

X

Boundary Conditions and Loads

This chapter includes material from the book “**Practical Finite Element Analysis**”. It also has been reviewed and has additional material added by Matthias Goelke.

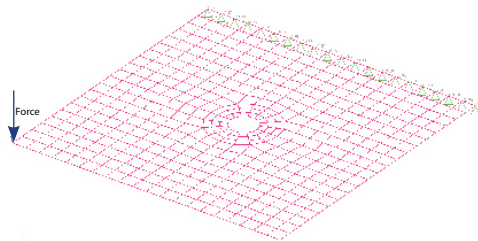
10.1 Boundary Conditions

The boundary condition is the application of a force and/or constraint. In HyperMesh, boundary conditions are stored within what are called load collectors. Load collectors may be created using the right click context menu in the Model Browser (Create > Load Collector).

Quite often (especially at the beginning) a load collector is needed for the constraints (also called SPC – Single Point Constraints) and a second one is needed for the forces and/or pressures. Keep in mind, you can place any constraints (e.g. nodes constraint) with respect to dof 1, or nodes with constraints dof123, etc. in a single load collector. The same rule applies for forces/pressures. They are stored within a single load collector regardless of their orientation and magnitude.

In the following you will learn some basic principles about the way forces may be applied to a structure.

1. Concentrated load (at a point or single node)

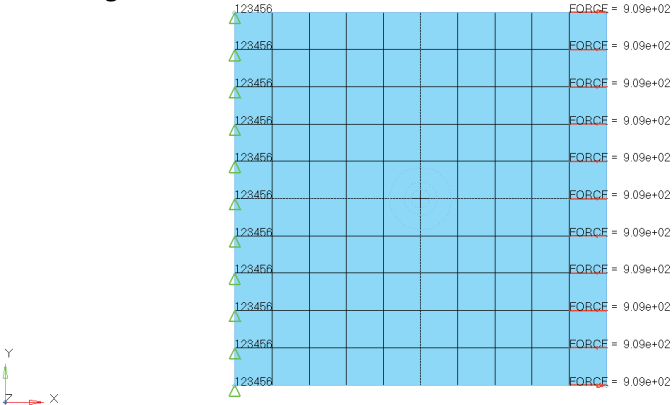


Applying forces to single nodes may cause irritating effects, especially while looking at the stresses in this area. Typically concentrated loads (i.e. forces on a single node) impose high stress gradients. Even though the high stresses are correct (i.e. force applied to an infinitesimal small area) one needs to ask whether this kind of loading is reasonable at all? In other words, which real-life loading scenario is represented in the model?

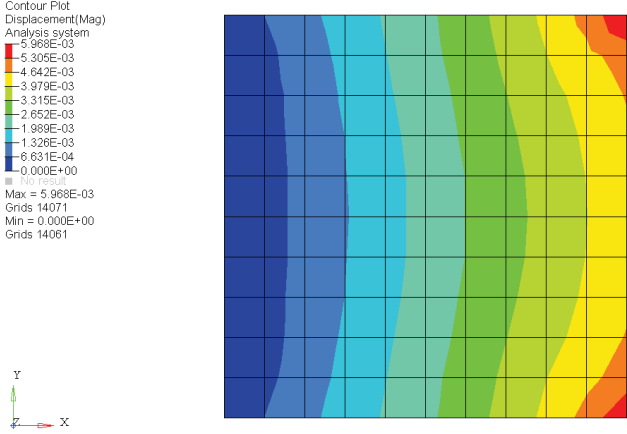
Therefore, forces are commonly applied as distributed loads, namely line loads, and surface loads which are “closer” to reality.



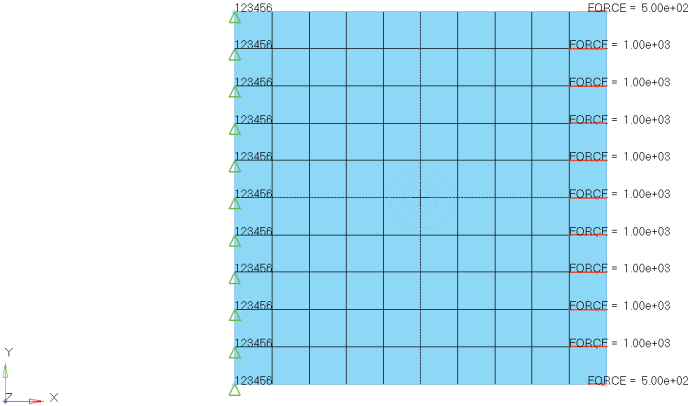
2. Force on line or edge



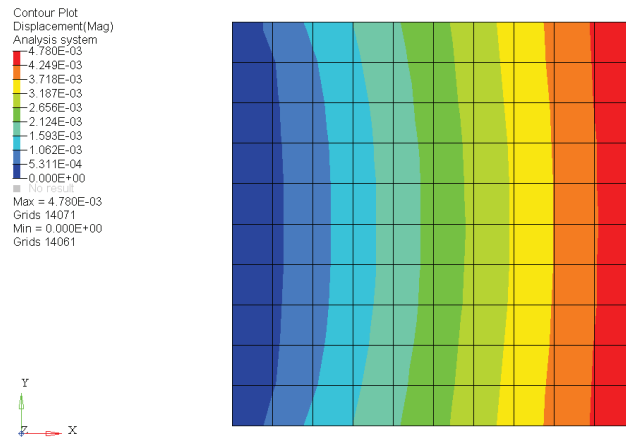
In the above figure, a plate subjected to 10.000 N. The force is equally applied to all (11) nodes at the model edge. Note that the forces at the corner will act only on 1/2 of the element edge.



The figure above is a displacement contour plot. Note the red "hotspots" located in the corners of the plate. The local displacement maximas are imposed by boundary effects (i.e. the forces applied to the corner nodes act only on 1/2 of the element edge), however we applied a constant magnitude along the plates edge.



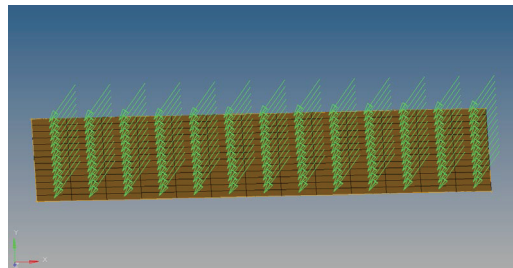
In the example above, the plate is also subjected to 10.000 N. This time the forces at the corners are just 1/2 the magnitude of the other applied forces.



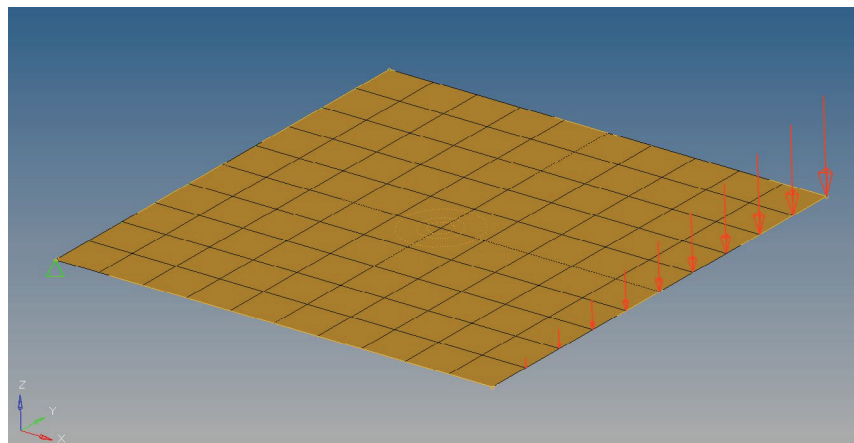
The figure above is the displacement contour plot from the plate_distributed.hm file. The displacements are more evenly distributed.

3. Traction (or “oblique” pressure)

Traction is a force acting on an area in any direction other than the normal direction. A force acting normal to an area is known as pressure.



4. Distributed load (Force varying as equation)

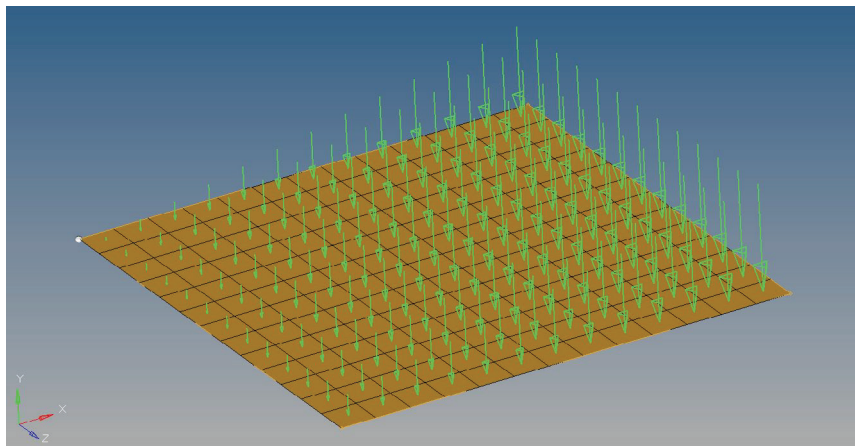


How to apply a force with “variable” magnitude?



