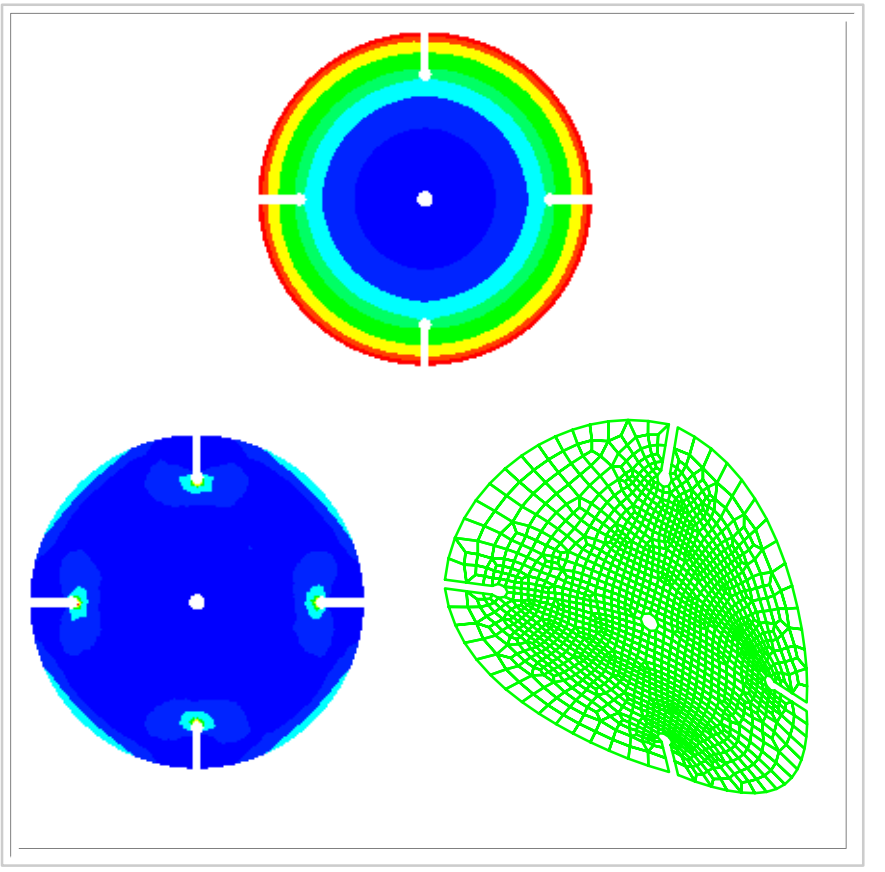


Thermal Analysis

I-DEAS™ Tutorials: Fundamental Skills

This tutorial demonstrates using linear conduction thermal analyses on a circular saw blade.

Learn how to:

- create a thermal model
- solve and display results
- calculate thermal stresses
- analyze thermal buckling

Before you begin...

Prerequisite tutorials:

- Getting Started (I-DEAS™ Multimedia Training)

  –or–

  Quick Tips to Using I-DEAS
  –and–
  Creating Parts
- Introduction to Simulation
- Managing Parts in Model Files
- Free Meshing
- Boundary Condition Sets
- Boundary Condition Surface Loads
- Boundary Condition Symmetry
- Displaying Results

If you didn't start I-DEAS with a new (empty) model file, open a new one now and give it a unique name.

File
Open

Open Model File form

Model File name: any unique name
OK

Make sure you're in the following application and task:

Simulation
Master Modeler

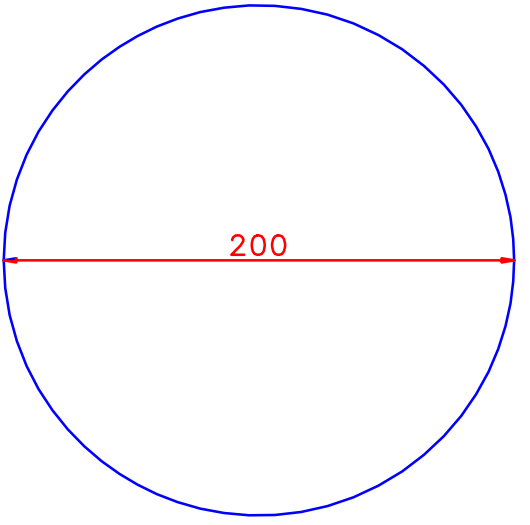
Set your units to mm.

