



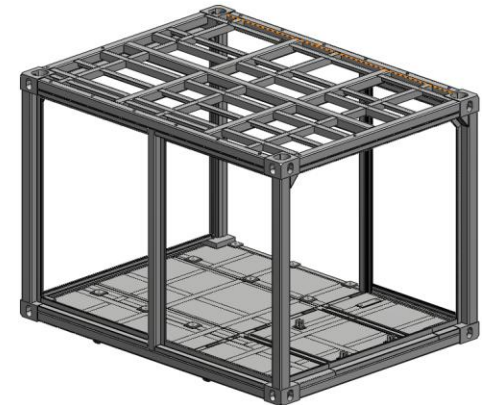
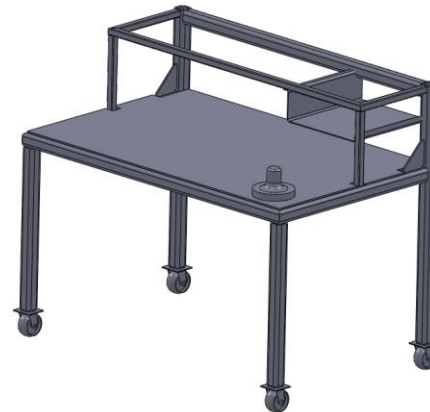
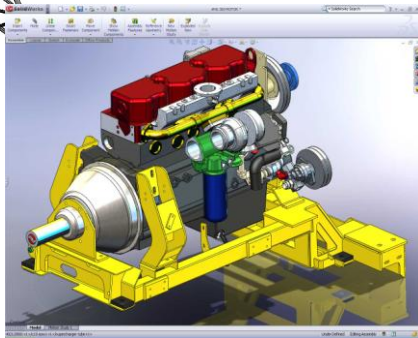
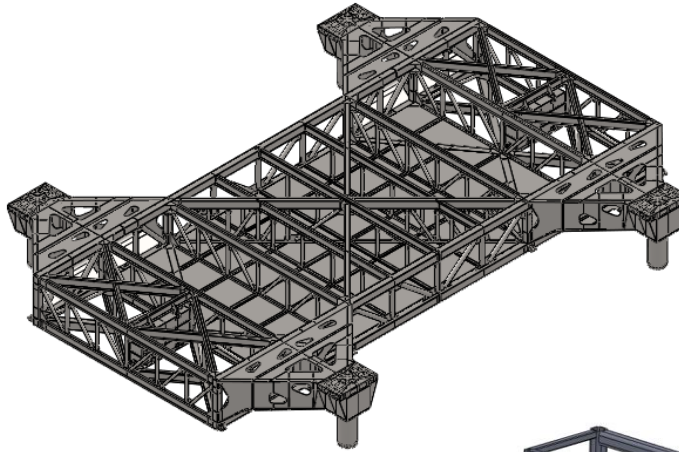
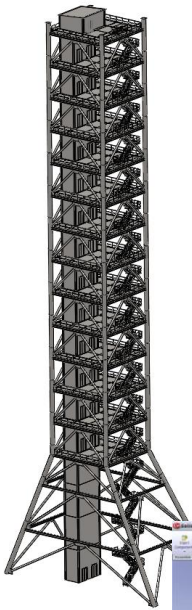
**SolidWorks**  
**WORLD 2010**

## **Intermediate Weldments in SolidWorks**

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Dassault Systems SolidWorks Corp.

## Introduction

- This presentation is designed to cover the weldments functionality beyond the basics that can be used to create more complex designs.

