

## ANSYS Class3 Error Report

**ERROR NO:**

2012-16

**Keywords:**

CREO PARAMETRIC      Pro/ENGINEER      NAMED SELECTION      DECOMPOSE  
DISJOINT FACES

**Description of Error:**

Solid or Surface bodies imported by the Creo Parametric Associative Geometry Interface using the technique described in Capturing a Solid Body in a Named Selection, will be missing N-1 faces from the named selection, where N is the number of external contours belonging to the face. Only one of the disjoint faces will be included in the named selection. This error only occurs when the preference "Decompose Disjoint Faces" is enabled.

**Typical GUI Path(s):**

The Decompose Disjoint Faces preference is accessible from the Workbench Project page via any system containing Geometry. Right click on Geometry and select Properties. The preference appears under Advanced Geometry Options. Additionally, the preference is adjustable from DesignModeler, see the details view Advanced Geometry Options. Named Selection import control and filter keys are accessible under Basic Geometry Options of both DesignModeler and Workbench System Geometry Properties. Creo Parametric and Pro/ENGINEER Wildfire layers are accessible within the CAD system by setting the Tree view to the Layer Tree.

**Other Comments:**

For details on Capturing a Solid Body in a Named Selection use this help path: See Help: // CAD Integration // File Format Support // Creo Parametric (formerly Pro/ENGINEER) (\*.prt, \*.asm) // Creo Parametric (formerly Pro/ENGINEER) Associative Geometry Interface (\*.prt, \*.asm)

**First Incorrect Version:**

Release 14.0

**Corrected In:**

Release 14.5

**Suggested User Action For Running on Uncorrected Version:**

Disable the preference Decompose Disjoint Faces.

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**Reviewed By QA:**

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**Approval:**

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