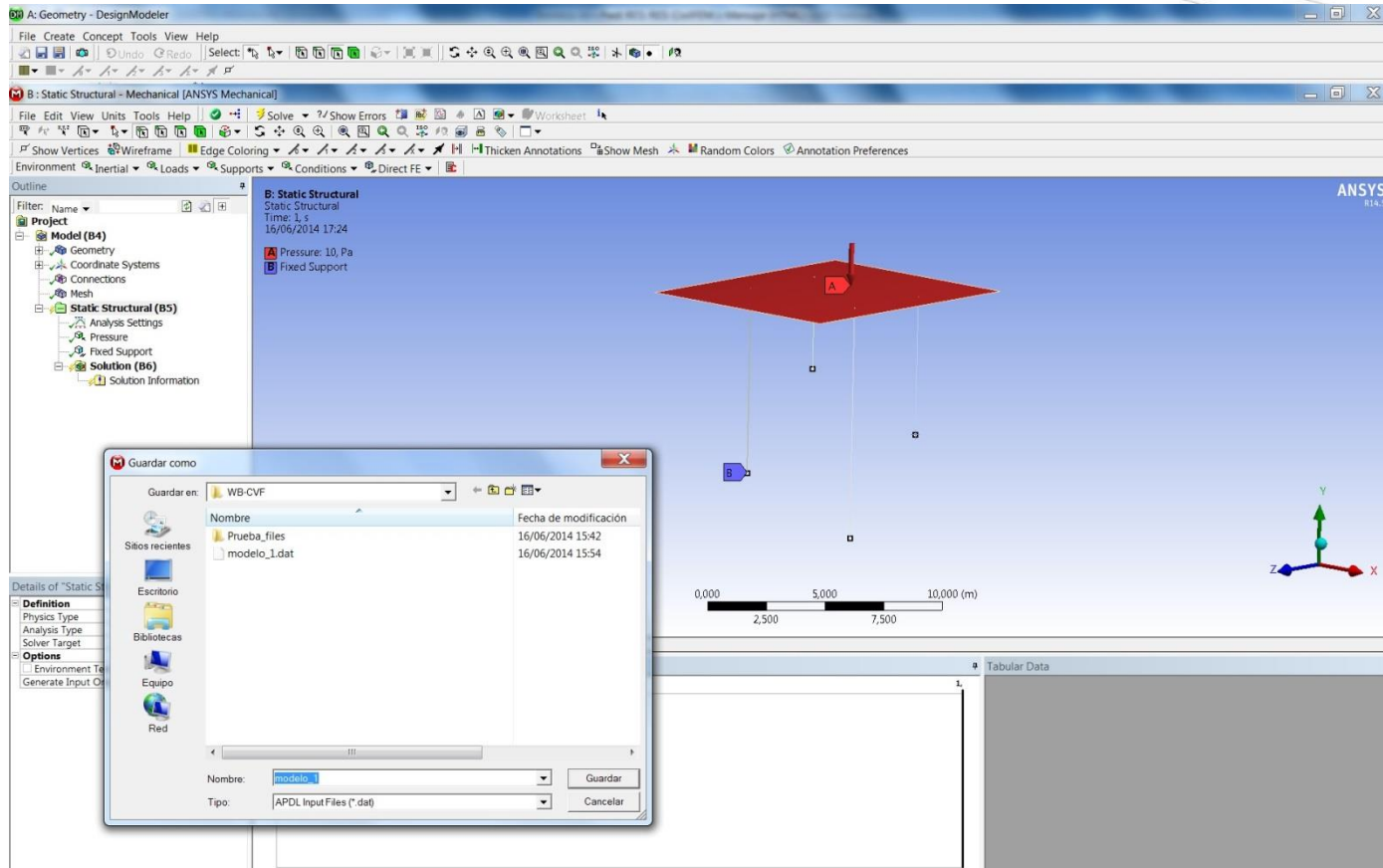- 3. Read .anf file with Ansys Mechanical APDL (read imput from…)

(II) Import all FE Model from Mechanical Environment to CivilFEM

The approach II: Once we had created all the FE model with the mesh, loads and Boundary Conditions in Mechanical Environment :