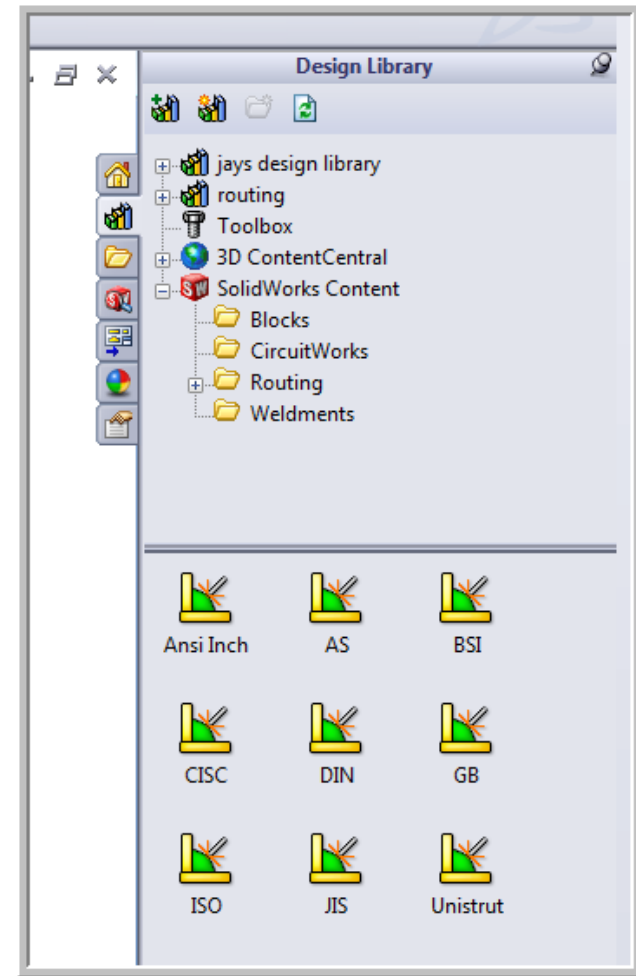


## Introduction

- Topics we will cover are:
  - Expanding your structural members library
  - Using groups in structural members
  - Inserting existing parts into weldments
  - Using patterns and mirrors with weldment bodies
  - Using Surfaces as tools in weldments
  - Using Sheetmetal in weldments
  - Using weldments and detailed cutlists in assemblies
  - Creating drawing views of sections of weldments
  - New functionality in SolidWorks 2011 –
    - Weld beads
    - 3D grids

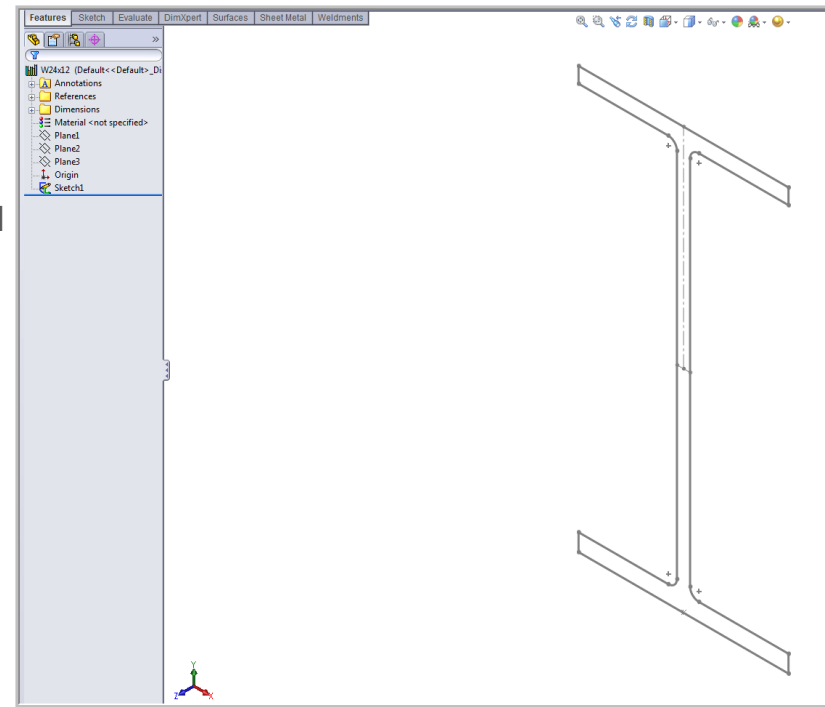
## First things; Get the structural shapes you need.

- As it comes, SolidWorks has a limited set of structural member shapes.
- You can download from the internet by going to the SolidWorks Content in the Design Library:
- CTRL + click on a type and download the zip file to the directory specified in Tools-Options-System Options-file locations-Weldment profiles
- Unzip the file and the new standard will be added.