Options
Units

mm (*milli newton*)

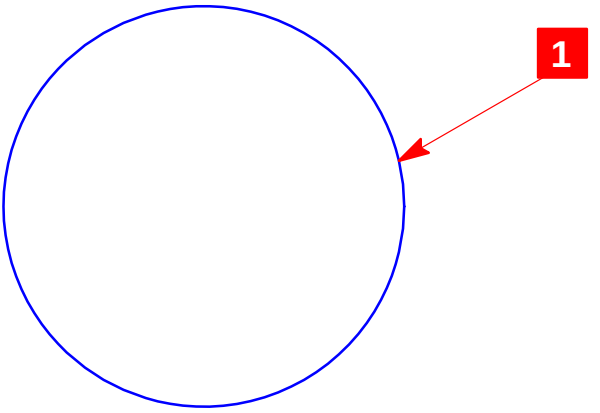
What: Sketch a circle with a radius of 100mm.

Hint



What: Create a surface bounded by the circle.

Hint

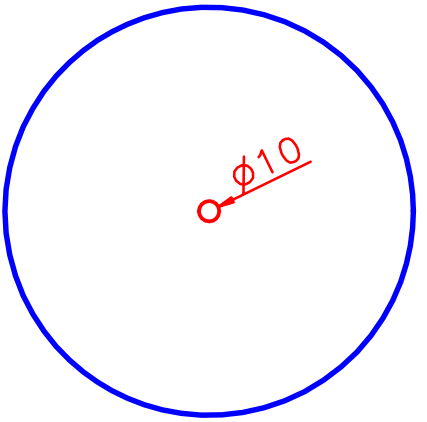


1

Yes

What: Sketch an inner circle on the surface with a radius of 5mm.

Hint



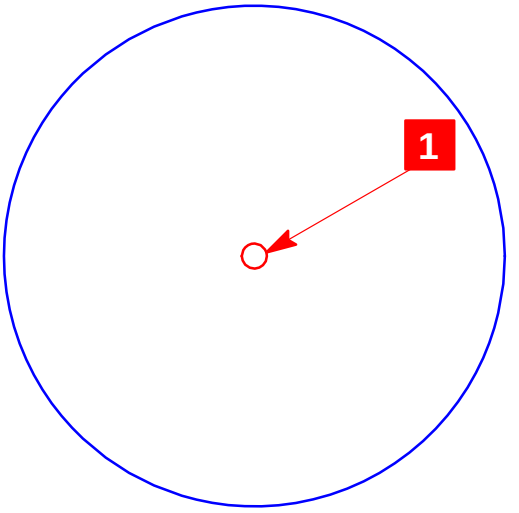
What: Cut the inner circle from the part.

Hint

1

Done

Cutout



What: Name the part.

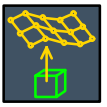
Hint

Name: Blade

What: Create an FE model.

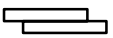
Hint


Boundary Conditions




Geometry Based Analysis Only

Save your model file.


File
Save

Warning!
If you are prompted by I-DEAS to save your model file, respond:


No

Save only when the tutorial instructions tell you to—not when I-DEAS prompts for a save.

Why:
If you make a mistake at any time between saves and can't recover, you can reopen your model file to the last save and start over from that point.

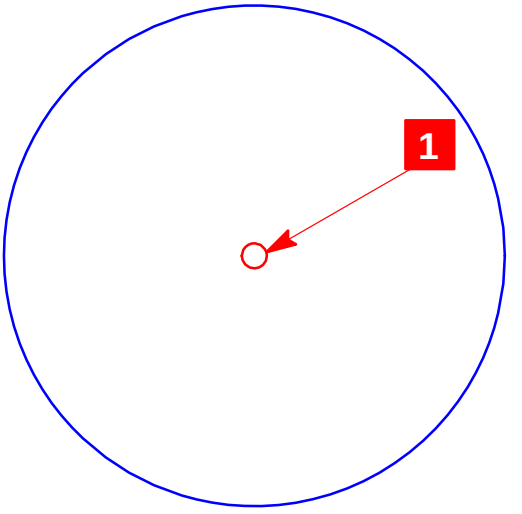
Hint
To reopen your model file to the previous save, press Control-Z.