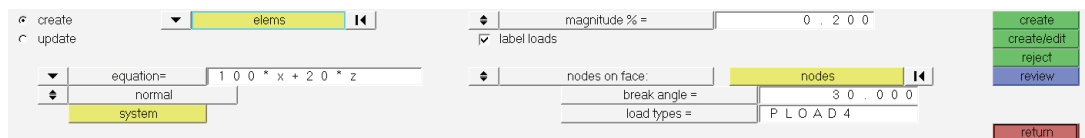
Distributed loads (varying with respects to the coordinates of the nodes or elements) can be applied by means of an equation. In the displayed example, the magnitude of the applied forces varies with respect to the nodal y-coordinate (i.e. the force is acting in negative z-direction and increases along positive y-direction by a factor of 10 respectively).

5. Pressure and Vacuum



In the image above, a distributed load (pressure) is shown. The origin of the plate is located at the highlighted node in the left upper corner.

How to apply a pressure with “variable” magnitude?



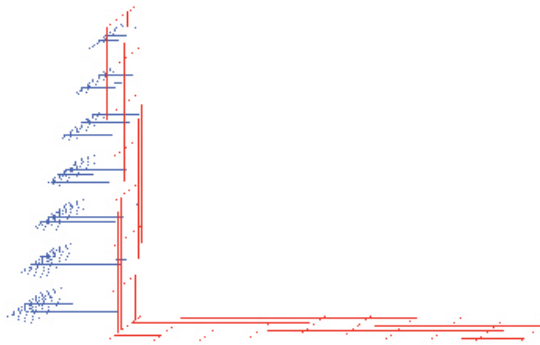
In the example above, the magnitude of the applied pressure depends on the x- and z- direction of the elements centroid.



6. Hydrostatic pressure

Civil engineering applications: Dam design. Mechanical engineering applications: Vessels / tanks containing liquid.

Hydrostatic pressure is zero at the top surface of the liquid and is maximum ($= \rho * g * h$) at the bottom surface. It varies linearly as shown in the following figure:

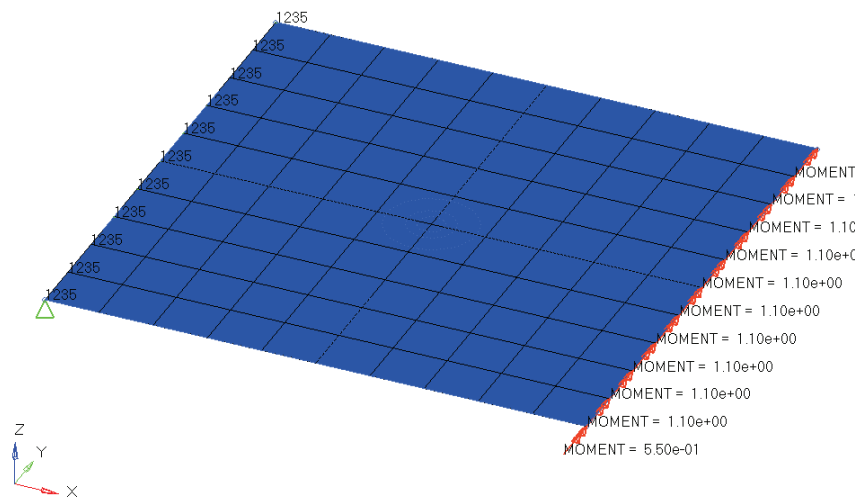


The hydrostatic pressure is applied taking into account the element centroids location (vertical position, h).

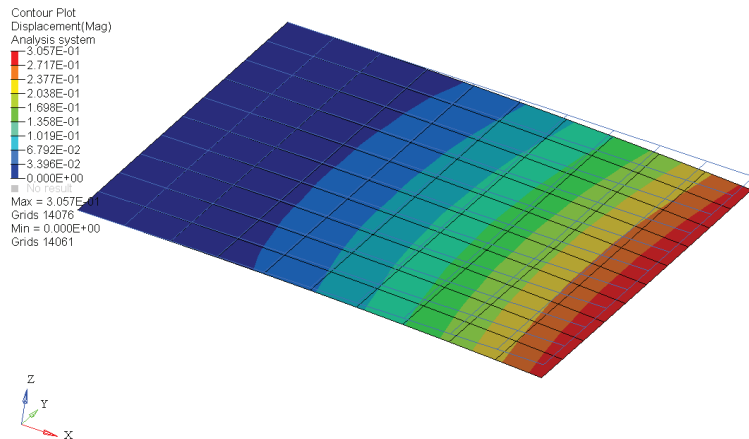
7. Bending moments

The convention for representing a force is a single arrow (\rightarrow), pointed towards the direction of the force .

A moment is represented by a double arrow, where the direction of the moment is decided by the right hand rule.

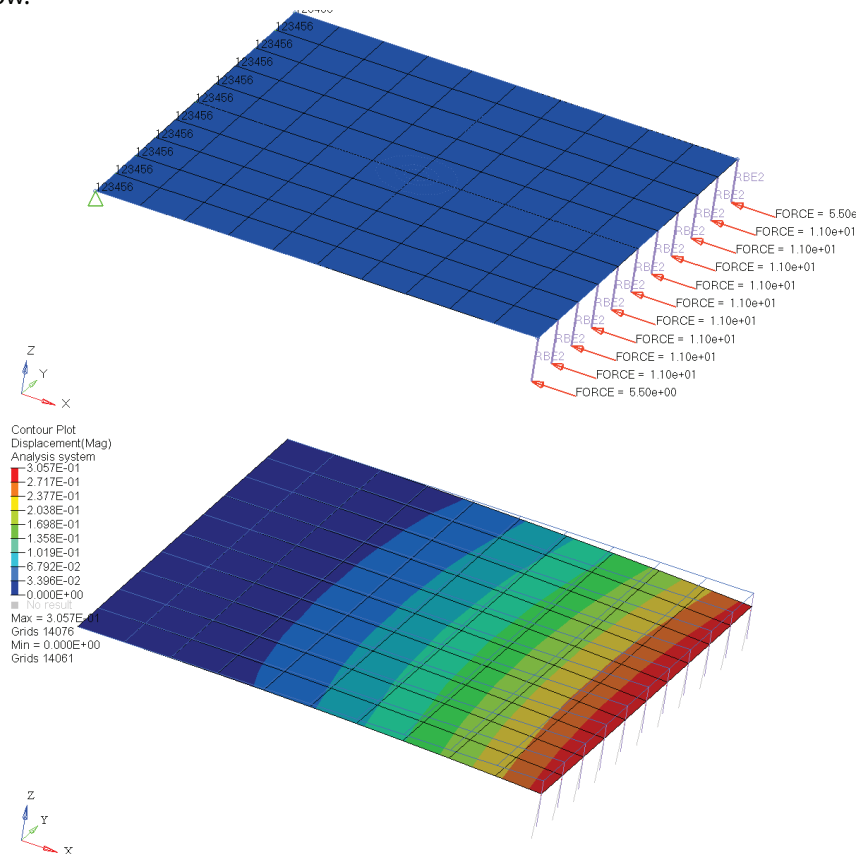


The nodes along the plates edge are subjected to moments. As a consequence, the nodes will tend to rotate with respect to the y-axis (dof 5).



In the image above, the plate is subjected to moments along its edge on the right. The displacements are scaled by 100 and the initial model is displayed as wire frame.

The moment applied to the nodes in the figure above can also be modeled by adding rigid elements (RBE2) to each node which are then subjected to corresponding forces. In this example, the RBE2 would be oriented in the z-direction and subjected to a force acting in x-direction as shown in the figures below.

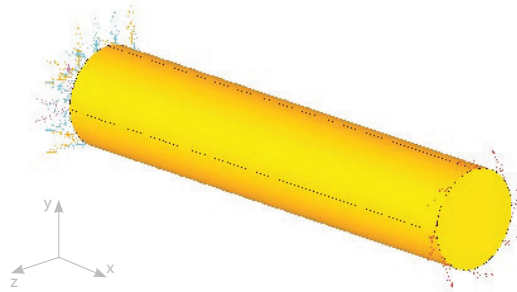


While postprocessing the results make sure that the RBE2 results (i.e. nodal position) are not postprocessed (just display the displacements of the shell elements).

8. Torque:



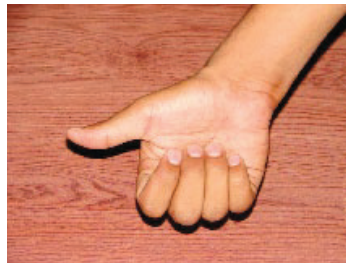
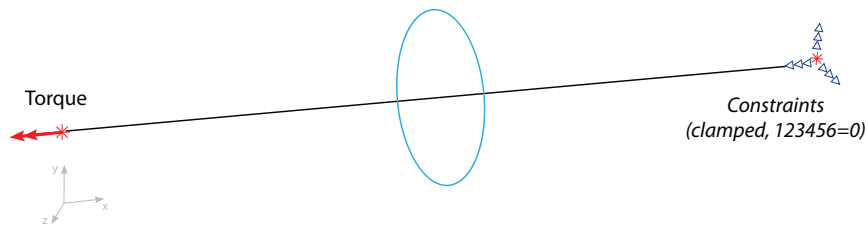
What is torque? Are torque and bending moments different?
Torque is a bending moment applied parallel to the axis of a shaft (M_x).



Torque or M_x causes shear stresses and angular deformation, while the effect of the other two moments (M_y, M_z) is the normal stress and longitudinal deformation.

How to decide the direction of torque (clockwise or anticlockwise)

It is based on the right hand rule. Point the thumb of your right hand towards the arrow direction. The direction of your fingers indicate the direction of torque.



