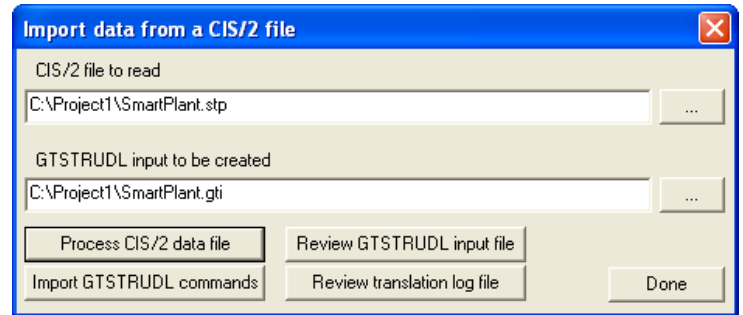


Since June 2002, the **Georgia Tech - CASE Center** has been providing functionality to **Import & Export CIS/2 files with GT STRUDL**. These functions translate **analysis** models, as opposed to design or manufacturing models. This functionality allows the user to import files, created by other software programs that support the **CIS/2** analysis model, into **GT STRUDL**, or export models created in **GT STRUDL** to other software packages that understand **CIS/2** analysis models. You can now take advantage of this flexibility, incorporate the strengths of multiple software packages, and increase your productivity.

IMPORT OF CIS/2 FILES

GT STRUDL can read the following information from a CIS/2 analysis model:

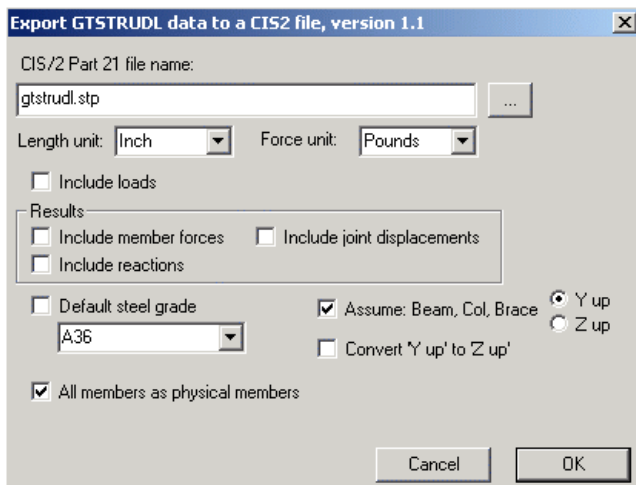
- < Joint/Node Location
- < Joint/Node Support Condition + Release Status
- < Member (Beam Element) Incidences
- < Member Profile + Cardinal Point
- < Member Beta Angle
- < Member End Releases (Pinned Connections)
- < Member Eccentricities
- < Load Cases
 - Applied Joint Loads
 - Applied Member Loads (Concentrated/Distributed, Local/Global)
- < Load Combinations
- < "Assembly Maps" - Which Analytical Members Compose a Single Physical Member



EXPORT OF CIS/2 FILES

GT STRUDL can create a CIS/2 analysis model containing all the data listed for import, plus analysis results:

- < Support Reactions
- < Joint/Node Displacements
- < Member End Forces (also called "loads" in some detailing packages)



When exporting a CIS/2 analysis model, the user can select:

- < Units (length and force)
- < Whether to Include Applied Loads or Results
- < A Default Steel Grade
- < Assumed Member Type (Beam, Column, Brace)
- < Convert from a Y Vertical Axis to a Z Vertical Axis
- < Write Analysis Elements as Physical Elements (if not already specified)