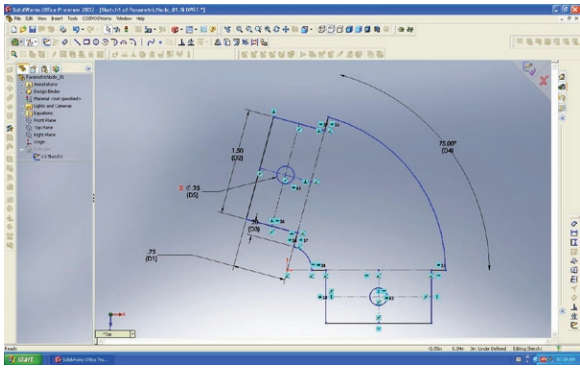
SolidWorks Workshop

by: Proxy (10.08.07)

Workshop: Solidworks Analysis of Parts (Micro Anaylsis)

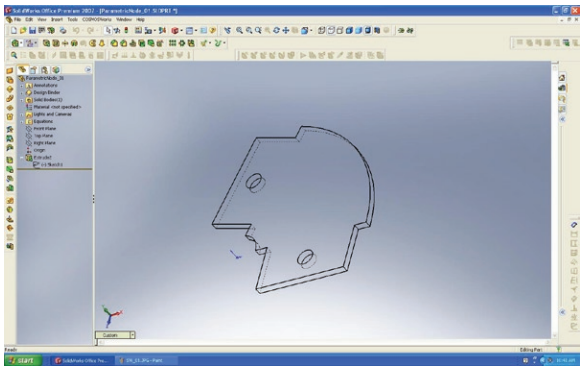
Why Solidworks?

The advancement of technology has enhanced us to shift the method of construction in architecture from repetitive components to variability in the components. the concept of objects with more variability makes us re-think in terms of topological relationships of components. we start to look at a part in a hierarchical way of relationships of what is driven and what is a driver.



Solidworks work flow

The basic idea is how geometric information is related to each other in a drawing. The majority of the work in solid works is about building the relationships. It is best to think of what your detail has as a variable (angle, length, etc) and basic construction outline.

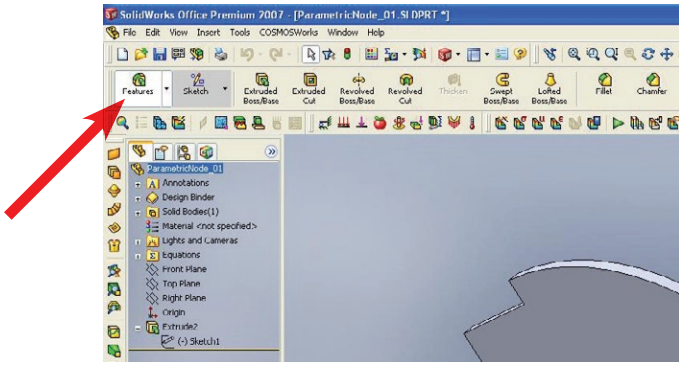


Basic modeling structure

In Solidworks and in any modeling software, the process of drawing and then giving form is the same. ‘sketch’ is all your drawing applications such as lines, circle, arc etc. and ‘feature’ is all your post draw operations, such as extrude, revolve, etc.

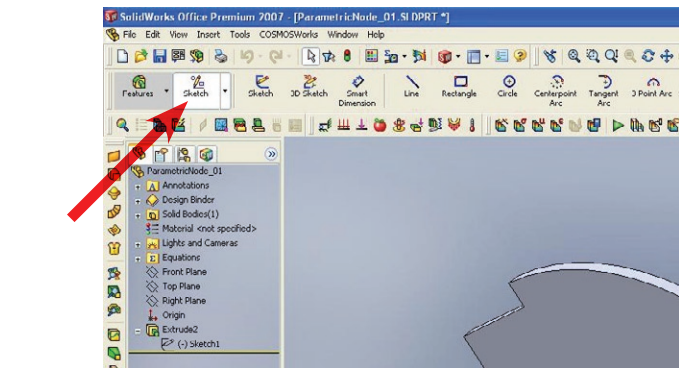
- Features
- Sketch

- Add Relation
- Display/Delete Relations
- Quick Snaps
- Mirror Entities
- Convert Entities
- Offset Entities
- Trim Entities
- etc



Simple relationships within ‘sketch’