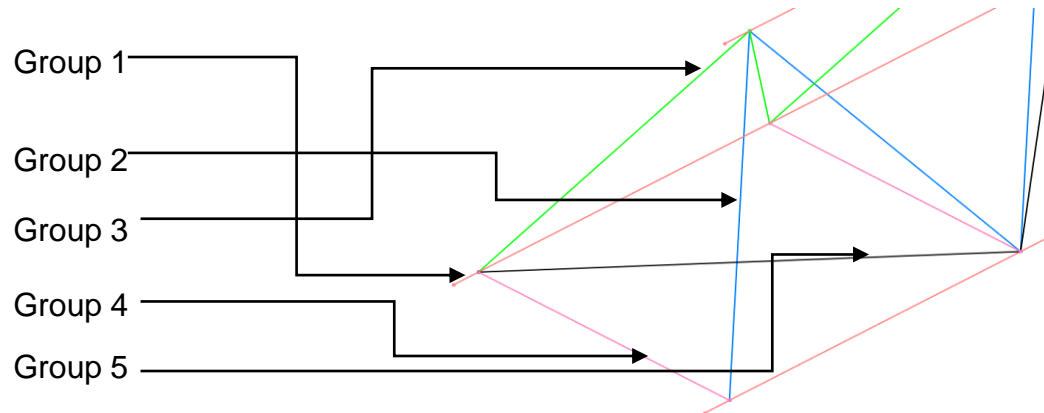
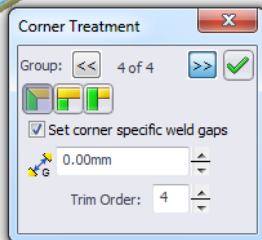
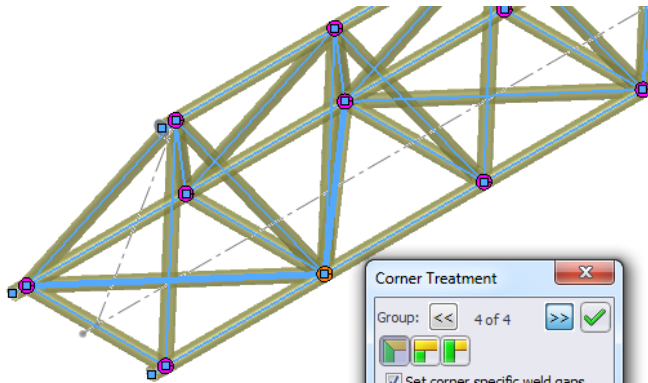
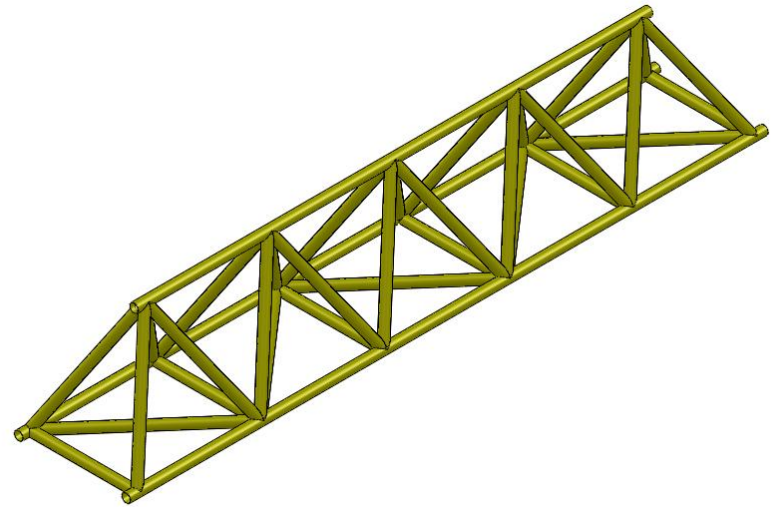
## First things; Get the structural shapes you need.

- You can also create your own structural shapes as library feature parts.
- Create 2 new folders in the Weldments profile directory one for the type and one under it for the sub type.
- Make a new part and sketch the profile you want on the front plane.
- Make sure to add points in your sketch where you want connection points
- Select the sketch in the feature tree and save the part.
- Change the file type to Library feature part (.sldlfp)
- Browse to the weldments profile section and save your part in the sub-type folder.
- Now you can use that profile in your design.