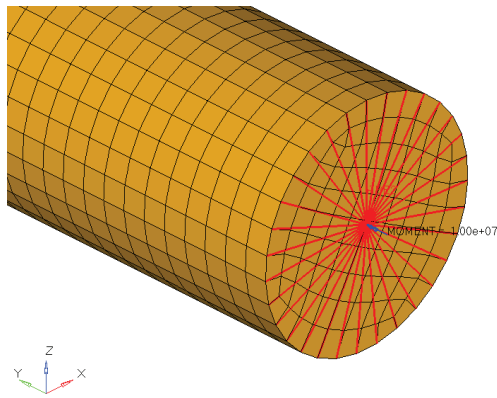
How to apply torque for solid elements (brick /tetra) :

As solid elements have no rotational stiffness at the grid points (only 3 translational dofs), a common mistake is to apply torques and moments to the grid points of solid elements directly.

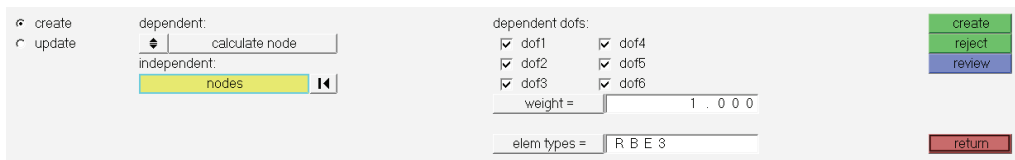
The correct way to apply a moment to a solid model is to use an RBE2 or RBE3 rigid-body element. The rigid-body element distributes an applied moment into the solid element model as forces.

Rigid element connection RBE2

A center node is connected to the outer edge nodes using a rigid element (RBE2). The torque is then applied at the center node.

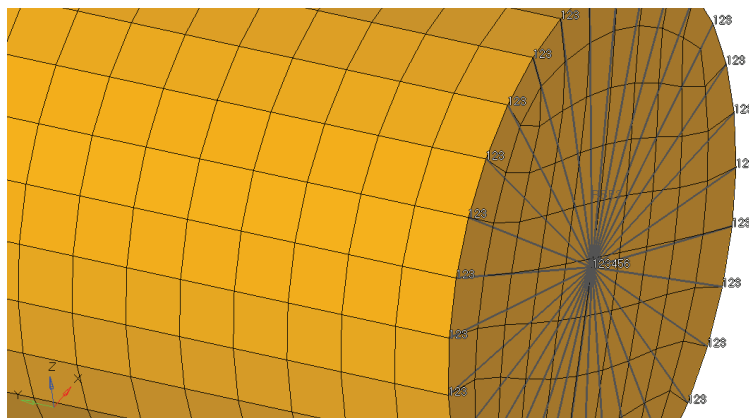


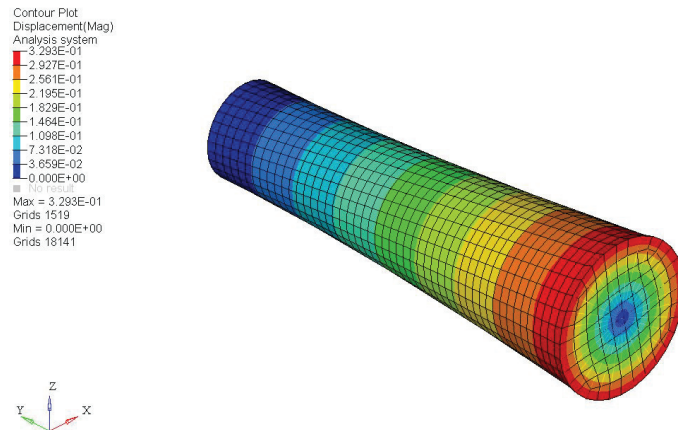
Alternatively, you may use an RBE3 elements instead:



Select the nodes at the outer contour of the shaft as independent nodes. The dependent node may then be determined automatically. It is pretty easy.

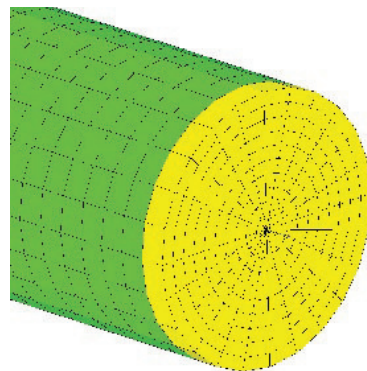
However, care needs to be taken with respect to the dof's being referenced. The node of the solid shaft possess translational degrees of freedom only (dof 123). The dependent node also allows for rotational displacements (dof 123456). If the rotational dof (in this example dof 5; rotation y-axis) of the dependent node is not "activated", the moment will not be transferred to the independent nodes.






Shell element coating:

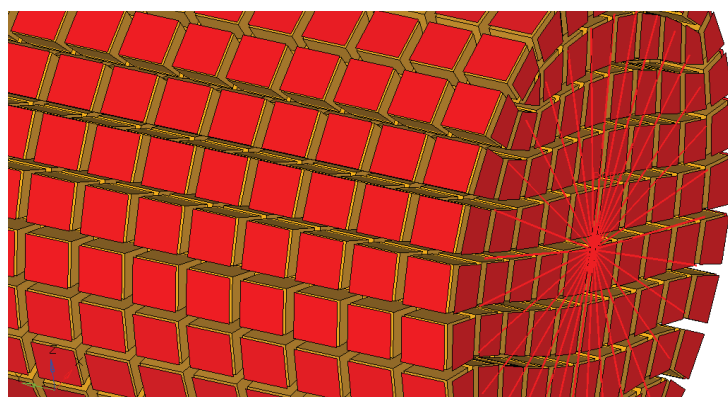
On the brick/tetra element outer face additional quad/tria (2-D) elements coating the solid elements are created. The thickness of these shell element should be negligible (so that it would not affect the results). Moment could now be applied on all the face nodes (moment per node = total moment / no. of nodes on the face).




The shell element coating can be easily created within HyperMesh. Create faces using the Faces panel.

This panel can be accessed through the  toolbar icon which is displayed using View > Toolbars > Checks.

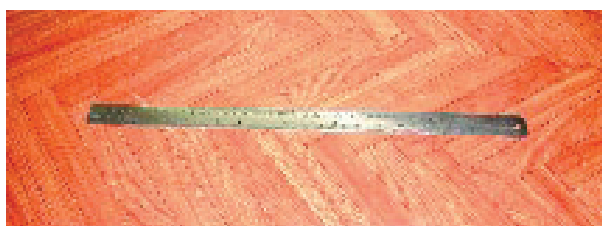
The faces (nothing more than 2-D plot elements) are automatically created and stored in a component collector named ^faces. All you need to do is treat these elements as regular elements (i.e. rename the component collector, assign materials and properties)



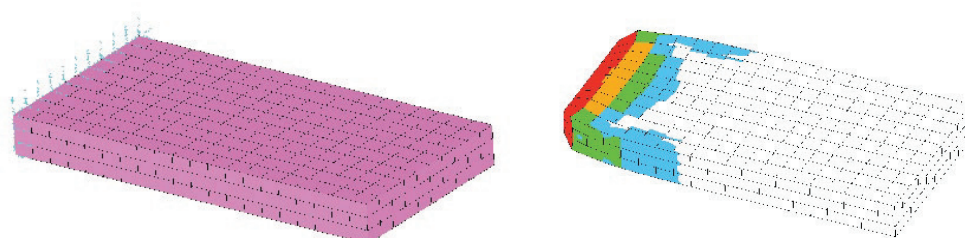
In the figure above, the elements of the shaft are displayed by means of the shrink element command . The orange elements are the hexahedral elements (3-D), and the red elements are the 2-D elements placed on the free faces of the 3-D elements.

9. Temperature loading

Suppose a metallic ruler is lying on the ground freely as shown in the figure below. If the temperature of the room is increased to 50 degrees, would there be any stress in the ruler due to temperature?



There will be no stress in the ruler. It will just expand (thermal strain) due to the higher temperature. Stress is caused only when there is a hindrance or resistance to deformation. Consider another case, this time one end of the metallic object is fixed on a rigid wall (non conductive material). Now if the temperature is increased, it will produce thermal stress (at the fixed end) as shown below.



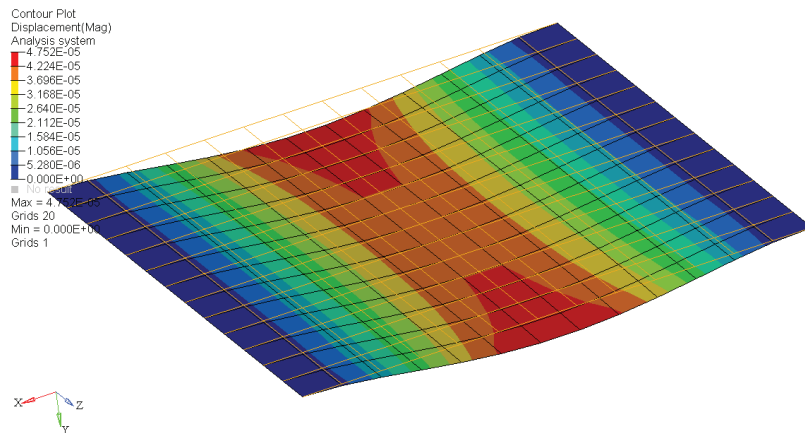
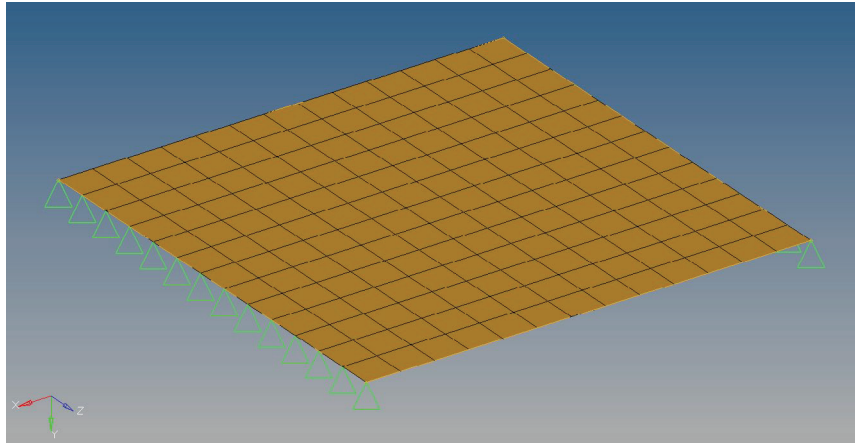
For thermal stress calculations, the input data needed is the temperature value on nodes, the ambient temperature, thermal conductivity, and the coefficient of linear thermal expansion.

