A beginner's tutorial for FE modelling of tubular joints using MSC.Patran

by

Liang Juxiang

September 2003

Introduction

When I tried to create my first Finite Element model for a tubular joint, I did not know how to get started, even after I had read through MSC.Patran user's manual. There are no examples that a beginner can follow.

This tutorial aims to lead beginners into the world of Patran by creating an FE model from scratch. Simple as it is, the tutorial covers almost all essential steps to build FE models for tubular joints and provides useful information that I obtained through three years of practice. Those who want to learn Patran may find it helpful.

Liang Juxiang
Department of Civil Engineering
National University of Singapore

Problem description

Create a solid Finite Element model for a tubular X-joint subjected to brace axial compression. The main dimensions of the joint are:

Outer diameter of the chord $d_0 = 508$ mm
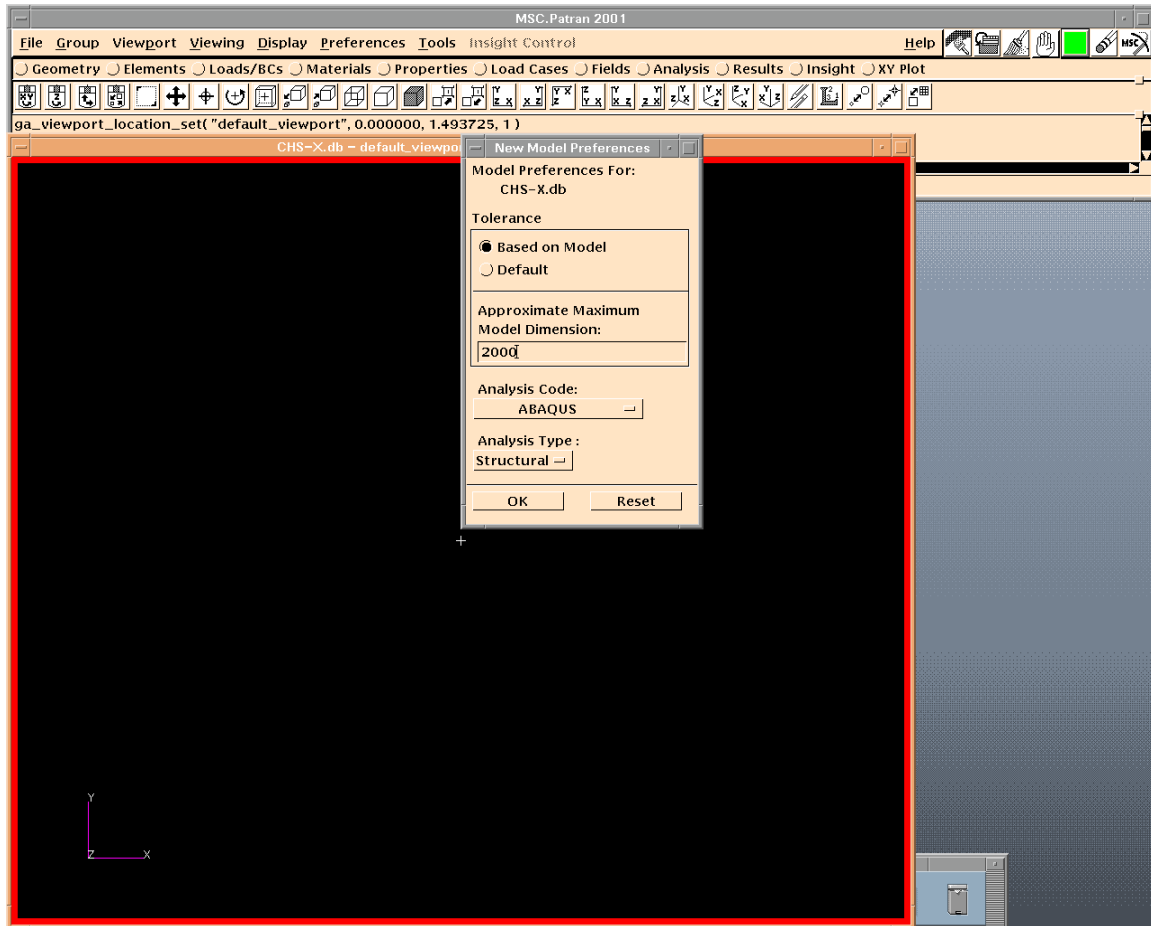
Outer diameter of the brace $d_1 = 324$ mm

Length of the chord $l_0 = 3048$ mm

Length of the brace $l_1 = 1296$ mm

Thickness of the chord wall $t_0 = 10$ mm

Thickness of the brace wall $t_1 = 10$ mm

Due to symmetry in geometry and loading, only one eighth of the overall joint is modelled. The analysis is supposed to be carried out by ABAQUS.

Create a new model

- ➢ Start MSC.Patran.
- ➢ Create a new file named CHS-X.db.
- ➢ Input Approximate Maximum Model Dimension.
- ➢ Choose ABAQUS for Analysis Code, and click OK.

Tips

- ➢ The Default value of Tolerance is 0.005. The Approximate Maximum Model Dimension affects the global tolerance, if Based on Model is selected.
- ➢ To change Tolerance and Analysis Code during modelling, go to Preferences.
- ➢ Point at the icons to find out the function of each icon.
- ➢ Units do not affect the analysis results, as long as all units are kept consistent throughout the modelling.
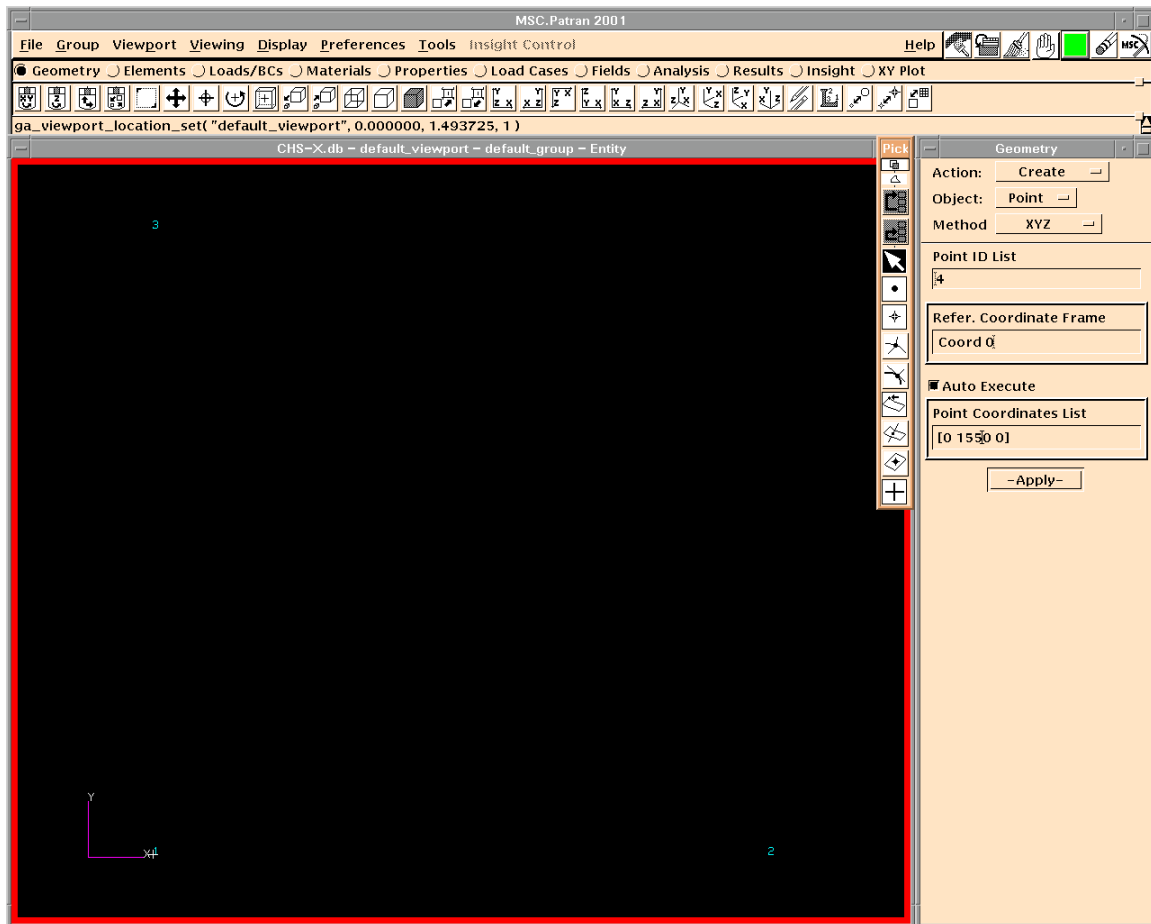
Create points

➢ Input Point Coordinates List.

➢ Create Points 1, 2 and 3 at the intersection and ends of the chord and brace axes.

➢ To display the point labels, go to Display ⟹ Geometry…

Tips

➢ You can create several points at a time.

➢ From points to curves, to surfaces and to solids to create the geometries, before you can mesh on the geometries.

➢ It is convenient to locate the intersection of brace and chord axes at the origin of the global coordinate system. Global coordinate frame is identified as Coord. 0.

➢ You can cancel Auto Excute, if you do not want Patran to create the point without clicking Apply.
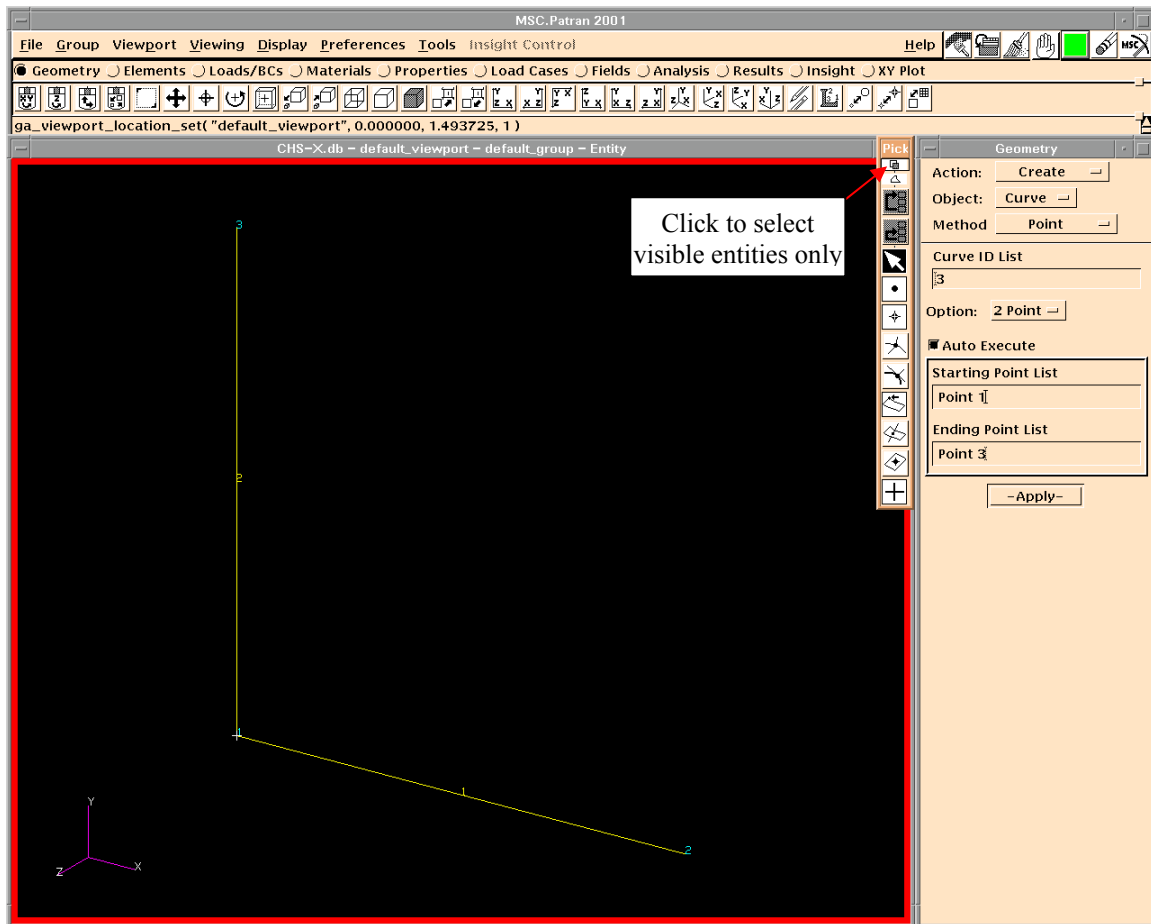
Create curves

> ➢ Select starting and ending points.
>
> ➢ Create Curves 1 and 2 as the chord and brace axes.

Tips

> ➢ The sequence of the two points determines the direction of the curve.
>
> ➢ The file is automatically saved.
>
> ➢ Patran can only undo for once.
>
> ➢ The icons beside the form help to select specific type of entities.
>
> ➢ Sometimes it is useful to click the icon indicated above to select visible entities only.
>
> ➢ To find online Patran manual, go to Help ⟹ Document Library…
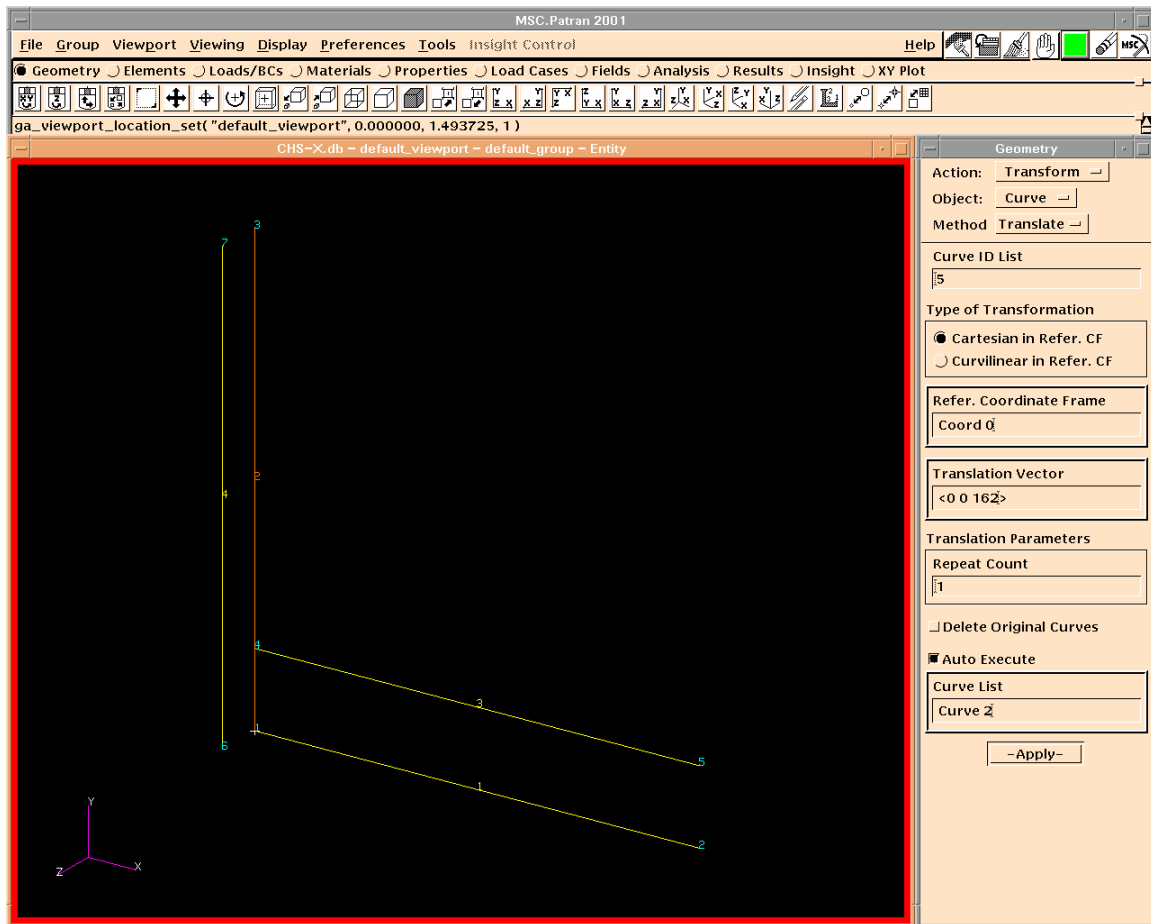
Translate curves

- ➢ Input Translation Vector.
- ➢ Translate Curves 1 and 2 with distances of the chord and brace outer radii to create Curves 3 and 4.

Tips

- ➢ For Circular Hollow Section joints, d_0 and d_1 often refer to outer diameters of the chord and brace.
- ➢ You can find a journal file (CHS-X.db.jou) in the working directory, which records each command executed and can be used to rebuild a model.
- ➢ Create a local coordinate frame, and choose it as Refer. Coordinate Frame, if you want to translate curves in directions other than the global axes.
- ➢ You do not have to delete the original curves, as you may use them later.