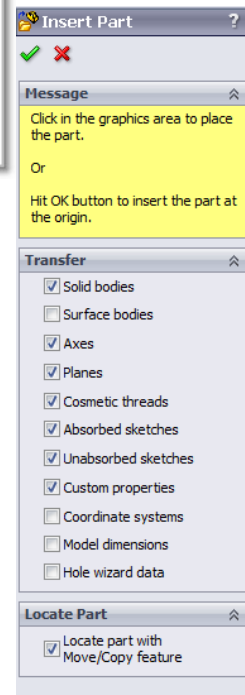
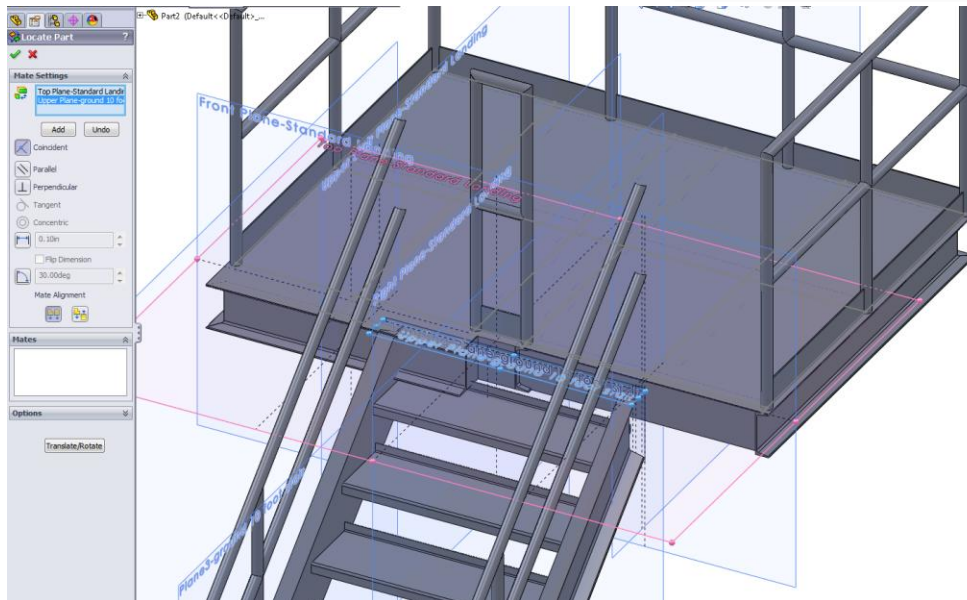
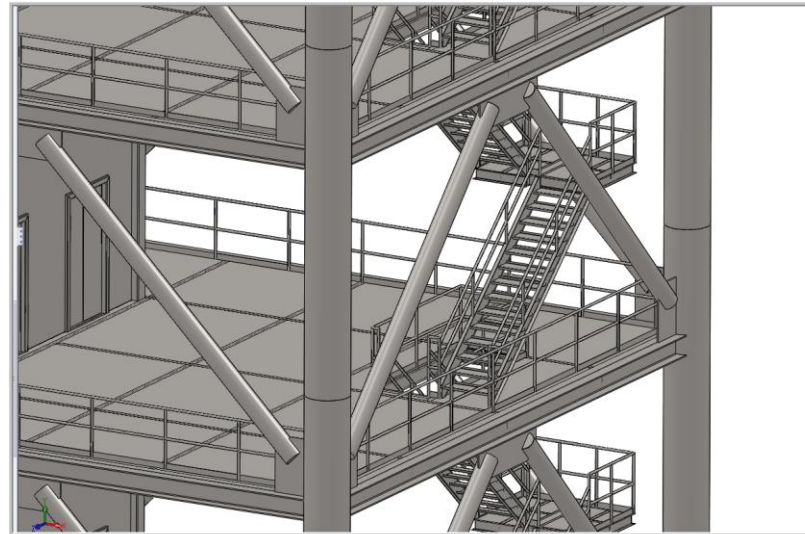
## Using groups in structural members

- Groups allow you to create complex shapes made from several members that all are the same shape and size
- You can use the trim order function of the corner treatment dialog box to control how the members meet at the corners