What: Create a temperature restraint on the inner edge.

How:

Things to notice

By defining the type of analysis first, icons that don't apply are made inactive.



1

Continued on next page...

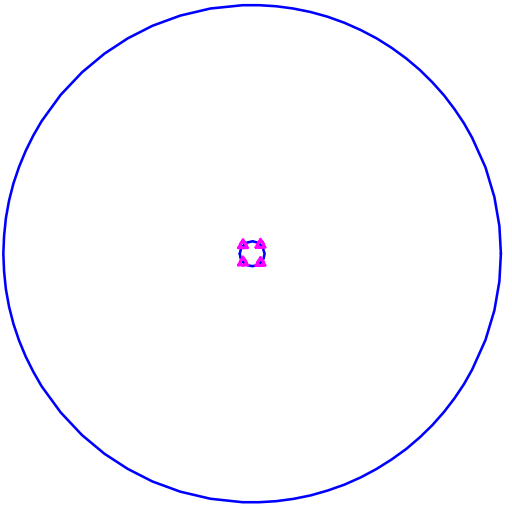
Temperature Restraint on Edge form

Restraint Set: Thermal Restraint

On Edge of...: *Thin Shell*

Temperature (Middle): 20

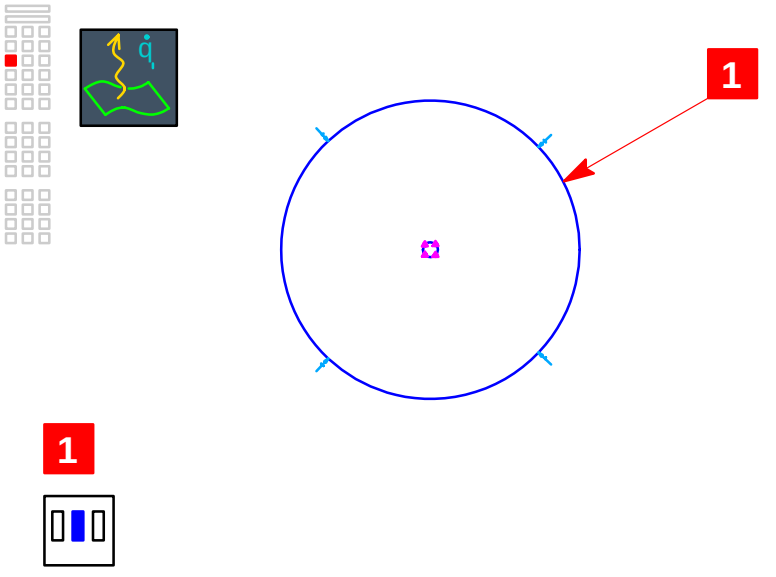
OK



Why: The blade is bolted tightly to a solid shaft which will conduct heat from the blade. This is modeled as a temperature restraint.

What: Create a heat flux on the outer edge.

How:



Heat Flux on Edge form

Load Set: Thermal Load

Heat Flux: 400000

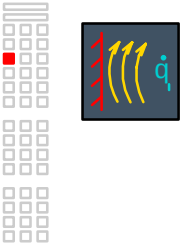
OK

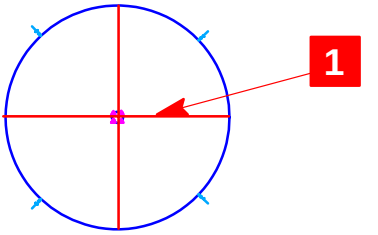
Why: The heat flux value for the edge represents 250 watts of power being dissipated by the cutting friction divided by a circumference of (.628m) = 400 W/(m sec) in SI units. In (mm–milli Newton) units, this value is 400,000.