10. Gravity loading : Specify direction of gravity and material density.



A load collector with the card image GRAV is needed. Please keep your unit system in mind.

```
GRAV      2      0.9810.00  0.000  1.000  0.000
```



11. Centrifugal load

The user has to specify the angular velocity, axis of rotation, and material density as input data.

```
RFORCE    3      0      0      0.000  0.000  0.000  
RACC  
0.000
```

The RFORCE Card Image defines a static loading condition due to a centrifugal force field.



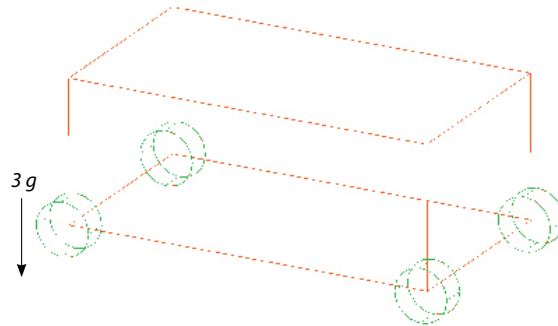
12. 'g' values (General rules) for full vehicle analysis :

- Vertical acceleration (Impact due to wheel passing over speed braker or pot holes): 3g
- Lateral acceleration (Cornering force, acts when vehicle takes a turn on curves): 0.5 to 1 g
- Axial acceleration (Braking or sudden acceleration): 0.5 to 1 g

12. One wheel in ditch

The FE model should include all of the components (non critical components could be represented by a concentrated mass). The mass of the vehicle and FE model mass, as well as actual wheel vehicle reactions and FE model wheel reactions, should match.

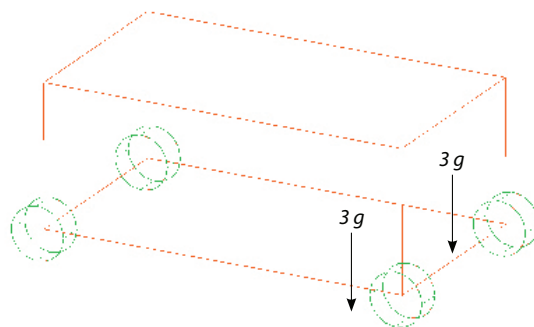
While applying a constraint, the vertical dof of the wheel, which is in considered in the ditch, should be released. The appropriate constraints should be applied on the other wheels so as to avoid rigid body motion. Specify the gravity direction as downward and magnitude = $3 \times 9810 \text{ mm/sec}^2$



Another simple but approximate approach (since most of the time either we do not have all the CAD data for the entire vehicle or sufficient time for a detailed FE model) is to apply 3 times the reaction force on the wheel which is in the ditch. Suppose the wheel reaction (as per test data) is 1000 N. Therefore, apply 3×1000 or 3000 N force in the vertically upward direction and constraint the other three wheels enough to avoid rigid body motion. This approach works well for relative design (for comparison of two designs).

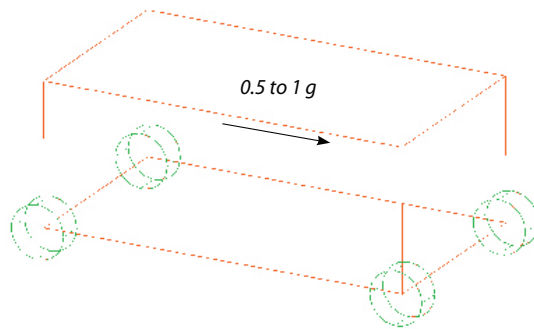
13. Two wheels in ditch:

Same as discussed above, except now instead of one wheel, two wheels are in the ditch. One wheel in the ditch causes twisting, while two wheels in the ditch produces a bending load.



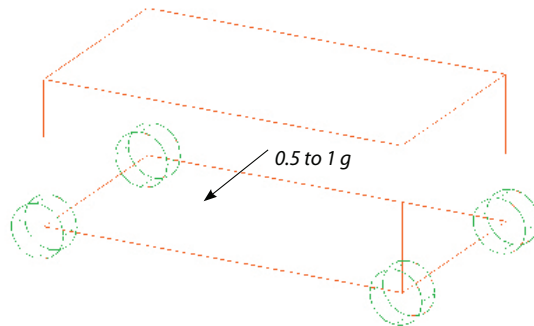
14. Braking :

Linear acceleration (or gravity) along the axial direction (opposite to vehicle motion) = 0.5 to 1 g.



15. Cornering:

Linear acceleration along the lateral direction = 0.5 to 1 g.



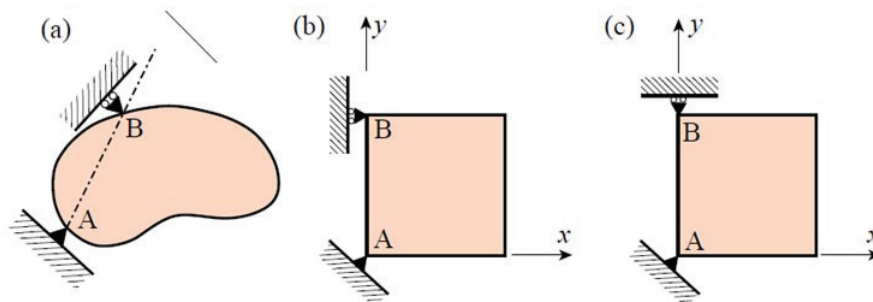
10.2 How to Apply Constraints

A beginner may find it difficult to apply boundary conditions, and in particular, constraints. Everyone who starts a career in CAE faces the following two basic questions:

- i) For a single component analysis, should forces and constraints be applied on the individual component (as per the free body diagram) or should the surrounding components also be considered.
- ii) At what location and how many dof should be constrained.

Constraints (supports) are used to restrain structures against relative rigid body motions.

Supporting Two Dimensional Bodies





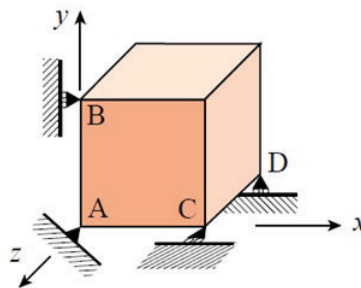
The figure above depicts two-dimensional bodies that move in the plane of the paper (taken from: <http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/IFEM.Ch07.d/IFEM.Ch07.pdf>).

If a body is not restrained, an applied load will cause infinite displacements (i.e. the FEM program will report a rigid body motion and will abort the run with an error message). Hence, regardless of loading conditions, the body must be restrained against two translations along x and y , and one rotation about z . Thus the minimum number of constraints that has to be imposed in two dimensions is three.

In figure (a) above, the constraint at A fixes (pins) the body with respect to translational displacements, whereas the constrain at B, together with A, provides rotational restraint. This body is free to distort in any manner without the supports imposing any deformation constraints.

Figure (b) is a simplification of figure (a). Here the line AB is parallel to the global y -axis. The x and y translations at point A, and the x translation at point B are restrained, respectively. If the roller support at B is modified as in figure (c), a rotational motion about point A is possible (i.e. the rolling direction is normal to AB). This will result in a singular modified stiffness matrix (i.e. rigid body motion).

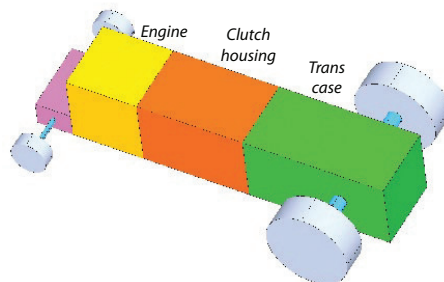
Supporting Three Dimensional Bodies



The figure above (taken from: <http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/IFEM.Ch07.d/IFEM.Ch07.pdf>) illustrates the extension of the freedom restraining concept to three dimensions. The minimal number of freedoms that have to be constrained is now six and many combinations are possible.

In the example above, all three degrees of freedom at point A have been fixed. This prevents all rigid body translations, and leaves three rotations to be taken care of. The x displacement component at point B is constrained to prevent rotation about z , and the z component is fixed at point C to prevent rotation about y . The y component is constrained at point D to prevent rotation about x .

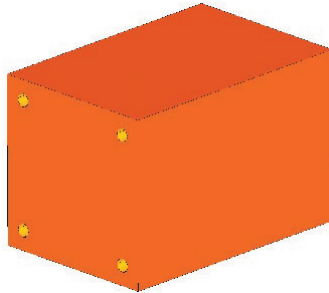
1. Clutch housing analysis





The aim is to analyse (only) the clutch housing. The clutch housing is connected to the engine and transmission case by bolts. There are 2 possibilities for analysis:

Approach 1) Only the clutch housing is considered for the analysis. Therefore, apply forces and moments as per the free body diagram and constrain the bolt holes on both the side faces (all dof = fix).