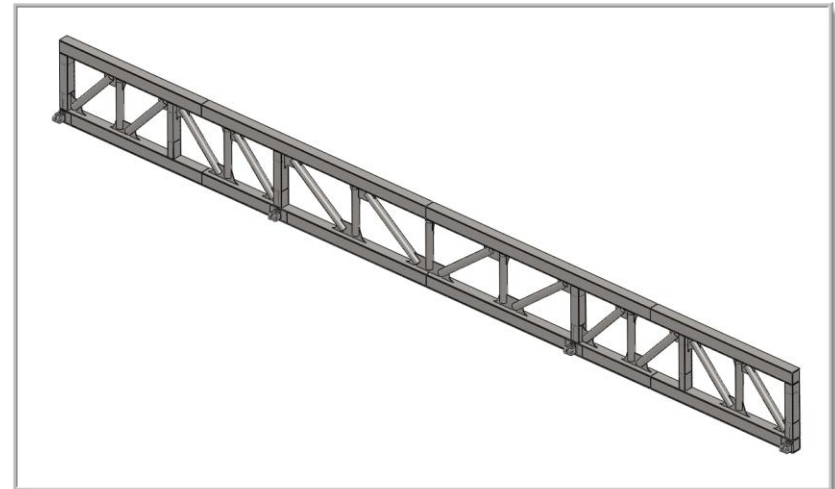
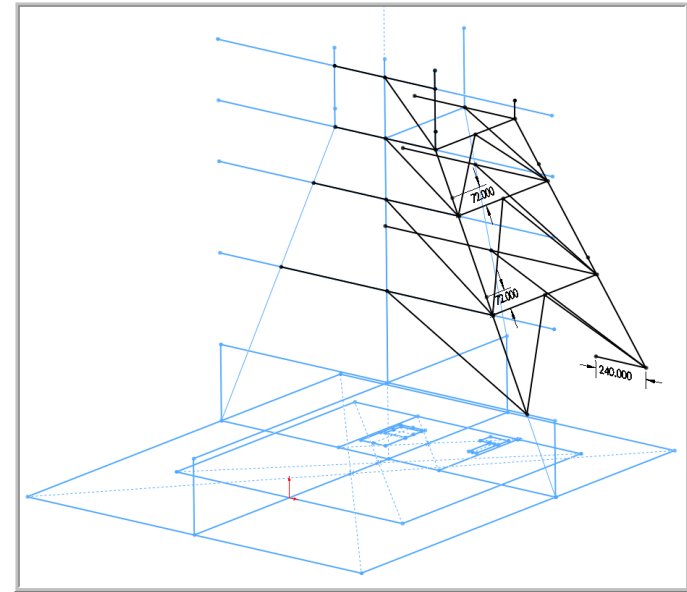
## Inserting existing parts into weldment parts

- You can take existing parts, including existing weldments, and insert them into your weldment part and “mate” them in place.
- This way you can re-use items instead of having to recreate them over and over.



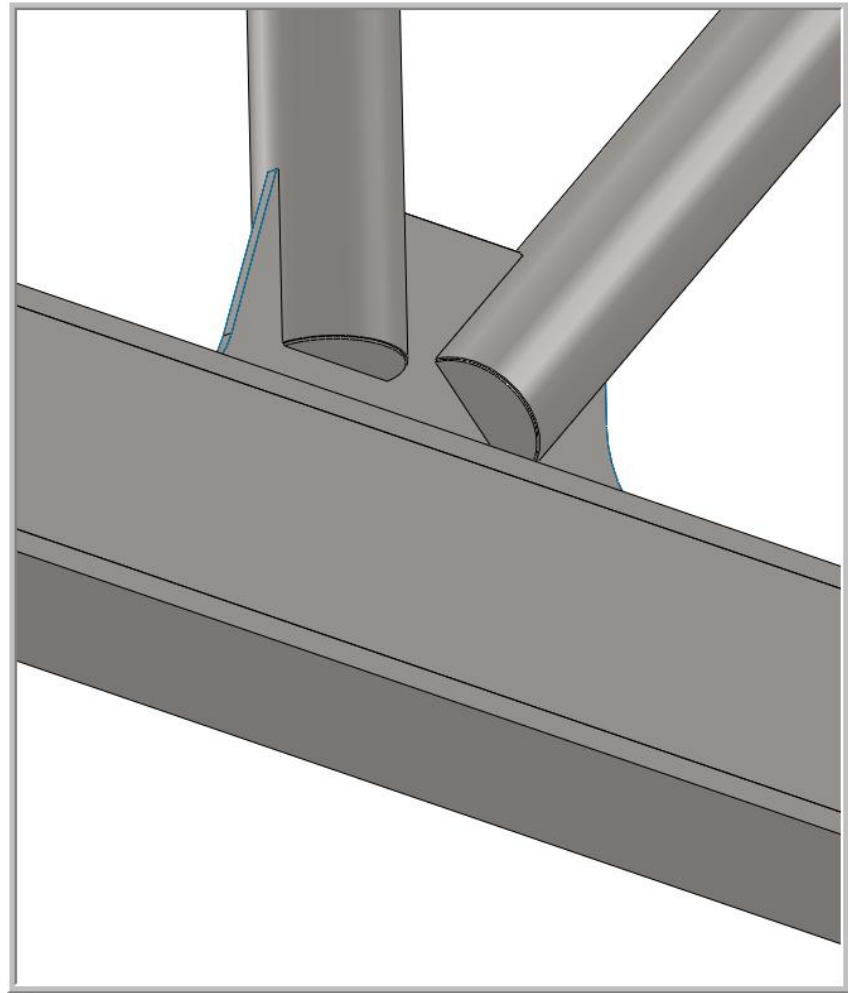
## Using mirrors and patterns of bodies to simplify your work