5
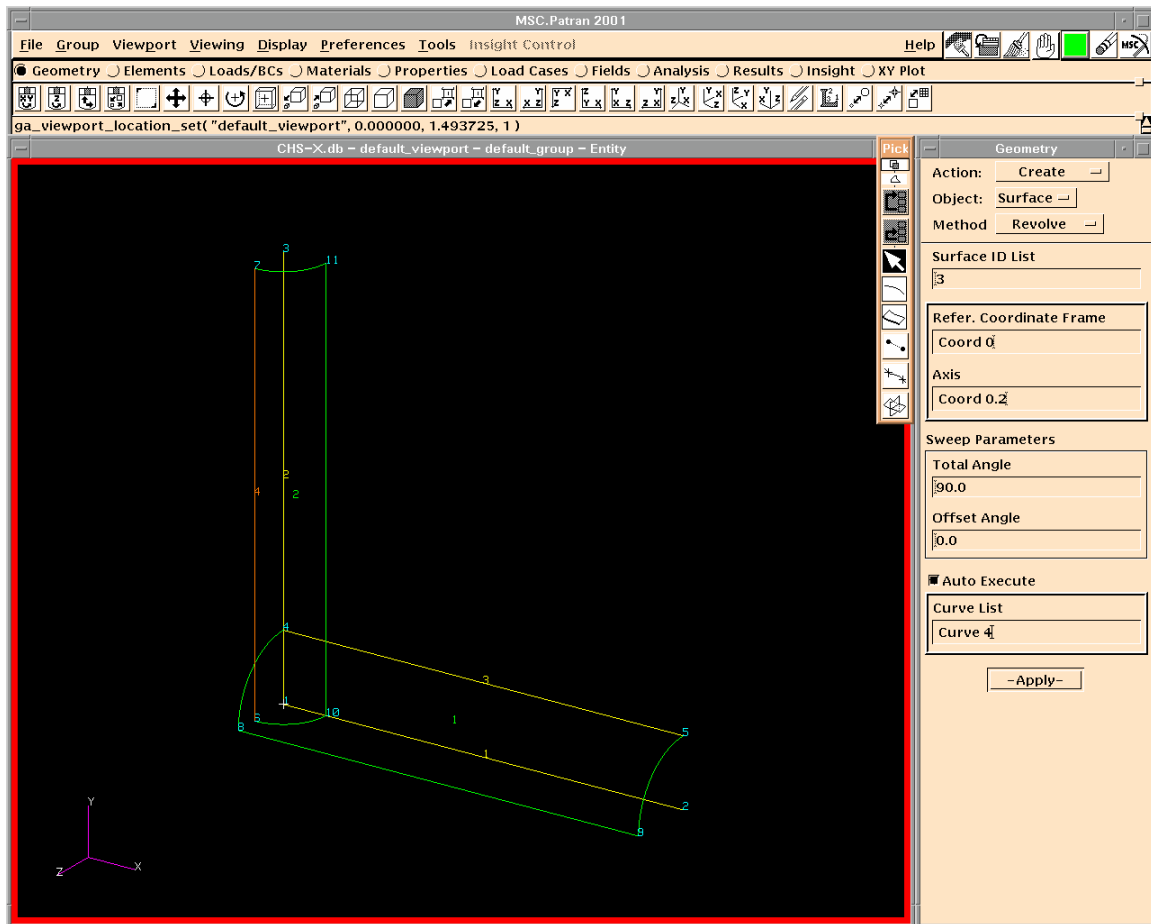
Create surfaces

➢ Choose the appropriate axis for revolving.

➢ Revolve Curves 3 and 4 for 90 degrees to create Surfaces 1 and 2 as the outer surfaces of the chord and brace.

Tips

➢ Coord 0.2 is the global Y axis.

➢ A negative value of Total Angle revolves the curve anticlockwise.

➢ Both surfaces created here have a normal towards the axis.

➢ Create local coordinate frames to revolve around axes other than the global ones.

➢ There are many ways to create a surface. Trying different ways to do the same thing will help you find the best way.
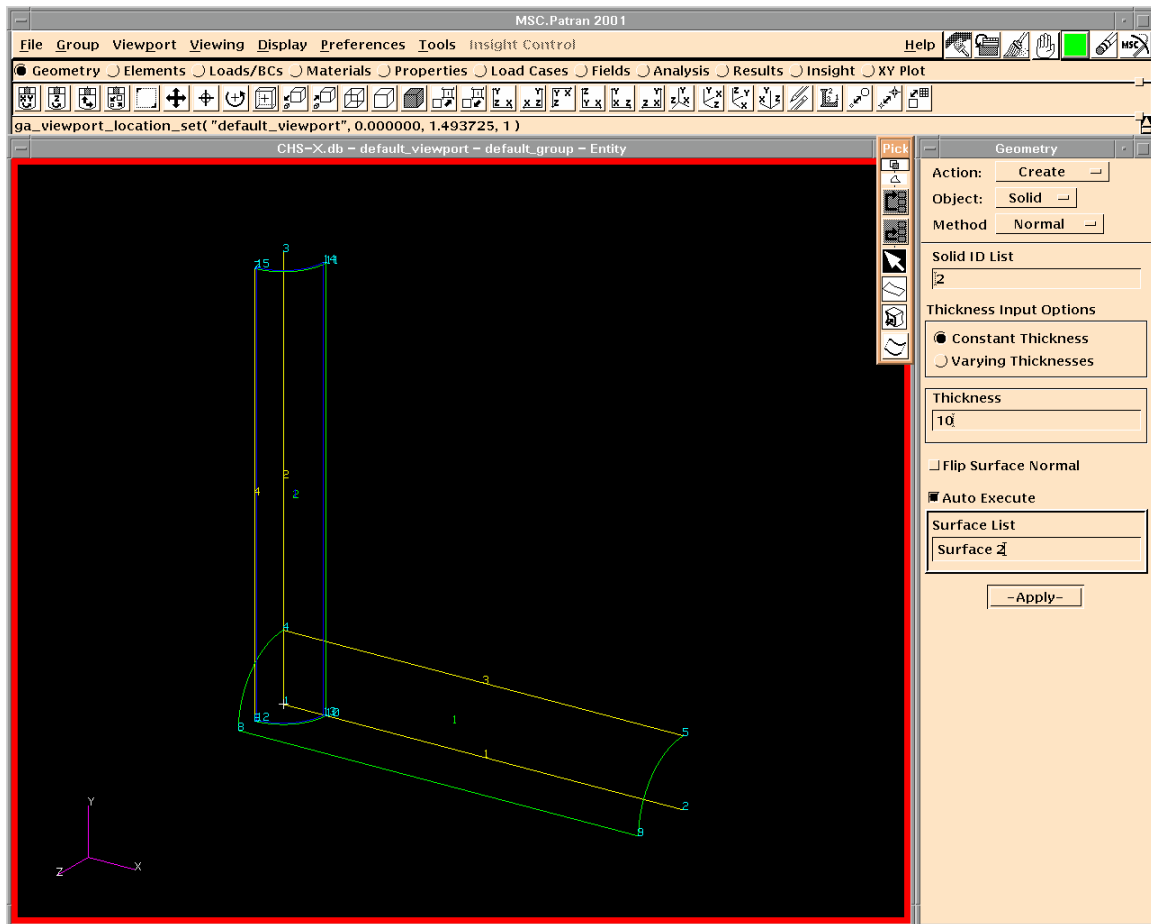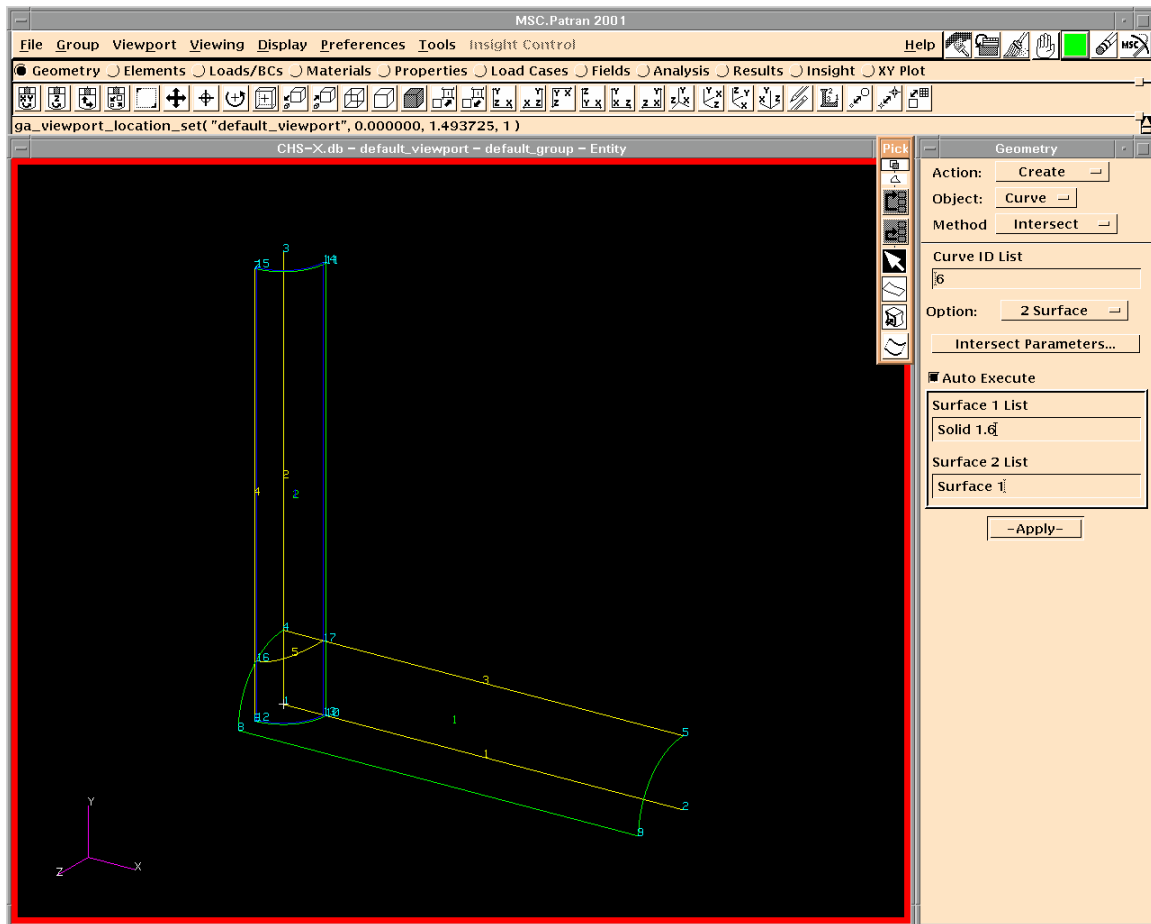
Create a solid

- ➢ Input Thickness.
- ➢ Select Surface 2 to create Solid 1 for the brace member.

Tips

- ➢ A negative value of the thickness sweeps the surface in the negative direction of the surface normal. Flip Surface Normal has the same effect.
- ➢ You may give a specific number to the solid ID before the solid is created. It helps to identify the solid when there are many.

Create a curve at the intersection

> ➢ Select the inner face of Solid 1 and Surface 1.

> ➢ Create Curve 5 at the intersection between the inner surface of the brace and the outer surface of the chord.

Tips

> ➢ Choose the icon of Solid Face to select the inner face of Solid 1.

> ➢ A solid face is identified as Solid A.B, where A is the solid ID and B is the face number. A solid face is equivalent to a surface.
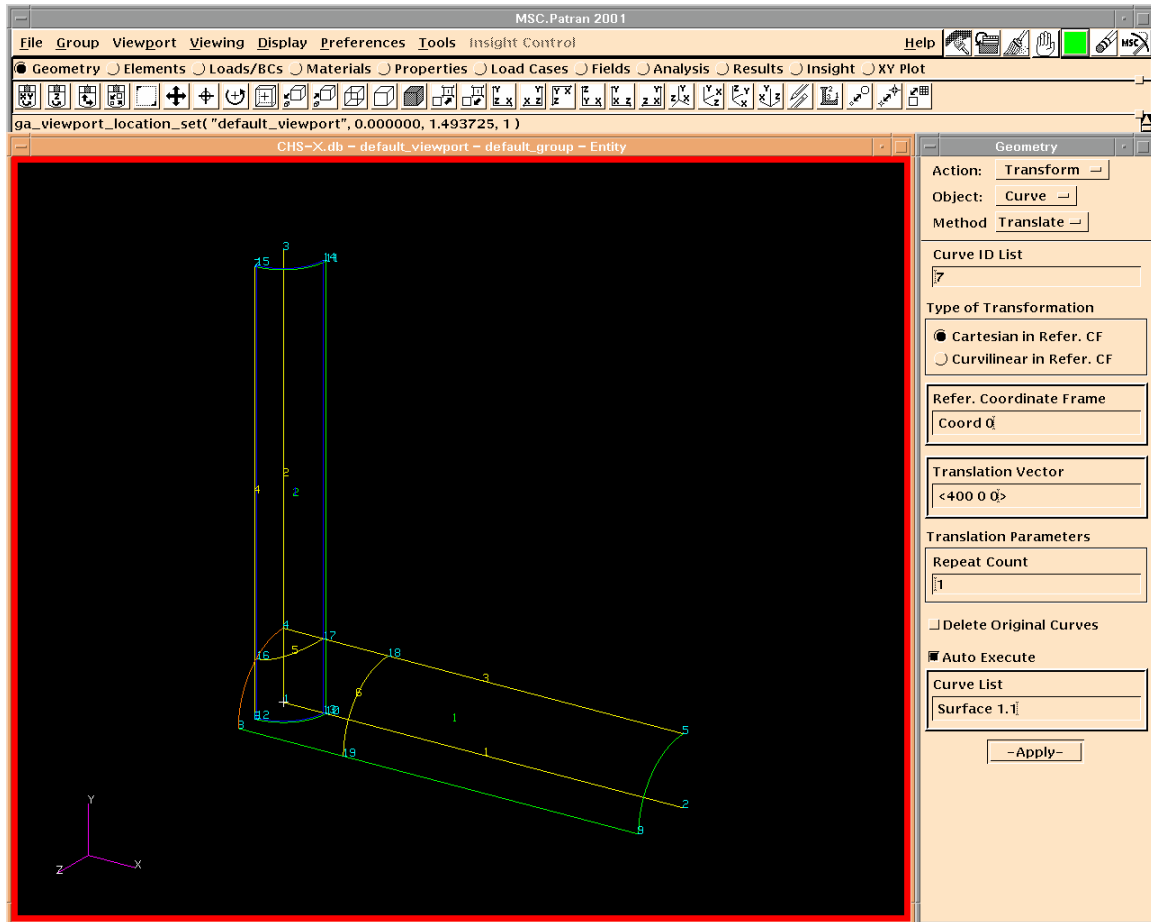
Translate an edge of a surface

- ➤ Input Translation Vector.
- ➤ Translate an edge of Surface 1 along axis X to create Curve 6.

Tips

- ➤ A surface edge is identified as Surface A.B, where A is the surface ID and B is the edge number. A surface edge is equivalent to a curve. A solid edge is identified as Solid A.B.C.
- ➤ Curve 6 will be used to break surface 1, so that different mesh strategies can be applied for the chord.
- ➤ Allow enough distance for the transition of different mesh strategies.
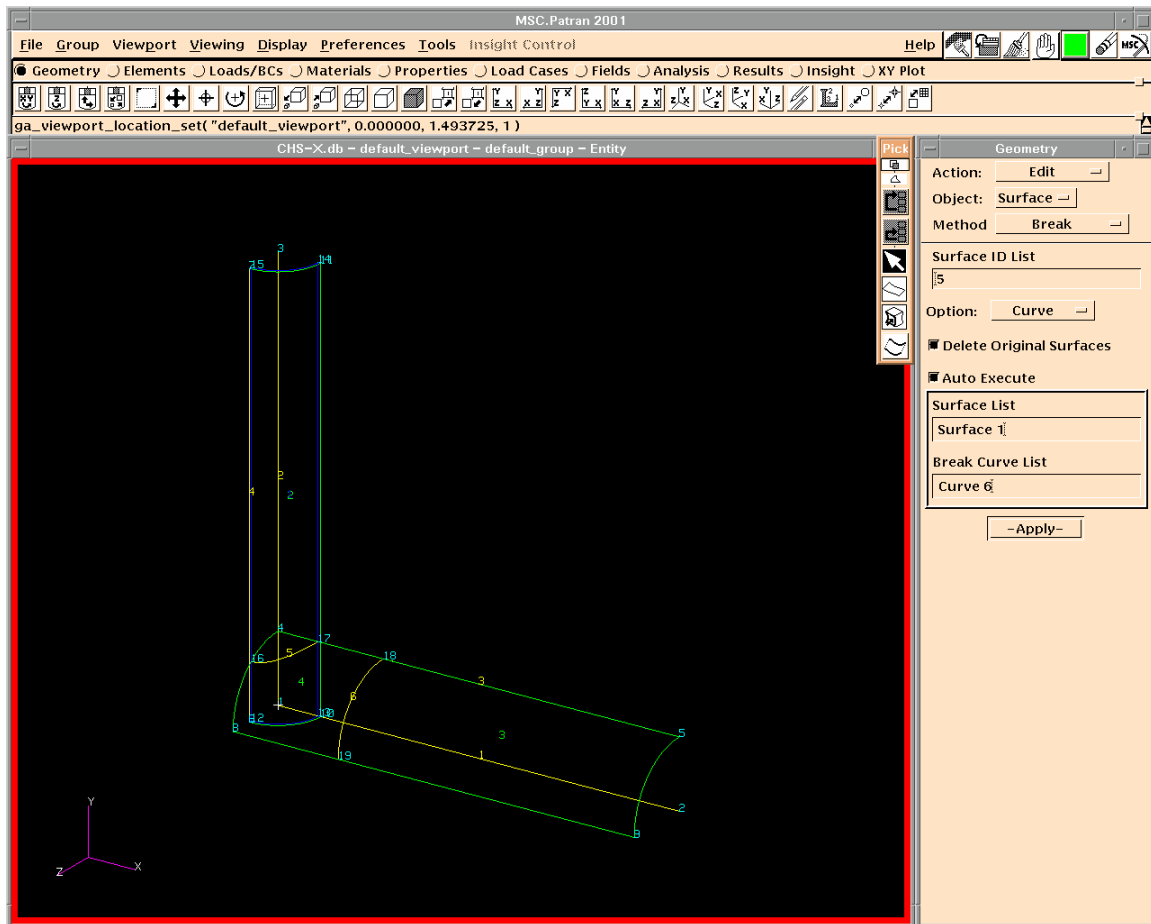
Break a surface using a curve

➢ Select Curve 6 to break Surface 1 into Surfaces 3 and 4.