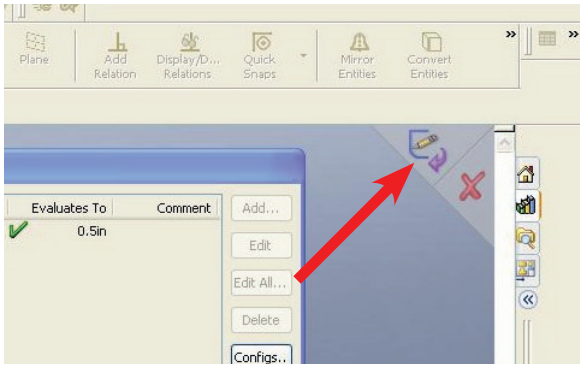
The difference between a CAD application drawing functions and Solidworks, is the ability to build a relationship amoungst entities (line, circle, arc etc.)



- Horizontal
- Vertical
- Collinear
- Perpendicular
- Parallel
- Equal
- Fix

Basic procedure to draw in Solidworks

1. go into Sketch > Line
2. it will ask you to select a plane to work in > select Top Plane
3. click on top right conner when done
4. in your FeatureManager design tree you have a Sketch1
5. select Sketch1, right click and select Edit Sketch
6. you can either draw more lines are build a relationship
7. select both lines, a properties window will pop up
8. under Add Relations choose any of the relationship you want



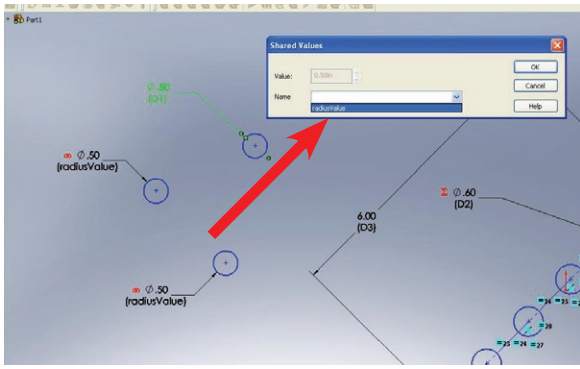
Making your drawing smart

There are several ways to building relationships in Solidworks. We will go through the following methods of building relationships.

- smart dimension
- link value
- making an equation

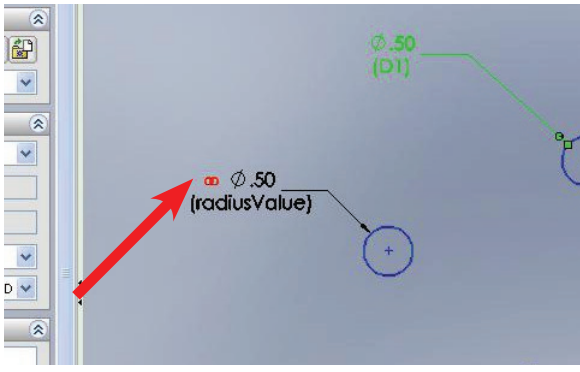
Smart dimension

1. any drawn entity (line, circle, etc) can be made a smart dimension
2. under sketch, select the smart dimension and select a line. it is like drawing a dimension but, it is `smart`, you can double click the dimension and type in a new value



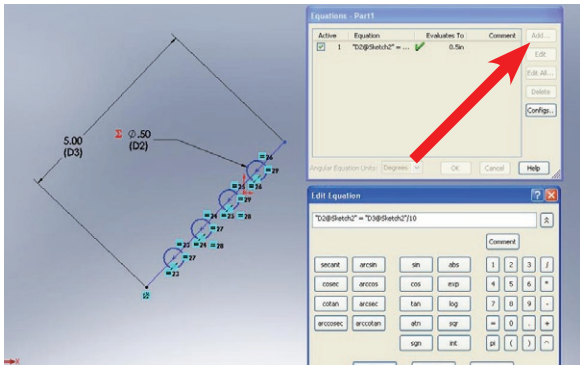
Link value

1. previously we have made a relationship through selecting the actual entities and assigning generic geometric relationships, but did not have dimensional relationship. we can also assign relationships between smart dimensions as `link value`, making them the same value, or through `equation` having a specific reference dimension and applying a mathematical property as add, subtract, multiply, divide, sin, cos, etc.
2. how to make `link value`
3. select a smart dimension, right click and select `link value`. a window will pop up called Shared Values. in the Name bar, type in any kind of name, later to be used.
4. if you had another entity that you wanted the same value, select the smart dimension of that entity, right click and select `link value`.
5. this time, you will see the name you have previously typed in as a selection. select and ok.