- Only sketch what you need. Most structural designs are symmetrical across 1 or 2 axis. You only need to sketch  $\frac{1}{2}$  or  $\frac{1}{4}$  of the complete model
- Even when there are differences, a lot of times that can still be modeled by building it symmetrically and then modifying the other side.
- Even when the that would result in a part being in 2 pieces when it should be one, you can always come back and combine bodies together.



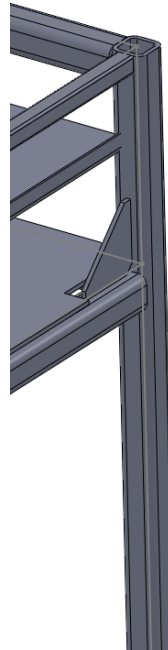
## Modeling Techniques; Don't be afraid to use other tools.

- Other tools in SolidWorks can be used with weldments to help with complex tasks.
- Surfaces can be used for cutting and trimming.
- Sheet metal can be used to make bent parts in 2010



## Using weldments and detailed cutlists in assemblies

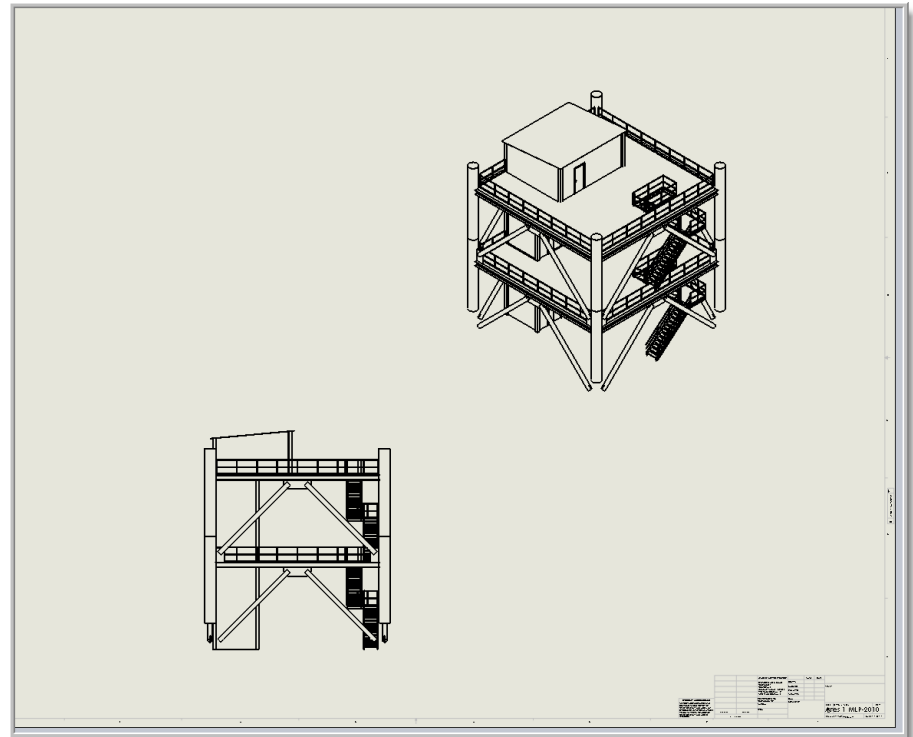
- Sometimes you want to create a weldment where some of the pieces are actually sub-assemblies.
- You can create an assembly containing the weldment and the sub assembly components and still create a drawing with one integrated cutlist