Tips

➢ Surfaces can be broken by a curve, another surface, a plane or...

➢ Surfaces are often broken at the intersections.

➢ Surfaces are also often broken for different mesh strategies to be used.

➢ If sometimes you can not break a surface using a curve, try other methods, e.g. using a plane to break a surface.

➢ Do not delete the original surface, you might find later that you have to restart from this point.
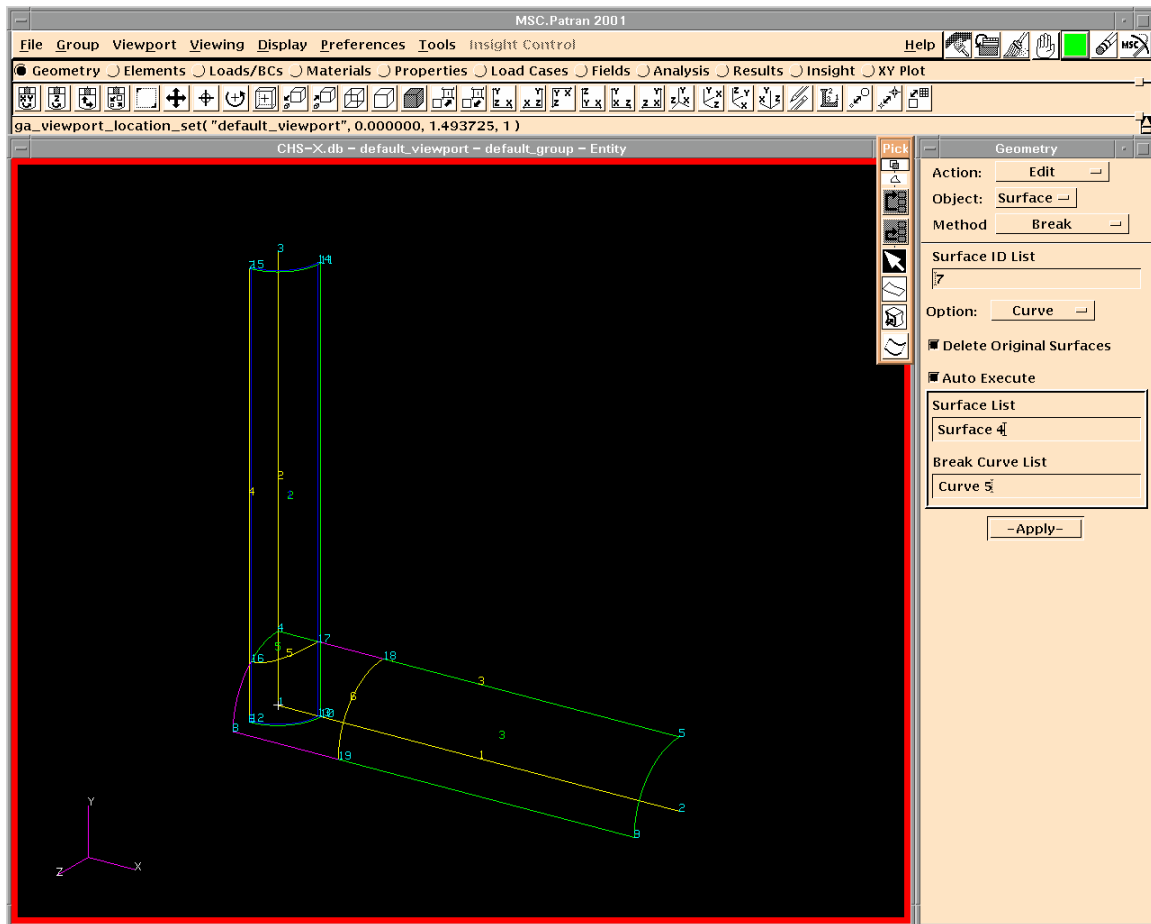
Break another surface

> ➢ Break Surface 4 using curve 5.

Tips

> ➢ Surfaces with three or four edges are indicated in green, while other surfaces are indicated in purple.
> ➢ The ID of the surface with purple edges is not shown.
> ➢ Surfaces with purple edges can not be meshed properly using IsoMesh.
> ➢ If you can not break a surface using another surface, try to create a curve at the intersection and break the surface using the curve.
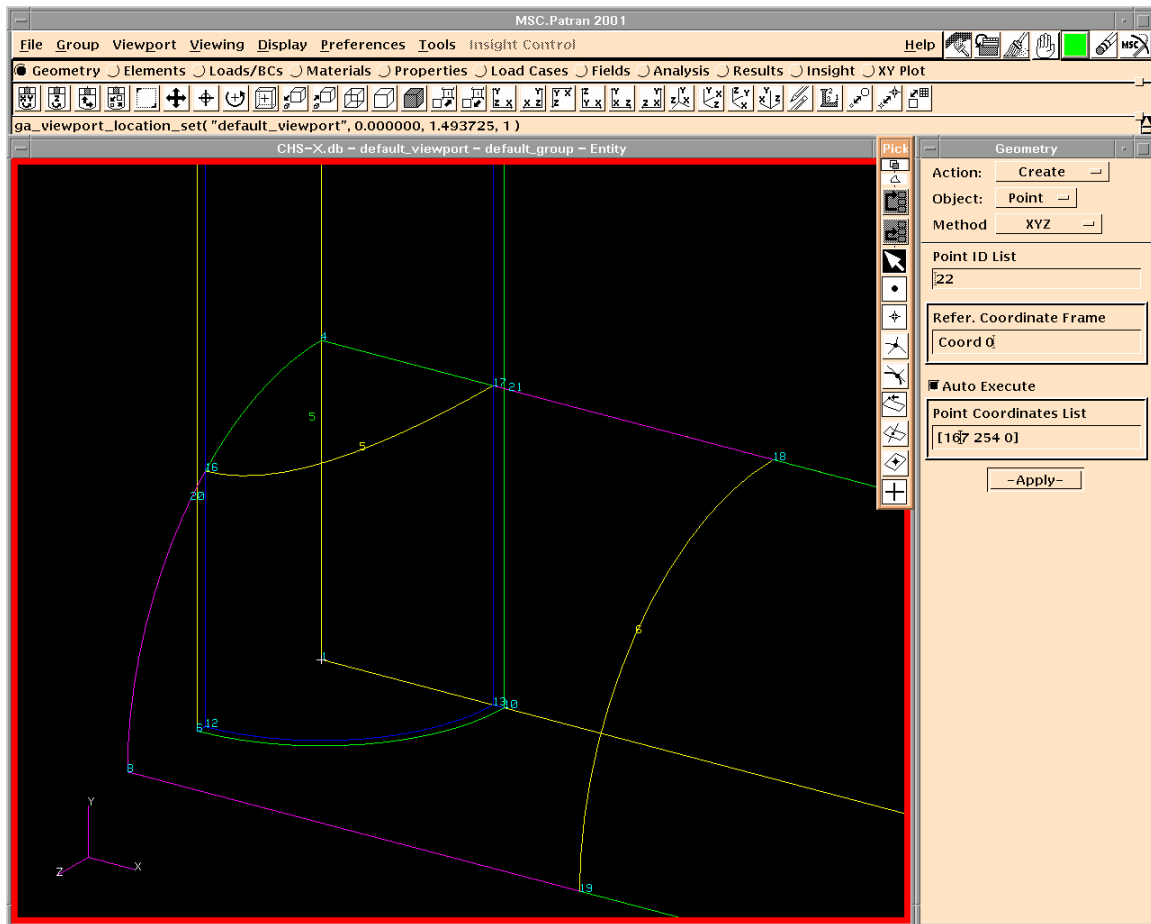
Begin to create the weld geometry

➢ Create two points at the outer edge of the weld footprint. The distance between Points 17 and 21 is about 1.5 times of the thickness of the brace $(1.5t_1)$ and the curved length between Points 16 and 20 is about $1.75t_1$.

Tips

➢ Weld geometries are usually the most difficult part of FE modelling for tubular joints.

➢ The geometries of welds can follow "Structural Welding Code, AWS D1.1-98. American Welding Society" (available in NUS Central Library).

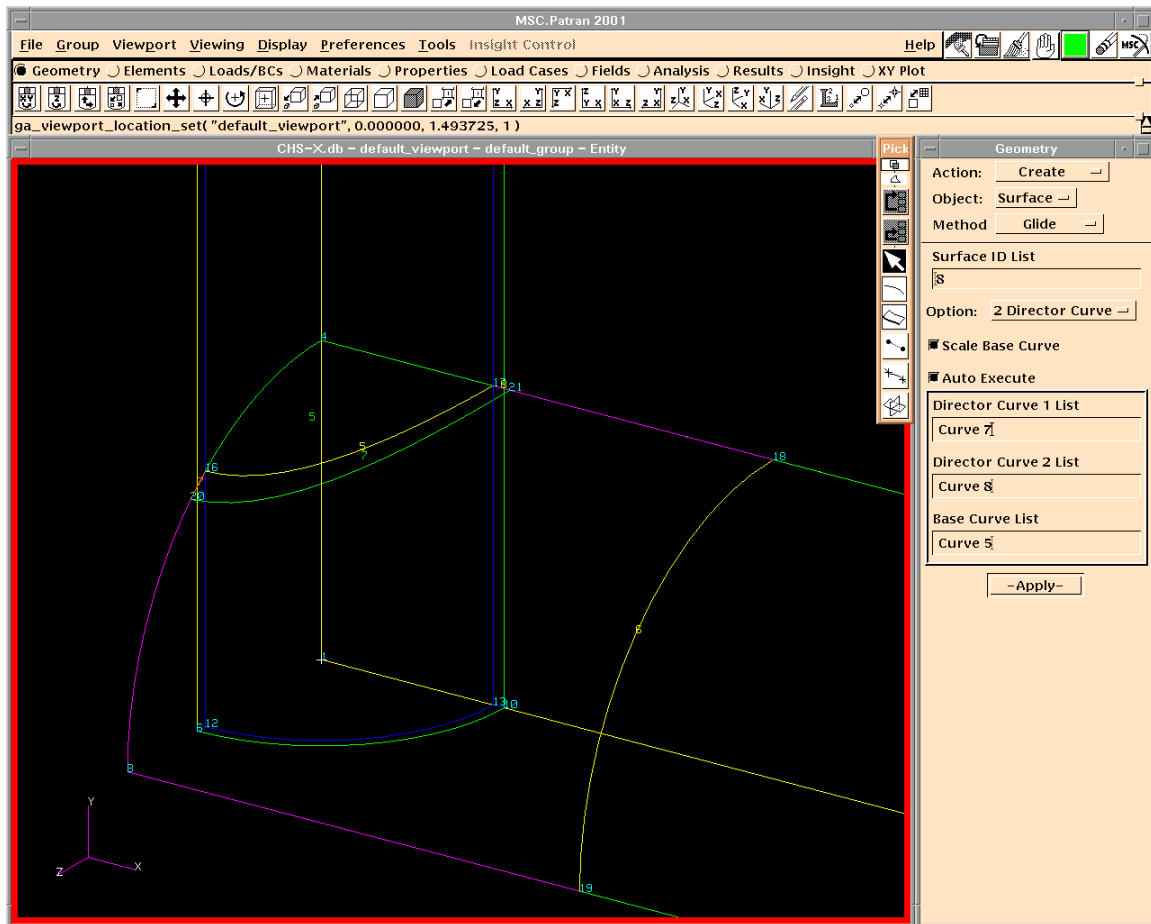➢ You may need more than two points to model the weld footprint.
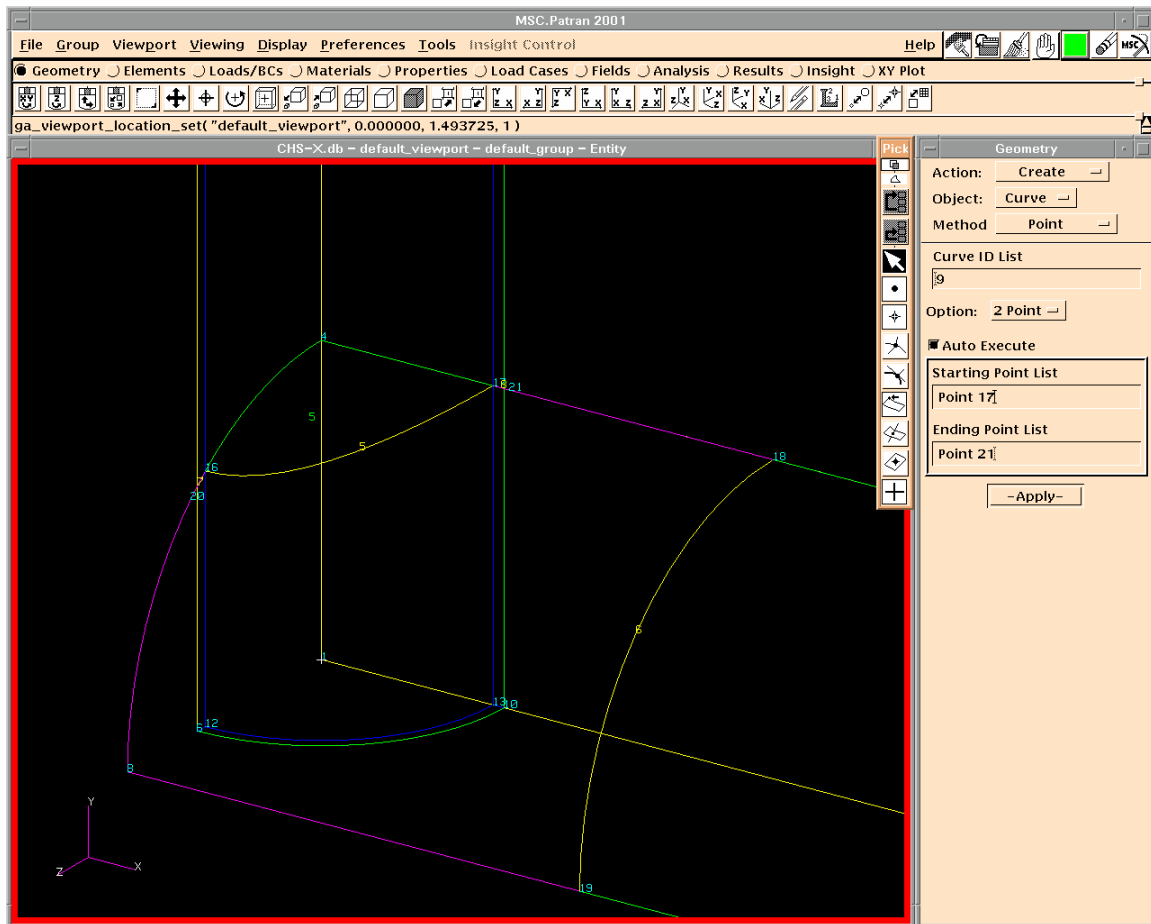
Create curves for the weld footprint

➢ Create curves 7 and 8 between Points 16 and 20, and Points 17 and 21.

Tips

➢ Although curve 7 is not straight and does not exactly fall on the chord surface, it is within the tolerance.