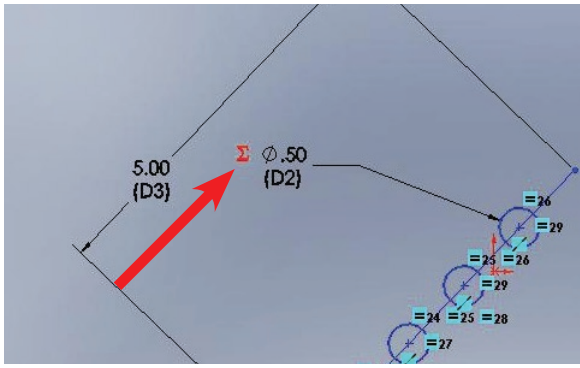


Equations

1. now you can see in your work space, a link icon beside the smart dimension and you can see the values are the same.
2. how to make `equation` relations
3. go to Tools > Equations... an Equation List Window will pop up.
4. select a smart dimension in your work space that you want to be driven by an equation.

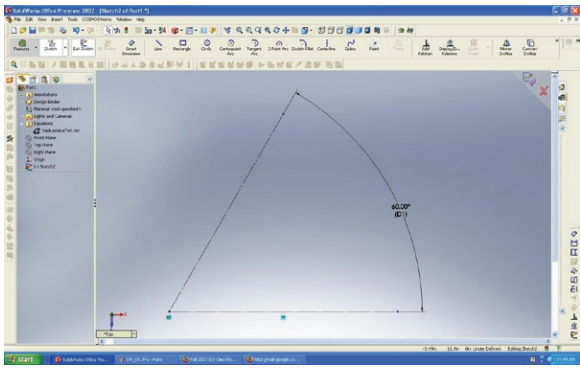


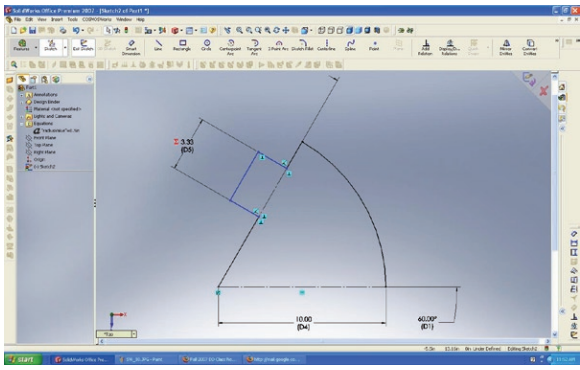
5. select Add... and and you will be able to add an equation.
6. “D2@Sketch2” = “D3@Sketch2”/10 Solidworks will automatically produce the syntax for the dimension names. in this case D2 is D3/10, so if the dimension of D3 is to be changed, D2 will update accordingly to the equations.

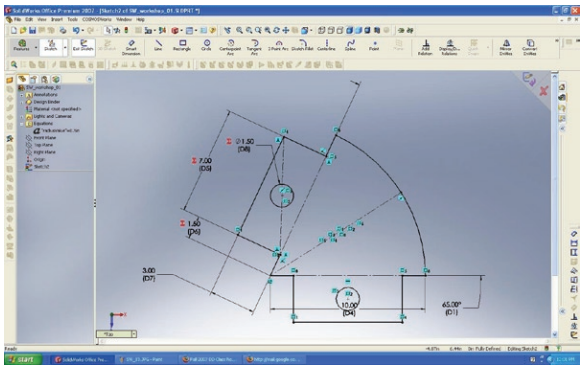


Detail example: parametric angle bracket between two planes

1. shifting from a repetitive component construct, we will build a smart component that has variability.
2. we must first see what the variability of the component are. in this case, an angle is a basic variable.
3. drawing a basic construction line that the angle is to be assigned through a smart dimension.
4. any other entity that is drawn with this construction lines will be driven by the construction line which is driven by the smart dimension.
5. once all the relationship are set and working, exit sketch.
6. select the sketch under the FeatureManager design tree.
7. unde Features > Extruded Boss/Base
8. draw a smart dimension on the thickness of our model.