Check the element library documentation for a description of units for the type of element you're using. Heat flux is expressed differently on surfaces or edges.

What: Define convection on both sides of the surface.

How:

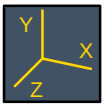


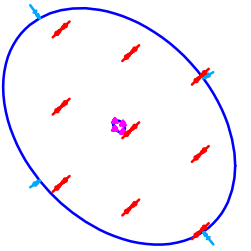


1 pick surface

Convection on Surface form

On Face of...: *Thin Shell*
Convection Coef (Top): 100
Surrounding Temp(Top): 20
Convection Coef (Bottom): 100
Surrounding Temp(Bottom): 20
OK





For help calculating convection values, see Maya's Thermal Lab at www.mayahtt.com.

What: Create a thermal analysis boundary condition set.

How:

Boundary Condition Set Management form

 Thermal

 *Heat Transfer*

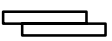
 *Restraint Set*

 *Restraint Set*: Thermal Restraint

 *Load Sets*: Thermal Load

 *OK*

Recovery Point

 *File*
Save

What: Create a physical property table.

Hint

Meshing

1st value for thickness: 1.4

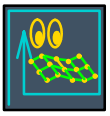
What: Mesh the surface.

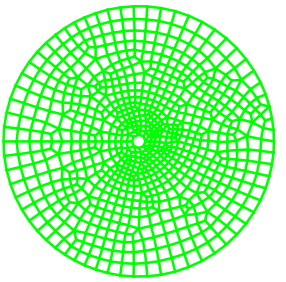
Hint

Element Length: 10

Free Options...

Percent Deviation: 10





Keep Mesh

Recovery Point

File
Save

What: Create a solution set.

Hint

Model Solution

Create

Solution Set form

Name: Thermal Solution

Type of Solution: *Heat Transfer*

OK

Dismiss

Recovery Point

File
Save

What: Solve the model.

Hint

What: Check for errors.

Hint

Things to notice

The informational warning states that a temperature load set has been stored for subsequent structural analysis. This set will be used to compute stresses and buckling load factors later in this tutorial.

To modify the model, you'll need to delete this temperature set as well as the result sets.

What: Display the resulting temperatures.

Hint

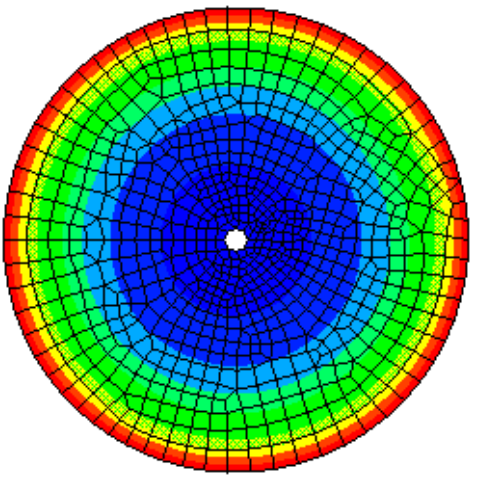

Post Processing


Temperature


Display Results


Deformation Results: *Clear*


OK



Things to notice

What's the maximum temperature at the edge of the blade?

What: Display the heat flux as an arrow plot.

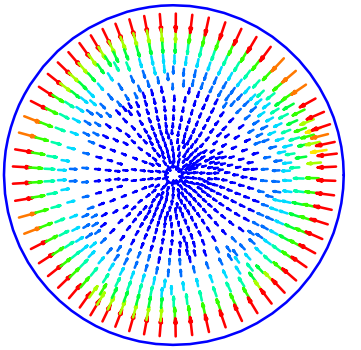
Hint


Heat Flux

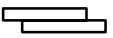

Display Results


OK


Arrow



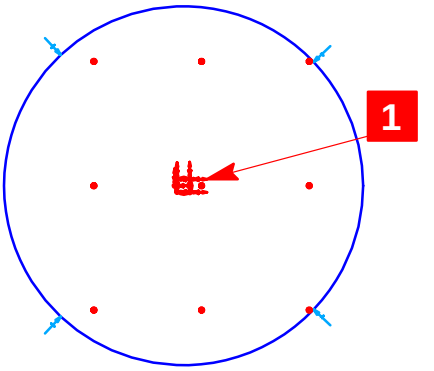
Recovery Point


File
Save

What: Restrain the center edge for the static analysis.