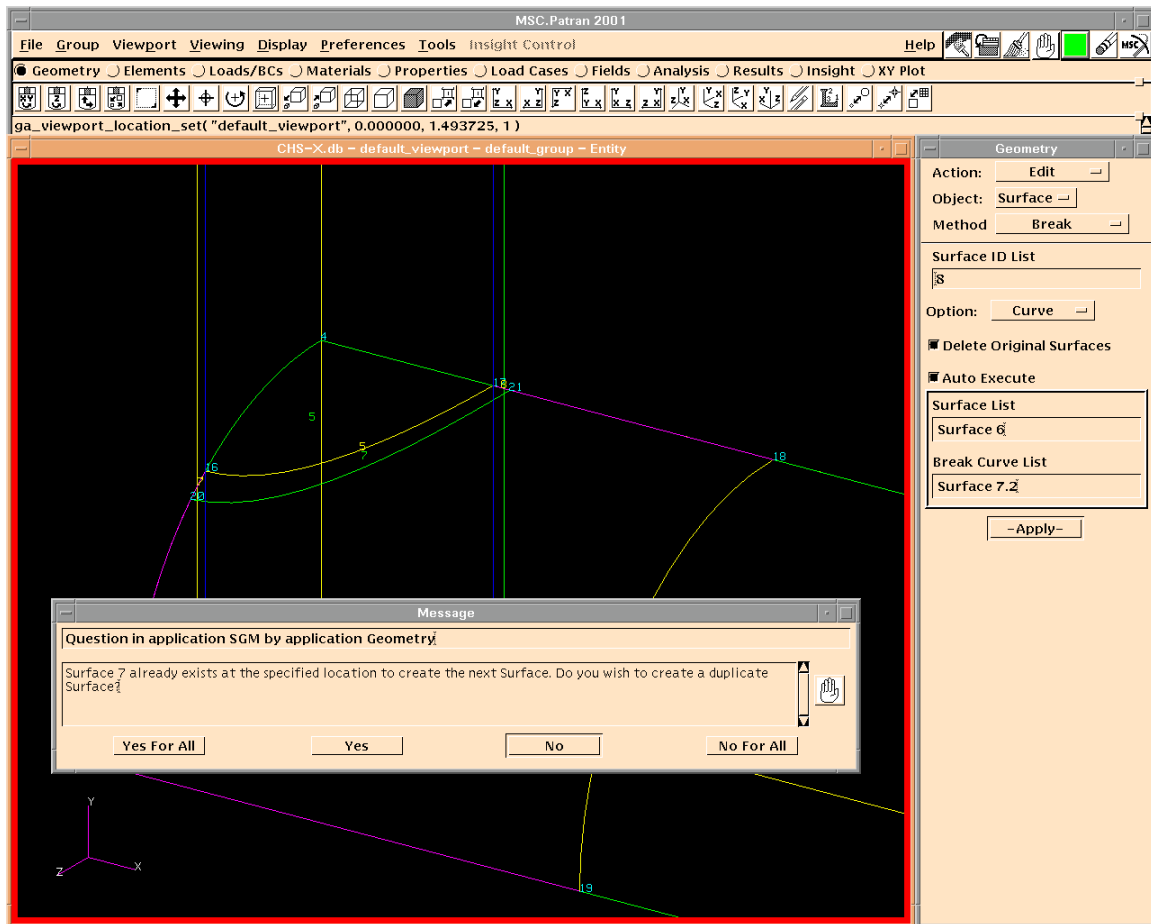➢ Find your own way to create the welds for other types of joints.

Create a surface using Glide

➢ Choose 2 Director Curve.

➢ Select Curves 7 and 8 as director curves and Curve 5 as base curve.

➢ Create Surface 7.

Tips

➢ Surface 7 falls on surface 6 (the one with purple edges) within tolerance.

➢ Building a new surface is often easier than breaking an existing one.

Break surface using an edge of another surface.

> ➢ Break Surface 6 using Surface 7.2.

> ➢ If you see a pop-up message as shown above, choose No to avoid duplicate surfaces.

Tips

> ➢ The pop-up message suggests that Surface 7 is on Surface 6 within tolerance.

> ➢ The sequence in which you break surfaces may affect the properties of the surface. In this case, if you break the surface at the outer edge of the weld footprint first, and then at the inner edge, you will find the surface of the weld footprint indicated in purple, although it has four edges.
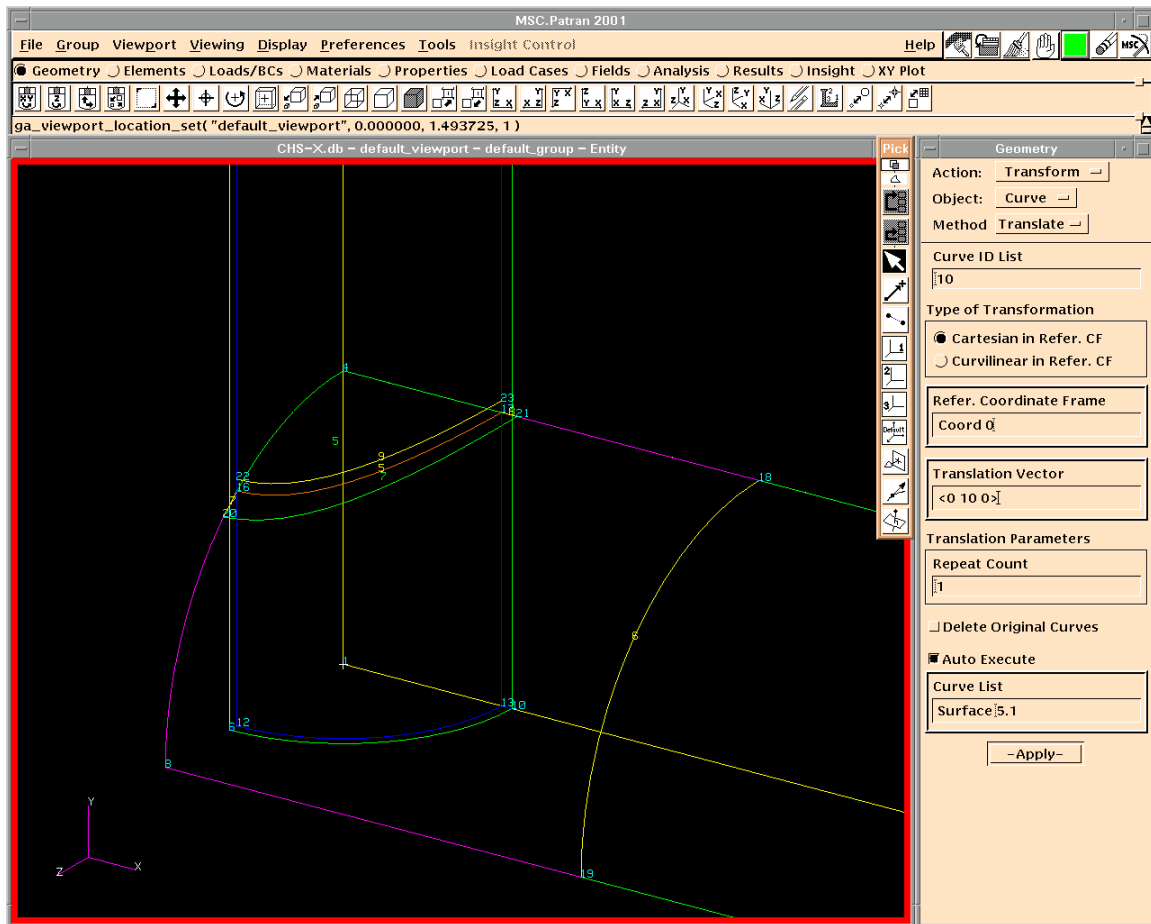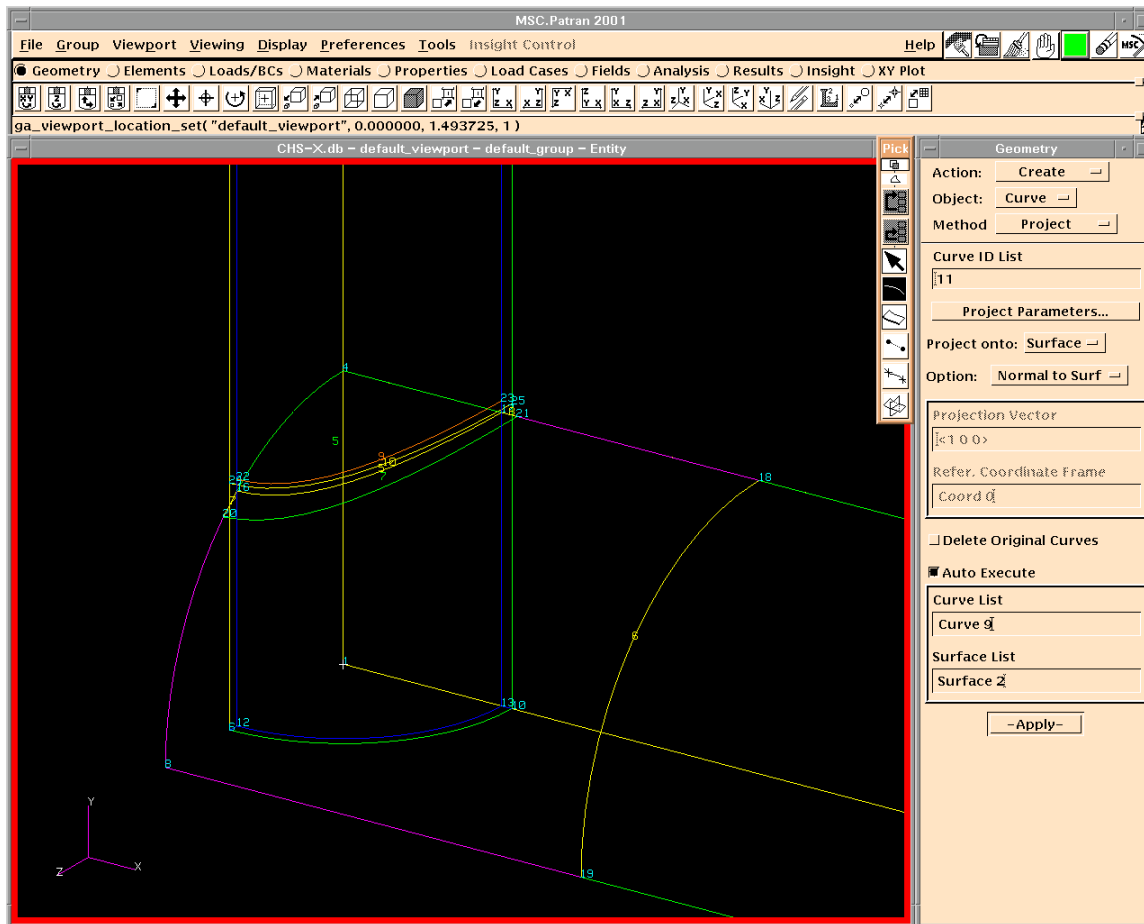
Translate a curve

➢ Input Translation Vector.

➢ Translate Curve 5 upwards to create Curve 9.

Tips

➢ An entity may have different IDs if it is associated with more than one entity.

➢ A command can be interrupted if the icon at the upper right corner is blue.

➢ When the icon turns red and keeps on beating, Patran ignores any of your interruptions.
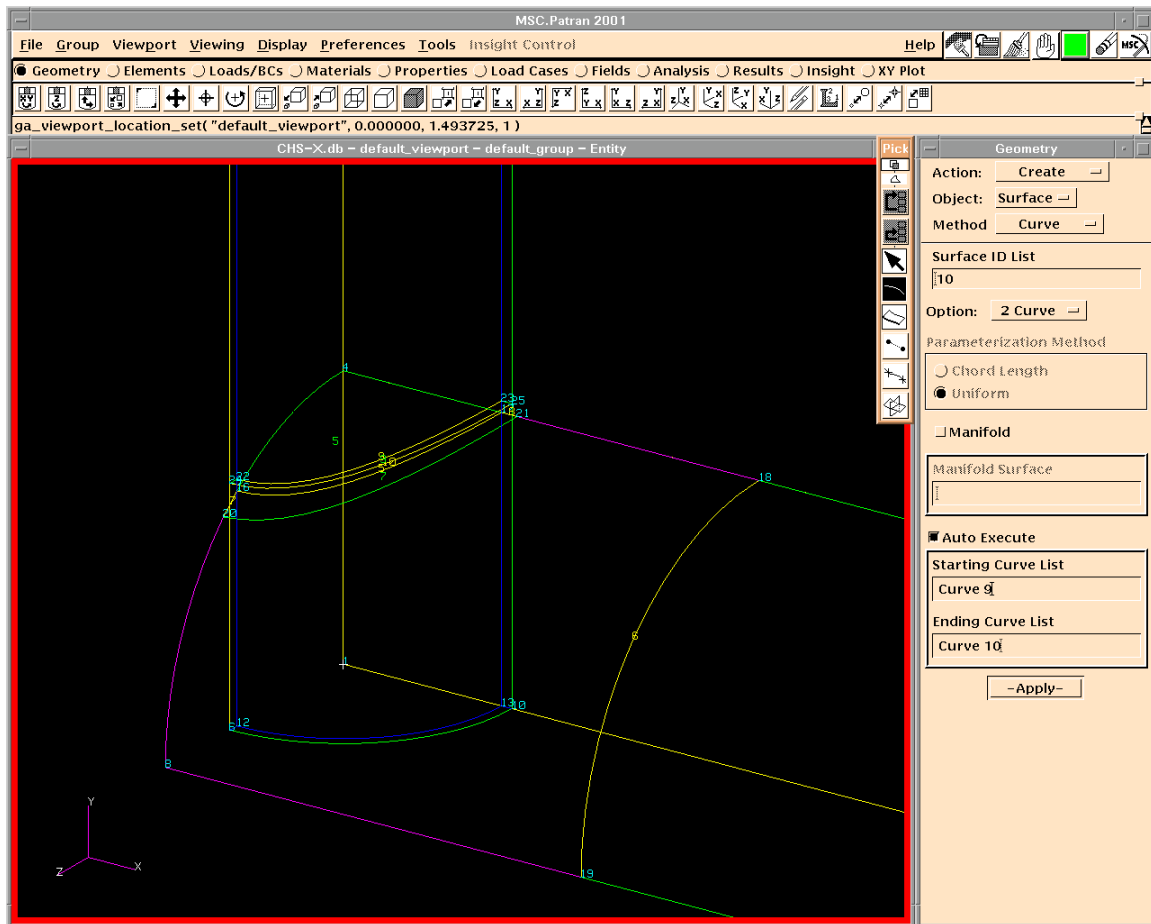
Project a curve

➢ Select Curve 9.

➢ Project Curve 9 on Surface 2 to create Curve 10.

Tips

➢ Using projecting is to make sure that equivalence of the FE model can be achieved at this position.

➢ You can click the icon at the upper right corner to Reset Graphics at anytime.

Create a surface using two curves

- ➢ Select Curves 9 and 10.
- ➢ Create Surface 9.