(II) Import all FE Model from Mechanical Environment to CivilFEM
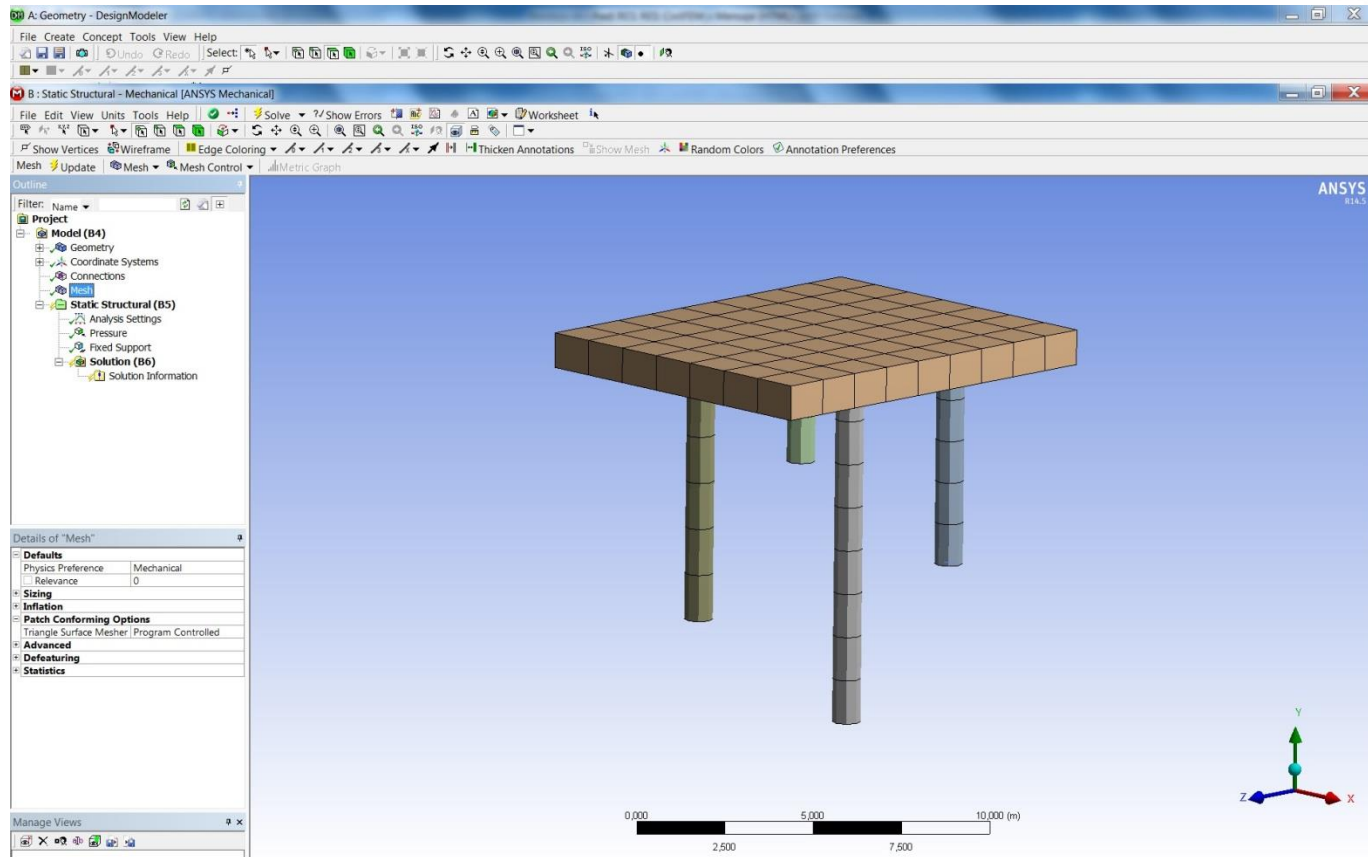
1. Export all FE model as .inp or .dat file

(II) Import all FE Model from Mechanical Environment to CivilFEM

- Read .inp or .dat with Ansys Mechanical APDL (read input from…)