	A	B	C	D
1	ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
2	1	Cutlist_table		1
3	2		TS2x2x0.25	1
4	3		TS1x1x0.1	1
5	4			4
6	5			2
7	6			1
8	7			1
9	8			1
10	9		TS2x2x0.25	2
11	10		TS2x2x0.25	2
12	11		TS1x1x0.1	2
13	12		TS1x1x0.1	2
14	13		TS2x2x0.25	2
15	14		TS2x2x0.25	2
16	15	caster test	Caster	4
17	16	Part6^caster test	Base Plate	1
18	17	Part7^caster test	yoke	1
19	18	Part8^caster test	wheel	1
20	19	Part1^Cutlist table test		1
21	20		TS1x1x0.1	2
22	21		TS1x1x0.1	1
23	22		TS1x1x0.1	1
24	23		TS1x1x0.1	4

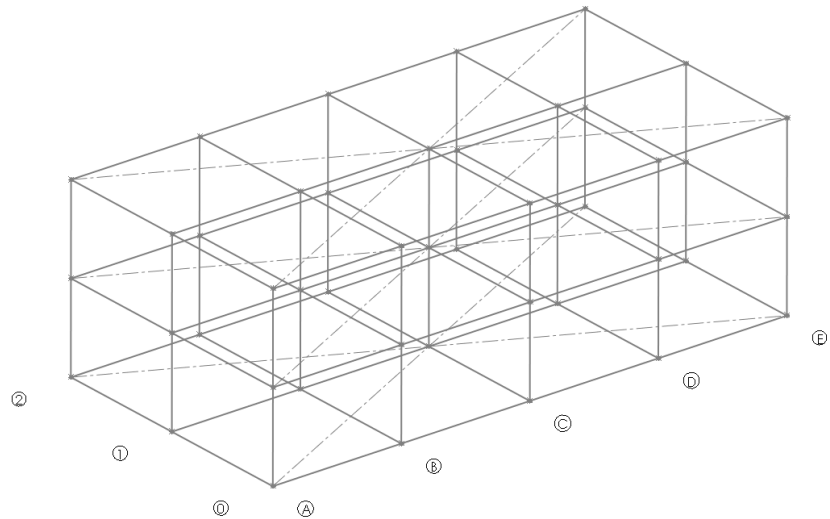
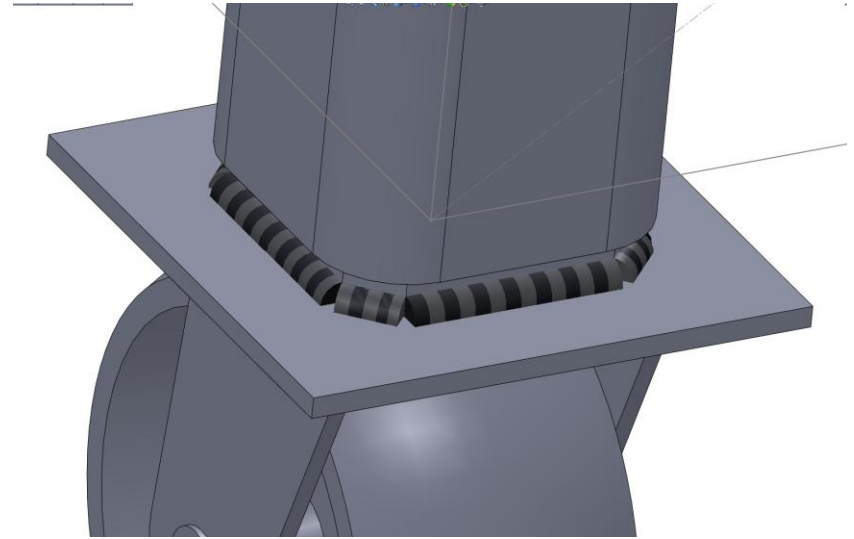
## Drawing Technique: Make drawing of just selected bodies

- This is new for SolidWorks 2010.
- You can now make a drawing of just selected bodies of a part.
- This way you can have multiple drawings of different sections of a weldment all referencing a single file.