Tips

- ➢ You may like to save the file now, as some steps will not be saved if Patran exits unexpectedly (this can occur due to several reasons).
- ➢ Saving copies at different stages during modelling is a good idea for beginners.
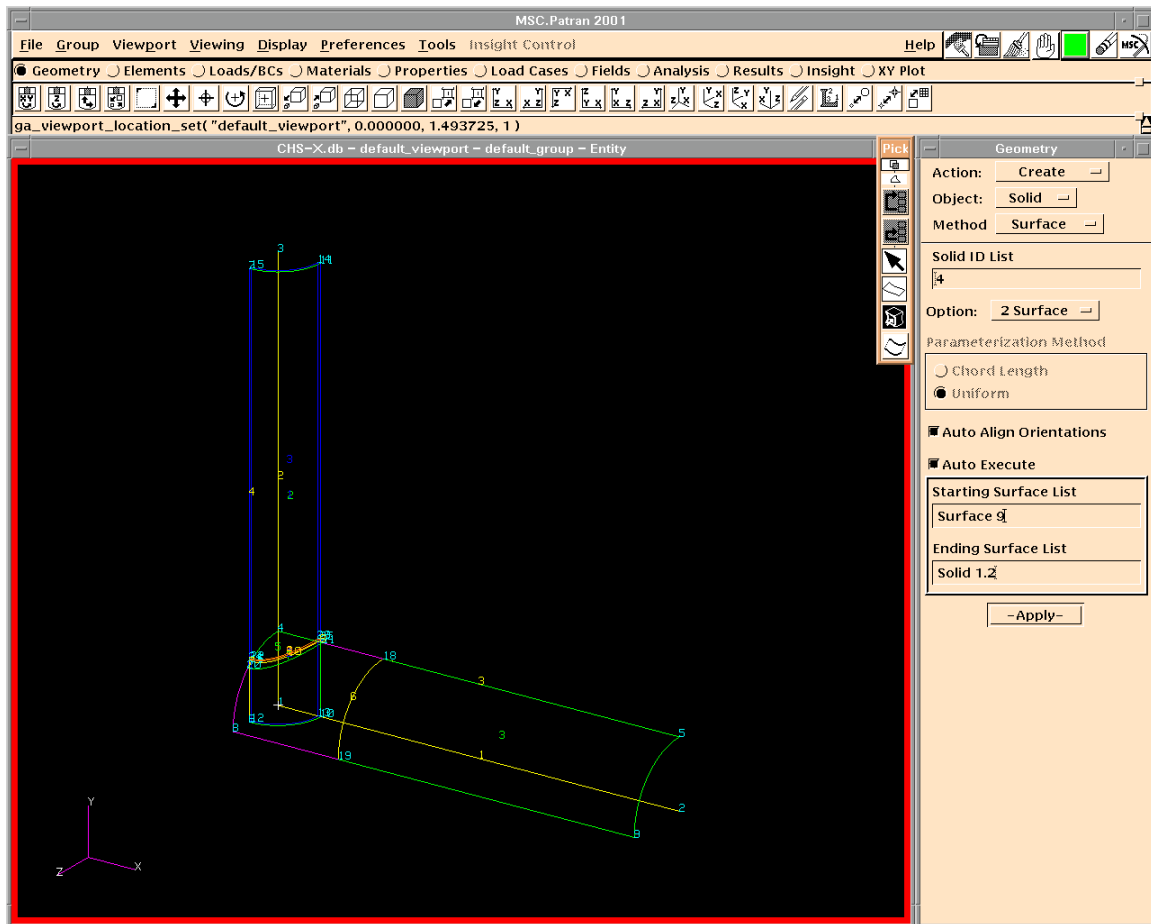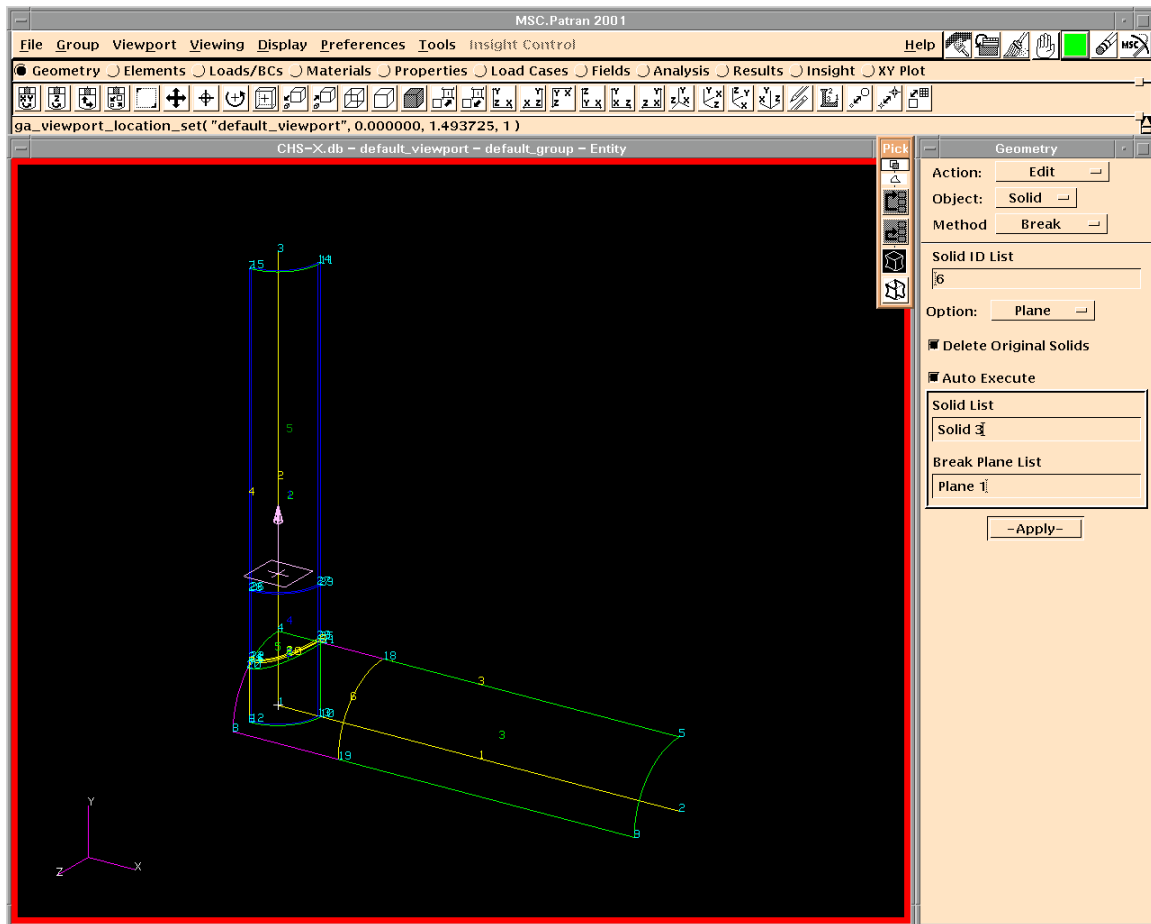
Create solids using two surfaces

- ➢ Select Surfaces 7 and 8.
- ➢ Create Solid 2 as the geometry of the weld.
- ➢ Select Surface 9 and Solid 1.2
- ➢ Create Solid 3 for the brace.

Tips

- ➢ It is always easier to create a new solid than to break an existing solid using a surface.
- ➢ Do not delete the temporary entities, unless you feel them confusing, as you may use them later.
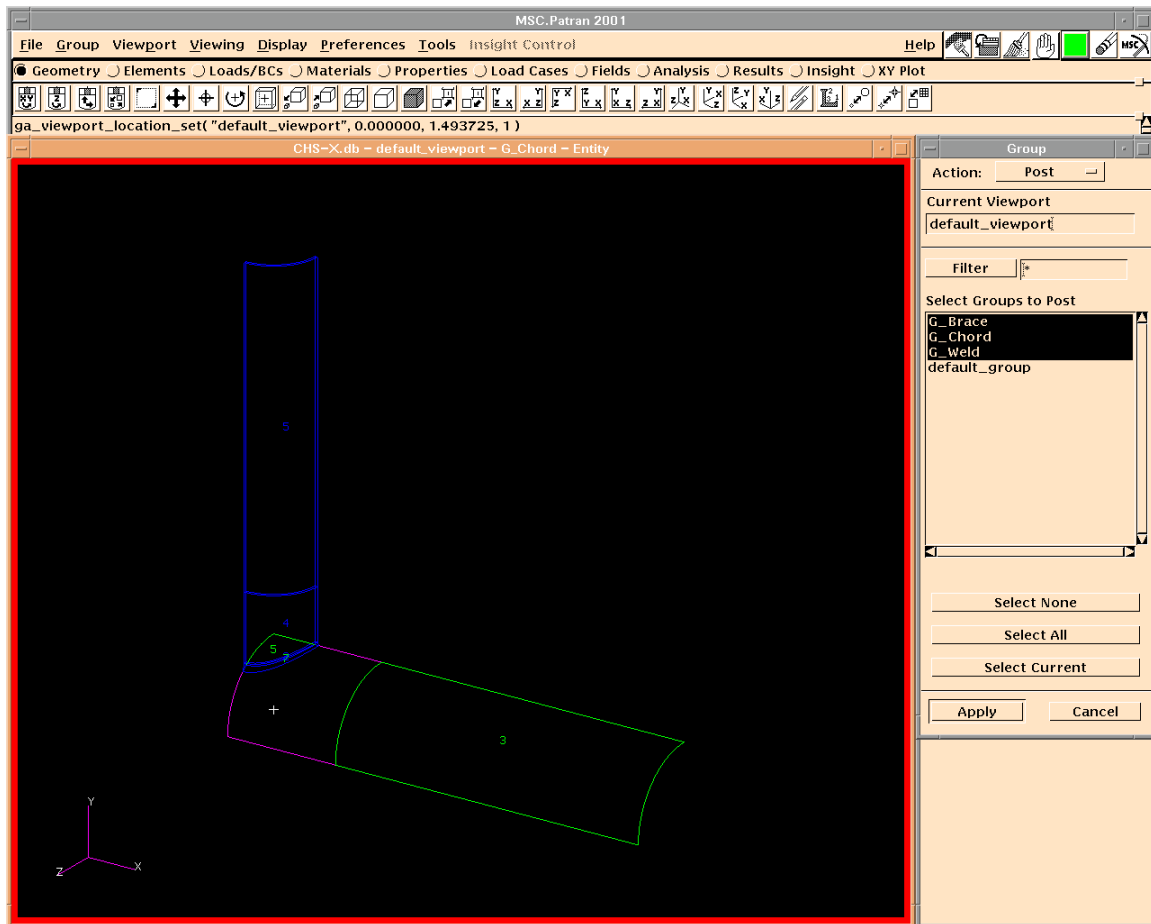
Break a solid using a plane

  ➢ Create Plane 1 at the position near the connection.

  ➢ Break Solid 3 using Plane 1 into Solids 4 and 5.

Tips

  ➢ Find the easiest way to create a plane.

  ➢ Solid 3 is broken for different mesh strategies to be used on the brace.

 ➢ Allow enough distance for the transition of different mesh strategies.

Create groups

> ➢ Go to Group ⟹ Create… to create a group for each joint member.
> ➢ Add only the necessary geometries (surfaces and solids in this case) to each group.

Tips

> ➢ Creating a group for each member makes following work easier.
> ➢ Leave temporary entities in the default group.
> ➢ Newly created group is automatically set current.
> ➢ Entities are created in the current group.
> ➢ Name of the current group is shown in the view title.
> ➢ To change the current group, go to Group ⟹ Set Current…
> ➢ Now it is ready for meshing.
> ➢ It is a good practice to follow the sequence of the menus during modelling.
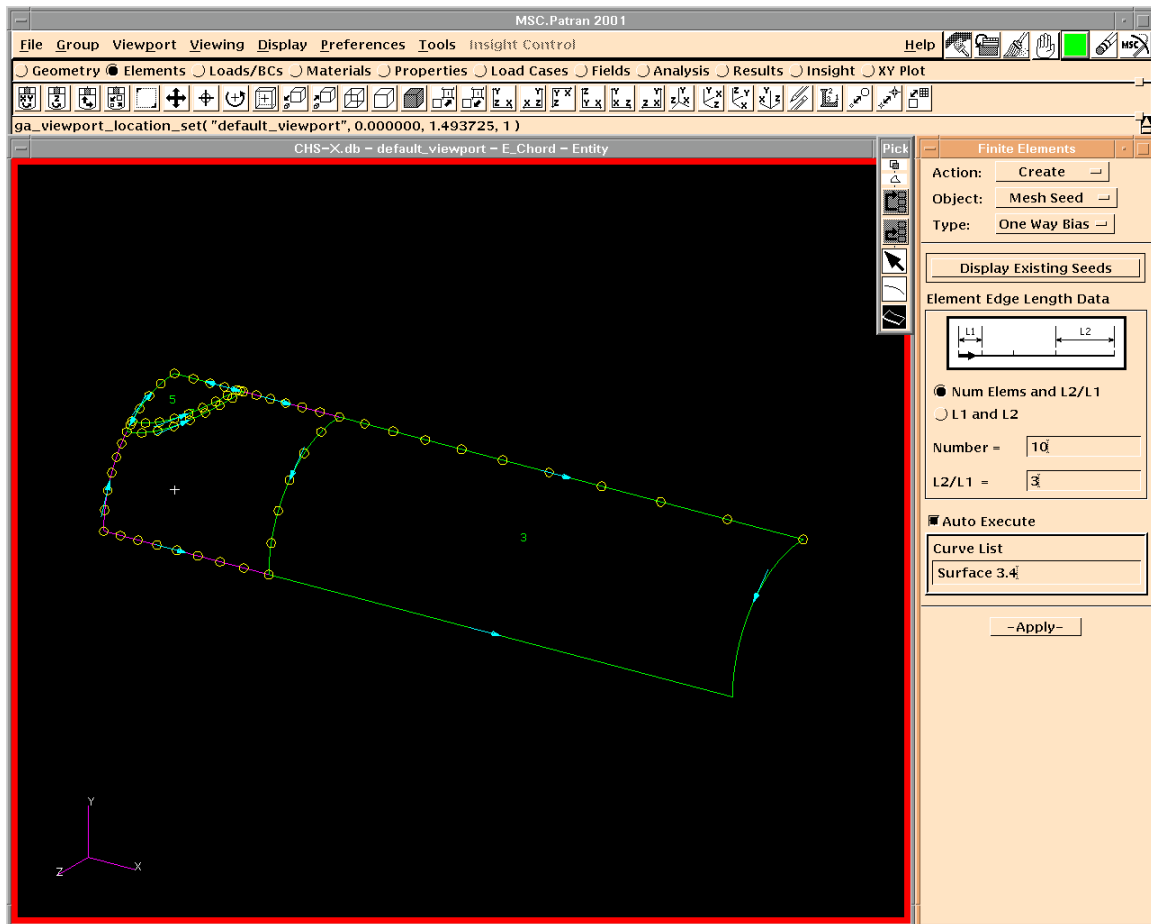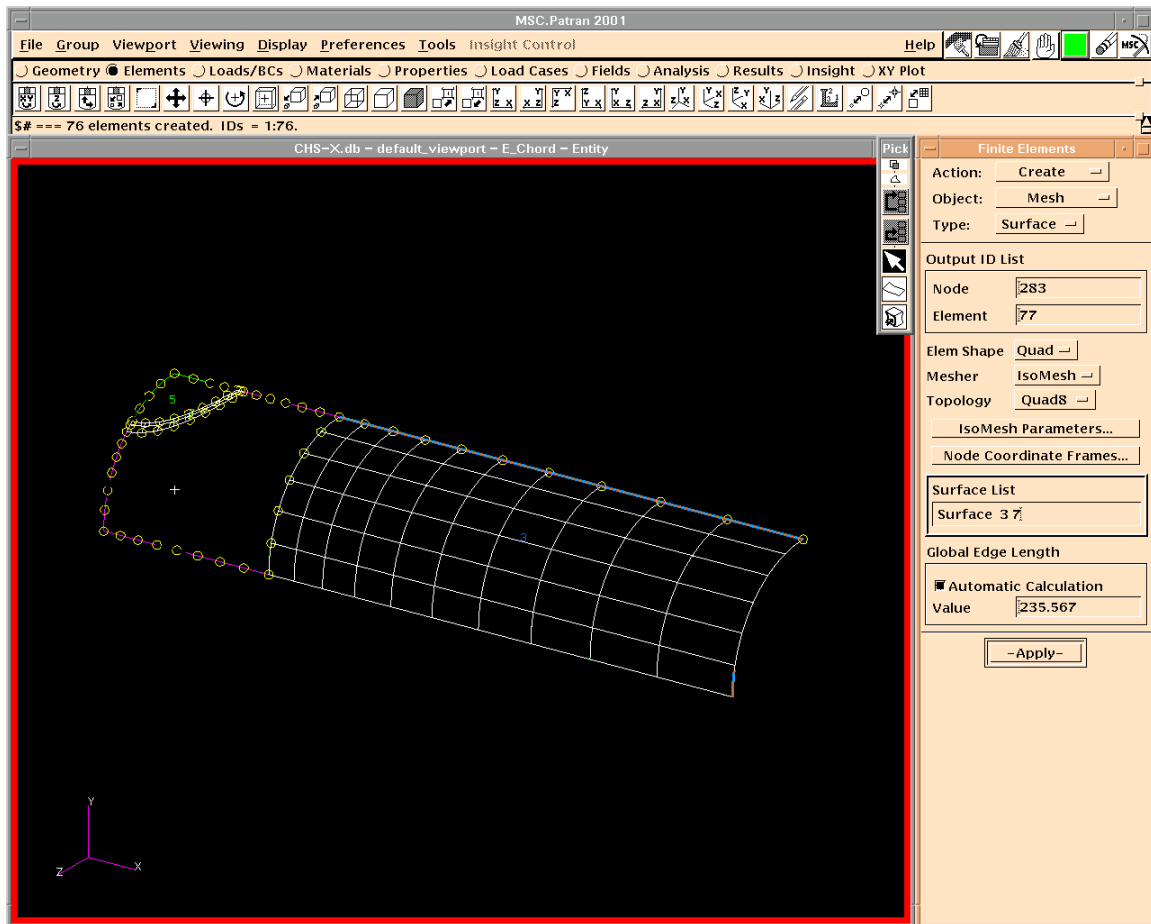
Create mesh seeds

> ➢ Post the group of the chord geometry only, and create a new group E_Chord.

> ➢ Group E_Chord is now the current group.

> ➢ Create mesh seeds on the edges

Tips

> ➢ Normally, smaller elements should be used at more critical parts. For tubular joints, mesh density may decrease with the distance from the intersection.

> ➢ Uniform is often used to create mesh seeds in the circular direction, while One Way Bias is often used in the longitudinal direction.

> ➢ For One Way Bias, an appropriate value of L_2/L_1 should be used, where L_2 is the mesh size at the end and L_1 is the mesh size at the beginning of the edge.

> ➢ Change element size gradually to make the FE model nice.

Create mesh using IsoMesh

➢ Select appropriate Element Shape, Mesher and Topology.

➢ Create shell elements on Surfaces 3 and 7 using IsoMesh.

Tips

➢ IsoMesh creates more regular meshes.

➢ Surfaces with green edges can be meshed using IsoMesh. For these surfaces, mesh seeds only need to be created on one edge in each direction.

➢ Quadratic elements are often used to study tubular joints.

➢ Try to avoid distorted elements, which, in ABAQUS, are those elements with interior angles less than 45° or greater than 135°, and may reduce accuracy of the analysis result.