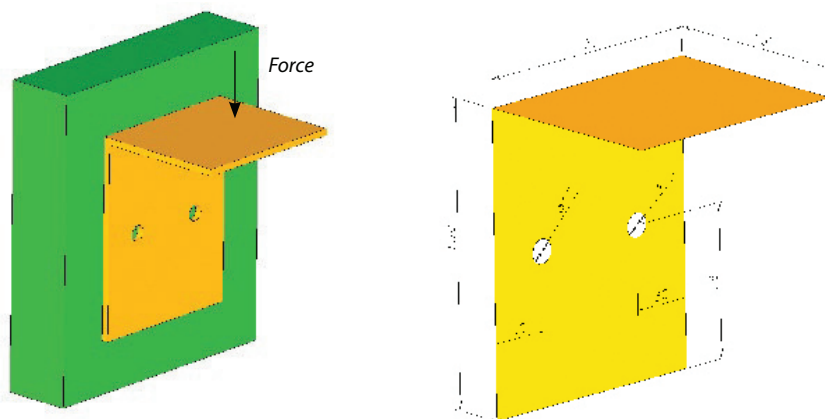


Approach 2) Model at least some portion of the engine and transmission case at the interface (otherwise model both these components completely with a coarse mesh by neglecting small features). Then, represent the other components like the front axle and rear axle using beam elements (approximate the cross sections). Apply constraints at the wheels (not all the dof are fixed but only minimum dofs required to avoid rigid motion or otherwise inertia relief / kinematic dof approach). Please note that the clutch housing, being the critical area, should be meshed fine.

The second option is recommended as the stiffness representation, as well as constraints, are more realistic. The first approach, fixing both sides of the clutch housing, results in over constraining and will lead to safer results (less stress and displacement). Also, it is not possible to consider special load cases like one wheel in ditch, two wheels in ditch, etc.

2. Bracket analysis

Problem definition: A bracket fixed in a rigid wall is subjected to a vertically downward load (180 kg).

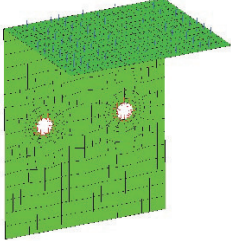
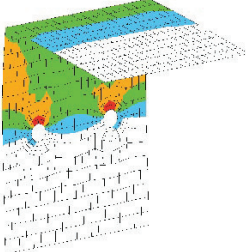
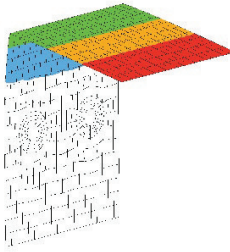
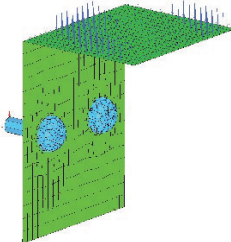
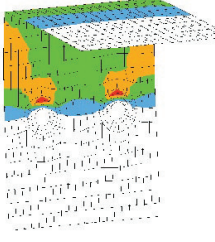

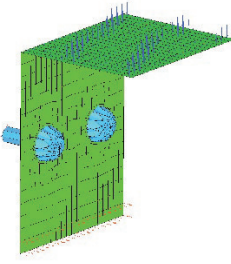
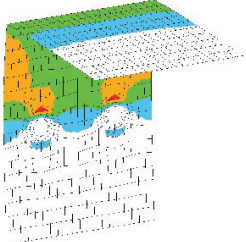
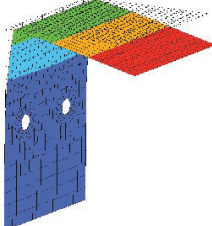


If above problem is given to CAE engineers working in different companies then you will find different CAE engineers applying constraints differently:

- i. Fix the bolt hole edge directly.



- ii. Model the bolt using rigid / beam elements and clamp bolt end.
- iii. Model the bolt, clamp bolt end, bracket bottom edge translations perpendicular to surface fixed.

	Stress N/mm ²	Displacement mm
Bolt holes edge fixed 	993 	15.5 
Bolt – modeled by beam elements 	770 	16.2 
Bolt by beams and Bracket bottom edge only z-dof fixed 	758 	15.8 

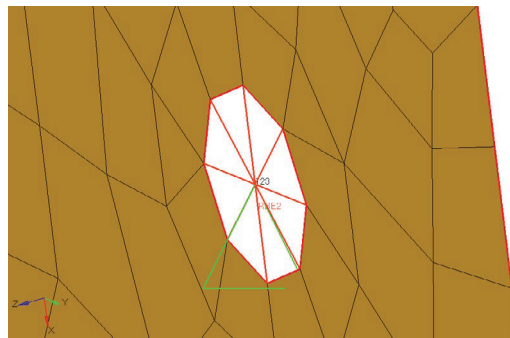
Applying a direct constraint on the hole edge causes very high stress. The second method shows the deformation of the bracket bottom edge, which is not realistic. Method 3 is recommended. Please observe the difference in the magnitude of the stress and displacement.

In some organizations it is a standard practice to neglect stress at the washer layer elements (washer area and one more layer surrounding the beam/rigid connection) due to high stresses observed at the beam/rigid and shell/solid connections.

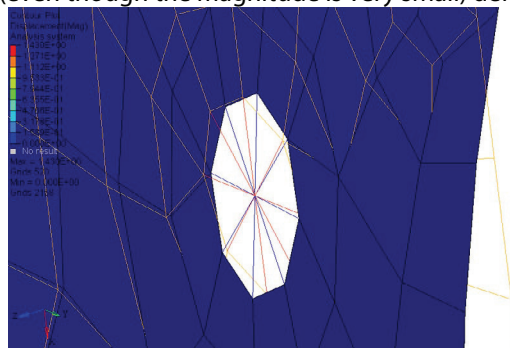
A modified version of the bracket is depicted in the figure below. This time the bracket is pinned to the “wall” by means of 3 simplified screws/bolts. The bolts are represented by rigid elements (RBE2). The center of the rigid elements is fixed/constrained with respect to any translational displacements



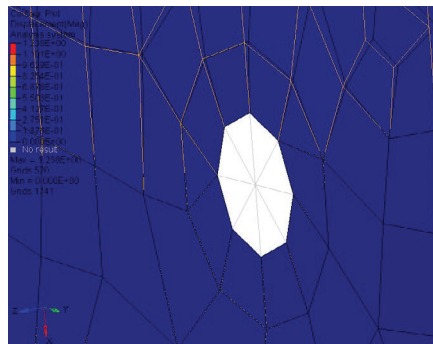
(dof 1-3).



What would you expect to happen? Even though this kind of constraint seems reasonable (i.e. bracket mounted to the wall), the unconstrained rotational dofs allow the central node to rotate. Thus, the hole itself (even though the magnitude is very small) deforms (see figure below).



In the figure above, the displacements are scaled by 100. The undeformed shape is plotted in a wireframe mode. Note the deformation of the hole. Is/was this effect really intended?

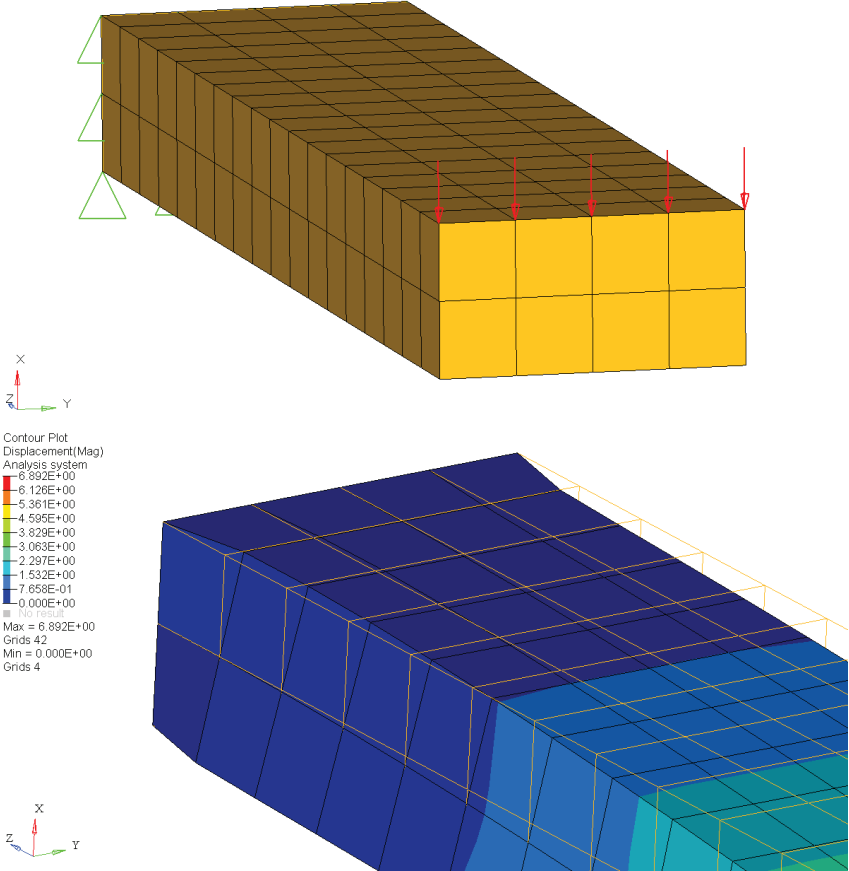


In the figure above, the displacements are scaled by 100. The undeformed shape is plotted in a wireframe mode. All the dofs of the central node are constrained. The hole remains its initial shape and "position".