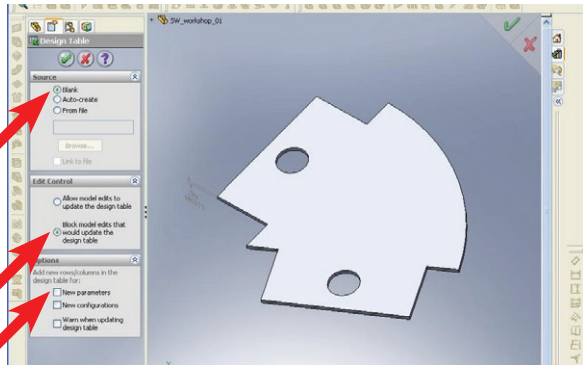


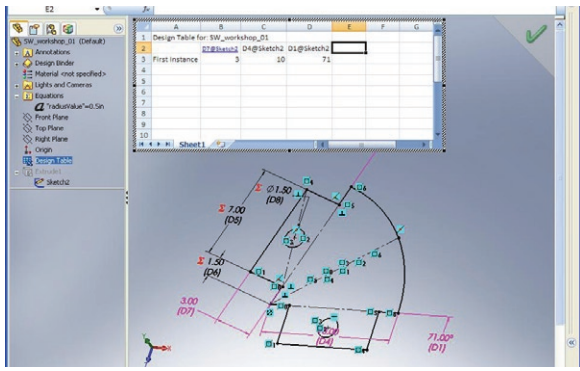


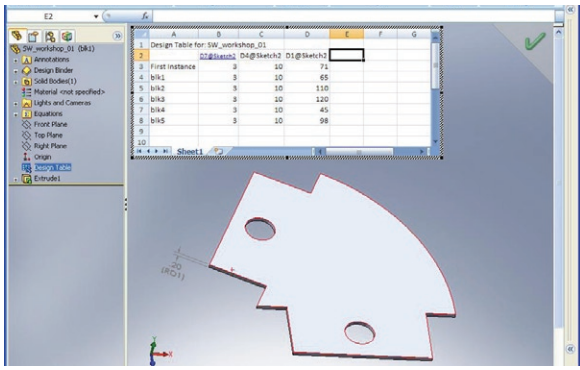


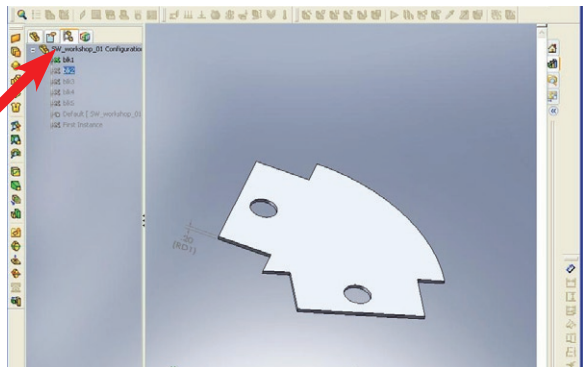
Making an excel spreadsheet to organize and assign different configuration

1. under Insert > Design Table...
2. chose source as Blank, the rest should be default.
3. In the PropertyManager: - Under Source, select Blank to insert a blank design table. - Under Edit Control, select Block model edits that would update the design table so you are not allowed to change the model if these changes would update the design table. - Under Options, clear New parameters and New configurations so that any changes you make to the model do not update the design table.
4. we will make a list of smart dimension that we want to input different configurations in the column direction
5. to do so we will double click on the smart dimension in the workspace.
6. once we have successfully made out spread sheet, as a test we will make different configurations in the spreadsheet.
7. in our ConfigurationManager we can see the names of the new instances we have made in our spreadsheet.
8. if we double click on our new instance names in our ConfigurationManager, we can see our instances update in our workspace.