➢ Sometimes, it is better to use Paver for green surfaces, especially triangular surfaces, to reduce distorted elements.
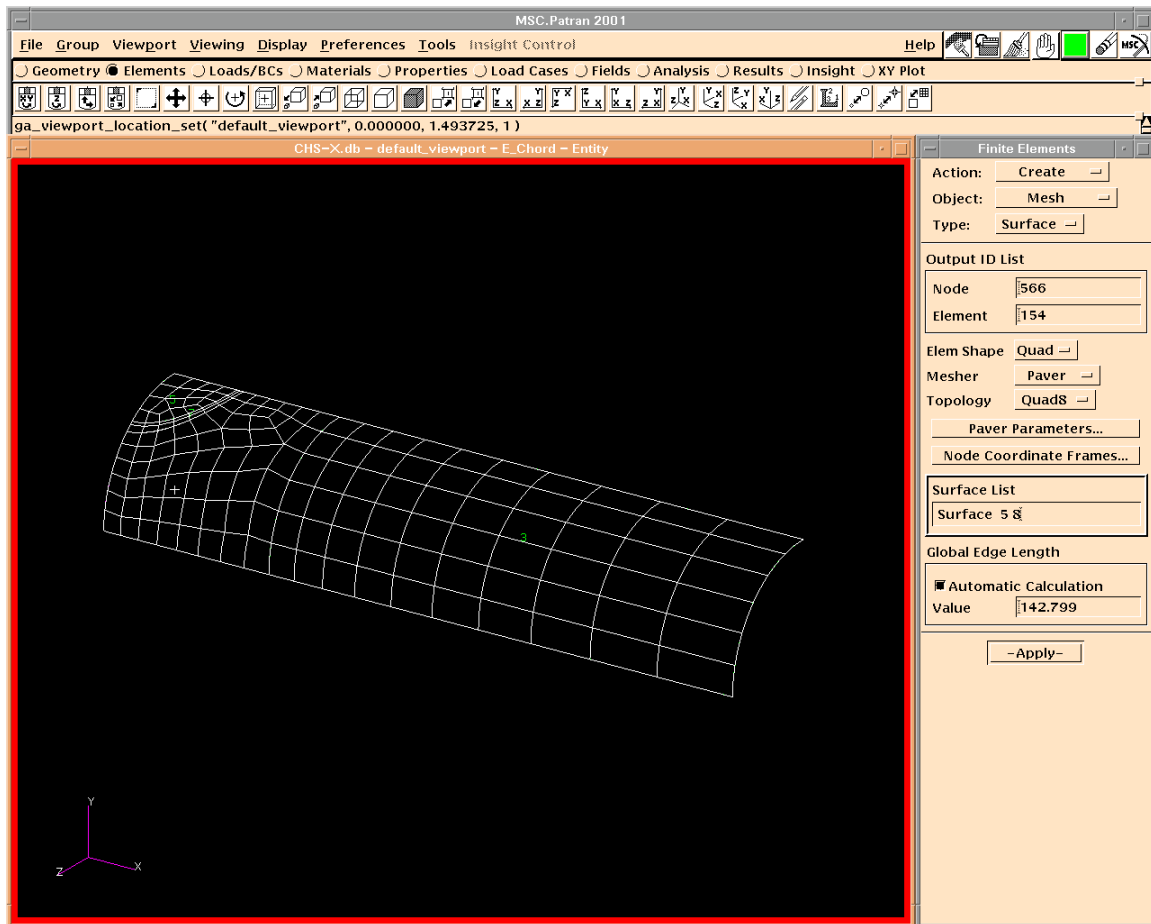
Create mesh using Paver

➢ Create Mesh Control on Surfaces 5 and 8.

➢ Select Paver as Mesher.

➢ Create shell elements on Surfaces 5 and 8.

Tips

➢ Use Paver to create proper mesh for surfaces with purple edges. Mesh Control is to determine the mesh size at the inner region of the surface.

➢ Mesh Control does not have effect on IsoMesh.

➢ For Paver, always try to create even number of mesh seeds on each edge to achieve proper mesh. If you want to use odd number of mesh seeds, use them on an even number of edges.
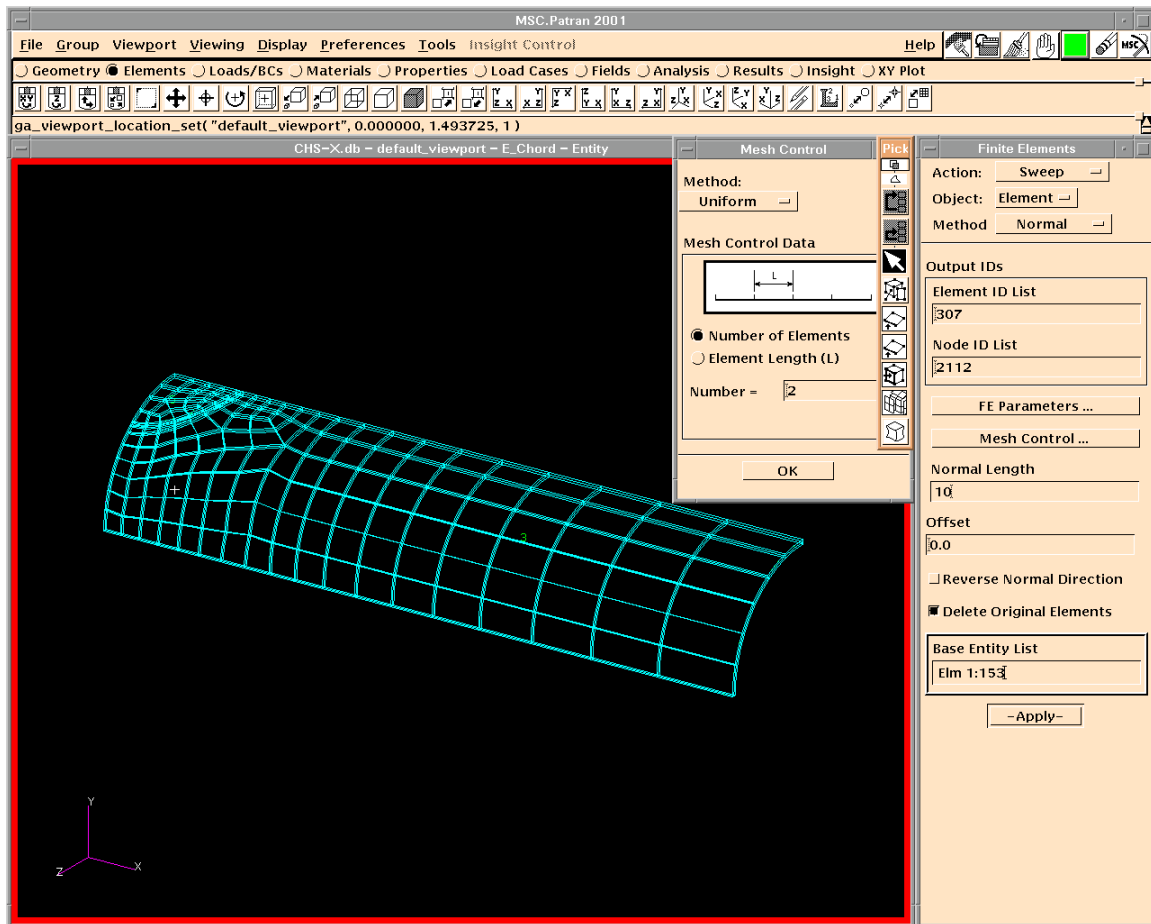
Sweep shell elements

  ➢ Input chord wall thickness as Normal Length.

  ➢ Toggle Delete Original Elements

  ➢ In Mesh Control, use two layers of elements.

  ➢ Select all shell elements to create solid elements.

Tips

  ➢ If a positive value of Normal Length is used, the shell elements sweep in their normal directions, which are the same as the normals of the associated surfaces.

  ➢ To make normal direction of all elements in the current group the same, go to Verify, Element, Normals and choose Reverse Elements.

  ➢ When you create surfaces, try to have same normal direction for all surfaces of a joint member. To display and change the normal of a surface, go back to Geometry.
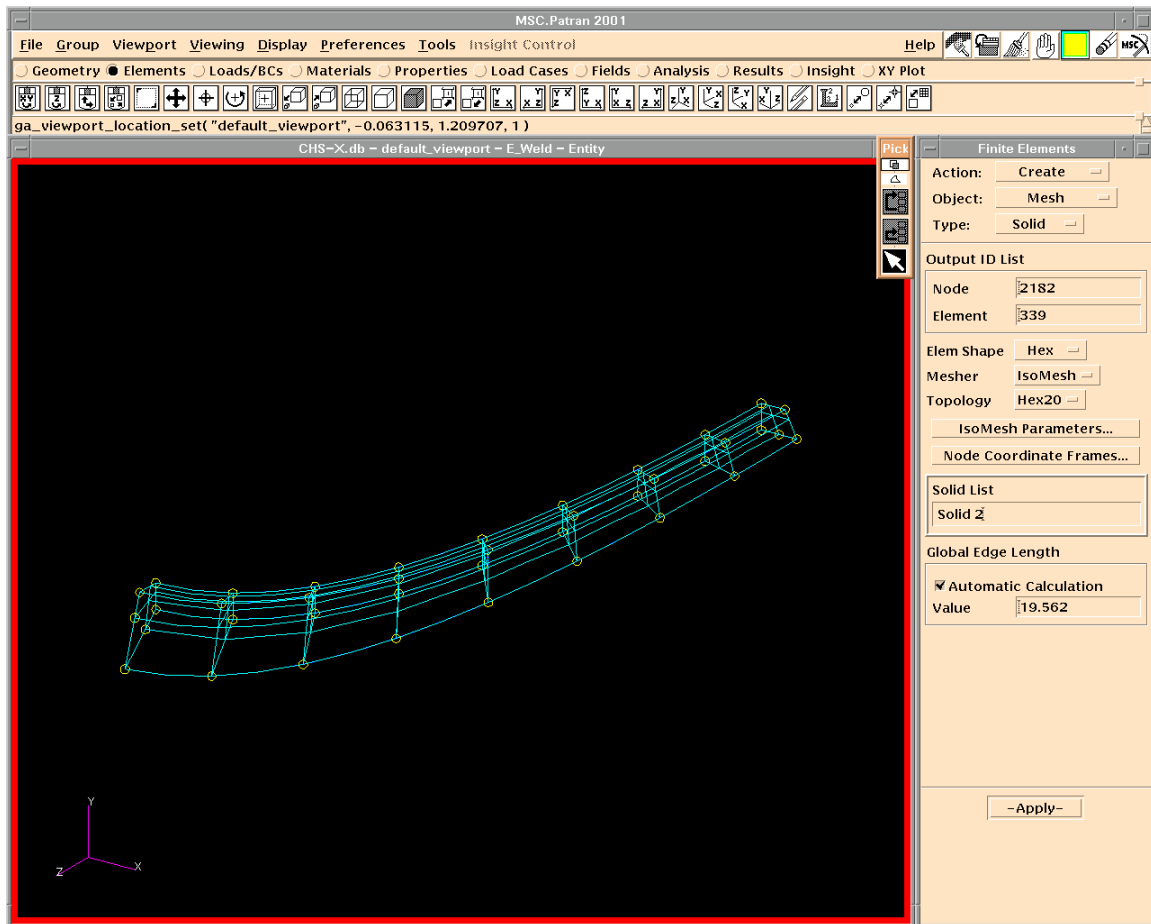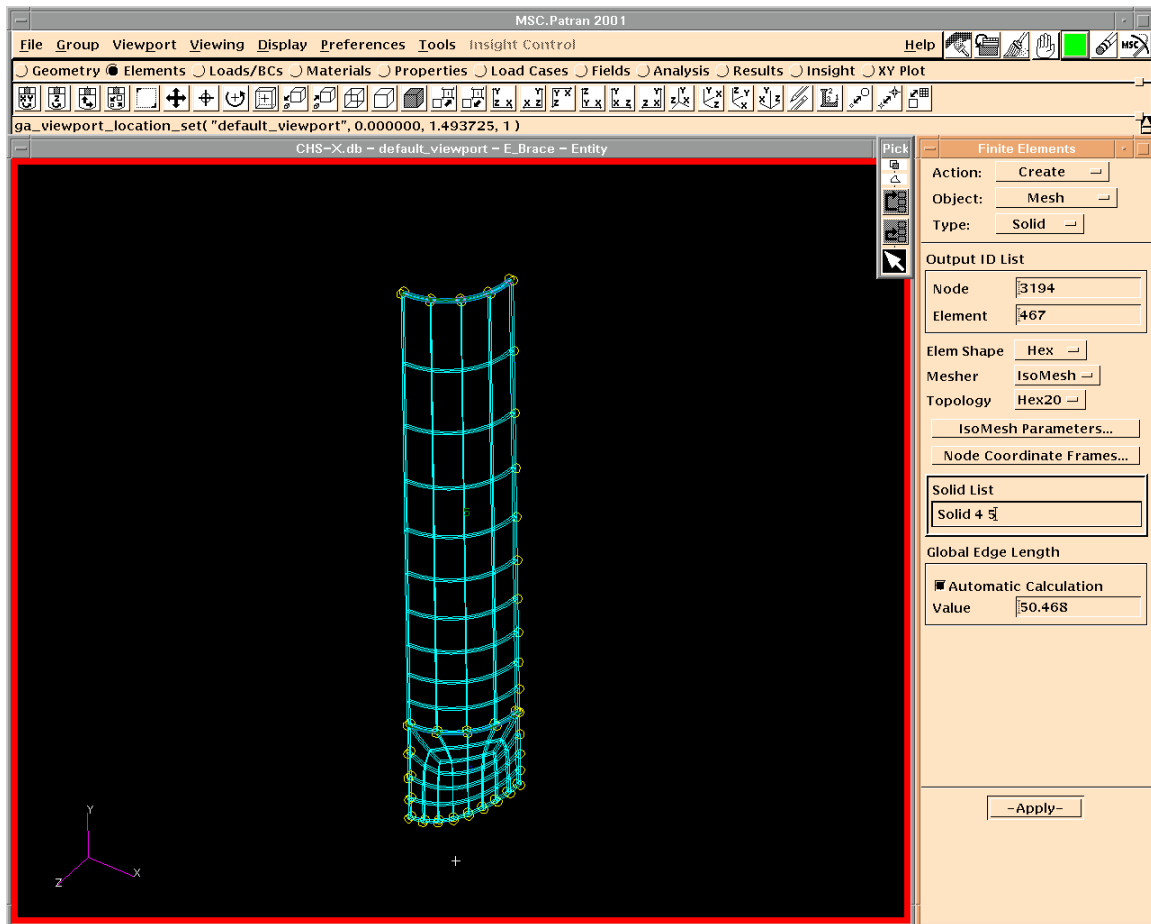
Create mesh on a solid

➢ Post the group of the weld geometry only.

➢ Create Group E_weld for the FE mesh of the weld.

➢ Create mesh seeds on edges. Use two layers of elements through the thickness.

➢ Use IsoMesh to create Hex20 elements on Solid 2.

Tips

➢ For IsoMesh, you may create mesh seed only on one edge in each direction.

➢ Make sure that same numbers of mesh seeds are used at the connections between the weld and the chord.

➢ It is a good idea to separate the FE meshes from geometries to make your model clean.

➢ Nodes and elements in each group will form node and element sets in the ABAQUS input file.

Create mesh on solids

➢ Post the group of the brace geometry only.

➢ Create Group E_Brace for the FE mesh of the brace.

➢ Create mesh seeds on edges. Use two layers of elements through the thickness.

➢ Use IsoMesh to create Hex20 elements on Solids 4 and 5.

Tips

➢ Use fewer elements at the part of the brace away from the connection to reduce the size of the FE model.

➢ Make sure that same numbers of mesh seeds are used at the connections between the brace and the weld.

➢ Using transition elements may reduce distorted elements and make your model nicer.
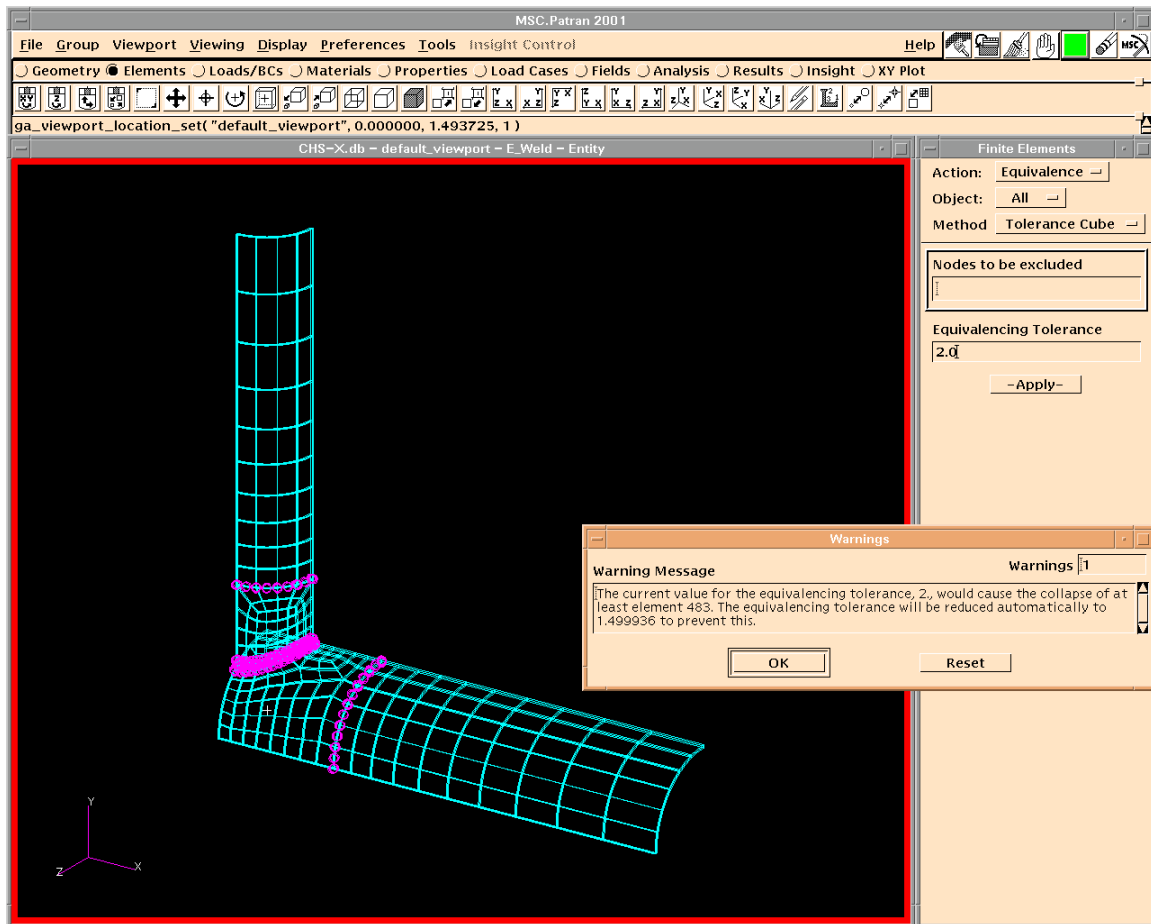
Equivalence

➢ Post the three FE groups you have created for each member of the joint.

