



3D models I usually make these days in Spaceclaim Engineer, to me the most proficient between those that I know, Others such Autodesk Autocad, Autodesk Inventor, Solidworks, Creo Elements, Abaqus, Ansys etc may be as proficient in solid modeling as this if not more. I use to work in mm and pass the model in ACIS .sat format to Autodesk Simulation Multiphysics.

In our non english european environment I have discovered that one avoids some dimensional setbacks if one keeps tight to the mm unit when planning to pass some model to one FEM program. So for these things except that I neglect of forget, I model in mm.

Autodesk Simulation Multiphysics 2012 [FEA Editor - [InsetThrough.fem]]

Mesh Draw Setup Analysis Selection View Tools Getting Started

Generate 3D Mesh 3D Mesh Settings Generate 2D Mesh Joint Inventor Parameters External Fluid Bolt Internal Fluid 4 Point Rectangular Between 2 Objects 3 Point Triangular Divide 1 Object 4 Object 3D 8 Point 3D Add to Selected Nodes Specify Automatic Visibility Refinement Points

FEA Editor Results Report

1 < Design Scenario 1 >

- Unit Systems < Metric mks (SI) >
- Analysis Type < Static Stress with Linear I
- Planes
- Load and Constraint Groups
- Parts
 - Part 1 < Hormigón.Sólido >
 - Element Type < Brick >
 - Element Definition
 - Material < Concrete (Fairly High S
 - CAD Mesh Options
 - Surfaces
 - Part 2 < Acero.Sólido >
 - Element Type < Brick >
 - Element Definition
 - Material < Steel (ASTM - A572) >
 - CAD Mesh Options
 - Surfaces
- Mesher
- Coordinate Systems
- Contact (Default: Bonded)

We will run first an elastic analysis where the two materials keep **BONDED** and elastic all along the deformation range, whichever the level of stress is attained in any of them. This is usually an optimistic and unconservative approach for RC models.

Here we have the model imported. The two materials show immediately. Do not forget to check the scale to see if dimensions look ok. Shown in meters now, they are.

To diminish input work, I accept a default concrete of $E_c=31000$ MPa instead of the 29500 that more properly corresponds to our 5 ksi concrete.

Similarly I choose A572 for the steel since there is scarce variation between common steels Young Modulus, and this program is not going to check any strength.

0.000 0.558 m 1.115 1.673

Ready SCRL

21:15 27/05/2011

Autodesk Simulation Multiphysics 2012 [FEA Editor - [InsetThrough.fem]]

Mesh Draw Setup Analysis Selection View Tools Getting Started

Generate 3D Mesh 3D Mesh Settings Generate 2D Mesh Joint Inventor Parameters Bolt Internal Fluid External Fluid 4 Point Rectangular 2 Objects 3 Point Triangular 4 Object 3D 8 Point 3D Divide 1 Object Add to Selected Nodes Specify Automatic Visibility Refinement Points

FEA Editor Results Report

1 < Design Scenario 1 >

- Unit Systems < Metric mks (SI) >
- Analysis Type < Static Stress with Linear I
- Planes
- Load and Constraint Groups
 - 1 < Surface Boundary Conditions >
 - Surface Boundary Condition 1 < U
 - Surface Boundary Condition 2 < U
 - Surface Boundary Condition 3 < U
 - Surface Boundary Condition 4 < U
 - 2 < Surface Forces >
 - Surface Force 3 < Unnamed >
 - 3 < Surface Forces >
 - 4 < Surface Forces >

Modifying 1 Surface Force Object

Magnitude: 598110 N

Direction:

- ☐ Normal
- ☐ X X: 0.922204735533475
- ☐ Y Y: 0.386701985719795
- ☐ Z Z: 0
- ☒ Custom Vector Selector...

Description:

OK Cancel

Even inconsequential to this analysis, I have set this time gravity in -Y direction. Now I have fixed the 4 sides of the concrete and applied the loads to the upper plate. 2 opposed forces in the long faces give entry to the moment, whereas the vectorial compound of the shear makes for the last one.

0.000 0.433 m 0.866 1.299

Ready

ES 21:32 27/05/2011