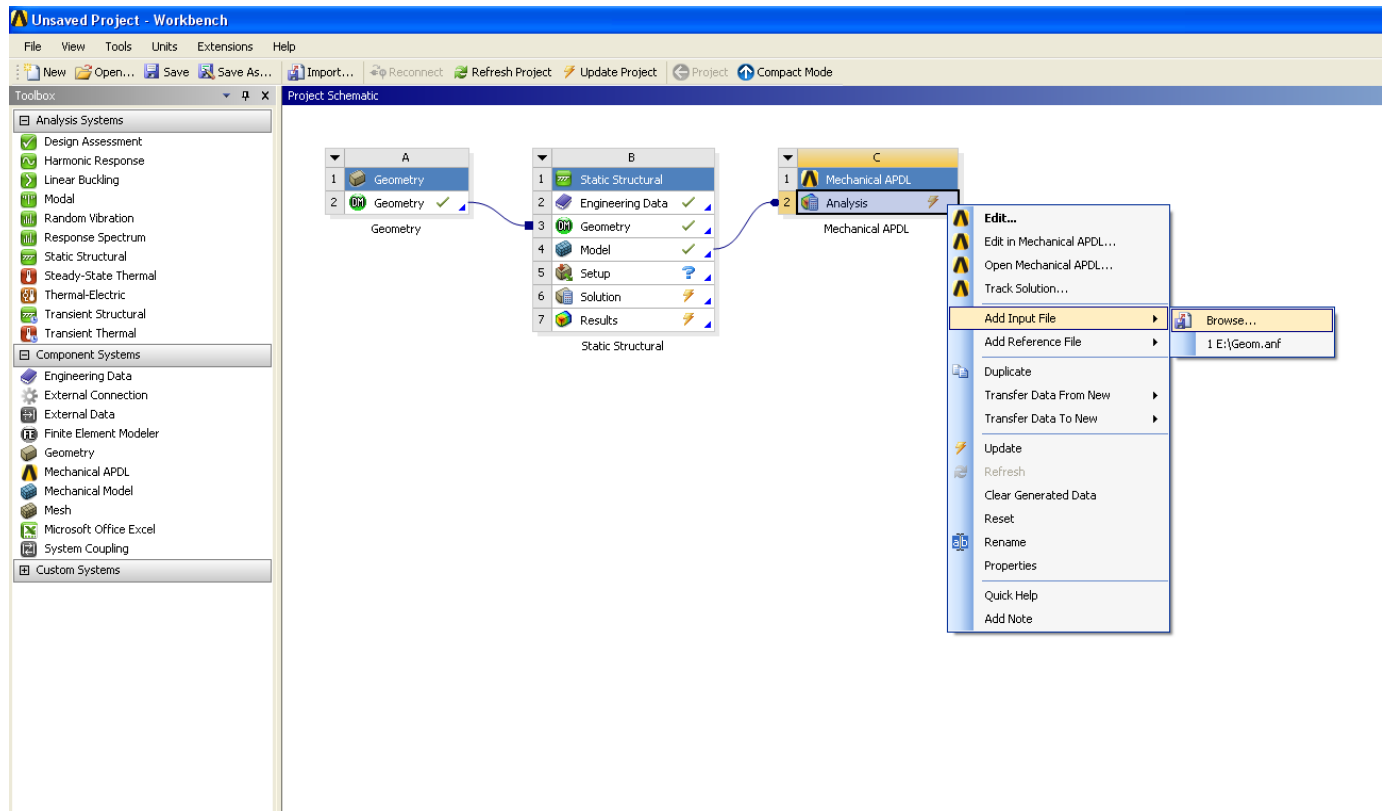# (III)Export geometry or FEM model directly from Workbench Environment to ANSYS Mechanical APDL

- • 1. Create Geometry or FEM model in Workbench

(III)Export geometry or FEM model directly from Workbench Environment to ANSYS Mechanical APDL

- 2. Select Mechanical APDL for the analysis type
- 3. Click right button on Analysis
- 4. Click on "Add Input File"

(III)Export geometry or FEM model directly from Workbench Environment to ANSYS Mechanical APDL