Hint


Boundary Conditions



1 pick center edge

Displacement Restraint on Edge form


Restraint Set: Static Restraint


OK

Why: You must enter a new name for the restraint set, to create a new set. Otherwise, you'll be attempting to add the static restraint to the thermal restraint set, which is locked by the results.

Things to notice

Elements are not shown here to simplify the illustration.

What: Create a boundary condition set for linear statics.

How:

Boundary Condition Set Management form

 Static

 *Linear Statics*

 *Restraint Set* 

Restraint Set form

 Static Restraint

 *OK*

Temperature Set 

Temperature Set form

 Load 1 Temps/Grads

 *OK*

 Don’t select any load set.

 *OK*

What: Create a solution set.

Hint

Model Solution

Manage Solutions Sets form

Create

Solution Set form

Name: Static Solution

Type of Solution: *Linear Statics*

OK

Dismiss

Recovery Point

File
Save

What: Solve the static analysis.

Hint

What: Display the resulting stresses.

Hint

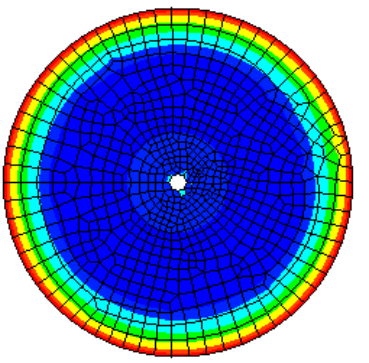

Post Processing


Stress


Display Results


Deformation Results: *Clear*


Contour



Things to notice

Is the blade in compression or tension in the outer edge? (Display maximum and minimum principal stresses to see the sign.)

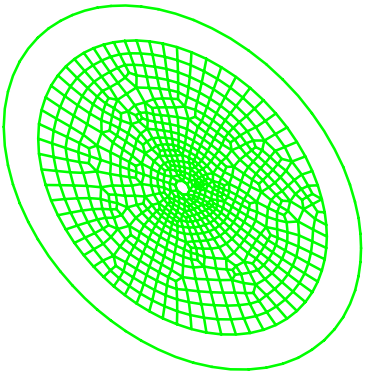
What: Display the displacements.

Hint

Display Results: *Clear*

Displacement

Deformation Results



Things to notice

The blade expands in the plane. Buckling behavior isn't predicted unless you perform a buckling analysis.

Analyze thermal buckling 1 of 4

What: Prepare to create a boundary condition set for linear buckling.

How:

Boundary Conditions

What: Create a new load set.

How:

Set Management form

Load

Set: Buckling Load Set

Create

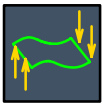
Make Current

Dismiss

Why: A load set is required for buckling, even though no loads will be supplied in addition to the thermal load set.

What: Create a boundary condition set for linear buckling.

Hint



Boundary Condition Set Management form

 Buckling

 *Linear Buckling*

 *Restraint Set* 

Restraint Set form

 Static Restraint

 *OK*

Temperature Set 

Temperature Set form

 Load 1 Temps/Grads

 *OK*

 *Load Sets*: Buckling Load Set

 *OK*

What: Create a solution set.

Hint

Model Solution

Manage Solutions Sets form

Create

Solution Set form

Name: Buckling Solution

Warning!
Be sure to define the type of solution, since buckling may not be the default in this case.

Type of Solution: *Linear Buckling*

OK

Dismiss

What: Solve the buckling analysis.

Hint

What: Display the buckling shapes and load factors.