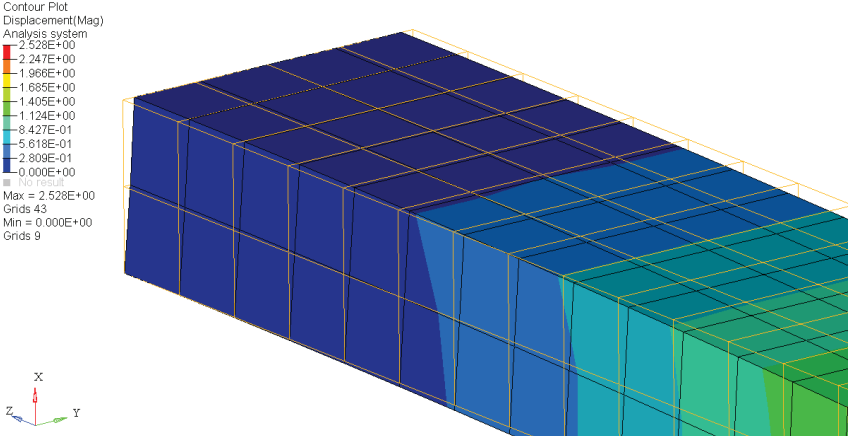
Another illustrative example regarding the effects of the boundary constraints on the modelling results is shown in the figure below. The translational degrees of freedom (dof 123) of the nodes at the rear end of the cantilever are constrained. At the tip a uniform force is applied in negative x-direction.



What would you expect to happen, especially in the vicinity of the constraints?



In the figure above, all the nodes at the rear are constrained with respect to any translational displacements. The displacements in the y- and x-direction are scaled by a factor of 200 and 5, respectively. The undeformed mesh is shown as wireframe (orange). Note the thickening at the base of the cantilever accompanied with thinning at the top of the cantilever.

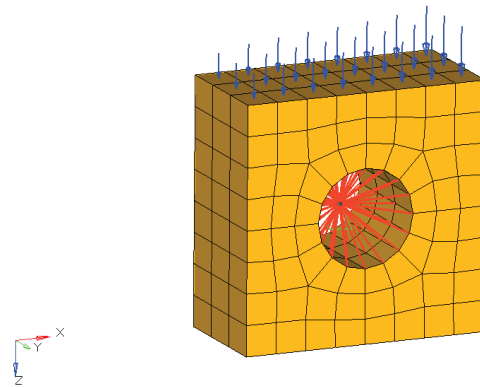


In the figure above, there are modified constraints at the rear of the cantilever. The nodes at the rear part are constraint in the x- and z- direction. In addition, nodes located at the symmetry axis (at the rear) are constraint in the y-direction. The displacements in the y- and x- direction are scaled



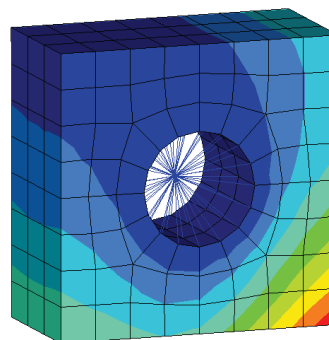
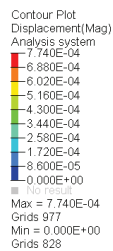
by a factor of 200 and 5, respectively. The undeformed mesh is shown as wireframe (orange). The displacements are quite different compared to the earlier figure. The question to be answered is which model is correct?

Imposing boundary conditions (constraints and forces) by means of RBE2 and RBE3. What are the differences?



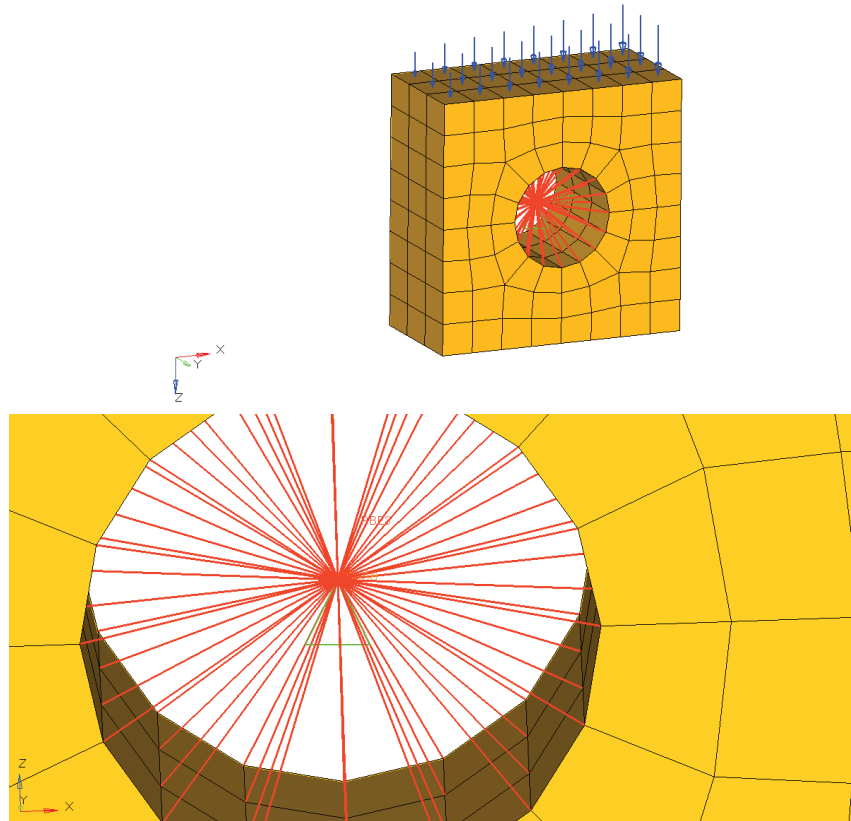
In the figure above, the nodes at the hole are constrained by an RBE2 element where the independent node is fixed with respect to all degrees of freedom (dof123456).

Before looking at the simulation results, ask yourself, what will the displacement contours look like?

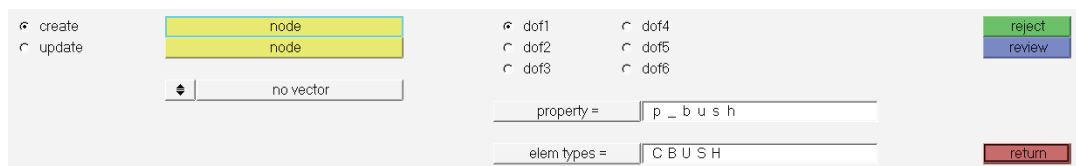


The figure above is a plot of the displacement field. Note that the displacements at the hole are zero. In other words, the RBE2 element “artificially” stiffens the hole.

Next, we are constraining the model by means of a RBE3 element. Note, that the dependent node of the RBE3 element (i.e. the one in the center of the spider) can't be constraint directly. This is because this node would depend on the nodes at the hole and the SPC. A work around is to attach the dependent node to a CBUSH element (with zero length and high stiffness values). The free node of the CBUSH element is then constrained with respect to all degrees of freedom.

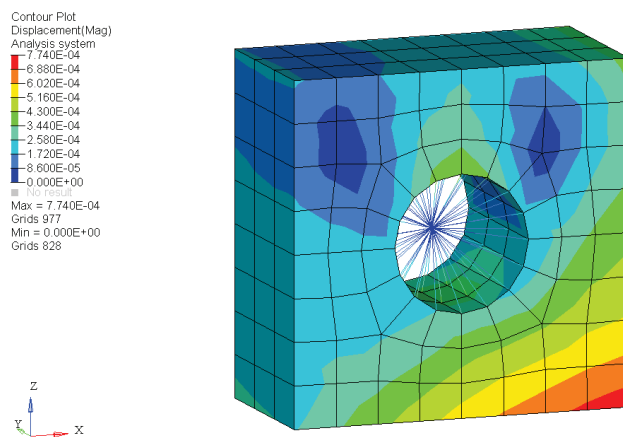
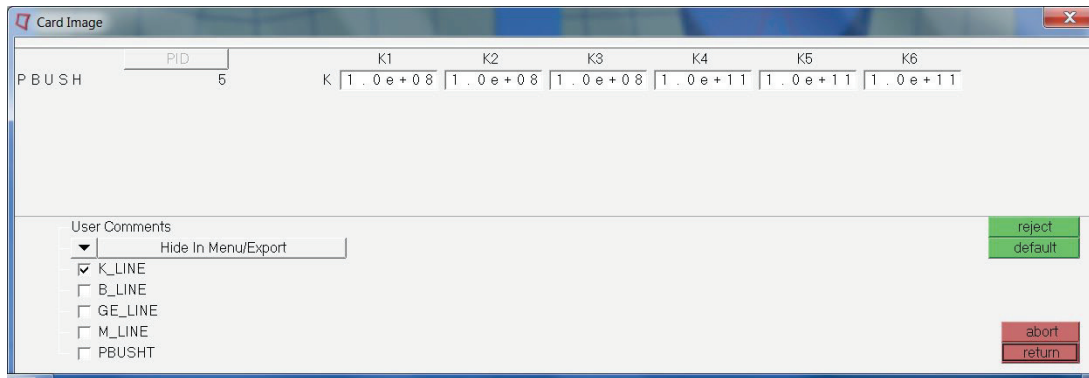


The above figure illustrates how the dependent node of the RBE3 is grounded to an CBUSH element with zero length. All degrees of freedom of the free node of the CBUSH are constrained. CBUSH elements are created in the spring panel which can be accessed by selecting Mesh > Create > 1D Elements > Springs.