5. Click on "Edit Mechanical APDL". The model is automaticaly opened in ANSYS Mechanical APDL with CivilFEM
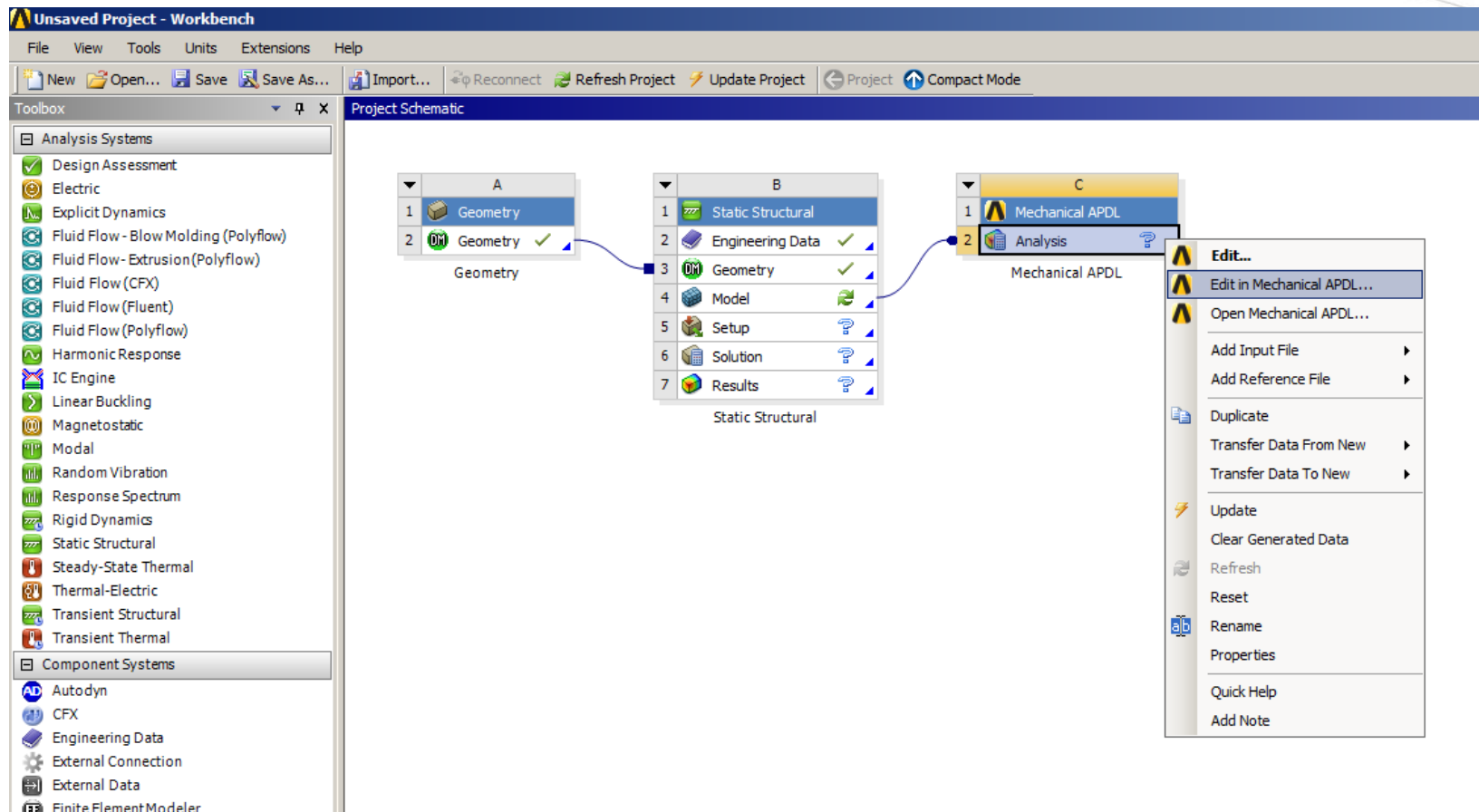
Preprocessing with CivilFEM

Once we had the mesh model with loads and BC in CivilFEM

- CivilFEM Set Up, set the corresponding Design Codes or Standards to be used
- CivilFEM preprocessor: define Code/Standard materials properties, create beam and shell properties to automatically overwrite Ansys materials and Real constants.
- CivilFEM preprocessor: create CivilFEM cross sections and shell vertices.
- ~cfsave (if you wish to save CivilFEM model)
- Solve if the model was not solved before or use ~RCVWRT command to generate the CivilFEM results file (.rcv) if model was previously solved/generated in Ansys WB.

Postprocess with CivilFEM

Before solving the model

- Write results into database after the solution. These command lines must be input in the beginning of the Ansys input file before solving:

  /CONFIG,NOELDB,0

  OUTRES,ALL,ALL

Once we have solved the model, postprocess with all the CivilFEM tools and features.

Please note that CivilFEM has a friendly preprocessor and several very useful wizards to generate directly typical civil engineering models too.