## New for 2011 – graphic weld beads and 3D grids

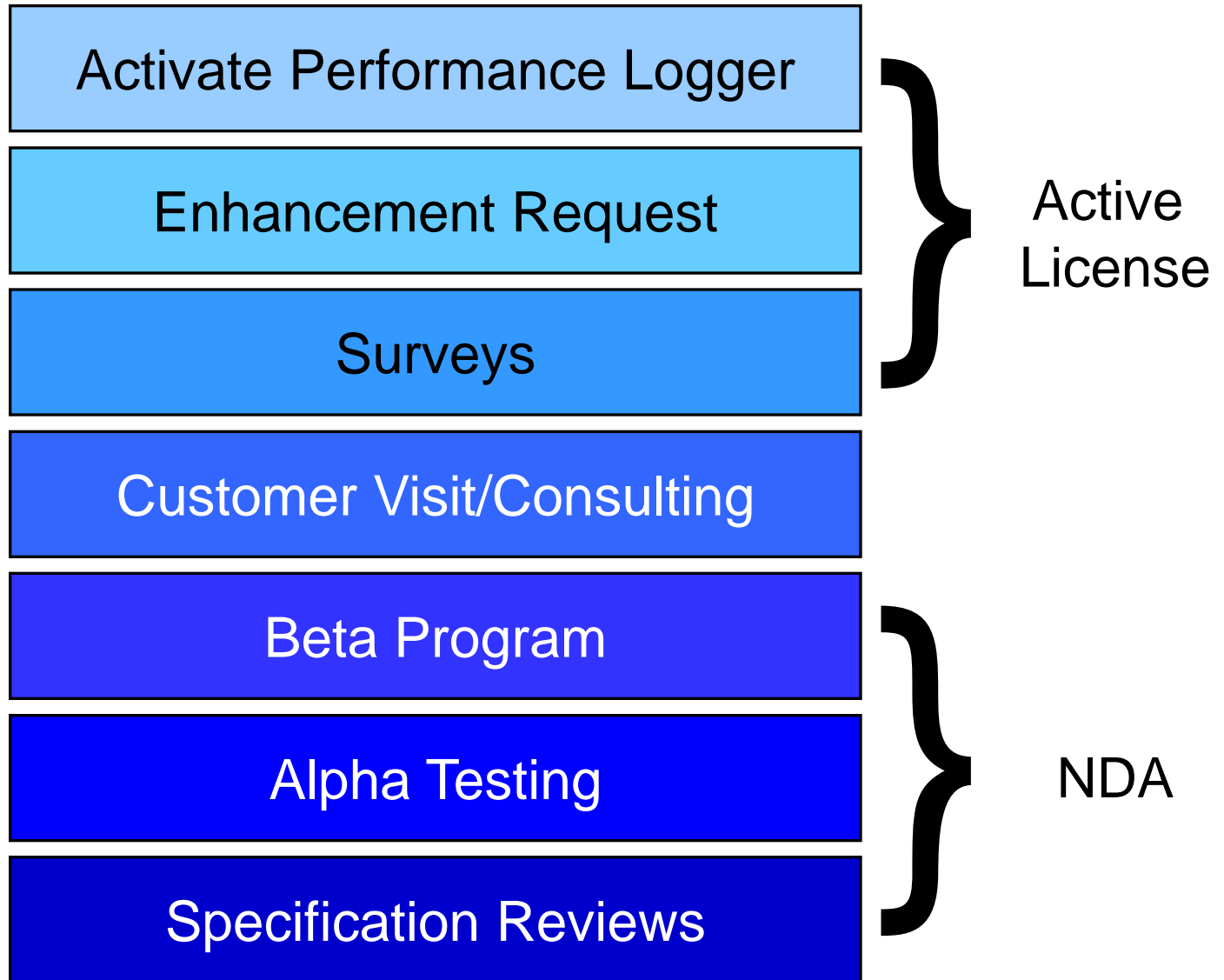
- Now you can add welds other than just fillet beads as graphical weld beads
- They display in your model and on your drawings but do not actually create 3D geometry so they don't have a negative affect on your models performance.
- You can also use the 3D grid tool to create the skeleton of your structure



Any Questions?



## Helping SolidWorks Help You



## Helping SolidWorks Help you

- Submit an Enhancement Request
  - Your vote for what is important to you.
- Turn on the performance logger
  - **Tools, Options, System Options, General**, check Enable performance email.
- Participate in the Beta program
- Respond to surveys
  - Beta program surveys
  - Surveys sent to Enhancement Request participants

**SolidWorks** does not re-distribute your personal information

## Helping SolidWorks Help you

- Host a Product Definition customer visit
  - Each product definition engineer visits 30 customers a year.
  - Give me your card with “Customer Visit” written on the back. We are always looking for Weldments users.
- Alpha Testing – Requires NDA
- Specification Reviews – Requires NDA
- Product Definition CAD consulting
  - Free to the customer
  - 3-5 days
  - Get help working with SW
  - The experience heavily influences the development of SW
- [www.solidworks.com/usability](http://www.solidworks.com/usability)

The End

# Thank You