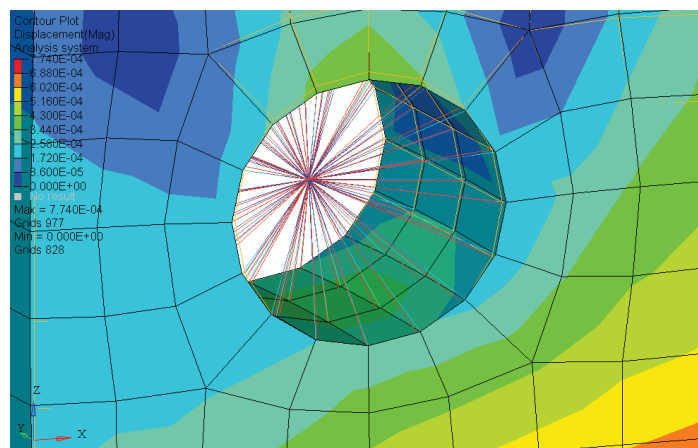


In order to create CBUSH elements, you first need to determine the corresponding “elem types” (default is CELAS), reference a property (can also be assigned later), and select the two end nodes of the spring. The options dof1-6 are irrelevant.

The property definition of the CBUSH element is shown in the following image



The figure above is a displacement contour plot. The hole is “supported” by a RBE3 element, which in turn is grounded to a CBUSE element with high stiffness values. All degrees of freedom of the free node of the CBUSE element are constrained.

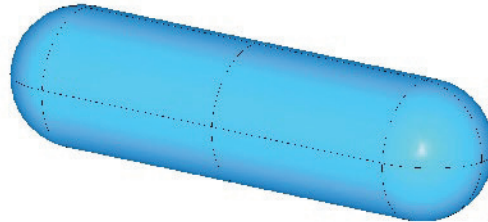


The figure above shows the displacement contour plot scaled by a factor 200. The hole deforms due to the applied pressure. The RBE3 element thus allows us to model a kind of soft support (whereas the RBE2 approach can be considered as a rigid/stiff support).

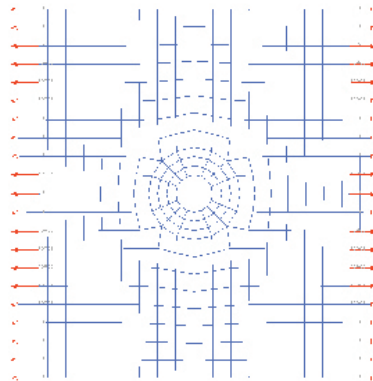


3. Pressure vessel lying freely on the ground and plate subjected to tensile load on both the sides

Sometimes a situation demands for an unconstrained structural analysis like a pressure vessel lying freely on the ground (just placed, no constraint/fixing) or a plate subjected to tensile load on the opposite edges without any constraint. A static analysis problem cannot be solved for an unconstrained structure. It should be fixed at atleast one node, or alternatively, at a few nodes so as to restrict rigid body motion.



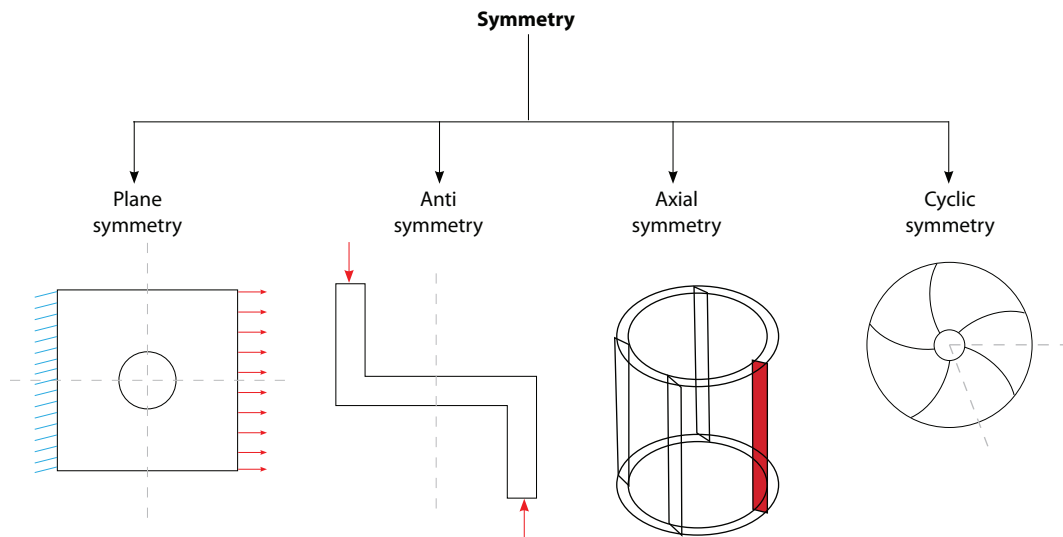
If the problem of the pressure vessel (subjected to internal pressure) or the plate with tensile load is solved without specifying any constraints, then either the solver will quit giving a singularity error message or otherwise report very that there is a high stress at an unrealistic location (if the auto singularity option is switched on).



To solve an unconstrained structural problems there are 2 ways:

- 1) **Approximate approach:** Create spring / beam elements (negligible stiffness value) on the entire circumference (outer edges or surface nodes) and apply the constraint at the free end of spring or beam.
- 2) **Recommended :** Inertia relief method or defining kinematic dofs in the model (see tutorial RD-1030: 3D Inertia Relief Analysis using RADIOSS)

10.3 Symmetry



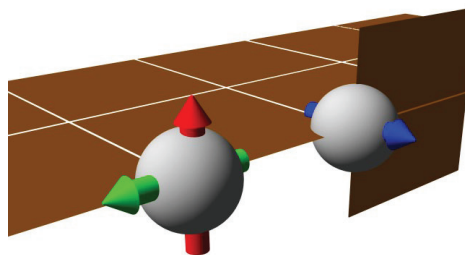
Condition for using any type of symmetry

Symmetric conditions could be used only when both the following conditions are fulfilled.

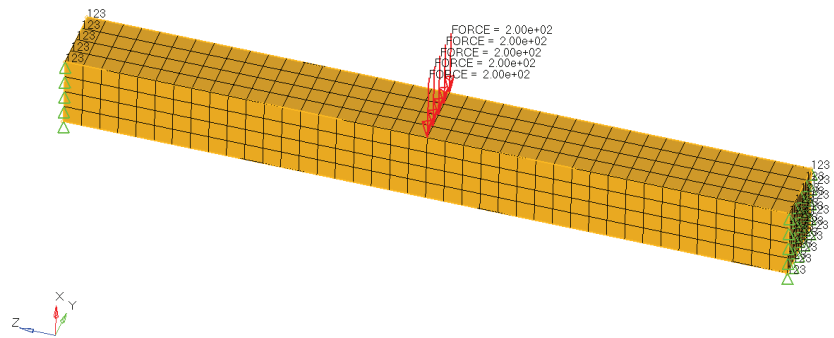
- 1) Geometry is symmetric
- 2) Boundary conditions (forces and constraints) are symmetric.

Advantages: Half, quarter or a portion of the model could be used for analysis, resulting in fewer dofs and computational cost.

Which dof must be constrained at the symmetry level?

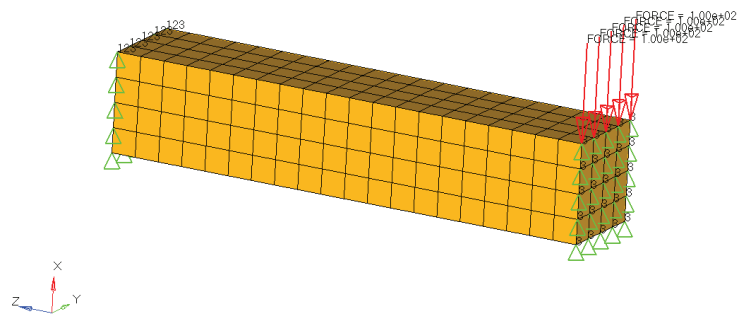


In the figure above, the dark vertically oriented plane represents the plane of symmetry. The finite elements nodes are colored gray, whereas possible nodal rotations are shown by means of blue, green and red arrows. Nodal rotations with respect to the green and red axis/arrows would "move/rotate" the node out of the plane of symmetry (just imagine the arrows would be glued to the nodes). Hence these degree of freedoms (dof) must be constrained. In contrast, nodal rotations with respect to the blue axis/arrow are not needed to be constrained. As the nodes of solid elements do only allow translational displacements, one just need to constrain any out of symmetry plane motions.

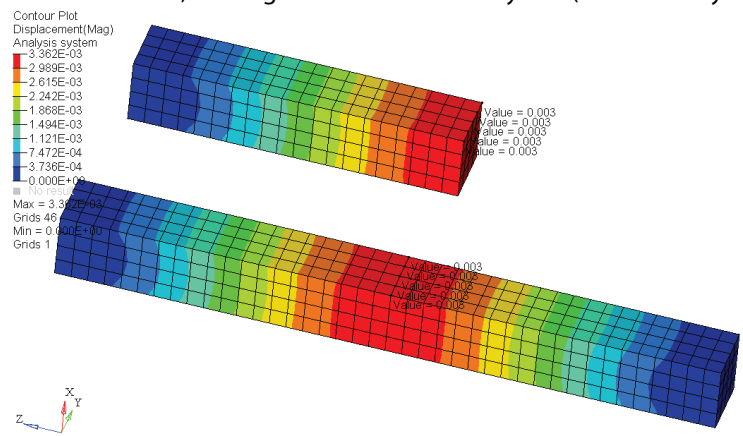


The figure above is considered the full model. The beam ends are constrained with respect to any translational displacements (dof 123). A vertical load of 200 N is applied at its center.