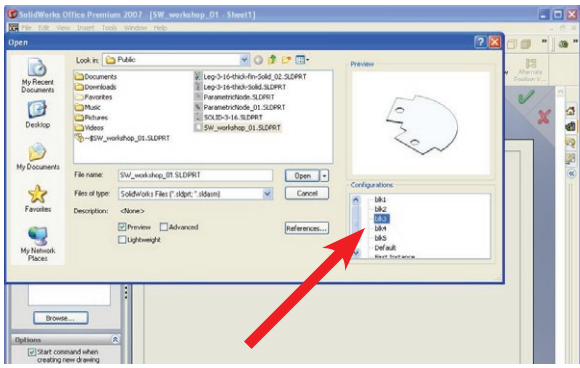


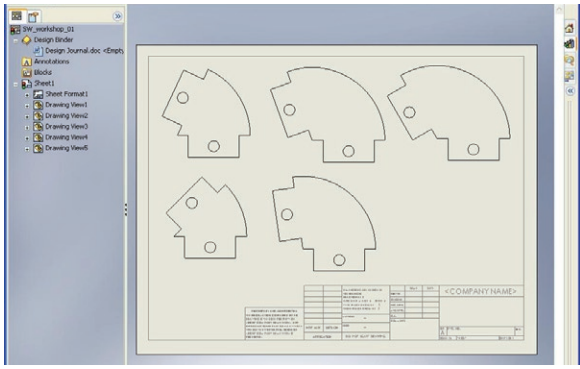




Making a dwg file to export to Rhino

1. File > New... > Draw
2. Model View > Part/Assembly to Insert > Browse... > choose the part file we've made.
3. When opening we can also choose which configuration we would like to draw.
4. place in screen and repeat the process of above to draw the rest.
5. once placed all components, File > Save As... > save it as a dwg file.

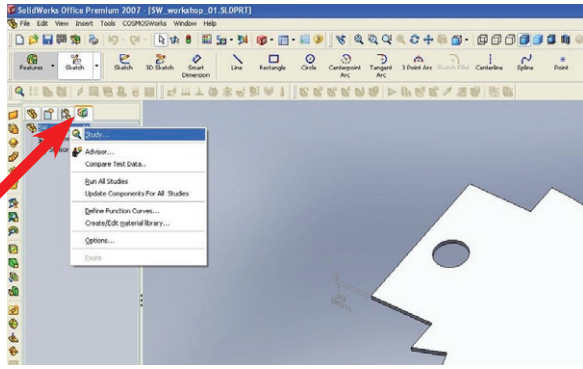


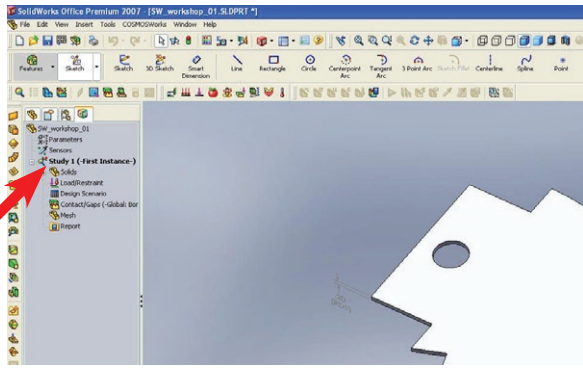


Finite Element Analysis (FEA) of component

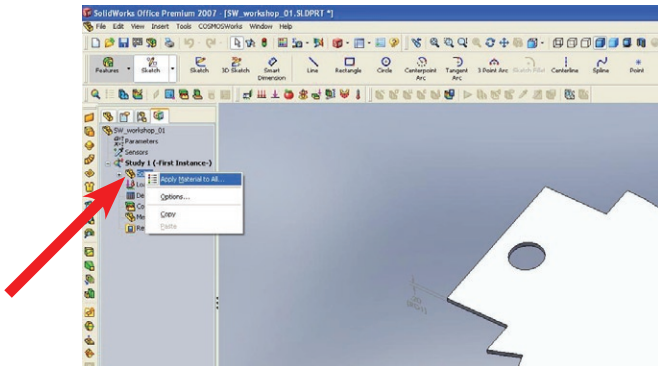
basic import check points from Rhino
- closed object
- scaled by units

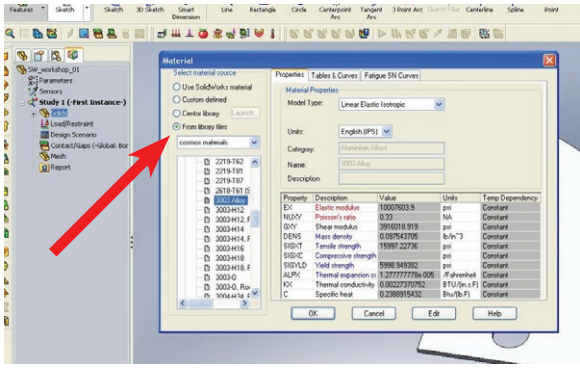
1. go to your COSMOSWorks Manager
2. right click on your object name and select Study...
3. you well then have a tree called Study 1 (-First Instance-) we will now set up the following elements:
    - assigning material
    - assigning restraints
    - assigning forces
    - running the analysis