➢ Apply Equivalence on the overall model.

Tips

➢ Equivalence is to delete duplicate nodes, and to connect different parts of the joints.

➢ Red marks indicate the positions of deleted nodes. Click Apply again if you do not like these marks.

➢ You may select Nodes to be Excluded, if you do not want to delete some duplicate nodes.

➢ You may also apply Equivalence on selected groups. This is especially useful when contact problems are involved.

➢ It is usually safer to give a relatively larger value of Equivalencing Tolerance. Patran will prompt you if this value is greater than the minimum element size.
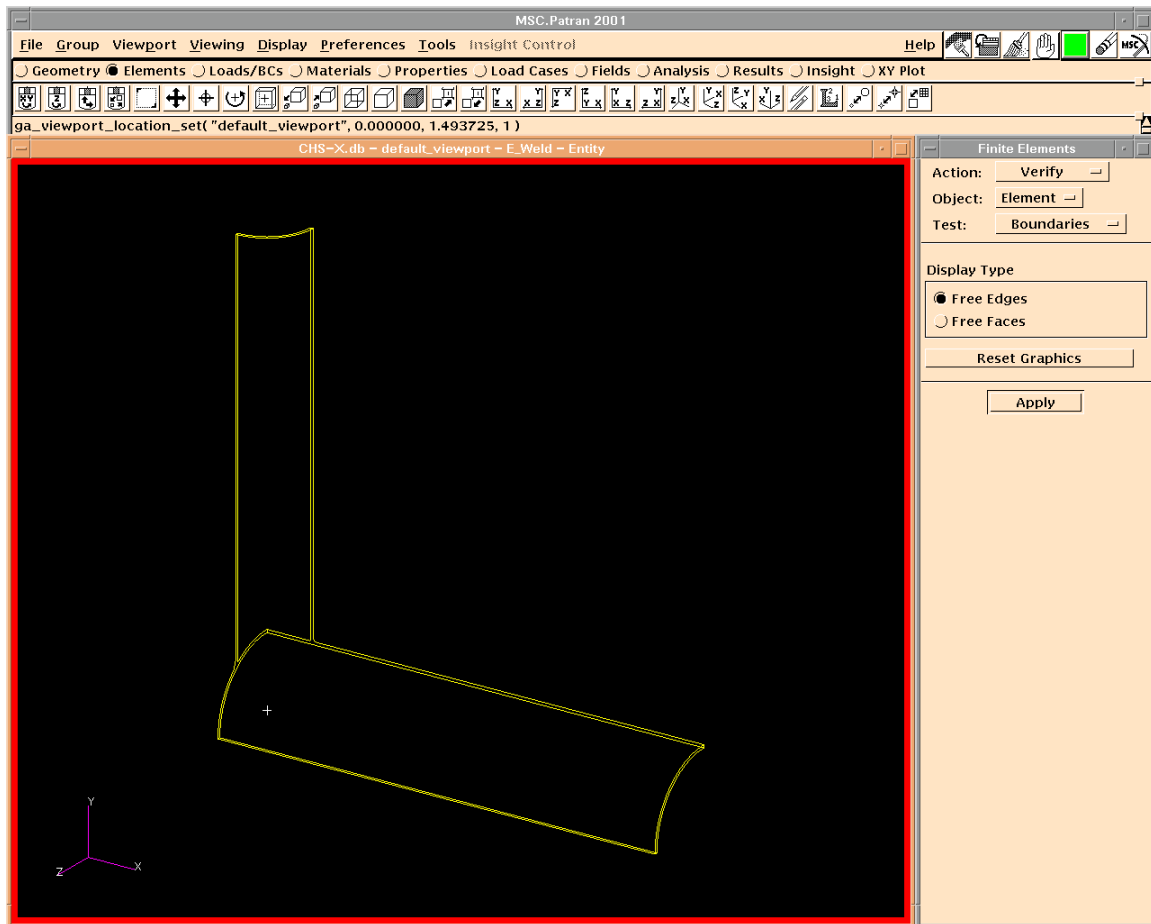
Verify boundaries

➢ Verify Free Edges and Free Face of the model.

Tips

➢ It is very important to make sure that different parts of the model are really connected.

➢ Try to find the problem if you have observed unexpected free edges. Also be careful not to connect separated parts, especially when contact problems are involved.

➢ Problems may rise from a too small global tolerance, different numbers of nodes on the common edges and different mesh strategies adopted on nearby parts.

➢ Click Reset Graphics to restore the FE mesh.
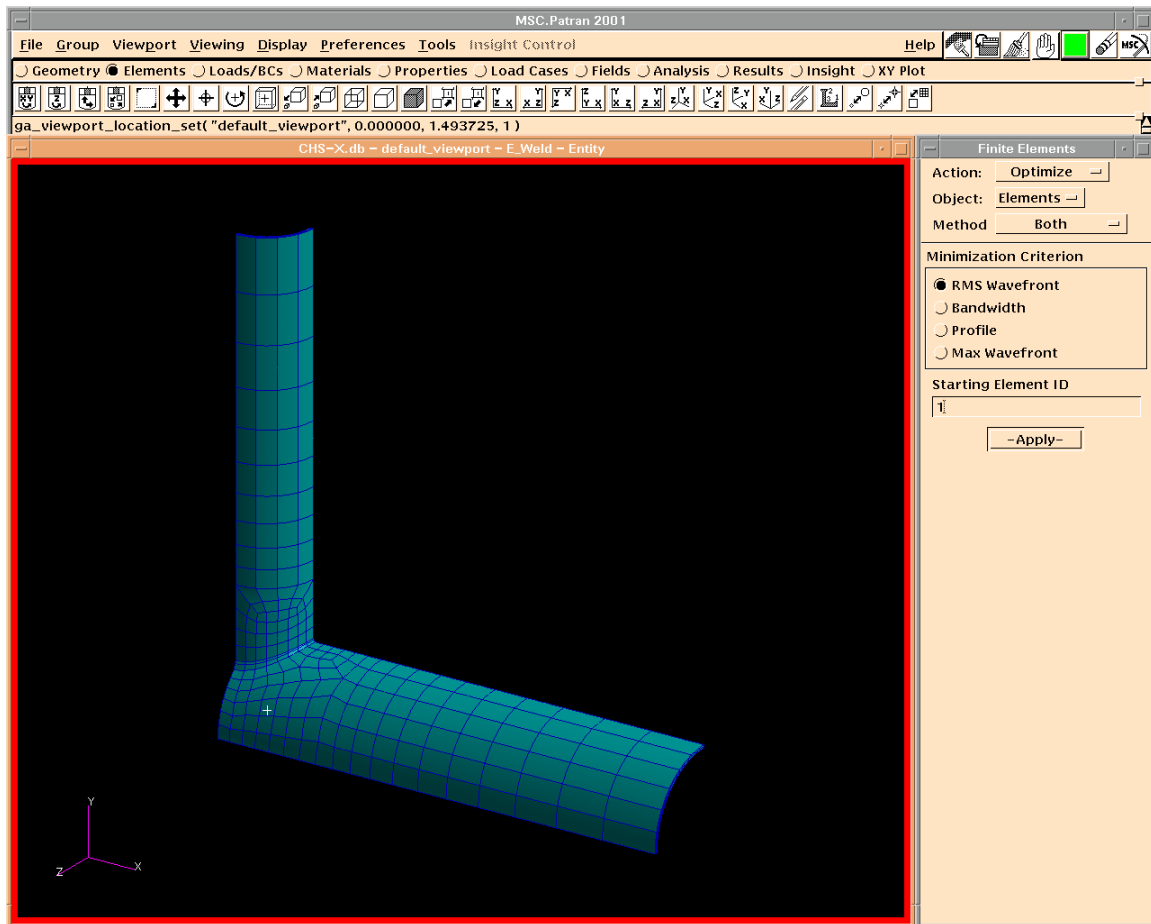
Optimize

➢ Click the icon of Smooth Shaded to have a clearer view.

➢ Optimize nodes and elements.

Tips

➢ Optimize will renumber nodes and elements.

➢ Optimize can help to reduce analysis time.

➢ You may also need to create a group for nodes at specific locations so that you can have a node set in the ABAQUS input file for you to monitor the behaviour of the loaded joint.

➢ In this case, the displacement at the Crown position of the joint is usually used to monitor the deformation of the joint.
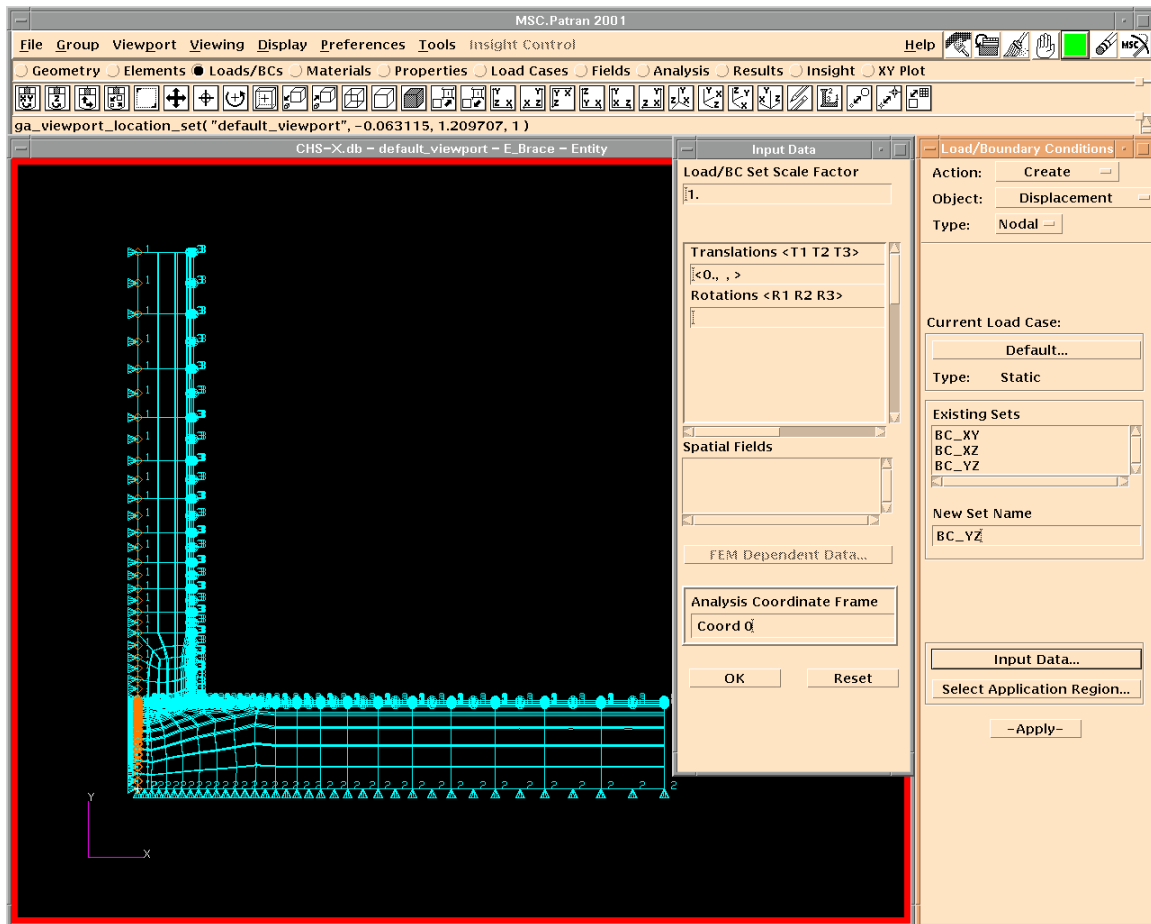
Apply boundary conditions

➢ Input prescribed displacements.

➢ Select all nodes on the plane of symmetry.

➢ Apply boundary conditions on each plane of symmetry.

Tips

➢ In this case movements of the nodes on the planes of symmetry are restrained in the corresponding planes. No additional boundary conditions are needed.

➢ Giving each set a meaningful name makes you model more readable.

➢ Solid elements have three degrees of freedom.

➢ Choose appropriate view to facilitate selecting nodes.

➢ If the elements on the boundaries have associated geometries, boundary conditions can also be applied on the geometries.
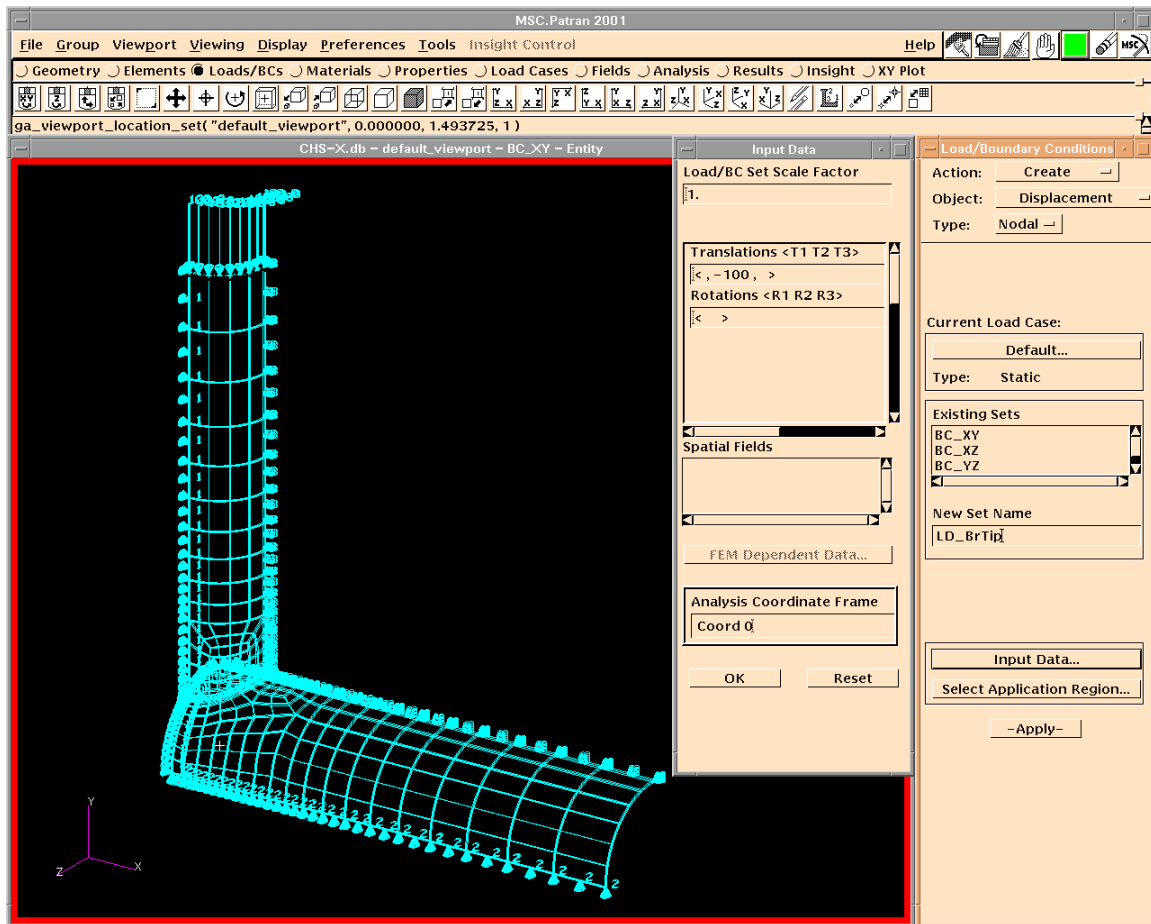
Apply Loads

- ➢ Input prescribed displacements.
- ➢ Select all nodes at the brace tip.
- ➢ Apply a downward displacement of 100 mm on the selected nodes.