If the symmetry plane is in the x-y plane then the translational displacements in its normal direction i.e. z-direction (dof 3) need to be constrained. On the other hand, we don't need to fix/delete rotations with respect to z-axis as solid elements do not allow nodal rotations. Remember, the nodes at the symmetry plane are not allowed to move (or rotate) out of the plane of symmetry.

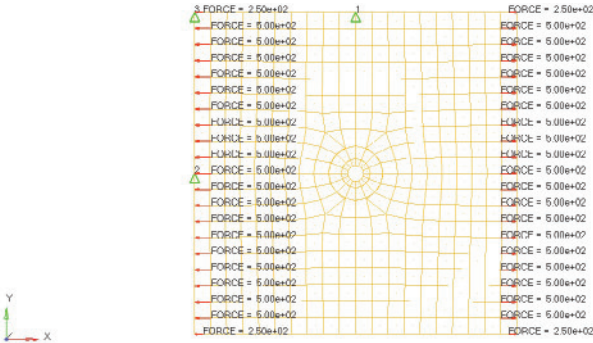


The figure above is considered the half model. At the plane of symmetry, the z-displacements (dof 3) are constrained. In addition, the original force is divided by two (as it acts only on half of the structure).

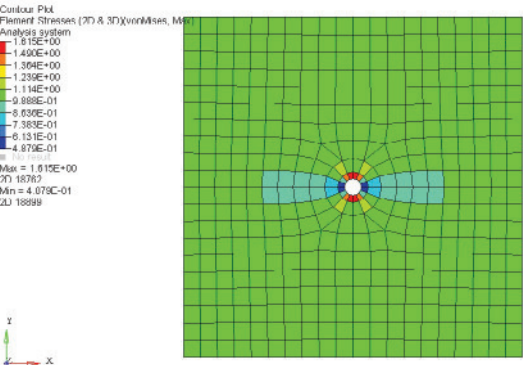




Let us consider a symmetrical plate with a hole subjected to symmetrical loads on the two opposing edges.

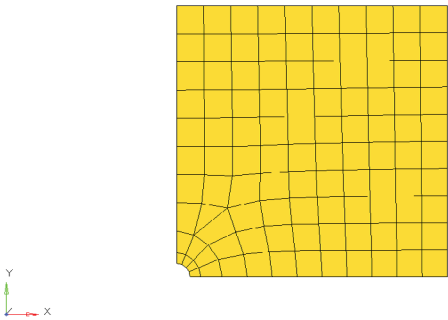


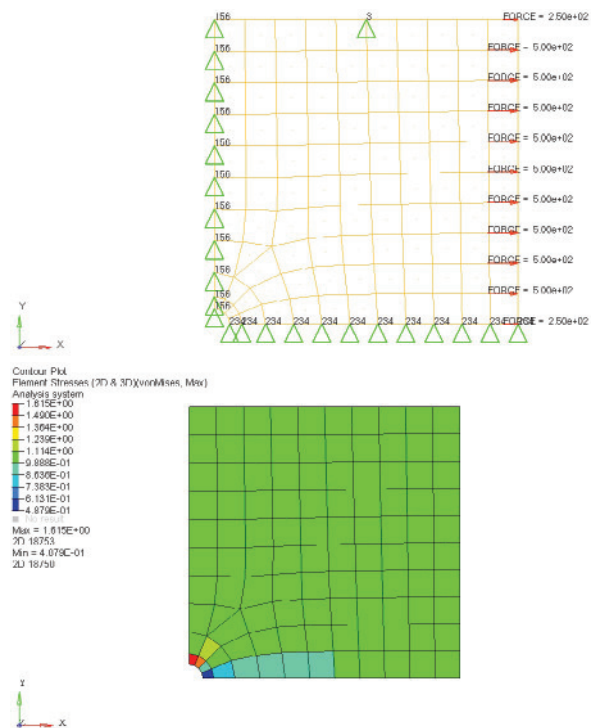
The figure above is the full plate model and serves as a reference model



The figure above is a contour plot of the element stresses (von Mises).

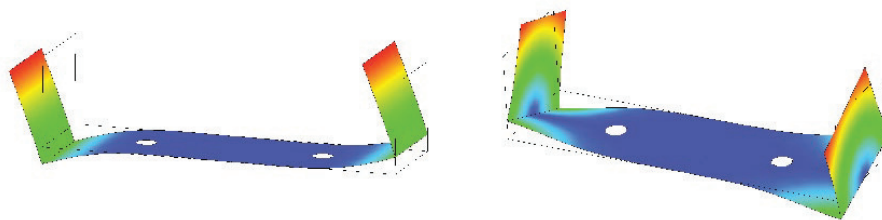
In the next step, a quarter segment of the plate is investigated. The corresponding loads and constraints are depicted in the figure below.



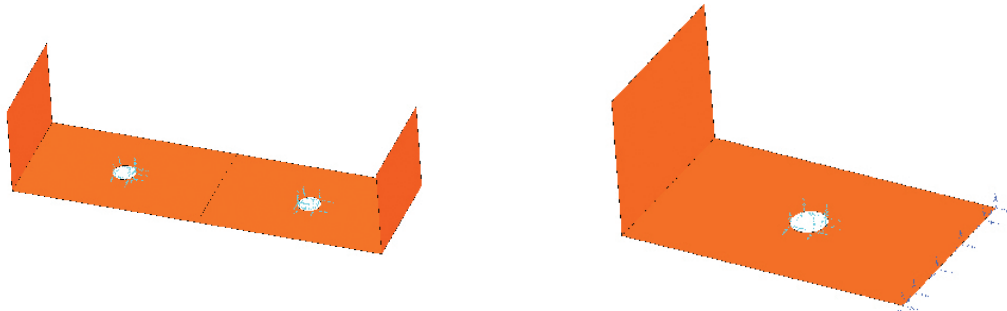


The figure above is a contour plot of the element stresses (von Mises).

Limitations: Symmetric boundary conditions should not be used for dynamic analysis (natural frequency and modal superposition solver). A symmetric model (half or portion of part) would miss some of the modes (anti nodes or out of phase modes) as shown below:



Natural Frequency comparison for Full and Half symmetric model



Mode No.	Full model Hz	Half symmetric model Hz
1	448	448
2	449	718
3	718	1206
4	719	2078
5	1206	2672
6	1211	4397

Question : We need to do a simulation of a casting component to decide the best option out of two casting materials. What will be the differentiating parameters, since the modulus of elasticity and poisson's ratio for both of the materials are the same. Secondly, we do not have stress strain data for both of the materials.

1) If the nature of the loading is static (most of the time static and rarely dynamic) and the company has only a linear static solver : Different grades of cast iron have different ultimate strength (or proportional yield strength). Linear static stress is independent of material and both the materials would report the same stress. The decision could be made based on the ultimate strength (and endurance strength). Say the maximum stress as per the FEA is 300 N/mm². For material 1, the ultimate stress is 350 and for material 2 it is 500 N/mm². In this situation, the second material is clearly the choice. Consider another case. Suppose the maximum stress reported by the FEA is 90. One can calculate the approximate value of endurance strength. For grey cast iron, the endurance strength = 0.3 * ultimate strength. Therefore, for material 1 the endurance strength is 105 N/mm² and should be preferred over the second one.

2) If the nature of the loading is dynamic and subjected to a sever load : Fatigue analysis is strongly recommended. Commercial fatigue analysis software usually provide material library. Specifying the appropriate material grade, surface finish, etc. and calculations for the life / endurance factor of safety would lead to realistic and optimum selection out of the two options.

10.4 Creating Loadsteps in HyperMesh

And finally, once the constraints and loads are specified, a corresponding loadstep needs to be created (otherwise the Finite Element program would not know what to do with these entities). In HyperMesh, this is done by selecting **Setup > Create > LoadSteps**.



The image above shows the Load Step panel in HyperMesh. This panel is used to define a loadstep. First, specify the name and the type of the loadstep. The load collector with the constraints are referenced using SPC and the forces/pressure are referenced using LOAD.

The loadstep will then be listed in the **Model Browser** along with all the other model entities.

Now the model can be exported and the analysis can be started!

10.5 Tutorials and Interactive Videos

To view the following videos and tutorials, you first need to register at the HyperWorks Client Center using your university E-Mail address. Once you have a password, log into the Client Center and then access the videos and tutorials using the links below.

Recommended Tutorials:

These tutorials can be accessed within the installation inside the Help Document.:

From the menu bar select Help > HyperWorks Desktop > Tutorials > HyperWorks Tutorials > HyperWorks Desktop > HyperMesh > Analysis Setup.

Tutorials can also be accessed within the Online Help which can be accessed using the links below. You will need to log into the HyperWorks Client Center to access these tutorials online.

- [HM-4000: Setting up Loading Conditions](#)
- [HM-4040: Working with Loads on Geometry](#)

Recommended Videos

Product Videos (10-15 minute; no HyperWorks installation required)

- [How to apply loads and constraints?](#)
- [How to define a load step?](#)

Recommended Tips and Tricks

- [Boundary Conditions – Loads Summary Tool](#)
- [Beam - Pressure Loading](#)

Presentation

- [Suspension and Chassis Loads](#)