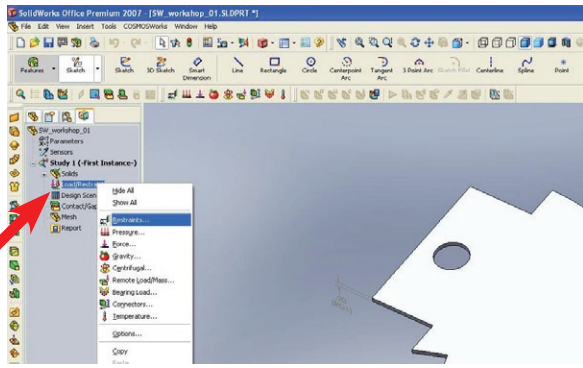


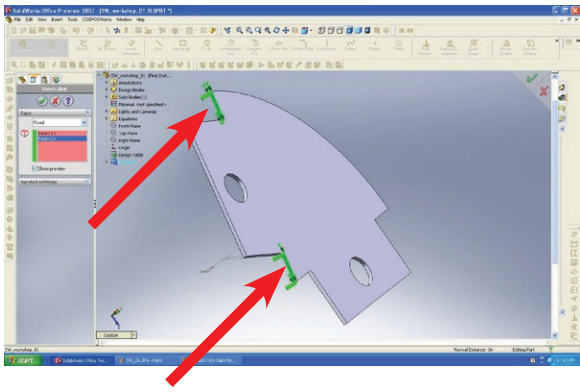
4. right click on Solids in your tree and select `Apply Material to All...` and assign the material from library files.





5. right click on Load/Restraint in your tree and select `Restraints...` and select the appropriate elements.
6. right click on Load/Restraint in your tree and select `Forces...` and select the appropriate elements.

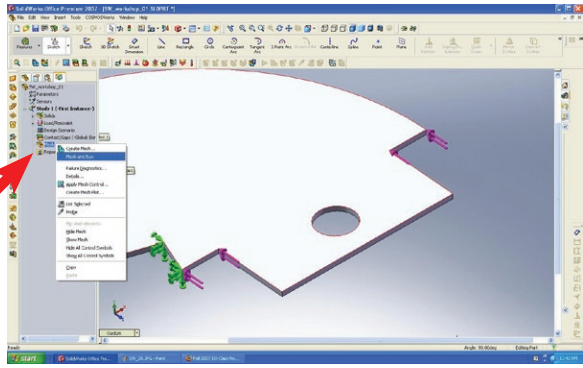


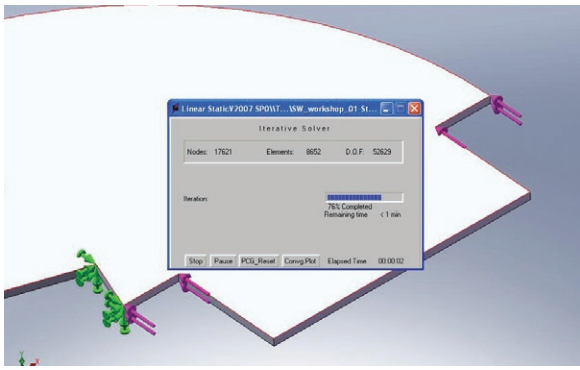


7. now we are set to run the FEA. right click on Mesh in your tree

and select `Mesh and Run`

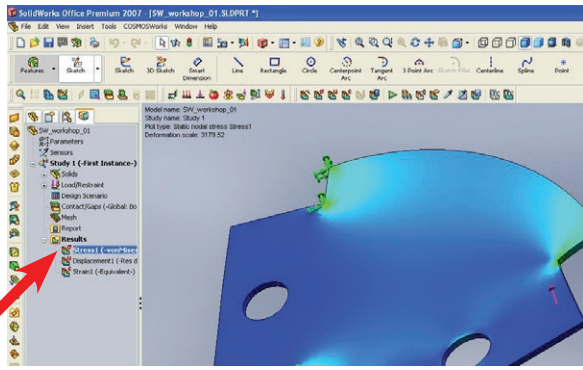
8. if it is successfully analyzed we have have under our Results the following analysis:
   - Stress1
   - Displacement1
   - Strain1
9. right click on the any of the above and select `Show`
10. again right click and select `Animate...` and this will show you the deformation in animation.
11. to make an animation, check the `Save as AVI FIle` in the property bar and set Options... and Browse... to were to save.





Structural Analysis of NURBS surface

1. import a surface from rhino.
2. give a thickness by Feature > Shell
3. assign a force to the whole surface, choose the plane you want the direction of the force to be applied (top plane)



The rough allowable stress for aluminum is 100 MPa (100,000,000 Pascals, or 1x10e8 Pascal or 1x10e8 N/m2).