Tips

- ➢ Loads can be applied either using displacement control or load control, they are equivalent from a physical point of view. Displacement control method is often preferred.
- ➢ It is a good practice to create groups of nodes and directly add and modify loading and boundary conditions on node sets in the AUBAQUS input file.
- ➢ Loading and boundary conditions specified on node sets are much easier to modify than those applied on individual nodes.
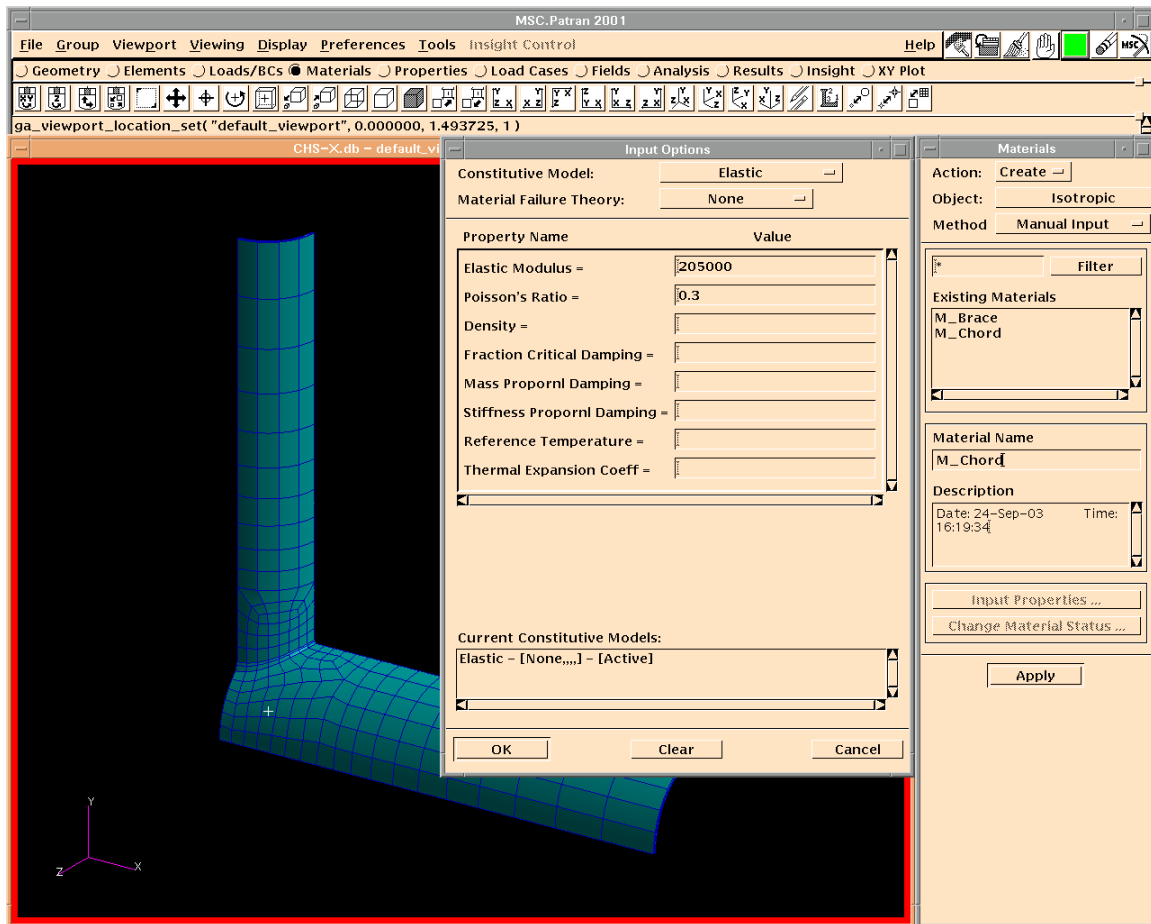
Create Materials

> ➢ Input material properties.

> ➢ Create materials for the chord and brace.

Tips

> ➢ Give each material a good name.

> ➢ Material nonlinearity may be considered by adding the plasticity part of the material property in the ABAQUS input file.

> ➢ Create material for each member, although they may have same properties. You will find it helpful in case you want to change the material properties of certain members.
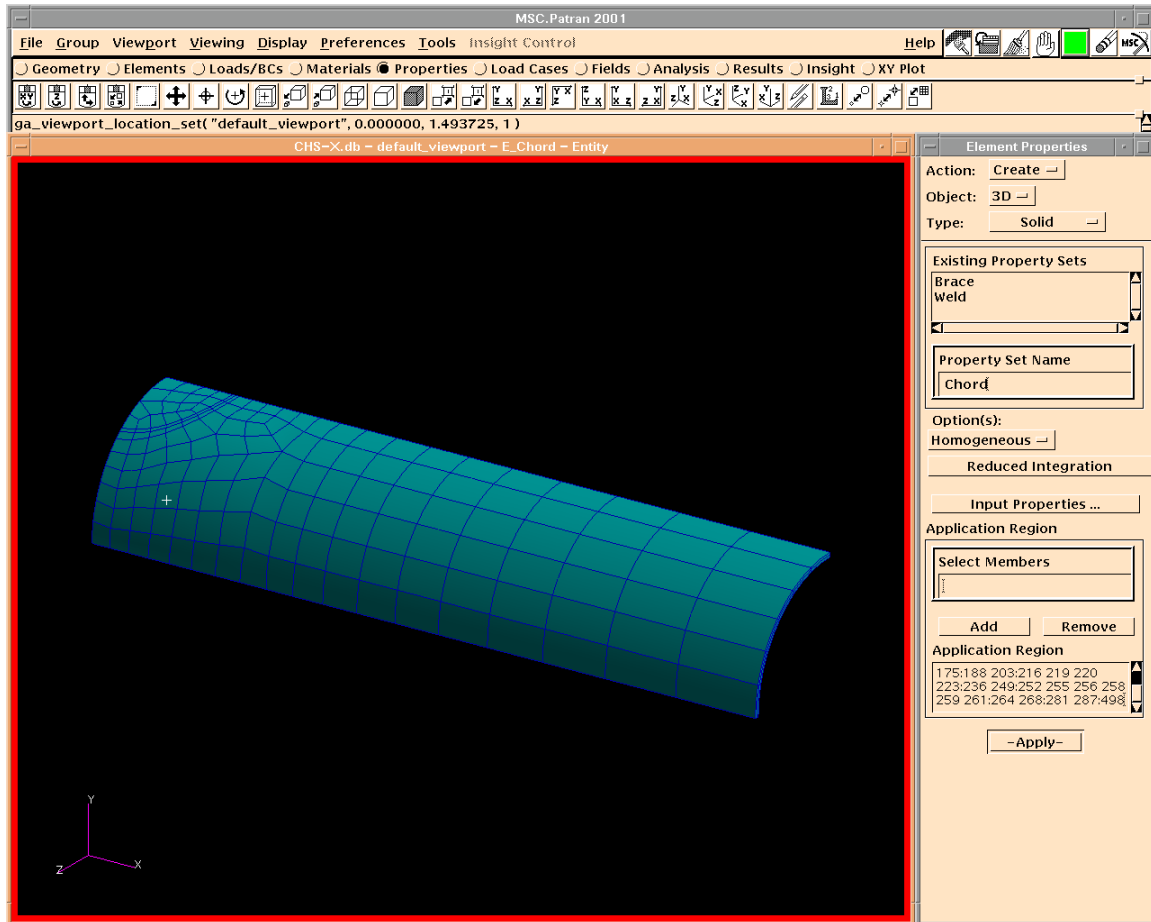
Create properties

- ➢ Choose the proper material.
- ➢ Select the corresponding solid elements.
- ➢ Create a property set for each member.

Tips

- ➢ Properties can also apply on geometries, if all elements are associated with geometries.
- ➢ Solid elements with Reduced Integration are often used for tubular joints.
- ➢ Same material can be used for the brace and the weld.
- ➢ Post only the relevant group for easier selection.
- ➢ Each set of property has a corresponding element set in the ABAQUS input file.
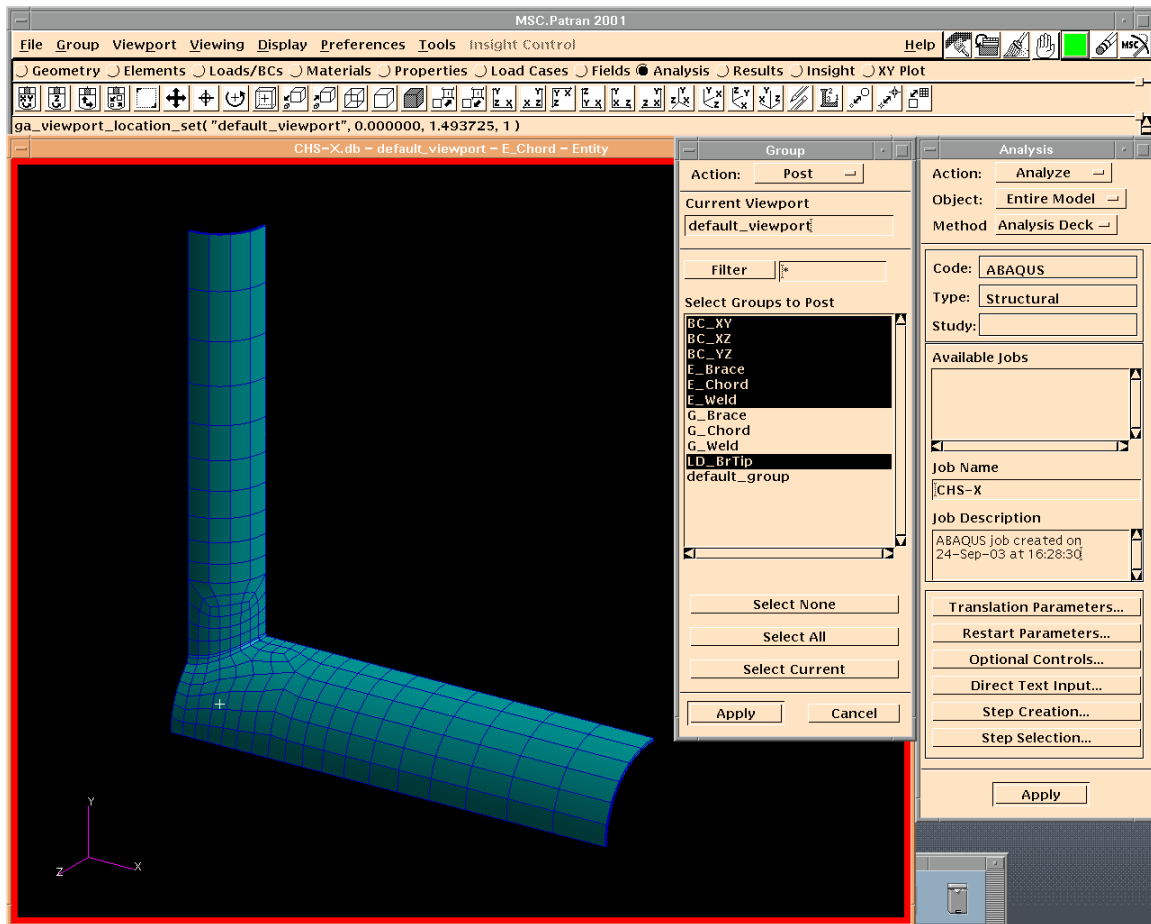
Generate ABAQUS input file

➢ Go directly to Analysis.

➢ Select Analysis Deck, since we only need the input file for ABAQUS.

➢ Use the default values for other parts on the form and click apply.

➢ Find the file "CHS-X.inp" in the working directory and modify the file where necessary.

Tips

➢ The default Load Case will be used. You can add Load Cases in the ABAQUS input file, if you need more than one case.

➢ Fields are seldom used for the FE modelling of tubular joints.

➢ It is easier to modify the .inp file than to specify everything in Patran.

➢ You can use Stream Editor to modify a textual file automatically. Type "man sed" under Unix prompt to find more information about Stream Editor.

An example of the *history data* of an ABAQUS input file

```
**
** Nonlinear analysis of a CHS X-joint under brace axial compression
**
*STEP, AMPLITUDE=RAMP, INC=50, NLGEOM
**
*STATIC
     0.02,      1.0,    0.001,      0.1
**
** Boundary condition on Plane YZ
**
*BOUNDARY, OP=NEW
  BC_YZ, 1, ,        0.
**
** Boundary condition on Plane XZ
**
*BOUNDARY, OP=NEW
  BC_XZ, 2, ,        0.
**
** Boundary condition on Plane XY
**
*BOUNDARY, OP=NEW
  BC_XY, 3, ,        0.
**
** Displacement at the brace tip
**
*BOUNDARY, OP=NEW
  LD_BRTIP, 2, ,     -100.0
**
** Print the vertical displacement of the node at the chord crown position to the data file
**
*NODE PRINT, NSET=ND_CHCRW, SUMMARY=NO, FREQ=1
U2,
**
** Print the vertical reaction force of each node at the brace tip and the total reaction force
**
*NODE PRINT, NSET=LD_BRTIP, SUMMARY=NO, TOTAL=YES, FREQ=1
RF2,
**
** Output ABAQUS pre-selected variables to the ODB file for contour plots in ABAQUS/CAE
**
*OUT PUT, FIELD, VARIABLE=PRESELECT
**
*OUT PUT, HISTORY
**
** Output selected variables to the ODB file for history plots in ABAQUS/CAE
**
*NODE OUTPUT, NSET=ND_CHCRW
U2,
**
*NODE OUTPUT, NSET=LD_BRTIP
RF2
**
*END STEP
```

About Patran Command Language (PCL)

Having created the first model and got familiar with most of the Patran commands, you may still feel frustrated if you need to create dozens of FE models, which is a common practice for parametric studies of tubular joints. It is indeed rather boring to stare at the screen and repeatedly create models of little difference: mostly, only the dimensions of the joints may change. You will be happy to learn about Patran Command Language (PCL).

PCL is a programming language embedded in MSC.Patran, which provides functions to create forms and write customized applications. A well developed PCL program can generate valid FE models automatically within minutes. An example of PCL programs for EHS joints is shown below. Contact Mr. Liang Juxiang (cveljx@nus.edu.sg) for sample PCL programs.
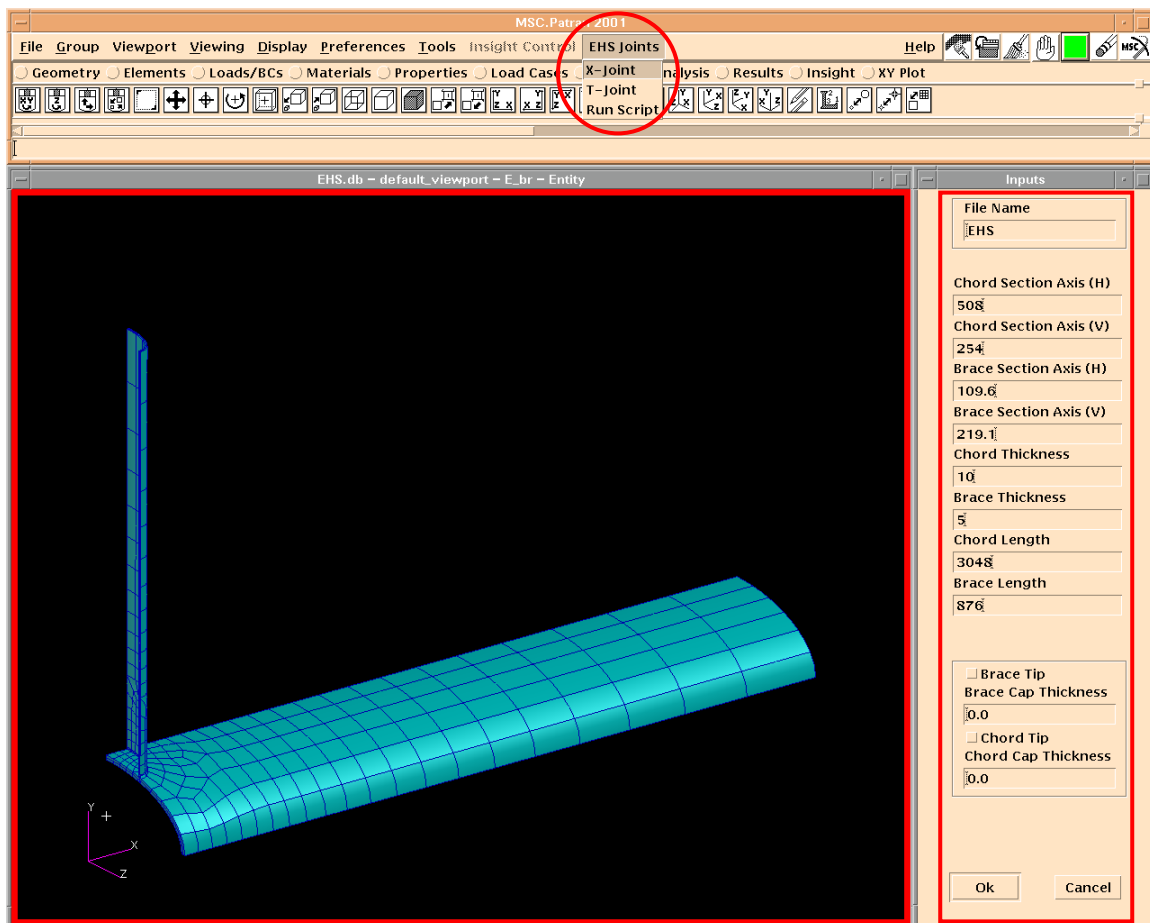
An Automatic mesh generator for Elliptical Hollow Section joints