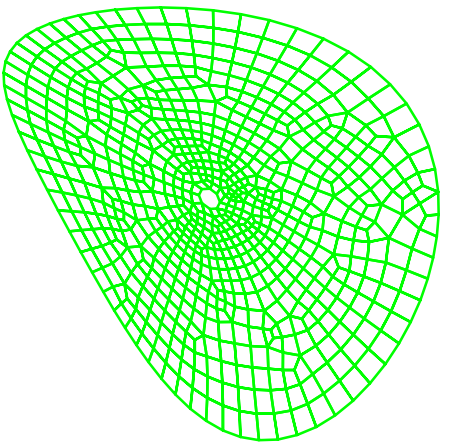
Hint


Post Processing


Display Results: *Clear*


Normal Mode 1


Deformation Results

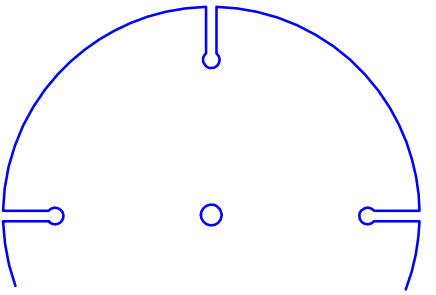


Things to notice

What's the buckling load factor? If it's less than 1.0, buckling is predicted.

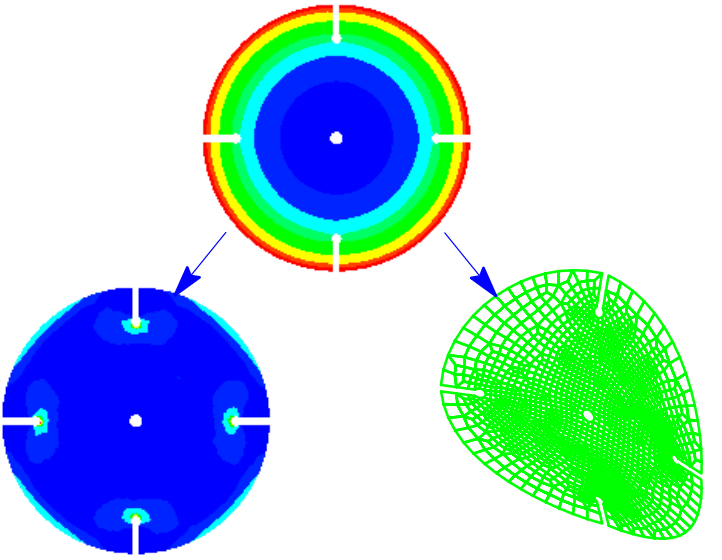
On your own...

What: To review the steps covered in this tutorial, modify the blade and repeat the analysis.



Hint

Make a copy of the blade part and add stress relief cuts. Rerun the thermal analysis, then the static and buckling analyses, using the new temperature loads.



Things to notice

Although the stress relief cuts will increase the buckling load factor, you must also make sure that the cuts don't introduce high stress concentrations.

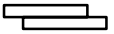
Tutorial wrap-up

You've completed the Thermal Analysis tutorial.

Delete or put away the FE model and the part. This part is not used in any other tutorials.

See also...

For additional information on the concepts covered in this tutorial, see the following:

 *Help, Manuals, Table of Contents*

Simulation: Finite Element Modeling User's Guide
Solving (heat transfer structural loads)
Applying Boundary Conditions
Applying Heat Transfer Loads
Defining Temperatures (as a structural load)
Simulation: Model Solution/Optimization User's Guide
Using the Solvers
Using Heat Transfer Analysis

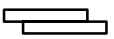
Related information for TMG and ESC is found in:

Simulation: Thermal Analysis User's Guide
TMG Thermal Analysis,
ESC I-DEAS Electronic System Cooling

What's next?

Other Advanced Projects tutorials introduce different element types and solution methods, including a series of tutorials on TMG and ESC.

To exit this tutorial, select:

 *File*
Exit

Warning!
Do not use the menu in the *I-DEAS Icons* window to exit. Use the File, Exit menu in the Acrobat Reader window.

I-DEAS Master Series™ Online Tutorials

Structural Dynamics Research Corporation
2000 Eastman Drive
Milford, Ohio 45150
(513) 576−2400